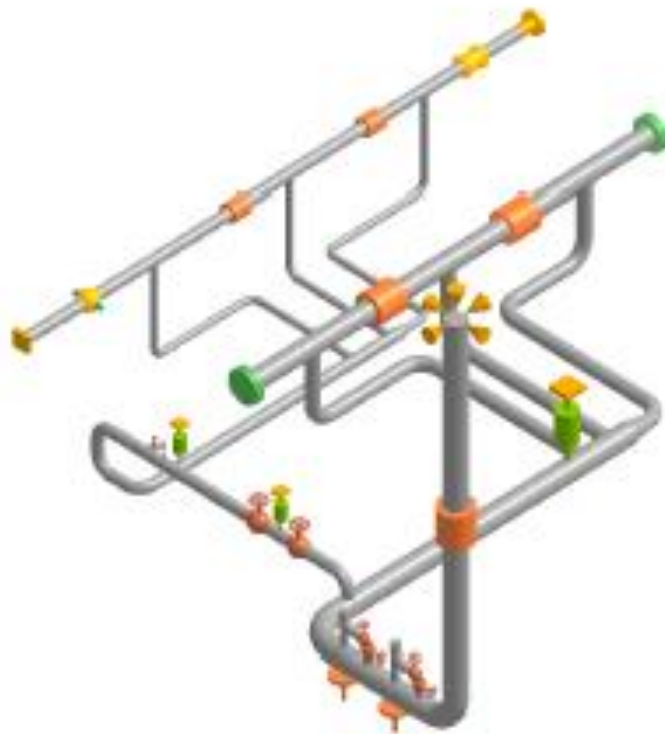


# **HOTclash™**

## **User's Manual**



From the CAdvantagE® Library

## Disclaimer

Please read the following carefully:

This software and this document have been developed and checked for correctness and accuracy by SST Systems, Inc. and SST India Pvt. Ltd. However, no warranty, expressed or implied, is made by SST Systems, Inc., and SST India Pvt. Ltd., as to the accuracy and correctness of this document or the functioning of the software and the accuracy and correctness.

Users must carry out all necessary tests to assure the proper functioning of the software and the applicability of its results.

HOTclash is a trademark of SST Systems, Inc and SST India Pvt. Ltd.

For Technical queries, contact

---

SST Systems, Inc.  
5674 Stoneridge Drive, Suite 107  
Pleasanton, CA 94588  
USA.



Tel: (408) 452-8111  
Fax: (408) 452-8388  
Email: [info@sstusa.com](mailto:info@sstusa.com)

SST India Pvt. Ltd.  
7, Crescent Road  
Bangalore – 560 001  
India.



Tel: +91-80-40736999  
Fax: +91-80-41120695  
Email: [sales@sstindia.co.in](mailto:sales@sstindia.co.in)  
[www.sstindia.co.in](http://www.sstindia.co.in)

# Table of Contents

<b>1.0 Introduction .....</b>	<b>1</b>
1.1 Recommended HOTclash Procedure .....	2
<b>2.0 Installing the Program .....</b>	<b>4</b>
<b>3.0 Limitations .....</b>	<b>4</b>
<b>4.0 Reference .....</b>	<b>5</b>
4.1 Loads.....	5
4.2 Pipe .....	5
4.3 Bend / Elbow .....	5
4.4 Valve .....	5
4.5 Reducer.....	5
4.6 Instrument .....	5
4.7 Flange .....	5
4.8 Olet.....	6
4.9 Tee .....	6
4.10 Three Way Valve.....	6
4.11 Cross .....	6
4.12 Four way Valve.....	6
4.13 Material.....	6
4.14 Support.....	6
4.15 Thermal Anchor Movement (TAM).....	7
4.16 Units .....	7
<b>Appendix A .....</b>	<b>8</b>
3D Plant Design-to-HOTclash Component Mapping.....	8
<b>Appendix B .....</b>	<b>9</b>
Material.....	9
Material DB Modification / Creation.....	9
Sample “Code” DB .....	10

Sample “Material DB” (B315.mdb) .....	10
Modification of “Config.ini” File .....	12
<b>Appendix C .....</b>	<b>13</b>
Schedule Table .....	13
Schedule Table Creation / Modification .....	13
<b>Appendix D .....</b>	<b>14</b>
Supports .....	14
With the use of mapping DB .....	14
Without using Mapping DB.....	14
<b>Appendix E .....</b>	<b>20</b>
Possible Restraints Types and Hangers .....	20
<b>Appendix F.....</b>	<b>24</b>
Errors and Descriptions.....	24

## 1.0 Introduction

It is common practice worldwide that piping designers/layout personnel route pipes with consideration given mainly to space constraints, process and flow constraints (such as pressure drop) and other requirements arising from constructability, operability and reparability. Often pipe stress requirements are not sufficiently considered while routing and supporting piping systems, especially in providing adequate flexibility to absorb expansion/contraction of pipes due to thermal loads. In order for piping designers to consider “first-level” of pipe stress requirements while routing and supporting pipe systems, the product checkSTRESS was developed by SST Systems, Inc. and SST India Pvt. Ltd. and released.

checkSTRESS is an add-on pipe stress check software module to 3D Plant Design Systems such as PDMS, CADMATIC, etc. It helps designers decide “Code Complaint” pipe routing during layout stage and is meant for first level of stress check by designers. So, by using checkSTRESS, the designer is able to send mostly “Code Complaint” piping systems to pipe stress engineers for detailed analysis and stress report preparation.

In order to comply with space constraints, piping designers check for interferences between piping and the adjacent structures, concrete buildings, equipment and reserve volumes for walkthrough, maintenance space etc. either manually on General Arrangement drawings or nowadays in a 3D Plant model. Unfortunately, such “Clash-check” is performed using only “as-designed” drawings/3D model. In reality, piping systems deform due to its own dead-weight during installed/shut-down condition, during hydrotest as well as during operating condition. Clash-checks for piping systems under such “deformed conditions” are usually never performed till date. This product “HOTclash”, developed by SST Systems, Inc. and SST India Pvt. Ltd., eliminates this deficiency in the plant design process. HOTclash is an add-on software module to 3D Plant Design Systems (currently available only on PDMS and CADMATIC). It transfers “deformed shapes” for the piping system under three different loading conditions to the 3D Plant Design software, namely

- “Shut-down” condition
- “Hydrotest” condition
- “Operating” condition

The 3D Plant Design Software, in turn, could then check for interference under the above three loading conditions as well as under the traditional “as-designed” model condition.

### “Shut-down” Condition

Under Shut-down condition, it is assumed that the following loads are applied on the piping system under consideration.

- the pressure specified by the user inside the 3D Plant Design software
- weight of the system including dead weight of pipes, fittings, components such as valves, instruments and flanges as well as weight of operating fluid (assumed filled fully throughout the piping system).

In order to avoid any interference between the “deformed shape” of the system under the above-defined shut-down condition and the adjacent structures, concrete buildings, equipment, reserved volumes etc.,

It may be necessary to support the piping system vertically. Typical vertical supports to carry dead-weight are:

- a. Variable spring hangers.
- b. Constant support hangers,
- c. Rod hangers and
- d. Resting steel supports.

Both rod hangers and resting steel supports fully restrain downward pipe movement but permit pipe to lift up at such supports.

## **“Hydrotest” Condition**

Under Hydrotest condition, the following loads are applied on the system

- the pressure specified by the user in the graphical user interface
- weight of test fluid entered as specific gravity in the GUI with respect to water (assumed filled fully through out the piping system)

Since variable spring hangers are always pinned during hydro-test, HOTclash internally replaces any such variable spring hangers with rigid 2-way vertical restraints for calculating “deflected shape” for hydro-test condition.

## **“Operating” Condition**

Under Operating condition, the following loads are applied on the system selected for HOTclash.

- the pressure specified by the user inside the 3D Plant Design software
- weight of the system including dead weight of pipes, fittings, components such as valves, instruments and flanges as well as weight of operating fluid (assumed filled fully throughout the piping system).
- the temperature specified by the user inside the 3D Plant Design software

## **1.1 Recommended HOTclash Procedure**

The steps given below may normally be followed to perform first-level interference checks for piping systems against the adjacent structures, concrete buildings, equipment, etc.

### **Step1:**

Apply HOTclash on the piping system under consideration in the 3D model, without exiting the 3D plant design system.

### **Step2:**

Execute HOTclash and from the Results menu, select shut-down condition and transfer its deflected shape to 3D Plant Design Software.

### **Step 3:**

Again execute HOTclash and this time, on the Results menu, select Hydro-test condition and transfer its deflected shape to 3D Plant Design software.

### **Step 4:**

Repeat Step 3 above for Operating condition and transfer the deflected shape to 3D Plant Design Software.

### **Step 5:**

Perform Clash check in 3D Plant Design Software for Original “as-designed” piping layout and also for the three hypothetical set of piping representing the three loading conditions namely “shut-down”, “hydro-test” and “operating” conditions.

### **Step 6:**

Once the clash check is performed, the hypothetical set of piping can be deleted from the 3D model.

This manual describes the operational details of HOTclash. It is assumed that the user is already familiar with the principles of Plant Design Software Piping Application and the practices followed in Plant Design Piping catalogue and specifications, and the user has used Plant Design Software to generate the piping by using available facilities in Plant Design Software.

The working sequence of the software is listed below.

1. The pipe branches modeled in the Plant Design Software are read and passed onto HOTclash.
2. HOTclash then, from the material mapping database (see Note 1) provided with HOTclash, identifies valid materials (which will be used for first level interference check) that would

correspond to the material specifications given for those branches in the 3D Plant Design Software. This executable finally carries out stress analysis and transfers the deflected shapes for shut-down, hydrottest and operating load cases to 3D Plant Design software.

The sequence of HOTclash operation is shown diagrammatically in Figure 1-1.

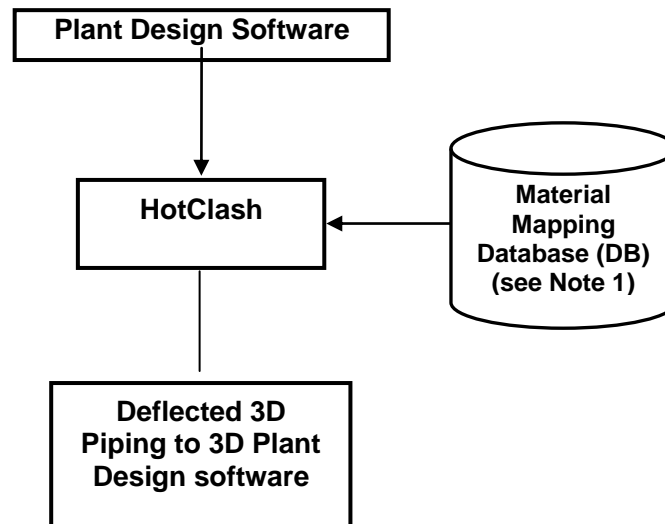


Figure 1.1

**Note 1:**

Refer [Appendix B](#) for more details on Material Property.

## 2.0 Installing the Program

Refer Section 1 in SST License Manager User's Manual for details.

## 3.0 Limitations

Limitations defined in this Chapter are generic in nature and is not specific to any Plant Design application software. Limitations specific to each Plant Design application is listed in the corresponding Readme file (\*.hlp) supplied along with the product for that Plant Design application.

The present version of the HOTclash has the following limitations.

- 3.1 In Case of unavailability of material detail corresponding to Plant Design material description (available in the neutral file), then program takes the Material information specified in the first row of the material Mapping DB selected for HOTclash. Refer [Appendix B](#) for more details. User can also modify the Material properties in HOTclash through Edit Layout->Misc->Materials.
- 3.2 If OD (outer diameter) or Thickness (Wall thickness) for an element is not available in the Plant Design, then OD and Thickness shall be extracted from the Mapping DB corresponding to the specified Nominal Size for that element.

**Note:** Thickness corresponding to Standard Schedule (ANSI) is entered in the Mapping DB and supplied with the standard product. User can change these values to suit their project requirement, if necessary. Refer [Appendix C](#) for more details.

In addition to the above, user can modify the OD and Thickness of a component through Edit Layout->Misc->Sections.

- 3.3 The following items are currently not transferred from Plant Design to HOTclash at this time. However, user can add these items in HOTclash through Edit Layout->Misc->Sections
  - a. Insulation Density and Insulation Thickness of the section.
  - b. Corrosion allowance and Mill tolerance of the piping section and
  - c. Lining Density and Lining Thickness of the piping section.



## 4.0 Reference

This section describes in detail, the methodology followed for transferring the piping components from Plant Design software to HOTclash.

### 4.1 Loads

Temperature (Deg F or Deg C) and Pressure (psi or kg/cm<sup>2</sup>) entered in 3D Plant Design software is transferred to HOTclash for all the elements. If the specific gravity of the fluid (with respect to water) is specified during run time, the same will be transferred to HOTclash. If left blank then it will be set as 0.0.

### 4.2 Pipe

Pipe from 3D Plant Design software is transferred as Pipe to HOTclash. OD and Thickness (in or mm) is read from the Plant Design Database and transferred to HOTclash for each element. If OD and/or Thickness are not available / entered in the 3D Plant Design software, then the program will read the OD and / or Thickness from the Mapping DB corresponding to the Nominal Size specified in the 3D Plant Design for that element. Material name for each pipe element is read from the 3D Plant Design Database the program then gets the material properties corresponding to the 3D Plant Design material name through the Mapping DB and writes the same to HOTclash for that element.

### 4.3 Bend / Elbow

Bend/Elbow from 3D Plant Design software is transferred as "Bend" to HOTclash. The radius (in or mm) of the bend is extracted from the database, if available otherwise; it is calculated as the distance between the Near/Far end of the bend and Tangent Intersection Point divided by  $\tan(\theta/2)$ , Where  $\theta$  is the included angle of the bend. The value thus obtained above shall be written to bend radius field in HOTclash.

### 4.4 Valve

Valve from 3D Plant Design software is transferred as "Valve" to HOTclash. Dry weight of valve (without Fluid weight [lb or kg]) is read from 3D Plant Design Database and transferred to "Weight" field of HOTclash Valve element.

### 4.5 Reducer

Reducer (Concentric and Eccentric) from 3D Plant Design software is transferred as Reducer to HOTclash. The OD (in or mm) and Thickness (in or mm) obtained from the Arrive position shall be written to "OD1" and "THK1" fields of HOTclash. On the other hand, the OD and Thickness obtained from the Leave position shall be transferred to "OD2" and "THK2" fields in HOTclash.

### 4.6 Instrument

Instruments from 3D Plant Design software are transferred as "Rigid" element to HOTclash. Dry weight (lb or kg) of Instrument is read from 3D Plant Design database and the same is transferred to "Weight" field of the rigid element in HOTclash.

### 4.7 Flange

Flange from 3D Plant Design software can be transferred as "Pipe with Flange" or "Rigid element" to HOTclash. If the user selects "Pipe with Flange" option from HOTclash dialog during run-time, then the program creates a pipe for a length equivalent to length of flange and creates a Flange node at the pipe end in HOTclash with flange type as "Weld Neck". The OD and Thickness (in or mm) corresponding to Flange Nominal Size is transferred to pipe in HOTclash. Dry weight of flange is read from 3D Plant Design database and the same is transferred to "Weight" field of the Flange Data Type in HOTclash

On the other hand, If the user selects “Rigid element” option during run-time then, the program will transfer the same as “Rigid Element” to HOTclash. Dry weight of flange is read from 3D Plant Design database and the same is transferred to “Weight” field of the Rigid Element in HOTclash.

#### **4.8 Olet**

Olet from 3D Plant Design software is transferred as Pipe to HOTclash with a Branch SIF defined at the intersection (i.e., where the branch pipe intersect the main run of the pipe).

#### **4.9 Tee**

Tee from 3D Plant Design software is transferred as three pipes connecting the near end & center, far end & center and branch point & center with a Branch SIF (Welding Tee) specified at the center point of the Tee. OD and Thickness read from the 3D Pant Design database for this component is transferred to each HOTclash pipe fields thus created as mentioned above.

#### **4.10 Three Way Valve**

Three Way Valve from 3D Plant Design is transferred as “Three Rigid Elements” or “Three Pipes with one Concentrated Mass at its Centre” to HOTclash 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 3D Plant Design 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.

#### **4.11 Cross**

Cross from 3D Plant Design software is transferred as four pipes connecting near end & center, far end & center and two branch points & center with Branch SIF (Welding Tee) at the pipes intersection in HOTclash. OD and Thickness read from the 3D Pant Design database for this component are transferred to each pipe fields thus created as mentioned above.

#### **4.12 Four way Valve**

Four Way Valve from 3D Plant Design is transferred as “Four Rigid Elements” or “Four Pipes with one Concentrated Mass at its Centre” to HOTclash by connecting the near end & center, far end & center and two branch ends & 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 3D Plant Design 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 Four Way valve is transferred as weight of the Concentrated Mass at the intersection of the three pipes.

#### **4.13 Material**

Material name from Plant Design software is mapped to get the corresponding HOTclash material and then transferred to HOTclash. Refer [Appendix B](#) for details.

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

#### **4.14 Support**

Support information (Location and its type) from Plant Design software can be transferred to HOTclash in two ways viz.

- a. With the use of mapping DB and
- b. Without using Mapping DB

Refer [Appendix D](#) for details.

## 4.15 Thermal Anchor Movement (TAM)

Thermal Anchor Movement (TAM) values entered in global X, Y and Z directions at Equipment Nozzle where the piping layout (selected for HOTclash) is connecting to. These values thus entered can be transferred to HOTclash. TAM values should be defined in “mm” for SI units and in “Inch” for English units. If the TAM values are not available / entered in 3D Plant Design software at Equipment Nozzle then they can be entered into HOTclash 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.

## 4.16 Units

This section describes the units of measurement used to transfer the information from 3D Plant Design software to HOTclash.

### Length (Inches or mm)

Length related dimensions such as OD, Wall thickness, Insulation thickness, Bend radius and Nominal Size from 3D Plant Design software are transferred as Inches or mm for English and SI units respectively.

### Temperature (Deg F or Deg C)

Temperature from Plant Design software is transferred as Deg F or Deg C for English / SI units respectively.

### Pressure (entered in psi or kg/cm2 in Plant Design)

Pressure from Plant Design software is transferred as psi or bar for English / SI units respectively.

### Mass or Weight (lb or Kg)

Dry weight and Wet weight of components from Plant Design software are transferred as lb or Kg for English and SI Units respectively.

### Density (lb/in3 or kg/m3)

Material Density, Insulation Density and Fluid Density from Plant Design software are transferred as lb/in3 or Kg/m3 for English and SI units respectively.

### Translational Stiffness (lb/in or N/mm)

Translational stiffness from Plant Design software is transferred as lb/in or N/mm for English and SI Units respectively to CAEPIPE.

### Rotational Stiffness (in-lb/deg or N-m/deg)

Rotational Stiffness from Plant Design software is transferred as lb-in/deg or N-m/deg for English and SI Units respectively to HOTclash.

## Appendix A

### 3D Plant Design-to-HOTclash Component Mapping

All components available in Plant Design software are mapped with HOTclash component. The details are tabulated below.

Type of Component in Plant Design software	Type of Component in HOTclash	Keywords used in Neutral File
Pipe	Pipe	PI
Valve	Valve	VA
Flange	Rigid Element / Pipe with Flange	FL
Instrument	Rigid Element	RB
Reducer Concentric	Reducer Concentric	RD
Reducer Eccentric	Reducer Eccentric	ER
Cross	Four Pipes with Branch SIF (Welding Tee)	CR
Elbow / Bend	Bend	EL
Three way Valve	Three Rigid Elements / Three Pipes with Concentrated Mass	3W
Four way Valve	Four Rigid Elements / Four Pipes with Concentrated Mass	4W
Tee	Three Pipes with Branch SIF (Welding Tee)	TW
Olet	Pipe with Branch SIF (Weldolet)	TO
Support	Restraint Data Type(s)	HA
Hanger	Hanger Data Type	HA

**Note:** Refer Plant Design software specific Readme file (.pdf) supplied with the product for detailed component mapping.

## Appendix B

### Material

Material name for each element is read from the 3D Plant Design catalogue database. The program then gets the material properties corresponding to 3D Plant Design material information as follows.

- a. Reads the Material Mapping DB file name from the table “code” available in Codedb.mdb corresponding to the Analysis Code specified in the dialog during run-time.
- b. Gets the Material properties from the Material Mapping DB corresponding to 3D Plant Design material description.

In case of unavailability of material property corresponding to 3D Plant Design material description then program reads the Material property specified in the first row of the Material Mapping DB (obtained above) and uses them in HOTclash.

### Material DB Modification / Creation

User is allowed to create their own material table and can use the same by modifying/adding the name of Material Mapping DB file name in table “code” of “Codedb.mdb” supplied along with this software. This table contains four fields viz. PdCode, KpCode, KpMat and KpSect. The first field “PdCode” contains the name of Piping Codes that can be specified in 3D Plant Design Software. The second field “KpCode” contains the name of Piping Codes that are available in HOTclash corresponding to Analysis Code defined during run time. Third field “KpMat” defines the name of the Material DB file to be read (to get the Material property) during run time. Fourth field “KpSect” defines the name of the Schedule Table to be read (to extract OD and Thickness if not available in the 3D Plant Design for that element) during run time. Please note, the Material DB must exist before it is used in the Codedb.mdb. The procedure for creating a Material DB and modifying the Codedb.mdb is listed below.

- a. Copy the existing file (B311.mdb) and rename the file with a desired name by pasting it to the directory where the source file was stored.
- b. The newly created DB contains four tables viz. MaterialE, MaterialS, MdetailE and MdetailS. The MaterialE and MdetailE table in the DB are used to define material properties in English units whereas MaterialS and MdetailS tables are used to define the material properties in Metric units.
- c. Enter the Plant Design Material description (available in 3D Plant Design Database) into the field “MatName” of “MaterialE/MaterialS” table and enter the engineering property of the material such as Density, Nu, Joint factor, etc., depending upon the availability of the information in the code selected and leave the rest of the fields as “None”. For example, the fields Tensile, CircFactor and Yield is not valid for B31.1 and hence it should left as “None”.
- d. Enter the Temperature related property such as Young’s Modulus, Alpha, Allowable, etc., into “MdetailE/MdetailS” table by expanding it using the button “+”. Fill the fields that are relevant to the Material Code selected and leave the rest as “None”.
- e. Modify the contents of each table with new values and save the DB. Fill the table fields with the appropriate values depending upon the type of piping code. Other fields can be left as “None”.
- f. After successful creation of material Mapping DB as explained above in steps a to e, open the DB codedb.mdb and enter the name of the Material DB file (without extension) in the field “KpMat”. For example, if you have created your own material Mapping DB file corresponding to B31.5 as B315.mdb, then enter the name of the file (B315.mdb) in the field “KpMat” as “B315” where the value of the field “PdCode” is equal to B31.5 and then enter the corresponding Section details table name in the field “KpSect”.
- g. A sample “Code” mapping DB with Material DB is given below for reference.

## Sample “Code” DB

PdCode	KpCode	kpMat	Kpsect
ASME CLASS2(1980)	ASME	B311	STDsch
ASME CLASS2(1986)	ASME-86	B311	STDsch
ASME CLASS2(1992)	ASME-92	B311	STDsch
B31.1	B311	B311	STDsch
B31.1(1967)	B311-67	B311	STDsch
B31.3	B313	B313	STDsch
B31.4	B314	B314	STDsch
B31.5	B315	B315	STDsch
B31.8	B318	B318	STDsch
BS806	BS806	B311	STDsch
CODETI(1995)	SNCT	B313	STDsch
Default	B311	B311	STDsch
EN13480	EUROPEAN	EN13480	STDsch
Norwegian(1983)	NORWEGIAN-83	B313	STDsch
Norwegian(1990)	NORWEGIAN-90	B313	STDsch
RCC-M(1985)	RCC-M	B311	STDsch
Stoomwezen(1989)	STOOMWEZEN	B311	STDsch
Swedish(1978)	SWEDISH	B311	STDsch
Z183(1990)	Z183	B313	STDsch
Z184(1992)	Z184	B313	STDsch

### Fields in Code DB Table:

PdCode - Name of the Piping Code than can be specified in Plant Design Software.

KpCode - Analysis code corresponding to Plant Design Code.

KpMat - Material DB name from where the material properties are to be read.

KpSect - Section DB name from where the Schedule details are to be read.

## Sample “Material DB” (B315.mdb)

### Sample Table “MaterialE” in English Units

	Index	MatName	Density	Nu	JointFactor	MaterialType	Tensile	CircFactor	Yield
+	1	A53-A	0.283	0.3	1	CS	None	None	None
+	2	A53-B	0.283	0.3	1	CS	None	None	None
+	3	A106-A	0.283	0.3	1	CS	None	None	None
+	4	A106-B	0.283	0.3	1	CS	None	None	None
+	5	A106-C	0.283	0.3	1	CS	None	None	None
+	6	API-5L-A	0.283	0.3	1	CS	None	None	None
+	7	API-5L-B	0.283	0.3	1	CS	None	None	None
+	8	A312-TP304	0.283	0.3	1	AS	None	None	None
+	9	A312-TP304L	0.283	0.3	1	AS	None	None	None
+	10	MonelB165-Annealed	0.283	0.3	1	CS	None	None	None
+	11	CopperB42-Annealed	0.283	0.3	1	CA	None	None	None
+	12	CopperB42-Drawn	0.283	0.3	1	CC	None	None	None
+	13	RedBrass-B43	0.283	0.3	1	CA	None	None	None
+	14	Aluminum-B241-A96061T6	0.283	0.3	1	AL	None	None	None
+	15	Aluminum-B241-A96063T6	0.283	0.3	1	AL	None	None	None

Fields in each table and their descriptions are given below in detail.

**Fields in MaterialE/MaterialS Tables:**

Index	- Unique Material Id
MatName	- Material Name
Density	- Density of the Material in English units (lb/in <sup>3</sup> or kg/m <sup>3</sup> )
Nu	- Poisson Ratio
JointFactor	- Joint Factor of the Material
MaterialType	- Type of Material (2 chars)
Tensile	- Tensile Strength (psi or MPa)
CircFactor	- Circular Factor
Yield	- Yield Strength (psi or MPa)

**Sample Table "MaterialE" from B315.mdb in English Units**

MatName	Temperature	E	Alpha	Allowable	Yield	Rupture	Design	Proof	fh	fCR
A106-A	-20	2.99E+07	5.89E-06	12000	None	None	None	None	None	None
A106-A	70	2.95E+07	6.08E-06	12000	None	None	None	None	None	None
A106-A	100	2.93E+07	6.14E-06	12000	None	None	None	None	None	None
A106-A	300	2.83E+07	6.88E-06	12000	None	None	None	None	None	None
A106-A	350	2.80E+07	6.85E-06	12000	None	None	None	None	None	None
A106-A	400	2.77E+07	6.82E-06	12000	None	None	None	None	None	None
A106-A	150	2.91E+07	6.24E-06	12000	None	None	None	None	None	None
A106-A	200	2.88E+07	6.35E-06	12000	None	None	None	None	None	None
A106-A	250	2.86E+07	6.62E-06	12000	None	None	None	None	None	None
A106-B	300	2.83E+07	6.88E-06	15000	None	None	None	None	None	None
A106-B	350	2.80E+07	6.85E-06	15000	None	None	None	None	None	None
A106-B	400	2.77E+07	6.82E-06	15000	None	None	None	None	None	None
A106-B	-20	2.99E+07	5.89E-06	15000	None	None	None	None	None	None
A106-B	70	2.95E+07	6.08E-06	15000	None	None	None	None	None	None
A106-B	100	2.93E+07	6.14E-06	15000	None	None	None	None	None	None
A106-B	150	2.91E+07	6.24E-06	15000	None	None	None	None	None	None
A106-B	200	2.88E+07	6.35E-06	15000	None	None	None	None	None	None
A106-B	250	2.86E+07	6.62E-06	15000	None	None	None	None	None	None

**Fields in MDetailE/MDetailS Tables:**

MatName	- Material name
Temperature	- Material Temperature (deg. F or deg. C)
E	- Young's Modulus (psi or MPa)
Alpha	- Alpha value for material (in/in/F or mm/mm/C)
Allowable	- Allowable Stress (psi or MPa)
Yield	- Yield Strength (psi or MPa)
Rupture	- Rupture Stress (psi or MPa)
Design	- Design Stress (psi or MPa)
Proof	- Proof Stress (psi or MPa)
fh	- Allowable Stress at Maximum Temperature (psi or MPa)
fCR	- Allowable Creep Stress (psi)

For FRP/GRP materials (with “MaterialType” field = “FR” or “F1” or “F2”), the fields given below should be entered with the values specified.

E	- Axial Modulus of FRP/GRP (psi or MPa)
Alpha	- Alpha value for material (in/in/F or mm/mm/C)
Allowable	- Hoop Allowable for FRP/GRP (psi or MPa)
Yield	- Torsional Allowable for FRP/GRP (psi or MPa)
Rupture	- Axial Allowable for FRP/GRP (psi or MPa)
Design	- Hoop Modulus for FRP/GRP (psi or MPa)
Proof	- Shear Modulus for FRP/GRP (psi or MPa)

Material properties for piping codes B31.1, B31.3, B31.4, B31.5, B31.8, ISO 14692, ASME-NM1 and EN13480 are available in the “Materials” folder.

## Modification of “Config.ini” File

Material Mapping DB’s and Support Mapping DB’s are stored in the Application directory of the Product. The customized Material Mapping DB’s and Support Mapping DB’s can be shared among the users of the product by copying these files in a shared location and modifying the “config.ini” file to point to the new path.

For clarity, “config.ini” file contains the path of the Material Mapping DB’s and Support Mapping DB’s. By default, this will point to the application directory. Copying these files to a shared location and modifying the path in the “config.ini” to reflect the new location will help users to share the customized DB’s. The content of the file is listed below for reference.

*[Config]*

*Product Name=HOTclash*

*Product Type=Server Version*

*Materials\_DB=*

*Code\_DB=*

*Support\_DB=*

*leaving the above fields empty will use the default path*

Assuming the Material Mapping DB’s, Code DB and Support DB are stored in the shared location DBS of machine “InfoP025”, modify the Materials\_DB, Code\_DB and Support\_DB as follows.

*[Config]*

*Product Name=HOTclash*

*Product Type=Server Version*

*Materials\_DB=\\InfoP025\DBS*

*Code\_DB=\\InfoP025\DBS*

*Support\_DB=\\InfoP025\DBS*

*leaving the above fields empty will use the default path*

**Warning: Care should be taken while entering the fields of the CodeDb.mdb and the Material Mapping DB. Wrong entry or leaving the field empty may lead to malfunction of the software.**



## Appendix C

### Schedule Table

The table "code" in Codedb.mdb contains a field named "KpSect" to specify the thickness to be used during run time, if the OD and/or thickness of a component is not available in the 3D Plant Design software.

Program reads the OD and/or thickness from the standard schedule table and uses them in HOTclash if the OD and/or thickness is not available in the 3D Plant Design software for a component. In case the value of the field "KpSect" is not defined or left empty in the table, then by default, the program uses the values of "Standard Schedule (STD) for ANSI".

### Schedule Table Creation / Modification

The procedure for creating the user defined Standard Schedule Table is listed below

1. Copy the table "STDsch" and then paste it as new table in the same "Codedb.mdb" by specifying a new name for the table.
2. Modify the contents of the table with new values.
3. Open the table "Code" and then enter the "KpSect" field with the name of the table created above corresponding to the "PdCode". For e.g. assuming the name of the new Standard Schedule table created as "Sch40" corresponding to "B31.1" PdCode, change the value of field "STDsch" as "Sch40".

PdCode	KpCode	kpMat	Kpsect
B31.1	B311	B311	STDsch

**Sample Schedule Table**

NPD_E	NPD_M	OD	THK
0.125	4	10.287	1.7272
0.25	8	13.716	2.2352
0.375	10	17.145	2.3114
0.5	15	21.336	2.7686
0.75	20	26.67	2.8702
1	25	33.401	3.3782
1.25	32	42.164	3.556
1.5	40	48.26	3.683
2	50	60.325	3.9116

Fields in standard schedule table and their descriptions are given below in detail.

NPD_E	- Nominal Piping Diameter in Inches
NPD_M	- Nominal Piping Diameter in Millimeters
OD	- Outside Diameter in mm
THK	- Wall thickness in mm

**Warning: Care should be taken while filling the fields of the Schedule DB, as the wrong entry may lead to malfunction of the software.**

## Appendix D

### Supports

Support information (Location and its type) from Plant Design software can be transferred to HOTclash in two ways viz.

1. With the use of mapping DB
2. Without using Mapping DB

#### With the use of mapping DB

The support details (entered via attributes) and its location specified in the Plant Design software are transferred to HOTclash. The values of the attributes filled at support locations shall be in accordance with the values specified in the field #1 of tables "Zvertical" and "Yvertical" of "SupportType.mdb" built into the application. The values from field #1 of table "Zvertical" shall be referred and entered at the support locations (via attributes), if the Global Vertical Axis to be used in the Stress Model is "Z". On the other hand, values from field #1 of table "Yvertical" shall be referred and entered at the support locations (via attributes), if the Global Vertical Axis to be used in the Stress Model is "Y".

Fortunately, the values entered/available in the field #1 of tables "Zvertical" and "Yvertical" are kept identical, because most Plant Design software always consider the vertical direction as Z-axis. On the other hand, pipe stress engineers in different parts of the world use either Z-axis as vertical or Y-axis as Vertical. So, the values entered in the field "KpSupport" are different for "Zvertical" and "Yvertical". Program always uses the value entered in the field "KpSupport" corresponding to the value entered in field "PdSupport", for its stress model file generations.

User can modify the values available in the field "PdSupport" of tables "Zvertical" and "Yvertical" to suit their requirements. It is recommended to keep the values entered in the field "PdSupport" of tables "Zvertical" and "Yvertical" identical as much as possible. This will help to avoid the user in reentering/changing the values at support locations for different Global Vertical Axis.

In case, the HOTclash support information corresponding to the attribute value entered in the Plant Design is not available/defined in the mapping DB, then the translator skips that support at the location.

#### Without using Mapping DB

In smaller organizations, the Plant Design Engineer would be responsible for carrying out the Stress Analysis and hence, he would be familiar with the usage of HOTclash software and the customs followed in generating the Stress Models. He also would be familiar in arriving at the mathematical model equivalent to the geometric model from Plant Design Software. i.e., can define the support conditions (boundary conditions) in HOTclash equivalent to the Physical Support configurations from Plant Design software. In such cases, the Plant Design Engineer can transfer the support details (locations and types) by entering them at the support locations directly and transfer the same to HOTclash without using the mapping DB. Refer the sections listed below for details.

#### Transfer of Supports

The support details (entered via attributes) and their locations specified in the Plant Design software are transferred to HOTclash. Refer .hlp file for details on attributes to be filled. The values of the attributes that can be filled at support locations shall be in the format as listed below.

#### Hangers

Hangers can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

##### Syntax:

Hanger(Type of Hanger: Number of Hangers: Allowable Travel Limit[in or mm]: Load Variation[%]: Short Range)

**Example:**

Hanger(Grinell:2: :20:1)

**Note:** Allowable Travel Limit option is not enabled at this time. Please skip that field while entering the values.

**Rod Hanger**

Rod Hangers can also be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

**Syntax:**

Hanger(ROD:Number of Hangers)

**Example:**

Hanger(ROD:1)

**Constant Support**

Constant Support can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

**Syntax:**

Hanger(CONSTSUPPORT:Number of Hangers)

**Example:**

Hanger(CONSTSUPPORT:3)

**User Hanger**

User specified Hangers could also be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

**Syntax:**

USERHANGER(Number of Hangers:Spring Rate[lb/in or N/m]:Cold Load:Hot Load)

**Example:**

For example one number of user hanger with spring rate of 1E8N/m, and hot load 1000 can be specified as follows,

USERHANGER (1:1E8: :1000)

**Guide**

Guide Restraint can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

**Syntax:**

GUI(Stiffness[lb/in or N/mm]:Gap[in or mm]:Friction Coefficient)

**Example:**

Guide with Rigid stiffness, 50mm gap between guide and pipe and 0.3-friction coefficient can be specified as GUI(R:50:0.3)

**Skewed Restraints**

Skewed restraint(s) with different directional vectors can be transferred from 3D Plant Design software to HOTclash by specifying vector details with the following at support locations.

**Syntax:**

SKEW(VecX:VecY:VecZ:Stiffness[lb/in or N/mm]:Gap[in or mm]:Friction Coefficient: Type)

Type can be defined as "R" (Rotational) or "T" (Translational)

**Example:**

SKEW(0.707:0.707:0.707:1E10: : :R)

**Note:** Gap and Friction Coefficient options are not enabled at this time. Please leave those fields blank while entering the values.

**Skewed Restraints in Local Axes**

Skewed restraint(s) in Axial, Shear X and Shear Y can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations. For definition on Local Coordinates, refer to Section titled “Local Coordinates” in HOTclash Technical Reference Manual.

**Syntax:**

SKEWA(Stiffness[ib/in or N/mm]:Gap[in or mm]:Friction Coefficient:Type) – Skewed Restraint in Local Axial direction of element

SKEWY(Stiffness[ib/in or N/mm]:Gap[in or mm]:Friction Coefficient:Type) – Skewed Restraint in Local Shear Y direction of element

SKEWZ(Stiffness[ib/in or N/mm]:Gap[in or mm]:Friction Coefficient:Type) – Skewed Restraint in Local Shear Z direction of element

Type can be defined as “R” (Rotational) or “T” (Translational)

**Example:**

SKEWA/SKEWY/SKEWZ(1E10::R) or SKEWA(Rigid::R)

**Double acting Translational Restraints**

Double acting Translational restraint(s) can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

If Double acting Translational restraint(s) with both Stiffness and Gap are specified at a location, then the program will transfer the same as “Anchor” in that direction with Gap as equal to Anchor Displacement in that direction. Refer examples for details.

If Double acting Translational restraints are specified with Stiffness and without Gap, then the program will transfer the same as “Anchor” in that direction with stiffness value as defined in the attribute.

Lastly, if Double acting Translational restraints are specified without Stiffness and without Gap then the program will transfer the same as Restraint in that direction.

**Syntax:**

Translational Restraint Type(Stiffness[ib/in or N/mm]:Gap[in or mm])

**Example:**

X(1E6,10) will be Transferred as Anchor with X stiffness as “1E6” and Displacement in X as 10 mm.

X(1E6);Y(1E6) will be transferred as Anchor with X and Y Translational stiffnesses as “1E6”.

X;Y will be transferred as Restraint in X and Y (i.e., X and Y Restraints)

**Double acting Rotational Restraints**

Double acting Rotational restraint(s) can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

If Double acting Rotational restraint(s) with both Stiffness and Gap specified at a location, then the program will transfer the same as “Anchor” with flexible stiffness in that direction with Gap as equal to Anchor Rotational Displacement in that direction. Refer examples for details.

If Double acting Rotational restraint(s) are specified with Stiffness and without Gap, then the program will transfer the same as “Anchor” with flexible stiffness in that rotational direction.

Lastly, if Double acting rotational restraint(s) are specified without Stiffness and without Gap, then the program will transfer the same as Skewed Restraint with stiffness as “Rigid” in that direction.

**Syntax:**

Rotational Restraint Type(Stiffness[lb/in or N/mm]:Gap[in or mm])

**Example:**

RX(1E6,10) will be Transferred as Anchor with RX stiffness as “1E6” and Displacement in XX as 10 mm.

RX(1E6);RY(1E6) will be transferred as two Skewed Rotational Restraints one in X direction and other in Y direction with stiffness as “1E6”.

RX;RY will be transferred as Skewed Rotational Restraints with Stiffness as “Rigid” in X and Y (i.e., X and Y Rotational Restraints)

**Double Acting Limit Stop**

Limit stop can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations. Directional components are must while specifying limit stop.

**Syntax:**

LIM(Stiffness[lb/in or N/mm]:Gap[in or mm]:Friction Coefficient:Xcomp:Ycomp:Zcomp)

LIM(Stiffness[lb/in or N/mm]:Lower Gap/Upper Gap[in or mm]:Friction Coefficient:Xcomp:Ycomp:Zcomp)

**Example:**

1. Limit stop in Y direction with Rigid stiffness and equal lower and upper gap of 50 mm (i.e., lower limit = -50 mm and upper limit = 50 mm) with coefficient of friction 0.2 can be transferred by specifying the support attribute as LIM(RIGID:50:0.2:0:1:0).
2. Limit stop in Y direction with Rigid stiffness and Lower gap of -50 mm and Upper Gap of 25 mm with coefficient of friction 0.2 can be transferred by specifying the support attribute as LIM(RIGID:-50/20:0.2:0:1:0).

Please note, Rigid stiffness means 1E+12 lb/in will be assigned internally in HOTclash software.

**Single / Double Limit Stop in Local Axes**

Limit stop in Local Axial, Shear Y and Shear Z directions can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations. For definition on Local Coordinates, refer to Section titled “Local Coordinates” in HOTclash Technical Reference Manual.

**Syntax:**

LIMA/LIMY/LIMZ(Stiffness[lb/in or N/mm]:Gap[in or mm]:Friction Coefficient]

LIMA/LIMY/LIMZ(Stiffness[lb/in or N/mm]:Lower Gap/Upper Gap[in or mm]:Friction Coefficient]

LIMA – Limit Stop in local Axial direction of element

LIMY– Limit Stop in local Shear Y direction of element

LIMZ – Limit Stop in local Shear Z direction of element

**Example:****Single Acting Local Limit Stops**

1. Single Acting Limit stop in Local Axial direction with Rigid stiffness and gap of 50 mm and coefficient of friction 0.2 can be transferred by specifying the support attribute as LIMA(RIGID:-50/None:0.2).
2. Single Acting Limit stop in Local Axial direction with Rigid stiffness and gap of 0 mm and coefficient of friction 0.2 can be transferred by specifying the support attribute as LIMA(RIGID:0/None:0.2).

### Double Acting Local Limit Stops

1. Double Acting Limit stop in Local Shear Y direction with stiffness of 1E+05 N/mm and equal lower and upper gap of 50 mm (lower limit = -50 mm and upper limit = 50 mm) and coefficient of friction 0.2 can be transferred by specifying the support attribute as LIMY(RIGID:50:0.2).
2. Double Acting Limit stop in Local Shear Y direction with stiffness of 1E+08 N/mm and lower gap of 0 mm, upper gap of 20 mm and coefficient of friction 0.2 can be transferred by specifying the support attribute as LIMA(RIGID:0/None:0.2).

Please note, Rigid stiffness means 1E+12 lb/in will be assigned in HOTclash.

### Single acting Translational Restraints

Single acting Translational Restraints are transferred as Limit Stop to HOTclash. If the user specifies single acting restraints with "+" and "-" in the same axis (direction), then the program will transfer it as "Limit Stop" with direction of restraint as equal to axis positive direction and Gap specified in positive direction as "Lower Limit" and Gap entered in negative axis direction as "Upper Limit".

If positive directional restraint is specified with Gap, then the program will transfer the same as Limit Stop with Gap value as "Lower Limit".

Similarly, if negative directional restraint is specified with Gap, then the program will transfer the same as Limit Stop with Gap value as "Upper Limit".

Refer the examples for details.

#### Syntax:

Single Acting Restraint Type(Stiffness[lb/in or N/mm]:Gap[in or mm]:Friction Coefficient)

#### Example:

+X(1e10:-15:0.25);-X(1e10:10:0.25) will be transferred as "Limit Stop" with Stiffness = 1E10, Lower Limit=-15 mm, Upper limit=10 mm and Direction = 1,0,0.

+Z(1E10:10) will be transferred as "Limit Stop" with Stiffness=1E10; UpperLimit=None, Lower Limit=10 and Direction=0,0,1.

-Y(1E10:10) will be transferred as "Limit Stop" with stiffness=1E10; UpperLimit=10;Lower Limit=None and Direction=0,-1,0.

### Snubber

Snubbers can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

#### Syntax:

Types of Snubber(Stiffness[lb/in or N/mm])

#### Example:

YSNB(1E10) or ZSNB(1E6)

### Skewed Snubber

Skewed Snubbers can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations.

#### Syntax:

SNB(VecX:VecY:VecZ:Stiffness[lb/in or N/mm])

#### Example:

Skewed Snubber with stiffness 1E+9 can be specified with directional vectors as follows,

SNB(0.707:0:0.707:1E9)

### **Snubber in Local Axes**

Snubbers in Local Axial, Shear Y and Shear Z directions can be transferred from 3D Plant Design software to HOTclash by specifying the following at support locations. For definition on Local Coordinates, refer to Section titled “Local Coordinates” in HOTclash Technical Reference Manual.

#### **Syntax:**

SNBA(Stiffness[lb/in or N/mm]) – Snubber in Local Axial direction of element

SNBY(Stiffness[lb/in or N/mm]) – Snubber in Shear Y direction of element

SNBZ(Stiffness[lb/in or N/mm]) – Snubber in Shear Z direction of element

#### **Example:**

SNBA(1E10) or SNBY or SNBZ(R)

### **Force / Moment**

Force and Moments can be transferred from Plant Design to HOTclash by specifying the following at support locations.

#### **Syntax:**

FORCE(Fx:Fy:Fz[lb or N])

MOMENT(Mx:My:Mz[lb-in or Nm])

#### **Example:**

1000N Force acting in Y direction can be specified as follows

FORCE(0:1000:0)

500Nm Moment acting in Z direction can be specified as follows

MOMENT(0:0:500)

### **Threaded Joint**

Threaded Joint can be assigned to nodes by specifying the following at support locations.

#### **Syntax:**

TJOINT

### **User SIF**

User SIF can be assigned for a node by specifying the following at support locations in Plant design software.

#### **Syntax:**

UserSIF(Value)

#### **Example:**

UserSIF(100)

## Appendix E

### Possible Restraints Types and Hangers

Particulars	Syntax	Example
<b>Anchor</b>		
Anchor	ANC(Stiffness:Gap)	ANC or ANC(1E12:0.0) or ANC(1E12)
<b>Double Acting Translational Restraints</b>		
X	X(Stiffness:Gap)	X or X(1E12) or X(1E12:25)
Y	Y(Stiffness:Gap)	Y or X(1E10) or Y(R:50)
Z	Z(Stiffness:Gap)	Z or Z(RIGID) or X(RIGID:35)
<b>Double Acting Rotational Restraints</b>		
RX	RX(Stiffness:Gap)	RX or RX(1E12) or RX(1E12:0.0)
RY	RY(Stiffness:Gap)	RY or RY(R) or RY(1E12:25)
RZ	RZ(Stiffness:Gap)	RZ or RZ(RIGID) or RZ(R:50)
<b>Double Acting Snubbers</b>		
XSNB	XSNB(Stiffness)	XSNB or XSNB(1E12)
YSNB	YSNB(Stiffness)	YSNB or YSNB(R)
ZSNB	ZSNB(Stiffness)	ZSNB or ZSNB(RIGID)
Skewed Snubbers	SNB(VecX:VecY:VecZ:Stiffness)	SNB(0.707:0.0:0.707:1e12) or SNB(0:0:0.707:RIGID)
<b>Double Acting Snubbers in Local Axes*</b>		
SNBA	SNBA(Stiffness)	SNBA or SNBA (1E12)
SNBY	SNBY(Stiffness)	SNBY or SNBY(R)
SNBZ	SNBZ(Stiffness)	SNBZ or SNBZ(RIGID)
<b>Single Acting Translational Restraints</b>		
+X and -X	Restraint Type(Stiffness:Gap:Friction Co-efficient)	+X(1E10:35:0.35) or -X(RIGID:25)
+Y and -Y	Restraint Type(Stiffness:Gap:Friction Co-efficient)	+Y(R:50:0.2) or -Y(:15:0.28)
+Z and Z	Restraint Type(Stiffness:Gap:Friction Co-efficient)	+Z(:45) or -Z(RIGID::0.26) or +Z(:25)



<b>Double Acting Limit Stops</b>		
LIM	LIM(Stiffness:Gap:Friction Co-efficient:Xcomp:Ycomp:Zcomp)	LIM(1E12:30::0:1:0) or LIM(RIGID:50:0.4:0.707:0.707:0)
<b>Single Acting Limit Stops in Local Axes*</b>		
Axial	LIMA(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMA(1E12:30/NONE) or LIMA(RIGID:-50/NONE:0.3)
Shear Y	LIMY(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMY(1E12:30/NONE) or LIMY(RIGID:-50/NONE:0.3)
Shear Z	LIMZ(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMZ(1E12:30/NONE) or LIMZ(RIGID:-50/NONE:0.3)
<b>Double Acting Limit Stops in Local Axes*</b>		
Axial	LIMA(Stiffness:Gap:Friction Co-efficient) or LIMA(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMA(1E12:-30/50) or LIMA(RIGID:-50/20:0.3) or LIMA(R::0.2) [Lower Limit = Upper Limit = 0.0]
Shear Y	LIMY(Stiffness:Gap:Friction Co-efficient) or LIMY(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMY(1E12:-20/30) or LIMY(RIGID:20:0.3) [Lower Limit = -20 and Upper Limit = 20] or LIMY(R::0.2) [Upper Limit = Upper Limit = 0.0]
Shear Z	LIMZ(Stiffness:Gap:Friction Co-efficient) or LIMZ(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMZ(1E12:-20/30) or LIMZ(RIGID:20:0.3) [Lower Limit = -20 and Upper Limit = 20] or LIMZ(R:0/20:0.2) [Lower Limit = 0.0 and Upper Limit = 20]
<b>Double Acting Skewed Restraints</b>		
Skewed Restraints	Skew(VecX:VecY:VecZ:Stiffness:Gap:Friction coefficient:Type of Restraint)	Skew(0.707:0.707:0.0:1E12: : :R)
<b>Double Acting Skewed Restraints in Local Axes*</b>		
Axial	SKEWA(Stiffness:Gap:Friction Co-efficient:Type)	SKEWA (1E12:::R) or SKEWA (1E12:::T) R = Rotational Restraint T = Translational Restraint
Shear Y	SKEWY(Stiffness:Gap:Friction Co-efficient:Type)	SKEWY (1E12:::R) or SKEWY (1E12:::T) R = Rotational Restraint T = Translational Restraint
Shear Z	SKEWZ(Stiffness:Gap:Friction Co-efficient:Type)	SKEWZ (1E12:::R) or SKEWZ (RIGID:::T) R = Rotational Restraint T = Translational Restraint
<b>Guide</b>		
GUI	GUI(Stiffness:Gap:Friction Co-efficient)	GUI or GUI(1E12) or GUI(R:50) or GUI(RIGID:25:0.25)

<b>Spring Hangers</b>		
Hanger	Hanger(Type:No.of Hangers:All.Travel Limit:Load Variation:Short Range)	Hanger or Hanger(Grinnell :1) or Hanger(Grinnell :1: :25) or Hanger(Grinnell :1: :25:1)
Constant Support Hanger	Hanger(CONSTSUPPORT:No.of Hangers)	Hanger(CONSTSUPPORT) or Hanger(CONSTSUPPORT:2)
Rod Hanger	Hanger(ROD:No. of Hangers)	Hanger(ROD) or Hanger(ROD:1)
<b>User Hangers</b>		
User Hangers	UserHanger(Spring Rate:No.of Hangers: Cold Load:Hot Load)	UserHanger(200:1:1131) or UserHanger(200:1:0.0:1088)
<b>Force / Moment</b>		
Force	Force(Fx:Fy:Fz)	Force(1200:800:0.0)
Moment	Moment(Mx:My:Mz)	Moment(0:500:250)
<b>Threaded Joint</b>		
Threaded Joint	TJOINT	TJOINT
<b>User SIF</b>		
User SIF	UserSIF(Value)	UserSIF(100)

\* Valid and available for HOTclash 9.40 or later, checkSTRESS, checkSTRESS II & checkSTRESS Nuke Version 9.40 or later, CAEPIPE & CAEPIPE 3D+ Version 10.50 or later, Plant Design to CAEPIPE version 9.30 or later.

**Note:**

1. Stiffness, Gap and Friction Coefficient are optional values. If not defined, then it will be transferred as 1E12 lb/in i.e. RIGID, 0.0 in, and 0.0 respectively to HOTclash.
2. For SI units, the Stiffness and Gap should be specified in N/mm and mm respectively.
3. The Hanger Type, Number of hanger, Allowable Travel Limit (not applicable at this time), Load variation and Short range are optional value. If the above information are not defined, then the program will assume the following
  - a. Hanger Type = Hanger Type is selected/Specified in the HOTclash dialog box during transfer.
  - b. Number of Hanger = 1
  - c. Allowable Travel Limit = 0.00 (not applicable at this time)
  - d. Load Variation = 25 %
  - e. Short range = 1 (Use short range)
4. For SI units, the Spring Rate, Cold Load and Hot Load should be specified in N/mm, Kg and Kg respectively.

5. For defining more than one support at each support location use “;” in between support definitions. Example, +X(1e12:0.25);-X(1e12:0.25).

Hanger Types		
ABB-PBS	Fee & Mason	Nordon
Basic Engineers	Flexider (30-60-120)	NPS Industries
Berger-Paterson	Flexider (50-100-200)	Piping Services
Bergen-Paterson (L)	Fronek	Piping Tech & Products
BHEL Hyderabad	Grinell	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)	

## Appendix F

### Errors and Descriptions

This Appendix presents the list of errors, their descriptions and the necessary actions to be taken.

1. **"Invalid Entry. Starting Node number should be a Numeric value."**  
Starting Node number specified is not a numeric value. Please enter only numeric value.
2. **"Invalid Entry. Node Increment should be a Numeric value."**  
Node Increment specified is not a numeric value. Please enter only numeric value.
3. **"Invalid Entry. Starting Node number should be < 10000."**  
Node number cannot be more than or equal to 10000. Please reenter the starting Number below 10000.
4. **"Invalid Entry. Node Increment should be < 10000."**  
Node Increment value is too high. Node number cannot be more than or equal to 10000. Please reduce the Node Increment value.
5. **"Cannot determine product. Contact Program vendor for details"**  
Some of the files required for the translator either moved or deleted. Please reinstall the product or contact program vendor for details.
6. **"Cannot initialize application. Contact Program vendor for details."**  
Contact program vendor immediately.
7. **"Invalid Data Type. Expected = 'Real' Available = 'String'"**
  - a. When reading the Plant Design neutral file, one of the field values in a line is expected to be Real number format. But the field is filled with string format. Translator unable to read that field, so it will show an error message with a line number and Entire line Data and the above message. User needs to check that particular field and modify to Real number format and need to transfer the file.
  - b. For example, Outer Diameter of a pipe is expected in Real Number format like "4", But in neutral file it may be like "4inch". In this case translator will give the above error message. User needs to remove "inch" from that field and save that neutral file then need to transfer.
8. **"Improper Bore or Weight Units. Check the Neutral File."**  
Bore and Weight units entered in neutral file are invalid. Translator will expect Bore Unit as either "IN" or "MM" and Weight unit as either "KG" or "LB". If any value other than the above is specified, the translator will show error message containing the line number, Entire line and the above message. User needs to check unit used then needs to transfer.
9. **"Wrong Neutral File. No Piping Elements available to Read..."**  
Translator expects at least any one piping component present in the Neutral file. If not, it will show the above error message. User needs to check the Plant Design Neutral file.
10. **"Number of Fields available in the above Line < The Required Fields."**  
Translator expects some of the fields in a line from the Plant Design Neutral file. If not available, it will show the above said message with Line number and that particular line. User needs to check and correct that line and then need to transfer or contact program vendor for more details.
11. **"Error in Mapping Data Base. Check the Data Bases."**  
Improper changes were made to the Mapping Database. Check the entries made carefully.

**12. "Node number <Numner> is defined twice in the Neutral file. Check the neutral file and proceed."**

The node number specified above is defined two times i.e., for the same node number X, Y and Z coordinate values specified in two places in the Plant Design Neutral file. User needs to check the neutral file.

**13. "The Node number exceeded 10000. Check the 'Start Node' and 'Node Increment'."**

During rearrange of node number, the new node is crossing 10000, which is not correct. Please reduce the "Start Node number" and "Node number Increment" values and try again.