checkSTRESS™

(for CADMATIC)

Introduction

Under the "checkSTRESS" brand name, three products are available namely checkSTRESS (which generates only CAEPIPE input file), checkSTRESS II (which generates input files for both CAEPIPE and CAESAR II) and checkSTRESS Nuke (which generates input files for both CAEPIPE and PIPESTRESS).

The following table lists the checkSTRESS products currently available for different 3D Plant Design Systems.

Sl. No	3D Plant Design System	Product Name	Status
1	PDMS	checkSTRESS	Available
2	PDMS	checkSTRESS II	Available
3	PDMS	checkSTRESS Nuke	Available
4	CADMATIC	checkSTRESS	Available
5	CADMATIC	checkSTRESS II	Available
6	CADMATIC	checkSTRESS Nuke	Available
7	PDS	checkSTRESS	Available
8	PDS	checkSTRESS II	Available
9	PDS	checkSTRESS Nuke	Available
10	PCF	checkSTRESS	Available
11	PCF	checkSTRESS II	Available
12	PCF	checkSTRESS Nuke	Available

This document describes the common features of checkSTRESS product(s) currently available for PDMS. Details related to the generation of CAESAR II input files by checkSTRESS II and PIPESTRESS input files by checkSTRESS Nuke are covered in a separate document titled "CIIModel.pdf" and "PSModel.pdf" respectively.

Installing Program

To install checkSTRESS on Windows, download the product from the website and execute the followings steps:

1 Run the program "SETUP.EXE" and follow the instructions as they appear on the screen.

2 After successful installation of the program, create an environmental variable with the name "checkSTRESS_CADMATIC" and set the value of the variable as "checkSTRESS_Installed_Path". For example, assuming the checkSTRESS is installed in "C:\Program Files\checkSTRESSCADMATIC", the value would be "C:\Progra~1\ checkSTRESSCADMATIC").

3 The environmental variable can be set as given below

- Open the Control Panel window and open the System Properties window by double clicking the "SYSTEM" icon from within the Control Panel window. Select the "Environment Variables" button from the "Advanced" tab for Windows 2000 or select the Environment tab and click "New" button.

- Then enter in the Variable prompt "checkSTRESS_CADMATIC", and "checkSTRESS_installed_path" (where the checkSTRESS is installed) in the Value prompt, for example ""C:\Progra~1\ checkSTRESSCADMATIC".

- Once entered, select the "Set" button to confirm

Note: For the program to work properly, avoid using space or special characters while entering the path or file name.

How to use checkSTRESS?

1 Pipes selected for Transfer need to be checked for the following.

a. Pressure and Temperature of the Pipeline defined using Pipeline Tags or Model Object Tags.

b. Support type can be transferred to checkSTRESS by creating a user-defined tag to the CADMATIC project with the name "sty" (Support Type) or any other tag name and assigning the same tag to the component connecting the pipe and the pipe support such as U-clamp, O-clamp, etc., at each pipe support location. The tag created as stated above should be set a value as given in the field #1 of the access db "SupportType.mdb" available in the installed directory of checkSTRESS to transfer the support type.

c. If the MMT_LEAVE_GASKET_GAPS is set to 1, then the translator treats the same as real gaps and might put an anchor at that location.

2 Load the CADMATIC Plant Modeler and run the macro "Setup\checkSTRESS.mac" available in the installed path of checkSTRESS through "Run Script". The following form appears.

a. Select the piping code to be used for the analysis of the model from the list box.

b. Enter/modify the Static Seismic Load (g's). Available only for checkSTRESS Nuke.

c. Enter the Starting Node number and Node increment value in the appropriate field. By default, the Starting Node number and Node increment are set to 10 and 10 respectively.

d. Enter the Specific Gravity of operating fluid. By default the specific gravity of water (i.e. 1) will be used for analysis.

e. Select Default Hanger to be used in checkSTRESS model. This Hanger type will be used only when the hanger type in checkSTRESS corresponding to Plant Design hanger is not specified /available in the Mapping DB.

f. Select the Boundary Conditions to be used from the form shown above. Refer section 4.0 titled "Reference" for details.

g. Choose the Global Vertical Axis for checkSTRESS model from the vertical Axis option. By default, "Z Axis" shall be taken as Global Vertical Axis if not specified.

Note: Since CADMATIC always uses vertical global axis as "Z" and Stress Engineers from different parts of the world use different coordinate systems, it is recommended that the designers should consult with the stress department and choose the "Vertical Global Axis" before saving the stress model in CAEPIPE format (for Stress department to carry out the detailed analysis). On the other hand, for performing the first level stress check using checkSTRESS, designers shall continue to use "Z" as Global Vertical Axis.

h. If the user wishes to eliminate those elements whose Nominal size (NS) is less than 50mm / 2in, then they can specify the value as "50" for SI Units and "2" for English Units in the field "Process elements, if dia >". In other words, the program will process only those elements whose nominal size is greater than the value specified in the field above.

i. Specify how the 3 Way and 4 Way valves to be transferred to checkSTRESS. Refer section 4.0 for details.

j. Specify how the Flanges are to be transferred to checkSTRESS. Refer section 3.0 for details.

k. Specify whether the Mapping DB shall be used during transfer or not. Refer checkSTRESS.pdf for details.

l. The Function 'Pick Pipe Line' lets the user to select the objects from the graphics. The user can select entire Pipelines or the Portion of the Pipes. Pipelines that are selected for transfer will be shown in "PipeLine Name Lists" box as shown in figure above.

m. Press the button "Apply" to perform the Stress Check.
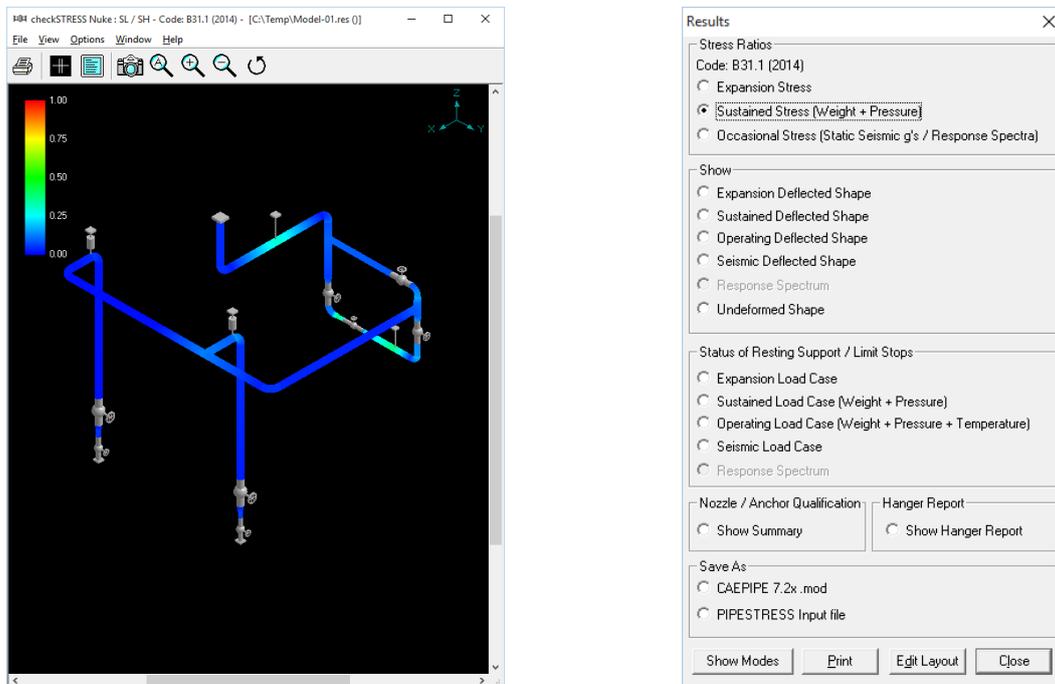
3   Response Spectrum load data can be added through Edit Layout > Misc > Response Spectrum. Once added, the same can be applied in stress model through Edit Layout > Loads > Spectrum. See Appendix A for details on adding Spectrum.

4   Figure above shows the stress contour plot for the Expansion case. Sustained Stress plot and Occasional Stress plot can be seen by selecting the option "Sustained Stress (Weight + Pressure)" and "Occasional Stress" respectively as shown in the figure right above.

5   Designers can also plot and view the deflected shape of the piping system under different loading condition using the options available in the heading titled "Deflected Shape" as shown in the figure right above.

6   Designers can review the Nozzle / Anchor Qualification using the Option "Show Summary".

7   Hanger selected by checkSTRESS for "to be designed" hangers can be viewed by selecting the option "Show Hanger Report".

8   Designers can generate the CAEPIPE mod file using the option "CAEPIPE 7.2x mod" for the Stress Department to carry out the detailed analysis.

9   Similarly, in order for the Stress Department to carry out the detailed analysis including stress report preparation, designers can generate CAESAR II input files in checkSTRESS II product and PIPESTRESS input files in checkSTRESS Nuke product respectively, by selecting the appropriate option from the dialog box as shown above.

10 Using the option "Print", designers can print the model data, code compliance for ten (10) high stress nodes, Nozzle / Anchor Qualification and Status of Resting Support / Limit Stops.

11 Mode shapes reports using the option "Show Modes" are available only for checkSTRESS Nuke product.

3.0 Limitations

The following are not transferred to checkSTRESS from CADMATIC database at this time. However user can add these in checkSTRESS through Edit Layout->Misc->Section

1   Insulation density and Insulation thickness.

2   Corrosion allowance and Mill tolerance.

3   Lining Density and Lining Thickness.

4.0 Reference

Loads

By default, Temperature and Pressure values entered at Pipes and Standard components via OpT and opP shall be transferred to checkSTRESS. If the Temperature and Pressure values for Pipes and Standard components and entered at different tags names other than opT and opP, then designer has to specify the name of the Tags in the dialog box as shown in the figure above. Designer should make sure that these tags are filled with appropriate values depending upon the Units of transfer. I.e., If you wish to transfer the model in SI units, then the value enter for Temperature and Pressure should be in Deg C and kg/cm2 respectively. On the other hand, if you wish to transfer the model in English Units, then the Temperature and Pressure values shall be entered in Deg F and psi respectively.

User can also add / modify the Temperature and Pressure for the piping system in checkSTRESS through Edit Layout->Misc->Loads

Fluid Density

Specify the Specific Gravity of the fluid (with respect to water) during the transfer of the model.

Weight

The weights of Valves, Instruments, Flanges, etc. are extracted from the Dimension Table through first mass quantity type attribute. If the values are defined /available in the database, then the program extracts the information and transfers the same to checkSTRESS.

User can add / modify the weight of components by selecting the component type such as Valve, Rigid, Flange, etc. through Edit Layout->Window->List.

OD, Wall Thickness and Nominal Size

Nominal Size is extracted from the first NS Quantity type of Dimension Table. Similarly, OD and Wall Thickness is extracted from first diameter and first wall thickness quantity type respectively from CADMATIC Dimension Table. For reducers and tees, the arrive OD and Thickness will be read from first Diameter and Wall Thickness quantity type and leave OD and Thickness will be read from second Diameter quantity type and Wall Thickness quantity type.

User can also modify OD, Wall thickness and Nominal size of the piping components directly in checkSTRESS through Edit Layout->Misc->Section.

Flange

Flange from CADMATIC can be transferred as "Pipe with Flange" or "Rigid element" to checkSTRESS. If the user selects "Pipe with Flange" option from the form, then the program creates a pipe with the length of the pipe equivalent to length of flange and creates a Flange at the pipe end in checkSTRESS with flange type as "Weld Neck". The OD and Thickness (in or mm) corresponding to Flange Nominal Size is transferred to pipe in checkSTRESS.

If the user selects "Rigid element" option then, the program will transfer the same as "Rigid Element" to checkSTRESS. Dry weight of flange extracted from CADMATIC will be transferred to "Weight" field of the Rigid Element in checkSTRESS.

User can also modify the property of Flange through Edit Layout->Window->List->Flanges.

Boundary Condition

Selecting the option "Anchor end if connected to Nozzle or Pipelines" will Anchor the Pipe Ends only when they are connected to a Nozzle or Pipelines, otherwise it leaves them as Free. On the other hand, if one end of the pipe is connected to a Nozzle and the other end of Pipe is connected to a Pipe and if that Pipe is not included as the part of the Stress model, then the program will anchor both the end automatically.

For more clarity, consider the following examples. If one end of the pipe is connected to a Pump Nozzle and the other end is not connected to any Nozzle or Pipeline, then the program will anchor the first end and leave the other end as free (i.e., do not create any support). On the other hand, if one end of the pipe is connected to a Nozzle and the other end is connected to a Pipe and that Pipe is not included in the Stress model, then the program will anchor both the ends automatically.

Similarly selecting the option "Anchor all Free Ends" will "Anchor" the Pipe ends automatically irrespective of whether they are connected to Nozzle or Pipelines.

In addition to the above, user can also add / modify supports in piping system in checkSTRESS directly through Edit Layout->Misc->Data Types.

Supports

- Checks for user defined "sty" or the tag name specified by the user in the dialog box as shown above. If available, then reads the details of support from this attribute and write them to the neutral file.

- Tag defined above should be set a value as given in the field #1 i.e., "PdSupport" of the access db "SupportType.mdb", when the user selects the option "Use Mapping DB" for transferring the support details to checkSTRESS. Otherwise, the attribute should be filled with values as specified in **Appendix E** of checkSTRESS User's Manual supplied along with the product. For more details refer to Appendix D of checkSTRESS User's Manual on transferring supports "Without Using Mapping DB".

- User can also add / modify support through Edit Layout > Misc > Data Types or by double clicking the Data Type in the Layout window.

Three Way Valve

Three Way Valve from CADMATIC can be transferred as "Three Rigid Elements" or "Three Pipes with one Concentrated Mass at its Centre" to checkSTRESS by connecting the near end & center, far end & center and branch end & center. From the section property, weight of fluid (kg/lb) is calculated and is added to the dry weight (kg/lb) of Three Way valve read from CADMATIC database. The total weight thus obtained is transferred to each rigid element in proportion to its length.

In the later option, the dry weight of Three Way valve is transferred as weight of the Concentrated Mass at the intersection of the three pipes.

User can modify the weight of rigid element through Edit Layout->Misc->Element Types and Concentrated Mass through Edit Layout->Misc->Data Types.

Four way Valve

Four Way Valve from CADMATIC can be transferred as "Four Rigid Elements" or "Four Pipes with one Concentrated Mass at its Centre" to checkSTRESS by connecting the near end & center, far end & center and two branch end & center. From the section property, weight of fluid (kg/lb) is calculated and is added to the dry weight (kg/lb) of Four Way valve read from CADMATIC. The total weight thus obtained is transferred to each rigid element in proportion to its length.

In the later option, the dry weight of Four Way valve is transferred as weight of the Concentrated Mass at the intersection of the three pipes.

User can modify the weight of rigid element through Edit Layout->Misc->Element Types and Concentrated Mass through Edit Layout->Misc->Data Types.

Material

Material description entered at Material tag of corporate catalogue is extracted from the database and the material property is then obtained from the intermediate mapping db. Refer Appendix B of checkSTRESS User's Manual for more details.

User can add / modify the Material properties in checkSTRESS through Edit Layout > Misc > Materials.

Thermal Anchor Movement (TAM)

Thermal Anchor Movement (TAM) values entered at User defined attributes HNX, HNY and HNX or TNX, TNY and TNZ (attributes need to be added and assigned for Model Object through CADMATIC COS and their values to be assigned in Plant Modeller using the option "Whole Pipeobject") corresponding to Start or End of pipe run is transferred to checkSTRESS. In other words, TAM at starting of the Pipe run should be entered at HNX, HNY and HNZ of Pipe Run. On the other hand, TAM at the end of the Pipe run should be entered at TNX, TNY and TNZ of Pipe run. TAM values should be defined in "mm" for CADMATIC projects in SI units and in "Inch" for CADMATIC project in English units.

For more clarity, consider the following example shown in figure below. Point A is connecting to a Tank. The TAM values corresponding to this point should be defined by selecting the Pipe run 1 and their values should be entered to attributes "HNX", "HNY" and "HNZ". Similarly, TAM values for Point B and C should be defined at attributes "TNX", "TNY" and "TNZ" of the Pipe runs 2 and 3 respectively.
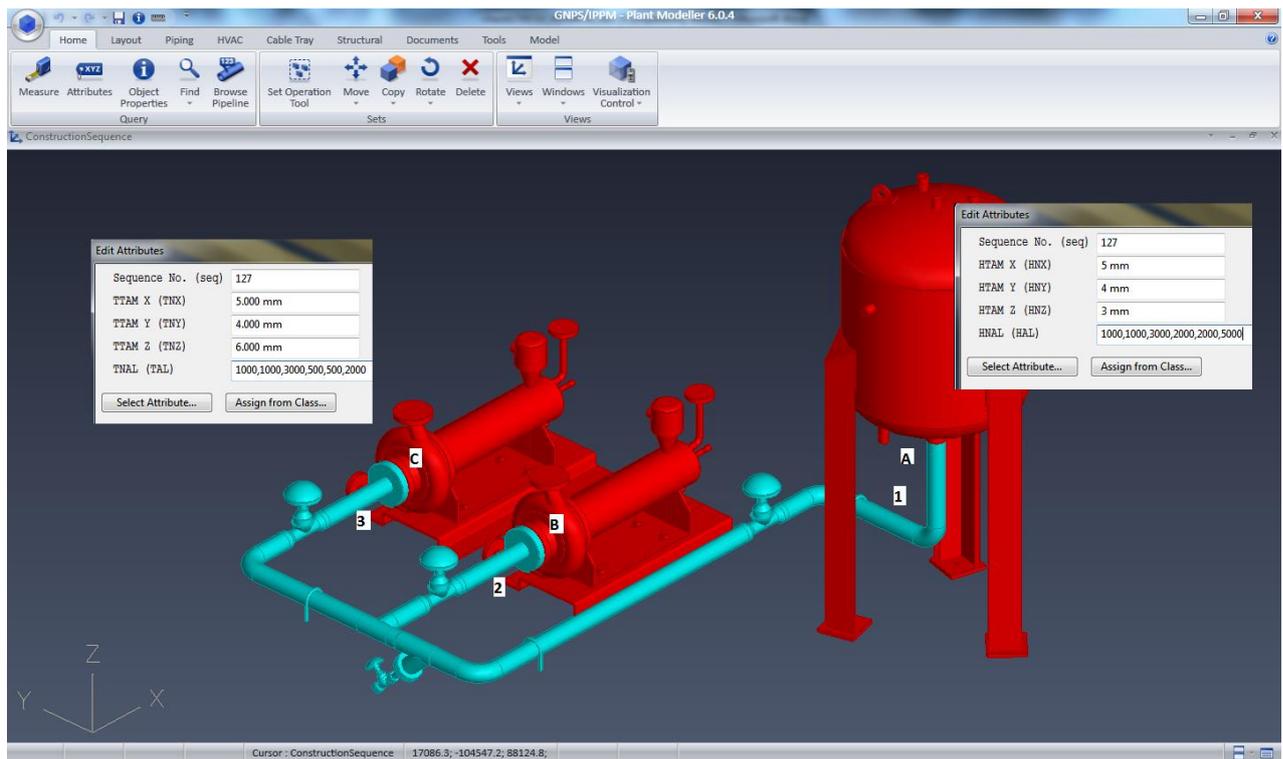
User can also add / modify the Thermal Anchor Movement values through Edit Layout option. i.e., Click Edit Layout then select "Anchor" and select "Edit data" through Layout > Edit. From the dialog box shown, click on "Specified Displacement" and enter the TAM values.

User defined Equipment Nozzle Allowable Loads

Equipment Nozzle Allowable Loads (forces and moments) provided by the equipment manufacturer or calculated using Applicable codes / Finite Element Methods can also be entered at Pipe runs as using the User Defined attributes HAL and TAL in global X, Y and Z directions at Start or End of Pipe run respectively in the format FX, FY, FZ, MX, MY and MZ as explained above for entering TAM values. Please note, the Equipment Nozzle Allowable Loads entered at HAL / TAL should be separated using "," as shown in the figure above. In addition, the force values should be entered in "lb" for English units and in "N" for SI units. Similarly, the moment values should be entered in "ft-lb" for English units and in "Nm" for SI units.

User can add / modify the Equipment Nozzle Allowable Load values through Edit Layout > Misc > User Allowable.

The allowable thus defined are compared against calculated loads and shown / printed in Support Load Summary outputs (can be viewed / printed through "Show Summary" option). If the calculated loads exceed the allowable, they are highlighted in red.

5.0 CADMATIC to checkSTRESS Component Mapping

The types of components available in Cadmatic are simulated as tabulated below in checkSTRESS.

Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
If modeled as Piping Components				
Straight Pipes	DM_GT_PIPE = TRUE & Bend Angle = 0	DM_GT_PIPE	Pipe	PI
Free Bends	DM_GT_PIPE = TRUE & Bend Angle > 0	DM_GT_PIPE	Bend	EL
Flexible Curves	DM_GT_FLXCURVE = TRUE	DM_GT_FLXCURV	Bend	EL
Flanges	DM_GT_2P = True & Primary Connection Code = 1 or 3 (Connection Type is Flange)	DM_GT_2	Rigid Element or Pipe with Flange	FL
Conc. Reducer	DM_GT_2P = True & 2nd Nominal Size > 0	DM_GT_2	Reducer Concentric	RD
Straight Pipes	DM_GT_2P = True & Keyword in corporate catalogue of CADMATIC has "*PIPE*" e.g. Seamless Straight Pipe	DM_GT_2	Pipe	PI
Flow meters, Adaptor, Flexible hoses, Caps	DM_GT_2P = True	DM_GT_2	Rigid Element	RB
Eccentric Reducer	DM_GT_3PDIRFIX = True		Reducer Eccentric	ER
Fixed Angle Curves	DM_GT_FIXCURVE = True	DM_GT_FIXCURVE	Bend	EL
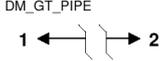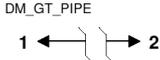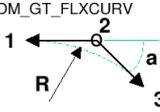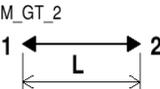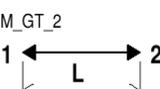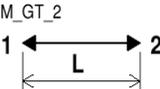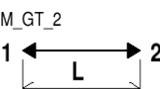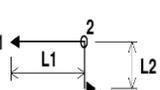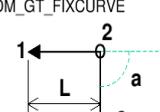
Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
Asymmetric Curves	DM_GT_ASYMCURVE = True		Bend	EL
U-Piece	DM_GT_RETURN = True	DM_GT_RETUR	Bend	EL
Sliding Sockets, Unions	DM_GT_PENETR = True	DM_GT_PENETR	Rigid Element	RB
If modeled as Standard Components				
Filter / Strainer	DM_GT_FLXCURVE = TRUE	DM_GT_FLXCURV	Rigid Element	RB
Some Flanges	DM_GT_2P = True & Primary Connection Code = 1 or 3 (Connection Type is Flange)	DM_GT_2	Rigid Element or Pipe with Flange	FL
Straight Pipes	DM_GT_2P = True & Keyword in corporate catalogue of CADMATIC has "*PIPE*" eg Seamless Straight Pipe	DM_GT_2	Pipe	PI
Valves	DM_GT_2P = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Ball check valve PVC	DM_GT_2	Valve	VA
Conc. Reducer	DM_GT_2P = True & 2nd Nominal Size > 0	DM_GT_2	Reducer Concentric	RD
Flow meters, Adaptor, Flexible hoses	DM_GT_2P = True	DM_GT_2	Rigid Element	RB
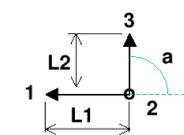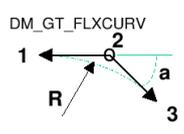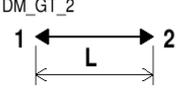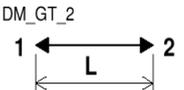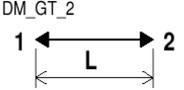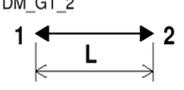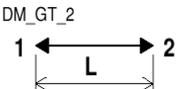
Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
Eccentric Reducer	DM_GT_3PDIRFIX = True		Reducer Eccentric	ER
Fixed Angle Valves	DM_GT_FIXCURVE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Pressured rig-valve WDP		Valve	VA
Straight Pipes	DM_GT_FIXCURVE = True & Keyword in corporate catalogue of CADMATIC has "*PIPE*" eg Seamless Straight Pipe		Pipe	PI
Filter / Strainer	DM_GT_FIXCURVE = True		Rigid Element	RB
3 Way Valves	DM_GT_TEE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Three-way valve Wafer		Three Rigid Elements or Three Pipes with Concentrated Mass	3W
Welding TEE	DM_GT_TEE = True		Pipes with Branch SIF (Welding TEE)	TW
4 Way Valves	DM_GT_CROSS = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Four-way valve Wafer		Four Rigid Elements or Four Pipes with Concentrated Mass	4W
Crosses	DM_GT_CROSS = True		Four Pipes with Branch SIF (Welding TEE)	CR
Lateral TEE	DM_GT_LATERAL = True		Pipes with Branch SIF (Welding TEE)	TW
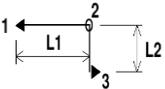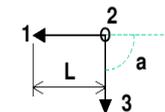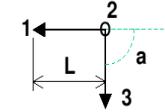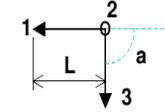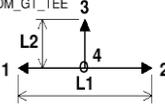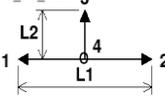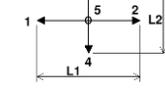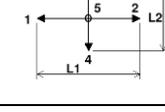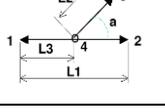
| Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
Valves	DM_GT_VALVE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Ball Valve		Valve	VA
Reduced Valves	DM_GT_VALVE = True & 2nd Nominal Size > 0 & Key word in corporate catalogue should not have "*valve*".		Reducer Concentric	RD
Flow meters, Pressure gauges, Manometers, Strainers	DM_GT_VALVE = True		Rigid Element	RB
Asymmetric Valves	DM_GT_ASYMCURVE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Pressured rig-valve WDP		Valve	VA
Asymmetric Curves	DM_GT_ASYMCURVE = True		Bend	EL
U-Piece	DM_GT_RETURN = True		Bend	EL
Y-piece	DM_GT_YPIECE = True		Pipes with Branch SIF (Welding TEE)	TW
Sliding Sockets, Unions	DM_GT_PENETR = True		Rigid Element	RB
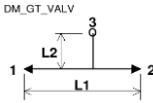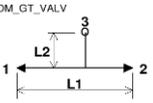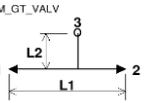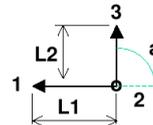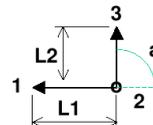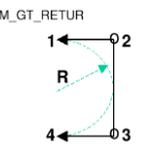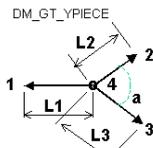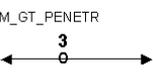
Appendix A
Response Spectrum

.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.
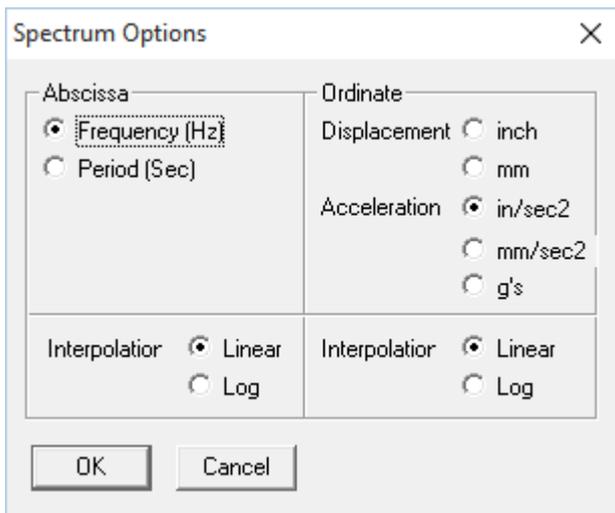
### 1. Input spectrums directly into the model

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; checkSTRESS will sort them later. You can input as many pairs of values as required.

2. 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.

a) 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.

b) 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.

c) 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 checkSTRESS.

d) Now, open the checkSTRESS model that needs these spectrums. Select Read Spectrum from the File menu, read all text files you created one after the other.
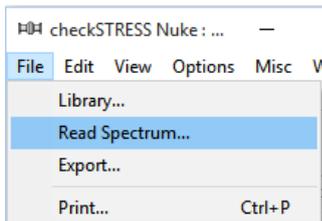
The format of a spectrum text file is shown below. The spectrum that is read appears on the row where the yellow highlight is placed (under the Name column). You can use the Edit menu commands to insert an empty row or delete an existing spectrum. Ensure that no two spectrums share the same Name. checkSTRESS 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, checkSTRESS issues a warning and uses the value corresponding to the closest frequency or period in the spectrum table (for example, checkSTRESS 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.