

checkSTRESS™

(for Elomatic's 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.

SI. 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 NT, load the product CD supplied by SST Systems and execute the followings steps:

- 1 Browse the CD, and 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.

checkSTRESS

Analysis Code: B31.1 Specific Gravity: 1.0

Starting Node Number: 10 Node Number Increment: 10

Default Hanger Design to be Used: Grinnell

Boundary Condition: Anchor ends if Connected to Nozzle or PipeLines

Vertical Axis: Y Direction Z Direction

Transfer 3 Way and 4 Way Valve as: Rigid Elements Pipes With Concen. Mass

Transfer Model in: SI units English Units

Transfer Flange as: Pipe With Flange Rigid Element

Transfer Supports: Using MDB Without Using MDB

Process Element, If Dia. is > 0 mm

Read Pressure and Temperature from PipeLine Tags:

Pressure: opP Temperature: opT

Read Pressure and Temperature from Model Object Tags:

Pressure: opP Temperature: opT

Read Support Type from Model Object Tags:

Support Type: sty

Merge Nodes if length of component is less than 5.0 mm

Functions:

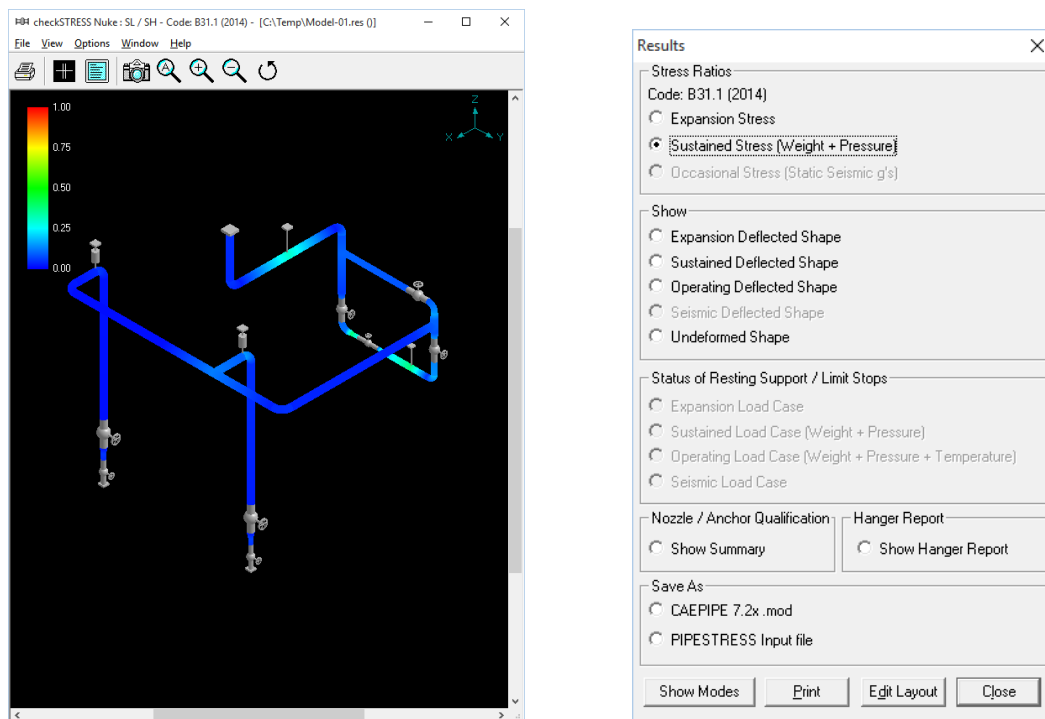
PipeLine Name Lists :

3-200-W-3420-416

- 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 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.
- 4 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.
- 5 Designers can review the Nozzle / Anchor Qualification using the Option “Show Summary”.
- 6 Hanger selected by checkSTRESS for “to be designed” hangers can be viewed by selecting the option “Show Hanger Report”.
- 7 Designers can generate the CAEPIPE mod file using the option “CAEPIPE 7.2x mod” for the Stress Department to carry out the detailed analysis.
- 8 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.
- 9 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.
- 10 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

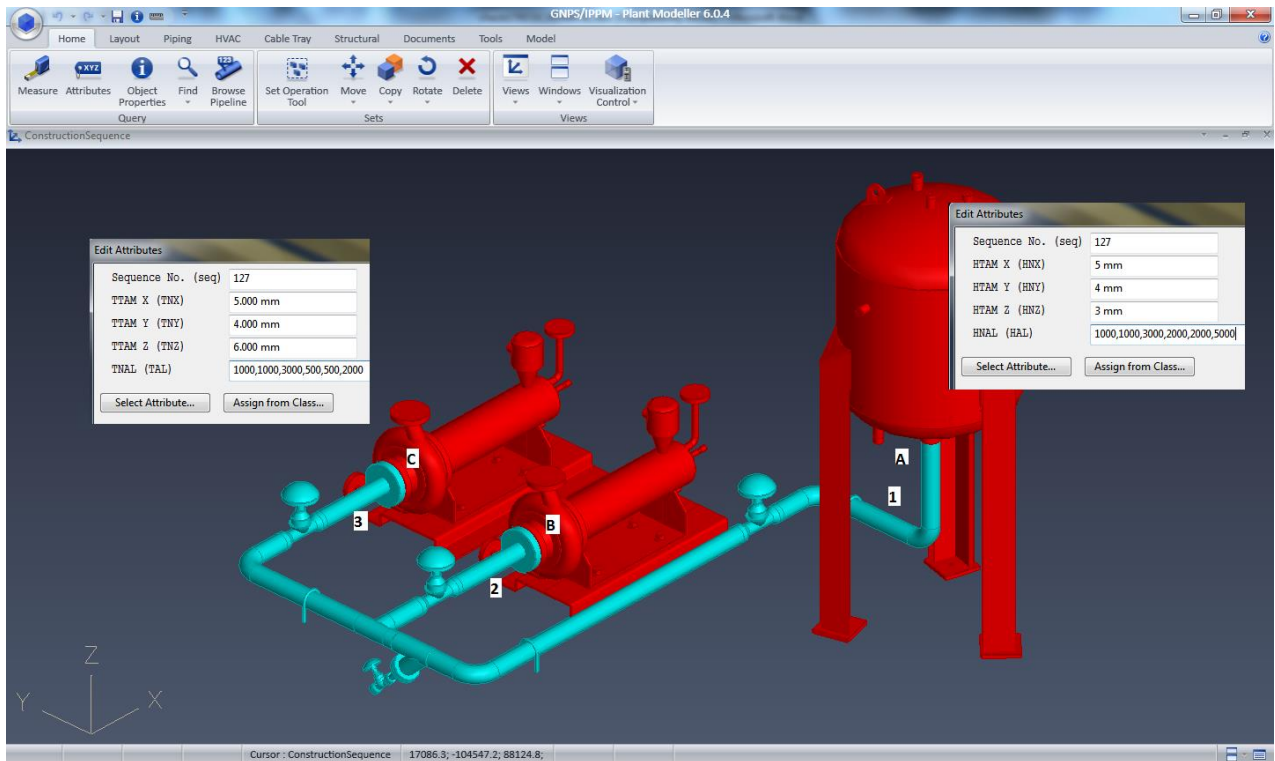
Material description entered at Material tag of corporate catalogue is extracted from the database and the material property is then obtained from the intermediate mapping db. Refer Appendix B of checkSTRESS User’s Manual for more details.

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 HNZ 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

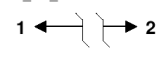
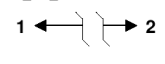
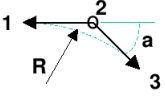

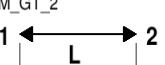
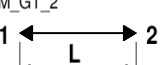
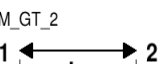
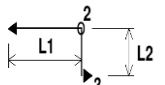
Equipment Nozzle Allowable Loads (forces and moments) provided by the equipment manufacturer or calculated using Applicable codes / Finite Element Methods can also be entered at Pipe runs as using the User Defined attributes HAL and TAL in global X, Y and Z directions at Start or End of Pipe run respectively 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.

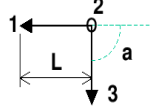
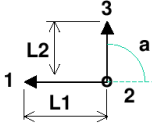
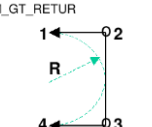
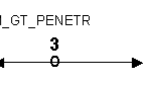
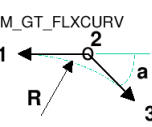
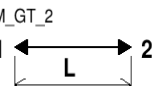
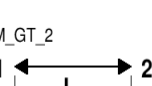
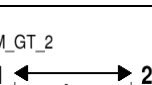
User can add / modify the Equipment Nozzle Allowable Load values through Edit Layout > Misc > User Allowable.

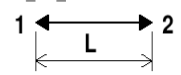
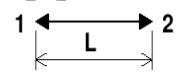
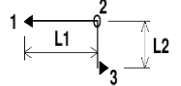
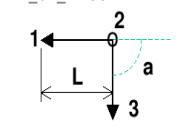
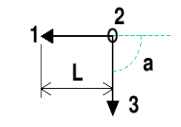
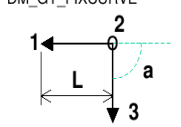
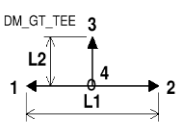
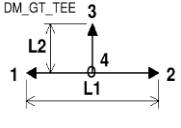
The allowable thus defined are compared against calculated loads and shown / printed in Support Load Summary outputs (can be viewed / printed through “Show Summary” option). If the calculated loads exceed the allowable, they are highlighted in red.

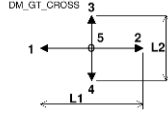
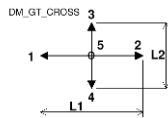
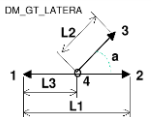
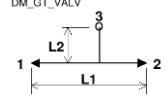
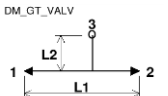
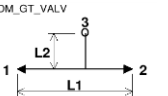
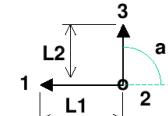
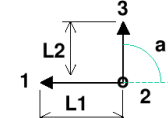
5.0 CADMATIC to checkSTRESS Component Mapping

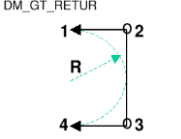
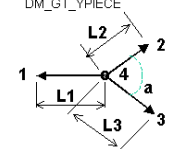
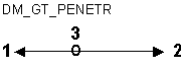
The types of components available in Cadmatic are simulated as tabulated below in checkSTRESS.

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 & 2 nd 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

Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
Fixed Angle Curves	DM_GT_FIXCURVE = True	DM_GT_FIXCURVE 	Bend	EL
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

Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
Conc. Reducer	DM_GT_2P = True & 2 nd Nominal Size > 0	DM_GT_2 	Reducer Concentric	RD
Flow meters, Adaptor, Flexible hoses	DM_GT_2P = True	DM_GT_2 	Rigid Element	RB
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	DM_GT_FIXCURVE 	Valve	VA
Straight Pipes	DM_GT_FIXCURVE = True & Keyword in corporate catalogue of CADMATIC has “*PIPE*” eg Seamless Straight Pipe	DM_GT_FIXCURVE 	Pipe	PI
Filter / Strainer	DM_GT_FIXCURVE = True	DM_GT_FIXCURVE 	Rigid Element	RB
3 Way Valves	DM_GT_TEE = True & Keyword in corporate catalogue of CADMATIC has “*VALVE*” eg Three-way valve Wafer	DM_GT_TEE 	Three Rigid Elements or Three Pipes with Concentrated Mass	3W
Welding TEE	DM_GT_TEE = True	DM_GT_TEE 	Pipes with Branch SIF (Welding TEE)	TW

Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
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
Valves	DM_GT_VALVE = True & Keyword in corporate catalogue of CADMATIC has “*VALVE*” eg Ball Valve		Valve	VA
Reduced Valves	DM_GT_VALVE = True & 2 nd 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_ASYMCCURVE = True & & Keyword in corporate catalogue of CADMATIC has “*VALVE*” eg Pressured rig-valve WDP		Valve	VA
Asymmetric Curves	DM_GT_ASYMCCURVE = True		Bend	EL

Cadmatic Component Description	Constraints	Geometric Types	checkSTRESS Component	Key Word in Neutral File
U-Piece	DM_GT_RETURN = True	 <p>DM_GT_RETURN</p> <p>1 ← 2</p> <p>R</p> <p>4 ← 3</p>	Bend	EL
Y-piece	DM_GT_YPIECE = True	 <p>DM_GT_YPIECE</p> <p>1 ← 2</p> <p>L2</p> <p>L1</p> <p>L3</p> <p>4</p> <p>3</p> <p>a</p>	Pipes with Branch SIF (Welding TEE)	TW
Sliding Sockets, Unions	DM_GT_PENETR = True	 <p>DM_GT_PENETR</p> <p>1 ← 2</p> <p>3</p> <p>4</p>	Rigid Element	RB