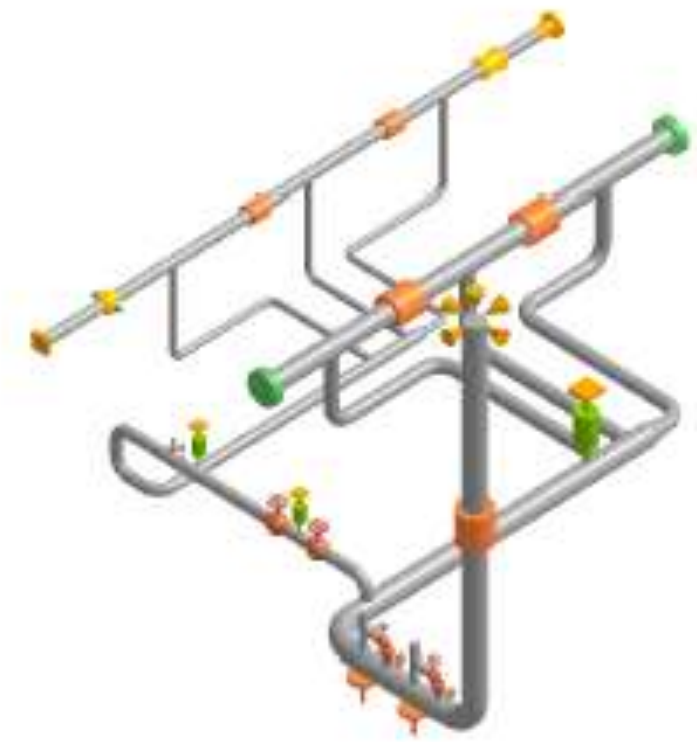
CAEPIPE 3D+™

User's Manual

[for 3D Piping Designers and Pipe Stress Engineers]



From the CAdvantagE™ Library

Innovative. Efficient. Effective.

Disclaimer

Please read the following carefully:

This software and this manual 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 the manual or the functioning of the software and the accuracy, correctness and utilization of its calculations. Users must carry out all necessary tests to assure the proper functioning of the software and the applicability of its results. All information presented by the software is for review, interpretation, approval and application by a Registered Professional Engineer.

CAEPIPE and CAdvantagE are trademarks of SST Systems, Inc. and SST India Pvt. Ltd. All other product names mentioned in this document are trademarks or registered trademarks of their respective companies/holders.

For Queries, contact

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

Tel: +1 408 452 8111
Email: info@sstusa.com
www.sstcae.com

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

Tel: +91 80 4215 2769
Email: sales@sstindia.co.in
www.sstcae.com

Table of Contents

Introduction **1**
Why CAEPIPE 3D+? 1
Overview of CAEPIPE 3D+ 4
Flexible Usage of CAEPIPE 3D+ 4
Benefits of CAEPIPE 3D+ 5
CAEPIPE 3D+ as Piping Designers' Tool **6**
Basic Pipe Stress Concepts 6
Recommended Procedure for first-level stress checks by 3D Piping Designers 8
Sample Pipe Stress Problems and Solutions 11
CAEPIPE 3D+ for PDMS/E3D **33**
Installing and Configuring Program 33
How to use CAEPIPE 3D+? 33
Limitations 37
Reference 37
Specifying Wall Thickness 41
CAEPIPE 3D+ for CADMATIC **42**
Installing and Configuring Program 42
How to use CAEPIPE 3D+? 42
Limitations 45
Reference 45
CAEPIPE 3D+ for PCF **49**
Installing and Configuring Program 49
How to use CAEPIPE 3D+? 49
Limitations 51
Reference 51
Export PCF from MEP Systems using CAEPIPE 3D+ Add-In from Revit 55
Exporting PCFs from MEP Systems 56
Reference (Revit Add-In) 59
Export PCF from Fabrication Module of Revit 61
CAEPIPE 3D+ for PDS **66**
Installing and Configuring Program 66
How to use CAEPIPE 3D+? 66
Reference 68
Appendix A **69**

PDMS/E3D to CAEPIPE 3D+ Component Mapping 69
CADMATIC to CAEPIPE 3D+ Component Mapping 75
PCF to CAEPIPE 3D+ Component Mapping 80
Appendix B 82
Material 82
Material DB Modification / Creation 82
Sample “Code” DB 83
Sample “Material DB” (B315.mdb) 84
Modification of “Config.ini” File 86
Appendix C 87
Schedule Table 87
Schedule Table Creation / Modification 87
Appendix D 88
Supports 88
With the use of mapping DB 88
Without using Mapping DB 88
Appendix E 95
Possible Restraints Types and Hangers 95
Appendix F 99
Errors and Descriptions 99

Introduction

Why CAEPIPE 3D+?

It is common practice worldwide that piping layout designers route pipes in 3D Plant Design systems with consideration given mainly to space constraints, process and flow constraints (such as pressure drop) and other requirements arising from constructability, operability and reparability. Most often, while routing piping systems, pipe stress requirements are not sufficiently considered.

In order to meet pipe stress requirements, pipe stress engineers have all along felt the following needs.

Need 1: Instead of creating pipe stress models by manually entering layout, pipe sizes and materials, supports, loads etc. in pipe stress software such as CAEPIPE, such stress models should be generated directly from 3D Plant Design systems.

Need 2: The layouts of the stress models so generated from 3D Plant Design systems should have already gone through first-level stress compliance checks performed by 3D piping designers, so that the piping layouts received by pipe stress engineers are already flexible enough to absorb expansion/contraction of pipes due to thermal loads.

Need 3: In addition to the traditional “clash checks” carried out on 3D plant model, piping designers should be able to check for “interference” when piping is deformed under “hot operating condition”.

Need 4: Immediate visualization of the 3D plant model while performing detailed analyses would immensely help pipe stress engineers to identify (a) all possible locations for pipe supports and (b) vacant space available in case re-routing of pipes is required.

Need 5: Piping designers should be able to import any layout changes made by stress engineers to 3D plant model as “reference geometry” for re-routing, and update pipe supports in 3D plant model by referring to finalized stress models.

SST Systems’ disruptive product CAEPIPE 3D+ fulfils all 5 Needs of pipe stress engineers as outlined in the respective features below.

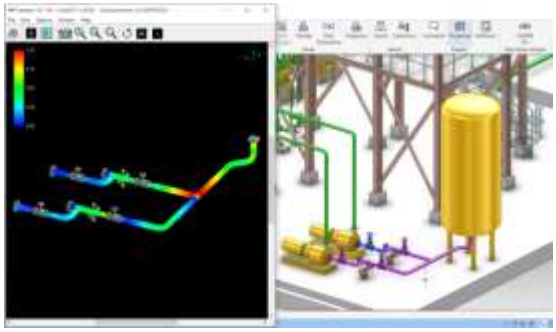
Feature 1:

CAEPIPE 3D+ meets Need 1 by generating pipe stress models directly from 3D Plant Design systems such as E3D/PDMS, SmartPlant, AutoCAD Plant 3D, AutoPlant, CADMATIC, CATIA, etc.

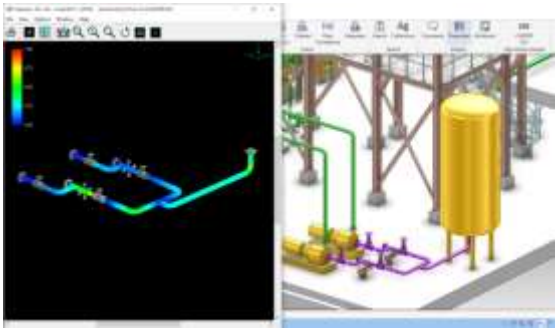
Stress Model generated by Designer from 3D Plant Design System

Feature 2:

Need 2 is met by first having 3D piping designers use CAEPIPE 3D+ as first-level piping flexibility check software. In other words, while pipe routing, 3D piping designers use CAEPIPE 3D+ to perform a few basic stress compliance checks, specifically to make sure the layout is flexible enough to absorb thermal expansion/contraction of pipes.



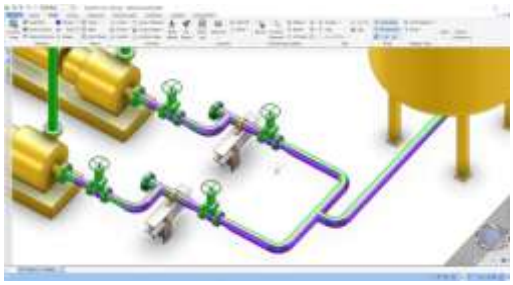
First-level Stress Check on Original Layout by Designer



First-level Stress Check on Revised Layout by Designer

Feature 3:

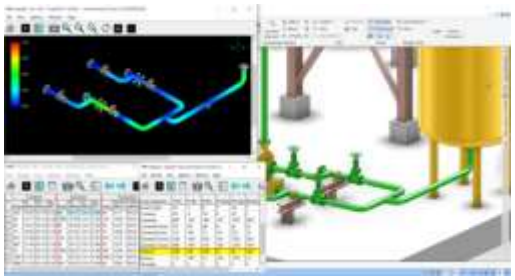
Once 3D piping designers finalize the layout after performing first-level piping flexibility checks, they can meet Need 3 by using CAEPIPE 3D+ to check for interference between the "deformed" layout of piping under "hot operating condition" and adjacent objects. This feature is available only on CAEPIPE 3D+ for E3D/PDMS and CADMATIC. To avoid any interference under hot condition, designers can make minor layout changes.



Clash Check under HOT Condition by Designer

Feature 4:

The “thermal stress-compliant” and “hot-clash-free” CAEPIPE 3D+ models resulting from Features 2 and 3 are transferred to pipe stress engineers (a) to carry out detailed analyses for loads such as deadweight, thermal, wind, seismic and dynamic loads, (b) to arrive at proper support scheme to meet all aspects of pipe stress requirements, and (c) to generate detailed stress reports.

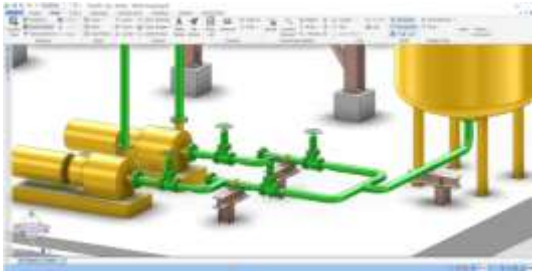


Detailed Analysis by Stress Engineer with 3D Plant Model in view

CAEPIPE 3D+ meets Need 4 of pipe stress engineers by providing immediate visualization of the 3D plant model while performing analyses. Such instant visualization helps (a) to identify all possible locations for pipe supports, and (b) to identify vacant space available in case re-routing of pipe is required.

Feature 5:

Once stress models are finalized by pipe stress engineers, CAEPIPE 3D+ meets Need 5 by allowing piping designers to import all layout changes made by stress engineers into 3D plant model as “reference geometry”, over which designers would re-route pipes. Designers would then update pipe supports in 3D plant model by referring to finalized stress models.



Finalized Layout & Supports transferred to 3D Plant by Designer

Overview of CAEPIPE 3D+

Pipe stress software CAEPIPE 3D+ has two (2) parts built-in as listed below.

Part 1 generates pipe stress models in CAEPIPE 3D+ format from:

- Plant database of 3D Plant Design software E3D/PDMS and CADMATIC, and
- "PCF" files generated from plant database of 3D Plant Design software such as SmartPlant, AutoCAD Plant 3D, AutoPlant, CATIA, CADWORX, etc.

by reading the following from the plant database / PCF files of the above 3D Plant Design software:

- Piping geometry, pipe sections and material properties,
- Temperature, pressure and weight of pipe fittings,
- Thermal anchor movements at equipment nozzles,
- Allowable loads at equipment nozzles, and
- Pipe support details to create the corresponding hangers, guides, etc. in CAEPIPE 3D+.

Part 2 of CAEPIPE 3D+ is the same as the full-fledged internationally accepted pipe stress software CAEPIPE, which reads the stress models generated through Part 1. For modeling and analysis features of CAEPIPE, review CAEPIPE User's Manual, Technical Reference Manual and Code Compliance Manual available at the link www.sstusa.com/caepipe-docs.php.

Flexible Usage of CAEPIPE 3D+

Each licensed seat of CAEPIPE 3D+ currently allows the use of the following 5 modules, with the only restriction being only one module can be used at any point in time. As listed below, CAEPIPE 3D+ can be used as a stand-alone CAEPIPE in case 3D plant database or PCF files are not available to work with.

3D Plant Design System	Module Name	Status
E3D/PDMS*	CAEPIPE 3D+ for E3D/PDMS	Available
CADMATIC*	CAEPIPE 3D+ for CADMATIC	Available
3D Plant Design System that produces PCF files (such as SmartPlant, AutoCAD Plant 3D, AutoPlant, CATIA, CADWORX etc.)	CAEPIPE 3D+ for PCF	Available
PDS	CAEPIPE 3D+ for PDS	Available
Nil	CAEPIPE (stand-alone)	Available

**CAEPIPE 3D+ runs as an on-line, add-on module to E3D/PDMS/CADMATIC, whereby a 3D piping designer transfers piping layouts and all available data from E3D/PDMS/CADMATIC to CAEPIPE 3D+ directly and performs first-level stress compliance checks without exiting E3D/PDMS/CADMATIC. Similarly, a pipe stress engineer performs detailed analyses of stress models received from the 3D piping designer without exiting E3D/PDMS/CADMATIC.*

In short, all five modules are available for use by a licensed CAEPIPE 3D+ seat, but only one module at any point in time.

Benefits of CAEPIPE 3D+

1. The same full-fledged pipe stress software CAEPIPE 3D+ is used with many 3D Plant Design systems or as a stand-alone stress software, based on the requirements of projects.
2. CAEPIPE 3D+ input files can be generated directly from 3D Plant Design systems, thereby saving time and avoiding errors in re-creating stress models manually.
3. CAEPIPE 3D+ can be used first by 3D piping designers as a first-level piping flexibility check software. This step substantially reduces the number of design iterations between the piping layout and stress departments, resulting in huge time saving during design.
4. Using CAEPIPE 3D+ module for E3D/PDMS/CADMATIC, 3D piping designers can check for interference between “piping deformed under hot operating condition” and adjacent objects.
5. When CAEPIPE 3D+ is used on E3D/PDMS/CADMATIC, 3D plant model can be instantly viewed by 3D designers during their first-level stress checks and by pipe stress engineers during detailed analyses.
6. Stress analysts can identify all possible locations for pipe supports on 3D plant model, as it contains all supporting objects such as steel and concrete structures adjacent to the concerned piping systems. These possible pipe support locations, once marked in the 3D plant model, are automatically transferred as “nodes” in the CAEPIPE 3D+ input files.
7. Piping designers can import layout changes made by stress engineers into 3D plant model and update pipe supports by viewing final stress models, resulting in significant time saving during design.

In summary, CAEPIPE 3D+ helps 3D piping designers and stress engineers to arrive at substantially better piping layouts that also meet all stress requirements, thereby saving costs associated with design, materials, construction, operation and maintenance.

CAEPIPE 3D+ as Piping Designers' Tool

Basic Pipe Stress Concepts

Piping systems experience different loadings, categorized into three basic loading types listed below.

Sustained Load:

It mainly consists of internal pressure and dead-weight. Dead-weight is from weight of pipes, fittings, components such as valves, operating fluid, insulation, cladding, lining etc.

Internal design/operating pressure develops uniform circumferential stresses in the pipe wall, based on which pipe wall thickness is determined during the process/P&ID stage of plant design such that piping "failure by rupture" is avoided. In addition, internal pressure develops axial stresses in the pipe wall. *These axial pressure stresses vary only with pressure, pipe diameter and wall thickness, which are already pre-set at the P&ID stage and hence these axial pressure stresses cannot be reduced by changing the piping layout or the pipe support scheme*.

On the other hand, dead-weight causes the pipe to bend (generally downward) between supports and nozzles, producing axial stresses in the pipe wall (also called "bending stresses"); these bending stresses linearly vary across the pipe cross-section, being tensile at either the top or bottom surface and compressive at the other surface. If the piping system is not supported in the vertical direction (i.e., in the gravity direction) excepting at equipment nozzles, bending of the pipe due to dead-weight may develop excessive stresses in the pipe and impose large loads on equipment nozzles, increasing the susceptibility to piping "failure by collapse".

Various international piping codes impose limits, also called "allowable stresses for sustained loads", on these axial stresses generated by dead-weight and pressure in order to avoid "failure by collapse".

For the calculated axial stresses to be below such allowable stresses for sustained loads, it may be necessary to support the piping system vertically. Typical vertical supports to carry dead-weight are:

a) Resting steel supports,

b) Rod hangers,

c) Variable spring hangers, and

d) Constant support hangers..

Both rod hangers and resting steel supports fully restrain downward pipe movement but permit pipe to lift up at such supports. If pipe lifts up at any of the rod hangers / resting supports during operating condition, then that support does not carry any pipe weight and hence will not serve its purpose.

In the below Section titled "Sample Pipe Stress Problems and Solutions", a couple of sample layouts are presented to illustrate how piping can be supported by spring hangers and resting steel supports to comply with the code requirements for sustained loads.

Thermal Load (also referred as Expansion Load):

It refers to the "cyclic" thermal expansion/contraction of piping as the system goes from one thermal state to another thermal state (for example, from "shut-down" to "normal operations" and then back to "shut-down"). If the piping system is not restrained in the thermal growth/contraction directions (for example, in the axial direction of a straight pipe), then for such cyclic thermal load, the pipe expands/contracts freely; in this case, no internal forces, moments and resulting stresses and strains are generated in the piping.

If, on the other hand, the pipe is "restrained" in the directions it wants to thermally deform (such as at equipment nozzles and pipe supports), such constraint on free thermal deformation generates cyclic thermal stresses and strains throughout the system as the system goes from one thermal state to another. When such calculated thermal stress ranges exceed the "allowable thermal stress range"

specified by various international piping codes, then the system is susceptible to “failure by fatigue”. So, in order to avoid “fatigue failure” due to cyclic thermal loads, the piping system should be made flexible (and not stiff). This is normally accomplished as follows:

a) Introduce bends/elbows in the layout, as bends/ elbows “ovalize” when bent by end-moments, which increases piping flexibility.

b) Introduce as much “offsets” as possible between equipment nozzles (which are normally modeled as anchors in pipe stress analysis). For example, if two equipment nozzles (which are to be connected by a pipeline) are in line, then the straight pipe connecting these nozzles is “very stiff”. If, on the other hand, the two equipment are located with an “offset”, then their nozzles will have to be connected by an “L-shaped” pipeline which includes a bend/elbow; such “L-shaped” pipeline is much more flexible than the straight pipeline mentioned above.

c) Introduce expansion loops (with each loop consisting of four bends/elbows) to absorb thermal growth/contraction.

d) Lastly, introduce expansion joints such as bellows, slip joints etc., if warranted.

In addition to generating thermal stress ranges in the piping system, cyclic thermal loads impose loads on static and rotating equipment nozzles. By following one or more of the steps from (a) to (d) above and steps (e) and (f) listed below, such nozzle loads can be reduced.

e) Introduce “axial restraints” (which restrain pipe in its axial direction) at appropriate locations such that thermal growth/contraction is directed away from equipment nozzles, especially critical ones.

f) Introduce “intermediate anchors” (which restrain pipe movement in the three translational and three rotational directions) at appropriate locations such that thermal deformation is absorbed by regions (such as expansion loops) away from equipment nozzles.

In the below Section titled “Sample Pipe Stress Problems and Solutions”, a few sample layouts are presented to illustrate how loops/offsets, axial restraints and intermediate anchors are used to reduce thermal stresses in piping (and resulting nozzle loads).

Occasional Loads:

These are the third type of loads, which are imposed on piping by occasional events such as earthquake, wind etc. To protect piping from wind (which normally blows in horizontal plane), it is normal practice to attach “lateral supports” to piping systems. During an earthquake, the earth may also move vertically. To protect piping against both horizontal and vertical movement during earthquake, “integral 2-way vertical and lateral restraints” are required.

In summary, to carry sustained loads, normally vertical weight supports (as those listed under the Section titled “Sustained Load” above) are required. To safeguard piping from wind, lateral restraints can be added to some of the vertical weight supports (especially resting steel supports). Similarly, to withstand static seismic ‘g’ loads, some of the vertical weight supports can be modified as “integral 2-way vertical and lateral restraints”. Lastly, for thermal loads, zero supports give zero stresses. So, thermal stresses and equipment nozzle loads will normally decrease as the number of supports goes down. Axial restraints and intermediate anchors are recommended only to direct thermal growth away from equipment nozzles.

Recommended Procedure for first-level stress checks by 3D Piping Designers

The steps given below may normally be followed to perform preliminary visual stress checks of piping systems designed using 3D plant design systems.

Step1: Generating stress model

Apply CAEPIPE 3D+ on the piping system under consideration in the 3D plant model.

In case some of the important inputs (such as Thermal Anchor Movements at equipment nozzles and Allowable Nozzle Loads) are not available in the 3D plant database/PCF file, they can be manually entered using the "Edit Layout" button provided in CAEPIPE 3D+.

For modeling and analysis features of CAEPIPE 3D+, review CAEPIPE User's Manual, Technical Reference Manual and Code Compliance Manual available in the link www.sstusa.com/caepipe-docs.php.

Step2: Studying Thermal Stress results for the Initial Layout

Review first stress contour plot for thermal stresses. The plot is color-coded such that "blue" region denotes areas with the least stress ratios (where stress ratio equals to actual computed stress divided by allowable thermal stress), "green" region with higher stress ratios, "yellow" region with even higher stress ratios, and "red" region with the highest stress ratios. Intermediate areas between these distinct colors will be of "bluish-green", "greenish-yellow" and "orange" colors.

Since thermal stresses generated are directly dependent on how "flexible" the layout is, it may be necessary to make the layout as "flexible" as possible (by including bends, offsets, loops etc.) to reduce thermal stresses. So, the Designer's goal is to arrive at a "flexible" layout for which thermal stress ratios remain within "blue" to "yellow" range and not get into "orange" and "red" zones. For a more "flexible" layout, even "yellow" zone may be avoided. That would leave even more thermal margin for stress engineers to meet other pipe stress criteria not considered under first-level stress checks.

Step 3: Finalizing Layout to meet Thermal Stress criteria

In case thermal stress ratios exceed "yellow" zone (i.e., "orange" and "red" zones appear in one or more areas of the piping system), it is important to study the deformed shape for "thermal" load case in order to understand how the piping deforms for "pure thermal" load (where only temperature change is considered). By studying such deformed shape, it is possible to arrive at a layout with appropriate bends, offsets and loops and/or with appropriately located axial restraints/intermediate anchors such that thermal stress ratios do not exceed "yellow" zone. This process may require the Designer to perform several iterations on layout and/or locations for axial restraints/intermediate anchors.

Step 4: Studying Results for Sustained Load

After finalizing piping layout under Steps 2 and 3 for thermal loading, the next task is to support the system vertically to carry its own deadweight under operating condition. In this connection, first review stress contour plot shown in color codes from "blue" to "red" (as in Step 2 above) for sustained stress ratios generated by deadweight and pressure for the system without any vertical supports (excepting those provided by equipment nozzles and intermediate anchors introduced in Step 3 above).

The Designer's goal is to arrive at a vertical support scheme consisting of (a) resting steel supports, (b) rod hangers, (c) variable spring hangers and (d) constant support hangers, at appropriate locations (where such pipe supports can be attached to adjacent concrete/steel structures, platforms etc.) so that stress contour plot for sustained stress ratios avoids "orange" and "red" zones and remains within "blue to yellow" range.

Step 5: Finalizing Vertical Supports to carry Sustained Load

In case sustained stresses exceed “yellow” zone in one or more areas of the piping system, study the deformed shape provided by CAEPIPE 3D+ for sustained load case in order to understand how the piping responds to its own deadweight. Next, identify pipe locations in the 3D model where the pipe can be vertically supported by the support types listed under Step 4 above. Based on this input, vertically support the piping such that sustained stresses do not exceed “yellow” zone. This step may require the Designer to execute CAEPIPE 3D+ on the system with several different locations for weight supports.

CAEPIPE 3D+ automatically sizes spring hangers wherever such hangers are located in the 3D model.

In case resting steel supports are selected to provide vertical support for piping under sustained load, it is to be made sure that piping continues to rest on such steel supports even during operating condition (= weight + pressure + thermal) and does not lift off from these supports. If pipe lifts up at any of these resting supports during operating condition, then that support does not carry any pipe weight under operating condition and hence will not serve its purpose. Similarly, at rod hanger locations, the tendency of piping should be to deform downward for operating load case, so that the rod hangers carry the pipe weight under tension. On the other hand, if pipe lifts up at any of the rod hangers, then that rod hanger goes into compression thereby not carrying the weight of the piping during operating condition. Whether the pipe weight is being carried during operation by resting steel supports and/or rod hangers (both types are mathematically modeled as one-way vertical Limit Stops in CAEPIPE 3D+) or whether the pipe lifts up at those support locations is shown in the report titled “Status of Limit Stops – Operating Load”. The goal is to make sure the status is shown as “Reached” at all vertical Limit Stops for Operating Load case.

Step 6: Studying Results for Occasional Load

After arriving at a final layout with an acceptable pipe support scheme under Steps 2 to 5 for thermal and sustained loads, the next task is to protect piping against large horizontal and vertical movements that could occur due to static seismic “g” load.

To accomplish this goal, first review stress contour plot for occasional stresses generated by deadweight, pressure and static seismic “g” load shown in color codes from “blue” to “red” (as in Step 2 above).

The Designer’s goal is to replace some of the weight supports (for example, resting supports) located in the “yellow” to “red” zones with “integral 2-way vertical and lateral restraints”, so that stress contour plot for occasional stresses avoids “orange” and “red” zones and remains within “blue to yellow” range.

Step 7: Finalizing 2-way Vertical and Lateral Restraints to withstand Occasional Load

In case occasional stresses exceed “yellow” zone in one or more areas of the piping system, study the seismic deformed shape provided by CAEPIPE 3D+ in order to understand how the piping responds to static seismic “g” load. Next, identify those weight support locations (for example, resting supports) in the “yellow” to “red” zones where the pipe can also be laterally supported and replace those weight supports with “integral 2-way vertical and lateral restraints”, such that occasional stresses do not exceed “yellow” zone. This step may require the Designer to execute CAEPIPE 3D+ on the system with several different locations for “integral 2-way vertical and lateral restraints”.

Step 8: Meeting Allowable Loads at Nozzles / Anchors

After locating relevant supports (a) to minimize thermal stresses, (b) to carry weight of the piping during operation, and (c) to withstand static seismic “g” load, the Designer should check the calculated loads at nozzles/anchors in the Support Load Summary. If the calculated loads at nozzles/anchors exceed the corresponding Allowable Loads, by studying the deformed shapes provided by CAEPIPE 3D+ for different load cases, it is possible to further modify the layout and/or support scheme such that the calculated loads at nozzles/anchors do not exceed the Allowable Loads.

As a minimum, the above said Nozzle Load compliance should be carried out for Operating Load case.

Any such changes made to the layout and/or support scheme at this stage (i.e., at Step 8) should not adversely affect the stresses for thermal, sustained and occasional load cases (i.e., all the 3 stress contour plots should continue to avoid “orange” and “red” zones and remain within “blue to yellow” range).

This process may require the Designer to perform several iterations on layout and/or support scheme.

Step 9: Clash check during HOT Condition

If Designers wish to transfer the Deflected Shape back to PDMS/E3D/CADMATIC to perform Clash Check during HOT Operating condition, the Designers can output the Deflected Shape for operating load case by pressing the button “Export Deflected Shape” or through “Results Window > File > Export to 3D Plant Design” while being in Operating Load Displacement Results. Select the file type as “PDMS Geometry Macro file (*.mac)” for PDMS/E3D or select the file type as “CADMATIC 3D Dump file (*.3dd)” for CADMATIC. The Deflected Shape file thus exported can then be imported into PDMS/E3D/CADMATIC for Clash check.

Step 10: Key Results to confirm Validity of Layout with Support Scheme finalized by Designers

Designers are to perform Step 1 to Step 8 (and Step 9 if needed) for all relevant piping systems of the project.

Once the layout and support scheme are finalized for a system, the Designer confirms the validity of that design by submitting key results generated by CAEPIPE 3D+ as listed below.

a) Sorted stress ratios and their locations,

b) Equipment nozzle load compliance with allowable loads,

c) Report listing spring sizes, hot loads, cold loads and travel for variable spring hangers sized by CAEPIPE 3D+,

d) Status of Piping at resting supports during operation (i.e., is the pipe resting on or lifting off a resting support?),

e) Bill of Materials, Weight and Centre of Gravity, Table of Contents, and

f) Relevant stress contour plots and deflected shapes.

Sample Pipe Stress Problems and Solutions

This Appendix provides a few sample layouts (specifically, Sample1, Sample 2, Sample 3 and Sample 5) to illustrate how bends, offsets, loops, axial restraints and/or intermediate anchors are used to reduce thermal stresses in piping (and resulting nozzle loads).

Sample 4 and Sample 5 illustrate how piping can be supported by spring hangers and resting steel supports to comply with the code requirements for sustained loads.

The CAEPIPE 3D+ model files created for the sample problems listed in this section are stored in the directory CAEPIPE3D_Insallation_Directory\Samples for reference.

<u>Sample: 1</u> (Loop_00 and Loop_01)

This problem illustrates the use of expansion loops to reduce thermal stresses.

A 8” NB Schedule 80 pipe (see Fig. 1A) connects two equipment at nodes 10 and 30 with an offset of 4’ (i.e., equal to distance between nodes 20 and 30). The pipe is of A53 Grade A carbon steel and is heated to 300 ^{0} F.

Pipe between nodes 10 and 20 grows thermally to the right towards node 20, while pipe between nodes 30 and 20 grows up towards node 20, as illustrated in Fig. 1B.

This thermal deformation generates large thermal stresses (orange and red zones) in the bend at node 20 and at anchor node 30, as shown in Fig. 1C.

Fig.1D shows a revised layout with a loop, introducing 2 additional bends at nodes 14 and 18, thereby making the layout more flexible. So, thermal growth of X-directional pipes between nodes 10 and 14 and then between 18 and 20 as well as the growth of Z-directional pipe between nodes 30 and 20 are absorbed by the 3 bends at nodes 14, 18 and 20, as seen in Fig. 1E.

The corresponding stress contour plots for thermal and sustained load cases are shown in Fig.1F and Fig. 1G, confirming code compliance.

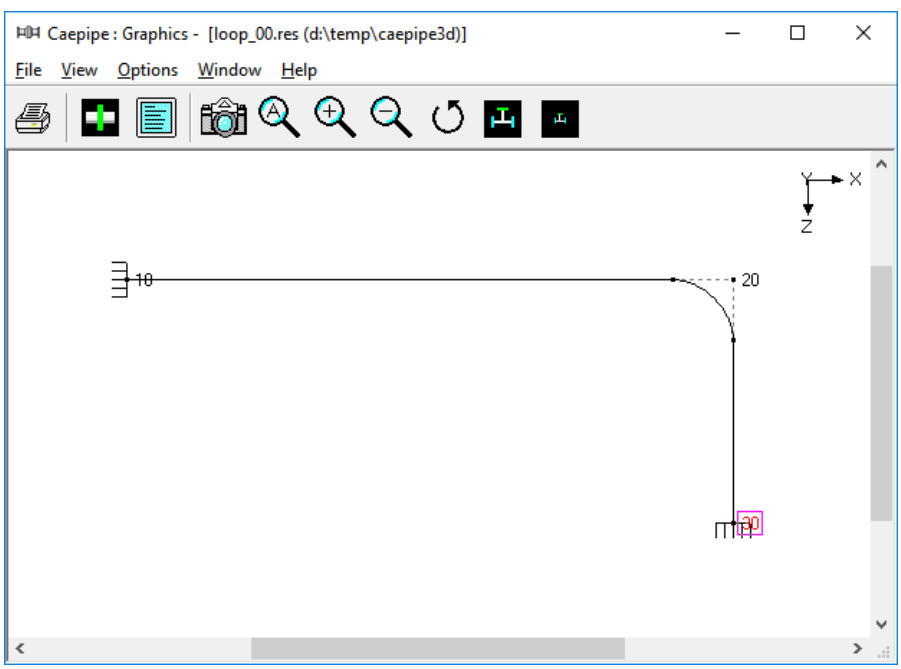


Fig. 1A. Layout with Node Numbers

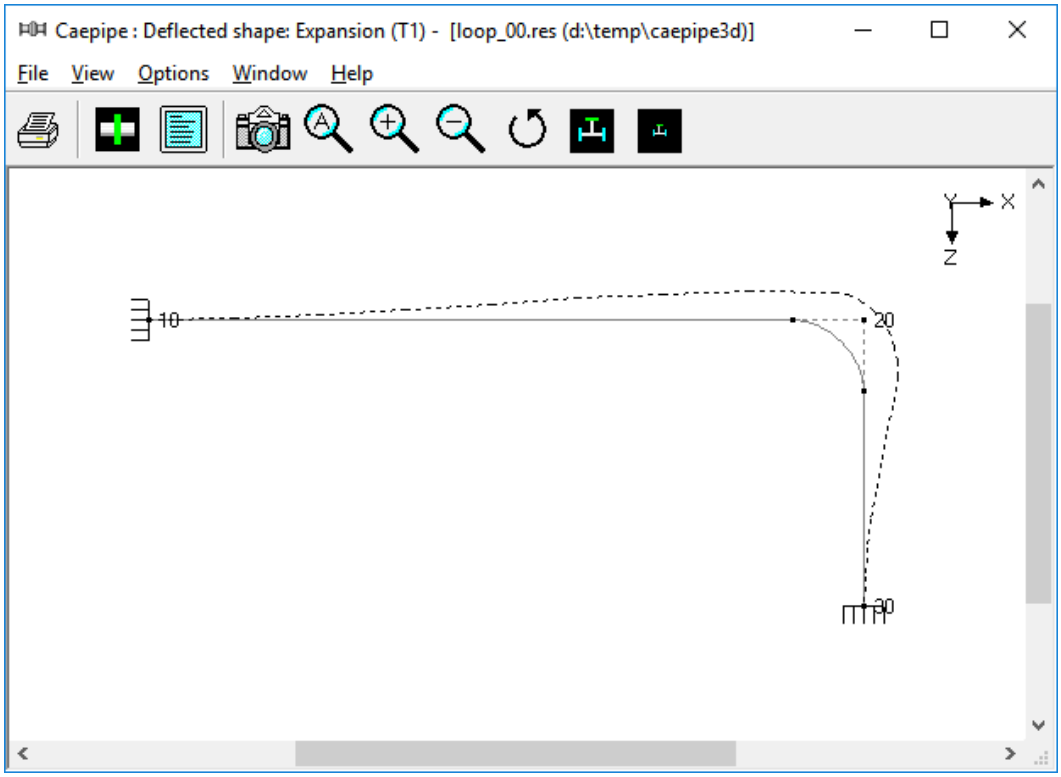


Fig. 1B Thermal Deformation Plot

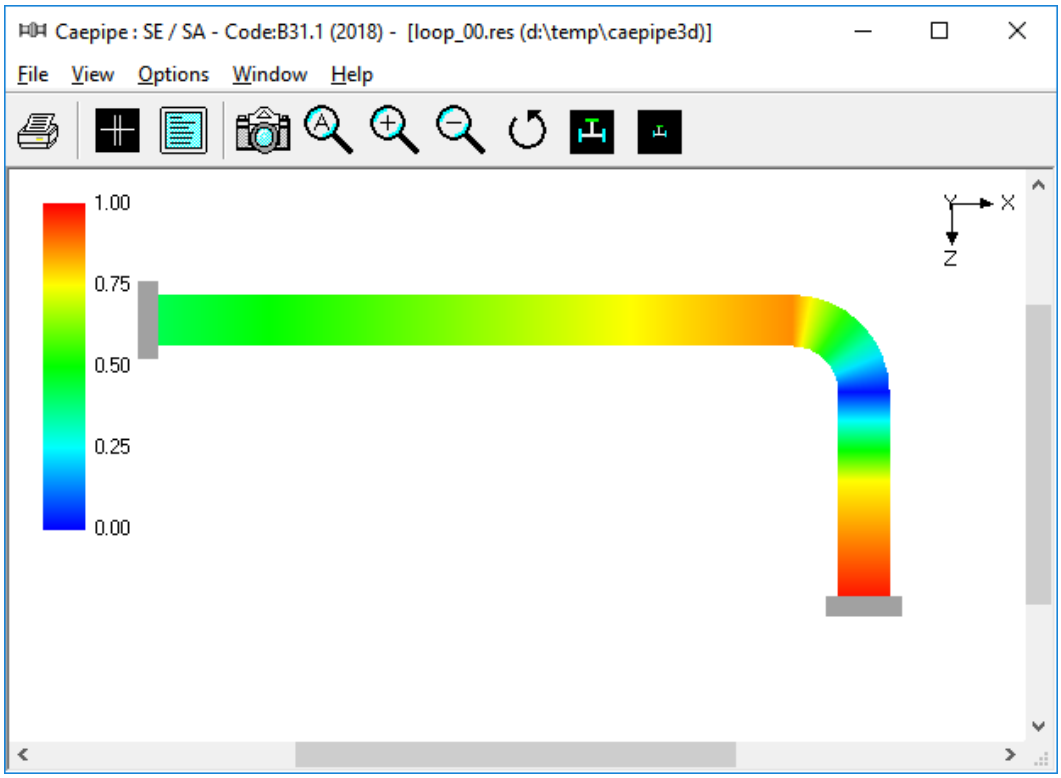


Fig. 1C Thermal Stress Contour Plot

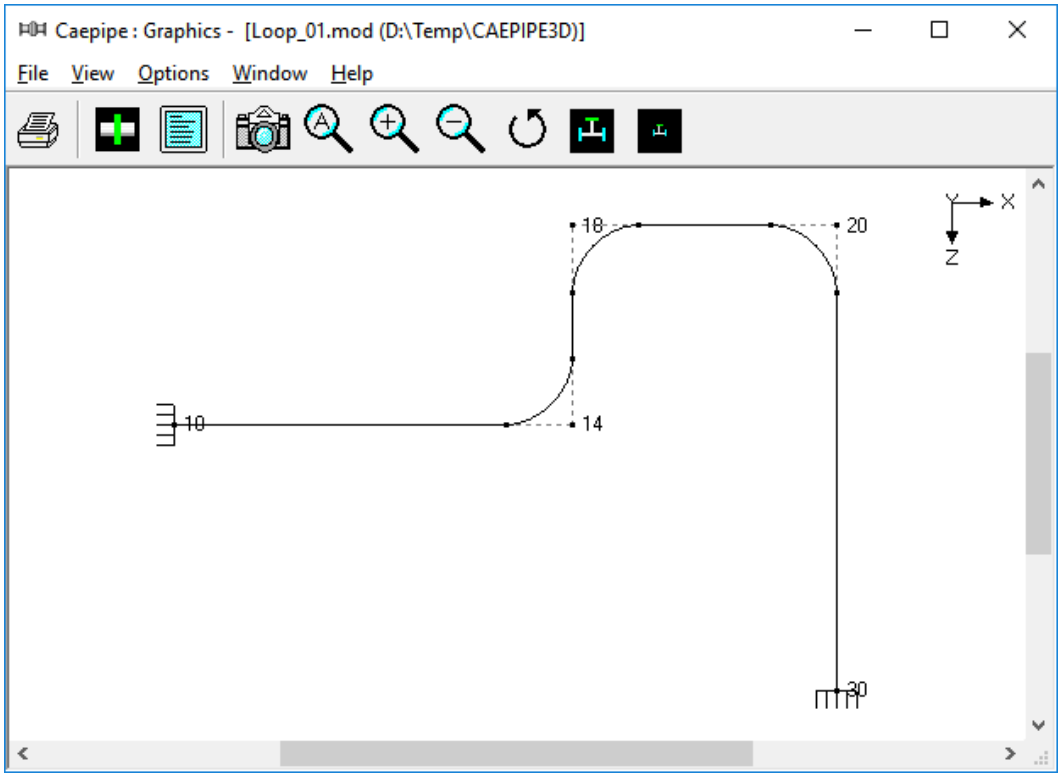


Fig. 1D Revised Layout with Node Numbers

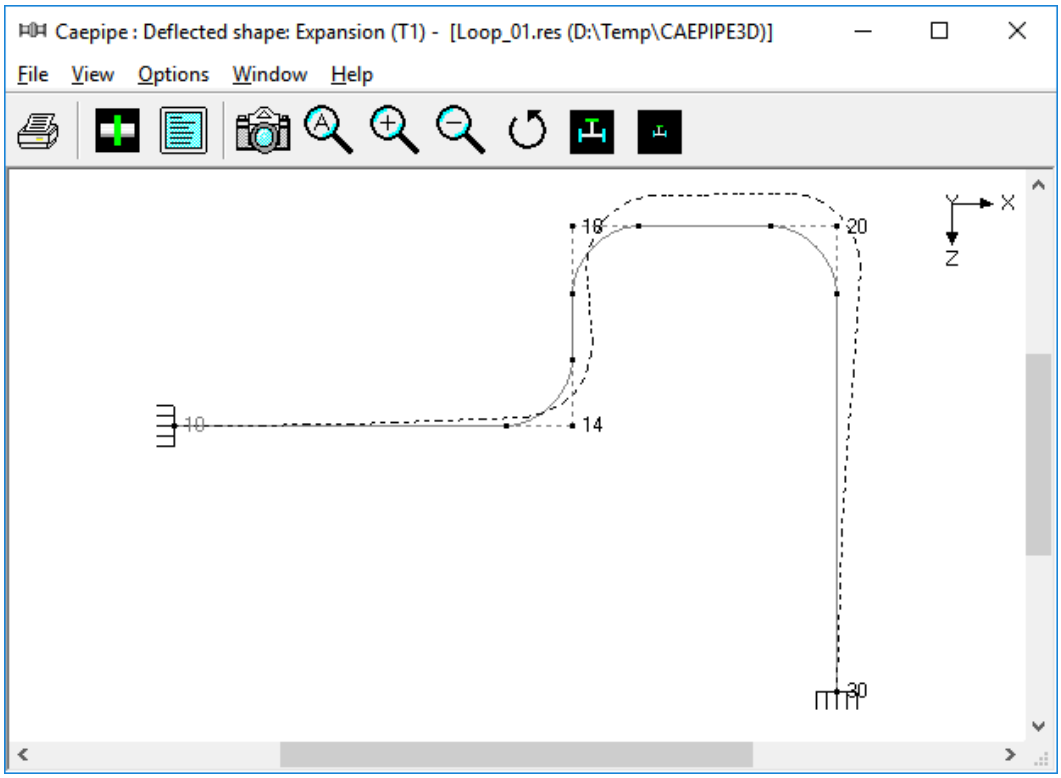


Fig. 1E Thermal Deformation Plot for Revised Layout

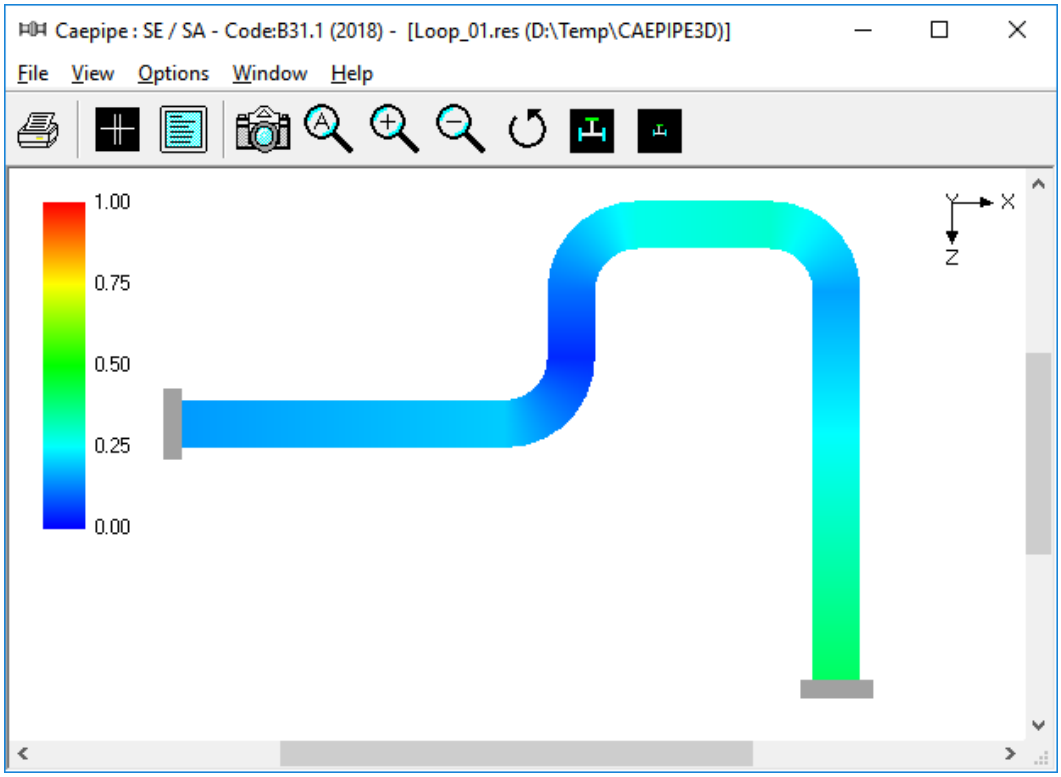


Fig. 1F Thermal Stress Contour Plot for Revised Layout

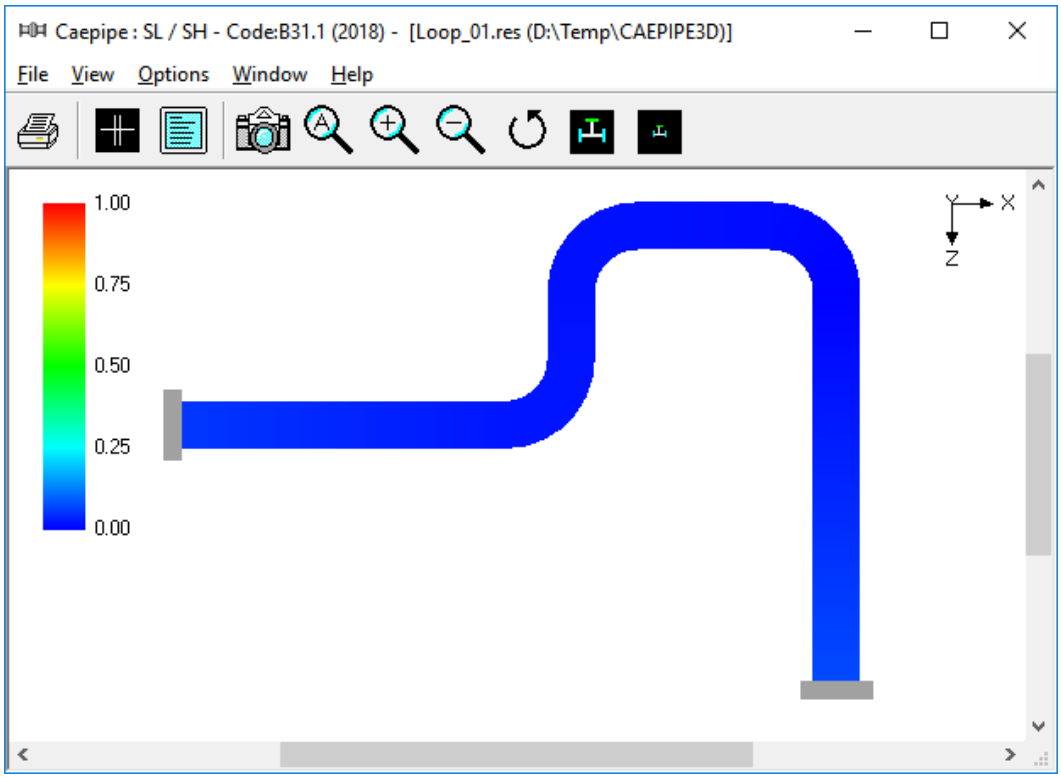


Fig. 1G Sustained Stress Contour Plot for Revised Layout

Sample: 2 (IntermediateAnchor_00 and 01)

This system shown in Fig. 2A is made of 3 pipe sizes, 4” NB Schedule 40 between the anchor node 10 and the first reducer starting at node 50, 6” NB Schedule 40 between the first reducer and the second reducer ending at node 90, and 8” Schedule 40 between the nodes 90 and anchor node 130. The system temperature is 470^0 F.

Since the loop between nodes 10 and 40 is much more flexible (as it is made of 4” NB pipe) than the loop between nodes 100 and 130, the straight pipe between nodes 40 and 100 will thermally grow mostly towards the 4” NB loop, as shown in Fig. 2B, straining the pipe between nodes 10 and 40; this, in turn, produces large thermal stresses (i.e., orange and red zones) in the 4” NB loop and at anchor node 10, as observed in Fig. 2C. In other words, the thermal growth of pipe between nodes 40 and 100 is mostly absorbed by the 4” NB loop and very little by the 8” NB loop, defeating the very purpose of the 8” NB loop.

In order to alleviate thermal stresses in the 4” NB loop, introduce an intermediate anchor at node 95 immediately after the second reducer, so that the thermal growth of straight pipe from node 95 to node 100 is absorbed by the 8” NB loop, while the thermal expansion of straight pipe between nodes 40 and 95 is absorbed by the 4” NB loop, thereby making both loops achieve their intended purpose. The corresponding thermal displacement and thermal stress contour plots are given in Fig. 2D and Fig. 2E respectively.

Fig. 2F confirms that for the deadweight of piping under operating condition, the present configuration with only two equipment nozzles at nodes 10 and 130 and an intermediate anchor at node 95 safely meet the code stress requirement for sustained load.

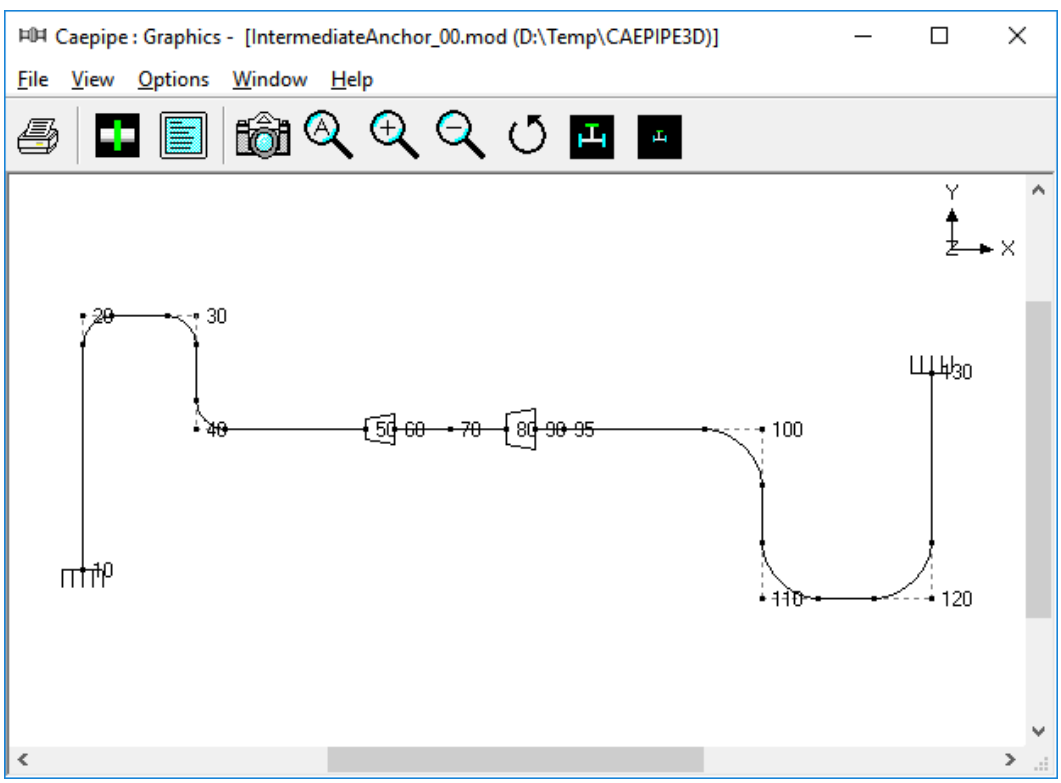


Fig. 2A Layout with Node Numbers

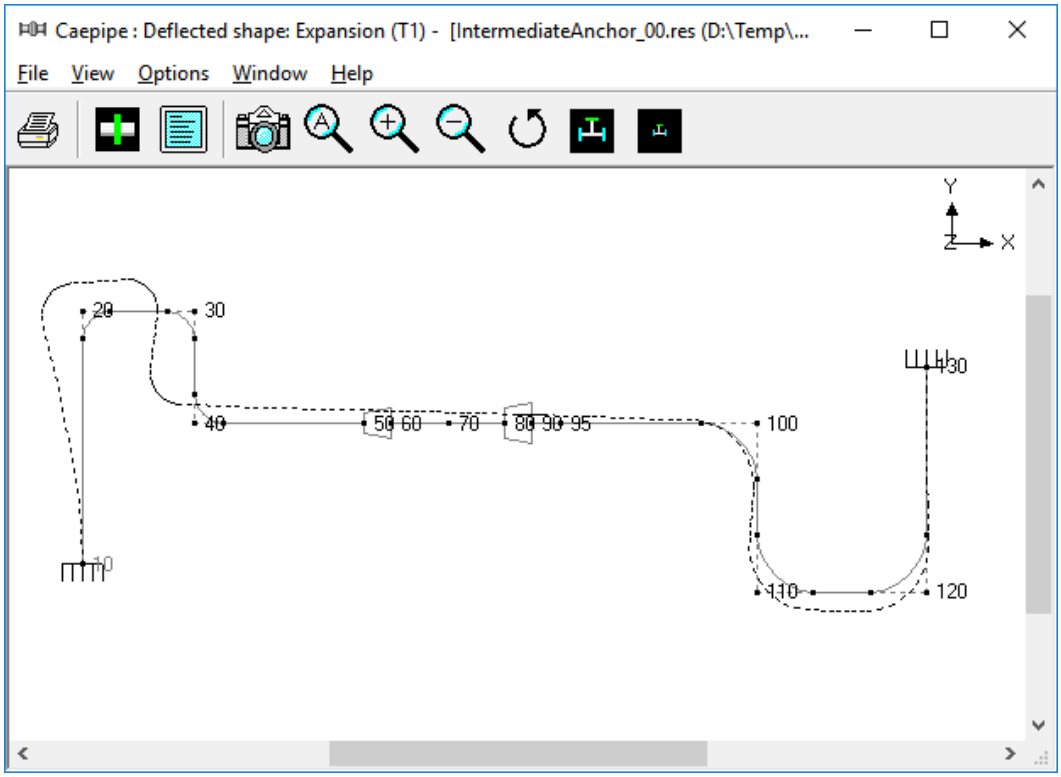


Fig. 2B Thermal Deformation Plot

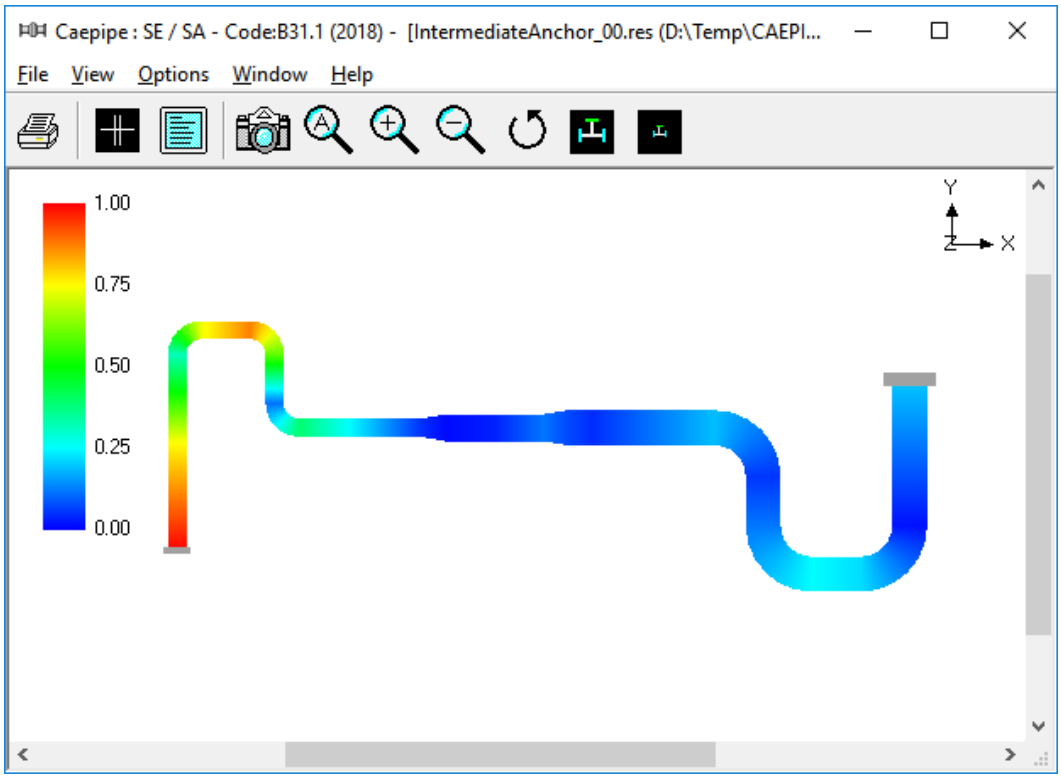


Fig. 2C Thermal Stress Contour Plot

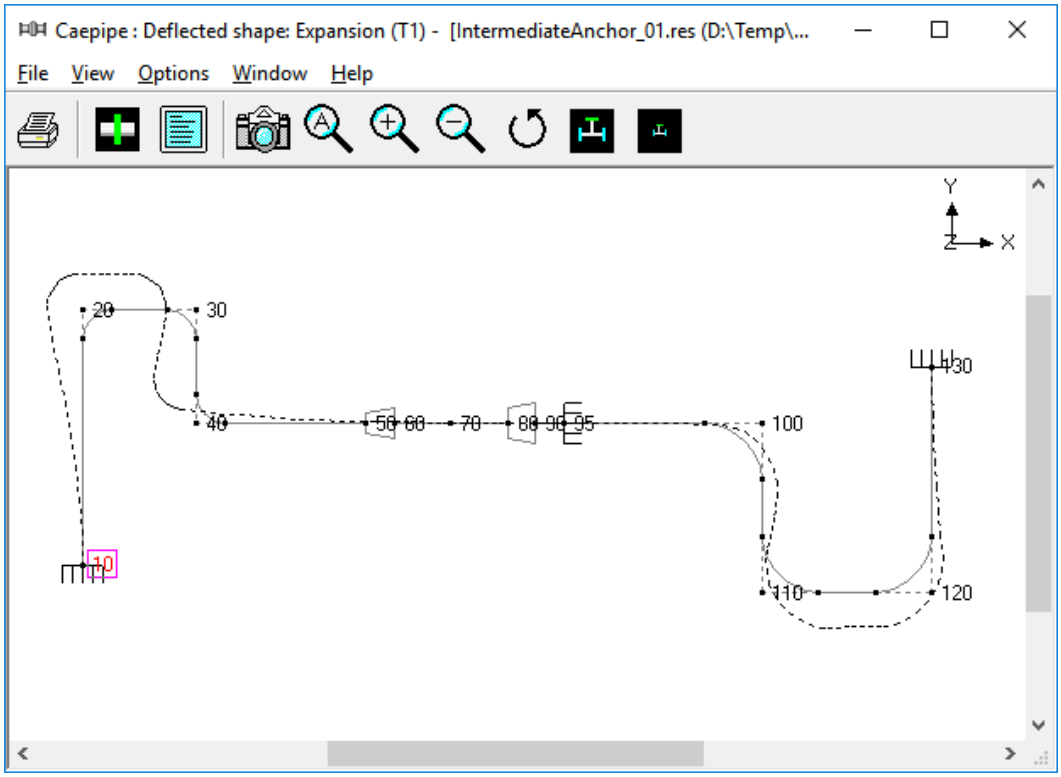


Fig. 2D Thermal Deformation Plot for Layout with Intermediate Anchor

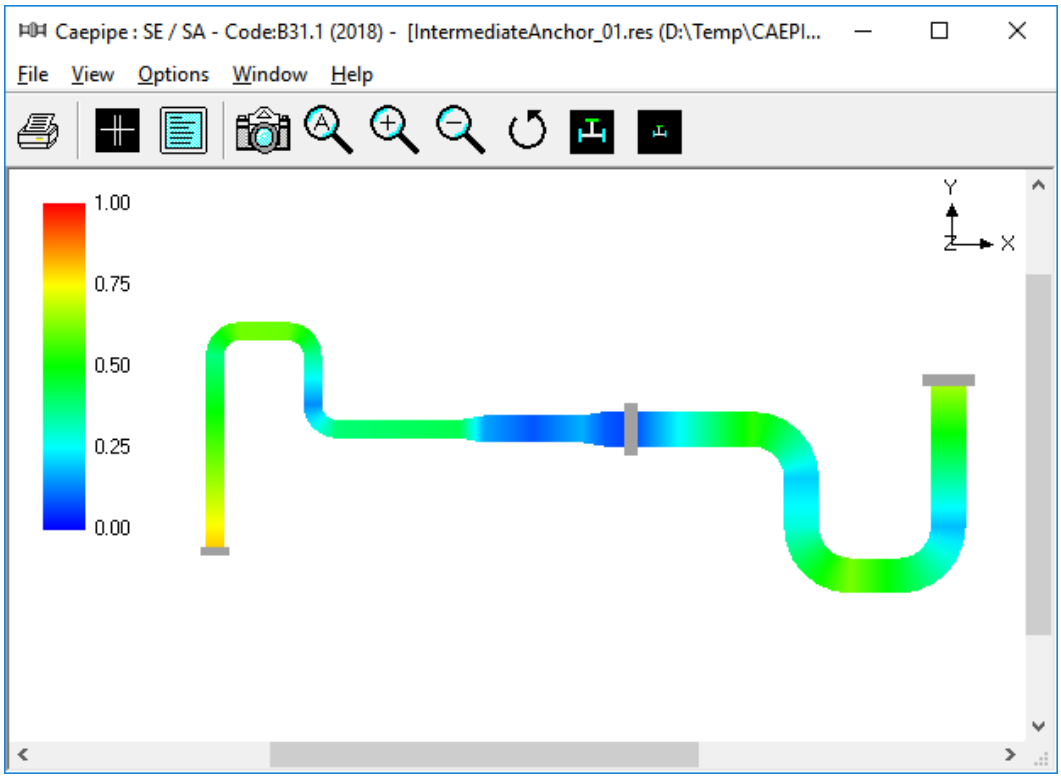


Fig. 2E Thermal Stress Contour Plot for Layout with Intermediate Anchor

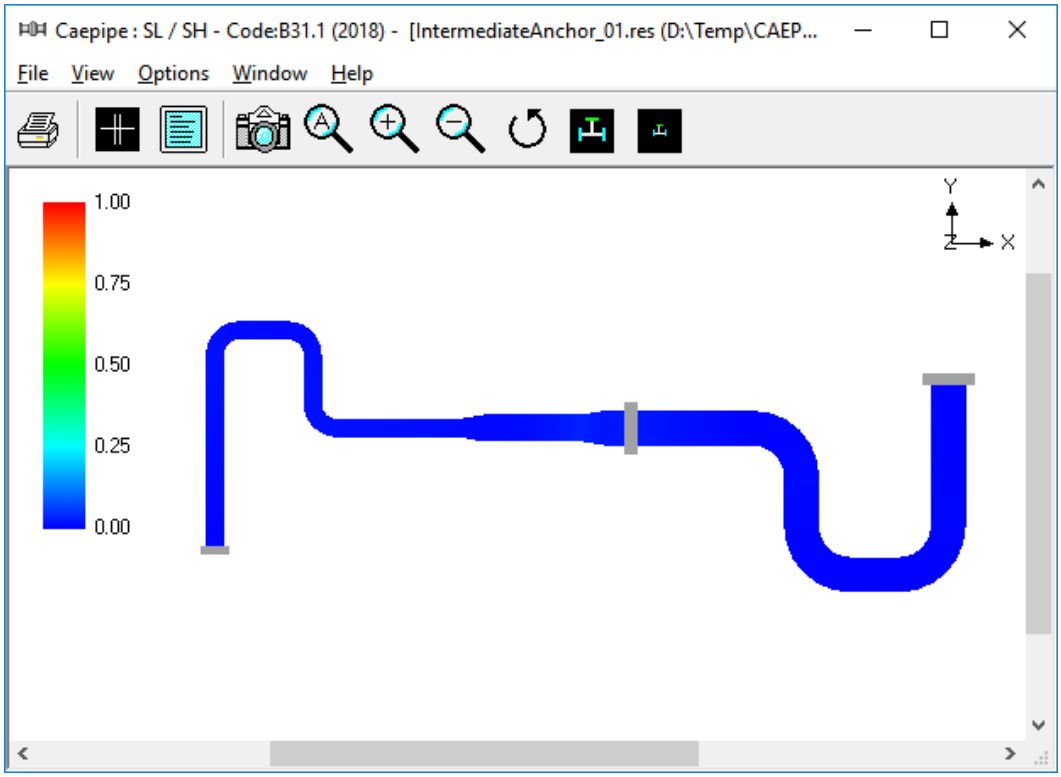


Fig. 2F Sustained Stress Contour Plot for Layout with Intermediate Anchor

Sample: 3 (AxialSupport_00 and 01)

This problem shows how axial restraints (i.e., supports that prevent movement in the pipe axial direction) can be effectively used to direct thermal growth towards expansion loops and to split thermal growth in a line such that the two piping portions of that line grow in opposing directions.

Fig. 3A shows the same problem as in Fig. 2D, with a 6” NB branch line added at the welding tee at node 70 (i.e., from node 70 to node 240).

The deformed geometry for thermal load is shown in Fig. 3B, where it is observed that the tee node 70 does not move up in +Y-direction. Since the intermediate anchor at node 95 restraints the vertical riser (between bend node 220 and tee node 70) from thermally growing upward towards node 70, this riser grows downward producing large bending moments and stresses at and around equipment nozzle at node 240; in addition, since the upward growth of this vertical riser is effectively restrained at the tee node 70 due to presence of intermediate anchor at node 95, large localized thermal stress is generated at the welding tee. This is observed in the thermal stress contour plot given in Fig. 3C.

Fig. 3D shows the same piping system with the intermediate anchor replaced by two axial restraints; the axial restraint in the horizontal line at node 95 splits and directs its thermal growth towards the 4” NB and 8” NB loops and does permit the horizontal line to move up in +Y-direction at tee node 70, whereas the axial restraint at node 210 splits the thermal growth of the vertical riser between nodes 220 and 70. From the thermal deformation plot given in Fig. 3E, it is observed that much less forces and moments and hence stresses would be generated at the equipment nozzle node 240 and welding tee node 70. Fig. 3F and Fig 3G show the thermal stress and sustained stress (in this case sustained stress is due to only deadweight as pressure is zero) contour plots, confirming code compliant system for both loading cases.

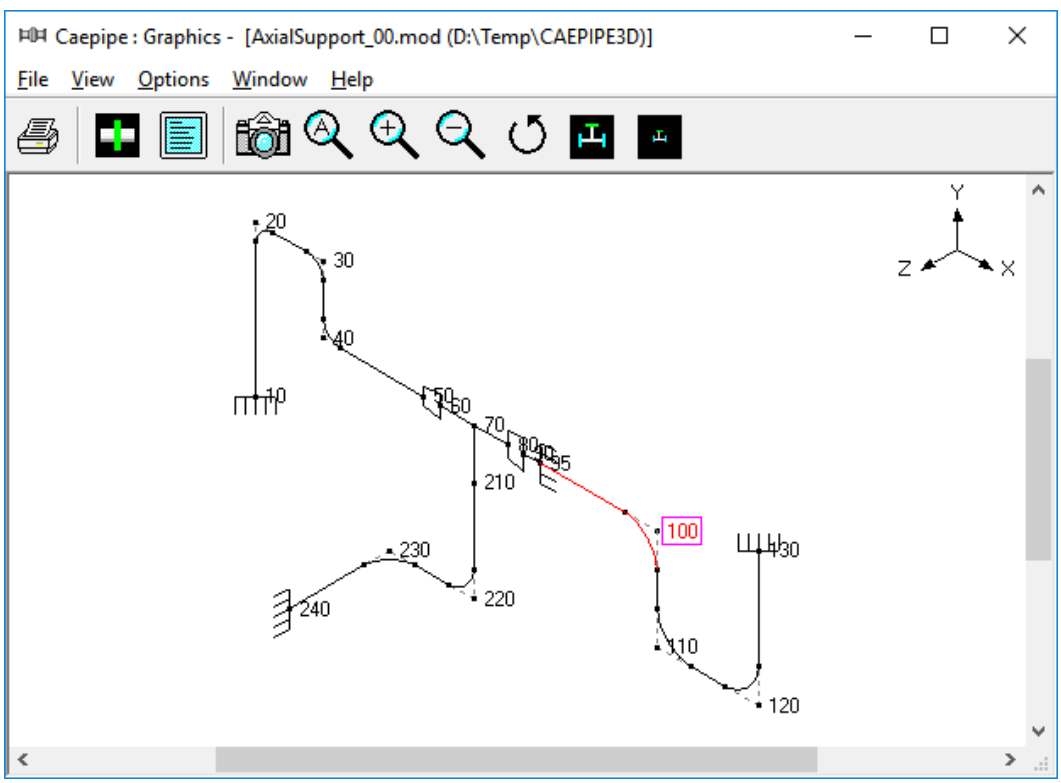


Fig. 3A Layout with Intermediate Anchor at Node 95

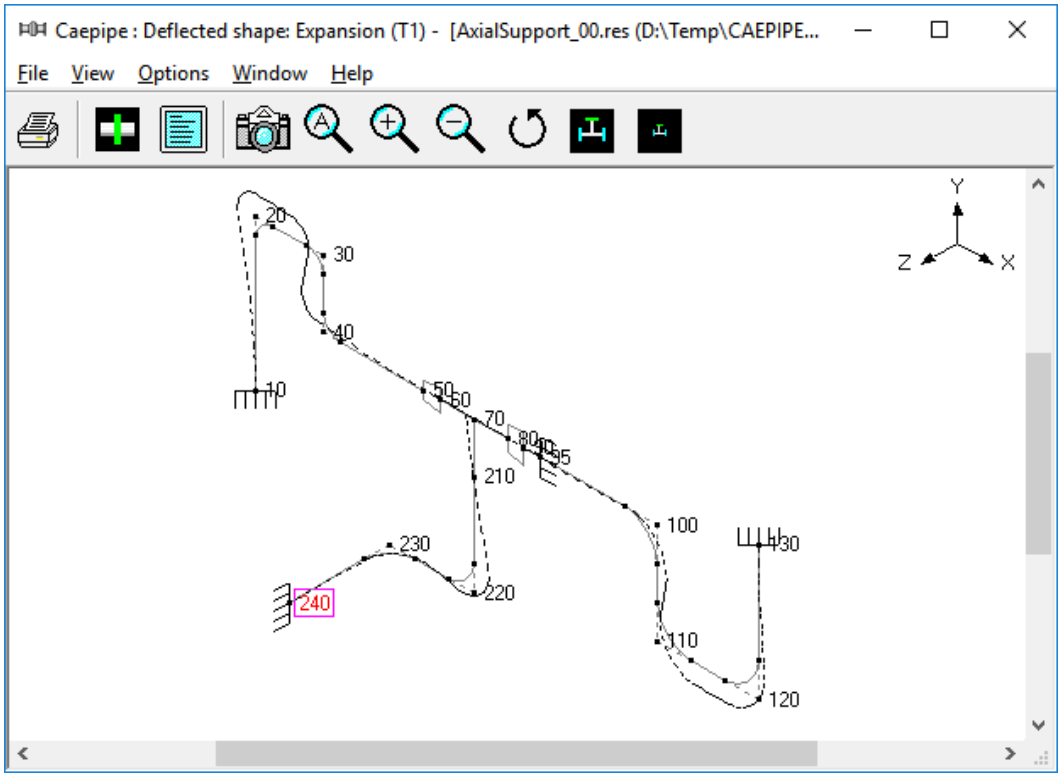


Fig. 3B Thermal Deformation Plot

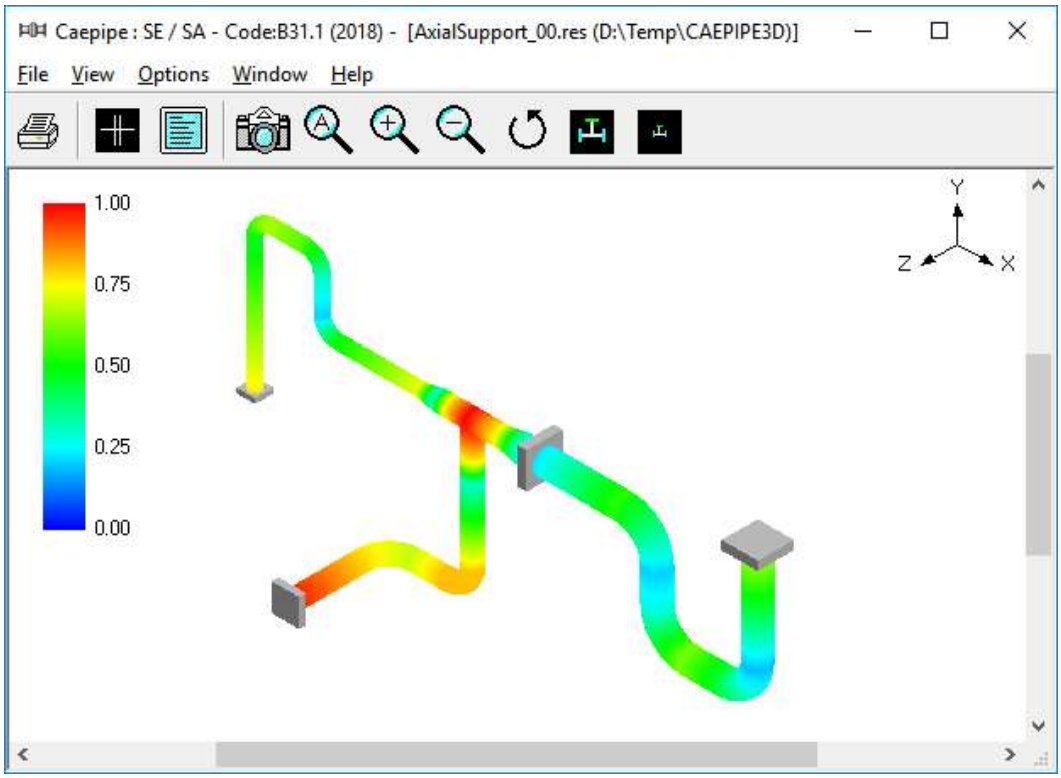


Fig. 3C Thermal Stress Contour Plot

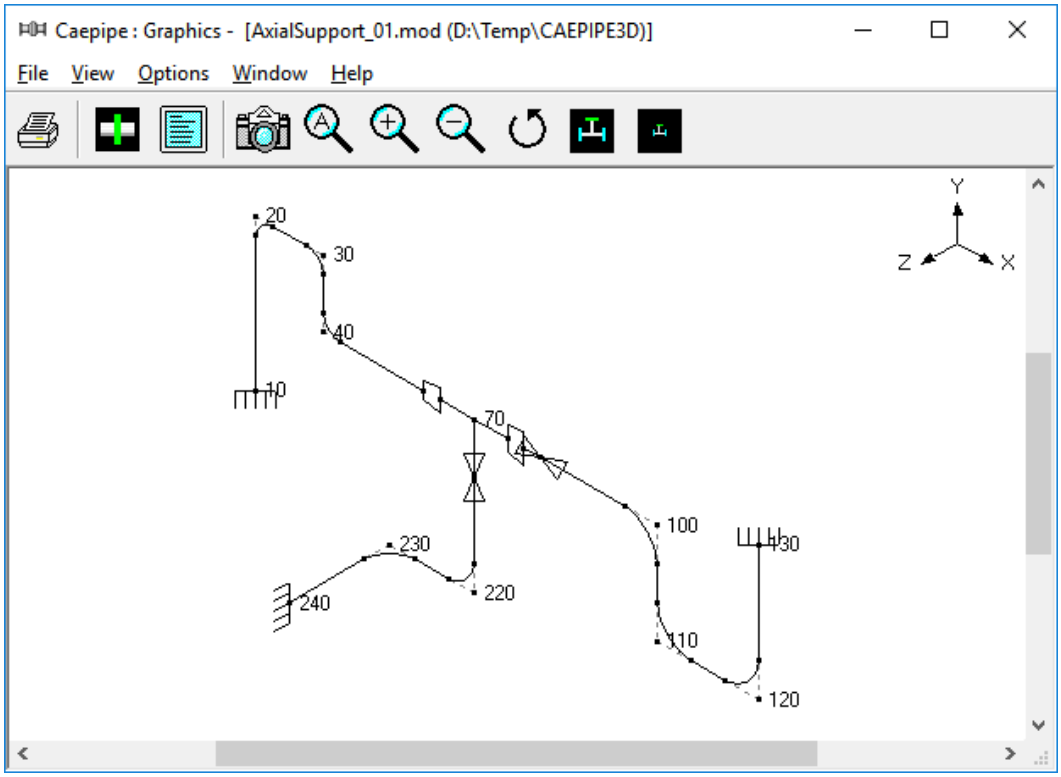


Fig. 3D Layout with Axial Restraints at Node 95 and 210

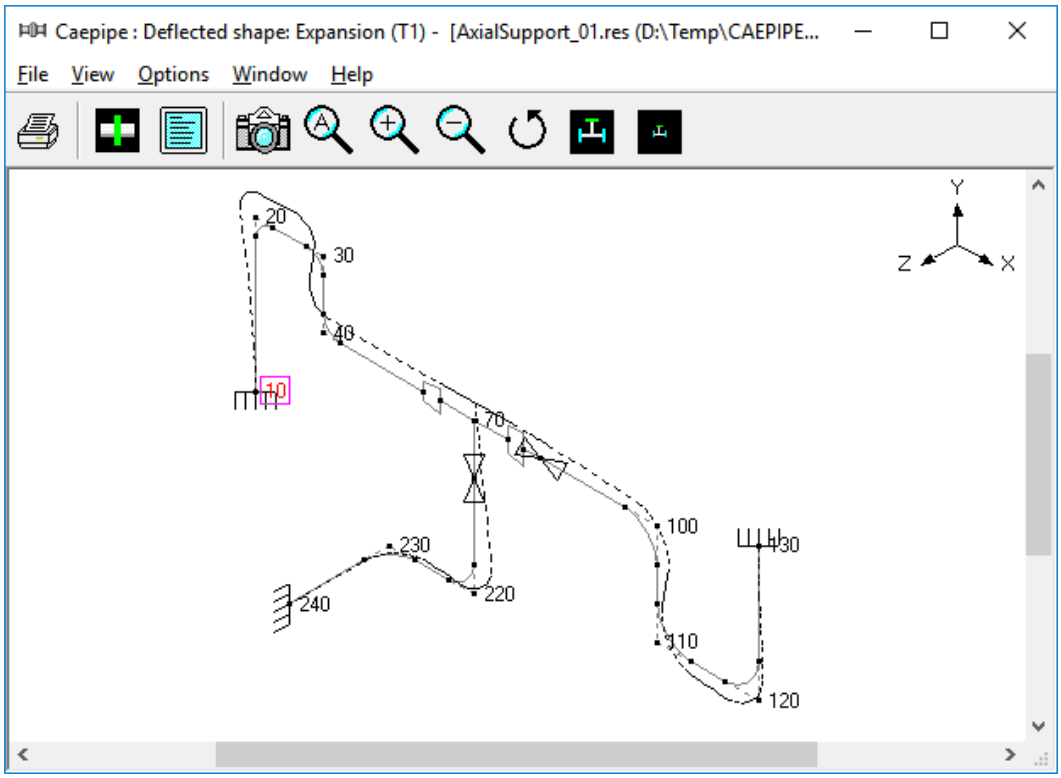


Fig. 3E Thermal Deformation Plot for Layout with Axial Restraints

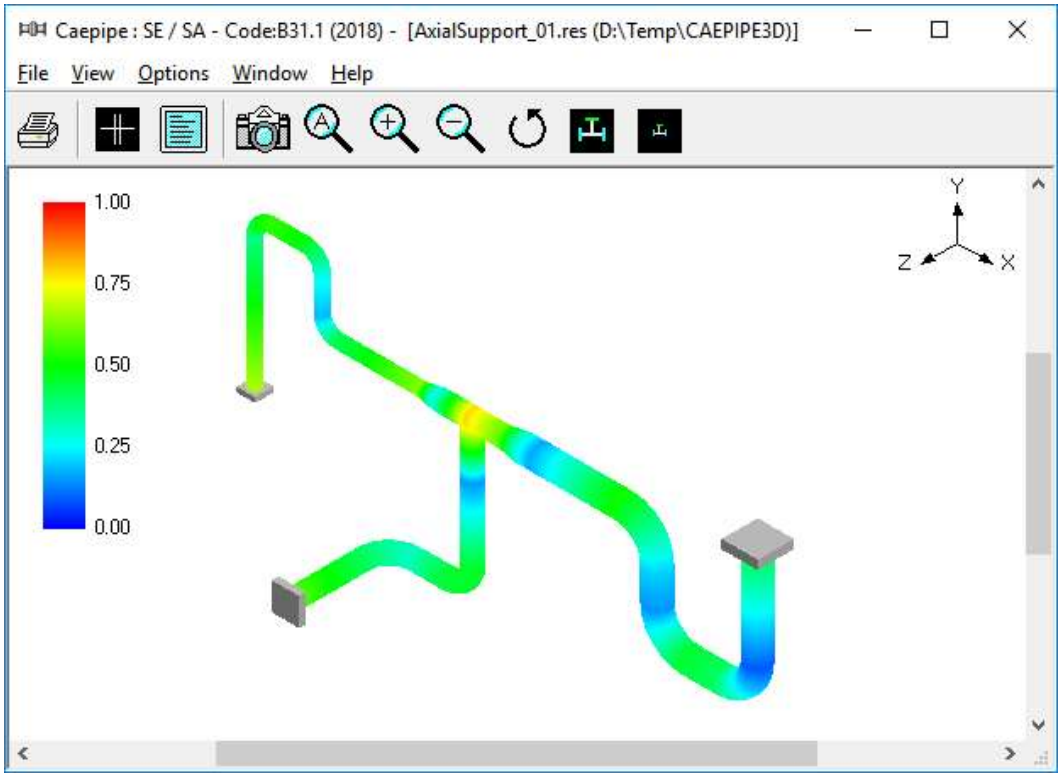


Fig. 3F Thermal Stress Contour Plot for Layout with Axial Restraints

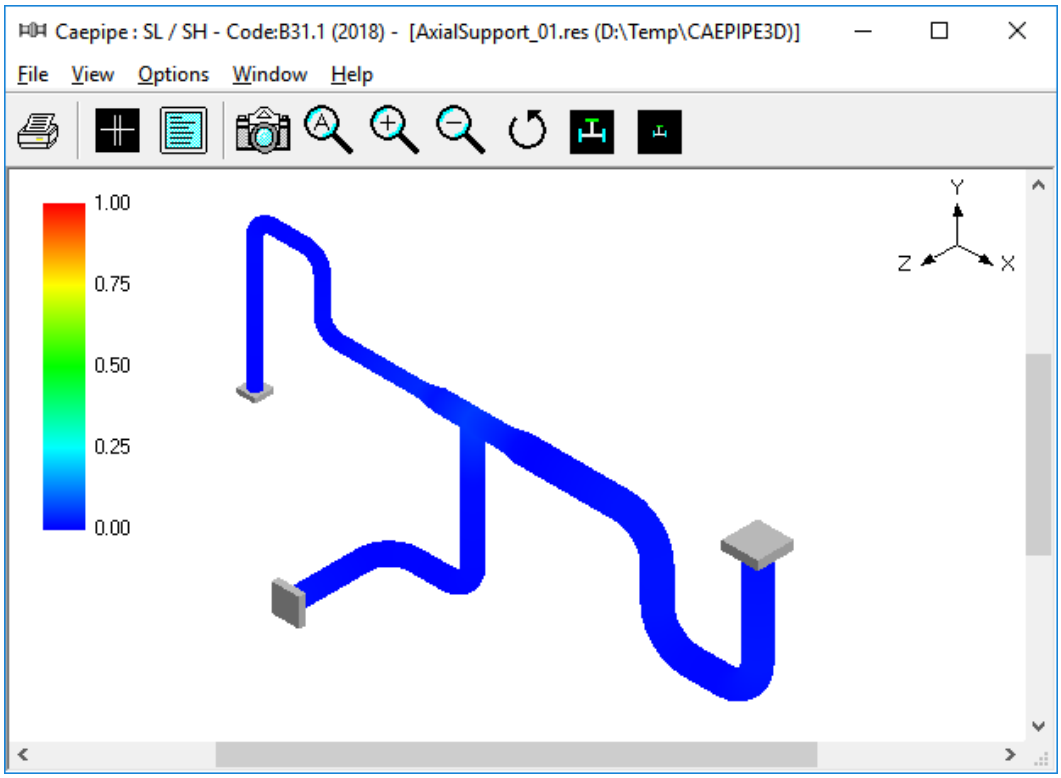


Fig. 3G Sustained Stress Contour Plot for Layout with Axial Restraints

Sample: 4 (WeightSupports_00 and 01)

This problem illustrates how to select and locate vertical supports to carry piping deadweight at operating condition.

Fig. 4A shows a practical problem with 10” NB Standard schedule pipe from equipment nozzle at node 5 upto the reducer starting at node 30, 8” NB Standard schedule pipe from this reducer to the pump nozzle at node 40, and a 6” NB Standard schedule branch line from the welding tee at node 25 to the equipment nozzle at node 125.

The thermal stress contour plot given in Fig. 4B confirms that the piping system is highly flexible and hence meets the code requirement for thermal load. Fig. 4C shows the deflected shape for sustained load (i.e., mainly deadweight). It is observed that the weight of (i) the horizontal line from node 5 to node 15 and (ii) a major portion of the vertical riser from node 15 to node 20 is carried by the equipment nozzle at node 5; on the other hand, the pump nozzle at node 40 carries the weight of (i) the horizontal line from node 20 to node 40, (ii) the valve portion of the branch line from node 25 to node 125 and (iii) a small portion of the vertical riser from node 15 to node 20. The deformation response for deadweight, in turn, generates large forces and moments and hence large sustained stresses at nozzle nodes 5 and 40 as shown in Fig. 4D for sustained stress contour plot.

Fig. 4E shows the same layout with variable spring hangers attached at the bends at nodes 20 and 115, which carry piping deadweight and provide negligible restraint to thermal movement from cold to hot condition and vice versa.

The thermal stress and sustained stress contour plots given in Fig. 4F and Fig. 4G confirm that the piping system with hangers is code complaint for both sustained and thermal load cases.

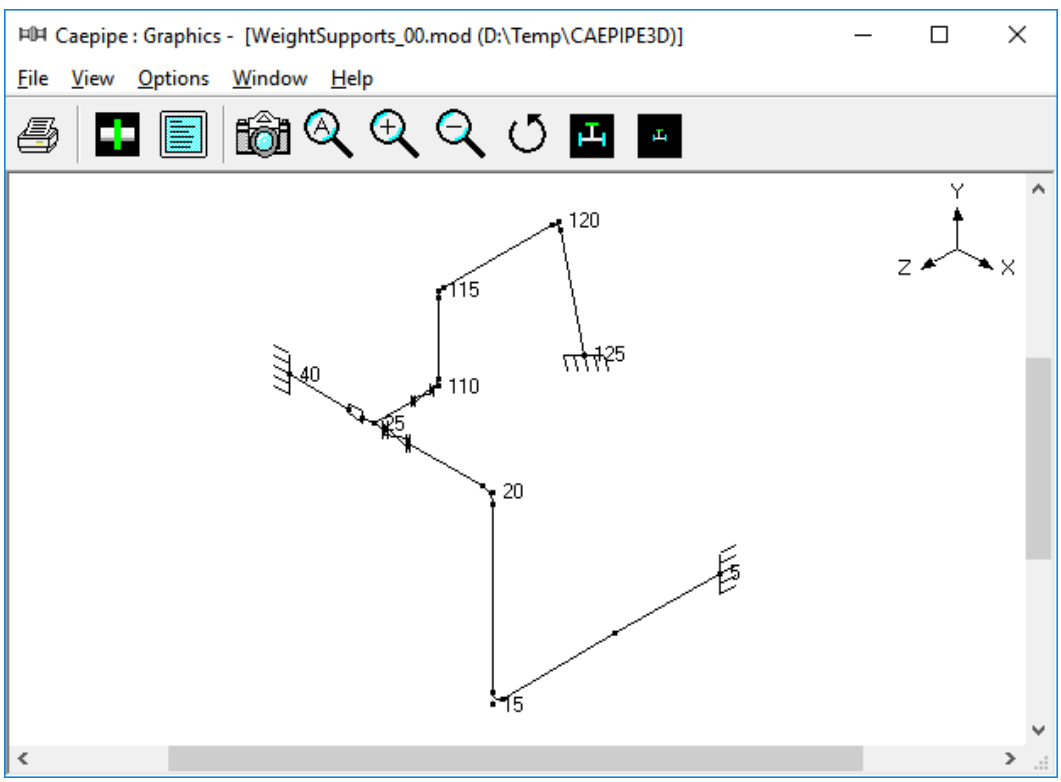


Fig. 4A Layout with Node Numbers

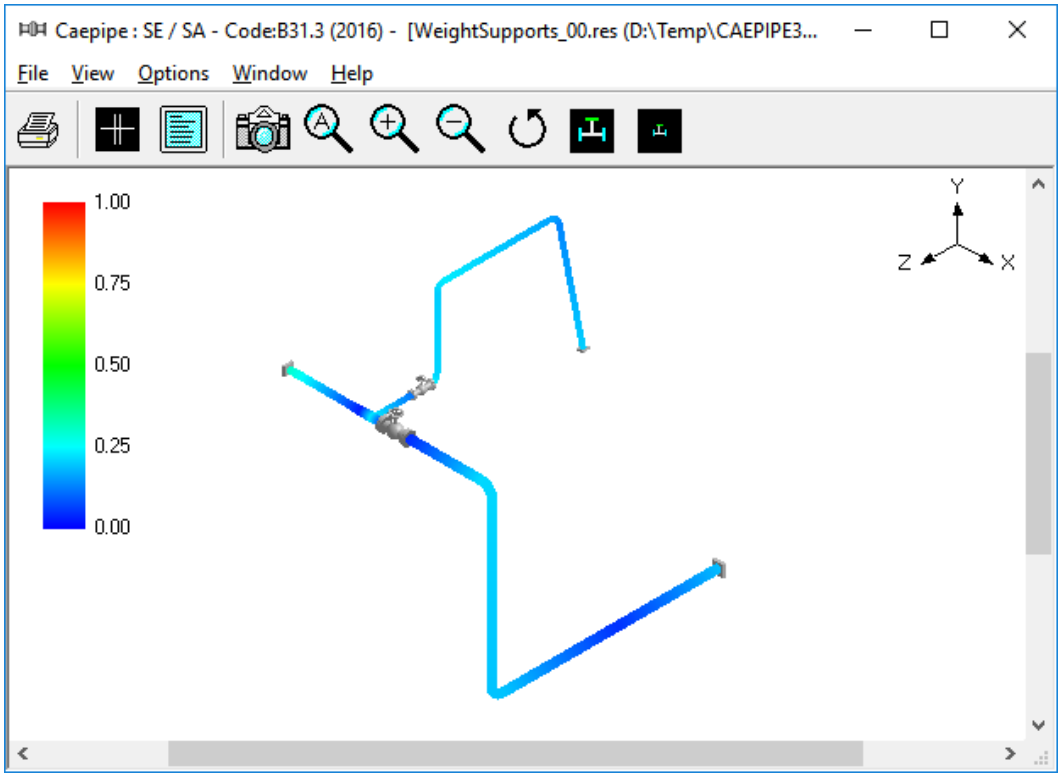


Fig. 4B Thermal Stress Contour Plot

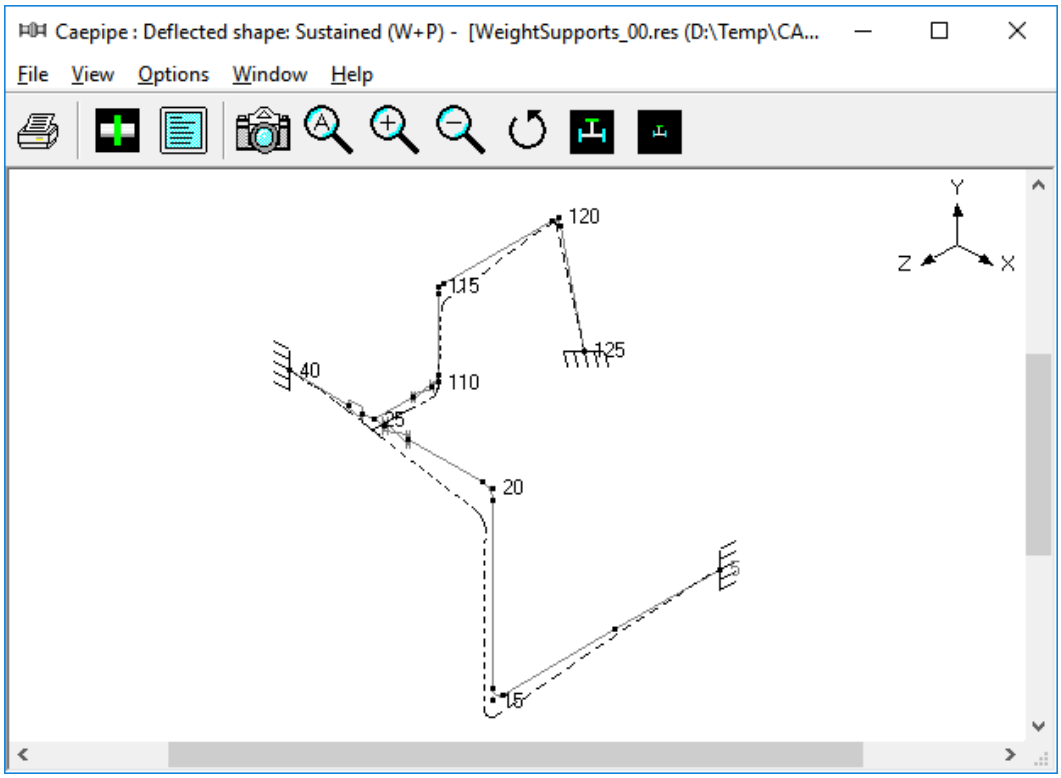


Fig 4C Sustained Load Deflected Shape

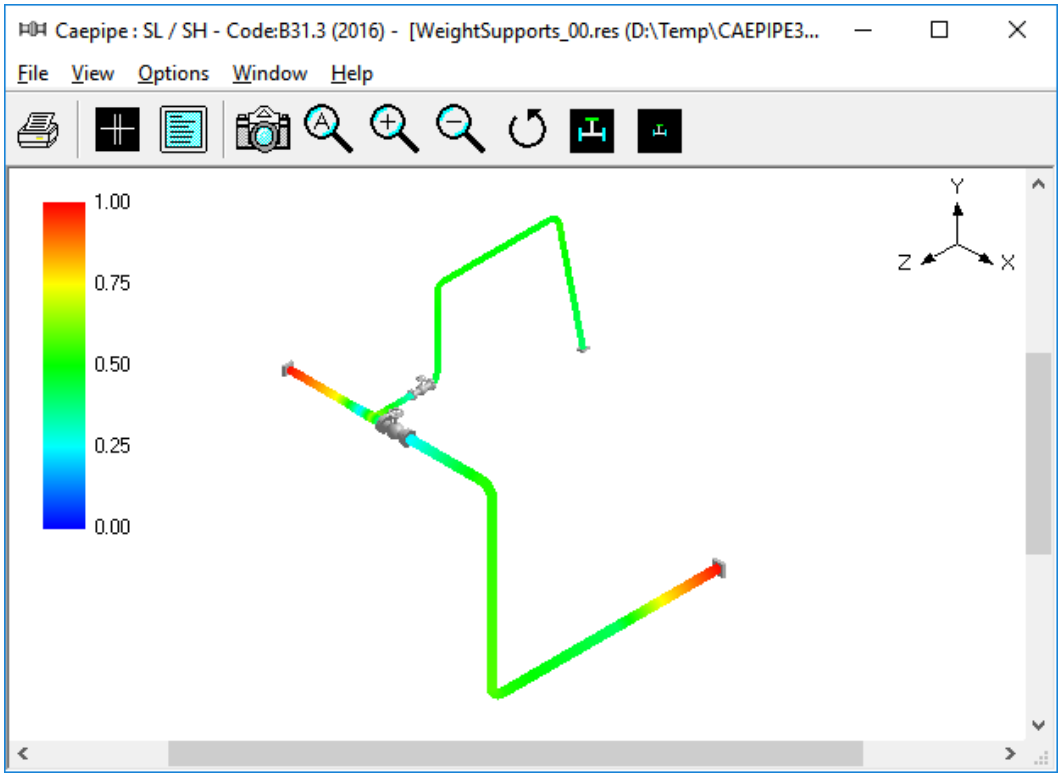


Fig. 4D Sustained Stress Contour Plot

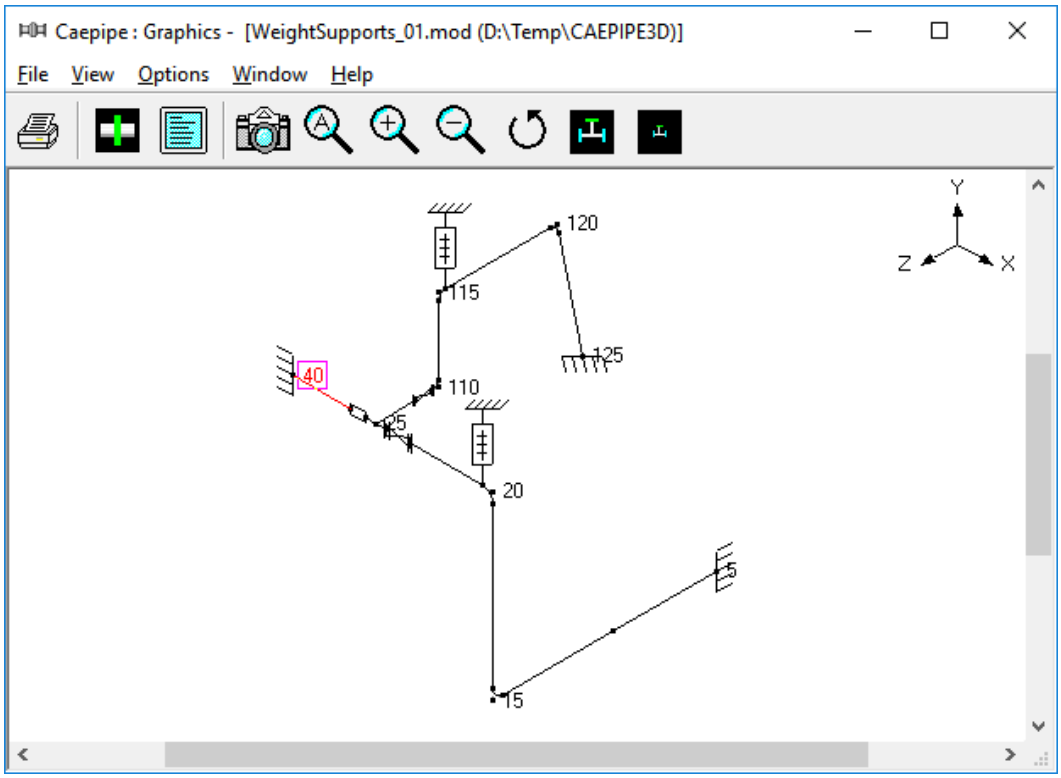


Fig. 4E Layout with Hangers

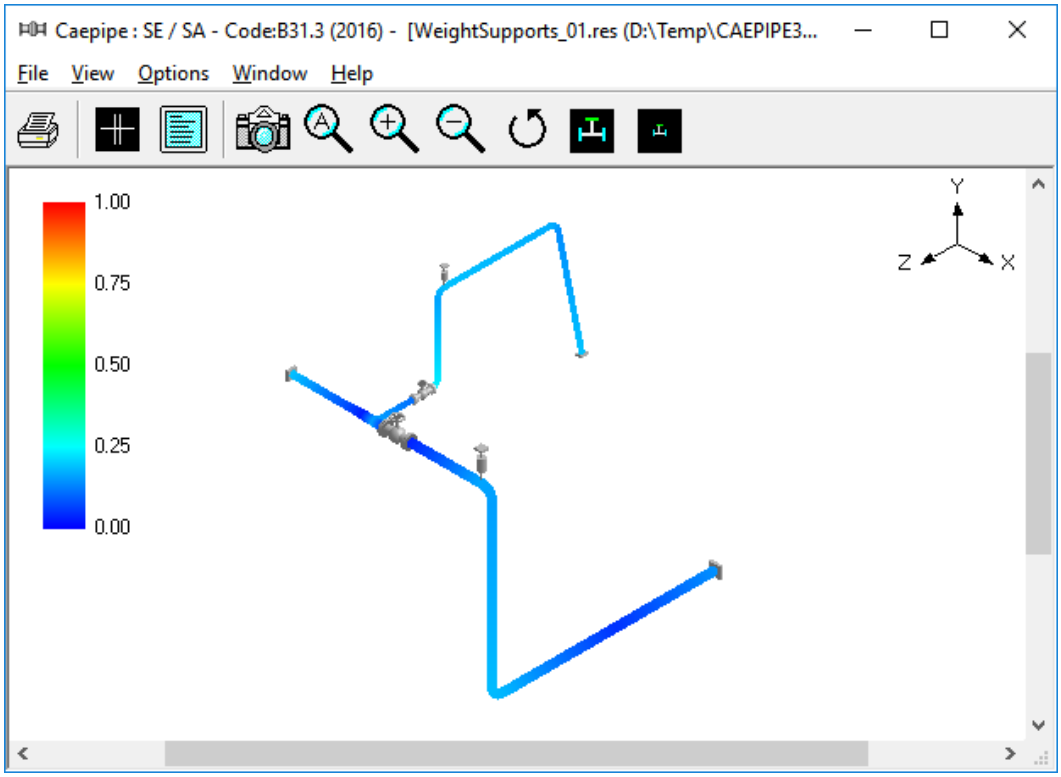


Fig. 4F Thermal Stress Contour Plot for Layout with Hangers

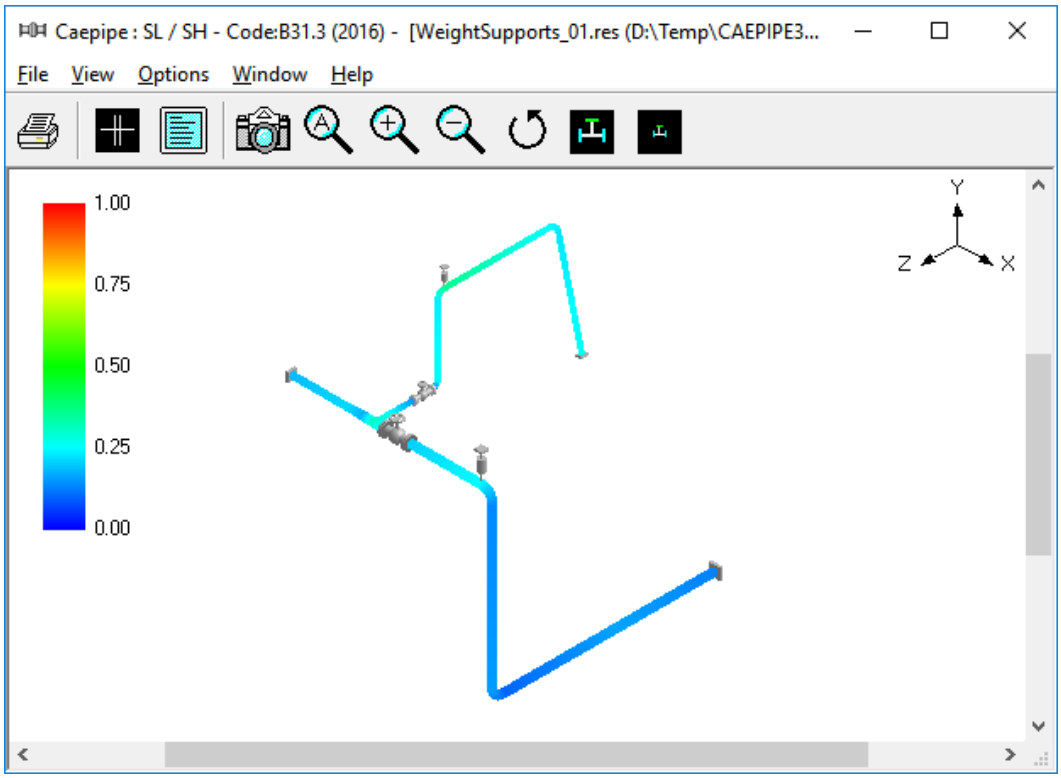


Fig. 4G Sustained Stress Contour Plot for Layout with Hangers

Sample: 5 (Condensate_00, 01 and 02)

This practical problem illustrates how to place resting steel supports to carry the weight of the system with operating fluid as well as to modify the layout in order to re-direct thermal growth to comply with code stress requirements.

Fig.5A shows the initial layout where condensate from a tank is extracted by the pump suction lines. When one pump is operating, the other one is 100% standby.

It is observed from the Fig. 5B that the pipeline from node 10 to node 100 thermally grows in the –Z direction, whereas the two pump suction lines, one from node 120 to node 180 and the other from node 110 to node 250, thermally grow in the +Z direction. So, the straight pipe between nodes 100 and 120 (with a welding tee at node 110) experiences two opposing deflection patterns. The pipe portion between nodes 110 and 120 is being deflected in the +Z direction like a rigid stick; on the other hand, the portion between nodes 100 and 110 is being bent at tee node 110 as the node 100 deflects in –Z direction. This deflection response, in turn, produces high strains and thermal stresses locally at the tee node 110, as shown in Fig. 5C.

In order to reduce the high local thermal stresses at node 110, we cut the straight pipe between nodes 100 and 120 into two parts; one part is the pipe from node 100 to node 110 and the second part is from node 110 to node 120. We then shifted the second part downstream towards the two pumps, resulting in the modified layout shown in Fig. 5D. Fortunately, this shift of pipe downstream would not adversely increase the pressure drop between the tank at node 10 and the pumps at nodes 180 and 250.

From the thermal deformation plot for this revised layout shown in Fig. 5E, it is observed that the two pump suction lines from the suction nozzles to the welding tee at node 111 have almost equal thermal growth in the +Z direction, thereby moving the branch pipe between nodes 111 and 300 as a rigid stick resulting in low thermal stresses in that branch pipe as seen in Fig. 5F. In addition, it is observed that the pump suction lines from the bend node 100 to the pump suction nozzles thermally grow in the +Z direction, whereas the pipe from the tank node 10 to the bend node 90 grow in the –Z direction; this opposing deflections rotate the inter connecting pipe between nodes 90 and 100 like a “see-saw” in the horizontal XZ plane, resulting in low thermal stresses in this region, as observed in Fig. 5F.

Although the thermal stress criteria have been met, the weight stresses exceed the sustained stress allowable, as illustrated by many red and orange areas in the sustained stress contour plot given in Fig 5G. This is because there are no vertical supports (excluding the 3 nozzles and a variable spring hanger at node 52) to carry the weight of the system.

Now, vertical resting supports are introduced as shown in Fig.5H and the corresponding sustained stress (i.e., weight + pressure) contour plot (with most areas in blue) shown in Fig.5J confirms that the sustained stresses are well below the allowable values.

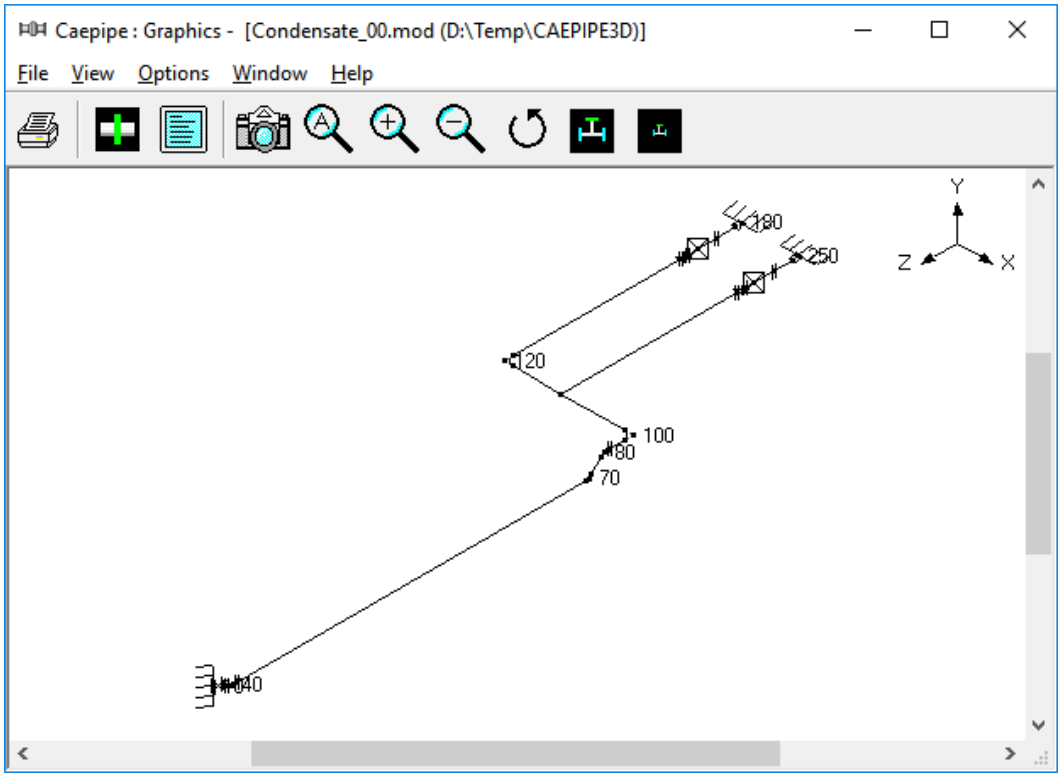


Fig. 5A Layout with Node Numbers

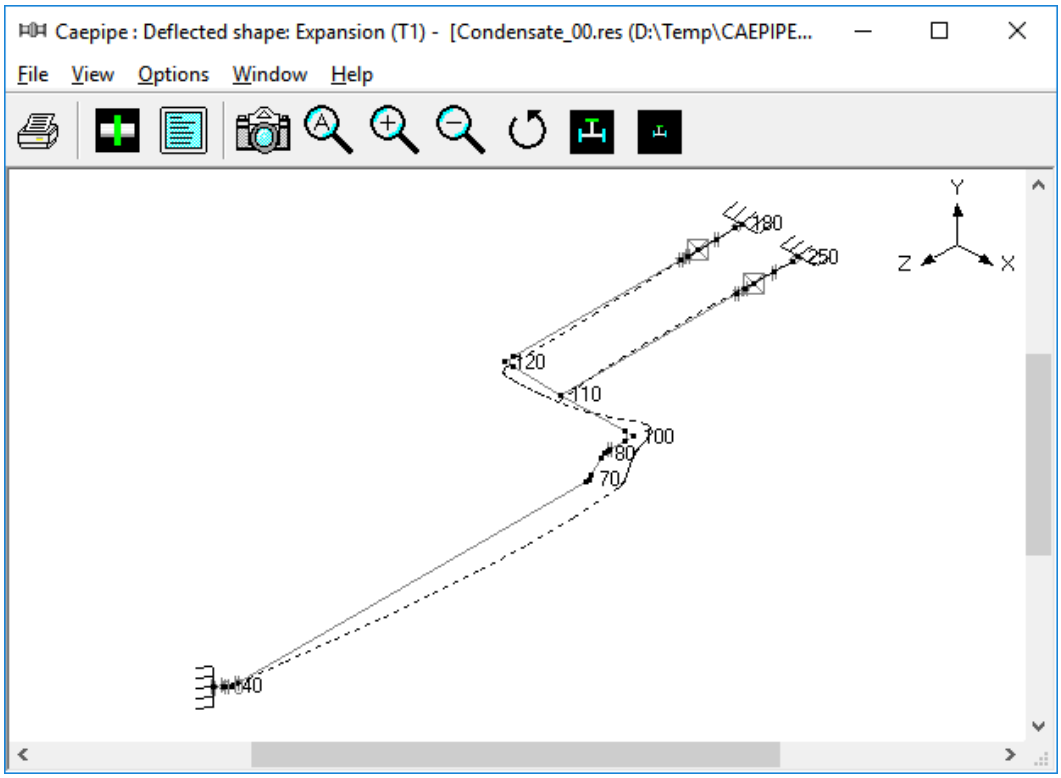


Fig. 5B Thermal Deformation Plot

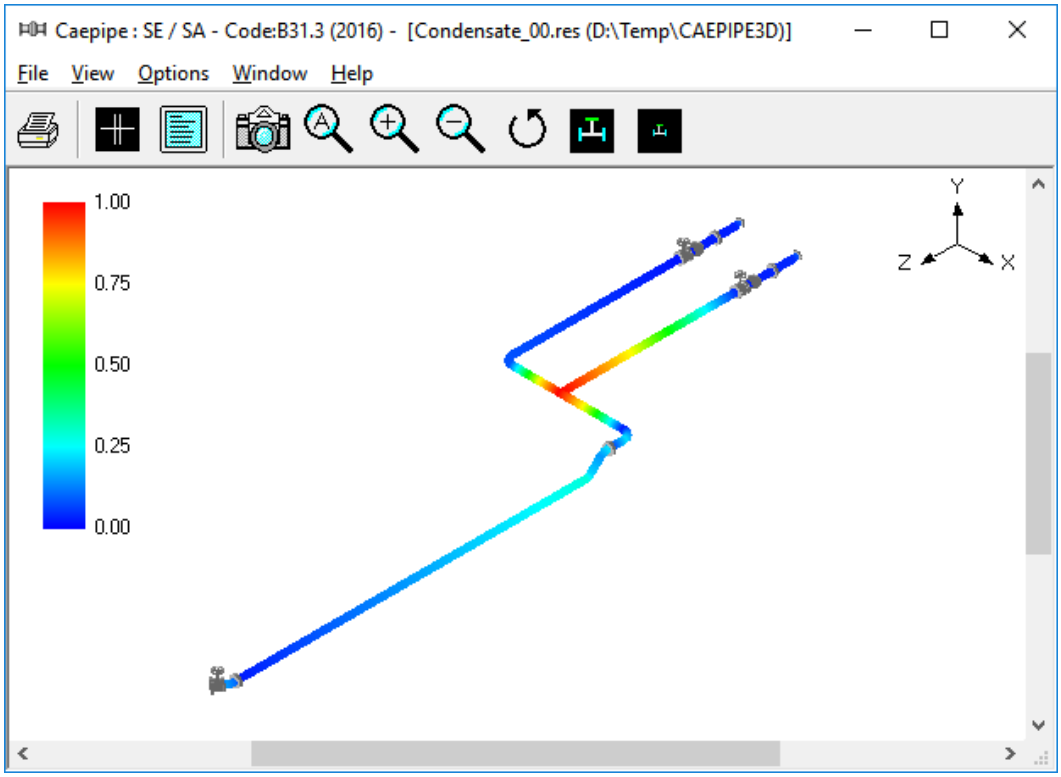


Fig. 5C Thermal Stress Contour Plot

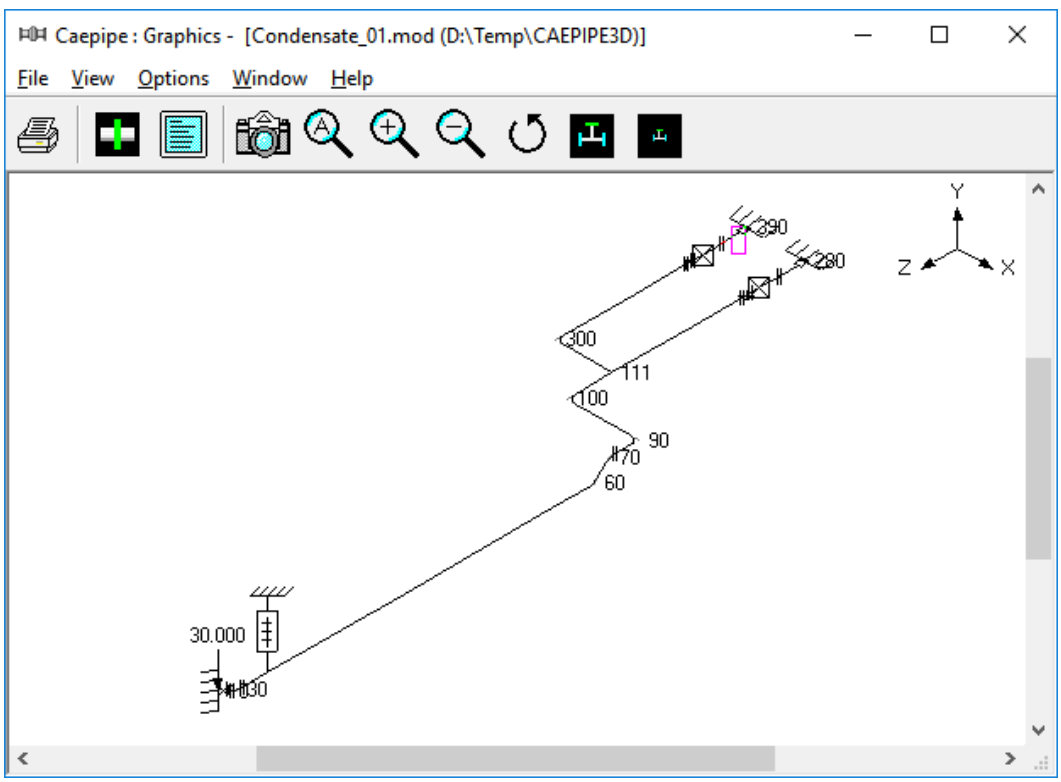


Fig. 5D Revised Layout with Node Numbers

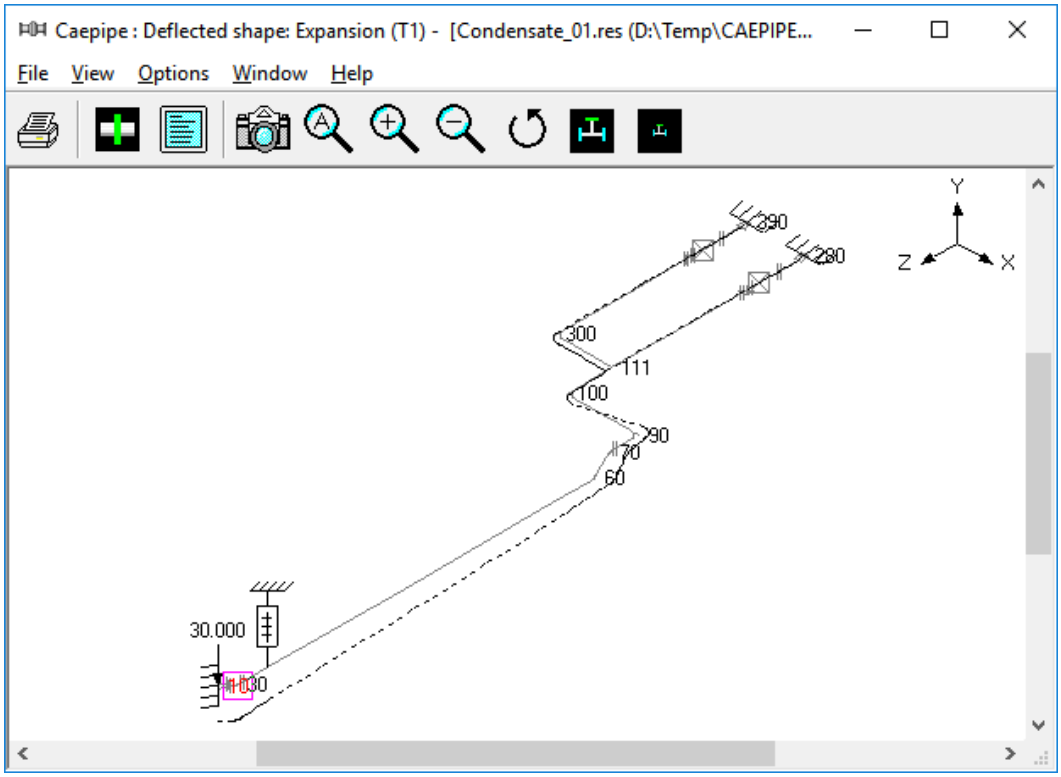


Fig. 5E Thermal Deformation Plot for Revised Layout

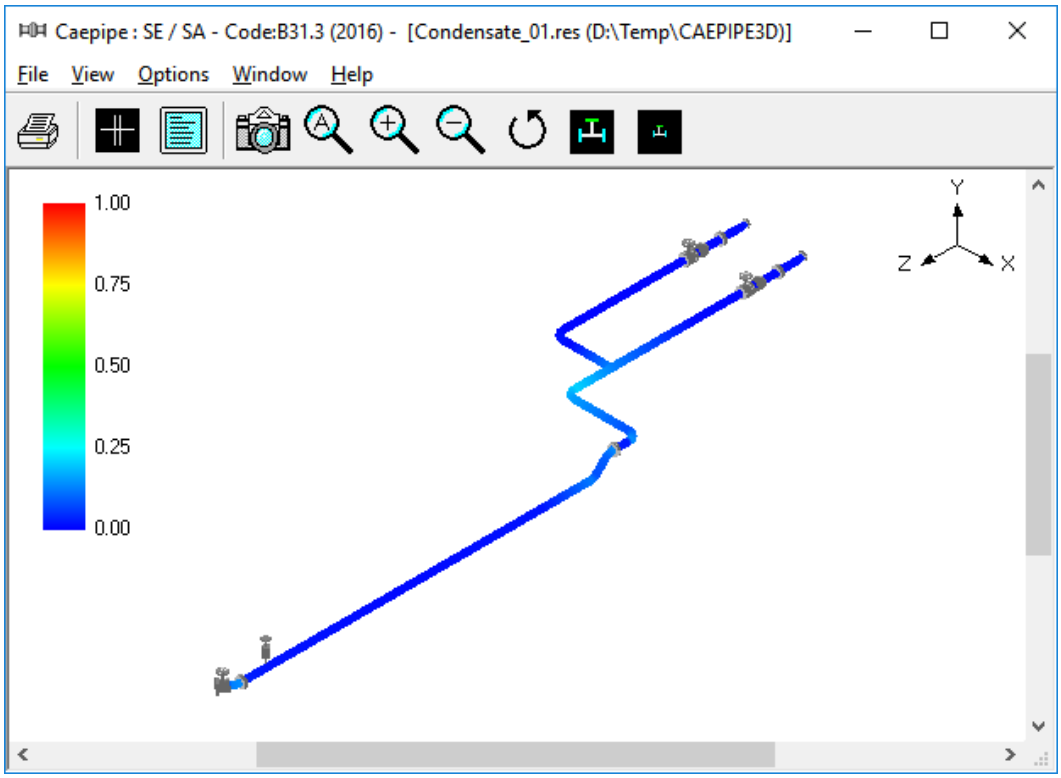


Fig. 5F Thermal Stress Contour Plot for Revised Layout

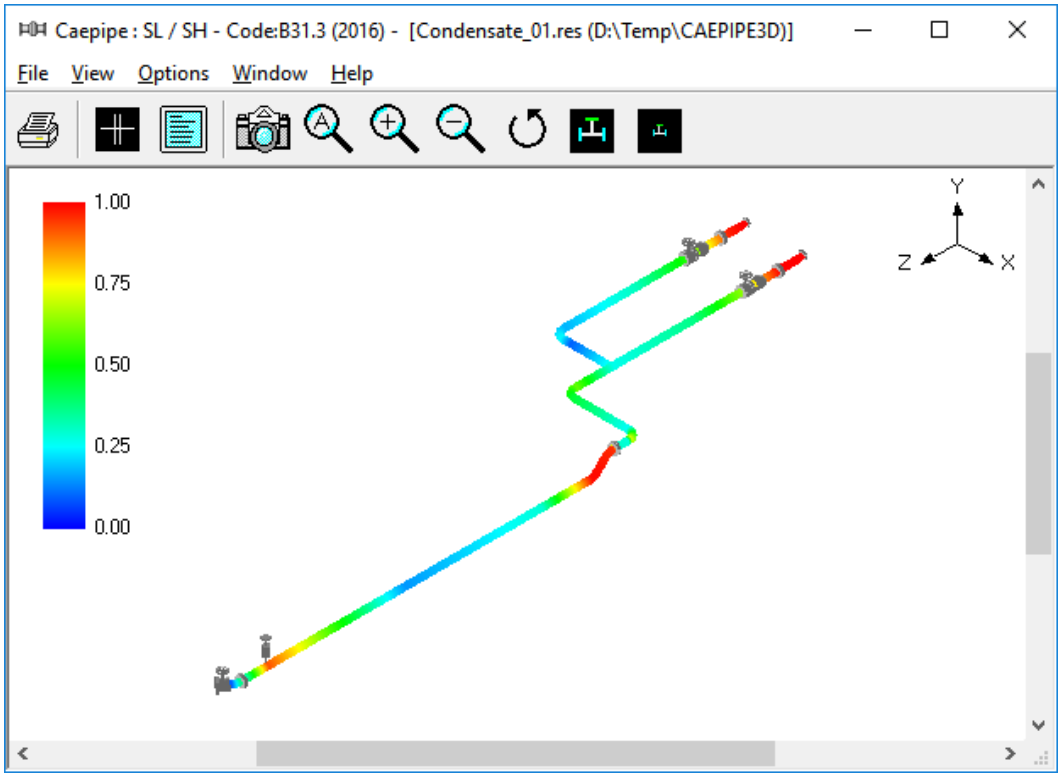


Fig. 5G Sustained Stress Contour Plot for Revised Layout

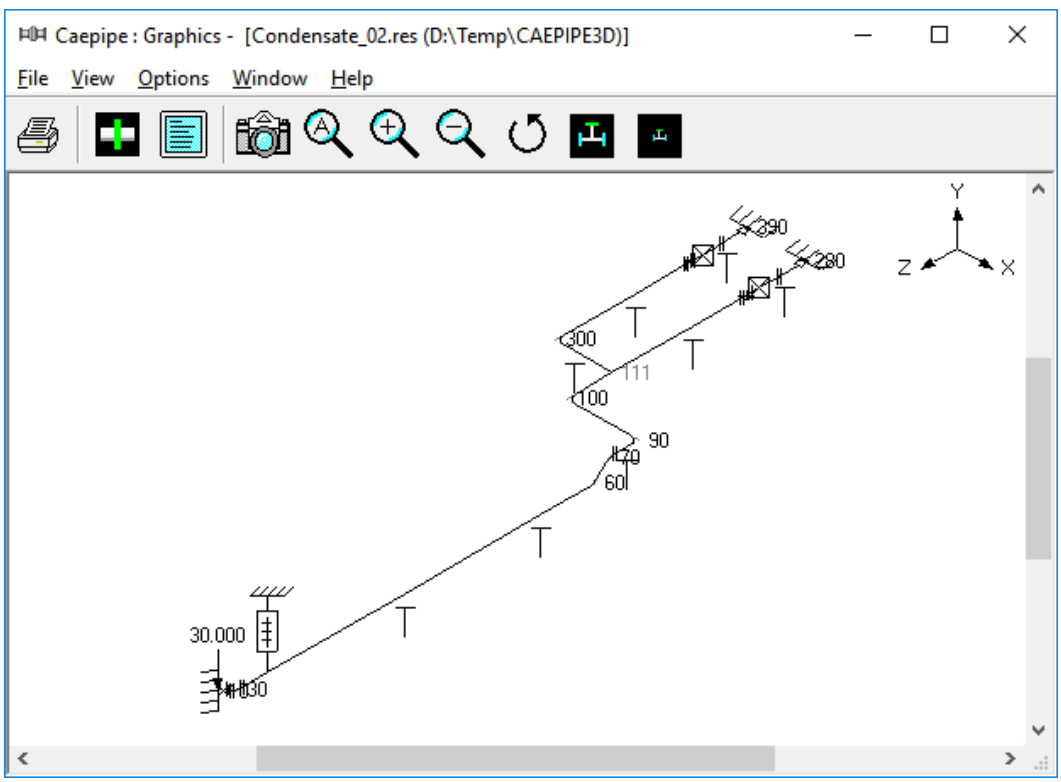


Fig. 5H Revised Layout with Resting Supports

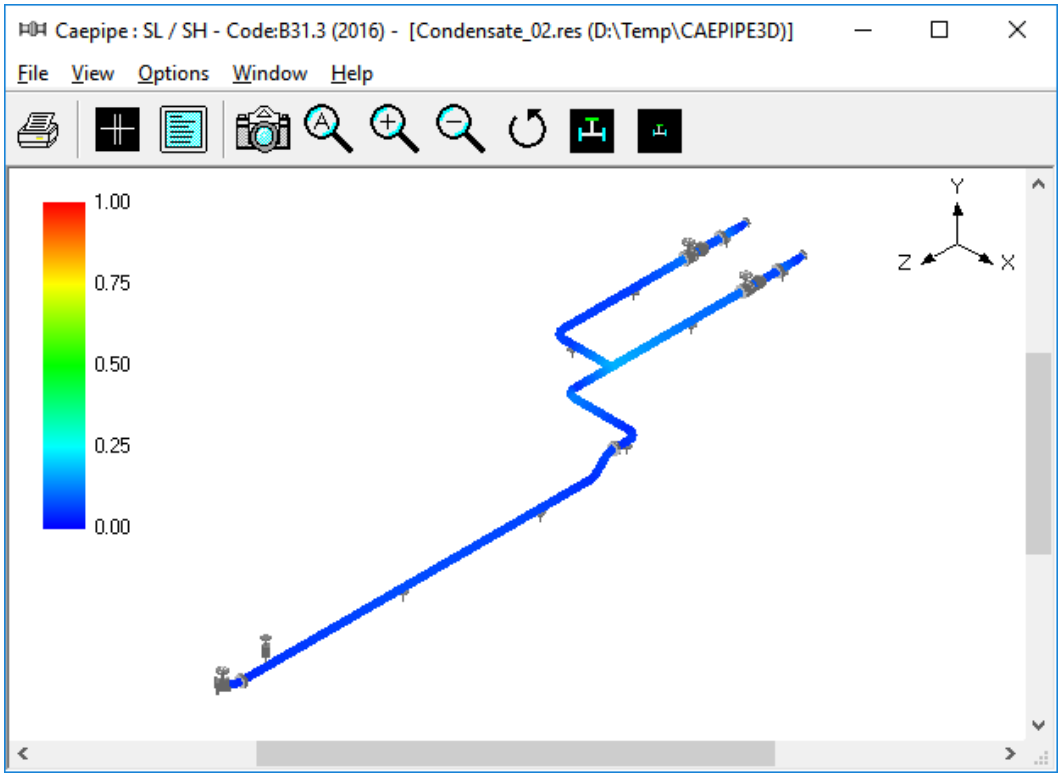


Fig. 5J Sustained Stress Contour Plot for Revised Layout with Resting Supports

CAEPIPE 3D+ for PDMS/E3D

Installing and Configuring Program

To install CAEPIPE 3D+ (for PDMS/E3D) on Windows OS, follow the steps given below:

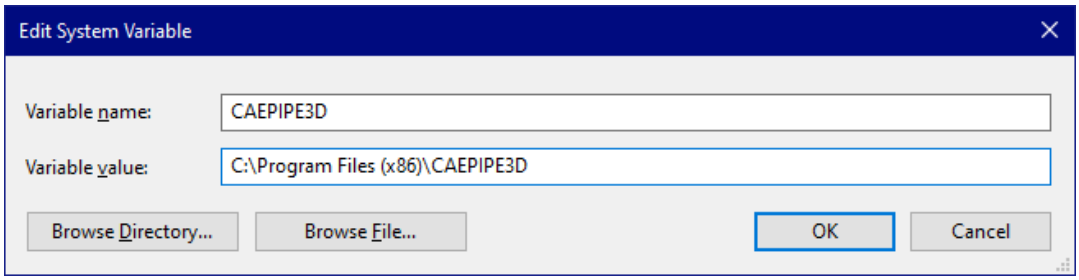
1 Download and run the program "Setup.exe" and follow the instructions as they appear on the screen.

2 Add the path of the installed directory to the "pmllib" path of the PDMS/E3D.

3 For example SET PMLLIB=C:\Program Files (x86)\CAEPIPE3D;%pmllib% (assuming CAEPIPE 3D+ is installed in the directory C:\Program Files (x86)\CAEPIPE3D).

Note: Since PML required for CAEPIPE 3D+ is accessed directly from the installation folder of CAEPIPE 3D+ as per the above statement, E3D will create a PML reference file with name pml.index in the installation folder. So, ensure that the user has write access to the installation folder of CAEPIPE 3D+. If the write access to the installation folder of CAEPIPE 3D+ is not available to the users, then copy the folder “Setup” available in the installation folder of CAEPIPE 3D+ to the PMLLIB folder of E3D installation directory and rename it from “Setup” to “CAEPIPE3D”.

4 Define an environmental variable with name as “CAEPIPE3D” and its value as Installation path. For example, if the product is installed in “C:\Program Files (x86)\CAEPIPE3D” then input the value as “C:\Program Files (x86)\CAEPIPE3D” in the Edit System Variable dialog as shown below.



5 The environment variables required by PDMS/E3D are also required for CAEPIPE3D. The PDMS/E3D environment must exist as outlined in the appropriate PDMS/E3D Installation Guide.

6 Load PDMS/E3D and enter the Lexicon Module as a user who has Read-Write access to DICT Database and run the datal file CAEPIPE3D.DAT, to create the UDA: SUPCODE for ATTA/ANCI elements, UDA :NX, :NY, :NZ and :NAL for Nozzle elements, UDA: :HNX, :HNY, :HNZ, :TNX, :TNY, :TNZ, :HNAL and :TNAL for Branch elements.

Note: Skip Step 6, if you have configured checkSTRESS or PDMS/E3D to CAEPIPE Translator with the PDMS/E3D project on which you will be using CAEPIPE 3D+.

How to use CAEPIPE 3D+?

1 Add the ZONE/PIPES/BRANCHES using “Add CE” or CE Members” button.

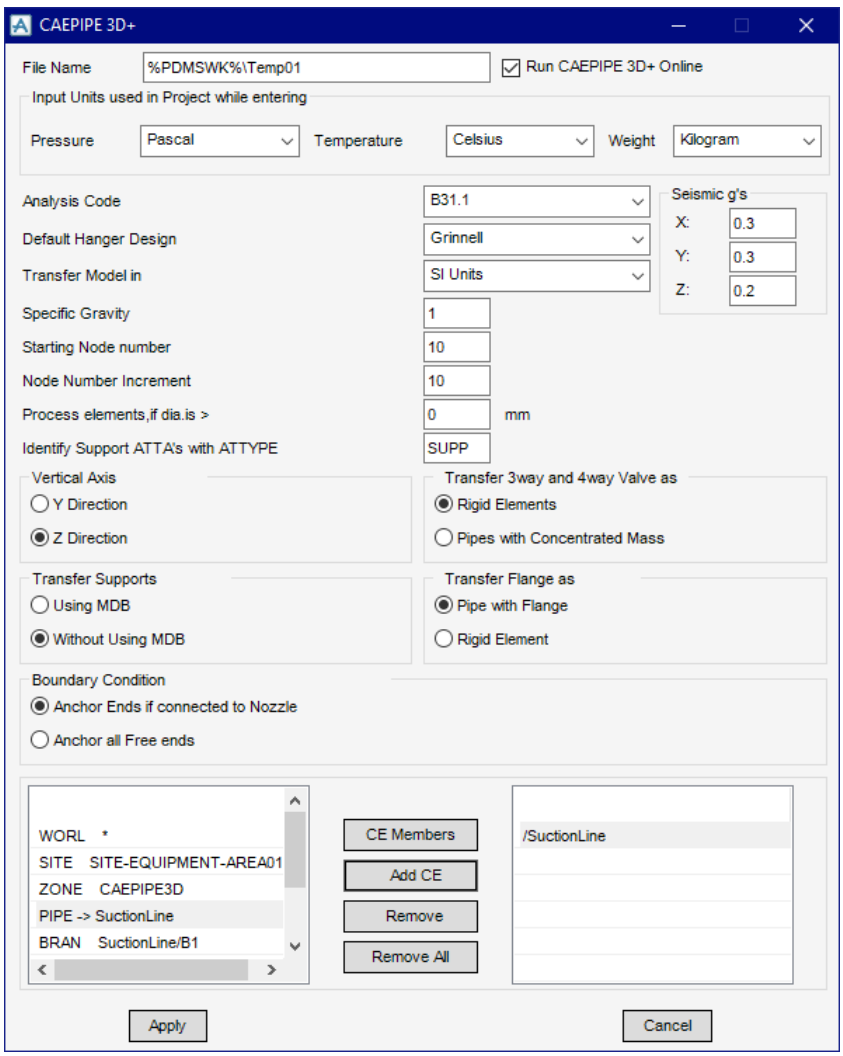
2 Branches selected for transfer need to be checked for the following.

- 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 analysis.

3 For transferring the support details, ATTA’s have to be created at support locations and the UDA (:SUPCODE or :KPSUPCODE) should be set a value as given in the field #1 (i.e., “PdSupport”) of the access db “SupportType.mdb”.

4 To use CAEPIPE 3D+ (for PDMS/E3D), type “show !!CAEPIPE3D” on the command line of PDMS/E3D Design Module. The following form appears,



- 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 it finds a space in the file name / path.

- To run **CAEPIPE 3D+** offline in PDMS or E3D, uncheck the "Run CAEPIPE 3D+ Online" option. This will prompt **CAEPIPE 3D+** to export the piping layout from E3D or PDMS into a neutral file with the `.n` extension. You can then input the exported file by launching **CAEPIPE 3D+ for PDMS/E3D** through the "CAEPIPE 3D+ for PDMS" or "CAEPIPE 3D+ for E3D" shortcuts, which are available in the Windows Start menu under **CAEPIPE3D**.
- 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 result in incorrect output.
- Select the piping code to be used for the analysis of the model from the list box.
- Enter/modify Static Seismic g’s.
- 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 CAEPIPE 3D+ model. This Hanger type will be used only when the hanger type in CAEPIPE 3D+ corresponding to Plant Design hanger is not specified /available in the Mapping DB. For details on transferring support from Plant Design to CAEPIPE 3D+, refer Appendix D.
- This option is valid in both E3D/PDMS when ATTA is used in piping to define supports. i.e., When the option “Identify Support ATTA’s with ATTYPE attribute set as” is entered with a Keyword, then it will transfer support locations and its information defined via ATTA 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. On the other hand, if ANCIs are used in piping to define Supports in E3D, then all ANCIs corresponding to each Branch will be transferred automatically by default.
- Select the units to be used for the CAEPIPE 3D+ model.

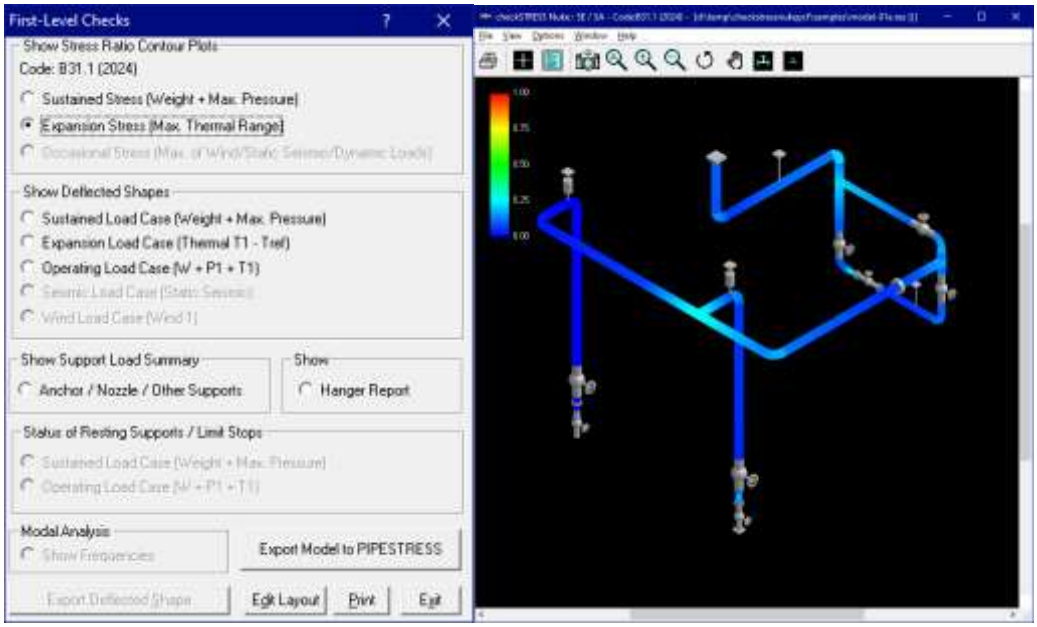
  Selecting the option “SI Units” transfers the Pipes/Branches details in SI units i.e. Length related dimensions such as OD, Nominal Size etc in mm, Temperature in Deg C, Pressure in bar, Weight in Kg, Density in Kg/m3, Translational Stiffness in N/mm, Rotational stiffness in N-m/deg to the neutral file.

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” from this document for more 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 CAEPIPE 3D+. Refer the section titled “Reference” from this document for more details.
- Navigate to the required ZONE/PIPE/BRANCH through the member list as shown in the figure above and press the button “Add CE” to include and to perform the Stress Analysis for the selected ZONE/PIPE/BRANCH.
- User can also use the button “CE Members” to include all the members of SITE/ZONE/PIPE and then exclude the members that are not required for analysis using the “Remove” button. 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 CAEPIPE 3D+ model from the vertical Axis option. “Z Axis” shall be taken as Global Vertical Axis by default.

  Note: Since E3D/PDMS consistently uses the vertical global axis as “Z,” and stress engineers from different regions may use different coordinate systems, it is advised that designers consult with the stress department to define the “Vertical Global Axis” before saving the stress model in CAEPIPE format for detailed analysis. However, for performing the first level stress check using CAEPIPE 3D+, designers should continue to use “Z” as the Global Vertical Axis.
- Press the button “Apply” to perform the Analysis.

5. Figure below shows the stress contour plot for the Expansion case.



Limitations

The following are not transferred to CAEPIPE 3D+ from PDMS/E3D at this time. However, user can add these in CAEPIPE 3D+ 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 CAEPIPE 3D+. 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 CAEPIPE 3D+ through Edit Layout->Misc->Loads.

Fluid Density

Even though the provision is available in PDMS/E3D 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 CAEPIPE 3D+. 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 manually by selecting the component type such as Valve, Rigid, Flange, etc. through Edit Layout->Window->List.

Wall Thickness

Wall thickness of the Piping components is generally not available in PDMS/E3D. Hence, they are set to 0.0 by default. However, user can transfer the Wall Thickness information to CAEPIPE 3D+ properly from PDMS/E3D (if they are specified in the project) by modifying the variables defined in the program. Refer section titled “Specifying Wall Thickness” below for details on modifying the program to extract Wall Thickness from PDMS/E3D.

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 this manual.

In addition to the above options, user can also modify the Wall thickness of the piping components directly in CAEPIPE 3D+ through Edit Layout->Misc->Section.

OD and Nominal Size

OD and Nominal Size are extracted from the Paragon Database of PDMS/E3D. For reducers, the arrive OD and Thickness shall be transferred to OD1 and Thk1 of the element in CAEPIPE 3D+ and the leave OD and Thickness shall be transferred as OD2 and Thk2 of the element in CAEPIPE 3D+.

User can also modify the OD and Nominal size of the piping components directly in CAEPIPE 3D+ 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 CAEPIPE 3D+ directly through Edit Layout->Misc->Data Types.

Supports

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

Step 1: Include/Exclude ATTA’s for Transfer

PDMS

Define ATTA and fill the attribute “ATTYPE” 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 “ATTYPE” 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 CAEPIPE 3D+.

- 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 CAEPIPE 3D+.
- 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 CAEPIPE 3D+ is selected. Otherwise, the attribute should be filled with values as specified in **Appendix E** of this manual. For details on transferring supports "Without Using Mapping DB", refer Appendix D of this manual

In addition to the above, user can also add / modify supports in piping system in CAEPIPE 3D+ 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 this manual.

User can add / modify the Material properties in CAEPIPE3D+ 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 CAEPIPE 3D+. TAM values should be defined in "mm" for PDMS/E3D projects in SI units and in "Inch" for PDMS/E3D project in English units.

If the Equipment Nozzle to which the Piping Layout is connecting to is not modeled in PDMS/E3D, then these TAM values can be entered at UDA :HNX, :HNY and :HNX or :TNX, :TNY and :TNZ at PDMS/E3D 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 PDMS/E3D Branch. On the other hand, TAM at the end of the Pipe run should be entered at :TNX, TNY and TNZ of PDMS/E3D 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 PDMS/E3D, then these Equipment Nozzle Allowable Loads can be entered at UDA :HNAL or :TNAL at PDMS/E3D 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 PDMS/E3D Branch. On the other hand, Allowable Loads at the end of the Pipe run should be entered at :TNAL of PDMS/E3D Branch.

Example:

For Nozzle, enter the values of FX, FY, FZ, MX, MY and MZ as “:NAL ‘3000,3000,6000,500,500,2000’”

For PDMS/E3D 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.

Specifying Wall Thickness

This section explains how to extract the wall thickness for PDMS/E3D piping components. To specify the wall thickness for piping components like Pipe, Olet, Tee, Elbow, Reducer and Cross, navigate to “CAEPIPE 3D+” 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 PDMS/E3D, 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()
```

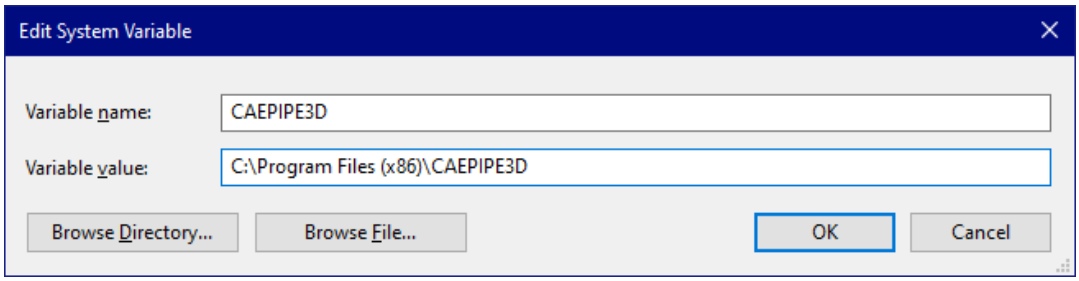
CAEPIPE 3D+ for CADMATIC

Installing and Configuring Program

To install CAEPIPE 3D+ on Windows OS, follow the steps given below:

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

2. After successful installation of the program, define an environmental variable with name as "CAEPIPE3D" and its value as Installation path. For example, if the product is installed in "C:\Program Files (x86)\CAEPIPE3D" then input the value as "C:\Program Files (x86)\CAEPIPE3D" in the Edit System Variable dialog as shown below.



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 or select the Environment tab and click "New" button.

- Then enter in the Variable prompt "CAEPIPE3D", and "CAEPIPE 3D+_installed_path" (where the CAEPIPE 3D+ is installed) in the Value prompt, for example "C:\Program Files (x86)\CAEPIPE3D".

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

How to use CAEPIPE 3D+?

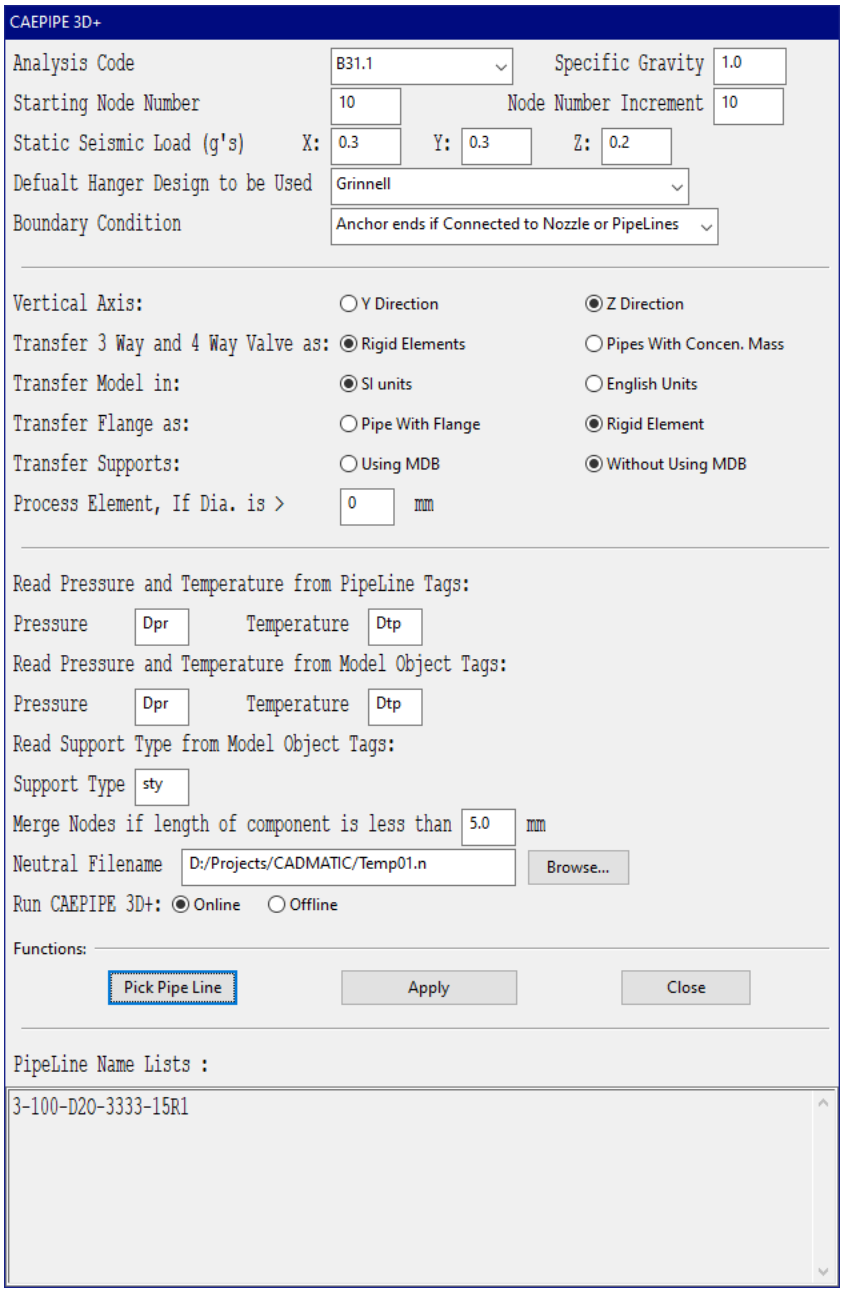
1 Pipes selected for Transfer need to be checked for the following.

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

b. Support type can be transferred to CAEPIPE 3D+ 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 CAEPIPE 3D+ 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.

Load the CADMATIC Plant Modeler and run the macro “Setup\CAEPIPE3D.mac” available in the installed path of CAEPIPE 3D+ through “Run Script”. The following form appears.



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

b. Enter/modify the Static Seismic Load (g’s).

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 CAEPIPE 3D+ model. This Hanger type will be used only when the hanger type in CAEPIPE 3D+ 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 CAEPIPE 3D+ model from the vertical Axis option. By default, “Z Axis” shall be taken as Global Vertical Axis if not specified.

   Note: Since CADMATIC consistently uses the vertical global axis as “Z,” and stress engineers from different regions may use different coordinate systems, it is advised that designers consult with the stress department to define the “Vertical Global Axis” before saving the stress model in CAEPIPE format for detailed analysis. However, for performing the first level stress check using CAEPIPE 3D+, designers should continue to use “Z” as the Global Vertical Axis.

   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.

h. Specify how the 3 Way and 4 Way valves to be transferred to CAEPIPE 3D+. Refer section 4.0 for details.

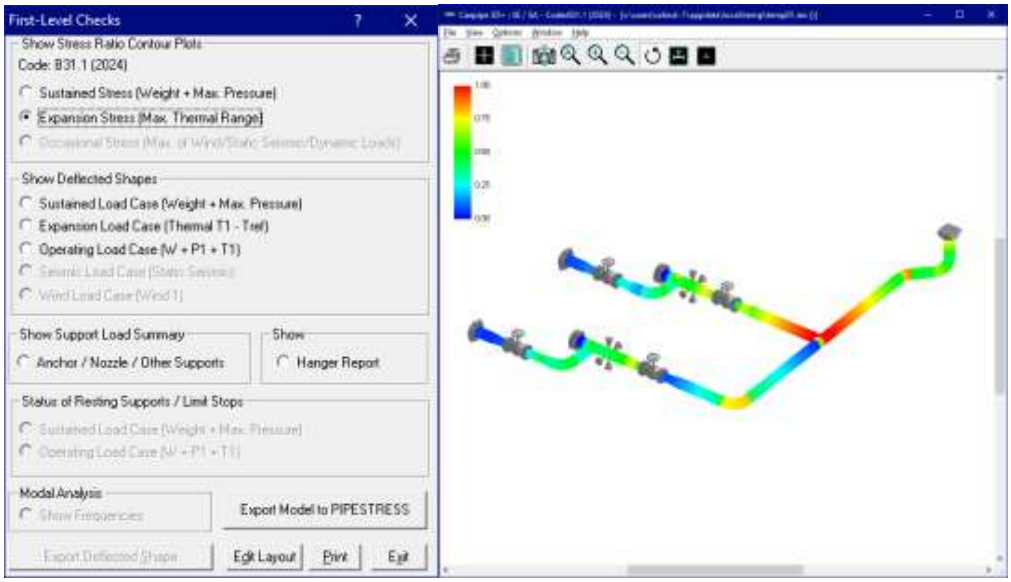
i. Specify how the Flanges are to be transferred to CAEPIPE 3D+. Refer section 3.0 for details.

j. Specify whether the Mapping DB shall be used during transfer or not.

k. To run **CAEPIPE 3D+** offline in CADMATIC, select the option "Run CAEPIPE 3D+ Offline". This will prompt **CAEPIPE 3D+** to export the piping layout from CADMATIC into a neutral file with the `.n` extension. You can then input the exported file by launching **CAEPIPE 3D+** through the "CAEPIPE 3D+ for CADMATIC" shortcut, which is available in the Windows Start menu under **CAEPIPE3D**.

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. Figure below shows the stress contour plot for the Expansion case.



Limitations

The following are not transferred to CAEPIPE 3D+ from CADMATIC database at this time. However user can add these in CAEPIPE 3D+ through Edit Layout->Misc->Section

3 Insulation density and Insulation thickness.

4 Corrosion allowance and Mill tolerance.

5 Lining Density and Lining Thickness.

Reference

Loads

By default, Temperature and Pressure values entered at Pipes and Standard components via OpT and opP shall be transferred to CAEPIPE 3D+. If the Temperature and Pressure values for Pipes and Standard components are 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 CAEPIPE 3D+ 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 CAEPIPE 3D+.

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 CAEPIPE 3D+ through Edit Layout->Misc->Section.

Flange

Flange from CADMATIC can be transferred as “Pipe with Flange” or “Rigid element” to CAEPIPE 3D+. 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 CAEPIPE 3D+ with flange type as “Weld Neck”. The OD and Thickness (in or mm) corresponding to Flange Nominal Size is transferred to pipe in CAEPIPE 3D+.

If the user selects “Rigid element” option then, the program will transfer the same as “Rigid Element” to CAEPIPE 3D+. Dry weight of flange extracted from CADMATIC will be transferred to “Weight” field of the Rigid Element in CAEPIPE 3D+.

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 CAEPIPE 3D+ 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 CAEPIPE 3D+. Otherwise, the attribute should be filled with values as specified in **Appendix E** of this manual. For more details refer to Appendix D of this 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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

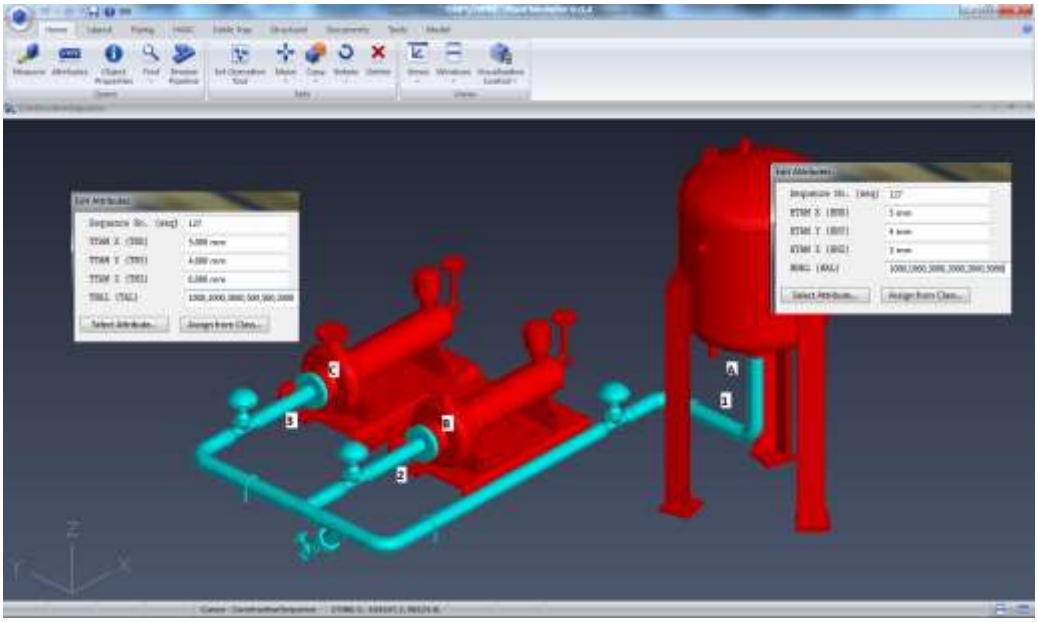
Material description entered at Material tag of corporate catalogue is extracted from the database and the material property is then obtained from the intermediate mapping db. Refer Appendix B of this manual for more details.

User can add / modify the Material properties in CAEPIPE 3D+ through Edit Layout > Misc > Materials.

Thermal Anchor Movement (TAM)

Thermal Anchor Movement (TAM) values entered at User defined attributes HNX, HNY and HNX or TNX, TNY and TNZ (attributes need to be added and assigned for Model Object through CADMATIC COS and their values to be assigned in Plant Modeller using the option “Whole Pipeobject”) corresponding to Start or End of pipe run is transferred to CAEPIPE 3D+. 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.

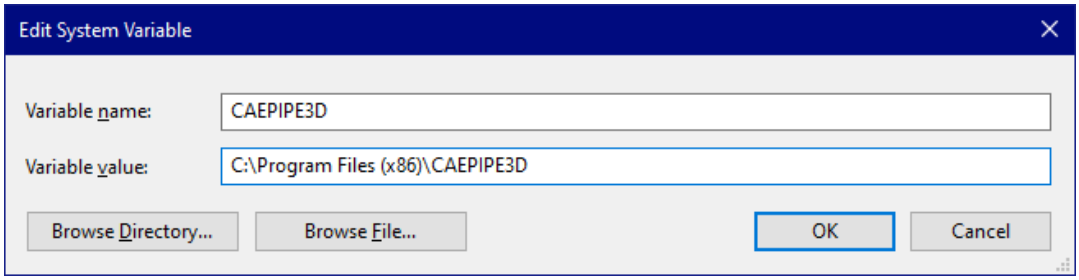
The allowable thus defined are compared against calculated loads and shown / printed in Support Load Summary outputs (can be viewed / printed through “Show Summary” option). If the calculated loads exceed the allowable, they are highlighted in red.

CAEPIPE 3D+ for PCF

Installing and Configuring Program

1 Run the program "Setup.exe" downloaded and follow the instructions as they appear on the screen.

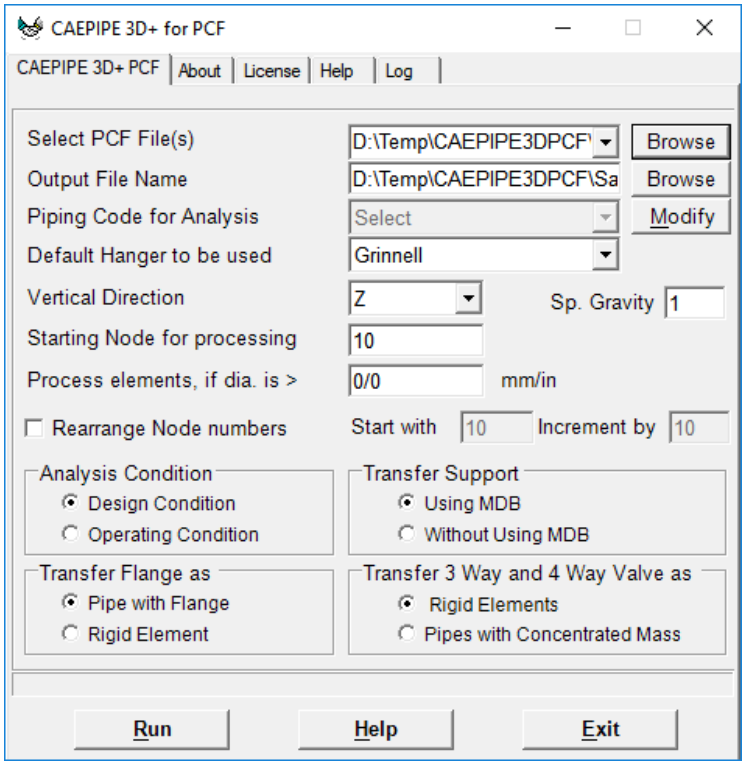
2 Define an environmental variable with name as “CAEPIPE3D” and its value as Installation path. For example, if the product is installed in “C:\Program Files (x86)\CAEPIPE3D” then input the value as “C:\Program Files (x86)\CAEPIPE3D” in the Edit System Variable dialog as shown below.



How to use CAEPIPE 3D+?

1 Select the PCF file(s). Use “shift” key for selecting multiple PCF files. CAEPIPE 3D+ combines the PCF files in to one stress analysis model, when multiple files are selected.

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 Enter the Specific Gravity of fluid with respect to water. By default the specific gravity is set to 1.0.

6 Select Default Hanger to be used in CAEPIPE 3D+. This selected Hanger type will be used in CAEPIPE 3D+, only when the hanger type in CAEPIPE 3D+ corresponding to Plant Design hanger is not specified /available in the Mapping DB. For details on transferring support from Plant Design to CAEPIPE 3D+, refer **Appendix D** of this manual.

7 Specify the Starting Node number for processing network. This is set to 10 by default. If the user is not sure about the node number, then this can be left as 10 or enter as 0.

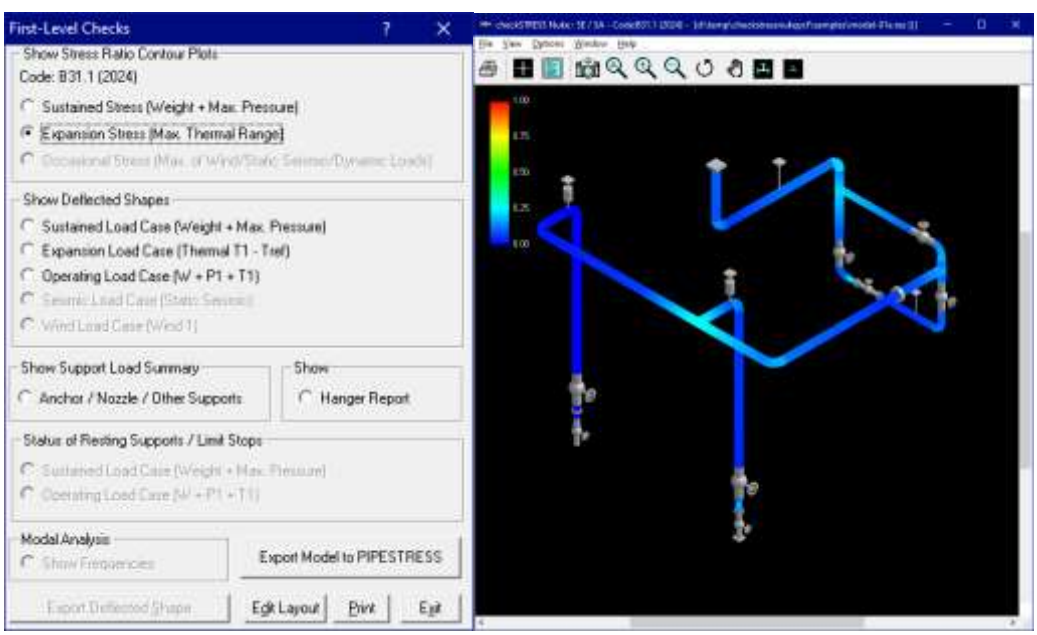
8 If the user wishes to eliminate those elements whose Nominal size (NS) is less than 50mm / 2in, then they can specify the value as “50/2” in the field “Process elements, if dia >”. In other words, the program will process only those elements whose nominal size is greater than the value specified in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

9 Specify how the Flange to be transferred to CAEPIPE 3D+.

10 Specify how the 3 Way and 4 Way Valves are to be transferred to CAEPIPE 3D+.

11 Choose the Global Vertical Axis for CAEPIPE 3D+ 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. Figure below shows the stress contour plot for the Expansion case.



Limitations

The following are not transferred to CAEPIPE 3D+ from Plant Design database at this time. However user can add these in CAEPIPE 3D+ through Edit Layout >Misc >Section

1 Insulation density and Insulation thickness.

2 Corrosion allowance and Mill tolerance.

3 Lining Density and Lining Thickness.

Reference

Units

Piping system from PCF will be transferred to CAEPIPE 3D+ in **SI** units when the attribute “UNITS-CO-ORDS” is defined as “MM” in PCF. Otherwise, the piping system will be transferred to CAEPIPE 3D+ in **ENGLISH** units.

Loads

Temperature for the piping system is read from “PIPELINE-TEMP” attribute of Pipeline Header information (PIPELINE-REFERENCE attribute) in PCF and will be transferred to CAEPIPE 3D+, if available. If not available in PCF, then the Temperature for the piping system will be transferred as 21 deg C / 70 deg F for **SI / English** units.

Pressure for the piping system is read from “PIPELINE-PRES” attribute of Pipeline Header information (PIPELINE-REFRENCE attribute) in PCF and will be transferred to CAEPIPE 3D+. If not available in PCF then the Pressure will be transferred as 0 to CAEPIPE 3D+.

User can add / modify the Temperature and Pressure for the Piping System through Edit Layout->Misc->Loads

Note:

Program will assume that the value of the attribute “PIPELINE-TEMP” and “PIPELINE_PRES” available in PCF as in deg.F and psi, if the “UNITS-CO-ORDS” attribute is defined as “ENGLISH” in PCF. On the other hand, the program will assume that the value of the attribute “PIPELINE-TEMP” and “PIPELINE-PRES” available in PCF as deg.C and bar, if the “UNITS-CO-ORDS” attribute is defined as “MM” or “MM-HUNDREDTH” in PCF.

Fluid Density

Specific Gravity of the fluid (with respect to water) entered in the form as shown above will be used by CAEPIPE 3D+ for calculating the fluid weight in the piping system.

Weight

The weights of Valves, Instruments, Flanges, etc. are read from “WEIGHT” attribute of component data from PCF and will be transferred to CAEPIPE 3D+, if available. If not available, then the weight will be transferred as 0.0 to CAEPIPE 3D+.

User can add / modify the weight of components through Edit Layout->Window->List.

Wall Thickness

The wall thicknesses of piping components are not typically included in the PCF file by default. However, if the Plant Design software is customized to export the wall thickness for each component node using attributes labelled WALL-THICKNESS<n> (where n ranges from 1 to 2 depending on the number of

nominal sizes the component has) then the program will read these values and transfers them to CAEPIPE 3D+. See also the sub-section titled **"OD and Wall Thickness via config_pcf.ini"** below.

If these attributes are not available in the PCF file, CAEPIPE 3D+ will then refer to the external Mapping DB, which is configured outside the PCF. This database contains the corresponding wall thickness information based on the Nominal Size data for each node. For further details, please refer to Appendix C of this manual.

User can modify the Wall thickness of the piping components through Edit Layout->Misc->Section.

OD and Nominal Size

Nominal size of a component is read from END-POINT attribute of PCF. The outside diameters of piping components are not typically included in the PCF file by default. However, if the Plant Design software is customized to export the outside diameter for each component node using attributes labelled OUTER-DIAMETER<n> (where n ranges from 1 to 2 depending on the number of nominal sizes the component has), the program will read these values and transfers them to CAEPIPE 3D+. See also the sub-section titled **"OD and Wall Thickness via config_pcf.ini"** below.

If these attributes are not available in the PCF file, CAEPIPE 3D+ will then refer to the external Mapping DB, which is configured outside the PCF. This database contains the corresponding outside diameter information based on the Nominal Size data for each node. For further details, please refer to Appendix C of this manual.

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

Boundary Conditions

The present version of the CAEPIPE 3D+ (for PCF) will "Anchor" all the free ends irrespective of whether they are connected to Equipment Nozzle or other Pipelines as the details of pipeline connections are not available in PCF file at this time.

User can add / modify supports in piping system through Edit Layout->Misc->Data Types

Supports

Support details including their locations will be transferred to CAEPIPE 3D+ as follows.

- Checks for attribute "SUPPORT" in PCF.
- If the attribute "SUPPORT" is defined in PCF, then reads the attribute "CO-ORDS" to locate the support in the specified coordinate.
- Reads the support type information from the attribute "SKEY" (if available) and generates the equivalent support type in CAEPIPE 3D+ by reading the Mapping DB (SupportType.mdb available with the product) when the option "Transfer Support using MDB" is selected from the dialog shown above. User can also customize and transfer support information along with stiffnesses, gap and friction by selecting the option "Transfer Support without using MDB". Refer **Appendix D** and **Appendix E** of this manual for details.
- If the attribute "SKEY" is not defined / available, then the support type will be transferred as "Hanger" with Hanger Type as specified in "Default Hanger to be used" option.

User can add / modify supports in piping system through Edit Layout->Misc->Data Types

Material

Material description entered for attribute “ITEM-CODE” will be read and the material property is then obtained from the intermediate mapping db. The material property thus obtained will be transferred to CAEPIPE 3D+. Refer Appendix B of this manual for details.

Below is information from a PCF file for an ELBOW. The material type and grade for this ELBOW are referenced via the “ITEM-CODE” attribute (highlighted in yellow).

In the ITEM-CODE section of the PCF file (highlighted in red), the “Material” attribute specifies the material type. In this example, the material for the ELBOW is “CS” (highlighted in yellow with red characters).

Following the procedure provided in Appendix B of this manual, enter "CS" as the Material Name in the Database along with its required mechanical properties. During PCF file transfer, the material properties for this ELBOW will be retrieved from the Database and written to CAEPIPE 3D+.

```
ELBOW
   END-POINT       7558.0000    549.5000    110.3486    4.0000  BW
   END-POINT       7564.0000    549.5000    104.3486    4.0000  BW
   CENTRE-POINT     7558.0000    549.5000    104.3486
   SKEY  ELBW
   ITEM-CODE  CS300-280
   ITEM-DESCRIPTION  ELL 90 LR, BW, ASME B16.9, ASTM A234 Gr WPB SMLS, Sch 40
   FABRICATION-ITEM
   ANGLE   9000
   PIPING-SPEC  CS300

ITEM-CODE  CS300-280
   DESCRIPTION  ELL 90 LR, BW, ASME B16.9, ASTM A234 Gr WPB SMLS, Sch 40
   Schedule  40
   PressureClass
   Material  CS
```

User can add / modify the Material properties in CAEPIPE 3D+ through Edit Layout->Misc->Materials.

Note:

Please ensure that you do not have more than one BLANK character inside the Material Description. For example, material description “Steel - ASTM A234/A234M Type E Grade WPB” is OK as BLANK characters between each word is one.

On the other hand, if you have a Material Description as “Forged Steel” in the PCF file (i.e., with more than one BLANK characters within the description) then while entering the Material Name inside the Mapping Db, please input as “Forged Steel” as the program will convert multiple BLANK characters into one BLANK character between each word.

Thermal Anchor Movement (TAM)

Thermal Anchor Movement (TAM) values entered at attribute “ATTRIBUTE1” under “END-CONNECTION-EQUIPMENT” of PCF file in the format “X Y Z” in global X, Y and Z directions respectively will be transferred to CAEPIPE 3D+ if available. These values should be defined in “mm” for SI units and in “Inch” for English units.

If TAM is not available in PCF file, then user can also add / modify the TAM 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 entered at attribute “ATTRIBUTE2” under “END-CONNECTION-EQUIPMENT” of PCF file in the format FX, FY, FZ, MX, MY and MZ will be transferred to CAEPIPE 3D+. These values should be separated using a BLANK space. 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 Allowable Loads not available in the PCF then user can add these values through Edit Layout > Misc > User Allowables.

The allowables thus defined/transferred 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.

OD and Wall Thickness via config_pcf.ini

If the outer diameter and wall thickness data are exported using attributes other than OUTER-DIAMETER<n> and WALL-THICKNESS<n> in the PCF, the user can define the names of these attributes in the config_pcf.ini file located in the installation folder.

For example, Reducer has two nominal sizes, accordingly the number of outer diameters and wall thicknesses will be 2 corresponding to each nominal size. Similarly, TEE has two nominal sizes (i.e., one for Run and the other for Branch). Accordingly, the number of outer diameters and wall thicknesses will be 2.

For instance, if the outer diameters (1 and 2), as well as the wall thicknesses (1 and 2), are exported through the attributes COMPONENT-ATTRIBUTE1, COMPONENT-ATTRIBUTE2, COMPONENT-ATTRIBUTE3, and COMPONENT-ATTRIBUTE4, then the user should update the config_pcf.ini file accordingly, as demonstrated below in **RED**. Do not modify strings given before the character “:” in each line of the config_pcf.ini file. The lines starting with * are comments in the ini file.

*For components having one Nominal size, Outer diameter 1 and Wall thickness 1 will be used
*and for components having two Nominal sizes, all the four attributes will be used
*i.e., Outer Diameter 1, Outer Diameter 2, Wall Thickness 1 and Wall Thickness 2.

OD1:COMPONENT-ATTRIBUTE1
OD2:COMPONENT-ATTRIBUTE2
WTK1:COMPONENT-ATTRIBUTE3
WTK2:COMPONENT-ATTRIBUTE4

The LAPJOINT-STUBEND can be transferred to CAEPIPE 3D+ based on specific instructions. By default, the LAPJOINT-STUBEND is transferred as a Flange to CAEPIPE 3D+. However, if the 3D Plant Design software exports the LAPJOINT-STUBEND as a combination of a STUB-END and a Flange, the user can include a specific line of code (as shown below) to ensure that the STUB-END is transferred as a Pipe. This will result in the user seeing a Pipe with a Flange at the end of the Pipe in CAEPIPE 3D+.

LAPJOINT-STUBEND:Pipe

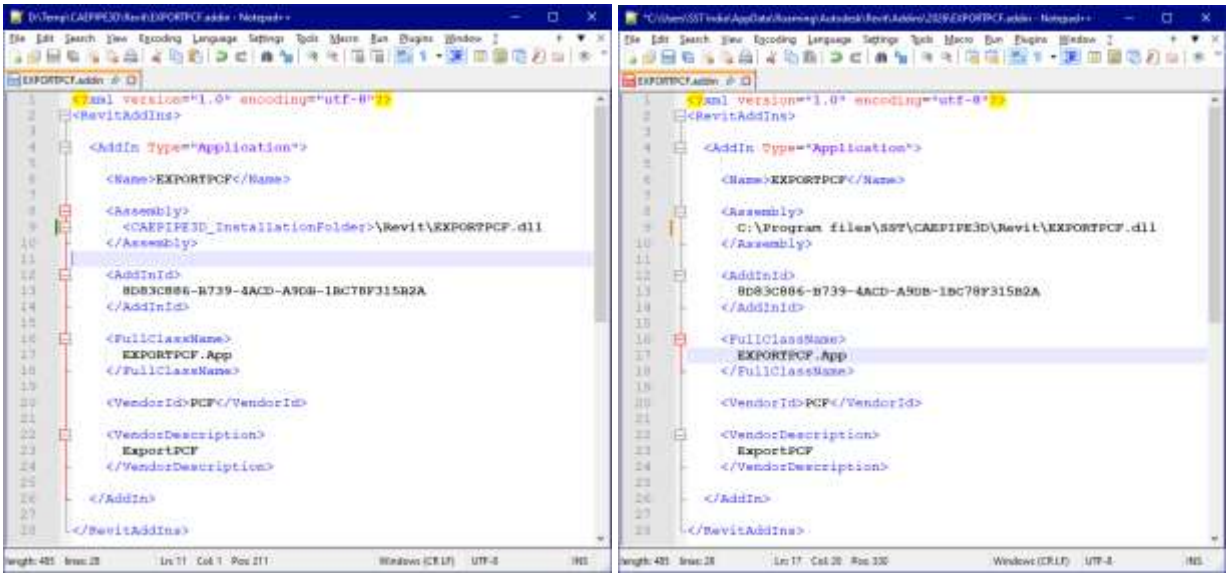
Export PCF from MEP Systems using CAEPIPE 3D+ Add-In from Revit

A CAEPIPE 3D+ Add-In for Revit is provided to enable the export of PCF files from Revit MEP (Plumbing) systems. To configure and use the Add-In, follow the steps below:

a. Navigate to the CAEPIPE 3D+ installation directory and open the Revit folder.

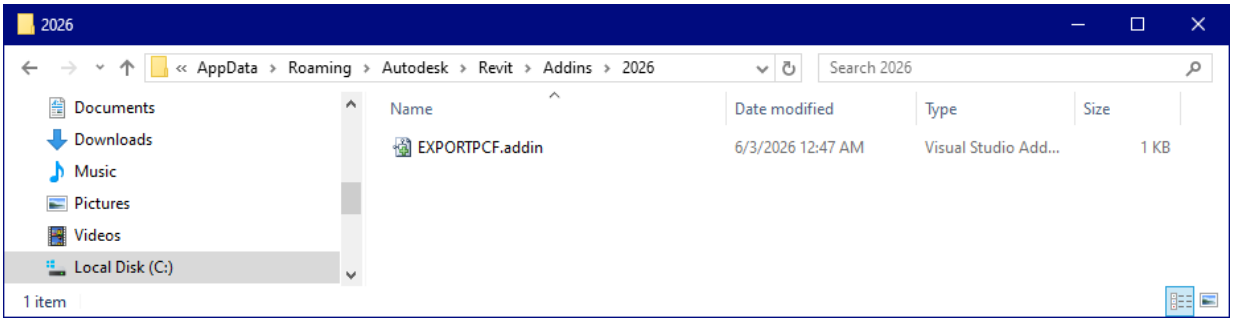
b. Open the file EXPORTPCF.addin using a text editor such as Notepad or WordPad.

c. Locate the entry containing <CAEPIPE3D_InstallationFolder> (Line 9 in the figure shown below) and replace it with the actual installation path of CAEPIPE 3D+. For example, if CAEPIPE 3D+ is installed in C:\Program Files\SST\CAEPIPE3D, replace <CAEPIPE3D_InstallationFolder> with: C:\Program Files\SST\CAEPIPE3D.

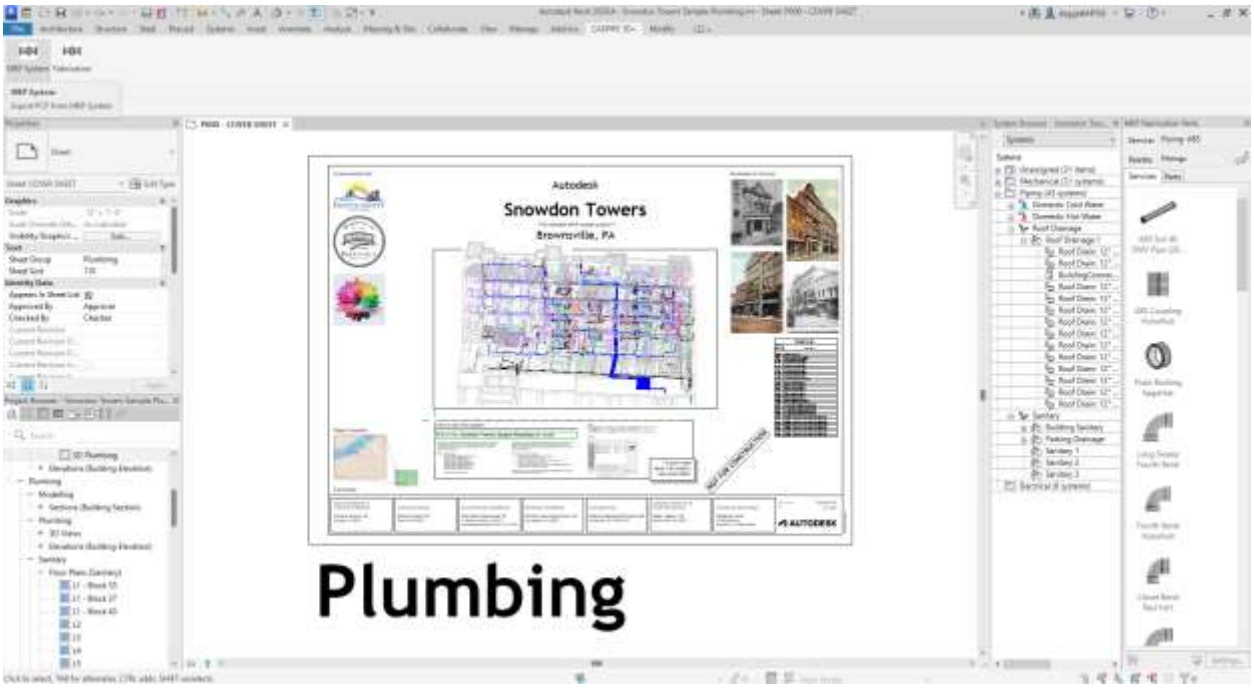


d. After completing the modifications described above, copy the EXPORTPCF.addin file to the Revit Addins directory. The Revit Addins directory is typically located within the %AppData% folder. To access it, enter %AppData% in the Windows File Explorer address bar and press Enter. From the AppData location, navigate to the following folder: Autodesk\Revit\Addins\<Version>. Version corresponds to the installed Revit version (for example, 2026).

In the example shown below, the Addin file is copied to the Revit 2026 Addins directory.



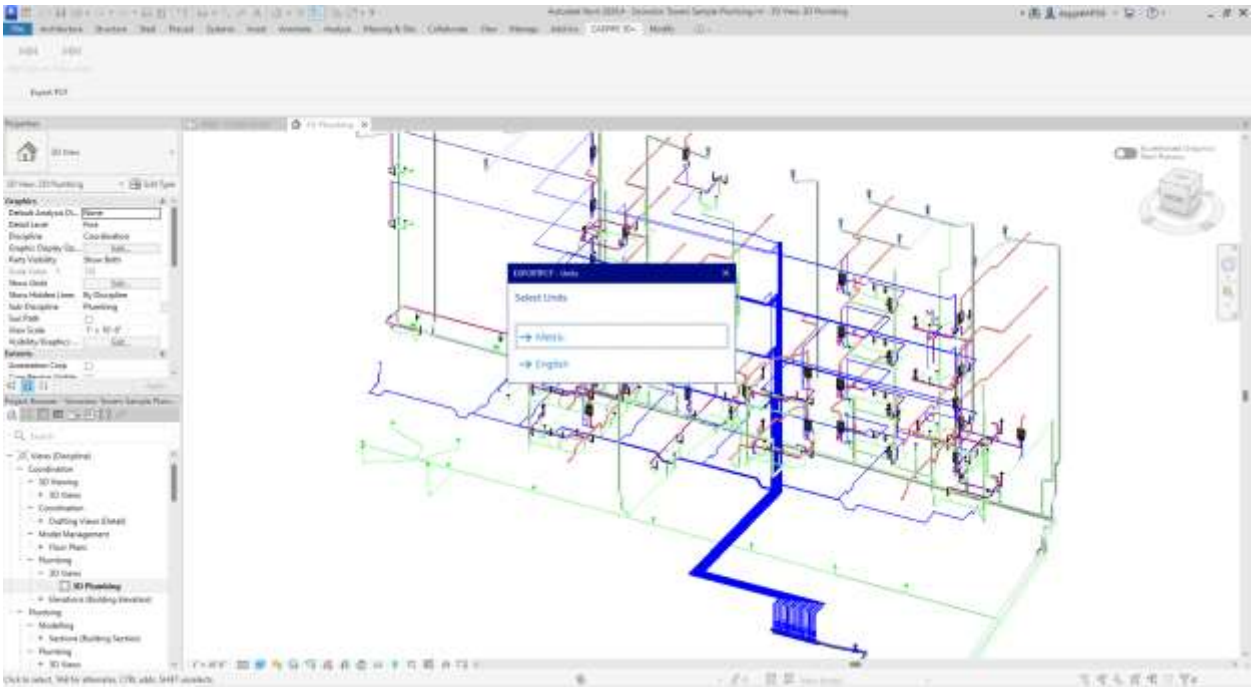
e. Launch Revit and open the desired project. Upon successful installation of the Add-Ins, a new tab named “CAEPIPE 3D+” will be available in the Revit ribbon. This tab contains the Export PCF command, as shown in the figure below.



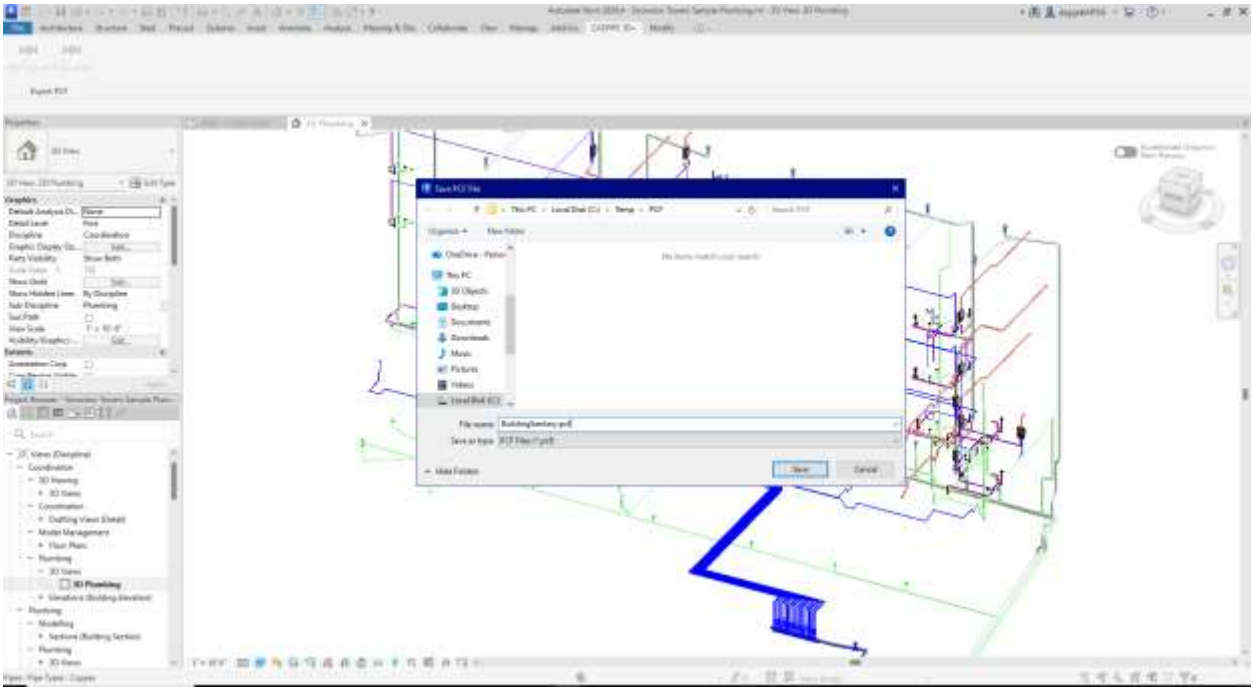
Exporting PCFs from MEP Systems

Click the “MEP System”. After the CAEPIPE 3D+ license is successfully validated, the Add-In prompts the user to provide the required export settings, including:

a. The units to be used for the PCF export.

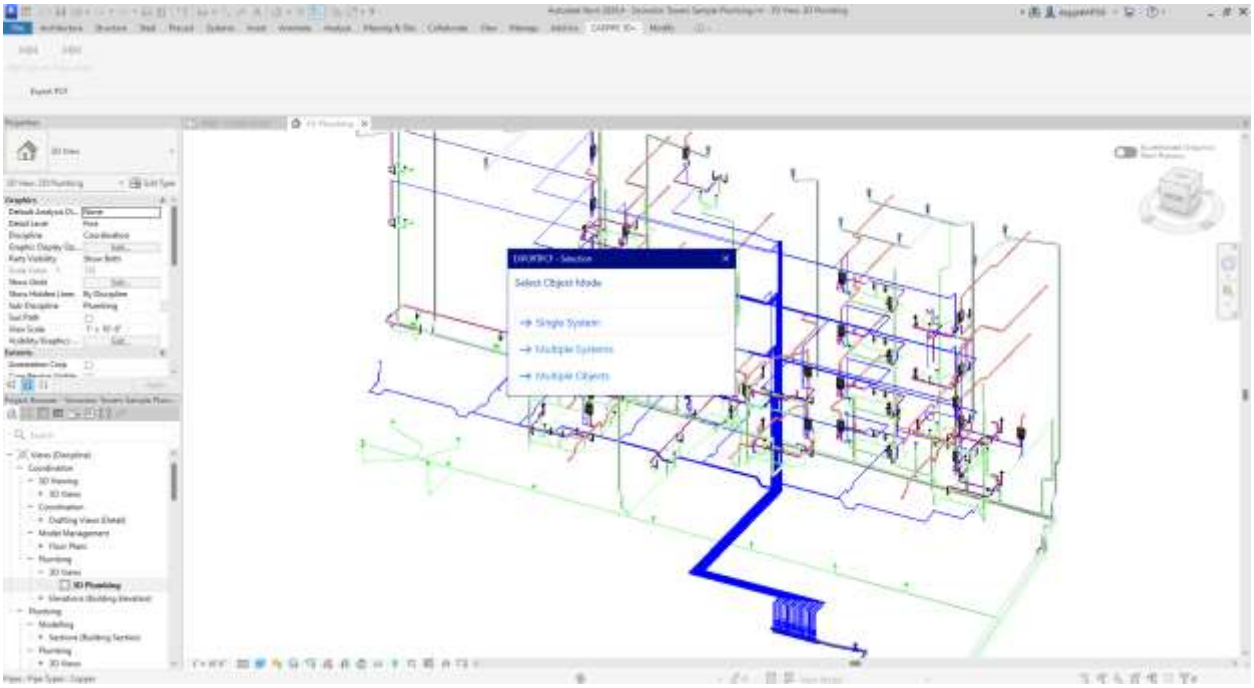


b. Specify the name of the PCF file and the folder where the exported file will be saved.

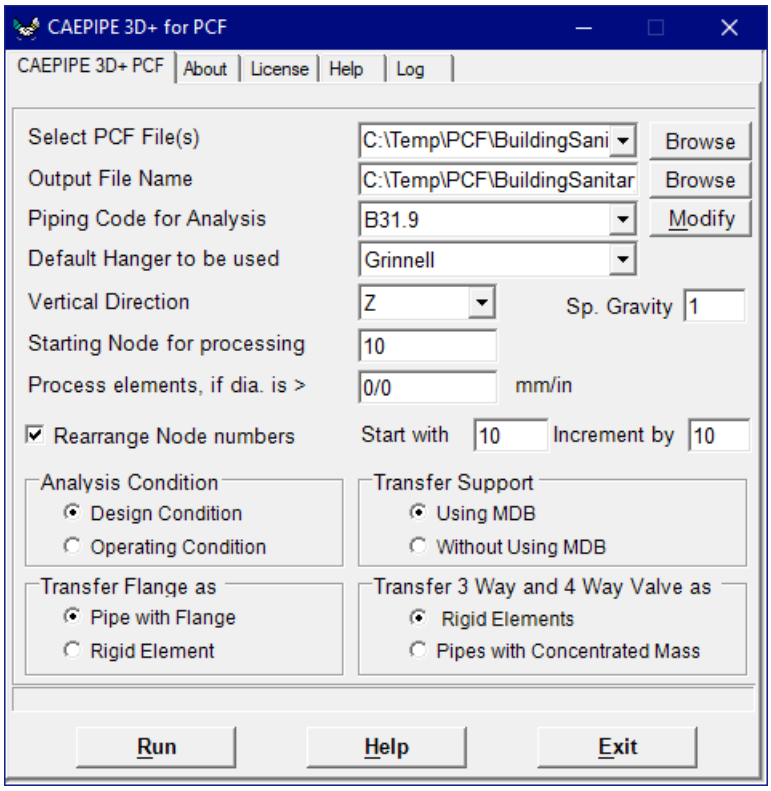


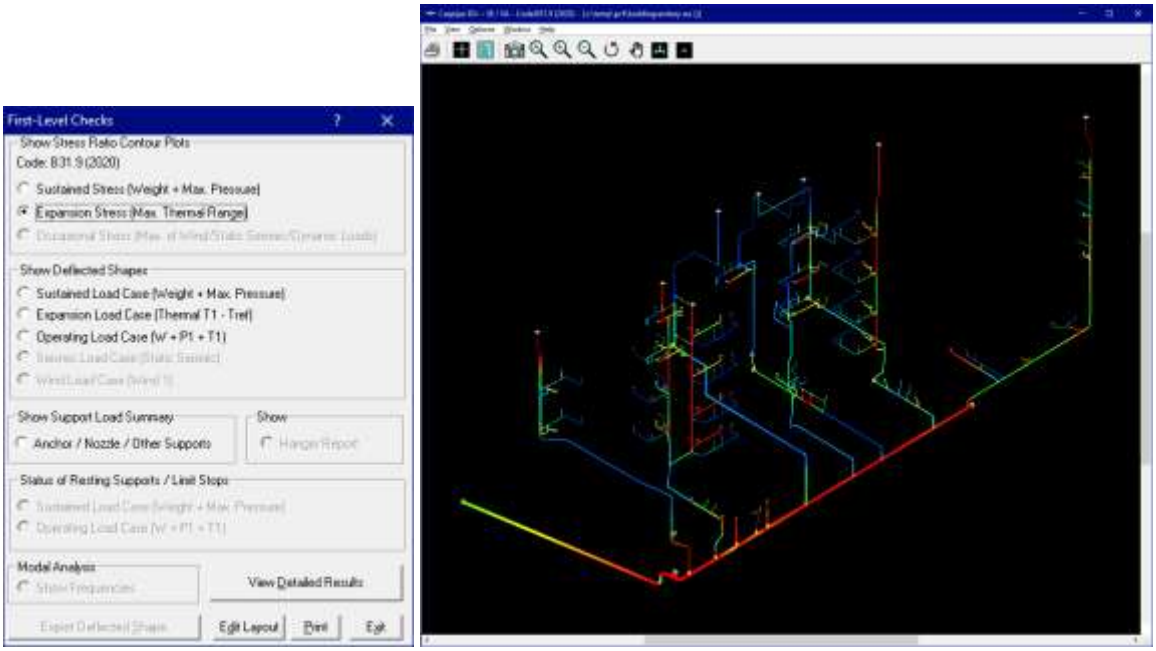
c. Object Selection Mode - Select one of the following options:

- Single System: Allows the user to select an object belonging to a piping system. All components associated with the selected system are exported to the PCF file.
- Multiple Systems: Allows the user to select objects from multiple piping systems. All components from All selected systems are exported to a single PCF file.
- Multiple Objects: Allows the user to select multiple objects regardless of their associated piping systems. Only the selected objects are exported to the PCF file.



d. Launch CAEPIPE 3D+ for PCF and select the PCF file generated to create the stress layout.





Reference (Revit Add-In)

Loads

The piping system temperature is obtained from the Temperature attribute assigned to the Revit Piping System. If this attribute is not defined, a default temperature of 21°C for SI units or 70°F for English units is transferred to CAEPIPE 3D+.

Similarly, the piping system pressure is obtained from the Pressure attribute assigned to the Revit Piping System. If the attribute is not available, a default pressure value of 0 bar or 0 psi is transferred.

After importing the model into CAEPIPE 3D+, users can review and modify the temperature and pressure values through Edit Layout > Misc > Loads

Note:

If temperature and pressure data are stored in Revit attributes other than Temperature and Pressure, the corresponding attribute names can be configured in the config_revit.ini file located in the CAEPIPE 3D+\Revit folder.

For example, if the temperature is defined using the attribute Fluid Temperature, replace the entry Temperature with Fluid Temperature in the config_revit.ini file.

Weight

The weights of components such as valves, instruments, flanges, and other fittings are obtained from the WEIGHT attribute defined for the corresponding fitting in Revit. If the attribute is not available, a default weight value of 0.0 is transferred to CAEPIPE 3D+.

After importing the model, users can review and modify component weights in CAEPIPE 3D+ through:

Edit Layout > Window > List

Note:

If weight information is stored in a Revit attribute other than WEIGHT, the corresponding attribute name can be configured in the config_revit.ini file located in the CAEPIPE 3D+\Revit folder.

For example, if the weight is defined using the attribute Empty Weight, replace the entry Weight with Empty Weight in the config_revit.ini file.

Nominal Size, Outside Diameter and Wall Thickness

For pipe elements, the Nominal Size, Outside Diameter (OD), and Wall Thickness values are obtained from the Revit parameters RBS_PIPE_DIAMETER_PARAM, RBS_PIPE_OUTER_DIAMETER, and RBS_PIPE_WALL_THICKNESS, respectively.

For fittings such as elbows, reducers, valves, and tees, these properties are derived from the connected pipe identified through the fitting connectors. The extracted values are then written to the PCF file. Outside diameter and wall thickness information are exported using the attributes OUTSIDE-DIAMETER1, OUTSIDE-DIAMETER2, WALL-THICKNESS1, and WALL-THICKNESS2, as applicable to each component.

If the required diameter and wall thickness data are not available in the PCF file, CAEPIPE 3D+ retrieves the information from the external Mapping Database configured separately from the PCF file. This database contains the corresponding outside diameter and wall thickness values for each nominal pipe size. For additional information, refer to Appendix C of this manual.

After importing the model, users can review and modify the nominal size and outside diameter of piping components through: Edit Layout > Misc > Section

Boundary Conditions and Supports

Due to limitations, the Add-In is unable to automatically identify and export connecting equipment and pipe supports from the Revit model.

Material

For pipe elements, the material name is obtained from the Revit parameter RBS_PIPE_MATERIAL_PARAM. For fittings, the material information is read from the Material attribute. The extracted material name is then written to the PCF file using the ITEM-CODE attribute.

During the import process, CAEPIPE 3D+ retrieves the corresponding material properties from the intermediate Mapping Database. The mapped material properties are then assigned to the piping components in CAEPIPE 3D+.

For additional information, refer to the Section Material (given above) above and Appendix B of this manual.

Note:

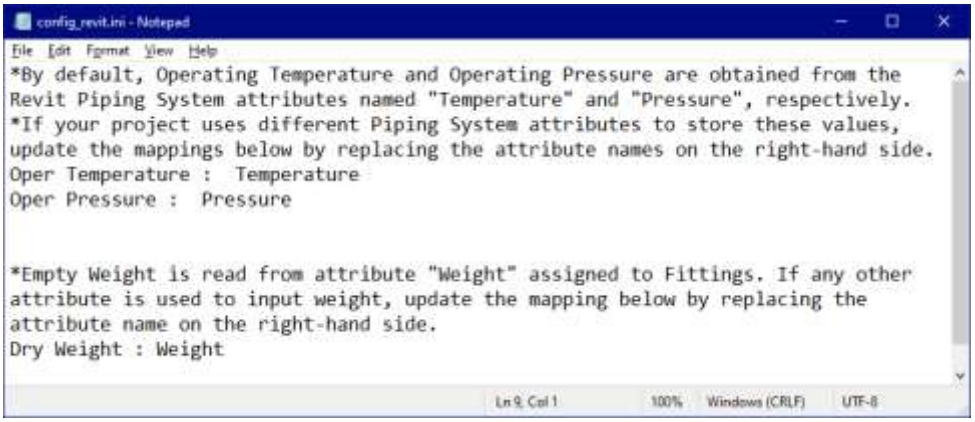
Please ensure that you do not have more than one BLANK character inside the Material Description. For example, material description “Steel - ASTM A234/A234M Type E Grade WPB” is OK as BLANK characters between each word is one.

On the other hand, if you have a Material Description as “Forged Steel” in the PCF file (i.e., with more than one BLANK characters within the description) then while entering the Material Name inside the Mapping Db, please input as “Forged Steel” as the program will convert multiple BLANK characters into one BLANK character between each word.

Temperature, Pressure and Weight via config_revit.ini

If temperature, pressure or weight information is stored in Revit attributes other than the default Temperature, Pressure, and Weight attributes, the corresponding attribute names can be configured in the config_revit.ini file located in the CAEPIPE 3D+\Revit installation folder.

Update the entries in the configuration file to match the attribute names used in your Revit project. The Add-In will then use the specified attributes when exporting data to the PCF file.



Identification of components from REVIT Plumbing Module

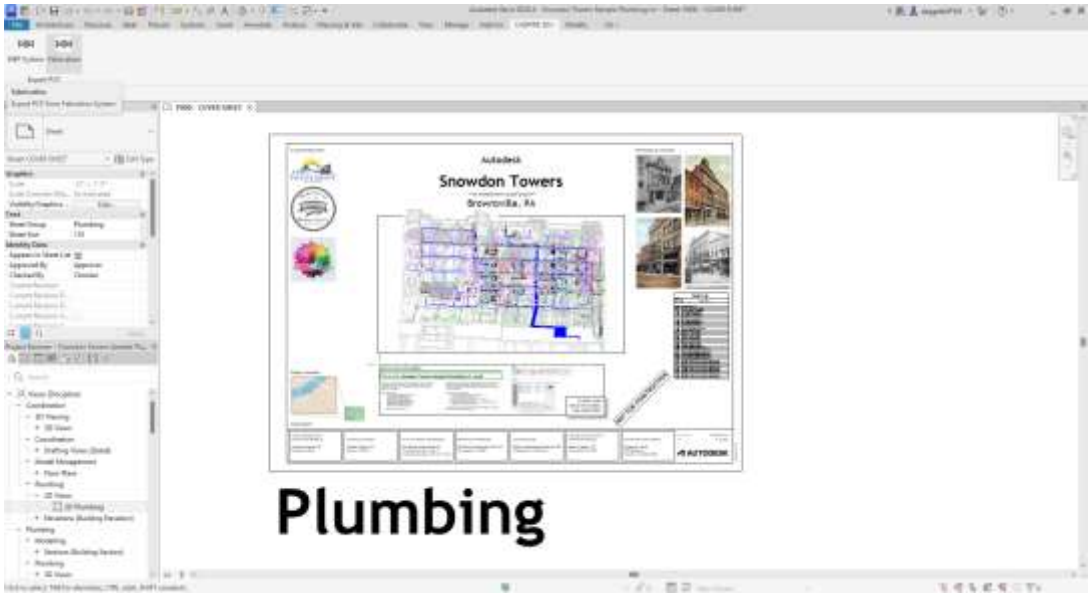
During PCF file generation, fittings are identified from the Revit Plumbing module by evaluating the parameters **Symbol Family Name**, **Symbol Name**, **Element Name**, and **Description**. If any of these parameters contain one of the keywords listed below, the corresponding component is classified as specified.

Sl. No.	No. of Connections	Keyword	Component in PCF	Remarks
1	1	CAP or PLUG	CAP	
2	1	Support	SUPPORT	
3	2	Radius, Elbow or Bend	ELBOW	
4	2	Eccentric	ECCENTRIC-REDUCER	
5	2	Reducer	REDUCER	
6	2	Flange	FLANGE	
7	2	Valve	VALVE	
8	2	Coupling	COUPLING	
9	2	Filter	FILTER	
10	2	Instrument	INSTRUMENT	
11	3	Valve	VALVE-3WAY	
12	3	Tee	TEE	
13	3	Instrument	INSTRUMENT-3WAY	
14	4	Cross	CROSS	
15	4	Valve	VALVE-4WAY	
16	4	Instrument	INSTRUMENT-4WAY	

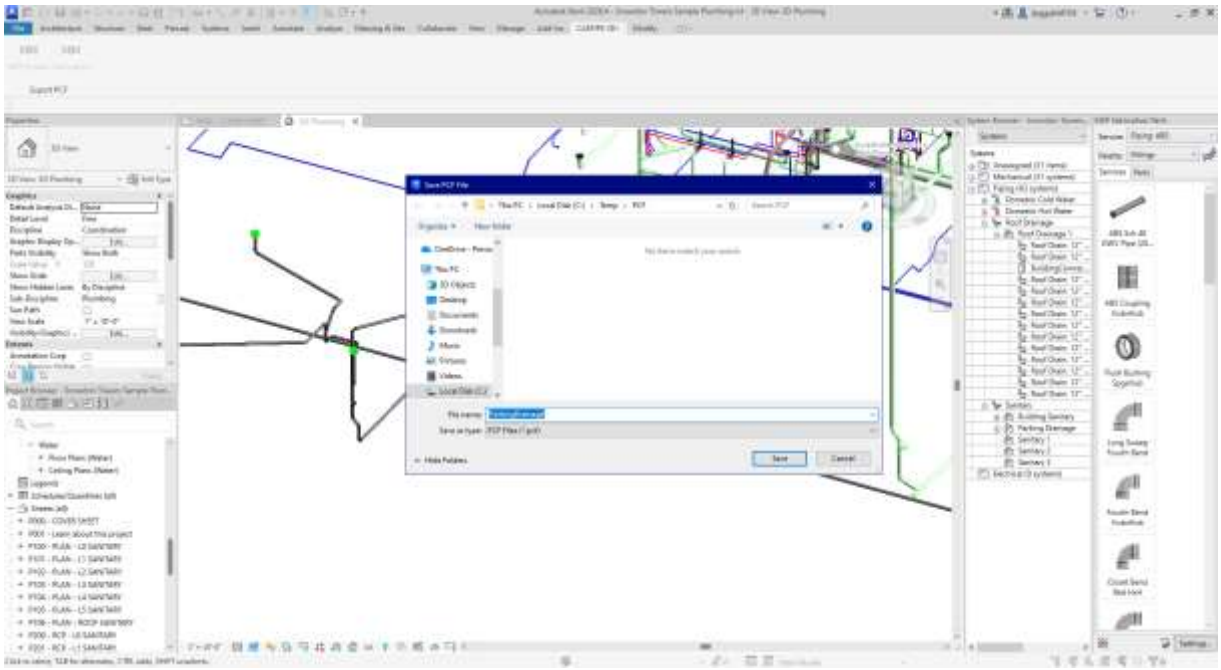
Export PCF from Fabrication Module of Revit

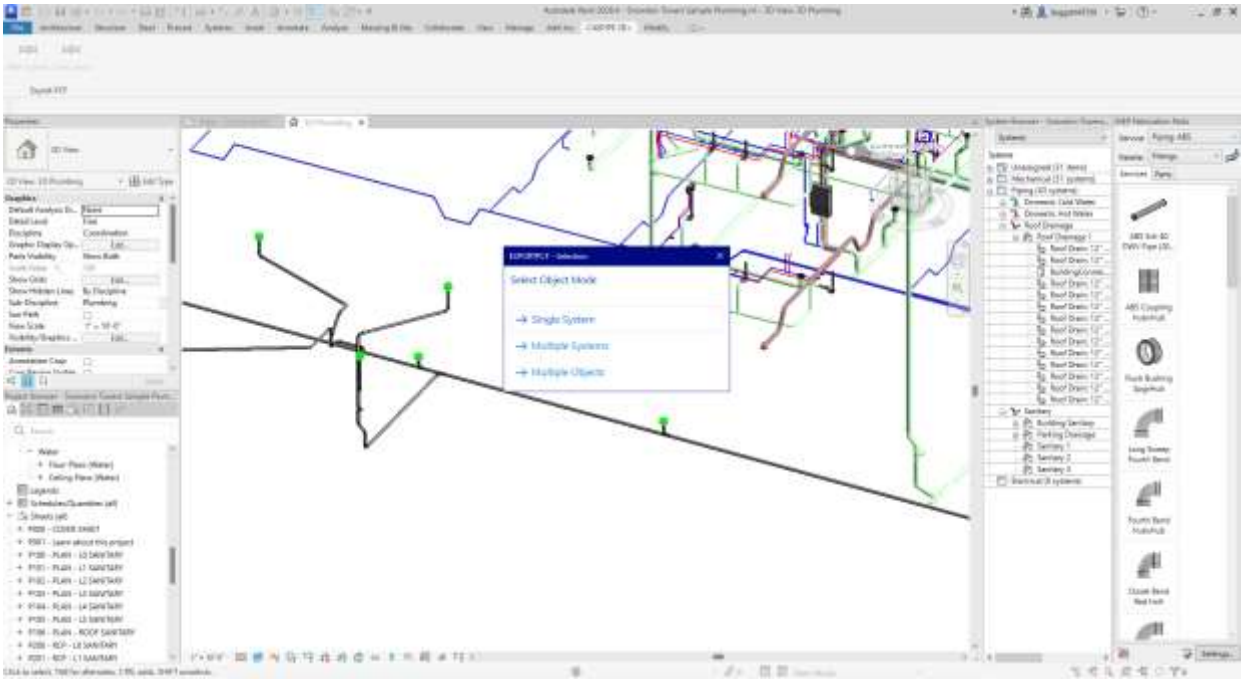
To export a Piping Component File (PCF) from Revit Fabrication parts, the Add-In provides a dedicated Fabrication button.

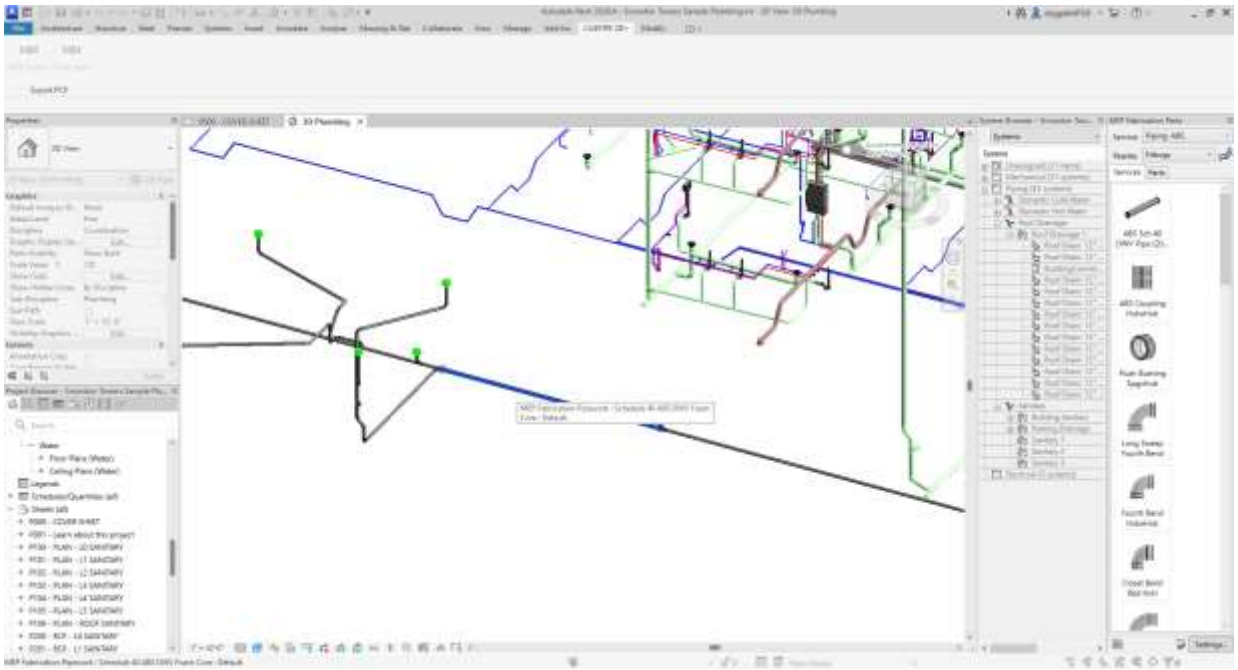
1. Click the Fabrication button, as shown in the figure below. Select the Fabrication parts to be included in the export. The Add-In uses Revit's built-in EXPORTPCF functionality to generate a PCF file containing the selected components. Import the generated PCF file into CAEPIPE 3D+ for further processing and stress analysis. For information on importing PCF files into CAEPIPE 3D+, refer to the Section titled “Reference” given under CAEPIPE 3D+ for PCF above.



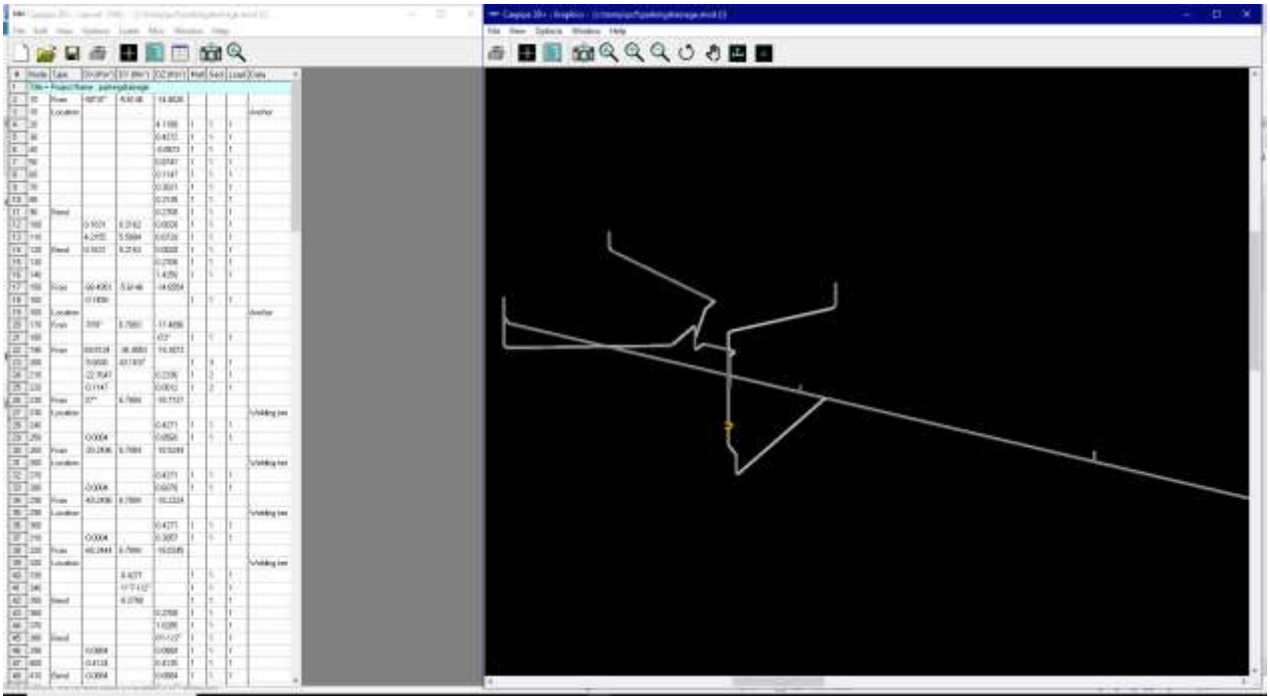
2. Repeat Steps “b” through “d” given in Section titled “Exporting PCFs from MEP Systems” above.




Select Object Mode
Single System
Multiple Systems
Multiple Objects



4. Once the PCF file is exported, then use CAEPIPE 3D+ for PCF to import the layout into CAEPIPE.

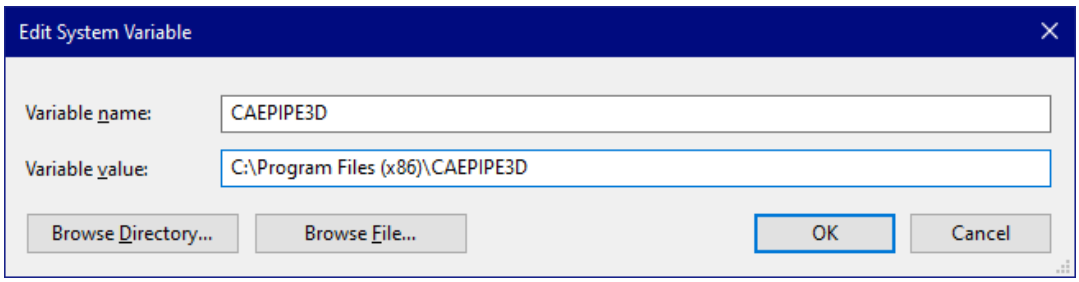


CAEPIPE 3D+ for PDS

Installing and Configuring Program

1 Run the program "Setup.exe" downloaded and follow the instructions as they appear on the screen.

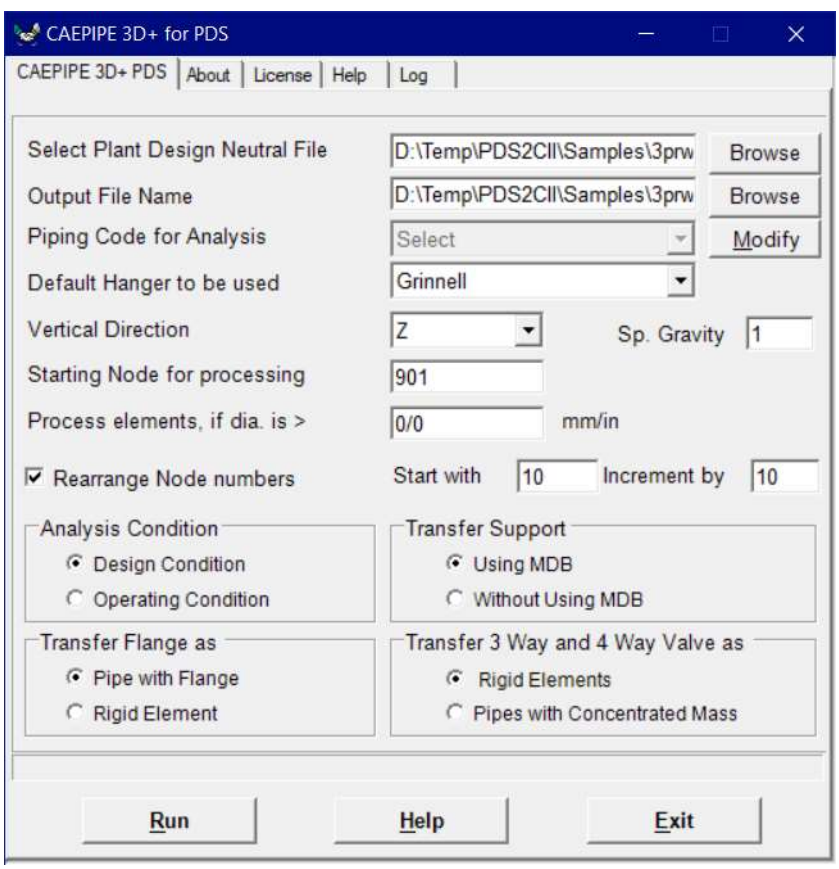
2 Define an environmental variable with name as “CAEPIPE3D” and its value as Installation path. For example, if the product is installed in “C:\Program Files (x86)\CAEPIPE3D” then input the value as “C:\Program Files (x86)\CAEPIPE3D” in the Edit System Variable dialog as shown below.



How to use CAEPIPE 3D+?

1 Select the PDS Neutral file.

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 Enter the Specific Gravity of fluid with respect to water. By default the specific gravity is set to 1.0.

6 Select Default Hanger to be used in CAEPIPE 3D+. This selected Hanger type will be used in CAEPIPE 3D+, only when the hanger type in CAEPIPE 3D+ corresponding to Plant Design hanger is not specified /available in the Mapping DB. For details on transferring support from Plant Design to CAEPIPE 3D+, refer **Appendix D** of this manual.

7 Specify the Starting Node number for processing network. This is set to 10 by default. If the user is not sure about the node number, then this can be left as 10 or enter as 0.

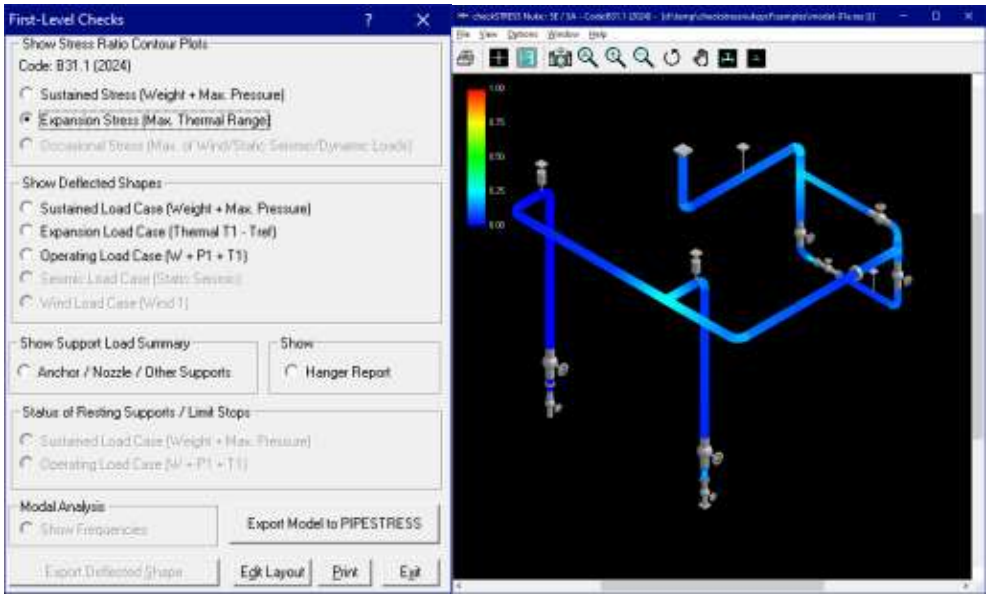
8 If the user wishes to eliminate those elements whose Nominal size (NS) is less than 50mm / 2in, then they can specify the value as “50/2” in the field “Process elements, if dia >”. In other words, the program will process only those elements whose nominal size is greater than the value specified in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

9 Specify how the Flange to be transferred to CAEPIPE 3D+.

10 Specify how the 3 Way and 4 Way Valves are to be transferred to CAEPIPE 3D+.

11 Choose the Global Vertical Axis for CAEPIPE 3D+ 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. Figure below shows the stress contour plot for the Expansion case.



Reference

Procedure for extracting neutral files from PDS software

This program is compatible with PDS versions released after May 1998 and up to PDS Version 8.xx.

1. CAEPIPE3D is supplied with the PDSTRESS.TBL and DEFAULTS.DAT. These files are available in the folder CAEPIPE 3D Installation folder\Setup.

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
! 1234567890123456789012345678901234567890
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 product support, so that we could incorporate such details in the next release of the product.

6. If the support detail is not specified at support location then it will be transferred as "Hanger" to CAEPIPE3D.

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.

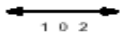
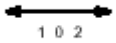
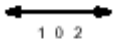
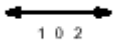
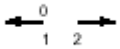
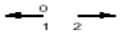
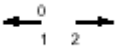
8. Thermal Anchor Movement (TAM) values extracted and written to stress neutral file through "LNOD" or "MLNOD" with type value as "MV" in global X, Y and Z directions can be transferred to CAEPIPE. TAM values should be defined in "mm" for PDS projects in SI units and in "Inch" for PDS project in English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

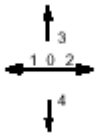
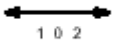
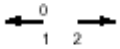
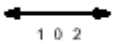
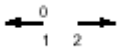
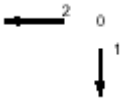
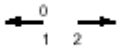
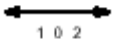
9. Equipment Nozzle Allowable Loads (forces and moments) provided by the equipment manufacturer or calculated using Applicable codes / Finite Element Methods should be entered in PDS at Equipment. They are then to be extracted and written through "LNOD" and "MLNOD" with type as "AL" in global X, Y and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

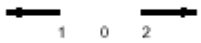
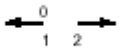
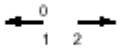
Appendix A

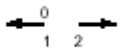
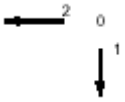
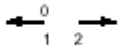
PDMS/E3D to CAEPIPE 3D+ Component Mapping

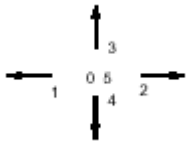
The types of components available in PDMS/E3D are simulated as tabulated below in CAEPIPE 3D+.

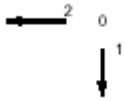
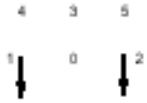
PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
Cap			
Butt weld		Rigid Element	RB
Cap compression		Rigid Element	RB
Cap Screwed		Rigid Element	RB
Cap Socket Weld		Rigid Element	RB
Closure			
Pipe Block		Rigid Element	RB
Pipe Block Variable Length		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

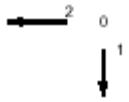
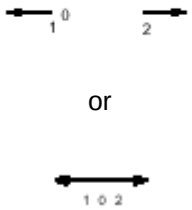
PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
Set on, Set on Reinforced		Four Pipes with Branch SIF (Welding TEE)	CR
Blind Flange			
Flange – Blind		Rigid Element or Pipe with Flange	FL
Flange - Reducing Concentric		Rigid Element or Pipe with Flange	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, Restrictor Plate, Tundish		Pipe	PI
Plug		Pipe	PI

PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
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
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

PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
			
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
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

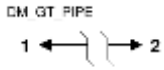
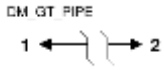
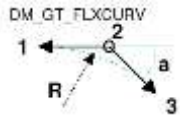
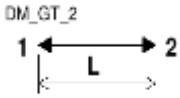
PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
Valve – Check		Valve	VA
VENT			
Rupture Disk		Rigid Element	RB
Four Way Valve			
Valve – 4 Way		Four Rigid Elements or Four Pipes with Concentrated Mass	4W
Three Way Valve			
Valve – 3 Way		Three Rigid 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

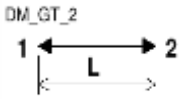
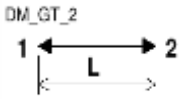
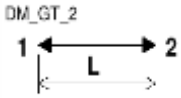
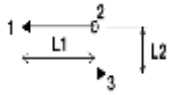
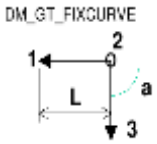
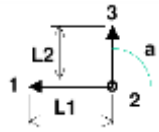
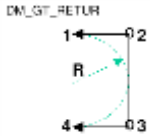
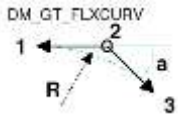
PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
Control, Control Square Indicator, Control Motorised Indicator, Hand Indicator, Pressure Reducing and Relief/Vent		Rigid Element	RB
3 Way Control, 3 Way Control Square, 3 Way Control Motorised Indicator and 3 Way Control Hand Indicator		Three Rigid Elements or Three Pipes with Concentrated Mass	3W
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

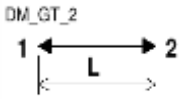
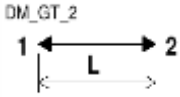
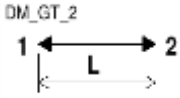
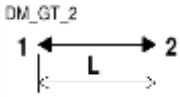
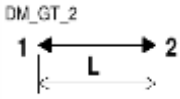
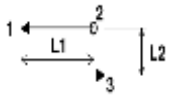
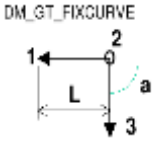
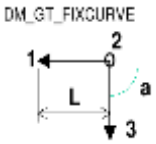
PDMS/E3D Component Description	Connection Points	Component in CAEPIPE 3D+	Keyword used in Neutral File
Filter / Strainer Angle		Rigid Element	RB
Flange			
Flange Blind, Flared/Loose Backing, Flange Backing, Flange Reducing Concentric, Screwed, Slip On, Slip On with 'J' Type Weld, Socket Weld and Weld Neck		Rigid Element or Pipe with Flange	FL
Orifice Slip On Flange and Orifice Weld Neck		Rigid Element with Tapping	OF

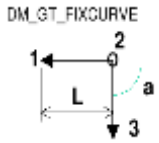
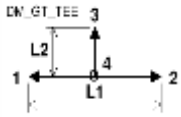
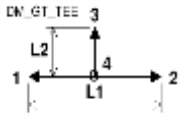
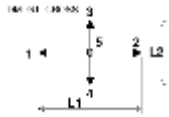
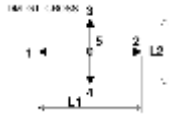
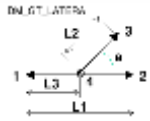
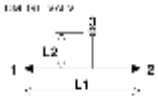
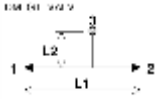
CADMATIC to CAEPIPE 3D+ Component Mapping

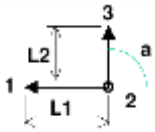
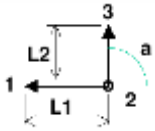
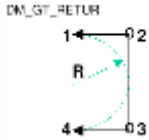
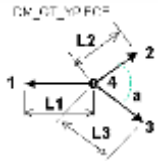
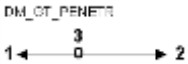
The types of components available in Cadmatic are simulated as tabulated below in CAEPIPE 3D+.

Cadmatic Component Description	Constraints	Geometric Types	CAEPIPE 3D+ Component	Key Word in Neutral File
If modeled as Piping Components				
Straight Pipes	DM_GT_PIPE = TRUE & Bend Angle = 0		Pipe	PI
Free Bends	DM_GT_PIPE = TRUE & Bend Angle > 0		Bend	EL
Flexible Curves	DM_GT_FLXCURV = TRUE		Bend	EL
Flanges	DM_GT_2P = True & Primary Connection Code = 1 or 3 (Connection Type is Flange)		Rigid Element or Pipe with Flange	FL

Cadmatic Component Description	Constraints	Geometric Types	CAEPIPE 3D+ Component	Key Word in Neutral File
Conc. Reducer	DM_GT_2P = True & 2nd Nominal Size > 0		Reducer Concentric	RD
Straight Pipes	DM_GT_2P = True & Keyword in corporate catalogue of CADMATIC has "*PIPE*" e.g. Seamless Straight Pipe		Pipe	PI
Flow meters, Adaptor, Flexible hoses, Caps	DM_GT_2P = True		Rigid Element	RB
Eccentric Reducer	DM_GT_3PDIRFIX = True		Reducer Eccentric	ER
Fixed Angle Curves	DM_GT_FIXCURVE = True		Bend	EL
Asymmetric Curves	DM_GT_ASYMCURVE = True		Bend	EL
U-Piece	DM_GT_RETURN = True		Bend	EL
Sliding Sockets, Unions	DM_GT_PENETR = True		Rigid Element	RB
If modeled as Standard Components				
Filter / Strainer	DM_GT_FLXCURVE = TRUE		Rigid Element	RB

Cadmatic Component Description	Constraints	Geometric Types	CAEPIPE 3D+ Component	Key Word in Neutral File
Some Flanges	DM_GT_2P = True & Primary Connection Code = 1 or 3 (Connection Type is Flange)		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		Pipe	PI
Valves	DM_GT_2P = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Ball check valve PVC		Valve	VA
Conc. Reducer	DM_GT_2P = True & 2nd Nominal Size > 0		Reducer Concentric	RD
Flow meters, Adaptor, Flexible hoses	DM_GT_2P = True		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		Valve	VA
Straight Pipes	DM_GT_FIXCURVE = True & Keyword in corporate catalogue of CADMATIC has "*PIPE*" eg Seamless Straight Pipe		Pipe	PI

Cadmatic Component Description	Constraints	Geometric Types	CAEPIPE 3D+ Component	Key Word in Neutral File
Filter / Strainer	DM_GT_FIXCURVE = True		Rigid Element	RB
3 Way Valves	DM_GT_TEE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Three-way valve Wafer		Three Rigid Elements or Three Pipes with Concentrated Mass	3W
Welding TEE	DM_GT_TEE = True		Pipes with Branch SIF (Welding TEE)	TW
4 Way Valves	DM_GT_CROSS = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Four-way valve Wafer		Four Rigid Elements or Four Pipes with Concentrated Mass	4W
Crosses	DM_GT_CROSS = True		Four Pipes with Branch SIF (Welding TEE)	CR
Lateral TEE	DM_GT_LATERAL = True		Pipes with Branch SIF (Welding TEE)	TW
Valves	DM_GT_VALVE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Ball Valve		Valve	VA
Reduced Valves	DM_GT_VALVE = True & 2nd Nominal Size > 0 & Key word in corporate catalogue should not have "*valve*".		Reducer Concentric	RD

Cadmatic Component Description	Constraints	Geometric Types	CAEPIPE 3D+ Component	Key Word in Neutral File
Flow meters, Pressure gauges, Manometers, Strainers	DM_GT_VALVE = True		Rigid Element	RB
Asymmetric Valves	DM_GT_ASYMCURVE = True & Keyword in corporate catalogue of CADMATIC has "*VALVE*" eg Pressured rig-valve WDP		Valve	VA
Asymmetric Curves	DM_GT_ASYMCURVE = True		Bend	EL
U-Piece	DM_GT_RETURN = True		Bend	EL
Y-piece	DM_GT_YPIECE = True		Pipes with Branch SIF (Welding TEE)	TW
Sliding Sockets, Unions	DM_GT_PENETR = True		Rigid Element	RB

PCF to CAEPIPE 3D+ Component Mapping

The types of components available in PCF are simulated as tabulated below in CAEPIPE 3D+.

Component identifier in PCF	Component in CAEPIPE 3D+
PIPE	Pipe
GASKET	Pipe
UNION	Pipe
CONNECTOR	Rigid Element
CAP	Rigid Element
FILTER	Rigid Element
INSTRUMENT	Rigid Element
INSTRUMENT with SKEY BLEW**	Bellow**
INSTRUMENT-ANGLE	Rigid Element
MISC-COMPONENT	Rigid Element
MISC-COMPONENT-ANGLE	Rigid Element
MISC-HYGENIC	Rigid Element
SAFETY-DISC	Rigid Element
TRAP	Rigid Element
TRAP-ANGLE	Rigid Element
VALVE-ANGLE	Rigid Element
REDUCER-CONCENTRIC	Reducer
COUPLING	Reducer
REDUCER-ECCENTRIC	Reducer
FILTER-OFFSET	Reducer
INSTRUMENT-OFFSET	Reducer
MISC-COMPONENT-OFFSET	Reducer
TRAP-OFFSET	Reducer
CROSS	Four Rigid Elements or Four Pipes with Concentrated Mass
VALVE-3WAY	Three Rigid Elements or Three Pipes with Concentrated Mass
INSTRUMENT-3WAY	Three Rigid Elements or Three Pipes with Concentrated Mass
VALVE-4WAY	Four Rigid Elements or Four Pipes with Concentrated Mass
INSTRUMENT-4WAY	Four Rigid Elements or Four Pipes with Concentrated Mass

Component identifier in PCF	Component in CAEPIPE 3D+
SUPPORT	HANGER (by default)
FLANGE	Flange
LAPJOINT-RING	Flange
LAPJOINT-STUBEND	Flange
TEE	Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)
ELBO-TEED	Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)
INSTRUMENT-TEE	Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)
REDUCER-CONCENTRIC-TEED	Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)
Y-PIECE-FITTING	Pipe(s) with Branch SIF (Welding TEE) or (Weldolet)
ELBOW	Bend
FILTER-ANGLE	Bend
OLET**	Pipe with Branch SIF (Weldolet)
CLAMP with SKEY CLVT**	Ball Joint***

** Available and valid for PCF-to-CAEPIPE, CATIA to CAEPIPE, AutoPlant to CAEPIPE Versions 9.30 or later, checkSTRESS PCF, checkSTRESS II PCF, checkSTRESS Nuke PCF Version 9.40 or later, CAEPIPE 3D+ for PCF Version 10.50 or later.

*** Available and valid for PCF-to-CAEPIPE, CATIA to CAEPIPE, AutoPlant to CAEPIPE Versions 13.00 or later, checkSTRESS PCF, checkSTRESS II PCF, checkSTRESS Nuke PCF Version 13.00 or later, CAEPIPE 3D+ for PCF Version 12.20.1 or later.

Stiffness of the Ball Joint can be modified by editing the field “KPSUPPORT” corresponding to the field “PdSupport” with value “CLVT” in the SupportType.mdb file for both ZVertical and YVertical tables.

By default, Victaulic Coupling is transferred as a “Flexible” Coupling as given here “**RK(Rigid,1), ROTL(None,1)”**. If it is to be transferred as a “Rigid” Coupling then modify the value as “**RK(Rigid,Rigid), ROTL(None,None)”.** For further details on RK and ROTL, see the section titled “MBF Import” in CAEPIPE User’s Manual.

Appendix B

Material

Material name for each element is read from the 3D Plant Design database and is written to the neutral file. The program then gets the CAEPIPE 3D+ material information corresponding to 3D Plant Design material information as follows.

- CAEPIPE 3D+ reads the Material Mapping DB file name from the table “code” available in Codedb.mdb corresponding to the Analysis Code specified in the form during transfer.

  If no Analysis Code is selected, the program automatically retrieves the code associated with “Default” from CodeDb.mdb. In this case, the Analysis Code is assigned as “B319”, and the materials are mapped from B311.mdb, as these settings are configured against the “Default” entry in CodeDb.mdb.

- Gets the CAEPIPE 3D+ Material properties from the other tables available in the Material Mapping DB corresponding to 3D Plant Design material description.

In case of unavailability of CAEPIPE 3D+ material property corresponding to 3D Plant Design material description available in the transfer file, then program reads the CAEPIPE 3D+ Material property specified in the first row of the Material Mapping DB and transfers the same to CAEPIPE 3D+.

Note:

The material data provided in the Access database are sample values intended only to demonstrate the Material Mapping process from Plant Design to CAEPIPE 3D+. The Access database should be updated with the actual material properties required as per the project in the corresponding material files. Alternatively, after transferring the layout to CAEPIPE 3D+, user can replace the material properties with the correct values supplied in the CAEPIPE 3D+ Material Library (*.mat files located in the CAEPIPE 3D+ Installation Directory > Material_Library) using Misc > Material Library.

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 CAEPIPE 3D+ corresponding to Analysis Code defined in 3D Plant Design Software. Third field “KpMat” defines the name of the Material DB file to be used (to get the CAEPIPE 3D+ Material property) during transfer. Fourth field “KpSect” defines the name of the Schedule Table to be used (to extract OD and Thickness if not available in the Neutral file for an element) during transfer. 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/MdetaiS" 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 available 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 thus created above 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 CAEPIPE 3D+ Section details table name in the field "KpSect". 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 details are specified.

KpSect - Section DB name from where the Schedule details are specified.

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/in3 or kg/m3)
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=CAEPIPE 3D+

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=CAEPIPE 3D+

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 as the 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 schedule to be used during transfer. In case of unavailability of OD and/or thickness values in transfer file, translator reads the OD and/or thickness from the standard schedule table and transfers the same to CAEPIPE 3D+. In case the value of the field “KpSect” is not defined or left empty in the table, then the program will use the “Standard Schedule (STD)” for ANSI standard by default.

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 the 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 CAEPIPE 3D+ 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 CAEPIPE 3D+. 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 filed “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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+. 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ by specifying the following at support locations.

Syntax:

USERHANGER(Number of Hangers:Spring Rate[ib/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 CAEPIPE 3D+ by specifying the following at support locations.

Syntax:

GUI(Stiffness[ib/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 CAEPIPE 3D+ by specifying vector details with the following at support locations.

Syntax:

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

Type can 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 CAEPIPE 3D+ by specifying the following at support locations. For definition on Local Coordinates, refer to Section titled “Local Coordinates” in CAEPIPE 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 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ by specifying the following at support locations. For definition on Local Coordinates, refer to Section titled “Local Coordinates” in CAEPIPE 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 CAEPIPE 3D+.

Single acting Translational Restraints

Single acting Translational Restraints are transferred as Limit Stop to CAEPIPE 3D+. 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ 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 CAEPIPE 3D+ by specifying the following at support locations. For definition on Local Coordinates, refer to Section titled “Local Coordinates” in CAEPIPE 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 CAEPIPE 3D+ 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
Anchor		
Anchor	ANC(Stiffness:Gap)	ANC or ANC(1E12:0.0) or ANC(1E12)
Double Acting Translational Restraints		
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)
Double Acting Rotational Restraints		
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)
Double Acting Snubbers		
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)
Double Acting Snubbers in Local Axes*		
SNBA	SNBA(Stiffness)	SNBA or SNBA (1E12)
SNBY	SNBY(Stiffness)	SNBY or SNBY(R)
SNBZ	SNBZ(Stiffness)	SNBZ or SNBZ(RIGID)
Single Acting Translational Restraints		
+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)

Double Acting Limit Stops		
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)
Single Acting Limit Stops in Local Axes*		
Axial	LIMA(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMA(1E12:30:0/NONE) or LIMA(RIGID:-50/NONE:0.3)
Shear Y	LIMY(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMY(1E12:30:0/NONE) or LIMY(RIGID:-50/NONE:0.3)
Shear Z	LIMZ(Stiffness:Lower Gap/Upper Gap:Friction Co-efficient)	LIMZ(1E12:30:0/NONE) or LIMZ(RIGID:-50/NONE:0.3)
Double Acting Limit Stops in Local Axes*		
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]
Double Acting Skewed Restraints		
Skewed Restraints	Skew(VecX:VecY:VecZ:Stiffness:Gap: Friction coefficient:Type of Restraint)	Skew(0.707:0.707:0.0:1E12: : :R)
Double Acting Skewed Restraints in Local Axes*		
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

Guide		
GUI	GUI(Stiffness:Gap:Friction Co-efficient)	GUI or GUI(1E12) or GUI(R:50) or GUI(RIGID:25:0.25)
Spring Hangers		
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)
User Hangers		
User Hangers	UserHanger(Spring Rate:No.of Hangers: Cold Load:Hot Load)	UserHanger(200:1:1131) or UserHanger(200:1:0.0:1088)
Force / Moment		
Force	Force(Fx:Fy:Fz)	Force(1200:800:0.0)
Moment	Moment(Mx:My:Mz)	Moment(0:500:250)
Threaded Joint		
Threaded Joint	TJOINT	TJOINT
User SIF		
User SIF	UserSIF(Value)	UserSIF(100)
Weldolet**		
Weldolet	OLET	OLET

* Valid and available for 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.

** Valid and available for checkSTRESS PCF, checkSTRESS II PCF & checkSTRESS Nuke PCF Version 9.40 or later, CAEPIPE 3D+ for PCF Version 10.50 or later, AutoPlant to CAEPIPE, CATIA to CAEPIPE and PCF 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 CAEPIPE 3D+.

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 selected/Specified in the Plant Design to CAEPIPE 3D+ 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. Staring 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 <Number> 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.