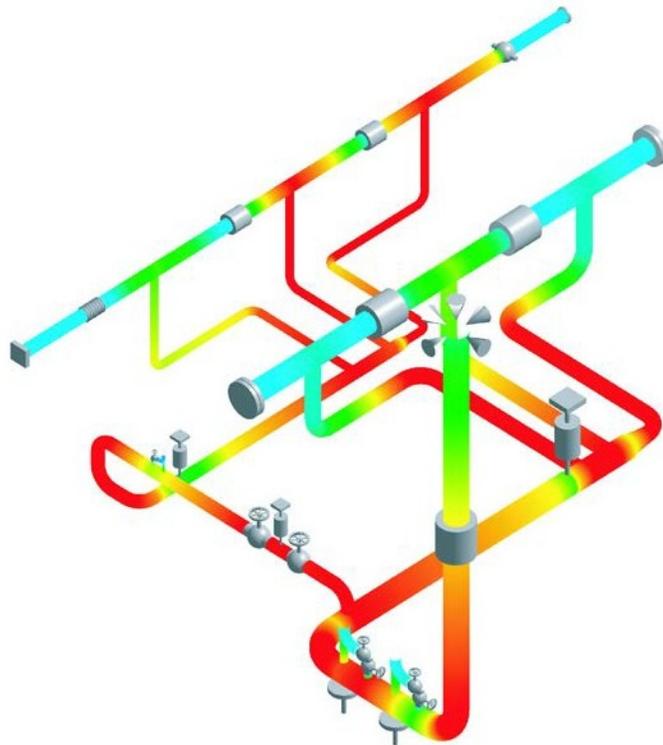


CAEPIPE™

User's Manual



SYSTEMS, INC.

The FASTEST Solutions for Piping Design and Analysis

Disclaimer

Please read the following carefully:

This software and this manual have been developed and checked for correctness and accuracy by SST Systems, Inc. However, no warranty, expressed or implied, is made by the authors or by SST Systems, Inc., as to the accuracy and correctness of the manual or the functioning of the software and the accuracy, correctness and utilization of its calculations. Users must carry out all necessary tests to assure the proper functioning of the software and the applicability of its results. All information presented by the software is for review, interpretation, approval and application by a Registered Professional Engineer.

CAEPIPE and CAdvantagE are trademarks of SST Systems, Inc. All other product names mentioned in this document are trademarks or registered trademarks of their respective companies/holders.

SST Systems, Inc.
1798 Technology Drive, Ste. 236
San Jose, CA 95110, USA
Tel: (408) 452-8111
Fax: (408) 452-8388

info@sstusa.com
www.sstusa.com

Table of Contents

INTRODUCTION	1
INSTALLATION	2
MENUS	
MAIN WINDOW MENUS	12
File	12
Help	18
LAYOUT WINDOW MENUS	19
File	19
Edit	29
View	54
Options.....	56
Loads	71
Misc.....	83
Window.....	98
GRAPHICS WINDOW MENUS	100
File	100
View	105
Options.....	111
RESULTS WINDOW MENUS	114
File.....	114
Results.....	122
View	126
REFERENCE	
ANCHOR	134
BEAM	142
BELLOWS	155
BEND	157
BRANCH SIF	174
BURIED PIPING	177
COLD SPRING (CUT PIPE)	186
COMMENT	188
COMPRESSOR.....	189
CONCENTRATED MASS.....	192
CONSTANT SUPPORT.....	193
DATA TYPES	194
DIRECTION	195
DYNAMIC ANALYSIS.....	198
DYNAMIC SUSCEPTIBILITY	205
ELASTIC ELEMENT	207
FIBER REINFORCED PLASTIC PIPING (FRP)	225
FLANGE.....	230
FLANGE AND BOLT STRESSES	234
FORCE	246
FORCE SPECTRUM.....	248
FROM (NODE)	254
GENERIC SUPPORT	256
GRAPHICS WINDOW.....	259
GUIDE	269
HANGER	271
HINGE JOINT	276
HYDROTEST	282

Table of Contents

JACKET END CAP.....	284
JACKETED PIPING.....	285
LAYOUT WINDOW.....	291
LIMIT STOP.....	301
LOAD.....	311
LOCAL COORDINATE SYSTEM.....	315
LOCATION.....	319
MATERIAL.....	321
MISSING MASS CORRECTION.....	331
MITER BEND.....	333
NODE.....	342
NONLINEARITIES.....	343
NOZZLE.....	347
PUMP.....	355
REDUCER.....	363
RESTRAINT.....	370
RESULTS WINDOW.....	371
RIGID ELEMENT.....	396
ROD HANGER.....	398
SECTION.....	400
SKewed RESTRAINT.....	403
SLIP JOINT.....	409
SNUBBER.....	412
SPIDER.....	413
SUPPORTS.....	414
TEES.....	417
TIE ROD.....	420
TURBINE.....	422
USER HANGER.....	426
USER SIF.....	428
VALVE.....	429
WELD.....	435

APPENDICES

A - CODE COMPLIANCE

B31.1 (2012).....	438
B31.3 (2012).....	447
B31.1 (1967).....	455
B31.4 (2012).....	458
B31.5 (2013).....	465
B31.8 (2012).....	470
B31.9 (2008).....	477
ASME (1980).....	479
ASME (1986).....	484
ASME (1992).....	489
BS 806 (1986).....	494
EN 13480 (2012).....	498
NORWEGIAN (1983).....	507
NORWEGIAN (1990).....	515
RCC-M (1985).....	526
CODETI (1985).....	531
STOOMWEZEN (1989).....	543
SWEDISH (1978).....	545
Z183 (1990).....	549

Table of Contents

Z184 (1992)	553
FRP STRESS.....	557
OPERATING STRESS FOR NDE	558
B - ROTATING EQUIPMENT REPORTS	
API 610	564
NEMA SM-23.....	572
API 617	574
C - IMPORT/EXPORT	
IMPORT	578
EXPORT.....	600
D - NOZZLE STIFFNESS CALCULATIONS	603
E - DYNAMIC SUSCEPTIBILITY METHOD.....	618
F - FLANGE MODULE.....	630
INDEX	648

This page is blank

Introduction

Thank you for licensing CAEPIPE (pronounced kay-pipe), the simple yet powerful software for solving a variety of piping design and stress analysis problems in several industries such as energy, process and aerospace.

CAEPIPE performs linear and nonlinear static and linear dynamic analyses of piping systems by imposing various loads such as deadweight, thermal, seismic, wind, spectrum, time history or harmonic, and calculates displacements, forces, moments, stresses, support loads etc. Further, it checks whether the piping system is piping code and guideline compliant (ASME, B31, European, Swedish, API 610, etc.) and produces concise, formatted and easy to understand reports.

For rapid modeling, CAEPIPE offers you a friendly and productive user interface that rigorously adheres to Windows standards. Open up to four windows simultaneously to get feedback on different aspects of the model. Extensive graphical display capabilities allow you to zoom, pan, rotate the image and see the model from different viewpoints. CAEPIPE uses the industry standard OpenGL[®] to render 3D images realistically for easy visualization. As the model is input and modified, CAEPIPE updates the graphics simultaneously to provide visual feedback. It animates deflected shapes and mode shapes, and shows color-coded stress contours, among others.

A true powerhouse in its speed of operation, CAEPIPE uses advanced Windows programming techniques such as intelligent repainting, scroll box tracking, multithreading, memory-mapped files for faster data access, among others, to make your job easier and faster. Every effort is made to keep the program and data file sizes small (e.g., program size is ~2 MB! And a 665-element piping model is 85 KB!).

Many thoughtful and useful details in the program allow you to work more productively. For example, you can annotate your model with copious comments for enhanced documentation, or duplicate repetitive input with one hotkey combination or rotate sections of the model with one operation. No unnecessary buttons clutter the toolbar nor are you forced to use a mouse unnecessarily. The many thoughtful keyboard shortcuts, too, add to your productivity.

Overall, CAEPIPE stands peerless among the tools available today for piping design and stress analysis. We invite you to explore the software so that you can make full use of its capabilities. Our friendly and knowledgeable support engineers are available to assist you. Should you need to reach them, please email: support@sstusa.com.

Three sections make up this manual:

1. Explanation of menus from the different CAEPIPE windows,
2. An alphabetically arranged Technical Reference section, and
3. Appendices with related information.

The manual ends with an index.

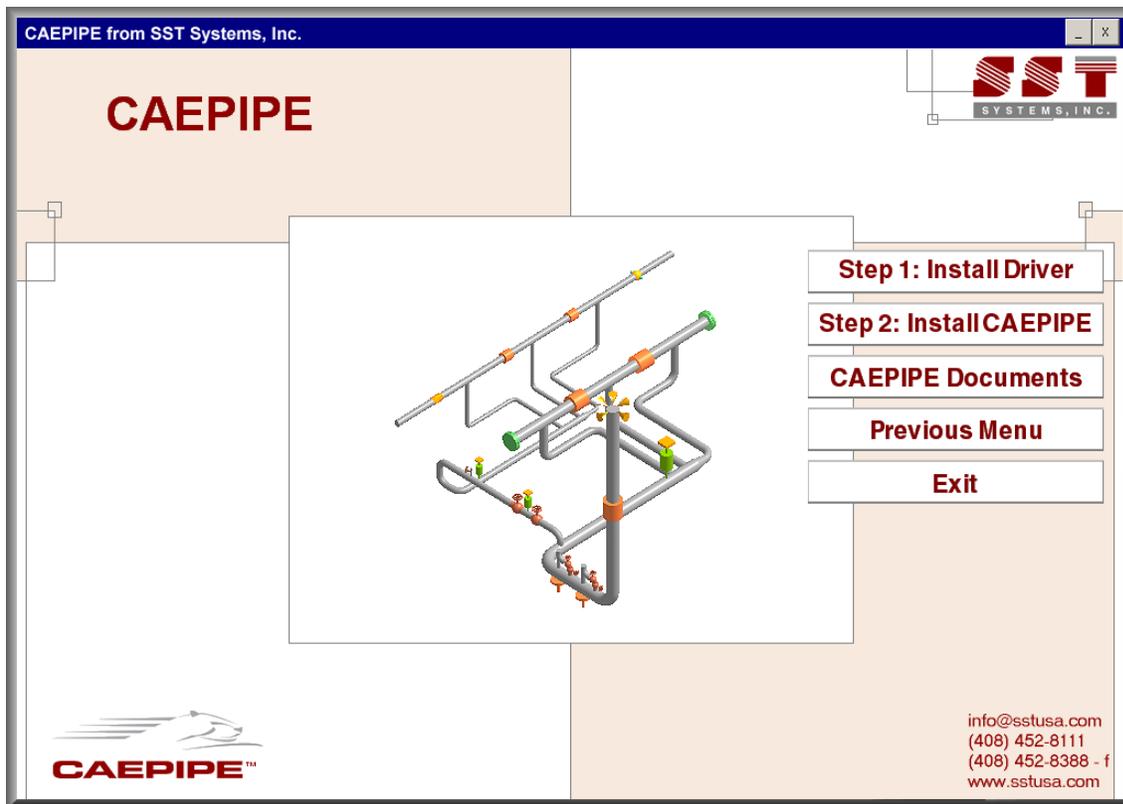
Installation

Installing CAEPIPE takes two easy and quick steps after you connect the hardware key securely to a USB port of your computer.

1. Install the hardware key driver and server software (from the CD or executable), and
2. Install CAEPIPE software.

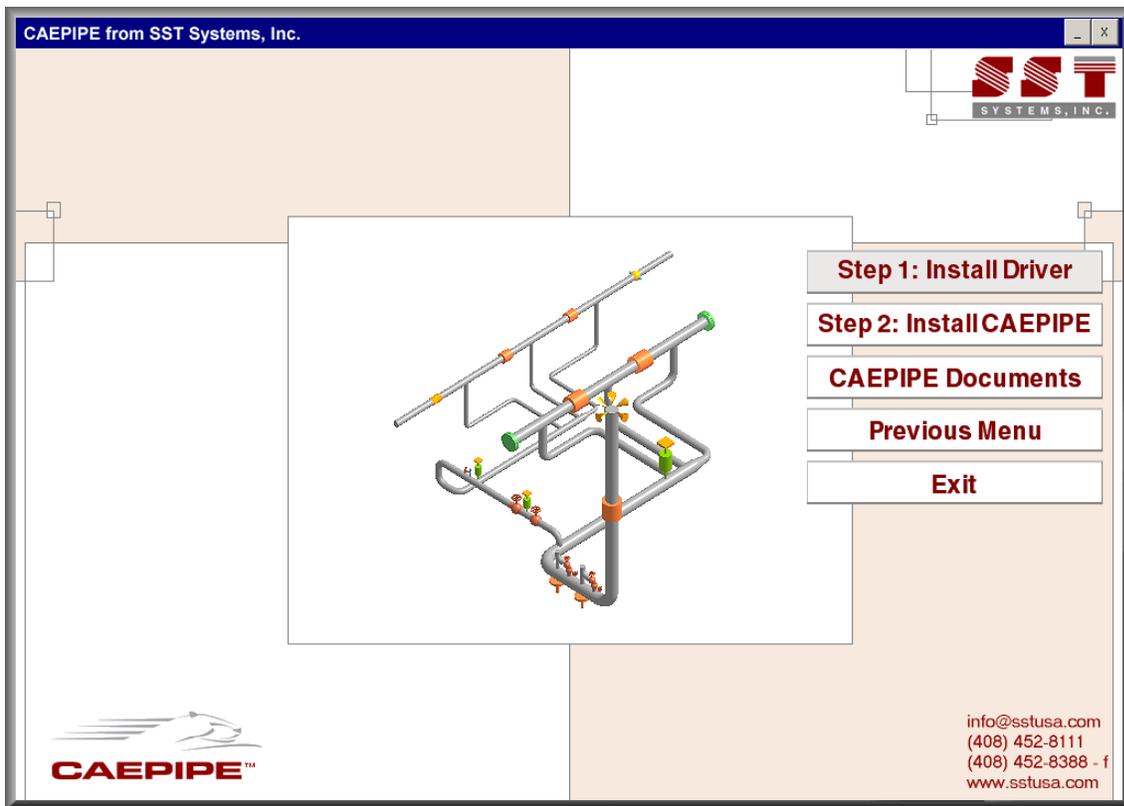
In addition to the software required for steps 1 and 2 above, the supplied Compact Disc (CD) also contains other helpful material such as an Information document with a list of technical features, a CAEPIPE Tutorial and utilities such as a hardware key finder if you have trouble. Feel free to browse and install whichever is required, or contact our support staff for assistance.

When you insert and “Autoplay” the CAEPIPE CD, it will open a window similar to that shown below on the left. Click on the CAEPIPE button (on the right). You will be shown another window similar to the image shown below on the right.

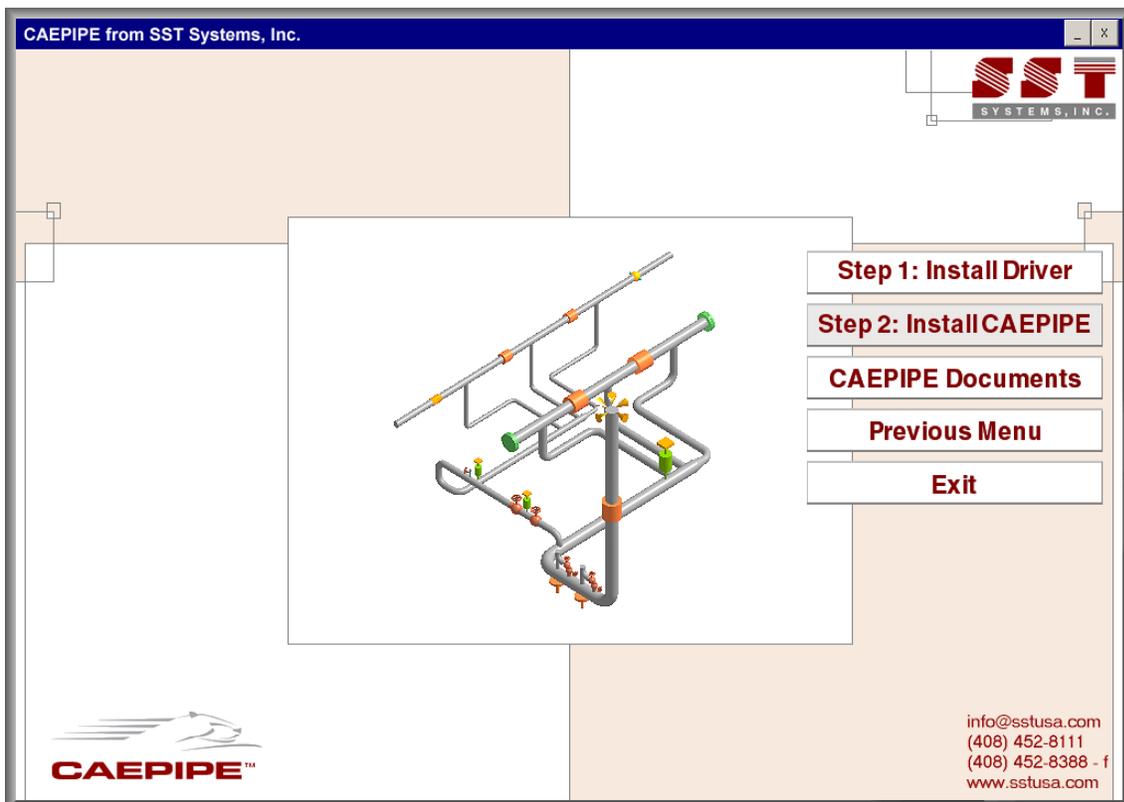


First, click on “Step 1: Install Driver” button to install the hardware key driver and server (Sentinel Protection Installer xxx.exe) and follow all the steps shown until you see confirmation of driver installation.

Installation

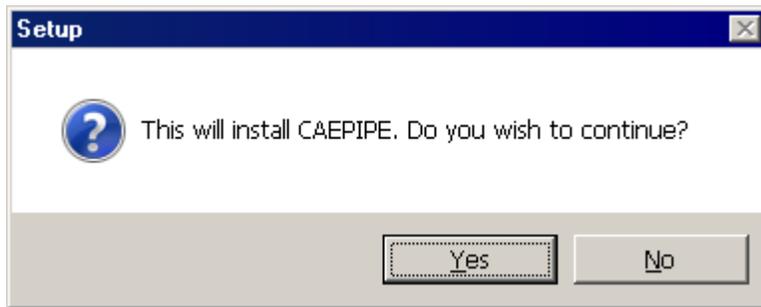


Second, click on “Step 2: Install CAEPIPE” button to install CAEPIPE software.

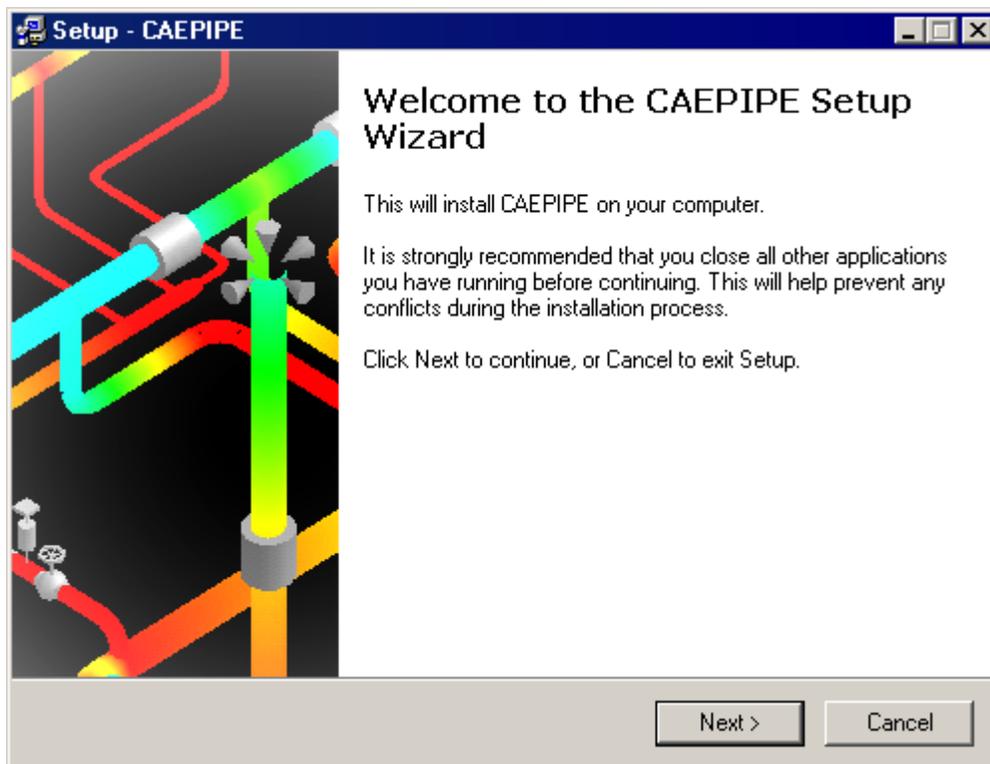


Installation

You will now see the following interactive screens one after another in which you will have to answer the usual installation questions. Respond appropriately.

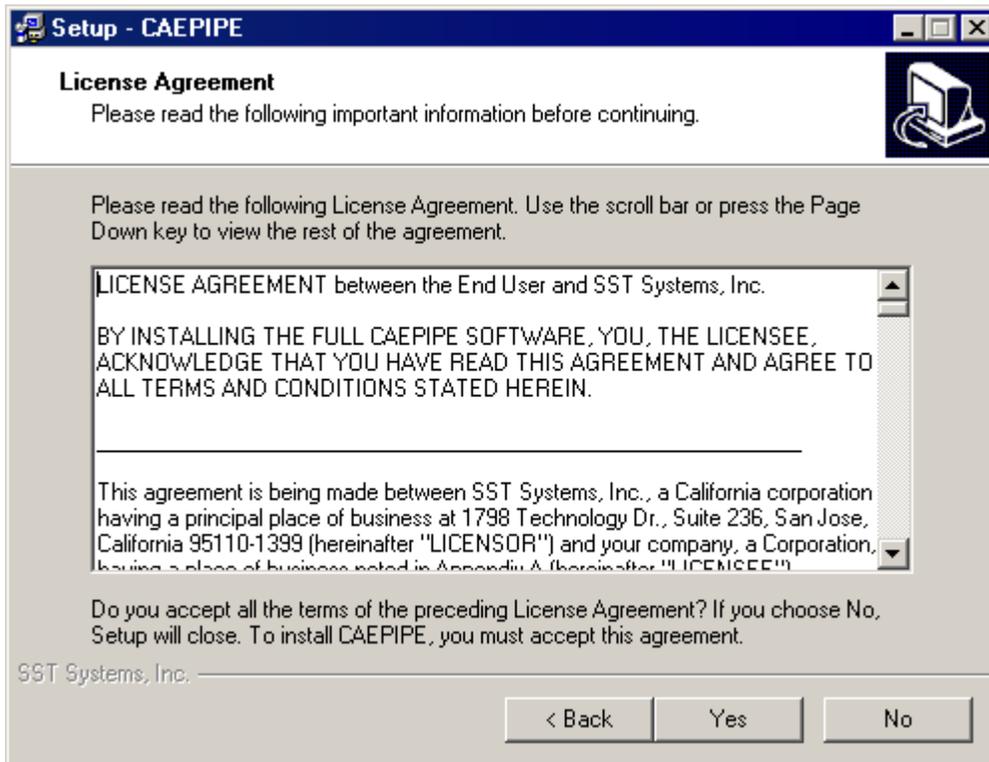


Click Next to continue.



Installation

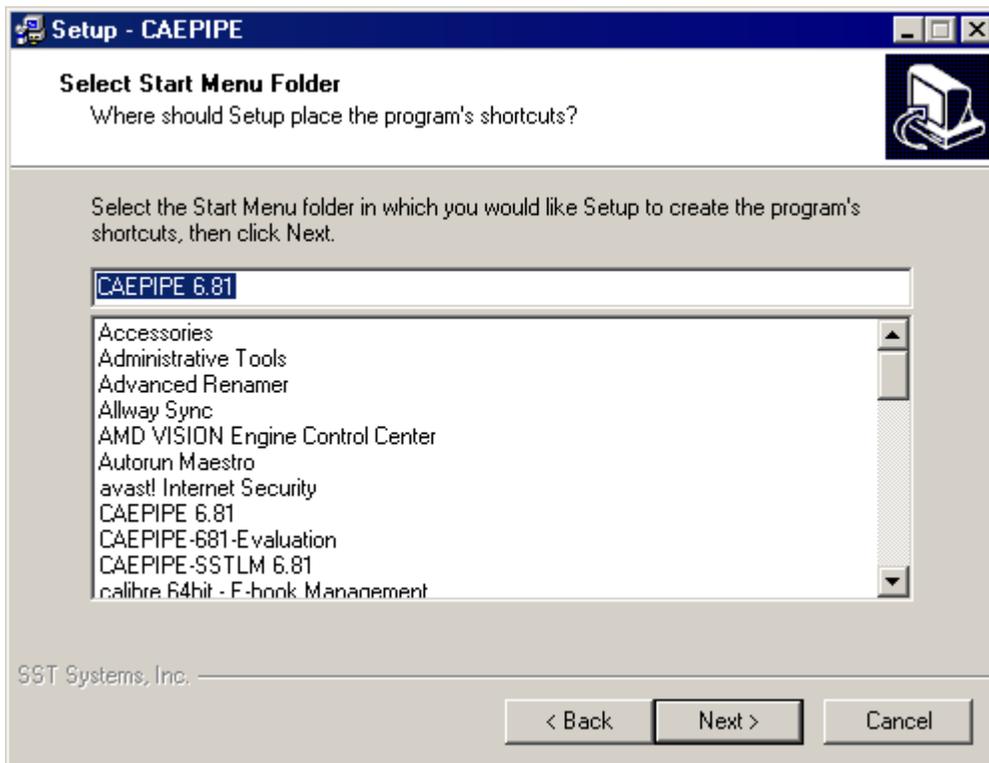
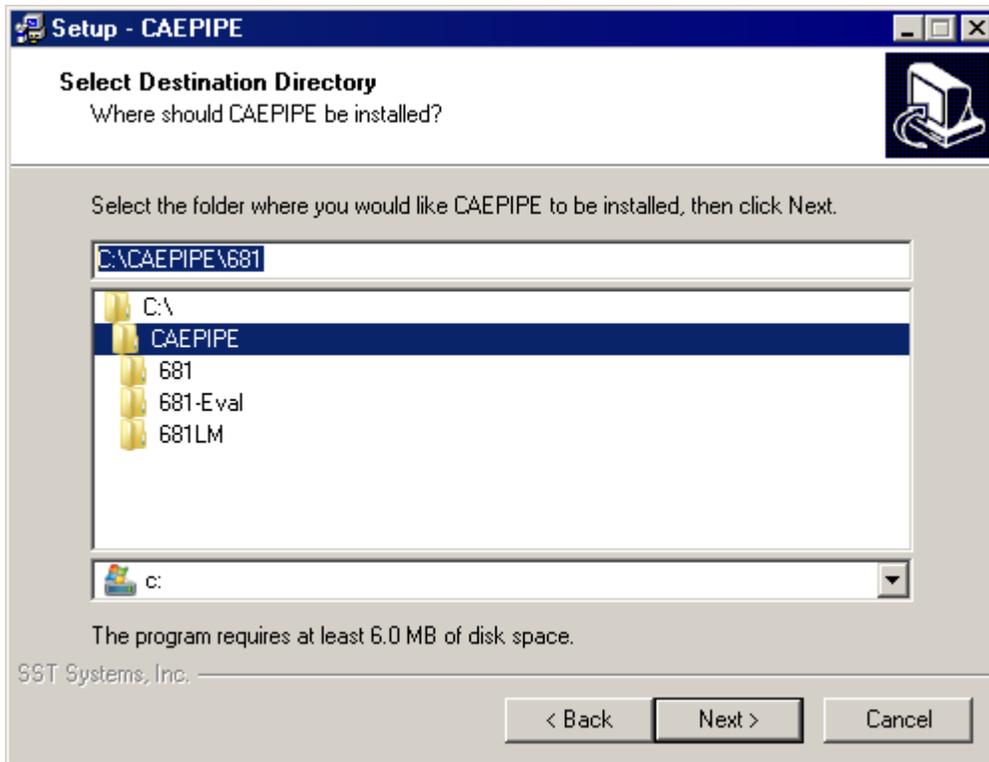
You must accept the terms of the License agreement by clicking on Yes before proceeding.



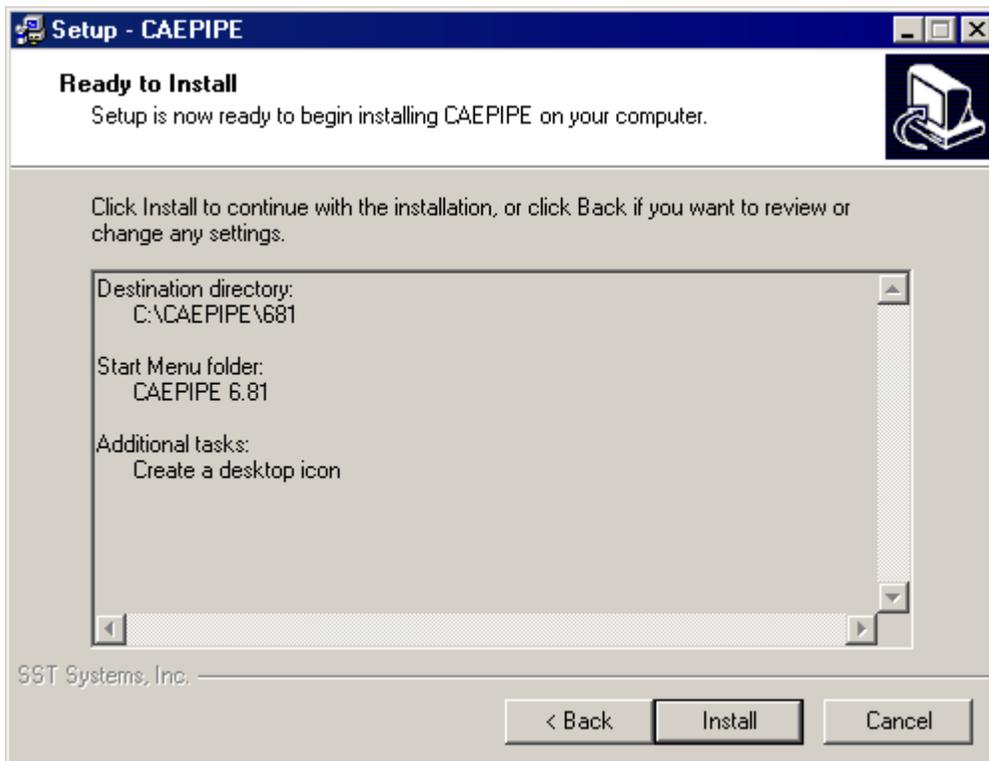
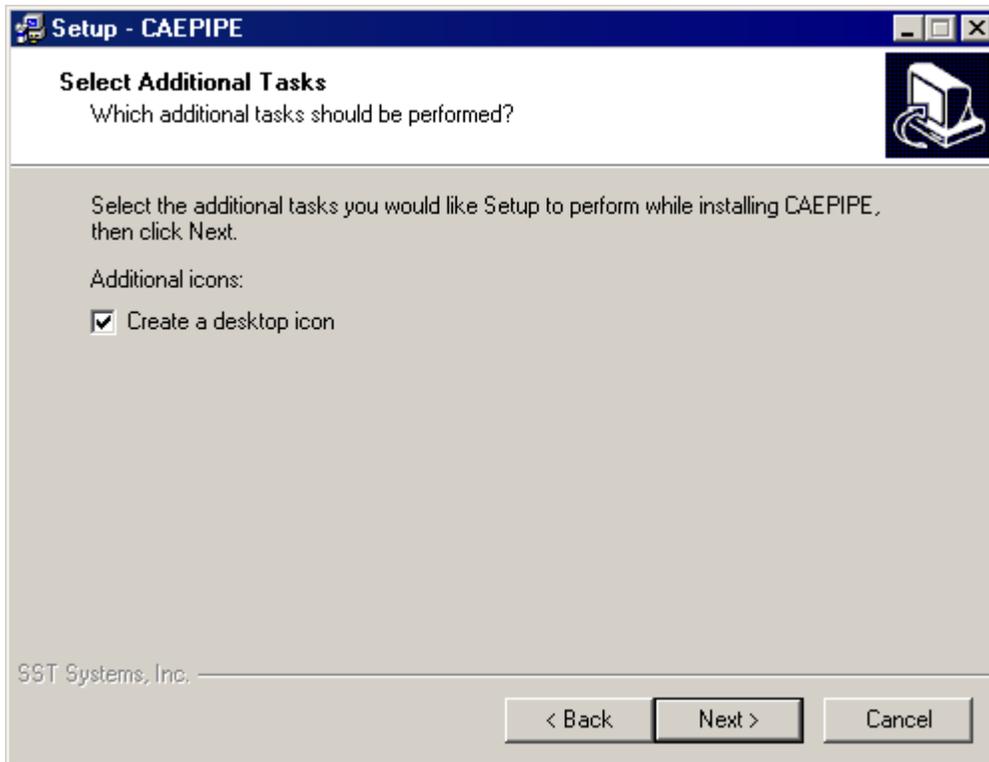
A copy of the license agreement (PDF file) is available on the distribution CD.

We recommend installing the software to the default folder shown during installation. If you have previous installations of CAEPIPE, it will be helpful to name the Start menu folder differently for the new version (e.g., CAEPIPE-681 for version 6.81).

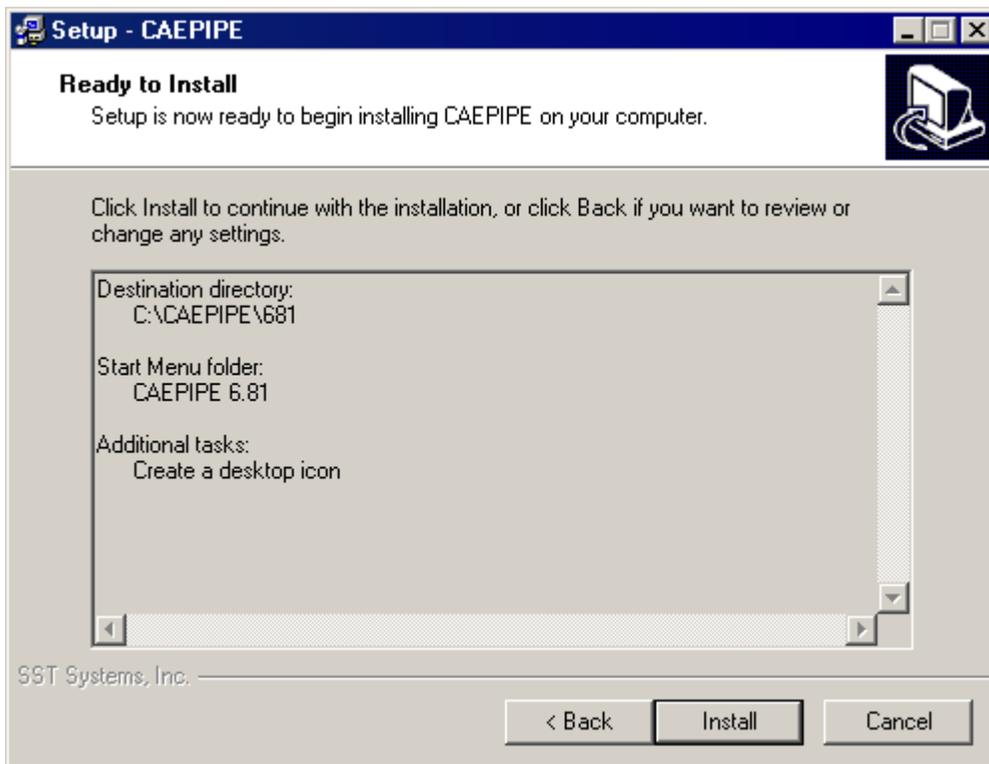
Installation



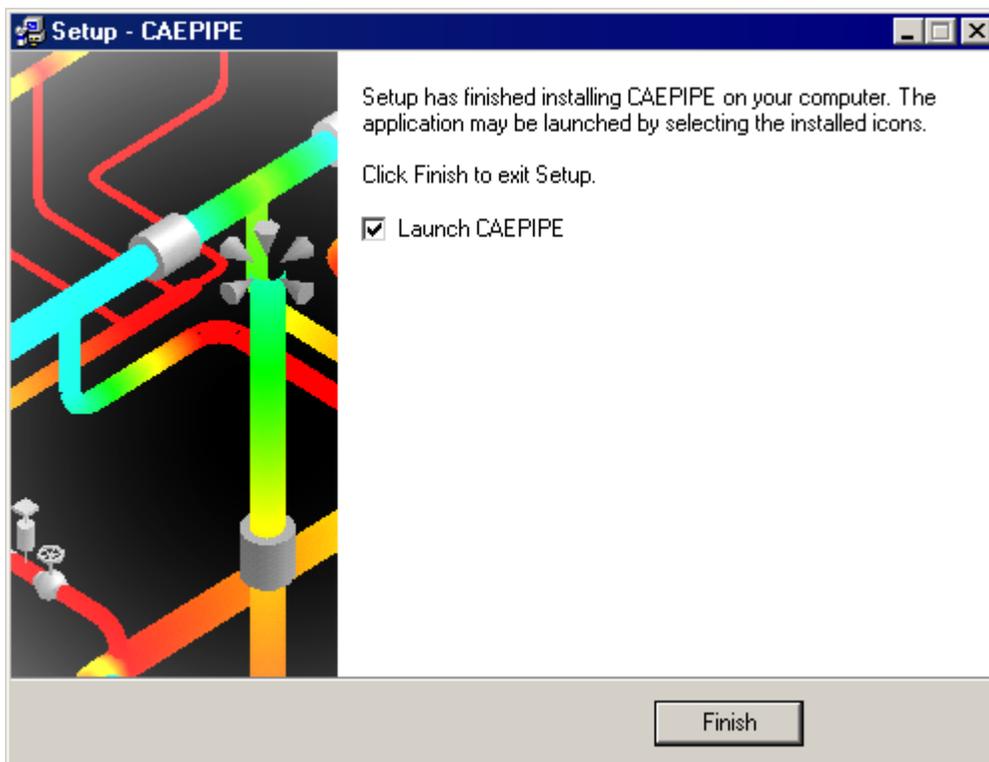
Installation



Installation



Once you have picked your choices, you can now begin to install by clicking on “Install” or use the “<Back” button to return to previous screens to change your choices.



Installation

After installation, you can execute CAEPIPE by double clicking on the Desktop icon or by selecting CAEPIPE from the newly created CAEPIPE folder on the Start menu. To uninstall CAEPIPE from your computer, select Uninstall CAEPIPE from the CAEPIPE folder on the Start menu.

NETWORK VERSION SETUP

For a network license too, you need to perform the same two steps as shown earlier in this topic. Step 1 is performed, however, on a server or a PC that is always available as it will act as the CAEPIPE license server. Once step 1 finishes, connect the USB hardware key, and ensure that the LED on the key lights up. Then, using the supplied CAEPIPE CD, install CAEPIPE (Step 2) on any PC on your network.

LICENSE ACCESS ACROSS NETWORK

To access a CAEPIPE license, for hardware keys marked "SuperProNet", you must define an environment variable called NSP_HOST on the client PC and point it to the server or PC that has the hardware key connected to it (from Step 1).

Create a new User/System variable in the Environment Variables dialog with the following configuration:

Variable: NSP_HOST

Value: Computer name or Computer IP Address (of server/PC where key is connected)

For detailed instructions on CAEPIPE network license installations, go to the following support pages on our website:

<http://www.sstusa.com/caepipe-install.php>

<http://www.sstusa.com/caepipe-nsphost.php>

<http://www.sstusa.com/hardware-key-faq.php>

WAN setup note: For WAN access, you must open any UDP packet communication among your routers on UDP port 6001.

If you don't open the UDP port, or don't set the environment variable (NSP_HOST), there may be a packet loss leading to access error. If no NSP_HOST is configured, then CAEPIPE's broadcast to the network via the UDP packet looking for the server might get lost, timing out the client with the "Status = 3, Key not found." Setting the NSP_HOST to the IP address of the machine with the key avoids the packet broadcast, and finds the specific machine with the specified IP address quickly.

This page is blank

Menus

This section explains the commands under the different menus in the four independent CAEPIPE windows: Layout, Graphics, List and Results. Each window contains its own menu and toolbar.

Many commands have keyboard shortcuts, shown next to the Command in the drop-down menu and also in online Help.

The menu items (commands) are shown as

Command

For the dialogs, the tabs are shown as

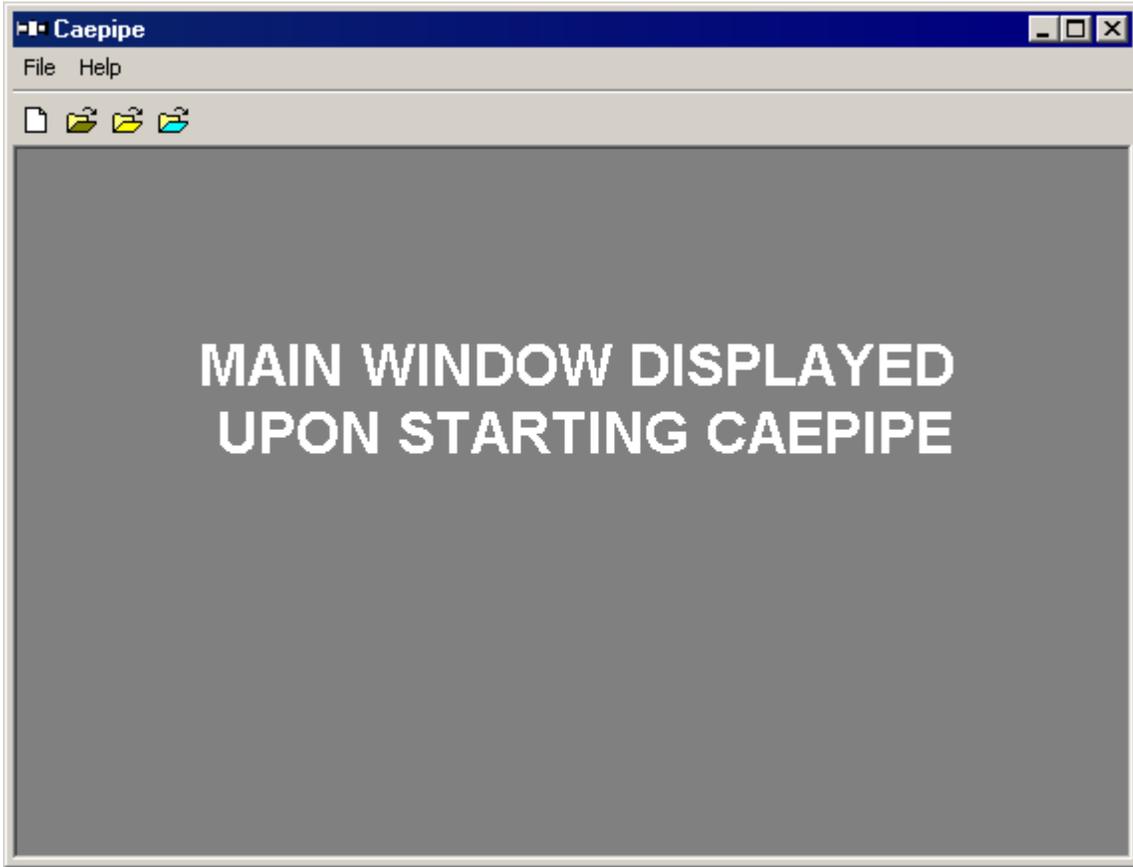
Tab

and the buttons are shown as

Button

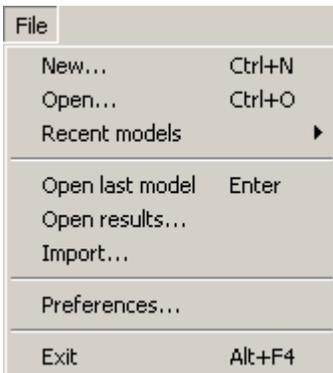
Main Window File Menu

The Main window is the first window you encounter when you start CAEPIPE.



File Menu

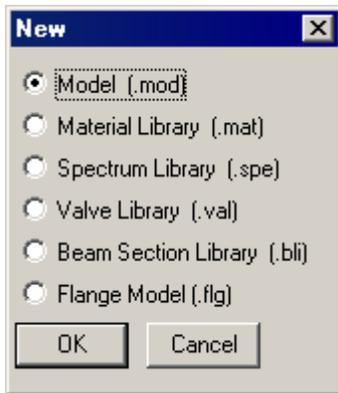
This menu contains commands to open files and set user preferences.



New (Ctrl+N)

This command opens a New File type dialog window. You can create any of the following file types associated with CAEPIPE from this window.

Main Window File Menu



Model (.mod) - Creates a blank layout in CAEPIPE for a new piping model file.

Material Library (.lib) - Creates a user-defined material library.

Spectrum Library (.spe) - Creates a user-defined spectrum library.

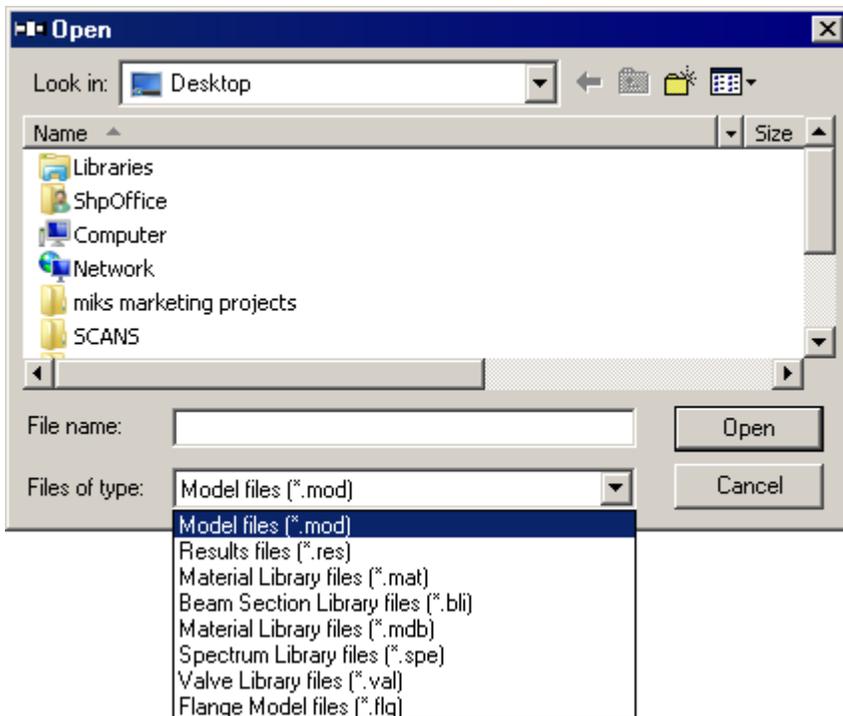
Valve Library (.val) - Creates a user-defined valve library.

Beam Section (.bli) - Creates a user-defined beam section library

Flange Model (.flg) - Creates a user-defined flange model.

Open (Ctrl+O)

From the Open command you can open any number of CAEPIPE compatible files.



Main Window File Menu

Model files (*.mod) - Opens CAEPIPE model files

Results files (*.res) - Opens CAEPIPE results files

Material Library files (*.bli) - Opens CAEPIPE material library files

Beam Section Library files (*.mod) - Opens CAEPIPE beam section library files

Material Library files (*.mdb) - Opens CAEPIPE material library files

Spectrum Library files (*.spe) - Opens CAEPIPE spectrum library files

Valve Library files (*.val) - Opens CAEPIPE valve library files

Flange Model files (*.flg) - Opens Flange Module model files

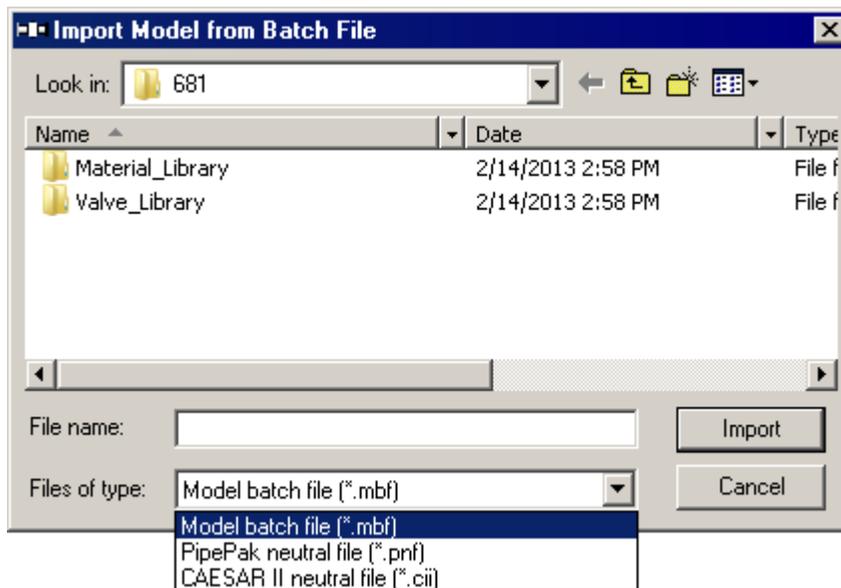
Open last model (Enter)

This command can be useful when you need to open the same model repeatedly. Simply pressing Enter from the Main window will open the model for you. There is no need to go through the Open File dialog.

Import

You can import a CAEPIPE neutral file (.mbf, similar neutral file), an Algor (Autodesk) PipePak neutral file (.pnf) or a COADE (Intergraph) neutral file (.cii) by using this command.

See Appendix C for details.

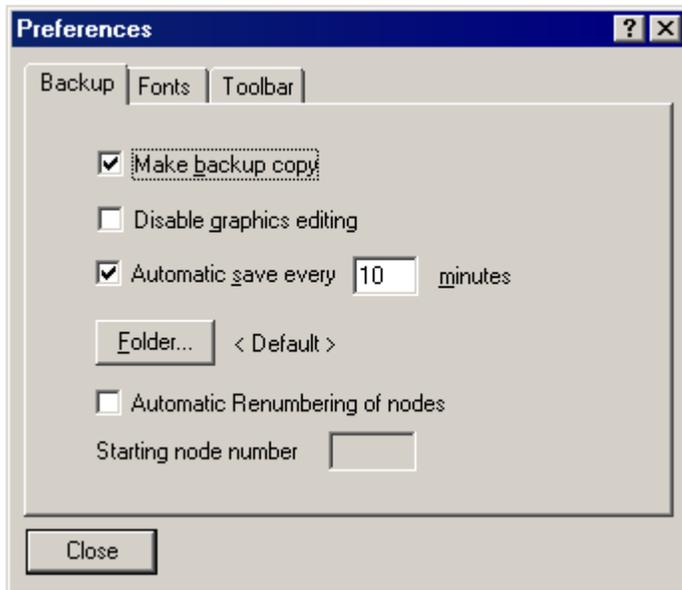


Preferences

Use this command to set your preferences for automatic saving and backup of the model, setting text and graphics fonts and toolbar preferences.

Preferences command has three tabs - Backup, Fonts and Toolbar.

Backup



Make backup copy

With this checked, every time you save your model, CAEPIPE copies the previously saved model data to another file (for backup, with a .bak extension). You can select the location where you want these files saved (click on Folder button). To open a backup file, rename the file to somename.mod before opening it in CAEPIPE.

Disable Graphics Editing

Graphics editing feature (in the Graphics window) can be turned ON/OFF from here. You may want to do this if your windows get garbled while using this feature.

Automatic Save

With this checked, and a time value entered, CAEPIPE will periodically save model data to a file named yourmodelname.sav, in the directory pointed to by Folder, or to untitled.sav.

If the software or hardware crashes, you can easily retrieve the most recently saved data. Locate the file (yourmodelname.sav) and rename the extension from .sav to .mod before opening it in CAEPIPE.

Folder

Select any folder to which you (as a user) have write permission to. Please make sure you check with your system administrator. CAEPIPE will try to issue you a warning if the folder you select does not have write permission. *<Default> points to the folder that contains the model file (.MOD).*

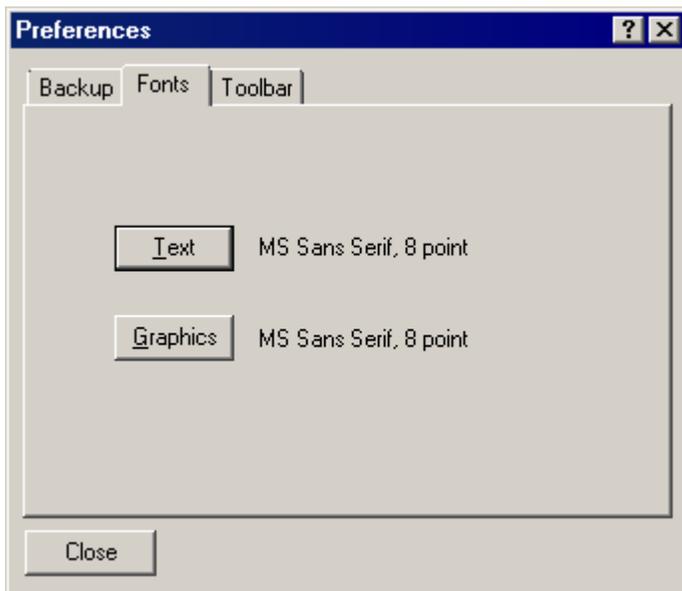
Main Window File Menu

Example: You can specify a new folder called CAEPIPE.BAK as a backup and save folder, so that CAEPIPE writes all backup files (.bak) and timed saves (.sav) to this folder.

Automatic Renumbering of nodes

When you delete (an) existing node(s) from a model in Layout, CAEPIPE automatically renumbers nodes sequentially from the “Starting Node Number” you specify. So, if you do not want CAEPIPE to automatically renumber nodes when you delete any node in the Layout window, uncheck this option here. Also note that the renumbering operation upon node deletion is instantaneous making it hard to identify which rows changed.

Fonts



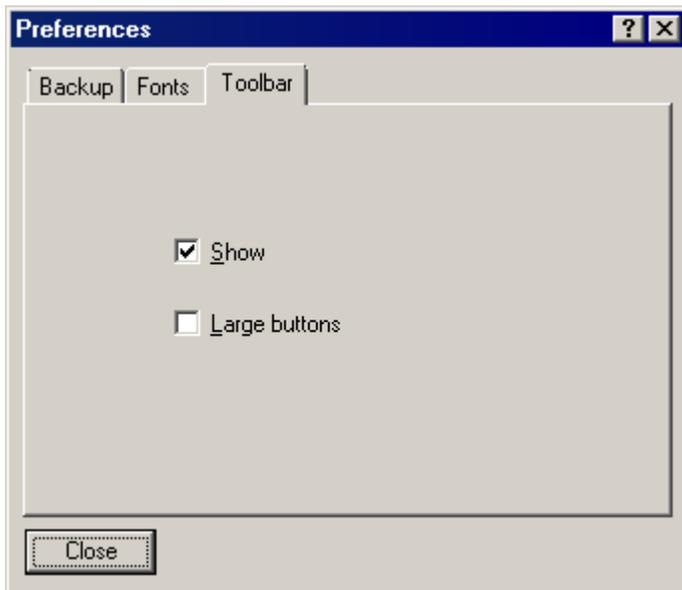
Text

Select a font face and size to use inside all CAEPIPE text windows (Layout, List and Results).

Graphics

Select a font face and size to use inside the Graphics window.

Toolbar



Show

You may choose to display or not display the toolbar in CAEPIPE windows.

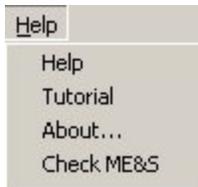
Large Buttons

Two sizes for toolbar buttons are available. Selecting this command will show the toolbar with large buttons.

Main Window Help Menu

Help Menu

From any CAEPIPE window - Main, Layout, List, Graphics or Results, you have access to the same Help menu.



Help

Clicking on Help command opens the CAEPIPE on-line help file.

Tutorial

We have three tutorials that are accessible from Start Menu > CAEPIPE:

1. Basic Pipe Stress Analysis Tutorial (SST-Basic_Pipe_Stress_Analysis_Tutorial.pdf)
2. Tutorial 1 (Tutorial_1.pdf)
3. Tutorial 2 (Tutorial_1.pdf)

The CAEPIPE Tutorials are available separately as an Adobe Acrobat (PDF) file. If you can't locate them inside your CAEPIPE program files folder, then visit our web site www.sstusa.com to get a copy.

About

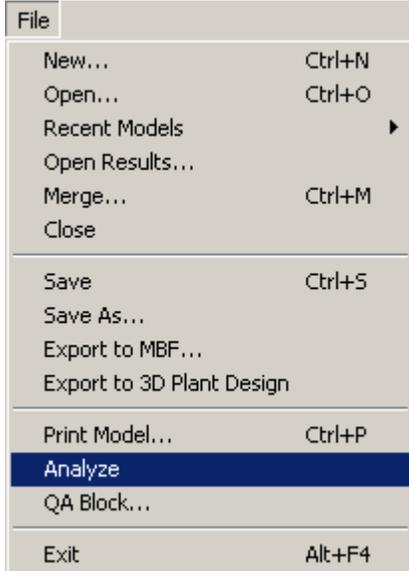


The About command shows the program version number and (your) hardware key serial number. Our support staff might ask for this information when you contact them.

Also from the About dialog, you can access the SST website by clicking on www.sstusa.com or send email to SST Technical Support by clicking on support@sstusa.com.

Layout Window File Menu

File Menu



This menu contains commands for standard file operations with a few special ones to note:

Open (Ctrl+O)

You can open the following CAEPIPE file types from the Open menu.

1. Model files (*.mod)
2. Results files (*.res)
3. Material Library files (*.mat)
4. Beam Section Library files (*.bli)
5. Material Library files (*.mdb) [old format]
6. Spectrum Library files (*.spe)
7. Valve Library files (*.val)
8. Flange Model files (*.flg)

For a description of each file type listed above, see page 14.

Open Results

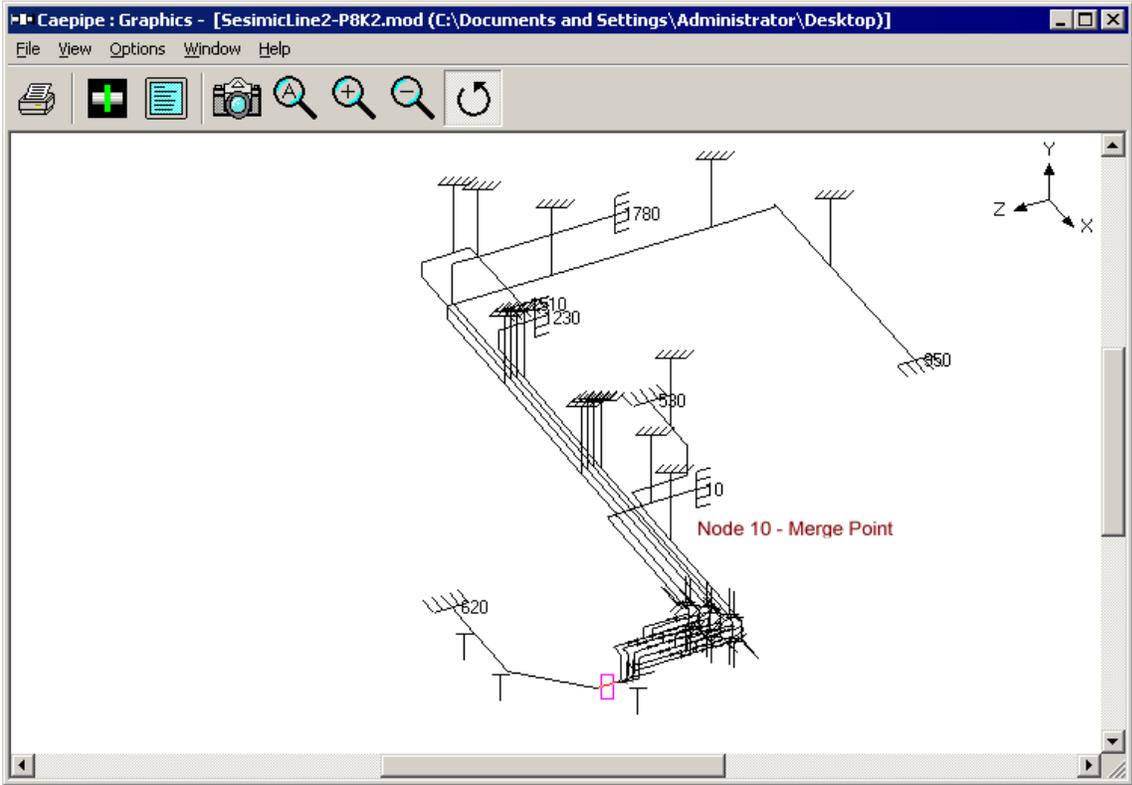
This command opens the Results files (*.res). The corresponding model file (.mod) must be present in the same directory as the results file (.res). For your convenience, the Open Results dialog shows the results filename for the currently opened model if it is already analyzed; if not, the name of the last opened results file is filled in the name field.

Merge

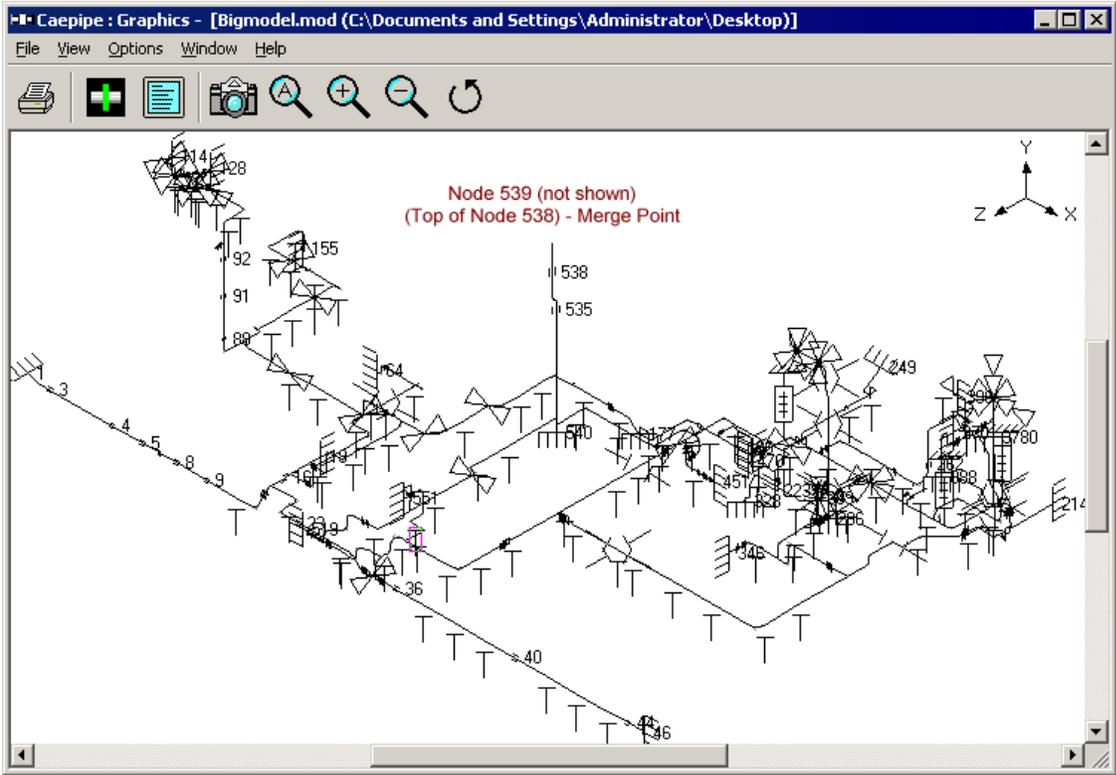
You can merge two or more piping models into one file using this command. This feature merges Materials, Sections, Loads, Beam Materials, Beam Sections, Beam Loads, Pumps, Compressors, Turbines and Elements from the selected model into the currently opened model. The process, however, skips Analysis Options, Load Cases, Wind Loads, Time History, Spectrums and Force Spectrums from the selected model during merging. Also, CAEPIPE checks for duplicate nodes and provides an option for renumbering the elements. Pressing “Yes” will renumber and merge the model; “No” will merge without renumbering, and you will have to manually identify and remove duplicates and/or renumber. See example under “From” in Reference for a suggestion on how to model separate files for merging later.

Example:

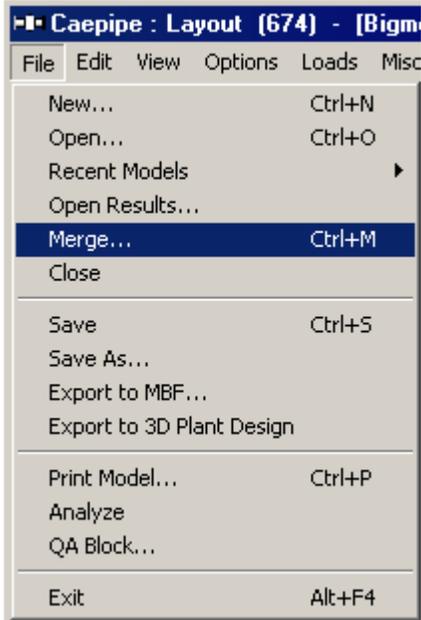
Here are two models with the note showing where the common merge point is (node 10 in the first model and node 539 in the second model).



Layout Window File Menu

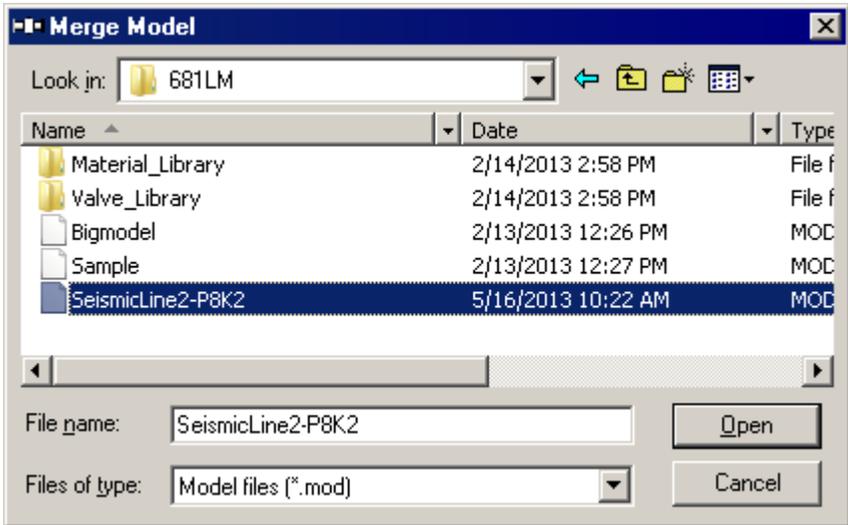


Open Bigmodel.mod first. Use menu File > Merge to start the process.

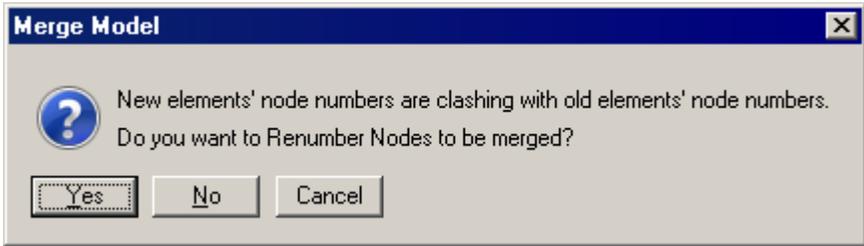


Select the second model as shown below and click on Open.

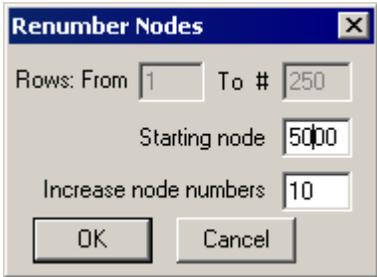
Layout Window File Menu



To avoid duplication of node numbers, names of materials, sections, loads, etc., click on Yes as shown below.



Choose a starting node number which is clearly not used in the first model. CAEPIPE will use the starting node number to number the nodes sequentially using the increment shown.



Models are merged now. The second model starts from row 675 (the row with the yellow highlight).

Layout Window File Menu

Caepipe : Layout (924) - [Bigmodel.mod (C:\Documents and Settings\Administrator\Desktop...)]

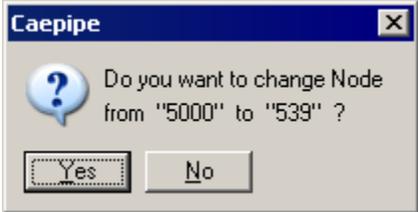
File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
671	556	Bend			-3'6"	1	3	3	
672	557	Bend		-5'3"		1	3	3	
673	558	Bend	-0.8000	-0.8000		1	3	3	
674	528					1	3	3	
675	Title = FWH21 Upstream								
676	5000	From							Anchor
677	5010			-0.0375	3'7-3/16"	106	24	7	Rod hanger
678	5020	Bend		-0.0375	3'7-5/16"	106	24	7	
679	5030		0'4"	-0.0035		106	24	7	
680	5040		11'8-3/4"	-0.1217		106	24	7	
681	5050	Valve	0'9"	-0.0078		106	24	7	
682	5060	Bend	0'4"	-0.0035		106	24	7	
683	5070			-0.2275		106	24	7	
684	5080			-0'2"		106	24	7	
685	5090	Reducer		-0'2"		106	26	7	
686	5100	Valve		-0'6"		106	26	7	
687	5110	Bend		-0.1442		106	26	7	
688	5120				1'0"	106	26	7	
689	5130	Valve			0'6"	106	26	7	
690	5140				0'2"	106	26	7	
691	5150	Valve			0'6"	106	26	7	
692	5160				0'2"	106	26	7	
693	5170	Valve			0'6"	106	26	7	
694	5180				0'3"	106	26	7	
695	5190	Reducer			0'2"	106	26	7	
696	5200				0'6"	106	24	7	
697	5210				0'6"	106	24	7	
698	5220	Valve			0'9"	106	24	7	
699	5230	Bend			1'2-1/8"	106	24	7	

Notice that node 10 in the first model and node 5000 in the second model start from the same global origin (0,0,0).

Layout Window File Menu

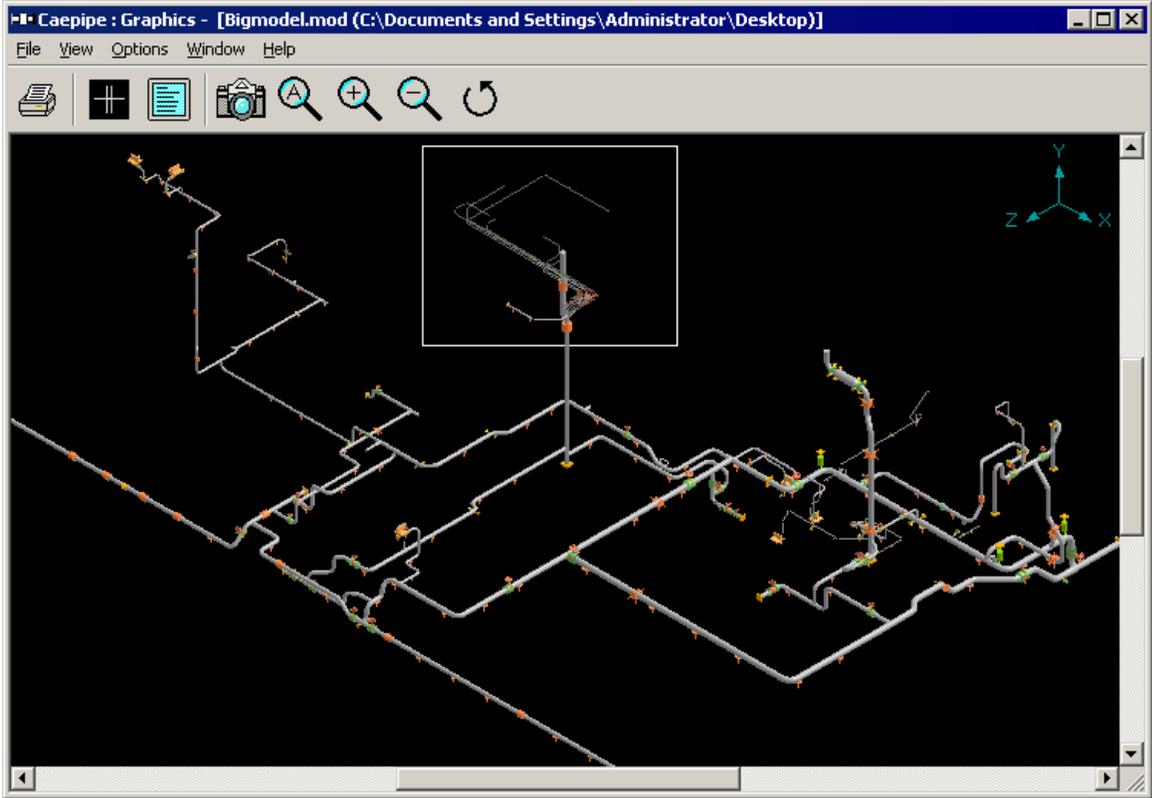
So, you need to change node 5000 to the connecting node number in the first model (i.e., node 539). Upon doing so, you are asked



Click on "Yes" to make the change. Also, remove the anchor from the data column on row 676.

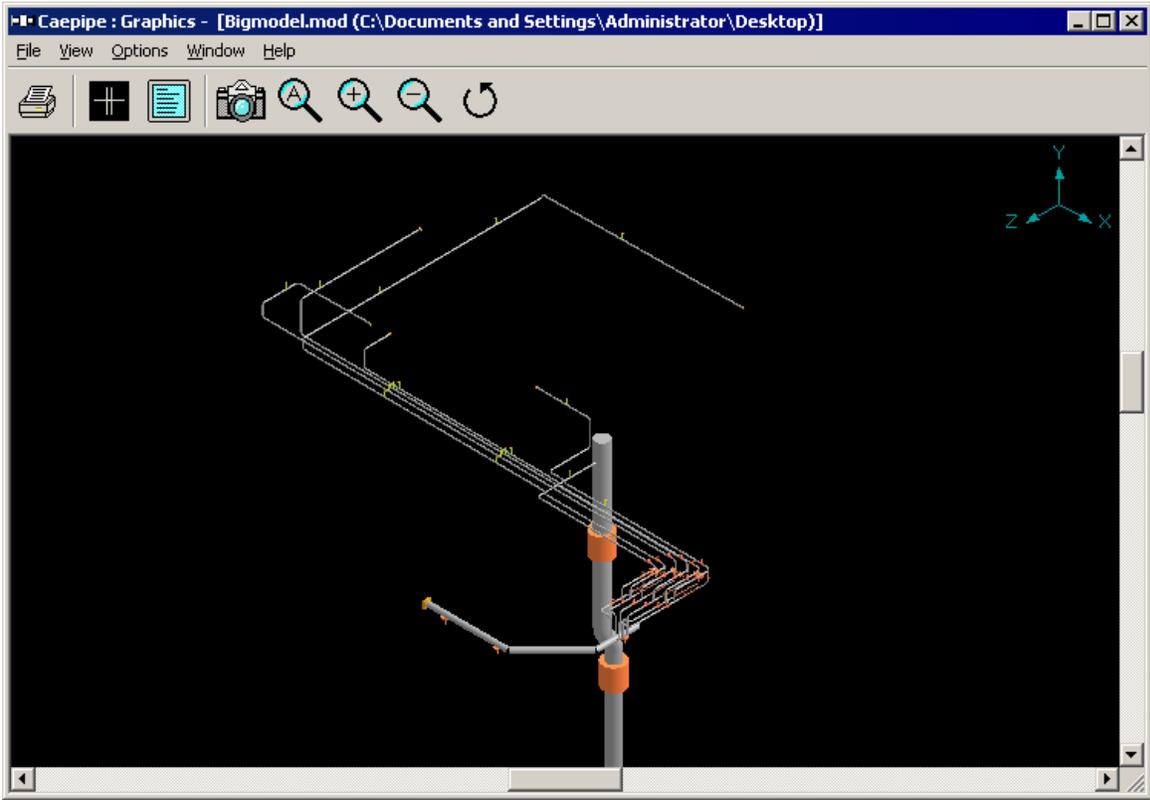
672	557	Bend		-5'3"		1	3	3	
673	558	Bend	-0.8000	-0.8000		1	3	3	
674	528					1	3	3	
675	Title = FWH21 Upstream								
676	539	From							
677	5010			-0.0375	3'7-3/16"	106	24	7	Rod hanger
678	5020	Bend		-0.0375	3'7-5/16"	106	24	7	
679	5030		0'4"	-0.0035		106	24	7	
680	5040		11'8-3/4"	-0.1217		106	24	7	

The combined model is shown in graphic view below (with a box – for illustration only - around the first model).



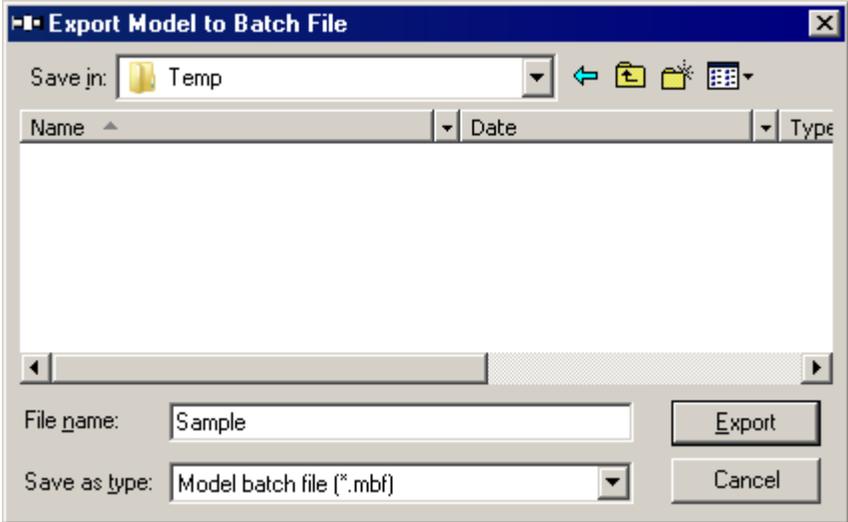
Layout Window File Menu

A zoomed image of the connecting node is shown below.



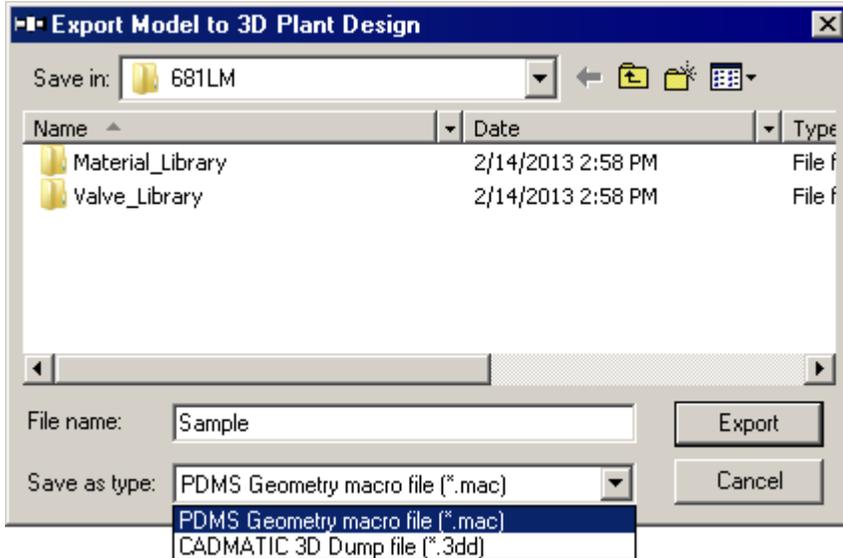
Export to .mbf

Presently, you can export the model data to CAEPIPE’s neutral file format (called a model batch file, .mbf). See Appendix C for format details.



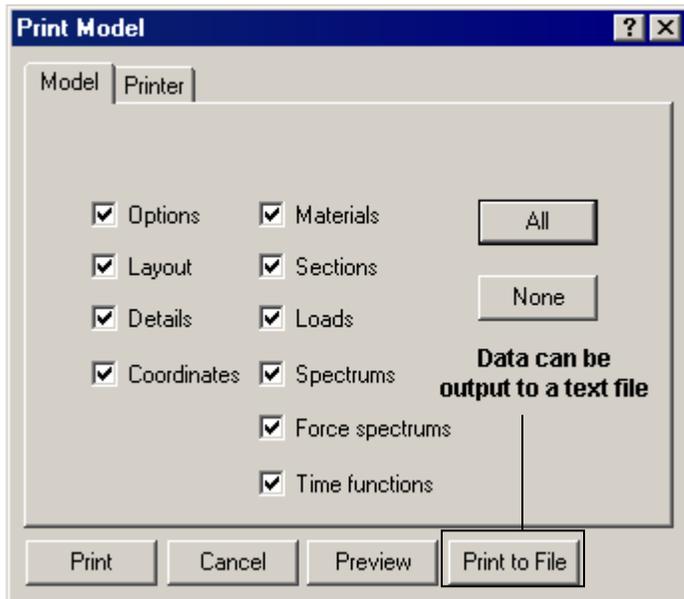
Export to 3D Plant Design

This feature exports the 3D CAEPIPE model to PDMS (.mac file) or CADMATIC (.3dd file) as structural components (only graphics without attributes).



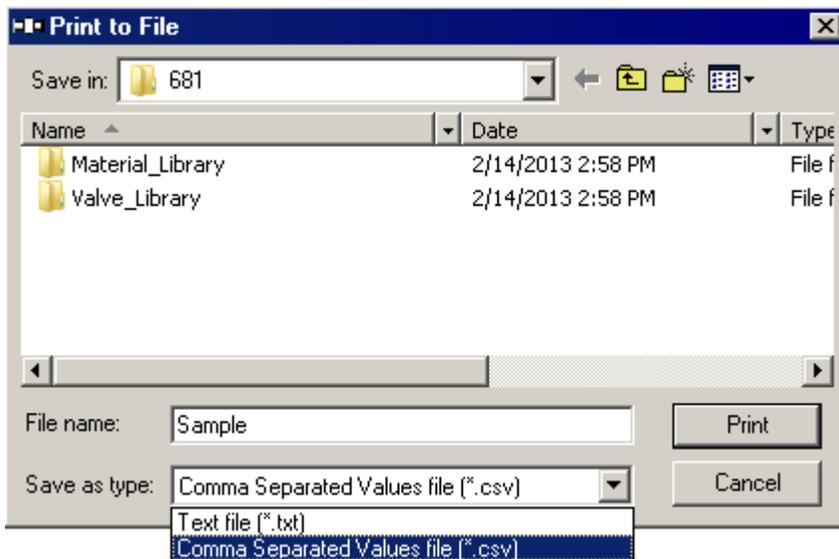
Print Model (Ctrl+P)

The Print command, flexible and customizable, is specific to each window. Here in Layout, you can print the model input data only (under the Model tab) to a printer, a PDF file (with a free PDF utility or Adobe® Acrobat installed), and change printer settings (under the Printer tab). You can customize the look of your report by changing the font, page sizes or margins.

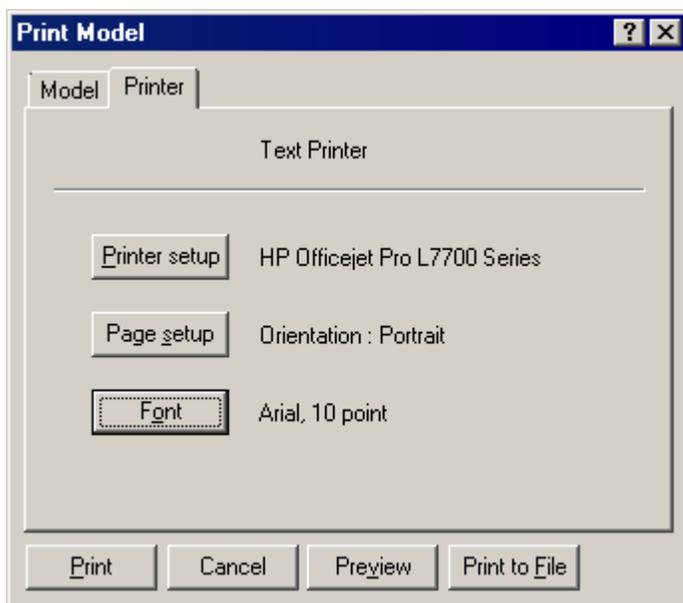


Data can be output to either a text file or a CSV (comma separated values) file that can be opened in a spreadsheet software (e.g., MS-Excel).

Layout Window File Menu



You could change the selected printer and customize other settings in the Printer tab (shown next).



Save (Ctrl+S)

This command saves model data to a .mod file format. It will also issue warning(s) if errors are found in the layout data.

Analyze

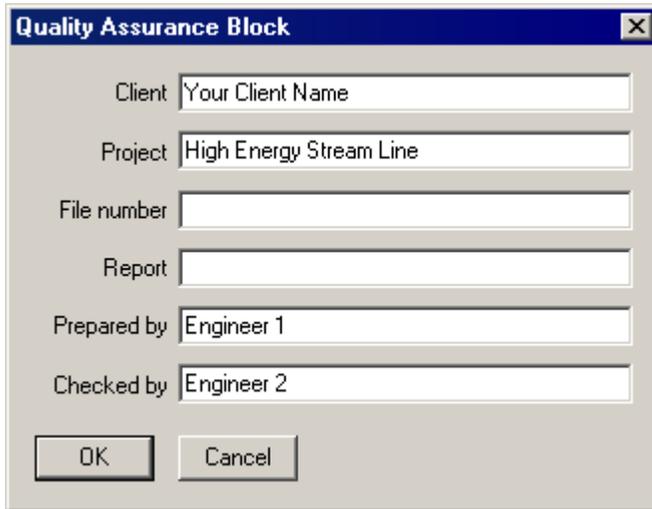
Once you are done modeling, use this command to analyze the model. At the end of analysis, you are asked whether you want to “View results?” Click on OK to see results or Cancel to return to the main window.

QA Block

Type in project-specific information here, which is printable only from the Results window. You cannot change this information after analysis in the Results window.

Layout Window File Menu

If you like to modify any information here, do it from the Layout window. You will then need to reanalyze the model because CAEPIPE will have deleted the existing results file for this model since the old data *may* no longer apply.



The image shows a dialog box titled "Quality Assurance Block" with a close button (X) in the top right corner. The dialog box contains several text input fields and two buttons at the bottom. The fields are labeled as follows:

- Client: Your Client Name
- Project: High Energy Stream Line
- File number: (empty)
- Report: (empty)
- Prepared by: Engineer 1
- Checked by: Engineer 2

At the bottom of the dialog box, there are two buttons: "OK" and "Cancel".

Layout Window Edit Menu

Edit Menu

Edit	
Edit type...	Ctrl+T
Edit data...	Ctrl+D
Copy...	Ctrl+C
Paste...	Ctrl+V
Find and Replace...	Ctrl+H
Insert	Ctrl+Ins
Delete...	Ctrl+X
Split...	
Multiple Split...	
Slope...	
Rotate...	
Change...	
Combine...	Ctrl+B
Renumber nodes...	
Generate...	
Regenerate	
Duplicate last row	Ctrl+Enter
Undo	Ctrl+Z
Redo	Ctrl+Y

Using this menu, you can edit properties for elements (under Type column) and data items (under Data column), insert and delete rows, split an element, rotate and renumber parts of the model, change a few properties like friction coefficient for a range of rows, and generate copies of existing rows.

Edit type (Ctrl+T)

Double click on the element name in the Type column or press Ctrl+T to edit properties (if applicable) of an element (bend, valve, etc.). The appropriate dialog is shown.

Edit data (Ctrl+D)

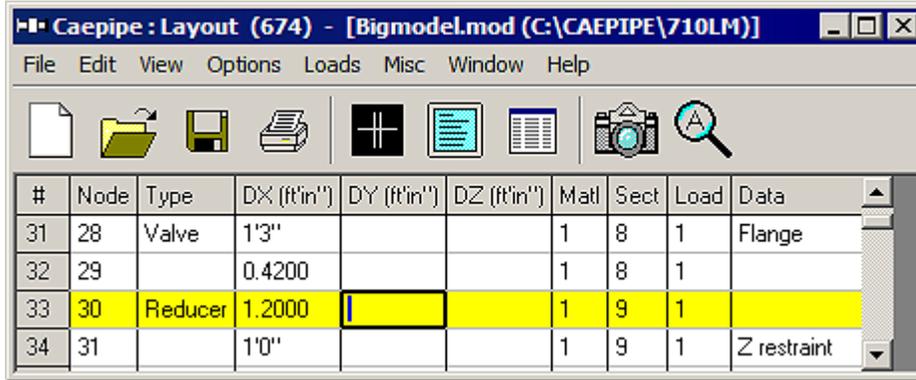
Double click on the Data item or press Ctrl+D to edit properties (if applicable) of the item (flange, anchor, etc.). The appropriate dialog is shown.

Copy (Ctrl+C)

You can copy a specific item on a row, an entire row or a block of rows, and Paste all, or individual parts of a row such as Material, Data or Type column entries. The copy command will copy all row data irrespective of the specific cursor location within that row. To copy, move the cursor to a row (with data) and press Ctrl+C (or select Edit menu > Copy command) to copy all data in that row. Multiple rows of Copy and Paste can be done only in Layout and NOT for Materials, Loads and Section properties.

Layout Window Edit Menu

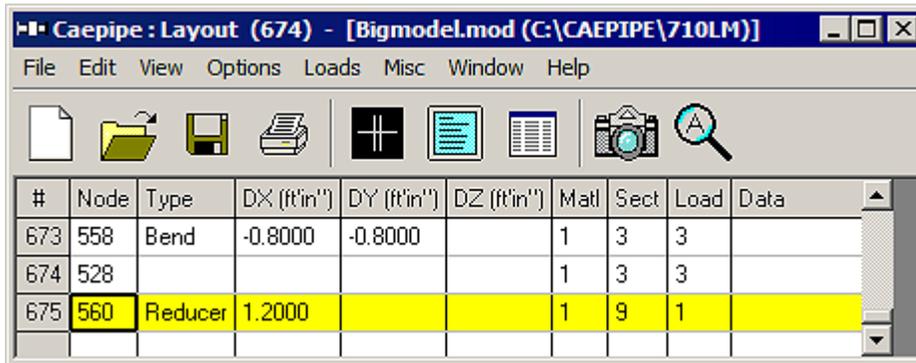
Example:



#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
31	28	Valve	1'3"			1	8	1	Flange
32	29		0.4200			1	8	1	
33	30	Reducer	1.2000			1	9	1	
34	31		1'0"			1	9	1	Z restraint

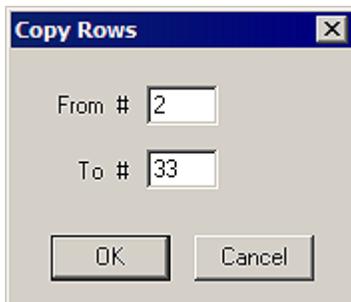
To copy the row at #33 (30, Reducer, 1.2ft, , ,1, 9, 1,), you may position the cursor anywhere before you invoke the Copy command. But, to paste ALL of this information, you must *position the cursor under the Node column of a (new) row*. Note that any existing information on a row will be overwritten.

If you want to paste, however, only the type 'Reducer' then place cursor under 'Type', for DX of 1.2', place cursor under DX, and so on.



#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
673	558	Bend	-0.8000	-0.8000		1	3	3	
674	528					1	3	3	
675	560	Reducer	1.2000			1	9	1	

To copy a block of rows (that represents a section of piping), invoke the command through Edit menu > Copy...



Copy Rows

From #

To #

OK Cancel

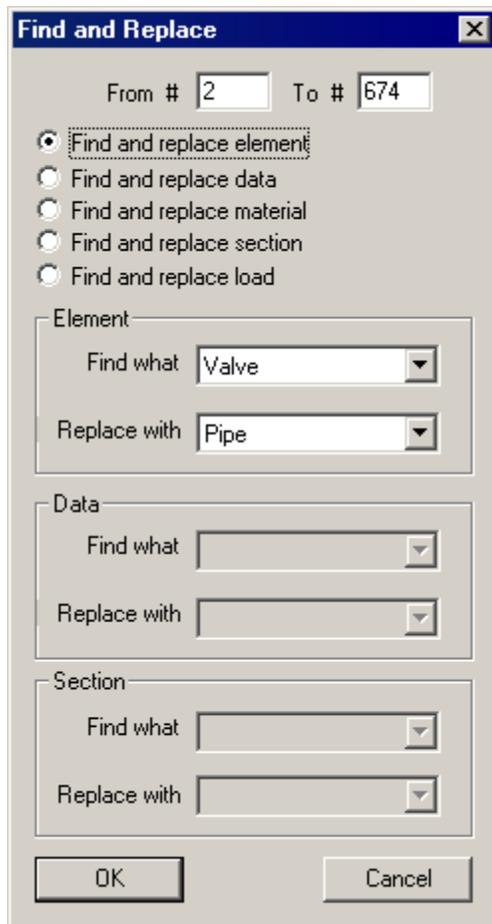
You need to specify (a range of) row number(s). Of course, you could accept the default row shown (which copies only one row).

Paste (Ctrl+V)

Once you have copied a row, move your cursor to the location where you want to paste the copied information and press Ctrl+V (or select Edit menu > Paste command). When pasting an entire row, move the cursor to the Node column of the row where you will be pasting. To paste an individual part of a copied row, move the cursor to a specific column within the row where you wish to paste the specific datum. The copied information is available for only one operation. Also, note that when a row is pasted, it will overwrite the existing row (if present).

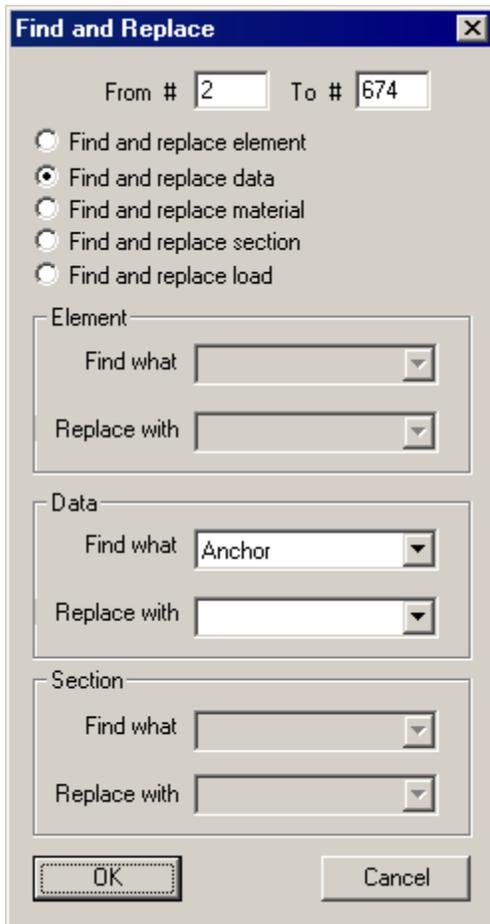
Find and Replace (Ctrl+H)

Find and replace Element / Data types, Material, Section, and Load by selecting the Find and Replace command from the Edit menu or press Ctrl+H. The Find and Replace dialog box is shown below.



In the shown dialog above, all Valves will be replaced with pipes between rows 2 and 674. Use this command, for example, to change all LR elbows to SR elbows. Note that you cannot leave the "Replace with" field blank.

Layout Window Edit Menu

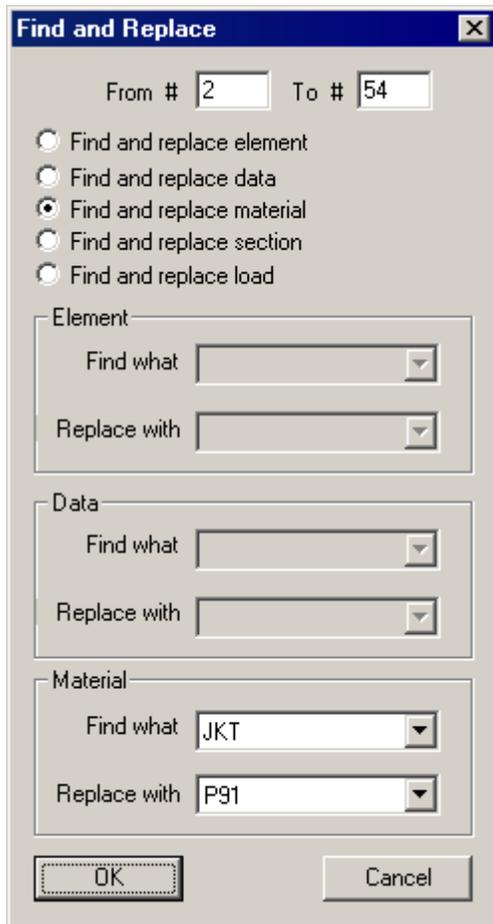


The image shows a "Find and Replace" dialog box with a blue title bar and a close button. It contains the following fields and controls:

- From # To #
- Radio buttons for search criteria:
 - Find and replace element
 - Find and replace data
 - Find and replace material
 - Find and replace section
 - Find and replace load
- Element section:
 - Find what
 - Replace with
- Data section:
 - Find what
 - Replace with
- Section section:
 - Find what
 - Replace with
- Buttons: and

You could change to different support types while evaluating different “what-if” conditions, or *remove* a specific support type from the model (select “blank” in the Replace with field).

Layout Window Edit Menu

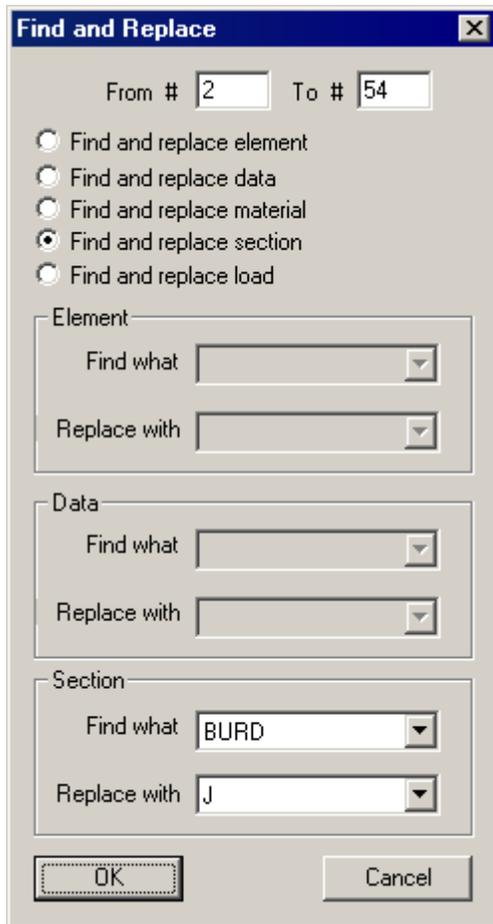


The image shows a "Find and Replace" dialog box with a blue title bar and a close button. It contains the following fields and options:

- From # To #
- Radio buttons for search criteria:
 - Find and replace element
 - Find and replace data
 - Find and replace material
 - Find and replace section
 - Find and replace load
- Element section:
 - Find what
 - Replace with
- Data section:
 - Find what
 - Replace with
- Material section:
 - Find what
 - Replace with
- Buttons: and

You could change to a different material type while evaluating different “what-if” conditions.

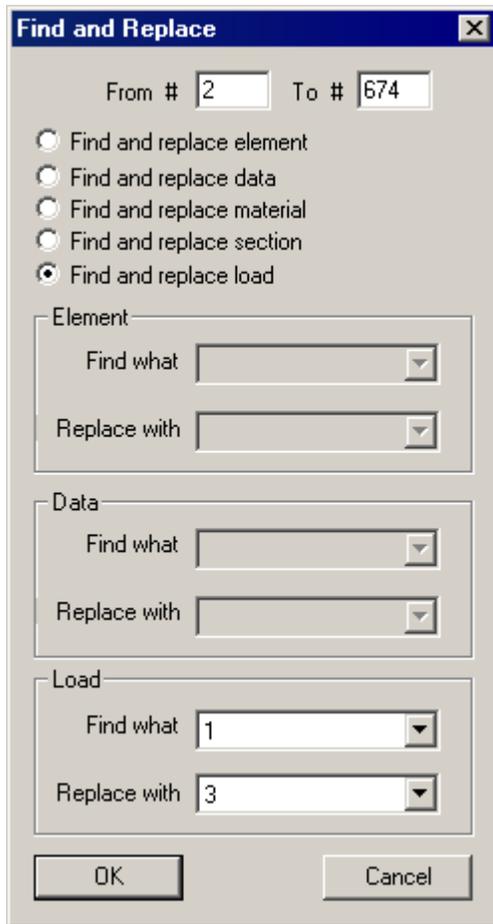
Layout Window Edit Menu



The image shows a "Find and Replace" dialog box with a blue title bar and a close button (X). At the top, there are two input fields: "From #" with the value "2" and "To #" with the value "54". Below these are five radio button options: "Find and replace element", "Find and replace data", "Find and replace material", "Find and replace section" (which is selected), and "Find and replace load". The dialog is divided into three sections: "Element", "Data", and "Section". Each section has a "Find what" and a "Replace with" dropdown menu. In the "Section" section, "Find what" is set to "BURD" and "Replace with" is set to "J". At the bottom, there are "OK" and "Cancel" buttons.

You could change to a different section while evaluating different “what-if” conditions.

Layout Window Edit Menu



You could change to a different load while evaluating different “what-if” conditions.

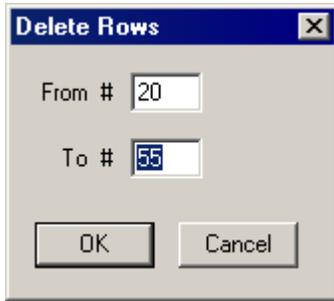
Insert (Ctrl+Ins)

Use Ctrl+Ins to insert an empty row above the highlighted row in the Layout window. This command is also available from the context menu, by right-clicking in the Offsets (DX, DY, DZ) columns, or via the Edit menu.

Delete (Ctrl+X)

Use Ctrl+X on a row to delete it. To delete a range of rows, you must select the Delete command from the Edit menu, and then type the row numbers to mark the range for deletion. If you want to delete till the end of the model, either type in a big number for “To #” such as 9999 or type in the (total) number (of rows) shown in the title bar in parentheses.

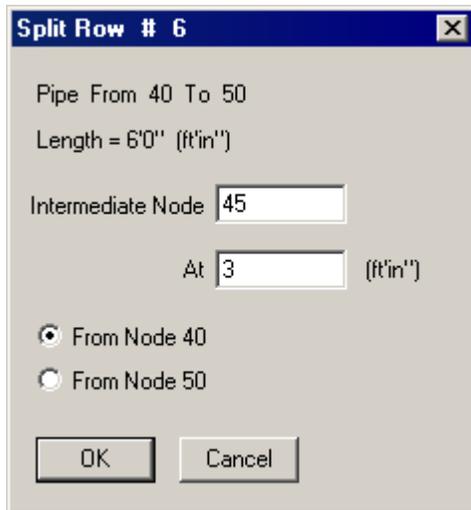
This command is also available from the context menu (by right clicking), in the Offsets (DX, DY, DZ) columns. You can undelete using the Undo command.



Split

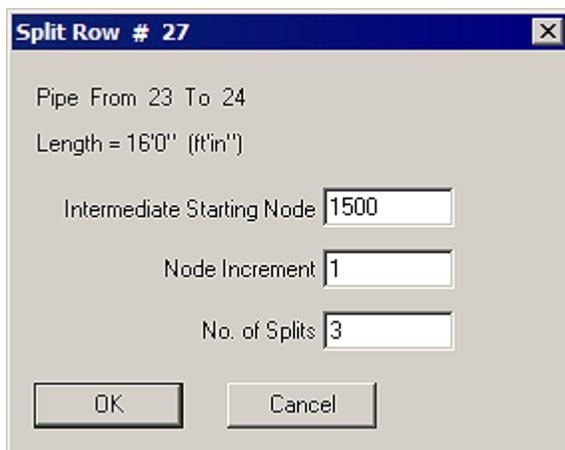
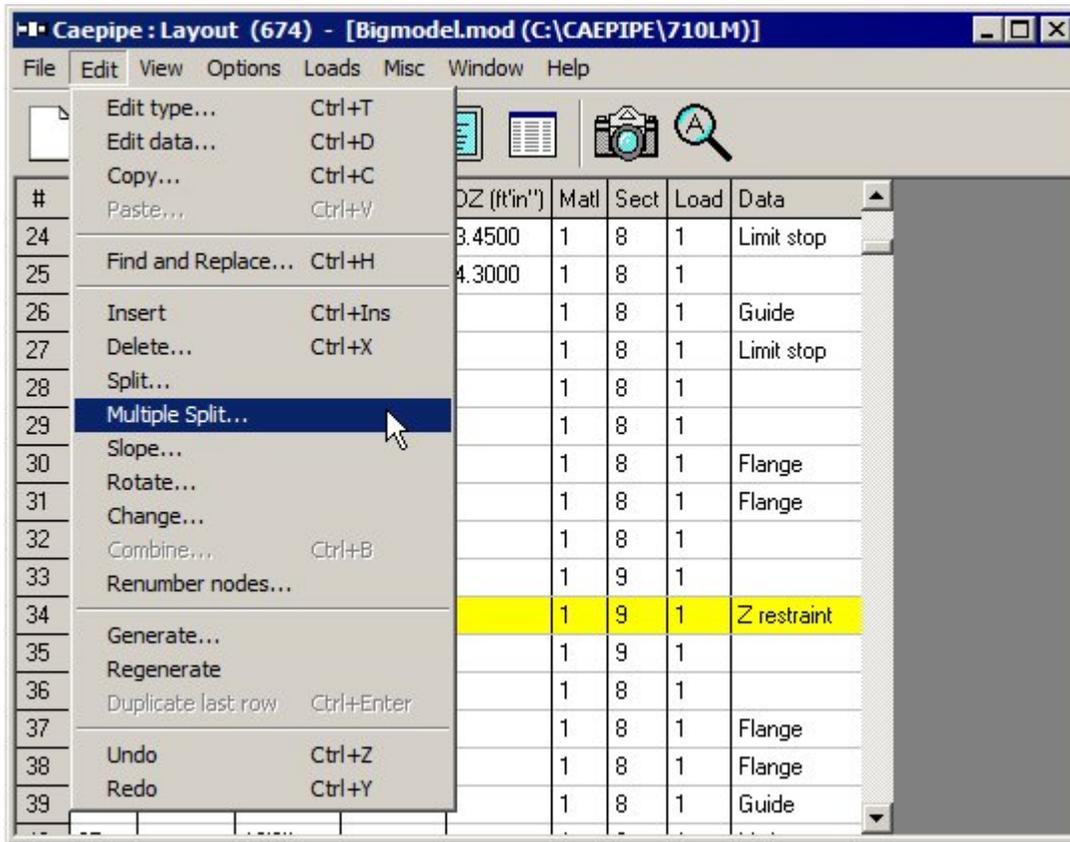
Use this command to split a straight element or a curved section of a bend into two parts. This can be useful to create nodes for supports, applying forces at intermediate locations or introducing intermediate mass points. To split curved sections of bends (including jacketed bends), specify intermediate nodes at desired angles in the bend dialog. This command is also available from the context menu by right clicking on the Offsets (DX, DY, DZ) columns.

When you have a long sloping pipe, you can create one long element with the slope built into it and then use the Split command to break it down into smaller elements. Or, of course, you could use the Slope command.



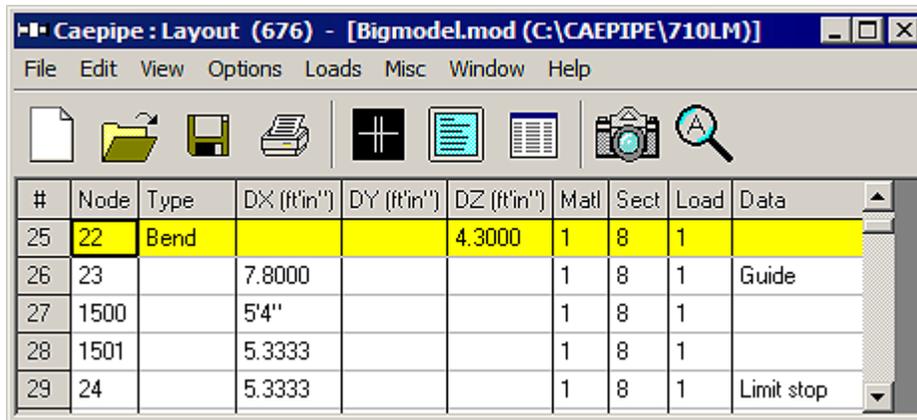
Multiple Split

CAEPIPE can split an element into multiple segments.



The above shown command parameters will split the 16' long element between nodes 23 and 24 into three *equal* segments and number them starting from 1500 (with an increment of 1). If you split a non-pipe element (such as a bend, valve, reducer, etc.), you will get multiple pipe elements and the non-pipe element as the result of this operation.

Layout Window Edit Menu



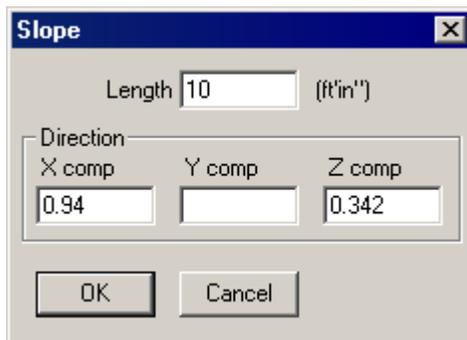
The screenshot shows the 'Caepipe : Layout (676) - [Bigmodel.mod (C:\CAEPIPE\710LM)]' window. The menu bar includes File, Edit, View, Options, Loads, Misc, Window, and Help. Below the menu bar is a toolbar with icons for file operations and viewing. A table displays the following data:

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
25	22	Bend			4.3000	1	8	1	
26	23		7.8000			1	8	1	Guide
27	1500		5'4"			1	8	1	
28	1501		5.3333			1	8	1	
29	24		5.3333			1	8	1	Limit stop

Slope

You can specify the slope of the line you want to model for an element, in terms of its direction cosines (DCs).

This command calculates the offsets automatically for a sloping element (that does not align with one of the major global axes). When a pipe slopes (or routes at an angle) with respect to the global axes, it becomes necessary to calculate the offsets (DX, DY and DZ) using those faithful sine and cosine functions. But, therein lies the problem. Sometimes, calculations get tricky and time-consuming.



The 'Slope' dialog box contains the following fields and buttons:

- Length: 10 (ft'in')
- Direction section with three input fields:
 - X comp: 0.94
 - Y comp: (empty)
 - Z comp: 0.342
- Buttons: OK, Cancel

Let us see a few examples.

Example 1: A 20° line in the XZ plane, 10 feet long

Method 1:

Calculate the offsets:

$DX = 10 \cos(20) = 9.4$ (ft.), $DZ = 10 \sin(20) = 3.42$ (ft.). Simple!

Method 2:

Find direction cosines (DCs) for the line (vector). Direction cosines are simply cosines of the angles the vector makes with the global X-, Y- and Z-axes. In this case, $A=20^\circ$ (X), $B=90^\circ$ (Y), $C=70^\circ$ (Z). Cosines of these angles are: X comp=0.94, Y comp=0, Z comp=0.342.

Now, position the cursor on the sloping element row, right click under DX/DY/DZ. Select Slope from the menu, enter these numbers into the X comp, Y comp and the Z comp fields

Layout Window Edit Menu

of the Slope dialog that opens, and type the length, 10 ft. Press Enter. CAEPIPE calculates the respective offsets using the DCs you input.

Once you input the slope, follow this pipe with other elements down the line by inputting different lengths while you retain the same direction cosines.

Example 2: A 40° line in the X(-Y) plane

Method 2:

Find direction cosines, if the line is at 40 deg. to X-axis in X(-Y) plane, it makes 50° with the (-Y) axis. So, the DCs are: $\cos(40^\circ) = 0.766$, $\cos(50^\circ) = 0.643$, So, X comp = 0.766, Y comp = -0.643, Z comp = 0.0. Type these DCs and a length into the Slope dialog. Press Enter.

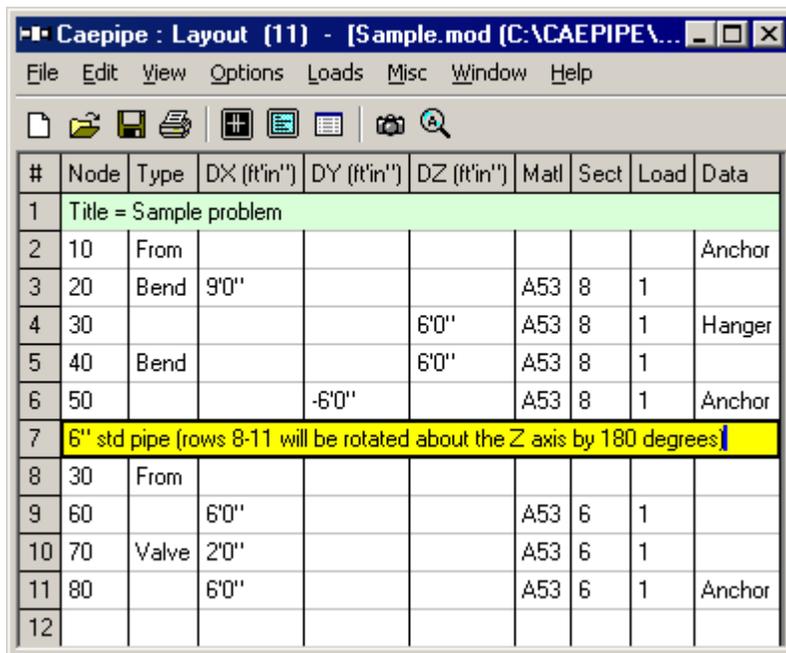
More information about Direction cosines is available under the section on Direction (Reference)

Rotate

You can rotate a model or a part of it about the global axes. CAEPIPE will adjust the offsets for new orientation.

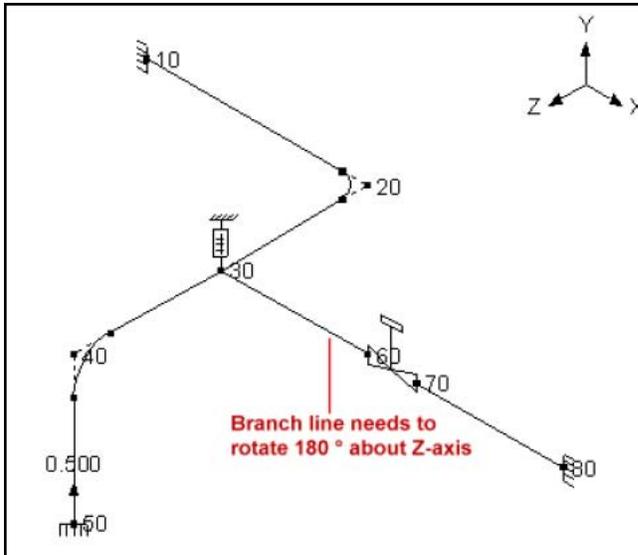
Example

Let us rotate the branch line of the familiar Sample model. The branch line is between nodes 30 to 80 (rows 8 to 11 in the Layout window). Say, we wanted to route this branch line in the -X direction (i.e., it needs to rotate 180°).

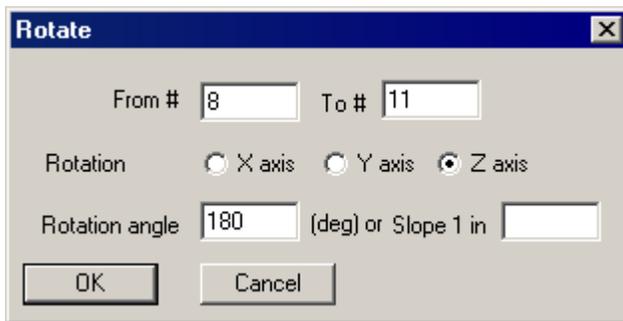


#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe (rows 8-11 will be rotated about the Z axis by 180 degrees)								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									

Layout Window Edit Menu



Select Rotate from the Edit menu and type in the shown values.

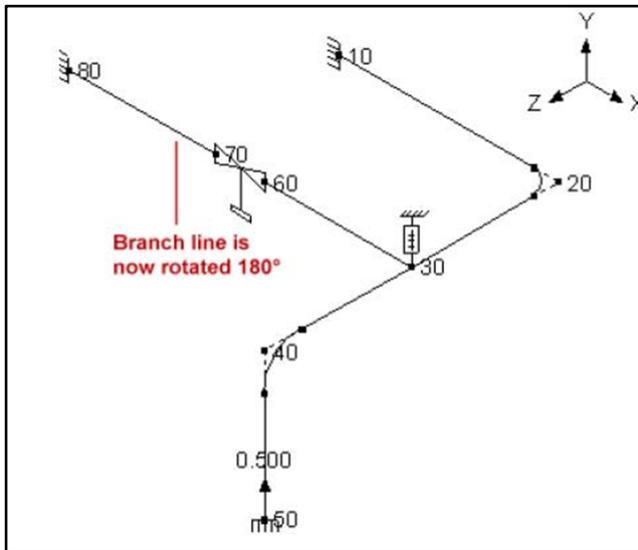


Notice the rotated offsets (now they are in the -X direction).

Note: The valve additional weight is shown in the negative direction because of the rotation. To make it positive, edit the Valve (Ctrl+T) properties and change its DY offset from -18 to 18 inches.

Layout Window Edit Menu

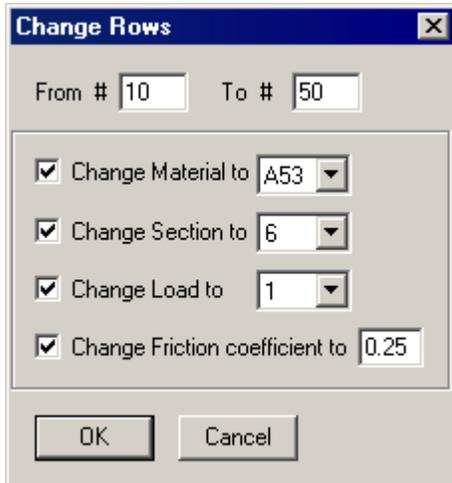
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe (rows 8-11 will be rotated about the Z axis by 180 degrees)								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									



Layout Window Edit Menu

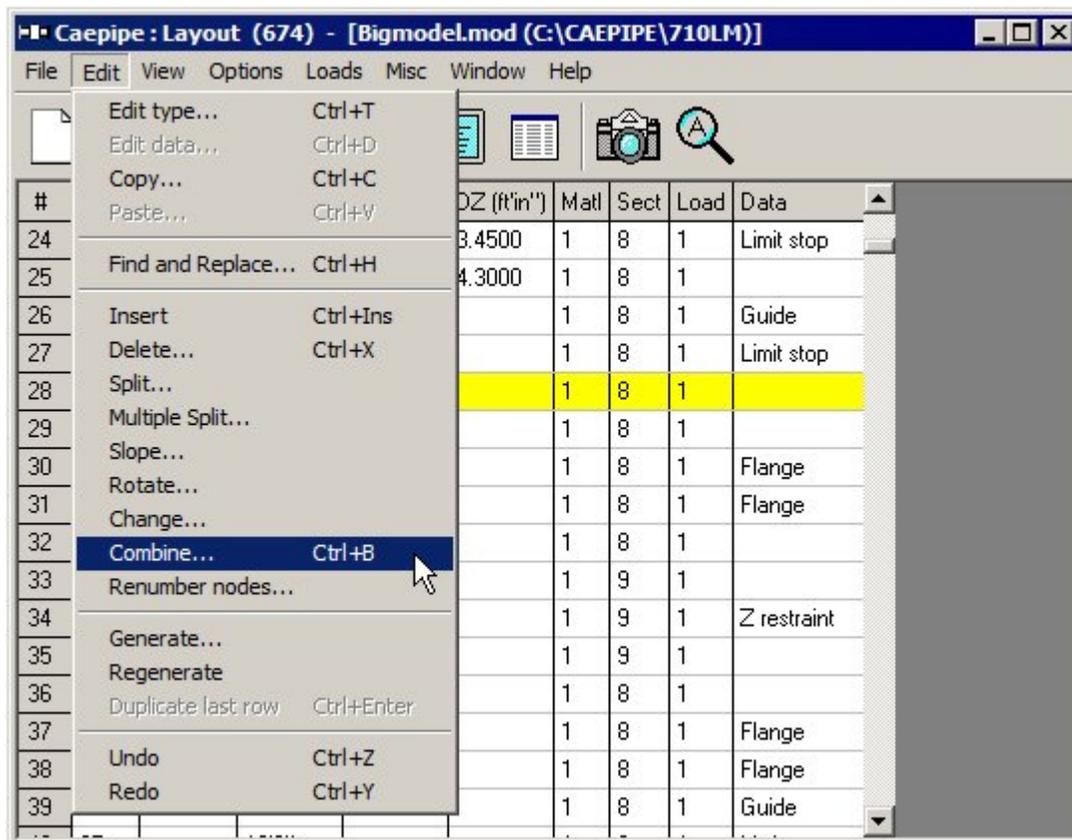
Change

Use this command for block operations such as changing friction coefficient for all guides and limit stops within a range of rows, or changing the material, section or load for a range of rows. You are asked for the range when you select this command from the Edit menu.



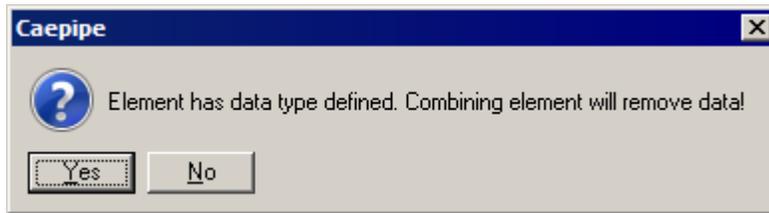
Combine (Ctrl+B)

You can combine current PIPE/BEND elements with previous PIPE elements.



Layout Window Edit Menu

If you have a Data type (such as a support) defined on one of the “to-be-combined” rows, CAEPIPE will alert you to the fact that you will lose that data type definition during the Combine process.



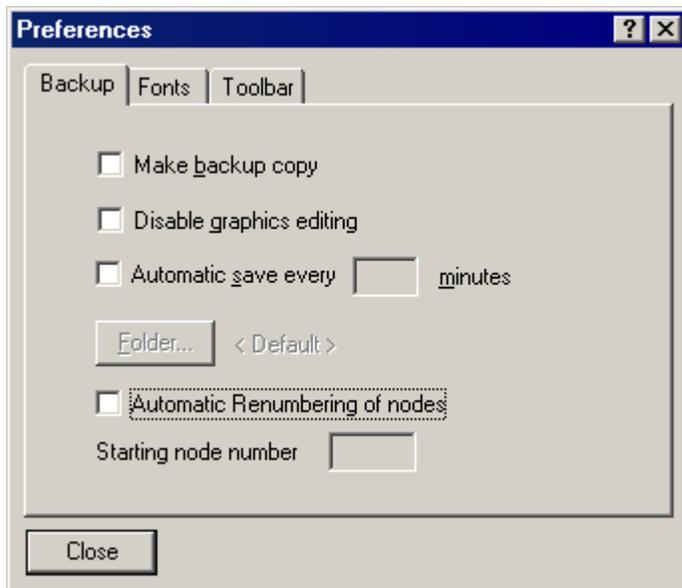
If you accidentally combine two elements, you can always UNDO the operation.

Renumber Nodes

You can renumber existing node numbers in your model using this handy feature; really useful when you have a big model and want to adopt a consistent node numbering system after multiple edits. You can renumber the whole or parts of the model keeping in mind that the greatest node number in a model cannot exceed 99,999.

Renumbering a segment of a piping system also renumbers all those nodes of the segment that appear as “Connected to” nodes in the remaining segments of the piping system.

Before using this feature, please make a copy of your model and work on the copy. Note: This feature is unrelated to “Automatic Renumbering of Nodes” found in the main window>Preferences.



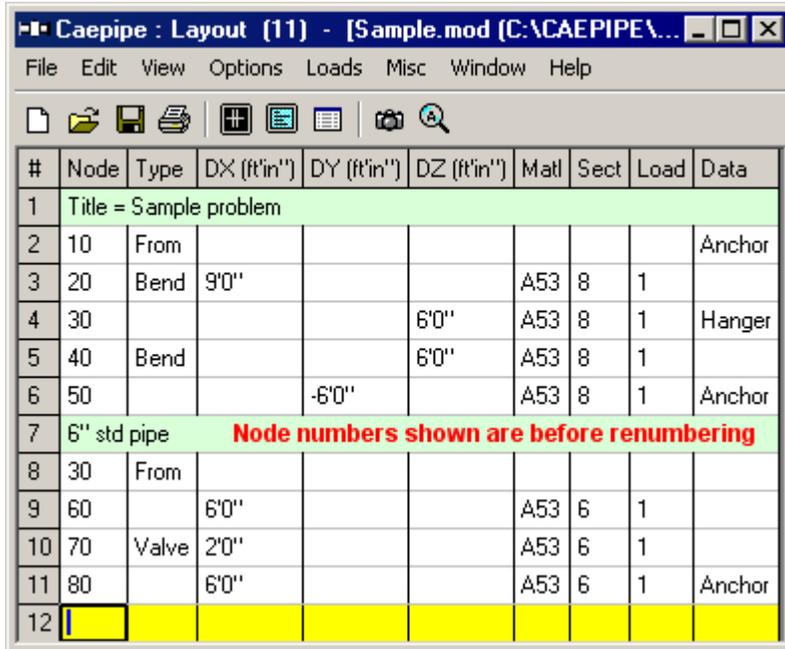
Automatic Renumbering in the Main window > Preferences

When this feature is turned on, deleting row(s) in Layout triggers an automatic (and instantaneous) renumbering operation, while the topic being discussed here allows you to renumber selectively for a range of rows.

Layout Window Edit Menu

Example

Assume that for the CAEPIPE Sample model, you wanted to change the node numbers for the header to begin from 100, and the branch line to begin from 1000. This can be easily done as shown next:

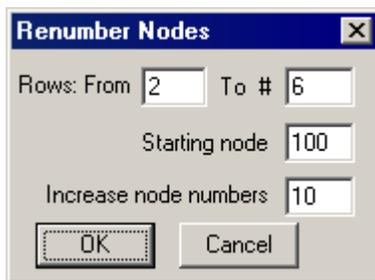


#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe Node numbers shown are before renumbering								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									

1. Changing header node numbers:

Select the menu Edit > Renumber nodes. The Renumber Nodes dialog will open. In the dialog, type in:

- 2 and 6 for From Row# and To Row#,
- 100 for Starting node number and
- 10 for Increase node numbers by.

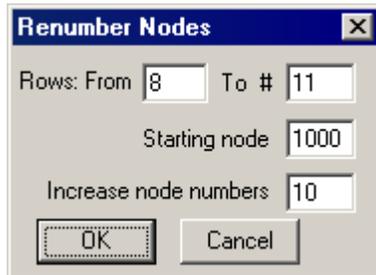


Press Enter (or click on OK) and CAEPIPE changes the node numbers.

2. Changing branch line numbering:

Again, use the menu Edit > Renumber nodes, in the Renumber Nodes dialog, type in

- 8 and 11 for From Row# and To Row#,
- 1000 for Starting node number and
- 10 for Increase node numbers by.



Press Enter (or click on OK) and CAEPIPE changes the node numbers.

The final layout looks like the following. Note that CAEPIPE tracks all occurrences of a specific node number (e.g., 1000, the hanger node, occurs on two rows 4 and 8) throughout the model so you do not have to remember to change every occurrence of the same node number in multiple places in your model.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
2	10	From							Anch
3	20	Bend	9'0"			A53	8	1	
4	1000				6'0"	A53	8	1	Hang
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anch
7	Node numbers shown are after numbering								
8	1000	From							
9	1010		6'0"			A53	6	1	
10	1020	Valve	2'0"			A53	6	1	
11	1030		6'0"			A53	6	1	Anch
12									

Other reasons you may want to use the Renumbering feature:

1. In a big model, multiple edits sometimes bring about confusing node numbering. Use this feature to organize the numbering system.
2. You may need to follow a guideline for numbering just to make parts of the piping system easily recognizable; all branch lines could get node numbers greater than 1000, all bypass lines could get node numbers greater than 2000, and so on.

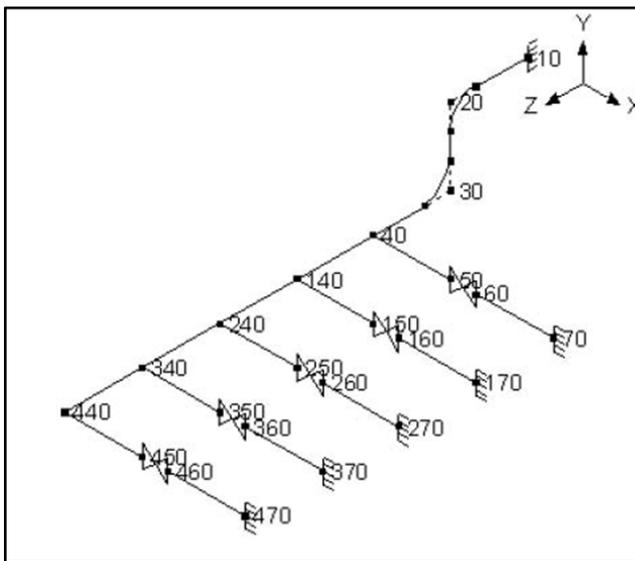
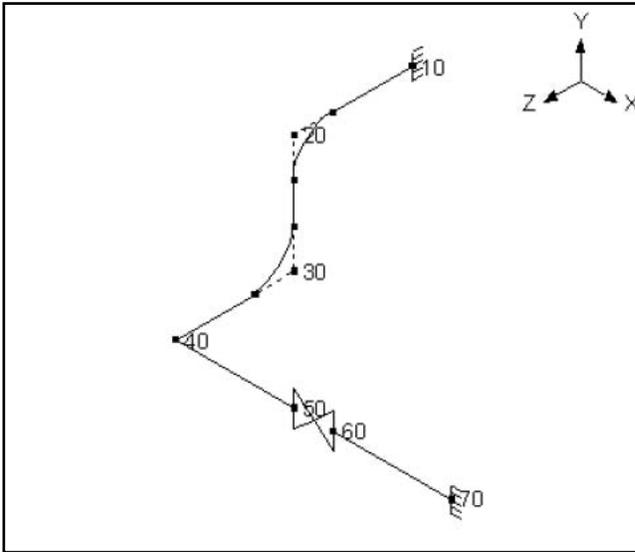
Generate

This powerful command allows you to generate copies of the whole or parts of the model while automatically taking care of node numbering and offsets. Its power is best illustrated by an example.

During “Generate” command execution, CAEPIPE issues a warning and stops when it encounters a Node number that already exists.

Example 1:

Assume that we want to go from the model shown in the first figure to the one shown in the second figure. There are two ways to reach the result. The first is by manually modeling every line like shown in the second figure. The second is the faster way using the Generate command, as you will see.

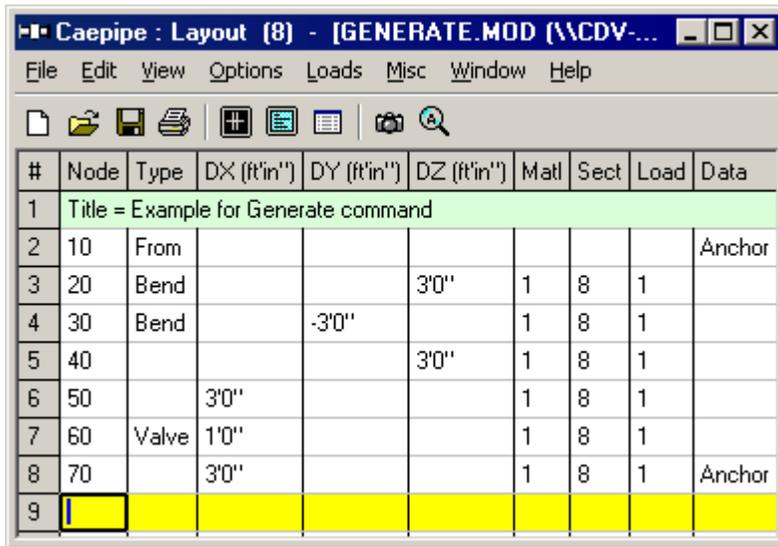


Layout Window Edit Menu

Summary of using the Generate command for the example:

1. First identify the part of the model that repeats and model it. In this case, it is the part comprising of nodes 40 to 70 (rows 5 to 8).
2. Next, use the Generate command to create four additional branches, each starting with say 140, 240, 340 and 440 (these node numbers could be any numbers).
3. Then, create one connecting element (from node 40 to 140).
4. Further generate three additional connecting elements.

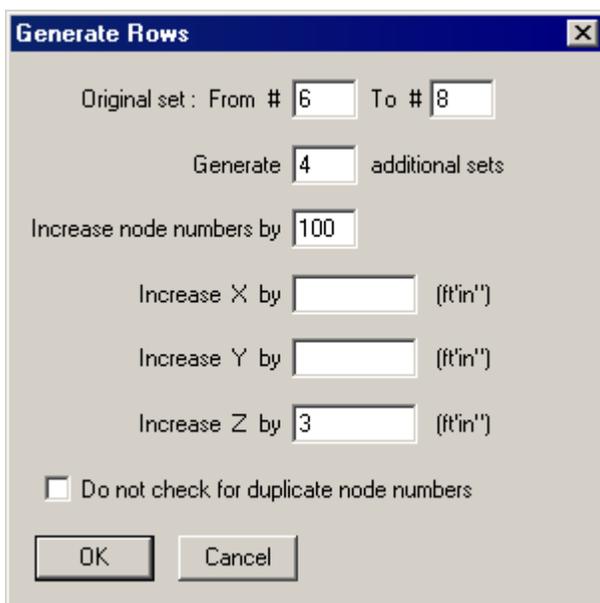
That is it. A number of figures follow to depict the above process pictorially.



The screenshot shows a window titled "Caepipe : Layout (8) - [GENERATE.MOD (\CDV-...". The window contains a menu bar (File, Edit, View, Options, Loads, Misc, Window, Help) and a toolbar with icons for file operations and editing. Below the toolbar is a table with the following data:

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Example for Generate command								
2	10	From							Anchor
3	20	Bend			3'0"	1	8	1	
4	30	Bend		-3'0"		1	8	1	
5	40				3'0"	1	8	1	
6	50		3'0"			1	8	1	
7	60	Valve	1'0"			1	8	1	
8	70		3'0"			1	8	1	Anchor
9									

The above window shows the layout for the starting point of the model. Use the Generate command.



The screenshot shows the "Generate Rows" dialog box with the following settings:

- Original set : From # To #
- Generate additional sets
- Increase node numbers by
- Increase X by (ft'in')
- Increase Y by (ft'in')
- Increase Z by (ft'in')
- Do not check for duplicate node numbers
- Buttons: OK, Cancel

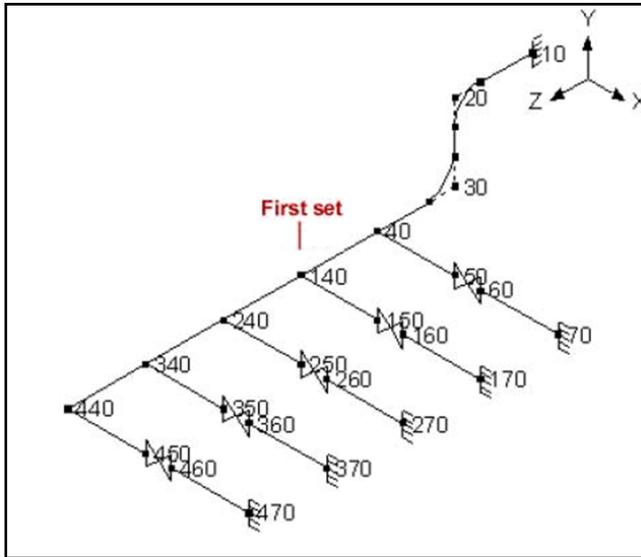
Layout Window Edit Menu

Notice the four sets that are generated (only the first one marked in the following figure).

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Example for Generate command								
2	10	From							Anchor
3	20	Bend			3'0"	1	8	1	
4	30	Bend		-3'0"		1	8	1	
5	40				3'0"	1	8	1	
6	50		3'0"			1	8	1	
7	60	Valve	1'0"			1	8	1	
8	70		3'0"			1	8	1	Anchor
9	140	From		-3'0"	9'0"				
10	150		3'0"			1	8	1	
11	160	Valve	1'0"			1	8	1	
12	170		3'0"			1	8	1	Anchor
13	240	From		-3'0"	12'0"				
14	250		3'0"			1	8	1	
15	260	Valve	1'0"			1	8	1	
16	270		3'0"			1	8	1	Anchor
17	340	From		-3'0"	15'0"				
18	350		3'0"			1	8	1	
19	360	Valve	1'0"			1	8	1	
20	370		3'0"			1	8	1	Anchor
21	440	From		-3'0"	18'0"				
22	450		3'0"			1	8	1	
23	460	Valve	1'0"			1	8	1	
24	470		3'0"			1	8	1	Anchor
25	40	From							

Look at the graphics that corresponds to the above Layout window.

Layout Window Edit Menu



Next, create one connecting element between 40 and 140, and generate the remaining three.

24	470		3'0"			1	8	1	Anchor
25	40	From	} First connecting element						
26	140			1	8	1			

Type 40 on an empty row, Tab to Type, press “F” for “From,” press Enter (rows 25 and 26 above). Now, generate the rest.

Generate Rows ✕

Original set : From # To #

Generate additional sets

Increase node numbers by

Increase X by (ft'in')

Increase Y by (ft'in')

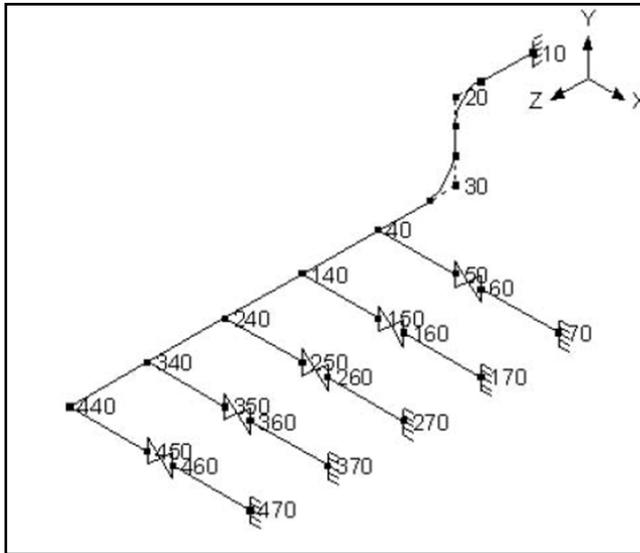
Increase Z by (ft'in')

Do not check for duplicate node numbers

Layout Window Edit Menu

Caepipe : Layout (29) - [GENERATE.MOD (C:\Use...)]									
File Edit View Options Loads Misc Window Help									
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Example for Generate command								
2	10	From							Anchor
3	20	Bend			3'0"	1	8	1	
4	30	Bend		-3'0"		1	8	1	
5	40				3'0"	1	8	1	
6	50		3'0"			1	8	1	
7	60	Valve	1'0"			1	8	1	
8	70		3'0"			1	8	1	Anchor
9	140	From		-3'0"	9'0"				
10	150		3'0"			1	8	1	
11	160	Valve	1'0"			1	8	1	
12	170		3'0"			1	8	1	Anchor
13	240	From		-3'0"	12'0"				
14	250		3'0"			1	8	1	
15	260	Valve	1'0"			1	8	1	
16	270		3'0"			1	8	1	Anchor
17	340	From		-3'0"	15'0"				
18	350		3'0"			1	8	1	
19	360	Valve	1'0"			1	8	1	
20	370		3'0"			1	8	1	Anchor
21	440	From		-3'0"	18'0"				
22	450		3'0"			1	8	1	
23	460	Valve	1'0"			1	8	1	
24	470		3'0"			1	8	1	Anchor
25	40	From							
26	140					1	8	1	
27	240					1	8	1	
28	340					1	8	1	
29	440					1	8	1	
30									

Full model layout



Final model as shown is the desired result. So, as you saw, Generate command allowed us to create four branch lines from one existing line in three operations.

Example 2:

See example under Beam section in Reference: (Example 1: Pipe Rack using Beams) p. 107.

Regenerate

Sometimes, owing to multiple edits, the model may be in an indeterminate stage. If you have made many edits, use this command to regenerate the model which will reinterpret the input and bring the model to a consistent state.

Duplicate Last Row (Ctrl+Enter)

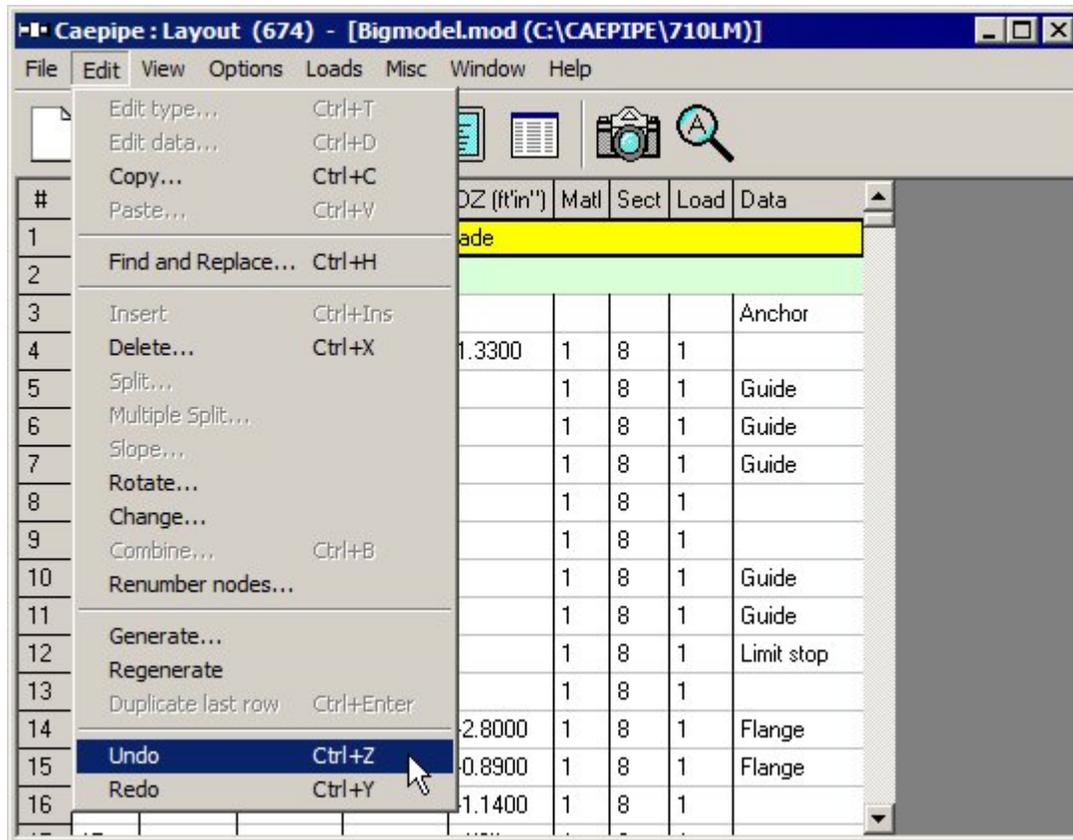
This is another handy feature that increases your productivity. For rows that repeat themselves (except for node number), this is a time saver. This is useful in modeling pipe racks or straight runs of pipe. Simply input all data on to a row and press Ctrl+Enter to duplicate this row on to the next with the node number automatically incremented.

CAEPIPE cannot duplicate a Location/Comment/Hydrotest row, nor can it duplicate a row between two existing rows. This command works only on new empty rows at the end of a model. If you press Ctrl+Enter right after a Location/Comment/Hydrotest row, CAEPIPE will duplicate the row right before the Location/Comment/Hydrotest row.

Undo (Ctrl+Z)

CAEPIPE can perform an Undo operation over 1000 times (1024)! This operation includes most user-input/edit actions through the Layout, List and Graphics windows.

Layout Window Edit Menu



Example 1: In the figure above, if you delete entire rows 10-14 one by one (see 1st col. for row #) and the first three guides (on rows 5-7), you can use the Undo command (Ctrl+Z) eight times to undo all of these operations.

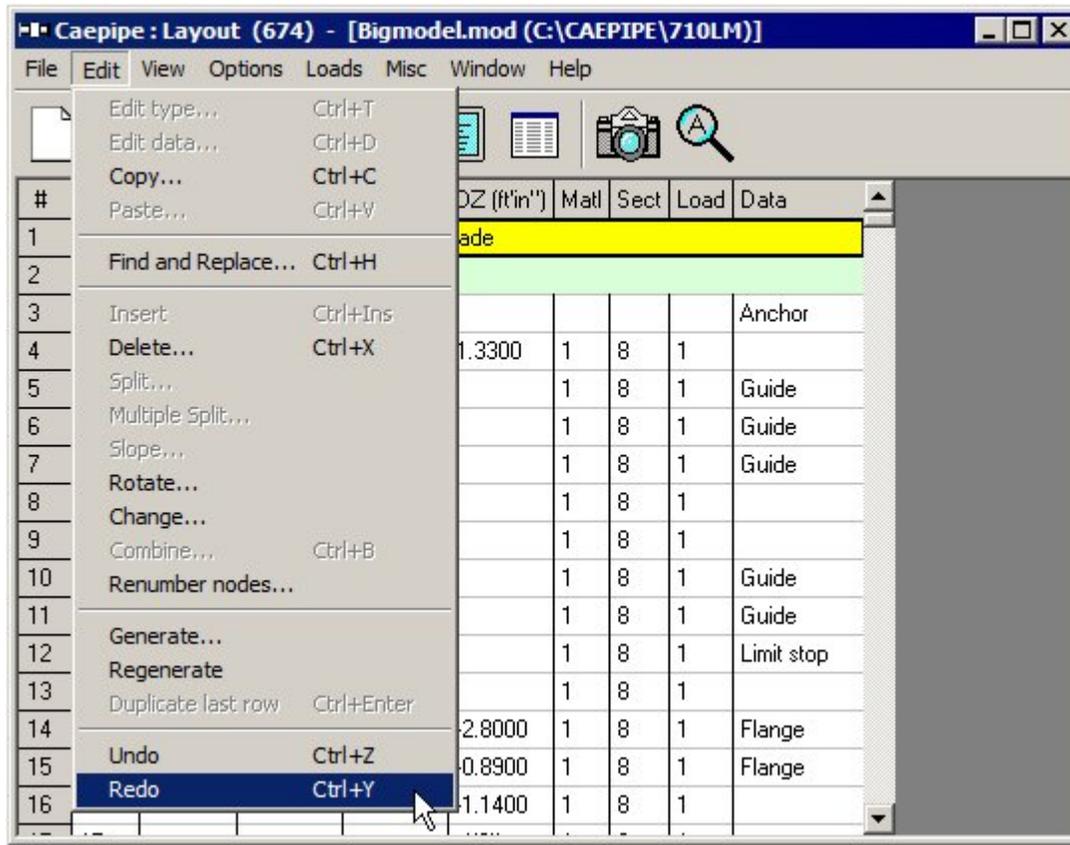
Example 2: If you were to change a few of the analysis options (such as piping code, # of thermal loads and Bourdon effect through the menus), you can undo these changes by using this command three times, even though these changes are not made visible.

Redo (Ctrl+Y)

Conversely, CAEPIPE can perform a Redo operation over a 1000 times (1024), which can include most user-input/edit actions through Layout, List and Graphics windows, including recently performed Undo operations.

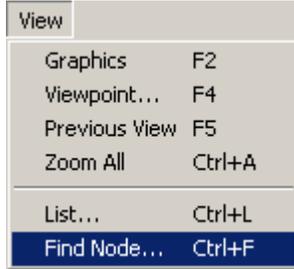
For example 1 under Undo: You can use the Redo operation to delete what you undid in the above example! In other words, delete rows 10-14 and the first three guides.

Layout Window Edit Menu



Layout Window View Menu

View Menu

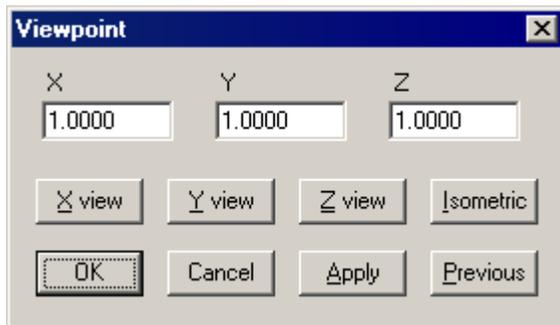


A few commands for the Graphics window can be carried out from here.

Graphics (F2)

Use this command to move focus to the Graphics window. If the window is not open, then F2 will open the Graphics window and move focus to it.

Viewpoint (F4)



Use this command to set the graphics viewpoint. Several useful buttons inside the dialog allow you to change viewpoint to a preset one. For example, if you want to see the “plan” view (Y-vertical), click on “Y view” button.

Previous View (F5)

Use this command to display the previously viewed graphics (zoom level and image area) in the Graphics window.

Zoom All (Ctrl+L)

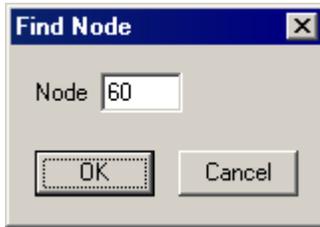
Use this command to view the whole model inside the Graphics window. Note that this command does not move focus to the Graphics window but brings the whole model graphics into view.

List (Ctrl+L)

This is one of CAEPIPE’s unique features that allows you to view itemized lists of every input data (pipes, bends, materials, valves, spectra, etc.) in a separate window. See more detailed explanation under the section Layout Window in Reference.

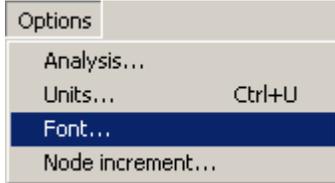
Find Node (Ctrl+F)

Use this command to search for a node number inside this window. You may also search for jacket and bend end nodes (e.g., 20A, 30C, 160J), *if* they are referenced on the layout screen.



Layout Window Options Menu

Options Menu



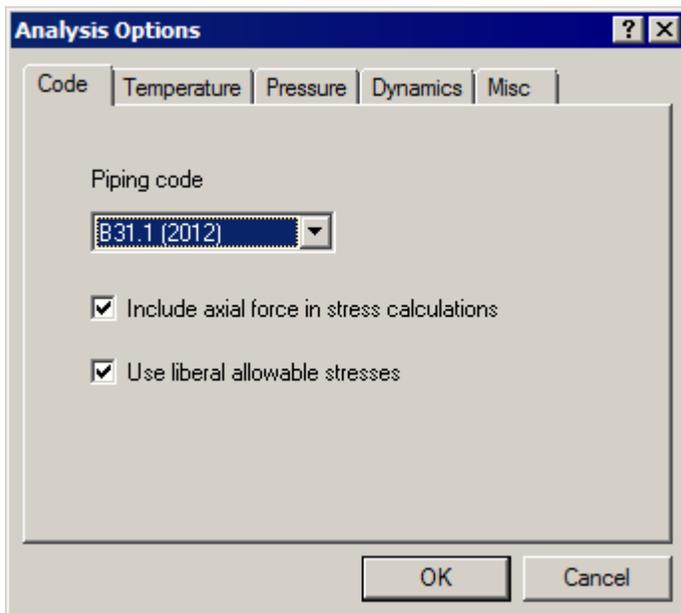
This menu allows you to specify analysis options, units, font and the node number increment (for automatic generation of the next node number while inputting a model).

Analysis

The Analysis Options dialog is shown. Here you can specify analysis options related to piping codes, temperature, pressure, dynamic analysis, etc., in various tabs of the dialog.

Code

On this tab you can choose the piping code and also set options for that piping code.



Layout Window Options Menu

Piping Code

Select a piping code from the “Piping code” drop-down combo box. The following codes are currently available:

- None
- B31.1
- B31.1 (1967)
- B31.3
- B31.4
- B31.5
- B31.8
- B31.9
- ASME Section III Class 2 (1980, 1986 and 1992)
- BS 806 (British)
- EN 13480 (European)
- Norwegian (1983 and 1990)
- RCC-M (French)
- CODETI (French)
- Stoomwezen (Dutch)
- Swedish
- Z183 (Canadian)
- Z184 (Canadian)

Notes:

- When the selected piping code is “None,” a “Static analysis” load case, which includes weight, pressure, thermal, cold spring, static seismic and wind loads, all applied at the same time, is available for analysis. No dynamic loads (time history, response spectrum, etc.) are included. For “Piping code = None” and a non-FRP material, the in-plane and out-of-plane Stress Intensification Factors (SIF) for a 90° bend/elbow are taken to be the same as those computed using ANSI B31.3. For a bend with bend angle less than 90°, the in-plane SIF is computed using the procedure given in ASME/BPVC Section III, Division 1, Case N-319-2 “Alternate Procedure for Evaluation of Stresses in Butt Welding Elbows in Class 1 Piping.”
- For some piping codes, you need to select an additional option (such as Design Factor for B31.8, equation level for ASME and RCC-M, Location Factor for Z183 and Z184, Occasional load factor for EN13480). These are shown below.

Layout Window Options Menu

Piping code <input type="text" value="B31.8 (2012)"/>	Design Factor F <input type="text" value="0.80"/>															
<input checked="" type="checkbox"/> Include axial force in stress calculations	<input type="text" value="0.80"/> <input type="text" value="0.72"/> <input type="text" value="0.60"/> <input style="background-color: #000080; color: #ffffff;" type="text" value="0.50"/> <input type="text" value="0.40"/>															
<input checked="" type="checkbox"/> Use liberal allowable stresses																
Piping code <input type="text" value="ASME Class 2 (1980)"/>	Equation 10 Level <input type="text" value="B (Upset)"/>															
<input type="checkbox"/> Include axial force in stress calculations	<input type="text" value="B (Upset)"/> <input style="background-color: #000080; color: #ffffff;" type="text" value="C (Emergency)"/> <input type="text" value="D (Faulted)"/>															
Piping code <input type="text" value="RCC-M (1985)"/>	Equation 10 Level <input type="text" value="B (Upset)"/>															
<input type="checkbox"/> Include axial force in stress calculations	<input type="text" value="B (Upset)"/> <input style="background-color: #000080; color: #ffffff;" type="text" value="C (Emergency)"/> <input type="text" value="D (Faulted)"/>															
Piping code <input style="background-color: #000080; color: #ffffff;" type="text" value="Z183 (1990)"/>	Location Factor (L) <input checked="" type="radio"/> 1.0 <input type="radio"/> 0.8															
<input type="checkbox"/> Include axial force in stress calculations																
Piping code <input style="background-color: #000080; color: #ffffff;" type="text" value="Z184 (1992)"/>																
<input type="checkbox"/> Include axial force in stress calculations																
Location Factor (L)																
<table border="0" style="width: 100%; border-collapse: collapse;"> <tr> <td style="padding-right: 10px;">Class location</td> <td><input checked="" type="radio"/> 1</td> <td><input type="radio"/> 2</td> <td><input type="radio"/> 3</td> <td><input type="radio"/> 4</td> </tr> <tr> <td><input checked="" type="radio"/> Nonsour service</td> <td>1.000</td> <td>0.900</td> <td>0.700</td> <td>0.550</td> </tr> <tr> <td><input type="radio"/> Sour service</td> <td>0.900</td> <td>0.750</td> <td>0.625</td> <td>0.500</td> </tr> </table>		Class location	<input checked="" type="radio"/> 1	<input type="radio"/> 2	<input type="radio"/> 3	<input type="radio"/> 4	<input checked="" type="radio"/> Nonsour service	1.000	0.900	0.700	0.550	<input type="radio"/> Sour service	0.900	0.750	0.625	0.500
Class location	<input checked="" type="radio"/> 1	<input type="radio"/> 2	<input type="radio"/> 3	<input type="radio"/> 4												
<input checked="" type="radio"/> Nonsour service	1.000	0.900	0.700	0.550												
<input type="radio"/> Sour service	0.900	0.750	0.625	0.500												
Piping code <input style="background-color: #000080; color: #ffffff;" type="text" value="EN 13480 (2012)"/>	Occasional load factor (k) <input type="text" value="1.20"/>															
<input type="checkbox"/> Include axial force in stress calculations																

Include axial force in stress calculations

By checking or un-checking this box, you can include or exclude axial force in stress calculations (F/A term). This option applies to all stresses (Sustained, Occasional and Expansion) and is provided since many piping codes do not clearly mandate the axial term's inclusion. When included, CAEPIPE includes the axial term F/A (where F=axial load, A=Pipe cross-sectional area) in its calculation of all stress equations, S_L , S_O and S_E .

Note that this option is always turned on for codes such as B31.3, B31.4 and B31.8.

Use Liberal Allowable Stresses

For B31.1, B31.1(1967), B31.3, B31.5 and CODETI piping codes, the allowable expansion stress may be increased by a difference between the allowable sustained stress and the actual sustained stress times the stress range reduction factor. See Thermal Expansion Stress Range or equivalent under each piping code in Appendix A.

A piping code committee member opines: Perhaps the term “liberal allowable” is not the best to describe the allowable. The allowable stress in B31.1 Eq. 13, i.e., $S_A + f(S_h - S_L)$, can be used anytime. The only prerequisite is knowing the S_L stresses, which will only be the case after the supports are known or implicit. The Eq. 1 allowable stress, S_A , is the allowable stress traditionally used in the flexibility analysis when no supports other than the equipment anchors are known. The flexibility analysis is used to determine whether the layout of piping between the equipment anchors is adequate. The displacements determined in the flexibility analysis will allow the designer to devise the pipe weight supports to interfere with the flexibility of the pipe as little as possible, i.e., the designer will use rigid supports where the piping does not move much, use variable springs where the piping moves a small amount (most typically 1/4” to 4”), and use constant springs where the movement is great (again, typically over 4”). For lateral loads the same concept as used for the pipe weight supports is used for the lateral supports, i.e., if lateral displacements are small, rigid supports may be used, for larger lateral movements, gapped or shock suppressor supports are used (although shock suppressor supports require considerable maintenance attention and in the long run are usually not preferable to gapped supports).

General Note on Thermal Allowable Calculation for All Piping Codes

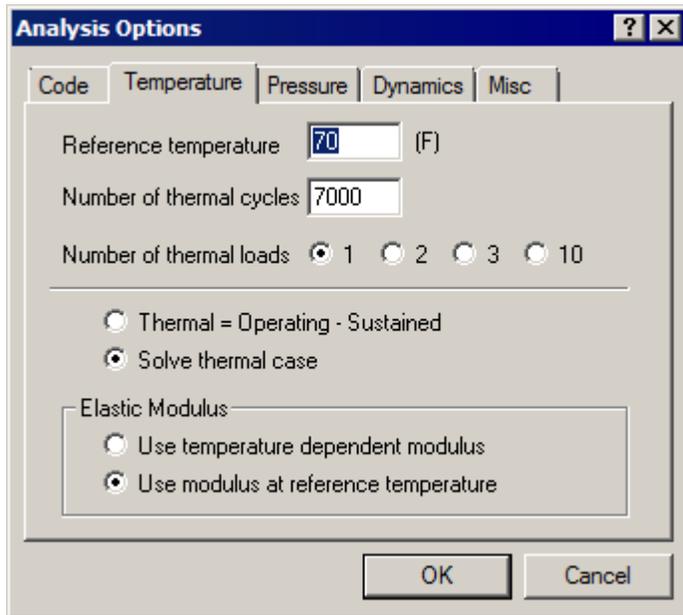
Cold allowable (S_c) and Hot allowable (S_h) stresses used in thermal stress calculations are calculated as follows for all piping codes:

- For Expansion Cases T1 through T10, Cold allowable (S_c) = Minimum (T_{ref} , T_n) and Hot allowable (S_h) = Maximum (T_{ref} , T_n), where $n = 1$ to 10
- For Thermal Ranges T_x - T_y , Cold allowable (S_c) = Minimum (T_x , T_y) and Hot allowable (S_h) = Maximum (T_x , T_y), where $x = 1$ to 10 and $y = 1$ to 10 and also x is not equal to y .

The specifics are listed for each code in the Section Modulus topic (See end of Appendix A).

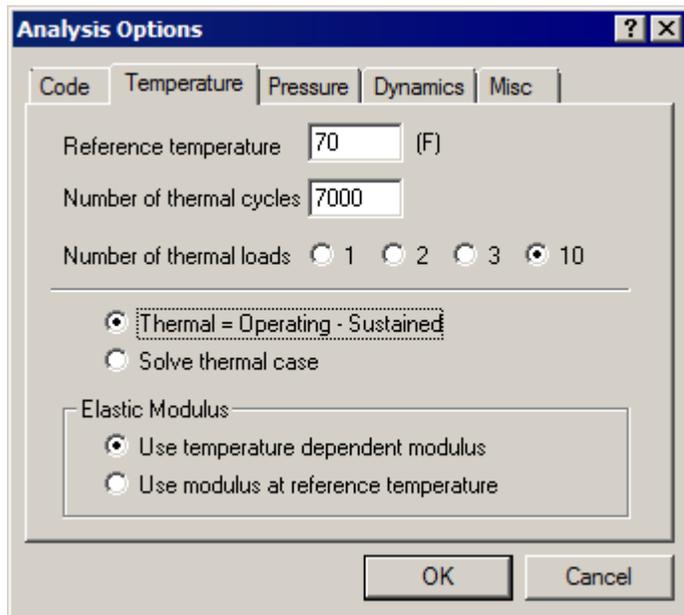
Temperature

On this tab you can set options related to thermal loads.



Reference Temperature

Type in the reference temperature here, usually 70°F (21.1°C). First, CAEPIPE uses this temperature to lookup the material properties table for the value of the cold modulus (cold = Reference). Second, this temperature's corresponding coefficient of thermal expansion is used for thermal expansion range calculations [e.g., $T_1 - T_2 = (T_1 - T_{ref}) - (T_2 - T_{ref})$].



Layout Window Options Menu

Number of Thermal cycles

CAEPIPE uses this number to determine the stress range reduction factor, f , which is used to reduce the allowable expansion stress range, S_A .

Number of Cycles	Factor (f)
7,000 and less	1.0
Over 7,000 to 14,000	0.9
Over 14,000 to 22,000	0.8
Over 22,000 to 45,000	0.7
Over 45,000 to 100,000	0.6
Over 100,000	0.5

The typical equation for calculating thermal expansion stress range (e.g., for B31.1 code) is:

$$S_A = f(1.25S_c + 0.25S_h)$$

where

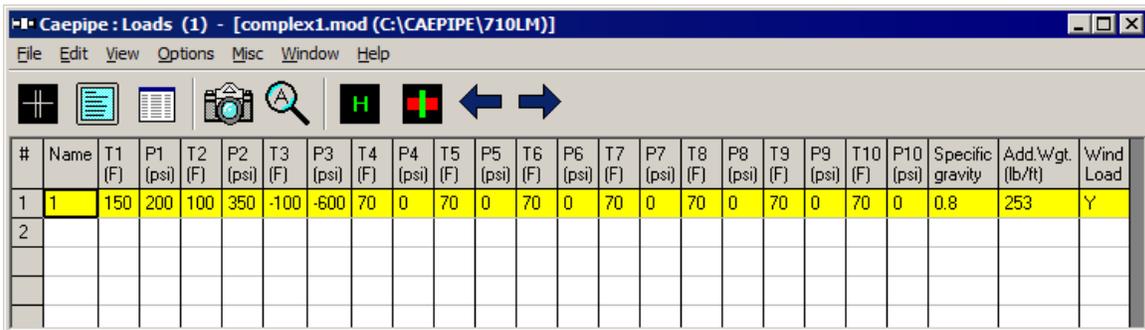
S_c = allowable stress at cold temperature

S_h = hot allowable stress

f = stress range reduction factor (from the above table).

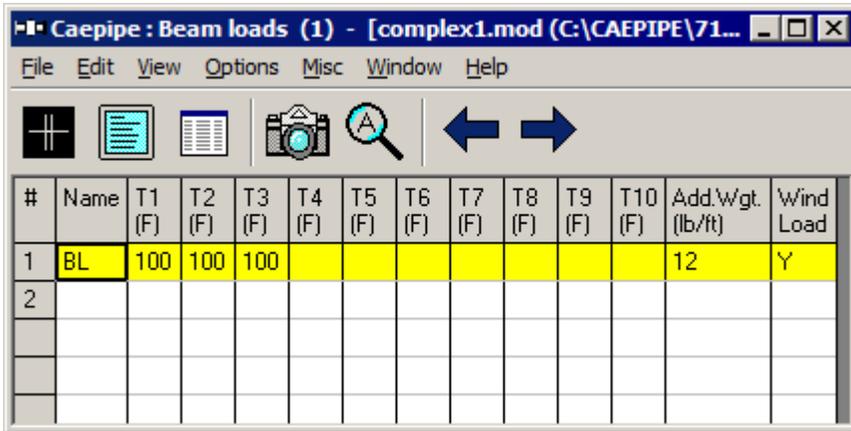
Number of Thermal Loads

You can enter up to 10 thermal loads. This feature must be first set through Layout window > Options > Analysis > Temperature. Select the number of thermal loads an element can see during operation. Up to 10 temperatures may be applied as part of Load (along with 10 pressures) under Misc menu > Loads.

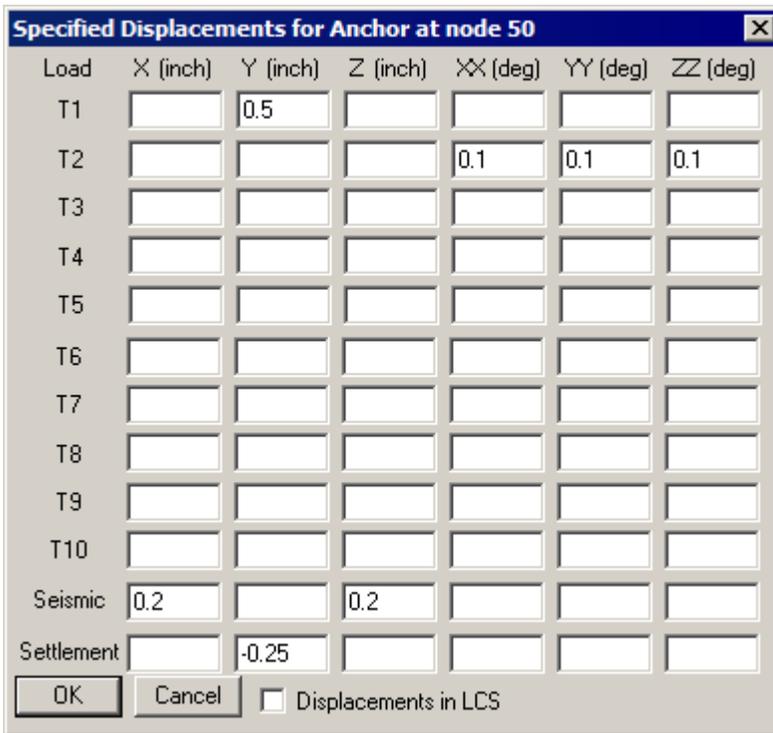


This also enables you to enter 10 temperatures under Misc menu > Beam Loads.

Layout Window Options Menu



Additionally, up to 10 specified thermal displacements can be entered for Anchor and Nozzle data types (Nozzle dialog not shown).



Thermal = Operating – Sustained

Thermal load case results can be calculated as the difference between the Operating and the Sustained load cases, i.e., the thermal load case is not solved independently. This is the recommended procedure to solve thermal cases (T1 through T10) especially when nonlinearities (limit stops, friction, etc.) are present.

When a model has no nonlinearities, the forces, moments and displacements at nodes will add up (i.e., Sustained load + Thermal load = Operating load). When a nonlinearity is present, each load case can be solved independently, i.e., the numbers are not added from Sustained (W+P) to Thermal (T1) to get Operating (W+P+T1); in other words, the results do not always add up. In the past, this used to be the only method of solving when nonlinearities were present. However, now you can use the new (recommended) option called "Thermal = Operating – Sustained." Use it to solve for the thermal case as the

Layout Window Options Menu

difference between Operating and Sustained cases, which will also ensure that sustained and thermal case results add up to operating case results.

Solve Thermal case

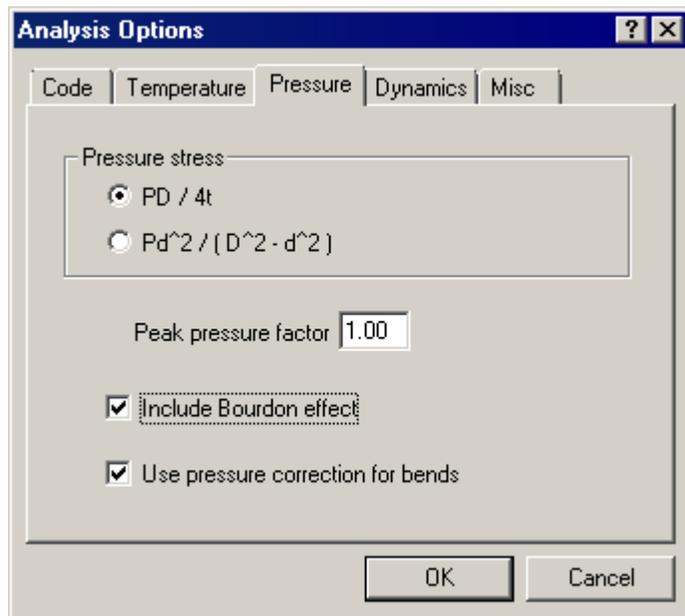
Results for the thermal load case are obtained from solving this case independently. See above discussion.

Elastic Modulus

CAEPIPE will use the elastic modulus at the recommended reference temperature (Cold modulus) for code compliance calculations. Or, it will use the temperature-dependent modulus (modulus at T1, T2, T3, T4, T5, T6, T7, T8, T9, or T10) for support load calculations for Sustained and Operating load cases. CAEPIPE uses the cold modulus always for the thermal load case.

Pressure

On this tab you can set options related to pressure loads.



Pressure Stress

The longitudinal pressure stress may be calculated as: $pD/4t$ or $pd^2/(D^2 - d^2)$, both given as options in a few ASME/B31 Codes. For B31.8, as per code, the latter pressure stress term is not an option.

Peak Pressure Factor

For occasional loads (seismic, spectrum and wind), the pressure 'p' in longitudinal pressure stress is multiplied by this factor.

Bourdon Effect

Bourdon effect is the tendency of straight pipes to elongate and bends to straighten due to internal pressure. This may be included or excluded from the analysis by checking or unchecking the box.

Layout Window Options Menu

Internal pressure will expand the pipe cross-section radially outward (i.e., bulge out) thereby contracting the length of the pipe due to Poisson's effect. On the other hand, external pressure will contract the pipe cross-section radially inward (i.e., shrinking inwards) thereby elongating the pipe length again due to Poisson's effect. In addition, due to end-cap force, bends open out due to internal pressure or deform inward due to external pressure. These physical phenomena are included in CAEPIPE calculations when the "Bourdon effect" button is turned ON. Pipe stresses and support loads generated due to such "Bourdon effect" deformations are generally considered as "Secondary Stresses and Support loads" and normally considered as part of the thermal expansion load case.

Therefore, by default, this effect is treated as an expansion load and included in the expansion and operating load cases. It is not applied to the sustained and occasional load cases.

If an environment variable ("BOURDONP") is set (to YES), however, the Bourdon effect is instead treated as a sustained load and included in the sustained and operating load cases. It is not applied to the expansion load case.

For a straight pipe, the following equation is used.

$$\text{Pressure deflection} = \frac{\text{Pressure} \times r_m \times L \times (1 - 2\nu)}{2 \times E \times t}$$

where

r_m = mean radius of the pipe

L = length of the pipe

ν = Poisson's Ratio

E = Elastic Modulus (Axial modulus in the case of FRP pipes)

t = pipe thickness

Pressure Correction for Bends (Pressure stiffening effect)

Pressure correction for bends is different from and unaffected by Bourdon effect. In large diameter thin-wall bends, pressure can significantly affect the flexibility and SIF. If pressure correction for bends is used: the Flexibility of the bend is divided by

$$1 + 6 \times \left(\frac{P}{E}\right) \left(\frac{r}{t}\right)^{7/3} \left(\frac{R}{r}\right)^{1/3}$$

and the SIF for the bend is divided by

$$1 + 3.25 \times \left(\frac{P}{E}\right) \left(\frac{r}{t}\right)^{5/2} \left(\frac{R}{r}\right)^{2/3}$$

where

P = pressure

E = elastic modulus

r = mean radius of matching pipe

t = nominal wall thickness

R = bend radius

It is recommended that this option be turned 'ON' whenever the internal or external pressure is high, so that the stiffening of the bend due to internal or external pressure is considered in the analysis. In other words, when a thin-walled pipe bend is bent by bending

Layout Window Options Menu

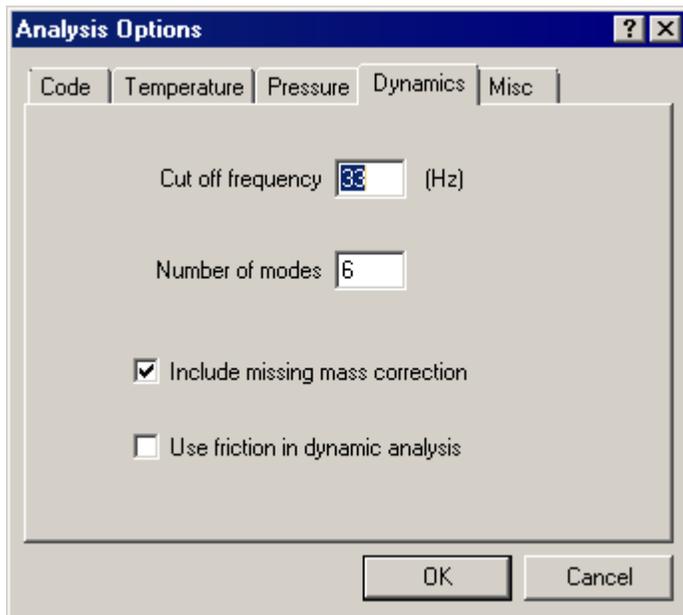
moments at its two ends, the bend will ovalize and increase its flexibility. Internal or external pressure will reduce this bend ovalization, thereby increasing its stiffness.

Pressure correction decreases the flexibility of the piping system (by increasing the stiffness of the system because of the stiffened elbows). Hence, the system frequencies (in modal analysis) tend to increase.

Related information can be found in Appendix D of most B31 piping codes.

Dynamics

On this tab you can set options related to dynamic analysis.



Cutoff Frequency / Number of Modes

These two options jointly control how the modal analysis routine works to extract the natural frequencies. Frequency (Hertz) and the chosen number of modes will determine the minimum number of modes extracted. The modal analysis will terminate either when the number of modes requested has been extracted or after extracting an additional frequency above the specified cutoff frequency value, whichever occurs first.

For earthquake analysis, a typical value for cutoff frequency is 33 Hz. The maximum frequency you can input is 9999 Hz.

When the selected piping code is "None," and the model contains nonlinearities (limit stops, certain expansion joints), a static analysis for a given model with and without a g-load input may produce different frequencies depending upon the status of the nonlinearities due to the presence or absence of the g-load in the static load case.

Include Missing Mass Correction

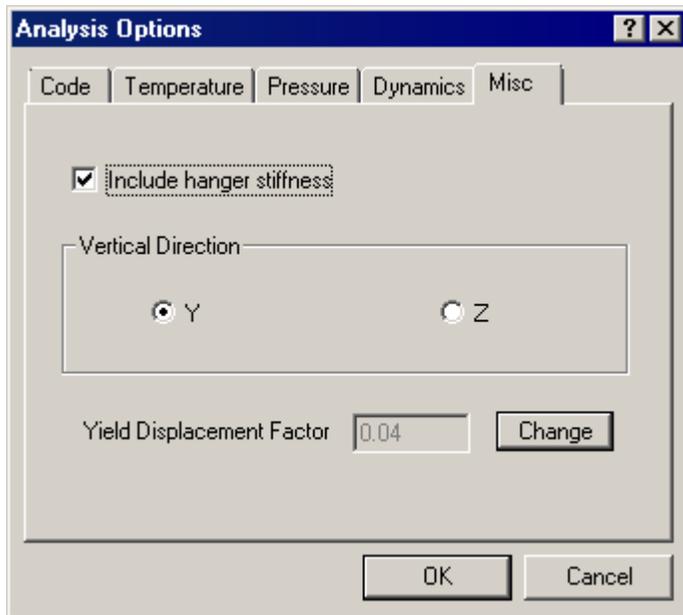
Missing mass correction to the response spectrum analysis can be included using this check box. See topic by the same name in Reference.

Use Friction in Dynamic Analysis

Friction is optional in dynamic analysis. If friction is included in dynamic analysis, the global stiffness matrix arrived at upon completion of the iterative calculations for the first operating case (W+P1+T1) is used to compute frequencies and mode shapes for the piping system.

Misc

On this tab you can set miscellaneous options.



Include Hanger stiffness

When the checkbox is checked, CAEPIPE adds the hanger spring rates for variable spring and user hangers to the overall stiffness matrix (recommended setting). Hanger spring rates for rod hangers (if active) are always included even when this checkbox is not checked.

If the checkbox is not checked, CAEPIPE does not include the hanger stiffnesses in the analysis. Some users prefer this as it more closely matches “hand calculations.”

In the 1970s, performing pipe stress analysis on mainframe computers was expensive (about \$1,000/run). So, pipe stress analysts would run a Thermal (T1) case alone first. Then, they would run a deadweight (DW) case with rigid supports at chosen hanger locations. Using the support loads and the hanger travel, analysts would select appropriate variable spring hangers from hanger manufacturer catalogs. To save on computing costs, they would not update the global stiffness matrix [K] with the hanger stiffnesses from the newly selected hangers because they would have to reanalyze the model which would cost more money. Many industrial plants built 30 or 40 years ago in the USA and presumably in other countries have pipe stress analyses done in this manner. These plants have archived such analysis reports that do not include hanger stiffnesses (as part of the global stiffness matrix [K]).

In later years, as and when the analysts in these plants needed to reanalyze those piping systems, they had to use modern programs like CAEPIPE. But, before they accepted the new results, they sometimes liked to verify that the new results (from CAEPIPE) matched the results from the old reports generated by the mainframe pipe stress analysis programs.

Layout Window Options Menu

This is why CAEPIPE provides the option “Do not include (or include) hanger stiffnesses.” It helps the engineers compare results from CAEPIPE with the old archived reports.

Today, with cheap PC computing power, there is no reason why hanger stiffnesses should be excluded from analysis. In fact, including hanger stiffnesses provides a more accurate picture of system behavior. As such, we recommend that you “include hanger stiffnesses” in every analysis.

Vertical Direction

You may specify either Y or Z as the vertical global axis for the model. Be careful when you change this setting on an existing model because all the offsets from the previous vertical axis will be transferred to the new vertical axis, and you will have to perform a rotate operation on the model to regain the old orientation.

Yield Displacement Factor

Soil (in Buried piping analysis) is modeled with an initial stiffness and an ultimate load, after reaching which, displacement continues without a further increase in load (i.e., yield stiffness is zero). You can change the initial stiffness by changing the Yield Displacement Factor (default=0.04, range between 0.04 to 1.0). No changes are recommended unless there is a problem with convergence during Buried Piping analysis.

Units (Ctrl+U)

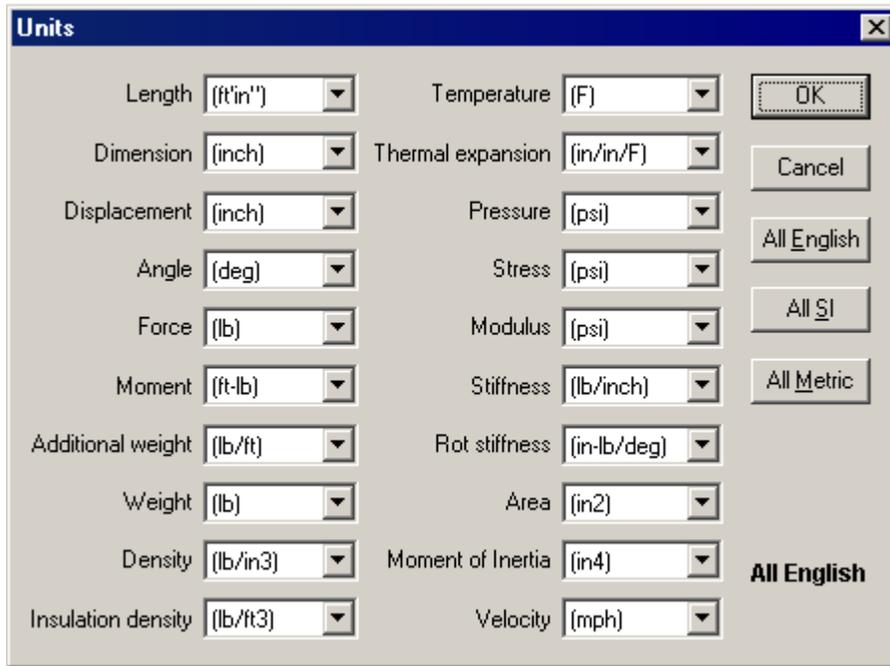
A flexible command, Units allows you to set any combination of units: English, SI or Metric, for any item. For convenience “All English,” “All SI” and “All Metric” buttons set all the units to that particular choice with a single button click.

As an example, engineers in the USA sometimes use lb./in. for stiffnesses but use SI units for the rest, for their international clients. The specific combination that you create is saved with the model so you do not have to reset them every time. You could do your analysis in English units and present your results to your client in SI units with the click of a button.

Layout Window Options Menu

All English

All English units are shown below.



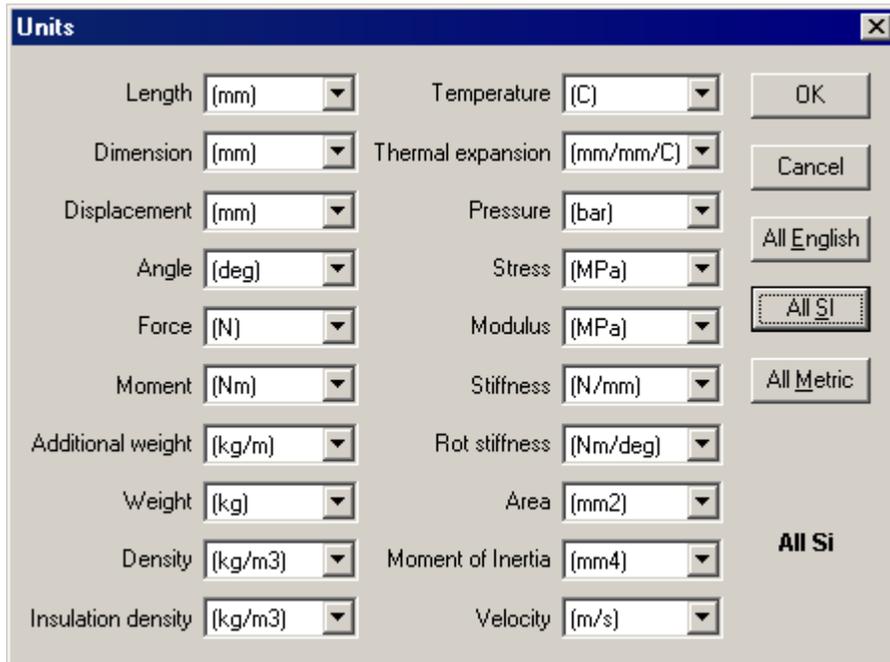
The screenshot shows the 'Units' dialog box with the following settings:

Property	Unit
Length	(ft'in")
Temperature	(F)
Dimension	(inch)
Thermal expansion	(in/in/F)
Displacement	(inch)
Pressure	(psi)
Angle	(deg)
Stress	(psi)
Force	(lb)
Modulus	(psi)
Moment	(ft-lb)
Stiffness	(lb/inch)
Additional weight	(lb/ft)
Rot stiffness	(in-lb/deg)
Weight	(lb)
Area	(in ²)
Density	(lb/in ³)
Moment of Inertia	(in ⁴)
Insulation density	(lb/ft ³)
Velocity	(mph)

Buttons on the right: OK, Cancel, All English, All SI, All Metric. The 'All English' button is highlighted.

All SI

All SI units are shown below.



The screenshot shows the 'Units' dialog box with the following settings:

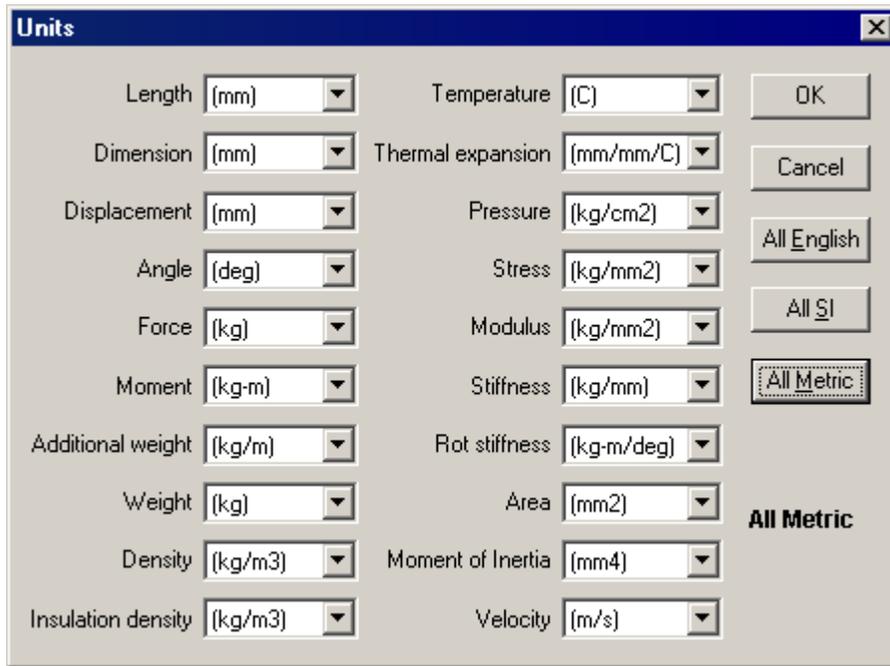
Property	Unit
Length	(mm)
Temperature	(C)
Dimension	(mm)
Thermal expansion	(mm/mm/C)
Displacement	(mm)
Pressure	(bar)
Angle	(deg)
Stress	(MPa)
Force	(N)
Modulus	(MPa)
Moment	(Nm)
Stiffness	(N/mm)
Additional weight	(kg/m)
Rot stiffness	(Nm/deg)
Weight	(kg)
Area	(mm ²)
Density	(kg/m ³)
Moment of Inertia	(mm ⁴)
Insulation density	(kg/m ³)
Velocity	(m/s)

Buttons on the right: OK, Cancel, All English, All SI, All Metric. The 'All SI' button is highlighted.

Layout Window Options Menu

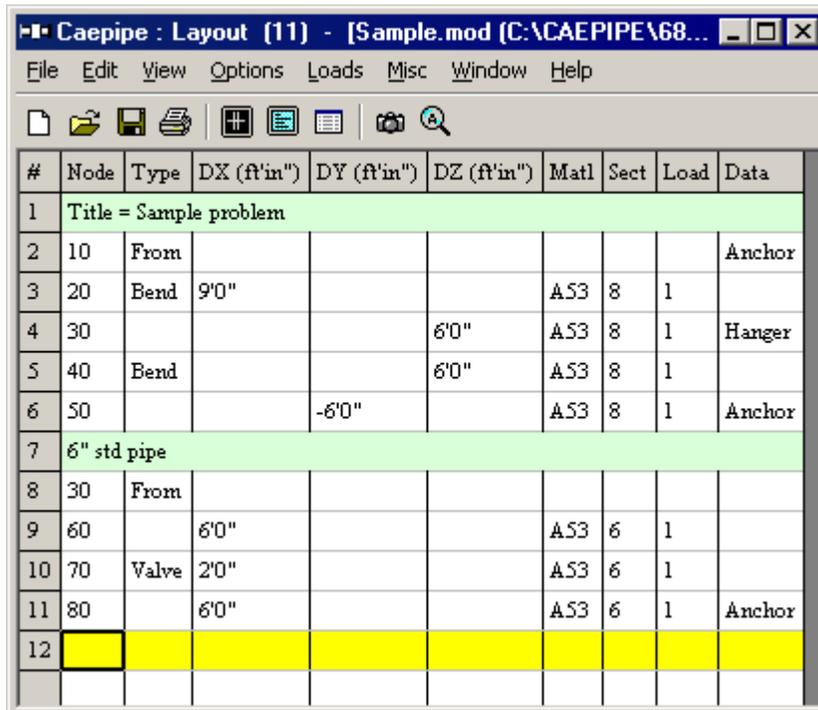
All Metric

All Metric units are shown below.



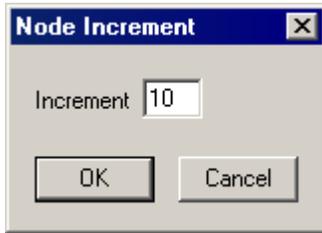
Font

You can display the text in all CAEPIPE text windows (Layout, List and Results) in a font face and size of your choice. Here, the font chosen is Times New Roman, 8 point:



Node Increment

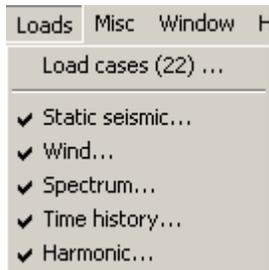
You can increase your productivity by having CAEPIPE automatically increment the node number when you model a piping system in CAEPIPE. You can turn this feature off by specifying a zero (0) increment here. The default is set to 10.



Layout Window Loads Menu

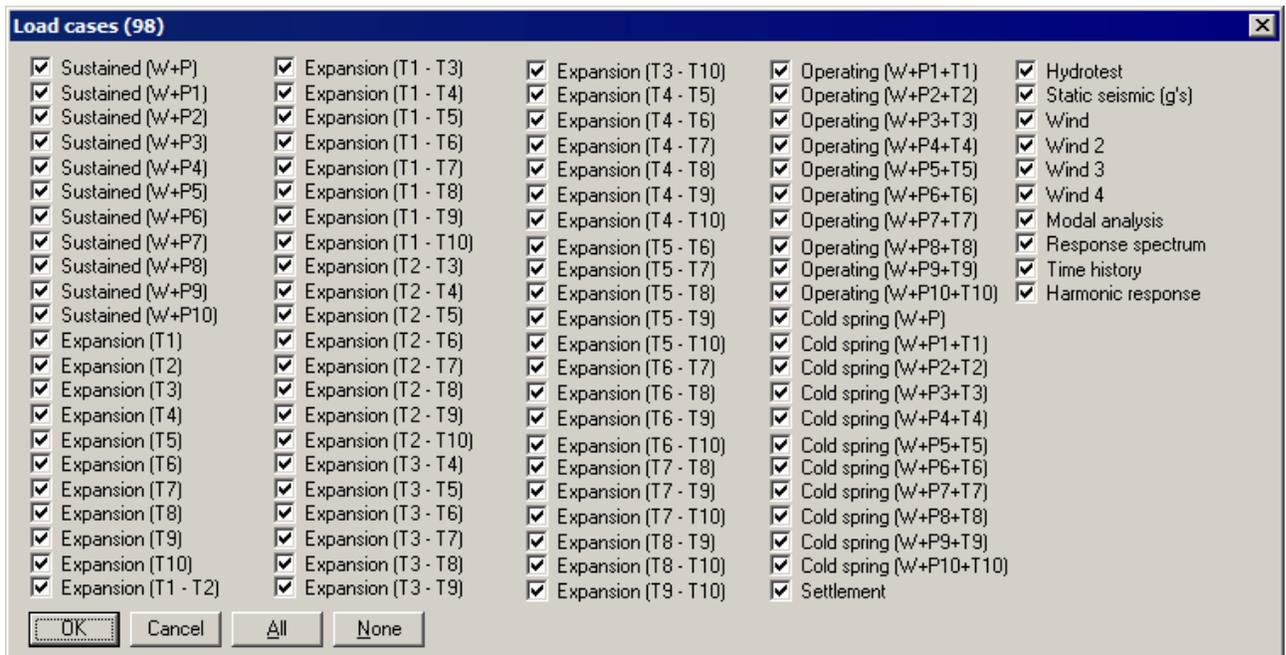
Loads Menu

From this menu, you can specify the load cases for analysis.



Load cases

Click on this command to select the different load cases for analysis. Most of the time, selecting a load case is as simple as checking the corresponding checkbox.



For a few others, load cases appear in this dialog *only after* you input their related data. For example, to perform a Response Spectrum analysis, you need to

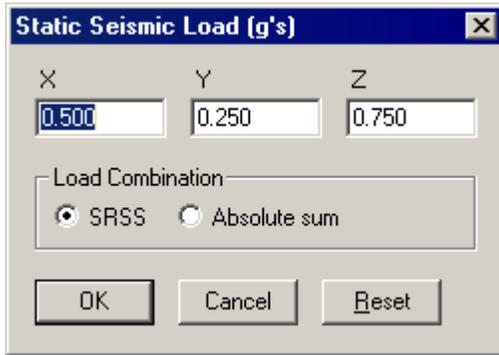
- First input Spectrums (under Misc menu)
- Then, select those spectrums under the menu Loads > Spectrum.
- Finally, you go back into menu Loads > Load Cases and check the box next to Response Spectrum for analysis.

As another example, the Cold Spring load cases do not appear in this dialog if you have not input a “Cut pipe” element in the layout window.

Further, multiple expansion and operating load cases appear here only when you set multiple thermal loads under Options > Analysis > Temperature in the Layout window.

Static Seismic Load

Use this load to apply static seismic loads (g-loads) to the model. CAEPIPE applies the input g-load to the piping system mass separately for X, Y and Z directions. The computed results (displacements, forces and moments) from each of the three (internal) load cases are combined using your choice of Absolute sum or Square root of sum of squares (SRSS). CAEPIPE performs only an algebraic sum when Piping code = None is selected.



Static seismic is an unsigned case and you will not see a sign (+ or -) in the results for this case.

Under menu Loads > Load cases, check the box next to Static Seismic (g's) to select this case for analysis. This load is treated as an Occasional load.

This g-load is applied only in the specified direction, i.e., a Z (g-load) of -0.189 is applied only in the -Z direction, not +Z.

A g-load input may affect the extracted natural frequencies when no piping code is selected (i.e., Piping code = None). See under "Dynamics > Cutoff Frequency" for how.

Acceleration in the vertical direction is generally taken to be 75% of the g-load in the horizontal direction.

The g-load can be calculated from any of the several sources available (ASCE A58.1, UBC, etc.). Guidelines from ANSI A58.1-1988, "Minimum Design Loads for Buildings and Other Structures" are used in the example below.

Piping is assumed to be equivalent to equipment thus giving a force coefficient, C_p as 0.3.

1. First, based on the map below (Contiguous 48 States+Alaska/Hawaii/Puerto Rico), identify the seismic zone and its corresponding coefficient (Z).

Layout Window Loads Menu

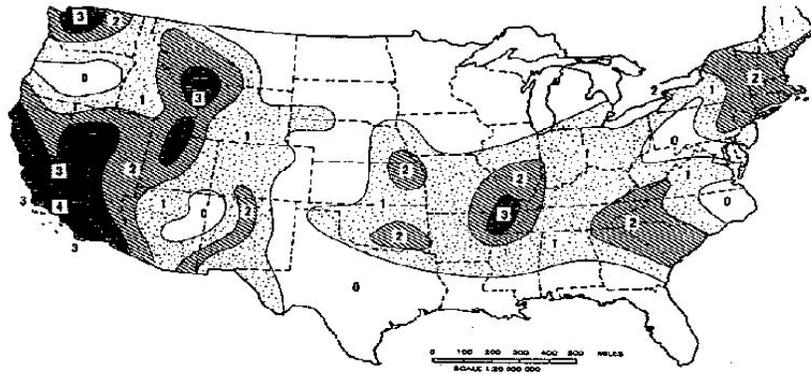


Fig. 14. Map for Seismic Zones—Contiguous 48 States (See Commentary Section 9.4)

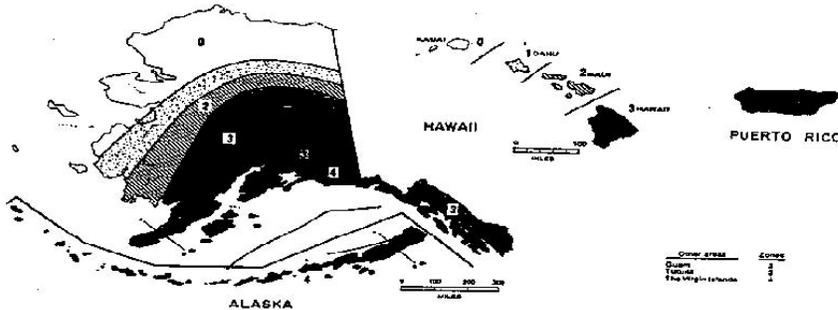


Fig. 15. Map for Seismic Zones—Hawaii, Alaska, and Puerto Rico—(See Commentary Section 9.4)

Seismic Zone	Coefficient Z
4	1.0000
3	0.7500
2	0.3750
1	0.1875
0	0.1250

Determine the Importance Factor, I as follows:

I = 1.5, For piping required in an emergency or piping with contents representing significant hazard to human life

I = 1.0, For other piping

Calculate the acceleration (g's) as $0.3 \times Z \times I$

Example:

For a power plant required to operate in an emergency, located in Anchorage, Alaska, calculate the design earthquake load coefficient for piping required to operate the plant.

1. From the map above, seismic zone is 4; from the table above, corresponding coefficient is 1.0.
2. For piping required to operate in an emergency, the Importance factor is 1.5. =
3. Calculate horizontal acceleration as $0.3 \times 1.0 \times 1.5 = 0.45g$, Vertical g-load (g) = $0.75 \times$ horizontal acceleration = $0.34g$.

The above accelerations need to be applied in the desired horizontal and vertical directions as long as the piping system is predominantly routed at grade level.

Static Seismic Analysis

CAEPIPE computes the inertia force (as mass * acceleration) for each direction and applies it as an occasional load. From the example above, assume we applied 0.45g in X, Y and Z directions. CAEPIPE applies an X acceleration (of 0.45g) first and solves the case. This procedure is then applied independently to the accelerations in Y and Z directions.

The above procedure results in three sets of solutions (displacements, element forces and support loads) to acceleration loads in X, Y and Z directions, which are typically combined in some manner. In CAEPIPE, two directional combination methods are available: SRSS (Square root of sum of squares) and ABS (Absolute).

In the SRSS method, displacements, element forces, and support loads from the three X, Y and Z accelerations are squared individually and added. The square roots of these respective sums are the displacement, element force and moment, and support load at the given node. In the ABS method, all absolute values of displacements, element forces and moments, and support loads are added to get the total displacements, element forces and support loads.

The occasional stresses (S_o) are added to sustained stresses (S_L) and shown under Occasional stresses ($S_L + S_o$).

CAEPIPE does not include the weight of the piping system in Seismic load case. A few reasons being:

1. The weight is already included in Sustained load case, and
2. Piping codes such as B31.3 specifically mention that MA and MB calculations (moments due to sustained and occasional loads respectively) be done separately before combining them to calculate S_o (Occasional stress).

If you can, avoid nonlinearities such as gaps and friction when you model seismic loads. As it is, Static Seismic is an approximation of Response Spectrum, which is an approximation of Time History analysis! So, the result you get from a static seismic analysis is a gross approximation.

CAEPIPE adds the support loads from a seismic case to those from Sustained and Operating to produce the combination loads. Look under the Support load summary to get the different combination loads at each support.

To evaluate different directional seismic analyses:

In linear analysis (i.e., no gaps nor friction), the results from a seismic load in +X and -X directions should be the same. So, the combination ranges can be seen by studying support load summary (e.g., Sustained+Seismic, Sustained-Seismic).

In nonlinear analysis, however, the results from a seismic load in +X may NOT be the same as those from -X. In such a case, you will have to make two runs, i.e., analyze the original (say, model1.mod) with +X g-load; copy the model to model2.mod, specify a -X g-load and compute results. Now you have two sets of results, one from each model. Compare the two carefully to identify the range (least to the most).

Or, for further post processing, export the results from each model to a CSV file via the Print > Print to File command (select file type as: CSV). Import each file into a spreadsheet (e.g., MS-Excel) and combine the results of interest manually.

Layout Window Loads Menu

In summary, you can set up as many runs as required for the g-loads (+ or -), depending on whether your model is linear or non-linear, and export results for those runs that you cannot combine inside CAEPIPE.

As another example, for a nonlinear model with g-loads only in the X-Y plane, you can analyze four runs (or four identical models with different seismic loads) thus:

- Run 1. Specify +X/+Y g-load
- Run 2. Specify -X/-Y g-load
- Run 3. Specify -X/+Y g-load
- Run 4. Specify +X/-Y g-load

Examine the seismic and combination forces and moments from all runs, or in a spreadsheet (using the .CSV files CAEPIPE exports for each model) to determine the most conservative of results (and range) for further interpretation.

As another example, in a linear model with g-loads only in the X-Y plane, the two runs

- Run 1. Specify +X/-Y g-load
- Run 2. Specify -X/+Y g-load

are the same. So, either case (run) alone is adequate.

Or, another example, for a nonlinear model with g-loads in all three global directions, you could set up eight runs/models:

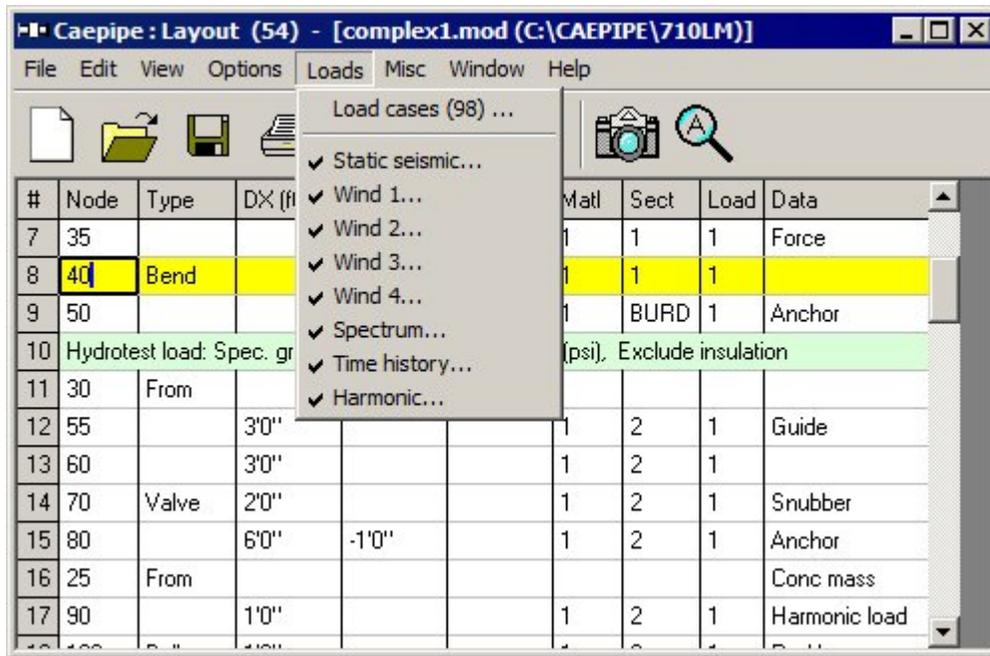
- Run 1. Specify +X/+Y/+Z g-load
- Run 2. Specify +X/+Y/-Z g-load
- Run 3. Specify +X/-Y/+Z g-load
- Run 4. Specify +X/-Y/-Z g-load
- Run 5. Specify -X/+Y/+Z g-load
- Run 6. Specify -X/+Y/-Z g-load
- Run 7. Specify -X/-Y/+Z g-load
- Run 8. Specify -X/-Y/-Z g-load

Each of these runs may yield different results. So, a careful evaluation of all eight runs is necessary to identify the most conservative solution and/or determine a range.

Layout Window Loads Menu

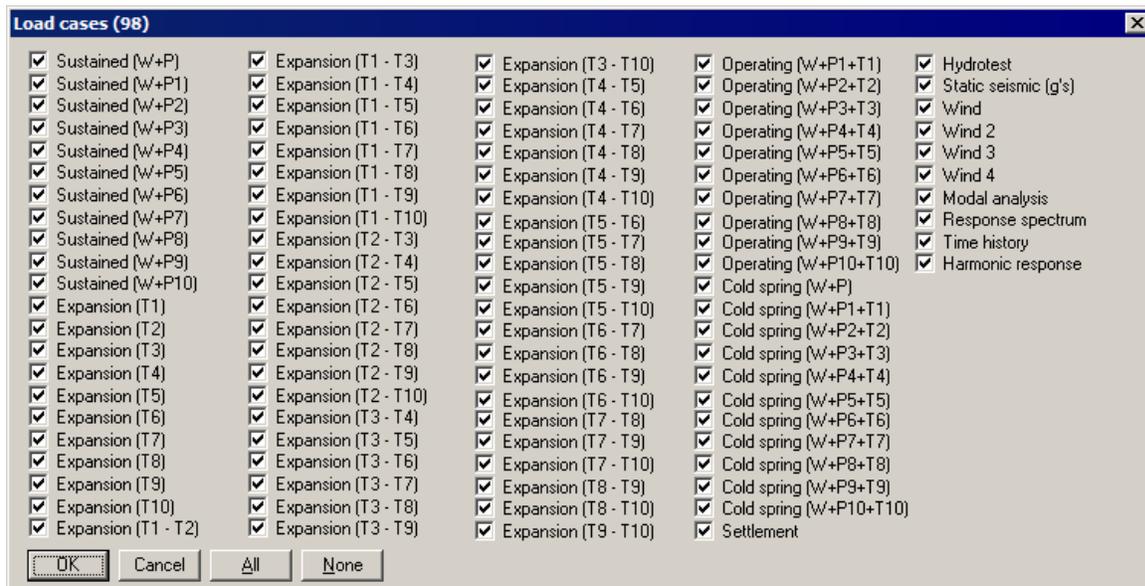
Wind Load

You can enter up to four separate wind load profiles named Wind, Wind 2, Wind 3 and Wind 4.



Load Combinations

With 10 thermal and four wind loads, you can analyze a total of up to 95+ load combinations (including 55 thermal load cases).



You can apply a wind load to the whole or parts of the model. CAEPIPE applies it as a lumped (concentrated force) load at the nodes (i.e., it is not a distributed load along the element). A wind profile is required for the region in which you (plan to) have the system installed.

Wind Load

Shape factor: 0.60

Direction:
 X comp:
 Y comp:
 Z comp: 1

Elevation (feet)	Velocity (mph)
0	0
50	45
100	75

Pressure vs Elevation
 Velocity vs Elevation

Units:
 Elevation: (feet)
 Pressure: (psf)
 Velocity: (mph)

OK Cancel Delete

Shape factor

Input a shape factor value here. CAEPIPE uses a constant value for shape factor. See ANSI A58.1, “Minimum Design loads for Buildings and other Structures” for more information.

For Pressure versus Elevation, one needs to multiply (or build) the shape factor into the input value for pressure. CAEPIPE does not use the shape factor when user selects Pressure versus Elevation.

Direction

Input the direction of the wind using the direction cosines (examples: for wind in Z direction, Z comp = 1; for wind in 45° X-Y plane, X comp=1, Y comp=1, Z comp=0). See section on Direction in Reference.

Wind Profile

You need to input values for wind velocity or pressure at different elevations. CAEPIPE interpolates the wind load on an element from these values based on its elevation.

Units

You can specify any combination of units for elevation, pressure and velocity.

Wind Pressure

The wind pressure in (lb/ft²) is calculated as:

$$\text{Wind pressure} = 0.0032 \times \text{Wind velocity}^2 \times \text{Shape factor}$$

where Wind velocity is in (mph).

To exclude an element(s) from wind loading, the Load definition dialog for that element should not have Wind load checked. See dialog below that shows “Wind load” unchecked.

The screenshot shows a dialog box titled "Load # 1". At the top, there is a "Load name" field containing the number "1" and a checked checkbox labeled "Wind load". Below this, there are three rows of input fields for temperature and pressure. The first row has "Temperature 1" set to "600" (F) and "Pressure 1" set to "200" (psi). The second and third rows have empty fields for "Temperature 2", "Pressure 2", "Temperature 3", and "Pressure 3". At the bottom, there is a "Spec. gravity" field set to "0.8" and an "Add. weight" field which is empty. Below these fields are "OK" and "Cancel" buttons, and a note that reads "Specific gravity is with respect to water".

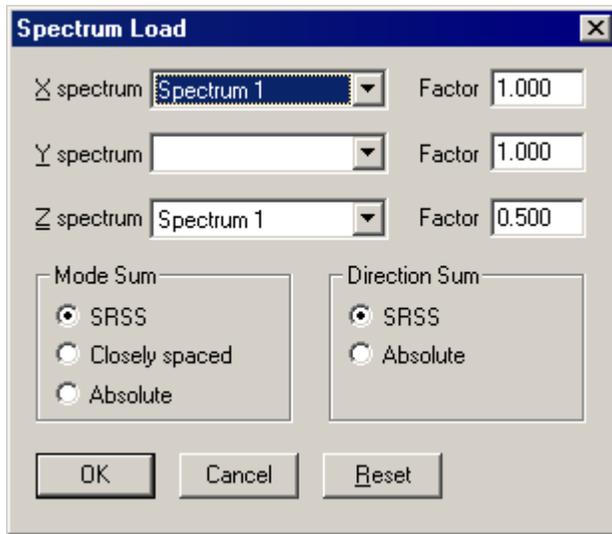
Spectrum Load

Shock waves (due to a seismic event), sometimes hundreds of feet long, travel through the soil causing ground motion at all supports in a piping system. Every support connected to the ground in a piping system will see the same excitation (hence uniform). These excitations are specified using a response spectrum, which is a table of maximum response versus natural frequency (with damping) for single degree-of-freedom systems.

A "Spectrum" load specified here will be applied simultaneously at all supports during a "Uniform response spectrum" analysis, following which, CAEPIPE will compute the modal and directional responses (to this uniform excitation), which are further combined in a manner you specify.

The spectrum load can only be input after at least one spectrum is defined (under menu Misc > Spectrums).

Layout Window Loads Menu



X, Y and Z spectrums

Select a spectrum from the drop-down combo box, which should have been input in the spectrum table for each direction.

Factor

The multiplying (scale) factor for the spectrum is input here. The same spectrum may be multiplied by different (Scale) factors to apply spectrum loads for different dynamic events.

Mode Sum

Pick one of three choices, SRSS (square root of sum of squares), Closely spaced or Absolute. See section on Dynamic Analysis in Reference for more information.

Direction Sum

Pick one of two choices, SRSS (square root of sum of squares) or Absolute. See section on Dynamic Analysis in Reference for more information.

See topic on Dynamic Analysis for more information.

Time History Load

A phenomenon that causes loads to vary with time can be applied in CAEPIPE for time history analysis. You may hand-calculate these values or use a simulation program (such as a transient fluid flow program for fluid hammer analysis) to get the variation of forces or moments in time at different points in the piping system. These loads are then input into CAEPIPE as time functions, which are later applied at the corresponding nodes of the piping system as “Time Varying Loads.”

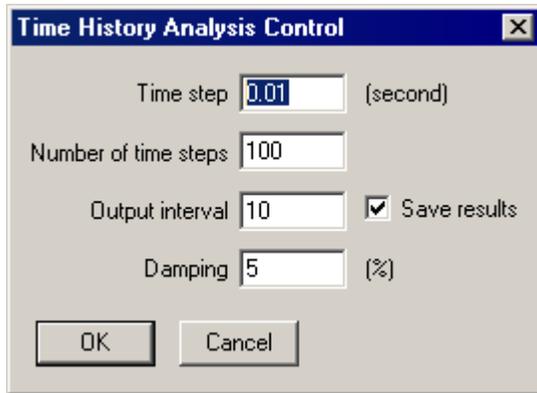
Time functions (input under menu Misc > Time Functions) are a series of nondimensional values versus time, which describe the variation of the forcing function with time. The actual value of the time function at any time is found by linear interpolation between time points.

When you input a “Time Varying Load” at a node, you may apply a scale factor, if necessary, along with the direction of the force or moment.

Layout Window Loads Menu

CAEPIPE will then apply these loads to compute the response of the piping system by performing a time history analysis.

Various parameters for time history analysis are specified in the Time History Analysis Control dialog under menu Loads > Time History.



Time step

The time step (time interval) at which the analysis is performed should be typically no more than 10% (smaller the better) of the period of the highest frequency of interest, i.e., higher the frequency, smaller the time step, e.g., for a 33 Hz maximum frequency, the time step would be less than 0.003 seconds.

Number of time steps

The time history response is calculated for a total time (seconds) of Time step \times Number of time steps. This is how long you want to study system response. The total time may exceed the range of available data in the time function. The time function is only a forcing function. But, CAEPIPE can compute system response to it well after the forcing function ceases. For example, the effect of a heavy steam hammer could linger on for a minute while your forcing function data could span only 8s (seconds).

Output interval

The output interval is a multiple of time step at which you want CAEPIPE to save (and later display) results. For example, assuming a time step of 0.001s, if you wanted to see results at every 100th time step (0.1s, 0.2s, 0.3s, and so on), enter 100 for output interval. You need to check the “Save Results” checkbox to see these time varying results which are saved in the file *modelname.rtb*. If the “Save Results” checkbox is not checked, only the enveloped results are available but not their variation in time. The .RTH files may be big for large models with many time steps (so ensure that you have adequate storage and permission settings).

Damping

Express the damping factor as a percentage (not as a fraction). Enter 5, not 0.05, for 5% damping.

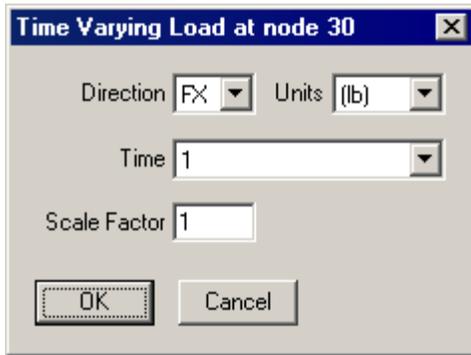
Example:

Input: Time step = 0.01s, Number of time steps: 1000, Output interval: 10, Damping: 5%, Save Results checkbox Checked.

Layout Window Loads Menu

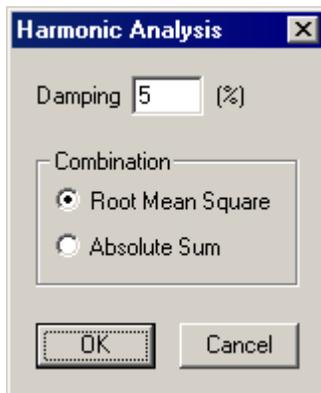
CAEPIPE calculates the solutions at every 0.01s up to 10s. The results are saved every 0.1s in the .rth file. Note however, that CAEPIPE calculates the enveloped (maximum) results at every time step interval and not the output interval. The output interval is strictly for display purpose in the results and does not affect analysis.

The Time Varying Load dialog is shown below:



The Time Varying Load at a node is specified by its direction, units, the associated time function and a scale factor. This scale factor is a scalar value, which when multiplied by the nondimensional time function will give the actual magnitude of the forcing function in the unit selected in the above dialog.

Harmonic Load



Harmonic loads can be loads from any sinusoidal loading, such as from rotating equipment or reciprocating pumps on a line. The magnitude of the loading should be determined before analysis. If only one compressor is on a line, then only one harmonic load is input. If more than one load is acting on the same line, then the phase (angle) or the separation in timing of application of each harmonic load becomes important (consider for example that the two loads may be equal and opposite thus canceling out any dynamic imbalance, or the two loads can be in the same direction, say +X and separated by 30° phase angle). So, the situation must be carefully analyzed before imposing these loads.

Harmonic analyses of linear structures are generally performed to determine the steady-state response to long-duration loads, which vary sinusoidally (harmonically) with time, thus enabling you to verify whether your designs will successfully overcome resonance, fatigue, and other harmful effects of harmonic vibrations. In a harmonic analysis, all long-duration loads and the structural response vary sinusoidally at the same frequency.

Layout Window Loads Menu

Various parameters for harmonic analysis are specified in this dialog shown above.

Damping

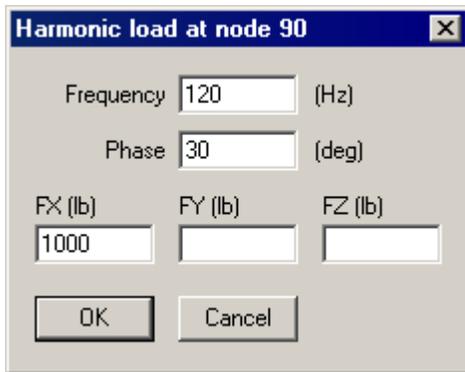
CAEPIPE applies this factor to all the modes. Enter value of critical damping as a percent, not as a fraction (Example: For 5%, enter 5, not 0.05).

Combination

CAEPIPE combines modal responses using one of two methods: Root mean square (RMS) or Absolute sum (which may be too conservative). Pick one.

After you input the data here, then apply Harmonic loads at nodes of interest. Then, select Harmonic under the menu Loads > Load cases for analysis.

The Harmonic Load dialog is shown below:



The screenshot shows a dialog box titled "Harmonic load at node 90". It contains the following fields and buttons:

- Frequency: 120 (Hz)
- Phase: 30 (deg)
- FX (lb): 1000
- FY (lb): (empty)
- FZ (lb): (empty)
- Buttons: OK, Cancel

The harmonic load can be imposed as a Force (FX/FY/FZ) at a specified frequency and phase angle. You may be able to get more information on the harmonic loading (mass, rpm, eccentricity, etc.) from the manufacturer of the equipment.

Since a harmonic load response is unsigned, results do not have signs (+ or -).

Layout Window Misc Menu

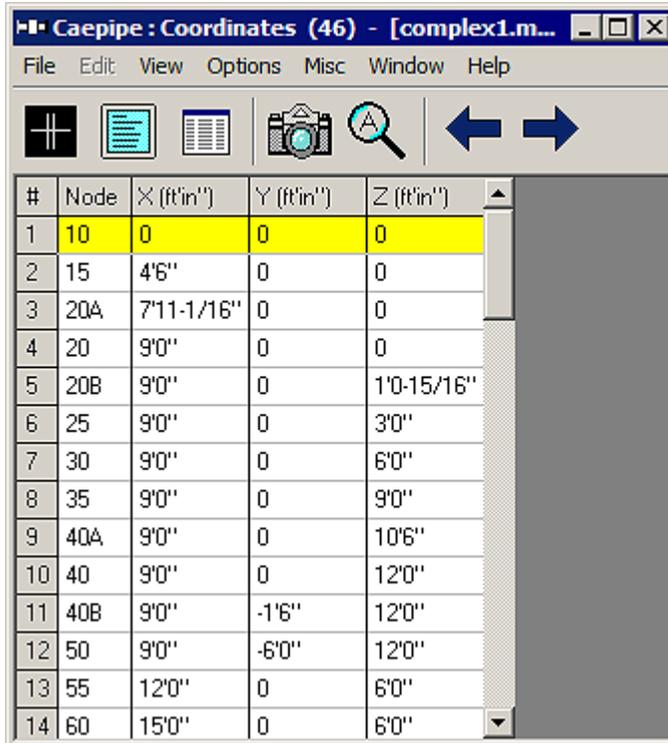
Misc Menu

A few utilities such as Check bends and connections, sundry items such as opening the list screens for materials, sections, loads, pumps, compressors and turbines, and commands to enter data for spectrums, time functions, etc., are among the commands on this menu. When Piping code is set to None under menu Options > Analysis, all items from Pumps to Soils (inclusive) are disabled.

Misc	Window	Help
Coordinates		Ctrl+Shift+C
Element types...		Ctrl+Shift+T
Data types...		Ctrl+Shift+D
Check Bends		
Check Connections		
Materials		Ctrl+Shift+M
Sections		Ctrl+Shift+S
Loads		Ctrl+Shift+L
Beam Materials		
Beam Sections		
Beam Loads		
Pumps		
Compressors		
Turbines		
Spectrums		
Force spectrums		
Time functions		
Soils		
User Allowables		

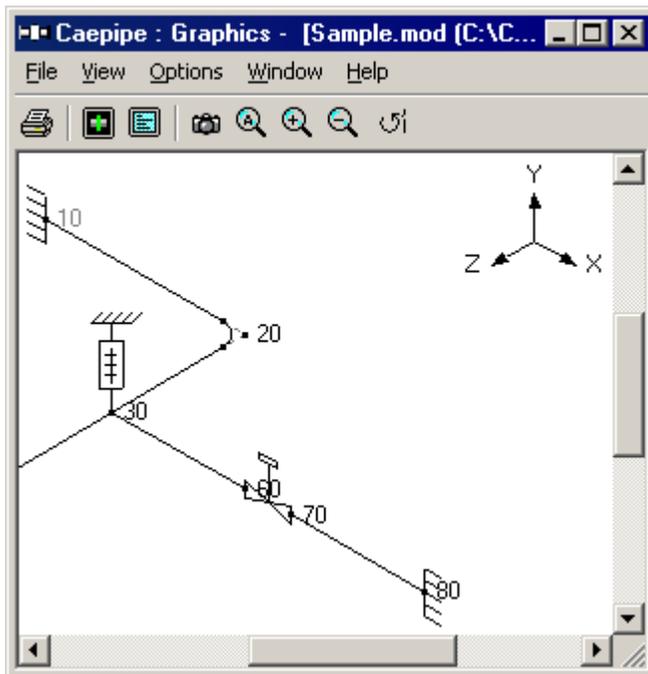
Coordinates (Ctrl+Shft+C)

This command lists the coordinates of all the nodes (including internally generated nodes such as A, B nodes) in the model.

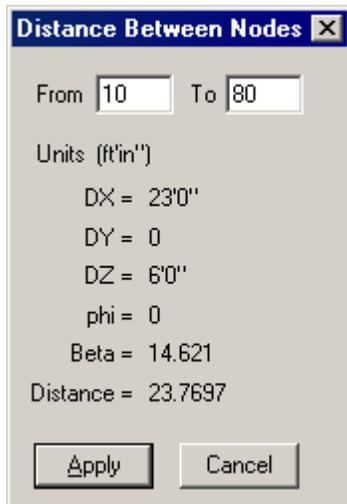


The screenshot shows the 'Caepipe : Coordinates (46)' window. The table below lists the coordinates for 14 nodes. Node 10 is highlighted in yellow.

#	Node	X (ft'in')	Y (ft'in')	Z (ft'in')
1	10	0	0	0
2	15	4'6"	0	0
3	20A	7'11-1/16"	0	0
4	20	9'0"	0	0
5	20B	9'0"	0	1'0-15/16"
6	25	9'0"	0	3'0"
7	30	9'0"	0	6'0"
8	35	9'0"	0	9'0"
9	40A	9'0"	0	10'6"
10	40	9'0"	0	12'0"
11	40B	9'0"	-1'6"	12'0"
12	50	9'0"	-6'0"	12'0"
13	55	12'0"	0	6'0"
14	60	15'0"	0	6'0"



Straight line distance between two nodes can be found by using the menu View > Distance command (Ctrl+D) in the coordinates window.



Also shown are the angles the straight line (between the two nodes) makes in the X-Y plane (phi) and from the X-Y plane (beta).

Element types (Ctrl+Shft+T) / Data types (Ctrl+Shft+D)

You can open these dialogs that contain the different elements or data items you can input at a node. You can also open the same dialogs by clicking on the header row in the Layout window or right clicking in the Type or Data column on an empty row.

More information about these commands is available under Layout Window in Reference.

Check Bends

CAEPIPE can identify incorrectly and incompletely modeled bends. If you have any, you must correct them before analysis.

Case 1 (Incorrect modeling)

An “Invalid Bend” occurs if the bend geometry is input incorrectly in the layout. For example, for 90° bends, the bend radius should be less than or equal to the shorter of the lengths (from the previous node to the bend node or from the bend node to the next node).

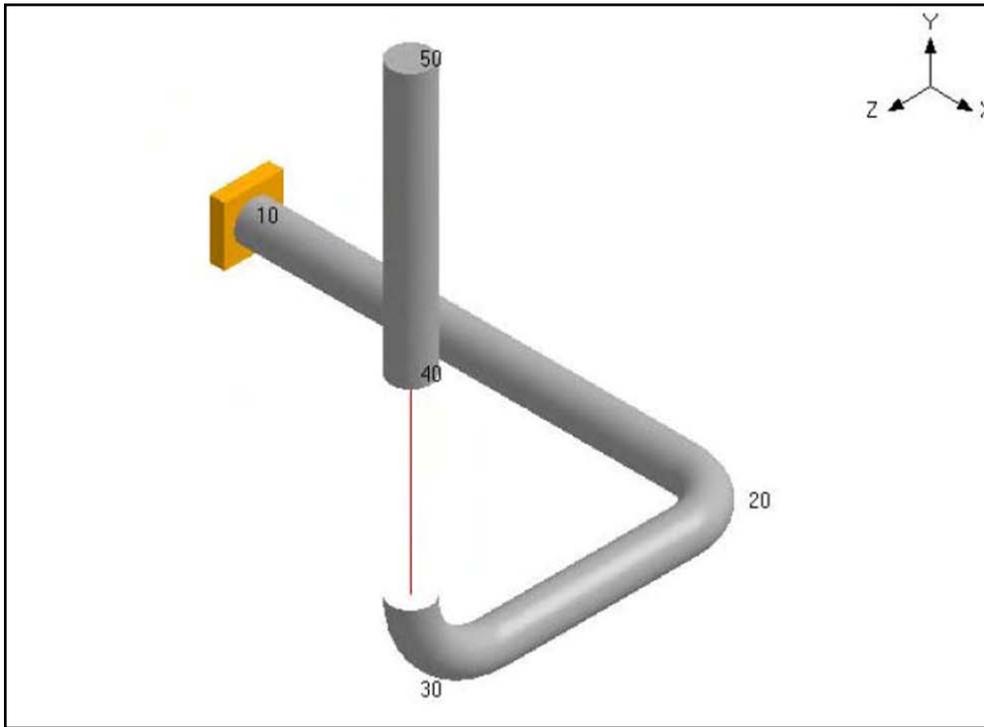
Case 2 (Incomplete modeling)

A bend needs to be given a change in direction from the previous direction on the row below it. If this change is either not specified or incorrectly specified, CAEPIPE flags this bend as an invalid bend. An example follows.

Layout Window Misc Menu

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Example of an invalid bend								
2	10	From							Anchor
3	20	Bend	9'0"			1	8	1	
4	30	Bend			6'0"	1	8	1	
5	40	Bend		5'0"		1	8	1	
6	50			5'0"		1	8	1	
7									

When you select the Check bends command under the Misc menu, you get the following message for this model.

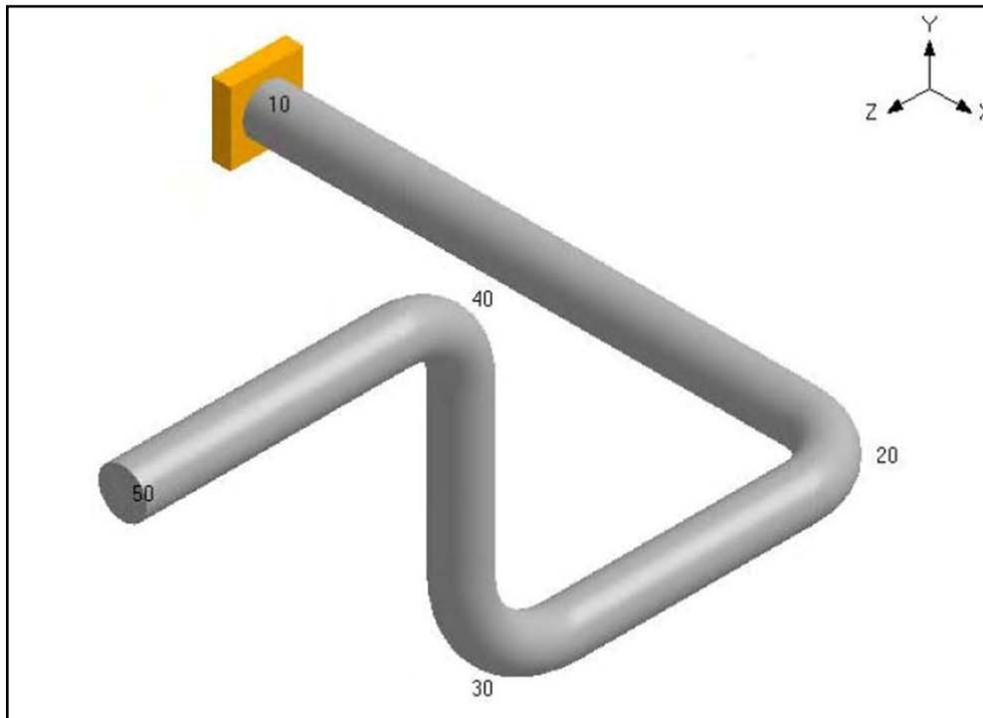


By studying the image and the Layout window above, you can see that the bend at node 20 turns from X direction into Z direction, and the bend at node 30 turns from Z direction into Y direction. But, the bend at node 40 incorrectly turns from Y direction into Y direction. To correct this, this bend needs to turn into any direction other than Y. The following screen shots show the correction. The bend at node 40 now turns into Z direction.

Layout Window Misc Menu

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Example of a valid bend								
2	10	From							Anchor
3	20	Bend	9'0"			1	8	1	
4	30	Bend			6'0"	1	8	1	
5	40	Bend		5'0"		1	8	1	
6	50				5'0"	1	8	1	
7									

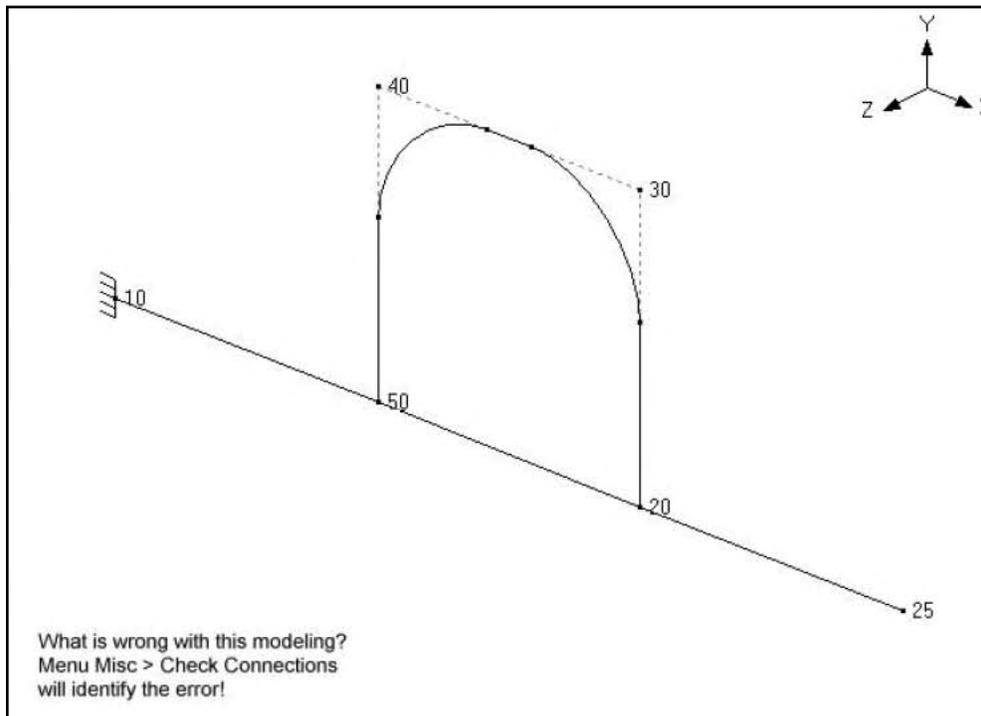
When you run the same Check bends command again, you get the following message.



Check Connections

While not foolproof, this command checks for missing connections between pipes. By studying the following two figures, can you identify what is wrong with the modeling shown below? Check connections command will.

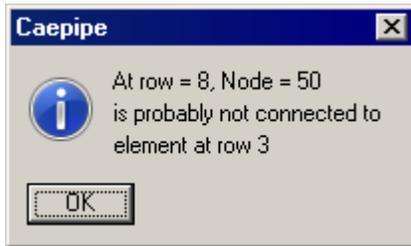
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data	
1	Title = Discontinuity in the model									
2	10	From							Anchor	
3	20		6'0"			A53	10	HI		
4	25		3'0"			A53	10	HI		
5	20	From								
6	30	Bend		3'0"		A53	10	HI		
7	40	Bend	-3'0"			A53	10	HI		
8	50			-3'0"		A53	10	HI		
9		What is wrong with this modeling?								



By studying the previous Layout window, you will notice that even though node 50 appears to be on the header (node 10 to node 25), it actually is not. Node 10 connects directly to node 20 which further connects to node 25. The loop begins from node 20 and goes up to node 50. But 50 is not defined on the header. So, there is no connection between the loop and the header!

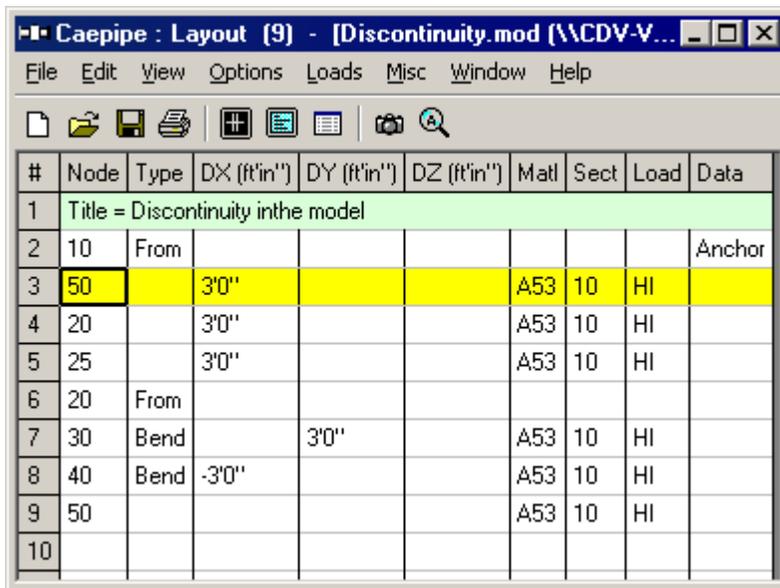
Layout Window Misc Menu

When you run the Check connections command, you get the following message.



So, you might want to run this command on complicated models with loops just to ensure that all such connections are properly modeled.

Corrected modeling is shown next. Notice that node 50 is now defined as part of the header from node 10 to node 25.



The screenshot shows the "Caepipe : Layout (9) - [Discontinuity.mod (\CDV-V...)]" window. It has a menu bar (File, Edit, View, Options, Loads, Misc, Window, Help) and a toolbar. Below is a table with columns: #, Node, Type, DX (ft'in'), DY (ft'in'), DZ (ft'in'), Matl, Sect, Load, Data.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Discontinuity in the model								
2	10	From							Anchor
3	50		3'0"			A53	10	HI	
4	20		3'0"			A53	10	HI	
5	25		3'0"			A53	10	HI	
6	20	From							
7	30	Bend		3'0"		A53	10	HI	
8	40	Bend	-3'0"			A53	10	HI	
9	50					A53	10	HI	
10									

Materials, Sections and Loads

Please see the respective sections with the same titles in Reference.

Beam Materials, Beam Sections and Beam Loads

These three can be found under Beam in Reference.

Pumps, Compressors and Turbines

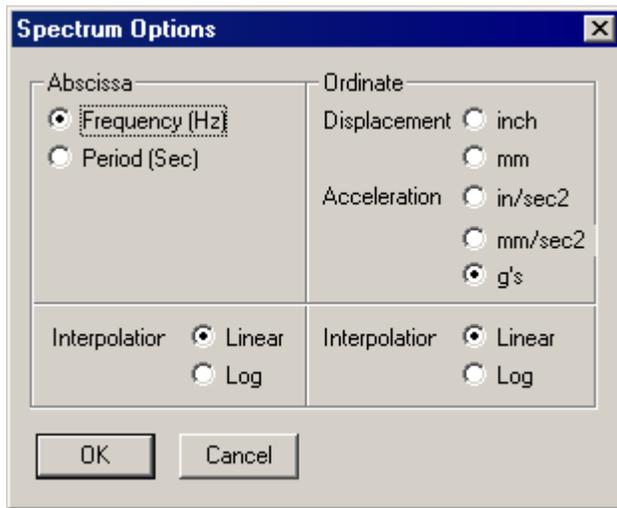
Please see the respective section with the same title in Reference.

Spectrums

A (uniform response) spectrum is a table of maximum response versus natural frequency for a specific excitation in single degree-of-freedom systems. You can input spectrums in three ways:

1. Input spectrums directly into the model.
2. Create a spectrum library and load spectrums from it.
3. Input spectrums from a text file.

When you use the first two methods, you may use menu Options > Spectrum command to set the different units for the X- and the Y-axes, and also choose the interpolation method.



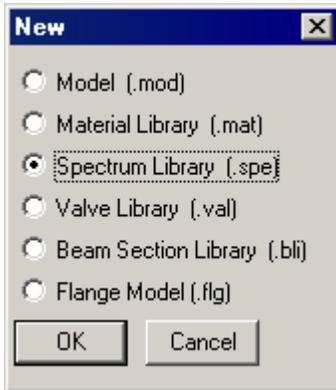
1. Input spectrums directly into the model

Select Spectrums from the Misc menu. You are shown the List window for spectrums. Start typing pairs of values into it. The frequencies or periods do not have to be in any order; CAEPIPE will sort them later. You can input as many pairs of values as required.

#	Name	#	Frequency (Hz)	Acceleration (g's)
1	X spectrum	1	1	0.00258799
2	Y spectrum	2	50	0.00258799
3	Z spectrum	3	62	0.00258799
4		4	75	0.00258799
		5	100	0.00258799
		6	139	0.00258799
		7	175	0.00258799

2. Create a spectrum library and load spectrums from it

From the Main, Layout or Results window, select menu File > New.

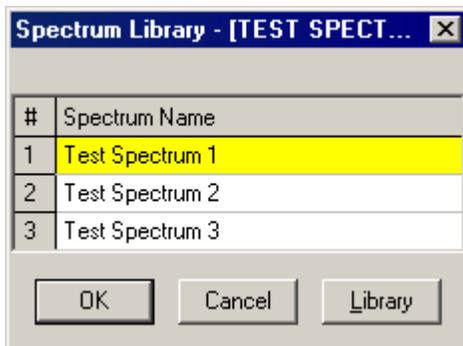


Select Spectrum library. The spectrum library List window is shown. Start typing pairs of values into it.

#	Name	#	Frequency (Hz)	Acceleration (g's)
1	Test Spectrum 1	1	1	190
2	Test Spectrum 2	2	2	220
3	Test Spectrum 3	3	3	250
4		4	4	1932
		5	5	1000
		6	6	1800
		7		

The frequencies or periods you type can be in any order. CAEPIPE will sort them later. You can input as many pairs of values as required. Be sure to save the file (it will be saved to a filename you specify, with a .spe extension) using the Save command in the File menu.

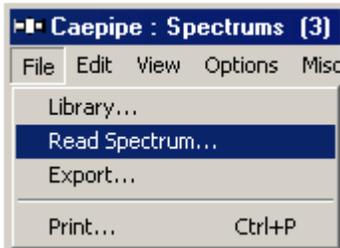
Now, open the CAEPIPE model that needs these spectrums. Open the saved library (menu File > Library) and input spectrums into the model from the shown list.



3. Input spectrums from a text file

This method involves creating a separate text file (in the format shown below) for each spectrum you want to input. The steps are given below.

- For each spectrum, on the first line, type the Name of the spectrum (up to 31 chars). On the second line, type units for Abscissa (X-axis) and Ordinate (Y-axis) axes, separated by a space.
- After these two lines, on each line, you can input a value pair. You can input as many of these value pair lines as required.
- Each spectrum should be saved to a separate text file. More than one spectrum should not be input in one text file as it cannot be read into CAEPIPE.
- Now, open the CAEPIPE model that needs these spectrums. Select Read Spectrum from the File menu, read all text files you created one after the other.



The format of a spectrum text file is shown below. The spectrum that is read appears on the row where the yellow highlight is placed (under the Name column). You can use the Edit menu commands to insert an empty row or delete an existing spectrum. Ensure that no two spectrums share the same Name. CAEPIPE issues a warning should such occur.

The spectrum text file should be in the following format:

```
Name (up to 31 characters
Abscissa units      Ordinate units
Abscissa value1    Ordinate value1
Abscissa value2    Ordinate value2
.                  .
.                  .
.                  .
```

Where

```
Abscissa units      0 = Frequency (Hz)
                   1 = Period (Sec)
```

```
Ordinate units      0 = Displacement (inch)
                   1 = Displacement (mm)
                   2 = Acceleration (in/sec2)
                   3 = Acceleration (mm/sec2)
                   4 = Acceleration (g's)
```

Example file

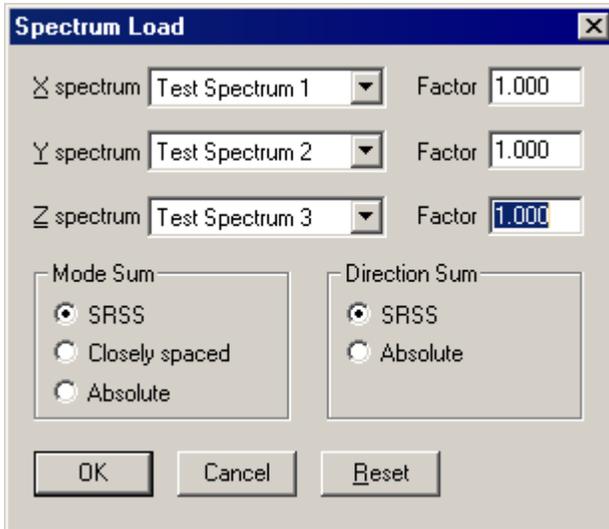
Test Spectrum

```
0 2
1 190
2 220
3 250
4 1932
5 1000
6 1800
. .
. .
```

Analysis note:

While analyzing the response spectrum case, when a mode (frequency or period) falls outside the spectrum table, CAEPIPE issues a warning and uses the value corresponding to the closest frequency or period in the spectrum table (for example, CAEPIPE uses the value corresponding to the lowest frequency if the calculated frequency is lower than the lowest in the spectrum table or uses the value corresponding to the highest frequency if the calculated frequency is higher than the highest in the spectrum table).

Once you are done inputting the different spectrums using any one of the three methods, you need to input the Spectrum load itself under the menu Loads > Spectrum in the Layout window. Details on inputting the Spectrum Load are provided earlier under the Loads menu.



Force Spectrums

See topic on Force Spectrum in Reference on page 248.

Time functions

Time functions are nondimensional tables (i.e., a series of time versus value pairs) which describe the variation of the forcing function with time. Usually, a transient fluid flow analysis program computes forces as a function of time at all changes in directions (bends/tees) and other points of interest. These forces/moments result from a transient event such as a fluid hammer. Separate force-time histories are then input as time functions and applied as Time Varying Loads within CAEPIPE at the corresponding nodes of interest in the piping model.

In CAEPIPE, the time function you define can have any interval between two time values. You can make that table as fine as you want it to be.

You must have a zero entry for the Value next to the first Time input. Time history analysis begins at time t=0.0. You can input as many time-value pairs as required.

There are two methods for inputting time functions.

1. Input time functions directly into the model.
2. Create a text file for each function and read it into CAEPIPE.

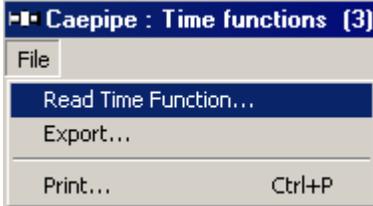
1. Input time functions directly into the model

Click on menu Misc > Time functions to type in time functions. Time is measured in seconds. Value is nondimensional. You can assign units to these Values when you input a Time varying load at a node. For now, simply start typing the time-value pairs. Next, input parameters for the Time History Analysis Control dialog, and then input Time Varying Loads at nodes of interest.

#	Name	#	Time (Sec)	Value
1	Time Function 1	1	0	0
2	Time Function 2	2	0.1	100
3	Time Function 3	3	0.2	0.025
4		4	0.3	26.65
		5	0.4	112
		6	0.5	806
		7	0.6	7984
		8	0.7	1882
		9	0.8	2531
		10	0.9	2480
		11	1	2109
		12		

2. Create a text file for each function and read it into CAEPIPE

Read these text files into CAEPIPE using the menu File > Read Time Function command. For each function, you need to type a Name (up to 31 characters) on the first line, type time followed by a Value (separated by a space) on the following line. You can input as many time-value pairs (one pair on each line) as required.



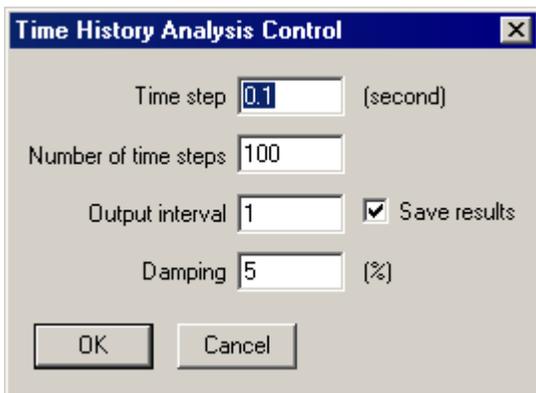
The format of the time function text file is shown below. The time function that is read from a file appears on the row where the yellow highlight is placed (under the Name column). You can use the Edit menu commands to insert an empty row or delete an existing time function. Ensure that no two time functions share the same Name. CAEPIPE issues a warning should such occur.

```

Time Function
0.0 0
0.1 0
0.2 0.025
0.3 26.65
0.4 112
0.5 806
0.6 7984
0.7 1882
0.8 2531
0.9 2480
1 2109
:
:
:
:

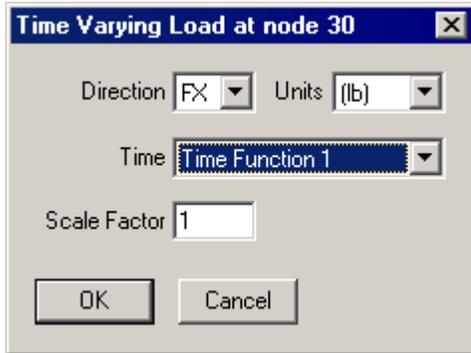
```

Once you are done entering time functions, specify the parameters in the Time History Analysis Control dialog under menu Loads > Time History, details of which are provided earlier under the Loads menu topic.



Layout Window Misc Menu

Then, specify Time Varying Loads at applicable nodes in the model. The following figure shows the time function as a time-varying force applied in the Y direction (FY) at node 100. Under Units, you can specify one of several depending on whether you are applying the Values in the time function as a force (FX, FY, FZ) or a moment (MX, MY, MZ). For more details, please see discussion under menu Loads > Time History.

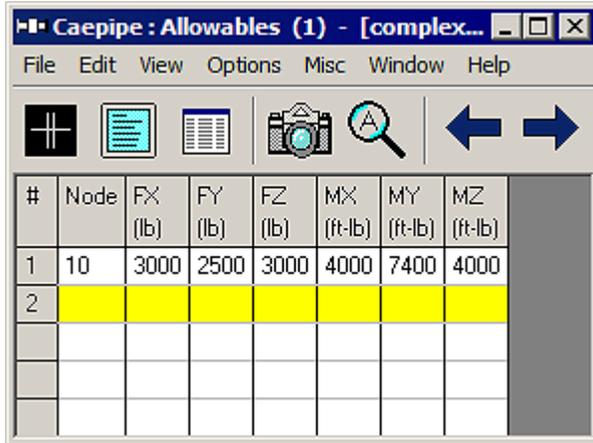


Soils

See Buried Piping Example in the topic on Buried Piping in Reference.

User-Defined Allowables

CAEPIPE allows you to define "User Allowables" for Anchors, Generic Supports, Nozzles and Restraints through Misc > User Allowables.



#	Node	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
1	10	3000	2500	3000	4000	7400	4000
2							

The allowables thus defined are compared against calculated loads and printed in Support Load Summary outputs.

Layout Window Misc Menu

Allowables
✕

Node

FX (lb) FY (lb) FZ (lb)

MX (ft-lb) MY (ft-lb) MZ (ft-lb)

If the calculated loads exceed the input allowables, they are highlighted in red.

Caepipe : Support load summary for anchor at node 10 - [Sample-usrAllowable.r...
▢ □ ✕

File Results View Options Window Help

🖨️
⊕
📄
📄
📷 🔍
☰
← →
📄 ↓
← →

Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)	Displacements (global)		
							X (inch)	Y (inch)	Z (inch)
Sustained	-13	-385	26	-365	-171	-1118	0.000	0.000	0.000
Operating1	-28550	1474	-13762	-6909	57834	16248	0.000	0.000	0.000
Maximum	-13	1474	26	-365	57834	16248	0.000	0.000	0.000
Minimum	-28550	-385	-13762	-6909	-171	-1118	0.000	0.000	0.000
Allowables	3000	2500	3000	4000	7400	4000	0.000	0.000	0.000

Layout Window Window Menu

Window Menu

For a single model file, CAEPIPE can keep up to four open windows simultaneously for your enhanced understanding of the presented information.

The windows are:

- Layout
- Graphics
- List, and
- Results

From any window, you can move focus to any other window (such as Graphics or List) using the F2 and F3 hotkeys. F2 will move focus between text (Layout, List or Results) and Graphics windows, and F3 between open text windows (between Results and Layout or Results and List).

Users who have a smaller monitor and work with maximized windows for input, list, graphics and results, will see more the advantage of these hotkeys (F2 and F3). In such a setting, one key press (F2 or F3) will quickly move the focus to another window (without having to either minimize the one where the focus is or switch to another window through the taskbar).

From the Layout window:

Window	
Graphics	F2
List	F3

From the List window:

Window	Help
Graphics	F2
Layout	F3

From the Graphics window:

Window	Help	Window	
Layout	F2	List	F2

From the Results window:

Window	Help
Graphics	F2
List	F3
Layout	

Layout Window Window Menu

Note: In the above Window menu from the Results window, if you moved focus to the Layout window, then the hotkey F3 would be assigned to Layout (not List as shown). So, the Window menu would look like:



Graphics Window File Menu

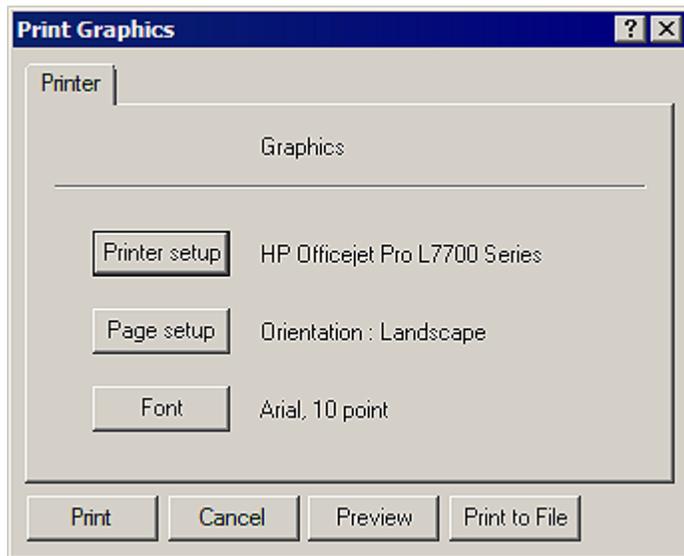
File Menu

This menu has commands for printing and copying graphics images to the clipboard. You can also set the drawing size for a printer/plotter/DXF file and type in a new title for the image (different from the Title entered on row 1 of the Layout window).

File	
Print...	Ctrl+P
Copy	Ctrl+C
Drawing Size...	
Plot Title...	

Print (Ctrl+P)

The Print command here opens the Print Graphics dialog which allows you to select a printer and customize other print settings.



Printer

On the Printer tab, you can setup the printer and the graphics print page as well as select the font for printing graphics.

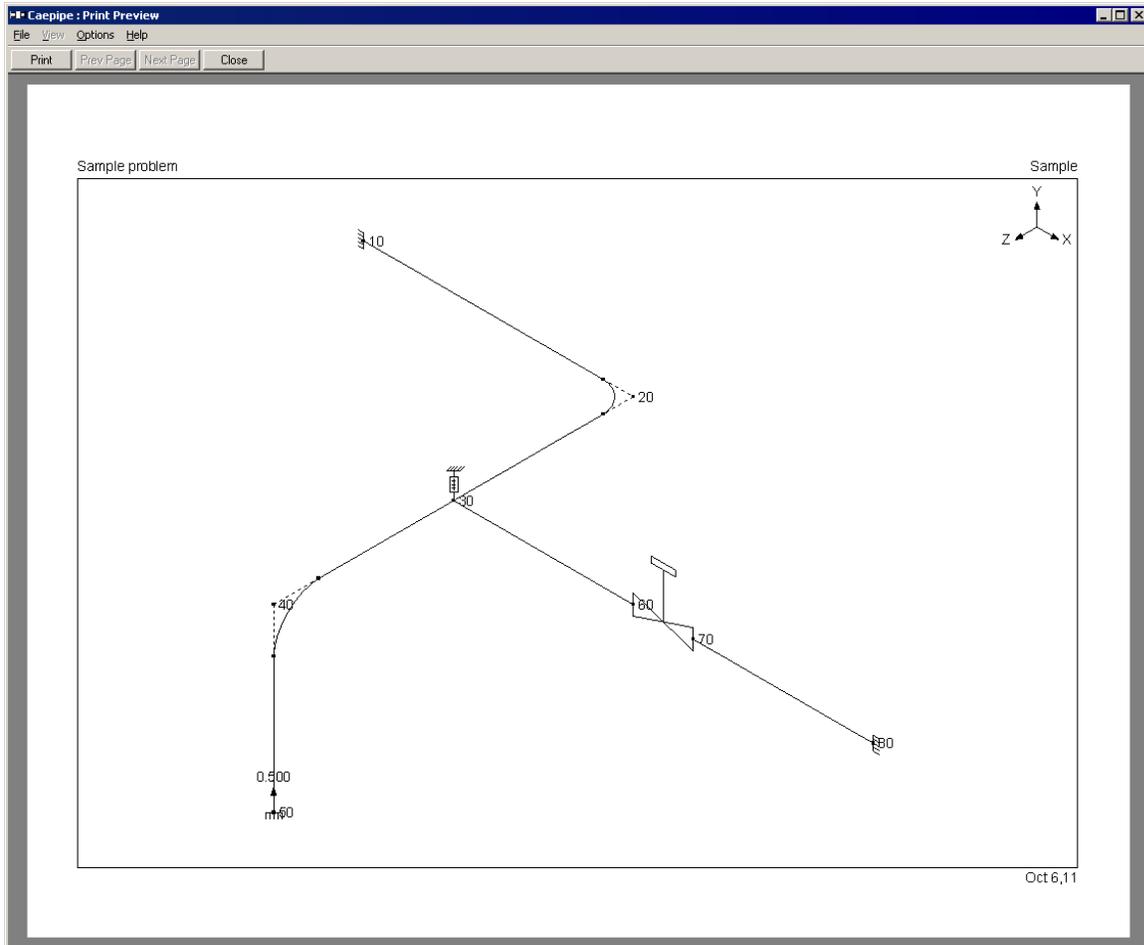
Font

The font you select here is different from the Font selected in the Graphics Font dialog under the Options menu. The Font command here sets the font for printing only, and the one under the Options menu sets the font to use for Graphics display only as explained on p. 111.

Graphics Window File Menu

Preview

Before you print an image, you can preview it by clicking on this button.

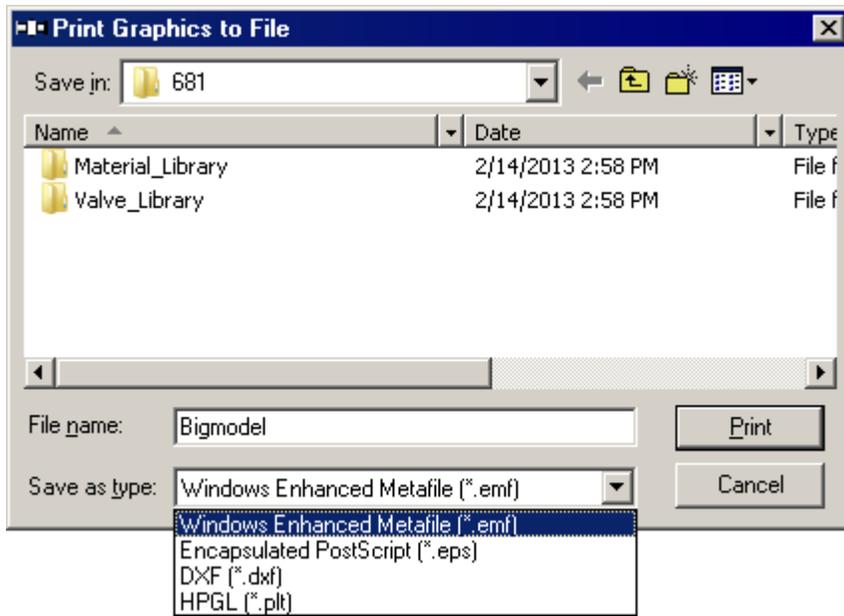


While in the Preview mode, click on Close to return to the Graphics window or click on Print to send the job to the printer.

Print to File

This can be a useful command to convert the line drawing (vector) graphics to other vector formats such as Encapsulated Postscript (EPS), Windows Enhanced Metafile (EMF), AutoCAD's DXF or HPGL's PLT. You can specify a filename for the graphics file. CAEPIPE can also convert other vector graphics such as non-rendered stress and stress ratio plots, deflected and mode shapes to the above vector formats.

Graphics Window File Menu

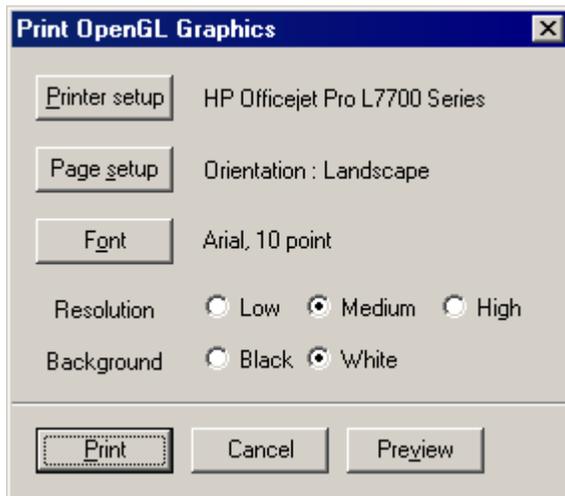


Note on DXF output

CAEPIPE sends only the model information including the line drawing into a DXF file in different layers. It does not send any results (like stresses or deflected shapes) to a DXF file.

Print Dialog for OpenGL Rendered Images

For rendered images, you will see a different Print dialog. Here, as in the earlier Print dialog, you can select a printer and customize certain print settings. The resolution and image background can be set, too. A black background will consume more of your printer ink.

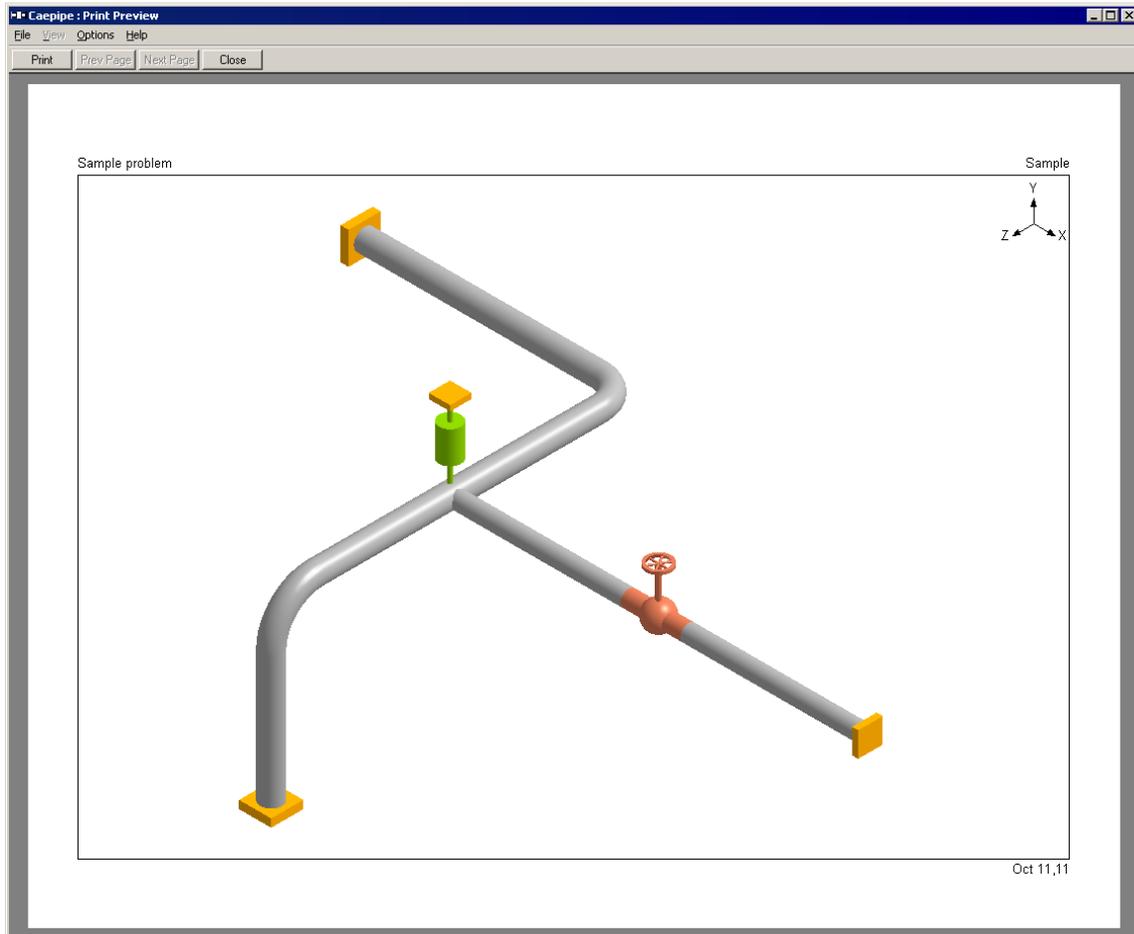


Graphics Window File Menu

Setting the resolution to High might take up a lot of system memory and slow your system down. Should such happen, use the Low or Medium setting.

Preview

Preview the OpenGL graphics rendering by clicking on this button.



Copy (Ctrl+C)

You can copy the displayed graphics image to the clipboard and then paste this image into any other program that accepts it. For example, you can copy the rendered view of a model and paste it into your paint or a graphics processing program so that you can use the finished image (possibly with your annotations) in a word processing program of your choice to generate a report.

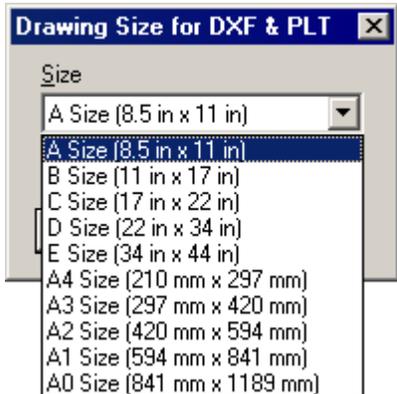
Drawing Size

The drawing size command applies only to DXF and PLT files. Before you generate a DXF or a PLT from menu File > Print > Print to File, you need to set the drawing size here. The default is set to US A size (Letter – 8.5” x 11”).

Graphics Window File Menu



There are several US and ISO sizes available.



Plot Title

For the graphics image printout (or for the image copied through the Copy command), you may type in an image title different from the model title (in the Layout window). Clicking on Reset will restore the title to the original model Title from row 1 in the Layout window.



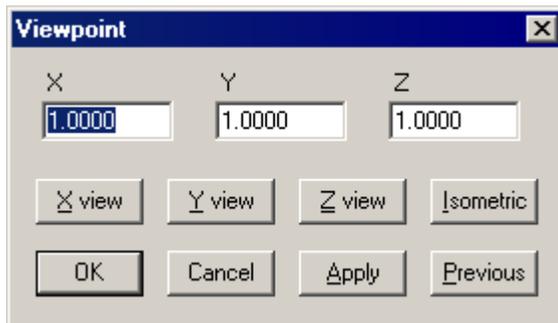
Graphics Window View Menu

View Menu

This menu contains commands for graphics operations.

View	Options	Window	Help
Viewpoint...			F4
Previous View			F5
Center			Ctrl+Shift+C
Turn			Ctrl+T
Zoom All			Ctrl+A
Zoom In			Page Up
Zoom Out			Page Dn
Show...			Ctrl+S
Show Nodes...			Ctrl+N
Show/Hide Dots + Numbers			F3
Redraw			Ctrl+D
Render			Ctrl+R
Hide Current Element			F6
Show All			F9
Make Transparent			F7
All Transparent			F10
All Opaque			F11
Show/Hide Selected Elements			F12
Freeze View			F8

Viewpoint (F4)



Use this command to set the graphics viewpoint. Several useful buttons inside the dialog allow you to change viewpoint to a preset one.

For example, if you want to see the “plan” view (Y-vertical), click on “Y view” button.

Previous View (F5)

Use this command to display the previously viewed graphics image in the Graphics window. The last used viewpoint, zoom level and area of the image are brought back into view.

Graphics Window View Menu

Center (Ctrl+Shift+C)

Use this command for centering the image around a particular location. When you select this command, the mouse pointer turns into a crosshair. It can be moved around by moving the mouse. Position the crosshair at the image location that you want centered and then left-click on the mouse to center the whole image around that point.

Turn (Ctrl+T)

Use this command to rotate the image about the horizontal or the vertical axis. Move the thumb on the scroll bar to rotate the image, or use the arrow keys on the keyboard.

Or, use

Shift + Mouse Scroll Button Up and Down for horizontal rotation of model

Ctrl + Mouse Scroll Button Up and Down for vertical rotation of model

In Translation mode, use:

Shift + Mouse Scroll Button Up and Down for horizontal translation of model

Ctrl + Mouse Scroll Button Up and down for vertical translation of model

Zoom All (Ctrl+A)

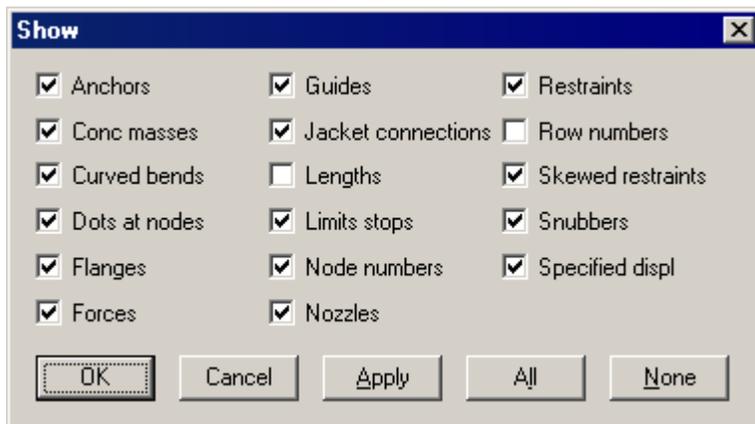
Use this command to view the whole model in the Graphics window.

Zoom In, Zoom Out

Use these commands to increase or decrease the magnification level for the image. Hotkeys are Page Up (for Zoom in) and Page Down (for Zoom out). You can also use the mouse wheel button to Zoom in and Zoom out.

Show

Use this command to either display or suppress display of various items in the Graphics window. This feature helps in reducing clutter in the displayed image. In addition, it enables you to display selectively one or more items in the Graphics window. Select or deselect items and click on OK or Apply (to see the effect immediately while keeping the dialog open). Click on All or None button to select all or deselect all the items with one click.

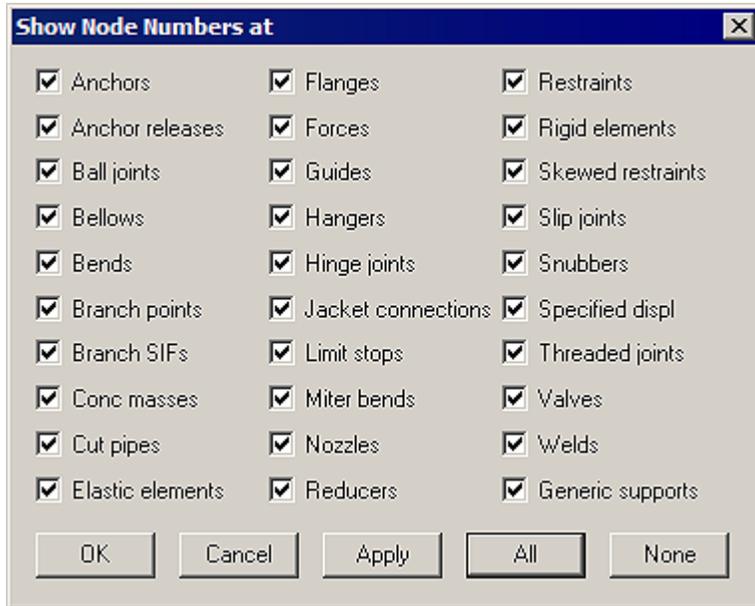


The Local Coordinate system can be shown for all the elements while viewing Results.

Show/Hide Dots and Numbers (F3)

Use the hotkey F3 to remove Dots and Node numbers from the graphics display.

Show Nodes



Use this feature to show node numbers selectively at different locations (such as anchors, limit stops, valves, etc.) to check your input or to display node numbers selectively for one or more items (anchors, hangers, etc.) in the Graphics window. To show node numbers, select or deselect the required items and click on OK or Apply (to see the effect immediately while keeping the dialog open). Click on All or None button to select all or deselect all the items with one click.

Redraw (Ctrl+D)

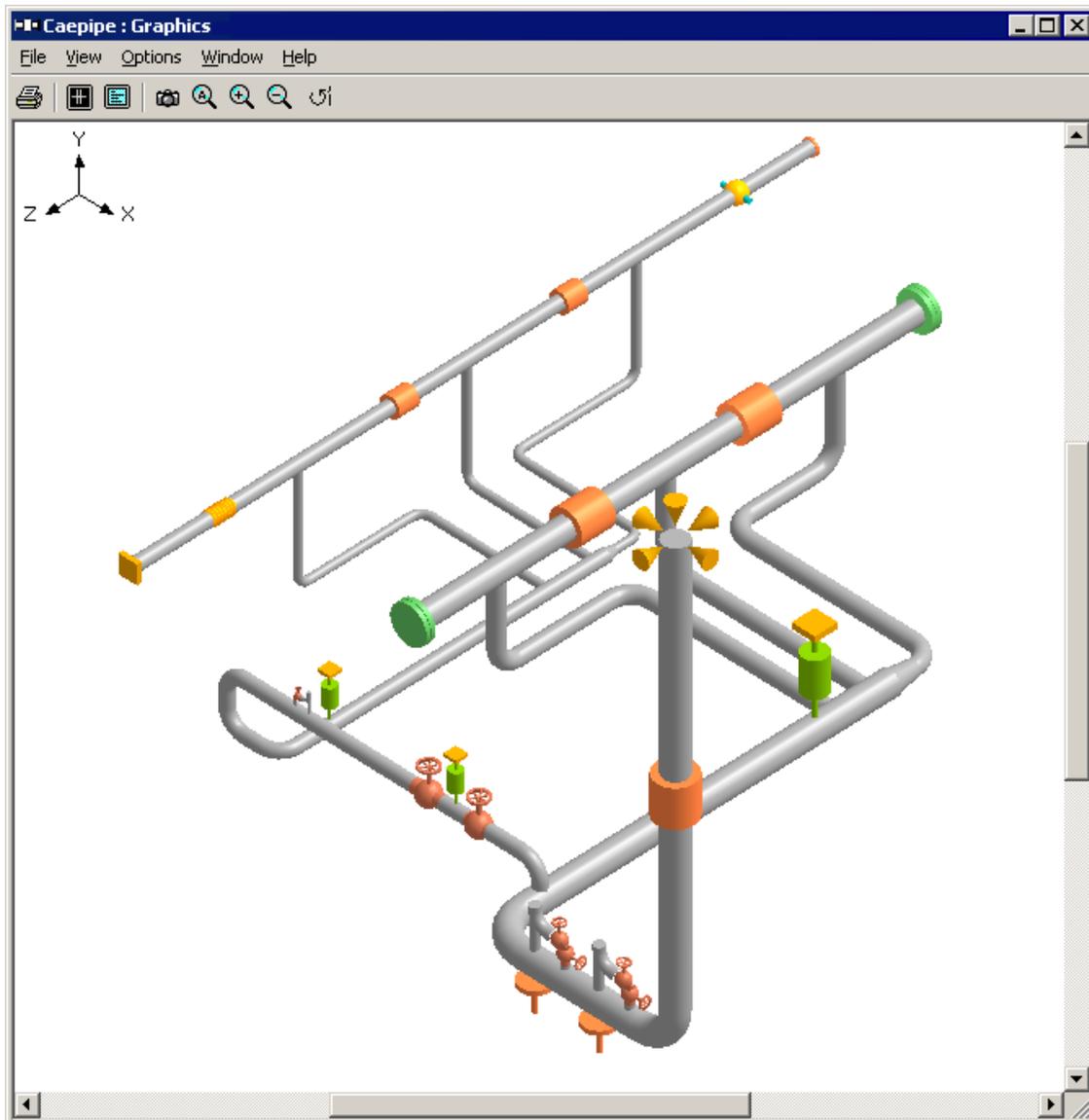
Use this command to redraw the entire graphics image. This will refresh the window with the existing image. This feature is useful when you are working with a two-monitor system.

Render (Ctrl+R)

Use this command to generate a realistic 3-D image of the model. Once you have a rendered image, you can use the Zoom All, Zoom in, Zoom out, Turn and other graphics commands on it. You can use the Show and the Show Nodes commands too to show specific items and/or node numbers. This image can be printed to a (color) printer.

Materials, Sections and Loads are shown graphically using Color Codes in rendered graphics. Press the “Color Graphics” icon available in Materials, Sections and Loads List windows to execute the command.

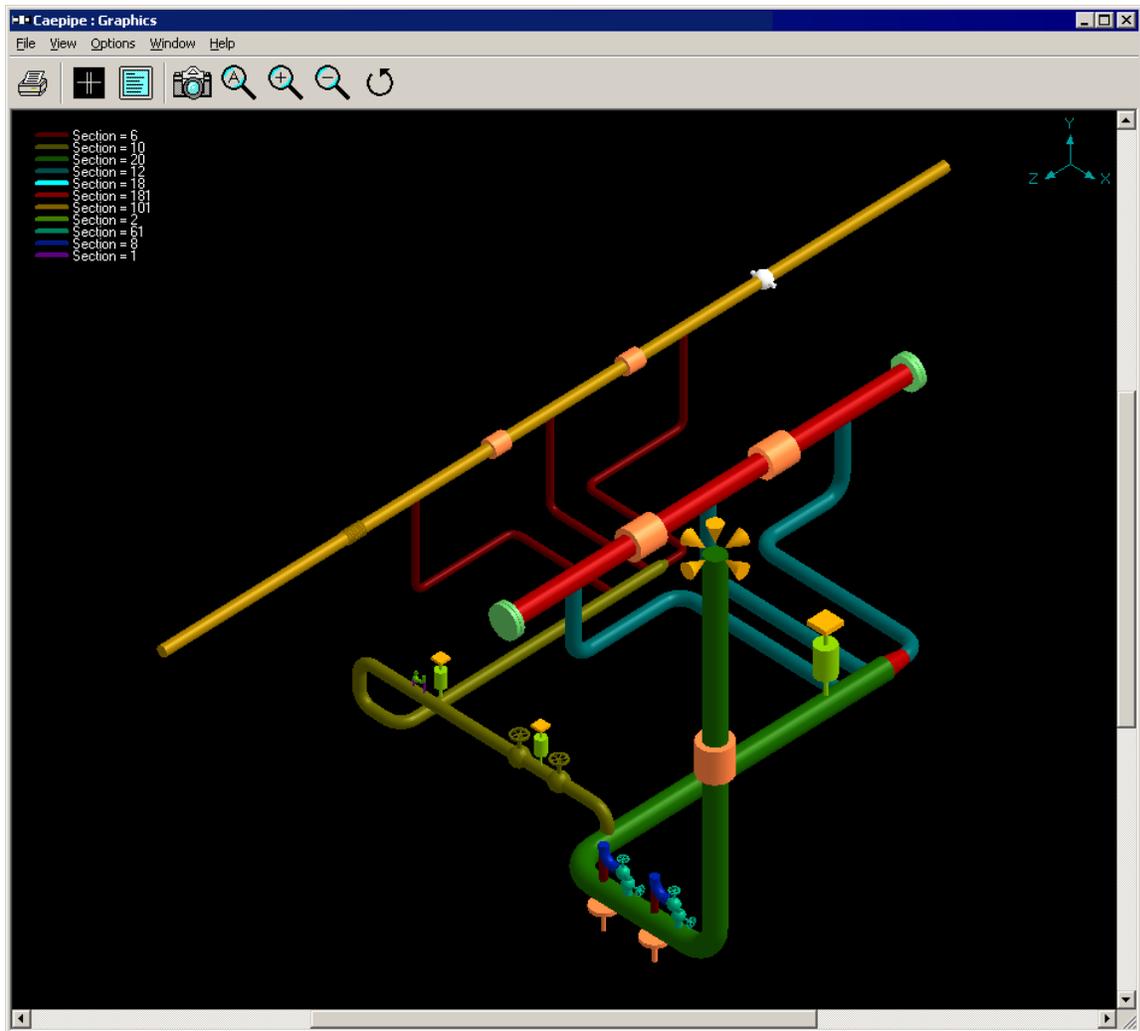
Graphics Window View Menu



The screenshot shows the 'Caepipe : Pipe Sections (11)' window. The title bar includes 'Caepipe : Pipe Sections (11)' and standard window controls. The menu bar contains 'File', 'Edit', 'View', 'Options', 'Misc', 'Window', and 'Help'. Below the menu bar is a toolbar with icons for view manipulation, and one icon (a red crosshair) is circled in red. Below the toolbar is a table with 10 columns and 3 rows of data.

#	Name	Nom Dia	Sch	OD (mm)	Thk (mm)	Cor.Al (mm)	M.Tol (%)	Ins.Dens (kg/m3)	Ins.Thk (mm)
1	5	6"	40	168.27	7.112		12.5	136.16	90
2	10	10"	40	273.05	9.271			136.16	90

Graphics Window View Menu



Hide Current Element (F6)

With the graphics cursor flashing on the element of interest, press F6 (or select this command) to hide it from view (in rendered mode).

Show All (F9)

Press F9 (or select this command) to display all elements in the Graphics window including those that were hidden using the “Hide Current Element” command.

Make Transparent (F7)

Press F7 (or select this command) to make an element transparent so you can see what is behind it; useful in congested models.

All Transparent (F10)

Press F10 (or select this command) to make all elements transparent. F11 will make them opaque.

All Opaque (F11)

This is the opposite of the previous command in that it makes all transparent elements opaque. F10 will make all elements transparent.

Show/Hide Selected Elements (F12)

This command works on showing or hiding a group of elements defined in a range. See figure for the different choices.

Freeze View (F8)

This key is a toggle. Pressing F8 freezes or unfreezes the display so you do not lose your point of view while modeling.

You may still perform the Zoom, Pan, Rotate, etc., with the frozen view. Also, the option “Freeze View” disables the dynamic updating of view (i.e., view remains unchanged) when you scroll in the Layout window.

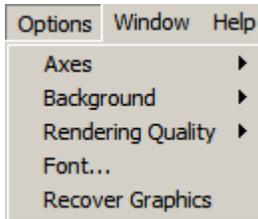
Show/Hide Dots and Numbers (F3)

Use the hotkey (F3) to remove the Dots and Node numbers from the Graphics window simultaneously. Pressing F3 (toggle) again will bring them back.

Graphics Window Options Menu

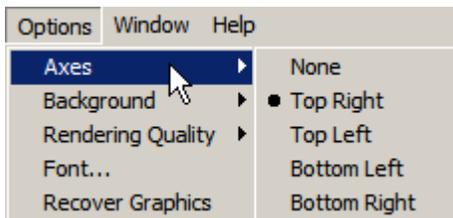
Options Menu

This menu contains commands to change the axes symbol location, color of graphics background, OpenGL rendering quality and the graphics font.



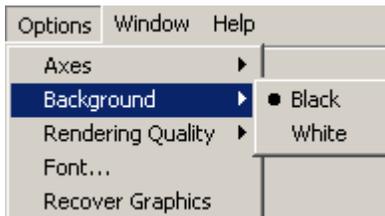
Axes

The axes symbol can be displayed in any one of the four corners of the Graphics window or not at all.



Background

The default background color for the Graphics windows is black. You can change it to white if you like. Changing so will also use less printer ink when you print a graphics image.



Rendering Quality

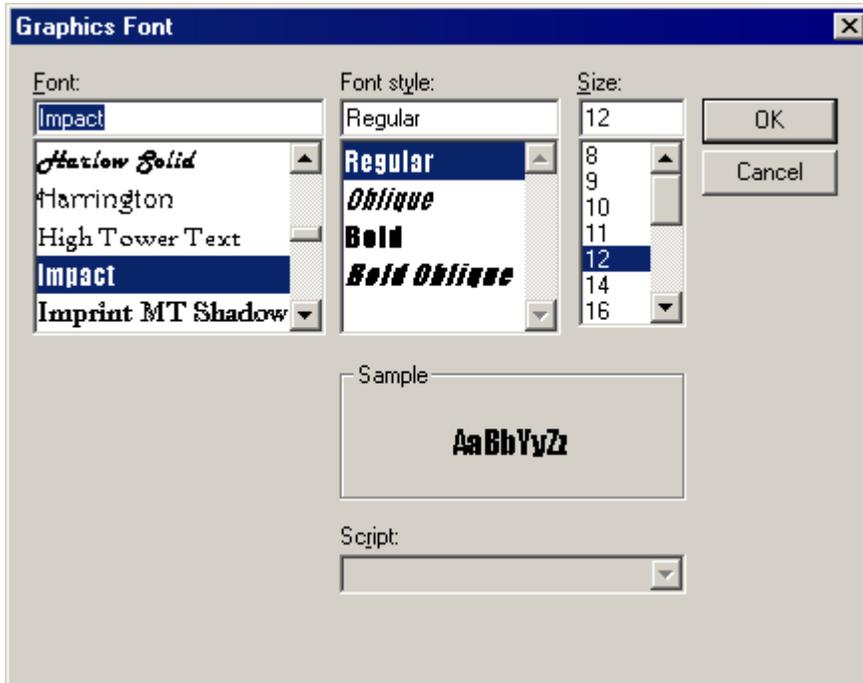
This feature allows you to set the quality of the rendered image. It is best to set this to High unless your computer renders images noticeably slowly.



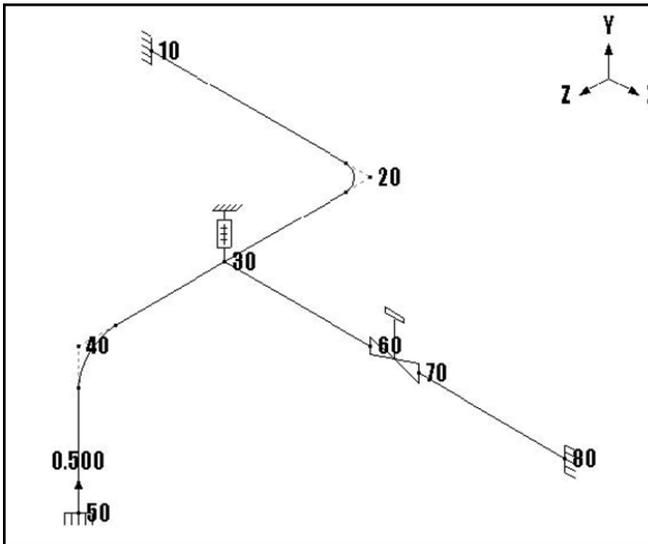
Font

You can select any available font for graphics display here. Note: Font for printing graphics is selected in the Print Graphics dialog as explained on p. 100.

Graphics Window Options Menu



The text in the image below is shown in Impact font.



Graphics Window Options Menu

Recover Graphics

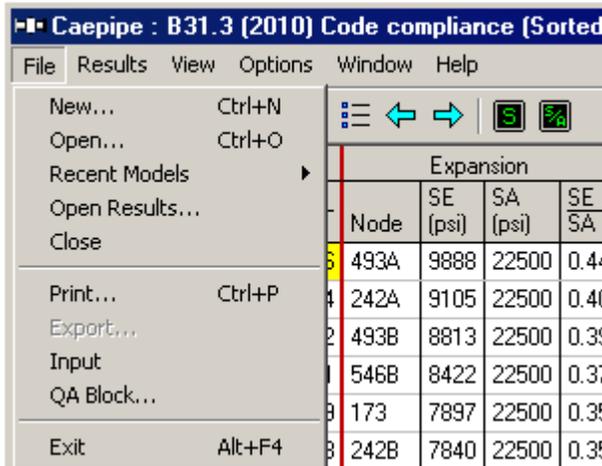
If CAEPIPE terminates abnormally, the graphics image may get garbled. You can recover the graphics image, however, using one of two methods:

- a) Open the model, enter the data to complete the inputs and save. Re-open the model to recover the graphics, OR
- b) Open the model, enter the data to complete the inputs and save. Select the command "Recover graphics" through Graphics Window > Options.



Results Window File Menu

File Menu



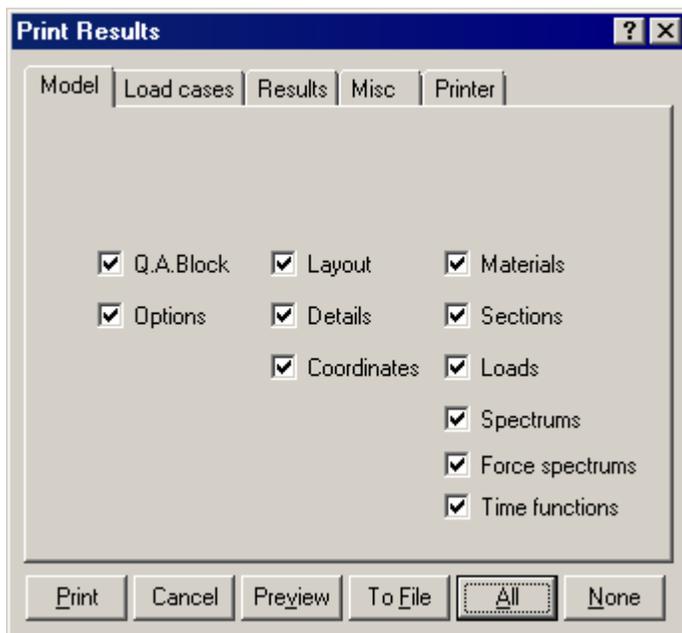
The File menu here is similar to the File menu in the Layout window. You can open or close any CAEPIPE model, material, spectrum library or CAEPIPE results file from here.

Print (Ctrl+P)

This is a comprehensive command that lets you print all model input and computed results. This dialog has five tabs and six buttons.

Model

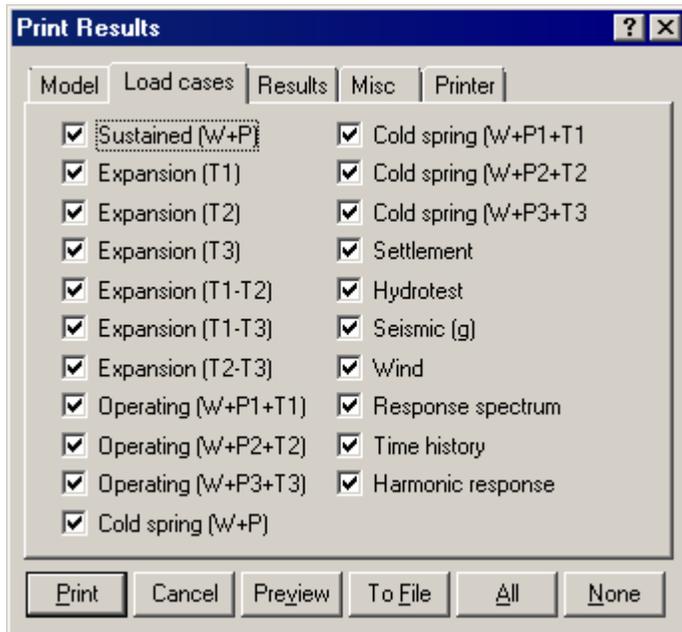
On this tab, check those input data items that you would like to print.



Results Window File Menu

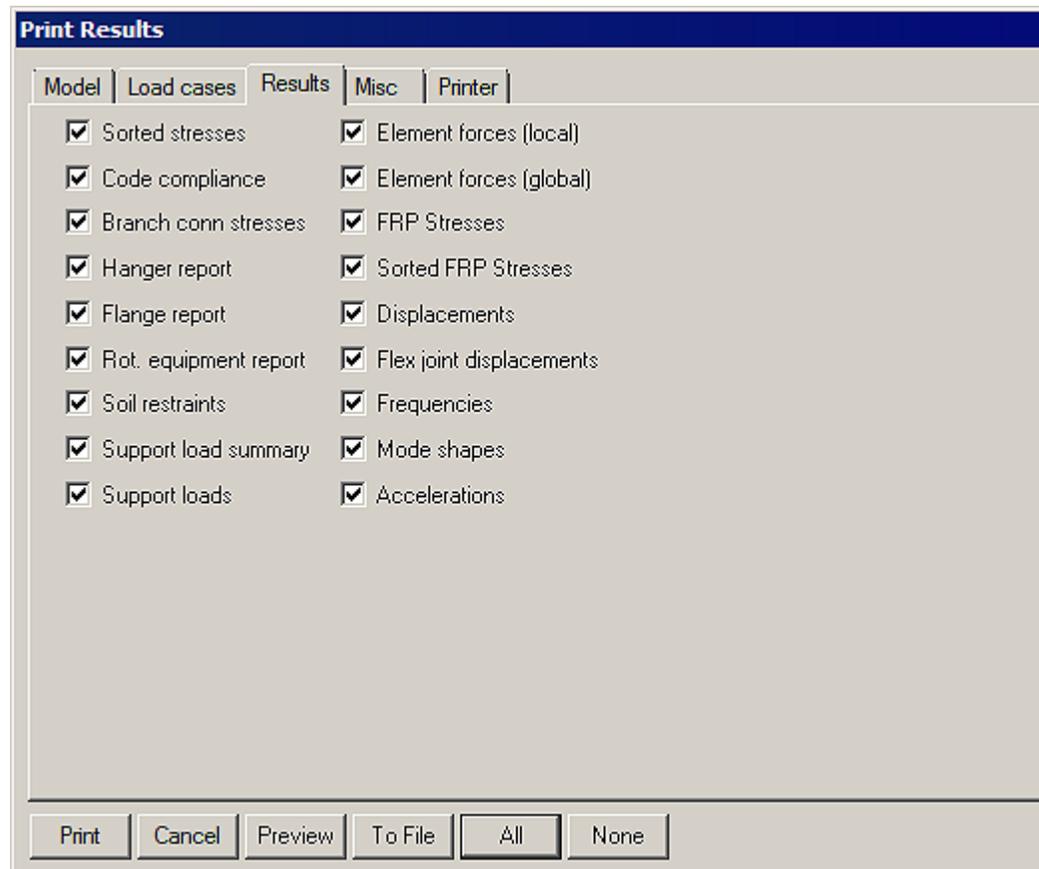
Load cases

On this tab, check the different load cases for which you want to print results.



Results

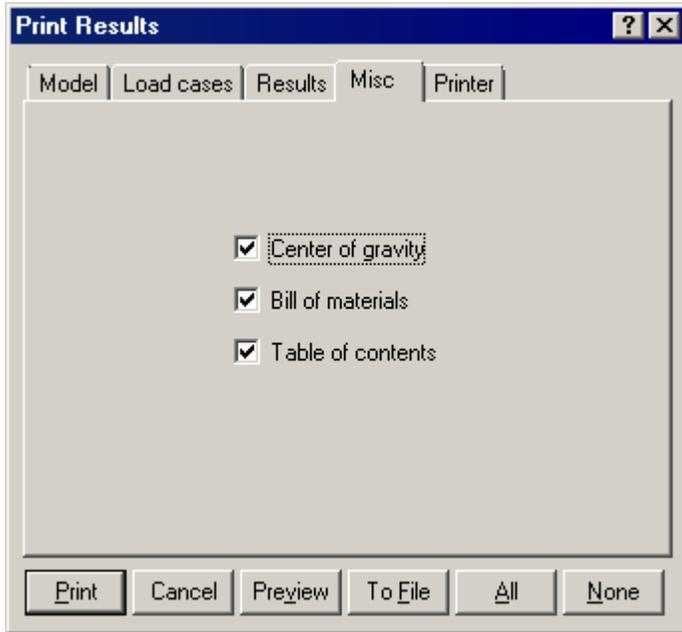
On this tab, check the different results items that you want to print.



Results Window File Menu

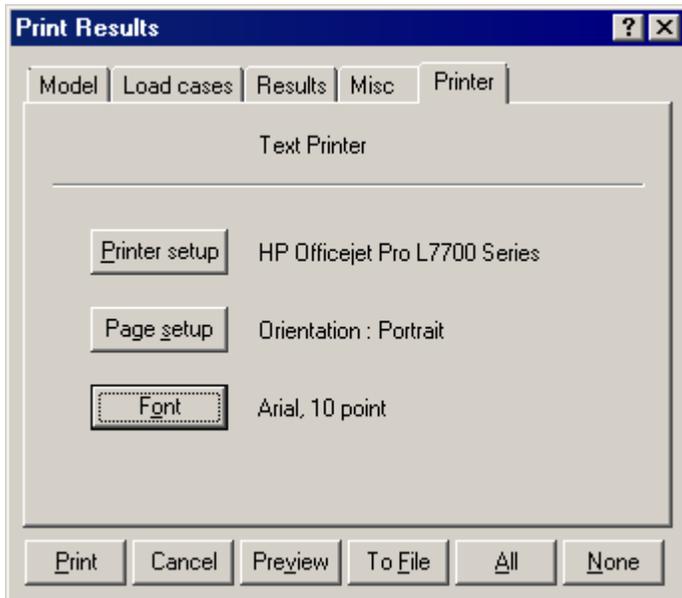
Misc

On this tab, select the items of interest that you want to print.



Printer

On this tab, you can customize your printer settings such as selecting a different printer, setting different margins, and the print font. By selecting a custom Adobe Acrobat-compatible printer (if available on your system), you can generate an Adobe PDF (Portable Document Format) file that contains the whole model input and output.



Print Dialog Buttons

Print

After selecting the items you want to print, click on Print button to send output to the printer selected under the Printer tab.

Cancel

You can click on Cancel to exit the dialog without printing.

Preview

Before printing, you can preview how the printed output will appear by clicking on Preview. If necessary, you can adjust the printer settings (under the Printer tab) to change the print font and margins, and preview again.

Results Window File Menu

Caepipe : Print Preview, Page 4 (of 15)
_ □ ×

File View Options Help

Print
Prev Page
Next Page
Close

Caepipe
Sample problem
Page 4

Pipe Sections (2)											
Name	Nom Dia	Sch	O.D (Inch)	Thk (Inch)	Cor. Al (Inch)	M.Tot (%)	Ins.Dens (lb/ft3)	Ins.Thk (Inch)	Un.Dens (lb/ft3)	Un.Thk (Inch)	Soil
8	8"	80	8.625	0.5			15	2			
6	6"	STD	6.625	0.28			15	2			

Pipe Loads (1)					
Name	T1 (F)	P1 (psf)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load
1	600	200	0.8		

831.3 (2010) Code compliance (Sustained stresses)

Node	Sustained			Expansion			SE	SA	SE
	SL (psf)	SR (psf)	SL SR	SE (psf)	SA (psf)	SE SA			
80	2538	17900	0.14	30	52765	29475	1.79		
60	2205	17900	0.12	50	50355	29475	1.71		
70	2133	17900	0.12	20A	47368	29475	1.61		
30	2033	17900	0.11	20B	38541	29475	1.14		
10	1448	17900	0.08	10	32123	29475	1.09		
40B	1053	17900	0.06	80	27166	29475	0.92		
20B	980	17900	0.05	40A	18838	29475	0.64		
20A	937	17900	0.05	60	17572	29475	0.60		
90	924	17900	0.05	70	12275	29475	0.42		
40A	902	17900	0.05	40B	10284	29475	0.35		

831.3 (2010) Code Compliance

Node	Press (psf)	Sustained			Expansion			SE	SA	SE
		SL (psf)	SR (psf)	SL SR	SE (psf)	SA (psf)	SE SA			
10	200	1448	17900	0.08	32123	29475	1.09			
20A	207.5	931	17900	0.05	28144	29475	0.95			
20A	200	937	17900	0.05	47368	29475	1.61			
20B	207.5	980	17900	0.05	38541	29475	1.14			
20B	200	968	17900	0.05	19655	29475	0.67			
30	207.5	1767	17900	0.10	52765	29475	1.79			
30	200	1758	17900	0.10	47439	29475	1.61			
40A	207.5	902	17900	0.05	15933	29475	0.54			
40A	200	902	17900	0.05	18838	29475	0.64			
40B	207.5	1053	17900	0.06	10284	29475	0.35			
40B	200	1053	17900	0.06	9293	29475	0.32			
90	207.5	924	17900	0.05	50355	29475	1.71			
30	200	2033	17900	0.11	37359	29475	1.27			
60	1513	2205	17900	0.12	17572	29475	0.60			
70	200	2133	17900	0.12	12275	29475	0.42			
80	1513	2538	17900	0.14	27166	29475	0.92			

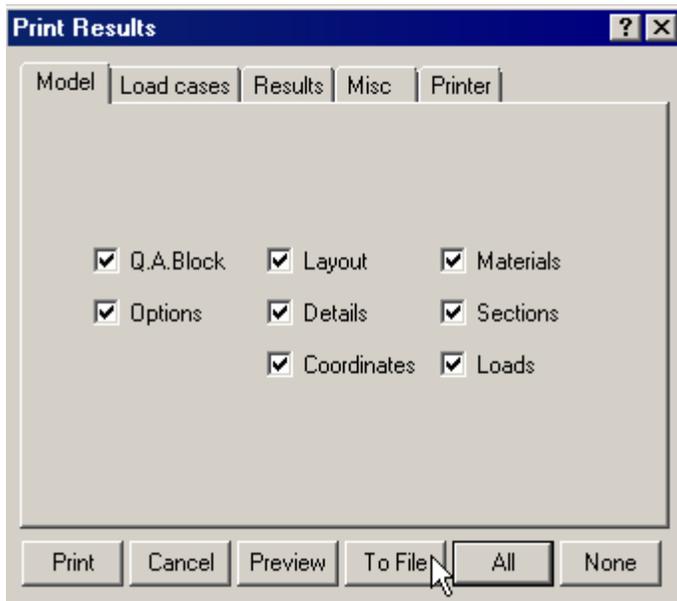
Hanger Report										
Node	No of	Type	Figure No.	Size	Spring rate (lb/Inch)	Ver travel (Inch)	Horz travel (Inch)	Hot load (lb)	Cold load (lb)	Var (%)
30	1	Grinnell	8-268	10	250	0.601	0.606	1287	1443	12

Version 6.81
Sample
May 15, 13

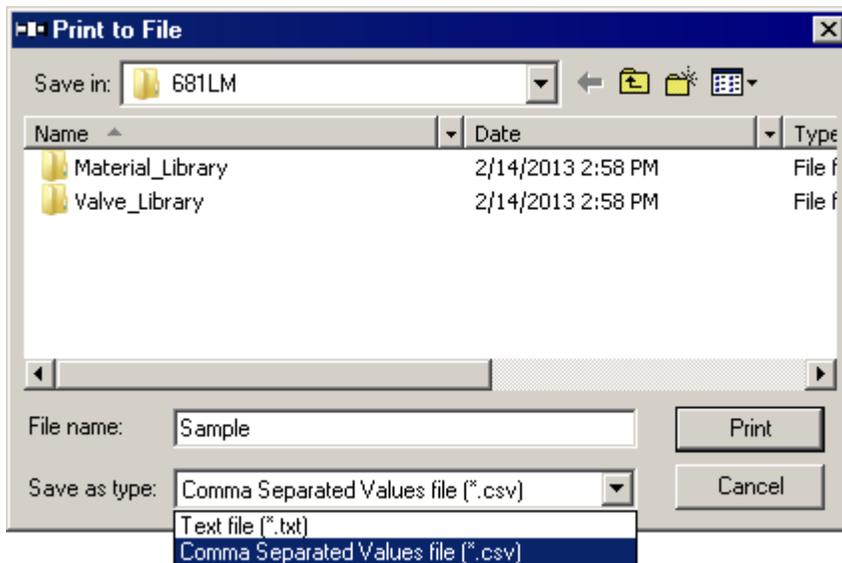
Results Window File Menu

To File

If you want to send all of your selected items (input and results) click on the To File button.



You can export the data to a .txt file or to a .csv file.



Results Window File Menu

Here's an example of what the .csv file looks like with the exported data.

	A	B	C	D	E
16	Report Number				
17					
18	Model Name: complex1				
19					
20	Title: Sample problem				
21					
22	Options	B31.1 (2010) Code Compliance (Sorted Stresses)			
23					
24	Piping code = B31.1 (2010)		Sustained		
25	Do not use liberal allowable stresses		SL		
26	Do not include axial force in stress calculations	Node	(psi)	SL/SH	Node
27	Reference temperature = 70 (F)				
28	Number of thermal cycles = 7000	70	35335	3.02	20A
29	Number of thermal loads = 3	150	26356	2.25	30
30	Solve thermal case	1000	19870	1.7	50
31	Use temperature dependent modulus	80	18792	1.61	35
32	Include hanger stiffness	125	18006	1.54	20B
33	Include Bourdon effect	124	18006	1.54	10
34	Use pressure correction for bends	133	17422	1.49	55
35	Pressure stress = PD / 4t	127	14616	1.25	70
36	Peak pressure factor = 1.00	126	14616	1.25	40A
37	Cut off frequency = 999 Hz	1010	13046	1.12	80
38	Number of modes = 6	55	12030	1.03	40B
39	Include missing mass correction	60	11139	0.95	150
40	Do not use friction in dynamic analysis	120	10562	0.9	1000
41	Vertical direction = Y	30	10506	0.9	25
42		122	10331	0.88	133

All

To select all items, click on the All button.

None

To deselect all items, click on the None button.

Input

This useful command allows you to go to the model Layout window quickly without having to go through a model File > Open... command.

QA Block

This command shows you the QA information you input in the Layout window. As the shown information is not editable here, to make changes to it, you need to go back to the Layout window and use the same command under the File menu. After you make edits, you need to save the model and reanalyze.

Results Window File Menu

Quality Assurance Block ✕

Client

Project

File number

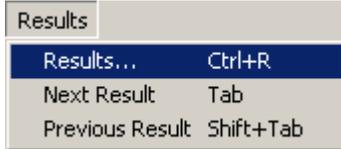
Report

Prepared by

Checked by

Results Window Results Menu

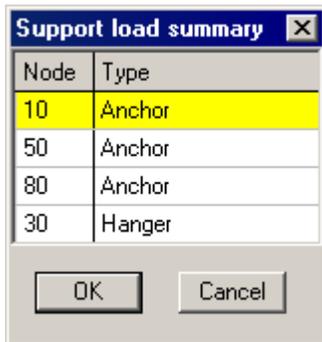
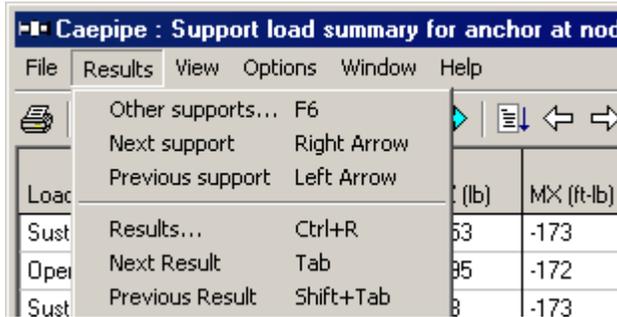
Results Menu



You can open the list of all results computed by selecting Results command (Ctrl+R). Press Tab or Shift+Tab to move forward or backward through the different results items, each of which will be shown in the same Results window.

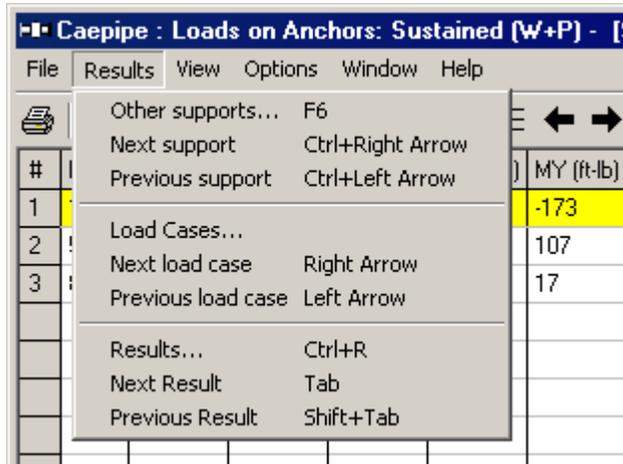
This menu may change with the displayed results item. For example, the menu shows like in the above figure for Sorted Stresses, Code Compliance and Hanger Report.

For Support Load summary, the menu shows similar to the following figure. The first set of commands is for showing the support load summary for other supports, one at a time. The second set of commands is as before.



For Support Loads, the menu changes to show related commands for the new results item.

Results Window Results Menu

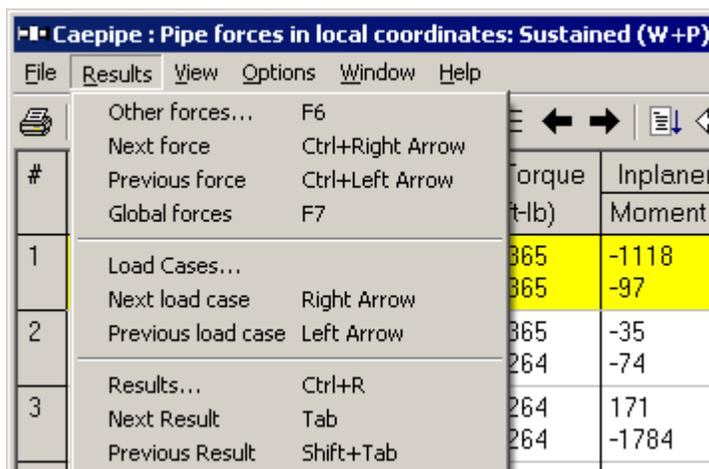


The first set of commands is for showing the support loads for other supports (hangers, limit stops, etc.), one at a time (you can also click on the white right or left arrow).



The second set of commands (Load cases, etc.) is for changing the load case for the displayed support. For example, the title bar of the figure shows Loads on Anchors: Sustained. You can change the load case to Expansion load case for this support by selecting the Next load case command (you can also click on the black right or left arrow). The last set of commands is as before, moving through different Results screens.

For Element forces, the menu changes to show related commands.



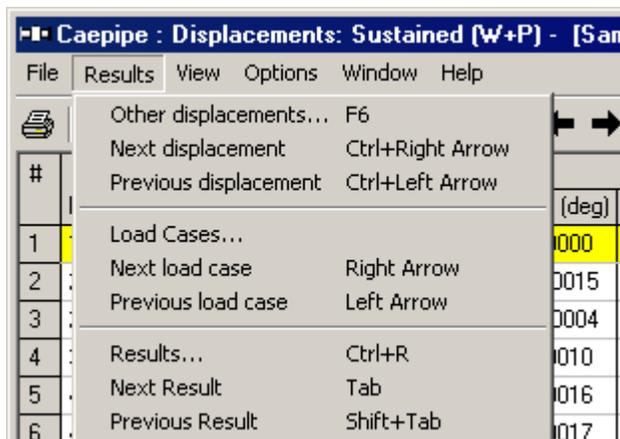
The first set of commands is for showing other forces (at other elements such as expansion joints and valves), one at a time (you can also click on the white right or left arrow).

Results Window Results Menu



The second set of commands (Load cases, etc.) is for changing the load case for the displayed support. For example, if the title bar of the figure showed Sustained load case, then by selecting the Next Load case command, you can display forces for the Expansion load case (you can also click on the black right or left arrow). The last set of commands is as before, moving through different Results screens.

For Displacements, the menu changes to show related commands as follows:



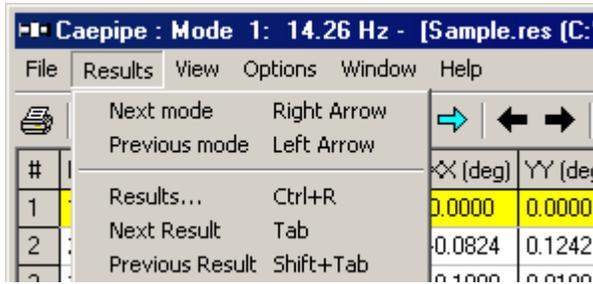
The first set of commands is for showing other displacements (such as minimum and maximum displacements and at other supports) you can also click on the white right or left arrow.



The second set of commands (Load cases, etc.) is for changing the load case for the displayed displacements. For example, if the title bar of the figure showed Sustained load case, then you can show displacements for the Expansion load case by selecting the Next Load case command. (You can also click on the black right or left arrow). The last set of commands is as before.

For Mode Shapes, the menu changes to show two commands specific to the new results.

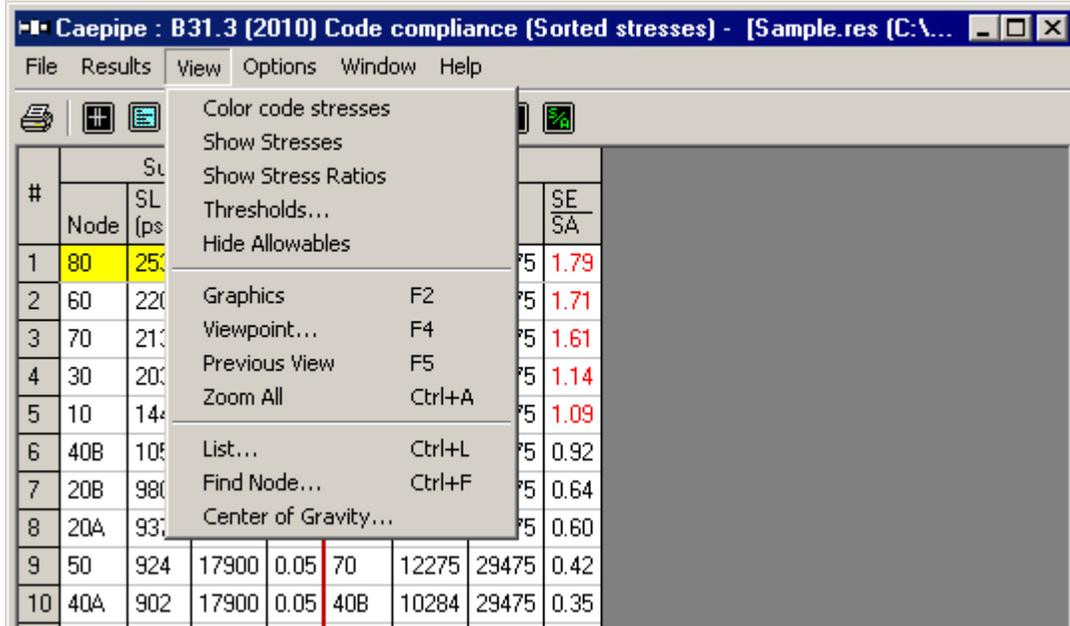
Results Window Results Menu



They are to move forward or backward through the available mode shape details. The last set of commands is as before.

So, depending on the displayed results item, the Results menu (Ctrl+R) will show different menus with at least one common set of commands for moving forward or backward through the different results.

View Menu



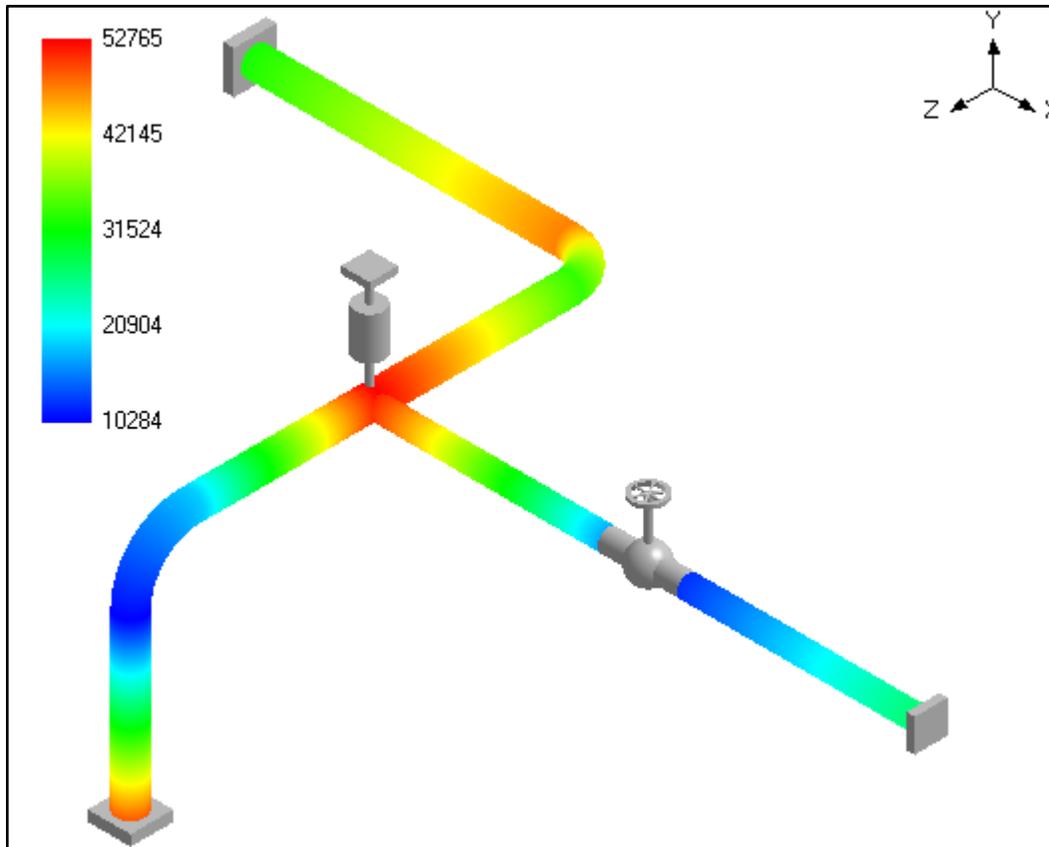
This menu contains commands mostly for graphics operations. The List and Find Node commands function like they do in Layout window > View menu. Center of Gravity command is covered later in this section. As with the Results menu, this menu too changes its offerings depending on the results item being viewed.

The first menu shown here is available from the Sorted Stresses results. The first set of commands helps you view stresses graphically in the Graphics window.

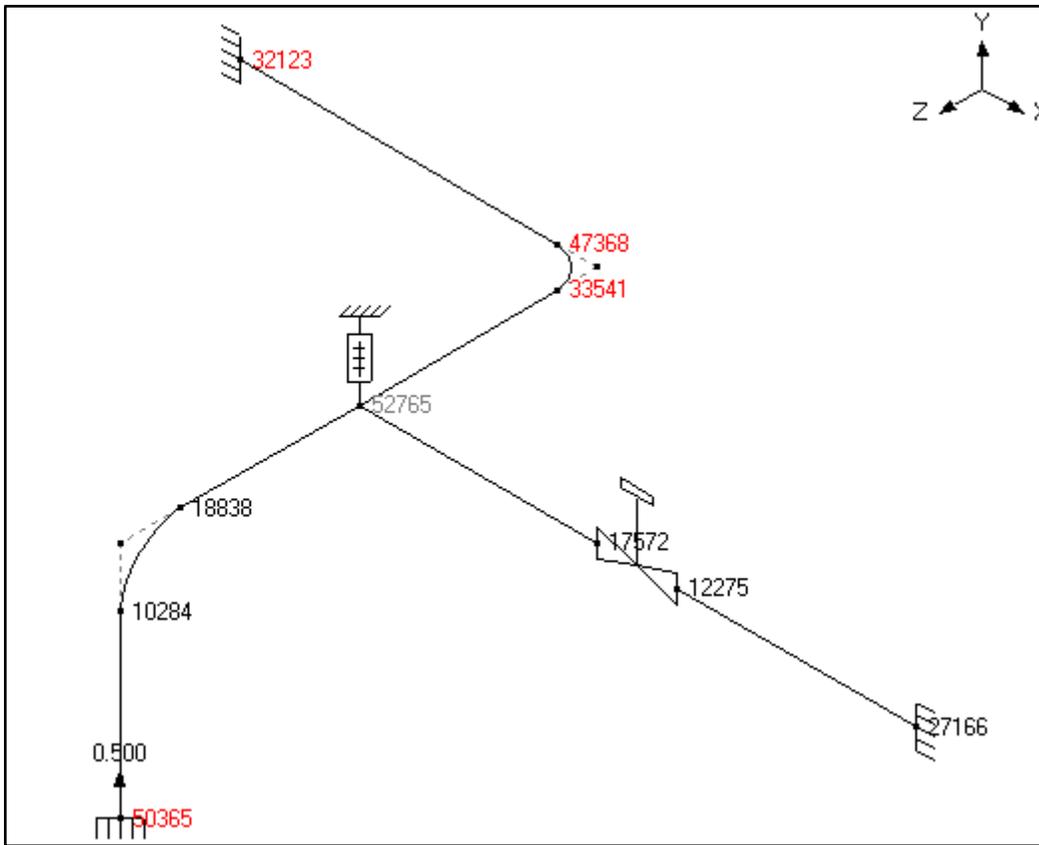
Color code stresses / No color coding

Displays or removes the color coded stress (or stress ratio) contour in the Graphics window. To display a color coded contour, place the yellow highlight on the type of stress you want to see graphically, for example, on Expansion stress. Then, select Color code stresses. Next, click on Show stresses or stress ratios button on the toolbar. The Graphics window will show the color coded contour plot, also available for jacketed piping.

Results Window View Menu



When you select No color coding, numeric values for stresses or stress ratios are shown see next image).



Thresholds

Use this command to display stresses (or stress ratios) over a specified stress or stress ratio value.

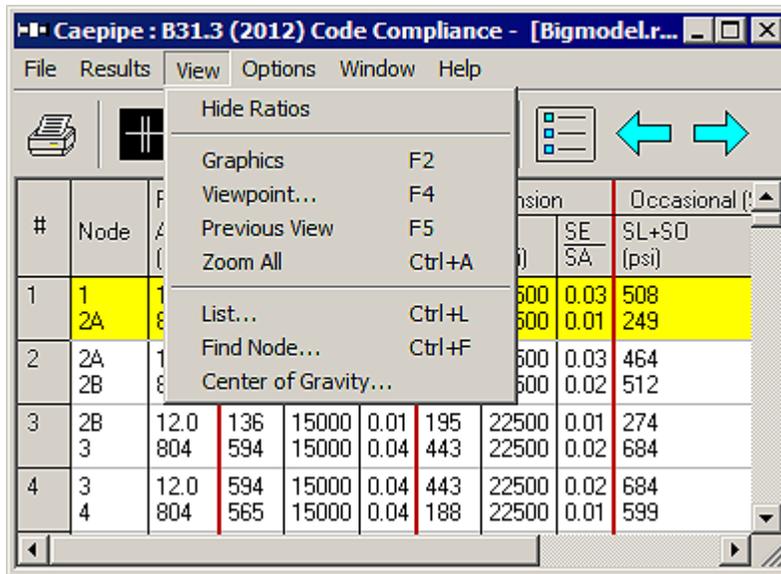
Hide Allowables

Selecting this command will remove the Allowables columns from the Results display. For some codes that have too many columns, this command helps by lessening the number of columns displayed.

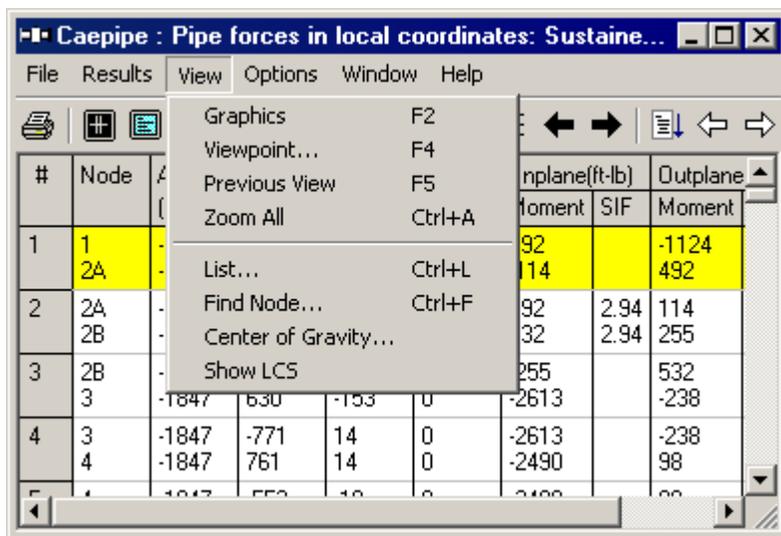
All commands (except Hide Ratios and Center of Gravity) function like they do under the Layout window > View menu.

Under the Code Compliance results display, the View menu shows only one command to Hide (stress) ratios (such as SL/SH, SE/SA). The remaining commands are the same as before.

Results Window View Menu

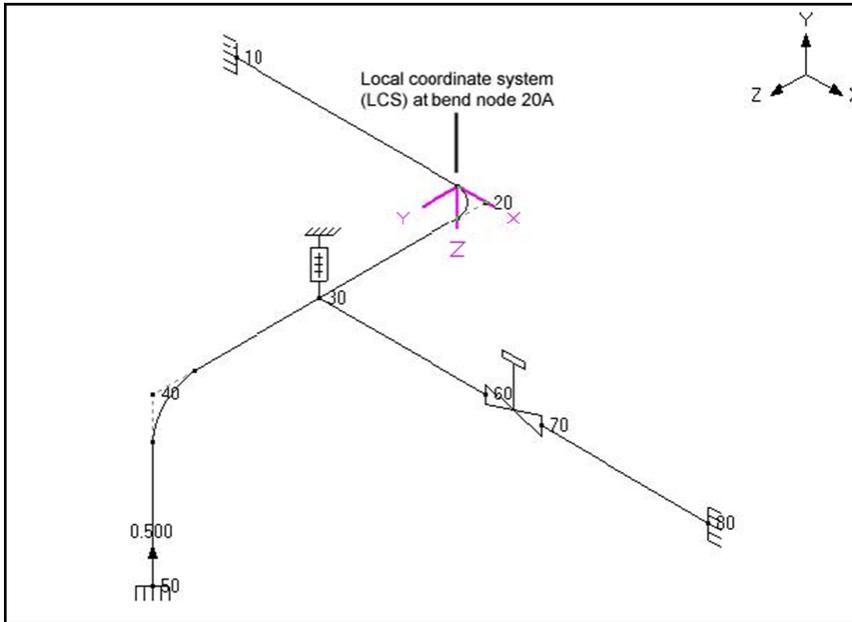


For the Pipe element (or Other element) forces display, the following View menu is shown.

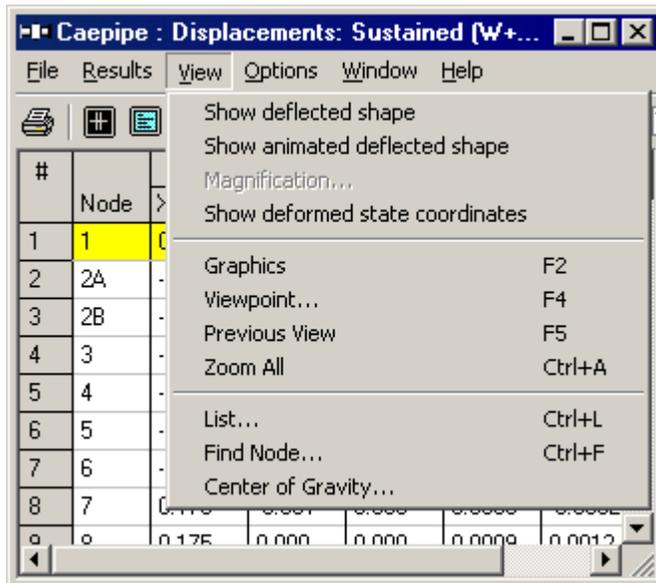


The only item new here is a command to Show LCS (Local Coordinate system)/Hide LCS in the Graphics window for each element. As you keep scrolling through the different elements in the results display, the corresponding element in the Graphics window will show a small coordinate symbol that depicts the element's LCS. The remaining commands are the same as before.

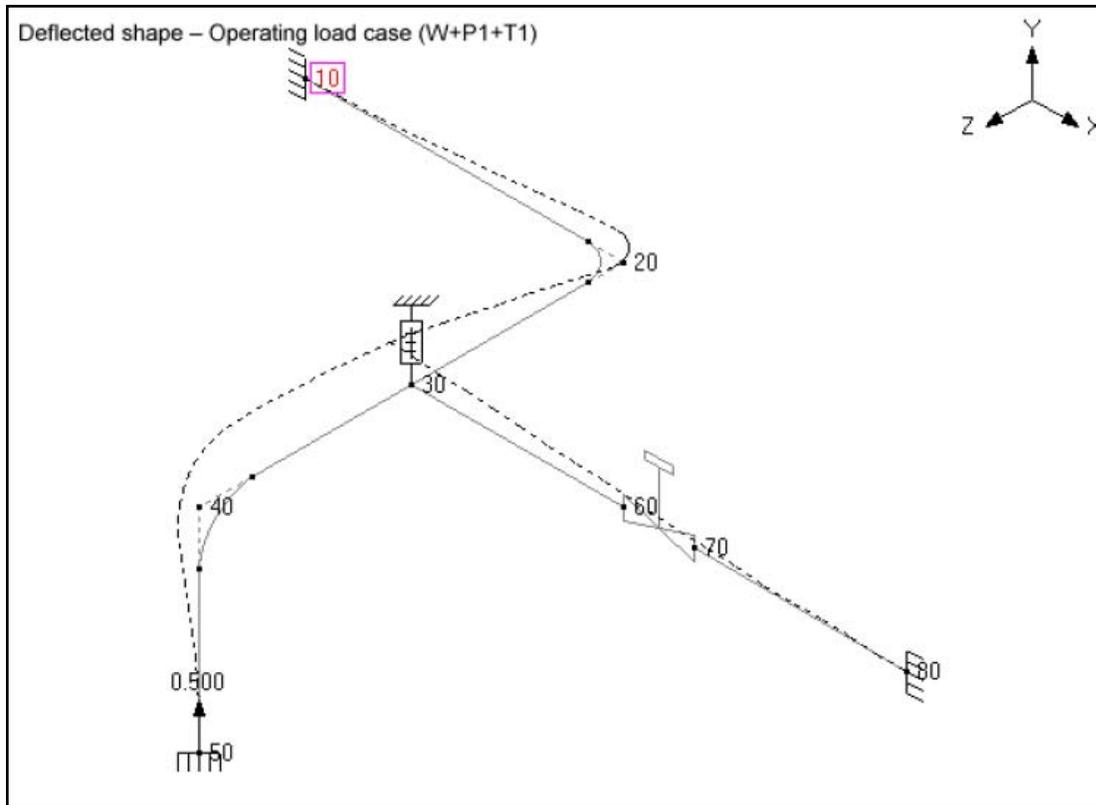
Results Window View Menu



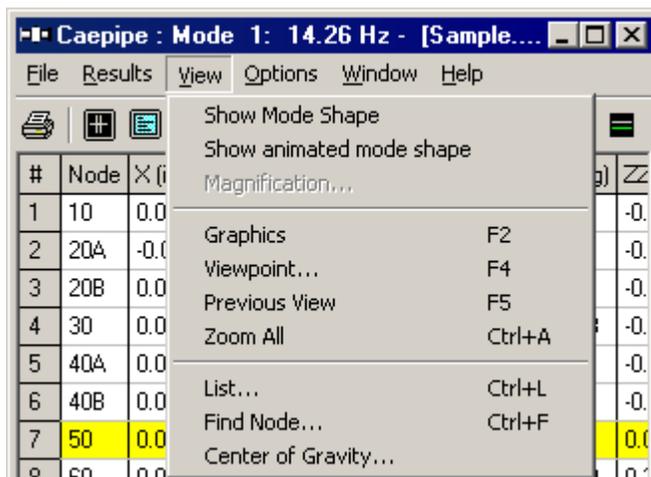
For Displacements results display, two commands to show and animate the deflected shape are shown. The remaining commands are the same as before.



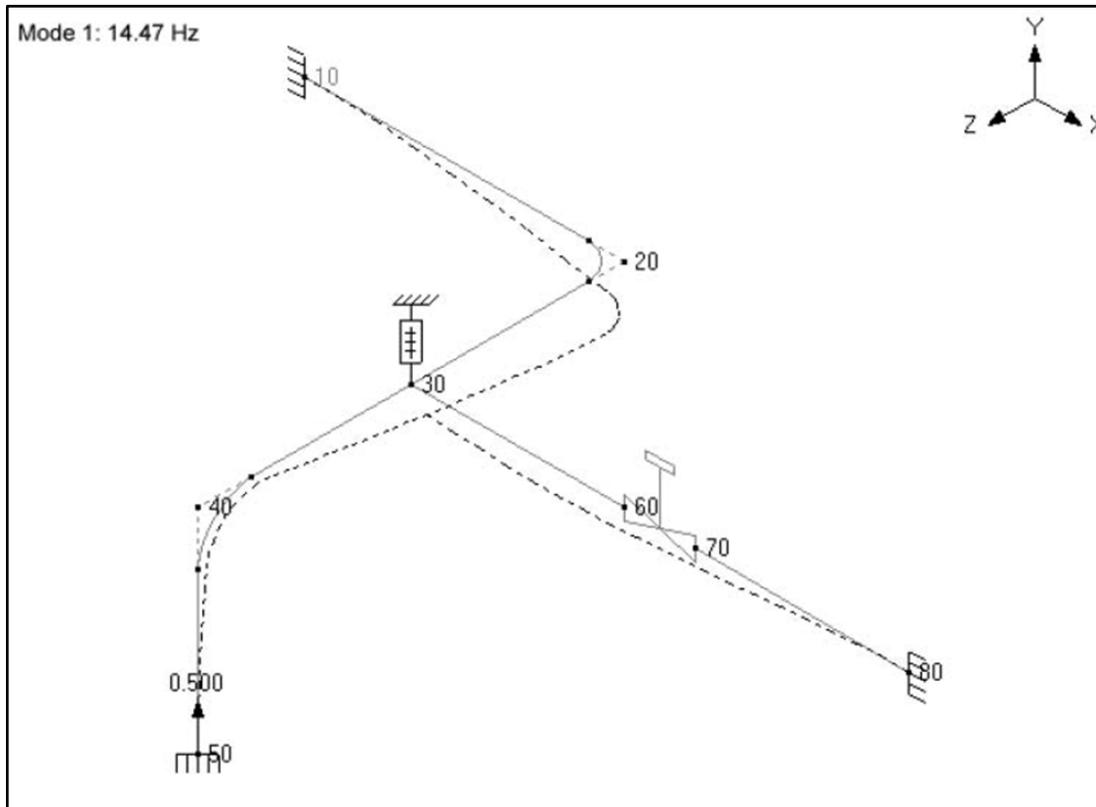
Results Window View Menu



For Frequencies and Mode Shapes results display, two commands to show and animate the mode shape are shown. The remaining commands are the same as before.

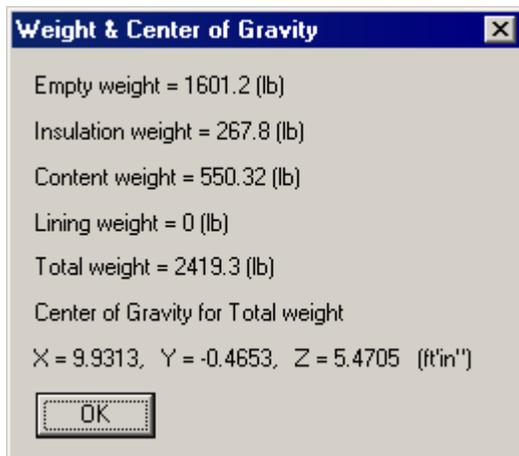


Results Window View Menu



Center of Gravity

This command displays the center of gravity and total weight of the piping system as modeled. This information can be useful when you are comparing the weight of this model with total weight obtained from another source.

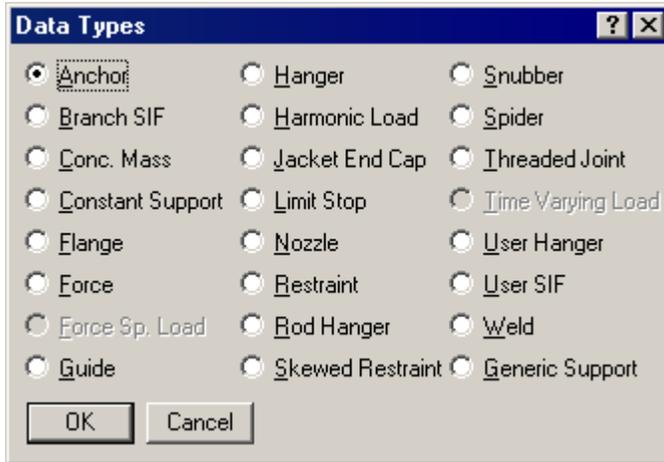


The Total weight is the sum of empty, insulation, content and lining weights.

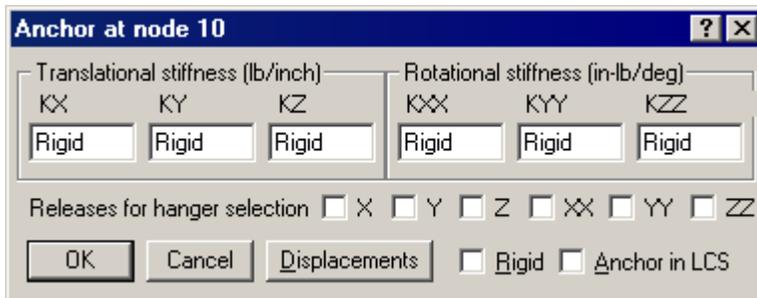
Reference

Anchor

An Anchor is a support that restrains the movement at a node in the three translational and the three rotational directions (each direction is a degree of freedom). In a physical piping system, this node may be on an anchor block or a foundation, or a location where the piping system ties into a wall or a large piece of equipment like a pump.



An Anchor is input by typing “A” in the Data column or selecting Anchor from the Data Types dialog. A rigid anchor is entered (i.e., an anchor with rigid stiffnesses in all six directions) by default. To change the default settings, edit the anchor by double-clicking on the anchor or pressing Ctrl+D in the row that has the anchor. An Anchor dialog is shown next.



Uncheck the “Rigid” checkbox to make the anchor non-rigid (i.e., a flexible anchor), and enter numerical values for stiffnesses for the six degrees of freedom. An anchor can be oriented along its local coordinate system (LCS) by checking “Anchor in LCS”, which will be useful if you have to input anchor movements in LCS.

Stiffness

The stiffness for a direction may be rigid (specified by typing the letter “r” in the stiffness field), or some value or be left blank. If it is left blank, there is no stiffness in that direction, i.e., the pipe is free to move in that direction. Internally, the rigid stiffness value is set to 1×10^{12} (lb./inch) for translational stiffness and 1×10^{12} (in.-lb./radian) for rotational stiffness.

Releases for Hanger Selection

These selections apply only during the automatic selection of a hanger by CAEPIPE. They do not influence the anchor stiffness in the released direction (if you checked any of the Release checkboxes). An Anchor by default acts in the six directions. You may release any combination of directions during (the automatic) hanger design. This feature is useful when

Anchor

hangers are used around equipment, where you want the hangers to take most of the weight load and thus reduce the load acting on the nearby equipment. CAEPIPE, during hot load calculation (preliminary sustained load case) in hanger design, releases the anchors (if you selected any of the Release checkboxes) so that the loads are taken by the hangers rather than by the anchors (which are modeling the equipment).

After the hot load calculation, CAEPIPE restores the original values to the released anchors before continuing analysis. Release anchors when they are (typically) within four (4) pipe diameters away from the hangers.

You may release any combination of translations or rotations. Typically, either the vertical translation or all translations and rotations are released. To release the Anchor in a particular direction, uncheck the corresponding checkbox.

Displacements

The Specified Displacements for Anchor dialog is shown.

Load	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
T1		0.5				
T2				0.1	0.1	0.1
T3						
T4						
T5						
T6						
T7						
T8						
T9						
T10						
Seismic	0.2		0.2			
Settlement		-0.25				

OK Cancel Displacements in LCS

Three types of translations and/or rotations at an anchor in the global X, Y and Z directions may be specified:

1. Thermal (10 displacements can be specified, one each for thermal loads T1 through T10). Applied only to the Expansion and Operating load cases.
2. Seismic (available for B31.1, ASME Section III Class 2, RCC-M and EN 13480 codes only). Solved as a separate internal load case and added absolutely to static seismic and response spectrum load cases.
3. Settlement (available under ASME Section III Class 2, RCC-M and EN 13480 codes only). Applied to a separate load case called Settlement.

Anchor

You may specify a displacement only if you *also* specify a corresponding non-zero or rigid stiffness in that direction, i.e., the corresponding stiffness should not be left blank.

Check “Displacements in LCS” if you want to enter anchor movements in the local coordinate system. These local movements are transformed into the global coordinate system and displayed in results.

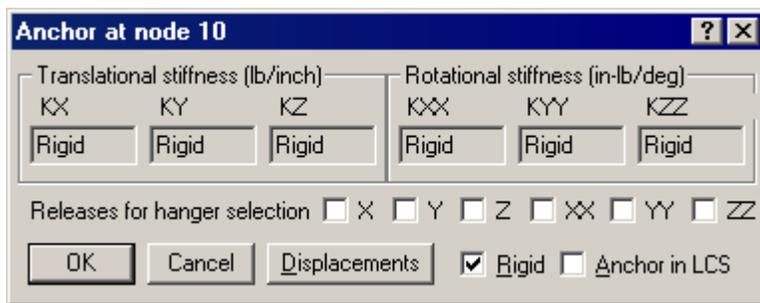
Settlement

For certain piping codes (ASME Section III Class 2, RCC-M and EN13480), an anchor settlement, which is a single non-repeated anchor movement (e.g., due to settlement of foundation), may be specified. This is applied to the Settlement load case. For those codes that do not have a provision for settlement (like B31.1), specify the anchor settlement as a thermal displacement (which tends to be a conservative approach).

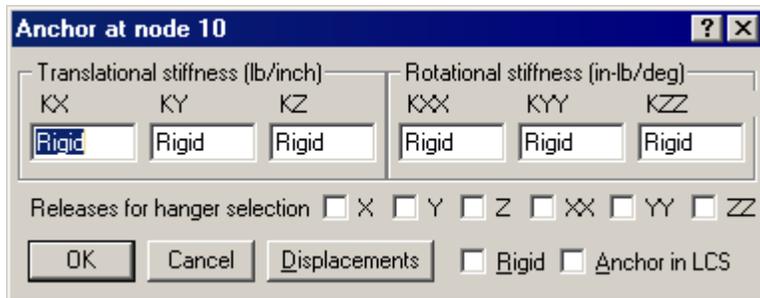
Example 1: Flexible Anchor

Nodes on most large equipment are modeled as rigid anchors. If you need to specify a non-rigid (i.e., flexible) anchor (for example at a nozzle to include vessel flexibility), you can input those stiffnesses by editing the anchor.

Double click on the anchor to show the anchor dialog.



By default the anchor has all stiffnesses rigid, no releases for hanger selection and no specified displacements. The stiffness fields are *grayed*, i.e., non-editable and the Rigid checkbox is checked. Click on the Rigid checkbox to uncheck it. The stiffness fields now become editable.



Type in the required stiffness values and press Enter or click on OK. The anchor definition shown in the next figure is now modified to be a flexible anchor.

Anchor

Anchor at node 10

Translational stiffness (lb/inch)			Rotational stiffness (in-lb/deg)		
KX	KY	KZ	KXX	KYY	KZZ
Rigid	0	1.200E+5	Rigid	50000	Rigid

Releases for hanger selection X Y Z XX YY ZZ

OK Cancel **Displacements** Rigid Anchor in LCS

Example 2: Rigid Vertical Support with Foundation Settlement

Assume that you need to model a vertical support on a foundation that has settled using ASME Section III Class 2 (1980) code for code compliance.

Vertical settlement ($-Y$) = 6 inches.

First, set the piping code to ASME Section III, which has a provision for Settlement load case, using the menu Options > Analysis > Code in the Layout window.

Next, create a rigid vertical support at the required node. Press “a” in the Data field to input a default anchor.

Next, edit the anchor so that it acts as a vertical support only, by modifying the stiffnesses similar to Example 1 so that only a Rigid stiffness KY in the Y direction remains.

Anchor at node 10

Translational stiffness (lb/inch)			Rotational stiffness (in-lb/deg)		
KX	KY	KZ	KXX	KYY	KZZ
	Rigid				

Releases for hanger selection X Y Z XX YY ZZ

OK Cancel **Displacements** Rigid Anchor in LCS

Now the anchor is modified to act as a Vertical 2-way rigid support. Click on the Displacements button and type in -6 (inch) for Settlement under Y. You could also input thermal and seismic displacements if required.

Specified Displacements for Anchor at node 10

Load	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
T1						
Seismic						
Settlement		-6				

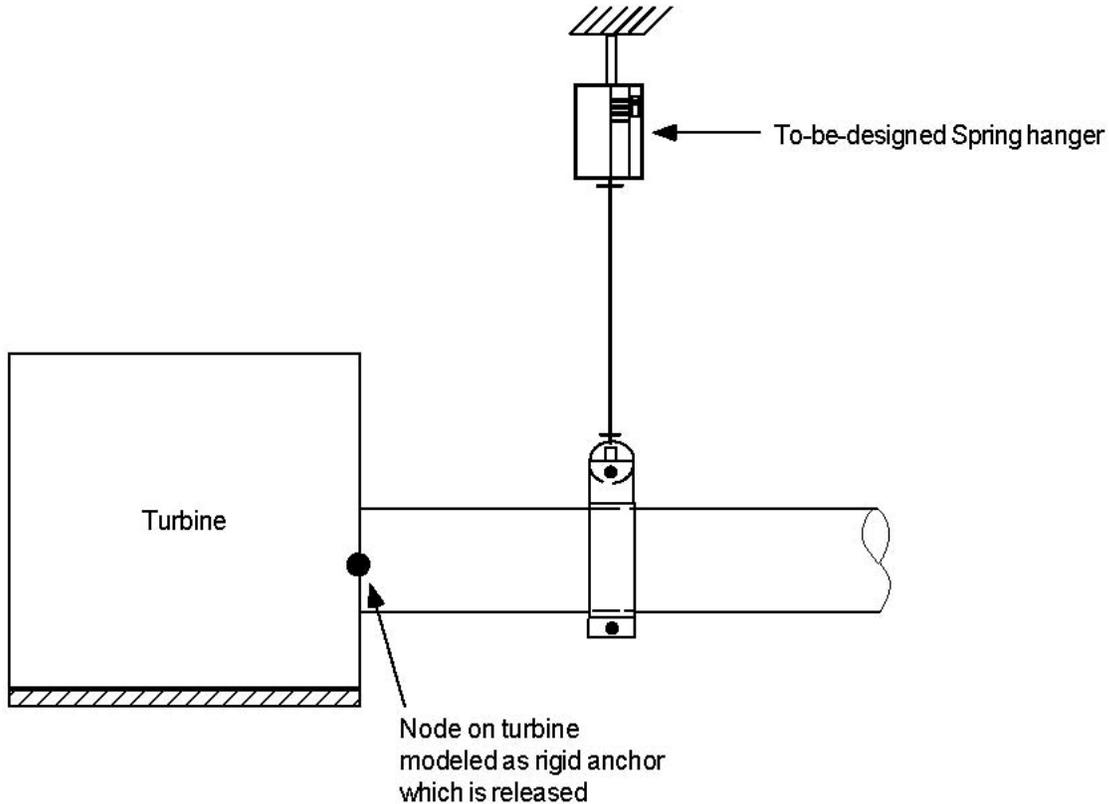
OK Cancel Displacements in LCS

The anchor is now modified to be a rigid vertical support with a specified settlement displacement.

Anchor

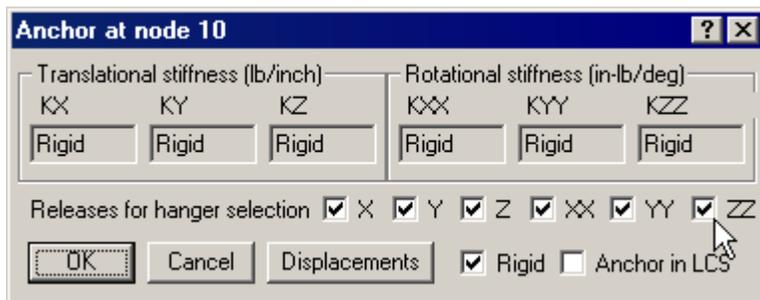
Check the box “Anchor in LCS” if you want to orient the anchor along a skewed line using its local coordinate system (LCS), which also aids you in specifying Displacements in LCS. Notice the naming convention changes (Kxx changes to kxx, X changes to x, and so on).

Example 3: Anchor Release during Hanger Design



CAEPIPE lets you model equipment nozzles as anchors. Assume that you had one on a turbine, as shown above, and that you have placed a hanger nearby. The main purpose of this hanger would be to take most of the load coming on the nearby turbine nozzle. To let CAEPIPE do that, you will have to release all directions of the Anchor so that the hanger will be designed to take most of the load.

First, enter an anchor for the node and then double click on it to edit it.



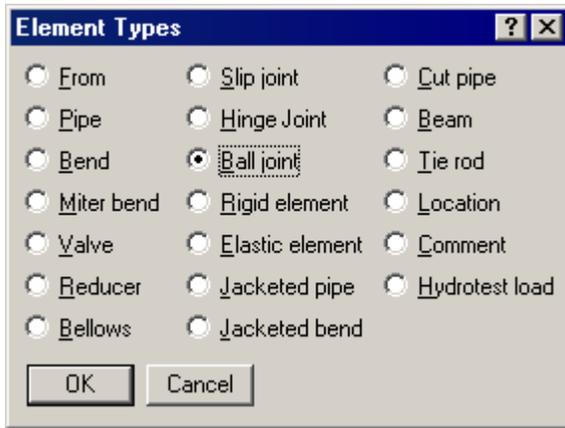
Click on the checkboxes for Releases for hanger selection in the required directions (X, Y, Z, XX, YY, ZZ). The anchor will be released in the specified directions during hanger design.

CAEPIPE restores the anchor to its original state (of no releases) after the preliminary hot load calculation during hanger design. See Hanger Design Procedure on p. 273.

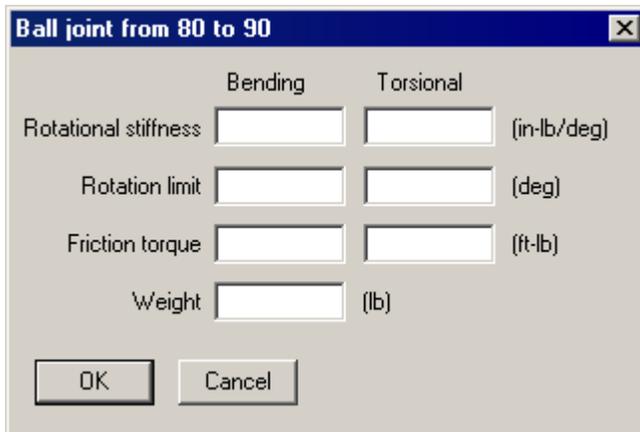
Ball Joint

A ball joint is a zero-length pipe element that allows rotations about the three orthogonal global axes (similar to a universal coupling joint in the rear axle of a motor vehicle) while still allowing the fluid to flow through it. If you do not want rotation in the torsional or (the two) bending directions, input “Rigid” for the respective stiffnesses. Since the ball joint is a zero-length element, the “From” and “To” nodes are coincident. Hence, you should leave the DX, DY and DZ fields in the Layout window blank (CAEPIPE will not let you enter a value).

A ball joint is input by typing “Ba” or “Ball” in the Type column or selecting Ball joint from the Element Types dialog.



The Ball joint dialog is shown.



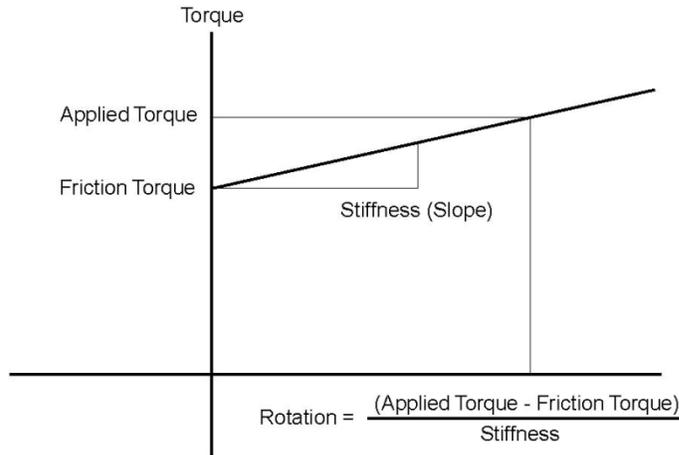
The rotational stiffnesses, rotation limits and the friction torques are specified independently in the bending and torsional directions. The torsional direction (local x) is determined by the preceding element’s local x. If a preceding element is unavailable, the following element is used to determine the torsional direction. The bending directions (local y and z) are orthogonal to the torsional direction (local x). Bending friction is determined by a resultant of torques in local y and z directions. Similarly, bending rotation is determined by a resultant of rotations in local y and z directions.

The stiffness, rotation limit and friction torque values are available from the manufacturer of the ball joint or from their test results. Otherwise, you must use engineering judgment.

The stiffness values may be left blank, in which case CAEPIPE uses a very small value (1 in.-lb./deg) internally to avoid dividing by zero during internal computation.

Ball Joint

A rotation limit of zero (0.0) means that the ball joint cannot rotate (i.e., it is rigid) in that direction. A rotation limit of “None” or Blank means that rotation is not limited to a finite value.



When the applied torque is less than the friction torque, there is no rotation. When the applied torque exceeds the friction torque, rotation is calculated as shown above. When rotation limit is reached, there is no further rotation irrespective of the applied torque.

When the option “Use friction in dynamic analysis” is selected, for modal analysis, CAEPIPE uses three different stiffnesses for a ball joint depending on the magnitude of the applied moment/torque in comparison to the user-specified friction torque. They are as follows:

- Case 1: When the applied moment at the ball joint (for the first operating load case when a piping code is selected or for the static case when “Piping code=None” is selected) **is less than** the friction torque, the friction is not overcome and the ball joint stiffness is internally set to a large number (in effect, infinity).
- Case 2: When the applied moment **is more than** the friction torque, the friction is overcome and the ball joint starts rotating (with the user-specified rotational stiffness being applied). This rotational deformation takes place until the user-specified rotational limit is reached.

So, from the time friction is overcome to the time when the rotational limit is reached, CAEPIPE internally sets an “equivalent stiffness” for the ball joint as given below.

If K_b is the user-specified stiffness for the ball joint,

$$\text{Rotation} = (\text{applied moment} - \text{friction torque}) / K_b \dots\dots\dots (1)$$

The “equivalent stiffness” chosen by CAEPIPE is the slope of the straight line from origin to the point (rotation, applied moment) in the figure given above. In other words,

$$K_{be} = (\text{applied moment}/\text{rotation}) \dots\dots\dots (2)$$

Combining (1) and (2), you get

$$K_{be} = (\text{applied moment}) \times K_b / (\text{applied moment} - \text{friction torque}) \dots\dots\dots (3)$$

Ball Joint

Case 3: When the actual rotation reaches the user-specified rotational limit for the first operating load case/static case, the ball joint stiffness is again set to "infinity."

CAEPIPE uses "equivalent stiffness" for ball joints as described under Case 2 above, when the friction torque is exceeded and the computed rotation is yet to reach the specified rotational limit, *as long as* the option "Use friction in dynamic analysis" is selected. When this option is NOT selected, CAEPIPE ignores the nonlinearities of the ball joint (namely, friction torque and rotational limit) and uses only the user-specified stiffnesses. That is why, despite very small moments, large rotations may be computed (i.e., a check against user-specified rotational limit is not performed).

See the topic Expansion Joints for examples.

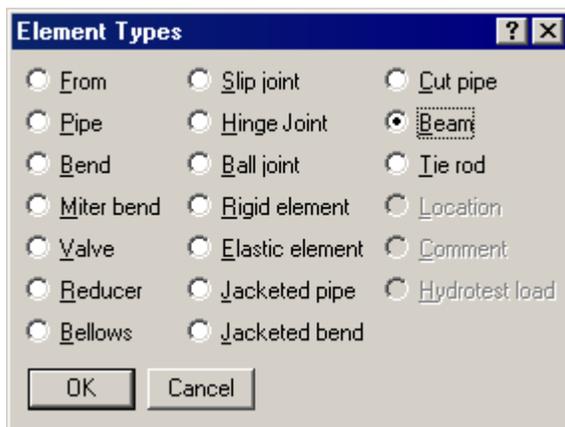
Beam

You can model elaborate structural systems inside CAEPIPE alongside the piping to be supported. In simple situations, if the structure is stiffer than the piping is, you may not need to model the structure at all but simply treat it as rigid (for example: input Rigid for Stiffness in Limit stop when simulating a stiff beam using a Limit stop). But, in cases where you need to account for structural flexibility, use the Beam element to model structural support systems alongside piping systems.

The material, section and load for a beam are different from those for a pipe. Just as you would define a material/section/load for a pipe, so too should you define a separate material/section/load for a beam. Look for Beam Material, Beam Section, and Beam Load (under Misc menu).

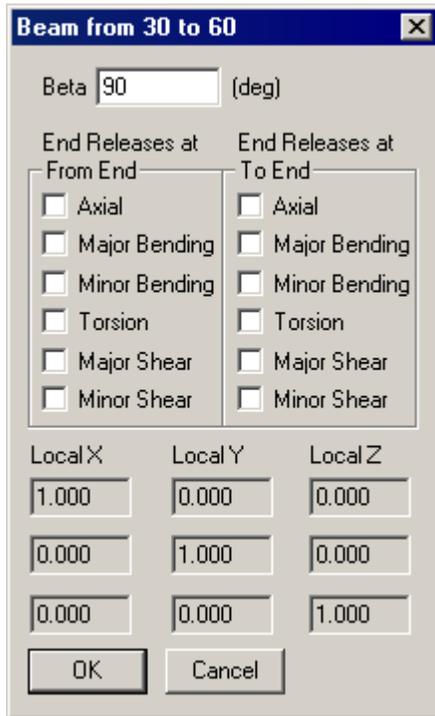
Upon analysis, CAEPIPE reports forces and moments for beam elements.

A beam is input by typing “bea” in the Type column or by selecting “Beam” from the Element type dialog.



Beam

The Beam dialog is shown.



Beta angle

Beta angle is used to define the orientation of a beam's local axes. See Beam orientation later in this section.

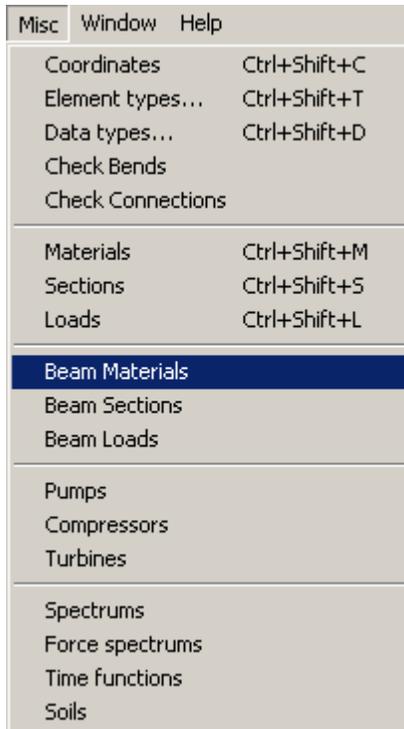
Beam End Releases

Each end of the beam (From and To ends) can be released to simulate the type of structural support you want to model. That is, you can use a combination of releases to specify whether a beam end is fixed, pinned, etc.

Beam material

Before you input a beam element, you must define a beam material, section and load. Select Beam materials from the Miscellaneous (Misc) menu in the Layout or List window.

Beam



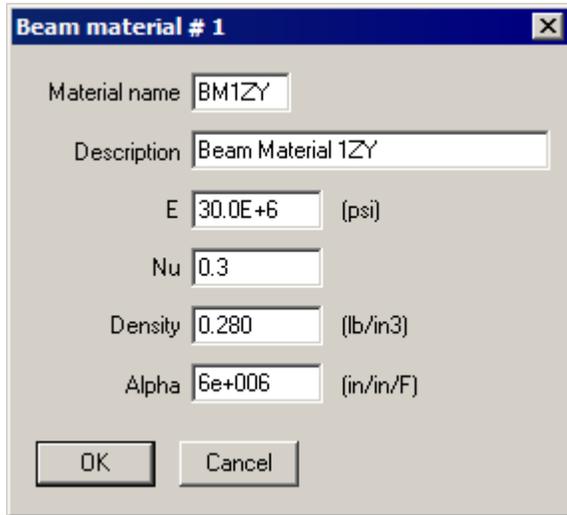
A beam material list window is shown. Double click on an empty row to input a new beam material.

The image shows a dialog window titled 'Caepipe : Beam materials (1) - [BigMod...]' with a menu bar (File, Edit, View, Options, Misc, Window, Help) and a toolbar with icons for grid, list, camera, search, and navigation. Below the toolbar is a table with the following data:

#	Name	Description	E (psi)	Nu	Density (lb/in ³)	Alpha (in/in/F)
1	BM1ZY	Beam Material	30.0E+6	0.3	0.280	6e+006
2						

A dialog for inputting beam material is shown.

Beam



The image shows a dialog box titled "Beam material # 1" with a close button (X) in the top right corner. It contains several input fields for material properties:

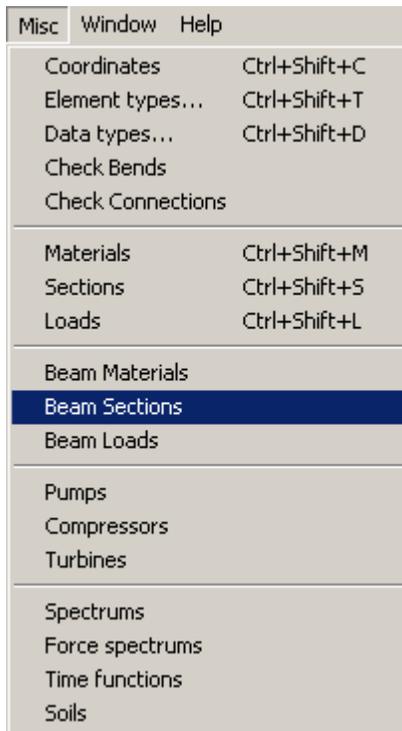
- Material name:
- Description:
- E: (psi)
- Nu:
- Density: (lb/in³)
- Alpha: (in/in/F)

At the bottom, there are two buttons: "OK" and "Cancel".

The material name can be up to three alpha-numeric characters to identify the beam material. A more complete description can be entered under Description. Enter modulus of elasticity, Poisson's ratio (ν), density of the material and mean coefficient of thermal expansion between T_{ref} and $T1/T2/T3$ in beam load.

Beam section

Select Beam Sections from the Misc(ellaneous) menu in the Layout or List window.



A list of beam sections is shown. Double click on an empty row to input a new beam section.

Beam

#	Name	Description	Axial area (in2)	Moment of inertia		Torsional inertia (in4)	Shear area		Depth (inch)	Width (inch)
				Major (in4)	Minor (in4)		Major (in2)	Minor (in2)		
1	BM1ZY	W 8x28	8.25	98	21.7	0.54				
2										

A dialog to input beam sections is shown.

Section name

Description

Axial area (in2)

Major moment of inertia (in4)

Minor moment of inertia (in4)

Torsional inertia (in4)

Major shear area (in2)

Minor shear area (in2)

Depth (inch)

Width (inch)

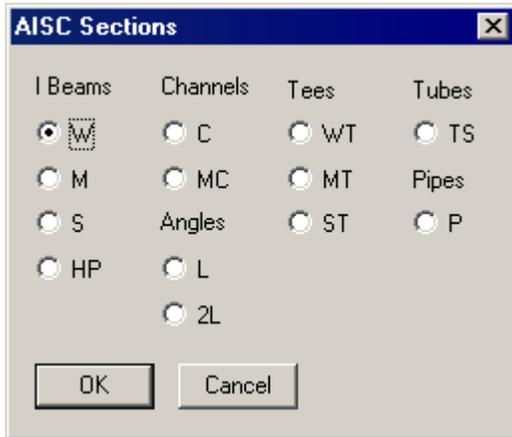
You can either input the data yourself or click on the Library button for a listing of different AISC I-beams, channels, tees, etc., that are built into CAEPIPE. Be sure to verify the properties that are shown in the fields after you select a section from the library.

The name can be up to three alpha-numeric characters to identify the beam section. A more complete description can be entered in the Description field.

The axial area, major and minor moments of inertia must be input. Input of torsional inertia is optional. If it is not input, it defaults to the sum of major and minor moments of inertia. Input of shear areas is optional. If it is not input, shear deflection is not included. Input of depth and width are optional. Presently, they are used only for rendered plots of the beam.

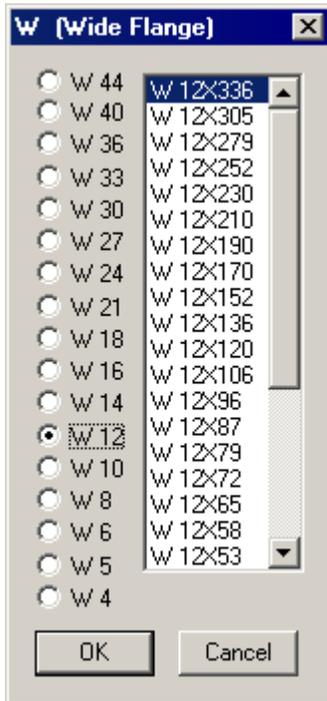
Dialogs for selecting a beam section from the AISC library are shown below:

Beam



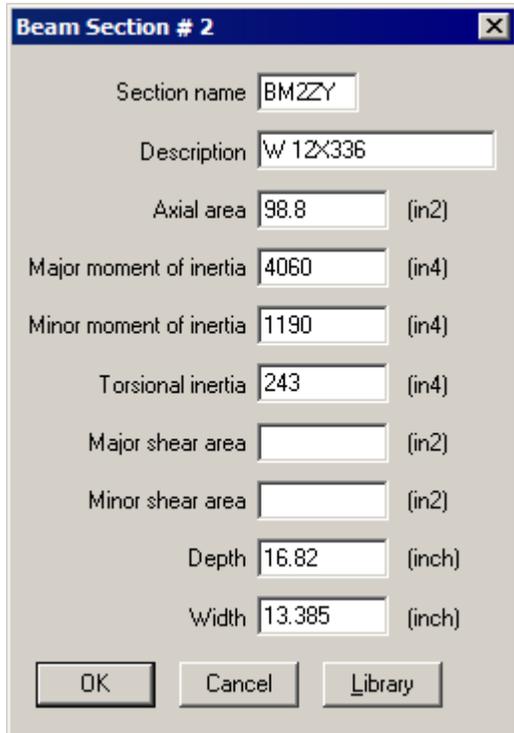
The type of the beam section (e.g., I beam, W (Wide Flange)) is selected from this dialog.

Another dialog which shows various available sections for the particular beam section type is then shown.



After selecting the section, click on OK and the section properties will be entered in the Beam section dialog.

Beam

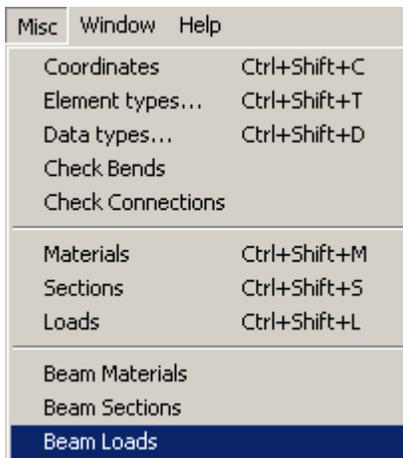


The image shows a dialog box titled "Beam Section # 2" with a close button (X) in the top right corner. The dialog contains several input fields for defining a beam section. The fields are: Section name (BMZZY), Description (W 12X336), Axial area (98.8 in²), Major moment of inertia (4060 in⁴), Minor moment of inertia (1190 in⁴), Torsional inertia (243 in⁴), Major shear area (empty in²), Minor shear area (empty in²), Depth (16.82 inch), and Width (13.385 inch). At the bottom, there are three buttons: OK, Cancel, and Library.

Section name	BMZZY	
Description	W 12X336	
Axial area	98.8	(in ²)
Major moment of inertia	4060	(in ⁴)
Minor moment of inertia	1190	(in ⁴)
Torsional inertia	243	(in ⁴)
Major shear area		(in ²)
Minor shear area		(in ²)
Depth	16.82	(inch)
Width	13.385	(inch)

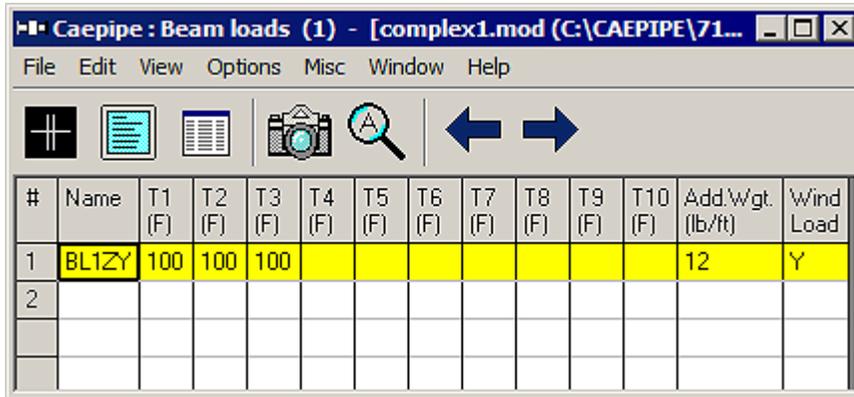
Beam load

Select Beam Loads from the Misc menu in the Layout or List window.



A list of beam loads is shown.

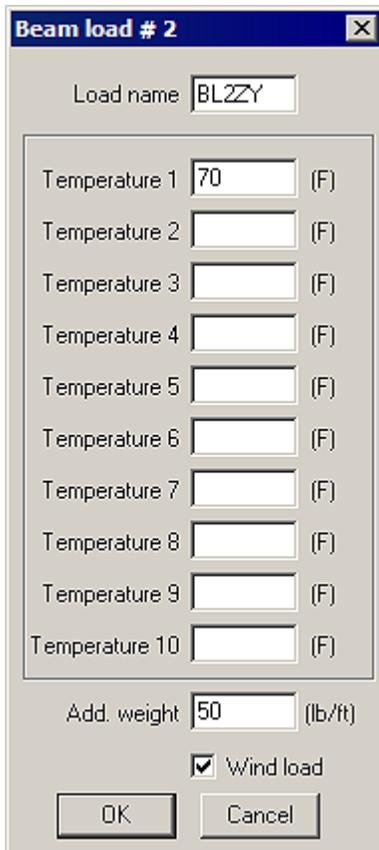
Beam



The screenshot shows a software window titled "Caepipe : Beam loads (1) - [complex1.mod (C:\CAEPIPE\71...". The window contains a menu bar (File, Edit, View, Options, Misc, Window, Help) and a toolbar with icons for a grid, list, camera, magnifying glass, and navigation arrows. Below the toolbar is a table with the following data:

#	Name	T1 (F)	T2 (F)	T3 (F)	T4 (F)	T5 (F)	T6 (F)	T7 (F)	T8 (F)	T9 (F)	T10 (F)	Add.Wgt. (lb/ft)	Wind Load
1	BL1ZY	100	100	100								12	Y
2													

Double click on an empty row to input a new beam load through the beam load dialog or start typing into the fields.



The screenshot shows a dialog box titled "Beam load # 2". It contains the following fields and controls:

- Load name:
- Temperature 1: (F)
- Temperature 2: (F)
- Temperature 3: (F)
- Temperature 4: (F)
- Temperature 5: (F)
- Temperature 6: (F)
- Temperature 7: (F)
- Temperature 8: (F)
- Temperature 9: (F)
- Temperature 10: (F)
- Add. weight: (lb/ft)
- Wind load
- OK button
- Cancel button

The Load name can be up to five alpha-numeric characters to identify the beam load. You can enter up to 10 temperatures depending on the preset number of thermal loads. The additional weight is a uniform weight per unit length added to the weight of the beam. This could for example be used to add snow load to the beam. Wind load may or may not be applied to the beam element by using the check box for Wind load in the dialog or typing "Y" or "N" for Wind load in the List window.

Beam orientation

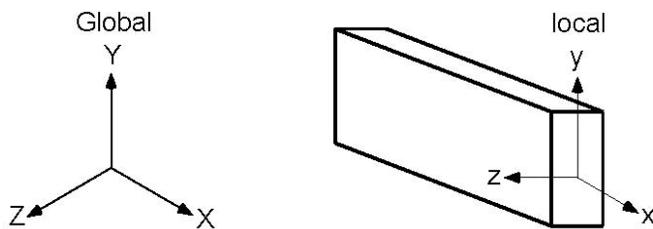
The Beam orientation is determined by the locations of the “From” and “To” nodes and the beta angle of the beam element. The local x-axis of the beam is always from the “From” node to the “To” node. The reference orientation corresponds to $\beta = 0.0$.

A nonzero beta angle (measured from the reference position) rotates the local y- and z-axes of the beam about the local x-axis of the beam in the counter clockwise direction.

The local coordinate system for beams can be displayed for each beam element through the List window (Ctrl+L, select Beams, menu View > Show LCS [for Local Coordinate System]).

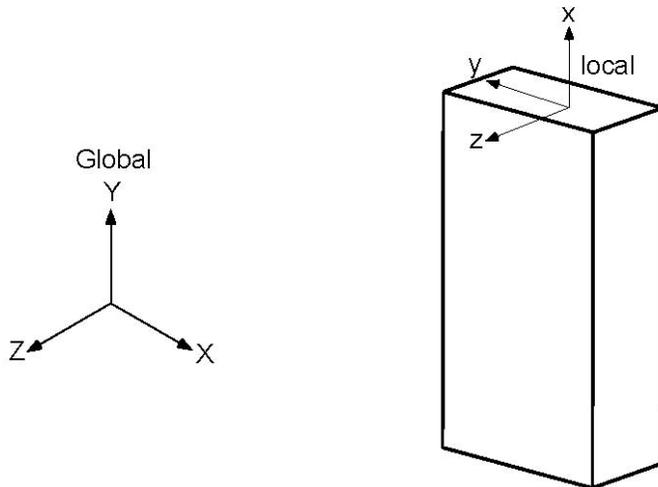
Global vertical axis is Y

Beam is not Vertical



The local y-axis of the beam lies in the local x - global Y plane (i.e., vertical plane) and is in the same positive direction as the global Y axis. The local z-axis is the cross product of the local x and y-axes. Major bending plane is local x-y, that is, $I_{zz} = \text{Major moment of inertia}$ and $I_{yy} = \text{Minor moment of inertia}$.

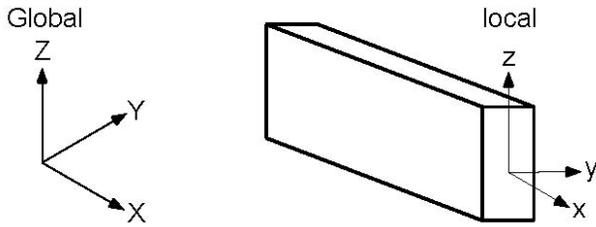
Beam is Vertical



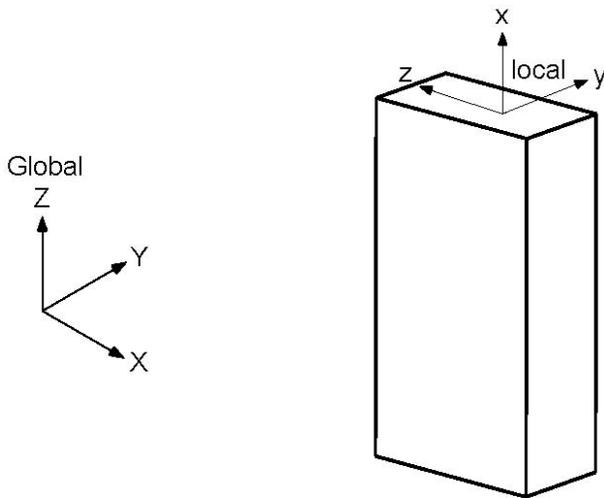
The local z-axis of the beam is in the global Z direction. The local y-axis is in the global - X direction. Major bending plane is x-y, i.e., $I_{zz} = \text{Major moment of inertia}$ and $I_{yy} = \text{Minor moment of inertia}$.

Global vertical axis is Z

Beam is not Vertical



The local z-axis of the beam lies in the local x - global Z plane (i.e., vertical plane) and is in the same positive direction as the global Z-axis. The local y-axis is the cross product of the local z and x-axes. Major bending plane is x-z, that is, I_{yy} = Major moment of inertia and I_{zz} = Minor moment of inertia.



The local y-axis of the beam is in the global Y direction. The local z-axis is in the global - X direction. Major bending plane is x-z, i.e., I_{yy} = Major moment of inertia and I_{zz} = Minor moment of inertia.

Beam

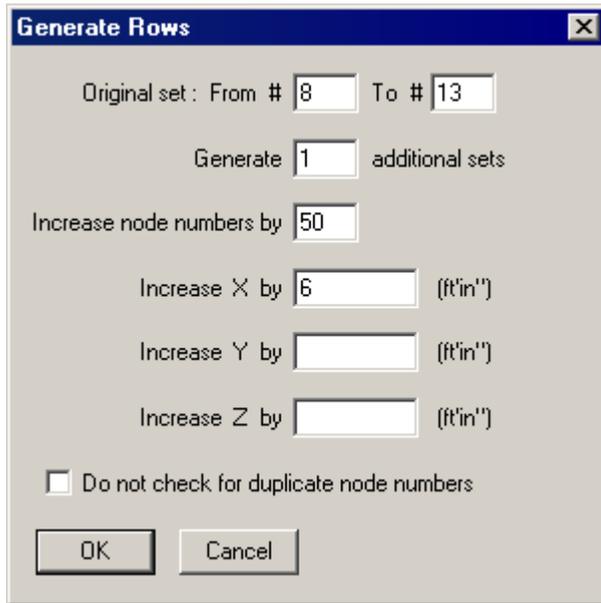
Example 1: Pipe Rack using Beams

Here, you see how to use a beam element to construct a pipe rack and connect the beam to the pipe so that CAEPIPE can account for the rack's flexibility. The procedure is simple. First, you need to create a beam material, section and load in addition to pipe material, section and load.

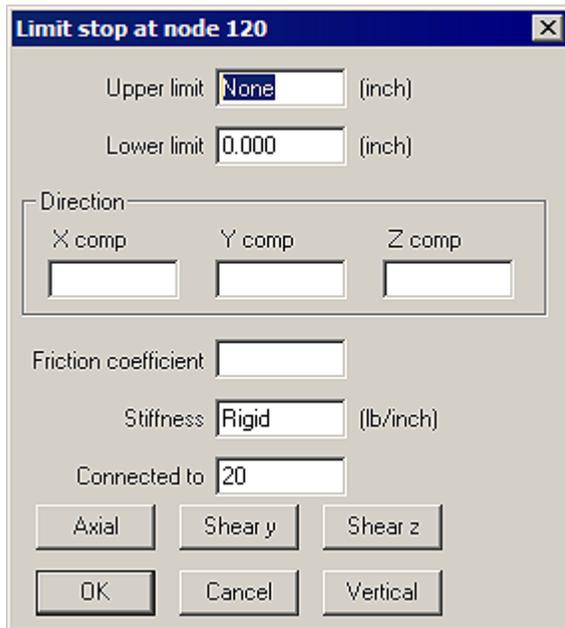
As the Layout window shows, model the piping (nodes 10 to 40) and the first beam support (nodes 100 to 140). Then, create the second beam support (nodes 150 to 190) using the Generate command (under Edit menu in the Layout window). Finally, connect piping at nodes 20 and 30 to beam nodes 120 and 170 using limit stops.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Pipe rack (using beam elements)								
2	Sample model illustrating structure-pipe interaction.								
3	Note limit stops connect the beams at 120/170 to pipes at 20/30								
4	10	From							Anchor
5	20		6'0"			1	1	1	
6	30		6'0"			1	1	1	
7	40		6'0"			1	1	1	
8	100	From	6'0"	-6'0"	3'0"				Anchor
9	110	Beam		5.3048		BM1	BM1	BM1	
10	Limit stop here restrains only downward movement.								
11	120	Beam			-3'0"	BM1	BM1	BM1	Limit stop
12	130	Beam			-3'0"	BM1	BM1	BM1	
13	140	Beam		-5.3048		BM1	BM1	BM1	Anchor
14	150	From	12'0"	-6'0"	3'0"				Anchor
15	160	Beam		5.3048		BM1	BM1	BM1	
16	Limit stops here and at row 20 simulate a U-bolt at node 30 (0.125"gap)								
17	170	Beam			-3'0"	BM1	BM1	BM1	Limit stop
18	180	Beam			-3'0"	BM1	BM1	BM1	
19	190	Beam		-5.3048		BM1	BM1	BM1	Anchor
20	170	Location							Limit stop
21									

The “Generate” dialog is shown below:

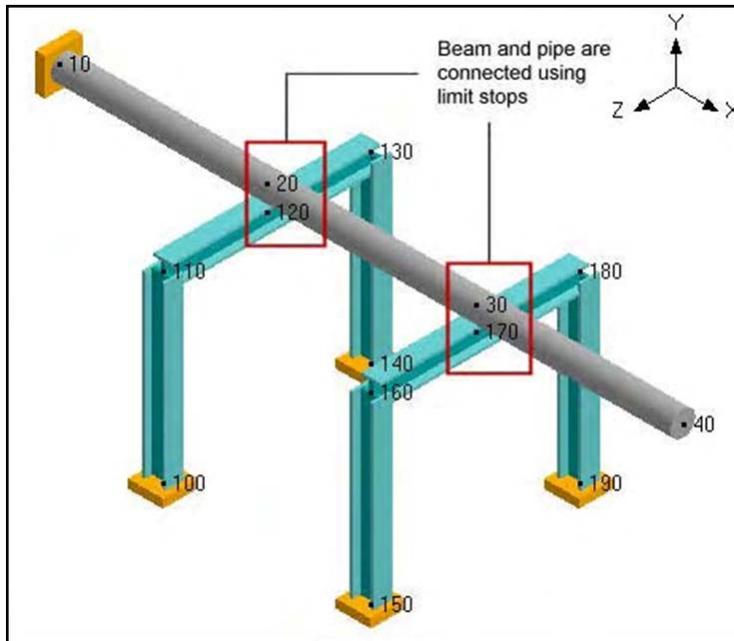


The “Limit Stop” dialog is shown below:



The graphics is shown below:

Beam



Example 2: Base Supported Bend using a Beam

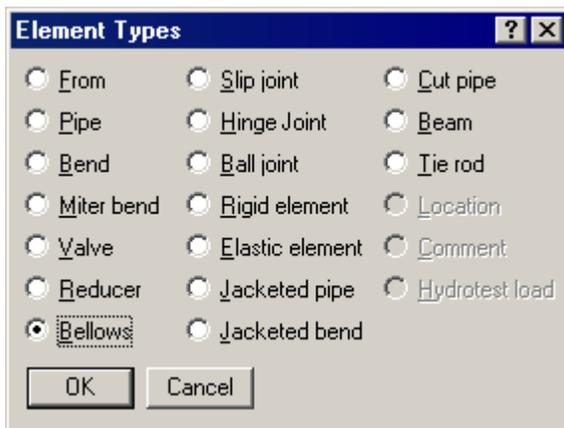
See Example 7 in the Bend section for modeling a base supported bend using a beam on page 170.

Bellows

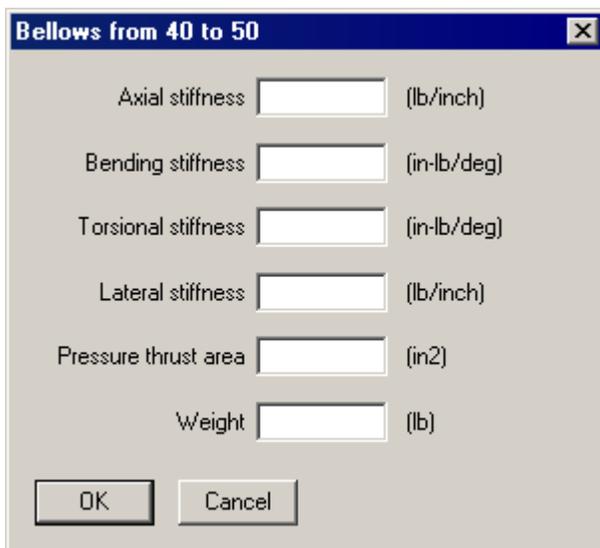
Bellows expansion joints are flexible elements included in high temperature piping systems to absorb primarily thermal movement. A Bellows contains one or more convolutions designed to withstand the internal pressure while still flexible enough to absorb the axial, lateral and bending deflections. Before use, you should note the critical pressure at which the bellows becomes unstable. The B31.1 piping code suggests that expansion joints may be employed only “when piping bends, loops, and offsets are not able to provide adequate flexibility.” (Para. 11.5, 2010).

Usually manufacturers of these expansion joints publish product catalogs that contain technical information about the joints you could use in your systems. The EJMA (Expansion Joint Manufacturers Association) also publishes a standards catalog with guidelines that “assist users, designers, and others in the selection and application of expansion joints for safe and reliable piping and vessel installation.”

A Bellows joint is input by typing “bel” in the Type column or selecting “Bellows” from the Element Types dialog.



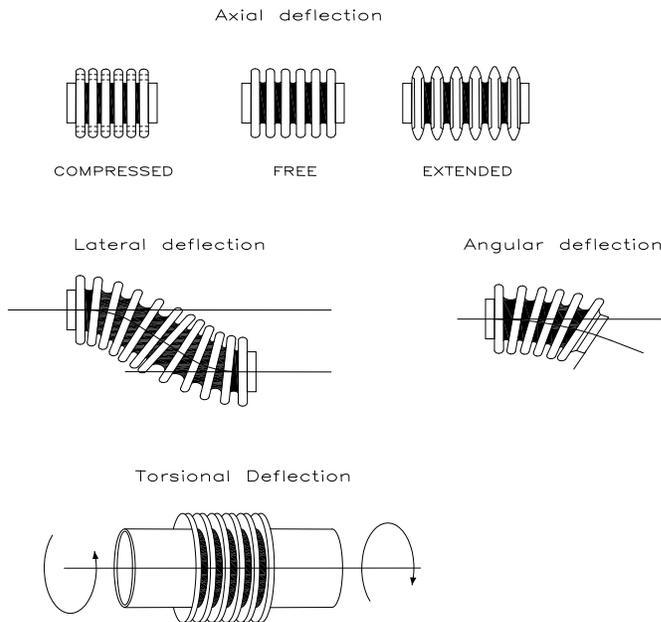
The Bellows dialog is shown.



Expansion joints are mainly modeled using the above shown four types of stiffnesses – axial, bending, torsional and lateral. The required stiffness values, pressure thrust area and weight should be taken from the manufacturer’s catalog.

Bellows

For a rigid stiffness (for example, torsional), enter “r” for Rigid; if highly flexible, enter 1 (lb./in.) as a minimum, to avoid dividing by zero during internal computation.



Axial: Refers to axial extension (as in cryogenic systems) or compress (as in high-temperature systems) axially along its centerline while in operation.

Bending (angular): Refers to the bellows bending about its center point on the centerline. Bending can be in any plane that passes through the centerline.

Lateral: Refers to the direction perpendicular to the centerline of the bellows. The two ends of the bellows remain parallel to each other while their centerlines are displaced causing an offset. This direction is also called transverse or parallel offset direction.

Torsional: Usually very stiff, refers to a twisting moment at one bellows end while the other end either relatively is stationary or twists in the other direction, about the bellows centerline.

The pressure thrust area will impose a thrust load of: (pressure \times thrust area), on both nodes of the bellows. Even if the bellows is tied, it is recommended that the pressure thrust area be input. The weight is the empty weight. CAEPIPE adds the weight of the contents, insulation and additional weight to the empty weight.

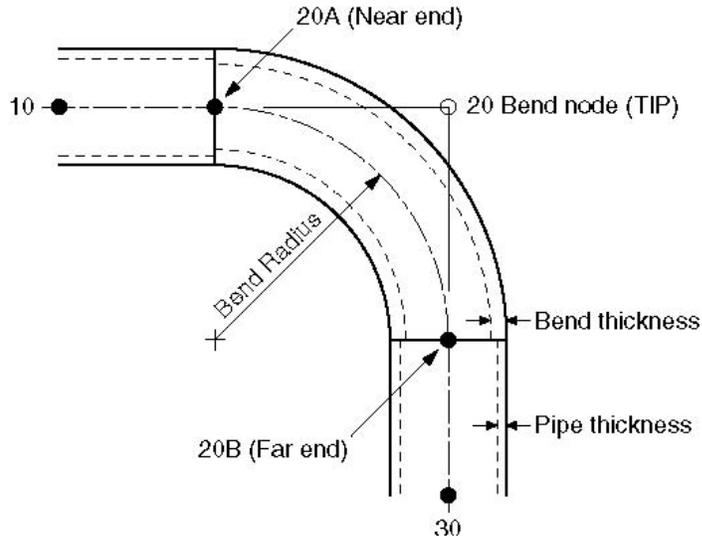
Pipe guides are needed adjacent to the bellows because of its inherent flexibility and the compressive loading on the adjacent pipes due to the pressure thrust of the joint. Moreover, proper guiding is necessary to direct thermal movement into the joint and prevent buckling of the line. Depending on the bellows behavior, you should place the first guide no farther than four pipe diameters from the joint. Place additional ones appropriately after studying the nearby deflections and loads.

Also, consider vessel and anchor movements, which may cause a misalignment at the joint.

See the topic on Expansion Joints for examples.

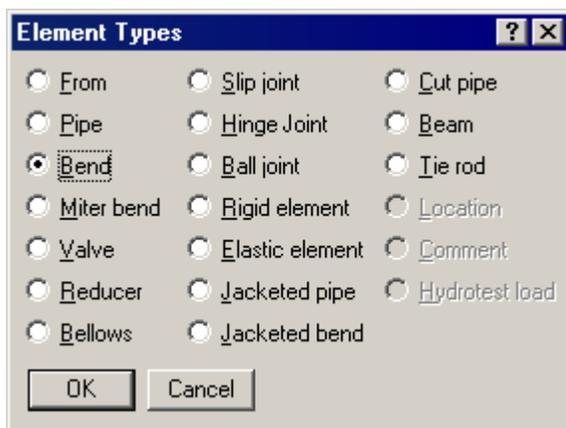
Bend

In CAEPIPE, the term Bend refers to all elbows and bends (custom-bent pipes). An elbow comes prefabricated with a standard bend radius (short or long radius) whereas a bend is custom- made from bending a straight pipe with a specified bend radius. Geometrically, a bend is a curved pipe segment which turns at an angle (typically 90° or 45°) from the direction of the run of the pipe. Some of the items associated with a bend are shown below.



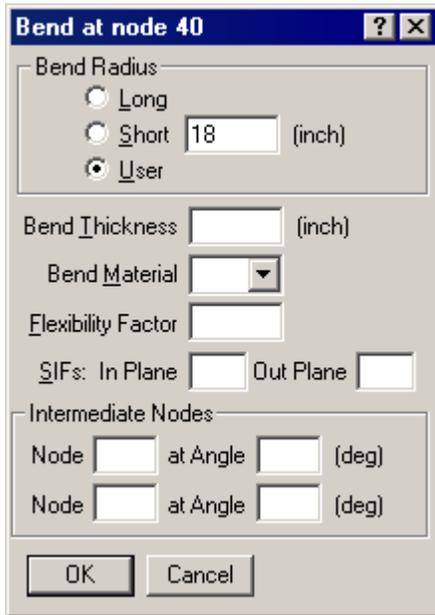
Node 20 is the Bend node, also referred to as the Tangent Intersection Point (TIP). As you can see from the figure, it is not physically located on the bend. Its only purpose is to define the bend. CAEPIPE automatically generates the end nodes of the curved portion of the bend (nodes 20A and 20B, called the near and far ends of the bend. The bend end nodes (20A and 20B in the figure) may be used to specify data items such as flanges, hangers, forces, etc.

A bend is input by typing “b” in the Type column or selecting “Bend” from the Element Types dialog.



Bend

If you need to modify an existing bend, double click on it or press Ctrl+T (Edit type) to bring up the Bend dialog.



Bend Radius

The radius of a bend (measured along the centerline of the bend) can be specified as Long, Short, or User (defined) by one of the radio buttons for Bend Radius. CAEPIPE has long and short radii built-in for standard ANSI, JIS, DIN and ISO pipe sizes. For nonstandard pipe sizes, Long radius is equal to 1.5 times the pipe OD and Short radius equal to the pipe OD.

Bend Thickness

Input the wall thickness of the bend if different from the preceding pipe's thickness. If specified, the Bend Thickness applies only to the curved portion of the bend (node 20A to node 20B in the figure above).

Bend Material

If the material of the bend is different from that of the adjoining pipe, select the Bend Material from the drop down combo box. The Bend Material, if specified, applies only to the curved portion of the bend (node 20A to node 20B in the figure above).

Flexibility Factor

This factor is automatically calculated for standard elbows according to the piping code chosen. If you have your own Flexibility Factor, enter it here instead of the piping code specified Flexibility Factor. A value of 2.0, for e.g., will mean that the bend is twice as flexible as a pipe of the same length.

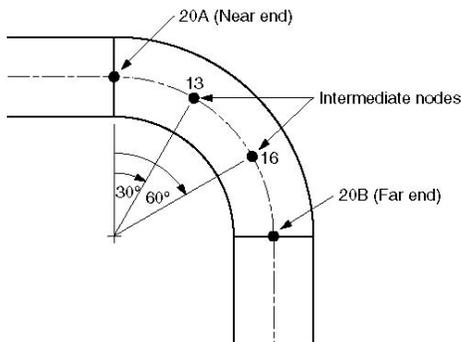
Bend

SIFs

These factors are automatically calculated for standard elbows according to the piping code chosen. If you have your own, specify them here (useful for FRP bends, for example), which will be used instead of the piping code specified SIFs. If User SIFs are also specified at bend nodes (A and/or B nodes), they will be used instead of the bend SIFs or code specified SIFs.

Intermediate Nodes

An intermediate node, located in between the ends of the bend, may be required in some situations to specify data items such as flanges, hangers, forces, etc. You can create an intermediate node by giving a (new) node number and an angle for it, which is measured from the near end of the bend (node 20A in figure). Up to two such nodes may be input. Note that the intermediate nodes 13 and 16 shown below are at angles of 30° and 60° respectively from node 20A (near end). The intermediate nodes can be used for specifying data items such as flanges, hangers, forces, etc.



Bend Examples

Some examples follow. They illustrate some common modeling requirements.

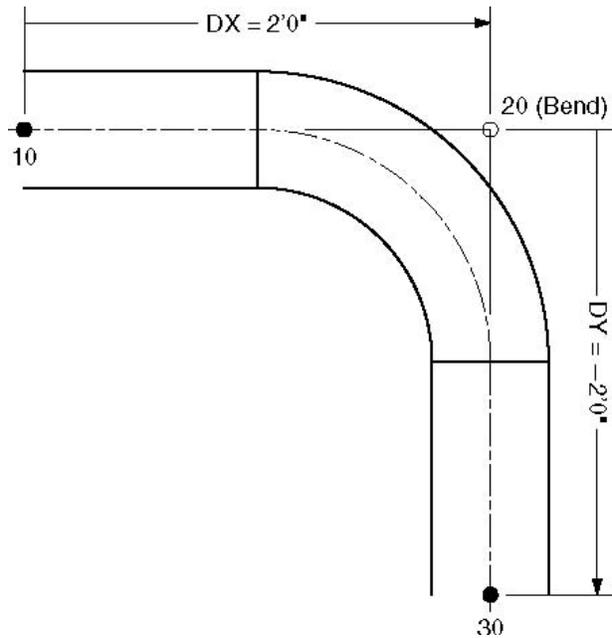
- Example 1: 90° Bend
- Example 2: 45° Bend
- Example 3: 180° Bend
- Example 4: Flanged Bend
- Example 5: Reducing Bend
- Example 6: Bend Supported by a Hanger
- Example 7: Base Supported Bend

To simplify the discussion of bend modeling, it is assumed that the material, section (8" STD), load and the first node (10) are already defined. It is also assumed that the bend has long radius (12") and the cursor is placed in row #3.

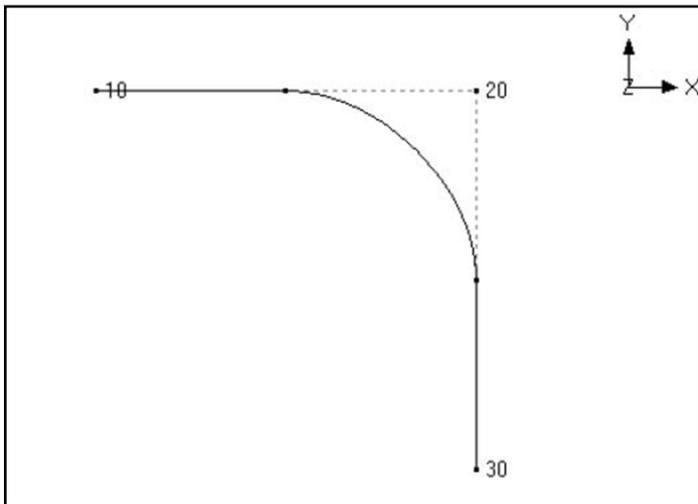
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Bend examples								
2	10	From							
3									

Bend

Example 1: 90° Bend

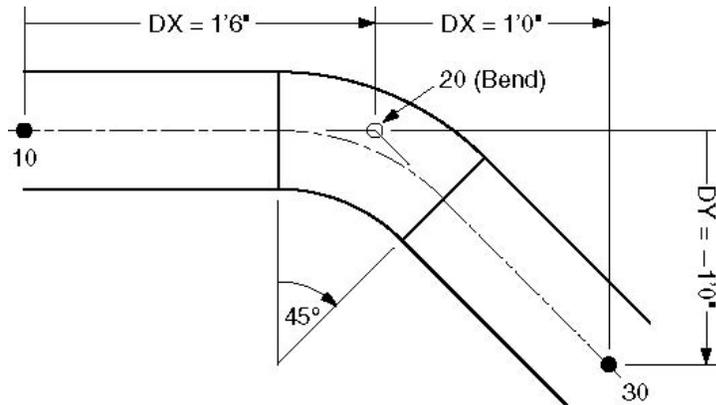


- ▶ Press Tab in row #3. Node 20 will be automatically assigned and the cursor will move to the Type column, type "B" (for Bend), Tab to DX, type 2. Enter material, section, load and press Enter. The cursor moves to the next row (#4).
- ▶ Tab to the DY column. The next Node 30 is automatically assigned. In DY column, type -2 and press Enter. This completes the bend input.

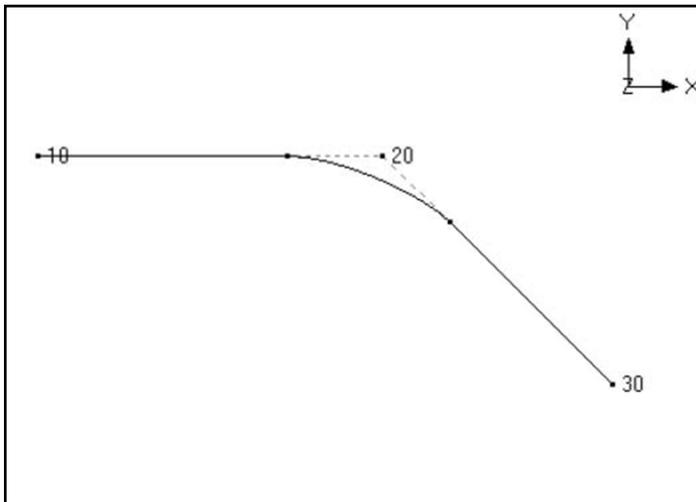


Bend

Example 2: 45° Bend



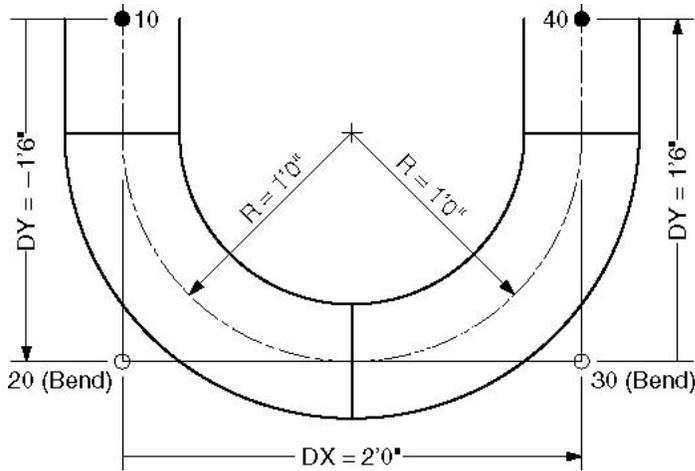
- ▶ Press Tab in row #3. Node 20 will be automatically assigned and the cursor will move to the Type column. type “B” (for Bend), Tab to DX, type 1’6”. Enter material, section, load and press Enter. The cursor moves to the next row (#4).
- ▶ Tab to the DX column. The next Node 30 is automatically assigned. In the DX column, type 1, Tab to DY and type -1, then press Enter. This completes the bend input.



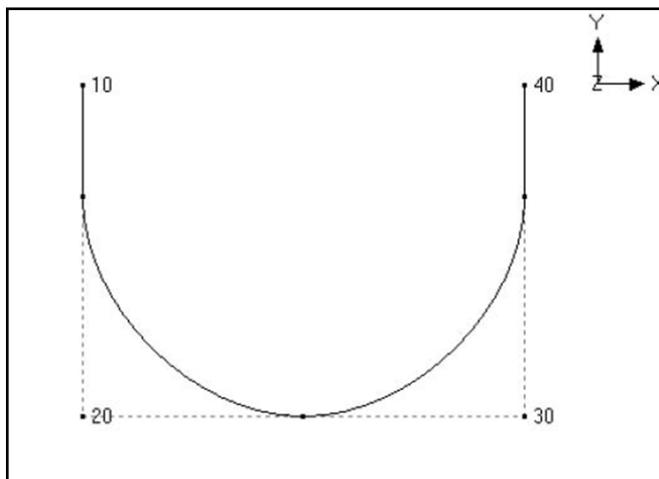
Bend

Example 3: 180° Bend

A 180° bend or U-bend, is often used in an expansion loop to relieve thermal stresses in the piping system. It is modeled as two 90° bends back-to-back.

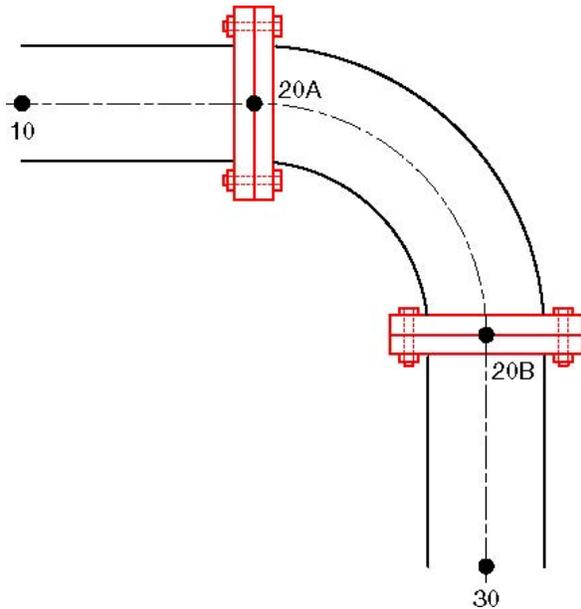


- ▶ Press Tab in row #3. Node 20 will be automatically assigned and the cursor will move to the Type column. type "B" (for Bend), Tab to DY, type $-1'6"$. Enter material, section, load and press Enter. The cursor moves to the next row (#4).
- ▶ Press Tab. Node 30 will be automatically assigned and the cursor will move to the Type column. type "B" (for Bend), Tab to DX, type 2. (DX is 2' because 8" std long radius bend has 12" radius and since these two bends are back to back, $DX = 2R$). Press Enter and the cursor moves to the next row (#5).
- ▶ Tab to the DY column. The next Node 40 is automatically assigned. In DY column, type $1'6"$, then press Enter. This completes the bend input.

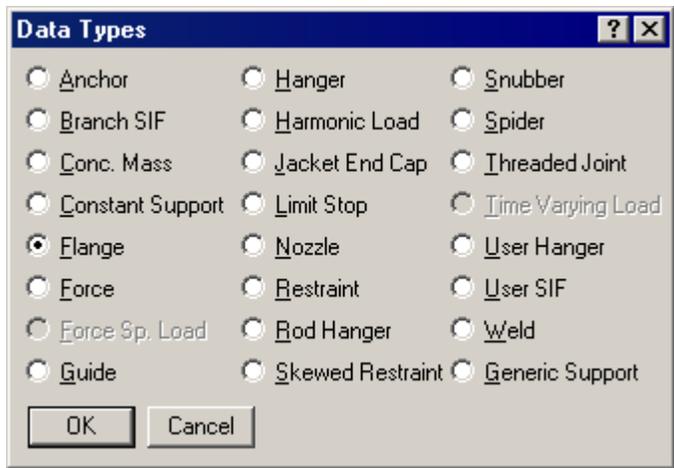


Example 4: Flanged Bend

Bends are often connected to the adjacent pipe sections with flanges. A flange may exist on one or both sides of the bend. Flange weight may have a significant effect on the pipe stresses. Also, the stress intensification and flexibility factors for a bend will decrease if one or both of the ends are flanged.



Model the bend as in Example 1. Then input flanges at nodes 20A and 20B. Since these are internally generated nodes, i.e., they do not *normally* appear in the Layout window, it is necessary to specify input at these nodes using the Location type. To input the flange at node 20A, in row #5, type 20A for Node, Tab to Type column and type “L” for Location. This opens the Data Types dialog.



Bend

Select Flange as the data type and click on OK. This opens the Flange dialog.

Select the Type of the flange from the drop-down combo box, e.g., Single welded slip-on flange. To get the weight of the flange, click on the Library button.

Rating	Weight (lb)
150	84
300	152
400	208
600	274
900	444
1500	668
2500	1384

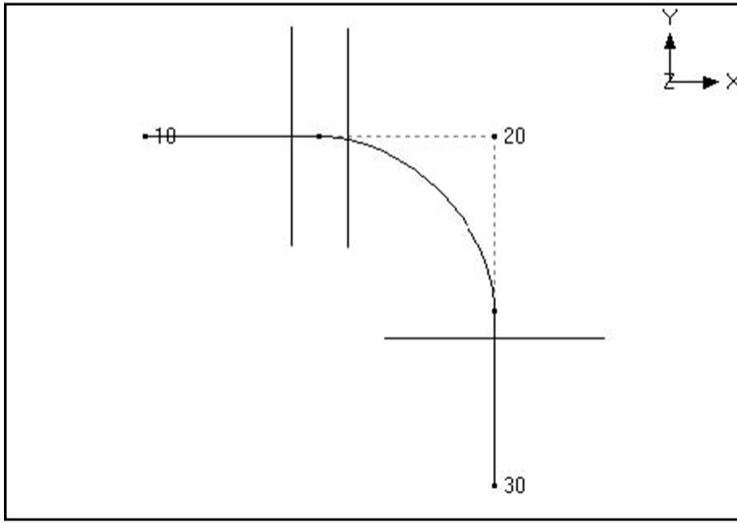
Select the pressure rating for the flange (e.g., 600) and press Enter. The weight of the flange is automatically entered in the Flange dialog. Press Enter again to input the flange.

Repeat the same procedure for the flange at node 20B.

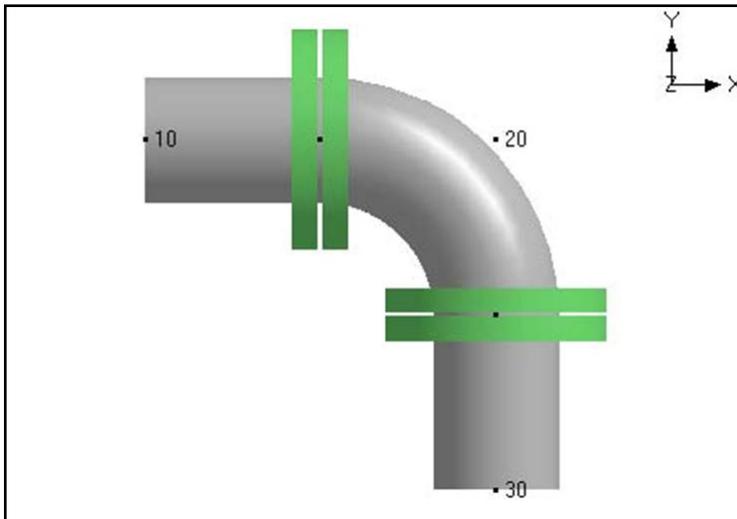
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Flanged bend								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			-2'0"		A53	8	1	
5	20A	Location							Flange
6	20B	Location							Flange

Bend

The graphics is shown below:



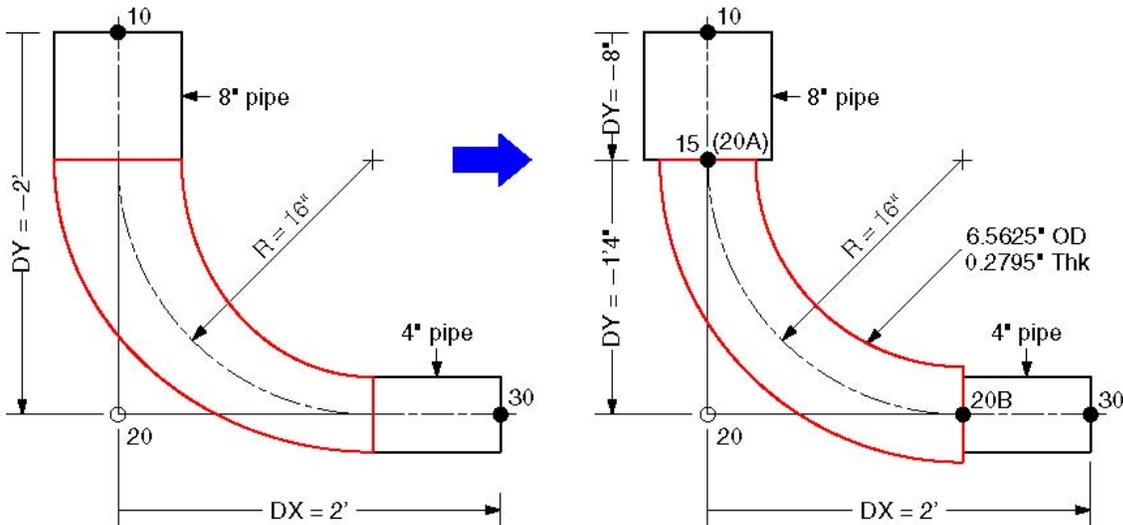
The rendered graphics is shown below:



Bend

Example 5: Reducing Bend

CAEPIPE does not have a reducing bend element. A reducing bend may be modeled using an average OD (outside diameter) and average thickness of the large and small ends of the bend. The bend radius of the reducing bend should input as user bend radius. The stress Intensification Factor (SIF) of the reducing bend, if available, should be input as Bend SIF.



- ▶ The 8" std pipe (OD = 8.625", Thk = 0.322") with Name = 8 is already defined. Now define a 4" std pipe (OD = 4.5", Thk = 0.237") with Name = 4.

The average OD of the two sections is $(8.625 + 4.5) / 2 = 6.5625$ " and the average Thickness is $(0.322 + 0.237) / 2 = 0.2795$ ".

Define a Non std section with Name = AVG, OD = 6.5625" and Thickness = 0.2795".

The list of sections is shown below.

#	Name	Nom Dia	Sch	OD (inch)	Thk (inch)	Cor. Al (inch)	M. Tol (%)	Ins. Dens (lb/ft3)	Ins. Thk (inch)
1	8	8"	STD	8.625	0.322				
2	AVG	Non Std		6.5625	0.2795				
3	4	4"	STD	4.5	0.237				

- ▶ Note that the section specified on the Bend row in the Layout window applies to the curved portion of the Bend (between the A and B nodes) as well as to the straight portion from the preceding node to the A node. In this case, we want to assign the section "AVG" only to the curved portion and assign the section "8" to the straight portion. This can be done by defining an additional node that is coincident with the A node thus making the straight portion of the bend zero length.
- ▶ In row #2, the first node (10) is already defined and the cursor is placed in row #3. Type 15 for Node, Tab to DY, type -8". Enter material (1), section (8), load (1) and press Enter. The cursor moves to the next row (#4).
- ▶ Press Tab. Node 20 is automatically assigned and the cursor will move to the Type column, type "B" (for Bend), Double click in the Type column to edit the Bend. Click on the User Bend Radius button and type 16 for bend radius. Press Enter to modify the Bend and return to the Layout window. Tab to DY, type -1'4". Tab to the

Bend

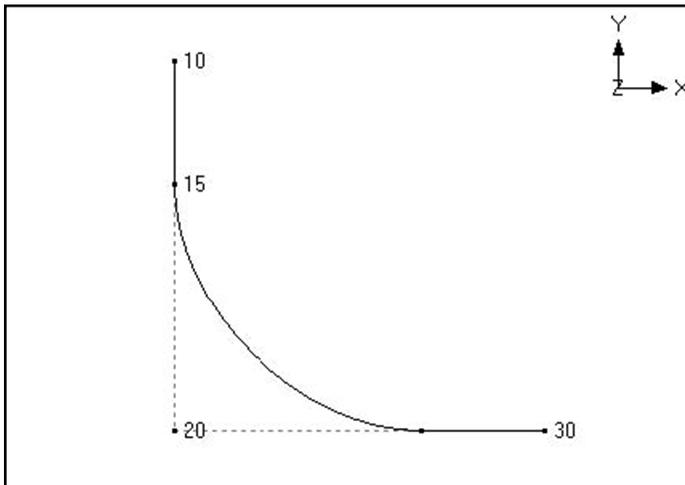
section column and type "AVG". Then press Enter. The material and load are copied from the previous row and the cursor moves to the next row(#5).

- ▶ Tab to the DX column. The next Node 30 is automatically assigned. In DX column, type 2'. Tab to section column, type 4 and then press Enter. This completes the reducing bend input.

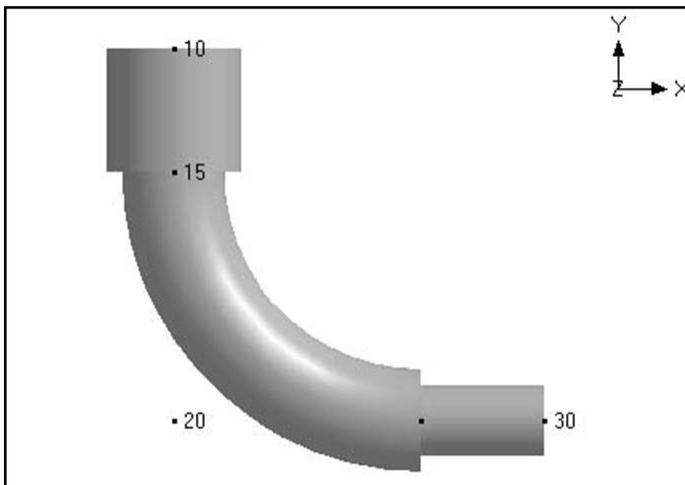
The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'n")	Matl	Sect	Load	Data
1	Title = Reducing bend								
2	10	From							
3	15			-0'8"		A53	8	1	
4	20	Bend		-1'4"		A53	AVG	1	
5	30		2'0"			A53	4	1	

The graphics is shown below:

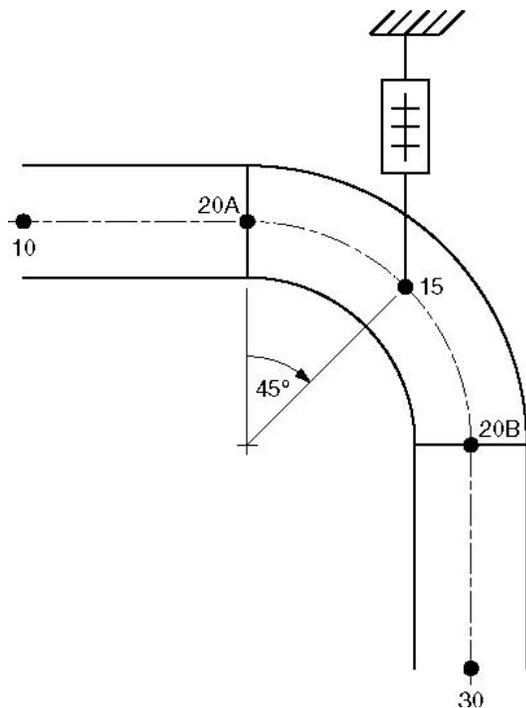


The rendered graphics is shown below:



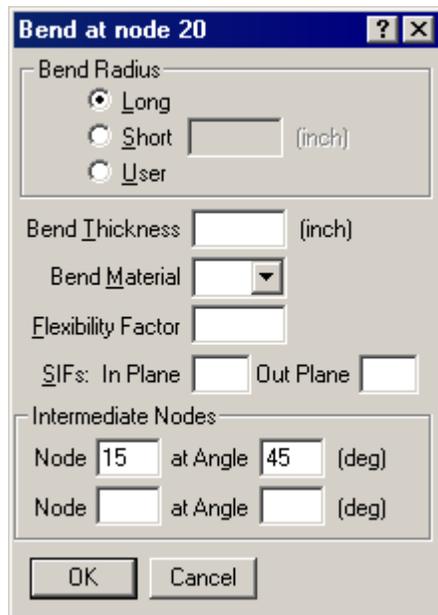
Example 6: Bend Supported by a Hanger

Assume that the bend is supported in the middle by a hanger.



Model the bend as in Example 1. The hanger is input at node 15, which is in the middle of the bend. Node 15 is created as an intermediate node on the bend as follows:

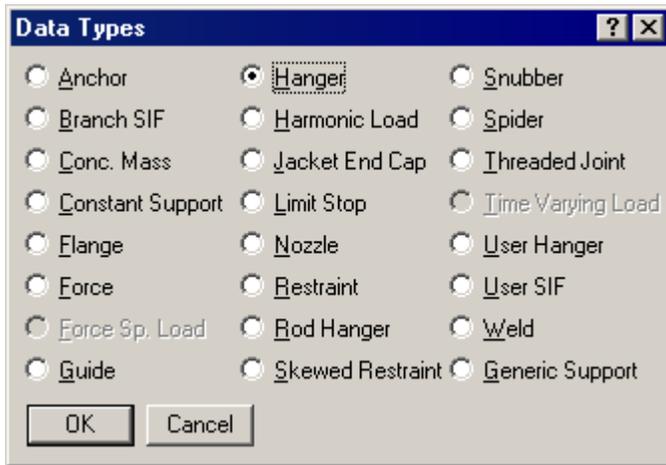
Double click on the bend (in the type column of the Layout window) to edit it. The bend dialog is shown.



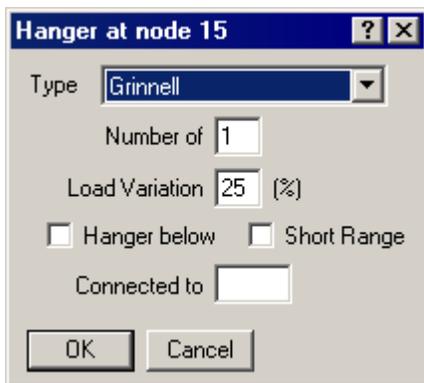
Under intermediate nodes, type 15 for node and 45 for its angle, then click on OK. This creates an intermediate node 15 at 45° from the node 20A (near end of the bend) as shown in the figure above.

Bend

Since node 15 does *not* show in the Layout window, it is necessary to specify data input at this node using the Location type. To input the hanger at this node, in row #5, type 15 for Node, Tab to Type column and type "L" for Location. This opens the Data Types dialog.



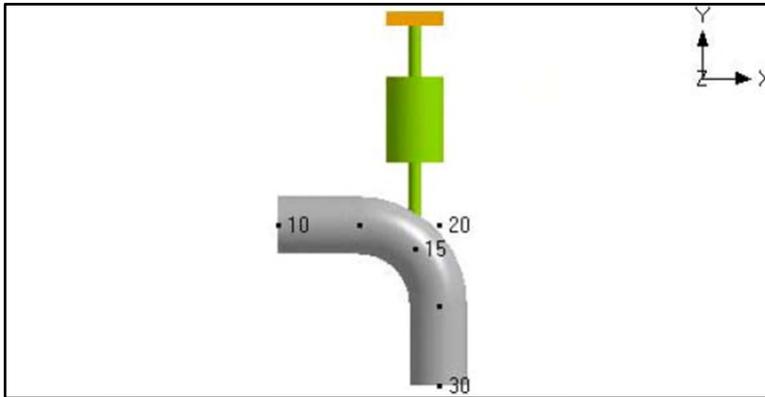
Click on Hanger, the hanger dialog is shown.



Click on OK to accept the default hanger and a hanger is entered at node 15.

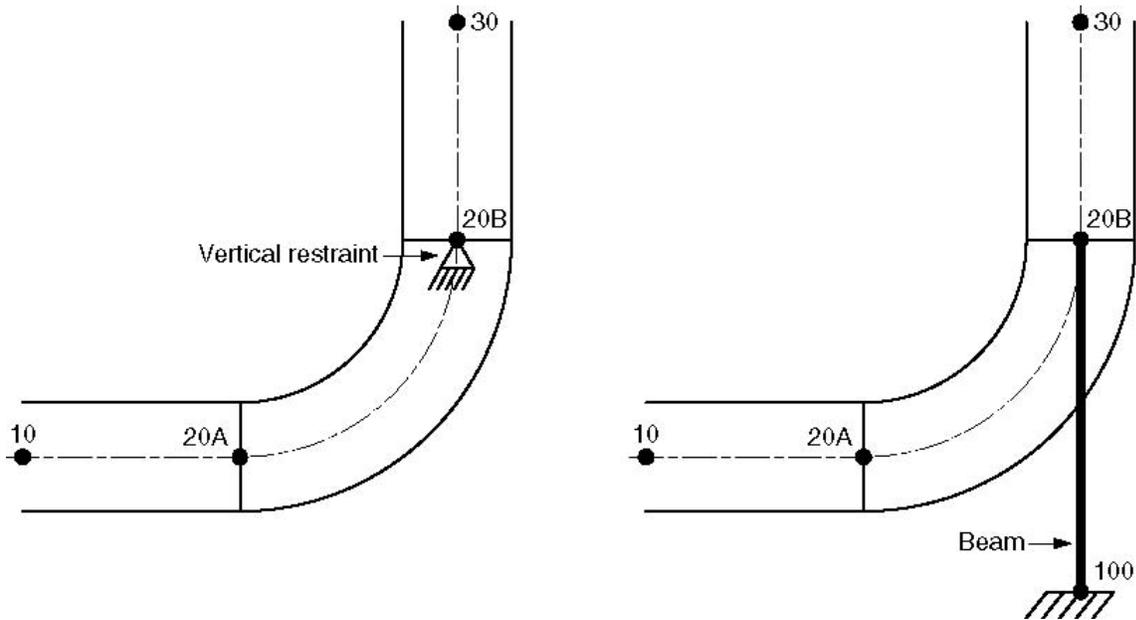
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Bend supported by a hanger								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			-2'0"		A53	8	1	
5	15	Location							Hanger

Bend



Example 7: Base Supported Bend

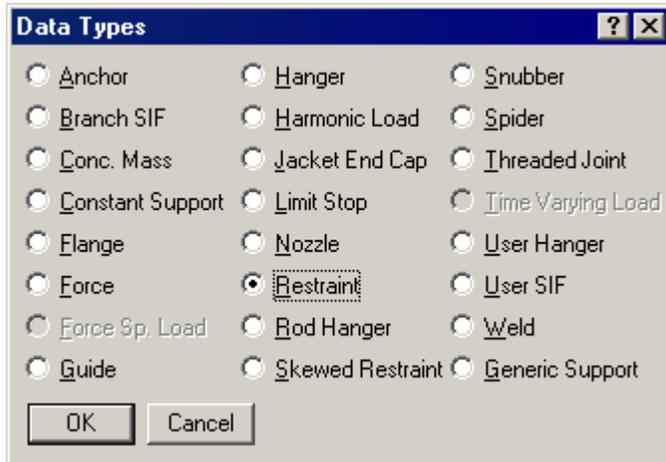
Two examples of base supported bends are shown below. In the figure on the left, the support is modeled using a rigid vertical restraint. In the figure on the right the support is modeled using a beam element.



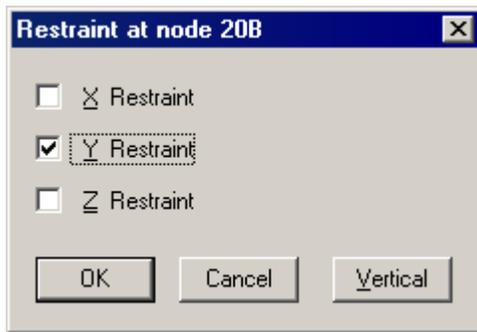
Vertical Restraint Support

- ▶ Model the bend at node 20 as before.
- ▶ To put a vertical restraint at node 20B, type 20B for node and “L” for Location. This will open the Data types dialog.

Bend



Double click on Restraint. This will open the Restraint dialog.



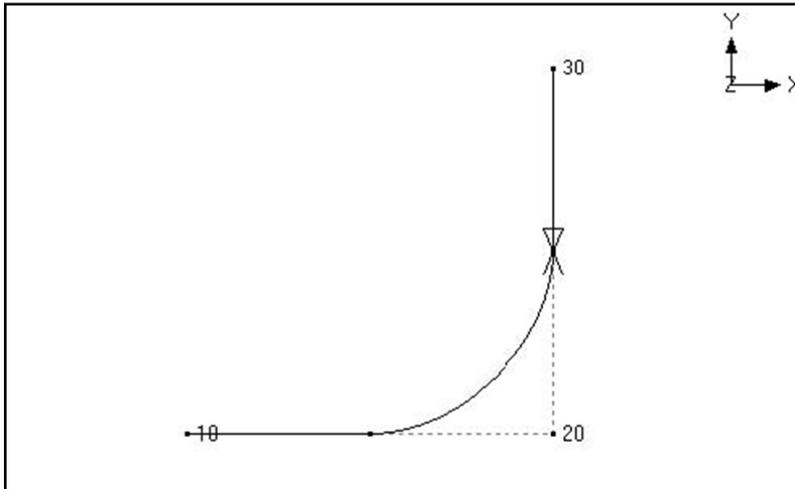
- ▶ Click on the Vertical button to check the Y or Z restraint (depending on the vertical axis) and click on OK.

The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Base support using a vertical restraint								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			2'0"		A53	8	1	
5	20B	Location							Y restraint

Bend

The graphics is shown below:



Beam Support

- ▶ Model the bend at node 20 as before.
- ▶ Create a beam material, section and load as described in the Beam section earlier in the Reference.
- ▶ Input a beam element from node 20B to node 100.

Type 20B in the Node column and “F” (for From) in the Type column to create a starting point. Press Enter to move to the next row.

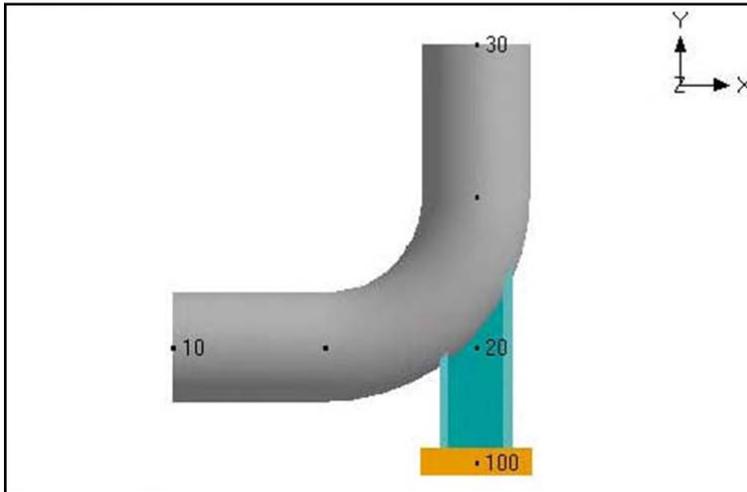
Type 100 in the Node column and “bea” (for Beam) in the Type column. In the DY column, type the beam length with a negative sign (since the beam is going downward from node 20B to node 100). Type the beam material, beam section and beam load names in the Matl, Sect and Load columns. In the Data column type “a” to input an Anchor.

The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in')	Matl	Sect	Loac	Data
1	Title = Base support using a beam								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			2'0"		A53	8	1	
5	20B	From							
6	100	Beam		-1'9"		BM1	BS1	BL1	Anchor

Bend

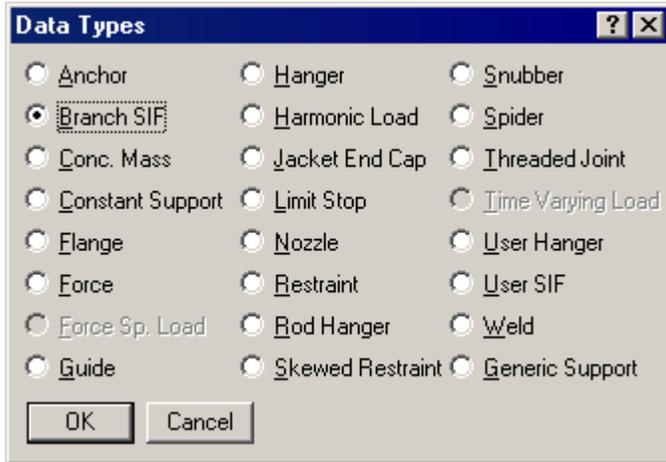
The graphics is shown below:



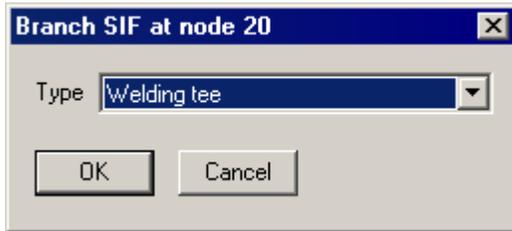
Branch SIF

A tee is modeled using three pipes that come together at a node, which should be designated as a “Tee” using the Branch SIF data type. If not, then CAEPIPE cannot calculate a code-specified SIF for the tee to use in stress calculations.

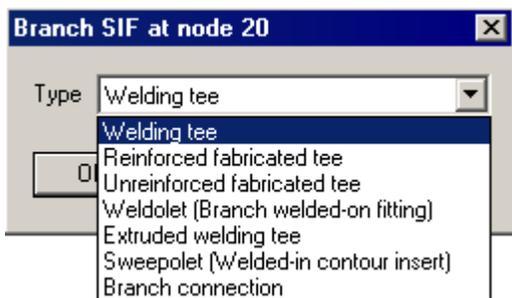
A Stress Intensification Factor (SIF) type for a tee can be input by typing “br” in the Data column or selecting “Branch SIF” from the Data types dialog.



The Branch SIF dialog is shown.

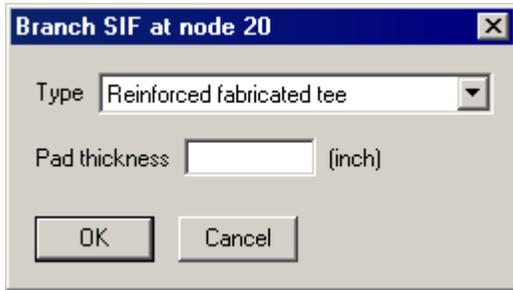


The type of the branch SIF can be selected from the Type drop-down combo box. Depending on the piping code different types of branch SIFs may be available. Typical branch SIF types (for B3.1 piping code) are shown below.



A few branch SIFs may need additional input; for example, in the case of a reinforced fabricated tee, a pad thickness is required.

Branch SIF



CAEPIPE differentiates between a header (run) and a branch line based on their ODs. So, when CAEPIPE finds two lines with ODs of 8 inches and 6 inches, it designates the 8 inch line as the header (or main) line with the 6 inch line designated as the branch line (this is also how you would model a reducing tee).

When the header and the branch lines have the same ODs, reduce the branch OD slightly so that the header and the branch lines are properly designated (e.g., $OD_{\text{header}} = 168.4 \text{ mm}$, $OD_{\text{branch}} = 168.3 \text{ mm}$).

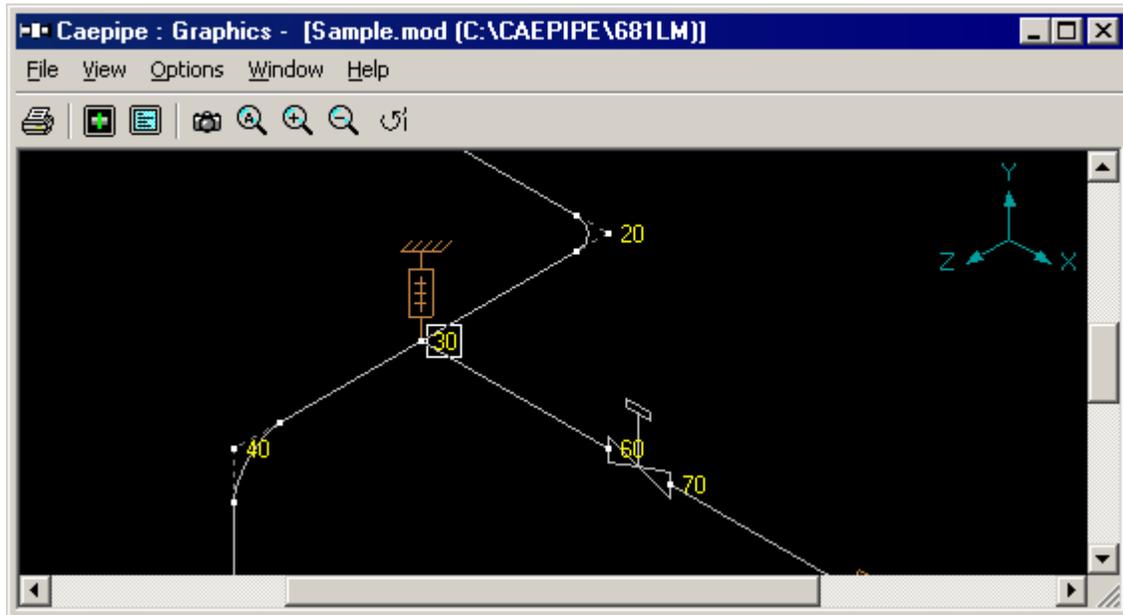
Examples:

Say you want to model a reducing tee as shown in the provided "Sample.mod". It's an 8"x6" reducing tee. First, we need to model the three pipes – two for the run (nodes 20 to 30 and 30 to 40), one for the branch (nodes 30 to 60), which will be assigned the 6" section. Each pipe section can have its own OD/Thickness. Lastly, designate the common node as a (Butt) Welding Tee.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe								
8	30	From							Welding tee
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	

Branch SIF

The Tee is as shown at node 30 in the graphics window below.



A “latrolet” or a lateral tee (commonly, the branch is at a 45° angle from the header) or a Y-joint is modeled like a regular tee (as above). But, the SIF for this joint needs to be input by you after consultation with the tee/joint manufacturer. Use a “User-SIF” data type at the common node to specify the SIF. See topic on Tees for more information.

Buried Piping

Soil in Buried piping analysis is modeled by using bilinear restraints with an initial stiffness and an ultimate load. After the ultimate load is reached the displacement continues without any further increase in load, i.e., the yield stiffness is zero. The initial stiffness is calculated by dividing the ultimate load by the yield displacement which is assumed to be $D/25$ where D is outside diameter of the pipe.

Soil modeling is based on Winkler's soil model of infinite closely spaced elastic springs. Soil stiffness is calculated for all three directions at each node. Pressure value in the load is suitably modified to consider the effect of static overburden soil pressure. Model is analyzed for operating (W+P1+T1) condition and the displacements in the three directions are noted. A check is made for whether skin friction is mobilized and the soil has attained the yield state. If true, then the spring is released in that direction indicating that soil no longer offers resistance in that direction. This modified model is again analyzed and checked for yield stage. The iterative process is continued till the percentage difference between displacements at each node for two successive iterations is less than 1%. The final stiffness is the true resistance offered by the soil to the pipe.

General Procedure to model buried piping

1. First define soils using the command Misc > Soils in the Layout or List window.
2. Next, tie these defined soils with pipe sections (Ctrl+Shft+S to list Sections, double click on an empty row, you will see the field Soil in the bottom right corner. Pick the soil name from the drop-down combo box).
3. Use this modified section for each element on the Layout window that is buried with this soil around it.

CAEPIPE presently does not discretize long sections of buried piping. A future release is anticipated to provide more comprehensive soil modeling. Discretization is required near "regions of discontinuities." These regions are changes in directions (bends, tees), or where piping exits to open space, or piping enters the buried region. In these areas, you need to subdivide long spans into smaller ones as much as possible. This will contribute to lowering vertical deflections.

It is possible to specify different soil characteristics for different portions of the pipe model. Here is how.

1. Define different soils using the command Misc > Soils.
2. Then, create as many pipe sections as the number of soils.
3. Associate each soil type with a section (to be used for the different portions).
4. Model piping using the different sections for different portions of the model.

Note: For vertical sections of pipe with varying buried depths, you can use only one soil tied to one section and CAEPIPE will calculate appropriately

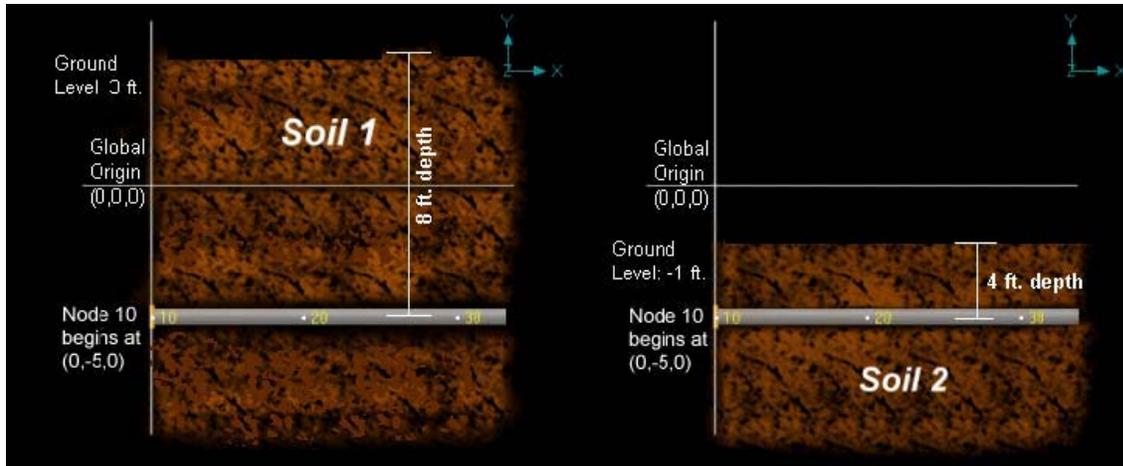
Buried Piping

Ground Level

Ground level for a soil is the height of the soil surface from the global origin (height could be positive or negative). It is NOT a measure of the depth of the pipe's centerline.

In the figure, the height of the soil surface is 3 feet from the global origin. Pipe node 10 [model origin] is defined at (0,-5,0). So, the pipe is buried 8' (3' - [-5']) deep into the soil. Define similarly for the other soil.

The pipe centerline is calculated by CAEPIPE from the given data



Two Soil types

Two types of soils can be defined - Cohesive and Cohesionless. Soil density and Ground level are input for both cohesive and cohesionless soils. The Ground level is used to calculate depth of the buried section. For cohesive soil, Strength is the un-drained cohesive strength (Cs). For cohesionless soil, Delta (δ) is angle of friction between soil and pipe, and Ks is Coefficient of horizontal soil stress. See nomenclature below for more information.

Highlight buried sections of the model in graphics

If your model contains sections that are above ground and buried, then you can selectively see only the buried sections of piping in CAEPIPE graphics by highlighting the section that is tied to the soil. Use the Highlight feature under the Section List window and place highlight on the buried piping section (see Highlight under List window>View menu, or press Ctrl+H). The Graphics window should highlight only that portion of the model that is using that specific section/soil.

Buried Piping

Nomenclature

D = Outside diameter of pipe

K_s = Coefficient of horizontal soil stress which depends on the relative density and state of consolidation of soil. K_s is empirical in nature and may be estimated from $N_q/50$. K_s can vary depending on the compaction of the soil from 0.25 (for loose soil) to 1.0 (really compacted soil).

N_q = Bearing capacity factor = $0.98414e^{(0.107311\phi)}$

ϕ = $\delta + 5^\circ$

δ = angle of friction between soil and pipe
Normal values for delta ranges between $25^\circ - 45^\circ$ (for sand).
Clean granular sand is 30° . With a mix of silt in it, the angle is 25°

S_p = soil pressure = soil density \times depth

C_s = Undrained cohesive strength (input for cohesive soil), (C_s in kN/m²) ≤ 1.0

A_f = Adhesion factor = $1.7012775 e^{(-0.00833699 C_s)}$

k_p = Coefficient of passive earth pressure
= $(1 + \sin \phi) / (1 - \sin \phi)$

bottom depth = depth + D/2
top depth = depth - D/2

N_r = $(N_q - 1.0) \tan (1.4 \phi)$

$d_q = d_r = 1.0 + 0.1 \tan (\pi/4 + \phi/2) \times \text{depth} / D$, for $d > 10^\circ$, otherwise $d_q = d_r = 1.0$

Calculation of Ultimate Loads

The ultimate loads (per unit length of pipe for axial and transverse directions and per unit projected length of pipe for vertical direction) are calculated as shown below.

Different equations are used for cohesive (clayey) and cohesionless (sandy) soils.

Axial direction

Cohesive soil: Axial load = $\pi \times D \times A_f \times C_s$

Cohesionless soil: Axial load = $\pi \times D \times K_s \times S_p \times \tan \delta$

Transverse direction

Cohesive soil: Transverse load = $D \times (2 C_s + S_p + 1.5 C_s \times \text{depth} / D)$

Cohesionless soil: Transverse load = $k_p \times k_p \times S_p \times D$

Vertically downward direction

Cohesive soil: Downward load = $D \times (5.7182 C_s + \text{Soil density} \times \text{bottom depth})$

Cohesionless soil: Downward load = $D \times (\text{Soil density} \times \text{bottom depth} \times N_q \times d_q + 0.5 \text{Soil density} \times D \times N_r \times d_r)$

Buried Piping

Vertically upward direction

Cohesive soil: Upward load = $D \times \text{Soil density} \times \text{top depth} + 2 C_s \times \text{top depth}$

Cohesionless soil: Upward load = $D \times \text{Soil density} \times \text{top depth}$

Buried Piping Example

Ultimate Loads and Stiffnesses computed by CAEPIPE for this example are verified later in this section.

Example data:

A 12" Std pipe 6' long is buried, 3' in cohesionless and 3' in cohesive soils.

Soil properties are as follows:

Cohesionless (Name of soil: S1, associated with pipe section 12A):

Density = 120 lb / ft³

Delta (δ) = 20°

Ks = 0.29 (calculated from $N_q/50$, where $N_q = 14.394$)

Ground level = 3'

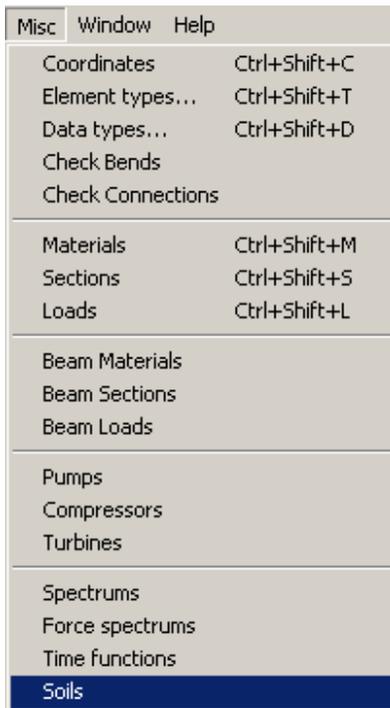
Cohesive (Name of soil: S2, associated with pipe section 12B):

Density = 150 lb / ft³

Strength = 100 psi

Ground level = -1'

1. Define soils using the command Misc > Soils.

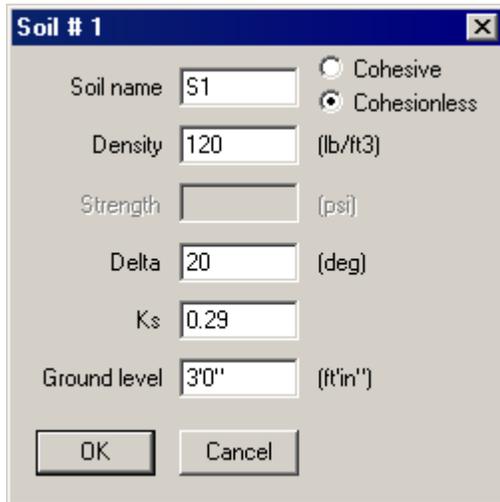


Buried Piping

A List window for soils will be displayed. Double click on an empty row to define a new soil.

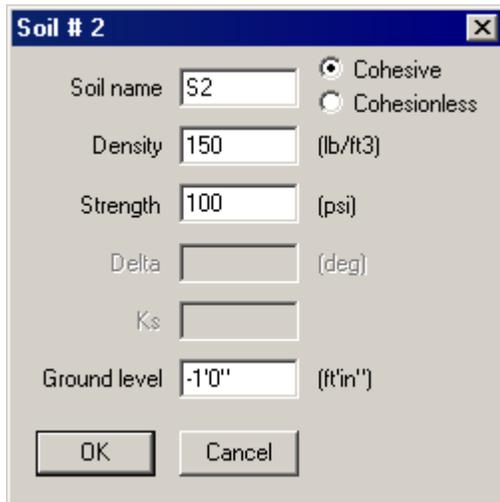
For our example, define two soils - one cohesive and the other cohesionless with properties as shown in the following dialogs.

Dialog for cohesionless soil:



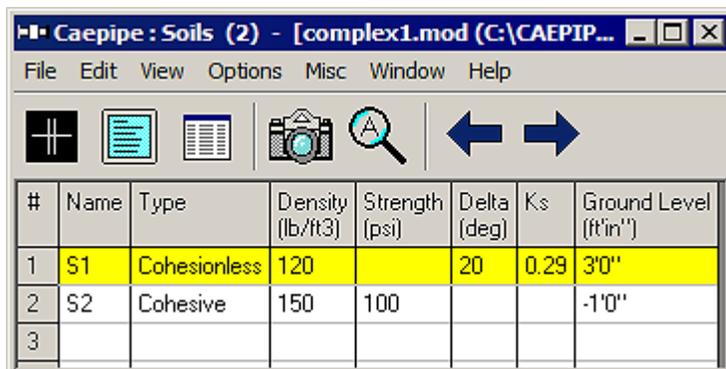
Soil # 1 dialog box showing properties for a cohesionless soil. The dialog includes fields for Soil name (S1), Density (120 lb/ft3), Strength, Delta (20 deg), Ks (0.29), and Ground level (3'0" ft'in"). Radio buttons are set to Cohesionless.

Dialog for cohesive soil:



Soil # 2 dialog box showing properties for a cohesive soil. The dialog includes fields for Soil name (S2), Density (150 lb/ft3), Strength (100 psi), Delta, Ks, and Ground level (-1'0" ft'in"). Radio buttons are set to Cohesive.

After you define the soils, you should see the two soils listed in the List window.



Caepipe : Soils (2) - [complex1.mod (C:\CAEPIP...)] window showing a list of defined soils. The table below represents the data shown in the window.

#	Name	Type	Density (lb/ft3)	Strength (psi)	Delta (deg)	Ks	Ground Level (ft'in')
1	S1	Cohesionless	120		20	0.29	3'0"
2	S2	Cohesive	150	100			-1'0"
3							

Buried Piping

2. Define pipe sections and then associate the soils with these sections.

Define two pipe sections, both 12"/STD pipe sections (name them 12A and 12B), and select the correct soil in the pipe section dialog box using the Soil drop-down combo box.

Soil S1 is associated with section 12A:

Section # 1

Section name: 12A ANSI DIN JIS ISO

Nominal diameter: 12" Schedule: STD

Outside diameter: 12.75 (inch) Thickness: 0.375 (inch)

Corrosion allowance: (inch) Mill tolerance: (%)

Insulation : Density: (lb/ft3) Thickness: (inch)

Lining : Density: (lb/ft3) Thickness: (inch)

OK Cancel Insulation Soil: S1

Soil S2 is associated with section 12B:

Section # 2

Section name: 12D ANSI DIN JIS ISO

Nominal diameter: 12" Schedule: STD

Outside diameter: 12.75 (inch) Thickness: 0.375 (inch)

Corrosion allowance: (inch) Mill tolerance: (%)

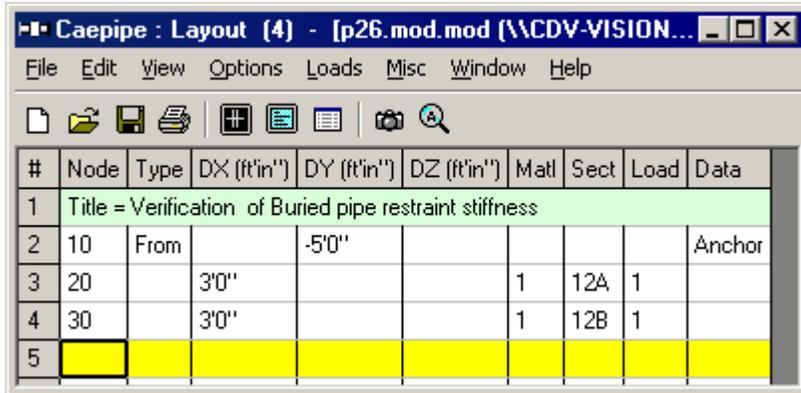
Insulation : Density: (lb/ft3) Thickness: (inch)

Lining : Density: (lb/ft3) Thickness: (inch)

OK Cancel Insulation Soil: S2

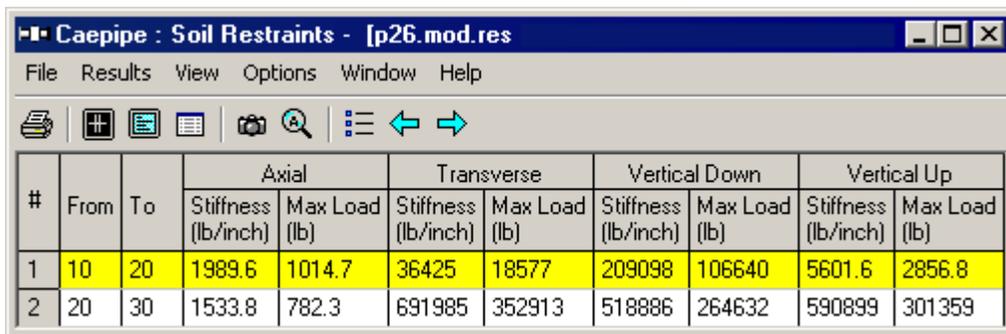
Buried Piping

3. Define the layout from 10 to 20 to 30; the first pipe element from 10 to 20 uses section 12A (Cohesionless soil type S1), and the next pipe element 20 to 30 uses section 12B (Cohesive soil type S2). Check Operating load case under Loads menu > Load cases for analysis.



#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Verification of Buried pipe restraint stiffness								
2	10	From		-5'0"					Anchor
3	20		3'0"			1	12A	1	
4	30		3'0"			1	12B	1	
5									

Save the model and analyze. Choose yes to view the results. From the Results dialog, pick Soil Restraints. The different loads and stiffnesses will be shown. The numbers found here below are the ones that are verified later in this section.



#	From	To	Axial		Transverse		Vertical Down		Vertical Up	
			Stiffness (lb/inch)	Max Load (lb)						
1	10	20	1989.6	1014.7	36425	18577	209098	106640	5601.6	2856.8
2	20	30	1533.8	782.3	691985	352913	518886	264632	590899	301359

Example Verification

Verification of cohesionless restraints (for pipe element 10 to 20)

S_p = soil pressure = soil density \times depth

depth = 3' - (-5') = 8' (since the pipe centerline is at -5' and ground level is at 3').

S_p = 120 lb/ft³ \times 8 ft = 960 lb / ft² = 6.6667 lb/in²

Axial direction

Axial load = $\pi \times D \times K_s \times S_p \times \tan \delta$

= $\pi \times 12.75 \times 0.29 \times 6.6667 \times \tan (20)$

= 28.1861 lb/in

= 1014.7 lb (for 36", length of pipe) (CAEPIPE: 1014.7)

Assuming yield displacement = D/25,

Axial stiffness = $25 \times 1014.7 / 12.75$ = 1989.6 (lb /in) (CAEPIPE: 1989.6)

Buried Piping

Transverse direction

$$\phi = \delta + 5^\circ = 20^\circ + 5^\circ = 25^\circ$$

$$\begin{aligned} k_p &= \text{Coefficient of passive earth pressure} \\ &= (1 + \sin \phi) / (1 - \sin \phi) \\ &= 2.4639 \end{aligned}$$

$$\begin{aligned} \text{Transverse load} &= k_p \times k_p \times S_p \times D \\ &= 2.4639 \times 2.4639 \times 6.6667 \times 12.75 \\ &= 516.0239 \text{ lb / in} \\ &= 18576.88 \text{ lb (for 36")} \text{ (CAEPIPE: 18577)} \end{aligned}$$

$$\begin{aligned} \text{Transverse stiffness} &= 25 \times 18576.88 / 12.75 \\ &= 36425 \text{ lb / in (CAEPIPE: 36425)} \end{aligned}$$

Vertically downward direction

$$\text{bottom depth} = 96'' + 12.75''/2 = 102.375''$$

$$N_q = \text{Bearing capacity factor} = 0.98414 e^{(0.107311 \phi)} = 14.39366$$

$$N_r = (N_q - 1.0) \times \tan(1.4 \phi) = 9.37834$$

since $\delta > 10^\circ$

$$d_q = d_r = 1.0 + 0.1 \times \tan(\pi/4 + \phi/2) \times \text{depth} / D = 2.18188$$

$$\begin{aligned} \text{Downward load} &= D \times (\text{Soil density} \times \text{bottom depth} \times N_q \times d_q + 0.5 \times \text{Soil density} \times D \times \\ &N_r \times d_r) \\ &= 12.75 \times ((120/1728) \times 102.375'' \times 14.39366 \times 2.18188 + 0.5 \times (120/1728) \times 12.75'' \\ &\times 9.37834 \times 2.18188) \\ &= 12.75 \times 232.3305 \text{ lb / in} \\ &= 106639.7 \text{ lb (for 36")} \text{ (CAEPIPE: 106640 lb)} \end{aligned}$$

$$\begin{aligned} \text{Downward stiffness} &= 25 \times 106639.7 / 12.75 \\ &= 209097 \text{ lb/in (CAEPIPE: 209098)} \end{aligned}$$

Vertically Upward Direction

$$\text{top depth} = 96'' - 12.75''/2 = 89.625''$$

$$\text{Upward load} = D \times \text{Soil density} \times \text{top depth}$$

$$= 12.75'' \times (120/1728) \times 89.625$$

$$= 79.35547 \text{ lb / in}$$

$$= 2856.7968 \text{ lb (for 36")} \text{ (CAEPIPE: 2856.8)}$$

$$\text{Upward stiffness} = 25 \times 2856.7968 / 12.75 = 5601.56 \text{ lb/in (CAEPIPE: 5601.6)}$$

Verification of cohesive restraints (for pipe element 20 to 30)

$$S_p = \text{soil pressure} = \text{soil density} \times \text{depth}$$

$$\text{depth} = -1' - (-5') = 4' \text{ (since the pipe centerline is at } -5' \text{ and ground level is at } 3').$$

$$S_p = 150 \text{ lb/ft}^3 \times 4 \text{ ft} = 600 \text{ lb / ft}^2 = 4.16667 \text{ lb/in}^2$$

Buried Piping

Axial direction

$$\text{Soil strength} = C_s = 100 \text{ psi} = 100 \times 6.89476 \text{ KN/M}^2 = 689.476 \text{ KN/M}^2$$

$$\begin{aligned} A_f &= \text{Adhesion factor} = 1.7012775 e^{(-0.00833699 C_s)} \\ &= 5.424795E-3 \end{aligned}$$

$$\begin{aligned} \text{Axial load} &= \pi \times D \times A_f \times C_s \\ &= \pi \times 12.75'' \times 5.424795E-3 \times 100 \text{ psi} \\ &= 21.72982 \text{ lb / in} \\ &= 782.25 \text{ lb (for 36'')} \text{ (CAEPIPE: 782.3)} \end{aligned}$$

$$\text{Axial stiffness} = 25 \times 782.25 / 12.75 = 1533.82 \text{ lb /in (CAEPIPE: 1533.8 lb/in)}$$

Transverse direction

$$\begin{aligned} \text{Transverse load} &= D \times (2 C_s + S_p + 1.5 C_s \times \text{depth} / D) \\ &= 12.75'' \times (2.0 \times 100 + 4.166667 + 1.5 \times 100 \times 48 / 12.75) \\ &= 9803.125 \text{ lb/in} \\ &= 352912.5 \text{ lb for 36'' (CAEPIPE: 352913)} \end{aligned}$$

$$\begin{aligned} \text{Transverse stiffness} &= 25 \times 352912.5 / 12.75 \\ &= 691985.3 \text{ lb /in (CAEPIPE: 691985)} \end{aligned}$$

Vertically Downward direction

$$\text{bottom depth} = 48'' + 12.75''/2 = 54.375''$$

$$\begin{aligned} \text{Downward load} &= D \times (5.7182 C_s + \text{Soil density} \times \text{bottom depth}) \\ &= 12.75'' \times (5.7182 \times 100 + (150/1728) \times 54.375'') \\ &= 7350.8857 \text{ lb/in} \\ &= 264631.88 \text{ lb for 36'' (CAEPIPE: 264632)} \end{aligned}$$

$$\begin{aligned} \text{Downward stiffness} &= 25 \times 264631.88 / 12.75 \\ &= 518886 \text{ lb / in (CAEPIPE: 518886)} \end{aligned}$$

Vertically Upward Direction

$$\text{top depth} = 48'' - 12.75''/2 = 41.625''$$

$$\begin{aligned} \text{Upward load} &= D \times \text{Soil density} \times \text{top depth} + 2 C_s \times \text{top depth} \\ &= 12.75'' \times ((150/1728) \times 41.625'' + 2 \times 100 \times 41.625'') \\ &= 8371.069 \text{ lb / in} \\ &= 301358.49 \text{ lb for 36'' (CAEPIPE: 301359)} \end{aligned}$$

$$\begin{aligned} \text{Upward stiffness} &= 25 \times 301358.49 / 12.75 \\ &= 590899 \text{ lb / in (CAEPIPE: 590899)} \end{aligned}$$

References

1. Tomlinson, M. J., Pile Design and Construction Practice. Fourth Edition. London: E & FN Spon, 1994.
2. Fleming, W.G.K., et al. Piling Engineering. Second Edition. Blackie Academic and Professional. (Chapters 4 and 5).

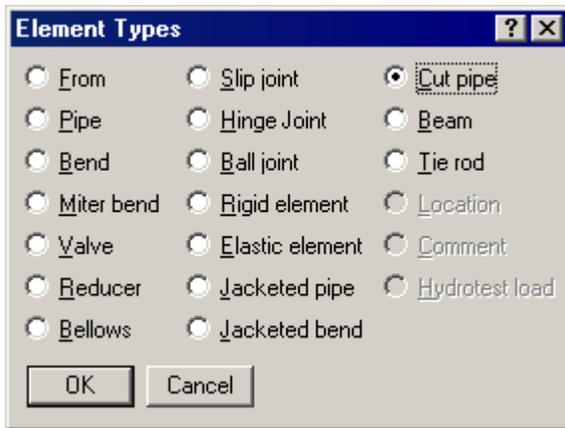
Cold Spring (Cut Pipe)

Cold spring (cut short or cut long) is used to reduce thermal forces on equipment connected to the piping system. When lengths of pipes are cut short or extended by design, they are pulled together or pushed apart to join them during installation, giving rise to a “cold-sprung” system.

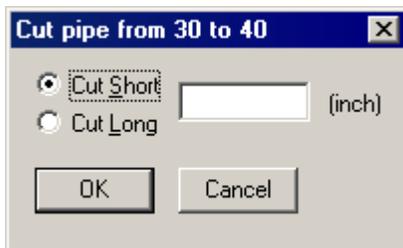
Such an installation process (cold condition) obviously introduces stresses, which are relieved when the system starts up (hot condition). Note however, that the piping codes do not allow credit for stresses since the displacement range is unaffected (similar to self-springing. See B31.1 para. 119.2 for details). But, allowance is made for the support loads due to cold spring (which can be helpful at the equipment).

This feature should be used only with a proper understanding of the implications.

Cold spring for a straight pipe is input by typing “c” in the Type column or selecting “Cut pipe” from the Element Types dialog.



The Cut pipe dialog is shown.



Select “Cut short” or “Cut long” using the radio buttons. The amount of cut (short or long) should be positive.

Since the piping codes do not allow credit for cold spring in stress calculations, a cold spring is used in additional sustained and operating load cases (designated “Cold Spring (W+P), Cold Spring (W+P1+T1)” etc.) which are not used in stress calculations but are used for support loads and rotating equipment reports.

Cold Spring load cases appear in the Loads menu (under Load cases) after a cold spring (Cut pipe element) is input into the model. The Load cases menu is shown next:

Cold Spring (Cut Pipe)

Load Case	Selected
<input type="checkbox"/> Sustained (W+P)	<input type="checkbox"/>
<input type="checkbox"/> Expansion (T1)	<input type="checkbox"/>
<input type="checkbox"/> Expansion (T2)	<input type="checkbox"/>
<input type="checkbox"/> Expansion (T3)	<input type="checkbox"/>
<input type="checkbox"/> Expansion (T1 - T2)	<input type="checkbox"/>
<input type="checkbox"/> Expansion (T1 - T3)	<input type="checkbox"/>
<input type="checkbox"/> Expansion (T2 - T3)	<input type="checkbox"/>
<input type="checkbox"/> Operating (W+P1+T1)	<input type="checkbox"/>
<input type="checkbox"/> Operating (W+P2+T2)	<input type="checkbox"/>
<input type="checkbox"/> Operating (W+P3+T3)	<input type="checkbox"/>
<input checked="" type="checkbox"/> Cold spring (W+P)	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Cold spring (W+P1+T1)	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Cold spring (W+P2+T2)	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Cold spring (W+P3+T3)	<input checked="" type="checkbox"/>
<input type="checkbox"/> Modal analysis	<input type="checkbox"/>

Buttons: OK, Cancel, All, None

For analysis, select the desired Cold Spring load cases from those shown. The Hanger selection procedure does not consider the cold spring since the selection is based on the first Operating (W+P1+T1) load case. However, if Cold Spring is used, the hanger loads for the Cold spring load cases [for example, Cold Spring (W+P1+T1)] will include the effect of the Cold spring.

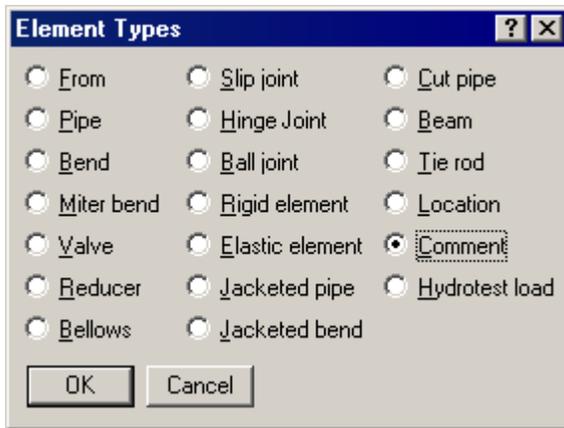
Comment

Most piping models and projects undergo many changes over their lifecycle including the engineers who work on them. CAEPIPE allows you to document each and every change within the model file so that anyone in the future can benefit from your notes, changes and their reasons, design decisions, and comments on the piping system modeled in the file. Use the Comment feature to write as many notes and comments as required anywhere in the CAEPIPE Layout window. They are saved with the model, and can be printed along with the layout data.

Two ways for putting in a comment are:

1. Simply press “c” first (in the Node column) on an empty row, or
2. On an empty row, select “Comment” from the Element Types dialog (Ctrl+Shft+T).

Use menu Edit > Insert (Ctrl+Ins) to insert an empty row between two existing rows of data.



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
10	7			0'3"		CRP	RT	HW	
11	End of reactor vertical shell								
12	...from centerline of Reactor								
13	12	From							
14	60	Rigid	3.2374		7.9481	CRP	RN	HW	
15	...Reactor nozzle AD1...								
16	65		0.9116		2.2381	CRP	RN	HW	
17	End of Reactor Nozzle AD1...projection to Reactor C.L. is 11'-0"								
18	70		0.3419		0.8393	CSP	RN1	CW	
19	75	Reducer	0.2829		0.6946	CSP	RN1	CW	

Rows 11, 12, 15 and 17 are the comment lines (highlighted with a light green background).

Compressor

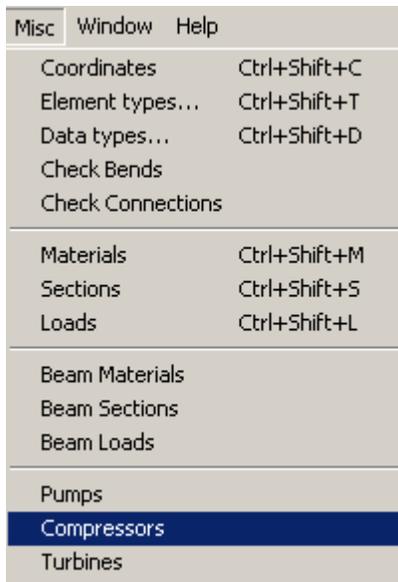
Pumps, compressors and turbines in CAEPIPE, referred to as rotating equipment, are each governed by an industry publication — API (American Petroleum Institute) publishes an API 610 for pumps and an API 617 for compressors while NEMA (National Electrical Manufacturers Association) publishes the NEMA SM-23 for turbines. These publications provide guidelines for evaluating nozzles connected to equipment among other technical information including the items relevant to piping stress analysis – criteria for piping design and a table of allowable loads.

Modeling the equipment is straightforward since it is assumed rigid (relative to connected piping) and modeled only through its end points (connection nozzles).

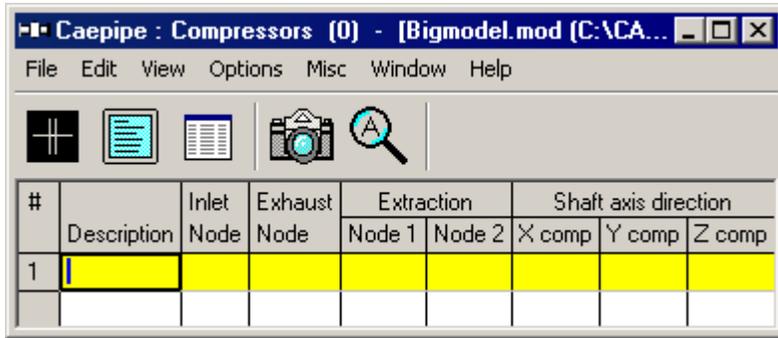
1. In your model, anchor all the nozzles (on the equipment) that need to be included in the analysis.
2. Specify these anchored nodes during the respective equipment definition via Misc. menu > Pumps/Compressors/Turbines in the Layout window.

CAEPIPE does not require you to model all the nozzles nor their connected piping. For example, you may model simply one inlet nozzle of a pump with its piping. Or, you may model one pump with both nozzles (with no connected piping) and impose external forces on them (if you have that data). Further, there is no need to connect the two anchors of the equipment with a rigid massless element like required in some archaic methods.

A compressor (like a turbine or a pump) is input by selecting “Compressors” from the Misc menu in the Layout or List window. Upon analysis, an API 617 compressor compliance report is produced. See Appendix B (API 617, for Compressors) for related information.

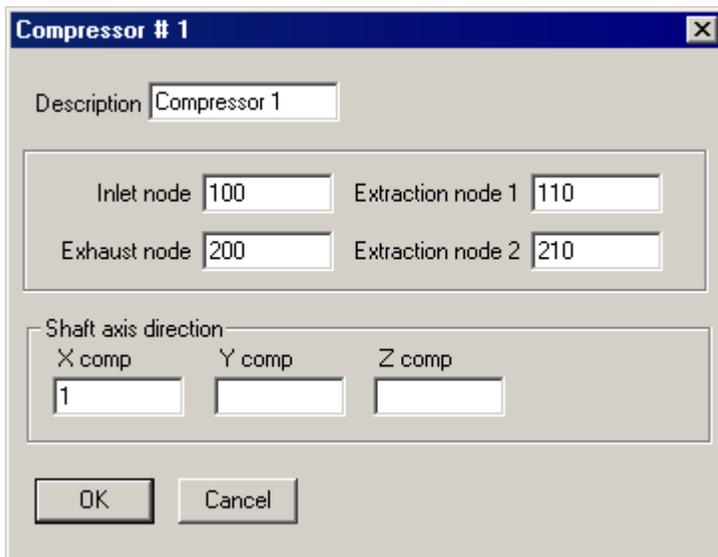


Compressor



#	Description	Inlet Node	Exhaust Node	Extraction		Shaft axis direction		
				Node 1	Node 2	X comp	Y comp	Z comp
1								

Once you see the Compressor List window, double click on an empty row for the Compressor dialog and enter the required information.



Compressor # 1

Description

Inlet node Extraction node 1

Exhaust node Extraction node 2

Shaft axis direction

X comp Y comp Z comp

A short description to identify the compressor may be entered for Description. The nozzle nodes must be anchors and the shaft axis must be in the horizontal plane. Some of the nozzle nodes may be left blank if they are not considered as a part of the piping system being analyzed (e.g., extraction nodes).

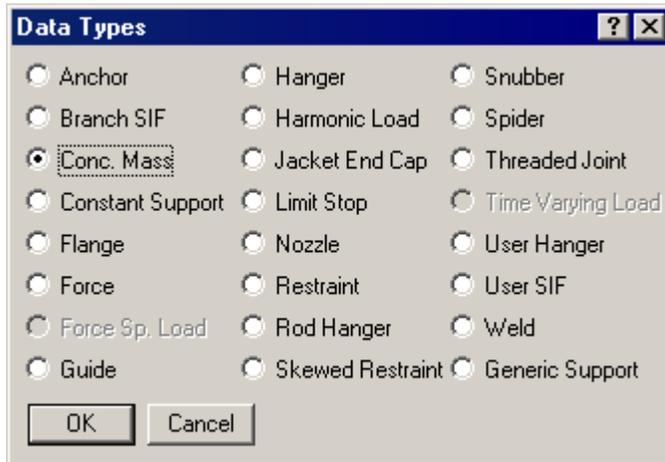
Compressor

Caepipe : Rotating Equipment Report - [p25.res (C:\Documents and Se...]							
File Results View Options Window Help							
API 617 (7th Ed. 2003) report for compressor : P24-API 617							
Load case: Operating (W+P1+T1)							
Shaft axis: Xcomp = 1.000, Ycomp = 0.000, Zcomp = 0.000							
----- Forces (lb) -----							
----- Moments (ft-lb) -----							
Node	Type	fx	fy	fz	mx	my	mz
5	Inlet	567	-301	497	2527	1981	-29
25	Exhaust	-252	-337	-549	-1075	2854	-2830
30	Extr.1	38	-140	-74	133	-200	420
----- Forces (lb) -----							
----- Moments (ft-lb) -----							
Node	Type	Size (inch)	--- Resultant ---		Allow		
			F(lb)	M(ft-lb)	3F + M	able	Ratio
5	Inlet	8.000	812	3211	5647	7400	0.763
25	Exhaust	12.000	692	4161	6236	8633	0.722
30	Extr.1	4.000	163	484	972	3700	0.263
Combined resultants at largest connection 25							
----- Forces (lb) -----							
----- Moments (ft-lb) -----							
		fx	fy	fz	mx	my	mz
	Calculated	353	-778	-126	1789	6848	330
	Allowable	1016	2541	2033	5082	2541	2541
	Ratio	0.347	0.306	0.062	0.352	2.695	0.130
----- Forces (lb) -----							
----- Moments (ft-lb) -----							
		--- Resultant ---		Allow			
		F(lb)	M(ft-lb)	2F + M	able	Ratio	
	Combined	864	7086	8813	5082	1.734	

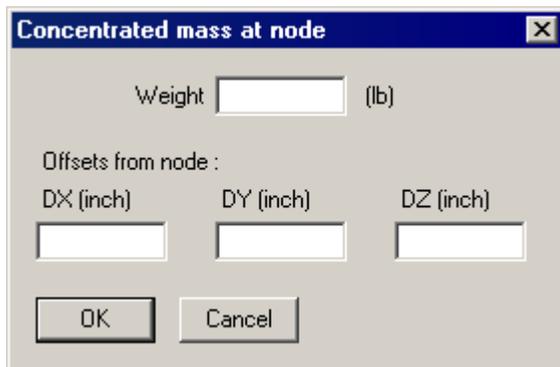
If you have input multiple temperatures, corresponding reports for additional operating load cases will be shown.

Concentrated Mass

A concentrated mass is input by typing “conc” in the Data column or selecting “Conc. Mass” from the Data Types dialog.



The Concentrated Mass dialog is shown.

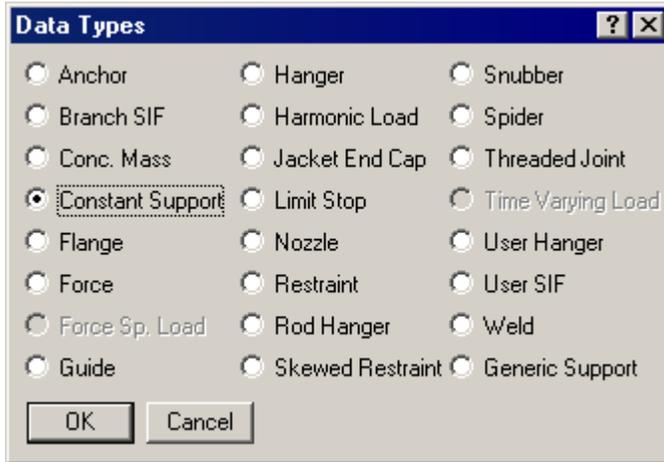


The weight of the concentrated mass should be input for Weight. The concentrated mass is located at the offset (DX, DY, DZ) from the node. Deadweight, seismic and dynamic loads due to concentrated masses are applied to the model.

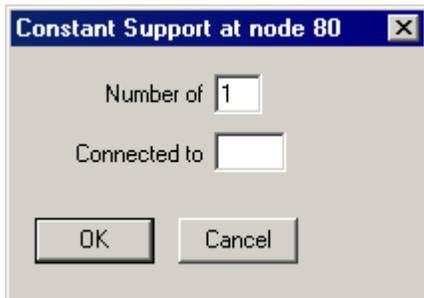
Constant Support

A constant support hanger exerts a constant vertical supporting force on the piping. The constant support load is automatically calculated by CAEPIPE. To analyze an existing constant support with a known load, input it as a user hanger with a zero spring rate.

A constant support is input by typing “cons” in the Data column or selecting “Constant Support” from the Data Types dialog.



The Constant Support dialog is shown.



Number of Hangers

The number of hangers is the number of separate hangers connected in parallel at this node.

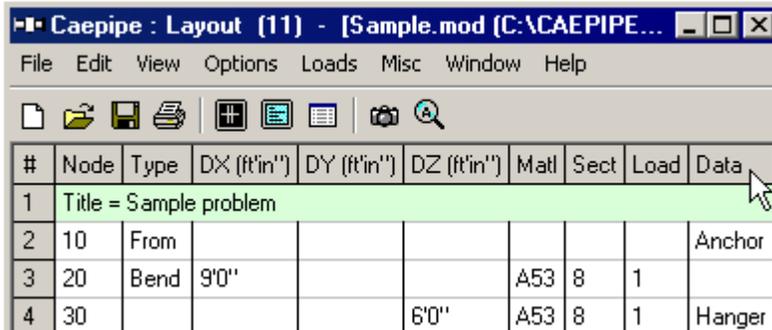
Connected to Node

By default the hanger is connected to a fixed ground point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the “Connected to node” field. This node must be directly above or below the hanger node.

Data Types

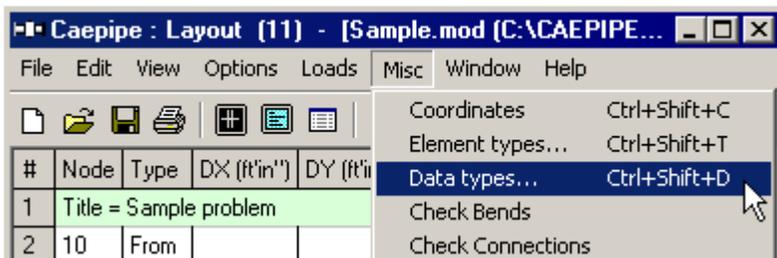
Items such as anchors, hangers and external forces, which are defined at nodes, are input in the Data column as Data types; different from inline elements such as pipes, bends and valves that *connect nodes*, and are input in the Type column as Element types.

The Data items can be selected from the Data Types dialog which is opened when you click on the Data header in the Layout window.

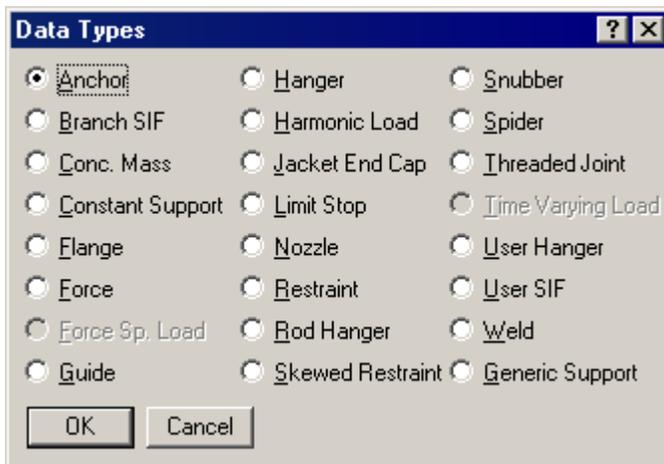


#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger

You may also use the command: Misc > Data types,



or press Ctrl+Shift+D to open the Data Types dialog.



You can select the data type by clicking on the radio button or pressing the underlined letter of the item, e.g., press “f” for Flange, Force or Force spectrum load. Or, you may simply start typing the first few letters of the item in the Data column. (For example, typing “fo” automatically opens a Force data type dialog).

Direction

Direction is required for several items such as Pump, Compressor, Turbine, Nozzle (for vessel axis), Limit Stop, Skewed restraint, Elastic element and Hinge joint.

The axis or the orientation of an item (listed above), is called the direction vector which is described in terms of the vector's global X, Y and Z components.

The angles the vector makes with the X, Y and Z axes are called Direction angles, whose cosines are called Direction cosines (or global X, Y and Z components used in CAEPIPE).

There are two methods of computing the X, Y and Z components.

First method: When you know the direction angles (see examples 1, 2 and 3).

Second method: When you know the coordinates of the end points of the vector (see example 4).

Example 1: Vertical Vessel

Assume a vertical vessel with axis in the Y direction, and α , β , γ as the direction angles the axis of the vessel makes with global X, Y and Z axes.

The angles are $\alpha = 90^\circ$, $\beta = 0^\circ$ (since axis is parallel to Y axis) and $\gamma = 90^\circ$.

So, the direction cosines or X, Y and Z components are

$$\text{X comp} = \cos(\alpha = 90^\circ) = 0,$$

$$\text{Y comp} = \cos(\beta = 0^\circ) = 1,$$

$$\text{Z comp} = \cos(\gamma = 90^\circ) = 0.$$

For Z vertical: X comp = 0, Y comp = 0 and Z comp = 1.

Example 2: Limit Stop at 45° from the X-axis in the X-Y plane

For a limit stop whose axis is oriented at 45° from the X-axis in the X-Y plane, the angles are $\alpha = 45^\circ$, $\beta = 45^\circ$ and $\gamma = 90^\circ$.

So, the direction cosines or X, Y and Z components are

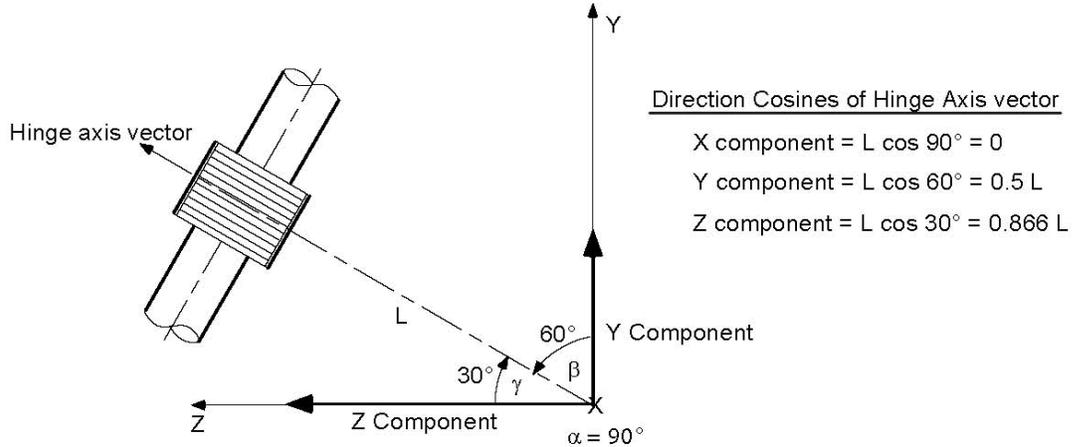
$$\text{X comp} = \cos(\alpha = 45^\circ) = 0.70711,$$

$$\text{Y comp} = \cos(\beta = 45^\circ) = 0.70711,$$

$$\text{Z comp} = \cos(\gamma = 90^\circ) = 0.0$$

Direction

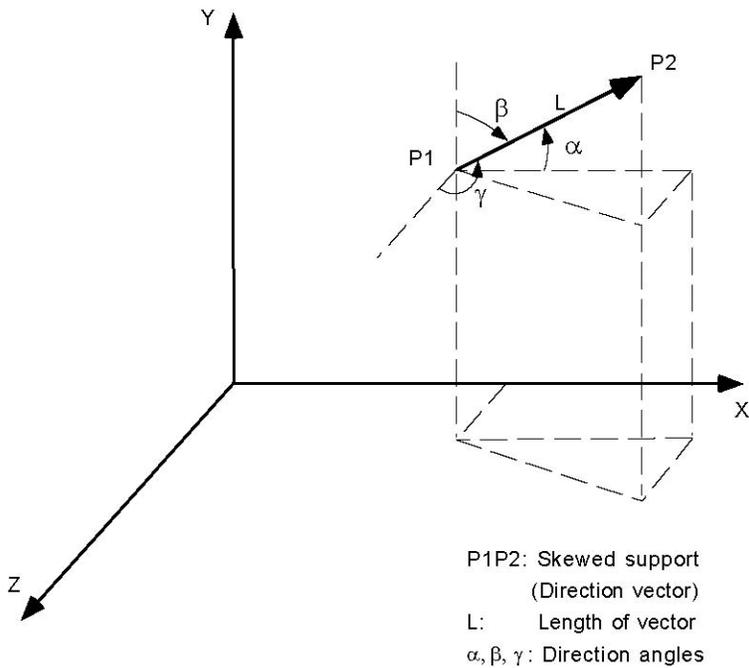
Example 3: Hinge axis at 30° from the Z-axis in the Y-Z plane



From the above figure, we have the angles $\alpha = 90^\circ$, $\beta = 60^\circ$ and $\gamma = 30^\circ$. Assuming $L = 1$ (or any length), the direction cosines or X, Y and Z components are

$$\begin{aligned} X \text{ comp} &= \cos (\alpha = 90^\circ) = 0.0, \\ Y \text{ comp} &= \cos (\beta = 60^\circ) = 0.5, \\ Z \text{ comp} &= \cos (\gamma = 30^\circ) = 0.866 \end{aligned}$$

Example 4: Skewed Support



Assume that we have a skewed support along P1P2 (which is the direction vector) shown in the figure above, Assume that the coordinates of these two points are $P1 = (12', 12', 12')$ and $P2 = (15', 16', 14')$.

Let us calculate this vector's global X, Y and Z components. There are two methods here

Short method

$$X \text{ comp} = (X_2 - X_1) = (15 - 12) = 3$$

$$Y \text{ comp} = (Y_2 - Y_1) = (16 - 12) = 4$$

$$Z \text{ comp} = (Z_2 - Z_1) = (14 - 12) = 2$$

Long method

First, let us calculate the length of the vector, L.

$$L = \sqrt{(X_2 - X_1)^2 + (Y_2 - Y_1)^2 + (Z_2 - Z_1)^2} = 5.385'$$

The angles α , β and γ which the vector makes with the global X, Y and Z axes are called the Direction angles of the vector; The cosines of these angles are called Direction cosines.

$$\cos \alpha = \frac{X_2 - X_1}{L}, \cos \beta = \frac{Y_2 - Y_1}{L}, \cos \gamma = \frac{Z_2 - Z_1}{L},$$

The direction cosines are

$$X \text{ comp} = \cos \alpha = 0.55709, Y \text{ comp} = \cos \beta = 0.74278, Z \text{ comp} = \cos \gamma = 0.37139$$

For information, the direction angles are $\alpha = 56^\circ 8''$, $\beta = 42^\circ 1''$ and $\gamma = 68^\circ 11''$.

To verify the results, the sum of the squares of the direction cosines must be 1.0. Thus,

$$\cos^2 \alpha + \cos^2 \beta + \cos^2 \gamma = 0.557092^2 + 0.742782^2 + 0.371392^2 = 1.0$$

Dynamic Analysis

Modal Analysis

The equations of motion for an undamped lumped mass system may be written as:

$$[M]\{\ddot{u}\} + [K]\{u\} = \{F(t)\} \quad (1)$$

Where $[M]$ = diagonal mass matrix

$\{u\}$ = displacement vector

$\{\ddot{u}\}$ = acceleration vector

$[K]$ = stiffness matrix

$\{F(t)\}$ = applied dynamic force vector

If the system is vibrating in a normal mode (i.e., free not forced vibration), we may make the substitutions

$$\begin{aligned} \{u\} &= \{a_n\} \sin \omega_n t \\ \{\ddot{u}\} &= -\omega_n^2 \{a_n\} \sin \omega_n t \\ \{F(t)\} &= 0 \end{aligned}$$

to obtain

$$-\omega_n^2 [M]\{a_n\} + [K]\{a_n\} = 0$$

or

$$[K]\{a_n\} = \omega_n^2 [M]\{a_n\} \quad (2)$$

where $\{a_n\}$ is the vector of modal displacements of the n^{th} mode (eigenvector).

Thus we have an eigenvalue (characteristic value) problem, and the roots of equation (2) are the eigenvalues (characteristic numbers), which are equal to the squares of the natural frequencies of the modes.

In CAEPIPE, the eigenvalue problem is solved using a determinant search technique. The solution algorithm combines triangular factorization and vector inverse iteration in an optimum manner to calculate the required eigenvalues and eigenvectors. These are obtained in sequence starting from the lowest eigenpair $\omega_1^2, \{a_1\}$. An efficient accelerated secant procedure which operates on the characteristic polynomial

$$p(\omega^2) = \det([K] - \omega^2 [M])$$

is used to obtain a shift near the next unknown eigenvalue. The eigenvalue separation theorem (Sturm sequence property) is used in this iteration. Each determinant evaluation requires a triangular factorization of the matrix $[K] - \omega^2 [M]$. Once a shift near the unknown eigenvalue has been obtained, inverse iteration is used to calculate the eigenvector. The eigenvalue is obtained by adding the Rayleigh quotient correction to the shift value. The eigenvector $\{a_n\}$, has an arbitrary magnitude and represents the characteristic shape of that mode.

Orthogonality

For any two roots corresponding to the n^{th} and m^{th} modes, we may write equation (2) as which is the orthogonality condition for eigenvectors.

$$[K]\{a_n\} = \omega_n^2 [M]\{a_n\} \quad (3)$$

$$[K]\{a_m\} = \omega_m^2 [M]\{a_m\} \quad (4)$$

If we postmultiply the transpose of (3) by $\{a_m\}$, we obtain

$$([K]\{a_n\})^T \{a_m\} = (\omega_n^2 [M]\{a_n\})^T \{a_m\} \quad (5)$$

or
$$\{a_n\}^T [K]^T \{a_m\} = \omega_n^2 \{a_n\}^T [M]^T \{a_m\}$$

Premultiplying (4) by $\{a_n\}^T$,
$$\{a_n\}^T [K]\{a_m\} = \omega_m^2 \{a_n\}^T [M]\{a_m\} \quad (6)$$

Since $[M]$ is a diagonal matrix, $[M] = [M]^T$. Also, since $[K]$ is a symmetric matrix, $[K] = [K]^T$. The left sides of equations (5) and (6) are therefore equal.

Subtracting (6) from (5),

$$(\omega_n^2 - \omega_m^2) \{a_n\}^T [M]\{a_m\} = 0 \quad (7)$$

Since $\omega_n \neq \omega_m$,

$$\{a_n\}^T [M]\{a_m\} = 0 \quad (8)$$

which is the orthogonality condition for eigenvectors.

Modal Equations

Since the eigenvectors (modal displacements) may be given any amplitude, it is convenient to replace $\{a_n\}$ by $\{\phi_n\}$ such that

$$\{\phi_n\}^T [M]\{\phi_n\} = 1 \quad (9)$$

The eigenvectors are evaluated so as to satisfy equation (9) and at the same time keep the displacements in the same proportion as those in $\{a_n\}$. The eigenvectors are then said to be *normalized*. Note that equation (7) is still satisfied since, if $n = m$, $\omega_n^2 - \omega_m^2 = 0$, and the remaining terms may be given any desired value.

Equation (2) now may be written for the n^{th} mode as

$$[K]\{\phi_n\} = \omega_n^2 [M]\{\phi_n\}$$

Let $[\Phi]$ be a square matrix containing all normalized eigenvectors such that the n^{th} column is the normalized eigenvector for the n^{th} mode. We can therefore write the matrix equation so as to include all modes as follows:

$$[K]\{\Phi\} = [M][\Phi][\omega_n^2] \quad (10)$$

Where $[\omega_n^2]$ is a diagonal matrix of eigenvalues. We now premultiply both sides of (10) by $[\Phi]^T$ to obtain

$$[\Phi]^T [K][\Phi] = [\Phi]^T [M][\Phi][\omega_n^2] \quad (11)$$

It may be shown that

$$[\Phi]^T [M] [\Phi] = [I] \quad (12)$$

Where I is the unit diagonal matrix. equation (12) can easily be verified by expansion and follows from the orthogonality condition and the fact that $[\Phi]$ has been normalized. equation (11) therefore can be written as

$$[\Phi]^T [K] [\Phi] = [\omega_n^2] \quad (13)$$

Returning now to the equation of motion (1),

$$\begin{aligned} \text{let } \{u\} &= [\Phi]\{A_n\} \\ \text{and } \{\ddot{u}\} &= [\Phi]\{\ddot{A}_n\} \end{aligned} \quad (14)$$

Where $\{A_n\}$ is the modal amplitude of the n^{th} mode. This merely states that the true modal displacements equal the characteristic displacements (eigenvector displacements) times the modal amplitude determined by the response calculations and, further that the total displacements are linear combinations of the modal values. If we now premultiply equation (1) by $[\Phi]^T$ and substitute equations (14), we obtain

$$[\Phi]^T [M] [\Phi] \{\ddot{A}_n\} + [\Phi]^T [K] [\Phi] \{A_n\} = [\Phi]^T \{F(t)\} \quad (15)$$

Substituting from equations (12) and (13) in equation (15),

$$\{\ddot{A}_n\} + [\omega_n^2] \{A_n\} = [\Phi]^T \{F(t)\} \quad (16)$$

which represents the modal equations of motion.

Support Motion

Solutions for support motion may be obtained if $\{F(t)\}$ is replaced by $-\ddot{u}_s(t)\{M\}$ where $\ddot{u}_s(t)$ is the prescribed support acceleration. Thus the modal equations of motion may be written as

$$\{\ddot{A}_n\} + [\omega_n^2] \{A_n\} = -\ddot{u}_s(t) [\Phi]^T \{M\} \quad (17)$$

Where A_n is the relative modal displacement for the n^{th} mode with respect to the support.

The participation factors for the modes are given by

$$\{\Gamma_n\} = [\Phi]^T \{M\} \quad (18)$$

Then the modal amplitude for the n^{th} mode is given by

$$A_n = \Gamma_n u_n^0 \quad (19)$$

where u_n^0 is the response of a single degree of freedom system having circular frequency ω_n . Using equations (14) and (19), the displacements are given by

$$\{u\} = [\Phi] \{A_n\} = [\Phi] \{\Gamma_n u_n^0\} \quad (20)$$

Effective Modal Mass

Effective modal mass is defined as the part of the total mass responding to the dynamic loading in each mode. When the participation factor is calculated using normalized eigenvectors as in equation (18), the effective modal mass for the n^{th} mode is simply the square of the normalized participation factor,

$$M_n = \Gamma_n^2$$

Effective modal mass is useful to verify if all the significant modes of vibration are included in the dynamic analysis by comparing the total effective modal mass with the total actual mass.

Response Spectrum

The concept of response spectrum, in recent years has gained wide acceptance in structural dynamics analysis, particularly in seismic design. Stated briefly, the response spectrum is a plot of the maximum response (maximum displacement, velocity, acceleration or any other quantity of interest), to a specified loading for all possible single degree-of-freedom systems. The abscissa of the spectrum is the natural frequency (or period) of the system, and the ordinate, the maximum response.

In general, response spectra are prepared by calculating the response to a specified excitation of single degree-of-freedom systems with various amounts of damping. Numerical integration with short time steps is used to calculate the response of the system. The step-by-step process is continued until the total earthquake record is completed and becomes the response of the system to that excitation. Changing the parameters of the system to change the natural frequency, the process is repeated and a new maximum response is recorded. This process is repeated until all frequencies of interest have been covered and the results plotted. Typically the El Centro, California earthquake of 1940 is used for this purpose.

Since the response spectra give only maximum response, only the maximum values for each mode are calculated and then superimposed (modal combination) to give total response. A conservative upper bound for the total response may be obtained by adding the absolute values of the maximum modal components (absolute sum). However this is excessively conservative and a more probable value of the maximum response is the square root of the sum of squares (SRSS) of the modal maxima.

To calculate response of the piping system, for each natural frequency of the piping system, the input spectrum is interpolated (linearly or logarithmically). The interpolated spectrum values are then combined for the X, Y and Z directions (direction sum) either as absolute sum or SRSS sum to give the maximum response of a single degree-of-freedom system: u_{\max}^0 at that frequency.

From equation (20), the maximum displacement vector for the n^{th} mode can be calculated from the maximum response of a single degree-of-freedom system,

$$\{u_n\}_{\max} = \{\Phi_n\} \Gamma_n u_n^0$$

The maximum values of element and support load forces per mode are calculated from the maximum displacements calculated per mode as above using the stiffness properties of the structure.

The total response (displacements and forces) is calculated by superimposing the modal responses according to the specified mode sum method which can be absolute sum, square root of sum of squares (SRSS) or closely spaced (10%) modes method.

Closely Spaced Modes

Studies have shown that SRSS procedure for combining modes can significantly underestimate the true response in certain cases in which some of the natural frequencies of a structural system are closely spaced. The ten percent method is one of NRC approved methods (Based on NRC Guide 1.92) for addressing this problem.

$$R = \sqrt{\sum_{n=1}^N R_n^2 + 2 \sum |R_i R_j|}$$

where R = Total (combined) response
 R_n = Peak value of the response due to the n^{th} mode
 N = Number of significant modes

The second summation is to be done on all i and j modes whose frequencies are closely spaced to each other. Let ω_i and ω_j be the frequencies of the i^{th} and j^{th} modes. The modes are closely spaced if:

$$\frac{\omega_j - \omega_i}{\omega_i} \leq 0.1 \text{ and } 1 \leq i \leq j \leq N$$

Time History

Time history analysis requires the solution to the equations

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F(t)\} \quad (21)$$

where $[M]$ = diagonal mass matrix
 $[C]$ = damping matrix
 $[K]$ = stiffness matrix
 $\{u\}$ = displacement vector
 $\{\dot{u}\}$ = velocity vector
 $\{\ddot{u}\}$ = acceleration vector
 $\{F(t)\}$ = applied dynamic force vector

Dynamic Analysis

The time history analysis is carried out using mode superposition method. It is assumed that the structural response can be described adequately by the p lowest vibration modes out of the total possible n vibration modes and $p < n$. Using the transformation $u = \Phi X$, where the columns in Φ are the p mass normalized eigenvectors, equation (21) can be written as

$$\ddot{X} + \Delta \dot{X} + \Omega^2 X = \Phi^T F \quad (22)$$

Where $\Delta = \text{diag}(2\omega_i \xi_i)$
 $\Omega^2 = \text{diag}(\omega_i^2)$

In equation (22), it is assumed that the damping matrix $[C]$ satisfies the modal orthogonality condition

$$\{\phi_i\}^T [C] \{\phi_j\} = 0 \quad (i \neq j)$$

Equation (22) therefore represents p uncoupled second order differential equations. These are solved using the Wilson θ method, which is an unconditionally stable step-by-step integration scheme. The same time step is used in the integration of all equations to simplify the calculations.

Harmonic Analysis

A harmonic analysis is performed to analyze the response to sinusoidal loads. Harmonic forces can arise from unbalanced rotating equipment, acoustic vibrations caused by reciprocating equipment, flow impedance, and other sources. These forces can be damaging to a piping system if their frequency is close to the system's natural frequency, thereby introducing resonant conditions. The equation of dynamic equilibrium associated with the response of the structure subjected to harmonic forces is:

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \sin(\omega t)F \quad (23)$$

where $[M]$ = diagonal mass matrix

$[C]$ = damping matrix

$[K]$ = stiffness matrix

$\{u\}$ = displacement vector

$\{\dot{u}\}$ = velocity vector

$\{\ddot{u}\}$ = acceleration vector

ω = frequency of the applied force

t = time

F = maximum magnitude of the applied force

More complex forms of vibration, such as those caused by the fluid flow, may be considered as superposition of several simple harmonics, each with its own frequency, magnitude, and phase.

A harmonic analysis uses the results from the modal analysis to obtain a solution. A single damping factor is used for all modes.

Dynamic Analysis

First, the maximum response of each harmonic is obtained separately. Then the total response of the system is determined by combining the individual responses. The combination method may be specified as the Root Mean Square (RMS) or Absolute Sum.

Dynamic Susceptibility

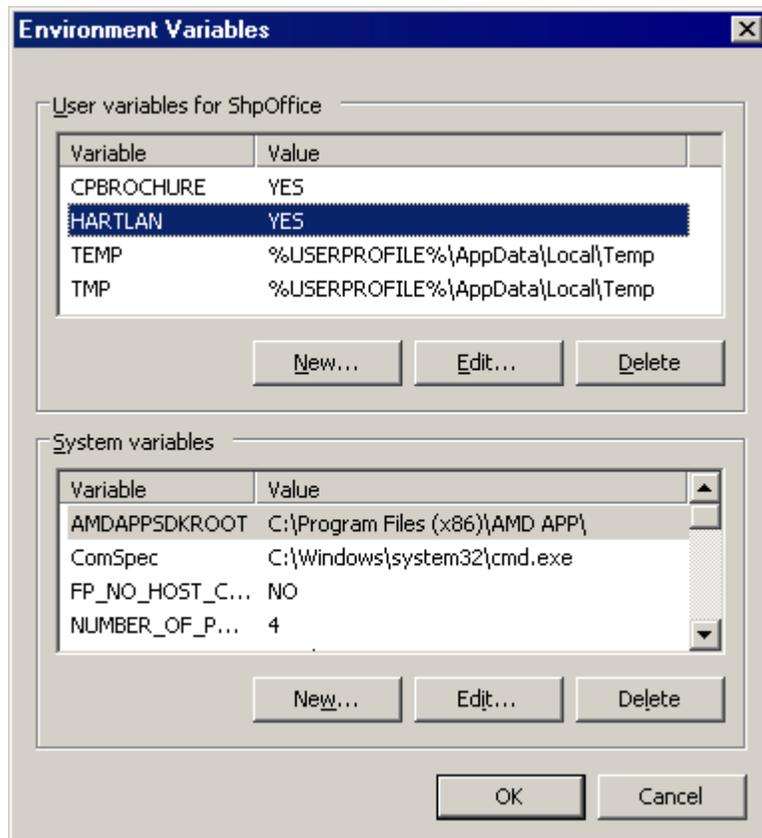
Dynamic Susceptibility feature is a screening tool for potentially large alternating stresses. The dynamic stresses are the dynamic bending stresses associated with vibration in a natural mode. In other words, the modal analysis result has been generalized to include the alternating bending stresses associated with the vibration in a natural mode. The dynamic susceptibility for any mode is the ratio of the maximum alternating bending stress to the maximum vibration velocity. This “susceptibility ratio” provides an indicator of the susceptibility of the system to large dynamic stresses. Also, the associated animated mode shapes include color-spot-markers identifying the respective locations of maximum vibration and maximum dynamic bending stress. The susceptibility ratio and the graphics feature provide incisive insights into the reasons for high susceptibility and how to make improvements.

The “Modal Analysis” output load case in CAEPIPE has been enhanced. In addition to the modal frequencies and mode shapes, you will see two new results items called “dynamic stresses” and “dynamic susceptibility.”

In case you do not see these two items in the results dialog, you need to activate this feature by defining an environment variable. See Appendix E for a detailed discussion.

First method:

An environment variable “HARTLEN” needs to be declared under My Computer > Properties > Environment > Variable (HARTLEN), and its Value set to (YES). Please check with your System Admin because different versions of Windows have slightly different methods of doing it.

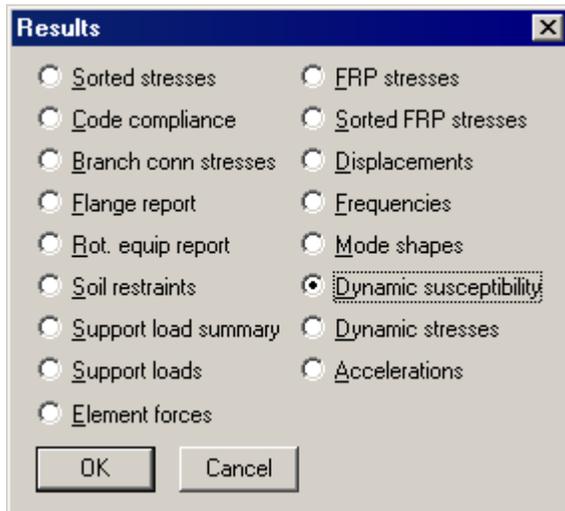


Dynamic Susceptibility

Second method:

Open the MS-DOS Command Prompt. Type “SET HARTLEN=YES” (enter), change directory (using CD command) to where CAEPIPE program files are located, start CAEPIPE.EXE.

Upon (modal) analysis, the Results dialog will display the required results (dynamic stresses and dynamic susceptibility).



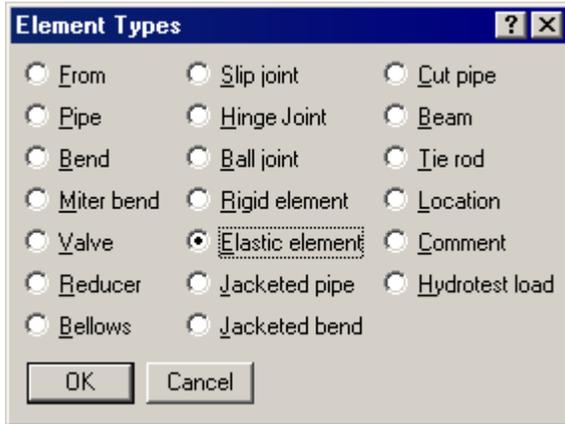
#	Mode	Frequency (Hz)	Maxima Velocity	Nodes Stress	Susceptibility (psi / ips)
1	3	3.593	130	70	3486
2	5	4.014	100	128	662
3	1	0.774	220B	250	330
4	2	0.830	210	250	298
5	6	4.781	240	240	282
6	4	3.601	210	240	81

#	Node	Displacement	Stress
Susceptibility = 282			
1	10	0.0000E+00	3.4879E+01
2	15	3.1792E-04	4.8861E+01
3	20A	9.0036E-04	8.4700E+01
4	20B	6.8968E-04	1.8018E+01
5	25	1.9210E-03	8.8395E+01
6	30	2.5995E-03	3.6917E+02

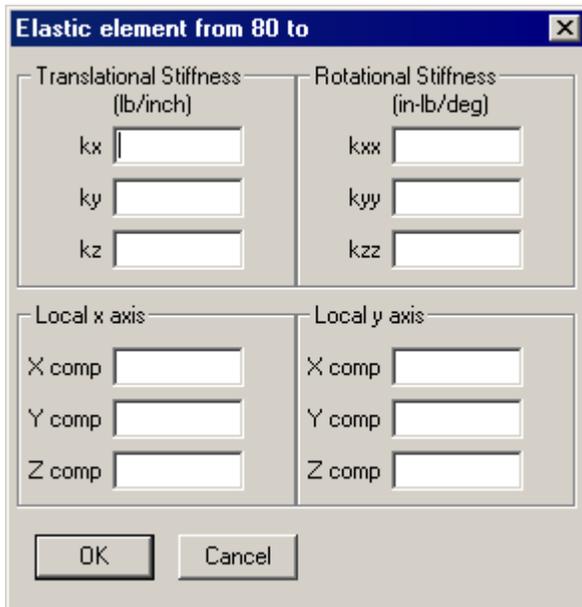
Elastic Element

The elastic element is a general 6×6 stiffness matrix, with nonzero diagonal terms and zero off-diagonal terms. Use this element to model the stiffness of a component unavailable in CAEPIPE.

An elastic element is input by typing “e” in the Type column or selecting “Elastic element” from the Element Types dialog.



The Elastic element dialog is shown.



The stiffnesses are in the local coordinate system defined by the directions of the local x-, y- and z-axes. The local y-axis should be perpendicular to the local x-axis (i.e., their dot product should be zero). The local z-axis is internally calculated as cross product of the local x- and y-axes.

The elastic element is not subjected to any sustained or thermal expansion loads.

See Slotted Hinge joint example under Expansion Joints topic.

Expansion Joints

CAEPIPE provides the following types of expansion joints:

1. Ball joint,
2. Bellows,
3. Hinge joint and
4. Slip joint

Using these types, you can also model tied bellows, a gimbal, a dual gimbal, a pressure-balanced elbow and a tee, a slotted hinge joint and a universal joint among other complex arrangements.

Before selecting the types of and locations for the expansion joints, you must study a piping system for the direction and magnitude of the thermal movements to be absorbed, availability of support structures for anchoring, and guiding of the piping. EJMA and manufacturer catalogs contain technical information that can guide you through this process. CAEPIPE becomes an ideal “what-if” tool for such rapid studies.

This topic will show how to model the following types of joints: Tied bellows, universal, gimbal, dual gimbal, pressure balanced joints and a slotted hinge joint. Also see discussion on Bellows.

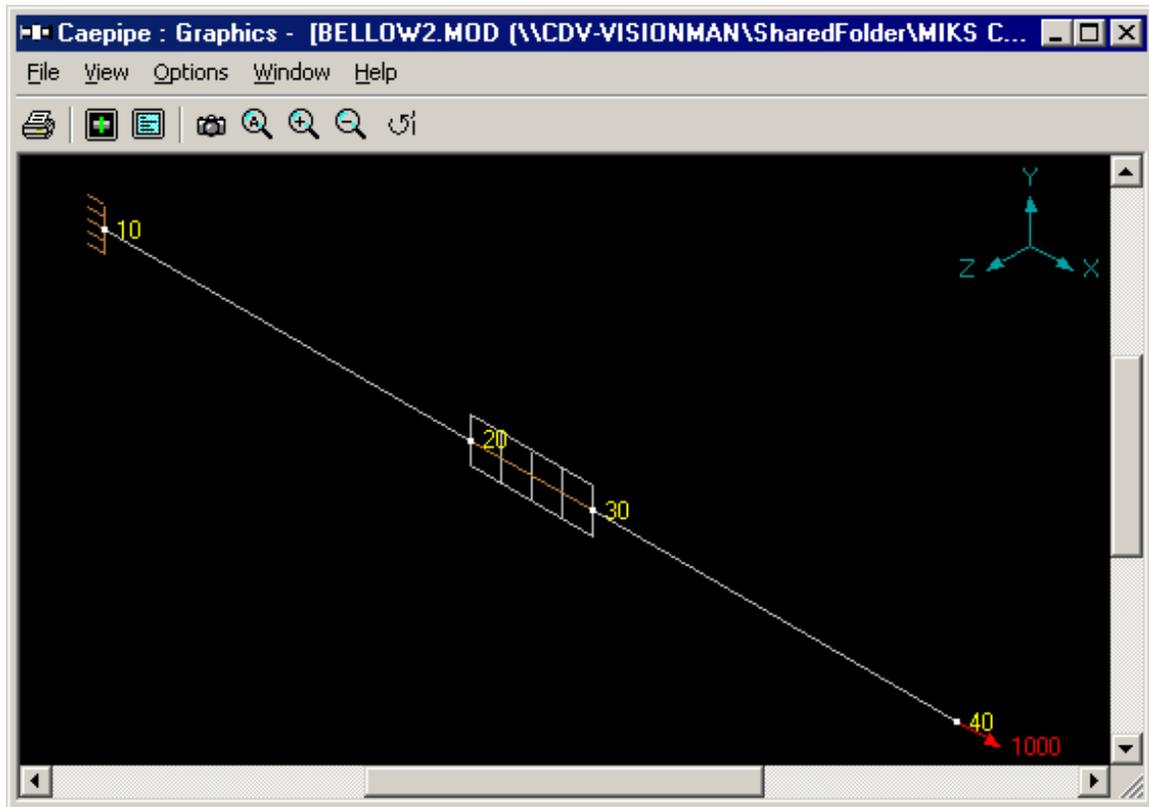
Tied Bellows:

Tie rods around bellows allow lateral deflection and absorb the full pressure thrust force in the event of an anchor failure. During normal operation, the adjacent equipment will see the pressure thrust force. Pressure thrust area must be input. One way of modeling the tie rods is to lump all four tie rods into a single limit stop (with limit stop stiffness = 4 x stiffness of tie rod = 4 x EA/L) at the center of the bellows. The reason for modeling the four tie rods as a single limit stop is that these rods act only in tension and not in compression. Simpler tie rod element could also be used. See last example in this section.

The screenshot shows the Caepipe software interface with a table of expansion joint components. The table has columns for Node, Type, DX (ft'in'), DY (ft'in'), DZ (ft'in'), Mat, Sect, Load, and Data. The first row is highlighted in yellow and contains the title 'Tied Bellows Example'. The subsequent rows define the components: Node 10 (From), Node 20 (3'0" DX), Node 30 (Bellows, 1'0" DX), Node 40 (3'0" DX), and Node 20 (Location, Limit stop). The table also includes columns for Material (Mat), Section (Sect), and Load, with values of 1 for all three in the relevant rows.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Mat	Sect	Load	Data
1	Title = Tied Bellows Example								
2	10	From							Anchor
3	20		3'0"			1	1	1	
4	30	Bellows	1'0"			1	1	1	
5	40		3'0"			1	1	1	Force
6	20	Location							Limit stop
7									

Expansion Joints



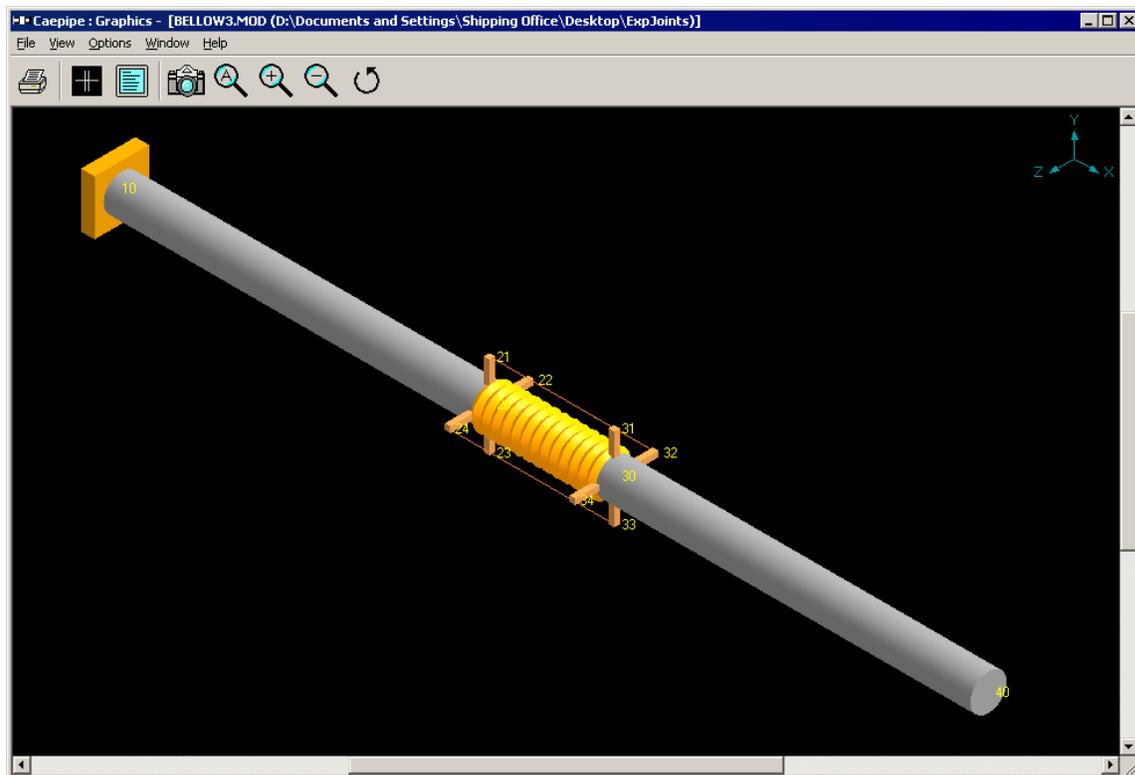
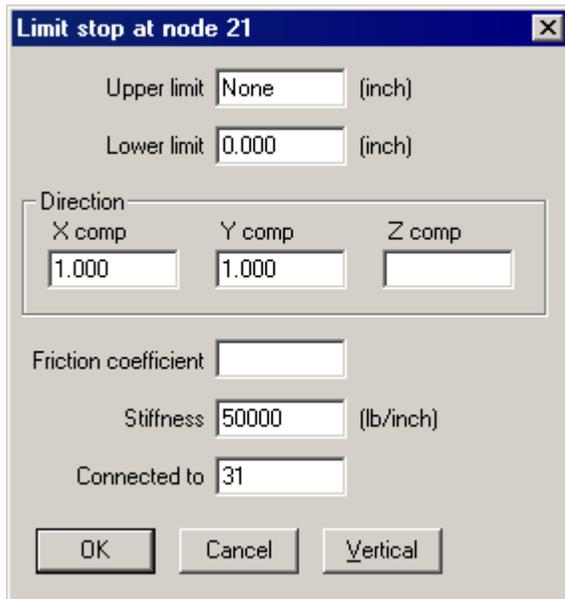
An elaborate method of modeling the above is to create four limit stops (instead of one) placed at their actual locations with rigid elements. As noted before, instead of using Limit stops, you could also use the simpler tie rod element(s). See next example.

Expansion Joints

Caepipe : Layout (25) - [BELLOW3.MOD (\\CDV-VISIO...]									
File Edit View Options Loads Misc Window Help									
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Tied Bellows Example								
2	10	From							Anchor
3	20		3'0"			1	1	1	
4	30	Bellows	1'0"			1	1	1	
5	40		3'0"			1	1	1	Force
6	20	From							
7	21	Rigid		0'4"		1	2	1	
8	20	From							
9	22	Rigid			-0'4"	1	2	1	
10	20	From							
11	23	Rigid		-0'4"		1	2	1	
12	20	From							
13	24	Rigid			0'4"	1	2	1	
14	30	From							
15	31	Rigid		0'4"		1	2	1	
16	30	From							
17	32	Rigid			-0'4"	1	2	1	
18	30	From							
19	33	Rigid		-0'4"		1	2	1	
20	30	From							
21	34	Rigid			0'4"	1	2	1	
22	21	Location							Limit stop
23	22	Location							Limit stop
24	23	Location							Limit stop
25	24	Location							Limit stop
26									

Expansion Joints

The limit stops at nodes 21 (shown below), 22, 23 and 24 should be modeled thus.



Universal joint:

This joint is composed of two single bellows with a center spool, with tie rods (90° apart) across the whole assembly (end to end). This is commonly used in multi-plane Z bend configurations to absorb large lateral deflections. Axial growth in the vertical leg (that contains the joint) will introduce bending in the horizontal legs which may require other supports.

Expansion Joints

The amount of lateral deflection can be increased by increasing the length of the center pipe section or the spool piece. The angulation or bending of the bellows can be controlled by the number of tie rods around the joint. See a manufacturer's catalog for details.

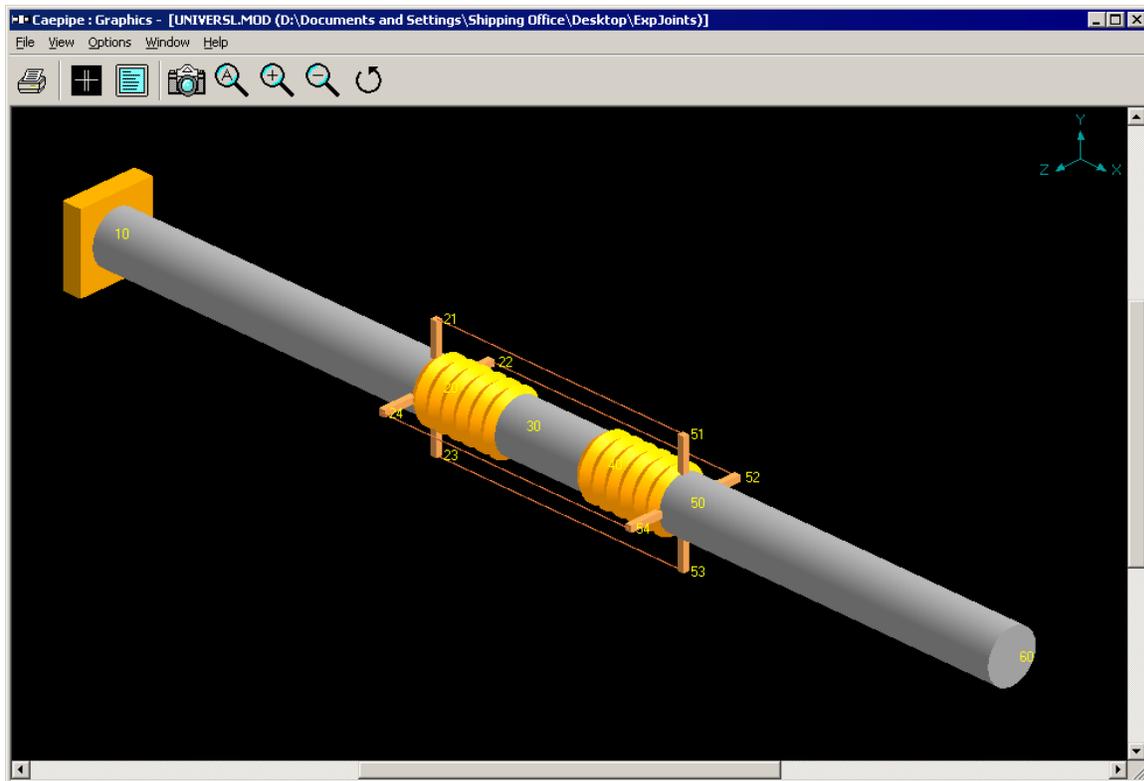
If a joint comes with no tie rods, then the thermal movement of the piping must be properly guided so as to have the joint behave as expected.

As mentioned in the earlier example, you can use tie rod elements instead of limit stops.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Tied Universal Expansion Joint								
2	10	From							Anchor
3	20		2'0"			1	1	1	
4	30	Bellows	0'6"			1	1	1	
5	40		0'6"			1	1	1	
6	50	Bellows	0'6"			1	1	1	
7	60		2'0"			1	1	1	Force
8	20	From							
9	21	Rigid		0'4"		1	2	1	
10	20	From							
11	22	Rigid			-0'4"	1	2	1	
12	20	From							
13	23	Rigid		-0'4"		1	2	1	
14	20	From							
15	24	Rigid			0'4"	1	2	1	
16	50	From							
17	51	Rigid		0'4"		1	2	1	
18	50	From							
19	52	Rigid			-0'4"	1	2	1	
20	50	From							
21	53	Rigid		-0'4"		1	2	1	
22	50	From							
23	54	Rigid			0'4"	1	2	1	
24	21	Location							Limit stop
25	22	Location							Limit stop
26	23	Location							Limit stop
27	24	Location							Limit stop
28									

Limit stops are modeled as before.

Expansion Joints



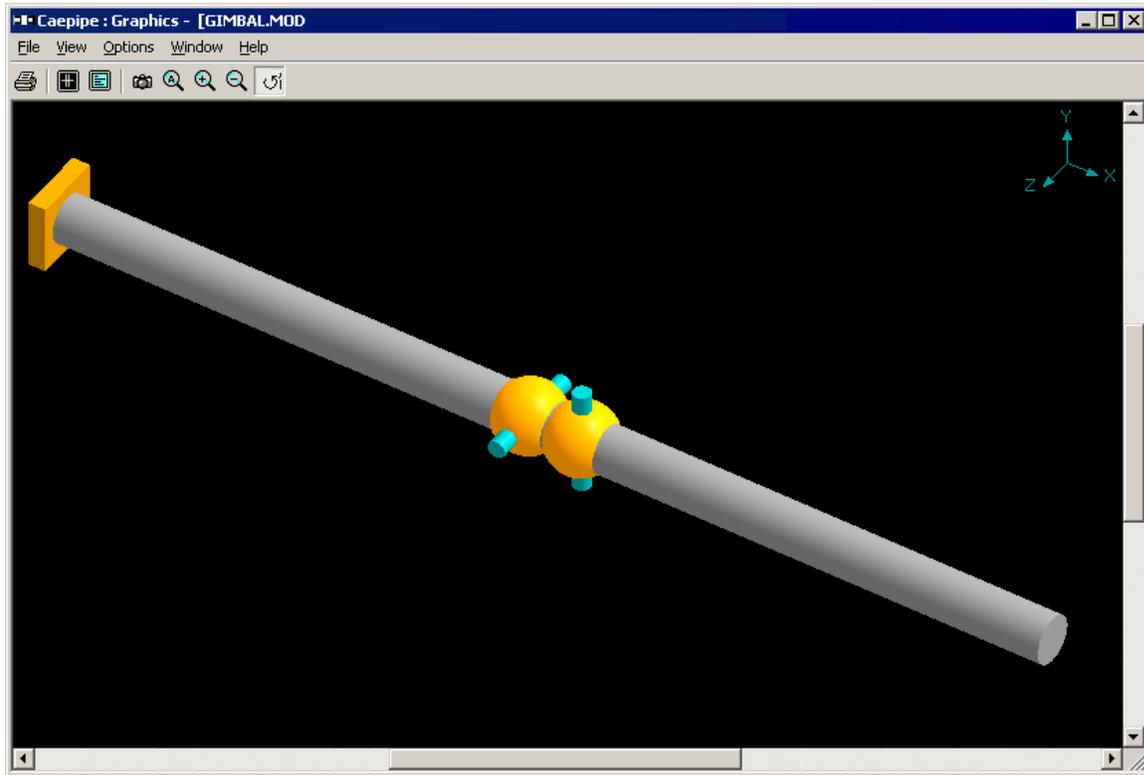
Gimbal, Dual Gimbal Joints:

These allow angular motion in all planes and resist axial and pressure thrust forces. These are modeled in CAEPIPE as two back to back hinge joints with perpendicular axes. The minimum distance between the adjacent gimbal joints is a function of the required angular and rotational deformation. The 4" shown in the next two examples is only for graphical depiction, and not representative of any functional requirement.

A gimbal joint example follows.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Gimbal Example								
2	10	From							Anchor
3	20		3'0"			1	1	1	
4	30	Hinge							
5	35		0'4"			1	1	1	
6	40	Hinge							
7	50		3'0"			1	1	1	Force
8									

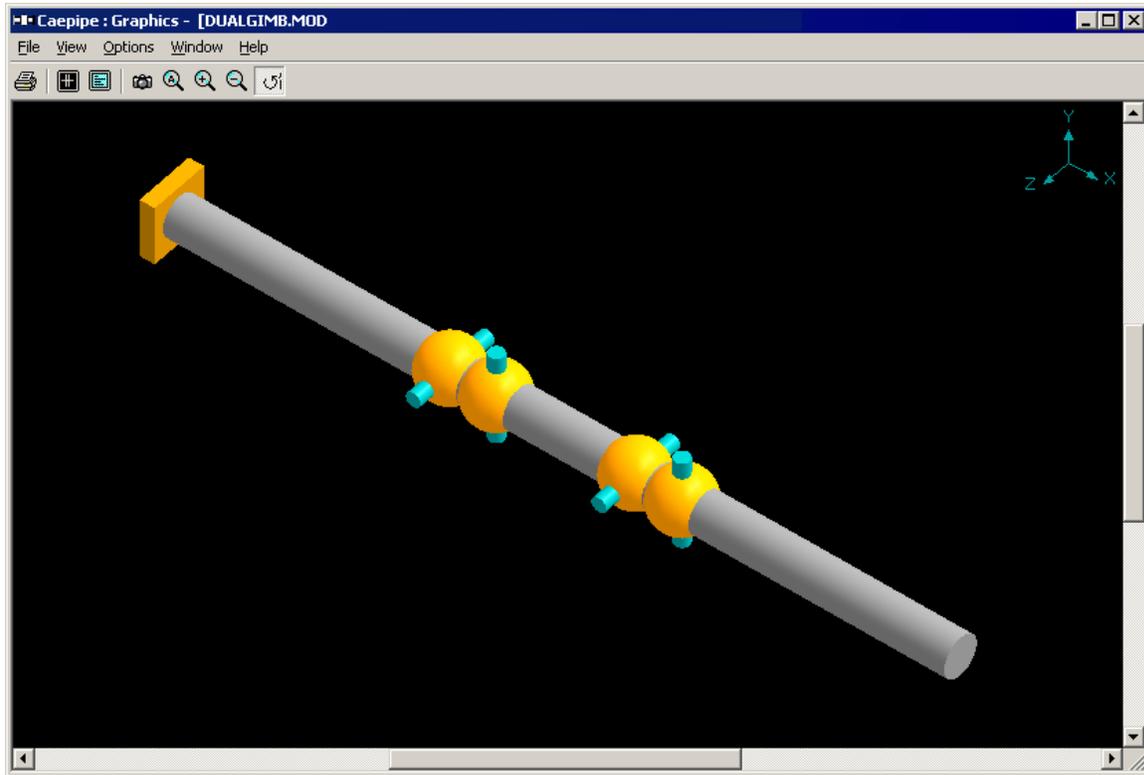
Expansion Joints



A Dual gimbal joint example follows:

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Dual Gimbal Joint								
2	10	From							Anchor
3	20		2'0"			1	1	1	
4	30	Hinge							
5	40	Hinge							
6	50		1'0"			1	1	1	
7	60	Hinge							
8	65		0'4"			1	1	1	
9	70	Hinge							
10	80		2'0"			1	1	1	Force
11									

Expansion Joints



Pressure-balanced Elbows and Tees:

These joints are used to absorb small amounts of axial and lateral movement at a change in direction while restraining the pressure thrust by means of tie rods. It is often used to reduce the pressure thrust forces coming on nearby equipment (rotating) whose allowable nozzle loads necessitate elimination of pressure thrust forces. You can use tie rods instead of limit stops. Two examples follow - a pressure-balanced tee followed by a pressure-balanced elbow.

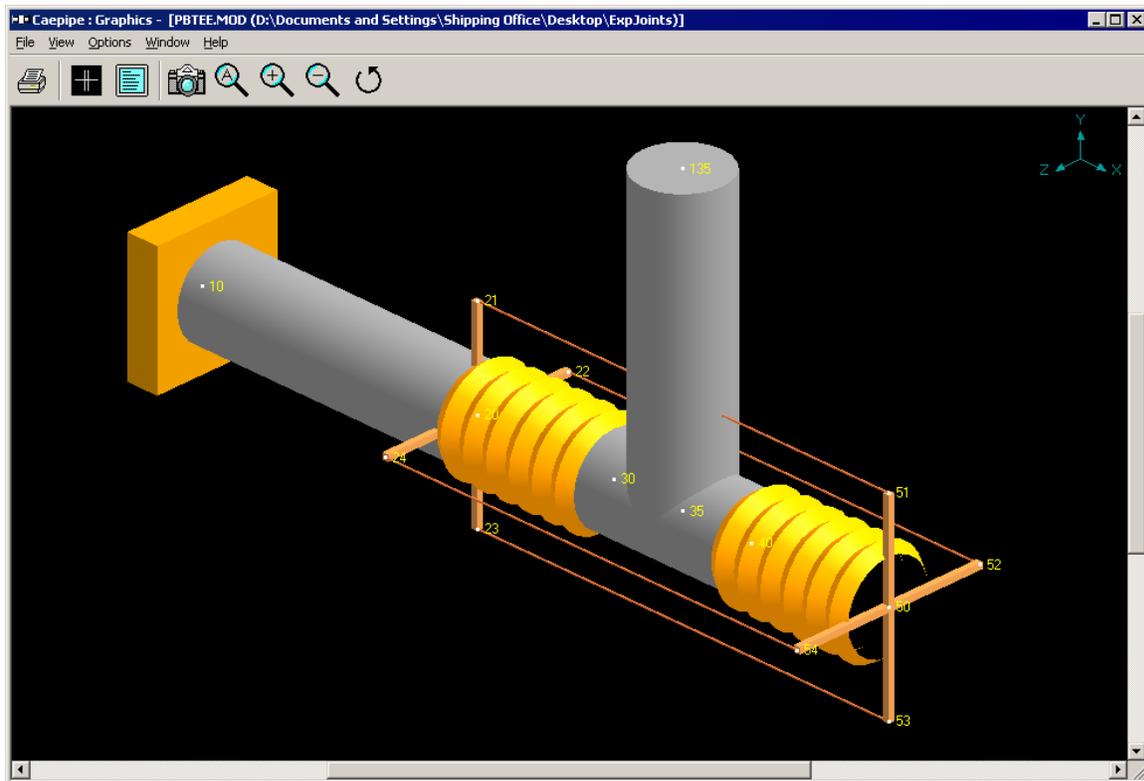
Expansion Joints

Pressure-balanced Tee:

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Pressure Balanced Tee								
2	10	From							Anchor
3	20		1'0"			1	1	1	
4	30	Bellows	0'6"			1	1	1	
5	35		0'3"			1	1	1	
6	40		0'3"			1	1	1	
7	50	Bellows	0'6"			1	1	1	
8	35	From							Welding tee
9	135			1'0"		1	1	1	
10	20	From							
11	21	Rigid		0'4"		1	2	1	
12	20	From							
13	22	Rigid			-0'4"	1	2	1	
14	20	From							
15	23	Rigid		-0'4"		1	2	1	
16	20	From							
17	24	Rigid			0'4"	1	2	1	
18	50	From							
19	51	Rigid		0'4"		1	2	1	
20	50	From							
21	52	Rigid			-0'4"	1	2	1	
22	50	From							
23	53	Rigid		-0'4"		1	2	1	
24	50	From							
25	54	Rigid			0'4"	1	2	1	
26	21	Location							Limit stop
27	22	Location							Limit stop
28	23	Location							Limit stop
29	24	Location							Limit stop
30									

Limit stops are modeled as before.

Expansion Joints



Pressure-balanced Elbow:

A pressure-balanced elbow absorbs axial and lateral movements at a change in direction while it restrains the axial pressure thrust on nearby (rotating) equipment.

The joint comprises of two bellows, one on either side of the elbow. Flow bellows, as the name suggests, is on the flow side, while the balancing bellows extends from the elbow out, subject to the same line pressure, secured to the flow bellows by tie rods. The balancing bellows expands by the same amount that the flow bellows compresses (because of the tie rod arrangement), with no volume change thus keeping the pressure forces in balance; hence the name pressure-balanced elbow.

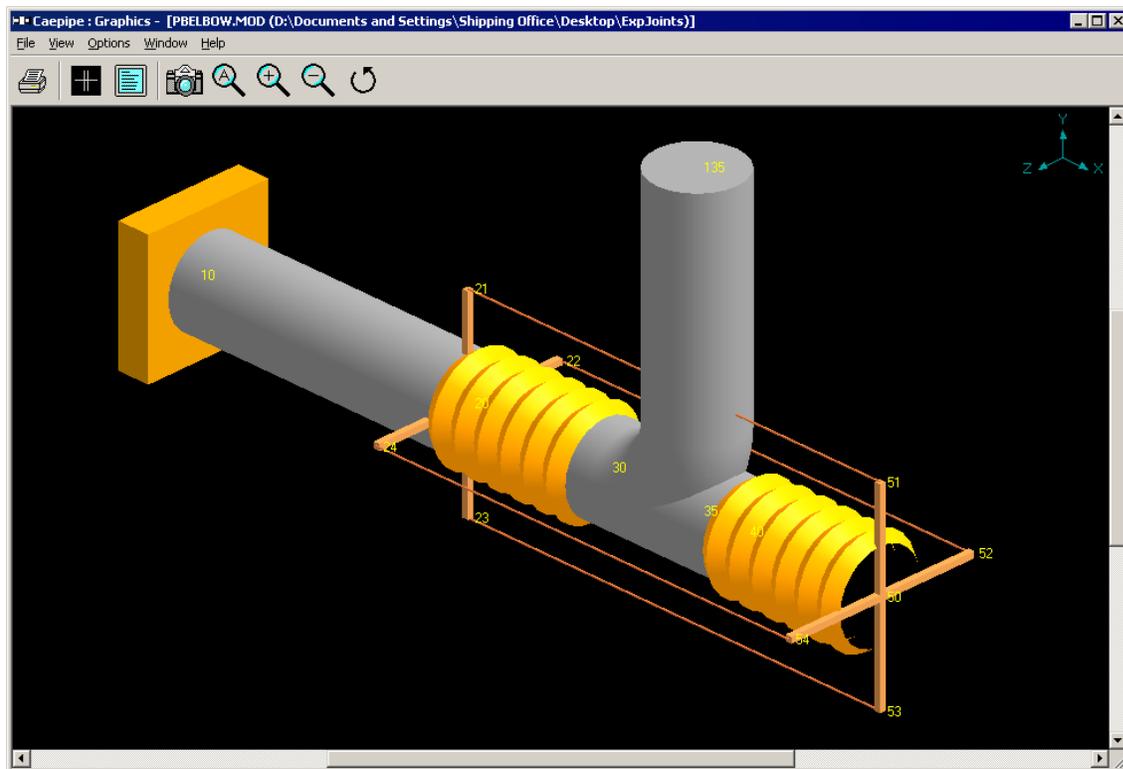
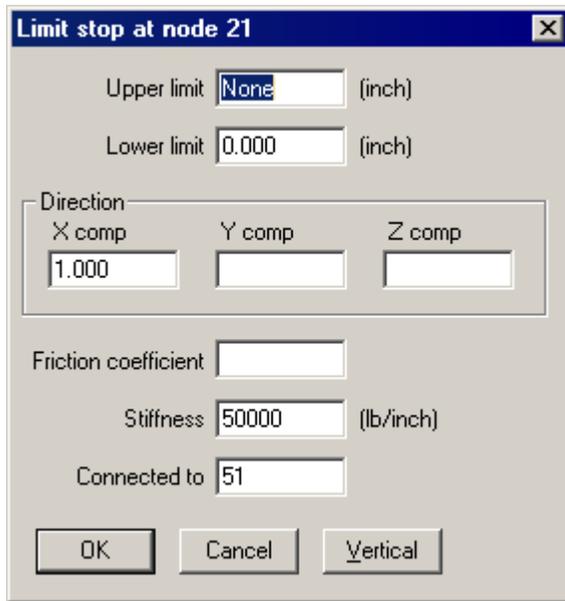
To allow a larger lateral movement, one should use a pressure-balanced universal expansion joint. The difference here is that there are two bellows on the flow side instead of one, with still one balancing bellows.

Modeling either of these joints is simple once you become aware of the above information.

Expansion Joints

Caepipe : Layout (29) - [PBELBOW.MOD (\CDV-VISION...)]									
File Edit View Options Loads Misc Window Help									
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Pressure Balanced Elbow								
2	10	From							Anchor
3	20		1'0"			1	1	1	
4	30	Bellows	0'6"			1	1	1	
5	40		0'6"			1	1	1	
6	50	Bellows	0'6"			1	1	1	
7	30	From							
8	35	Bend	0'4"			1	1	1	
9	135			1'0"		1	1	1	
10	20	From							
11	21	Rigid		0'4"		1	2	1	
12	20	From							
13	22	Rigid			-0'4"	1	2	1	
14	20	From							
15	23	Rigid		-0'4"		1	2	1	
16	20	From							
17	24	Rigid			0'4"	1	2	1	
18	50	From							
19	51	Rigid		0'4"		1	2	1	
20	50	From							
21	52	Rigid			-0'4"	1	2	1	
22	50	From							
23	53	Rigid		-0'4"		1	2	1	
24	50	From							
25	54	Rigid			0'4"	1	2	1	
26	21	Location							Limit stop
27	22	Location							Limit stop
28	23	Location							Limit stop
29	24	Location							Limit stop
30									

Expansion Joints



A few points to note:

1. The Bend node (35) is NOT on the pipe element 30-40. Notice the rows in the layout window. The bellows modeling is from nodes 20-30-40-50. Bend modeling starts on row 7 "From" 30-35-135.
2. Depending on how precise you want to be in your modeling of the tie rods, you have three choices:
 - a) Simply input one tie rod from 20-50. Input the combined stiffnesses of all the tie rods around the bellows, or

Expansion Joints

- b) As shown in the model graphics, use massless rigid elements to model the offsets (such as from 20-21, 20-24 etc.) on both sides of the joint, and tie corresponding nodes with Limit stops. Note that a limit stop takes only one stiffness input for both in tension and in compression, or
- c) Replace limit stops in item b. with tie rods. A tie rod allows stiffness inputs for tension and compression, allowing for more flexibility in input. Note that CAEPIPE does not require Section and Material input for tie rods.

Slotted Hinge joint:

In CAEPIPE, this joint is modeled using an elastic element, a limit stop and a hinge joint. The limit stop models the slot (+ and – 1 inch, for example) thereby allowing for limited axial travel. The elastic element has very low axial stiffness and all other stiffnesses rigid to model the slot. The hinge provides rotational stiffness about the Z axis. Due to the slots, these joints will not resist any pressure thrust forces thereby necessitating anchoring.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Mat	Sect	Load	Data
1	Title = Slotted Hinge Joint								
2	10	From							Anchor
3	15		3'0"			1	1	1	Limit stop
4	20	Elastic	0.0100						
5	30	Hinge							
6	40		3'0"			1	1	1	Force
7									

Elastic element from 15 to 20 [X]

<p>Translational Stiffness (lb/inch)</p> <p>kx <input type="text" value="1"/></p> <p>ky <input type="text" value="Rigid"/></p> <p>kz <input type="text" value="Rigid"/></p>	<p>Rotational Stiffness (in-lb/deg)</p> <p>kxx <input type="text" value="Rigid"/></p> <p>kyy <input type="text" value="Rigid"/></p> <p>kzz <input type="text" value="Rigid"/></p>
<p>Local x axis</p> <p>X comp <input type="text" value="1.0000"/></p> <p>Y comp <input type="text" value="0.0000"/></p> <p>Z comp <input type="text" value="0.0000"/></p>	<p>Local y axis</p> <p>X comp <input type="text" value="0.0000"/></p> <p>Y comp <input type="text" value="1.0000"/></p> <p>Z comp <input type="text" value="0.0000"/></p>

Expansion Joints

Hinge joint from 20 to 30 ✕

Rotational stiffness (in-lb/deg)

Rotation limit (deg)

Friction torque (ft-lb)

Weight (lb)

Axis direction

X comp	Y comp	Z comp
<input type="text"/>	<input type="text"/>	<input type="text" value="1.000"/>

Limit stop at node 15 ✕

Upper limit (inch)

Lower limit (inch)

Direction

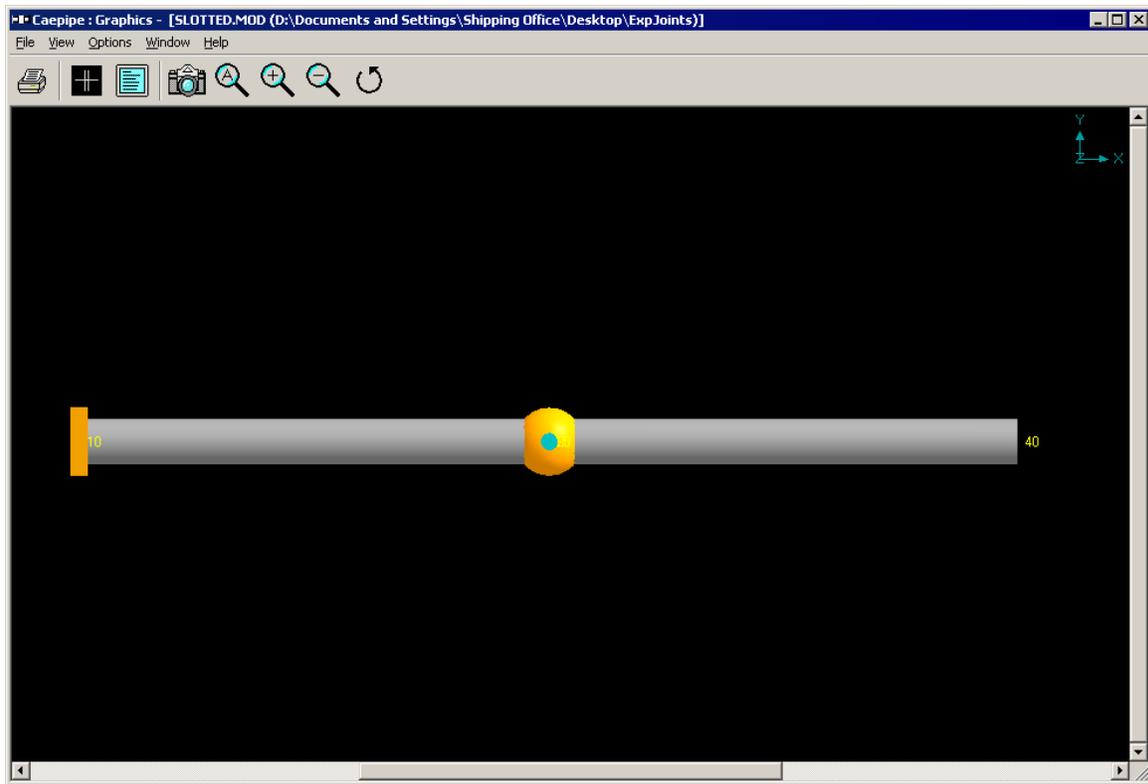
X comp	Y comp	Z comp
<input type="text"/>	<input type="text"/>	<input type="text"/>

Friction coefficient

Stiffness (lb/inch)

Connected to

Expansion Joints



Modeling Tie Rods:

A tie rod's chief function is to restrain the axial pressure thrust at an expansion joint (e.g., bellows). A tie rod between the ends of an expansion joint could be modeled to allow for axial compression but prevent axial expansion due to pressure thrust. To create this effect, you would need to model a tie rod *and* enter pressure thrust area for the bellows expansion joint.

For a tie rod, enter stiffnesses in tension and compression. To prevent the expansion joint from going into tension, input a Rigid stiffness (1E12 lb/in) for the Tension Stiffness field. Leave the Compression Stiffness blank if you want to allow the expansion joint to freely compress. Instead of making the Tension stiffness rigid, you could calculate the total stiffness of all the rods and enter it for Tension Stiffness, too.

In some cases, the expansion joint may extend or contract a specific amount before it stops completely. To model this behavior, i.e., if you want to prevent the expansion joint from extending or contracting freely in a direction, input a gap in that direction and a corresponding stiffness.

How to model:

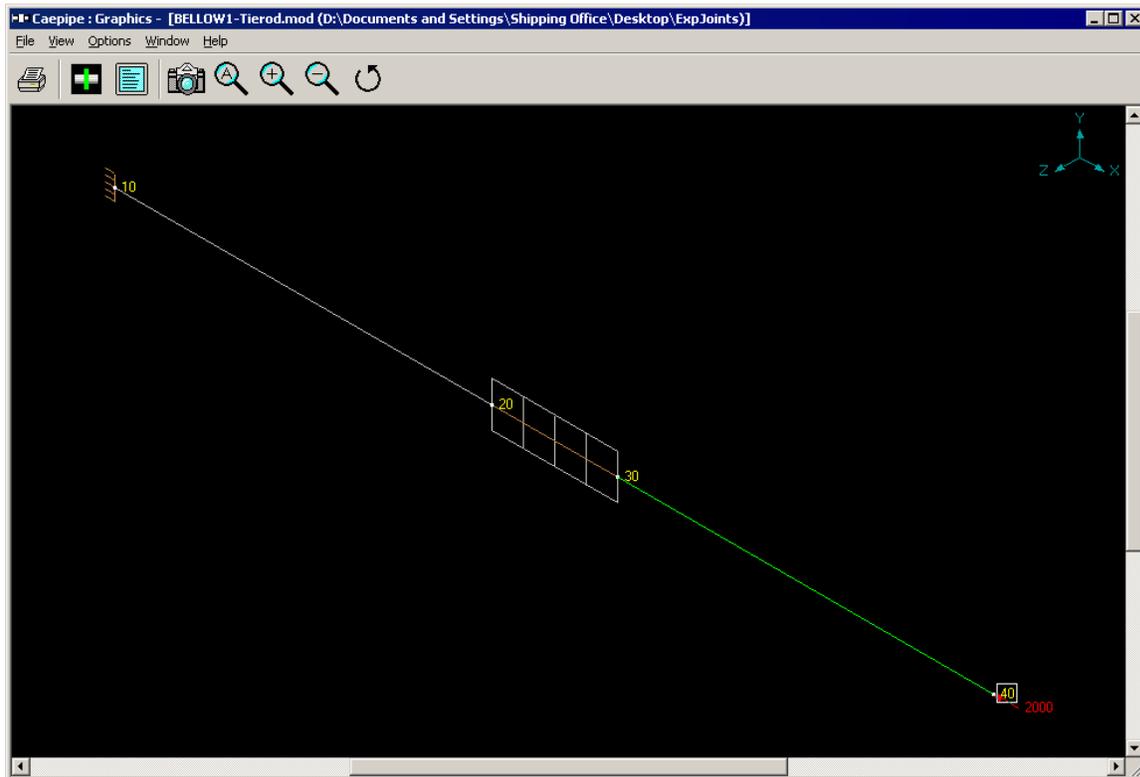
A tie rod is modeled between the same nodes as the expansion joint is. Next, go to a new row, type in the starting node number of the joint, specify "From" under "Type" followed by Enter. On the next new row, type the other node number of the joint, followed by "Tie Rod" under "Type". There is no need to input the offsets (DX, DY or DZ) as CAEPIPE already has the coordinates of these nodes from the previous expansion joint definition.

Expansion Joints

Caepipe : Layout (7) - [BELLOW1.MOD (\ACDV-VISI...]

File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Tied Bellows Example								
2	10	From							Anchor
3	20		3'0"			1	1	1	
4	30	Bellows	1'0"			1	1	1	
5	40		3'0"			1	1	1	Force
6	Tierod modeled here between the bellows nodes								
7	20	From							
8	30	Tie rod							



Tie rod from 20 to 30 [?] [X]

	Tension	Compression	
Stiffness	<input type="text" value="1e12"/>	<input type="text" value="0"/>	(lb/inch)
Gap	<input type="text" value="0"/>	<input type="text" value="0"/>	(inch)

Expansion Joints

Bellows from 20 to 30 ✕

Axial stiffness (lb/inch)

Bending stiffness (in-lb/deg)

Torsional stiffness (in-lb/deg)

Lateral stiffness (lb/inch)

Pressure thrust area (in²)

Weight (lb)

Fiber Reinforced Plastic Piping (FRP)

FRP piping has gained wide acceptance in many industries due to its lightweight nature, superior corrosion resistance, temperature capabilities and mechanical strength. Several manufacturers produce different types of FRP pipes and fittings and provide technical assistance to their customers on

design matters through installation. You can model FRP materials in CAEPIPE and have it calculate deflections, forces, moments and stresses.

To define the FRP material, click on “Matl” in the header row in the Layout window.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
33	150		1'0"			1	3	1	Nozzle
34	35	From							Reinf tee

In the Material List window that is shown, double click on an empty row to input a new material or on a material description to edit the material properties.

#	Name	Description	Type	Density (lb/in ³)	Nu	Joint factor	#	Temp (F)	Axial Mod. (psi)	Hoop Mod. (psi)	Shear Mod. (psi)
1	1	A106 Grade A	CS	0.280	0.3	1.00	1	20	1.16E+6	2.64E+6	1.09E+6
2	JKT	Titanium B337 Grade 12	TI	0.164	0.3	1.00	2	700	1.16E+6	2.64E+6	1.09E+6
3	P91	A335 Grade P91 (9Cr-1Mo-V)	CS	0.280	0.3	1.00	3				
4	FRP1	Duct	FR	0.052	0.25	1.00					
5											

The Material dialog will be shown.

Material # 4

Material name: FRP1

Description: Duct

Type: FR: Fiber Reinf. Plastic (FRP)

Density: 0.052 (lb/in³)

Nu: 0.25

Joint factor: 1.00

Select Type: FR here before clicking on properties

Buttons: OK, Cancel, Properties, Library

Fiber Reinforced Plastic Piping (FRP)

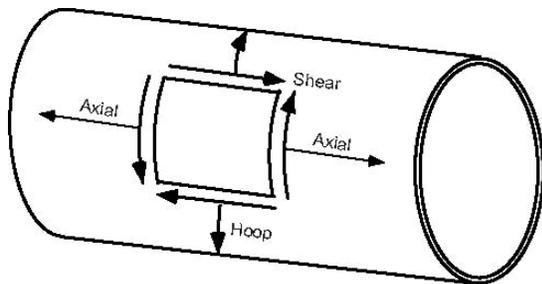
The material name can be up to three alpha-numeric characters. Enter description and density. You need to select “FR: Fiber Reinf. Plastic (FRP)” from the Type drop-down combo box before you click on the Properties button. The Poisson’s ratio (ν) is the ratio of the strain in the axial direction resulting from the stress in the hoop direction.

When you click on the Properties button, you are shown the table below where you enter temperature-dependent properties. Additionally, you can define the Axial and Torsional allowable stresses so that CAEPIPE can use them to compare with calculated stresses under the FRP "Sorted Stresses" results.

Material Properties							
#	Temp (F)	Axial Mod. (psi)	Hoop Mod. (psi)	Shear Mod. (psi)	Alpha (in/in/F)	Axial All. (psi)	Torsional All. (psi)
1	20	1.16E+6	2.64E+6	1.09E+6	16.26E-6	3000	7500
2	700	1.16E+6	2.64E+6	1.09E+6	16.26E-6	1500	2500

FRP Material Moduli

CAEPIPE requires three moduli for an FRP material:



- Axial or Longitudinal (this is the most important one)
- Hoop (used in Bourdon effect calculations). If this modulus is not available, use axial modulus.
- Shear or Torsional. If this modulus is not available, use engineering judgment in putting 1/2 of axial modulus or a similar value. Note that a high modulus will result in high stresses, and a low modulus will result in high deflections.

For FRP bends, a Flexibility factor of 1.0 is used unless you override it by specifying a Flexibility factor inside the bend dialog. Also for FRP bends, CAEPIPE uses a default SIF of 2.3. You can override this too by specifying User-SIFs at the bend end nodes (A and B nodes).

Stiffness matrix

The stiffness matrix for an FRP material is formulated in the following manner:

The stiffness matrix for a pipe is calculated using the following terms:

$$\text{Axial term} = L / EA$$

$$\text{Shear term} = \text{shape factor} \times L / GA$$

$$\text{Bending term} = L / EI$$

$$\text{Torsion term} = L / 2GI$$

where L = length, A = area, I = moment of inertia

Fiber Reinforced Plastic Piping (FRP)

E = Elastic modulus, G = shear modulus

For an isotropic material,

$$G = E / 2(1 + \nu)$$

where ν = Poisson's ratio,

For an FRP material, however,

E = axial modulus and G is independently specified (i.e., it is not calculated using E and ν).

The hoop modulus and FRP Poisson's ratio are only used in Bourdon effect calculation where,

Poisson's ratio used = FRP Poisson's ratio input x axial modulus / hoop modulus

Results

CAEPIPE calculates deflections, forces, moments and stresses. Each item can be seen under the respective title in Results. FRP element stresses can be seen, sorted or unsorted. These different stresses are computed as per formulas given towards the end of Appendix A.

#	Node	Hoop (psi)	Axial (psi)	Bending (psi)	Longitudinal Max (psi)	Longitudinal Min (psi)	Torsional (psi)
1	10 20	11538 11538	5500 5500	3121 923	8620 6423	2379 4577	141 141
2	20 25	11538 11538	5500 5500	923 1265	6423 6765	4577 4235	141 141
3	25 30A	11538 11538	5500 5500	1265 1814	6765 7314	4235 3686	141 141
4	30A 30B	11538 11538	5502 5497	4172 4248	9675 9745	1330 1249	141 141
5	30B 50	11538 11538	5494 5494	1847 1306	7341 6801	3647 4188	141 141
6	50 60	11538 11538	5494 5494	1306 898	6801 6392	4188 4597	141 141
7	60 70	11538 11538	5494 5494	898 3187	6392 8681	4597 2308	141 141

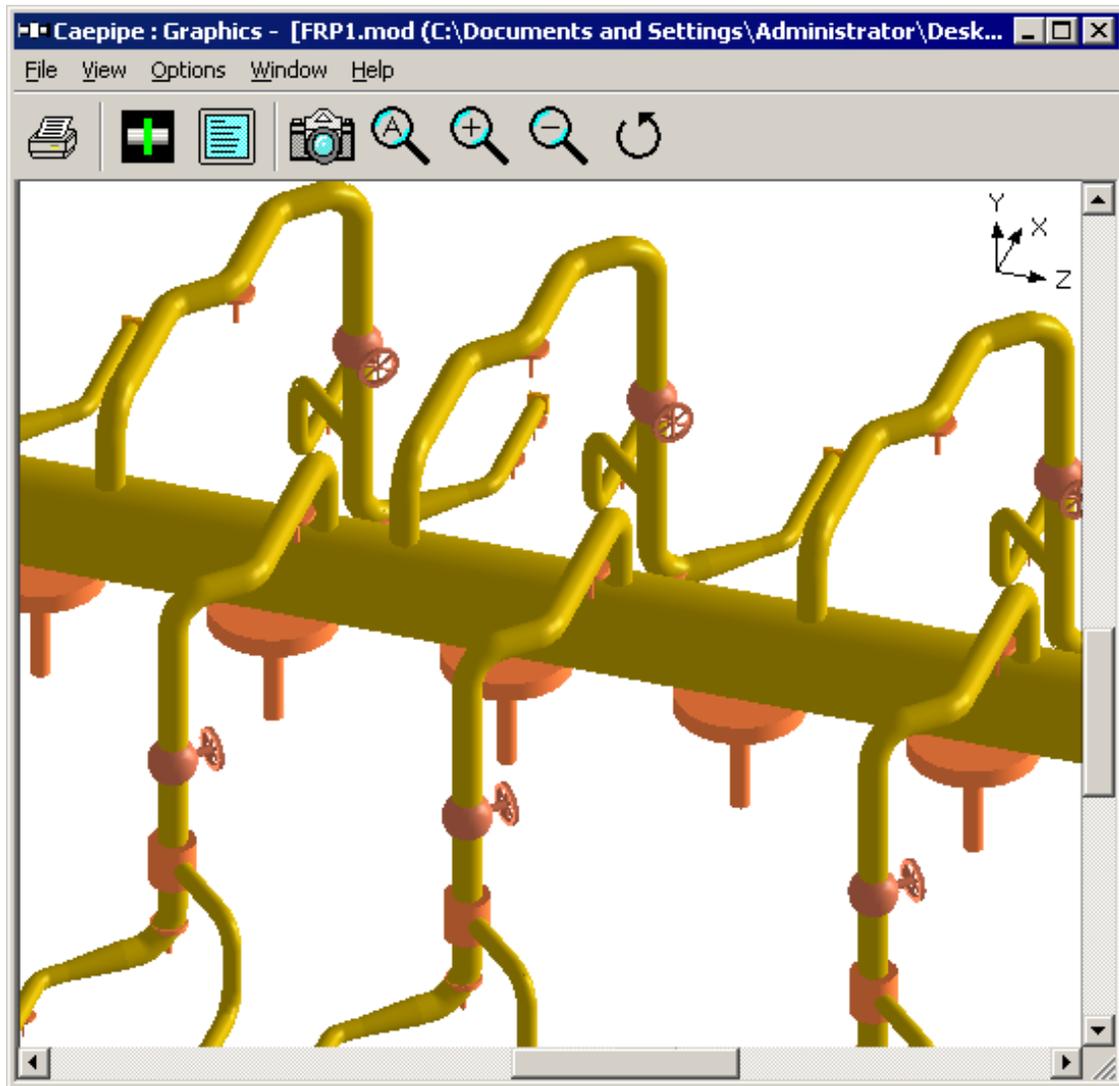
Fiber Reinforced Plastic Piping (FRP)

#	Hoop				Max Long				Min Long				Torsion			
	Node	Stress (psi)	Allow (psi)	Stress/Allow	Node	Stress (psi)	Allow (psi)	Stress/Allow	Node	Stress (psi)	Allow (psi)	Stress/Allow	Node	Stress (psi)	Allow (psi)	Stress/Allow
1	20	11538	2500	4.62	30B	9745	2500	3.90	60	4597	2500	1.84	60	141	7500	0.02
2	20	11538	2500	4.62	30A	9675	2500	3.87	60	4597	2500	1.84	20	141	7500	0.02
3	25	11538	2500	4.62	70	8681	2500	3.47	20	4577	2500	1.83	50	141	7500	0.02
4	50	11538	2500	4.62	10	8620	2500	3.45	20	4577	2500	1.83	25	141	7500	0.02
5	50	11538	2500	4.62	30B	7341	2500	2.94	25	4235	2500	1.69	30B	141	7500	0.02
6	60	11538	2500	4.62	30A	7314	2500	2.93	25	4235	2500	1.69	10	141	7500	0.02
7	10	11538	2500	4.62	50	6801	2500	2.72	50	4188	2500	1.68	30A	141	7500	0.02
8	30B	11538	2500	4.62	50	6801	2500	2.72	50	4188	2500	1.68	30A	141	7500	0.02
9	25	11538	2500	4.62	25	6765	2500	2.71	30A	3686	2500	1.47	25	141	7500	0.02
10	60	11538	2500	4.62	25	6765	2500	2.71	30B	3647	2500	1.46	20	141	7500	0.02
11	30A	11538	2500	4.62	20	6423	2500	2.57	10	2379	2500	0.95	60	141	7500	0.02
12	70	11538	2500	4.62	20	6423	2500	2.57	70	2308	2500	0.92	50	141	7500	0.02
13	30A	11538	2500	4.62	60	6392	2500	2.56	30A	1330	2500	0.53	70	141	7500	0.02
14	30B	11538	2500	4.62	60	6392	2500	2.56	30B	1249	2500	0.50	30B	141	7500	0.02

Axial and Torsional allowables may be entered under material properties so that they can be used to compare against calculated stresses in “FRP Sorted Stresses.” Forces, Stresses and Sorted stresses for FRP piping may be printed to a .CSV file (spreadsheet-compatible).

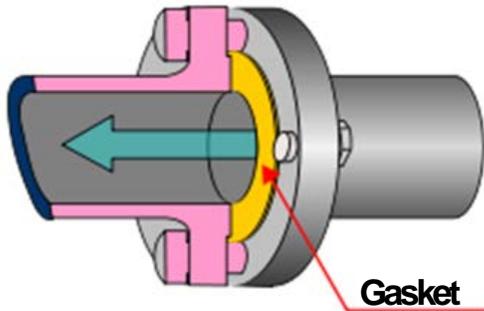
CAEPIPE renders FRP piping in golden color.

Fiber Reinforced Plastic Piping (FRP)

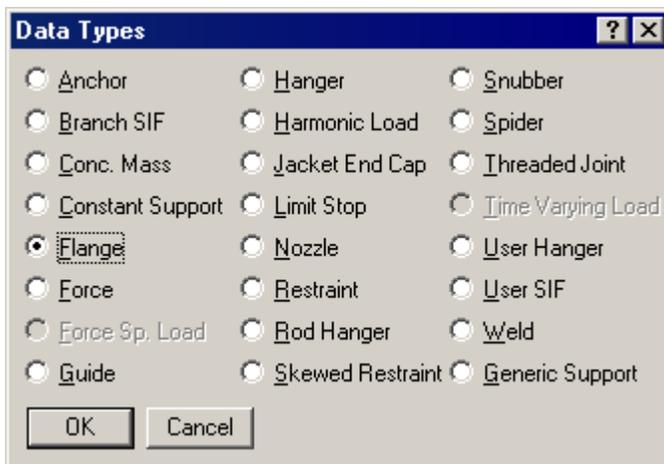


Flange

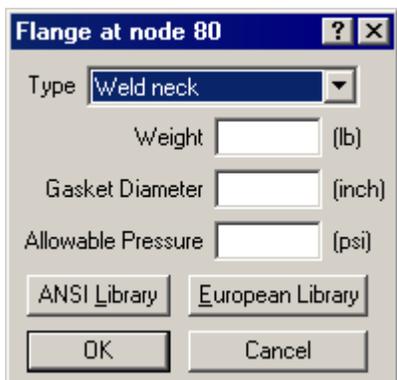
Flanges are used to stiffen the ends of pipes/valves/bends/etc. and act as connections to adjacent elements. ASME B16.5 is a standard in the USA to which flange manufacturers produce flanges for ½” to 23” NPS (nominal pipe size), while the B16.47 standard is used for 26” to 60” NPS. These are classified according to (pressure) classes, flanges in which are further classified according to materials with different pressures and temperatures.



A flange is input by typing “fl” in the Data column or selecting “Flange” from the Data Types dialog. If flanges are located at the bend end nodes (A, B nodes), or jacketed bend nodes (C, D nodes), the bend flexibility and SIF are modified.

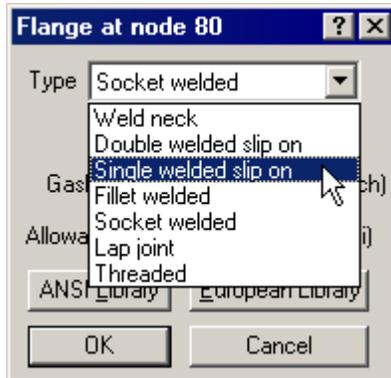


The Flange dialog is shown.



Several flange types are available – weld neck, socket welded, threaded, lap joint, etc. Use the Type drop-down combo box to select one.

Flange



Weight

The weight you provide should be the total weight of flanges, i.e., if there are two flanges the weight should be the weight of two flanges.

Gasket Diameter

The gasket diameter is used in calculating equivalent flange pressure in the flange report.

Allowable Pressure

CAEPIPE provides an approximation for the tendency of a flange to leak by calculating an “equivalent flange pressure” and comparing it to the (user-input) allowable pressure for the flange in the flange report. Often, the allowable pressure may be conservatively set to the flange rating. The allowable pressure can come from B16.5 (or a similar standard) for the flange class, material, pressure and temperature.

Flange Library

You may access the flange library by clicking on the Library button of the flange dialog. The default weight in the Flange Library is for two Flanges. However, each library dialog has an option to include weight for a single flange. See checkbox in the Flange library dialog below to include weight for a single flange.

Flange

ANSI Library

Flange Library [X]

Size = 8"

Rating	Weight (lb)
150	84
300	152
400	208
600	274
900	444
1500	668
2500	1384

Weight is of 2 flanges

Include weight for single flange

OK Cancel

European Library

Flange Library [X]

Size = 8"

Rating	Weight (lb)
PN16	48.5
PN25	74.96
PN40	94.8
PN64	153.88
PN100	232.8

Weight is of 2 flanges

Include weight for single flange

OK Cancel

The default weight in the library is the weight of two weld neck flanges (including bolts).

Flange Report

Caepipe : Flange Report - [complex1.res (C:\Users\S...)]

File Results View Options Window Help

Print Add Refresh Find Home Back Forward

#	Node	Pressure (psi)	Bending moment (ft-lb)	Axial force (lb)	Gasket diameter (inch)	Flange Pressure (psi)	Allowable Pressure (psi)	Flange Pressure Allowable
1	40B	200	20761	11589	6	6484	3000	2.161
2	40A	200	31298	15811	7.5	5092	5000	1.018

Flange

CAEPIPE lists every flange in a model in the flange report. The “Flange Pressure” is an equivalent pressure calculated from the actual pressure in the piping element, the bending moment and the axial force on the flange from the first operating case (W+P1+T1), as follows:

$$FlangePressure = Pressure + \frac{16 \times BendingMoment}{\pi \times Gasketdia^3} + \frac{4 \times AxialForce}{\pi \times Gasketdia^2}$$

The Gasket diameter is at the gasket loading location (if it is not input, it is conservatively assumed to be the internal diameter of the pipe).

The above shown equation is based on the article NC3658.1 of ASME Section III Class 2, 1992 or later editions, around page 176. The computed equivalent flange pressure is compared with the allowable pressure.

If you have input more than one temperature load, the flange equivalent pressure is calculated for all the applicable operating load cases, the worst of which is reported in the Flange report.

A flange report is generated even when no piping code is chosen. The flange report is shown in the results.

Suggestions for dealing with high flange pressure to allowable ratios

The Flange report in the CAEPIPE results window shows the loads at each flange location for the operating case (W+P+T).

The “equivalent” flange pressure is the sum of three terms from the flange equation as shown above. The last column in the Flange report shows a ratio of this equivalent flange pressure to a user-input allowable pressure. This ratio is flagged in red when greater than 1.0.

There are several ways of dealing with a high ratio, as suggested below.

- Ensure that you input an allowable pressure for the flange by looking up B16.5 or a similar code (as a function of design temperature and pressure).
- The equation from the previous page is generally regarded as a conservative equation. Being so, you can discard the axial stress term in it to arrive at a new “equivalent flange pressure.” This should be done outside CAEPIPE, maybe in a spreadsheet program.
- If the fluid within the pipe is non-hazardous, you can increase the allowable pressure by 10-20%, depending on your judgment.
- Since a flange is unlikely to fail by collapse, the key idea of the flange report is to “quantify” the tendency of a flange to leak its contents. Engineering judgment will play an important role in interpreting this report.
- If none of the above work, then you will have to reduce the bending moment at those flange locations. Be sure to examine all load cases, but frequently the excessive moments come from the expansion case. If so, consider introducing loops, bends and offsets as required.

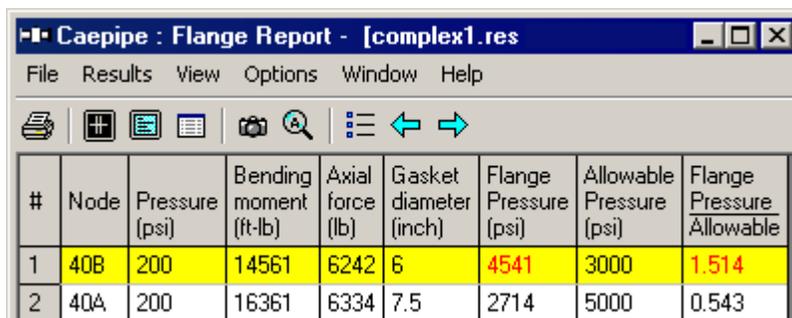
Flange and Bolt Stresses

Flange joints are essential components in all pressurized systems; they are also one of the most complex. Many factors are involved in determining the successful design and operation of a bolted flange joint service, namely, the interaction between the bolting, flange, and gasket as well as important non-linear variables such as friction and gasket properties. The Pressure Vessel and Piping Codes were developed with safety in mind; they provide a method for sizing the flange and bolts to be structurally adequate for the specified design conditions.

This Flange module implemented in CAEPIPE addresses the design rules contained in the ASME Section VIII, Division 1, Appendix 2 on bolted flange connections with gaskets.

These design rules will help you to obtain better insight into a flange joint's tendency to leak, beyond that provided by the rudimentary (yet indicative) flange report produced by a piping analysis, as seen in the earlier topic. You can examine the flange and bolt stresses arising from the bolt tightening loads required for a leakage-free joint.

This module assumes that you already have flanges and gaskets picked out for your system and performed a piping flexibility analysis using them, which will have produced a flange report as shown below.



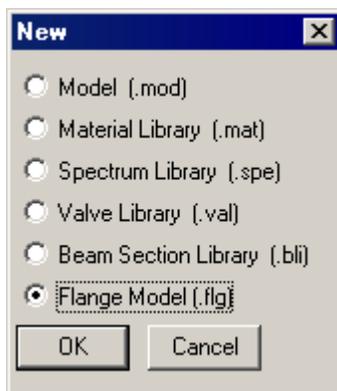
The screenshot shows a window titled "Caepipe : Flange Report - [complex1.res]". The window contains a menu bar (File, Results, View, Options, Window, Help) and a toolbar with various icons. Below the toolbar is a table with the following data:

#	Node	Pressure (psi)	Bending moment (ft-lb)	Axial force (lb)	Gasket diameter (inch)	Flange Pressure (psi)	Allowable Pressure (psi)	Flange Pressure Allowable
1	40B	200	14561	6242	6	4541	3000	1.514
2	40A	200	16361	6334	7.5	2714	5000	0.543

Note that this flange stress module to calculate flange and bolt stresses is separate from a piping model file and can be accessed from File Menu > Open/New command.

Because this module accepts axial load and bending moment at a flange as input among many others, you will need to first create your piping model that includes flanges (which you need to validate) and generate a Flange Report as shown above. Such a report will contain the information you can now use in the Flange stress module to calculate stresses.

When you first create a new flange model file, it comes populated with default values for a sample flange (see example 1 later on in this topic).



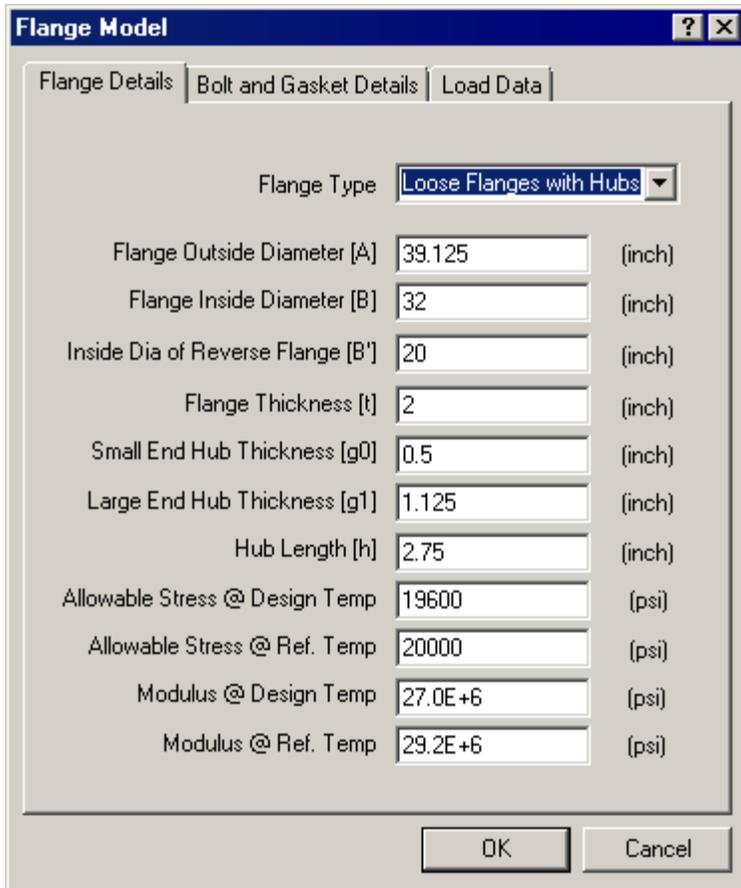
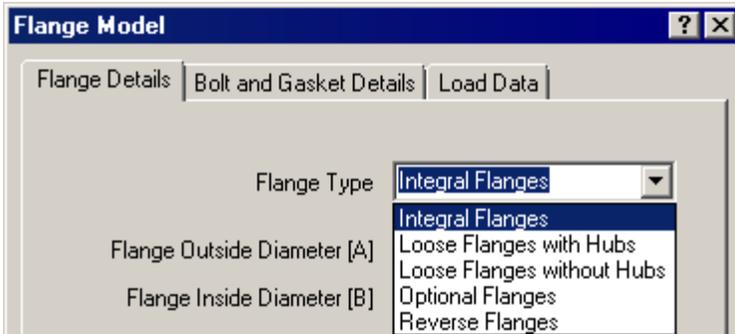
Flange and Bolt Stresses

Flange and Bolt Stresses

Flange Stress Calculation as per ASME Section VIII, Div. 1 - Appendix 2	
Flange Details:	
Flange Type :	Integral Flanges
Flange Outside Diameter [A] =	39.125 (inch)
Flange Inside Diameter [B] =	32 (inch)
Inside Diameter of Reverse Flange [B'] =	20 (inch)
Flange Thickness [t] =	2 (inch)
Small End Hub Thickness [g0] =	0.5 (inch)
Large End Hub Thickness [g1] =	1.125 (inch)
Hub Length [h] =	2.75 (inch)
Allowable Stress @ Design Temp [sf] =	19600 (psi)
Allowable Stress @ Ref. Temp [sfa] =	20000 (psi)
Modulus @ Design Temp [E] =	27.0E+6 (psi)
Modulus @ Ref. Temp [Ea] =	29.2E+6 (psi)
Bolting Information:	
Bolt Circle Diameter =	37 (inch)
Number of Bolts =	36
Bolt Diameter =	1 (inch)
Allowable Stress @ Ref. Temp [sa] =	25000 (psi)
Allowable Stress @ Design Temp [sb] =	25000 (psi)
Gasket Information:	
Gasket Outside Diameter =	35.5 (inch)
Gasket Inner Diameter =	33.5 (inch)
Leak Pressure Ratio [m] =	3.00
Gasket Seating Stress [y] =	10000 (psi)
Facing Sketch =	1
Facing Column =	1
Load Data:	
Design Pressure =	414 (psi)
Design Temperature =	500 (F)
Axial Force =	1000 (lb)
Bending Moment =	200 (ft-lb)
Allowable Pressure =	665 (psi)

Flange and Bolt Stresses

Double-clicking anywhere in the previous screen (or Edit menu > Edit (Ctrl+E)) opens a dialog with input fields (with default values) you can edit. You will need to enter all of your flange data in this dialog. A legend of the different parameters you see here are explained in detail in Appendix F of this manual.



Required flange input information is organized into three Property tabs – Flange Details, Bolt and Gasket Details, and Load Data, the last of which accepts data from a piping model's Flange Report. Once all the data is input, save the model (flange model filenames will have a .flg extension). Now, select File menu > Analyze to calculate flange stresses, which will be shown *right below* the input information.

Flange and Bolt Stresses

Flange Equivalent Pressure as per NC3658.1 of ASME Section III Class 2

Equivalent Pressure = 415 (psi)
 ANSI B16.5 Flange Allowable Pressure Rating = 665 (psi)

Stress Safety Factor: (If greater than one, then joint failure is predicted)
 Equivalent Pressure / Allowable Pressure = 0.62

Flange Stresses as per ASME Section VIII. Div. 1 - Appendix 2
 According to S-2(d) of Appendix S, Rigidity factors (J) should be < 1.0.

ASME Rigidity Factor 'J', Operating Case = 1.017185
 ASME Rigidity Factor 'J', Seating Case = 0.594627

J < 1.0 minimizes the possibility of flange leakage.

Calculated Stresses as per ASME Section VIII. Div. 1

Flange Stresses (psi) - Operating Condition

	Calculated	Allowed	Ratio	Status
Longitudinal Hub (SH)	24152	29400	0.821	OK
Radial Flange (SR)	11590	19600	0.591	OK
Tangential Flange (ST)	7232	19600	0.369	OK
0.5(SH + SR)	17871	19600	0.912	OK
0.5(SH + ST)	15692	19600	0.801	OK
Bolt Stress	24673	25000	0.987	OK

Flange Stresses (psi) - Gasket Seating Condition

	Calculated	Allowed	Ratio	Status
Longitudinal Hub (SH)	15269	30000	0.509	OK
Radial Flange (SR)	7327	20000	0.366	OK
Tangential Flange (ST)	4572	20000	0.229	OK
0.5(SH + SR)	11298	20000	0.565	OK
0.5(SH + ST)	9921	20000	0.496	OK
Bolt Stress	19482	25000	0.779	OK

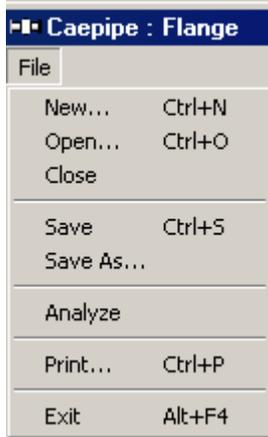
There are three main sections in the shown results:

- Flange Equivalent Pressure (same as the one shown in piping model results > Flange Report),
- Flange stresses in the Operating condition, and
- Flange stresses in the Gasket seating condition

Flange Stress Module Menus

Use these numbers in accordance with the ASME Section VIII publication and your engineering judgment to qualify these flanges for use in your piping systems.

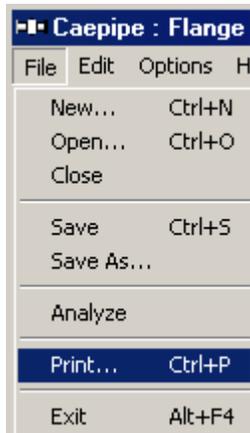
File Menu



Analyze

Analyze command calculates the flange and bolt stresses and compares them to the input allowable stresses.

Print



You can print a Flange Report by using the Print command. You can also preview the report by clicking the Preview button on the print dialog.

Flange and Bolt Stresses

Caepipe : Print Preview, Page 1 (of 2)

File View Options Help

Print Prev Page Next Page Close

Caepipe Page 1

Flange Stress Calculation as per ASME Section VIII, Div. 1 - Appendix 2

Flange Details:
 Flange Type : Integral Flanges
 Flange Outside Diameter [A] = 39.125 (inch)
 Flange Inside Diameter [B] = 32 (inch)
 Inside Diameter of Reverse Flange [B'] = 20 (inch)
 Flange Thickness [t] = 2 (inch)
 Small End Hub Thickness [g0] = 0.5 (inch)
 Large End Hub Thickness [g1] = 1.125 (inch)
 Hub Length [h] = 2.75 (inch)
 Allowable Stress @ Design Temp [sf] = 19600 (psi)
 Allowable Stress @ Ref. Temp [sfa] = 20000 (psi)
 Modulus @ Design Temp [E] = 27.0E+6 (psi)
 Modulus @ Ref. Temp [Ea] = 29.2E+6 (psi)

Bolting Information:
 Bolt Circle Diameter = 37 (inch)
 Number of Bolts = 36
 Bolt Diameter = 1 (inch)
 Allowable Stress @ Ref. Temp [sa] = 25000 (psi)
 Allowable Stress @ Design Temp [sb] = 25000 (psi)

Gasket Information:
 Gasket Outside Diameter = 35.5 (inch)
 Gasket Inner Diameter = 33.5 (inch)
 Leak Pressure Ratio [m] = 3.00
 Gasket Seating Stress [y] = 10000 (psi)
 Facing Sketch = 1
 Facing Column = 1

Load Data:
 Design Pressure = 414 (psi)
 Design Temperature = 500 (F)
 Axial Force = 1000 (lb)
 Bending Moment = 200 (ft-lb)
 Allowable Pressure = 665 (psi)

Flange Equivalent Pressure as per NC3658.1 of ASME Section III Class 2

Equivalent Pressure = 415 (psi)
 ANSI B16.5 Flange Allowable Pressure Rating = 665 (psi)

Stress Safety Factor: (If greater than one, then joint failure is predicted)
 Equivalent Pressure / Allowable Pressure = 0.62

Flange Stresses as per ASME Section VIII, Div. 1 - Appendix 2

According to S-2(d) of Appendix S, Rigidity factors (J) should be < 1.0.

ASME Rigidity Factor 'J', Operating Case = 1.017185
 ASME Rigidity Factor 'J', Seating Case = 0.594627

J < 1.0 minimizes the possibility of flange leakage.

Calculated Stresses as per ASME Section VIII, Div. 1

Flange Stresses (psi) - Operating Condition

Version 6.81 Untitled Jun 14, 13

Edit Menu

Edit

You can edit the Flange, Bolt, and Gasket Details, as well as Load Data by clicking the Edit command.

Edit

Edit Ctrl+E

Flange and Bolt Stresses

Flange Model Window

You can edit the Flange, Bolt, and Gasket Details, as well as Load Data by clicking the Edit command.

Flange Details Tab

Modify the flange details using this window.

The screenshot shows a dialog box titled "Flange Model" with three tabs: "Flange Details", "Bolt and Gasket Details", and "Load Data". The "Flange Details" tab is active. It contains the following fields:

Parameter	Value	Unit
Flange Type	Integral Flanges	
Flange Outside Diameter [A]	39.125	(inch)
Flange Inside Diameter [B]	32	(inch)
Inside Dia of Reverse Flange [B']	20	(inch)
Flange Thickness [t]	2	(inch)
Small End Hub Thickness [g0]	0.5	(inch)
Large End Hub Thickness [g1]	1.125	(inch)
Hub Length [h]	2.75	(inch)
Allowable Stress @ Design Temp	19600	(psi)
Allowable Stress @ Ref. Temp	20000	(psi)
Modulus @ Design Temp	27.0E+6	(psi)
Modulus @ Ref. Temp	29.2E+6	(psi)

At the bottom of the dialog box are "OK" and "Cancel" buttons.

Flange and Bolt Stresses

Bolt and Gasket Details Tab

Modify the bolt and gasket details using this window.

The image shows a software dialog box titled "Flange Model" with three tabs: "Flange Details", "Bolt and Gasket Details", and "Load Data". The "Bolt and Gasket Details" tab is active. It contains two sections: "Bolting Information" and "Gasket Information".

Bolting Information:

- Bolt Circle Diameter: 37 (inch)
- Number of Bolts: 36
- Bolt Diameter: 1" (dropdown menu)
- Allowable Stress @ Design Temp: 25000 (psi)
- Allowable Stress @ Ref. Temp: 25000 (psi)

Gasket Information:

- Gasket Outer Diameter: 35.5 (inch)
- Gasket Inner Diameter: 33.5 (inch)
- Leak Pressure Ratio [m]: 3.00
- Gasket Seating Stress [y]: 10000 (psi)
- Facing Sketch: 1
- Facing Column: 1

At the bottom of the dialog box are "OK" and "Cancel" buttons.

Load Data Tab

Modify the load data using this window.

The screenshot shows a dialog box titled "Flange Model" with three tabs: "Flange Details", "Bolt and Gasket Details", and "Load Data". The "Load Data" tab is active. It contains five input fields with their respective units:

Parameter	Value	Unit
Design Pressure	414	(psi)
Design Temperature	500	(F)
Axial Force	1000	(lb)
Bending Moment	200	(ft-lb)
Allowable Pressure	665	(psi)

At the bottom of the dialog box are "OK" and "Cancel" buttons.

Options Menu

The screenshot shows a menu with the following items:

- Options
- Units... Ctrl+U
- Font...

Units

See Units in the Layout Window Options Menu section at the beginning of this manual.

Font

See Font in the Layout Window Options Menu section at the beginning of this manual.

Sample Problem

Problem 1:

(Example on page 19 Chapter 40 “Bolted-Flange Joints and Connections” by William J. Koves on “Companion Guide to the ASME Boiler & Pressure Vessel Code” by K .R. Rao [2001], American Society of Mechanical Engineers, U.S.)

Flange Details:

Flange Type : **Integral Flanges**

Flange Outside Diameter [A] = 39.125 (inch)

Flange Inside Diameter [B] = 32 (inch)

Flange Thickness [t] = 2 (inch)

Small End Hub Thickness [g0] = 0.5 (inch)

Large End Hub Thickness [g1] = 1.125 (inch)

Hub Length [h] = 2.75 (inch)

All. Stress @ Design Temp [sf] = 19600 (psi)

All. Stress @ Ref. Temp [sfa] = 20000 (psi)

Modulus @ Design Temp [E] = 2.7E+7 (psi)

Modulus @ Ref. Temp [Ea] = 2.92E+7 (psi)

Bolting Information:

Bolt Circle Diameter = 37 (inch)

Number of Bolts = 36

Bolt Diameter = 1 (inch)

All. Stress @ Ref. Temp [sa] = 25000 (psi)

All. Stress @ Design Temp [sb] = 25000 (psi)

Gasket Information:

Gasket Outside Diameter = 35.5 (inch)

Gasket Inner Diameter = 33.5 (inch)

Leak Pressure Ratio [m] = 3.00

Gasket Seating Stress [y] = 10000 (psi)

Facing Sketch = 1

Facing Column = 1

Load Data:

Design Pressure = 414 (psi)

Design Temperature = 500 (F)

Axial Force = 1000 (lb)

Bending Moment = 200 (ft-lb)

Allowable Pressure = 665 (psi)

Flange and Bolt Stresses

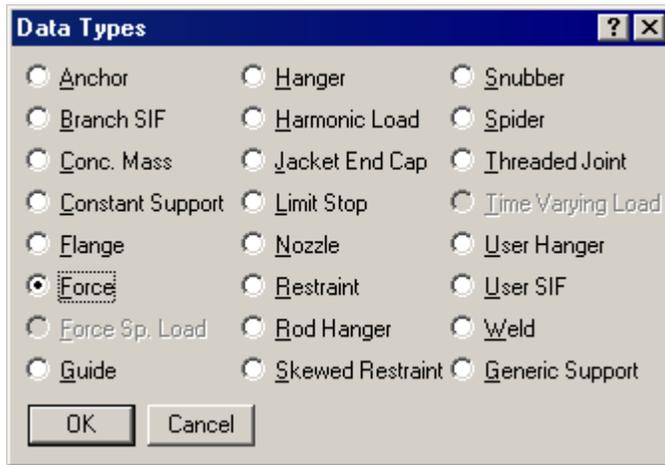
Comparison of Results

Flange Stresses	Text Book Results (psi)	CAEPIPE (psi)	CAESAR II (psi)
Operating condition			
Longitudinal Hub (SH)	24150	24152	24227
Radial Flange (SR)	11590	11590	11636
Tangential Flange (ST)	7230	7232	7205
0.5(SH + SR)	17870	17871	17932
0.5(SH + ST)	15690	15692	15716
Gasket Seating Condition			
Longitudinal Hub (SH)	15270	15269	15292
Radial Flange (SR)	7330	7327	7345
Tangential Flange (ST)	4570	4572	4547
0.5(SH + SR)	11300	11298	11318
0.5(SH + ST)	9900	9921	9920

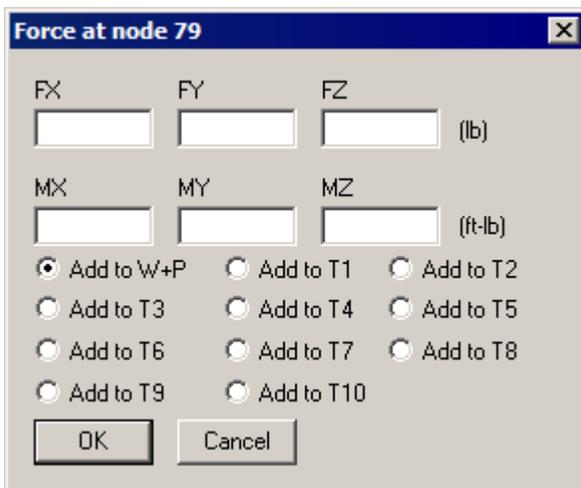
Legend for the different parameters and more examples are given in Appendix F.

Force

External forces and moments on the piping system in the global X, Y and Z directions may be input at any location. Type “fo” in the Data column or select “Force” from the Data Types dialog.



The Force dialog is shown.



If you select “Add to W+P”, the specified forces and moments are applied to the sustained and operating load cases.

If you select any of the thermal cases (“Add to T1”, “Add to T2”, “Add to T3”, all the way up to “Add to T10”), the specified forces and moments are applied to the selected thermal load case (T1/T2/T3) and its operating load case counterpart (for e.g., T1 and W+P1+T1, or T2 and W+P2+T2, or T3 and W+P3+T3).

You may use this data type to apply external forces and moments (from other sources) on specific nodes. Or, you may use this to apply the pre-calculated relief valve force at the end of a valve. If you use it for the latter, then input:

1. A dummy 0.01g-load, under menu Load > Static Seismic
2. Check the Static Seismic case for analysis under menu Load > Load Case

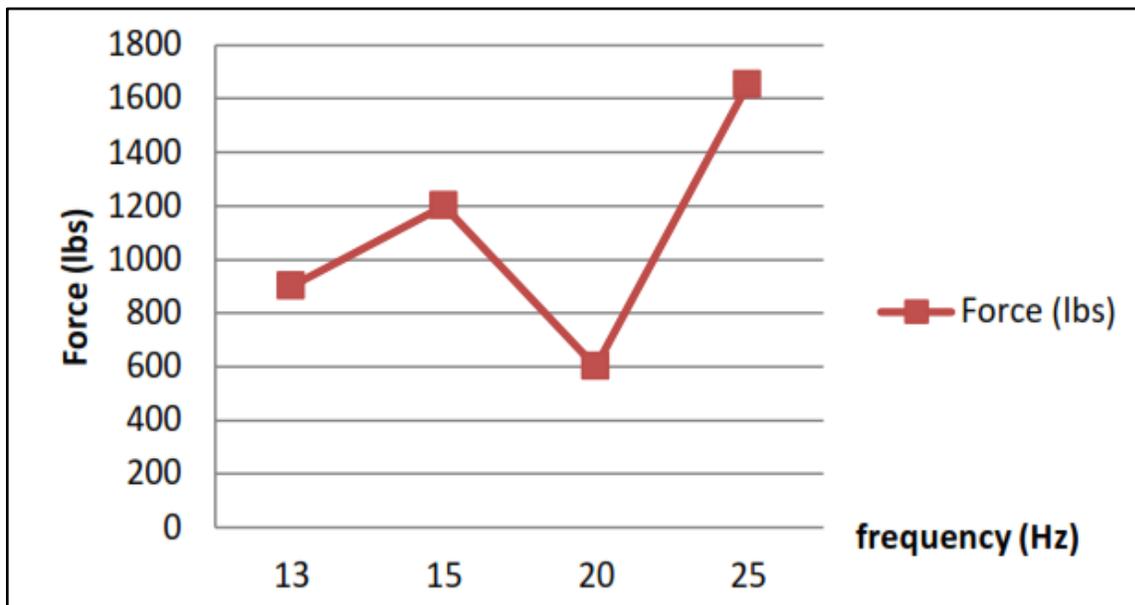
Force

Upon analysis, you may study the effect of the relief valve force under the occasional stresses – which are due to the combination of Sustained and occasional loads (relief valve force) as most B31.x Codes stipulate.

Force Spectrum

Force spectrum analyses (different from harmonic analyses) are generally performed to determine the response of the piping system to short-duration impulsive loads such as fluid hammer, safety relief valve (SRV) and slug flow loads. For an actual short-duration impulsive dynamic load exerted on a piping system, a fluid transient analysis is first carried out in order to arrive at the “time-history loads” (i.e., force vs. time) acting in the three global directions (namely global X, Y and Z) at all affected points in the piping system. The time-history load sets so computed are then applied, one time-history load set at a time, on a single degree-of-freedom (dof) spring-mass system with a pre-set natural frequency, to determine the maximum dynamic response of this single dof system with that natural frequency. Such dynamic analysis for that time-history load is repeated on the same single dof system with different pre-set natural frequencies. The force spectrum for that time-history load would then be a table of maximum dynamic response computed for the single dof system with different natural frequencies. In other words, the force spectrum is a table of force spectral values vs frequencies that captures the maximum intensity and frequency content of that time-history load. Similarly, force spectrum tables are determined for all other time-history load sets. The above force spectrum tables (i.e., maximum dynamic force vs frequency) are then applied as inputs at the respective piping nodes of the CAEPIPE stress model to compute displacements, forces and stresses.

For any piping system, there are as many natural modes of vibrations as the number of dynamic degrees of freedom for that system. The force spectral value corresponding to a natural frequency of the piping system is arrived at by interpolating the force spectrum vs frequency table as determined above. For better understanding, as an example, refer to the graph shown next as well as the natural frequencies computed for a piping system at 10 Hz, 14 Hz, 21 Hz, 29 Hz and 33.8 Hz.



Force Spectrum

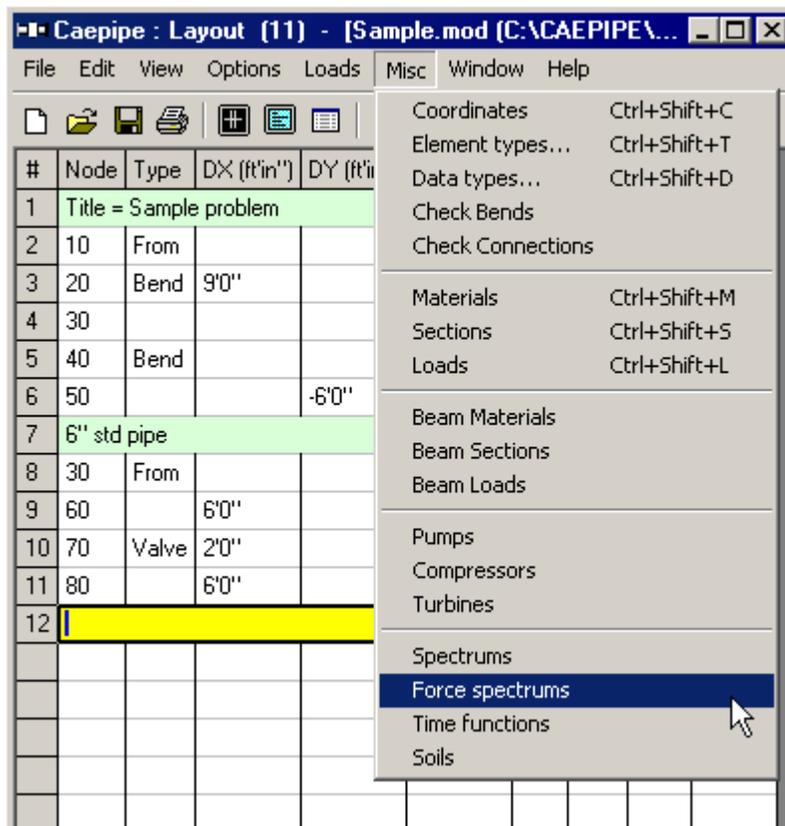
From the above graph, to arrive at a force value corresponding to a natural frequency of 14 Hz, CAEPIPE interpolates the force spectral values between 13 and 15 Hz. Similarly, to arrive at a force value corresponding to a natural frequency of 21 Hz, CAEPIPE interpolates the force spectral values between 20 Hz & 25 Hz. Since force spectral values above 25 Hz are not defined in the graph shown above, CAEPIPE arrives at a force value of 1650 lb. (i.e., the spectral value corresponding to the maximum frequency of 25 Hz in the above plot) even for natural frequencies of 29 and 33.8 Hz. Similarly, CAEPIPE arrives at a force value of 900 lb. for a natural frequency of 10 Hz (i.e., the spectral value corresponding to the minimum frequency of 13 Hz in the above plot).

If only one set of force versus frequency is input (for example, 1000 lb. at 14 Hz) in the force spectrum table for your model, CAEPIPE applies the same force (1000 lb.) for all natural frequencies computed for that piping system. Note that the displacement produced at a node will remain unchanged even when the sole frequency in the force spectrum table is changed from 14 Hz to any other frequency.

Here, the results of the modal analysis are used with force spectrum loads to calculate the response (displacements, support loads and stresses) of the piping system. It is often used in place of a time-history analysis to determine the response of the piping system to sudden impulsive loads such as water hammer, relief valve and slug flow. The force spectrum is a table of spectral values versus frequencies that captures the intensity and frequency content of the time-history loads. It is a table of Dynamic Load Factors (DLF) versus natural frequencies. DLF is the ratio of the maximum dynamic displacement divided by the maximum static displacement. Note that Force spectrum is a non-dimensional function (since it is a ratio) defining a curve representing force versus frequency. The actual force spectrum load at a node is defined using this force spectrum in addition to the direction (FX, FY, FZ, MX, MY, MZ), units (lb, N, kg, ft-lb, in-lb, Nm, kg-m) and a scale factor.

The Force spectrums are input from the Layout or List menu: Misc > Force spectrums.

Force Spectrum



The Force spectrum list appears.

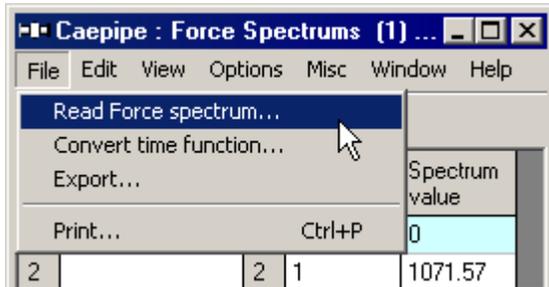
The screenshot shows the 'Caepipe : Force spectrums (1) ...' window. The table displays the following data:

#	Name	#	Frequency (Hz)	Spectrum value
1	Relief valve load	1	0	0
2		2	1	1071.57
		3	2	1654.55
		4	3	1730.73
		5	4	1646.81
		6	5	1544.29
		7	6	1431.83
		8	7	1315.82
		9		

Enter a name for the force spectrum and spectrum values versus frequencies table. In addition to inputting the force spectrum directly, it can also be read from a text file or converted from a previously defined time function.

To read a force spectrum from a text file:

use the List menu: File > Read force spectrum.



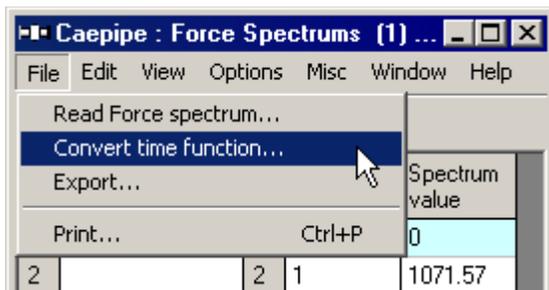
The text file should be in the following format:

```
Name (up to 31 characters)
Frequency (Hz) Spectrum value
Frequency (Hz) Spectrum value
Frequency (Hz) Spectrum value
.
.
.
```

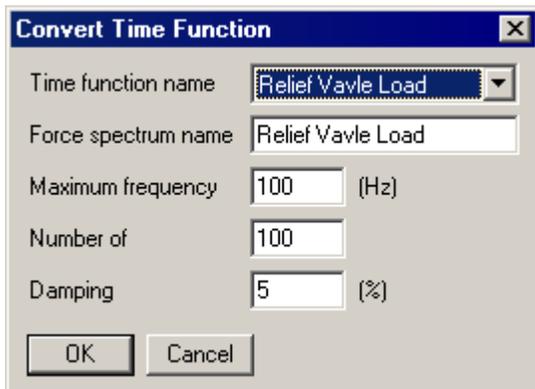
The frequencies can be in any order. They will be sorted in ascending order after reading from the file.

To convert a previously defined time function to force spectrum:

use the List menu: File > Convert time function.



The Convert Time Function dialog appears.



Force Spectrum

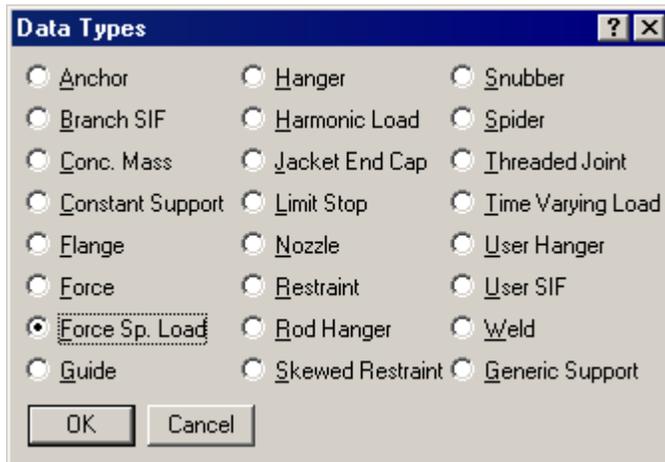
Select the time function to convert from the Time function name drop down combo box. Then input the Force spectrum name (defaults to the Time function name), Maximum frequency, Number of frequencies and the Damping. When you press Enter or click on OK, the time function will be converted to a force spectrum and entered into the force spectrum list.

The time function is converted to a force spectrum by solving the dynamic equation of motion for a damped single spring mass system with the time function as a forcing function at each frequency using Duhamel's integral and dividing the absolute maximum dynamic displacement by the static displacement.

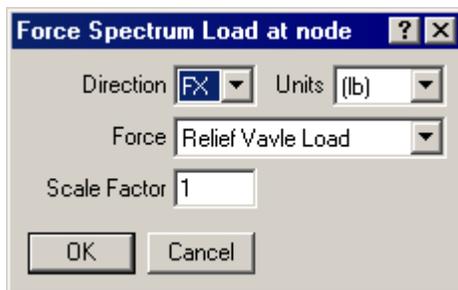
Force Spectrum Load

The force spectrum loads are applied at nodes (in Data column in Layout window). At least one force spectrum must be defined before a force spectrum load at a node can be input.

To apply the force spectrum load at a node click on the Data heading or press Ctrl+Shift+D for Data Types dialog.



Select "Force Sp. Load" as the data type and click on OK. This opens the Force Spectrum Load dialog.

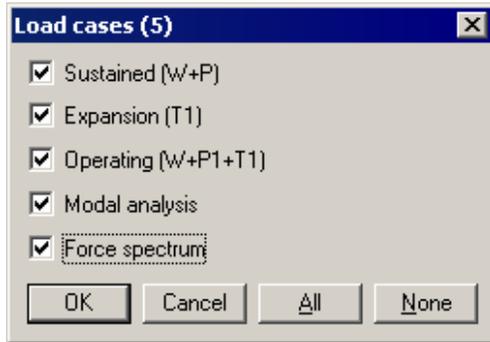


Select the direction, units and force spectrum using the drop down combo boxes and input appropriate scale factor. The scale factor can be a scalar value, which, when multiplied by the non-dimensional force spectrum, will give the actual magnitudes of the force versus frequency in the global direction and unit selected in the above dialog. Then click on OK to enter the force spectrum load at that node.

Force Spectrum

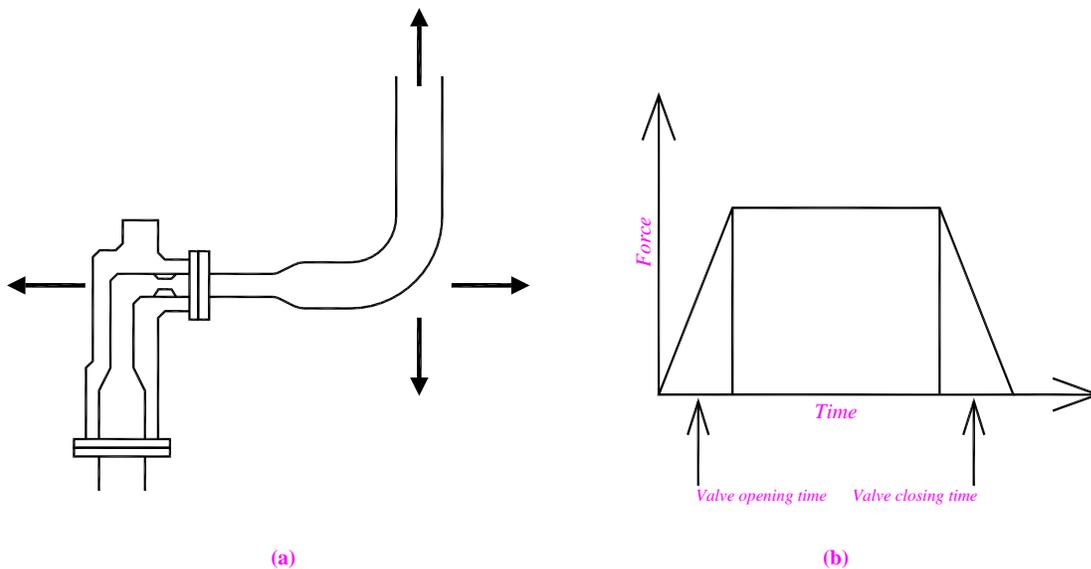
7	6" std pipe								
8	30	From							
9	60		6'0"			A53	6	1	Force sp load
10	70	Valve	2'0"			A53	6	1	

Input force spectrum loads at other nodes similarly. Then select the force spectrum load case for analysis using the Layout menu: Loads > Load cases.



Note that Modal analysis and Sustained (W+P) load cases are automatically selected when you select Force spectrum load case. The force spectrum load case is analyzed as an Occasional load.

A Relief Valve Analysis may be performed by first obtaining the data about valve opening and subsequent behavior, as a force versus time history profile. Enter the profile as a time function. See under Time History Load for how to.



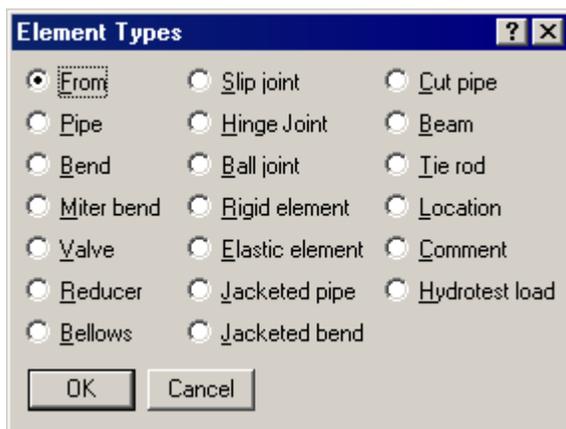
Then under Force Spectrum, use “Convert time function” to convert the force-time history profile into a Force-Spectrum. Input loads and analyze.

From (Node)

“From” is a special Element Type used to start a new (branch) line. For a new node number, any values you type in under the DX, DY and DZ columns are taken to be coordinates (and not offsets). If you use “From” for an existing node number, then you do not have to specify values for the DX, DY and DZ fields.

If you specify a new node number other than the global origin [that has coordinates (0, 0, 0)], then you must specify the coordinates for the new node using “From” and DX, DY and DZ fields. If you do not, then the new node number will have the same location as the node that is the global origin.

“From” is input by typing “f” in the Type column or selecting “From” from the Element Types dialog.

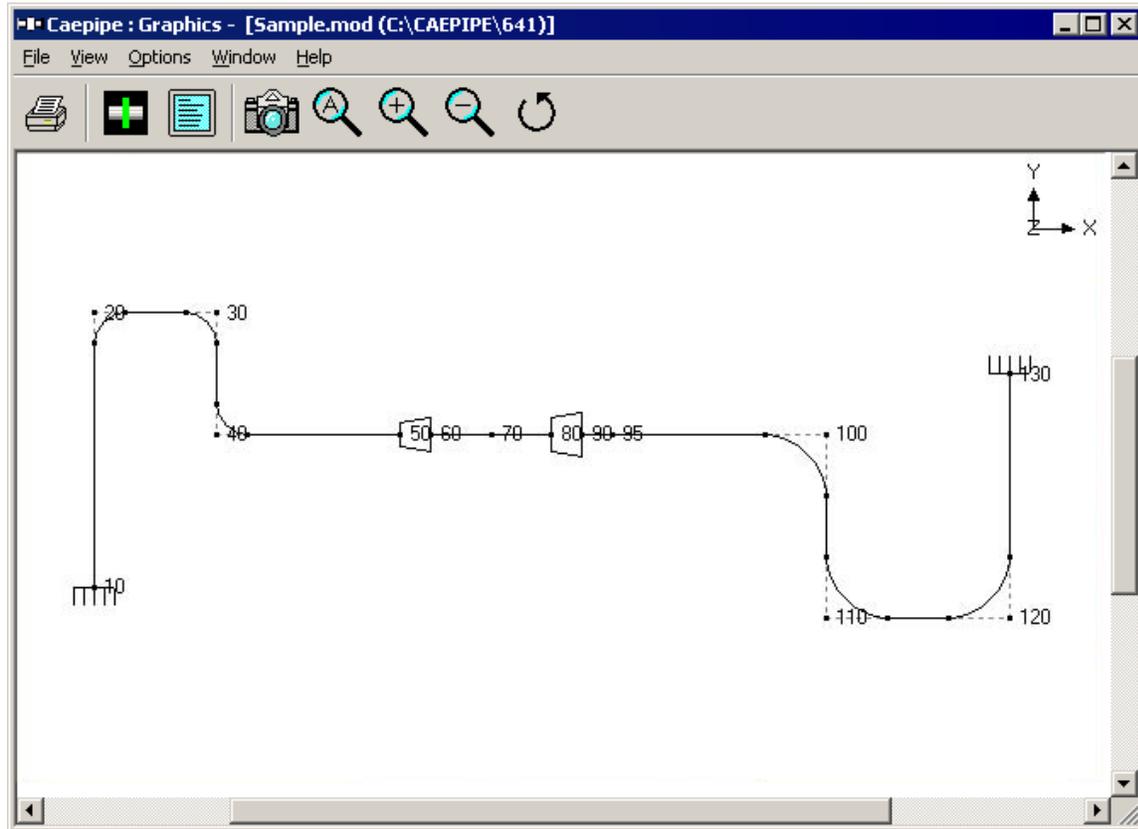


The first node of a model is always a fixed “From” node since you have to start the model from a point. The DX, DY, DZ fields for this node may be left blank to mean the global origin (0,0,0) or coordinates may be specified in the DX, DY and DZ fields to have a nonzero reference point for the model.

Values specified for DX, DY and DZ for any other node other than a “From” node are interpreted as offsets (not as coordinates) unless the node is suffixed with an asterisk “*”. See the topic “Node” for more information.

It is helpful to create a model from the global origin and continue in an orderly manner from there, especially when you plan to merge (see File menu) two models later.

Example:



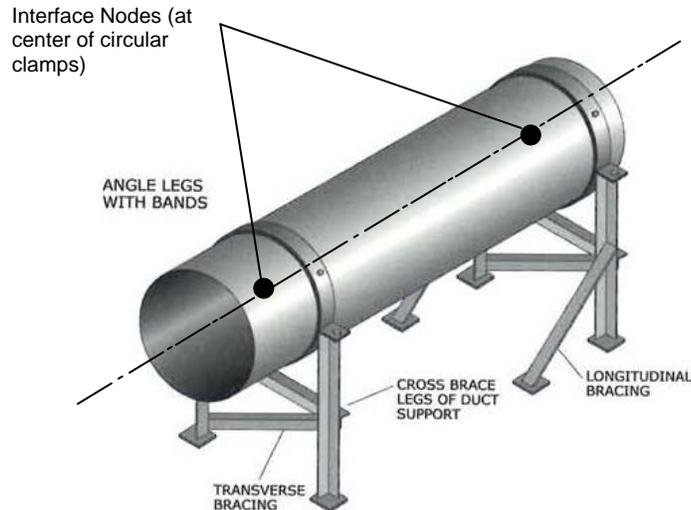
Assume you had two separate models you wanted to combine later.

First model: In this first model, you would start modeling from node 10 up to node 100.

Second model: You would model from node 100 up to node 130, which is a better way to model rather than modeling from node 130 to node 100, which would complicate the "Merge" process.

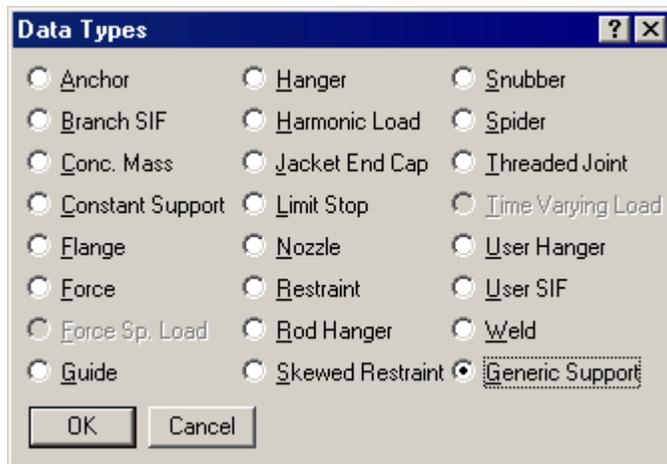
Generic Support

A generic data type called “Generic support” is available to define a complex support fitting for which the support stiffnesses are obtained from an external source, using a reduced 6x6 stiffness matrix. An example would be a support assembly as shown below.

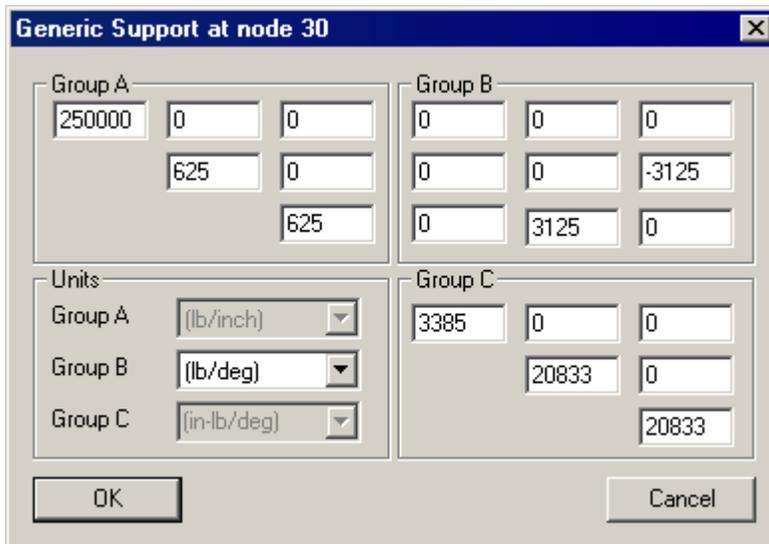


A reduced 6x6 stiffness matrix representative of this support at a single interface node (as shown in figure above) will first need to be arrived at using any general purpose FEA software package (such as ANSYS, NASTRAN, etc.) Then, those stiffness values representing the support assembly can then be input into CAEPIPE model for further analysis.

“Generic Support” is input by typing “ge” in the Data column or selecting “Generic Support” from the Data Types dialog.



Generic Support



The dialog box titled "Generic Support at node 30" contains the following data:

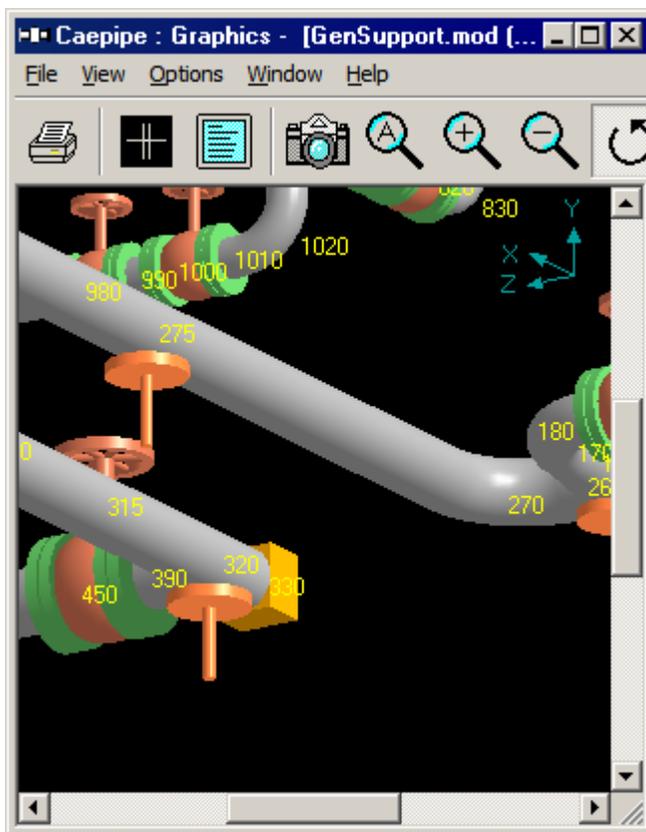
Group	Stiffness 1	Stiffness 2	Stiffness 3
Group A	250000	0	0
	625	0	
		625	
Group B	0	0	0
	0	0	-3125
	0	3125	0
Group C	3385	0	0
		20833	0
			20833

Units are set as follows:

Group	Units
Group A	(lb/inch)
Group B	(lb/deg)
Group C	(in-lb/deg)

Units for Groups A and C are controlled centrally through the Units command (Ctrl+U in Layout window). Units for Group B need to be set in this dialog.

Each stiffness field in the dialog is editable. The default “rigid” stiffness is shown for all the diagonal terms. If you have a stiffness value for any of these including the off-diagonals, enter them here (ensure units integrity). Graphically, the support is shown as a solid block (330 in the next figure).



Displacements, support loads and support load summaries are shown for this support type.

Generic Support

Caepipe : Support load summary for generic support at node 30 - [complex1.res (\...\...)]

File Results View Options Window Help









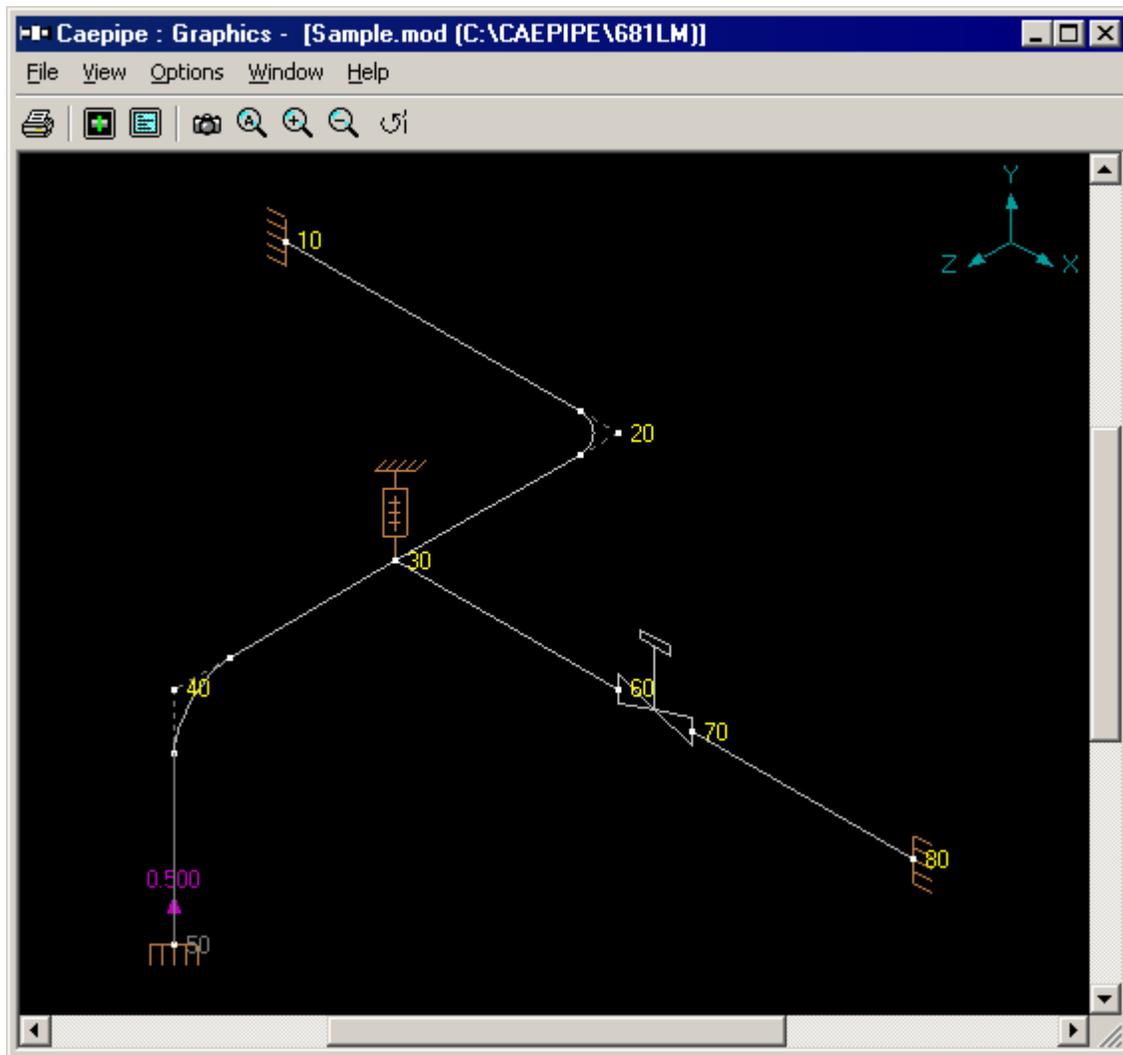





Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)	Displacements (global)		
							X (inch)	Y (inch)	Z (inch)
Sustained	-2663	-75	6	-16	309	734	-0.011	-0.048	0.021
Operating1	-16802	346	-234	-91	-998	-2247	-0.067	0.141	-0.060
Operating2	6509	3	-30	-18	592	605	0.026	-0.042	0.042
Operating3	25788	56	349	-3	1033	1067	0.103	-0.075	0.058
Sustained+Settlement	-2692	-317	35	19	723	2171	-0.011	-0.137	0.048
Operating1+Settlement	-16831	103	-206	-56	-584	-809	-0.067	0.052	-0.032
Operating2+Settlement	6480	-239	-2	17	1006	2043	0.026	-0.131	0.069

Graphics Window

When you open a model, CAEPIPE automatically opens the Layout and the Graphics windows. Many commands to manipulate the image such as zoom, turn (rotate), pan and print are available in this window.



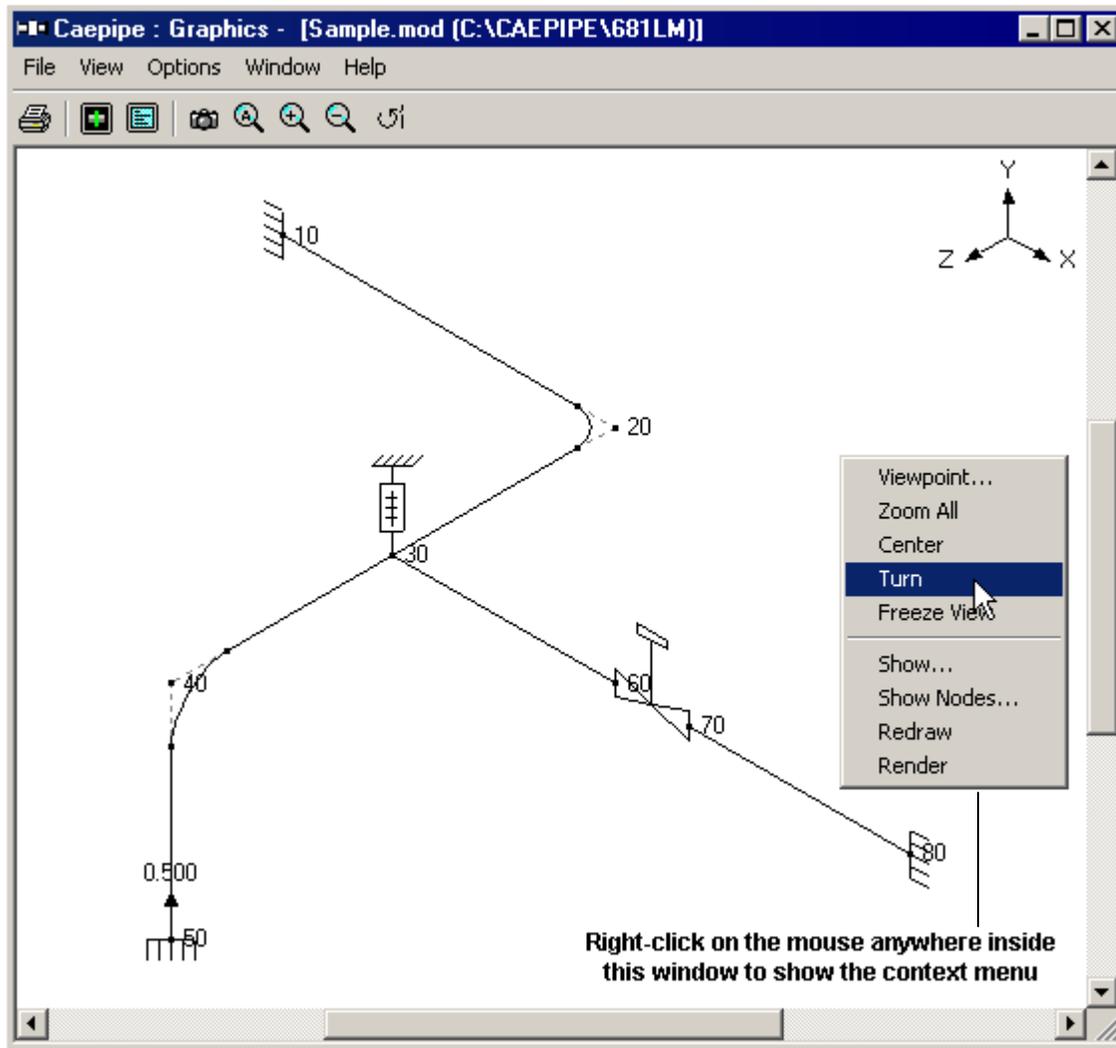
You could opt to display the image with a white background too (use menu Options > Background > White).

When a model is opened, CAEPIPE displays the last saved model view in the Graphics window.

Graphics Window

Context Menu

Right-clicking anywhere inside this window displays the context menu.



In addition to a few commands constantly available on this menu, other commands appear on the menu depending on the results item shown in the Results window. For example, when Sorted stresses are shown, the context menu shows three commands specific to Sorted stresses (Show Stresses, Show stress ratios and [set stress/ratio] Thresholds).

Graphics Window



Editing in Graphics Window

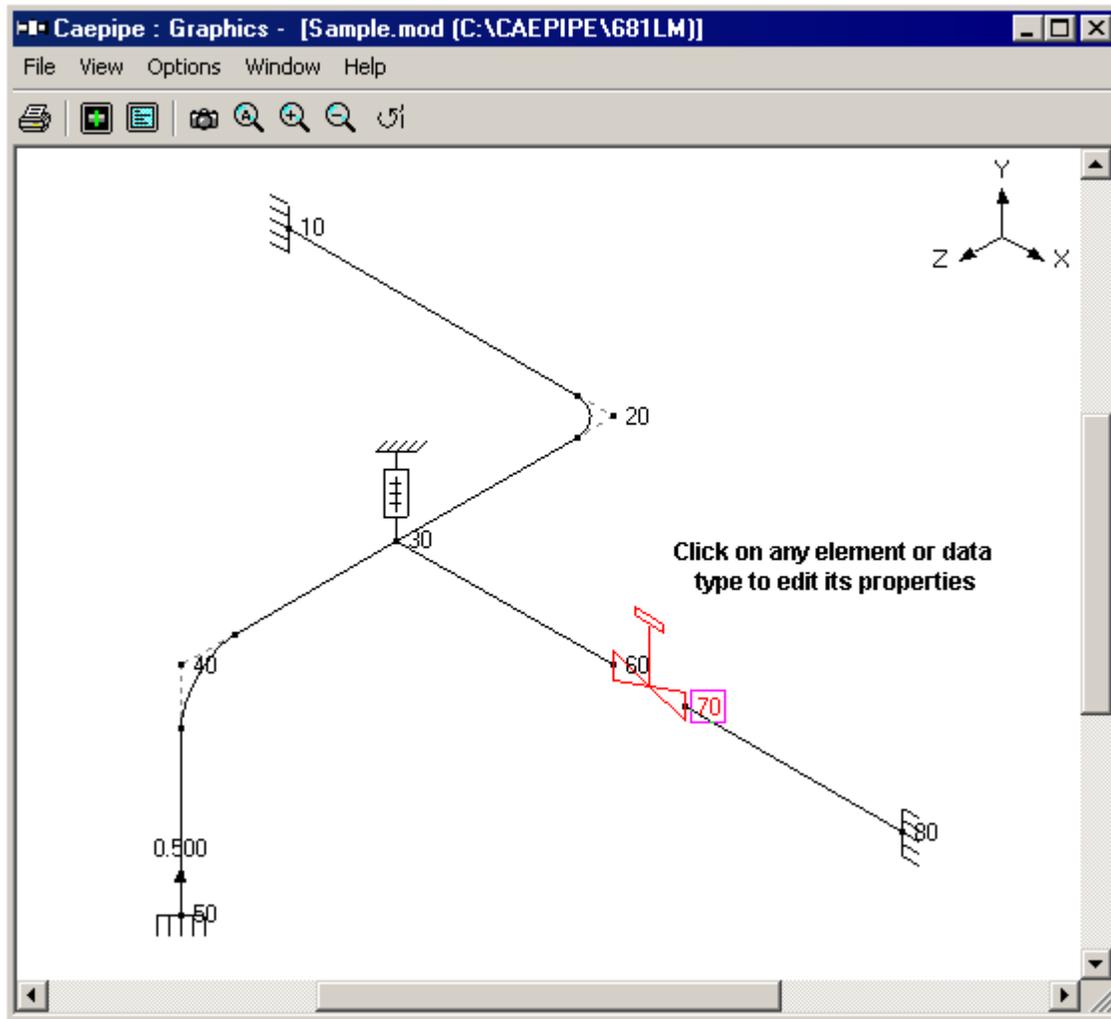
Clicking on an editable element or data item brings up the related dialog (same as that you see opened in the Layout window). You can enter or modify properties inside the dialog just as if you were editing in the Layout window. This feature gives you more flexibility during editing your model. See next page for illustrative images.

Modified or newly entered data is immediately updated in the Layout and the List windows. When you click on any graphics symbol, CAEPIPE automatically synchronizes the highlights in the Layout and the List windows to the row that contains the element/data item you clicked on, so that you see all the pertinent data about that element at the same time.

This feature works in Render mode too. Sometimes when symbols are closely grouped, you may want to zoom in on that area before clicking on a symbol. That way, CAEPIPE presents you the correct dialog box. Dialog boxes are opened only for elements/data that have editable data. For example, clicking on a pipe element will do nothing but will still synchronize the highlights anyway in the other windows to the same pipe element.

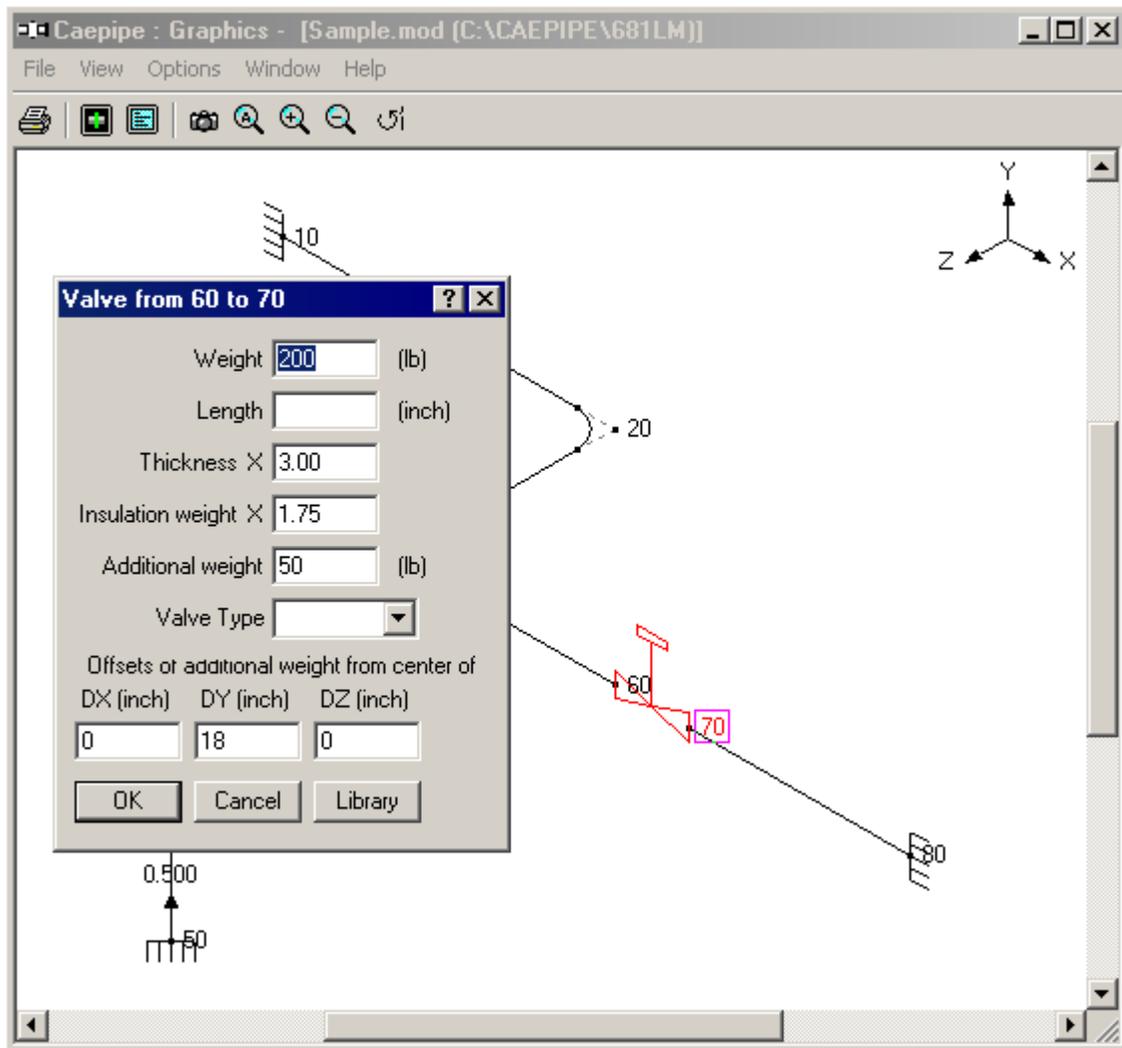
When you are viewing results, you can still click on an element/data item, except now the data you see presented are read-only (you cannot edit them).

Graphics Window



Clicking on a valve (as shown above) will open the dialog in the window shown next. You may modify any value inside the dialog and have it updated across all open windows.

Graphics Window



Hotkeys You Can Use

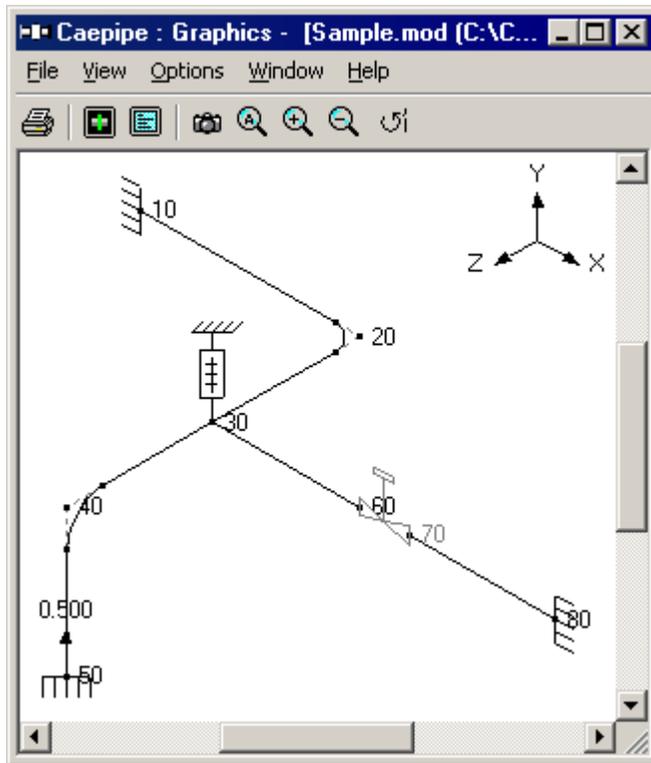
In addition to conventional MS-Windows hotkeys (such as Ctrl+P for Print, Ctrl+C for copy image, etc.), you may use the following keys.

Ctrl+A	Zoom All
Ctrl+Shift+C	Center image around a chosen point
Ctrl+T	Turn (rotate) image
Ctrl+S	Show
Ctrl+N	Show node numbers
Ctrl+D	Redraw image
Ctrl+R	Render image
Page Up	Zoom in
Page Down	Zoom out
F2	Move focus to Layout or List window (wherever the focus was before)
F3	Show/Hide Dots + Numbers
F4	Viewpoint
F5	Previous view
F6	Hide Current Element
F7	Make Transparent
F8	Freeze View
F9	Show All
F10	All Transparent
F11	All Opaque
F12	Show/Hide Selected Elements

Dynamic Resizing of Window Image

When you resize the Graphics window, CAEPIPE will automatically resize the image proportionately isotropically. Dynamic scaling makes it unnecessary for you to “Zoom All” every time you resize the window.

Graphics Window



Using Scroll Thumbs

When you want to move or pan the image, simply drag the scroll thumbs in the scrollbar and the image moves accordingly. If you have activated the Turn command (menu View > Turn, or Ctrl+T), then the scroll thumbs when moved using the mouse will rotate the image.

Graphics Synchronization with Open Text Windows

The graphics image has a flashing cursor (such as on an anchor or bend) that always synchronizes with the text window that has the focus. In congested models, the flashing cursor points precisely to the location of the element you are working on by acting as a flashing beacon. Example: Move the highlight in Layout to an anchor. The Graphics and the List windows (if open) automatically show or highlight the same anchor.

Dynamic Updating of Data in All Open windows: Layout, Graphics and List

Example: Delete an Anchor or change a Material name in the List window. The Layout and the Graphics windows are automatically updated immediately, showing the changes. You are not forced to perform edits from any one given window. You can get instant feedback.

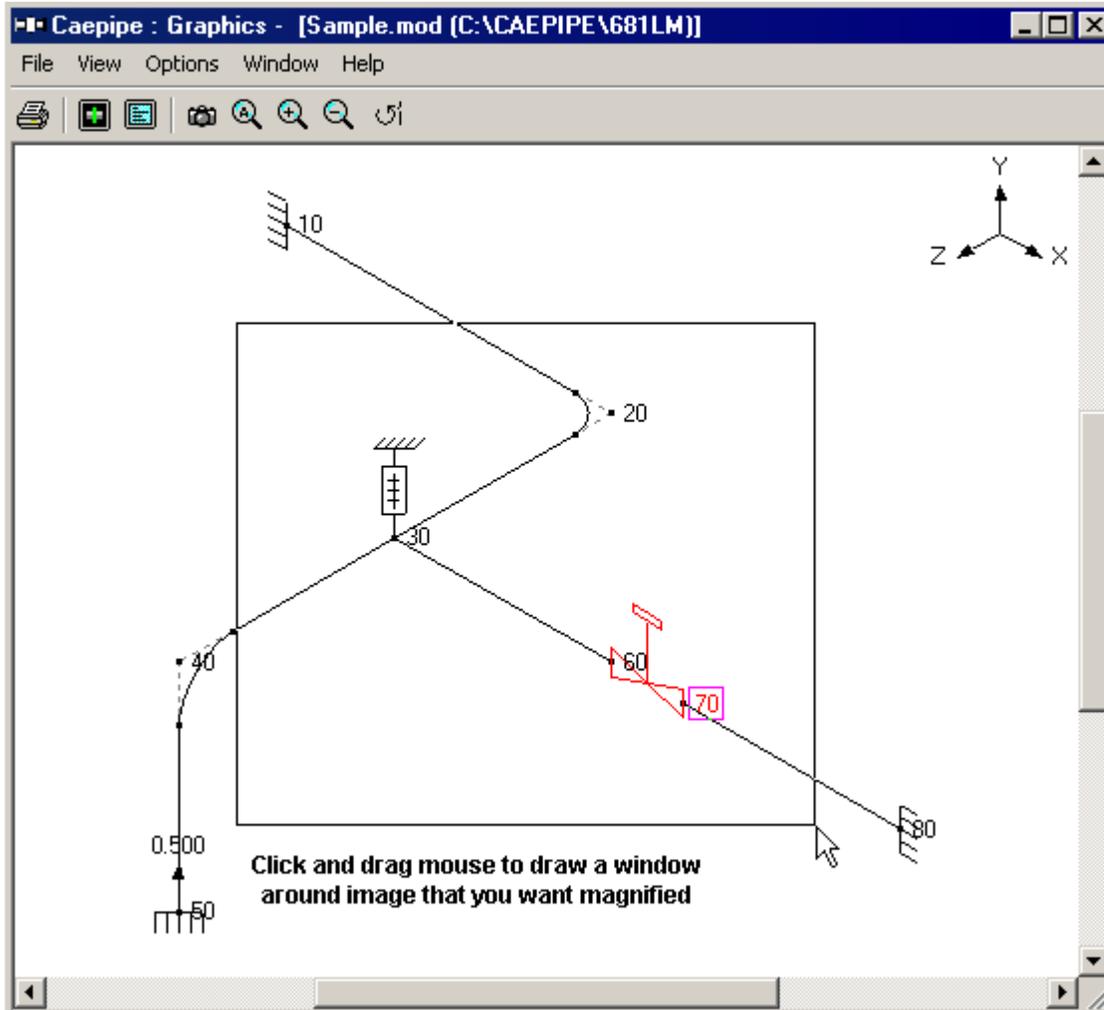
Simultaneous Visual Updates of Deflected and Mode Shapes

Simply switch between different load cases (or mode shapes) in the Results window to see the respective deflected/mode shape change too in the Graphics window.

Graphics Window

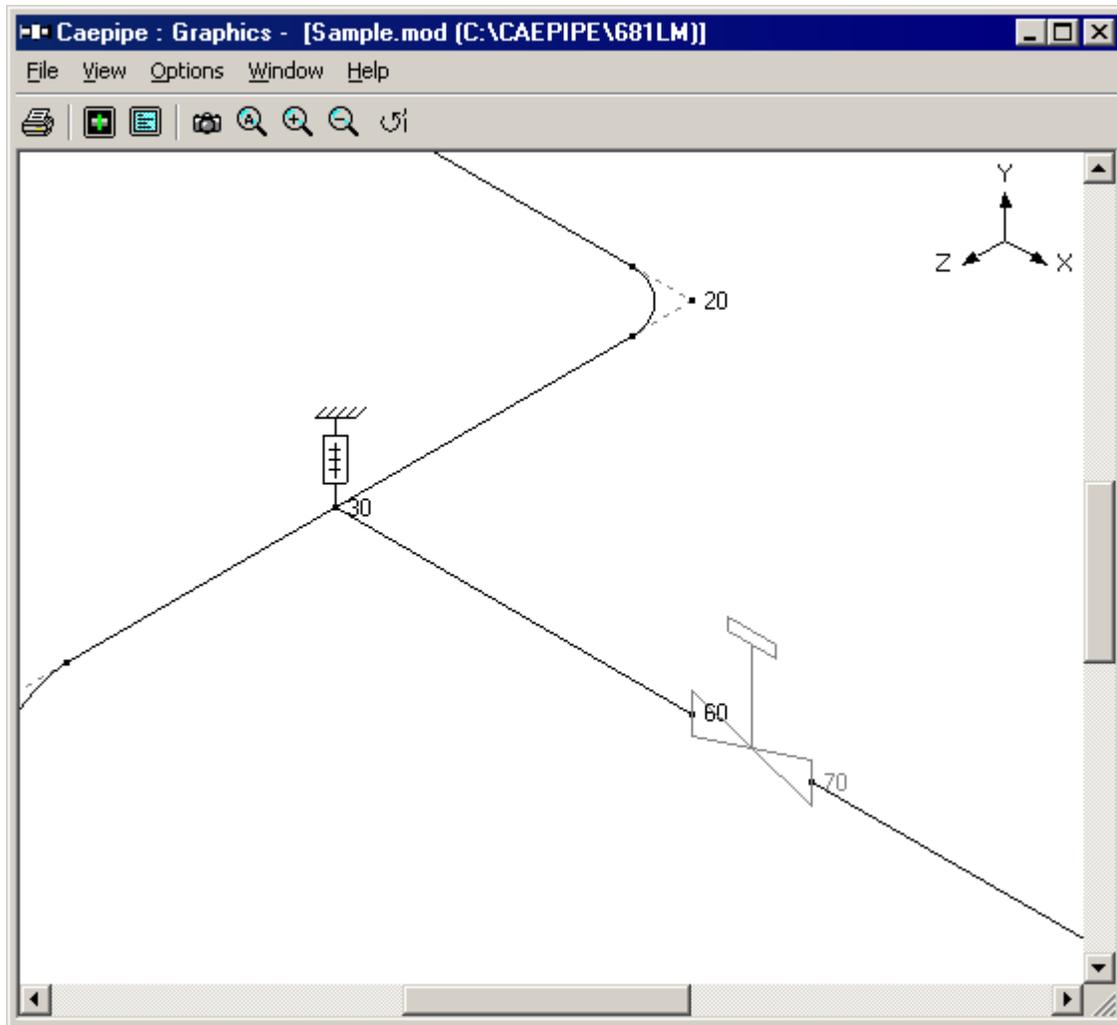
Zoom Window

To enlarge an area of image, simply use the mouse to draw a window (a box) around an area you are interested in. CAEPIPE enlarges the image (inside the drawn box) when you release the mouse button.



The zoomed image is shown next.

Graphics Window



Menus and Toolbar

The different menus in this window are explained in detail in the Menu section. A summary is given here.



File menu:

- You may print the image to a printer or a file (formats supported are Windows Enhanced Metafile (EMF), Encapsulated Postscript (EPS), HPGL plot file (PLT) and AutoCAD DXF). You can change the printer font from here.

Before printing to a printer or a file, you can change the title for the printout using the Plot Title command. Before printing a rendered image, you may want to select a white background and a quality of rendering (default is High). For DXF & PLT files use the "Drawing Size" command to change size.

Graphics Window

View menu:

- Mainly for Graphics operations (Viewpoint, Previous view and Zoom All).
- Selective display of node numbers and graphic symbols makes CAEPIPE flexible and powerful (use Show and Show Nodes commands). In addition, other standard graphical operations such as pan, zoom, previous view, viewpoint change, center image, etc., are available through the menus or a readily available context menu.
- Selective display of elements is possible. Use the commands such as “Make Transparent”, “All Opaque”, etc.
- Render Mode: Use this mode for realistic visualization of your model using OpenGL®.

Even in this mode, you can selectively display symbols and/or node numbers. Use a combination of Show and Show Nodes commands available in the context menu (or from View menu). Image can be printed, too.

Options menu:

- Customize the display image by choosing a display font, repositioning the Axes symbol, and changing the size of the toolbar icons.

Window menu:

- From here, you can move the focus to the text window from which it came. F2 will move focus between text and Graphics windows. The real advantage of this hotkey (F2) is for users who have a small monitor and work with maximized windows for input, list, graphics and results. In such a setting, one keystroke (F2) will quickly move the focus to another window (without having to either minimize the one where the focus is or switch windows through the taskbar).

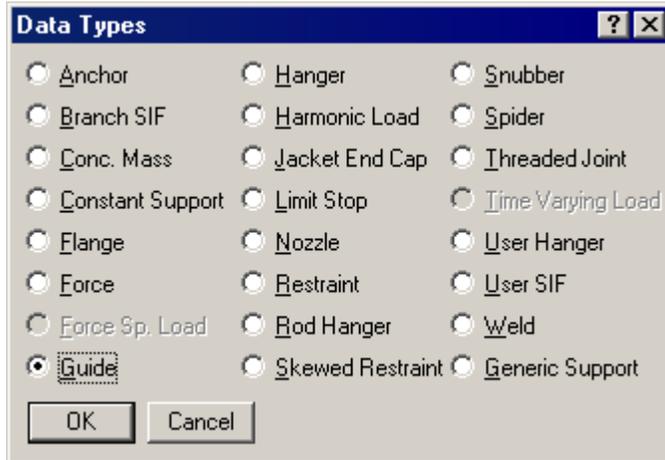
Help menu:

- For on-line help. For on-line help and information pertaining to the remaining period of your yearly Maintenance, Enhancement and Support (ME&S) agreement with SST.

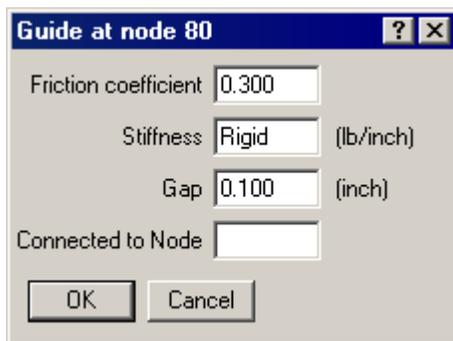
Guide

A guide is used in the field to control or direct pipe movement due to thermal loads. Likewise, its CAEPIPE equivalent allows axial movement while restraining the pipe against lateral translations (but not rotations). A guide restricts the translational movement normal to its axis, i.e., displacements are restrained in the local y and z directions.

A guide is input by typing “g” in the Data column or selecting “Guide” from the Data Types dialog.



The Guide dialog is shown.



Friction Coefficient

When a friction coefficient is entered, a nonlinear analysis is performed. In each iteration, the friction force is calculated which is friction coefficient times the normal force (the vector sum of y and z reaction forces). This force is applied in the local x direction opposing the axial motion of pipe. The solution converges when the displacement changes by less than 1% between successive iterations.

Stiffness

The default stiffness is rigid which is input by typing “r” or “Rigid” in the Stiffness field. A non-rigid stiffness may be entered by typing the value of the stiffness in the Stiffness field.

Gap

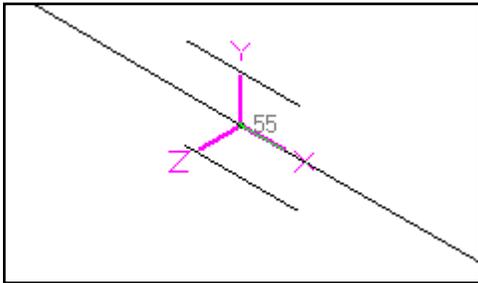
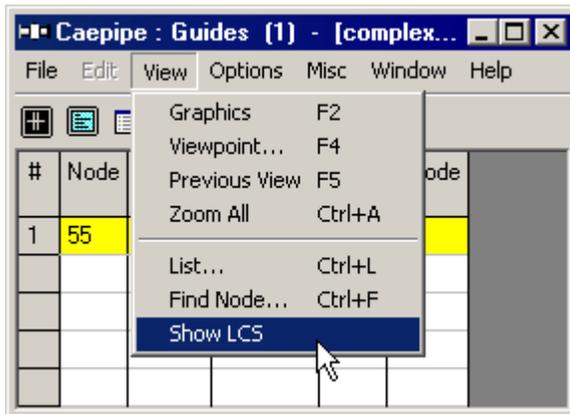
A clearance between the pipe and the guide, if present, may be entered as a Gap. The gap is assumed to be symmetric about the guide axis. This gap must be closed before any restraint forces can develop. If there is no gap leave this field blank or enter it as 0.0.

Connected to Node

By default the guide is assumed to be connected to a fixed *ground* point which is not a part of the piping system. A guide can be connected to another node in the piping system by entering the node number in the “Connected to Node” field.

Local Coordinate System (LCS)

A guide’s local x-axis is based on the preceding element. If a preceding element is unavailable, the following element is used to calculate the guide’s axes. The local coordinate system (LCS) may be viewed graphically from the Guide List window using the menu: View > Show LCS.

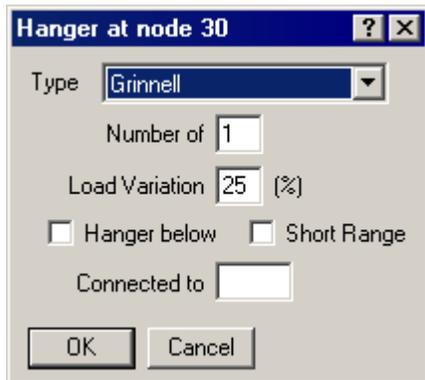
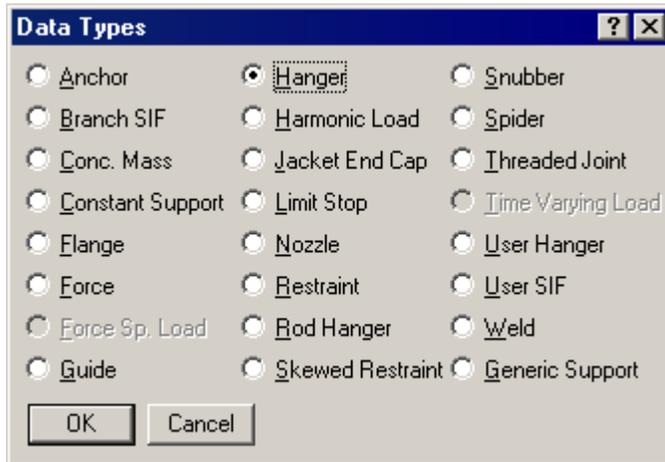


Guide forces in global coordinate system are “Print(ed) to file” (in addition to forces in local coordinate system), accessible from the Print command dialog.

Hanger

Variable spring hangers support a specific load while allowing vertical and lateral thermal movement from the installed to the operating condition. CAEPIPE assumes a spring hanger to always act in the vertical direction.

Also called a “To be designed” hanger, it is input by typing “h (Enter)” in the Data column (or typing “Han”) or selecting “Hanger” from the Data Types dialog. The Hanger dialog is shown.



Hanger

Type

The type (i.e., manufacturer) of the hanger can be selected from the drop-down combo box “Type.” The following hanger types are currently available:

Hanger Types		
ABB-PBS	Fee & Mason	Nordon
Basic Engineers	Flexider (30-60-120)	NPS Industries
Bergen-Paterson	Flexider (50-100-200)	Piping Services
Bergen-Paterson (L)	Fronck	Piping Tech & Products
BHEL Hyderabad	Grinnell	Power Piping
BHEL Trichy	Hydra	Sanwa Tekki(30-60-120)
Borrello	Lisega	Sanwa Tekki(85-170)
Carpenter & Paterson	Mitsubishi (30-60-120)	Sarathi
Comet	Mitsubishi (80-160)	Spring Supports
Corner & Lada	Myricks	SSG
Dynax	NHK (30-60-120)	
Elcen	NHK (80-160)	

Number of Hangers

The number of hangers is the number of separate hangers connected in parallel at this node. The stiffness and load capacity of each hanger are multiplied by the number of hangers to find the effective stiffness and load capacity of the hanger support at this node.

Load Variation

The load variation (in percent) is the maximum variation between the cold and hot loads. Typical value is 25%.

Hanger Below

Changes the graphical depiction only.

This should be used to specify whether the Hanger is connected below the pipe. Graphical symbol changes accordingly. This selection is valid only for Hanger and User Hanger and not for Constant Support Hanger nor Rod Hanger.

Short Range

Short range hangers are used if the available space is not enough for installing mid-range hangers. They are considered, however, a specialty item and generally not used. If a short range hanger is to be designed, check the Short Range check box.

Connected to Node

By default the hanger is connected to a fixed *ground* point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the “Connected to node” field. This node *must be directly above or below* the hanger node.

Hanger Design Procedure

1. Calculate Hot Load

Hot load is the load which balances the piping system under sustained loads. To calculate hot load, a preliminary sustained load analysis is performed in which all hanger locations are restrained vertically. If any anchor is to be released (so that the hanger rather than the nearby equipment takes the sustained load), it is released. The reactions at the hanger locations from this preliminary sustained load analysis are the hanger hot loads.

2. Calculate Hanger Travel

Vertical restraints (applied in step 1) at hanger locations are removed. Released anchors (if any) are restored. A preliminary operating load case analysis is performed. If multiple thermal load cases are specified only the first thermal load is used for this operating load case. The hot loads (calculated in step 1) are applied as upward forces at the hanger locations. Vertical displacements at the hanger locations obtained from this operating load case analysis are the hanger travels. If limit stops are present, the hot loads are recalculated with the status of the limit stops at the end of the preliminary operating load case. Then the hanger travels are recalculated using the recalculated hot loads.

3. Select Hanger

The hanger is selected from the manufacturer's catalog based on the hot load and hanger travel. The cold load is calculated as: $\text{cold load} = \text{hot load} + \text{spring rate} \times \text{hanger travel}$. The hanger load variation is calculated as: $\text{Load Variation} = 100 \times \text{Spring rate} \times \text{travel} / \text{Hot load}$. The calculated load variation is checked against specified maximum load variation. The hanger for which both the hot and cold loads are within the hanger's allowable working range and the load variation is less than the allowed load variation is selected. The hanger is selected such that the hot load is closest to the average of the minimum and maximum loads.

4. Install Hangers

If "Include hanger stiffness" is chosen in the Analysis options: The hanger spring rates are added to the overall stiffness matrix. The hanger cold loads are used in the sustained and operating load cases. If "Do not include hanger stiffness" is chosen in the Analysis options: The hanger spring rates are not added to the overall stiffness matrix. The hanger hot loads are used in the sustained and operating load cases.

For Lisega hangers, size column will report as Hanger Number, Type and Range instead of Hanger Number, Range and Type. For example, hanger having a spring rate of 2.1 N/mm, vertical travel of 30mm with load 440N will be reported as 21D.193 instead of 21D319.

Sustained Displacement During Hanger Selection

An Example Application

Sometimes, rotating equipment vendors (e.g. turbine vendors) require that there be no weight load imposed on the turbine connections after welding/bolting the pipe to the nozzle but prior to plant start-up. This can be accomplished only if the pipe is hung by variable spring hangers like a swing at the nozzle and if the pipe end of such "hung" pipe is almost perfectly aligned with the turbine nozzle prior to welding or bolting that pipe end to the

Hanger

turbine nozzle. In other words, the spring hangers that carry the weight of the “hung” pipe near the nozzle are to be sized and placed such that the resulting displacements (i.e., 3 translations and 3 rotations) at the pipe end are nearly zero, so that the pipe end need not be forcibly deformed in order to weld/bolt it to the turbine nozzle. This, in turn, makes sure that the weight load of the pipe is not imposed on the turbine nozzle prior to plant start-up.

CAEPIPE can be used to perform the above study by carrying out the following steps.

- a) During Step 1 of the Hanger Design procedure given above, release all 6 degrees of freedom of the anchor corresponding to that turbine nozzle, so that piping weight load will not be transferred to the nozzle during the initial Sustained load analysis (in which all hangers are pinned, thereby restraining the pipe vertically at the hanger locations).
- b) Review the preliminary Sustained Load Displacements computed during Hanger Selection via Results > Displacements > Mouse right click > Show sust. disp. during hanger selection. If the Sustained Load translations and rotations at the concerned pipe end are NOT nearly zero, try out different hanger locations till the preliminary Sustained Load displacements are nearly zero.
- c) When the Preliminary Sustained Load displacements at the concerned pipe end are nearly zero, the Support Loads at that Nozzle reported by CAEPIPE for Sustained Load case would be close to zero.

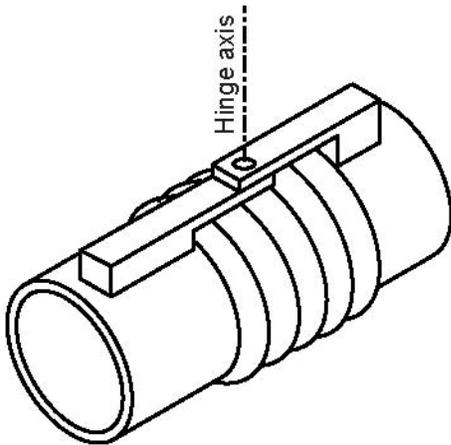
In results, the sustained displacements during hanger selection when hangers are pinned can be shown via Results > Displacements > Mouse right click > Show sust. disp. during hanger selection.

The screenshot shows the CAEPIPE software interface. The title bar reads "Caepipe : Displacements: Sustained (W+P) - [Sample.res (C:\CAEPIPE\681)]". The menu bar includes "File", "Results", "View", "Options", "Window", and "Help". Below the menu bar is a toolbar with various icons. The main window displays a table of displacements for 10 nodes. A context menu is open over the table, listing several options. The option "Show sust. disp. during hanger selection" is highlighted at the bottom of the menu.

#	Node	Displacements (global)					
		X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	20A	0.000	-0.008	0.002	-0.0103	-0.0015	-0.0000
3	20B	-0.000	-0.007	0.002	-0.0129	-0.0004	-0.0000
4	1000	0.000	0.004	0.002	-0.0038	0.0010	-0.0000
5	40A	0.002	0.002	0.002	0.0056	0.0016	-0.0000
6	40B	0.001	0.000	0.001	0.0022	0.0017	-0.0000
7	50	0.000	0.000	0.000	0.0000	0.0000	0.0000
8	1010	0.000	-0.013	0.001	-0.0020	0.0008	-0.0000
9	1020	0.000	-0.012	0.001	-0.0018	0.0008	0.0000
10	1030	0.000	0.000	0.000	0.0000	0.0000	0.0000

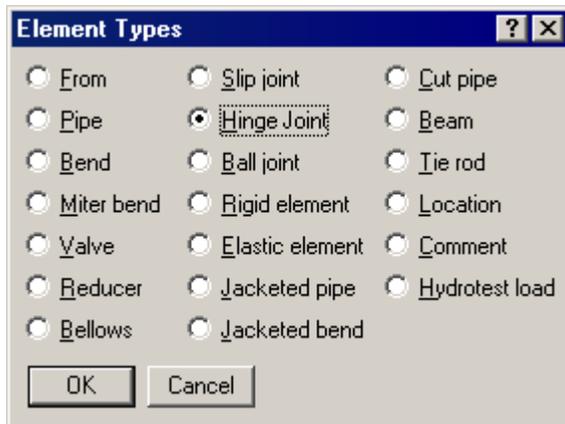
Hinge Joint

Hinge joint is an expansion joint designed to permit angular rotation in a single plane by use of a pair of pins that pass through plates attached to the expansion joint ends. Hinge joints are used in sets of two or three to absorb pipe movement in one or more directions in a single-plane piping system. A pair of hinge joints, separated by a section of piping, will act together to absorb lateral deflection. Hinge joints are designed to take the full pressure thrust.



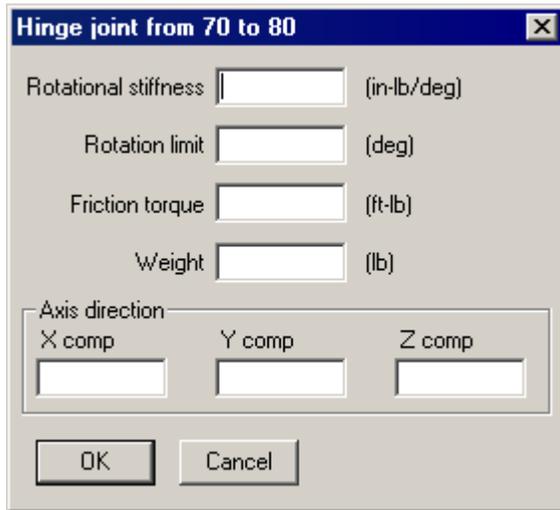
The two sides of the hinge joint shown are joined by hinge pins which are along the hinge axis shown in the figure. A hinge is modeled by two nodes, one on each side of the hinge joint. The two nodes of the hinge joint are coincident. So, it is a zero length element, i.e., the “From” and “To” nodes are coincident. Hence, the DX, DY and DZ fields in the Layout window should be left blank.

A hinge joint is input by typing “h” in the Type column or selecting “Hinge joint” from the Element Types dialog.



Hinge Joint

The Hinge joint dialog is shown.



The image shows a software dialog box titled "Hinge joint from 70 to 80". It contains several input fields for configuring a hinge joint. The fields are: "Rotational stiffness" (in-lb/deg), "Rotation limit" (deg), "Friction torque" (ft-lb), and "Weight" (lb). Below these is a section for "Axis direction" with three input fields: "X comp", "Y comp", and "Z comp". At the bottom of the dialog are "OK" and "Cancel" buttons.

Rotational Stiffness

Also called Angular stiffness. Input the stiffness around the rotational (hinge) axis. The stiffness value may be available from the manufacturer of the hinge joint or from test results. Otherwise engineering judgment may be used. The stiffness values may be left blank. In that case a very small value (1 in.-lb./deg.) is used internally to avoid dividing by zero.

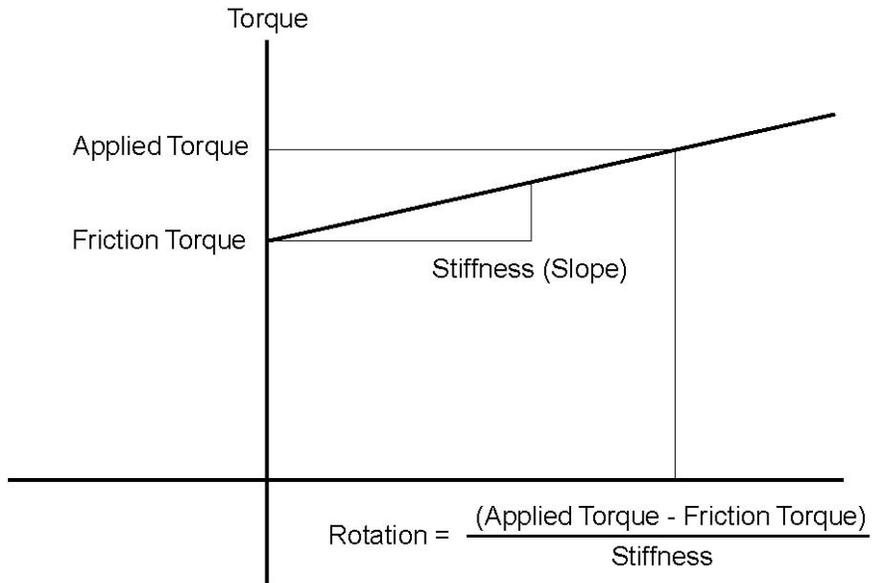
Rotation Limit

Rotation limit is an upper limit on the rotation of hinge joint in the plus or minus directions. Rotation limit of 0.0 (zero) means it is unable to rotate (i.e., it is rigid). Rotation limit of "None" or Blank means that there is no limit.

Friction Torque

The hinge joint will rotate only if the external torque exceeds the friction torque. Beyond that the rotation is proportional to the rotational stiffness of the hinge joint. The friction torque value may be available from the manufacturer of the hinge joint or from test results. Otherwise engineering judgment may be used. If you do not want friction in the hinge joint, the friction torque value may be left blank.

Hinge Joint



When the applied torque is less than friction torque, there is no rotation. When the applied torque exceeds friction torque, the rotation is calculated as shown above. When rotation limit is reached, there is no further rotation irrespective of the applied torque. See discussion for K_{bc} under Ball joint for additional information.

Weight

This is the total weight of the Hinge joint assembly.

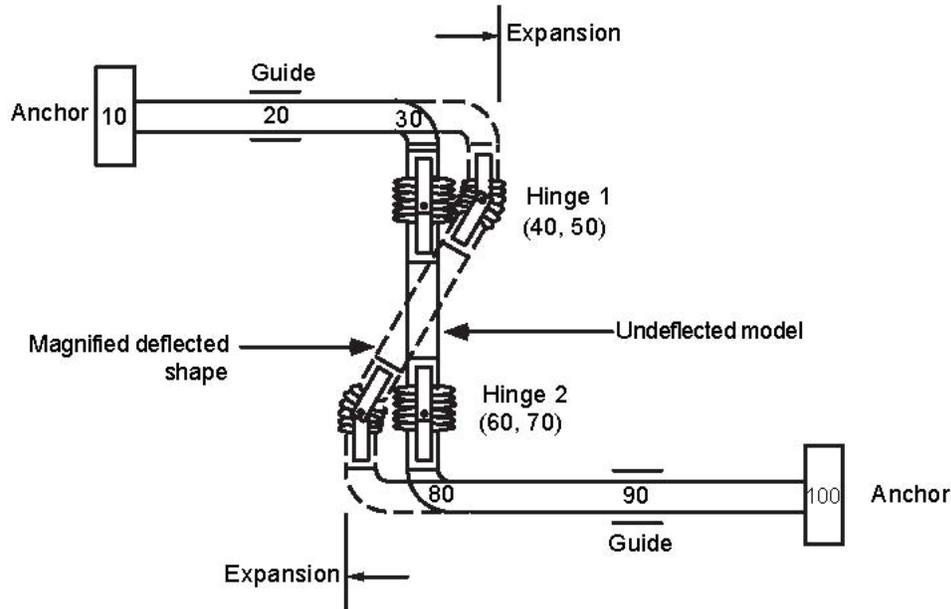
Axis direction

The hinge axis is specified by the “Axis direction.” See “Direction,” for more information on specifying a direction using X comp, Y comp and Z comp.

Hinge Joint

Example

Assume that we had the model shown below (with an exaggerated deflection) containing 6" piping with a pair of Hinge joints. Each Hinge joint has the following data: rotational stiffness of 66 in.-lb./degree, and weight of 35 lb.



The following steps describe the modeling procedure (with auto node incrementing, you do not have to type in the node numbers below):

- ▶ The first node 10 is already defined as an anchor. Press Enter to move to the next row.
- ▶ Type 20 for Node, 3'6" for DX, enter material, section and load names, Guide for Data. Press Enter to move to the next row.
- ▶ Input bend at node 30: Type 30 for Node, press Tab to move to the Type field. Type "b" and Tab to next column to enter a bend, 2' for DX, press Enter to move to the next row.
- ▶ Type 40 for Node, enter -1'6" for DY (as the offset from node 30 to node 40), press Enter to complete the bend and move to next row.
- ▶ Type 50 for Node. Type "h" in the Type column. This shows the hinge dialog.

Hinge Joint

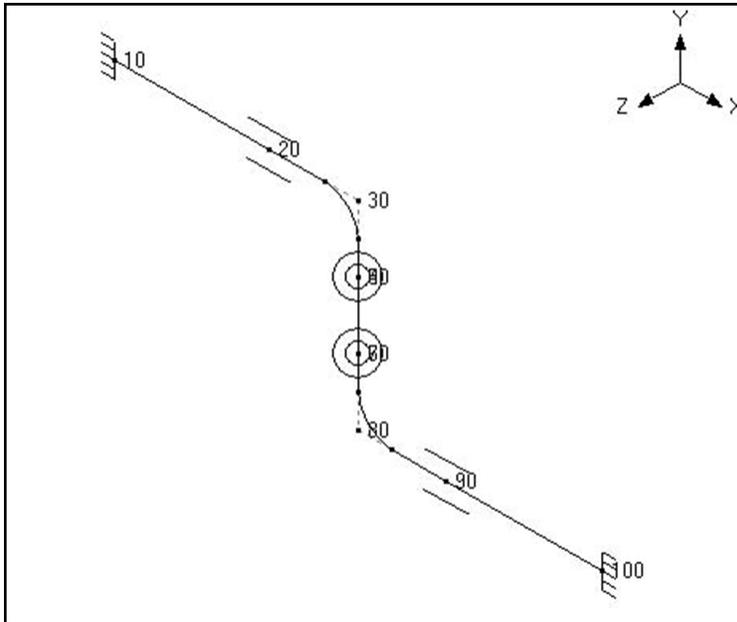
Enter 66 (in.-lb./deg.) for Rotational stiffness, 35 (lb.) for Weight, 1.0 for Z comp (axis direction), press Enter or click on OK to close the dialog. This completes the hinge input. Since there cannot be any offsets (DX, DY, DZ) for the hinge node from the previous node, the cursor automatically moves to the next row.

- ▶ Type 60 for Node, enter -1'6" for DY (as the offset from node 50 to node 60), press Enter to move to next row.
- ▶ Type 70 for Node. Type "h" in the Type column. This shows the hinge dialog. Enter the hinge data as before and click on OK to move to the next row.
- ▶ Input bend at node 80: type 80 for Node, press Tab to move to Type field, type "b" for Type and Tab to next column to enter a bend. Type -1'6" for DY (offset from node 70 to bend node 80), press Enter.
- ▶ Complete the model through nodes 90 and 100 similar to steps 1 and 2 above.

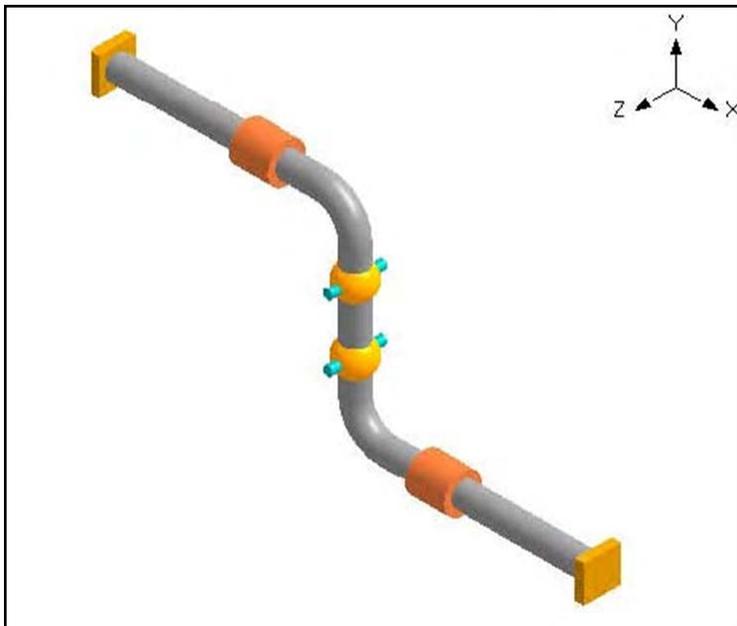
The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Loac	Data
1	Title = Hinge Joint								
2	10	From							Anchor
3	20		3'6"			1	6	1	Guide
4	30	Bend	2'0"			1	6	1	
5	40			-1'6"		1	6	1	
6	50	Hinge							
7	60			-1'6"		1	6	1	
8	70	Hinge							
9	80	Bend		-1'6"		1	6	1	
10	90		2'0"			1	6	1	Guide
11	100		3'6"			1	6	1	Anchor
12									

Hinge Joint



The rendered graphics is shown below:



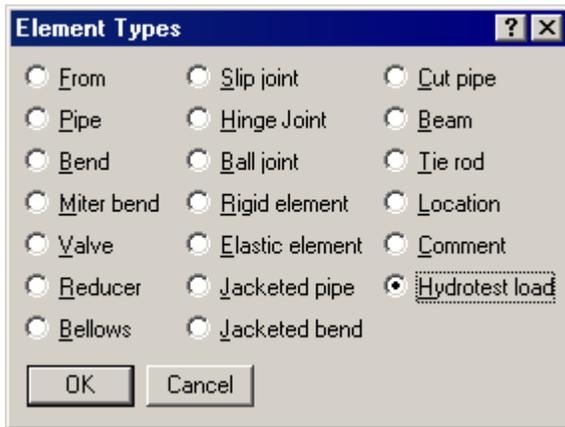
See under Expansion Joints for more examples.

Hydrotest

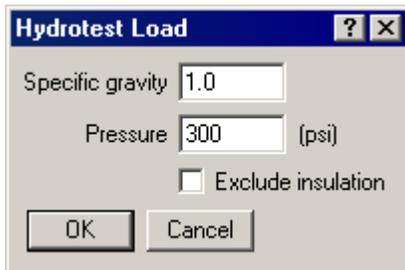
Use this load to analyze loading from a hydrostatic test which is performed by filling the piping system with a pressurized fluid (typically water) to check for leaks, etc., before putting the system in service.

During hydrotest, all hangers are assumed pinned (i.e., they act as rigid vertical supports). The hydrotest load is defined by the specific gravity of the test fluid (1.0 for water), test pressure and whether to include or exclude the insulation weight (because many times the hydrotest is performed before applying the insulation).

The hydrotest load is input by pressing “h” on an empty row in the Layout window (similar to pressing “c” for a comment) or on an empty row, selecting “Hydrotest load” from the Element Types dialog (Ctrl+Shft+T).



The “Hydrotest Load” dialog appears.



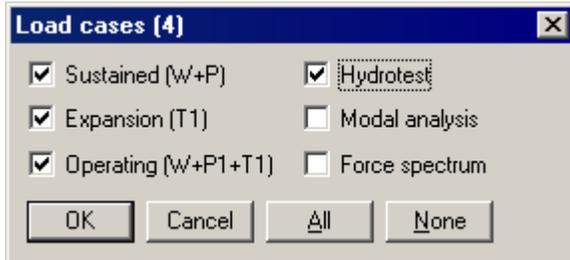
After the hydrotest load is input by pressing Enter or clicking on OK, the hydrotest load appears in the Layout window.

Caepipe : Layout (13) - [Sample1.mod (C:\CAEPIPE\681LM)]									
File Edit View Options Loads Misc Window Help									
[Icons: File, Folder, Print, Add, List, Table, Camera, Magnifying Glass]									
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	Hydrotest load: Spec. gravity = 1.0, Pressure = 300 (psi)								
3	10	From							Anchor
4	20	Bend	9'0"			A53	8	1	

Hydrotest

If you need to modify an existing hydrotest load, double click on the row that defines the hydrotest load to bring up the Hydrotest Load dialog. The hydrotest load is applied to the rows that follow until changed by another hydrotest load. The hydrotest load can be constant over the whole model or can be changed or even excluded in parts of the model.

To analyze the hydrotest load case, the Hydrotest load case must be selected using the command Loads > Load cases from the Layout window.



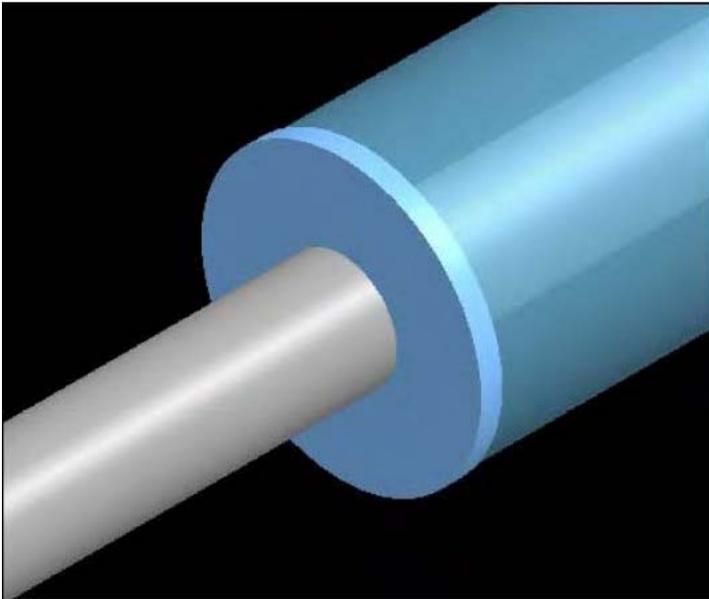
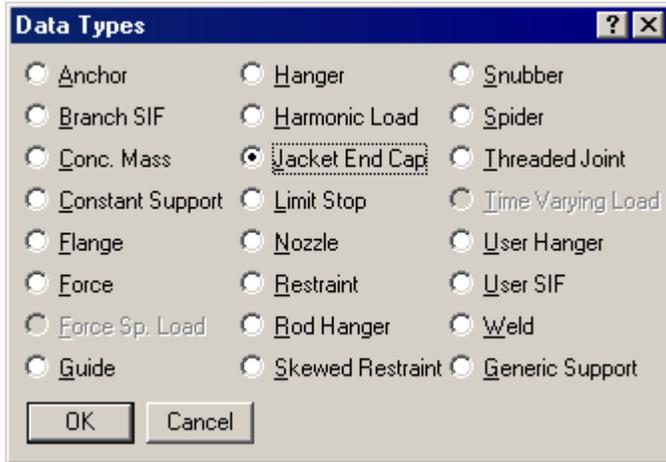
The hydrotest load case is analyzed as a Sustained load.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = 52# Let Down System								
2	10	From							Anchor
3	15	Bend		3'9"		CS	16	165	
4	20	Bend	-2'10"		2'10"	CS	16	165	
5	25			-2'5-3/4"		CS	16	165	Flange
6	30	Valve		-3'3"		CS	16	165	Force sp load
7	35	Bend		-2'5-3/4"		CS	16	165	
8	40		20'0"			CS	16	165	Hanger
9	45	Bend	1'0"			CS	16	165	
10	50	Bend	5'0"		-5'0"	CS	16	165	
11	55				-12'0"	CS	16	165	Hanger
12	60				-5'0"	CS	16	165	Weldolet
13	65				-10'0"	CS	16	165	Hanger
14	70				-5'0"	CS	16	165	Anchor
15	New piping								
16	Hydrotest load: Spec. gravity = 1.0, Pressure = 425 (psi), Exclude insulation								
17	60	From							
18	75		1'5-3/8"			CS	10	165	Flange
19	80	Valve	2'7"			CS	10	165	Flange
20	85		0'6-3/8"			CS	10	165	
21	90	Reducer	0'8"			CS	12	165	
22	95	Bend	5.5400			CS	12	165	
23	100	Bend			6'3"	CS	12	165	
24	105		-4'0"			CS	12	165	Hanger

Jacket End Cap

Use a jacket end cap to rigidly connect all six degrees of freedom of the coincident nodes of a jacketed pipe (i.e., the node on the core pipe and the corresponding node on the jacket pipe (“J” node) are tied together so that both nodes have the same displacements and rotations).

A jacket end cap is input at a jacketed pipe node by typing “j” in the Data column or selecting “Jacket End Cap” from the Data Types dialog.



You will need this to secure the jacket pipe to the core pipe rigidly. Sometimes (depending on the combination of restraints), you may get a “Stiffness Matrix not positive definite” error, which may be corrected by inserting a jacket end cap.

Jacketed Piping

Jacketed piping is used when the primary state of the pipe contents (fuel, chemicals such as resins, etc.) needs to be maintained during transport. An outer (jacket) pipe surrounds the inner (core) pipe that contains the fluid or the chemical. The jacket provides external heating or cooling as required along the length of the core pipe.

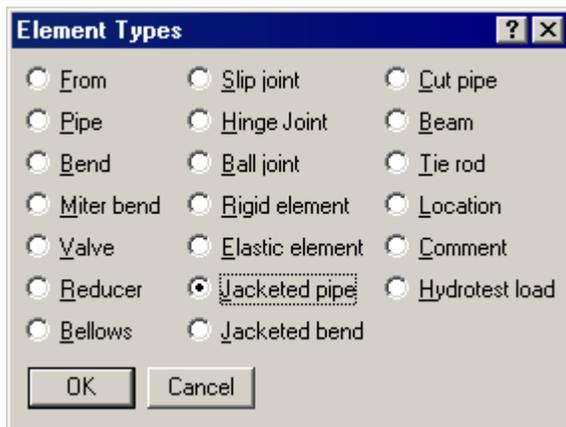
The terminology used here is as follows:

- *Jacketed* piping refers to the entire assembly, i.e., a core pipe with a jacket on the outside.
- *Jacket* pipe refers only to the outside pipe.
- *Core* pipe refers only to the inside pipe that contains the fluid.

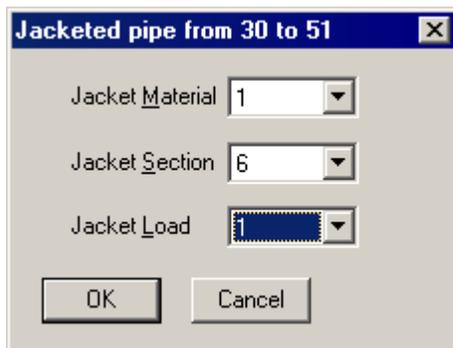
In CAEPIPE, jacketed piping need only be modeled once, not twice (as in some other programs). CAEPIPE models the outer jacket pipe *along with* the inner core pipe on the Layout window. Each row defines a jacketed piping element. The jacket and the core pipes may have different materials, sections and (P, T) loads.

Jacketed Pipe

A Jacketed pipe is input by typing “JP” under Type or selecting “Jacketed pipe” from the Element types dialog. The material, section and load specified in the Jacketed Pipe dialog apply to the jacket pipe while the ones mentioned on the layout row (next to offsets) apply to the core pipe.



The Jacketed Pipe dialog is shown.



Jacketed Piping

The jacket's material, section, and load names are input here (1, 2 and 1 as shown). CAEPIPE retains the properties of a jacket pipe until changed so there is no need to retype the names of the jacket properties every time you input a jacketed pipe.

The ends of the jacket and core pipes need to be rigidly connected using the "Jacket End cap" data type (see previous topic). Also, "Spiders" need to be input at locations found in the physical assembly. You may have to break up the piping into smaller elements to insert spiders at appropriate locations. See example given later in this section.

In case you are analyzing for wind, it may be more accurate to specify a different load for the core pipe alone that does not specify the Wind load since the core pipe is not exposed. Same applies to the core pipe insulation if the core pipe does not have insulation.

Internal nodes

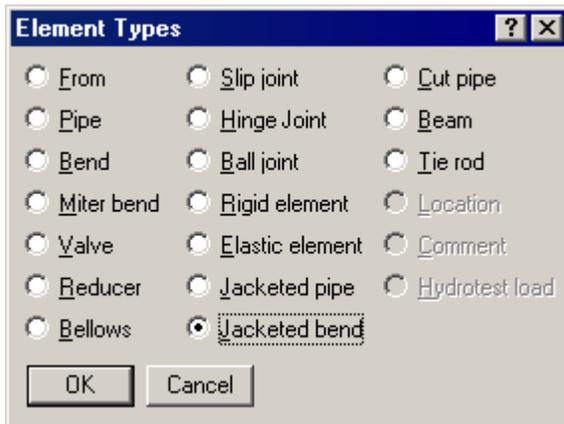
CAEPIPE generates a "J" node for jacket pipes. For example, if you had a jacketed pipe from node 10 to 20, CAEPIPE generates 10J and a 20J as (internal) jacket nodes (that may be referenced on the layout screen).

These internally generated nodes may be used to specify data items such as a spider, jacket end cap, support (hanger, restraint), forces on the jacket. See example later in this topic.

Jacketed Bend

A Jacketed bend consists of a core bend (with a straight portion) surrounded by a jacket bend (with a straight portion of jacket pipe).

A Jacketed bend is input by typing "JB" in the Type column or by selecting "Jacketed bend" from the Element types dialog.



Jacketed Piping

The Jacketed Bend dialog is shown.

The screenshot shows a dialog box titled "Jacketed Bend at node 51". It is divided into several sections:

- Jacket:** Material (A53), Section (6), Load (1).
- Core:** Material, Section, Load (all empty).
- Jacket Bend Radius (inch):** Radio buttons for Long, Short, and User. The Short option is selected.
- Core Bend Radius (inch):** Radio buttons for Long, Short, and User. The Short option is selected.
- Bend Thickness (inch):** Jacket and Core (both empty).
- Intermediate Bend Nodes on Jacket:** Two rows, each with Node and at Angle (deg) fields (all empty).

At the bottom are OK and Cancel buttons.

Jacket (properties)

The jacket's material, section, and load names are input here. The properties of a jacketed pipe are retained until changed. So, there is no need to retype the names of the jacket properties every time you input a jacketed pipe.

Core (properties)

Presently these properties are disabled. You need to enter them on the layout row under Material, Section and Load.

Bend radius

Separate bend radii may be specified for the core and the jacket pipes.

Note that CAEPIPE does not check for interference between the core and the jacket arising out of differently specified bend radii.

The bend radius for the core pipe is the same as that of the jacket pipe, since the two bends are generally concentric. Use the Render feature in the Graphics window to check visually for interference, however.

Bend thickness

Separate bend thicknesses may be specified for the core and the jacket bends, if they are not default jacket and core section thicknesses.

Intermediate nodes

You can define additional nodes on the outside jacket of a jacketed bend for locating supports. You may also use internal nodes generated by CAEPIPE. See Internal nodes below.

Internal nodes

CAEPIPE generates “C” and “D” nodes for the Jacketed bend on the jacket at the near and far ends of the bend. The core pipe (bend) has its own “A” and “B” nodes. Example: When you define a Jacketed bend from node 20 to node 30, 30A, 30B (nodes on core bend), 30C and 30D (nodes on jacket) are generated. Nodes (30A, 30C) and (30B and 30D) are coincident only if the core and the jacket pipes have the same bend radii. See figure.

These internal nodes may be used to specify data items such as a spider, jacket end cap, supports, forces, etc.

Split a Bend/Pipe

A jacketed pipe/bend may be split by using the Split option in the Edit menu in the Layout window.

Contents Weight

The weight of the contents between the jacket and the core pipes is calculated in the following manner:

(a) Twice the insulation thickness on the core pipe is added to the outer diameter of the core pipe. (b) Area is calculated by using the above diameter (a). (c) The internal area of the jacket pipe is calculated. (d) The area (b) is subtracted from the internal area of the jacket pipe (c) and this result is further used to compute weight.

Jacketed Reducer

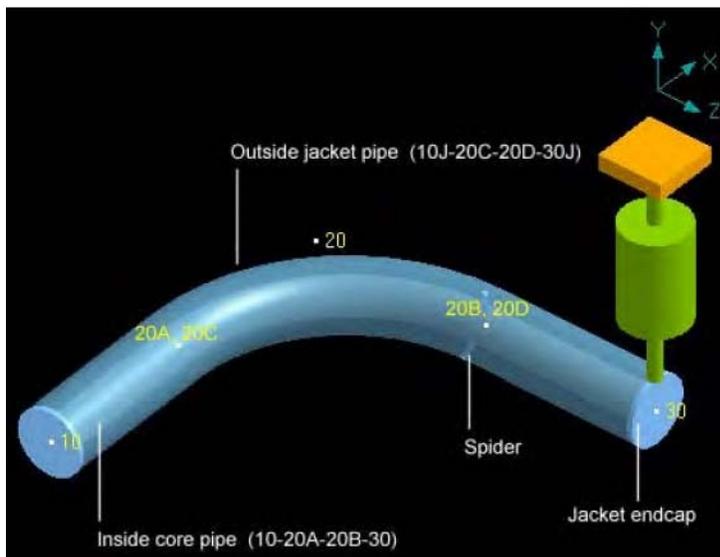
See modeling procedure in topic on the Reducer.

Example: Jacketed Pipe/Bend

The figure below shows a Jacketed pipe with a Jacketed bend (at node 20-TIP). Observe the spider at the far end of the bend (node 20B).

Jacketed Piping

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Jacketed Pipe								
2	10	From							
3	20	Jbend	3'0"			1	3	1	
4	30	Jpipe			3'0"	1	3	1	
5	10J	Location							Anchor
6	20B	Location							Spider
7	30J	Location							Hanger
8	30	Location							Jacket endcap
9									



The nodes 10J, 20C, 20D, 30J, 20A, 20B are internally generated nodes. You may use them for specifying data items such as spiders, supports (hanger, restraint), forces, etc.

The nodes (10, 10J), (20A, 20C), (20B, 20D) and (30, 30J) are coincident. The nodes 20A and 20B are coincident with 20C and 20D respectively only if the core and the jacket pipes have the same bend radii.

Note that the core and jacket nodes are not connected even though they are coincident. The pipes have to be supported and connected using supports and jacket connections. An anchor each at nodes 10 and 10J is specified. The hanger is at node 30J since it is attached to the jacket.

A jacket connection of the type spider at node 20B acts as an internal guide between the core pipe and the jacket pipe, that is, it prevents any radial movement but allows sliding, rotating and bending movement between core and jacket pipes.

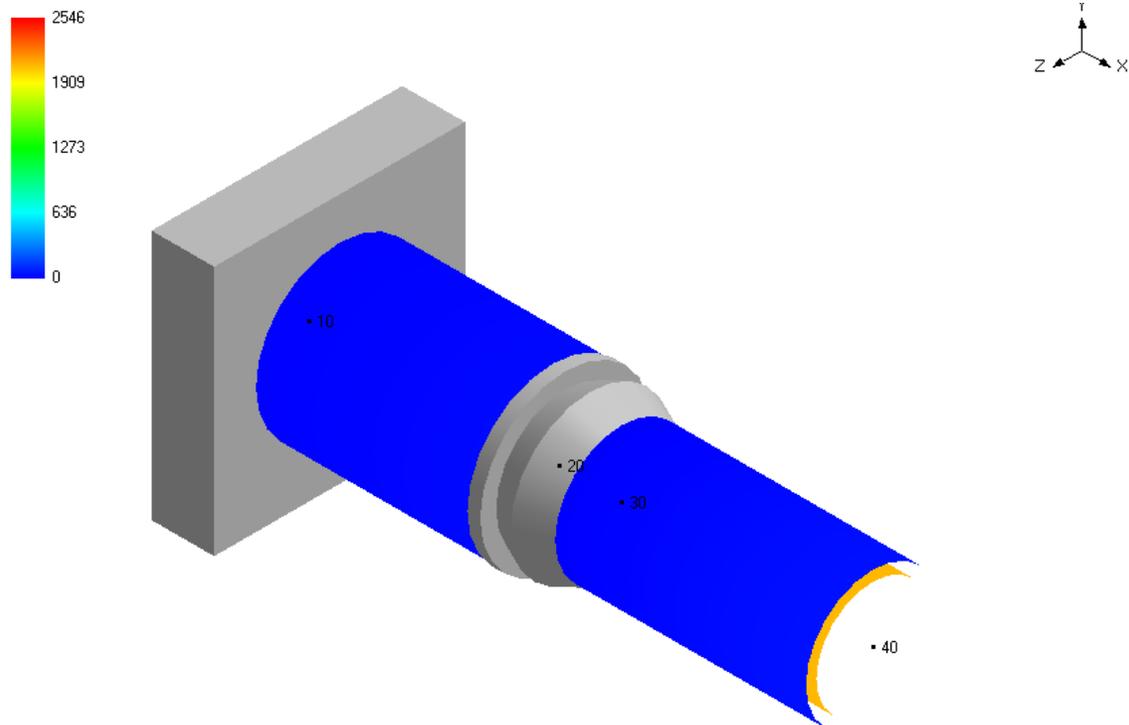
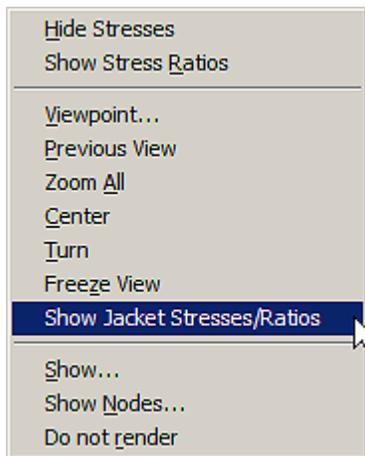
Jacketed Piping

In case forces transmitted from core pipe to jacket through spiders are required, then spiders specified on core pipe can be replaced by guides with “connected to” nodes specified as the corresponding nodes on the jacket pipe.

The end cap at node 30 connects the core and jacket pipes rigidly.

Jacketed Piping Stresses/Ratios

CAEPIPE provides an option for you to display the color-coded stress/ratio contour for jacketed piping in the graphics window context menu. The default stress contour is for the core piping. Upon selecting the command for Jacket stresses as shown below:



Layout Window

The text window displayed (that contains the piping layout) when you open a piping model is called the Layout window.

A Layout window, as the name indicates, allows you to create or modify the layout of your piping system.

Other things you can do here are:

- Specify material types, pipe sections and loads.
- Use some graphics features (zoom all, viewpoint, etc.) to view the model.
- Define analysis options such as piping code to use, reference temperature, cutoff frequency for mode extraction, as well as set units, etc.
- Define load cases (Sustained, Expansion, Response Spectrum, etc.) for analysis.

In the Layout window, the row you are working on is highlighted by the yellow highlight bar (in figure below, the highlight is on the Title row). The following illustration names the different fields in the Layout window. Presently, there are ten fields (columns) in the Layout window.

The Header row with column headings (see “Clickable Header row” in the illustration) is a special row that allows you to click on it to perform a related operation. For example, you can click on Type to show Element types.

The screenshot shows a software window titled "Caepipe : Layout (11) - [Sample.mod (D:\kpipe\510j)]". It has a menu bar (File, Edit, View, Options, Loads, Misc, Window, Help) and a toolbar. Below is a table with 12 rows and 10 columns. The first row is highlighted in yellow and contains the text "Title = Sample problem". The second row is highlighted in green and contains "6" std pipe" in the "Load" column. The rest of the table contains numerical and text data. To the right of the table is a legend with 8 numbered items: 1 Node number, 2 Type of Element (pipe, bend, valve, etc.), 3 X, Y and Z offsets (units can be changed), 4 Material name, 5 Pipe section name, 6 Load (T,P) name, 7 Additional data at each node (flange etc.), 8 Comment row (can be inserted anywhere). The legend items are numbered in red. The table columns are numbered 1 through 7 in red at the bottom row.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data	
1	Title = Sample problem									
2	10	From							Anchor	
3	20	Bend	9'0"			A53	8	1		
4	30				6'0"	A53	8	1	Hanger	
5	40	Bend			6'0"	A53	8	1		
6	50			-6'0"		A53	8	1	Anchor	
7	6" std pipe							8		
8	30	From								
9	60		6'0"			A53	6	1		
10	70	valv	0'			A53	6	1		
11	80		6'0"			A53	6	1	Anchor	
12	1	2	3	3	3	4	5	6	7	

Description of Fields (numbers in parentheses refer to numbers in the illustration above).

(Row Number)

The first field contains the row number, automatically incremented and non-editable.

(1) Node

This is the node number field where you type in node numbers as you model the piping system. The numbers should be numeric except when you are using internally generated nodes (A, B, C, D, & J).

To change an existing node number, click in this field and use Backspace or Delete key. Type in a new node number. Press Tab or Enter and confirm the change.

Layout Window

You need not enter each node number. It is automatically generated when you press Tab to move to the next field. You can set an increment under the menu Options > Node increment command.

(2) Type

This is the element type field. An element (in-line) is a fitting or a component between two nodes. This could be a pipe, a bend, a valve, an expansion joint or most items listed under the Element types dialog. See a description later in this section under Element Types.

(3) DX, DY, DZ

These fields are offsets (also called relative coordinates) in the three global X, Y and Z directions. They are measured from the previous node's location.

Offsets are components of lengths of elements (exceptions are zero-length elements like a ball and a hinge joint). For example, to specify the offsets of a 6 feet long pipe routed 30 degrees to global X-axis (in X-Y plane), the offsets are: DX=5.196 ft. ($6 * \text{Cosine } 30^\circ$), DY=3 ft. ($6 * \text{Sine } 30^\circ$).

If the element is in a skewed direction, then calculate the X, Y and the Z components (using basic trigonometry) and input them here, or use the Slope command to have CAEPIPE calculate the components.

When you specify values for the DX, DY and DZ fields for a "From" element type row, they are treated as (absolute) coordinates of the node rather than (relative) offsets from the previous node. If the model begins at the global origin (0,0,0), you do not need to specify coordinates for the first From node. For any other From row with a previously defined node number, you do not need to enter coordinates either because CAEPIPE already knows the coordinates of this node (for example, node 30 on row 8 in the previous figure).

The offsets can be negative depending on the way you route your piping.

The default number of decimals you can input for a length value is four. With an environment variable "INCREASE_DIGITS" set to "YES", the number of digits you can input after the decimal can be increased to seven. This data can be printed other than to TXT and CSV formats.

(4) Matl (Material)

Type in the material name of the material you want to use for this element. See Find and Replace command under Layout window > Edit menu, for editing options.

(5) Sect (Section)

Type in the section name of the pipe section you want to use for this element. See Find and Replace command under Layout window > Edit menu, for editing options.

(6) Load

Type in the load name of the load you want to use for this element. See Find and Replace command under Layout window > Edit menu, for editing options.

Layout Window

(7) Data

Use this field to enter data items at a node such as anchors, flanges, nozzles, hangers, SIFs, etc. See Data types below for how to specify an item.

(8) Comment

You can create a comment anywhere you like in the Layout window by simply pressing “c” (first) on an empty row. You can use this feature to annotate the model with notes. Comments print along with the layout data so you can use this feature to document (revisions to) the model.

You can specify a specialized form of a comment to input a Hydrostatic test load by pressing “h” first on an empty row. The load is applied to the model from that row onwards until modified again (in the next figure, the Hydrostatic load is applied to model from row 8 onwards).

5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	Hydrotest load: Spec. gravity = 1.0, Pressure = 300 (psi), Exclude insulation								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor

Menu bar

File Edit View Options Loads Misc Window Help

File menu:

- Standard file operations with a few special ones to note - Open Results (model file(.MOD) must be present for this operation), Print (to file), Analyze (model), QA Block, Recently opened file list. You can open not only the CAEPIPE model file, but also material and spectrum library files, and the CAEPIPE Results files.

Edit menu:

- You can edit all types of data (elements, supports, range of rows) from here. Use Ctrl T to edit element (bend, valve), Ctrl D to edit Data (flange, anchor). You can change properties for groups of elements, split elements and generate copies of existing rows.

View menu:

- Mainly for Graphics (Viewpoint, Previous view and Zoom All). Other functions are to open the List window (Ctrl L) by selecting one of the displayed items, and search for a node number.

Options menu:

- For setting Analysis options such as Piping code, Reference Temperature, Pressure correction, cutoff frequency for mode extraction, etc., set Units (SI, Metric, English or a combination), and customize the look of CAEPIPE by choosing a display font.

Layout Window

Loads menu:

- Set the load cases for analysis, and specify loads for the model such as static seismic, wind, spectrum, time history, etc.

Misc menu:

- View model coordinates, define/view materials, sections, loads (also for beams), pumps, compressors and turbines, time functions and spectra.

Window menu:

- From here, you can move the focus to another window (such as Graphics or List). F2 will move focus between text and Graphics windows and F3 between open text windows. The real advantage of these hotkeys (F2 and F3) is for users who have a smaller monitor and work with maximized windows for input, list, graphics and results. In such a setting, one key press (F2 or F3) will quickly move the focus to another window (without having to either minimize the one where the focus is or switch windows through the taskbar).

Help menu:

- For on-line help and information pertaining to the remaining period of your yearly Maintenance, Enhancement and Support (ME&S) agreement with SST.

Toolbar



The toolbar presently is organized into three groups: File, Window and Graphics operations.

- File operations are New (model), Open (a model), Save (model), and Print.
- Window operations are (move focus to) Graphics window, List window, and display a dialog of List of items to show an item in the List window.
- Graphics operations are Viewpoint and Zoom all.

In the opening window (when you first start CAEPIPE), there is a button to open the last opened model file (hotkey: Enter). This is not available from other windows.

Hotkeys You Can Use

You can use following hotkeys in the Layout window:

Tab	Move cursor from field to field (left to right)	Home	Move cursor to the first character in a field
Shift Tab	Same as Tab but right to left	End	Move cursor to the last character in a field
Page Up	Move highlight up one page	Ctrl+Home	Top of model
Page Down	Move highlight down one page	Ctrl+End	End of model
Delete/BkSp	Delete characters	Arrow keys	Move highlight one row/field at a time
Ctrl+Insert	Insert an empty row	Ctrl+X	Delete a row
Ctrl+Enter	Duplicates last layout row	F1	Help
F2	Graphics window	F3	List window

Layout Window

Other hotkeys listed under menus.

Hotkey combination: CTRL+ENTER (for duplicating the last layout row), works for Beam Elements (in addition to PIPE and JPIPE).

Function Keys

F1: Opens on-line help.

F2: This key moves the focus to and from the Graphics window.

F3: This key moves focus to and from the List window.

F4: Opens the Viewpoint dialog which you can set for the Graphics window.

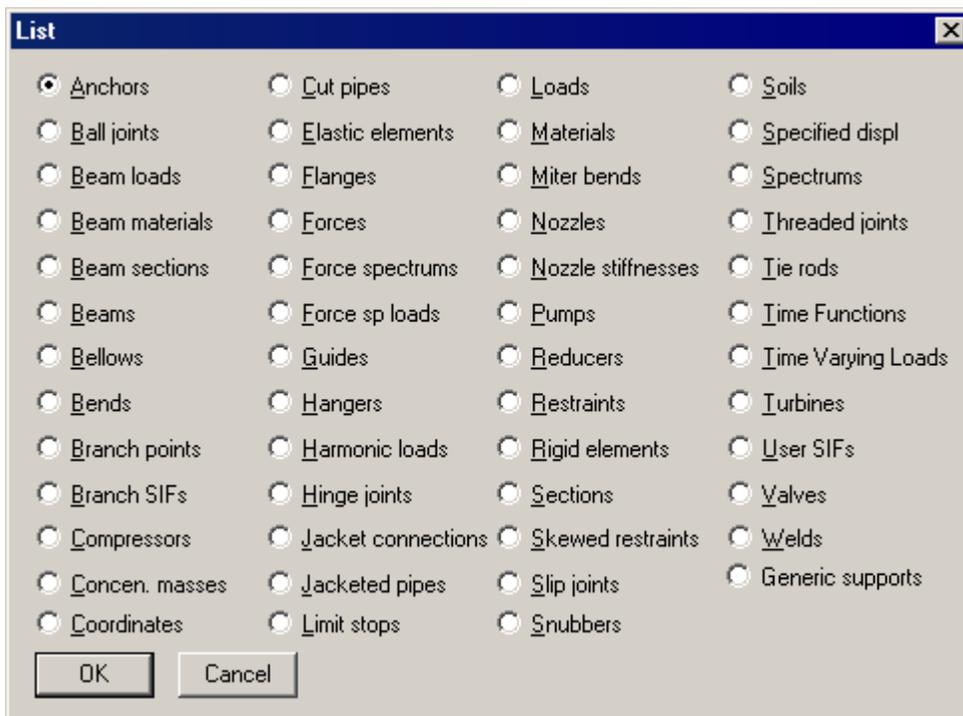
F5: This function key restores the Graphics window to the previous graphics view.

List (Ctrl+L)

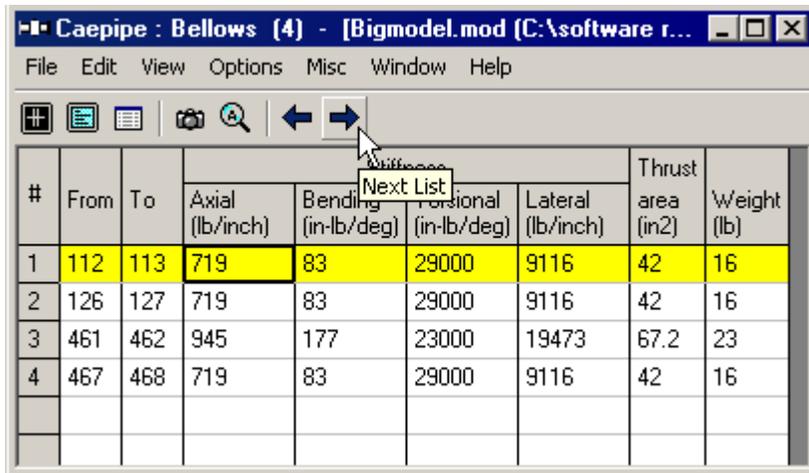
List is a powerful feature that helps you see itemized lists of information in the model. All input data can be viewed through this command. For example, you can view all element and data types specified in the model, list Coordinates of the nodes of the model, Specified displacements, Nozzle stiffnesses, Branch nodes, Threaded joints, Pumps, Turbines, Compressors, Materials, Sections and Loads, etc.

This is a useful tool to verify information after modeling is finished, for example, to check whether all bends have the same bend radius or whether limit stops are aligned in the correct directions or how many anchors have been specified in the model and so on.

This command is “intelligent” in that the list dialog shows only those items that exist in the model. Consequently, List can display as small a list as in the first figure below or as comprehensive a list as shown in the next figure.



Layout Window



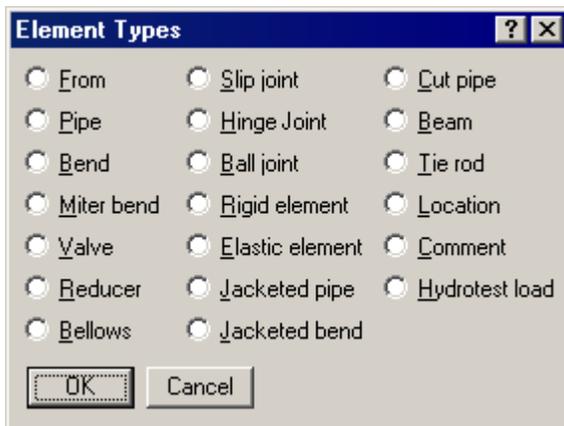
#	From	To	Axial (lb/inch)	Bending (in-lb/deg)	Torsional (in-lb/deg)	Lateral (lb/inch)	Thrust area (in ²)	Weight (lb)
1	112	113	719	83	29000	9116	42	16
2	126	127	719	83	29000	9116	42	16
3	461	462	945	177	23000	19473	67.2	23
4	467	468	719	83	29000	9116	42	16

You can navigate through the different itemized lists by clicking on the two blue-colored arrows on the toolbar.

Element types

How to input an Element

To input an element (between two nodes) in a new row, either right click in the Type field of the row or left-click on Type (in the Header row) to display the Element types dialog (or press Ctrl+Shift+T). Select the element you want from this dialog to insert into the model.



Notice the different types of elements available. Any of these can be input for “Type.” Some of these are strictly not elements such as From, Location, Comment and Hydrostatic test load but need to be input from here anyway. All of these have keyboard shortcuts. You do not have to necessarily open this dialog to input any of these. You can type the first letter or the first few letters (example: Be for bend, M for miter, V for valve, etc.) to input an element type.

Shortcut keys for Element types

b (Tab)	Bend
ben	Bend (no Tab required, and typing “ben” moves the cursor over to the DX field.)
bea	Beam
bel	Bellows
ba	Ball joint
c	Cut pipe (Cold spring)
e	Elastic element
f	From
h	Hinge joint
j	Jacketed pipe
jb	Jacketed Bend
l (L)	Location
m	Miter Bend
P	Pipe (you don’t have to type this because a Pipe is the default element type)
r (Tab)	Reducer
re	Reducer
ri	Rigid element
s	Slip joint
v	Valve
t	Tierod

Once input, each element name appears under the Type field. The only exception is the Pipe element which is indicated by a blank (see the first annotated figure in this section). The field under Type is blank on the row that contains Node 30 signifying a pipe.

How to change the Element type

If you wanted to change a Valve element to an Elastic element, then you have to do the following: Move the highlight bar to the row that contains the Valve (or click once on Valve). Delete the word “Valve” (Backspace) and press “E”, confirm the change;

or

Assuming the highlight is on the Valve row, click on Type (in the header row) to display the Element types dialog and then select Elastic element from it. The Valve changes to an Elastic element after you confirm the change.

How to change Element properties

From the keyboard: Press Ctrl+T (from anywhere on the row) to edit the element. Note: A pipe has no editable property, and Ctrl+T does nothing.

Mouse: Right clicking on the element gives you applicable choices. For example, for a Bend, you are shown Edit Bend and List Bends as the choices.

Notes:

1. A pipe element is the default element type. So, leave the element type field blank for that row.
2. An existing element type (say, a bend) cannot be changed to a Location type.
3. To change the element type, you can erase (use Backspace or Delete key) the existing element and specify a new one (using an accelerator key), or use the mouse to specify

Layout Window

another element by clicking on Type in the header row. Or, use Ctrl+X to delete the entire row.

Data types

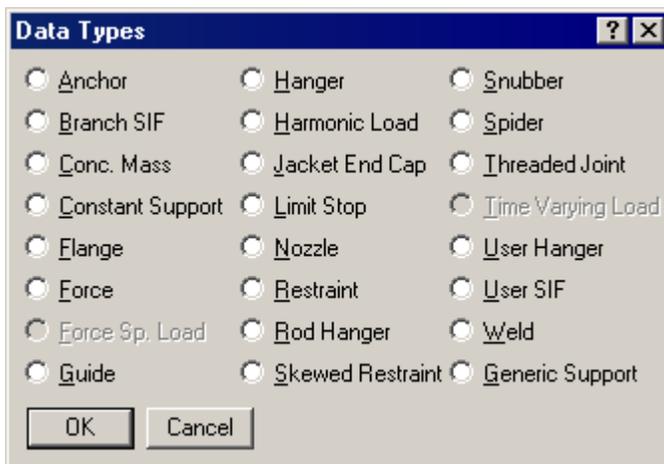
How to input a Data item

When you want to input a support, a restraint, a flange, an SIF, etc., at a node, use the mouse or the keyboard (several key accelerators are available to make your job easy) to open the Data types dialog and select an item from it.

Keyboard: Press Ctrl+Shift+D to open the Data types dialog to select an item, or, type one of the following keys (under Shortcut keys) that will automatically enter the respective Data type (press Ctrl D to position the cursor in this field):

Mouse: To open the Data types dialog,

- Left click on Data (in the header row), or
- Right click in the Data column, or
- Select Misc menu >> Data Types.



Shortcut keys for Data types

a	Anchor (rigid)
br	Branch SIF (Welding tee, Branch connection, etc.)
c (Enter)	Concentrated Mass
conc	Concentrated Mass
cons	Constant Support
f (Enter)	Flange
fl	Flange
fo	Force
gu	Guide
ge	Generic Support
h (Enter)	Hanger (to be designed)
han	Hanger (to be designed)
har	Harmonic load
j	Jacket end cap
l (L)	Limit stop

Layout Window

n	Nozzle
r (Enter)	Restraint (2-way), see also X, Y, Z below
ro	Rod hanger
s (Enter)	Skewed restraint
sk	Skewed restraint
sn	Snubber
sp	Spider
th	Threaded joint
t (Enter)	Time varying load
ti	Time varying load
u (Enter)	User hanger
user s	User SIF
w	Weld
x	X restraint (2-way)
y	Y restraint (2-way)
z	Z restraint (2-way)

Notice the different types available. For example, press “a” in the Data field to input an Anchor (you do not have to go through the long-winded mouse method of clicking on Data in the header row to display the Data types dialog, and then selecting Anchor from it).

How to change the Data type

Assume that you wanted to change an Anchor to a Nozzle. Before you change an existing Data type (Anchor), you have to delete it. A deletion cannot be undone presently.

To delete, click on Anchor, delete the word “Anchor” (backspace) and then type “n”, or right click on the Anchor, and select “Delete Anchor.”

Alternately, with the cursor placed on the row for which you need to change the Anchor, click on Data on the header row, and pick Nozzle and confirm the change.

How to edit the Data item's properties

To edit properties for anchors, flanges, etc.

Keyboard: Press Ctrl+D (from anywhere on the row) to edit the item. Note: A Threaded Joint has no editable property, and Ctrl D does nothing.

Mouse: Right clicking on the item gives you applicable choices. Example: For an Anchor, you are shown Edit Anchor.

Notes:

1. You cannot specify a data item for a Bend, a Miter Bend or a Jacketed Bend, as the node number is a tangent intersection point, and does not exist physically on the element. Instead, create an intermediate node for the Bend or Jacketed Bend and then specify a data item at the new node using Location type. You cannot create an intermediate node for a miter bend.
- 2a. You can delete an existing data item at anytime:
 - Use the Delete or the Backspace key to erase the item and press Enter.
 - Right click on the item and select Delete Item.

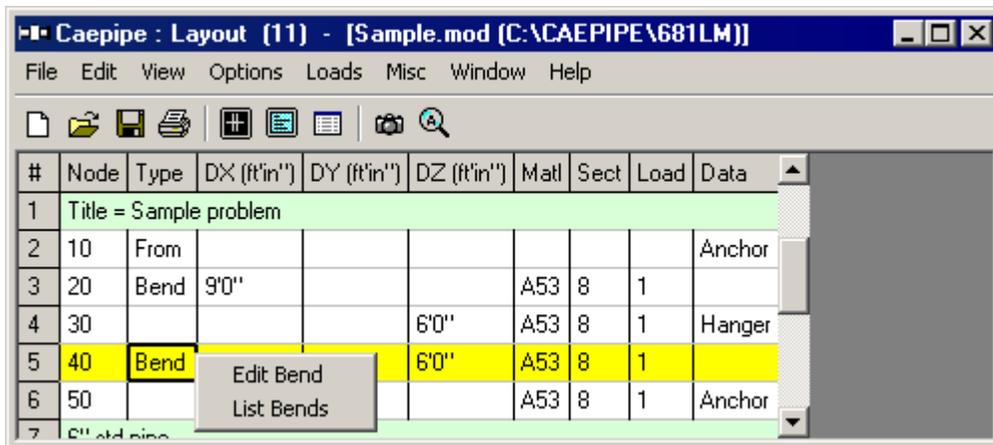
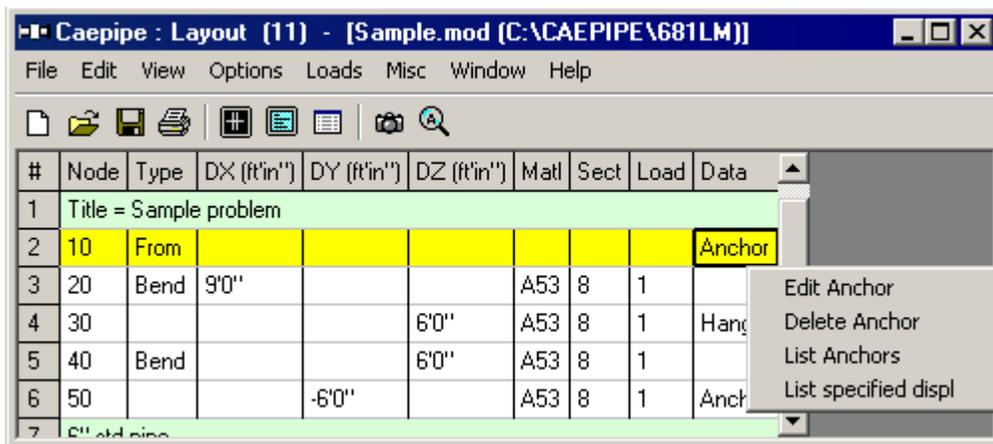
Layout Window

2b. Alternately, to change the data type, you can erase (use Backspace or Delete key) the existing item and type a new one (accelerator), or use the mouse to specify another data item by clicking on Data in the header row.

A deletion (whether a row or a field entry) cannot be undone presently.

Context Menus

Context menus (right click menus) can be found everywhere in CAEPIPE. For example, when you right click on an Anchor, or a Bend, you are shown the following context menus. You can perform any listed action that more than often is a shortcut (e.g., Delete Anchor or List Anchors).

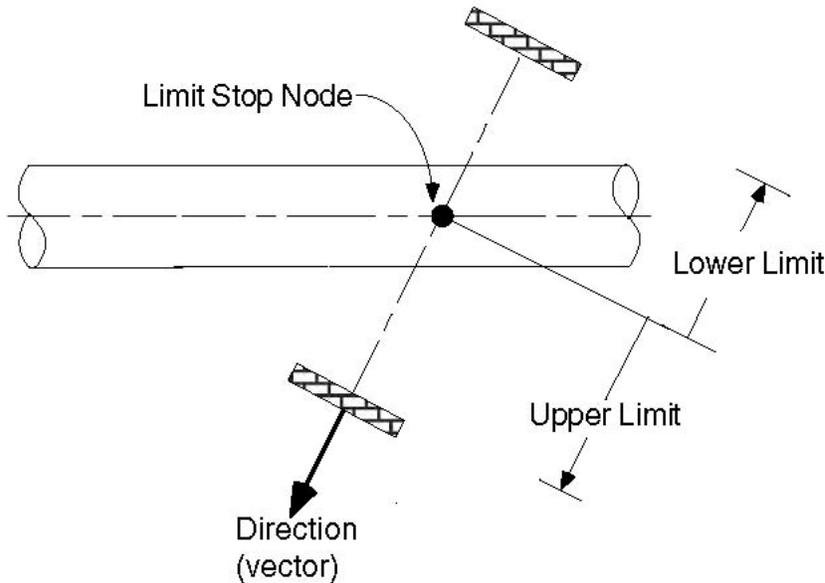


Right clicking on DX/DY/DZ shows you, for example, Insert, Delete (rows), Split (an element) and (specify) Slope for an element. Right click in the Matl, Sect or Load column to see the respective context menu.

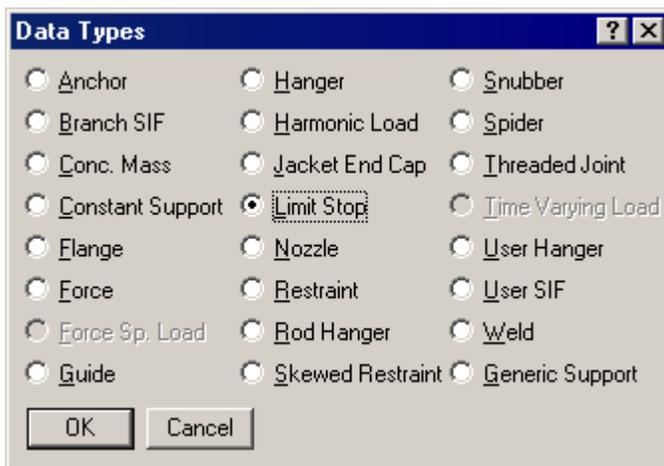
Limit Stop

In CAEPIPE, a limit stop is capable of modeling several types of physical supports including a guide, an anchor, a resting support, a two-way rigid restraint and a rod hanger. Using a combination of upper and lower gaps (limits), friction coefficient, support stiffness and a direction for its axis, you can model the above mentioned physical support types.

A limit stop prevents a node from moving beyond a certain distance (called a gap or a limit) in a certain direction. The node can move freely within the gap. After the gap closes, a limit stop acts as a rigid or flexible restraint (depending on your input for stiffness) resisting further movement of the node in the specified direction. If friction is specified, after the limit is reached, the friction force will oppose movement in the plane normal to the limit stop direction.



A limit stop is input by typing "1(L)" in the Data column or selecting "Limit Stop" from the Data Types dialog.



Limit Stop

The Limit Stop dialog is shown.

Limit stop at node 30

Upper limit (inch)

Lower limit (inch)

Direction

X comp Y comp Z comp

Friction coefficient

Stiffness (lb/inch)

Connected to

Limits

Also called gaps, these limits, upper and lower, are the gaps present on either side of the node. The gap in front of the node in the direction of the vector is called the upper limit, and the gap to the rear of the node is called the lower limit. The gap is measured from the undeflected position of the node.

Typically, the upper limit is positive and the lower limit negative. In some situations, it is possible to have a positive lower limit or a negative upper limit, which is the same as forcefully displacing the node in that direction by the gap specified.

The algebraic value of the upper limit must be greater than the lower limit. For example, upper limit = $-0.125''$, lower limit = $-0.25''$, or upper limit = $0.25''$, lower limit = $-0.25''$.

If a particular limit does not exist (that is, a node can move freely on that side of the node), then that limit should be left blank (as in the case of a resting support, the upper limit should be blank assuming Y-vertical and Y comp = 1, as in the figure shown).

If there is no gap at all, then the corresponding limit should be explicitly input as zero. When zero is entered, the limit stop acts as a one-way restraint in that direction.

Direction

The direction in which the limit stop is oriented must be specified in terms of its global X, Y and Z components. Or use one of the preset buttons to orient the limit stop axis (see above image):

1. Vertical: To set the axis in the (global) vertical direction
2. Axial: To set the axis along the local-x direction (pipe axis)
3. Shear y: To set the axis in the local-y direction
4. Shear z: To set the axis in the local-z direction

Limit Stop

If you have connected the limit stop node to another node in the same piping model, then unless the connected node is coincident with the limit stop node, the direction must not be input. It is calculated from the locations of the connected node and the limit stop node, and its *direction oriented from the connected node to the limit stop node*.

Friction coefficient

If friction coefficient is specified, a friction force will oppose the movement in the plane normal to the limit stop direction when the gap is closed. This friction force is displayed in results under Limit stop support loads.

If you had several limit stops with friction coefficients specified, and you wanted to change all of those friction coefficients to the same value, use the Change command under the Edit menu.

Stiffness

The default is set to Rigid stiffness. Other values may be input by estimating the stiffness of the support. (e.g., for a rod, stiffness = AE/L).

where

A = Cross-sectional area of rod

E = Young's modulus

L = Length of rod

Connected to Node

You can connect a limit stop node to another node in the same piping model. During gap and friction calculations, the relative displacements of the limit stop node are calculated with respect to the connected node. If you connect the limit stop node to an external fixed point (Ground point), leave the "Connected to Node" blank. See "Direction" above for information about how the direction is calculated depending on whether the connected node is coincident or not with the limit stop node.

Limit stop at node 425

Upper limit (inch)

Lower limit (inch)

Direction

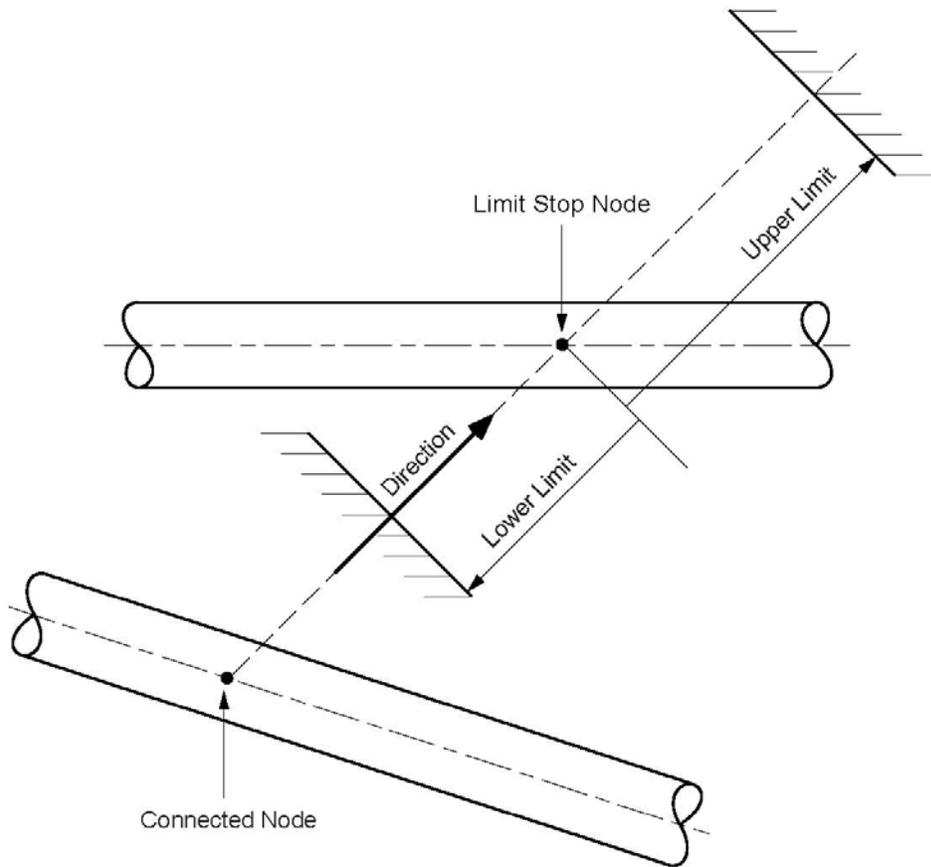
X comp	Y comp	Z comp
<input type="text"/>	<input type="text" value="1.000"/>	<input type="text"/>

Friction coefficient

Stiffness (lb/inch)

Connected to

Limit Stop



Solution procedure

Limit stops require a nonlinear iterative solution. If you specify a friction coefficient, the following procedure is used for convergence: If the lower or upper limit is reached, the corresponding reaction force is calculated. The maximum friction force is the product of friction coefficient and the reaction force. The solution converges when the displacement varies by less than 1% between successive iterations.

Limit stops are included in dynamic analysis. The status of the limit stops at arrived at upon completion of all iterative calculations for the first operating case (W+P1+T1) is used during dynamic analysis. If the limit is reached at the end of iterations for the first operating case, then the limit stop is treated as a rigid two-way restraint in the direction of the limit stop during dynamic analysis.

Example 1: Vertical 1-way restraint

Assume that you have a vertical 1-way support with the following data: Upper limit = None, Lower limit = 0, Friction coefficient = 0.3, Direction vector of the limit stop is vertical along +Y axis.

Model the pipe up to the Limit stop node 30. At node 30, type "1(L)" in the Data column. The limit stop dialog will be shown.

Limit Stop

Limit stop at node 30
✕

Upper limit (inch)

Lower limit (inch)

Direction

X comp	Y comp	Z comp
<input type="text"/>	<input type="text" value="1.000"/>	<input type="text"/>

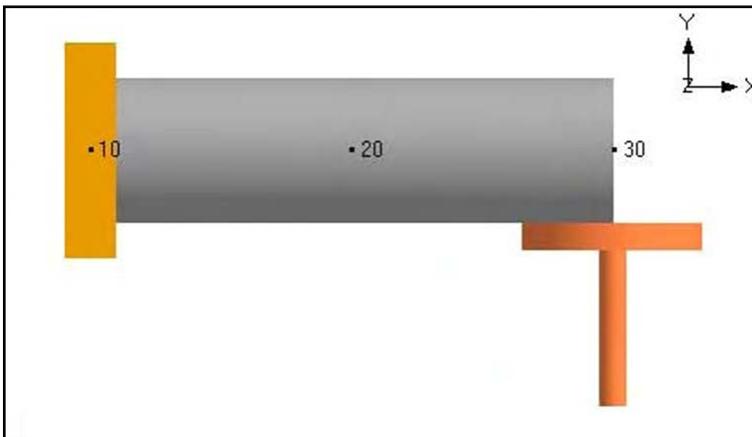
Friction coefficient

Stiffness (lb/inch)

Connected to

In the limit stop dialog, press “Vertical” button. The data is automatically entered [0.000 for Lower limit, No Upper limit (blank), 1.000 for Y comp (vertical)]. Enter 0.3 for friction coefficient (this acts in the X-Z plane which is perpendicular to the direction of the limit stop).

#	Node	Type	DX (ft/in)	DY (ft/in)	DZ (ft/in)	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20		1'0"			1	6	1	
4	30		1'0"			1	6	1	Limit stop



Example 2: Pipe Slide/Shoe Assembly

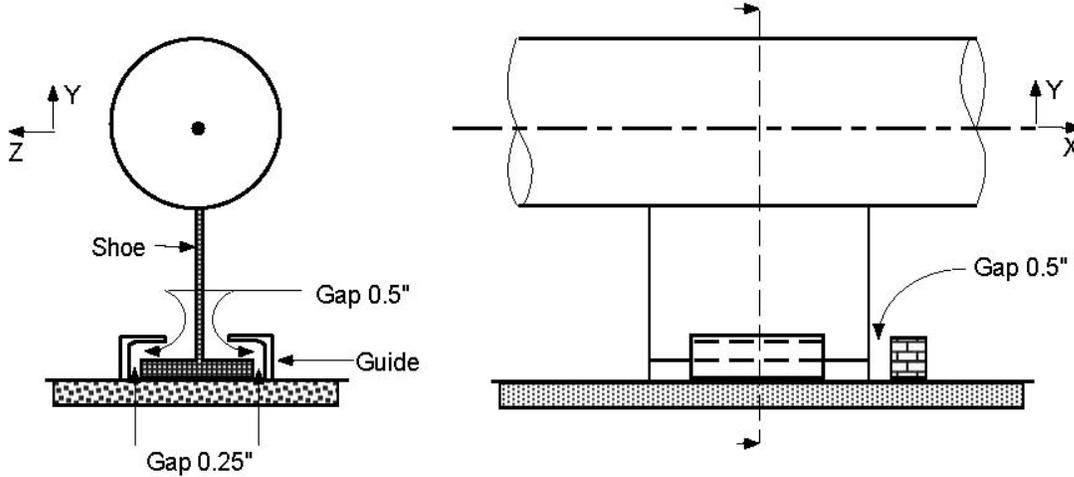


Figure not to scale

The assembly is modeled using three limit stops, one in each direction.

- ▶ When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- ▶ Press Tab in the Node column which puts the node number 20 automatically. Type 5' for DX, enter material, section and load names, press Enter.
- ▶ Press Tab in the Node column which puts the node number 30 automatically. Type 5' for DX, Tab to Data column and type "l(L)" to open limit stop dialog. Input line stop with gap (0.5") along X axis (Notice that this is a one-way restraint, there is only one stop block along +X). Type Upper limit 0.5", leave lower limit blank, Direction as (X comp = 1, Y comp = 0, Z comp = 0), Friction coefficient=0.3, press Enter.

Limit Stop

Limit stop at node 30

Upper limit (inch)

Lower limit (inch)

Direction

X comp	Y comp	Z comp
<input type="text" value="1.000"/>	<input type="text"/>	<input type="text"/>

Friction coefficient

Stiffness (lb/inch)

Connected to

- ▶ Create a limit stop in the Y direction. Type 30 for Node, press Tab to move to Type field, type "L" for Location, choose Limit stop from the Data Types dialog, in the Limit stop dialog, type Upper limit = 0.5", Lower limit = 0.0, Direction (X comp = 0, Y comp = 1, Z comp = 0), Friction coefficient=0.3, press Enter.

Limit stop at node 30

Upper limit (inch)

Lower limit (inch)

Direction

X comp	Y comp	Z comp
<input type="text"/>	<input type="text" value="1.000"/>	<input type="text"/>

Friction coefficient

Stiffness (lb/inch)

Connected to

- ▶ Create a limit stop in the Z direction. Type 30 for Node, press Tab to move to Type field, type "L" for Location, choose Limit stop from the Data Types dialog, in the Limit stop dialog, type Upper limit = 0.25", Lower limit = -0.25", Direction (X comp = 0, Y comp = 0, Z comp = 1), Friction coefficient = 0.3, press Enter.

Limit Stop

Limit stop at node 30
✕

Upper limit (inch)

Lower limit (inch)

Direction

X comp	Y comp	Z comp
<input style="width: 60px;" type="text"/>	<input style="width: 60px;" type="text"/>	<input style="width: 60px;" type="text" value="1.000"/>

Friction coefficient

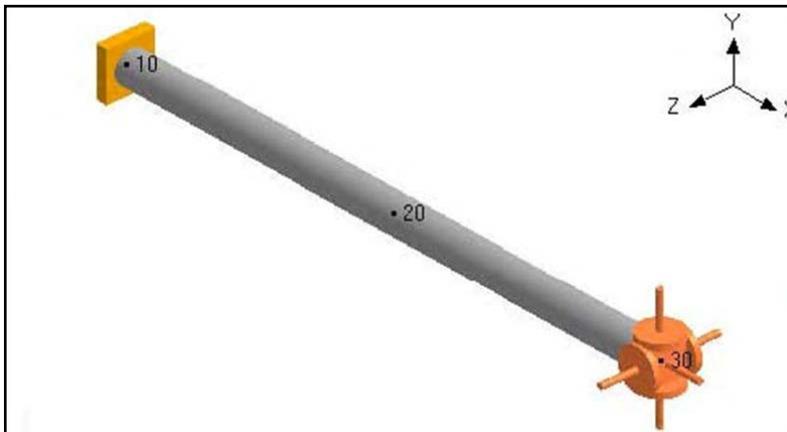
Stiffness (lb/inch)

Connected to

The Layout window is shown below.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Pipe slide/shoe assembly								
2	10	From							Anchor
3	20		5'0"			1	6	1	
4	30		5'0"			1	6	1	Limit stop
5	30	Location							Limit stop
6	30	Location							Limit stop

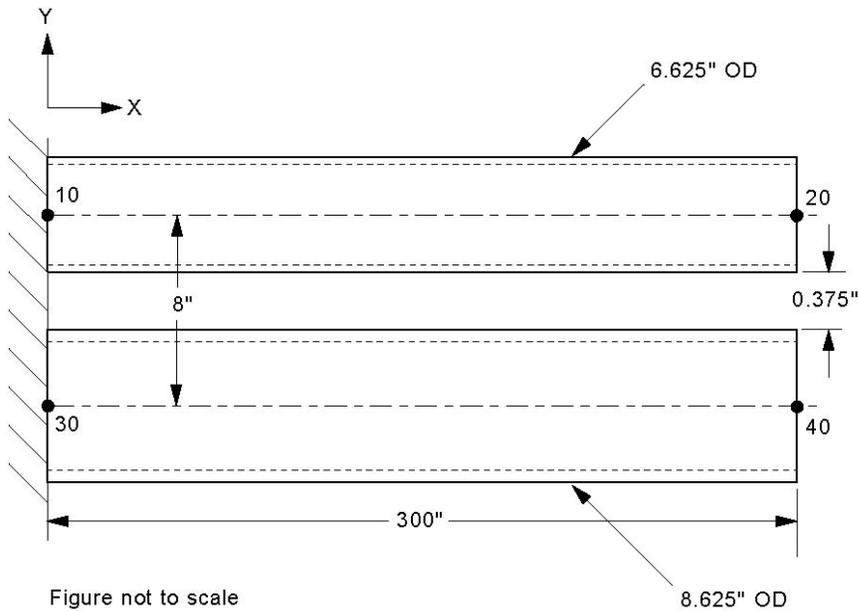
The Graphics window is shown below.



Limit Stop

Example 3: Limit Stop Connected to Another Node

As shown in the figure below, two 300" long cantilever pipes are connected with an 8" separation between center lines.



The gap between the bottom of the top pipe and the top of the bottom pipe is

$$8'' - (6.625'' + 8.625'') / 2 = 0.375''.$$

If pipes were free to deflect downward due to deadweight, the top pipe will deflect 1.908" and the bottom pipe will deflect 1.116" at the free ends. The relative deflection between them will be $1.908'' - 1.116'' = 0.792''$. This however is not possible because when the relative deflection exceeds 0.375" the pipes will touch. This situation can be modeled using a limit stop connecting the free ends of the pipes. In this case the top pipe deflects 1.608", i.e., less than the 1.908" free deflection because it is resisted by the bottom pipe. The bottom pipe deflects 1.233", i.e., more than the 1.116" free deflection because additional load is imposed on it by the top pipe when they touch. The difference between the deflections is $1.608 - 1.233 = 0.375''$ as expected.

- ▶ When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- ▶ Press Tab in the Node column which puts the node number 20 automatically. Type 300" for DX, enter material, 6" section and load names, press Enter.
- ▶ For the bottom pipe, start with node 30 of Type "From" at DY = -8" and make it an anchor. On the next row enter node 40 with DX = 300" and 8" section. Press Enter to go to the next row.
- ▶ Enter the limit stop at node 20. Type 20 for Node, press Tab to move to Type field, type "L" for Location, choose Limit stop from the Data Types dialog, in the Limit stop dialog, leave the Upper limit blank and input Lower limit = -0.375". Input the Connecting Node as 40. Since this limit stop connects two nodes, the direction should be left blank. The direction is implicitly from node 40 to node 20.

Limit Stop

Limit stop at node 20
✕

Upper limit (inch)

Lower limit (inch)

Direction

X comp	Y comp	Z comp
<input style="width: 50px;" type="text"/>	<input style="width: 50px;" type="text"/>	<input style="width: 50px;" type="text"/>

Friction coefficient

Stiffness (lb/inch)

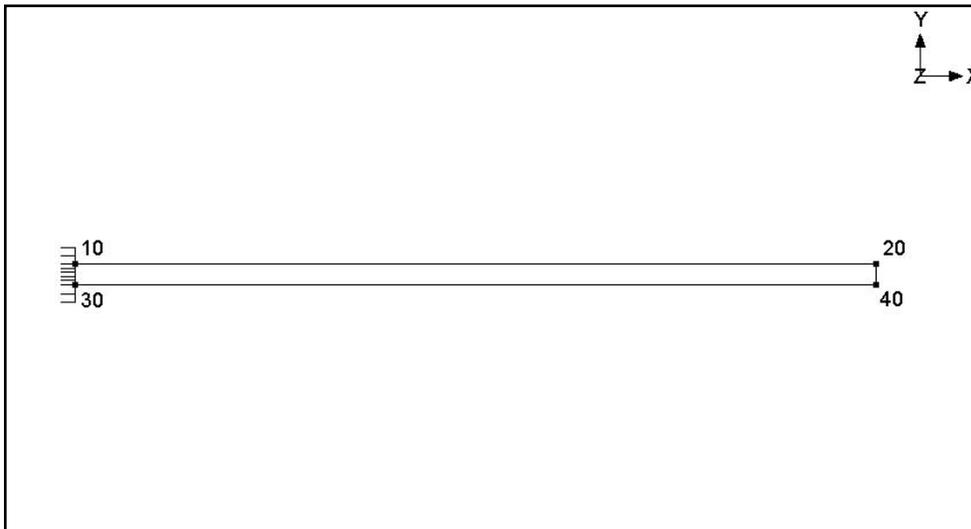
Connected to

Alternatively the limit stop could be specified at node 40 connected to node 20. The direction now would be from node 20 to 40. The limits would still be no upper limit and lower limit = $-0.375''$ since the direction is now reversed compared to the previous case. Both these cases will give identical results.

The Layout window is shown below.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Limit stop connected to another node								
2	10	From							Anchor
3	20		300			1	6	1	
4	30	From		-8					Anchor
5	40		300			1	8	1	
6	20	Location							Limit stop

The Graphics is shown below.



Load

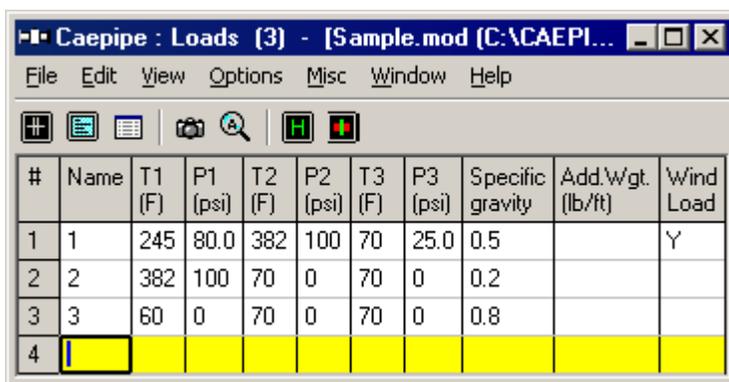
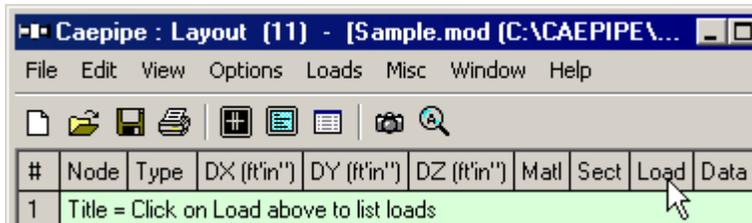
Loads on a piping system can be many and varied along its routing. Different piping segments may experience different pressures and temperatures depending on process requirements, different loads (snow, wind, etc.) depending on their physical locations and carry different states of a fluid between different points in a piping system. CAEPIPE offers a flexible method to input as many loads as required for as many segments or elements as needed with the least effort.

So, Load allows you to apply a temperature and pressure, specify weight of the fluid and add additional weight (e.g., due to snow load) on each element (if required) or for a range of elements in the model. Also the wind load can be turned “on” or “off” for each element (if required) or for a range of elements in the model.

After specifying the requested information here, including a name, use it under the Load column on the Layout window to associate the load information with an element.

Each load allows up to 10 operating conditions for Temperature and Pressure depending on the “Number of Thermal Loads” under Options > Analysis > Temperature. This Load is not to be confused with Load cases [which are combinations of load(s)] found under the Loads menu in the Layout window. Load cases are analysis cases (Sustained case, Thermal case, Operating case, Static Seismic case, etc.) for which CAEPIPE computes a set of results.

To define a new load, click on Load in the Header row in the Layout window (or select Loads under the Misc menu, hotkey: Ctrl+Shift+L). This opens a List window that displays currently defined loads.



Either you can start typing the load data directly here into the fields or double click on an empty row to enter data through a dialog.

Depending on the number of thermal loads specified (under Options > Analysis > Temperature), up to 10 temperature/pressure load sets (T1/P1, T2/P2, T3/P3,...,T10/P10) are available which correspond with the 10 operating load cases for each element.

Load

#	Name	T1 (F)	P1 (psi)	T2 (F)	P2 (psi)	T3 (F)	P3 (psi)	T4 (F)	P4 (psi)	T5 (F)	P5 (psi)	T6 (F)	P6 (psi)	T7 (F)	P7 (psi)	T8 (F)	P8 (psi)	T9 (F)	P9 (psi)	T10 (F)	P10 (psi)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load
1	1	150	200	100	350	-100	-600	70	0	70	0	70	0	70	0	70	0	70	0	70	0	0.8	253	Y
2																								

Up to 10 specified thermal displacements can be entered for Anchor and Nozzle data types.

Load	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
T1		0.5				
T2				0.1	0.1	0.1
T3						
T4						
T5						
T6						
T7						
T8						
T9						
T10						
Seismic	0.2		0.2			
Settlement		-0.25				

Displacements in LCS

Load

Temperature	Pressure
150 (F)	200 (psi)
100 (F)	350 (psi)
-100 (F)	-600 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)
70 (F)	0 (psi)

Spec. gravity: 0.8 Add. weight: 253 (lb/ft)

OK Cancel Specific gravity is with respect to water

Load Name

Type an alphanumeric name (up to five characters) in this field. The name can be changed later.

Temperatures

Type up to 10 operating temperatures. The maximum of the 10 temperatures is the Design temperature, used to look up the corresponding allowable stress (S_t) for the material.

The other quantities looked up using these temperatures are the thermal expansion coefficients (α) and the temperature-dependent moduli.

Make sure to select number of thermal loads equal to two, three or 10 under Options > Analysis > Temperature, when you have more than one set of temperature and pressure.

Pressures

Type up to 10 operating pressures that correspond to the 10 operating temperatures above. The maximum of the 10 pressures is the Design pressure, used to calculate the pressure stress term [$PD/4t$ or $Pd^2/(D^2 - d^2)$], specified under Options > Analysis > Pressure.

Specify gauge pressures for Pressures input. Negative (external) pressures may be specified, too. But, the longitudinal pressure stress term ($pD/4t$) will still be positive according to the piping codes. Internal pressure will expand the pipe cross-section radially outward while external (negative) pressure will contract the pipe cross-section radially inward due to Poisson's effect.

Load

Specific Gravity

Specific gravity is the ratio of the density of a fluid to the density of a reference substance (in this case, water). Type in the specific gravity of content inside the pipe. This input is used to calculate the weight of the contents, which is added to the weight of the pipe. Specific gravity is with respect to water.

Additional weight

The value you enter here is taken as weight per unit length of the element and this total additional weight is added to the weight of the pipe. (Total additional weight = Length x Additional weight per unit length).

For example, this could be used to apply the weight of the snow on the pipe.

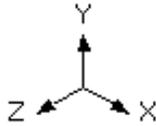
Wind load

Type Y(es) or N(o) to apply or not apply the wind load for this element. When you press Y(es), the wind load (entered as a separate load under Loads menu) is applied to this element. The default is Y(es).

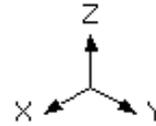
For example, this is useful when you have a part of a piping system exposed to wind with the remaining part inside a building. In such a case, you should define two Loads, all data the same except that one has Wind load and the other does not. The Load with the wind load is applied to those elements that are affected by wind.

Local Coordinate System

CAEPIPE allows you to create your physical piping system in a mathematically equivalent 3D Cartesian coordinate space with a global origin, which is the point of intersection of three planes orthogonal to each other, with three axes commonly denoted X, Y and Z (with either of the latter two vertical).



Y Vertical



Z Vertical

Once you begin creating your system from a given point (usually the global origin), you route your piping system one element at a time until you get to the end of the line(s). An element's orientation could be different from another element's, thereby necessitating an element's own "elemental coordinate system," which is commonly referred to as the Local Coordinate System (LCS), provided for the purpose of understanding the local forces and moments on each element. This system can be turned on (graphically) through the View menu > Show LCS command while you are viewing "Element Forces in Local Coordinates" in the Results window.

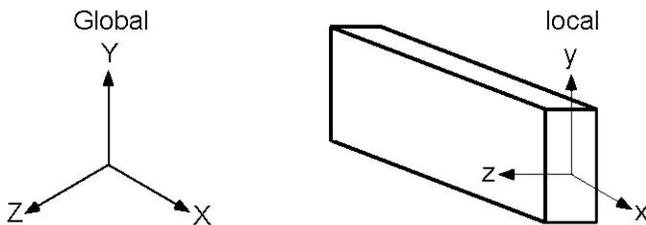
For a straight element (such as a pipe or a beam), the "local x" axis is along the element, from the "From" node to the "To" node. For a node location such as a guide, the local axes are based on the previous connected element. If such an element does not exist, the following element is used. The local y- and z-axes are calculated differently depending on whether the vertical direction is Y or Z and also depending on whether the element is in the vertical direction.

The local coordinate system may be displayed graphically (for beams and guides in the input processor and for all the elements in the output processor) by selecting the "Show LCS (Local coordinate system)" command from the View menu.

In CAEPIPE, the local coordinate system is indicated by lower case x, y and z letters. The global coordinate system is indicated by upper case X, Y and Z letters.

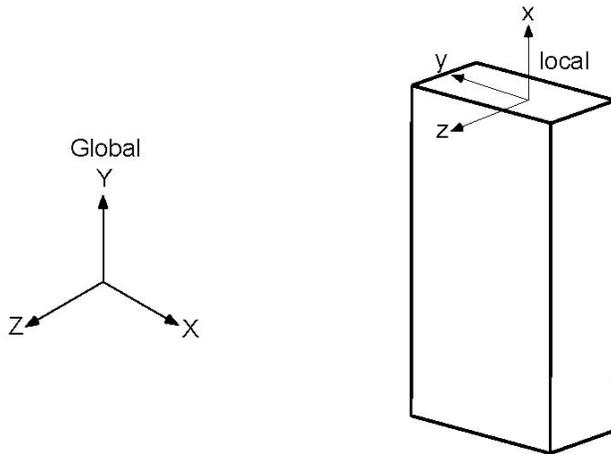
Global vertical axis is Y

Element is not Vertical



The local y-axis of the element lies in the local x - global Y plane (i.e., vertical plane) and is in the same positive direction as the global Y axis. The local z-axis is the cross product of the local x and y-axes.

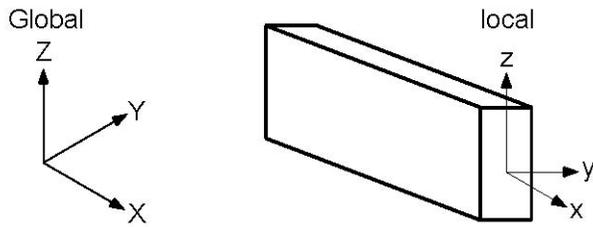
Element is Vertical



The local z-axis of the element is in the global Z direction. The local y-axis is in the global -X direction.

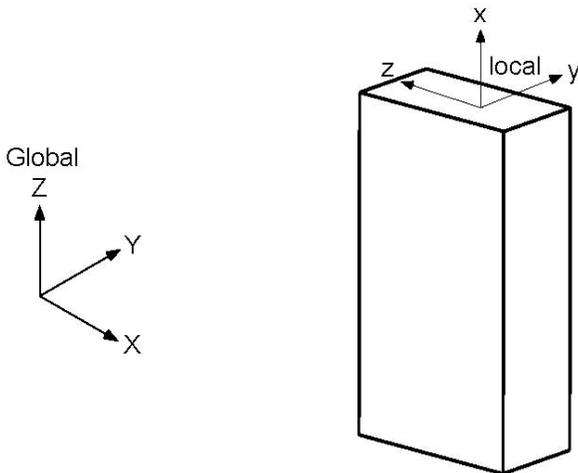
Global vertical axis is Z

Element is not Vertical



The local z-axis of the element lies in the local x - global Z plane (i.e., vertical plane) and is in the same positive direction as the global Z-axis. The local y-axis is the cross product of the local z and x-axes.

Element is Vertical



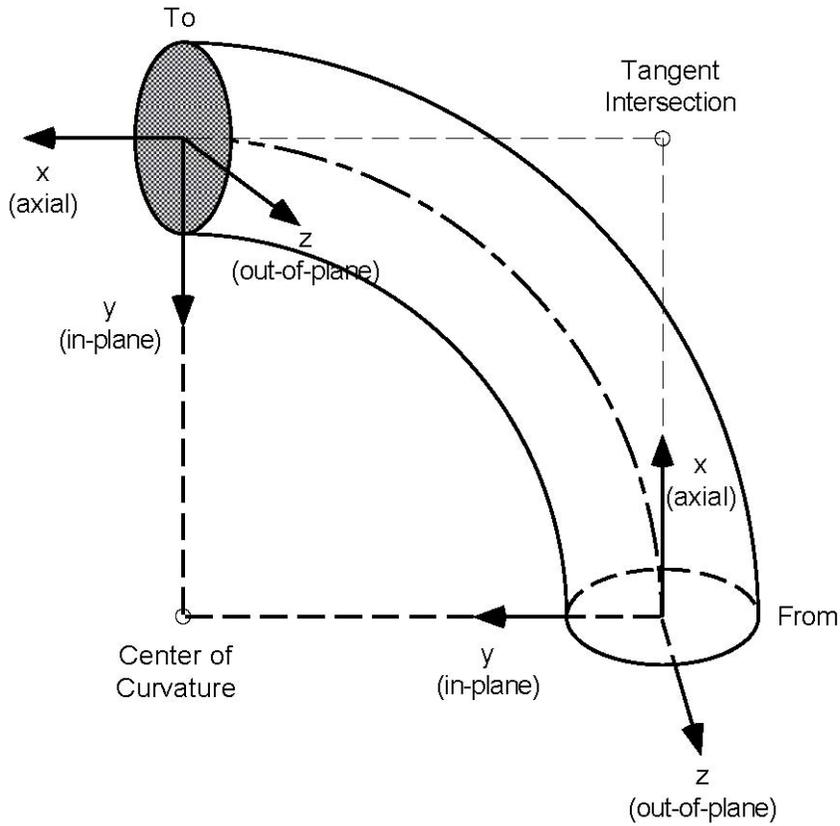
The local y-axis of the element is in the global Y direction. The local z-axis is in the global -X direction.

Local Coordinate System

Local Coordinate System for a Bend

For a bend, at the “From” node, the local x axis is along the tangent from the “From” node to the tangent intersection point. The local y-axis is along the radius and points to the center of curvature. The local z-axis is cross product of the local x- and y-axes.

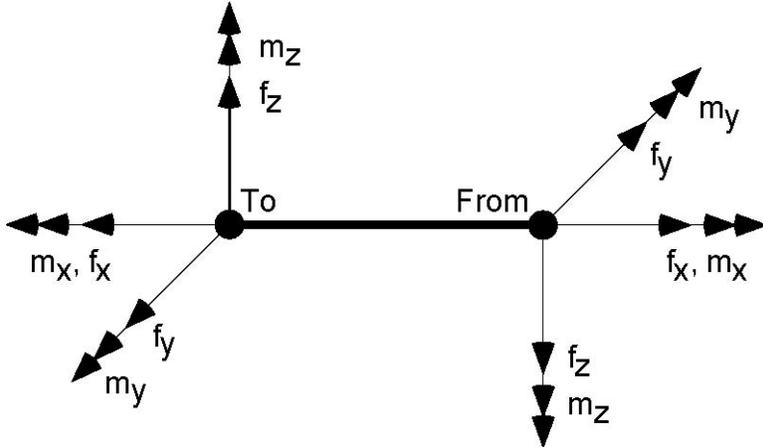
Similarly, at the “To” node, the local x-axis is along the tangent line from the tangent intersection point to the “To” node. The local y-axis is along the radius and points to the center of curvature. The local z-axis is the cross product of the local x- and y-axes.



Local Coordinate System

Element Forces and Moments

The sign conventions for the element forces and moments in the local coordinate system follow strength of materials conventions, i.e., forces and moments are positive in the positive local axes directions at the “To” node and they are negative in the positive local axes directions at the “From” node.



Positive sign conventions for local forces and moments are shown above at the “From” and “To” nodes of an element. Note that positive directions at the “From” node are reversed compared to the positive directions at the “To” node.

Location

Several times, you may need to put in more than one data item at a node, e.g., a hanger and a Branch SIF to designate the type of tee. At those times, use Location (press “L” in the Type field or pick Location from the Element types dialog) to input more than one data item at a node. For example, you may want to input a force at an existing hanger (see Example 1 below) or three limit stops (in different directions) at one node. See Example 2 under Limit stop.

Another use for Location is when you want to input a data item at an internally generated node. Nodes are internally generated by CAEPIPE for bends (A, B nodes, e.g., 20A) and Jacketed pipes/bends (J, C, D nodes, e.g., 10J).

By design, each row in the Layout window allows only one data item to be inserted under the Data field. Additional data items can be input only through Location.

Ensure that the node you use for Location has already been defined on an earlier row: defined earlier as an intermediate node for a bend or is an internally generated node. In other words, you cannot use Location on an undefined node.

Example 1: Pipe and Shoe assembly

See example 2 under the topic Limit Stop on page 306.

Example 2: Data at internally generated nodes

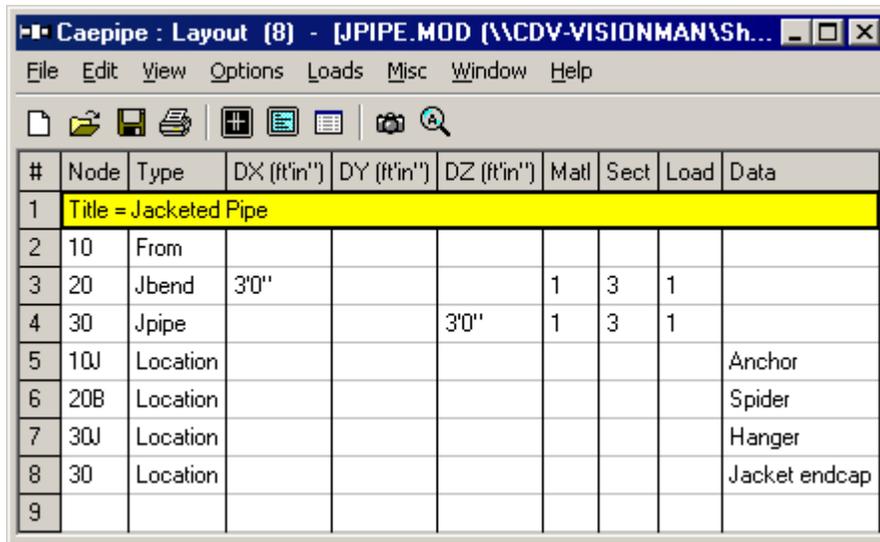
Let us take the example given under Jacketed pipe/bend. For a Jacketed pipe, CAEPIPE internally generates the J node on the jacket. For a Jacketed Bend, CAEPIPE generates the C and D nodes on the jacket.

There are four data items in this example:

1. Node 10J is the starting Jacket node which is anchored (node 10 is separately anchored, since it is the node on the core pipe).
2. There is a hanger at node 30J since the hanger is connected to the Jacket.
3. There is a spacer (spider) at the far end of the bend, node 20B (which is on the core pipe). Remember that the bend has a jacket on the outside.
4. Node 30 has a jacket end cap

The following sample Layout window contains all these data as Location data.

Location



The screenshot shows the 'Caepipe : Layout (8) - [JPIPE.MOD (\CDV-VISIONMAN\Sh...)]' window. The menu bar includes File, Edit, View, Options, Loads, Misc, Window, and Help. The toolbar contains icons for file operations and navigation. The main data table is as follows:

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Jacketed Pipe								
2	10	From							
3	20	Jbend	3'0"			1	3	1	
4	30	Jpipe			3'0"	1	3	1	
5	10J	Location							Anchor
6	20B	Location							Spider
7	30J	Location							Hanger
8	30	Location							Jacket endcap
9									

Example 3: Bend supported by a Hanger

See Example 6 in the Bend section for an example of using Location at an intermediate node on p. 168.

Material

For a project, a piping material engineer usually produces different material lists for different process and utility systems based on process design conditions, list of components, fluid type (corrosivity, viscosity), end conditions and temperature, pressure and size ranges.

As a result, a piping system will have its materials list, which will specify the materials a stress analyst will have to define inside CAEPIPE before analysis. CAEPIPE is so flexible that it allows each element (such as a pipe, beam, elbow, valve, jacket, bellows, etc.) to have its own material definition.

Once you define and name a material type, you type in that name on the Layout window under the column “Matl” to specify the material for an element. A material may be metallic or not (FRP). You need to obtain properties (density, Poisson’s ratio, Young’s modulus, mean coefficient of thermal expansion and allowable stress as a function of temperature) for a new material not found in the supplied libraries. You will need properties for at least two temperatures (reference and design) for each material you define. Subsequently, the temperatures you specify for an element that uses this material must fall within this temperature range.

The material name you specify on the layout applies to the piping element on that row. For jacketed piping, you must specify two materials - one for the core pipe (on the Layout window), and the other for the jacket pipe in its own dialog. The material you specify for a bend, a jacketed bend, and for a miter bend applies only to that specific element on that input row.

There are two ways in which you can define materials:

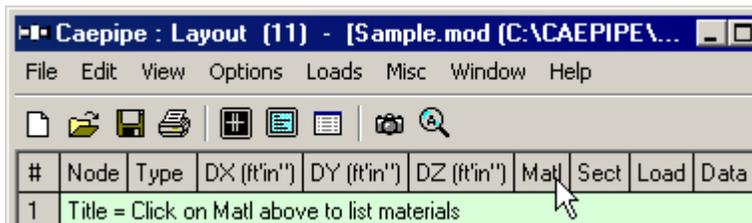
1. By defining a material inside the CAEPIPE model, or
2. By picking a material from an existing material library.

For the sake of convenience, we suggest that you create a separate material library for your piping system project so that you can share it with your team members.

Below, you will see how to create a material inside a model and how to create a new or modify an existing material library.

Define a Material inside a CAEPIPE model

From the Layout window, click on “Matl” on the header row (or select Materials from the Misc menu, Ctrl+Shift+M).



Material

The List window for materials is shown.

The screenshot shows a window titled "Caepipe : Materials (5) - [Sample.mod (C:\CAEPIPE\681LM)]". The window contains a table with the following columns: #, Name, Description, Type, Density (lb/in³), Nu, Joint factor, Yield (psi), #, Temp (F), E (psi), Alpha (in/in/F), and Allowable (psi). The table lists materials 1 through 18, with material 3 highlighted in yellow.

#	Name	Description	Type	Density (lb/in ³)	Nu	Joint factor	Yield (psi)	#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	A53	A53 Grade B	CS	0.283	0.3	1.00	35000	1	-20	31.0E+6	6.25E-6	17100
2	106	A106 Grade C	CS	0.283	0.3	1.00	40000	2	70	30.6E+6	6.40E-6	17100
3	355	A335 Grade P22 (2 1/4Cr-1Mo)	CS	0.283	0.3	1.00		3	200	29.8E+6	6.70E-6	17100
4	B42	B42 Annealed	CA	0.163	0.3	1.00	9000	4	300	29.4E+6	6.90E-6	16600
5	TTN	Titanium B337 Grade 12	TI	0.163	0.3	1.00		5	400	28.8E+6	7.10E-6	16600
6								6	500	28.3E+6	7.30E-6	16600
								7	600	27.7E+6	7.40E-6	16600
								8	650	27.4E+6	7.50E-6	16600
								9	700	27.1E+6	7.60E-6	16600
								10	750	26.7E+6	7.70E-6	16600
								11	800	26.3E+6	7.80E-6	16600
								12	850	26.0E+6	7.85E-6	16600
								13	900	25.6E+6	7.90E-6	13600
								14	950	25.1E+6	8.00E-6	10800
								15	1000	24.6E+6	8.10E-6	8000
								16	1050	24.2E+6	8.15E-6	5700
								17	1100	23.7E+6	8.20E-6	3800
								18				

In the Material List window, you can edit inside both panes - the left pane contains Name, Description, Type of material, Density, Poisson's ratio (ν) and Joint factor and the right pane contains material properties (usually modulus of Elasticity, mean Coefficient of Linear thermal expansion [Alpha], not instantaneous nor total thermal, and the code-specific Allowable stress) as a function of Temperature. While entering the temperature-dependent material properties, you do not need to necessarily enter temperatures in an ascending order, although recommended. CAEPIPE will sort the entries later.

After you are done entering properties for one material, be sure to press Enter when the cursor is in the left pane, to move it to the next row so you can start entering the next material. You can insert, delete and edit any material (see under Edit menu).

These panes may change depending on the piping code chosen. For example, for the Swedish and Norwegian codes, the following window is displayed. This window contains additional columns for Tensile strength and instead of a single Joint factor, it has Longitudinal and Circumferential joint factors.

Material

Caepipe : Materials (1) - [Sample.mod (C:\CAEPIPE\681LM)]

File Edit View Options Misc Window Help

#	Name	Description	Type	Density (kg/m3)	Nu	Joint factor	Tensile (MPa)	#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	fh (MPa)	fCR (MPa)
1	A53	A53 Grade B	CS	7833	0.3	1.00	241.3	1	-198.3	216495	9.00E-6	137.9	
2								2	-128.9	212359	9.63E-6	137.9	
								3	-73.33	208222	10.17E-6	137.9	
								4	21.11	203395	10.93E-6	137.9	
								5	93.33	198569	11.48E-6	137.9	
								6	148.9	195122	11.88E-6	137.9	
								7	204.4	190985	12.28E-6	137.2	
								8	260	188227	12.64E-6	131.0	
								9	315.6	184090	13.01E-6	123.4	
								10	343.3	179953	13.19E-6	119.3	

The European (EN 13480) code has a column for Tensile strength and temperature-dependent properties have an additional column for fCR (allowable creep stress).

Caepipe : Materials (1) - [Sample.mod (C:\CAEPIPE\681LM)]

File Edit View Options Misc Window Help

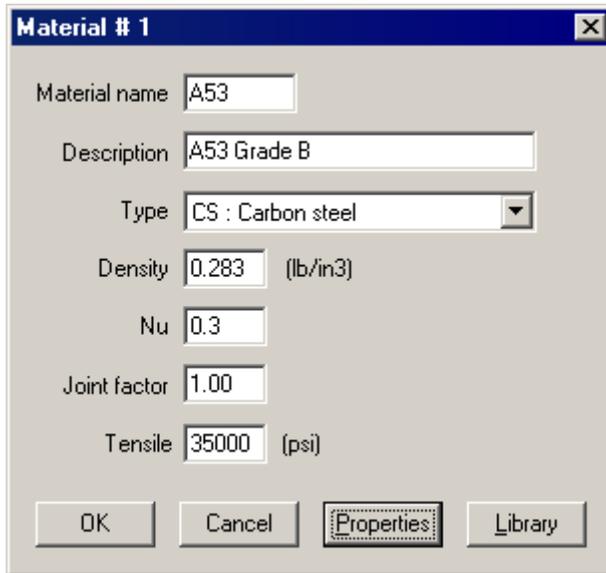
#	Name	Description	Type	Density (lb/in3)	Nu	Joint factor	Tensile (psi)	#	Temp (F)	E (psi)	Alpha (in/in/F)	fh (psi)	fCR (psi)
1	A53	A53 Grade B	CS	0.283	0.3	1.00	35000	1	-325	31.4E+6	5.00E-6	20000	
2								2	-200	30.8E+6	5.35E-6	20000	
								3	-100	30.2E+6	5.65E-6	20000	
								4	70	29.5E+6	6.07E-6	20000	
								5	200	28.8E+6	6.38E-6	20000	
								6	300	28.3E+6	6.60E-6	20000	
								7	400	27.7E+6	6.82E-6	19900	
								8	500	27.3E+6	7.02E-6	19000	
								9	600	26.7E+6	7.23E-6	17900	
								10	650	26.1E+6	7.33E-6	17300	

Material

To Input a New Material

You can input a new material in three ways:

1. Start typing directly into the fields in the Materials List window.
2. Input through dialogs (shown below for some but not all piping codes).



The dialog box titled "Material # 1" contains the following fields and controls:

- Material name: A53
- Description: A53 Grade B
- Type: CS : Carbon steel (dropdown menu)
- Density: 0.283 (lb/in³)
- Nu: 0.3
- Joint factor: 1.00
- Tensile: 35000 (psi)
- Buttons: OK, Cancel, Properties (highlighted), Library

#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	-325	31.4E+6	5.00E-6	20000
2	-200	30.8E+6	5.35E-6	20000
3	-100	30.2E+6	5.65E-6	20000
4	70	29.5E+6	6.07E-6	20000
5	200	28.8E+6	6.38E-6	20000
6	300	28.3E+6	6.60E-6	20000
7	400	27.7E+6	6.82E-6	19900
8	500	27.3E+6	7.02E-6	19000
9	600	26.7E+6	7.23E-6	17900
10	650	26.1E+6	7.33E-6	17300
11	700	25.5E+6	7.44E-6	16700

For the Swedish and Norwegian piping codes, the Material dialog has Longitudinal and Circumferential joint factors and a Tensile strength field.

Material

Material # 1

Material name:

Description:

Type:

Density: (lb/in3) Nu:

Long. joint: Circ. joint factor:

Tensile strength: (psi)

OK Cancel Properties Library

For the European (EN 13480) code, the Material dialog has a single Joint factor and a Tensile strength field, while the properties window has fCR (creep stress allowable).

Material # 1

Material name:

Description:

Type:

Density: (lb/in3)

Nu:

Joint factor:

Tensile: (psi)

OK Cancel Properties Library

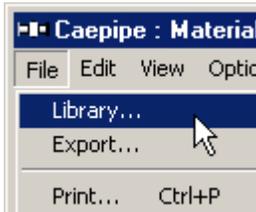
#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	f _h (MPa)	f _{CR} (MPa)
1	-198.3	216495	9.00E-6	137.9	
2	-128.9	212359	9.63E-6	137.9	
3	-73.33	208222	10.17E-6	137.9	
4	21.11	203395	10.93E-6	137.9	
5	93.33	198569	11.48E-6	137.9	
6	148.9	195122	11.88E-6	137.9	
7	204.4	190985	12.28E-6	137.2	
8	260	188227	12.64E-6	131.0	
9	315.6	184090	13.01E-6	123.4	

Material

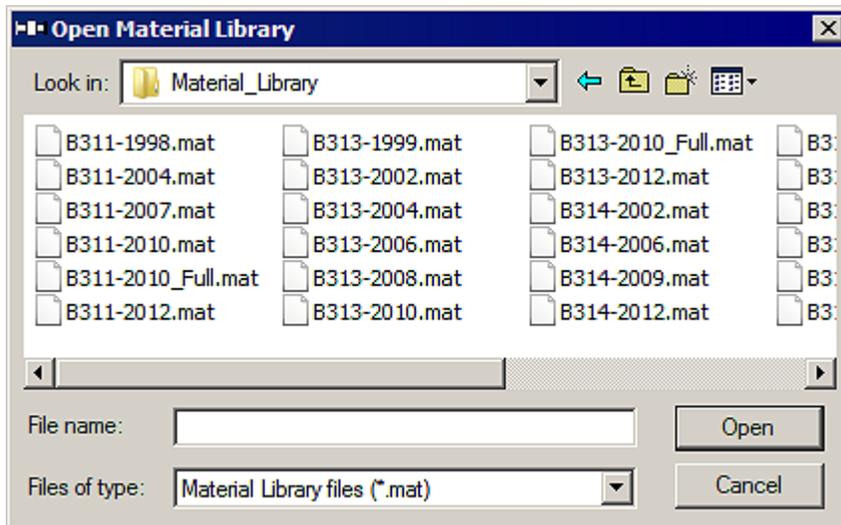
- Pick a material from an existing material library (supplied with CAEPIPE or your own). Click on the Library button on the toolbar to open the library:



(or select Library command under the File menu):

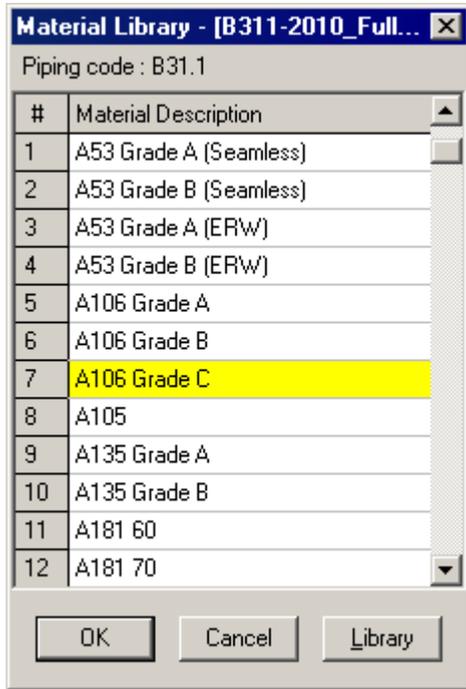


You will have to open a library file first if it was not previously opened. Select the one of interest. Note that the libraries with filenames ending with “_Full” (e.g., B313-2010_Full.mat) have approx. 200 materials each (from the respective piping code).



You can select a material from the opened library by double clicking on it or highlighting it and clicking on OK.

Material



Name

Type a Material name, up to five alpha-numeric characters.

Description

Type a description for the material, up to 31 characters.

Type

- AL for Aluminum
- AS for Austenitic Stainless Steel
- CA for Copper alloys annealed
- CC for Copper alloys cold worked
- CS for Carbon Steel
- FR for Fiber Reinforced Plastic piping
- FS for Ferritic steel
- NA for Nickel alloys 800, 800H, 825
- SS for Stainless Steel
- TI for Titanium

These material types are used in calculation of the Y factor for allowable pressure at high temperatures for certain piping codes. Swedish and Norwegian piping codes also use it for calculating allowable expansion stress range. These codes also need tensile strength.

For Fiber Reinforced Plastic piping, you need to select the material type “FR” to enter FRP material properties. More information can be found under the section Fiber Reinforced Plastic.

Density

Density of the material is used to calculate weight load and also mass for dynamic analysis.

Material

Nu

The Poisson's ratio (Nu) defaults to 0.3 if not input.

Joint factor

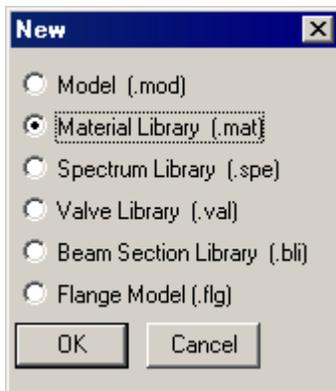
The joint factor is the longitudinal weld joint factor used to calculate allowable pressure. For Swedish and Norwegian piping codes, a circumferential joint factor is also input to calculate longitudinal pressure stress.

Tensile strength

For Swedish, Norwegian and European (EN13480) piping codes, tensile strength is used in the calculation of the allowable expansion stress range. For Stoomwezen piping code, tensile strength is used in the calculation of hot allowable stress.

To create or modify a material library

CAEPIPE offers you flexibility in creating your own material libraries (user-defined libraries). That way, you do not feel restricted by the offered choices in materials and can continually keep updating the material libraries with your own materials. To create a library: From the Main window, select File > New and click on Material Library.



A List window for materials is shown.

#	Description	Type	Density (lb/in3)	Nu	Joint factor	Yield (psi)	#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	UserMat	CS	0.283	0.3	1.00	50	1	70	30.0E+6	5.85E-6	24000
2							2	100			

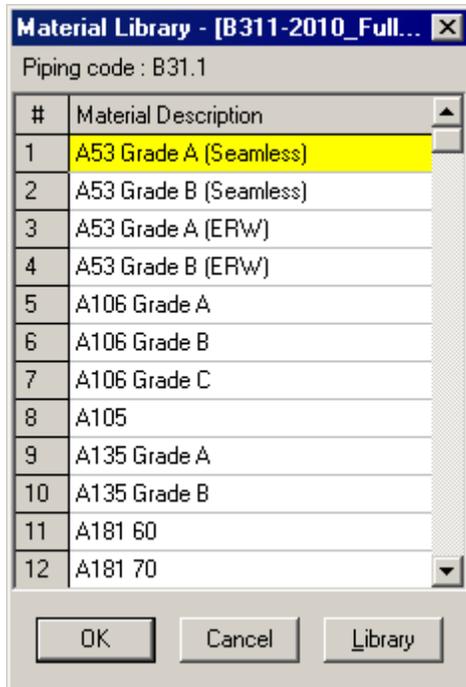
You must select a piping code first, using the menu command Options > Piping code, before you start entering properties.

You can, as before, start typing directly into the fields, or enter properties through a dialog. The only difference is that materials in the library do not have names whereas those in a model have names.

Material

Therefore, first make sure that you have selected the correct piping code (under Options > Analysis > Piping code) in the Layout window. Then, ensure that you use the correct properties from that code.

CAEPIPE comes with many libraries from several piping codes. When inside a CAEPIPE model, you can open any library and pick a material from it. Make sure to pick the proper library (with the proper year), especially between B31.1 and B31.3 libraries because they have significantly different allowable stresses for the same materials. Also verify the material properties in these libraries before you use them.



Missing Mass Correction

In dynamic analysis using modal superposition, usually an approximate solution is obtained because only a limited number of modes is considered. (For seismic analysis, typically all modes up to 33 Hz). The errors in pipe displacements and stresses are usually small because they are affected relatively little by high modes. The error in support loads may be substantial because the influence of high modes on support loads can be important. In stiff piping systems with few low frequency modes, stresses may also be affected significantly.

Using limited number of modes results in some mass of the system being ignored. The distribution of this “missing mass” is such that the inertia forces associated with it will usually produce small displacements and stresses. However these forces will often produce significant support loads, and in stiff systems can produce significant stresses.

A correction can be made by determining the modal contributions to the mass of the system and obtaining the “missing mass” as the difference between these contributions and the actual mass.

The inertial force vector for the n^{th} mode is given by

$$\{F_n\} = -[M]\{\ddot{u}_n\} = \omega_n^2 [M]\{\phi_n\}A_n \quad (1)$$

Where $[M]$ = diagonal mass matrix
 $\{\ddot{u}_n\}$ = acceleration vector
 ω_n = circular frequency
 $\{\phi_n\}$ = mass normalized eigenvector
 A_n = modal displacement for mode n

For X seismic excitation

$$A_n = \{\phi_n\}^T [M]\{r_x\} \frac{S_{nx}^a}{\omega_n^2} = \Gamma_{nx} \frac{S_{nx}^a}{\omega_n^2} \quad (2)$$

Where $\{r_x\}$ = displacement vector due to a unit displacement in the X direction
 S_{nx}^a = spectral acceleration for the n^{th} mode for excitation in the X direction
 Γ_{nx} = mass participation factor in the X direction for mode n

Let m = number of modes used in analysis
 N = total number of modes

If it is assumed that the higher modes: $m+1$ through N are in phase and have a common spectral acceleration S_{ox}^a (conservatively taken as the maximum spectral acceleration after the m^{th} mode), the total inertial force contribution of these higher modes (also known as “Rigid body force” or “Left out force”) is

$$\{F_x^R\} = S_{ox}^a [M] \sum_{n=m+1}^N \{\phi_n\} \Gamma_{nx} \quad (3)$$

Missing Mass Correction

It can be shown that

$$\{r_x\} = \sum_{n=1}^N \{\phi_n\} \Gamma_{nx} = \sum_{n=1}^m \{\phi_n\} \Gamma_{nx} + \sum_{n=m+1}^N \{\phi_n\} \Gamma_{nx} \quad (4)$$

Substituting from (4) for the summation in (3)

$$\{F_x^R\} = S_{ox}^a [M] \left[\{r_x\} - \sum_{n=1}^m \{\phi_n\} \Gamma_{nx} \right] \quad (5)$$

Note that there will be missing mass inertia forces in the Y and Z directions, in addition to the X direction, for X excitation.

The missing mass force vectors for the Y and Z directions are similarly calculated. The response to each of these three force vectors is calculated and these additional response vectors are combined with the responses of the first “m” modes.

This feature is currently not available for Time History and Harmonic analyses.

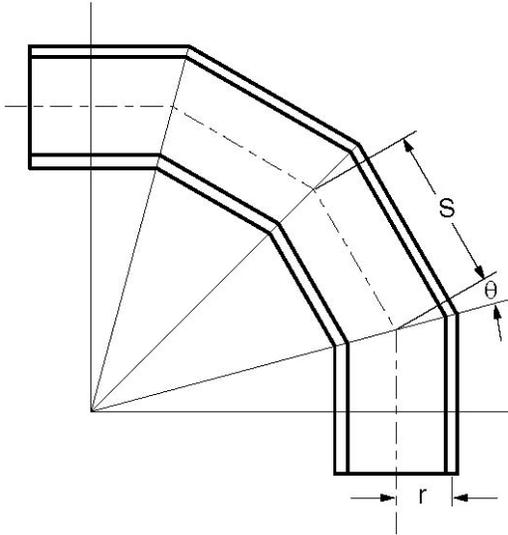
The above described method is based on the technical paper by Powell. See below for details.

Powell, G.H. “Missing Mass Correction in Modal Analysis of Piping Systems.” Transactions of the 5th International Conference on Structural Mechanics in Reactor Technology. August 1979: Berlin, Germany.

Miter Bend

Miter bends are typically used when space limitations do not allow the use of regular bends (elbows), or when a miter is more economical to use than a regular bend. Mitters are not fittings. They are fabricated from pipe, to requirements. “The use of miters to make changes in direction is practically restricted to low-pressure lines, 10-inch and larger if the pressure drop is unimportant...” (Sherwood 1980).

See figure below for Miter bend parameters.

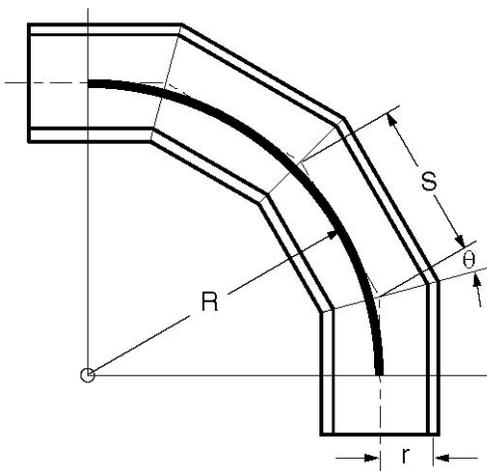


In this figure, r = mean radius of pipe
 S = miter spacing at center line
 θ = one-half angle between adjacent miter axes ($\leq 22.5^\circ$)

Before modeling a miter bend, you should determine whether it is closely or widely spaced.

Closely Spaced Miter

A miter bend is closely spaced when $S < r(1 + \tan \theta)$.



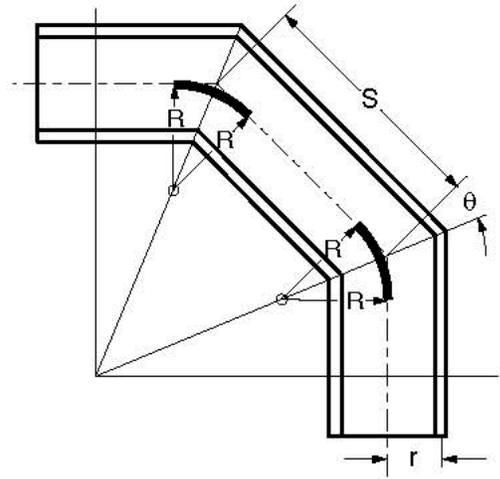
A closely spaced miter bend is input as a single miter bend element.

The Bend Radius (R) is calculated as: $R = 0.5 S \cot \theta$.

Miter Bend

Widely Spaced Miter

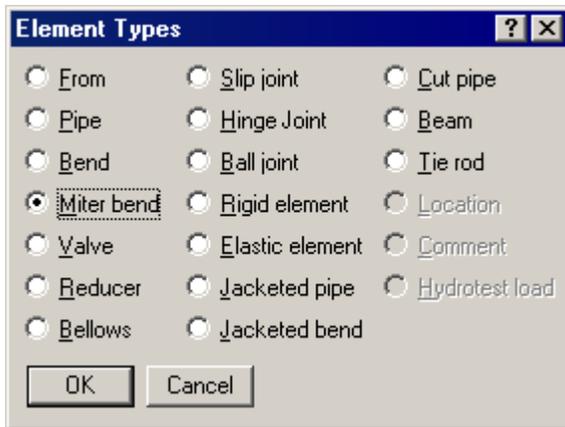
A miter bend is widely spaced when $S \geq r(1 + \tan \theta)$.



A widely spaced miter bend is modeled with as many miter bend elements as there are miter cuts.

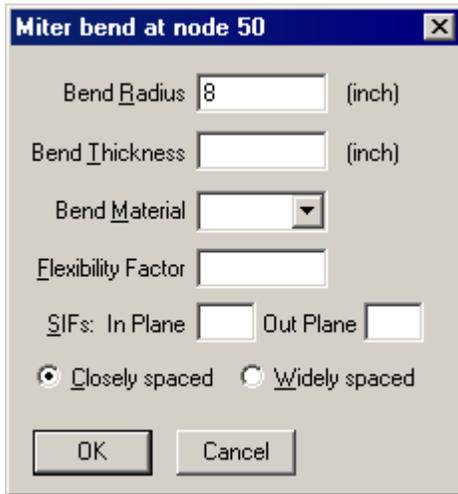
The Bend Radius (R) is calculated as: $R = 0.5 r (1 + \cot \theta)$.

A miter bend is input by typing “m” in the Type column or selecting “Miter bend” from the Element types dialog.



Miter Bend

The Miter bend dialog is shown.



Bend Radius

The bend radius (R) depends on the type of miter (Closely or Widely spaced). It is calculated as explained previously and input in this field.

Bend Thickness

Input the wall thickness of the miter bend if it is different from that of the adjoining pipe thicknesses. The Bend Thickness, if specified, applies only to the curved portion(s) of the equivalent bend(s) of the miter bend.

Bend Material

If the material of the miter bend is different from that of the adjoining pipe, select the Bend Material from the drop down combo box. The Bend Material, if specified, applies only to the curved portion(s) of the equivalent bend(s) of the miter bend.

Flexibility Factor

Specify a flexibility factor for the miter bend if different from the piping code's factor. If you specify one, CAEPIPE uses it instead of the piping code specified Flexibility Factor. A value of 2.0, for e.g., will mean that the miter bend is twice as flexible as a pipe of the same length.

Closely spaced

To specify the miter bend as closely spaced, click on the "Closely spaced" radio button.

Widely spaced

To specify the miter bend as widely spaced, click on the "Widely spaced" radio button.

Parameters for 90° Miter Bends

The parameters (dimensions) for 90° miter bends (with 2, 3, 4 miter cuts) in terms of dimension (A), mean pipe radius (r) and number of miter cuts (N) are shown in the following table.

Miter Bend

Use the table to determine whether the miter bend is Closely spaced or Widely spaced, and if it is Widely spaced and has 2, 3 or 4 miter cuts, to calculate the dimensions B, C, D, E and R (equivalent miter bend radius) before modeling it.

A closely spaced miter requires only the miter bend radius (same as dimension A shown in the figures).

			Dimensions for Widely spaced miters only				
N	S	Closely spaced if	B	C	D	E	R
2	0.828427 A	$A < 1.707107 r$	0.414214 A	0.585786 A	-	-	1.707107 r
3	0.535898 A	$A < 2.366025 r$	0.267949 A	0.464102 A	-	-	2.366025 r
4	0.397825 A	$A < 3.013670 r$	0.198912 A	0.367542 A	0.281305 A	0.152241 A	3.013670 r

N = Number of miter cuts

Half angle $\theta = 90^\circ / (2N)$

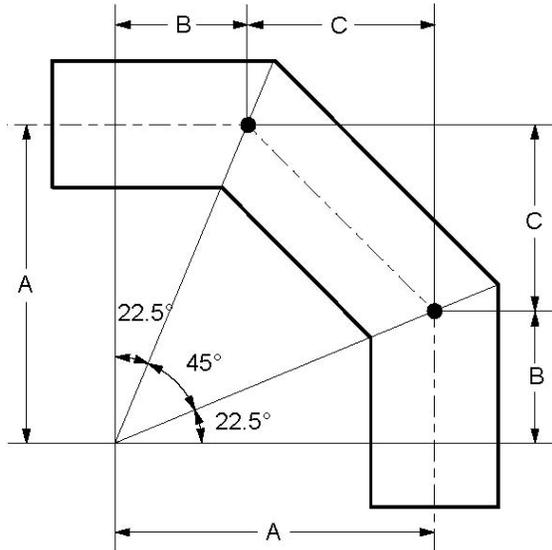
The following pages show the details of how dimensions B, C, D and E were calculated, and are provided only for your information. The above table is important for your modeling requirements. For miters with more than 4 cuts, you have to calculate the required dimensions similar to those shown on the following pages. The next table outlines the modeling procedure for either miter type.

Miter Modeling Procedure

Determine Miter Type			
Closely Spaced Miter	Widely Spaced Miter		
Any number of cuts	2 cuts	3 cuts	4 cuts
Model miter as a single Closely spaced miter with Bend radius as dimension A.	Calculate R, B and C from above table. Calculate offsets of nodes using B and C. See Example 2 Widely spaced miter.		Calculate R, B, C, D and E from above table. Calculate offsets of nodes using B, C, D and E.
See Example 1, Closely spaced miter.	Model this miter as 2 widely spaced miters with R as the bend radius.	Model this miter as 3 widely spaced miters with R as the bend radius.	Model this miter as 4 widely spaced miters with R as the bend radius.

Miter Bend

Two Miter Cuts (N=2)



$$\theta = 90^\circ / (2 \times 2) = 22.5^\circ$$

$$S = 2 A \tan \theta = 0.828427 A$$

$$B = A \tan \theta = 0.414214 A$$

$$C = 2 A \tan \theta \cos 2\theta = 0.585786 A$$

Closely Spaced

The miter is Closely spaced if,

$$S < r (1 + \tan \theta)$$

Substituting for S,

$$2A \tan \theta < r (1 + \tan \theta)$$

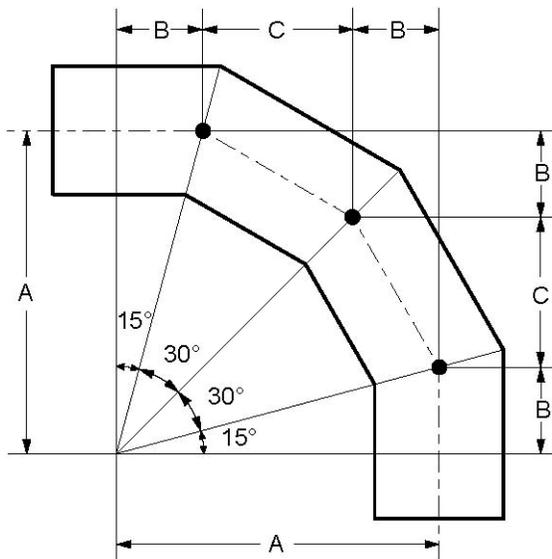
$$A < 1.707107 r$$

$$\text{Bend radius, } R = 0.5 S \cot \theta = A$$

Widely Spaced

$$R = 0.5 r (1 + \cot \theta) = 1.707107 r$$

Three Miter Cuts (N=3)



$$\theta = 90^\circ / (2 \times 3) = 15^\circ$$

$$S = 2 A \tan \theta = 0.535898 A$$

$$B = A \tan \theta = 0.267949 A$$

$$C = 2 A \tan \theta \cos 2\theta = 0.464102 A$$

Closely Spaced

The miter is Closely spaced if,

$$S < r (1 + \tan \theta)$$

Substituting for S,

$$2A \tan \theta < r (1 + \tan \theta)$$

$$A < 2.366025 r$$

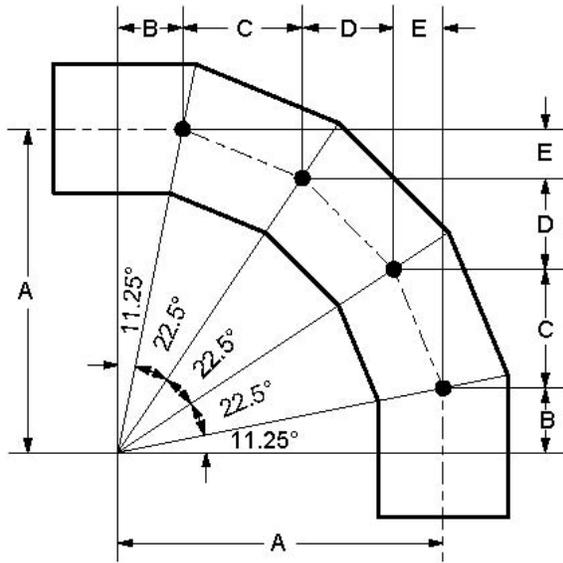
$$\text{Bend radius, } R = 0.5 S \cot \theta = A$$

Widely Spaced

$$R = 0.5 r (1 + \cot \theta) = 2.366025 r$$

Miter Bend

Four Miter Cuts (N=4)



$$\theta = 90^\circ / (2 \times 4) = 11.25^\circ$$

$$S = 2 A \tan \theta = 0.397825 A$$

$$B = A \tan \theta = 0.198912 A$$

$$C = 2 A \tan \theta \cos 2\theta = 0.367542 A$$

$$D = 2 A \tan \theta \sin 4\theta = 0.281305 A$$

$$E = 2 A \tan \theta \sin 2\theta = 0.152241 A$$

Closely Spaced

The miter is Closely spaced if,

$$S < r (1 + \tan \theta)$$

Substituting for S,

$$2A \tan \theta < r (1 + \tan \theta)$$

$$A < 3.013670 r$$

$$\text{Bend radius, } R = 0.5 S \cot \theta = A$$

Widely Spaced

$$R = 0.5 r (1 + \cot \theta) = 3.013670 r$$

Example 1: Closely Spaced Miter

Example Data

Pipe OD = 8.625", thickness, $t = 0.322$ "

Mean pipe radius, $r = (8.625 - 0.322) / 2 = 4.1515$ ",

Number of miter cuts = 3,

Dimension A = 8", See 3-cut miter figure.

Look up table (Miter modeling procedure), for an outline of the modeling procedure.

First, determine the type of miter (Closely or Widely spaced) before modeling it.

Look up Summary of Miter parameters, for $N = 3$. A miter is Closely spaced if $A < 2.366025r$. This condition is true for $r = 4.1515$ ". Hence, this is a Closely spaced miter.

Steps for Example 1

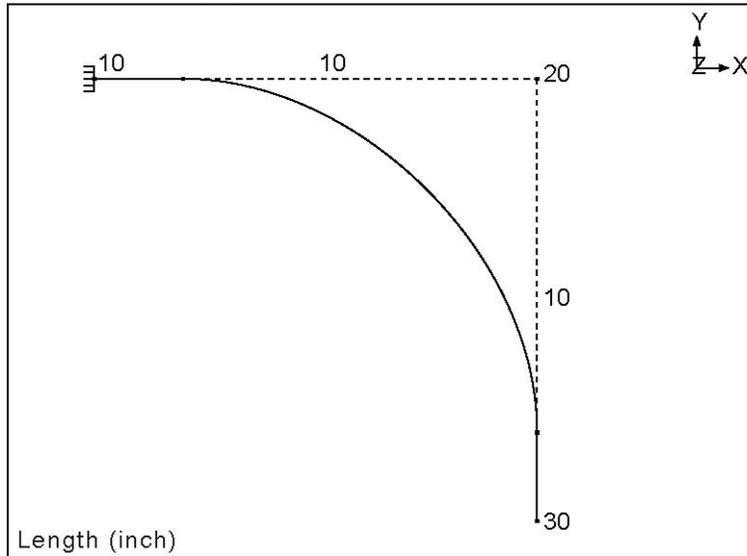
- ▶ Create From node: When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- ▶ Construct miter bend: type 20 for Node (simply pressing Tab puts this node number automatically for you), type "m" under the Type column, type 8" for bend radius, select Closely spaced, click on Ok. Type 10" for DX, enter material, section and load names, press Enter.
- ▶ Finish the miter bend: type 30 for Node and -10" for DY, press Enter.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Example 1: Closely spaced Miter								
2	10	From							Anchor
3	20	Miter	10			1	8	1	
4	30			-10		1	8	1	

Miter Bend

Example 1: Closely Spaced Miter

MITER1



Example 2: Widely Spaced Miter

Let us assume the same data as in Example 1 (Closely spaced miter) with only one change, namely, dimension A (see next figure).

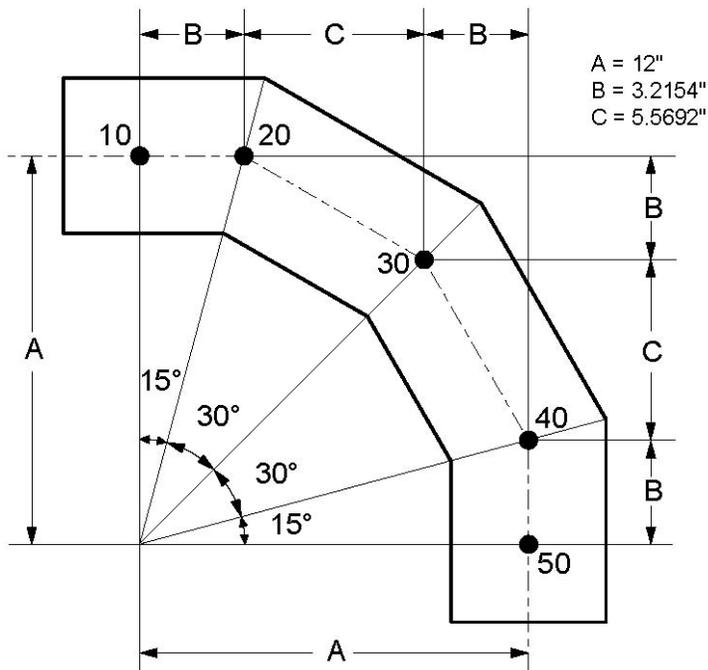
Pipe OD = 8.625", thickness, $t = 0.322$ "

Mean pipe radius, $r = (8.625 - 0.322) / 2 = 4.1515$ ",

Number of miter cuts = 3,

Dimension A = 12".

Look up table (for Miter modeling procedure) for an outline of the modeling procedure.



It is essential to determine the type of miter (Closely or Widely spaced) before modeling it.

Miter Bend

Determine miter type

Look up table, Summary of Miter parameters, for $N = 3$. A miter is Closely spaced if $A < 2.366025r$. This condition is false for $r = 4.1515''$. Hence, this is a Widely spaced miter. This miter bend has to be modeled as a series of 3 miters. Next, observe in this table that dimensions B, C and R (Equivalent miter bend radius) can be calculated for $N = 3$ cuts.

Calculate required dimensions

With $r = 4.1515''$,
Equivalent miter bend radius, $R = 2.366025 \times r = 9.8225''$,
Dimension B = $0.267949 \times A = 3.2154''$,
Dimension C = $0.464102 \times A = 5.5692''$

See previous figure. After calculating B, C and R, let us now calculate the offsets of nodes 20, 30 and 40 (the 3 nodes correspond with the 3 miter cuts).

Calculate Offsets

Offsets of node 20 from 10: (First miter cut)
 $DX = B = 3.2154''$
 $DY = 0''$ (because node 20 is on the horizontal axis).

Offsets of node 30 from 20: (Second miter cut)
 $DX = C = 5.5692''$
 $DY = -B = -3.2154''$

Offsets of node 40 from 30: (Third miter cut)
 $DX = B = 3.2154''$
 $DY = -C = -5.5692''$

Offsets of node 50 from 40:
 $DX = 0''$ (because node 50 is on the vertical axis).
 $DY = -B = -3.2154''$

Now, start to build the model in CAEPIPE as shown in Example 1 but with different data (bend radius = $9.8225''$, select Widely Spaced miter, and offsets as shown above).

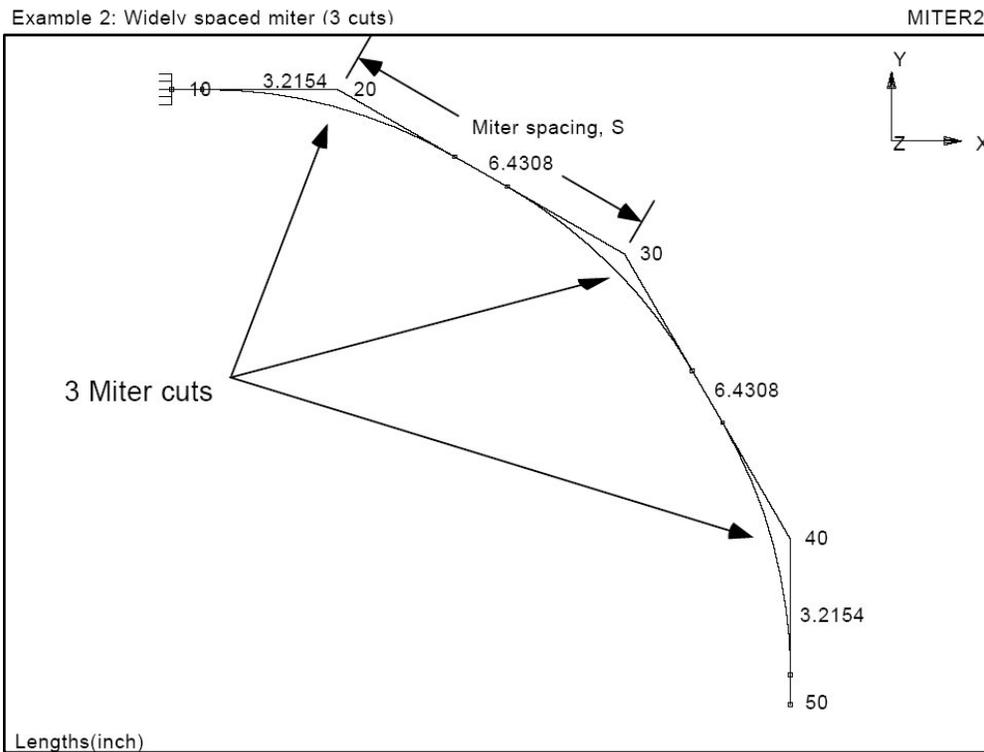
Steps for Example 2

- ▶ Create From node: When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- ▶ Construct first miter bend: type 20 for Node (simply pressing Tab puts this node number automatically for you), type "m" under the Type column, type $9.8225''$ for bend radius, select Widely spaced, click on Ok. Type $3.2154''$ for DX, enter material, section and load names, press Enter.
- ▶ Construct second miter bend: type 30 for Node (simply pressing Tab puts this node number automatically for you), type "m" under the Type column, type $9.8225''$ for bend radius, select Widely spaced, click on Ok. Type $5.5692''$ for DX, $-3.2154''$ for DY, press Enter.

Miter Bend

- ▶ Construct third miter bend: type 40 for Node (simply pressing Tab puts this node number automatically for you), type “m” under the Type column, type 9.8225” for bend radius, select Widely spaced, click on Ok. Type 3.2154” for DX, -5.5692” for DY, press Enter.
- ▶ Finish the miter bend: type 50 for Node and -3.2154” for DY, press Enter.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Example 2: Widely spaced miter (3 cuts)								
2	10	From							Anchor
3	20	Miter	3.2154			1	8	1	
4	30	Miter	5.5692	-3.2154		1	8	1	
5	40	Miter	3.2154	-5.5692		1	8	1	
6	50			-3.2154		1	8	1	



Sherwood, David. R., and Dennis J. Whistance. THE “PIPING GUIDE” A Compact Reference for the Design and Drafting of Industrial Piping Systems. First Edition (revised). San Francisco: Syntek Books Co., 1980.

Node

A Node refers to a connecting point between two elements such as pipes, reducers, valves or expansion joints. The maximum # of nodes in a model cannot exceed 9,999, while the maximum node number itself cannot be greater than 99,999. Usually, a node has a numeric designation. In CAEPIPE, occasionally, you may have a need to reference a node followed by a letter such as A/B/C/D/J. These are automatically generated internal nodes. A and B nodes (e.g., 20A, 20B) designate the near and far ends of a Bend node (see section on Bend). J, C and D (e.g., 10J, 10C, 10D) designate a Jacketed pipe and a Jacketed bend (see section on Jacketed Piping).

In the Layout window, node numbers are typed under the column titled Node. A node number may be typed as an integer or an integer followed by one of the letters A/B/C/D/J. Use “Location” type to specify more than one data item at a node (See section on Location).

The starting node in a piping system is always a node of type “From,” which is usually anchored.

Nodes not only act as connect points for elements but also act as locations for providing supports or applying restraints/external forces and moments to the piping system. Each node has six static degrees of freedom (three dynamic), three translational (in x, y and z directions) and three rotational (about x, y and z axes). Any or all of them may be restrained using supports.

Specifying Coordinates

The values typed in the DX, DY and DZ fields on the Layout window are interpreted as offsets from the previous node. If required to specify absolute coordinates for a node (i.e., fix the location of a point in space), the node has to be of type “From” or should have an asterisk (*) at the end of it (e.g., 20*). In these cases, the numbers entered in the DX, DY and DZ fields are interpreted as absolute coordinates of the node rather than offsets from the previous node. If the coordinates for a particular node are duplicated the second set of values is ignored. An asterisk (*) for a “From” node is ignored too.

You can list all coordinates by selecting Coordinates under Misc menu (or click on the Node header or right click on any node number). This feature can be helpful for verifying correctness of the input.

In the Layout window, you can search for a node by using the “Find node” command (under the View menu, or use Ctrl+F), specify a “Node increment” for automatic node numbering (under Options menu), and renumber nodes for a range of rows (under Edit menu)

Automatic Renumbering

When you delete a row with a node number in the Layout window, CAEPIPE automatically renumbers nodes from the top starting with the number you have specified under the main window > File > Preferences > Automatic Renumbering. Note that this is different from the user-performed selective renumbering of a range of rows from the Edit menu.

When turned on, deleting a row triggers an automatic renumbering operation inside the Layout window. So if you do not want such to happen, turn the feature off from the main window.

Nonlinearities

There are several types of nonlinearities in CAEPIPE:

1. Gaps in limit stops, guides.
2. Rotational limits in ball and hinge joints.
3. Rod hangers as one-way restraints.
4. Tie rods with different stiffnesses and gaps in tension and compression.
5. Friction in limit stops, guides, slip joints, hinge joints and ball joints.
6. Buried piping.

An iterative solution is performed when nonlinearities are present. At each iteration, the stiffness matrix and the load vector are reformulated because both of these depend on the solution (displacements) from the previous iteration.

Limit stop

Limit stops are input by specifying direction, the upper and lower limits and optionally a friction coefficient. The upper and lower limits are along the direction of the limit stop and measured from the undeflected position of the node. Typically the upper limit is positive and the lower limit negative. The upper limit should be algebraically always greater than the lower limit. In some situations it is possible to have a positive lower limit or a negative upper limit. If a particular limit does not exist (i.e., a node can move unrestrained in that direction), that limit should be left blank. If a gap does not exist then the limit should be explicitly input as zero.

Solution Procedure

At each iteration, the displacements at the limit stop node are resolved along the limit stop direction. The resolved displacement is compared with the upper and lower limits. If the displacement exceeds the upper limit or is less than the lower limit the gap is closed otherwise it is open. The solution is converged when the displacement is within 1% of previous displacement.

If the gap is open, no stiffness or force is applied due to the limit stop at the node.

If the gap is closed, a high stiffness (1×10^{12} lb./inch) along with a force = gap x high stiffness is applied in the direction of the limit stop at the node. The limit stop in this case acts like a specified displacement.

After the solution has converged and if the gap is closed, the support load at the limit stop is calculated by: (resolved displacement - gap distance) x high stiffness. During hanger design, the hot loads are recalculated with the status of the limit stops at the end of the preliminary operating load case. Then the hanger travels are recalculated using the recalculated hot loads.

In dynamic analysis, the status of the limit stops for the first operating case (W+P1+T1), is used. If either the upper or lower limit is reached for the first operating case, the limit stop is treated as a rigid two-way restraint in the direction of the limit stop. If the limit is not reached, then it is treated as having no restraint.

Friction

Friction is specified by entering coefficient of friction for limit stop and guide, entering friction force and/or friction torque for slip joint, entering friction torque for hinge joint and entering bending and/or torsional friction torque for ball joint.

Friction is modeled using variable equivalent stiffnesses (fictitious restraints) in CAEPIPE. The stiffness of these fictitious restraints is estimated from the results of previous iteration. If friction is included in dynamic analysis, the values of the equivalent stiffnesses from the first operating case are used in modal and dynamic analyses.

Friction in Limit Stop

If the gap is not closed there is no normal force and hence no friction. If the gap is closed, the normal force (limit stop support load) is calculated as explained above. The maximum friction force is friction coefficient * normal force. The displacement of the limit stop node is resolved into a plane normal to the limit stop direction (let us call this resolved displacement: y). Also let k_y = equivalent friction stiffness which is assumed to be zero for first iteration.

If y is non zero or $y * k_y >$ maximum friction force

then $k_y =$ maximum friction force / y

otherwise $k_y =$ high stiffness (1×10^{12} lb/inch) [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the stiffness matrix. The iterations are continued till the displacement y is within 1% of displacement from the previous iteration. The friction force is $y * k_y$.

Friction in Guide

A guide is modeled by adding high stiffnesses perpendicular to the direction of the pipe. The normal force in the guide is calculated by the vector sum of the local y and z support loads. Maximum friction force is friction coefficient * normal force. The displacements at the guide node are resolved in the direction of the guide. Let us call this displacement: x . Also let $k_x =$ equivalent friction stiffness which is assumed to be zero for first iteration.

If x is non zero or $x * k_x >$ maximum friction force

then $k_x =$ maximum friction force / x

otherwise $k_x =$ high stiffness (1×10^{12} lb/inch) [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the stiffness matrix. The iterations are continued till the displacement x is within 1% of displacement from the previous iteration. The friction force is $x * k_x$.

Friction in Slip Joint

The relative displacements (From node, To node) for the slip joint are resolved in the direction of the slip joint. Let us call this relative displacement: x . Also let k_x = equivalent friction stiffness which is assumed to be zero for first iteration.

If x is non zero or $x * k_x >$ friction force

then $k_x =$ friction force / x

otherwise $k_x =$ high stiffness (1×10^{12} lb/inch) [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the stiffness matrix. The iterations are continued till the displacement x is within 1% of displacement from the previous iteration.

Similar technique is used for friction torque (using rotations instead of translations).

Friction in Hinge Joint

The relative rotations (From node, To node) for the hinge joint are resolved in the direction of the hinge axis. Let us call this relative rotation: x . Also let k_x = equivalent friction stiffness which is assumed to be zero for first iteration. Maximum friction torque = friction torque + hinge stiffness * x .

If x is non zero or $x * k_x >$ maximum friction torque

then $k_x =$ maximum friction torque / x

otherwise $k_x =$ high stiffness [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the stiffness matrix. The iterations are continued till the displacement x is within 1% of displacement from the previous iteration.

Friction in Ball Joint

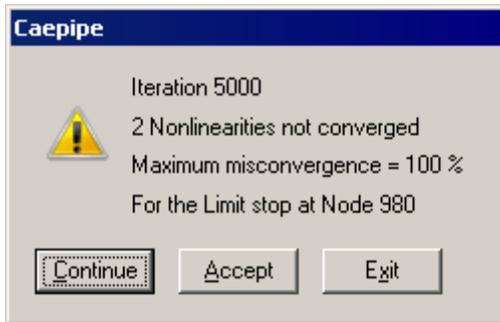
For a ball joint friction in bending (transverse) and torsional (axial) directions is treated independently. For the bending case, the resultant of the local y and z directions is used. Otherwise a procedure similar to the one used for hinge joint is used.

Friction in Dynamic Analysis

Friction is optional in dynamic analysis. Friction is mathematically modeled by using equivalent stiffnesses. If friction is included in dynamic analysis, the values of the equivalent stiffnesses from the first operating case are used.

Misconvergence

During the iterative solution procedure for nonlinearities, a misconception is reported in the following manner:



You have three options:

- Continue the iterative procedure for 5000 more iterations to see whether the solution converges, or
- Accept the misconception (maximum misconception is reported, 100% in the above dialog), or
- Exit the analysis processor completely.

An environment variable "CPITER" may be defined to change number of iterations from the default 5000. For example, when "CPITER=10000", iterations will continue up to 10,000 before showing the dialog: Continue, Accept or Exit if there is nonconvergence.

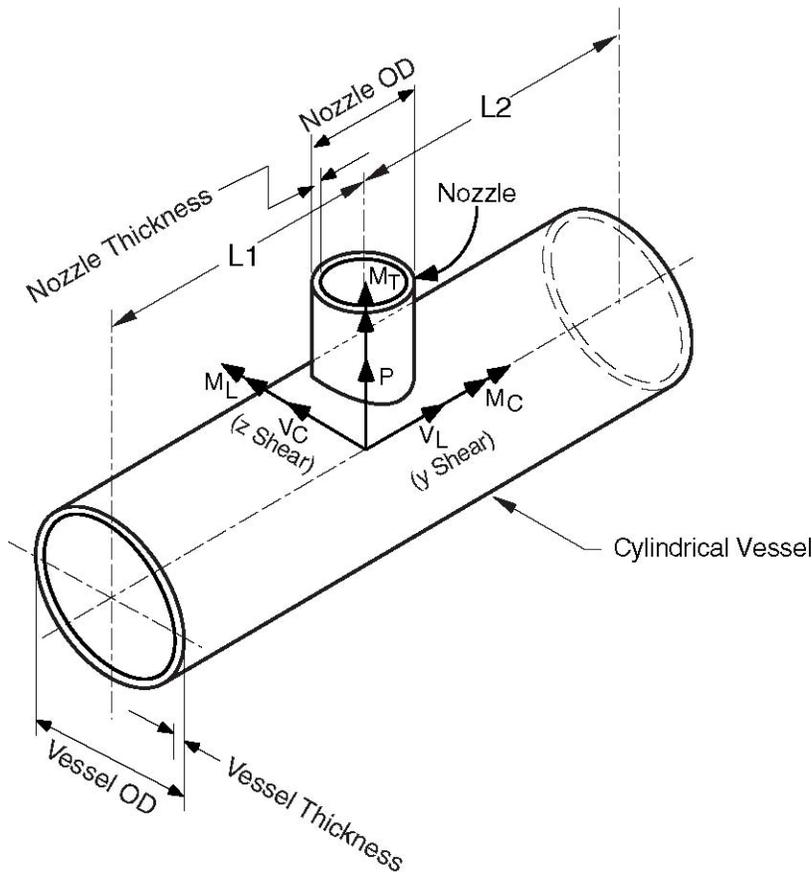
The shown misconception in the solution is really a quantification of how much off the results will be, IF you accept it. The maximum misconception (100%) is shown along with its location where such is happening. For example, a misconception of 100% could mean that the results at that location (say support loads etc.) may be off by 100%. So, for example, if the support load is 1,200 lb., the true range could be within a bracket of (1200 +/- 192 lb.); similarly with other results. Results at the other nonconverged locations (191 more in this example) will be below this reported number (100%).

So, the results are not necessarily unusable. If the support load were 27,000 lb., then either the support should be designed to handle the true range, or other changes should be made to the model to reduce the amount of misconception. Sometimes, you may observe during analysis that the misconception reported begins to oscillate among a few values for # of nonlinearities not converged (e.g., 32, 8, 16, 4, 32, 8, 16, 4, ad infinitum) in which case, you will know that this process will not converge. Now you have, as before, two options: 1. Change parameters (mainly gaps and friction values, or removal of unneeded supports) inside the model to influence the convergence routine, or, 2. Increase the # of iterations. In case neither works, then use engineering judgment to accept or reject the merit of such a solution with variations as described above.

Nozzle

Nozzles are integral attachments of vessels (such as a storage tank) which connect with external piping. Nozzles transmit the shell (vessel) flexibility to the piping system and hence included in piping stress analysis.

Two types of nozzles can be modeled: A nozzle attached to a cylindrical vessel relatively far from the ends of the cylinder, and a nozzle attached to a cylindrical shell with a flat bottom and close to the flat-bottom. CAEPIPE calculates the nozzle stiffnesses (local flexibility components) according to WRC 297 and API 650 guidelines. See Appendix D for the procedures.



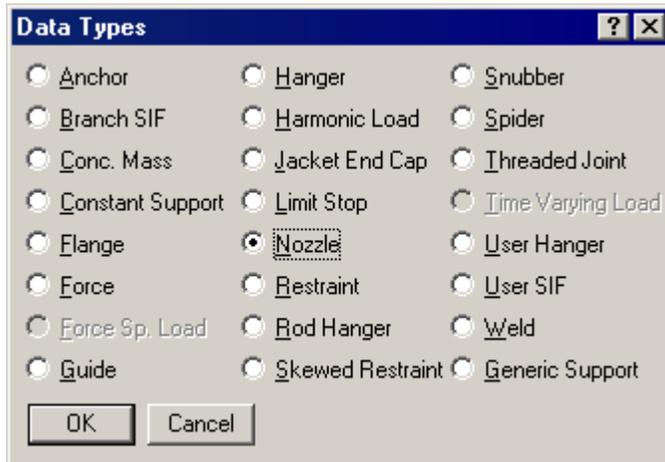
The coordinate system is as shown in the figure. The six components of the forces and moments at the nozzle-vessel interface are:

P = Radial load	M_C = Circumferential moment
V_C = Circumferential load	M_T = Torsional moment
V_L = Longitudinal load	M_L = Longitudinal moment

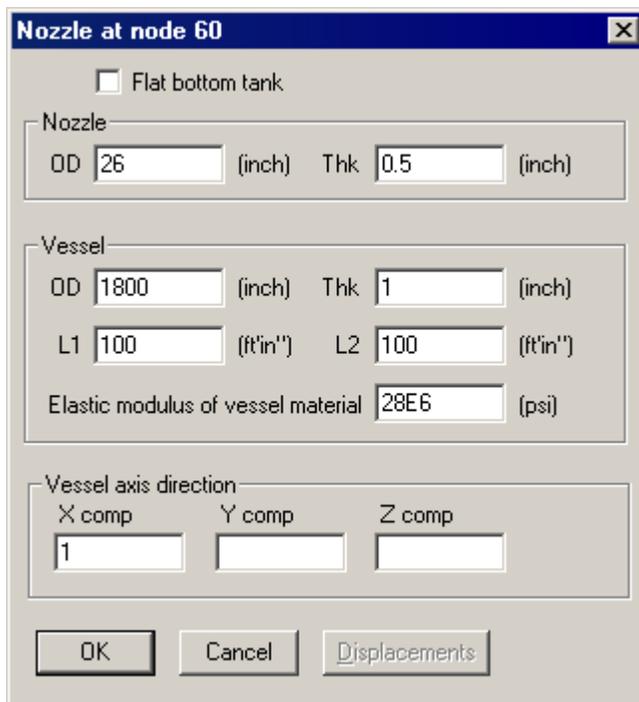
Of the six components of stiffnesses, only three stiffnesses, axial (K_x), circumferential (K_{yy}), and longitudinal (K_{zz}), are computed. The remaining three are assumed to be rigid.

Nozzle

A nozzle is input by typing “n” in the Data column or selecting “Nozzle” from the Data Types dialog.



The Nozzle dialog is shown. Note that the Displacements button is disabled. It is only enabled after the nozzle is input (i.e., existing nozzle).



Nozzle

OD: Outside diameter of the nozzle.

Thk: Thickness of the nozzle.

Vessel

OD: Outside diameter of the vessel.

Thk: Thickness of the vessel.

L1, L2: Distances from the nozzle to the nearest stiffening ring, tubesheet or the vessel end.

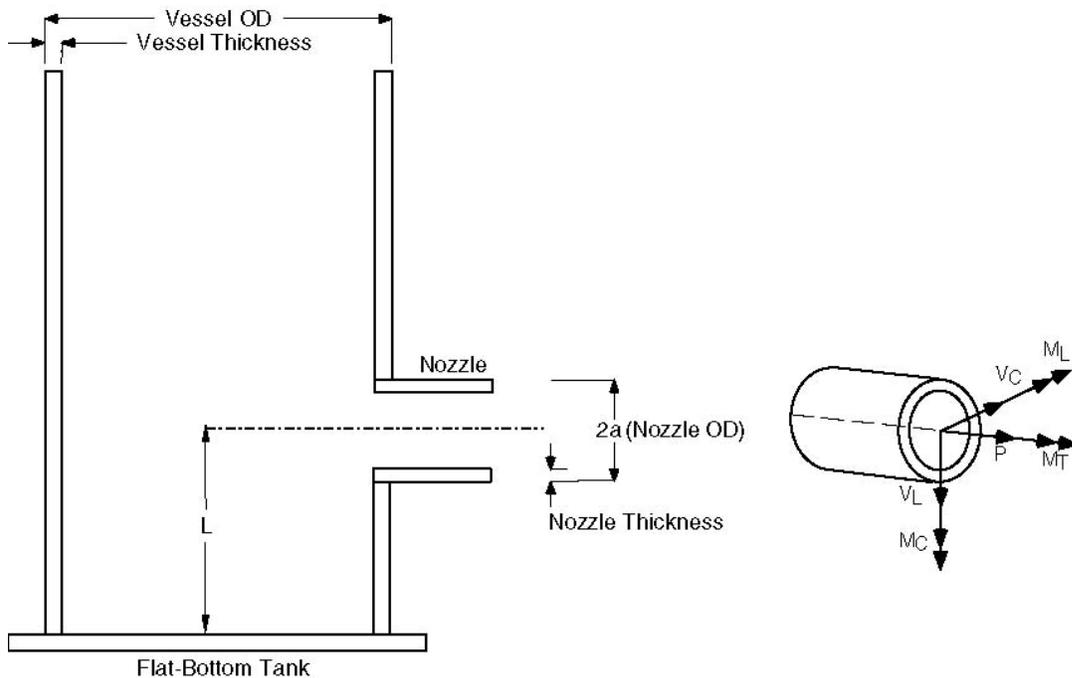
Nozzle

Vessel axis direction

The orientation of the vessel axis in terms of its global X, Y and Z components are entered here. See example under “Specifying a Direction.”

Flat-Bottom Tanks

For a nozzle on a flat-bottom tank and close to the flat-bottom, check the Flat-bottom tank checkbox.



A slightly modified Nozzle dialog is displayed.

Nozzle at node 60 [X]

Flat bottom tank Reinforcing pad

Nozzle

OD (inch) Thk (inch)

Vessel

OD (inch) Thk (inch)

L (ft'in')

Elastic modulus of vessel material (psi)

Vessel axis direction

X comp Y comp Z comp

OK Cancel Displacements

Nozzle

All the input fields are the same as before except:

L

L is the distance from the flat-bottom to the nozzle centerline.

Reinforcing pad

If the nozzle is reinforced, check this box.

Stiffness Calculation

Here again, only three stiffnesses, axial, circumferential and longitudinal are computed according to API 650 guidelines. See Appendix D for the procedure.

The flat-bottom tank nozzles are subject to the following limitations (API 650):

Limitations

- ▶ Nozzle OD/Vessel OD ratio must be between 0.005 and 0.04.
- ▶ L/Nozzle OD ratio must be between 1.0 and 1.5. See graphs in Appendix D, Figures D-3 through D-14.

Displacements

Displacements (translations and/or rotations) in the global X, Y and Z directions may be specified for nozzles (for thermal, settlement and seismic cases). Click on the Displacements button. You will see a dialog similar to one shown below. Type in specified displacements and press Enter.

Load	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
T1	-0.5	0.25	0.15	0.1	-0.2	-0.9
T2						
T3						
T4						
T5						
T6						
T7						
T8						
T9						
T10						

Nozzle

There are three types of displacements which can be specified:

1. Thermal (up to 10 displacements can be specified, one each for thermal loads T1 through T10). Applied only to the Expansion and Operating load cases.
2. Seismic (available for B31.1, ASME Section III Class 2, RCC-M and EN 13480 codes only). Solved as a separate internal load case and added absolutely to static seismic and response spectrum load cases.
3. Settlement (available under ASME Section III Class 2, RCC-M and EN 13480 codes only) Applied to a separate load case called Settlement.

Settlement

For certain piping codes (ASME Section III Class 2, RCC-M and EN 13480), a settlement, which is a single nonrepeated movement (e.g., due to settlement of foundation), may be specified. This is applied to the Settlement load case. For those codes which do not have a provision for settlement (like B31.1), specify the settlement as a thermal displacement (a conservative approach).

Example 1: Nozzle on a cylindrical vessel

Assume the following data:

Vessel OD = 1800 in.

Vessel Thickness = 1.0 in.

Nozzle OD = 26 in.

Nozzle Thickness = 0.5 in.

L1 = L2 = 1200 in.

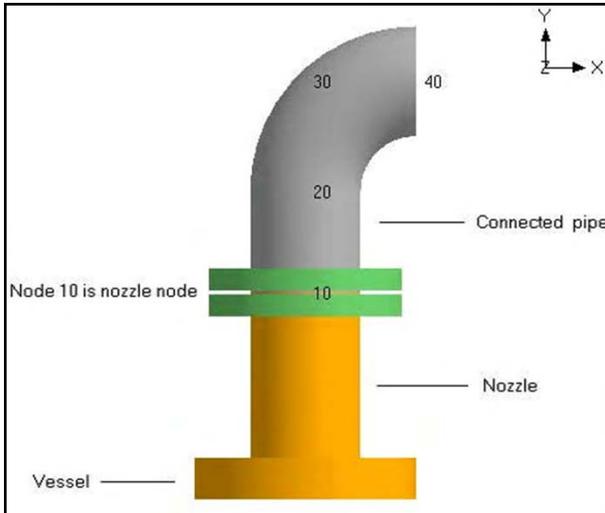
Elastic modulus for vessel material = 28×10^6 psi.

The first node (10) is already defined as an anchor. To replace the anchor by a nozzle, right click on the Anchor in the Data column, then select Delete Anchor. Then type “n” in the Data column to input the nozzle. The Nozzle dialog will be shown. Input the nozzle data in the dialog. The Layout window looks like the following:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example 1: Nozzle on a cylindrical vessel								
2	10	From							Nozzle
3	20			20"		P11	26	1	
4	30	Bend		22"		P11	26	1	
5	40		22"			P11	26	1	
6	10	Location							Flange

The graphics is shown next.

Nozzle



Nozzle Stiffnesses

The three stiffnesses computed can be viewed using the List command (Ctrl+L in the Layout window). In the List dialog, select Nozzle Stiffnesses. The following window is displayed.

#	Node	Flat Bot.	Axial (lb/inch)	Circumferential (in-lb/deg)	Longitudinal (in-lb/deg)
1	10	No	12534	3.549E+5	6.980E+5

The nozzle/vessel data may be edited here (double click to edit).

Example 2: Nozzle on a flat-bottom storage tank

Assume the following data:

Vessel OD = 1800 in.

Vessel Thickness = 1.0 in.

Nozzle OD = 26 in.

Nozzle Thickness = 0.5 in.

L = 36 in.

Elastic modulus of the vessel material = 28×10^6 psi.

No reinforcing pad on the vessel.

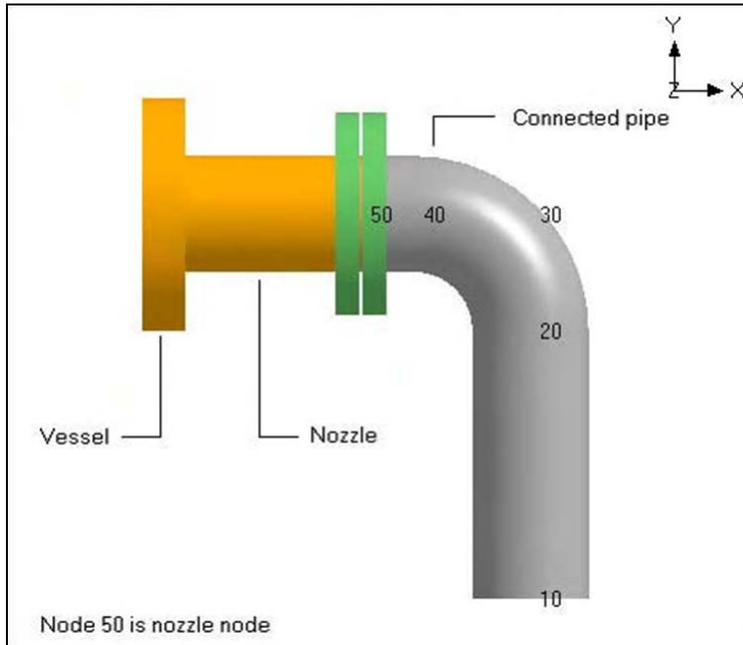
Create the piping till the nozzle node. At the nozzle node, enter a Nozzle by typing “n” and provide the required data. Check the Flat-bottom tank checkbox.

The Layout window looks like the following.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Example 2: Nozzle on a flat-bottom tank								
2	10	From							
3	20			60		P11	26	1	
4	30	Bend		26		P11	26	1	
5	40		-26			P11	26	1	
6	50		-12			P11	26	1	Nozzle
7	50	Location							Flange

Nozzle

The graphics is shown next.

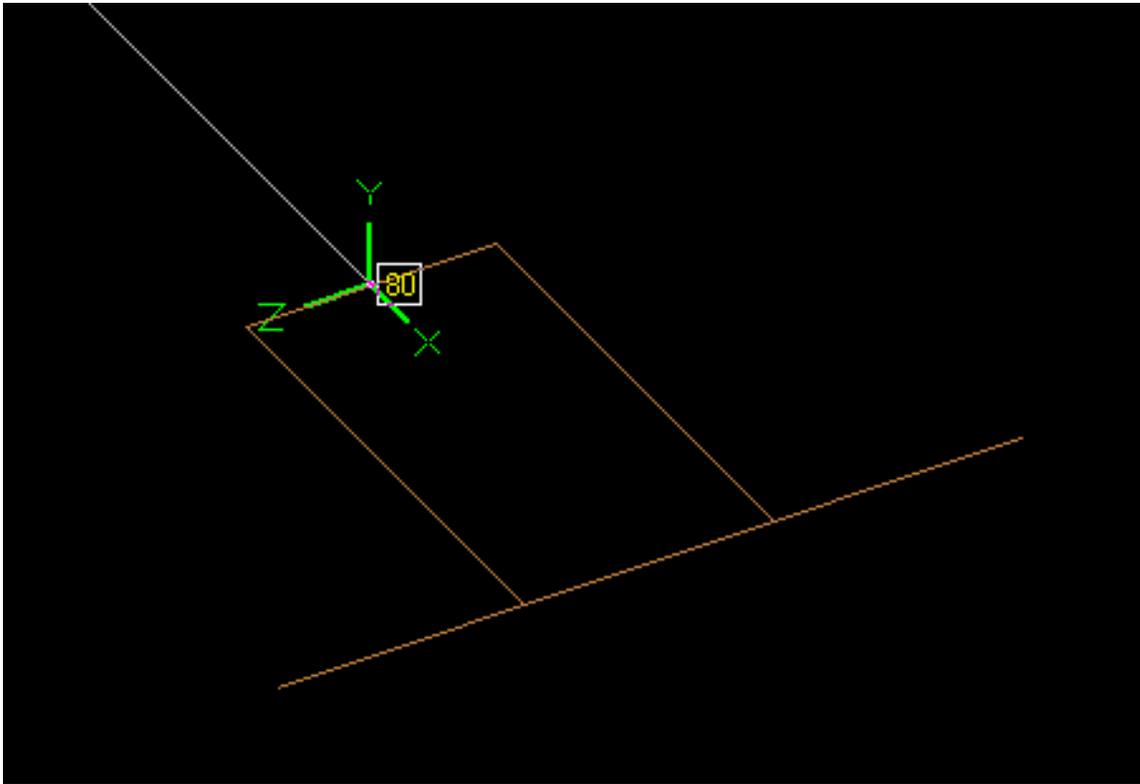


The three nozzle stiffnesses computed can be viewed as before by using the List command (Ctrl+L) and selecting Nozzle Stiffnesses.

LCS (Local Coordinate System) is now shown for Nozzle element. It can be accessed a few ways:

1. List (Ctrl+L) > Nozzles > View menu > Show LCS, -OR-
2. List (Ctrl+L) > Nozzles > Mouse Right Click on the listed Nozzle > Show LCS, -OR-
3. Results window > Support Loads > Other Support Loads > Nozzles > View menu > Show LCS, -OR-
4. Results window > Support Loads > Other Support Loads > Nozzles > Mouse Right Click > Show LCS.

Nozzle



Pump

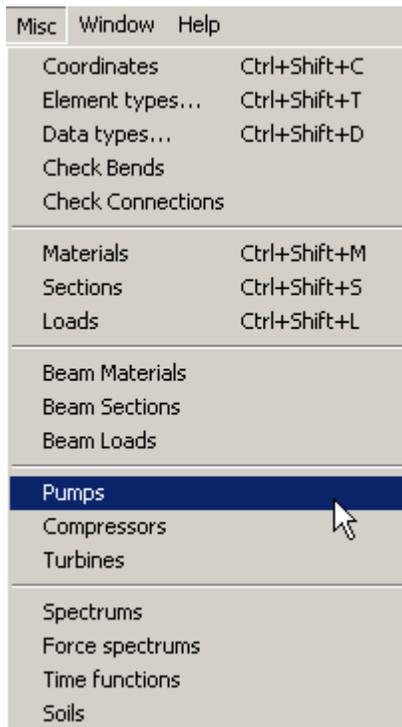
Pumps, compressors and turbines in CAEPIPE, referred to as rotating equipment, are each governed by an industry publication — API (American Petroleum Institute) publishes an API 610 for pumps and an API 617 for compressors while NEMA (National Electrical Manufacturers Association) publishes the NEMA SM-23 for turbines. These publications provide guidelines for evaluating nozzles connected to equipment among other technical information including the items relevant to piping stress analysis – criteria for piping design and a table of allowable loads.

Modeling the equipment is straightforward since it is assumed rigid (relative to connected piping) and modeled only through its end points (connection nozzles).

1. In your model, anchor all of the nozzles (on the equipment) that need to be included in the analysis.
2. Specify these anchored nodes during the respective equipment definition via Misc. menu > Pumps/Compressors/Turbines in the Layout window.

CAEPIPE does not require you to model all of the nozzles nor their connected piping. For example, you may model simply one inlet nozzle of a pump with its piping. Or, you may model one pump with both nozzles (with no connected piping) and impose external forces on them (if you have that data). Further, there is no need to connect the two anchors of the equipment with a rigid massless element like required in some archaic methods. A flange and an anchor may coexist.

A pump is input by selecting “Pumps” from the Misc menu in the Layout or List window. CAEPIPE produces an API 610 pump compliance report after analysis. You may model two types of pumps: Horizontal or Vertical inline. *See Appendix B (API 610, for Pumps) for related information.*



Pump

#	Description	Vert. Inline	Suction		Discharge		Shaft axis direction			Center of pump		
			Node	Loc	Node	Loc	X comp	Y comp	Z comp	X (ft'in')	Y (ft'in')	Z (ft'in')
1												

Once you see the Pump List window (as shown above), double click on an empty row for a new Pump definition dialog that allows you to enter the required information.

Pump # 1

Description: Horizontal Vertical inline

Suction Node: Location: Top Side End

Discharge Node: Location: Top Side End

Shaft axis direction:

X comp: Y comp: Z comp:

Location of the center of pump:

X: Y: Z: (ft'in')

Type a short description to identify the pump in Description. You must designate the pump nozzles as anchors, and the shaft axis must be in the horizontal plane. The nozzle locations (top, side or end) should be specified for suction and discharge nodes.

See section on specifying a Direction for information on X comp/Y comp/Z comp.

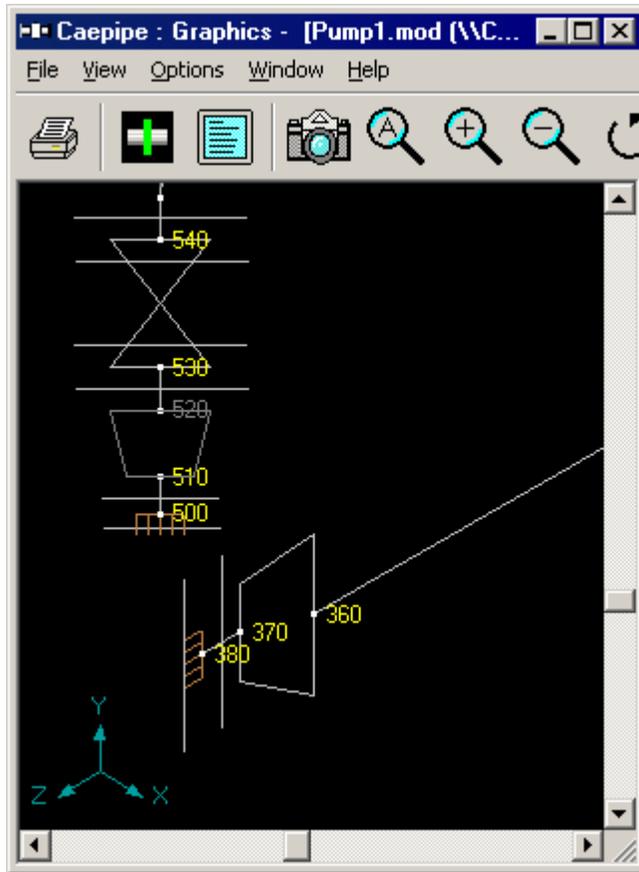
For horizontal pumps, you must enter coordinates for the center of the pump with respect to global origin. For pumps with two support pedestals, API 610 defines the center by the “intersection of the pump shaft centerline and a vertical plane passing through the center of the two support pedestals.” For pumps with four support pedestals, the center is defined by the “pump shaft centerline and a vertical plane passing midway between the four pedestals.” See Appendix B for illustrative figures.

You might find it helpful to first model the nozzles as anchors. In some situations, you might not have the discharge or suction side piping. In that case, here is how you can fix the location of the other side. Look up the coordinates of the nozzle (anchor) you have already modeled.

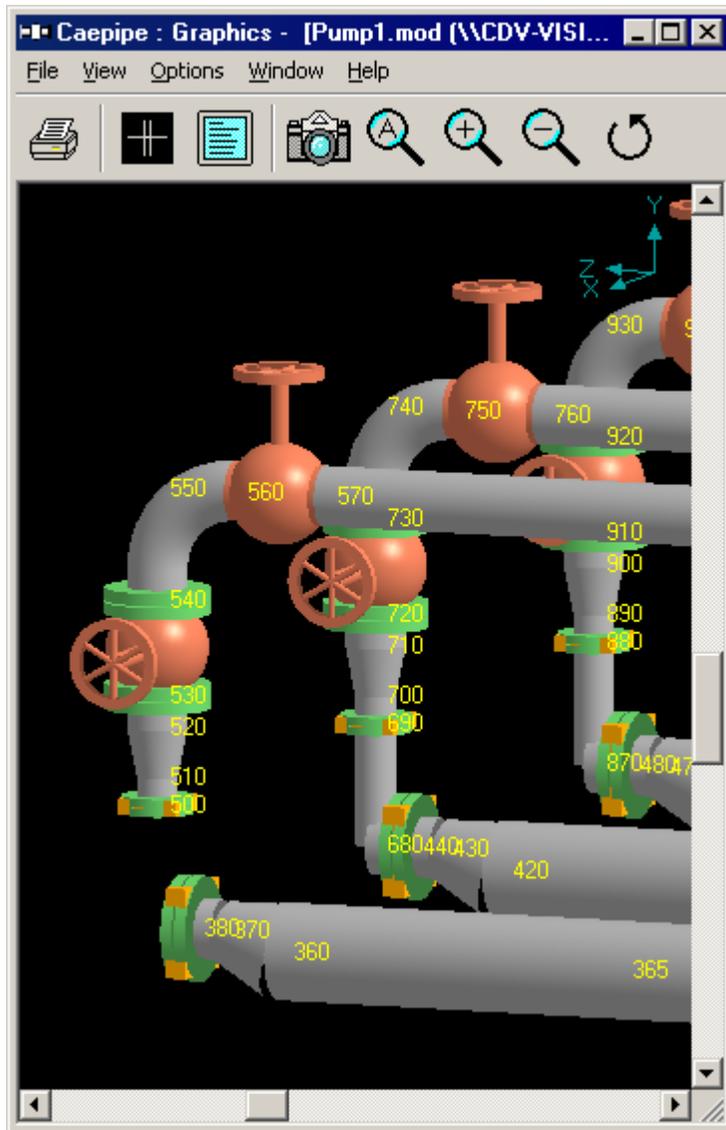
Pump

Then, use the coordinates command (menu Misc > Coordinates) to note the (X, Y, Z) coordinates of this pump nozzle node number. Using its coordinates, you can now arrive at the required coordinates of the other side's nozzle and the center of the pump.

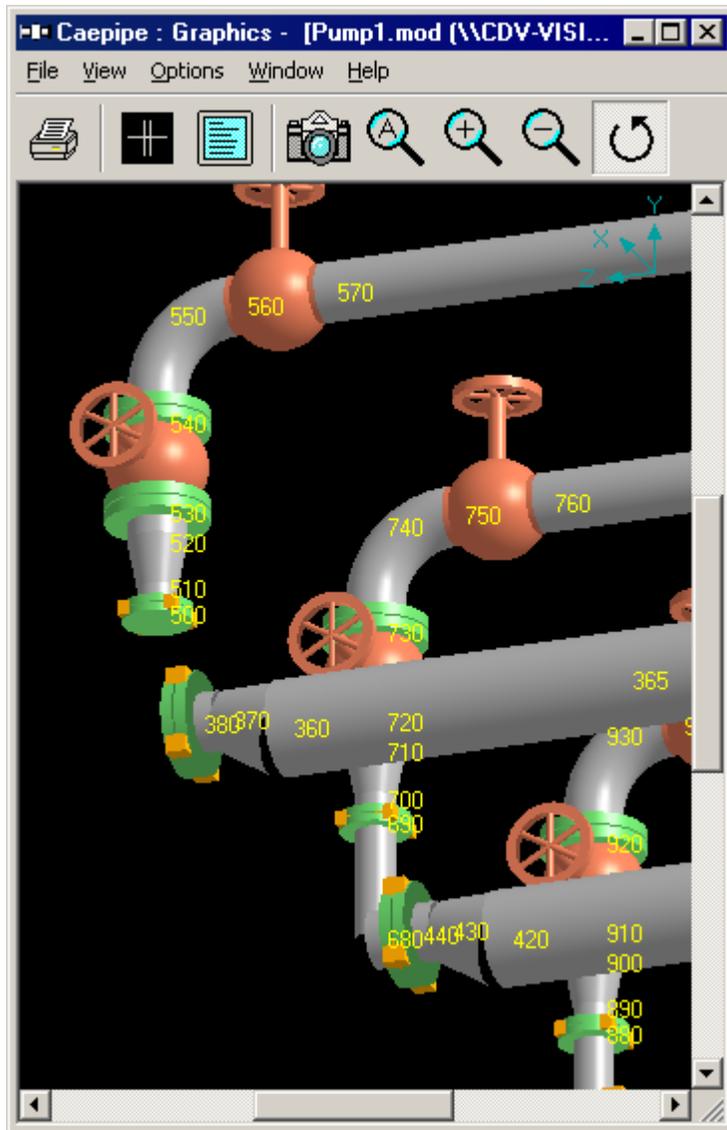
Example



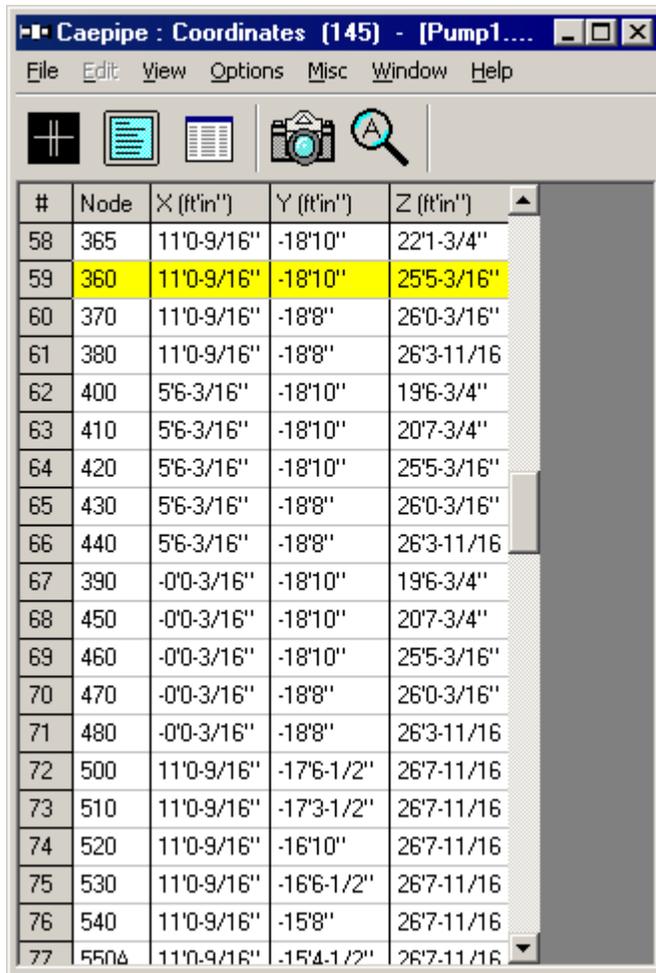
Pump



Pump



Pump



#	Node	X (ft'in')	Y (ft'in')	Z (ft'in')
58	365	11'0-9/16"	-18'10"	22'1-3/4"
59	360	11'0-9/16"	-18'10"	25'5-3/16"
60	370	11'0-9/16"	-18'8"	26'0-3/16"
61	380	11'0-9/16"	-18'8"	26'3-11/16"
62	400	5'6-3/16"	-18'10"	19'6-3/4"
63	410	5'6-3/16"	-18'10"	20'7-3/4"
64	420	5'6-3/16"	-18'10"	25'5-3/16"
65	430	5'6-3/16"	-18'8"	26'0-3/16"
66	440	5'6-3/16"	-18'8"	26'3-11/16"
67	390	-0'0-3/16"	-18'10"	19'6-3/4"
68	450	-0'0-3/16"	-18'10"	20'7-3/4"
69	460	-0'0-3/16"	-18'10"	25'5-3/16"
70	470	-0'0-3/16"	-18'8"	26'0-3/16"
71	480	-0'0-3/16"	-18'8"	26'3-11/16"
72	500	11'0-9/16"	-17'6-1/2"	26'7-11/16"
73	510	11'0-9/16"	-17'3-1/2"	26'7-11/16"
74	520	11'0-9/16"	-16'10"	26'7-11/16"
75	530	11'0-9/16"	-16'6-1/2"	26'7-11/16"
76	540	11'0-9/16"	-15'8"	26'7-11/16"
77	550A	11'0-9/16"	-15'4-1/2"	26'7-11/16"

In the above image, suction side piping ends at node 380. Discharge piping starts at node 500. In the Pump definition dialog (shown next), you can see that the center of the pump is just behind the suction nozzle (node 380) coordinates. The reducer between nodes 370 and 380 is a vertically offset eccentric reducer; hence the graphics at the reducer shows a break.

Pump

The screenshot shows the 'Pump # 1' dialog box with the following settings:

- Description: #1 Pump
- Orientation: Horizontal, Vertical inline
- Suction Node: 380, Location: Top, Side, End
- Discharge Node: 500, Location: Top, Side, End
- Shaft axis direction: X comp (empty), Y comp (empty), Z comp: 1.000
- Location of the center of pump: X: 11'0-3/4", Y: -18'8", Z: 26'7-11/16 (ft'in')
- Buttons: OK, Cancel

A simpler example is when you do not have piping on both sides of the pump. Consider the network below consisting of two pipe segments connected by a pump.

(suction side) 10-20-30-...-90-100 → PUMP ← 200-210-...-280-290-300 (discharge side)

The suction side of the pump ends at node 100. The discharge side begins at node 200. Make nodes 100 and 200 as anchors so that equipment loads can be calculated. A similar method applies to turbines and compressors too.

A different dialog is shown for vertical in-line pumps. Only Description, Suction and Discharge nodes are required.

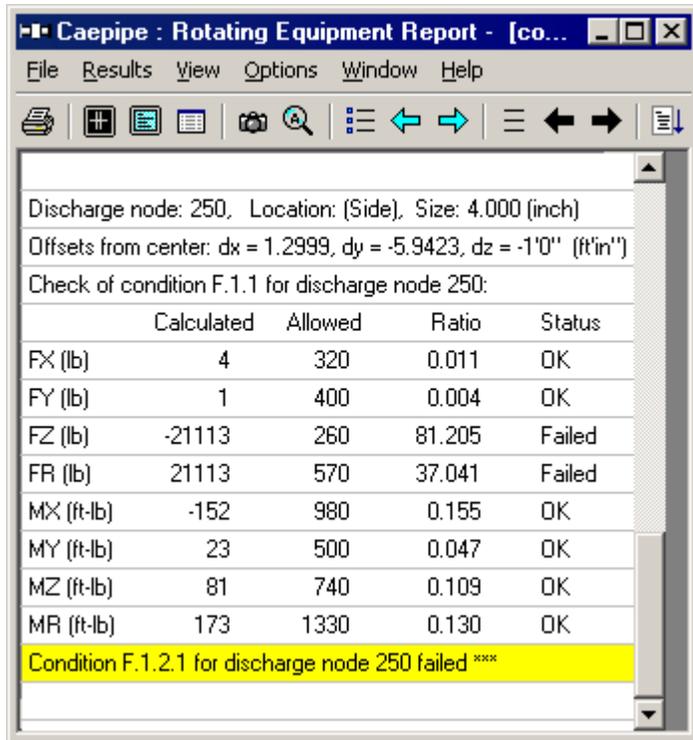
The screenshot shows the 'Pump # 1' dialog box with the following settings:

- Description: #1 Pump
- Orientation: Horizontal, Vertical inline
- Suction Node: 380
- Discharge Node: 500
- Buttons: OK, Cancel

Pump

API 610 REPORT

Upon analysis, you will see CAEPIPE produce API 610 reports under “Rotating Equipment Reports” in Results.



Caepipe : Rotating Equipment Report - [co...]

File Results View Options Window Help

Discharge node: 250, Location: (Side), Size: 4.000 (inch)
Offsets from center: dx = 1.2999, dy = -5.9423, dz = -1'0" (ft'in')

Check of condition F.1.1 for discharge node 250:

	Calculated	Allowed	Ratio	Status
FX (lb)	4	320	0.011	OK
FY (lb)	1	400	0.004	OK
FZ (lb)	-21113	260	81.205	Failed
FR (lb)	21113	570	37.041	Failed
MX (ft-lb)	-152	980	0.155	OK
MY (ft-lb)	23	500	0.047	OK
MZ (ft-lb)	81	740	0.109	OK
MR (ft-lb)	173	1330	0.130	OK

Condition F.1.2.1 for discharge node 250 failed ***

In addition to the input details, for a specific load case, the calculated forces and moments (on the nozzle and those at the center of the pump), API allowables, ratios (of calculated to allowable) and status for all of them are reported.

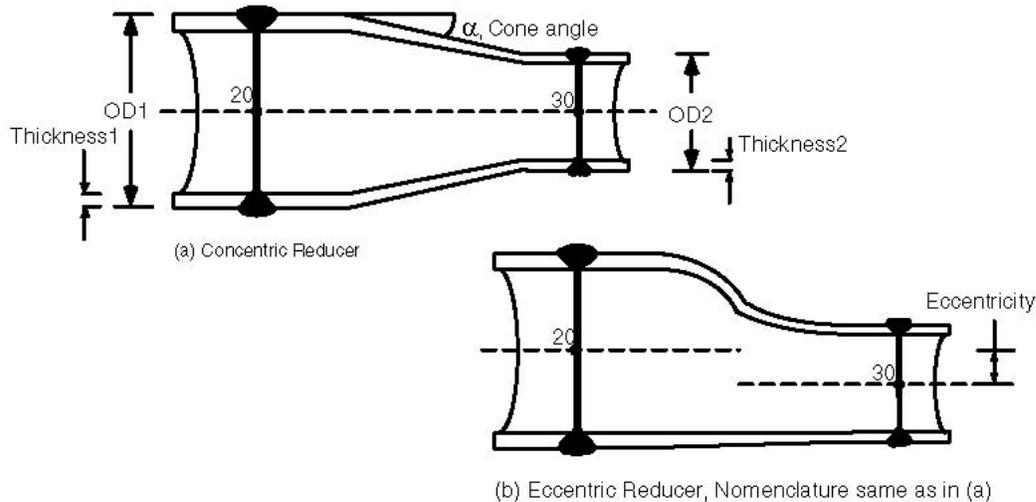
When you see “Failed” entries in this report, you will need to examine the cause of the high force or moment for that line item. Generally, the high numbers come from the expansion load but may well come from the weight load. You must reduce these excessive forces and moments by making the system or intersections more flexible before this pump can become compliant.

Note: If you have input multiple temperatures and pressures, corresponding reports for additional operating load cases will be shown. Use the black right arrow key to see them.

Reducer

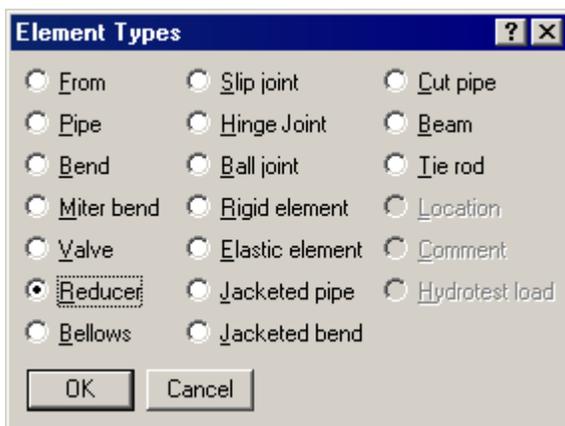
Use a reducer to join a larger pipe to a small pipe to meet hydraulic flow requirements. A reducer is Concentric when the axes at the reducer ends are collinear; Eccentric, when they are not. Use an eccentric reducer only when necessary to keep the top or bottom of the line level.

In the figures shown below, observe that the two ends of the reducers are of different diameters. The larger end (at node 20) has the outside diameter and thickness as OD1 and Thickness 1 with the smaller end (at node 30) having OD2 and Thickness 2. In case of the eccentric reducer, the eccentricity as shown is between the two axes of the ends of the reducer. The cone angle, α , is also shown in the following figure.



An eccentric reducer's eccentricity is modeled by a change in offsets of the "To" node (node 30 in Figure (b) above). Eccentricity is $(ID1-ID2) / 2$. See example 2 later in this topic.

A reducer (concentric or eccentric) is input by typing "re" in the Type column or selecting "Reducer" from the Element Types dialog.



Reducer

The Reducer dialog is shown.

Reducer from 20 to 30

OD1 8.625 Thk1 0.5 (inch) Section 1

OD2 Thk2 (inch) Section 2

Cone_angle (deg)

OK Cancel

OD1, OD2, Thk1, Thk2

These are the cross-sectional properties at the two ends of the reducer. OD stands for outside diameter and Thk stands for thickness. By default, OD1 and Thk1 contain preceding section's outside diameter and thickness, but different values may be typed here. The Section1 and Section2 buttons can be used to quickly input OD and Thk values from previously defined sections.

Cone angle

Shown in Figure (a), it is used to calculate SIF at the ends of the reducer for certain piping codes (B31.1, ASME Section III Class 2, RCC-M, Swedish and Norwegian). For these codes, if the cone angle is left blank, the maximum value of the SIF (2.0) is used. For all other piping codes in CAEPIPE, the cone angle is not used.

SIF Calculation

For B31.1, ASME Section III Class 2, EN 13480, RCC-M, Swedish and Norwegian, as mentioned above, the cone angle (if input) is used to calculate the SIF. If the cone angle is not input, the maximum value of the SIF (2.0) is used.

For Swedish and Norwegian piping codes, additional input is required which affects the calculation of SIF.

Reducer from 30 to 40

OD1 8.625 Thk1 0.5 (inch) Section 1

OD2 6.625 Thk2 0.28 (inch) Section 2

Cone_angle 20 (deg)

Knuckles Delta (inch)

OK Cancel

Knuckles

If the reducer is with knuckles check this box.

Reducer

Delta

If the reducer is without knuckles, specify delta, which is the mismatch (difference in mean radii across the weld at the smaller end of the reducer). If the reducer is with knuckles, delta is not used.

For other codes, if the code is not specific about a reducer's SIF, then a value of 1.0 is used.

Weight, Stiffness and Stress Calculation

The properties such as weight of the reducer, stiffness, contents weight and insulation weight are based on the average diameter (of OD1 and OD2) and average thickness (of Thk1 and Thk2).

The stresses at each end, however, are calculated using the actual dimensions at each end.

Example 1: Concentric Reducer

To model a concentric reducer as shown in Figure (a) with the following data: 8"x4" reducer, OD1=8.625", Thk1=0.322", OD2=4.5", Thk2=0.237".

Create two sections, 8"/STD and 4"/STD.

- ▶ The first node (10) is already defined. Press Enter to move to the next row.
- ▶ Complete pipe run till node 20: type 20 for Node, type 1 (ft.) for DX, enter material, 8" section and load names, press Enter.
- ▶ Input reducer: Type 30 for Node, press Tab to move to the Type field, type "Re" (to open the Reducer dialog box, note that CAEPIPE displays the preceding 8" section's properties for OD1 and Thk1).

The dialog box titled "Reducer from 20 to 30" has the following fields and buttons:

- OD1: 8.625 (inch)
- Thk1: 0.5 (inch)
- Section 1 button
- OD2: (empty) (inch)
- Thk2: (empty) (inch)
- Section 2 button
- Cone angle: (empty) (deg)
- OK button
- Cancel button

Press the "Section 2" button to select the section at "To" node.

Name	Nominal Diameter	Sch	OD (inch)	Thk (inch)
8	8"	80	8.625	0.5
6	6"	STD	6.625	0.28

Buttons: OK, Cancel

Highlight the 4" section and press OK. 4.5" for OD2 and 0.237" for Thk2 will be entered in the Reducer dialog.

Reducer

Reducer from 20 to 30
✕

OD1 Thk1 (inch)

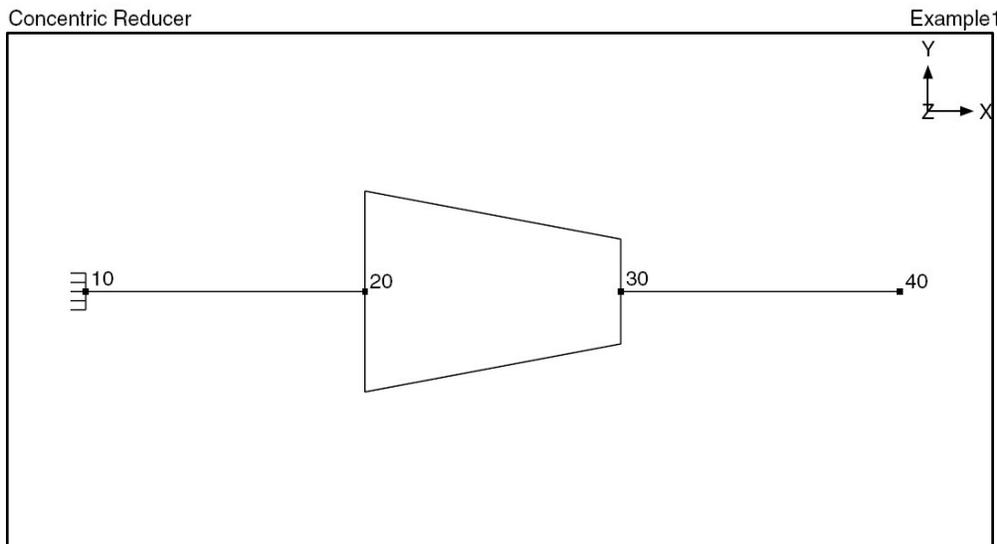
OD2 Thk2 (inch)

Cone angle (deg)

Press OK, type 11" for DX (reducer's length) and press Enter. Now you are asked if you want to change section, press Yes. "Select Section" dialog will be shown. Highlight the 4" section and press Enter. Press Enter again on Layout to move to the next row.

Type 40 for Node, 1 (ft.) for DX, press Enter.

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Concentric Reducer								
2	10	From							Anchor
3	20		1'0"			A53	8	1	
4	30	Reducer	0'11"		6'0"	A53	4	1	
5	40		1'0"		6'0"	A53	4	1	
6									



Example 2: Eccentric Reducer

To model an eccentric reducer (as shown in Figure (b) earlier in the topic) with the following data: 8"x6" reducer, OD1=8.625", Thk1=0.322", OD2=6.625", Thk2=0.28", eccentricity = $(ID1-ID2) / 2 = 0.958$ " which is modeled as change in elevation.

Create two sections, 8"/STD and 6"/STD.

- ▶ The first node (10) is already defined. Press Enter to move to the next row.

Reducer

- ▶ Complete pipe run till node 20: type 20 for Node, type 12” for DX, enter material, 8” section and load names, press Enter.
- ▶ Input reducer: Type 30 for Node, press Tab to move to the Type field, type “Re” (to open the Reducer dialog box, note that the preceding 8” section properties are already displayed for OD1 and Thk1).

Reducer from 20 to 30

OD1 8.625 Thk1 0.5 (inch) Section 1

OD2 Thk2 (inch) Section 2

Cone angle (deg)

OK Cancel

Press the “Section 2” button to select the section at “To” node.

Select Section 2

Name	Nominal Diameter	Sch	OD (inch)	Thk (inch)
8	8"	80	8.625	0.5
6	6"	STD	6.625	0.28

OK Cancel

Highlight the 6” section and press OK.

6.625” for OD2 and 0.28” for Thk2 will be entered in the Reducer dialog.

Reducer from 20 to 30

OD1 8.625 Thk1 0.5 (inch) Section 1

OD2 6.625 Thk2 0.28 (inch) Section 2

Cone angle (deg)

OK Cancel

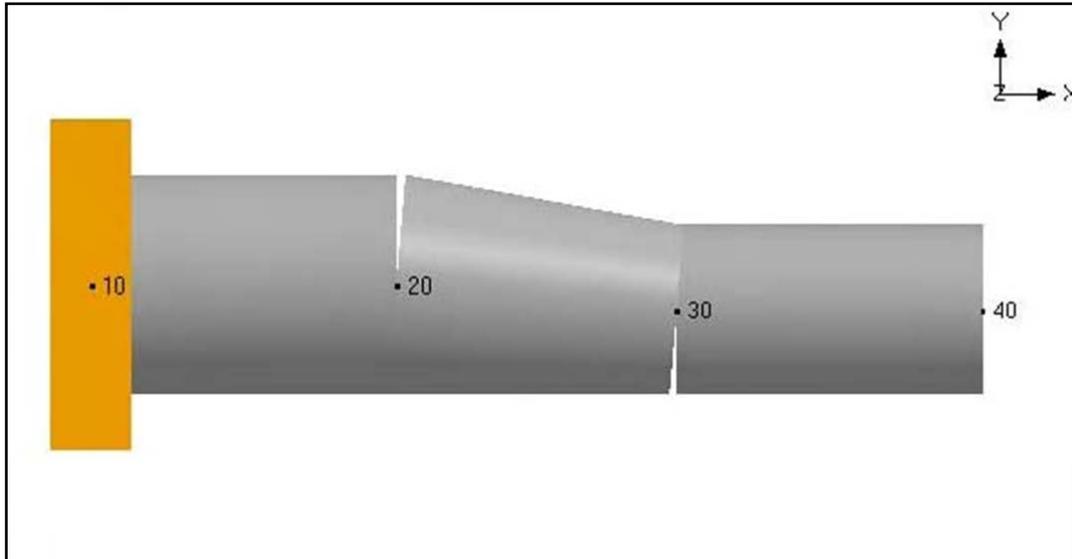
Press OK, type 11” for DX (reducer’s length along X axis), -0.958” for DY (this is the eccentricity), then press Enter. Now you are asked if you want to change section, press Yes. “Select Section” dialog will be shown. Highlight the 6” section and press Enter. Press Enter again on Layout to move to the next row.

- ▶ Type 40 for Node, 12” for DX, press Enter.

Reducer

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Loac	Data
1	Title = Eccentric Reducer								
2	10	From							Anchor
3	20		12			A53	8	1	
4	30	Reducer	11	-0.958		A53	6	1	
5	40		12			A53	6	1	
6									

The graphics is shown below:



Example 3: Jacketed Reducer

A jacketed reducer may be modeled in the following manner: Calculate the averages of OD1 and OD2, and of Thk1 and Thk2 of the reducer. Create a new pipe section with these averages as OD and Thickness. Insert a pipe at that location which has the calculated average OD and thickness. You may have to input an SIF (at the two ends) using code guidelines for a reducer.

Alternate Method: In a jacketed piping system (e.g., 10-20 is a JPIPE, model a reducer next as you would normally do in a non-jacketed system, i.e., 20-30 is a reducer, followed by a JPIPE between 30-40). Then, connect the jacket nodes (nodes with the suffix J) together (on either side of a reducer) with an outside reducer (shown below) or a JPIPE (jacket pipe). In other words, on new rows, connect 20J to 30J with a reducer or a JPIPE.

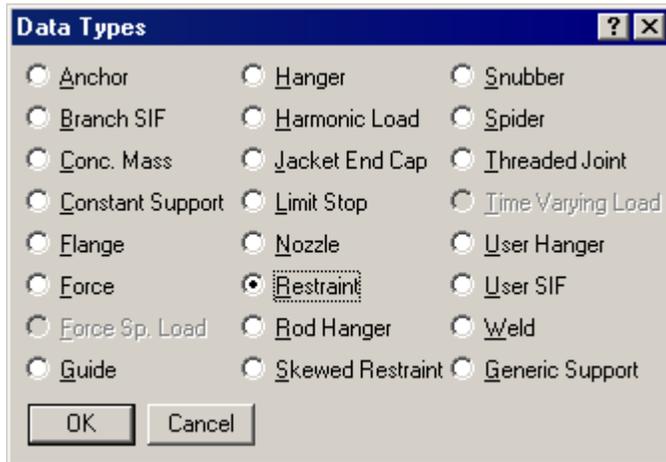
Reducer

Caepipe : Layout (11) - [Sample.mod [C:\CAEPIPE\681LM]]									
File Edit View Options Loads Misc Window Help									
#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Jpipe	2'0"			A53	6	1	
4	First reducer is on the core pipe								
5	30	Reducer	0'6"			A53	4	1	
6	40	Jpipe	2'0"			A53	4	1	
7	Second reducer is on the jacket pipe. Note use of jacket node suffix J								
8	20J	From							
9	30J	Reducer				A53	10	1	
10	Continue from the last jacket pipe, i.e., from node 40								
11	40	From							Jacket endcap
12									

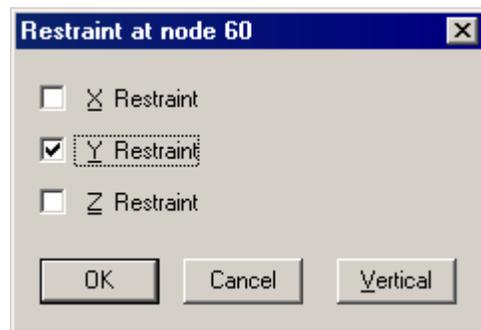
Restraint

This support type is a convenient way to specify a translational two-way rigid restraint in the global X, Y and Z directions. This is most commonly used to specify a rigid 2-way vertical restraint.

A restraint is input by typing “re” in the Data column or selecting “Restraint” from the Data Types dialog. Alternately, simply typing “X” or “Y” or “Z” in the data type field inputs a restraint in the respective direction and moves the cursor to the next row.



The Restraint dialog is shown.



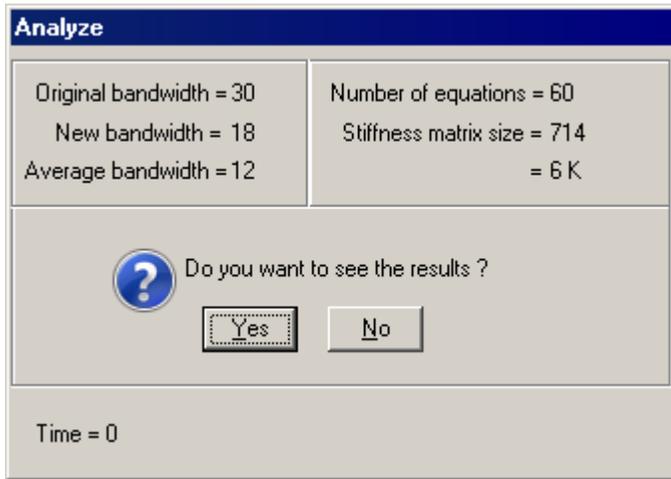
Use the check boxes to apply the restraint in a particular direction (both ways). Click on the vertical button for a rigid vertical restraint. All three directions may be checked too.

Rigid restraint has a stiffness of 1×10^{12} (lb./in.).

Results Window

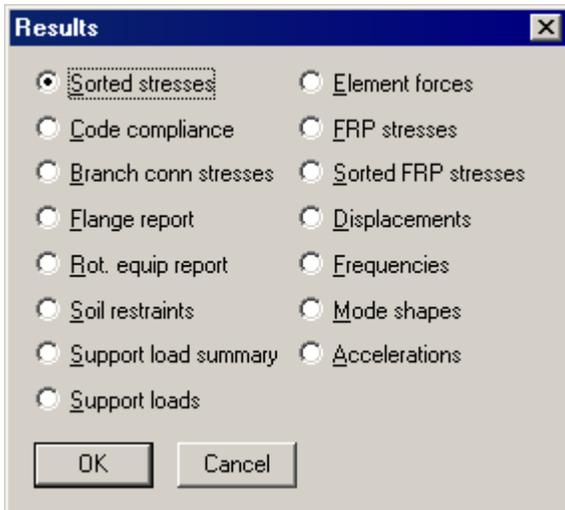
CAEPIPE opens the Results window, one among the four independent windows, either after analysis of a model or after you open a results file (.res) from the File menu in the Main, Layout or Results windows.

On successful completion of model analysis, CAEPIPE asks you whether you want to see results. Click on Yes to continue.



When you do so, the Results window opens, in which you can view every computed result for the piping model. This window displays results in an organized manner conducive to easy understanding of the results. The other windows (Layout and List) may be opened from here in read-only mode for better comprehension, i.e., any detail in these windows may be viewed but cannot be modified. To modify the input data, you must open the Layout window in input mode (by selecting menu File > Input).

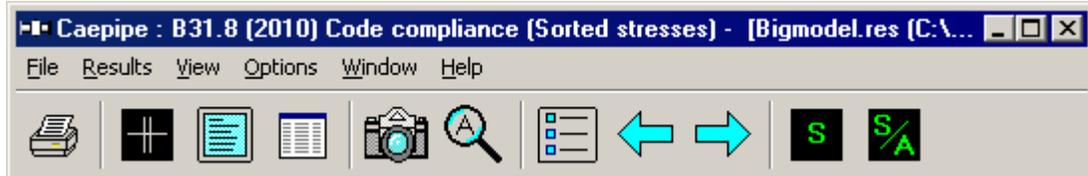
A dialog listing the different results for a model is shown below. Select an item to see it.



Results Window

While viewing results, it is helpful to be aware of the following:

- The name of the results item and the load case (if applicable) are always shown in the title bar of the Results window. See image below.



- Synchronized highlight: When you move the highlight to a node number (or an element) in the Results window, the highlight in the Graphics window is automatically synchronized to the corresponding node number (or element).
- One key navigation: By merely pressing the Tab key, you can move forward through the different results one at a time (Sorted stresses, Code Compliance, Support Loads, Element forces, etc.). Move backward by using Shift+Tab, or by clicking on the cyan colored arrows (left and right). In this window, the display order of results is circular, i.e., if you press Tab from Sorted Stresses, you will see Code compliance next, or if you press Shift+Tab, you will see Time History results (or whichever is the last item on the list of computed results).
- Changing Toolbar and Menus: As you move forward or backward through the results, the toolbar changes with the displayed result. Also, the menu commands in Results and View menus change. They will contain commands relevant to the displayed result.
- Simultaneous graphics: You can view graphics simultaneously with results (and input).

Caepipe : ASME Class 2 (1992) Code compliance (Sorted stresses) - [All.res (F:\UMan\LayoutWindow)]																				
File Results View Options Window Help																				
< Menubar																				
< Toolbar																				
Displays Previous or Next results item																				
#	Sustained (8)				Expansion (10)			Expansion (11)			Occasional (9)			Settlement (10a)						
	Node	SL (psi)	1.5SH (psi)	SL 1.5SH	Node	SE (psi)	SA (psi)	SE SA	Node	SL+SE (psi)	SH+SA (psi)	SL+SE SH+SA	Node	SL+SO (psi)	1.8SH (psi)	SL+SO 1.8SH	Node	SS (psi)	3.0SC (psi)	SS 3.0SC
7	124	4090	25950	0.16	220A	45712	29325	1.56	220A	48743	46625	1.05	40B	21508	31140	0.69	220A	7444	60000	0.12
8	125	4090	25950	0.16	220B	42917	29325	1.46	40A	46679	46625	1.00	230	17553	31140	0.56	40A	7345	60000	0.12
9	240	3988	25950	0.15	40A	42824	29325	1.46	220B	46593	46625	1.00	40A	16960	31140	0.54	240	5469	60000	0.09
10	25	3966	25950	0.15	13	42003	29325	1.43	13	43935	46625	0.94	51	14295	31140	0.46	50	5185	60000	0.09
11	51	3923	25950	0.15	20A	40100	29325	1.37	20A	42020	46625	0.90	220B	9635	31140	0.31	51	4648	60000	0.08
12	15	3775	25950	0.15	16					39312	46625	0.84	25	8984	31140	0.29	230	4151	60000	0.07
13	220B	3675	25950	0.14	247D					38811	46625	0.83	120	6550	31140	0.21	55	2022	60000	0.03
14	55	3368	25950	0.13	25					35596	46625	0.76	110	6168	31140	0.20	60	1131	60000	0.02
15	40A	3289	25950	0.13	40B					34109	46625	0.73	15	5101	31140	0.16	10	1063	60000	0.02
16	129	3251	25950	0.13	10					32586	46625	0.70	125	5070	31140	0.16	70	957	60000	0.02
17	122	3242	25950	0.12	230					30940	46625	0.66	124	5070	31140	0.16	80	956	60000	0.02
18	128	3181	25950	0.12	20B					29810	46625	0.64	20A	4262	31140	0.14	15	828	60000	0.01
19	80	3140	25950	0.12	246					25635	46625	0.55	20B	4055	31140	0.13	20B	672	60000	0.01
20	60	3078	25950	0.12	210	20310	29325	0.69	51	21081	46625	0.45	55	4050	31140	0.13	20A	646	60000	0.01
21	123	3053	25950	0.12	80	17322	29325	0.59	246	20782	46625	0.45	13	3983	31140	0.13	16	643	60000	0.01
22	70	3042	25950	0.12	51	17158	29325	0.59	80	20462	46625	0.44	60	3790	31140	0.12	13	638	60000	0.01
23	220A	3031	25950	0.12	70	14860	29325	0.51	70	17902	46625	0.38	245	3541	31140	0.11	245	521	60000	0.01
24	126	3016	25950	0.12	247B	12800	29325	0.44	246J	13890	46625	0.30	247D	3510	31140	0.11	245J	163	60000	0.00

Results Window

- For a few items, you will notice left and right black arrows. By clicking on these arrow buttons (or pressing the corresponding arrow keys on the keyboard), you can change the load case. Similarly, in some other results, you will notice left and right white arrows. By clicking on these arrow buttons (or pressing the corresponding arrow keys on the keyboard with the CTRL key), you can display results for other related items (for example, while viewing Support loads, you can change over to other supports [like limit stops, hangers, restraints, etc.]).



Depending on the model configuration, you might see more choices within a topic. For example, you might see the Other Forces dialog from Pipe Forces results screen. You might also see the Other Supports dialog from Support Loads results.

Each results item has a context menu as shown in the sorted stresses window (see previous image). Right-click in the results window to see what the context menu offers.

Sorted Stresses

For sorted stresses, you are shown Show stresses, Show stress ratios, Thresholds and Hide Allowables (see image on previous page). The resulting actions of the first three commands apply to the Graphics window and the last one applies to the Results window. CAEPIPE removes four columns (1.5SH, SA, SH+SA and 3.0SC) when you Hide Allowables thereby allowing you to reduce the information presented. See next figure (in which “Allowables” are hidden).

#	Sustained (8)			Expansion (10)			Expansion (11)			Occasional (9)			Settlement (10a)		
	Node	SL (psi)	SL 1.5SH	Node	SE (psi)	SE SA	Node	SL+SE (psi)	SL+SE SH+SA	Node	SL+SO (psi)	SL+SO 1.8SH	Node	SS (psi)	SS 3.0SC
7	124	4090	0.16	220A	45712	1.56	220A	48743	1.05	122	24110	0.77	220A	7444	0.12
8	125	4090	0.16	220B	42917	1.46	40A	46679	1.00	220A	23666	0.76	40A	7345	0.12
9	240	3988	0.15	40A	42824	1.46	220B	46593	1.00	123	23440	0.75	240	5469	0.09
10	25	3966	0.15	13	42003	1.43	13	43935	0.94	125	21602	0.69	50	5185	0.09
11	51	3923	0.15	20A	40100	1.37	20A	42020	0.90	124	21602	0.69	51	4648	0.08
12	15	3775	0.15	16	37514	1.28	16	39312	0.84	40B	21508	0.69	230	4151	0.07
13	220B	3675	0.14	247D	35831	1.22	247D	38811	0.83	230	17553	0.56	55	2022	0.03
14	55	3368	0.13	25	31542	1.08	25	35596	0.76	40A	16960	0.54	60	1131	0.02
15	40A	3289	0.13	40B	31066	1.06	40B	34109	0.73	51	14295	0.46	10	1063	0.02
16	129	3251	0.13	10	30238	1.03	10	32586	0.70	110	10652	0.34	70	957	0.02
17	122	3242	0.12	230	27996	0.95	230	30940	0.66	220B	9635	0.31	80	956	0.02
18	128	3181	0.12	20B	27863	0.95	20B	29810	0.64	25	8984	0.29	15	828	0.01
19	80	3140	0.12	246	20658	0.70	210	25635	0.55	15	5101	0.16	20B	672	0.01
20	60	3078	0.12	210	20310	0.69	51	21081	0.45	128	4481	0.14	20A	646	0.01
21	123	3053	0.12	80	17322	0.59	246	20782	0.45	20A	4262	0.14	16	643	0.01
22	70	3042	0.12	51	17158	0.59	80	20462	0.44	129	4058	0.13	13	638	0.01

Results Window

A few screenshots of Results window with the different results items are shown next.

Code Compliance

Here, CAEPIPE displays the stresses on an element-by-element basis. Sorted stresses screen (i.e., the previous results item) shows stresses at nodes sorted in the descending order of stress ratios. In this results item, CAEPIPE displays stresses for each element (highlight is on element 19, nodes 120 and 122).

#	Node	Press. Allow. (psi)	Sustained (8)			Expansion (10)			Expansion (11)			Occasional (9)			Settlement (10a)		
			SL (psi)	1.5SH (psi)	SL 1.5SH	SE (psi)	SA (psi)	SE SA	SL+SE (psi)	SH+SA (psi)	SL+SE SH+SA	SL+SO (psi)	1.8SH (psi)	SL+SO 1.8SH	SS	3.0SC	SS 3.0SC
10	40A 40B	500 2103	2716 4963	25950 25950	0.10 0.19	42824 31066	29325 29325	1.46 1.06	46679 34109	46625 46625	1.00 0.73	16960 21508	31140 31140	0.54 0.69	7345 7669	60000 60000	0.12 0.13
11	40B 50	500 2103	4566 2687	25950 25950	0.18 0.10	31063 48664	29325 29325	1.06 1.66	34106 51251	46625 46625	0.73 1.10	17301 31614	31140 31140	0.56 1.02	7669 5186	60000 60000	0.13 0.09
12	30 51	500 1514	5806 3923	25950 25950	0.22 0.15	85251 17158	29325 29325	2.91 0.59	93122 21081	46625 46625	2.00 0.45	24200 14254	31140 31140	0.78 0.46	11767 4648	60000 60000	0.20 0.08
13	51 55	500 1514	3923 3368	25950 25950	0.15 0.13	17158 6327	29325 29325	0.59 0.22	21081 9694	46625 46625	0.45 0.21	14295 4050	31140 31140	0.46 0.13	4648 2022	60000 60000	0.08 0.03
14	55 60	500 1514	3368 3078	25950 25950	0.13 0.12	6327 5896	29325 29325	0.22 0.20	9694 8974	46625 46625	0.21 0.19	4050 3790	31140 31140	0.13 0.12	2022 1131	60000 60000	0.03 0.02
15	70 80	500 1514	3042 3140	25950 25950	0.12 0.12	14860 17322	29325 29325	0.51 0.59	17902 20462	46625 46625	0.38 0.44	3204 3340	31140 31140	0.10 0.11	957 956	60000 60000	0.02 0.02
16	25 90	500 1514	3966 2977	25950 25950	0.15 0.11	28011 458	29325 29325	0.96 0.02	32708 3435	46625 46625	0.70 0.07	6709 3021	31140 31140	0.22 0.10	9265 28	60000 60000	0.15 0.00
17	100 110	500 1514	2971 3077	25950 25950	0.11 0.12	183 72	29325 29325	0.01 0.00	3153 3149	46625 46625	0.07 0.07	3000 3211	31140 31140	0.10 0.10	17 24	60000 60000	0.00 0.00
18	110 120	500	6034 5712	25950 25950	0.23 0.22	130 361	29325 29325	0.00 0.01	3249 3748	46625 46625	0.07 0.08	6168 6550	31140 31140	0.20 0.21	43 98	60000 60000	0.00 0.00
19	120 122	500 1514	4801 3242	25950 25950	0.19 0.12	137 61	29325 29325	0.00 0.00	3478 3303	46625 46625	0.07 0.07	5119 3398	31140 31140	0.16 0.11	37 10	60000 60000	0.00 0.00
20	123	500	3053	25950	0.12	41	29325	0.00	3094	46625	0.07	3231	31140	0.10	5	60000	0.00

Branch Connection Stresses

Branch connection stresses are available for only a few piping codes such as ASME Section III. You can change the load case by pressing the left or right black arrow key.

#	Node	Type	B1	Pressure stress (psi)	Run / Branch			Total Stress (psi)	Total Stress 1.5SH
					B2	Moment (ft-lb)	Stress (psi)		
1	35	Reinf tee	0.500	-2588	2.100	1537	1494	-554	-0.03
					2.100	555	540		

Results Window

Hanger Report

The screenshot shows the 'Caepipe : Hanger Report' window. The table contains the following data:

#	Node	No of	Type	Figure No.	Size	Spring rate (lb/inch)	Vert travel (inch)	Horz travel (inch)	Hot load (lb)	Cold load (lb)	Var (%)
19	158	1	Rod Hanger			Rigid					
20	306	1	Rod Hanger			Rigid					
21	312	1	Rod Hanger			Rigid					
22	338	1	Rod Hanger			Rigid					
23	346	1	Grinnell	B-268	10	260	0.547	0.034	1391	1534	10
24	380	1	User hanger			200	0.168	0.406	775	809	4
25	418	1	Rod Hanger			Rigid					
26	440	1	Rod Hanger			Rigid					
27	475	1	Grinnell	B-268	11	340	0.112	0.566	1668	1706	2
28	517	1	Rod Hanger			Rigid					
29	518	1	Rod Hanger			Rigid					
30	522	1	Rod Hanger			Rigid					
31	531	1	Grinnell	B-268	5	63	0.229	1.181	280	294	5

Flange Report

The screenshot shows the 'Caepipe : Flange Report' window. The table contains the following data:

#	Node	Pressure (psi)	Bending moment (ft-lb)	Axial force (lb)	Gasket diameter (inch)	Flange Pressure (psi)	Allowable Pressure (psi)	Flange Pressure Allowable
1	40B	200	20765	11002	6	6465	3000	2.155
2	40A	200	31333	15822	7.5	5097	5000	1.019

See the end of the “Flange” section for suggestions on how to reduce the ratio in the last column.

Rotating Equipment Reports

The rotating equipment reports are produced for all Operating load cases. The example model shown below has several cold spring operating load cases and hence you see the left and right black arrow keys.

The left and right white arrow keys advance (or go back) to the next (or previous) rotating equipment report.

Results Window

Discharge node: 250, Location: (Side), Size: 4.000 (inch)
 Offsets from center: dx = 1.2999, dy = -5.9423, dz = -1'0" (ft'in")
 Check of condition F.1.1 for discharge node 250:

	Calculated	Allowed	Ratio	Status
FX (lb)	4	320	0.011	OK
FY (lb)	1	400	0.004	OK
FZ (lb)	-21113	260	81.205	Failed
FR (lb)	21113	570	37.041	Failed
MX (ft-lb)	-152	980	0.155	OK
MY (ft-lb)	23	500	0.047	OK
MZ (ft-lb)	81	740	0.109	OK
MR (ft-lb)	173	1330	0.130	OK

Condition F.1.2.1 for discharge node 250 failed ****

Soil Restraints

Soil restraints are shown when you have buried piping in your model.

#	From	To	Axial		Transverse		Vertical Down		Vertical Up	
			Stiffness (lb/inch)	Max Load (lb)						
1	10	20	1989.6	1014.7	36425	18577	209098	106640	5601.6	2856.8
2	20	30	1533.8	782.3	691985	352913	518886	264632	590899	301359

Support Load Summary for 150+ Load Combinations

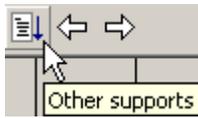
This gives you a summary of many combinations of different load cases by showing the algebraically minimum and maximum loads at each support. This table can be of use to a support designer (not all are shown below).

Results Window

Caepipe : Support load summary for anchor at node 10 - [complex1.res [\\CDV-VISIO...]									
File Results View Options Window Help									
Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)	Displacements (global)		
							X (inch)	Y (inch)	Z (inch)
Cold spring1+Settlement	-19755	4678	-2644	-1082	-8746	6836	0.000	0.000	0.000
Cold spring2+Settlement	-21765	2995	-3742	2427	-4196	4535	0.000	0.000	0.000
Cold spring3+Settlement	16248	1918	17258	4100	-99573	3063	0.000	0.000	0.000
Hydrotest	-328	2813	148	885	-1360	4286	0.000	0.000	0.000
Cold spring0+Wind	7104	2872	11109	1498	-68550	4366	0.000	0.000	0.000
Cold spring1+Wind	-19906	5633	-2950	-3273	-6597	8141	0.000	0.000	0.000
Cold spring2+Wind	-21916	3949	-4048	237	-2047	5840	0.000	0.000	0.000
Cold spring3+Wind	16097	2873	16952	1909	-97424	4368	0.000	0.000	0.000
Cold spring0+Seismic	8869	3162	12452	2539	-61037	4763	0.000	0.000	0.000
Cold spring0-Seismic	5342	2597	9641	481	-75763	3991	0.000	0.000	0.000
Cold spring1+Seismic	-18141	5923	-1606	-2231	916	8538	0.000	0.000	0.000
Cold spring1-Seismic	-21668	5358	-4418	-4289	-13811	7766	0.000	0.000	0.000
Cold spring2+Seismic	-20151	4240	-2704	1278	5466	6237	0.000	0.000	0.000
Cold spring2-Seismic	-23678	3675	-5516	-780	-9261	5465	0.000	0.000	0.000
Cold spring3+Seismic	17862	3163	18296	2951	-89911	4765	0.000	0.000	0.000
Cold spring3-Seismic	14335	2598	15484	893	-104638	3992	0.000	0.000	0.000
Cold spring0+Response	7934	3077	11783	2351	-63331	4646	0.000	0.000	0.000
Cold spring0-Response	6277	2682	10310	669	-73469	4107	0.000	0.000	0.000
Cold spring1+Response	-19076	5838	-2275	-2419	-1378	8421	0.000	0.000	0.000
Cold spring1-Response	-20733	5443	-3748	-4101	-11516	7882	0.000	0.000	0.000
Cold spring2+Response	-21085	4154	-3374	1090	3171	6120	0.000	0.000	0.000
Cold spring2-Response	-22743	3760	-4847	-592	-6967	5581	0.000	0.000	0.000
Cold spring3+Response	16927	3078	17626	2763	-92206	4648	0.000	0.000	0.000
Cold spring3-Response	15270	2683	16153	1081	-102344	4109	0.000	0.000	0.000
Cold spring0+Time history	7105	2879	11046	1511	-68395	4376	0.000	0.000	0.000
Cold spring1+Time history	-19905	5640	-3013	-3260	-6442	8151	0.000	0.000	0.000
Cold spring2+Time history	-21915	3957	-4111	250	-1893	5850	0.000	0.000	0.000
Cold spring3+Time history	16098	2880	16889	1922	-97270	4378	0.000	0.000	0.000
Cold spring0+Harmonic	7106	2879	11046	1510	-68400	4377	0.000	0.000	0.000
Cold spring0-Harmonic	7105	2879	11046	1510	-68401	4377	0.000	0.000	0.000
Cold spring1+Harmonic	-19904	5640	-3012	-3260	-6447	8152	0.000	0.000	0.000
Cold spring1-Harmonic	-19905	5640	-3012	-3260	-6448	8152	0.000	0.000	0.000
Cold spring2+Harmonic	-21914	3957	-4110	249	-1897	5851	0.000	0.000	0.000
Cold spring2-Harmonic	-21914	3957	-4110	249	-1898	5851	0.000	0.000	0.000
Cold spring3+Harmonic	16099	2881	16890	1922	-97274	4379	0.000	0.000	0.000
Cold spring3-Harmonic	16098	2881	16890	1922	-97275	4378	0.000	0.000	0.000
Maximum	17862	5923	18296	4100	5466	8538	0.000	0.000	0.000
Minimum	-23678	1917	-5516	-4289	-104638	3061	0.000	0.000	0.000

Results Window

The supports at which a load summary is available can be shown by clicking on the Other Supports button (immediately to the left of the left white arrow button). Select the support from the list of available supports.

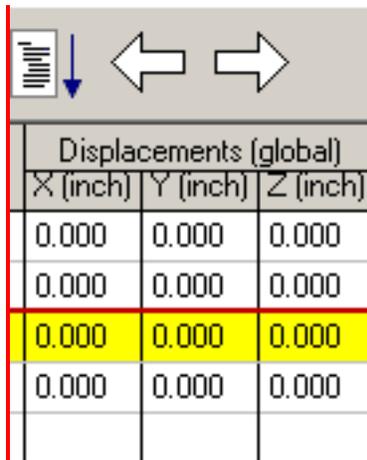


Node	Type
10	Anchor
50	Anchor
80	Anchor
30	Hanger

OK Cancel

Here, support load summaries are available for the four supports shown. Summaries may be included in the “Print to file” option (in addition to inclusion in formatted reports).

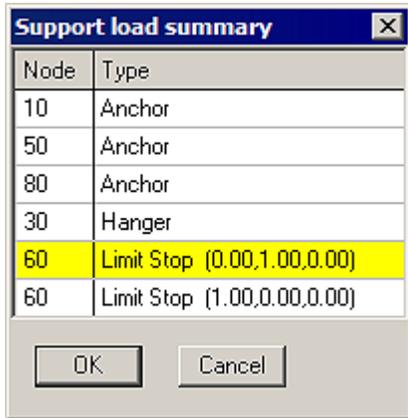
Support Load Summary includes Global displacements at each support for load cases and combinations.

A screenshot of a software interface showing a table titled "Displacements (global)". The table has three columns: X (inch), Y (inch), and Z (inch). The table contains five rows of data, all showing 0.000. The third row is highlighted in yellow. Above the table are navigation icons: a list icon, a downward arrow, and two horizontal arrows (left and right).

Displacements (global)		
X (inch)	Y (inch)	Z (inch)
0.000	0.000	0.000
0.000	0.000	0.000
0.000	0.000	0.000
0.000	0.000	0.000

To clearly show the direction of each limit stop among multiple at the same node, the direction is shown alongside the limit stop (see node 60 below).

Results Window

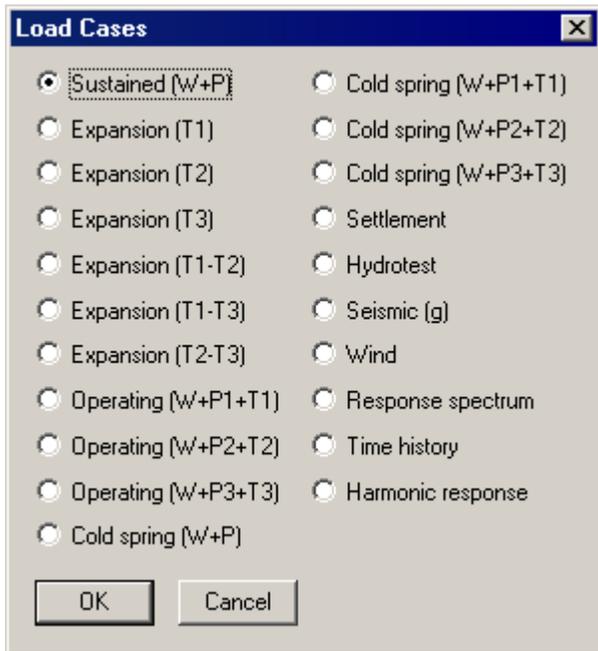


The dialog box titled "Support load summary" contains a table with two columns: "Node" and "Type". The table lists support types for nodes 10, 50, 80, 30, 60, and 60. The row for node 60 with type "Limit Stop (0.00,1.00,0.00)" is highlighted in yellow. Below the table are "OK" and "Cancel" buttons.

Node	Type
10	Anchor
50	Anchor
80	Anchor
30	Hanger
60	Limit Stop (0.00,1.00,0.00)
60	Limit Stop (1.00,0.00,0.00)

Support Loads

Support Loads are shown for all support types. For a support type, the support loads can be shown for all load cases by clicking on the black right/left arrow or simply press arrow keys on the keyboard. A list of load cases can be shown by clicking on the Load cases button.

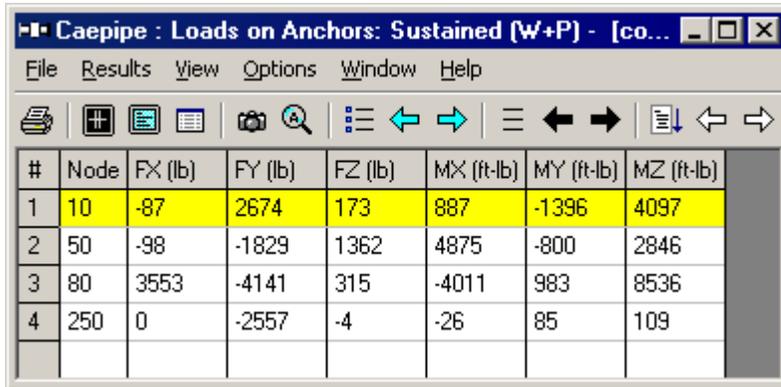


The "Load Cases" dialog box displays a list of radio button options for selecting a load case. The "Sustained (W+P)" option is selected. Other options include Expansion (T1-T3), Operating (W+P1-T3), Cold spring (W+P1-T3), Settlement, Hydrotest, Seismic (g), Wind, Response spectrum, Time history, and Harmonic response. "OK" and "Cancel" buttons are at the bottom.

<input checked="" type="radio"/> Sustained (W+P)	<input type="radio"/> Cold spring (W+P1+T1)
<input type="radio"/> Expansion (T1)	<input type="radio"/> Cold spring (W+P2+T2)
<input type="radio"/> Expansion (T2)	<input type="radio"/> Cold spring (W+P3+T3)
<input type="radio"/> Expansion (T3)	<input type="radio"/> Settlement
<input type="radio"/> Expansion (T1-T2)	<input type="radio"/> Hydrotest
<input type="radio"/> Expansion (T1-T3)	<input type="radio"/> Seismic (g)
<input type="radio"/> Expansion (T2-T3)	<input type="radio"/> Wind
<input type="radio"/> Operating (W+P1+T1)	<input type="radio"/> Response spectrum
<input type="radio"/> Operating (W+P2+T2)	<input type="radio"/> Time history
<input type="radio"/> Operating (W+P3+T3)	<input type="radio"/> Harmonic response
<input type="radio"/> Cold spring (W+P)	

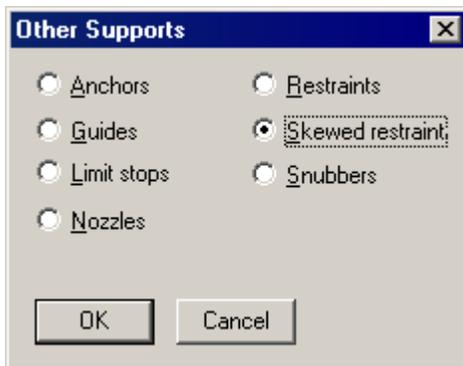
Support loads on all the supports in the model are shown in the next window. The window first shows anchor loads, if present. Then, when you click on the left or right white arrow, it shows you loads at different supports.

Results Window



#	Node	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
1	10	-87	2674	173	887	-1396	4097
2	50	-98	-1829	1362	4875	-800	2846
3	80	3553	-4141	315	-4011	983	8536
4	250	0	-2557	-4	-26	85	109

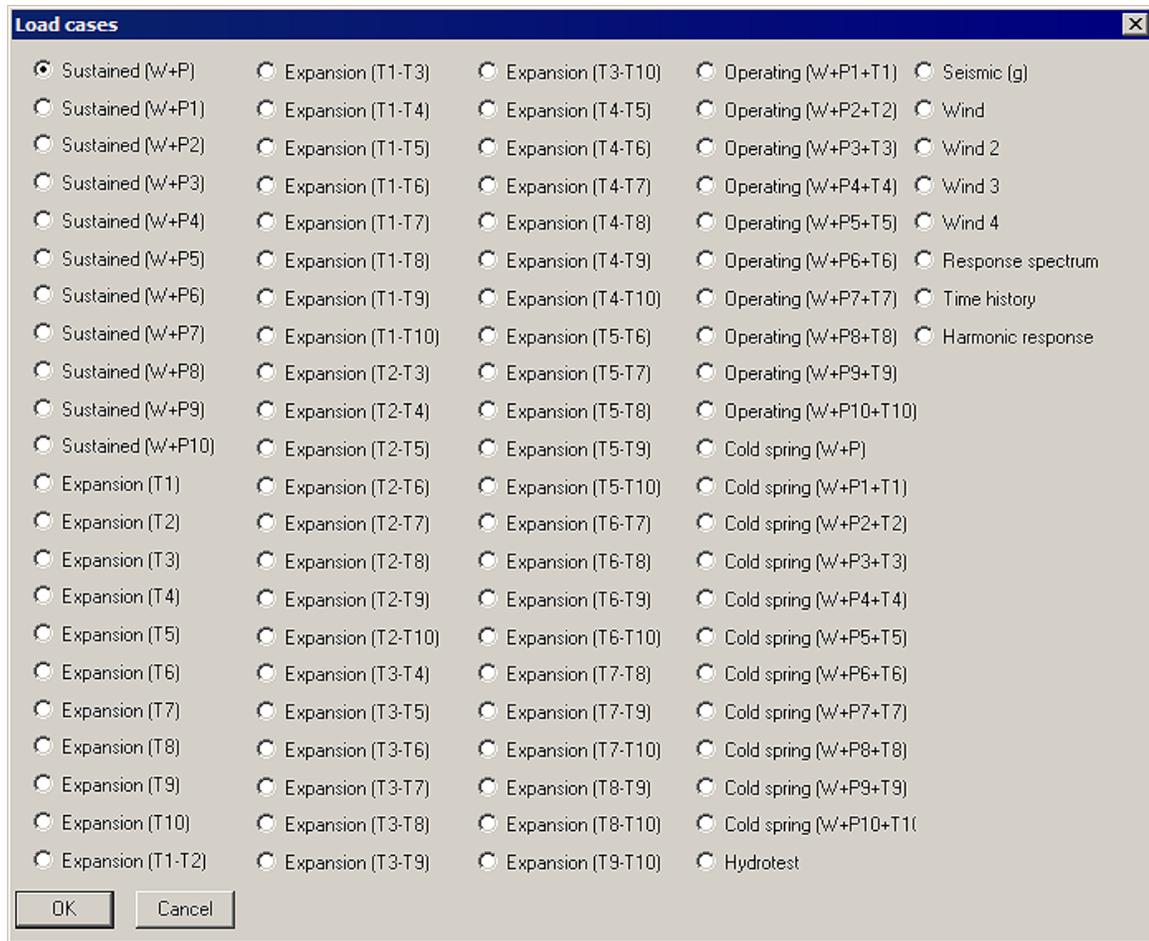
You can find out which other supports are in the model by clicking on the Other Supports button (immediately to the left of the left white arrow button). When you click on this button, the list of available support types is shown. Select the type.



Results from Multiple Thermal and Wind Loads

Displacements, Support Loads, Element Forces, etc., are calculated for the below shown thermal and wind load cases, among others. Expansion stresses under the Sorted Stresses screen will show the worst thermal case. Similarly, if any of the four wind load cases happens to be the worst among all occasional loads, then it will be shown under occasional stresses.

Results Window



Element Forces

Forces and moments on all pipe elements are shown next. If you had other elements such as a valve, you can display forces and moments for them too by clicking on the Other Forces button on the toolbar.

Caepipe : Pipe forces in local coordinates: Sustained (W+P) - [comple...]

File Results View Options Window Help

#	Node	fx (lb)	fy (lb)	fz (lb)	mx (ft-lb)	my (ft-lb)	mz (ft-lb)	B1	B2	SL (psi)
1	10	-87	2674	173	887	-1396	4097	0.50	1.00	4750
	15	-87	4100	173	887	-617	-11146	0.50	1.00	8069
2	15	-87	-4082	173	887	-617	-11146	0.50	1.00	8069
	20A	-87	-2998	173	887	-25	967	0.50	1.00	3230
3	20A	-87	173	2998	887	967	25	0.06	2.43	1853
	20B	173	87	2461	3989	1977	-256	0.06	2.43	5593
4	20B	173	-2461	87	3989	256	1977	0.50	1.00	4771
	25	173	-1852	87	3989	423	6122	0.50	1.00	6171
5	25	168	92	393	4256	419	5052	0.50	1.00	5828
	30	168	1042	393	4256	1598	3352	0.50	1.00	5352

Global Forces

Results Window

The default view for forces and moments is in the local coordinate system. They can be shown in the global coordinate system too by clicking on the Global forces button (or selecting Global Forces command from the Results menu, hotkey: F7). When you select Global forces, the forces and moments are shown in global coordinates and the button changes in the same location (on the toolbar) to an “L” for Local forces. So, should you want to return to pipe forces in local coordinates, simply click again at the same location on the Local forces button.

#	Node	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
1	10	87	-2674	-173	-887	1396	-4097
	15	-87	4100	173	887	-617	-11146
2	15	87	4082	-173	-887	617	11146
	20A	-87	-2998	173	887	-25	967
3	20A	87	2998	-173	-887	25	-967
	20B	-87	-2461	173	-1977	256	3989
4	20B	87	2461	-173	1977	-256	-3989
	25	-87	-1852	173	-6122	423	3989
5	25	393	-92	-168	5052	-419	-4256
	30	-393	1042	168	-3352	1598	4256

The convention in CAEPIPE is to display the headings for global forces and moments in uppercase (FX, FY, FZ, etc.) while display the headings for local forces in lower case (fx, fy, fz, etc.).

Sorted FRP Stresses

#	Hoop				Max Long				Min Long				Torsion			
	Node	Stress (psi)	Allow (psi)	Stress/Allow	Node	Stress (psi)	Allow (psi)	Stress/Allow	Node	Stress (psi)	Allow (psi)	Stress/Allow	Node	Stress (psi)	Allow (psi)	Stress/Allow
1	220A	-5696	1500	3.80	250	-3236	1500	2.16	220A	-5935	1500	3.96	220B	158	2500	0.06
2	220A	-5696	1500	3.80	240	-2833	1500	1.89	210	-4837	1500	3.22	240	158	2500	0.06
3	220B	-5696	1500	3.80	240	-2833	1500	1.89	220A	-4190	1500	2.79	230	158	2500	0.06
4	250	-5696	1500	3.80	230	-2751	1500	1.83	250	-4071	1500	2.71	240	158	2500	0.06
5	210	-5696	1500	3.80	230	-2751	1500	1.83	220B	-4009	1500	2.67	250	158	2500	0.06
6	230	-5696	1500	3.80	220B	-2669	1500	1.78	240	-3647	1500	2.43	230	158	2500	0.06
7	240	-5696	1500	3.80	220B	-2140	1500	1.43	240	-3647	1500	2.43	220B	158	2500	0.06
8	240	-5696	1500	3.80	220A	-1506	1500	1.00	230	-3564	1500	2.38	210	10	2500	0.00
9	220B	-5696	1500	3.80	210	-859	1500	0.57	230	-3564	1500	2.38	220A	10	2500	0.00
10	230	-5696	1500	3.80	220A	239	1500	0.16	220B	-3481	1500	2.32	220A	10	2500	0.00

Results Window

The various FRP stresses such as hoop, maximum and minimum longitudinal and torsional are sorted here by descending order of stress ratios assuming you have input the material allowables.

Displacements

Displacements for all load cases can be shown (operating load case shown here). Click on the black left/right arrow to show displacements for other load cases. You can show the deflected shape in the graphics window for any load case by clicking on the button to the left of the "A" button.

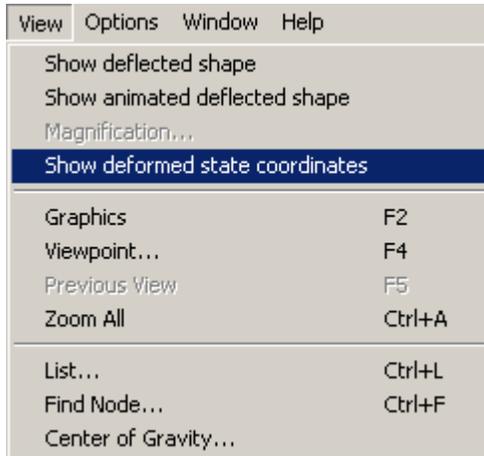
Results Window

Caepipe : Displacements: Operating (W+P1+T1) - [...]							
File Results View Options Window Help							
#	Node	Displacements (global)					
		X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	15	0.203	0.000	-0.274	-0.0668	0.4128	-0.0715
3	20A	0.358	-0.101	-0.515	-0.1176	0.1520	-0.1682
4	20B	0.282	-0.099	-0.410	-0.2209	-0.8817	-0.1048
5	25	-0.104	0.002	-0.323	-0.2967	-0.9392	-0.0659
6	30	-0.590	0.237	-0.186	-0.4598	-0.4421	0.0053
7	35	-0.665	0.559	-0.049	-0.5018	0.0920	0.3197
8	40A	-0.611	0.706	0.020	-0.4067	0.2051	0.4397
9	40B	-0.357	0.706	0.109	0.0913	0.2151	0.5604
10	50	0.000	0.500	0.000	0.0000	0.0000	0.0000
11	55	-0.463	0.115	0.011	-0.4610	-0.2150	-0.1881
12	60	-0.336	0.074	0.103	-0.4621	-0.0910	0.0148
13	70	-0.246	0.082	0.138	-0.4623	-0.0771	0.0143
14	80	0.000	0.000	0.200	0.0000	0.0000	0.0000
15	90	-0.058	-0.014	-0.126	-0.2990	-0.9355	-0.0780
16	100	-0.388	-0.564	-0.088	-0.7964	-0.0626	0.0288
17	110	-0.342	-0.557	-0.075	-0.7988	-0.0615	0.0377
18	120	-0.295	-0.547	-0.062	-0.8031	-0.0618	0.0642
19	122	-0.249	-0.527	-0.049	-0.8123	-0.0674	0.1288
20	123	-0.249	-0.500	-0.035	-0.8123	-0.0674	0.1288
21	124	-0.203	-0.467	-0.019	-0.8216	-0.0864	0.2185
22	125	-0.203	-0.467	-0.019	-0.8216	-0.0864	0.2185
23	126	-0.158	-0.407	0.002	-0.8308	-0.1174	0.3447
24	127	-0.158	-0.407	0.002	-0.8308	-0.1174	0.3447
25	128	-0.112	-0.325	0.030	-0.8400	-0.1604	0.4103
26	129	-0.066	-0.239	0.064	-0.8400	-0.1604	0.4103
27	130	0.075	-0.190	0.273	-0.8387	-0.1599	0.4683
28	131	-0.635	0.086	0.036	0.0987	-0.1805	0.4400
29	132	-0.589	0.176	0.073	0.0991	-0.1810	0.3790
30	133	-0.543	0.240	0.112	0.0996	-0.1826	0.1886
31	150	-0.497	0.250	0.150	0.1000	-0.1853	-0.1430
32	210	-1.075	0.212	-0.281	-0.5017	0.4673	0.3866
33	220A	-0.964	0.342	-0.369	-0.4775	0.3822	0.8151
34	220B	-0.664	0.435	-0.345	-0.4396	0.2956	0.9052
35	230	-0.488	0.327	-0.259	-0.3883	0.2534	0.7759
36	240	-0.339	0.220	-0.183	-0.3332	0.2112	0.6466
37	250	0.000	0.000	0.000	0.0000	0.0000	0.0000
38	70J	-0.248	0.117	0.100	-0.8407	-0.1616	0.4099
39	1000	-0.211	-0.037	0.098	-0.6638	-0.1327	0.1137
40	1000J	-0.205	-0.037	0.074	-0.8408	-0.1616	0.4099
41	1010	-0.185	-0.104	0.083	-0.7359	-0.1484	0.1822

Results Window

You can animate the same deflected shape by clicking on Show animated deflected shape (last) button in the toolbar.

You have the option of showing the deformed state coordinates in the global coordinate system along with the displacements at each node.

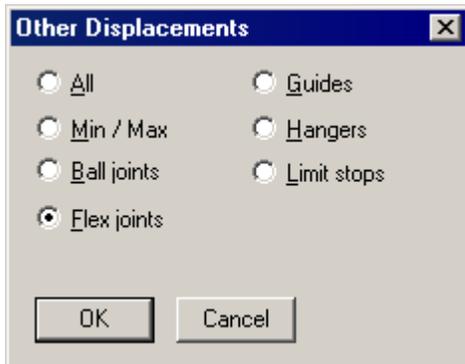


Results Window

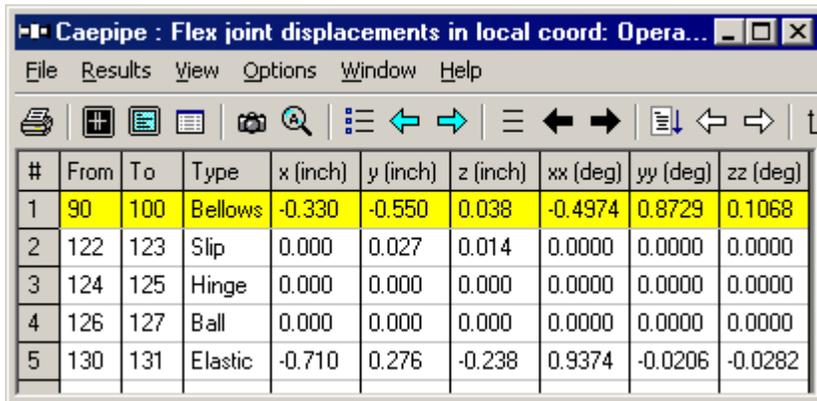
Caepipe : Displacements: Operating (W+P1+T1) - [complex1.res (\CDV-VISIONMAN\Sh...]										
File Results View Options Window Help										
#	Node	Displacements (global)						Deformed state coordinates		
		X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)	X (ft'in')	Y (ft'in')	Z (ft'in')
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000	0	0	0
2	15	0.203	0.000	-0.274	-0.0668	0.4128	-0.0715	4.5169	0	-0.0228
3	20A	0.358	-0.101	-0.515	-0.1176	0.1520	-0.1682	7.9517	-0.0084	-0.0429
4	20B	0.282	-0.099	-0.410	-0.2209	-0.8817	-0.1048	9.0235	-0.0083	1.0439
5	25	-0.104	0.002	-0.323	-0.2967	-0.9392	-0.0659	8.9913	0.0001596	2.9731
6	30	-0.590	0.237	-0.186	-0.4598	-0.4421	0.0053	8.9508	0.0198	5.9845
7	35	-0.665	0.559	-0.049	-0.5018	0.0920	0.3197	8.9446	0.0466	8.9959
8	40A	-0.611	0.706	0.020	-0.4067	0.2051	0.4397	8.9491	0.0588	10.5016
9	40B	-0.357	0.706	0.109	0.0913	0.2151	0.5604	8.9703	-1.4412	12.0091
10	50	0.000	0.500	0.000	0.0000	0.0000	0.0000	9'0"	-5'11-1/2"	12'0"
11	55	-0.463	0.115	0.011	-0.4610	-0.2150	-0.1881	11.9615	0.0095	6'0"
12	60	-0.336	0.074	0.103	-0.4621	-0.0910	0.0148	14.9720	0.0062	6.0086
13	70	-0.246	0.082	0.138	-0.4623	-0.0771	0.0143	16.9795	0.0068	6.0115
14	80	0.000	0.000	0.200	0.0000	0.0000	0.0000	23'0"	-1'0"	6.0167
15	90	-0.058	-0.014	-0.126	-0.2990	-0.9355	-0.0780	9.9952	-0.001162	2.9895
16	100	-0.388	-0.564	-0.088	-0.7964	-0.0626	0.0288	10.9677	-0.0470	2.9927
17	110	-0.342	-0.557	-0.075	-0.7988	-0.0615	0.0377	11.9715	-0.0465	2.9937
18	120	-0.295	-0.547	-0.062	-0.8031	-0.0618	0.0642	12.9754	-0.0456	2.9948
19	122	-0.249	-0.527	-0.049	-0.8123	-0.0674	0.1288	13.9792	-0.0439	2.9959
20	123	-0.249	-0.500	-0.035	-0.8123	-0.0674	0.1288	14.9792	-0'0-1/2"	2.9971
21	124	-0.203	-0.467	-0.019	-0.8216	-0.0864	0.2185	15.9830	-0.0389	2.9984
22	125	-0.203	-0.467	-0.019	-0.8216	-0.0864	0.2185	15.9830	-0.0389	2.9984
23	126	-0.158	-0.407	0.002	-0.8308	-0.1174	0.3447	16.9869	-0.0339	3'0"
24	127	-0.158	-0.407	0.002	-0.8308	-0.1174	0.3447	16.9869	-0.0339	3'0"
25	128	-0.112	-0.325	0.030	-0.8400	-0.1604	0.4103	17.9907	-0.0271	3.0025
26	129	-0.066	-0.239	0.064	-0.8400	-0.1604	0.4103	18.9945	-0.0199	3.0053
27	130	0.075	-0.190	0.273	-0.8387	-0.1599	0.4683	20.0063	-1.0158	3.0228
28	131	-0.635	0.086	0.036	0.0987	-0.1805	0.4400	19.9471	-0.9928	3.0030
29	132	-0.589	0.176	0.073	0.0991	-0.1810	0.3790	20.9510	-0.9853	3.0061
30	133	-0.543	0.240	0.112	0.0996	-0.1826	0.1886	21.9548	-0.9800	3.0093
31	150	-0.497	0.250	0.150	0.1000	-0.1853	-0.1430	22.9586	-0'11-3/4"	3.0125
32	210	-1.075	0.212	-0.281	-0.5017	0.4673	0.3866	9.9104	0.0177	8.9766
33	220A	-0.964	0.342	-0.369	-0.4775	0.3822	0.8151	10.9196	0.0285	8.9692
34	220B	-0.664	0.435	-0.345	-0.4396	0.2956	0.9052	11.9447	-0.9638	8.9712
35	230	-0.488	0.327	-0.259	-0.3883	0.2534	0.7759	11.9594	-1.9727	8.9785
36	240	-0.339	0.220	-0.183	-0.3332	0.2112	0.6466	11.9718	-2.9817	8.9848
37	250	0.000	0.000	0.000	0.0000	0.0000	0.0000	12'0"	-8'0"	9'0"
38	70J	-0.248	0.117	0.100	-0.8407	-0.1616	0.4099	16.9793	0.0098	6.0083
39	1000	-0.211	-0.037	0.098	-0.6638	-0.1327	0.1137	17.2324	-0.0031	5.0082
40	1000J	-0.205	-0.037	0.074	-0.8408	-0.1616	0.4099	17.2329	-0.0031	5.0062
41	1010	-0.185	-0.104	0.083	-0.7359	-0.1484	0.1822	17.4846	-0.0086	4.5069

Results Window

Also, you can show Minimum/Maximum displacements, displacements at flexible joints (if any) by clicking on the Other displacements button and selecting the item of interest (or by clicking on the left/right white arrows).



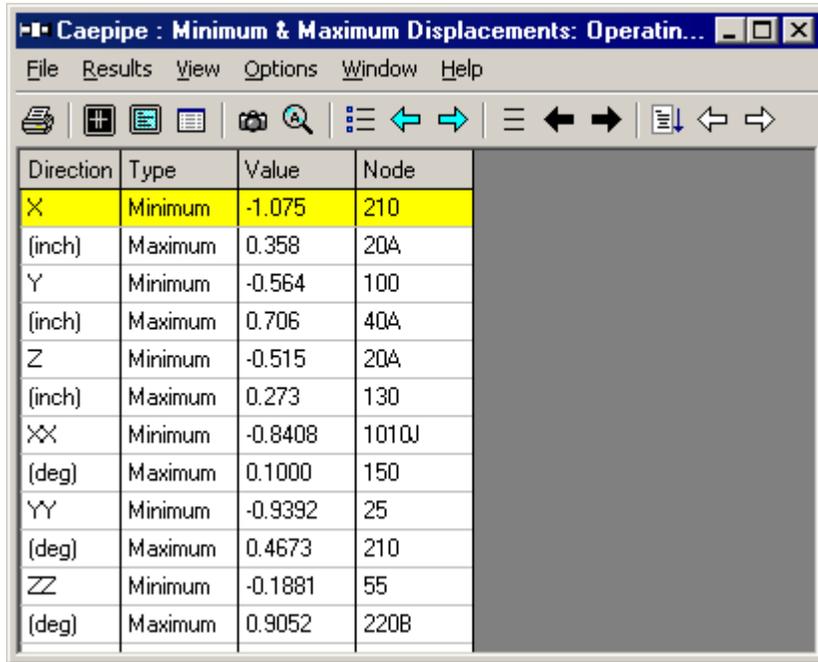
Displacements in local coordinates for all flexible joints (bellows, ball, hinge, etc.) are shown under Flex joints.



#	From	To	Type	x (inch)	y (inch)	z (inch)	xx (deg)	yy (deg)	zz (deg)
1	90	100	Bellows	-0.330	-0.550	0.038	-0.4974	0.8729	0.1068
2	122	123	Slip	0.000	0.027	0.014	0.0000	0.0000	0.0000
3	124	125	Hinge	0.000	0.000	0.000	0.0000	0.0000	0.0000
4	126	127	Ball	0.000	0.000	0.000	0.0000	0.0000	0.0000
5	130	131	Elastic	-0.710	0.276	-0.238	0.9374	-0.0206	-0.0282

Minimum/Maximum displacements for each load case can be shown (Sustained load case shown below). You can show minimum/maximum displacements for other load cases by clicking on the black arrows.

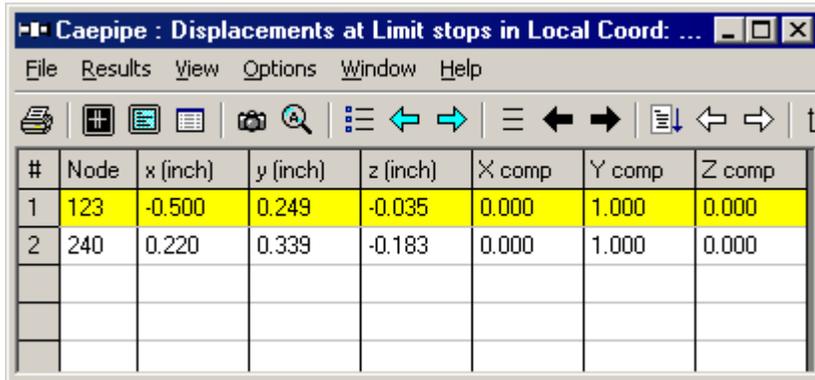
Results Window



The screenshot shows a software window titled "Caepipe : Minimum & Maximum Displacements: Operatin...". The window contains a menu bar (File, Results, View, Options, Window, Help) and a toolbar with various icons. Below the toolbar is a table with the following data:

Direction	Type	Value	Node
X	Minimum	-1.075	210
(inch)	Maximum	0.358	20A
Y	Minimum	-0.564	100
(inch)	Maximum	0.706	40A
Z	Minimum	-0.515	20A
(inch)	Maximum	0.273	130
XX	Minimum	-0.8408	1010J
(deg)	Maximum	0.1000	150
YY	Minimum	-0.9392	25
(deg)	Maximum	0.4673	210
ZZ	Minimum	-0.1881	55
(deg)	Maximum	0.9052	220B

Displacements at “Other supports” can be shown (limit stops shown here).



The screenshot shows a software window titled "Caepipe : Displacements at Limit stops in Local Coord: ...". The window contains a menu bar (File, Results, View, Options, Window, Help) and a toolbar. Below the toolbar is a table with the following data:

#	Node	x (inch)	y (inch)	z (inch)	X comp	Y comp	Z comp
1	123	-0.500	0.249	-0.035	0.000	1.000	0.000
2	240	0.220	0.339	-0.183	0.000	1.000	0.000

Frequencies

A list of natural frequencies, periods, modal participation factors and modal mass fractions is shown next. You can show each frequency’s mode shape graphically or animate it by clicking on Show mode shape or Show animated mode shape button in the toolbar.

Results Window

#	Frequency (Hz)	Period (second)	Participation factors			Modal mass / Total mass		
			X	Y	Z	X	Y	Z
1	0.774	1.2912	2.1450	0.0612	-0.2165	0.0633	0.0001	0.0006
2	0.830	1.2052	0.2255	0.0064	2.1547	0.0007	0.0000	0.0639
3	2.435	0.4107	-2.0835	-0.0087	0.0083	0.0598	0.0000	0.0000
4	3.593	0.2783	-4.1546	-0.2973	0.1725	0.2376	0.0012	0.0004
5	3.601	0.2777	-0.2704	0.1901	0.3373	0.0010	0.0005	0.0016
6	4.014	0.2491	-0.9339	0.0478	-4.5433	0.0120	0.0000	0.2842
7					Total	0.3745	0.0018	0.3507

Each frequency's mode shape detail is shown in the next window. As in the earlier window, you can show graphically the mode shape or animate it by clicking on the appropriate button.

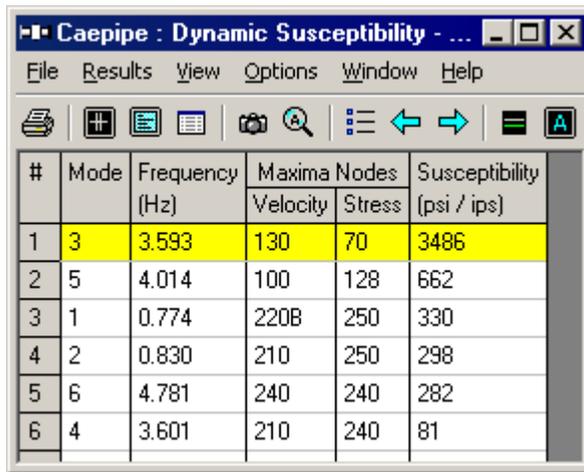
Results Window

Caepipe : Mode 1: 0.77 Hz - [complex1.res (\...)]							
File Results View Options Window Help							
#	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	15	0.000	0.000	0.000	-0.0001	-0.0000	-0.0000
3	20A	0.000	0.000	0.000	-0.0002	-0.0001	-0.0001
4	20B	0.000	0.000	0.000	-0.0002	-0.0000	-0.0004
5	25	0.000	0.000	0.000	-0.0001	-0.0000	-0.0005
6	30	0.000	0.000	0.000	-0.0000	0.0002	-0.0008
7	35	0.000	0.000	0.000	0.0001	0.0004	-0.0015
8	40A	0.000	0.000	0.000	0.0001	0.0004	-0.0012
9	40B	0.000	0.000	0.000	0.0001	0.0003	-0.0005
10	50	0.000	0.000	0.000	0.0000	0.0000	0.0000
11	55	0.000	0.000	0.000	-0.0000	0.0001	-0.0000
12	60	0.000	0.000	0.000	0.0000	0.0000	0.0001
13	70	0.000	0.000	0.000	0.0000	-0.0000	0.0001
14	80	0.000	0.000	0.000	0.0000	0.0000	0.0000
15	90	0.000	0.000	0.000	-0.0001	-0.0000	-0.0005
16	100	0.000	0.000	0.000	-0.0001	0.0000	0.0001
17	110	0.000	0.000	0.000	-0.0001	0.0000	0.0001
18	120	0.000	0.000	0.000	-0.0001	0.0000	0.0001
19	122	0.000	0.000	0.000	-0.0001	0.0000	0.0001
20	123	0.000	0.000	0.000	0.0001	0.0000	0.0001
21	124	0.000	0.000	0.000	0.0001	0.0000	0.0001
22	125	0.000	0.000	0.000	0.0001	0.0001	-0.0000
23	126	0.000	0.000	0.000	0.0001	0.0001	-0.0000
24	127	0.000	0.000	0.000	0.0001	0.0000	0.0000
25	128	0.000	0.000	0.000	0.0001	0.0000	0.0000
26	129	0.000	0.000	0.000	0.0001	0.0000	0.0000
27	130	0.000	0.000	0.000	0.0001	0.0000	0.0000
28	131	0.000	0.000	0.000	0.0000	0.0000	-0.0000
29	132	0.000	0.000	0.000	0.0000	0.0000	-0.0000
30	133	0.000	0.000	0.000	0.0000	0.0000	-0.0000
31	150	0.000	0.000	0.000	0.0000	0.0000	-0.0000
32	210	0.558	0.060	-0.091	0.0000	-0.1312	-0.0249
33	220A	0.558	0.043	-0.066	-0.0076	-0.1105	-0.1395
34	220B	0.505	0.000	-0.042	-0.0184	-0.0868	-0.3083
35	230	0.432	0.000	-0.037	-0.0290	-0.0744	-0.3913
36	240	0.343	0.000	-0.030	-0.0362	-0.0620	-0.4441
37	250	0.000	0.000	0.000	0.0000	0.0000	0.0000
38	70J	0.000	0.000	0.000	0.0001	0.0000	0.0000
39	1000	0.000	0.000	0.000	0.0000	0.0000	0.0001

Results Window

Dynamic Susceptibility

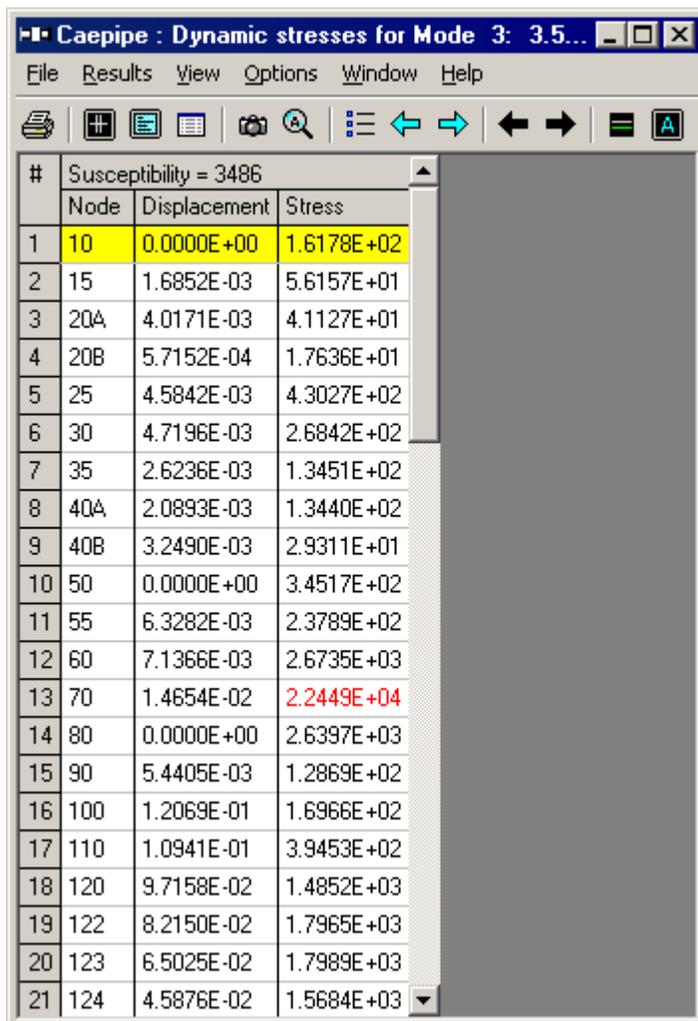
Please see topic by this name in Reference and also Appendix E.



Caepipe : Dynamic Susceptibility - ...

File Results View Options Window Help

#	Mode	Frequency (Hz)	Maxima		Susceptibility (psi / ips)
			Velocity	Stress	
1	3	3.593	130	70	3486
2	5	4.014	100	128	662
3	1	0.774	220B	250	330
4	2	0.830	210	250	298
5	6	4.781	240	240	282
6	4	3.601	210	240	81



Caepipe : Dynamic stresses for Mode 3: 3.5...

File Results View Options Window Help

Susceptibility = 3486

#	Node	Displacement	Stress
1	10	0.0000E+00	1.6178E+02
2	15	1.6852E-03	5.6157E+01
3	20A	4.0171E-03	4.1127E+01
4	20B	5.7152E-04	1.7636E+01
5	25	4.5842E-03	4.3027E+02
6	30	4.7196E-03	2.6842E+02
7	35	2.6236E-03	1.3451E+02
8	40A	2.0893E-03	1.3440E+02
9	40B	3.2490E-03	2.9311E+01
10	50	0.0000E+00	3.4517E+02
11	55	6.3282E-03	2.3789E+02
12	60	7.1366E-03	2.6735E+03
13	70	1.4654E-02	2.2449E+04
14	80	0.0000E+00	2.6397E+03
15	90	5.4405E-03	1.2869E+02
16	100	1.2069E-01	1.6966E+02
17	110	1.0941E-01	3.9453E+02
18	120	9.7158E-02	1.4852E+03
19	122	8.2150E-02	1.7965E+03
20	123	6.5025E-02	1.7989E+03
21	124	4.5876E-02	1.5684E+03

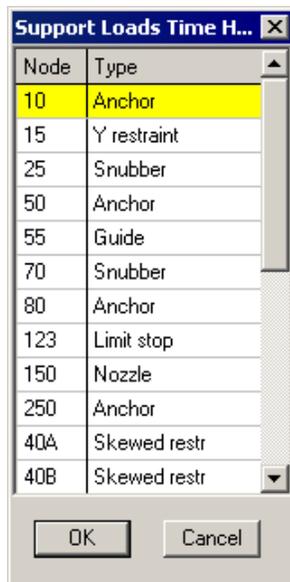
Results Window

Time History

For time history results, you are shown the following dialog from which you need to select an item.



Then, you are shown a list of supports in the model from which you need to select one.



Once you select a support from the list, then you are shown the time history at that location.

Results Window

#	Time (Sec)	X (lb)	Y (lb)	Z (lb)	XX (ft-lb)	YY (ft-lb)	ZZ (ft-lb)
1	0	0	0	0	0	0	0
2	0.1	0	0	0	0	0	0
3	0.2	0	0	0	0	0	0
4	0.3	0	3	0	2	0	10
5	0.4	0	12	0	9	0	42
6	0.5	3	85	0	62	2	304
7	0.6	32	843	-4	614	18	3007
8	0.7	8	203	-1	148	5	726
9	0.8	10	267	-1	194	6	952
10	0.9	10	263	-1	192	6	939
11	1	8	224	-1	163	5	798
12	1.1	8	224	-1	163	5	797
13	1.2	8	224	-1	163	5	798
14	1.3	8	224	-1	163	5	798
15	1.4	8	224	-1	163	5	798
16	1.5	8	224	-1	163	5	798
17	1.6	8	224	-1	163	5	798
18	1.7	8	224	-1	163	5	798
19	1.8	8	224	-1	163	5	798

While viewing these results, you can export time history results to a comma separated values (.csv) file that can be read by a spreadsheet program such as MS-Excel (see menu File > Export) for charting, etc.

Hotkeys You Can Use

For keyboard operations, the following list of hotkeys (in addition to MS-Windows keys for open, print, exit, etc.) can make you more productive.

Tab	Next results item
(Shift) Tab	Previous results item
Right Arrow Key	Next load case or mode shape
Left Arrow Key	Previous load case or mode shape
Ctrl+Right Arrow Key	Next item of same type (e.g., next support)
Ctrl+Left Arrow Key	Previous item of same type (e.g., previous support)
Ctrl+F	Find node
Ctrl+L	List
Ctrl+U	Units
Ctrl+A	Graphics - Zoom all
F2	Move focus to Graphics window
F3	Move focus to Layout, List or Results window (wherever the focus was before)
F4	Graphics - Set Viewpoint
F5	Graphics - Previous view
F6	Show list of Other supports, forces, displacements, etc.
F7	Show forces, displacements in global/local coordinates

Results Window

Open Other Windows

You can open the remaining text windows from here - List and Layout - to have all four windows open at the same time to enhance your understanding of the model results. Use the Window menu to open the other windows. They are displayed in non-editable mode.

Menus and Toolbar

These items are explained in detail under Menus. A summary is given here.



File menu

This menu contains commands for file operations including starting a new model, opening a Results file, printing, etc.

Results menu

This is one of the menus that changes with the displayed result. At all times, it contains commands for

- Display list of Results
- Move forward to Next Result (Tab key)
- Move backward to Previous Result (Shift+Tab key)

An example of the additions to the Results menu:

When Support Loads are displayed, the menu displays, in addition to the above, the following commands:

- Other supports
- Next Support
- Previous Support
- Load cases
- Next Load case
- Previous Load case

All of the above commands also appear on the (context-sensitive) toolbar.

View menu

Commands for graphics operations are available from here. This menu, like the Results menu, changes with displayed results.

An example of the changes:

When Sorted Stresses are displayed, the menu displays, in addition to the above, the following commands:

Results Window

- Show stresses in the Graphics window
- Show stress ratios in the Graphics window
- Set stress or stress ratio thresholds (to display stresses or stress ratios that are above a specific threshold value).

You do not have to move focus to the Graphics window to execute these commands.

Options menu

You can change units or font from the options menu. Any change here in units or font will affect all text windows. But the change in units is not saved when you close the Results window (they will be saved when you change units in input mode and save the model).

Window menu

You can move focus to other windows. Use F2 to move between text and Graphics windows, and F3 to move focus between open text windows. This is helpful to those who work with maximized windows (perhaps, because of a lower monitor resolution).

Help menu

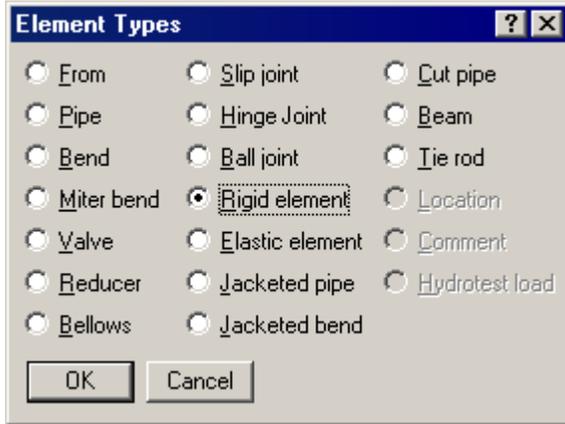
For on-line help and information pertaining to the remaining period of your yearly Maintenance, Enhancement and Support (ME&S) agreement with SST.

Rigid Element

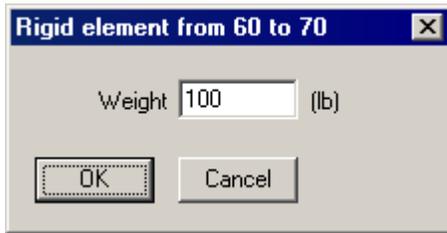
Use this element to model any “stiff” (relative to pipe) inline component.

The stiffnesses of 1×10^{12} (lb/inch) in translational directions (axial and shear), and 1×10^{12} (inch-lb./rad.) in rotational directions (bending and torsional) are used.

A rigid element is input by typing “ri” in the Type column or selecting “Rigid element” from the Element Types dialog.



The Rigid element dialog is shown.



Weight

The only required input is weight. It is applied as a distributed load along the length of the rigid element. To this weight, the Additional weight from Load is added. Contents and insulation weight are not used. Material density does not affect the weight of the rigid element.

Thermal expansion of the rigid element is calculated using the coefficient of thermal expansion from the material and temperatures from the load. Wind load is calculated using the section properties.

Rigid, weightless Elements

These may be needed when you want to account for some hard-to-model element’s thermal growth, or to connect the center line of a huge pipe to its outside surface, again to account for its (radial) thermal growth/contraction (which impacts the branch line), or to model a rigid link between two points (such as the arms around bellows, which can be tied using tie rods. See the second modeling method under “Tied Bellows” example under Expansion Joints topic).

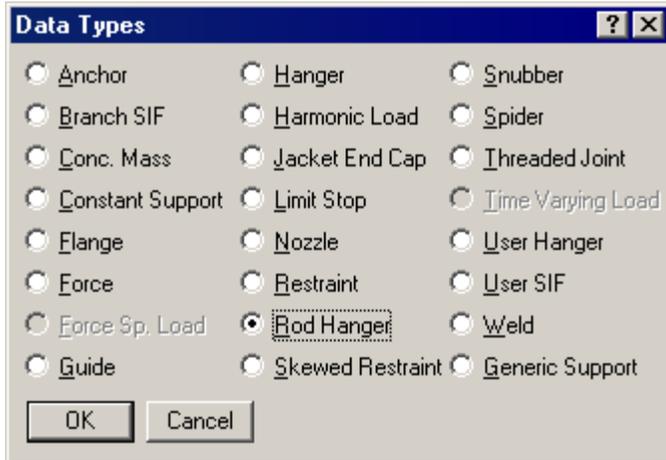
Rigid Element

To model any of these, input a rigid element, type zero for weight and ensure that the corresponding Load (specified on the Layout window under the Load column) used for this element does not have any Additional weight specified.

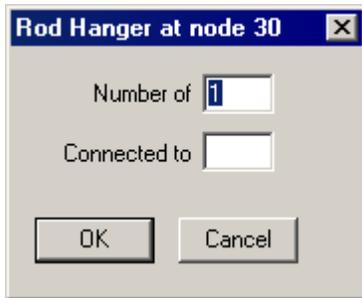
Rod Hanger

A rod hanger is a rigid one-way vertical support. The rod hanger node is rigidly supported against downward movement but able to move freely in the upward direction. That is, the rod hanger is rigid in tension (downward movement) and has no stiffness in compression (upward movement). *A rod hanger always acts in the vertical direction.*

A rod hanger is input by typing “ro” in the Data column or selecting “Rod Hanger” from the Data Types dialog.



By default, one rod hanger without a connected node is input. The number of hangers and the node to which it may be connect to may be specified in the Rod Hanger dialog.



Number of Hangers

The number of hangers is the number of separate hangers connected in parallel at this node.

Connected to Node

By default the hanger is connected to a fixed *ground* point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the “Connected to node” field. This node must be directly above the hanger node.

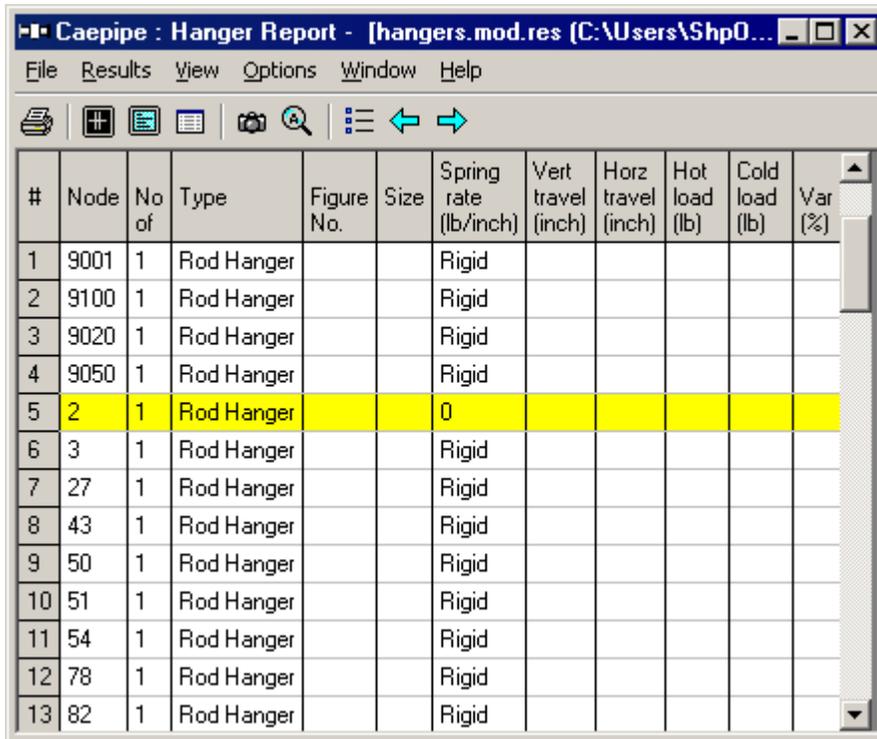
A rod hanger in CAEPIPE functions as a limit stop, that is, it functions as a nonlinear one-way restraint. It is rigid in -Y direction and fully flexible in +Y direction (in a Y-vertical system). The rod hanger offers no resistance in +Y direction.

Rod hanger results are included in the hanger report, which reports results for the first operating case (W+P1+T1). In the hanger report, a rod hanger’s spring rate may be shown either as Rigid or zero, the latter potentially confusing to the user.

Rod Hanger

It simply means that there is possibly liftoff at the hanger location for the first operating case. You can confirm this by studying vertical displacement (Y or Z) at the support for the first operating case (which will be 0 or positive). You can find reports for other operating load cases under Support Loads > Other Supports > Rod Hangers.

Liftoff (i.e., zero spring rate and a zero or positive operating condition displacement) indicates that the support may not be needed and hence could be removed. You will need to study the effect on the system at other supports after removing the rod hanger.



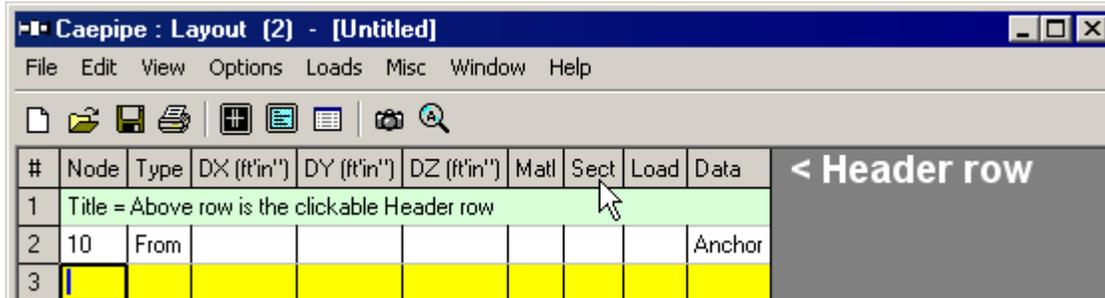
#	Node	No of	Type	Figure No.	Size	Spring rate (lb/inch)	Vert travel (inch)	Horz travel (inch)	Hot load (lb)	Cold load (lb)	Var (%)
1	9001	1	Rod Hanger			Rigid					
2	9100	1	Rod Hanger			Rigid					
3	9020	1	Rod Hanger			Rigid					
4	9050	1	Rod Hanger			Rigid					
5	2	1	Rod Hanger			0					
6	3	1	Rod Hanger			Rigid					
7	27	1	Rod Hanger			Rigid					
8	43	1	Rod Hanger			Rigid					
9	50	1	Rod Hanger			Rigid					
10	51	1	Rod Hanger			Rigid					
11	54	1	Rod Hanger			Rigid					
12	78	1	Rod Hanger			Rigid					
13	82	1	Rod Hanger			Rigid					

In dynamic analysis, the status of the rod hanger from the first operating case (W+P1+T1) is used, i.e., if the rod hanger is in tension in the first operating case, a rigid vertical restraint is used in dynamic analysis. If the rod hanger is in compression in the first operating case (possible liftoff), no vertical restraint is used at that location in dynamic analysis.

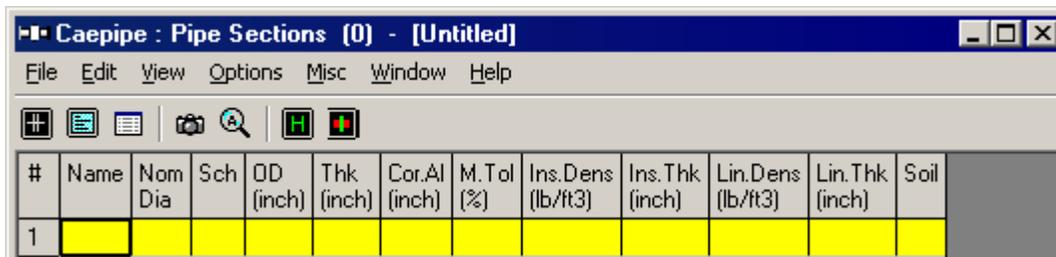
Section

A Section denotes the cross-sectional properties of a pipe used to build a piping model. You may define as many sections as needed. To define each section, you will need properties such as outside diameter (or Nominal Dia.), thickness of pipe, corrosion allowance, insulation, inside lining, and a name that is used under the Section column on the Layout window while building your model.

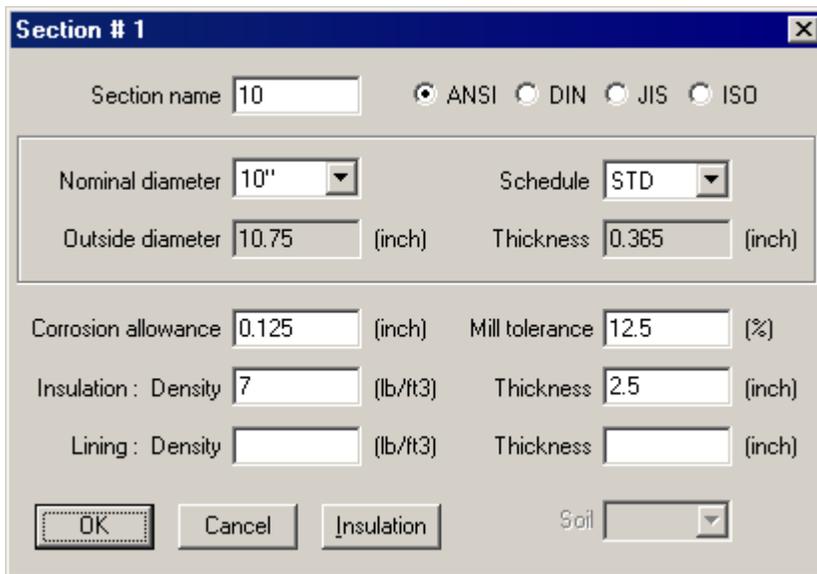
Click on “Sect” on the Header row or select Sections from the Misc menu.



CAEPIPE presents a List window that lists all defined sections in the model (none defined yet in the image below). Double click on an empty row to define a new one.



The section dialog is shown.



Section

Section name

Type an alpha-numeric name (up to five characters long) in this field. Example: If you have three 8" sections with different schedules, you could name them thus: 8-STD, 8-80 and 8-80S.

Nominal Dia, Schedule

Four databases of pipe sizes are built-into CAEPIPE — ANSI (American National Standards Institute, default), DIN (Deutsche Industrie Norm), JIS (Japanese Industrial Standard) and ISO (International Organization for Standardization).

When you click on the drop-down combo box for pipe sizes, CAEPIPE shows the list of pipe sizes that pertains to the selected database (ANSI, DIN, JIS or ISO). ANSI pipe sizes range from 1/8" to 48", DIN from 15 to 1600, JIS from 8A to 1500A, and ISO from 15 to 1000. Select the required nominal pipe size and schedule (wall thickness). To change to a different database (JIS, DIN or ISO), click on the appropriate radio button. On selection, CAEPIPE populates the correct OD and Thickness.

For pipe sizes you do not see on the list, each database allows you a nonstandard definition ("Non std" in the pipe sizes list) too. In other words, you are not restricted only to the choices available in the databases. You may define any size and thickness as needed. You will need to enter the Outside diameter and Thickness of such a nonstandard pipe, in addition to the other parameters.

Corrosion Allowance

The corrosion allowance reduces the wall thickness of the pipe and used to calculate the allowable pressure for the pipe section. Additionally, for some piping codes (B31.3, B31.4, B31.5, B31.8, B31.1 (1967), CODETI, Canadian Z183 and Z184), corrosion allowance is used for reducing the section modulus and axial area only for calculating sustained and occasional stresses.

Mill Tolerance

The mill tolerance (in percent) is also used to reduce the wall thickness of the pipe while calculating allowable pressure. For example, if the mill tolerance is input as 12.5 (%), the pipe thickness (while calculating allowable pressure) is = 0.875 x nominal thickness.

Reduced thickness = (1 - Mill tolerance/100) × nominal thickness - Corrosion allowance

If defined while modeling, corrosion allowance, mill tolerance, insulation and lining densities are carried forward only for a new section definition.

Insulation

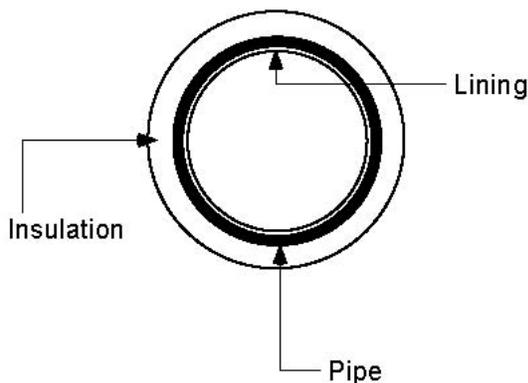
Type the pipe insulation density and thickness here. Click on the Insulation button for the insulation library, or enter your own.

Insulation Material	Density (lb/ft ³)
Amosite Asbestos	16
Calcium Silicate	15
Careytemp	10
Cellular Glass	9
Fiberglass	7
High Temperature	24
Kaylo 10	12.5
Mineral Wool	8.5
Perlite	13
Poly Urethane	2.2
Styro Foam	1.8
Super-X	25

Highlight the desired insulation material and press Enter. The insulation density is entered on the section property dialog. CAEPIPE uses these values to calculate the insulation weight which is added to the weight of the pipe.

Lining

Lining is used to prevent internal corrosion that might occur during transportation of a gas or a fluid. CAEPIPE has the ability to model these protective coatings inside the pipe.



Lining is different from insulation. Insulation is around and outside of the pipe. Lining is on the inside of the pipe. Both have respective thicknesses and densities. See previous figure.

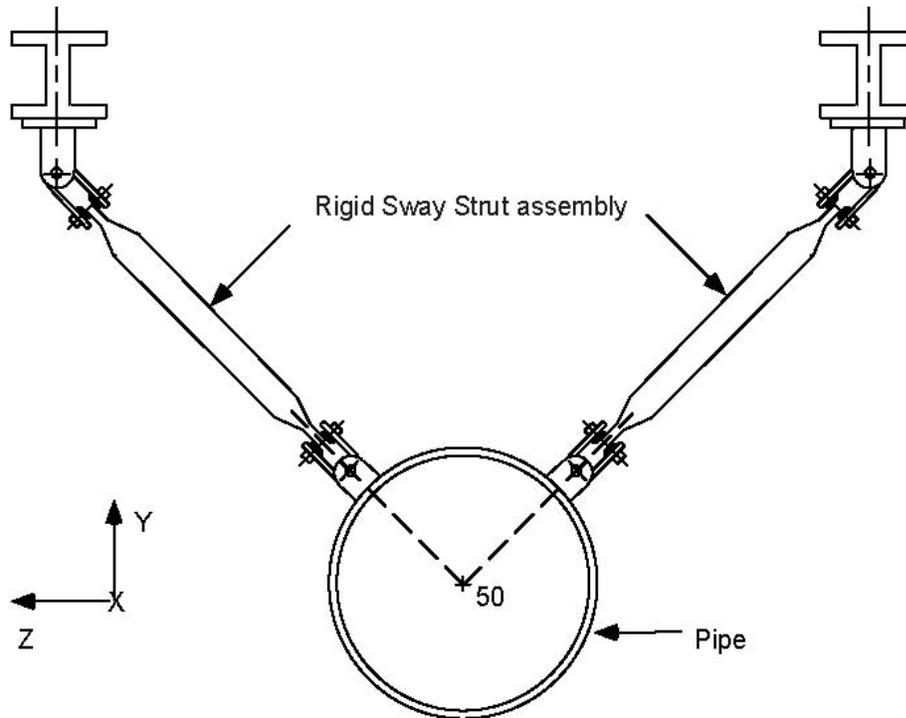
While calculating the weight of the fluid/gas inside the pipe, CAEPIPE accounts for lining thickness by reducing the pipe's internal diameter by twice the lining thickness.

Skewed Restraint

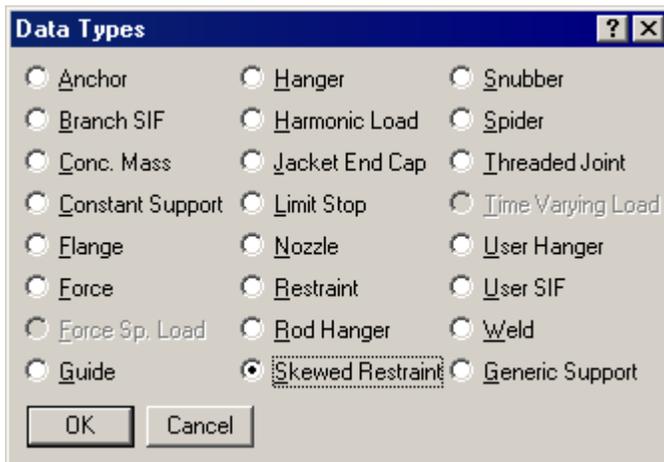
A Skewed Restraint is a two-way support that resists translation along or rotation about any specified direction at a node. You have to use either a manufacturer-supplied stiffness or calculate it for the support you want to model.

Use this restraint to model sway braces, sway struts and similar supports. You can also use this to model vertical/horizontal supports, though it is used more commonly to resist lateral forces.

The figure below shows an application.

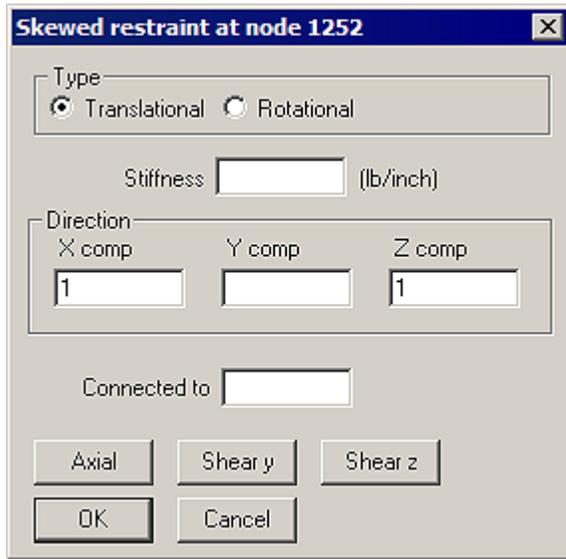


A skewed restraint is input by typing "sk" in the Data column or selecting "Skewed restraint" from the Data Types dialog.



Skewed Restraint

The Skewed Restraint dialog is shown.



Type

Translational: Use this type to restrain translation along the specified direction.

Rotational: Use this type to restrain rotation about the specified direction.

Stiffness

Type in the translational or rotational stiffness of the support. As an illustration, assume that you had a rod (in tension only) which you were modeling as a skewed restraint. You can calculate the stiffness (required to be input) in the following manner: Assume a 2.5 in. dia. rod 2 feet long, modulus of elasticity of rod material = 30×10^6 psi.

The translational (axial) stiffness is $AE/L = \frac{\pi}{4} (2.5)^2 \times 30 \times 10^6 / 24 = 6,135,925$ lb./in.

The rotational stiffness is $GJ/L = \frac{E}{2(1+\nu)} \times \frac{J}{L} = 1,843,727$ in.-lb./rad., where G is the shear modulus, ν is the Poisson's ratio and J is polar moment of inertia.

Direction

If you have no "connected to node," the direction in which the skewed restraint is oriented must be specified in terms of its global X, Y and Z components. See topic on specifying a Direction.

Or use one of the preset buttons to orient the skewed restraint axis:

1. **Axial:** To set the axis along the local-x direction (pipe axis)
2. **Shear y:** To set the axis in the local-y direction
3. **Shear z:** To set the axis in the local-z direction

If you have connected the skewed restraint node to another node, then the direction must not be input. It is calculated from the locations of the skewed restraint node and the connected node, and it is oriented from the skewed restraint node towards the connected

Skewed Restraint

node. In order for CAEPIPE to calculate the direction, the skewed restraint node and the connected node must not be coincident.

Connected to node

If the skewed restraint node is connected to an externally fixed point (ground), leave the “Connected to node” blank. You may connect a skewed restraint node to another node that is not coincident with the skewed restraint node. Note that during skewed restraint force calculations, the relative displacement of the skewed restraint node is calculated with respect to the connected node.

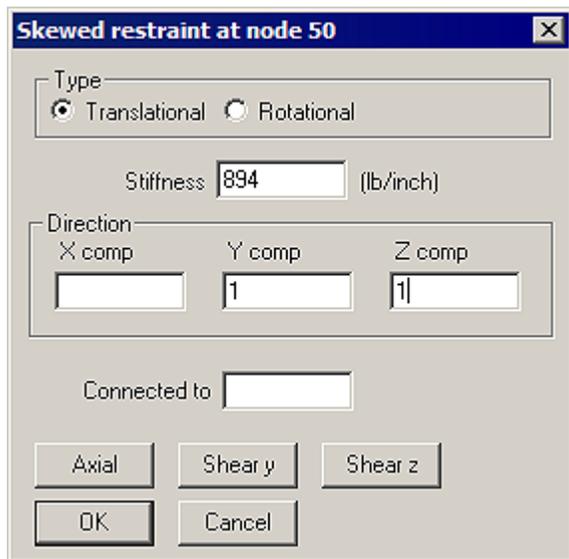
Example: Modeling a Sway Brace

Assume that we need to model two sway braces in the same arrangement as shown in the figure at the beginning of this section. The translational stiffness of the sway braces is given as 894 lb./in. As can be surmised from the figure, the orientation of the sway braces (sway struts in the figure) is at 45° from the X- and Y-axes. We shall model the support on the right hand side first followed by the support on the left hand side.

The following steps describe the modeling procedure:

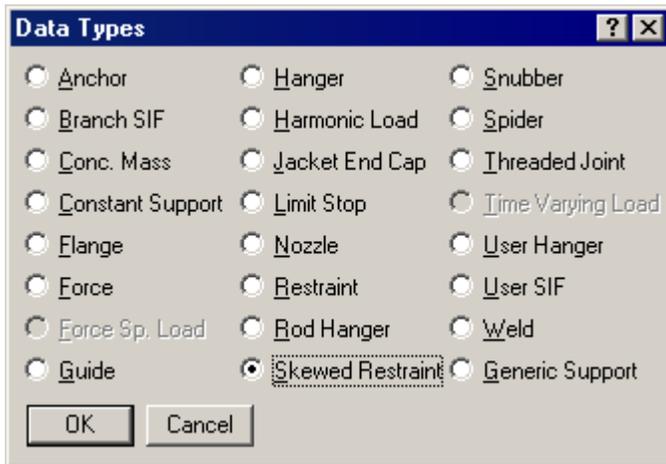
- ▶ Create node (on pipeline) where support is required. In this case, the node is 50. Position highlight on this row.
- ▶ First support (right): Type “sk” in the Data column to open the skewed restraint dialog box.

Ensure that Type is set to Translational; if not, click on the Translational radio button. Type 894 for Stiffness, type 1 for Y comp and -1 for Z comp, press Enter.



Skewed Restraint

- ▶ Second support (left): type 50 for Node on an empty row, press Tab to move to next field, press "l(L)" for Location. This will open the Data types dialog.



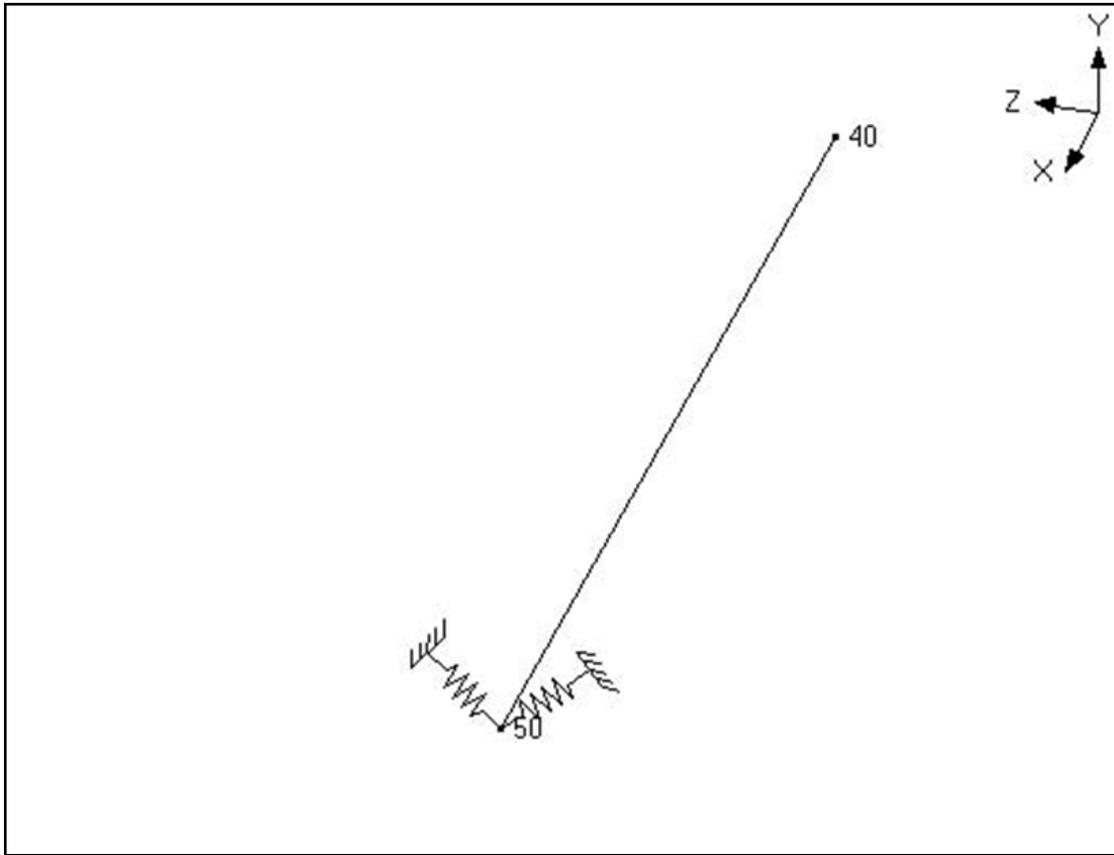
Select Skewed restraint by clicking on it to open the skewed restraint dialog. Enter the skewed restraint dialog similar to the first skewed restraint except in this case type 1 for Z comp, press Enter.

The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sway brace								
2	40	From							
3	50		5'0"			1	8	1	Skewed restr
4	50	Location							Skewed restr
5									

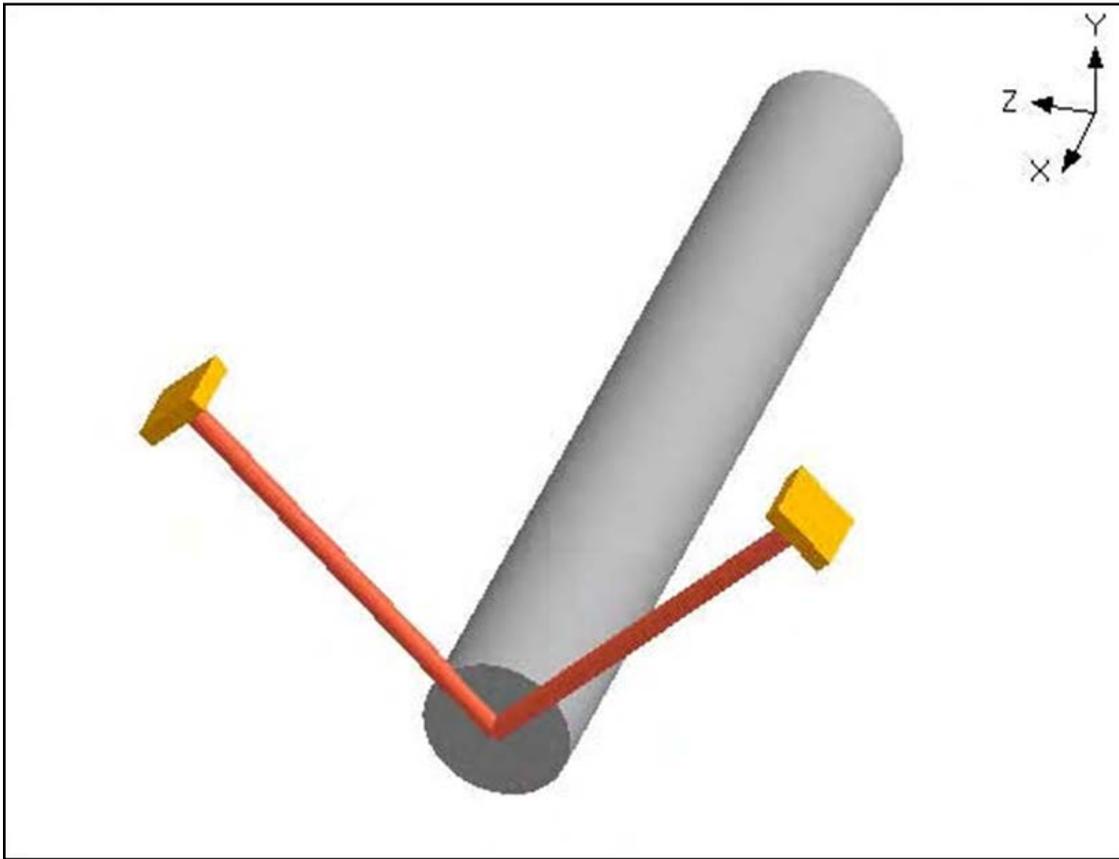
The graphics is shown below:

Skewed Restraint



Skewed Restraint

The rendered graphics is shown below:

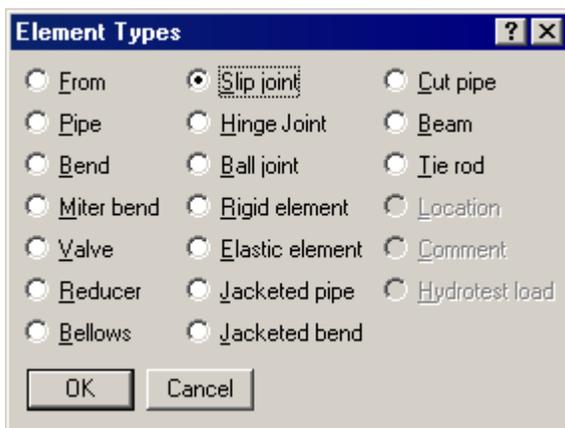


Slip Joint

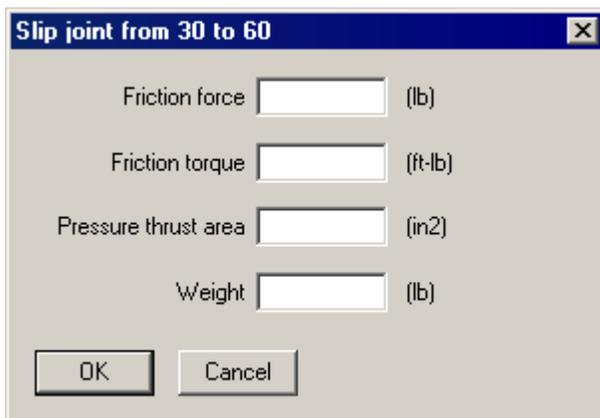
A slip joint allows for axial (through telescopic action) and torsional movement between adjacent pipes (due to thermal expansion or contraction). The joint itself can be fixed using an anchor if so designed. Slip joints are susceptible to lateral buckling due to internal pressure, and may become less effective when subjected to small bending loads. Proper guiding to prevent buckling and keeping the two telescopic parts concentric are therefore necessary.

Since the primary purpose of a slip joint is to absorb axial growth, the joint is ideal for placing it towards the end of long pipe runs, while its growth is directed axially by the use of one or more guides.

A Slip Joint is input by typing “s” under the Type column or by selecting “Slip joint” from the Element types dialog.



The Slip joint dialog is shown.



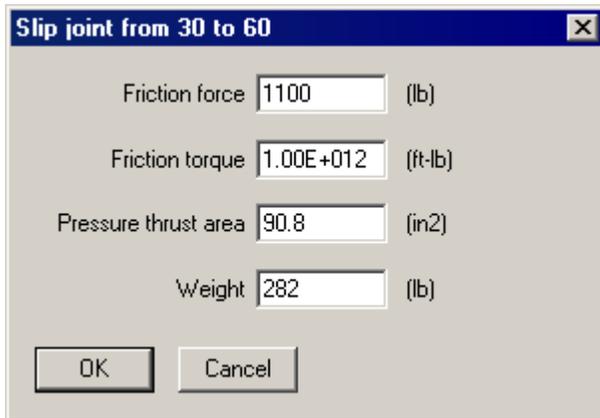
A slip joint manufacturer should be able to provide you the required data for a slip joint.

A slip joint will have axial deflection or rotation only when the external forces exceed the friction force or friction torque respectively. If the pressure thrust area is input, CAEPIPE imposes a thrust load of: Pressure x Thrust area on both nodes of the slip joint. The weight is the empty weight of the joint. The contents, insulation and additional weight are added to the empty weight. A slip joint is rigid in lateral directions.

Slip Joint

Example:

Assume that we want to model a (telescoping action) slip joint that allows only axial movement with no torsion. The outer sleeve of the joint is anchored to hold it in place while the other end is free to translate axially. The axial (friction) force has to exceed 1,100 lb. (you will need to get this datum from a manufacturer's catalog) to make the slip joint move and the slip joint cannot rotate about the axial direction. So, the data would look similar to that shown next between nodes 30 and 40.



Slip joint from 30 to 60

Friction force (lb)

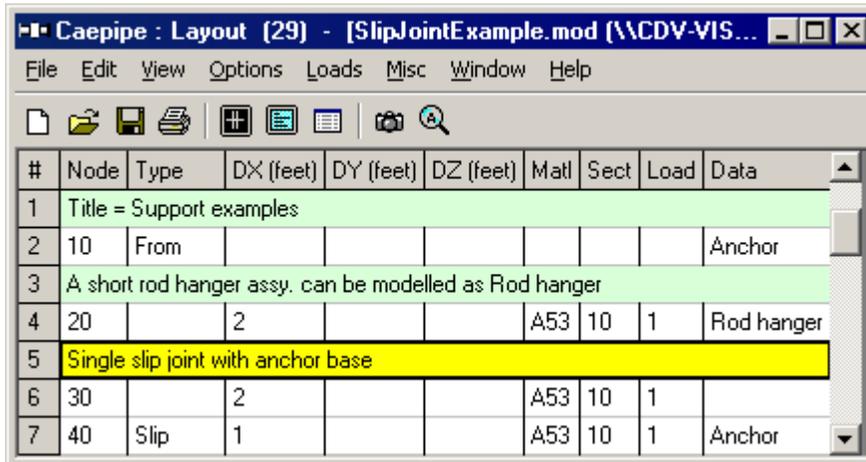
Friction torque (ft-lb)

Pressure thrust area (in²)

Weight (lb)

OK Cancel

And, the modeling on the layout screen would look thus:



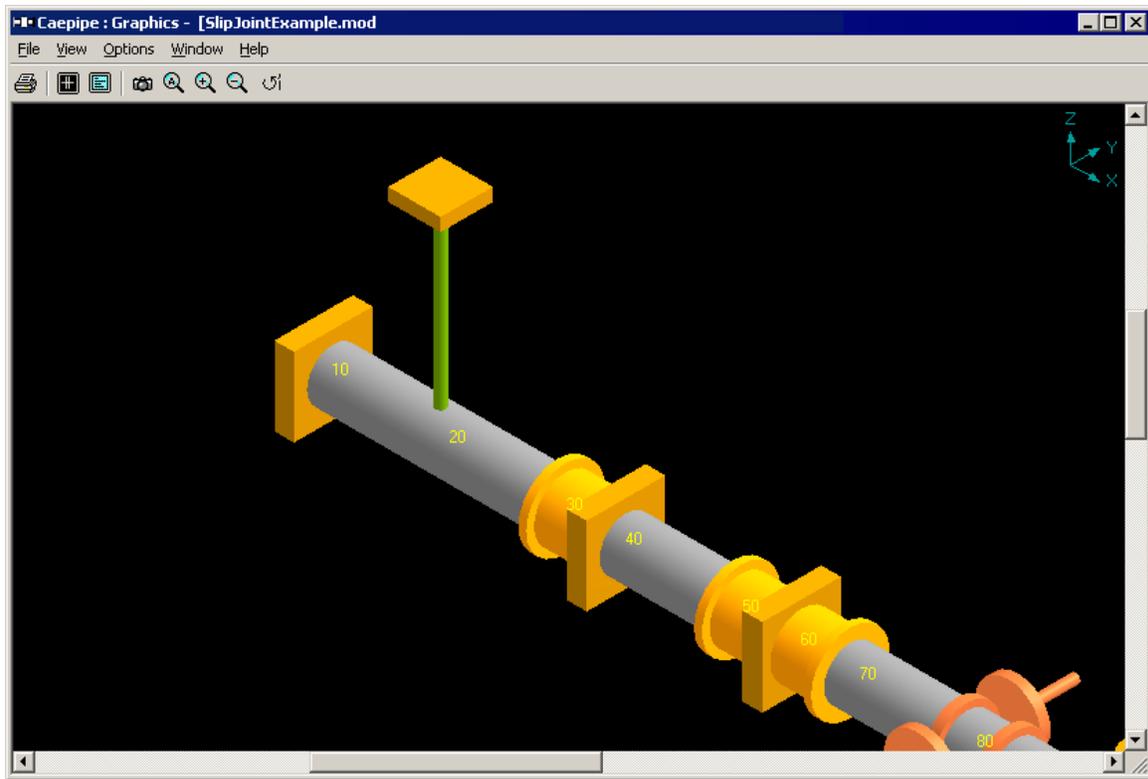
Caepipe : Layout (29) - [SlipJointExample.mod (\CDV-VIS...]

File Edit View Options Loads Misc Window Help

File Edit View Options Loads Misc Window Help

#	Node	Type	DX (feet)	DY (feet)	DZ (feet)	Matl	Sect	Load	Data
1	Title = Support examples								
2	10	From							Anchor
3	A short rod hanger assy. can be modelled as Rod hanger								
4	20		2			A53	10	1	Rod hanger
5	Single slip joint with anchor base								
6	30		2			A53	10	1	
7	40	Slip	1			A53	10	1	Anchor

Slip Joint

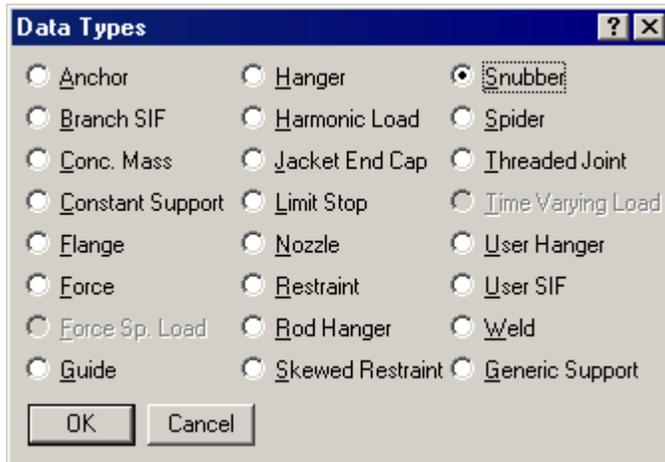


See the topic on Nonlinearities for related information.

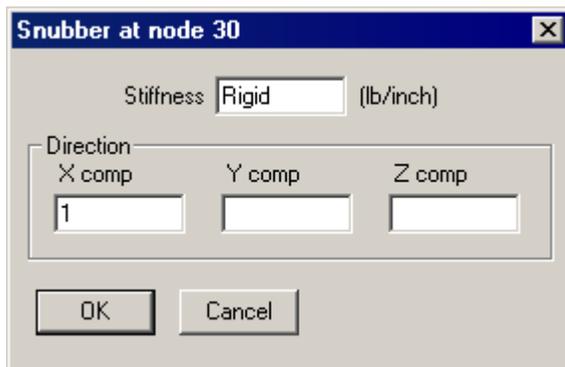
Snubber

A snubber provides only translational restraint in a specified direction for seismic and dynamic cases only. In other words, a snubber engages only during movements caused by a dynamic load. It does not restrain against static loads such as weight and thermal.

A snubber is input by typing “sn” in the Data column or selecting “Snubber” from the Data Types dialog.



The Snubber dialog is shown.

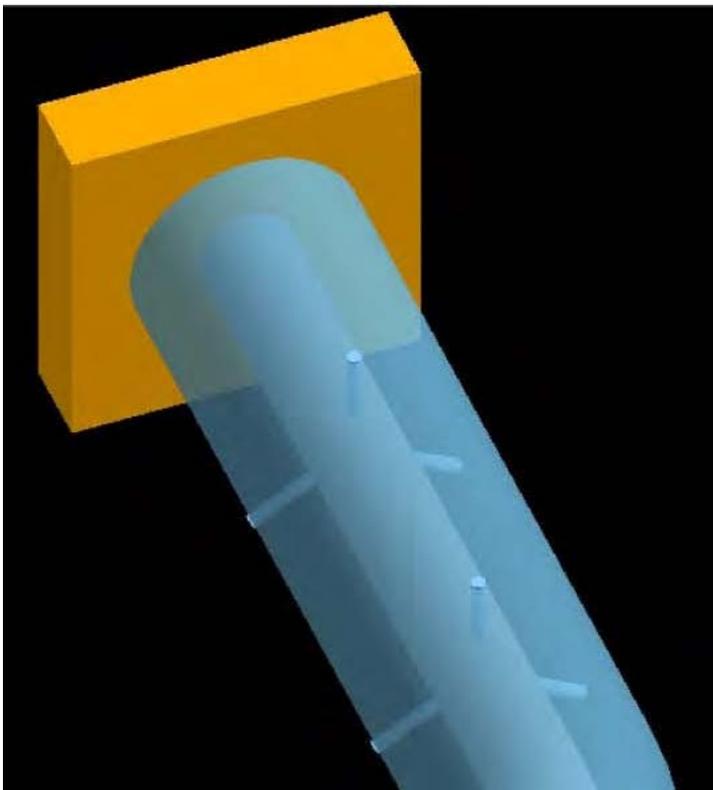
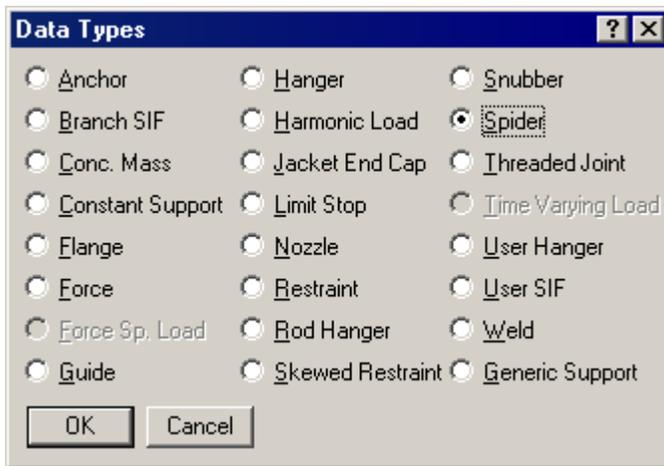


The stiffness defaults to Rigid, however a stiffness may be input for flexible snubbers. See section on specifying a Direction for information on X comp, Y comp, Z comp. A snubber can be made active in any direction by using this combination of Direction Cosines (X comp, Y comp, and Z comp).

Spider

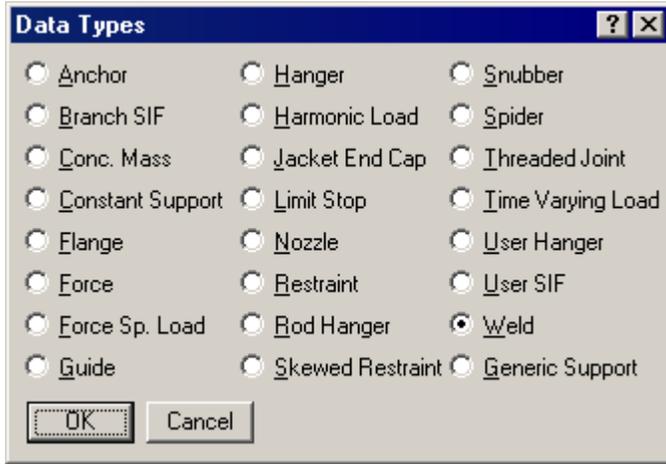
Use a spider (also called a spacer) to connect the coincident nodes of a jacketed pipe (i.e., the node on the core pipe and the corresponding node on the jacket pipe). The spider acts as an internal guide. At the spider location, the local x-axis is calculated along the pipe direction. The spider connects the local y and z translations for the core and jacket nodes. It prevents any radial movement but allows sliding, rotating and bending movement between core and jacket pipes. No gap is allowed between the core pipe and the spider. See section on Jacketed pipe for related information.

A spider is input at a jacketed pipe node by typing “sp” in the Data column or selecting “Spider” from the Data Types dialog.



Supports

CAEPIPE provides many support types, which enable you to model available support hardware. These support types are available under the Data types menu (See section on Data Types).



You can use one or more types of supports to simulate a real-world support. For example, you could use two lateral limit stops with unequal gaps to simulate a pipe shoe (see example under Limit Stop).

To input more than one support at a node, use the “Location” data type.

Anchor

An anchor can be modeled as a flexible or rigid support which by default restrains the three global translations and rotations at the applied node (six degrees of freedom). Use this to model all anchor blocks, and nodes where piping connects to equipment (pumps, compressors, turbines, etc.).

Restraint

A restraint is a two-way rigid support which restrains the translations (negative and positive directions) along the specified global directions. You can apply a restraint in all the three directions at the same time.

Skewed Restraint

This is a flexible two-way support that can be oriented in any direction. Use this support to resist either translational movement along or rotational movement about the specified direction (as in a pipe sleeve which resists nonaxial rotation). Use this to model rigid or flexible sway struts and sway braces.

Hanger, User hanger, Rod hanger, Constant support

These should be used as vertical supports only. Use a Hanger when you want to design a variable spring hanger(s) for your piping system (there are 30+ hanger catalogs built-in to CAEPIPE for your convenience). Use a User hanger when you want to analyze hangers in an existing piping system. Use a Rod hanger for a rod hanger assembly. Use a Constant support to design a constant support or a constant effort hanger.

“Bottomed-out” Springs

To analyze this situation, use a spring hanger and a limit stop at this node. Type in the maximum allowable hanger travel for one of the limits of the limit stop. Once the hanger traverses the maximum distance allowed, the limit stop becomes active.

Guide

A Guide is a rigid or flexible restraint which resists lateral pipe movements (in directions perpendicular to the axis of the pipe). You can specify an annular gap, if required, inside the guide. A friction coefficient is optional. Use a Guide to model U-straps, U-Bolts, pipe guide assemblies, pipe slides and similar supports.

Limit stop

A limit stop, a nonlinear restraint, can be oriented in any direction with a gap specified on both sides of the pipe. A limit stop allows free movement for the distance of the gap and then acts as a rigid or flexible restraint.

A Line Stop is a support that restricts axial movement of pipe. This support can be modeled using a limit stop with its direction oriented along the pipe's axis. Use this support to model pipe slide assemblies, pipe skirts and similar arrangements. See topic on Limit Stop for an example.

A limit stop can be used to model 1-way supports for pipe racks where vertical downward movement is restrained while upward movement is not. See example for pipe rack modeling in the Beam topic.

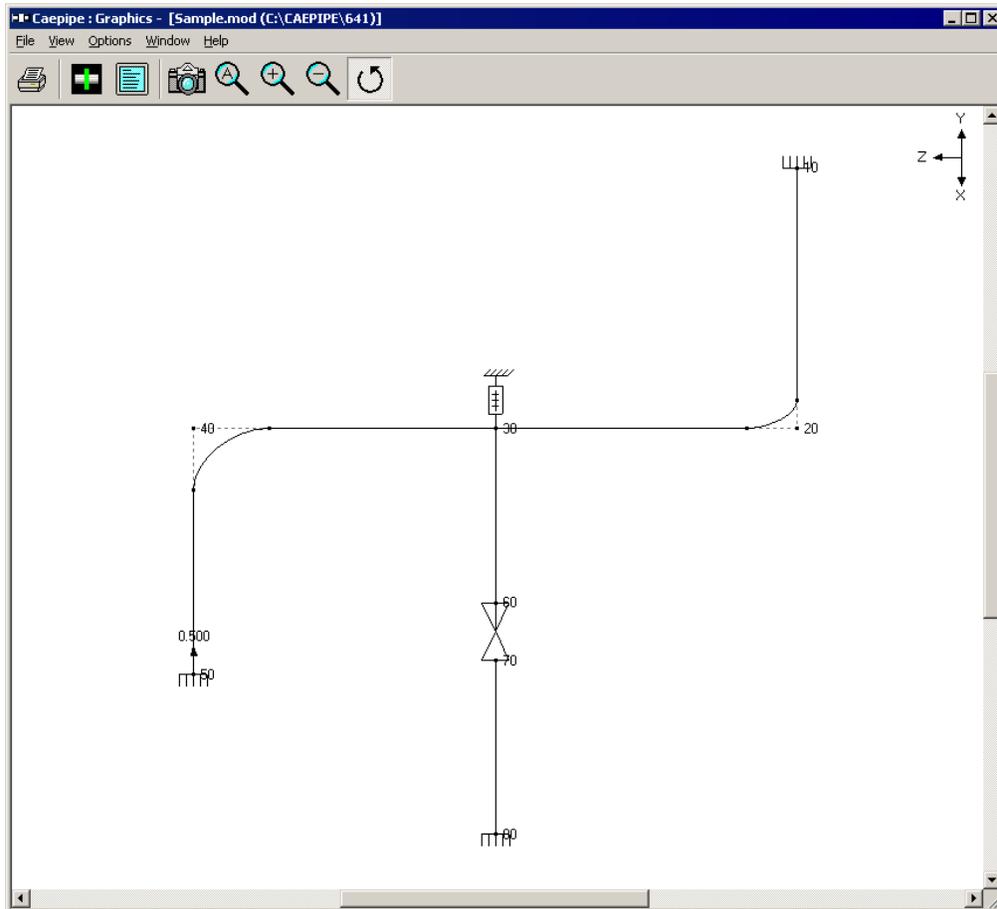
Refer to the respective CAEPIPE support term in Reference for a detailed explanation on it.

Supports

Caepipe : Layout (29) - [SlipJointExample.mod (\\CDV-V...)]									
File Edit View Options Loads Misc Window Help									
#	Node	Type	DX (feet)	DY (feet)	DZ (feet)	Matl	Sect	Load	Data
1	Title = Support examples								
2	10	From							Anchor
3	A short rod hanger assy. can be modelled as Rod hanger								
4	20		2			A53	10	1	Rod hanger
5	Single slip joint with anchor base								
6	30		2			A53	10	1	
7	40	Slip	1			A53	10	1	Anchor
8	Double slip joint with anchor base								
9	50		2			A53	10	1	
10	60	Slip	1			A53	10	1	Anchor
11	70	Slip	1			A53	10	1	
12	Pipe slide base modelled with 3 limit stops								
13	First limit stop is in vertical direction, Free to lift but not go down								
14	80		2			A53	10	1	Limit stop
15	Second limit stop is for lateral movement +/- 3"								
16	80	Location							Limit stop
17	Third limit stop is for axial movement +8", -0"								
18	80	Location							Limit stop
19	Pipe slide guide modelled with restraint and a limit stop								
20	The restraints are against vertical and lateral movements								
21	The limit stop is for axial movement +8", -0"								
22	90		3			A53	10	1	YZ restraint
23	90	Location							Limit stop
24	Pipe spider guide is modelled as guide								
25	100		3			A53	10	1	Guide
26	Roller support is modelled as vertical limit stop and lateral restraint								
27	110		3			A53	10	1	Limit stop
28	110	Location							Y restraint
29	120		3			A53	10	1	Anchor
30									

Tees

Physical tees, even though integral components, are not modeled as such in CAEPIPE. Instead, they are modeled as three pipes coming together at a common node.



In this figure, you can see that three pipe elements (20-30, 30-40, and 30-60) come together at node 30 to form a tee.

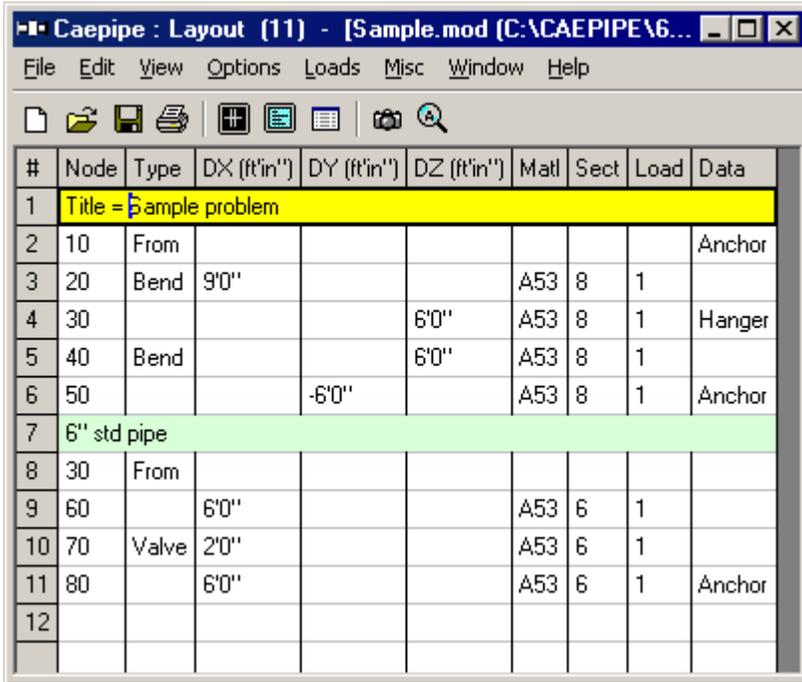
Tees

Modeling

The method of modeling is as simple as its representation.

Step 1:

You could model the pipe run first from 20 to 30 (1st element), then from 30 to 40 (2nd element), and finally, from 30 to 60 (3rd element). See example layout window below.



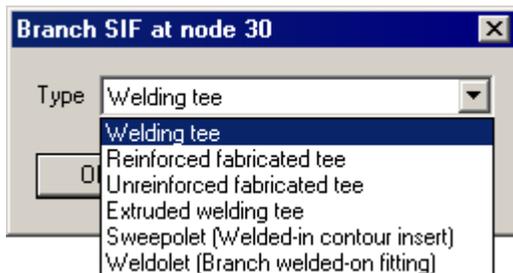
The screenshot shows the 'Caepipe : Layout (11) - [Sample.mod (C:\CAEPIPE\6...)]' window. It features a menu bar (File, Edit, View, Options, Loads, Misc, Window, Help) and a toolbar with icons for file operations and navigation. Below the toolbar is a table with the following data:

#	Node	Type	DX (ft'in')	DY (ft'in')	DZ (ft'in')	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									

Alternately, you could model from 20 – 30 – 60, and then 30 – 40. Or from 60 – 30 – 20, and then 30 – 40. Modeling order is immaterial to analysis (but sometimes complicate the merge files process. See Merge under Layout window > Merge).

Step 2:

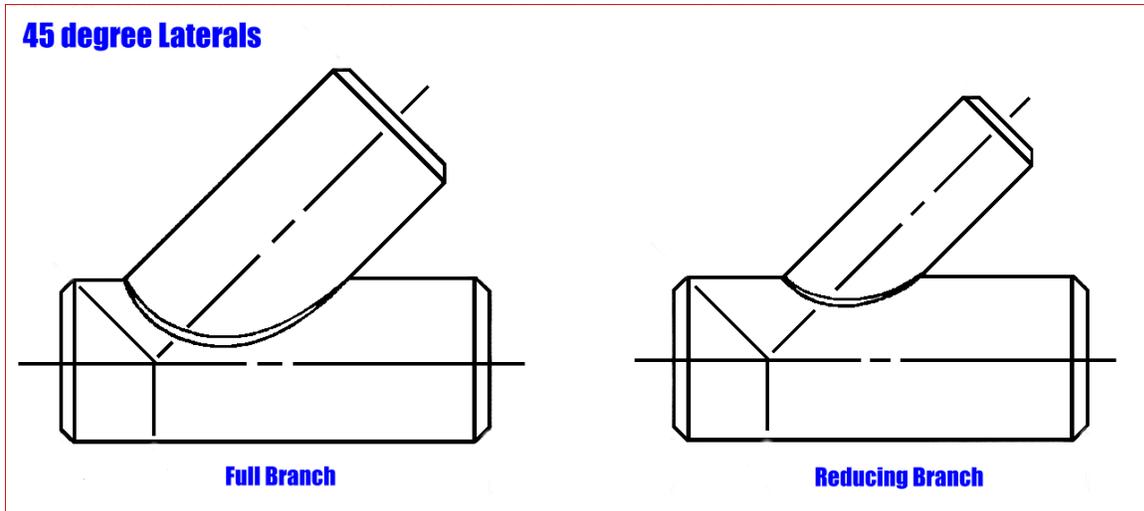
There is one final step remaining. You need to designate the type of tee connection it is. In this example, the tee is a “Welding Tee”. So, on any row that contains node 30 (i.e., row #4 or #8), type “Br” (Branch SIF) and select “Welding Tee” from the drop down list.



The shown list is piping code-dependent. In other words, the list of tee types shown comes from the selected piping code that dictates how an SIF for each of the listed tee types is calculated. See Appendix D of B31 codes for more information (or see Appendix A in this manual which contains the same information from the B31 codes).

Tees

Based on a more rigorous analysis, if you have another more accurate SIF value for a joint you want to insert (instead of the code's), then skip this step (2) and use User-SIF data type at the same node to input your own SIF.

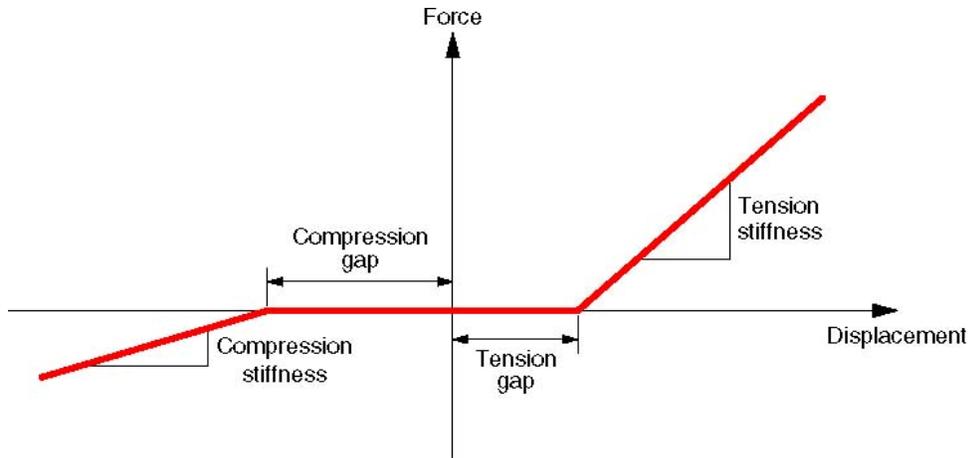


For 45° laterals, a piping code committee member opines: Use the SIF for an unreinforced fabricated tee and evaluate the branch stress using the section modulus of the branch. The connection footprint on the run pipe itself has a section modulus greater than the branch section modulus and this will compensate for the unreinforced fabricated tee SIF which should be lower than the unreinforced fabricated lateral SIF. Ignore the fact that a reduced outlet branch connection requires an “effective section modulus” in accordance with B31.1 Para. 104.8.4(C) or B31.3 Para. 319.4.4(c) and just calculate the intensified stress as the unreinforced fabricated tee SIF x the branch moment / branch section modulus (whether you use the SIF times the resultant moment approach of B31.1 or the in-plane and out-plane SIFs times the in-plane and out-plane moments, respectively, approach of B31.3, use the same philosophy of ignoring the “effective modulus.”

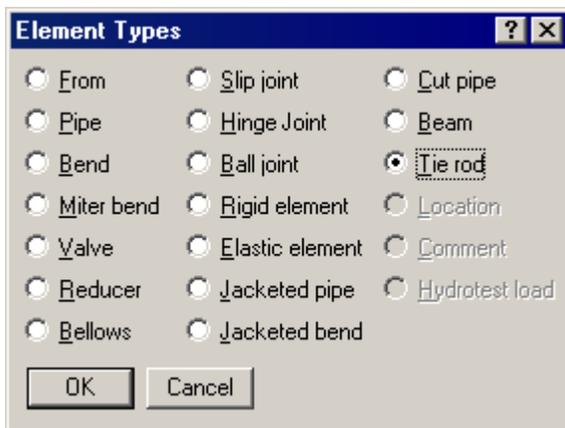
Pay particular attention to the fabrication of the lateral making sure that the Code required cover fillet dimensioned $t(c)$ in B31.1 Fig. 127.4.8(D)(a) or B31.3 Fig. 328.5.4D(1) meets the required size. If the cover fillet is larger than $t(c)$, that improves the lateral SIF. The figures and dimensions shown are a bit unfortunate, especially for a lateral because a strict reading of the Code would seem to require a bigger weld on the obtuse side of the branch and a smaller weld on the acute side of the branch when just the opposite is true. Having a constant cover fillet weld leg length all around the branch would improve the requirement and the committee has been working on that. Personally, I think the larger the run and branch pipes are, the larger the cover fillet should be. Ask for or calculate whether area replacement requirements B31.1 Para. 104.3.1(D) or B31.3 Para. 304.3.3 are met noticing that the required reinforcement for a lateral is greater by the factor $(2 - \sin \alpha)$.

Tie Rod

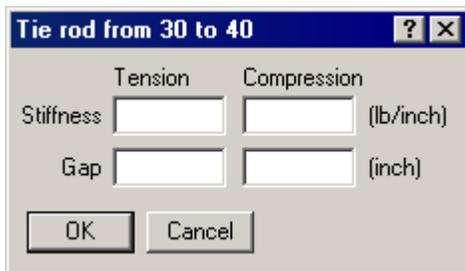
Tie rod is a nonlinear element with different stiffnesses and gaps in tension and compression, used to model tie rods in bellows, chains, etc. The force versus displacement relationship for a tie rod is shown below. Around expansion joints, a tie rod continuously restrains the full pressure thrust while allowing only lateral deflection, bending and torsional rotation.



When the tie rod is in tension, and the displacement is greater than the tension gap, tension stiffness is used. If the displacement is less than the tension gap, zero stiffness is used. Similarly when the tie rod is in compression, and the displacement is greater than the compression gap, compression stiffness is used. If the displacement is less than the compression gap, zero stiffness is used. A Tie rod is input by typing “t” in the Type column or selecting “Tie rod” from the Element Types dialog.



The Tie Rod dialog is shown.



Tie Rod

A tie rod can be made “Tension only” by setting the compression stiffness to zero. Similarly it can be made “Compression only” by setting the tension stiffness to zero. Both Tension and Compression stiffnesses cannot be zero. If there is no tension or compression gap, leave it blank or specify it as zero. See the earlier “Expansion Joints” topic for examples.

Turbine

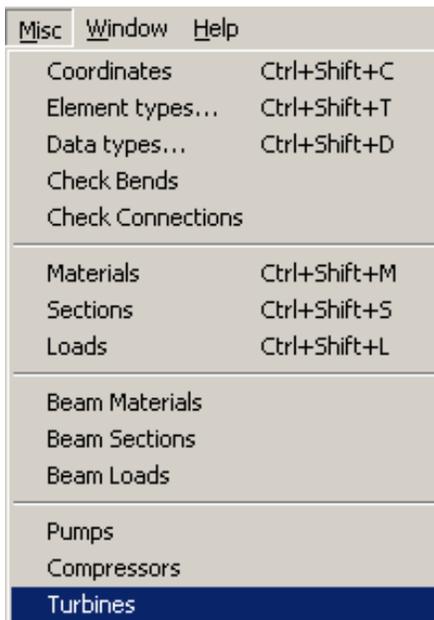
Pumps, compressors and turbines in CAEPIPE, referred to as rotating equipment, are each governed by an industry publication — API (American Petroleum Institute) publishes an API 610 for pumps and an API 617 for compressors while NEMA (National Electrical Manufacturers Association) publishes the NEMA SM-23 for turbines. These publications provide guidelines for evaluating nozzles connected to equipment among other technical information including the items relevant to piping stress analysis – criteria for piping design and a table of allowable loads.

Modeling the equipment is straightforward since it is assumed rigid (relative to connected piping) and modeled only through its end points (connection nozzles).

1. In your model, anchor all of the nozzles (on the equipment) that need to be included in the analysis.
2. Specify these anchored nodes during the respective equipment definition via Misc. menu > Pumps/Compressors/Turbines in the Layout window.

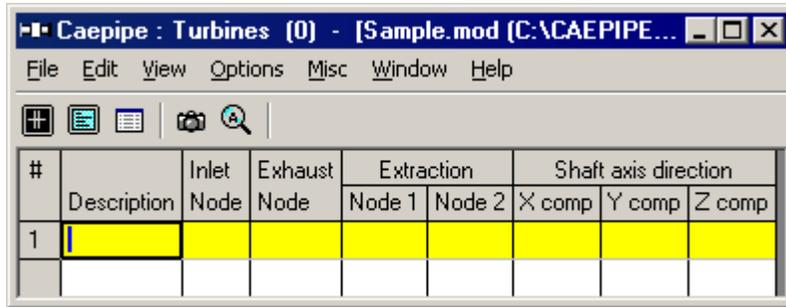
CAEPIPE does not require you to model all of the nozzles nor their connected piping. For example, you may model simply one inlet nozzle of a pump with its piping. Or, you may model one pump with both nozzles (with no connected piping) and impose external forces on them (if you have that data). Further, there is no need to connect the two anchors of the equipment with a rigid massless element like required in some archaic methods. A flange and an anchor may coexist.

A turbine (like a pump or a compressor) is input by selecting “Turbines” from the Miscellaneous (Misc) menu in the Layout window. CAEPIPE, upon analysis, produces a NEMA SM-23 turbine compliance report. See Appendix B (NEMA SM-23, for Turbines) for related information.



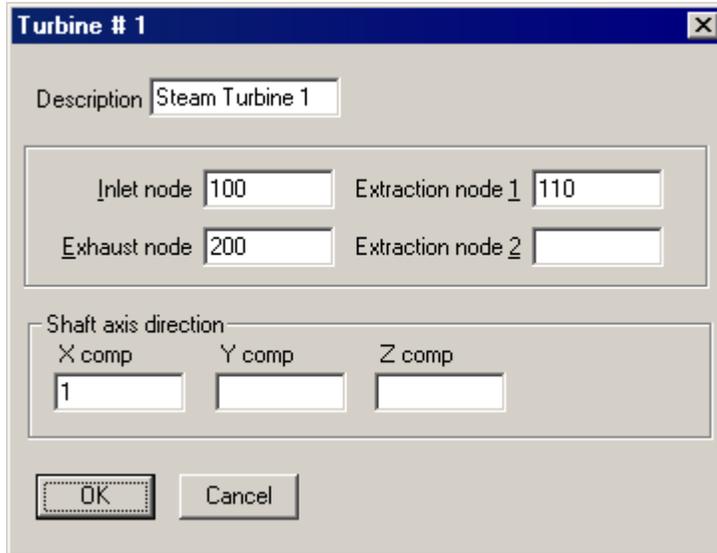
Once you see the Turbine List window, double click on an empty row for the Turbine dialog and enter the required information.

Turbine



The screenshot shows the 'Caepipe : Turbines (0) - [Sample.mod (C:\CAEPIPE...)]' window. It features a menu bar (File, Edit, View, Options, Misc, Window, Help) and a toolbar with icons for adding, deleting, and searching. Below is a table with the following structure:

#	Description	Inlet Node	Exhaust Node	Extraction		Shaft axis direction		
				Node 1	Node 2	X comp	Y comp	Z comp
1								



The 'Turbine # 1' dialog box contains the following fields and controls:

- Description: Steam Turbine 1
- Inlet node: 100
- Exhaust node: 200
- Extraction node 1: 110
- Extraction node 2: (blank)
- Shaft axis direction:
 - X comp: 1
 - Y comp: (blank)
 - Z comp: (blank)
- Buttons: OK, Cancel

A short description to identify the turbine may be entered for Description. The nozzle nodes must be anchors and the shaft axis must be in the horizontal plane. Some of the nozzle nodes may be left blank if they are not on the turbine (e.g., extraction nodes).

See under Pumps for related modeling tips and under Specifying a Direction for information on how to specify X comp/Y comp/ Z comp for Shaft axis.

Turbine

Caepipe : Rotating Equipment Report - [complex1.res (C:\CAEPIPE\700LM)]

File Results View Options Window Help

API 610 (10th ed.), Oct 2004 / ISO 13709 report for pump : pump1

Load case: Operating (W+P10+T10)

Shaft axis: Xcomp = 1.000, Ycomp = 0.000, Zcomp = -2.500

Center location: X = 6'0", Y = -7'0", Z = 8'0" (ft'in")

Suction node: 50, Location: (Top), Size: 8.000 (inch)

Offsets from center: dx = -2.5997, dy = -4.2710, dz = 1'0" (ft'in")

Check of condition F.1.1 for suction node 50:

	Calculated	Allowed	Ratio	Status
FX (lb)	-1226	850	1.442	----
FY (lb)	1292	700	1.845	----
FZ (lb)	-5476	1100	4.978	Failed
FR (lb)	5758	1560	3.691	Failed
MX (ft-lb)	447	2600	0.172	OK
MY (ft-lb)	179	1300	0.137	OK
MZ (ft-lb)	603	1900	0.317	OK
MR (ft-lb)	771	3500	0.220	OK

Condition F.1.2.a for suction node 50 failed ****

Discharge node: 250, Location: (Side), Size: 4.000 (inch)

Offsets from center: dx = 1.2999, dy = -5.9423, dz = -1'0" (ft'in")

Check of condition F.1.1 for discharge node 250:

	Calculated	Allowed	Ratio	Status
FX (lb)	10	320	0.032	OK
FY (lb)	-26	400	0.064	OK
FZ (lb)	-4	260	0.016	OK
FR (lb)	28	570	0.049	OK
MX (ft-lb)	0	980	0.000	OK
MY (ft-lb)	0	500	0.000	OK
MZ (ft-lb)	0	740	0.000	OK
MR (ft-lb)	0	1330	0.000	OK

Condition F.1.1 for discharge node 250 passed

Turbine

NEMA SM-23 Report

NEMA SM23 (1991) report for turbine : ST

Load case: Operating (W+P1+T1)

Shaft axis: Xcomp = 1.000, Ycomp = 0.000, Zcomp = 0.000

		---- Forces (lb) ----			---- Moments (ft-lb) ----		
Node	Type	fx	fy	fz	mx	my	mz
210	Inlet	-106	107	436	58	-535	-323
300	Exhaust	-168	-2607	0	0	0	240

		Size	--- Resultant ---		Allow		
Node	Type	(inch)	F(lb)	M(ft-lb)	3F + M	able	Ratio
210	Inlet	10.000	462	628	2013	4333	0.465
300	Exhaust	36.000	2612	240	8077	8667	0.932

Combined resultants at Exhaust node 300

		---- Forces (lb) ----			---- Moments (ft-lb) ----		
		fx	fy	fz	mx	my	mz
Calculated		-273	-2499	436	-1424	1221	-874
Allowable		923	2307	1845	4614	2307	2307
Ratio		0.296	1.083	0.237	0.309	0.529	0.379

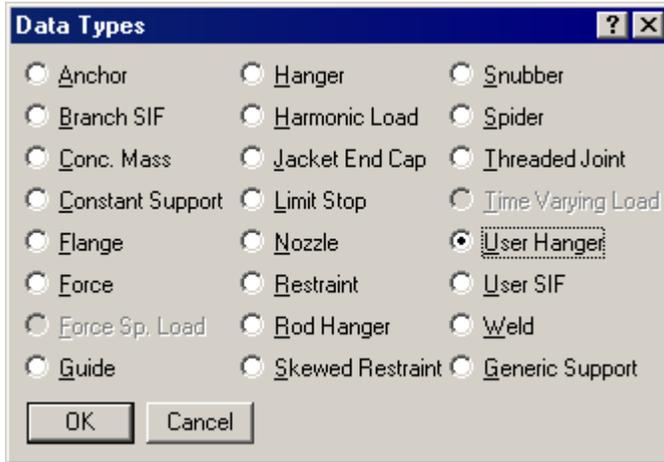
		--- Resultant ---		Allow		
		F(lb)	M(ft-lb)	2F + M	able	Ratio
Combined		2552	2069	7173	4614	1.555

See Appendix B, NEMA SM-23 for more information on how to interpret the report.

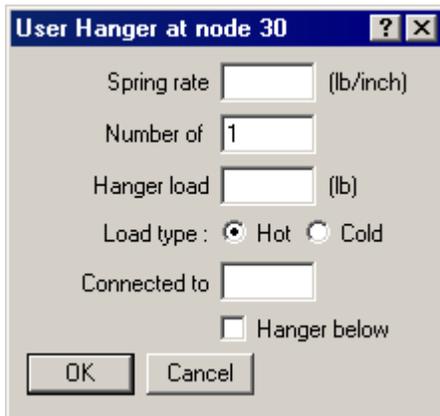
Note: If you have input multiple temperatures and pressures, corresponding reports for additional operating load cases will be shown. Use the black left/right arrow key to see them.

User Hanger

Use the “User hanger” type for analyzing existing hangers different from hangers that need “to be designed” (for which you use the “Hanger” data type). For a hanger already designed and working, the spring rate and load are known and should be input here. If not, leave them blank so that CAEPIPE can calculate them. A user hanger is input by typing “u” in the Data column and pressing Enter or selecting “User Hanger” from the Data Types dialog.



The User Hanger dialog is shown.



Spring Rate

The spring rate is required. *For a constant support user hanger, input the spring rate as zero.*

Number of Hangers

Type in the number of separate hangers connected in parallel at this node. The stiffness and load of each hanger are multiplied by the number of hangers to find the effective stiffness and load of the hanger support at this node.

Hanger Load

Input the hanger load, if known. Otherwise, leave it blank and CAEPIPE will calculate the load.

Hanger Load

The hanger load may be specified as hot or cold using the Load type radio buttons.

User Hanger

Connected to Node

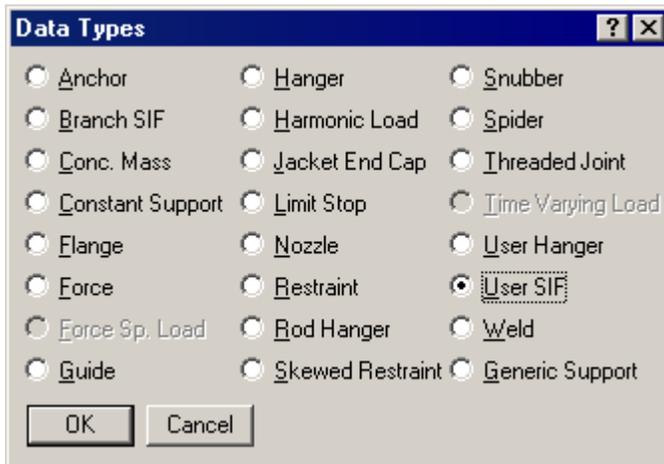
By default the hanger is connected to a fixed *ground* point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the “Connected to node” field. This node must be directly above or below the hanger node.

User SIF

User SIF (Stress Intensification Factor) may be used to specify SIF at a node where there is normally no SIF (i.e., non bend or non tee node) or to override calculated SIF at the node.

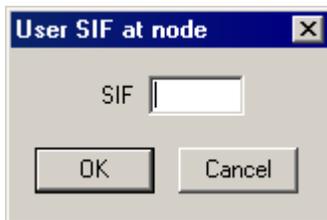
Use this for any component that needs an SIF value – nonright angle tees, nonstandard tee or branch connection, flanges, etc. – for which the chosen piping code does not specify an SIF, or you want to override the code’s SIF. For example, in case of a bend or a tee, CAEPIPE calculates the SIF according to the selected piping code. To override the calculated SIF, specify a User SIF. *Note that a User SIF is applied to all elements that come together at this node.*

A User SIF is input by typing “user s” in the Data column or selecting “User SIF” from the Data types dialog.



Depending on the piping code, either a single value (B31.1) or both in-plane and out-of-plane values of SIF (B31.3) may be required. The corresponding dialog will be shown.

B31.1 code



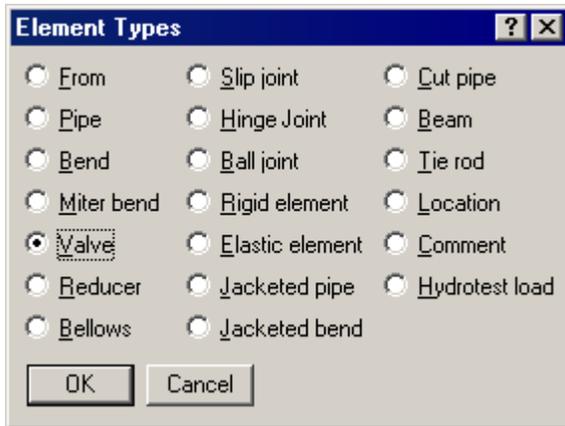
B31.3 code



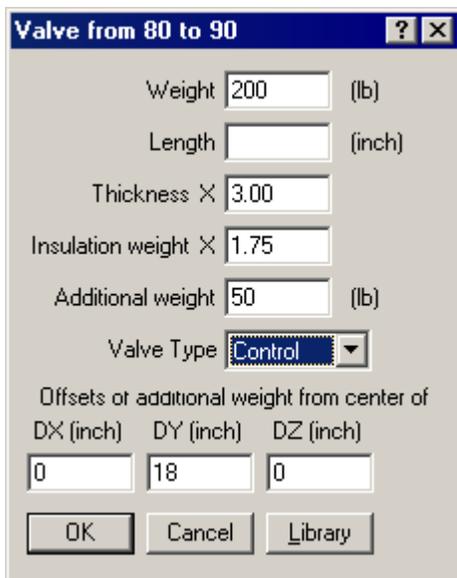
Valve

Use this element to model any type of valve. A valve is relatively more rigid than a pipe. CAEPIPE uses the data input to calculate the rigidity.

A Valve is input by typing “v” in the Type column or selecting “Valve” from the Element Types dialog.



The Valve dialog is shown.



Weight

The weight is the empty weight (without contents, insulation, etc.). CAEPIPE applies this weight as a uniformly distributed load along the length of the valve. Additional weight, if specified, is treated as a concentrated weight offset from the center of the valve.

Length

If the valve length is input, the DX, DY, DZ in Layout is adjusted to match the valve length. If the valve length is left blank, the valve length is calculated from DX, DY, DZ input in Layout.

Thickness X

The thickness multiplier (Thickness X) is used for stiffness calculation (i.e., the thickness of the pipe section is multiplied by Thickness multiplier in the calculation of the valve stiffness). Typical value for Thickness multiplier is 3 which is the default value if left blank.

Insulation weight X

The insulation weight multiplier (Insulation weight X) is used if the valve has additional insulation compared to adjacent pipe (i.e., weight of insulation calculated from the insulation thickness of the pipe section is multiplied by Insulation weight X multiplier). Typical value for insulation weight multiplier is 1.75 which is the default value if left blank.

Additional weight

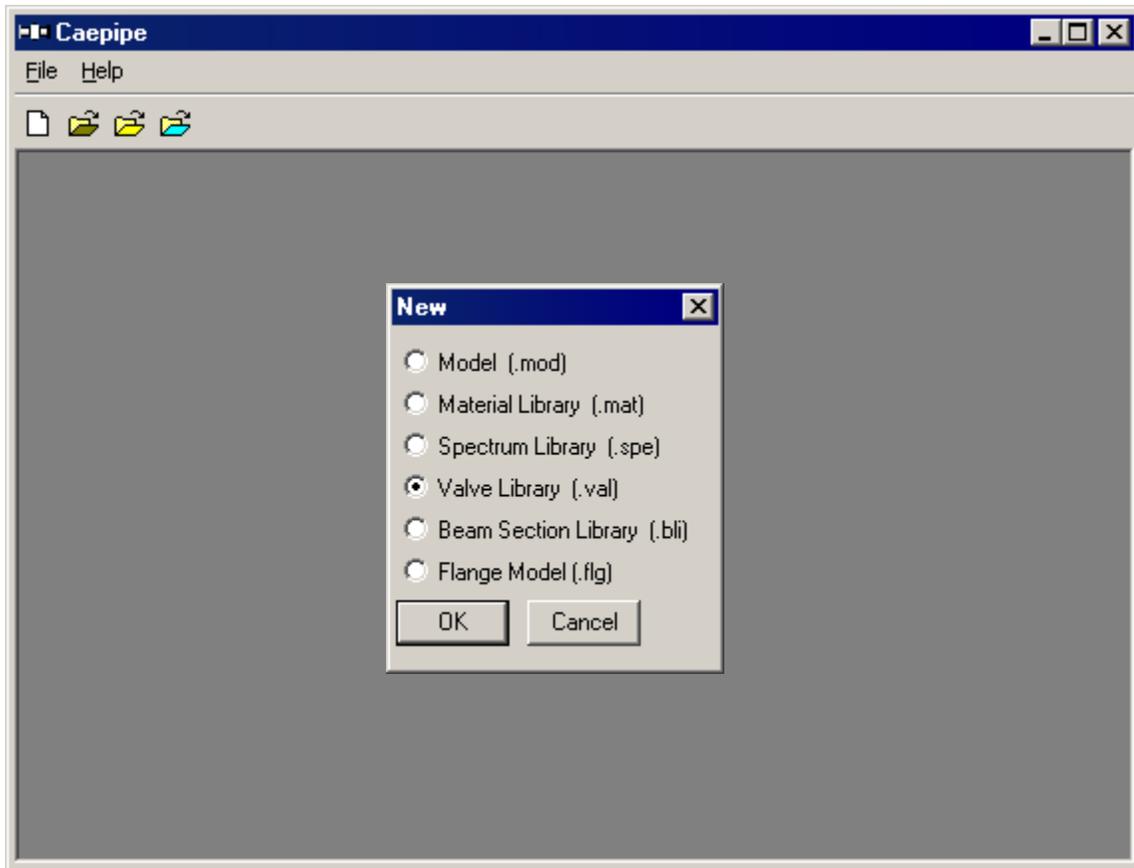
The additional weight is a concentrated weight which may be specified at an offset from the center of the valve, such as for a valve operator.

Valve Library

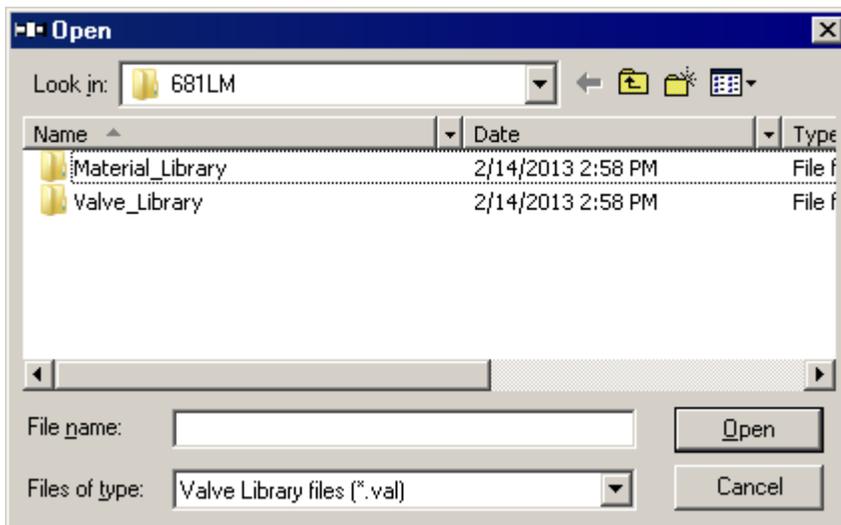
Cast Iron, Steel and Alloy valve (Flanged and Butt Welding ends) libraries are provided. The Type of Valve, Connection Type and Rating are indicative in the filenames listed in the libraries. You need to input only the weight of a valve. If necessary, you may create your own user-definable valve library. A new valve library can be created from menu File > New > Valve Library in the main opening CAEPIPE window.

Cast Iron, Steel and Alloy valve (Flanged and Butt Welding ends) libraries are provided. The Type of Valve, Connection Type and Rating are indicative in the filenames listed in the libraries. Valve weights are included for different categories of valves. If necessary, you may create your own user-definable valve library. A new valve library can be created from menu File > New > Valve Library in the main opening CAEPIPE window.

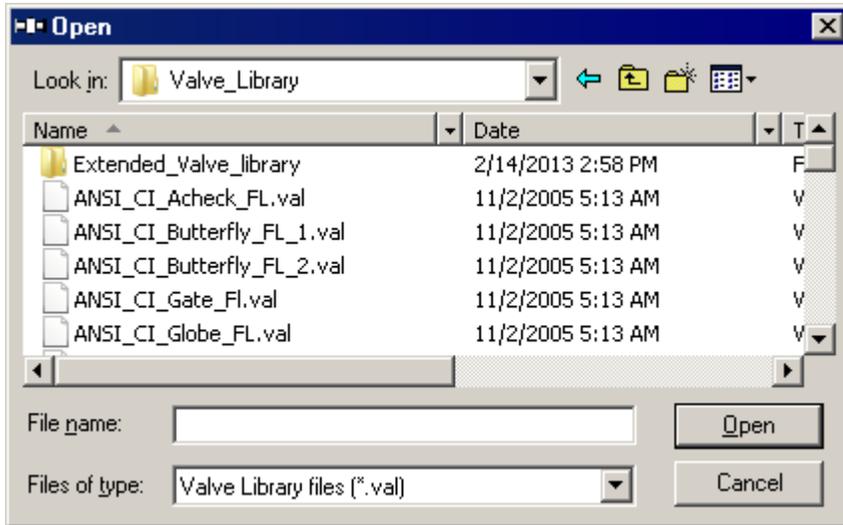
Valve



The valve library may be accessed by clicking on the Library button of the Valve dialog. Navigate to the folder called “Valve_Library” or similar under your CAEPIPE program files folder.



Valve

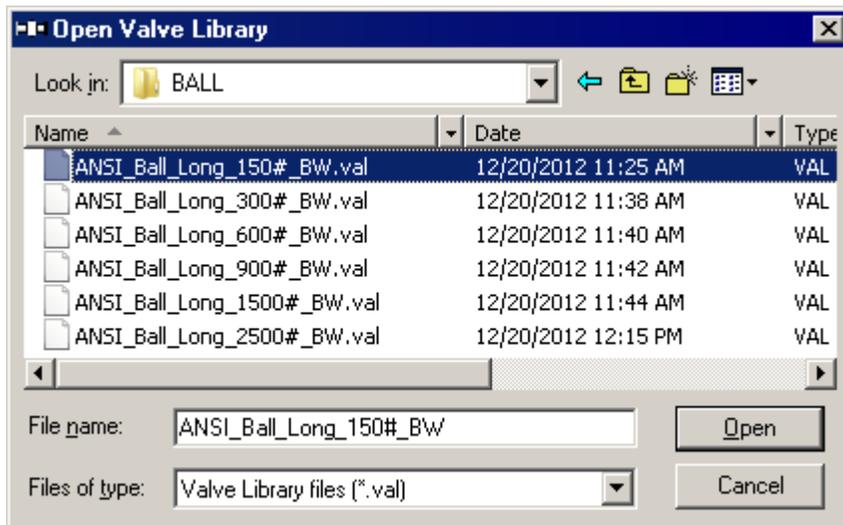
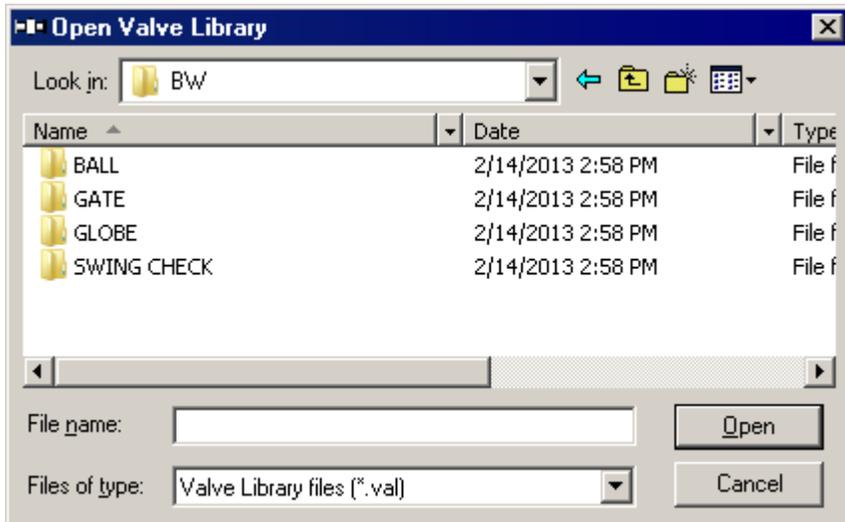
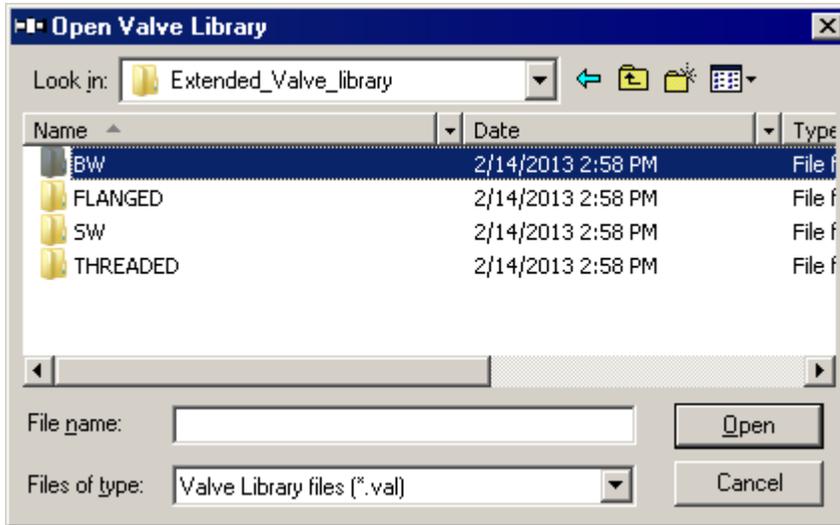


Selecting one of these valve types will display a list of valves for you to select from.

#	Description	Nom Dia	OD (inch)	Rating	Connection Type	Length (ft'in')	Weight (lb)
5	Gate Sol. Wed. & Dou.Disk	1"	1.315	150	FL	0'5"	
6	Gate Sol. Wed. & Dou.Disk	1-1/4"	1.66	150	FL	0.4593	
7	Gate Sol. Wed. & Dou.Disk	1-1/2"	1.9	150	FL	0.5413	
8	Gate Sol. Wed. & Dou.Disk	2"	2.375	150	FL	0'7"	
9	Gate Sol. Wed. & Dou.Disk	2-1/2"	2.875	150	FL	0.6234	
10	Gate Sol. Wed. & Dou.Disk	3"	3.5	150	FL	0.6660	
11	Gate Sol. Wed. & Dou.Disk	4"	4.5	150	FL	0.7513	
12	Gate Sol. Wed. & Dou.Disk	5"	5.563	150	FL	0'10"	
13	Gate Sol. Wed. & Dou.Disk	6"	6.625	150	FL	0.8760	
14	Gate Sol. Wed. & Dou.Disk	8"	8.625	150	FL	0.9580	
15	Gate Sol. Wed. & Dou.Disk	10"	10.75	150	FL	1.0827	
16	Gate Sol. Wed. & Dou.Disk	12"	12.75	150	FL	1.1680	

Select one from the displayed list. The weights of the valves (in the Extended Library) are provided. If a valve is flanged, the flanges at the two ends must be separately input using the Flange data type and their weight accounted for. If you however import a flanged valve from the library, it may already have that weight built-in. Please confirm the data with your valve manufacturer's catalog, and input the correct weight.

Valve



Calculation of Moment of Inertia:

The inside diameter of the valve is calculated from the section O.D. (outside dia.) and the section thickness.

$$\text{I.D.} = \text{O.D.} - 2 \times \text{Thickness}$$

Then a new O.D. is calculated as:

$$\text{New O.D.} = \text{I.D. (inside dia.)} + 2 \times \text{Thickness} \times \text{Multiplier}$$

The moment of inertia is based on the new O.D. and Thickness x Multiplier. For a thin-walled pipe, Thickness and Inertia multipliers are approximately the same. The weight of the contents of the valve is based on the I.D. as calculated above. The weight of the insulation is the weight calculated from the section O.D. and insulation thickness and density (from section properties) multiplied by the insulation weight multiplier.

Angle/Relief Valve

Angle and relief valves which have the outlet pipe at some angle (typically 90°) from the inlet pipe may be modeled by two valves one after the other and at that angle. The total weight of the actual valve must be divided between these two modeled valves.

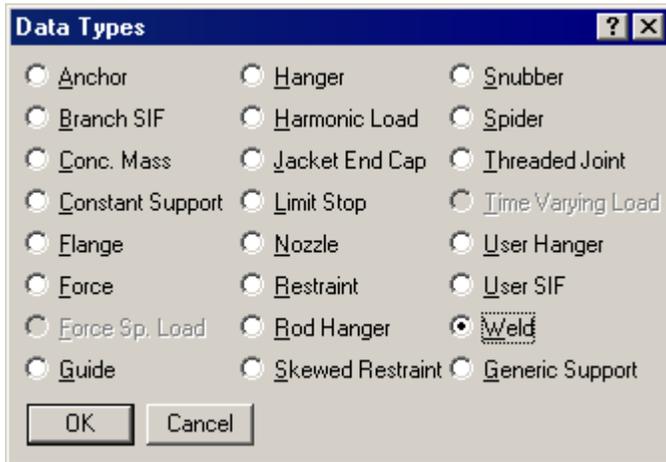
Analysis note: For a relief valve, you need to calculate the relief valve force and input it at the valve node as a Force (CAEPIPE treats it as a sustained load). Then, input a dummy g-load (e.g., 0.01g) and select Static Seismic for analysis. In results, review the occasional stresses which will include the effects of Sustained and Occasional loads. You will have to ignore the excessive Sustained stress at the node where the relief valve load (via a Force) was input. Examine the occasional stress with respect to the occasional allowable ($1.2S_h$), not the Sustained allowable (S_h).

Performing the above analyses in two separate runs will help.

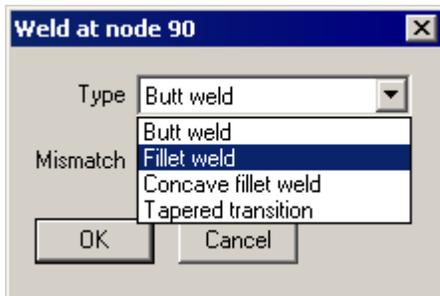
Also, see discussion towards end of the “Force Spectrum Analysis” topic.

Weld

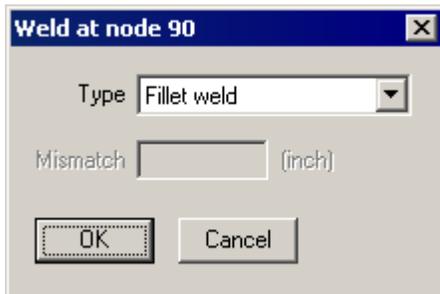
Use this data type to input a Weld at a node. Type “w” in the Data column or select “Weld” from the Data types dialog.



The Weld dialog is shown. Four types of welds are available: Butt, Fillet, Concave Fillet, and a Tapered transition. The type of the weld should be selected from the “Type” drop-down combo box.



Butt weld and Tapered transition require the input of weld mismatch. Mismatch is the difference in the mean radii across the weld.



The SIF for a weld is calculated according to the selected piping code (see Appendix A for details). If you have an unlisted weld type, you could specify the SIF for it using the User SIF data type.

This page is blank

Appendix A

Code Compliance

Allowable Pressure

At this time, there is no provision in CAEPIPE to specify the type of pipe construction, i.e., whether the pipe is a seamless or longitudinal welded or spiral welded. Accordingly, irrespective of the type of pipe construction, CAEPIPE calculates allowable pressure as follows.

For straight pipes and bends with seamless construction or designed for sustained operation below the creep range, Eq. (9) of para.104.1.2 is used as given below to compute allowable pressure.

$$P_a = \frac{2SEt_a}{D_o - 2Yt_a}$$

For straight pipes and bends designed for sustained operation within the creep range, Eq. (11) of para.104.1.4 is used as given below to calculate allowable pressure.

$$P_a = \frac{2SEWt_a}{D_o - 2Yt_a}$$

where

P_a = allowable pressure

SE = allowable stress as given in Appendix A of B31.1 (2012) Code, where

E = weld joint efficiency factor or casting quality factor as given in Table 102.4.3

t_a = available thickness for pressure design = $t_n \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

(Any additional thickness required for threading, grooving, erosion, corrosion, etc., should be included in corrosion allowance in CAEPIPE)

t_n = nominal pipe thickness

D_o = outside diameter of pipe

d = inside diameter of pipe

The Pressure coefficient Y is implemented as per Table 104.1.2 (A). In addition,

Y = 0.0, for cast iron and non-ferrous materials.

$Y = Y = \frac{d}{d+D_o}$, if $D_o/t_n < 6$, for ferritic and austenitic steels designed for temperatures of 900°F (480°C) and below

W = weld strength reduction factor as per Table 102.4.7. Refer to Annexure B for details on Weld strength reduction factor implemented in CAEPIPE.

For closely spaced miter bends, the allowable pressure is calculated from Eq. (C.3.1) of para.104.3.3.

$$P_a = \frac{SEt_a(R - r)}{r(R - r/2)}$$

where

r = mean radius of pipe = $(D_o - t_n)/2$

R = equivalent bend radius of the miter

For widely spaced miter bends, the allowable pressure is calculated from Eq. (C.3.2) of para. 104.3.3.

$$P_a = \frac{SEt_a^2}{r(t_a + 1.25 \tan \theta \sqrt{rt_a})}$$

Where, θ = miter half angle

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Eq. 15 of para.104.8.1

$$S_L = \frac{PD_o}{4t_n} + \frac{0.75iM_A}{Z} \leq S_h$$

where

P = maximum of CAEPIPE pressures P1 through P10

D_o = outside diameter

t_n = nominal wall thickness

i = stress intensification factor. The product $0.75i$ shall not be less than 1.0.

M_A = resultant bending moment due to weight and other sustained loads

Z = uncorroded section modulus; for reduced outlets, effective section modulus as per para. 104.8.4

S_h = hot allowable stress at maximum CAEPIPE temperature [i.e., at max (Tref, T1 through T10)]

Occasional Stress

The stress (S_{Lo}) due to occasional loads is calculated from Eq. 16 of para.104.8.2 as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_o) such as earthquake or wind. Wind and earthquake are not considered concurrently.

$$S_{Lo} = \frac{P_{peak} D_o}{4t_n} + \frac{0.75iM_A}{Z} + \frac{0.75iM_B}{Z} \leq 1.2S_h$$

where

M_B = resultant bending moment on the cross-section due to occasional loads such as thrusts from relief / safety valve loads, from pressure and flow transients, earthquake, wind etc.

P_{peak} = peak pressure = (peak pressure factor in CAEPIPE) x P

Expansion Stress Range (i.e., Stress due to Displacement Load Range)

The stress (S_E) due to thermal expansion is calculated from Eq. 17 of para.104.8.3.

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

M_C = resultant moment due to thermal expansion

S_A = $f(1.25S_C + 0.25S_h)$, from Eq. (1A) of para. 102.3.2 (B)

f = cyclic stress range reduction factor from Eq.(1C) of para. 102.3.2(B),

$f = \frac{6}{N^{0.2}} \leq 1.0$ and $f \geq 0.15$ with N being the total number of equivalent reference displacement stress range cycles expected during the service life of the piping

S_C = basic allowable stress as minimum metal temperature expected during the displacement cycle under analysis

S_h = basic allowable stress as maximum metal temperature expected during the displacement cycle under analysis

When S_h is greater than S_L , the allowable stress range may be calculated as

$$S_A = f[1.25(S_C + S_h) - S_L], \text{ from Eq. (1B) of para. 102.3.2 (B)}$$

This is specified as an analysis option: “Use liberal allowable stresses”, in the menu Options->Analysis on the Code tab of CAEPIPE.

Note:

Refer to Thickness and Section Modulus towards the end of this appendix.

MANDATORY APPENDIX D

Table D-1 Flexibility and Stress Intensification Factors

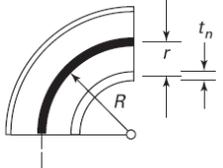
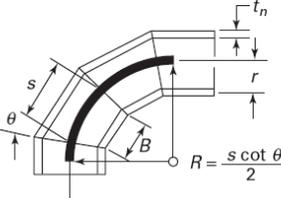
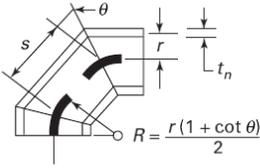
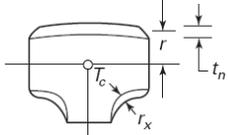
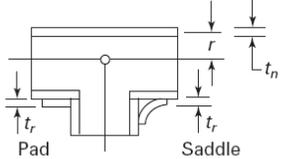
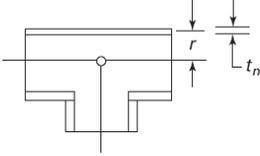
Description	Flexibility Characteristic, h	Flexibility Factor, k	Stress Intensification Factor, i	Sketch
Welding elbow or pipe bend [Notes (1), (2), (3), (4), (5)]	$\frac{t_n R}{r^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{2/3}}$	
Closely spaced miter bend [Notes (1), (2), (3), (5)] $s < r(1 + \tan \theta)$ $B \geq 6 t_n$ $\theta \leq 22\frac{1}{2}$ deg	$\frac{s t_n \cot \theta}{2r^2}$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	
Widely spaced miter bend [Notes (1), (2), (5), (6)] $s \geq r(1 + \tan \theta)$ $\theta \leq 22\frac{1}{2}$ deg	$\frac{t_n (1 + \cot \theta)}{2r}$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	
Welding tee per ASME B16.9 [Notes (1), (2), (7)]	$\frac{3.1 t_n}{r}$	1	$\frac{0.9}{h^{2/3}}$	
Reinforced fabricated tee [Notes (1), (2), (8), (9)]	$\frac{(t_n + \frac{t_r}{2})^{5/2}}{r (t_n)^{3/2}}$	1	$\frac{0.9}{h^{2/3}}$	
Unreinforced fabricated tee [Notes (1), (2), (9)]	$\frac{t_n}{r}$	1	$\frac{0.9}{h^{2/3}}$	

Table D-1 Flexibility and Stress Intensification Factors (Cont'd)

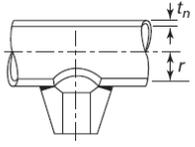
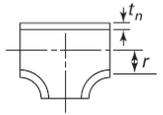
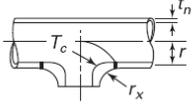
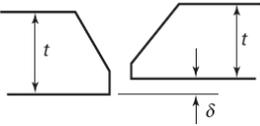
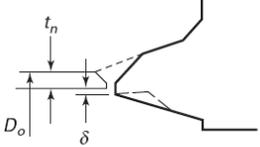
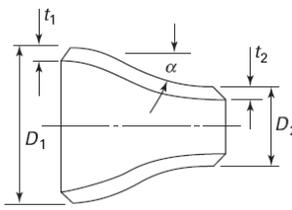
Description	Flexibility Characteristic, h	Flexibility Factor, k	Stress Intensification Factor, i	Sketch
Branch welded-on fitting (integrally reinforced) per MSS SP-97 [Notes (1), (2)]	$\frac{3.3t_n}{r}$	1	$\frac{0.9}{h^{2/3}}$	
Extruded outlet meeting the requirements of para. 104.3.1(G) [Notes (1), (2)]	$\frac{t_n}{r}$	1	$\frac{0.9}{h^{2/3}}$	
Welded-in contour insert [Notes (1), (2), (7)]	$3.1 \frac{t_n}{r}$	1	$\frac{0.9}{h^{2/3}}$	
Description	Flexibility Factor, k	Stress Intensification Factor, i		Sketch
Branch connection [Notes (1), (10)]	1	For checking branch end $1.5 \left(\frac{R_m}{t_{nh}} \right)^{2/3} \left(\frac{r'_m}{R_m} \right)^{1/2} \left(\frac{t_{nb}}{t_{nh}} \right) \left(\frac{r'_m}{r_p} \right)$		See Fig. D-1
Butt weld [Note (1)] $t \geq 0.237$ in., $\delta_{\max} \leq \frac{1}{16}$ in., and $\delta_{\text{avg}}/t \leq 0.13$	1	1.0 [Note (11)]		
Butt weld [Note (1)] $t \geq 0.237$ in., $\delta_{\max} \leq \frac{1}{8}$ in., and $\delta_{\text{avg}}/t = \text{any value}$	1	1.9 max. or $[0.9 + 2.7(\delta_{\text{avg}}/t)]$, but not less than 1.0 [Note (11)]		
Butt weld [Note (1)] $t < 0.237$ in., $\delta_{\max} \leq \frac{1}{16}$ in., and $\delta_{\text{avg}}/t \leq 0.33$	1			
Fillet welds	1	1.3 [Note (12)]		See Figs. 127.4.4(A), 127.4.4(B), and 127.4.4(C)
Tapered transition per para. 127.4.2(B) and ASME B16.25 [Note (1)]	1	1.9 max. or $1.3 + 0.0036 \frac{D_o}{t_n} + 3.6 \frac{\delta}{t_n}$		

Table D-1 Flexibility and Stress Intensification Factors (Cont'd)

Description	Flexibility Factor, k	Stress Intensification Factor, i	Sketch
Concentric reducer per ASME B16.9 [Note (13)]	1	2.0 max. or $0.5 + 0.01\alpha \left(\frac{D_2}{t_2}\right)^{1/2}$	
Threaded pipe joint or threaded flange	1	2.3	...
Corrugated straight pipe, or corrugated or creased bend [Note (14)]	5	2.5	...

NOTES:

- (1) The following nomenclature applies to Table D-1:
 - B = length of miter segment at crotch, in. (mm)
 - D_o = outside diameter, in. (mm)
 - D_{ob} = outside diameter of branch, in. (mm)
 - R = bend radius of elbow or pipe bend, in. (mm)
 - r = mean radius of pipe, in. (mm) (matching pipe for tees)
 - r_x = external crotch radius of welded-in contour inserts and welding tees, in. (mm)
 - s = miter spacing at centerline, in. (mm)
 - T_c = crotch thickness of welded-in contour inserts and welding tees, in. (mm)
 - t_n = nominal wall thickness of pipe, in. (mm) (matching pipe for tees)
 - t_r = reinforcement pad or saddle thickness, in. (mm)
 - α = reducer cone angle, deg
 - δ = mismatch, in. (mm)
 - θ = one-half angle between adjacent miter axes, deg
- (2) The flexibility factors k and stress intensification factors i in Table D-1 apply to bending in any plane for fittings and shall in no case be taken less than unity. Both factors apply over the effective arc length (shown by heavy centerlines in the sketches) for curved and miter elbows, and to the intersection point for tees. The values of k and i can be read directly from Chart D-1 by entering with the characteristic h computed from the formulas given.
- (3) Where flanges are attached to one or both ends, the values of k and i in Table D-1 shall be multiplied by the factor c given below, which can be read directly from Chart D-2, entering with the computed h : one end flanged, $c = h^{1/6}$; both ends flanged, $c = h^{1/3}$.
- (4) The designer is cautioned that cast butt welding elbows may have considerably heavier walls than those of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (5) In large diameter thin-wall elbows and bends, pressure can significantly affect magnitudes of k and i . Values from the Table may be corrected by dividing k by

$$\left[1 + 6 \left(\frac{P}{E_c}\right) \left(\frac{r}{t_n}\right)^{7/3} \left(\frac{R}{r}\right)^{1/3} \right]$$

and dividing i by

$$\left[1 + 3.25 \left(\frac{P}{E_c}\right) \left(\frac{r}{t_n}\right)^{5/2} \left(\frac{R}{r}\right)^{2/3} \right]$$

- (6) Also includes single miter joints.
- (7) If $r_x \geq D_{ob}/8$ and $T_c \geq 1.5t_n$, a flexibility characteristic, h , of $4.4t_n/r$ may be used.
- (8) When $t_r > 1.5t_n$, $h = 4.05t_n/r$.
- (9) The stress intensification factors in the Table were obtained from tests on full size outlet connections. For less than full size outlets, the full size values should be used until more applicable values are developed.

Table D-1 Flexibility and Stress Intensification Factors (Cont'd)

NOTES (Cont'd):

- (10) The equation applies only if the following conditions are met:
- (a) The reinforcement area requirements of para. 104.3 are met.
 - (b) The axis of the branch pipe is normal to the surface of run pipe wall.
 - (c) For branch connections in a pipe, the arc distance measured between the centers of adjacent branches along the surface of the run pipe is not less than three times the sum of their inside radii in the longitudinal direction or is not less than two times the sum of their radii along the circumference of the run pipe.
 - (d) The inside corner radius r_1 (see Fig. D-1) is between 10% and 50% of t_{nh} .
 - (e) The outer radius r_2 (see Fig. D-1) is not less than the larger of $T_b/2$, $(T_b + y)/2$ [shown in Fig. D-1 sketch (c)], or $t_{nh}/2$.
 - (f) The outer radius r_3 (see Fig. D-1) is not less than the larger of:
 - (1) $0.002\theta d_o$;
 - (2) $2(\sin \theta)^3$ times the offset for the configurations shown in Fig. D-1 sketches (a) and (b).
 - (g) $R_m/t_{nh} \leq 50$ and $r'_m/R_m \leq 0.5$.
- (11) The stress intensification factors apply to girth butt welds between two items for which the wall thicknesses are between $0.875t$ and $1.10t$ for an axial distance of $\sqrt{D_o t}$. D_o and t are nominal outside diameter and nominal wall thickness, respectively. δ_{avg} is the average mismatch or offset.
- (12) For welds to socket welded fittings, the stress intensification factor is based on the assumption that the pipe and fitting are matched in accordance with ASME B16.11 and a full weld is made between the pipe and fitting as shown in Fig. 127.4.4(C). For welds to socket welding flanges, the stress intensification factor is based on the weld geometry shown in Fig. 127.4.4(B) and has been shown to envelop the results of the pipe to socket welded fitting tests. Blending the toe of the fillet weld, with no undercut, smoothly into the pipe wall, as shown in the concave fillet welds in Fig. 127.4.4(A) sketches (b) and (d), has been shown to improve the fatigue performance of the weld.
- (13) The equation applies only if the following conditions are met:
- (a) Cone angle α does not exceed 60 deg, and the reducer is concentric.
 - (b) The larger of D_1/t_1 and D_2/t_2 does not exceed 100.
 - (c) The wall thickness is not less than t_1 throughout the body of the reducer, except in and immediately adjacent to the cylindrical portion on the small end, where the thickness shall not be less than t_2 .
- (14) Factors shown apply to bending; flexibility factor for torsion equals 0.9.

Chart D-1 Flexibility Factor, k , and Stress Intensification Factor, i

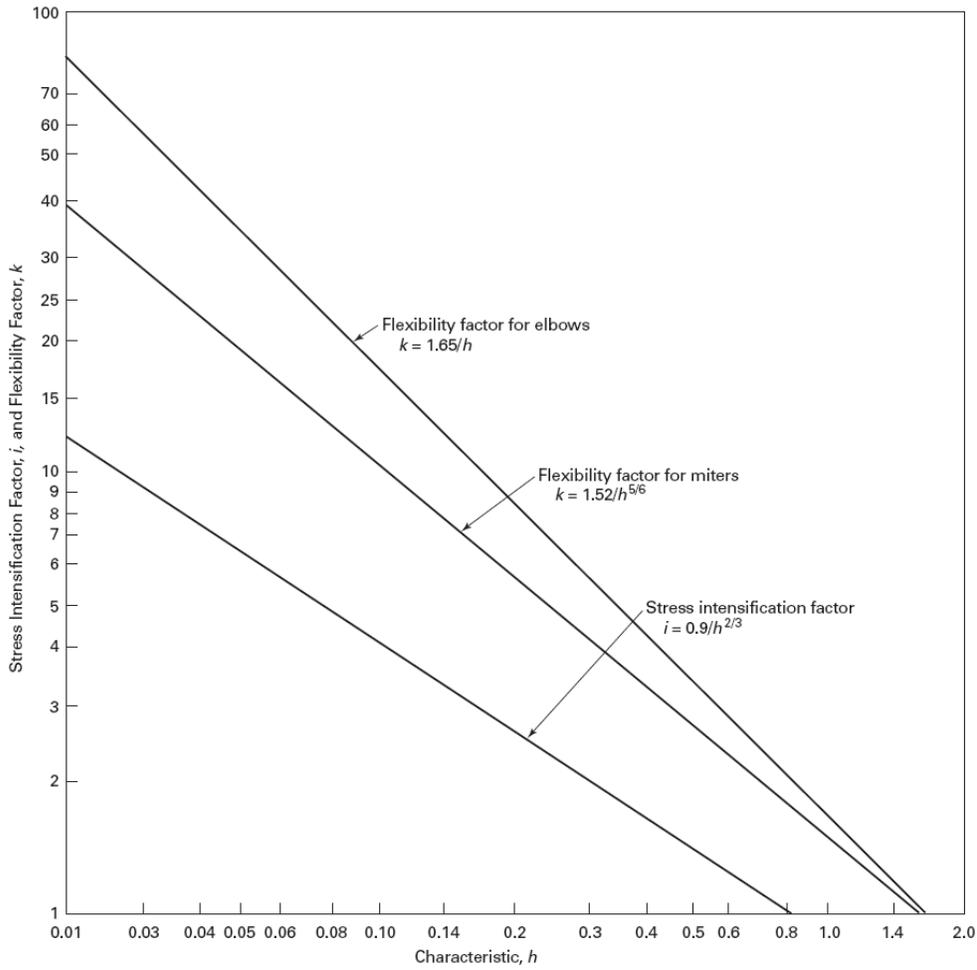
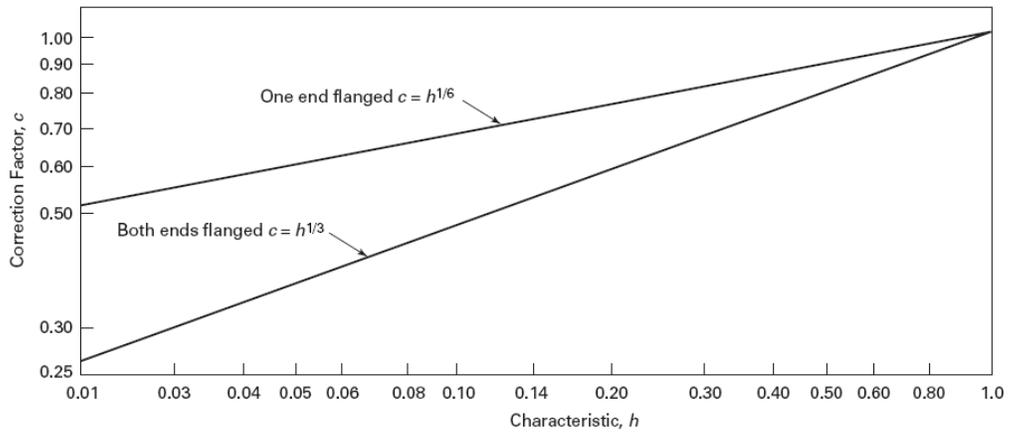


Chart D-2 Correction Factor, c



B31.1 (2012)

Weld Strength Reduction Factors applied for calculating the Allowable Design Pressure of components (extracted from Table 102.4.7 of ASME B31.1-2012).

Sl. No.	Steel Group	Weld Strength Reduction Factor for Temperature, °F (°C)										
		700	750	800	850	900	950	1000	1050	1100	1150	1200
		(371)	(399)	(427)	(454)	(482)	(510)	(538)	(566)	(593)	(621)	(649)
1	Carbon Steel (CS) with Max. Temp in CAEPIPE is ≤ 800 [see note 1 below]	1.00	0.95	0.91	NP							
2	Carbon Steel (CS) with Max. Temp in CAEPIPE > 800	-	-	1.00	0.95	0.91	0.86	0.82	0.77	0.73	0.68	0.64
3	Ferritic Steels (FS)	1.00	1.00	1.00	1.00	1.00	1.00	0.95	0.91	0.86	0.82	0.77
4	Austenitic Steel (AS) [contd. in note 2 below]	1.00	1.00	1.00	1.00	1.00	1.00	0.95	0.91	0.86	0.82	0.77
5	Materials other than those stated from Sl. Nos. 1 to 4	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00

Notes:

1. NP = Not permitted
2. For Austenitic Steels (including 800H and 800 HT) the values up to 1500°F are as follows:

Temperature, °F	Temperature, °C	Weld Strength Reduction Factor
1250	677	0.73
1300	704	0.68
1350	732	0.64
1400	760	0.59
1450	788	0.55
1500	816	0.50

Allowable Internal Pressure

For straight pipes and bends, the allowable pressure is calculated using Eq. (3a) for straight pipes and Eq. (3c) with $I = 1.0$ for bends from paras. 304.1.2. and 304.2.1. respectively.

$$P_a = \frac{2SEWt_a}{D - 2Yt_a}$$

where

P_a = allowable pressure

S = allowable stress as provided in para. 302.3.1 (a) and as per Table A-1

E = joint factor (input as material property) from Table A-1A or A-1B from para. 302.3.3. and para. 302.3.4.

W = Weld Joint Strength Reduction Factor from para. 302.3.5 (e) and as per Table 302.3.5 is implemented in CAEPIPE as follows. T_{max} below denotes maximum operating temperature (i.e., max of T_1 through T_{10} and T_{ref} in CAEPIPE).

With Material Type in CAEPIPE = CS [CrMo]

$W = 1.0$ with $T_{max} \leq 800^{\circ}F$ (or $427^{\circ}C$)

$W = 0.64$ with $T_{max} > 1200^{\circ}F$ (or $649^{\circ}C$) and

For $T_{max} > 800^{\circ}F$ (or $427^{\circ}C$) and $\leq 1200^{\circ}F$ (or $649^{\circ}C$), the values of W are taken from Table 302.3.5.

W for intermediate temperatures are linearly interpolated.

With Material Type in CAEPIPE = FS [CSEF (Subcritical)]

$W = 1.0$ with $T_{max} \leq 900^{\circ}F$ (or $482^{\circ}C$)

$W = 0.5$ with $T_{max} > 900^{\circ}F$ (or $482^{\circ}C$)

With Material Type in CAEPIPE = AS or NA

$W = 1.0$ with $T_{max} \leq 950^{\circ}F$ (or $510^{\circ}C$)

For $T_{max} > 950^{\circ}F$ (or $510^{\circ}C$), the values of W are taken as per Table 302.3.5.

W for intermediate temperatures are linearly interpolated.

With Material Type in CAEPIPE = SS

$W = 1.0$ with $T_{max} \leq 1500^{\circ}F$ (or $816^{\circ}C$)

For Other Material Types in CAEPIPE

$W = 1.0$ with $T_{max} \leq 800^{\circ}F$ (or $427^{\circ}C$)

$W = 1 - 0.000909 (T_{max} - T_{cr})$ for $T_{max} > 800^{\circ}F$ (or $427^{\circ}C$) and $\leq 1500^{\circ}F$ (or $816^{\circ}C$)

where, T_{cr} is taken as $800^{\circ}F$

t_a = available thickness for pressure design

= $t_n \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance "c"}$

(Any additional thickness required for threading, grooving, erosion, corrosion, etc. should be included in corrosion allowance in CAEPIPE)

t_n = nominal pipe thickness

D = outside diameter

d = inside diameter

Y = Pressure coefficient from Table 304.1.1, valid for $t_a < D/6$, and

$Y = \frac{d+2c}{D+d+2c}$, valid for $t_a \geq D/6$

For closely spaced miter bends, the allowable pressure is calculated using Eq. (4b) from para. 304.2.3.

$$P_a = \frac{SEWt_a(R - r)}{r(R - r/2)}$$

where

r = mean radius of pipe = $(D - t_n)/2$

R = effective bend radius of the miter (see para. 304.2.3 of code for definition)

For widely spaced miter bends, the allowable pressure is calculated using Eq. (4c) from para. 304.2.3 as

$$P_a = \frac{SEWt_a^2}{r(t_a + 1.25 \tan \theta \sqrt{rt_a})}$$

where

θ = miter half angle

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated using Eq. (23a) and (23b) from para. 320.2 and para. 302.3.5 (c).

$$S_L = \sqrt{(|S_a| + S_b)^2 + (2S_t)^2} \leq S_h$$

where

$$S_a = \left[\frac{I_a F_a}{A_p} \right]_{Sustained} = \left[\frac{PD}{4t_s} + \frac{R}{A_p} \right]_{Sustained}$$

$$S_b = \left[\frac{\sqrt{(I_i M_i)^2 + (I_o M_o)^2}}{Z_m} \right]_{Sustained}$$

$$S_t = \left[\frac{I_t M_t}{2Z_m} \right]_{Sustained}$$

P = maximum of CAEPIPE input pressures P1 through P10

D = outside diameter

t_s = wall thickness used for sustained stress calculation after deducting corrosion allowance from the nominal thickness

t_n = nominal thickness – corrosion allowance in CAEPIPE, as per para. 320.1

A_p = corroded cross-sectional area of the pipe computed using t_s as per para. 320.1.

I_a = longitudinal force index = 1.0

F_a = longitudinal force due to sustained loads (pressure and weight)

R = axial force due to weight

I_i = in-plane stress intensification factor; the product of $0.75i_i$ shall not be less than 1.0

I_o = out-of-plane stress intensification factor; the product of $0.75i_o$ shall not be less than 1.0

I_t = torsional moment index = 1.0

M_i = in-plane bending moment due to sustained loads e.g., pressure and weight

M_o = out-of-plane bending moment due to sustained loads e.g., pressure and weight

M_t = torsional moment due to sustained loads e.g., pressure and weight

Z_m = corroded section modulus as per para. 320.1; for reduced outlets / branch connections, effective section modulus

Z_m = hot allowable stress at maximum temperature [i.e., at Max(T_{ref}, T₁ through T₁₀)]

Sustained plus Occasional Stress

The stress (S_{Lo}) due to sustained and occasional loads is calculated as the sum of stress due to sustained loads such as due to pressure and weight (S_L) and stress due to occasional loads (S_O) such as due to earthquake or wind. Wind and earthquake are not considered concurrently (see para. 302.3.6(a)).

For temp ≤ 427° C or 800° F

$$S_{Lo} \leq 1.33S_h$$

For temp > 427° C or 800° F

$$S_{Lo} \leq 0.9WS$$

where

$S_{Lo} = S_L + S_O$, where S_L is computed as above, and

$$S_O = \sqrt{(|S_{ao}| + S_{bo})^2 + (2S_{to})^2}$$

$$S_{ao} = \left[\frac{I_a F_a}{A_p} \right]_{Occasional} = \left[\frac{(P_{peak} - P)D}{4t_s} + \frac{R}{A_p} \right]_{Occasional}$$

$$S_{bo} = \left[\frac{\sqrt{(I_i M_i)^2 + (I_o M_o)^2}}{Z_m} \right]_{Occasional}$$

$$S_{to} = \left[\frac{I_t M_t}{2Z_m} \right]_{Occasional}$$

P_{peak} = peak pressure = (peak pressure factor in CAEPIPE) x P

R = axial force due to occasional loads such as earthquake or wind

M_i = in-plane bending moment due to occasional loads such as earthquake or wind

M_o = out-of-plane bending moment due to occasional loads such as earthquake or wind

M_t = torsional moment due to occasional loads such as earthquake or wind

S_y = yield strength at maximum temperature, i.e., max (T_{ref}, T₁ through T₁₀)

W = 1.0 for Austenitic stainless steel and 0.8 for all other materials as per para. 302.3.6(a)

Expansion Stress

The stress (S_E) due to thermal expansion is calculated using Eq. 17 from para. 319.4.4

$$S_E = \sqrt{(|S_a| + S_b)^2 + (2S_t)^2}$$

where

$$S_a = \left[\frac{I_a F_a}{A_p} \right]_{Expansion}$$

$$S_b = \left[\frac{\sqrt{(I_i M_i)^2 + (I_o M_o)^2}}{Z} \right]_{Expansion}$$

$$S_t = \left[\frac{I_t M_t}{2Z} \right]_{Expansion}$$

A = un-corroded cross-sectional area of the pipe/fitting computed using nominal thickness t_n and outer diameter D , as per para. 319.3.5.

I_a = axial stress intensification factor = 1.0 for elbows, pipe bends and miter bends and $I_a = I_o$ for other components as listed in Appendix D of B31.3 (2012)

F_a = range of axial forces due to displacement strains between any two thermal conditions being evaluated

I_i = in-plane stress intensification factor; the product of $0.75i_i$ shall not be less than 1.0

I_o = out-of-plane stress intensification factor; the product of $0.75i_o$ shall not be less than 1.0

I_t = torsional stress intensification factor = 1.0

M_i = in-plane bending moment

M_o = out-of-plane bending moment

M_t = torsional moment

Z = uncorroded section modulus as per para. 319.3.5; for reduced outlets/branch connections, effective section modulus as per para. 319.4.4 (c)

S_A = $S_A = f(1.25S_C + 0.25S_h)$, Eq. (1a) of para. 302.3.5(d)

f = stress range reduction factor from Eq. (1c) of para. 302.3.5 (d) = $6N^{-0.2}$ where $f \geq 0.15$ and $f \leq 1.0$ (see Note 1 below)

S_C = basic allowable stress as minimum metal temperature expected during the displacement cycle under analysis

S_h = basic allowable stress as maximum metal temperature expected during the displacement cycle under analysis

When S_h is greater than S_L , the allowable stress range may be calculated as

$S_A = f[1.25(S_C + S_h) - S_L]$, Eq. (1b) of para. 302.3.5(d).

This is specified as an analysis option “Use liberal allowable stresses”, in the menu Layout > Options > Analysis > Code tab.

Notes:

1. As per para. 302.3.5 (d), f = maximum value of stress range factor; 1.2 for ferrous materials with specified minimum tensile strengths ≤ 517 MPa (75 ksi) and at Metal temperatures $\leq 3710^\circ\text{C}$ (7000°F). This criterion is not implemented in CAEPIPE as the provision for entering the minimum tensile strength in material property is not available at this time. Hence $f \leq 1.0$ for all materials including Ferrous materials.
2. Refer end of this appendix for the details of “Thickness and Section Modulus used for weight, pressure and stress calculations”.

APPENDIX D FLEXIBILITY AND STRESS INTENSIFICATION FACTORS

Table D300¹ Flexibility Factor, *k*, and Stress Intensification Factor, *i*

Description	Flexibility Factor, <i>k</i>	Stress Intensification Factor [Notes (2), (3)]		Flexibility Characteristic, <i>h</i>	Sketch
		Out-of-Plane, <i>i_o</i>	In-Plane, <i>i_i</i>		
Welding elbow or pipe bend [Notes (2), (4)-(7)]	$\frac{1.65}{h}$	$\frac{0.75}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{TR_1}{r_2^2}$	
Closely spaced miter bend $s < r_2 (1 + \tan \theta)$ [Notes (2), (4), (5), (7)]	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{\cot \theta}{2} \left(\frac{sT}{r_2^2} \right)$	
Single miter bend or widely spaced miter bend $s \geq r_2 (1 + \tan \theta)$ [Notes (2), (4), (7)]	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{1 + \cot \theta}{2} \left(\frac{T}{r_2} \right)$	
Welding tee in accordance with ASME B16.9 [Notes (2), (4), (6), (8), (9)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4}i_o + \frac{1}{4}$	$3.1 \frac{T}{r_2}$	
Reinforced fabricated tee with pad or saddle [Notes (2), (4), (9), (10), (11)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4}i_o + \frac{1}{4}$	$\frac{(T + \frac{1}{2} T_r)^{2.5}}{T^{1.5} r_2}$	

Table D300¹ Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

(1)

Description	Flexibility Factor, k	Stress Intensification Factor [Notes (2), (3)]		Flexibility Characteristic, h	Sketch
		Out-of-Plane, i_o	In-Plane, i_i		
Unreinforced fabricated tee [Notes (2), (4), (9), (11)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4}i_o + \frac{1}{4}$	$\frac{T}{r_2}$	
Extruded welding tee with $r_x \geq 0.05 D_b$ $T_c < 1.5 \bar{T}$ [Notes (2), (4), (9)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4}i_o + \frac{1}{4}$	$\left(1 + \frac{r_x}{r_2}\right) \frac{\bar{T}}{r_2}$	
Welded-in contour insert [Notes (2), (4), (8), (9)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4}i_o + \frac{1}{4}$	$3.1 \frac{\bar{T}}{r_2}$	
Branch welded-on fitting (integrally reinforced) [Notes (2), (4), (11), (12)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$3.3 \frac{\bar{T}}{r_2}$	

Description	Flexibility Factor, k	Stress Intensification Factor, i [Note (1)]
Butt welded joint, reducer, or weld neck flange	1	1.0
Double-welded slip-on flange	1	1.2
Fillet or socket weld	1	1.3 [Note (13)]
Lap joint flange (with ASME B16.9 lap joint stub)	1	1.6
Threaded pipe joint or threaded flange	1	2.3
Corrugated straight pipe, or corrugated or creased bend [Note (14)]	5	2.5

Table D300¹ Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

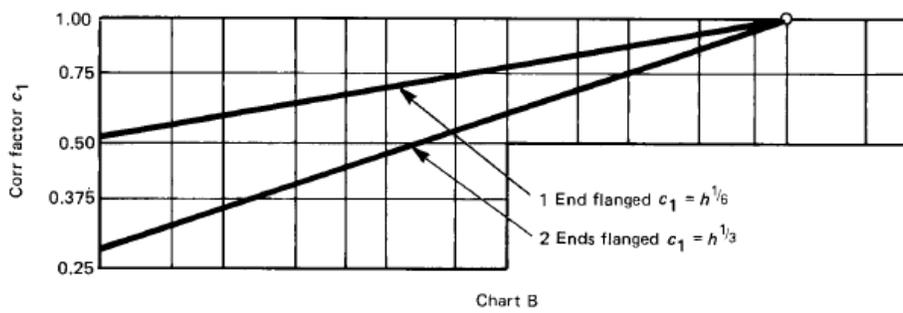
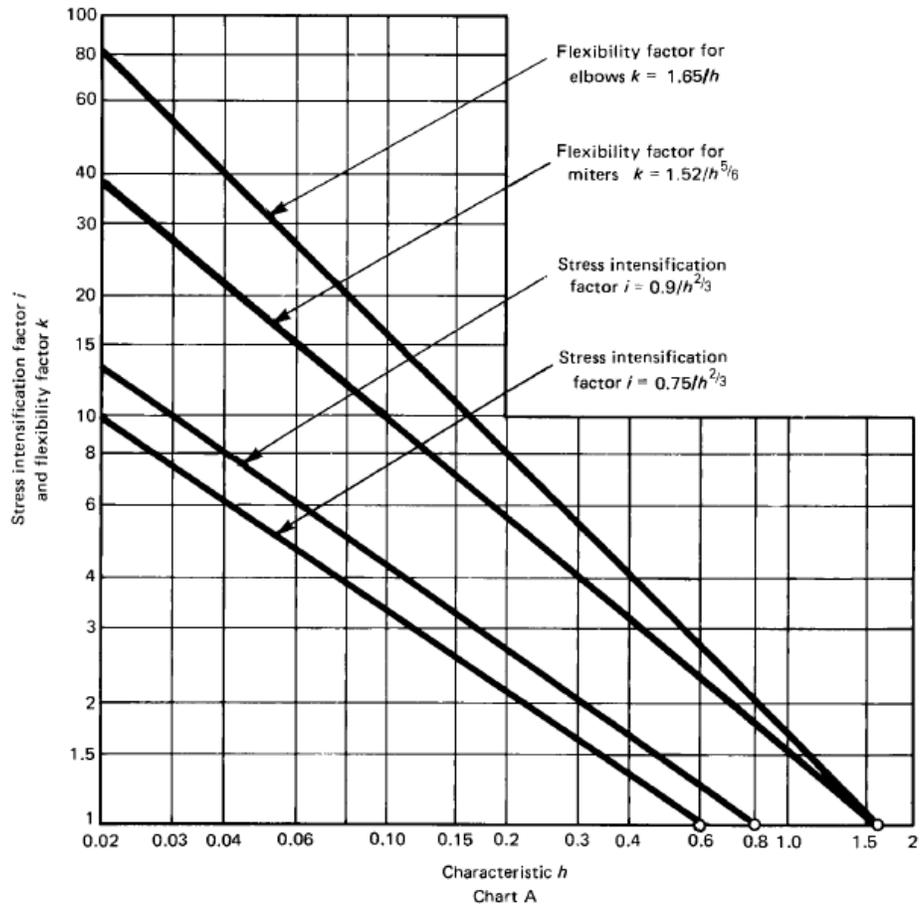


Table D300¹ Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

(11)

NOTES:

- (1) Stress intensification and flexibility factor data in Table D300 are for use in the absence of more directly applicable data (see para. 319.3.6). Their validity has been demonstrated for $D/\bar{T} \leq 100$.
- (2) The flexibility factor, k , in the Table applies to bending in any plane. The flexibility factors, k , and stress intensification factors, i , shall not be less than unity; factors for torsion equal unity. Both factors apply over the effective arc length (shown by heavy centerlines in the sketches) for curved and miter bends, and to the intersection point for tees.
- (3) A single intensification factor equal to $0.9/h^{2/3}$ may be used for both i_i and i_o if desired.
- (4) The values of k and i can be read directly from Chart A by entering with the characteristic h computed from the formulas given above. Nomenclature is as follows:
 - D_b = outside diameter of branch
 - R_1 = bend radius of welding elbow or pipe bend
 - r_x = see definition in para. 304.3.4(c)
 - r_2 = mean radius of matching pipe
 - s = miter spacing at centerline
 - \bar{T} = for elbows and miter bends, the nominal wall thickness of the fitting
 - = for tees, the nominal wall thickness of the matching pipe
 - T_c = crotch thickness of branch connections measured at the center of the crotch where shown in the sketches
 - \bar{T}_r = pad or saddle thickness
 - θ = one-half angle between adjacent miter axes
- (5) Where flanges are attached to one or both ends, the values of k and i in the Table shall be corrected by the factors C_1 , which can be read directly from Chart B, entering with the computed h .
- (6) The designer is cautioned that cast butt-welded fittings may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (7) In large diameter thin-wall elbows and bends, pressure can significantly affect the magnitudes of k and i . To correct values from the Table, divide k by

$$1 + 6 \left(\frac{P_i}{E_i} \right) \left(\frac{r_2}{\bar{T}} \right)^{2/3} \left(\frac{R_1}{r_2} \right)^{1/3}$$

divide i by

$$1 + 3.25 \left(\frac{P_i}{E_i} \right) \left(\frac{r_2}{\bar{T}} \right)^{5/2} \left(\frac{R_1}{r_2} \right)^{2/3}$$

For consistency, use kPa and mm for SI metric, and psi and in. for U.S. customary notation.

- (8) If $r_x \geq \frac{1}{8} D_b$ and $T_c \geq 1.5\bar{T}$, a flexibility characteristic of $4.4\bar{T}/r_2$ may be used.
- (9) Stress intensification factors for branch connections are based on tests with at least two diameters of straight run pipe on each side of the branch centerline. More closely loaded branches may require special consideration.
- (10) When \bar{T}_r is $> 1\frac{1}{2}\bar{T}$, use $h = 4\bar{T}/r_2$.
- (11) The out-of-plane stress intensification factor (SIF) for a reducing branch connection with branch-to-run diameter ratio of $0.5 < d/D < 1.0$ may be nonconservative. A smooth concave weld contour has been shown to reduce the SIF. Selection of the appropriate SIF is the designer's responsibility.
- (12) The designer must be satisfied that this fabrication has a pressure rating equivalent to straight pipe.
- (13) For welds to socket welded fittings, the stress intensification factor is based on the assumption that the pipe and fitting are matched in accordance with ASME B16.11 and a fillet weld is made between the pipe and fitting as shown in Fig. 328.5.2C. For welds to socket welded flanges, the stress intensification factor is based on the weld geometry shown in Fig. 328.5.2B sketch (3) and has been shown to envelope the results of the pipe to socket welded fitting tests. Blending the toe of the fillet weld smoothly into the pipe wall, as shown in the concave fillet welds in Fig. 328.5.2A, has been shown to improve the fatigue performance of the weld.
- (14) Factors shown apply to bending. Flexibility factor for torsion equals 0.9.

B31.1 (1967)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from 104.1.2.

$$P = \frac{2SEt_m}{D - 2Yt_m}$$

where

P = allowable pressure

S = allowable stress

E = joint factor (input as material property)

t_m = minimum required thickness, including mechanical, corrosion, and erosion allowances

= $t \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

t = nominal pipe thickness

D = outside diameter

d = inside diameter

Y = pressure coefficient from Table 104.1.2(a)2

Sustained Stress

The stress S_L due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from 102.3.2(d)

$$S_L = \frac{PD}{4t_m} + \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z} \leq S_h$$

where

P = maximum pressure

D = outside diameter

t_m = minimum wall thickness

i_i = in-plane stress intensification factor

i_o = out of plane stress intensification factor

M_i = in-plane bending moment

M_o = out of plane bending moment

Z = section modulus

S_h = hot allowable stress

Occasional Stress

The stress S_{LO} is calculated as the sum of stress due to sustained loads S_L and stress due to occasional loads S_O such as earthquake or wind. Wind and earthquake are not considered concurrently (102.3.1).

$$S_{LO} = S_L + \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z} \leq 1.2S_h$$

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from (119.6.4).

$$S_E = \sqrt{S_b^2 + 4S_t^2} \leq S_A$$

where

$$S_b = \text{resultant bending stress} = \sqrt{(i_i M_i)^2 + (i_o M_o)^2} / Z$$

$$S_t = \text{torsional stress} = M_t / 2Z$$

Z = uncorroded section modulus, for a branch, effective section modulus

$$S_A = f(1.25S_c + 0.25S_h)$$

S_c = allowable stress at cold temperature

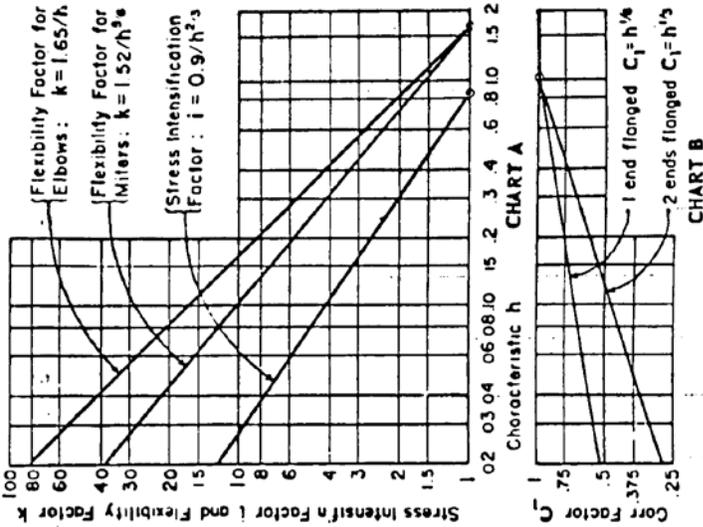
f = stress range reduction factor from Table 102.3.2(c).

When S_h is greater than S_L , the allowable stress range may be calculated as

$$S_A = f[1.25(S_c + S_h) - S_L]$$

This is specified as an option (Use liberal allowable stresses) in the menu Options > Analysis on the Code tab.

FLEXIBILITY AND STRESS INTENSIFICATION FACTORS



Description	Flexibility Factor k	Stress Int. Factor i	Flexibility Characteristic h	Sketch
Welding elbow, ^{1,2,3} or pipe bend	$\frac{1.65}{h}$	$\frac{0.9}{h^{2.3}}$	$\frac{\bar{T}R_1}{(r_2)^2}$	
Closely spaced miter bend ^{1,2,3} $s < r_2 (1 + \tan \theta)$	$\frac{1.52}{h^{1.5}}$	$\frac{0.9}{h^{2.5}}$	$\frac{\cot \theta \bar{T} s}{2(r_2)^2}$	
Widely spaced miter bend ^{1,2,4} $s \geq r_2 (1 + \tan \theta)$	$\frac{1.52}{h^{1.5}}$	$\frac{0.9}{h^{2.5}}$	$\frac{1 + \cot \theta \bar{T}}{2} \frac{\bar{T}}{r_2}$	
Welding tee ^{1,2,6} per USAS B16.9	1	$\frac{0.9}{h^{2.5}}$	$4.4 \frac{\bar{T}}{r_2}$	
Reinforced fabricated tee, ^{1,2,6,8} with pad or saddle	1	$\frac{0.9}{h^{2.5}}$	$\frac{(\bar{T} + \frac{1}{2} t_e)^{3/2}}{\bar{T}^{3/2} r_2}$	
Unreinforced ^{1,2,6} fabricated tee	1	$\frac{0.9}{h^{2.5}}$	$\frac{\bar{T}}{r_2}$	
Butt-welded joint, reducer, or welded neck flange	1	1.0		
Double welded slip-on flange	1	1.2		
Fillet welded joint, socket welded flange, or single welded slip-on flange	1	1.3		
Lap joint flange (with USAS B16.9 lap joint stub)	1	1.6		
Threaded pipe joint, or threaded flange	1	2.3		
Corrugated straight pipe, or corrugated or creased bend ⁵	5	2.5		

¹The flexibility factors k and stress intensification factors i in the table apply to bending in any plane for fittings shall in no case be taken less than unity; factors for torsion equal unity. Both factors apply over the effective arc length (shown by heavy center lines in the sketches) for curved and miter elbows, and to the intersection point for tees.

²The values of k and i can be read directly from Chart A above by entering with the characteristic h computed from the formulas given, where:

r_2 = mean radius of matching pipe, inches
 \bar{T} = for elbows and miter bends, the nominal wall thickness of the fitting (see note 7), inches.
 \bar{T} = for tees, the nominal wall thickness of the matching pipe, inches.

³Where flanges are attached to one or both ends, the values of k and i in the table shall be corrected by the factors C_1 given below, which can be read directly from Chart B, entering with the computed h :

One end flanged: $C_1 = \frac{1}{2}$
 Both ends flanged: $C_1 = \frac{1}{4}$

⁴Also includes single-miter joint.
⁵Factors shown apply to bending; flexibility factor for torsion equals 0.9.

⁶The stress intensification factors i in the table were obtained from tests on full size outlet connections. For less than full size outlets, the full size values should be used until more applicable values are developed.

⁷The engineer is cautioned that cast butt-welding elbows may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.

⁸When t_e is $> 1\frac{1}{2} \bar{T}$, use $h = 4.05 \frac{\bar{T}}{r_2}$

Allowable Pressure

For straight pipes and bends (including closely spaced and widely spaced miter bends), the allowable pressure is calculated from para. 403.2.1.

$$P_i = \frac{2SEt_a}{D}$$

where

P_i = allowable pressure

S = allowable stress = $0.72 S_y$

S_y = specified minimum yield strength of pipe

E = weld joint factor as defined in Table 403.2.1-1

t_a = available thickness for pressure design

= $t_n \times (1 - \text{mill tolerance}/100) - \text{sum of allowances, as per para. 403.2.1, for corrosion, threading, grooving and erosion.}$

D = outside diameter

Stress due to Sustained Loads (Unrestrained Piping)

For Pipes (as per para. 402.6.2)

$$S_L = \left| \frac{PD}{4t_n} + \frac{F_a}{A} \right|_{Sustained} + \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2 + (M_t)^2}}{Z} \right]_{Sustained} \leq 0.75 S_y$$

as per Table 403.3.1-1

where

i_i = in-plane stress intensification factor = 1.0 for pipes

i_o = out-of-plane stress intensification factor = 1.0 for pipes

For Fittings & Components (as per para. 402.6.2)

$$S_{L(fc)} = \left| \frac{PD}{4t_n} + \frac{F_a}{A} \right|_{Sustained} + \left[\frac{\sqrt{(0.75i_i M_i)^2 + (0.75i_o M_o)^2 + (M_t)^2}}{Z} \right]_{Sustained} \leq 0.75 S_y$$

as per Table 403.3.1-1

where

P = maximum operating pressure = max of CAEPIPE input pressures (P1 through P10). Due considerations shall be given as per para. 401.2.2.2 while inputting pressure values in CAEPIPE.

D = outside diameter

t_n = nominal thickness as per para. 402.1

i_i = in-plane stress intensification factor; the product $0.75i_i$ shall not be less than 1.0

i_o = out-of-plane stress intensification factor; the product $0.75i_o$ shall not be less than 1.0

M_i = in-plane bending moment

M_o = out-of-plane bending moment

M_t = torsional moment

- Z = uncorroded section modulus; for reduced outlets, effective section modulus
 F_a = axial force component for external loads
 A = nominal cross-section area
 S_y = specified minimum yield strength of pipe

Stress due to Sustained Loads + Occasional Loads (Unrestrained Piping)

For Pipes (as per para. 402.6.2)

$$S_{LO} = S_L + \left| \frac{(P_{peak} - P)D}{4t_n} + \frac{F_a}{A} \right|_{occasional} + \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2 + (M_t)^2}}{Z} \right]_{occasional} \leq 0.8S_y$$

as per Table 403.3.1-1

For Fittings & Components (as per para. 402.6.2)

$$S_{LO} = S_{L(fc)} + \left| \frac{(P_{peak} - P)D}{4t_n} + \frac{F_a}{A} \right|_{occasional} + \left[\frac{\sqrt{(0.75i_i M_i)^2 + (0.75i_o M_o)^2 + (M_t)^2}}{Z} \right]_{occasional} \leq 0.8S_y$$

as per Table 403.3.1-1

where

P_{peak} = peak pressure = (peak pressure factor x P) where P = maximum operating pressure, as defined above with $1.0 \leq$ peak pressure factor ≥ 1.1 as per para. 403.3.4

Expansion Stress (Unrestrained Piping)

The stress (S_E) due to thermal expansion is calculated from para.402.5.2

$$S_E = \sqrt{S_b^2 + 4S_t^2} \leq S_A \text{ as per Table 403.3.1-1 and para. 403.3.2}$$

where

S_b = resultant bending stress = $\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z}$

S_t = torsional stress = $\frac{M_t}{2Z}$

M_t = torsional moment

Z = uncorroded section modulus; for reduced outlets, effective section modulus

Please note, "Liberal allowable" option is always turned ON for ANSI B31.4

$S_A = f[1.25(S_C + S_h) - S_L]$

f = stress range reduction factor = $6/N^{0.2}$, where N = number of equivalent full range cycles

where $f \leq 1.2$ (from para. 403.3.2).

$S_C = 0.67S_y$ at the lower of the installed temperature or minimum operating temperature

= $0.67S_y$ at the higher of the installed temperature or maximum operating temperature

where

S_C = specified minimum yield strength of pipe

Stress due to Sustained, Thermal and Occasional Loads (Restrained Piping)

The Net longitudinal stress (S_L) due to sustained, thermal expansion and occasional loads for restrained piping is calculated from para. 402.6.1

$$S_L = \max(|S_p + S_x + S_B|, |S_p + S_x - S_B|)_{sustained} + \max(|S_p + S_x + S_B|, |S_p + S_x - S_B|)_{occasional} + \max(|S_T|_{warmest}, |S_T|_{coldest}) \leq 0.9S_y$$

where

Pressure stress = $S_p = v \frac{PD}{2t_n}$ where $v = 0.3$ as per para. 402.2.3 and can be either positive or negative

Stress due to axial loading (other than temperature and pressure) = $S_x = \frac{F_a}{A}$

and can be positive or negative.

Nominal bending stress S_B from Weight and / or other External loads for

For Pipes

$$S_B = \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2 + (M_t)^2}}{Z}$$

For Fittings & Components

$$S_B = \frac{\sqrt{(0.75 i_i M_i)^2 + (0.75 i_o M_o)^2 + (M_t)^2}}{Z}$$

Thermal expansion stress $S_T = E\alpha(T_i - T_o)$, which can either be positive or negative

where

P = maximum operating pressure = max (P_1 through P_{10})

D = outside diameter

t_n = nominal thickness

i_i = in-plane stress intensification factor; the product $0.75i_i$ shall not be less than 1.0

i_o = out-of-plane stress intensification factor; the product $0.75i_o$ shall not be less than 1.0

M_i = in-plane bending moment

M_o = out-of-plane bending moment

M_t = torsional moment

F_a = axial force component for external loads

A = nominal cross-section area

Z = uncorroded section modulus; for reduced outlets, effective section modulus

S_y = specified minimum yield strength of pipe

T_i = installation temperature = T_{ref} in CAEPIPE

T_o = warmest or coldest operating temperature

α = coefficient of thermal expansion at T_o defined above

E = young's modulus at ambient (reference) temperature

Note:

1. Para. 402.6.2 of B31.4 (2012) states that “Longitudinal stress from pressure in an unrestrained line should include consideration of bending stress or axial stress that may be caused by elongation of the pipe due to internal pressure and result in stress at bends and at connections and produce additional loads on equipment and on supports”.
The above statement seems to imply that “elongation of pipe and opening of bends due to Bourdon effect” are to be included in the Sustained load case (and hence in Operating case and Sustained plus Occasional load case).
On the other hand, since the deformation due to Bourdon effect is being constrained by piping supports, CAEPIPE includes the Bourdon effect as part of the results for Thermal Expansion (when “Solve Thermal Case” is opted) or as part of the Operating Case (when “Thermal = Operating – Sustained is opted).
2. Young’s modulus of elasticity corresponding to reference temperature (T_{ref}) is used to form the stiffness matrix in accordance with para. 402.2.2.
3. Refer Annexure B for the details of “Thickness” and the “Section Modulus” used for weight, pressure and stress calculations.

Table 402.1-1 Flexibility Factor, *k*, and Stress Intensification Factor, *i*

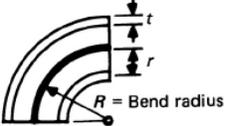
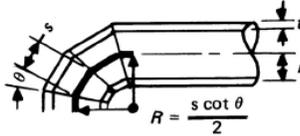
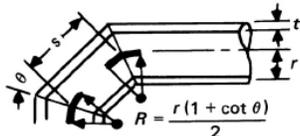
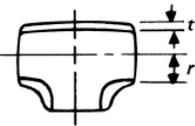
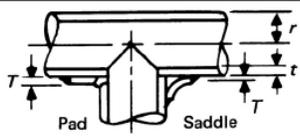
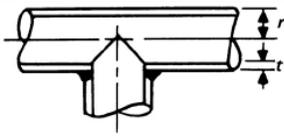
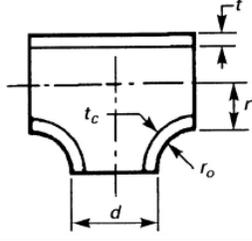
Description	Flexibility Factor, <i>k</i>	Stress Intensification Factor		Flexibility Characteristic, <i>h</i>	Sketch
		<i>i_i</i> [Note (1)]	<i>i_o</i> [Note (2)]		
Welding elbow, or pipe bend [Notes (3)–(7)]	$\frac{1.65}{h}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.75}{h^{2/3}}$	$\frac{tR}{r^2}$	
Closely spaced miter bend, [Notes (3)–(5), and (7)] <i>s</i> < <i>r</i> (1 + tan θ)	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.75}{h^{2/3}}$	$\frac{\cot \theta}{2} \frac{ts}{r^2}$	
Widely spaced miter bend, [Notes (3),(4), (7), and (8)] <i>s</i> ≥ <i>r</i> (1 + tan θ)	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.75}{h^{2/3}}$	$\frac{1 + \cot \theta}{2} \frac{t}{r}$	
Welding tee [Notes (3) and (4)] per ASME B16.9	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{2/3}}$	$4.4 \frac{t}{r}$	
Reinforced tee [Notes (3),(4), and (9)] with pad or saddle	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{2/3}}$	$\frac{(t + 1/2 T)^{5/2}}{t^{3/2} r}$	
Unreinforced fabricated tee [Notes (3) and (4)]	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{2/3}}$	$\frac{t}{r}$	
Extruded welding tee [Notes (3),(4), and (10)] <i>r_o</i> ≥ 0.05 <i>d</i> <i>t_c</i> < 1.5 <i>t</i>	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{2/3}}$	$\left(1 + \frac{r_o}{r}\right) \frac{t}{r}$	
Butt welded joint, reducer, or welding neck flange	1	1.0
Double welded slip-on flange	1	1.2
Fillet welded joint (single welded), or single welded slip-on flange	1	1.3

Table 402.1-1 Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

Description	Flexibility Factor, k	Stress Intensification Factor		Flexibility Characteristic, h	Sketch
		i_i [Note (1)]	i_o [Note (2)]		
Lapped flange (with ANSI B16.9 lap-joint stub)	1	1.6	
Threaded pipe joint, or threaded flange	1	2.3	
Corrugated straight pipe, or corrugated or creased bend [Note (11)]	5	2.5	

NOTES:

- (1) In-plane.
- (2) Out-of-plane.
- (3) For fittings and miter bends, the flexibility factors, k , and stress intensification factors, i , in the Table apply to bending in any plane and shall not be less than unity; factors for torsion equal unity. Both factors apply over the effective arc length (shown by heavy center lines in the sketches) for curved and miter elbows, and to the intersection point for tees.
- (4) The values of k and i can be read directly from Chart A by entering with the characteristic, h , computed from the equations given, where

- d = outside diameter of branch
- R = bend radius of welding elbow or pipe bend, in. (mm)
- r = mean radius of matching pipe, in. (mm)
- r_o = see Note (10)
- s = miter spacing at center line
- T = pad or saddle thickness, in. (mm)
- t = nominal wall thickness of: part itself, for elbows and curved or mited bends; matching pipe, for welding tees; run or header, for fabricated tees (provided that if thickness is greater than that of matching pipe, increased thickness must be maintained for at least one run O.D. to each side of the branch O.D.)
- t_c = the crotch thickness of tees
- θ = one-half angle between adjacent miter axes, deg

- (5) Where flanges are attached to one or both ends, the values of k and i in this Table shall be corrected by the factors C_1 given below, which can be read directly from Chart B, entering with the computed h : one end flanged, $h^{1/6} \geq 1$; both ends flanged, $h^{1/3} \geq 1$.
- (6) The engineer is cautioned that cast butt welding elbows may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (7) In large diameter thin wall elbows and bends, pressure can significantly affect the magnitude of flexibility and stress intensification factors. To correct values obtained from this Table for the pressure effect, divide

$$\text{Flexibility factor, } k, \text{ by } 1 + 6 \frac{P}{E_c} \left(\frac{r}{t}\right)^{7/3} \left(\frac{R}{r}\right)^{1/3}$$

$$\text{Stress Intensification factor, } i, \text{ by } 1 + 3.25 \frac{P}{E_c} \left(\frac{r}{t}\right)^{5/2} \left(\frac{R}{r}\right)^{2/3}$$

where

- E_c = cold modulus of elasticity
- P = gage pressure

- (8) Also includes single miter joint.
- (9) When $T > 1\frac{1}{2}t$, use $h = 4.05 t/r$.
- (10) Radius of curvature of external contoured portion of outlet measured in the plane containing the axes of the run and branch. This is subject to the following limitations:
 - (a) minimum radius, r_o : the lesser of $0.05d$ or 38 mm (1.5 in.)
 - (b) maximum radius, r_o shall not exceed
 - (1) for branches DN 200 (NPS 8) and larger, $0.10d + 13$ mm (0.50 in.)
 - (2) for branches less than DN 200 (NPS 8), 32 mm (1.25 in.)
 - (c) when the external contour contains more than one radius, the radius on any arc sector of approximately 45 deg shall meet the requirements of (a) and (b) above
 - (d) machining shall not be employed in order to meet the above requirements
- (11) Factors shown apply to bending; flexibility factor for torsion equals 0.9.

Table 402.1-1 Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

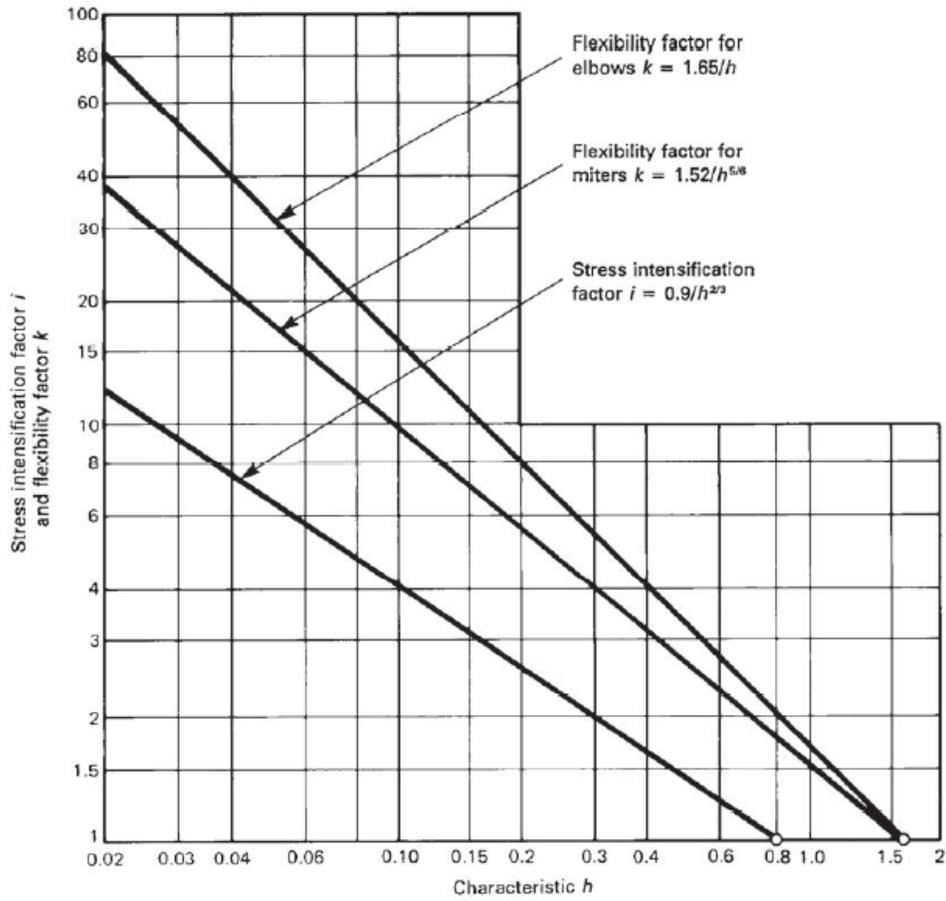


Chart A

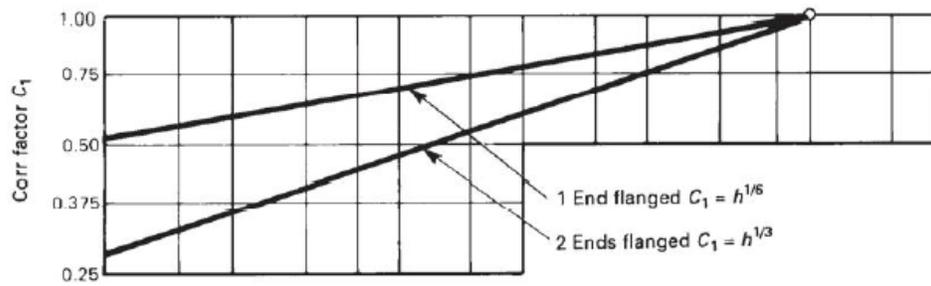


Chart B

Allowable Pressure

For straight pipes and bends (including closely spaced and widely spaced miter bends), the allowable pressure is calculated from para. 504.1.2.

$$P = \frac{2SEt_a}{D - 2Yt_a}$$

where

- P = allowable pressure
- S = basic allowable stress at maximum of CAEPIPE input temperatures T_1 through T_{10}
- E = longitudinal or spiral joint factor (input as material property) from para. 502.3.1 and Table 502.3.1

Table 502.3.1 provides maximum allowable hoop stress values (SE) as a function of metal temperature and includes Longitudinal or Spiral Joint Factor (E) for various materials. Divide SE value by E value provided in Table 502.3.1 to obtain basic allowable stress S . For materials where E is not given explicitly in Table 502.3.1, use $E = 1.0$.

Hence, SE in the above formula for allowable pressure P is the allowable hoop stress per para. 502.3.1 and Table 502.3.1.

- t_a = available thickness for pressure design (as per para. 504.1.1)
- = $t_n \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

(Any additional thickness required for threading, grooving, erosion, corrosion, etc., should be included in corrosion allowance)

- t_n = nominal pipe thickness
- D = outside diameter
- d = inside diameter
- Y = pressure coefficient

For ductile non-ferrous materials and ferritic and austenitic steels,

$$Y = 0.4 \text{ for } D/t_a \geq 6 \text{ and } Y = \frac{d}{d+D}, \text{ for } 4 \leq D/t_a < 6$$

For Cast Iron, $Y = 0.0$

Sustained Stress (in corroded condition)

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from para. 502.3.2(d). Also, refer to Note 1 below.

$$S_L = \frac{PD}{4t_c} + \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z_c} \leq S_h$$

where

- P = maximum of CAEPIPE input pressures P_1 through P_{10}
- D = outside diameter
- t_c = nominal thickness – corrosion allowance, as per para. 502.3.2 (d)
- i_i = in-plane stress intensification factor
- i_o = out-of-plane stress intensification factor
- M_i = in-plane bending moment
- M_o = out-of-plane bending moment
- Z_c = corroded section modulus as per para. 502.3.2 (d)

S_h = basic allowable stress at maximum of CAEPIPE input temperatures T_1 through T_{10}

Occasional Stress (in corroded condition)

The stress (S_{Lo}) due to occasional loads is calculated as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_o) such as earthquake or wind. Wind and earthquake are not considered concurrently (see para. 502.3.3 (a)). Also, refer to Note 1 below.

$$S_{Lo} = \frac{P_{peak} D}{4t_c} + \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z_c} \right]_{sustained} + \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z_c} \right]_{occasional} \leq 1.33S_h$$

where

P_{peak} = peak pressure = (peak pressure factor) x P, where P is defined above

Expansion Stress (in uncorroded condition)

The stress (S_E) due to thermal expansion is calculated from para. 519.4.5 and para. 519.3.5.

$$S_E = \sqrt{S_b^2 + 4S_t^2} \leq S_A$$

where

S_b = resultant bending stress = $\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z}$

S_t = torsional stress = $\frac{M_t}{2Z}$

M_t = torsional moment

Z = uncorroded section modulus; for reduced outlets, effective section modulus

S_A = $f(1.25S_{cold} + 1.25S_{hot})$ (see para. 502.3.2 (c))

f = stress range reduction factor from Figure 502.3.2

S_{cold} = basic allowable stress as minimum metal temperature expected during the displacement cycle under analysis

S_{hot} = basic allowable stress as maximum metal temperature expected during the displacement cycle under analysis

When S_h is greater than S_L , the allowable stress range may be calculated as

$$S_A = S_h + f(S_h - S_L)$$

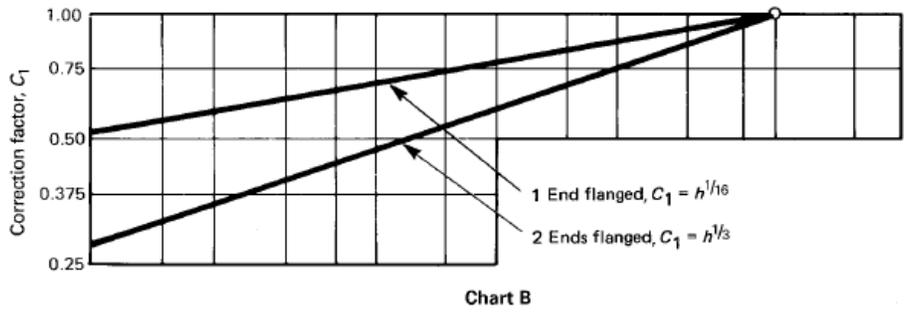
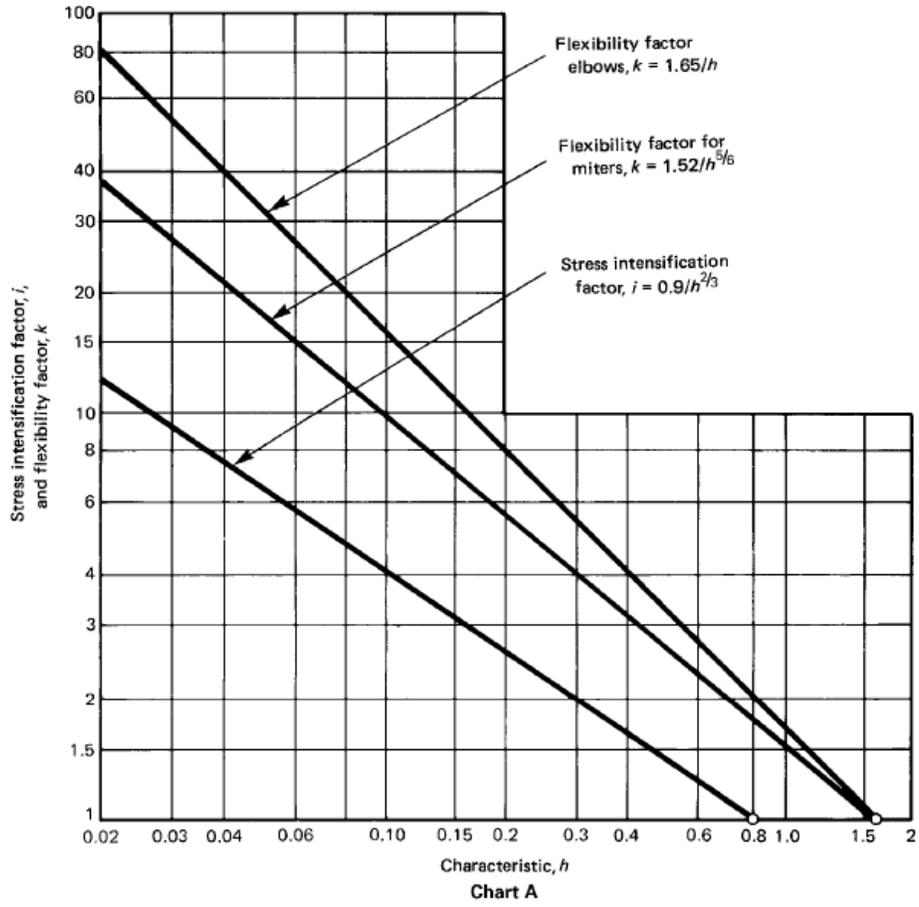
where, S_h = basic allowable stress at maximum of CAEPIPE input temperatures T_1 , T_2 and T_3

This is specified as an analysis option: “Use liberal allowable stresses”, in the CAEPIPE menu Options->Analysis on the Code tab.

Notes:

1. As per para. 502.3.2 (d), the pressure stress should be calculated using the formula $Pd^2/(D^2 - d^2)$, where d is the internal diameter = $D - 2t_c$. This can be selected through Options > Analysis > Pressure
2. Refer Annexure B for the details of “Thickness” and the “Section Modulus” used for weight, pressure and stress calculations.

Table 519.3.6 Illustration



B31.5 (2013)

Table 519.3.6 Flexibility Factor, k , and Stress Intensification Factor, i

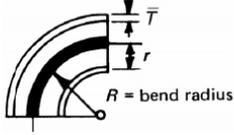
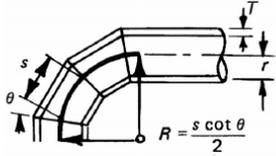
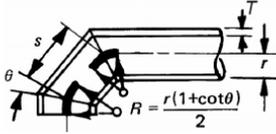
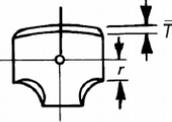
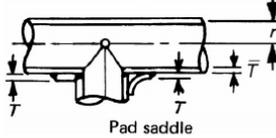
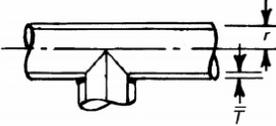
Description	Flexibility Characteristic, h	Flexibility Factor, k	Stress Intensification Factor		Illustration
			i_i [Note (1)]	i_o [Note (2)]	
Welding elbow or pipe bend [Notes (3)–(7)]	$\frac{\bar{T}R}{r^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{3/5}}$	$\frac{0.75}{h^{3/5}}$	
Closely spaced miter bend [Notes (3), (4), (5), and (7)], $s < r(1 + \tan \theta)$	$\frac{\bar{T}s}{r^2} \left(\frac{\cot \theta}{2} \right)$	$\frac{1.52}{h^{3/4}}$	$\frac{0.9}{h^{3/5}}$	$\frac{0.75}{h^{3/5}}$	
Widely spaced miter bend [Notes (3), (4), (7), and (8)], $s \geq r(1 + \tan \theta)$	$\frac{\bar{T}}{r} \left(\frac{1 + \cot \theta}{2} \right)$	$\frac{1.52}{h^{3/4}}$	$\frac{0.9}{h^{3/5}}$	$\frac{0.75}{h^{3/5}}$	
Welding tee ASME B16.9 [Notes (3) and (4)]	$4.4 \frac{\bar{T}}{r}$	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{3/5}}$	
Reinforced fabricated tee with pad or saddle [Notes (3), (4), and (9)]	$\frac{(\bar{T} + \frac{1}{2}T)^{3/2}}{t^{3/2}r}$	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{3/5}}$	
Unreinforced fabricated tee [Notes (3) and (4)]	$\frac{\bar{T}}{r}$	1	$0.75i_o + 0.25$	$\frac{0.9}{h^{3/5}}$	
Butt welded joint, reducer, or welding neck flange	...	1	1.0	1.0	...
Double-welded slip-on flange	...	1	1.2	1.2	...

Table 519.3.6 Flexibility Factor, *k*, and Stress Intensification Factor, *i* (Cont'd)

Description	Flexibility Characteristic, <i>h</i>	Flexibility Factor, <i>k</i>	Stress Intensification Factor		Illustration
			<i>i_i</i> [Note (1)]	<i>i_o</i> [Note (2)]	
Fillet welded joint (single-welded), socket welded flange, or single-welded slip-on flange	...	1	1.3	1.3	...
Lap flange (with ASME B16.9 lap-joint stub)	...	1	1.6	1.6	...
Threaded pipe joint or threaded flange	...	1	2.3	2.3	...
Corrugated straight pipe, or corrugated or creased bend [Note (10)]	...	5	2.5	2.5	...

GENERAL NOTE: For reference, see Table 519.3.6 Illustration on page 41.

NOTES:

- (1) In-plane.
- (2) Out-plane.
- (3) For fittings and miter bends the flexibility factors, *k*, and stress intensification factors, *i*, in the Table apply to bending in any plane and shall not be less than unity; factors for torsion equal unity.
- (4) Both factors apply over the effective arc length (shown by heavy centerlines in the sketches) for curved and miter elbows and to the intersection point for tees. The values of *k* and *i* can be read directly from Chart A by entering with the characteristic, *h*, computed from the equations given where
 - R* = bend radius of welding elbow or pipe bend, in. (mm)
 - r* = mean radius of matching pipe, in. (mm)
 - s* = miter spacing at centerline, in. (mm)
 - T* = pad or saddle thickness, in. (mm)
 - \bar{T} = nominal wall thickness, in. (mm), of: part itself for elbows and curved or miter bends; matching pipe for welding tees; run or header for fabricated tees (provided that if thickness is greater than that of matching pipe, increased thickness must be maintained for at least one run outside diameter to each side of the branch outside diameter).
 - θ = one-half angle between adjacent miter axes, deg
- (5) Where flanges are attached to one or both ends, the values of *k* and *T* in the Table shall be corrected by the factors *C*₁ given below, which can be read directly from Chart B; entering with the computed *h*: one end flanged, $h^{1/4} \geq 1$; both ends flanged, $h^{1/2} \geq 1$.
- (6) The engineer is cautioned that cast butt welding elbows may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (7) In large-diameter thin-wall elbows and bends, pressure can significantly affect the magnitude of flexibility and stress intensification factors. To correct values obtained from the Table for the pressure effect, divide
 - (a) flexibility factor, *k*, by

$$1 + 6 \frac{P}{E_c} \left(\frac{r}{T} \right)^{1/2} \left(\frac{R}{r} \right)^{1/3}$$

- (b) stress intensification factor, *i*, by

$$1 + 3.25 \frac{P}{E_c} \left(\frac{r}{T} \right)^{3/2} \left(\frac{R}{r} \right)^{3/5}$$

where

- E_c* = cold modulus of elasticity, ksi (MPa)
- P* = internal design pressure, psi (kPa)

- (8) Also includes single-miter joint.
- (9) When $T > 1.5\bar{T}$, use $h = 4.05 \bar{T}/r$.
- (10) Factors shown apply to bending; flexibility factor for torsion equals 0.9.

Allowable Pressure

For straight pipes and bends (including closely spaced and widely spaced miter bends), the allowable pressure is calculated from para. 841.1.1.

$$P = \frac{2SEt_nFT}{D}$$

where

P = allowable pressure

P = specified minimum yield strength from para. 817.1.3 (h) and para. 841.1.4 (a)

E = longitudinal joint factor (input as material property), obtained from Table 841.1.7-1 and para. 817.1.3 (d)

t_n = nominal pipe thickness

D = nominal outside diameter

F = construction type design factor, obtained from Table 841.1.6-1

T = temperature derating factor, obtained from Table 841.1.8-1 and para. 841.1.8

Stress due to Sustained and Occasional Loads (Unrestrained Piping)

The sum of longitudinal pressure stress and the bending stress due to external loads, such as weight of the pipe and contents, seismic or wind, etc. is calculated according to paras. 833.6 (a) and 833.6 (b) along with paras. 805.2.3, 833.2 (b), 833.2 (d), 833.2 (e) and 833.2 (f).

Please note, the “include axial force in stress calculations” option is turned ON by default for ANSI B31.8.

Sustained Stress S_L (required to compute Expansion Stress Allowable S_A):

For Pipes and Long Radius Bends

$$S_L = \left[\frac{PD}{4t_n} + \frac{R}{A} \right]_{\text{Sustained}} + \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z} \right]_{\text{Sustained}}$$

For other Fittings or Components.

$$S_{L(fc)} = \left[\frac{PD}{4t_n} + \frac{R}{A} \right]_{\text{Sustained}} + \left[\frac{\sqrt{(0.75i_i M_i)^2 + (0.75i_o M_o)^2 + (M_t)^2}}{Z} \right]_{\text{Sustained}}$$

Sustained + Occasional Stress S_{LO} :

For Pipes and Long Radius Bends

$$S_{LO} = S_L + \left[\frac{(P_{\text{peak}} - P)D}{4t_n} + \frac{R}{A} \right]_{\text{occasional}} + \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z} \right]_{\text{occasional}} \leq 0.75ST$$

For Fittings or Components

$$S_{LO} = S_{L(fc)} + \left[\frac{(P_{\text{peak}} - P)D}{4t_n} + \frac{R}{A} \right]_{\text{occasional}} + \left[\frac{\sqrt{(0.75i_i M_i)^2 + (0.75i_o M_o)^2 + (M_t)^2}}{Z} \right]_{\text{occasional}} \leq 0.75ST$$

where

P = maximum operating pressure = max (P_1 through P_{10})

P_{peak} = Peak pressure factor x P

D = nominal outside diameter

t_n	= nominal thickness
i_i	= in-plane stress intensification factor; the product $0.75i_i$ shall not be less than 1.0
i_o	= out-of-plane stress intensification factor; the product $0.75i_o$ shall not be less than 1.0
M_i	= in-plane bending moment
M_o	= out-of-plane bending moment
M_t	= torsional moment
Z	= uncorroded section modulus; for reduced outlets, effective section modulus
R	= axial force component for external loads (other than thermal expansion and pressure)
A	= corroded cross-section area (i.e., after deducting for corrosion)
S	= specified minimum yield strength from para. 841.1.1(a)
T	= temperature derating factor, obtained from para. 841.1.8 and Table 841.1.8-1

Note:

Young's modulus of elasticity corresponding to the lowest operating temperature [=min (T_1 through T_{10} , T_{ref})] is used to form the stiffness matrix for Sustained and Occasional load calculations.

Expansion Stress (Unrestrained Piping)

The stress (S_E) due to thermal expansion is calculated from para.833.8.

$$S_E = \sqrt{S_b^2 + 4S_t^2} \leq S_A$$

where

$$S_b = \text{resultant bending stress} = \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z}$$

$$S_t = \text{torsional stress} = \frac{M_t}{2Z}$$

M_t = torsional moment

Z = uncorroded section modulus; for reduced outlets, effective section modulus

Please note, "Liberal allowable" option is always turned ON for ANSI B31.8.

$$S_b = f[1.25(S_C + S_h) - S_L]$$

f = stress range reduction factor = $6/\sqrt{N}$, where N = number of equivalent full range cycles

where $f \leq 1.0$ (from para. 833.8 (b)).

f = 0.33SuT at the minimum installed or operating temperature

S_h = 0.33SuT at the maximum installed or operating temperature

where

S_u = specified minimum ultimate tensile strength = 1.5 S_y (assumed), and

S_y = specified minimum yield strength as per para. 841.1.1(a)

T = temperature derating factor, obtained from para. 841.1.8 and Table 841.1.8-1

Note:

Young's modulus of elasticity corresponding to the lowest operating temperature [=min(T_1 through T_{10} , T_{ref})] is used to form the stiffness matrix for Expansion load calculations.

Stress due to Sustained, Thermal and Occasional Loads (Restrained Piping)

The Net longitudinal stress (S_L) due to sustained, thermal expansion and occasional loads for restrained piping is calculated from paras. 833.3 (a), 833.3 (b) along with paras. 805.2.3, 833.2 (a), 833.2 (c), 833.2 (d), 833.2 (e) and 833.2 (f)

$$S_L = \max(|S_p + S_x + S_B|, |S_p + S_x - S_B|)_{sustained} + \max(|S_p + S_x + S_B|, |S_p + S_x - S_B|)_{occasional} + \max(|S_T|_{warmest}, |S_T|_{coldest}) \leq 0.9ST$$

where

$$\text{Internal pressure stress} = S_p = 0.3 \frac{PD}{2t_n}$$

Stress due to axial loading (other than thermal expansion and pressure) $= S_x = \frac{R}{A}$, and may be positive or negative

Nominal bending stress S_B from Weight and / or other External loads for

For Pipes and Long Radius Bends

$$S_B = \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z}$$

For other Fittings or Components.

$$S_B = \frac{\sqrt{(0.75i_i M_i)^2 + (0.75i_o M_o)^2 + (M_t)^2}}{Z}$$

Thermal expansion stress $= S_T = E\alpha(T_i - T_o)$, and may be positive or negative

Where

- P = maximum operating pressure = max(P1 through P10)
- D = nominal outside diameter
- t_n = nominal thickness
- i_i = in-plane stress intensification factor; the product $0.75i_i$ shall not be less than 1.0
- i_o = out-of-plane stress intensification factor; the product $0.75i_o$ shall not be less than 1.0
- M_i = in-plane bending moment
- M_o = out-of-plane bending moment
- M_t = torsional moment
- R = axial force component for external loads (other than thermal expansion and pressure)
- A = corroded cross-sectional area (i.e., after deducting for corrosion)
- Z = uncorroded section modulus; for reduced outlets, effective section modulus
- S = Specified Minimum Yield Strength (SMYS) from para. 841.1.1 (a)
- T = Temperature derating factor from para. 841.1.8 and Table 841.1.8-1
- E = Young's modulus at ambient (reference) temperature
- T_i = installation temperature = T_{ref} in CAEPIPE
- T_o = warmest or coldest operating temperature
- α = coefficient of thermal expansion at T_o defined above

Table E-1 Flexibility Factor, k , and Stress Intensification Factor, i

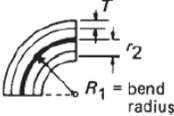
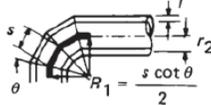
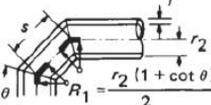
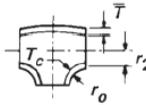
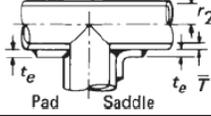
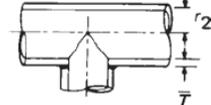
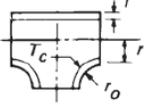
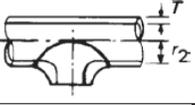
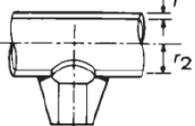
Description	Flexibility Factor, k	Stress Intensification Factor, i [Notes (1) and (2)]		Flexibility Characteristic, h	Sketch
		Out-plane, i_o	In-plane, i_i		
Welding elbow or pipe bend [Notes (1)–(5)]	$\frac{1.65}{h}$	$\frac{0.75}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{\bar{T} R_1}{r_2^2}$	
Closely spaced miter bend [Notes (1), (2), (3), and (5)] $s < r_2 (1 + \tan \theta)$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{\cot \theta \bar{T} s}{2 r_2^2}$	
Single miter bend or widely spaced miter bend $s \geq r_2 (1 + \tan \theta)$ [Notes (1), (2), and (5)]	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{1 + \cot \theta \bar{T}}{2 r_2}$	
Welding tee per ASME B16.9 with $r_o \geq \frac{3}{8} \bar{T}$ $T_c \geq 1.5 \bar{T}$ [Notes (1), (2), and (6)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4} i_o + \frac{1}{4}$	$4.4 \frac{\bar{T}}{r_2}$	
Reinforced fabricated tee with pad or saddle [Notes (1), (2), (7)–(9)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4} i_o + \frac{1}{4}$	$\frac{(\bar{T} + \frac{1}{2} t_d)^{5/2}}{\bar{T}^{3/2} r_2}$	
Unreinforced fabricated tee [Notes (1), (2), and (9)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4} i_o + \frac{1}{4}$	$\frac{\bar{T}}{r_2}$	
Extruded outlet $r_o \geq 0.05d$ $T_c < 1.5 \bar{T}$ [Notes (1), (2), and (6)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4} i_o + \frac{1}{4}$	$\left(1 + \frac{r_o}{r_2}\right) \frac{\bar{T}}{r_2}$	
Welded-in contour insert $r_o \geq \frac{3}{8} \bar{T}$ $T_c \geq 1.5 \bar{T}$ [Notes (1), (2), and (10)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4} i_o + \frac{1}{4}$	$4.4 \frac{\bar{T}}{r_2}$	
Branch welded-on fitting (integrally reinforced) [Notes (1), (2), (9), and (11)]	1	$\frac{0.9}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$3.3 \frac{\bar{T}}{r_2}$	

Table E-1 Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

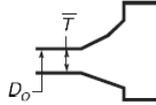
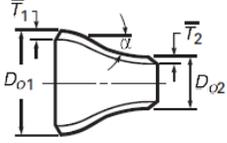
Description	Flexibility Factor, k	Stress Intensification Factor, i	Sketch
Buttweld [Notes (1) and (12)] $\bar{T} \geq 0.237$ in. (6.02 mm), $\delta_{max} \leq \frac{1}{16}$ in. (1.59 mm), and $\delta_{avg}/\bar{T} \leq 0.13$	1	1.0	
Buttweld [Notes (1) and (12)] $\bar{T} \geq 0.237$ in. (6.02 mm), $\delta_{max} \leq \frac{1}{8}$ in. (3.18 mm), and $\delta_{avg}/\bar{T} =$ any value	1	1.9 max. or $(0.9 + 2.7(\delta_{avg}/\bar{T}))$, but not less than 1.0	
Buttweld [Notes (1) and (12)] $\bar{T} \leq 0.237$ in. (6.02 mm), $\delta_{max} \leq \frac{1}{16}$ in. (1.59 mm), and $\delta_{avg}/\bar{T} \leq 0.33$	1	1.9 max. or $1.3 + 0.0036 \frac{D_o}{\bar{T}} + 3.6 \frac{\delta}{\bar{T}}$	
Concentric reducer per ASME B16.9 [Notes (1) and (13)]	1	2.0 max. or $0.5 + 0.01\alpha \left(\frac{D_{o1}}{T_2}\right)^{1/2}$	
Double-welded slip-on flange [Note (14)]	1	1.2	
Socket welding flange or fitting [Notes (14) and (15)]	1	2.1 max or $2.1 \bar{T}/C_s$ but not less than 1.3	
Lap joint flange (with ASME B16.9 lap joint stub) [Note (14)]	1	1.6	
Threaded pipe joint or threaded flange [Note (14)]	1	2.3	
Corrugated straight pipe, or corrugated or creased bend [Note (16)]	5	2.5	

Table E-1 Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

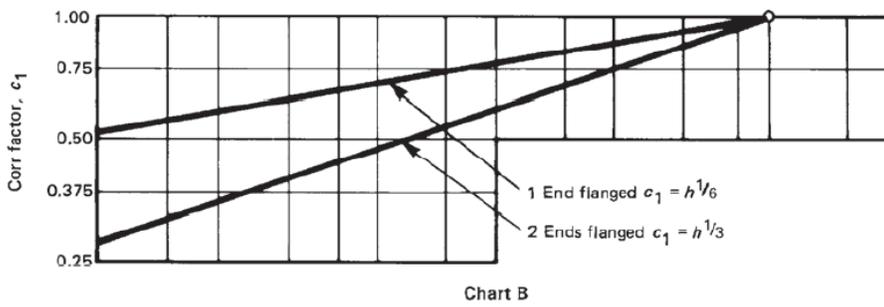
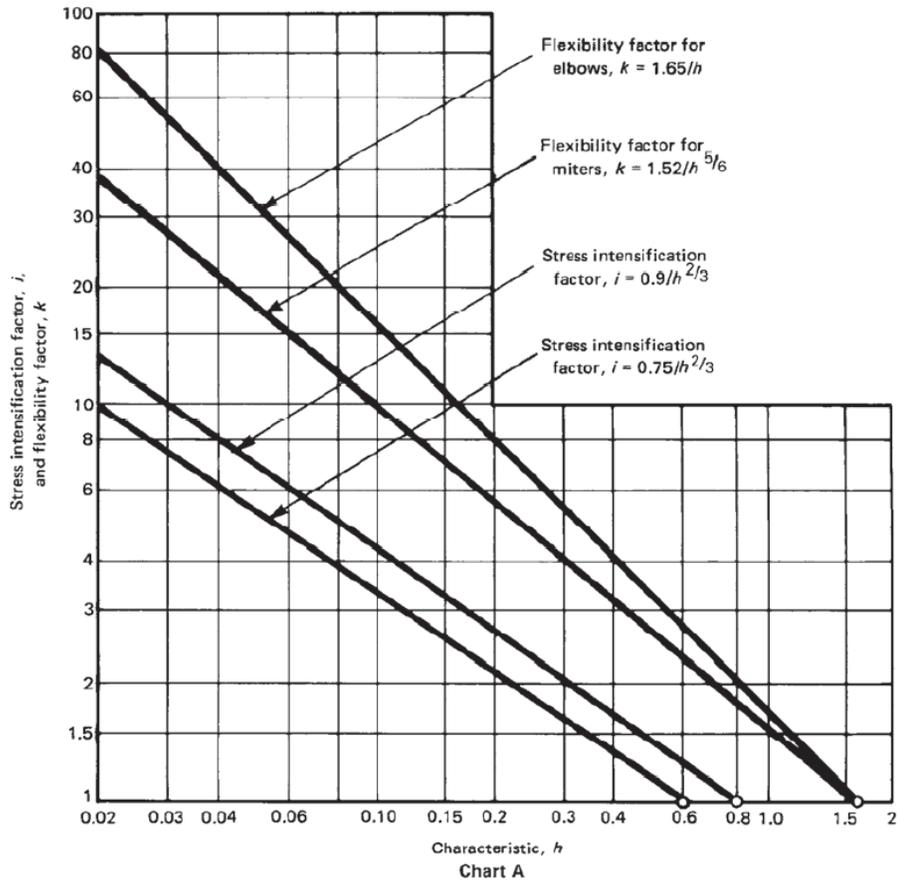


Table E-1 Flexibility Factor, k , and Stress Intensification Factor, i (Cont'd)

NOTES:

- (1) The nomenclature is as follows:
- d = outside diameter of branch, in. (mm)
 - R_1 = bend radius of welding elbow or pipe bend, in. (mm)
 - r_o = radius of curvature of external contoured portion of outlet, measured in the plane containing the axes of the header and branch, in. (mm)
 - r_2 = mean radius of matching pipe, in. (mm)
 - s = miter spacing at centerline, in. (mm)
 - \bar{T} = nominal wall thickness of piping component, in. (mm)
 - = for elbows and miter bends, the nominal wall thickness of the fitting, in. (mm)
 - = for welding tees, the nominal wall thickness of the matching pipe, in. (mm)
 - = for fabricated tees, the nominal wall thickness of the run or header (provided that if thickness is greater than that of matching pipe, increased thickness must be maintained for at least one run outside diameter to each side of the branch outside diameter), in. (mm)
 - T_c = the crotch thickness of tees, in. (mm)
 - t_e = pad or saddle thickness, in. (mm)
 - α = reducer cone angle, deg
 - θ = one-half angle between adjacent miter axes, deg
- (2) The flexibility factor, k , applies to bending in any plane. The flexibility factors, k , and stress intensification factors, i , shall not be less than unity; factors for torsion equal unity. Both factors apply over the effective arc length (shown by heavy centerlines in the sketches) for curved and miter bends and to the intersection point for tees.
The values of k and i can be read directly from Chart A by entering with the characteristic, h , computed from the formulas given.
- (3) Where flanges are attached to one or both ends, the values of k and i shall be corrected by the factors, C_w , which can be read directly from Chart B, entering with the computed h .
- (4) The designer is cautioned that cast butt-welded fittings may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (5) In large diameter thin-wall elbows and bends, pressure can significantly affect the magnitudes of k and i . To correct values from the table, divide k by

$$\left[1 + 6 \left(\frac{P}{E_e} \right) \left(\frac{r_2}{\bar{T}} \right)^{2/3} \left(\frac{R_1}{r_2} \right)^{1/3} \right]$$

divide i by

$$\left[1 + 3.25 \left(\frac{P}{E_e} \right) \left(\frac{r_2}{\bar{T}} \right)^{5/2} \left(\frac{R_1}{r_2} \right)^{2/3} \right]$$

where

E_e = cold modulus of elasticity, psi (MPa)
 P = gage pressure, psi (MPa)

- (6) If the number of displacement cycles is less than 200, the radius and thickness limits specified need not be met. When the radius and thickness limits are not met and the number of design cycles exceeds 200, the out-plane and in-plane stress intensification factors shall be calculated as $1.12/h^{2/3}$ and $(0.67/h^{2/3}) + 1/4$, respectively.
- (7) When $t_e > 1/2 T$, use $h = 4.05 T / r_2$.
- (8) The minimum value of the stress intensification factor shall be 1.2.
- (9) When the branch-to-run diameter ratio exceeds 0.5, but is less than 1.0, and the number of design displacement cycles exceeds 200, the out-plane and in-plane stress intensification factors shall be calculated as $1.8/h^{2/3}$ and $(0.67/h^{2/3}) + 1/4$, respectively, unless the transition weld between the branch and run is blended to a smooth concave contour. If the transition weld is blended to a smooth concave contour, the stress intensification factors in the table still apply.
- (10) If the number of displacement cycles is less than 200, the radius and thickness limits specified need not be met. When the radius and thickness limits are not met and the number of design displacement cycles exceeds 200, the out-plane and in-plane stress intensification factors shall be calculated as $1.8/h^{2/3}$ and $(0.67/h^{2/3}) + 1/4$, respectively.
- (11) The designer must be satisfied that this fabrication has a pressure rating equivalent to straight pipe.
- (12) The stress intensification factors apply to girth butt welds between two items for which the wall thicknesses are between $0.875 \bar{T}$ and $1.10 \bar{T}$ for an axial distance of $\sqrt{D_o \bar{T}}$. D_o and \bar{T} are nominal outside diameter and nominal wall thickness, respectively. $\bar{\delta}_{avg}$ is the average mismatch or offset.
- (13) The equation applies only if the following conditions are met.
- (a) Cone angle α does not exceed 60 deg, and the reducer is concentric.
 - (b) The larger of D_{o1}/\bar{T} and D_{o2}/\bar{T} does not exceed 100.
 - (c) The wall thickness is not less than \bar{T}_1 throughout the body of the reducer, except in and immediately adjacent to the cylindrical portion on the small end, where the thickness shall not be less than \bar{T}_2 .
- (14) For some flanged joints, leakage may occur at expansion stresses otherwise permitted herein. The moment to produce leakage of a flanged joint with a gasket having no self-sealing characteristics can be estimated by the following equation:

$$M_L = (C/4) (S_b A_b - P A_p)$$

A_b = total area of flange bolts, in.² (mm²)
 A_p = area to outside of gasket contact, in.² (mm²)
 C = bolt circle, in. (mm)
 M_L = moment to produce flange leakage, in.-lb (mm·N)
 P = internal pressure, psi (MPa)
 S_b = bolt stress, psi (MPa)

- (15) C_x is the fillet weld length. For unequal lengths, use the smaller leg for C_x .
- (16) Factors shown apply to bending. Flexibility factor for torsion equals 0.9.

Allowable Pressure

For straight pipes and bends, the calculation of allowable pressure is based on Eq. 2 of para. 904.1 and 904.2.1.

$$P = \frac{2SE(t_m - A)}{D}$$

where

P = allowable pressure

SE = allowable hoop stress, given in Appendix I of B31.9 (2008) Code, where

E = longitudinal or spiral weld joint efficiency factor or casting quality factor

t_m = minimum required pipe thickness as per para. 904.1.1(a)

= $t_n \times (1 - \text{mill tolerance}/100)$

t_n = nominal pipe thickness

A = corrosion allowance

(Any additional thickness required for threading, grooving, erosion, corrosion, etc., should be included in “corrosion allowance” in CAEPIPE)

D = outside diameter

For closely and widely spaced miter bends, the allowable pressure shall be the lower positive value calculated from Eqs. (3A) and (3B) of para. 904.2.2(a)

$$P = \frac{SET}{r} \left(\frac{T}{T + 0.64 \tan \theta \sqrt{rT}} \right) \quad \text{Eq. (3A)}$$

$$P = \frac{SET}{r} \left(\frac{R-r}{R-r/2} \right) \quad \text{Eq. (3B)}$$

where

r = mean radius of pipe = $(D - t_n)/2$

T = $t_m - A$, where t_m and A are defined above

R = effective bend radius of the miter

θ = miter half angle

Sustained Stress (in uncorroded condition)

The longitudinal stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated as mentioned in para. 902.3.2(d)

$$S_L = \frac{PD_o}{4t_n} + \frac{0.75iM_A}{Z} \leq S_h$$

where

P = maximum of CAEPIPE pressures P_1 , P_2 and P_3

D_o = outside diameter

t_n = nominal wall thickness

i = stress intensification factor. The product $0.75i$ shall not be less than 1.0.

M_A = resultant bending moment due to weight and other sustained loads

Z = uncorroded section modulus; for reduced outlets, effective section modulus

S_h = hot allowable stress at maximum of CAEPIPE input temperatures T_1 , T_2 and T_3

Occasional Stress (in uncorroded condition)

The longitudinal stress (S_{LO}) due to occasional loads is calculated as mentioned in para. 902.3.3 (a) as the sum of stresses due to pressure, live and dead loads and stress due to occasional loads (S_O) such as earthquake or wind. Wind and earthquake are not considered to occur concurrently.

$$S_{LO} = \frac{P_{peak} D_o}{4t_n} + \frac{0.75iM_A}{Z} + \frac{0.75iM_B}{Z} \leq 1.33S_h$$

where

M_B = resultant bending moment due to occasional loads

P_{peak} = peak pressure = (peak pressure factor) x P

Expansion Stress (in uncorroded condition)

The stress (S_E) due to thermal expansion is calculated from para. 902.3.2(c), para. 919.2.1 and para. 914.4.1(b)

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

M_C = resultant moment due to thermal expansion

$S_A = f(1.25S_C + 0.25S_h)$

f = stress range reduction factor $6/N^{0.2}$, where N being the total number of equivalent reference displacement stress range cycles expected during the service life of the piping. Also $0.15 \leq f \leq 1.0$

S_C = allowable stress at cold temperature, i.e. at minimum of CAEPIPE input temperatures T_1, T_2, T_3 and T_{ref}

When S_h is greater than S_L , the allowable stress range may be calculated as per para. 902.3.2 (d).

$$S_A = f[1.25(S_C + S_h) - S_L]$$

This is specified as an analysis option: “Use liberal allowable stresses”, in the CAEPIPE menu Options>Analysis on the “Code” tab.

Note:

Refer to the topic towards the end of this appendix for the details of “Thickness” and the “Section Modulus” used for weight, pressure and stress calculations.

ASME Section III, Class 2 NC (1980)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 5 of NC-3641.1.

$$P = \frac{2SEt_m}{D - 2Yt_m}$$

where

P = allowable pressure

S = allowable stress

E = joint factor (input as material property)

t_m = minimum required thickness, including mechanical and corrosion allowances

= $t \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

t = nominal pipe thickness

D = outside diameter

d = inside diameter

Y = pressure coefficient

= 0.4 for $t < D/6$

= $d/(D + d)$ for $t < D/6$

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 9 of NC-3652.

$$S_L = \frac{PD}{4t} + \frac{0.75iM_A}{Z} \leq S_h$$

where

P = maximum pressure

D = outside diameter

t = nominal wall thickness

i = stress intensification factor. The product $0.75i$, shall not be less than 1.0

M_A = resultant bending moment due to sustained loads = $\sqrt{M_X^2 + M_Y^2 + M_Z^2}$

Z = section modulus, for reduced outlets, effective section modulus

S_h = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_O) such as earthquake or wind from Equation 10 of NC-3652.2. Wind and earthquake are not considered concurrently.

$$S_{LO} = S_L + \frac{0.75iM_B}{Z} \leq 1.2S_h$$

where

M_B = resultant moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 11 of NC-3652.3.

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

M_C = resultant moment due to thermal expansion

S_A = $f(1.25S_c + 0.25S_h)$

S_c = allowable stress at cold temperature

f = stress range reduction factor from Table NC-3611.2(e)-1

The stress due to pressure, weight, other sustained loads and thermal expansion is calculated from Equation 13 of NC-3652.2.

$$S_{TE} = S_L + S_E \leq S_h + S_A$$

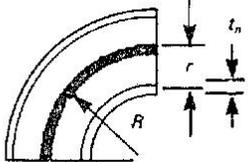
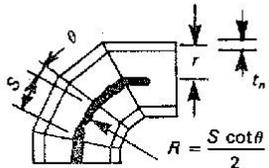
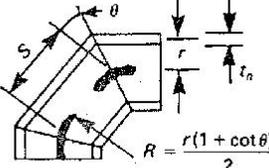
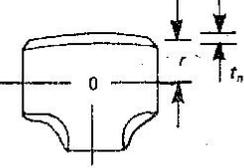
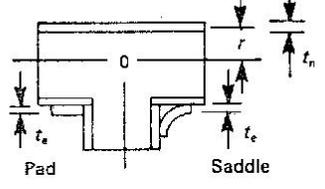
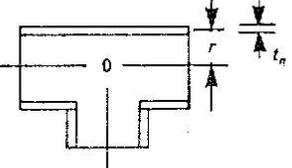
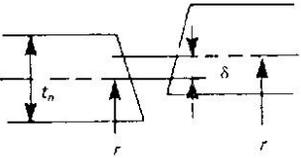
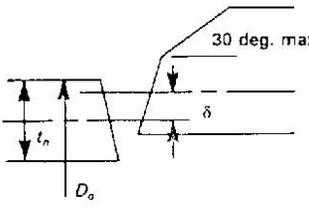
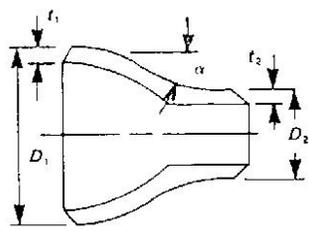
Description	Flexibility Characteristic h	Flexibility Factor k	Stress Intensification Factor i	Sketch
Welding elbow or pipe bend [Notes (1), (2), (3)]	$\frac{t_n R}{r^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{3/2}}$	
Closely spaced miter bend [Notes (1), (2), (3)] $s < r(1 + \tan \theta)$	$\frac{st_n \cot \theta}{2r^2}$	$\frac{1.52}{h^{3/2}}$	$\frac{0.9}{h^{3/2}}$	
Widely spaced miter bend [Notes (1), (2), (4)] $s \geq r(1 + \tan \theta)$	$\frac{t_n(1 + \cot \theta)}{2r}$	$\frac{1.52}{h^{3/2}}$	$\frac{0.9}{h^{3/2}}$	
Welding tee per ANSI B16.9 [Notes (1), (2)]	$\frac{4.4 t_n}{r}$	1	$\frac{0.9}{h^{3/2}}$	
Reinforced fabricated tee [Notes (1), (2), (5)]	$\frac{(t_n + \frac{t_s}{2})^{3/2}}{r(t_n)^{3/2}}$	1	$\frac{0.9}{h^{3/2}}$	
Unreinforced fabricated tee [Notes (1), (2)]	$\frac{t_n}{r}$	1	$\frac{0.9}{h^{3/2}}$	

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$)

ASME Section III, Class 2 NC (1980)

Fig. NC-3673.2(b)-1

SECTION III, DIVISION 1 — SUBSECTION NC

Description	Flexibility Factor k	Stress Intensification Factor i	Sketch
Branch connection [Note (6)]	1	For checking branch end $Z = \pi (r'_m)^2 T_b$ $i = 1.5 \left(\frac{R_m}{T} \right)^{2/3} \left(\frac{r'_m}{R_m} \right)^{1/2} \left(\frac{T_b}{T_r} \right) \left(\frac{r_m}{r_p} \right)$ For checking run ends $Z = \pi (R_m) r^2 T_r$ $i = 0.4 \left(\frac{R_m}{T_r} \right)^{2/3} \left(\frac{r'_m}{R_m} \right)$ but not less than 1.5	Fig. NC-3673.2(b)-2
Butt weld [Note (1)] $t_n > 3/16$ and $\frac{\delta}{t_n} \leq 0.1$	1	1.0	
Butt weld [Note (1)] $t_n \leq 3/16$ or $\frac{\delta}{t_n} > 0.1$	1	1.0 for flush weld 1.8 for as-welded	
Fillet welded joint, socket welded flange, or single welded slip-on flange	1	2.1	Fig. NC-3673.2(b)-3 sketches (a), (b), (c), (e), and (f)
Braze joint	1	2.1	Fig. NC-4511-1
Full fillet weld	1	1.3	Fig. NC-3673.2(b)-3 sketch (d)
30 deg. tapered transition (ANSI B16.25) [Note (1)]	1	1.9 max. or $1.3 + 0.0036 \frac{D_o}{t_n} + 3.6 \frac{\delta}{t_n}$	
Concentric reducer [Note (7)] (ANSI B16.9 or MSS SP48)	1	2.0 max. or $0.5 + 0.01 \alpha \left(\frac{D_2}{t_2} \right)^{1/2}$	
Threaded pipe joint or threaded flange	1	2.3	
Corrugated straight pipe or corrugated or creased bend [Note (8)]	5	2.5	

(See notes on next page)

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$) (CONT'D)

NOTES TO FIG. NC-3673.2(b)-1:

- (1) The following nomenclature applies.
- r = mean radius of pipe, in. (matching pipe for tees and elbows)
 - t_n = nominal wall thickness of pipe, in. [matching pipe for tees and elbows, see Note (9)]
 - R = bend radius of elbow or pipe bend, in.
 - θ = one-half angle between adjacent miter axes
 - s = miter spacing at center line, in.
 - t_s = reinforced thickness, in.
 - δ = mismatch, in.
 - D_o = outside diameter, in.
- (2) The flexibility factors k and stress intensification factors i apply to bending in any plane for fittings and shall in no case be taken less than unity. Both factors apply over the effective arc length (shown by heavy center lines in the sketches) for curved and miter elbows, and to the intersection point for tees. The values of k and i can be read directly by entering with the characteristic h computed from the equations given.
- (3) Where flanges are attached to one or both ends, the values of k and i shall be corrected by the factor c given below, which can be read directly from Fig. NC-3673.2(b)-5, entering with the computed h .
- (a) One end flanged, $c = h^{3/4}$
 - (b) Both ends flanged, $c = h^{3/2}$
- (4) Also includes single miter joints.
- (5) When $t_s > 1.5t_n$, $h = 4.05t_n/r$
- (6) The equation applies only if the following conditions are met:
- (a) The reinforcement area requirements of NC-3643 are met.
 - (b) The axis of the branch pipe is normal to the surface of run pipe wall.
 - (c) For branch connections in a pipe, the arc distance measured between the centers of adjacent branches along the surface of the run pipe is not less than three times the sum of their inside radii in the longitudinal direction or is not less than two times the sum of their radii along the circumference of the run pipe.
 - (d) The inside corner radius r_1 [Fig. NC-3673.2(b)-2] is between 10% and 50% of T_r .
 - (e) The outer radius r_2 is not less than the larger of $T_o/2$, $(T_o + Y)/2$ [Fig. NC-3673-2(b)-2 sketch (c)] or $T_r/2$.
 - (f) The outer radius r_3 is not less than the larger of
 - (1) $0.002\theta d_o$
 - (2) $2(\sin \theta)^2$ times the offset for the configurations shown in Figs. NC-3673.2(b)-2 sketches (a) and (b).
 - (g) $R_m/T_r \leq 50$ and $r'_m/R_m \leq 0.5$.
- (7) The equation applies only if the following conditions are met:
- (a) Cone angle α does not exceed 60 deg., and the reducer is concentric.
 - (b) The larger of D_1/t_1 and D_2/t_2 does not exceed 100.
 - (c) The wall thickness is not less than t_1 throughout the body of the reducer, except in and immediately adjacent to the cylindrical portion on the small end, where the thickness shall not be less than t_2 .
- (8) Factors shown apply to bending; flexibility factor for torsion equals 0.9.
- (9) The designer is cautioned that cast butt welding elbows may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.

ASME Section III, Class 2 NC (1986)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 5 of NC-3641.1.

$$P = \frac{2SEt_m}{D - 2Yt_m}$$

where

P = allowable pressure

S = allowable stress

E = joint factor (input as material property)

t_m = minimum required thickness, including mechanical and corrosion allowance
= $t \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

t = nominal pipe thickness

D = outside diameter

d = inside diameter

Y = pressure coefficient

= 0.4 for $t < D/6$

= $d/(D + d)$ for $t \geq D/6$

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 8 of NC-3652.

$$S_L = B1 \frac{PD}{2t} + B2 \frac{M_A}{Z} \leq 1.5S_h$$

where

$B1, B2$ = primary stress indices from NB-3680

P = maximum pressure

D = outside diameter

t = nominal wall thickness

M_A = resultant bending moment due to sustained loads = $\sqrt{M_X^2 + M_Y^2 + M_Z^2}$

Z = section modulus, for reduced outlets, effective section modulus

S_h = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_O) such as earthquake or wind from Equation 9 of NC-3653.1. Wind and earthquake are not considered concurrently.

$$S_{LO} = S_L + B2 \frac{M_B}{Z} \leq 1.8S_h$$

where

M_B = resultant moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 10 of NC-3653.2.

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

i = stress intensification factor

M_C = resultant moment due to thermal expansion

$S_A = f(1.25S_c + 0.25S_h)$

S_c = allowable stress at cold temperature

f = stress range reduction factor from Table NC-3611.2(e)-1

The stress due to pressure, weight, other sustained loads and thermal expansion is calculated from Equation 11 of NC-3653.2(c).

$$S_{TE} = \frac{PD}{4t} + \frac{0.75iM_A}{Z} + \frac{iM_C}{Z} \leq S_h + S_A$$

$0.75i$ shall not be less than 1.0.

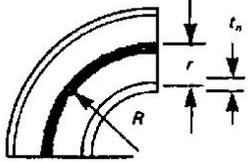
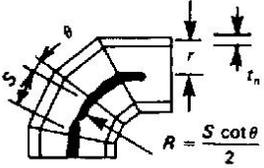
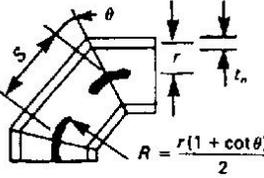
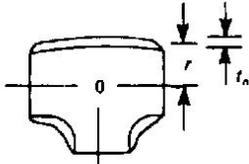
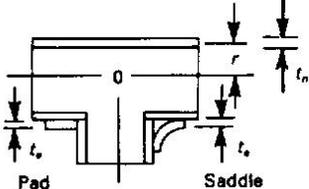
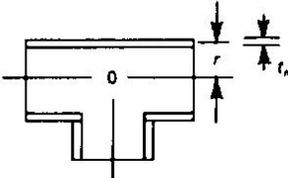
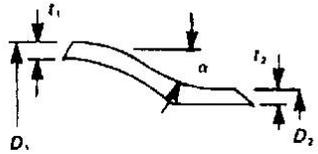
Description	Flexibility Characteristic h	Flexibility Factor k	Stress Intensification Factor i	Sketch
Welding elbow or pipe bend [Notes (1), (2), (3)]	$\frac{t_n R}{r^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{1/2}}$	
Closely spaced miter bend [Notes (1), (2), (3)] $s < r(1 + \tan \theta)$	$\frac{st_n \cot \theta}{2r^2}$	$\frac{1.52}{h^{1/2}}$	$\frac{0.9}{h^{1/2}}$	
Widely spaced miter bend [Notes (1), (2), (4)] $s \geq r(1 + \tan \theta)$	$\frac{t_n(1 + \cot \theta)}{2r}$	$\frac{1.52}{h^{1/2}}$	$\frac{0.9}{h^{1/2}}$	
Welding tee per ANSI B16.9 [Notes (1), (2)]	$\frac{4.4 t_n}{r}$	1	$\frac{0.9}{h^{1/2}}$	
Reinforced fabricated tee [Notes (1), (5), (10)]	$\frac{(t_n + \frac{t_s}{2})^{1/2}}{r(t_n)^{1/2}}$	1	$\frac{0.9}{h^{1/2}}$	
Unreinforced fabricated tee [Notes (1), (10)]	$\frac{t_n}{r}$	1	$\frac{0.9}{h^{1/2}}$	

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$)

Fig. NC-3673.2(b)-1

SECTION III, DIVISION 1 — SUBSECTION NC

1986 Edition

Description	Flexibility Factor k	Stress Intensification Factor i	Sketch
Branch connection [Note (6)]	1	For checking branch end $Z = \pi (r'_m)^2 T'_b$ $i = 1.5 \left(\frac{R_m}{T_r}\right)^{2/3} \left(\frac{r'_m}{R_m}\right)^{1/3} \left(\frac{T'_b}{T_r}\right) \left(\frac{r'_m}{r_p}\right)$ For checking run ends $Z = \pi (R_m)^2 T_r$ $i = 0.4 \left(\frac{R_m}{T_r}\right)^{2/3} \left(\frac{r'_m}{R_m}\right)$ but not less than 1.5	Fig. NC-3673.2(b)-2
Girth butt weld [Note (1)] $t_n \geq 0.237$ in.	1	1.0	
Girth butt weld [Note (1)] $t_n < 0.237$ in.	1	1.9 max. or $0.9(1 + 38/t_n)$ but not less than 1.0	
Circumferential fillet welded or socket welded joints [Note (11)]	1	$2.1/(C_x/t_n)$ but not less than 1.3	Fig. NC-4427-1 sketches (c-1), (c-2), and (c-3)
Braze joint	1	2.1	Fig. NC-4511-1
30 deg. tapered transition (ANSI B16.25) [Note (1)]	1	1.9 max. or $1.3 + 0.0036 \frac{D_o}{t_n} + 3.6 \frac{\delta}{t_n}$	
Concentric and eccentric reducers [Note (7)] (ANSI B16.9)	1	2.0 max. or $0.5 + 0.01 \alpha \left(\frac{D_2}{t_2}\right)^{1/2}$	
Threaded pipe joint or threaded flange	1	2.3	
Corrugated straight pipe or corrugated or creased bend [Note (8)]	5	2.5	

(See notes on next page)

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$) (CONT'D)

FIG. NC-3673.2(b)-1 (CONT'D):

NOTES:

- (1) The following nomenclature applies:
- r = mean radius of pipe, in. [matching pipe for tees and elbows]
 - t_n = nominal wall thickness of pipe, in. [matching pipe for tees and elbows, see Note (9)]
 - R = bend radius of elbow or pipe bend, in.
 - θ = one-half angle between adjacent miter axes, deg.
 - s = miter spacing at center line, in.
 - t_r = reinforced thickness, in.
 - δ = average permissible mismatch at girth butt welds as shown in Fig. NC-4233-1. A value of δ less than $1/32$ in. may be used provided the smaller mismatch is specified for fabrication. For "flush" welds, as defined in Fig. NB-3683.1(c)-1, δ may be taken as zero, $i = 1.0$, and flush welds need not be ground.
 - D_o = outside diameter, in.
- (2) The flexibility factors k and stress intensification factors i apply to bending in any plane for fittings and shall in no case be taken less than unity. Both factors apply over the effective arc length (shown by heavy center lines in the sketches) for curved and miter elbows, and to the intersection point for tees. The values of k and i can be read directly by entering with the characteristic h computed from the equations given.
- (3) Where flanges are attached to one or both ends, the values of k and i shall be corrected by the factor c given below, which can be read directly from Fig. NC-3673.2(b)-5, entering with the computed h .
- (a) One end flanged, $c = h^{1/8}$
 - (b) Both ends flanged, $c = h^{3/8}$
- (4) Also includes single miter joints.
- (5) When $t_r > 1.5t_n$, $h = 4.05t_n/r$
- (6) The equation applies only if the following conditions are met.
- (a) The reinforcement area requirements of NC-3643 are met.
 - (b) The axis of the branch pipe is normal to the surface of run pipe wall.
 - (c) For branch connections in a pipe, the arc distance measured between the centers of adjacent branches along the surface of the run pipe is not less than three times the sum of their inside radii in the longitudinal direction or not less than two times the sum of their inside radii along the circumference of the run pipe.
 - (d) The inside corner radius r_i [Fig. NC-3673.2(b)-2] for nominal branch pipe size greater than 4 in. shall be between 10% and 50% T_r . The radius r_i is not required for nominal branch pipe size smaller than 4 in.
 - (e) The outer radius r_2 is not less than the larger of $T_b/2$, $(T_b + Y)/2$ [Fig. NC-3673-2(b)-2 sketch (c)] or $T_r/2$.
 - (f) The outer radius r_3 is not less than the larger of
 - (1) $0.002\theta d_o$
 - (2) $2(\sin \theta)^3$ times the offset for the configurations shown in Fig. NC-3673.2(b)-2 sketches (a) and (b).
 - (g) $R_m/T_r \leq 50$ and $r'_m/R_m \leq 0.5$.
 - (h) The outer radius r_2 is not required provided an additional multiplier of 2.0 is included in the equations for branch end and run end stress intensification factors. In this case, the calculated value of i for the branch or run shall not be less than 2.1.
- (7) The equation applies only if the following conditions are met.
- (a) Cone angle α does not exceed 60 deg.
 - (b) The larger of D_1/t_1 and D_2/t_2 does not exceed 100.
 - (c) The wall thickness is not less than t_1 throughout the body of the reducer, except in and immediately adjacent to the cylindrical portion on the small end, where the thickness shall not be less than t_2 .
 - (d) For eccentric reducers, α is the maximum cone angle.
- (8) Factors shown apply to bending; flexibility factor for torsion equals 0.9.
- (9) The designer is cautioned that cast butt welding elbows may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (10) The stress intensification factor i shall in no case be taken as less than 2.1.
- (11) C_s is the fillet weld length. For unequal leg lengths, use the smaller leg length for C_s .

ASME Section III, Class 2 NC (1992)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 5 of NC-3641.1.

$$P = \frac{2SEt_m}{D - 2Yt_m}$$

where

P = allowable pressure

S = allowable stress

E = joint factor (input as material property)

t_m = minimum required thickness, including mechanical and corrosion allowance

= $t \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

t = nominal pipe thickness

D = outside diameter

d = inside diameter

Y = pressure coefficient

= 0.4 for $t < D/6$

= $d/(D + d)$ for $t \geq D/6$

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 8 of NC-3652.

$$S_L = B1 \frac{PD}{2t} + B2 \frac{M_A}{Z} \leq 1.5S_h$$

where

$B1, B2$ = primary stress indices from NB-3680

P = maximum pressure

D = outside diameter

t = nominal wall thickness

M_A = resultant bending moment due to sustained loads = $\sqrt{M_X^2 + M_Y^2 + M_Z^2}$

Z = section modulus, for reduced outlets, effective section modulus

S_h = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_O) such as earthquake or wind from Equation 9 of NC-3653.1. Wind and earthquake are not considered concurrently.

$$S_{LO} = S_L + B2 \frac{M_B}{Z} \leq 1.8S_h$$

where

M_B = resultant moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 10 of NC-3653.2.

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

i = stress intensification factor

M_C = resultant moment due to thermal expansion

$S_A = f(1.25S_c + 0.25S_h)$

S_c = allowable stress at cold temperature

f = stress range reduction factor from Table NC-3611.2(e)-1

The stress due to pressure, weight, other sustained loads and thermal expansion is calculated from Equation 11 of NC-3653.2(c).

$$S_{TE} = \frac{PD}{4t} + \frac{0.75iM_A}{Z} + \frac{iM_C}{Z} \leq S_h + S_A$$

$0.75i$ shall not be less than 1.0.

Fig. NC-3673.2(b)-1

1992 SECTION III, DIVISION I — NC

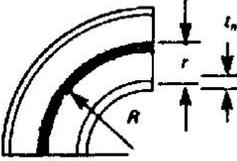
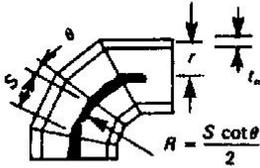
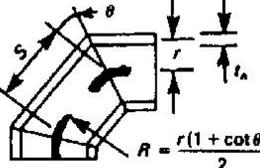
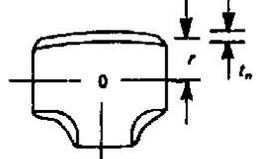
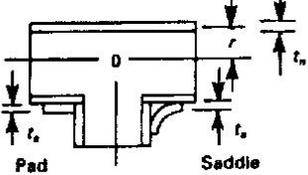
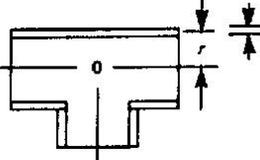
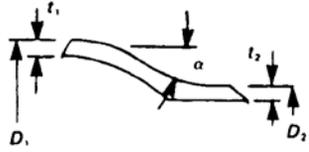
Description	Flexibility Characteristic h	Flexibility Factor k	Stress Intensification Factor i	Sketch
Welding elbow or pipe bend [Notes (1), (2), (3)]	$\frac{t_n R}{r^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{1/2}}$	
Closely spaced miter bend [Notes (1), (2), (3)] $s < r(1 + \tan \theta)$	$\frac{st_n \cot \theta}{2r^2}$	$\frac{1.52}{h^{1/2}}$	$\frac{0.9}{h^{1/2}}$	
Widely spaced miter bend [Notes (1), (2), (4)] $s \geq r(1 + \tan \theta)$	$\frac{t_n(1 + \cot \theta)}{2r}$	$\frac{1.52}{h^{1/2}}$	$\frac{0.9}{h^{1/2}}$	
Welding tee per ANSI B16.9 [Notes (1), (2)]	$\frac{4.4 t_n}{r}$	1	$\frac{0.9}{h^{1/2}}$	
Reinforced fabricated tee [Notes (1), (5), (10)]	$\frac{(t_n + \frac{t_s}{2})^{3/2}}{r(t_n)^{1/2}}$	1	$\frac{0.9}{h^{1/2}}$	
Unreinforced fabricated tee [Notes (1), (10)]	$\frac{t_n}{r}$	1	$\frac{0.9}{h^{1/2}}$	

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$)

Description	Flexibility Factor k	Stress Intensification Factor i	Sketch
Branch connection [Note (6)]	1	For checking branch end $Z = \pi (r'_m)^2 T'_b$ $i = 1.5 \left(\frac{R_m}{T_r}\right)^{2/3} \left(\frac{r'_m}{R_m}\right)^{1/2} \left(\frac{T'_b}{T_r}\right) \left(\frac{r'_m}{r_a}\right)$	Fig. NC-3673.2(b)-2
		For checking run ends $Z = \pi (R_m)^2 T_r$ $i = 0.4 \left(\frac{R_m}{T_r}\right)^{2/3} \left(\frac{r'_m}{R_m}\right)$ but not less than 1.5	
Girth butt weld [Note (1)] $t_n \geq 0.237$ in.	1	1.0	
Girth butt weld [Note (1)] $t_n < 0.237$ in.	1	1.9 max. or $0.9(1 + 3\delta/t_n)$ but not less than 1.0	
Circumferential fillet welded or socket welded joints [Note (11)]	1	$2.1/(C_x/t_n)$ but not less than 1.3	Fig. NC-4427-1 sketches (c-1), (c-2), and (c-3)
Brazed joint	1	2.1	Fig. NC-4511-1
30 deg. tapered transition (ANSI B16.25) [Note (1)]	1	1.9 max. or $1.3 + 0.0036 \frac{D_o}{t_n} + 3.6 \frac{\delta}{t_n}$	
Concentric and eccentric reducers [Note (7)] (ANSI B16.9)	1	2.0 max. or $0.5 + 0.01 \alpha \left(\frac{D_2}{t_2}\right)^{1/2}$	
Threaded pipe joint or threaded flange	1	2.3	
Corrugated straight pipe or corrugated or creased bend [Note (8)]	5	2.5	

(See notes on next page)

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$) (CONT'D)

Fig. NC-3673.2(h)-1

1992 SECTION III, DIVISION 1 — NC

NOTES:

- (1) The following nomenclature applies:
- r = mean radius of pipe, in. [matching pipe for tees and elbows]
 - t_n = nominal wall thickness of pipe, in. [matching pipe for tees and elbows, see Note (9)]
 - R = bend radius of elbow or pipe bend, in.
 - θ = one-half angle between adjacent miter axes, deg.
 - s = miter spacing at center line, in.
 - t_r = reinforced thickness, in.
 - D_o = outside diameter, in.
- (2) The flexibility factors k and stress intensification factors i apply to bending in any plane for fittings and shall in no case be taken less than unity. Both factors apply over the effective arc length (shown by heavy center lines in the sketches) for curved and miter elbows, and to the intersection point for tees.
- (3) Where flanges are attached to one or both ends, the values of k and i shall be corrected by the factor c given below.
- (a) One end flanged, $c = h^{3/4}$
 - (b) Both ends flanged, $c = h^{3/2}$
- (4) Also includes single miter joints.
- (5) When $t_r > 1.6t_n$, $h = 4.05t_n/r$
- (6) The equation applies only if the following conditions are met.
- (a) The reinforcement area requirements of NC-3643 are met.
 - (b) The axis of the branch pipe is normal to the surface of run pipe wall.
 - (c) For branch connections in a pipe, the arc distance measured between the centers of adjacent branches along the surface of the run pipe is not less than three times the sum of their inside radii in the longitudinal direction or not less than two times the sum of their inside radii along the circumference of the run pipe.
 - (d) The inside corner radius r_1 [Fig. NC-3673.2(b)-2] for nominal branch pipe size greater than 4 in. shall be between 10% and 50% T_r . The radius r_1 is not required for nominal branch pipe size smaller than 4 in.
 - (e) The outer radius r_2 is not less than the larger of $T_n/2$, $(T_n + Y)/2$ [Fig. NC-3673.2(b)-2 sketch (c)] or $T_r/2$.
 - (f) The outer radius r_3 is not less than the larger of
 - (1) $0.002\theta d_o$
 - (2) $2(\sin \theta)^2$ times the offset for the configurations shown in Fig. NC-3673.2(b)-2 sketches (a) and (b).
 - (g) $R_n/T_r \leq 50$ and $r'_n/R_n \leq 0.6$.
 - (h) The outer radius r_3 is not required provided an additional multiplier of 2.0 is included in the equations for branch end and run end stress intensification factors. In this case, the calculated value of i for the branch or run shall not be less than 2.1.
- (7) The equation applies only if the following conditions are met.
- (a) Cone angle α does not exceed 60 deg.
 - (b) The larger of D_1/t_1 and D_2/t_2 does not exceed 100.
 - (c) The wall thickness is not less than t_1 throughout the body of the reducer, except in and immediately adjacent to the cylindrical portion on the small end, where the thickness shall not be less than t_2 .
 - (d) For eccentric reducers, α is the maximum cone angle.
- (8) Factors shown apply to bending; flexibility factor for torsion equals 0.8.
- (9) The designer is cautioned that cast butt welding elbows may have considerably heavier walls than that of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.
- (10) The stress intensification factor i shall in no case be taken as less than 2.1.
- (11) In Fig. NC-4427-1(c-1) and (c-2), C_r shall be taken as X_{min} and $C_r \geq 1.25 t_n$. In Fig. NC-4427-1 (c-3), $C_r \geq 0.75 t_n$. For unequal leg lengths use the smaller leg length for C_r .

FIG. NC-3673.2(b)-1 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$) (CONT'D)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from 4.2(4).

$$P = \frac{2fet_f}{D - 2t_f}$$

where

P = allowable pressure

f = allowable stress

e = joint factor (input as material property)

t_f = minimum required thickness, including mechanical and corrosion allowances

= nominal pipe thickness \times (1 – mill tolerance/100) – corrosion allowance

D = outside diameter

$$P = \frac{fe}{X(1 + 0.6427 \tan \alpha \sqrt{X})}$$

where

α = miter angle

$X = (D - 2t_f)/2t_f$

Stresses

Stresses are calculated according to 4.11.

Sustained stresses are due to pressure, weight and other sustained mechanical loads. Maximum value is 0.8 \times proof stress value or rupture stress value in hot condition.

Expansion stresses are due to thermal (including specified displacements) and pressure loadings. Maximum value is lower of

- 1) 0.9 \times proof stress at room temperature plus 0.9 \times proof stress at design temperature.
- 2) 0.9 \times proof stress at room temperature plus rupture stress at design temperature.

Hot stresses are due to pressure, deadweight, other sustained mechanical loads, thermal and cold pull loadings. Maximum value is the rupture stress at design temperature.

Occasional stresses (which are due to seismic, dynamic or wind loadings) are not calculated. If these are to be evaluated, ASME B31.1 code should be used according to Inquiry Case 806/2 : November 1986 to BS 806 : 1986.

Stresses on Straight Pipes and Bends (4.11.4)

The combined stress f_c on straight pipes and bends including miter bends is calculated from (28).

$$F_c = F^2 + 4f_s^2$$

where

F = greater of f_T or f_L

f_T = transverse stress (4.11.4.2)

= transverse pressure stress + transverse bending stress

f_L = longitudinal stress (4.11.4.3)

= longitudinal pressure stress + longitudinal bending stress

f_s = torsional stress (4.11.4.4)

Transverse Stress f_T (4.11.4.2)

The transverse pressure stress on both straight pipes and bends excluding miter bends is calculated from (29).

$$\frac{pd}{2t} + 0.5p$$

The transverse pressure stress on miter bends is calculated from (30)

$$\left(\frac{pd}{2t} + 0.5p\right) \left(1 + 0.6427 \tan \alpha \sqrt{\frac{d+t}{2t}}\right)$$

The transverse bending stress on straight pipe is zero.

The transverse bending stress on bends including miter bends is calculated from (31)

$$\frac{r}{I} (M_i F_{Ti})^2 + (M_o F_{To})^2$$

where

d = inside diameter

t = thickness

p = pressure

M_i = in-plane bending moment

M_o = out-of-plane bending moment

F_{Ti} = in-plane transverse stress intensification factor

F_{To} = out-of-plane transverse stress intensification factor

r = mean radius of pipe

I = moment of inertia

D = outside diameter

α = miter angle

Longitudinal Stress f_L (4.11.4.3)

The longitudinal pressure stress on both straight pipes and bends including miter bends is calculated from (32)

$$\frac{pd^2}{4t(d+t)}$$

The longitudinal bending stress on straight pipe is calculated from (33).

$$\frac{d+2t}{2I} \sqrt{M_i^2 + M_o^2}$$

The longitudinal bending stress on bends including miter bends is calculated from (34).

$$\frac{r}{I} \sqrt{(M_i F_{Li})^2 + (M_o F_{Lo})^2}$$

where

F_{Li} = in-plane longitudinal stress intensification factor

F_{Lo} = out-of-plane longitudinal stress intensification factor

Torsional Stress f_S (4.11.4.4)

The torsional stress for both straight pipes and bends including miter bends is calculated from (35).

$$f_S = M_t \frac{d+2t}{4I}$$

where

M_t = torsional moment

Stress at Branch Junction (4.11.5)

The combined stress at a branch junction is calculated from (36)

$$f_{CB} = f_B^2 + 4f_{SB}^2$$

where

f_B = transverse pressure stress at the junction plus nondirectional bending stress

f_{SB} = torsional stress at the junction

Transverse Pressure Stress at Branch Junction (4.11.5.2)

The transverse pressure stress at branch junction is calculated from (37)

$$pm \left(\frac{d_1 + t_a}{2t_a} \right)$$

where

d_1 = mean radius of main pipe

t_a = minimum thickness of main pipe

p = design pressure

m = stress multiplier, or equal to:

(a) for branch junctions where both r_2/r_1 and t_2/t_1 are less than or equal to 0.3

$$m = 1.8 + \frac{2.8r_2}{r_1} \sqrt{\frac{r_1}{t_1}}$$

(b) for branch junctions where either r_2/r_1 and t_2/t_1 are greater than 0.3

$$m = 2.5Z_1^{0.2042} \text{ for } d_2/d_1 \leq 0.7$$

$$m = 2.5Z_1^{0.2415} \text{ for } d_2/d_1 > 0.7$$

$$Z_1 = (r_2/t_2)^2 t_1/r_1$$

r_1 = mean radius of main pipe

r_2 = mean radius of branch pipe

t_1 = thickness of main pipe

t_2 = thickness of branch pipe

Nondirectional Bending Stress at Branch Junction (4.11.5.3)

For the main pipe, the bending stress is calculated from (40)

$$\frac{r_1}{I} \sqrt{(M_i B_i)^2 + (M_o B_o)^2}$$

where

r_1 = mean radius of pipe

I = moment of inertia of main pipe

M_i = in-plane bending moment

M_o = out-of-plane bending moment

B_i = in-plane branch stress intensification factor (Fig.4.11.1(6))

B_o = out-of-plane branch stress intensification factor (Fig.4.11.1(6))

For the branch pipe, the bending stress is calculated from

$$\frac{r_2}{I} \sqrt{(M_i B_i)^2 + (M_o B_o)^2}$$

where

r_2 = mean radius of branch pipe

I = calculated from $\pi r_2^3 B_o t_2$ or $\pi r_2^3 t_1$ whichever gives the lower value

Allowable Pressure

The allowable pressure for straight pipes is calculated from equation 6.1-1 or 6.1-3 depending on the ratio between inner and outer diameter.

For $D_o/D_i \leq 1.7$

$$P = \frac{2fze}{D_o - e}$$

For $D_o/D_i > 1.7$

$$P = fz \frac{(1 - a^2)}{(1 + a^2)}$$

where

P = allowable pressure

f = allowable stress

z = joint factor (input as material property in CAEPIPE)

e = nominal pipe thickness \times (1 – mill tolerance %/100 – corrosion allowance)“c”

(Any additional thickness required for threading, grooving, erosion, corrosion, etc. should be included in corrosion allowance in CAEPIPE)

D_o = outer diameter

D_i = inner diameter

$$a = 1 - \frac{2e}{D_o}$$

For pipe bends the maximum allowable pressure is calculated using the equivalent pipe wall thickness e_{equi} –

$$e_{equi} = \frac{e}{t_f}$$

where

$$t_f = \left(\frac{R/D - 0.25}{R/D - 0.50} \right)$$

R = radius on bend

For closely spaced miter bends, the allowable pressure is calculated from equations 6.3.4-1 and 6.3.4-2.

$$P = \min \left[\frac{fze^2}{r(e + 0.643 \tan \theta \sqrt{re})}, \frac{fze(R_s - r)}{r(R_s - r/2)} \right] \text{ with } \theta \leq 22.5$$

For widely spaced miter bends, the allowable pressure is calculated from equations 6.3.4-1, 6.3.4-2 and 6.3.5-1

$$P = \min \left[\frac{fze^2}{r(e + 0.643 \tan \theta \sqrt{re})}, \frac{fze(R_s - r)}{r(R_s - r/2)} \right] \text{ with } \theta \leq 22.5$$

$$P = \frac{fze^2}{r(e + 1.25 \tan \theta \sqrt{re})} \text{ with } \theta \leq 22.5$$

where

r = mean radius of pipe = $(D - t)/2$

R_S = effective bend radius of the miter

θ = miter half angle

Sustained Stress

The stress (σ_1) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from equation (12.3.2-1)

$$\sigma_1 = \frac{PD_o}{4e_n} + \frac{0.75iM_A}{Z} \leq f_h$$

where

P = maximum of CAEPIPE input pressures P1, P2 and P3

D_o = outside diameter

e_n = nominal pipe wall thickness

i = stress intensification factor. The product $0.75i$, shall not be less than 1.0

M_A = resulting bending moment due to sustained loads

Z = uncorroded section modulus; for reduced outlets / branch connections, effective section modulus

f_h = hot allowable stress

Sustained plus Occasional Stress

The stress (σ_2) due to sustained and occasional loads is calculated from equation (12.3.3-1) as the sum of stress due to sustained loads such as due to pressure, weight and other sustained mechanical loads and stress due to occasional loads such as earthquake or wind. Wind and earthquake are not considered concurrently

$$\sigma_2 = \frac{PD_o}{4e_n} + \frac{0.75iM_A}{Z} + \frac{0.75iM_B}{Z} \leq kf_h$$

where

M_B = resultant bending moment due to occasional load

Z = 1.2 if the occasional load is acting less than 1% in any 24 hour operating period. In CAEPIPE

k = 1.2; user can change this input in Layout window > Options > Analysis > Code > EN13480.

Expansion Stress

The stress (σ_3) due to thermal expansion is calculated from equation (12.3.4-1)

$$\sigma_3 = \frac{iM_C}{Z} \leq f_a$$

where

M_C = resultant moment due to thermal expansion and alternating loads

Z = uncorroded section modulus; for reduced outlets / branch connections, effective section modulus

$$f_a = U(1.25f_c + 0.25f_h) + \frac{E_h}{E_c}$$

- U = cyclic stress range reduction factor taken from table 12.1.3-1
 f_h = allowable stress at hot temperature
 E_c = elastic modulus at cold temperature
 E_h = elastic modulus at hot temperature

If the above condition in equation (12.3.4-1) is not met, equation (12.3.4-2) may be used.

$$\sigma_4 = \frac{PD_o}{4e_n} + \frac{0.75iM_A}{Z} + \frac{iM_C}{Z} \leq f_h + f_h$$

Additional Conditions for the Creep Range

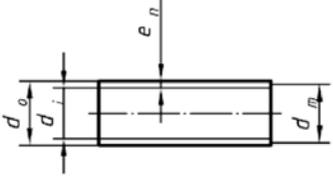
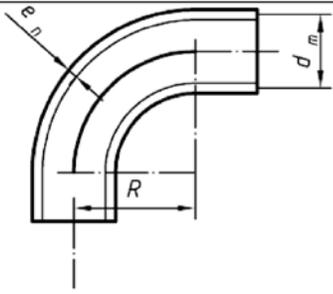
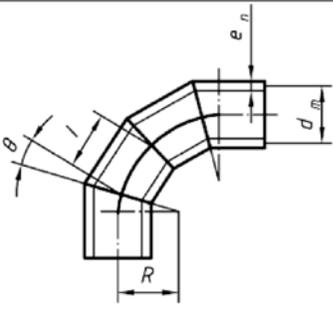
For piping operating within the creep range, the stress, (σ_5), due to sustained, thermal and alternating loadings shall satisfy the equation (12.3.5-1) below.

$$\sigma_5 = \frac{PD_o}{4e} + \frac{0.75iM_A}{Z} + \frac{0.75iM_C}{Z} \leq f_{cr}$$

where

f_{cr} = allowable creep stress value

Table H.1 — Flexibility characteristics, flexibility and stress intensification factors and section moduli for general cases

N°	Designation	Sketch	Flexibility characteristic h	Flexibility factor k_B^a	Stress intensification factor i	Section modulus Z
1	straight pipe		1	1	1	
2	plain bend		$\frac{4Re_n}{d_m^2}$	$\frac{1,65}{h}$	$\frac{0,9}{h^{2/3}}^{b,c}$	$\frac{\pi}{32} \frac{d_o^4 - d_i^4}{d_o}$
3	Closely spaced mitre bend $l < r(1 + \tan \theta)$ $(l = 2R \tan \theta)$		$\frac{4Re_n}{d_m^2}$ with $R = \frac{l \cot \theta}{2}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}^{b,c}$	

(to be continued)

Table H.1 (continued)

N°	Designation	Sketch	Flexibility characteristic h	Flexibility factor k_B^a	Stress intensification factor i	Section modulus Z
4	Single mitre bend or widely spaced mitre bend $l \geq r(1 + \tan \theta)$		$\frac{4Re_n}{d_m^2}$ with $R = \frac{d_m(1 + \cot \theta)}{4}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}^b$	
5	forged welded-in reducer		Shape conditions : $\alpha \leq 60^\circ$ $e_n \geq d_o/100$ $e_2 \geq e_1$	1	$0,5 + \frac{\alpha}{100} \left(\frac{d_o}{e_n} \right)^{1/2}$ max. 2,0 (α in deg.) ^d	
6	tee with welded-on, welded-in or extruded nozzle		$\frac{2e_n}{d_m}$	1	$\frac{0,9}{h^{2/3}}^{be}$	Header : $\frac{\pi}{32} \frac{d_o^4 - d_i^4}{d_o}$
7	as above, however, with additional reinforcing ring		$\frac{2(e_n + 0,5e_{pl})^{5/2}}{d_m e_n^{3/2}}$ with $e_{pl} \leq e_n$	1	$\frac{0,9}{h^{2/3}}^{be}$	Nozzle $\frac{\pi}{4} d_{n,b}^2 e_x$
8	forged welded-in tee with e_n and $e_{n,b}$ as connecting wall thickness		$\frac{8,8e_n}{d_m}$	1	$\frac{0,9}{h^{2/3}}^{bg}$	with e_x as smaller value of $e_{x1} = e_n$ and $e_{x2} = i e_{n,b}$ resp.
9	butt weld		$e_n \leq 5\text{mm}$ and $\delta \leq 0,1e_n^f$ $e_n < 5\text{mm}$ and $\delta > 0,1e_n^f$	1 1	1,0 ^f 1,8 ^f	

(to be continued)

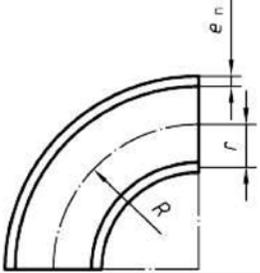
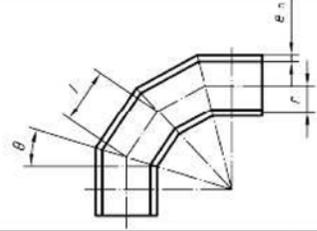
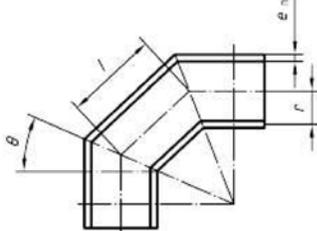
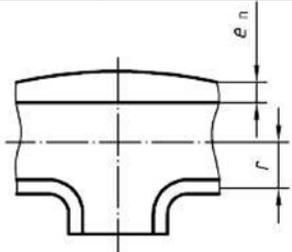
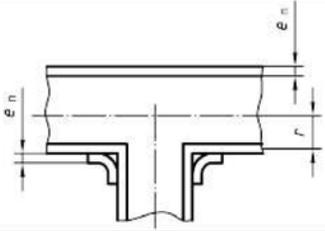
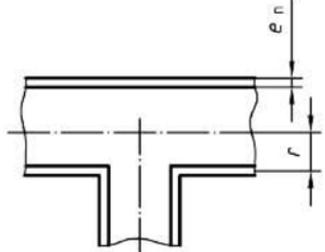
N°	Designation	Sketch	Flexibility characteristic h	Flexibility factor k_B	Stress intensification factor i	Section modulus Z
4	Single mitre bend or widely spaced mitre bend $l \geq r(1 + \tan \theta)$		$\frac{4Re_n}{d_m^2}$ with $R = \frac{d_m(1 + \cot \theta)}{4}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}$	
5	forged welded-in reducer		Shape conditions : $\alpha \leq 60^\circ$ $e_n \geq d_o/100$ $e_2 \geq e_1$	1	$0,5 + \frac{\alpha}{100} \left(\frac{d_o}{e_n} \right)^{1/2}$ max. 2,0 (α in deg.)	
6	tee with welded-on, welded-in or extruded nozzle		$\frac{2e_n}{d_m}$	1	$\frac{0,9}{h^{2/3}}$	Header : $\frac{\pi}{32} \frac{d_o^4 - d_i^4}{d_o}$
7	as above, however, with additional reinforcing ring		$\frac{2(e_n + 0,5e_{pl})^{5/2}}{d_m e_n^{3/2}}$ with $e_{pl} \leq e_n$	1	$\frac{0,9}{h^{2/3}}$	Nozzle $\frac{\pi}{4} d_{m,b}^2 e_x$
8	forged welded-in tee with e_n and $e_{n,b}$ as connecting wall thickness		$\frac{8,8e_n}{d_m}$	1	$\frac{0,9}{h^{2/3}}$	with e_x as smaller value of $e_{x1} = e_n$ and $e_{x2} = i e_{n,b}$ resp.
9	butt weld		$e_n \leq 5\text{mm}$ and $\delta \leq 0,1e_n$ $e_n < 5\text{mm}$ and $\delta > 0,1e_n$	1 1	1,0 1,8	

Table H.1 (concluded)

N°	Designation	Sketch	Flexibility characteristic h	Flexibility factor k_B^a	Stress intensification factor i	Section modulus Z
10	wall thickness transitions		$\alpha \leq 30^\circ$ $\beta \leq 15^\circ$ (without circumferential weld at transitions $\delta = 0$)	1	$1,3 + 0,0036 \frac{d_o}{e_n} + 3,6 \frac{\delta}{e_n}$ max 1,9 ^f	$\frac{\pi}{32} \frac{d_o^4 - d_i^4}{d_o}$
11	fillet welds at set-in connections		concave shape with continuous transition to pipe	1	1,3	smaller value of $\frac{\pi}{32} \frac{d_o^4 - d_i^4}{d_o}$ and
12				1	2,1	$\frac{\pi}{4} d_o^2 a$

- ^a The flexibility factor k_B applies to bending in all planes. The factor related to torsion is equal to 1 in all cases.
- ^b The factors k_B and i apply over the whole effective length of the elbows and bends and at the intersection of the axes in case of tees and nozzles.
- ^c If these components are fitted with :
- flange at one extremity, k_B and i are multiplied by $h^{1/6}$;
 - flange at each of the extremities, k_B and i are multiplied by $h^{1/3}$.
- ^d The wall thickness of the reducer is not less than e_1 except in the vicinity of the small end where however the thickness is not less than e_n .
- ^e Other values may be used subject to justification.
- ^f The factor applies if the fabrication tolerances are met. Otherwise the determination of the factors is the responsibility of the designer.
- ^g The factors only apply to nozzles with convergent axes.

Table H.3 — Flexibility characteristics and stress intensification factors for out-of-plane and in-plane bending

Component description	Out-of-plane i_o	In-plane i_i	Flexibility characteristic	Sketch
Welding elbow or pipe bend	$\frac{0,75}{h^{2/3}}_{abc}$	$\frac{0,9}{h^{2/3}}_{abc}$	$\frac{e_n R}{r^2}$	
Closely spaced mitre bend $l < r(1 + \tan\theta)$ ($l = 2R \tan\theta$)	$\frac{0,9}{h^{2/3}}_{abc}$	$\frac{0,9}{h^{2/3}}_{abc}$	$\frac{\cot\theta}{2} \frac{e_n l}{r^2}$	
Single mitre bend or widely spaced mitre bend $l \geq r(1 + \tan\theta)$	$\frac{0,9}{h^{2/3}}_{abc}$	$\frac{0,9}{h^{2/3}}_{abc}$	$\frac{e_n}{r} \left(\frac{1 + \cot\theta}{2} \right)$	
Forged tee to be welded, designed with a burst pressure greater than or equal to the burst pressure of the connected pipes	$\frac{0,9}{h^{2/3}}_{aefgi}$	$0,75i_o + 0,25_{aefgi}$	$\frac{4,4e_n}{r}$	
Reinforced fabricated tee with pad or saddle	$\frac{0,9}{h^{2/3}}_{adei}$	$0,75i_o + 0,25_{adei}$	$\frac{(e_n + 0,5e_r)^{5/2}}{r(e_n^{3/2})}$	
Unreinforced fabricated tee	$\frac{0,9}{h^{2/3}}_{adei}$	$0,75i_o + 0,25_{adei}$	$\frac{e_n}{r}$	

(to be continued)

Table H.3 (concluded)

Component description	Out-of-plane i_o	In-plane i_i	Flexibility characteristic	Sketch
Extruded welding tee	$\frac{0,9}{h^{2/3}}_{ae}$	$0,75i_o + 0,25_{ae}$	$\left(1 + \frac{r_1}{r}\right) \frac{e_n}{r}$	
Welded in contour insert	$\frac{0,9}{h^{2/3}}_{aefgi}$	$0,75i_o + 0,25_{aefgi}$	$\frac{4,4e_n}{r}$	
Branch welded on fitting (integrally reinforced)	$\frac{0,9}{h^{2/3}}_{adfh}$	$0,75i_o + 0,25_{adfh}$	$\frac{3,3e_n}{r}$	

- ^a The factors i_o and i_i apply over the whole effective length of the elbows and bends and at the intersection of the axes in case of tees and nozzles.
- ^b If these components are fitted with :
- flange at one extremity, i_o and i_i are multiplied by $h^{1/6}$;
 - flange at each of the extremities, i_o and i_i are multiplied by $h^{1/3}$.
- ^c If the pressure is likely to correct ovality (large diameter, small thickness), the factors i_o and i_i shall be divided by $1 + 3,25 \left(\frac{P}{E}\right) \left(\frac{r_m}{e_r}\right)^{5/2} \left(\frac{R}{r_n}\right)^{2/3}$, where P is the operating pressure and E the modulus of elasticity at 20 °C.
- ^d For a nozzle with a ratio of branch diameter to pipe diameter exceeding 0,5, the out-of-plane stress intensification factor may be non-conservative. In addition a smooth transition by a concave shaped weld is proved to reduce the value of this factor. Consequently the selection of an appropriate value for this factor remains the responsibility of the designer.
- ^e The stress intensification factors regarding the branch connections are based on tests carried out with at least two diameters of straight pipe on either side of the branch axis. The case of closer branches requires a particular attention.
- ^f The forgings shall be suitable with regard to the operating conditions.
- ^g When the limitations with respect to radius and thickness are not met and reliable data are not available, the flexibility characteristic is taken as $\frac{e_n}{r}$.
- ^h The designer shall check that the design against pressure is at least equivalent to that for a straight pipe.
- ⁱ The factors only apply to nozzles with convergent axes.

Norwegian (1983)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 6.3.

$$p = \frac{2fz_L T_{eff}}{D_m - T_{eff}}$$

where

p = allowable pressure

f = allowable stress

z_L = joint efficiency of longitudinal weld (input as material property)

T_{eff} = nominal pipe thickness \times (1 – mill tolerance/100) – corrosion allowance

D = outside diameter

d = inside diameter

m = pressure coefficient

= 1.0 for $D/d < 1.6$

= $0.25(D/d) + 0.6$ for $1.6 \leq D/d \leq 2.0$

Sustained Stress

Stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 9.32.

$$S_L = \frac{pD}{4T_{eff}z_C} + \frac{0.75k_1M_A}{W} \leq f_2$$

where

p = maximum pressure

z_C = joint efficiency of circumferential weld (input as material property)

k_1 = stress intensification factor. The product $0.75k_1$ shall not be less than 1.0

M_A = resultant bending moment due to sustained loads

W = section modulus, for reduced outlets, effective section modulus

f_2 = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of the stress due to sustained loads (S_L) and the stress (S_O) due to occasional loads such as earthquake or wind from Equation 9.33. Wind and earthquake are not considered concurrently.

$$S_{LO} = \frac{pD}{4T_{eff}z_C} + \frac{0.75k_1(M_A + M_B)}{W} \leq 1.2f_2$$

where

M_B = resultant bending moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 9.34.

$$S_E = \frac{k_1 M_C}{W} \leq S_r$$

Alternatively, Equation 9.35 may be used.

$$S_L + S_E \leq f_2 + S_r$$

where

M_C = resultant bending moment due to thermal expansion.

S_r = $f_r(1.17R_1 + 0.17R_2)$

f_r = stress range reduction factor taken from Table 9.1.

R_1 = smaller of f_1 and $0.267R_m$

R_2 = smaller of f_2 and $0.367R_m$

f_1 = allowable stress at cold condition

f_2 = allowable stress at hot condition

R_m = tensile strength at room temperature

At moderate temperatures (up to 370° C) for carbon steel, low alloy steel and chromium steel (specified as CS material type) and up to 425° C for austenitic stainless steel (specified as AS material type), the limits $0.267R_m$ and $0.367R_m$ are disregarded and S_r is selected as smaller of S_r' and S_r'' ,

where

$S_r' = 1.17f_1 + 0.20f_2$

$S_r'' = 290f_r - f_2$

Table 9.2 Stress and flexibility coefficients for pipe and pipe components (fittings)

Item	Type of pipe component	Characteristic value h	Flexibility coefficient k	Stress coefficient k_1	Figure
1	Pipe bend, see notes 1 and 2	$\frac{T \cdot R}{r_m^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{2/3}}$	
2	Pipe bend of welded segments with a segment length $L < R (1 + \tan^2 \theta_h)$ see notes 1 and 2	$\frac{L \cdot T \cdot \cot \theta_h}{2 r_m}$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h}$	
3	Pipe bend of welded segments with a segment length $L \geq r_m (1 + \tan^2 \theta_h)$ or inclined circumferential welds, see notes 1 and 3	$\frac{T (1 + \cot^2 \theta_h)}{2 r_m}$	1	$\frac{0.9}{h^{2/3}}$	

Stress and flexibility coefficients for pipe and pipe components (fittings)

Item	Type of pipe component	Characteristic value h	Flexibility coefficient k	Stress coefficient k_1	Figure
4	Tee bend according to MS 5594, see note 1	$4.4 \frac{T_h}{r_h}$	1	$\frac{0.9}{h} \frac{273}{h}$	
5	Branch without local reinforcement of the main pipe, see note 1	$\frac{T_h}{r_h}$	1	$\frac{0.9}{h} \frac{273}{h}$	
6	Branch or main pipe with raised edges radiused at joint between main pipe and branch, see note 1	$\frac{T_e}{T_h} \frac{3}{2} \frac{T_e \cdot R_e}{r_h}$	1	$\frac{0.9}{h} \frac{273}{h}$	

Stress and flexibility coefficients for pipe and pipe components (fittings)

Item	Type of pipe component	Characteristic value h	Flexibility coefficient k	Stress coefficient k _t	Figure
7	Branch on locally thickened main pipe, see note 1	$\frac{T_h + T}{2} \cdot \frac{5/2}{r_h \cdot T_h}$	1	$\frac{0.9}{h^{2/3}}$	
	$L < \frac{d_j}{2} + 1.8 \sqrt{0 \cdot T_f}$ $L \geq \frac{d_j}{2} + 1.8 \sqrt{0 \cdot T_f}$	$\frac{T_f^{5/2}}{r_h \cdot T_h^{3/2}}$	1	$\frac{0.9}{h^{2/3}}$	
8	Branch with plate reinforcement, see notes 1 and 4	$\frac{T_p^{5/2}}{(T_h + \frac{T_p^2}{2}) \cdot \frac{3/2}{r_h \cdot T_h}}$	1	$\frac{0.9}{h^{2/3}}$	

Symbols (items 4 - 8)

r = fillet radius (mm)

r_g = mean radius of branch = $(\frac{d - t_g}{2})$ (mm)

R_e = effective radius = r + r_h (mm)

r_h = mean radius of main pipe = $(\frac{D - T_h}{2})$ (mm)

T_g = specified wall thickness of branch (mm)

T_h = specified wall thickness of main (mm)

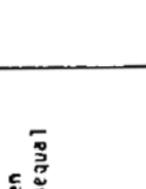
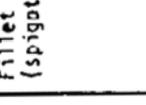
T_e = effective wall thickness = 1/2 (T_p + T_s) (mm)

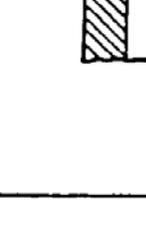
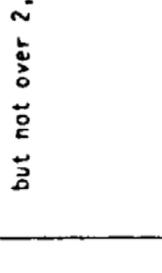
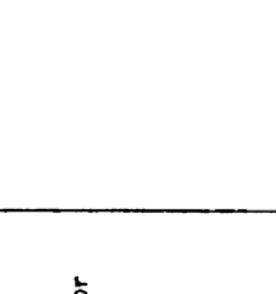
T_f = wall thickness at fillet radius (see item 5) (mm)

T_s = wall thickness of main pipe at a point located diametrically opposite of the branch (see item 5) (mm)

T_r = wall thickness of locally thickened main pipe (mm)

T_p = thickness of plate reinforcement (mm)

Item	Type of pipe component	Flexibility coefficient k	Stress coefficient k_1	Figure
9	Butt weld $T > 4,5$ mm and $d_e/T_{nom} \leq 0,1$ $T \leq 4,5$ mm or $d_e/T_{nom} > 0,1$	1	1,0 1,8 (If the weld is ground flush inside and out, the value 1,0 may be used)	
10	Butt weld between components of unequal thickness	1	$1,3 - 0,0036 \frac{D}{T} + 3,6 \frac{d_e}{T}$ but not over 1,9	
11	Fillet weld (spigot type)	1	2,1	 <p>Flange $a \geq 1,2 T$</p> <p>Welding of Pipe Joint $a \geq 1,05 T$</p>

Item	Type of pipe component	Flexibility coefficient k	Stress coefficient k_1	Figure
12	Concave fillet weld (spigot type) with even transition to the pipe. (no undercut permitted)	1	1,3	 <p>$a \geq 1,05 T$</p>
13	Conical reducer with defined knuckles r_1 and r_2	1	$0,5 + 0,01 \cdot a_1 \sqrt{\frac{D_{min}}{t_1}}$ but not over 2,0	
14	Conical reducer without knuckles, see notes 1 and 5 $t_1 > 4,5$ mm and $d_e/t_1 \leq 0,1$ $t_1 \leq 4,5$ mm or $d_e/t_1 > 0,1$		$0,9 + 0,17 \cdot a_1 \sqrt{\frac{D_{min}}{t_1}}$ $1,25 + 0,023 \cdot a_1 \sqrt{\frac{D_{min}}{t_1}}$ but not less than 1,8	

Item	Type of component	Flexibility coefficient k	Stress coefficient k ₁
15	Threaded joint or connection to threaded flange	1	2,3

Symbols (items 13 - 14)

D_{min} , D_{maks} , α_1 see clause 6.5

T_1 = specified wall thickness of the smaller pipe (mm)

T = specified wall thickness of the larger pipe (mm)

d_e = difference in mean radius at each side of the welding groove (mm)

Notes relating to illustrations shown in table 9.2

All dimensions are nominal

- The coefficients k and k₁ apply to bending in any plane and shall never be given a value less than 1. For plain bends and bends made of welded segments, the coefficients shall be applied to the effective length of the arc (shown by a heavy centreline in the illustrations). For branches and tees, they shall be applied up to the intersection point between the main pipe and the branch, see fig. 9.3.

k and k₁ as functions of h can be taken from fig. 9.4.

- If a flange is joined to one end or both, the coefficients k and k₁ shall be multiplied by the factor F as shown below.

$F = h^{1/6}$ if one end has a flange

$F = h^{1/3}$ if both ends have flanges

F as function of h can be taken from fig. 9.5.

- An inclined circumferential weld at the angle t_h degrees so that

$$t_h \leq 9 \cdot \sqrt{\frac{T}{r_m}}$$

may be considered as a perpendicular circumferential weld.

- When $T_p \geq 2,5 T_h$ is set to $h = 4,05 \cdot \frac{T_h}{r_h}$

- The formula for k₁ applies only when the following conditions are satisfied:

- the reducer is concentric
- the angle α_1 does not exceed 60°
- neither D_{max}/T nor D_{min}/T_1 exceeds 100.

Norwegian (1990)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 6.3.

$$p = \frac{2fz_L T_{eff}}{D_m - T_{eff}}$$

where

p = allowable pressure

f = allowable stress

z_L = joint efficiency of longitudinal weld (input as material property)

T_{eff} = nominal pipe thickness \times (1 – mill tolerance/100) – corrosion allowance

D = outside diameter

d = inside diameter

m = pressure coefficient

= 1.0 for $D/d < 1.6$

= $0.25(D/d) + 0.6$ for $1.6 \leq D/d \leq 2.0$

Sustained Stress

Stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 10.7.

$$S_L = \frac{pD}{4T_{eff}z_C} + \frac{0.75k_1M_A}{W} \leq f_2$$

where

p = maximum pressure

z_C = joint efficiency of circumferential weld (input as material property)

k_1 = stress intensification factor. The product $0.75k_1$ shall not be less than 1.0

M_A = resultant bending moment due to sustained loads

W = section modulus, for reduced outlets, effective section modulus

f_2 = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of the stress due to sustained loads (S_L) and the stress (S_O) due to occasional loads such as earthquake or wind from Equation 10.8. Wind and earthquake are not considered concurrently.

$$S_{LO} = \frac{pD}{4T_{eff}z_C} + \frac{0.75k_1(M_A + M_B)}{W} \leq 1.2f_2$$

where

M_B = resultant bending moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 10.9.

$$S_E = \frac{k_1 M_C}{W} \leq S_r$$

Alternatively, Equation 10.10 may be used.

$$S_L + S_E \leq f_2 + S_r$$

where

M_C = resultant bending moment due to thermal expansion.

S_r = smaller of S_r' and S_r''

S_r' = $1.25f_1 + 0.25f_2$

S_r'' = $f_r R_s - f_2$

f_r = stress range reduction factor

f_1 = allowable stress at cold condition

f_2 = allowable stress at hot condition

R_s = permissible extent of stress for 7000 load cycles for different materials

R_s is determined by material type as follows:

Material	Type	R_s
Carbon and low alloy steel	CS	290
Austenitic stainless steel	AS	400
Copper alloys, annealed	CA	150
Copper alloys, cold worked	CC	100
Aluminum	AL	130
Titanium	TI	200

The stress range reduction factor f_r depends on the number of thermal cycles (NE). For moderate high temperatures ($\leq 370^\circ$ C for carbon, low alloy and chromium steel and $\leq 425^\circ$ C for austenitic stainless steel), f_r is calculated from

$$f_r = (7000/N_E)^{0.2} N_E > 100 \text{ (i.e., } f_r \leq 2.34)$$

At higher temperatures, S_r shall not be greater than $f_r(1.25R_1 + 0.25R_2)$ and f_r shall not be greater than 1.0.

where

R_1 = smaller of f_1 and $0.25R_m$

R_2 = smaller of f_2 and $0.25R_m$

R_m = tensile strength at room temperature

Tabell 10.1 Spennings- og fleksibilitets-koeffisient for rør og rørdeler

Table 10.1 Stress and flexibility coefficients for pipe and pipe components (fittings)

Betegnelser (detalj 4 - 8)

Symbols (item 4-8)

r	= hulkeilradius (mm)
r_g	= grenens middelradius = $(d - T_g)/2$ (mm)
r_h	= hovedrørets middelradius = $(d - T_h)/2$ (mm)
R_e	= $r + r_h$ (mm)
T_e	= $0,5 \times (T_r + T_s)$ (mm)
T_f	= veggtykkelse i lokalt tykkere hovedrør (mm)
T_g	= grenens nominelle veggtykkelse (mm)
T_h	= hovedrørets nominelle veggtykkelse (mm)
T_p	= tykkelsen av plateforsterkning (mm)
T_r	= veggtykkelse ved hulkeilradien (se detalj 6) (mm)
T_s	= veggtykkelse i hovedrørets side rett ovenfor grenen (se detalj 6) (mm)

r	= fillet radius (mm)
r_g	= mean radius of branch = $(d - T_g)/2$ (mm)
r_h	= mean radius of main pipe = $(d - T_h)/2$ (mm)
R_e	= $r + r_h$ (mm)
T_e	= $0,5 \times (T_r + T_s)$ (mm)
T_f	= wall thickness of locally thickened main pipe
T_g	= nominal wall thickness of branch (mm)
T_h	= nominal wall thickness of main pipe (mm)
T_p	= thickness of plate reinforcement (mm)
T_r	= wall thickness at fillet radius (see item 6) (mm)
T_s	= wall thickness of pipe at a point located diametrically opposite of the branch (see item 6) (mm)

Betegnelser (detalj 13 -14)

Symbols (item 13-14)

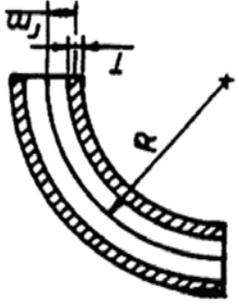
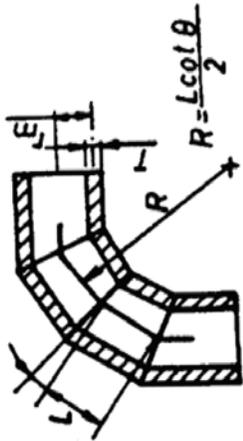
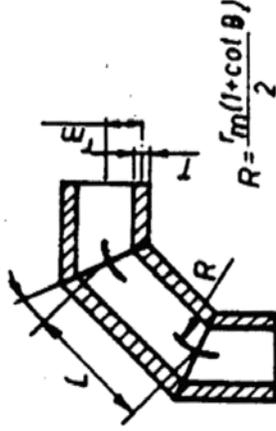
D_{\min} og D_{\max} , se punkt 7.5

D_{\min} and D_{\max} , see clause 7.5.

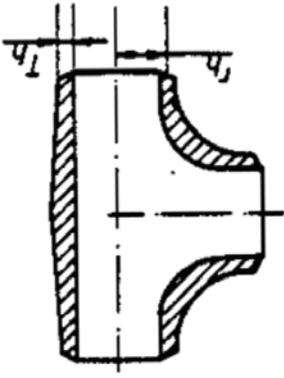
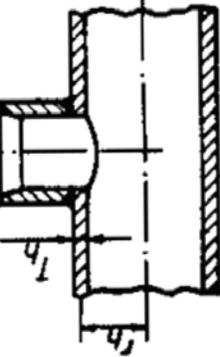
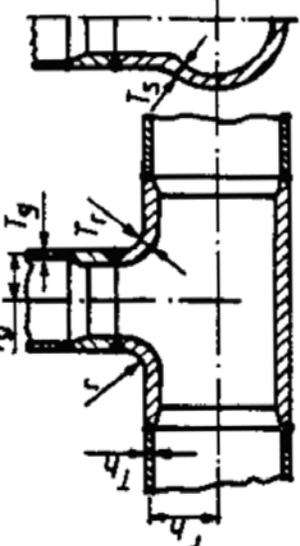
d_e	= forskjell i middelradius på hver side av sveisefuge (mm)
T	= nominell veggtykkelse i det største røret (mm)
T_1	= nominell veggtykkelse i det minste røret (mm)
α	= vinkel, i grader

d_e	= difference in mean radius on each side of the welding groove (mm)
T	= nominal wall thickness of the larger pipe (mm)
T_1	= nominal wall thickness of the smaller pipe (mm)
α	= angle, in degrees

Tabell 10.1 Spennings- og fleksibilitetskoeffisient for rør og rørdeler
 Table 10.1 Stress and flexibility coefficient for pipe and pipe components (fittings)

Detail/Item	Type rørdel Type of pipe component	Karakteristisk verdi h Characteristic value h	Fleksibilitets- koeffisient k Flexibility coefficient k	Spennings- koeffisient k_1 Stress coefficient k_1	Figur Figure
1	Rørbend, se fotnote 1, 2 og 3 Pipe bend, see notes 1, 2 and 3	$\frac{T \cdot R}{r_m^2}$	$\frac{1,65}{h}$	$\frac{0,9}{h^{2/3}}$	
2	Segmentsveist rørbend med segmentlengde $L < r_m \cdot (1 + \tan \theta)$ se fotnote 1, 2 og 3 Pipe bend of welded segments with a segment length $L < r_m \cdot (1 + \tan \theta)$ see notes 1, 2 and 3	$\frac{L \cdot T \cdot \cot \theta}{2 \cdot r_m^2}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}$	
3	Segmentsveist rørbend med segmentlengde $L \geq r_m \cdot (1 + \tan \theta)$ eller skrå rundsveis, se fotnote 1 og 4 Pipe bend of welded segments with a segment length $L \geq r_m \cdot (1 + \tan \theta)$ or inclined circum-ferential welds, see notes 1 and 4	$\frac{T \cdot (1 + \cot \theta)}{2 \cdot r_m}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}$	

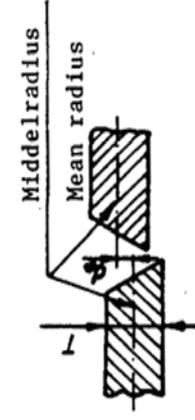
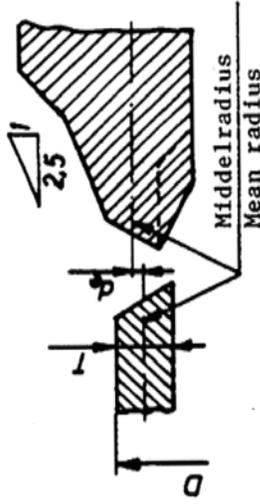
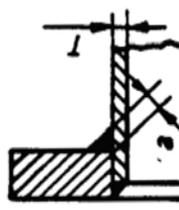
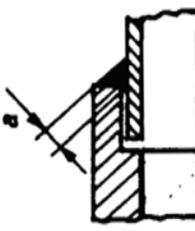
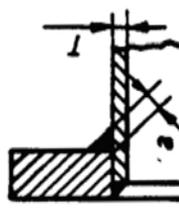
Tabell 10.1 fortsatt
Table 10.1 continued

Detail Item	Type of pipe component	Characteristic value h	Flexibilitets- koeffisient k	Spennings- koeffisient k ₁	Figur Figure
4	T-rør etter NS 5594, se fotnote 1 Tee bend according to NS 5594, see note 1	$4,4 \cdot \frac{T_h}{r_h}$	1	$\frac{0,9}{h^{2/3}}$	
5	Gren uten lokal forsterkning av hovedrøret, se fotnote 1 Branch without local reinforcement of the main pipe, see note 1	$\frac{T_h}{r_h}$	1	$\frac{0,9}{h^{2/3}}$	
6	Gren eller utkraging med radius i overgang mellom hovedrør og grenrør, se fotnote 1 Branch or main pipe with raised edges radiused at joint between main pipe and branch, see note 1	$\left(\frac{T_e}{T_h}\right)^{3/2} \cdot \frac{T_e \cdot R_e}{r_h^2}$	1	$\frac{0,9}{h^{2/3}}$	

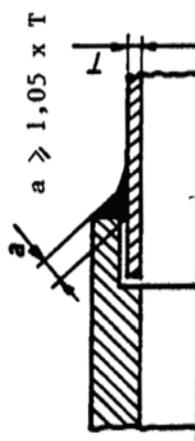
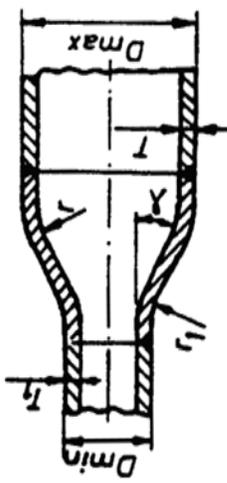
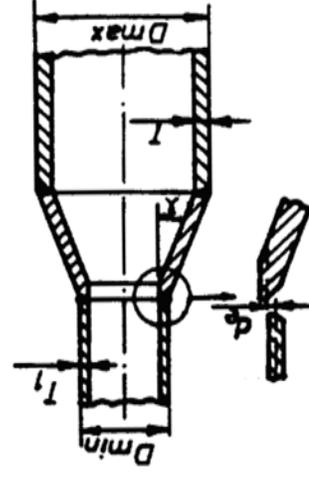
Tabell 10.1 fortsatt
Table 10.1 continued

Detalj Item	Type rørdel Type of pipe component	Karakteristisk verdi h Characteristic value h	Fleksibilitets- koeffisient k Flexibility coefficient k	Spennings- koeffisient k ₁ Stress coefficient k ₁	Figur Figure
7	Gren med lokalt tykkere hoved- rør, se fotnote 1 Branch on locally thickened main pipe, see note 1 $L < \frac{d_1}{2} + 1,8 \sqrt{D \cdot T_f}$ $L \geq \frac{d_i}{2} + 1,8 \sqrt{D \cdot T_f}$	$\frac{\left(\frac{T_h}{2} + \frac{T_f}{2}\right)^{5/2}}{r_h \cdot T_h^{3/2}}$ $\frac{T_f^{5/2}}{r_h \cdot T_h^{3/2}}$	1	$\frac{0,9}{h^{2/3}}$ $\frac{0,9}{h^{2/3}}$	
8	Gren med plateforsterkning, se fotnote 1 og 5 Branch with plate reinforcement, see notes 1 and 5	$\frac{\left(\frac{T_p}{2} + \frac{T_h}{2}\right)^{5/2}}{r_h \cdot T_h^{3/2}}$	1	$\frac{0,9}{h^{2/3}}$	

Tabell 10.1 fortsatt
Table 10.1 continued

Detail Item	Type of pipe component	Flexibilitets-koeffisient k	Spenningskoeffisient k_1 Stress coefficient k_1	Figur Figure
9	Buttsveis/Butt weld $T > 4,5$ mm og/and $d_e/T \leq 0,1$ $T \leq 4,5$ mm eller/or $d_e/T > 0,1$	1 1	1,0 1,8 (Slipes sveisen jevn inn-vendig og utvendig kan verdien 1,0 brukes) (If the weld is ground flush inside and out, the value 1,0 may be used)	 
10	Buttsveis ved ulik godstykkelse etter avsnitt 11.3.6 Butt weld between components of unequal thickness according to clause 11.3.6	1	$1,3 - 0,0036 \cdot \frac{D}{T} + 3,6 \cdot \frac{d_e}{T}$ men maks/but max. 1,9	 
11	Kiltsveis (Innstukksveis) Fillet weld (spigot type)	1	2,1	 

Tabell 10.1 fortsatt
Table 10.1 continued

Detail Item	Type of pipe component	Flexibilitets-koeffisient k	Spenningskoeffisient k_1 Stress coefficient k_1	Figur Figure
12	Konkav kilsveis (innstikkveis) med jevn overgang til røret (smelte "diken" ikke tillatt) Concave fillet weld (spigot type) with even transition to the pipe. (no undercut permitted)	1	1,3	
13	Konisk rørovergang med definerte hulkeiler r og r ₁ , se fotnote 1 og 4 Conical reducer with defined knuckles r and r ₁ , see note 1 and 4	1	$0,5 + 0,01 \cdot \alpha \cdot \sqrt{\frac{D_{\min}}{T_1}}$ men maks./but max. 2,0	
14	Konisk rørovergang uten hulkeiler, se fotnote 1 og 4 Conical reducer without knuckles, see notes 1 and 4 $T_1 > 4,5$ mm og/and $d_e/T_1 \leq 0,1$ $T_1 \leq 4,5$ mm eller/for $d_e/T_1 > 0,1$	1	$0,9 + 0,017 \cdot \alpha \cdot \sqrt{\frac{D_{\min}}{T_1}}$ $1,25 + 0,023 \cdot \alpha \cdot \sqrt{\frac{D_{\min}}{T_1}}$ men min./but min. 1,8	

Tabell 10.1 fortsatt
Table 10.1 continued

Detalj Item	Type r�rdel Type of pipe component	Fleksibilitetskoeffisient k Flexibility coefficient k	Spenningskoeffisient k_1 Stress coefficient k_1
15	Gjengeforbindelse eller tilslutning til gjengeflens Threaded joint or connection to threaded flange	1	2,3

Fotnoter til figurene i tabell 10.1

Notes relating to illustrations shown in table 10.1

Alle dimensjoner er nominelle.

All dimensions are nominal.

- 1) Koeffisientene k og k_1 gjelder for bøyning i vilkårlig plan og skal aldri være mindre enn 1. For glatte og segmentsveiste rør bend skal koeffisientene gjelde over den effektive buelengde (markert med tykk senterlinje i figurene). For gren og T-rør skal de gjelde frem til skjæringspunktet mellom hovedrør og gren, se figur 10.5.

- 1) The coefficients k and k_1 apply to bending in any plane and shall never be given a value less than 1. For plain bends and bends made of welded segments, the coefficients shall be applied to the effective length of the arc (shown by a heavy centre-line in the illustrations). For branches and tees, they shall be applied up to the intersection point between the main pipe and the branch, see figure 10.5.

k og k_1 som funksjon av h kan tas fra figur 10.3. Tillatte verdier for h er angitt i figur 10.3.

k and k_1 as functions of h can be taken from figure 10.3. Permissible values for h is given in figure 10.3.

- 2) Hvis en eller begge ender har standard flenstilslutning skal koeffisientene k og k_1 multipliseres med faktoren F som vist nedenfor

- 2) If a standard flange is joined to one end or both, the coefficients k and k_1 shall be multiplied by the factor F as shown below.

$F = h^{1/6}$ om en ende har flens

$F = h^{1/6}$ if one end has a flange

$F = h^{1/3}$ om begge ender har flens

$F = h^{1/3}$ if both ends have flanges

F som funksjon av h kan tas fra figur 10.4. Tillatte verdier for h er angitt i figur 10.4.

F as function of h can be taken from figure 10.4. Permissible values for h is given in figure 10.4.

- 3) For albuer under innvendig trykk kan k endres på følgende måte:

- 3) For elbows under internal pressure k may be changed in the following way:

Under innvirkning av det innvendige overtrykk p minsker koeffisienten k til k_p .

The coefficient k is reduced to k_p due to the effect of the internal pressure, p :

$$k_p = \frac{k}{1 + \frac{R_t}{E} \cdot 6 \left(\frac{r_m}{T}\right)^{4/3} \cdot \left(\frac{R}{r_m}\right)^{1/3}}$$

der

where

E = elastisitetsmodul (N/mm²)

E = modulus of elasticity (N/mm²)

p = innvendig overtrykk (MPa)

p = internal pressure (MPa)

<p>$r_m = (D - T)/2$, rørtverrsnittets middelradius (mm)</p>	<p>$r_m = (D-T)/2$, mean radius of the pipe section (mm)</p>
<p>$R_t = (p \times r_m)/T$, tangentiell spenning</p>	<p>$R_t = (p \times r_m)/T$, tangential stress (mm)</p>
<p>Minste verdi for k_p er 1.</p>	<p>The minimum value of k_p is 1.</p>
<p>4) Skrårundsveis med vinkelen θ i grader slik at</p>	<p>4) An inclined circumferential weld with the angle θ in degrees so that</p>
$\theta \leq 9 \cdot \sqrt{\frac{T}{r_m}}$	
<p>kan betraktes som rett rundsveis.</p>	<p>may be considered as a perpendicular cir- cumferential weld.</p>
<p>5) Når $T_p \geq 2,5 \times T_h$ settes $h = 4,05 \times (T_h/r_h)$</p>	<p>5) When $T_p \geq 2,5 \times T_h$ is set $h = 4,05 \times (T_h/r_h)$</p>
<p>6) Formelen for k_1 gjelder bare under forutsetning av at følgende krav er oppfylt:</p> <ul style="list-style-type: none"> - overgangsstykket er konsentrisk - vinkelen $\alpha \leq 60^\circ$ - D_{maks}/T respektive D_{min}/T_1 ikke er større enn 100. 	<p>6) The formula for k_1 applies only when the following conditions are satisfied:</p> <ul style="list-style-type: none"> - the reducer is concentric - the angle $\alpha \leq 60^\circ$ - neither D_{maks}/T nor D_{min}/T_1 exceeds 100

RCC-M (1985)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 2 of C 3641.1.

$$P = \frac{2SEt_m}{D - 2Yt_m}$$

where

P = allowable pressure

S = allowable stress

E = joint factor (input as material property)

t_m = minimum required thickness, including mechanical and corrosion allowance
= $t \times (1 - \text{mill tolerance}/100) - \text{corrosion allowance}$

t = nominal pipe thickness

D = outside diameter

d = inside diameter

Y = pressure coefficient

= 0.4 for $t < D/6$

= $d/(D + d)$ for $t \geq D/6$

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 6 of C 3652.

$$S_L = \frac{PD}{4t} + \frac{0.75iM_A}{Z} \leq S_h$$

where

P = maximum pressure

D = outside diameter

t = nominal wall thickness

i = stress intensification factor. The product $0.75i$, shall not be less than 1.0

M_A = resultant bending moment due to sustained loads = $\sqrt{M_X^2 + M_Y^2 + M_Z^2}$

Z = section modulus, for reduced outlets, effective section modulus

S_h = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_O), such as earthquake or wind from Equation 10 of C 3654. Wind and earthquake are not considered concurrently.

$$S_{LO} = S_L + \frac{0.75iM_B}{Z} \leq 1.2S_h$$

where

M_B = resultant moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 7 of C 3653.2.

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

M_C = resultant moment due to thermal expansion

$S_A = f(1.25S_c + 0.25S_h)$

S_c = allowable stress at cold temperature

f = stress range reduction factor from Table C 3653.3

The stress due to pressure, weight, other sustained loads and thermal expansion is calculated from Equation 8 of C 3653.2.

$$S_{TE} = S_L + S_E \leq S_h + S_A$$

FIGURE C 3680.1
 FLEXIBILITY AND STRESS INTENSIFICATION FACTORS ($D_o/t_n \leq 100$)

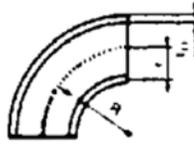
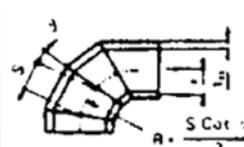
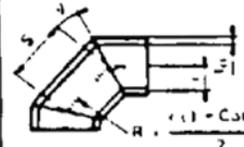
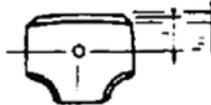
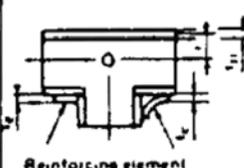
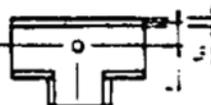
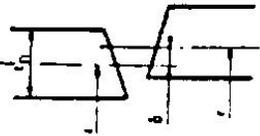
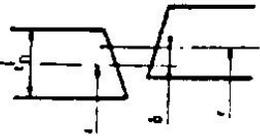
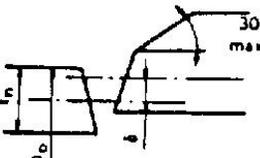
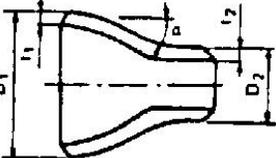
Description	Flexibility characteristic h	Flexibility factor k	Stress intensification factor i	Sketch
Welding elbow or pipe bend. (Notes (1), (2), (3)).	$\frac{t_n R}{r^2}$	$\frac{1.65}{h}$	$\frac{0.9}{h^{2.3}}$	
Closely spaced miter bend. (Notes (1), (2), (3)) $s < r(1 + \tan \theta)$	$\frac{\pi t_n \cot \theta}{2r^2}$	$\frac{1.52}{h^{2.6}}$	$\frac{0.9}{h^{2.3}}$	
Widely spaced miter bend. (Notes (1), (2), (4)) $s \geq r(1 + \tan \theta)$	$\frac{t_n (1 + \cot \theta)}{2r}$	$\frac{1.52}{h^{2.6}}$	$\frac{0.9}{h^{2.3}}$	
Welding tee per ANSI B.16.9 (Notes (1), (2)).	$\frac{4.4 t_n}{r}$	1	$\frac{0.9}{h^{2.3}}$	
Reinforced fabricated tee (Notes (1), (2), (5)).	$\frac{(t_n + \frac{t_b}{2})^{3/2}}{\pi t_n^{3/2}}$	1	$\frac{0.9}{h^{2.3}}$	
Unreinforced fabricated tee (Notes (1), (2)).	$\frac{t_n}{r}$	1	$\frac{0.9}{h^{2.3}}$	

FIGURE C 3680.1
FLEXIBILITY AND STRESS INTENSIFICATION FACTORS (cont'd)

Description	Flexibility factor K	Stress intensification factor i	Sketch
Branch connections [Note (6)]	1	- Analysis of run pipe: $Z = \pi (R_m)^2 T_r$ $i = 0,4 \left(\frac{R_m}{T_r} \right)^{1/2} \left(\frac{r'_m}{R_m} \right)$ with a minimum value of 1,5 - Analysis of branch pipe $Z = \pi (r'_m)^2 T'_b$ $i = 1,5 \left(\frac{R_m}{T_r} \right)^{1/2} \left(\frac{r'_m}{R_m} \right)^{1/2} \left(\frac{T'_b}{T_r} \right) \left(\frac{r'_m}{r_p} \right)$	Fig. C 3680.2
Butt welds [Note (1)] $t_n > 4,75 \text{ mm}$ and $\frac{\delta}{t_n} < 0,1$	1	1,0	
Butt weld [Note (1)] $t_n \leq 4,75 \text{ mm}$ or $\frac{\delta}{t_n} > 0,1$	1	for flush weld 1,0 for re-welded 1,8	
Fillet welded joint, socket welded flange or single welded slip-on flange	1	2,1	Figure C 3661.2 Sketches (C), (D), (E), (F) and (G)
Braze joint	1	2,1	Conformity to C 3671.4.b)
Full fillet weld	1	1,3	Figure C 3661.2 Sketch (H)
Tapered transition $\leq 30^\circ$ (ANSI B 16.25) [Note (1)]	1	1,9 max. or $1,3 + 0,0036 \frac{D_n}{t_n} + 3,6 \frac{\delta}{t_n}$	
Concentric reducers (ANSI B.16.9 or MSS SP 48) [Note (7)]	1	2,0 max. or $0,5 + 0,01 \left(\frac{D_1}{D_2} \right)^{1/2}$	
Threaded pipe joint or threaded flange	1	2,3	—
Corrugated straight pipe or corrugated or creased bend [Note (8)]	5	2,5	—

NOTES TO FIGURE C 3680.1:

- (1) The following nomenclature applies
- r = mean radius of pipe (matching pipe for tees and elbows)
 - t_n = nominal wall thickness of pipe (matching pipe for tees and elbows see note (9))
 - R = bend radius of elbow or pipe bend
 - θ = one-half angle between adjacent miter axes
 - s = miter spacing at centreline
 - t_e = reinforced thickness
 - δ = mismatch
 - D_o = outside diameter
- (2) The flexibility factors k and stress intensification factors i apply to bending in any plane for fittings and shall in no case be taken less than unity. Both factors apply over the effective arc length (shown by heavy centrelines in the sketches) for curved and miter elbows and to the intersection point for tees.
- (3) Where flanges are attached to one or both ends the values of k and i shall be corrected by the factors given below
- one end flanged $c = h/\delta$
 - both ends flanged $c = r/\delta$
- (4) Also includes single miter joints
- (5) When $t_e > 1.5 t_n$ $h = 4.05 t_n/r$
- (6) The equation applies only if the following conditions are met
- a) The reinforcement area requirements of C 3643 are met
 - b) The axis of the branch pipe is normal to the surface of run pipe wall
 - c) For branch connections in a pipe the arc distance measured between the centres of adjacent branches along the surface of the run pipe is not less than three times the sum of their inside radii in the longitudinal direction or is not less than two times the sum of their inside radii along the circumference of the run pipe
 - d) The inside corner radius r_1 (figure C 3680.2) is between 10% and 50% of T_r
 - e) The outer radius r_2 is not less than the larger of $T_b/2$ ($T_b + y/2$) (fig. C 3680.2 sketch (c)) or $T_r/2$
 - f) The outer radius r_3 is not less than the larger of
 - (1) $0.002 \theta D_o$
 - (2) $2 (\sin \theta)^2$ times the offset for the configurations shown in fig. C 3680.2 sketches (a) and (b)
 - g) $R_m/T_r \leq 50$ and $t_m/R_m \leq 0.5$
- (7) The equations apply only if the following conditions are met
- a) Cone angle α does not exceed 60° and the reducer is concentric
 - b) The larger of D_1/t_1 and D_2/t_2 does not exceed 100
 - c) The wall thickness is not less than t_1 throughout the body of the reducer except in and immediately adjacent to the cylindrical portion on the small end where the thickness shall not be less than t
- (8) Factors shown apply to bending flexibility factor for tension equals 0.9
- (9) The designer is cautioned that cast butt welding elbows may have considerably heavier walls than those of the pipe with which they are used. Large errors may be introduced unless the effect of these greater thicknesses is considered.

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from

$$P = \frac{2fze}{D - e}$$

where

P = allowable pressure

S = allowable stress

E = joint factor (input as material property in CAEPIPE)

t_m = nominal pipe thickness \times (1 – mill tolerance/100) – corrosion allowance “c”

(Any additional thickness required for threading, grooving, erosion, corrosion, etc. should be included in corrosion allowance in CAEPIPE)

D = outside diameter

For pipe bends the maximum allowable pressure is calculated using the equivalent pipe wall thickness e_{equi} –

$$e_{equi} = \frac{e}{t_f}$$

Where

$$t_f = \frac{R(D - 0.25)}{R(D - 0.25)}$$

For closely spaced miter bends, the allowable pressure is calculated from

$$P = \min \left[\frac{fze^2}{r(e + 0.643 \tan \theta \sqrt{re})}, \frac{fze^2(R - r)}{r(R - r/2)} \right] \text{ with } \theta \leq 22.5$$

For widely spaced miter bends, the allowable pressure is calculated from

$$P = \min \left[\frac{fze^2}{r(e + 0.643 \tan \theta \sqrt{re})}, \frac{fze^2(R - r)}{r(R - r/2)} \right] \text{ with } \theta \leq 22.5$$

$$P = \frac{fze^2}{r(e + 1.25 \tan \theta \sqrt{re})} \text{ with } \theta \leq 22.5$$

Where

r = mean radius of pipe = (D-t)/2

R_s = effective bend radius of the miter

θ = miter half angle

Y = pressure

coefficient

Sustained Stress

The stress (σ_1) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from

$$\sigma_1 = \frac{PD_o}{4e} + \frac{0.75iM_A}{Z} \leq f_h$$

Where

P = maximum of CAEPIPE input pressures P1, P2 and P3

D_o = outside diameter

e = nominal pipe thickness x [1 - mill tolerance%/100] - corrosion allowance "c"

i = stress intensification factor; the product of 0.75i shall not be less than 1.0

M_A = resulting bending moment due to sustained loads

Z = uncorroded section modulus; for reduced outlets / branch connections, effective section modulus

f_h = hot allowable stress

Sustained plus Occasional Stress

The stress (σ_2) due to sustained and occasional loads is calculated as the sum of stress due to sustained loads such as due to pressure, weight and other sustained mechanical loads and stress due to occasional loads such as earthquake or wind. Wind and earthquake are not considered concurrently

$$\sigma_2 = \frac{PD_o}{4e} + \frac{0.75iM_A}{Z} + \frac{0.75iM_B}{Z} \leq kf_h$$

M_B = resultant bending moment due to occasional load

k = 1.2 if the occasional load is acting less than 1% in any 24 hour operating period. In CAEPIPE $k=1.2$ is used for occasional loading.

Expansion Stress

The stress (σ_3) due to thermal expansion is calculated from

$$\sigma_3 = \frac{iM_C}{Z} + \leq f_a$$

Z = uncorroded section modulus; for reduced outlets / branch connections, effective section modulus

$$f_a = U(1.25f_C + 0.25f_h) \frac{E_h}{E_c}$$

U = cyclic stress range reduction factor as mentioned below

Number of cycle	Stress range reduction factor U
$f_a \leq 7000$	1.0
7001 to 140000	0.9
14001 to 220000	0.8
22001 to 0.7	0.7
45001 to 100000	0.6
100001 to 2000000	0.5

- f_c = allowable stress at cold temperature
 f_h = allowable stress at hot temperature
 E_c = elastic modulus at cold temperature
 E_h = elastic modulus at hot temperature

$$\sigma_4 = \frac{PD_o}{4e} + \frac{0.75iM_A}{Z} + \frac{iM_C}{Z} \leq f_h + f_a$$

Additional Conditions for the Creep Range

For piping operating within the creep range, the stress, (σ_5), due to sustained, thermal and alternating loadings shall satisfy the equation below.

$$\sigma_5 = \frac{PD_o}{4e} + \frac{0.75iM_A}{Z} + \frac{0.75iM_C}{3Z} \leq f_{cr}$$

Where

- f_{cr} = allowable creep stress value

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte

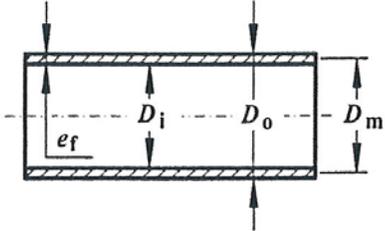
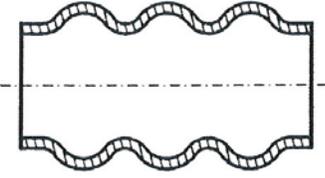
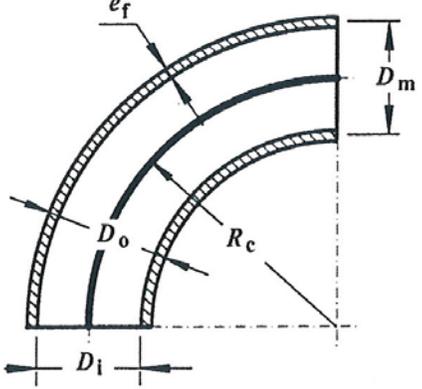
Composant	Caractéristique de flexibilité h	Coefficient de flexibilité k Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			i	i_0 hors du plan	i_i dans le plan	
<p>1. Tuyau droit soudé bout-à-bout</p> 		1	1	1	1	$\frac{\pi}{32} \frac{D_e^4 - D_i^4}{D_e}$
<p>2. Tuyau droit ondulé ou coude ondulé</p> 		5	2,5	2,5	2,5	
<p>3. Coude ou cintre soudé bout-à-bout</p>  <p>Note : Pour les intersections tubulure-coude voir § C2.2.7.4.4.</p>	$\frac{4 e_f R_c}{D_m^2}$	$\frac{1,65}{h}$	$\frac{0,9}{h^{2/3}}$ Notes (2) & (3)	$\frac{0,75}{h^{2/3}}$ Notes (2), (3) & (4)	$\frac{0,9}{h^{2/3}}$ Notes (2), (3) & (4)	$\frac{\pi}{32} \frac{D_e^4 - D_i^4}{D_e}$

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

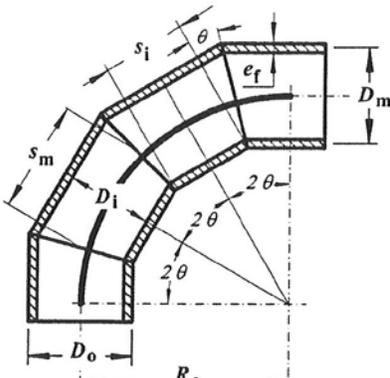
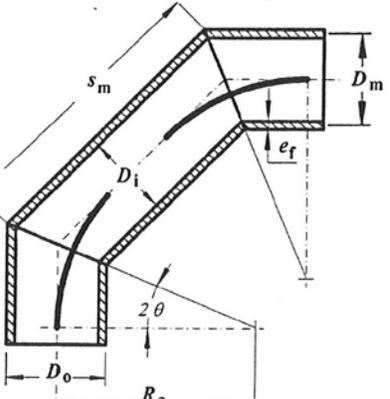
Composant	Caractéristique de flexibilité h	Coefficient de flexibilité k Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			i	i_0 hors du plan	i_i dans le plan	
<p>4.1 Coude à sections rapprochées ou à sections multiples</p>  <p> $s_m \leq \frac{D_m}{2} (1 + \tan \theta)$ $R_c = \frac{s_m \cot \theta}{2}$ </p>	$\frac{4 R_c e_f}{D_m^2}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}$ $s_i \geq 6 e_f$ $\theta \leq 22,5^\circ$ Notes (2) & (3)	$\frac{0,9}{h^{2/3}}$ (hors du plan) $\frac{0,9}{h^{2/3}}$ (dans le plan) Notes (2),(3) & (4)	$\frac{\pi D_o^4 - D_i^4}{32 D_o}$	
<p>4.2 Coude à sections espacées et coude à un onglet</p>  <p> $s_m > \frac{D_m}{2} (1 + \tan \theta)$ $R_c = \frac{D_m (1 + \cot \theta)}{4}$ </p>	$\frac{4 R_c e_f}{D_m^2}$	$\frac{1,52}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}$ $\theta \leq 22,5^\circ$ Note (2)	$\frac{0,9}{h^{2/3}}$ (hors du plan) $\frac{0,9}{h^{2/3}}$ (dans le plan) Notes (2), (3) & (4)	$\frac{\pi D_o^4 - D_i^4}{32 D_o}$	

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

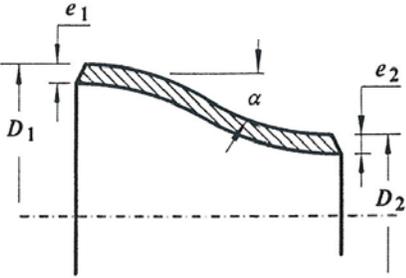
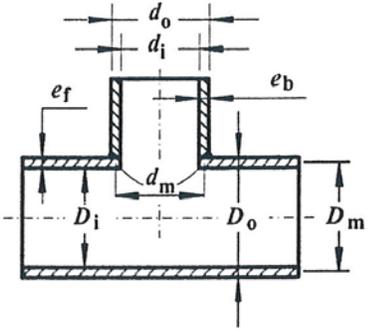
Composant	Caractéristique de flexibilité h	Coefficient de flexibilité k Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			i	i_0 hors du plan	i_i dans le plan	
<p>5.1 Réduction forgée</p> 		1		$0,5 + \frac{\alpha}{100} \left(\frac{D_2}{e_2} \right)^{1/2}$ MAX = 2,0 (α en $^\circ$) $\alpha \leq 60^\circ$ $d_1/e_1 \leq 100$ $d_2/e_2 \leq 100$ Note (5)		
<p>5.2 Réduction chaudronnée voir figure C2.2.4</p>				Note (14)		
<p>6. Té reconstitué non renforcé avec tubulure posée ou pénétrante</p> 	$\frac{2 e_f}{D_m}$	1	$\frac{0,9}{h^{2/3}}$ Notes (2) & (12)	$\frac{0,9}{h^{2/3} (\sin \alpha)^{3/2}}$ $\alpha =$ angle d'inclinaison de la tubulure par rapport au collecteur $\frac{3 i_0}{4} + 0,25$ Notes (2), (6), (7) & (11)	Collecteur $\frac{\pi D_e^4 - D_i^4}{32 D_e}$ Dérivation $\frac{\pi}{4} d_m^2 e_x$ $e_x = \text{MIN}(e_f; i e_b)$	

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

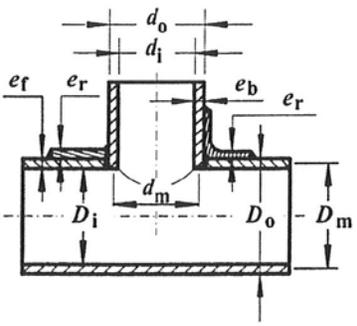
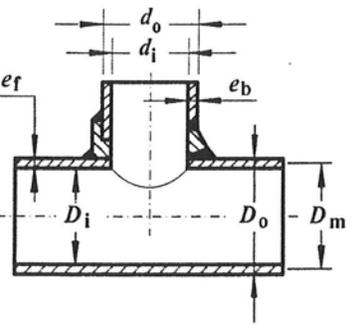
Composant	Caractéristique de flexibilité h	Coefficient de flexibilité k Note (1)	Coefficients d'intensification de contrainte		Module d'inertie
			i	i_0 hors du plan i_i dans le plan	
<p>7. Té reconstitué avec tubulure posée ou pénétrante et renforcé par anneau ou selle de renfort</p>  <p>avec anneau renfort avec selle de renfort</p>	<p>si $e_r \leq 1,5 e_f$</p> $\frac{2(e_f + 0,5 e_r)^{5/2}}{e_f^{3/2} D_m}$ <p>si $e_r > 1,5 e_f$</p> $8 \frac{e_r}{D_m}$	1	$\frac{0,9}{h^{2/3}}$ Notes (2) & (12)	$\frac{0,9}{h^{2/3} (\sin \alpha)^{3/2}}$ $\alpha =$ angle d'inclinaison de la tubulure par rapport au collecteur $\frac{3 i_0}{4} + 0,25$	<p>Collecteur</p> $\frac{\pi D_e^4 - D_i^4}{32 D_e}$ <p>Dérivation</p> $\frac{\pi}{4} d_m^2 e_x$ <p>$e_x = \text{MIN}(e_f ; i e_b)$</p>
<p>8. Piquage avec pièces forgées</p>  <p>Type manchon forgé Type weldolets etc.</p>	$6,6 \frac{e_f}{D_m}$	<p>1 Type manchon forgé</p> <p>1 Type weldolets etc.</p>	$\frac{0,9}{h^{2/3}}$ Notes (2), (6), (8) & (10)	<p>2,1 2,1</p> <p>$\frac{3 i_0}{4} + 0,25$</p>	

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

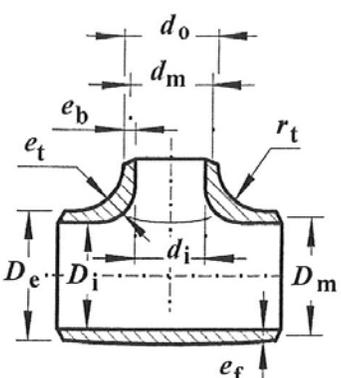
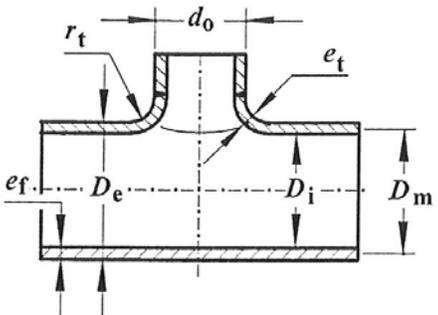
Composant	Caractéristique de flexibilité h	Coefficient de flexibilité k Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			i	i_0 hors du plan	i_i dans le plan	
<p>11. Té forgé</p> 	$\frac{8,8 e_f}{D_m}$	1	$\frac{0,9}{h^{2/3}}$	$\frac{0,9}{h^{2/3}}$	$\frac{3 i_0}{4} + 0,25$	<p>Collecteur</p> $\frac{\pi D_e^4 - D_i^4}{32 D_e}$ <p>Dérivation</p> $\frac{\pi}{4} d_m^2 e_x$ <p>$e_x = \text{MIN}(e_f; i e_b)$</p> <p>Notes (2), (7), (8), (9) & (11)</p> <p>$e_t \geq 1,5 e_f$</p>
<p>12. Té à souder extrudé</p> 	$\left(1 + \frac{2 r_t}{D_m}\right) \frac{2 e_f}{D_m}$	1	Note (13)	$\frac{0,9}{h^{2/3}}$	$\frac{3 i_0}{4} + 0,25$	<p>$r_t \geq 0,05 d_o$</p> <p>$e_t \leq 1,5 e_f$</p> <p>Notes (2) & (7)</p>

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

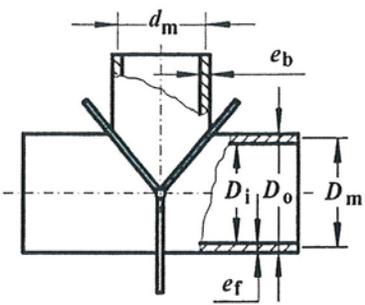
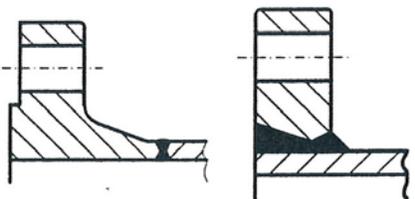
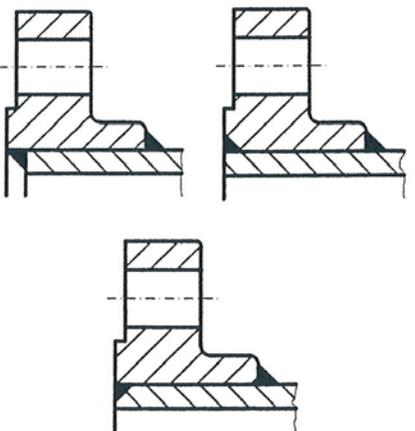
Composant	Caractéristique de flexibilité h	Coefficient de flexibilité k Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			i	i_0 hors du plan	i_i dans le plan	
<p>13. Triform</p> 	$\frac{2 e_f}{D_m}$	1	Note (13)	$\frac{0,9}{h^{2/3}}$	$\frac{3 i_0}{4} + 0,25$	<p>Collecteur $\frac{\pi}{32} \frac{D_o^4 - D_i^4}{D_o}$</p> <p>Dérivation $\frac{\pi}{4} d_m^2 e_x$</p> <p>$e_x = \text{MIN}(e_f ; i e_b)$</p>
<p>14.1 Bride à assembler bout-à-bout</p> 		1		1		
<p>14.2 Bride à emmancher et à souder (soudée des deux côtés)</p> 		1		1,2		

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

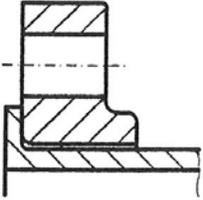
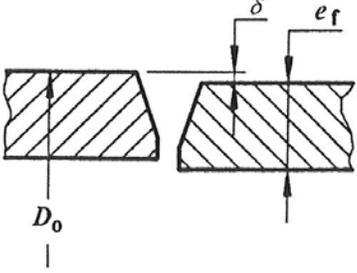
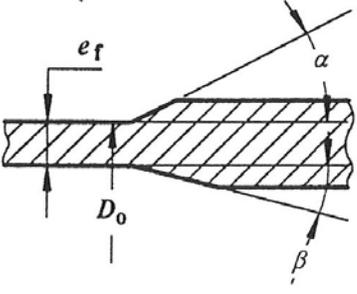
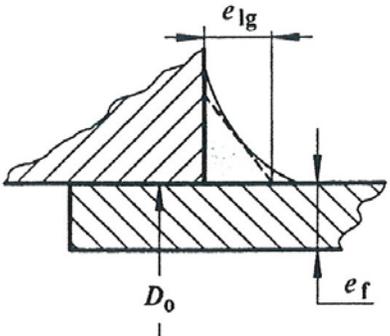
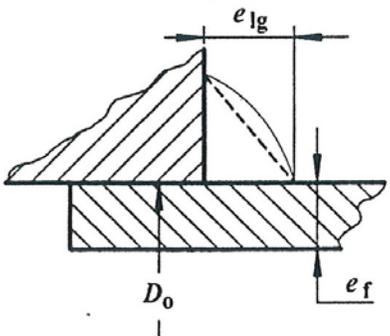
Composant	Caractéristique de flexibilité <i>h</i>	Coefficient de flexibilité <i>k</i> Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			<i>i</i>	<i>i</i> ₀ hors du plan	<i>i</i> _i dans le plan	
14.3 Bride tournantes 		1	1,6			
14.4 Bride à visser Voir figure C2.2.8.3.7-2		1	2,3			
15.1 Soudure bout-à-bout 			1 $e_n \geq 5 \text{ mm}$ et $\delta \leq 0,1 e_f$ Note (15)			$\frac{\pi}{32} \frac{D_o^4 - D_i^4}{D_o}$
			1,8 $e_n < 5 \text{ mm}$ ou $\delta > 0,1 e_f$ Note (15)			
15.2 Transition d'épaisseur de paroi 			$1,3 + 0,0036 D_o / e_f + 3,6 \delta / e_f$ avec un maximum de 1,9 Note (15)			$\frac{\pi}{32} \frac{D_o^4 - D_i^4}{D_o}$
$\alpha \leq 30^\circ$ $\beta \leq 15^\circ$ sans soudure circconférentielle au niveau de la transition $\delta = 0$						

Tableau C3.2.6-1 - Caractéristique de flexibilité, coefficient de flexibilité et coefficients d'intensification de contrainte (suite)

Composant	Caractéristique de flexibilité <i>h</i>	Coefficient de flexibilité <i>k</i> Note (1)	Coefficients d'intensification de contrainte			Module d'inertie
			<i>i</i>	<i>i</i> ₀ hors du plan	<i>i</i> _i dans le plan	
<p>15.3 Composant emmanché-soudé</p>  <p>forme concave avec raccordement régulier au tuyau</p>		1	1,3	$\frac{2,1 e_f}{e_{lg}}$ <p>avec un minimum de 1,3 et un maximum de 2,1</p>	$\frac{\pi}{32} \frac{D_o^4 - D_i^4}{D_o}$ $\frac{\pi}{4} D_o^2 e_f$	
<p>15.4 Composant emmanché-soudé</p>  <p>forme convexe</p>		1	2,1	$\frac{2,1 e_f}{e_{lg}}$ <p>avec un minimum de 1,3 et un maximum de 2,1</p>	$\frac{\pi}{32} \frac{D_o^4 - D_i^4}{D_o}$ $\frac{\pi}{4} D_o^2 e_f$	

Notes du tableau C3.2.6-1

(1) Le coefficient de flexibilité k s'applique à la flexion dans tous les plans. Le coefficient pour la torsion est égal à 1 dans tous les cas à l'exception du cas 2 (tuyau droit et coude ondulé) pour lequel il est égal à 0,9.

(2) Les coefficients k, i, i_0, i_1 s'appliquent sur toute la longueur effective des coudes et des cintres (ligne épaissie sur les schémas) et à l'intersection des axes pour les tés et les piquages.

(3) Si ces composants sont munis :

- d'une bride à l'une de leurs extrémités, k, i, i_0, i_1 sont à multiplier par $h^{1/6}$,
- d'une bride à leurs deux extrémités, k, i, i_0, i_1 sont à multiplier par $h^{1/3}$.

(4) Si la pression est susceptible d'apporter une correction d'ovalisation (grand diamètre, petite épaisseur), les coefficients sont divisés par :

$$- 1 + 6 \left(\frac{P}{E} \right) \left(\frac{r_m}{e_r} \right)^{7/3} \left(\frac{R_c}{r_m} \right)^{1/3} \quad \text{pour le coefficient } k.$$

$$- 1 + 3,25 \left(\frac{P}{E} \right) \left(\frac{r_m}{e_r} \right)^{5/2} \left(\frac{R_c}{r_m} \right)^{2/3} \quad \text{pour les coefficients } i, i_0, i_1.$$

P étant la pression de service et E le module d'élasticité à 20 °C.

(5) L'épaisseur de la paroi de la réduction ne doit pas être inférieure à e_1 sauf au voisinage de l'extrémité de petit diamètre où, toutefois, l'épaisseur de la paroi ne doit pas être inférieure à e_2 .

(6) Le coefficient d'intensification de contrainte « hors du plan », pour un piquage dont le rapport des diamètres de la tubulure et du collecteur est supérieur à 0,5, peut être non-conservatif. Par ailleurs, il a été démontré qu'un raccordement régulier par une soudure de forme concave réduit la valeur de ce coefficient. Le choix d'une valeur appropriée pour ce coefficient reste donc de la responsabilité du Concepteur.

(7) Les coefficients d'intensification de contrainte pour les raccords de tubulures sont basés sur des essais avec au moins deux diamètres de tuyau droit de chaque côté de l'axe de la tubulure. Le cas de tubulures plus proches requiert une attention particulière.

(8) Les pièces forgées utilisées doivent être appropriées aux conditions de service.

(9) Lorsque les limitations portant sur le rayon et l'épaisseur ne sont pas respectées et en l'absence de données fiables la caractéristique de flexibilité doit être prise égale à $\frac{2 e_f}{D_m}$.

(10) Le Concepteur doit vérifier que le dimensionnement en fonction de la pression est au moins équivalent à celui du tuyau droit.

(11) Les coefficients ne s'appliquent qu'aux piquages à axes concourants.

(12) D'autres valeurs peuvent être utilisées sous réserve de justification.

(13) En l'absence de données fiables, la détermination des coefficients est de la responsabilité du Concepteur.

(14) En l'absence de données plus précises, les coefficients d'intensification de contrainte peuvent être pris égaux à 2,5.

(15) Le coefficient s'applique dans le cas où les tolérances de fabrication (voir partie F) sont respectées. Dans le cas contraire, la détermination des coefficients est de la responsabilité du Concepteur.

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from

$$S_L = \frac{P(D - t)}{4t} + \frac{0.75iM_A}{Z} \leq S_h$$

where

P = maximum pressure

D = outside diameter

t = nominal wall thickness

i = stress intensification factor. The product $0.75i$, shall not be less than 1.0

M_A = resultant bending moment due to sustained loads = $\sqrt{M_X^2 + M_Y^2 + M_Z^2}$

Z = section modulus, for reduced outlets, effective section modulus

S_h = hot allowable stress: smaller of:

0.67 yield stress at temperature

0.44 tensile strength at 20° C

0.67 rupture stress (after 100,000 hours at temperature)

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained loads (S_L) and stress due to occasional loads (S_O), such as earthquake or wind. Wind and earthquake are not considered concurrently.

$$S_{LO} = S_L + \frac{0.75iM_B}{Z} \leq 1.2S_h$$

where

M_B = resultant moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from

$$S_E = \frac{iM_C}{Z} \leq S_A$$

where

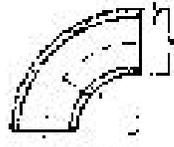
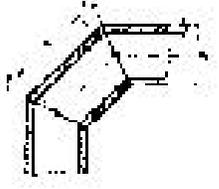
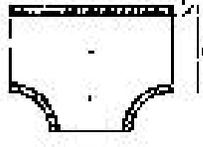
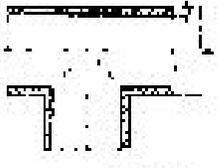
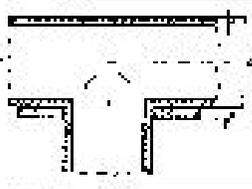
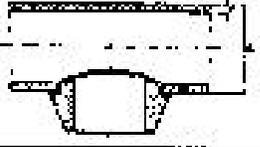
M_C = resultant moment due to thermal expansion

S_A = smaller of:

C_f (0.8 yield strength at 20° C + 0.13 hot yield strength)

C_f (0.8 yield strength at 20° C + 0.2 x 0.67 rupture stress)

C_f = stress range reduction factor

		In-afgevoert		Pipe & Component		n		k		i		
		gladde bocht met m (< 3) flanges			$\frac{4L \cdot d}{(D_o - d)^2}$	$1,05m^{-0,4}$	$0,94 \frac{m^{-0,4}}{k}$					
		keuze segmentbocht met m (< 2) flenzen			$\frac{2d \cdot s}{(D_o - d)^2} (\cos \varphi)$	$1,22m^{-0,2}$	$0,72m^{-0,4}$					
		lange segmentbocht widely spaced mitre bend			$\frac{d}{D_o - d} (1 - \cos \varphi)$	$1,22m^{-0,1}$	$0,72m^{-0,1}$					
		gevoerd of gevormd forged or pressed			$\frac{1,8L}{D_o - d}$							
		opgebouwd, oversterkt fabricated, overreinforced			$\frac{L}{D_o - d}$							
Versterkt en afgeklingd Thick end to anchor		opgebouwd met ringversterking	$d_1 \leq 1,2d$		$\frac{2(d + 0,2d_1)^{2,5}}{(D_o - d) \cdot d^{1,5}}$	1	$0,94m^{-0,5}$					
		fabricated, with pad or weld reinforcement	$d_1 > 1,2d$		$\frac{L \cdot d}{D_o - d}$							
		opgebouwd, versterkt fabricated, reinforced			$\frac{L}{D_o - d}$							
Bijzondere aansluitingen pipe unusual joints		stomp las butt weld			-			1				
		schroefverbinding screwed connection			-			1,1				
		blat flens met twee bocklassen				-			1,0			
		double welded slip-on flange				-			1,0			

Swedish (1978)

Allowable Pressure

The allowable pressure for straight pipes and bends is calculated from Equation 6.3.

$$p = \frac{2\sigma_{tn}z_L s_{eff}}{D_y m - s_{eff}}$$

where

p = allowable pressure

σ_{tn} = allowable stress

z_L = joint efficiency of longitudinal weld (input as material property)

s_{eff} = nominal pipe thickness \times (1 – mill tolerance/100) – corrosion allowance

D_y = outside diameter

D_i = inside diameter = $D_y - s_{eff}$

m = pressure coefficient

= 1.0 for $D_y/D_i < 1.6$

= $0.25(D_y/D_i) + 0.6$ for $1.6 \leq D_y/D_i \leq 2.0$

Sustained Stress

Stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) is calculated from Equation 9.37.

$$S_L = \frac{p_{max} D_y}{4s_{eff} z_C} + \frac{0.75k_1 M_A}{W_y} \leq \sigma_{tn2}$$

where

p_{max} = maximum pressure

z_C = joint efficiency of circumferential weld (input as material property)

k_1 = stress intensification factor. The product $0.75k_1$ shall not be less than 1.0

M_A = resultant bending moment due to sustained loads

W_y = section modulus, for reduced outlets, effective section modulus

σ_{tn2} = hot allowable stress

Occasional Stress

The stress (S_{LO}) is calculated as the sum of the stress due to sustained loads (S_L) and the stress (S_O) due to occasional loads such as earthquake or wind from Equation 9.38. Wind and earthquake are not considered concurrently.

$$S_{LO} = \frac{p_{max} D_y}{4s_{eff} z_C} + \frac{0.75k_1 (M_A + M_B)}{W_y} \leq 1.2\sigma_{tn2}$$

where

M_B = resultant bending moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from Equation 9.39.

$$S_E = \frac{k_1 M_C}{W_y} \leq S_r$$

where

M_C = resultant bending moment due to thermal expansion

S_r = $f(1.17\sigma_1 + 0.17\sigma_2)$ from Equation 9.43

f = stress range reduction factor taken from Table 9.1

σ_1 = smaller of σ_{tn1} and $0.267R_m$

σ_2 = smaller of σ_{tn2} and $0.367R_m$

σ_{tn1} = allowable stress at cold condition

σ_{tn2} = allowable stress at hot condition

R_m = tensile strength at room temperature

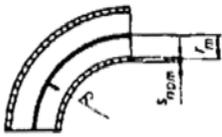
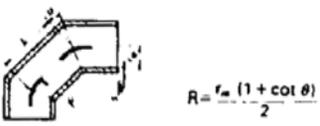
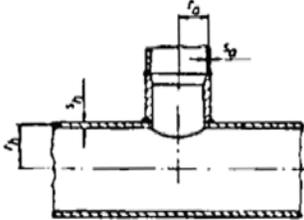
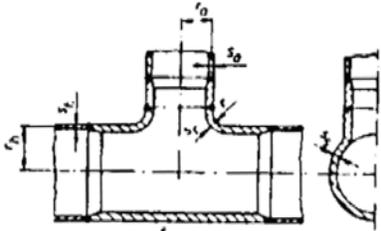
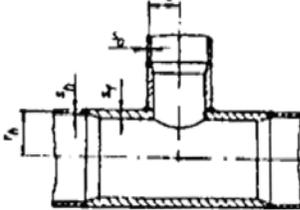
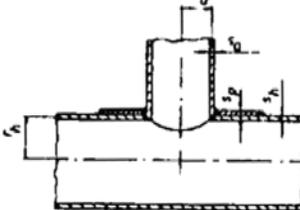
At moderate temperatures (up to 370° C) for carbon steel, low alloy steel and chromium steel (specified as CS material type) and up to 425° C for austenitic stainless steel (specified as AS material type), the limits $0.267R_m$ and $0.367R_m$ are disregarded and S_r is selected as smaller of S_r' and S_r'' ,

where

$$S_r' = 1.17\sigma_{tn1} + 0.20\sigma_{tn2}$$

$$S_r'' = 290f - \sigma_{tn2}$$

Table 9:2 k and k_1

Item	Type of fitting	h	k	k_1	Figure
1	Bend, see notes 1 and 2	$\frac{s_{nom} \cdot R}{r_m^2}$	$\frac{1,65}{h}$	$\frac{0,9}{h^{2/3}}$	
2	Mitre bend of segment length L smaller than $r_m (1 + \tan \theta)$; see notes 1 and 2	$\frac{L \cdot s_{nom} \cdot \cot \theta}{2 r_m^2}$	$\frac{1,5}{h^{5/6}}$	$\frac{0,9}{h^{2/3}}$	
3	Mitre bend of segment length L not less than $r_m (1 + \tan \theta)$ or inclined circumferential welds; see notes 1 and 3	$\frac{s_{nom} (1 + \cot \theta)}{2 r_m}$	1	$\frac{0,9}{h^{2/3}}$	
4	Branch without local reinforcement of the main pipe; see note 1	$\frac{s_b}{r_b}$	1	$\frac{0,9}{h^{2/3}}$	
5	Raised edges on main pipe or branch radiused at joint; see note 1	$\left(\frac{s_b}{s_h}\right)^{3/2} \cdot \frac{s_b \cdot R_e}{r_b^2}$	1	$\frac{0,9}{h^{2/3}}$	
6	Branch on locally thickened main pipe; see note 1	$L \cdot \frac{d_f}{2} + 1,8 \sqrt{D_v \cdot s_f}$	$\frac{\left(\frac{s_h + s_f}{2}\right)^{5/2}}{r_b \cdot s_h^{3/2}}$	$\frac{0,9}{h^{2/3}}$	
	$L \cdot \frac{d_f}{2} + 1,8 \sqrt{D_v \cdot s_f}$	$\frac{s_f^{5/2}}{r_b \cdot s_h^{3/2}}$	1	$\frac{0,9}{h^{2/3}}$	
7	Branch with plate reinforcement; see notes 1 and 4	$\frac{\left(s_h \frac{s_p}{2}\right)^{5/2}}{r_b \cdot s_h^{3/2}}$	1	$\frac{0,9}{h^{2/3}}$	

Symbols (items 4-7)

r = fillet radius
 r_m = mean radius of branch $\left[= \frac{d \cdot s - s_m}{2} \right]$
 R_e = effective radius $= r + r_b$
 r_b = mean radius of main pipe $\left[= \frac{D_v - s_b}{2} \right]$
 s_b = nominal wall thickness of branch

s_b = nominal wall thickness of main pipe
 s_e = effective wall thickness $= 0,5(s_b + s_1)$
 s_f = wall thickness in fillet (see item 5)
 s_h = wall thickness of main pipe side at branch (see item 5)
 s_l = wall thickness of locally thickened main pipe
 s_p = thickness of plate reinforcement

Table 9:2 (cont.)

Item	Type of fitting	h	k	k_1	Figure
8	Butt weld $s_{nom} > 4,5$ mm and $\delta/s_{nom} \leq 0,1$ $s_{nom} \leq 4,5$ mm or $\delta/s_{nom} > 0,1$		1 1	1,0 1,8 (If the weld is ground flush inside and out, k_1 may be set = 1,0)	
9	Butt weld between dissimilar thicknesses according to Section 11.2.2		1	$1,3 + 0,0036 \frac{D_r}{s_{nom}} + 3,6 \frac{\delta}{s_{nom}}$ but not over 1,9	
10	Fillet weld (spigot type)		1	2,1	
11	Concave fillet weld (spigot type) with even transition to the pipe (no undercut allowed)		1	1,3	
12	Conical reducer with defined knuckles r and r_1 , complying with Section 6.5.2.3; see notes 1 and 5		1	$0,5 + 0,01 \alpha \sqrt{\frac{D_{r, min}}{s_{1nom}}}$ but not over 2,0	
13	Conical reducer without knuckles; see notes 1 and 5 $s_{1nom} > 4,5$ mm and $\delta/s_{nom} \leq 0,1$ $s_{1nom} \leq 4,5$ mm or $\delta/s_{1nom} > 0,1$		1	$0,9 + 0,017 \cdot \alpha \sqrt{\frac{D_{r, min}}{s_{1nom}}}$ $1,25 + 0,023 \cdot \alpha \sqrt{\frac{D_{r, min}}{s_{1nom}}}$ but not less than 1,8	
14	Thread joint or connection to threaded flange		1	2,3	

Symbols (items 12-13)

$D_{r, min}$, $D_{r, max}$, α : see Section 6.5
 s_{1nom} = nominal wall thickness of the smaller pipe
 s_{nom} = nominal wall thickness of the larger pipe
 δ = difference in mean radius across a weld

Allowable Pressure

The allowable pressure is calculated from 5.3.2.1.

$$P = 2StFLJT/D$$

where

P = allowable pressure

S = specified minimum yield strength

t = minimum wall thickness

= nominal wall thickness \times (1 - mill tolerance/100) - corrosion allowance

F = design factor = 0.8 for steel pipe (from 5.2.3)

L = location factor (from Table 5.1)

J = joint factor (from Table 5.2), input as material property

T = temperature derating factor for steel pipe (from Table 5.3)

D = outside diameter

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) and wind loads, is calculated from (5.6.3.5)

$$S_L = \frac{PD}{4t} + i \frac{M}{Z} \leq SFLJT$$

where

P = maximum pressure

D = outside diameter

t = minimum wall thickness

i = stress intensification factor

M = resultant bending moment due to sustained loads

Z = corroded section modulus

S = specified minimum yield strength

F = design factor

L = location factor

J = joint factor

T = temperature derating factor

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained and wind loads (S_L) and stress due to occasional loads (S_O), such as earthquake. The allowable stress for occasional stress is not given in the code, therefore it is conservatively taken the same as the sustained stress allowable.

$$S_{LO} = S_L + i \frac{M}{Z} \leq SFLJT$$

where

M = resultant bending moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from (5.6.3.3).

$$S_E = \sqrt{S_b^2 + 4S_t^2} \leq 0.72ST$$

where

S_b = resultant bending stress = iM_b/Z

S_t = torsional stress = $M_t/2Z$

M_b = resultant bending moment due to expansion loads

M_t = torsional moment due to expansion loads

Z = uncorroded section modulus

Table 5.5
Flexibility and Stress Intensification Factors

Description	Flex- ibility Factor <i>k</i>	Stress Intens. Factor <i>i</i>	Description	Flex- ibility Factor <i>k</i> *	Stress Intens. Factor <i>i</i> *	Flexibility Character- istic <i>h</i>	See sketch
Buttwelded joint, reducer, or welding neck flange	1	1.0	Welding elbow, or pipe bend†	$\frac{1.65}{h}$	$\frac{0.9}{h^{2/3}}$	$\frac{1R}{r^2}$	A
Double-welded slip-on or socket welding flange	1	1.2	Mitre bend with close spacing‡ $s < r(1 + \tan \alpha)$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{\cot \alpha}{2} \cdot \frac{ts}{r^2}$	B
Fillet welded joint, or single-welded socket welding flange	1	1.3	Mitre bend with wide spacing ‡§ $s \geq r(1 + \tan \alpha)$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{1 + \cot \alpha}{2} \cdot \frac{t}{r}$	C
Lap joint flange (with CSA Standard CAN3-Z245 12) (lap joint stub)	1	1.6	Welding tee per CSA Standard CAN3-Z245 11	1	$\frac{0.9}{h^{2/3}}$	$4.4 \frac{1}{r}$	D
Screwed pipe joint, or screwed flange	1	2.3	Reinforced fabricated tee, with pad or saddle	1	$\frac{0.9}{h^{2/3}}$	$\frac{(t + 1/2 T)^{5/2}}{t^{3/2} r}$	E
Corrugated pipe, straight or curved, or creased bend**	5	2.5	Unreinforced fabricated tee	1	$\frac{0.9}{h^{2/3}}$	$\frac{t}{r}$	F

*The flexibility factors and stress intensification factors apply to fittings of the same nominal wall thickness as the pipe used in the pipeline system, and shall in no case be taken as less than unity. They apply over the effective arc length (shown by dash-dot lines in the sketches) for curved and mitre elbows, and to the intersection point for tees.

Where flanges are attached to one or both ends, the values of *k* and *i* in the Table shall be multiplied by the following factors:

One end flanged: $(h)^{1/6}$

Both ends flanged: $(h)^{1/3}$

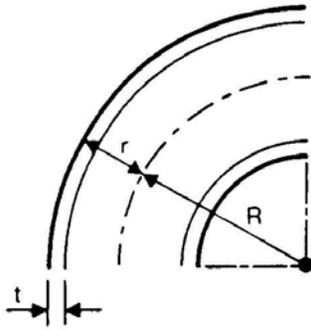
‡Subject to the limitations of Clause 5.4.2.2(d).

§Also includes single-mitre joint.

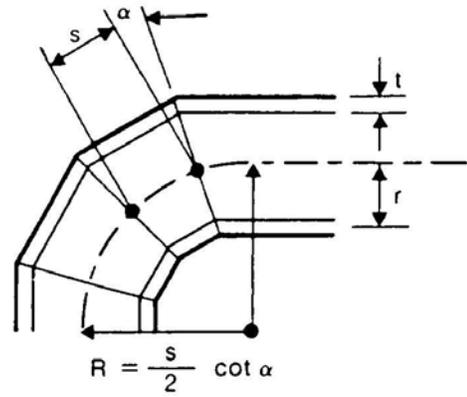
**Factors shown apply to bending. Flexibility factor for torsion equals 0.9.

Note: Designers are cautioned that specific cases may require more comprehensive analysis than that specified in Clause 5.6.

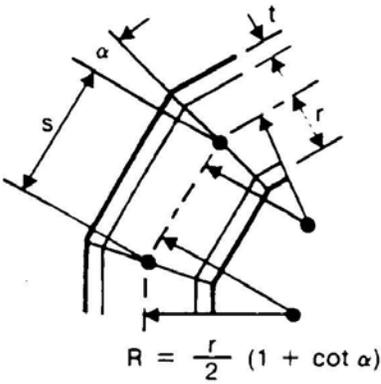
Table 5.5 (Concluded)



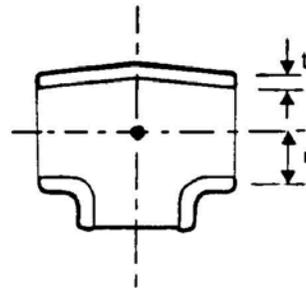
Sketch A



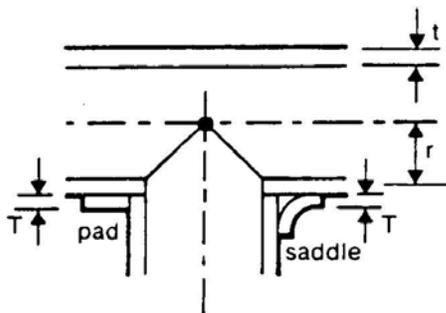
Sketch B



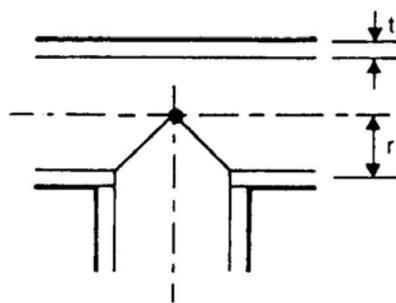
Sketch C



Sketch D



Sketch E



Sketch F

Allowable Pressure

The allowable pressure is calculated from 5.4.2.1.

$$P = 2StFLJT/D$$

where

P = allowable pressure

S = specified minimum yield strength

t = minimum wall thickness

= nominal wall thickness \times (1 - mill tolerance/100) - corrosion allowance

F = design factor = 0.8 for steel pipe (from 5.2.3)

L = location factor (from Table 5.1)

J = joint factor (from Table 5.2), input as material property

T = temperature derating factor for steel pipe (from Table 5.3)

D = outside diameter

Sustained Stress

The stress (S_L) due to sustained loads (pressure, weight and other sustained mechanical loads) and wind loads, is calculated from (5.19.3.5)

$$S_L = \frac{PD}{4t} + i \frac{M}{Z} \leq SFLJT$$

where

P = maximum pressure

D = outside diameter

t = minimum wall thickness

i = stress intensification factor

M = resultant bending moment due to sustained loads

Z = corroded section modulus

S = specified minimum yield strength

F = design factor

L = location factor

J = joint factor

T = temperature derating factor

Occasional Stress

The stress (S_{LO}) is calculated as the sum of stress due to sustained and wind loads (S_L) and stress due to occasional loads (S_O), such as earthquake. The allowable stress for occasional stress is not given in the code, therefore it is conservatively taken the same as the sustained stress allowable.

$$S_{LO} = S_L + i \frac{M}{Z} \leq SFLJT$$

where

M = resultant bending moment due to occasional loads

Expansion Stress

The stress (S_E) due to thermal expansion is calculated from (5.6.3.3).

$$S_E = \sqrt{S_b^2 + 4S_t^2} \leq 0.72ST$$

where

S_b = resultant bending stress = iM_b/Z

S_t = torsional stress = $M_t/2Z$

M_b = resultant bending moment due to expansion loads

M_t = torsional moment due to expansion loads

Z = uncorroded section modulus

Table 5.8
Flexibility and Stress Intensification Factors

Description	Flex- ibility Factor <i>k</i>	Stress Intens. Factor <i>i</i>	Description	Flex- ibility Factor <i>k</i> *	Stress Intens. Factor <i>i</i> *	Flexibility Character- istic <i>h</i>	See sketch
Buttwelded joint, reducer, or welding neck flange	1	1.0	Welding elbow, or pipe bend†	$\frac{1.65}{h}$	$\frac{0.9}{h^{2/3}}$	$\frac{1R}{r^2}$	A
Double-welded slip-on or socket welding flange	1	1.2	Mitre bend with close spacing†† $s < r(1 + \tan \alpha)$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{\cot \alpha}{2} \cdot \frac{ts}{r^2}$	B
Fillet welded joint, or single-welded socket welding flange	1	1.3	Mitre bend with wide spacing †§ $s \geq r(1 + \tan \alpha)$	$\frac{1.52}{h^{5/6}}$	$\frac{0.9}{h^{2/3}}$	$\frac{1 + \cot \alpha}{2} \cdot \frac{t}{r}$	C
Lap joint flange (with ANSI B16.9 lap joint stub)	1	1.6	Welding tee per CSA Standard CAN/CSA-Z245.11	1	$\frac{0.9}{h^{2/3}}$	$4.4 \frac{t}{r}$	D
Screwed pipe joint, or screwed flange	1	2.3	Reinforced fabricated tee, with pad or saddle	1	$\frac{0.9}{h^{2/3}}$	$\frac{(t + 1/2 T)^{5/2}}{t^{3/2} r}$	E
Corrugated pipe, straight or curved, or creased bend**	5	2.5	Unreinforced fabricated tee	1	$\frac{0.9}{h^{2/3}}$	$\frac{t}{r}$	F

*The flexibility factors and stress intensification factors apply to fittings of the same nominal wall thickness as the pipe used in the pipeline system, and shall in no case be taken as less than unity. They apply over the effective arc length (shown by dash-dot lines in the sketches) for curved and mitre elbows, and to the intersection point for tees.

† Where flanges are attached to one or both ends, the values of *k* and *i* in the Table shall be multiplied by the following factors:

One end flanged: $(h)^{1/6}$

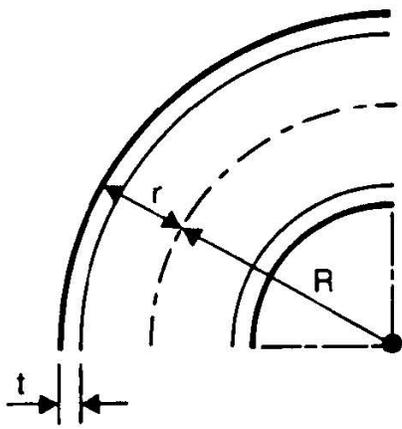
Both ends flanged: $(h)^{1/3}$

‡ Subject to the limitations of Clause 7.2.3(e).

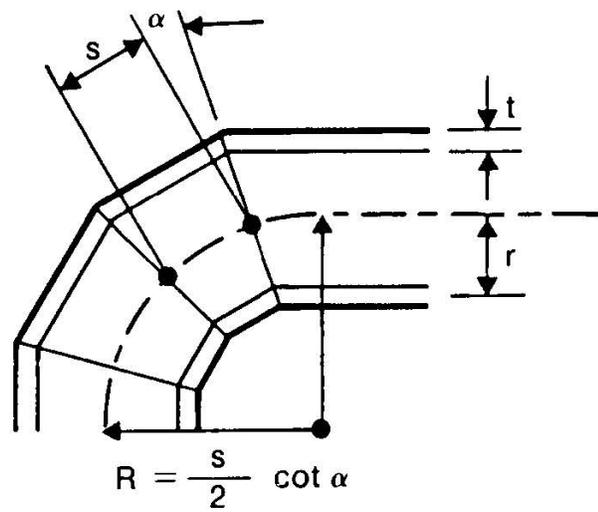
§ Also includes single-mitre joint.

** Factors shown apply to bending. Flexibility factor for torsion equals 0.9.

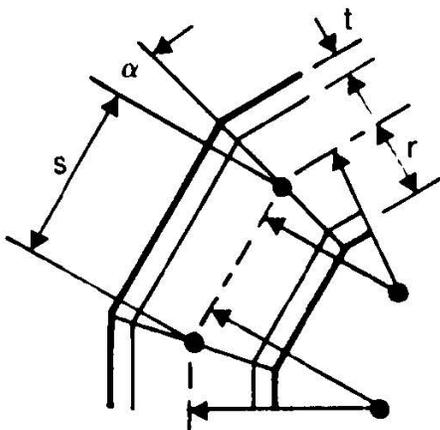
Note: Designers are cautioned that specific cases may require more comprehensive analysis than that specified in Clause 5.19.



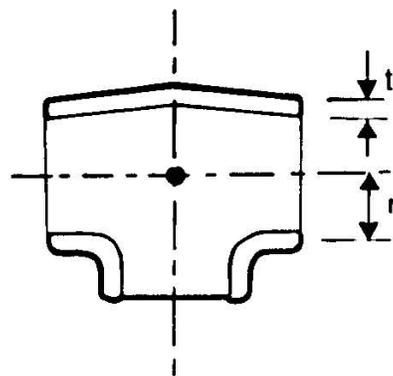
Sketch A



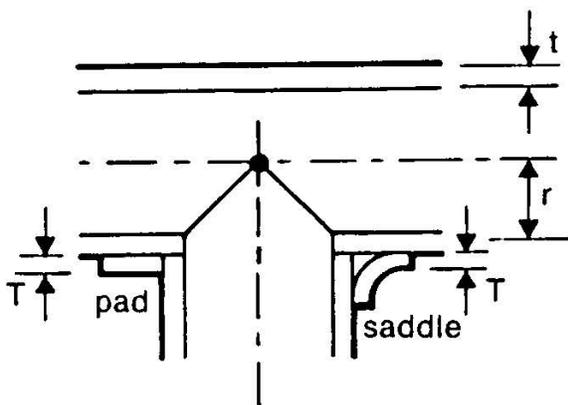
Sketch B



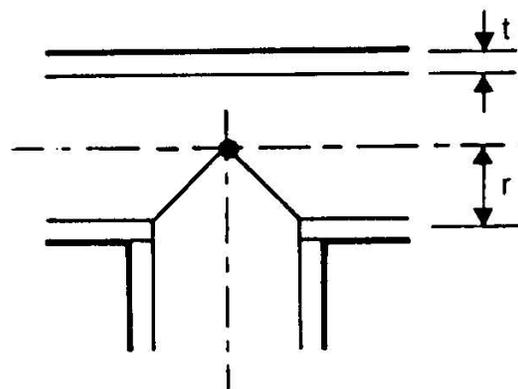
Sketch C



Sketch D



Sketch E



Sketch F

FRP Stress

Fiber Reinforced Plastic Pipe Stress

$$\text{Hoop stress: } S_H = \frac{PD}{2t_m}$$

$$\text{Longitudinal stress: } S_L = \frac{PD}{4t_m} + \frac{F}{A} + \frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z}$$

$$\text{Torsional stress: } S_T = \frac{M_t}{2Z}$$

$$\text{Shear stress: } S_{SH} = \sqrt{\left(\frac{S_L - S_H}{2}\right)^2 + \left(\frac{M_t}{2Z}\right)^2}$$

$$\text{Maximum principal stress: } S_P = \frac{S_L + S_H}{2} + S_{SH}$$

$$\text{Maximum shear stress: } S_S = \max\left(\frac{S_P}{2}, S_{SH}\right)$$

$$\text{Octahedral shear stress: } S_O = \frac{\sqrt{2}}{3} \sqrt{\left(\frac{S_L + S_H}{2}\right)^2 + 3(S_{SH})^2}$$

where

P = pressure

D = outside diameter

t_m = minimum thickness

= nominal thickness x (1 – mill tolerance/100) – corrosion allowance

i_i = in-plane stress intensification factor

i_o = out of plane stress intensification factor

M_i = in plane bending moment

M_o = out of plane bending moment

M_t = torque

Z = section modulus

F = axial force

A = cross-section area

Operating Stress for NDE

This function is available to users who wish to inspect or examine high stress areas (Non-Destructive Evaluation, NDE) of piping that have high “operating stresses.”

*While B31.1 and B31.3 piping codes do not consider the operating condition (W+P+T) as a required load case for code compliance, you may, however, evaluate this load case the same way as you would the other code compliance cases such as Sustained (W+P) and Expansion (T). CAEPIPE will show columns displaying Operating Stress (S_{opr}) and Allowable stress (S_{all}) alongside the code required stresses even though the *Operating Stress is not a code requirement.**

To display these columns, you will have to turn them on first: (mouse right click) > Show Operating Stress for NDE.

#	Sustained				Expansion			
	Node	SL (psi)	SH (psi)	SL SH	Node	SE (psi)	SA (psi)	SE SA
1	80	2538	17900	0.14	30	52765	29475	1.79
2	60	2205	17900	0.12	50	50365	29475	1.
3	70	2133	17900	0.12	20A	47368	29475	1.
4	30	2033	17900	0.11	20B	33541	29475	1.
5	10	1448	17900	0.08	10	32123	29475	1.
6	40B	1053	17900	0.06	80	27166	29475	0.
7	20B	980	17900	0.05	40A	18838	29475	0.
8	20A	937	17900	0.05	60	17572	29475	0.
9	50	924	17900	0.05	70	12275	29475	0.
10	40A	902	17900	0.05	40B	10284	29475	0.35

Operating Stress for NDE

#	Sustained				Expansion				Operating Stress for NDE			
	Node	SL (psi)	SH (psi)	SL/SH	Node	SE (psi)	SA (psi)	SE/SA	Node	Sopr (psi)	Sall (psi)	Sopr/Sall
1	80	2538	17900	0.14	30	52765	29475	1.79	30	51580	47375	1.09
2	60	2205	17900	0.12	50	50365	29475	1.71	50	50571	47375	1.07
3	70	2133	17900	0.12	20A	47368	29475	1.61	20A	43725	47375	0.92
4	30	2033	17900	0.11	20B	33541	29475	1.14	20B	32188	47375	0.68
5	10	1448	17900	0.08	10	32123	29475	1.09	10	28281	47375	0.60
6	40B	1053	17900	0.06	80	27166	29475	0.92	40A	17801	47375	0.38
7	20B	980	17900	0.05	40A	18838	29475	0.64	80	10496	47375	0.22
8	20A	937	17900	0.05	60	17572	29475	0.60	40B	10383	47375	0.22
9	50	924	17900	0.05	70	12275	29475	0.42	70	3072	47375	0.06
10	40A	902	17900	0.05	40B	10284	29475	0.35	60	1509	47375	0.03

This allowable stress is NOT specified nor recommended by B31.1/B31.3 piping codes but, as mentioned above, may aid you in identifying regions of high combined stress in your piping system, which may warrant a closer examination in your engineering judgment.

The equation used to calculate the operating stresses is modeled along the lines of a piping code specified stress equation. The three components of the equation are:

1. Axial stress combined with the longitudinal pressure stress,
2. Bending stress from in-plane and out-of-plane bending moments, and
3. A torsional stress from the torsion term.

The allowable stress for this case is simply a combination of the sustained and the thermal allowable stresses.

Note that the Operating Stress is calculated and shown only for the first Operating case (W+P1+T1). Details follow.

Operating Stress for NDE

The stress (S_{opr}) due to operating loads (pressure, weight and thermal load T_1) is calculated as

$$S_{opr} = S_a + \sqrt{(S_a)^2 + (2S_t)^2} \leq S_{all}$$

where

$$S_a = \left[\frac{PD}{4t} + \frac{F}{A} \right]_{Operating\ 1}$$

$$S_b = \left[\frac{\sqrt{(i_i M_i)^2 + (i_o M_o)^2}}{Z} \right]_{Operating\ 1}$$

$$S_t = \left[\frac{M_t}{2Z} \right]_{Operating\ 1}$$

P = maximum of CAEPIPE input pressures P1, P2 and P3

D = outside diameter

t = nominal wall thickness

A = un-corroded cross-sectional area of the pipe

F = longitudinal force

i_i = in-plane stress intensification factor according to analysis code selected in CAEPIPE

i_o = out-of-plane stress intensification factor according to analysis code selected in CAEPIPE

Note: If the analysis code selected provides only the stress intensification i , then $i_i = i_o = i$.

M_i = in-plane bending moment

M_o = factor for loose type flanges (from Fig. 2-7.5)

M_t = factor for loose type flanges (from Fig. 2-7.4)

Z = un-corroded section modulus; for reduced outlets / branch connections, effective section modulus

$S_{all} = f(1.25S_{cold} + 0.25S_{hot}) + S_{hot}$

f = stress range reduction factor = $6/N^{0.2}$

N = Number of equivalent full-range thermal cycles

S_{cold} = basic allowable stress at T_{ref}

S_{hot} = basic allowable stress at CAEPIPE input temperature T_1

Thickness and Section Modulus used in Weight, Pressure and Stress Calculations for ANSI B31.x Codes

Particulars	Allowable Pressure	Pipe Weight	Sustained Stress	Expansion Stress	Occasional Stress
B31.1 (2012)					
Pipe Thickness used	Nominal Thk. x (1-mill tolerance/100) – Corrosion allowance	Nominal Thickness	Nominal Thickness	-	Nominal Thickness
Section Modulus used	-	-	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus
B31.3 (2012)					
Pipe Thickness used	Nominal Thk. x (1-mill tolerance/100) – Corrosion allowance	Nominal Thickness	Nominal Thickness - Corrosion allowance	-	Nominal Thickness – Corrosion allowance
Section Modulus used	-	-	<i>Corroded</i> Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus	<i>Corroded</i> Section Modulus; For Branch, effective section modulus
B31.4 (2012)					
Pipe Thickness used	Nominal Thk. x (1-mill tolerance/100) – Corrosion allowance	Nominal Thickness	Nominal Thickness	-	Nominal Thickness
Section Modulus used	-	-	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch effective section modulus
B31.5 (2013)					
Pipe Thickness used	Nominal Thk. x (1-mill tolerance/100) – Corrosion allowance	Nominal Thickness	Nominal Thickness – Corrosion allowance	-	Nominal Thickness – Corrosion allowance

Thickness and Section Modulus used in Weight, Pressure and Stress Calculations for ANSI B31.x Codes

Particulars	Allowable Pressure	Pipe Weight	Sustained Stress	Expansion Stress	Occasional Stress
Section Modulus used	-	-	Corroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus	Corroded Section Modulus; For Branch, effective section modulus
B31.8 (2012)					
Pipe Thickness used	Nominal Thk.	Nominal Thickness	Nominal Thickness	-	Nominal Thickness
Section Modulus used	-	-	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus
B31.9 (2008)					
Pipe Thickness used	Nominal Thk. x (1-mill tolerance/100) – Corrosion allowance	Nominal Thickness	Nominal Thickness	-	Nominal Thickness
Section Modulus used	-	-	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus	Uncorroded Section Modulus; For Branch, effective section modulus

Note:

1. Corrosion allowance includes thickness required for threading, grooving, erosion, corrosion etc.
2. Uncorroded section modulus = section modulus calculated using the nominal thickness.
3. Corroded section modulus = section modulus calculated using the “corroded thickness”
corroded thickness = nominal thickness – corrosion allowance
4. Effective section modulus = section modulus calculated using effective branch thickness, which is lesser of $i_1 t_b$ or t_h
where, t_b = branch nominal thickness, t_h = header nominal thickness, i_1 = in-plane SIF at branch

Appendix B

Rotating Equipment Reports

API Standards 610 / ISO 13709

API 610 (Tenth Edition, 2004) / ISO 13709 for Pumps

The allowable nozzle forces and moments for pumps are taken from Table 4 of the tenth edition of API Standard 610 / ISO 13709.

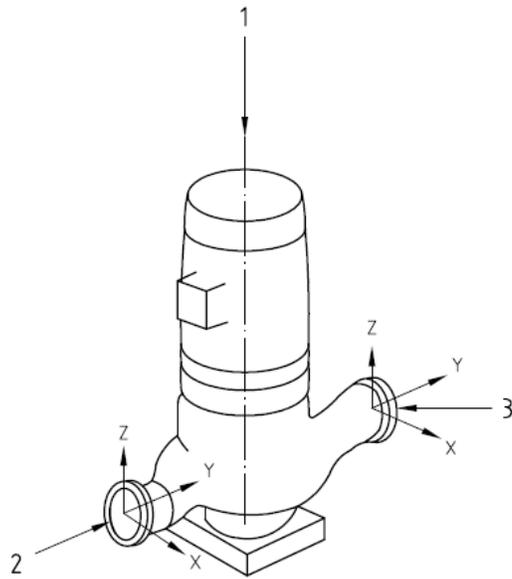
Table 4 — Nozzle loadings

	SI units								
	Nominal size of flange (DN)								
	≤ 50	80	100	150	200	250	300	350	400
	Forces (N)								
Each top nozzle									
F_X	710	1 070	1 420	2 490	3 780	5 340	6 670	7 120	8 450
F_Y	580	890	1 160	2 050	3 110	4 450	5 340	5 780	6 670
F_Z	890	1 330	1 780	3 110	4 890	6 670	8 000	8 900	10 230
F_R	1 280	1 930	2 560	4 480	6 920	9 630	11 700	12 780	14 850
Each side nozzle									
F_X	710	1 070	1 420	2 490	3 780	5 340	6 670	7 120	8 450
F_Y	890	1 330	1 780	3 110	4 890	6 670	8 000	8 900	10 230
F_Z	580	890	1 160	2 050	3 110	4 450	5 340	5 780	6 670
F_R	1 280	1 930	2 560	4 480	6 920	9 630	11 700	12 780	14 850
Each end nozzle									
F_X	890	1 330	1 780	3 110	4 890	6 670	8 000	8 900	10 230
F_Y	710	1 070	1 420	2 490	3 780	5 340	6 670	7 120	8 450
F_Z	580	890	1 160	2 050	3 110	4 450	5 340	5 780	6 670
F_R	1 280	1 930	2 560	4 480	6 920	9 630	11 700	12 780	14 850
	Moments (N·m)								
Each nozzle									
M_X	460	950	1 330	2 300	3 530	5 020	6 100	6 370	7 320
M_Y	230	470	680	1 180	1 760	2 440	2 980	3 120	3 660
M_Z	350	720	1 000	1 760	2 580	3 800	4 610	4 750	5 420
M_R	620	1 280	1 800	3 130	4 710	6 750	8 210	8 540	9 820

Table 4 — Nozzle loadings (continued)

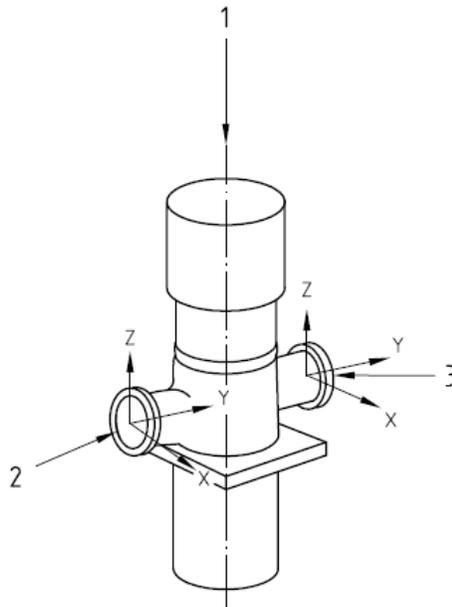
	US Customary units								
	Nominal size of flange (NPS)								
	≤ 2	3	4	6	8	10	12	14	16
Forces (lbf)									
Each top nozzle									
F_X	160	240	320	560	850	1 200	1 500	1 600	1 900
F_Y	130	200	260	460	700	1 000	1 200	1 300	1 500
F_Z	200	300	400	700	1 100	1 500	1 800	2 000	2 300
F_R	290	430	570	1 010	1 560	2 200	2 600	2 900	3 300
Each side nozzle									
F_X	160	240	320	560	850	1 200	1 500	1 600	1 900
F_Y	200	300	400	700	1 100	1 500	1 800	2 000	2 300
F_Z	130	200	260	460	700	1 000	1 200	1 300	1 500
F_R	290	430	570	1 010	1 560	2 200	2 600	2 900	3 300
Each end nozzle									
F_X	200	300	400	700	1 100	1 500	1 800	2 000	2 300
F_Y	160	240	320	560	850	1 200	1 500	1 600	1 900
F_Z	130	200	260	460	700	1 000	1 200	1 300	1 500
F_R	290	430	570	1 010	1 560	2 200	2 600	2 900	3 300
Moments (ft-lbf)									
Each nozzle									
M_X	340	700	980	1 700	2 600	3 700	4 500	4 700	5 400
M_Y	170	350	500	870	1 300	1 800	2 200	2 300	2 700
M_Z	260	530	740	1 300	1 900	2 800	3 400	3 500	4 000
M_R	460	950	1 330	2 310	3 500	5 000	6 100	6 300	7 200
NOTE 1 See Figures 20 through 24 for orientation of nozzle loads (X, Y and Z).									
NOTE 2 Each value shown above indicates range from minus that value to plus that value; for example 160 indicates a range from -160 to +160.									

The coordinate systems and nozzle orientations for various pump configurations are shown next.



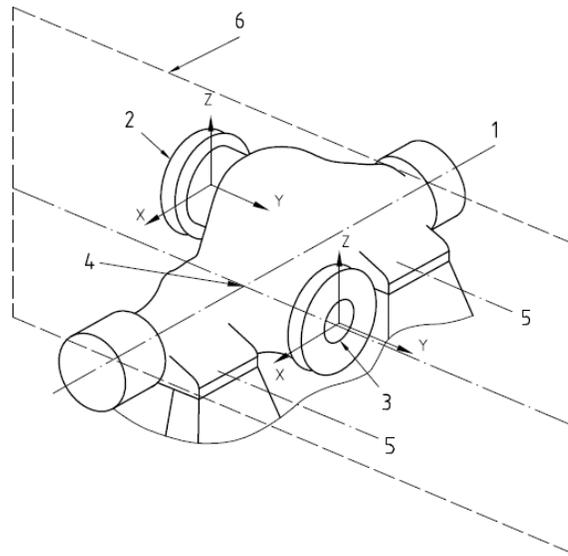
- Key**
- 1 shaft centreline
 - 2 discharge
 - 3 suction

Figure 20 — Coordinate system for the forces and moments in Table 4 — Vertical in-line pumps



- Key**
- 1 shaft centreline
 - 2 discharge
 - 3 suction

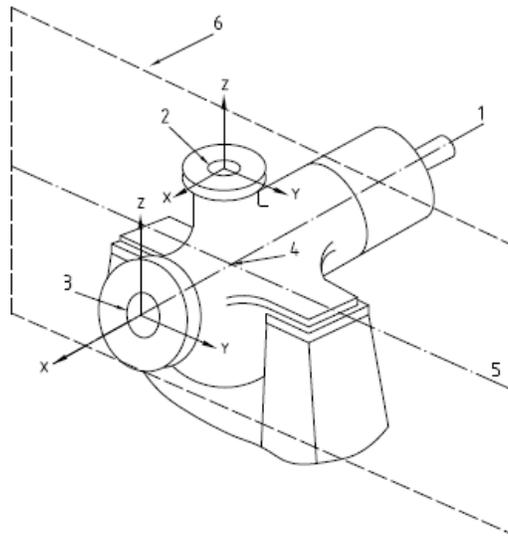
Figure 21 — Coordinate system for the forces and moments in Table 4 — Vertically suspended double-casing pumps



Key

- 1 shaft centreline
- 2 discharge nozzle
- 3 suction nozzle
- 4 centre of pump
- 5 pedestal centreline
- 6 vertical plane

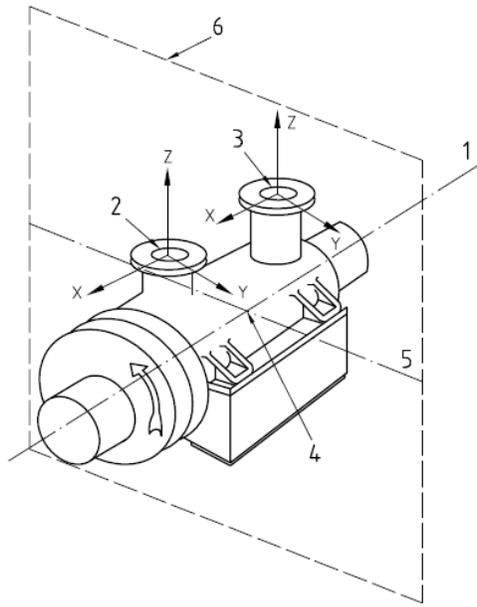
Figure 22 — Coordinate system for the forces and moments in Table 4 — Horizontal pumps with side suction and side discharge nozzles



Key

- 1 shaft centreline
- 2 discharge nozzle
- 3 suction nozzle
- 4 centre of pump
- 5 pedestal centreline
- 6 vertical plane

Figure 23 — Coordinate system for the forces and moments in Table 4 — Horizontal pumps with end suction and top discharge nozzles



Key

- 1 shaft centreline
- 2 discharge nozzle
- 3 suction nozzle
- 4 centre of pump
- 5 pedestal centreline
- 6 vertical plane

Figure 24 — Coordinate system for the forces and moments in Table 4 — Horizontal pumps with top nozzles

Criteria for Piping Design

The criteria for piping design are taken from Appendix F of the API 610.

F.1 Horizontal pumps

F.1.1 Acceptable piping configurations should not cause excessive misalignment between the pump and driver. Piping configurations that produce component nozzle loads lying within the ranges specified in Table 4 limit casing distortion to one-half the pump vendor's design criterion (see 5.3.3) and ensure pump shaft displacement of less than 250 μm (0,010 in).

F.1.2 Piping configurations that produce loads outside the ranges specified in Table 4 are also acceptable without consultation with the pump vendor if the conditions specified in F.1.2 a) through F.1.2 c) below are satisfied. Satisfying these conditions ensures that any pump casing distortion will be within the vendor's design criteria (see 5.3.3) and that the displacement of the pump shaft will be less than 380 μm (0,015 in).

- a) The individual component forces and moments acting on each pump nozzle flange shall not exceed the range specified in Table 4 (T4) by a factor of more than 2.
- b) The resultant applied force (F_{RSA} , F_{RDA}) and the resultant applied moment (M_{RSA} , M_{RDA}) acting on each pump nozzle flange shall satisfy the appropriate interaction equations below.

$$[F_{RSA} / (1,5 \times F_{RST4})] + [M_{RSA} / (1,5 \times M_{RST4})] \leq 2 \quad (\text{F.1})$$

$$[F_{RDA} / (1,5 \times F_{RDT4})] + [M_{RDA} / (1,5 \times M_{RDT4})] \leq 2 \quad (\text{F.2})$$

- c) The applied component forces and moments acting on each pump nozzle flange shall be translated to the centre of the pump. The magnitude of the resultant applied force (F_{RCA}), the resultant applied moment (M_{RCA}), and the applied moment shall be limited by Equation (F.3), Equation (F.4) and Equation (F.5) (the sign convention shown in Figure 20 through Figure 24 and the right-hand rule should be used in evaluating these equations).

$$F_{RCA} < 1,5 (F_{RST4} + F_{RDT4}) \quad (\text{F.3})$$

$$|M_{YCA}| < 2,0 (M_{YST4} + M_{YDT4}) \quad (\text{F.4})$$

$$M_{RCA} < 1,5 (M_{RST4} + M_{RDT4}) \quad (\text{F.5})$$

where

$$F_{RCA} = [(F_{XCA})^2 + (F_{YCA})^2 + (F_{ZCA})^2]^{0,5}$$

where

$$F_{XCA} = F_{XSA} + F_{XDA}$$

$$F_{YCA} = F_{YSA} + F_{YDA}$$

$$F_{ZCA} = F_{ZSA} + F_{ZDA}$$

$$M_{RCA} = [(M_{XCA})^2 + (M_{YCA})^2 + (M_{ZCA})^2]^{0,5}$$

where

$$M_{XCA} = M_{XSA} + M_{XDA} - [(F_{YSA})(zS) + (F_{YDA})(zD) - (F_{ZSA})(yS) - (F_{ZDA})(yD)]/1\ 000$$

$$M_{YCA} = M_{YSA} + M_{YDA} + [(F_{XSA})(zS) + (F_{XDA})(zD) - (F_{ZSA})(xS) - (F_{ZDA})(xD)]/1\ 000$$

$$M_{ZCA} = M_{ZSA} + M_{ZDA} - [(F_{XSA})(yS) + (F_{XDA})(yD) - (F_{YSA})(xS) - (F_{YDA})(xD)]/1\ 000$$

In USC units, the constant 1 000 shall be changed to 12. This constant is the conversion factor to change millimetres to metres or inches to feet.

F.1.3 Piping configurations that produce loads greater than those allowed in F.1.2 shall be approved by the purchaser and the vendor.

F.2 Vertical in-line pumps

Vertical in-line pumps that are supported only by the attached piping may be subjected to component piping loads that are more than double the values shown in Table 4 if these loads do not cause a principal stress greater than 41 N/mm² (5 950 psi) in either nozzle. For calculation purposes, the section properties of the pump nozzles shall be based on Schedule 40 pipe whose nominal size is equal to that of the appropriate pump nozzle. Equation (F.6), Equation (F.7), and Equation (F.8) can be used to evaluate principal stress, longitudinal stress and shear stress, respectively, in the nozzles.

For SI units, the following equations apply:

$$\sigma_p = (\sigma/2) + (\sigma^2/4 + \tau^2)^{0.5} < 41 \quad (F.6)$$

$$\sigma_l = [1,27 \times F_Y / (D_o^2 - D_i^2)] + [10\,200 \times D_o (M_X^2 + M_Z^2)^{0.5}] / (D_o^4 - D_i^4) \quad (F.7)$$

$$\tau = [1,27 \times (F_X^2 + F_Z^2)^{0.5}] / (D_o^2 - D_i^2) + [5\,100 \times D_o (|M_Y|)] / (D_o^4 - D_i^4) \quad (F.8)$$

For USC units, the following equations apply:

$$\sigma_p = (\sigma/2) + (\sigma^2/4 + \tau^2)^{0.5} < 5\,950 \quad (F.9)$$

$$\sigma_l = [1,27 \times F_Y / (D_o^2 - D_i^2)] + [122 \times D_o (M_X^2 + M_Z^2)^{0.5}] / (D_o^4 - D_i^4) \quad (F.10)$$

$$\tau = [1,27 \times (F_X^2 + F_Z^2)^{0.5}] / (D_o^2 - D_i^2) + [61 \times D_o (|M_Y|)] / (D_o^4 - D_i^4) \quad (F.11)$$

where

σ_p is the principal stress, expressed in MPa (lbf/in²);

σ_l is the longitudinal stress, expressed in MPa (lbf/in²);

τ is the shear stress, expressed in MPa (lbf/in²);

F_X is the applied force on the X axis;

F_Y is the applied force on the Y axis;

F_Z is the applied force on the Z axis;

M_X is the applied moment on the X axis;

M_Y is the applied moment on the Y axis;

M_Z is the applied moment on the Z axis;

D_i, D_o are the inner and outer diameters of the nozzles, expressed in millimetres (inches).

$F_X, F_Y, F_Z, M_X, M_Y,$ and M_Z represent the applied loads acting on the suction or discharge nozzles; thus, subscripts S_A and D_A have been omitted to simplify the equations. The sign of F_Y is positive if the load puts the nozzle in tension; the sign is negative if the load puts the nozzle in compression. One should refer to Figure 20 and the applied nozzle loads to determine whether the nozzle is in tension or compression. The absolute value of M_Y should be used in Equations (F.8) to (F.11).

F.3 Nomenclature

The following definitions apply to the sample problems in F.4

where

- C is the centre of the pump. For pump types OH2 and BB2 with two support pedestals, the centre is defined by the intersection of the pump shaft centreline and a vertical plane passing through the centre of the two pedestals (see Figure 23 and Figure 24). For pump types BB1, BB3, BB4 and BB5 with four support pedestals, the centre is defined by the intersection of the pump shaft centreline and a vertical plane passing midway between the four pedestals (see Figure 22);
- D is the discharge nozzle;
- D_i is the inside diameter of Schedule 40 pipe whose nominal size is equal to that of the pump nozzle in question, expressed in millimetres (inches);
- D_o is the outside diameter of Schedule 40 pipe whose nominal size is equal to that of the pump nozzle in question, expressed in millimetres (inches);
- F is the force, expressed in newtons (pounds force);
- F_R is the resultant force. (F_{RSA} and F_{RDA} are calculated by the square root of the sum of the squares method using the applied component forces acting on the nozzle flange. F_{RST4} and F_{RDT4} are extracted from Table 4, using the appropriate nozzle size);
- M is the moment, expressed in newton metres (foot-pounds force);
- M_R is the resultant moment. (M_{RSA} and M_{RDA} are calculated by the square root of the squares method using the applied component moments acting on the nozzle flange. M_{RST4} and M_{RDT4} are extracted from Table 4 using the appropriate nozzle size);
- σ_p is the principal stress, expressed in megapascals (pounds force per square inch);
- σ_l is the longitudinal stress, expressed in newtons per square millimetre (pounds per square inch);
- τ is the shear stress, expressed in newtons per square millimetre (pounds per square inch);
- S is the suction nozzle;
- x, y, z are the location coordinates of the nozzle flanges with respect to the centre of the pump, expressed in millimetres (inches);
- X, Y, Z are the directions of the load (see Figures 20 to 24);
- Subscript A is an applied load;
- Subscript T4 is a load extracted from Table 4.

NEMA SM-23 (1991) for Turbines

There are two types of allowables.

1. Allowables for each nozzle and
2. Combined allowables for each turbine.

1. Allowables for each Nozzle

The resultant force and the resultant moment at any connection must not exceed

$$3F_R + M_R \leq 500D_e$$

where

F_R = resultant force at the nozzle (lb)

M_R = resultant moment at the nozzle (ft.-lb)

D_e = nominal pipe size (inches) of the connection up to 8" in diameter
 = $(16 + D_{nom})/3$ If the size is greater than 8"

2. Combined Allowables for the Turbine

The combined resultants of the forces and moments at the inlet, exhaust and extraction nozzles resolved at the centerline of the exhaust nozzle must not exceed the following two conditions:

- (a) These resultant should not exceed:

$$2F_C + M_C \leq 250D_C$$

where

F_C = combined resultant of inlet, exhaust and extraction forces (lb)

M_C = combined resultant of inlet, exhaust and extraction moments (ft.-lb.)

D_C = diameter (in inches) of a circular opening equal to the total areas of the inlet, exhaust and extraction nozzles up to a value of 9" in diameter
 = $(18 + \text{Equivalent diameter}) / 3$ (in.) For values > 9"

- (b) The components of these resultants shall not exceed:

$$\begin{aligned} F_X &= 50D_C M_X = 250D_C \\ F_Y &= 125D_C M_Y = 125D_C \\ F_Z &= 100D_C M_Z = 125D_C \end{aligned}$$

where

F_X = horizontal component of F_R parallel to the turbine shaft

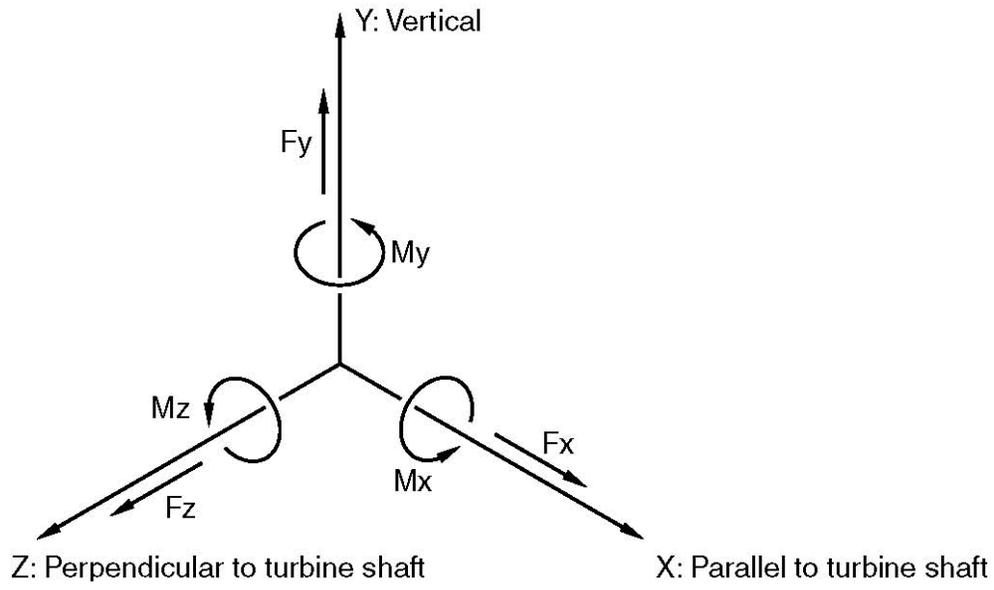
F_Y = vertical component of F_R

F_Z = horizontal component of F_R at right angles to the turbine shaft

M_X = horizontal component of M_R parallel to the turbine shaft

M_Y = vertical component of M_R

M_Z = horizontal component of M_R at right angles to the turbine shaft



API 617 (Seventh edition, June 2003) for Compressors

1. Allowables for each Nozzle

The total resultant force and resultant moment imposed on the compressor at any connection should not exceed

$$3F_r + M_r \leq 927D_e$$

Where

F_r = resultant force at the Nozzle (lb)

M_r = resultant moment at the Nozzle (ft.-lb)

D_e = nominal pipe size (inches) of the connection up to 8" in diameter

= $(16 + D_{nom})/3$ If the size is greater than 8"

2. Combined Allowables for Compressors

The combined resultants of the forces and moments of the inlet, sidestream, and discharge connections resolved at the centerlines of the largest connection should not exceed the following two conditions:

(a) The resultant should not exceed:

$$2F_c + M_c \leq 462D_c$$

Where

F_c = combined resultant of inlet, sidestream, and discharge forces (lb)

M_c = combined resultant of inlet, sidestream, and discharge moments, and moments resulting from forces (ft.-lb)

D_c = diameter of one circular opening equal to the total areas of the inlet, sidestream, and discharge openings. If the equivalent nozzle diameter is greater than 9", use a value of D_c equal to $(18 + \text{Equivalent Diameter}) / 3$

(b) The components of these resultants shall not exceed:

$$F_x = 92D_c \quad M_x = 462D_c$$

$$F_y = 231D_c \quad M_y = 231D_c$$

$$F_z = 185D_c \quad M_z = 231D_c$$

Where

F_x = horizontal component of F_c parallel to the compressor shaft (lb)

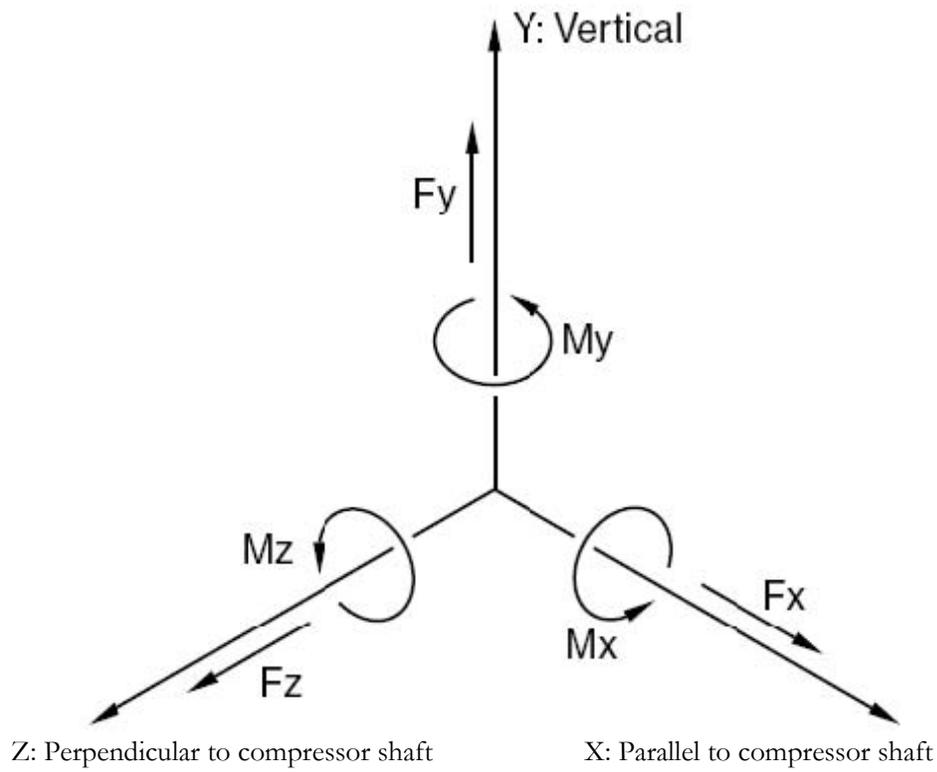
F_y = vertical component of F_c (lb)

F_z = horizontal component of F_c at right angles to be compressor shaft (lb)

M_x = component of M_c around the horizontal axis (ft.-lb)

M_y = component of M_c around the vertical axis (ft.-lb)

M_z = component of M_c around the horizontal axis at right angles to the compressor (ft.-lb)



This page is blank

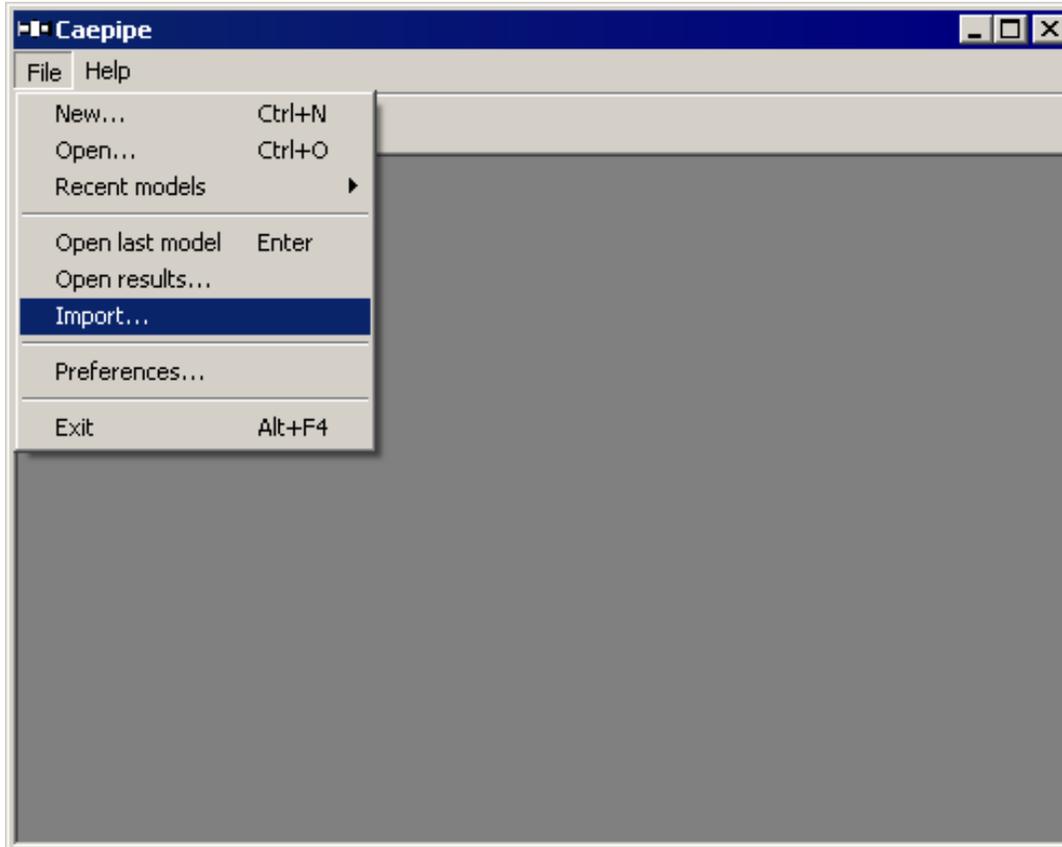
Appendix C

Import / Export

Import

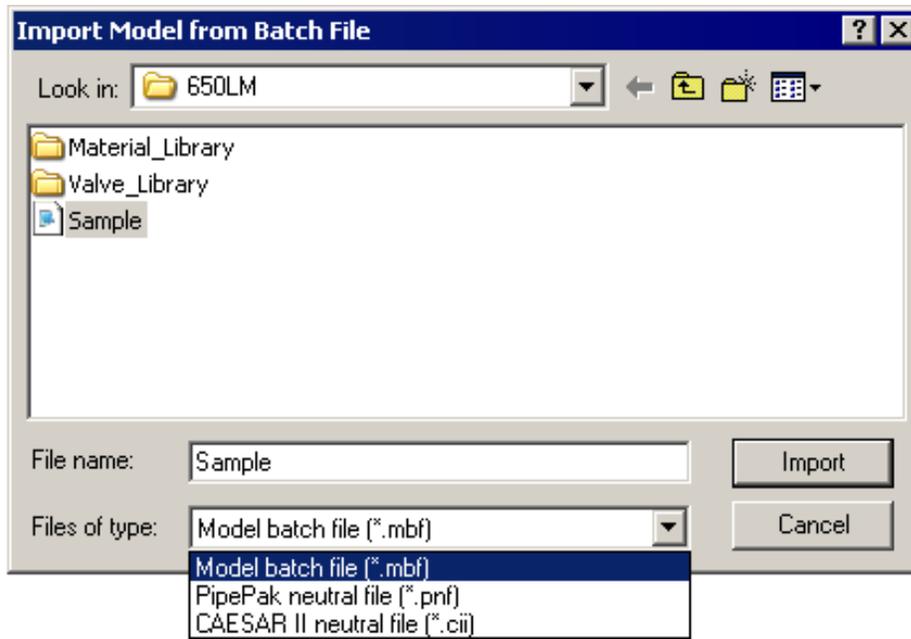
CAEPIPE can import model data from a text file, which may be created using a text editor and should have the extension: .mbf (model batch file). This text file may also be created for an existing model using the Export command from the Layout window.

To import a model, select the menu command File > Import from the Main window.



The Import Model dialog is shown.

Import



Import

Select the model batch file (.mbf) and then click on the Import button. The batch file will be read and the corresponding model file (.mod) will be created and then shown in the Layout window which can be further modified or analyzed.

For detailed information on the .pnf and .cii file import capabilities please contact us at support@sstusa.com.

Command line operation

Dragging and dropping an .mbf file on CAEPIPE.exe will import, analyze and produce the results in .csv format and close the program. On the other hand, dragging and dropping the .mod file on CAEPIPE.exe will open the .mod file. Using the command line, you may have CAEPIPE analyze and produce a CSV results file thus:

```
C:\CAEPIPE\681> caepipe.exe sample.mbf (Enter)
```

Format of .MBF

The above command will produce a “sample.csv” file in the same folder as the .mbf file.

The input data is given in the following order. Except for the heading the start of each section is indicated by a keyword. The data for that section follow. Only the first three characters of the keyword are significant. See an example MBF at the end of this appendix.

<u>Keyword</u>	<u>Data</u>
	Heading
OPTIONS	Program options
MATERIAL	Material properties
SOILS	Soil properties
PIPE	Section properties
LOADS	Loads data
BMATERIALS	Beam Materials
BSECTIONS	Beam section properties
BLOADS	Beam loads
LAYOUT	Layout data
PUMPS	Pump data
COMPRESSORS	Compressor data
TURBINES	Turbine data
SEISMIC	Seismic load
WIND	Wind load

Heading

Any number of heading lines may be given. Only the first will be used as Title in the CAEPIPE model.

OPTIONS

Hanger design

HGRA	Grinnell (Default)
HGRB	Bergen-Paterson
HGRC	Fee and Mason
HGRD	Basic Engineers

Import

HGRE Power Piping
HGRF Nordon
HGRG Carpenter and Paterson
HGRH NPS
HGRI PTP
HGRJ Corner and Lada
HGRK Elcen

Piping Code

B311	ANSI B31.1
B311-67	USAS B31.1 (1967)
B313	ANSI B31.3
B314	ANSI B31.4
B315	ANSI B31.5
B318	ANSI B31.8
B319	ANSI B31.9
ASME	ASME Section III, Class 2 (1980)
ASME-86	ASME Section III, Class 2 (1986)
ASME-92	ASME Section III, Class 2 (1992)
BS806	British code
NORWEIGIAN-83	Norwegian code (1983)
NORWEIGIAN-90	Norwegian code (1990)
RCC-M	French code (1985)
SNCT	CODETI (1995)
SWEDISH	Swedish code (1978)
STOOMWEZEN	Dutch code (1989)
Z183	Z183 (1990)
Z184	Z183 (1992)
EUROPEAN	EN 13480 (2002)

Units

SI SI units (default is English units)
RAD Radians for angles (default is degrees)

Vertical Axis

Z Vertical axis is Z (Default is Y)

OPTIONS Example

```
OPTIONS  
HGRA , B311 , RAD
```

MATERIAL

Name is Material Name (up to 3 characters). The values in square brackets ([...]) are optional.

English units

First line: Name, Density (lb/in³), Poisson's ratio, [Long. joint factor], [circ. joint factor]

Import

Following lines: Name, Temp (F), E (psi), alfa (in/in/F), [allowable stress (psi)], [yield stress (psi)], [rupture stress (psi)]

.
.
.

SI units

First line: Name, Density (kg/m³), Poisson's ratio, [Long. joint factor], [circ.joint factor]

Following lines: Name, Temp (C), E (N/mm²), alfa (mm/mm/C), [allowable stress (N/mm²)], [yield stress (N/mm²)], [rupture stress (N/mm²)]

.
.
.

MATERIAL Example (English units)

MATERIAL

```
A53,0.283,0.300,1.00
A53,70,29.5E+6,6.07E-6,20000
A53,200,28.8E+6,6.38E-6,20000
A53,300,28.3E+6,6.60E-6,20000
A53,400,27.7E+6,6.82E-6,20000
A53,500,27.3E+6,7.02E-6,18900
ALU,0.098,0.330,1.00
ALU,70,10.0E+6,12.25E-6,10000
ALU,150,9.80E+6,12.67E-6,10000
ALU,200,9.60E+6,12.95E-6,9800
ALU,250,9.40E+6,13.12E-6,9000
ALU,300,9.20E+6,13.28E-6,6600
```

SOILS (Soil properties)

Soil properties are input as follows:

Note:

Name is Soil Name (up to 3 characters)

Type must be 0 or 1 (1 = Cohesive; 0 = Cohesive less)

If Type = 0, then Delta and Ks is required

If Type = 1, then Strength is required

English units

Name, Type (0 or 1), Density (lb/ft³), Ground level (ft'in"), [Strength (psi) /Delta (deg or rad)], [Ks]

SI units

Name, Type (0 or 1), Density (kg/m³), Ground level (mm), [Strength (N/mm²) / Delta (deg or rad)], [Ks]

PIPE (Section properties)

Section properties are input as follows for pipe sections:

Import

Note:

Name is Section Name (up to 5 characters).

OD must be actual OD and not Nominal pipe size (e.g., input 2.125 and not 2.0 for 2" NPS)

Thk must be an actual numerical value and not a schedule.

Soil name is Name of the soil (up to 3 characters). It should be defined in the soil section before use in this section.

English units

Name, OD (inch), Thk (inch), Corrosion allowance (inch), Mill tolerance (%), Insulation density (lb/ft³), Insulation Thk (inch)

SI units

Name, OD (mm), Thk (mm), Corrosion allowance (mm), Mill tolerance (%), Insulation density (kg/m³), Insulation Thk (mm), Lining density (kg/m³), Lining Thk (mm), Soil name

PIPE Example (English units)

PIPE

8, 8.625, 0.5, 0, 0.0, 11, 2

6, 6.625, 0.28, 0, 0.0, 11, 2

LOADS (Load data)

Load data are input as follows:

Note:

Name is Load Name (up to 3 characters).

T2, P2, T3 and P3 are Temperature 2, Pressure 2, Temperature 3 and Pressure 3 respectively.

English units

Name, Temperature 1 (F), Pressure 1 (psi), Specific Gravity, [T2,P2,T3,P3, Additional weight (lb/ft)]

SI units

Name, Temperature 1 (c), Pressure 1 (bar), Specific Gravity, [T2,P2,T3,P3, Additional weight (kg/m)]

BMATERIALS (Beam Materials)

Name is Material Name (up to 3 characters). The values in square brackets ([...]) are optional.

English units

Name, Young's Modulus (psi), [Poisson's ratio, Density (lb/in³), Alfa (in/in/F)]

SI units

Name, Young's Modulus (Mpa), [Poisson's ratio, Density (kg/m³), Alfa (mm/mm/c)]

BSECTIONS (Beam Sections)

Name is Beam Section name (up to 3 characters). The values in square brackets ([...]) are optional.

Import

English units

Name, Axial area (in²), Major moment of inertia (in⁴), Minor moment of inertia (in⁴), [Torsional inertia (in⁴), Major shear area (in²), Minor shear area (in²), Depth (inch), Width (inch)]

SI units

Name, Axial area (mm²), Major moment of inertia (mm⁴), Minor moment of inertia (mm⁴), [Torsional inertia (mm⁴), Major shear area (mm²), Minor shear area (mm²), Depth (mm), Width (mm)]

BLOADS (Beam Loads)

Name is Beam Load name (up to 3 characters). The values in square brackets ([...]) are optional.

English units

Name, T1 (F), [T2 (F), T3 (F), Additional weight (lb/ft)]

SI units

Name, T1 (c), [T2 (c), T3 (c), Additional weight (kg/m)]

LAYOUT

The piping layout is entered as a (key letter-data) pair, which may be input in any order. The exceptions to this convention are the comments which must always appear last. The pairs are separated by commas. A data line which contains only continued comments (from the previous line) is acceptable. A list of key letters follows:

F From node
T To node
H Hydro test Load
L Location node
K Code for node
M Material number
P Pipe (Section) property number
J Joint code
X X offset
Y Y offset
Z Z offset
B Bend radius
C Comments (separated by commas)
* Model comments (Example: * Node 230 = P-427B Suction)

LAYOUT Example (English units)

```
LAYOUT
F10,KA,M5,P12,CT=650,P=500
T20,KI,X10'8-1/2,B20
T30,Z8
T40,JR,X1'6,CWGT=250
```

Import

- F (From) From node is specified. When a new branch is started, the first node of the branch is specified as a “From” node. The X, Y and the Z fields are taken as coordinates rather than offsets from the previous node.
- T (To) To node is specified. This is a “To” node from the previous “From” node or the previous “To” node (but not from the previous “Location node”).
- L (Location) Location node is used to input additional data at a node when the node has more than one data item such as a hanger/force, etc.
- K (Code) The following codes may be used:

<u>Code</u>	<u>Description</u>
A	Anchor
B	Branch connection
H	Hinge (To node only)
M	Miter bend (To node only)
I	Tangent intersection (To node only)
T	Welding Tee
S	Sweepolet
W	Weldolet
F	Fabricated Tee
E	Extruded Tee
R	Radiused branch
P	Branch on thickened pipe

M (Material) A material is retained until altered. Another material should be entered only when there is a change.

P (Pipe) A pipe (section property) is retained until altered. Another section property should be entered only when there is a change.

J (Joint Code)

<u>Code</u>	<u>Description</u>
B	Ball joint
C	Cut pipe
D	Reducer
E	Expansion
I	Jacket bend
L	Elastic element
M	Beam
P	Jacket pipe
R	Rigid
S	Slip joint
T	Tierod
V	Valve

The weight of a rigid joint or a valve should be entered using a WGT comment. The stiffnesses of the expansion joint should be entered using an ES comment and the pressure thrust area should be entered using the TA comment.

Import

X, Y and Z The offsets May be entered in combination of feet, inches and fractions of an inch for English units and mm for SI units.

Example (English units)

Entry	Value
-10	-10 ft
10'8 or 10-8	10 ft 8 in
0'8 or 0-8	8 in
10.5	10 ft 6 in
1'6-3/8 or 1-6-3/8	1 ft 6.375 in

B (Bend Radius) The bend radius (inch or mm) is entered only if a tangent intersection(i.e., Bend, Jbend and Miter Bend) has been specified. The default is the long radius for the bend and jacketed bend.

C (Comment) The comment section allows entry of data related to a particular node or element. For example, a pipe end specified as a hinge would have the rotational spring constant and the direction vector entered in the comment section. A line temperature can be entered as comment and is retained until changed. Multiple comments may be entered separated by commas. If the last comment is followed by a comma, the comment is continued on the next line.

*(Model comment)The model comment section allows entry of notes relevant to analysis model. The model comment can e up to 70 characters.

Example: *Load cases considered for analysis are T1 and P1.

The comments are as follows:

AMB	Ambient or reference temperature (F or C) Default is 70 F. Example: AMB=80
AWGT	Additional weight for valve (lb. or kg) Use only for valve. Example: AWGT=100
BTHK	Bend thickness (inch or mm) Use for Bend and Jbend (core). Example: BTHK=6.01
BMAT	Name of Bend Material (Up to 3 characters) Should be defined in Material section before use. Example: BMAT=BM1
BK	Bending stiffness (in-lb/deg or Nm/deg) Use only for Bellows Example: BK=10000
BSIF	Bend SIF In-plane Out-plane

Import

	Example: BSIF(1.5,2.0) For Piping code with one SIF, use in-plane=Out-plane=SIF Example: BSIF (2.0,2.0)
BETA	Beta angle for Beam (deg or rad) Use only for Beam Example: BETA=90
CONE	Cone angle for reducer (deg. or rad.)
CNOD	Connected to Node Use for Guide, Hanger, Limit Stop, Rod Hanger, Skewed Restraint, User Hanger and Constant Support Example: CNOD=150
CLD	Cold load Use only for User Hanger. Example: CLD=1
CRTCH	Crotch radius for an extruded tee (inch or mm) Example: CRTCH=1.25
CS	Constant support spring Example: CS=2: Two constant support springs
CWGT	Concentrated weight (lb or kg) Use only for concentrated weight. Example: CWGT=200
D or DIS	Specified displacements (Inch or mm) Note: Entry of zero is ignored and not treated as a specified displacement. Example: D(0.1,-0.25,0) or DIS(0.1,-0.25,0) DX=0.1,DY=-0.25 or DISX=0.1,DISY=-0.25
DV	Direction vector. DV(xcomp, ycomp, zcomp) Use for Hinge, Snubber and Nozzle Example: DV(1,-2,0)
DIS2	Specified Displacement for Temperature 2 (inch or mm) Use only for Anchor Note: Entry of zero is ignored and not treated as a specified displacement. Example: DIS2(1,-2,0)
DIS3	Specified Displacement for Temperature 3 (inch or mm) Use only for Anchor Note: Entry of zero is ignored and not treated as a specified displacement. Example: DIS3(1,-2,0)
E	Young's modulus (psi or Mpa) Use only for Nozzle Example: E=28E6
ES	Expansion joint stiffnesses Axial(lb/in or N/mm)

Import

	Lateral(lb/in or N/mm) Torsional(in-lb/deg or NM/deg) Example: ES(1000,5000,200)																
F or FIXD	Translational restraint Example: FIXD(1,0,1) : Restrain X and Z translations FIXDX=1 or FX or FIXDX																
FF	Bend Flexibility factor Use for Bends and Miter bends Example: FF=1.5																
FIXR	Rotational restraint Example: FIXR(0,1,0) : Restrain Y rotation. FIXRY=1 or FIXRY																
FLANGE	FLANGE = Type, where <table border="0" style="margin-left: 20px;"> <thead> <tr> <th style="text-decoration: underline;">Type</th> <th style="text-decoration: underline;">Description</th> </tr> </thead> <tbody> <tr> <td>WN</td> <td>Weld neck flange</td> </tr> <tr> <td>SO</td> <td>Single welded slip on</td> </tr> <tr> <td>DW</td> <td>Double welded slip on</td> </tr> <tr> <td>SW</td> <td>Socket welded</td> </tr> <tr> <td>FW</td> <td>Fillet welded</td> </tr> <tr> <td>IJ</td> <td>Lap joint</td> </tr> <tr> <td>TH</td> <td>Threaded</td> </tr> </tbody> </table> Example: FLANGE=TH (Threaded flange)	Type	Description	WN	Weld neck flange	SO	Single welded slip on	DW	Double welded slip on	SW	Socket welded	FW	Fillet welded	IJ	Lap joint	TH	Threaded
Type	Description																
WN	Weld neck flange																
SO	Single welded slip on																
DW	Double welded slip on																
SW	Socket welded																
FW	Fillet welded																
IJ	Lap joint																
TH	Threaded																
FOR	Force (lb or N) Use for Force and Harmonic load. Example: FOR(100,0,-200) or FORX=100,FORZ=-200																
FFOR	Friction force (lb or N) Use for Slip joint Example: FFOR=100																
FTOR	Friction torque (ft-lb or Nm) Use for Slip joint Example: FTOR=100																
FRCT	Bending and Torsional friction Torque (ft-lb or Nm). FCRT(Bending, Torsional) Use only for Ball joint Example: FCRT(100,150)																
FREE	Free anchor during hanger design Example: FREE: Free all directions FREEY: Free Y direction																
FRE	Frequency (Hz) Use only for Harmonic Load. Example: FRE=30																

Import

G	Guide
GAP	Tension and Compression gap (inch or mm) GAP(Tension, Compression) Use only for Tie rod. Example: GAP(10,15)
GGAP	Guide Gap (inch or mm) Use only for guide Example: GGAP=5
HSG	Hydrotest Specific Gravity Use only for Hydrotest Load Example: HSG=0.7
HPRES	Hydrostatic Pressure (psi or bar) Use only for Hydrotest Load Example: HPRES=3
HTYP	Defines the Hanger Type. Refer to end of this appendix for details. Use only for Hanger Example: HTYP=16 (Grinnell)
IN1	Intermediate Node 1 for Bends Node Number (>1 and < 9999) Angle (deg or rad) Use for Bend and Jbend Example: IN1(300,30)
IN2	Intermediate Node 2 for Bends Node Number (>1 and < 9999) Angle (deg or rad) Use for Bend and Jbend Example: IN2(400,30)
INSF	Insulation factor Use only for Valve Example: INSF=3.0
JCAP	Jacked End Cap Defines the data type Jacked End Cap Example: JCAP
JMAT	Jacket Material (up to 3 characters) Use for Jpipe and Jbend Example: JMAT=A53
JSEC	Jacket Section (up to 3 characters) Use for Jpipe and Jbend Example: JSEC=N10
JLOAD	Jacket Load (up to 3 characters) Use for Jpipe and Jbend Example: JLOAD=L3

Import

JTHK	Jacket Thickness (inch or mm) Use only for Jbend
JR	Jacket Radius (inch or mm) Use only for Jbend. Example: JR=6.75
K	Translational stiffness (lb/inch or N/mm) Use for Skewed restraint Example: K=500,DV(1.5,-0.75,0.25)
KR	Rotational stiffness (in.-lb./deg. or N-m/deg) Use for Skewed restraint Example: KR=1200,DV(1.2,2.5,0)
KTIE	Tension and Compression stiffness. KTIE(Tension, Compression) (lb/in or N/mm). Use only for Tie rod Example: KTIE(1000,1500)
LS	Limit stop LS(M1,M2) M1=allowable movement in negative direction (in. or mm) M2=allowable movement in positive direction (inch or mm) Example : LS(-1.0,1.5), DV(0,1,0), MU=0.3
LOAD	Beam load reference (up to 3 characters) Note: Beam load should be defined in BLOADS section before use. Example: LOAD=B1
LEN	Length (inch or mm) Use only for Branch SIF with type Branch on Thickened Pipe Example: LEN=5
L1	Length 1 (ft-in or mm) Use to define "L" for API 650 Nozzle and "L1" for WRC 297 Nozzle Example: L1=3'0" or L1=900
L2	Length 2 (ft-in or mm) Use only for WRC 297 Nozzle to define "L2" Example: L2=4'0" or L2=1200
LONG	Cut long (inch or mm) Use only for Cut pipe. Example: LONG=100
LXAX	Local X axis. LXAX(xcomp, ycomp, zcomp) Use only for Elastic Element Example: LXAX(1,0,0)
LYAX	Local Y axis. LYAX(xcomp, ycomp, zcomp) Use only for Elastic Element Example: LYAX(0,1,0)

Import

MAT	Beam material reference (up to 3 characters) Note: Beam material should be defined in Beam material (BMATERIALS) section before use. Example: MAT=M1
MM	Mismatch (inch or mm) Use only for weld Example: MM=5
MLV	Maximum load variation (%) in hanger design Default is 25%. Example: MLV=30
MOM	Moment (ft-lb or NM) Example: MOM(200,-100,0) or MOMX=200, MOMY=-100
MU	Friction co-efficient Example: MU=0.3
NOD	Nozzle outside diameter (inch or mm) Example: NOD=104
NTHK	Nozzle thickness (inch or mm) Example: NTHK=6.01
NOZZLE	Defines the Nozzle data type. (650 or 297) Example: NOZZLE=650 (API 650) or NOZZLE=297 (WRC 297)
OD1	Outer diameter at from end for the reducer (inch or mm)
OD2	Outer diameter at to end for reducer (inch or mm)
OFFSET	Offset of concentrated weight from node or additional weight of valve from the center of valve (inch or mm) OFFSET(X offset, Y offset, Z offset). Example: OFFSET(0,18,0)
PAD	Thickness of reinforcement for fabricated tee (inch or mm) Example: PAD=0.25
PH	Phase (deg or rad) Use only for Harmonic Load Example: PH=10
P or PRES	Pressure (psig or bar) Example: P=500
ROT	Specified rotation (deg or rad) Note: Entry of zero is ignored and not treated as a specified rotation. Example: ROT(1.5,0,-0.25) ROTX=1.5,ROTZ=-0.25
RLIM	Rotation limit (deg or rad) Use only for Hinge Example: RLIM=10

Import

ROTL	Rotational limit in Bending and Torsion ROTL(Bending, Torsion) (deg or rad) Use only for Ball joint Example: ROTL(10,20)
ROTK	Rotational stiffness. ROTK(kxx, kyy, kzz) (deg or rad) Use for Anchor and Elastic element Example: ROTK(5,6,3)
ROT2	Rotational Displacement for Temperature 2. (deg or rad) ROT2(kxx, kyy, kzz) Use only for Anchor Example: ROT2(10,15,10)
ROT3	Rotational Displacement for Temperature 3. (deg or rad) ROT3(kxx,kyy,kzz) Use only for Anchor Example: ROT3(10,15,10)
R	Fillet radius (inch or mm) Use only for Branch SIF (Radiused Branch and Branch Connection) Example: R=10
RPAD	Reinforcing Pad (0 or 1) Use only for Nozzle (API 650) Example RPAD=1
RK	Rotational Stiffness in Bending and Torsion. RK(Bending, Torsion) (ft-lb or Nm) Use only for Ball joint Example: RK(100,150)
SG	Specific Gravity Example: SG=0.8
SIF	Stress intensification factor at node. SIF=value or SIF(in-plane, Out-plane) Example: SIF=1.3 or SIF(1.5,2.0)
SHORT	Cut short (inch or mm). Use only for Cut pipe. Example: SHORT=100
SEC	Beam section reference (up to 3 characters) Note: Beam section should be defined in Beam section (BSECTIONS) before use. Example: SEC=BS1
SEIS	Seismic displacement (inch or mm) (x,y,z) Example: SEIS (-0.25, 0.00, -0.25)
SERO	Seismic rotation (deg or rad) (rx,ry,rz) Example: SERO (0.25, 0.00, 0.25)
SETT	Settlement displacement (inch or mm) (x,y,z) Example: SETT (-0.25, -0.25, -0.25)

Import

SPIDER	Defines the data type SPIDER Example: SPIDER
SR	Turn on the option Short range. Use only for Hanger. Example: SR
STIFF	Stiffness (lb/in or N/mm) Use for Guide, Limit stop and Snubbers Example: STIFF=1000
STRO	Settlement rotation (deg or rad) (rx,ry,rz) Example: STRO (0.15, 0.15, 0.15)
T or TEMP	Temperature (F or C) Example: T=650
TA	Pressure thrust area for bellows and Slip joints (in ² or mm ²). Example: TA=12.3
THK	Thickness (inch or mm) Use only for Branch SIF (Radiused Branch & Branch on Thickened Pipe) Example: THK=10
THK1	Thickness at from end for reducer (inch or mm)
THK2	Thickness at to end for reducer (inch or mm)
THKF	Thickness factor Use only for Valve. Example: THKF=3.0
TRAK	Translational Stiffness (lb/in or N/mm). TRAK(kx,ky,kz) Use for Anchor and Elastic element. Example: TRAK(1000,1500,2000)
TJOINT	Defines the Threaded Joint Example: TJOINT
U or UNIF	Uniform load (lb/ft or Kg/m) Example: U=200
US	User-defined spring hanger US(No. of hangers, spring rate(lb./inch or N/mm), hot load(lb. or N)) Examples: US(2,600,1540) US(1,0,2300) : Constant support
VS	Variable spring hanger Example : VS, VS=2: two variable spring hangers
VOD	Vessel outside diameter (inch or mm) Use only for Nozzle. Example: VOD=250
VTHK	Vessel thickness (inch or mm)

Import

	Use only for Nozzle Example: VTHK=10
VWGT	Valve weight (lb or kg) Example: VWGT=100
WGT	Weight of an item (ball joint, flange, Slip joint, etc.) (lb or kg) Example: WGT=50
WS	Widely Spaced Use only for Miter bend Example: WS
WTYPE	Weld type Example: WTYPE=1 (1 = Butt weld, 2 = Fillet weld, 3 = Concave fillet weld, 4 = Tapered Transition)

PUMPS

Description (up to 16 characters)
Suction/Discharge location (0 = Top, 1 = Side, 2 = End)
Shaft axis (xcomp, ycomp, zcomp)
Center of pump (x, y, z)

English units

Desc, Horizontal / Vertical inline (0 or 1), Shaft axis, Center of pump (ft-in), Suction node, Suction location, Discharge Node, Discharge Location

SI units

Desc, Horizontal / Vertical inline (0 or 1), Shaft axis, Center of pump (mm), Suction node, Suction location, Discharge Node, Discharge Location

Horizontal Pumps

Desc, 0, Shaft axis, Center of pump, Suction node, Suction location, Discharge Node, Discharge Location

Vertical Inline Pumps

Desc, 1, Suction node, Discharge Node

Example

```
PUMPS
427BSOUTH,1,350
427ANORTH,0,0.0000,1.0000,0.0000,-3.8299,-13'8",44.33,390,2,410,2
427WEST,1,300
```

COMPRESSORS

Description (up to 16 characters)
Shaft axis (xcomp, ycomp, zcomp)

Import

Description, Shaft axis direction, Inlet Node, Exhaust Node, Extraction Node 1, Extraction Node 2.

Example

```
COMPRESSORS  
Compressor1,1.0000,0.0000,0.0000,210,300
```

TURBINES

Description (up to 16 characters)
Shaft axis (xcomp, ycomp, zcomp)

Description, Shaft axis direction, Inlet Node, Exhaust Node, Extraction Node 1, Extraction Node 2.

Example

```
TURBINES  
TURBINE1,1.0000,0.0000,0.0000,250,360
```

SEISMIC

X,Y,Z Static equivalent seismic loads in g's.

SEISMIC Example: Static seismic loads of 0.25 and 0.3 g's in X and Z directions.

```
SEISMIC  
0.25,0.00,0.30
```

WIND

Wind velocity (MPH or m/s), Shape factor (defaults to 0.6), X, Y, Z components of wind direction.

WIND Example: A wind load of 100 mph in the X direction.

```
WIND  
100,0.600,1.000,0.000,0.000
```

Example:

An example of a model batch file (.mbf file) is shown below:

```
Desalter Pumps  
  
OPTIONS  
HGRA,B313  
MATERIAL  
A53,0.283,0.300,1.00  
A53,-325,31.4E+6,5.00E-6,20000  
A53,-200,30.8E+6,5.35E-6,20000  
A53,-100,30.2E+6,5.65E-6,20000  
A53,70,29.5E+6,6.07E-6,20000  
A53,200,28.8E+6,6.38E-6,20000  
A53,300,28.3E+6,6.60E-6,20000  
A53,400,27.7E+6,6.82E-6,20000  
A53,500,27.3E+6,7.02E-6,18900
```

Import

A53,600,26.7E+6,7.23E-6,17300
A53,650,26.1E+6,7.33E-6,17000
A53,700,25.5E+6,7.44E-6,16500
A53,750,24.9E+6,7.54E-6,13000
A53,800,24.2E+6,7.65E-6,10800
A53,850,23.3E+6,7.75E-6,8700
A53,900,22.4E+6,7.84E-6,6500
A53,950,21.4E+6,7.91E-6,4500
A53,1000,20.4E+6,7.97E-6,2500
A53,1050,19.2E+6,8.05E-6,1600
A53,1100,18.0E+6,8.12E-6,1000
PIPE
4,4.5,0.237,0,12.5,11,2
6,6.625,0.28,0,12.5,11,2
8,8.625,0.322,0,12.5,11,2
10,10.75,0.365,0,12.5,11,2
12,12.75,0.406,0,12.5,11,2
LOADS
1,400,280,0.85,400,280,350,190
1N,400,280,0.85,100,280,350,190
1S,100,280,0.85,400,280,350,190
1C,100,280,0.85,100,280,100,190
4,400,600,0.85,400,600,285,450
4N,400,600,0.85,100,600,285,450
4S,100,600,0.85,400,600,285,450
SO1,350,190,0.85,70,0,70,0
SOA,230,145,0.85,70,0,70,0
SD1,400,280,0.85,70,0,70,0
DO1,285,450,0.85,70,0,70,0
DOA,225,485,0.85,70,0,70,0
DD1,400,600,0.85,70,0,70,0
LAYOUT
F10,CNOZZLE=297,L2=10'0",NOD=17.5,NTHK=1.3125,VOD=144,VTHK=1.375,
L1=10'0",E=29.0E+6,DV(0.0000,0.0000,1.0000)
L10,CFLANGE=WN,WGT=220

T20,JV,MA53,P10,Y1'6-1/4",CL=1,VWGT=625,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=220
T30,Y0'4-5/8"
T40,JD,P8,Y0'7",COD1=10.75,THK1=0.365,OD2=8.625,THK2=0.322
T50,KI,B12,Y6'0-1/8"
T60,KI,B12,X-1.0017,Z1.0017
T70,KI,B12,X-5'8-15/16
T80,KI,B12,Y-11'3"
T90,Z6'6",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T100,Z20'0",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T110,JI,B6.75,P4,Z17'6",CJMAT=A53,JSEC=12,JLOAD=1,JR=19.125
T120,P8,X10'11",CLS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T130,KI,B12,X2'0"
*
T240,KT,JE,Z6'4-9/16",CES(100,400,300),TA=500,BK=11459
*

Import

```
T160,X-2'0",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T170,KI,B12,X-8'0"
T180,Y-8'2-13/16,CFLANGE=WN,WGT=152
T190,JV,Y-1'4-3/4",CVWGT=590,THKF=3.0000,INSF=1.7500,FLANGE=WN,
WGT=152
T200,KI,B12,Y-1'4-3/8",CL=1N
T210,JS,Z-1'0",CFFOR=1000
T220,JD,P6,Y0'0-15/16",Z-0'6",COD1=8.625,THK1=0.322,OD2=6.625,
THK2=0.28,FRE=30.000
L200A,CUS(1,900,2250)
*
F240
T250,KT,P8,Z1'2",CL=1
*
T260,KT,Z1'2"
T270,Z0'11"
*
F260
T280,X-2'0",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T290,KI,B12,X-8'0"
T300,Y-8'2-13/16,CFLANGE=WN,WGT=96
T310,JV,Y-1'4-3/4",CVWGT=590,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=152
T320,KI,B8,Y-1'4-3/8",CL=1S
T330,Z1'0"
T340,JD,Y0'0-
15/16",Z0'6",COD1=8.625,THK1=0.322,OD2=6.625,THK2=0.28
*Node 230 = P-427B Suction
T350,KA,P6,Z1'5-3/16",CDIS(0,0.027,0),DIS2(0,0.027,0),
DIS3(0,0.027,0)
L350,CFLANGE=WN,WGT=96
*Node 550 = P-427A Suction
F220
T390,KA,Z-2'7",CL=1N,DIS(0,0.027,0),DIS3(0,0.027,0)
L390,CFLANGE=WN,WGT=96

T400,Z-1'6-3/8"
T410,JL,Z-1'5-3/8",CTRAK(1,1,1),ROTK(1,1,1),
LXAX(1.0000,0.0000,0.0000),LYAX(0.0000,1.0000,0.0000)
*From node 200 2'11.5" in the Z direction
F370,KA,X-3.8300,Y-13'8",Z59.6267,CDIS(0,0.027,0),
DIS2(0,0.027,0),DIS3(0,0.027,0)
L370,CFLANGE=WN,WGT=58
T420,P4,Z0'3-3/8",CL=4S
T430,JD,P6,Z0'5-1/2",COD1=4.5,THK1=0.237,OD2=6.625,THK2=0.28
T440,KI,B9,Z2'0-5/8"
T450,Y1'0-7/8",CFLANGE=WN,WGT=96
T460,JV,Y1'5-5/8",CVWGT=204,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=96
T470,JV,Y1'4-1/8",CVWGT=334,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=96
T480,KI,B9,Y7'0-3/8"
T490,X8'0",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
```

Import

```
T500,KT,X3'6"
*
*From node 340 -2'11.5" in the -Z direction
F410,KA,CDIS(0,0.027,0),DIS3(0,0.027,0)
L410,CFLANGE=WN,WGT=58
T510,P4,Z-0'3-3/8",CL=4N
T520,JD,P6,Z-0'5-1/2",COD1=4.5,THK1=0.237,OD2=6.625,THK2=0.28
T530,KI,B9,Z-4'0-5/8"
T540,Y1'0-7/8",CFLANGE=WN,WGT=96
T550,JV,Y1'5-5/8",CVWGT=204,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=96
T560,JV,Y1'4-1/8",CVWGT=334,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=96
T570,KI,B9,Y7'0-3/8"
T580,X8'0",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T590,KI,B9,X3'6"
T500
L530B,CUS(1,520,1392)
F500
T600,KI,B9,Z1'7",CL=4
T610,X-3'6",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T620,KI,B9,X-10'11"
T630,KI,B9,Y-4'6-11/16
T640,KI,B9,Z7'4"
T650,Y1'1",CFLANGE=WN,WGT=96
T660,Y0'11"
T670,JD,P8,Y0'6",COD1=6.625,THK1=0.28,OD2=8.625,THK2=0.322
T680,KT,Y0'7"
*
F250
T685,P4,X-2'0",CL=1C,LS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T690,KI,B6,X-10'11"
T700,KI,B6,Y-1'5-11/16
T710,Z1'10",CFLANGE=WN,WGT=58

*
F680
T720,Z-1'6",CL=4,FLANGE=WN,WGT=58
T730,JV,Z-1'0-1/4",CVWGT=170,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=58
T735,Z-6'6-3/4",CL=1C,LS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T710,CL=1
*
F680
T800,KI,B12,P8,Y3'0",CL=4
T810,Z6'1-5/8",CFLANGE=WN,WGT=152
T820,Z0'4-3/8",CLS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T830,KI,B12,Z1'0"
T840,KI,B12,X7'0"
T850,KI,B12,X1'6",Y-1'6"
```

Import

```
T860,X6'0",CFLANGE=WN,WGT=152
T870,JV,X1'4-3/4",CVWGT=410,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=152
T880,KT,X1'4"
*
T890,KI,B12,Y1'7"
T900,KI,B12,Z2'0"
T910,Y-1'1-1/4",CFLANGE=WN,WGT=152
T920,JV,Y-1'4-3/4",CVWGT=410,THKF=3.0000,INSF=1.7500,FLANGE=WN,
WGT=152
T930,KI,B12,Y-1'1-1/4"
T940,KT,Z-2'0"
*
T950,X1'0",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T960,X1'0"
*
F940
T970,X-1'4"
*
F880
T980,KI,B12,X10'6"
T990,KI,B12,Y4'6"
T1000,X1'10",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1010,X26'8",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1020,KI,B12,X6'2"
T1030,KI,B12,Z2'0"
T1040,Y4'2"
T1050,JD,P10,Y0'7",COD1=6.625,THK1=0.28,OD2=8.625,THK2=0.322
T1060,KI,B15,Y1'3"
T1070,Z7'2",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1080,Z18'8",CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1090,KI,B15,Z15'2"
T1100,KI,B15,Y-25'6"
T1110,KI,B15,X-3.8971,Z-2'3"
T1120,Y1'7-7/8",CFLANGE=WN,WGT=220

PUMPS
427BSOUTH,1,350
427ANORTH,0,0.0000,1.0000,0.0000,3.8299,13'8",44.3351,390,2,410,2
COMPRESSORS
Compressor_1,1.0000,0.0000,0.0000,210,300
SEISMIC
0.25,0.00,0.30
WIND
100,0.600,1.000,0.000,0.000
```

Hanger Type

This section describes the hanger type (HTYP under comments section) number to be used for defining variable spring hanger type.

Hanger Description Type	Value
ABB-PBS	1

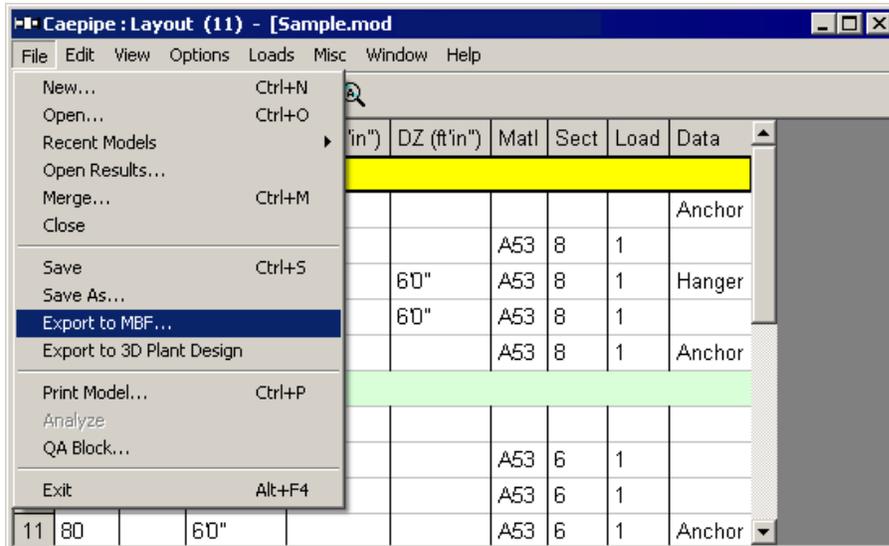
Import

Basic Engineers	2
Bergen Paterson	3
Bergen Paterson (L)	4
BHEL Hyderabad	5
BHEL Trichy	6
Borrello	7
Carpenter & Paterson	8
Corner & Lada	9
Dynax	10
Elcen	11
Fee and Mason	12
Flexider (30-60-120)	13
Flexider (50-100-200)	14
Fronck	15
Grinell	16
Hydra	17
Lisega	18
Mitsubishi (30-60-120)	19
Mitsubishi (80-160)	20
Myricks	21
NHK (30-60-120)	22
NHK (80-160)	23
Nordon	24
NPS Industries	25
Piping Services	26
Piping Tech & Products	27
Power Piping	28
Sanwa Tekki (30-60-120)	29
Sanwa Tekki (85-170)	30
Sarathi	31
Spring Supports	32
SSG	33
Comet	34

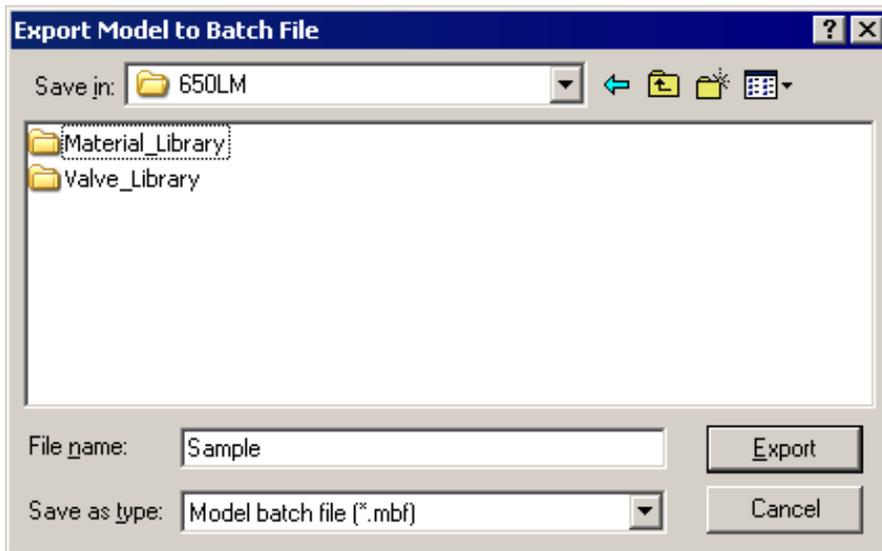
CAEPIPE can export model data from inside the Layout window to a text file with the extension .mbf (model batch file), which may be edited using a text editor. The edited text file may then be read back into CAEPIPE by using the Import feature. But note that the existing .mod file will be overwritten. So, you must rename the edited text file before importing it back into CAEPIPE.

To export a model, select the menu command File > Export from the Layout window.

Import



The Export Model dialog is shown. Click on Export button to write to a .mbf file.



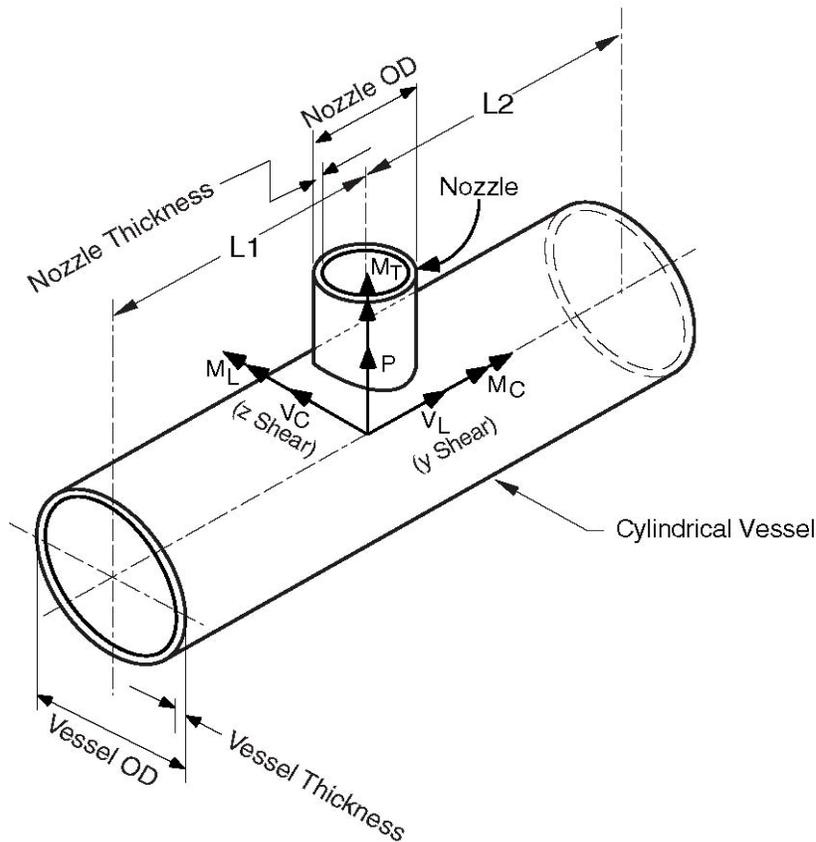
This page is blank

Appendix D

Nozzle Stiffness Calculations

Nozzle Stiffness Calculations (WRC 297)

Six stiffnesses are shown below at the nozzle-vessel interface — three are calculated and the other three are assumed rigid.



The coordinate system is as shown in the figure. The six components of the forces and moments at the nozzle-vessel interface are:

- | | |
|---------------------------------------|---|
| P = Radial load | M _C = Circumferential moment |
| V _C = Circumferential load | M _T = Torsional moment |
| V _L = Longitudinal load | M _L = Longitudinal moment |

Of the six components of stiffnesses, only three stiffnesses, axial (K_x), circumferential (K_{yy}), and longitudinal (K_{zz}), are calculated. The remaining three are assumed to be rigid.

Several graphs are given at the end of this appendix. The stiffness coefficients are obtained by interpolating logarithmically from these graphs.

The first two, Figures D-1 and D-2, are used to calculate nozzle stiffness coefficients for Nozzles on cylindrical vessels. Figure D-1 is used to calculate the axial stiffness coefficient and Figure D-2 is used to calculate circumferential and longitudinal stiffness coefficients.

Nozzle Stiffness Calculations

Nomenclature

- D = mean diameter of vessel
 d = outside diameter of nozzle
 T = thickness of vessel
 t = thickness of nozzle
 λ = $(d/D)\sqrt{D/T}$
 Λ = L/\sqrt{DT}
 L = unsupported length of cylinder
= $8L_1L_2/(\sqrt{L_1} + \sqrt{L_2})^2$
 L_1 = distance from nozzle center line to vessel end
 L_2 = distance from nozzle center line to vessel end
 E = modulus of elasticity of vessel material

Axial Stiffness(K_x)

$$K_x = \alpha \times \frac{4.95ET^2}{D\sqrt{\Lambda}} \quad (1)$$

where

α = stiffness coefficient read from Figure D-1

Circumferential Stiffness(K_{yy})

$$K_{yy} = \beta \times ET^3 \quad (2)$$

where

β = stiffness coefficient read from Figure D-2.

The bottom three curves in Figure D-2, marked Circumferential moment M_C are used to find β .

Longitudinal Stiffness(K_{zz})

$$K_{zz} = \gamma \times ET^3 \quad (3)$$

where

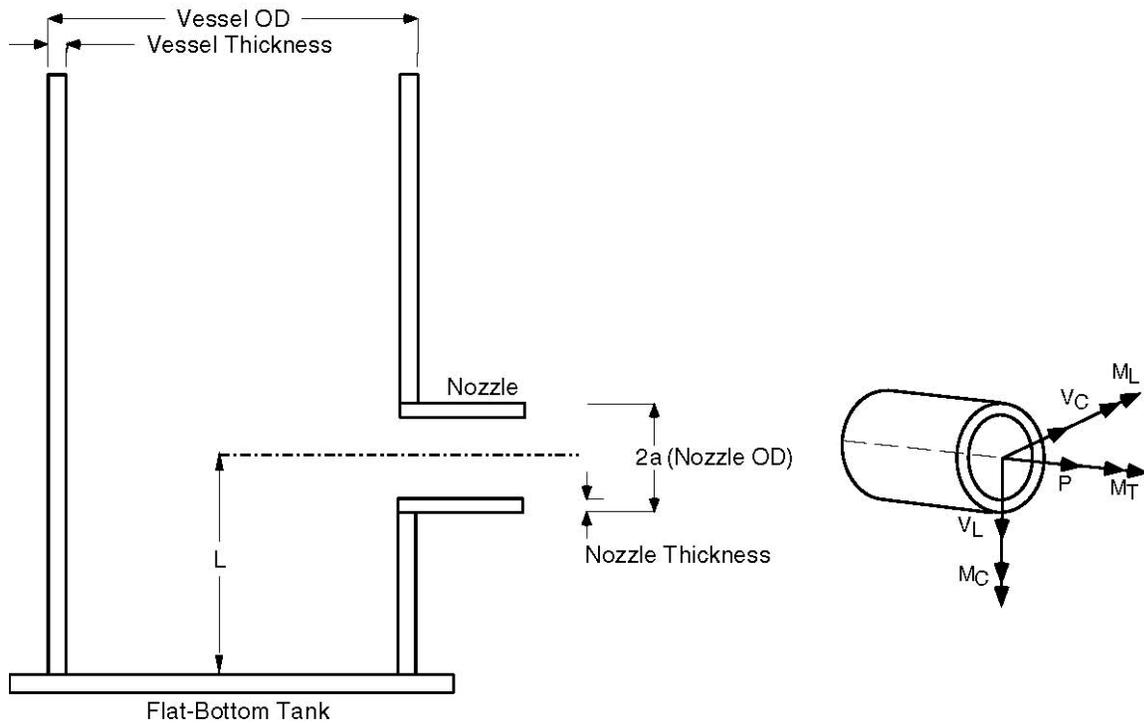
γ = stiffness coefficient read from Figure D-2.

The top two curves in Figure D-2, marked Longitudinal moment M_L are used to find γ .

Nozzle Stiffness Calculations

Calculation of Nozzle stiffnesses for Nozzles on Flat-bottom tanks

This procedure is similar to the previous one.



As before, only three stiffnesses are calculated as the other three are assumed to be rigid. The ones that are calculated are axial (K_x), circumferential (K_{yy}), and longitudinal (K_{zz}).

For Nozzles on flat-bottom tanks, twelve graphs are given at the end of this appendix, Figures D-3 through D-14. Six are for “with reinforcing pad (on vessel)” with the other six for no reinforcing pad on the vessel. The stiffness coefficients are obtained using the appropriate graph.

Nozzle Stiffness Calculations

Nomenclature

R = Mean radius of vessel
 t = thickness of vessel
 $2a$ = outside diameter of nozzle

Axial Stiffness(K_x)

$$K_x = K_R \times E \times (2a) \quad (4)$$

where

K_R = axial stiffness coefficient.

Circumferential Stiffness(K_{yy})

$$K_{yy} = K_C \times E \times (2a)^3 \quad (5)$$

where

K_C = circumferential stiffness coefficient.

Longitudinal Stiffness(K_{zz})

$$K_{zz} = K_L \times E \times (2a)^3 \quad (6)$$

where

K_L =longitudinal stiffness coefficient.

The graphs for stiffness coefficients follow:

Nozzle Stiffness Calculations

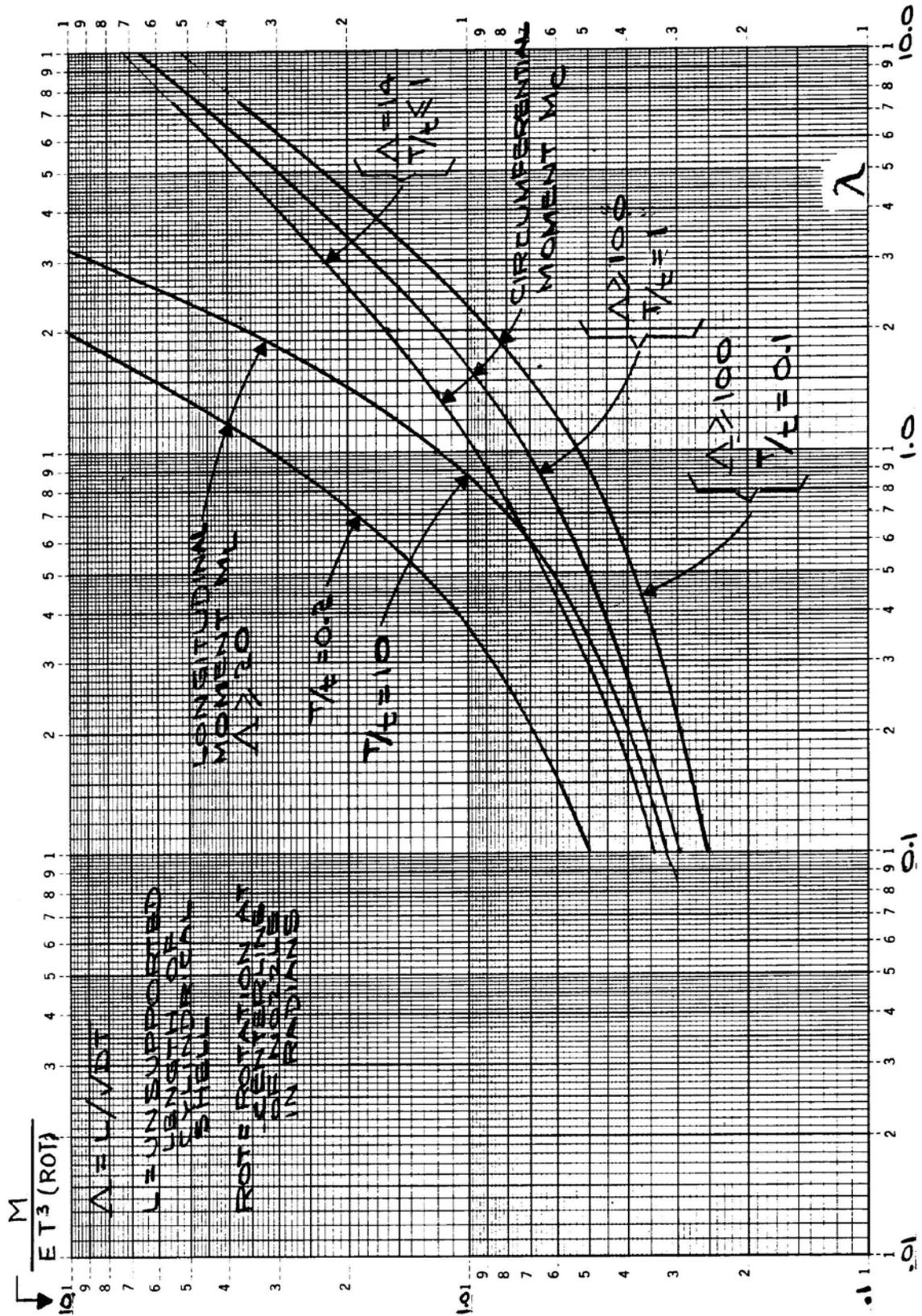


Figure D-2: Stiffness coefficients for moment loads on nozzle

Nozzle Stiffness Calculations

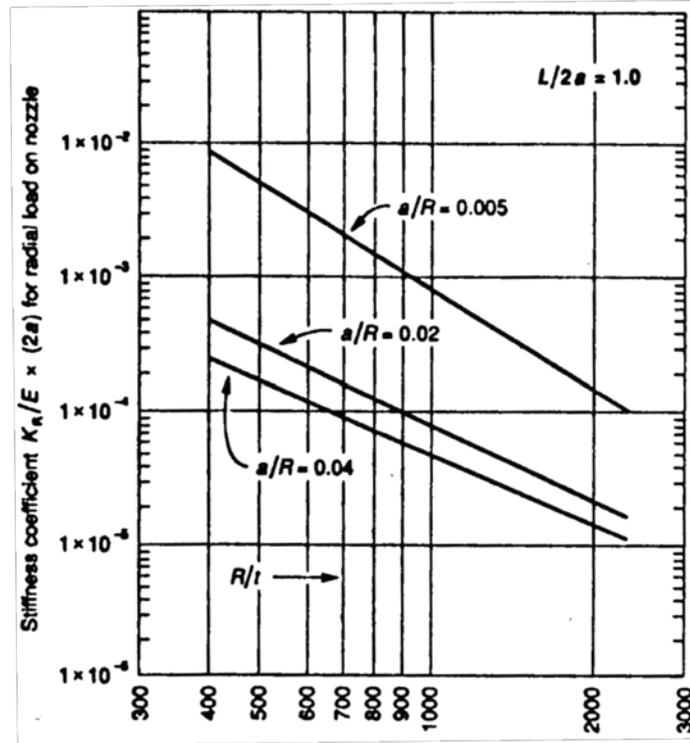


Figure D-3: Stiffness coefficient for axial load (with reinforcing pad) ($L/2a = 1.0$)

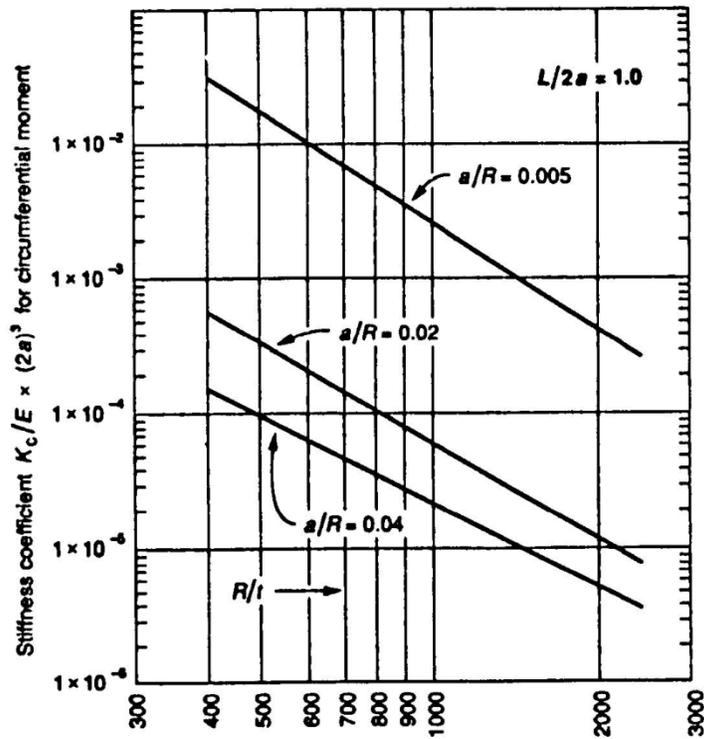


Figure D-4: Stiffness coefficient for circumferential moment (with reinforcing pad) ($L/2a = 1.0$)

Nozzle Stiffness Calculations

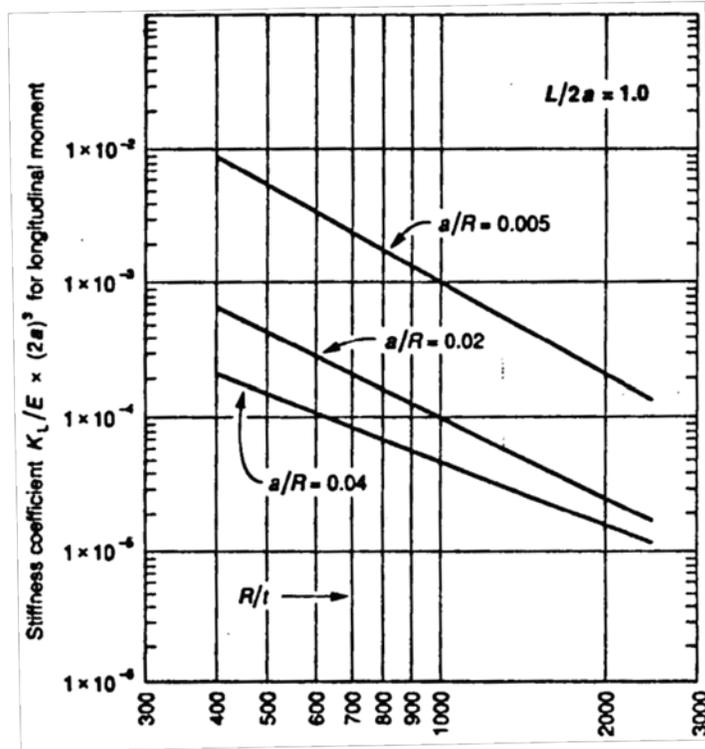


Figure D-5: Stiffness coefficient for longitudinal moment (with reinforcing pad) ($L/2a = 1.0$)

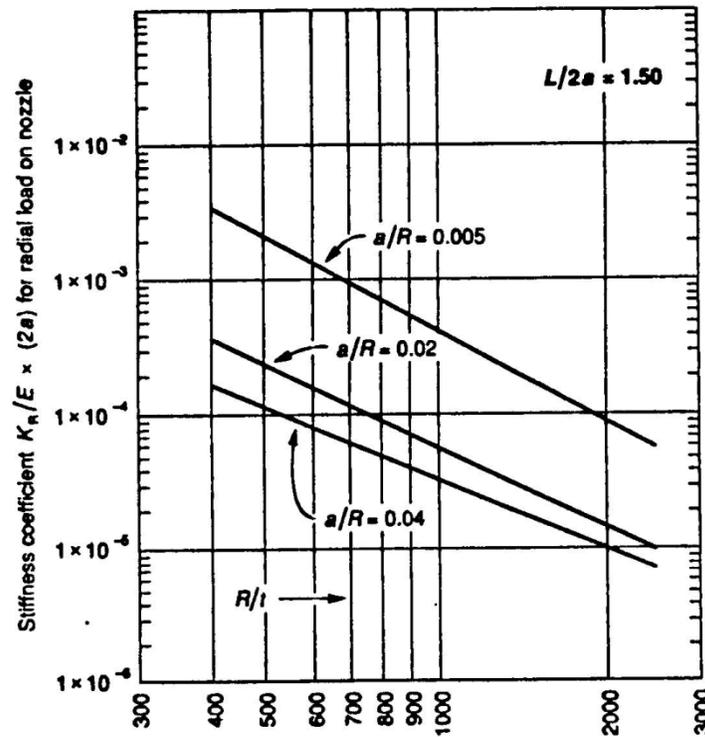


Figure D-6: Stiffness coefficient for axial load (with reinforcing pad) ($L/2a = 1.5$)

Nozzle Stiffness Calculations

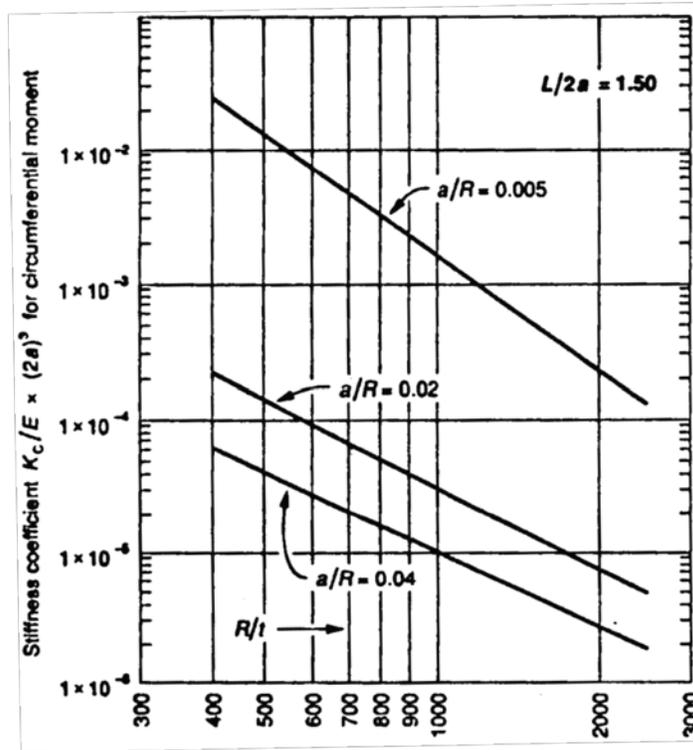


Figure D-7: Stiffness coefficient for circumferential moment (with reinforcing pad)
($L/2a = 1.5$)

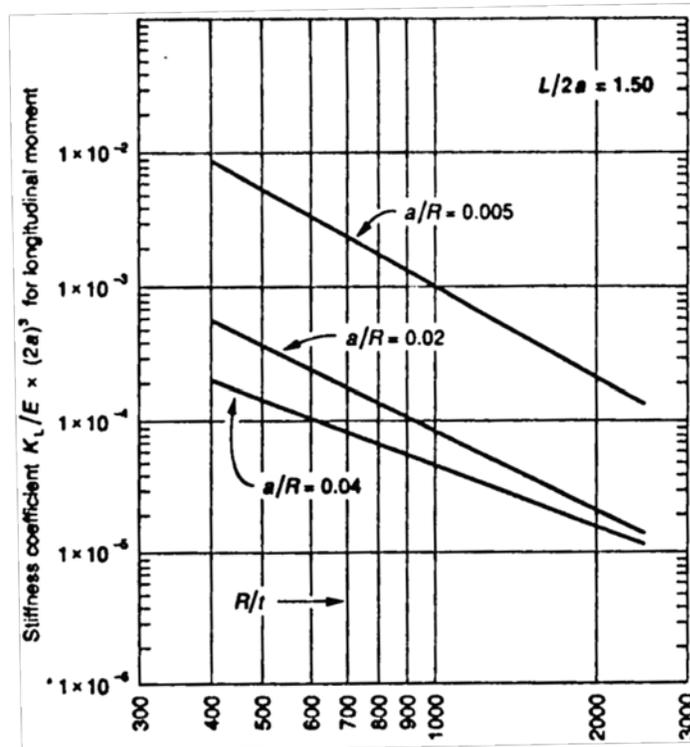


Figure D-8: Stiffness coefficient for longitudinal moment (with reinforcing pad) ($L/2a = 1.5$)

Nozzle Stiffness Calculations

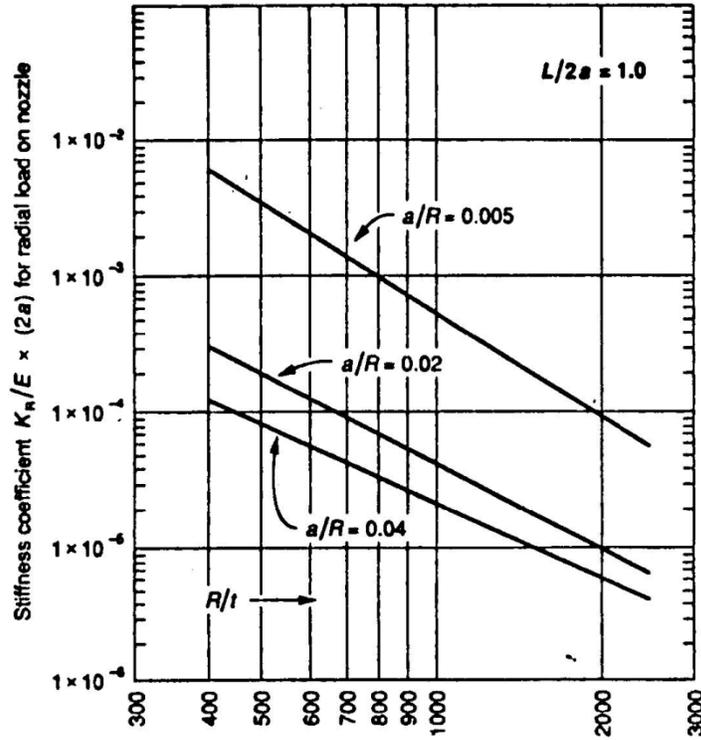


Figure D-9: Stiffness coefficient for axial load (no reinforcing pad) ($L/2a = 1.0$)

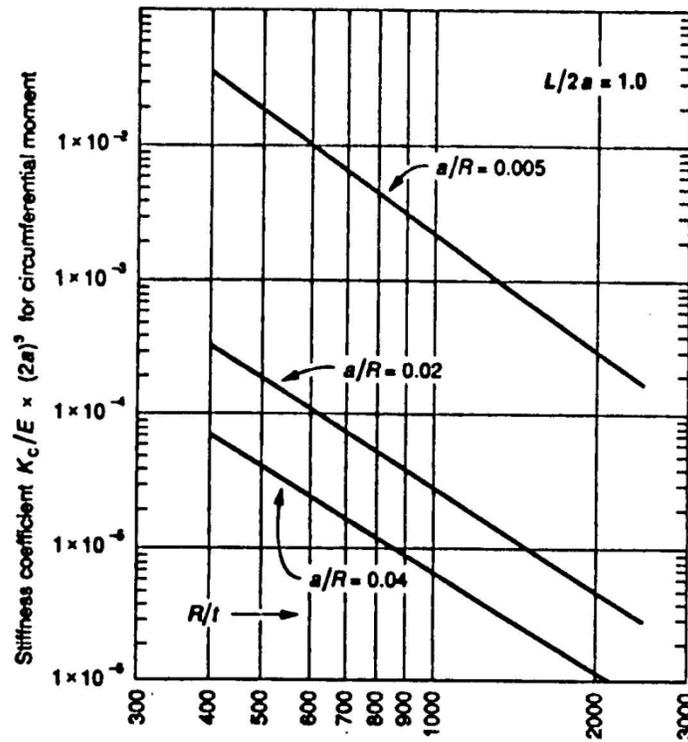


Figure D-10: Stiffness coefficient for circumferential moment (no reinforcing pad) ($L/2a = 1.0$)

Nozzle Stiffness Calculations

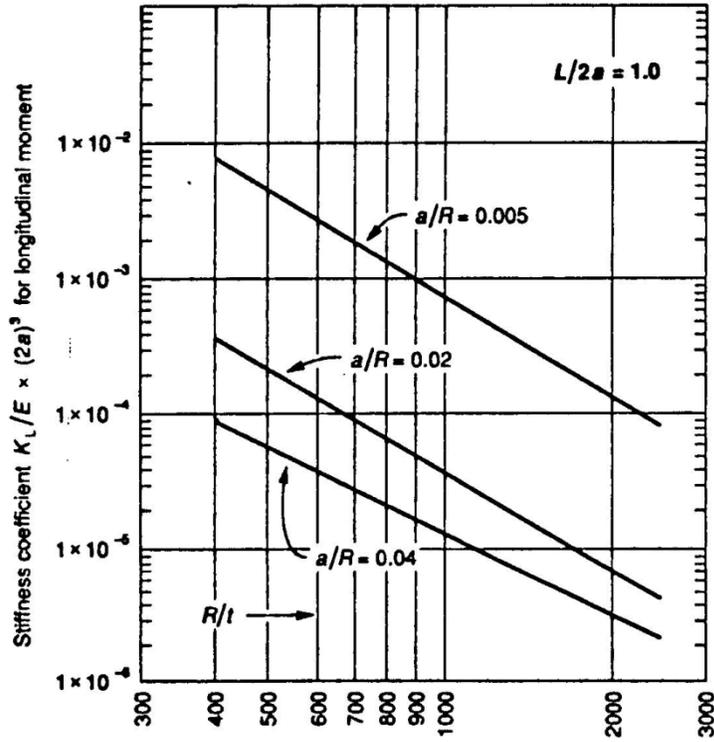


Figure D-11: Stiffness coefficient for longitudinal moment (no reinforcing pad) ($L/2a = 1.0$)

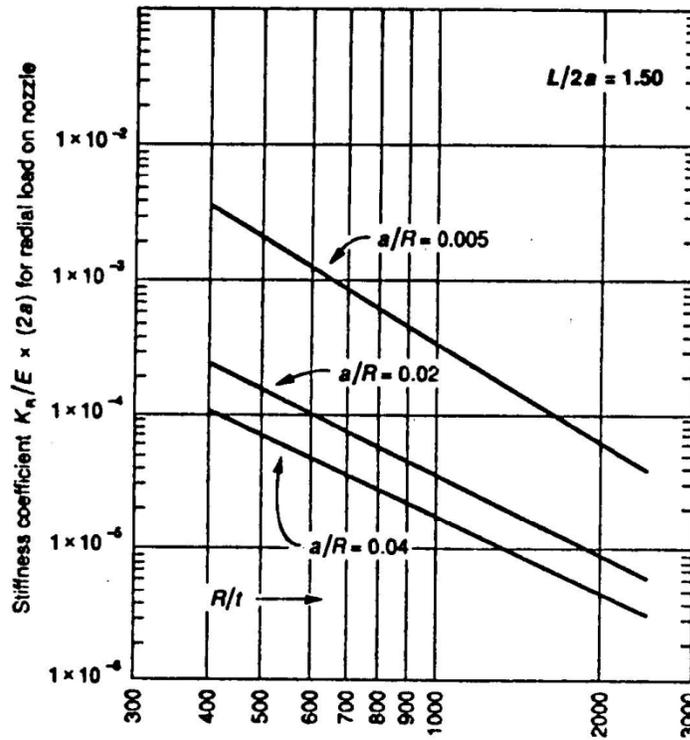


Figure D-12: Stiffness coefficient for axial load (no reinforcing pad) ($L/2a = 1.5$)

Nozzle Stiffness Calculations

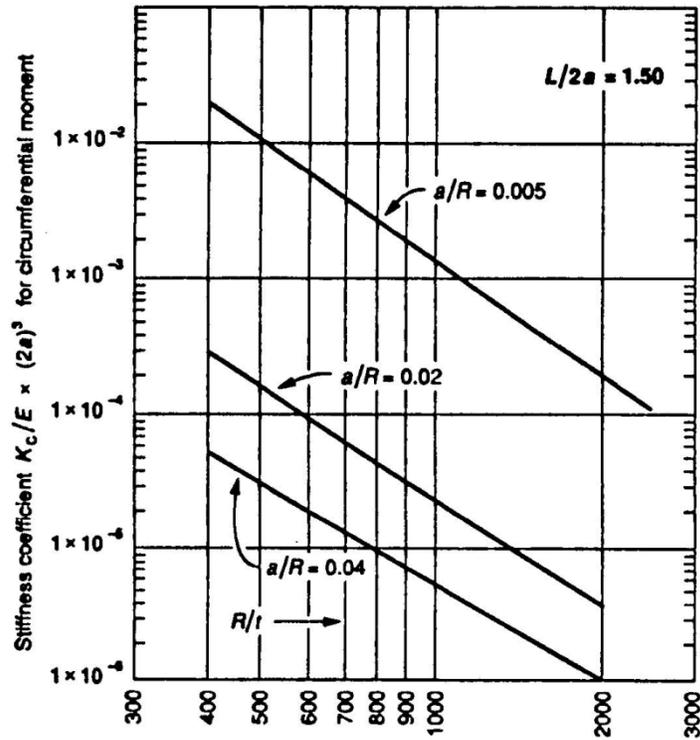


Figure D-13: Stiffness coefficient for circumferential moment (no reinforcing pad)
($L/2a = 1.5$)

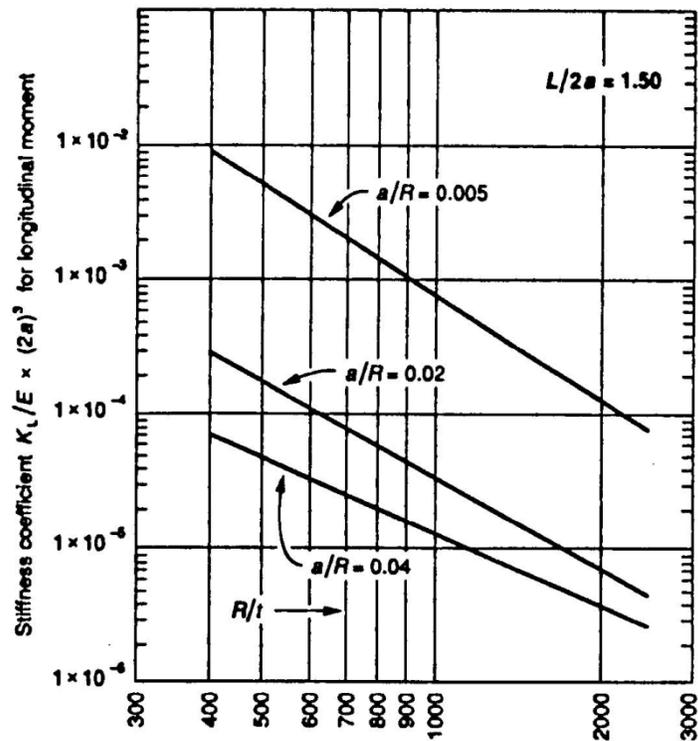


Figure D-14: Stiffness coefficient for longitudinal moment (no reinforcing pad) ($L/2a = 1.5$)

This page is blank

Appendix E

Dynamic Susceptibility Method

Dynamic Susceptibility Method

The “Dynamic Susceptibility” Method for Piping Vibration –A Screening Tool for Potentially Large Alternating Stresses

Dr. R. T. Hartlen, Plant Equipment Dynamics, Ontario, CANADA

Summary

The enhanced output of the “Modal Analysis” load case in CAEPIPE shows modal frequencies and mode shapes AND now two new outputs called “*dynamic stresses*” and “*dynamic susceptibility*.” The *dynamic stresses* are the dynamic bending stresses associated with vibration in a natural mode. That is to say, the *modal analysis result has been generalized* to include the alternating bending stresses associated with the vibration in a natural mode. The *dynamic susceptibility* for any mode is the ratio of the maximum alternating bending stress to the maximum vibration velocity. This “susceptibility ratio” provides an indicator of the susceptibility of the system to large dynamic stresses. Also, the associated animated mode shapes include color-spot-markers identifying the respective locations of maximum vibration and maximum dynamic bending stress. The susceptibility ratio and the graphics feature provide incisive insights into the reasons for high susceptibility and how to make improvements. This new feature is illustrated by application to the CAEPIPE “Sample problem” system.

1. Dynamic Susceptibility: New Analytical Tool Available for Vibration of Piping

When addressing vibration issues, the piping designer does not have the specific requirements, nor the analytical tools and technical references typically available for other plant equipment such as rotating machinery. Typically, piping vibration problems only become apparent at the time of commissioning and early operation, after a fatigue failure or degradation of pipe supports. Discovery of a problem is then followed by an ad hoc effort to assess, diagnose and correct as required. The “Dynamic Susceptibility” analysis, now included in CAEPIPE, provides a new analytical tool to assist the piping designer at any stage, from preliminary layout to resolution of field problems.

CAEPIPE’s Dynamic Susceptibility feature utilizes the “Stress per Velocity” method, an incisive analytical tool for “screening” the vibration modes of a system. It readily identifies which modes, if excited, could potentially cause large dynamic stresses. Furthermore, it reveals which features of the system layout and support are responsible for the susceptibility to large dynamic stresses. At the design stage, the method allows the designer to quickly identify and correct features that could lead to large dynamic stresses at frequencies likely to be excited. Where problems are encountered in the field, the method provides quick and incisive support to efforts of observation, measurement, assessment, diagnosis and correction.

The technical foundation of this method lies in an underlying fundamental relationship between the kinetic energy of vibratory motion, and the corresponding potential energy stored in elastic stresses. That is to say, the kinetic energy at zero displacement and maximum system velocity must equal the stored elastic energy at zero velocity and maximum displacement. This implies a fundamental relationship between vibration velocity and dynamic bending stresses, which is the foundation of the stress per velocity approach for “susceptibility screening” of vibration modes.

The key analytical step is to determine, mode by mode, the ratio of maximum dynamic stress to maximum vibration velocity. This ratio will lie in a lower “baseline range” for

uncomplicated systems such as classical uniform-beam configurations. For more complex systems, the stress / velocity ratio will increase due to typical complications such as three-dimensional layout, discrete heavy masses, changes of cross-section and susceptible branch connections. *System modes with large stress-velocity ratios are the potentially susceptible modes.*

The Stress / Velocity method, implemented in CAEPIPE as the Dynamic Susceptibility feature, *automatically and quickly finds these modes and quantifies the susceptibility.* Evaluation of the results, including special-purpose color animation, helps to identify which details of layout and support are responsible for the large stresses.

This technical note is to present and explain the “dynamic susceptibility” outputs now included in the modal analysis load case, and to illustrate by application to the standard CAEPIPE “Sample problem” system.

2. Underlying Fundamental Basis of the Method

2.1 Kinetic Energy and Potential Energy; Vibration Velocity and Dynamic Stresses

The underlying theoretical basis for the Stress / Velocity method is a deceptively straightforward but universally-applicable relationship between kinetic energy and potential (elastic) energy for vibrating systems. Stated simply, *for vibration at a system natural frequency*, the ***kinetic energy at maximum velocity and zero displacement*** must then be stored as ***elastic (strain) energy at maximum displacement and zero velocity***. Since the strain energy and kinetic energy are respectively proportional to the squares of stress and velocity, it follows that dynamic stress, σ , will be proportional to vibration velocity, v . For idealized straight-beam systems, consisting of thin-walled pipe and with no contents, insulation or concentrated mass, the ratio σ / v is *dependent primarily upon material properties, (density ρ and modulus E), and is remarkably independent of system-specific dimensions, natural-mode number and vibration frequency.* For real continuous systems of course, the kinetic and potential energies are distributed over the structure in accordance with the respective modes shapes. However, *integrated over the structure*, the underlying energy- equality holds true. Provided the spatial distributions are sufficiently similar, i.e. harmonic functions, the RMS or maximum stress will still be directly related to the RMS or maximum vibration velocity.

2.2 The “Screening” Approach

As stated above, for idealized pure beam systems the stress-velocity ratio will depend primarily upon material properties.

For real systems, the spatial patterns of the mode shapes will depart from the idealized harmonic functions, and the *stress-velocity ratios accordingly increase above the theoretical minimum or baseline value.* System details causing the ratios to increase would include the three-dimensional layout, large unsupported masses, high-density contents in thin-walled pipe, susceptible branch connections, changes of cross section, etc. *The more “unfavorable” the system layout and details are, the larger the σ / v ratios for some modes will be.*

Thus, the general susceptibility of a system to large dynamic stresses can be assessed by *determining the extent to which the σ / v ratios for any mode exceed the baseline range.*

Furthermore, by determining which particular modes have the high ratios, and whether these modes are known or likely to be excited, the at-risk vibration frequencies and mode shapes are identified for further assessment and attention. This is the basis of the Stress / Velocity

method of analysis and its implementation as the “dynamic susceptibility” feature in CAEPIPE.

2.3 Relation to Velocity-based Vibration Acceptance Criteria

There are various general and application-specific acceptance criteria based upon vibration velocity as the quantity of record. Some, in order to cover the worst case scenarios, are overly conservative for many systems. Others are presented as being applicable only to the first mode of simple beams, leading to the misconception that the stress / velocity relationship does not apply at all to higher modes. In any case, there are real and perceived limitations on the use of screening acceptance criteria based upon a single value of vibration velocity.

The dynamic susceptibility method turns this apparent limitation into a useful analytical tool! Specifically, large stress / velocity ratios, well above the baseline values, are recognized as a “warning flag.” Large values indicate that some feature(s) of the system make it particularly susceptible to large dynamic stresses in specific modes.

3 What the Dynamic Susceptibility Method Does

3.1 General Approach

The Dynamic Susceptibility method is essentially a post processor to fully exploit the modal analysis results of the system. Mode shape tables of dynamic bending stress and vibration velocity are searched for their respective maxima. Dividing the maximum stress by the maximum velocity yield the “ σ/v ratio” for each mode. That ratio is the basis for assessing the susceptibility to large dynamic stresses. Larger values indicate higher susceptibility associated with specific details of the system.

3.2 Specific Implementation in CAEPIPE

The Stress / Velocity method has been implemented as additional analysis and output of the CAEPIPE modal analysis. The modal analysis load case now includes additional outputs and features as follows:

Dynamic Stresses	This output provides the “mode shapes” of dynamic bending stresses, tabulated along with the conventional mode shape of vibration magnitude.
------------------	--

Dynamic Susceptibility	This output is a table of s/v ratios, in psi / ips, mode by mode, in rank order of decreasing magnitude. In addition to modal frequencies and s/v ratios, the table also includes the node locations of the maxima of vibration amplitude and bending stresses.
------------------------	---

With the dynamic susceptibility output selected, the animated graphic display of the vibration mode shape includes the added feature of color spot markers showing the locations of maximum vibration and maximum dynamic bending stress.

These outputs will assist the designer through a more-complete understanding of the system’s dynamic characteristics. They provide incisive quantified insights into how specific details of components, layout and support could contribute to large dynamic stresses, and into how to make improvements.

4 What the Dynamic Susceptibility Method Does Not Do Directly

The Stress / Velocity method of assessment, and its implementation in CAEPIPE as dynamic susceptibility, is *based entirely upon the system's dynamic characteristics per se*. Thus the vibration velocities and dynamic stresses employed in the analysis, although directly related to each other, are of *arbitrary magnitude*. There is no computation of the response to a prescribed forcing function, and no attempt to calculate actual dynamic stresses. Thus the dynamic susceptibility results ***do not factor directly into a pass-fail code compliance consideration***. Rather, they assist the designer to assess and reduce susceptibility to large dynamic stresses if necessary, in order to meet whatever requirements have been specified.

An example follows next.

5 Illustrative Example of the Dynamic Susceptibility Analysis

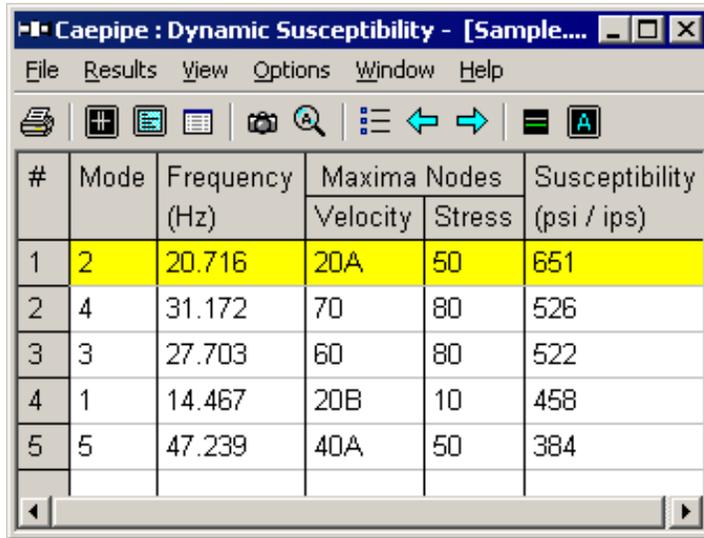
The “dynamic susceptibility” feature of CAEPIPE will be illustrated here by application to the standard CAEPIPE Example system. The modal analysis was performed for frequencies up to 200 Hz, resulting in a reporting-out for 12 modes.

#	Frequency (Hz)	Period (second)	Participation factors			Modal mass / Total mass		
			X	Y	Z	X	Y	Z
1	14.52	0.0689	0.0232	-1.7571	-0.7834	0.0001	0.5112	0.1016
2	20.779	0.0481	-0.0004	0.5862	-1.9915	0.0000	0.0569	0.6567
3	27.750	0.0360	0.0826	0.4566	0.0823	0.0011	0.0345	0.0011
4	31.229	0.0320	-0.1751	-0.0063	0.110	0.0051	0.0000	0.0020
5	47.406	0.0211	-1.0989	-0.1355	0.0559	0.2000	0.0030	0.0005
6	52.422	0.0191	0.1787	-0.3770	0.3007	0.0053	0.0235	0.0150
7	129.69	0.0077	-0.0446	-0.1827	0.0125	0.0003	0.0055	0.0000
8	132.987	0.0075	0.1195	0.0090	-0.1265	0.0024	0.0000	0.0026
9	138.665	0.0072	1.4714	-0.0020	0.0228	0.3585	0.0000	0.0001
10	163.986	0.0061	-0.0379	-0.3694	0.1358	0.0002	0.0226	0.0031
11	173.053	0.0058	0.4440	-0.2272	0.1155	0.0326	0.0085	0.0022
12	191.852	0.0052	0.2435	0.1143	-0.0572	0.0098	0.0022	0.0005

The frequencies range from mode 1 at 14.5 Hz to mode 12 at 192 Hz. In two instances, very similar horizontal and vertical modes appear in pairs, i.e. modes 3 & 4 and 7 & 8.

The relevant features of this system can be readily identified and understood, by reference to the dynamic-susceptibility table and the animated graphic display of mode shape. Results will be considered here in the order of decreasing susceptibility.

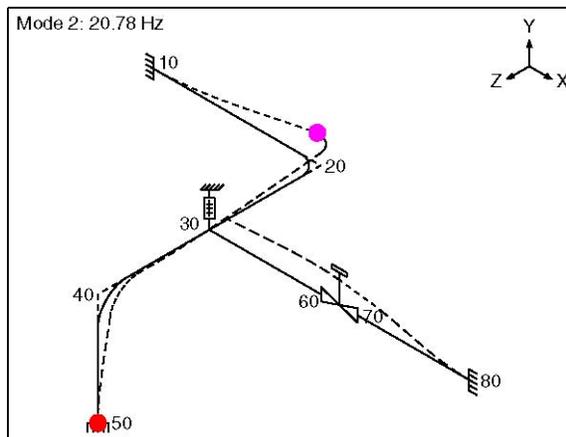
Dynamic Susceptibility Method



#	Mode	Frequency (Hz)	Maxima Nodes		Susceptibility (psi / ips)
			Velocity	Stress	
1	2	20.716	20A	50	651
2	4	31.172	70	80	526
3	3	27.703	60	80	522
4	1	14.467	20B	10	458
5	5	47.239	40A	50	384

5.1 Axial movement of long pipe run (large added mass in motion)

From the dynamic susceptibility table, the top of the list is mode 2 at 20.8 Hz, having a dynamic susceptibility of 649 psi / ips.

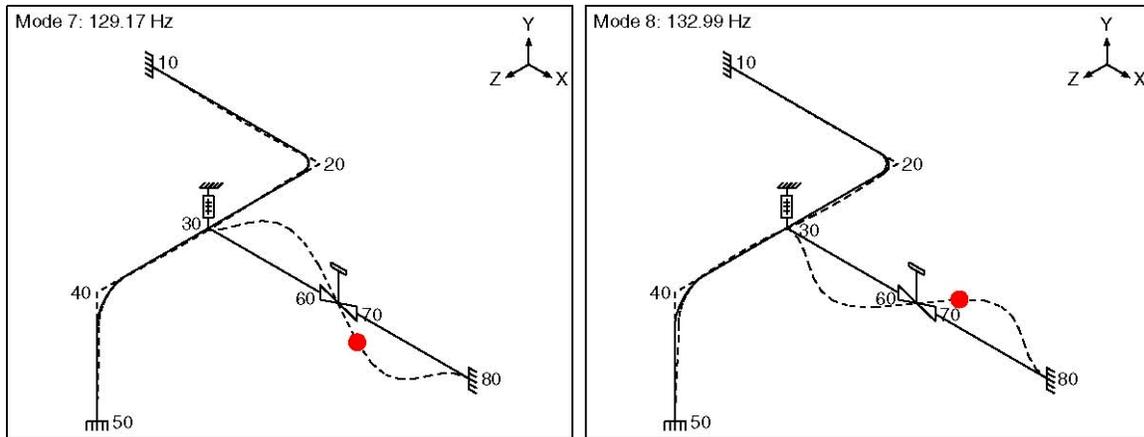


From the animated graphic display, note that the maximum dynamic bending stresses are at the anchored point, node 50. Note also that the dominant motion is a “Z” motion of the straight run between nodes 20 and 40 (i.e., in effect an axial motion of that run as a rigid body). The designer’s interpretation here is that the vertical rise from node 50 to node 40 is effectively a cantilevered beam with an effective large added mass at the tip; that feature of layout accounts for the high susceptibility.

5.2 Effects associated with the valve (local rigidity to bending, and added mass)

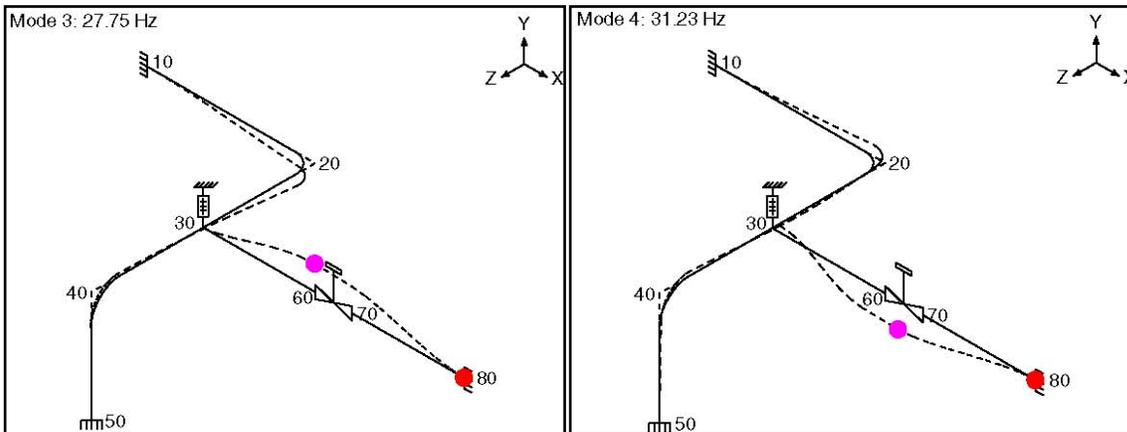
The next-highest values of susceptibility are for the two pairs of modes, modes 7 & 8 at 129 and 133 Hz, and modes 3 & 4 at 27.8 and 31.2 Hz. As will be shown here, these are associated with effects of the valve,

Dynamic Susceptibility Method



The susceptibility for modes 7 & 8, respectively 594 and 589 psi / ips, is attributable to the rigidity of the valve element within an otherwise flexible pipe run. This can be seen from a close look at the animated graphic. Notice that these relatively high frequency modes feature a reversal of bending curvature along the run between nodes 30 and 80. Notice also that there is a stronger localized curvature on approach to the valve body. The designer's interpretation here is that, since there cannot be any curvature of the rigid valve itself, there must be a more concentrated curvature of the adjacent pipe.

The dynamic susceptibility of modes 3 & 4, respectively 521 and 526 psi / ips, is associated with the more straightforward “concentrated mass” effect of the valve.

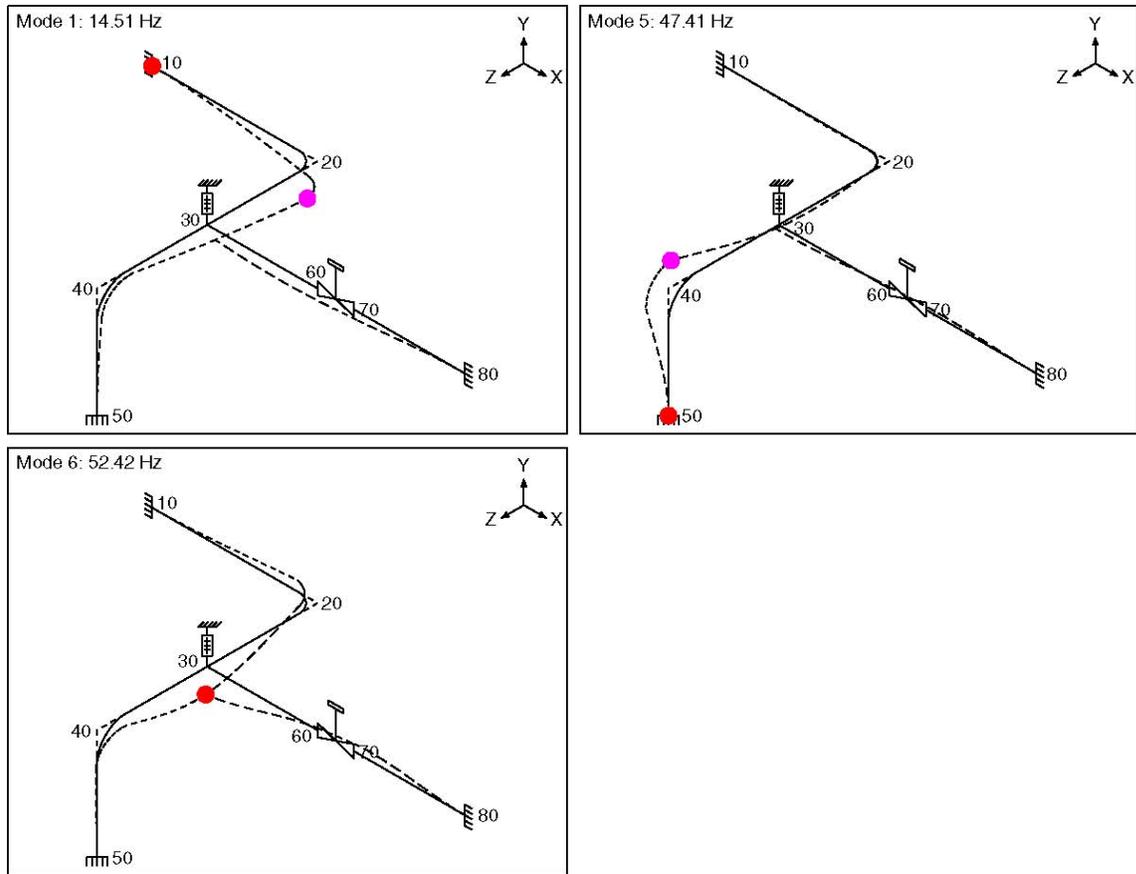


From the animated graphic, these modes feature a large amplitude vibration at the valve. The kinetic energy of this added mass must be stored as strain energy in the flexing (i.e., spring) element, resulting in elevated dynamic stresses.

5.3 Beam modes with “moderate” added mass effects of adjacent spans

Modes 1,5 and 6, with frequencies of 14.5, 47.4 and 52.4 Hz, show progressively decreasing “intermediate to low” values of susceptibility at respectively 456, 383 and 339 psi / ips.

Dynamic Susceptibility Method

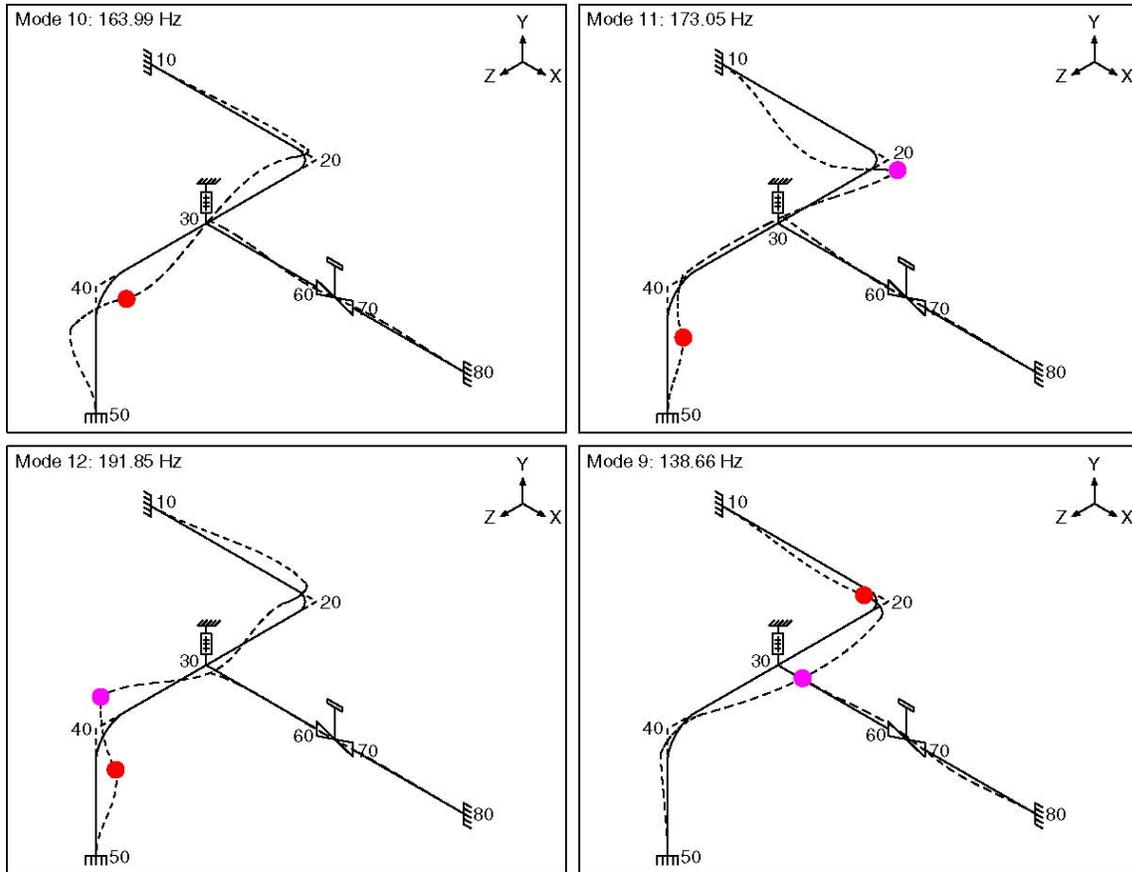


Reference to the animated graphics shows that these modes involve predominantly *transverse* vibration (as contrasted with the prominent *axial* movement of mode 2) and involve *little participation at the valve* (which accounted for the elevated susceptibility of modes 7 & 8 and 3 & 4). Notice that these modes, 1, 5, and 6, involve varying degrees of the influence of effective added mass of adjacent spans, and of length of the cantilevered span contributing most to stiffness.

5.4 Modes approaching the “simple-beam baseline” behavior

Modes 10, 11 and 12 have significantly higher frequencies, 164 to 192 Hz, and correspondingly short wavelengths. Consequently, the vibration pattern tends to be transverse beam vibration “within the span,” with little or no effect from connected spans or the valve. For these modes, the susceptibility ratios range from 256 to 272 psi / ips. These values are approaching the baseline values for uncomplicated mode shapes of the pipe section and pipe contents of this system.

Dynamic Susceptibility Method



NOTE: Mode 9, at 138 Hz, is clearly an exception, with a susceptibility of only 104 psi / ips, well below the baseline level. From the animated display, it can be seen that this is not really a “bending” mode; rather, the spring effect for this mode is an axial stretching of the run between nodes 80 and 30. Consequently, the bending stresses are low, as reflected in the abnormal susceptibility ratio. In effect, this mode lies outside the intended application of the dynamic susceptibility approach. Notice however, that the low susceptibility ratio has in effect “flagged” this mode as “not a bending mode”; that in itself provides the designer additional insight into system characteristics and behavior.

5.5 Summary Comment

As per paragraphs 5.1 to 5.4, the dynamic susceptibility method has incisively identified the key features of the Sample model, with respect to potentially large dynamic stresses. This of course is a relatively simple system. An experienced designer, with some appreciation of dynamics, might view the results as obvious. However, the method will do the same job, automatically and directly, on any larger or more complex system for which nothing is obvious!

6 Summary of “Dynamic Susceptibility” Analytical Capability

The stress / velocity method, implemented in CAEPIPE as the “Dynamic Susceptibility” feature, provides quantified insights into the stress versus vibration characteristics of the system layout per se.

In particular, the dynamic susceptibility table identifies specific modes that are susceptible to large dynamic stresses for a given level of vibration. The larger the stress / velocity ratio, the

stronger the indication that some particular feature of layout, mass distribution, supports, stress raisers, etc., is causing susceptibility to large dynamic stresses.

The animated mode-shape display identifies, by the color-spot-markers, the locations of the respective maxima in dynamic stress and vibration velocity. Review of these animated plots will reveal the offending pattern of motion, and provide immediate insight into what features of the system are responsible for the large dynamic stresses.

Finally, the “dynamic stresses” table provides the distribution of dynamic stresses around the system, i.e., in effect, the mode shape of dynamic stresses to go along with the conventional mode shape of vibration. This information allows identification of other parts of the system, if any, with dynamic stresses comparable to the identified maximum.

7 Suggested Applications and Associated Benefits

7.1 At the Design Stage

At the design stage, the dynamic susceptibility feature allows the designer to quickly determine whether the system may be susceptible to very large dynamic stresses. This could be a broad look at all frequencies, or *could be focused on particular frequencies where excitation is likely to occur*. On identifying high susceptibility, the designer can then make changes to improve the design. It is important to note that this method is based upon the dynamic-stress versus vibration-velocity characteristics of the system per se. There is no need to specify a forcing function and perform a response calculation and stress / fatigue analysis. However, *where such analysis is a requirement*, the dynamic susceptibility module can assist the designer to *achieve a system layout that will meet the requirements and criteria*.

7.2 Commissioning, Acceptance Testing

The dynamic susceptibility feature can also contribute to planning acceptance testing and associated measurements where these are undertaken whether by formal requirement or by choice. Locations for measurement of vibration or dynamic strain can be selected based upon knowing the locations of the maxima and the distribution of vibration and dynamic stress. Reference to the dynamic susceptibility results can *help assure that the modes of most potential concern are well covered by the minimum set of practically-achievable measurements*. Furthermore, mode-specific acceptance criteria can be readily established to *avoid the restrictions of generally over-conservative guideline type criteria, while providing assurance that any highly-susceptible situations are identified and addressed*.

7.3 Troubleshooting and Correction

As mentioned earlier, when vibration and/or fatigue problems are recognized at start up or early operation, there is typically an ad hoc program of observation, measurement, assessment, diagnosis and correction. It is not uncommon for there to be some uncertainty about what to measure and what is acceptable. The dynamic susceptibility module can contribute very effectively in these situations.

Normally, the overall symptoms, approximate frequency and pattern of vibration are known to some extent from observation and/or a few measurements. After modeling the system, and obtaining the dynamic susceptibility results, the subsequent steps can be *highly focused on specific frequencies and locations*, the optimum measurements, and system-specific acceptance criteria.

Dynamic Susceptibility Method

Equally or more importantly, the *proposed solution options can be modeled and evaluated* to make sure they will achieve the required improvement.

7.4 General

The dynamic susceptibility module *does not apply directly to meeting code or other formal stress analysis requirements*. However, it is an incisive analytical tool to help the designer understand the stress / vibration relationship, assess the situation and to decide how to modify the design if necessary. It can be used for design, planning acceptance tests, and troubleshooting and correction.

8 Information for Reference

The Stress /Velocity method for screening piping system modes was developed and brought to the attention of SST Systems by Dr. R.T Hartlen of Plant Equipment Dynamics Inc.

The background material provided here is intended to provide only a concise summary of the underlying fundamentals, the universality for idealized systems, and the expected detail-dependent variations for real systems. The stress / velocity method, although not yet widely known and applied, is fundamentally theoretically sound. However, complete theoretical rigor is beyond the scope of this note.

For users who may wish to independently examine and validate the underlying theoretical fundamentals, a few key references are provided. References 1, 2 and 3 deal with fundamentals. References 4 and 5 deal with application to piping. The CEA research projects reported in References 3 and 4 were initiated and guided by Dr. Hartlen.

References

1. F.V. Hunt, Stress and Strain Limits on the Attainable Velocity in Mechanical Systems, JASA, 32(9) 1123-1128, 1960
1. E.E. Ungar, Maximum Stresses in Beams and Plates Vibrating at Resonance, ASME Journal of Engineering for Industry, v84, n1, pp149-155, 1962
2. R. Elmaraghy et al, Correlation of Vibratory Stress, Velocity and Sound, Canadian Electrical Association Project, G197, Feb 1982
3. J.D. Tulk, Correlation Between Dynamic Stress and Vibration Velocity in Complex Piping Systems, Canadian Electrical Association Project G521, March 1988
4. Michael P. Norton, Acoustically Induced Structural Vibration and Fatigue - A Review,
5. Third International Congress on Air-and Structure-borne Sound and Vibration, June 1994, Montreal, Canada

This page is blank

Appendix F

Flange Module

Flange Module

Notation based on ASME Section VIII, Division 1, Appendix 2

The symbols described below are used in the formulas for the design of flanges

A = outside diameter of flange

A_b = cross-sectional area of the bolts using the root diameter of the thread

A_m = total required cross-sectional area of bolts taken as greater of A_{m1} and A_{m2}

A_{m1} = total cross-sectional area of bolts at root of thread or section of least diameter under stress, required for the operating conditions

$$= \frac{W_{m1}}{S_b}$$

A_{m2} = total cross-sectional area of bolts at root of thread or section of least diameter under stress

$$= \frac{W_{m2}}{S_a}$$

B = inside diameter of flange

B' = inside diameter of reverse flange

b = effective gasket or joint-contact-surface seating width

b_0 = basic gasket seating width (from Table 2-5.2)

C = bolt-circle diameter

c = basic dimension used for the minimum sizing of welds

e = factor = $\frac{F}{h_o}$

d = factor = $\frac{U}{V} h_o g_0^2$ for integral type flanges

d = factor = $\frac{U}{V_L} h_o g_0^2$ for loose type flanges

e = factor = $\frac{F}{h_o}$ for integral type flanges

c = factor = $\frac{F_L}{h_o}$ for loose type flanges

F = factor for integral type flanges (from Fig. 2-7.2)

F_L = factor for loose type flanges (from Fig. 2-7.4)

f = hub stress correction for integral flanges from Fig. 2-7.6 (when greater than one, this is the ratio of the stress in the small end of hub to the stress in the large end), (for values below limit of figure, use $f = 1$.)

G = diameter at location of gasket load reaction

g_0 = thickness of hub at small end

g_1 = thickness of hub at back of flange

H = total hydrostatic end force = $0.785G^2P$

H_D = hydrostatic end force on area inside of flange = $0.785B^2P$

H_G = gasket load (difference between flange design bolt load and total hydrostatic end force) = $W - H$

H_P = total joint-contact surface compression load = $2b \times 3.14 G_m P$

H_T = difference between total hydrostatic end force and the hydrostatic end force on area inside of flange = $H - H_D$

h = hub length

Flange Module

- h_D = radial distance from the bolt circle, to the circle on which H_D acts, as prescribed in Table 2-6
 h_G = radial distance from gasket load reaction to the bolt circle = $\frac{C-G}{2}$
 h_O = factor = $\sqrt{B_{g0}}$
 h_T = distance from the bolt circle, to the circle on which H_T acts, as prescribed in Table 2-6
 K = ratio of outside diameter of flange to inside diameter of flange = A / B
 L = factor = $\frac{(t_e+1)}{T} + \frac{t^3}{d}$
 M_D = component of moment due to $H_D = H_D h_D$
 M_G = component of moment due to $H_G = H_G h_G$
 M_T = component of moment due to $H_T = H_T h_T$
 M_O = total moment acting upon the flange for the operating conditions or gasket seating as may apply
 $M_O = W \frac{(C-G)}{2}$ for gasket seating condition
 $M_O = H_D h_D + H_G h_G + H_T h_T$ for operating condition
 N = width used to determine the basic gasket seating with b_0 , based upon the possible contact width of the gasket (see Table 2-5.2)
 P = internal design pressure
 R = radial distance from bolt circle to point of intersection of hub and back of flange. For integral and hub flanges, $R = (C-B / 2) - g_1$
 S_a = allowable bolt stress at reference temperature
 S_b = allowable bolt stress at design temperature
 S_f = allowable stress for material of flange at design temperature (operating condition)
 S_H = calculated longitudinal stress in hub
 S_R = calculated radial stress in flange
 S_T = calculated tangential stress in flange
 T = factor involving K (from Fig. 2-7.1)
 t = flange thickness
 U = factor involving K (from Fig. 2-7.1)
 V = factor for integral type flanges (from Fig. 2-7.3)
 V_L = factor for loose type flanges (from Fig. 2-7.5)
 W = flange design bolt load, for the operating condition or gasket seating
 = W_{m1} for operating condition
 = $\frac{(A_m + A_b) S_a}{2}$ for gasket seating condition
 W_{m1} = minimum required bolt load for the operating conditions
 W_{m1} = minimum required bolt load for gasket seating
 w = width used to determine the basic gasket seating width b_0 , based upon the contact width between the flange and the gasket (see Table 2-5.2)
 Y = factor involving K (from Fig. 2-7.1)
 y = gasket or joint-contact-surface unit seating load
 Z = factor involving K (from Fig. 2-7.1)

Calculation of Flange Stresses

The stresses in the flange shall be determined for both the operating conditions and gasket seating condition, in accordance with the following formulas:

(1) Integral type flanges

$$\text{Longitudinal hub stress } S_H = \frac{fM_0}{Lg_1^2B} < 1.5S_f$$

$$\text{Radial flange stress } S_R = \frac{(1.33te+1)M_0}{Lt^2B} < S_f$$

$$\text{Tangential flange stress } S_T = \frac{YM_0}{t^2B} - ZS_R < S_f$$

Combined stress

$$\frac{S_H + S_R}{2} < S_f \text{ and}$$

$$\frac{S_H + S_T}{2} < S_f$$

(2) Loose type flanges with hubs

$$\text{Longitudinal hub stress } S_H = \frac{fM_0}{Lg_1^2B} < 1.5S_f$$

$$\text{Radial flange stress } S_R = \frac{(1.33te+1)M_0}{Lt^2B} < S_f$$

$$\text{Tangential flange stress } S_T = \frac{YM_0}{t^2B} - ZS_R < S_f$$

Combined stress

$$\frac{S_H + S_R}{2} < S_f \text{ and}$$

$$\frac{S_H + S_T}{2} < S_f$$

where,

$$L = \text{factor} = \frac{(te+1)}{T} + \frac{t^3}{d}$$

$$d = \text{factor} = \frac{U}{V_L} h_o g_0^2 \text{ for integral type flanges}$$

$$e = \text{factor} = \frac{F_L}{h_o}$$

V_L = factor for loose type flanges (from Fig. 2-7.5)

F_L = factor for loose type flanges (from Fig. 2-7.4)

(3) Loose type flanges without hubs

$$\text{Longitudinal hub stress } S_H = 0$$

$$\text{Radial flange stress } S_R = 0$$

$$\text{Tangential flange stress } S_T = \frac{YM_0}{t^2B} < S_f$$

Combined stress

$$\frac{S_H + S_R}{2} < S_f \text{ and}$$

$$\frac{S_H + S_T}{2} < S_f$$

where,

$$L = \text{factor} = \frac{(te+1)}{T} + \frac{t^3}{d}$$

$$d = \text{factor} = \frac{U}{V_L} h_0 g_0^2$$

$$e = \text{factor} = \frac{F_L}{h_0}$$

V_L = factor for loose type flanges (from Fig. 2-7.5)

F_L = factor for loose type flanges (from Fig. 2-7.4)

(4) Optional type flanges (calculated as loose flanges without hubs)

Longitudinal hub stress $S_H = 0$

Radial flange stress $S_R = 0$

Tangential flange stress $S_T = \frac{YM_0}{t^2 B} < S_f$

Combined stress

$$\frac{S_H + S_R}{2} < S_f \text{ and}$$

$$\frac{S_H + S_T}{2} < S_f$$

where,

$$L = \text{factor} = \frac{(te+1)}{T} + \frac{t^3}{d}$$

$$d = \text{factor} = \frac{U}{V_L} h_0 g_0^2$$

$$e = \text{factor} = \frac{F_L}{h_0}$$

V_L = factor for loose type flanges (from Fig. 2-7.5)

F_L = factor for loose type flanges (from Fig. 2-7.4)

(5) Reverse type flanges

$$\text{Longitudinal hub stress } S_H = \frac{fM_0}{L_r g_1^2 B'} < 1.5S_f$$

$$\text{Radial flange stress } S_R = \frac{(1.33te_r+1)M_0}{L_r t^2 B'} < S_f$$

$$\text{Tangential flange stress } S_r = \frac{YM_0}{t^2 B} - ZS_R \frac{0.67te_r+1}{1.33te_r+1} < S_f$$

Combined stress

$$\frac{S_H+S_R}{2} < S_f \text{ and}$$

$$\frac{S_H + S_T}{2} < S_f$$

where,

$$L_r = \text{factor} = \frac{(te_r+1)}{T} + \frac{t^3}{d_r}$$

$$d_r = \text{factor} = \frac{U_r}{V} h_o g_0^2$$

$$e_r = \text{factor} = \frac{F}{h_o}$$

$$h_o = \text{factor} = \sqrt{Ag_0}$$

$$\alpha_r = \frac{1+0.668\frac{(K+1)}{Y}}{K^2}$$

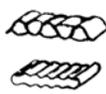
$$T_r = \frac{Z+0.3}{Z-0.3} \alpha_r T$$

$$U_r = \alpha_Y U$$

$$Y_r = \alpha_Y Y$$

Flange Module

TABLE 2-5.1
GASKET MATERIALS AND CONTACT FACINGS¹
 Gasket Factors m for Operating Conditions and Minimum Design Seating Stress y

Gasket Material	Gasket Factor m	Min. Design Seating Stress y , psi (MPa)	Sketches	Facing Sketch and Column in Table 2-5.2
Self-energizing types (O rings, metallic, elastomer, other gasket types considered as self-sealing)	0	0 (0)
Elastomers without fabric or high percent of asbestos fiber: Below 75A Shore Durometer	0.50	0 (0)		(1a),(1b),(1c),(1d), (4),(5); Column II
75A or higher Shore Durometer	1.00	200 (1.4)		
Asbestos with suitable binder for operating conditions: $\frac{1}{8}$ in. (3.2 mm) thick	2.00	1,600 (11)		(1a),(1b),(1c),(1d), (4),(5); Column II
$\frac{1}{16}$ in. (1.6 mm) thick	2.75	3,700 (26)		
$\frac{1}{32}$ in. (0.8 mm) thick	3.50	6,500 (45)		
Elastomers with cotton fabric insertion	1.25	400 (2.8)		(1a),(1b),(1c),(1d), (4),(5); Column II
Elastomers with asbestos fabric insertion (with or without wire reinforcement):				
3-ply	2.25	2,200 (15)		(1a),(1b),(1c),(1d), (4),(5); Column II
2-ply	2.50	2,900 (20)		
1-ply	2.75	3,700 (26)		
Vegetable fiber	1.75	1,100 (7.6)		(1a),(1b),(1c),(1d), (4),(5); Column II
Spiral-wound metal, asbestos filled:				
Carbon	2.50	10,000 (69)		(1a),(1b); Column II
Stainless, Monel, and nickel-base alloys	3.00	10,000 (69)		
Corrugated metal, asbestos inserted, or corrugated metal, jacketed asbestos filled:				
Soft aluminum	2.50	2,900 (20)		(1a),(1b); Column II
Soft copper or brass	2.75	3,700 (26)		
Iron or soft steel	3.00	4,500 (31)		
Monel or 4%–6% chrome	3.25	5,500 (38)		
Stainless steels and nickel-base alloys	3.50	6,500 (45)		

Flange Module

TABLE 2-5.1
GASKET MATERIALS AND CONTACT FACINGS¹ (CONT'D)
 Gasket Factors m for Operating Conditions and Minimum Design Seating Stress y

Gasket Material	Gasket Factor m	Min. Design Seating Stress y , psi (MPa)	Sketches	Facing Sketch and Column in Table 2-5.2
Corrugated metal:				
Soft aluminum	2.75	3,700 (26)		(1a),(1b),(1c),(1d); Column II
Soft copper or brass	3.00	4,500 (31)		
Iron or soft steel	3.25	5,500 (38)		
Monel or 4%–6% chrome	3.50	6,500 (45)		
Stainless steels and nickel-base alloys	3.75	7,600 (52)		
Flat metal, jacketed asbestos filled:				
Soft aluminum	3.25	5,500 (38)		(1a),(1b),(1c), ² (1d) ² ;(2) ² ; Column II
Soft copper or brass	3.50	6,500 (45)		
Iron or soft steel	3.75	7,600 (52)		
Monel	3.50	8,000 (55)		
4%–6% chrome	3.75	9,000 (62)		
Stainless steels and nickel-base alloys	3.75	9,000 (62)		
Grooved metal:				
Soft aluminum	3.25	5,500 (38)		(1a),(1b),(1c),(1d), (2),(3); Column II
Soft copper or brass	3.50	6,500 (45)		
Iron or soft metal	3.75	7,600 (52)		
Monel or 4%–6% chrome	3.75	9,000 (62)		
Stainless steels and nickel-base alloys	4.25	10,100 (70)		
Solid flat metal:				
Soft aluminum	4.00	8,800 (61)		(1a),(1b),(1c),(1d), (2),(3),(4),(5); Column I
Soft copper or brass	4.75	13,000 (90)		
Iron or soft steel	5.50	18,000 (124)		
Monel or 4%–6% chrome	6.00	21,800 (150)		
Stainless steels and nickel-base alloys	6.50	26,000 (180)		
Ring joint:				
Iron or soft steel	5.50	18,000 (124)		(6); Column I
Monel or 4%–6% chrome	6.00	21,800 (150)		
Stainless steels and nickel-base alloys	6.50	26,000 (180)		

NOTES:

(1) This Table gives a list of many commonly used gasket materials and contact facings with suggested design values of m and y that have generally proved satisfactory in actual service when using effective gasket seating width b given in Table 2-5.2. The design values and other details given in this Table are suggested only and are not mandatory.

(2) The surface of a gasket having a lap should not be against the nubbin.

Flange Module

TABLE 2-5.2
EFFECTIVE GASKET WIDTH²

Facing Sketch (Exaggerated)		Basic Gasket Seating Width b_o	
		Column I	Column II
(1a)			
(1b)	<p style="text-align: center;">See Note (1)</p>	$\frac{N}{2}$	$\frac{N}{2}$
(1c)	<p style="text-align: center;">$w \leq N$</p>	$\frac{w + T}{2}; \left(\frac{w + N}{4} \text{ max} \right)$	$\frac{w + T}{2}; \left(\frac{w + N}{4} \text{ max} \right)$
(1d)	<p style="text-align: center;">See Note (1)</p> <p style="text-align: center;">$w \leq N$</p>		
(2)	<p style="text-align: center;">$\frac{1}{64}$ in. (0.4 mm) nubbin</p> <p style="text-align: center;">$w \leq N/2$</p>	$\frac{w + N}{4}$	$\frac{w + 3N}{8}$
(3)	<p style="text-align: center;">$\frac{1}{64}$ in. (0.4 mm) nubbin</p> <p style="text-align: center;">$w \leq N/2$</p>	$\frac{N}{4}$	$\frac{3N}{8}$
(4)	<p style="text-align: center;">See Note (1)</p>	$\frac{3N}{8}$	$\frac{7N}{16}$
(5)	<p style="text-align: center;">See Note (1)</p>	$\frac{N}{4}$	$\frac{3N}{8}$
(6)		$\frac{w}{8}$...

TABLE 2-6
MOMENT ARMS FOR FLANGE LOADS UNDER
OPERATING CONDITIONS

	h_D	h_T	h_G
Integral type flanges [see Fig. 2-4 sketches (5), (6), (6a), (6b), and (7)]; and optional type flanges calculated as integral type [see Fig. 2-4 sketches (8), (8a), (9), (9a), (10), (10a), and (11)]	$R + 0.5g_1$	$\frac{R + g_1 + h_G}{2}$	$\frac{C - G}{2}$
Loose type, except lap-joint flanges [see Fig. 2-4 sketches (2), (2a), (3), (3a), (4), and (4a)]; and optional type flanges calculated as loose type [see Fig. 2-4 sketches (8), (8a), (9), (9a), (10), (10a), and (11)]	$\frac{C - B}{2}$	$\frac{h_D + h_G}{2}$	$\frac{C - G}{2}$
Lap-type flanges [see Fig. 2-4 sketches (1) and (1a)]	$\frac{C - B}{2}$	$\frac{C - G}{2}$	$\frac{C - G}{2}$

Flange Module

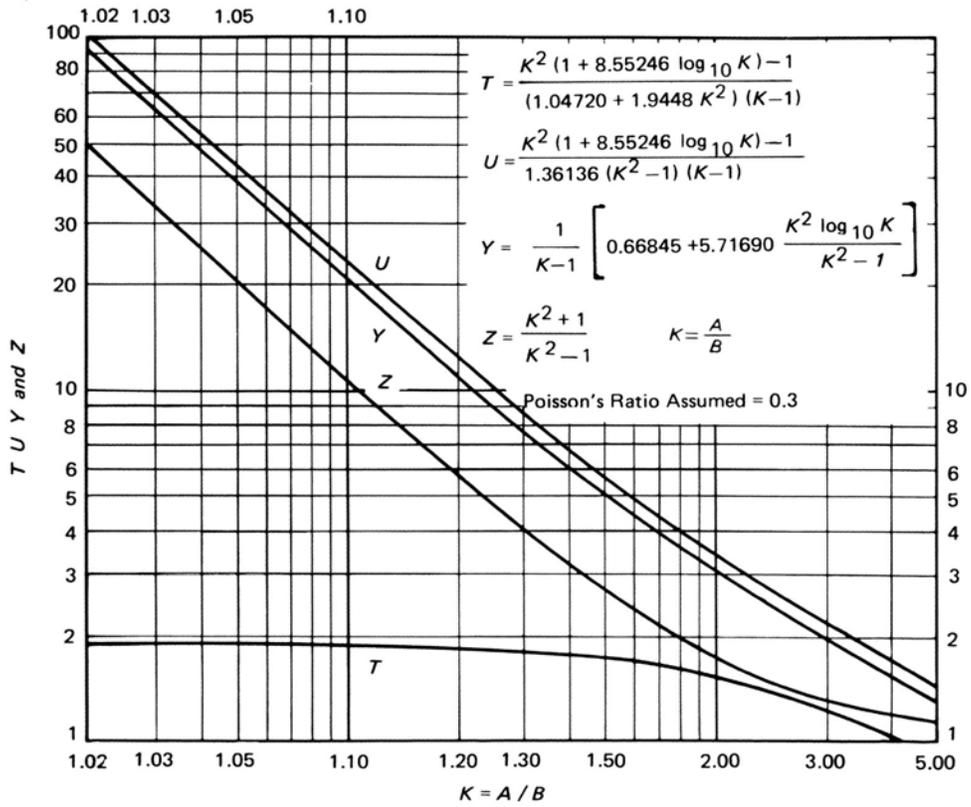


FIG. 2-7.1 VALUES OF T , U , Y , AND Z
(Terms Involving K)

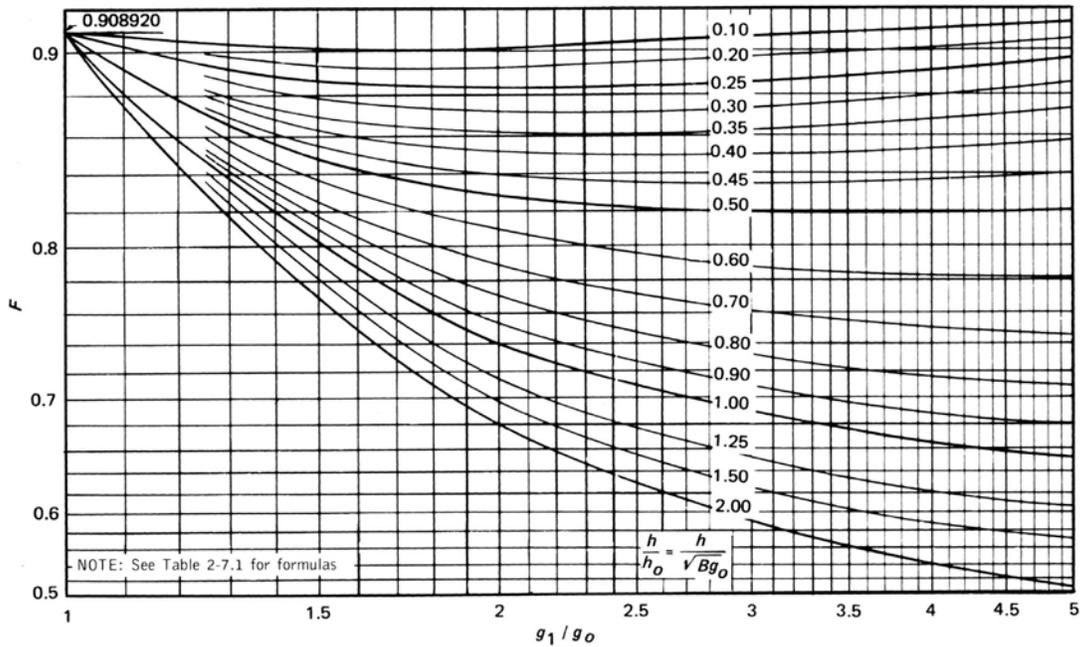


FIG. 2-7.2 VALUES OF F
(Integral Flange Factors)

Flange Module

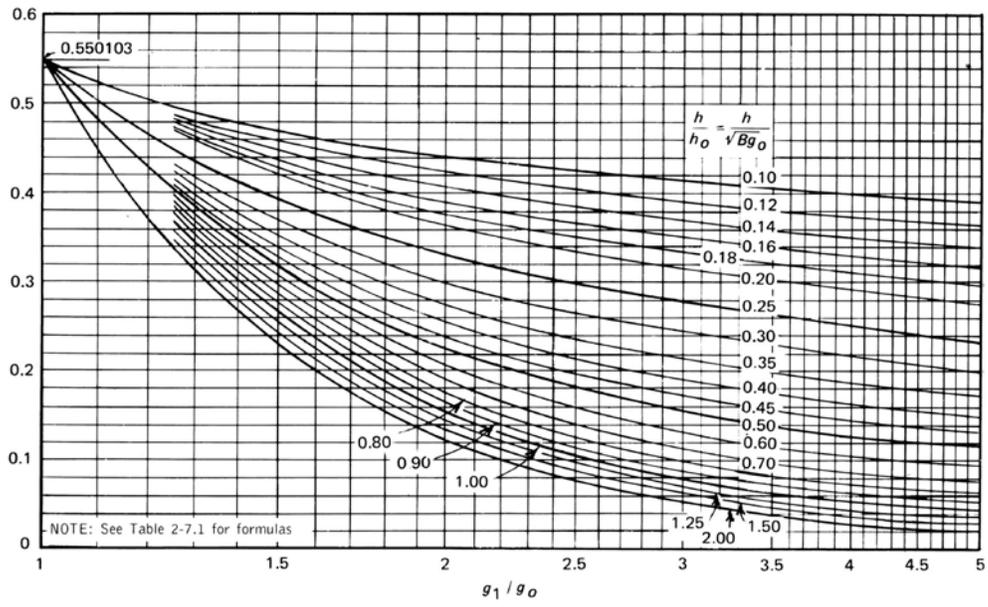


FIG. 2-7.3 VALUES OF V
(Integral Flange Factors)

Flange Module

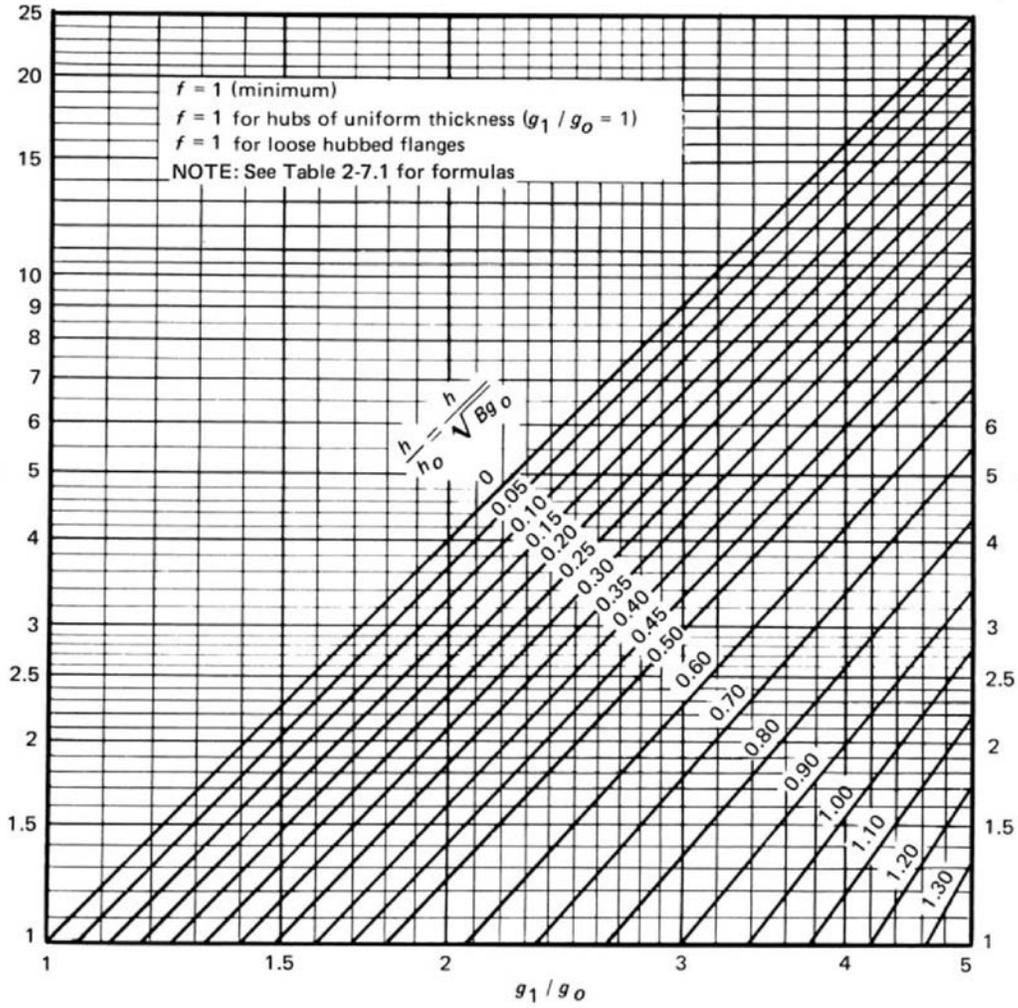


FIG. 2-7.6 VALUES OF f
(Hub Stress Correction Factor)

Problem 2:

(Example from “Taylor Forge & Pipe Works, 1961”)

Flange Details:

Flange Type : Loose Flanges with Hubs
Flange Outside Diameter [A] = 40.375 (inch)
Flange Inside Diameter [B] = 33.25 (inch)
Inside Diameter of Reverse Flange [B'] = 20 (inch)
Flange Thickness [t] = 2.125 (inch)
Small End Hub Thickness [g0] = 0.875 (inch)
Large End Hub Thickness [g1] = 1.125 (inch)
Hub Length [h] = 2.5 (inch)
All. Stress @ Design Temp [sf] = 17500 (psi)
All. Stress @ Ref. Temp [sfa] = 17500 (psi)
Modulus @ Design Temp [E] = 2.7E+7 (psi)
Modulus @ Ref. Temp [Ea] = 2.92E+7 (psi)

Bolting Information:

Bolt Circle Diameter = 38.25 (inch)
Number of Bolts = 44
Bolt Diameter = 1 (inch)
All. Stress @ Ref. Temp [sa] = 20000 (psi)
All. Stress @ Design Temp [sb] = 20000 (psi)

Gasket Information:

Gasket Outside Diameter = 35.75 (inch)
Gasket Inner Diameter = 34.25 (inch)
Leak Pressure Ratio [m] = 2.75
Gasket Seating Stress [y] = 3700 (psi)
Facing Sketch = 1
Facing Column = 1

Load Data:

Design Pressure = 400 (psi)
Design Temperature = 500 (F)
Axial Force = 1000 (lb)
Bending Moment = 200 (ft-lb)
Allowable Pressure = 665 (psi)

Flange Module

Comparison of Results:

Flange Stresses	Text Book Results (psi)	CAEPIPE (psi)	CAESAR II (psi)
Operating condition			
Longitudinal Hub (SH)	20800	21153	21214
Radial Flange (SR)	11100	11110	11155
Tangential Flange (ST)	13800	13826	13797
0.5(SH + SR)	15950	16132	16185
0.5(SH + ST)	17300	17489	17506
Gasket Seating Condition			
Longitudinal Hub (SH)	14400	14623	15095
Radial Flange (SR)	7660	7681	7938
Tangential Flange (ST)	9500	9558	9818
0.5(SH + SR)	11030	11152	11517
0.5(SH + ST)	11950	12091	12457

Problem 3:

(Example 10.5 on page 209 Chapter 10 on “CASTI Guidebook to ASME Section VIII Div.1 – Pressure Vessels – Third Edition”)

Flange Details:

Flange Type : Reverse Flanges
 Flange Outside Diameter [A] = 49 (inch)
 Flange Inside Diameter [B] = 48.25 (inch)
 Inside Diameter of Reverse Flange [B'] = 20.25 (inch)
 Flange Thickness [t] = 5.25 (inch)
 Small End Hub Thickness [g0] = 0.375 (inch)
 Large End Hub Thickness [g1] = 1.375 (inch)
 Hub Length [h] = 6 (inch)
 All. Stress @ Design Temp [sf] = 12000 (psi)
 All. Stress @ Ref. Temp [sfa] = 20000 (psi)
 Modulus @ Design Temp [E] = 2.7E+7 (psi)
 Modulus @ Ref. Temp [Ea] = 2.92E+7 (psi)

Bolting Information:

Bolt Circle Diameter = 44 (inch)
 Number of Bolts = 32
 Bolt Diameter = 1.25 (inch)
 All. Stress @ Ref. Temp [sa] = 25000 (psi)
 All. Stress @ Design Temp [sb] = 21000 (psi)

Gasket Information:

Gasket Outside Diameter = 24 (inch)
 Gasket Inner Diameter = 22 (inch)
 Leak Pressure Ratio [m] = 2.50
 Gasket Seating Stress [y] = 10000 (psi)
 Facing Sketch = 1

Flange Module

Facing Column = 1

Load Data:

Design Pressure = 150 (psi)

Design Temperature = 800 (F)

Axial Force = 1000 (lb)

Bending Moment = 200 (ft-lb)

Allowable Pressure = 665 (psi)

Comparison of Results:

Flange Stresses	Text Book Results (psi)	CAEPIPE (psi)	CAESAR II (psi)
Operating condition			
Longitudinal Hub (SH)	2060	2257	2055
Radial Flange (SR)	280	307	280
Tangential Flange (ST)	1340	1314	1336
0.5(SH + SR)	1170	1282	1168
0.5(SH + ST)	1700	1785	1695
Gasket Seating Condition			
Longitudinal Hub (SH)	9220	10082	9220
Radial Flange (SR)	1260	1372	1257
Tangential Flange (ST)	6000	7004	5997
0.5(SH + SR)	5240	5727	5239
0.5(SH + ST)	7610	8543	8703

Problem 4:

(Example from KEDKEP CONSULTING, INC. dated May 27, 2008)

Flange Details:

Flange Type : Loose Flanges without Hubs / Optional Flanges

Flange Outside Diameter [A] = 38.4 (inch)

Flange Inside Diameter [B] = 32 (inch)

Inside Diameter of Reverse Flange [B'] = 32 (inch)

Flange Thickness [t] = 4 (inch)

Small End Hub Thickness [g0] = 0.001 (inch)

Large End Hub Thickness [g1] = 0.001 (inch)

Hub Length [h] = 0.001 (inch)

All. Stress @ Design Temp [sf] = 20000 (psi)

All. Stress @ Ref. Temp [sfa] = 20000 (psi)

Modulus @ Design Temp [E] = 2.7E+7 (psi)

Modulus @ Ref. Temp [Ea] = 2.92E+7 (psi)

Bolting Information:

Bolt Circle Diameter = 36 (inch)

Number of Bolts = 28

Bolt Diameter = 1 (inch)

Flange Module

All. Stress @ Ref. Temp [sa] = 25000 (psi)
 All. Stress @ Design Temp [sb] = 25000 (psi)

Gasket Information:

Gasket Outside Diameter = 32.75 (inch)
 Gasket Inner Diameter = 32 (inch)
 Leak Pressure Ratio [m] = 0.50
 Gasket Seating Stress [y] = 0 (psi)
 Facing Sketch = 2
 Facing Column = 2

Load Data:

Design Pressure = 300 (psi)
 Design Temperature = 295 (F)
 Axial Force = 1000 (lb)
 Bending Moment = 200 (ft-lb)
 Allowable Pressure = 665 (psi)

Comparison of Results:

Flange Stresses	Text Book Results (psi)	CAEPIPE (psi)	CAESAR II (psi)
Operating Condition			
Longitudinal Hub (SH)	0	0	3
Radial Flange (SR)	0	0	0
Tangential Flange (ST)	10577	10569	10618
0.5(SH + SR)	0	0	1.5
0.5(SH + ST)	0	0	5310.5
Bolt Stress	16378	16371	16445
Gasket Seating Condition			
Longitudinal Hub (SH)	0	0	3
Radial Flange (SR)	0	0	0
Tangential Flange (ST)	12147	11987	12166
0.5(SH + SR)	0	0	1.5
0.5(SH + ST)	0	0	6085
Bolt Stress	0	0	124

This page is blank

Index

Index

A

absolute sum, 72, 82, 201, 202
acceleration, 72, 73, 74, 331
 spectral, 331
 vector, 198, 202, 203, 331
additional weight, 149, 311, 314, 396, 429, 430
analyze, 27
anchor, 134
 displacements, 135
 releases for hanger selection, 135
 settlement, 136
 stiffness, 134
angle
 valve, 434
animate
 deflected shape, 130
 mode shape, 131, 205
ANSI pipe sizes, 158, 401
API, 189, 355
API 610 (Tenth Edition, 2004), 564
API 610 report, 355, 362
API 617, 355
API 617 report, 189, 574
API 650, 347, 350
automatic
 save, 15
automatic node renumbering, 16, 43
axes, 111
 elastic element, 207
 local beam, 143
 local coordinate system, 315
 symbol location in graphics, 111
axial force, 59, 233
axial stiffness, 345, 587, 590

B

B31.1 (1967) code, 455
B31.1 (2012) code, 438
B31.3 (2012) code, 447
B31.4 (2012) code, 458
B31.5 (2013) code, 465
B31.8 (2012) code, 470
B31.9 (2008) code, 477
backup, 15
ball joint, 139
 displacements, 387
 friction, 343
 rotation limit, 140
 stiffness, 139
batch file, 580
 example, 595
 export, 25, 600
 import, 578
beam, 142
 additional weight, 149
 AISC library, 146

 beta angle, 143
 end releases, 143
 load, 148
 local coordinate system, 150
 material, 143
 orientation, 150
 section, 145
beam section library
 files, 19
bellows, 155
 displacements, 387
 stiffness, 155
 tie rods, 420
bend, 157
 check, 85
 examples, 159
 180° bend, 162
 45° bend, 161
 90° bend, 160
 base supported, 170
 flanged, 163
 reducing, 166
 supported by a hanger, 168
flexibility factor, 158
incomplete, 85
intermediate nodes, 159
invalid, 85
long radius, 157, 158
material, 158
pressure correction, 64
pressure stiffening, 64
radius, 158
short radius, 157, 158
SIF, 159, 166
tangent intersection point, 157
thickness, 158
bottomed-out springs, 415
Bourdon effect, 63
branch connection, 298
 nonstandard, 428
 stresses, 374
branch line, 39, 175, 254
branch nodes, 295
branch SIF, 174
BS 806 code, 494
buried piping, 177
 general procedure, 177
 ground level, 178
 nomenclature, 179

C

center, 106
center of gravity, 132
change, 42
check
 bend, 85
 connections, 88

Index

- circumferential
 - joint factor, 322, 324, 328
 - stiffness, 347, 604, 605, 606, 607
- closely spaced modes, 202
- code compliance, 374
- cold load, 273
- cold spring, 186
- color code stresses, 126
- combine, 42
- command line operation, 580
- comment, 188
- compressor, 189
- concentrated mass, 192
- cone angle, 363, 364
- connected node, 303, 398
 - hanger, 272
 - skewed restraint, 404
- constant support, 193
 - user defined hanger, 426
- coordinates, 84, 254, 342
- copy graphics, 103
- core pipe, 284, 288, 319, 413
- core properties, 287
- corrosion allowance, 400
- cut pipe, 71, 186, 297
- cutoff frequency, 291, 293

D

- damping
 - harmonic load, 82
 - time history, 80
- data types, 194, 298
- deflected shape, 130
 - animated, 130
- delete, 35
- density, 327
- design
 - factor, 57
 - hanger, 271
 - pressure, 313
 - temperature, 313
- DIN pipe sizes, 158, 401
- direction, 195
- direction sum, 79
- disable graphics editing, 15
- displacement vector, 198, 201
- displacements
 - anchor, 135
 - nozzle, 348
- drawing size, 103
- duplicate last row, 51
- DXF file, 102, 267
- dynamic analysis, 198
 - closely spaced modes, 202
 - effective modal mass, 201
 - friction, 66, 344
 - modal analysis, 198
 - modal equations, 199
 - orthogonality, 199

- participation factors, 200
- response spectrum, 201
- support motion, 200
- time history, 202

- dynamic susceptibility, 205
 - environment variable, 205
 - method, 618

E

- earthquake, 201
- edit
 - copy, 29
 - data, 29
 - find and replace, 31
 - menu, 29
 - paste, 31
 - type, 29
- effective modal mass, 201
- eigenvalues, 198
- eigenvector, 198
 - normalized, 331
- elastic element, 207
 - stiffness, 207
- elastic modulus, 63
- elbow, 157
 - pressure balanced, 217
- element forces, 123, 381
- element forces and moments
 - in local coordinate system, 318
- element stresses
 - FRP, 225
- element types, 296
- EMF file, 101, 267
- EN 13480 (2012) code, 498
- environment variable
 - BOURDONP, 64
 - CPITER, 346
 - dynamic susceptibility, 205
 - HARTLEN, 205
- EPS file, 101, 267
- exit, 346, 393
- expansion joints, 155, 208
 - bellows, 208
 - dual gimbal, 213
 - gimbal, 213
 - slotted hinge, 220
 - tied bellows, 208
 - universal joint, 211
- export
 - 3D Plant Design, 26
 - mbf, 25

F

- fiber reinforced plastic piping (FRP), 225, 226
 - flexibility factor, 226
 - moduli, 226
 - SIF, 226
 - stiffness matrix, 226

Index

- find and replace, 31
 - find node, 55
 - flange, 230
 - allowable pressure, 231
 - equivalent pressure, 231
 - flange and bolt stresses, 234
 - gasket diameter, 231
 - library, 231
 - model
 - files, 19
 - rating, 231
 - report, 375
 - flange and bolt stresses, 234
 - flange module, 234
 - flexibility factor
 - bend, 158
 - FRP, 226
 - miter bend, 335
 - font
 - graphics, 16, 111
 - printer, 100
 - printer, 26, 116
 - text, 15, 16, 69, 395
 - force, 246
 - force spectrum, 93, 248
 - convert time function, 251
 - load, 252
 - read from a text file, 251
 - frequencies, 388
 - closely spaced, 202
 - frequency
 - circular, 200, 331
 - natural, 619
 - friction, 344
 - ball joint, 345
 - guide, 344
 - hinge joint, 345
 - in dynamic analysis, 66, 344
 - limit stop, 344
 - slip joint, 345
 - friction coefficient, 29, 269, 303, 343, 415
 - friction force, 269, 301, 303, 344, 409
 - friction torque, 139, 277, 344, 345, 409
 - From node, 254
 - FRP Stress, 557
- G**
- gasket diameter, 231
 - generate, 46
 - generic support, 256
 - g-load, 72, 434
 - global
 - axes, 38
 - coordinate system, 315
 - forces, 382
 - origin, 178, 292
 - vertical axis, 67
 - graphics, 54
 - background, 111
 - center, 106
 - context menu, 260
 - editing, 261
 - font, 111
 - hotkeys, 264
 - previous view, 105
 - print, 100
 - recover, 113
 - render, 107
 - rotate, 264
 - show, 106
 - turn, 106
 - viewpoint, 105, 264
 - zoom, 106
 - graphics window, 259, 264
 - guide
 - friction, 344
 - friction coefficient, 269
 - gap, 269
 - local coordinate system, 270
 - stiffness, 269
- H**
- hanger, 271
 - below, 272
 - catalog, 273, 414
 - cold load, 272
 - connected node, 272
 - constant support, 193
 - design procedure, 273
 - hot load, 273
 - load variation, 273
 - manufacturer, 273
 - number of, 272
 - report, 375, 398
 - rod, 398
 - short range, 272
 - stiffness, 272
 - to be designed, 271, 298
 - travel, 273, 343, 415
 - type, 272
 - user, 426
 - harmonic analysis, 203
 - harmonic load, 81
 - combination, 82
 - damping, 82
 - frequency, 82
 - phase angle, 81
 - harmonic response, 82
 - hinge joint, 276
 - angular stiffness, 277
 - axis direction, 278
 - displacements, 387
 - example, 279
 - friction, 345
 - friction torque, 277
 - rotational limit, 277
 - rotational stiffness, 277
 - Weight, 278

Index

hotkeys
 graphics window, 264
 layout window, 294
 results window, 393

hydrotest
 load, 282
 load case, 283

I

import, 14
import model, 578
increase digits, 292
input
 new material, 324
insert, 35
installation, 2
insulation, 402
 density, 402
 thickness, 402
intermediate nodes, 159
internal nodes, 288

J

jacket end cap, 284
jacket pipe, 321
jacket properties, 287
jacketed bend, 286
 internal nodes, 288
jacketed pipe, 285
 internal nodes, 286
jacketed piping
 jacketed bend, 286
 jacketed reducer, 288
JIS pipe sizes, 158, 401
joint factor, 328
 circumferential, 328
 longitudinal, 324, 328

K

knuckles
 reducer, 364

L

lateral tee, 176
layout window, 291
left out force, 331
liberal allowable stresses, 59
limit stop, 301
 friction, 344
 nonlinearities, 343
 solution procedure, 343
lining, 402
list, 54
load
 additional weight, 314
 combinations, 76

 pressure, 313
 snow, 314
 specific gravity, 314
 spectrum, 78
 static seismic, 72
 temperature, 313
 wind, 314
load cases, 71
 cold spring, 186
 force spectrum, 252
 hydrotest, 283
 print results, 115
loads
 acceleration, 73
 force spectrum, 249
 thermal, 71
local coordinate system, 315
 element forces and moments, 318
 for a beam, 150
 for a bend, 317
 for a guide, 270
 for a pipe, 315
local forces, 318
location, 319
logarithmic interpolation, 201, 604
long radius, 157, 158, 586
longitudinal
 joint factor, 324, 328
 pressure stress, 63, 328
longitudinal stiffness, 347, 604, 605, 607

M

material
 create library, 328
 define, 321
 density, 327
 description, 327
 input, 324
 name, 327
 select from library, 326
 type, 327
menus, 11
 graphics window menus, 100
 file menu, 100
 options menu, 111
 view menu, 105
 layout window menus, 19
 edit menu, 29, 293
 file menu, 19, 293
 help menu, 294
 loads menu, 71, 294
 misc menu, 83, 294
 options menu, 56, 293
 view menu, 54, 293
 window menu, 98, 294
 results window menus, 114
merge, 20
missing mass correction, 331
miter bend

Index

- closely spaced, 335
- flexibility factor, 335
- modeling procedure, 336
- parameters, 335
- widely spaced, 334, 335

modal analysis, 198

modal displacements, 199

mode shapes, 101, 124, 131, 265

mode sum, 79

model file

- automatic save, 15
- backup, 15
- save, 19

modulus

- cold, 63
- elastic, 63
- hot, 63

multiple split, 37

N

NDE, 558

NEMA SM-23 report, 572

node, 342

- find, 55
- increment, 70
- internal, 288
- list coordinates, 342
- specifying coordinates, 342

nominal diameter, 401

nonlinearities, 343

- ball joint, 345
- friction, 344
- guide, 344
- hinge joint, 345
- limit stop, 344
- misconvergence, 346
- number of iterations, 346
- slip joint, 345
- types, 343

nonstandard

- branch connection, 428
- pipe, 401

Norwegian (1983) code, 507

Norwegian (1990) code, 515

nozzle, 347

- axial stiffness, 347, 604, 605, 607
- circumferential stiffness, 347, 604, 605, 607
- coordinate system, 347, 604
- displacements, 350
- example
 - on a cylindrical vessel, 351
 - on a flat-bottom storage tank, 352
- longitudinal stiffness, 347, 604, 605, 607
- on a cylindrical vessel, 347
- reinforcing pad, 350
- settlement, 351
- stiffness calculations, 604
- stiffness coefficients, 604

NRC Guide 1.92, 202

- number of modes, 331
- number of thermal cycles, 516
- number of thermal loads, 61, 149, 311
- number of time steps, 80

O

occasional load, 63, 72, 253

occasional load., 74

Operating Stress for NDE, 558

options menu, 56

other element forces, 129

P

pad thickness, 174

participation factor, 200, 331, 388

period, 201, 388

phase angle

- harmonic load, 81

pipe skirt, 415

pipe sleeve, 414

pipe slide assembly, 415

pipe slide/shoe assembly, 306

pipng code, 56

plot title, 104

PLT file, 101, 267

Poisson's ratio, 328

Postscript, 101, 267

pressure

- gauge, 313
- negative, 313

pressure correction for bends, 64

pressure stiffening in bends, 64

pressure stress, 63

previous view, 54, 105

print, 100

- graphics, 100
- model, 26
- preview, 101, 103
- results, 115
 - load cases, 115
 - misc, 116
 - model, 114
 - to file, 119
- to file, 101

printer, 100, 116

- font, 26, 116

pump, 355

Q

QA block, 27

R

recover graphics, 113

redo, 52

redraw, 107

reducer, 363

Index

- concentric, 363
 - example, 365
 - cone angle, 363, 364
 - delta, 365
 - eccentric, 363
 - example, 366
 - example
 - concentric, 365
 - eccentric, 366
 - jacketed, 368
 - knuckles, 364
 - SIF, 364
 - stresses, 365
 - weight, stiffness and stress calculation, 365
 - reducing tee, 175
 - reference temperature, 60, 291, 293, 586
 - regenerate, 51
 - releases for hanger selection, 134
 - relief valve, 434
 - render, 107
 - renumber nodes, 43
 - report
 - flange, 233, 375
 - hanger, 375
 - rotating equipment, 375
 - response spectrum, 65, 71, 93, 201, 351
 - restraint, 370
 - stiffness, 370
 - results window, 371
 - rigid body force, 331
 - rigid element, 396
 - stiffness, 396
 - weight, 396
 - rod hanger, 398
 - stiffness, 398
 - rotate, 39
 - rotation limit
 - ball joint, 140
 - hinge joint, 277
- S**
- save, 27, 294
 - automatic, 15
 - section, 400
 - ANSI pipe sizes, 401
 - corrosion allowance, 401
 - DIN pipe sizes, 401
 - insulation, 402
 - JIS pipe sizes, 401
 - lining, 402
 - mill tolerance, 401
 - name, 401
 - nominal diameter, 401
 - nonstandard, 401
 - schedule, 401
 - section modulus, 401
 - settlement
 - anchor, 136
 - nozzle, 351
 - shape factor, 77
 - shear
 - area, 146
 - deflection, 146
 - shear deflection, 146
 - short radius, 157, 158
 - show nodes, 107
 - SIF
 - branch, 174, 298
 - FRP, 226
 - reducer, 364
 - user, 299, 428
 - weld, 435
 - sign conventions, 318
 - skewed restraint, 403
 - connected node, 405
 - direction, 404
 - example
 - sway brace, 405
 - rotational, 404
 - stiffness, 404
 - translational, 404
 - type, 404
 - slip joint, 409
 - friction, 345
 - friction force, 409
 - friction torque, 409
 - pressure thrust area, 409
 - slope, 38
 - snow load, 149, 311, 314
 - snubber, 412
 - stiffness, 412
 - soil restraints, 376
 - sorted stresses, 122, 126, 260, 372, 374, 394
 - specific gravity, 314
 - spectrum
 - direction sum, 79
 - load, 78
 - mode sum, 79
 - spectrum library, 90, 91
 - files, 19, 114, 293
 - spectrums, 90
 - spider, 413
 - split, 36
 - multiple, 37
 - spring rate, 66, 193, 273, 398, 426, 593
 - SRSS, 72, 74, 79, 201
 - static analysis, 57
 - static seismic analysis, 74
 - static seismic load, 72
 - stiffness
 - anchor, 134
 - ball joint, 139
 - bellows, 155
 - elastic element, 207
 - guide, 269
 - hanger, 272
 - hinge joint, 277
 - include hanger stiffness, 66
 - limit stop, 303

Index

- nozzle, 347, 604
- reducer, 365
- restraint, 370
- rigid element, 396
- rod hanger, 398
- snubber, 412
- tie rod, 420
- user hanger, 426
- valve, 430
- stiffness matrix, 66, 198, 202, 203, 273, 343, 344
 - FRP, 226
- Stoomwezen code, 543
- stress range reduction factor, 59, 61
- stresses
 - color coded, 126
 - FRP, 227
 - liberal allowable, 59
 - occasional, 74, 401
 - reducer, 365
 - sorted, 122, 126, 260
 - sustained, 401
 - thresholds, 128
- support load summary, 376
- support loads, 379
- support motion, 200
- supports, 414
 - anchor, 414
 - generic, 256
 - guide, 415
 - hangers, 414
 - limit stop, 415
 - restraint, 414
 - skewed restraint, 414
- sway brace, 403, 405, 414
- sway struts, 403, 414
- Swedish code, 545
- sweepolet, 585

T

- tangent intersection point, 157, 299, 317, 585
- tee
 - extruded, 585
 - fabricated, 585
 - pressure-balanced, 216
 - SIF, 174
 - welding, 298, 585
- Tee, 215, 417
- tensile strength, 328
- thermal cycles, 516
- thermal loads, 71, 135, 311, 351
- threaded joint, 295, 299
- tie rod, 420
 - modeling, 222
 - stiffness, 420
- time functions, 94
- time history, 202
 - damping, 80
 - output interval, 80
 - time step, 80

- time history load, 79
- title
 - plot, 104
- turbine, 422
- turn, 106

U

- u-bolt, 415
- undo, 51
- units
 - English, 68
 - Metric, 69
 - SI, 68
- user allowables, 96
- user hanger, 426
 - constant support, 426
 - load, 426
 - spring rate, 426
 - stiffness, 426
- user SIF, 428
- u-strap, 415

V

- valve, 429
 - additional weight, 430
 - angle, 434
 - calculation of moment of inertia, 434
 - insulation weight X, 430
 - length, 429
 - library, 431
 - files, 19
 - relief, 434
 - stiffness, 430
 - thickness X, 430
 - weight, 429
- velocity vector, 202
- view
 - previous, 105
- viewpoint, 54, 105

W

- weight multiplier, 434
- weld, 435
- weldolet, 585
- wind load, 76, 314
 - shape factor, 77
- wind pressure, 78
- wind profile, 77
- WRC 297, 347

Y

- Y factor, 327

Z

- Z183 code, 549

Index

Z184 code, 553

zero length element, 139, 276

zoom

 all, 54, 106

in, 106

out, 106

window, 266