

checkSTRESS™

(for AVEVA's E3D/PDMS)

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	E3D/PDMS	checkSTRESS	Available
2	E3D/PDMS	checkSTRESS II	Available
3	E3D/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 E3D/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

- 1 To install checkSTRESS (for E3D/PDMS) on Windows OS, download the program from the website and execute the followings steps:
- 2 Run the program "Setup.exe" and follow the instructions as they appear on the screen.
- 3 Add the path of the installed directory to the "pmlib" path of the E3D/PDMS.
- 4 For example SET PMLLIB=C:\Progra~1\SST_Systems\checkSTRESSPDMS %1\pmlib (assuming checkSTRESS is installed in the directory C:\Program Files\SST_Systems\checkSTRESSPDMS).
Note: For the program to work properly, avoid using space or special characters while entering the path or file name.
- 5 Define an environmental variable with name as "checkSTRESS_PDMS" and the value as checkSTRESS Installation path.
For example, set the value as "C:\Progra~1\SST_Systems\checkSTRESSPDMS" (assuming that the product is installed in the directory "C:\ProgramFiles\SST_Systems\checkSTRESSPDMS").
- 6 The environment variables required by E3D/PDMS are also required for checkSTRESSPDMS. The E3D/PDMS environment must exist as outlined in the appropriate E3D/PDMS Installation Guide.
- 7 Load E3D/PDMS and enter the Lexicon Module as a user who has Read-Write access to DICT Database and run the data file checkSTRESS.DAT, to create the UDA: SUPCODE for ATTA elements in PDMS and ANCI element in E3D, UDA :NX, :NY, :NZ and :NAL for Nozzle elements, UDA: :HNX, :HNY, :HNZ, :TNX, :TNY, :TNZ, :HNAL and :TNAL for Branch elements.

How to use checkSTRESS?

- 1 Add the ZONE/PIPES/BANCHES need to be checked for stress using "Add CE" or CE Members" button.
- 2 Branches selected for transfer need to be checked for the following.
 - TEMP, PRES attributes of the BRAN element
 - Each branch should have a valid PH OD
 - Each branch in the network should have a valid HSTU
 - Each branch should have a component
 - It is recommended to perform "DATA CONSISTENCY CHECK" for all Pipes/Branches selected for transfer.
- 3 For transferring the support details, ATTA's have to be created at support locations and the UDA (:SUPCODE) should be set a value as given in the field #1 (i.e., "PdSupport") of the access db "SupportType.mdb".
- 4 To use checkSTRESS (for E3D/PDMS), type "show !!checkSTRESS" on the command line of E3D/PDMS Design Module. The following form appears,

checkSTRESS

File Name: %PDMSWK%\Temp01

Input Units used in Project while entering

Pressure: Bar Temperature: Celsius Weight: Kilogram

Analysis Code: B31.1

Default Hanger Design: Grinnell

Transfer Model in: SI Units

Specific Gravity: 1

Starting Node number: 10

Node Number Increment: 10

Process elements, if dia is >: 0 mm

Identify Support ATTA's with ATTYPE: SUPP

Vertical Axis

☐ Y Direction

☒ Z Direction

Transfer 3way and 4way Valve as

☒ Rigid Elements

☐ Pipes with Concentrated Mass

Transfer Supports

☒ Using MDB

☐ Without Using MDB

Transfer Flange as

☒ Pipe with Flange

☐ Rigid Element

Boundary Condition

☒ Anchor Ends if connected to Nozzle

☐ Anchor all Free ends

WORL *

SITE STABILIZER

ZONE PIPES

PIPE 100-B-1

BRAN -> 100-B-1-B1

GASK 1

FLAN 1

CE Members

Add CE

Remove

Remove All

Apply

Cancel

- Specify/Change the Name of the file if you wish. Do not leave a blank space while giving the file name / path. Program checks and generates error, if finds a space in the file name / path.
- Select the Input units used in Project while entering “Pressure”, “Temperatures” and “Weight” using the options as shown in the figure above. Please note, selecting wrong input units used in project will results in incorrect output.

- Select the piping code to be used for the analysis of the model from the list box.
- Enter/modify the Static Seismic g's. Available only for checkSTRESS Nuke.
- 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.
- Enter the Specific Gravity of operating fluid with respect to water. By default the specific gravity is set to 1.0.
- 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. For details on transferring support from Plant Design to checkSTRESS, refer Appendix D of checkSTRESS User's manual (checkSTRESS.pdf).
- When the "Identify Support ATTA's with ATTYPE attribute set as" is entered with a Keyword, then it will transfer support locations and its information from E3D/PDMS, only when the attribute "ATTYPE" is filled with the string that matches the Keyword entered in the form. On the other hand, leaving this text box empty (unfilled) will transfer all the ATTA from the E3D/PDMS for the selected Pipes/Branches without checking its "ATTYPE" attribute. For example, if the Keyword is specified as "SUPP" then the program will include ATTA's for transfer, only when the ATTYPE attribute is filled with the same Keyword "SUPP", otherwise, it excludes them from transfer.
- Select the units to be used for the checkSTRESS model.

Selecting the option "SI Units" transfers the Pipes/Branches details in SI units i.e. Length related dimensions such as OD, Nominal Size etc in mm, Temperature in Deg C, Pressure in bar, Weight in Kg, Density in Kg/m³, Translational Stiffness in N/mm, Rotational stiffness in N-m/deg to the neutral file.

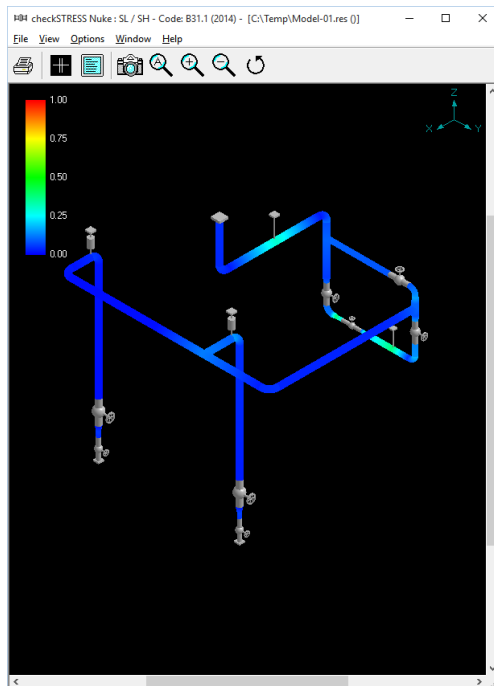
Similarly Selecting the option "ENGLISH Units" transfers Pipes/Branches details in ENGLISH units i.e. Length related dimensions such as OD, Nominal Size etc in Inch, Temperature in Deg F, Pressure in psi, Weight in lb, Density in lb/in³, Translational Stiffness in lb/in, Rotational Stiffness in lb-in/deg to the neutral file.

- Select the Boundary Conditions to be used from the form shown above. Refer the section titled "Reference" for details.
- 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.
- Specify how the 3 Way and 4 Way Valves to be transferred to checkSTRESS. Refer the section titled "Reference" for details.

- Navigate to the required ZONE/PIPE/BANCH through the member list as shown in the figure above.
- Press the button “Add CE” to include and to perform the Stress Check for the selected ZONE/PIPE/BANCH.
- User can also use the button “CE Members” to include all the members of SITE/ZONE/PIPE and then exclude a few members using the “Remove” button if required. For example, if you wish to add 8 PIPES out of 10 PIPES under a particular ZONE, navigate to the ZONE and press the button “CE Members”. Highlight the PIPE (one at a time) in the included list and press the button “Remove” to exclude them.
- 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 E3D/PDMS 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/CAESAR II 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.

- Press the button “Apply” to perform the Stress Check.



First-Level Checks

Stress Ratios
Code: B31.9 (2020)

☒ Sustained Stress (Weight + Max. Pressure)
☐ Expansion Stress (Max. Thermal Range)
☐ Occasional Stress (Max. of Wind/Seismic/Dynamic Loads)

Show

☐ Sustained Deflected Shape (Weight + Max. Pressure)
☐ Expansion Deflected Shape (Thermal T1 - Tref)
☐ Operating Deflected Shape (W + P1 + T1)
☐ Seismic Deflected Shape (Static Seismic)
☐ Wind Deflected Shape (Wind 1)

Nozzle / Anchor Qualification: ☐ Show Summary

Hanger Report: ☐ Show Hanger Report

Status of Resting Supports / Limit Stops

☐ Sustained Load Case (Weight + Max. Pressure)
☐ Operating Load Case (W + P1 + T1)

Modal Analysis
☐ Show Frequencies

Export Model to PIPESTRESS

Export Deflected Shape Edit Layout Print Exit

- 5 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. Refer to Appendix B for details on adding Spectrum.
- 6 Figure above shows the stress contour plot for the Expansion case. Sustained Stress plot can be seen by selecting the option “Sustained Stress (Weight + Pressure)” as shown in the figure right above.
- 7 Designers can also plot and view the deflected shapes of the piping system under different loading conditions using the options available in the heading titled “Deflected Shape” as shown in the figure on the right above.
- 8 Designers can review the Nozzle / Anchor Qualification using the Option “Show Summary”.
- 9 Hanger selected by checkSTRESS for “to be designed” hangers can be viewed by selecting the option “Show Hanger Report”.
- 10 Designers can generate the CAEPIPE mod file using the option “Edit Layout > File > Save or Save as” for the Stress Department to carry out the detailed analysis including stress report preparation.
- 11 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.
- 12 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.

Limitations

The following are not transferred to checkSTRESS from E3D/PDMS at this time. However user can add these in checkSTRESS through Edit Layout->Misc->Section

- 1 Corrosion allowance and Mill tolerance.
- 2 Lining Density and Lining Thickness.

Reference

Loads

Temperature and Pressure values entered at Pipe/Branch level via TEMP and PRES attribute shall be transferred to checkSTRESS. Hence, user should fill these attributes with appropriate values depending upon the Units of transfer. I.e., If you wish to transfer the model in SI units, then the value entered 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 values in checkSTRESS through Edit Layout->Misc->Loads.

Fluid Density

Even though the provision is available in E3D/PDMS Propcon Database to enter the Fluid density, most of the users do not use such facility. Hence, provision is given to 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 Propcon Database through the “Cweight” attribute. If defined/available in the Propcon Database, the program extracts the information and transfer to checkSTRESS. Care should be taken while filling the “Cweight” attribute in Propcon. i.e., the values should be specified in Kg / lb, to transfer the same in SI / English units respectively.

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

Wall Thickness

Since the Wall thickness of the Piping components is generally not available in E3D/PDMS, they are set to 0.0 by default. However, user can transfer the Wall Thickness information to checkSTRESS properly from E3D/PDMS (if they are specified) by modifying the variables defined in the program. Refer **Appendix A** for details on modifying the program to extract Wall Thickness from E3D/PDMS.

User can also specify wall thickness of piping components through Mapping DB (Code.mdb) supplied along with the project. For details refer to **Appendix C** of checkSTRESS User's Manual (checkSTRESS.pdf).

In addition to the above options, user can also modify the Wall thickness of the piping components directly in checkSTRESS through Edit Layout->Misc->Section.

OD and Nominal Size

OD and Nominal Size are extracted from the Paragon Database of E3D/PDMS. For reducers, the arrive OD and Thickness shall be transferred to OD1 and Thk1 of the element in checkSTRESS and the leave OD and Thickness shall be transferred as OD2 and Thk2 of the element in checkSTRESS.

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

Boundary Conditions

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 Branch and if that Branch 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 Branch and that Branch 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” all the ends of the Branches 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

Supports locations and its information are included for transfer from E3D/PDMS when the following conditions are met.

Step 1: Include/Exclude ATTA's for Transfer

PDMS

Define ATTA and fill the attribute “ATTTYPE” with an appropriate Keyword. Use the same Keyword in the text box “Identify Support ATTA's with ATTYPE attribute set as” (as shown in figure above). Mismatching of Keywords will exclude the ATTA's from transfer (or) fill the attribute “ATTTYPE” with an appropriate Keyword and leave the text box “Identify Support ATTA's with ATTYPE attribute set as” empty (unfilled) to include all ATTA's for Transfer.

Everything 3D (E3D)

Create ANCI using the Support Design module of E3D at Support locations. Input the Support Type information at the UDA :SUPCODE of ANCI.

Step 2: Transfer Support Details

The program performs the following for transferring the support details to checkSTRESS.

- Checks for User Defined Attribute (UDA) “:SUPCODE” at ATTA for PDMS. If the User Defined Attribute :SUPCODE is NOT available, then the program will read the support details from the “STEXT” attribute of ATTA from PDMS and transfer the same to checkSTRESS.
- Check for User Defined Attribute (UDA) “:SUPCODE” at ANCI for E3D. If available, then the program will read the details of support from this attribute and write them to the neutral file. If NOT available, then the program will write the Support Type as “HS1” by default in the Neutral file.
- For both PDMS/E3D, UDA :SUPCODE should be set a value as given in the field #1 i.e., “PdSupport” of the access db “SupportType.mdb”, when the option “Use Mapping DB” for transferring the support details to checkSTRESS is selected. Otherwise, the attribute should be filled with values as specified in **Appendix E** of checkSTRESS User's Manual supplied along with the product. For details on transferring supports “Without Using Mapping DB”, refer to Appendix D of checkSTRESS User's Manual (checkSTRESS.pdf).

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

Material

Material description entered at XTEXT attribute of SMTE is extracted from the Paragon Database and the material property is then obtained from the intermediate mapping db. For more details, refer **Appendix B** of checkSTRESS User's Manual supplied along with the product.

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 UDA :NX, :NY, and :NZ in global X, Y and Z directions respectively at Equipment Nozzle where the piping layout (selected for transfer) is connecting to. These values are then transferred to checkSTRESS. TAM values should be defined in "mm" for E3D/PDMS projects in SI units and in "Inch" for E3D/PDMS project in English units.

If the Equipment Nozzle to which the Piping Layout is connecting to is not modeled in E3D/PDMS, then these TAM values can be entered at UDA :HNX, :HNY and :HNZ or :TNX, :TNY and :TNZ at E3D/PDMS Branch corresponding to Head or Tail of pipe run. In other words, TAM at starting of the Pipe run should be entered at :HNX, :HNY and :HNZ of E3D/PDMS Branch. On the other hand, TAM at the end of the Pipe run should be entered at :TNX, TNY and TNZ of E3D/PDMS Branch.

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 be entered at UDA :NAL in global X, Y and Z directions at Equipment Nozzle in the format FX, FY, FZ, MX, MY and MZ. These values should be separated using ",". Please note, 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.

If the Equipment Nozzle to which the Piping Layout is "Connecting To" is not modeled in E3D/PDMS, then these Equipment Nozzle Allowable Loads can be entered at UDA :HNAL or :TNAL at E3D/PDMS Branch corresponding to Head or Tail of the Pipe run. In other words, Allowable Loads at starting of the Pipe run should be entered at :HNAL of E3D/PDMS Branch. On the other hand, Allowable Loads at the end of the Pipe run should be entered at :TNAL of E3D/PDMS Branch.

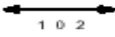
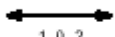
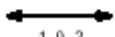
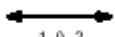
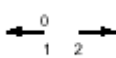
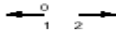
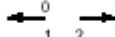
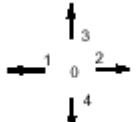
Example:

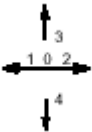
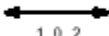
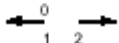


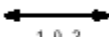
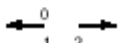
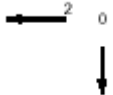
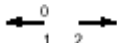
For Nozzle, enter the values of FX, FY, FZ, MX, MY and MZ as ":NAL '3000,3000,6000,500,500,2000'"

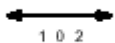

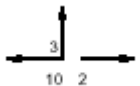
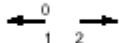
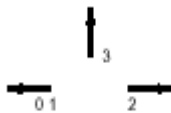
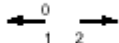
For E3D/PDMS Branch Head/Tail, enter the values of FX, FY, FZ, MX, MY and MZ as ":HNAL '3000,3000,6000,500,500,2000'". The allowables thus defined are compared against calculated loads and shown / printed in Support Load Summary outputs (can be viewed / printed through "Show Summary" option). If the calculated loads exceed the allowables, they are highlighted in red.


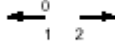
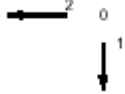
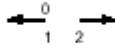

E3D/PDMS to checkSTRESS component Mapping



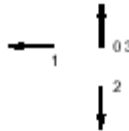




The types of components available in E3D/PDMS are simulated as tabulated below in checkSTRESSPDMS.








E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Cap			
Butt weld		Rigid Element	RB
Cap compression		Rigid Element	RB
Cap Screwed		Rigid Element	RB
Cap Socket Weld		Rigid Element	RB
Closure			
Pipe Block		Rigid Element	RB
Pipe Block Variable Length		Rigid Element	RB
Coupling			
Nipple – Screwed, Coupling Compression, Coupling Screwed, Coupling Socket weld, Butt weld Elbolet, Socket Weld Elbolet, Screwed Elbolet		Reducer Concentric	RD
Cross			
Butt Weld, Socket Weld, Compression, Flanged, Screwed		Four Pipes with Branch SIF (Welding TEE)	CR


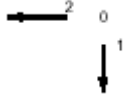
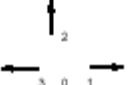


E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Set on, Set on Reinforced		Four Pipes with Branch SIF (Welding TEE)	CR
Blind Flange			
Flange – Blind		Rigid Element or Pipe with Flange	FL
Flange - Reducing Concentric		Rigid Element or Pipe with Flange	FL
Flange Reducing Eccentric		NA	NA
Fixed Length Tube			
Fixed length Pipe with Flanged ends and without Flanged ends.		Pipe	PI
Lap Joint and Stub End			
Lap Joint Ring Loose		Rigid Element or Pipe with Flange	FL
Lap Joint Stub End Loose		Rigid Element or Pipe with Flange	FL
PCOMponent			
Block – Angle		Pipe	PI
Expansion Bellows, Flame Trap, Flexible Hose, Hose Coupling, Non – Category item,		Pipe	PI

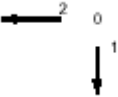



E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Restrictor Plate, Tundish			
Plug		Pipe	PI
Sight Glass		Pipe	PI
Slip Plate, Slip Ring, Spectacle Blind		Pipe	PI
Reducer Concentric			
Reducer Concentric Butt Weld, Fabricated from Plate, Swaged from Pipe, Compression, Flanged, Fabricated from Plate Flanged, Swaged from Pipe Flanged, Nipple, Screwed, Socket Weld Bush, Socket Weld & Special Reducing Flange		Reducer Concentric	RD
Reducer Concentric with a Butt Weld Connection, Fabricated Plate Connection, Connection Swaged from Pipe, Connection Flanged, Connection Fabricated from Plate Flanged, Connection Swaged from Pipe Flanged, Screwed Connection and Connection Socket Weld		Reducer Concentric	RD
Reducer Eccentric			
Reducer eccentric Buttweld, Fabricated from Plate and Swaged from Pipe.		Reducer Eccentric	ER

E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Reducer Eccentric with a Connection Butt Weld, Connection Fabricated from Plate, Connection Swaged from Pipe, Connection Flanged, Connection fabricated from Plate Flanged and Connection Swaged from Pipe Flanged	NA	NA	NA
Eccentric Screwed, Flanged, Fabricated from Plate Flanged and Swaged from Pipe Flanged		Reducer Eccentric	ER
TRAP			
Trap – In Line		Pipe	PI
Trap – Angle		Pipe	PI
Trap – Offset, Trap – Return	NA	NA	NA
UNION			
Union – Screwed, Socket Weld		Rigid Element	RB
TEE or OLET			
Olet - Half Coupling, Nipolet (Screwed, Plain Ended), Sockolet, Thredolet, Weldolet, Tee- Set-on, Set-on Reinforced and Tee Pulled		Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)	TU/TW/TF/TO

E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Olet – Latrolet (Butt Weld, Screwed & Socket Weld)		Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)	TU/TW/TF/TO
Olet – Instrument Flanged, Tee – Butt Weld, Compression, Flanged, Screwed, Socket Weld, Swept Branch Butt Weld, Swept Branch Flanged, Swept Branch Compression and Swept Branch Socket Weld		Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)	TU/TW/TF/TO
Valve			
Valve – Angle (Pressure Reducing, Relief/Vent,		Valve	VA
Valve – Ball, Basic, Butterfly, Cock, Diaphragm, Gate, Globe, Needle, Plug, Pressure Reducing, Relief/Vent and Slide		Valve	VA
Valve – Check		Valve	VA
VENT			
Rupture Disk		Rigid Element	RB
Four Way Valve			
Valve – 4 Way		Four Rigid Elements or Four Pipes with Concentrated Mass	4W

E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Three Way Valve			
Valve – 3 Way		Three Rigid Elements or Three Pipes with Concentrated Mass	3W
INSTRUMENT			
Instrument, Orifice Plate, Restrictor Plate and Rupture Disk		Rigid Element	RB
Instrument Angle		Rigid Element	RB
Instrument Dial and Dial Flanged		Rigid Element	RB
Valve – Angle Relief/Vent, Angle Pressure Reducing, Angle Control, Angle Control Square Indicator, Angle Control Motorised Indicator, Angle Control Hand Indicator		Rigid Element	RB
Control, Control Square Indicator, Control Motorised Indicator, Hand Indicator, Pressure Reducing and Relief/Vent		Rigid Element	RB
3 Way Control, 3 Way Control Square, 3 Way Control Motorised Indicator and 3 Way Control Hand Indicator		Three Rigid Elements or Three Pipes with Concentrated Mass	3W

E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
4 Way Control Square Indicator, 4 Way Control, 4 Way Control Motorised Indicator, 4 Way Control Hand Indicator		Four Rigid Elements or Four Pipes with Concentrated Mass	4W
ELBOW or BEND			
Elbow – Butt Weld(90 and 45 Deg), Screwed with Female Ends, Elbow Compression, Screwed with Male Ends, Elbow Socket Weld, Elbow Reducing, Bend – Mitre, Lobster Back Butt weld and Lobster Back Bend and Pulled		Bend	EL
Butt Weld with a Connection, Compression with a Connection, Screwed Female with a Connection, Socket Weld with a Connection, Bend Flanged with a Connection, Bend – Mitre with a Connection Flanged, Butt Weld Bend – Mitre with a Connection, Lobster Back with a Connection and Pulled with a Connection		Pipes with Branch SIF (Welding TEE)	TW
Butt Weld 180 Deg Return, Flanged 180 Deg Return and Pulled 180 Deg Return		Bend	EL
FILTER			
Filter/Strainer- Straight Through		Rigid Element	RB

E3D/PDMS Component Description	Connection Points	Component in checkSTRESS	Keyword used in Neutral File
Filter / Strainer Angle		Rigid Element	RB
Flange			
Flange Blind, Flared/Loose Backing, Flange Backing, Flange Reducing Concentric, Screwed, Slip On, Slip On with 'J' Type Weld, Socket Weld and Weld Neck	 or 	Rigid Element or Pipe with Flange	FL
Orifice Slip On Flange and Orifice Weld Neck		Rigid Element with Tapping	OF

Appendix A

Specifying Wall Thickness

This section explains how to extract the wall thickness for E3D/PDMS piping components. To specify the wall thickness for piping components like Pipe, Olet, Tee, Elbow, Reducer and Cross, navigate to “checkSTRESS” installed path and locate the folder “Setup”. Navigate to the folder “Objects” and then open the corresponding .pml object files in which the thickness needs to be specified.

Locate the variables “!This.WallThickness[1] and “!This.WallThickness[2]” in the file and modify the values to suit the project requirement. By default these variables are assigned to ‘0.00’.

Catalogue Parameter

If wall thickness is defined using Catalogue Parameter of SCOM in E3D/PDMS, then the variable can be modified to read the value from Catalogue Parameter. For example, if the Wall Thickness is specified in Catalogue Parameter 3, modify the lines as follows.

```
!This.WallThickness[1] = !DbRef.catref.Param[3].String()  
!This.WallThickness[2] = !DbRef.catref.Param[3].String()
```

Design Parameter

If Design Parameter (DESPARAM) is used to specify the Wall Thickness, then the variable can be modified to read the value from Design Parameter. For example, if the Wall Thickness is specified in Design Parameter 3, modify the lines as follows.

```
!This.WallThickness[1] = !DbRef.spref.Desparam[3].String()  
!This.WallThickness[2] = !DbRef.spref.Desparam[3].String()
```

User Defined Attributes (UDA) at Specification component

If User Defined Attribute is used to specify the Wall thickness of the piping component at SPCO level, modify the variable listed below to read the value from UDA. For example, if the Wall Thickness is specified in the variable “WallThickness” of SPCO element, modify the lines as follows.

```
!This.WallThickness[1] = !DbRef.Spref.:WallThickness.string()  
!This.WallThickness[2] = !DbRef.Spref.:WallThickness.string()
```

User Defined Attributes (UDA) at Piping Component

If User Defined Attribute is used to specify the Wall thickness at piping component, modify the variable listed below to read the value from UDA. For example, if the Wall Thickness is specified in the variable “WallThickness” of piping element, modify the lines as follows.

```
!This.WallThickness[1] = !DbRef.:WallThickness.string()  
!This.WallThickness[2] = !DbRef.:WallThickness.string()
```

Appendix B

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.

Spectrum Options

Abscissa: ☒ Frequency (Hz) ☐ Period (Sec)

Ordinate: Displacement ☐ inch ☐ mm; Acceleration ☒ in/sec2 ☐ mm/sec2 ☐ g's

Interpolation: ☒ Linear ☐ Log

OK Cancel

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.

checkSTRESS Nuke: ...

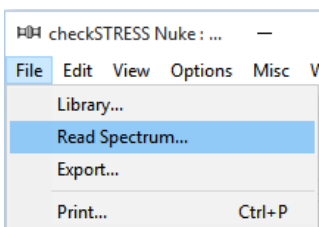
File Edit View Options Misc Window Help

#	Name	#	Frequency (Hz)	Acceleration (in/sec2)
1	X spectrum	1	1	0.0025
2	Y spectrum	2	50	0.0025
3	Z spectrum	3	62	0.0025
4		4	75	0.0025
		5	100	0.0025
		6		

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.

- For each spectrum, on the first line, type the Name of the spectrum (up to 31 chars). On the second line, type units for Abscissa (X-axis) and Ordinate (Y-axis) axes, separated by a space.
- After these two lines, on each line, you can input a value pair. You can input as many of these value pair lines as required.
- Each spectrum should be saved to a separate text file. More than one spectrum should not be input in one text file as it cannot be read into checkSTRESS.
- 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.

Spectrum Load [X]

X spectrum [X spectrum] Factor 1.000

Y spectrum [Y spectrum] Factor 1.000

Z spectrum [Z spectrum] Factor 1.000

Mode Sum

- ☒ SRSS
- ☐ Closely spaced
- ☐ Absolute

Direction Sum

- ☒ SRSS
- ☐ Absolute

OK Cancel Reset