checkSTRESS™
[for Intergraph's PDS]

Introduction

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

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

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

This document describes the common features of checkSTRESS products currently available for PDS.

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

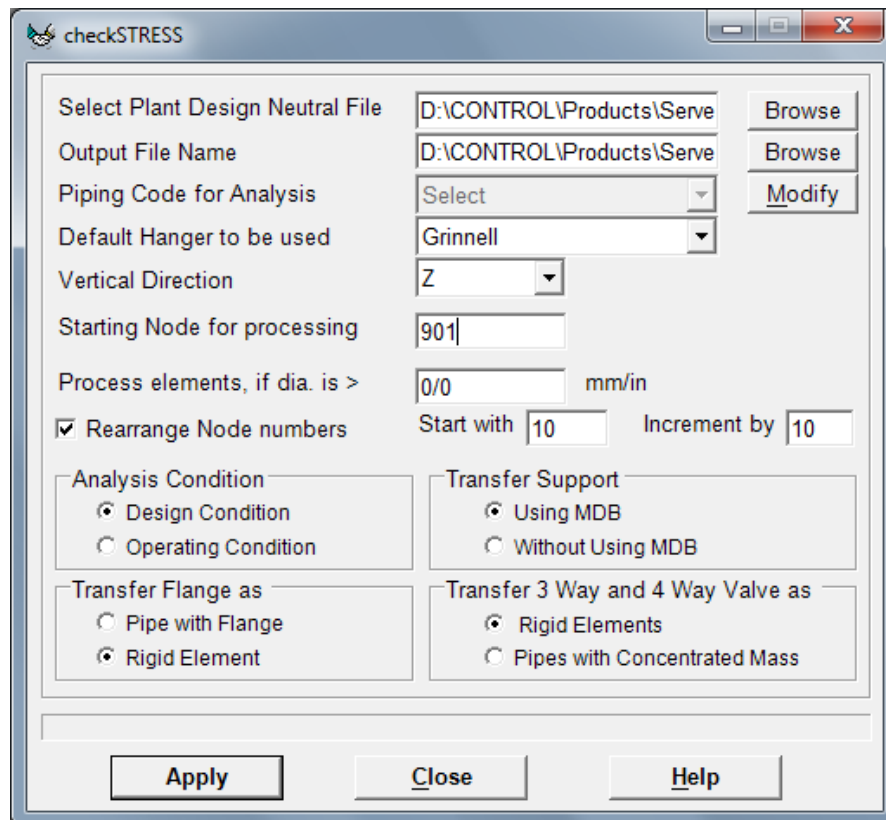To install checkSTRESS on Windows OS, load the product CD supplied and execute the followings steps:

1. Browse the CD, and run the program "Setup.exe" and follow the instructions as they appear on the screen.

2. Define an environmental variable with name "checkSTRESS_PDS" and the value as checkSTRESS Installation path. For example, set the value as "C:\Progra~1\checkSTRESSNukePDS (assuming checkSTRESS is installed in the directory "C:\Program Files\checkSTRESSNukePDS).

Note:

Use only short names while defining the Environmental variable and do not use 'space' or 'special characters' while defining the absolute path.

How to use checkSTRESS?

1. Select the Neutral file generated out of PDS.

2. Specify the name of the output file.

3. Select the piping code to be used for the analysis from the list box.

4. Enter the Starting Node number and Node increment value in the appropriate field by selecting the option "Rearrange Node Numbers". By default, the Starting Node number and Node increment are set to 10 and 10 respectively.

5. By default, specific gravity of the fluid with respect to water is set to 1.0. User can modify the same through Edit Layout->Misc->Loads after launching checkSTRESS.

6. Select Default Hanger to be used in checkSTRESS. This selected Hanger type will be used in checkSTRESS, 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).

7. Specify the Starting Node number (equivalent to minimum anchor node specified in the defaults.dat while extracting neutral file from PDS) for processing network. This is set to 901 by default. User can change this node number to match with the minimum anchor node specified in defaults.dat file. If the user is not sure about the node number, this can be left as 901 or 0.

8. If the user wishes to eliminate those elements whose Nominal size (NS) is less than 50mm / 2in, then they can specify the value as "50/2" 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.

9. Specify how the Flange to be transferred to checkSTRESS.

10. Specify how the 3 Way and 4 Way Valves are to be transferred to checkSTRESS.

11. Choose the Global Vertical Axis for checkSTRESS from the Vertical Direction option. By default, "Z Axis" shall be taken as Global Vertical Axis.

12. Press the button "Apply" to perform the Stress Check.
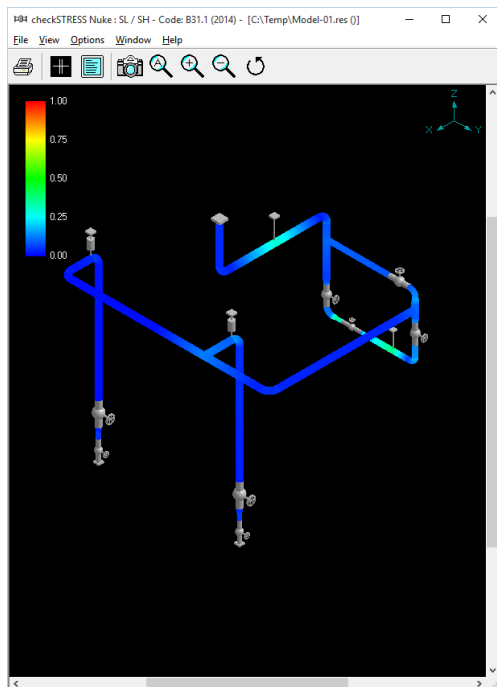
13. Figure above shows the stress contour plot for the Sustained case. Expansion Stress plot can be seen by selecting the option "Expansion Stress" as shown in the figure right above.

14. 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.

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

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

17. Designers can generate the CAEPIPE mod file using the option "CAEPIPE 7.2x mod" for the Stress Department to carry out the detailed analysis including stress report preparation.

18. 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.

19. 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.

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

Limitations

The following are not transferred to checkSTRESS from PDS 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.

Procedure for extracting neutral files from PDS software

This version of checkSTRESS is compatible with PDS versions released after May 1998 and upto PDS Version 8.xx.

1. The present version of the checkSTRESS is supplied with the PDSTRESS.TBL and DEFAULTS.DAT.

2. Replace the files PDSTRESS.TBL and DEFAULTS.DAT currently available in the directory PDS_Installed_Dir\win32app\INGR\pdstress\dat\ with the files PDSTRESS.TBL and DEFAULTS.DAT supplied with this version.

3. Replace the PDS to STRESS Map path available in EXTERNAL FILES block of DEFAULTS.DAT as shown below with your current path of PDSTRESS.TBL (path in which PDSTRESS.TBL is available) by opening the file DEFAULTS.DAT.

EXTERNAL FILES
!Note: maximum 5 files, 40 characters maximum for each file
! 123456789012345678901234567890123456789 0
e:\win32app\ingr\pdstress\dat\pdstress.tbl   !PDS TO STRESS MAP (path is user defined)
ENDOF EXTERNAL FILES

4. *Refer to PDS Stress Interface User's Guide for more details on how to modify the option files PDSTRESS.TBL and DEFAULTS.DAT for adding new components.*

5. For User-defined components, the PDS Administrator has to ensure that the corresponding details are added in PDSTRESS.TBL and DEFAULTS.DAT. For new component categories, the PDS Administrator has to send the details of such component categories to SST Systems / InfoPlant, so that SST Systems / InfoPlant could incorporate such details in the next release of the checkSTRESS.

6. If the support detail is not specified at support location then it will be transferred as "Hanger" to checkSTRESS. Refer to Appendix D and Appendix E of checkSTRESS User's Manual for details on transferring supports to checkSTRESS.

7. Load the PDS software and click on "Stress Interface" and type in the name of the pipeline and name of the Neutral File in the form and click "Apply". This will create a Neutral File at the specified location.

8. Thermal Anchor Movement (TAM) values extracted and written to stress neutral file through "LNOD" or "MLNOD" with type value as "MV" in global X, Y and Z directions can be transferred to checkSTRESS. TAM values should be defined in "mm" for PDS projects in SI units and in "Inch" for PDS project in English units.

   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.

9. Equipment Nozzle Allowable Loads (forces and moments) provided by the equipment manufacturer or calculated using Applicable codes / Finite Element Methods should be extracted and written through "LNOD" and "MLNOD" with type as "AL" in global X, Y and Z directions 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.

   User can also add / modify the Equipment Nozzle Allowable Loads through Edit Layout > Misc > User Allowables.

   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.