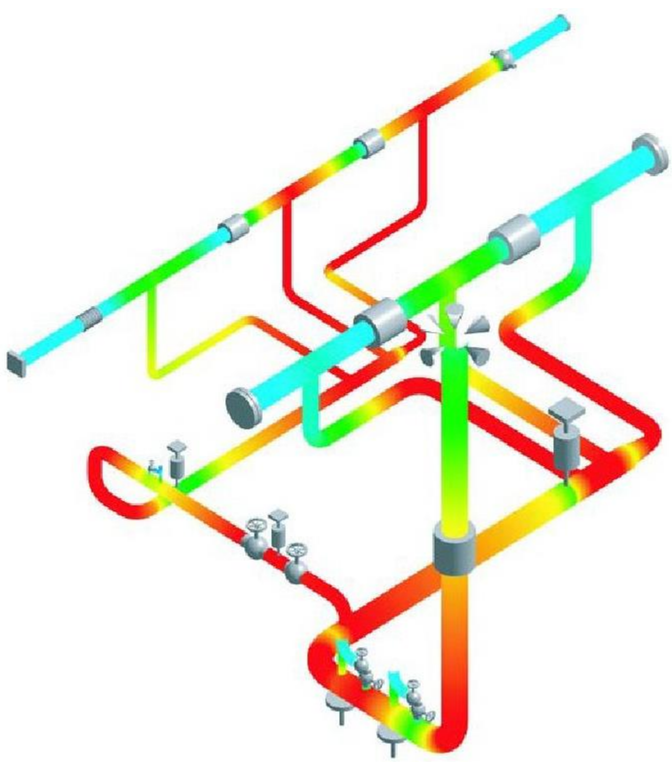
CAEPIPE™

Technical Reference Manual



From the CAdvantagE™ Library

SST SYSTEMS, INC.

The *FASTEST* Solutions for Piping Design and Analysis

Disclaimer

Please read the following carefully:

This software and this manual have been developed and checked for correctness and accuracy by SST Systems, Inc. However, no warranty, expressed or implied, is made by SST Systems, Inc., 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. All other product names mentioned in this document are trademarks or registered trademarks of their respective companies/holders.

For Info and Support on CAEPIPE, contact:

SST Systems, Inc.
1798 Technology Drive, Ste. 236
San Jose, CA 95110, USA
Tel: (408) 452-8111
Fax: (408) 452-8388

info@sstusa.com
support@sstusa.com
www.sstusa.com

Table of Contents

INTRODUCTION **1**

TECHNICAL REFERENCE **2**

ANCHOR 3
BALL 9
BEAM 12
BELLOWS 24
BEND 26
BRANCH SIF 47
BURIED PIPING 50
COLD SPRING (CUT PIPE) 66
COMMENT 68
COMPRESSOR 69
CONCENTRATED MASS 72
CONSTANT SUPPORT 73
DATA TYPES 74
DIRECTION 75
DYNAMIC ANALYSIS 78
DYNAMIC SUSCEPTIBILITY 85
ELASTIC ELEMENT 87
EXPANSION JOINTS 88
FIBER REINFORCED PLASTIC PIPING (FRP) 103
FLANGE 107
FLANGE AND BOLT STRESSES 115
FORCE 128
FORCE SPECTRUM 129
FROM (NODE) 135
GENERIC SUPPORT 137
GUIDE 140
HANGER 142
HINGE JOINT 147
HYDROTEST 153
JACKET END CAP 155
JACKETED PIPING 156
LIMIT STOP 162
LOAD 172
LOCAL COORDINATE SYSTEM 176
LOCATION 185
LUG EVALUATION 186
MATERIAL 195
MISSING MASS CORRECTION 205
MITER BEND 207
NODE 216
NONLINEARITIES 217
NOZZLE 221
NOZZLE EVALUATION 231
PRESSURE DESIGN - PIPE & PIPE FITTINGS 243
PUMP 256
REDUCER 266
REFINE NODAL MESH 273
RELIEF VALVE LOAD ANALYSIS 287
RESTRAINT 296
RIGID ELEMENT 297
ROD HANGER 299

Table of Contents

SECTION ... 301
SKEWED RESTRAINT ... 305
SLIP JOINT ... 310
SNUBBER ... 313
SPIDER ... 314
STATIC SEISMIC LOAD-ASCE/SEI 7-16 ... 315
STATIC SEISMIC LOAD-ANSI A58.1-1988 ... 320
SUPPORTS ... 322
TEES ... 325
TIE ROD ... 328
TURBINE ... 329
USER HANGER ... 332
USER SIF ... 345
VALVE ... 346
WELD ... 351
WIND LOAD – ANSI A58.1-1982 ... 352
WIND LOAD – ASCE/SEI 7-16 ... 353
WIND LOAD – EN 1991-1-4 (2010) ... 357

ANNEXURE I- DYNAMIC SUSCEPTIBILITY METHOD ... 363

ANNEXURE II - NOZZLE STIFFNESS CALCULATIONS ... 375

INDEX ... 387

This page is blank

Introduction

Thank you for licensing CAEPIPE (pronounced kay-pipe), the simple yet powerful software for solving a variety of piping design and stress analysis problems in numerous industries listed below.

Power(fossil & nuclear)	Oil & Gas production (onshore & offshore)
Refinery	Chemical & Petrochemical
Fertilizers	Pharmaceutical
Sugar & Food Processing	Paper & Pulp
Steel / Metal Process	Water & Waste Treatment
Aircraft and Aerospace	Building Services
Defense Industries	Ship Building

CAEPIPE performs linear and nonlinear static and linear dynamic analyses of piping systems by imposing various loads such as deadweight, thermal, seismic, wind, spectrum, time history or harmonic, and calculates displacements, forces, moments, stresses, support loads etc. Further, it checks whether the piping system is piping code and guideline compliant (ASME, B31, European, Swedish, API 610, etc.) and producesconcise, formatted and easy to understand reports.

For rapid modeling, CAEPIPE offers you a friendly and productive user interface that rigorously adheres to Windows standards. Open up to four windows simultaneously to get feedback on different aspects of the model. Extensive graphical display capabilities allow you to zoom, pan, rotate the image and see the model from different viewpoints. CAEPIPE uses the industry standard OpenGL® to render 3D images realistically for easy visualization. As the model is input and modified, CAEPIPE updates the graphics simultaneously to provide visual feedback. It animates deflected shapes and mode shapes, and shows color-coded stress contours, among others.

A true powerhouse in its speed of operation, CAEPIPE uses advanced Windows programming techniques such as intelligent repainting, scroll box tracking, multithreading, memory-mapped files for faster data access, among others, to make your job easier and faster. Every effort is made to keep the program and data file sizes small (e.g., program size is ~2 MB! And a 665-element piping model is 85 KB!).

Many thoughtful and useful details in the program allow you to work more productively. For example, you can annotate your model with copious comments for enhanced documentation, or duplicate repetitive inputwith one hotkey combination or rotate sections of the model with one operation. No unnecessary buttons clutter the toolbar nor are you forced to use a mouse unnecessarily. The many thoughtful keyboard shortcuts, too, add to your productivity.

Overall, CAEPIPE stands peerless among the tools available today for piping design and stress analysis. We invite you to explore the software so that you can make full use of its capabilities. Our friendly and knowledgeable support engineers are available to assist you.Should you need to reach them, please email: support@sstusa.com.

Two sections make up this manual:

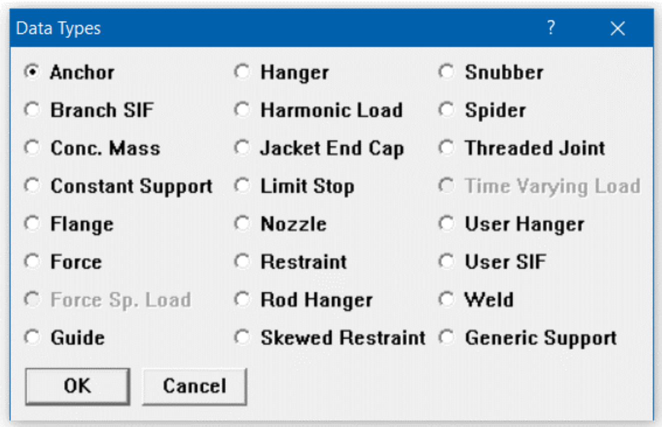
1. Explanation of menus from the different CAEPIPE windows,
2. Appendices with related information.

The manual ends with an index.

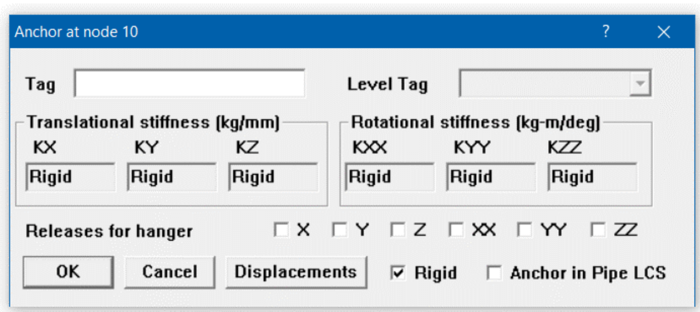
Technical Reference

Anchor

An Anchor is a support that restrains the pipe movement at a node in the three translational and the three rotational directions (i.e., restrains the node in all six degrees of freedom). In a physical piping system, this node may be on an anchor block or a foundation, or a location where the piping system ties into a wall or a large piece of equipment like a pump.



An Anchor is input by typing "A" in the Data column or selecting Anchor from the Data Types dialog. A rigid anchor is entered (i.e., an anchor with rigid stiffnesses in all six degrees of freedom) by default. To change the default, edit the anchor by double-clicking on the anchor or pressing Ctrl+D in the row that has the anchor. An Anchor dialog is shown next.



Uncheck the "Rigid" checkbox to make the anchor non-rigid (i.e., a flexible anchor), and enter numerical values for stiffnesses in the six degrees of freedom.

Stiffness

The stiffness for a degree of freedom may be rigid (specified by typing the letter "r" in the stiffness field), or some value or be left blank. If it is left blank, there is no stiffness in that degree of freedom, i.e., the pipe is free to move in that degree of freedom. Internally, the rigid stiffness value is set to 1×10^{12} (lb./inch) for translational stiffness and 1×10^{12} (inch-lb./radian) for rotational stiffness.

Releases for Hanger Selection

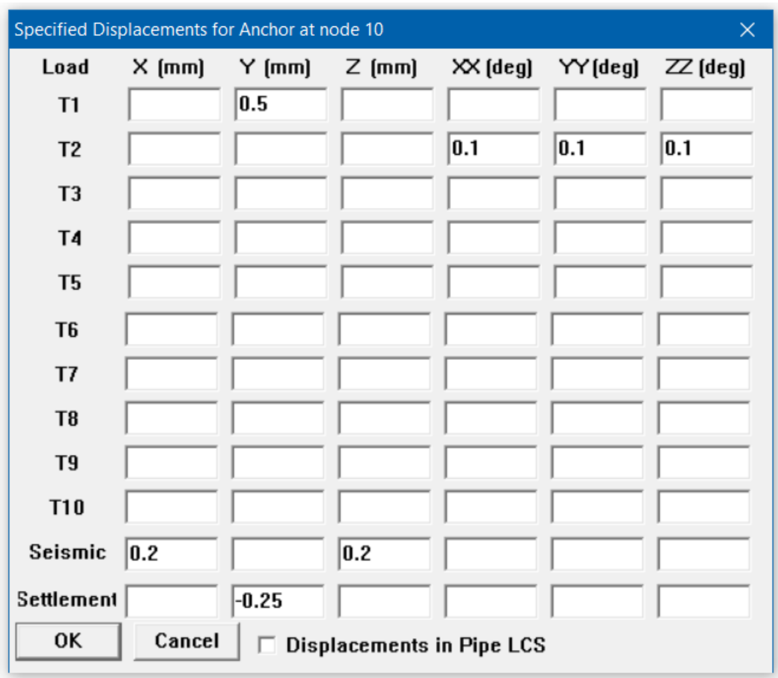
These releases apply only during the automatic selection of a hanger by CAEPIPE. If you checked any of the Release check boxes, the pipe is assumed to be free in that released degree of freedom during the automatic hanger selection by CAEPIPE. You may release any combination of degrees of freedom at a hanger node during such automatic hanger selection. This feature is useful when hangers are located near equipment, where you want the hangers to carry most of the weight load and thus reduce the load acting on the nearby equipment. CAEPIPE, during hot load calculation (preliminary sustained load case) in hanger design, releases the anchors (if you selected any of the Release checkboxes) so that the weight loads are taken by the hangers rather than by the anchors (which represent the equipment).

After the hot load calculation, CAEPIPE restores the original values of stiffnesses to the released anchors before continuing analysis. Release anchors when they are (typically) within four (4) pipe diameters away from the nearest hangers.

You may release any combination of translations or rotations. Typically, either the vertical translation or all translations and rotations are released. To release the Anchor in a particular degree of freedom, check the corresponding checkbox.

Displacements

The dialog for Specified Displacements at an Anchor is shown below.



At an anchor, three types of translations and/or rotations in the global X, Y and Z directions may be specified as listed below.

1. Thermal displacements (up to 10sets can be specified, one each for thermal loads T1 through T10). Applied only to the corresponding Expansion and Operating load cases.
2. Seismic displacement (were available for B31.1, B31.9, ASME Section III Class 2, RCC-M and EN 13480 codes only. Starting Version 10.50 of CAEPIPE, Seismic displacement is available for all codes). Solved as a separate internal load case, the results of which are added absolutely to static seismic and response spectrum load cases.
3. Settlement (available under ASME B31.1, ASME Section III Class 2, RCC-M and EN 13480 codes only). Applied as a separate load case called Settlement.

You may specify a displacement only if you *also* specify a corresponding non-zero or rigid stiffness in that degree of freedom, i.e., the corresponding stiffness should not be left blank.

Check "Displacements in LCS" if you want to enter anchor movements in the local coordinate system. These local movements are transformed into the global coordinate system and displayed in results.

Settlement

For certain piping codes (ASME B31.1, ASME Section III Class 2, RCC-M and EN13480), an anchor settlement, which is a single non-repeated anchor movement (e.g., due to settlement of foundation), may be specified. This is applied to the Settlement load case. For those codes that do not have a provision for settlement (like B31.3), specify the anchor settlement as a thermal displacement (which tends to be a conservative approach) for one of the temperature load and define that temperature as equal to reference temperature.

Anchor in Local Coordinate System (LCS)

Check the box "Anchor in LCS" if you want to orient the anchor along a skewed line using its local coordinate system (LCS), which also aids you in specifying Displacements in LCS. Notice the naming convention changes (KXX changes to kxx, X changes to x, and so on).

Note:

1. Pressure Thrust (End-cap Force) of Pressure P x Inner Area (A) of pipe is not included in the Support Loads for Anchors displayed by CAEPIPE at this time. Since CAEPIPE's results for numerous problems compare well with the results from other third-party software, it confirms that the other stress programs are also not including the Pressure Thrust (End-cap Force) of pipe in the Anchor Loads at this time. Refer to the Verification Manual supplied with CAEPIPE for comparison of results with other stress programs.

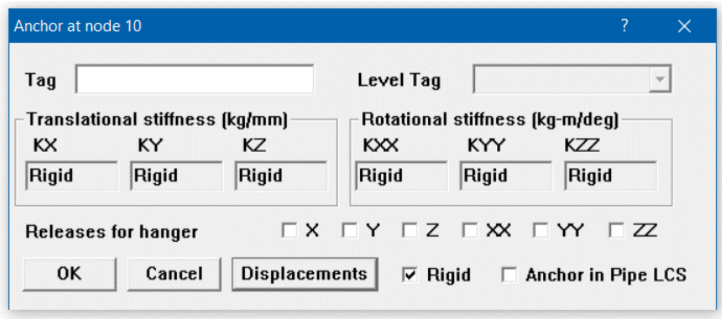
   If you wish to include the effect of Pressure Thrust (End-cap force) due to internal pressure in your piping on the Anchor loads, then you will have to compute the same manually (= P x A) and input it as an external force at the Anchor Nodes using the Force data type available with CAEPIPE. Please choose the option "Add to W+P" in the Force data type dialog. By doing so, the End-cap force (= P x A) will be included in all relevant load cases and combinations of CAEPIPE. Of course, when the "None" code is selected under Options>Analysis> Code, this End-cap force is included in the only case of "Static".

2. *At an Anchor defined in space to which a pipe support is attached, a "dummy"elementhasto be added. This additional element should be defined such that the Local Coordinate System forthis element should be the same as the Local Coordinate System of the attached Anchor.*

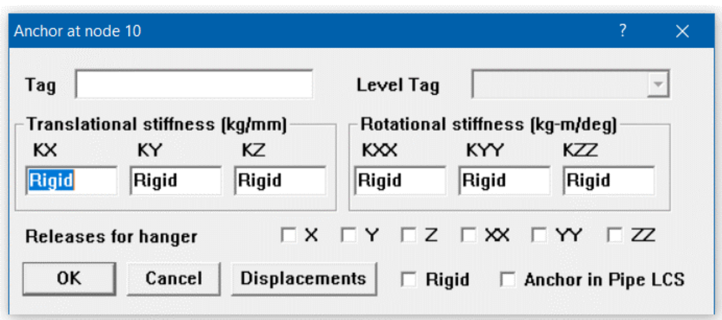
Example 1: Flexible Anchor

Nodes on most large equipment are modeled as rigid anchors. If you need to specify a non-rigid (i.e., flexible) anchor (for example at a nozzle to include vessel flexibility), you can input those stiffnesses by editing the anchor.

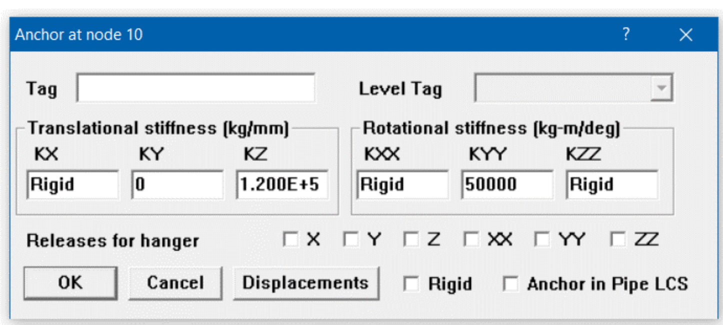
Double click on the anchor to show the anchor dialog.



By default the anchor has all stiffnesses rigid, no releases for hanger selection and no specified displacements. The stiffness fields are *grayed,* i.e., non-editable and the Rigid checkbox is checked. Click on the Rigid checkbox to uncheck it. The stiffness fields now become editable.



Type in the required stiffness values and press Enter or click on OK. The anchor definition shown in the next figure is now modified to be a flexible anchor.



Example 2: Rigid Vertical Support with Foundation Settlement

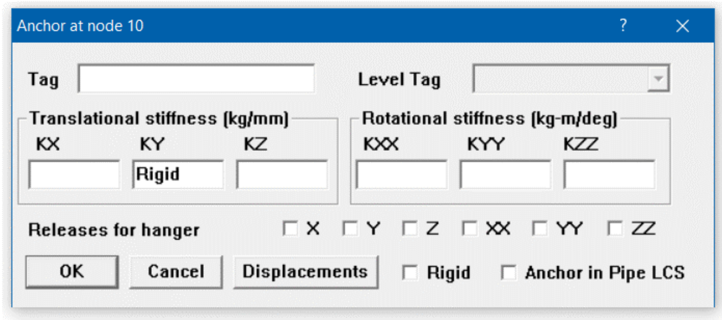
Assume that you need to model a vertical support on a foundation that has settled using ASME Section III Class 2 (1980) code for code compliance.

Vertical settlement (–Y) = 6 inches.

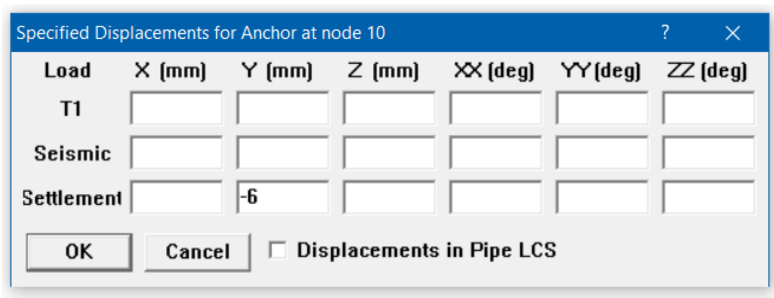
First, set the piping code to ASME Section III, which has a provision for Settlement load case, using the menu Options > Analysis > Code in the Layout window.

Next, create a rigid vertical support at the required node. Press "a" in the Data field to input a default anchor.

Next, edit the anchor so that it acts as a vertical support only, by modifying the stiffnesses similar to Example 1 so that only a Rigid stiffness KY in the Y direction remains.

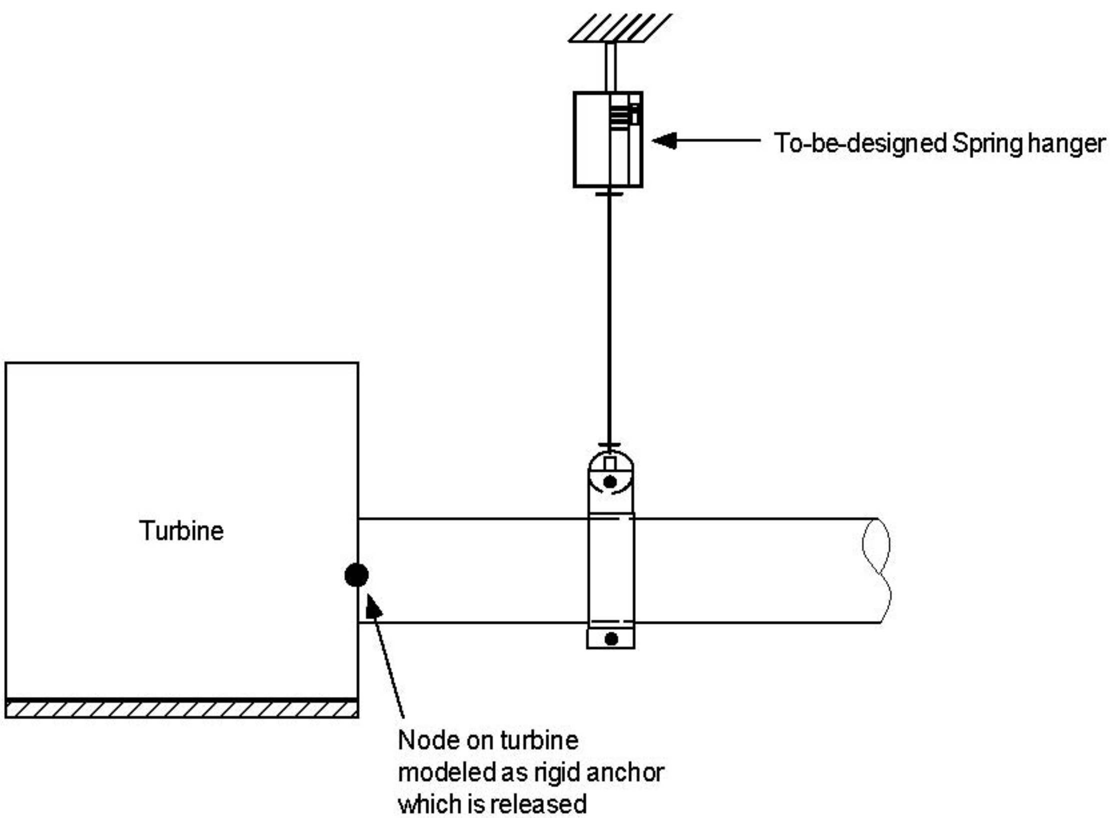


Now the anchor is modified to act as a Vertical 2-way rigid support. Click on the Displacements button and type in –6 (inch) for Settlement under Y. You could also input thermal and seismic displacements if required.



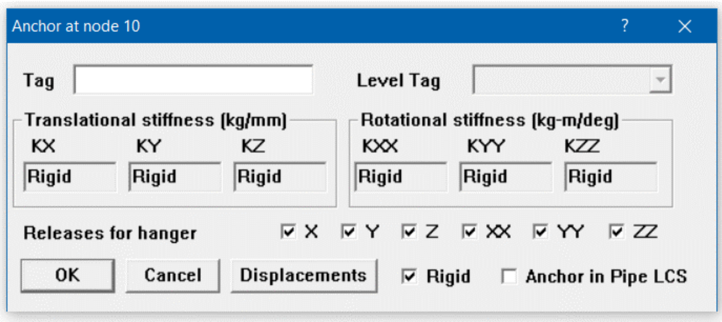
The anchor is now modified to be a rigid vertical support with a specified settlement displacement.

Example 3: Anchor Release during Hanger Design



CAEPIPE lets you model equipment nozzles as anchors. Assume that you had one on a turbine, as shown above, and that you have placed a hanger nearby. The main purpose of this hanger would be to carry most of the piping weight that would have been imposed on the nearby turbine nozzle if not for the hanger. To let CAEPIPE do that, you will have to release all six degrees of freedom of the Anchor during hanger design so that the hanger will be designed to carry most of the piping weight.

First, enter an anchor for the node and then double click on it to edit it.

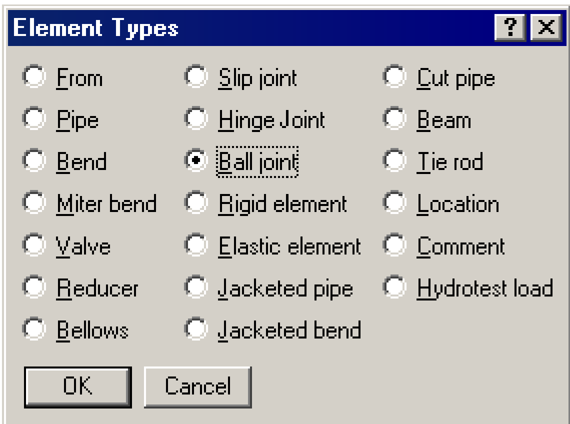


Click on the checkboxes for Releases for hanger selection in the required directions (X, Y, Z, XX, YY, ZZ). The anchor will be released in the specified directions during hanger design.

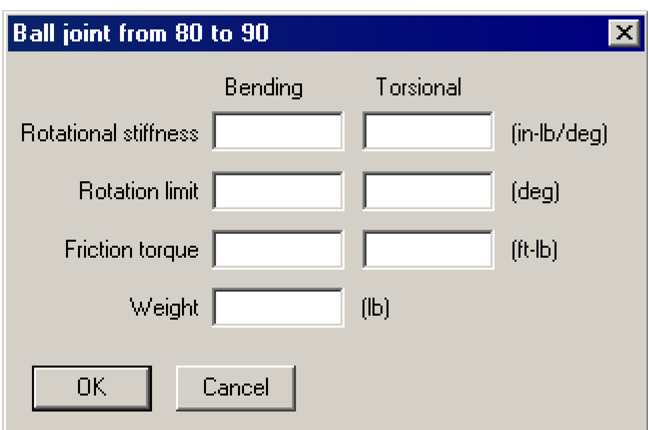
CAEPIPE restores the anchor to its original state (of no releases) after completing the preliminary hot load calculation during hanger design. Refer to the section on Hanger Design Procedure for further details.

A ball joint is a zero-length pipe element that allows rotations about the three orthogonal global axes (similar to a universal coupling joint in the rear axle of a motor vehicle) while still allowing the fluid to flow through it. If you do not want rotation in the torsional or (the two) bending directions, input "Rigid" for the respective stiffnesses. Since the ball joint is a zero-length element, the "From" and "To" nodes are coincident. Hence, you should leave the DX, DY and DZ fields in the Layout window blank (CAEPIPE will not let you enter a value).

A ball joint is input by typing "Ba" or "Ball" in the Type column or selecting Ball joint from the Element Types dialog.



The Ball joint dialog is shown.



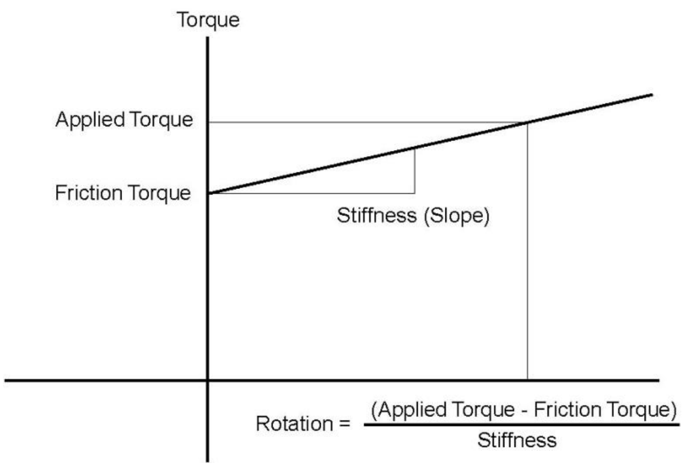
Weight of the ball joint is input in lbf or kgf and NOT its mass. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

The rotational stiffnesses, rotation limits and the friction torques are specified independently in the bending and torsional directions. The torsional direction (local x) is determined by the preceding element's local x. If a preceding element is unavailable, the following element is used to determine the torsional direction. The bending directions (local y and z) are orthogonal to the torsional direction (local x). Bending friction is determined by a resultant of frictiontorques in local y and z directions. Similarly, bending rotation limit is determined by a resultant of rotational limits in local y and z directions.

The stiffnesses, rotation limits and friction torque values are available from the manufacturer of the ball joints or from their test results. Otherwise, you must use engineering judgment.

The stiffness values may be left blank, in which case CAEPIPE uses a very small value (1 in.-lb./rad) internally to avoid dividing by zero during internal computation.

A rotation limit of zero (0.0) means that the ball joint cannot rotate (i.e., it is rigid) in that direction. A rotation limit of “None” or Blank means that rotation is not limited to a finite value.



When the applied torque is less than the friction torque, there is no rotation. When the applied torque exceeds the friction torque, rotation is calculated as shown above. When rotation limit is reached, there is no further rotation irrespective of the applied torque.

When the option “Use friction in dynamic analysis” is selected, for modal analysis, CAEPIPE uses three different stiffnesses for a ball joint depending on the magnitude of the applied moment/torque in comparison to the user-specified friction torque. They are as follows:

Case 1: When the applied moment at the ball joint (for the first operating load case when a piping code is selected or for the static case when “Piping code=None” is selected) **is less than** the friction torque, the friction is not overcome and the ball joint stiffness is internally set to "rigid”, i.e., 1×10^{12} (inch-lb./radian).

Case 2: When the applied moment **is more than** the friction torque, the friction is overcome and the ball joint starts rotating (with the user-specified rotational stiffness being applied). This rotational deformation takes place until the user-specified rotational limit is reached.

So, from the time friction is overcome to the time when the rotational limit is reached, CAEPIPE internally sets an “equivalent stiffness” for the ball joint as given below.

If K_b is the user-specified stiffness for the ball joint,

Rotation = (applied moment – friction torque)/ K_b ……… (1)

The “equivalent stiffness” chosen by CAEPIPE is the slope of the straight line from origin to the point (rotation, applied moment) in the figure given above. In other words,

K_{be} = (applied moment/rotation) …….. (2)

Combining (1) and (2), you get

K_{be} = (applied moment) x K_b / (applied moment – friction torque) ……. (3)

Case 3: When the actual rotation reaches the user-specified rotational limit for the first operating load case/static case, the rotational stiffness for the ball joint is again set to "rigid", i.e., 1×10^{12} (inch-lb./radian).

For modal analysis, CAEPIPE uses "equivalent stiffness" for ball joints as described under Case 2 above, when the friction torque is exceeded and the computed rotation is yet to reach the specified rotational limit, *as long as* the option "Use friction in dynamic analysis" is selected. When this option is NOT selected, CAEPIPE ignores the nonlinearities of the ball joint (namely, friction torque and rotational limit) and uses only the user-specified stiffnesses for modal analysis. That is why, despite very small moments, large rotations may be computed (i.e., a check against user-specified rotational limit is not performed).

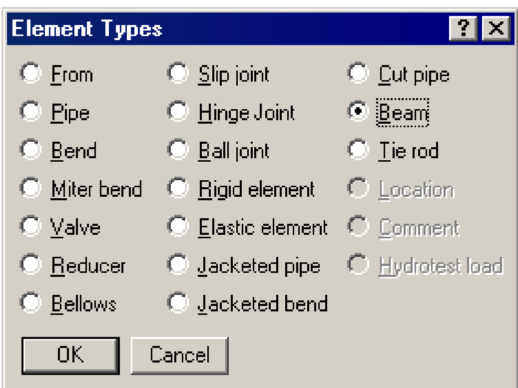
See the topic Expansion Joints for examples.

You can model elaborate structural systems inside CAEPIPE alongside the piping to be supported. In simple situations, if the structure is much stiffer than the piping is, you may not need to model the structure at all but simply treat it as rigid (for example: input Rigid for Stiffness in a vertical Limit stop when simulating a support where pipe could rest on a stiff beam). But, in cases where you need to account for structural flexibility, use the Beam element to model structural support systems alongside piping systems.

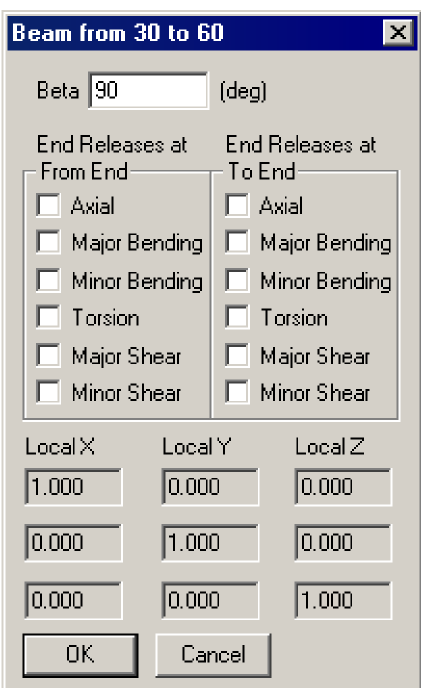
The material, section and load for a beam are different from those for a pipe. Just as you would define a material/section/load for a pipe, so too should you define a separate material/section/load for a beam. Look for Beam Material, Beam Section, and Beam Load (under Misc menu).

Upon analysis, CAEPIPE reports forces and moments for beam elements.

A beam is input by typing "bea" in the Type column or by selecting "Beam" from the Element type dialog.



The Beam dialog is shown.



Beta angle

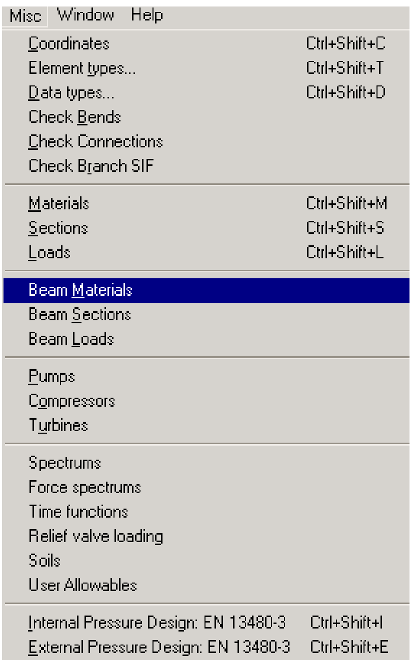
Beta angle is used to define the orientation of a beam's local axes. See Beam orientation later in this section.

Beam End Releases

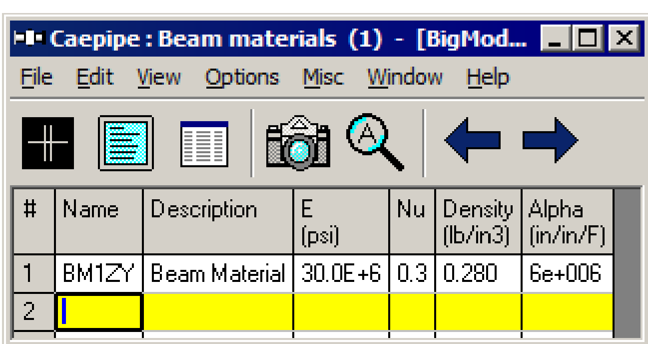
Each end of the beam (From and To ends) can be released to simulate the type of structural support you want to model. That is, you can use a combination of releases to specify whether a beam end is fixed, pinned, etc.

Beam material

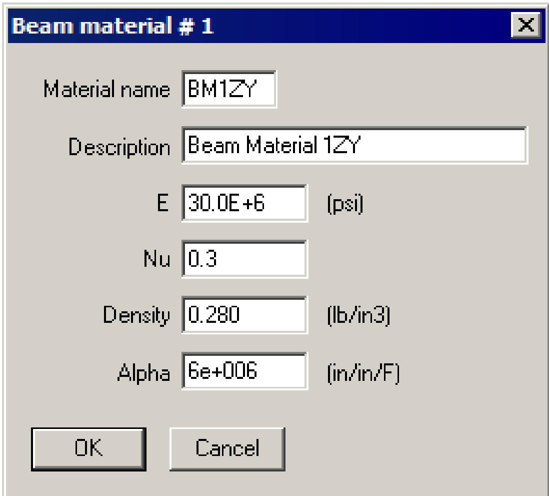
Before you input a beam element, you must define a beam material, section and load. Select Beam materials from the Miscellaneous (Misc) menu in the Layout or List window.



A beam material list window is shown. Double click on an empty row to input a new beam material.



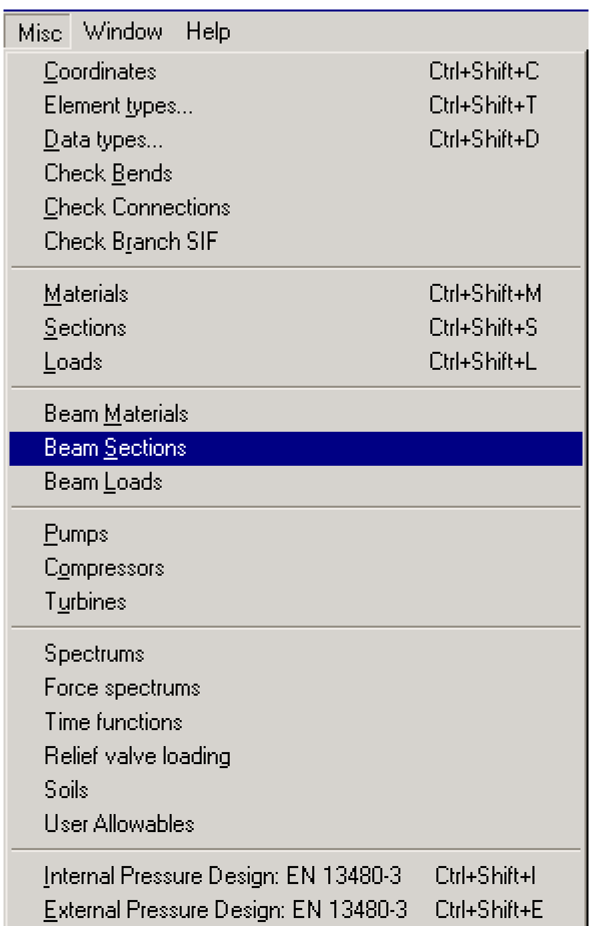
A dialog for inputting beam material is shown.



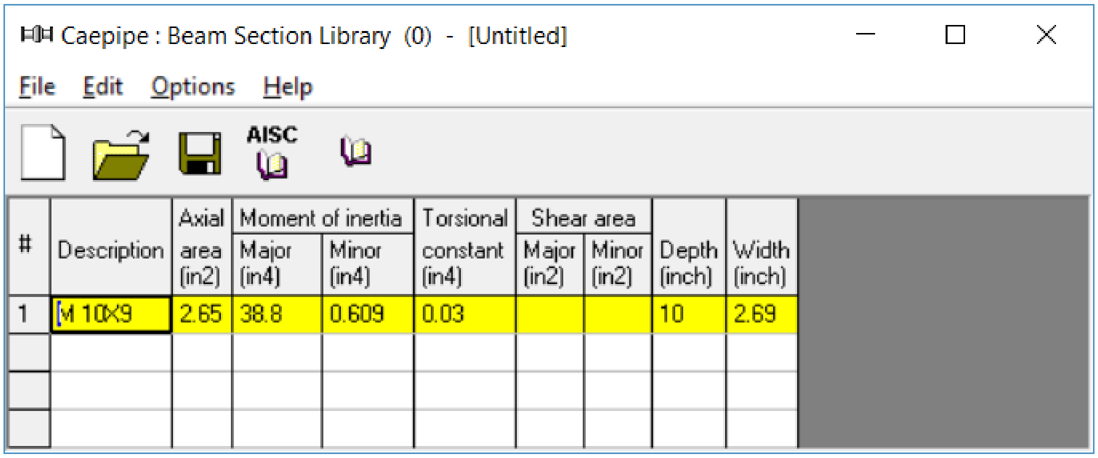
The material name can be up to five alpha-numeric characters to identify the beam material. A more complete description can be entered under Description. Enter modulus of elasticity, Poisson's ratio (Nu), density of the material and mean coefficient of thermal expansion between T_{ref} and T1/T2/T3/…./T10 in beam load.

Beam section

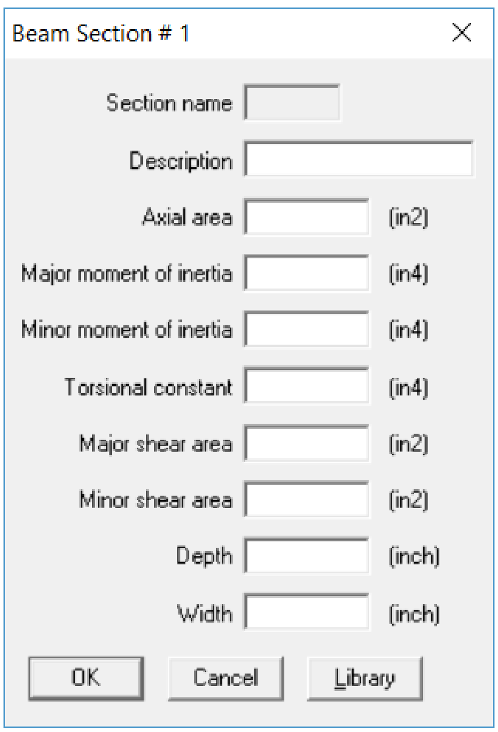
Select Beam Sections from the "Misc" menu in the Layout or List window.



A list of beam sections is shown. Double click on an empty row to input a new beam section.



A dialog to input beam sections is shown.



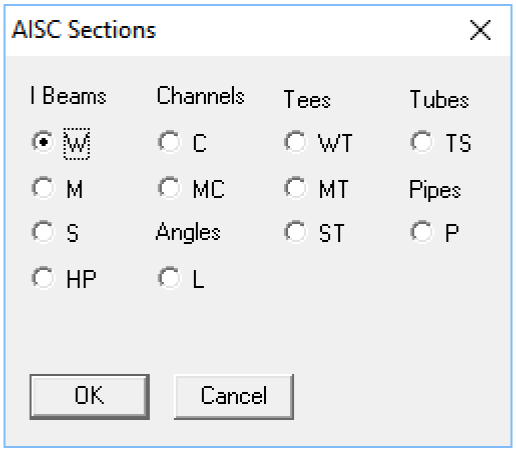
You can either input the data yourself or click on the AISC Library button for a listing of different AISC I-beams, channels, tees, etc., that are built into CAEPIPE or click on the Library button (next to AISC Library) for a listing of different User Defined Beams. Be sure to verify the properties that are shown in the fields after you select a section from the library.

The name can be up to five alpha-numeric characters to identify the beam section. A more complete description can be entered in the Description field.

The axial area, major and minor moments of inertia must be input. Input of torsional constant is optional. If it is not input, it defaults to the sum of major and minor moments of inertia. Input of shear areas is optional. If they are not input, shear deflection is not included.

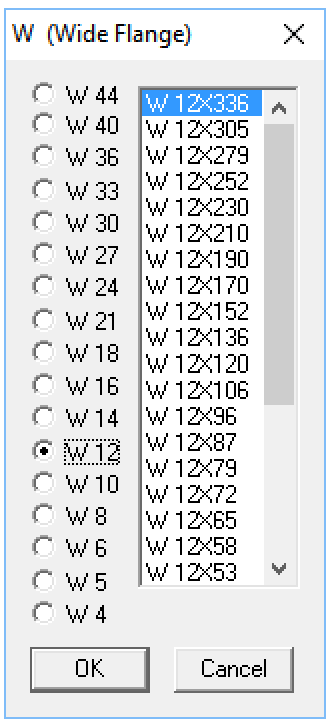
Input of depth and width are optional. Presently, they are used only for rendered plots of the beam.

Dialogs for selecting a beam section from the AISC library are shown below:

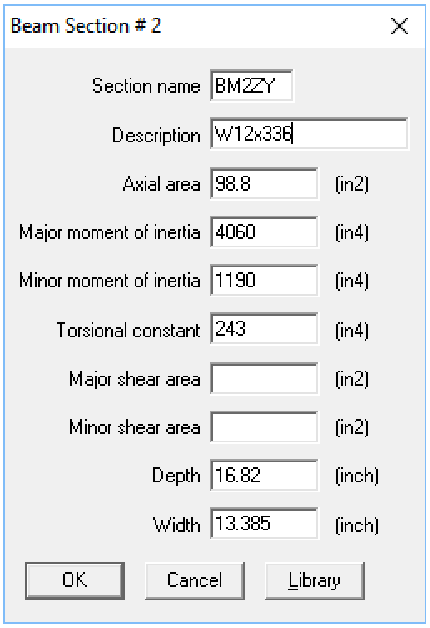


The type of the beam section (e.g., I beam, W (Wide Flange)) is selected from this dialog.

Another dialog which shows various available sections for the particular beam section type is then shown.

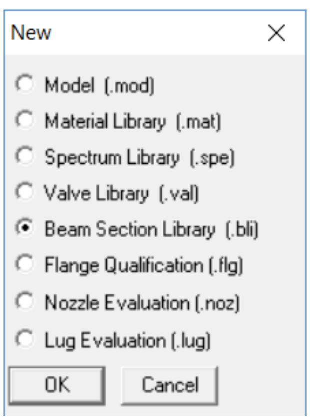


After selecting the section, click on OK and the section properties will be entered in the Beam section dialog.

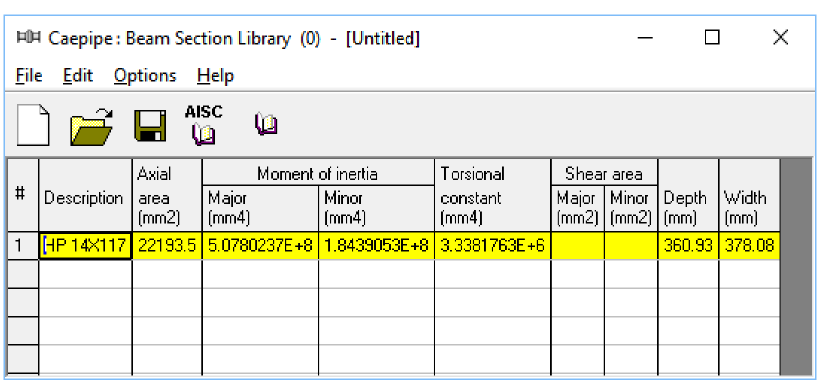


To create or modify a Beam section library

CAEPIPE offers you flexibility in creating your own Beam section libraries (user-defined libraries). That way, you do not feel restricted by the offered choices in Beam sections and can continually keep updating / adding Beam section libraries with your own sections. To create a library: From the Main window, select File > New and click on Beam Section Library.

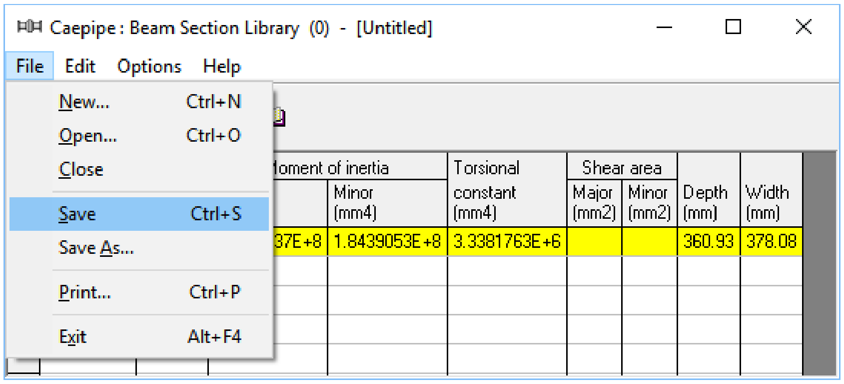


A List window for Beam section is shown.

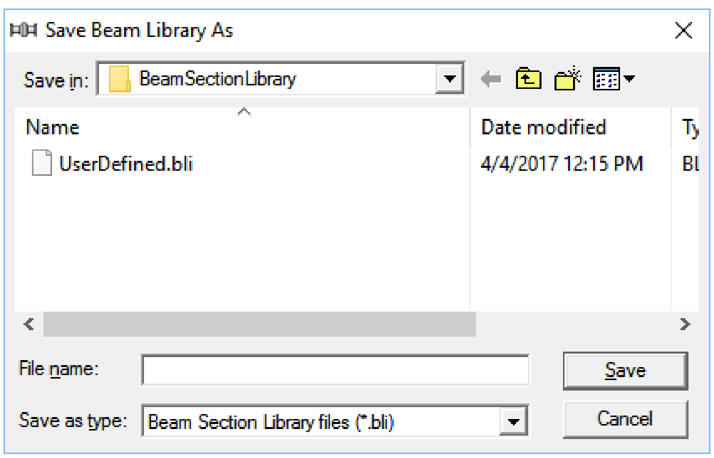


You can, as before, start typing directly into the fields, or enter properties through a dialog. The only difference is that sections in the library do not have names whereas those in a model have names.

After you are done entering sections, you must save to a Beam Section library file by using the File > Save command.

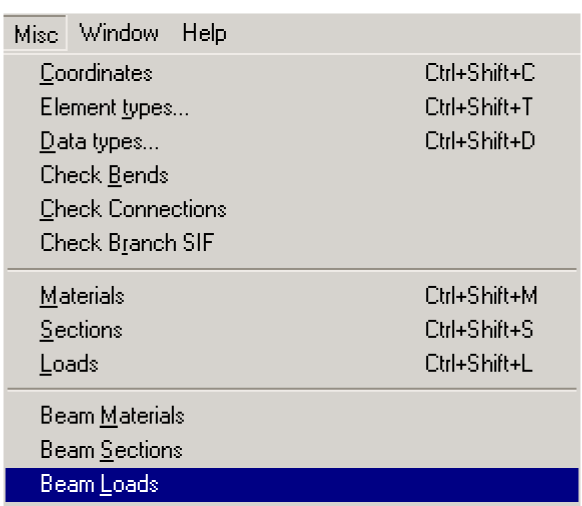


Give the file a suitable name. The file will be saved with a .bli extension.

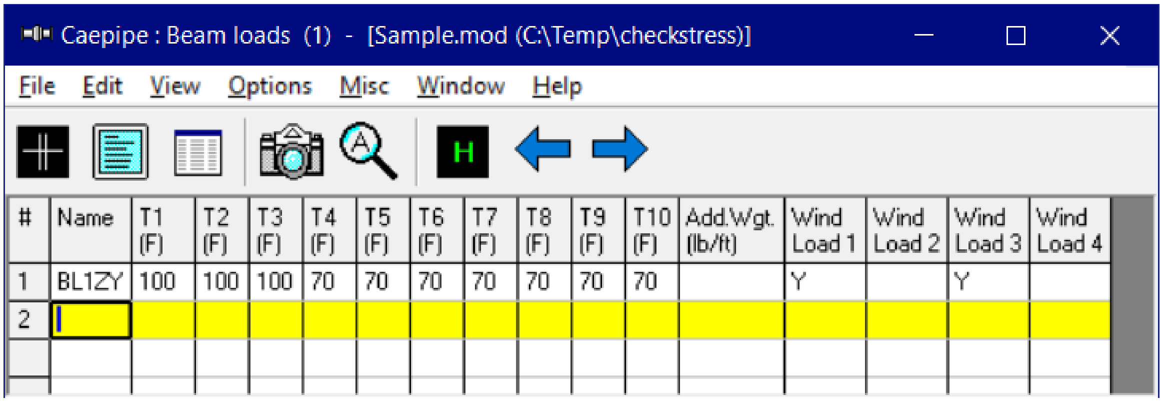


Beam load

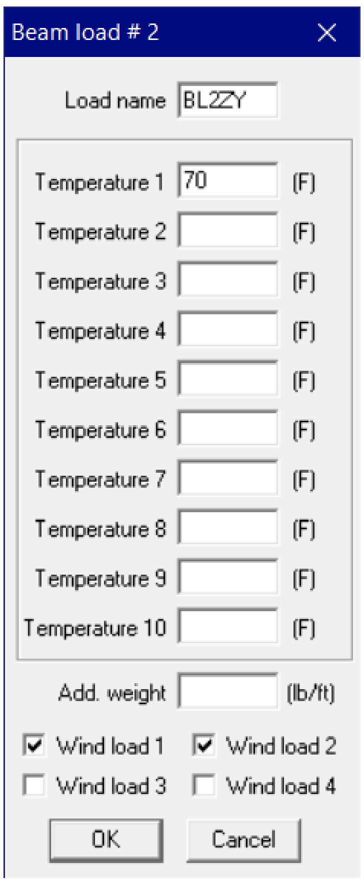
Select Beam Loads from the Misc menu in the Layout or List window.



A list of beam loads is shown.



Double click on an empty row to input a new beam load through the beam load dialog or start typing into the fields.



The Load name can be up to five alpha-numeric characters to identify the beam load. You can enter up to 10 temperatures depending on the preset number of thermal loads. The additional weight is a uniform weight per unit length added to the weight of the beam. This could for example be used to add snow load to the beam. Wind load may or may not be applied to the beam element by using the check box for Wind load 1/2/3/4 in the dialog or typing “Y” or “N” for Wind loads in the List window.

Beam orientation

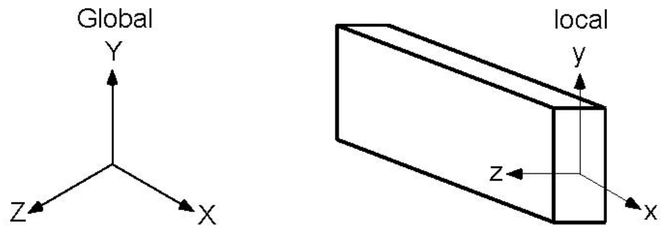
The Beam orientation is determined by the locations of the "From" and "To" nodes and the beta angle of the beam element. The local x-axis of the beam is always from the "From" node to the "To" node. The reference orientation corresponds to beta = 0.0.

A nonzero beta angle (measured from the reference position) rotates the local y- and z-axes of the beam about the local x-axis of the beam in the counter clockwise direction.

The local coordinate system for beams can be displayed for each beam element through the List window (Ctrl+L, select Beams, menu View > Show LCS [for Local Coordinate System]).

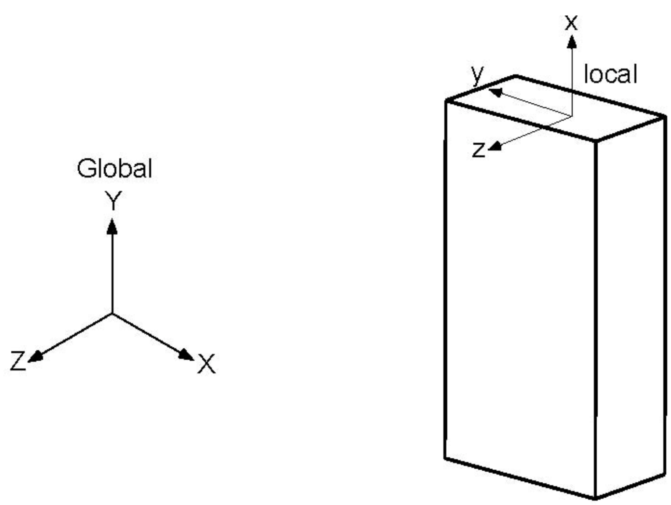
Global vertical axis is Y

Beam is not Vertical



The local y-axis of the beam lies in the local x - global Y plane (i.e., vertical plane) and is in the same positive direction as the global Y axis. The local z-axis is the cross product of the local x and y-axes. Major bending plane is local x-y, that is, Izz = Major moment of inertia and Iyy = Minor moment of inertia.

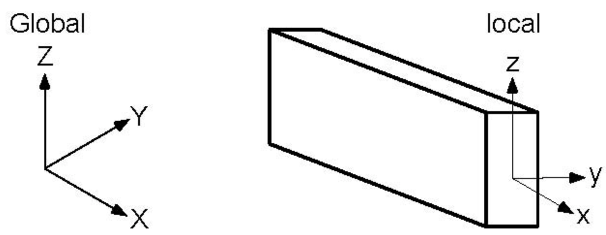
Beam is Vertical



The local z-axis of the beam is in the global Z direction. The local y-axis is in global –X direction. Major bending plane is x-y, i.e., Izz = Major moment of inertia and Iyy = Minor moment of inertia.

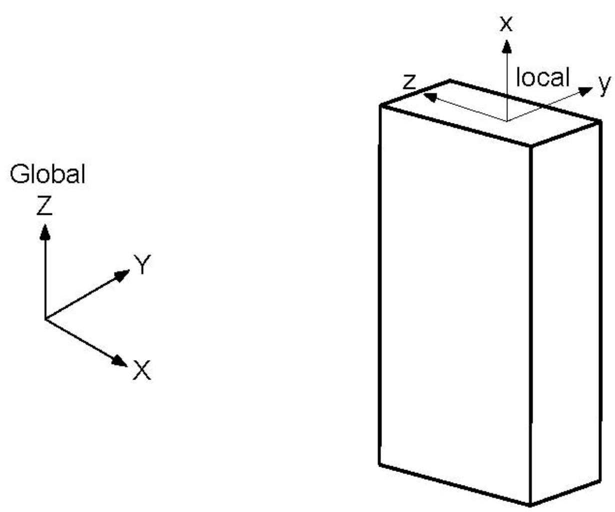
Global vertical axis is Z

Beam is not Vertical



The local z-axis of the beam lies in the local x - global Z plane (i.e., vertical plane) and is in the same positive direction as the global Z-axis. The local y-axis is the cross product of the local z and x-axes. Major bending plane is x-z, that is, Iyy = Major moment of inertia and Izz = Minor moment of inertia.

Beam is Vertical



The local y-axis of the beam is in the global Y direction. The local z-axis is in global –X direction. Major bending plane is x-z, i.e., Iyy = Major moment of inertia and Izz = Minor moment of inertia.

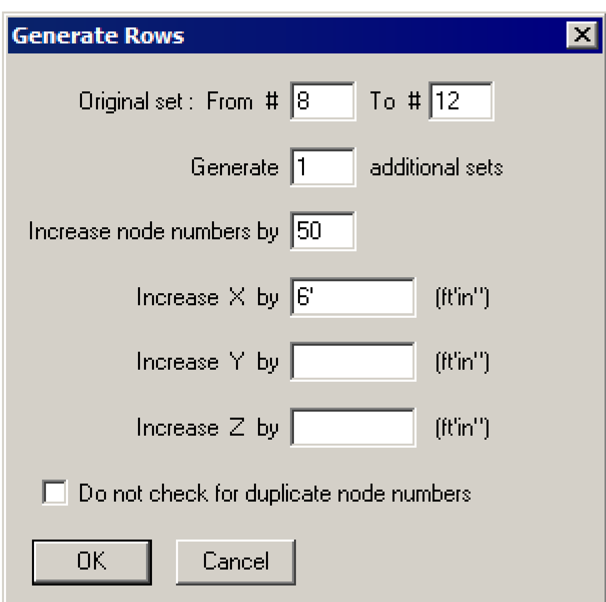
Example 1: Pipe Rack using Beams

Here, you see how to use a beam element to construct a pipe rack and connect the beam to the pipe so that CAEPIPE can account for the rack's flexibility. The procedure is simple. First, you need to create a beam material, section and load in addition to pipe material, section and load.

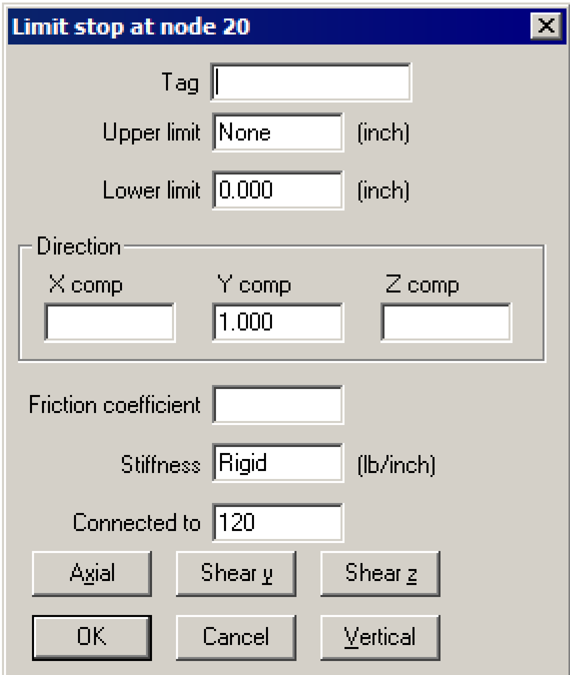
As the Layout window shows, model the piping (nodes 10 to 40) and the first beam support (nodes 100 to 140). Then, create the second beam support (nodes 150 to 190) using the Generate command (under Edit menu in the Layout window). Finally, connect piping at nodes 20 and 30 to beam nodes 120 and 170 using limit stops.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Pipe rack (using Beam elements)								
2	Sample model illustrating structure-pipe interaction								
3	10	From							Anchor
4	20		6'0"			1	1	1	Limit stop
5	30		6'0"			1	1	1	Limit stop
6	40		6'0"			1	1	1	
7	Limit Stop at Node 20 is connecting to Beam at Node 120								
8	100	From	6'0"	-6'0"	3'0"				Anchor
9	110	Beam		5.3048		BM1	BS1	BL1	
10	120	Beam			-3'0"	BM1	BS1	BL1	
11	130	Beam			-3'0"	BM1	BS1	BL1	
12	140	Beam		-5.3048		BM1	BS1	BL1	Anchor
13	Limit Stop at Node 30 is connecting to Beam at Node 170								
14	150	From	12'0"	-6'0"	3'0"				Anchor
15	160	Beam		5.3048		BM1	BS1	BL1	
16	170	Beam			-3'0"	BM1	BS1	BL1	
17	180	Beam			-3'0"	BM1	BS1	BL1	
18	190	Beam		-5.3048		BM1	BS1	BL1	Anchor

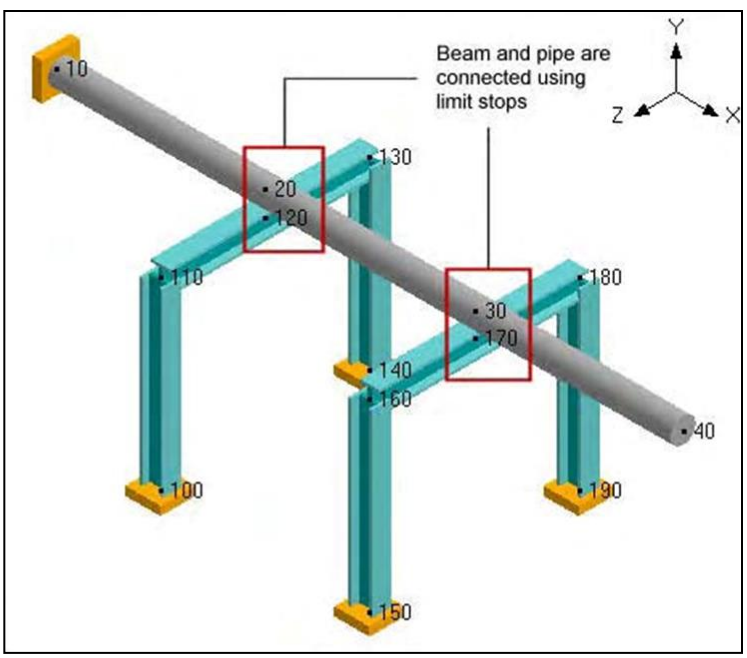
The "Generate" dialog is shown below:



The "Limit Stop" dialog is shown below:



The graphics is shown below:



Example 2: Base Supported Bend using a Beam

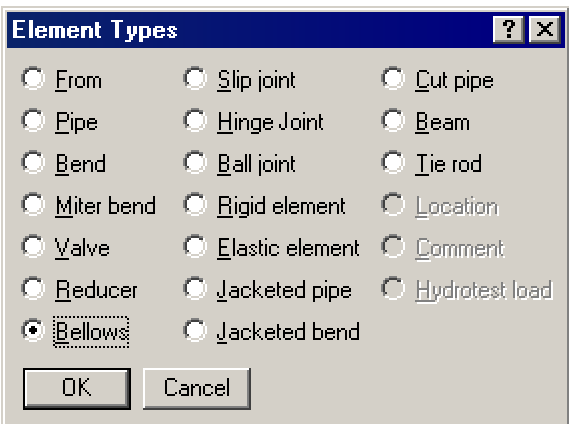
See Example 7 in the Bend section of this manual for modeling a base supported bend using a beam.

Bellows

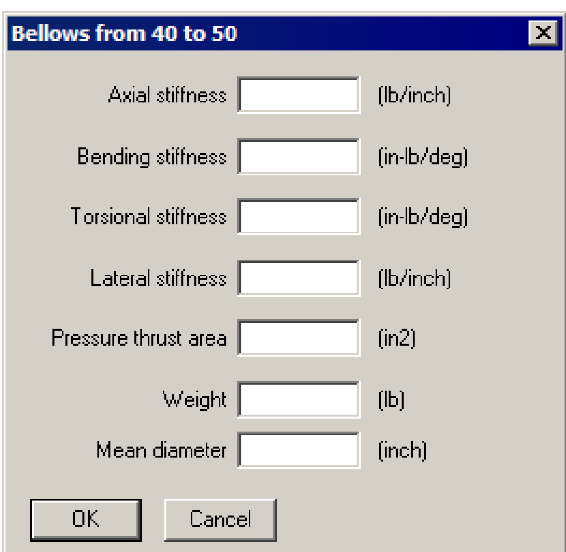
Bellows expansion joints are flexible elements included in high temperature piping systems to absorb primarily thermal movement. A Bellows contains one or more convolutions designed to withstand the internal pressure while still flexible enough to absorb the axial, lateral and bending deflections. Before use, you should note the critical pressure at which the bellows becomes unstable. The B31.1 piping code suggests that expansion joints may be employed only “when piping bends, loops, and offsets are not able to provide adequate flexibility.” (Para. 11.5, 2010).

Usually manufacturers of these expansion joints publish product catalogs that contain technical information about the joints you could use in your systems. The EJMA (Expansion Joint Manufacturers Association) also publishes a standards catalog with guidelines that “assist users, designers, and others in the selection and application of expansion joints for safe and reliable piping and vessel installation.”

A Bellows joint is input by typing “bel” in the Type column or selecting “Bellows” from the Element Types dialog.

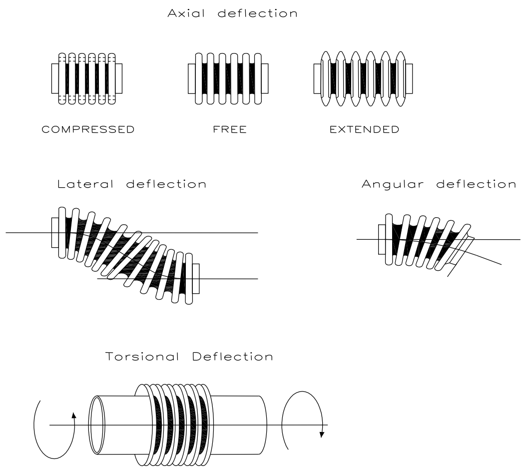


The Bellows dialog is shown.



Expansion joints are mainly modeled using the above shown four types of stiffnesses – axial, bending, torsional and lateral. The required stiffness values, pressure thrust area and weight should be taken from the manufacturer’s catalog.

For a rigid stiffness (for example, torsional), enter "r" for Rigid; if highly flexible, enter 1 (lb/in.) as a minimum, to avoid dividing by zero during internal computation.



Axial: Refers to axial extension (as in cryogenic systems) or contraction (as in high-temperature systems) axially along its centerline while in operation.

Bending (angular): Refers to the bellows bending about its center point on the centerline. Bending can be in any plane that passes through the centerline.

Lateral: Refers to the direction perpendicular to the centerline of the bellows. The two ends of the bellows remain parallel to each other while their centerlines are displaced causing an offset. This direction is also called transverse or parallel offset direction.

Torsional: Usually very stiff, refers to a twisting moment at one bellows end while the other end either relatively is stationary or twists in the other direction, about the bellows centerline.

The pressure thrust area will impose a thrust load of: (pressure × thrust area), on both nodes of the bellows. Even if the bellows is tied, it is recommended that the pressure thrust area be input. The weight is the empty weight. CAEPIPE adds the weight of the contents, insulation and additional weight to the empty weight.

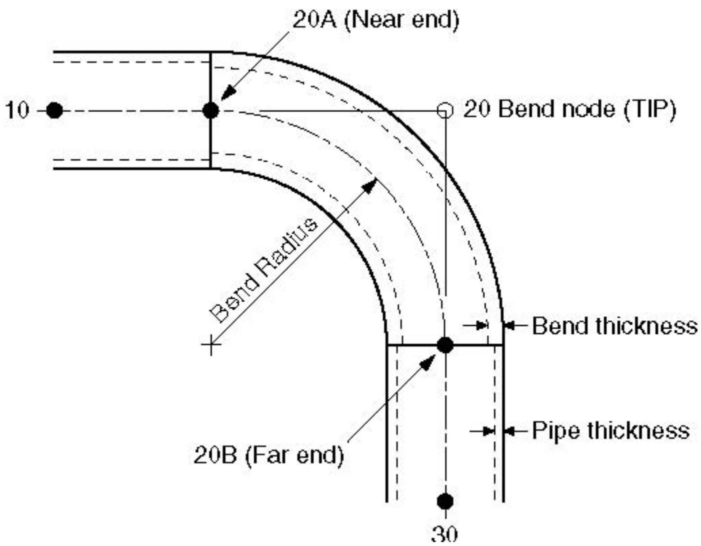
Weight of the bellows joint is input in lbf or kgf and NOT its mass. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

Mean diameter is the "mean" between the outer and inner diameters of any Convolution of the bellows. Since outer and inner diameters of all convolutions of the bellows are the same, the Mean diameter is the same for all convolutions of that bellows.

Pipe guides are needed adjacent to the bellows because of its inherent flexibility and the compressive loading on the adjacent pipes due to the pressure thrust of the joint. Moreover, proper guiding is necessary to direct thermal movement into the joint and prevent buckling of the line. Depending on the bellows behavior, you should place the first guide no farther than four pipe diameters from the joint. Place additional ones appropriately after studying the nearby deflections and loads.

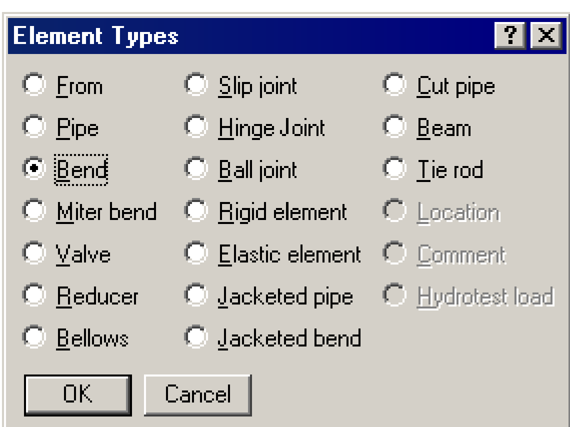
Also, consider vessel and anchor movements, which may cause a misalignment at the joint.See the topic on Expansion Joints for examples.

In CAEPIPE, the term Bend refers to all elbows and bends (custom-bent pipes). An elbow comes prefabricated with a standard bend radius (short or long radius) whereas a bend is custom- made from bending a straight pipe with a specified bend radius. Geometrically, a bend is a curved pipe segment which turns at an angle (typically 90° or 45°) from the direction of the run of the pipe. Some of the items associated with a bend are shown below.

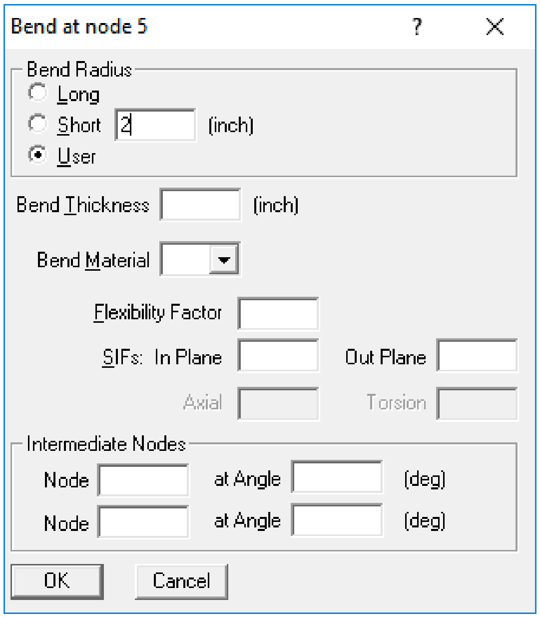


Node 20 is the Bend node, also referred to as the Tangent Intersection Point (TIP). As you can see from the figure, it is not physically located on the bend. Its only purpose is to define the bend. CAEPIPE automatically generates the end nodes of the curved portion of the bend (nodes 20A and 20B), called the near and far ends of the bend. The bend end nodes (20A and 20B in the figure) may be used to specify data items such as flanges, hangers, forces, etc.

A bend is input by typing "b" in the Type column or selecting "Bend" from the Element Types dialog.



If you need to modify an existing bend, double click on it or press Ctrl+T (Edit type) to bring up the Bend dialog.



Bend Radius

The radius of a bend (measured along the centerline of the bend) can be specified as Long, Short, or User (defined) by one of the radio buttons for Bend Radius. CAEPIPE has long and short radii built-in for standard ANSI, JIS, DIN and ISO pipe sizes. For nonstandard pipe sizes, Long radius is equal to 1.5 times the pipe OD and Short radius equal to the pipe OD.

Bend Thickness

Input the wall thickness of the bend if different from the preceding pipe's thickness. If specified, the Bend Thickness applies only to the curved portion of the bend (node 20A to node 20B in the figure above).

Bend Material

If the material of the bend is different from that of the preceding pipe, select the Bend Material from the drop down combo box. The Bend Material, if specified, applies only to the curved portion of the bend (node 20A to node 20B in the figure above).

Flexibility Factor

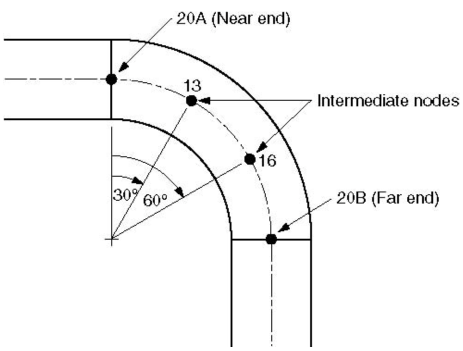
This factor is automatically calculated for standard elbows according to the piping code chosen. If you have your own Flexibility Factor, enter it here instead of the piping code specified Flexibility Factor. A value of 2.0, for e.g., will mean that the bend is twice as flexible as a pipe of the same length.

SIFs

These factors are automatically calculated for standard elbows according to the piping code chosen. If you have your own, specify them here (useful for FRP bends, for example), which will be used instead of the piping code specified SIFs. If User SIFs are also specified at bend nodes (A and/or B nodes), they will be used instead of the bend SIFs or code specified SIFs.

Intermediate Nodes

An intermediate node, located in between the ends of the bend, may be required in some situations to specify data items such as flanges, hangers, forces, etc. You can create an intermediate node by giving a (new) node number and an angle for it, which is measured from the near end of the bend (node 20A in figure). Up to two such nodes may be input. Note that the intermediate nodes 13 and 16 shown below are at angles of 30° and 60° respectively from node 20A (near end). The intermediate nodes can be used for specifying data items such as flanges, hangers, forces, etc.



Note:

CAEPIPE will issue a message "Angle is too short" when the User tries to add an intermediate node at an Angle less than 4.5 degree. Similarly, will issue a message "Angle is too large" when the user tries to add an intermediate node at an Angle less than 4.5 from the Far end.

Bend Examples

Some examples follow. They illustrate some common modeling requirements.

Example 1: 90° Bend

Example 2: 45° Bend and Pipe routing along a smooth curve

Example 3: 180° Bend

Example 4: Flanged Bend

Example 5: Reducing Bend

Example 6: Bend Supported by a Hanger

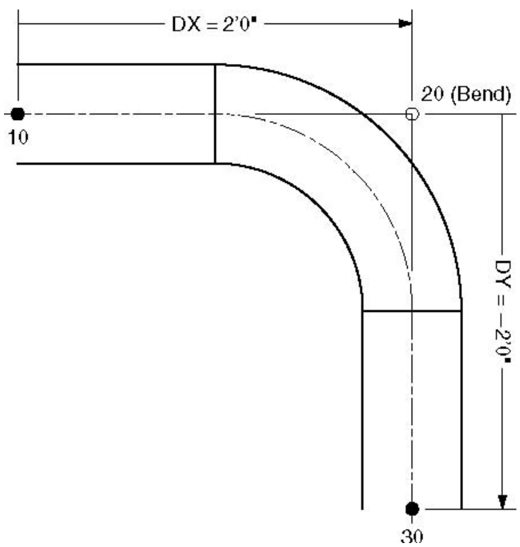
Example 7: Base Supported Bend

Example 8: Circular Piping

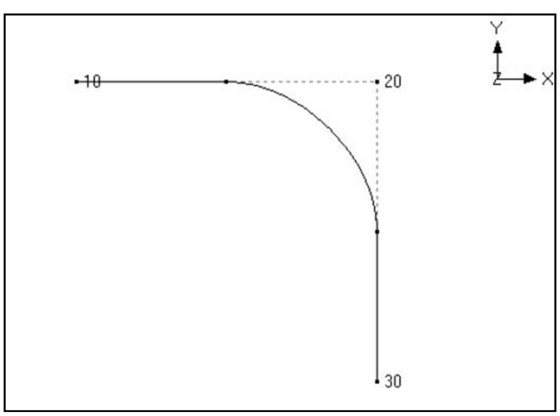
To simplify the discussion of bend modeling, it is assumed that the material, section (8" STD), load and the first node (10) are already defined. It is also assumed that the bend has long radius (12") and the cursor is placed in row #3.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Bend examples								
2	10	From							
3									

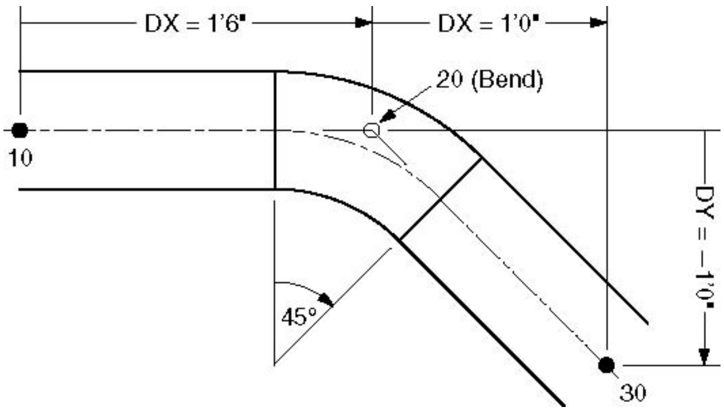
Example 1: 90° Bend



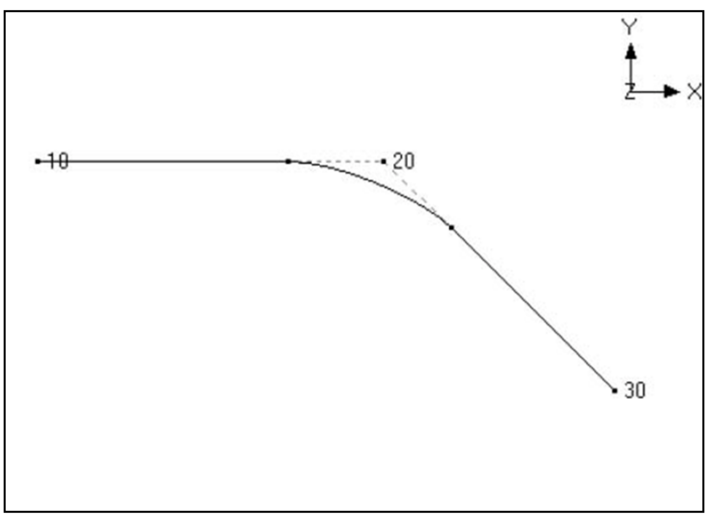
- Press Tab in row #3. Node 20 will be automatically assigned and the cursor will move to the Type column, type "B" (for Bend), Tab to DX, type 2. Enter material, section, load and press Enter. The cursor moves to the next row (#4).
- Tab to the DY column. The next Node 30 is automatically assigned. In DY column, type -2 and press Enter. This completes the bend input.



Example 2: 45° Bend and Pipe routing along a smooth curve



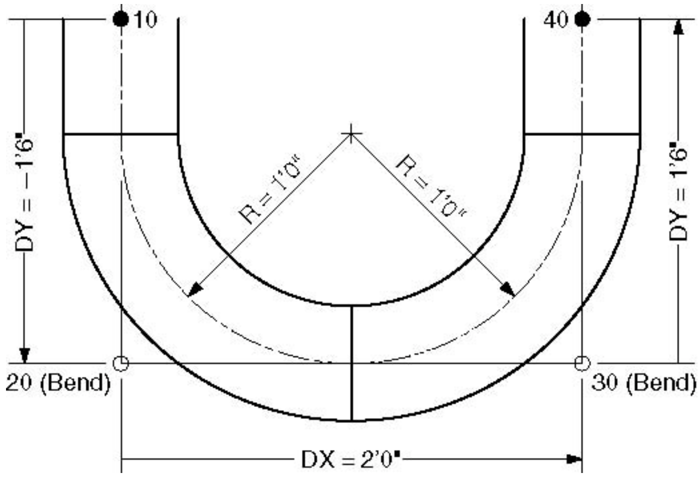
- Press Tab in row #3. Node 20 will be automatically assigned and the cursor will move to the Type column. type "B" (for Bend), Tab to DX, type 1'6". Enter material, section, load and press Enter. The cursor moves to the next row (#4).
- Tab to the DX column. The next Node 30 is automatically assigned. In the DX column, type 1, Tab to DY and type -1, then press Enter. This completes the bend input.



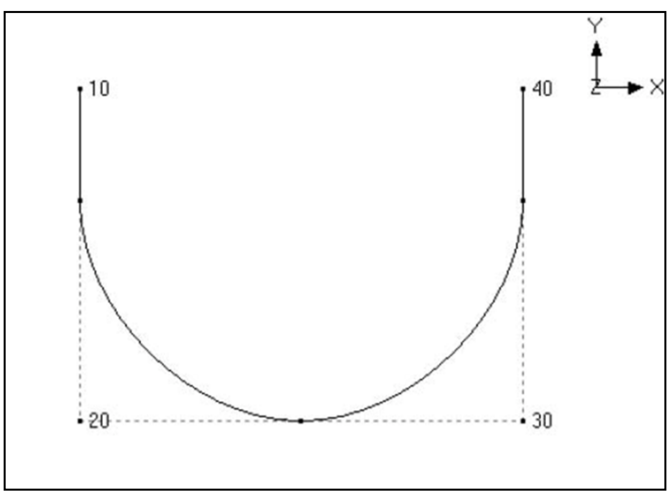
To route a pipe along a smooth curve, you need to split that curve into a number of circular arc segments. Each circular arc segment is then defined as a bend element by giving the offsets (DX, DY and DZ) from the previous Point or Bend Tangent Intersection Point (TIP) to the next TIP, as done for the 45^0 bend above. You continue this process until the routing along the smooth curve is completed.

Example 3: 180° Bend

A 180° bend or U-bend, is often used in an expansion loop to relieve thermal stresses in the piping system. It is modeled as two 90° bends back-to-back.

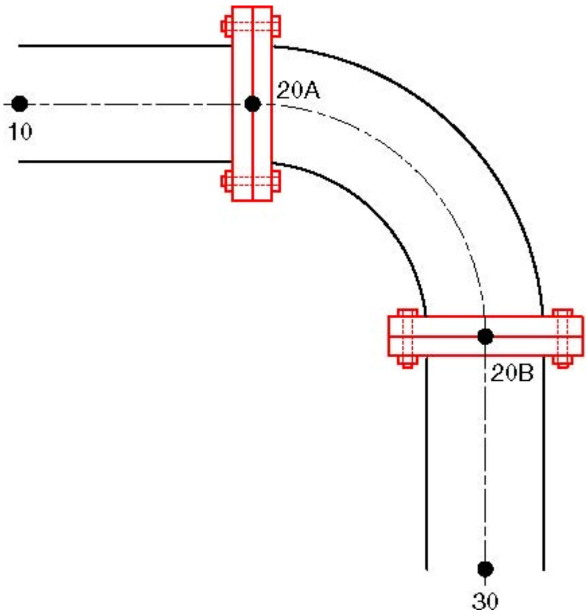


- Press Tab in row #3. Node 20 will be automatically assigned and the cursor will move to the Type column. type "B" (for Bend), Tab to DY, type –1'6". Enter material, section, load and press Enter. The cursor moves to the next row (#4).
- Press Tab. Node 30 will be automatically assigned and the cursor will move to the Type column. type "B" (for Bend), Tab to DX, type 2. (DX is 2' because 8" std long radius bend has 12" radius and since these two bends are back to back, DX = 2R). Press Enter and the cursor moves to the next row (#5).
- Tab to the DY column. The next Node 40 is automatically assigned. In DY column, type 1'6", then press Enter. This completes the bend input.

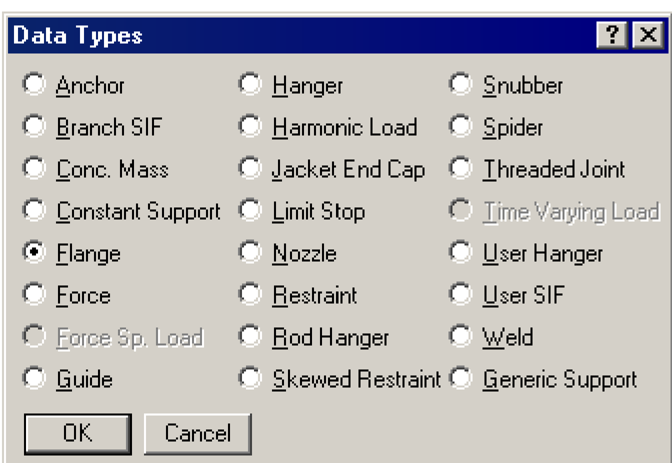


Example 4: Flanged Bend

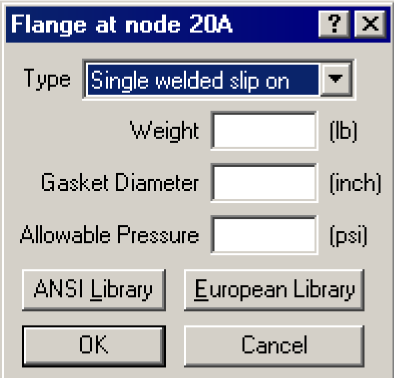
Bends are often connected to the adjacent pipe sections with flanges. A flange may exist on one or both sides of the bend. Flange weight may have a significant effect on the pipe stresses. Also, the stress intensification and flexibility factors for a bend will decrease if one or both of the ends are flanged.



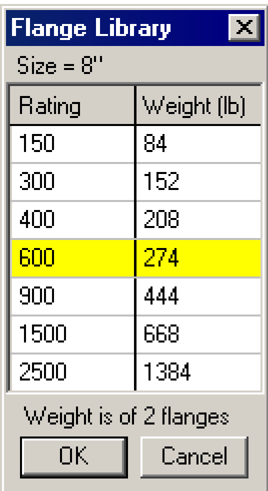
Model the bend as in Example 1. Then input flanges at nodes 20A and 20B. Since these are internally generated nodes, i.e., they do not *normally* appear in the Layout window, it is necessary to specify input at these nodes using the Location type. To input the flange at node 20A, in row #5, type 20A for Node, Tab to Type column and type "L" for Location. This opens the Data Types dialog.



Select Flange as the data type and click on OK. This opens the Flange dialog.



Select the Type of the flange from the drop-down combo box, e.g., Single welded slip-on flange. To get the weight of the flange, click on the Library button.

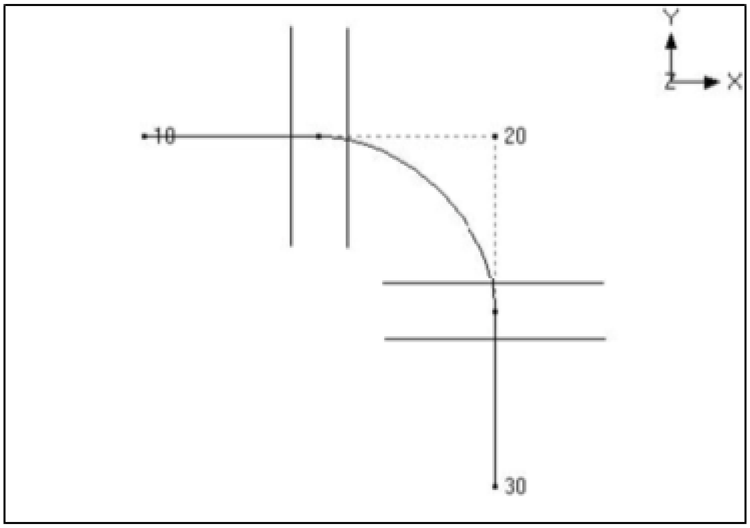


Select the pressure rating for the flange (e.g., 600) and press Enter. The weight of the flange is automatically entered in the Flange dialog. Press Enter again to input the flange.

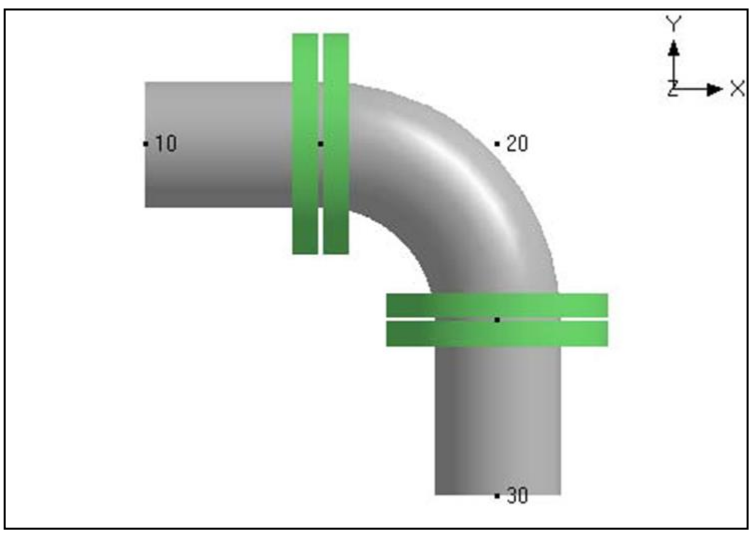
Repeat the same procedure for the flange at node 20B.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Flanged bend								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			-2'0"		A53	8	1	
5	20A	Location							Flange
6	20B	Location							Flange

The graphics is shown below:

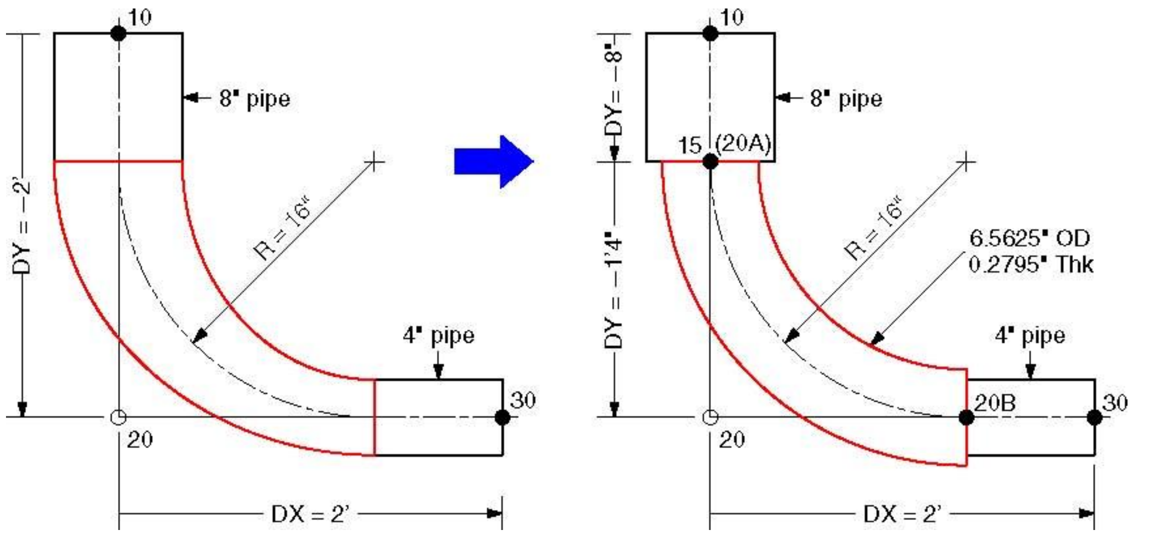


The rendered graphics is shown below:



Example 5: Reducing Bend

CAEPIPE does not have a reducing bend element. A reducing bend may be modeled using an average OD (outside diameter) and average thickness of the large and small ends of the bend. The bend radius of the reducing bend should be input as user bend radius. The Stress Intensification Factor (SIF) of the reducing bend, if available, should be input as Bend SIF.



- The 8" std pipe (OD = 8.625", Thk = 0.322") with Name = 8 is already defined. Now define a 4" std pipe (OD = 4.5", Thk = 0.237") with Name = 4.

The average OD of the two sections is (8.625 + 4.5) / 2 = 6.5625" and the average Thickness is (0.322 + 0.237) / 2 = 0.2795".

Define a "Non Std" section with Name = AVG, OD = 6.5625" and Thickness = 0.2795".

The list of sections is shown below.

#	Name	Nom Dia	Sch	OD (inch)	Thk (inch)	Cor.Al (inch)	M.Tol (%)	Ins.Dens (lb/ft3)	Ins.Thk (inch)
1	8	8"	STD	8.625	0.322				
2	AVG	Non Std		6.5625	0.2795				
3	4	4"	STD	4.5	0.237				

- Note that the section specified on the Bend row in the Layout window applies to the curved portion of the Bend (between the A and B nodes) as well as to the straight portion from the preceding node to the A node. In this case, we want to assign the section "AVG" **only** to the curved portion and assign the section "8" to the straight portion. This can be done by defining an additional node that is coincident with the A node thus making the straight portion of the bend zero length.
- In row #2, the first node (10) is already defined and the cursor is placed in row #3. Type 15 for Node, Tab to DY, type -8". Enter material (1), section (8), load (1) and press Enter. The cursor moves to the next row (#4).
- Press Tab. Node 20 is automatically assigned and the cursor will move to the Type column, type "B" (for Bend), Double click in the Type column to edit the Bend. Click on the User Bend Radius button and type 16 for bend radius. Press Enter to modify the Bend and return to the Layout window. Tab to DY, type -1'4". Tab to the

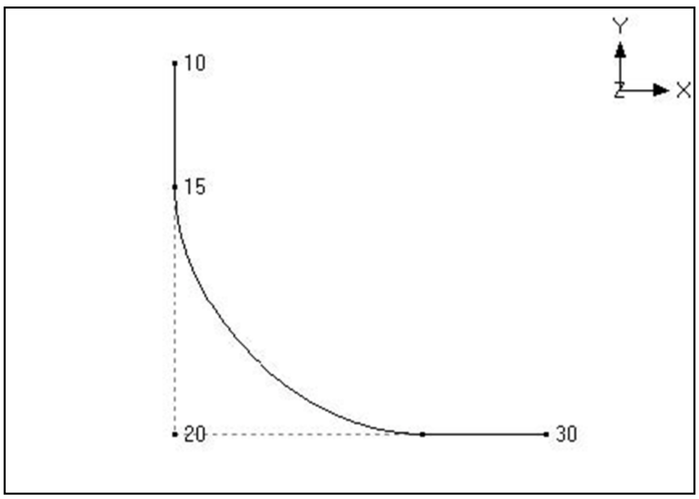
section column and type "AVG". Then press Enter. The material and load are copied from the previous row and the cursor moves to the next row(#5).

- Tab to the DX column. The next Node 30 is automatically assigned. In DX column, type 2'. Tab to section column, type 4 and then press Enter. This completes the reducing bend input.

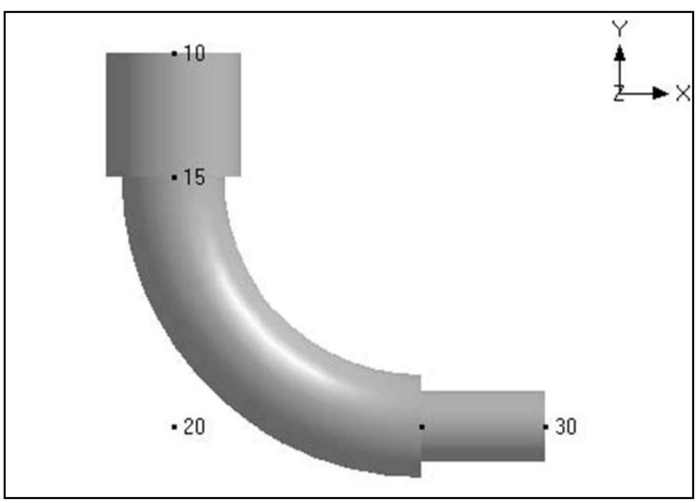
The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'n")	Matl	Sect	Load	Data
1	Title = Reducing bend								
2	10	From							
3	15			-0'8"		A53	8	1	
4	20	Bend		-1'4"		A53	AVG	1	
5	30		2'0"			A53	4	1	

The graphics is shown below:

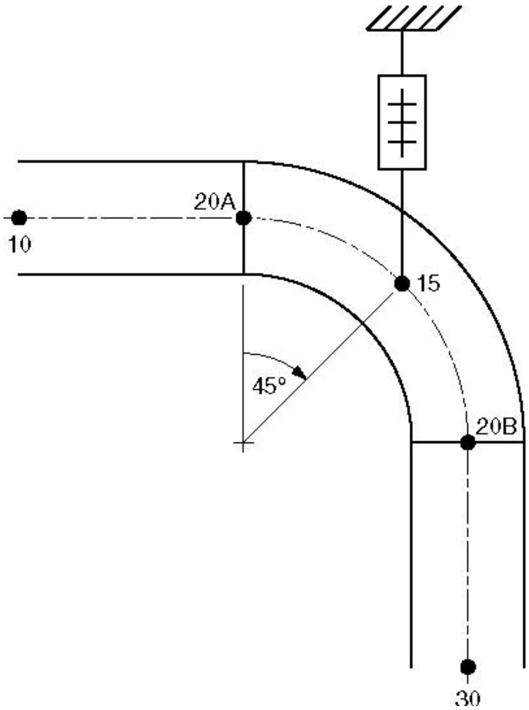


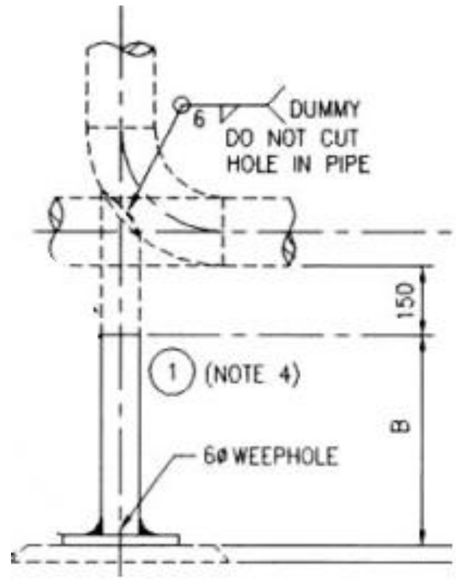
The rendered graphics is shown below:



Example 6: Bend Supported by a Hanger or Resting Support

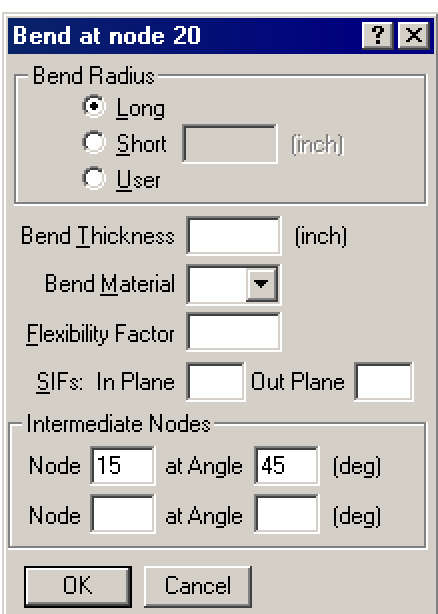
Bend supported in the middle by a hanger or resting support as shown in the figures below can be modeled by defining an intermediate node at Bend.





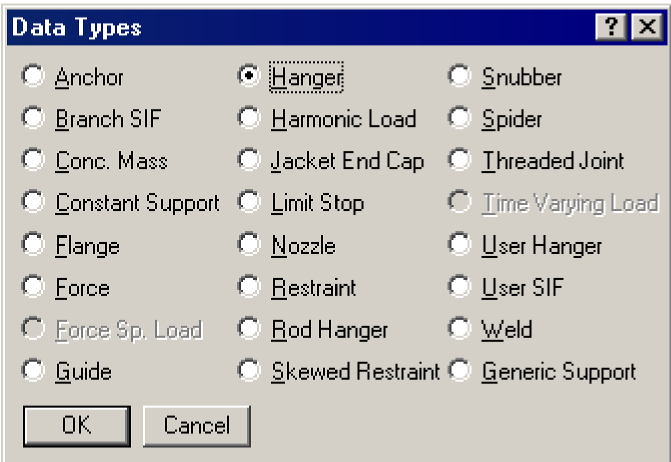
Assuming that the Bend is supported in the middle of the hanger, Model the bend as in Example 1. The hanger is input at node 15, which is in the middle of the bend. Node 15 is created as an intermediate node on the bend as follows:

Double click on the bend (in the type column of the Layout window) to edit it. The bend dialog is shown.

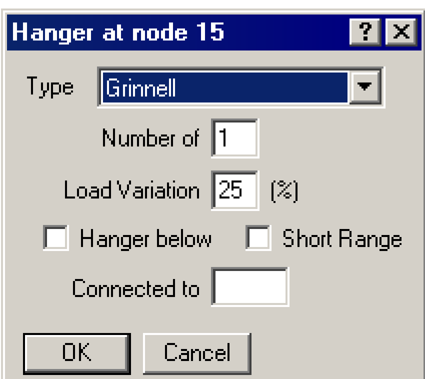


Under intermediate nodes, type 15 for node and 45 for its angle, then click on OK. This creates an intermediate node 15 at 45° from the node 20A (near end of the bend) as shown in the figure above.

Since node 15 does *not* show in the Layout window, it is necessary to specify data input at this node using the Location type. To input the hanger at this node, in row #5, type 15 for Node, Tab to Type column and type "L" for Location. This opens the Data Types dialog.



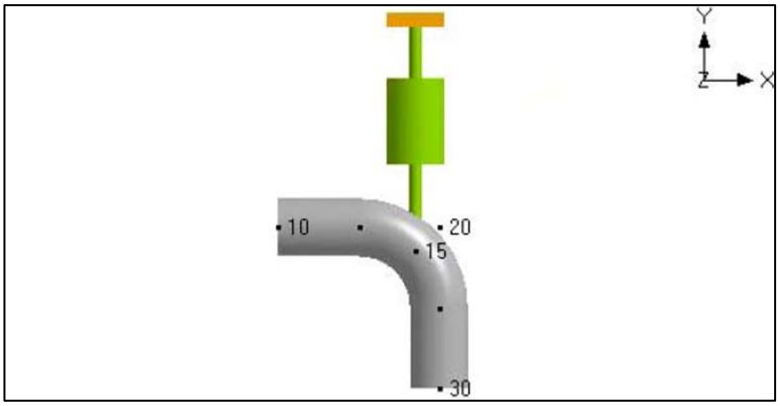
Click on Hanger, the hanger dialog is shown.



Click on OK to accept the default hanger and a hanger is entered at node 15.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Bend supported by a hanger								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			-2'0"		A53	8	1	
5	15	Location							Hanger

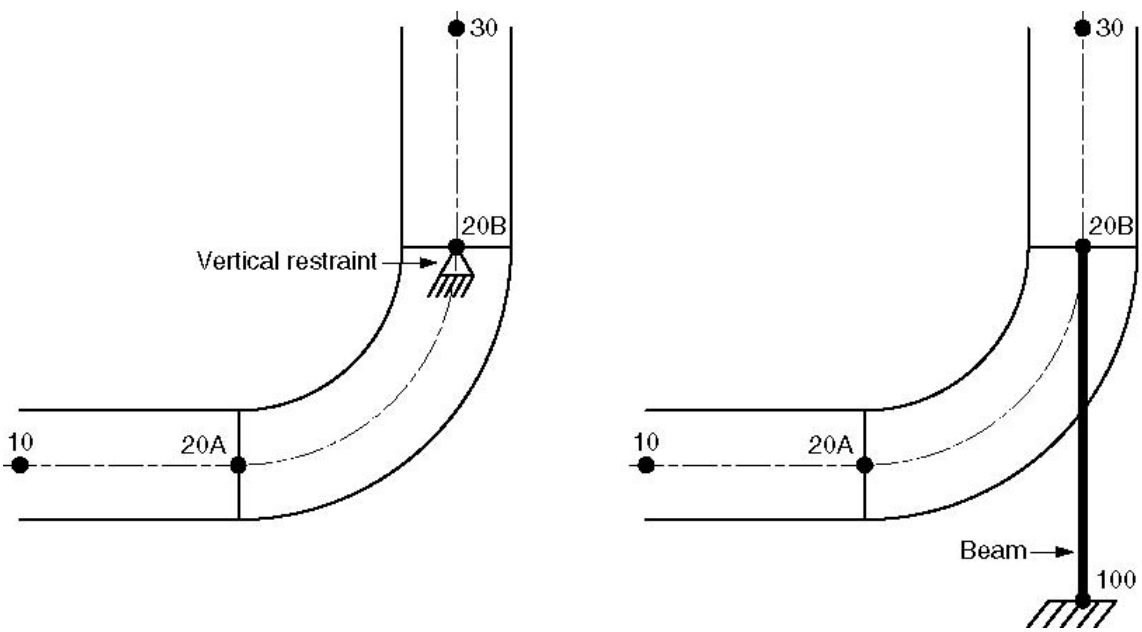
The rendered graphics is shown next.



Follow a similar approach above to model a resting support at the middle of the Bend.

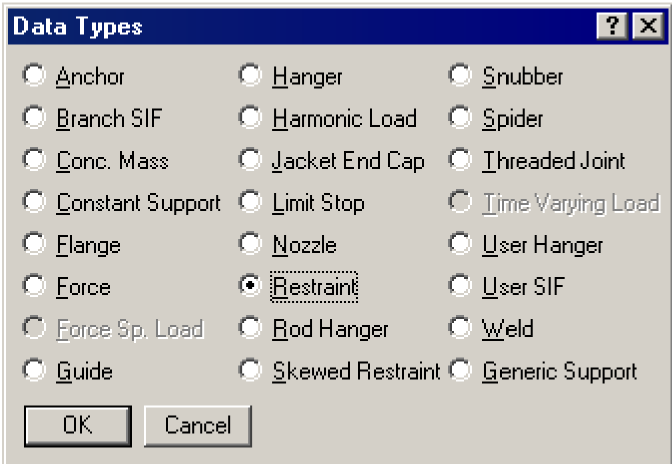
Example 7: Base Supported Bend

Two examples of base supported bends are shown below. In the figure on the left, the support is modeled using a rigid vertical restraint. In the figure on the right the support is modeled using a beam element.

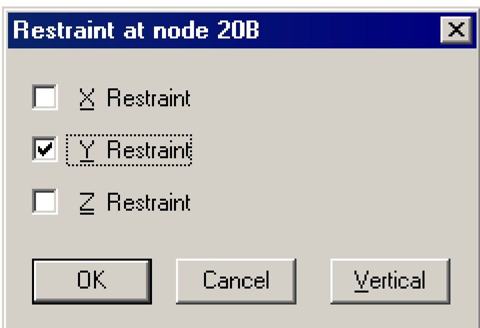


Vertical Restraint Support

- Model the bend at node 20 as before.
- To put a vertical restraint at node 20B, type 20B for node and "L" for Location. This will open the Data types dialog.



Double click on Restraint. This will open the Restraint dialog.

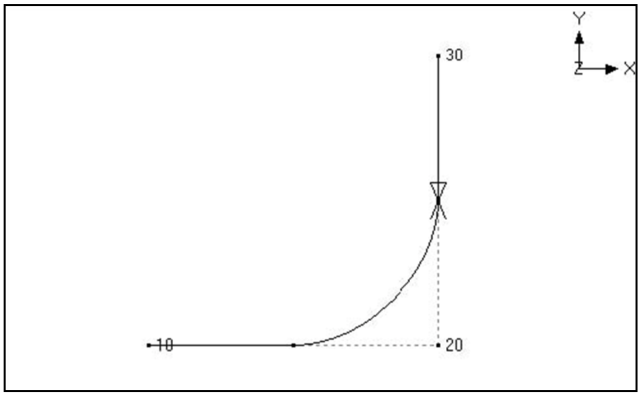


- Click on the Vertical button to check the Y or Z restraint (depending on the vertical axis) and click on OK.

The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in"	Matl	Sect	Load	Data
1	Title = Base support using a vertical restraint								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			2'0"		A53	8	1	
5	20B	Location							Y restraint

The graphics is shown below:



Beam Support

- Model the bend at node 20 as before.
- Create a beam material, section and load as described earlier under the Beam section in the Technical Reference manual.
- Input a beam element from node 20B to node 100.

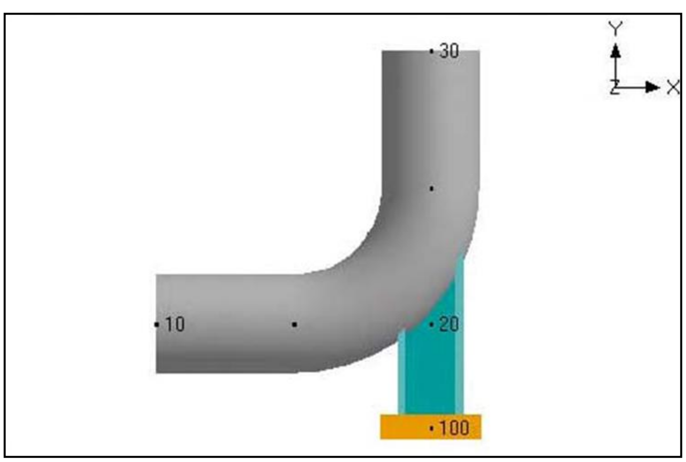
Type 20B in the Node column and "f" (for From) in the Type column to create a starting point. Press Enter to move to the next row.

Type 100 in the Node column and "bea" (for Beam) in the Type column. In the DY column, type the beam length with a negative sign (since the beam is going downward from node 20B to node 100). Type the beam material, beam section and beam load names in the Matl, Sect and Load columns. In the Data column type "a" to input an Anchor.

The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in')	Matl	Sect	Loac	Data
1	Title = Base support using a beam								
2	10	From							
3	20	Bend	2'0"			A53	8	1	
4	30			2'0"		A53	8	1	
5	20B	From							
6	100	Beam		-1'9"		BM1	BS1	BL1	Anchor

The rendered graphics is shown below:



Example 8: Circular Piping

Circular piping around the Nuclear Containment or Tank can be modeled using a series of Bends. Depending upon the number of nodes required to represent the layout and its supports, one can decide the required bend angle.

In this example, Bends with 22.5degrees are chosen to represent the layout. The center line diameter of the Circular pipe layout is 96" and the Section property is 3" Schedule 40.

Origin of the Circular piping is at absolute coordinates Xo=150", Yo=30" and Zo=0.0".

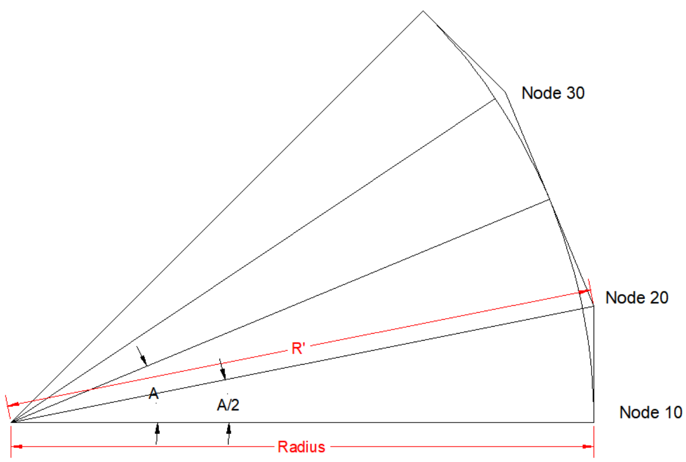
Vertical Axis is chosen as "Z".

With reference to the Origin, absolute coordinate for Node 10 is

X' =150"+Radius (96") = 246"; Y'=30" and Z' =0.0".

Bend Angle chosen (A) = 22.5 deg.

Distance between Origin and Tangent Intersection Point (TIP) = R' = Radius / COS(A/2) = 96/COS(11.25) = 97.88".



Using the above information, absolute coordinates and the offset distances for TIP nodes are computed and presented in the Table below.

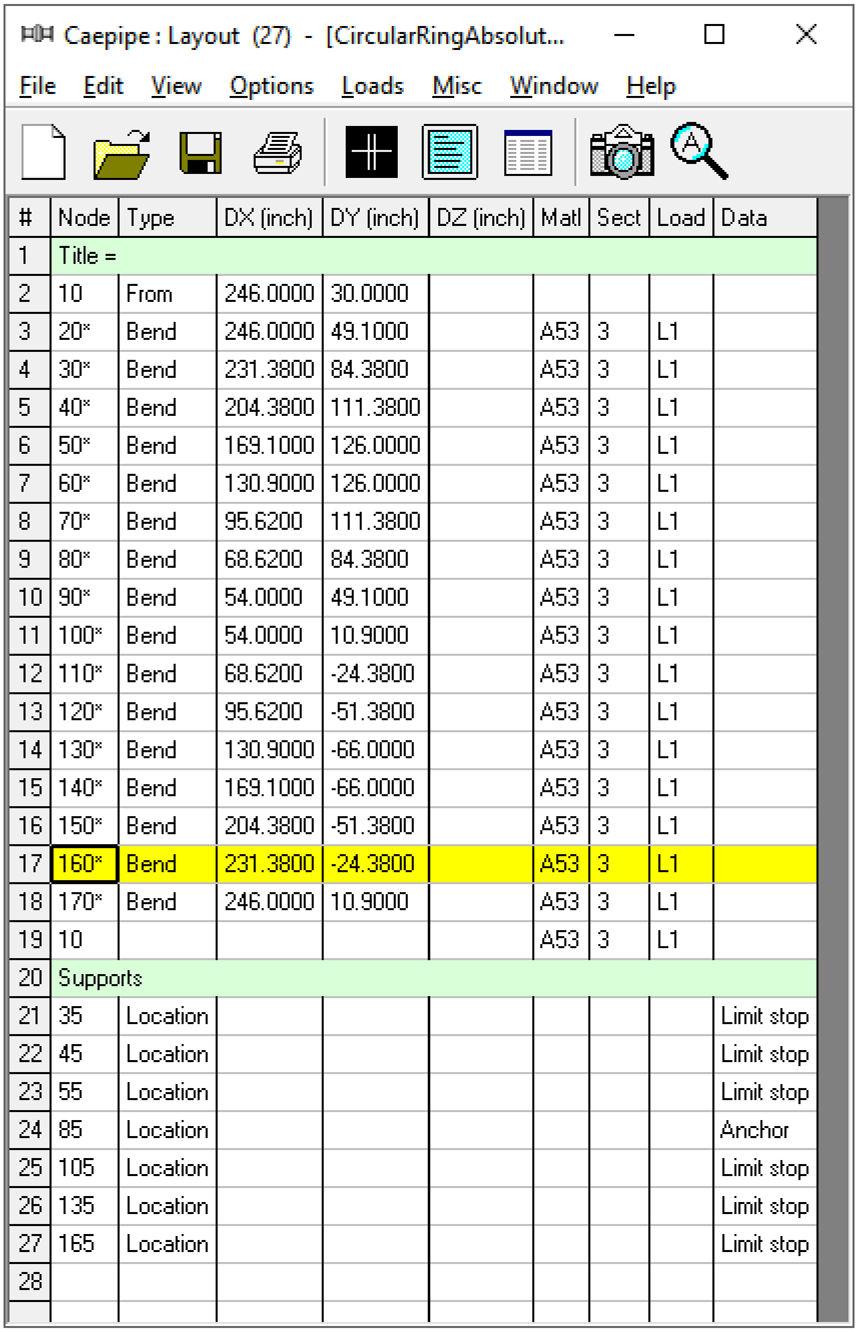
Two layouts are generated.

The first layout is generated using absolute coordinates and hence under the Node column you will see a "*" after the Node number in CAEPIPE layout, meaning that DX, DY and DZ values are not offsets, instead they are absolute coordinates for the Node being input..

The second layout is generated using the Offset Distances listed in the table below.

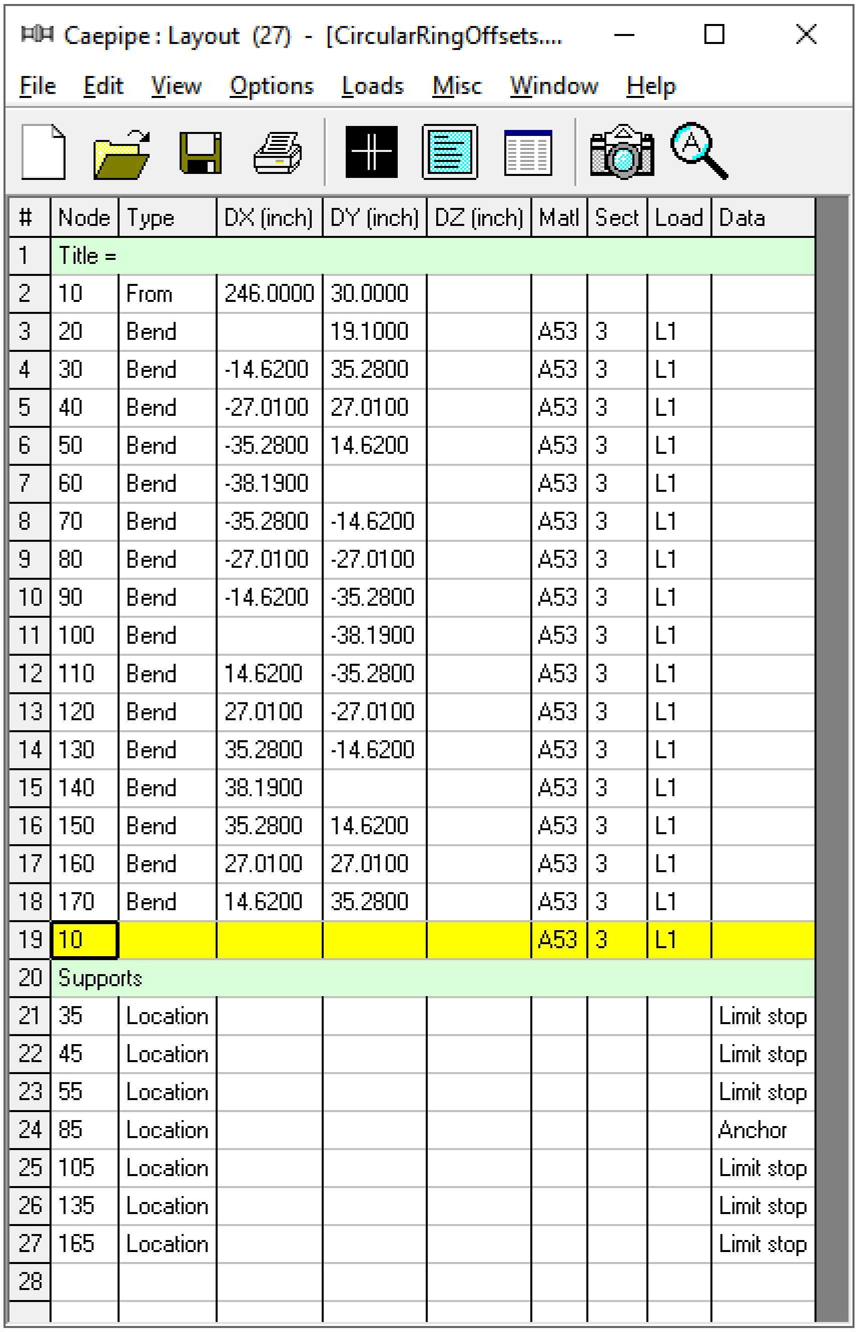
Ring Radius = Bend Radius (in)				96	Bend Angle	22.5
				Xo (in)	Yo (in)	Zo (in)
Absolute Coordinates of Circular Ring Center				150	30	0
				X' (in)	Y' (in)	Z' (in)
Absolute Coordinates for Node 10				246	30	0
			Absolute Coordinates		**Offset Distances**	
TIP Node	**Angle (B) @ Tangent Intersection Point (TIP) (Deg)**	**R' = Distance between Origin and TIP (in)**	**X' = Xo + [COS(B) x R'] (in)**	**Y' = Yo + [SIN(B) x R'] (in)**	**DX (in)**	**DY (in)**
20	11.25	97.88	246.00	49.10	0.00	19.10
30	33.75	97.88	231.38	84.38	-14.62	35.28
40	56.25	97.88	204.38	111.38	-27.01	27.01
50	78.75	97.88	169.10	126.00	-35.28	14.62
60	101.25	97.88	130.90	126.00	-38.19	0.00
70	123.75	97.88	95.62	111.38	-35.28	-14.62
80	146.25	97.88	68.62	84.38	-27.01	-27.01
90	168.75	97.88	54.00	49.10	-14.62	-35.28
100	191.25	97.88	54.00	10.90	0.00	-38.19
110	213.75	97.88	68.62	-24.38	14.62	-35.28
120	236.25	97.88	95.62	-51.38	27.01	-27.01
130	258.75	97.88	130.90	-66.00	35.28	-14.62
140	281.25	97.88	169.10	-66.00	38.19	0.00
150	303.75	97.88	204.38	-51.38	35.28	14.62
160	326.25	97.88	231.38	-24.38	27.01	27.01
170	348.75	97.88	246.00	10.90	14.62	35.28

Layout using Absolute Coordinates

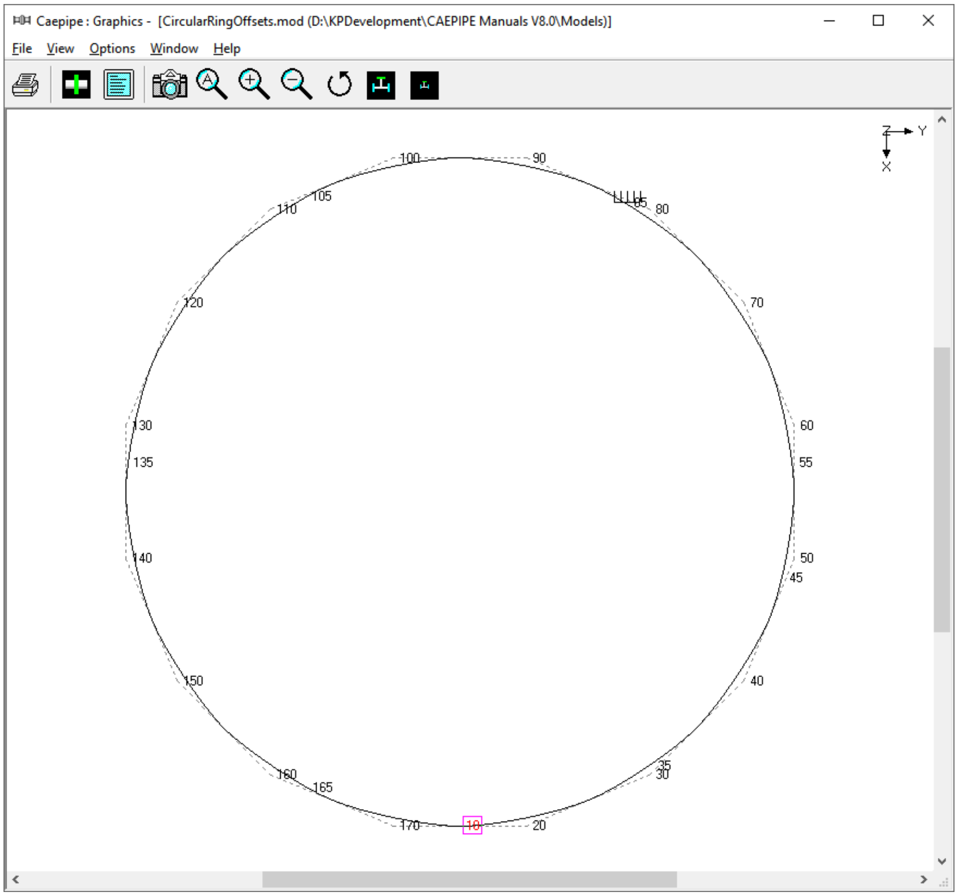


#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title =								
2	10	From	246.0000	30.0000					
3	20*	Bend	246.0000	49.1000		A53	3	L1	
4	30*	Bend	231.3800	84.3800		A53	3	L1	
5	40*	Bend	204.3800	111.3800		A53	3	L1	
6	50*	Bend	169.1000	126.0000		A53	3	L1	
7	60*	Bend	130.9000	126.0000		A53	3	L1	
8	70*	Bend	95.6200	111.3800		A53	3	L1	
9	80*	Bend	68.6200	84.3800		A53	3	L1	
10	90*	Bend	54.0000	49.1000		A53	3	L1	
11	100*	Bend	54.0000	10.9000		A53	3	L1	
12	110*	Bend	68.6200	-24.3800		A53	3	L1	
13	120*	Bend	95.6200	-51.3800		A53	3	L1	
14	130*	Bend	130.9000	-66.0000		A53	3	L1	
15	140*	Bend	169.1000	-66.0000		A53	3	L1	
16	150*	Bend	204.3800	-51.3800		A53	3	L1	
17	160*	Bend	231.3800	-24.3800		A53	3	L1	
18	170*	Bend	246.0000	10.9000		A53	3	L1	
19	10					A53	3	L1	
20	Supports								
21	35	Location							Limit stop
22	45	Location							Limit stop
23	55	Location							Limit stop
24	85	Location							Anchor
25	105	Location							Limit stop
26	135	Location							Limit stop
27	165	Location							Limit stop
28									

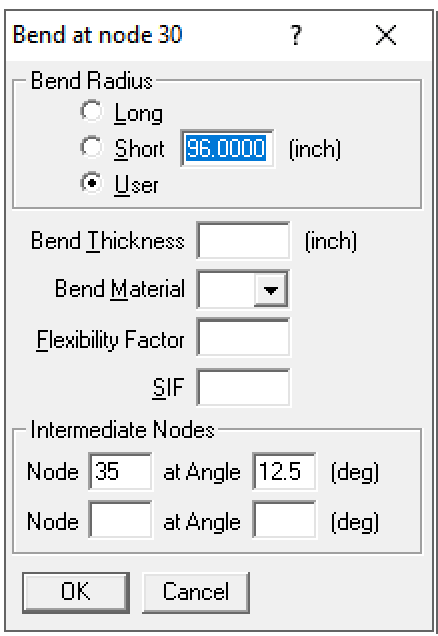
Layout using Offset Distances

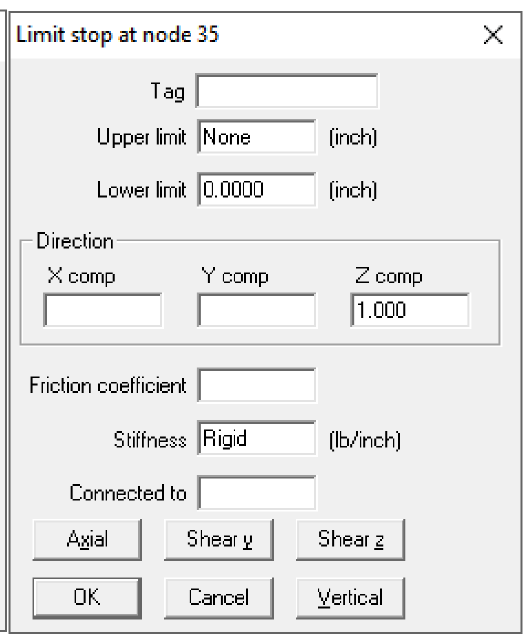


#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title =								
2	10	From	246.0000	30.0000					
3	20	Bend		19.1000		A53	3	L1	
4	30	Bend	-14.6200	35.2800		A53	3	L1	
5	40	Bend	-27.0100	27.0100		A53	3	L1	
6	50	Bend	-35.2800	14.6200		A53	3	L1	
7	60	Bend	-38.1900			A53	3	L1	
8	70	Bend	-35.2800	-14.6200		A53	3	L1	
9	80	Bend	-27.0100	-27.0100		A53	3	L1	
10	90	Bend	-14.6200	-35.2800		A53	3	L1	
11	100	Bend		-38.1900		A53	3	L1	
12	110	Bend	14.6200	-35.2800		A53	3	L1	
13	120	Bend	27.0100	-27.0100		A53	3	L1	
14	130	Bend	35.2800	-14.6200		A53	3	L1	
15	140	Bend	38.1900			A53	3	L1	
16	150	Bend	35.2800	14.6200		A53	3	L1	
17	160	Bend	27.0100	27.0100		A53	3	L1	
18	170	Bend	14.6200	35.2800		A53	3	L1	
19	10					A53	3	L1	
20	Supports								
21	35	Location							Limit stop
22	45	Location							Limit stop
23	55	Location							Limit stop
24	85	Location							Anchor
25	105	Location							Limit stop
26	135	Location							Limit stop
27	165	Location							Limit stop
28									



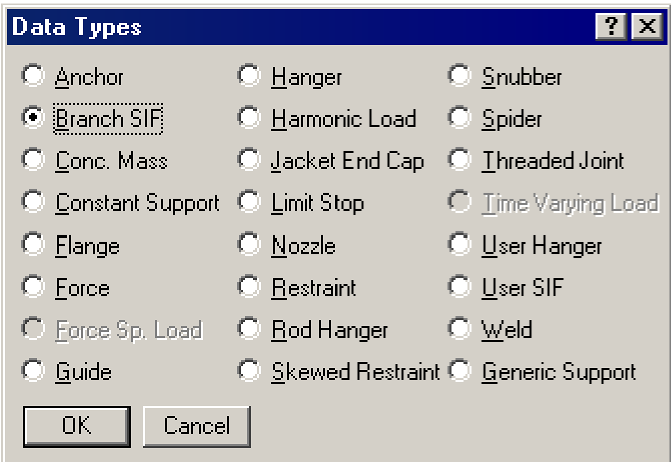
As an example, shown below are the snap shots to split the Bend at Node 30 with an intermediate Node 35 and to place a Limit Stop at that intermediate Node 35.



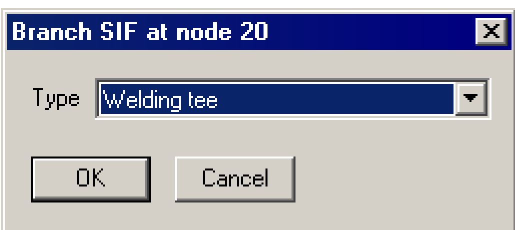


A tee is modeled using three pipes that come together at a node, which should be designated as a "Tee" using the Branch SIF data type. If not, then CAEPIPE cannot calculate a code-specified SIF for the tee to use in stress calculations.

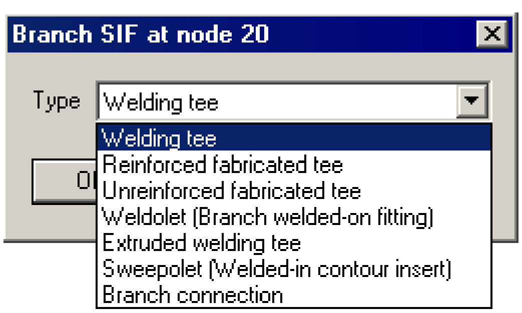
A Stress Intensification Factor (SIF) type for a tee can be input by typing "br" in the Data column or selecting "Branch SIF" from the Data types dialog.



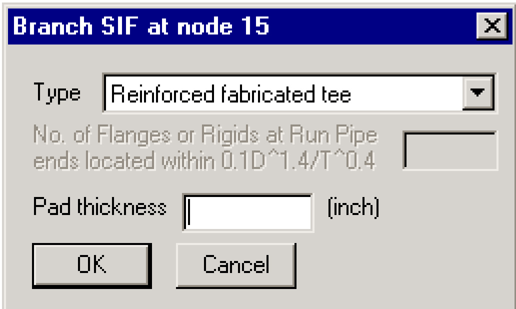
The Branch SIF dialog is shown.



The type of the branch SIF can be selected from the Type drop-down combo box. Depending on the piping code selected, different types of branch SIFs may be available. Typical branch SIF types (for B31.1 piping code) are shown below.



A few branch SIFs may need additional input; for example, in the case of a reinforced fabricated tee, a pad thickness is required.



The field "No. of Flanges or Rigids at Run Pipe ends…" shown above is available only when the Option "Use B31J for SIFs and Flexibility Factors" is turned ON through Layout window > Options > Analysis for B31.x codes. This field can be blank or entered as 1 or 2.

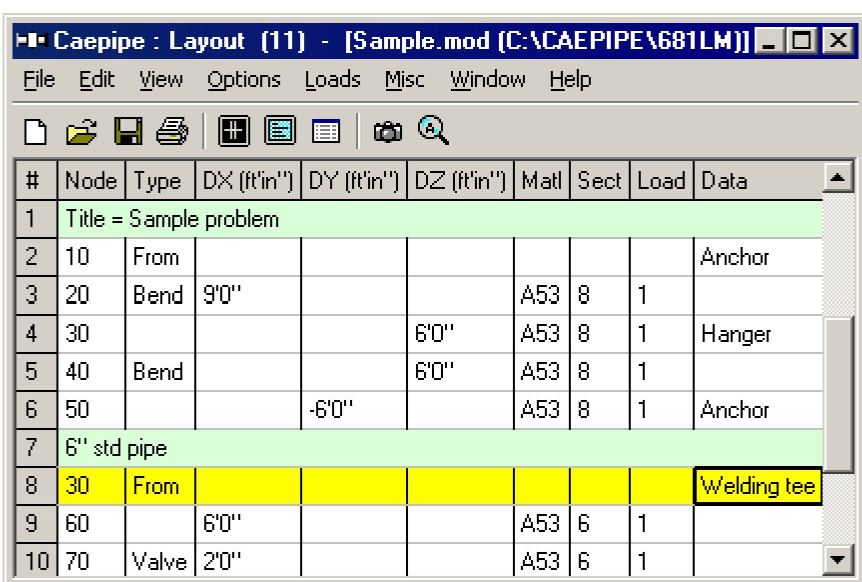
When entered as either 1 or 2, CAEPIPE will multiply the flexibility factors for the Branch by a factor 'c' provided in "Table 1-3 – Flanged End Corrections" of ASME B31J-2017. For more details, see the section titled "ASME B31J-2017" from Code Compliance Manual.

CAEPIPE differentiates between a header (run) and a branch line based on their ODs. So, when CAEPIPE finds two lines with ODs of 8 inches and 6 inches coming together at a node, it designates the 8 inch line as the header (or main) line with the 6 inch line designated as the branch line (this is also how you would model a reducing tee).

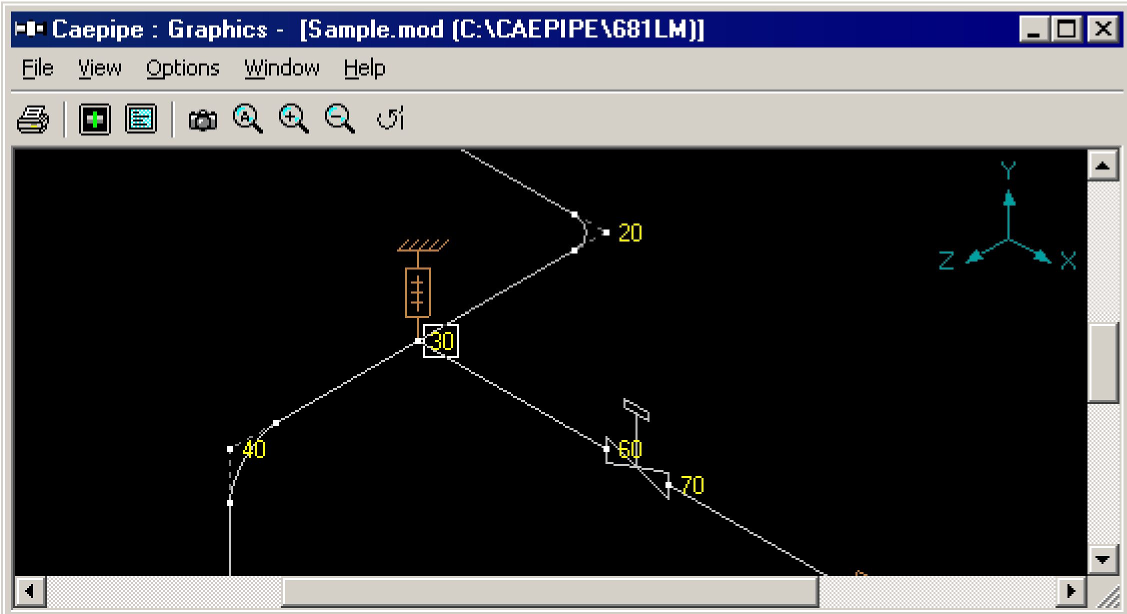
When the header and the branch lines have the same ODs, reduce the branch OD slightly so that the header and the branch lines are properly designated (e.g., OD_{header} = 168.4 mm, OD_{branch} = 168.3 mm).

Examples:

Say you want to model a reducing tee as shown in the provided "Sample.mod". It's an 8"x6" reducing tee. First, we need to model the three pipes – two for the run (nodes 20 to 30 and 30 to 40), one for the branch (nodes 30 to 60), which will be assigned the 6" section. Each pipe section can have its own OD/Thickness. Lastly, designate the common node as a (Butt) Welding Tee.



The Tee is as shown at node 30 in the graphics window below.



A "latrolet" or a lateral tee (commonly, the branch is at a 45° angle from the header) or a Y-joint is modeled like a regular tee (as above). But, the SIF for this joint needs to be input by you after consultation with the tee/joint manufacturer. Use a "User-SIF" data type at the common node to specify the SIF. See topic on Tees for more information.

Soil in Buried piping analysis is modeled by using bilinear restraints with an initial stiffness and an ultimate load. After the ultimate load is reached, the displacement continues without any further increase in load, i.e., the yield stiffness is zero. The initial stiffness is calculated by dividing the ultimate load by the yield displacement which is assumed to be D/25 where D is the outside diameter of the pipe.

Soil modeling is based on Winkler's soil model of infinite, closely spaced elastic springs. Soil stiffness is calculated for all three directions at each node. The pressure value in the load is suitably modified to consider the effect of static overburden soil pressure. The model is analyzed for operating (W+P1+T1) condition and the displacements in the three directions are noted. A check is made for whether skin friction is mobilized and the soil has attained the yield state. If true, then the spring is released in that direction, indicating that soil no longer offers resistance in that direction. This modified model is again analyzed and checked for the yield stage. The iterative process is continued until the percentage difference between displacements at each node for two successive iterations is less than 1%. The final stiffness is the true resistance offered by the soil to the pipe.

General Procedure to model buried piping

1. First, define soils using the command Misc > Soils in the Layout or List window.
2. Next, tie these defined soils with pipe sections (Ctrl+Shft+S to list Sections, double click on an empty row, you will see the field Soil in the bottom right corner. Pick the soil name from the drop-down combo box).
3. Use this modified section for each element on the Layout window that is buried with this soil around it.
4. Discretize long sections of buried piping (Refine Nodal Mesh) through Layout window > Edit > Refine Nodal Mesh > Buried Piping.

   It is at the bends, elbows, and branch connections that the highest stresses are found in buried piping subjected to thermal expansion. These stresses are due to the soil forces that bear against the transverse runs. The stresses are proportional to the amount of soil deformation at the elbows or branch connections. Hence, piping elements adjoining to bends, elbows and branch connections are to be discretized in the stress model.

   In addition, to best simulate Winkler's soil model, it is recommended to discretize even the remaining long straight buried pipe sections in the stress layout

The details of such discretization are explained below.

Refinement of Nodal Mesh for Buried Piping

Modulus of Subgrade Reaction (k)

This factor k defines the resistance of the soil or backfill to pipe movement due to the bearing pressure at the pipe/soil interface. Several methods for calculating modulus of subgrade reaction (k) have been developed in recent years. As per Trautmann, C.H., and O'Rourke, T.D., "Lateral Force-Displacement Response of Buried Pipes," Journal of Geotechnical Engineering, ASCE, Vol. 111, No. 9 Sep 1985, pp. 1077-1092, the modulus of subgrade reaction, k, can be calculated as per Eq. (2) in Appendix VII of ASME B31.1-2018 code.

$$k = C_k N_h w D$$

where,

C_k = a dimensionless factor for estimating horizontal stiffness of compacted backfill. C_k may be estimated at 20 for loose soil, 30 for medium soil, and 80 for dense or compacted soil. *In the current version of CAEPIPE, the value of C_k is internally set as 80 for both cohesive and cohesionless soil.*

D = pipe outside diameter

w = soil density

N_h = a dimensionless horizontal force factor from Fig. 8 of above stated technical paper. For a typical value where the soil internal friction angle is 30 deg. the curve from Fig. 8 may be approximated by a straight line defined by

N_h = 0.285H/D + 4.3

where

H = the depth of pipe below grade at the pipe centerline

Influence Length (L_k)

The influence length is defined as the portion of a transverse pipe run which is deflected or "influenced" by pipe thermal expansion along the axis of the longitudinal run.

From Hetenyi's theory, (*Beams on Elastic Foundation, The University of Michigan Press, Ann Arbor, Michigan 1967*) (also, see Section VII-3.3.2 of Appendix VII of ASME B31.1-2018 code)

$$L_k = \frac{3\pi}{4\beta}$$

where,

Pipe / Soil System Characteristics = $\beta = \left[\frac{k}{4EI}\right]^{1/4}$

E = modulus of elasticity of pipe at reference temperature

I = moment of inertia of pipe cross section

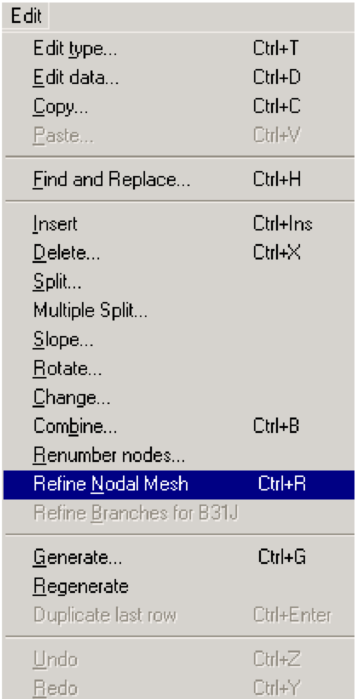
k = modulus of subgrade reaction of soil as detailed above.

Implementation in CAEPIPE

As stated earlier, it is at the bends, elbows, and branch connections that the highest stresses are found in buried piping subjected to thermal expansion of the pipe. These stresses are due to the soil forces that bear against the transverse runs. The stresses are proportional to the amount of soil deformation at the elbows or branch connections. Hence, piping elements adjoining to bends, elbows and branch connections are to be discretized in the stress model.

In addition, to best simulate Winkler's soil model, it is recommended to discretize even the remaining long straight buried pipe sections in the stress layout

This can be performed through Layout window > Edit > Refine Nodal Mesh > Buried Piping.



When the command is selected, CAEPIPE will refine the piping layout as detailed below.

1. Calculate modulus of subgrade reaction (k) as detailed above. While calculating k, the value of C_k is taken as 80 for both cohesive and cohesionless soil.

2. Calculate influence length (L_k) for the element that is fully buried.

3. If the length of the pipe element near bend / elbow / branch connection is greater than or equal to the influence length (L_k), then the pipe element will be split into a number of short elements with length of each short element being equal to 2 x OD of that pipe section until the Influence length (L_k).

4. On the other hand, if the length of the pipe element near bend / elbow / branch connection is less than the influence length (L_k) and greater than 2 x OD of the pipe, then the pipe element will be split into a number of short elements with length of each short element being equal to 2 x OD of that pipe section.

5. If any buried straight pipe element is longer than the influence length (L_k), then the straight pipe element will be split into a number of equal length elements, where the number of such equal length elements will be computed as [(int)(Original Straight Pipe length/Influence Length) + 1]. For example, if the total length of straight pipe element is equal to 1843" and influence length is 400", then the straight pipe will be split into 5 equal length elements [= (int)(1843/400) + 1 = ((int)(4.61) + 1) = (4 + 1)] with each element having a length of 368.6" [= 1843/5].

Note:

While refining the layout, the new node number will be generated by adding the node increment specified (through Layout Window > Options > Node increment) to the available free node number. Hence, set the node increment value as required before refining the buried piping layout.

It is possible to specify different soil characteristics for different portions of the pipe model. Here is how.

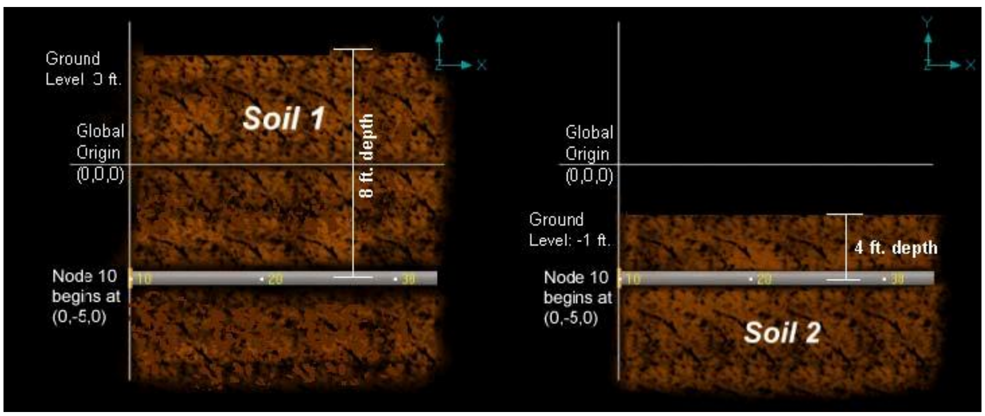
1. Define different soils using the command Misc > Soils.
2. Associate each soil type with those sections that are buried in that soil.
3. Model the buried layout using the different sections for different buried portions.

Ground Level

Ground level for soil is the height of the soil surface from the global origin (height could be positive or negative). It is NOT a measure of the depth of the pipe's centerline.

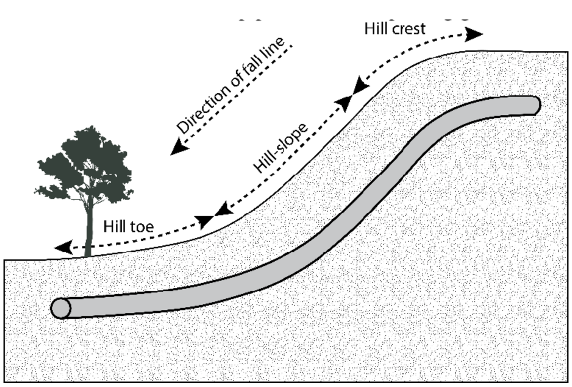
In the figure, the height of the soil surface is 3 feet above the global origin. Pipe node 10 [model origin] is defined at (0,-5,0). So, the pipe is buried 8' (3' - [-5']) deep into the soil. Define similarly for the other soil.

The pipe centerline is calculated by CAEPIPE from the given data



Depth of Soil above Pipe's Centerline

When the option "Value entered is Depth of Soil above pipe centerline" is turned ON in Soil input then CAEPIPE will compute maximum soil loads for the sections buried using the Depth entered. This option will be helpful for modeling pipes that are running up or down a hill with same depth of soil filled above pipe's centerline as shown in the figure given below.



Warning:

Assign Soil only to those elements that are really buried in soil when the option "Value entered is Depth of Soil above pipe centerline" is turned ON.

Two Soil types

Two types of soils can be defined - Cohesive and Cohesionless. Soil density and Ground level are input for both cohesive and cohesionless soils. The Ground level is used to calculate the depth of the buried section. For cohesive soil, Strength is the un-drained cohesive strength (Cs). For cohesionless soil, Delta (δ) is the angle of friction between soil and pipe, and Ks is the Coefficient of horizontal soil stress. See the nomenclature below for more information.

Highlight buried sections of the model in graphics

If your model contains sections that are above ground and buried, then you can selectively see only the buried sections of piping in CAEPIPE graphics by highlighting the section that is tied to the soil. Use the Highlight feature under the Section List window and place highlight on the buried piping section (see Highlight under List window>View menu, or press Ctrl+H). The Graphics window should highlight only that portion of the model that is using that specific section/soil.

Nomenclature

When the option "Include Insulation Thickness" in "Soil" input is turned OFF, then

D = Outside diameter of the pipe

When the option "Include Insulation Thickness" in "Soil" input is turned ON, then

D = Outside diameter of pipe + (2.0 x Insulation Thickness)

Ks = Coefficient of horizontal soil stress, which depends on the relative density and state of consolidation of soil. Ks is empirical in nature and may be estimated from Nq/50. Ks can vary depending on the compaction of the soil from 0.25 (for loose soil) to 1.0 (really compacted soil).

Nq = Bearing capacity factor = $0.98414e^{(0.107311\phi)}$

$\phi = \delta + 5°$

δ = angle of friction between soil and pipe
Normal values for delta ranges between 25° – 45° (for sand).
Clean granular sand is 30° . With a mix of silt in it, the angle is 25°

Sp = soil pressure = soil density $\times$ depth

Cs = Undrained cohesive strength (input for cohesive soil), (Cs in kN/m2) $\geq$1.0

Af = Adhesion factor = $1.7012775\ e^{(-0.00833699\ Cs)}$

kp = Coefficient of passive earth pressure
= $(1 + \sin\phi) / (1 - \sin\phi)$

bottom depth = depth + D/2
top depth = depth – D/2

Nr = $(Nq - 1.0) \tan(1.4\phi)$

dq = dr = 1.0 + 0.1 tan (π/4 + ϕ/2) × depth / D, for δ> 10° , otherwise dq = dr = 1.0

Calculation of Ultimate Loads

The ultimate loads (per unit length of pipe for axial and transverse directions and per unit projected length of pipe for vertical direction) are calculated as shown below.

Different equations are used for cohesive (clayey) and cohesionless (sandy) soils.

Axial direction

Cohesive soil: Axial load = π × D × Af × Cs
Cohesionless soil: Axial load = π× D × Ks × Sp × tan δ

Transverse direction

Cohesive soil: Transverse load = D × (2 Cs + Sp + 1.5 Cs × depth / D)
Cohesionless soil: Transverse load = kp × kp × Sp × D

Vertically downward direction

Cohesive soil: Downward load = D × (5.7182 Cs + Soil density × bottom depth)
Cohesionless soil: Downward load = D × (Soil density × bottom depth × Nq × dq + 0.5 Soil density × D × Nr × dr)

Vertically upward direction

Cohesive soil: Upward load = D × Soil density × top depth + 2 Cs × top depth
Cohesionless soil: Upward load = D × Soil density × top depth

Buried Piping Example

Ultimate Loads and Stiffnesses computed by CAEPIPE for this example are verified later in this section.

Example data:

A 12" Std pipe 6' long is buried, 3' in cohesionless and 3' in cohesive soils with No Insulation properties

Soil properties are as follows:

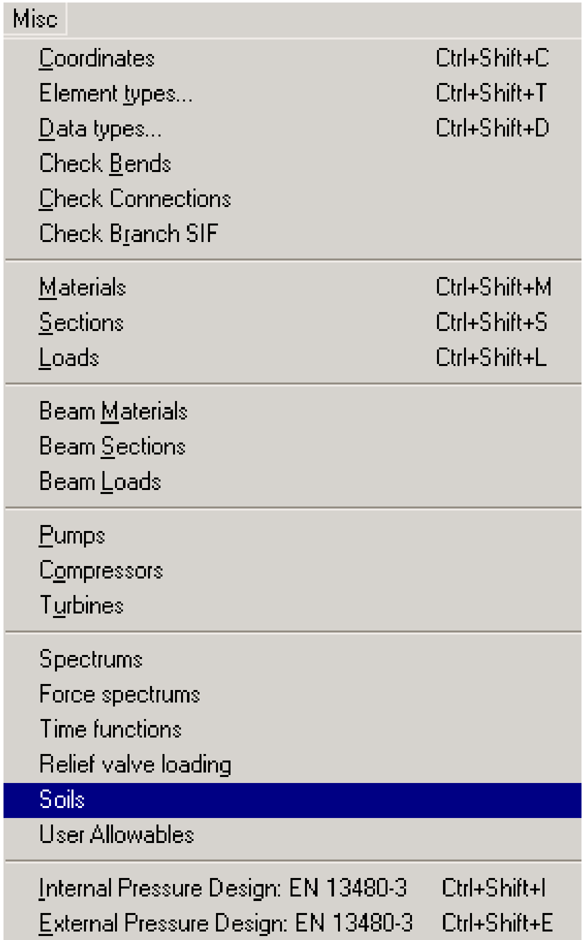
Cohesionless (Name of soil: S1, associated with pipe section 12A):

Density = 120 lbf / ft3
Delta (δ) = 20°
Ks = 0.29 (calculated from Nq/50, where Nq = 14.394)
Ground level = 3'

Cohesive (Name of soil: S2, associated with pipe section 12B):

Density = 150 lbf / ft3
Strength = 100 psi
Ground level = -1'

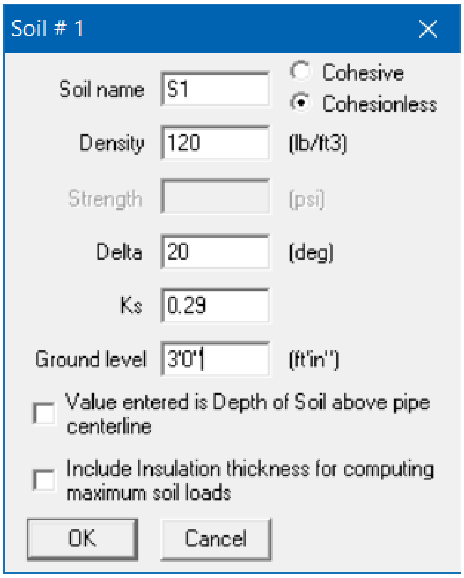
1. Define soils using the command Misc > Soils.



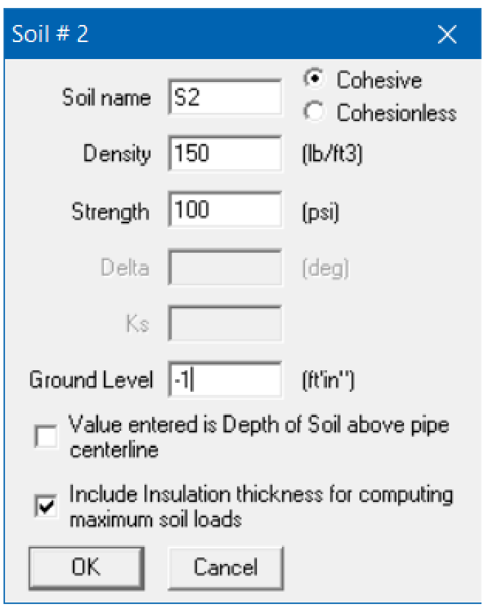
A List window for soils will be displayed. Double click on an empty row to define a new soil.

For our example, define two soils - one cohesionlessand the other cohesive with properties as shown in the following dialogs.

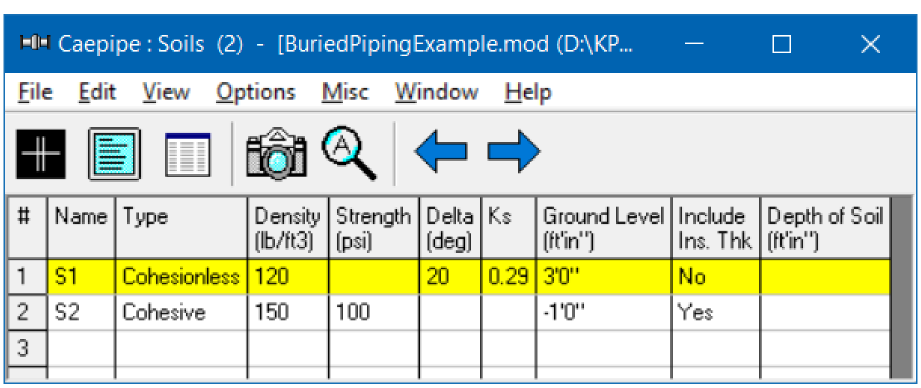
Dialog for cohesionless soil:



Dialog for cohesive soil:



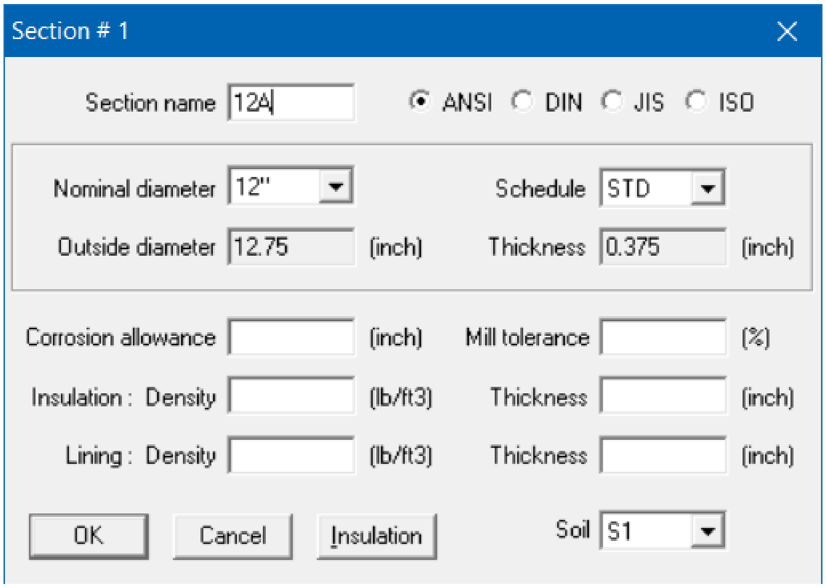
After you have defined the soils, you should see the two soils listed in the List window.



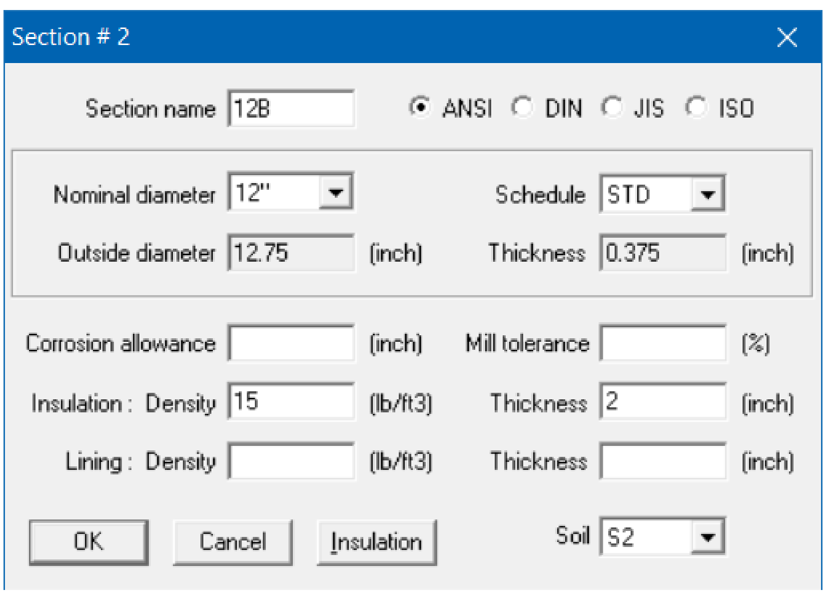
2. Define pipe sections and then associate the soils with these sections.

Define two pipe sections, both 12"/STD pipe sections (name them 12A and 12B), and select the correct soil in the pipe section dialog box using the Soil drop-down combo box.

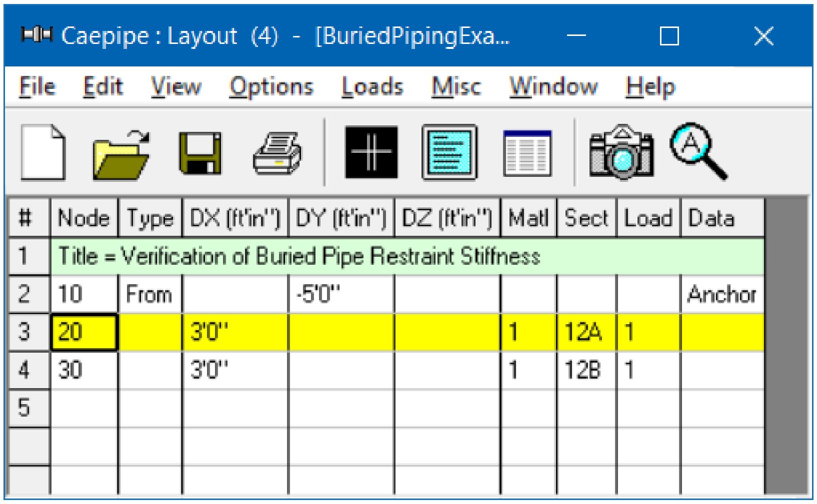
Soil S1 is associated with section 12A:



Soil S2 is associated with section 12B:

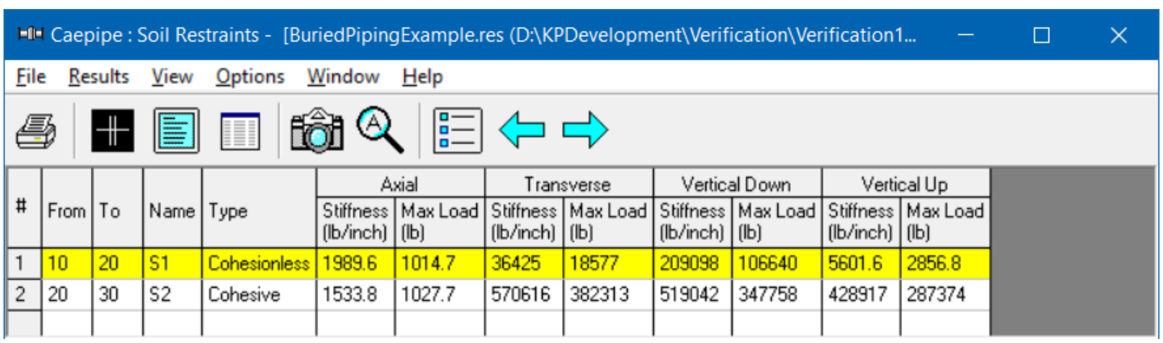


3. Define the layout from 10 to 20 to 30; the first pipe element from 10 to 20 uses section 12A (Cohesionless soil type S1), and the next pipe element 20 to 30 uses section 12B (Cohesive soil type S2). Check Operating load case under Loads menu > Load cases for analysis.



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Verification of Buried Pipe Restraint Stiffness								
2	10	From		-5'0"					Anchor
3	20		3'0"			1	12A	1	
4	30		3'0"			1	12B	1	
5									

Save the model and analyze. Choose yes to view the results. From the Results dialog, pick Soil Restraints. The soil loads and soil stiffnessesin different directions as computed by CAEPIPE are now shown, which are verified below in this section.



#	From	To	Name	Type	Axial		Transverse		Vertical Down		Vertical Up	
					Stiffness (lb/inch)	Max Load (lb)	Stiffness (lb/inch)	Max Load (lb)	Stiffness (lb/inch)	Max Load (lb)	Stiffness (lb/inch)	Max Load (lb)
1	10	20	S1	Cohesionless	1989.6	1014.7	36425	18577	209098	106640	5601.6	2856.8
2	20	30	S2	Cohesive	1533.8	1027.7	570616	382313	519042	347758	428917	287374

Example Verification

Verification of cohesionless restraints (for pipe element 10 to 20)

Sp = soil pressure = soil density × depth
depth = 3' - (-5') = 8' (since the pipe centerline is at -5' and ground level is at 3').
Sp = 120 lb/ft3 × 8 ft = 960 lb / ft2 = 6.6667 lb/in2

Axial direction

Axial load = π× D × Ks × Sp × tan δ
= π ×12.75×0.29×6.6667×tan (20)
= 28.1861 lb/in
= 1014.7 lb (for 36", length of pipe) (CAEPIPE: 1014.7)

Assuming yield displacement = D/25,
Axial stiffness = 25 × 1014.7 / 12.75 = 1989.6 (lb /in) (CAEPIPE: 1989.6)

Transverse direction

$\phi = \delta + 5° = 20° + 5° = 25°$
kp = Coefficient of passive earth pressure
= (1+ sin ϕ) / (1 - sin ϕ)
= 2.4639

Transverse load = kp × kp × Sp × D
= 2.4639 × 2.4639 × 6.6667 × 12.75
= 516.0239 lb /in
= 18576.88 lb (for 36") (CAEPIPE: 18577)

Transverse stiffness = 25 × 18576.88 / 12.75
= 36425 lb / in (CAEPIPE: 36425)

Vertically downward direction

bottom depth = 96" + 12.75"/2 = 102.375"

Nq = Bearing capacity factor = 0.98414 $e^{(0.107311\ \phi)}$ = 14.39366
Nr = (Nq – 1.0) × tan (1.4 ϕ) = 9.37834
since δ> 10°

dq = dr = 1.0 + 0.1 × tan(π/4 + ϕ/2) × depth / D = 2.18188

Downward load = D × (Soil density × bottom depth × Nq × dq + 0.5 × Soil density × D × Nr × dr)
= 12.75 × ((120/1728) × 102.375" × 14.39366 × 2.18188 + 0.5 × (120/1728) × 12.75" ×9.37834×2.18188)
= 12.75 × 232.3305 lb / in
= 106639.7 lb (for 36") (CAEPIPE: 106640 lb)

Downward stiffness = 25 × 106639.7 / 12.75
= 209097 lb/in (CAEPIPE: 209098)

Vertically Upward Direction

top depth = 96" – 12.75"/2 = 89.625"
Upward load = D × Soil density × top depth
= 12.75" × (120/1728) × 89.625
= 79.35547 lb / in
= 2856.7968 lb (for 36") (CAEPIPE: 2856.8)
Upward stiffness = 25 × 2856.7968 / 12.75 = 5601.56 lb/in (CAEPIPE: 5601.6)

Verification of cohesive restraints (for pipe element 20 to 30)

Sp = soil pressure = soil density × depth

depth = –1' – (–5') = 4' (since the pipe centerline is at -5' and ground level is at -1').

Sp = 150 lbf/ft3 × 4 ft = 600 lb / ft2 = 4.16667 lb/in2

D = 12.75" + (2 x 2.0) = 16.75" (as insulation thickness is defined as 2" andthe option "Include Insulation Thickness" is turned ON in Soil S2 input).

Axial direction

Soil strength = Cs = 100 psi = 100 × 6.89476 KN/m^2 = 689.476 KN/m^2

Af = Adhesion factor = 1.7012775 $e^{(-0.00833699\ Cs)}$[where Cs should be in KN/m^2]
= 5.424795E-3

Axial load = π× D × Af × Cs
= π×16.75”×5.424795E-3×100 psi
= 28.54 lb / in
= 1027.66 lb (for 36”) (CAEPIPE: 1027.7)

Axial stiffness = 25 × 1027.66 / 16.75 = 1533.82 lb /in (CAEPIPE: 1533.8 lb/in)

Transverse direction

Transverse load = D × (2 Cs + Sp + 1.5 Cs × depth / D)
= 16.75” × [(2.0×100 + 4.166667 + (1.5×100×48/16.75)]
= 10619.79 lb/in
= 382312.5 lb for 36” (CAEPIPE: 382313)

Transverse stiffness = 25 ×382312.5 / 16.75
= 570615.67 lb /in (CAEPIPE: 570616)

Vertically Downward direction

bottom depth = 48” + 16.75”/2 = 56.375”

Downward load = D × (5.7182 Cs + Soil density × bottom depth)
= 16.75” × [(5.7182×100) + ((150/1728) × 56.375”)]
= 9659.95 lb/in
= 347758.34 lb for 36” (CAEPIPE: 347758)

Downward stiffness = 25 ×347758.34 / 16.75
= 519042.30 lb / in (CAEPIPE: 519042)

Vertically Upward Direction

top depth = 48” – 16.75”/2 = 39.625”

Upward load = D × Soil density × top depth + 2 Cs × top depth
= 16.75” × (150/1728) × 39.625” + (2×100×39.625”)
= 7982.61 lb / in
= 287374.12 lb for 36” (CAEPIPE: 287374)

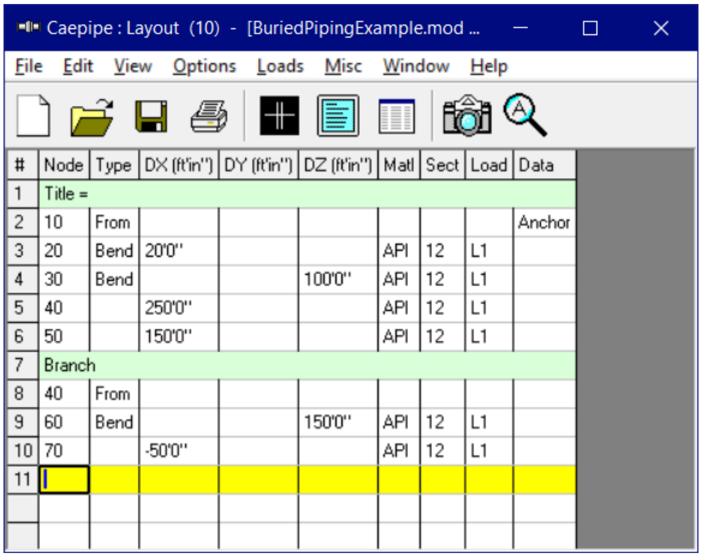
Upward stiffness = 25 ×287374.12 / 16.75
= 428916.60 lb / in (CAEPIPE: 428917)

References

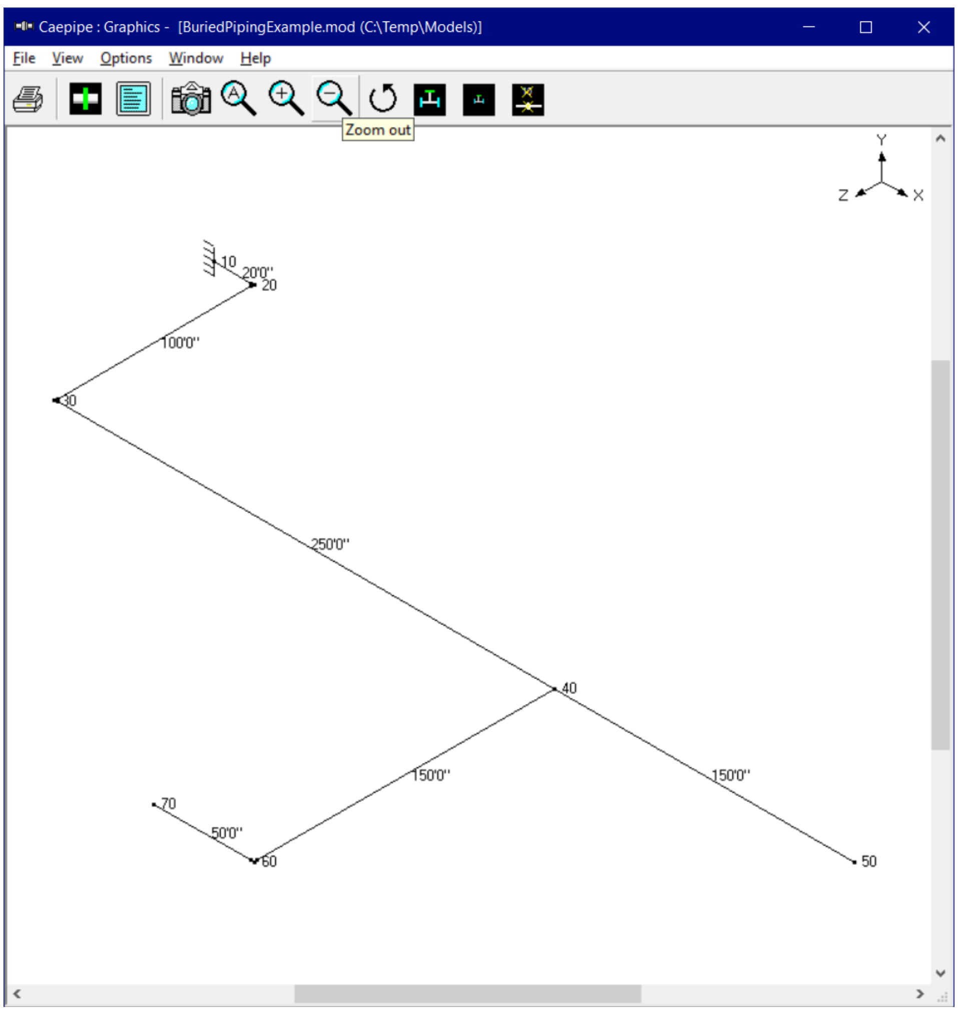
1. Tomlinson, M. J., Pile Design and Construction Practice. Fourth Edition. London: E & FN Spon, 1994.

2. Fleming, W.G.K., et al. Piling Engineering. Second Edition. Blackie Academic and Professional. (Chapters 4 and 5).

Discretization Example


Caepipe : Layout (10) - [BuriedPipingExample.mod ...
File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title =								
2	10	From							Anchor
3	20	Bend	20'0"			API	12	L1	
4	30	Bend			100'0"	API	12	L1	
5	40		250'0"			API	12	L1	
6	50		150'0"			API	12	L1	
7	Branch								
8	40	From							
9	60	Bend			150'0"	API	12	L1	
10	70		-50'0"			API	12	L1	
11									



Soil characteristics

Soil density, w = 130 lbf/ft3 = 0.075 lb/in3

Pipe depth below grade, H = 12 ft (144 in)

Type of backfill, dense sand (cohesionless soil)

C_k = 80

Calculation of Modulus of subgrade reaction (k)

N_h = 0.285H/D + 4.3

N_h = (0.285 x 144 / 12.75) + 4.3 = 7.518

$k = C_k N_h w D$ = 80 x 7.518 x 0.075 x 12.75 = 575.127 psi

Calculation of Influence Length (L_k)

Moment of inertia, I = 279.3 in4

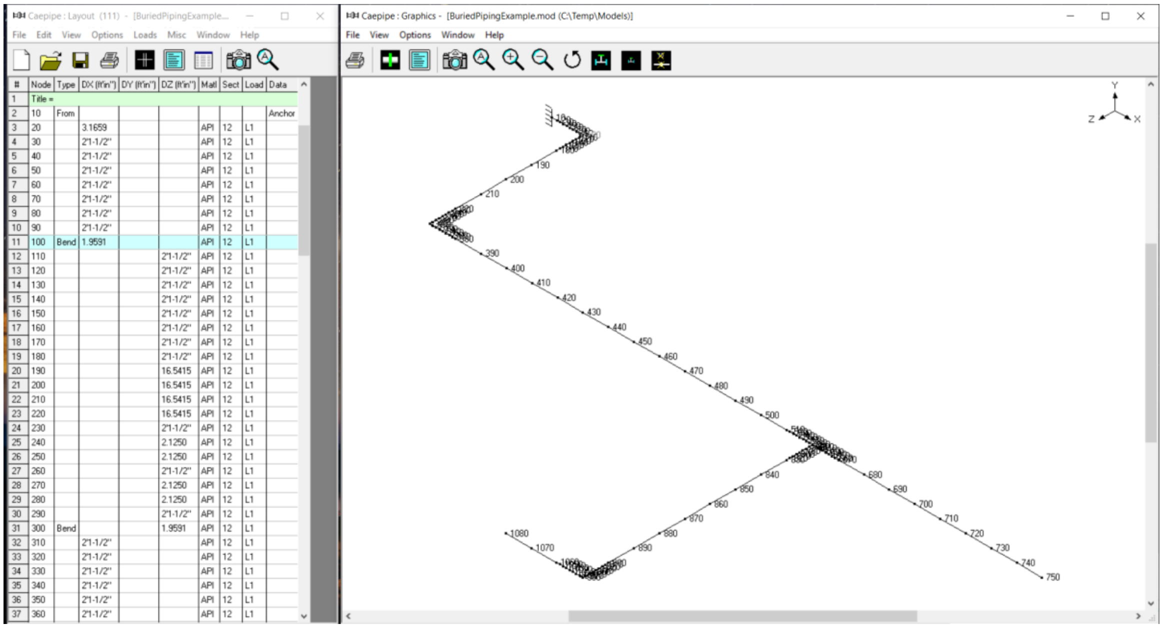
Modulus of elasticity, E = 27.9 x 10^6 psi

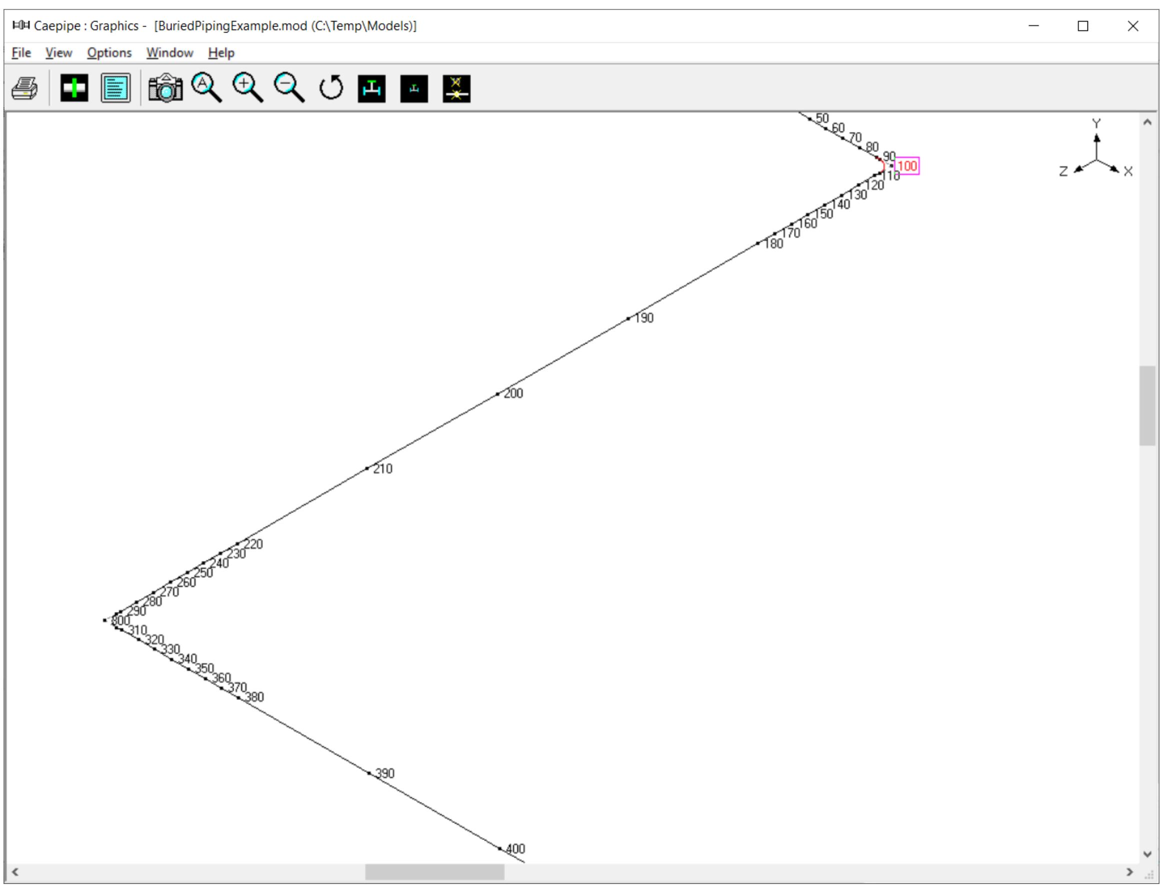
$$L_k = \frac{3\pi}{4\beta}$$

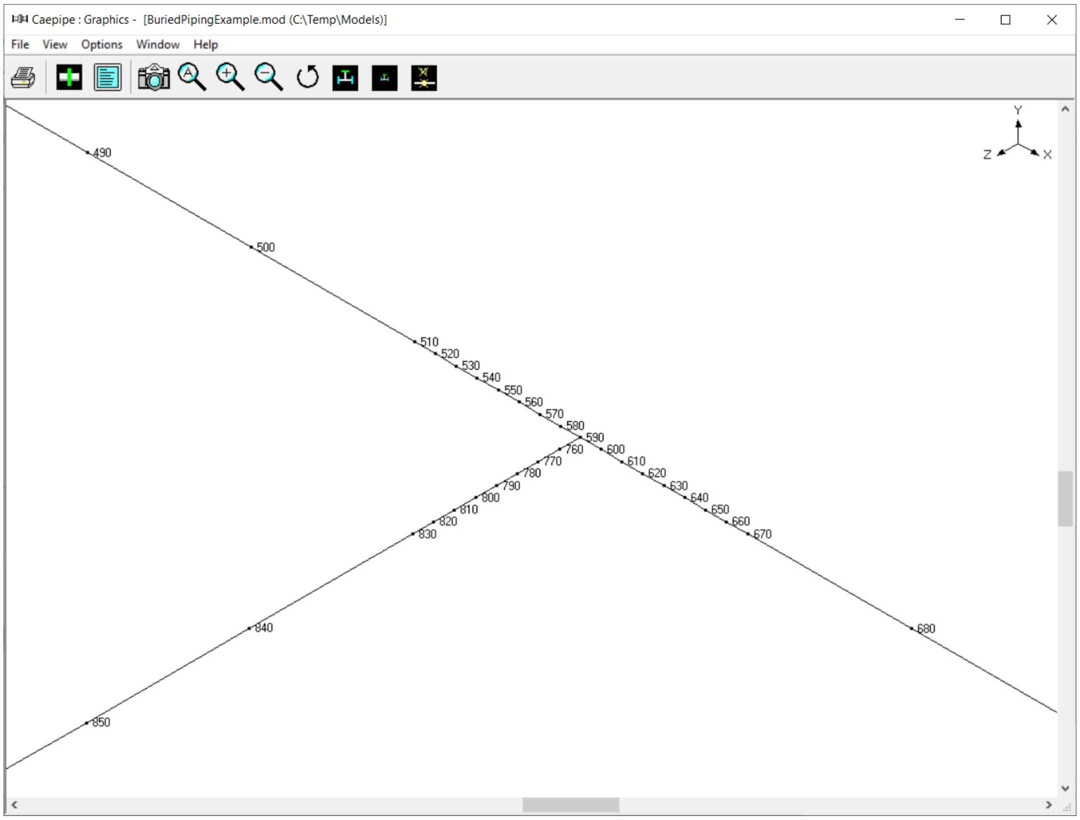
Pipe / Soil System Characteristics = $\beta = \left[\frac{k}{4EI}\right]^{1/4}$ = [575.127 / (4 x 27.9 x 10^6 x 279.3)] $^{1/4}$ = 0.01165

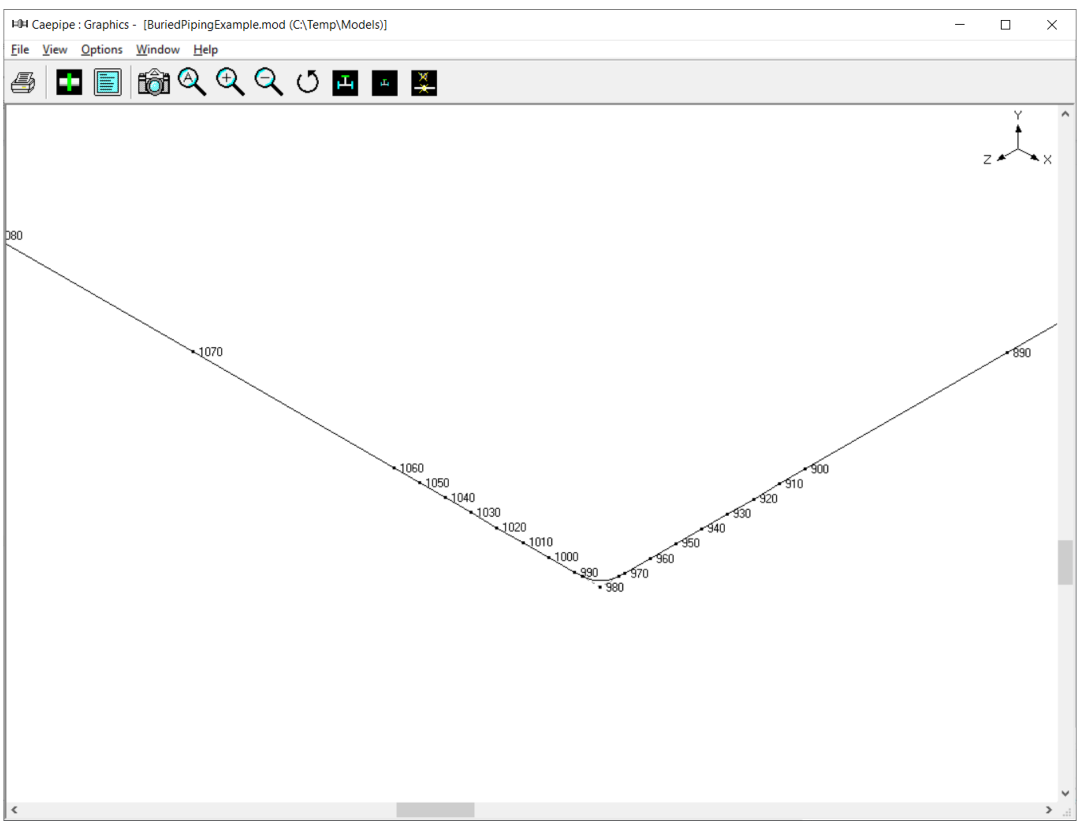
Influence Length (L_k) = 3 x 3.14 / (4 x 0.01165) = 202.145 in

As the lengths of pipe elements near the bends and branch connection are greater than the influence length (L_k = 202.145 in), the pipe elements near the bends and branch connection are split into a number of short elements with length of each short element being equal to 2 x OD = 2 x 12.75 = 25.5 in until the influence length (L_k). See figures given below for details.


Caepipe : Layout (111) - [BuriedPipingExample...
File Edit View Options Loads Misc Window Help
Caepipe : Graphics - [BuriedPipingExample.mod (C:\Temp\Models)]
File View Options Window Help


Caepipe : Graphics - [BuriedPipingExample.mod (C:\Temp\Models)]
File View Options Window Help


Caepipe : Graphics - [BuriedPipingExample.mod (C:\Temp\Models)]
File View Options Window Help
Y
Z
X
490
500
510
520
530
540
550
560
570
580
590
600
610
620
630
640
650
660
670
680
760
770
780
790
800
810
820
830
840
850


Caepipe : Graphics - [BuriedPipingExample.mod (C:\Temp\Models)]
File View Options Window Help
Y
Z
X
1070
1060
1050
1040
1030
1020
1010
1000
990
980
970
960
950
940
930
920
910
900
890

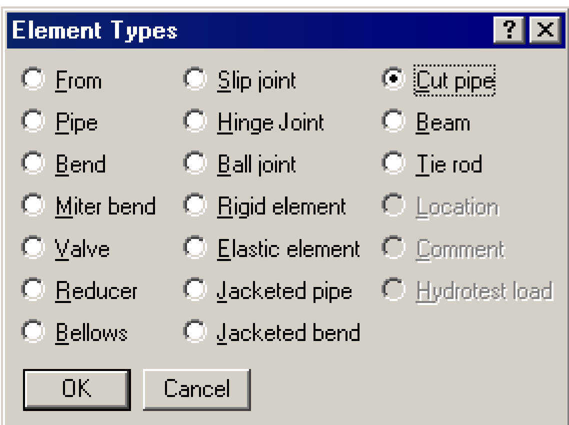
Cold Spring (Cut Pipe)

Cold spring (cut short or cut long) is used to reduce thermal forces on equipment connected to the piping system. When lengths of pipes are cut short or extended by design, they are pulled together or pushed apart to join them during installation, giving rise to a "cold-sprung" system.

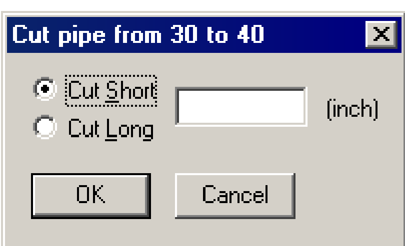
Such an installation process (cold condition) obviously introduces stresses, which are relieved when the system starts up (hot condition). Note however, that the piping codes do not allow credit for any reduction in stresses due to cold spring since the displacement range is unaffected (similar to self-springing. See B31.1 para. 119.2 fordetails). But, codes allow reduction in support loads due to cold spring (which can be helpful at the equipment).

This feature should be used only with a proper understanding of the implications.

Cold spring for a straight pipe is input by typing "c" in the Type column or selecting "Cut pipe" from the Element Types dialog.



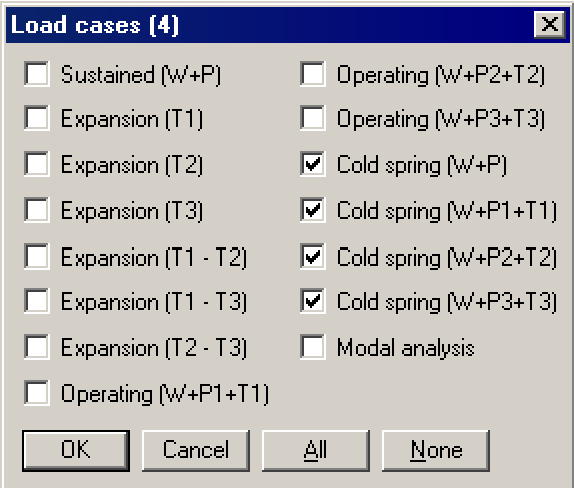
The Cut pipe dialog is shown.



Select "Cut short" or "Cut long" using the radio buttons. The amount of cut (short or long) should be positive.

Since the piping codes do not allow credit for cold spring in stress calculations, a cold spring is used in additional sustained and operating load cases (designated "Cold Spring (W+P), Cold Spring (W+P1+T1)" etc.) which are not used in stress calculations but are used for support loads and rotating equipment reports.

Cold Spring load cases appear in the Loads menu (under Load cases) after a cold spring (Cut pipe element) is input into the model. The Load cases menu is shown next:



For analysis, select the desired Cold Spring load cases from those shown. The built-in Hanger selection procedure does not consider the cold spring since the selection is based on the first Operating (W+P1+T1) load case. However, if Cold Spring is used, the hanger loads for the Cold spring load cases [for example, Cold Spring (W+P1+T1)] will include the effect of the Cold spring.

For an example on Cold Spring modeling and its use in reducing Anchor loads for operating load case(s), refer http://www.sstusa.com/caepipe-tutorials.php.

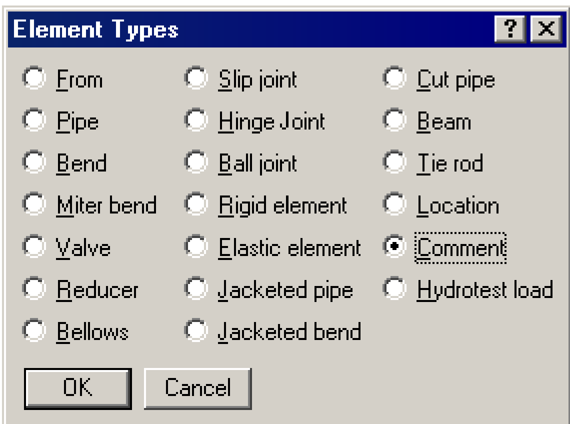
Comment

Just as a computer programmer benefits immensely from clear documentation about the program, so too will an engineer benefit from clear notes, design decisions and comments about a piping system. Use the Comment feature to write as many notes and comments as required anywhere in the CAEPIPE Layout window. They can be printed along with the layout data.

Two ways for putting in a comment:

1. Simply type “c” first (in the Node column) on an empty row, or
2. On an empty row, select “Comment” from the Element Types dialog (Ctrl+Shft+T).

Use menu Edit > Insert (Ctrl+Ins) to insert an empty row between two existing rows of data.



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
10	17			0'3"		CRP	RT	HW	
11	End of reactor vertical shell								
12	...from centerline of Reactor								
13	12	From							
14	60	Rigid	3.2374		7.9481	CRP	RN	HW	
15	...Reactor nozzle A01...								
16	65		0.9116		2.2381	CRP	RN	HW	
17	End of Reactor Nozzle A01...projection to Reactor C.L. is 11'-0"								
18	70		0.3419		0.8393	CSP	RN1	CW	
19	75	Reducer	0.2829		0.6946	CSP	RN1	CW	

Rows 11, 12, 15 and 17 are the comment lines (highlighted with a light green background).

Compressor

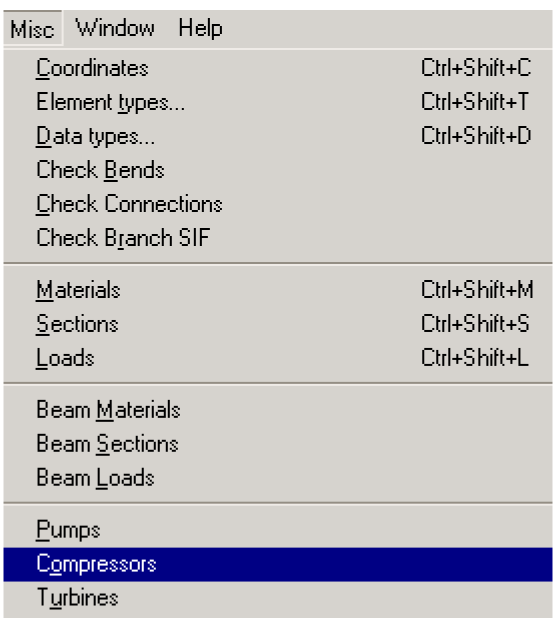
Pumps, compressors and turbines in CAEPIPE, referred to as rotating equipment, are each governed by an industry publication - API (American Petroleum Institute) publishes an API 610 for pumps, ANSI (American National Standards Institute) publishes an ANSI/HI 9.6.2 for Rotodynamic Pumps, API 617 for compressors and NEMA (National Electrical Manufacturers Association) publishes the NEMA SM-23 for turbines. These publications provide guidelines for evaluating nozzles connected to equipment among other technical information including the items relevant to piping stress analysis – criteria for piping design and a table of allowable loads.

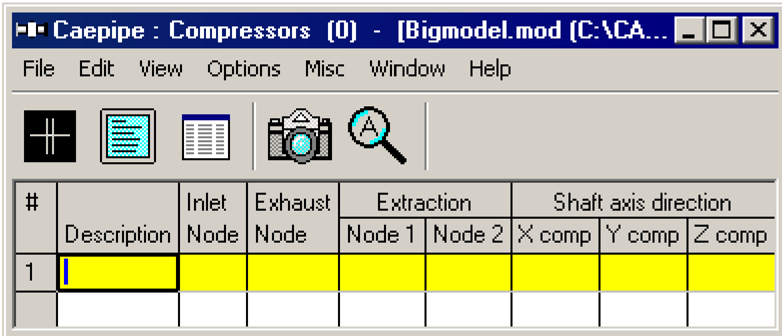
Modeling the equipment is straightforward since it is assumed rigid (relative to connected piping) and modeled only through its end points (connection nozzles).

1. In your model, anchor all the nozzles (on the equipment) that need to be included in the analysis.
2. Specify these anchored nodes during the respective equipment definition via Misc. menu > Pumps/Compressors/Turbines in the Layout window.

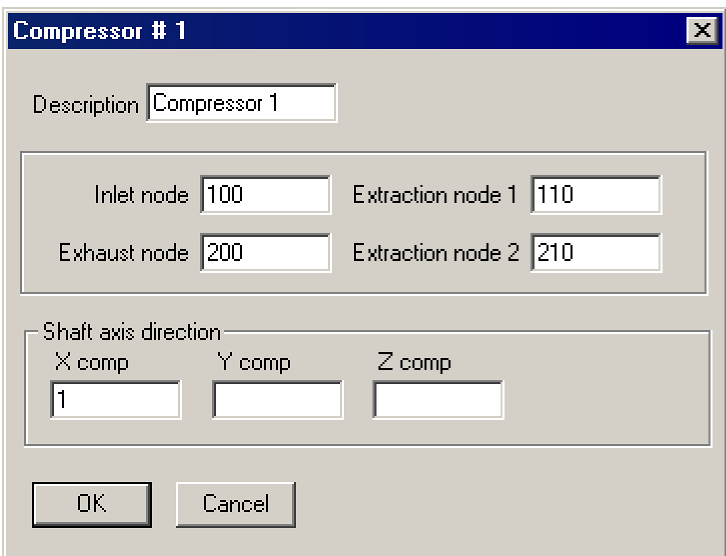
CAEPIPE does not require you to model all the nozzles nor their connected piping. For example, you may model simply one inlet nozzle of a pump with its piping. Or, you may model one pump with both nozzles (with no connected piping) and impose external forces on them (if you have that data). Further, there is no need to connect the two anchors of the equipment with a rigid massless element like required in some archaic methods.

A compressor (like a turbine or a pump) is input by selecting "Compressors" from the Misc menu in the Layout or List window. Upon analysis, an API 617 compressor compliance report is produced. See the section titled "Rotating Equipment Qualification" from Code Compliance Manual for related information.





Once you see the Compressor List window, double click on an empty row for the Compressor dialog and enter the required information.



A short description to identify the compressor may be entered for Description. The nozzle nodes must be anchors and the shaft axis must be in the horizontal plane. Some of the nozzle nodes may be left blank if they are not considered as a part of the piping system being analyzed (e.g., extraction nodes).

Caepipe : Rotating Equipment Report - [p25.res (C:\Documents and Se...

File Results View Options Window Help

API 617 (7th Ed. 2003) report for compressor : P24-API 617

Load case: Operating (W+P1+T1)

Shaft axis: Xcomp = 1.000, Ycomp = 0.000, Zcomp = 0.000

		----- Forces (lb) -----			----- Moments (ft-lb) -----		
Node	Type	fx	fy	fz	mx	my	mz
5	Inlet	567	-301	497	2527	1981	-29
25	Exhaust	-252	-337	-549	-1075	2854	-2830
30	Extr.1	38	-140	-74	133	-200	420

		Size	--- Resultant ---			Allow	
Node	Type	(inch)	F(lb)	M(ft-lb)	3F + M	able	Ratio
5	Inlet	8.000	812	3211	5647	7400	0.763
25	Exhaust	12.000	692	4161	6236	8633	0.722
30	Extr.1	4.000	163	484	972	3700	0.263

Combined resultants at largest connection 25

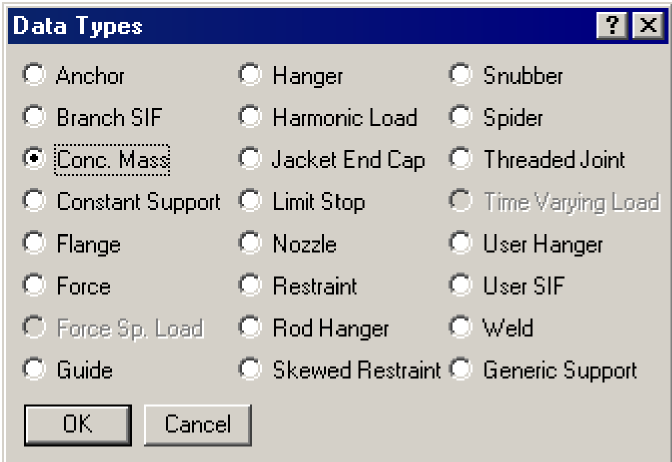
	----- Forces (lb) -----			----- Moments (ft-lb) -----		
	fx	fy	fz	mx	my	mz
Calculated	353	-778	-126	1789	6848	330
Allowable	1016	2541	2033	5082	2541	2541
Ratio	0.347	0.306	0.062	0.352	2.695	0.130

	--- Resultant ---			Allow	
	F(lb)	M(ft-lb)	2F + M	able	Ratio
Combined	864	7086	8813	5082	1.734

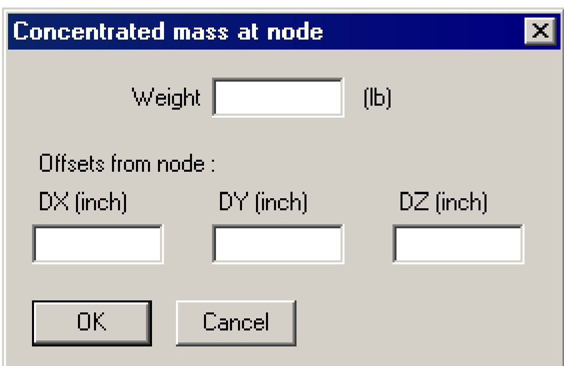
If you have input multiple temperatures, corresponding reports for additional operating load cases are shown.

Concentrated Mass

A concentrated mass is input by typing "conc" in the Data column or selecting "Conc. Mass" from the Data Types dialog.



The Concentrated Mass dialog is shown.



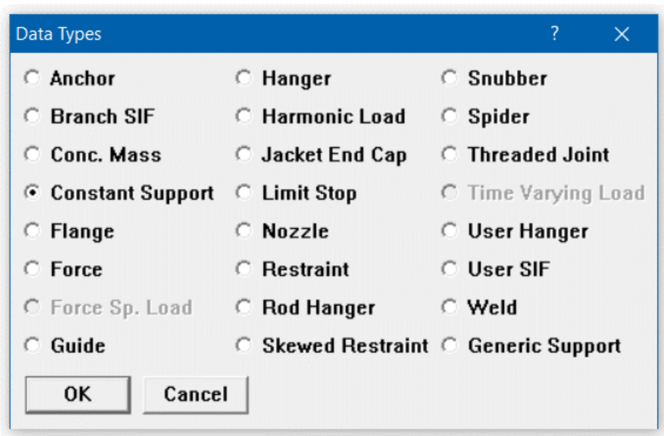
The weight of the concentrated mass should be input for Weight. Weight is to be input in lbf or kgf and NOT in mass units. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

The concentrated mass is located at the offset (DX, DY, DZ) from the node. Deadweight, seismic and dynamic loads due to concentrated masses are applied to the model.

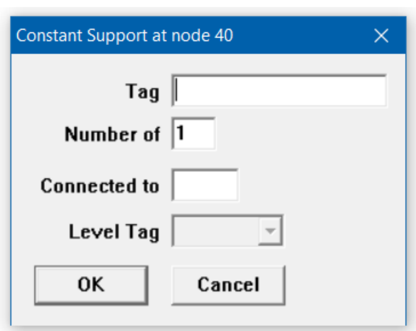
Constant Support

A constant support hanger exerts a constant vertical supporting force on the piping, irrespective of whether the pipe is in hot or cold condition. It is equivalent to a weight pulling up the pipe through a pulley, where the same upward force is exerted on the pipe irrespective of the position of the pipe. The constant support load is automatically calculated by CAEPIPE. To analyze an existing constant support with a known load, input it as a user hanger with a zero spring rate.

A constant support is input by typing "cons" in the Data column or selecting "Constant Support" from the Data Types dialog.



The Constant Support dialog is shown.



Tag

Tag can be 14characters long. Tags are useful in identifying a support while modeling, reviewing of reports and in field erection. Tag Name entered in this field is shown in all reports.

Number of Hangers

The number of hangers is the number of separate hangers connected in parallel at this node.

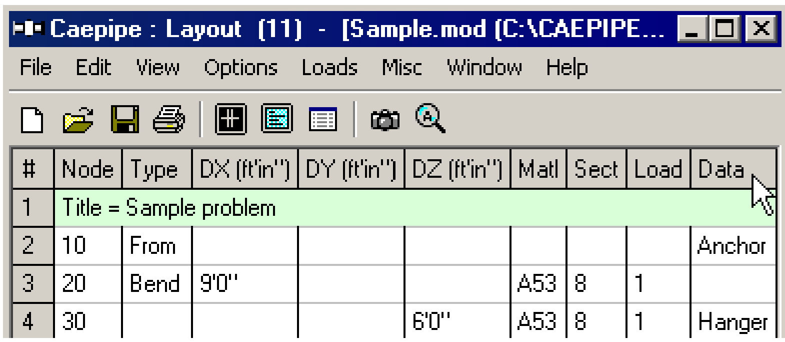
Connected to Node

By default the hanger is connected to a fixed ground point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the "Connected to node" field. This node must be above the hanger node.

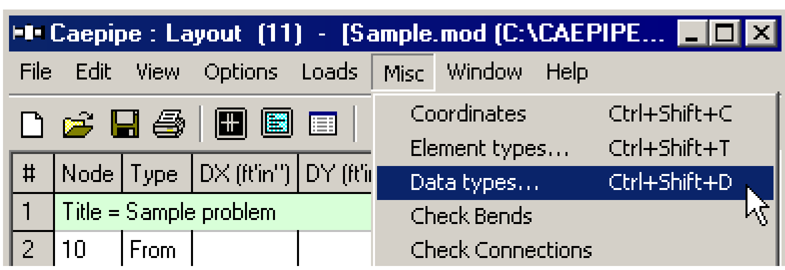
Data Types

Items such as anchors, hangers and external forces, which are defined at nodes, are input in the Data column as Data types; different from inline elements such as pipes, bends and valves that *connect nodes,* and are input in the Type column as Element types.

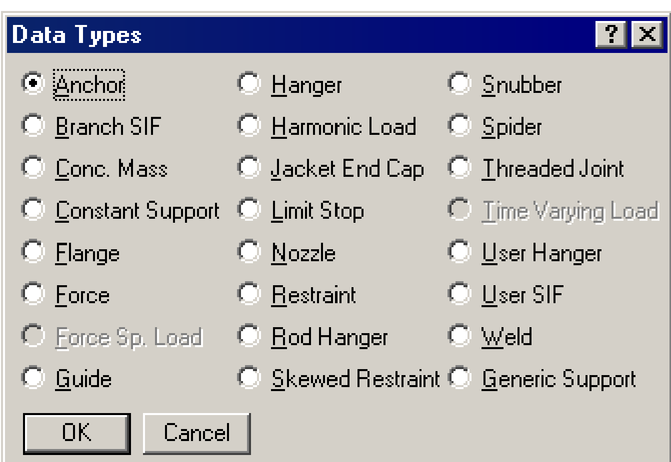
The Data items can be selected from the Data Types dialog which is opened when you click on the Data header in the Layout window.



You may also use the command: Misc > Data types,



or press Ctrl+Shift+D to open the Data Types dialog.



You can select the data type by clicking on the radio button or pressing the underlined letter of the item, e.g., press "f" for Flange, Force or Force spectrum load. Or, you may simply start typing the first few letters of the item in the Data column. (For example, typing "fo" automatically opens a Force data type dialog).

Direction is required for several items such as Pump, Compressor, Turbine, Nozzle (for vessel axis), Limit Stop, Skewed restraint, Elastic element and Hinge joint.

The axis or the orientation of an item (listed above), is called the direction vector which is described in terms of the vector's global X, Y and Z components.

The angles the vector makes with the X, Y and Z axes are called Direction angles, whose cosines are called Direction cosines (or global X, Y and Z components used in CAEPIPE).

There are two methods of computing the X, Y and Z components.

First method: When you know the direction angles (see examples 1, 2 and 3).

Second method: When you know the coordinates of the end points of the vector (see example 4).

Example 1: Vertical Vessel

Assume a vertical vessel with axis in the Y direction, and α, β, γ as the direction angles the axis of the vessel makes with global X, Y and Z axes.

The angles are $\alpha = 90°$, $\beta = 0°$ (since axis is parallel to Y axis) and $\gamma = 90°$.

So, the direction cosines or X, Y and Z components are

X comp = cos ($\alpha = 90°$) = 0,
Y comp = cos ($\beta = 0°$) = 1,
Z comp = cos ($\gamma = 90°$) = 0.

For Z vertical: X comp = 0, Y comp = 0 and Z comp = 1.

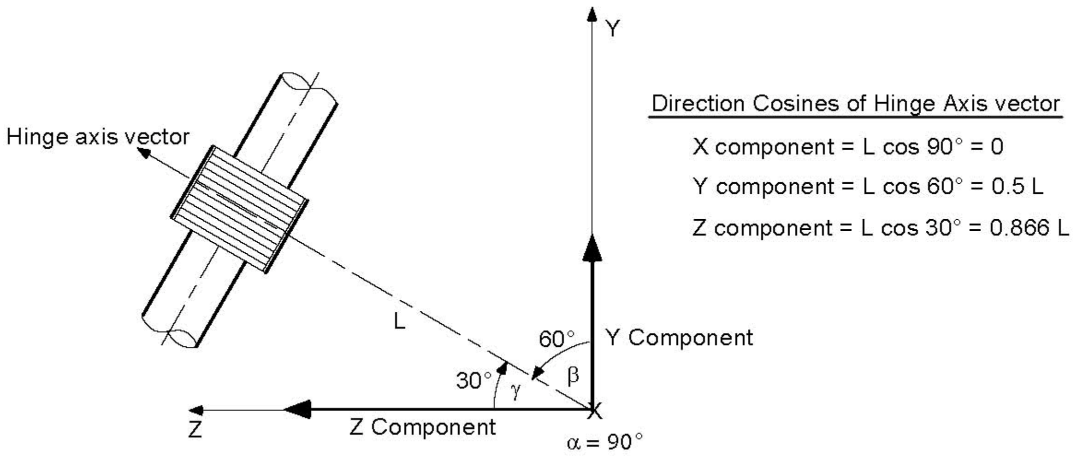
Example 2: Limit Stop at 45° from the X-axis in the X-Y plane

For a limit stop whose axis is oriented at 45° from the X-axis in the X-Y plane, the angles are $\alpha = 45°$, $\beta = 45°$ and $\gamma = 90°$.

So, the direction cosines or X, Y and Z components are

X comp = cos ($\alpha = 45°$) = 0.70711,
Y comp = cos ($\beta = 45°$) = 0.70711,
Z comp = cos ($\gamma = 90°$) = 0.0

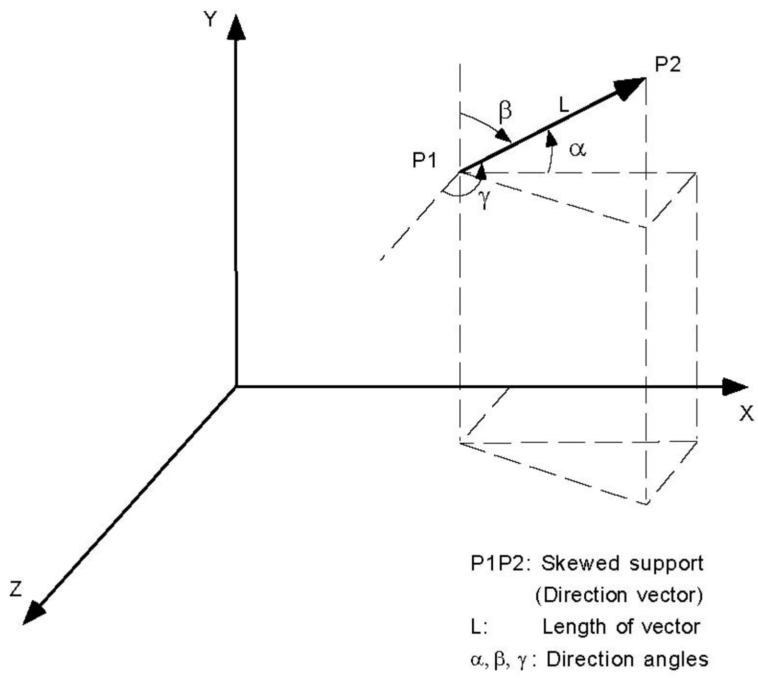
Example 3: Hinge axis at 30° from the Z-axis in the Y-Z plane



From the above figure, we have the angles α = 90°, β = 60° and γ = 30°. Assuming L = 1 (or any length), the direction cosines or X, Y and Z components are

X comp = cos (α = 90°) = 0.0,
Y comp = cos (β = 60°) = 0.5,
Z comp = cos (γ = 30°) = 0.866

Example 4: Skewed Support



Assume that we have a skewed support along P1P2 (which is the direction vector) shown in the figure above, Assume that the coordinates of these two points are P1 = (12',12',12') and P2 =(15',16',14').

Let us calculate this vector's global X, Y and Z components. There are two methods here

Short method

X comp = $(X_2 - X_1) = (15 - 12) = 3$
Y comp = $(Y_2 - Y_1) = (16 - 12) = 4$
Z comp = $(Z_2 - Z_1) = (14 - 12) = 2$

Long method

First, let us calculate the length of the vector, L.

$$L = \sqrt{(X_2 - X_1)^2 + (Y_2 - Y_1)^2 + (Z_2 - Z_1)^2} = 5.385'$$

The angles α, β and γ which the vector makes with the global X, Y and Z axes are called the Direction angles of the vector; The cosines of these angles are called Direction cosines.

$$\cos\alpha = \frac{X_2 - X_1}{L},\ \cos\beta = \frac{Y_2 - Y_1}{L},\ \cos\gamma = \frac{Z_2 - Z_1}{L},$$

The direction cosines are

X comp = $\cos\alpha = 0.55709$, Y comp = $\cos\beta = 0.74278$, Z comp = $\cos\gamma = 0.37139$

For information, the direction angles are α = 56°8'0", β = 42°1'0" and γ = 68°11'0".

To verify the results, the sum of the squares of the direction cosines must be 1.0. Thus,

$\cos^2\alpha + \cos^2\beta + \cos^2\gamma = 0.557092^2 + 0.742782^2 + 0.371392^2 = 1.0$

Modal Analysis

The equations of motion for an undamped lumped mass system may be written as:

$$[M]\{\ddot{u}\} + [K]\{u\} = \{F(t)\} \tag{1}$$

Where $[M]$= diagonal mass matrix

$\{u\}$ = displacement vector
$\{\ddot{u}\}$ = acceleration vector
$[K]$ = stiffness matrix
$\{F(t)\}$ = applied dynamic force vector

If the system is vibrating in a normal mode (i.e., free not forced vibration), we may make the substitutions

$$\{u\} = \{a_n\}\sin\omega_n t$$
$$\{\ddot{u}\} = -\omega_n^2\{a_n\}\sin\omega_n t$$
$$\{F(t)\} = 0$$

to obtain

$$-\omega_n^2[M]\{a_n\} + [K]\{a_n\} = 0$$

or

$$[K]\{a_n\} = \omega_n^2[M]\{a_n\} \tag{2}$$

where $\{a_n\}$ is the vector of modal displacements of the n^{th} mode (eigenvector).

Thus we have an eigenvalue (characteristic value) problem, and the roots of equation (2) are the eigenvalues (characteristic numbers), which are equal to the squares of the natural frequencies of the modes.

In CAEPIPE, the eigenvalue problem is solved using a determinant search technique. The solution algorithm combines triangular factorization and vector inverse iteration in an optimum manner to calculate the required eigenvalues and eigenvectors. These are obtained in sequence starting from the lowest eigen-pair $[\omega_1^2,\{a_1\}]$. An efficient accelerated secant procedure which operates on the characteristic polynomial

$$p(\omega^2) = det([K] - \omega^2[M])$$

is used to obtain a shift near the next unknown eigenvalue. The eigenvalue separation theorem (Sturm sequence property) is used in this iteration. Each determinant evaluation requires a triangular factorization of the matrix$([K] - \omega^2[M])$. Once a shift near the unknown eigenvalue has been obtained, inverse iteration is used to calculate the eigenvector. The eigenvalue is obtained by adding the Rayleigh quotient correction to the shift value. The eigenvector $\{a_n\}$, has an arbitrary magnitude and represents the characteristic shape of that mode.

Starting from Version 10.40, the determinant of the characteristic polynomial is stored in the power form with an exponent of the order of matrix size. With this improvement in Modal analysis algorithm, CAEPIPE can now extract much higher modes (Eigen vectors) up to the frequency of 9999 Hz.

Orthogonality

For any two roots corresponding to the $n^{th}$and m^{th}modes, we may write equation (2) as which is the orthogonality condition for eigenvectors.

$$[K]\{a_n\} = \omega_n^2[M]\{a_n\} \tag{3}$$
$$[K]\{a_m\} = \omega_m^2[M]\{a_m\} \tag{4}$$

If we postmultiply the transpose of (3) by $\{a_m\}$, we obtain

$$([K]\{a_n\})^T\{a_m\} = (\omega_n^2[M]\{a_n\})^T\{a_m\}$$

or

$$\{a_n\}^T[K]^T\{a_m\} = \omega_n^2\{a_n\}^T[M]^T\{a_m\} \tag{5}$$

Premultiplying (4) by $\{a_n\}^T$, $\{a_n\}^T[K]\{a_m\} = \omega_m^2\{a_n\}^T[M]\{a_m\}$ (6)

Since $[M]$ is a diagonal matrix, $[M] = [M]^T$. Also, since $[K]$ is a symmetric matrix, $[K] = [K]^T$. The left sides of equations (5) and (6) are therefore equal.

Subtracting (6) from (5),

$$(\omega_n^2 - \omega_m^2)\{a_n\}^T[M]\{a_m\} = 0 \tag{7}$$

Since $\omega_n \neq \omega_m$,

$$\{a_n\}^T[M]\{a_m\} = 0 \tag{8}$$

which is the orthogonality condition for eigenvectors.

Modal Equations

Since the eigenvectors (modal displacements) may be given any amplitude, it is convenient to replace $\{a_n\}$ by $\{\emptyset_n\}$ such that

$$\{\emptyset_n\}^T[M]\{\emptyset_n\} = 1 \tag{9}$$

The eigenvectors are evaluated so as to satisfy equation (9) and at the same time keep the displacements in the same proportion as those in $\{a_n\}$. The eigenvectors are then said to be *normalized*. Note that equation (7) is still satisfied since, if n = m,$\omega_n^2 - \omega_m^2 = 0$, and the remaining terms may be given any desired value.

Equation (2) now may be written for the n^{th} mode as

$$[K]\{\emptyset_n\} = \omega_n^2[M]\{\emptyset_n\}$$

Let $[\Phi]$be a square matrix containing all normalized eigenvectors such that the n^{th} column is the normalized eigenvector for the n^{th} mode. We can therefore write the matrix equation so as to include all modes as follows:

$$[K]\{\Phi\} = [M][\Phi][\omega_n^2] \tag{10}$$

Where $[\omega_n^2]$ is a diagonal matrix of eigenvalues. We now premultiply both sides of (10) by $[\Phi]^T$to obtain

$$[\Phi]^T[K][\Phi] = [\Phi]^T[M][\Phi][\omega_n^2] \tag{11}$$

It may be shown that

$$[\Phi]^T[M][\Phi] = [I] \tag{12}$$

where $[I]$ is the unit diagonal matrix. Equation (12) can easily be verified by expansion and follows from the orthogonality condition and the fact that $[\Phi]$ has been normalized. Equation (11) therefore can be written as

$$[\Phi]^T[K][\Phi] = [\omega_n^2] \tag{13}$$

Returning now to the equation of motion (1),

$$\text{let } \{u\} = [\Phi]\{A_n\}$$

$$\text{and } \{\ddot{u}\} = [\Phi]\{\ddot{A}_n\} \tag{14}$$

Where $\{A_n\}$ is the modal amplitude of the nth mode. This merely states that the true modal displacements equal the characteristic displacements (eigenvector displacements) times the modal amplitude determined by the response calculations and, further that the total displacements are linear combinations of the modal values. If we now pre-multiply equation (1) by $[\Phi]^T$and substitute equations (14), we obtain

$$[\Phi]^T[M][\Phi]\{\ddot{A}_n\} + [\Phi]^T[K][\Phi]\{A_n\} = [\Phi]^T\{F(t)\} \tag{15}$$

Substituting from equations (12) and (13) in equation (15),

$$\{\ddot{A}_n\} + [\omega_n^2]\{A_n\} = [\Phi]^T\{F(t)\} \tag{16}$$

which represents the modal equations of motion.

Uniform Support Motion

Solutions for uniform support motion (when all supports experience the same excitation) may be obtained if $\{F(t)\}$ is replaced by$-\ddot{u}_s(t)\{M\}$where $\ddot{u}_s(t)$ is the prescribed support acceleration. Thus the modal equations of motion may be written as

$$\{\ddot{A}_n\} + [\omega_n^2]\{A_n\} = -\ddot{u}_s(t)[\Phi]^T\{M\} \tag{17}$$

where $A_n$is the relative modal displacement for the nth mode with respect to the support.

The participation factors for the modes are given by

$$\{\Gamma_n\} = [\Phi]^T\{M\}\{1\} \tag{18}$$

Then the modal amplitude contribution from the nth mode is given by

$$A_n = \Gamma_n u_n^0 \tag{19}$$

Where u_n^0 is the response of a single degree of freedom system having circular frequencyω_n. Using equations (14) and (19), the total displacements from M modes are given by

$$\{u\} = [\Phi]\{A_n\} = [\Phi]\{\Gamma_n u_n^0\} \tag{20}$$

$$= \sum_{n=1}^{M} \phi_n \Gamma_n u_n^0$$

Effective Modal Mass

Effective modal mass is defined as the part of the total mass responding to the dynamic loading in each mode. When the participation factor is calculated using normalized eigenvectors as in equation (18), the effective modal mass for the n^{th} mode is simply the square of the normalized participation factor,

$$M_n = \Gamma_n^2 \tag{21}$$

Effective modal mass is useful to verify if all the significant modes of vibration are included in the dynamic analysis by comparing the total effective modal mass with the total actual mass.

Independent Support Motion (Multi-level Response Spectrum Analysis)

When the piping is routed along the wall of a tall building, the piping supports at higher elevation floors of the building will experience higher seismic excitations, whereas the piping supports at the ground level will experience lesser seismic excitation. Likewise, a pipeline crossing over a bridge may experience different seismic excitations at the ends. Piping systems with such multiple seismic excitations can be analyzed in CAEPIPE using Multi-level Response Spectrum Analysis. For any number of levels in a system, the participation factor for the n^{th} mode with k^{th} level excitation is given by

$$\{\Gamma_{n,k}\} = [\Phi]^T\{M\}\{r_k\} \tag{22}$$

where, r_k is the influence vector representing the simultaneous displacements of all the supports located at k^{th} level.

Then the modal amplitude contribution from the n^{th} mode is given by

$$A_n = \sum_{k=1}^{L} \Gamma_{n,k} u_{n,k}^0 \tag{23}$$

where, L is the number of levels and $u_{n,k}^0$ is the response of a single degree of freedom system having a circular frequency ω_n. The total displacements are given by

$$\{u\} = [\Phi]\{A_n\} = \sum_{n=1}^{M} \phi_n \sum_{k=1}^{L} \left|\Gamma_{n,k} u_{n,k}^0\right| \tag{24}$$

Notes:

1) Two (2) types of combinations “SRSS”and “ABS” can be performed for Level Summations in CAEPIPE The combination over level contributions will be performed first, followed by interspatial and then intermodal combination without the consideration of closely spaced modes. This is consistent with present NRC guidelines.

2) Missing mass correction is not available for Multi-level Response Spectrum Analysis in CAEPIPE at this time. So, it is recommended to include a sufficient number of modes in the analysis and ensure that the “Modal mass / Total mass” is sufficiently high (thereby confirming that most of the mass of the system participates in the modes computed) in Global X, Y and Z directions through Results Window > Results > Results… > Frequencies.

References:

Bezler, P., Subudhi, M., & Hartzman, M. (1985). Piping benchmark problems: dynamic analysis independent support motion response spectrum method (NUREG/CR--1677-Vol2). United States

Nakamura, Y., Kiureghian, A.D., & Liu, D. (1993). Multiple-Support Response Spectrum analysis of the golden gate bridge.

Lin, C.W., & Loceff, F. (1980). A new approach to compute system response with multiple support response spectra input. Nuclear Engineering and Design, 60(3), 347-352.

Response Spectrum

The concept of response spectrum, in recent years has gained wide acceptance in structural dynamics analysis, particularly in seismic design. Stated briefly, the response spectrum is a plot of the maximum response (maximum displacement, velocity, acceleration or any other quantity of interest), to a specified loading for all possible single degree-of-freedom systems. The abscissa of the spectrum is the natural frequency (or period) of the system, and the ordinate, the maximum response.

In general, response spectra are prepared by calculating the response to a specified excitation of single degree-of-freedom systems with various amounts of damping. Numerical integration with short time steps is used to calculate the response of the system. The step-by-step process is continued until the total earthquake record is completed and becomes the response of the system to that excitation. Changing the parameters of the system to change the natural frequency, the process is repeated and a new maximum response is recorded. This process is repeated until all frequencies of interest have been covered and the results plotted. CAEPIPE provides fourteen (14) response spectra for your convenience. Refer to Appendix B titled "Response Spectrum Libraries" in CAEPIPE User's Manual for details.

Since the response spectra give only maximum response, only the maximum values for each mode are calculated and then superimposed (modal combination) to give total response. A conservative upper bound for the total response may be obtained by adding the absolute values of the maximum modal components (absolute sum). However this is excessively conservative and a more probable value of the maximum response is the square root of the sum of squares (SRSS) of the modal maxima.

To calculate response of the piping system, for each natural frequency of the piping system, the input spectrum is interpolated (linearly or logarithmically). The interpolated spectrum values are then combined for the X, Y and Z directions (direction sum) either as absolute sum or SRSS sum to give the maximum response of a single degree-of-freedom system: $u^0_{\max}$ at that frequency.

From equation (20) or (24) the maximum displacement vector for the n^{th} mode can be calculated from the maximum response $\{u_n\}_{max}$ of a single degree-of-freedom system.

The maximum values of element and support load forces per mode are calculated from the maximum displacements calculated per mode as above using the stiffness properties of the structure.

The total response (displacements and forces) is calculated by superimposing the modal responses according to the specified mode sum method which can be absolute sum, square root of sum of squares (SRSS) or closely spaced (10%) modes method.

Closely Spaced Modes

Studies have shown that SRSS procedure for combining modes can significantly underestimate the true response in certain cases in which some of the natural frequencies of a structural system are closely spaced. The ten percent method is one of NRC approved methods (Based on NRC Guide 1.92) for addressing this problem.

$$R = \sqrt{\sum_{n=1}^{N} R_n^2 + 2\sum \left|R_i R_j\right|}$$

where R = Total (combined) response
R_n = Peak value of the response due to the n^{th} mode
N = Number of significant modes

The second summation is to be done on all i and j modes whose frequencies are closely spaced to each other. Let ω_i and ω_j be the frequencies of the i^{th} and j^{th} modes. The modes are closely spaced if:

$$\frac{\omega_j - \omega_i}{\omega_i} \leq 0.1 \text{ and } 1 \leq i \leq j \leq N$$

Time History

Time history analysis requires the solution to the equations

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F(t)\} \tag{25}$$

where $[M]$ = diagonal mass matrix
$[C]$ = damping matrix
$[K]$ = stiffness matrix
$\{u\}$ = displacement vector
$\{\dot{u}\}$ = velocity vector
$\{\ddot{u}\}$ = acceleration vector
$\{F(t)\}$ = applied dynamic force vector

The time history analysis is carried out using mode superposition method. It is assumed that the structural response can be described adequately by the p lowest vibration modes out of the total possible n vibration modes and p < n. Using the transformation$u = \Phi X$, where the columns in Φ are the p mass normalized eigenvectors, equation (25) can be written as

$$\ddot{X} + \Delta\dot{X} + \Omega^2 \mathrm{X} = \Phi^T F \quad (26)$$

where Δ = diag $(2\omega_i \xi_i)$

Ω^2 = diag (ω_i^2)

In equation (26), it is assumed that the damping matrix $[C]$satisfies the modal orthogonalitycondition

$$\{\phi_i\}^T [C]\{\phi_j\} = 0 \quad (i \neq j)$$

Equation (26) therefore represents p uncoupled second order differential equations. These are solved using the Wilson θ method, which is an unconditionally stable step-by-step integration scheme. The same time step is used in the integration of all equations to simplify the calculations.

Harmonic Analysis

A harmonic analysis is performed to determine the response of a piping system to sinusoidal loads. Harmonic forces can arise from unbalanced rotating equipment, acoustic vibrations caused by reciprocating equipment, flow impedance, and other sources. These forces can be damaging to a piping system if their frequency is close to the piping system's natural frequency, thereby introducing resonant conditions. The equation of dynamic equilibrium associated with the response of the structure subjected to harmonic forces is:

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \sin(\omega t)F \quad (27)$$

where $[M]$ = diagonal mass matrix

$[C]$ = damping matrix

$[K]$ = stiffness matrix

$\{u\}$ = displacement vector

$\{\dot{u}\}$ = velocity vector

$\{\ddot{u}\}$ = acceleration vector

ω = frequency of the applied force

t = time

F = maximum magnitude of the applied force

It is feasible that multiple harmonic loads may be applied simultaneously at different locations of a piping system. More complex forms of vibration, such as those caused by the fluid flow, may be considered as superposition of several simple harmonics, each with its own frequency, magnitude, and phase.

A harmonic analysis uses the results from the modal analysis to obtain a solution. A single damping factor is used for all modes.

First, the maximum response for each harmonic load is obtained separately. Then, the total response for multiple simultaneous harmonic loads is determined by combining the individual responses. The combination method may be specified as the Root Mean Square (RMS) or Absolute Sum. Even in the case of a system with a single harmonic load, the said combination is always carried out, so that the resulting solution becomes an "unsigned" case. For an unsigned case, the actual values for displacements, element forces and moments, etc. computed internally by CAEPIPE prior to such combination can be +ve or -ve for the dynamic event. After the combination, the resulting values become "unsigned".

Dynamic Susceptibility

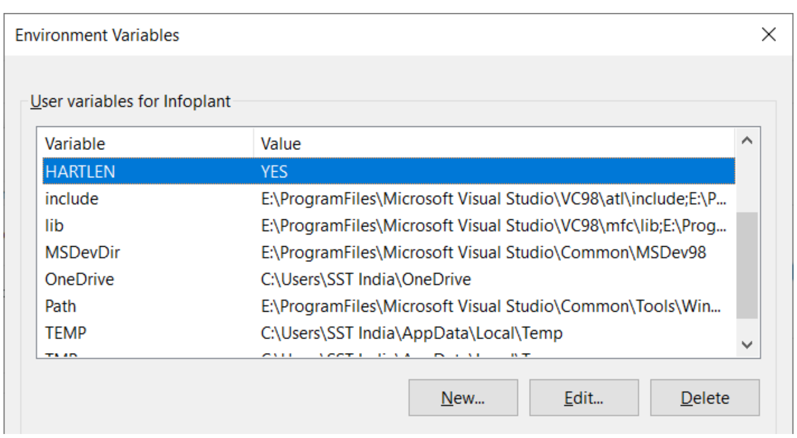
Dynamic Susceptibility feature is a screening tool for potentially large alternating stresses. The dynamic stresses are the dynamic bending stresses associated with vibration in a natural mode. In other words, the modal analysis result has been generalized to include the alternating bending stresses associated with the vibration in a natural mode. The dynamic susceptibility for any mode is the ratio of the maximum alternating bending stress to the maximum vibration velocity. This "susceptibility ratio" provides an indicator of the susceptibility of the system to large dynamic stresses. Also, the associated animated mode shapes include color-spot-markers identifying the respective locations of maximum vibration velocity and maximum dynamic bending stress. The susceptibility ratio and the graphics feature provide incisive insights into the reasons for high susceptibility and how to make improvements.

The "Modal Analysis" output load case in CAEPIPE has been enhanced. In addition to the modal frequencies and mode shapes, you will see two new results items called "dynamic stresses" and "dynamic susceptibility."

In case you do not see these two items in the results dialog, you need to activate this feature by defining an environment variable. See Annexure I for a detailed discussion.

First method:

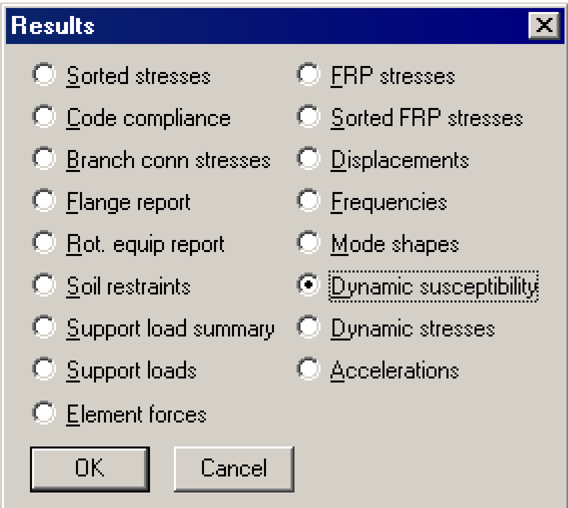
An environment variable "HARTLEN" needs to be declared under My Computer > Properties > Environment > Variable (HARTLEN), and its Value set to (YES). Please check with your System Admin because different versions of Windows have slightly different methods of doing it.

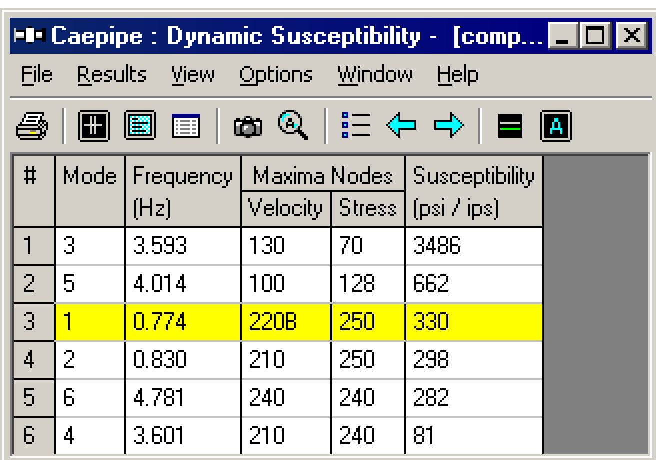


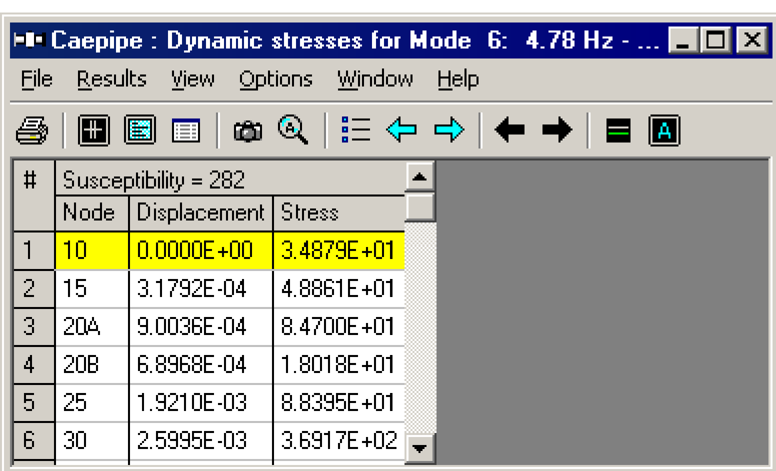
Second method:

Open the MS-DOS Command Prompt. Type "SET HARTLEN=YES" (enter), change directory (using CD command) to where CAEPIPE program files are located, start CAEPIPE.EXE.

Upon (modal) analysis, the Results dialog will display the required results (dynamic stresses and dynamic susceptibility).

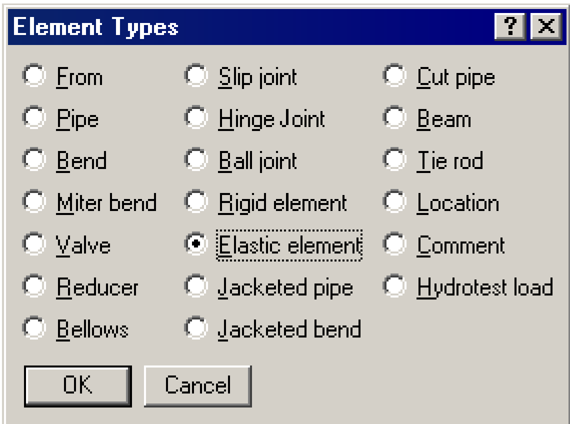




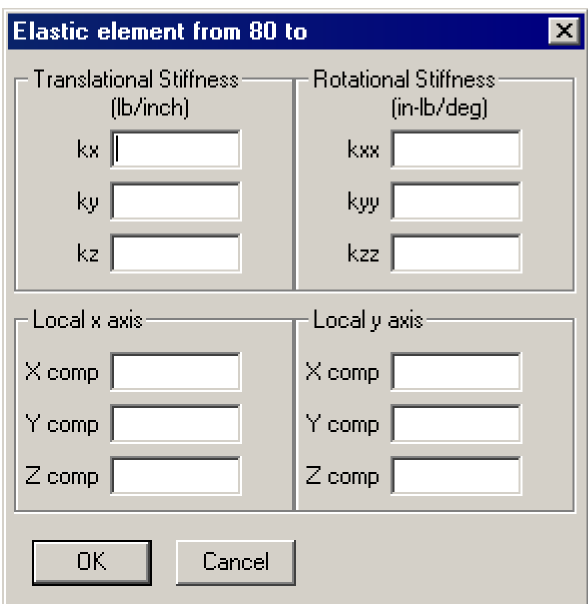


The elastic element is a general 6 × 6 stiffness matrix, with nonzero diagonal terms and zero off-diagonal terms. Use this element to model the stiffness of a component unavailable in CAEPIPE.

An elastic element is input by typing "e" in the Type column or selecting "Elastic element" from the Element Types dialog.



The Elastic element dialog is shown.



The stiffnesses are in the local coordinate system defined by the directions of the local x-, y- and z-axes. As done for all other element types, the positive local x-axis for the elastic element is along the element from the "from" node to the "to" node. The local y-axis should be perpendicular to the local x-axis (i.e., their dot product should be zero). The local z-axis is internally calculated as cross product of the local x- and y-axes.

The elastic element is not subjected to any sustained or thermal expansion loads.

CAEPIPE provides the following types of expansion joints:

1. Ball joint,
2. Bellows,
3. Hinge joint and
4. Slip joint

Using these types, you can also model tied bellows, a gimbal, a dual gimbal, a pressure-balanced elbow and a tee, a slotted hinge joint and a universal joint among other complex arrangements.

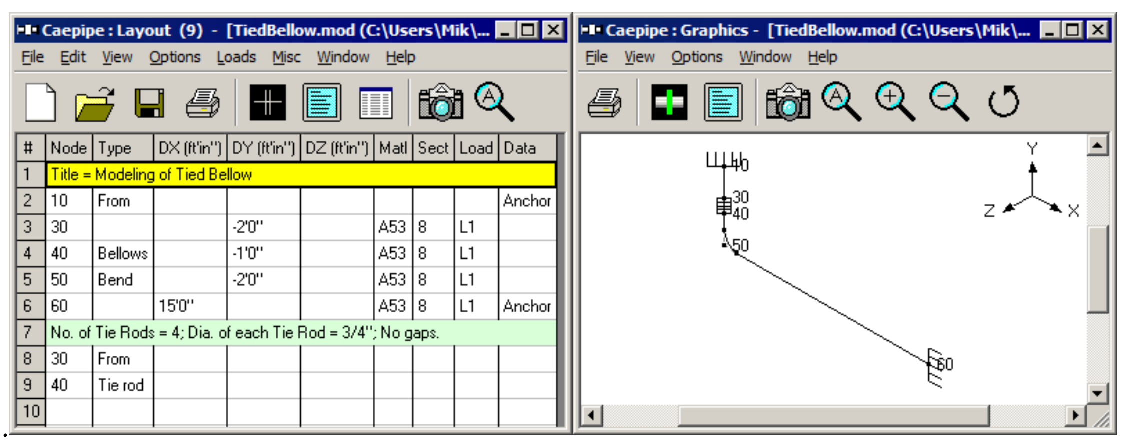
Before selecting the types and locations for the expansion joints, you must study a piping system for the direction and magnitude of the thermal movements to be absorbed, availability of support structures for anchoring, and guiding of the piping. EJMA and manufacturer catalogs contain technical information that can guide you through this process. CAEPIPE becomes an ideal "what-if" tool for such rapid studies.

This topic will show how to model the following types of joints: Tied bellows, hinged bellows, gimbal, universal hinged bellows and pressure balanced joints. Also see discussion on Bellows.

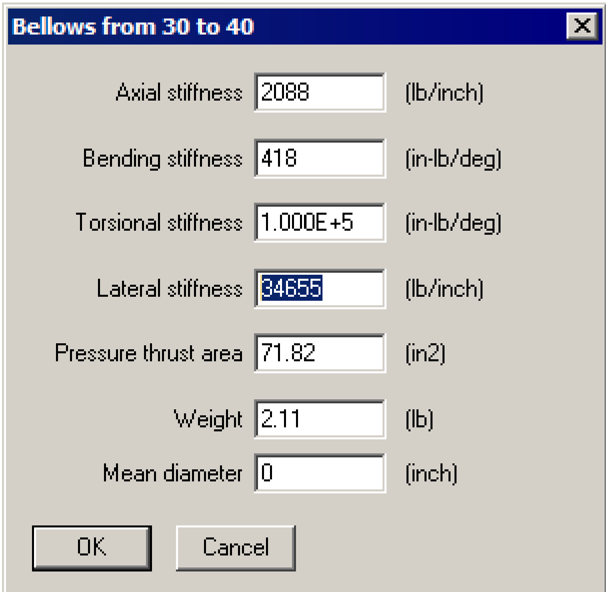
Example 1: Tied Bellow (without gaps)

Whenever a bellow is present in a piping system, the equipment nozzle/piping support adjacent to the bellow will experience a pressure thrust force (=pressure thrust area x pressure) generated by the bellow during normal operation. Tie rods can be added to the bellow in order to fully absorb such pressure thrust force, while still allowing the bellow to laterally deflect (i.e., allowing lateral displacement and lateral rotation).

In the example shown below, four tie rods are attached to the bellow without any "gaps" on tie rods on either side of the bellow. Because there are no "gaps", the tie rods offer the same stiffness under both tension and compression (as long as the compression is not large enough to buckle the tie rods). In order to determine the axial force carried by each tie rod, pressure thrust area for the bellow must be input. One way of modeling the tie rods is to lump all four tie rods into a single tie rod along the bellow center line (with tension stiffness = compression stiffness = n x stiffness of each tie rod = n x EA/L, where 'n' is the number of tie rods, E is the Young's Modulus of the tie rod material, A and L are the cross-sectional area and length of each tie rod).



In the example shown above, the properties of the Tied Bellow are as follows.



Note:

Weight is to be input in lbf or kgf and NOT in mass units. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

For Bending stiffness of the bellow, the following two options are provided.

Option 1: Input the Bending stiffness as specified by the manufacturer or as reasonably determined from industry standards such as EJMA. If a non-zero value for Bending stiffness is input, then leave the "Mean diameter" field blank or zero.

Option 2: If a non-zero value for Bending stiffness is not input as per Option 1 above and is left blank, then input the actual non-zero value for "Mean diameter", in which case CAEPIPE will internally calculate the Bending stiffness for the bellow based on the Mean diameter and other inputs provided for that bellow. In this case, the Mean diameter is the "mean" between the outer and inner diameters of any Convolution of the bellow. Since outer and inner diameters of all convolutions of the bellow are the same, the Mean diameter is the same for all convolutions of that bellow.

Among the above two options, Option 1 is recommended if you are able to specify a realistic non-zero value for the Bending stiffness of the bellow.

Tie Rods properties

No. of Tie Rods (n) = 4 Nos.

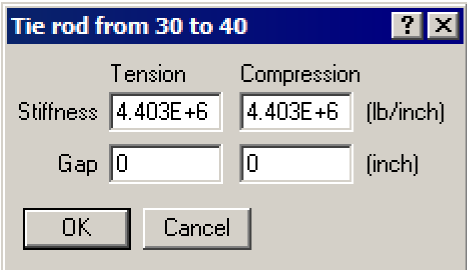
Diameter of Tie Rod (D) = 3/4"

Length of Tie Rod (L) = 12"

Young's Modulus of Tie Rod (E) = 29.9E+6 psi

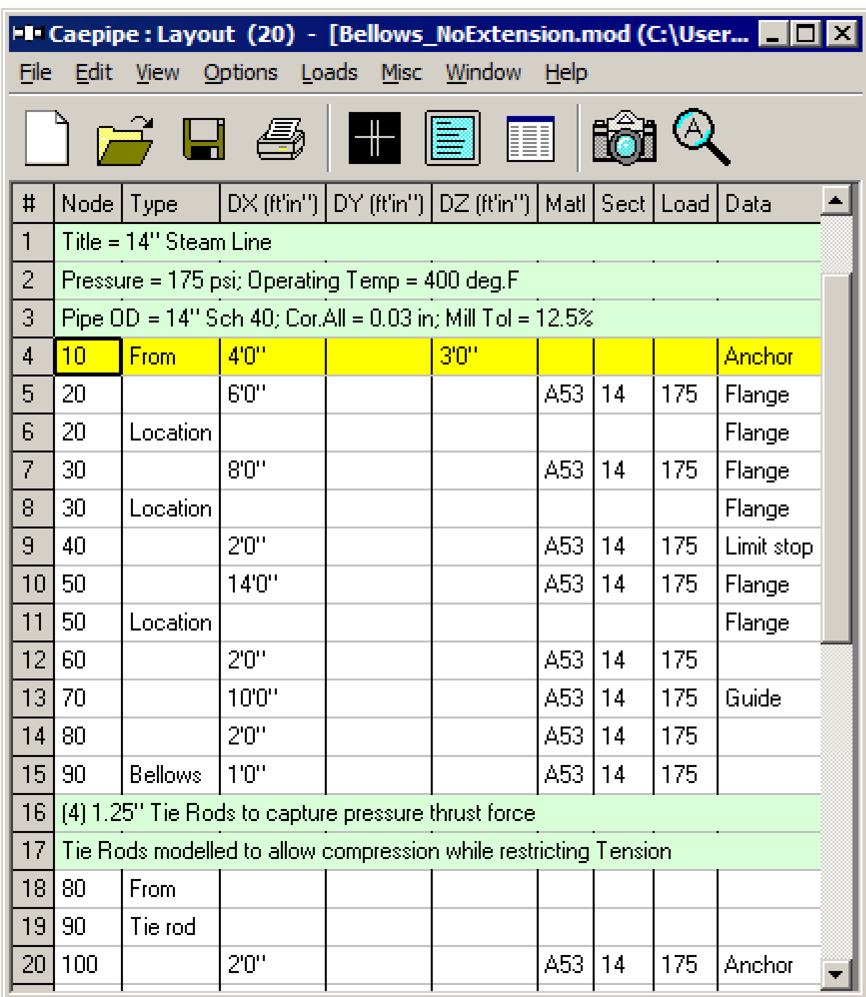
Stiffness of Tie Rods = n x AE/L = 4 x (π/4) x 0.75^2 x 29.9E+6 / (12") = 4.403E+6 lb/in

Accordingly, for Tie Rods, Tension Stiffness = Compression Stiffness = 4.403E+6 lb/in.

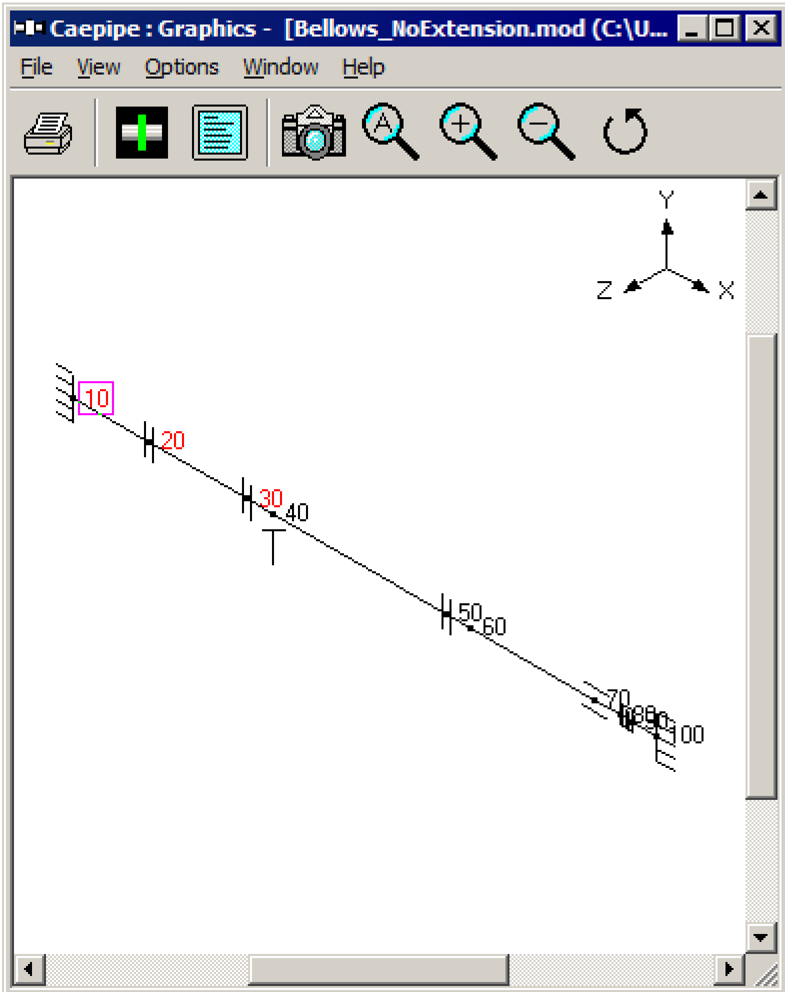


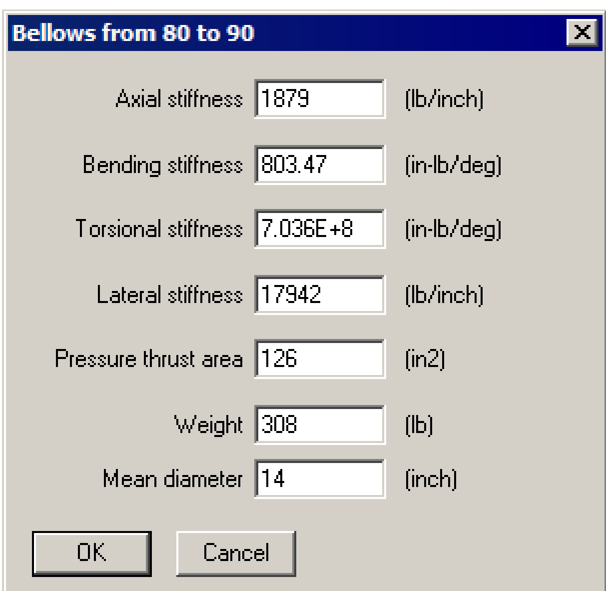
Example 2: Tied bellow with free compression

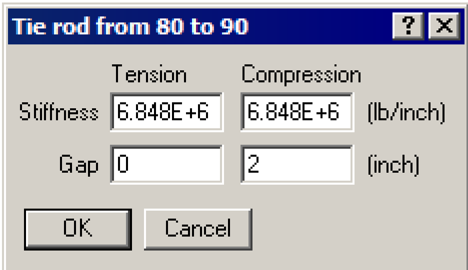
The model shown below has a tied bellow between Nodes 80 & 90. Tie Rod is defined with the same tension stiffness and compression stiffness of 6.848E+06 lb/in (equals to combined axial stiffness of 4 Nos. of 1.25" dia. tie rods). However, gaps are set differently in the tension and compression directions, namely 0.0" in the tension direction and 2.0" in the compression direction (assuming 2.0" as the maximum compression permitted by the manufacturer). This allows the bellow to compress freely up to 2.0" and at the same time restricts the bellow from extension. Beyond 2.0" of compression, compression stiffness of tie rods will come into play.



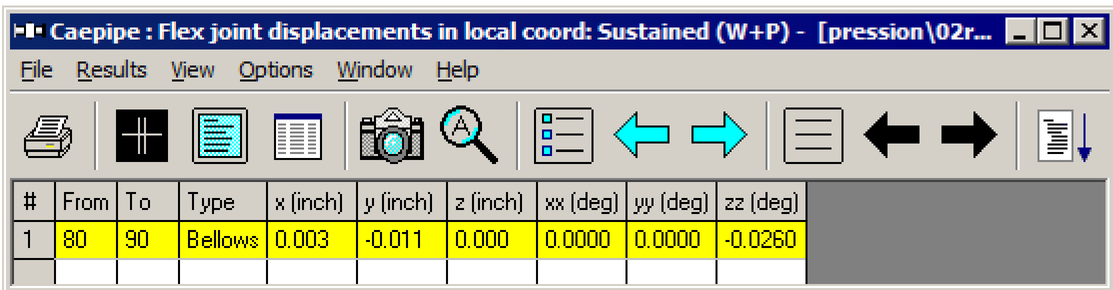
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = 14" Steam Line								
2	Pressure = 175 psi; Operating Temp = 400 deg.F								
3	Pipe OD = 14" Sch 40; Cor.All = 0.03 in; Mill Tol = 12.5%								
4	10	From	4'0"		3'0"				Anchor
5	20		6'0"			A53	14	175	Flange
6	20	Location							Flange
7	30		8'0"			A53	14	175	Flange
8	30	Location							Flange
9	40		2'0"			A53	14	175	Limit stop
10	50		14'0"			A53	14	175	Flange
11	50	Location							Flange
12	60		2'0"			A53	14	175	
13	70		10'0"			A53	14	175	Guide
14	80		2'0"			A53	14	175	
15	90	Bellows	1'0"			A53	14	175	
16	(4) 1.25" Tie Rods to capture pressure thrust force								
17	Tie Rods modelled to allow compression while restricting Tension								
18	80	From							
19	90	Tie rod							
20	100		2'0"			A53	14	175	Anchor


Caepipe : Graphics - [Bellows_NoExtension.mod (C:\U...
File
View
Options
Window
Help
Y
Z
X
10
20
30
40
50
60
70
100

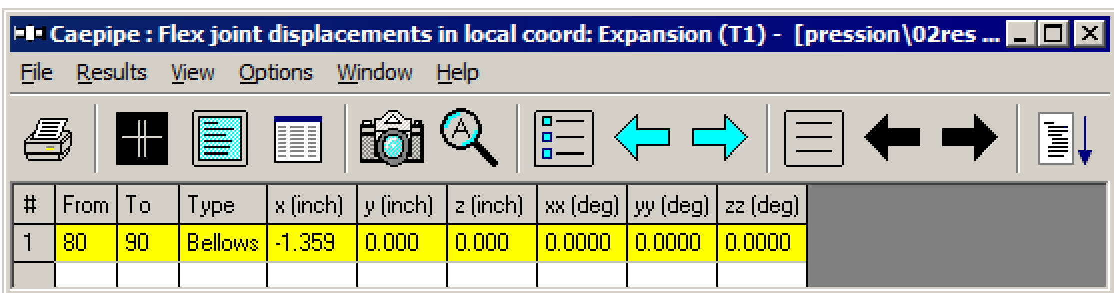

Bellows from 80 to 90
Axial stiffness 1879 (lb/inch)
Bending stiffness 803.47 (in-lb/deg)
Torsional stiffness 7.036E+8 (in-lb/deg)
Lateral stiffness 17942 (lb/inch)
Pressure thrust area 126 (in2)
Weight 308 (lb)
Mean diameter 14 (inch)
OK
Cancel



From“Flex. Joint” displacements results of CAEPIPE, it is observed that the deflection for bellow between Nodes 80 and 90 is +0.003” for Sustained Case and -1.359” for Expansion load case (which is less than the compression gap of 2.0” provided). Please observe that the bellow compresses for the Expansion load in this model as the bellow is in between two anchors. This confirms that the modeling of Tied bellow with 0.0” gap for tension and 2.0” gap for compression directions produces the expected results.



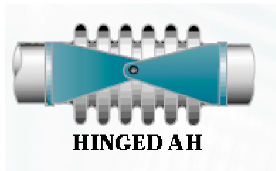
#	From	To	Type	x (inch)	y (inch)	z (inch)	xx (deg)	yy (deg)	zz (deg)
1	80	90	Bellows	0.003	-0.011	0.000	0.0000	0.0000	-0.0260



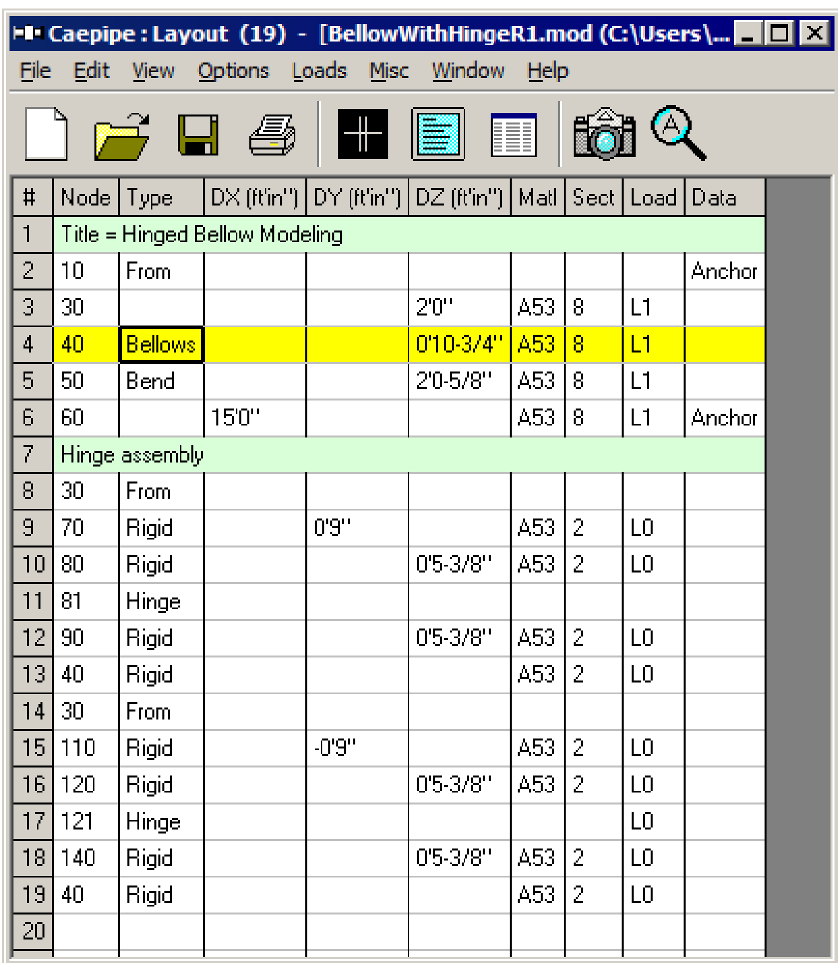
#	From	To	Type	x (inch)	y (inch)	z (inch)	xx (deg)	yy (deg)	zz (deg)
1	80	90	Bellows	-1.359	0.000	0.000	0.0000	0.0000	0.0000

Example 3: Hinged Bellow

A hinged expansion joint contains one bellow and is designed to permite angular rotation in one plane only, by the use of a pair of pins through hinge plates attached to the expansion joint ends. The hinges and hinge pins must be designed to restrain the thrust of the expansion joint due to internal pressure and extraneous forces, where applicable. See Figure shown below.

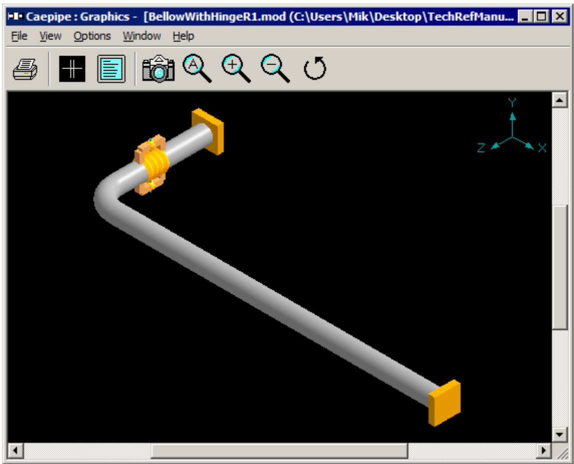


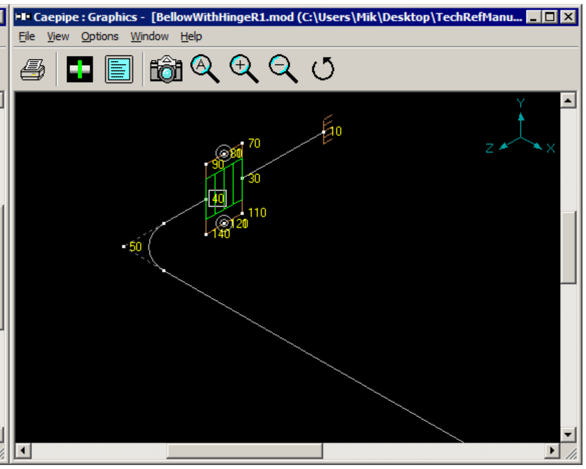
The sample model shown below has a Tied bellow between Nodes 30 and 40. The stiffnesses of the bellow in Axial = 2088 lb/in, Bending = 418 in-lb/deg, Torsion = 100000 in-lb/deg (in case of unavailability of data, set the Torsional stiffness of the bellow to be the same as the torsional stiffness of equivalent pipe), and Lateral = 34655 lb/in. The stiffnesses of the hinge plates are assumed to be "Rigid" in this example. Accordingly, to connect the Bellow Nodes 30 and 40 to Hinge plates, four (4) weightless "Rigid" elements are defined connecting the Nodes 30-70, 30-110, 40-90 and 40-140 with each one having its length as 9" (as the OD of the Flange is indicated as 18" in hinged bellow catalog referred). In addition, four (4) more weightless "Rigid" elements were defined connecting the Nodes 70-80, 81-90, 110-120 and 121-140 and two (2) hinges connecting nodes 80-81 and 120-121.

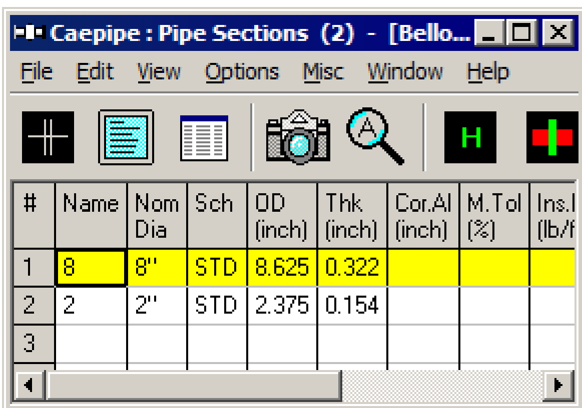

Caepipe : Layout (19) - [BellowWithHingeR1.mod (C:\Users\...

File Edit View Options Loads Misc Window Help

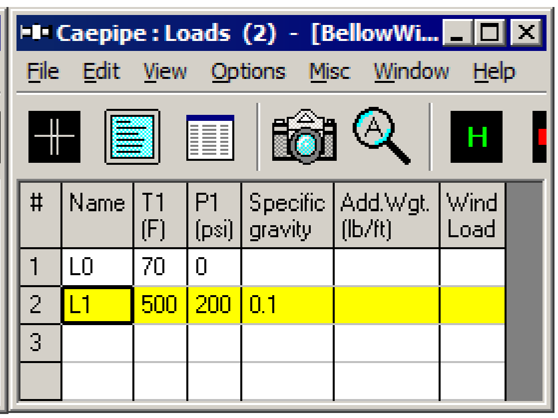
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Hinged Bellow Modeling								
2	10	From							Anchor
3	30				2'0"	A53	8	L1	
4	40	Bellows			0'10-3/4"	A53	8	L1	
5	50	Bend			2'0-5/8"	A53	8	L1	
6	60		15'0"			A53	8	L1	Anchor
7	Hinge assembly								
8	30	From							
9	70	Rigid		0'9"		A53	2	L0	
10	80	Rigid			0'5-3/8"	A53	2	L0	
11	81	Hinge							
12	90	Rigid			0'5-3/8"	A53	2	L0	
13	40	Rigid				A53	2	L0	
14	30	From							
15	110	Rigid		-0'9"		A53	2	L0	
16	120	Rigid			0'5-3/8"	A53	2	L0	
17	121	Hinge						L0	
18	140	Rigid			0'5-3/8"	A53	2	L0	
19	40	Rigid				A53	2	L0	
20									



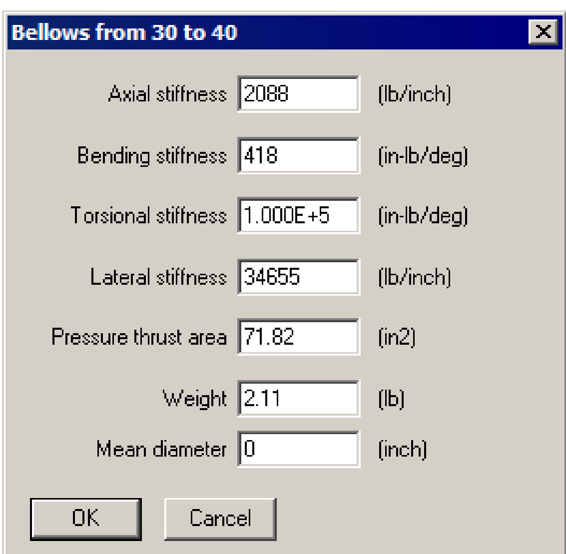


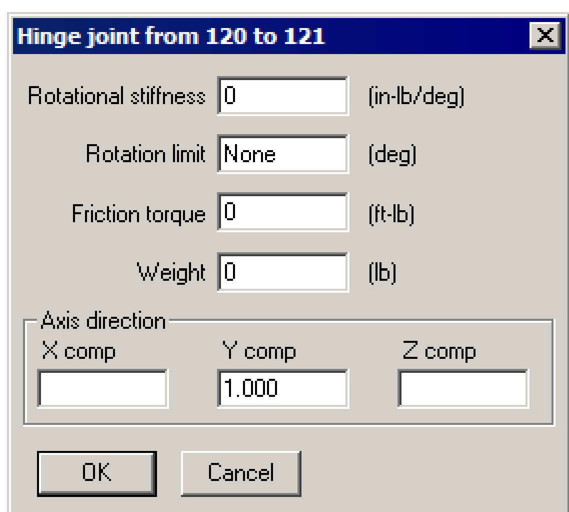


#	Name	Nom Dia	Sch	OD (inch)	Thk (inch)	Cor.Al (inch)	M.Tol (%)	Ins.I (lb/f
1	8	8"	STD	8.625	0.322			
2	2	2"	STD	2.375	0.154			
3								

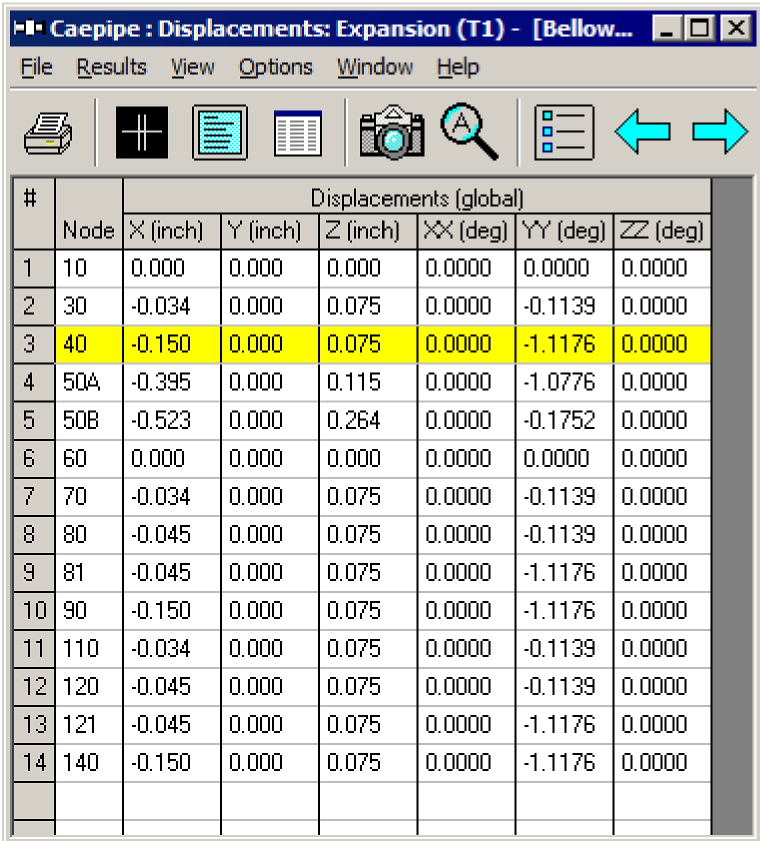


#	Name	T1 (F)	P1 (psi)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load
1	L0	70	0			
2	L1	500	200	0.1		
3						

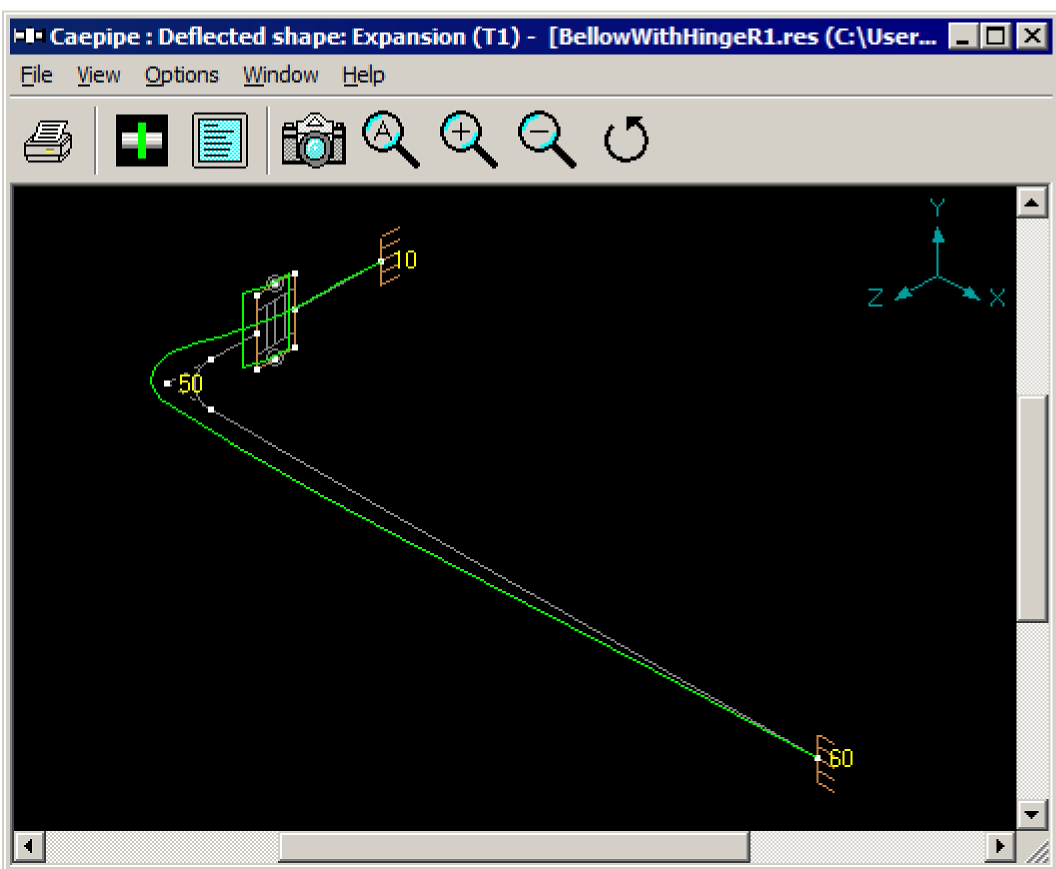




Now from the displacements results of CAEPIPE for Expansion load case, it is observed that the rotation at Node 40 is much larger than the rotation at Node 30 in YY direction. In other words, the hinges at Nodes 80 and 120 are allowing the two ends of the bellow to bend. This in effect confirms that the modeling of hinged bellow as shown in this model produces the expected results.



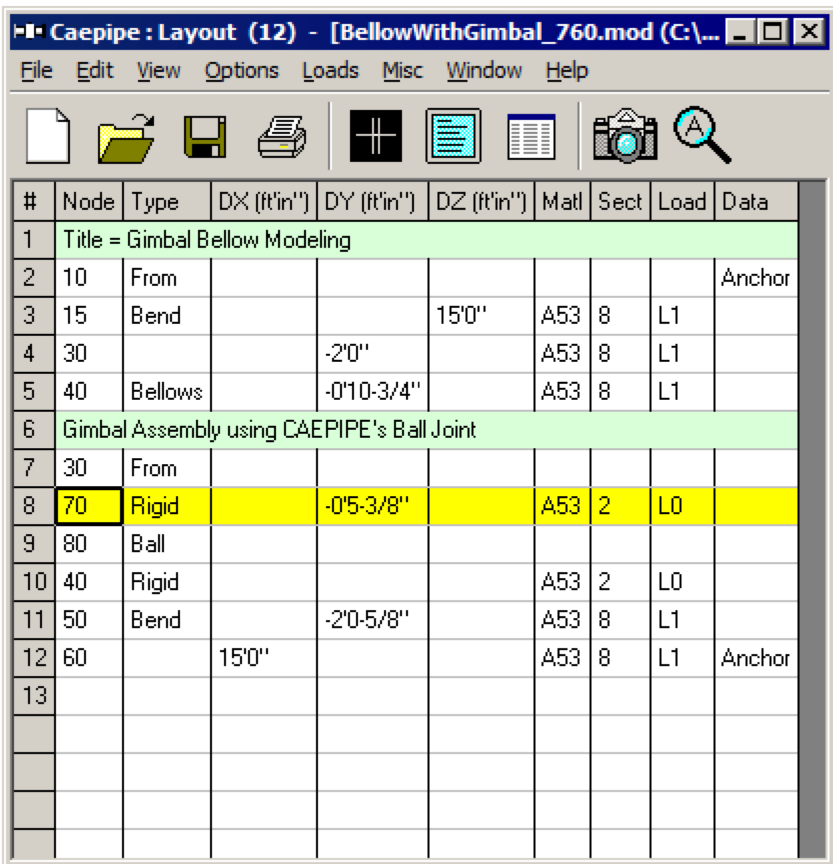
#	Node	Displacements (global)					
		X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	30	-0.034	0.000	0.075	0.0000	-0.1139	0.0000
3	40	-0.150	0.000	0.075	0.0000	-1.1176	0.0000
4	50A	-0.395	0.000	0.115	0.0000	-1.0776	0.0000
5	50B	-0.523	0.000	0.264	0.0000	-0.1752	0.0000
6	60	0.000	0.000	0.000	0.0000	0.0000	0.0000
7	70	-0.034	0.000	0.075	0.0000	-0.1139	0.0000
8	80	-0.045	0.000	0.075	0.0000	-0.1139	0.0000
9	81	-0.045	0.000	0.075	0.0000	-1.1176	0.0000
10	90	-0.150	0.000	0.075	0.0000	-1.1176	0.0000
11	110	-0.034	0.000	0.075	0.0000	-0.1139	0.0000
12	120	-0.045	0.000	0.075	0.0000	-0.1139	0.0000
13	121	-0.045	0.000	0.075	0.0000	-1.1176	0.0000
14	140	-0.150	0.000	0.075	0.0000	-1.1176	0.0000



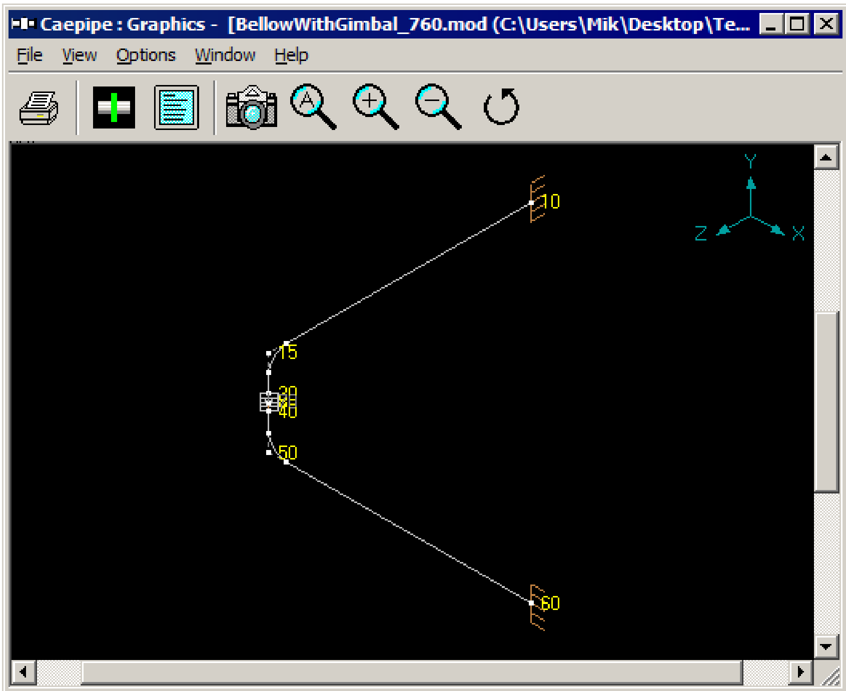
Example 4: Gimbal Bellow

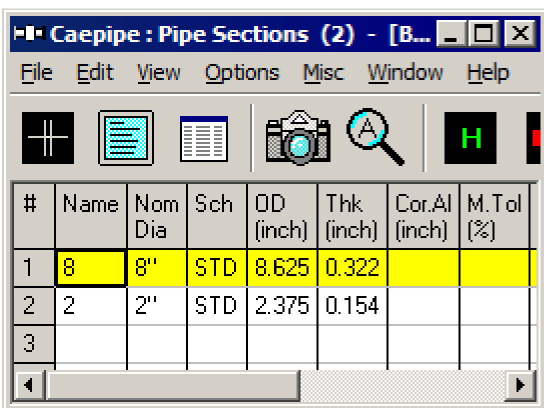
A gimbal expansion joint is designed to permit angular rotation in any plane by the use of two pairs of hinges affixed to a common floating gimbal ring. The gimbal ring, hinges and pins are designed to restrain the thrust of the expansion joint due to internal pressure and extraneous forces, where applicable.

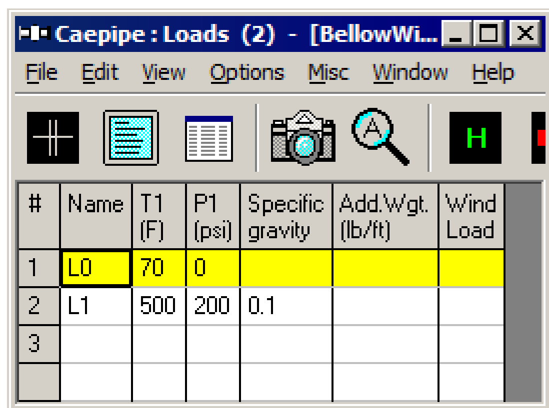
In this sample model, the Gimbal is simulated by connecting the Bellow Nodes 30 & 40 using two "massless" Rigid Elements and one Ball Joint (i.e., a Rigid Element from Nodes 30 to 70 followed by a Ball Joint connecting Nodes 70 & 80 and another Rigid Element from Nodes 80 to 40). All the stiffnesses of the Ball Joint are made as "Rigid" excepting the Bending Stiffness. The Bending Stiffness (the same applied in both "local y" and "local z" directions) is defined as "1" in-lb/deg. In addition, weight of this ball joint is left blank (i.e., equal to 0.0).

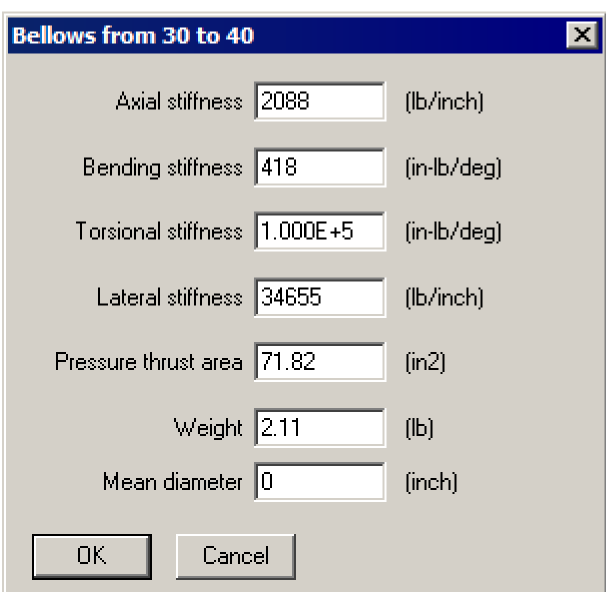


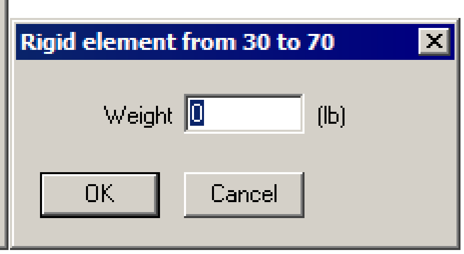
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Gimbal Bellow Modeling								
2	10	From							Anchor
3	15	Bend			15'0"	A53	8	L1	
4	30			-2'0"		A53	8	L1	
5	40	Bellows		-0'10-3/4"		A53	8	L1	
6	Gimbal Assembly using CAEPIPE's Ball Joint								
7	30	From							
8	70	Rigid		-0'5-3/8"		A53	2	L0	
9	80	Ball							
10	40	Rigid				A53	2	L0	
11	50	Bend		-2'0-5/8"		A53	8	L1	
12	60		15'0"			A53	8	L1	Anchor
13									

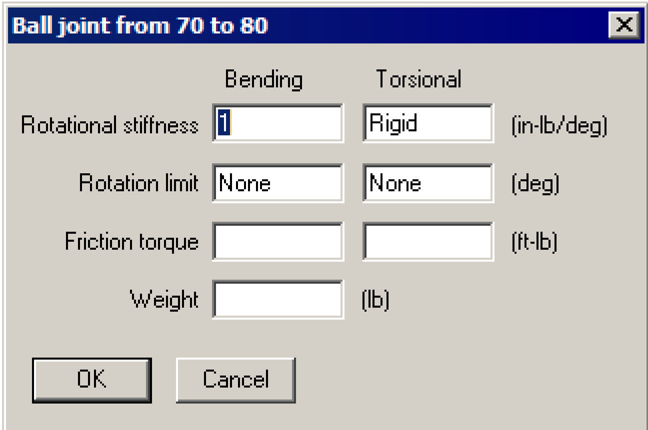

Caepipe : Graphics - [BellowWithGimbal_760.mod (C:\Users\Mik\Desktop\Te...
File View Options Window Help
10
15
30
40
50
60
Y
Z
X


Caepipe : Pipe Sections (2) - [B...
File Edit View Options Misc Window Help
H
Name Nom Dia Sch OD (inch) Thk (inch) Cor.Al (inch) M.Tol (%)
1 8 8" STD 8.625 0.322
2 2 2" STD 2.375 0.154
3

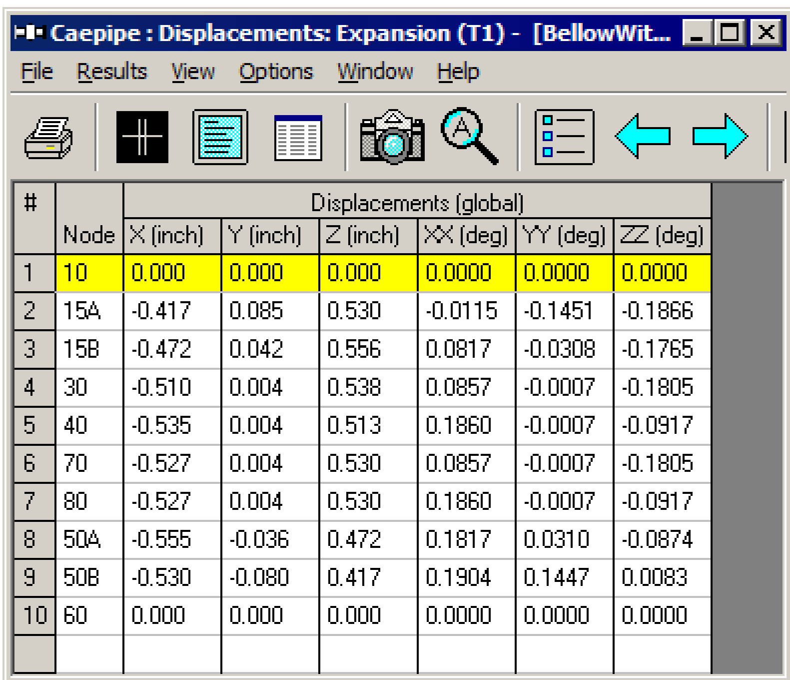

Caepipe : Loads (2) - [BellowWi...
File Edit View Options Misc Window Help
H
Name T1 (F) P1 (psi) Specific gravity Add.Wgt. (lb/ft) Wind Load
1 L0 70 0
2 L1 500 200 0.1
3


Bellows from 30 to 40
Axial stiffness 2088 (lb/inch)
Bending stiffness 418 (in-lb/deg)
Torsional stiffness 1.000E+5 (in-lb/deg)
Lateral stiffness 34655 (lb/inch)
Pressure thrust area 71.82 (in2)
Weight 2.11 (lb)
Mean diameter 0 (inch)
OK
Cancel


Rigid element from 30 to 70
Weight 0 (lb)
OK
Cancel



As expected, the "Displacements" results for the bellow displayed in CAEPIPE have a sudden change in XX and ZZ rotations, confirming the fact that the Gimbal is getting rotated in the two orthogonal directions due to the deformation of the two orthogonal lines.



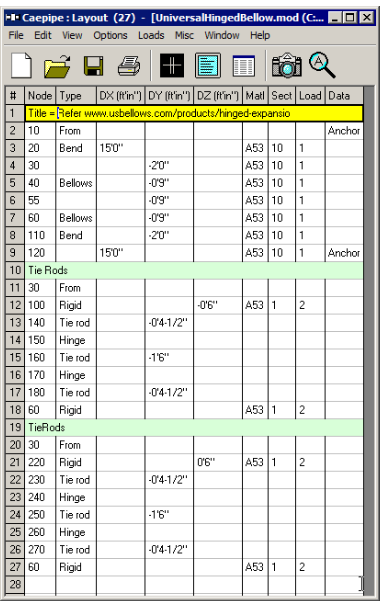
#		Displacements (global)					
	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	15A	-0.417	0.085	0.530	-0.0115	-0.1451	-0.1866
3	15B	-0.472	0.042	0.556	0.0817	-0.0308	-0.1765
4	30	-0.510	0.004	0.538	0.0857	-0.0007	-0.1805
5	40	-0.535	0.004	0.513	0.1860	-0.0007	-0.0917
6	70	-0.527	0.004	0.530	0.0857	-0.0007	-0.1805
7	80	-0.527	0.004	0.530	0.1860	-0.0007	-0.0917
8	50A	-0.555	-0.036	0.472	0.1817	0.0310	-0.0874
9	50B	-0.530	-0.080	0.417	0.1904	0.1447	0.0083
10	60	0.000	0.000	0.000	0.0000	0.0000	0.0000

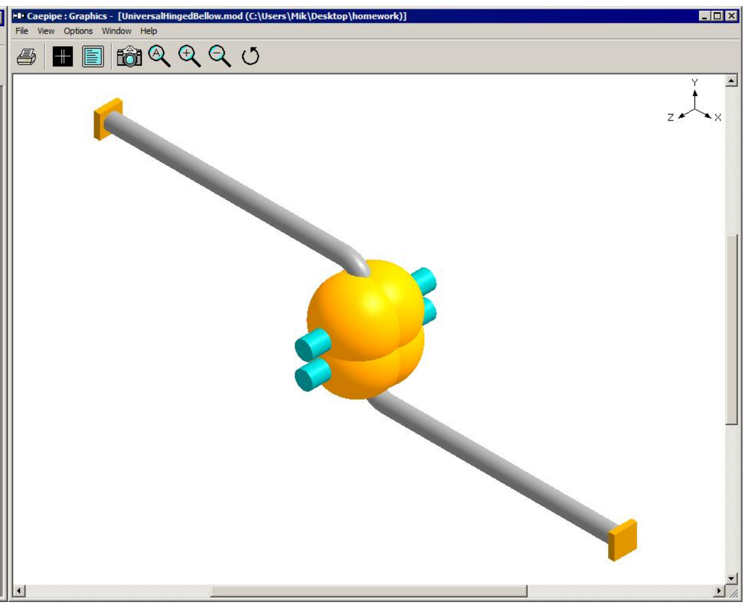
Example 5: Universal Hinged Expansion Joints

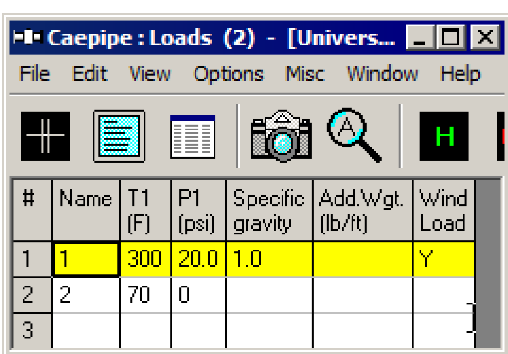
Universal Hinged Expansion Joints have two bellows separated by a pipe spool with overall length restrained by hinge hardware designed to contain pressure thrust. A hinged universal expansion joint accepts large lateral movements in a single plane with very low spring forces.

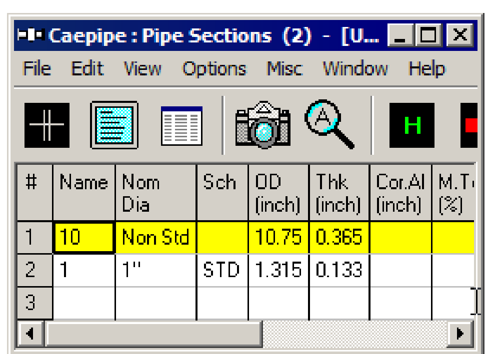
This sample model simulates the Universal Hinged Expansion Joints with two Tie Rods using the CAEPIPE's Tie Rod elements. The advantages of this model are (a) stiffness of the tie rods can be input explicitly (in this case, stiffness corresponding to 1" dia tie rod is input) and (b) gaps can be specified to simulate slotted holes.

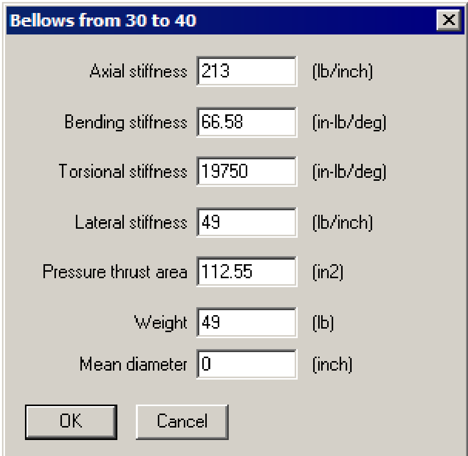
In this sample model, the Universal Hinged Expansion Joint is simulated by connecting the Bellow Nodes 30 & 60 using Tie Rods and “massless” Rigid Elements, namely four “massless” Rigid Elements connecting Nodes 30-100, 30-220, 60-180 and 60-270; two Tie Rods connecting Nodes 100-180 and 220-270 and four hinges connecting Nodes 140-150, 160-170, 230-240 and 250-260. See snap shots shown below for details.

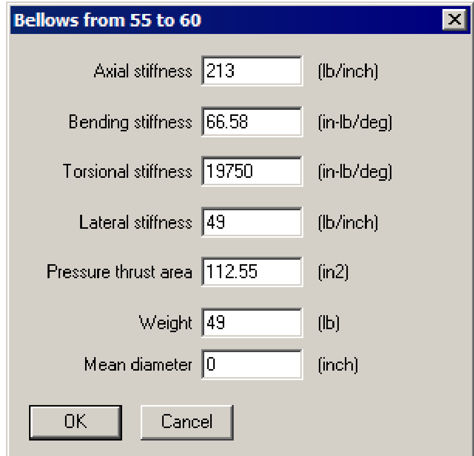


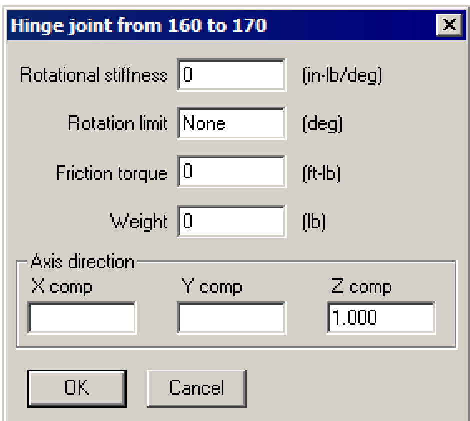


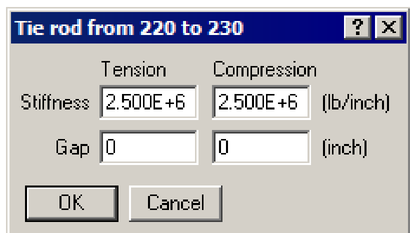


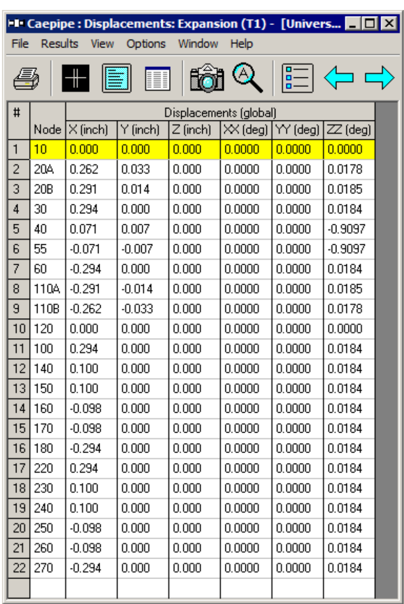




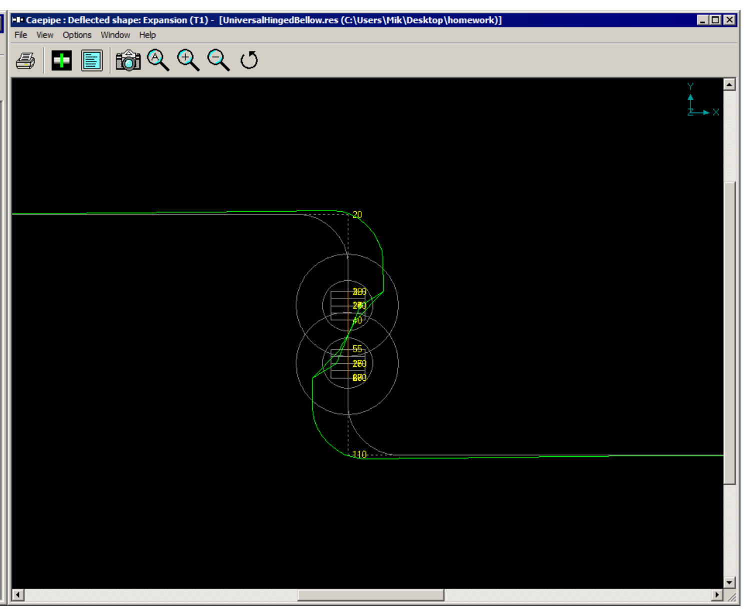








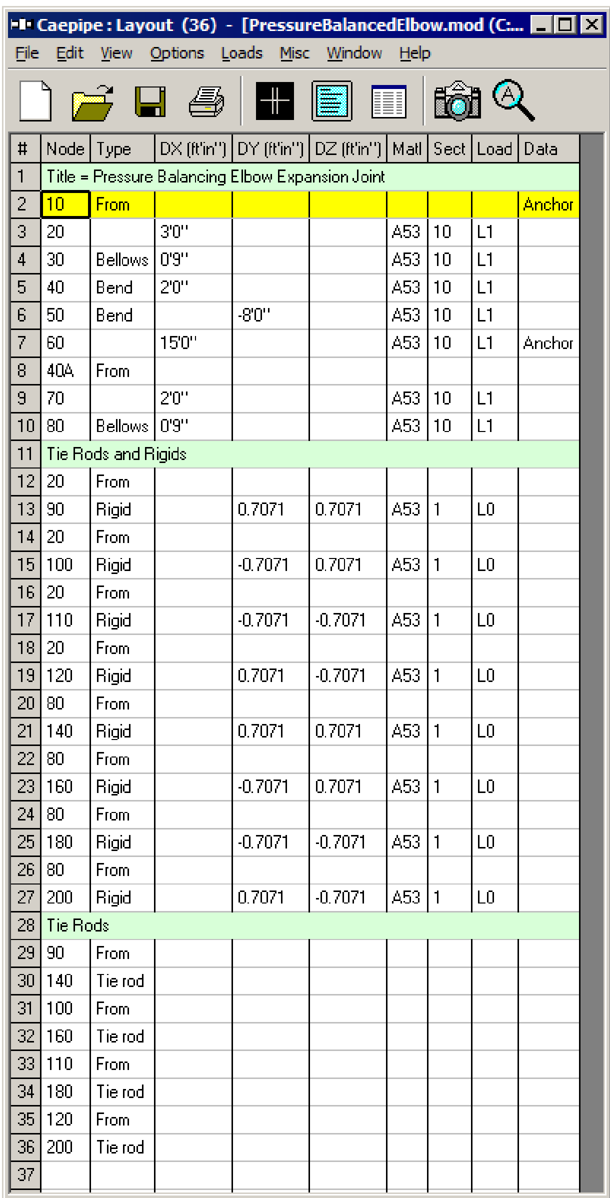
#		Displacements (global)					
	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	20A	0.262	0.033	0.000	0.0000	0.0000	0.0178
3	20B	0.291	0.014	0.000	0.0000	0.0000	0.0185
4	30	0.294	0.000	0.000	0.0000	0.0000	0.0184
5	40	0.071	0.007	0.000	0.0000	0.0000	-0.9097
6	55	-0.071	-0.007	0.000	0.0000	0.0000	-0.9097
7	60	-0.294	0.000	0.000	0.0000	0.0000	0.0184
8	110A	-0.291	-0.014	0.000	0.0000	0.0000	0.0185
9	110B	-0.262	-0.033	0.000	0.0000	0.0000	0.0178
10	120	0.000	0.000	0.000	0.0000	0.0000	0.0000
11	100	0.294	0.000	0.000	0.0000	0.0000	0.0184
12	140	0.100	0.000	0.000	0.0000	0.0000	0.0184
13	150	0.100	0.000	0.000	0.0000	0.0000	0.0184
14	160	-0.098	0.000	0.000	0.0000	0.0000	0.0184
15	170	-0.098	0.000	0.000	0.0000	0.0000	0.0184
16	180	-0.294	0.000	0.000	0.0000	0.0000	0.0184
17	220	0.294	0.000	0.000	0.0000	0.0000	0.0184
18	230	0.100	0.000	0.000	0.0000	0.0000	0.0184
19	240	0.100	0.000	0.000	0.0000	0.0000	0.0184
20	250	-0.098	0.000	0.000	0.0000	0.0000	0.0184
21	260	-0.098	0.000	0.000	0.0000	0.0000	0.0184
22	270	-0.294	0.000	0.000	0.0000	0.0000	0.0184



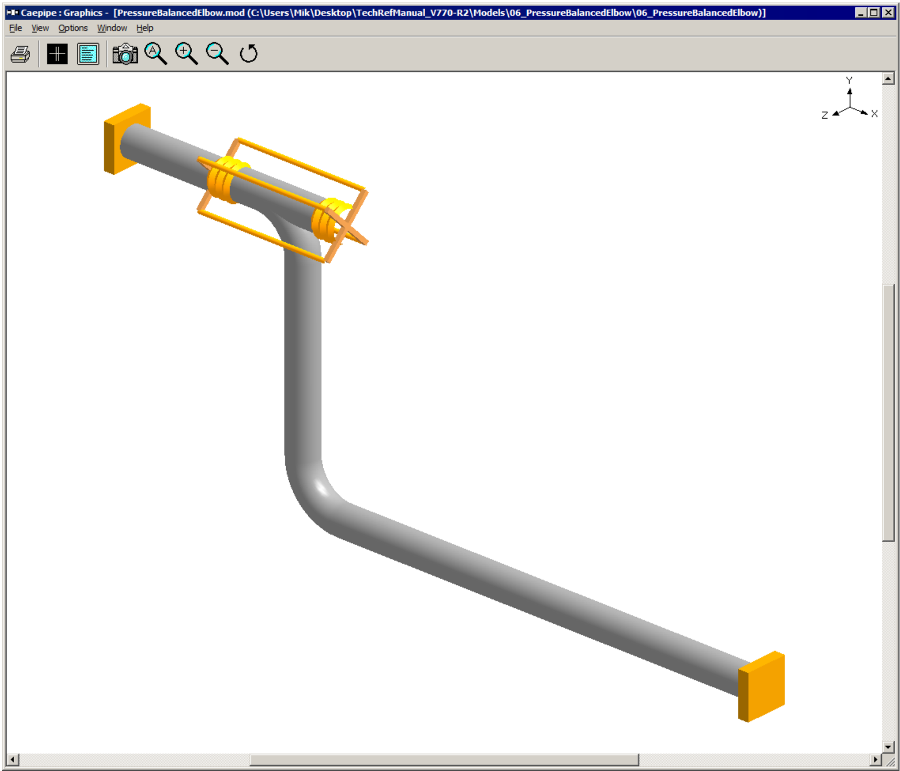
Example 6: Pressure Balanced Elbow Expansion Joint

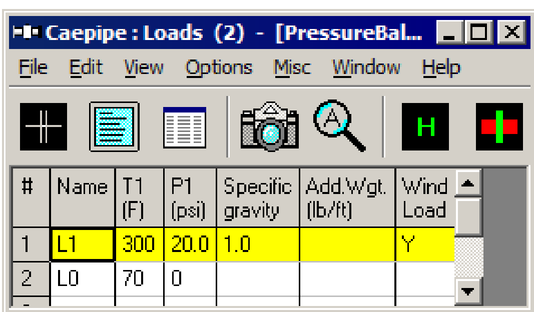
Pressure Balanced Elbow Expansion Joints can consist of a single or double bellows in the flow section, and a balancing bellow of equal area on the back side of the elbow. Tie rods attach the outboard end of the balancing bellow to the outboard end of the flow bellows. Under pressure, the tie rods are loaded with the pressure thrust force. If the flow bellows compresses in service, the balancing bellow extends by the same amount without exposing the adjacent anchors to pressure thrust forces. However, the spring forces associated with bellows movements are imposed on the adjacent equipment. A pressure balanced elbow type expansion joint can accept ***axial compression, axial extension, lateral movements and very limited angular motion***.

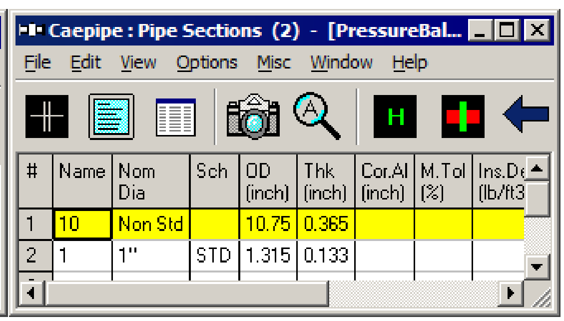
The sample model shown below simulates the Pressure Balanced Elbow Expansion Joint with Four Tie Rods using the CAEPIPE's Tie Rod elements. The stiffness of the tie rods can be input explicitly (in this case, stiffness corresponding to 1" dia tie rod is input). See snap shots below for details.

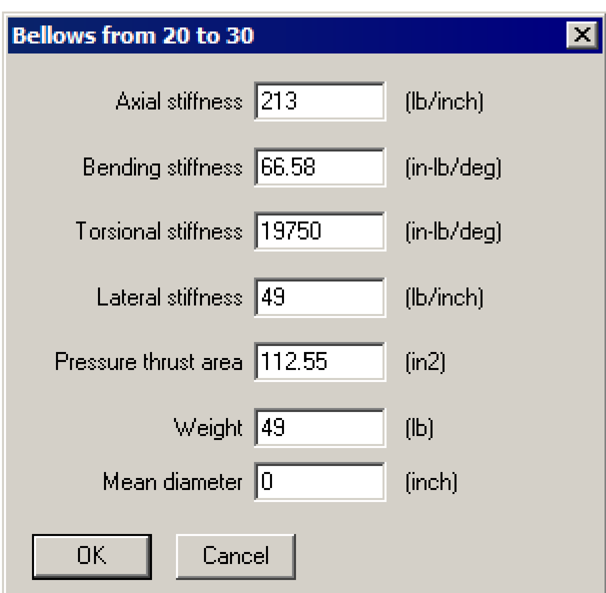


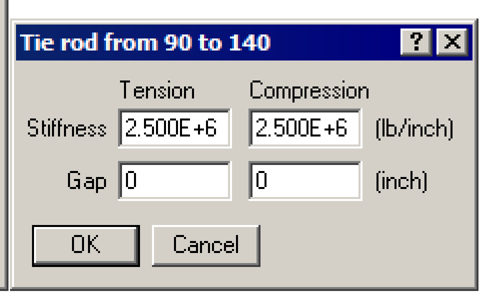
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Pressure Balancing Elbow Expansion Joint								
2	10	From							Anchor
3	20		3'0"			A53	10	L1	
4	30	Bellows	0'9"			A53	10	L1	
5	40	Bend	2'0"			A53	10	L1	
6	50	Bend		-8'0"		A53	10	L1	
7	60		15'0"			A53	10	L1	Anchor
8	40A	From							
9	70		2'0"			A53	10	L1	
10	80	Bellows	0'9"			A53	10	L1	
11	Tie Rods and Rigids								
12	20	From							
13	90	Rigid		0.7071	0.7071	A53	1	L0	
14	20	From							
15	100	Rigid		-0.7071	0.7071	A53	1	L0	
16	20	From							
17	110	Rigid		-0.7071	-0.7071	A53	1	L0	
18	20	From							
19	120	Rigid		0.7071	-0.7071	A53	1	L0	
20	80	From							
21	140	Rigid		0.7071	0.7071	A53	1	L0	
22	80	From							
23	160	Rigid		-0.7071	0.7071	A53	1	L0	
24	80	From							
25	180	Rigid		-0.7071	-0.7071	A53	1	L0	
26	80	From							
27	200	Rigid		0.7071	-0.7071	A53	1	L0	
28	Tie Rods								
29	90	From							
30	140	Tie rod							
31	100	From							
32	160	Tie rod							
33	110	From							
34	180	Tie rod							
35	120	From							
36	200	Tie rod							
37									


Caepipe : Graphics - [PressureBalancedElbow.mod (C:\Users\Mik\Desktop\TechRefManual_V770-R2\Models\06_PressureBalancedElbow\06_PressureBalancedElbow)]
File View Options Window Help
Y
Z
X


Caepipe : Loads (2) - [PressureBal...
File Edit View Options Misc Window Help
H
Name T1 (F) P1 (psi) Specific gravity Add.Wgt. (lb/ft) Wind Load
1 L1 300 20.0 1.0 Y
2 L0 70 0

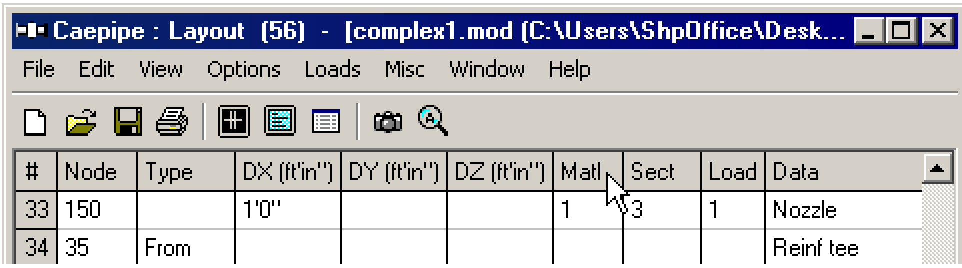

Caepipe : Pipe Sections (2) - [PressureBal...
File Edit View Options Misc Window Help
H
Name Nom Dia Sch OD (inch) Thk (inch) Cor.Al (inch) M.Tol (%) Ins.De (lb/ft3
1 10 Non Std 10.75 0.365
2 1 1" STD 1.315 0.133


Bellows from 20 to 30
Axial stiffness 213 (lb/inch)
Bending stiffness 66.58 (in-lb/deg)
Torsional stiffness 19750 (in-lb/deg)
Lateral stiffness 49 (lb/inch)
Pressure thrust area 112.55 (in2)
Weight 49 (lb)
Mean diameter 0 (inch)
OK
Cancel

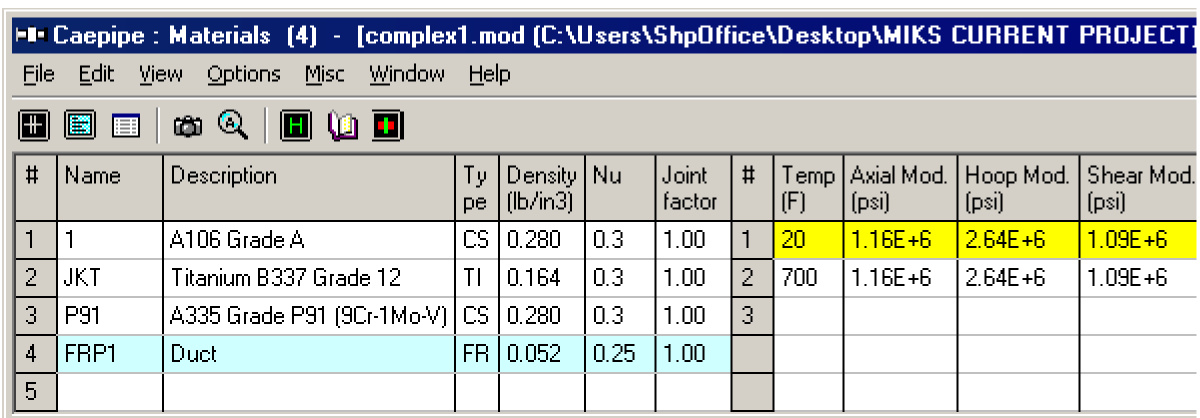

Tie rod from 90 to 140
Tension
Compression
Stiffness 2.500E+6 2.500E+6 (lb/inch)
Gap 0 0 (inch)
OK
Cancel

FRP piping has gained wide acceptance in many industries due to its lightweight nature, superior corrosion resistance, temperature capabilities and mechanical strength. Several manufacturers produce different types of FRP pipes and fittings and provide technical assistance to their customers on design matters through installation. You can model FRP materials in CAEPIPE and have it calculate deflections, forces, moments and stresses.

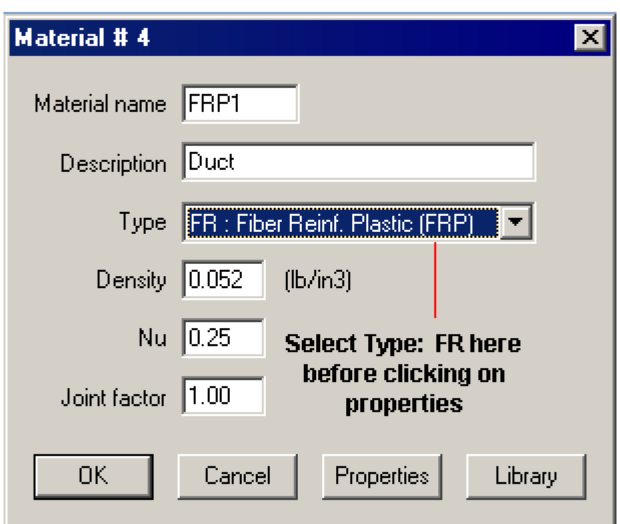
To define the FRP material, click on "Matl" in the header row in the Layout window.



In the Material List window that is shown, double click on an empty row to input a new material or on a material description to edit the material properties.



The Material dialog will be shown.



The material name can be up to five alpha-numeric characters. Enter description and density. You need to select "FR: Fiber Reinf. Plastic (FRP)" from the Type drop-down combo box before you click on the Properties button. Poisson's ratio (Nu) is a measure of the **Poisson effect**, the phenomenon in which a material tends to expand in directions perpendicular to the direction of compression. Conversely, if the material is stretched rather than compressed, it usually tends to contract in the directions transverse to the direction of stretching.

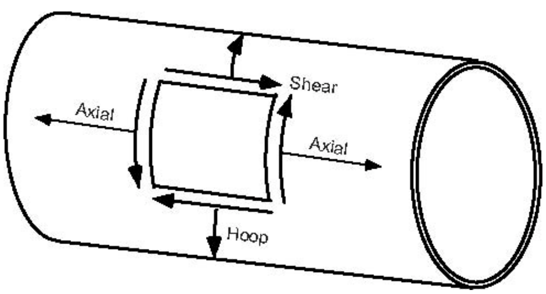
When you click on the Properties button, you are shown the table below where you enter temperature-dependent properties. Additionally, you can define the Axial and Torsional allowable stresses so that CAEPIPE can use them to compare with calculated stresses under the FRP "Sorted Stresses" results.

Material Properties

#	Temp (F)	Axial Mod. (psi)	Hoop Mod. (psi)	Shear Mod. (psi)	Alpha (in/in/F)	Axial All. (psi)	Torsional All. (psi)
1	20	1.16E+6	2.64E+6	1.09E+6	16.26E-6	3000	7500
2	700	1.16E+6	2.64E+6	1.09E+6	16.26E-6	1500	2500

FRP Material Moduli

CAEPIPE requires three moduli for a FRP material:



Axial or Longitudinal (this is the most important one)
Hoop (used in Bourdon effect calculations). If this modulus is not available, use axial modulus.
Shear or Torsional. If this modulus is not available, use engineering judgment in specifying 1/2 of axial modulus or a similar value. Note that a high modulus will result in high stresses, and a low modulus will result in high deflections.

For FRP bends, a Flexibility factor of 1.0 is used unless you override it by specifying a Flexibility factor inside the bend dialog. Also for FRP bends, CAEPIPE uses a default SIF of 2.3. You can override this too by specifying User-SIFs at the bend end nodes (A and B nodes).

Stiffness matrix

The stiffness matrix for an FRP material is formulated in the following manner:

The stiffness matrix for a pipe is calculated using the following terms:

Axial term = L / EA
Shear term = shape factor x L / GA
Bending term = L / EI
Torsion term = L / 2GI

where L = length, A = area, I = moment of inertia
E = Elastic modulus, G = shear modulus

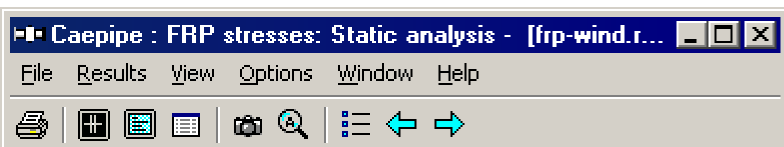
For an isotropic material,
$G = E / 2(1 + \nu)$
where ν = Poisson's ratio,

For a FRP material, however,
E = axial modulus and G is independently specified (i.e., it is not calculated using E and ν).

The hoop modulus and FRP Poisson's ratio are only used in Bourdon effect calculation where,
Poisson's ratio used = FRP Poisson's ratio input x (axial modulus / hoop modulus)

Results

CAEPIPE calculates deflections, forces, moments and stresses. Each item can be seen under the respective title in Results. FRP element stresses can be seen, sorted or unsorted. These FRP stresses are computed as per the formulae given in Section titled "Piping Code Compliance" in the Code Compliance Manual.



#	Node	Hoop (psi)	Axial (psi)	Bending (psi)	Longitudinal Max (psi)	Longitudinal Min (psi)	Torsional (psi)
1	10	11538	5500	3121	8620	2379	141
	20	11538	5500	923	6423	4577	141
2	20	11538	5500	923	6423	4577	141
	25	11538	5500	1265	6765	4235	141
3	25	11538	5500	1265	6765	4235	141
	30A	11538	5500	1814	7314	3686	141
4	30A	11538	5502	4172	9675	1330	141
	30B	11538	5497	4248	9745	1249	141
5	30B	11538	5494	1847	7341	3647	141
	50	11538	5494	1306	6801	4188	141
6	50	11538	5494	1306	6801	4188	141
	60	11538	5494	898	6392	4597	141
7	60	11538	5494	898	6392	4597	141
	70	11538	5494	3187	8681	2308	141

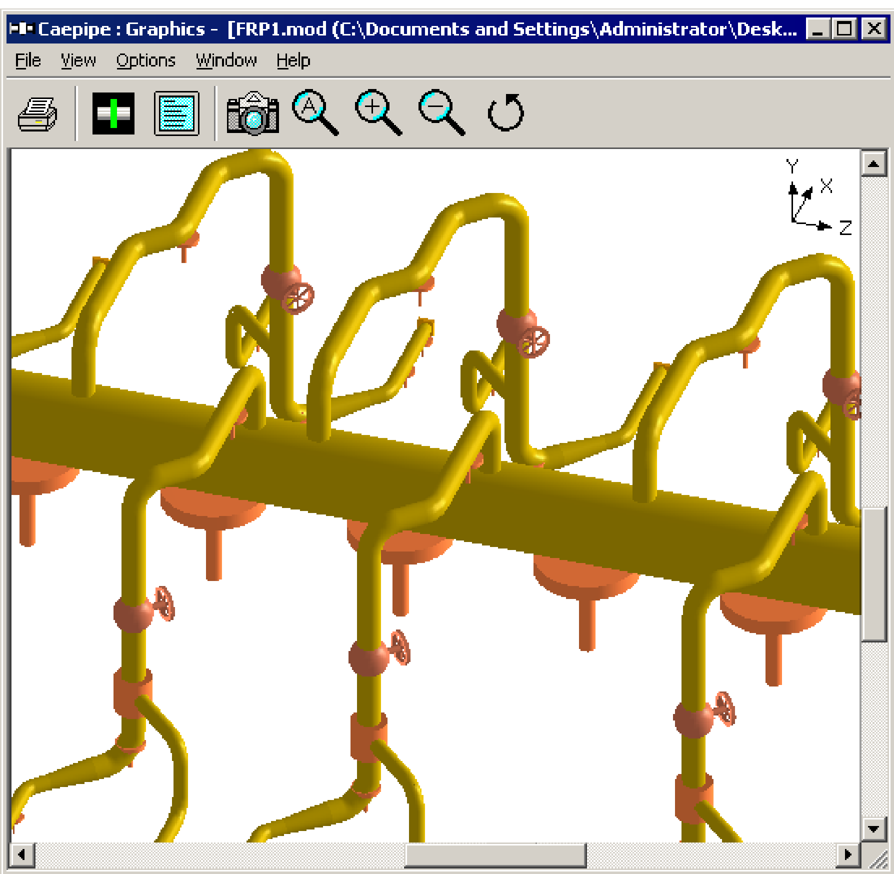
Caepipe : Sorted FRP stresses: Static analysis - [frp-wind.res

File Results View Options Window Help

#	Hoop				Max Long				Min Long				Torsion			
	Node	Stress (psi)	Allow (psi)	Stress/ Allow	Node	Stress (psi)	Allow (psi)	Stress/ Allow	Node	Stress (psi)	Allow (psi)	Stress/ Allow	Node	Stress (psi)	Allow (psi)	Stress/ Allow
1	20	11538	2500	4.62	30B	9745	2500	3.90	60	4597	2500	1.84	60	141	7500	0.02
2	20	11538	2500	4.62	30A	9675	2500	3.87	60	4597	2500	1.84	20	141	7500	0.02
3	25	11538	2500	4.62	70	8681	2500	3.47	20	4577	2500	1.83	50	141	7500	0.02
4	50	11538	2500	4.62	10	8620	2500	3.45	20	4577	2500	1.83	25	141	7500	0.02
5	50	11538	2500	4.62	30B	7341	2500	2.94	25	4235	2500	1.69	30B	141	7500	0.02
6	60	11538	2500	4.62	30A	7314	2500	2.93	25	4235	2500	1.69	10	141	7500	0.02
7	10	11538	2500	4.62	50	6801	2500	2.72	50	4188	2500	1.68	30A	141	7500	0.02
8	30B	11538	2500	4.62	50	6801	2500	2.72	50	4188	2500	1.68	30A	141	7500	0.02
9	25	11538	2500	4.62	25	6765	2500	2.71	30A	3686	2500	1.47	25	141	7500	0.02
10	60	11538	2500	4.62	25	6765	2500	2.71	30B	3647	2500	1.46	20	141	7500	0.02
11	30A	11538	2500	4.62	20	6423	2500	2.57	10	2379	2500	0.95	60	141	7500	0.02
12	70	11538	2500	4.62	20	6423	2500	2.57	70	2308	2500	0.92	50	141	7500	0.02
13	30A	11538	2500	4.62	60	6392	2500	2.56	30A	1330	2500	0.53	70	141	7500	0.02
14	30B	11538	2500	4.62	60	6392	2500	2.56	30B	1249	2500	0.50	30B	141	7500	0.02

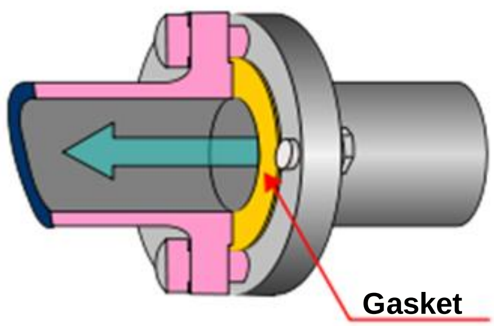
Axial and Torsional allowables may be entered under material properties so that they can be used to compare against calculated stresses in "Sorted FRP Stresses." Forces, Stresses and Sorted stresses for FRP piping may be printed to a .CSV file (spreadsheet-compatible).

CAEPIPE renders FRP piping in golden color.

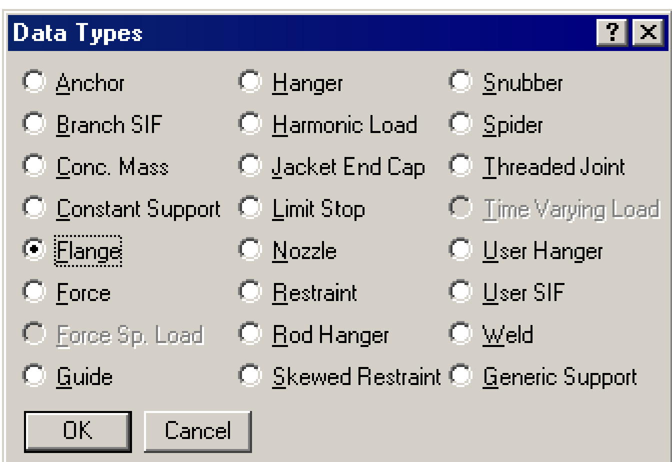


Flange

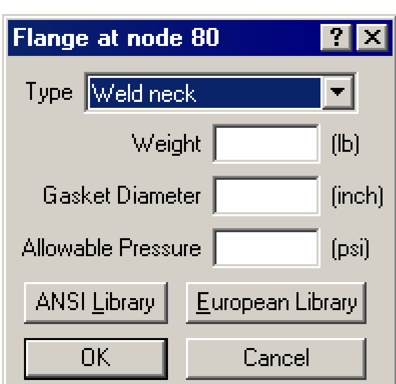
A flange is a method of connecting pipes, valves, pumps and other equipment to form a piping system. It also provides easy access for cleaning, inspection or modification. Flanges are usually welded or screwed. Flanged joints are made by bolting together two flanges with a gasket between them to provide a seal. The material of a flange, is basically set during the choice of the pipe, in most cases, a flange is of the same material as the pipe. There are many different flange standards being followed worldwide. To allow easy functionality and inter-changeability, these are designed to have standardized dimensions. Common world standards include ASA/ANSI/ASME (USA), PN/DIN (European), BS10 (British/Australian), and JIS/KS (Japanese/Korean).



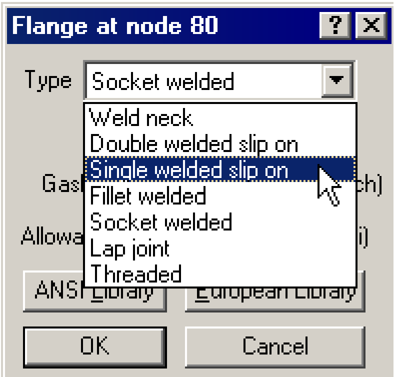
A flange is input by typing “fl” in the Data column or selecting “Flange” from the Data Types dialog. If flanges are located at the bend end nodes (A, B nodes), or jacket bend nodes (C, D nodes), the bend flexibility and SIF are internally modified in CAEPIPE.



The Flange dialog is shown.



Several flange types are available – weld neck, socket welded, threaded, lap joint, etc. Use the Type drop-down combo box to select one.



Weight

The weight you provide should be the total weight of flanges, i.e., if there are two flanges the weight should be the weight of two flanges.Weight is to be input in lbf or kgf and NOT in mass units. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

Gasket Diameter

The gasket diameter is used in calculating equivalent flange pressure in the flange report.

As stated in Section titled "Flange Report" below, Gasket Diameter (G) required is the "diameter at location of gasket load reaction". This Gasket Diameter (G) can be calculated as detailed in "Flange Report" section.

Allowable Pressure

CAEPIPE provides an approximation for the tendency of a flange to leak by calculating an "equivalent flange pressure" and comparing it to the (user-input) allowable pressure for the flange in the flange report. Often, the allowable pressure may be conservatively set to the flange rating. The allowable pressure can be taken from B16.5 (or a similar standard) for the flange class, material, pressure and temperature.

The temperature-pressure ratings provided in ASME/ANSI B16.5 are computed using the formula given in para. D2.1 of Annex D of ASME/ANSI B16.5 (given below). The values thus obtained are listed in a tabular form for all materials at different flange ratings.

$P_T = (P_r \text{ x } S_I) / 8750 <= P_c$

where,

P_c = Ceiling pressure as specified in D3 of Annex D at temperature.

P_T = Rated working pressure in psig for specified material at temperature.

P_r = Pressure rating as per Class in Psig.

S_I = Selected stress in Psig for specified material at temperature.

PIPE FLANGES AND FLANGED FITTINGS ASME B16.5a-1998

TABLES 2
PRESSURE–TEMPERATURE RATINGS FOR GROUPS 1.1 THROUGH 3.17 MATERIALS

TABLE 2-1.1 RATINGS FOR GROUP 1.1 MATERIALS

Nominal Designation	Forgings	Castings	Plates
C–Si	A 105 (1)	A 216 Gr. WCB (1)	A 515 Gr. 70 (1)
C–Mn–Si	A 350 Gr. LF2 (1)		A 516 Gr. 70 (1)(2) A 537 Cl. 1 (3)
C–Mn–Si–V	A 350 Gr. LF6 Cl. 1 (4)		

NOTES:
(1) Upon prolonged exposure to temperatures above 800°F, the carbide phase of steel may be converted to graphite. Permissible, but not recommended for prolonged use above 800°F.
(2) Not to be used over 850°F.
(3) Not to be used over 700°F.
(4) Not to be used over 500°F.

WORKING PRESSURES BY CLASSES, psig

Class Temp., °F	150	300	400	600	900	1500	2500
–20 to 100	285	740	990	1480	2220	3705	6170
200	260	675	900	1350	2025	3375	5625
300	230	655	875	1315	1970	3280	5470
400	200	635	845	1270	1900	3170	5280
500	170	600	800	1200	1795	2995	4990
600	140	550	730	1095	1640	2735	4560
650	125	535	715	1075	1610	2685	4475
700	110	535	710	1065	1600	2665	4440
750	95	505	670	1010	1510	2520	4200
800	80	410	550	825	1235	2060	3430
850	65	270	355	535	805	1340	2230
900	50	170	230	345	515	860	1430
950	35	105	140	205	310	515	860
1000	20	50	70	105	155	260	430

For example, from the values shown in Table 2.-1.1 of ASME B16.5 (1998) (shown above), for a 300# carbon steel flange with Material A105, the allowable pressure is 740 psig at 100°F. This is calculated as detailed below.

Allowable Stress = Minimum (60% of Min. Yield, 1.25 x Allowable Stress at Temperature) as per para. D2.2 of Annex D of ASME B16.5 (1998).

Accordingly, Allowable Stress (S_I) from ASME Boiler and Pressure Vessel Code, Section II, Part D for A105 at Temperature 100 deg. F = Minimum(60% of Min. Yield, 1.25 x Allowable Stress at Temp) = Minimum(60% of 36000, 1.25 x 20000) = Minimum(21600, 25000) = 21600 psi

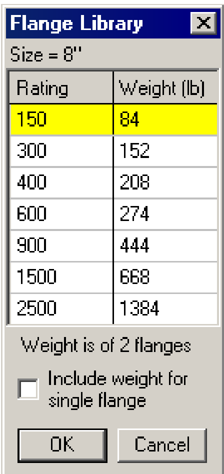
Pressure rating (P_r) = 300 #

So, rated working pressure (P_T) = (21600 x 300) / 8750 = 740 psi (same as the value shown in the snapshot).

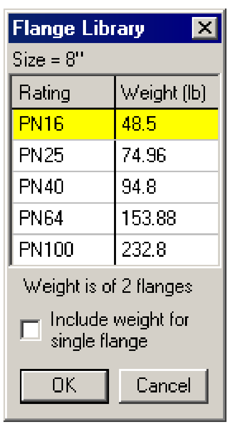
Flange Library

You may access the flange library by clicking on the Library button of the flange dialog. The default weight in the Flange Library is for two Flanges. However, each library dialog has an option to include weight for a single flange. See checkbox in the Flange library dialog below to include weight for a single flange.

ANSI Library



European Library



The default weight in the library is the weight of two weld neck flanges (including bolts).

Flange Report

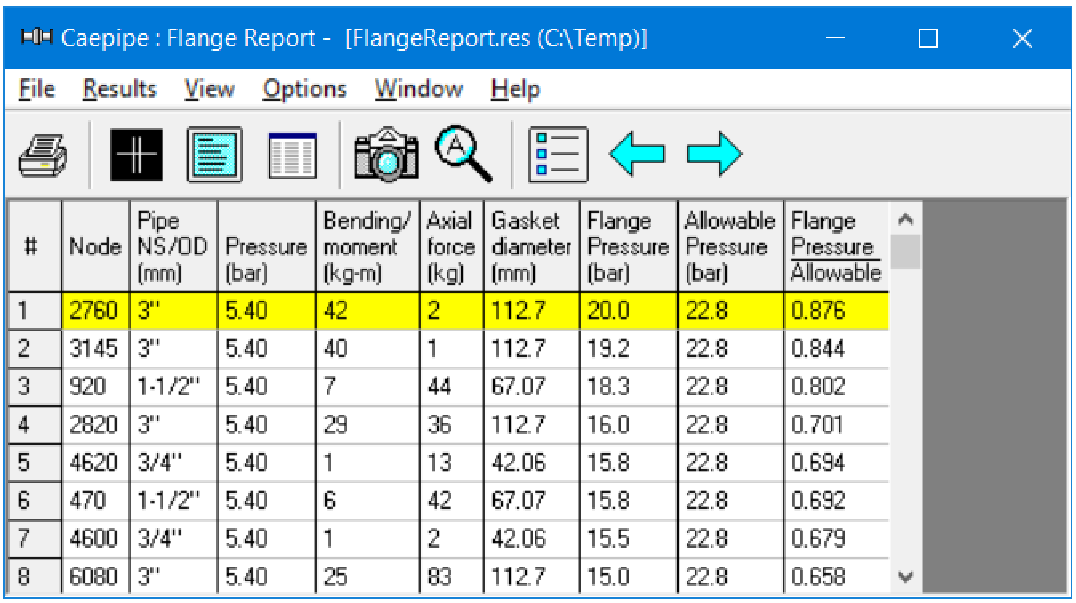
CAEPIPE lists every flange in a model in the flange report. The “Flange Pressure” is an equivalent pressure calculated from the actual pressure in the piping element, the bending moment and the axial force on the flange from the operating case(s), as follows:

For Piping codes such as BS 806, IGEM, Norwegian, RCC-M, CODETI, Stoomwezen, Swedish and EN 13480-3, equivalent pressure is calculated in accordance with Eq. 6.6.2-1 of EN 13480-3 (2020) as given below.

$$Flange\ Pressure = Pressure + \frac{16 \times BendingMoment}{\pi \times G^3} + \frac{4 \times AxialForce}{\pi \times G^2}$$

where,

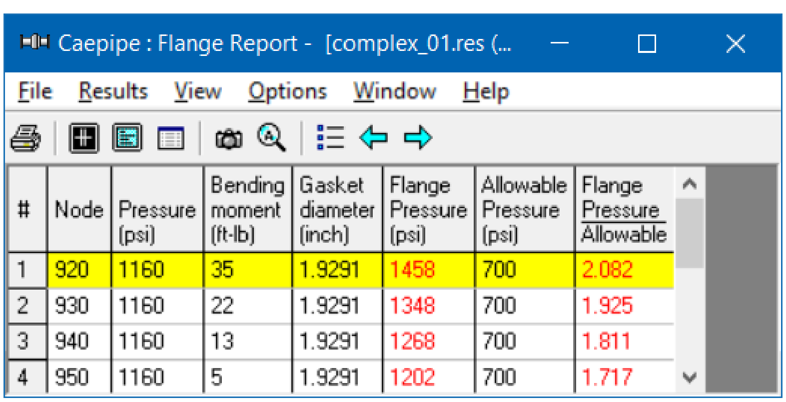
Bending Moment = resulting bending moment



#	Node	Pipe NS/OD (mm)	Pressure (bar)	Bending/ moment (kg-m)	Axial force (kg)	Gasket diameter (mm)	Flange Pressure (bar)	Allowable Pressure (bar)	Flange Pressure / Allowable
1	2760	3"	5.40	42	2	112.7	20.0	22.8	0.876
2	3145	3"	5.40	40	1	112.7	19.2	22.8	0.844
3	920	1-1/2"	5.40	7	44	67.07	18.3	22.8	0.802
4	2820	3"	5.40	29	36	112.7	16.0	22.8	0.701
5	4620	3/4"	5.40	1	13	42.06	15.8	22.8	0.694
6	470	1-1/2"	5.40	6	42	67.07	15.8	22.8	0.692
7	4600	3/4"	5.40	1	2	42.06	15.5	22.8	0.679
8	6080	3"	5.40	25	83	112.7	15.0	22.8	0.658

For all other Piping codes, equivalent pressure is calculated in accordance with NC.3658.3 of ASME Section III Class 2 (2017) as given below.

$$Flange\ Pressure = Pressure + \frac{16 \times M_{fs}}{\pi \times G^3}$$



#	Node	Pressure (psi)	Bending moment (ft-lb)	Gasket diameter (inch)	Flange Pressure (psi)	Allowable Pressure (psi)	Flange Pressure / Allowable
1	920	1160	35	1.9291	1458	700	2.082
2	930	1160	22	1.9291	1348	700	1.925
3	940	1160	13	1.9291	1268	700	1.811
4	950	1160	5	1.9291	1202	700	1.717

The Gasket diameter is at the gasket loading location (if it is not input, it is conservatively assumed to be the internal diameter of the pipe).“Gasket loading location” is the location where the gasket load reaction is acting. As per ASME Sec. VIII Division 1, “G” is the diameter at location of gasket load reaction and can be calculated as follows. See snap shots shown below for details.

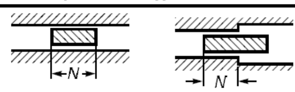
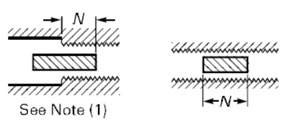
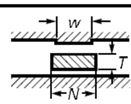
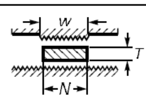
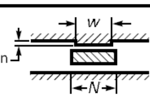
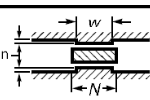
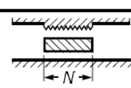
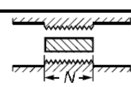
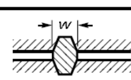
G = mean diameter of gasket contact face, inches, when $b_0 <= ¼$ inches or G = outside diameter of gasket contact face - 2b, inches, when $b_0 > ¼$ inches.

Mfs = Maximum(Resultant Bending Moment, Torsional Moment)

where,

b = effective gasket or joint-contact-surface seating width, inches.

b_0 = basic gasket seating width, inches.

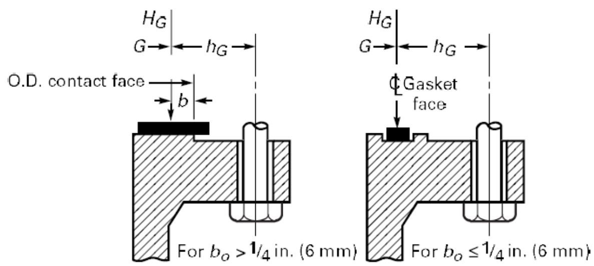
Facing Sketch (Exaggerated)	Basic Gasket Seating Width b_o Column I	Basic Gasket Seating Width b_o Column II
(1a) 	$\frac{N}{2}$	$\frac{N}{2}$
(1b)  See Note (1)		
(1c)  $w \le N$	$\frac{w+T}{2}$; $\left(\frac{w+N}{4}\text{ max}\right)$	$\frac{w+T}{2}$; $\left(\frac{w+N}{4}\text{ max}\right)$
(1d)  See Note (1) $w \le N$		
(2) 1/64 in. nubbin  $w \le N/2$	$\frac{w+N}{4}$	$\frac{w+3N}{8}$
(3) 1/64 in. nubbin  $w \le N/2$	$\frac{N}{4}$	$\frac{3N}{8}$
(4)  See Note (1)	$\frac{3N}{8}$	$\frac{7N}{16}$
(5)  See Note (1)	$\frac{N}{4}$	$\frac{3N}{8}$
(6) 	$\frac{w}{8}$	. . .

EFFECTIVE GASKET WIDTH[2] (CONT'D)

Effective Gasket Seating Width, b

$b = b_o$ when $b_o \le 1/4$ in. (6 mm); $b = 0.5\sqrt{b_o}$ when $b_o > 1/4$ in. (6 mm)

Location of Gasket Load Reaction



In the absence of seating width data, G can be taken as the mean diameter[= (ID + OD) / 2]. For example, as per the snap shot shown below, the mean diameter G for the "Ring Gaskets for 250/300# ANSI Pipe Flanges" for 8" is 10.375" [= (8.625" + 12.125") / 2].

Ring Gaskets for 250/300# ANSI Pipe Flanges

Nominal Pipe Size	ID	OD
1/2"	27/32	2-1/8
3/4"	1-1/16	2-5/8
1"	1-5/16	2-7/8
1-1/4"	1-21/32	3-1/4
1-1/2"	1-29/32	3-3/4
2"	2-3/8	4-3/8
2-1/2"	2-7/8	5-1/8
3"	3-1/2	5-7/8
3-1/2"	4	6-1/2
4"	4-1/2	7-1/8
4-1/2"	5	7-5/8
5"	5-9/16	8-1/2
6"	6-5/8	9-7/8
7"	7-5/8	10-7/8

Nominal Pipe Size	ID	OD
8"	8-5/8	12-1/8
10"	10-3/4	14-1/4
12"	12-3/4	16-5/8
14"	14	19-1/8
15"	15	20-1/8
16"	16	21-1/4
18"	18	23-1/2
20"	20	25-3/4
22"	22	27-5/8
24"	24	30-1/2
30"	30	37-1/2
36"	36	44
42"	42	50-3/4
48"	48	58-3/4

The computed equivalent flange pressure is compared with the flange allowable pressure.

If you have input more than one temperature load, the flange equivalent pressure is calculated for all the applicable operating load cases, the worst of which is reported in the Flange report.

A flange report is generated even when no piping code is chosen. The flange report is shown in the results.

Suggestions for dealing with high equivalent flange pressure to allowable ratios

The Flange report in the CAEPIPE results window shows the loads at each flange location for the worst operating load case (W+P+T).

The "equivalent" flange pressure is the sum of two terms from the flange equation as shown above. The last column in the Flange report shows a ratio of this equivalent flange pressure to a user-input allowable pressure. This ratio is flagged in red when greater than 1.0.

Ensure that you input an allowable pressure for the flange by looking up B16.5 or a similar code (as a function of design temperature and pressure).

Since a flange is unlikely to fail by collapse, the key idea of the flange report is to "quantify" the tendency of a flange to leak its contents. Engineering judgment will play an important role in interpreting this report.

If the ratio of equivalent flange pressure to allowable pressure is flagged in red, then try to reduce the bending moment at that flange location. Be sure to examine all load cases, but frequently the excessive moments come from the expansion case. If so, consider introducing loops, bends and offsets as required to reduce the bending moments at flange locations for expansion load case(s).If it is practical, move those flanges with high (flange pressure-to-allowable) ratios to piping locations where the bending moments are less.

Flange joints are essential components in all pressurized systems; they are also one of the most complex. Many factors are involved in determining the successful design and operation of a bolted flange joint service, namely, the interaction between the bolting, flange, and gasket as well as important non-linear variables such as friction and gasket properties. The Pressure Vessel and Piping Codes were developed with safety in mind; they provide a method for sizing the flange and bolts to be structurally adequate for the specified design conditions.

The Flange Qualification module implemented in CAEPIPE addresses the design rules contained in the ASME Section VIII, Division 1, Appendix 2 on bolted flange connections with gaskets.

These design rules will help you to obtain better insight into a flange joint's tendency to leak, beyond that provided by the rudimentary (yet indicative) flange report produced by a piping analysis, as seen in the previous section. You can examine the flange and bolt stresses arising from the bolt tightening loads required for a leakage-free joint.

The Flange Qualification module assumes that you already have flanges and gaskets picked out for your system and performed a piping flexibility analysis of that system with CAEPIPE, which will have produced a flange report as given above in the previous section depending upon the piping code selected.

Note that this Flange Qualification module to calculate flange and bolt stresses is separate from a piping stress model file and can be accessed from File Menu > Open/New command.

The Flange Qualification module performs three (3) qualifications namely,

1. Flange Allowable Moment as per NC3658.3 of ASME Section III Class 2,

2. Flange Stresses for Operating Case as per Appendix 2 of ASME Section VIII Division 1, and

3. Flange Stresses for Gasket Seating Case as per Appendix 2 of ASME Section VIII Division 1.

Out of the three (3) qualifications listed above, Bending / Torsional Moment entered in “Flange Qualification” module is used ONLY in calculating the Allowable Flange Moment as per NC3658.3 of ASME Section III Class 2. The same results are also shown in “Flange report” under CAEPIPE Results. The equation involved in Flange Joint analysis is given below.

Note: Flange Joint analysis as per NC 3658.3 is valid for ASME B16.5 flanges with Bolt Stress at 100 deg. F is greater than or equal to 20000 psi (138 MPa). So, please ensure that the Bolt Stresses at 100 deg.F is greater than or equal to 20000 psi.

(U.S. Customary Units)

$$M_{fs} \le 3,125\left(S_y / 36,000\right) C A_b$$

(SI Units)

$$M_{fs} \le 21.7\left(S_y / 250\right) C A_b$$

A_b = total cross-sectional area of bolts at root of thread of section of least diameter under stress, in2 or mm2

C = bolt circle diameter, in or mm.

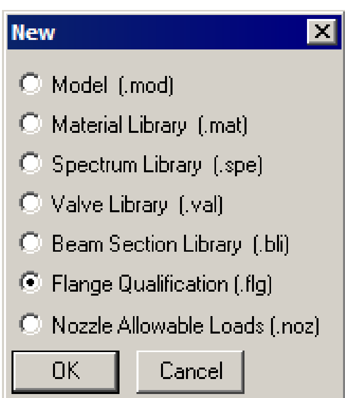
S_y = Yield stress of flange material at Design temperature = 1.5 * Flange Allowable Stress at Design Temperature (assumed inside CAEPIPE, as Allowable Stress is generally 2/3 of Yield Stress at temperatures well below creep)

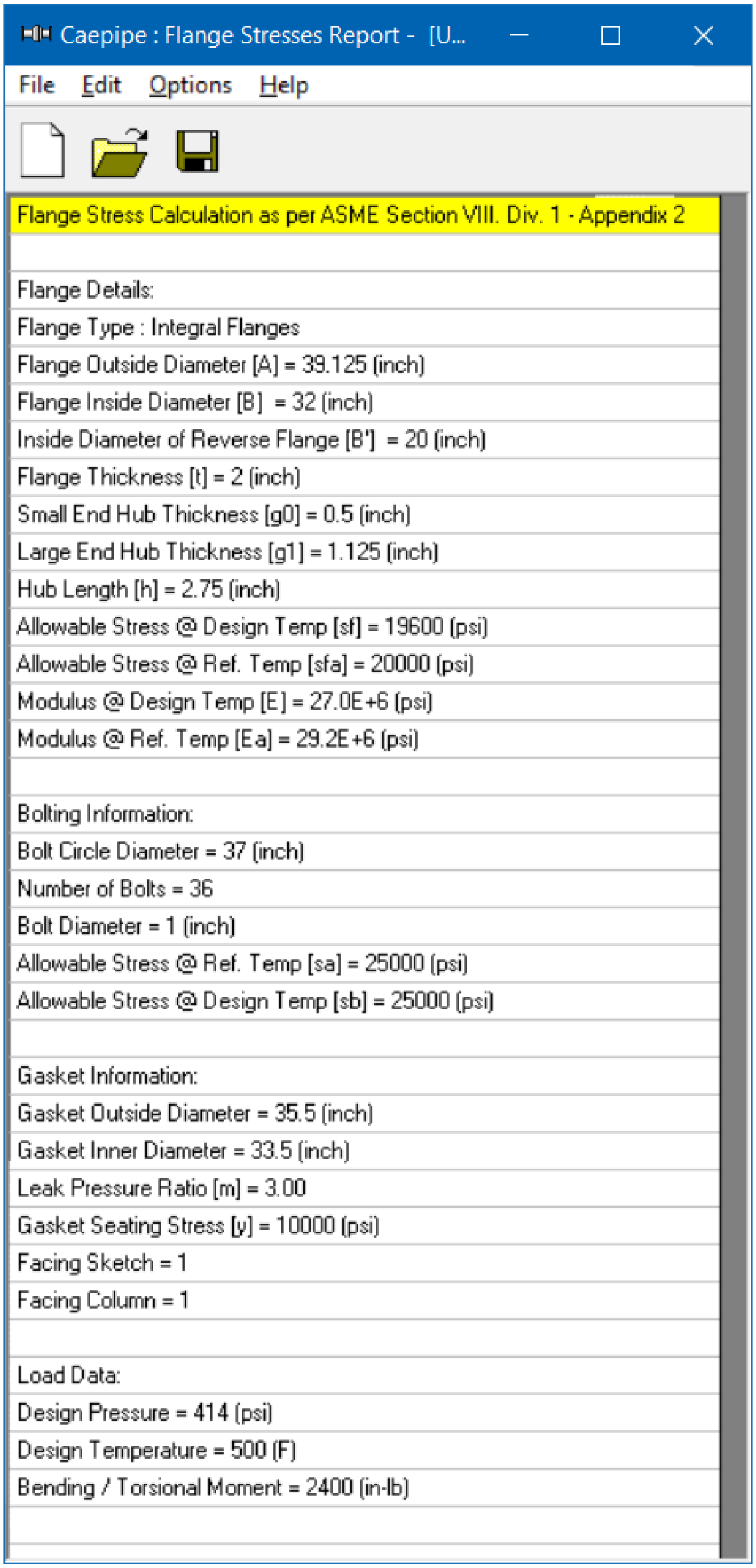
M_{fs} = bending or torsional moment applied to the joint due to weight, thermal expansion of the piping, sustained anchor movement, relief valve steady-state thrust and other sustained mechanical loads applied to the flanged joint during the design or service condition, in-lb or N.mm.

On the other hand, Flange Stresses and Flange Rigidity Factors computed as per ASME Section VIII Division 1 Appendix 2 are independent of Axial Force and Bending Moment as observed from the detailed write-up given in the Section titled "Flange Qualification" of CAEPIPE Code Compliance Manual.

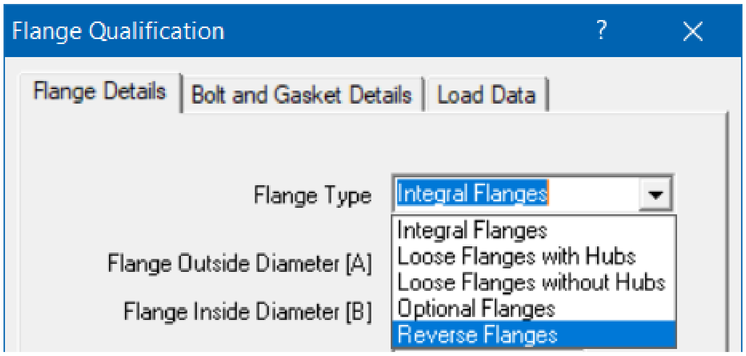
Because this module accepts bending / torsional moment at a flange as input among many others, you will need to first create in CAEPIPE your pipe stress model that includes flanges (which you need to validate) and generate a Flange Report as shown above. Such a report will contain the information you can now use in the Flange Qualification module to calculate flange and bolt stresses.

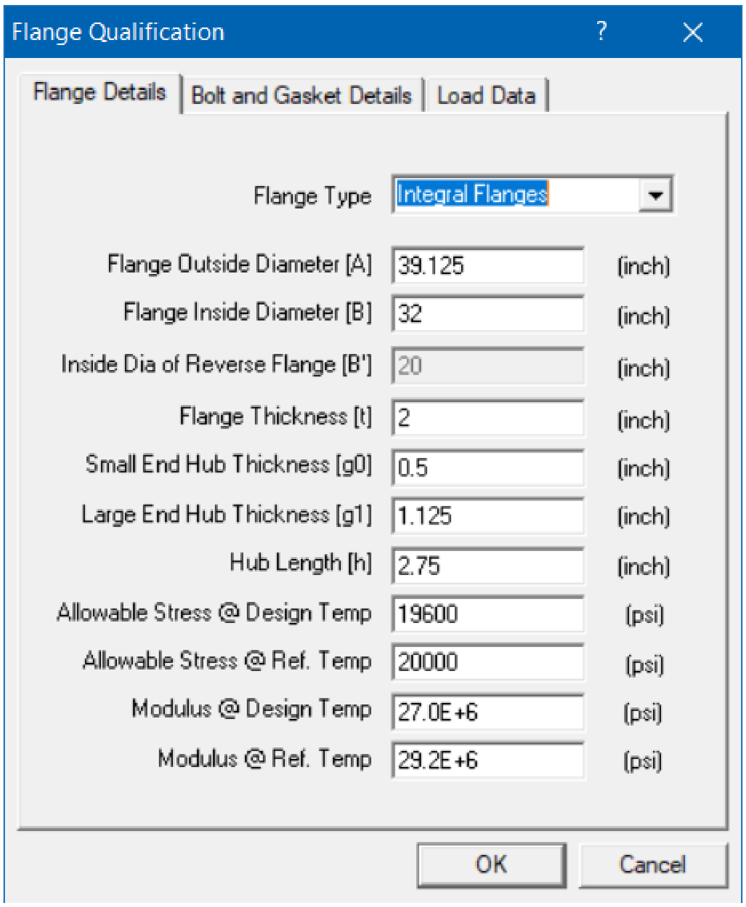
When you first create a new Flange Qualification file, it comes populated with default values for a sample flange (see example 1 later on in this topic).



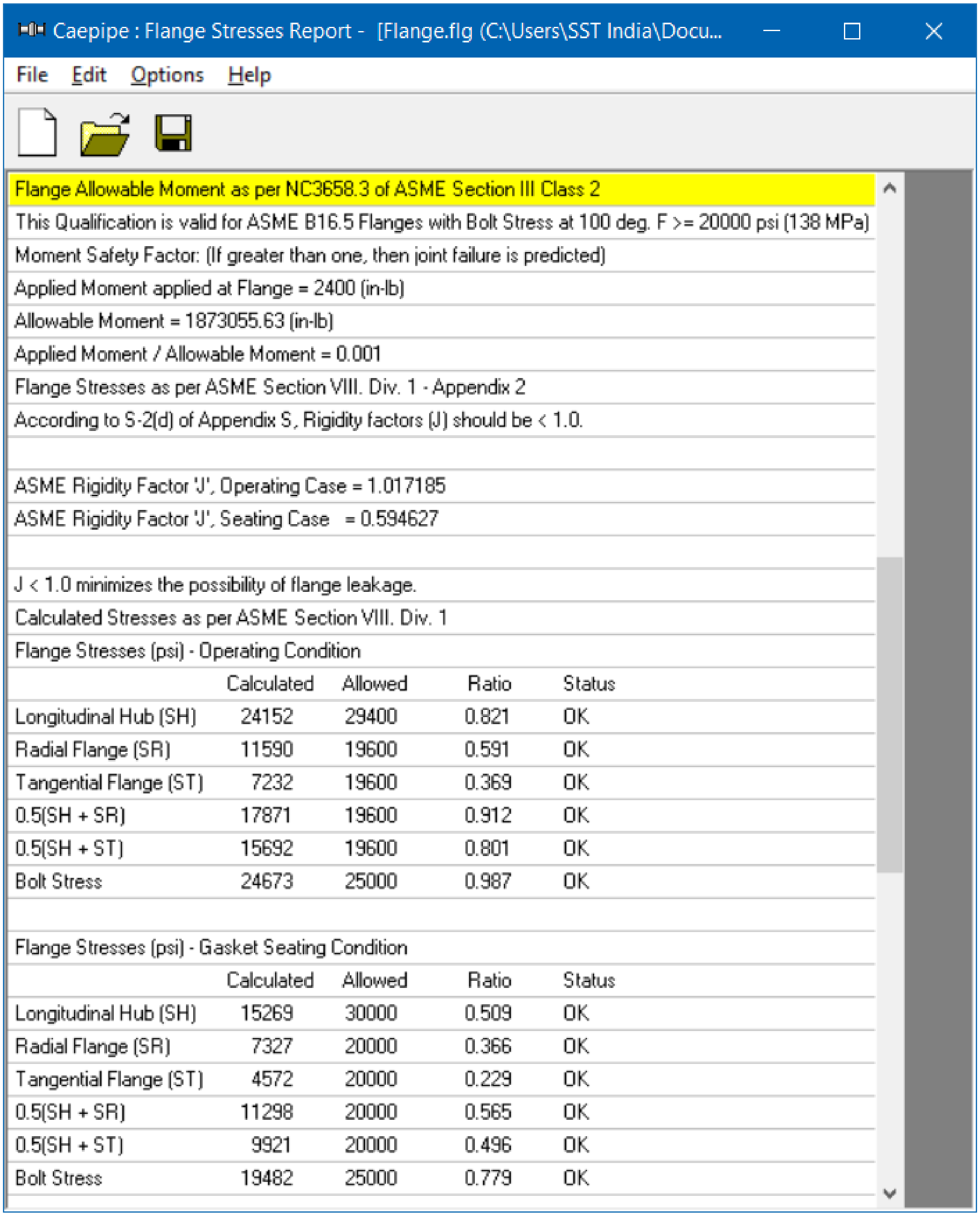


Double-clicking anywhere in the previous screen (or Edit menu > Edit (Ctrl+E)) opens a dialog with input fields (with default values) you can edit. You will need to enter all of your flange data in this dialog. The different parameters you see here are explained in detail in the Section titled "Flange Qualification" of the Code Compliance manual.





Required flange input information is organized into three Property tabs – Flange Details, Bolt and Gasket Details, and Load Data, the last of which accepts data from a piping model's Flange Report. Once all the data is input, save the model (Flange Qualification filenames will have a .flg extension). Now, select File menu > Analyze to calculate flange stresses, which will be shown *right below* the input information.



Flange Allowable Moment as per NC3658.3 of ASME Section III Class 2

This Qualification is valid for ASME B16.5 Flanges with Bolt Stress at 100 deg. F >= 20000 psi (138 MPa)

Moment Safety Factor: (If greater than one, then joint failure is predicted)

Applied Moment applied at Flange = 2400 (in-lb)

Allowable Moment = 1873055.63 (in-lb)

Applied Moment / Allowable Moment = 0.001

Flange Stresses as per ASME Section VIII. Div. 1 - Appendix 2

According to S-2(d) of Appendix S, Rigidity factors (J) should be < 1.0.

ASME Rigidity Factor 'J', Operating Case = 1.017185

ASME Rigidity Factor 'J', Seating Case = 0.594627

J < 1.0 minimizes the possibility of flange leakage.

Calculated Stresses as per ASME Section VIII. Div. 1

Flange Stresses (psi) - Operating Condition

	Calculated	Allowed	Ratio	Status
Longitudinal Hub (SH)	24152	29400	0.821	OK
Radial Flange (SR)	11590	19600	0.591	OK
Tangential Flange (ST)	7232	19600	0.369	OK
0.5(SH + SR)	17871	19600	0.912	OK
0.5(SH + ST)	15692	19600	0.801	OK
Bolt Stress	24673	25000	0.987	OK

Flange Stresses (psi) - Gasket Seating Condition

	Calculated	Allowed	Ratio	Status
Longitudinal Hub (SH)	15269	30000	0.509	OK
Radial Flange (SR)	7327	20000	0.366	OK
Tangential Flange (ST)	4572	20000	0.229	OK
0.5(SH + SR)	11298	20000	0.565	OK
0.5(SH + ST)	9921	20000	0.496	OK
Bolt Stress	19482	25000	0.779	OK

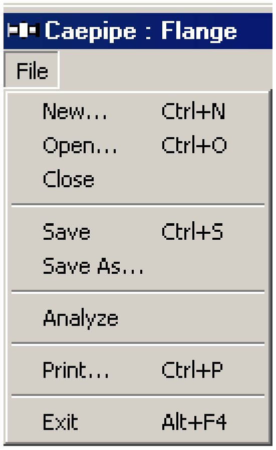
There are three main sections in the shown results:

- Flange Equivalent Pressure (same as the one shown in piping model results > Flange Report),
- Flange stresses in the Operating condition, and
- Flange stresses in the Gasket seating condition

Flange Qualification Module Menus

Use these numbers in accordance with the ASME Section VIII publication and your engineering judgment to qualify these flanges for use in your piping systems.

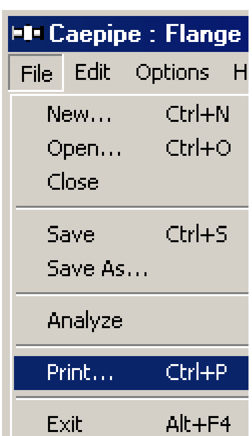
File Menu



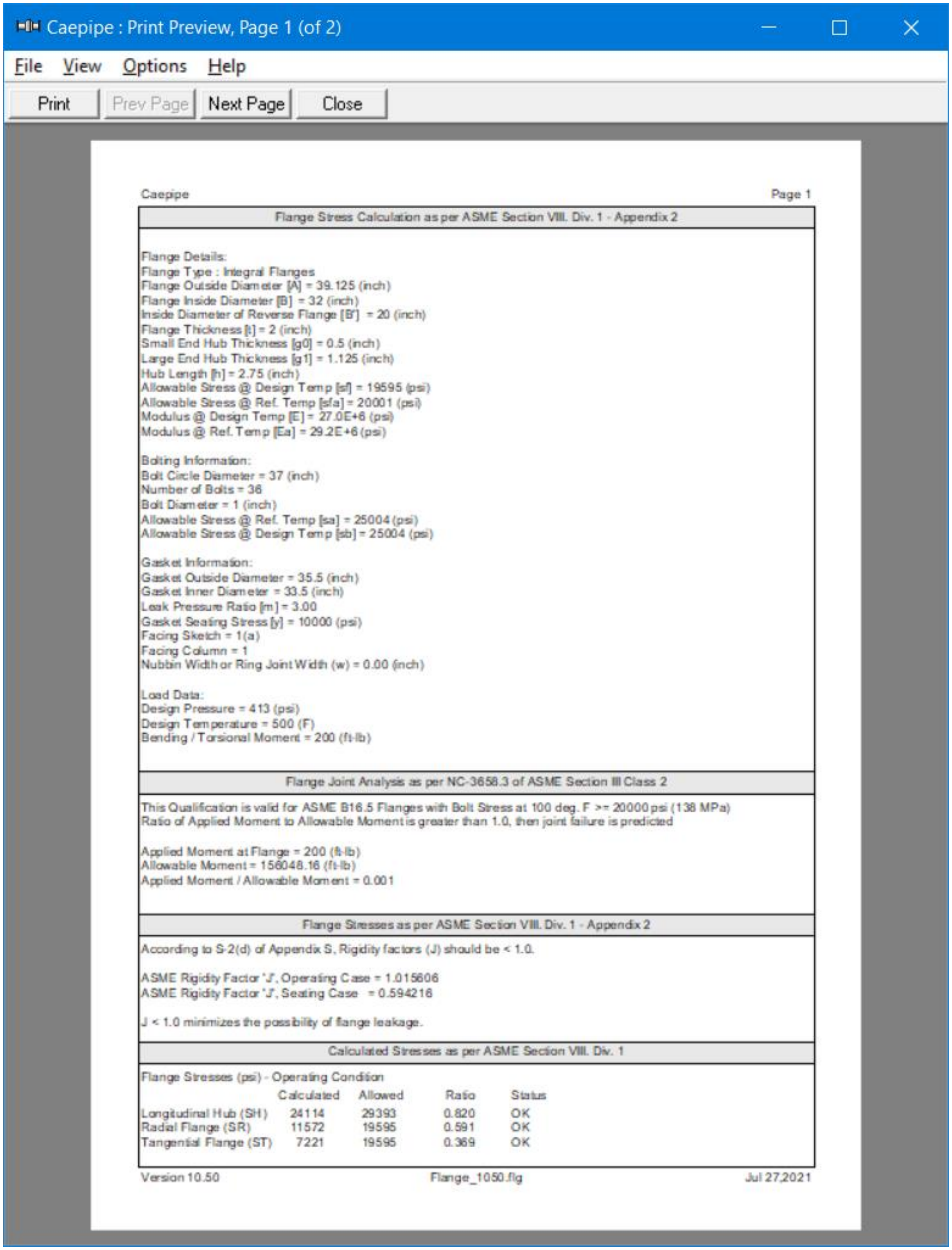
.Analyze.

Analyze command calculates the flange and bolt stresses and compares them to the input allowable stresses.

Print.



You can print a Flange Report by using the Print command. You can also preview the report by clicking the Preview button on the print dialog.



Caepipe Page 1

Flange Stress Calculation as per ASME Section VIII. Div. 1 - Appendix 2

Flange Details:
Flange Type : Integral Flanges
Flange Outside Diameter [A] = 39.125 (inch)
Flange Inside Diameter [B] = 32 (inch)
Inside Diameter of Reverse Flange [B'] = 20 (inch)
Flange Thickness [t] = 2 (inch)
Small End Hub Thickness [g0] = 0.5 (inch)
Large End Hub Thickness [g1] = 1.125 (inch)
Hub Length [h] = 2.75 (inch)
Allowable Stress @ Design Temp [sf] = 19595 (psi)
Allowable Stress @ Ref. Temp [sfa] = 20001 (psi)
Modulus @ Design Temp [E] = 27.0E+6 (psi)
Modulus @ Ref. Temp [Ea] = 29.2E+6 (psi)

Bolting Information:
Bolt Circle Diameter = 37 (inch)
Number of Bolts = 36
Bolt Diameter = 1 (inch)
Allowable Stress @ Ref. Temp [sa] = 25004 (psi)
Allowable Stress @ Design Temp [sb] = 25004 (psi)

Gasket Information:
Gasket Outside Diameter = 35.5 (inch)
Gasket Inner Diameter = 33.5 (inch)
Leak Pressure Ratio [m] = 3.00
Gasket Seating Stress [y] = 10000 (psi)
Facing Sketch = 1(a)
Facing Column = 1
Nubbin Width or Ring Joint Width (w) = 0.00 (inch)

Load Data:
Design Pressure = 413 (psi)
Design Temperature = 500 (F)
Bending / Torsional Moment = 200 (ft-lb)

Flange Joint Analysis as per NC-3658.3 of ASME Section III Class 2

This Qualification is valid for ASME B16.5 Flanges with Bolt Stress at 100 deg. F >= 20000 psi (138 MPa)
Ratio of Applied Moment to Allowable Moment is greater than 1.0, then joint failure is predicted

Applied Moment at Flange = 200 (ft-lb)
Allowable Moment = 156048.16 (ft-lb)
Applied Moment / Allowable Moment = 0.001

Flange Stresses as per ASME Section VIII. Div. 1 - Appendix 2

According to S-2(d) of Appendix S, Rigidity factors (J) should be < 1.0.

ASME Rigidity Factor 'J', Operating Case = 1.015606
ASME Rigidity Factor 'J', Seating Case = 0.594216

J < 1.0 minimizes the possibility of flange leakage.

Calculated Stresses as per ASME Section VIII. Div. 1

Flange Stresses (psi) - Operating Condition

	Calculated	Allowed	Ratio	Status
Longitudinal Hub (SH)	24114	29393	0.820	OK
Radial Flange (SR)	11572	19595	0.591	OK
Tangential Flange (ST)	7221	19595	0.369	OK

Version 10.50 Flange_1050.flg Jul 27,2021

Edit Menu

.Edit.

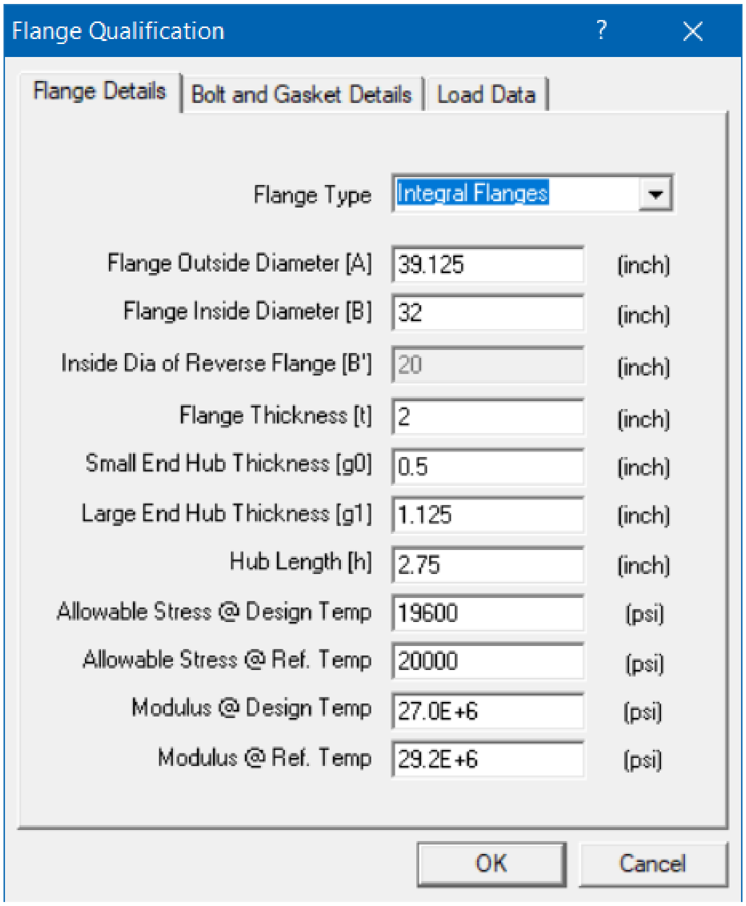
You can edit the Flange, Bolt, and Gasket Details, as well as Load Data by clicking the Edit command.

Flange Qualification Window

You can edit the Flange, Bolt, and Gasket Details, as well as Load Data by clicking the Edit command.

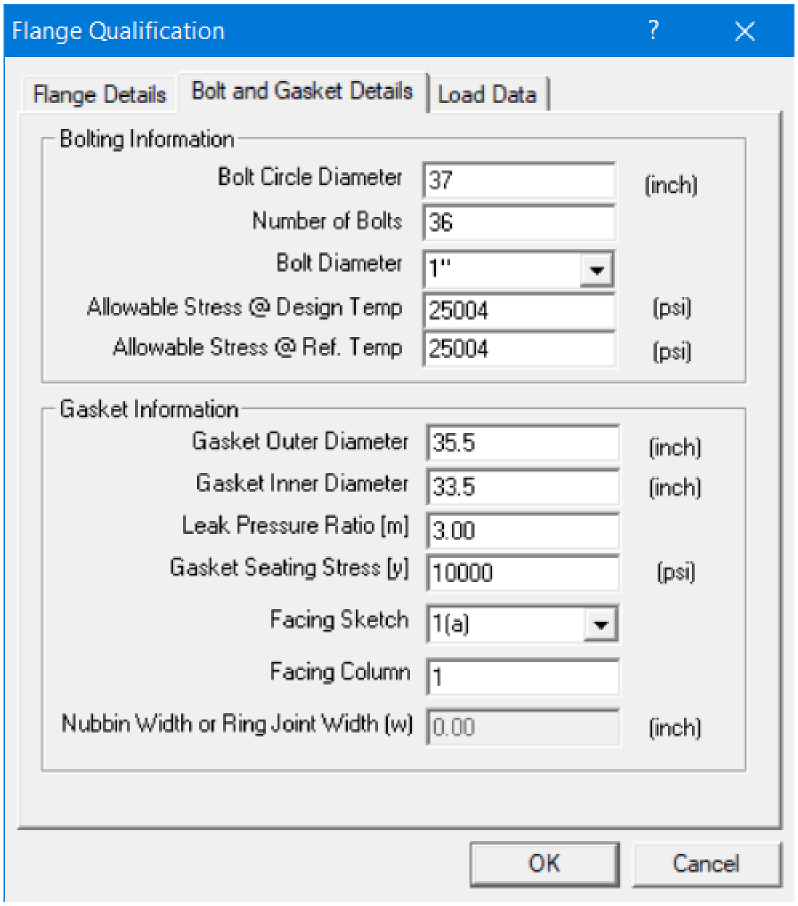
Flange Details Tab

Modify the flange details using this window.



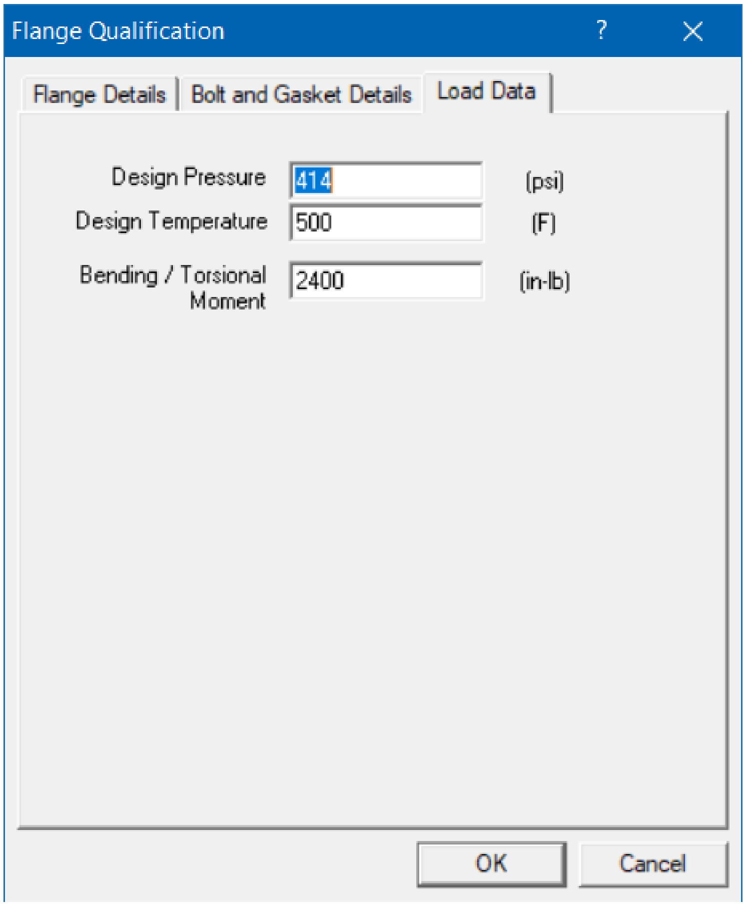
Bolt and Gasket Details Tab

Modify the bolt and gasket details using this window.

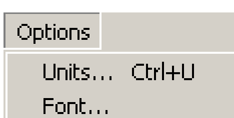


Load Data Tab

Modify the load data using this window.



Options Menu



.Units.

See Units in the Layout Window Options Menu section of the CAEPIPE User's Manual.

.Font.

See Font in the Layout Window Options Menu section of the CAEPIPE User's Manual.

Sample Problem

Problem 1:

(Example on page 19 in Chapter 40 "Bolted-Flange Joints and Connections" by William J. Koves on "Companion Guide to the ASME Boiler & Pressure Vessel Code" by K .R. Rao [2001], American Society of Mechanical Engineers, U.S.)

Flange Details:

Flange Type : **Integral Flanges**
Flange Outside Diameter [A] = 39.125 (inch)
Flange Inside Diameter [B] = 32 (inch)
Flange Thickness [t] = 2 (inch)
Small End Hub Thickness [g0] = 0.5 (inch)
Large End Hub Thickness [g1] = 1.125 (inch)
Hub Length [h] = 2.75 (inch)
All. Stress @ Design Temp [sf] = 19600 (psi)
All. Stress @ Ref. Temp [sfa] = 20000 (psi)
Modulus @ Design Temp [E] = 2.7E+7 (psi)
Modulus @ Ref. Temp [Ea] = 2.92E+7 (psi)

Bolting Information:

Bolt Circle Diameter = 37 (inch)
Number of Bolts = 36
Bolt Diameter = 1 (inch)
All. Stress @ Ref. Temp [sa] = 25000 (psi)
All. Stress @ Design Temp [sb] = 25000 (psi)

Gasket Information:

Gasket Outside Diameter = 35.5 (inch)
Gasket Inner Diameter = 33.5 (inch)
Leak Pressure Ratio [m] = 3.00
Gasket Seating Stress [y] = 10000 (psi)
Facing Sketch = 1
Facing Column = 1

Load Data:

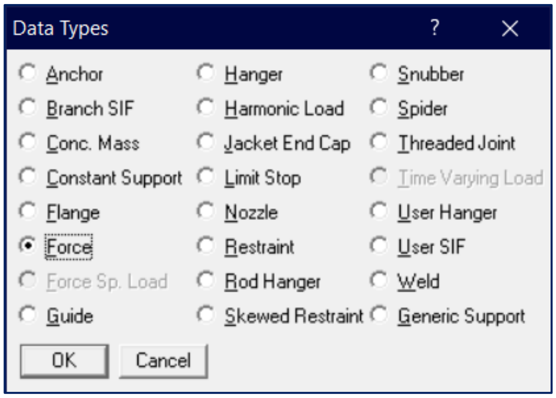
Design Pressure = 414 (psi)
Design Temperature = 500 (F)
Bending Moment = 200 (ft-lb)

Comparison of Results

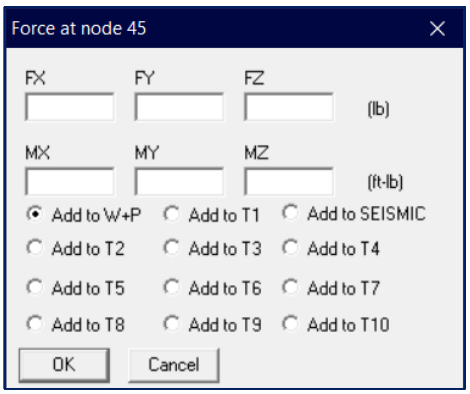
Flange Stresses	Text Book Results (psi)	CAEPIPE (psi)	CAESAR II (psi)
Operating condition			
Longitudinal Hub (SH)	24150	24152	24227
Radial Flange (SR)	11590	11590	11636
Tangential Flange (ST)	7230	7232	7205
0.5(SH + SR)	17870	17871	17932
0.5(SH + ST)	15690	15692	15716
Gasket Seating Condition			
Longitudinal Hub (SH)	15270	15269	15292
Radial Flange (SR)	7330	7327	7345
Tangential Flange (ST)	4570	4572	4547
0.5(SH + SR)	11300	11298	11318
0.5(SH + ST)	9900	9921	9920

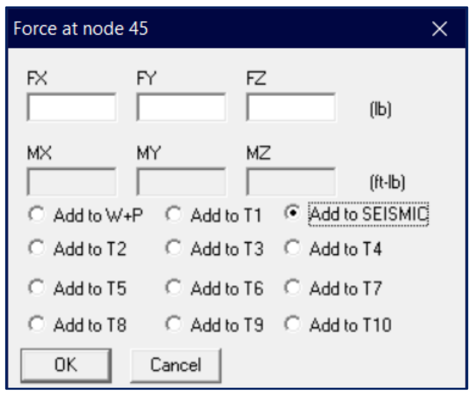
Legend for the different parameters and more examples are given in Section titled "Flange Qualification" in the Code Compliance Manual.

External forces and moments on the piping system in the global X, Y and Z directions may be input at any location. Type "fo" in the Data column or select "Force" from the Data Types dialog.



The Force dialog is shown.





If you select "Add to W+P", the specified forces and moments are applied to the sustained and operating load cases.

If you select "Add to SEISMIC", the specified forces are applied to the Static Seismic Load case. Moments cannot be defined for the Static Seismic Load case and hence moment input fields are disabled.

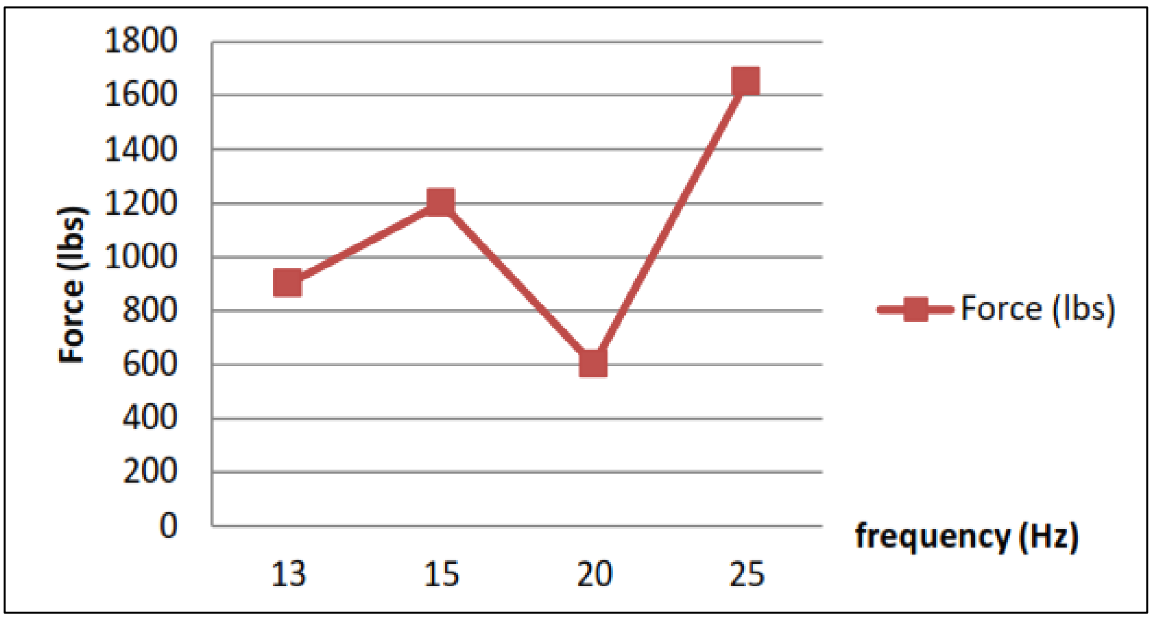
Force defined in Global X direction (FX) will be included only with x'g solution when x'g acceleration input is non-zero. Similarly, forces defined in Global Y (FY) and Global Z (FZ) directions will be included in y'g and z'g solutions respectively, when y'g and z'g accelerations are input as non-zero values.

See Section titled "Static Seismic Load" from CAEPIPE User's Manual for further details on how CAEPIPE performs Static Seismic Load analysis.

If you select any of the thermal cases ("Add to T1", "Add to T2", "Add to T3", all the way up to "Add to T10"), the specified forces and moments are applied to the selected thermal load case (T1/T2/T3/…/T10) and its operating load case counterpart (i.e.,T1 and W+P1+T1, or T2 and W+P2+T2, or T3 and W+P3+T3 or T4 and W+P4+T4 or T5 and W+P5+T5 or T6 and W+P6+T6 or T7 and W+P7+T7 or T8 and W+P8+T8 or T9 and W+P9+T9 or T10 and W+P10+T10).

Force spectrum analyses (different from harmonic analyses) are generally performed to determine the response of the piping system to short-duration impulsive loads such as fluid hammer, safety relief valve (SRV) and slug flow loads. For an actual short-duration impulsive dynamic load exerted on a piping system, a fluid transient analysis is first carried out in order to arrive at the "time-history loads" (i.e., force vs. time) acting in the three global directions (namely global X, Y and Z) at all affected points in the piping system. The time-history load sets so computed are then applied, one time-history load set at a time, on a single degree-of-freedomspring-mass system with a pre-set natural frequency, to determine the maximum dynamic response of this single degree-of-freedomsystem with that natural frequency. Such dynamic analysis for that time-history load is repeated on the same single degree-of-freedom system with different pre-set natural frequencies. The force spectrum for that time-history load would then be a table of maximum dynamic response computed for the single degree-of-freedom system with different natural frequencies. In other words, the force spectrum is a table of force spectral values vs frequencies that captures the maximum intensity and frequency content of that time-history load. Similarly, force spectrum tables are determined for all other time-history load sets. The above force spectrum tables (i.e., maximum dynamic force vs frequency) are then applied as inputs at the respective piping nodes of the CAEPIPE stress model to compute displacements, forces and stresses.

For any piping system, there are as many natural modes of vibrations as the number of dynamic degrees of freedom for that system. The force spectral value corresponding to a natural frequency of the piping system is arrived at by interpolating the force spectrum vs frequency table as determined above. For better understanding, as an example, refer to the graph shown next as well as the natural frequencies computed for a piping system at 10 Hz, 14 Hz, 21 Hz, 29 Hz and 33.8 Hz.

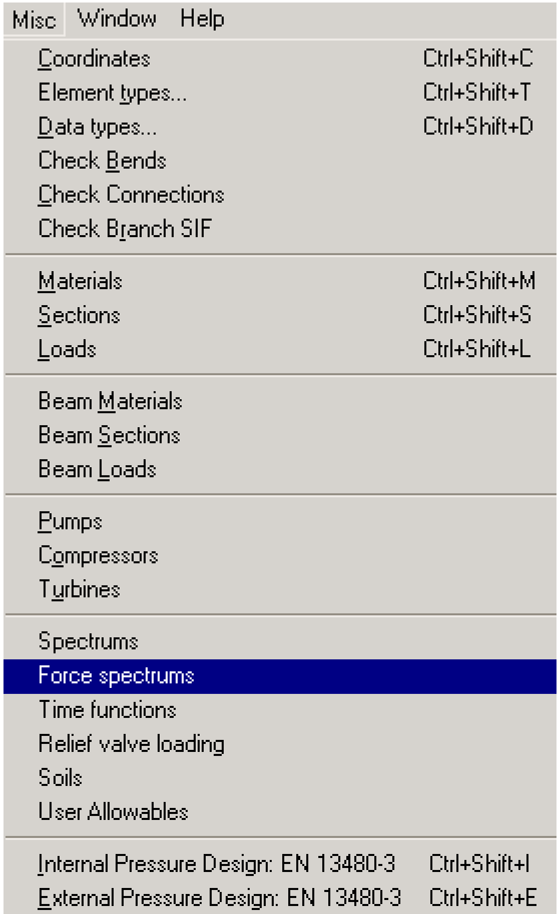


From the above graph, to arrive at a force value corresponding to a natural frequency of 14 Hz, CAEPIPE interpolates the force spectral values between 13 and 15 Hz. Similarly, to arrive at a force value corresponding to a natural frequency of 21 Hz, CAEPIPE interpolates the force spectral values between 20 Hz & 25 Hz. Since force spectral values above 25 Hz are not defined in the graph shown above, CAEPIPE arrives at a force value of 1650 lb. (i.e., the spectral value corresponding to the maximum frequency of 25 Hz in the above plot) even for natural frequencies of 29 and 33.8 Hz. Similarly, CAEPIPE arrives at a force value of 900 lb. for a natural frequency of 10 Hz (i.e., the spectral value corresponding to the minimum frequency of 13 Hz in the above plot).

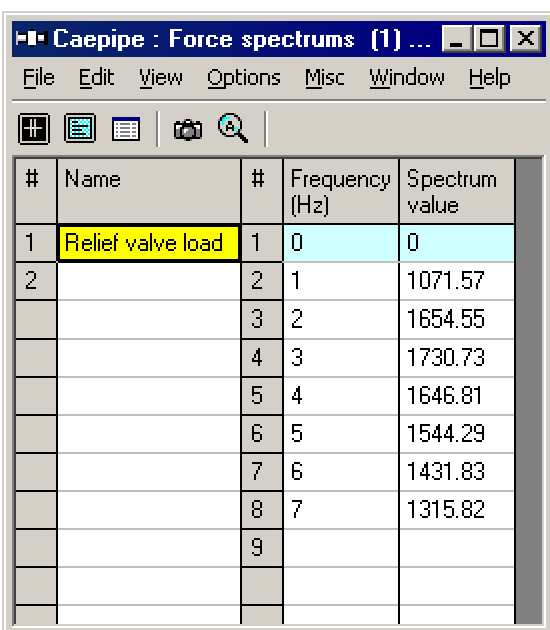
If only one set of force versus frequency is input (for example, 1000 lb. at 14 Hz) in the force spectrum table for your model, CAEPIPE applies the same force (1000 lb.) for all natural frequencies computed for that piping system. Note that the displacement produced at a node will remain unchanged even when the sole frequency in the force spectrum table is changed from 14 Hz to any other frequency.

Here, the results of the modal analysis are used with force spectrum loads to calculate the response (displacements, support loads and stresses) of the piping system. It is often used in place of a time-history analysis to determine the response of the piping system to sudden impulsive loads such as water hammer, relief valve and slug flow. The force spectrum is a table of spectral values versus frequencies that captures the intensity and frequency content of the time-history loads. It is a table of Dynamic Load Factors (DLF) versus natural frequencies. DLF is the ratio of the maximum dynamic displacement divided by the maximum static displacement. Note that Force spectrum is a non-dimensional function (since it is a ratio) defining a curve representing force versus frequency. The actual force spectrum load at a node is defined using this force spectrum in addition to the direction (FX, FY, FZ, MX, MY, MZ), units (lb, N, kg, ft-lb, in-lb, Nm, kg-m) and a scale factor.

The Force spectrums are input from the Layout or List menu: Misc > Force spectrums.



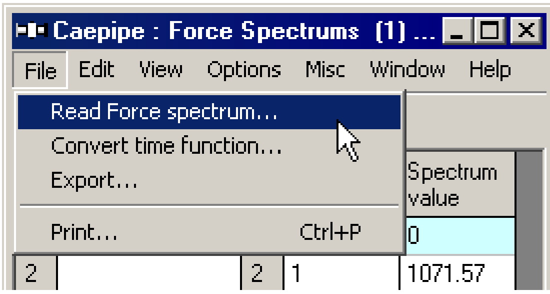
The Force spectrum list appears.



Enter a name for the force spectrum and spectrum values versus frequencies table. In addition to inputting the force spectrum directly, it can also be read from a text file or converted from a previously defined time function.

To read a force spectrum from a text file:

use the List menu: File > Read force spectrum.



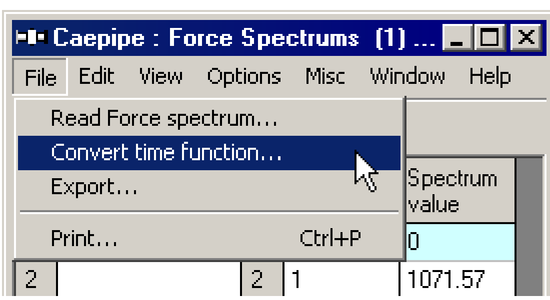
The text file should be in the following format:

Name (up to 31 characters)
Frequency (Hz) Spectrum value
Frequency (Hz) Spectrum value
Frequency (Hz) Spectrum value
. .
. .
. .

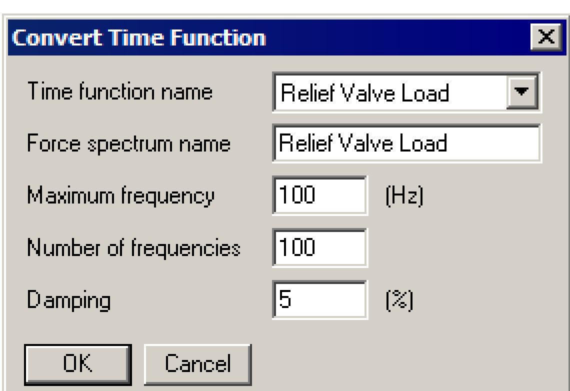
The frequencies can be in any order. They will be sorted in ascending order after reading from the file.

To convert a previously defined time function to force spectrum:

use the List menu: File > Convert time function.



The Convert Time Function dialog appears.



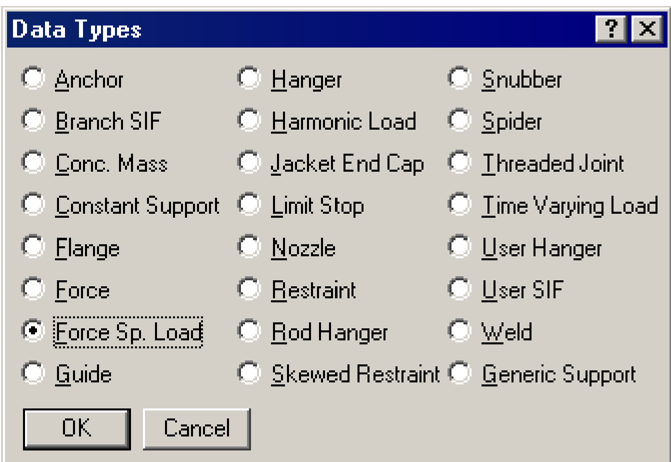
Select the time function to convert from the Time function name drop down combo box. Then input the Force spectrum name (defaults to the Time function name), Maximum frequency, Number of frequencies and the Damping. When you press Enter or click on OK, the time function will be converted to a force spectrum and entered into the force spectrum list.

The time function is converted to a force spectrum by solving the dynamic equation of motion for a damped single spring mass system with the time function as a forcing function at each frequency using Duhamel's integral and dividing the absolute maximum dynamic displacement by the static displacement.

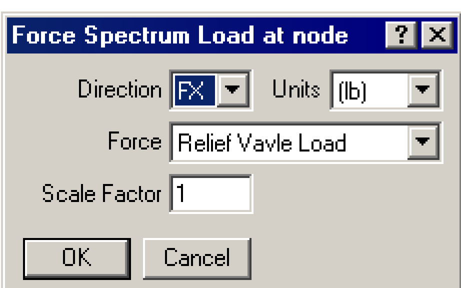
Force Spectrum Load

The force spectrum loads are applied at nodes (in Data column in Layout window). At least one force spectrum must be defined before a force spectrum load at a node can be input.

To apply the force spectrum load at a node click on the Data heading or press Ctrl+Shift+D for Data Types dialog.



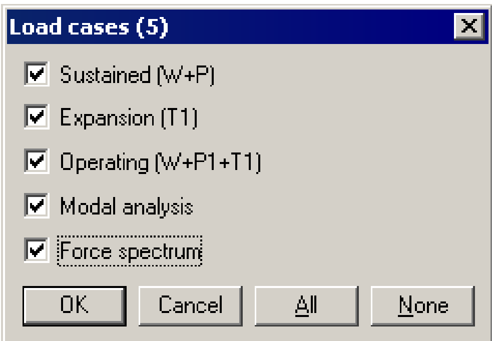
Select "Force Sp. Load" as the data type and click on OK. This opens the Force Spectrum Load dialog.



Select the direction, units and force spectrum using the drop down combo boxes and input appropriate scale factor. The scale factor can be a scalar value, which, when multiplied by the non-dimensional force spectrum, will give the actual magnitudes of the force versus frequency in the global direction and unit selected in the above dialog. Then click on OK to enter the force spectrum load at that node.

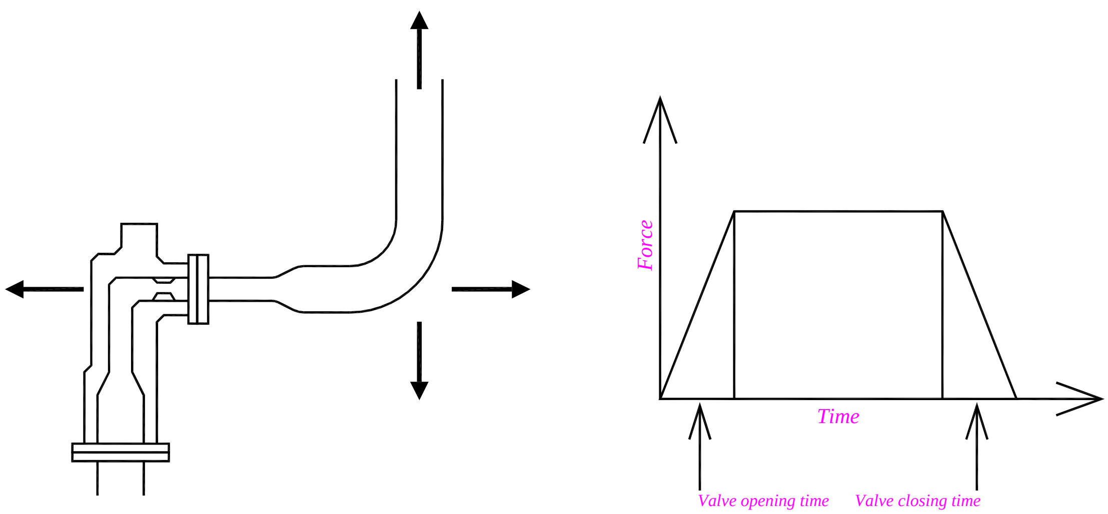
7	6" std pipe								
8	30	From							
9	60		6'0"			A53	6	1	Force sp load
10	70	Valve	2'0"			A53	6	1	

Input force spectrum loads at other nodes similarly. Then select the force spectrum load case for analysis using the Layout menu: Loads > Load cases.



Note that Modal analysis and Sustained (W+P) load cases are automatically selected when you select Force spectrum load case. The force spectrum load case is analyzed as an Occasional load.

A Relief Valve Analysis may be performed by first obtaining the data about valve opening and subsequent behavior, as a force versus time history profile. Enter the profile as a time function. See under Time History Load for how to.



(a) (b)

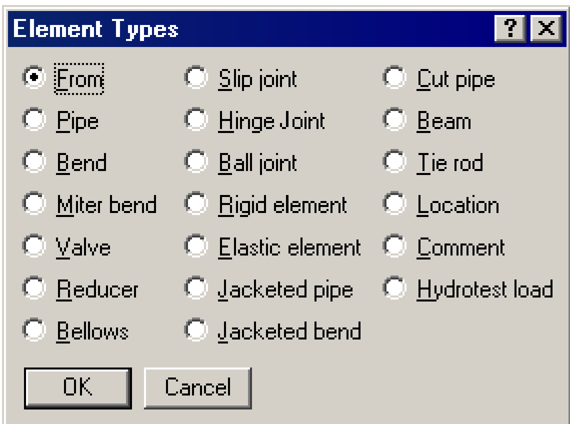
Then under Force Spectrum, use "Convert time function" to convert the force-time history profile into a Force-Spectrum. Input loads and analyze. See the topic Relief Valve Load Analysis for more details.

From (Node)

“From” is a special Element Type used to start a new (branch) line. For a new node number, any values you type in under the DX, DY and DZ columns are taken to be coordinates (and not offsets). If you use “From” for an existing node number, then you do not have to specify values for the DX, DY and DZ fields.

If you specify a new node number other than the global origin [that has coordinates (0, 0, 0)], then you must specify the coordinates for the new node using “From” and DX, DY and DZ fields. If you do not, then the new node number will have the same location as the node that is the global origin.

“From” is input by typing “f” in the Type column or selecting “From” from the Element Types dialog.

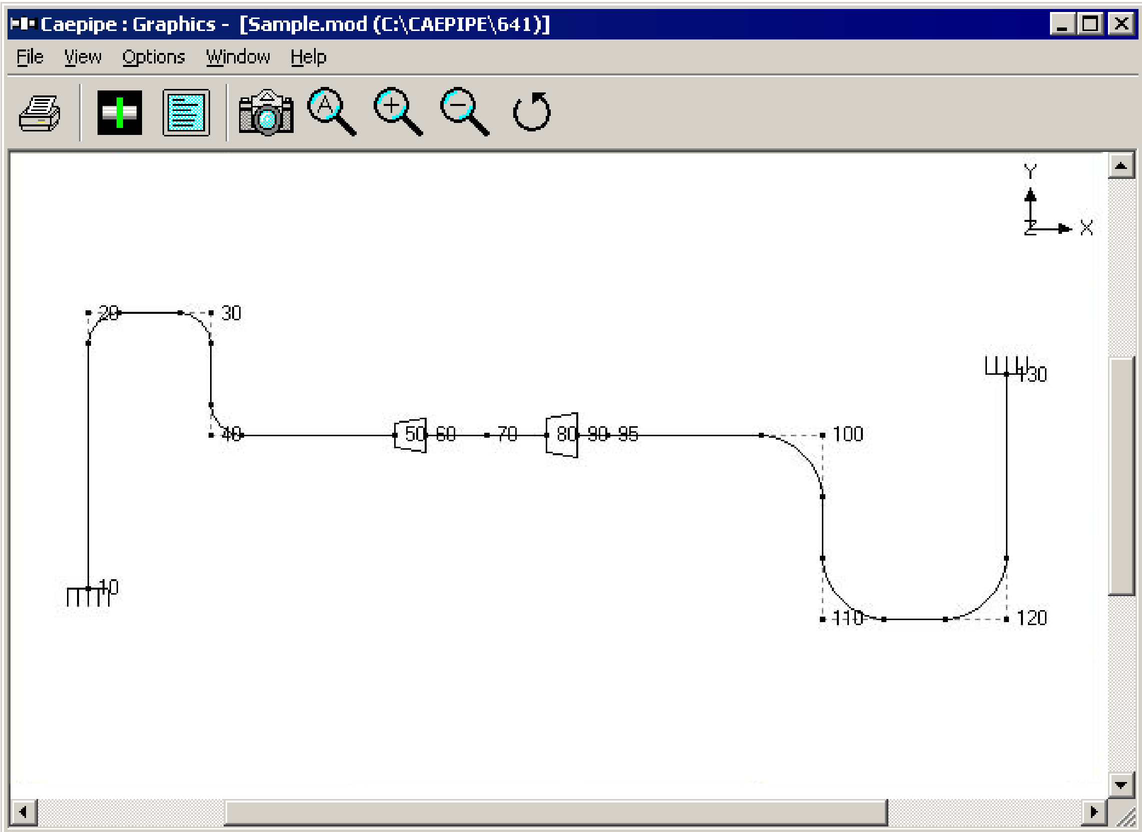


The first node of a model is always a fixed “From” node since you have to start the model from a point. The DX, DY, DZ fields for this node may be left blank to mean the global origin (0,0,0) or global coordinates may be specified in the DX, DY and DZ fields to have a nonzero reference point for the model.

Values specified for DX, DY and DZ for any other node other than a “From” node are interpreted as offsets (not as coordinates) unless the node is suffixed with an asterisk “*”. See the topic “Node” for more information.

It is helpful to create a model starting from a reference node located using its global coordinates and continue in an orderly manner from there, especially when you plan to merge (see File menu) two models later.

Example:

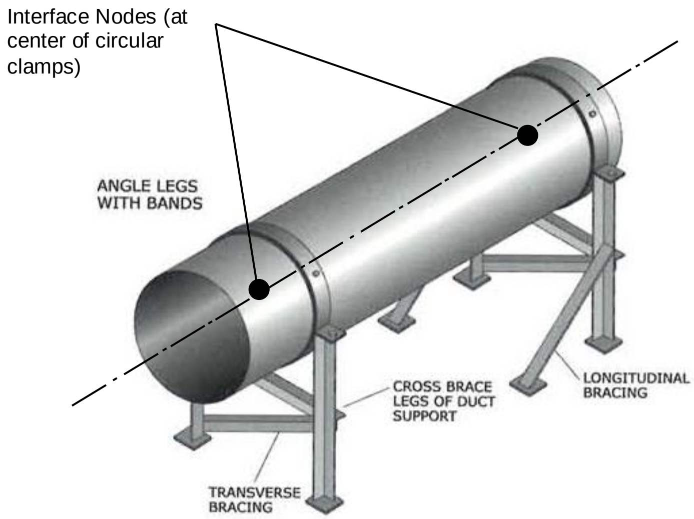


Assume you had two separate models you wanted to combine later.

First model: In this first model, you would start modeling from node 10 up to node 95. Since Node 100 in above is a Tangent-Intersection-Point (TIP), it is not a good idea to create a model upto TIP. That means, the bend at Node 100 would not know which way it should turn!!

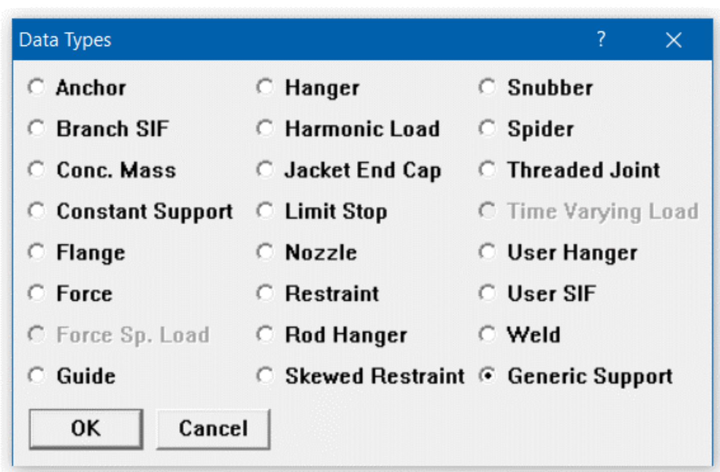
Second model: You would model from node 95 up to node 130, which is a better way to model rather than modeling from node 130 to node 95, which would complicate the "Merge" process.

A generic data type called "Generic support" is available to define a complex support fitting for which the support stiffnesses are obtained from an external source, using a reduced 6x6 stiffness matrix. An example would be a support assembly as shown below.

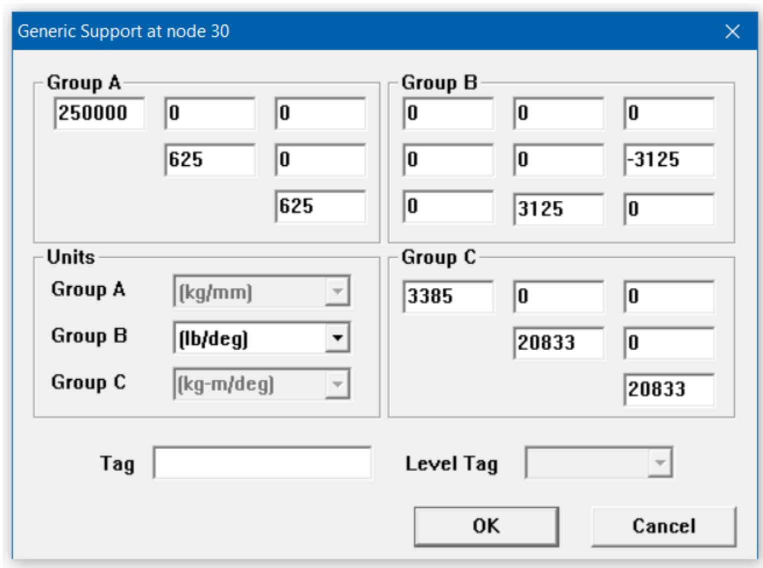


A reduced 6x6 stiffness matrix representative of this support at a single interface node (as shown in figure above) will first need to be arrived at using any general purpose FEA software package (such as ANSYS, NASTRAN, etc.) Then, those stiffness values representing the support assembly can then be input into CAEPIPE model for further analysis.

"Generic Support" is input by typing "ge" in the Data column or selecting "Generic Support" from the Data Types dialog.



Since the reduced 6x6 stiffness matrix representing the generic support is always a symmetric matrix, only the upper triangular stiffness values need to be input, with the diagonal terms having positive non-zero values. Diagonal terms are never zero or left blank. Off-diagonal terms can be zero, positive or negative values.



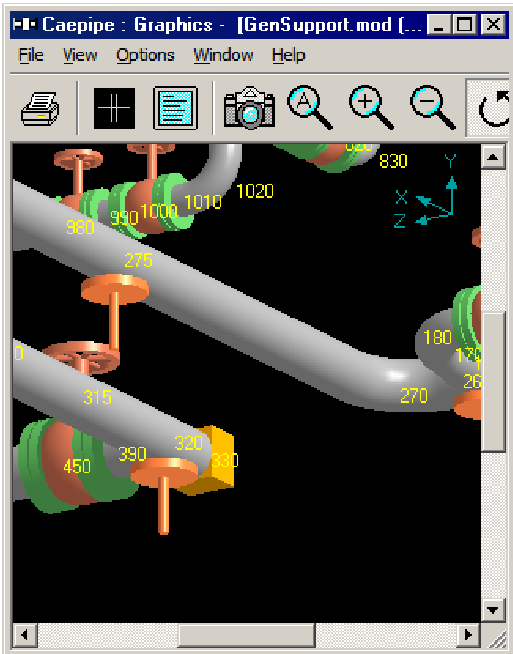
Units for Groups A and C are controlled centrally through the Units command (Ctrl+U in Layout window). Units for Group B need to be set in this dialog.

The Stiffnesses in the Groups A, B and C are to be computed in Global Coordinate System and entered.

The fields in Group A are translational stiffnesses. The fields in Group C are rotational stiffnesses.

The fields in Group B are the coupling stiffnesses. For example, a lateral force at the end of a Cantilever beam produces not only the displacement in the direction of that force, but also a rotation. Similarly, a moment at the end of a Cantilever beam produces not only the rotation in the direction of that moment, but also a lateral displacement.

Each stiffness field in the dialog is editable. The default "rigid" stiffness is shown for all the diagonal terms. If you have a stiffness value for any of these including the off-diagonals, enter them here (ensure units integrity). Graphically, the support is shown as a solid block (at node 330 in the next figure).



Displacements, support loads and support load summaries are shown for this support type.

Caepipe : Support load summary for generic support at node 30 - [complex1.res (\\...

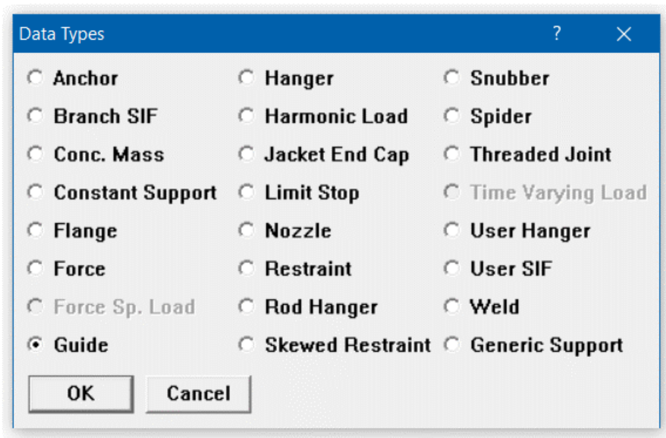
File Results View Options Window Help

Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)	Displacements (global) X (inch)	Y (inch)	Z (inch
Sustained	-2663	-75	6	-16	309	734	-0.011	-0.048	0.021
Operating1	-16802	346	-234	-91	-998	-2247	-0.067	0.141	-0.060
Operating2	6509	3	-30	-18	592	605	0.026	-0.042	0.042
Operating3	25788	56	349	-3	1033	1067	0.103	-0.075	0.058
Sustained+Settlement	-2692	-317	35	19	723	2171	-0.011	-0.137	0.048
Operating1+Settlement	-16831	103	-206	-56	-584	-809	-0.067	0.052	-0.032
Operating2+Settlement	6480	-239	-2	17	1006	2043	0.026	-0.131	0.069

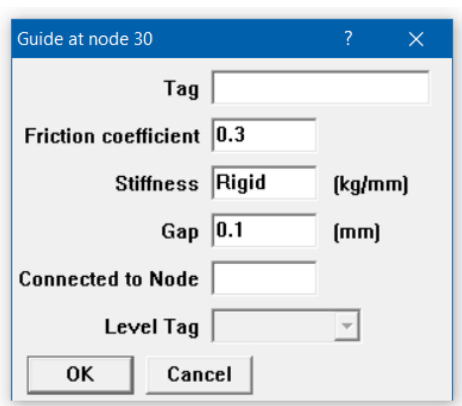
Guide

A guide is used in the field to control or direct pipe movement. Likewise, its CAEPIPE equivalent allows axial movement while restraining the pipe against lateral translations (but not rotations). A guide restricts the translational movement normal to its axis, i.e., displacements are restrained in the local y and z directions of the element to which the guide is attached. The Local Coordinate System (LCS) of the guide can be viewed through View > List > Guide and right-click mouse and select the option “Show LCS”.

A guide is input by typing “g” in the Data column or selecting “Guide” from the Data Types dialog.



The Guide dialog is shown.



Tag

Tag can be 12 characters long. Tags are useful in identifying a support while modeling, reviewing of reports and in field erection. Tag Name entered in this field is shown in all reports.

Friction Coefficient

When a friction coefficient is entered, a nonlinear analysis is performed. In each iteration, the friction force is calculated which is friction coefficient times the normal force (the vector sum of local y and local z reaction forces). This friction force is applied in the local x direction opposing the axial motion of pipe. The solution converges when the displacement changes by less than 1% between successive iterations.

Stiffness

The default stiffness is rigid which is input by typing "r" or "Rigid" in the Stiffness field. A non-rigid stiffness may be entered by typing the value of the stiffness in the Stiffness field.

Gap

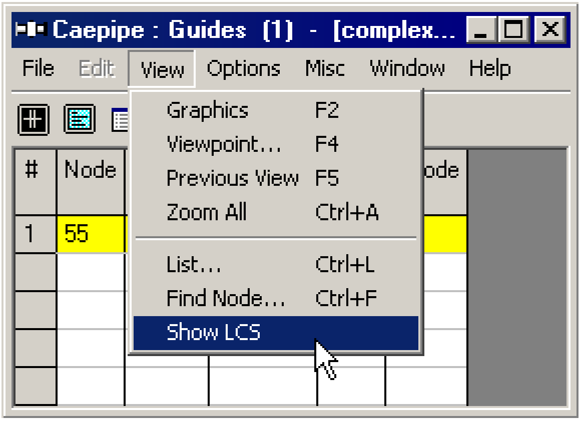
A clearance between the pipe and the guide, if present, may be entered as a Gap. The gap is assumed to be symmetric about the guide axis. This gap must be closed before any restraint forces can develop. If there is no gap, leave this field blank or enter it as 0.0.

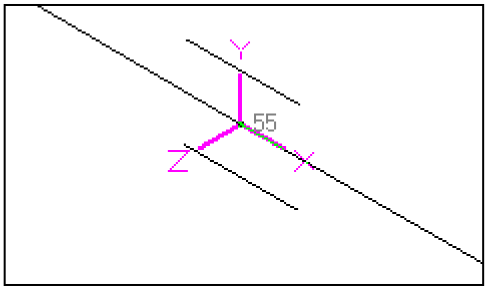
Connected to Node

By default the guide is assumed to be connected to a fixed *ground* point which is not a part of the piping system. A guide can be connected to another node in the piping system by entering the node number in the "Connected to Node" field.

Local Coordinate System (LCS)

A guide's local x-axis is based on the preceding element. If a preceding element is unavailable, the following element is used to determine the guide's local x-axis. The local coordinate system (LCS) may be viewed graphically from the Guide List window using the menu: View > Show LCS.



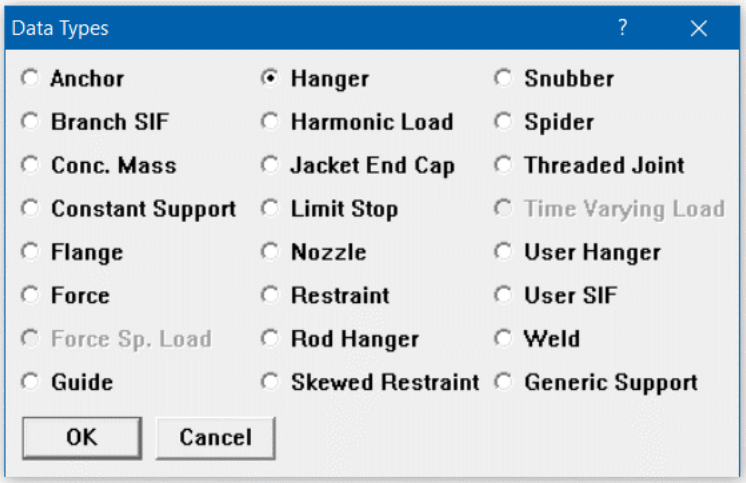


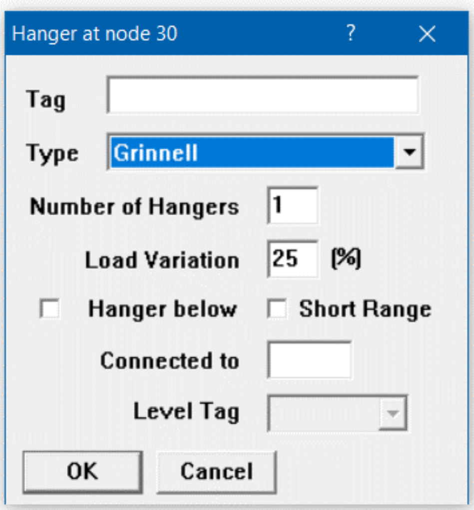
Guide forces in global coordinate system are "Print(ed) to file" (in addition to forces in local coordinate system), accessible from the Print command dialog.

Hanger

Variable spring hangers support the dead-weight of piping while allowing vertical and lateral thermal movement from the installed to the operating condition. CAEPIPE assumes a spring hanger to always act in the vertical direction.

Also called a "To be designed" hanger, it is input by typing "h (Enter)" in the Data column (or typing "Han") or selecting "Hanger" from the Data Types dialog. The Hanger dialog is shown.





Tag

Tag can be 12 characters long. Tags are useful in identifying a support while modeling, reviewing of reports and in field erection. Tag Name entered in this field is shown in all reports.

Type

The type (i.e., manufacturer) of the hanger can be selected from the drop-down combo box "Type." The following hanger types are currently available:

Hanger Types		
ABB-PBS	Fee & Mason	Nordon
Basic Engineers	Flexider (30-60-120)	NPS Industries
Bergen-Paterson	Flexider (50-100-200)	Piping Services
Bergen-Paterson (L)	Fronek	Piping Tech & Products
BHEL Hyderabad	Grinnell	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)	Gradior
Elcen	NHK (80-160)	

*CAEPIPE includes catalog from Carpenter Paterson Ltd. based in England.

Number of Hangers

The number of hangers is the number of separate hangers connected in parallel at this node. The stiffness and load capacity of each hanger are multiplied by the number of hangers to find the effective stiffness and load capacity of all the hanger supports at this node.

Load Variation

The load variation (in percent) is the maximum variation between the cold and hot loads. Typical value is 25%.

Hanger Below

Changesthe graphical depiction only.

This should be used to specify whether the Hanger is placed below the pipe. Graphical symbol changes accordingly. This selection is valid only for Hanger and User Hanger and not for Constant Support Hanger nor Rod Hanger.

Short Range

Short range hangers are used if the available space is not enough for installing mid-range hangers. They are considered, however, as a specialty item and generally not used. If a short range hanger is to be designed, check the Short Range check box.

Connected to Node

By default the hanger is connected to a fixed *ground* point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the "Connected to node" field. This node *must be directly above or below* the hanger node.

Hanger Design Procedure

1. Calculate Hot Load

Hot load for a variable spring hanger is the actual weight of the pipe (including the weights of content, lining and insulation) being carried by that hanger. To calculate hot load, a preliminary sustained load analysis is performed in which all hanger locations are restrained vertically. If any anchor is to be released (so that the hanger rather than the nearby equipment takes the sustained load), it is released. The reactions at the hanger locations from this preliminary sustained load analysis are the hanger hot loads.

2. Calculate Hanger Travel

Vertical restraints (applied in step 1) at hanger locations are removed. Released anchors (if any) are restored. A preliminary operating load case analysis is performed. If multiple thermal load cases are specified, only the first thermal load is used for this operating load case. The hot loads (calculated in step 1) are applied as upward forces at the hanger locations. Vertical displacements at the hanger locations obtained from this operating load case analysis are the hanger travels. If limit stops are present, the hot loads are recalculated with the status of the limit stops at the end of the preliminary operating load case. Then the hanger travels are recalculated using the recalculated hot loads.

3. Select Hanger

The hanger is selected from the manufacturer's catalog based on the hot load and hanger travel. The cold load is calculated as: cold load = hot load + spring rate x hanger travel. The hanger load variation is calculated as: Load Variation = 100 x Spring rate x travel / Hot load. The calculated load variation is checked against specified maximum load variation. The hanger for which both the hot and cold loads are within the hanger's allowable working range and the load variation is less than the allowed load variation is selected. The hanger is selected such that the hot load is closest to the average of the minimum and maximum loads.

4. Install Hangers

If "Include hanger stiffness" is chosen in the Analysis options: The hanger spring rates are added to the overall stiffness matrix. The hanger cold loads are used in the sustained and operating load cases. If "Do not include hanger stiffness" is chosen in the Analysis options: The hanger spring rates are not added to the overall stiffness matrix. The hanger hot loads are used in the sustained and operating load cases.

For Lisega hangers, size column will report as Hanger Number, Type and Range instead of Hanger Number, Range and Type. For example, hanger having a spring rate of 2.1 N/mm, vertical travel of 30mm with load 440N will be reported as 21D.193 instead of 21D319.

Sustained Displacement during Hanger Selection

An Example Application

Sometimes, rotating equipment vendors (e.g. turbine vendors) require that there be no weight load imposed on the turbine connections after welding/bolting the pipe to the nozzle but prior to plant start-up. This can be accomplished only if the pipe is hung by variable spring hangers like a swing at the nozzle <u>and</u> if the pipe end of such "hung" pipe is almost perfectly aligned with the turbine nozzle prior to welding or bolting that pipe end to the turbine nozzle. In other words, the spring hangers that carry the weight of the "hung" pipe near the nozzle are <u>to be sized and placed</u> such that the resulting displacements (i.e., 3 translations and 3 rotations) at the pipe end are nearly zero, so that the pipe end need not be forcibly deformed in order to weld/bolt it to the turbine nozzle. This, in turn, makes sure that the weight load of the pipe is not imposed on the turbine nozzle prior to plant start-up.

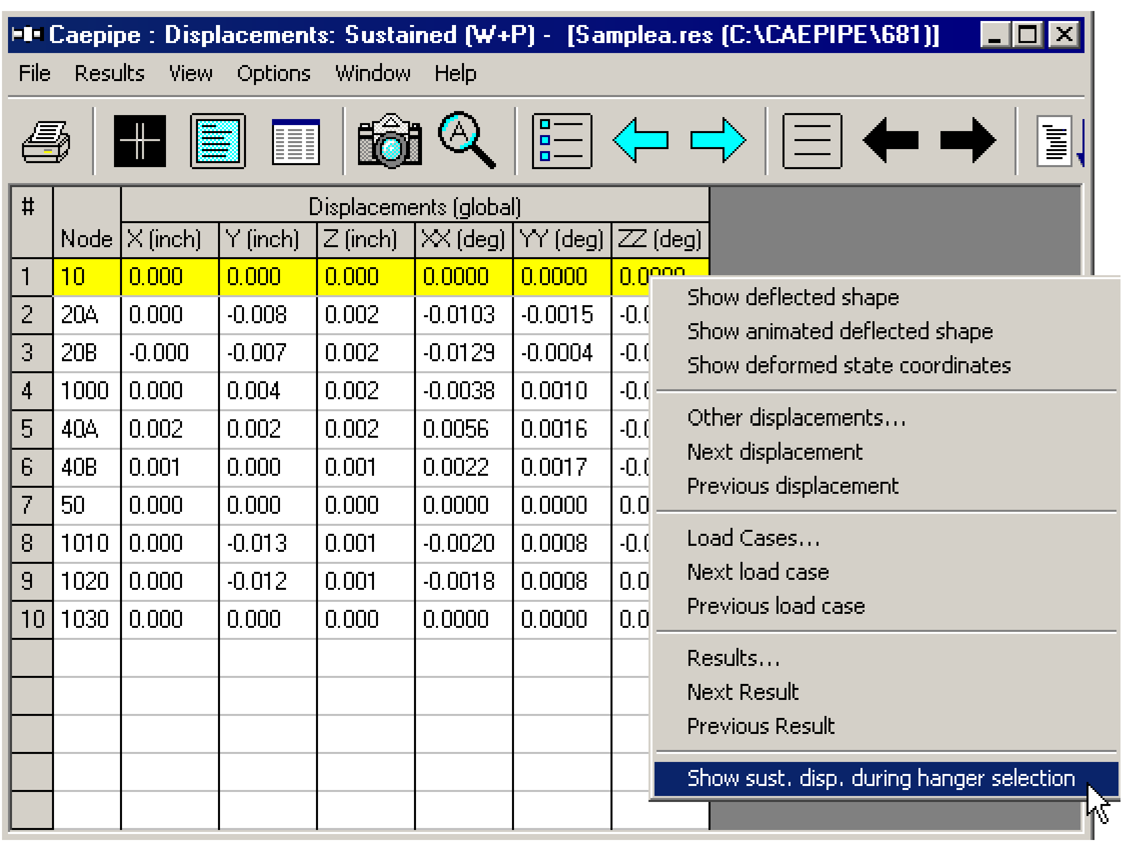
CAEPIPE can be used to perform the above study by carrying out the following steps.

a) During Step 1 of the Hanger Design procedure given above, release all 6 degrees of freedom of the anchor corresponding to that turbine nozzle, so that piping weight load will not be transferred to the nozzle during the initial Sustained load analysis (in which all hangers are pinned, thereby restraining the pipe vertically at the hanger locations).

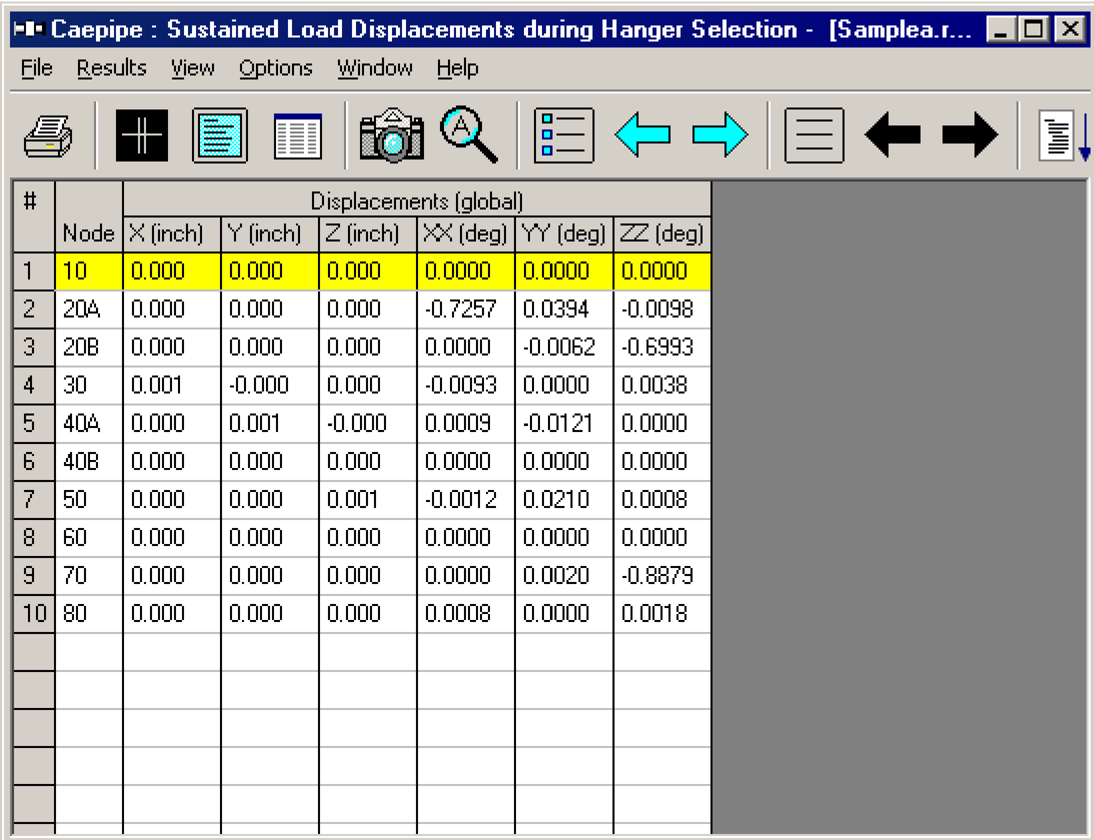
b) Review the preliminary Sustained Load Displacements computed during Hanger Selection via Results > Displacements > Mouse right click > Show sust. disp. during hanger selection. If the Sustained Load translations and rotations at the concerned pipe end are NOT nearly zero, try out different hanger locations till the preliminary Sustained Load displacements at that pipe end are nearly zero.

c) When the Preliminary Sustained Load displacements at the concerned pipe end are nearly zero, the Support Loads at that Nozzle reported by CAEPIPE for Sustained Load case would be close to zero.

In results, the sustained displacements during hanger selection when hangers are pinned can be shown via Results >Displacements>Mouse right click>Show sust. disp. during hanger selection.



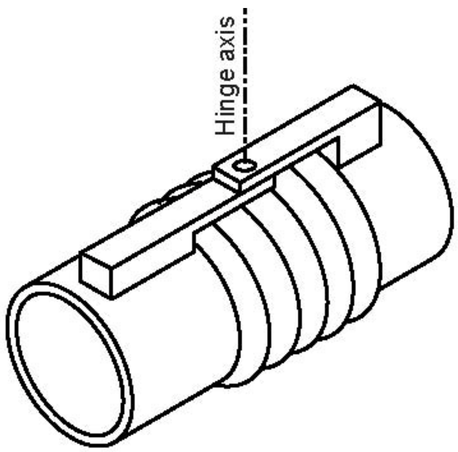
And here is the Results Window for sustained displacements during hanger selection after selecting this feature.



#	Node	Displacements (global)					
		X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	20A	0.000	0.000	0.000	-0.7257	0.0394	-0.0098
3	20B	0.000	0.000	0.000	0.0000	-0.0062	-0.6993
4	30	0.001	-0.000	0.000	-0.0093	0.0000	0.0038
5	40A	0.000	0.001	-0.000	0.0009	-0.0121	0.0000
6	40B	0.000	0.000	0.000	0.0000	0.0000	0.0000
7	50	0.000	0.000	0.001	-0.0012	0.0210	0.0008
8	60	0.000	0.000	0.000	0.0000	0.0000	0.0000
9	70	0.000	0.000	0.000	0.0000	0.0020	-0.8879
10	80	0.000	0.000	0.000	0.0008	0.0000	0.0018

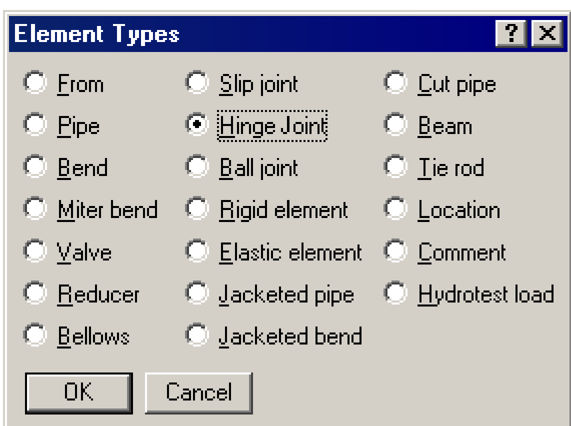
Hinge Joint

Hinge joint is an expansion joint designed to permit angular rotation in a single plane by use of a pair of pins that pass through plates attached to the expansion joint ends. Hinge joints are used in sets of two or three to absorb pipe movement in one or more directions in a single-plane piping system. A pair of hinge joints, separated by a section of piping, will act together to absorb lateral deflection. Hinge joints are designed to take the full pressure thrust.

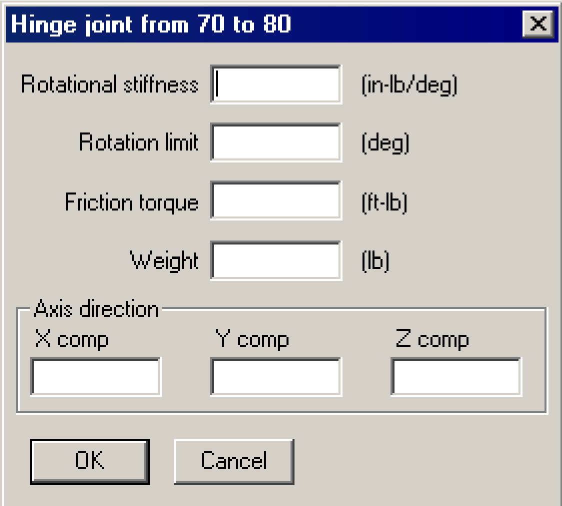


The two sides of the hinge joint shown are joined by hinge pins which are along the hinge axis shown in the figure. A hinge is modeled by two nodes, one on each side of the hinge joint. The two nodes of the hinge joint are coincident. So, it is a zero length element, i.e., the "From" and "To" nodes are coincident. Hence, the DX, DY and DZ fields in the Layout window should be left blank.

A hinge joint is input by typing "h" in the Type column or selecting "Hinge joint" from the Element Types dialog.



The Hinge joint dialog is shown.



Rotational Stiffness

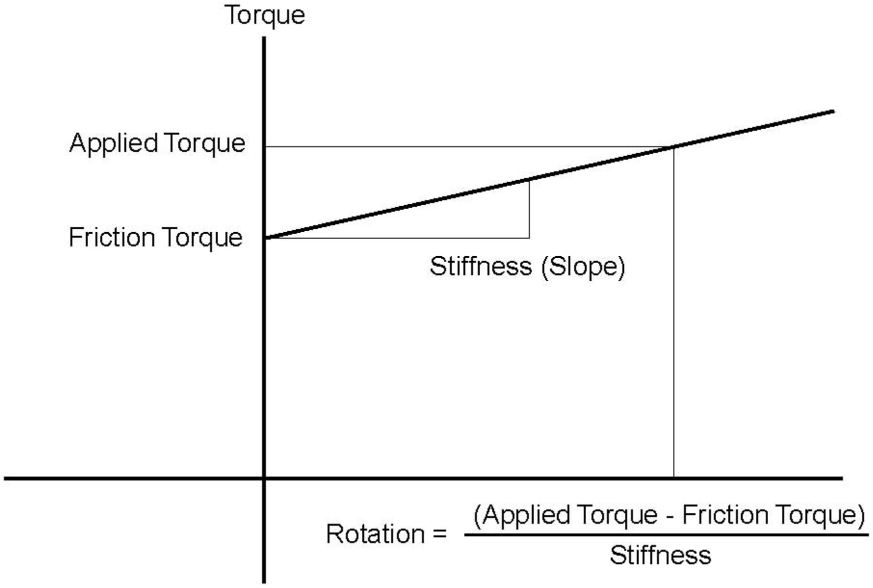
Also called Angular stiffness. Input the stiffness around the rotational (hinge) axis. The stiffness value may be available from the manufacturer of the hinge joint or from test results. Otherwise engineering judgment may be used. The stiffness values may be left blank. In that case a very small value (1 in-lb/rad.) is used internally to avoid dividing by zero.

Rotation Limit

Rotation limit is an upper limit on the rotation of hinge joint in the plus or minus directions. Rotation limit of 0.0 (zero) means it is unable to rotate (i.e., it is rigid). Rotation limit of "None" or Blank means that there is no limit on its rotation.

Friction Torque

The hinge joint will rotate only if the external torque exceeds the friction torque. Beyond that, the rotation is proportional to the rotational stiffness of the hinge joint. The friction torque value may be available from the manufacturer of the hinge joint or from test results. Otherwise engineering judgment may be used. If you do not want friction in the hinge joint, the friction torque value may be left blank.



When the applied torque is less than friction torque, there is no rotation. When the applied torque exceeds friction torque, the rotation is calculated as shown above. When rotation limit is reached, there is no further rotation irrespective of the applied torque. See discussion for K_{be} under Ball joint for additional information.

Weight

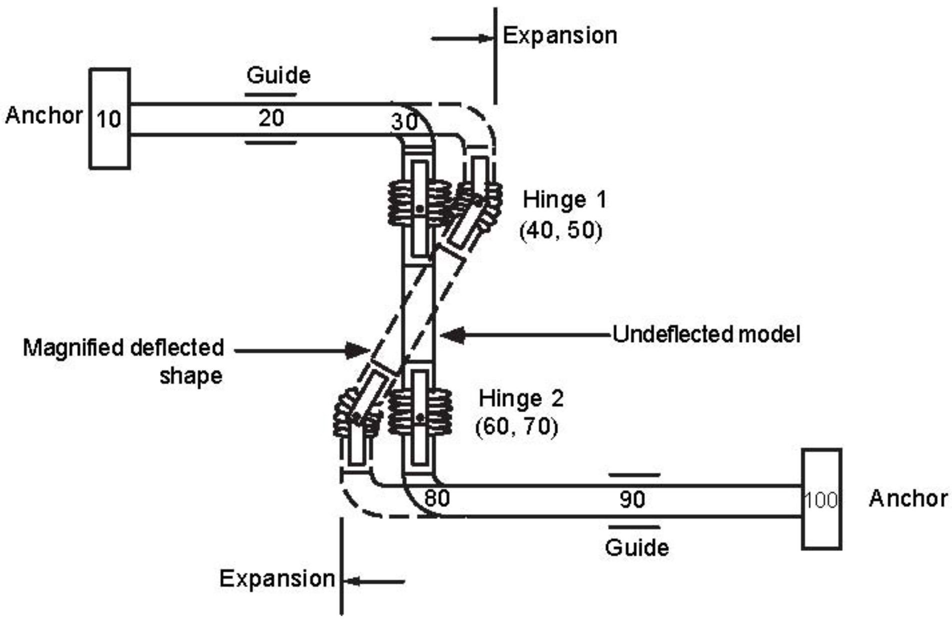
This is the total weight of the Hinge joint assembly.Weight is to be input in lbf or kgf and NOT in mass units. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

Axis direction

The hinge axis is specified by the "Axis direction." See "Direction," for more information on specifying a direction using X comp, Y comp and Z comp.

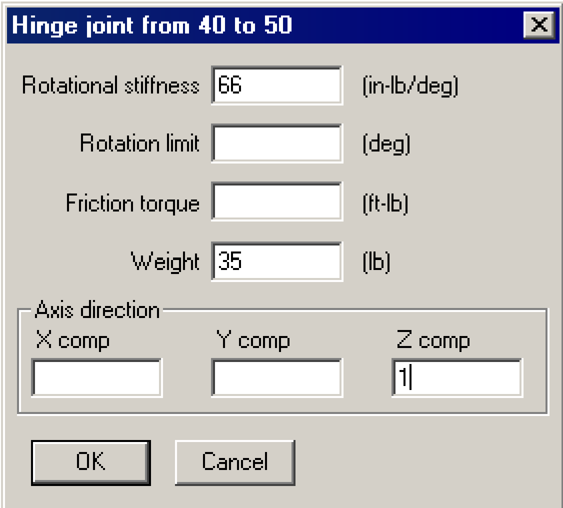
Example

Assume that we had the model shown below (with an exaggerated deflection) containing 6" piping with a pair of Hinge joints. Each Hinge joint has the following data: rotational stiffness of 66 in.-lb./degree, and weight of 35 lb.



The following steps describe the modeling procedure (with auto node incrementing, you do not have to type in the node numbers below):

- The first node 10 is already defined as an anchor. Press Enter to move to the next row.
- Type 20 for Node, 3'6" for DX, enter material, section and load names, Guide for Data. Press Enter to move to the next row.
- Input bend at node 30: Type 30 for Node, press Tab to move to the Type field. Type "b" and Tab to next column to enter a bend, 2' for DX, press Enter to move to the next row.
- Type 40 for Node, enter –1'6" for DY (as the offset from node 30 to node 40), press Enter to complete the bend and move to next row.
- Type 50 for Node. Type "h" in the Type column. This shows the hinge dialog.

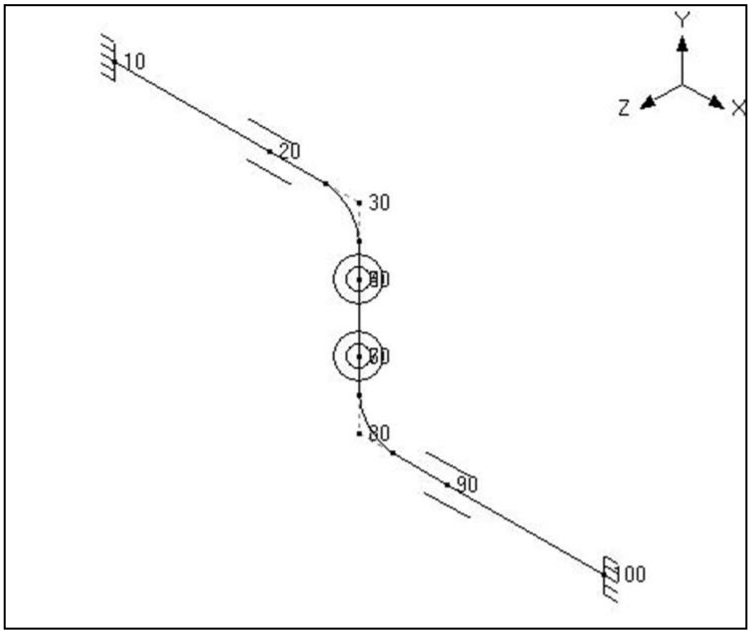


Enter 66 (in.-lb/deg.) for Rotational stiffness, 35 (lbf) for Weight, 1.0 for Z comp (axis direction), press Enter or click on OK to close the dialog. This completes the hinge input. Since there cannot be any offsets (DX, DY, DZ) for the hinge node from the previous node, the cursor automatically moves to the next row.

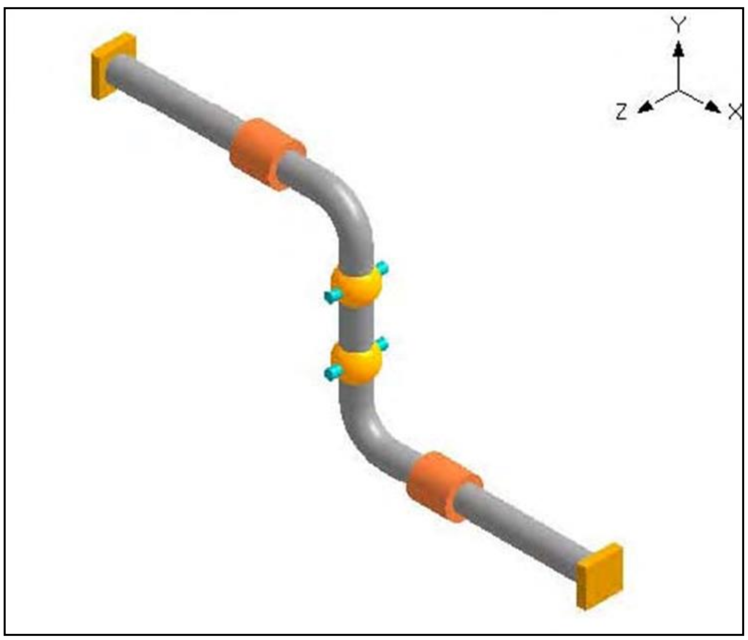
- Type 60 for Node, enter –1'6" for DY (as the offset from node 50 to node 60), press Enter to move to next row.
- Type 70 for Node. Type "h" in the Type column. This shows the hinge dialog. Enter the hinge data as before and click on OK to move to the next row.
- Input bend at node 80: type 80 for Node, press Tab to move to Type field, type "b" for Type and Tab to next column to enter a bend. Type –1'6" for DY (offset from node 70 to bend node 80), press Enter.
- Complete the model through nodes 90 and 100 similar to steps 1 and 2 above.

The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Loac	Data
1	Title = Hinge Joint								
2	10	From							Anchor
3	20		3'6"			1	6	1	Guide
4	30	Bend	2'0"			1	6	1	
5	40			-1'6"		1	6	1	
6	50	Hinge							
7	60			-1'6"		1	6	1	
8	70	Hinge							
9	80	Bend		-1'6"		1	6	1	
10	90		2'0"			1	6	1	Guide
11	100		3'6"			1	6	1	Anchor
12									



The rendered graphics is shown below:

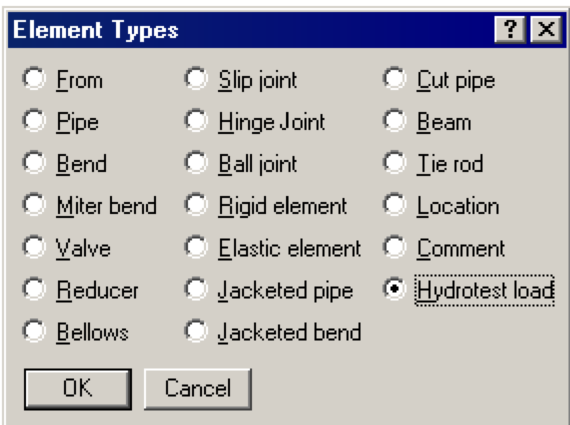


See under Expansion Joints for more examples.

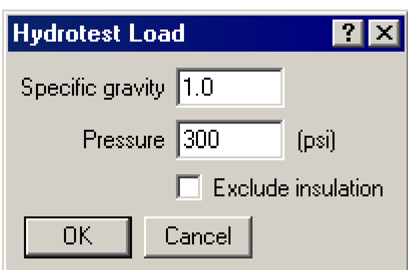
Use this load to analyze loading from a hydrostatic test which is performed by filling the piping system with a pressurized fluid (typically water) to check for leaks, etc., before putting the system into service.

During hydrotest, all hangers are assumed pinned (i.e., they act as rigid vertical supports). The hydrotest load is defined by the specific gravity of the test fluid (1.0 for water), test pressure and whether to include or exclude the insulation weight (because many times the hydrotest is performed before applying the insulation).

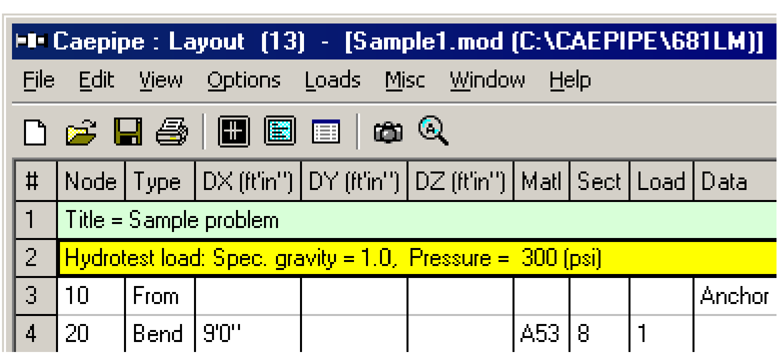
The hydrotest load is input by pressing "h" on an empty row in the Layout window (similar to pressing "c" for a comment) or on an empty row, selecting "Hydrotest load" from the Element Types dialog (Ctrl+Shft+T).



The "Hydrotest Load" dialog appears.

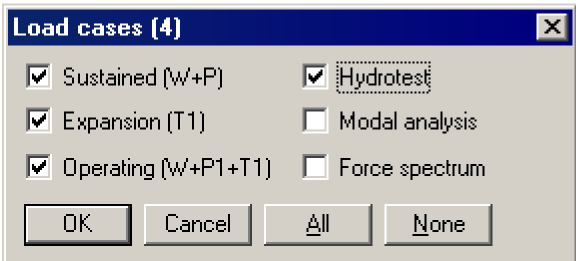


After the hydrotest load is input by pressing Enter or clicking on OK, the hydrotest load appears in the Layout window.

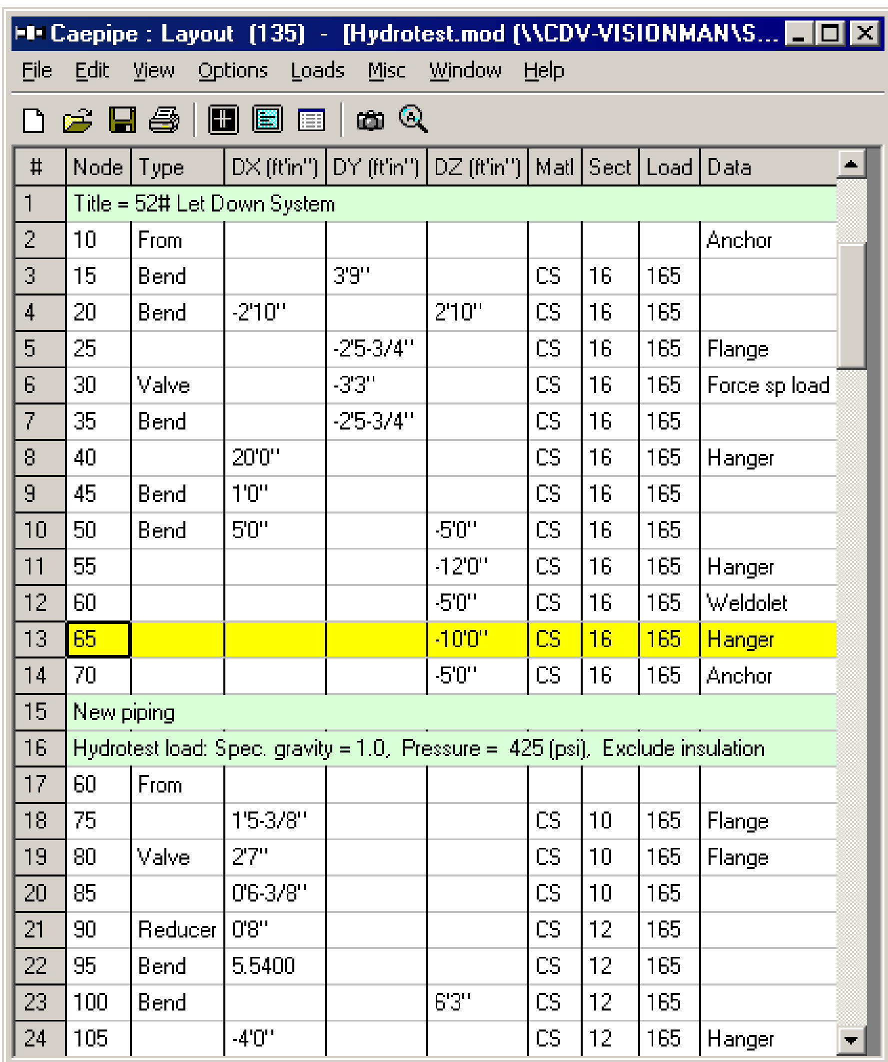


If you need to modify an existing hydrotest load, double click on the row that defines the hydrotest load to bring up the Hydrotest Load dialog. The hydrotest load is applied to the rows that follow until changed by another hydrotest load. The hydrotest load can be constant over the whole model or can be changed in parts of the model.

To analyze the hydrotest load case, the Hydrotest load case must be selected using the command Loads > Load cases from the Layout window.



The hydrotest load case is analyzed as a sustained load (with no temperature effect considered) and the resulting stresses are computed using the Sustained Stress (SL) equation corresponding to the piping code selected for analysis.

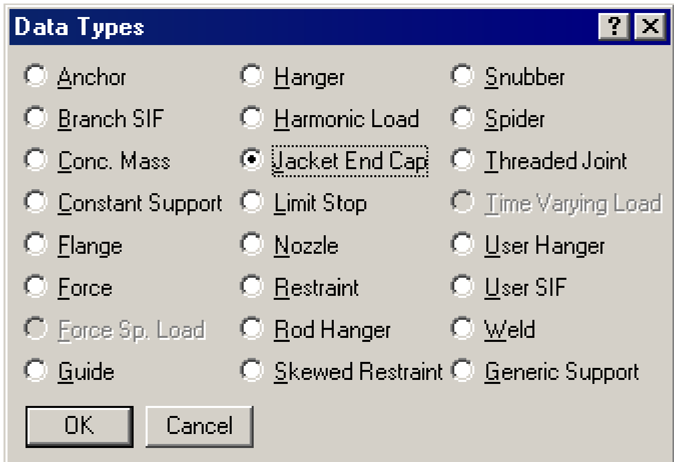


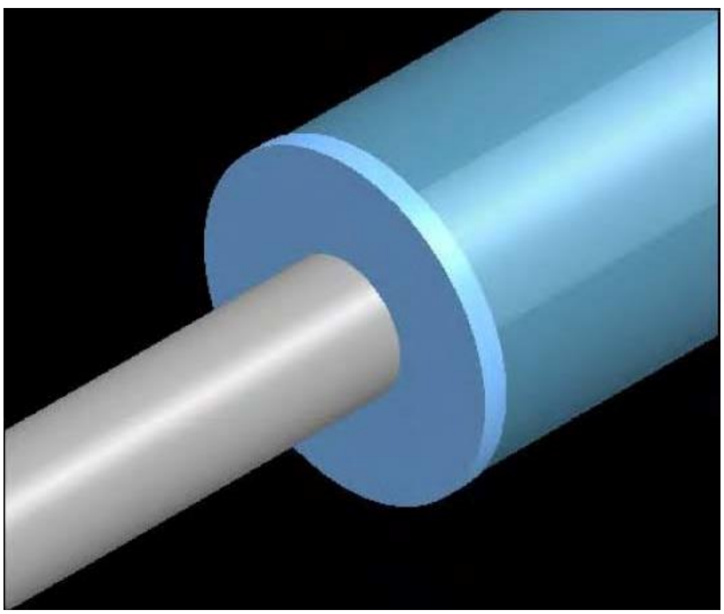
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = 52# Let Down System								
2	10	From							Anchor
3	15	Bend		3'9"		CS	16	165	
4	20	Bend	-2'10"		2'10"	CS	16	165	
5	25			-2'5-3/4"		CS	16	165	Flange
6	30	Valve		-3'3"		CS	16	165	Force sp load
7	35	Bend		-2'5-3/4"		CS	16	165	
8	40		20'0"			CS	16	165	Hanger
9	45	Bend	1'0"			CS	16	165	
10	50	Bend	5'0"		-5'0"	CS	16	165	
11	55				-12'0"	CS	16	165	Hanger
12	60				-5'0"	CS	16	165	Weldolet
13	65				-10'0"	CS	16	165	Hanger
14	70				-5'0"	CS	16	165	Anchor
15	New piping								
16	Hydrotest load: Spec. gravity = 1.0, Pressure = 425 (psi), Exclude insulation								
17	60	From							
18	75		1'5-3/8"			CS	10	165	Flange
19	80	Valve	2'7"			CS	10	165	Flange
20	85		0'6-3/8"			CS	10	165	
21	90	Reducer	0'8"			CS	12	165	
22	95	Bend	5.5400			CS	12	165	
23	100	Bend			6'3"	CS	12	165	
24	105		-4'0"			CS	12	165	Hanger

Jacket End Cap

Use a jacket end cap to rigidly connect all six degrees of freedom of the coincident nodes of a jacketed pipe (i.e., the node on the core pipe and the corresponding node on the jacket pipe ("J" node) are tied together so that both nodes have the same displacements and rotations).

A jacket end cap is input at a jacketed pipe node by typing "j" in the Data column or selecting "Jacket End Cap" from the Data Types dialog.





You will need this to secure the jacket pipe to the core pipe rigidly. Sometimes (depending on the combination of restraints), you may get a "Stiffness Matrix not positive definite" error, which may be corrected by inserting a jacket end cap.

Jacketed Piping

Jacketed piping is used when the primary state of the pipe contents (fuel, chemicals such as resins, etc.) needs to be maintained at a specific temperature during transport. An outer (jacket) pipe surrounds the inner (core) pipe that contains the operating fluid or the chemical. The jacket provides external heating or cooling as required along the length of the core pipe.

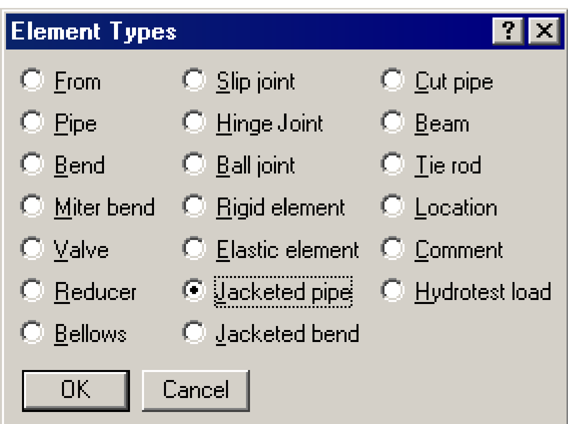
The terminology used here is as follows:

- *Jacketed* piping refers to the entire assembly, i.e., a core pipe with a jacket on the outside.
- *Jacket* pipe refers only to the outside pipe.
- *Core* pipe refers only to the inside pipe that contains the operating fluid.

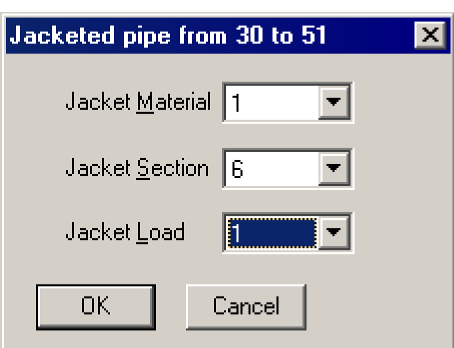
In CAEPIPE, jacketed piping need only be modeled once, not twice (as in some other pipe stress software programs). CAEPIPE models the outer jacket pipe *along with* the inner core pipe once on the Layout window. Each row defines a jacketed piping element. The jacket and the core pipes may have different materials, sections and (P, T) loads.

Jacketed Pipe

A Jacketed pipe is input by typing "JP" under Type or selecting "Jacketed pipe" from the Element Types dialog. The material, section and load specified in the Jacketed Pipe dialog apply to the jacket pipe while the ones mentioned on the layout row (next to offsets) apply to the core pipe.



The Jacketed Pipe dialog is shown.



The jacket's material, section, and load names are input here (1, 6 and 1 as shown). CAEPIPE retains the properties of a jacket pipe until changed so there is no need to retype the names of the jacket properties every time you input a jacketed pipe.

The ends of the jacket and core pipes need to be rigidly connected using the "Jacket End cap" data type (see previous topic). Also, "Spiders" need to be input at locations found in the physical assembly. You may have to break up the piping into smaller elements to insert spiders at appropriate locations. See example given later in this section.

In case you are analyzing for wind, it may be more accurate to specify a different load for the core pipe alone that does not specify the Wind load since the core pipe is not exposed to wind. Same applies to the core pipe insulation if the core pipe does not have insulation.

Internal nodes

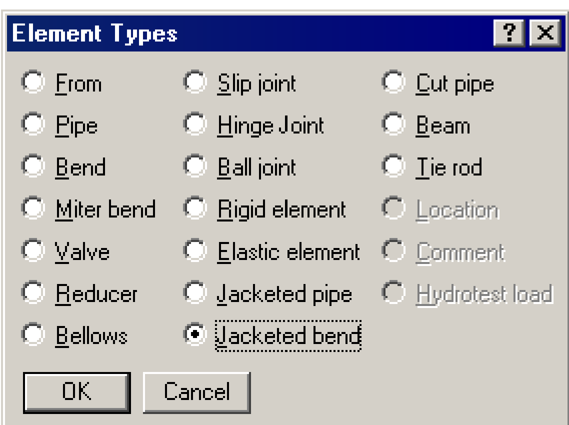
CAEPIPE generates a "J" node for jacket pipes. For example, if you had a jacketed pipe from node 10 to 20, CAEPIPE generates 10J and a 20J as (internal) jacket nodes (that may be referenced on the layout screen).

These internally generated nodes may be used to specify data items such as a spider, jacket end cap, support (hanger, restraint), forces on the jacket. See example later in this topic.

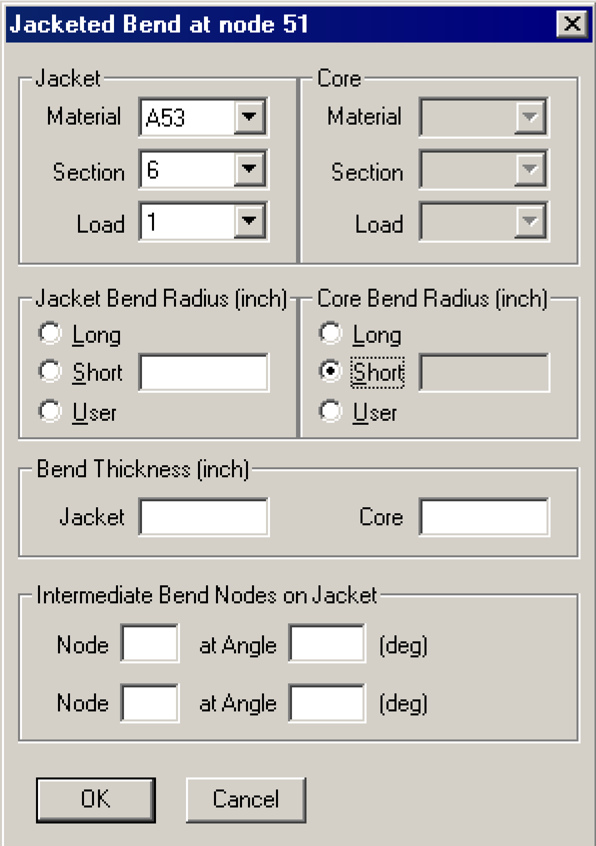
Jacketed Bend

A Jacketed bend consists of a core bend (with a straight portion) surrounded by a jacket bend (with a straight portion of jacket pipe).

A Jacketed bend is input by typing "JB" in the Type column or by selecting "Jacketed bend" from the Element types dialog.



The Jacketed Bend dialog is shown.



Jacket (properties)

The jacket's material, section, and load names are input here. The properties of a jacketed pipe are retained until changed. So, there is no need to retype the names of the jacket properties every time you input a jacketed pipe.

Core (properties)

Presently these properties are disabled. You need to enter them on the layout row under Material, Section and Load.

Bend radius

Separate bend radii may be specified for the core and the jacket pipes.

Note that CAEPIPE does not check for interference between the core and the jacket arising out of differently specified bend radii.

The bend radius for the core pipe is normally the same as that of the jacket pipe, since the two bends are generally concentric. Use the Render feature in the Graphics window to check visually for interference, however.

Bend thickness

Separate bend thicknesses may be specified for the core and the jacket bends, if they are not default jacket and core section thicknesses.

Intermediate nodes

You can define additional nodes on the outside jacket of a jacketed bend for locating supports. You may also use internal nodes generated by CAEPIPE. See Internal nodes below.

Internal nodes

CAEPIPE generates "C" and "D" nodes for the Jacketed bend on the jacket at the near and far ends of the bend. The core pipe (bend) has its own "A" and "B" nodes. Example: When you define a Jacketed bend from node 20 to node 30, 30A, 30B (nodes on core bend), 30C and 30D (nodes on jacket) are generated. Nodes (30A, 30C) and (30B and 30D) are coincident only if the core and the jacket pipes have the same bend radii. See figure.

These internal nodes may be used to specify data items such as a spider, jacket end cap, supports, forces, etc.

Split a Bend/Pipe

A jacketed pipe/bend may be split by using the Split option in the Edit menu in the Layout window.

Contents Weight

The weight of the contents between the jacket and the core pipes is calculated in the following manner:

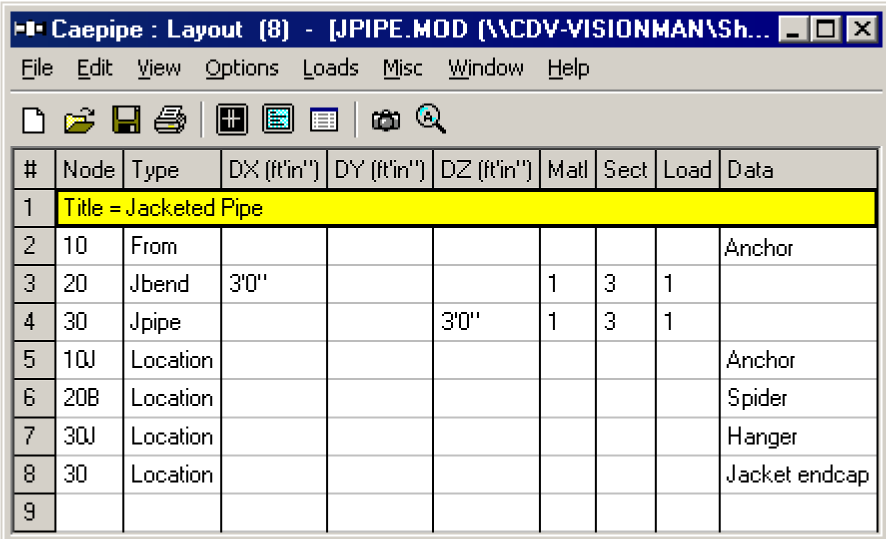
(a) Twice the insulation thickness on the core pipe is added to the outer diameter of the core pipe. (b) The external area of the core pipe is calculated by using the above diameter (a). (c) The internal area of the jacket pipe is calculated. (d) The external area of the core pipe (b) is subtracted from the internal area of the jacket pipe (c) and this result is further used to compute the weight of contents between the jacket and core pipes.

Jacketed Reducer

See modeling procedure in topic on the Reducer.

Example: Jacketed Pipe/Bend

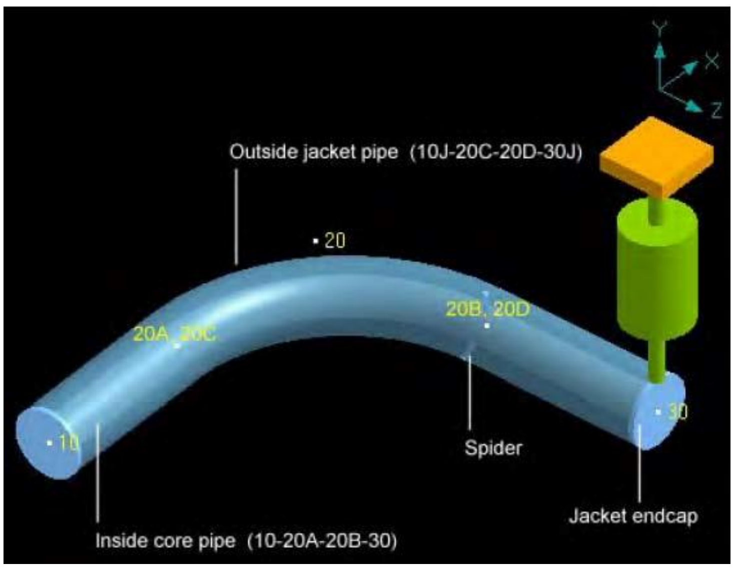
The figure below shows a Jacketed pipe with a Jacketed bend (at node 20-TIP). Observe the spider at the far end of the bend (node 20B).



Caepipe : Layout (8) - [JPIPE.MOD (\\CDV-VISIONMAN\Sh...

File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Jacketed Pipe								
2	10	From							Anchor
3	20	Jbend	3'0"			1	3	1	
4	30	Jpipe			3'0"	1	3	1	
5	10J	Location							Anchor
6	20B	Location							Spider
7	30J	Location							Hanger
8	30	Location							Jacket endcap
9									



The nodes 10J, 20C, 20D, 30J, 20A and 20B are internally generated nodes. You may use them for specifying data items such as spiders, supports (hanger, restraint), forces, etc.

The pairs of nodes (10, 10J)and (30, 30J) are coincident. The nodes 20A and 20B are coincident with the nodes 20C and 20D respectively only if the core and the jacket pipes have the same bend radii.

Note that the core and jacket nodes are not connected even though they are coincident. The core and jacket pipes have to be supported and connected using supportsand jacket connections (namely, spiders and jacket end caps). An anchor each at nodes 10 and 10J is specified. The hanger is at node 30J since it is attached to the jacket.

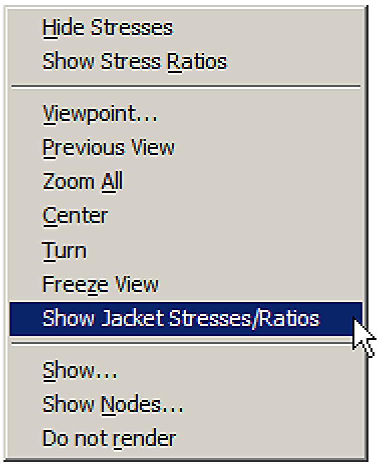
A jacket connection of the type spider at node 20B acts as an internal guide between the core pipe and the jacket pipe, that is, it prevents any radial movement but allows sliding, rotating and bending movement between core and jacket pipes.

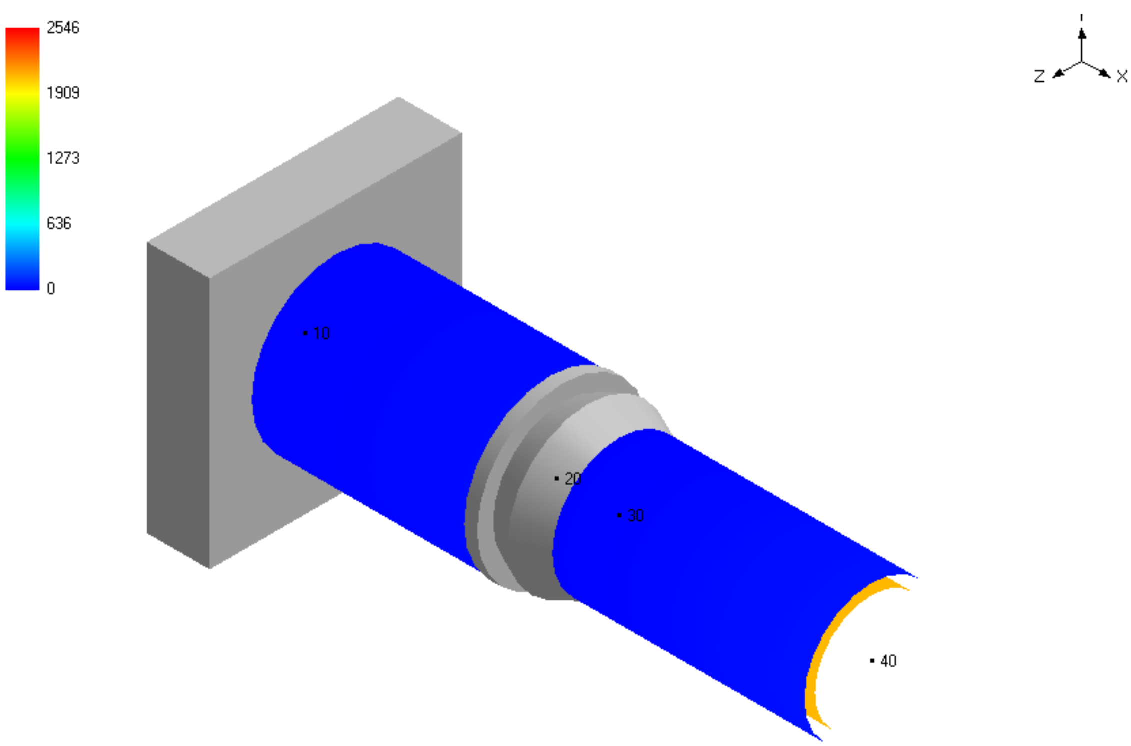
In case forces transmitted from core pipe to jacket through spiders are required, then spiders specified on core pipe can be replaced by guides with "connected to" nodes specified as the corresponding nodes on the jacket pipe.

The end cap at node 30 connects the core and jacket pipes rigidly.

Jacketed Piping Stresses/Ratios

CAEPIPE provides an option for you to display the color-coded stress/ratio contour for jacketed piping in the graphics window context menu. The default stress contour is for thecore piping. Upon selecting the command for Jacket stresses as shown below, stress contour plot for the jacket piping is displayed:

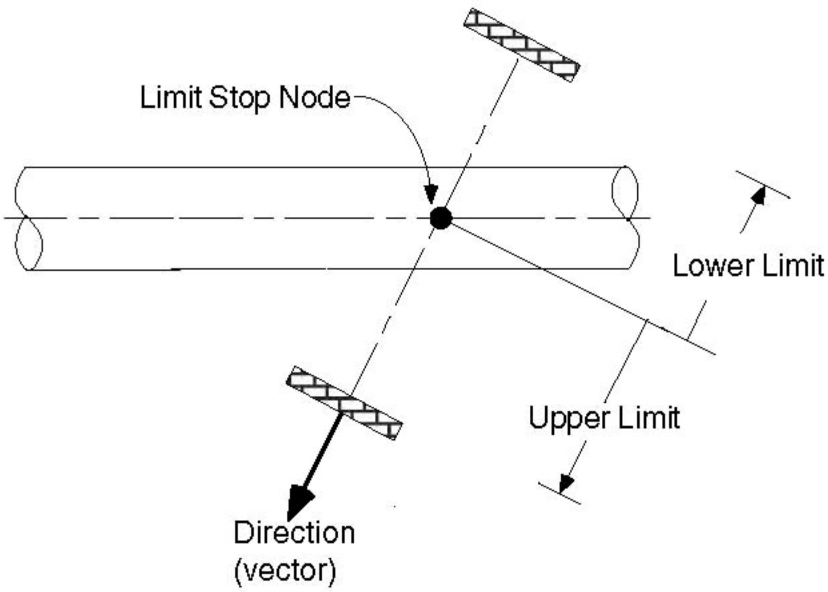




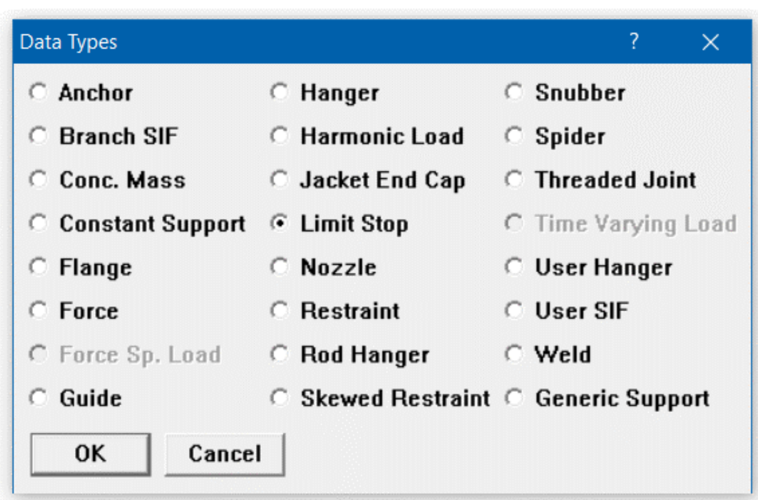
Limit Stop

In CAEPIPE, a limit stop is capable of modeling several types of physical supports including a guide, an anchor, a resting support, a two-way rigid restraint and a rod hanger. Using a combination of upper and lower gaps (limits), friction coefficient, support stiffness and a direction for its axis, you can model the above mentioned physical support types.

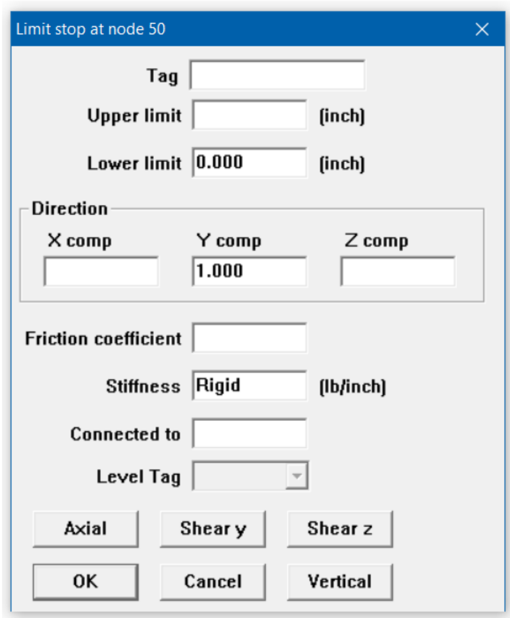
A limit stop prevents a node from moving beyond a certain distance (called a gap or a limit) in a certain direction. The node can move freely within the gap. After the gap closes, a limit stop acts as a rigid or flexible restraint (depending on your input for stiffness) resisting further movement of the node in the specified direction. If friction is specified, after the limit is reached, the friction force will oppose movement in the plane normal to the limit stop direction.



A limit stop is input by typing "l(L)" in the Data column or selecting "Limit Stop" from the Data Types dialog.



The Limit Stop dialog is shown.



Limits

Also called gaps, these limits, upper and lower, are the gaps present on either side of the node. The gap in front of the node in the direction of the vector is called the upper limit, and the gap to the rear of the node is called the lower limit. The gap is measured from the undeflected position of the node.

Typically, the upper limit is positive and the lower limit negative. In some situations, it is possible to have a positive lower limit or a negative upper limit, which is the same as forcefully displacing the node in that direction by the gap specified.

The algebraic value of the upper limit must be greater than the lower limit. For example, upper limit = –0.125”, lower limit= –0.25”, or upper limit = 0.25”, lower limit = –0.25”.

If a particular limit does not exist (that is, a node can move freely on that side of the node), then that limit should be left blank (as in the case of a resting support, the upper limit should be blank assuming Y-vertical and Y comp = 1, as in the figure shown).

If there is no gap at all, then the corresponding limit should be explicitly input as zero. When zero is entered, the limit stop acts as a one-way restraint in that direction.

Direction

The direction in which the limit stop is oriented must be specified in terms of its global X, Y and Z components. Or use one of the preset buttons to orient the limit stop axis (see above image):

1. Vertical: To set the limit stop axis in the(global) vertical direction
2. Axial: To set the limit stop axis along the local-x direction (pipe axis)
3. Shear y: To set the limit stop axis in the local-y directionof pipe
4. Shear z: To set the limit stop axis in the local-z directionof pipe

If you have connected the limit stop node to another node in the same piping model, then unless the connected node is coincident with the limit stop node, the limit stop direction must not be input. It is calculated from the locations of the connected node and the limit stop node, and its *direction is oriented from the connected node to the limit stop node*.

Friction coefficient

If friction coefficient is specified, a friction force will oppose the movement in the plane normal to the limit stop direction when the gap is closed. This friction force is displayed in results under Limit stop support loads.

If you had several limit stops with friction coefficients specified, and you wanted to change all of those friction coefficients to the same value, use the Change command under the Edit menu.

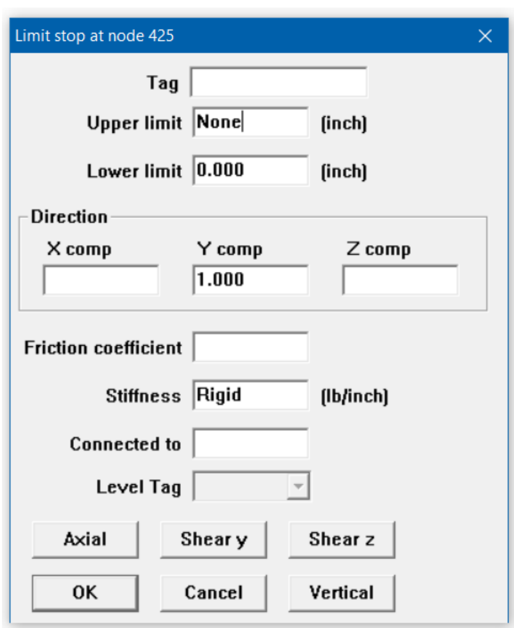
Stiffness

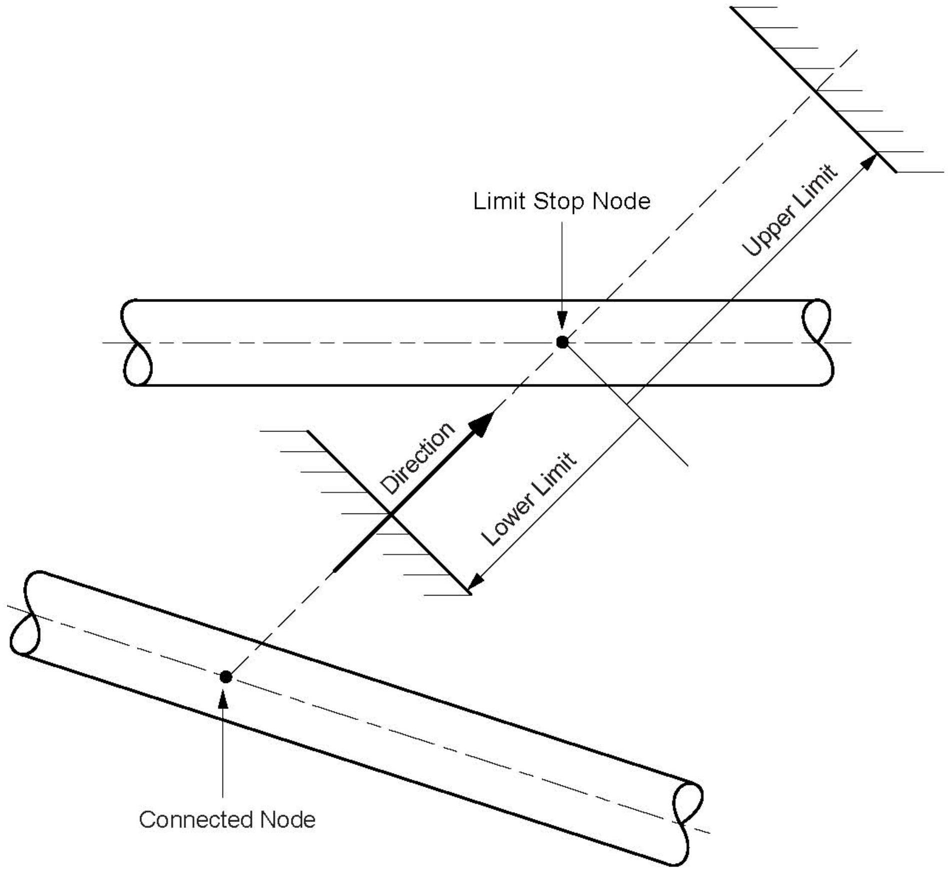
The default is set to Rigid stiffness. Other values may be input by estimating the stiffness of the support. (e.g., for a rod, stiffness = AE/L).

where
A = Cross-sectional area of rod
E = Young's modulus of rod material
L = Length of rod

Connected to Node

You can connect a limit stop node to another node in the same piping model. During gap and friction calculations, the relative displacements of the limit stop node are calculated with respect to the connected node. If you connect the limit stop node to an external fixed point (Ground point), leave the "Connected to Node" blank. See "Direction" above for information about how the direction is calculated depending on whether the connected node is coincident or not with the limit stop node.





Solution procedure

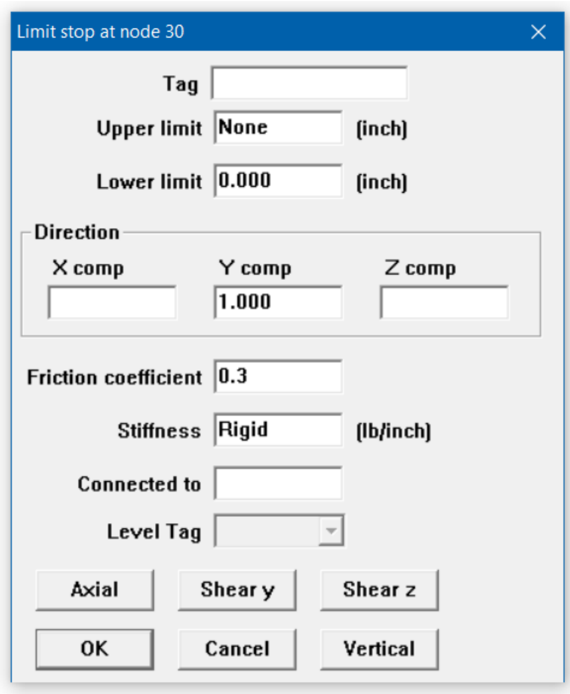
Limit stops require a nonlinear iterative solution. If you specify a friction coefficient, the following procedure is used for convergence: If the lower or upper limit is reached, the corresponding reaction force is calculated. The maximum friction force is the product of friction coefficient and the reaction force. The solution converges when the displacement varies by less than 1% between successive iterations.

Limit stops are included in dynamic analysis. The status of the limit stops arrived at upon completion of all iterative calculations for the first operating case (W+P1+T1) is used during dynamic analysis. If either the lower limit or the upper limit is reached at the end of iterations for the first operating case, then that limit stop is treated as a rigid two-way restraint in the direction of the limit stop during dynamic analysis. If both limitsare not reached, then that limit stop is ignored during dynamic analysis.

Example 1: Vertical 1-way restraint

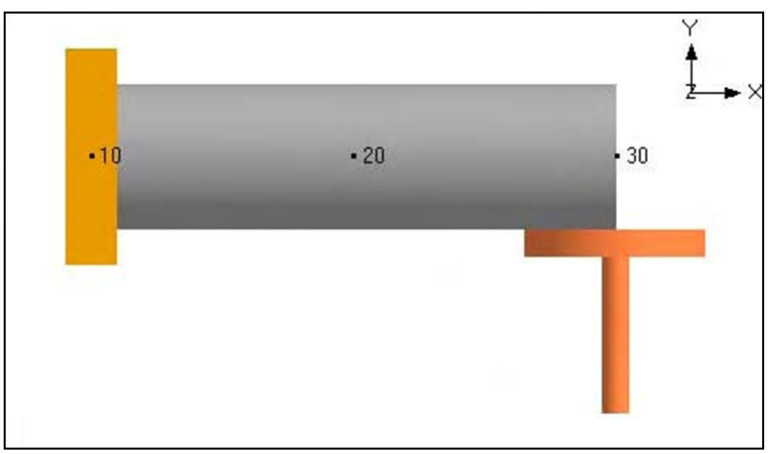
Assume that you have a vertical 1-way support with the following data: Upper limit = None, Lower limit = 0, Friction coefficient = 0.3, Direction vector of the limit stop is vertical along +Y axis.

Model the pipe up to the Limit stop node 30. At node 30, type “l(L)” in the Data column. The limit stop dialog will be shown.



In the limit stop dialog, press “Vertical” button. The data is automatically entered [0.000 for Lower limit, None for Upper limit (blank), 1.000 for Y comp (vertical)]. Enter 0.3 for friction coefficient (this acts in the X-Z plane which is perpendicular to the direction of the limit stop).

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20		1'0"			1	6	1	
4	30		1'0"			1	6	1	Limit stop



Example 2: Pipe Slide/Shoe Assembly

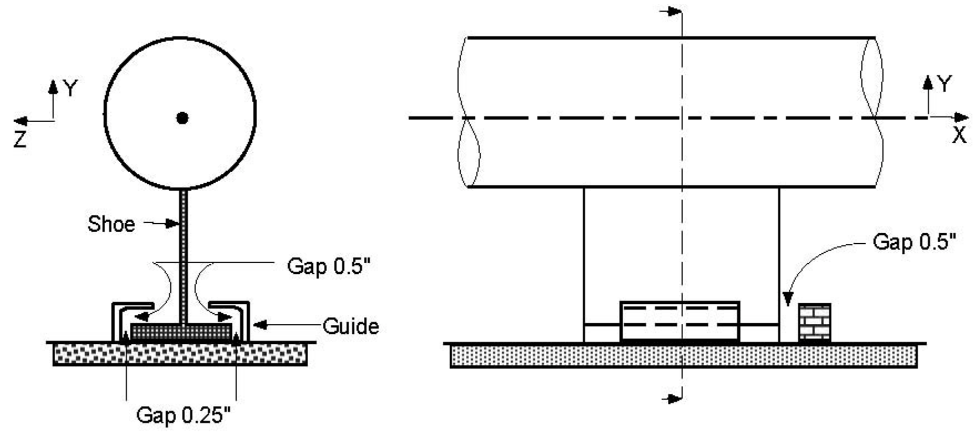
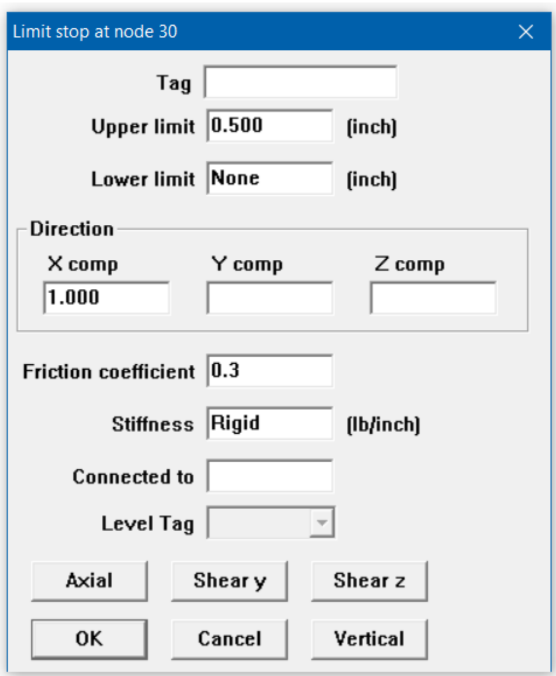


Figure not to scale

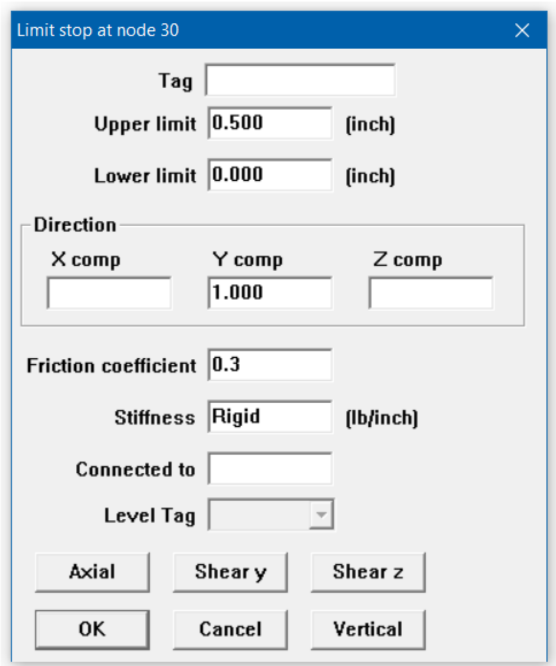
The assembly is modeled using three limit stops, one in each direction.

- When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- Press Tab in the Node column which puts the node number 20 automatically. Type 5' for DX, enter material, section and load names, press Enter.
- Press Tab in the Node column which puts the node number 30 automatically. Type 5' for DX, Tab to Data column and type "l(L)" to open limit stop dialog. Input line stop with gap (0.5") along X axis (Notice that this is a one-way restraint; there is only one stop block along +X). Type Upper limit 0.5", leave lower limit blank, Direction as (X comp = 1, Y comp = 0, Z comp = 0), Friction coefficient=0.3, press Enter.

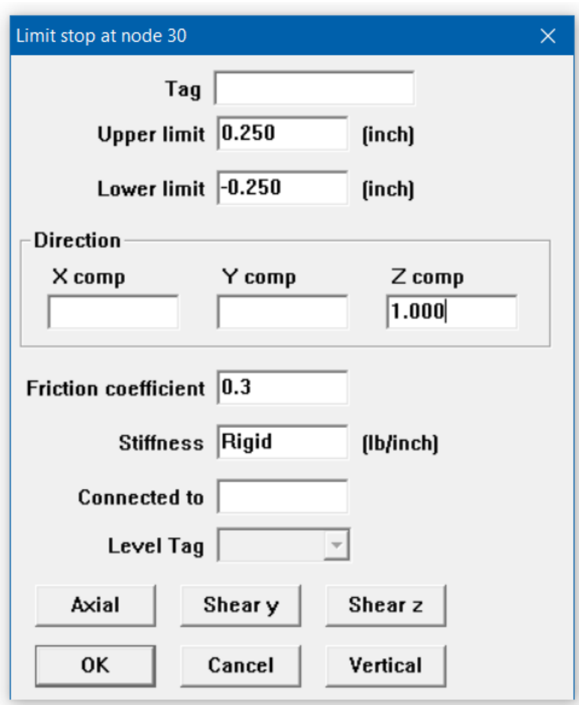


- Create a limit stop in the Y direction. Type 30 for Node, press Tab to move to Type field, type "L" for Location, choose Limit stop from the Data Types dialog, in the

Limit stop dialog, type Upper limit = 0.5”, Lower limit = 0.0, Direction (X comp =0, Y comp = 1, Z comp = 0), Friction coefficient=0.3, press Enter.



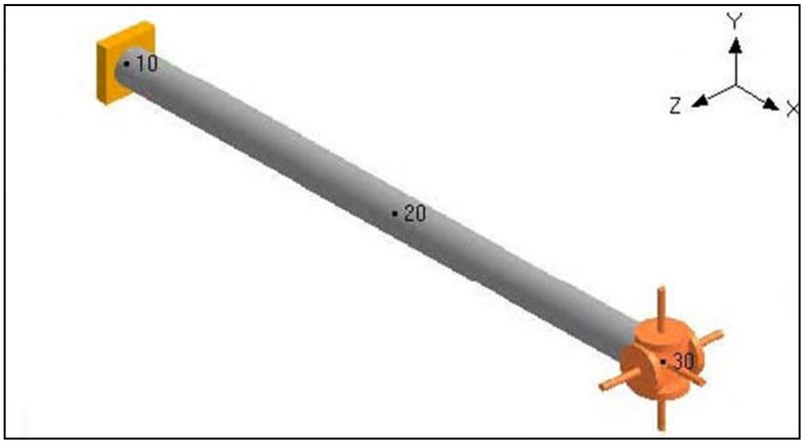
- Create a limit stop in the Z direction. Type 30 for Node, press Tab to move to Type field, type “L” for Location, choose Limit stop from the Data Types dialog, in the Limit stop dialog, type Upper limit = 0.25”, Lower limit = –0.25”, Direction (X comp =0, Y comp = 0, Z comp = 1), Friction coefficient =0.3, press Enter.



The Layout window is shown below.

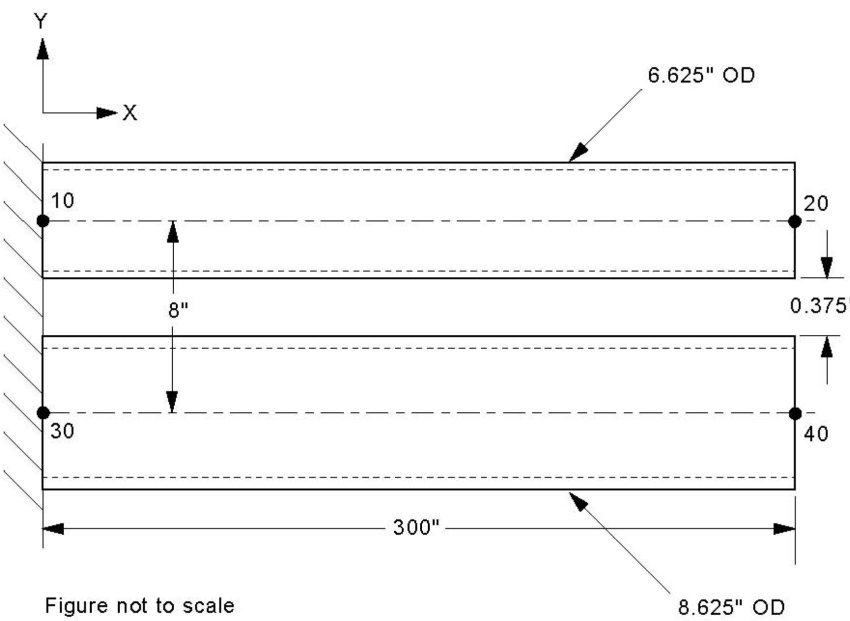
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Pipe slide/shoe assembly								
2	10	From							Anchor
3	20		5'0"			1	6	1	
4	30		5'0"			1	6	1	Limit stop
5	30	Location							Limit stop
6	30	Location							Limit stop

The Graphics window is shown below.



Example 3: Limit Stop Connected to Another Node

As shown in the figure below, two 300” long cantilever pipes are connected with an 8” separation between center lines.

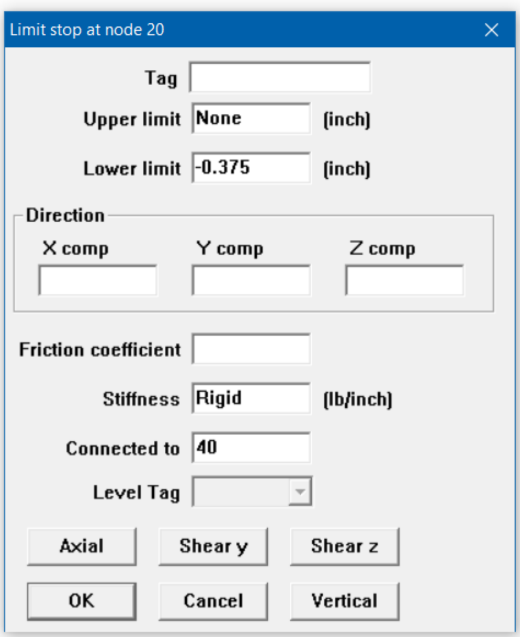


The gap between the bottom of the top pipe and the top of the bottom pipe is

8” – (6.625” + 8.625”) / 2 = 0.375”.

If pipes were free to deflect downward due to deadweight, the top pipe will deflect 1.908” and the bottom pipe will deflect 1.116” at the free ends. The relative deflection between them will be 1.908” – 1.116” = 0.792”. This however is not possible because when the relative deflection exceeds 0.375” the pipes will touch. This situation can be modeled using a limit stop connecting the free ends of the pipes. In this case the top pipe deflects 1.608”, i.e., less than the 1.908” free deflection because it is resisted by the bottom pipe. The bottom pipe deflects 1.233”, i.e., more than the 1.116” free deflection because additional load is imposed on it by the top pipe when they touch. The difference between the deflections is 1.608 – 1.233 = 0.375” as expected.

- When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- Press Tab in the Node column which puts the node number 20 automatically. Type 300” for DX, enter material, 6” section and load names, press Enter.
- For the bottom pipe, start with node 30 of Type “From” at DY = –8” and make it an anchor. On the next row enter node 40 with DX = 300” and 8” section. Press Enter to go to the next row.
- Enter the limit stop at node 20. Type 20 for Node, press Tab to move to Type field, type “L” for Location, choose Limit stop from the Data Types dialog, in the Limit stop dialog, leave the Upper limit blank and input Lower limit = -0.375”. Input the Connecting Node as 40. Since this limit stop connects two nodes, the direction should be left blank. The direction is implicitly from node 40 to node 20.



Alternatively the limit stop could be specified at node 40 connected to node 20. The direction now would be from node 20 to 40. The limits would then be Lower limit = None and Upper limit = 0.375” since the direction is now reversed compared to the previous case. Both these cases will give identical results.

The Layout window is shown below.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Limit stop connected to another node								
2	10	From							Anchor
3	20		300			1	6	1	
4	30	From		-8					Anchor
5	40		300			1	8	1	
6	20	Location							Limit stop

The Graphics is shown below.

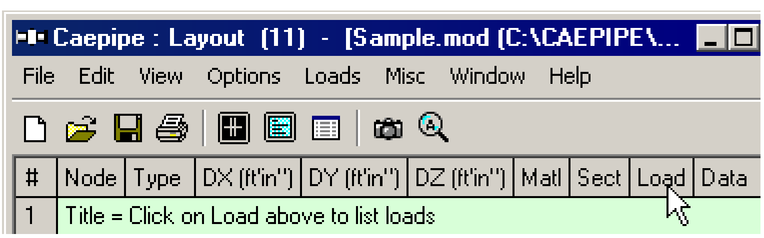
Loads on a piping system can be many and varied along its routing. Different piping segments may experience different pressures and temperatures depending on process requirements, different loads (snow, wind, etc.) depending on their physical locations and carry different states of a fluid between different points in a piping system. CAEPIPE offers a flexible method to input as many loads as required for as many segments or elements as needed with the least effort.

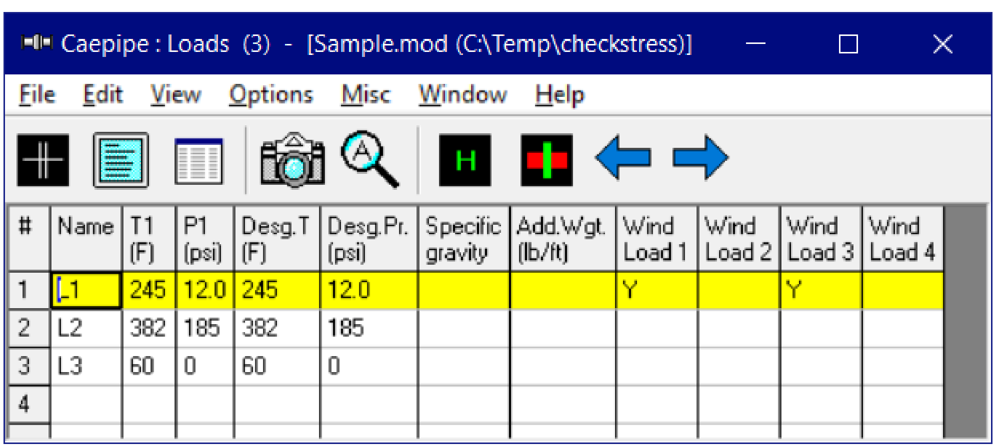
So, Load allows you to apply a temperature and pressure, specify weight of the operating fluid and add additional weight (e.g., due to snow load) on each element (if required) or for a range of elements in the model. Also the wind load can be turned "on" or "off" for each element (if required) or for a range of elements in the model.

After specifying the requested information here, including a name, use it under the Load column on the Layout window to associate the load information with an element.

Each load allows up to 10 operating conditions for Temperature and Pressure depending on the "Number of Thermal Loads" specified under Options > Analysis > Temperature. This Load is not to be confused with Load cases [which are combinations of load(s)] found under the Loads menu in the Layout window. Load cases are analysis cases (Sustained case, Thermal case, Operating case, Static Seismic case, etc.) for which CAEPIPE computes a set of results.

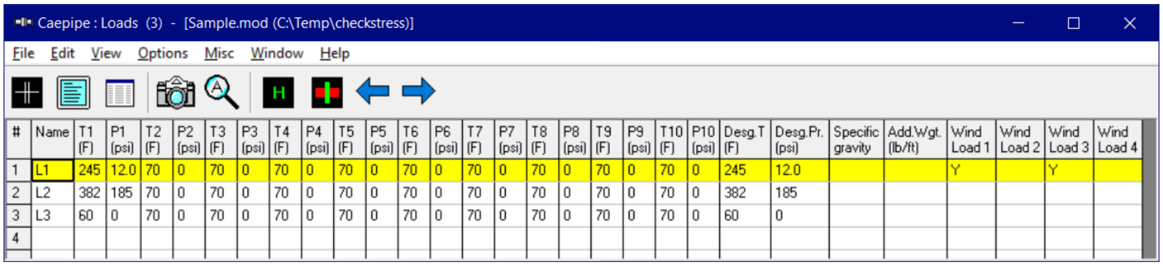
To define a new load, click on Load in the Header row in the Layout window (or select Loads under the Misc menu, hotkey: Ctrl+Shift+L). This opens a List window that displays currently defined loads.





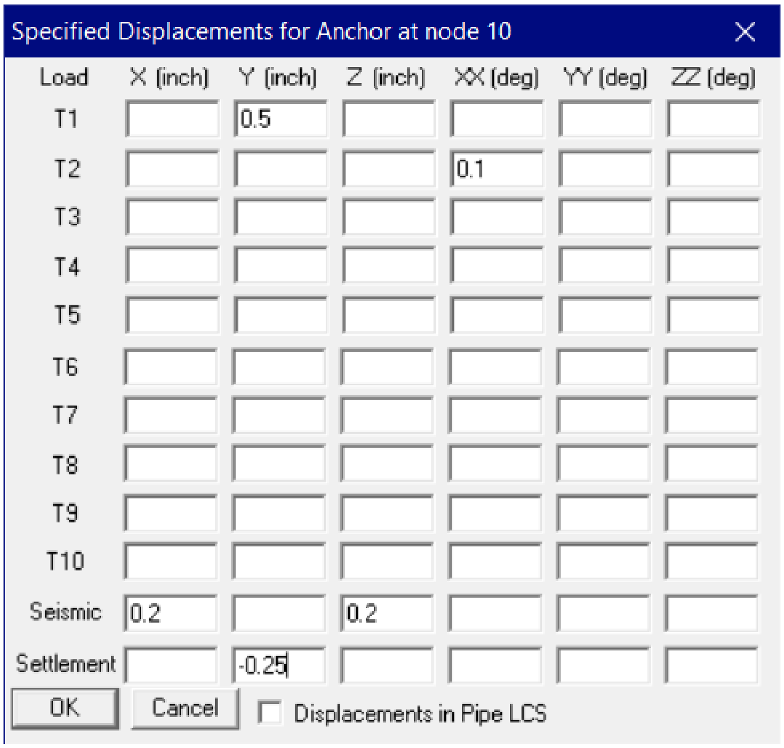
Either you can start typing the load data directly here into the fields or double click on an empty row to enter data through a dialog.

Depending on the number of thermal loads specified (under Options > Analysis > Temperature), up to 10 temperature/pressure load sets (T1/P1, T2/P2, T3/P3,…,T10/P10) can be input for each element or for a range of elements.

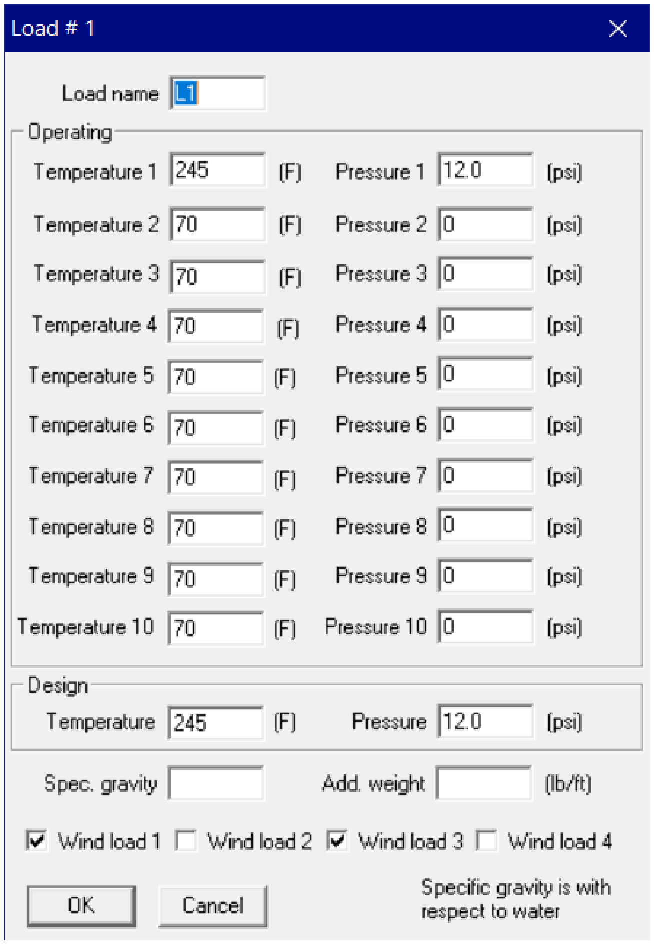


#	Name	T1 (F)	P1 (psi)	T2 (F)	P2 (psi)	T3 (F)	P3 (psi)	T4 (F)	P4 (psi)	T5 (F)	P5 (psi)	T6 (F)	P6 (psi)	T7 (F)	P7 (psi)	T8 (F)	P8 (psi)	T9 (F)	P9 (psi)	T10 (F)	P10 (psi)	Desg.T (F)	Desg.Pr. (psi)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load 1	Wind Load 2	Wind Load 3	Wind Load 4
1	L1	245	12.0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	245	12.0			Y		Y	
2	L2	382	185	70	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	382	185						
3	L3	60	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	70	0	60	0						
4																													

Up to 10 specified thermal displacements can be entered for Anchor and Nozzle data types.



Load	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
T1		0.5				
T2				0.1		
T3						
T4						
T5						
T6						
T7						
T8						
T9						
T10						
Seismic	0.2		0.2			
Settlement		-0.25				



Load Name

Type an alphanumeric name (up to five characters) in this field. The name can be changed later.

Temperatures

Type up to 10 operating temperatures. The maximum of the 10 temperatures is used to look up the corresponding allowable stress for the material used in code evaluation.

The other quantities looked up using these temperatures are the thermal expansion coefficients (alpha) and the temperature-dependent moduli.

Make sure to select number of thermal loads equal to two, three or 10 under Options > Analysis > Temperature, when you have more than one set of temperature and pressure.

Pressures

Type up to 10 operating pressures that correspond to the 10 operating temperatures above. The maximum of the 10 pressures is used to calculate the pressure stress term $[PD/4t \text{ or } Pd^2/(D^2 - d^2)]$, specified under Options > Analysis > Pressure.

Specify gauge pressures for Pressures input. Negative (external) pressures may be specified, too. But, the longitudinal pressure stress term (pD/4t) will still be positive according to the piping codes. Internal pressure will expand the pipe cross-section radially outward while external (negative) pressure will contract the pipe cross-section radially inward.

Design Pressure and Temperature

CAEPIPE requires that the Design Temperature to be entered should be equal to or greater than the algebraic maximum of all operatingtemperatures (T1 through T10). Similarly, Design Pressure to be entered should be equal to or greater than thealgebraic maximum of all operating pressures (P1 through P10).

Design Temperature so entered will be used to determinethe allowable stress for material, which is in turn used to compute the Allowable Pressure as per the piping code selected. Since Allowable Pressure reduces with decreasing allowable stress, to be conservative,the least allowable stress would then be obtained when the Design Temperatureis no less than the algebraic maximum of all operating temperatures (T1 through T10).

The Allowable Pressure so computed as per the piping code selected is then compared against the Design Pressure entered above and reported in the Code Compliance results.

Except as stated above, Design Temperature and Design Pressure are not used anywhere else during CAEPIPE analysis.

Specific Gravity

Specific gravity is the ratio of the density of a fluid to the density of a reference substance (in this case, water). Enter the specific gravity of the operating fluid inside the pipe. This input is used to calculate the weight of the operating fluid, which is added to the weight of the pipe. Specific gravity is with respect to water.

Additional weight

The value you enter here is taken as weight per unit length of the element and this total additional weight is added to the weight of the pipe. (Total additional weight = Length of element x Additional weight per unit length). Additional Weight is to be input in lbf/ft or kgf/m.

Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

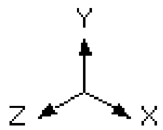
For example, Additional weight could be used to apply the weight of snow on the pipe.

Wind load 1/2/3/4

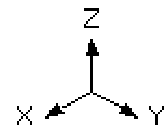
Type Y(es) or N(o) to apply or not apply the wind loads for this element. When you press Y(es), the wind load (entered as a separate load under Loads menu) is applied to this element.

For example, this is useful when you have a part of a piping system exposed to wind with the remaining part inside a building. In such a case, you should define two Loads, all data the same except that one has Wind load and the other does not. The Load with the wind load is applied to those elements that are affected by wind.

CAEPIPE allows you to create your physical piping system in a mathematically equivalent 3D Cartesian coordinate space with a global origin, which is the point of intersection of three planes orthogonal to each other, with three axes commonly denoted X, Y and Z (with either of the latter two vertical).



Y Vertical



Z Vertical

Once you begin creating your system from a given point (usually the global origin), you route your piping system one element at a time until you get to the end of the line(s). An element's orientation could be different from another element's, thereby necessitating an element's own "elemental coordinate system," which is commonly referred to as the Local Coordinate System (LCS), provided for the purpose of understanding the local forces and moments on each element. This system can be turned on (graphically) through the View menu > Show LCS command while you are viewing "Element Forces in Local Coordinates" in the Results window.

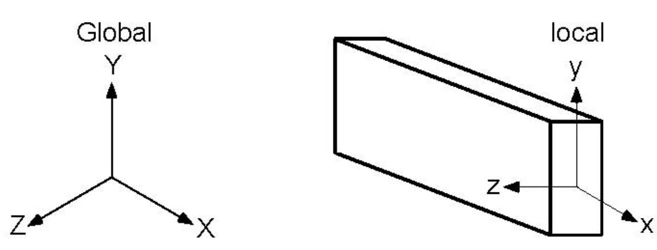
For a straight element (such as a pipe or a beam), the "local x" axis is along the element, from the "From" node to the "To" node. For a node location such as a guide, the local axes are based on the previous connected element. If the preceding element does not exist, the following element is used. The local y-axis and local z-axis are calculated differently depending on whether the vertical direction is Y or Z and also depending on whether the element is in the vertical direction.

The local coordinate system may be displayed graphically (for beams and guides in the input processor and for all the elements in the output processor) by selecting the "Show LCS (Local coordinate system)" command from the View menu.

In CAEPIPE, the local coordinate system is indicated by the lower case x, y and z letters, whereas the global coordinate system is indicated by the upper case X, Y and Z letters.

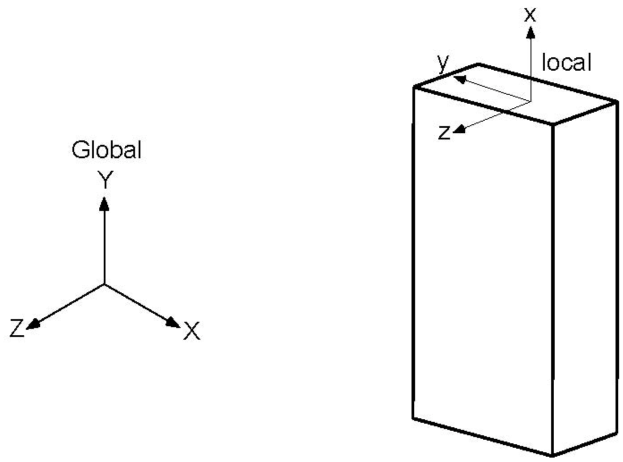
Global vertical axis is Y

Element is not Vertical



The local y-axis of the element lies in the local x - global Y plane (i.e., vertical plane) and is in the same positive direction as the global Y axis. The local z-axis is the cross product of the local x-axis and local y-axis.

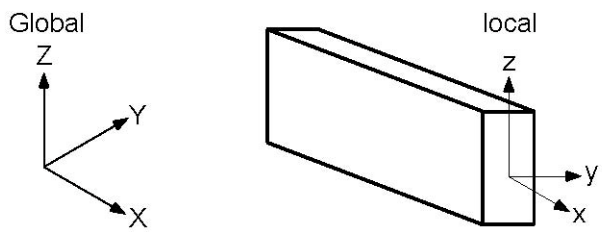
Element is Vertical



The local z-axis of the element is in the global Z direction. The local y-axis is in the global –X direction.

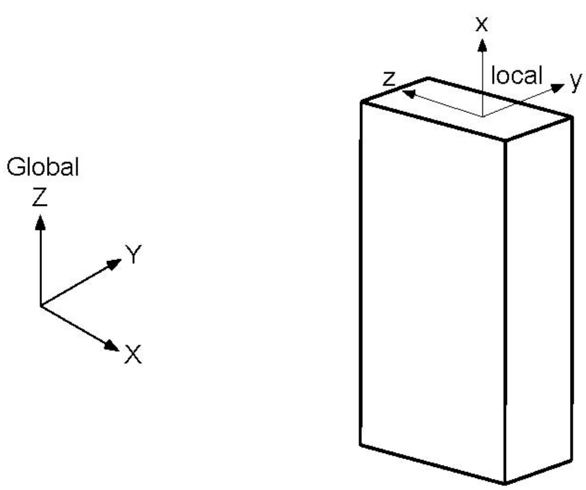
Global vertical axis is Z

Element is not Vertical



The local z-axis of the element lies in the local x - global Z plane (i.e., vertical plane) and is in the same positive direction as the global Z-axis. The local y-axis is the cross product of the local z-axis and local x-axis.

Element is Vertical

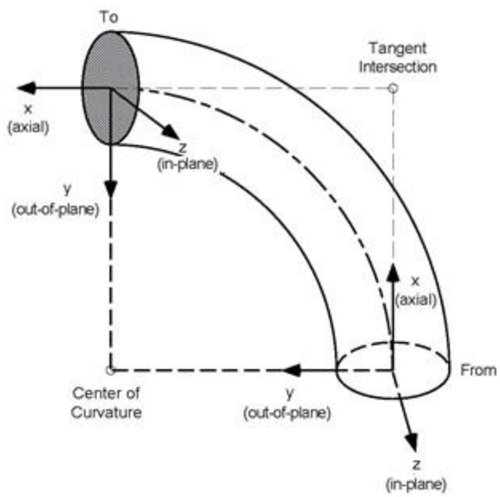


The local y-axis of the element is in the global Y direction. The local z-axis is in the global–X direction.

Local Coordinate System for a Bend

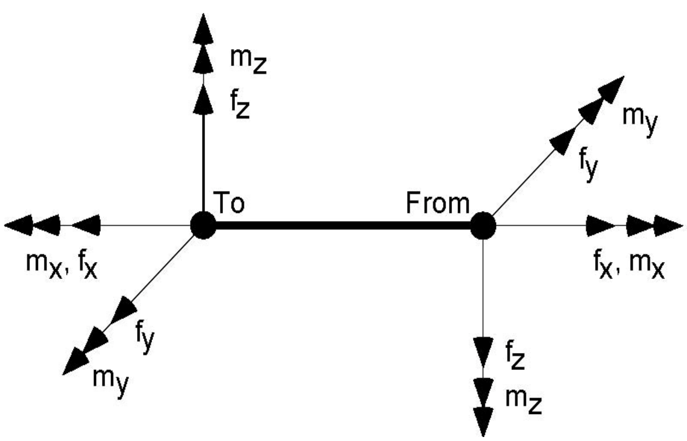
For a bend, at the "From" node, the local x axis is along the tangent from the "From" node to the tangent intersection point. The local y-axis is along the radius and points to the center of curvature. The local z-axis is the cross product of the local x-axis and local y-axis.

Similarly, at the "To" node, the local x-axis is along the tangent line from the tangent intersection point to the "To" node. The local y-axis is along the radius and points to the center of curvature. The local z-axis is the cross product of the local x-axis and local y-axis.



Sign Convention for Element Forces and Moments

The sign conventions for the element forces and moments in the local coordinate system follow strength of materials conventions, i.e., forces and moments at the "To" node of an element are positive if they are in the positive local axes directions of the element. On the other hand, forces and moments at the "From" node of the element are negativeif they are in the positive local axes directions of the element.



Positive sign conventions for local forces and moments are shown above at the "From" and "To" nodes of an element. Note that positive directions at the "From" node are reversed compared to the positive directions at the "To" node.

In-plane and Out-of-plane Moments

CAEPIPE outputs local moments in the form of Torsion, In-plane and Out-of-plane for a few piping analysis codes such as ASME B31.1 (2020) or later, ASME B31.3, etc.

To have graphical illustrations of the In-plane and Out-of-plane moments, shown below are the two sample CAEPIPE models created by keeping Global vertical axis as Z and Y respectively along with the following outputs.

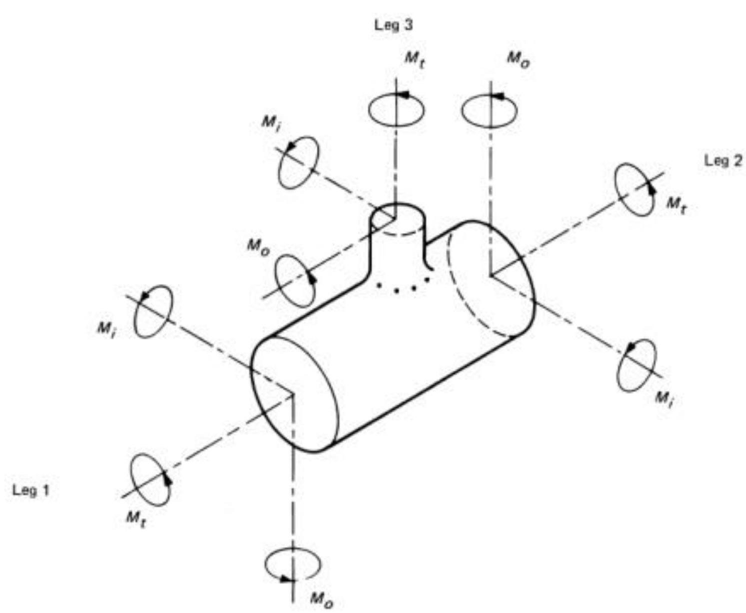
1. Local Coordinate System (LCS) captured for different elements by turning on the option Results Window > View menu > Show LCS command while in "Element Forces in Local Coordinates".

2. Local forces and moments output by CAEPIPE with piping analysis code selected as ASME B31.9, and

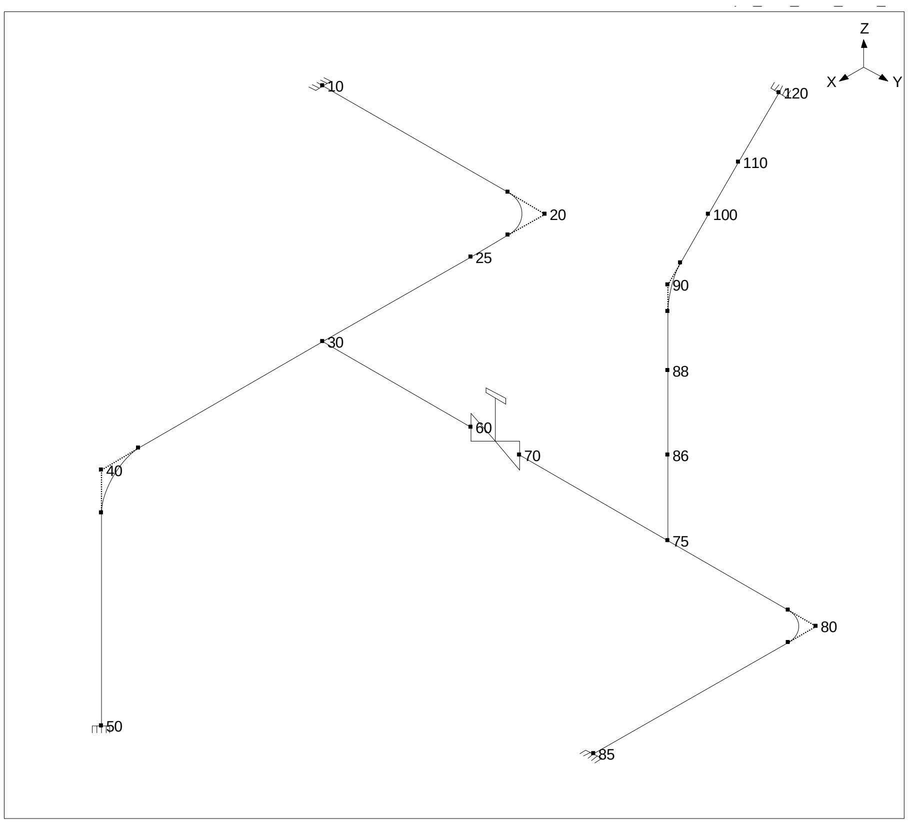
3. Local forces and moments output by CAEPIPE with piping analysis code as ASME B31.1.

From the graphical representation of LCS and the local forces and moments output by CAEPIPE using ASME B31.9 and ASME B31.1 codes, you may observe that the In-plane moment is about local z-axis and its corresponding rotation is in local x-y plane, and whereas the Out-of-plane moment is about local y-axis and its corresponding rotation is in local x-z plane for all element types such as Pipe, Bend, Mitre, Reducer, Valve, Rigid, etc. (excepting TEEs) available in CAEPIPE.

Similarly, from the LCS and the local forces and moments output by CAEPIPE using ASME B31.9 and ASME B31.1 codes, you may observe that the In-plane moment (Mi) of a Tee element is about the vector normal to the plane formed by connecting the two nodes on the Run side (Leg 1 and Leg 2) as well as a node on the Branch side (Leg 3) as shown in the figure below. Similarly, Out-of-plane moments (Mo) for the three legs of Tee are as shown below.



When Global Z Axis Vertical



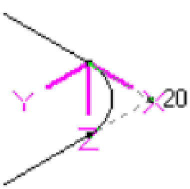


Fig. A.1

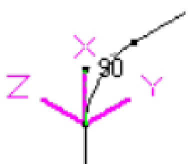


Fig. A.2

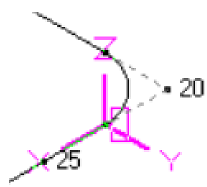


Fig. A.3

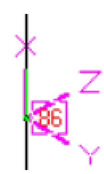


Fig. A.4

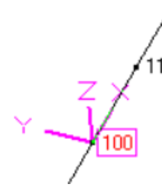


Fig. A.5

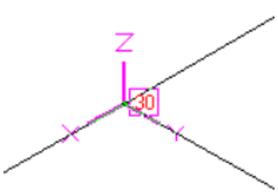


Fig. B.1

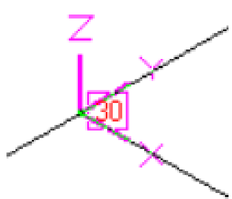


Fig. B.2

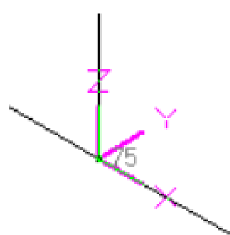


Fig. B.3

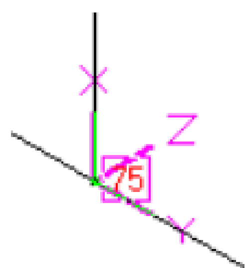


Fig. B.4

Fig. A.1 - Bend in Horizontal Plane
Fig. A.2 - Bend in Vertical Plane
Fig. A.3 - Pipe in Horizontal Plane
Fig. A.4 - Pipe Vertical Plane
Fig. A.5 - Pipe Skewed in 3D

Fig. B.1 - Tee in Horizontal Plane – Run Element
Fig. B.2 - Tee in Horizontal Plane – Branch Element
Fig. B.3 - Tee in Vertical Plane – Run Element
Fig. B.4 - Tee in Vertical Plane – Branch Element

ASME B31.9 Code Selected

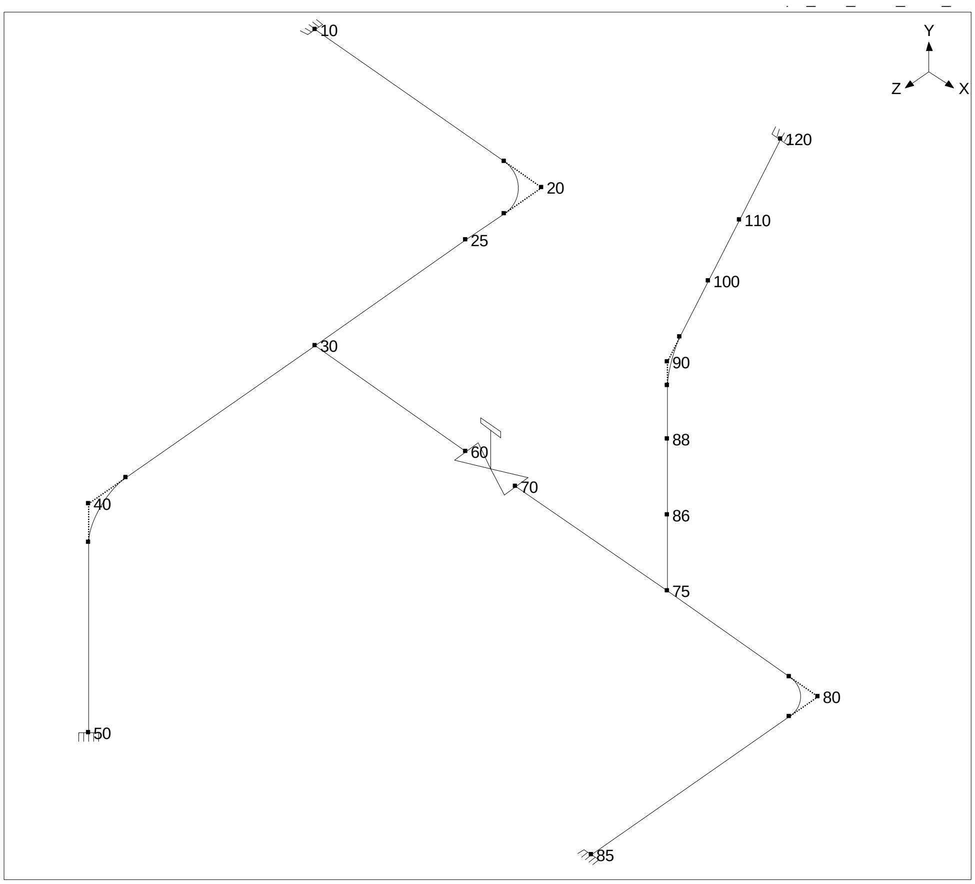
#	Node	fx (lb)	fy (lb)	fz (lb)	mx (ft-lb)	my (ft-lb)	mz (ft-lb)	SIF	Sopr (psi)
1	10 20A	-3426 -3426	6219 6219	-124 207	-2287 -2287	1413 1620	23832 -7262		11903 3973
2	20A 20B	-3426 -6219	-6219 3426	-207 -311	-2287 -1864	-1620 2014	7262 10054	1.77 1.77	6695 8860
3	20B 25	-6219 -6219	-3426 -3426	311 377	-1864 -1864	-2014 -1671	-10054 -6628		5046 3412
4	25 30	-6219 -6219	-3426 -3426	377 641	-1864 -1864	-1671 365	-6628 7077	 1.71	3412 5948
5	30 40A	-5840 -5840	479 479	1934 2265	2058 2058	-1029 9470	3528 1134	1.71	3212 4750
6	40A 40B	-5840 -2369	-2265 5840	479 479	2058 -655	-1134 -1580	9470 5932	1.77 1.77	8284 5565
7	40B 50	-2369 -2699	-479 -479	5840 5840	-655 -655	-5932 23271	-1580 814		3268 11623
8	30 60	-3905 -3905	378 378	-1293 -1155	1394 1394	3923 -973	3549 2036	1.71	7146 3638
9	70 75	-3905 -3905	378 378	-699 -561	1394 1394	-2200 -4720	1535 22	 2.14	4162 14264
10	75 80A	-3402 -3402	1296 1296	675 787	-1875 -1875	-2070 306	-313 -4526	2.14	6830 6915
11	80A 80B	-3402 -1296	-1296 3402	-787 -828	-1875 -907	-306 1265	4526 2947	2.27 2.27	14738 10701
12	80B 85	-1296 -1296	-3402 -3402	828 1010	-907 -907	-1265 3560	-2947 14912		5067 22051
13	75 86	-1236 -1166	-503 -503	-918 -918	335 335	3268 1432	-2650 -1643	2.14	13070 3498
14	86 88	-1166 -1097	-503 -503	-918 -918	335 335	1432 -404	-1643 -637		3498 1560
15	88 90A	-1097 -1050	-503 -503	-918 -918	335 335	-404 -1667	-637 56		1560 2807
16	90A 90B	-1050 -839	-665 875	808 808	335 1216	671 389	1527 1389	2.27 2.27	5763 5366
17	90B 100	-839 -831	808 808	-875 -828	1216 1216	1389 216	-389 -1502		3106 3190
18	100 110	-831 -821	808 808	-828 -777	1216 1216	216 -988	-1502 -2714		3190 4871
19	110 120	-821 -809	808 808	-777 -710	1216 1216	-988 -2458	-2714 -4312		4871 7665

ASME B31.1 Code selected

#	Node	Axial (lb)	y Shear (lb)	z Shear (lb)	Torsion(ft-lb)		Inplane(ft-lb)		Outplane(ft-lb)		Flex. Factors			Sopr (psi)
					Moment	SIF	Moment	SIF	Moment	SIF	FFi	FFo	FFt	
1	10 20A	-3426 -3426	6219 6219	-124 207	-2287 -2287		23832 -7262		1413 1620					11903 3973
2	20A 20B	-3426 -6219	-6219 3426	-207 -311	-2287 -1864	1.00 1.00	7262 10054	1.77 1.77	-1620 2014	1.47 1.47	3.53 3.53	3.53 3.53		6649 8808
3	20B 25	-6219 -6219	-3426 -3426	311 377	-1864 -1864		-10054 -6628		-2014 -1671					5046 3412
4	25 30	-6219 -6219	-3426 -3426	377 641	-1864 -1864	 1.43	-6628 7077	 2.31	-1671 365	 1.00				3412 8048
5	30 40A	-5840 -5840	479 479	1934 2265	2058 2058	1.43	3528 1134	2.31	-1029 9470	1.00				4243 4750
6	40A 40B	-5840 -2369	-2265 5840	479 479	2058 -655	1.00 1.00	9470 5932	1.77 1.77	-1134 -1580	1.47 1.47	3.53 3.53	3.53 3.53		8266 5511
7	40B 50	-2369 -2699	-479 -479	5840 5840	-655 -655		-1580 814		-5932 23271					3268 11623
8	30 60	-3905 -3905	378 378	-1293 -1155	1394 1394	1.00	-3549 2036	1.00	-3923 -973	1.25				8665 3638
9	70 75	-3905 -3905	378 378	-699 -561	1394 1394	 2.12	1535 4720	 1.66	-2200 22	 2.12	1.88 1.88	1.04 1.04	1.04 1.04	4162 11748
10	75 80A	-3402 -3402	1296 1296	675 787	-1875 -1875	2.12	2070 -4526	1.66	-313 306	2.12	1.88 1.88	1.04 1.04	1.04 1.04	7469 6915
11	80A 80B	-3402 -1296	-1296 3402	-787 -828	-1875 -907	1.00 1.00	4526 2947	2.27 2.27	-306 1265	1.89 1.89	6.59 6.59	6.59 6.59		14728 10456
12	80B 85	-1296 -1296	-3402 -3402	828 1010	-907 -907		-2947 14912		-1265 3560					5067 22051
13	75 86	-1236 -1166	-503 -503	-918 -918	335 335	2.12	-2650 -1643	1.66	3268 1432	2.12	1.88 1.88	1.04 1.04	1.04 1.04	12010 3498
14	86 88	-1166 -1097	-503 -503	-918 -918	335 335		-1643 -637		1432 -404					3498 1560
15	88 90A	-1097 -1050	-503 -503	-918 -918	335 335		-637 56		-404 -1667					1560 2807
16	90A 90B	-1050 -839	-665 875	808 808	335 1216	1.00 1.00	1527 1389	2.27 2.27	671 389	1.89 1.89	6.59 6.59	6.59 6.59		5629 5317
17	90B 100	-839 -831	808 808	-875 -828	1216 1216		-389 -1502		1389 216					3106 3190
18	100 110	-831 -821	808 808	-828 -777	1216 1216		-1502 -2714		216 -988					3190 4871
19	110 120	-821 -809	808 808	-777 -710	1216 1216		-2714 -4312		-988 -2458					4871 7665

Element Type	In-plane Moment	Bending in what plane?	Direction of +ve In-plane Moment Vector	Out-of-plane Moment	Bending in what plane?	Direction of +ve Out-of-plane Moment Vector
Bend (20A-20B) & (90A-90B)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
Pipe Horizontal (20B-25), Pipe Vertical (86-88), Pipe Skewed (100-110), Reducer, etc.	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
Pipe (20B-25), Pipe (86-88), Pipe Skewed (100-110), Reducer, etc.	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
Pipe (20B-25), Reducer, etc.	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
TEE - Run (30-40A)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
TEE - Branch (30-60)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
TEE - Run (75-80A)	(my)	x-z plane	(y axis)	(mz)	x-y plane	(z axis)
TEE - Branch (75-90A)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
The definition summarized in the above table can be verified by comparing the local moment values output in the form of In-plane and Out-of-plane by CAEPIPE for ASME B31.1 code against the local moments output in the form of mx, my and mz by CAEPIPE for ASME B31.9 code.						

When Global Y Axis Vertical



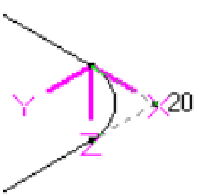


Fig. C.1

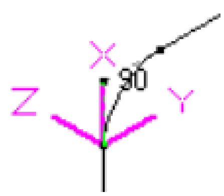


Fig. C.2

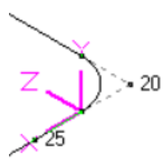


Fig. C.3

Fig. C.4

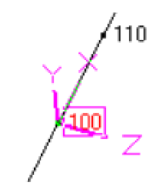


Fig. C.5

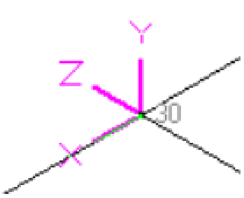


Fig. D.1

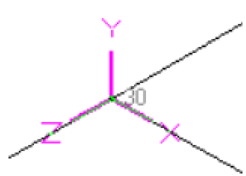


Fig. D.2

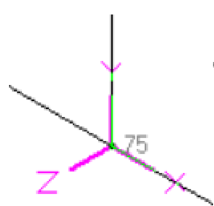


Fig. D.3

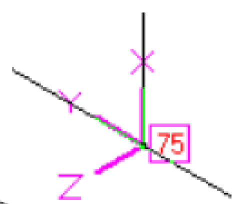


Fig. D.4

Fig. C.1 - Bend in Horizontal Plane
Fig. C.2 - Bend in Vertical Plane
Fig. C.3 - Pipe in Horizontal Plane
Fig. C.4 - Pipe Vertical Plane
Fig. C.5 - Pipe Skewed in 3D

Fig. D.1 - Tee in Horizontal Plane – Run Element
Fig. D.2 - Tee in Horizontal Plane – Branch Element
Fig. D.3 - Tee in Vertical Plane – Run Element
Fig. D.4 - Tee in Vertical Plane – Branch Element

ASME B31.9 Code Selected

#	Node	fx (lb)	fy (lb)	fz (lb)	mx (ft-lb)	my (ft-lb)	mz (ft-lb)	SIF	Sopr (psi)
1	10 20A	-3426 -3426	-124 207	-6219 -6219	-2287 -2287	23832 -7262	-1413 -1620		11903 3973
2	20A 20B	-3426 -6219	-6219 3426	-207 -311	-2287 -1864	-1620 2014	7262 10054	1.77 1.77	6695 8860
3	20B 25	-6219 -6219	311 377	3426 3426	-1864 -1864	-10054 -6628	2014 1671		5046 3412
4	25 30	-6219 -6219	377 641	3426 3426	-1864 -1864	-6628 7077	1671 -365	 1.71	3412 5948
5	30 40A	-5840 -5840	1934 2265	-479 -479	2058 2058	3528 1134	1029 -9470	1.71	3212 4750
6	40A 40B	-5840 -2369	-2265 5840	479 479	2058 -655	-1134 -1580	9470 5932	1.77 1.77	8284 5565
7	40B 50	-2369 -2699	-479 -479	5840 5840	-655 -655	-5932 23271	-1580 814		3268 11623
8	30 60	-3905 -3905	-1293 -1155	-378 -378	1394 1394	3549 2036	-3923 973	1.71	7146 3638
9	70 75	-3905 -3905	-699 -561	-378 -378	1394 1394	1535 22	2200 4720	 2.14	4162 14264
10	75 80A	-3402 -3402	675 787	-1296 -1296	-1875 -1875	-313 -4526	2070 -306	2.14	6830 6915
11	80A 80B	-3402 -1296	-1296 3402	-787 -828	-1875 -907	-306 1265	4526 2947	2.27 2.27	14738 10701
12	80B 85	-1296 -1296	828 1010	3402 3402	-907 -907	-2947 14912	1265 -3560		5067 22051
13	75 86	-1236 -1166	503 503	918 918	335 335	-3268 -1432	2650 1643	2.14	13070 3498
14	86 88	-1166 -1097	503 503	918 918	335 335	-1432 404	1643 637		3498 1560
15	88 90A	-1097 -1050	503 503	918 918	335 335	404 1667	637 -56		1560 2807
16	90A 90B	-1050 -839	-665 875	808 808	335 1216	671 389	1527 1389	2.27 2.27	5763 5366
17	90B 100	-839 -831	-875 -828	-808 -808	1216 1216	-389 -1502	-1389 -216		3106 3190
18	100 110	-831 -821	-828 -777	-808 -808	1216 1216	-1502 -2714	-216 988		3190 4871
19	110 120	-821 -809	-777 -710	-808 -808	1216 1216	-2714 -4312	988 2458		4871 7665

ASME B31.1 Code selected

#	Node	Axial (lb)	y Shear (lb)	z Shear (lb)	Torsion(ft-lb)		Inplane(ft-lb)		Outplane(ft-lb)		Flex. Factors			Sopr (psi)
					Moment	SIF	Moment	SIF	Moment	SIF	FFi	FFo	FFt	
1	10 20A	-3426 -3426	-124 207	-6219 -6219	-2287 -2287		-1413 -1620		23832 -7262					11903 3973
2	20A 20B	-3426 -6219	-6219 3426	-207 -311	-2287 -1864	1.00 1.00	7262 10054	1.77 1.77	-1620 2014	1.47 1.47	3.53 3.53	3.53 3.53		6649 8808
3	20B 25	-6219 -6219	311 377	3426 3426	-1864 -1864		2014 1671		-10054 -6628					5046 3412
4	25 30	-6219 -6219	377 641	3426 3426	-1864 -1864	 1.43	1671 7077	 2.31	-6628 365	 1.00				3412 8048
5	30 40A	-5840 -5840	1934 2265	-479 -479	2058 2058	1.43	3528 -9470	2.31	-1029 1134	1.00				4243 4750
6	40A 40B	-5840 -2369	-2265 5840	479 479	2058 -655	1.00 1.00	9470 5932	1.77 1.77	-1134 -1580	1.47 1.47	3.53 3.53	3.53 3.53		8266 5511
7	40B 50	-2369 -2699	-479 -479	5840 5840	-655 -655		-1580 814		-5932 23271					3268 11623
8	30 60	-3905 -3905	-1293 -1155	-378 -378	1394 1394	1.00	-3549 973	1.00	-3923 2036	1.25				8665 3638
9	70 75	-3905 -3905	-699 -561	-378 -378	1394 1394	 2.12	2200 4720	 1.66	1535 22	 2.12	1.88 1.88	1.04 1.04	1.04 1.04	4162 11748
10	75 80A	-3402 -3402	675 787	-1296 -1296	-1875 -1875	2.12	2070 -306	1.66	-313 -4526	2.12	1.88 1.88	1.04 1.04	1.04 1.04	7469 6915
11	80A 80B	-3402 -1296	-1296 3402	-787 -828	-1875 -907	1.00 1.00	4526 2947	2.27 2.27	-306 1265	1.89 1.89	6.59 6.59	6.59 6.59		14728 10456
12	80B 85	-1296 -1296	828 1010	3402 3402	-907 -907		1265 -3560		-2947 14912					5067 22051
13	75 86	-1236 -1166	503 503	918 918	335 335	2.12	-2650 1643	1.66	3268 -1432	2.12	1.88 1.88	1.04 1.04	1.04 1.04	12010 3498
14	86 88	-1166 -1097	503 503	918 918	335 335		1643 637		-1432 404					3498 1560
15	88 90A	-1097 -1050	503 503	918 918	335 335		637 -56		404 1667					1560 2807
16	90A 90B	-1050 -839	-665 875	808 808	335 1216	1.00 1.00	1527 1389	2.27 2.27	671 389	1.89 1.89	6.59 6.59	6.59 6.59		5629 5317
17	90B 100	-839 -831	-875 -828	-808 -808	1216 1216		-1389 -216		-389 -1502					3106 3190
18	100 110	-831 -821	-828 -777	-808 -808	1216 1216		-216 988		-1502 -2714					3190 4871
19	110 120	-821 -809	-777 -710	-808 -808	1216 1216		988 2458		-2714 -4312					4871 7665

Element Type	In-plane Moment	Bending in what plane?	Direction of +ve In-plane Moment Vector	Out-of-plane Moment	Bending in what plane?	Direction of +ve Out-of-plane Moment Vector
Bend (20A-20B) & (90A-90B)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
Pipe Horizontal (20B-25), Pipe (86-88), Pipe Vertical (86-88), Pipe Skewed (100-110), Reducer, etc.	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
TEE - Run (30-40A)	(my)	x-z plane	(y axis)	(mz)	x-y plane	(z axis)
TEE - Branch (30-60)	(my)	x-z plane	(y axis)	(mz)	x-y plane	(z axis)
TEE - Run (75-80A)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)
TEE - Branch (75-90A)	(mz)	x-y plane	(z axis)	(my)	x-z plane	(y axis)

The definition summarized in the above table can be verified by comparing the local moment values output in the form of In-plane and Out-of-plane by CAEPIPE for ASME B31.1 code against the local moments output in the form of mx, my and mz by CAEPIPE for ASME B31.9 code.

Summary:

From the above illustrations, you may note the following.

1. For an element, the "local x" axis is always along the element, from the "From" node to the "To" node. The local y-axis and local z-axis are calculated differently depending on whether the vertical direction is Y or Z and also depending on whether the element is in the vertical direction. Refer to the Section titled "Local Coordinate System" given above for details on how CAEPIPE computes local y-axis and z-axis for an element.

2. Local Coordinate System (LCS) of an element can be seen (turned on) graphically through the View menu > Show LCS command while you are viewing "Element Forces in Local Coordinates" in the Results window.

3. For all elements excepting Tees, irrespective of whether the element is horizontal / vertical / inclined (skewed in 3D) or whether the Global vertical axis is Y or Z, the In-plane moment is always about the local z-axis with its +ve direction along local +z axis and its corresponding rotation is in the local x-y plane; whereas, the Out-of-plane moment is always about local y-axis with its +ve direction along local +y axis and its corresponding rotation is in the local x-z plane.

4. For TEEs, the vectors for the In-plane moments (Mi) for the three legs of Tee are normal to the plane formed by connecting two nodes on the Run side (Leg 1 and Leg 2) as well as a node on the Branch side (Leg 3) as shown in the figure above. Similarly, the vectors for the Out-of-plane moments (Mo) for the three legs of Tee are as shown above.

5. Local x, y and z axes computed internally by CAEPIPE for an element (as defined in the Section titled "Local Coordinates System" given above) can be observed by inputting the "From" and "To" coordinates of that element in the excel macro that is available in the link www.sstusa.com/downloads/GCS_LCS.xlsx. This macro is used for computing Local forces and moments from Global forces and moments.

Several times, you may need to input more than one data item at a node, e.g., a hanger and a Branch SIF to designate the type of tee. At those times, use Location (press "L" in the Type field or pick Location from the Element Types dialog) to input more than one data item at a node. For example, you may want to input a lateral restraint at an existing hanger.

Another use for Location is when you want to input a data item at an internally generated node. Nodes are internally generated by CAEPIPE for bends (A, B nodes, e.g., 20A) and Jacketed pipes/bends (J, C, D nodes, e.g., 10J).

By design, each row in the Layout window allows only one data item to be entered under the Data field. Additional data items can be input only through Location (see Examples below).

Ensure that the node you use for Location has already been defined on an earlier row, ordefined earlier as an intermediate node for a bend or is an internally generated node. In other words, you cannot use Location on an undefined node.

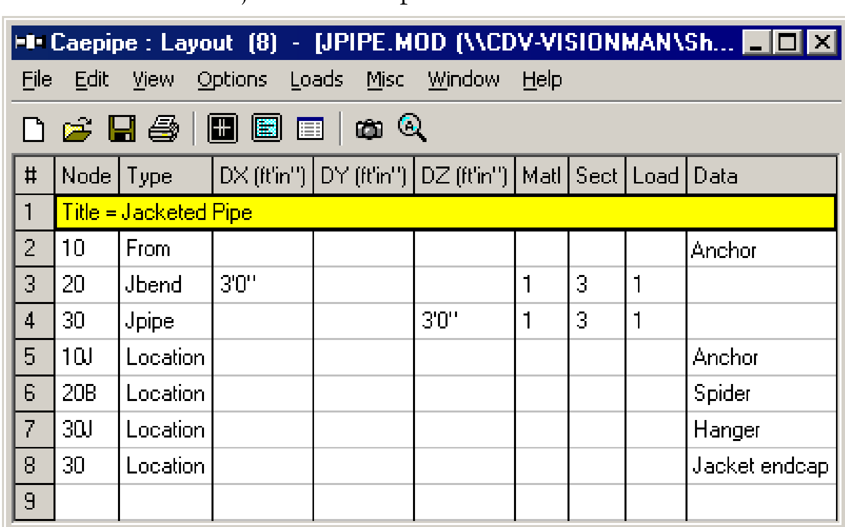
Example 1: Multiple Limit Stops at a node to modelPipe Slide and Shoe assembly

See Example 2 under the Limit Stoptopicinthis manual.

Example 2: Data at internally generated nodes

Refer to the example given underthe JacketedPiping in this manual. CAEPIPE internally generates J node on the jacket for a Jacketed pipe and the C and D nodes on the jacket for a Jacketed Bend. The following 4 data items present in that example are input using the Location type as shown in the Layout window below.

1. Node 10J is the starting Jacket node which is anchored (node 10 is separately anchored, since it is the node on the core pipe).
2. There is a hanger at node 30J since the hanger is connected to the Jacket.
3. There is a spacer (spider) at the far end of the bend, node 20B (which is on the core pipe). Remember that the bend has a jacket on the outside.
4. Node 30 has a jacket end cap



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Jacketed Pipe								
2	10	From							Anchor
3	20	Jbend	3'0"			1	3	1	
4	30	Jpipe			3'0"	1	3	1	
5	10J	Location							Anchor
6	20B	Location							Spider
7	30J	Location							Hanger
8	30	Location							Jacket endcap
9									

Example 3: Hanger at a Bend Intermediate Node

See Example 6 in the Bend section of this manual to locate a hanger at an intermediate node.

Lugs (integral attachments) are forged attachments or attachments welded on the pressure-loaded wall of a straight pipe which transfer piping loadings to the steel framework or concrete.

Loads on attachments cause local stresses in the pipe wall. Equations to determine these pipestresses at lug attachments are given in different codes. These local stresses are then added to the piping system stresses at the attachments. The combined stresses thus obtained are checked for compliance with the appropriate equations given in those codes.

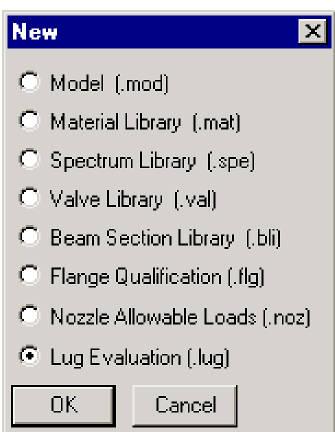
The Lug Evaluation module implemented in CAEPIPE computes local pipe stresses as per the following codes for Rectangular and Hollow Circular cross sectional attachments.

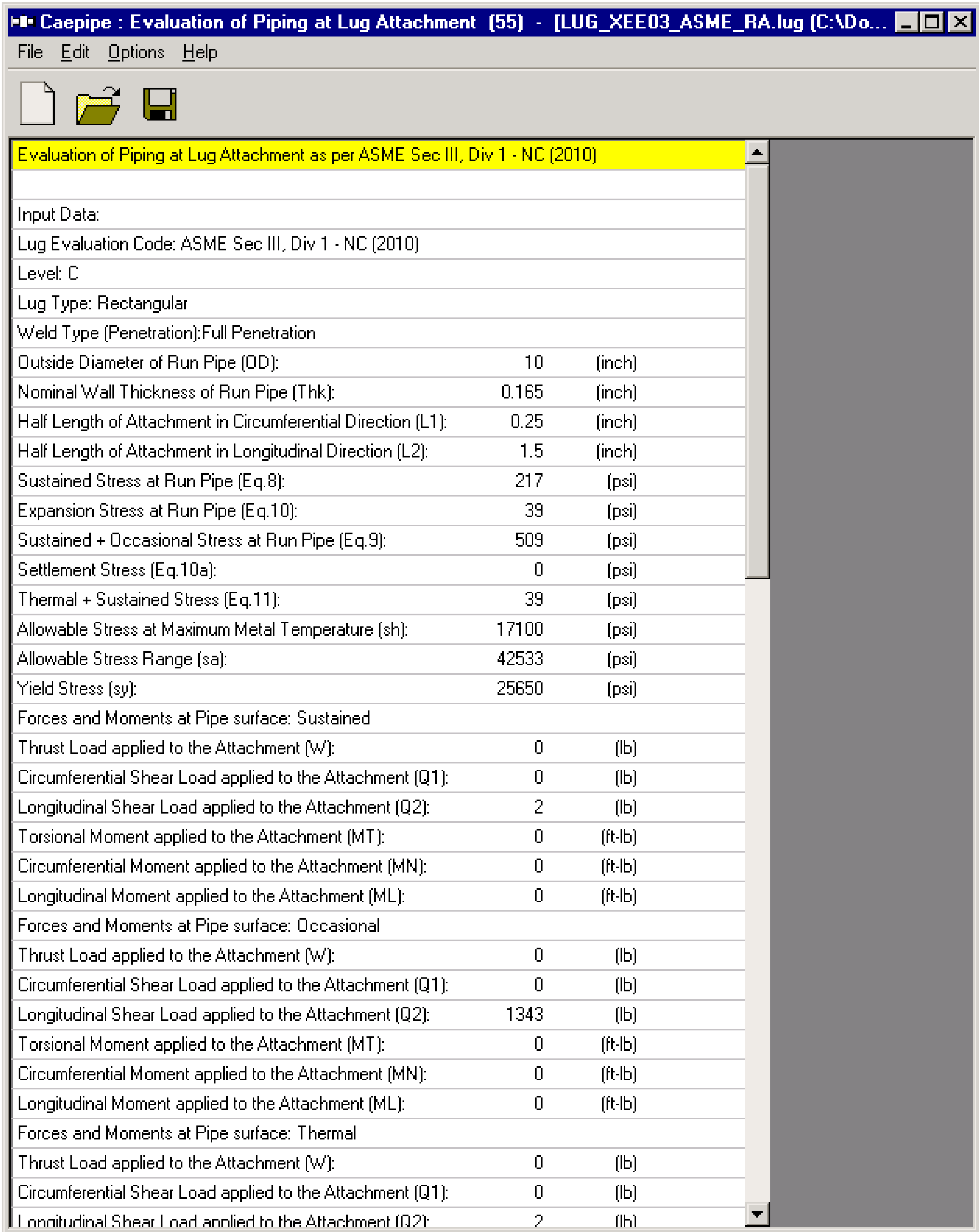
1. ASME Section III, Division 1 (2010) – Appendix Y (NC Piping – Class 2)
2. ASME Section III, Division 1 (2010) – Appendix Y (ND Piping – Class 3)
3. EN 13480-3 (2017), Section 11

The details on the implementation of this module are provided in the Section titled "Lug Evaluation" of the CAEPIPE Code Compliance Manual.

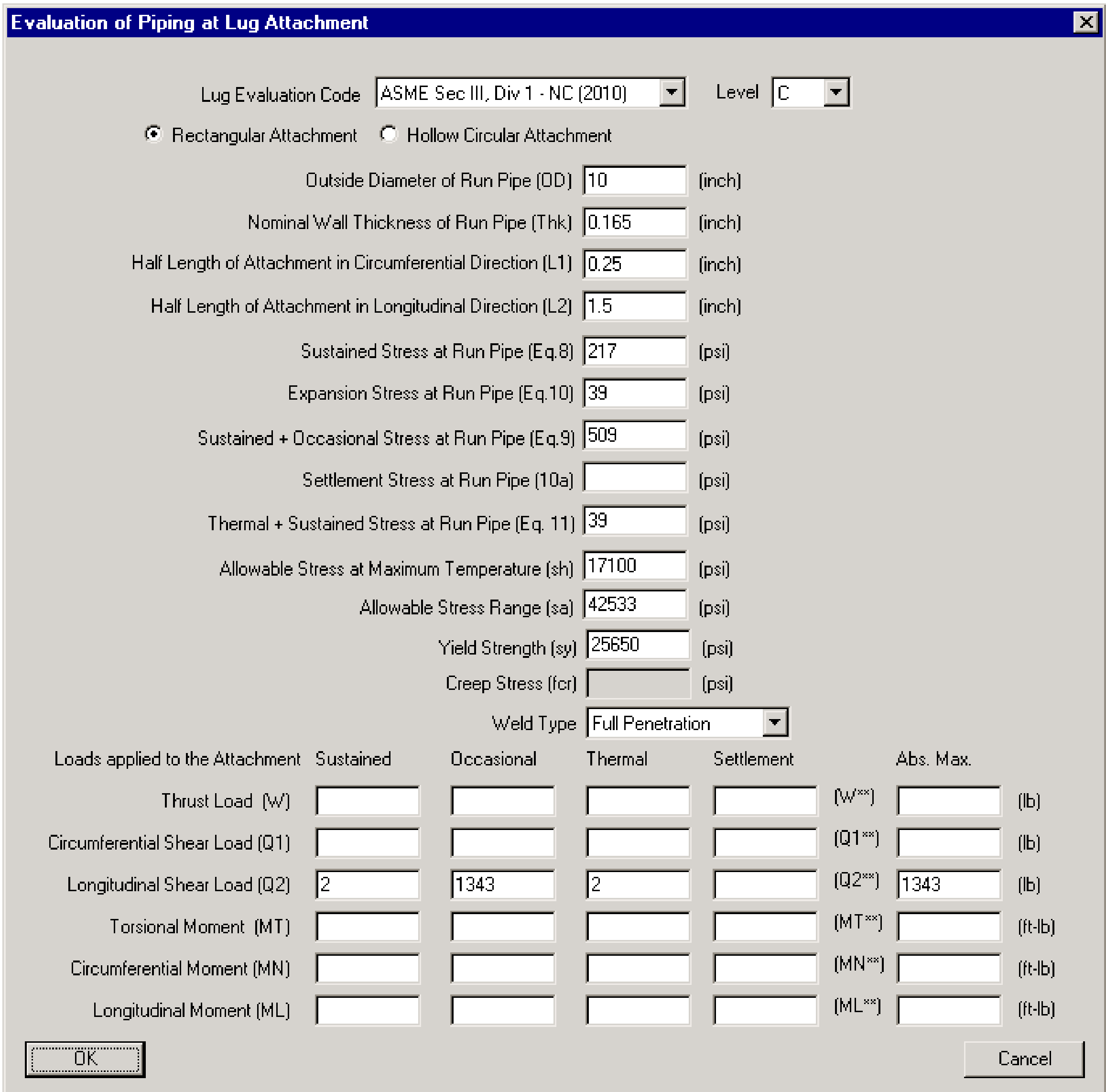
The Lug Evaluation module assumes that the flexibility analysis of that piping system with CAEPIPE has already been performed, which will have produced a stress report as well as the forces and moments at the location where the lug is attached to the piping.

Note that this module is separate from a piping stress model file and can be accessed from File Menu > Open/New command.

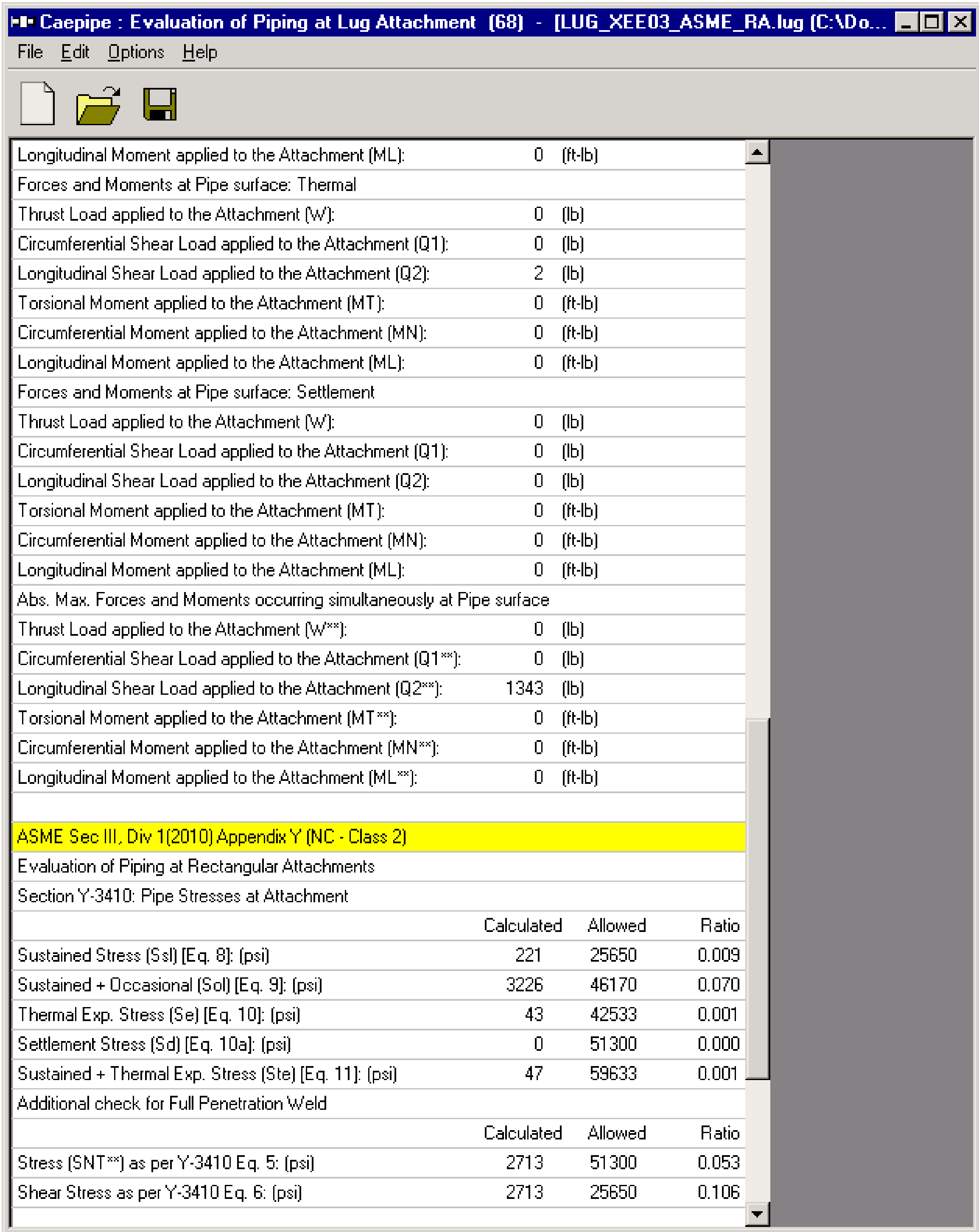




Double-clicking anywhere in the previous screen (or Edit menu > Edit (Ctrl+E)) opens a dialog with input fields you can edit. You will need to enter the data in this dialog. The different parameters required to be input are explained in detail in the Section titled "Lug Evaluation" of the Code Compliance manual.



Once the required data are input, save the file (Lug evaluation will have a .lug extension). Now, select File menu > Analyze to calculate stresses and perform code evaluation, which will be shown *right below* the input information.



Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)
Forces and Moments at Pipe surface: Thermal		
Thrust Load applied to the Attachment (W):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2):	2	(lb)
Torsional Moment applied to the Attachment (MT):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)
Forces and Moments at Pipe surface: Settlement		
Thrust Load applied to the Attachment (W):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2):	0	(lb)
Torsional Moment applied to the Attachment (MT):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)
Abs. Max. Forces and Moments occurring simultaneously at Pipe surface		
Thrust Load applied to the Attachment (W**):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1**):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2**):	1343	(lb)
Torsional Moment applied to the Attachment (MT**):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN**):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML**):	0	(ft-lb)

ASME Sec III, Div 1(2010) Appendix Y (NC - Class 2)

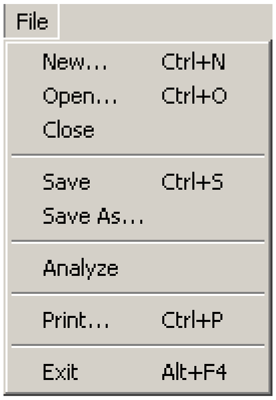
Evaluation of Piping at Rectangular Attachments

Section Y-3410: Pipe Stresses at Attachment

	Calculated	Allowed	Ratio
Sustained Stress (Ssl) (Eq. 8): (psi)	221	25650	0.009
Sustained + Occasional (Sol) (Eq. 9): (psi)	3226	46170	0.070
Thermal Exp. Stress (Se) (Eq. 10): (psi)	43	42533	0.001
Settlement Stress (Sd) (Eq. 10a): (psi)	0	51300	0.000
Sustained + Thermal Exp. Stress (Ste) (Eq. 11): (psi)	47	59633	0.001
Additional check for Full Penetration Weld			
	Calculated	Allowed	Ratio
Stress (SNT**) as per Y-3410 Eq. 5: (psi)	2713	51300	0.053
Shear Stress as per Y-3410 Eq. 6: (psi)	2713	25650	0.106

Lug Evaluation Module Menus

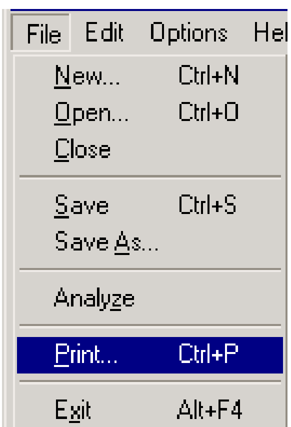
File Menu



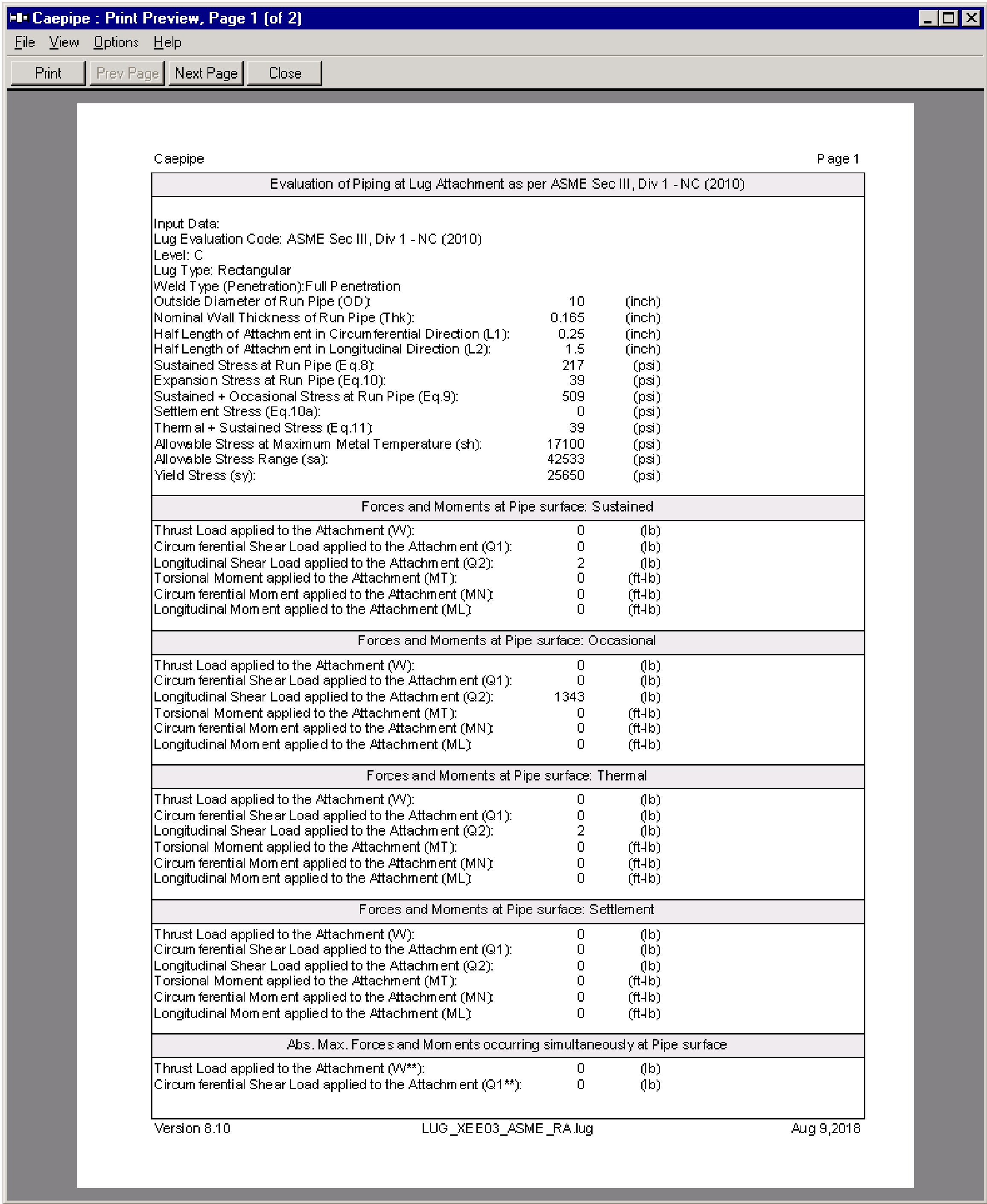
.Analyze.

Analyze command calculates pipestressesat the attachment and compares them to stress allowable specified by the selected code.

Print.



You can print a Report by using the Print command. You can also preview the report by clicking the Preview button on the print dialog.



Caepipe Page 1

Evaluation of Piping at Lug Attachment as per ASME Sec III, Div 1 - NC (2010)

Input Data:
Lug Evaluation Code: ASME Sec III, Div 1 - NC (2010)
Level: C
Lug Type: Rectangular
Weld Type (Penetration):Full Penetration

Outside Diameter of Run Pipe (OD):	10	(inch)
Nominal Wall Thickness of Run Pipe (Thk):	0.165	(inch)
Half Length of Attachment in Circumferential Direction (L1):	0.25	(inch)
Half Length of Attachment in Longitudinal Direction (L2):	1.5	(inch)
Sustained Stress at Run Pipe (Eq.8):	217	(psi)
Expansion Stress at Run Pipe (Eq.10):	39	(psi)
Sustained + Occasional Stress at Run Pipe (Eq.9):	509	(psi)
Settlement Stress (Eq.10a):	0	(psi)
Thermal + Sustained Stress (Eq.11):	39	(psi)
Allowable Stress at Maximum Metal Temperature (sh):	17100	(psi)
Allowable Stress Range (sa):	42533	(psi)
Yield Stress (sy):	25650	(psi)

Forces and Moments at Pipe surface: Sustained

Thrust Load applied to the Attachment (W):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2):	2	(lb)
Torsional Moment applied to the Attachment (MT):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)

Forces and Moments at Pipe surface: Occasional

Thrust Load applied to the Attachment (W):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2):	1343	(lb)
Torsional Moment applied to the Attachment (MT):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)

Forces and Moments at Pipe surface: Thermal

Thrust Load applied to the Attachment (W):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2):	2	(lb)
Torsional Moment applied to the Attachment (MT):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)

Forces and Moments at Pipe surface: Settlement

Thrust Load applied to the Attachment (W):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1):	0	(lb)
Longitudinal Shear Load applied to the Attachment (Q2):	0	(lb)
Torsional Moment applied to the Attachment (MT):	0	(ft-lb)
Circumferential Moment applied to the Attachment (MN):	0	(ft-lb)
Longitudinal Moment applied to the Attachment (ML):	0	(ft-lb)

Abs. Max. Forces and Moments occurring simultaneously at Pipe surface

Thrust Load applied to the Attachment (W**):	0	(lb)
Circumferential Shear Load applied to the Attachment (Q1**):	0	(lb)

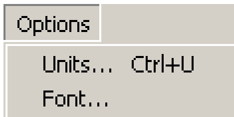
Version 8.10 LUG_XEE03_ASME_RA.lug Aug 9,2018

Edit Menu

.Edit.

You can edit the data by clicking the Edit command.

Options Menu



.Units.

See Units in the Layout Window Options Menu section of the CAEPIPE User's Manual.

.Font.

See Font in the Layout Window Options Menu section of the CAEPIPE User's Manual.

Sample Problem

```
Caepipe                    LUG_XEE03_ASME_RA.lug                  Page 1
Version 8.10                                                      Aug 10,2018
------------------------------------------------------------------------------------
    Evaluation of Piping at Lug Attachment as per ASME Sec III, Div 1 - NC (2010)
------------------------------------------------------------------------------------

Input Data:
Lug Evaluation Code: ASME Sec III, Div 1 - NC (2010)
Level: C
Lug Type: Rectangular
Weld Type (Penetration):Full Penetration
Outside Diameter of Run Pipe (OD):                                  10    (inch)
Nominal Wall Thickness of Run Pipe (Thk):                        0.165    (inch)
Half Length of Attachment in Circumferential Direction (L1):      0.25    (inch)
Half Length of Attachment in Longitudinal Direction (L2):          1.5    (inch)
Sustained Stress at Run Pipe (Eq.8):                               217    (psi)
Expansion Stress at Run Pipe (Eq.10):                               39    (psi)
Sustained + Occasional Stress at Run Pipe (Eq.9):                  509    (psi)
Settlement Stress (Eq.10a):                                          0    (psi)
Thermal + Sustained Stress (Eq.11):                                 39    (psi)
Allowable Stress at Maximum Metal Temperature (sh):              17100    (psi)
Allowable Stress Range (sa):                                     42533    (psi)
Yield Stress (sy):                                               25650    (psi)
------------------------------------------------------------------------------------
                    Forces and Moments at Pipe surface: Sustained
------------------------------------------------------------------------------------
Thrust Load applied to the Attachment (W):                           0    (lb)
Circumferential Shear Load applied to the Attachment (Q1):           0    (lb)
Longitudinal Shear Load applied to the Attachment (Q2):              2    (lb)
Torsional Moment applied to the Attachment (MT):                     0    (ft-lb)
Circumferential Moment applied to the Attachment (MN):               0    (ft-lb)
Longitudinal Moment applied to the Attachment (ML):                  0    (ft-lb)
------------------------------------------------------------------------------------
                   Forces and Moments at Pipe surface: Occasional
------------------------------------------------------------------------------------
Thrust Load applied to the Attachment (W):                           0    (lb)
Circumferential Shear Load applied to the Attachment (Q1):           0    (lb)
Longitudinal Shear Load applied to the Attachment (Q2):           1343    (lb)
Torsional Moment applied to the Attachment (MT):                     0    (ft-lb)
Circumferential Moment applied to the Attachment (MN):               0    (ft-lb)
Longitudinal Moment applied to the Attachment (ML):                  0    (ft-lb)
------------------------------------------------------------------------------------
                     Forces and Moments at Pipe surface: Thermal
------------------------------------------------------------------------------------
Thrust Load applied to the Attachment (W):                           0    (lb)
Circumferential Shear Load applied to the Attachment (Q1):           0    (lb)
Longitudinal Shear Load applied to the Attachment (Q2):              2    (lb)
Torsional Moment applied to the Attachment (MT):                     0    (ft-lb)
Circumferential Moment applied to the Attachment (MN):               0    (ft-lb)
Longitudinal Moment applied to the Attachment (ML):                  0    (ft-lb)
------------------------------------------------------------------------------------
                   Forces and Moments at Pipe surface: Settlement
------------------------------------------------------------------------------------
Thrust Load applied to the Attachment (W):                           0    (lb)
Circumferential Shear Load applied to the Attachment (Q1):           0    (lb)
Longitudinal Shear Load applied to the Attachment (Q2):              0    (lb)
Torsional Moment applied to the Attachment (MT):                     0    (ft-lb)
Circumferential Moment applied to the Attachment (MN):               0    (ft-lb)
Longitudinal Moment applied to the Attachment (ML):                  0    (ft-lb)
```

```
Caepipe                    LUG_XEE03_ASME_RA.lug                   Page 2
Version 8.10                                                       Aug 10,2018
-----------------------------------------------------------------------------------
       Abs. Max. Forces and Moments occurring simultaneously at Pipe surface
-----------------------------------------------------------------------------------
Thrust Load applied to the Attachment (W**):                          0    (lb)
Circumferential Shear Load applied to the Attachment (Q1**):          0    (lb)
Longitudinal Shear Load applied to the Attachment (Q2**):          1343    (lb)
Torsional Moment applied to the Attachment (MT**):                    0    (ft-lb)
Circumferential Moment applied to the Attachment (MN**):              0    (ft-lb)
Longitudinal Moment applied to the Attachment (ML**):                 0    (ft-lb)

-----------------------------------------------------------------------------------
                ASME Sec III, Div 1(2010) Appendix Y (NC - Class 2)
-----------------------------------------------------------------------------------
                  Evaluation of Piping at Rectangular Attachments
-----------------------------------------------------------------------------------
                    Section Y-3410: Pipe Stresses at Attachment
-----------------------------------------------------------------------------------
                                                        Calculated   Allowed   Ratio
                                                             (psi)     (psi)
  Sustained Stress (Ssl) [Eq. 8]: (psi)                        221     25650   0.009
  Sustained + Occasional (Sol) [Eq. 9]: (psi)                 3226     46170   0.070
  Thermal Exp. Stress (Se) [Eq. 10]: (psi)                      43     42533   0.001
  Settlement Stress (Sd) [Eq. 10a]: (psi)                        0     51300   0.000
  Sustained + Thermal Exp. Stress (Ste) [Eq. 11]: (psi)         47     59633   0.001
-----------------------------------------------------------------------------------
                    Additional check for Full Penetration Weld
-----------------------------------------------------------------------------------
                                                        Calculated   Allowed   Ratio
                                                             (psi)     (psi)
  Stress (SNT**) as per Y-3410 Eq. 5: (psi)                   2713     51300   0.053
  Shear Stress as per Y-3410 Eq. 6: (psi)                     2713     25650   0.106
```

For a project, a piping material engineer usually produces different material lists for different process and utility systems based on process design conditions, list of components, fluid type (corrosivity, viscosity), end conditions and temperature, pressure and size ranges.

As a result, a piping system will have its materials list, which will specify the materials a stress analyst will have to define inside CAEPIPE before analysis. CAEPIPE is so flexible that it allows each element (such as a pipe, beam, elbow, valve, jacket, bellows, etc.) to have its own material definition.

Once you define and name a material type, you type in that name on the Layout window under the column "Matl" to specify the material for an element. A material may be metallic or FRP. You need to obtain properties (density, Poisson's ratio, Young's modulus, mean coefficient of thermal expansion and allowable stress as a function of temperature) for a new material not found in the supplied libraries. You will need properties for at least two temperatures (reference and design) for each material you define. Subsequently, the temperatures you specify for an element that uses this material must fall within this temperature range.

The material name you specify on the layout applies to the piping element on that row. For jacketed piping, you must specify two materials - one for the core pipe (on the Layout window), and the other for the jacket pipe in its own dialog. The material you specify for a bend, a jacketed bend, and for a miter bend applies only to that specific element on that input row.

There are two ways in which you can define materials:

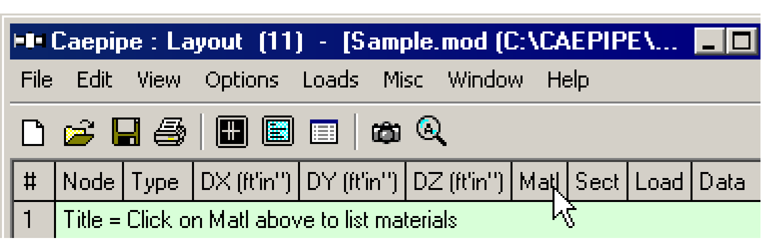
1. By defining a material inside the CAEPIPE model, or
2. By picking a material from an existing material library.

For the sake of convenience, we suggest that you create a separate material library for your pipe stress project so that you can share it with your team members.

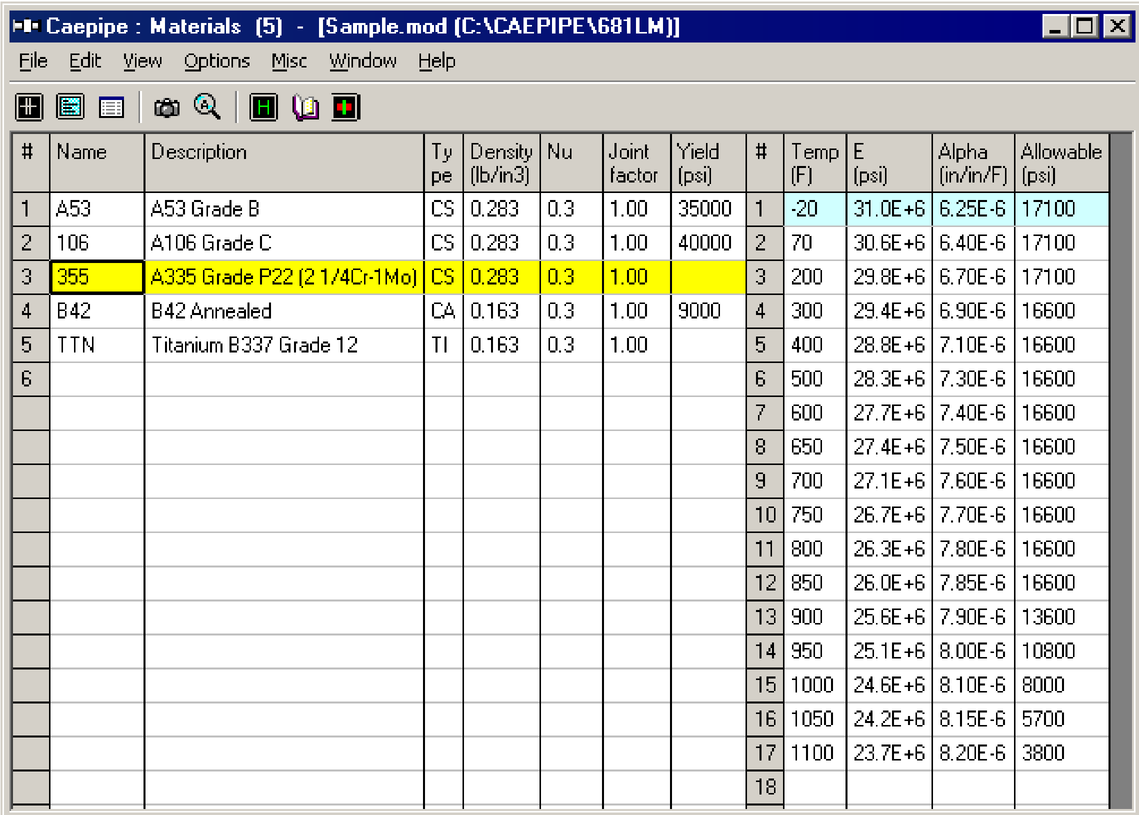
Below, you will see how to create a material inside a model and how to create a new or modify an existing material library.

Define a Material inside a CAEPIPE model

From the Layout window, click on "Matl" on the header row (or select Materials from the Misc menu, Ctrl+Shift+M).



The List window for materials is shown.



#	Name	Description	Type	Density (lb/in3)	Nu	Joint factor	Yield (psi)
1	A53	A53 Grade B	CS	0.283	0.3	1.00	35000
2	106	A106 Grade C	CS	0.283	0.3	1.00	40000
3	355	A335 Grade P22 (2 1/4Cr-1Mo)	CS	0.283	0.3	1.00	
4	B42	B42 Annealed	CA	0.163	0.3	1.00	9000
5	TTN	Titanium B337 Grade 12	TI	0.163	0.3	1.00	
6							

#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	-20	31.0E+6	6.25E-6	17100
2	70	30.6E+6	6.40E-6	17100
3	200	29.8E+6	6.70E-6	17100
4	300	29.4E+6	6.90E-6	16600
5	400	28.8E+6	7.10E-6	16600
6	500	28.3E+6	7.30E-6	16600
7	600	27.7E+6	7.40E-6	16600
8	650	27.4E+6	7.50E-6	16600
9	700	27.1E+6	7.60E-6	16600
10	750	26.7E+6	7.70E-6	16600
11	800	26.3E+6	7.80E-6	16600
12	850	26.0E+6	7.85E-6	16600
13	900	25.6E+6	7.90E-6	13600
14	950	25.1E+6	8.00E-6	10800
15	1000	24.6E+6	8.10E-6	8000
16	1050	24.2E+6	8.15E-6	5700
17	1100	23.7E+6	8.20E-6	3800
18				

In the Material List window, you can edit inside both panes - the left pane contains Name, Description, Type of material, Density, Poisson's ratio (nu), Joint factor etc. and the right pane contains material properties (usually Modulus of Elasticity E, mean Coefficient of Linear thermal expansion [Alpha], not instantaneous coefficient nor thermal expansion per unit length, and the code-specific Allowable stress) as a function of Temperature. CAEPIPE requires "Weight Density" to be input in lbf/in3 or kgf/m3 and NOT its mass density. While entering the temperature-dependent material properties, you need not enter temperatures in an ascending order, although recommended. CAEPIPE will sort the entries later.

After you are done entering properties for one material, be sure to press Enter when the cursor is in the left pane, to move it to the next row so you can start entering the next material. You can insert, copy & paste, delete and edit any material (see under Edit menu).

These panes may change depending on the piping code chosen. For example, for the Swedish and Norwegian codes, the following window is displayed. This window contains additional columns for Tensile strength and instead of one Joint factor, it has Longitudinal and Circumferential joint factors.

Caepipe : Material Library (23) - [EN13480-2012.mat (C:\CAEPIPE\770LM\Material_Library)]

File Edit Options Help

#	Description	Type	Density (kg/m3)	Nu	Long. factor	Circ. factor	Tensile (MPa)
1	EN 1.0345 (P235GH) MAX 60 MM	CS	7850	0.3	1.00	1.00	360.0
2	EN 1.0425 (P265GH) MAX 60 MM	CS	7850	0.3	1.00	1.00	410.0
3	EN 1.0481 (P295GH) MAX 60 MM	CS	7850	0.3	1.00	1.00	460.0
4	EN 1.4000 (X6CR13)	FS	7700	0.3	1.00	1.00	400.0
5	EN 1.4002 (X6CRAL13)	FS	7700	0.3	1.00	1.00	400.0
6	EN 1.4003 (X2CRNI12)	FS	7700	0.3	1.00	1.00	450.0
7	EN 1.4016 (X6CR17)	FS	7700	0.3	1.00	1.00	450.0
8	EN 1.4113 (X6CRMO17-1)	FS	7700	0.3	1.00	1.00	450.0
9	EN 1.4301 (X5XRNI18-10)	AS	7900	0.3	1.00	1.00	520.0
10	EN 1.4306 (X2CRNI19-11)	AS	7900	0.3	1.00	1.00	520.0
11	EN 1.4307 (X2CRNI18-9)	AS	7900	0.3	1.00	1.00	520.0
12	EN 1.4311 (X2CRNIN18-10)	AS	7900	0.3	1.00	1.00	550.0

#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	Allowable (MPa)
1	20	212000	11.90E-6	138.7
2	50	209500	12.20E-6	138.7
3	100	207000	12.50E-6	130.7
4	150	203000	12.80E-6	120.7
5	200	199000	13.00E-6	111.3
6	250	195500	13.30E-6	102.0
7	300	192000	13.60E-6	93.30
8	350	188000	13.90E-6	86.70
9	380	185600	14.00E-6	83.50
10	390	184800	14.10E-6	82.40
11	400	184000	14.10E-6	81.30
12	410	183100	14.10E-6	80.80

The European (EN 13480) code has a column for Tensile strength and temperature-dependent properties have an additional column for fCR (allowable creep stress).

Caepipe : Material Library (23) - [EN13480-2012.mat (C:\CAEPIPE\770LM\Material_Library)]

File Edit Options Help

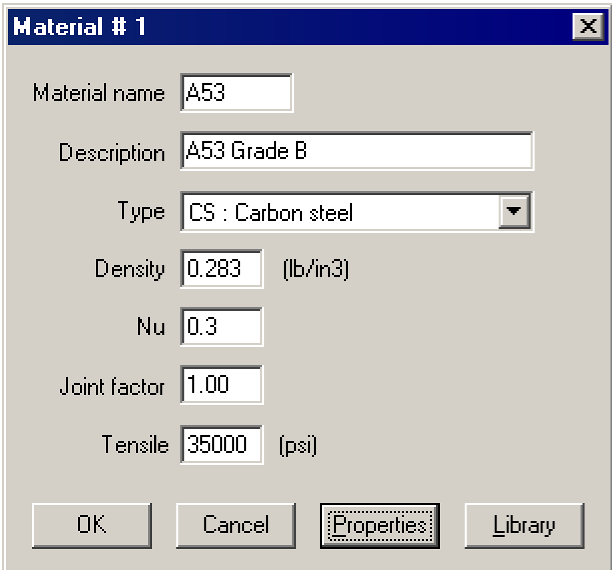
#	Description	Type	Density (kg/m3)	Nu	Joint factor	Tensile (MPa)
1	EN 1.0345 (P235GH) MAX 60 MM	CS	7850	0.3	1.00	360.0
2	EN 1.0425 (P265GH) MAX 60 MM	CS	7850	0.3	1.00	410.0
3	EN 1.0481 (P295GH) MAX 60 MM	CS	7850	0.3	1.00	460.0
4	EN 1.4000 (X6CR13)	FS	7700	0.3	1.00	400.0
5	EN 1.4002 (X6CRAL13)	FS	7700	0.3	1.00	400.0
6	EN 1.4003 (X2CRNI12)	FS	7700	0.3	1.00	450.0
7	EN 1.4016 (X6CR17)	FS	7700	0.3	1.00	450.0
8	EN 1.4113 (X6CRMO17-1)	FS	7700	0.3	1.00	450.0
9	EN 1.4301 (X5XRNI18-10)	AS	7900	0.3	1.00	520.0
10	EN 1.4306 (X2CRNI19-11)	AS	7900	0.3	1.00	520.0
11	EN 1.4307 (X2CRNI18-9)	AS	7900	0.3	1.00	520.0
12	EN 1.4311 (X2CRNIN18-10)	AS	7900	0.3	1.00	550.0

#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	f (MPa)	fCR (MPa)
1	20	212000	11.90E-6	138.7	
2	50	209500	12.20E-6	138.7	
3	100	207000	12.50E-6	130.7	
4	150	203000	12.80E-6	120.7	
5	200	199000	13.00E-6	111.3	
6	250	195500	13.30E-6	102.0	
7	300	192000	13.60E-6	93.30	
8	350	188000	13.90E-6	86.70	
9	380	185600	14.00E-6	83.50	116.0
10	390	184800	14.10E-6	82.40	103.2
11	400	184000	14.10E-6	81.30	92.00
12	410	183100	14.10E-6	80.80	80.80

To Input a New Material

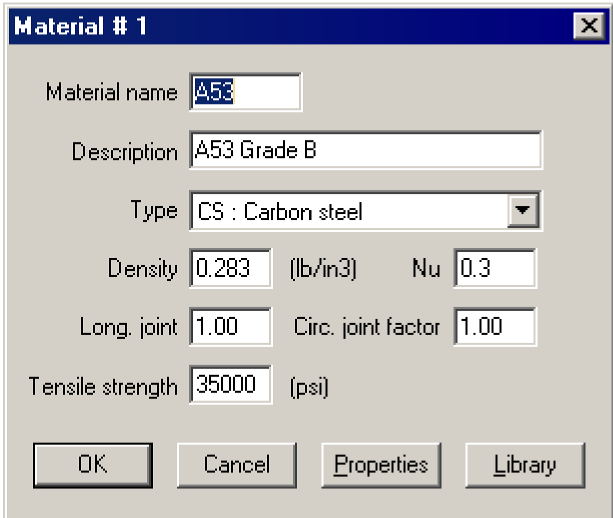
You can input a new material in three ways:

1. Start typing directly into the fields in the Materials List window.
2. Input through dialogs (shown below for some but not all piping codes).

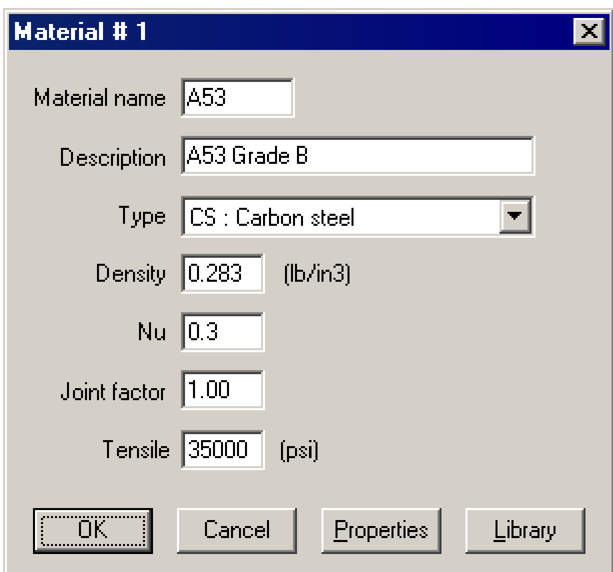


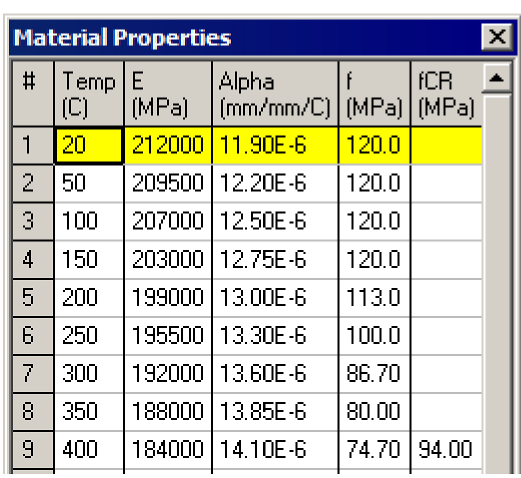
#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	-325	31.4E+6	5.00E-6	20000
2	-200	30.8E+6	5.35E-6	20000
3	-100	30.2E+6	5.65E-6	20000
4	70	29.5E+6	6.07E-6	20000
5	200	28.8E+6	6.38E-6	20000
6	300	28.3E+6	6.60E-6	20000
7	400	27.7E+6	6.82E-6	19900
8	500	27.3E+6	7.02E-6	19000
9	600	26.7E+6	7.23E-6	17900
10	650	26.1E+6	7.33E-6	17300
11	700	25.5E+6	7.44E-6	16700

For the Swedish and Norwegian piping codes, the Material dialog has Longitudinal and Circumferential joint factors and a Tensile strength field.



For the European (EN 13480) code, the Material dialog has a single Joint factor and a Tensile strength field, while the properties window has fCR (creep stress allowable).

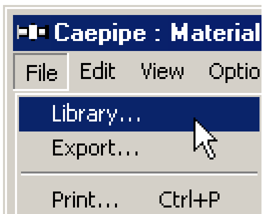



Material Properties

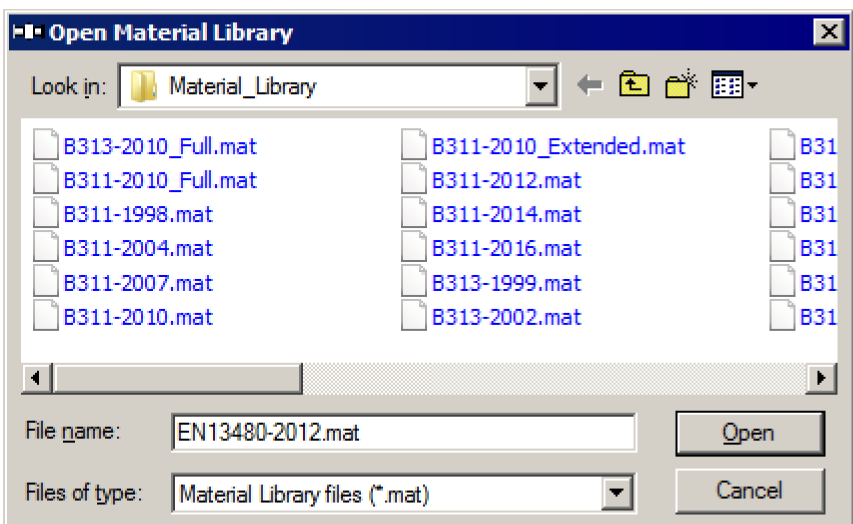
#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	f (MPa)	fCR (MPa)
1	20	212000	11.90E-6	120.0	
2	50	209500	12.20E-6	120.0	
3	100	207000	12.50E-6	120.0	
4	150	203000	12.75E-6	120.0	
5	200	199000	13.00E-6	113.0	
6	250	195500	13.30E-6	100.0	
7	300	192000	13.60E-6	86.70	
8	350	188000	13.85E-6	80.00	
9	400	184000	14.10E-6	74.70	94.00

3. Pick a material from an existing material library (supplied with CAEPIPE or your own). Click on the Library button on the toolbar to open the library:

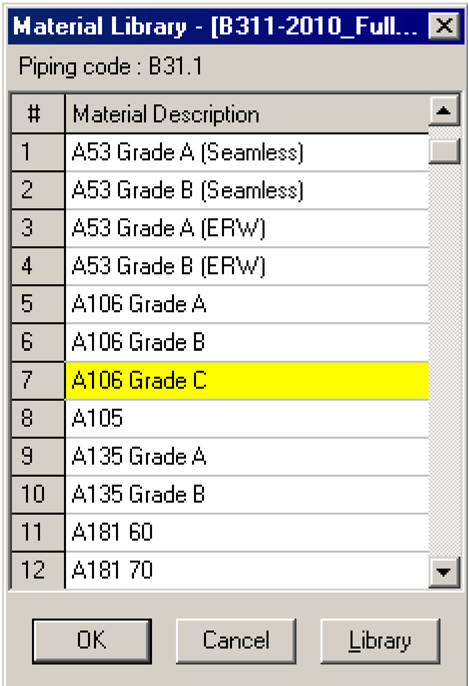
(or select Library command under the File menu):



You will have to open a library file first if it was not previously opened. Select the one of interest. Note that the libraries with filenames starting with B311 and B313 with year later than 2010 (e.g., B313-2012.mat) have approx. 200 materials each (from the respective piping code).



You can select a material from the opened library by double clicking on it or highlighting it and clicking on OK.



Name

Type a Material name, up to five(5) alpha-numeric characters.

Description

Type a description for the material, up to 31 characters.

Type

AL for Aluminum
AS for Austenitic Stainless Steel
CA for Copper alloys annealed
CC for Copper alloys cold worked
CS for Carbon Steel
FR for Fiber Reinforced Plastic piping
FS for Ferritic steel
NA for Nickel alloys 800, 800H, 825
SS for Stainless Steel
TI for Titanium

These material types are used in calculation of the Y factor for allowable pressure at high temperatures for certain piping codes. Swedish and Norwegian piping codes also use it for calculating allowable expansion stress range. These codes also need tensile strength.

For Fiber Reinforced Plastic piping, you need to select the material type “FR” to enter FRP material properties. More information can be found under the section Fiber Reinforced Plastic Piping in this manual.

Density

Density of the material is used to calculate weight load and also mass for dynamic analysis. CAEPIPE requires "Weight Density" to be input in lbf/in3 or lbf/ft3 or kgf/m3 or gf/cm3 and NOT its mass density. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

Nu

The Poisson's ratio (Nu) defaults to 0.3 if not input.

Joint factor

The joint factor is the longitudinal weld joint factor used to calculate allowable pressure. For Swedish and Norwegian piping codes, a circumferential joint factor is also input to calculate longitudinal pressure stress.

Tensile strength / Yield

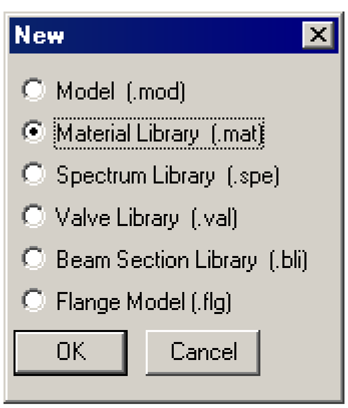
For Swedish, Norwegian and European (EN13480) piping codes, tensile strength is used in the calculation of the allowable expansion stress range. For Stoomwezen piping code, tensile strength is used in the calculation of hot allowable stress.

For B31.3 or B31.12 piping codes, yield is used in the calculation of sustained plus occasional allowable stress when the temperature is > 480^0 C or 800^0 F.

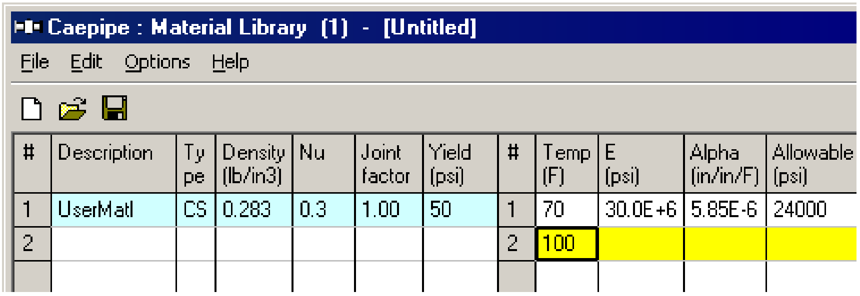
Note: Leaving the yield field blank will cause CAEPIPE to issue Assertion failure during analysis for stress system having the temperature of piping >480^0 C or 800^0 F.

To create or modify a material library

CAEPIPE offers you flexibility in creating your own material libraries (user-defined libraries). That way, you do not feel restricted by the offered choices in materials and can continually keep updating the material libraries with your own materials. To create a library: From the Main window, select File > New and click on Material Library.



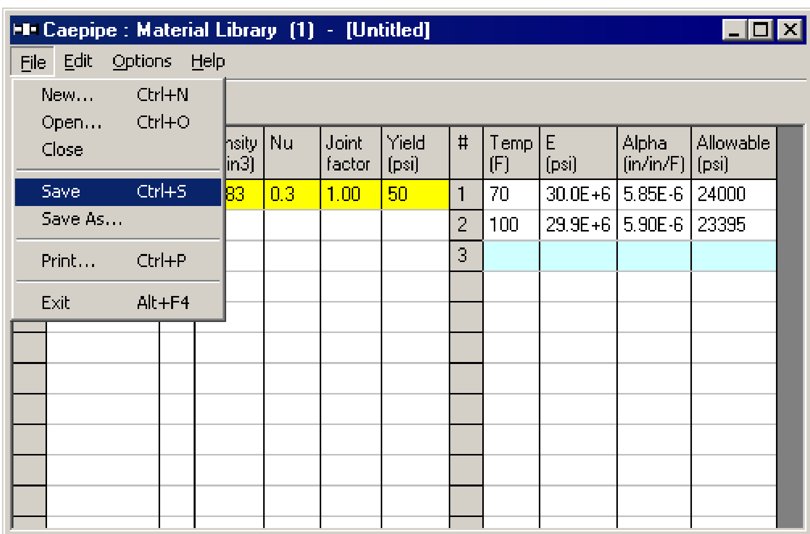
A List window for materials is shown.



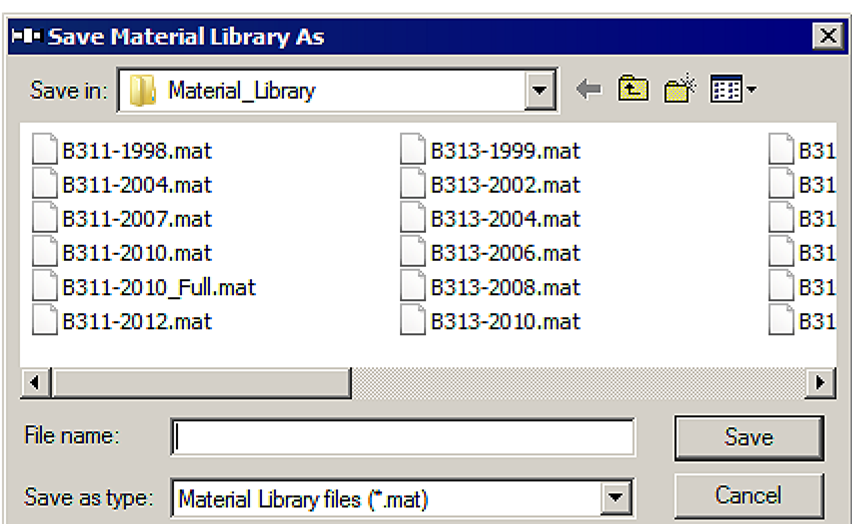
You must select a piping code first, using the menu command Options > Piping code, before you start entering properties.

You can, as before, start typing directly into the fields, or enter properties through a dialog. The only difference is that materials in the library do not have names whereas those in a model have names.

After you are done entering materials, you must save to a material library file by using the File > Save command.



Give the file a suitable name. The file will be saved with a .mat extension.



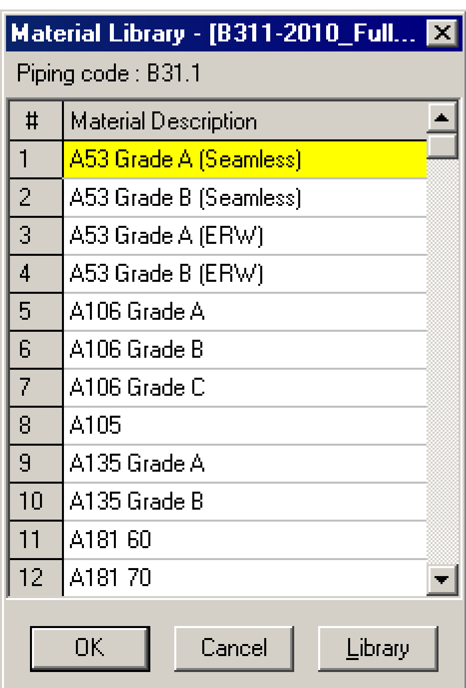
Note: It is also possible to Import/Export the material library through a Ascii Material Batch Library (.mlb) file. See the section titled Import/Export Material Library in Appendix A of CAEPIPE User's Manual for more details.

Should you need to change the piping code, then you will need to update all materials' properties (in this library) according to the new code, or load a new file. Better yet, if you do not see the library among the supplied files, create a new library for the new code.

Frequently, this issue confuses users and they end up using material properties that come from one code under another code (Example: A53 Grade B, common to B31.1 and B31.3, is used by mistake under the wrong code. Note that this material has different allowable stresses under the two codes!).

Therefore, first make sure that you have selected the correct piping code (under Options > Analysis > Piping code) in the Layout window. Then, ensure that you use the correct properties from that code.

CAEPIPE comes with many libraries from several piping codes. When inside a CAEPIPE model, you can open any library and pick a material from it. Make sure to pick the proper library (with the proper year), especially between B31.1 and B31.3 libraries because they have significantly different allowable stresses for the same materials. Also verify the material properties in these libraries before you use them.



In summary, CAEPIPE is supplied with many built-in material libraries applicable for different piping codes. Users should ensure that the correct piping code and the corresponding material library are selected for each CAEPIPE model created and the built-in material properties have been checked for correctness prior to their use.

Missing Mass Correction

In dynamic analysis using modal superposition, usually an approximate solution is obtained because only a limited number of modes is considered. (For seismic analysis, typically all modes up to 33 Hz). The errors in pipe displacements and stresses are usually small because they are affected relatively little by high modes. The error in support loads may be substantial because the influence of higher modes on support loads can be important. In stiff piping systems with few low frequency modes, stresses may also be affected significantly.

Using limited number of modes results in some mass of the system being ignored. The distribution of this "missing mass" is such that the inertia forces associated with it will usually produce small displacements and stresses. However these forces will often produce significant support loads, and in stiff systems can produce significant stresses.

A correction can be made by determining the modal contributions to the mass of the system and obtaining the "missing mass" as the difference between these contributions and the actual mass.

The inertial force vector for the nth mode is given by

$$\{F_n\} = -[M]\{\ddot{u}_n\} = \omega_n^2 [M]\{\emptyset_n\} A_n \tag{1}$$

Where $[M]$ = diagonal mass matrix
$\{\ddot{u}_n\}$ = acceleration vector
ω_n = circular frequency
$\{\emptyset_n\}$ = mass normalized eigenvector
A_n = modal displacement for mode n

For X seismic excitation

$$A_n = \{\emptyset_n\}^T [M]\{r_x\} \frac{S_{nx}^a}{\omega_n^2} = \Gamma_{nx} \frac{S_{nx}^a}{\omega_n^2} \tag{2}$$

Where $\{r_x\}$ = displacement vector due to a unit displacement in the X direction
S_{nx}^a = spectral acceleration for the nth mode for excitation in the X direction
Γ_{nx} = mass participation factor in the X direction for mode n

Let m = number of modes used in analysis
N = total number of modes

If it is assumed that the higher modes: m+1 through N are in phase and have a common spectral accelerationS_{ox}^a (conservatively taken as the maximum spectral acceleration after the mth mode), the total inertial force contribution of these higher modes (also known as "Rigid body force" or "Left out force") is

$$\{F_x^R\} = S_{ox}^a [M] \sum_{n=m+1}^{N} \{\phi_n\}\, \Gamma_{nx} \tag{3}$$

It can be shown that

$$\{r_x\} = \sum_{n=1}^{N} \{\phi_n\}\,\Gamma_{nx} = \sum_{n=1}^{m} \{\phi_n\}\,\Gamma_{nx} + \sum_{n=m+1}^{N} \{\phi_n\}\,\Gamma_{nx} \tag{4}$$

Substituting from (4) for the summation in (3)

$$\{F_x^R\} = S_{ox}^a\,[M]\left[\{r_x\} - \sum_{n=1}^{m} \{\phi_n\}\,\Gamma_{nx}\right] \tag{5}$$

Note that there will be missing mass inertia forces in the Y and Z directions, in addition to the X direction, for X excitation.

The missing mass force vectors for the Y and Z directions are similarly calculated. The response to each of these three force vectors is calculated and these additional response vectors are combined with the responses of the first "m" modes.

This feature is currently not available for Time History, Multi-Level Response Spectrum and Harmonic analyses.

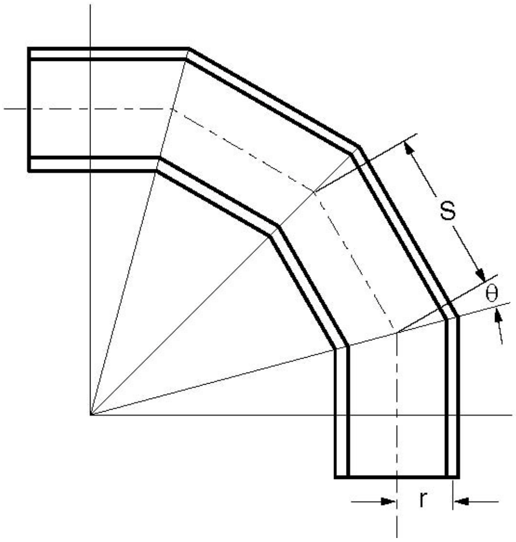
The above described method is based on the technical paper by Powell. See below for details.

Powell, G.H. "Missing Mass Correction in Modal Analysis of Piping Systems." Transactions of the 5th International Conference on Structural Mechanics in Reactor Technology. August 1979: Berlin, Germany.

Miter Bend

Miter bends are typically used when space limitations do not allow the use of regular bends (elbows), or when a miter is more economical to use than a regular bend. Miters are not fittings. They are fabricated from pipe, to requirements. "The use of miters to make changes in direction is practically restricted to low-pressure lines, 10-inch and larger if the pressure drop is unimportant..." (Sherwood 1980).

See figure below for Miter bend parameters.

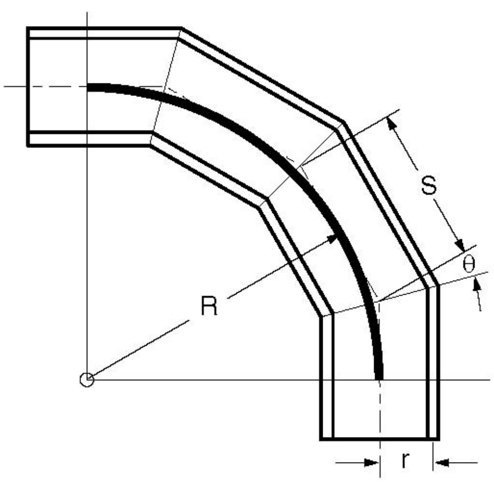


In this figure, r = mean radius of pipe
S = miter spacing at center line
θ = one-half angle between adjacent miter axes ($\leq 22.5°$)

Before modeling a miter bend, you should determine whether it is closely or widely spaced.

Closely Spaced Miter

A miter bend is closely spaced when $S < r(1 + \tan\theta)$.

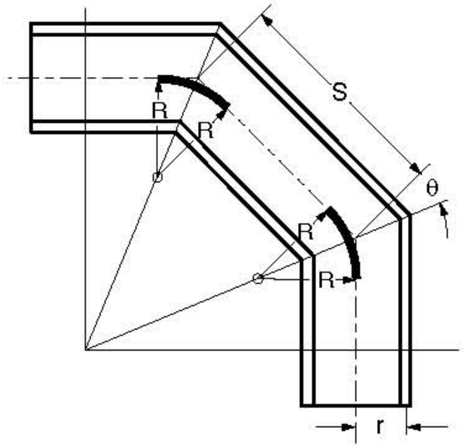


A closely spaced miter bend is input as a single miter bend element.

The Bend Radius (R) is calculated as: $R = 0.5\ S \cot\theta$.

Widely Spaced Miter

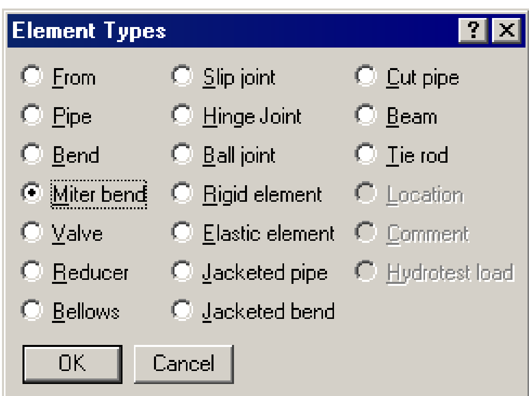
A miter bend is widely spaced when $S \geq r(1 + \tan\theta)$.



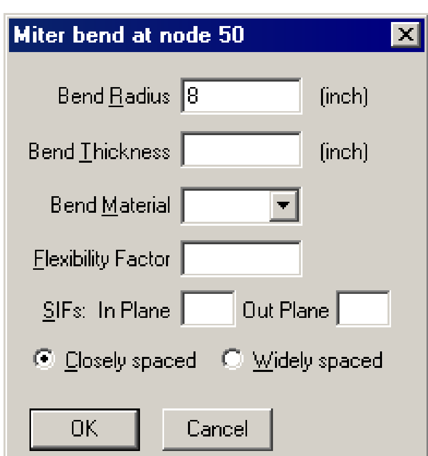
A widely spaced miter bend is modeled with as many miter bend elements as there are miter cuts.

The Bend Radius (R) is calculated as: $R = 0.5\, r\, (1 + \cot\theta)$.

A miter bend is input by typing "m" in the Type column or selecting "Miter bend" from the Element types dialog.



The Miter bend dialog is shown.



Bend Radius

The bend radius (R) depends on the type of miter (Closely or Widely spaced). It is calculated as explained previously and input in this field.

Bend Thickness

Input the wall thickness of the miter bend if it is different from that of the adjoining pipe thicknesses. The Bend Thickness, if specified, applies only to the curved portion(s) of the equivalent bend(s) of the miter bend.

Bend Material

If the material of the miter bend is different from that of the adjoining pipe, select the Bend Material from the drop down combo box. The Bend Material, if specified, applies only to the curved portion(s) of the equivalent bend(s) of the miter bend.

Flexibility Factor

Specify a flexibility factor for the miter bend if different from the piping code's factor. If you specify one, CAEPIPE uses it instead of the piping code specified Flexibility Factor. A value of 2.0, for e.g., will mean that the miter bend is twice as flexible as a pipe of the same length.

Closely spaced

To specify the miter bend as closely spaced, click on the "Closely spaced" radio button.

Widely spaced

To specify the miter bend as widely spaced, click on the "Widely spaced" radio button.

Parameters for 90° Miter Bends

The parameters (dimensions) for 90° miter bends (with 2, 3, 4 miter cuts) in terms of dimension (A), mean pipe radius (r) and number of miter cuts (N) are shown in the following table.

Use the table to determine whether the miter bend is Closely spaced or Widely spaced, and if it is Widely spaced and has 2, 3 or 4 miter cuts, to calculate the dimensions B, C, D, E and R (equivalent miter bend radius) before modeling it.

A closely spaced miter requires only the miter bend radius (same as dimension A shown in the figures).

			Dimensions for Widely spaced miters only				
N	S	Closely spaced if	B	C	D	E	R
2	0.828427 A	A < 1.707107 r	0.414214 A	0.585786 A	-	-	1.707107 r
3	0.535898 A	A < 2.366025 r	0.267949 A	0.464102 A	-	-	2.366025 r
4	0.397825 A	A < 3.013670 r	0.198912 A	0.367542 A	0.281305 A	0.152241 A	3.013670 r

N = Number of miter cuts
Half angle $\theta = 90° / (2N)$

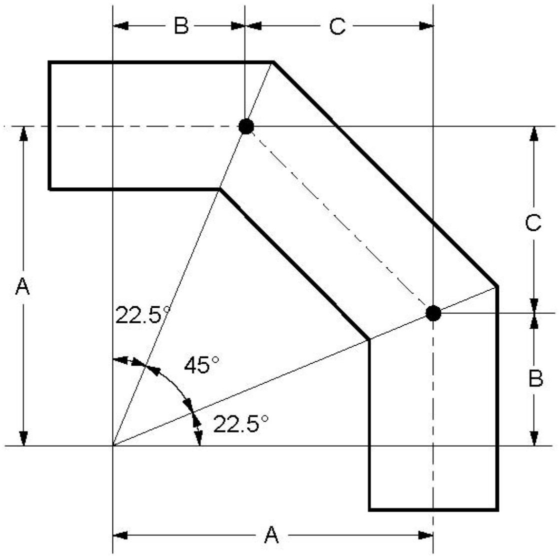
The following pages show the details of how dimensions B, C, D and E were calculated, and are provided only for your information. The above table is important for your modeling requirements. For miters with more than 4 cuts, you have to calculate the required dimensions similar to those shown on the following pages. The next table outlines the modeling procedure for either miter type.

Miter Modeling Procedure

<table>
<tr><td colspan="4">Determine Miter Type</td></tr>
<tr><td>Closely Spaced Miter</td><td colspan="3">Widely Spaced Miter</td></tr>
<tr><td>Any number of cuts</td><td>2 cuts</td><td>3 cuts</td><td>4 cuts</td></tr>
<tr><td rowspan="2">Model miter as a single Closely spaced miter with Bend radius as dimension A.

See Example 1, Closely spaced miter.</td><td colspan="2">Calculate R, B and C from above table. Calculate offsets of nodes using B and C. See Example 2 Widely spaced miter.</td><td>Calculate R, B, C, D and E from above table. Calculate offsets of nodes using B, C, D and E.</td></tr>
<tr><td>Model this miter as 2 widely spaced miters with R as the bend radius.</td><td>Model this miter as 3 widely spaced miters with R as the bend radius.</td><td>Model this miter as 4 widely spaced miters with R as the bend radius.</td></tr>
</table>

Two Miter Cuts (N=2)

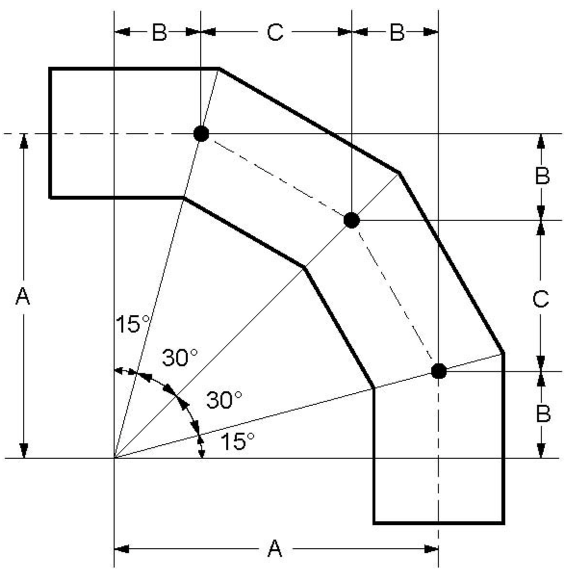


$\theta = 90° / (2 \times 2) = 22.5°$
$S = 2 A \tan\theta = 0.828427 A$
$B = A \tan\theta = 0.414214 A$
$C = 2 A \tan\theta \cos 2\theta = 0.585786 A$

Closely Spaced
The miter is Closely spaced if,
$S < r (1 + \tan\theta)$
Substituting for S,
$2A \tan\theta < r (1 + \tan\theta)$
$A < 1.707107 r$
Bend radius, $R = 0.5 S \cot\theta = A$

Widely Spaced
$R = 0.5 r (1 + \cot\theta) = 1.707107 r$

Three Miter Cuts (N=3)

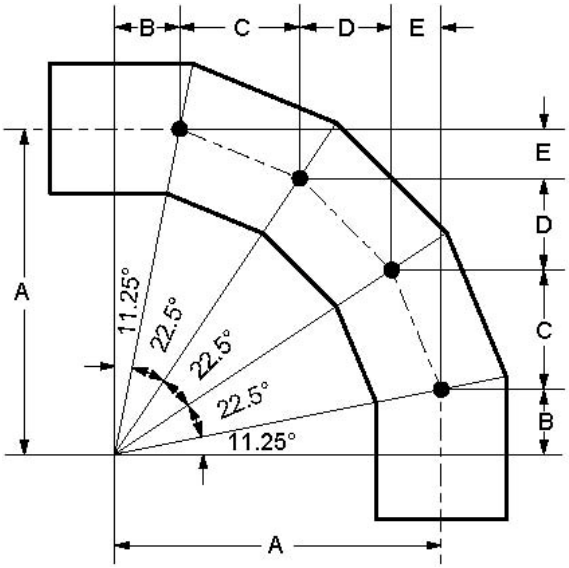


$\theta = 90° / (2 \times 3) = 15°$
$S = 2 A \tan\theta = 0.535898 A$
$B = A \tan\theta = 0.267949 A$
$C = 2 A \tan\theta \cos 2\theta = 0.464102 A$

Closely Spaced
The miter is Closely spaced if,
$S < r (1 + \tan\theta)$
Substituting for S,
$2A \tan\theta < r (1 + \tan\theta)$
$A < 2.366025 r$
Bend radius, $R = 0.5 S \cot\theta = A$

Widely Spaced
$R = 0.5 r (1 + \cot\theta) = 2.366025 r$

Four Miter Cuts (N=4)



$\theta = 90° / (2 \times 4) = 11.25°$
$S = 2 A \tan\theta = 0.397825 A$
$B = A \tan\theta = 0.198912 A$
$C = 2 A \tan\theta \cos 2\theta = 0.367542 A$
$D = 2 A \tan\theta \sin 4\theta = 0.281305 A$
$E = 2 A \tan\theta \sin 2\theta = 0.152241 A$

Closely Spaced
The miter is Closely spaced if,
$S < r (1 + \tan\theta)$
Substituting for S,
$2A \tan\theta < r (1 + \tan\theta)$
$A < 3.013670 r$
Bend radius, $R = 0.5 S \cot\theta = A$

Widely Spaced
$R = 0.5 r (1 + \cot\theta) = 3.013670 r$

Example 1: Closely Spaced Miter

Example Data

Pipe OD = 8.625”, thickness, t = 0.322”
Mean pipe radius, r = (8.625 - 0.322) / 2 = 4.1515”,
Number of miter cuts = 3,
Dimension A = 8”, See 3-cut miter figure.

Look up table (Miter modeling procedure), for an outline of the modeling procedure.

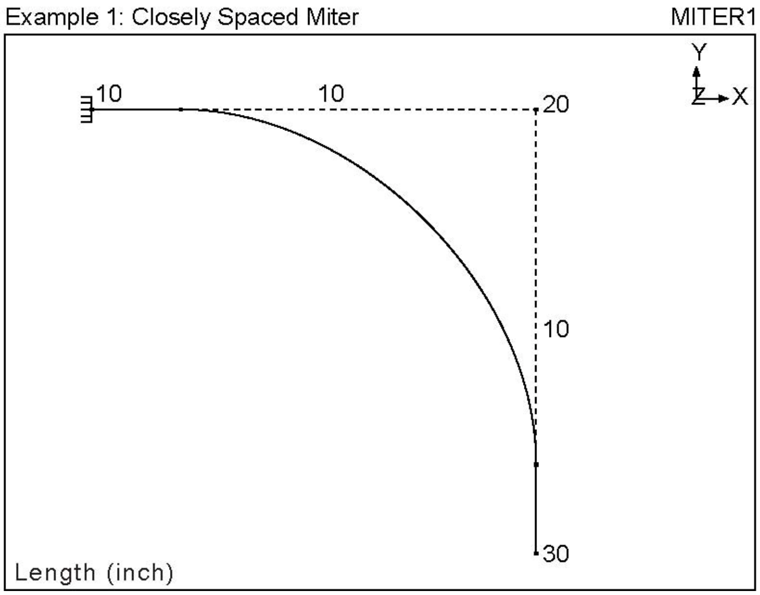
First, determine the type of miter (Closely or Widely spaced) before modeling it.

Look up Summary of Miter parameters, for N = 3. A miter is Closely spaced if A < 2.366025r. This condition is true for r = 4.1515”. Hence, this is a Closely spaced miter.

Steps for Example 1

- Create From node: When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- Construct miter bend: type 20 for Node (simply pressing Tab puts this node number automatically for you), type “m” under the Type column, type 8” for bend radius, select Closely spaced, click on Ok. Type 10” for DX, enter material, section and load names, press Enter.
- Finish the miter bend: type 30 for Node and -10” for DY, press Enter.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Example 1: Closely spaced Miter								
2	10	From							Anchor
3	20	Miter	10			1	8	1	
4	30			-10		1	8	1	

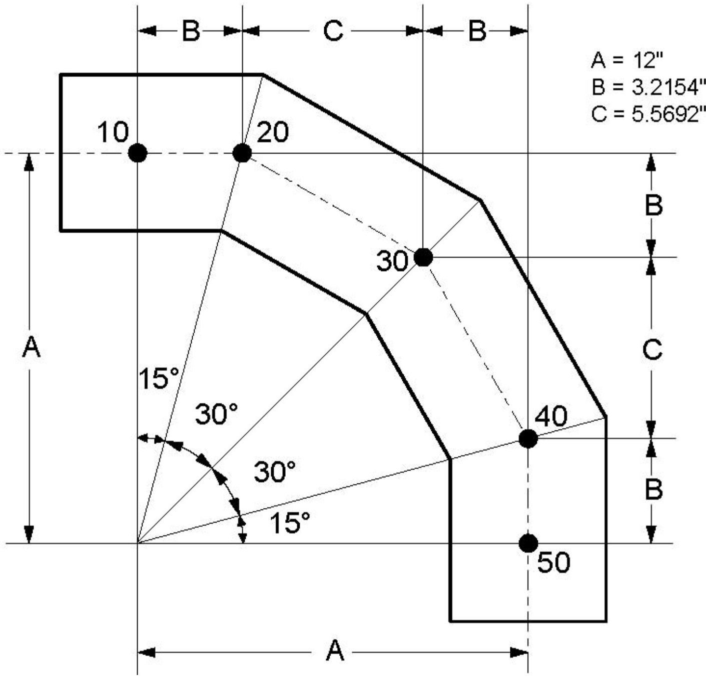


Example 2: Widely Spaced Miter

Let us assume the same data as in Example 1 (Closely spaced miter) with only one change, namely, dimension A (see next figure).

Pipe OD = 8.625", thickness, t = 0.322"
Mean pipe radius, r = (8.625 - 0.322) / 2 = 4.1515",
Number of miter cuts = 3,
Dimension A = 12".

Look up table (for Miter modeling procedure) for an outline of the modeling procedure.



It is essential to determine the type of miter (Closely or Widely spaced) before modeling it.

Determine miter type

Look up table, Summary of Miter parameters, for N = 3. A miter is Closely spaced if A < 2.366025r. This condition is false for r = 4.1515”. Hence, this is a Widely spaced miter. This miter bend has to be modeled as a series of 3 miters. Next, observe in this table that dimensions B, C and R (Equivalent miter bend radius) can be calculated for N = 3 cuts.

Calculate required dimensions

With r = 4.1515”,
Equivalent miter bend radius, R = 2.366025 × r = 9.8225”,
Dimension B = 0.267949 × A = 3.2154”,
Dimension C = 0.464102 × A = 5.5692”

See previous figure. After calculating B, C and R, let us now calculate the offsets of nodes 20, 30 and 40 (the 3 nodes correspond with the 3 miter cuts).

Calculate Offsets

Offsets of node 20 from 10: (First miter cut)
DX = B = 3.2154”
DY = 0” (because node 20 is on the horizontal axis).

Offsets of node 30 from 20: (Second miter cut)
DX = C = 5.5692”
DY = -B = -3.2154”

Offsets of node 40 from 30: (Third miter cut)
DX = B = 3.2154”
DY = -C = -5.5692”

Offsets of node 50 from 40:
DX = 0” (because node 50 is on the vertical axis).
DY = -B = -3.2154”

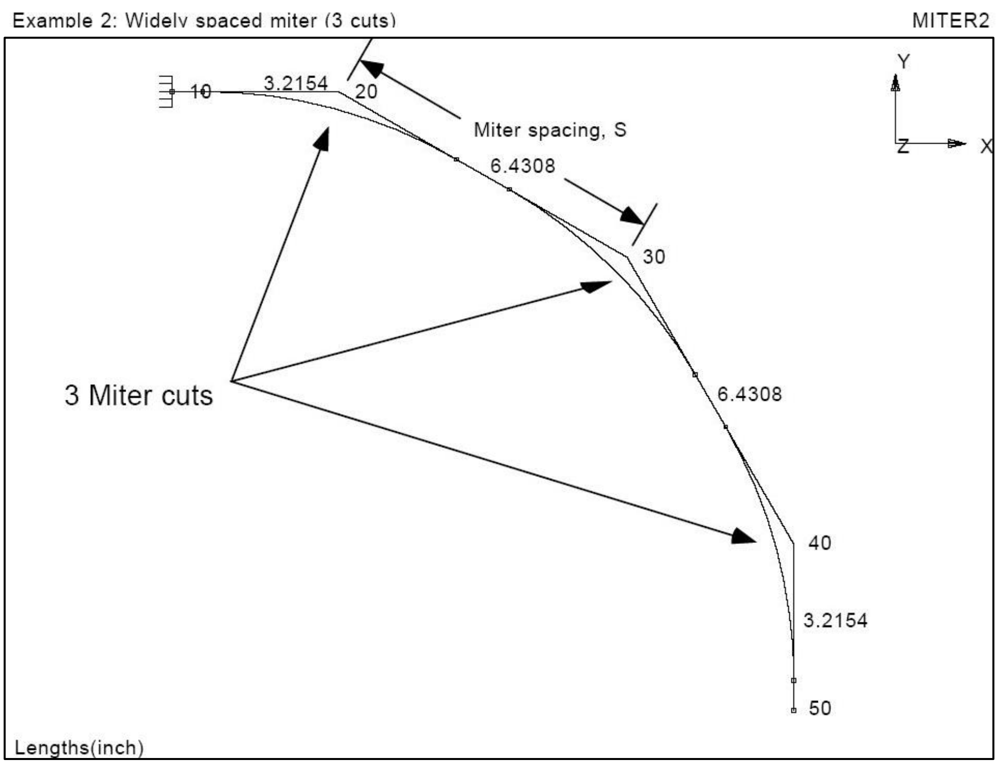
Now, start to build the model in CAEPIPE as shown in Example 1 but with different data (bend radius = 9.8225”, select Widely Spaced miter, and offsets as shown above).

Steps for Example 2

- Create From node: When you start a new model file, node 10 and an Anchor are automatically input, press Enter to move cursor to next empty row.
- Construct first miter bend: type 20 for Node (simply pressing Tab puts this node number automatically for you), type “m” under the Type column, type 9.8225” for bend radius, select Widely spaced, click on Ok. Type 3.2154” for DX, enter material, section and load names, press Enter.
- Construct second miter bend: type 30 for Node (simply pressing Tab puts this node number automatically for you), type “m” under the Type column, type 9.8225” for bend radius, select Widely spaced, click on Ok. Type 5.5692” for DX, -3.2154” for DY, press Enter.

- Construct third miter bend: type 40 for Node (simply pressing Tab puts this node number automatically for you), type "m" under the Type column, type 9.8225" for bend radius, select Widely spaced, click on Ok. Type 3.2154" for DX, -5.5692" for DY, press Enter.
- Finish the miter bend: type 50 for Node and -3.2154" for DY, press Enter.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Example 2: Widely spaced miter (3 cuts)								
2	10	From							Anchor
3	20	Miter	3.2154			1	8	1	
4	30	Miter	5.5692	-3.2154		1	8	1	
5	40	Miter	3.2154	-5.5692		1	8	1	
6	50			-3.2154		1	8	1	



Sherwood, David. R., and Dennis J. Whistance. THE "PIPING GUIDE" A Compact Reference for the Design and Drafting of Industrial Piping Systems. First Edition (revised). San Francisco: Syentek Books Co., 1980.

Node

A Node refers to a connecting point between two elements such as pipes, reducers, valves or expansion joints. The maximum # of nodes in a model cannot exceed 9,999; while the maximum node number itself cannot be greater than 99,999.Usually, a node has a numeric designation. In CAEPIPE, occasionally, you may have a need to reference a node followed by a letter such as A/B/C/D/J. These are automatically generated internal nodes. A and B nodes (e.g., 20A, 20B) designate the near and far ends of a Bend node (see section on Bend). J, C and D (e.g., 10J, 10C, 10D) designate a Jacketed pipe and a Jacketed bend (see section on Jacketed Piping).

In the Layout window, node numbers are typed under the column titled Node. A node number may be typed as an integer or an integer followed by one of the letters A/B/C/D/J. Use "Location" type to specify more than one data item at a node (See section on Location).

The starting node in a piping system is always a node of type "From," which is usually anchored.

Nodes not only act as connect points for elements but also act as locations for providing supports or applying restraints/external forces and moments to the piping system. Each node has six static degrees of freedom (three dynamic), three translational (in x, y and z directions) and three rotational (about x, y and z axes). Any or all of them may be restrained using supports.

Specifying Coordinates

The values typed in the DX, DY and DZ fields on the Layout window are interpreted as offsets from the previous node. If required to specify absolute coordinates for a node (i.e., fix the location of a point in space), the node has to be of type "From" or should have an asterisk (*) at the end of it (e.g., 20*). In these cases, the numbers entered in the DX, DY and DZ fields are interpreted as absolute coordinates of the node rather than offsets from the previous node. If the coordinates for a particular node are duplicated the second set of values is ignored. An asterisk (*) for a "From" node is ignored too.

You can list all coordinates by selecting Coordinates under Misc menu (or click on the Node header or right click on any node number). This feature can be helpful for verifying correctness of the input.

In the Layout window, you can search for a node by using the "Find node" command (under the View menu, or use Ctrl+F), specify a "Node increment" for automatic node numbering (under Options menu), and renumber nodes for a range of rows (under Edit menu)

Automatic Renumbering

When you delete a row with a node number in the Layout window, CAEPIPE automatically renumbers nodes from the top starting with the number you have specified under the main window > File > Preferences > Automatic Renumbering. Note that this is different from the user-performed selective renumbering of a range of rows from the Edit menu.

When turned on, deleting a row triggers an automatic renumbering operation inside the Layout window. So if you do not want such to happen, turn the feature off from the main window.

Nonlinearities

There are several types of nonlinearities in CAEPIPE:

1. Gaps in limit stops and guides.
2. Rotational limits in ball and hinge joints.
3. Rod hangers as one-way restraints.
4. Tie rods with different stiffnesses and gaps in tension and compression.
5. Friction in limit stops, guides, slip joints, hinge joints and ball joints.
6. Buried piping.

An iterative solution is performed when nonlinearities are present. At the start of each iteration, the overall global stiffness matrix and the load vector are reformulated based on the solution (displacements) obtained from the previous iteration.

Limit stop

Limit stops are input by specifying direction, the upper and lower limits (i.e., gaps) and optionally a friction coefficient and a support stiffness that comes into play when either gap is closed. The upper and lower limits are along the direction of the limit stop and measured from the undeflected position of the node. Typically the upper limit is positive and the lower limit negative. The upper limit should be algebraically always greater than or equal to the lower limit. In some situations it is possible to have a positive lower limit or a negative upper limit. If a particular limit does not exist (i.e., a node can move unrestrained in that direction), that limit should be left blank. If a gap does not exist, then the corresponding limit should be explicitly input as zero.

Solution Procedure

At the end of each iteration, the displacements computed at the limit stop node are resolved along the limit stop direction. The resolved displacement (in the limit stop direction) is compared with the upper and lower limits. If the resolved displacement exceeds the upper limit or is less than the lower limit, the gap is closed; otherwise it is open. The solution is converged when the displacements computed are within 1% of the corresponding displacementsobtained at the end of the previous iteration.

If both upper and lower limits (gaps) of a limit stop are open, no stiffness is applied at thatlimit stop node, and friction does not arise there.

If either one of the two gaps is closed,the support load along the limit stop direction is calculated as(resolved displacement - gap distance) x user-specified support stiffness. In this case, if friction is specified at the limit stop, the following iterative procedure is carried out.

1. The resultant displacement (d) in local yz plane is computed at this limit stop node.
2. The support load at the limit stop is calculated as (resolved displacement - gap distance) x user-specified support stiffness.
3. Using this calculated support load, the friction force (ff) is calculated as ff = friction coefficient (mu) x support load.
4. Using the friction force (ff) and the resultant in-plane displacement (d), the friction stiffness (kf) is computed as kf = ff / d.
5. The global stiffness matrix is then updated to include the friction stiffness computed.

6. Analysis iteration is continued with the updated global stiffness matrix. The solution is converged when the displacementscomputed are within 1% of the corresponding displacements obtained at the end of the previous iteration.
7. After the solution has converged and if the gap is closed, the support load and the friction force at the limit stop are calculated as:

   Support load at limit stop = (resolved displacement - gap distance) x user-specified support stiffness.

   y shear = fy = local y displacement x friction stiffness (kf)

   z shear = fz = local z displacement x friction stiffness (kf)

   Resultant friction force (ff) = sqrt(fy^2 + fz^2)

During hanger design, the hot loads are recalculated with the status of the limit stops at the end of the preliminary operating load case. Then the hanger travels are recalculated using the recalculated hot loads.

In dynamic analysis, the status of the limit stops at the end of the first operating load case (W+P1+T1) is used. If either the upper or lower limit is reached for the first operatingload case, the limit stop is treated as a rigid two-way restraint in the direction of the limit stop. If both limitsare not reached, then that limit stop is treated as having no restraint.

Friction

Friction is specified by entering coefficient of friction for limit stop and guide, entering friction force and/or friction torque for slip joint, entering friction torque for hinge joint and entering bending and/or torsional friction torque for ball joint.

Friction is modeled using variable equivalent friction stiffnesses (fictitious restraints) in CAEPIPE as described in the above Solution Procedure. The stiffnesses of these fictitious restraints are estimated from the results of previous iteration. If friction is included in dynamic analysis, these equivalent friction stiffnesses computed from the last iteration of the first operating load case are included in modal and dynamic analyses.

Friction in Limit Stop

If the gap is not closed, there is no normal force and hence no friction. If the gap is closed, the normal force (limit stop support load) is calculated as explained above. The maximum friction force is friction coefficient * normal force. The displacement of the limit stop node is resolved into a plane normal to the limit stop direction (let us call this resolved displacement: y). Also let k_y = equivalent friction stiffness which is assumed to be zero for the first iteration.

If y is non-zero or y * k_y> maximum friction force,

then k_y = maximum friction force / y

otherwise k_y = high stiffness (1×10^{12} lb/inch) [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the global stiffness matrix. The iterations are continued till the displacement y is within 1% of y displacement from the previous iteration. The friction force is y * k_y.

Friction in Guide

A guide is modeled by adding high stiffnesses perpendicular to the direction of the pipe. The normal force in the guide is calculated by the vector sum of the local y and z support loads. Maximum friction force is friction coefficient * normal force. The displacements at the guide node are resolved in the direction of the guide axis. Let us call this displacement: x. Also let k_x = equivalent friction stiffness which is assumed to be zero for the first iteration.

If x is non-zero or x * k_x> maximum friction force,

then k_x = maximum friction force / x

otherwise k_x = high stiffness (1×10^{12} lb/inch) [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the global stiffness matrix. The iterations are continued till the displacement x is within 1% of x displacement from the previous iteration. The friction force is x * k_x.

Friction in Slip Joint

The relative displacements (between From node and To node) for the slip joint are resolved in the direction of the slip joint. Let us call this relative displacement: x. Also let k_x = equivalent friction stiffness which is assumed to be zero for the first iteration.

If x is non-zero or x * k_x> friction force input,

then k_x = friction force input / x

otherwise k_x = high stiffness (1×10^{12} lb/inch) [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the global stiffness matrix. The iterations are continued till the displacement x is within 1% of x displacement from the previous iteration. The friction force is x * k_x.

Similar technique is used for friction torque (using rotations instead of translations).

Friction in Hinge Joint

The relative rotations (between From node and To node) for the hinge joint are resolved in the direction of the hinge axis. Let us call this relative rotation: x. Also let k_x = equivalent friction stiffness which is assumed to be zero for the first iteration. Maximum friction torque = friction torque input + hinge stiffness input * x.

If x is non-zero or x * k_x> maximum friction torque,

then k_x = maximum friction torque / x

otherwise k_x = high stiffness [This is the case of no sliding]

In the next iteration the equivalent friction stiffness is added to the global stiffness matrix. The iterations are continued till the relative rotation x is within 1% of x rotation from the previous iteration. The friction torque is x * k_x.

Friction in Ball Joint

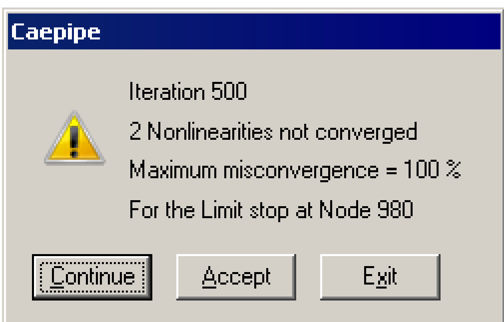
For a ball joint friction in bending (transverse) and torsional (axial) directions is treated independently. For the bending case, the resultant of the local y and z directions is used. Otherwise a procedure similar to the one used for hinge joint is used.

Friction in Dynamic Analysis

Friction is optional in dynamic analysis. Friction is mathematically modeled by using equivalent stiffnesses. If friction is included in dynamic analysis, these equivalent friction stiffnesses computed from the last iteration of the first operating load case are included in modal and dynamic analyses.

Misconvergence

During the iterative solution procedure for nonlinearities, a misconvergence is reported in the following manner:



You have three options:

Continue the iterative procedure for 500 more iterations to see whether the solution converges, or
Accept the misconvergence (maximum misconvergence is reported, 100% in the above dialog), or
Exit the analysis processor completely.

An environment variable “CPITER” may be defined to change number of iterations from the default 500. For example, when “CPITER=1000”, iterations will continue up to 1000 before showing the dialog: Continue, Accept or Exit if there is misconvergence.

The shown misconvergence in the solution is really a quantification of how much off the results will be, IF you accept it. The maximum misconvergence (100% in the above case) is shown along with its location where such is happening.

In case of a solution with large misconvergence, you have two options: 1. Change parameters (mainly gaps and friction values, or removal of unneeded supports) inside the model to influence the convergence routine, or, 2. Increase the # of iterations. In case neither works, then use engineering judgment to accept or reject the merit of such a solution.

Nozzles are integral attachments of vessels (such as pressure vessels, storage tanks etc.) which connect with external piping. Nozzles transmit the shell (vessel) flexibility to the piping system and hence are generally included in piping stress analysis.

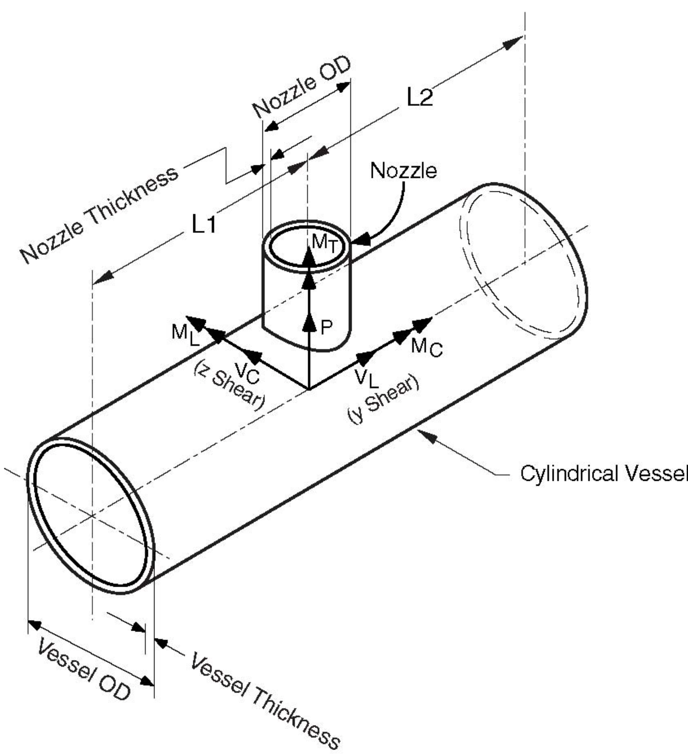
Three types of nozzles can be modeled in CAEPIPE, namely (i) a nozzle attached to a cylindrical vessel relatively far from the ends of the cylinder, (ii) a nozzle attached to a spherical vessel or torispherical head and (iii) a nozzle attached to a cylindrical shell with a flat bottom and close to the flat-bottom. CAEPIPE calculates the nozzle stiffnesses (local flexibility components) according to WRC 297, PD 5500 and API 650 guidelines. See Annexure II for the procedures according to WRC 297 and API 650.

Note:

Pressure Thrust (End-cap Force) of Pressure P x Inner Area (A) of pipe is not included in the Support Loads for Nozzles displayed by CAEPIPE at this time. Since CAEPIPE's results for numerous problems compare well with the results from other third-party software, it confirms that the other stress programs are also not including the Pressure Thrust (End-cap Force) of pipe in the Nozzle Loads at this time. Refer to the Verification Manual supplied with CAEPIPE for comparison of results with other stress programs.

If you wish to include the effect of Pressure Thrust (End-cap force) due to internal pressure in your piping on the Nozzle loads, then you will have to compute the same manually (= P x A) and input it as an external force at the Nozzle Nodes using the Force data type available with CAEPIPE. Please choose the option "Add to W+P" in the Force data type dialog. By doing so, the End-cap force (= P x A) will be included in all relevant load cases and combinations of CAEPIPE. Of course, when the "None" code is selected under Optionss > Analysis > Code, this End-cap force is included in the only case of "Static".

Nozzle attached to a cylindrical vessel

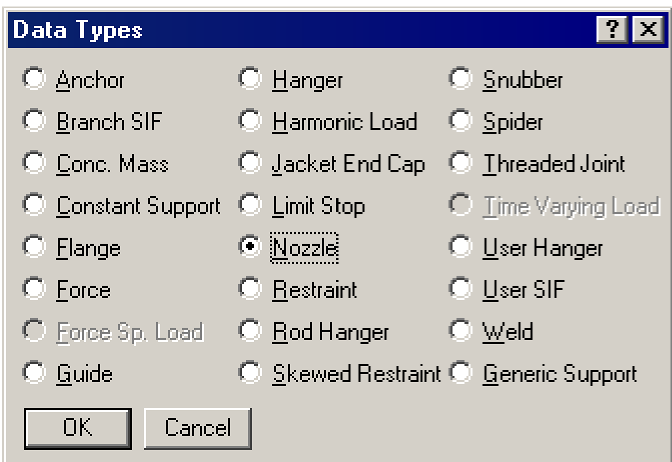


The coordinate system is as shown in the figure. The six components of the forces and moments at the nozzle-vessel interface are:

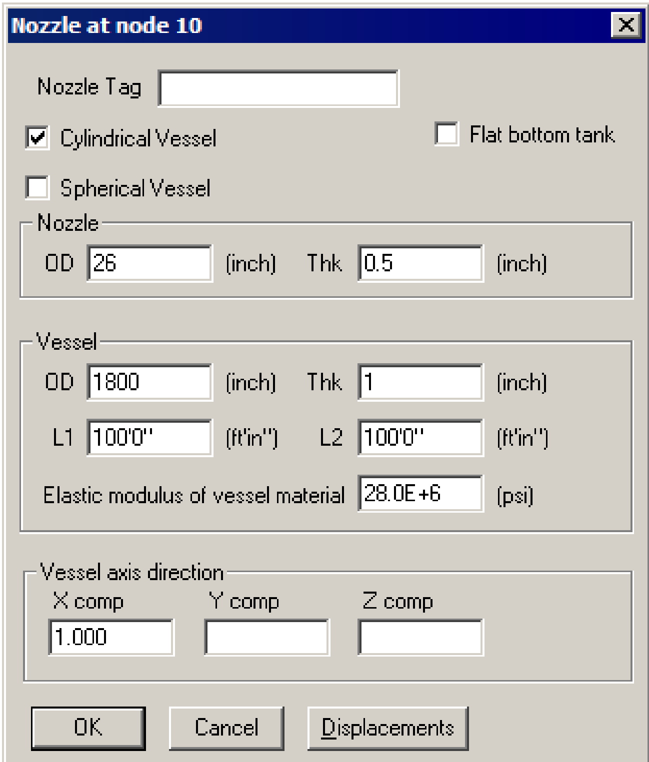
P = Radial load	M_C = Circumferential moment
V_C = Circumferential load	M_T = Torsional moment
V_L = Longitudinal load	M_L = Longitudinal moment

Of the six components of shell stiffnesses, only three stiffnesses, axial (Kx), circumferential (Kyy), and longitudinal (Kzz), are computed. The remaining three are assumed to be rigid.

A nozzle is input by typing "n" in the Data column or selecting "Nozzle" from the Data Types dialog.



The Nozzle dialog is shown. Note that the Displacements button is disabled. It is only enabled after the nozzle is input (i.e., existing nozzle).



Nozzle

OD: Outside diameter of the nozzle.
Thk: Thickness of the nozzle.

Vessel

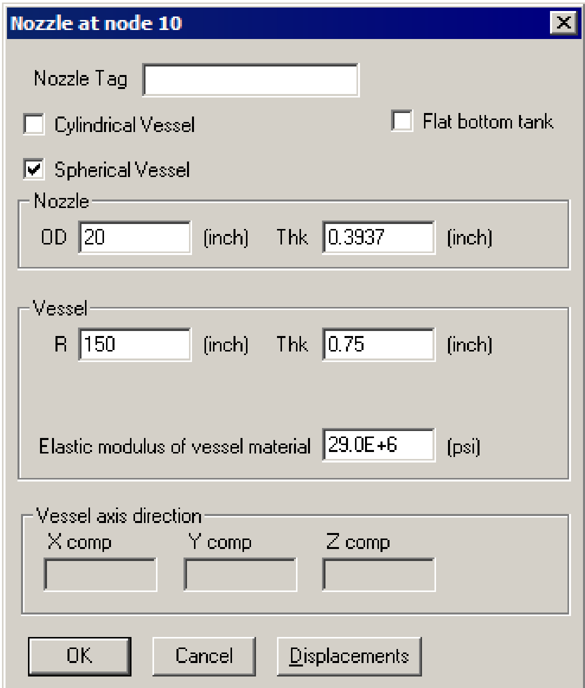
OD: Outside diameter of the vessel.
Thk: Thickness of the vessel.
L1, L2: Distances from the nozzle to the nearest stiffening ring, tubesheet or the vessel end.

Vessel axis direction

The orientation of the vessel axis in terms of its global X, Y and Z components are entered here. See example under "Specifying a Direction."

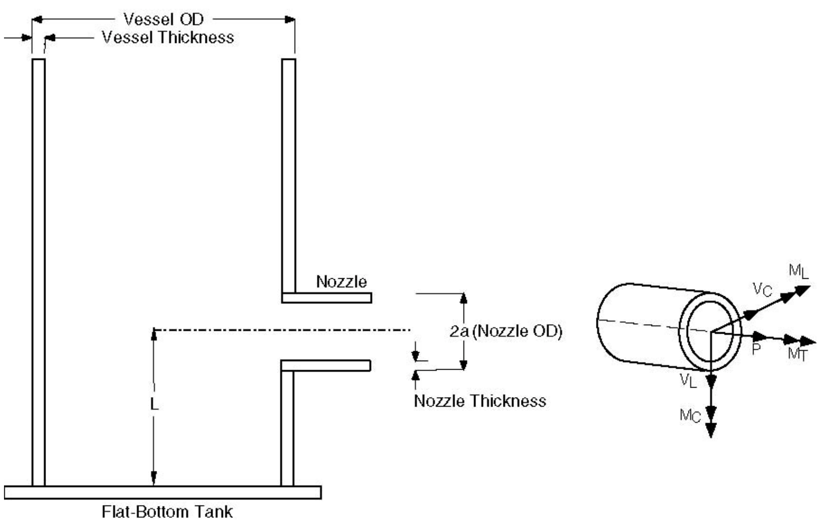
Nozzle to Spherical / Torispherical Shell

For a nozzle attached to a spherical shell or a torispherical head, check the Spherical Vessel checkbox and enter the required details.

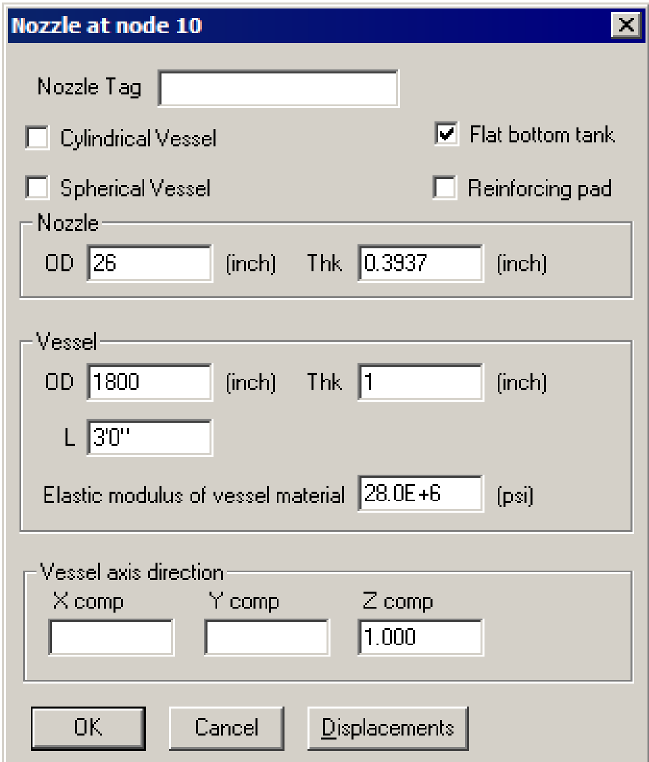


Nozzle attached to Flat-Bottom Tanks

For a nozzle on a flat-bottom tank and close to the flat-bottom, check the Flat-bottom tank checkbox.



A slightly modified Nozzle dialog is displayed.



All the input fields are the same as before except:

L

L is the distance from the flat-bottom to the nozzle centerline.

Reinforcing pad

If the nozzle is reinforced, check this box.

Stiffness Calculation

Here again, only three shell stiffnesses, axialtranslational (i.e., axial to the pipe, the same as radial to the shell), circumferential bending and longitudinal bending are computed according to API 650 guidelines. See Annexure II for the procedure. The remaining three shell stiffnesses are assumed to be rigid.

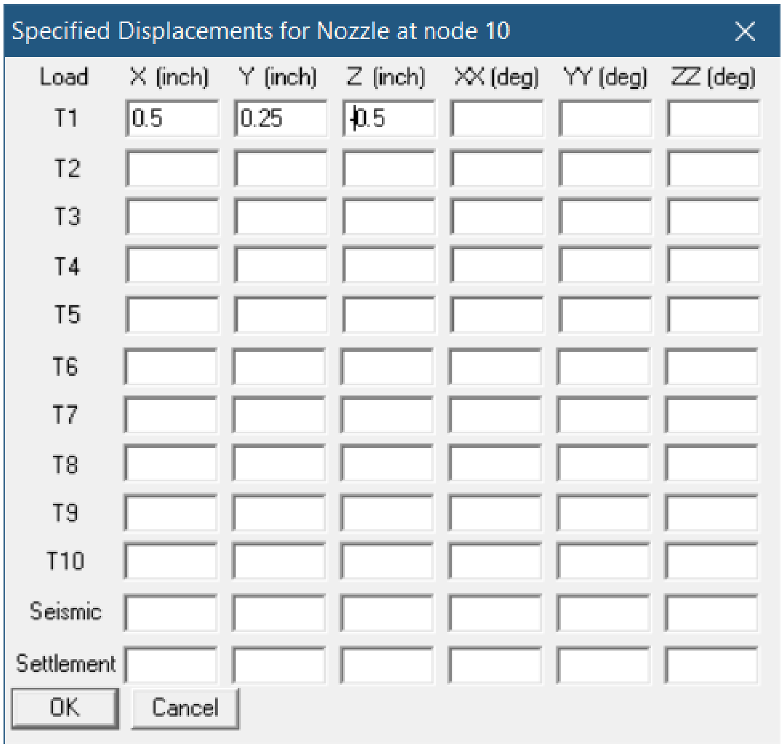
The flat-bottom tank nozzles are subject to the following limitations (API 650):

Limitations

- Nozzle OD/Vessel OD ratio must be between 0.005 and 0.04.
- L/Nozzle OD ratio must be between 1.0 and 1.5. See graphs in Annexure II, Figures D-3 through D-14.

Displacements

After a nozzle is input in a stress model, Displacements (translations and/or rotations) in the global X, Y and Z directions may be specified for that nozzle (for thermal, settlement and seismic cases). Click on the Displacements button. You will see a dialog similar to one shown below. Type in specified displacements and press Enter.



There are three types of displacements which can be specified:

1. Thermal (up to 10 displacements can be specified, one for each of the thermal loads T1 through T10). Applied only to the Expansion and Operating load cases.
2. Seismic (available for B31.1, ASME Section III Class 2, RCC-M and EN 13480 codes only). Solved as a separate internal load case and the results so obtained for this case are added absolutelyto the corresponding results from static seismic and response spectrum load cases.
3. Settlement (available under ASME B31.1, ASME Section III Class 2, RCC-M and EN 13480 codes only). Applied as a separate load case called Settlement as described below.

Settlement

For certain piping codes (ASME B31.1, ASME Section III Class 2, RCC-M and EN 13480), a settlement, which is a single non-repeated movement (e.g., due to settlement of foundation), may be specified. This is applied to the Settlement load case. For those codes which do not have a separate provision for settlement (like B31.3), specify the settlement as a thermal displacement (a conservative approach) for one of the temperature load and define that temperature as equal to reference temperature.

Example 1: Nozzle on a cylindrical vessel

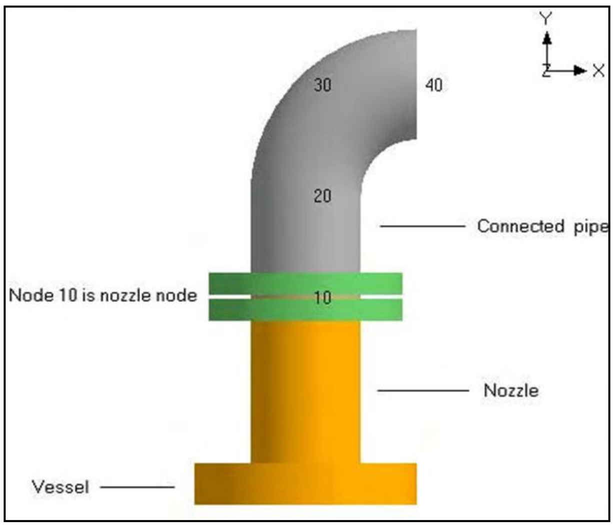
Assume the following data:
Vessel Radius = 900 in.
Vessel Thickness = 1.0 in.
Nozzle OD = 26 in.
Nozzle Thickness = 0.5 in.
L1 = L2 = 1200 in.

Elastic modulus for vessel material = 28×10^6 psi.

The first node (10) is already defined as an anchor. To replace the anchor by a nozzle, right click on the Anchor in the Data column, then select Delete Anchor. Then type "n" in the Data column to input the nozzle. The Nozzle dialog will be shown. Input the nozzle data in the dialog. The Layout window looks like the following:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example 1: Nozzle on a cylindrical vessel								
2	10	From							Nozzle
3	20			2'0"		P11	26	1	
4	30	Bend		2'2"		P11	26	1	
5	40		2'2"			P11	26	1	
6	10	Location							Flange

The graphics is shown next.



Nozzle Stiffnesses

The three local shell stiffnesses computed can be viewed using the List command (Ctrl+L in the Layout window) and selecting Nozzle Stiffnesses. The following window is displayed.

#	Node	Flat Bot.	Axial (lb/inch)	Circumferential (in-lb/deg)	Longitudinal (in-lb/deg)
1	10	No	12534	3.549E+5	6.980E+5

The nozzle/vessel data may be edited here (double click to edit).

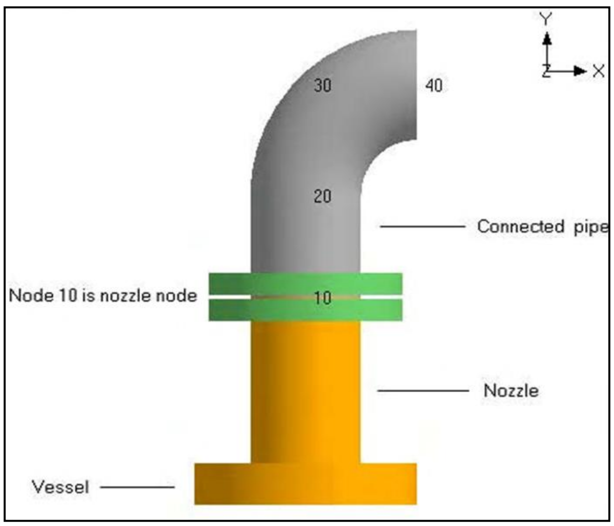
Example 2: Nozzle on a spherical vessel

Assume the following data:
Vessel OD = 1800 in.
Vessel Thickness = 1.0 in.
Nozzle OD = 26 in.
Nozzle Thickness = 0.5 in.
Elastic modulus for vessel material = 28×10^6 psi.

The first node (10) is already defined as an anchor. To replace the anchor by a nozzle, right click on the Anchor in the Data column, then select Delete Anchor. Then type "n" in the Data column to input the nozzle. The Nozzle dialog will be shown. Input the nozzle data in the dialog. The Layout window looks like the following:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title =								
2	10	From							Nozzle
3	20			2'0"		P11	26	1	
4	30	Bend		2'2"		P11	26	1	
5	40		2'2"			P11	26	1	
6	10	Location							Flange
7									

The graphics is shown next.



Nozzle Stiffnesses

The three local shell stiffnesses computed can be viewed using the List command (Ctrl+L in the Layout window) and selecting Nozzle Stiffnesses. The following window is displayed.

#	Node	Vess. Type	Radial (kp) (lb/inch)	Circumferential (kmc) (in-lb/deg)	Longitudinal (kml) (in-lb/deg)
1	10	Sph	1.455E+5	1.099E+6	1.099E+6

The nozzle/vessel data may be edited here (double click to edit).

Example 3: Nozzle on a flat-bottom storage tank

Assume the following data:

Vessel OD = 1800 in.
Vessel Thickness = 1.0 in.
Nozzle OD = 26 in.
Nozzle Thickness = 0.5 in.

L = 36 in.

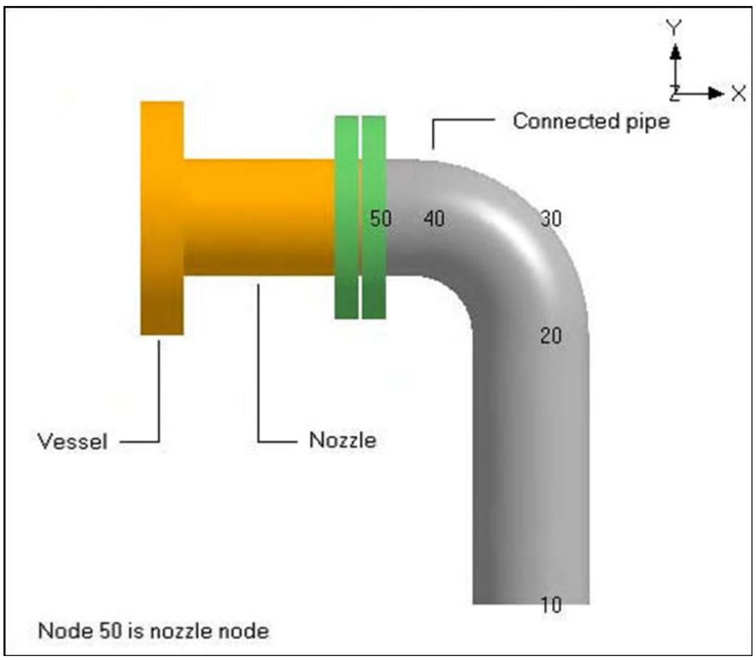
Elastic modulus of the vessel material = 28×10^6psi.
No reinforcing pad on the vessel.

Create the piping till the nozzle node. At the nozzle node, enter a Nozzle by typing "n", check the Flat-bottom tank checkboxand provide the required data.

The Layout window looks like the following.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Load	Data
1	Title = Example 2: Nozzle on a flat-bottom tank								
2	10	From							
3	20			60		P11	26	1	
4	30	Bend		26		P11	26	1	
5	40		-26			P11	26	1	
6	50		-12			P11	26	1	Nozzle
7	50	Location							Flange

The graphics is shown next.

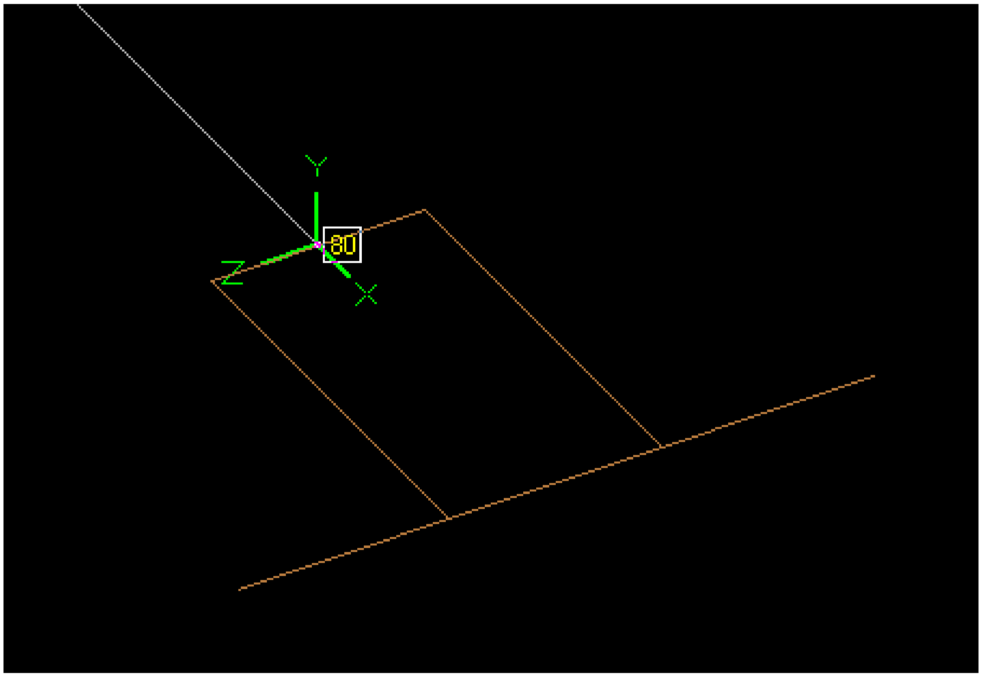


The three nozzle stiffnesses computed can be viewed as before by using the List command (Ctrl+L) and selecting Nozzle Stiffnesses.

LCS (Local Coordinate System) can be displayed for Nozzle element by a few ways:

1. List (Ctrl+L) > Nozzles > View menu > Show LCS, -OR-
2. List (Ctrl+L) > Nozzles > Mouse Right Click on the listed Nozzle > Show LCS, -OR-
3. Results window > Support Loads > Other Support Loads > Nozzles > View menu > Show LCS, -OR-
4. Results window > Support Loads > Other Support Loads > Nozzles > Mouse Right

Click > Show LCS.



One of the qualification requirements for a piping system is to keep the loads imparted by the piping on equipment nozzles within certain allowable limits. These loads consist of sets of three forces and three moments, for the various load combinations. There are basically two types of nozzle load limits: (1) nozzle loads applied to rotating equipment, and (2) nozzle loads applied to static equipment such as heat exchangers, tanks and vessels.

Nozzle Loads applied to rotating equipment

Rotating equipment consist of equipment with moving parts, such as pumps, compressors, turbines and fans. The pipe nozzle load limits are developed by the equipment manufacturer and are intended to prevent malfunction, such as shaft misalignment, or distortion of the casing that could impede the movement of impellers. These limits are typically based on actual testing of the equipment, and not on analysis.

Some pump standards have published standard nozzle load limits, but these are only valid for the particular pumps for which they are published. This is the case for the American Petroleum Institute's API-610 and the Hydraulic Institute's HI 9.6.2.11 standard.

See subsection titled "Pump" from this manual for more details.

Similarly, API 617 provides nozzle load limits for compressors and NEMA SM-23 for turbines.

See subsections titled "Compressor" and "Turbine" from this manual for further details.

Nozzle Loads applied to static equipment

For Nozzlesconnected to static equipment such as heat exchangers, tanks and vessels, the pipe load limits are based on stress or strain limits at the nozzle-to-shell intersection, both on the shell and nozzle sides.*At present, CAEPIPE considers pipe load limits based on stress or strain limits only at the shell side of the nozzle intersection.*

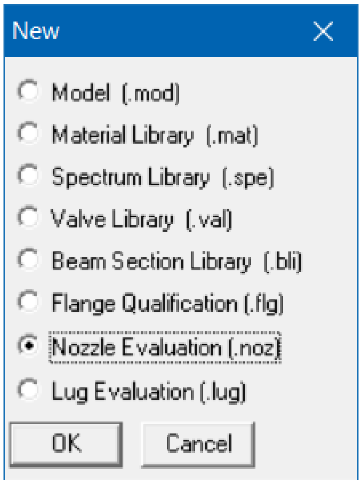
Nozzle Evaluation Module

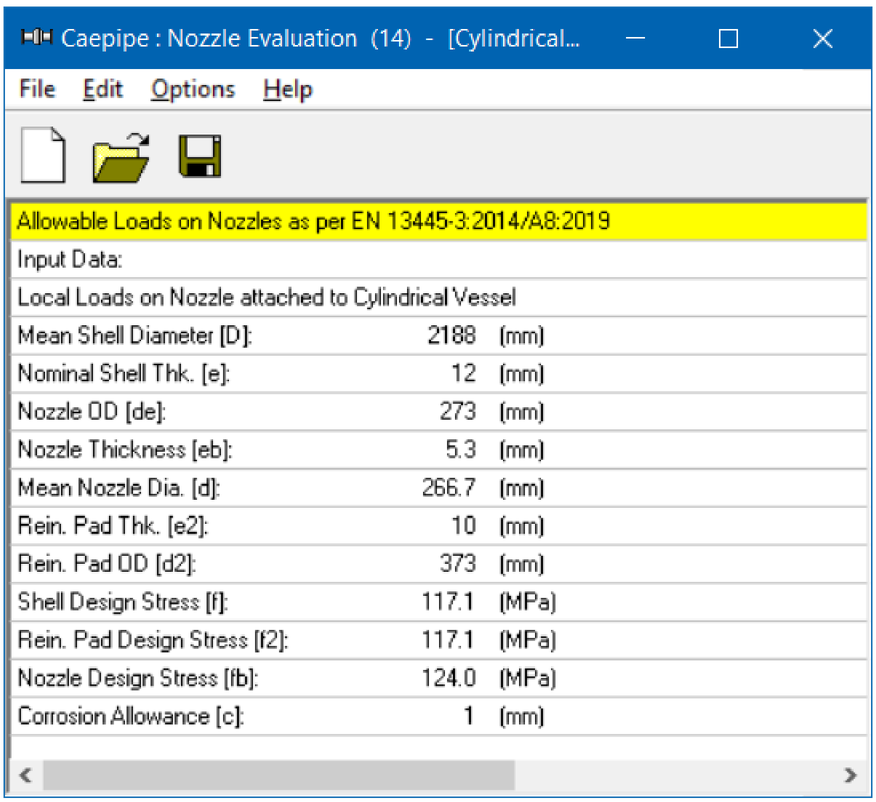
The Nozzle Evaluation module implemented in CAEPIPE computes Allowable Loads and Local Stresses at Shell for Nozzles connecting to Spherical and Cylindrical Vessels as per the following codes.

1. Allowable Loads on Nozzle – EN 13445-3:2014/A8:2019 (hereinafter referred as EN13445).
2. Local Shell Stresses at Nozzle– WRC Bulletin 537 (hereinafter referred as WRC 537).

The Nozzle Evaluation module assumes that the flexibility analysis of that piping system with CAEPIPE has already been performed, which will have produced a stress report as well as the forces and moments at the location where the nozzle is attached to the piping.

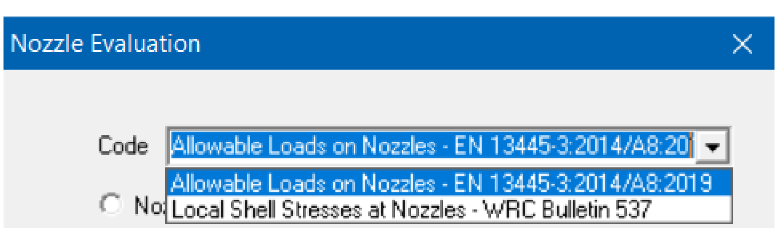
Note that this module is separate from a piping stress model file and can be accessed from File Menu > Open/New command.





Double-clicking anywhere in the previous screen (or Edit menu > Edit (Ctrl+E)) opens a dialog with input fields you can edit. You will need to enter the data in this dialog. The different parameters required to be input are explained in detail below.

Details on implementation for Calculation of Local Shell Stresses at Nozzles as per WRC Bulletin 537 are provided in Code Compliance Manual.



Code

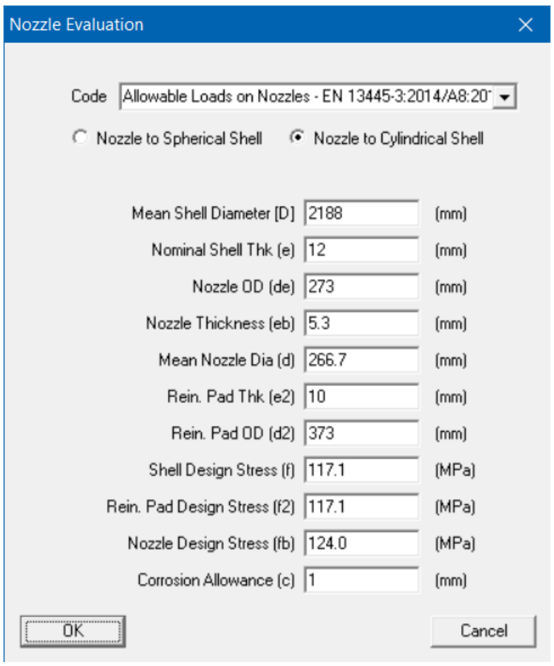
Selecting the Code as "Allowable Loads on Nozzles – EN 13445-3:2014/A8:2019" will allow user to compute the "Allowable Loads on Nozzles".

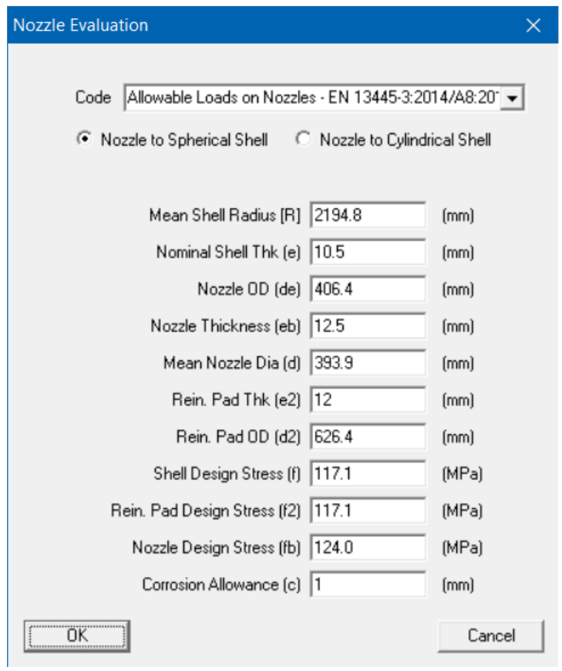
On the other hand, selecting the Code as "Local Shell Stresses at Nozzles – WRC Bulletin 537" will allow user to compute the Local Shell Stresses at Nozzles as per WRC 537 and perform Stress Compliance as per ASME Section VIII Division 2.

Nozzle to Spherical / Cylindrical Shell

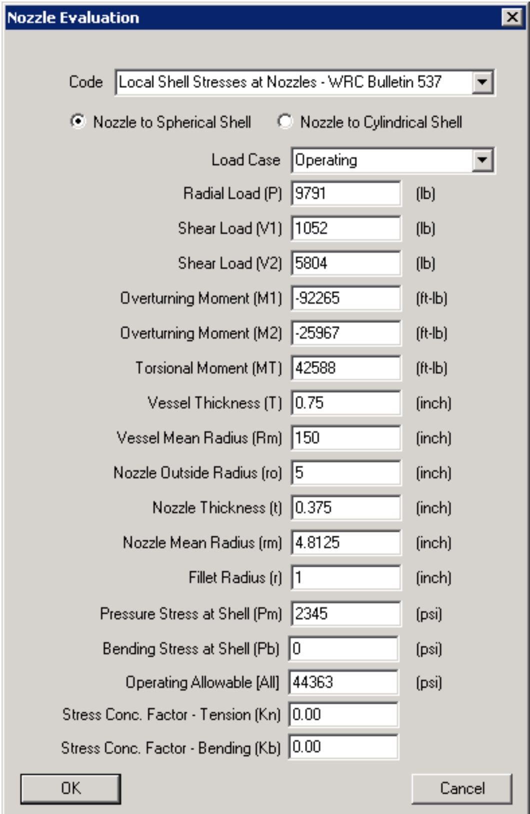
Selecting the option "Nozzle to Spherical / CylindricalShell" will show the dialog boxes as shown below for the two Codes.

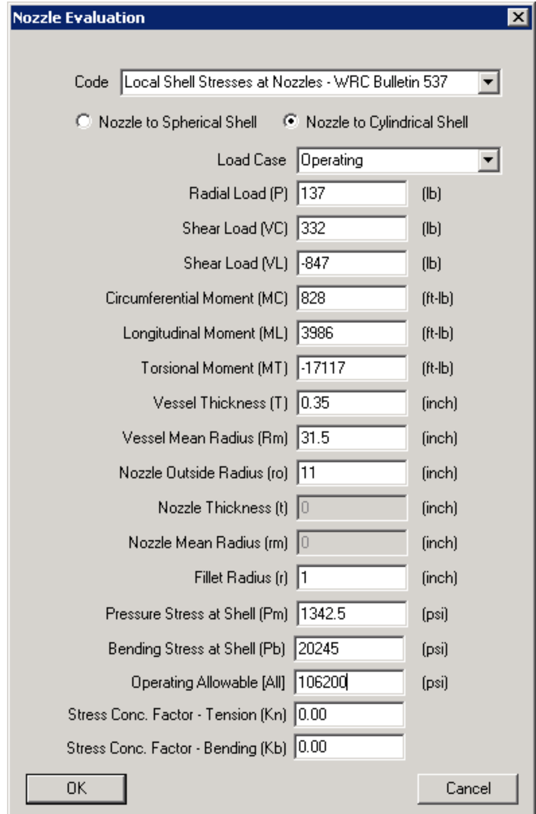
For EN13445, the input data required for the two types are shown below.





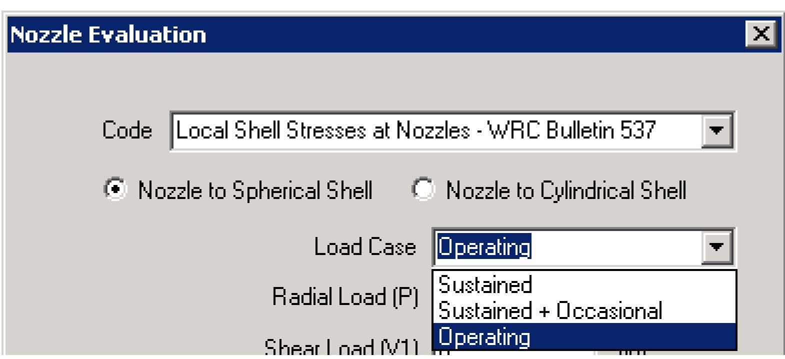
Similarly, for WRC 537, the data required to be input for the two types are shown below.





Load Case

From the option available, select the Load Case for which the three (3) forces and three (3) moments are being entered.



Depending upon the selection of "Load Case", CAEPIPE will compute Radial/Circumferential, Tangential/Longitudinal, Shear and Combined Stress Intensities as explained below.

Please note, the display text for "Allowable (All)"inputfield will change automaticallydepending on the "Load Case" selected.

For example, for "Sustained" Load Case, the display text for "Allowable (All)" input field will change to "Sustained Allowable (All)". Similarly for "Sustained + Occasional", the display text will change to "Sustained + Occasional Allowable (All)".

Sustained Allowable [All] 18000 (psi)

Sustained + Occasional Allowable [All] 22000 (psi)

Operating Allowable [All] 28000 (psi)

Sustained and Sustained + Occasional

Combined Stresses computed will<u>exclude</u> thefollowing BendingStresses from the Evaluation of Nozzle to Spherical / Cylindrical Shells respectively.

a. Radial/Circumferential Bending Stresses due to P

b. Radial/Circumferential Bending Stresses due to M_1/M_C

c. Radial/Circumferential Bending Stresses due to M_2/M_L

d. Tangential/Longitudinal Bending Stresses due to P

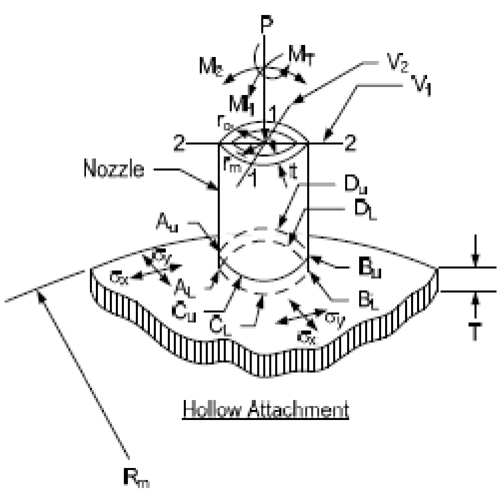
e. Tangential/Longitudinal Bending Stresses due to M_1/M_C

f. Tangential/Longitudinal Bending Stresses due to M_2/M_L

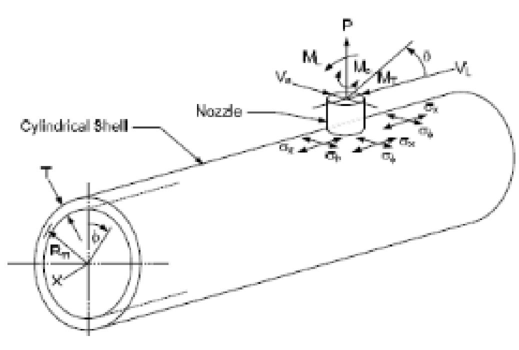
Operating

Combined Stresses computed will include all Membrane and Bendingstresses due to P,M_1 and M_2 / P, M_C and M_L, Torsional stresses due to $M_T$as well as Shear stressesdue to V_1/V_C and V_2/V_L.

Once the required data are input, save the file (Nozzle evaluation will have a .noz extension). Now, select File menu > Analyze to calculate loads or stressesas per the code selected, which will be shown *right below* the input information.



Hollow Attachment



Solid Attachment

Loads

For Nozzle to Spherical Vessel, enter the following loadscomputedin piping analysis for the selected load case.

1. Radial Load (P)
2. Shear Load (V_1)
3. Shear Load (V_2)
4. Overturning Moment (M_1)
5. Overturning Moment (M_2) and
6. Torsional Moment (M_T)

Similarly, for Nozzle to Cylindrical Vessel, enter the following loads computed in piping analysis for the selected load case.

1. Radial Load (P)
2. Shear Load (V_C)
3. Shear Load (V_L)
4. Circumferential Moment (M_C)
5. Longitudinal Moment (M_L) and
6. Torsional Moment (M_T)

Vessel and Nozzle Parameters

For Spherical Vessel, from the Vessel Drawing, read and enter the following parameters.

Vessel Thickness (T), Vessel Mean Radius (R_m), Nozzle Outside Radius (r_o), Nozzle Thickness (t) and Nozzle Mean Radius (r_m).

Similarly, for Cylindrical Vessel, from the Vessel Drawing, read and enter the following parameters.

Vessel Thickness (T), Vessel Mean Radius (R_m) and Nozzle Outside Radius (r_o).

Fillet Radius

Fillet Radius is required to compute the Stress Concentration Factors for Tension (K_n) and Bending (K_b) from Figure B-2 WRC Bulletin 537.

Pressure Stress at Shell (P_m)

Pm is the Average Primary Membrane Stress across the cross-section of the vessel away from Gross Structural Discontinuities such as a Nozzle.

For a Spherical Shell such as Enclosure/Head to a Vessel, Pm due to internalpressure would be PR/2T,where P = Internal Pressure, R is the Mean Radius of the Head and T is the Thickness of the Head.

For a Cylindrical Shell such as Pressure Vessel/Pre-heater/Tank, to be conservative, Pm due to Internal Pressure would be the Circumferential Stress = PD/2T, where D is the Mean Diameter of the Cylinder.

The Stress value thus calculated should be entered in this field.

Bending Stress at Shell (P_b)

Pb is the Primary Membrane Stress proportional to the distance from the Axis of the Vessel due to External Loads such as Weight, Wind, Earthquake, etc. away from Gross Structural Discontinuities such as a Nozzle.

For a Spherical Shell such as Enclosure/Head to a Vessel, being a free end, Bending Stress Pb due to external loads could be 0.0.

For a Cylindrical Shell such as Pressure Vessel/Pre-heater/Tank, Bending Stress Pb due to external loads could be calculated as M/Z, where M is the Bending Moment on the Shell at Nozzle location and Z is the Section Modulus of the Shell.

So, Bending Stress Pb could be manually calculated or determinedusing computer programs.

Allowable Stress (All)

Allowable Stress (All) is to be computed and entered depending on the load case selected.

For example, as perClause 5.2.2.4 of ASME Section VIII Division 2 (2017), the Allowable Stress (All) for both "Sustained" and "Sustained + Occasional" load case is to be entered as "1.5S_h", where S_h is the basic allowable stress at maximum metal temperature for Shell.

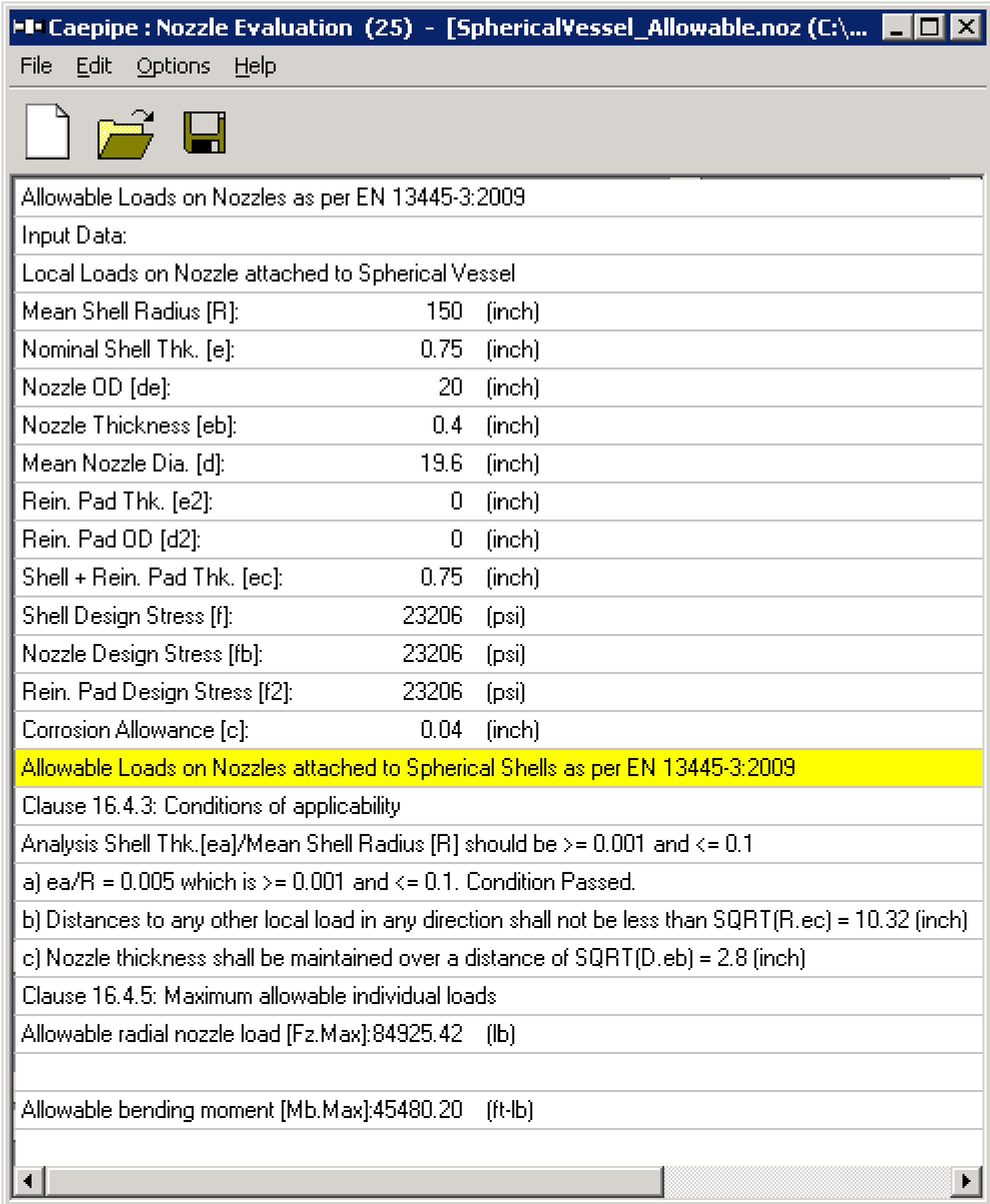
Similarly,as per Clause 5.5.6 of ASME Section VIII Division 2 (2017), the Allowable Stress (All) forOperating load case should be entered as "3 (S_C + S_h)/2 = 1.5 (S_C + S_h)", where S_c is the allowable stress at minimum metal temperature for Shell and S_h is defined above.

Stress Concentration Factors (K_n and K_b)

CAEPIPE will automatically compute the values of K_n and $K_b$using Figure B-2 of WRC 537 when these fields are entered as 0.0.

On the other hand, when thesefields are entered with a value great than or equal to 1.0, then CAEPIPE will use these values of K_n and $K_b$while computing the stresses as per WRC 537.

Once the required data are input, save the file (Nozzle evaluation will have a .noz extension). Now, select File menu > Analyze to perform the evaluation, which will be shown *right below* the input information.


Caepipe : Nozzle Evaluation (25) - [SphericalVessel_Allowable.noz (C:\...
File Edit Options Help
Allowable Loads on Nozzles as per EN 13445-3:2009
Input Data:
Local Loads on Nozzle attached to Spherical Vessel
Mean Shell Radius [R]: 150 (inch)
Nominal Shell Thk. [e]: 0.75 (inch)
Nozzle OD [de]: 20 (inch)
Nozzle Thickness [eb]: 0.4 (inch)
Mean Nozzle Dia. [d]: 19.6 (inch)
Rein. Pad Thk. [e2]: 0 (inch)
Rein. Pad OD [d2]: 0 (inch)
Shell + Rein. Pad Thk. [ec]: 0.75 (inch)
Shell Design Stress [f]: 23206 (psi)
Nozzle Design Stress [fb]: 23206 (psi)
Rein. Pad Design Stress [f2]: 23206 (psi)
Corrosion Allowance [c]: 0.04 (inch)
Allowable Loads on Nozzles attached to Spherical Shells as per EN 13445-3:2009
Clause 16.4.3: Conditions of applicability
Analysis Shell Thk.[ea]/Mean Shell Radius [R] should be >= 0.001 and <= 0.1
a) ea/R = 0.005 which is >= 0.001 and <= 0.1. Condition Passed.
b) Distances to any other local load in any direction shall not be less than SQRT(R.ec) = 10.32 (inch)
c) Nozzle thickness shall be maintained over a distance of SQRT(D.eb) = 2.8 (inch)
Clause 16.4.5: Maximum allowable individual loads
Allowable radial nozzle load [Fz.Max]:84925.42 (lb)
Allowable bending moment [Mb.Max]:45480.20 (ft-lb)

Caepipe : Nozzle Evaluation (54) - [441_31_Operating.noz (C:\Documents and Settings\TEMP\Desktop\NozzEval)]

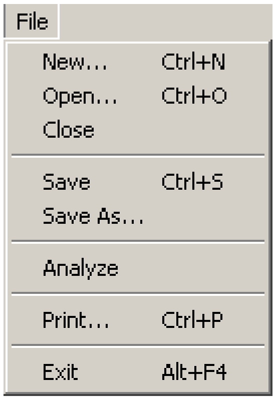
File Edit Options Help

Calculation of Local Stresses in Cylindrical Shells as per WRC Bulletin 537 (psi)

Stresses	Fig.No	Au	Al	Bu	Bl	Cu	Cl	Du	Dl
Circumferential Membrane (P)	3C	-45	-45	-45	-45	-45	-45	-45	-45
Circumferential Bending (P)	2C-1	-53	53	-53	53	0	0	0	0
Circumferential Bending (P)	1C	0	0	0	0	-230	230	-230	230
Circumferential Membrane (MC)	3A	0	0	0	0	-293	-293	293	293
Circumferential Bending (MC)	1A	0	0	0	0	-3113	3113	3113	-3113
Circumferential Membrane (ML)	3B	-2844	-2844	2844	2844	0	0	0	0
Circumferential Bending (ML)	1B-1	-2068	2068	2068	-2068	0	0	0	0
Circumferential Stresses (Sp)	-	-5010	-768	4815	784	-3680	3005	3130	-2635
Longitudinal Membrane (P)	4C	-134	-134	-134	-134	-134	-134	-134	-134
Longitudinal Bending (P)	1C-1	-153	153	-153	153	0	0	0	0
Longitudinal Bending (P)	2C	0	0	0	0	57	-57	57	-57
Longitudinal Membrane (MC)	4A	0	0	0	0	-1133	-1133	1133	1133
Longitudinal Bending (MC)	2A	0	0	0	0	-1185	1185	1185	-1185
Longitudinal Membrane (ML)	4B	-1517	-1517	1517	1517	0	0	0	0
Longitudinal Bending (ML)	2B-1	-3624	3624	3624	-3624	0	0	0	0
Longitudinal Stresses (Sx)	-	-5428	2126	4855	-2088	-2395	-138	2242	-243
Shear Stress (V1)	-	27	27	-27	-27	0	0	0	0
Shear Stress (V2)	-	0	0	0	0	70	70	-70	-70
Shear Stress (MT)	-	-772	-772	-772	-772	-772	-772	-772	-772
Shear Stresses (Z)	-	-744	-744	-799	-799	-702	-702	-842	-842
Combined Stress Intensity (PL+Q)	-	5992	3255	5635	3287	3989	3443	3638	2925
Stress Compliance as per ASME Section VIII Division 2 - Operating Load									
	Calculated	Allowed	Ratio	Status					
Stress (Pm+Pb+Pl+Q) (psi)	27580	106200	0.260	Pass					

Nozzle Evaluation Module Menus

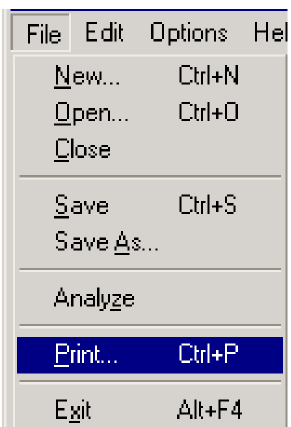
File Menu



.Analyze.

Analyze command calculates nozzle allowable loads as per EN 13445-3:2014/A8:2019 or shell stressesat the attachment as per WRC Bulletin 537 and compares them to stress allowable specified by ASME Section VIII Division 2 (2017).

Print.



You can print a Report by using the Print command. You can also preview the report by clicking the Preview button on the print dialog.

Caepipe : Print Preview

File View Options Help

Print | Prev Page | Next Page | Close

Caepipe — Page 1

Allowable Loads on Nozzles as per EN 13445-3:2009

Input Data:
Local Loads on Nozzle attached to Spherical Vessel

Mean Shell Radius [R]:	150 (inch)
Nominal Shell Thk. [e]:	0.75 (inch)
Nozzle OD [de]:	20 (inch)
Nozzle Thickness [eb]:	0.4 (inch)
Mean Nozzle Dia. [d]:	19.6 (inch)
Rein. Pad Thk. [e2]:	0 (inch)
Rein. Pad OD [d2]:	0 (inch)
Shell + Rein. Pad Thk. [ec]:	0.75 (inch)
Shell Design Stress [f]:	23206 (psi)
Nozzle Design Stress [fb]:	23206 (psi)
Rein. Pad Design Stress [f2]:	23206 (psi)
Corrosion Allowance [c]:	0.04 (inch)

Allowable Loads on Nozzles attached to Spherical Shells as per EN 13445-3:2009

Clause 16.4.3: Conditions of applicability

Analysis Shell Thk.[ea]/Mean Shell Radius [R] should be >= 0.001 and <= 0.1

a) ea/R = 0.005 which is >= 0.001 and <= 0.1. Condition Passed.
b) Distances to any other local load in any direction shall not be less than SQRT(R.ec) = 10.32 (inch)
c) Nozzle thickness shall be maintained over a distance of SQRT(D.eb) = 2.8 (inch)

Clause 16.4.5: Maximum allowable individual loads

Allowable radial nozzle load [Fz.Max]:84925.42 (lb)

Allowable bending moment [Mb.Max]:45480.20 (ft-lb)

Version 8.10 — SphericalVessel_Allowable.noz — Aug 22,2018

Caepipe : Print Preview, Page 1 (of 2)

File View Options Help

Print | Prev Page | Next Page | Close

Caepipe — Page 1

Local Shell Stresses at Nozzles

Input Data:
Local Shell Stresses at Nozzles attached to Cylindrical Shells
Load Case : Operating

Radial Load [P]:	137 (lb)
Shear Load [VC]:	332 (lb)
Shear Load [VL]:	-847 (lb)
Circumferential Moment [MC]:	828 (ft-lb)
Longitudinal Moment [ML]:	3986 (ft-lb)
Torsional Moment [MT]:	-17117 (ft-lb)
Vessel Thickness [T]:	0.35 (inch)
Vessel Radius: [Rm]:	31.5 (inch)
Attachment Radius [ro]:	11 (inch)
Fillet Radius [r]:	1 (inch)
Pressure Stress at Shell [Pm]:	1343 (psi)
Bending Stress at Shell [Pb]:	20246 (psi)
Operating Allowable [All]:	106200 (psi)
Stress Conc. Factor - Tension [Kn]:	0.00
Stress Conc. Factor - Bending [Kb]:	0.00

Stress Concentration Factor(s) Computed as per Fig. B-2

Tension [Kn]:	1.36
Bending [Kb]:	1.14

Calculation of Local Stresses in Cylindrical Shells as per WRC Bulletin 537 (psi)

Stresses	Fig. No	Au	Al	Bu	Bl	Cu	Cl	Du	Dl
Circumferential Membrane (P)	3C	-46	-45	-45	-46	-45	-46	-46	-45
Circumferential Bending (P)	2C-1	-53	53	-53	53	0	0	0	0
Circumferential Bending (P)	1C	0	0	0	0	-230	230	-230	230
Circumferential Membrane (MC)	3A	0	0	0	0	-293	-293	293	293
Circumferential Bending (MC)	1A	0	0	0	0	-3113	3113	3113	-3113
Circumferential Membrane (ML)	3B	-2844	-2844	2844	2844	0	0	0	0
Circumferential Bending (ML)	1B-1	-2068	2068	2068	-2068	0	0	0	0
Circumferential Stresses (Sp)	-	-5010	-768	4815	784	-3680	3005	3130	-2635
Longitudinal Membrane (P)	4C	-134	-134	-134	-134	-134	-134	-134	-134
Longitudinal Bending (P)	1C-1	-153	153	-153	153	0	0	0	0

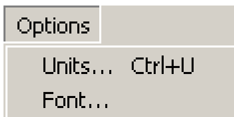
Version 8.10 — 441_31_Operating.noz — Sep 3,2018

Edit Menu

.Edit.

You can edit the data by clicking the Edit command.

Options Menu



.Units.

See Units in the Layout Window Options Menu section of the CAEPIPE User's Manual.

.Font.

See Font in the Layout Window Options Menu section of the CAEPIPE User's Manual.

Pressure Design of Pipe and Pipe Fittings according to EN 13480-3 (2017)

Pressure Design of Pipe and Pipe Fittings can be performed using the modules built into CAEPIPE which are independent of the piping flexibility analysis.

These modules can be launched through Layout frame > Misc > Internal Pressure Design: EN 13480-3 and Layout frame > Misc > External Pressure Design: EN 13480-3.

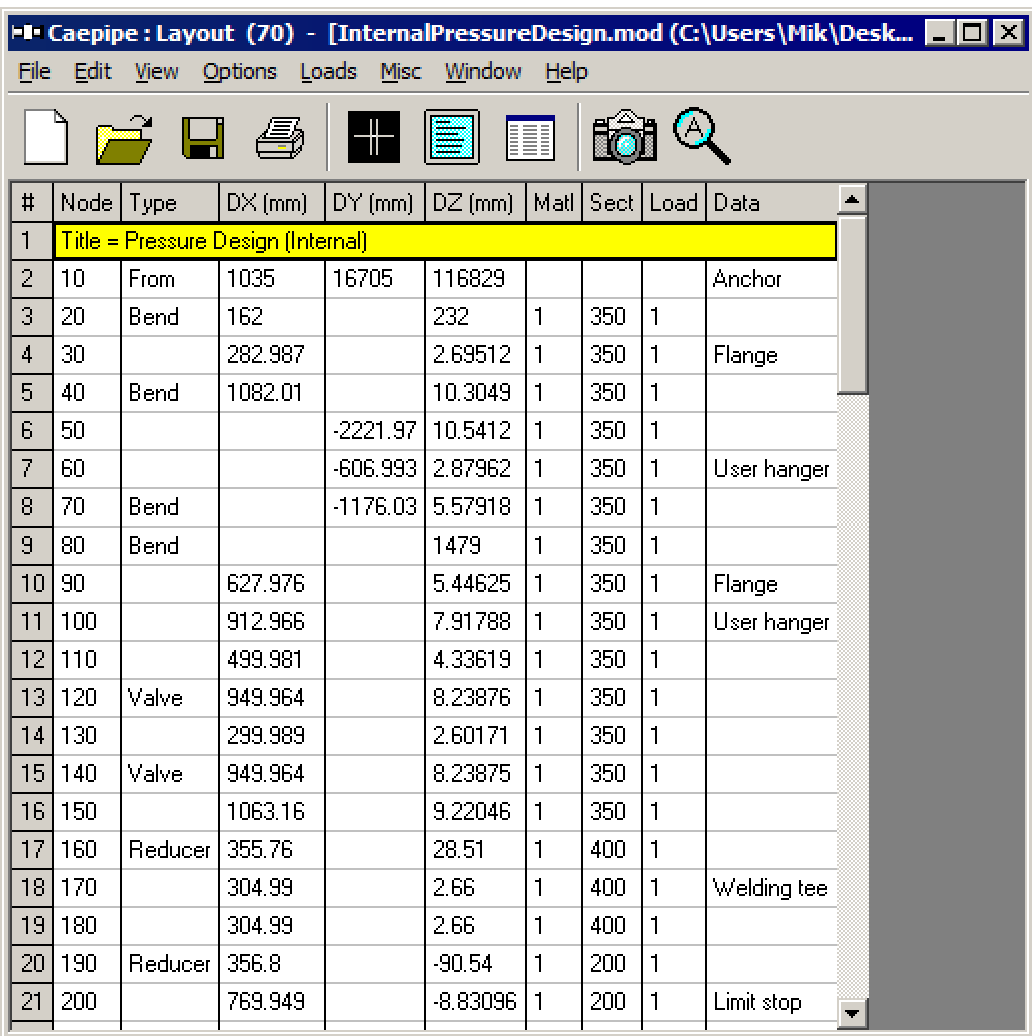
Note:

These modules perform Pressure Design of Pipe and Pipe Fittings ONLY using the equations given in the EN 13480-3 (2017) Code irrespective of the Analysis Code selected for piping flexibility analysis in CAEPIPE.

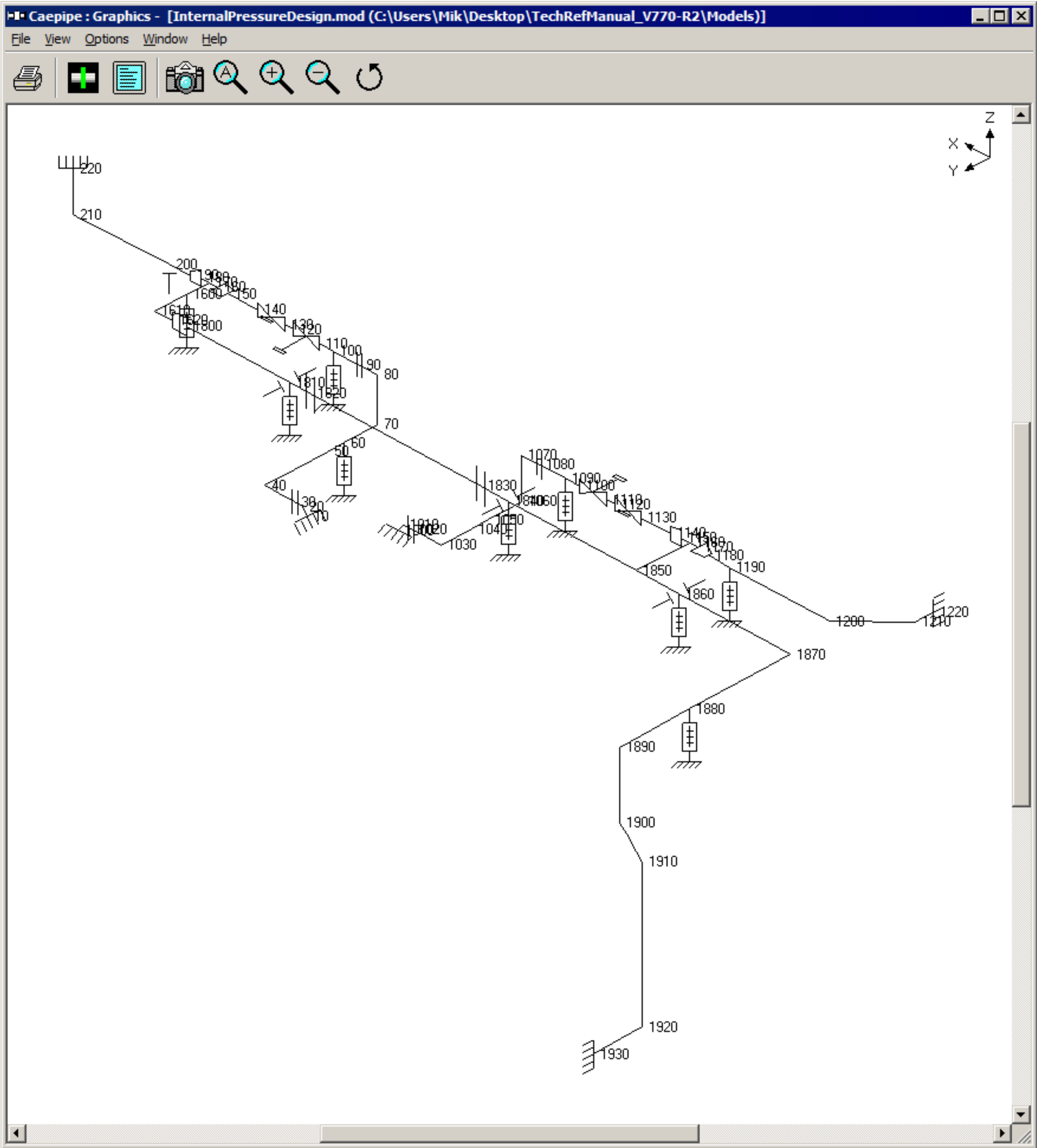
In case the pipe stress analysis is performed with an Analysis Code other than EN 13480-3 (2017), the Pressure Design modules will use the material allowable stresses corresponding to the maximum temperature T1 through T10 entered in the CAEPIPE stress model.

Internal Pressure Design of Pipe and Pipe Fittings

Snap shots shown below present a sample stress model developed to show the Internal Pressure Design calculations performed by CAEPIPE.



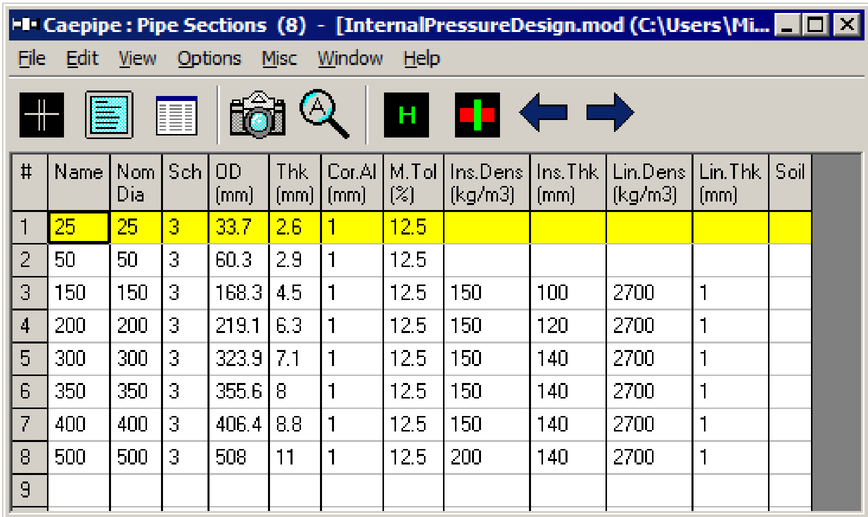
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
1	Title = Pressure Design (Internal)								
2	10	From	1035	16705	116829				Anchor
3	20	Bend	162		232	1	350	1	
4	30		282.987		2.69512	1	350	1	Flange
5	40	Bend	1082.01		10.3049	1	350	1	
6	50			-2221.97	10.5412	1	350	1	
7	60			-606.993	2.87962	1	350	1	User hanger
8	70	Bend		-1176.03	5.57918	1	350	1	
9	80	Bend			1479	1	350	1	
10	90		627.976		5.44625	1	350	1	Flange
11	100		912.966		7.91788	1	350	1	User hanger
12	110		499.981		4.33619	1	350	1	
13	120	Valve	949.964		8.23876	1	350	1	
14	130		299.989		2.60171	1	350	1	
15	140	Valve	949.964		8.23875	1	350	1	
16	150		1063.16		9.22046	1	350	1	
17	160	Reducer	355.76		28.51	1	400	1	
18	170		304.99		2.66	1	400	1	Welding tee
19	180		304.99		2.66	1	400	1	
20	190	Reducer	356.8		-90.54	1	200	1	
21	200		769.949		-8.83096	1	200	1	Limit stop



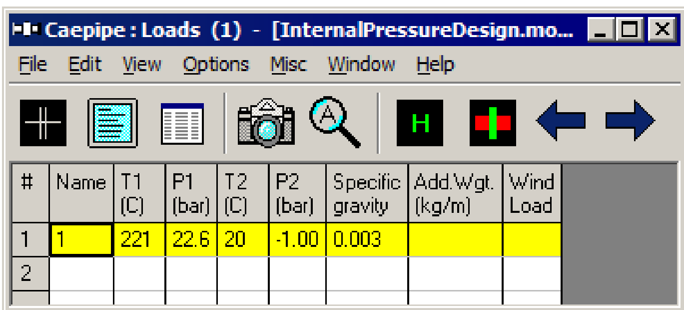
Caepipe : Materials (1) - [InternalPressureDesign.mod (C:\Users\Mik\Desktop\TechRefManual_V770-R2\Mod...

File Edit View Options Misc Window Help

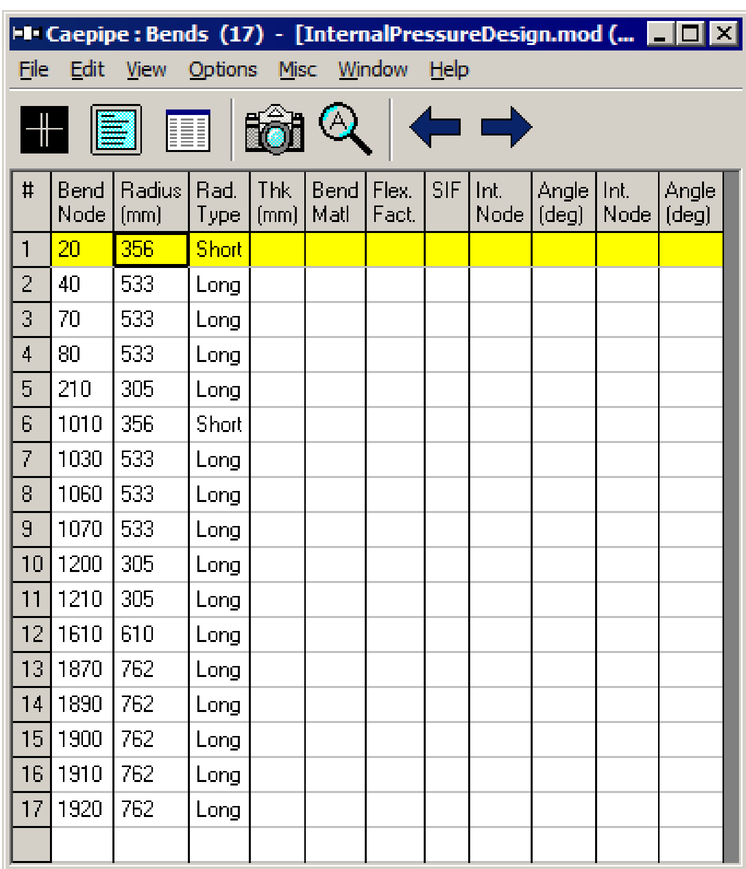
#	Name	Description	Type	Density (kg/m3)	Nu	Joint factor	Tensile (N/mm2)	#	Temp (C)	E (kN/mm2)	Alpha (mm/mm/C)	f (N/mm2)	fCR (N/mm2)
1	1	EN 10216-2 10CrMo9-10 Seamless	CS	7850	0.3	0.80	229.8	1	20	211	11.51E-6	137.1	
2								2	50	209	11.78E-6	136.9	
								3	100	206	12.10E-6	136.5	
								4	150	203	12.43E-6	132.8	
								5	200	199	12.75E-6	132.8	
								6	250	196	13.08E-6	132.8	
								7	300	192	13.22E-6	132.8	
								8					



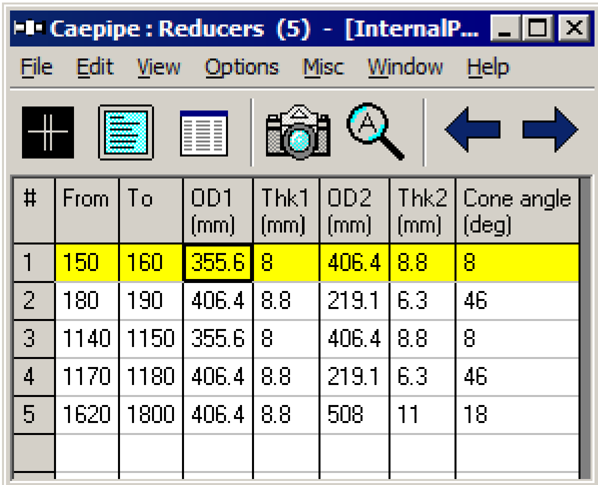
#	Name	Nom Dia	Sch	OD (mm)	Thk (mm)	Cor.Al (mm)	M.Tol (%)	Ins.Dens (kg/m3)	Ins.Thk (mm)	Lin.Dens (kg/m3)	Lin.Thk (mm)	Soil
1	25	25	3	33.7	2.6	1	12.5					
2	50	50	3	60.3	2.9	1	12.5					
3	150	150	3	168.3	4.5	1	12.5	150	100	2700	1	
4	200	200	3	219.1	6.3	1	12.5	150	120	2700	1	
5	300	300	3	323.9	7.1	1	12.5	150	140	2700	1	
6	350	350	3	355.6	8	1	12.5	150	140	2700	1	
7	400	400	3	406.4	8.8	1	12.5	150	140	2700	1	
8	500	500	3	508	11	1	12.5	200	140	2700	1	
9												



#	Name	T1 (C)	P1 (bar)	T2 (C)	P2 (bar)	Specific gravity	Add.Wgt. (kg/m)	Wind Load
1	1	221	22.6	20	-1.00	0.003		
2								



#	Bend Node	Radius (mm)	Rad. Type	Thk (mm)	Bend Matl	Flex. Fact.	SIF	Int. Node	Angle (deg)	Int. Node	Angle (deg)
1	20	356	Short								
2	40	533	Long								
3	70	533	Long								
4	80	533	Long								
5	210	305	Long								
6	1010	356	Short								
7	1030	533	Long								
8	1060	533	Long								
9	1070	533	Long								
10	1200	305	Long								
11	1210	305	Long								
12	1610	610	Long								
13	1870	762	Long								
14	1890	762	Long								
15	1900	762	Long								
16	1910	762	Long								
17	1920	762	Long								



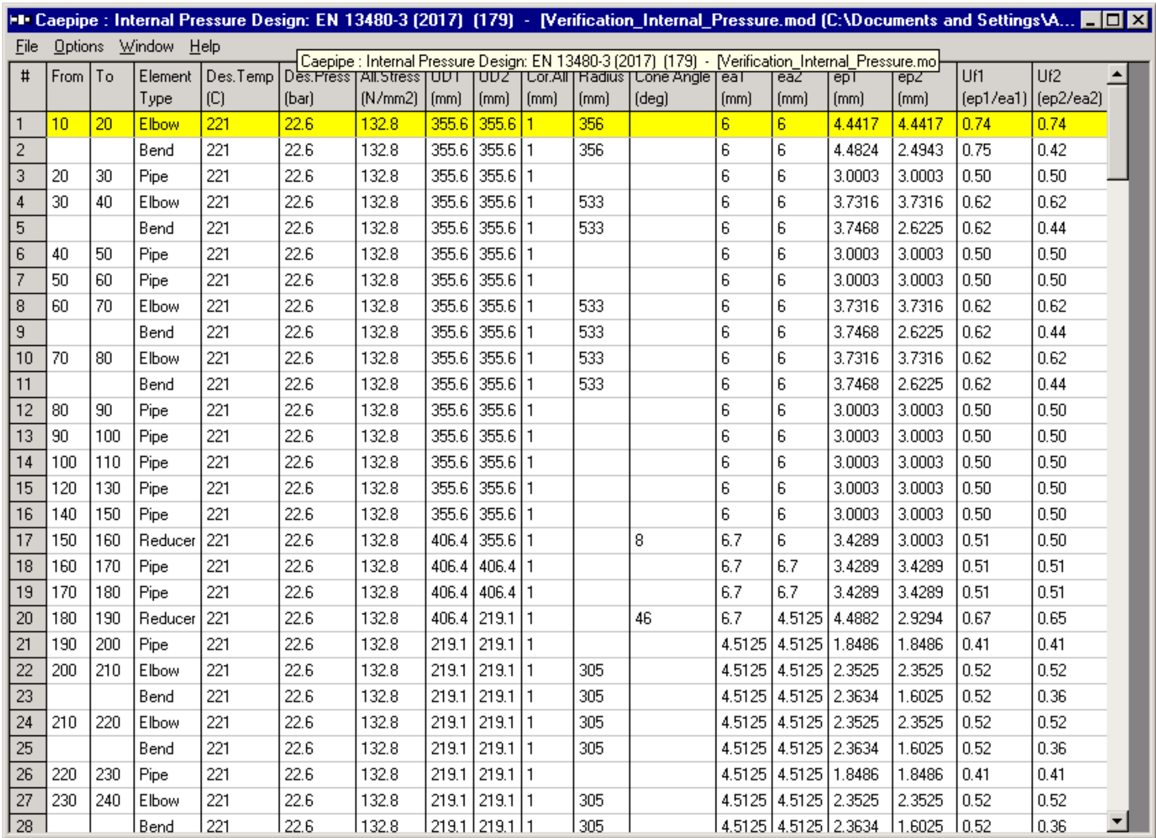
#	From	To	OD1 (mm)	Thk1 (mm)	OD2 (mm)	Thk2 (mm)	Cone angle (deg)
1	150	160	355.6	8	406.4	8.8	8
2	180	190	406.4	8.8	219.1	6.3	46
3	1140	1150	355.6	8	406.4	8.8	8
4	1170	1180	406.4	8.8	219.1	6.3	46
5	1620	1800	406.4	8.8	508	11	18

Internal pressure design calculations of pipe and pipe fittings according to EN 13480-3 (2017) are independent of element lengths entered. Hence, these calculations can be performed from the CAEPIPE model already developed for flexibility analysis. Equations used for performing Internal Pressure Design as per EN 13480-3 (2017) are provided in Section titled "Pressure Design of Pipe & Pipe Fittings" in the Code Compliance Manual.

Once the layout of the stress model as shown in the above snap shots is completed, the internal pressure design is performed through Layout window > Misc > Internal Pressure Design: EN 13480-3.

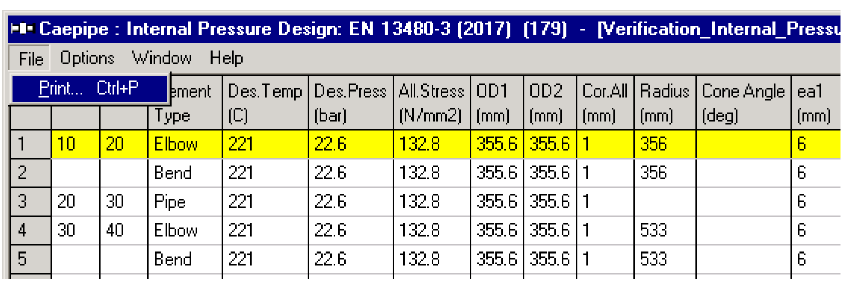
When executed, CAEPIPE automatically performs the pressure design calculations for Pipes, Elbows, Miters, Bends and Reducers for the entire stress model and displays the results as shown below.

It is observed that the ratios Uf1 and Uf2 are all less than 1.0, confirming that the Internal Pressure Design requirements of EN 13480-3 (2017) code are met for this stress model.

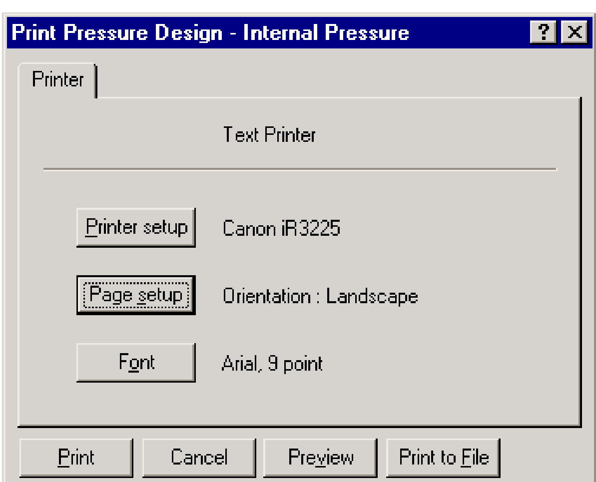


#	From	To	Element Type	Des.Temp (C)	Des.Press (bar)	All.Stress (N/mm2)	OD1 (mm)	OD2 (mm)	Cor.All (mm)	Radius (mm)	Cone Angle (deg)	ea1 (mm)	ea2 (mm)	ep1 (mm)	ep2 (mm)	Uf1 (ep1/ea1)	Uf2 (ep2/ea2)
1	10	20	Elbow	221	22.6	132.8	355.6	355.6	1	356		6	6	4.4417	4.4417	0.74	0.74
2			Bend	221	22.6	132.8	355.6	355.6	1	356		6	6	4.4824	2.4943	0.75	0.42
3	20	30	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
4	30	40	Elbow	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7316	3.7316	0.62	0.62
5			Bend	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7468	2.6225	0.62	0.44
6	40	50	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
7	50	60	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
8	60	70	Elbow	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7316	3.7316	0.62	0.62
9			Bend	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7468	2.6225	0.62	0.44
10	70	80	Elbow	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7316	3.7316	0.62	0.62
11			Bend	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7468	2.6225	0.62	0.44
12	80	90	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
13	90	100	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
14	100	110	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
15	120	130	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
16	140	150	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
17	150	160	Reducer	221	22.6	132.8	406.4	355.6	1		8	6.7	6	3.4289	3.0003	0.51	0.50
18	160	170	Pipe	221	22.6	132.8	406.4	406.4	1			6.7	6.7	3.4289	3.4289	0.51	0.51
19	170	180	Pipe	221	22.6	132.8	406.4	406.4	1			6.7	6.7	3.4289	3.4289	0.51	0.51
20	180	190	Reducer	221	22.6	132.8	406.4	219.1	1		46	6.7	4.5125	4.4882	2.9294	0.67	0.65
21	190	200	Pipe	221	22.6	132.8	219.1	219.1	1			4.5125	4.5125	1.8486	1.8486	0.41	0.41
22	200	210	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
23			Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36
24	210	220	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
25			Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36
26	220	230	Pipe	221	22.6	132.8	219.1	219.1	1			4.5125	4.5125	1.8486	1.8486	0.41	0.41
27	230	240	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
28			Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36

The results shown above can also be printed to the printer or to a file using the option File > Print.



#	From	To	...ement Type	Des.Temp (C)	Des.Press (bar)	All.Stress (N/mm2)	OD1 (mm)	OD2 (mm)	Cor.All (mm)	Radius (mm)	Cone Angle (deg)	ea1 (mm)
1	10	20	Elbow	221	22.6	132.8	355.6	355.6	1	356		6
2			Bend	221	22.6	132.8	355.6	355.6	1	356		6
3	20	30	Pipe	221	22.6	132.8	355.6	355.6	1			6
4	30	40	Elbow	221	22.6	132.8	355.6	355.6	1	533		6
5			Bend	221	22.6	132.8	355.6	355.6	1	533		6



Pressure Design of Pipe and Pipe fittings

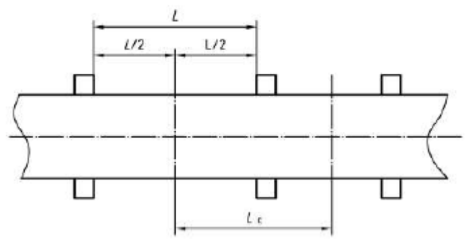
Internal Pressure Design: EN 13480-3 (2017) (179)																
From	To	Element Type	Des.Temp (C)	Des.Press (bar)	All.Stress (N/mm2)	OD1 (mm)	OD2 (mm)	Cor.All (mm)	Radius (mm)	Cone Angle (deg)	ea1 (mm)	ea2 (mm)	ep1 (mm)	ep2 (mm)	Uf1 (ep1/ea1)	Uf2 (ep2/ea2)
10	20	Elbow	221	22.6	132.8	355.6	355.6	1	356		6	6	4.4417	4.4417	0.74	0.74
		Bend	221	22.6	132.8	355.6	355.6	1	356		6	6	4.4824	2.4943	0.75	0.42
20	30	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
30	40	Elbow	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7316	3.7316	0.62	0.62
		Bend	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7468	2.6225	0.62	0.44
40	50	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
50	60	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
60	70	Elbow	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7316	3.7316	0.62	0.62
		Bend	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7468	2.6225	0.62	0.44
70	80	Elbow	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7316	3.7316	0.62	0.62
		Bend	221	22.6	132.8	355.6	355.6	1	533		6	6	3.7468	2.6225	0.62	0.44
80	90	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
90	100	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
100	110	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
120	130	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
140	150	Pipe	221	22.6	132.8	355.6	355.6	1			6	6	3.0003	3.0003	0.50	0.50
150	160	Reducer	221	22.6	132.8	406.4	355.6	1		8	6.7	6	3.4289	3.0003	0.51	0.50
160	170	Pipe	221	22.6	132.8	406.4	406.4	1			6.7	6.7	3.4289	3.4289	0.51	0.51
170	180	Pipe	221	22.6	132.8	406.4	406.4	1			6.7	6.7	3.4289	3.4289	0.51	0.51
180	190	Reducer	221	22.6	132.8	406.4	219.1	1		46	6.7	4.5125	4.4882	2.9294	0.67	0.65
190	200	Pipe	221	22.6	132.8	219.1	219.1	1			4.5125	4.5125	1.8486	1.8486	0.41	0.41
200	210	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
		Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36
210	220	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
		Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36
220	230	Pipe	221	22.6	132.8	219.1	219.1	1			4.5125	4.5125	1.8486	1.8486	0.41	0.41
230	240	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
		Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36
240	250	Elbow	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3525	2.3525	0.52	0.52
		Bend	221	22.6	132.8	219.1	219.1	1	305		4.5125	4.5125	2.3634	1.6025	0.52	0.36
250	260	Pipe	221	22.6	132.8	219.1	219.1	1			4.5125	4.5125	1.8486	1.8486	0.41	0.41

External Pressure Design of Pipe and Pipe Fittings

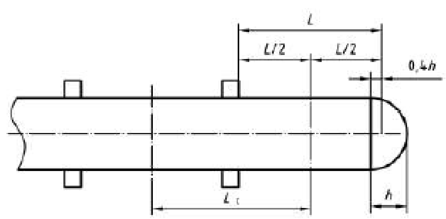
External Pressure Design module will function ONLY when the stress layout is defined with negative pressure (such as vacuum pressure).

This module first calculates collapse pressure (same as buckling pressure), which is a function of span length "L" between the stiffeners placed on the piping (shown in figures below). Since the collapse (buckling) mode of deformation for a pipe element between two adjacent stiffeners is restrained by these stiffeners, shorter the span length L between the stiffeners, higher the collapse (buckling) pressure.

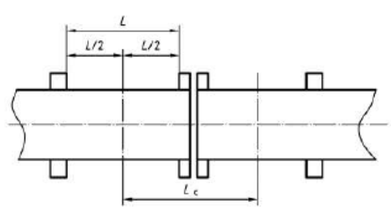
The External Pressure Design module assumes that a stiffener is located at each node of the CAEPIPE model. Hence, ensure that nodes are defined in CAEPIPE model only at locations where the stiffeners are attached to the piping. Even nodes where flanges or certain types of supports that restrain the collapse (buckling) mode of deformation should be included as "stiffener locations". All other nodes at which the collapse (buckling) mode of deformation is not restrained (such as resting supports) should not be included in the CAEPIPE model for external pressure design calculations. In other words, the CAEPIPE stress model (that was developed for flexibility analysis) needs to be edited before performing the external pressure design.



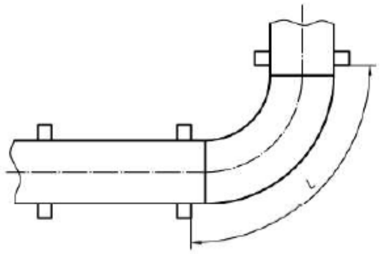
Single Pipe



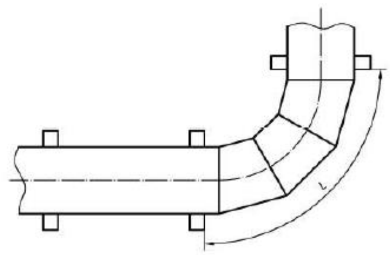
Pipe with bend



Pipe with flange connections



Pipe with bend or elbow with 'L' measured on extrados



Pipe with mitre with 'L' measured on extrados

The procedure given below will help in retaining ONLY those nodes of the CAEPIPE stress model (originally developed for flexibility analysis) prior to External Pressure Design calculations.

- Create a copy of the existing CAEPIPE stress model (developed for flexibility analysis).
- At whichever node the collapse (buckling) mode of deformation is NOT restrained, navigate to that element node in the layout window and use the option "Combine…" through Layout window > Edit. This action will remove that node by combining the two adjacent elements.
- Repeat the above step and remove all other nodes where the collapse (buckling) mode of deformation is NOT restrained, thereby retaining ONLY the stiffeners, flanges and supportsthat restrain the collapse (buckling) mode].
- Upon completion, save the model.

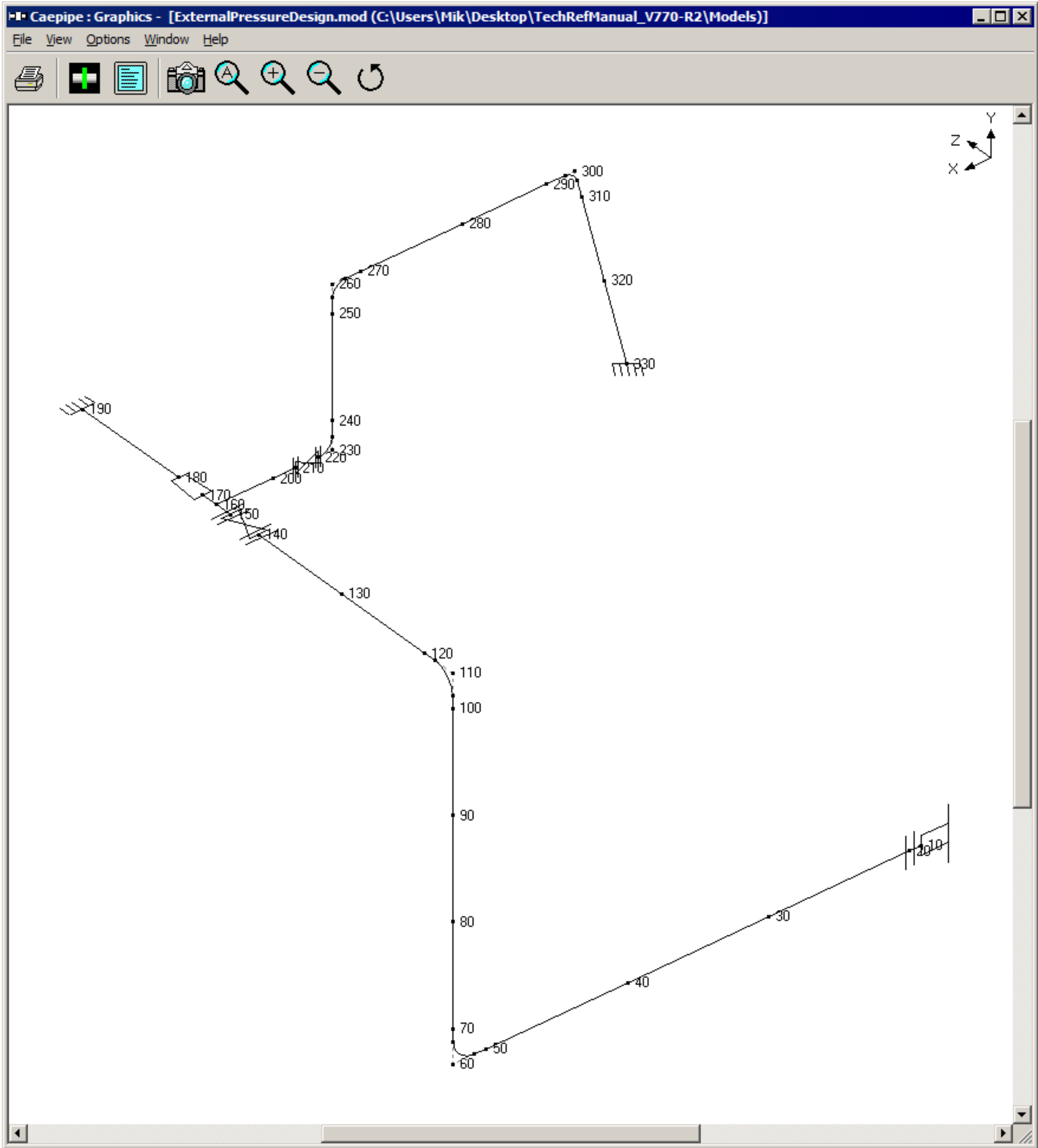
Snap shots shown below present a sample model developed to show the External Pressure Design calculations performed by CAEPIPE. As stated above, a copy of the original stress model was made and the model has been edited to include only those nodes on pipe where stiffeners, flanges and supports (that are equivalent to stiffeners from the point of view of restraining collapse mode of deformation) are attached.

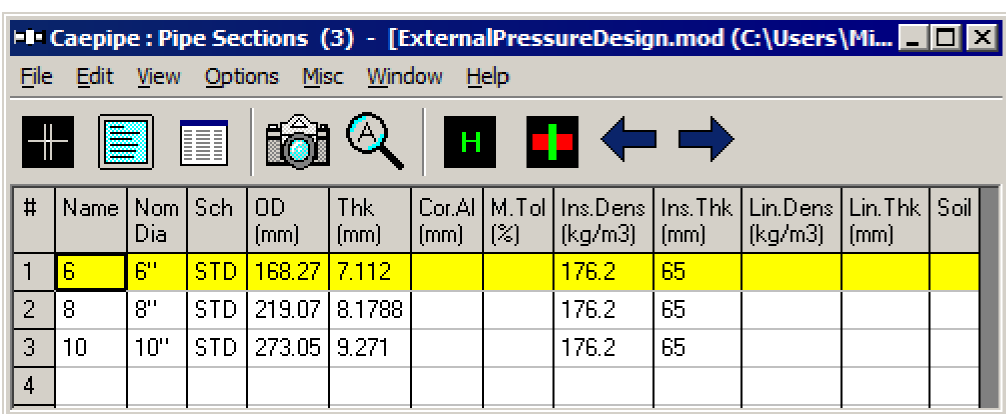
Refer Section titled "External Pressure Design according to SS EN 13480-3" in CAEPIPE Code compliance manual for details on implementation.

Caepipe : Layout (37) - [ExternalPressureDesign.mod (C:\...

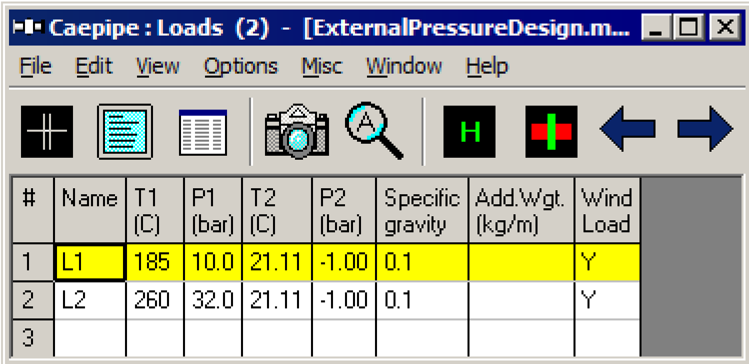
File Edit View Options Loads Misc Window Help

#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
1	Title = Sample Problem 2								
2	10	From							Nozzle
3	20		200			312	10	L1	Flange
4	30		2500			312	10	L1	
5	40		2500			312	10	L1	
6	50		2500			312	10	L1	
7	60	Bend	600			312	10	L1	
8	70			600		312	10	L1	
9	80			1800		312	10	L1	
10	90			1800		312	10	L1	
11	100			1800		312	10	L1	
12	110	Bend		600		312	10	L1	
13	120				600	312	10	L1	
14	130				1820	312	10	L1	
15	140				1820	312	10	L1	Flange
16	150	Valve			622.3	312	10	L1	Flange
17	160				300	312	10	L1	Welding tee
18	170				300	312	10	L1	
19	180	Reducer			530	312	8	L1	
20	190				2100	312	8	L1	Anchor
21	6" Branch								
22	160	From							
23	200		-1000			312	6	L1	
24	210		-400			312	6	L1	Flange
25	220	Valve	-403.23			312	6	L1	Flange
26	Change in Temperature and Pressure								
27	230	Bend	-255			312	6	L2	
28	240			500		312	6	L2	
29	250			1800		312	6	L2	
30	260	Bend		500		312	6	L2	
31	270		-500			312	6	L2	
32	280		-1800			312	6	L2	
33	290		-1490			312	6	L2	
34	300	Bend	-500			312	6	L2	
35	310		-120.644	-485.227		312	6	L2	
36	320		-394.678	-1587.39		312	6	L2	
37	330		-394.678	-1587.39		312	6	L2	Anchor
38									

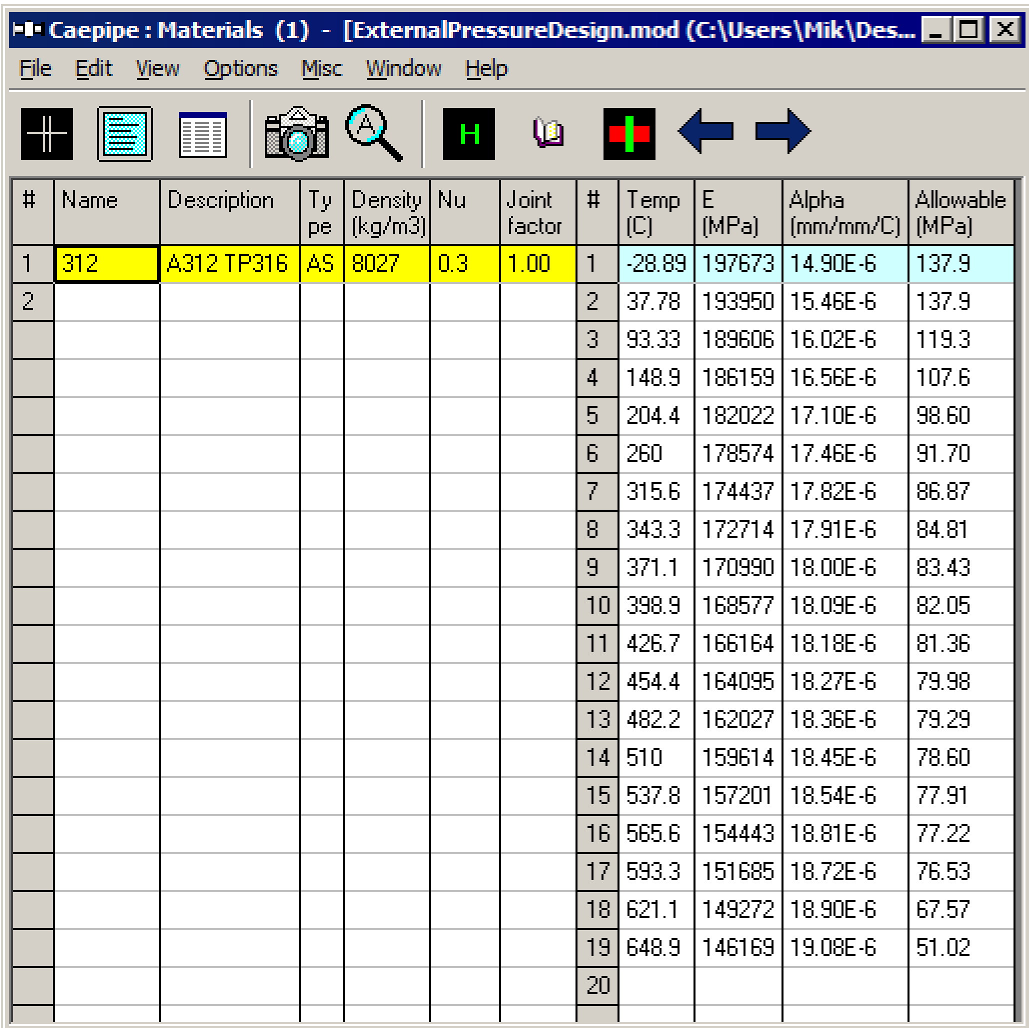




#	Name	Nom Dia	Sch	OD (mm)	Thk (mm)	Cor.Al (mm)	M.Tol (%)	Ins.Dens (kg/m3)	Ins.Thk (mm)	Lin.Dens (kg/m3)	Lin.Thk (mm)	Soil
1	6	6"	STD	168.27	7.112			176.2	65			
2	8	8"	STD	219.07	8.1788			176.2	65			
3	10	10"	STD	273.05	9.271			176.2	65			
4												



#	Name	T1 (C)	P1 (bar)	T2 (C)	P2 (bar)	Specific gravity	Add.Wgt. (kg/m)	Wind Load
1	L1	185	10.0	21.11	-1.00	0.1		Y
2	L2	260	32.0	21.11	-1.00	0.1		Y
3								



#	Name	Description	Type	Density (kg/m3)	Nu	Joint factor
1	312	A312 TP316	AS	8027	0.3	1.00
2						

#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	Allowable (MPa)
1	-28.89	197673	14.90E-6	137.9
2	37.78	193950	15.46E-6	137.9
3	93.33	189606	16.02E-6	119.3
4	148.9	186159	16.56E-6	107.6
5	204.4	182022	17.10E-6	98.60
6	260	178574	17.46E-6	91.70
7	315.6	174437	17.82E-6	86.87
8	343.3	172714	17.91E-6	84.81
9	371.1	170990	18.00E-6	83.43
10	398.9	168577	18.09E-6	82.05
11	426.7	166164	18.18E-6	81.36
12	454.4	164095	18.27E-6	79.98
13	482.2	162027	18.36E-6	79.29
14	510	159614	18.45E-6	78.60
15	537.8	157201	18.54E-6	77.91
16	565.6	154443	18.81E-6	77.22
17	593.3	151685	18.72E-6	76.53
18	621.1	149272	18.90E-6	67.57
19	648.9	146169	19.08E-6	51.02
20				

Caepipe : Bends (5) - [ExternalPressureDesign.mod (C:...

File Edit View Options Misc Window Help

#	Bend Node	Radius (mm)	Rad. Type	Thk (mm)	Bend Matl	Flex. Fact.	SIF	Int. Node	Angle (deg)	Int. Node	Angle (deg)
1	60	381	Long								
2	110	381	Long								
3	230	228.6	Long								
4	260	228.6	Long								
5	300	228.6	Long								

Caepipe : Reducers (1) - [ExternalPress...

File Edit View Options Misc Window Help

#	From	To	OD1 (mm)	Thk1 (mm)	OD2 (mm)	Thk2 (mm)	Cone angle (deg)
1	170	180	273.05	9.271	219.07	8.1788	

Once the layout of the stress model as shown in the above snap shots is completed, the external pressure design is performed through Layout window > Misc > External Pressure Design: EN 13480-3.

When executed, CAEPIPE automatically performs the external pressure design calculations for Pipes, Miters, Elbows, Bends and Reducers for the entire stress model and displays the results as shown below.

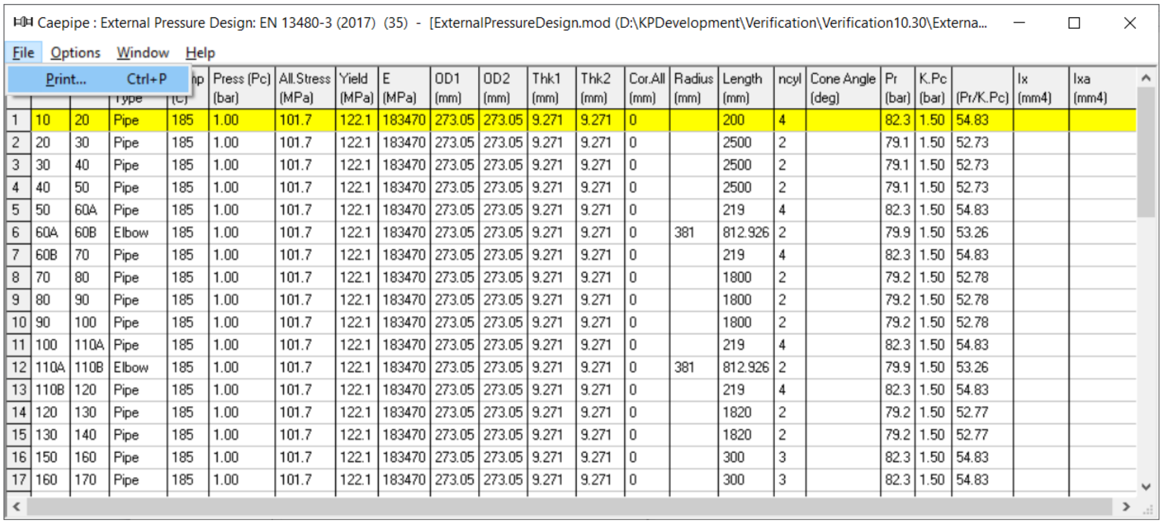
It is observed that the ratio [Pr/(KPc)] is much higher than 1.0 throughout the stress model, confirming that the collapse (buckling) pressures Pr calculated for all segments of the stress model are much higher than the corresponding peak negative pressures specified in the CAEPIPE model. In other words, the potential for any segment of this piping system to collapse (buckle) is very minimal.

Caepipe : External Pressure Design: EN 13480-3 (2017) (35) - [ExternalPressureDesign.mod (D:\KPDevelopment\Verification\Verification10.30\Externa...

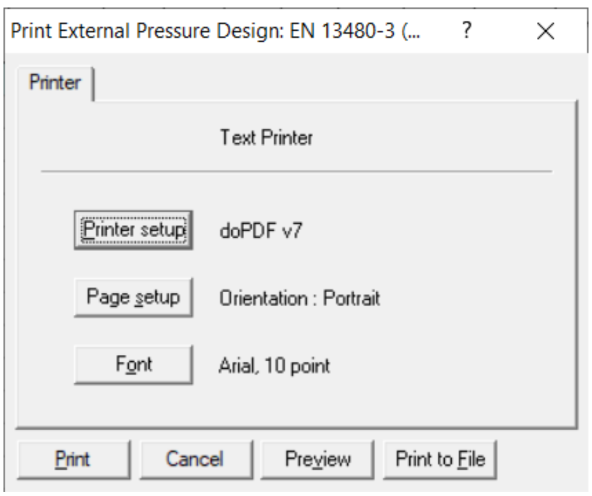
File Options Window Help

#	From	To	Element Type	Temp (C)	Press (Pc) (bar)	All.Stress (MPa)	Yield (MPa)	E (MPa)	OD1 (mm)	OD2 (mm)	Thk1 (mm)	Thk2 (mm)	Cor.All (mm)	Radius (mm)	Length (mm)	ncyl	Cone Angle (deg)	Pr (bar)	K.Pc (bar)	(Pr/K.Pc)	Ix (mm4)	Ixa (mm4)
1	10	20	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		200	4		82.3	1.50	54.83		
2	20	30	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		2500	2		79.1	1.50	52.73		
3	30	40	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		2500	2		79.1	1.50	52.73		
4	40	50	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		2500	2		79.1	1.50	52.73		
5	50	60A	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
6	60A	60B	Elbow	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0	381	812.926	2		79.9	1.50	53.26		
7	60B	70	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
8	70	80	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1800	2		79.2	1.50	52.78		
9	80	90	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1800	2		79.2	1.50	52.78		
10	90	100	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1800	2		79.2	1.50	52.78		
11	100	110A	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
12	110A	110B	Elbow	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0	381	812.926	2		79.9	1.50	53.26		
13	110B	120	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
14	120	130	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1820	2		79.2	1.50	52.77		
15	130	140	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1820	2		79.2	1.50	52.77		
16	150	160	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		300	3		82.3	1.50	54.83		
17	160	170	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		300	3		82.3	1.50	54.83		

The results shown above can also be printed to the printer or to a file using the option File > Print.



			Type	...p (C)	Press (Pc) (bar)	All.Stress (MPa)	Yield (MPa)	E (MPa)	OD1 (mm)	OD2 (mm)	Thk1 (mm)	Thk2 (mm)	Cor.All (mm)	Radius (mm)	Length (mm)	ncyl	Cone Angle (deg)	Pr (bar)	K.Pc (bar)	(Pr/K.Pc)	Ix (mm4)	Ixa (mm4)
1	10	20	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		200	4		82.3	1.50	54.83		
2	20	30	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		2500	2		79.1	1.50	52.73		
3	30	40	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		2500	2		79.1	1.50	52.73		
4	40	50	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		2500	2		79.1	1.50	52.73		
5	50	60A	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
6	60A	60B	Elbow	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0	381	812.926	2		79.9	1.50	53.26		
7	60B	70	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
8	70	80	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1800	2		79.2	1.50	52.78		
9	80	90	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1800	2		79.2	1.50	52.78		
10	90	100	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1800	2		79.2	1.50	52.78		
11	100	110A	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
12	110A	110B	Elbow	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0	381	812.926	2		79.9	1.50	53.26		
13	110B	120	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		219	4		82.3	1.50	54.83		
14	120	130	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1820	2		79.2	1.50	52.77		
15	130	140	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		1820	2		79.2	1.50	52.77		
16	150	160	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		300	3		82.3	1.50	54.83		
17	160	170	Pipe	185	1.00	101.7	122.1	183470	273.05	273.05	9.271	9.271	0		300	3		82.3	1.50	54.83		



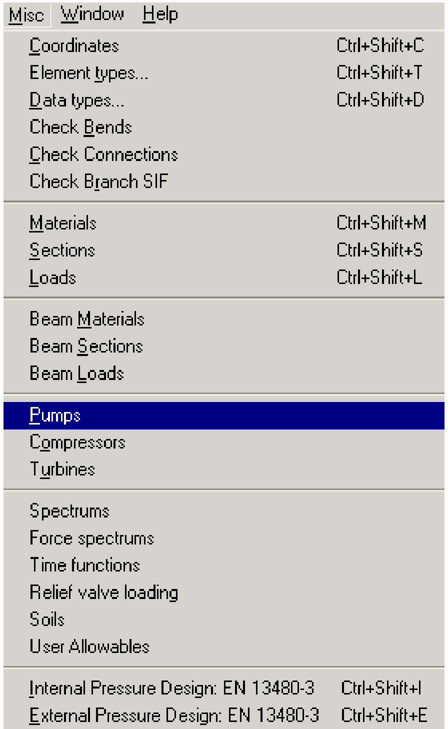
Pumps, compressors and turbines in CAEPIPE, referred to as rotating equipment, are each governed by an industry publication — API (American Petroleum Institute) publishes an API 610 for Horizontal and Vertical inline pumps, ANSI/HI 9.6.2 for Rotodynamic pumps, and API 617 for compressors, while NEMA (National Electrical Manufacturers Association) publishes the NEMA SM-23 for turbines. These publications provide guidelines for evaluating nozzles connected to rotating equipment among other technical information including the items relevant to piping stress analysis – criteria for piping design and a table of allowable loads.

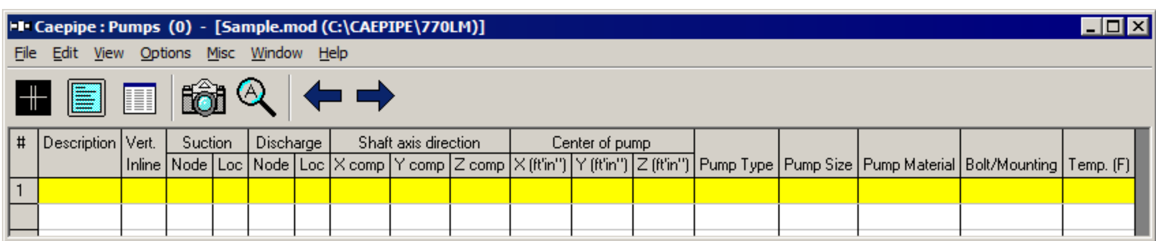
Modeling the rotating equipment is straightforward since it is assumed rigid (relative to connected piping) and modeled only through its end points (connection nozzles).

1. In your model, anchor all of the nozzles (on the equipment) that need to be included in the analysis.
2. Specify these anchored nodes during the respective equipment definition via Misc. menu > Pumps/Compressors/Turbines in the Layout window.

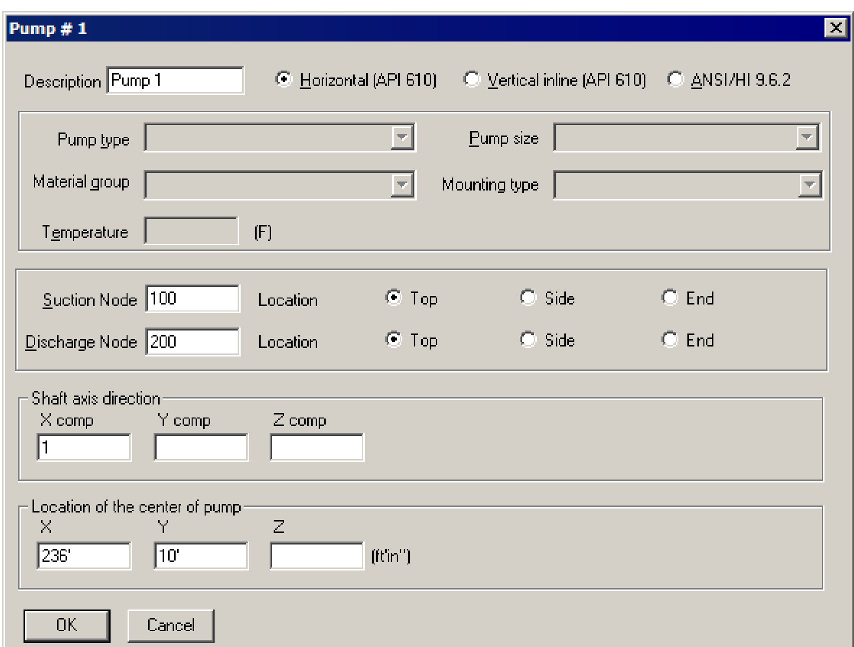
CAEPIPE does not require you to model all of the nozzles nor their connected piping. For example, you may model simply one inlet nozzle of a pump with its piping. Or, you may model one pump with both nozzles (with no connected piping) and impose external forces on them (if you have that data). Further, there is no need to connect the two anchors of the equipment with a rigid massless element like required in some archaic methods. A flange and an anchor may coexist.

A pump is input by selecting "Pumps" from the Misc menu in the Layout or List window. CAEPIPE produces API 610 pump compliance report after analysis for 2 types of pumps, namely Horizontal and Vertical inline. Also, CAEPIPE generates ANSI /HI 9.6.2 compliance report for four types of Rotodynamic pumps: Horizontal or Vertical inline or Axial split case or Vertical turbine short set pumps. *See Section titled "Rotating Equipment Qualification" from the Code Compliance Manual for related information.*





Once you see the Pump List window, double click on an empty row for the Pump dialog and enter the required information.



Type a short description to identify the pump in Description. You must designate the pump nozzles as anchors, and the shaft axis must be in the horizontal plane. The nozzle locations (top, side or end) should be specified for suction and discharge nodes.

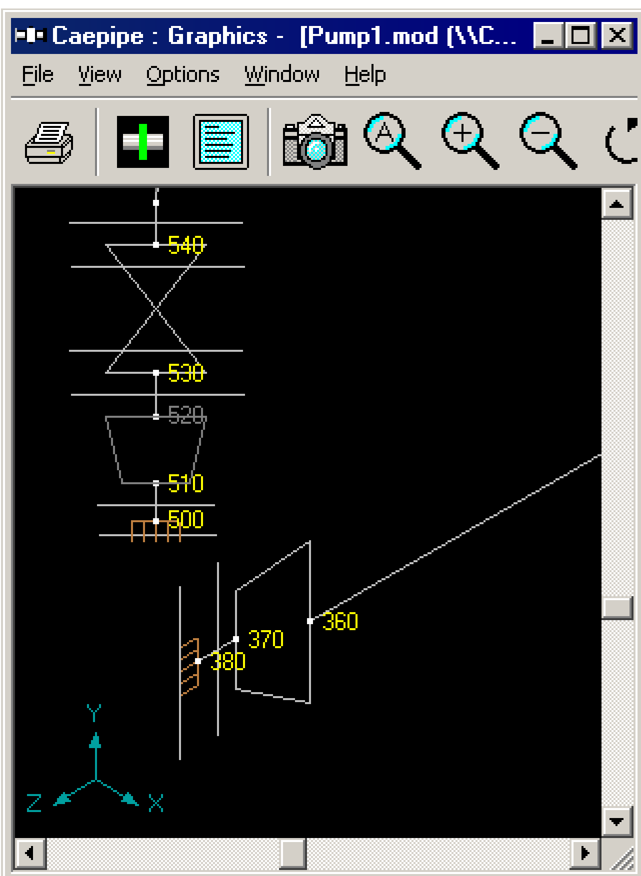
See section on specifying a Direction for information on X comp/Y comp/Z comp.

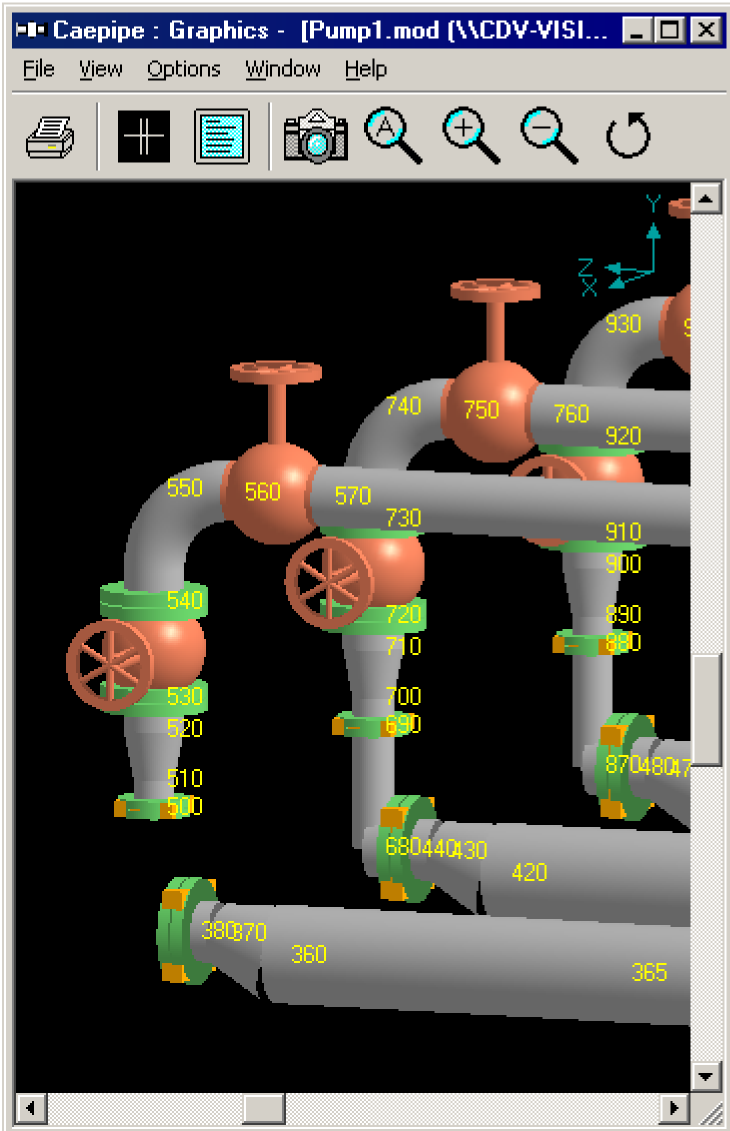
For horizontal pumps, you must enter coordinates for the center of the pump with respect to global origin. For pumps with two support pedestals, API 610 defines the center by the "intersection of the pump shaft centerline and a vertical plane passing through the center of the two support pedestals." For pumps with four support pedestals, the center is defined by the "intersection of the pump shaft centerline and a vertical plane passing midway between the four pedestals." See Section titled "Rotating Equipment Qualification" of the Code Compliance Manual for illustrative figures.

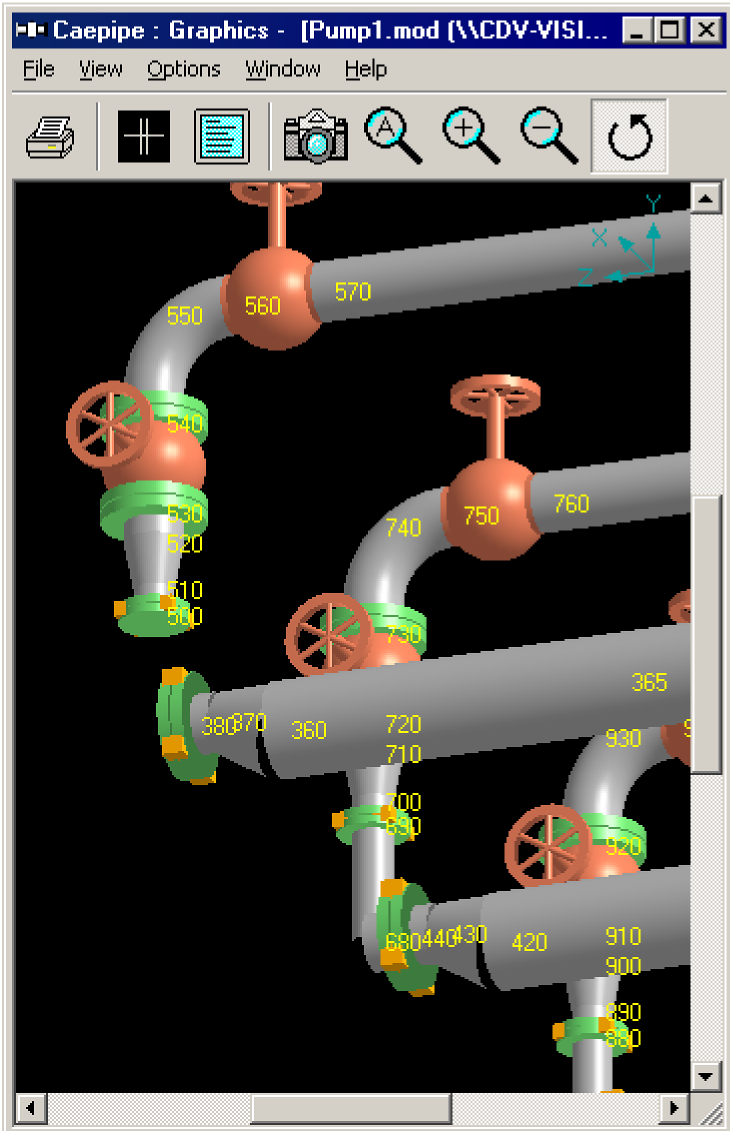
You might find it helpful to first model the nozzles as anchors. In some situations, you might not have the discharge or suction side piping. In that case, here is how you can fix the location of the other side. Look up the coordinates of the nozzle (anchor) you have already modeled.

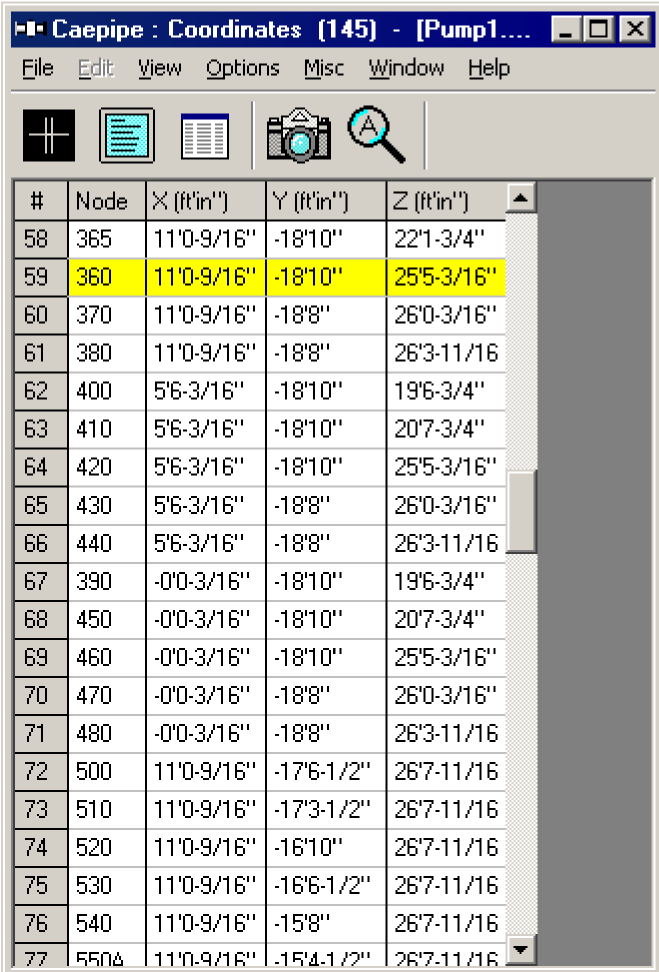
Then, use the coordinates command (menu Misc > Coordinates) to note the (X, Y, Z) coordinates of this pump nozzle node number. Using its coordinates, you can now arrive at the required coordinates of the other side's nozzle and the center of the pump.

Example – API 610 Pump Compliance



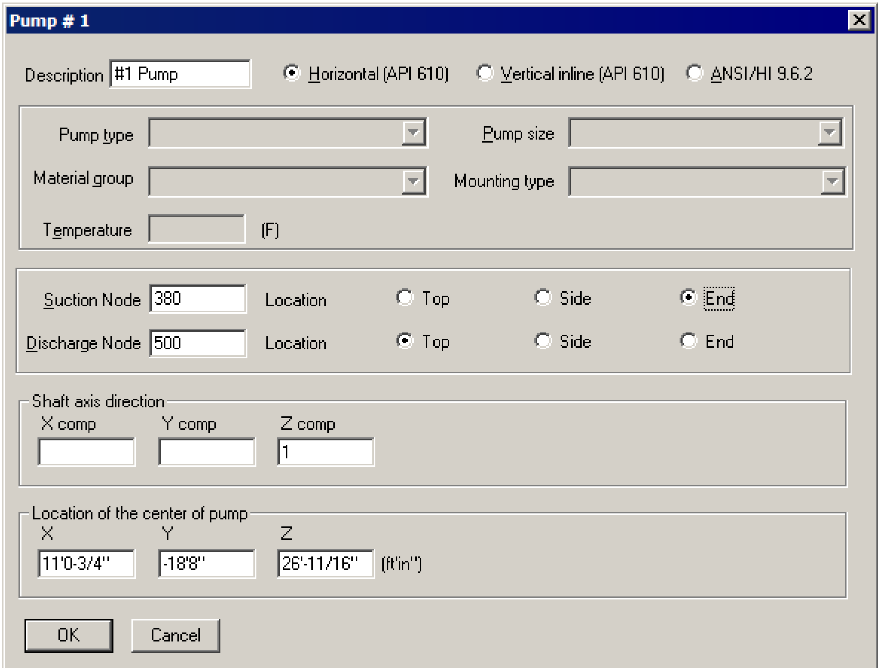

Caepipe : Graphics - [Pump1.mod (\\CDV-VISI...
File View Options Window Help
930
740
750
760
920
550
560
570
730
910
900
540
720
710
890
880
530
520
700
690
510
500
870480
680440430
420
380370
360
365


Caepipe : Graphics - [Pump1.mod (\\CDV-VISI...
File View Options Window Help
550
560
570
540
530
520
510
740
750
760
365
360
720
710
930
700
420
910
900



#	Node	X (ft'in")	Y (ft'in")	Z (ft'in")
58	365	11'0-9/16"	-18'10"	22'1-3/4"
59	360	11'0-9/16"	-18'10"	25'5-3/16"
60	370	11'0-9/16"	-18'8"	26'0-3/16"
61	380	11'0-9/16"	-18'8"	26'3-11/16
62	400	5'6-3/16"	-18'10"	19'6-3/4"
63	410	5'6-3/16"	-18'10"	20'7-3/4"
64	420	5'6-3/16"	-18'10"	25'5-3/16"
65	430	5'6-3/16"	-18'8"	26'0-3/16"
66	440	5'6-3/16"	-18'8"	26'3-11/16
67	390	-0'0-3/16"	-18'10"	19'6-3/4"
68	450	-0'0-3/16"	-18'10"	20'7-3/4"
69	460	-0'0-3/16"	-18'10"	25'5-3/16"
70	470	-0'0-3/16"	-18'8"	26'0-3/16"
71	480	-0'0-3/16"	-18'8"	26'3-11/16
72	500	11'0-9/16"	-17'6-1/2"	26'7-11/16
73	510	11'0-9/16"	-17'3-1/2"	26'7-11/16
74	520	11'0-9/16"	-16'10"	26'7-11/16
75	530	11'0-9/16"	-16'6-1/2"	26'7-11/16
76	540	11'0-9/16"	-15'8"	26'7-11/16
77	550A	11'0-9/16"	-15'4-1/2"	26'7-11/16

In the above image, suction side piping ends at node 380. Discharge piping starts at node 500. In the Pump definition dialog (shown next), you can see that the center of the pump is just behind the suction nozzle (node 380) coordinates. The reducer between nodes 370 and 380 is a vertically offset eccentric reducer; hence the graphics at the reducer shows a break.

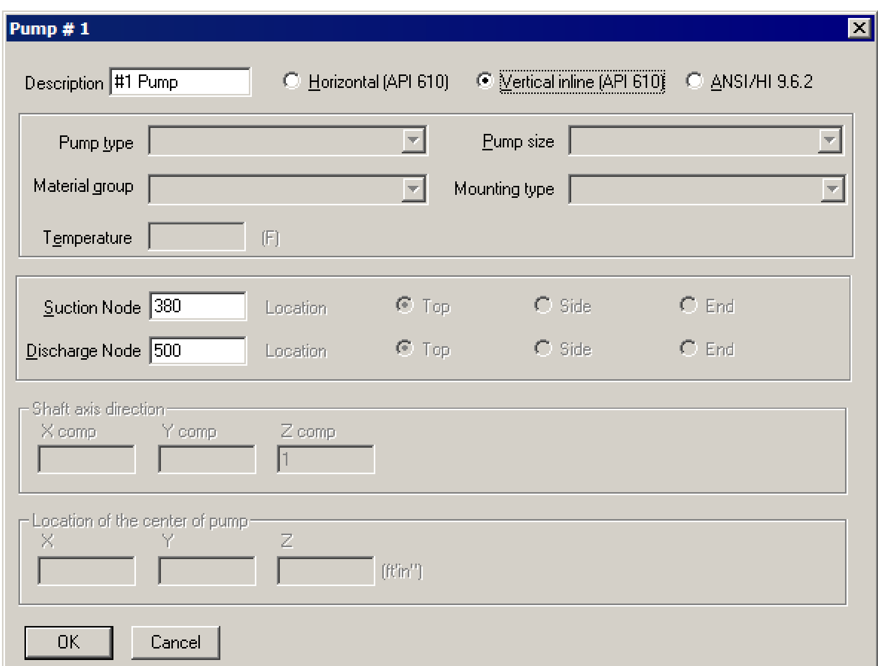


A simpler example is when you do have piping on both sides of the pump. Consider the network below consisting of two pipe segments connected by a pump.

(suction side) 10-20-30-…-90-100→PUMP←200-210-…-280-290-300 (discharge side)

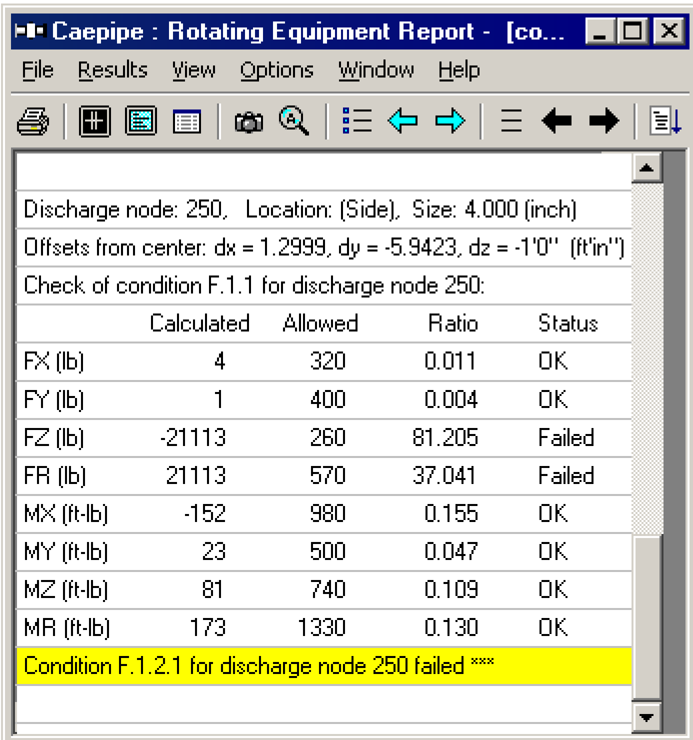
The suction side of the pump ends at node 100. The discharge side begins at node 200. Make nodes 100 and 200 as anchors so that equipment loads can be calculated. A similar method applies to turbines and compressors too.

A different dialog is shown for vertical inline pumps. Only Description, Suction and Discharge nodes are required.



API 610 Report

Upon analysis, you will see CAEPIPE produce API 610 reports under “Rotating Equipment Reports” in Results.



Discharge node: 250, Location: (Side), Size: 4.000 (inch)
Offsets from center: dx = 1.2999, dy = -5.9423, dz = -1'0" (ft'in")
Check of condition F.1.1 for discharge node 250:

	Calculated	Allowed	Ratio	Status
FX (lb)	4	320	0.011	OK
FY (lb)	1	400	0.004	OK
FZ (lb)	-21113	260	81.205	Failed
FR (lb)	21113	570	37.041	Failed
MX (ft-lb)	-152	980	0.155	OK
MY (ft-lb)	23	500	0.047	OK
MZ (ft-lb)	81	740	0.109	OK
MR (ft-lb)	173	1330	0.130	OK

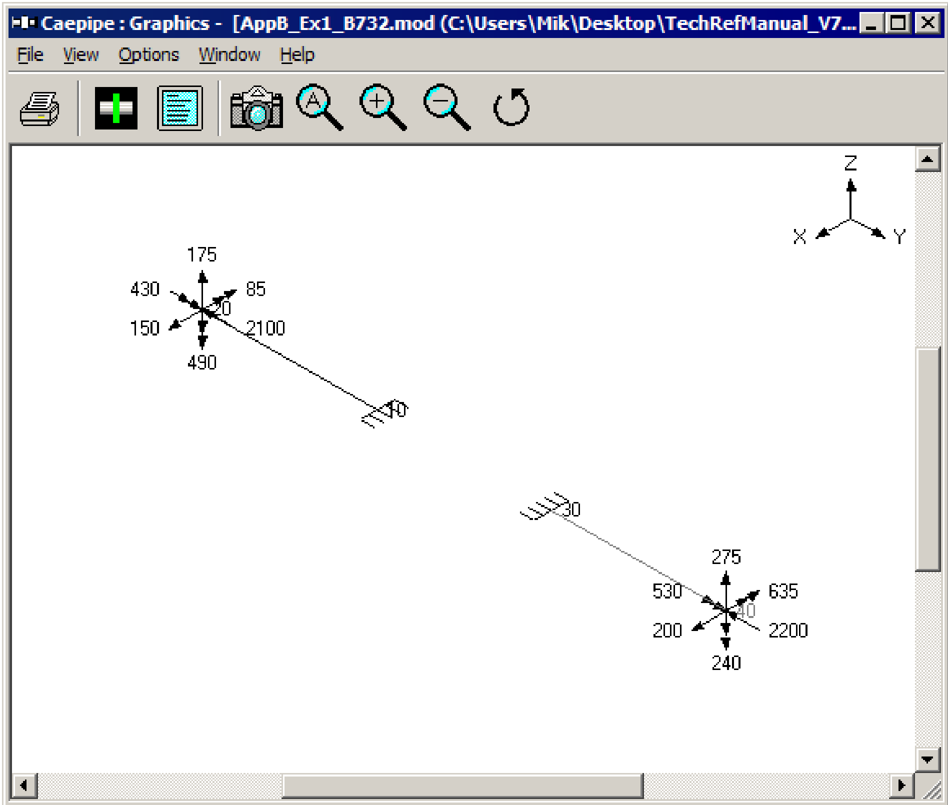
Condition F.1.2.1 for discharge node 250 failed ***

In addition to the input details, for a specific load case, the calculated forces and moments (on the nozzle and those at the center of the pump), API allowables, ratios (of calculated to allowable) and status for all of them are reported.

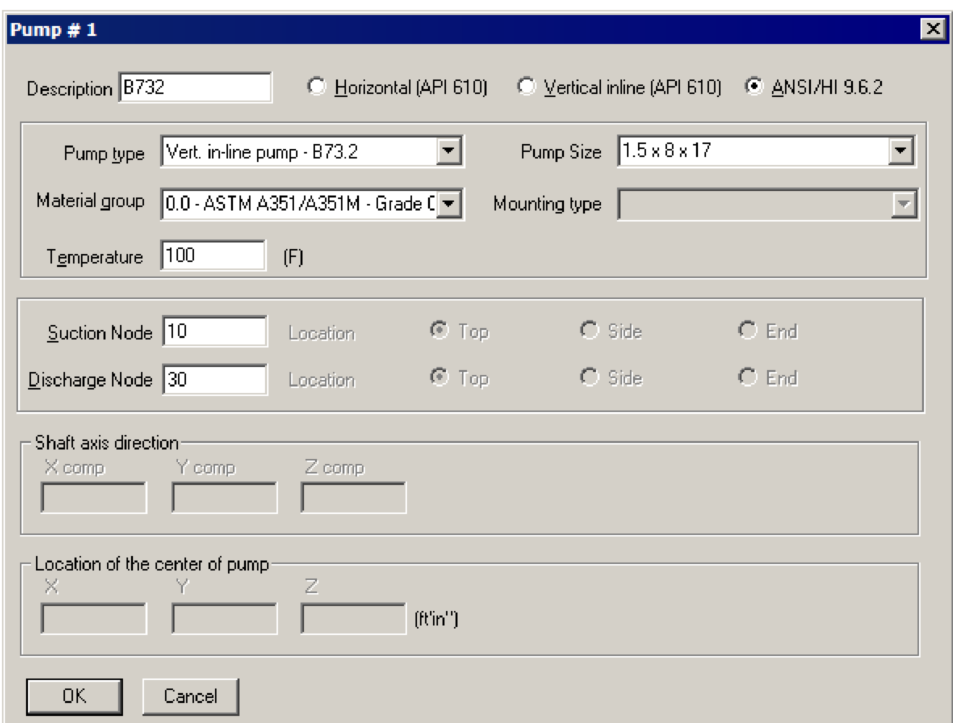
When you see “Failed” entries in this report, you will need to examine the cause of the high force or moment for that line item. Generally, the high numbers come from the expansion load but may well come from the weight load. You must reduce these excessive forces and moments by making the system or intersections more flexible before this pump can become compliant.

Note: If you have input multiple temperatures, corresponding reports for additional operating load cases are shown. Use the black right arrow key to see them.

Example – ANSI/HI 9.6.2 Pump Compliance

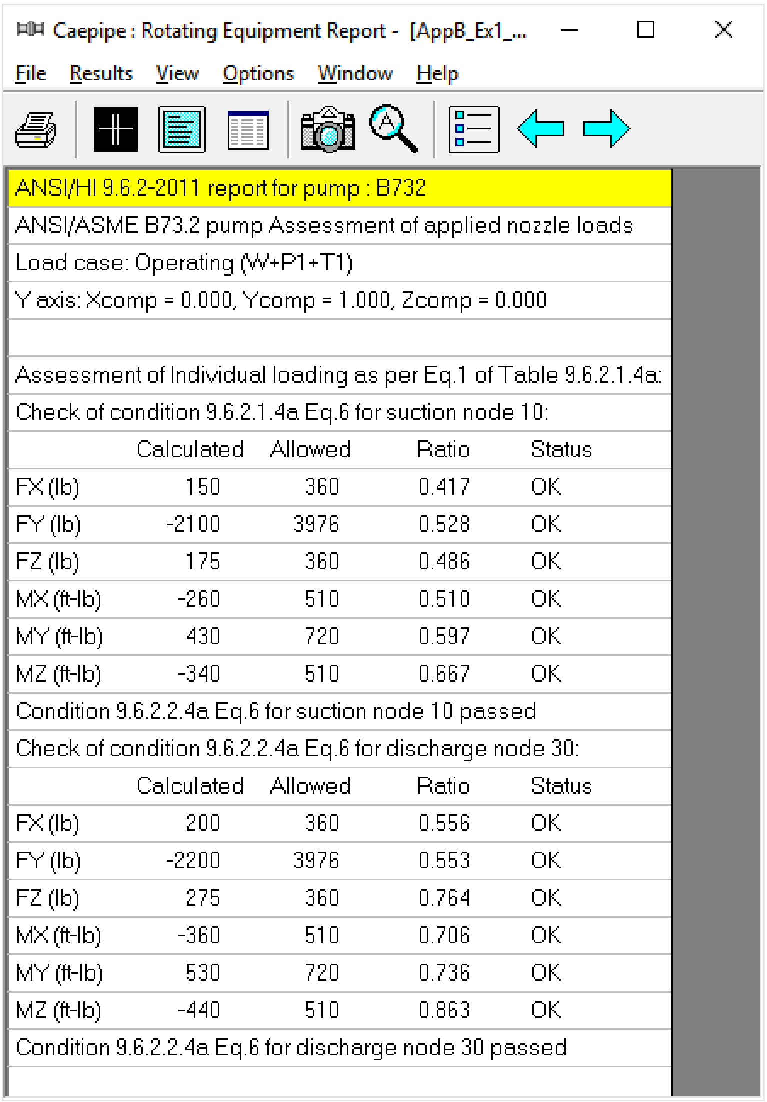


In the above image, suction side piping ends at node 10. Discharge piping starts at node 30. Pump size is 1.5 x 8 17 with Material Group ASTM A351/A351M – Grade CF8M. The temperature of the pump is set to 100^0 F.



ANSI/HI 9.6.2 Report

Upon analysis, you will see CAEPIPE produce ANSI/HI 9.6.2 reports under "Rotating Equipment Reports" in Results.



ANSI/HI 9.6.2-2011 report for pump : B732

ANSI/ASME B73.2 pump Assessment of applied nozzle loads

Load case: Operating (W+P1+T1)

Y axis: Xcomp = 0.000, Ycomp = 1.000, Zcomp = 0.000

Assessment of Individual loading as per Eq.1 of Table 9.6.2.1.4a:

Check of condition 9.6.2.1.4a Eq.6 for suction node 10:

	Calculated	Allowed	Ratio	Status
FX (lb)	150	360	0.417	OK
FY (lb)	-2100	3976	0.528	OK
FZ (lb)	175	360	0.486	OK
MX (ft-lb)	-260	510	0.510	OK
MY (ft-lb)	430	720	0.597	OK
MZ (ft-lb)	-340	510	0.667	OK

Condition 9.6.2.2.4a Eq.6 for suction node 10 passed

Check of condition 9.6.2.2.4a Eq.6 for discharge node 30:

	Calculated	Allowed	Ratio	Status
FX (lb)	200	360	0.556	OK
FY (lb)	-2200	3976	0.553	OK
FZ (lb)	275	360	0.764	OK
MX (ft-lb)	-360	510	0.706	OK
MY (ft-lb)	530	720	0.736	OK
MZ (ft-lb)	-440	510	0.863	OK

Condition 9.6.2.2.4a Eq.6 for discharge node 30 passed

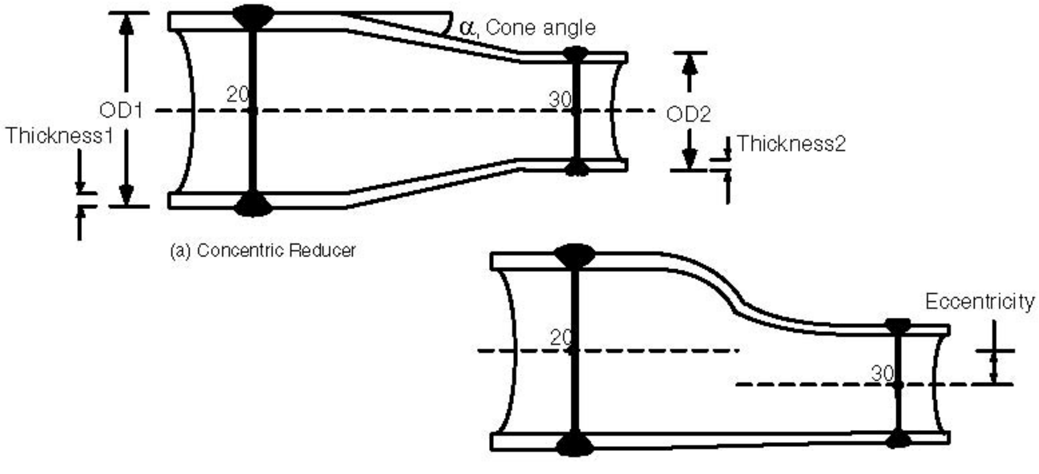
In addition to the input details, for a specific load case, the calculated forces and moments, ANSI/HI 9.6.2 allowables, ratios (of calculated to allowable) and status for all of them are reported.

When you see "Failed" entries in this report, you will need to examine the cause of the high force or moment for that line item. Generally, the high numbers come from the expansion load but may well come from the weight load. You must reduce these excessive forces and moments by making the system or intersections more flexible before this pump can become compliant.

Note: If you have input multiple temperatures, corresponding reports for additional operating load cases are shown. Use the black right arrow key to see them.

Use a reducer to join a larger pipe to a small pipe to meet fluid flow requirements. A reducer is Concentric when the axes at the reducer ends are collinear; Eccentric, when they are not. Use an eccentric reducer only when necessary to keep the top/bottom of the line level.

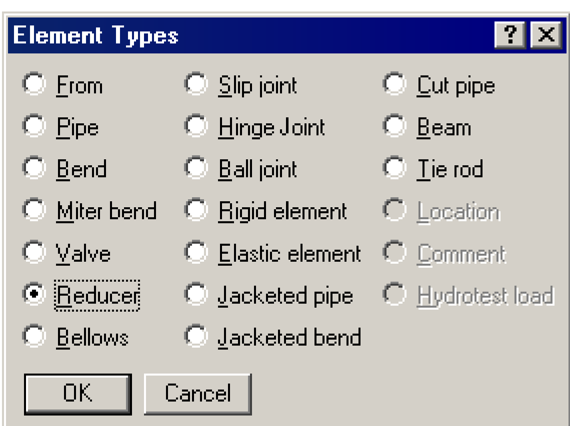
In the figures shown below, observe that the two ends of the reducers are of different diameters. The larger end (at node 20) has the outside diameter and thickness as OD1 and Thickness 1 (Thk1) with the smaller end (at node 30) having OD2 and Thickness 2 (Thk2). In case of the eccentric reducer, the eccentricity as shown is between the two axes of the ends of the reducer. The cone angle, α, is also as shown in the following figure.



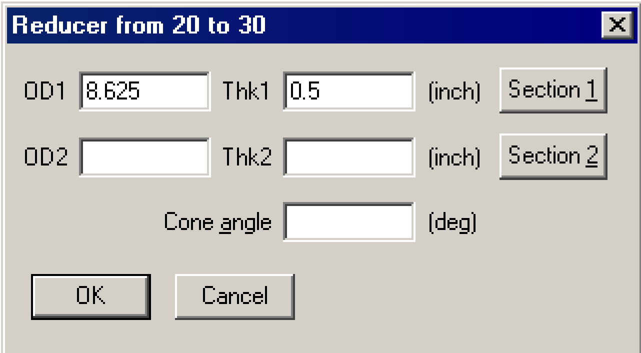
(b) Eccentric Reducer, Nomenclature same as in (a)

An eccentric reducer's eccentricity is modeled by a change in offsets of the "To" node (node 30 in Figure (b) above). Eccentricity is (ID1–ID2) / 2. See example 2 later in this topic.

A reducer (concentric or eccentric) is input by typing "re" in the Type column or selecting "Reducer" from the Element Types dialog.



The Reducer dialog is shown.



OD1, OD2, Thk1, Thk2

These are the cross-sectional properties at the two ends of the reducer. OD stands for outside diameter and Thk stands for thickness. By default, OD1 and Thk1 contain preceding section'soutside diameter and thickness, but different values may be typed here. The Section1 and Section2 buttons can be used to quickly input OD and Thk values from previously defined sections.

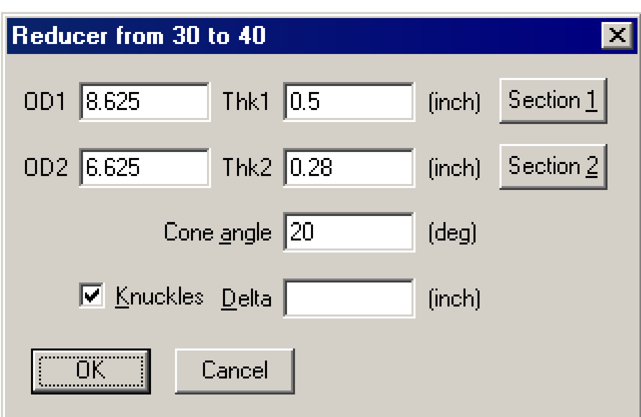
Cone angle

Shown in Figure (a) above, it is used to calculate SIF at the ends of the reducer for certain piping codes (B31.1, B31.9, ASME Section III Class 2, RCC-M, Swedish and Norwegian). For these codes, if the cone angle is left blank, the maximum value of the SIF (2.0) is used. For all other piping codes in CAEPIPE, the cone angle is not used.

SIF Calculation

For B31.1, B31.9, ASME Section III Class 2, EN 13480, RCC-M, Swedish and Norwegian, as mentioned above, the cone angle (if input) is used to calculate the SIF. If the cone angle is not input, the maximum value of the SIF (2.0) is used.

For Swedish and Norwegian piping codes, additional input is required which affects the calculation of SIF.



Knuckles

If the reducer is with knuckles,check this box.

Delta

If the reducer is without knuckles, specify delta, which is the mismatch (difference in mean radii across the weld at the smaller end of the reducer). If the reducer is with knuckles, delta is not used.

For other codes, if the code is not specific about a reducer's SIF, then a value of 1.0 is used.

Weight, Stiffness and Stress Calculation

The properties such as weight of the reducer, stiffness, contents weight and insulation weight are based on the average diameter (of OD1 and OD2) and average thickness (of Thk1 and Thk2).

The stresses at each end, however, are calculated using the actual dimensions at each end.

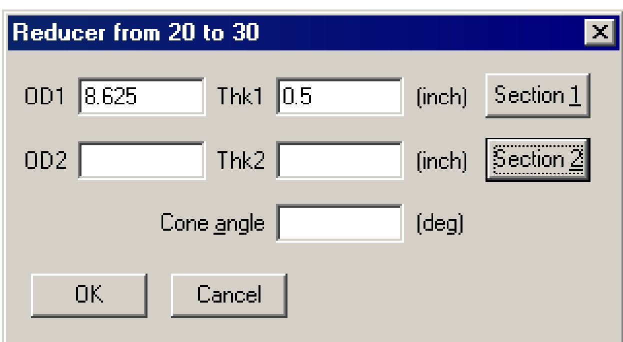
Example 1: Concentric Reducer

To model a concentric reducer as shown in Figure (a) earlier in the topicwith the data:

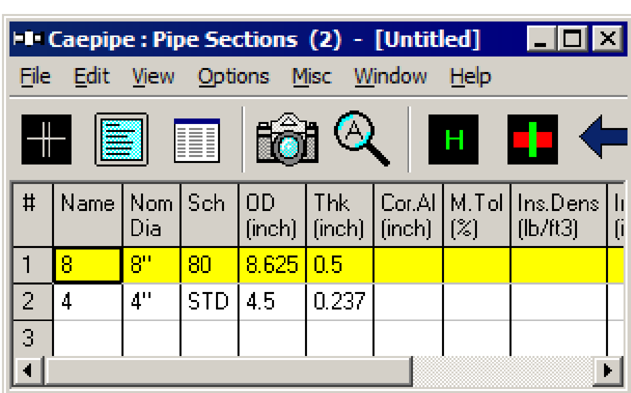
8"x4" reducer, OD1=8.625", Thk1=0.322", OD2=4.5", Thk2=0.237".

Create two sections, 8"/STD and 4"/STD.

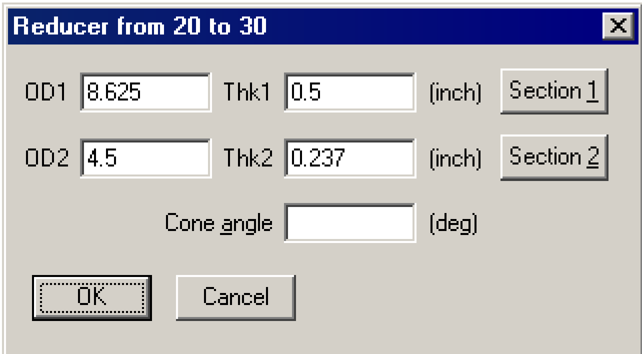
- The first node (10) is already defined. Press Enter to move to the next row.
- Complete pipe run till node 20: type 20 for Node, type 1 (ft.) for DX, enter material, 8" section and load names, press Enter.
- Input reducer: Type 30 for Node, press Tab to move to the Type field, type "Re" (to open the Reducer dialog box, note that by default CAEPIPE displays the preceding 8" section's properties for OD1 and Thk1).



Press the "Section 2" button to select the section at "To" node.



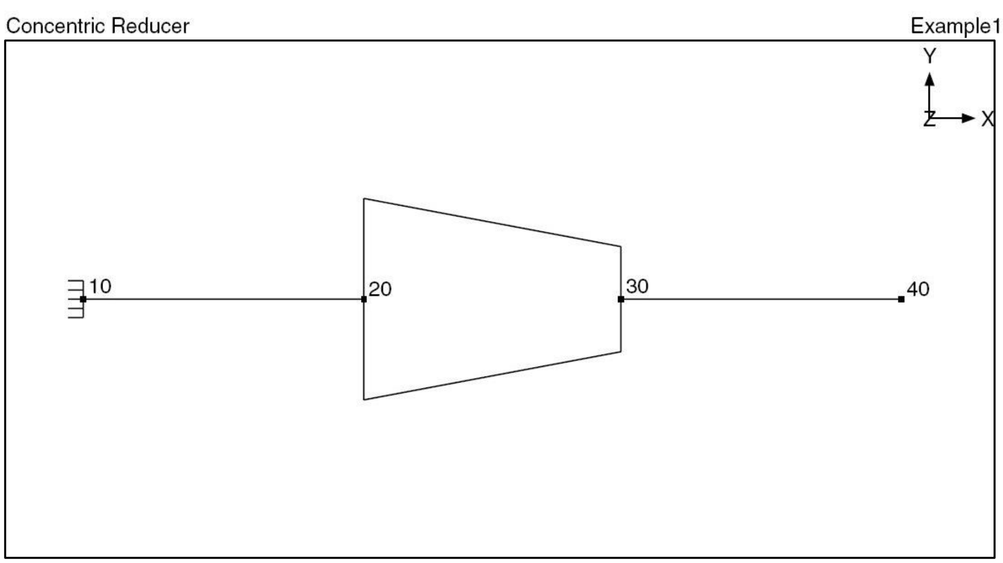
Highlight the 4" section and press OK. 4.5" for OD2 and 0.237" for Thk2 will be entered in the Reducer dialog.



Press OK, type 11" for DX (reducer's length) and press Enter. Now you are asked if you want to change section, press Yes. "Select Section" dialog will be shown. Highlight the 4" section and press Enter. Press Enter again on Layout to move to the next row.

Type 40 for Node, 1 (ft.) for DX, press Enter.

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Concentric Reducer								
2	10	From							Anchor
3	20		1'0"			A53	8	1	
4	30	Reducer	0'11"			A53	4	1	
5	40		1'0"			A53	4	1	
6									

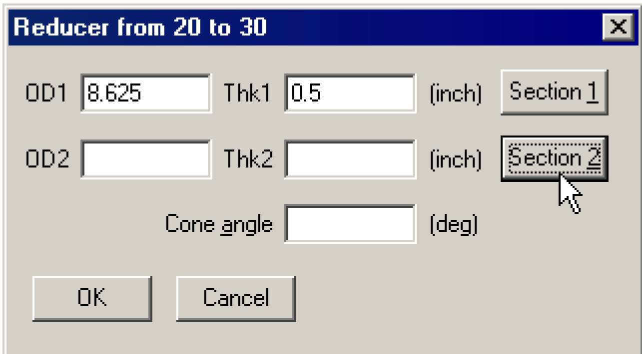


Example 2: Eccentric Reducer

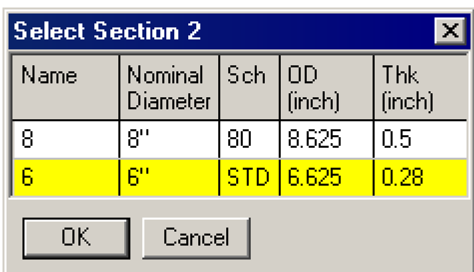
To model an eccentric reducer (as shown in Figure (b) earlier in the topic) with the following data: 8"x6" reducer, OD1=8.625", Thk1=0.322", OD2=6.625", Thk2=0.28", eccentricity = (ID1–ID2) / 2 = 0.958" which is modeled as change in elevation.

Create two sections, 8"/STD and 6"/STD.

- The first node (10) is already defined. Press Enter to move to the next row.
- Complete pipe run till node 20: type 20 for Node, type 12" for DX, enter material, 8" section and load names, press Enter.
- Input reducer: Type 30 for Node, press Tab to move to the Type field, type "Re" (to open the Reducer dialog box, note that the preceding 8" section properties are already displayed for OD1 and Thk1).

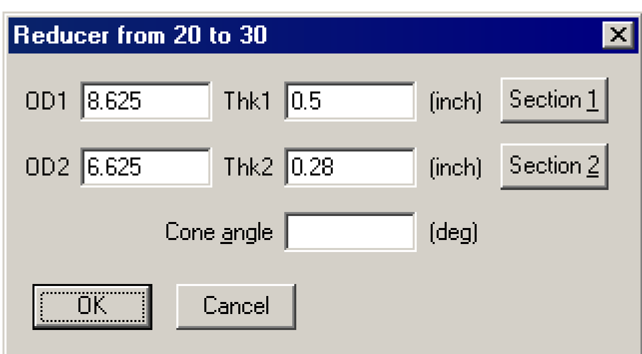


Press the "Section 2" button to select the section at "To" node.



Highlight the 6" section and press OK.

6.625" for OD2 and 0.28" for Thk2 will be entered in the Reducer dialog.

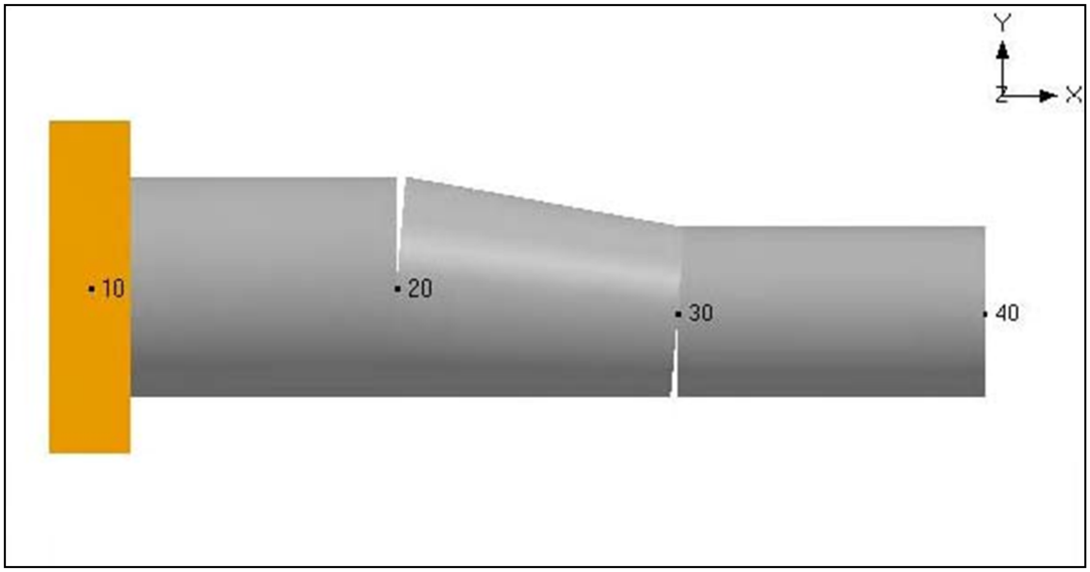


Press OK, type 11" for DX (reducer's length along X axis), -0.958" for DY (this is the eccentricity), then press Enter. Now you are asked if you want to change section, press Yes. "Select Section" dialog will be shown. Highlight the 6" section and press Enter. Press Enter again on Layout to move to the next row.

- Type 40 for Node, 12" for DX, press Enter.

#	Node	Type	DX (inch)	DY (inch)	DZ (inch)	Matl	Sect	Loac	Data
1	Title = Eccentric Reducer								
2	10	From							Anchor
3	20		12			A53	8	1	
4	30	Reducer	11	-0.958		A53	6	1	
5	40		12			A53	6	1	
6									

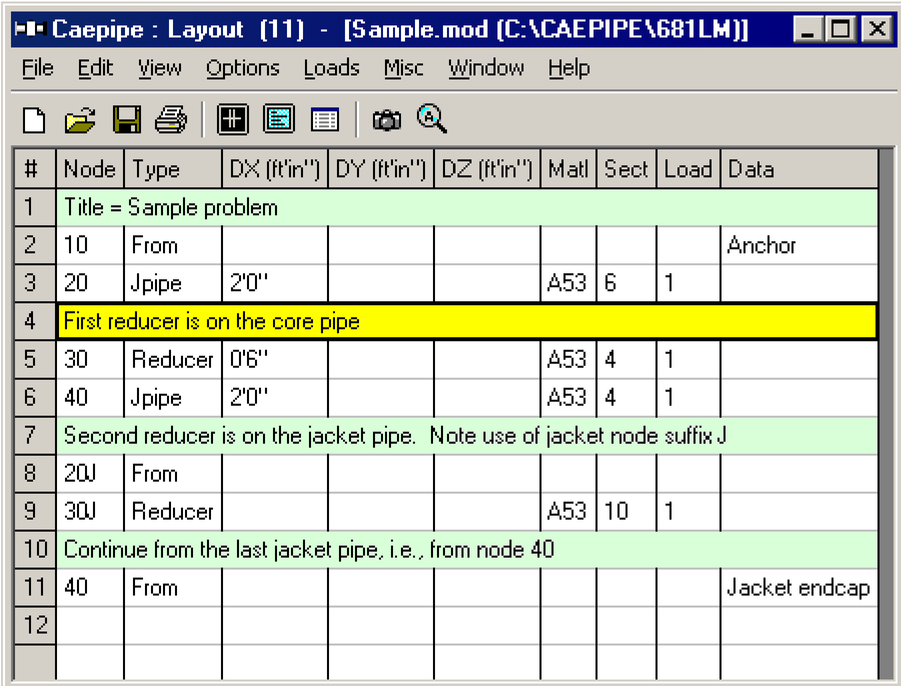
The rendered graphics is shown below:



Example 3: Jacketed Reducer

A jacketed reducer may be modeled in the following manner: Calculate the averages of OD1 and OD2, and of Thk1 and Thk2 for the two reducers, one for the core pipe and the other for the jacket pipe. Create two new pipe sections with these averages as OD and Thickness. Insert two pipes at the location of that jacketed reducer, one for the core pipe and the other for the jacket pipe, having the corresponding section with average OD and thickness. You may have to input SIF at the two ends of each reducer using code guidelines for a reducer.

Alternate Method: In a jacketed piping system, e.g., 10-20 is a JPIPE, model the reducer for the core pipe next as you would normally do in a non-jacketed system, i.e., 20-30 is the core pipe reducer, followed by a JPIPE between 30-40. Then, connect the jacket nodes 20J and 30J with a jacket reducer (shown below) or a JPIPE (jacket pipe). In other words, on new rows, connect 20J to 30J with the jacket reducer or a JPIPE with average OD and thickness as calculated above.



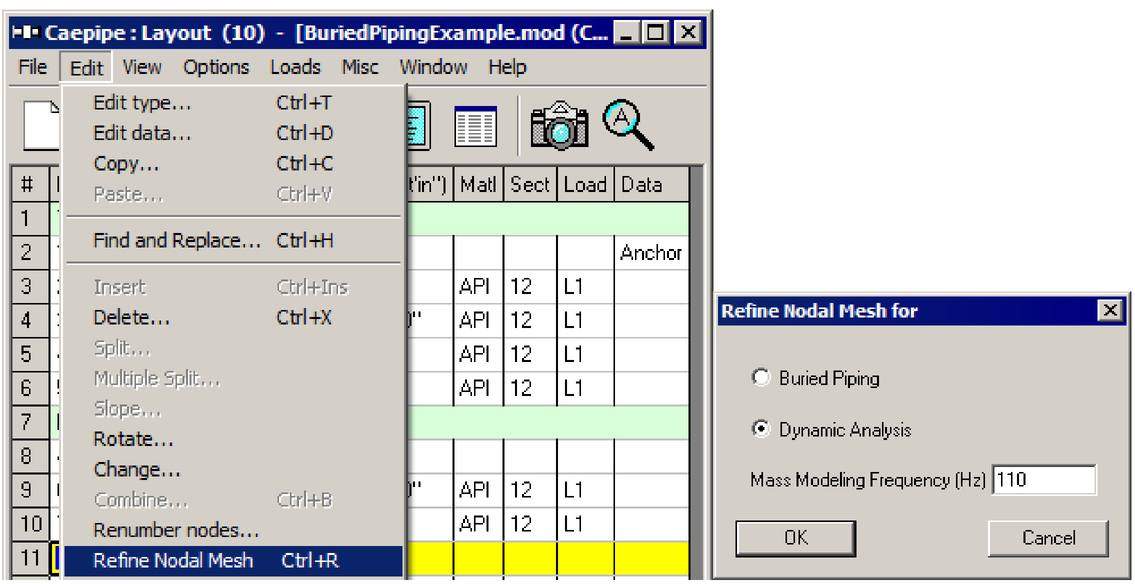
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Jpipe	2'0"			A53	6	1	
4	First reducer is on the core pipe								
5	30	Reducer	0'6"			A53	4	1	
6	40	Jpipe	2'0"			A53	4	1	
7	Second reducer is on the jacket pipe. Note use of jacket node suffix J								
8	20J	From							
9	30J	Reducer				A53	10	1	
10	Continue from the last jacket pipe, i.e., from node 40								
11	40	From							Jacket endcap
12									

Refinement of Nodal Mesh based on Mass Modeling Frequency

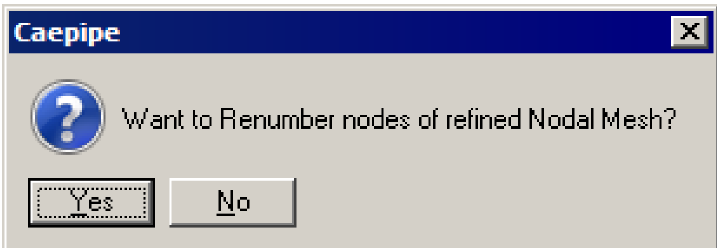
The purpose of this feature is to ensure that there are a sufficient number of mass points for an accurate dynamic model for the dynamic loading under consideration.
Intermediate mass points along a span are generated based on the free vibration of an equivalent simply supported beam. Optimum element length is calculated from:

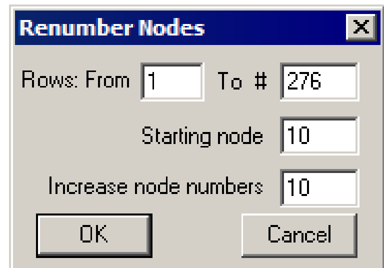
$$L_{opt} = \frac{1}{2}\sqrt{\frac{\pi}{2.f}}\cdot\sqrt[4]{\frac{E.I.g}{w}}$$

L_{opt} = Optimum length required to capture the span dynamic behavior
f = mass modeling frequency
g = acceleration due to gravity
E = modulus of elasticity of pipe material
Although the above equation is valid for any temperature, to generate intermediate nodes, E is taken at the reference temperature entered in CAEPIPE.
I = moment of inertia of pipe cross section
w = weight per unit length of pipe (including insulation, lining and content)
Intermediate mass points can be automatically generated in CAEPIPE by selecting the radio button "Dynamic Analysis" through Layout window > Edit > Refine Nodal Mesh. Enter the Mass modeling Frequency in the dialog box shown and press the button "OK". See figures shown below for details.



While refining the Nodal Mesh, the new node numbers will be generated by adding the node increment specified through Layout window > Options > Node increment to get the new node numbers (without affecting the original node numbers used in the Layout window). Hence, set the node increment value as required before refining the Nodal Mesh.
Upon refining the Nodal Mesh based on Mass modeling frequency, CAEPIPE will prompt for renumbering of nodes as shown below.

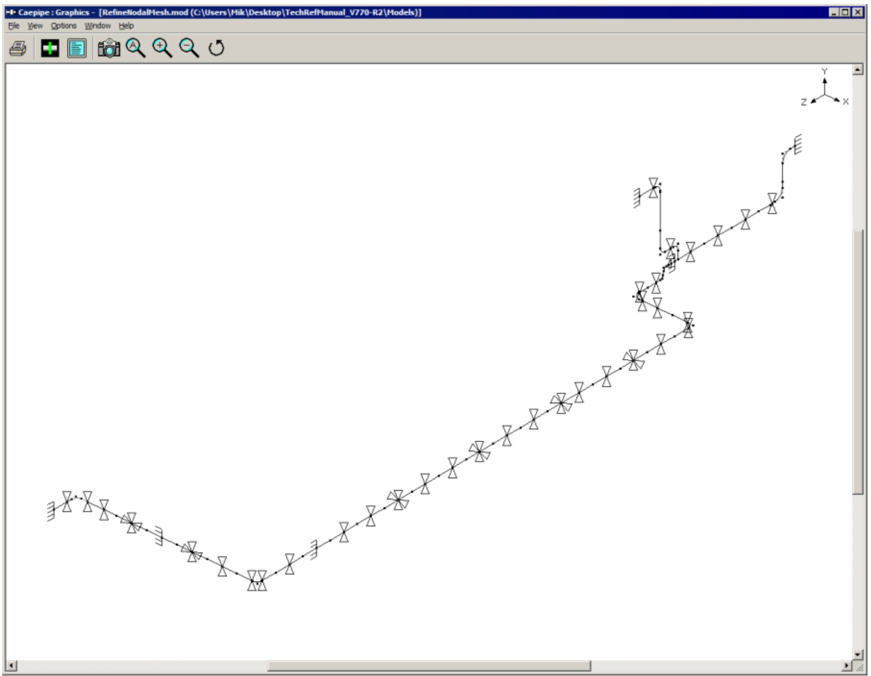




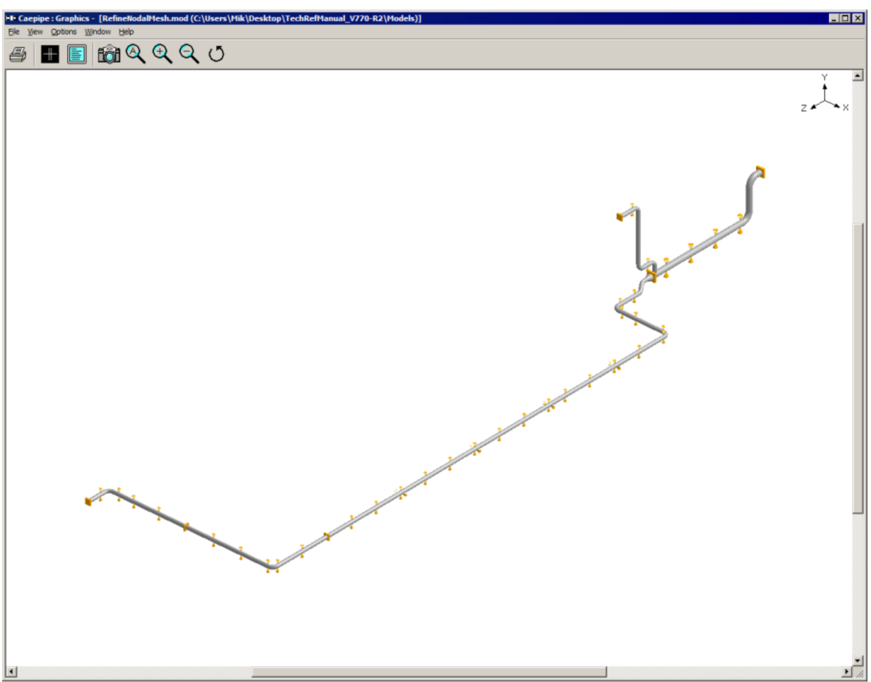
Press the button "Yes" to renumber the nodes and enter the details required by CAEPIPE in the dialog box. See snap shot shown above.

Example

A sample CAEPIPE model (with graphics and layout details as shown below) was chosen for verification of implementation. Modal analysis was then performed by defining the cut-off frequency and number of modes as “110 Hz” and 175 respectively in the CAEPIPE model through Layout window > Options > Analysis > Dynamics with the node points as defined by the stress analyst.



Wireframe layout without addition of mass points



Rendered layout without addition of mass points

Caepipe dynamic_model_original Page 1
Version 8.00 May 30,2018

Options

Piping code = B31.3 (2016)
Do not use liberal allowable stresses
Include axial force in stress calculations
Reference temperature = 40 (F)
Number of thermal cycles = 7000
Number of thermal loads = 1
Thermal = Operating - Sustained
Use modulus at reference temperature
Include hanger stiffness
Include Bourdon effect
Use pressure correction for bends
Pressure stress = PD / 4t
Peak pressure factor = 1.00
Cut off frequency = 110 Hz
Number of modes = 175
Include missing mass correction
Use friction in dynamic analysis
Vertical direction = Y

#	Node	Type	DX(ft'in")	DY(ft'in")	DZ(ft'in")	Mat	Sec	Load	Data
1	Title =								
2	10	From							Anchor
3	20	Bend			11'0"	API	54I	54I	
4	30			-20'0"		API	54I	54I	
5	40	Bend		-10'9"		API	54I	54I	
6	50				9'0"	API	54I	54I	Y restraint
7	60				2'0"	API	54O	54O	
8	65				10'0"	API	54O	54O	
9	70				12'0"	API	54O	54O	Y restraint
10	75				12'0"	API	54O	54O	
11	80				12'0"	API	54O	54O	Y restraint
12	85				12'0"	API	54O	54O	
13	90				12'0"	API	54O	54O	Y restraint
14	100				11'0"	API	54O	54O	
15	110				3'0"	API	54O	54O	Anchor
16	120				3'0"	API	54O	54O	
17	130	Reducer			3'0"	API	54O	54O	
18	140	Bend			3'8"	API	36I	36I	
19	150	Bend	2'11"	-4'6"	4'6"	API	36I	36I	
20	160				6'0"	API	36I	36I	Y restraint
21	165				7'0"	API	36I	36I	
22	170				7'0"	API	36I	36I	Y restraint
23	180	Bend			6'0"	API	36I	36I	
24	190		7'3"			API	36I	36I	Y restraint
25	200		12'0"			API	36I	36I	Y restraint
26	210		12'0"			API	36I	36I	
27	220		12'0"			API	36I	36I	Y restraint
28	230	Bend	4'5"			API	36I	36I	
29	240				4'5"	API	36I	36I	Y restraint
30	250				12'0"	API	36I	36I	
31	260				12'0"	API	36I	36I	Y restraint
32	270				12'0"	API	36I	36I	
33	280				12'0"	API	36I	36I	XY restraint

Caepipe dynamic_model_original Page 2
Version 8.00 May 30,2018

#	Node	Type	DX(ft'in")	DY(ft'in")	DZ(ft'in")	Mat	Sec	Load	Data
34	290				12'0"	API	36I	36I	
35	300				12'0"	API	36I	36I	Y restraint
36	310				12'0"	API	36I	36I	
37	320				12'0"	API	36I	36I	Y restraint
38	330				16'0"	API	36I	36I	XY restraint
39	340				12'0"	API	36I	36I	
40	350				12'0"	API	36I	36I	Y restraint
41	360				12'0"	API	36I	36I	
42	370				12'0"	API	36I	36I	Y restraint
43	380				12'0"	API	36I	36I	
44	390				12'0"	API	36I	36I	XY restraint
45	400				12'0"	API	36I	36I	
46	410				12'0"	API	36I	36I	Y restraint
47	420				12'0"	API	36I	36I	
48	430				12'0"	API	36I	36I	Y restraint
49	440				12'0"	API	36I	36I	
50	450				12'0"	API	36I	36I	XY restraint
51	460				12'0"	API	36I	36I	
52	470				12'0"	API	36I	36I	Y restraint
53	480				12'0"	API	36I	36I	
54	490				12'0"	API	36I	36I	Y restraint
55	500				12'0"	API	36I	36I	
56	510				12'0"	API	36I	36I	Anchor
57	520				12'0"	API	36I	36I	
58	530				12'0"	API	36I	36I	Y restraint
59	540				12'0"	API	36I	36I	
60	550				12'0"	API	36I	36I	Y restraint
61	560	Bend			4'5"	API	36I	36I	
62	570		-4'5"			API	36I	36I	Y restraint
63	580		-12'0"			API	36I	36I	
64	590		-11'7"			API	36I	36I	Y restraint
65	600		-12'0"			API	36I	36I	
66	610		-12'0"			API	36I	36I	XY restraint
67	620		-12'0"			API	36I	36I	
68	630		-12'0"			API	36I	36I	Anchor
69	640		-12'0"			API	36I	36I	
70	650		-12'0"			API	36I	36I	XY restraint
71	660		-12'0"			API	36I	36I	
72	670		-10'0"			API	36I	36I	Y restraint
73	680		-13'0"			API	36I	36I	Y restraint
74	690	Bend	-9'0"			API	36I	36I	
75	700				7'11"	API	36I	36I	Y restraint
76	710				12'0"	API	36I	36I	Anchor
77	100	From							
78	720	Bend		10'9"		API	36O	36O	
79	730				7'0"	API	36O	36O	Y restraint
80	740	Bend			9'0"	API	36O	36O	
81	750			16'9"		API	36O	36O	
82	760			26'11"		API	36O	36O	
83	770	Bend		5'8"		API	36O	36O	
84	780				6'0"	API	36O	36O	Y restraint
85	790				12'0"	API	36O	36O	Anchor

Caepipe dynamic_model_original Page 3
Version 8.00 May 30,2018

Anchors

Node	KX/kx	KY/ky	KZ/kz	KXX/kxx	KYY/kyy	KZZ/kzz	Releases X Y Z XXYYZZ	Anchor In Pipe
	(lb/inch)			(in-lb/deg)				
10	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid		GCS
110	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid		GCS
510	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid		GCS
630	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid		GCS
710	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid		GCS
790	Rigid	Rigid	Rigid	Rigid	Rigid	Rigid		GCS

Bends

Bend Node	Radius (inch)	Thickness (inch)	Bend Matl	Flex. Factor	Int. Node	Angle (deg)	Int. Node	Angle (deg)
20	81	L						
40	81	L						
140	54	L						
150	54	L						
180	54	L						
230	48	U						
560	48	U						
690	48	U						
720	54	L						
740	54	L						
770	54	L						

Reducers

From	To	OD1 (inch)	Thk1 (inch)	OD2 (inch)	Thk2 (inch)	Cone Angle (deg)	Knuckles	Delta (inch)
120	130	54	0.375	36	0.375			

Restraints

Node	X	Y	Z
50		Yes	
70		Yes	
80		Yes	
90		Yes	
160		Yes	
170		Yes	
190		Yes	
200		Yes	
220		Yes	
240		Yes	
260		Yes	
280	Yes	Yes	
300		Yes	
320		Yes	
330	Yes	Yes	
350		Yes	

Restraints

Node	X	Y	Z
370		Yes	
390	Yes	Yes	
410		Yes	
430		Yes	
450	Yes	Yes	
470		Yes	
490		Yes	
530		Yes	
550		Yes	
570		Yes	
590		Yes	
610	Yes	Yes	
650	Yes	Yes	
670		Yes	
680		Yes	
700		Yes	
730		Yes	
780		Yes	

Pipe material API: API 5L Grade B

Density = 0.283 (lb/in3), Nu = 0.300, Joint factor = 1.00, Type = CS
Yield strength = 35000 (psi)

Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
-325	31.4E+6	5.00E-6	20000
-200	30.8E+6	5.35E-6	20000
-100	30.2E+6	5.65E-6	20000
70	29.5E+6	6.07E-6	20000
200	28.8E+6	6.38E-6	20000
300	28.3E+6	6.60E-6	20000
400	27.7E+6	6.82E-6	19900
500	27.3E+6	7.02E-6	19000
600	26.7E+6	7.23E-6	17900
650	26.1E+6	7.33E-6	17300
700	25.5E+6	7.44E-6	16700
750	24.8E+6	7.54E-6	13900
800	24.2E+6	7.65E-6	11400
850	23.3E+6	7.75E-6	8700
900	22.4E+6	7.84E-6	5900
950	21.4E+6	7.91E-6	4000
1000	20.4E+6	7.97E-6	2500
1050	19.2E+6	8.05E-6	1600
1100	18.0E+6	8.12E-6	1000

Caepipe dynamic_model_original Page 5
Version 8.00 May 30,2018

Pipe Sections

Name	Nominal Dia.	Sch	O.D. (inch)	Thk (inch)	Cor.Al (inch)	M.Tol (%)	Ins.Dens (lb/ft3)	Ins.Th (inch)	Lin.Dens (lb/ft3)	Lin.Th (inch)
36I	36"	STD	36	0.375	0	0.0				
36O	36"	STD	36	0.375	0	0.0				
54O	Non Std		54	0.375	0	0.0				
54I	Non Std		54	0.375	0	0.0				

Loads

Static seismic load: X = -0.20, Y = -0.20, Z = -0.20 (g's)
Acceleration load combination = Square Root of Sum of Squares

X spectrum: Malta, NY_b Factor = 1.0000
Y spectrum: Malta, NY_b Factor = 0.0430
Z spectrum: Malta, NY_b Factor = 1.0000

Mode sum = SRSS Direction sum = SRSS

Wind Load 1

Shape factor = 0.60
Wind direction: X comp = 0.000, Y comp = 0.000, Z comp = 1.000

Elevation (feet)	Pressure (psf)
0	15
15	15
30	15
45	15
60	15

Pipe Loads

Load Name	T1 (F)	P1 (psi)	T2 (F)	P2 (psi)	T3 (F)	P3 (psi)	Specific gravity	Add.Wgt (lb/ft)	Wind Load
36O	100	125							Y
36I	100	125							
54O	100	125							Y
54I	100	125							

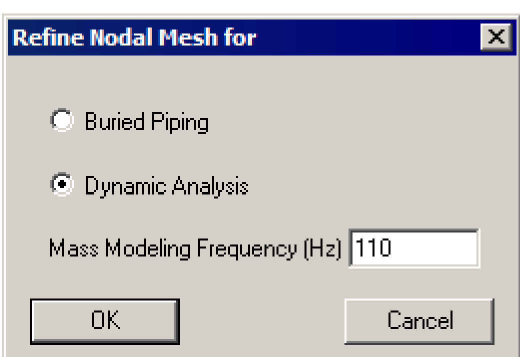
From the Modal Analysis results shown below, it was noted that the CAEPIPE was able to extract 92 modes with highest frequency being 95.967 Hz.

```
Caepipe                        dynamic_model_original              Page 1
Version 8.00                                                       May 30,2018
----------------------------------------------------------------------------------
     Frequency  Period    Participation factors    Modal mass / Total mass
 Mode    (Hz)    (sec)      X        Y        Z         X        Y       Z
----------------------------------------------------------------------------------
  1      1.875  0.5335  -5.7615  -0.0004   0.5255   0.0979  0.0000  0.0008
  2      3.288  0.3041  -6.7721  -0.0000   0.0003   0.1353  0.0000  0.0000
  3      5.971  0.1675  -4.1919   0.0000  -0.0001   0.0518  0.0000  0.0000
  4      6.147  0.1627   0.0909  -0.0000  -0.1021   0.0000  0.0000  0.0000
  5      7.800  0.1282  -0.4240  -0.0000   4.5032   0.0005  0.0000  0.0598
  6      8.102  0.1234   0.0029   0.0001   0.3321   0.0000  0.0000  0.0003
  7      9.933  0.1007   0.0861  -0.0000  -4.2655   0.0000  0.0000  0.0537
  8     10.146  0.0986  -0.6501   0.0002   1.6688   0.0012  0.0000  0.0082
  9     10.241  0.0976  -2.1689  -0.0000   0.0048   0.0139  0.0000  0.0000
 10     11.279  0.0887   3.2684  -0.0002  -7.9718   0.0315  0.0000  0.1874
 11     11.628  0.0860   6.7199  -0.0002   5.8034   0.1332  0.0000  0.0993
 12     13.116  0.0762  -0.0013   1.4576  -3.8221   0.0000  0.0063  0.0431
 13     13.276  0.0753   2.6364   0.0000  -0.0001   0.0205  0.0000  0.0000
 14     13.931  0.0718   3.3527   0.0030  -3.2879   0.0332  0.0000  0.0319
 15     16.027  0.0624  -3.6866   0.0000  -0.3181   0.0401  0.0000  0.0003
 16     16.038  0.0624  -2.9281  -0.0001  -0.7810   0.0253  0.0000  0.0018
 17     17.802  0.0562  -1.8832   0.0179   2.3004   0.0105  0.0000  0.0156
 18     21.780  0.0459   0.0000   2.7930   3.3879   0.0000  0.0230  0.0339
 19     22.412  0.0446  -0.0006  -3.2159  -1.3274   0.0000  0.0305  0.0052
 20     22.683  0.0441  -0.3117  -0.0005   0.0747   0.0003  0.0000  0.0000
 21     24.003  0.0417  -0.5955  -0.0000   0.1696   0.0010  0.0000  0.0001
 22     24.910  0.0401   2.7763  -0.0000  -0.0070   0.0227  0.0000  0.0000
 23     26.421  0.0378  -0.2314   0.0001   0.1348   0.0002  0.0000  0.0001
 24     27.698  0.0361  -1.3316   0.0000  -0.0002   0.0052  0.0000  0.0000
 25     29.713  0.0337  -1.2585  -0.0000  -0.0049   0.0047  0.0000  0.0000
 26     30.079  0.0332  -0.3437   0.0000  -0.1955   0.0003  0.0000  0.0001
 27     30.146  0.0332   0.0215  -0.0001   0.3494   0.0000  0.0000  0.0004
 28     31.946  0.0313  -0.6062   0.0012  -1.6290   0.0011  0.0000  0.0078
 29     33.670  0.0297   0.9928  -0.0040   2.8297   0.0029  0.0000  0.0236
 30     36.072  0.0277   0.0014   1.4997  -2.1277   0.0000  0.0066  0.0134
 31     37.812  0.0264  -0.4989  -0.0074   2.1572   0.0007  0.0000  0.0137
 32     39.097  0.0256   1.2371  -0.0000  -0.0086   0.0045  0.0000  0.0000
 33     41.172  0.0243  -0.0001   1.2001  -6.2487   0.0000  0.0042  0.1152
 34     41.250  0.0242  -0.3414  -0.0000   1.6185   0.0003  0.0000  0.0077
 35     42.488  0.0235   0.7544  -0.0001  -0.0047   0.0017  0.0000  0.0000
 36     43.804  0.0228  -0.4830   0.0108   1.2389   0.0007  0.0000  0.0045
 37     46.065  0.0217  -0.0011   0.0686  -0.0001   0.0000  0.0000  0.0000
 38     46.766  0.0214  -0.0098  -0.5622  -0.0023   0.0000  0.0009  0.0000
 39     47.504  0.0211  -0.2337  -0.0029  -0.2475   0.0002  0.0000  0.0002
 40     49.063  0.0204   0.0015   0.3511   0.0007   0.0000  0.0004  0.0000
 41     49.295  0.0203  -0.0000  -1.0018  -0.0000   0.0000  0.0030  0.0000
 42     49.985  0.0200   1.3732  -0.0000  -1.3809   0.0056  0.0000  0.0056
 43     50.052  0.0200  -0.1938   0.0000  -1.9635   0.0001  0.0000  0.0114
 44     52.178  0.0192  -0.9811  -0.0281  -0.6345   0.0028  0.0000  0.0012
 45     52.335  0.0191   0.0000  -0.8918  -0.0000   0.0000  0.0023  0.0000
 46     53.313  0.0188  -0.0715  -1.0283  -0.1169   0.0000  0.0031  0.0000
 47     53.496  0.0187  -1.0974  -0.0751  -1.8548   0.0036  0.0000  0.0101
 48     54.481  0.0184   0.0407  -1.8917   0.0845   0.0000  0.0106  0.0000
 49     56.408  0.0177   0.4159  -1.3502   0.0358   0.0005  0.0054  0.0000
 50     56.457  0.0177   2.6140   0.2176  -0.1253   0.0202  0.0001  0.0000
 51     56.920  0.0176   1.8036   0.0012   0.0177   0.0096  0.0000  0.0000
 52     57.929  0.0173  -0.2764  -0.0033  -0.0311   0.0002  0.0000  0.0000
 53     58.372  0.0171  -1.1204   0.0366   0.3623   0.0037  0.0000  0.0004
```

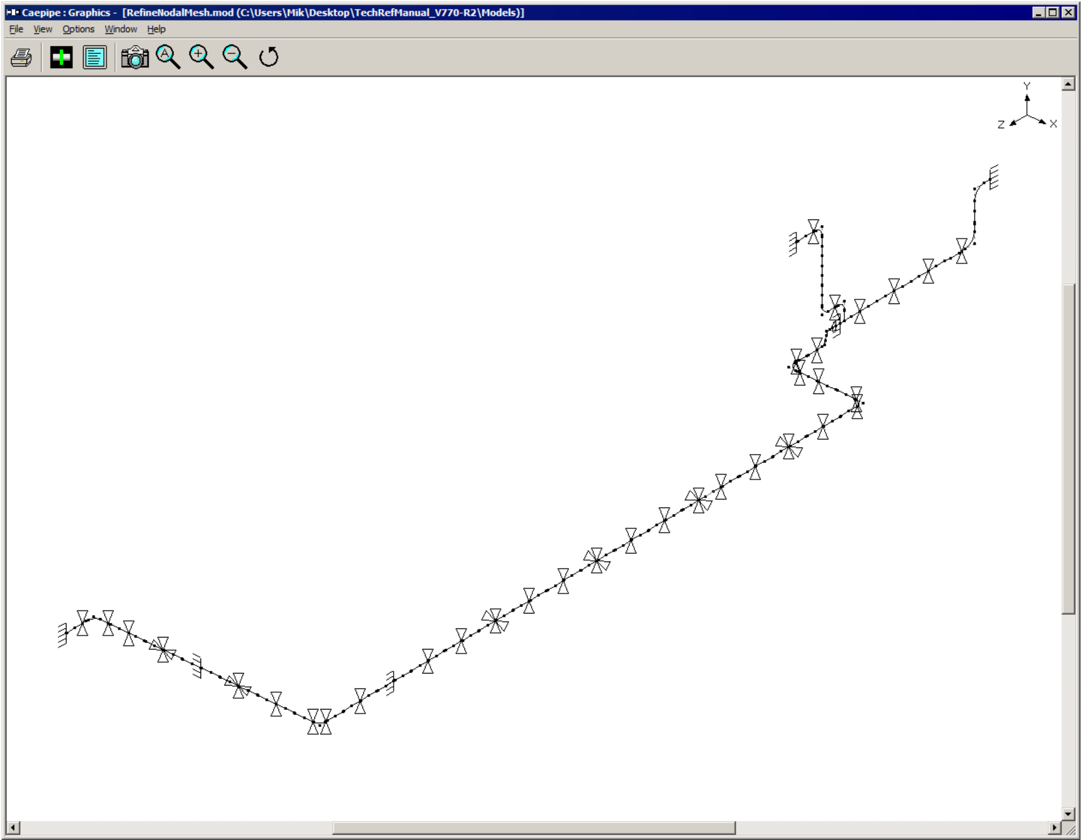
Caepipe dynamic_model_original Page 2
Version 8.00 May 30,2018

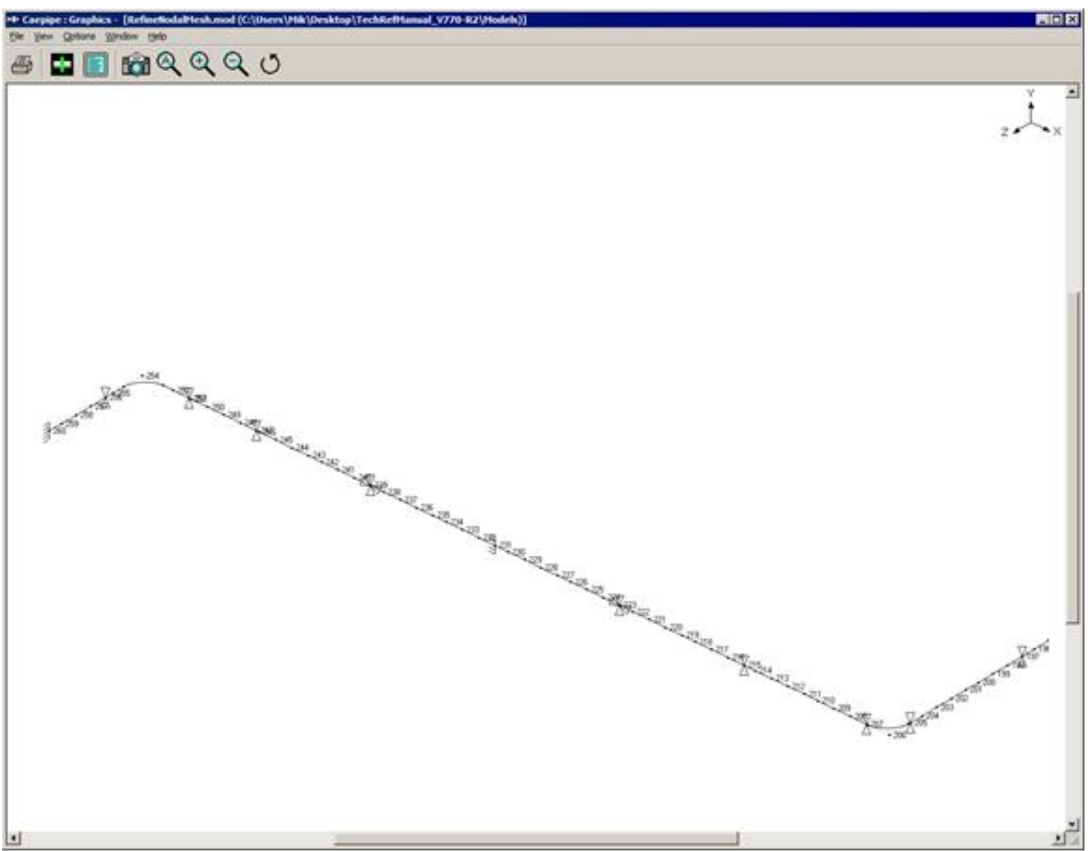
Mode	Frequency (Hz)	Period (sec)	Participation factors X	Y	Z	Modal mass / Total mass X	Y	Z
54	58.736	0.0170	0.0013	-0.4068	0.0005	0.0000	0.0005	0.0000
55	62.338	0.0160	-0.0004	1.7473	2.3502	0.0000	0.0090	0.0163
56	63.166	0.0158	1.4680	0.0042	1.3132	0.0064	0.0000	0.0051
57	63.180	0.0158	1.1531	0.0001	-1.1840	0.0039	0.0000	0.0041
58	63.512	0.0157	0.0000	-0.1612	0.0000	0.0000	0.0001	0.0000
59	64.003	0.0156	-0.0020	-0.5988	0.0007	0.0000	0.0011	0.0000
60	64.339	0.0155	-0.0000	-1.1254	0.0000	0.0000	0.0037	0.0000
61	65.107	0.0154	-1.0551	-0.0210	1.1888	0.0033	0.0000	0.0042
62	65.192	0.0153	0.0016	0.3043	0.5961	0.0000	0.0003	0.0010
63	65.893	0.0152	-0.0078	-0.0572	0.0105	0.0000	0.0000	0.0000
64	69.245	0.0144	-0.0005	-0.9214	0.0006	0.0000	0.0025	0.0000
65	70.044	0.0143	-2.3140	-0.0000	-1.0324	0.0158	0.0000	0.0031
66	70.911	0.0141	-3.2315	-0.0000	-1.9392	0.0308	0.0000	0.0111
67	73.355	0.0136	-0.9182	0.2454	0.2391	0.0025	0.0002	0.0002
68	73.445	0.0136	-0.0085	1.5999	0.0018	0.0000	0.0075	0.0000
69	74.015	0.0135	0.0021	1.9142	0.0703	0.0000	0.0108	0.0000
70	74.156	0.0135	-0.0000	2.9214	-0.0000	0.0000	0.0252	0.0000
71	74.905	0.0134	0.0017	-3.8949	0.0017	0.0000	0.0447	0.0000
72	76.249	0.0131	-0.1108	0.0489	0.0287	0.0000	0.0000	0.0000
73	76.553	0.0131	0.0001	1.4453	0.0002	0.0000	0.0062	0.0000
74	76.599	0.0131	0.0000	3.2442	0.0000	0.0000	0.0310	0.0000
75	78.151	0.0128	-0.0001	-5.7802	-0.0001	0.0000	0.0985	0.0000
76	78.399	0.0128	0.1428	-1.8259	0.2005	0.0001	0.0098	0.0001
77	79.572	0.0126	-0.0000	-3.1388	-0.0000	0.0000	0.0291	0.0000
78	80.352	0.0124	-0.0611	-0.0303	0.0321	0.0000	0.0000	0.0000
79	80.590	0.0124	-0.4718	0.0055	-0.0030	0.0007	0.0000	0.0000
80	81.366	0.0123	0.0014	-1.7246	1.7171	0.0000	0.0088	0.0087
81	81.537	0.0123	-0.6787	0.0000	2.6001	0.0014	0.0000	0.0199
82	83.538	0.0120	2.6049	-0.0000	0.2580	0.0200	0.0000	0.0002
83	83.555	0.0120	0.0186	0.0064	-0.0224	0.0000	0.0000	0.0000
84	85.143	0.0117	-0.9456	0.0093	-0.0136	0.0026	0.0000	0.0000
85	86.260	0.0116	-0.0033	-0.7082	1.6260	0.0000	0.0015	0.0078
86	86.553	0.0116	0.1187	-0.0708	1.3196	0.0000	0.0000	0.0051
87	87.389	0.0114	-0.0038	-0.2207	-0.7108	0.0000	0.0001	0.0015
88	87.678	0.0114	1.8995	-0.0000	-1.1843	0.0106	0.0000	0.0041
89	90.219	0.0111	-0.3088	-0.2777	-0.2039	0.0003	0.0002	0.0001
90	93.121	0.0107	-0.5735	-0.1326	0.4279	0.0010	0.0001	0.0005
91	95.224	0.0105	0.0000	4.5908	0.5409	0.0000	0.0622	0.0009
92	95.967	0.0104	0.7252	-0.0000	-0.6981	0.0016	0.0000	0.0014
					Total	0.7887	0.4496	0.8526

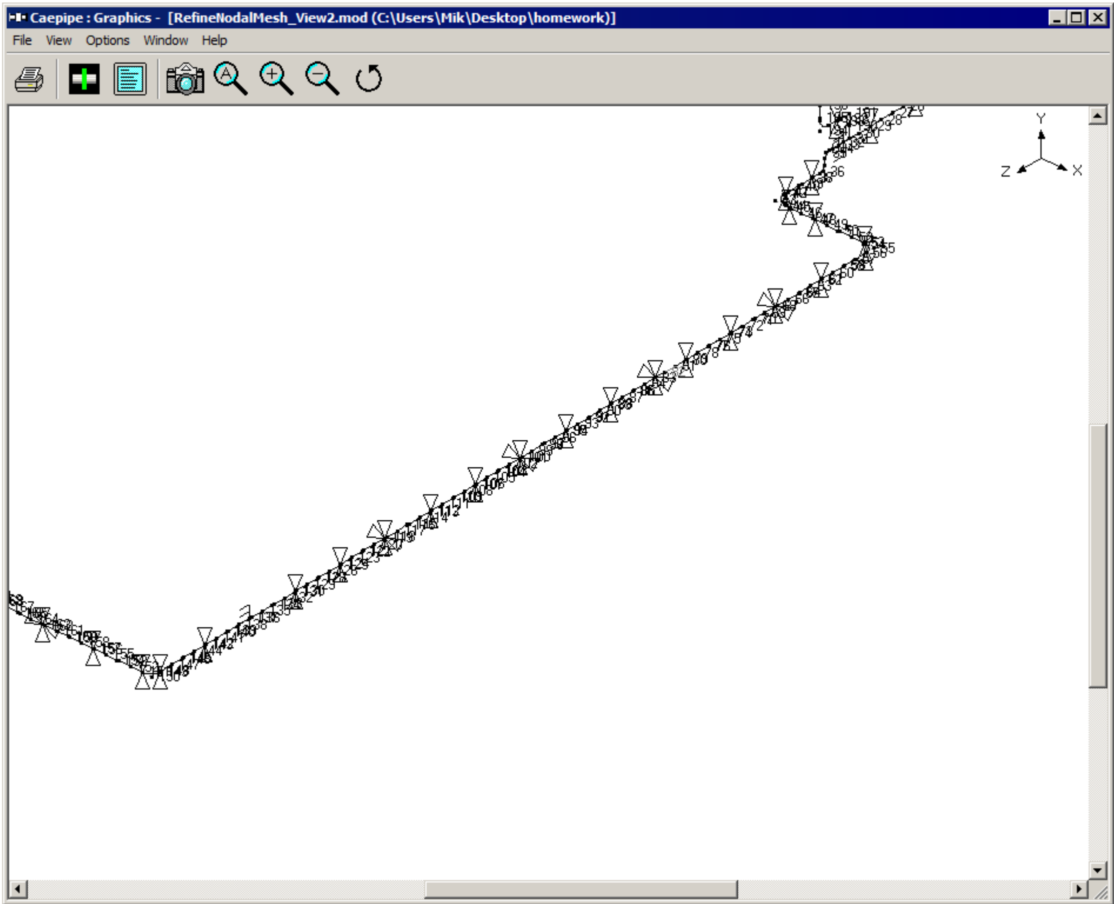
Now, the original model shown above was refined through the feature Layout window > Edit > Refine Nodal Mesh. The Mass modeling frequency was set to 110 Hz for refining the mesh for "Dynamic Analysis" as shown below.

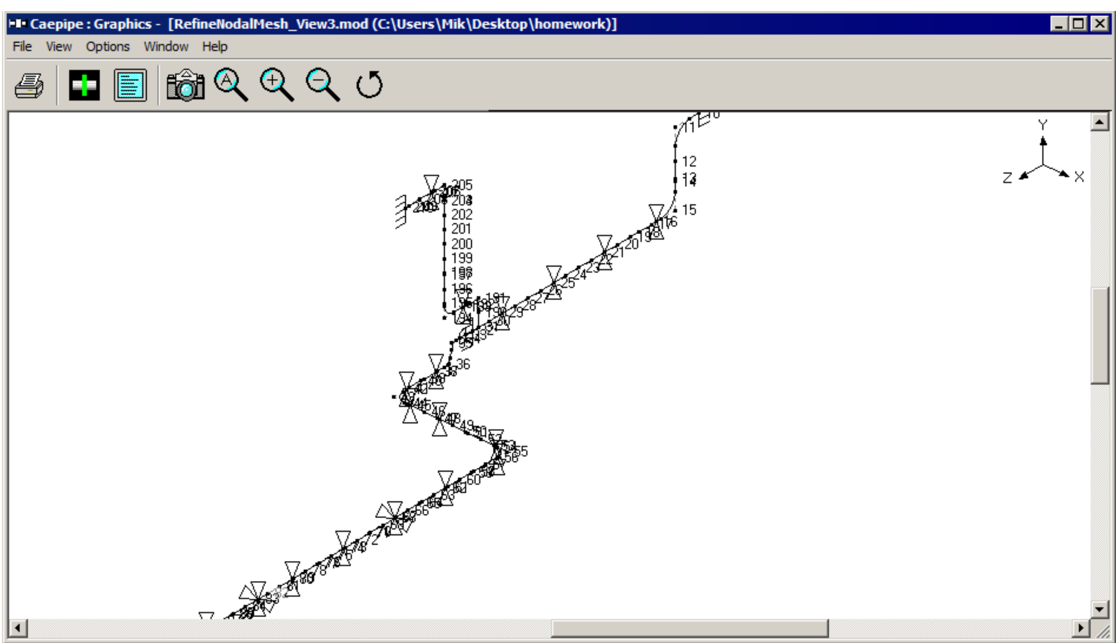


The resulting refined model with additional mass points added by CAEPIPE is shown in the snap shots below.


Caepipe : Graphics - [RefineNodalMesh.mod (C:\Users\Mik\Desktop\TechRefManual_V770-R2\Models)]
File View Options Window Help
Y
Z
X


Caepipe : Graphics - [RefineNodalMesh.mod (C:\Users\Mik\Desktop\TechRefManual_V770-R2\Models)]
File View Options Window Help
Y
Z
X


Caepipe : Graphics - [RefineNodalMesh_View2.mod (C:\Users\Mik\Desktop\homework)]
File View Options Window Help
Y
Z
X


Caepipe : Graphics - [RefineNodalMesh_View3.mod (C:\Users\Mik\Desktop\homework)]
File View Options Window Help
Y
Z
X
11
12
15
205
202
201
200
199
36
55

Modal analysis was then performed using CAEPIPE for the refined model (with additional mass points automatically added) for mass modeling frequency 110 Hz and found that the CAEPIPE was able to extract all the modes below the cut-off frequency 110 Hz specified in the analysis options. Please see the modal analysis results obtained from CAEPIPE shown below.

Caepipe dynamic_model_refined Page 1
Version 8.00 May 30,2018

Mode	Frequency (Hz)	Period (sec)	Participation factors X	Y	Z	Modal mass / Total mass X	Y	Z
1	1.755	0.5698	-5.7696	-0.0003	0.5155	0.0982	0.0000	0.0008
2	2.855	0.3502	6.7662	0.0000	-0.0002	0.1350	0.0000	0.0000
3	5.521	0.1811	-4.1344	0.0000	-0.0001	0.0504	0.0000	0.0000
4	6.151	0.1626	0.0878	-0.0000	-0.1088	0.0000	0.0000	0.0000
5	7.722	0.1295	0.4309	0.0000	-4.5502	0.0005	0.0000	0.0611
6	8.119	0.1232	0.0315	0.0001	0.3840	0.0000	0.0000	0.0004
7	9.824	0.1018	0.0839	-0.0000	-4.3165	0.0000	0.0000	0.0550
8	10.063	0.0994	2.3014	0.0000	0.0054	0.0156	0.0000	0.0000
9	10.134	0.0987	-0.6185	0.0003	2.3174	0.0011	0.0000	0.0158
10	11.111	0.0900	2.0288	0.0003	-8.3004	0.0121	0.0000	0.2032
11	11.613	0.0861	-7.1492	0.0006	-4.4135	0.1508	0.0000	0.0575
12	12.635	0.0791	-0.0006	1.5513	-3.8503	0.0000	0.0071	0.0437
13	13.233	0.0756	2.6877	0.0000	-0.0007	0.0213	0.0000	0.0000
14	13.640	0.0733	2.9690	0.0034	-4.0182	0.0260	0.0000	0.0476
15	15.725	0.0636	3.7891	-0.0000	0.3498	0.0423	0.0000	0.0004
16	15.805	0.0633	3.1354	0.0024	0.9356	0.0290	0.0000	0.0026
17	16.409	0.0609	2.2658	-0.0169	-2.5046	0.0151	0.0000	0.0185
18	17.349	0.0576	-0.0000	-2.7191	-3.1311	0.0000	0.0218	0.0289
19	20.661	0.0484	0.0007	3.2479	1.4079	0.0000	0.0311	0.0058
20	22.737	0.0440	0.2479	0.0001	-0.0877	0.0002	0.0000	0.0000
21	23.733	0.0421	0.7055	-0.0000	-0.0963	0.0015	0.0000	0.0000
22	23.782	0.0420	-2.9878	0.0000	0.0053	0.0263	0.0000	0.0000
23	26.507	0.0377	0.1027	0.0005	-0.1910	0.0000	0.0000	0.0001
24	27.187	0.0368	-0.6336	0.0000	-0.0045	0.0012	0.0000	0.0000
25	29.021	0.0345	-1.1191	0.0000	0.0002	0.0037	0.0000	0.0000
26	29.061	0.0344	0.3323	-0.0000	0.1679	0.0003	0.0000	0.0001
27	30.289	0.0330	-0.4261	0.0036	-1.0079	0.0005	0.0000	0.0030
28	31.215	0.0320	1.0457	-0.0087	2.3123	0.0032	0.0000	0.0158
29	33.583	0.0298	0.4478	-0.0064	1.5750	0.0006	0.0000	0.0073
30	36.362	0.0275	-1.1690	0.0000	-0.0052	0.0040	0.0000	0.0000
31	37.505	0.0267	-0.0033	1.4151	-2.1430	0.0000	0.0059	0.0135
32	37.885	0.0264	0.4183	0.0121	-2.4989	0.0005	0.0000	0.0184
33	40.194	0.0249	0.1951	-0.0000	-1.5005	0.0001	0.0000	0.0066
34	40.968	0.0244	0.0001	-1.1979	6.4128	0.0000	0.0042	0.1213
35	43.038	0.0232	0.3511	-0.0019	-1.2983	0.0004	0.0000	0.0050
36	43.372	0.0231	-0.4795	0.0000	-0.0105	0.0007	0.0000	0.0000
37	46.870	0.0213	0.0021	-0.0008	0.0010	0.0000	0.0000	0.0000
38	47.541	0.0210	-0.0144	-0.7462	-0.0080	0.0000	0.0016	0.0000
39	48.377	0.0207	-0.2963	-0.0021	-0.3487	0.0003	0.0000	0.0004
40	50.010	0.0200	-0.4830	0.0001	-1.8955	0.0007	0.0000	0.0106
41	50.132	0.0199	-0.0043	-0.3850	-0.0040	0.0000	0.0004	0.0000
42	50.285	0.0199	0.0000	1.2526	0.0000	0.0000	0.0046	0.0000
43	51.219	0.0195	1.2828	0.0000	-1.6462	0.0049	0.0000	0.0080
44	51.778	0.0193	1.4654	0.0565	1.8643	0.0063	0.0000	0.0103
45	53.561	0.0187	0.0000	-1.1367	-0.0000	0.0000	0.0038	0.0000
46	53.672	0.0186	-0.1396	0.0329	0.4238	0.0001	0.0000	0.0005
47	54.731	0.0183	0.0027	-1.2178	0.0180	0.0000	0.0044	0.0000
48	56.221	0.0178	-0.0024	-2.0037	0.0429	0.0000	0.0118	0.0000
49	57.051	0.0175	-1.6501	-0.0050	-0.0543	0.0080	0.0000	0.0000
50	57.365	0.0174	-0.2648	-0.0208	0.4221	0.0002	0.0000	0.0005
51	58.161	0.0172	-0.0620	-1.7053	0.0076	0.0000	0.0086	0.0000
52	58.480	0.0171	-2.8218	0.0672	0.2197	0.0235	0.0000	0.0001
53	59.590	0.0168	0.0011	-1.6797	-2.3900	0.0000	0.0083	0.0168

```
Caepipe                       dynamic_model_refined               Page 2
Version 8.00                                                      May 30,2018
--------------------------------------------------------------------------------
     Frequency  Period    Participation factors    Modal mass / Total mass
 Mode    (Hz)    (sec)      X        Y        Z        X       Y       Z
--------------------------------------------------------------------------------
  54    59.697  0.0168  1.3631  -0.0003  -0.0002   0.0055  0.0000  0.0000
  55    60.969  0.0164 -0.0001   1.4746   0.5326   0.0000  0.0064  0.0008
  56    61.166  0.0163 -0.0003   0.3240  -0.0009   0.0000  0.0003  0.0000
  57    63.791  0.0157  1.1014  -0.0167   1.6815   0.0036  0.0000  0.0083
  58    65.476  0.0153  0.0000  -0.0239  -0.0000   0.0000  0.0000  0.0000
  59    66.097  0.0151 -2.0302  -0.0513   0.5933   0.0122  0.0000  0.0010
  60    66.996  0.0149  0.6629   0.0000  -1.1896   0.0013  0.0000  0.0042
  61    67.413  0.0148  0.0051  -0.4949  -0.0031   0.0000  0.0007  0.0000
  62    67.702  0.0148  0.0000  -1.4164  -0.0000   0.0000  0.0059  0.0000
  63    69.973  0.0143 -0.0065  -0.2969   0.0052   0.0000  0.0003  0.0000
  64    70.117  0.0143  0.0002  -2.2385   0.7487   0.0000  0.0148  0.0017
  65    70.175  0.0143 -2.4152  -0.0000  -0.9261   0.0172  0.0000  0.0025
  66    74.234  0.0135  3.4185   0.0003   1.5639   0.0345  0.0000  0.0072
  67    74.560  0.0134  0.0042   1.0701  -0.0017   0.0000  0.0034  0.0000
  68    74.843  0.0134 -1.3329   0.8236   0.1534   0.0052  0.0020  0.0001
  69    76.927  0.0130 -0.4107  -1.5744   0.2386   0.0005  0.0073  0.0002
  70    78.576  0.0127 -0.1023   0.0017  -0.0013   0.0000  0.0000  0.0000
  71    80.082  0.0125  0.0006  -0.5746   1.6438   0.0000  0.0010  0.0080
  72    80.485  0.0124  0.0002  -2.1498   0.0002   0.0000  0.0136  0.0000
  73    81.701  0.0122 -0.2466  -0.0537   0.0757   0.0002  0.0000  0.0000
  74    81.749  0.0122 -0.0002   3.4973   0.0000   0.0000  0.0361  0.0000
  75    82.787  0.0121 -0.0004   4.3124  -0.0023   0.0000  0.0549  0.0000
  76    84.316  0.0119  1.6969   0.0000  -3.1197   0.0085  0.0000  0.0287
  77    85.065  0.0118 -0.0026  -0.4712   0.8256   0.0000  0.0007  0.0020
  78    85.440  0.0117  0.0000   3.7852  -0.0000   0.0000  0.0423  0.0000
  79    85.507  0.0117 -0.0001   1.4996   0.0009   0.0000  0.0066  0.0000
  80    87.100  0.0115 -0.0680   0.0853  -1.2077   0.0000  0.0000  0.0043
  81    87.358  0.0114 -0.0000   3.5186  -0.0000   0.0000  0.0365  0.0000
  82    87.548  0.0114  1.1884  -0.0035  -0.0409   0.0042  0.0000  0.0000
  83    87.820  0.0114  0.1975  -0.0174   0.6688   0.0001  0.0000  0.0013
  84    88.217  0.0113 -0.0001   6.8293  -0.0003   0.0000  0.1376  0.0000
  85    88.587  0.0113  0.0003  -0.6062  -1.1192   0.0000  0.0011  0.0037
  86    89.553  0.0112  2.9233  -0.0000   0.1706   0.0252  0.0000  0.0001
  87    91.995  0.0109 -0.2639  -0.2144  -0.3238   0.0002  0.0001  0.0003
  88    92.948  0.0108  0.1126   0.0508   0.0909   0.0000  0.0000  0.0000
  89    93.055  0.0107  1.1698  -0.0041   0.6341   0.0040  0.0000  0.0012
  90   102.283  0.0098  0.0000  -0.0302  -0.0000   0.0000  0.0000  0.0000
  91   103.208  0.0097  0.0131   5.0475   0.3381   0.0000  0.0751  0.0003
  92   103.322  0.0097 -0.6291  -0.0242   0.3139   0.0012  0.0000  0.0003
  93   109.288  0.0092 -0.7946  -0.1550   0.6055   0.0019  0.0001  0.0011
  94   110.299  0.0091  1.2405  -0.0000   0.3050   0.0045  0.0000  0.0003
                                        ------------------------------
                                          Total   0.8148  0.5606  0.8574
```

General

During an overpressure event, the discharge of a PRV imposes a load, referred to as a reaction force, on the collective installation. The flowrate and associated reaction force increase from nominally zero to some value, remain relatively constant at that value for the duration of the release, and then decrease to zero again, i.e., when the relief valve opens, the discharge fluid creates a jet force that acts on the piping system. This force increases from zero to its full value over a time frame similar to the opening time of the valve. The relief valve remains open until sufficient fluid is vented to relieve the overpressure situation. As the valve closes, the reduction in flow reduces the jet force to zero.

Simplified Analysis Approach

American Petroleum Institute's API 520, Part II (1994), provides a basis for calculation of the reaction force in the event of a vapor or a two-phase release directly to the atmosphere. There is no discussion in this section of API 520, Part II, about the reaction force developed during a liquid release. Furthermore, no guidance is presented with respect to applying these results or determining if an installation is acceptable; instead, the burden is placed on the designer to ensure that the installation is appropriately designed. While this may be reasonable for the design of new facilities, evaluating the adequacy of existing facilities becomes much more complicated.

The formula (section 2.4.1.1) in US Customary units from API 520, Part II (1994), for vapor relief devices discharging to the atmosphere, is shown below:

$$F = \frac{W}{366}\sqrt{\frac{kT}{(k+1)M}} + (AP)$$

where,

F = Reaction force at the point of discharge to the atmosphere,(lbf.)
k = Ratio of specific heats (CP/CV) at the outlet conditions
W = Flow rate of any gas or vapor, pound mass (lbm.)/hr
CP = Specific heat at constant pressure
CV = Specific heat at constant volume
T = Temperature at the outlet, °R
M = Molecular weight of the process fluid
A = Area of the outlet at the point of discharge, in2
P = Static pressure within the outlet at the point of discharge, psig
Using the reaction force computed from the above formula along with the following PRV parameters, namely

- Valve Opening Time,
- Valve Closing Time and
- Relief duration (all obtained from the PRV manufacturer),

One can generate a PRV load profile and apply it in CAEPIPE to perform a simplified analysis.

Detailed Analysis Approach

Section 2.4.2 of API 520, Part II (1994) also states the following.

"Pressure relief devices that relieve under steady-state flow conditions into a closed system usually do not create large forces and bending moments on the exhaust system. Only at points of sudden expansion will there be any significant reaction forces to be calculated. Closed discharge systems, however, do not lend themselves to simplified analytical techniques. A complex time history analysis of the piping system may be required to obtain the true values of the reaction forces and associated moments."

Such complex time history analysis of the piping system can be carried out as follows.

- Perform a fluid transient analysis on the piping system using a fluid dynamics software tool such as, "PipeNet", "RELAP", "ROLAST", etc.
- Apply the resulting output obtained (forces as a function of frequency) at the bend node after the relief valve in a pipe stress analysis software (CAEPIPE).
- Compute forces, moments and stresses in the piping system due to this loading.

As one can see, this method is detailed, time consuming and expensive and hence, not covered here.

Example 1:

Step 1:

By assuming the following data, one can apply the relief valve loading in CAEPIPE. Please see the model below for details.

1. Reaction force (F) computed using the formula above = 6854 lb.

2. Relief Valve Opening time = 8 ms (milliseconds)

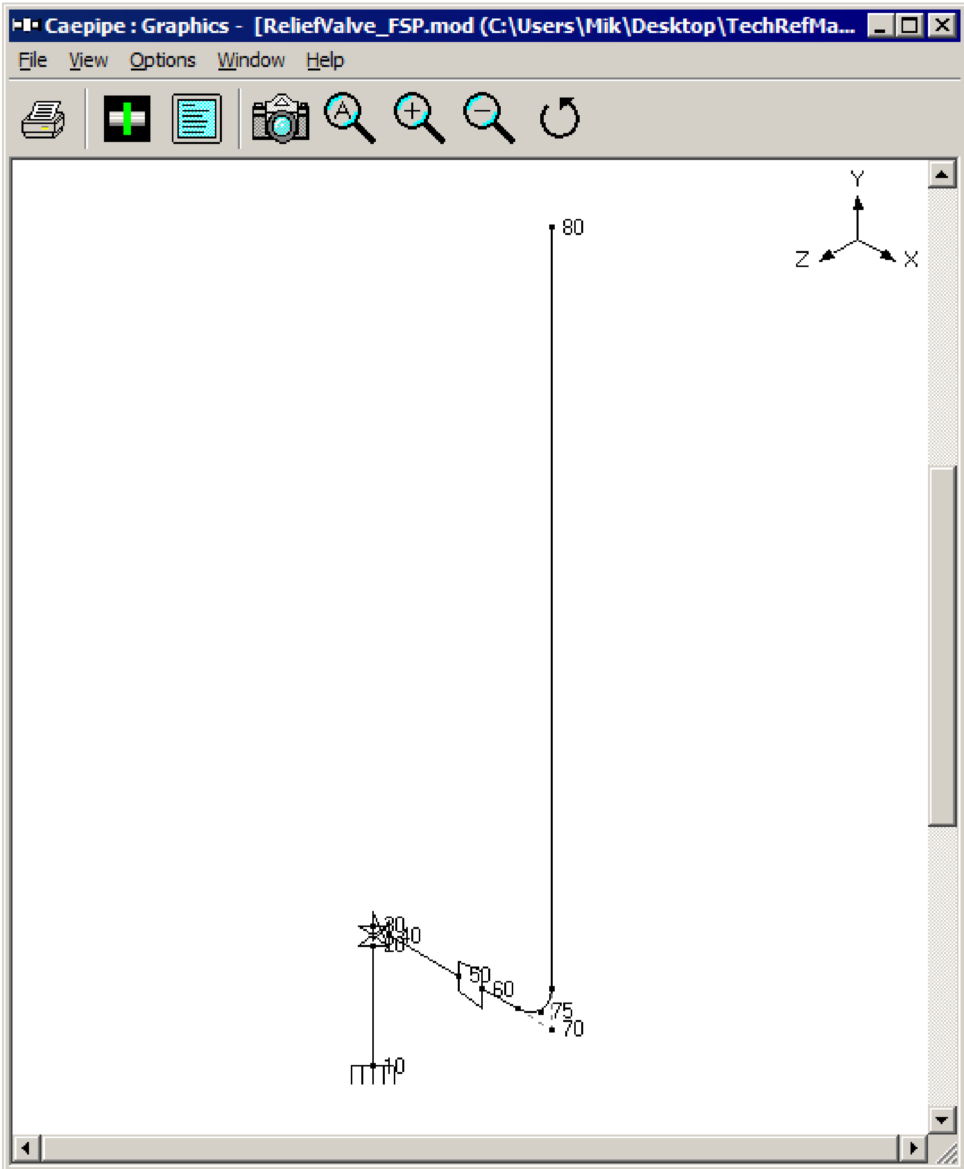
3. Relief Valve Closing time = 8 ms

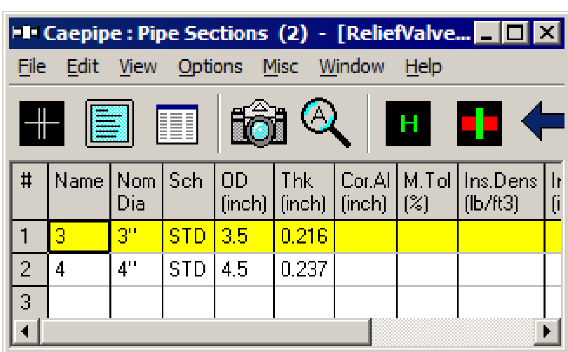
4. Relief duration = 1 s

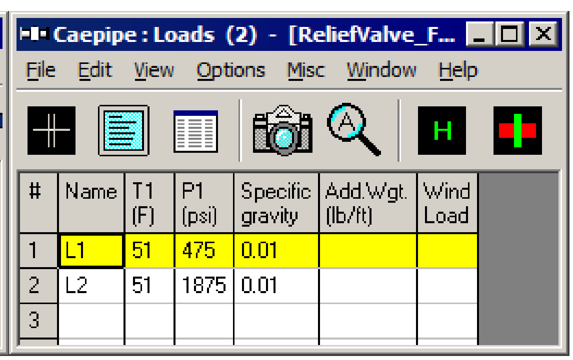
6. Pressure = 475 psig

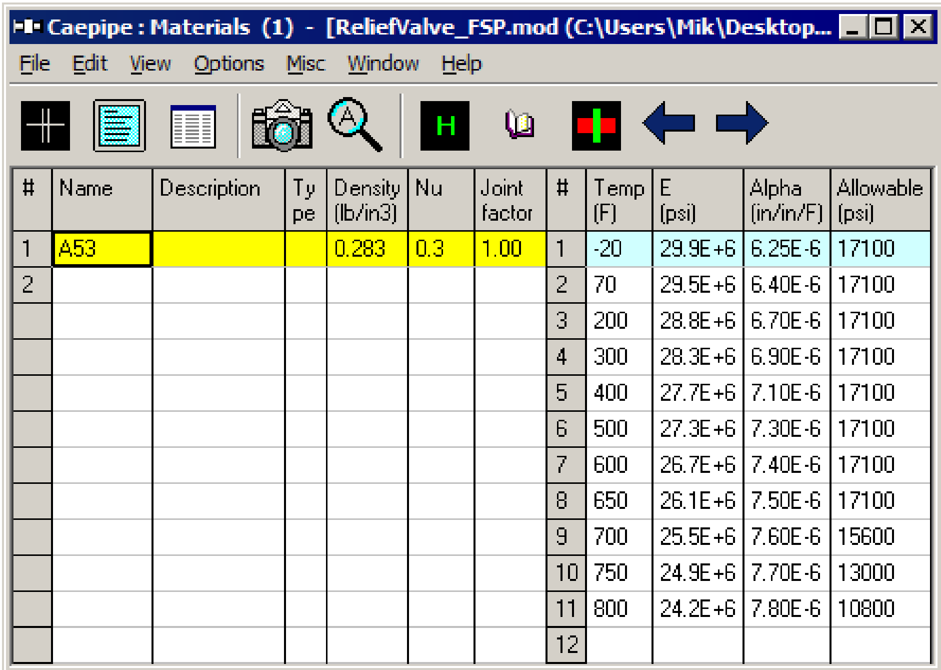
7. Temperature = 51°F

The steps followed in generating the model are given below.

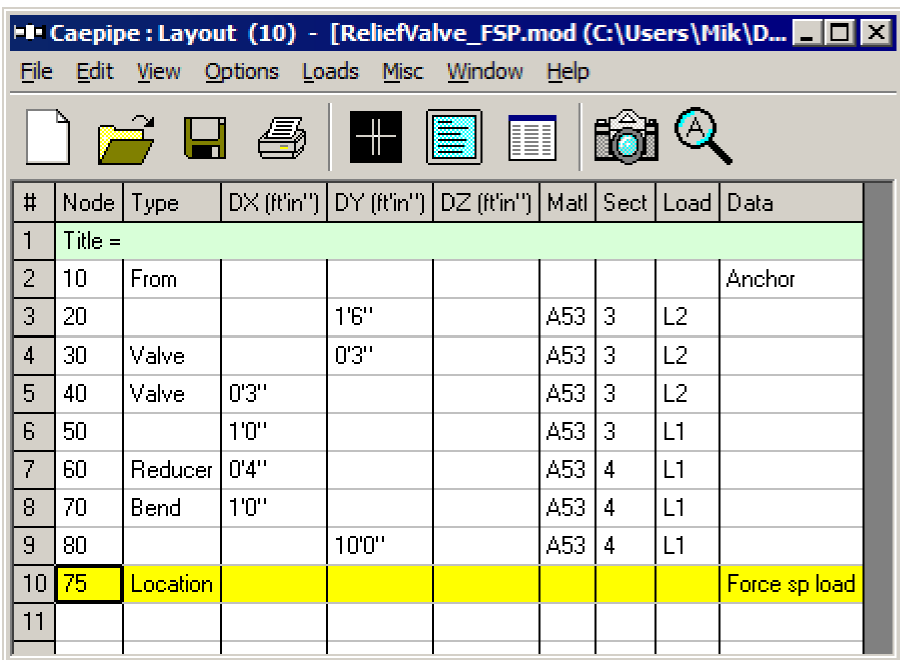

Caepipe : Graphics - [ReliefValve_FSP.mod (C:\Users\Mik\Desktop\TechRefMa...
File View Options Window Help
80
Y
Z
X
50
60
75
70
10


Caepipe : Pipe Sections (2) - [ReliefValve...
File Edit View Options Misc Window Help
Name Nom Dia Sch OD (inch) Thk (inch) Cor.Al (inch) M.Tol (%) Ins.Dens (lb/ft3)
1 3 3" STD 3.5 0.216
2 4 4" STD 4.5 0.237
3

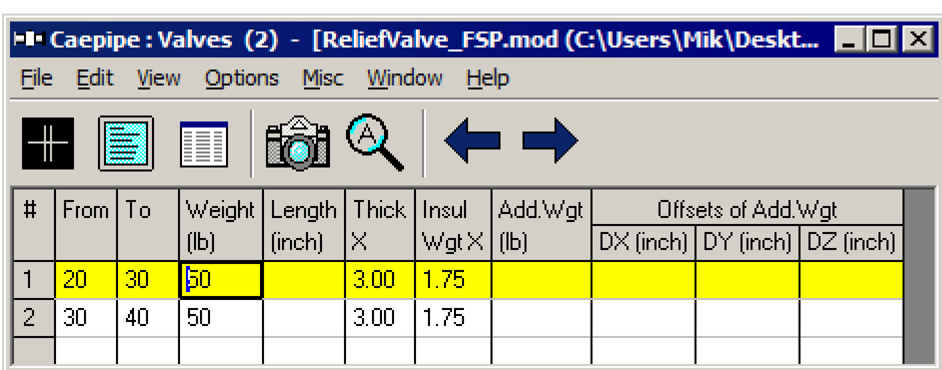

Caepipe : Loads (2) - [ReliefValve_F...
File Edit View Options Misc Window Help
Name T1 (F) P1 (psi) Specific gravity Add.Wgt. (lb/ft) Wind Load
1 L1 51 475 0.01
2 L2 51 1875 0.01
3



#	Name	Description	Type	Density (lb/in3)	Nu	Joint factor	#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	A53			0.283	0.3	1.00	1	-20	29.9E+6	6.25E-6	17100
2							2	70	29.5E+6	6.40E-6	17100
							3	200	28.8E+6	6.70E-6	17100
							4	300	28.3E+6	6.90E-6	17100
							5	400	27.7E+6	7.10E-6	17100
							6	500	27.3E+6	7.30E-6	17100
							7	600	26.7E+6	7.40E-6	17100
							8	650	26.1E+6	7.50E-6	17100
							9	700	25.5E+6	7.60E-6	15600
							10	750	24.9E+6	7.70E-6	13000
							11	800	24.2E+6	7.80E-6	10800
							12				



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title =								
2	10	From							Anchor
3	20			1'6"		A53	3	L2	
4	30	Valve		0'3"		A53	3	L2	
5	40	Valve	0'3"			A53	3	L2	
6	50		1'0"			A53	3	L1	
7	60	Reducer	0'4"			A53	4	L1	
8	70	Bend	1'0"			A53	4	L1	
9	80			10'0"		A53	4	L1	
10	75	Location							Force sp load
11									

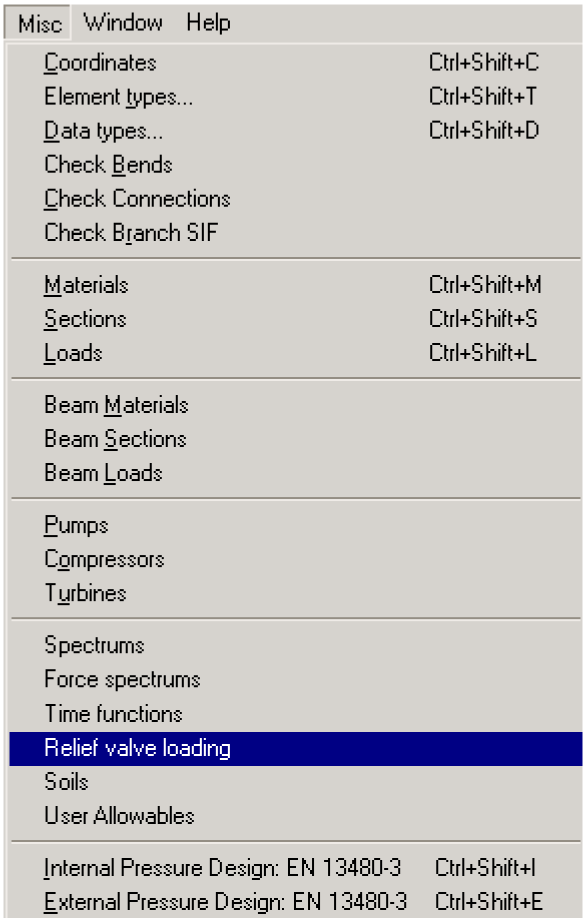


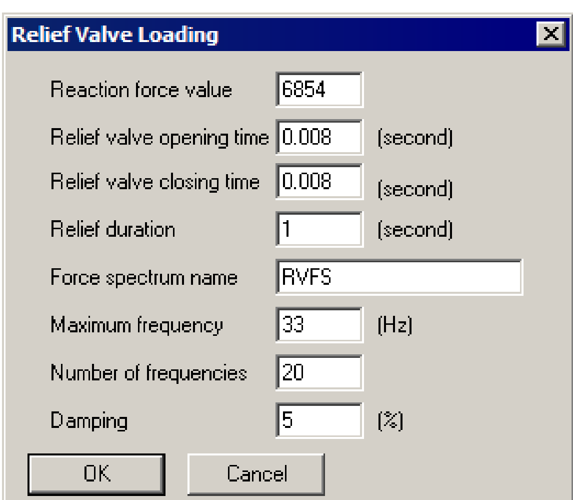
#	From	To	Weight (lb)	Length (inch)	Thick X	Insul Wgt X	Add.Wgt (lb)	Offsets of Add.Wgt DX (inch)	DY (inch)	DZ (inch)
1	20	30	50		3.00	1.75				
2	30	40	50		3.00	1.75				

Step 2:

After creating your piping model (with node 75 being the center node of the discharge bend where the PRV reaction force will be applied),

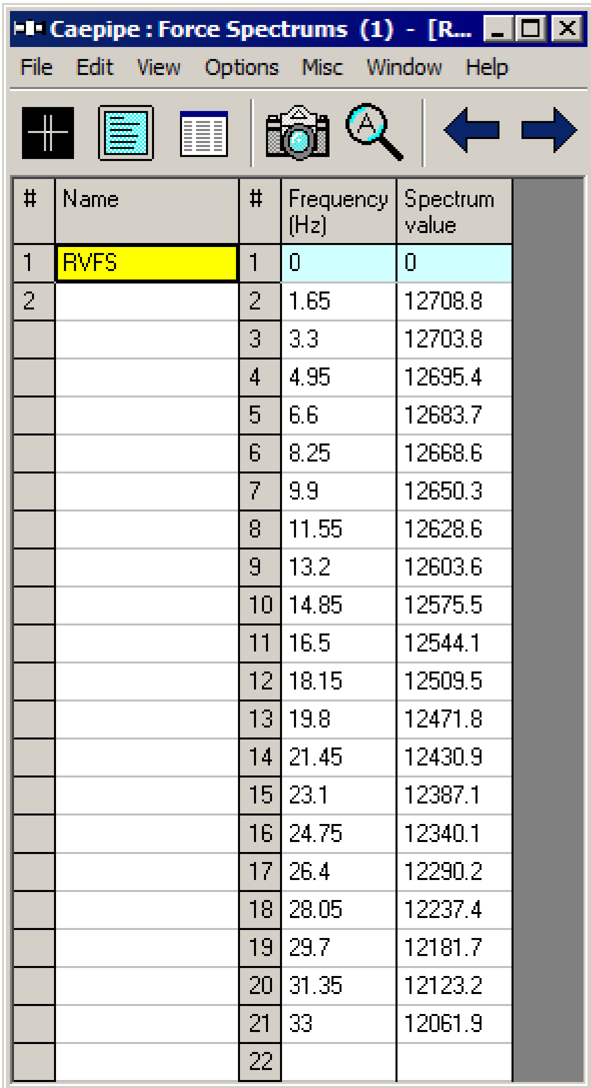
Select “Relief valve loading” from CAEPIPE Layout window > Misc and enter the data in the dialog box as shown in the figure below.





Step 3:

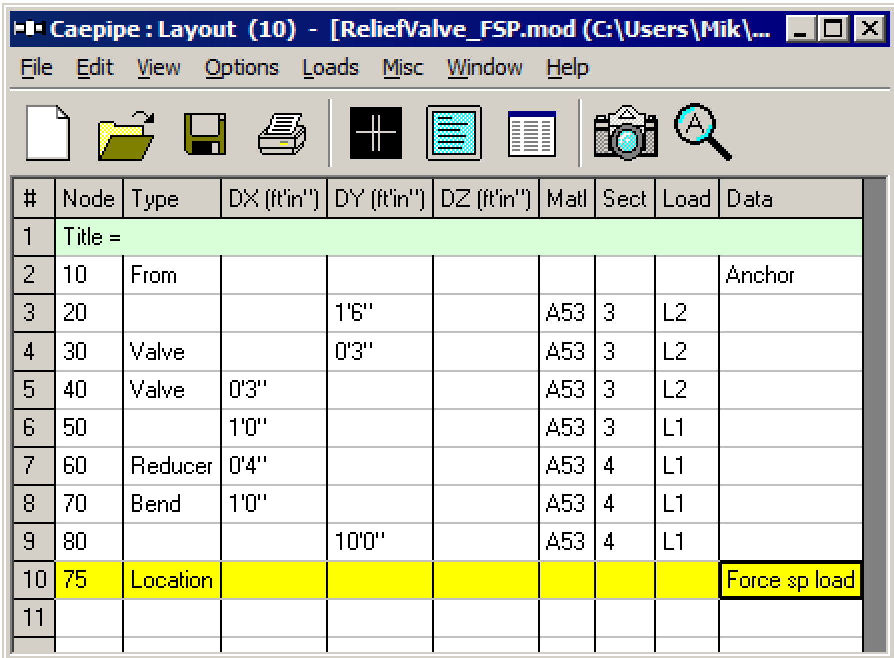
After entering the data as shown in the dialog above, press the button "OK". Using the above input values for Relief Valve Loading, CAEPIPE internally generates a time-history loading function, which is then applied on a single degree-freedom spring-mass system with each intermediate frequency (between 0.0 Hz and the maximum frequency) to generate the "Force Spectrum Load" shown below.



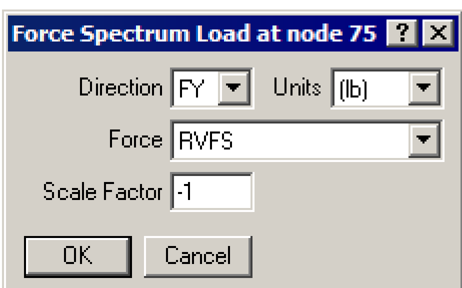
#	Name	#	Frequency (Hz)	Spectrum value
1	RVFS	1	0	0
2		2	1.65	12708.8
		3	3.3	12703.8
		4	4.95	12695.4
		5	6.6	12683.7
		6	8.25	12668.6
		7	9.9	12650.3
		8	11.55	12628.6
		9	13.2	12603.6
		10	14.85	12575.5
		11	16.5	12544.1
		12	18.15	12509.5
		13	19.8	12471.8
		14	21.45	12430.9
		15	23.1	12387.1
		16	24.75	12340.1
		17	26.4	12290.2
		18	28.05	12237.4
		19	29.7	12181.7
		20	31.35	12123.2
		21	33	12061.9
		22		

Step 4:

Apply the Force Spectrum Load thus generated at the bend center node 75 after the relief valve in downward direction (-FY by specifying negative Scale Factor) as shown below.

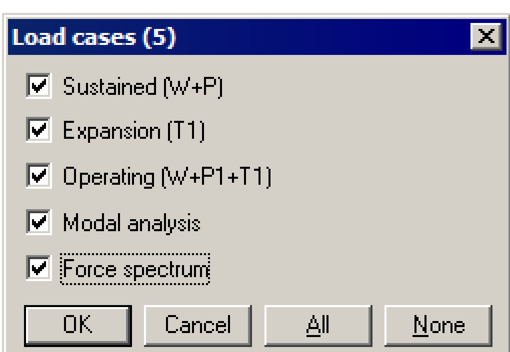


#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title =								
2	10	From							Anchor
3	20			1'6"		A53	3	L2	
4	30	Valve		0'3"		A53	3	L2	
5	40	Valve	0'3"			A53	3	L2	
6	50		1'0"			A53	3	L1	
7	60	Reducer	0'4"			A53	4	L1	
8	70	Bend	1'0"			A53	4	L1	
9	80			10'0"		A53	4	L1	
10	75	Location							Force sp load
11									



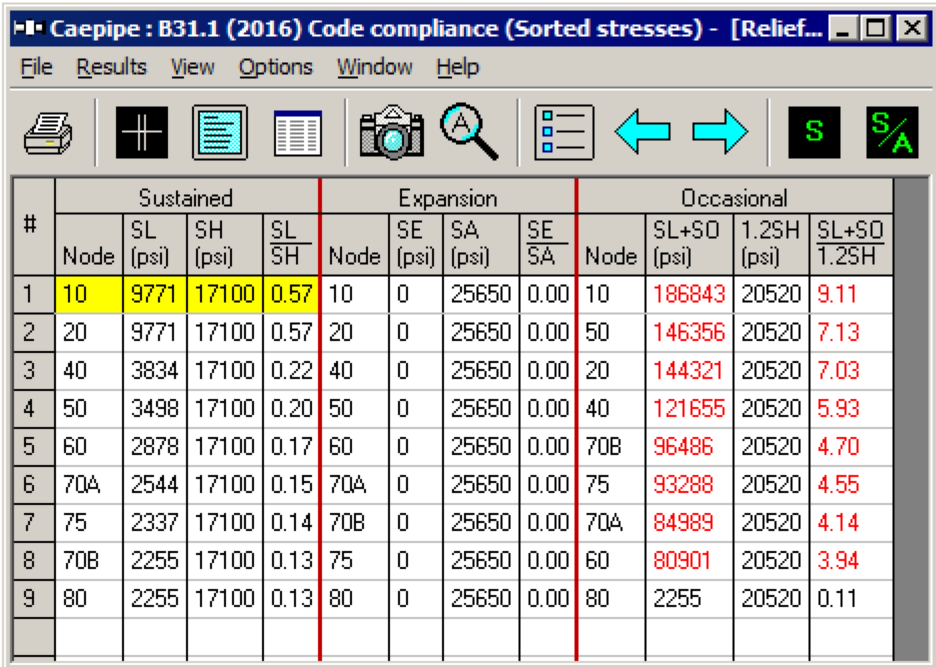
Step 5:

Check "Force Spectrum" for analysis through Layout window > Load cases. Click on OK.



Step 6:

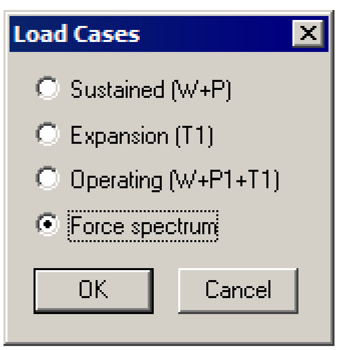
Save and Analyze the model. After analysis, CAEPIPE displays Occasional stresses which include the effects of the PRV load.

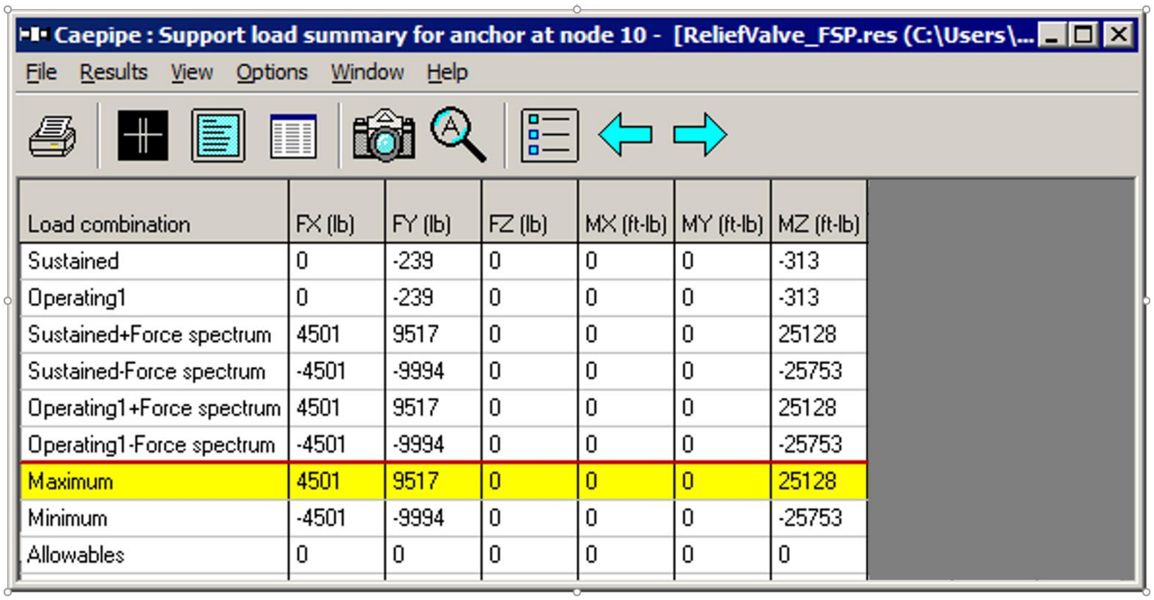


#	Sustained				Expansion				Occasional			
	Node	SL (psi)	SH (psi)	SL/SH	Node	SE (psi)	SA (psi)	SE/SA	Node	SL+SO (psi)	1.2SH (psi)	SL+SO/1.2SH
1	10	9771	17100	0.57	10	0	25650	0.00	10	186843	20520	9.11
2	20	9771	17100	0.57	20	0	25650	0.00	50	146356	20520	7.13
3	40	3834	17100	0.22	40	0	25650	0.00	20	144321	20520	7.03
4	50	3498	17100	0.20	50	0	25650	0.00	40	121655	20520	5.93
5	60	2878	17100	0.17	60	0	25650	0.00	70B	96486	20520	4.70
6	70A	2544	17100	0.15	70A	0	25650	0.00	75	93288	20520	4.55
7	75	2337	17100	0.14	70B	0	25650	0.00	70A	84989	20520	4.14
8	70B	2255	17100	0.13	75	0	25650	0.00	60	80901	20520	3.94
9	80	2255	17100	0.13	80	0	25650	0.00	80	2255	20520	0.11

Step 7:

Another load case called "Force Spectrum" will be available for which you can study displacements, support loads, support load summary (for sizing supports), etc.



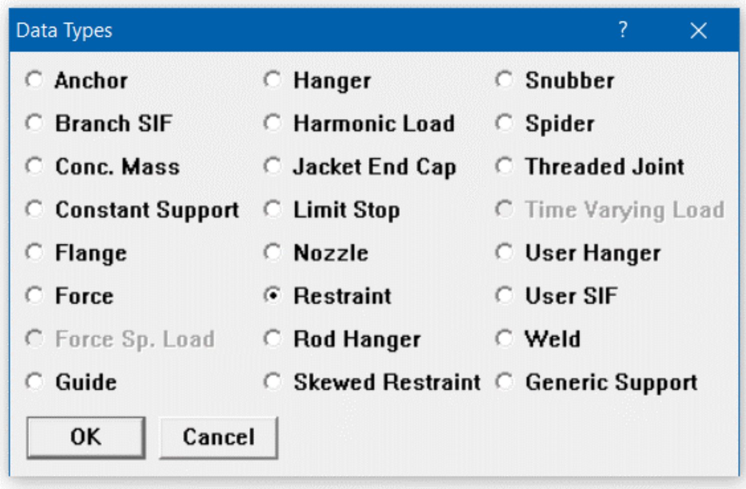


Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
Sustained	0	-239	0	0	0	-313
Operating1	0	-239	0	0	0	-313
Sustained+Force spectrum	4501	9517	0	0	0	25128
Sustained-Force spectrum	-4501	-9994	0	0	0	-25753
Operating1+Force spectrum	4501	9517	0	0	0	25128
Operating1-Force spectrum	-4501	-9994	0	0	0	-25753
Maximum	4501	9517	0	0	0	25128
Minimum	-4501	-9994	0	0	0	-25753
Allowables	0	0	0	0	0	0

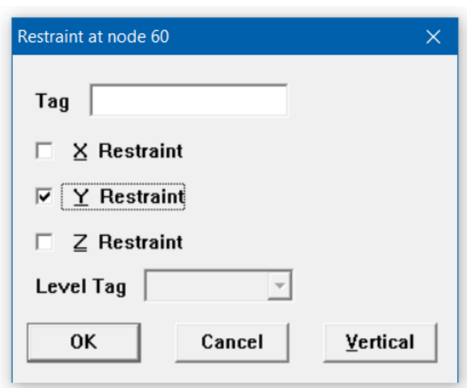
Restraint

This support type is a convenient way to specify a translational two-way rigid restraint in the global X, Y and Z directions.

A restraint is input by typing "re" in the Data column or selecting "Restraint" from the Data Types dialog. Alternately, simply typing "X" or "Y" or "Z" in the data type field inputs a restraint in the respective direction and moves the cursor to the next row.



The Restraint dialog is shown.



Use the check boxes to apply the restraint in a particular direction (both ways). Click on the vertical button for a rigid vertical restraint. All three directions may be checked too.

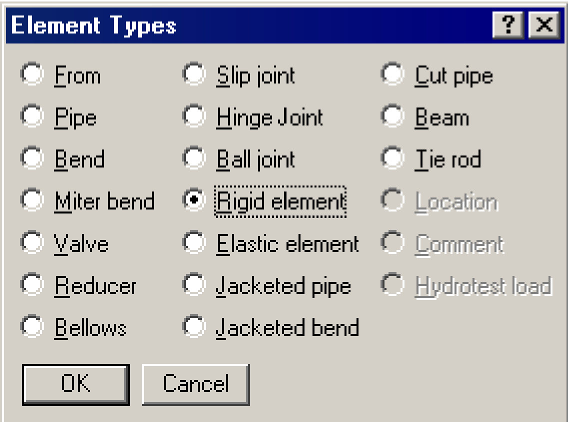
Rigid restraint has a stiffness of 1×10^{12} (lb/in.).

Rigid Element

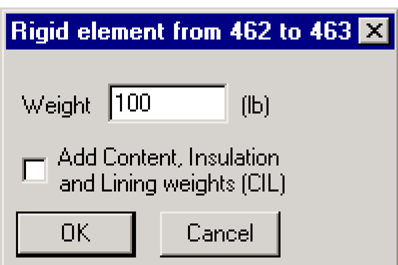
Use this element to model any "stiff" (relative to pipe) inline component.

The stiffnesses of 1 x 10^{12}(lb/inch) in translational directions (axial and shear), and 1 x 10^{12} (inch-lb./rad.) in rotational directions (bending and torsional) are used.

A rigid element is input by typing "ri" in the Type column or selecting "Rigid element" from the Element Types dialog.



The Rigid element dialog is shown.



Weight

The required input is weight. It is applied as a distributed load along the length of the rigid element. To this empty weight, the Additional weightspecified under Load column is added (to include weight of snow etc.).

Weight is to be input in lbf or kgf and NOT in mass units. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

For Sustained load case analysis, weights of content, insulation and lining (as calculated using insulation thickness and its density as well as lining thickness and density) are added internally in CAEPIPE when the option "Add Content, Insulation and Lining weights (CIL)" is turned ON.

For the Empty Weight load case analysis, only weights of insulation and lining will be included, while the weight of content will be excludedwhen the option "Add Content, Insulation and Lining weights (CIL)" is turned ON.

Material density does not affect the weight of the rigid element.

Thermal expansion of the rigid element is calculated using the coefficient of thermal expansion from the Material column and temperatures from the Load column. Wind load is calculated using the section properties inclusive of insulation thickness.

Rigid, weightless Elements

These may be needed when you want to account for some hard-to-model element's thermal growth, or to connect the center line of a large pipe to its outside surface, again to account for its (radial) thermal growth/contraction (which impacts the branch line), or to model a rigid, massless link between two points on the stress model.

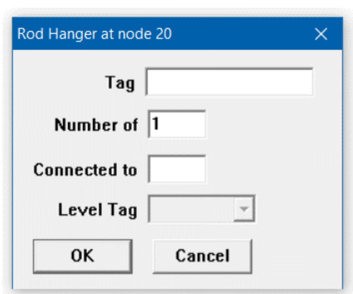
To model any of these, input a rigid element, type zero for weight and ensure that the corresponding Load (specified on the Layout window under the Load column) used for this element does not have any Additional weight specified.

Rod Hanger

A rod hanger is a rigid one-way vertical support. The rod hanger node is rigidly supported against downward movement but able to move freely in the upward direction. That is, the rod hanger is rigid in tension (downward movement) and has no stiffness in compression (upward movement). *CAEPIPE considers a rod hanger to always act in the vertical direction.*

A rod hanger is input by typing "ro" in the Data column or selecting "Rod Hanger" from the Data Types dialog.

By default, one rod hanger without a connected node is input. The number of hangers and the node to which it may be connected to may be specified in the Rod Hanger dialog.



Number of Hangers

The number of hangers is the number of separate rod hangers connected in parallel at this node.

Connected to Node

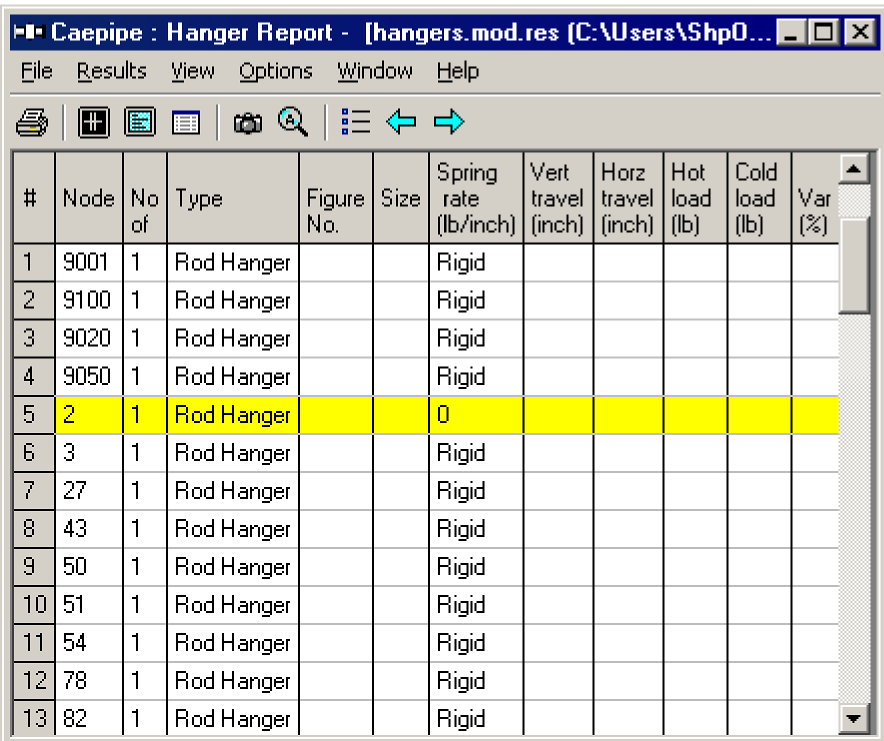
By default the rod hanger is connected to a fixed *ground* point which is not a part of the piping system. A rod hanger can be connected to another node in the piping system by entering the other node number in the "Connected to" nodefield. This node must be directly above the rod hanger node.

A rod hanger in CAEPIPE functions as a vertical limit stop, that is, it functions as a nonlinear one-way restraint. It is rigid in -Y direction and fully flexible in +Y direction (in a Y-vertical system). The rod hanger offers no resistance in +Y direction.

Rod hanger results are included in the hanger report, which reports results for the first operating case (W+P1+T1). In the hanger report, a rod hanger's spring rate may be shown either as Rigid or zero, the latter potentially confusing to the user.

It simply means that there is possibly liftoff at the rod hanger location for the first operating case. You can confirm this by studying vertical displacement (Y or Z) at the rod hanger for the first operating case (which will be 0 or positive). If this vertical displacement is zero, it means the rod hanger is in tension and its sprig rate is shown as Rigid; on the other hand, if this vertical displacement is positive, then the rod hanger is in compression and its spring rate is shown as zero. You can find reports for other operating load cases under Support Loads > Other Supports > Rod Hangers.

Liftoff (i.e., zero spring rate and a positive operating condition displacement) indicates that the rod hanger may not be needed and hence could be removed. You will need to study the effect on the system at other supports after removing the rod hanger.

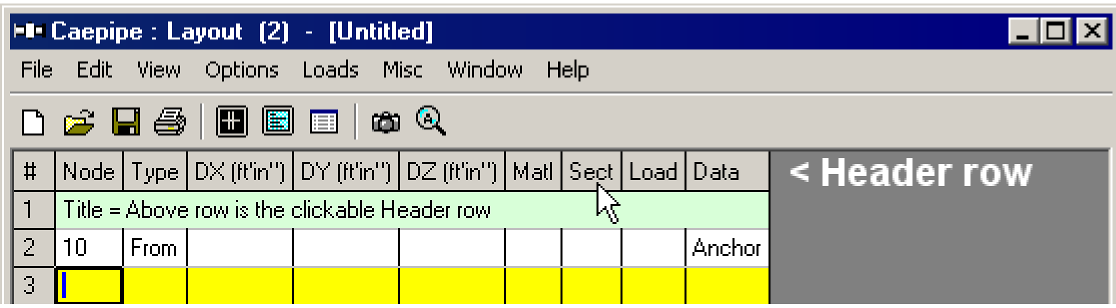


#	Node	No of	Type	Figure No.	Size	Spring rate (lb/inch)	Vert travel (inch)	Horz travel (inch)	Hot load (lb)	Cold load (lb)	Var (%)
1	9001	1	Rod Hanger			Rigid					
2	9100	1	Rod Hanger			Rigid					
3	9020	1	Rod Hanger			Rigid					
4	9050	1	Rod Hanger			Rigid					
5	2	1	Rod Hanger			0					
6	3	1	Rod Hanger			Rigid					
7	27	1	Rod Hanger			Rigid					
8	43	1	Rod Hanger			Rigid					
9	50	1	Rod Hanger			Rigid					
10	51	1	Rod Hanger			Rigid					
11	54	1	Rod Hanger			Rigid					
12	78	1	Rod Hanger			Rigid					
13	82	1	Rod Hanger			Rigid					

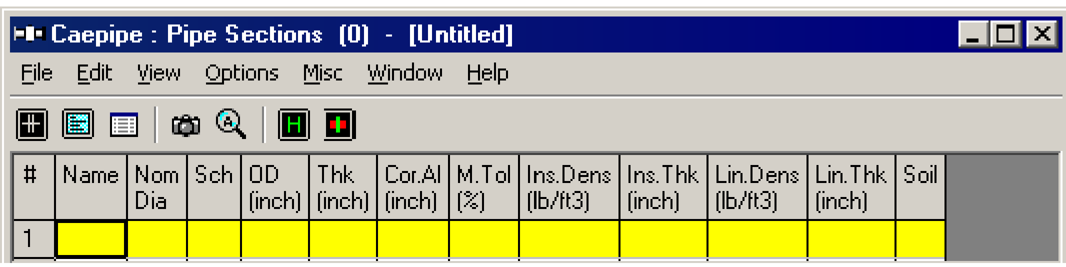
In dynamic analysis, the status of the rod hanger from the first operating case (W+P1+T1) is used, i.e., if the rod hanger is in tension in the first operating case, a rigid vertical two-way restraint is used in dynamic analysis. If the rod hanger is in compression in the first operating case (possible liftoff), no vertical restraint is used at that location in dynamic analysis.

A Section denotes the cross-sectional properties of a pipe used to build a piping model. You may define as many sections as needed. To define each section, you will need properties such as outside diameter (or Nominal Dia.), thickness of pipe, corrosion allowance, insulation, inside lining, and a name that is used under the Section column on the Layout window while building your model.

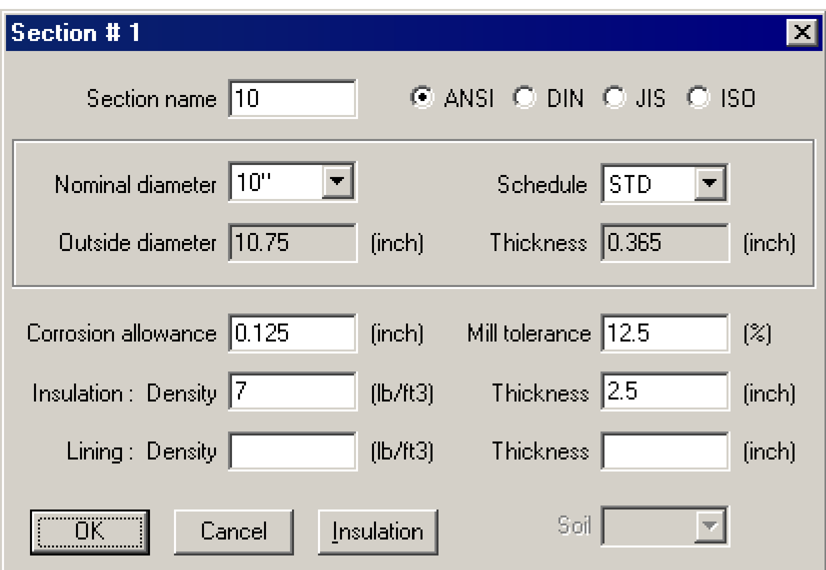
Click on “Sect” on the Header row or select Sections from the Misc menu.



CAEPIPE presents a List window that lists all defined sections in the model (none defined yet in the image below). Double click on an empty row to define a new one.



The section dialog is shown.



Section name

Type an alpha-numeric name (up to 5 characters long) in this field. Example: If you have three 8" sections with different schedules, you could name them thus: 8-STD, 8-80 and 8-80S.

Nominal Dia, Schedule

Four databases of pipe sizes are built-into CAEPIPE — ANSI (American National Standards Institute, default), DIN (Deutsche Industrie Norm), JIS (Japanese Industrial Standard) and ISO (International Organization for Standardization).

When you click on the drop-down combo box for pipe sizes, CAEPIPE shows the list of pipe sizes that pertains to the selected database (ANSI, DIN, JIS or ISO). ANSI pipe sizes range from 1/8" to 48", DIN from 15 to 1600, JIS from 8A to 1500A, and ISO from 15 to 1000. Select the required nominal pipe size and schedule (wall thickness). To change to a different database (JIS, DIN or ISO), click on the appropriate radio button. On selection, CAEPIPE populates the correct OD and Thickness.

For pipe sizes you do not see on the list, each database allows you a nonstandard definition ("Non std" in the pipe sizes list) too. In other words, you are not restricted only to the choices available in the databases. You may define any size and thickness as needed. You will need to enter the Outside diameter and Thickness of such a nonstandard pipe, in addition to the other parameters.

Corrosion Allowance

The corrosion allowance reduces the wall thickness of the pipe and is used to calculate the allowable pressure for the pipe section. Additionally, for some piping codes (B31.3, B31.4, B31.5, B31.8, B31.12, B31.1 (1967), CODETI, Canadian Z183 and Z184), corrosion allowance is used for reducing the section modulus and cross-sectional area only for calculating sustained and occasional stresses.

Mill Tolerance

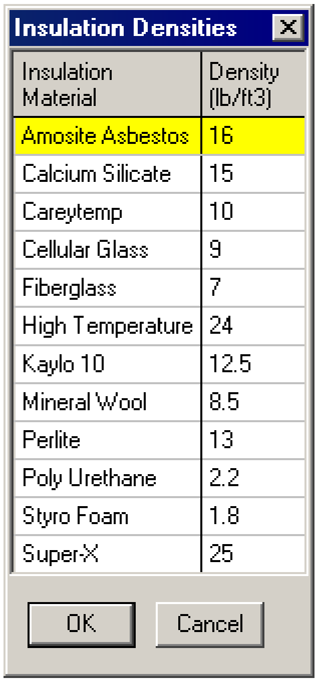
The mill tolerance (in percent) is also used to reduce the wall thickness of the pipe while calculating allowable pressure. For example, if the mill tolerance is input as 12.5 (%), the pipe thickness (while calculating allowable pressure) is = 0.875 x nominal thickness.

Reduced thickness = (1 − Mill tolerance/100) × nominal thickness − Corrosion allowance

If defined while modeling, corrosion allowance, mill tolerance, insulation and lining densities are automatically carried forward while defining a new section.

Insulation

Type the pipe insulation density and thickness here. Click on the Insulation button for the insulation library, or enter your own.



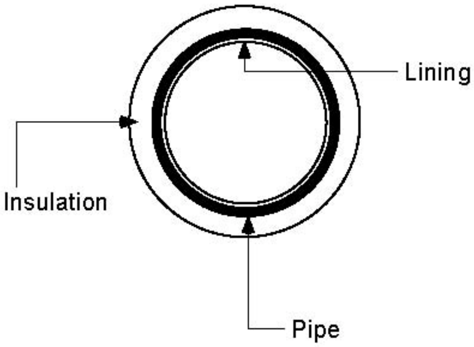
Insulation Densities

Insulation Material	Density (lb/ft3)
Amosite Asbestos	16
Calcium Silicate	15
Careytemp	10
Cellular Glass	9
Fiberglass	7
High Temperature	24
Kaylo 10	12.5
Mineral Wool	8.5
Perlite	13
Poly Urethane	2.2
Styro Foam	1.8
Super-X	25

Highlight the desired insulation material and press Enter. The insulation density is entered on the section property dialog. CAEPIPE uses insulation thickness and density to calculate the insulation weight which is added to the weight of the pipe. Insulation thickness is also used to calculate the projected area exposed to wind load(s).

Lining

Lining is used to prevent internal corrosion that might occur during transportation of a gas or a liquid. CAEPIPE has the ability to model these protective coatings inside the pipe.



Lining is different from insulation. Insulation is around and outside of the pipe. Lining is on the inside of the pipe. Both have respective thicknesses and densities, which are used to calculate the respective weight which is then added to the weight of the pipe. See previous figure.

While calculating the weight of the liquid/gas inside the pipe, CAEPIPE accounts for lining thickness by reducing the pipe's internal diameter by twice the lining thickness.

Note:

CAEPIPE requires "Weight Density" to be input in lbf/in3 or kgf/m3 and NOT its "Mass Density" for insulation and lining.

Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass for each item to be equal to (weight / g-value).

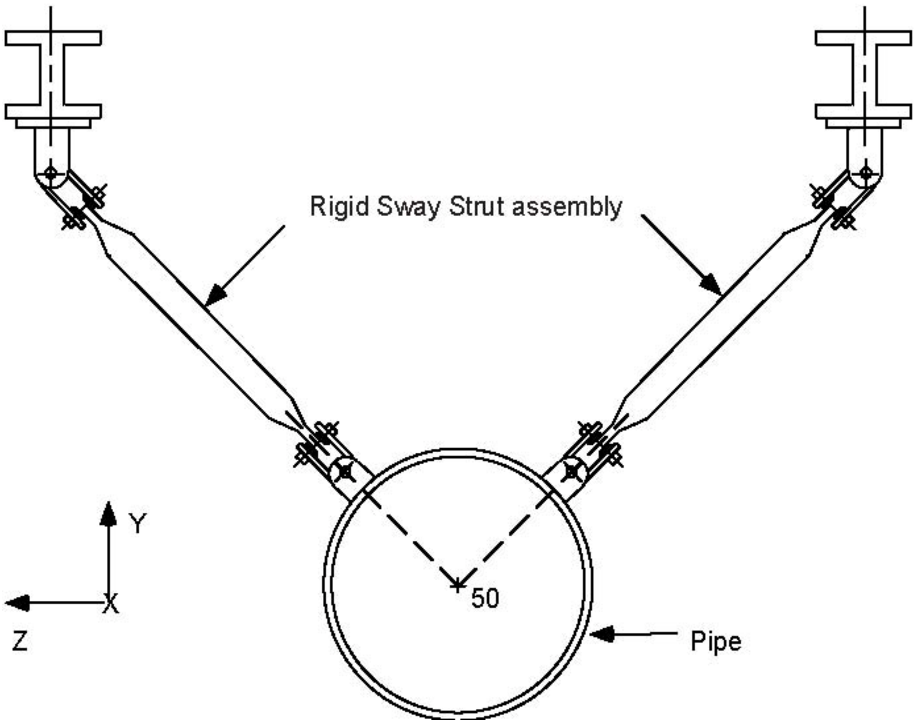
Using values input for Insulation Thickness, Insulation Density, Lining Thickness and Lining Density for each pipe section in astress model, CAEPIPE will compute their weight and include the same in theanalysis. In addition, CAEPIPE considers Insulationand/or Lining as integral parts of the piping, and that there is no relative motion between insulation and piping and lining and piping.Accordingly, CAEPIPE does not account for any friction between pipe and insulation and pipe and lining.

Skewed Restraint

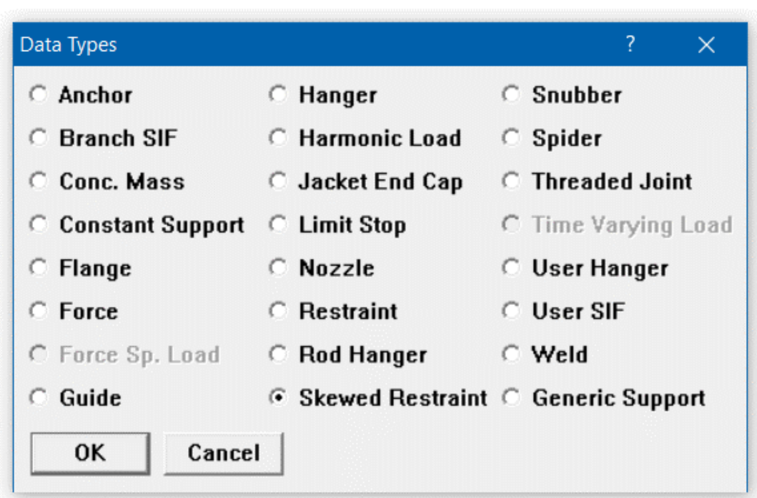
A Skewed Restraint is a two-way support that resists translation along or rotation about any specified direction at a node. You have to use either a manufacturer-supplied stiffness or calculate it for the support you want to model.

Use this restraint to model sway braces, sway struts and similar supports. You can also use this to model vertical/horizontal supports, though it is used more commonly to resist lateral forces.

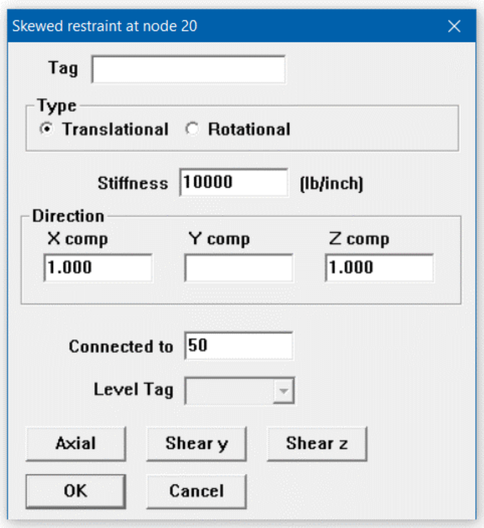
The figure below shows an application.



A skewed restraint is input by typing "sk" in the Data column or selecting "Skewed restraint" from the Data Types dialog.



The Skewed Restraint dialog is shown.



Type

Translational: Use this type to restrain translation along the specified direction.

Rotational: Use this type to restrain rotation about the specified direction.

Stiffness

Type in the translational or rotational stiffness of the support. As an illustration, assume that you had a rod (in tension only) which you were modeling as a skewed restraint. You can calculate the stiffness (required to be input) in the following manner: Assume a 2.5 in. dia. rod 2 feet long, modulus of elasticity of rod material = 30×10^6 psi.

The translational (axial) stiffness is $AE/L = \frac{\pi}{4}(2.5)^2 \times 30 \times 10^6/24$=6,135,925 lb./in.

The rotational stiffness is $GJ/L = \frac{E}{2(1+\nu)} \times \frac{J}{L} = 1{,}843{,}727$ in. -lb./rad., where G is the shear

modulus, ν is the Poisson's ration and J is polar moment of inertia.

Direction

If you have no "connected to node," the direction in which the skewed restraint is oriented must be specified in terms of its global X, Y and Z components. See topic on specifying a Direction.

If the skewed restraint node is connected to an externally fixed point (ground), then for the Direction vector components (X comp, Y comp, Z comp), you can specify the offsets (DX, DY, DZ) from the skewed restraint node to the fixed point.

Or use one of the preset buttons to orient the skewed restraint axis:

1. **Axial**: To set the axis along the local-x direction (pipe axis)
2. **Shear y**: To set the axis in the local-y direction
3. **Shear z**: To set the axis in the local-z direction

If you have connected the skewed restraint node to another node, then the direction must not be input. It is calculated from the locations of the skewed restraint node and the connected node, and it is oriented from the skewed restraint node towards the connected node. In order for CAEPIPE to calculate the direction, the skewed restraint node and the connected node must not be coincident.

Connected to node

If the skewed restraint node is connected to an externally fixed point (ground), leave the "Connected to node" blank. You may connect a skewed restraint node to another node that is not coincident with the skewed restraint node. Note that during skewed restraint force calculations, the relative displacement of the skewed restraint node is calculated with respect to the connected node.

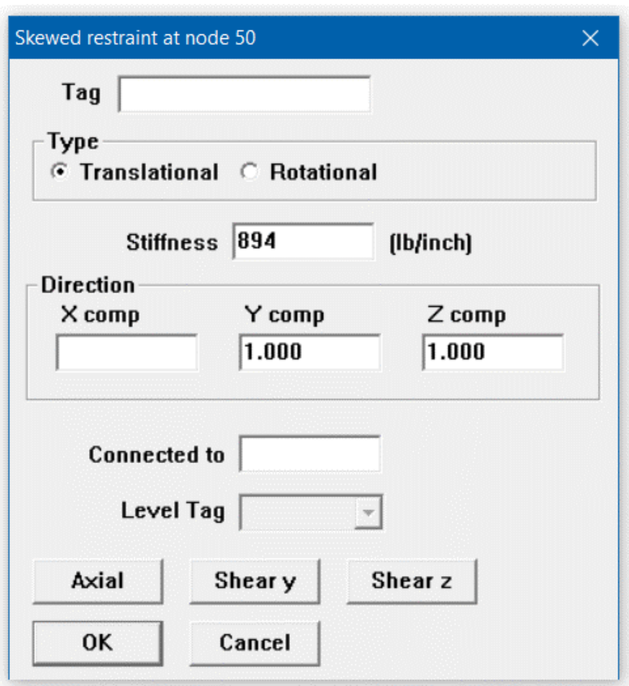
Example: Modeling a Sway Brace

Assume that we need to model two sway braces in the same arrangement as shown in the figure at the beginning of this section. The translational stiffness of the sway braces is given as 894 lb./in. As can be surmised from the figure, the orientation of the sway braces (sway struts in the figure) is at 45° from the Y- and Z-axes. We shall model the support on the right hand side first followed by the support on the left hand side.

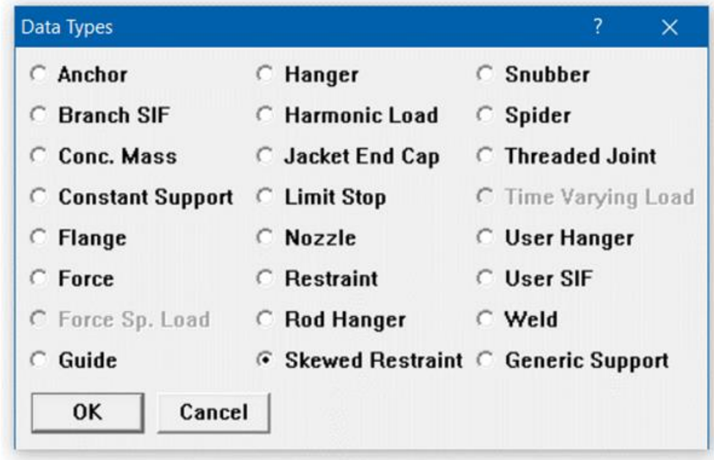
The following steps describe the modeling procedure:

- Create node (on pipeline) where support is required. In this case, the node is 50. Position highlight on this row.
- First support (right): Type "sk" in the Data column to open the skewed restraint dialog box.

Ensure that Type is set to Translational; if not, click on the Translational radio button. Type 894 for Stiffness, type 1 for Y comp and –1 for Z comp, press Enter.



- Second support (left): type 50 for Node on an empty row, press Tab to move to next field, press "l(L)" for Location. This will open the Data types dialog.

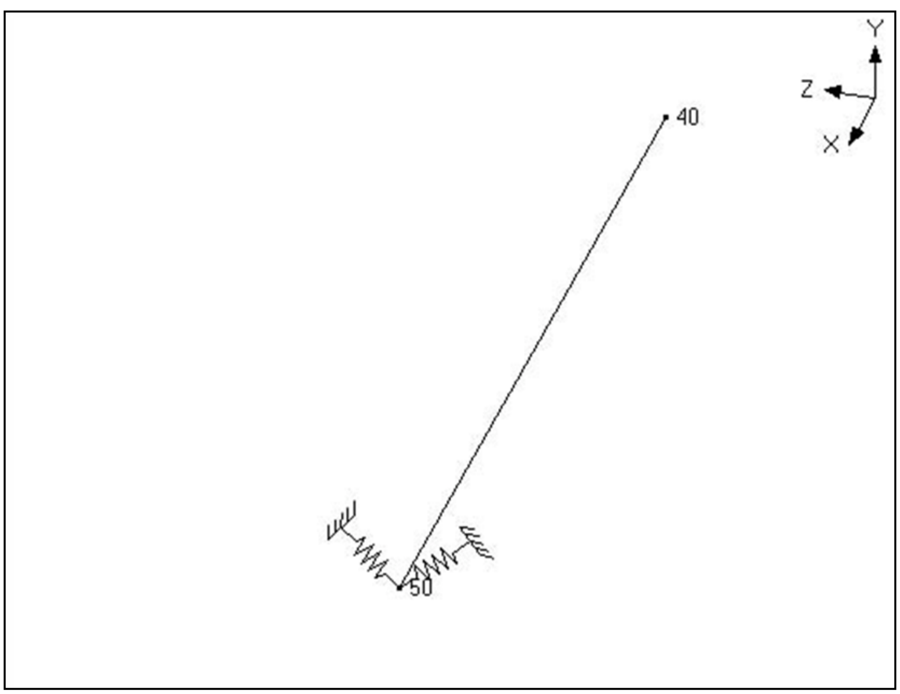


Select Skewed restraint by clicking on it to open the skewed restraint dialog. Enter the skewed restraint dialog similar to the first skewed restraint except in this case type 1 for Z comp, press Enter.

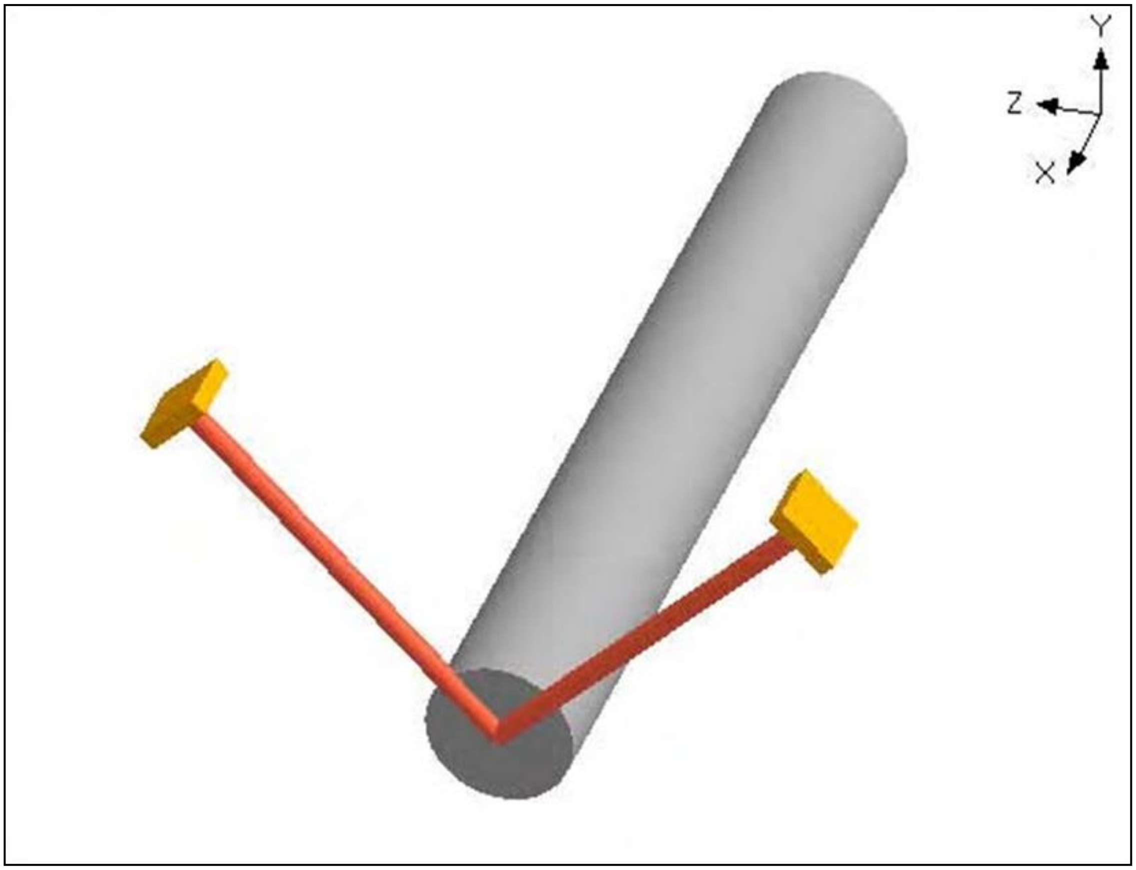
The Layout window is shown below:

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sway brace								
2	40	From							
3	50		5'0"			1	8	1	Skewed restr
4	50	Location							Skewed restr
5									

The graphics is shown below:



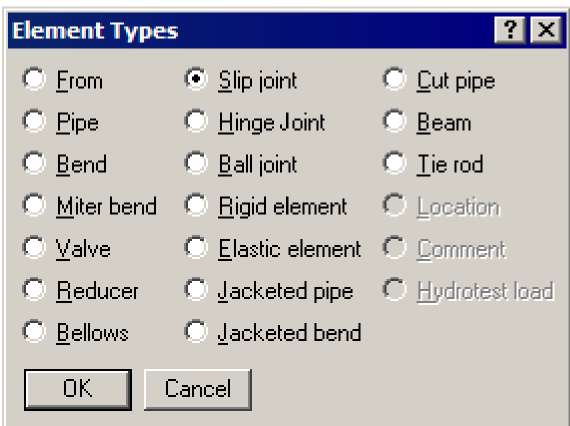
The rendered graphics is shown below:



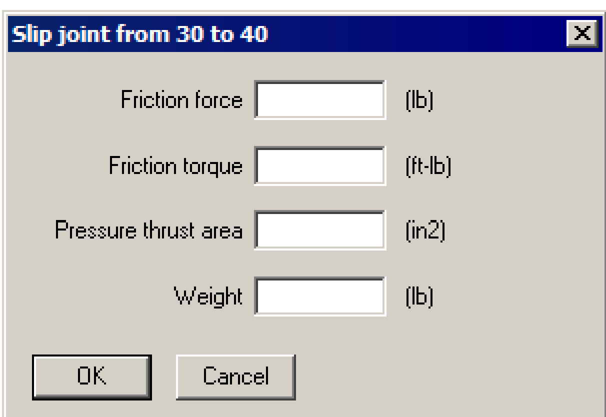
A slip joint allows for axial (through telescopic action) and torsional movement between adjacent pipes (due to thermal expansion or contraction). The joint itself can be fixed using an anchor if so designed. Slip joints are susceptible to lateral buckling due to internal pressure, and may become less effective when subjected to small bending loads. Proper guiding to prevent buckling and keeping the two telescopic parts concentric are therefore necessary.

Since the primary purpose of a slip joint is to absorb axial growth, the joint is ideal for placing it towards the end of long pipe runs, while its growth is directed axially by the use of one or more guides.

A Slip Joint is input by typing "s" under the Type column or by selecting "Slip joint" from the Element types dialog.



The Slip joint dialog is shown.



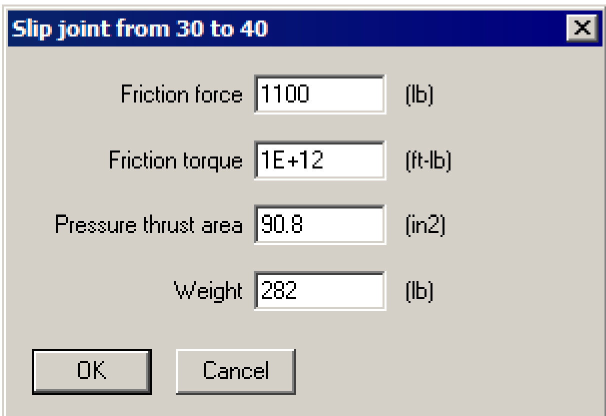
A slip joint manufacturer should be able to provide you the required data for a slip joint.

A slip joint will have axial deflection or rotation only when the external forces exceed the friction force or friction torque respectively. If the pressure thrust area is input, CAEPIPE imposes a thrust load of: Pressure x Thrust area on both nodes of the slip joint. The weight is the empty weight of the joint. The contents, insulation and additional weight are added to the empty weight. A slip joint is considered to be rigid in lateral directions in CAEPIPE.

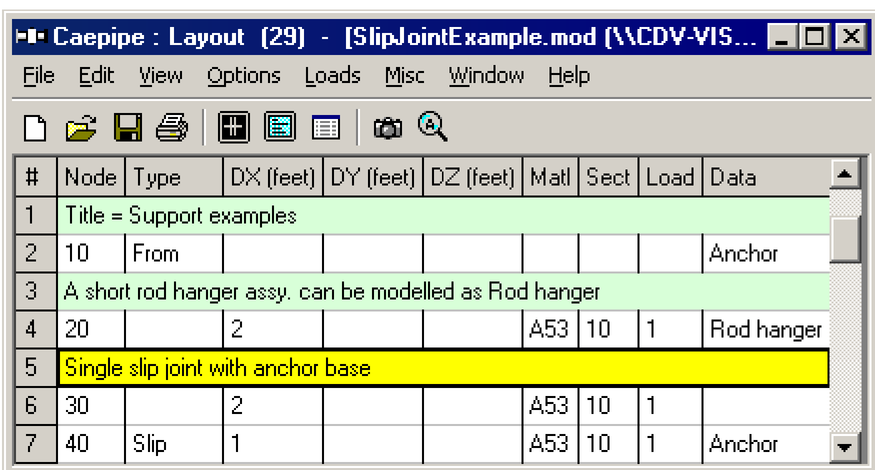
Weight of the slip joint is input in lbf or kgf and NOT its mass. Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

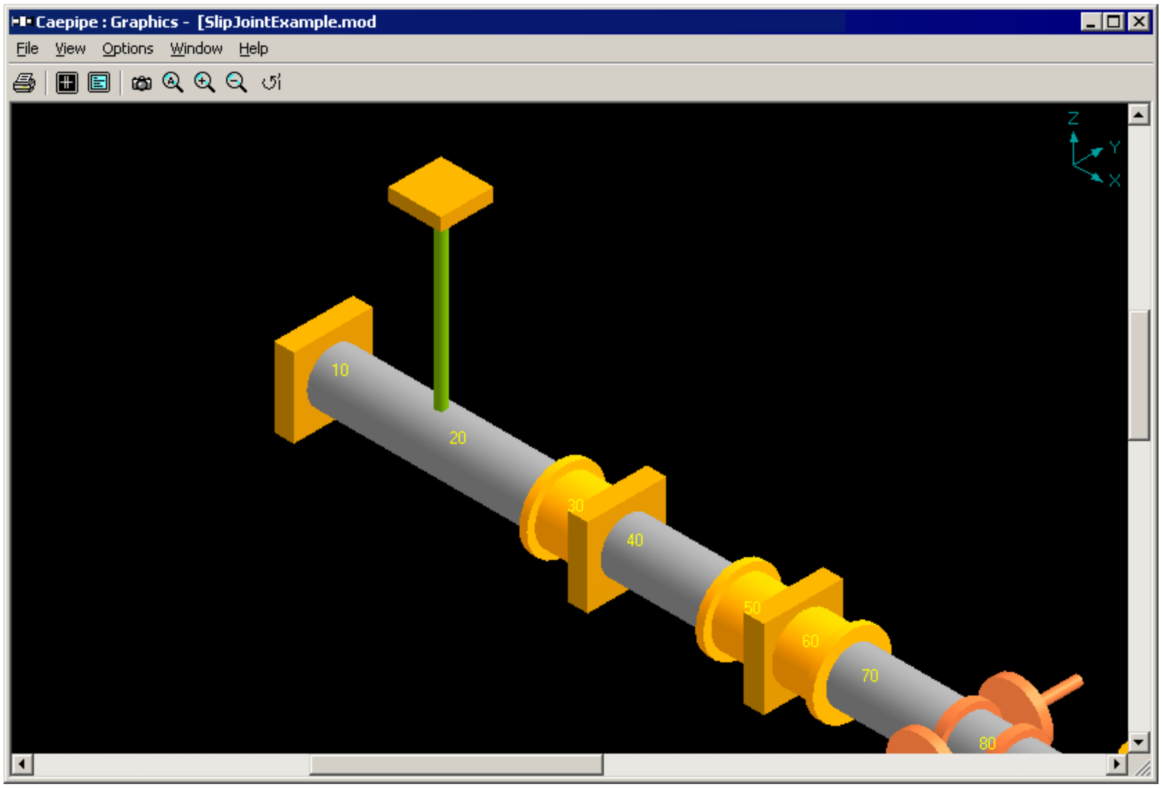
Example:

Assume that we want to model a (telescoping action) slip joint that allows only axial movement with no torsion. The outer sleeve of the joint is anchored to hold it in place while the other end is free to translate axially. The axial (friction) force has to exceed 1,100 lb. (you will need to get this datum from a manufacturer's catalog) to make the slip joint move and the slip joint cannot rotate about the axial direction. So, the data would look similar to that shown next between nodes 30 and 40.



And, the modeling on the layout screen would look thus:



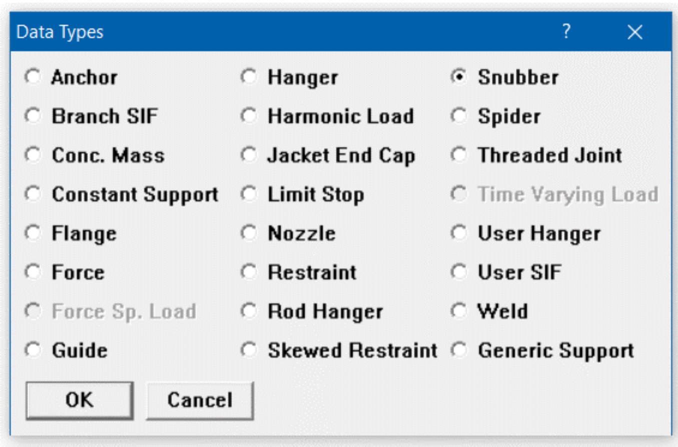


See the topic on Nonlinearities for related information.

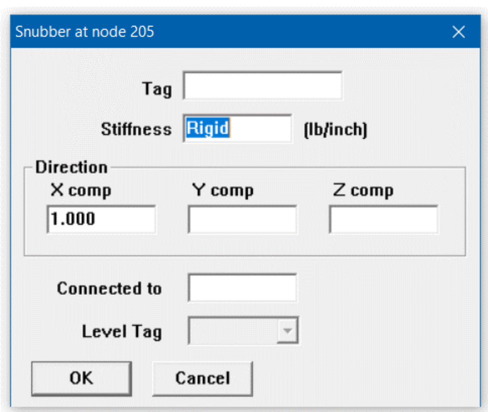
Snubber

A snubber provides only translational restraint in a specified direction for seismic and dynamic cases only. In other words, a snubber engages only during movements caused by a dynamic load. It does not restrain against static loads such as weight and thermal.

A snubber is input by typing "sn" in the Data column or selecting "Snubber" from the Data Types dialog.



The Snubber dialog is shown.



The stiffness defaults to Rigid, however a stiffness may be input for flexible snubbers. See section on specifying a Direction for information on X comp, Y comp, Z comp. A snubber can be made active in any direction by using this combination of Direction Cosines (X comp, Y comp, and Z comp).

Since the snubber is considered to be attached to an externally fixed point, for the Direction vector components (X comp, Y comp, Z comp), it is easier to specify the offsets (DX, DY, DZ) from the snubber node to the fixed point.

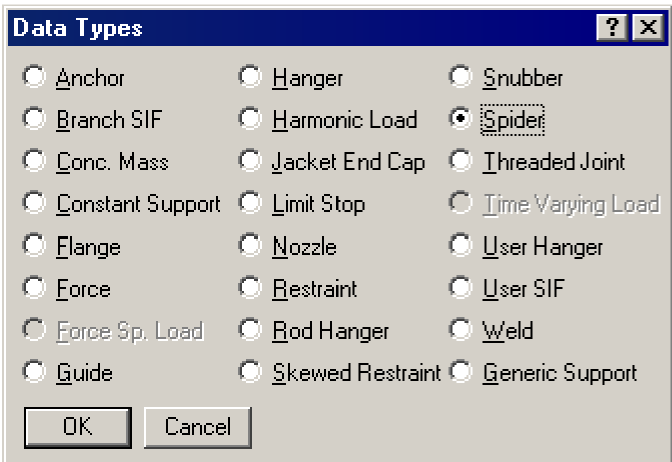
Connected to node

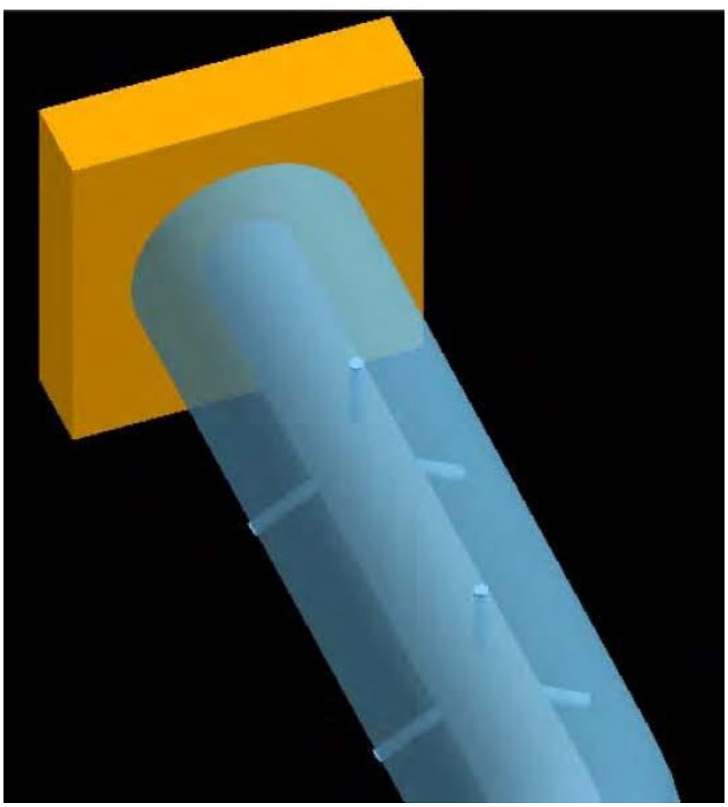
If the snubber is connected to an externally fixed point (ground), leave the "Connected to node" blank. You may connect a snubber node to another node that is not coincident with the snubber node. Note that during snubber force calculations, the relative displacement of the snubber node is calculated with respect to the connected node.

Spider

Use a spider (also called a spacer) to connect the coincident nodes of a jacketed pipe (i.e., the node on the core pipe and the corresponding node on the jacket pipe). The spider acts as an internal guide. At the spider location, the local x-axis is calculated along the pipe direction. The spider connects the local y and z translations for the core and jacket nodes. It prevents any radial movement but allows sliding, rotating and bending movement between core and jacket pipes. No gap is allowed between the core pipe and the spider. See section on Jacketed pipe for related information.

A spider is input at a jacketed pipe node by typing "sp" in the Data column or selecting "Spider" from the Data Types dialog.





Starting Version 10.30, CAEPIPE has the built-in feature to calculate g-load values using the procedure given in ASCE/SEI 7-16 as described below.

.Static Seismic Load – ASCE/SEI 7-16.

Guidelines from ASCE/SEI 7-16 "Minimum Design Loads for Buildings and Other Structures" are explained below.

Structure Occupancy Category (Risk Category):

Table 1.5-1 of ASCE/SEI 7-16 provides the Risk Category of Buildings and Other Structures for Flood, Wind, Snow, Earthquake, and Ice Loads Structure. Based upon your project specification, select the Structure Occupancy Category from the options available.

Table 1.5-1 Risk Category of Buildings and Other Structures for Flood, Wind, Snow, Earthquake, and Ice Loads

Use or Occupancy of Buildings and Structures	Risk Category
Buildings and other structures that represent a low risk to human life in the event of failure	I
All buildings and other structures except those listed in Risk Categories I, III, and IV	II
Buildings and other structures, the failure of which could pose a substantial risk to human life. Buildings and other structures, not included in Risk Category IV, with potential to cause a substantial economic impact and/or mass disruption of day-to-day civilian life in the event of failure. Buildings and other structures not included in Risk Category IV (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, hazardous waste, or explosives) containing toxic or explosive substances where their quantity exceeds a threshold quantity established by the authority having jurisdiction and is sufficient to pose a threat to the public if released.	III
Buildings and other structures designated as essential facilities. Buildings and other structures, the failure of which could pose a substantial hazard to the community. Buildings and other structures (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, or hazardous waste) containing sufficient quantities of highly toxic substances where the quantity exceeds a threshold quantity established by the authority having jurisdiction to be dangerous to the public if released and is sufficient to pose a threat to the public if released.[a] Buildings and other structures required to maintain the functionality of other Risk Category IV structures.	IV

[a]Buildings and other structures containing toxic, highly toxic, or explosive substances shall be eligible for classification to a lower Risk Category if it can be demonstrated to the satisfaction of the authority having jurisdiction by a hazard assessment as described in Section 1.5.2 that a release of the substances is commensurate with the risk associated with that Risk Category.

Site Class:

Based on the site soil properties, the site shall be classified as Site Class A, B, C, D, E & F.

Refer Chapter 20 of ASCE/SEI 7-16 for details on Site Class. Depending on your project specification, select the Site Class from the options provided.

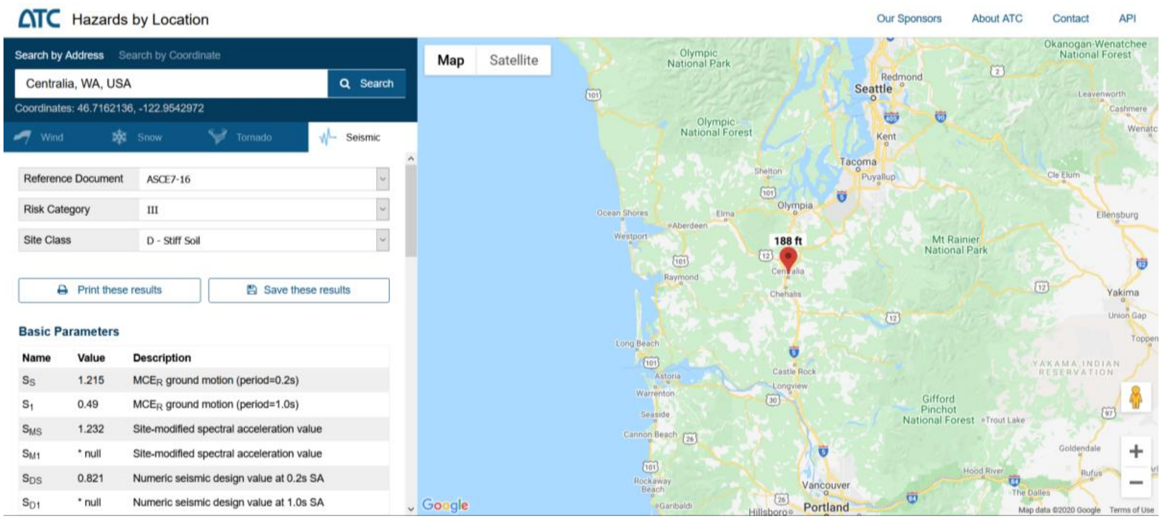
Note: Para. 11.4.3 of ASCE/SEI 7-16 states that "Where the soil properties are not known in sufficient detail to determine the site class, Site Class D shall be used unless the authority having jurisdiction or geotechnical data determines Site Class E or F soils are present at the site".

Mapped MCE Spectral Acceleration at Short Period S(S):

The USGS maintains a website http://earthquake.usgs.gov/designmaps with which the site latitude and longitude, as well as the value of S(S) can be retrieved.

Users can also retrieve the above said values from https://hazards.atcouncil.org.

For example, for Centralia, WA, the value of S(S) is retrieved as 1.125 with Structure Occupancy Category III and Site Class D from the link https://hazards.atcouncil.org as shown below.



Component Height in Structure (z)

Component Height in Structure (z) is the point of attachment of components with respect to the base. As per para. 13.3.1 of ASCE/SEI 7-16, for components at or below the base, z shall be taken as 0. In addition, the value of z/h need not exceed 1.0.

For example, for a Piping attached to a Boiler Nozzle located at an Elevation 150'0" with a Ground Level of 25'0", z can be computed as 125' (= 150' – 25').

Structure Height (h)

Structure height (h) is the average roof height of the structure with respect to the base. From the above example, the average roof height of structure (h) with respect to the base can be calculated as 125' (= 150' – 25').

Component Amplification Factor, a(p)

Table 13.6-1 Seismic Coefficients for Mechanical and Electrical Components of ASCE/SEI 7-16 provides values of Component Amplification Factor "a(p)" for various components. As per para. 13.3.1 of ASCE/SEI 7-16, the component amplification factor is from 1.00 to 2.50.

For example, Piping in accordance with ASME B31, including in-line components with joints made by welding or brazing, the value of Component Amplification Factor "a(p)" is listed as 2.50.

Component Response Modification Factor, R(p)

Table 13.6-1 Seismic Coefficients for Mechanical and Electrical Components of ASCE/SEI 7-16 provides values of Component Response Modification Factor “R(p)” for various components. As per para. 13.3.1 of ASCE/SEI 7-16, component response modification factor should be from 1.00 to 12.00

For example, Piping in accordance with ASME B31, including in-line components with joints made by welding or brazing, the value of Component Response Modification Factor “R(p)” is listed as 12.00.

Importance Factor, I(p)

Para. 13.1.1 of ASCE/SEI 7-16 provides a guideline to arrive at component Importance Factor “I(p)”. The value of “I(p)” can be between 1.0 and 1.5.

Allowable Stress Design Factor, ADS(a)

Enter Allowable Stress Design Factor, ADS(a) as specified in the project specification. If this data is not available, then it can be specified as 1.0.

Para. 13.1.8 of ASCE/SEI 7-16 states that the earthquake loads determined in accordance with Section 13.3.1 of ASCE/SEI 7-16 shall be multiplied by an Allowable Stress Design Factor ADS(a) of 0.7.

Example:

For a power plant required to operate in an emergency, located in Centralia, WA, USA, with Occupancy Category as III, Site Class as D (with Stiff soil) and Importance Factor 1.5, use ASCE/SEI 7-16 and compute the design earthquake load coefficient for piping required to operate the plant.

1. From the map above with Occupancy Category as III and Site Class as D, the Mapped MCE Spectral Acceleration at Short Period ‘S(S)” = 1.215.
2. Component Height in Structure (z) = 150’0” – 25’0” = 125’0”
3. Structure Height (h) = 150’0” – 25’0” = 125’0”
4. Component Amplification Factor, a(p) = 2.50 *(as per Table 13.6-1 of ASCE/SEI 7-16, for Piping in accordance with ASME B31, including in-line components with joints made by welding or brazing)*
5. Component Response Modification Factor, R(p) = 12.00 *(as per Table 13.6-1 of ASCE/SEI 7-16, for Piping in accordance with ASME B31, including in-line components with joints made by welding or brazing)*
6. Importance Factor, I(p) = 1.5 (given)

With the data provided above,

   a. Site Coefficient at Short Period, F(a) = 1.014 (as per Table 11.4-1 for S(S) = 1.215 is and Site Class D)

Table 11.4-1 Site Coefficient, F_a
Mapped Risk-Targeted Maximum Considered Earthquake (MCE_R) Spectral Response Acceleration Parameter at Short

SITE CLASS	$S_S \le 0.25$	$S_S = 0.5$	$S_S = 0.75$	$S_S = 1.0$	$S_S = 1.25$	$S_S \ge 1.5$
A	0.8	0.8	0.8	0.8	0.8	0.8
B	0.9	0.9	0.9	0.9	0.9	0.9
C	1.3	1.3	1.2	1.2	1.2	1.2
D	1.6	1.4	1.2	1.1	1.0	1.0
E	2.4	1.7	1.3	See Section 11.4.8		
F	See Section 11.4.8					

b. As per equation 11.4-1 of ASCE/SEI 7-16, Maximum MCE Spectral Acceleration at Short Period S(MS) = F(a)*S(S) = 1.014 * 1.215 = 1.232 g

c. As per equation 11.4-3 of ASCE/SEI 7-16, Design Spectral Acceleration at Short Period S(DS) = (2/3)*S(MS) = (2/3)*1.232 = 0.821 g

d. As per equation 13.3-1 of ASCE/SEI 7-16,

Horizontal Seismic g-load value = H(g) = 0.4*S(DS)*[a(p)/R(p)]*I(p)*[1 + 2(z/h)]

= 0.4*0.821*(2.5/12.0)*1.50*[1+(2*(125/125))] = 0.308 g

e. As per equation 13.3-2 of ASCE/SEI 7-16,

Maximum Horizontal Seismic g-load value H(g.max) = 1.6*S(DS)*I(p)

= 1.6*0.821*1.5 = 1.97 g

f. As per equation 13.3-3 of ASCE/SEI 7-16,

Minimum Horizontal Seismic g-load value H(g.min) = 0.30*S(DS)*I(p)

= 0.30*0.821*1.5 = **0.369 g**

g. As per Para 13.3.1 of ASCE/SEI 7-16,

Vertical Seismic g-load V(g) = 0.20*S(DS)

= 0.20*0.821 = **0.164 g**

As per Para. 13.3.1 of ASCE/SEI 7-16, **H(g) >= H(g.min) and H(g) <= H(g.max)**

Hence,

Horizontal Seismic g-load value H(g) = 0.369 g.

Vertical Seismic g-load value V(g) = 0.164 g

From the above, as per Para. 13.1.7 of ASCE/SEI 7 – 16,

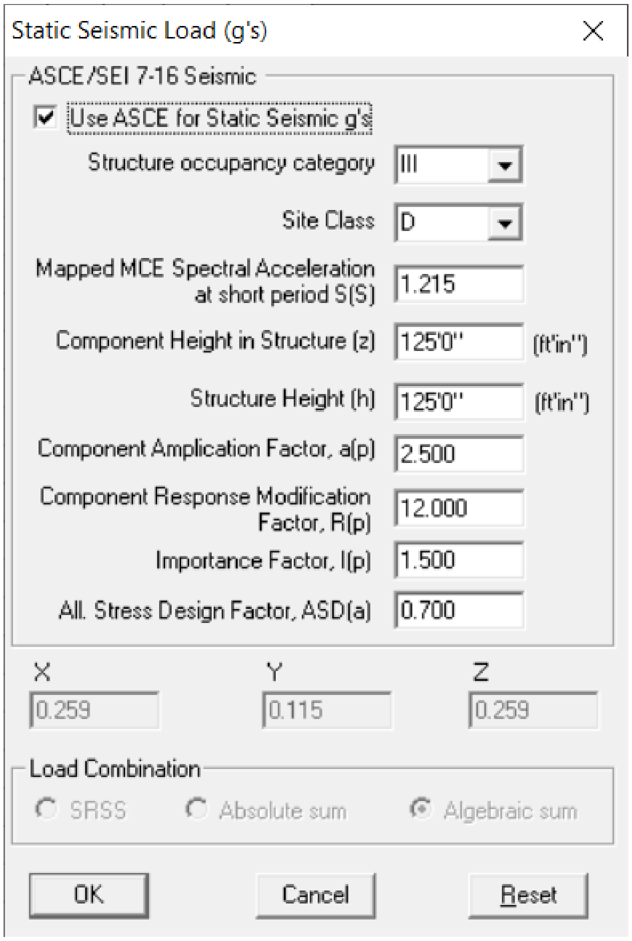
Allowable Stress Design Horizontal Seismic g-load value = H(g)*ADS(a)

= 0.369 * 0.70 = 0.258 g

Allowable Stress Design Vertical Seismic g-load value = V(g)*ADS(a)

= 0.164 * 0.70 = 0.115 g

CAEPIPE Output



.Static Seismic Load – ANSI A58.1-1988.

The procedure from ANSI A58.1-1988, "Minimum Design Loads for Buildings and Other Structures," is used in the below example to calculate the g-load values for input into CAEPIPE.

Piping is assumed to be equivalent to equipment, thus giving a force coefficient, Cp as 0.3.

1. First, based on the map below (Contiguous 48 States+Alaska/Hawaii/Puerto Rico), identify the seismic zone and its corresponding coefficient (Z).

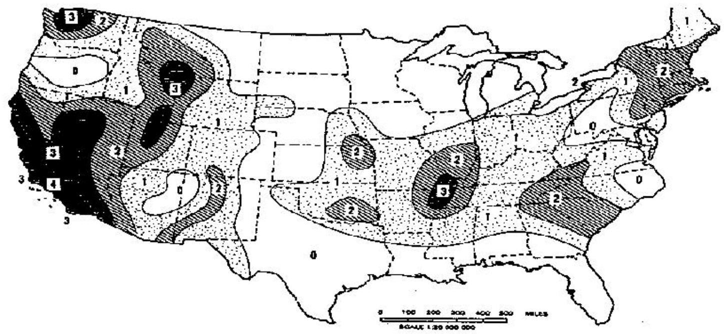


Fig. 14. Map for Seismic Zones—Contiguous 48 States (See Commentary Section 9.4)

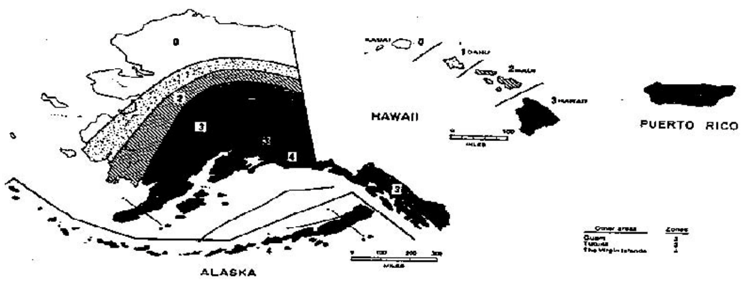


Fig. 15. Map for Seismic Zones—Hawaii, Alaska, and Puerto Rico—(See Commentary Section 9.4)

Seismic Zone	Coefficient Z
4	1.0000
3	0.7500
2	0.3750
1	0.1875
0	0.1250

Determine the Importance Factor, I as follows:

I = 1.5, For piping required in an emergency or piping with contents representing a significant hazard to human life

I = 1.0, For other piping

Calculate the acceleration (g's) as $0.3 \times Z \times I$

Example:

For a power plant required to operate in an emergency, located in Anchorage, Alaska, calculate the design earthquake load coefficient for piping required to operate the plant.

1. From the map above, the seismic zone is 4; from the table above, the corresponding Coefficient Z is 1.0.

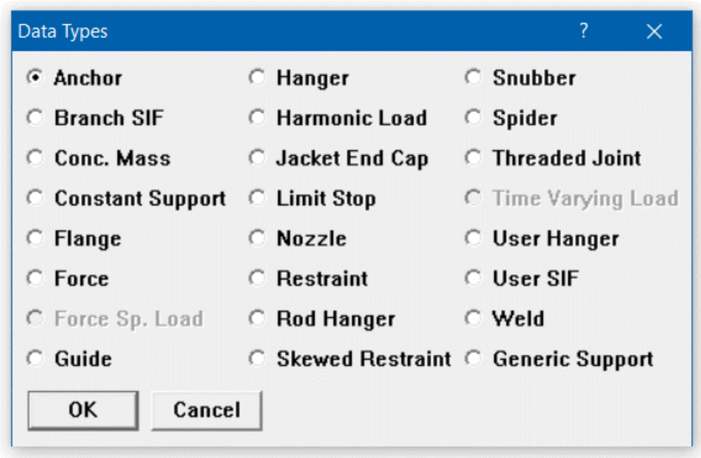
2. For piping required to operate in an emergency, the Importance Factor I = 1.5.

3. Calculate horizontal acceleration as $0.3 \times 1.0 \times 1.5 = 0.45g$, Vertical g-load (g) = 0.75 * horizontal acceleration = 0.34g.

The above accelerations need to be applied in the desired horizontal and vertical directions as long as the piping system is predominantly routed at grade level.

Supports

CAEPIPE provides many support types, as listed under the Data types menu (See section on Data Types in the CAEPIPE User's Manual).



You can use one or more such support types at one nodal location to model almost all types of real-world pipe support hardware, thereby incorporating accurate mathematical representation of those supports in the CAEPIPE stress model. For example, you could use two lateral limit stops with unequal gaps to simulate a pipe shoe (see example under Limit Stop).

To input more than one support at a node, use the "Location" data type.

Anchor

An anchor can be modeled as a flexible or rigid support which by default restrains the three translations and three rotations either in the global or local directions at the applied node (six degrees of freedom). Use this to model all anchor blocks, and nodes where piping connects to equipment (pumps, compressors, turbines, etc.). See section on Anchor for further details.

Restraint

A restraint is a two-way rigid support which restrains the translations (negative and positive directions) along the specified global directions. You can apply a restraint in all the three directions at the same time. See section on Restraint for further details.

Skewed Restraint

This is a flexible two-way support that can be oriented in any direction. Use this support to resist either translational movement along or rotational movement about the specified direction. Use this to model rigid or flexible sway struts and sway braces. See section on Skewed Restraint for further details.

Hanger, User hanger, Rod hanger, Constant support

These should be used as vertical supports only. Use a Hanger when you want to design (i.e., select from a built-in catalog) a variable spring hanger(s) for your piping system (there are 30+ hanger catalogs built-in to CAEPIPE for your convenience). Use a User Hanger when you want to analyze piping system with existing variable spring hangers. Use a Rod Hanger for a rod hanger assembly. Use a Constant Support to design a constant support or a constant force hanger. See corresponding sections for further details.

"Bottomed-out" Springs

To analyze this situation, use a variable spring hanger and a limit stop at this node. Type in the maximum allowable hanger travel for one of the limits of the limit stop. Once the hanger traverses the maximum distance allowed, the limit stop becomes active.

Guide

A Guide is a rigid or flexible restraint which resists lateral pipe movements (in directions perpendicular to the axis of the pipe). You can specify an annular gap, if required, inside the guide. A friction coefficient is optional. Use a Guide to model U-straps, U-Bolts, pipe guide assemblies, pipe slides and similar supports. See section on Guide for further details.

Limit stop

A limit stop, a nonlinear restraint, can be oriented in any direction with a gap specified on both sides of the pipe. A limit stop allows free movement for the distance of the gap and then acts as a rigid or flexible restraint.

A Line Stop is a support that restricts axial movement of pipe. This support can be modeled using a limit stop with its direction oriented along the pipe's axis. Use this support to model pipe slide assemblies, pipe skirts and similar arrangements. See topic on Limit Stop for an example.

A limit stop can be used to model 1-way supports for pipe racks where vertical downward movement is restrained while upward movement is not. See example for pipe rack modeling in the Beam topic.

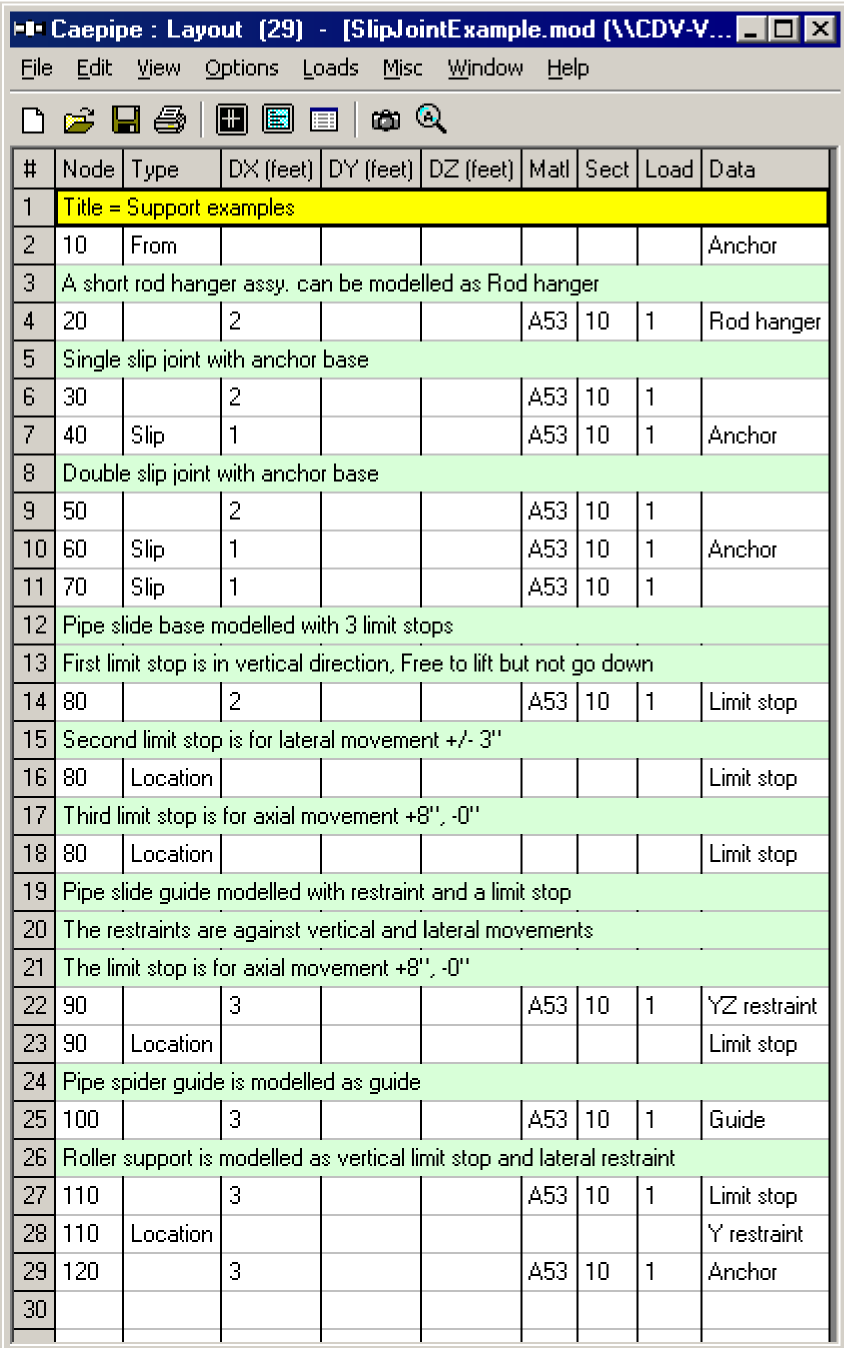
Support Tag

All the Supports described above and Nozzle can have Tags (Support Tags). Each Tag can be up to 14 characters long. Tags are useful in identifying supports while modeling, reviewing of reports and in field erection. Tag Name entered in this field is shown in all reports.

Level Tag

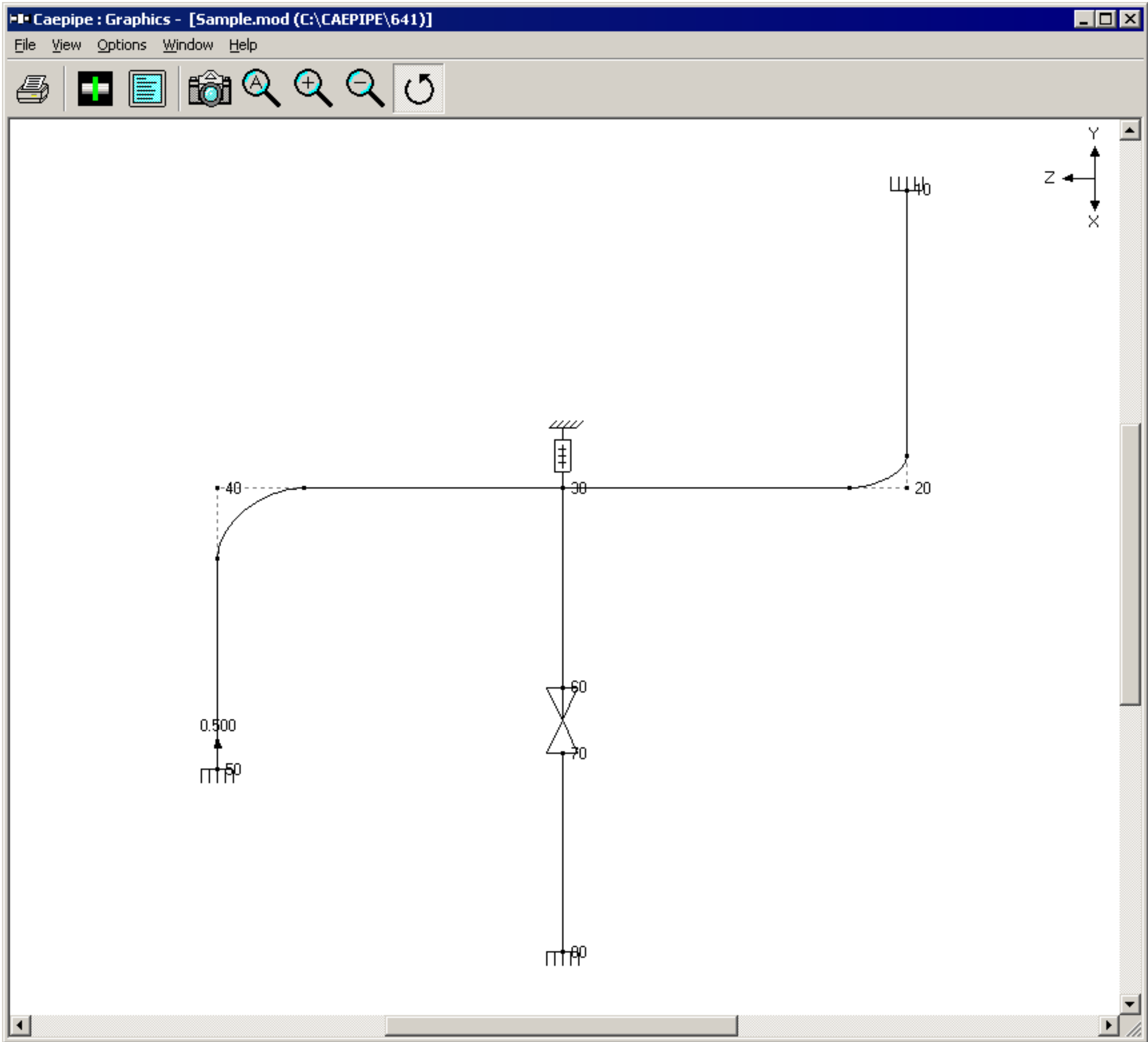
Level Tag shall be assigned for the multi-level response spectrum analysis. It will be disabled if a single level spectrum or no spectrum levels are defined. The CAEPIPE will automatically assign the level tag to all support for a single level. See section on Spectrum Load in Users Manual for more information.

Refer to the respective CAEPIPE support term in this manual for further details.



#	Node	Type	DX (feet)	DY (feet)	DZ (feet)	Matl	Sect	Load	Data
1	Title = Support examples								
2	10	From							Anchor
3	A short rod hanger assy. can be modelled as Rod hanger								
4	20		2			A53	10	1	Rod hanger
5	Single slip joint with anchor base								
6	30		2			A53	10	1	
7	40	Slip	1			A53	10	1	Anchor
8	Double slip joint with anchor base								
9	50		2			A53	10	1	
10	60	Slip	1			A53	10	1	Anchor
11	70	Slip	1			A53	10	1	
12	Pipe slide base modelled with 3 limit stops								
13	First limit stop is in vertical direction, Free to lift but not go down								
14	80		2			A53	10	1	Limit stop
15	Second limit stop is for lateral movement +/- 3"								
16	80	Location							Limit stop
17	Third limit stop is for axial movement +8", -0"								
18	80	Location							Limit stop
19	Pipe slide guide modelled with restraint and a limit stop								
20	The restraints are against vertical and lateral movements								
21	The limit stop is for axial movement +8", -0"								
22	90		3			A53	10	1	YZ restraint
23	90	Location							Limit stop
24	Pipe spider guide is modelled as guide								
25	100		3			A53	10	1	Guide
26	Roller support is modelled as vertical limit stop and lateral restraint								
27	110		3			A53	10	1	Limit stop
28	110	Location							Y restraint
29	120		3			A53	10	1	Anchor
30									

Physical tees, even though integral components, are not modeled as such in CAEPIPE. Instead, they are modeled as three pipes coming together at a common node.



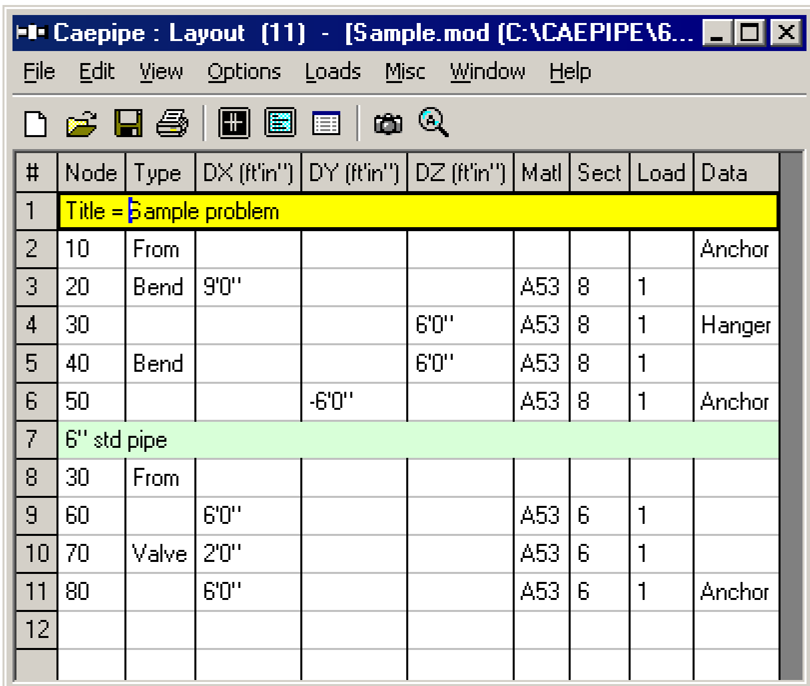
In this figure, you can see that three pipe elements (20-30, 30-40, and 30-60) come together at node 30 to form a tee.

Modeling

The method of modeling is as simple as its representation.

Step 1:

You could model the pipe run first from 20 to 30 (1st element), then from 30 to 40 (2nd element), and finally, from 30 to 60 (3rd element). See example layout window below.

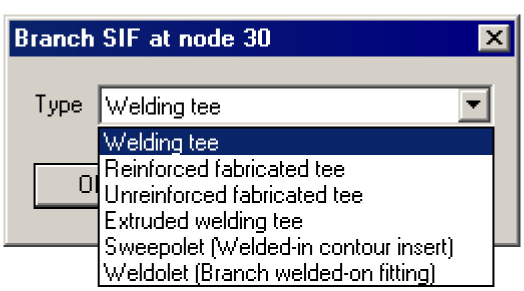


#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									

Alternately, you could model from 20 – 30 – 60, and then 30 – 40. Or from 60 – 30 – 20, and then 30 – 40. Modeling order is immaterial to analysis (but sometimes complicates the merge files process. See Merge under Layout window > Merge in the User's Manual).

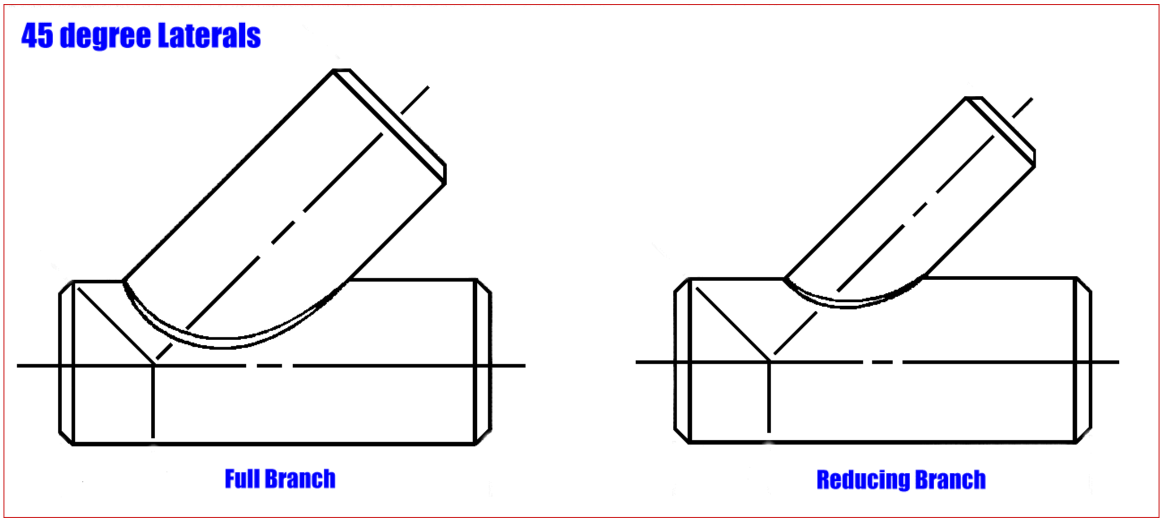
Step 2:

There is one final step remaining. You need to designate the type of tee connection it is. In this example, the tee is a "Welding Tee". So, on any row that contains node 30 (i.e., row #4 or row #8), type "Br" (Branch SIF) and select "Welding Tee" from the drop down list. Since row #4 already has a hanger specified, you can use row #8 to specify the Branch SIF.



The shown Branch SIF list is piping code-dependent. In other words, the list of tee types shown comes from the selected piping code that dictates how an SIF for each of the listed tee types is calculated. See Section titled "Piping Code Compliance" from the Code Compliance Manual for information on Branch SIFs for different piping codes.

Based on a more rigorous analysis, if you have another more accurate SIF value for a joint you want to insert (instead of the code's), then skip Step (2) above and use User-SIF data type at the same node to input your own SIF.

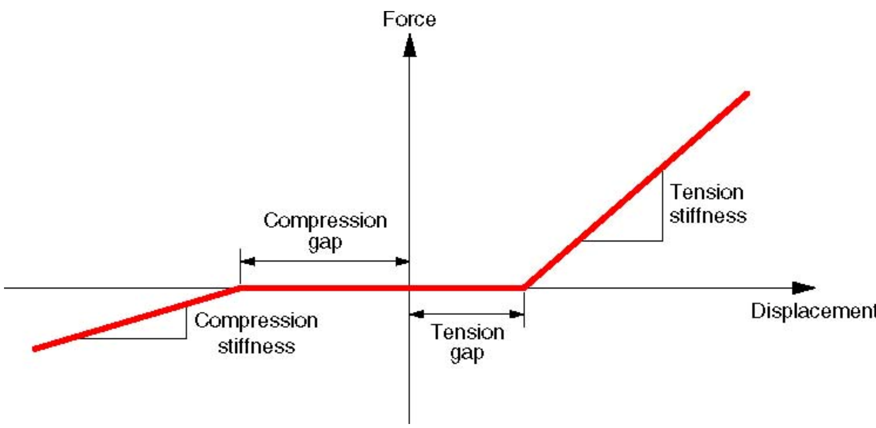


For 45° laterals, a piping code committee member opines as follows:

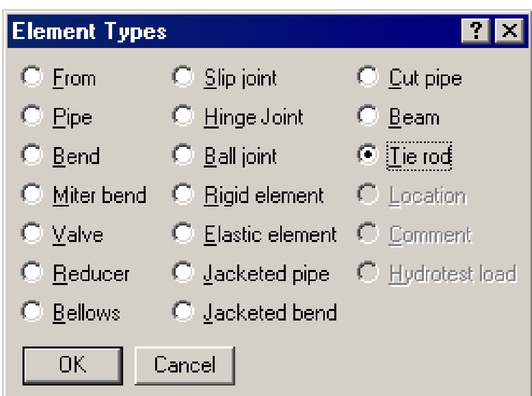
Use the SIF for an unreinforced fabricated tee and evaluate the branch stress using the section modulus of the branch. The connection footprint on the run pipe itself has a section modulus greater than the branch section modulus and this will compensate for the unreinforced fabricated tee SIF which should be lower than the unreinforced fabricated lateral SIF. Ignore the fact that a reduced outlet branch connection requires an "effective section modulus" in accordance with ASME B31.1 (2014) Para. 104.8.4(C) or ASME B31.3 (2014) Para. 319.4.4(c) and just calculate the intensified stress as the unreinforced fabricated tee SIF x the branch moment / branch section modulus (whether you use the SIF times the resultant moment approach of B31.1 or the in-plane and out-plane SIFs times the in-plane and out-plane moments, respectively, approach of B31.3, use the same philosophy of ignoring the "effective modulus."

Pay particular attention to the fabrication of the lateral making sure that the Code required cover fillet dimensioned t(c) in ASME B31.1 (2014) Fig. 127.4.8(D)(a) or ASME B31.3 (2014) Fig. 328.5.4D(1) meets the required size. If the cover fillet is larger than t(c), that improves the lateral SIF. The figures and dimensions shown are a bit unfortunate, especially for a lateral because a strict reading of the Code would seem to require a bigger weld on the obtuse side of the branch and a smaller weld on the acute side of the branch when just the opposite is true. Having a constant cover fillet weld leg length all around the branch would improve the requirement and the committee has been working on that. Personally, I think the larger the run and branch pipes are, the larger the cover fillet should be. Ask for or calculate whether area replacement requirements ASME B31.1 (2014) Para. 104.3.1(D) or ASME B31.3 (2014) Para. 304.3.3 are met noticing that the required reinforcement for a lateral is greater by the factor (2 - sin alpha).

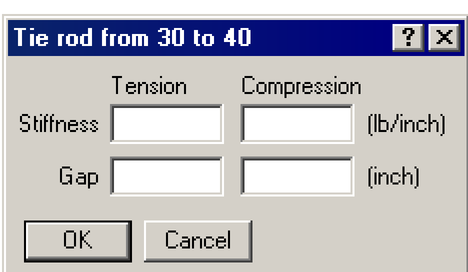
Tie rod is a nonlinear element with different stiffnesses and gaps in tension and compression, used to model tie rods in bellows, chains, etc. The force versus displacement relationship for a tie rod is shown below. Around expansion joints, a tie rod continuously restrains the full pressure thrust while allowing only lateral deflection, bending and torsional rotation.



When the tie rod is in tension, and the displacement is greater than the tension gap, tension stiffness is used. If the displacement is less than the tension gap, zero stiffness is used. Similarly when the tie rod is in compression, and the displacement is greater than the compression gap, compression stiffness is used. If the displacement is less than the compression gap, zero stiffness is used. A Tie rod is input by typing "t" in the Type column or selecting "Tie rod" from the Element Types dialog.



The Tie Rod dialog is shown.



A tie rod can be made "Tension only" by setting the compression stiffness to zero. Similarly it can be made "Compression only" by setting the tension stiffness to zero. Both Tension and Compression stiffnesses cannot be zero. If there is no tension or compression gap, leave it blank or specify it as zero. See the earlier "Expansion Joints" topic for examples.

Turbine

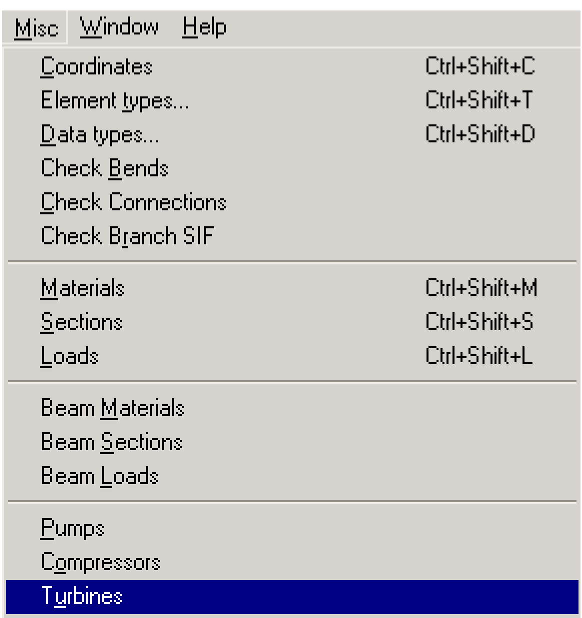
Pumps, compressors and turbines in CAEPIPE, referred to as rotating equipment, are each governed by an industry publication — API (American Petroleum Institute) publishes an API 610 for pumps and an API 617 for compressors while NEMA (National Electrical Manufacturers Association) publishes the NEMA SM-23 for turbines. These publications provide guidelines for evaluating nozzles connected to equipment among other technical information including the items relevant to piping stress analysis – criteria for piping design and a table of allowable loads.

Modeling the equipment is straightforward since it is assumed rigid (relative to connected piping) and modeled only through its end points (connection nozzles).

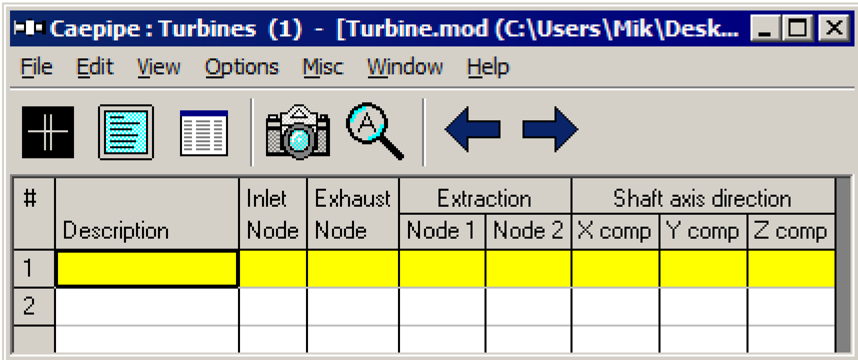
1. In your model, anchor all of the nozzles (on the equipment) that need to be included in the analysis.
2. Specify these anchored nodes during the respective equipment definition via Misc. menu > Pumps/Compressors/Turbines in the Layout window.

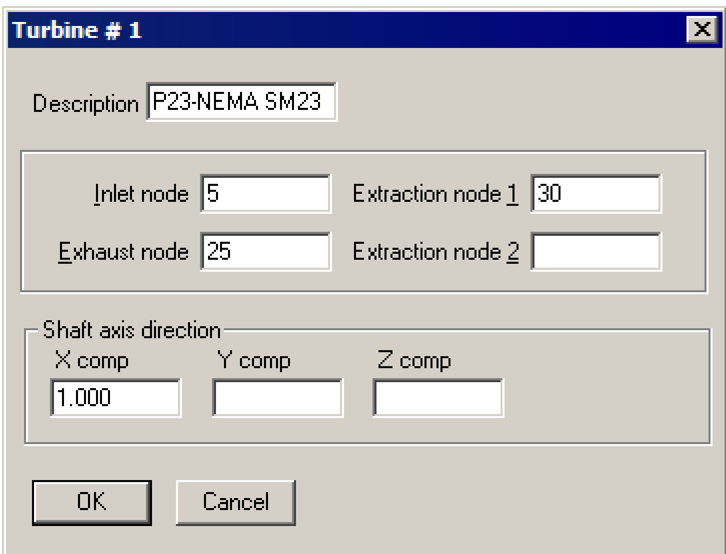
CAEPIPE does not require you to model all of the nozzles nor their connected piping. For example, you may model simply one inlet nozzle of a turbine with its piping. Or, you may model one turbine with all its nozzles (with no connected piping) and impose external forces on them (if you have that data). Further, there is no need to connect the anchors of the equipment with a rigid massless element like required in some archaic methods. A flange and an anchor may coexist.

A turbine (like a pump or a compressor) is input by selecting "Turbines" from the Miscellaneous (Misc) menu in the Layout window. CAEPIPE, upon analysis, produces a NEMA SM-23 turbine compliance report. See Section titled "Rotating Equipment Qualification" in the Code Compliance Manual for related information on NEMA SM-23, for Turbines.



Once you see the Turbine List window, double click on an empty row for the Turbine dialog and enter the required information.

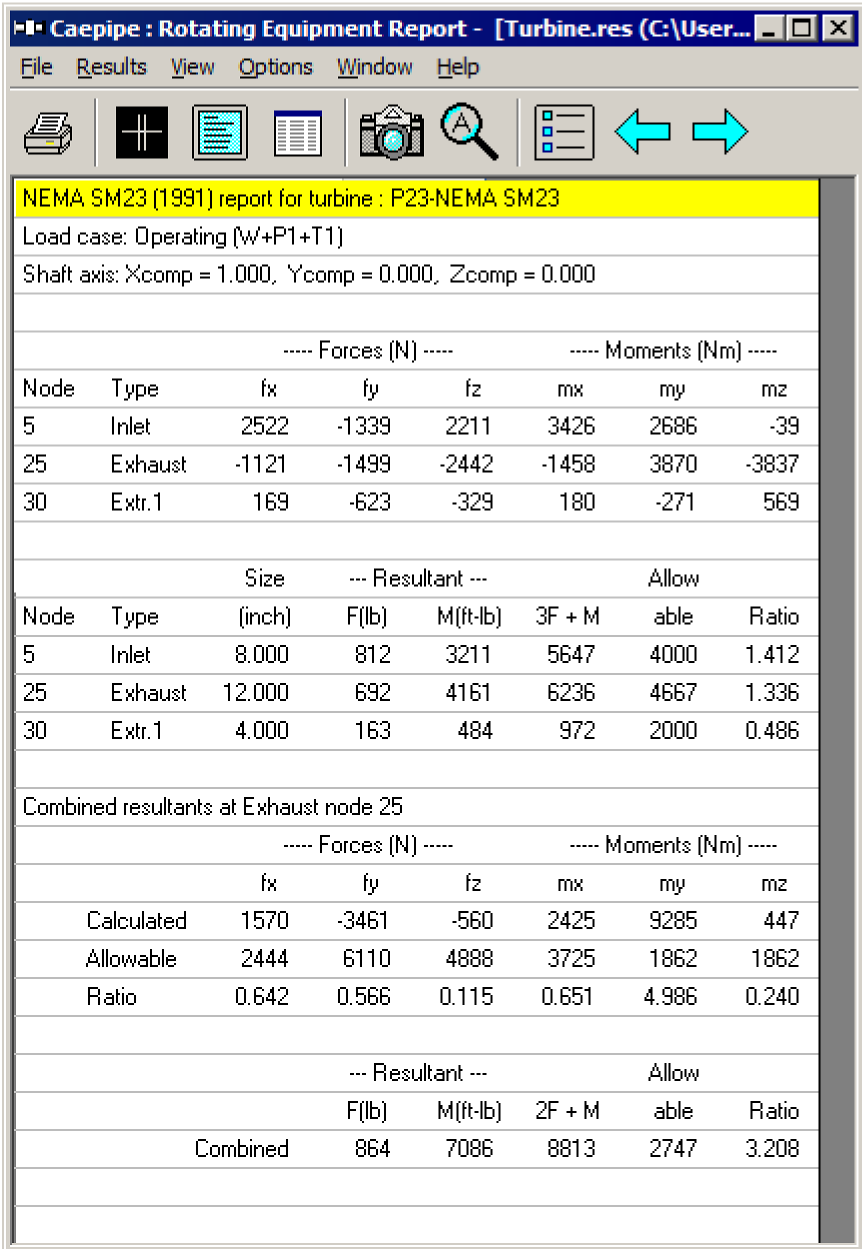




A short description to identify the turbine may be entered for Description. The nozzle nodes must be anchors and the shaft axis must be in the horizontal plane. Some of the nozzle nodes may be left blank if they are not on the turbine (e.g., extraction nodes).

See under Pumps for related modeling tips and the topic on specifying a Direction for information on how to specify X comp/Y comp/ Z comp for Shaft axis.

NEMA SM-23 Report



NEMA SM23 (1991) report for turbine : P23-NEMA SM23

Load case: Operating (W+P1+T1)

Shaft axis: Xcomp = 1.000, Ycomp = 0.000, Zcomp = 0.000

		----- Forces (N) -----			----- Moments (Nm) -----		
Node	Type	fx	fy	fz	mx	my	mz
5	Inlet	2522	-1339	2211	3426	2686	-39
25	Exhaust	-1121	-1499	-2442	-1458	3870	-3837
30	Extr.1	169	-623	-329	180	-271	569

		Size	--- Resultant ---			Allow	
Node	Type	(inch)	F(lb)	M(ft-lb)	3F + M	able	Ratio
5	Inlet	8.000	812	3211	5647	4000	1.412
25	Exhaust	12.000	692	4161	6236	4667	1.336
30	Extr.1	4.000	163	484	972	2000	0.486

Combined resultants at Exhaust node 25

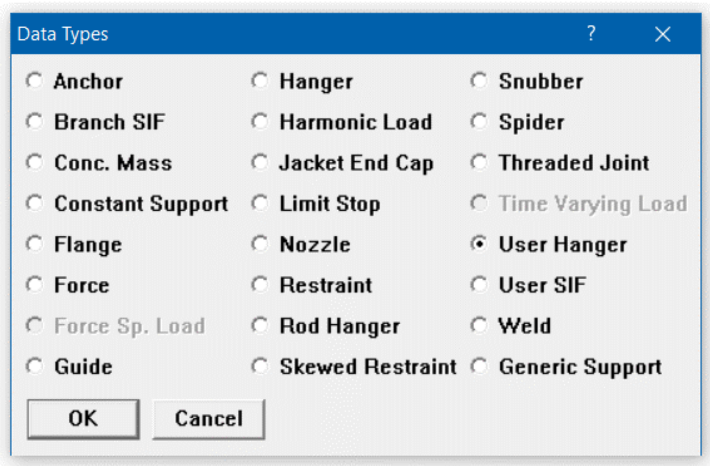
	----- Forces (N) -----			----- Moments (Nm) -----		
	fx	fy	fz	mx	my	mz
Calculated	1570	-3461	-560	2425	9285	447
Allowable	2444	6110	4888	3725	1862	1862
Ratio	0.642	0.566	0.115	0.651	4.986	0.240

	--- Resultant ---			Allow	
	F(lb)	M(ft-lb)	2F + M	able	Ratio
Combined	864	7086	8813	2747	3.208

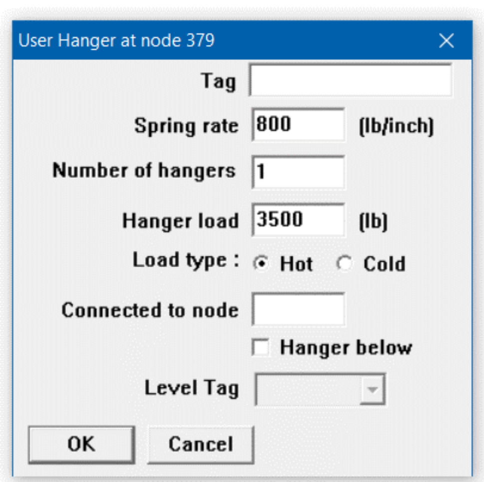
See Section titled "Rotating Equipment Qualification" of the Code Compliance Manual for more information on how to interpret the NEMA SM-23 report.

Note: If you have input multiple temperatures, corresponding reports for additional operating load cases are shown. Use the black right arrow key to see them.

Use the "User Hanger" type for analyzing piping systems with existing variable spring hangers, which are different from spring hangers that need "to be designed" (for which you use the "Hanger" data type).A user hanger is input by typing "u" in the Data column and pressing Enter or selecting "User Hanger" from the Data Types dialog.



The User Hanger dialog is shown.



Spring Rate

The spring rate is required. *For a constant support user hanger, input the spring rate as zero.*

Number of Hangers

Type in the number of separate hangers connected in parallel at this node. The stiffness and load of each hanger are multiplied by the number of hangers to find the effective stiffness and load of the hanger support at this node.

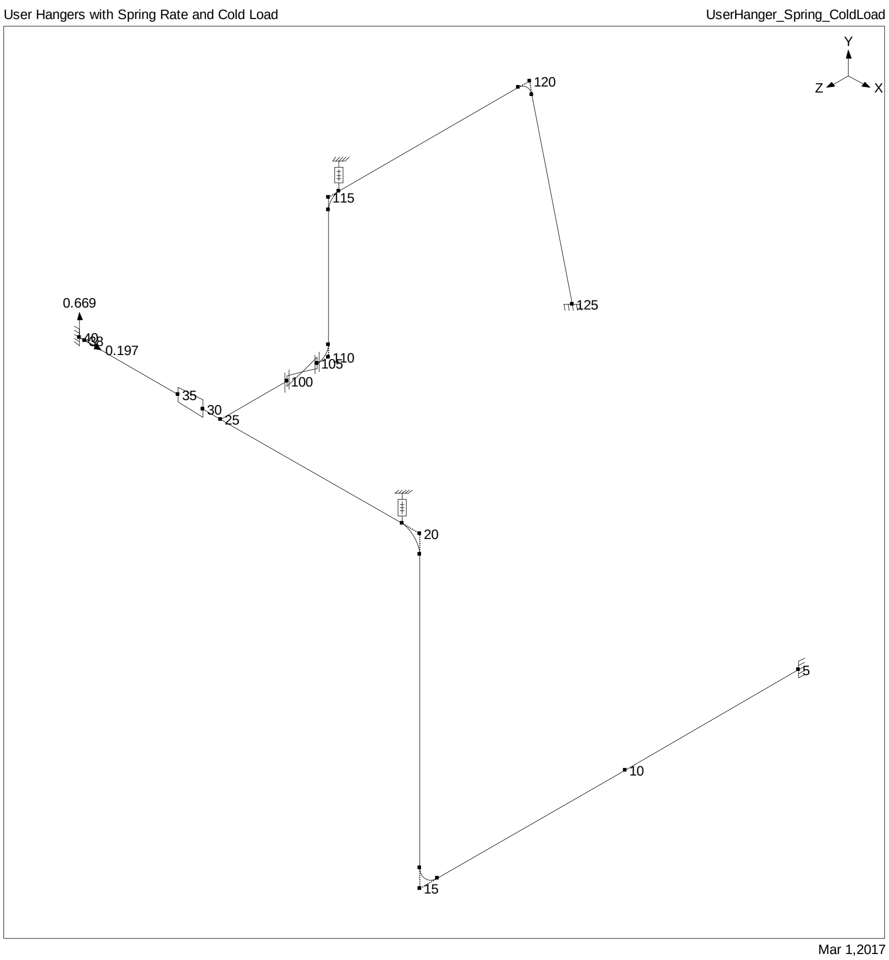
Hanger Load and Load type

Input the hanger load, if known. Otherwise, leave it blank and CAEPIPE will calculate the load.

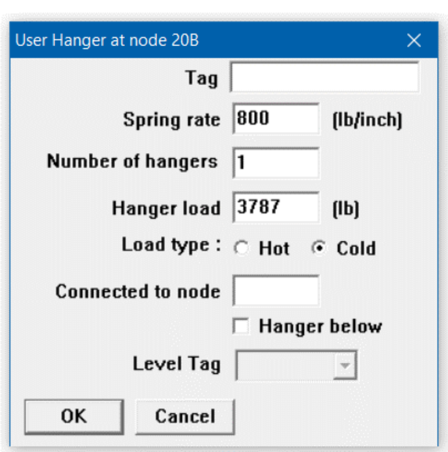
The hanger load may be specified as hot or cold using the Load type radio buttons.

When Cold Load and Spring Rate are input for User Hanger

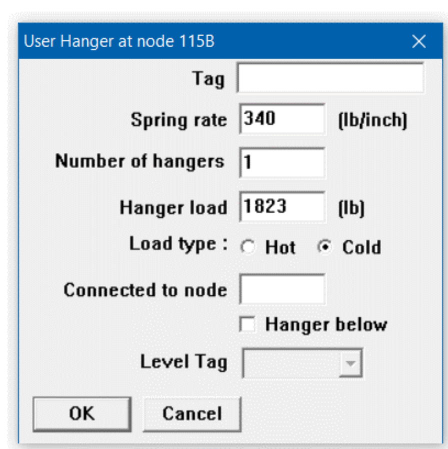
Snap shots shown below are from a CAEPIPE model with two (2) User Hangers defined at Nodes 20B and 115B. In this model, Spring rate and Cold load are input for each User Hanger as given below.



User Hanger at Node 20B



User Hanger at Node 115B



Analysis Options

Code : Piping code = B31.3 (2014)
Include axial force in stress calculations
Use liberal allowable stresses

Temperature : Reference temperature = 70 (F)
Number of thermal cycles = 7000
Number of thermal loads = 1
Thermal = Operating - Sustained
Use modulus at reference temperature

Pressure : Pressure stress = PD / 4t
Peak pressure factor = 1.00
Include Bourdon effect
Use pressure correction for bends

Dynamics : Cut off frequency = 33 Hz
Number of modes = 20
Do not include missing mass correction
Do not use friction in dynamic analysis

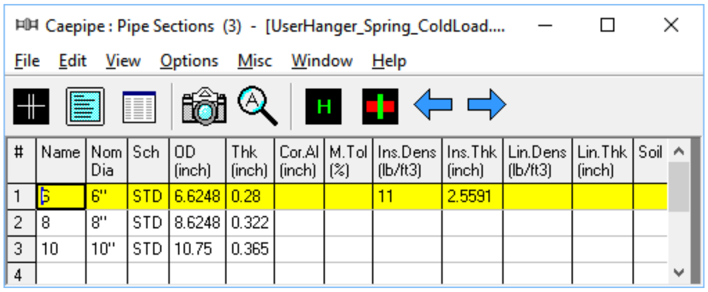
Misc. : Include hanger stiffness
Vertical direction = Y

Details of Layout

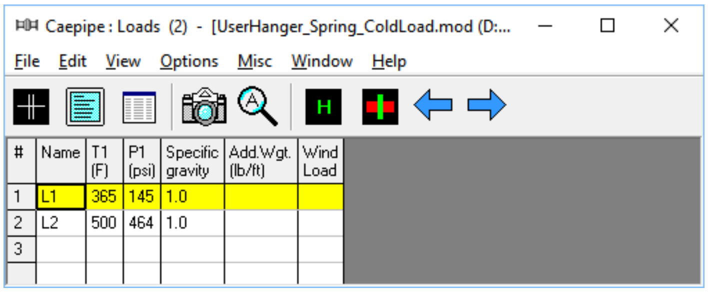
Caepipe : Layout (25) - [UserHanger_Spring_ColdLoad....

File Edit View Options Loads Misc Window Help

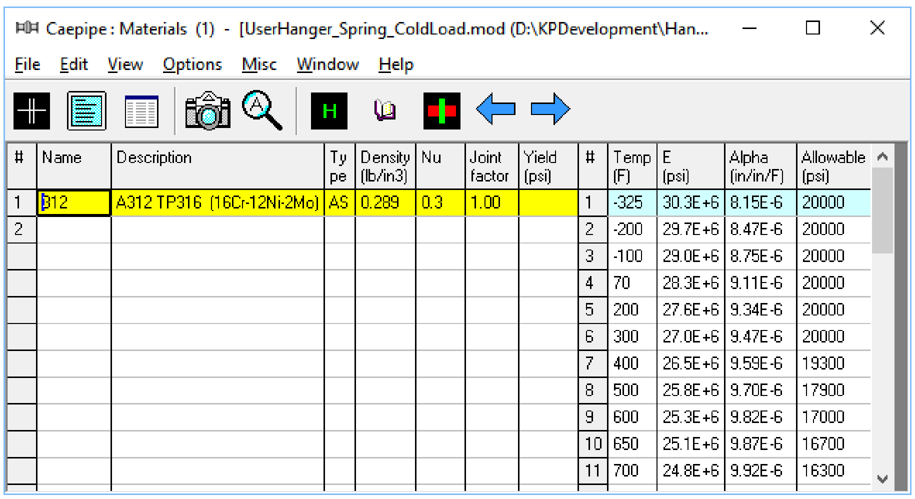
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = User Hangers with Spring Rate and Cold Load								
2	5	From							Anchor
3	10				12.1719	312	10	L1	
4	15	Bend			14.3373	312	10	L1	
5	15A	Location							
6	20	Bend		21.4895		312	10	L1	
7	20B	Location							
8	20B	Location							User hanger
9	25		-13.9108			312	10	L1	Welding tee
10	30		-1.2467			312	10	L1	
11	35	Reducer	-1.7388			312	10	L1	
12	38		-6.5617			312	8	L1	
13	40		-0.3281			312	8	L1	Harmonic load
14	40	Location							Anchor
15	6" Branch								
16	25	From							
17	100				-4.5932	312	6	L1	Flange
18	105	Valve			-2.1654	312	6	L1	Flange
19	105	Location							
20	110	Bend			-0.7382	312	6	L1	
21	115	Bend		9.6785		312	6	L2	
22	115B	Location							User hanger
23	115B	Location							
24	120	Bend			-14.0748	312	6	L2	
25	125		2.9856	-12.0079		312	6	L2	Anchor
26									



#	Name	Nom Dia	Sch	OD (inch)	Thk (inch)	Cor.Al (inch)	M.Tol (%)	Ins.Dens (lb/ft3)	Ins.Thk (inch)	Lin.Dens (lb/ft3)	Lin.Thk (inch)	Soil
1	6	6"	STD	6.6248	0.28			11	2.5591			
2	8	8"	STD	8.6248	0.322							
3	10	10"	STD	10.75	0.365							
4												



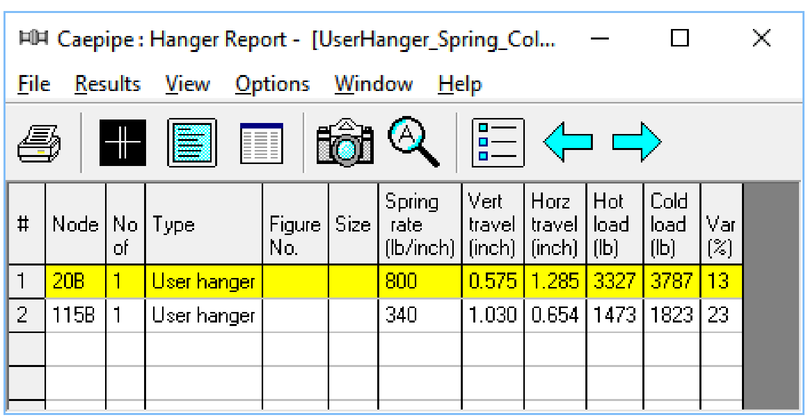
#	Name	T1 (F)	P1 (psi)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load
1	L1	365	145	1.0		
2	L2	500	464	1.0		
3						



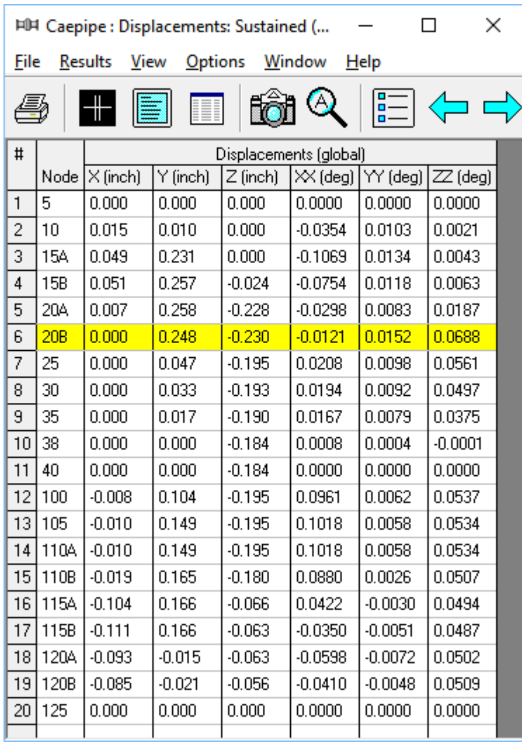
#	Name	Description	Type	Density (lb/in3)	Nu	Joint factor	Yield (psi)
1	312	A312 TP316 (16Cr-12Ni-2Mo)	AS	0.289	0.3	1.00	
2							

#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowable (psi)
1	-325	30.3E+6	8.15E-6	20000
2	-200	29.7E+6	8.47E-6	20000
3	-100	29.0E+6	8.75E-6	20000
4	70	28.3E+6	9.11E-6	20000
5	200	27.6E+6	9.34E-6	20000
6	300	27.0E+6	9.47E-6	20000
7	400	26.5E+6	9.59E-6	19300
8	500	25.8E+6	9.70E-6	17900
9	600	25.3E+6	9.82E-6	17000
10	650	25.1E+6	9.87E-6	16700
11	700	24.8E+6	9.92E-6	16300

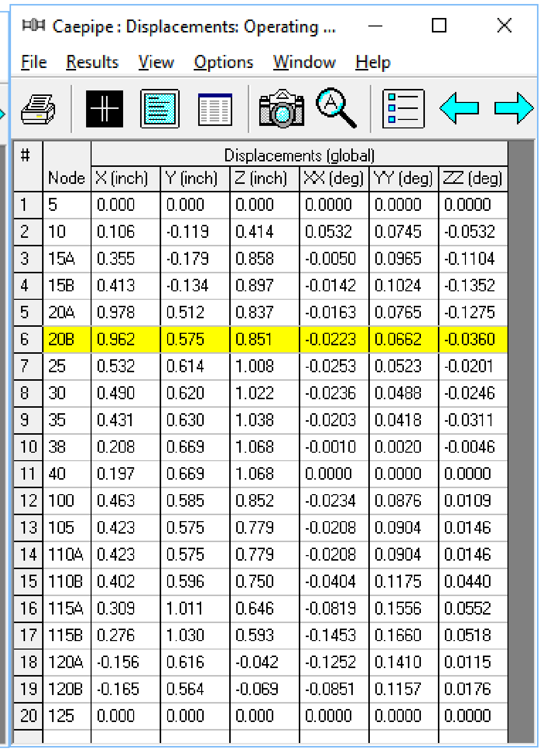
From the analysis results, Hanger Report as well as Displacements and Support Loads for Sustained & Operating load cases are presented below.



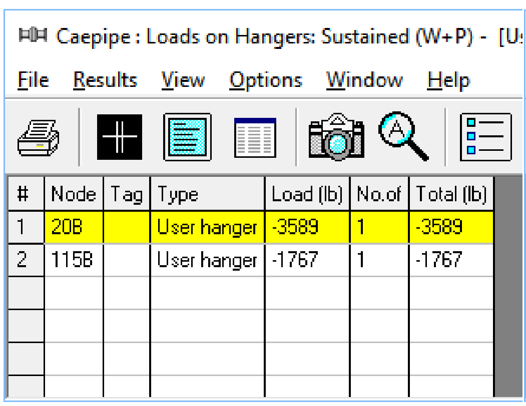
#	Node	No of	Type	Figure No.	Size	Spring rate (lb/inch)	Vert travel (inch)	Horz travel (inch)	Hot load (lb)	Cold load (lb)	Var (%)
1	20B	1	User hanger			800	0.575	1.285	3327	3787	13
2	115B	1	User hanger			340	1.030	0.654	1473	1823	23



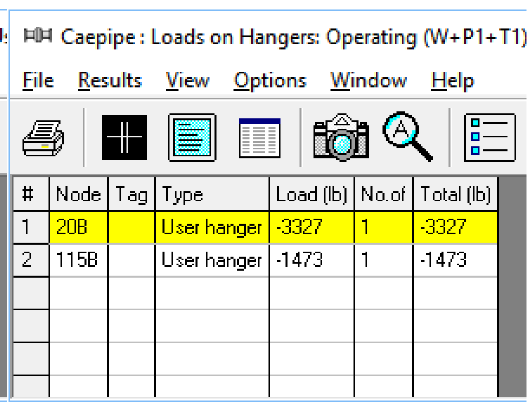
#	Displacements (global)						
	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	5	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	10	0.015	0.010	0.000	-0.0354	0.0103	0.0021
3	15A	0.049	0.231	0.000	-0.1069	0.0134	0.0043
4	15B	0.051	0.257	-0.024	-0.0754	0.0118	0.0063
5	20A	0.007	0.258	-0.228	-0.0298	0.0083	0.0187
6	20B	0.000	0.248	-0.230	-0.0121	0.0152	0.0688
7	25	0.000	0.047	-0.195	0.0208	0.0098	0.0561
8	30	0.000	0.033	-0.193	0.0194	0.0092	0.0497
9	35	0.000	0.017	-0.190	0.0167	0.0079	0.0375
10	38	0.000	0.000	-0.184	0.0008	0.0004	-0.0001
11	40	0.000	0.000	-0.184	0.0000	0.0000	0.0000
12	100	-0.008	0.104	-0.195	0.0961	0.0062	0.0537
13	105	-0.010	0.149	-0.195	0.1018	0.0058	0.0534
14	110A	-0.010	0.149	-0.195	0.1018	0.0058	0.0534
15	110B	-0.019	0.165	-0.180	0.0880	0.0026	0.0507
16	115A	-0.104	0.166	-0.066	0.0422	-0.0030	0.0494
17	115B	-0.111	0.166	-0.063	-0.0350	-0.0051	0.0487
18	120A	-0.093	-0.015	-0.063	-0.0598	-0.0072	0.0502
19	120B	-0.085	-0.021	-0.056	-0.0410	-0.0048	0.0509
20	125	0.000	0.000	0.000	0.0000	0.0000	0.0000



#	Displacements (global)						
	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	5	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	10	0.106	-0.119	0.414	0.0532	0.0745	-0.0532
3	15A	0.355	-0.179	0.858	-0.0050	0.0965	-0.1104
4	15B	0.413	-0.134	0.897	-0.0142	0.1024	-0.1352
5	20A	0.978	0.512	0.837	-0.0163	0.0765	-0.1275
6	20B	0.962	0.575	0.851	-0.0223	0.0662	-0.0360
7	25	0.532	0.614	1.008	-0.0253	0.0523	-0.0201
8	30	0.490	0.620	1.022	-0.0236	0.0488	-0.0246
9	35	0.431	0.630	1.038	-0.0203	0.0418	-0.0311
10	38	0.208	0.669	1.068	-0.0010	0.0020	-0.0046
11	40	0.197	0.669	1.068	0.0000	0.0000	0.0000
12	100	0.463	0.585	0.852	-0.0234	0.0876	0.0109
13	105	0.423	0.575	0.779	-0.0208	0.0904	0.0146
14	110A	0.423	0.575	0.779	-0.0208	0.0904	0.0146
15	110B	0.402	0.596	0.750	-0.0404	0.1175	0.0440
16	115A	0.309	1.011	0.646	-0.0819	0.1556	0.0552
17	115B	0.276	1.030	0.593	-0.1453	0.1660	0.0518
18	120A	-0.156	0.616	-0.042	-0.1252	0.1410	0.0115
19	120B	-0.165	0.564	-0.069	-0.0851	0.1157	0.0176
20	125	0.000	0.000	0.000	0.0000	0.0000	0.0000



#	Node	Tag	Type	Load (lb)	No.of	Total (lb)
1	20B		User hanger	-3589	1	-3589
2	115B		User hanger	-1767	1	-1767



#	Node	Tag	Type	Load (lb)	No.of	Total (lb)
1	20B		User hanger	-3327	1	-3327
2	115B		User hanger	-1473	1	-1473

Given below are the Steps performed by CAEPIPE to arrive at the results shown above for User Hangers when "Cold load" and "Spring rate" are input.

Step 1: Compute Hot Load from Preliminary Sustained Load Analysis

Performs a preliminary sustained load analysis and computes hot loads for the two User Hangers by replacing them with Vertical Restraints.

To verify this step, remove the User Hangers at nodes 20B and 115B and instead add Vertical Rigid Restraints at those two nodes and perform a sustained load analysis. From the Analysis results, you will observe that the "Loads on Restraints" at nodes 20B and 115B for Sustained load case will be identical to the hot loads reported above in the Hanger Report for the model with User Hangers.

Step 2: Compute Vertical Travel from Preliminary Operating Load Analysis

Vertical Restraints added in Step 1 above at User Hanger locations are removed. The hot loads (calculated in Step 1) are applied as upward forces at the hanger locations. Vertical displacements at the hanger locations obtained from the operating load case analysis are the hanger travels.

To verify this step, remove the Vertical Restraints at Nodes 20B and 115B from the model developed in Step 1 above. Apply the hot loads as upward forces at Nodes 20B and 115B by choosing "Add to W+P" in CAEPIPE's Force dialog and perform the analysis. Vertical displacement results at the User Hanger locations for the Operating load case will be identical to the Vertical Travel values reported above in the Hanger Report for the model with User Hangers.

Step 3: Perform Detailed Analysis

Performs once again the Sustained and Operating load case analyses by including the "Spring Rate" and "Cold load" input into CAEPIPE model at User Hanger locations.

To verify this step, to the model developed under Step 2 above add a skewed restraint in vertical direction at each User Hanger location with its stiffness equal to the "Spring Rate" for that User Hanger. In addition, add to the sustained load case (i.e., choose "Add to W+P" in CAEPIPE's Force dialog) an upward force at each skewed restraint node that is equal to the Cold load for the corresponding User Hanger. The resulting Step 3 model is then analyzed. Displacements for the Sustained and Operating load cases obtained from this Step 3 model will be identical to the displacements obtained for Sustained and Operating load cases reported above for the model with User Hangers.

Now, from the Step 3 model results, we observe the following at the two skewed restraints.

Sustained load case

Support load at Node 20B [A] = Spring Rate x Sustained displacement at Node 20B

= 800 lb/in x 0.248"

=198.4 lb (comparing well with CAEPIPE result of 198 lb)

Support load at Node 115B [B] = 340 lb/in x 0.166"

= 56.44 lb (comparing well with CAEPIPE result of 56 lb)

Operating load case

Support load at Node 20B [C] = Spring Rate x Operating displacement at Node 20B

= 800 lb/in x 0.575"

= 460 lb (comparing well with CAEPIPE result of 460 lb)

Support load at Node 115B [D] = 340 lb/in x 1.030"

= 350.2 lb (comparing well with CAEPIPE result of 350 lb)

Using the above Support load and Cold load, CAEPIPE is computing and reporting the Loads on User Hangers as follows.

When the option "Include hanger stiffness" is turned ON

Sustained load case

Load on Hanger at 20B = -Cold load + Spring Rate x Sustained displacement at Node 20B

= -Cold Load + A

= -3787 + 198.4 = **-3588.6 lb** (matches with Hanger loads results)

Operating load case

Load on Hanger at 20B = -Cold load + Spring Rate x Operating displacement at Node 20B

= -Cold Load + C

= -3787 + 460 = **-3327 lb** (matches with Hanger loads results)

When the option "Include hanger stiffness" is turned OFF

Sustained load case

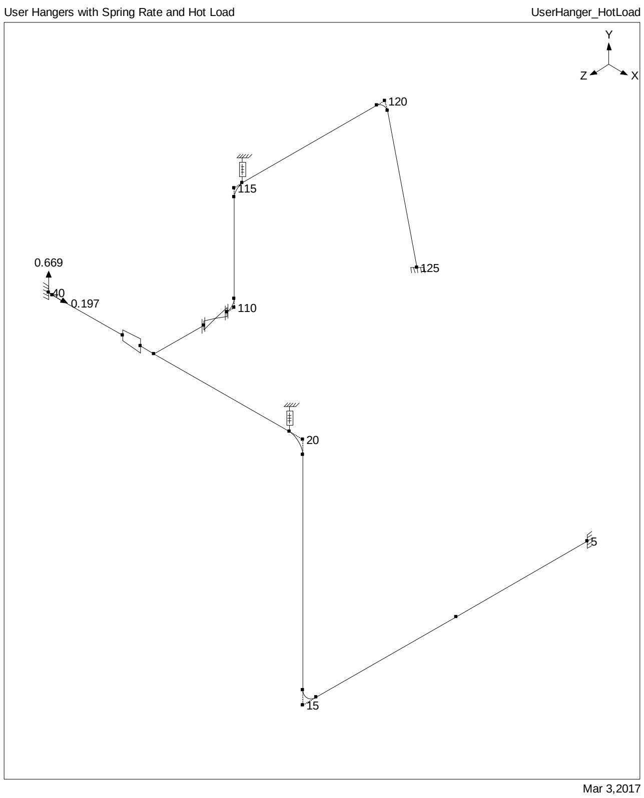
Load on Hanger @ 20B = **-Hot load = -3327 lb**

Operating load case

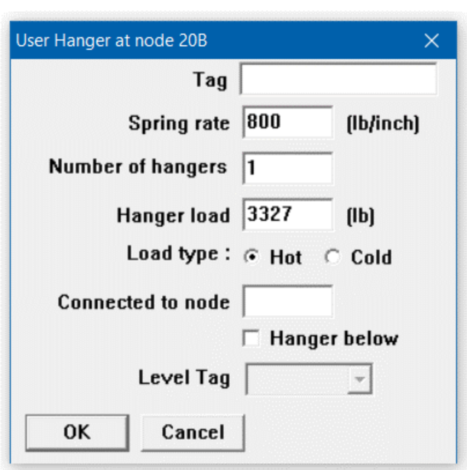
Load on Hanger @ 20B = **-Hot load = -3327 lb**

When Hot Load and Spring Rate are input for User Hanger

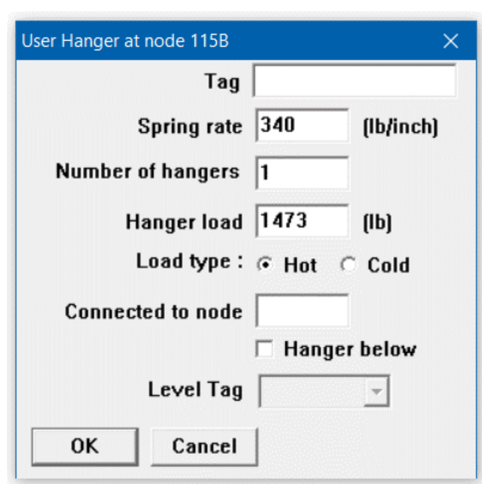
Snap shots shown below are from a CAEPIPE model with two (2) User Hangers defined at Nodes 20B and 115B. In this model, Spring rate and Hot load are input for each User Hanger as given below.



User Hanger at Node 20B



User Hanger at Node 115B



Analysis Options

Code	:	Piping code = B31.3 (2014) Include axial force in stress calculations Use liberal allowable stresses
Temperature	:	Reference temperature = 70 (F) Number of thermal cycles = 7000 Number of thermal loads = 1 Thermal = Operating - Sustained Use modulus at reference temperature
Pressure	:	Pressure stress = PD / 4t Peak pressure factor = 1.00 Include Bourdon effect Use pressure correction for bends
Dynamics	:	Cut off frequency = 33 Hz Number of modes = 20 Do not include missing mass correction Do not use friction in dynamic analysis
Misc.	:	Include hanger stiffness Vertical direction = Y

Details of Layout

Caepipe : Layout (25) - [UserHanger_HotLoad.mod ...

File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = User Hangers with Spring Rate and Hot Load								
2	5	From							Anchor
3	10				12.1719	312	10	L1	
4	15	Bend			14.3373	312	10	L1	
5	15A	Location							
6	20	Bend		21.4895		312	10	L1	
7	20B	Location							
8	20B	Location							User hanger
9	25		-13.9108			312	10	L1	Welding tee
10	30		-1.2467			312	10	L1	
11	35	Reducer	-1.7388			312	10	L1	
12	38		-6.5617			312	8	L1	
13	40		-0.3281			312	8	L1	Harmonic load
14	40	Location							Anchor
15	6" Branch								
16	25	From							
17	100				-4.5932	312	6	L1	Flange
18	105	Valve			-2.1654	312	6	L1	Flange
19	105	Location							Restraint
20	110	Bend			-0.7382	312	6	L1	
21	115	Bend		9.6785		312	6	L2	
22	115B	Location							User hanger
23	115B	Location							
24	120	Bend			-14.0748	312	6	L2	
25	125		2.9856	-12.0079		312	6	L2	Anchor
26									

Caepipe : Pipe Sections (3) - [UserHanger_HotLoad.mod ...

File Edit View Options Misc Window Help

#	Name	Nom Dia	Sch	OD (inch)	Thk (inch)	Cor.Al (inch)	M.Tol (%)	Ins.Dens (lb/ft3)	Ins.Thk (inch)	Lin.Dens (lb/ft3)	Lin.Thk (inch)	Soil
1	6	6"	STD	6.6248	0.28			11	2.5591			
2	8	8"	STD	8.6248	0.322							
3	10	10"	STD	10.75	0.365							
4												

Caepipe : Loads (2) - [UserHanger_HotLoad.mod...

File Edit View Options Misc Window Help

#	Name	T1 (F)	P1 (psi)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load
1	L1	365	145	1.0		
2	L2	500	464	1.0		
3						

Caepipe : Materials (1) - [UserHanger_HotLoad.mod (D:\KPDevelopment\Han...

File Edit View Options Misc Window Help

#	Name	Description	Type	Density (lb/in3)	Nu	Joint factor	Yie (ps
1	312	A312 TP316 (16Cr-12Ni-2Mo)	AS	0.289	0.3	1.00	
2							

#	Temp (F)	E (psi)	Alpha (in/in/F)	Allowat (psi)
1	-325	30.3E+6	8.15E-6	20000
2	-200	29.7E+6	8.47E-6	20000
3	-100	29.0E+6	8.75E-6	20000
4	70	28.3E+6	9.11E-6	20000
5	200	27.6E+6	9.34E-6	20000
6	300	27.0E+6	9.47E-6	20000
7	400	26.5E+6	9.59E-6	19300
8	500	25.8E+6	9.70E-6	17900
9	600	25.3E+6	9.82E-6	17000
10	650	25.1E+6	9.87E-6	16700
11	700	24.8E+6	9.92E-6	16300
12	750	24.5E+6	9.99E-6	16100
13	800	24.1E+6	10.05E-6	15900

From the analysis results, Hanger Report as well as Displacements and Support Loads for Sustained & Operating load cases are presented below.

Caepipe : Hanger Report - [UserHanger_HotLoad....

File Results View Options Window Help

#	Node	No of	Type	Figure No.	Size	Spring rate (lb/inch)	Vert travel (inch)	Horz travel (inch)	Hot load (lb)	Cold load (lb)	Var (%)
1	20B	1	User hanger			800	0.576	1.284	3327	3788	13
2	115B	1	User hanger			340	1.030	0.654	1473	1823	23

Caepipe : Displacements: Sustained (W+P) - ...

File Results View Options Window Help

#	Node	Displacements (global) X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	5	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	10	0.015	0.010	0.000	-0.0354	0.0103	0.0021
3	15A	0.049	0.231	0.000	-0.1070	0.0134	0.0043
4	15B	0.051	0.257	-0.024	-0.0754	0.0118	0.0063
5	20A	0.007	0.259	-0.228	-0.0298	0.0083	0.0187
6	20B	0.000	0.248	-0.230	-0.0121	0.0152	0.0689
7	25	0.000	0.047	-0.195	0.0208	0.0098	0.0562
8	30	0.000	0.033	-0.193	0.0195	0.0092	0.0498
9	35	0.000	0.017	-0.190	0.0167	0.0079	0.0376
10	38	0.000	0.000	-0.184	0.0008	0.0004	-0.0001
11	40	0.000	0.000	-0.184	0.0000	0.0000	0.0000
12	100	-0.008	0.104	-0.195	0.0962	0.0062	0.0538
13	105	-0.010	0.149	-0.195	0.1019	0.0058	0.0535
14	110A	-0.010	0.149	-0.195	0.1019	0.0058	0.0535
15	110B	-0.019	0.165	-0.180	0.0880	0.0026	0.0507
16	115A	-0.104	0.166	-0.067	0.0422	-0.0030	0.0494
17	115B	-0.111	0.166	-0.063	-0.0351	-0.0051	0.0487
18	120A	-0.093	-0.015	-0.063	-0.0599	-0.0073	0.0502
19	120B	-0.085	-0.021	-0.056	-0.0411	-0.0048	0.0509
20	125	0.000	0.000	0.000	0.0000	0.0000	0.0000

Caepipe : Displacements: Operating (W+P1+...

File Results View Options Window Help

#	Node	Displacements (global) X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	5	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	10	0.106	-0.119	0.414	0.0531	0.0745	-0.0532
3	15A	0.355	-0.179	0.858	-0.0051	0.0966	-0.1104
4	15B	0.413	-0.133	0.897	-0.0142	0.1025	-0.1352
5	20A	0.978	0.513	0.837	-0.0163	0.0765	-0.1274
6	20B	0.962	0.576	0.851	-0.0223	0.0662	-0.0359
7	25	0.532	0.614	1.008	-0.0253	0.0523	-0.0200
8	30	0.490	0.620	1.021	-0.0236	0.0488	-0.0246
9	35	0.431	0.630	1.038	-0.0203	0.0418	-0.0311
10	38	0.208	0.669	1.068	-0.0010	0.0020	-0.0046
11	40	0.197	0.669	1.068	0.0000	0.0000	0.0000
12	100	0.463	0.585	0.852	-0.0234	0.0876	0.0110
13	105	0.423	0.575	0.779	-0.0207	0.0904	0.0147
14	110A	0.423	0.575	0.779	-0.0207	0.0904	0.0147
15	110B	0.402	0.597	0.750	-0.0404	0.1175	0.0441
16	115A	0.309	1.011	0.646	-0.0819	0.1556	0.0552
17	115B	0.276	1.030	0.593	-0.1453	0.1660	0.0518
18	120A	-0.156	0.616	-0.042	-0.1252	0.1410	0.0115
19	120B	-0.165	0.564	-0.069	-0.0851	0.1157	0.0176
20	125	0.000	0.000	0.000	0.0000	0.0000	0.0000

Caepipe : Loads on Hangers: Sustained (W+P) - [U

File Results View Options Window Help

#	Node	Tag	Type	Load (lb)	No.of	Total (lb)
1	20B		User hanger	-3589	1	-3589
2	115B		User hanger	-1767	1	-1767

Caepipe : Loads on Hangers: Operating (W+P1+T1)

File Results View Options Window Help

#	Node	Tag	Type	Load (lb)	No.of	Total (lb)
1	20B		User hanger	-3327	1	-3327
2	115B		User hanger	-1473	1	-1473

Given below are the Steps performed by CAEPIPE to arrive at the results shown above for User Hangers when "Hot load" and "Spring rate" are input.

Step 1: Compute Vertical Travel from Preliminary Operating Load Analysis

Performs a preliminary operating load analysis by including the hot loads input as upward forces at the hanger locations. Vertical displacements at the hanger locations obtained from the operating load case analysis are the hanger travels.

To verify this step, remove the User Hangers at Nodes 20B and 115B and instead apply the respective hot loads as upward forces at those two nodes by choosing "Add to W+P" in CAEPIPE's Force dialog and perform the analysis. Vertical displacement results at the User Hanger locations for the Operating load case will be identical to the Vertical Travel values reported above in the Hanger Report for the model with User Hangers.

Step 2: Compute Cold Load

The Cold load is then computed using the Hot load entered and Vertical Travel obtained from Step 1 above as stated below.

Cold load = Hot load + Spring Rate x Vertical Travel

For example, at Node 20B,

Cold load = 3327 + 800 x 0.576 = 3787.8 lb.

Similarly, at Node 115B,

Cold load = 1473 + 340 x 1.030 = 1823.2 lb.

Step 3: Perform Detailed Analysis

Performs once again Sustained and Operating load case analyses by including the "Spring Rate" and "Cold load" (obtained in Step 2 above) into CAEPIPE model at User Hanger locations.

To verify this step, to the model developed under Step 1 above add a skewed restraint in vertical direction at each User Hanger location with its stiffness equal to the "Spring Rate" for that User Hanger. In addition, add to the sustained load case (i.e., choose "Add to W+P" in CAEPIPE's Force dialog) an upward force at each skewed restraint node that is equal to the Cold load computed in Step 2 above for the corresponding User Hanger. The resulting Step 3 model is then analyzed. Displacements for the Sustained and Operating load cases obtained from this Step 3 model will be identical to the displacements obtained for Sustained and Operating load cases reported above for the model with User Hangers.

Now, from the Step 3 model results, we observe the following at the two skewed restraints.

Sustained load case

Support load at Node 20B [A] = Spring Rate x Sustained displacement at Node 20B

= 800 lb/in x 0.248"

=198.4 lb (comparing well with CAEPIPE result of 198 lb)

Support load at Node 115B [B] = 340 lb/in x 0.166"

= 56.44 lb (comparing well with CAEPIPE result of 56 lb)

Operating load case

Support load at Node 20B [C] = Spring Rate x Operating displacement at Node 20B

= 800 lb/in x 0.576"

= 460.8 lb (matching with CAEPIPE result of 461 lb)

Support load at Node 115B [D] = 340 lb/in x 1.030"

= 350.2 lb (matching with CAEPIPE result of 350 lb)

Using the above Support load values and Cold load, CAEPIPE is computing and reporting the Loads on User Hangers as follows.

When the option "Include hanger stiffness" is turned ON

Sustained load case

Load on Hanger at 20B = -Cold load + Spring Rate x Sustained displacement at Node 20B

= -Cold Load + A

= -3787.8 + 198.4 = **-3589.4 lb** (matches with Hanger loads results)

Operating load case

Load on Hanger at 20B = -Cold load + Spring Rate x Operating displacement at Node 20B

= -Cold Load + C

= -3787.8 + 460.8 = **-3327 lb** (matches with Hanger loads results)

When the option "Include hanger stiffness" is turned OFF

Sustained load case

Load on Hanger at 20B = **-Hot load = -3327 lb**

Operating load case

Load on Hanger at 20B = **-Hot load = -3327 lb**

Connected to Node

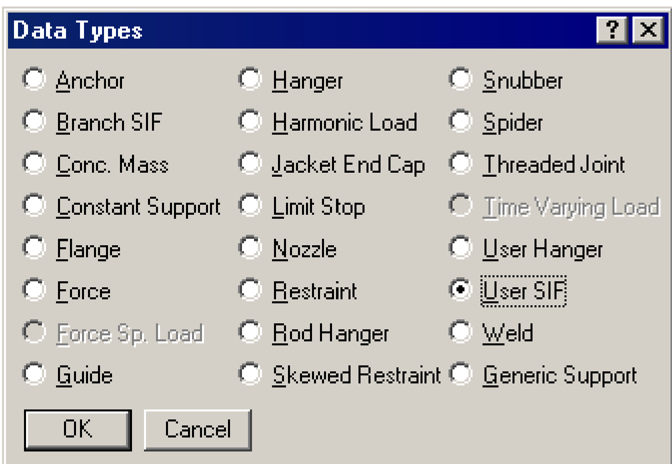
By default the hanger is connected to a fixed *ground* point which is not a part of the piping system. A hanger can be connected to another node in the piping system by entering the node number in the "Connected to" node field. This node *must be directly above or below* the hanger node.

User SIF

User SIF (Stress Intensification Factor) may be used to specify SIF at a node where there is normally no SIF internally calculated (i.e., at a non-bend or non-tee node) or to override any internally calculated SIF at the node.

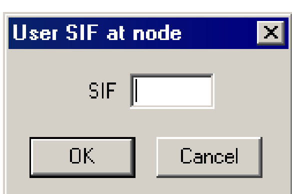
Use this for any component that needs an SIF value such as non-right angle tees, nonstandard tees or branch connections, flanges, etc., for which the chosen piping code does not specify a SIF, or you want to override the code's SIF. For example, in case of a bend or a tee, CAEPIPE calculates the SIF according to the selected piping code. To override the calculated SIF, specify a User SIF. *Note that a User SIF is applied to all elements that come together at this node.*

A User SIF is input by typing "user s" in the Data column or selecting "User SIF" from the Data types dialog.

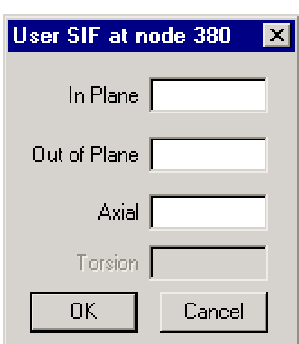


Depending on the piping code selected, either a single value User SIF (B31.1)or in-plane, out-of-plane and Axial values of User SIF (B31.3) or in-plane, out-of-plane, Axial and Torsion values of User SIF (B31J) may be input. The corresponding dialog will be shown.

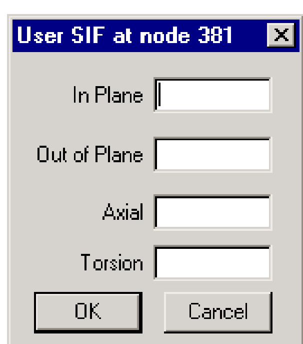
B31.1 code



B31.3 code

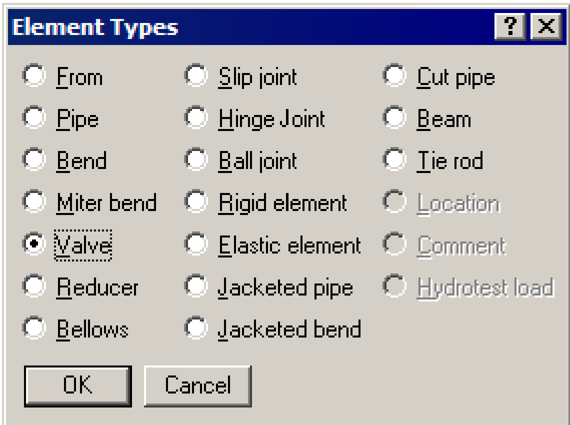


B31J Turned ON

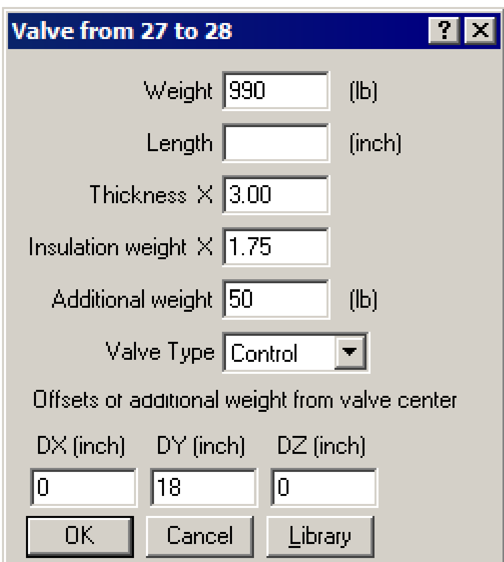


Use this element to model any type of valve. A valve is relatively more rigid than a pipe. CAEPIPE uses the data input to calculate the rigidity.

A Valve is input by typing "v" in the Type column or selecting "Valve" from the Element Types dialog.



The Valve dialog is shown.



Weight

The weight is the empty weight (without contents, insulation, etc.). CAEPIPE applies this weight as a uniformly distributed load along the length of the valve. Additional weight, if specified, is treated as a concentrated weight offset from the center of the valve.

CAEPIPE requires "Weight" to be input in lbf or kgf and NOT its "Mass". Whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass to be equal to (weight / g-value).

Length

If the valve length is input, the DX, DY, DZ in Layout is adjusted to match the valve length, (assuming that the local x-axis of valve is in the same direction as the local x-axis of the preceeding element). If the valve length is left blank, the valve length is calculated from DX, DY, DZ input in Layout.

Thickness X

The thickness multiplier (Thickness X) is used for stiffness calculation (i.e., the thickness of the pipe section is multiplied by Thickness multiplier by increasing only the OD of the valve and not changing its ID in the calculation of the valve stiffness). Typical value for Thickness multiplier is 3 which is the default value if left blank.

Insulation weight X

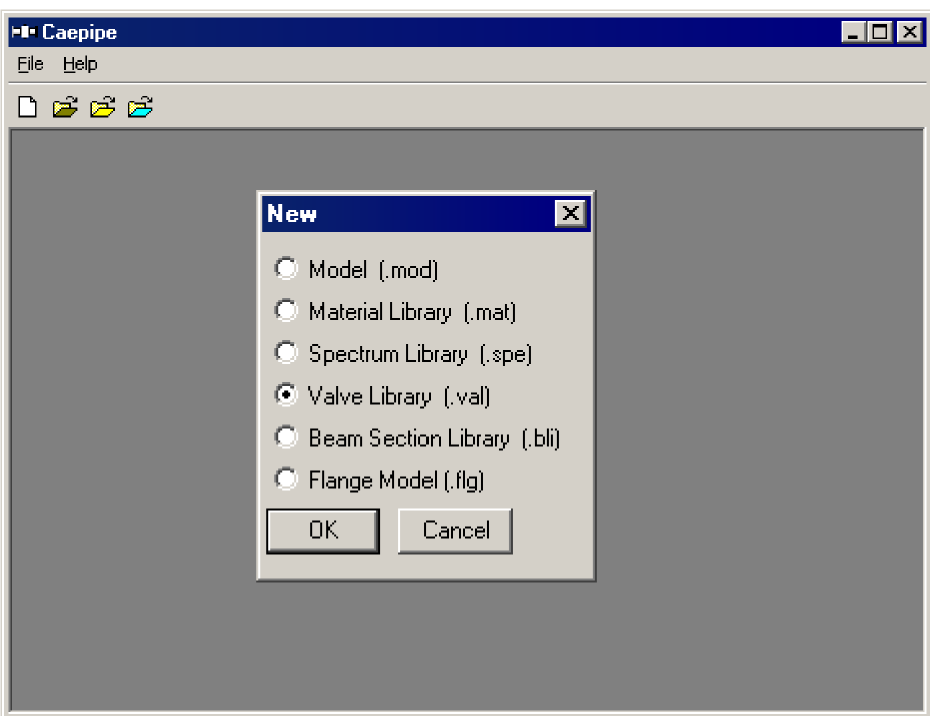
The insulation weight multiplier (Insulation weight X) is used if the valve has additional insulation compared to adjacent pipe (i.e., weight of insulation calculated from the insulation thickness of the pipe section is multiplied by Insulation weight X multiplier). Typical value for insulation weight multiplier is 1.75 which is the default value if left blank.

Additional weight

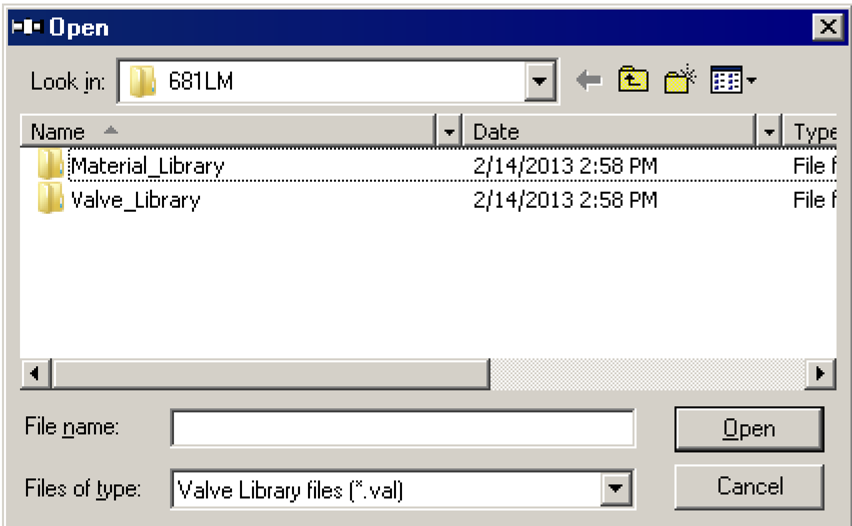
The additional weight is a concentrated weight which may be specified at an offset from the center of the valve, such as for a valve operator. As stated above, CAEPIPE requires "Additional Weight" to be input in lbf or kgf and NOT its "Mass".

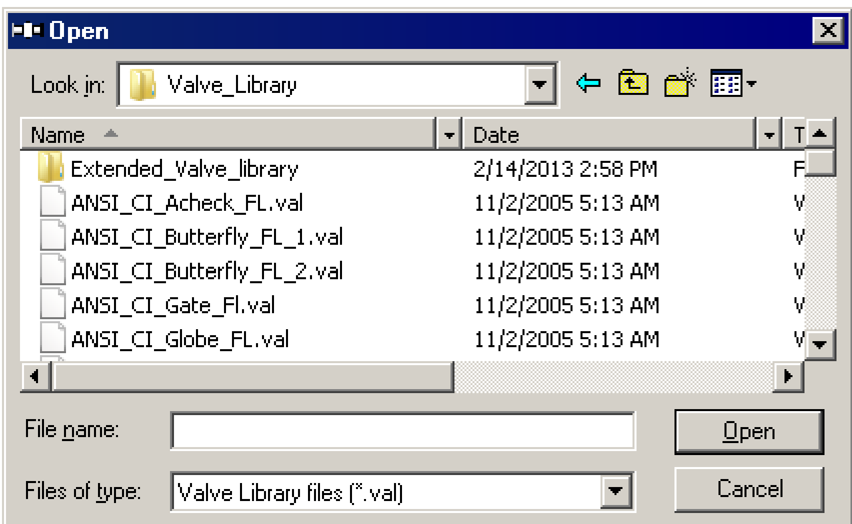
Valve Library

Cast Iron, Steel and Alloy valve (Flanged and Butt Welding ends) libraries are provided. The Type of Valve, Connection Type and Rating are indicative in the filenames listed in the libraries. Valve weights are included for different categories of valves. If necessary, you may create your own user-definable valve library. A new valve library can be created from menu File > New > Valve Library in the main opening CAEPIPE window.

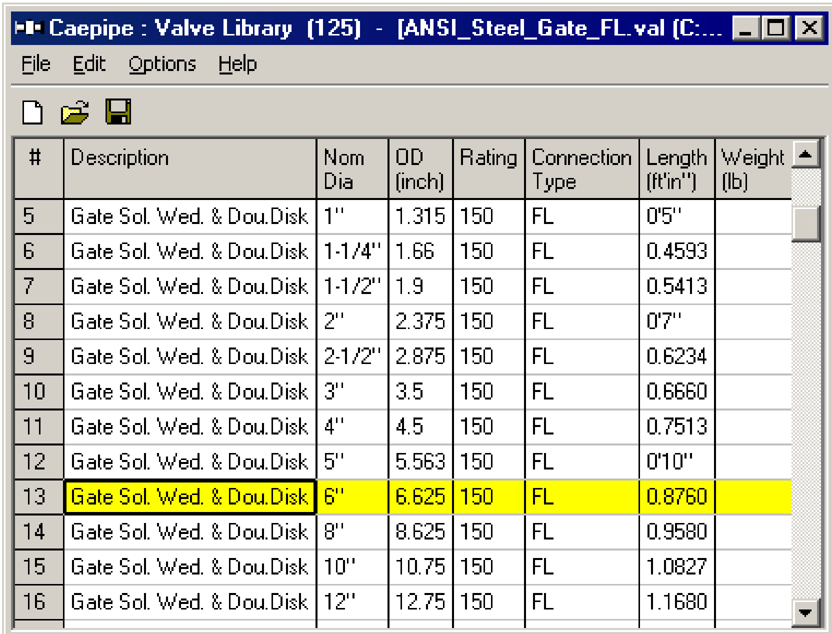


The valve library may be accessed by clicking on the Library button of the Valve dialog. Navigate to the folder called "Valve_Library" or similar under your CAEPIPE program files folder.



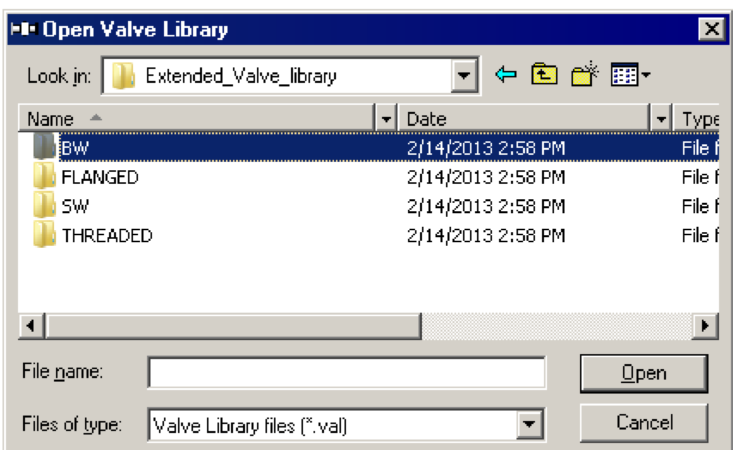


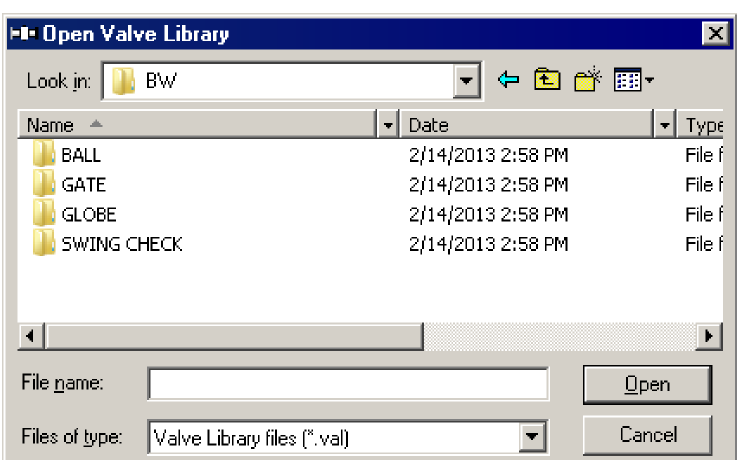
Selecting one of these valve types will display a list of valves for you to select from.

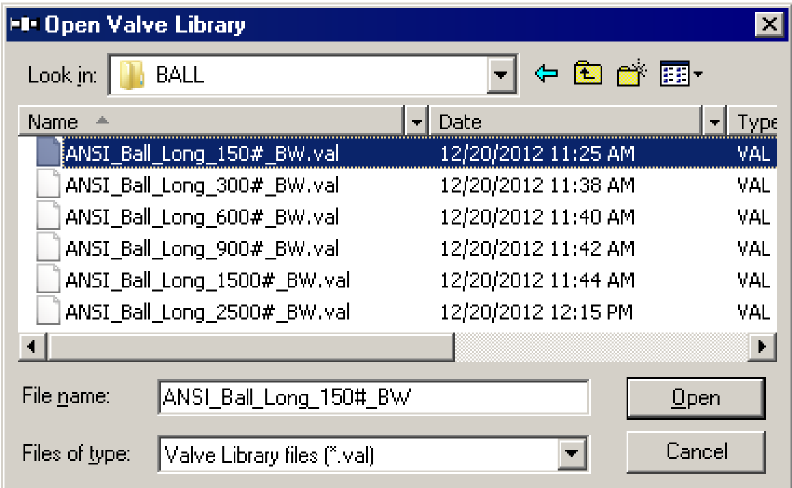


#	Description	Nom Dia	OD (inch)	Rating	Connection Type	Length (ft'in")	Weight (lb)
5	Gate Sol. Wed. & Dou.Disk	1"	1.315	150	FL	0'5"	
6	Gate Sol. Wed. & Dou.Disk	1-1/4"	1.66	150	FL	0.4593	
7	Gate Sol. Wed. & Dou.Disk	1-1/2"	1.9	150	FL	0.5413	
8	Gate Sol. Wed. & Dou.Disk	2"	2.375	150	FL	0'7"	
9	Gate Sol. Wed. & Dou.Disk	2-1/2"	2.875	150	FL	0.6234	
10	Gate Sol. Wed. & Dou.Disk	3"	3.5	150	FL	0.6660	
11	Gate Sol. Wed. & Dou.Disk	4"	4.5	150	FL	0.7513	
12	Gate Sol. Wed. & Dou.Disk	5"	5.563	150	FL	0'10"	
13	Gate Sol. Wed. & Dou.Disk	6"	6.625	150	FL	0.8760	
14	Gate Sol. Wed. & Dou.Disk	8"	8.625	150	FL	0.9580	
15	Gate Sol. Wed. & Dou.Disk	10"	10.75	150	FL	1.0827	
16	Gate Sol. Wed. & Dou.Disk	12"	12.75	150	FL	1.1680	

Select one from the displayed list. The weights of the valves (in the Extended Library) are provided. If a valve is flanged, the mating flanges at the two ends of the valve must be separately input using the Flange data type and their corresponding weight. Please confirm the data with your valve manufacturer's catalog, and input the correct weight.







Calculation of Moment of Inertia

The inside diameter of the valve is calculated from the section O.D. (outside dia.) and the section Thickness.

I.D. = Section O.D. − 2 ×Section Thickness

Then the new O.D. and Thickness for the valve are calculated as:

New Thickness = Section Thickness × Thickness X multiplier

New O.D. = I.D. (inside dia.) + 2 × New Thickness

The moment of inertia for the valve is now based on the New O.D. and New Thickness. For a thin-walled pipe, Thickness X and Inertia multipliers are approximately the same. The weight of the contents of the valve is based on the I.D. as calculated above.

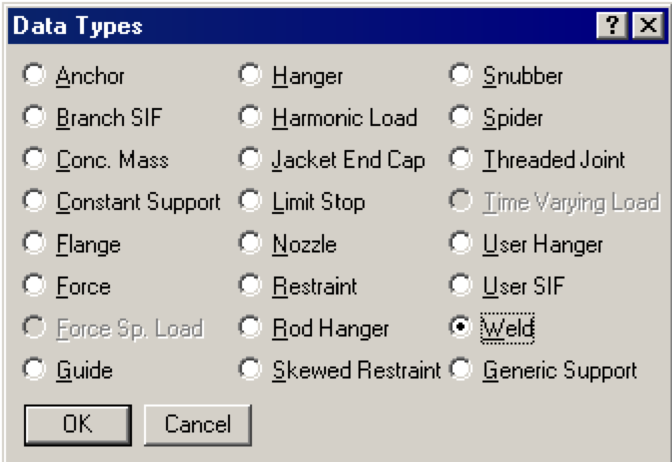
The weight of the insulation is the weight calculated from the section O.D. and insulation thickness and density (from section properties) multiplied by the insulation weight multiplier.

Angle/Relief Valve

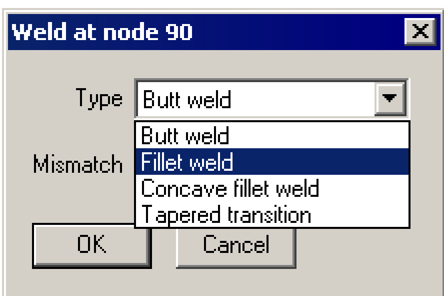
Angle and relief valves which have the outlet pipe at some angle (typically 90°) from the inlet pipe may be modeled by two valves one after the other and at that angle. The total weight of the actual valve must be divided between these two modeled valves.

Weld

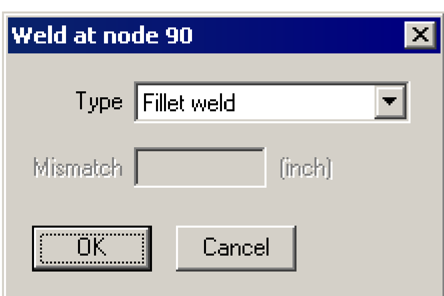
Use this data type to input a Weld at a node. Type "w" in the Data column or select "Weld" from the Data types dialog.



The Weld dialog is shown. Four types of welds are available: Butt, Fillet, Concave Fillet, and a Tapered transition. The type of the weld should be selected from the "Type" drop-down combo box.



Butt weld and Tapered transition require the input of weld mismatch. Mismatch is the difference in the mean radii across the weld.



The SIF for a weld is calculated according to the selected piping code (see Section titled "Piping Code Compliance" from the Code Compliance Manual for details) and is incorporated in the stress calculations. If you have an unlisted weld type, you could specify the SIF for it using the User SIF data type.

Any SIF value specified using the "User SIF" Data item will always overwrite any other SIF value calculated/determined at that node using any other method(s).

.Wind Load – ANSI A58.1 - 1982.

CAEPIPE calculates Design Wind Force F as follows to calculate wind load acting on each pipe element.

$$\mathbf{F = q \text{ x Shape factor x } A_f \text{ (lbs)}}$$

where,

A_f is the area of the piping plus insulation projected on a plane normal to the wind direction

q = Dynamic pressure due to wind (lbs/ft^2) = 0. 0.00256V^2 x I^2 x (K_z G_z C_f)

V = Basic Wind Speed, V (mph), from Basic Wind Speed Map, for your region.

Shape factor = 0.6 for Circular cross-section (to be input into CAEPIPE for Wind)

"Section 6" of ANSI A58.1 - 1982, Minimum Design Loads for Buildings and Other Structures can be referred to arrive at the values for [I^2 x (K_z G_z C_f)].

Importance Factor, I^2, is determined according to structure category and location:

Value of I^2 for:	**Other Areas**	**Within 100 mi of Ocean line**
Chemical plants, refineries, industrial facilities, and power plants not required inan emergency	1.00	1.11
Power Plants required in an emergency	1.15	1.24

Determine combined Velocity Coefficient, Gust Factor, and Force Coefficient, K_z G_z C_f) according to height of piping;

Height of Piping	**$K_zG_zC_f$**
At or below 50' above ground	1.23
Above 50' at or below 100' above ground	1.44
Above 100', at or below 200' above ground	1.68

Note:

If Wind Speed as a function of Elevation is input, the factor **SQRT**[(I^2 x (K_z G_z C_f))] has to be multipliedwith the Actual Wind Velocity as a function of elevation to arrive at the Wind Speed that is to be input at each elevation manually.

For example, if the Actual Wind Speed is 60 mph at an elevation of 50', I^2 as 1.00 and $K_z.G_z.C_f$ as 1.23, SQRT[1.00 x 1.23] = 1.109. So, V = 60 x 1.109 for up to 50' elevation = 66.54 mph. This should be input in the CAEPIPE Wind dialog.

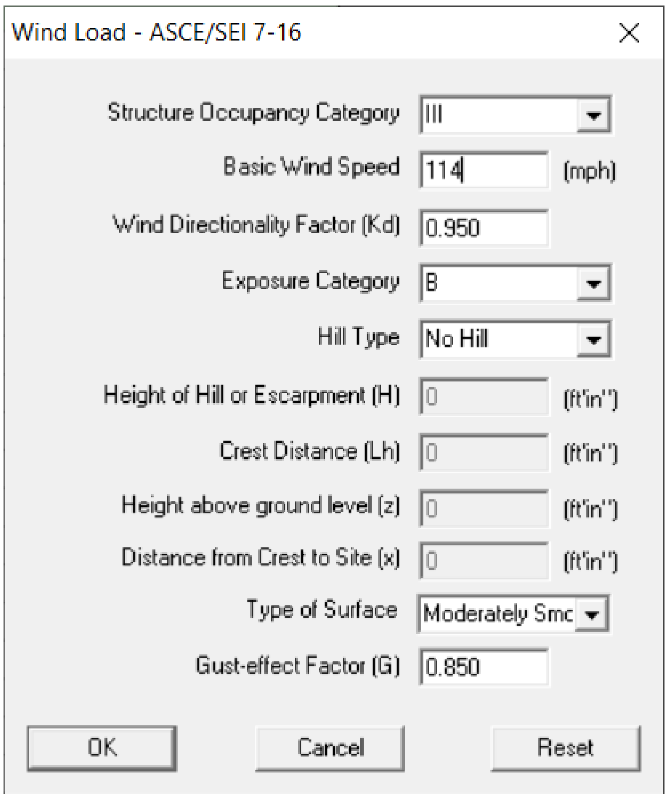
If Pressure as a function of Elevation is input in CAEPIPE, the factor [I^2 x (Kz Gz Cf)] and the Shape Factor have to be incorporated in the Pressure input at each elevation manually. This is because CAEPIPE currently uses the Shape Factor input into Wind dialog only for Velocity vs Elevation option and not for Pressure vs Elevation option.

.Wind Load – ASCE/SEI 7-16.

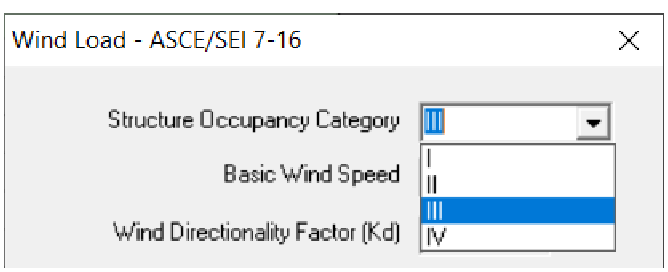
The determination of wind loads for the structural design of buildings is a complex subject that many building codes simplify by presenting tables of net wind pressures versus height above grade. Wind loads on a building in any particular locality depend on many factors, including recorded wind speeds in the area, the terrain around the building, and the shape and height of the building. It is now common for wind tunnel model tests to be conducted for tall buildings to determine wind loads - which may be more severe than the minimum code requirements.

American Society of Civil Engineers Standard (ASCE) SEI 7-16 contains detailed information and formulas for computing wind loads on buildings in various geographic locations. The procedure described in this code has been updated in CAEPIPE starting Version 10.30 and can be accessed through this option Layout Window > Misc > ASCE/SEI 7-16.

By defining the basic wind parameters in the dialog shown, CAEPIPE will compute the Design Wind Force internally as per ASCE/SEI 7-16 and apply the same to the piping system. For details on implementation, refer Appendix “ASCE/SEI 7-16” from Code Compliance Manual.



Structure Occupancy Category



Structure Occupancy Category (Risk category) can be I, II, III or IV as provided in the Table below.

Table 1.5-1 Risk Category of Buildings and Other Structures for Flood, Wind, Snow, Earthquake, and Ice Loads

Use or Occupancy of Buildings and Structures	Risk Category
Buildings and other structures that represent a low risk to human life in the event of failure	I
All buildings and other structures except those listed in Risk Categories I, III, and IV	II
Buildings and other structures, the failure of which could pose a substantial risk to human life. Buildings and other structures, not included in Risk Category IV, with potential to cause a substantial economic impact and/or mass disruption of day-to-day civilian life in the event of failure. Buildings and other structures not included in Risk Category IV (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, hazardous waste, or explosives) containing toxic or explosive substances where their quantity exceeds a threshold quantity established by the authority having jurisdiction and is sufficient to pose a threat to the public if released.	III
Buildings and other structures designated as essential facilities. Buildings and other structures, the failure of which could pose a substantial hazard to the community. Buildings and other structures (including, but not limited to, facilities that manufacture, process, handle, store, use, or dispose of such substances as hazardous fuels, hazardous chemicals, or hazardous waste) containing sufficient quantities of highly toxic substances where the quantity exceeds a threshold quantity established by the authority having jurisdiction to be dangerous to the public if released and is sufficient to pose a threat to the public if released.[a] Buildings and other structures required to maintain the functionality of other Risk Category IV structures.	IV

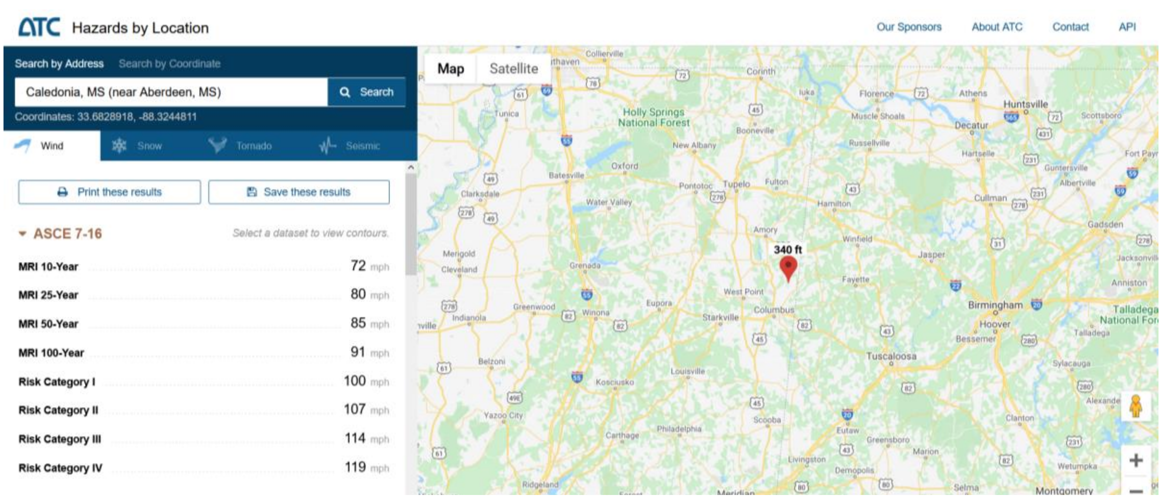
[a]Buildings and other structures containing toxic, highly toxic, or explosive substances shall be eligible for classification to a lower Risk Category if it can be demonstrated to the satisfaction of the authority having jurisdiction by a hazard assessment as described in Section 1.5.2 that a release of the substances is commensurate with the risk associated with that Risk Category.

Basic Wind Speed (V)

Depending upon the Risk Category, Basic Wind Speed (V) can be determined from the Figures 26.5.-1A, 26.5-1B and 26.5-1C provided in Chapter 26 of ASCE/SEI 7-16.

For example, Basic Wind Speed (V) at Caledonia, MS (near Aberdeen, MS) with Risk Category III is 114 mph (as per Figure 26.5-1B of ASCE/SEI 7-16).

Alternatively, one can use the link https://hazards.atcouncil.org/#/ to determine the Basic Wind Speed (V) as per ASCE/SEI 7-16 by entering the location as shown below.



Wind Directionality Factor (Kd)

This Wind Directionality Factor (Kd) accommodates the cross-sectional shape of the structure. The wind directionality factor (Kd) can be determined from Table 26.6-1 of ASCE/SEI 7-16. This table is provided below for quick reference.

Table 26.6-1 Wind Directionality Factor, K_d

Structure Type	Directionality Factor K_d
Buildings	
Main Wind Force Resisting System	0.85
Components and Cladding	0.85
Arched Roofs	0.85
Circular Domes	1.0[a]
Chimneys, Tanks, and Similar Structures	
Square	0.90
Hexagonal	0.95
Octagonal	1.0[a]
Round	1.0[a]
Solid Freestanding Walls, Roof Top Equipment, and Solid Freestanding and Attached Signs	0.85
Open Signs and Single-Plane Open Frames	0.85
Trussed Towers	
Triangular, square, or rectangular	0.85
All other cross sections	0.95

[a]Directionality factor $K_d = 0.95$ shall be permitted for round or octagonal structures with nonaxisymmetric structural systems.

For example, Chimneys, Tanks, and Similar Round Structures, Kd = 0.95.

Exposure Category

Exposure Category can be B, C or D.

For each wind direction considered, an exposure category that adequately reflects the characteristics of ground surface irregularities shall be determined for the site at which the building or structure is to be constructed. For a site located in the transition zone between categories, the category resulting in the largest wind forces shall apply. Account shall be taken of variations in ground surface roughness that arises from natural topography and vegetation as well as from constructed features.

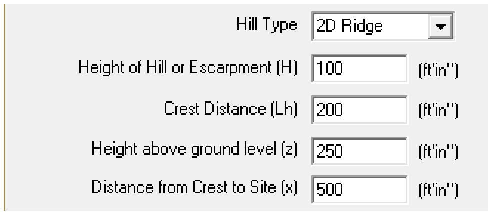
The exposure category for an individual structure shall be based upon the site conditions that will exist at the time when all adjacent structures on the site have been constructed, provided that their construction is expected to begin within one year of the start of construction for the structure for which the exposure category is determined.

For further details, refer para. 26.7.3 of ASCE/SEI 7-16 on Exposure Categories.

Topographical Parameters

Topography or large vertical displacements of the ground surface can have a significant effect on the wind speed profile. The wind flow in a realistic environment is not merely over a single ground feature such as hills, ridges, escarpment, but as well over undulating and mountainous terrain. It is important to understand that the flow over one hill will affect that around the next. The effects of undulating and mountainous terrain are almost similar to those of a very rough surface. Ridges and escarpments are mainly two dimensional land feature, and hills are mainly three dimensional.

Hills differ from ridges in that the wind can diverge over sides in addition to speeding up over crests. The speed-up effects of a hill are thus generally less than that those of a ridge of the identical slope. In general, the wind increases its speed when it moves up the windward slope of a hill or a ridge. The maximum increase in wind speed is usually experienced at or near the crest.



Parameters such as Hill Type, Height of Hill or Escarpment (H), Crest Distance (Lh), Height above ground level (z) and Distance from Crest to Site (x) are required by ASCE/SEI 7-16 for computing the Topographical Factor (Kzt) required in computing the Velocity Pressure (q_z).

Refer para. 26.8.2 of ASCE/SEI 7-16 for more details on Topographical Parameters.

Type of Surface

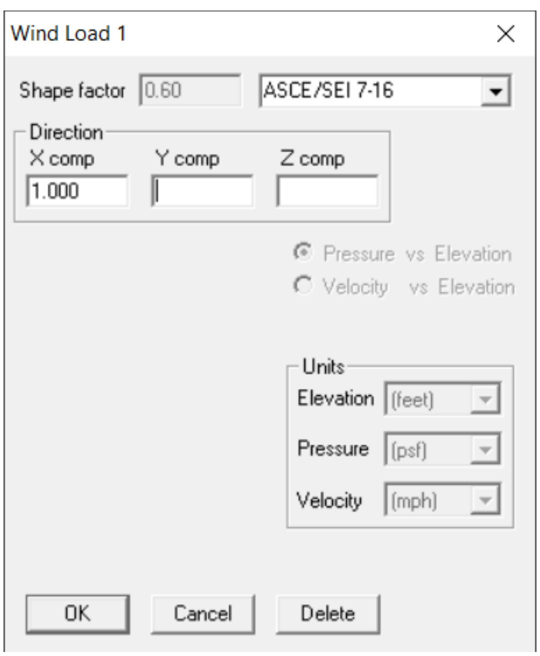
Type of Surface can be "Moderately Smooth", "Rough" or "Very Rough". This parameter is used to compute the force coefficient (Cf) required in computing the design wind force (F).

For further details, refer para. 29.5 from ASCE/SEI 7-16.

Gust-effect Factor (G)

As per para 26.9.1, Gust-effect factor for a rigid building and other structures is permitted to be taken as 0.85.

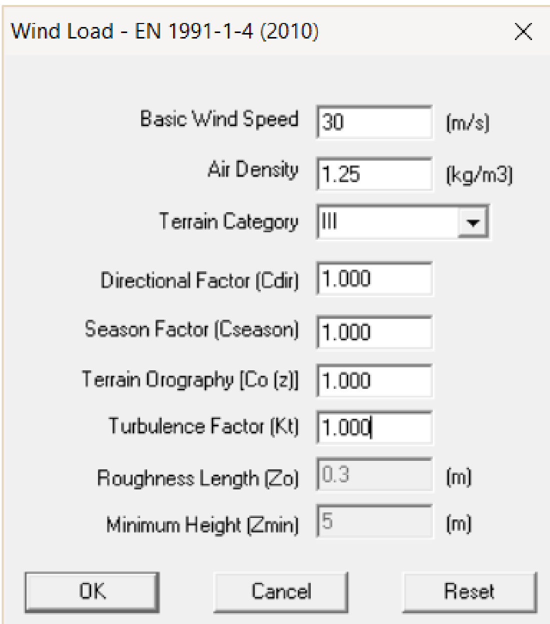
Upon defining the above parameters, the user can apply Wind Load by selecting the wind code as "ASCE/SEI 7-16" through Layout Window > Loads > Wind1/Wind2/Wind3/Wind4.



.Wind Load – EN 1991-1-4 (2010).

Eurocode 1 EN 1991-1-4 Action on structures (wind load) and EN 1991-1-4 (2010) contains detailed information and formulas for computing wind loads on buildings in various geographic locations. The procedure described in this code has been included in CAEPIPE starting Version 10.30 and can be accessed through this option Layout Window > Misc >Wind Load – EN 1991-1-4 (2010).

By defining the wind parameters in the dialog shown, CAEPIPE will compute the Design Wind Force internally as per EN 1991-1-4 (2010) and apply the same to the piping system. For details on implementation, refer to Appendix "EN 1991-1-4 (2010)" from Code Compliance Manual.



Fundamental value of the basic wind velocity can be obtained from National Annex for EN 1991-1-4.

Terrain Category

Based on the location of the structure, Terrain Category classification can be obtained from the Annex A and Table A.1 of EN 1991-1-4.

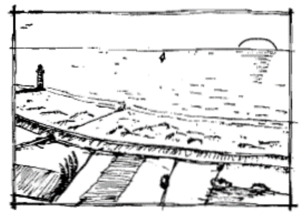
Table 4.1 — Terrain categories and terrain parameters

Terrain category		z_0 m	z_{min} m
0	Sea or coastal area exposed to the open sea	0,003	1
I	Lakes or flat and horizontal area with negligible vegetation and without obstacles	0,01	1
II	Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights	0,05	2
III	Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest)	0,3	5
IV	Area in which at least 15 % of the surface is covered with buildings and their average height exceeds 15 m	1,0	10
NOTE: The terrain categories are illustrated in A.1.			

A.1 Illustrations of the upper roughness of each terrain category

Terrain category 0

Sea, coastal area exposed to the open sea

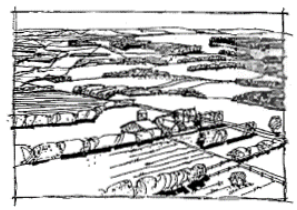


Terrain category I

Lakes or area with negligible vegetation and without obstacles

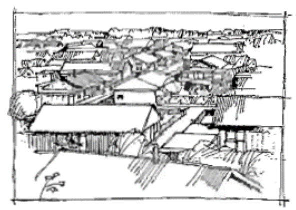
Terrain category II

Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights



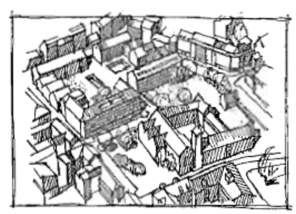
Terrain category III

Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest)



Terrain category IV

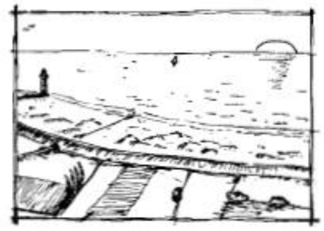
Area in which at least 15 % of the surface is covered with buildings and their average height exceeds 15 m



A.1 Illustrations of the upper roughness of each terrain category

Terrain category 0

Sea, coastal area exposed to the open sea

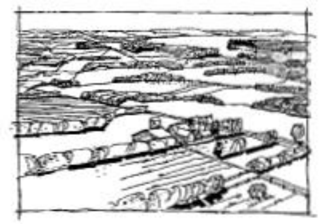


Terrain category I

Lakes or area with negligible vegetation and without obstacles

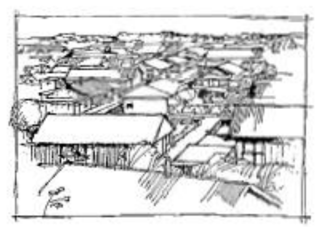
Terrain category II

Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights



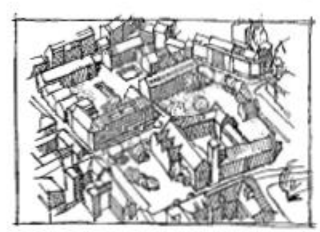
Terrain category III

Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest)



Terrain category IV

Area in which at least 15 % of the surface is covered with buildings and their average height exceeds 15 m



Directional and Seasonal Factors (Cdir & Cseason)

In order to calculate the Basic Wind Velocity as per Expression (4.1) of EN 1991-1-4 (2010), we need to determine the directional and seasonal factors (Cdir &Cseason). National Annex for EN 1991-1-4 simplifies this calculation as the suggested values of these factors are equal to 1.0.

Terrain Orography

The recommended value of the Terrain Orography factor is 1.0.

Where orography (e.g. hills, cliffs etc.) increases wind velocities by more than 5% the effects should be taken into account using the orography factor Co(z).

The procedure to be used for determining Co(z) may be given in the National Annex. The recommended procedure is given in A.3 of EN 1991-1-4 (2010).

The effects of orography may be neglected when the average slope of the upwind terrain is less than 3°. The upwind terrain may be considered up to a distance of 10 times the height of the isolated orographic feature.

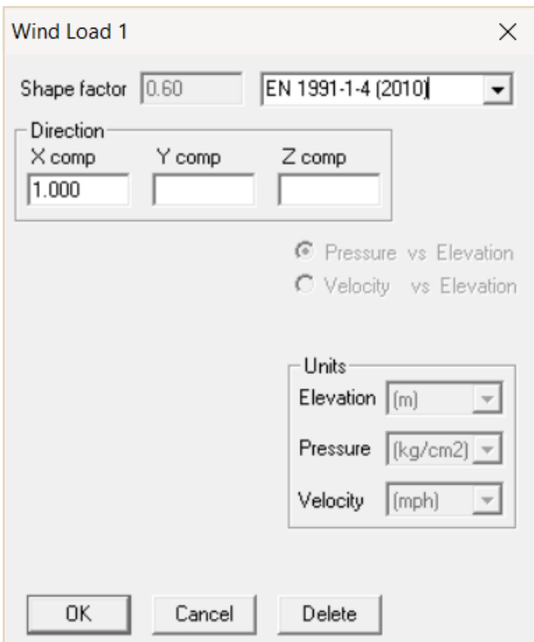
Turbulence Factor (Kt)

Turbulence factor (Kt) may be given in the National Annex. The recommended value for Kt is 1.0.

Roughness Length and Minimum Height

Roughness Length (Zo) and the Minimum Height (Zmin) are provided for each Terrain Category in Table 4.1 of EN 1991-1-4 (2010).

Upon defining the above parameters, the user can apply Wind Load by selecting the wind code as "EN 1991-1-4 (2010)" through Layout Window > Loads > Wind 1/Wind 2/Wind 3/Wind 4.



This page is blank

Annexure I

Dynamic Susceptibility Method

The "Dynamic Susceptibility" Method for Piping Vibration –*A Screening Tool for Potentially Large Alternating Stresses*

Dr. R. T. Hartlen, Plant Equipment Dynamics, Ontario, CANADA

Summary

The enhanced output of the "Modal Analysis" load case in CAEPIPE shows modal frequencies and mode shapes AND now two new outputs called ***"dynamic stresses"*** and ***"dynamic susceptibility".*** The *dynamicstresses* are the dynamic bending stresses associated with vibration in a natural mode. That is to say, the *modal analysis result has been generalized* to include the alternating bending stresses associated with the vibration in a natural mode. The *dynamic susceptibility* for any mode is the ratio of the maximum alternating bending stress to the maximum vibration velocity. This "susceptibility ratio" provides an indicator of the susceptibility of the system to large dynamic stresses. Also, the associated animated mode shapes include color-spot-markers identifying the respective locations of maximum vibration velocity and maximum dynamic bending stress. The susceptibility ratio and the graphics feature provide incisive insights into the reasons for high susceptibility and how to make improvements. This new feature is illustrated by application to the CAEPIPE "Sample problem" stress model.

1. Dynamic Susceptibility: New Analytical Tool Available for Vibration of Piping

When addressing vibration issues, the piping designer does not have the specific requirements, nor the analytical tools and technical references typically available for other plant equipment such as rotating machinery. Typically, piping vibration problems only become apparent at the time of commissioning and early operation, after a fatigue failure or degradation of pipe supports. Discovery of a problem is then followed by an ad hoc effort to assess, diagnose and correct as required. The "Dynamic Susceptibility" analysis, now included in CAEPIPE, provides a new analytical tool to assist the piping designer at any stage, from preliminary layout to resolution of field problems.

CAEPIPE's Dynamic Susceptibility feature utilizes the "Stress per Velocity" method, an incisive analytical tool for "screening" the vibration modes of a system. It readily identifies which modes, if excited, could potentially cause large dynamic stresses. Furthermore, it reveals which features of the system layout and support are responsible for the susceptibility to large dynamic stresses. At the design stage, the method allows the designer to quickly identify and correct features that could lead to large dynamic stresses at frequencies likely to be excited. Where problems are encountered in the field, the method provides quick and incisive support to efforts of observation, measurement, assessment, diagnosis and correction.

The technical foundation of this method lies in an underlying fundamental relationship between the kinetic energy of vibratory motion, and the corresponding potential energy stored in elastic stresses. That is to say, the kinetic energy at zero displacement and maximum system velocity must equal the stored elastic energy at zero velocity and maximum displacement. This implies a fundamental relationship between vibration velocity and dynamic bending stresses, which is the foundation of the stress per velocity approach for "susceptibility screening" of vibration modes.

The key analytical step is to determine, mode by mode, the ratio of maximum dynamic stress to maximum vibration velocity. This ratio will lie in a lower "baseline range" for uncomplicated systems such as classical uniform-beam configurations. For more complex systems, the stress / velocity ratio will increase due to typical complications such as three-dimensional layout, discrete heavy masses, changes of cross-section and susceptible branch connections. *System modes with large stress-velocity ratios are the potentially susceptible modes.*

The Stress / Velocity method, implemented in CAEPIPE as the Dynamic Susceptibility feature, *automatically and quickly finds these modes and quantifies the susceptibility*. Evaluation of the results, including special-purpose color animation, helps to identify which details of layout and support are responsible for the large stresses.

This technical note is to present and explain the "dynamic susceptibility" outputs now included in the modal analysis load case, and to illustrate by application to the standard CAEPIPE "Sample problem" system.

2. Underlying Fundamental Basis of the Method

2.1 Kinetic Energy and Potential Energy; Vibration Velocity and Dynamic Stresses

The underlying theoretical basis for the Stress / Velocity method is a deceptively straightforward but universally-applicable relationship between kinetic energy and potential (elastic) energy for vibrating systems. Stated simply, *for vibration at a system natural frequency,* the ***kinetic energy at maximum velocity and zero displacement*** must then be stored as ***elastic (strain) energy at maximum displacement and zero velocity.*** Since the strain energy and kinetic energy are respectively proportional to the squares of stress and velocity, it follows that dynamic stress, σ, will be proportional to vibration velocity, **v**. For idealized straight-beam systems, consisting of thin-walled pipe and with no contents, insulation or concentrated mass, the ratio σ / **v** is *dependent primarily upon material properties, (density* ρ *and modulus* **E***) ,and is remarkably independent of system-specific dimensions, natural-mode number and vibration frequency*. For real continuous systems of course, the kinetic and potential energies are distributed over the structure in accordance with the respective modes shapes. However, *integrated over the structure,* the underlying energy- equality holds true. Provided the spatial distributions are sufficiently similar, i.e. harmonic functions, the RMS or maximum stress will still be directly related to the RMS or maximum vibration velocity.

2.2 The "Screening" Approach

As stated above, for idealized pure beam systems the stress-velocity ratio will depend primarily upon material properties.

For real systems, the spatial patterns of the mode shapes will depart from the idealized harmonic functions, and the *stress-velocity ratios accordingly increase above the theoretical minimum or baseline value*. System details causing the ratios to increase would include the three-dimensional layout, large unsupported masses, high-density contents in thin-walled pipe, susceptible branch connections, changes of cross section, etc. *The more "unfavorable "the system layout and details are, the larger the* σ / **v** *ratios for some modes will be.*

Thus, the general susceptibility of a system to large dynamic stresses can be assessed by *determining the extent to which the* σ / **v** *ratios for any mode exceed the baseline range.*

Furthermore, by determining which particular modes have the high ratios, and whether these modes are known or likely to be excited, the at-risk vibration frequencies and mode shapes are identified for further assessment and attention. This is the basis of the Stress / Velocity method of analysis and its implementation as the "dynamic susceptibility" feature in CAEPIPE.

2.3 Relation to Velocity-based Vibration Acceptance Criteria

There are various general and application-specific acceptance criteria based upon vibration velocity as the quantity of record. Some, in order to cover the worst case scenarios, are overly conservative for many systems. Others are presented as being applicable only to the first mode of simple beams, leading to the misconception that the stress / velocity relationship does not apply at all to higher modes. In any case, there are real and perceived limitations on the use of screening acceptance criteria based upon a single value of vibration velocity.

The dynamic susceptibility method turns this apparent limitation into a useful analytical tool! Specifically, large stress / velocity ratios, well above the baseline values, are recognized as a "warning flag." Large values indicate that some feature(s) of the system make it particularly susceptible to large dynamic stresses in specific modes.

3 What the Dynamic Susceptibility Method Does

3.1 General Approach

The Dynamic Susceptibility method is essentially a post processor to fully exploit the modal analysis results of the system. Mode shape tables of dynamic bending stress and vibration velocity are searched for their respective maxima. Dividing the maximum stress by the maximum velocity yield the "**σ/v ratio**" for each mode. That ratio is the basis for assessing the susceptibility to large dynamic stresses. Larger values indicate higher susceptibility associated with specific details of the system.

3.2 Specific Implementation in CAEPIPE

The Stress / Velocity method has been implemented as additional analysis and output of the CAEPIPE modal analysis. The modal analysis load case now includes additional outputs and features as follows:

Dynamic Stresses	This output provides the "mode shapes" of dynamic bending stresses, tabulated along with the conventional mode shape of vibration magnitude.
Dynamic Susceptibility	This output is a table of s/v ratios, in psi / ips, mode by mode, in rank order of decreasing magnitude. In addition to modal frequencies and s/v ratios, the table also includes the node locations of the maxima of vibration amplitude and bending stresses. With the dynamic susceptibility output selected, the animated graphic display of the vibration mode shape includes the added feature of color spot markers showing the locations of maximum vibration and maximum dynamic bending stress.

These outputs will assist the designer through a more-complete understanding of the system's dynamic characteristics. They provide incisive quantified insights into how specific details of components, layout and support could contribute to large dynamic stresses, and into how to make improvements.

4 What the Dynamic Susceptibility Method Does Not Do Directly

The Stress / Velocity method of assessment, and its implementation in CAEPIPE as dynamic susceptibility, is *based entirely upon the system's dynamic characteristics per se.* Thus the vibration velocities and dynamic stresses employed in the analysis, although directly related to each other, are of *arbitrary magnitude.* There is no computation of the response to a prescribed forcing function, and no attempt to calculate actual dynamic stresses. Thus the dynamic susceptibility results ***do not factor directly into a pass-fail code compliance consideration.*** Rather, they assist the designer to assess and reduce susceptibility to large dynamic stresses if necessary, in order to meet whatever requirements have been specified.

An example follows next.

5 Illustrative Example of the Dynamic Susceptibility Analysis

The "dynamic susceptibility" feature of CAEPIPE will be illustrated here by application to the standard CAEPIPE Example system. The modal analysis was performed for frequencies up to 200 Hz, resulting in a reporting-out for 12 modes.

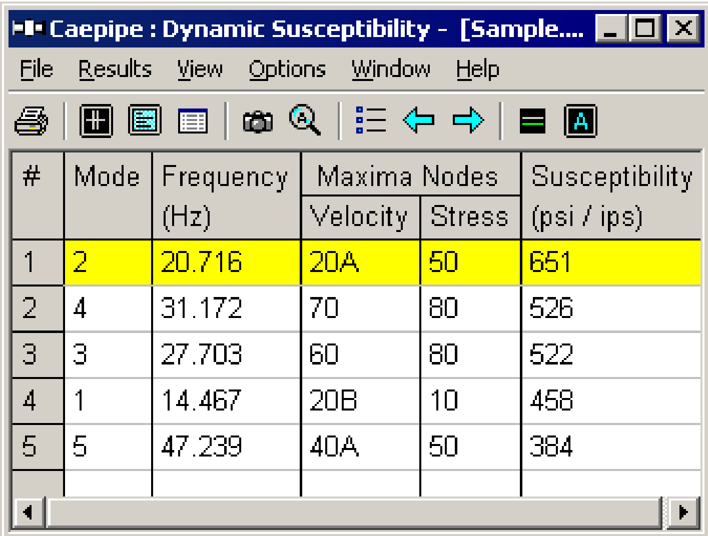
Caepipe : Frequencies - [S.res (D:\Manual\DynamicSusc...

File Results View Options Window Help

#	Frequency (Hz)	Period (second)	Participation factors			Modal mass / Total mass		
			X	Y	Z	X	Y	Z
1	14.5'2	0.0689	0.0232	-1.7571	-0.7834	0.0001	0.5112	0.1016
2	20.779	0.0481	-0.0004	0.5862	-1.9915	0.0000	0.0569	0.6567
3	27.750	0.0360	0.0826	0.4566	0.0823	0.0011	0.0345	0.0011
4	31.229	0.0320	-0.1751	-0.0063	0.110'	0.0051	0.0000	0.0020
5	47.406	0.0211	-1.0989	-0.1355	0.0559	0.2000	0.0030	0.0005
6	52.422	0.0191	0.1787	-0.3770	0.3007	0.0053	0.0235	0.0150
7	129.'69	0.0077	-0.0446	-0.1827	0.0125	0.0003	0.0055	0.0000
8	132.987	0.0075	0.1195	0.0090	-0.1265	0.0024	0.0000	0.0026
9	138.665	0.0072	1.4714	-0.0020	0.0228	0.3585	0.0000	0.0001
10	163.986	0.0061	-0.0379	-0.3694	0.1358	0.0002	0.0226	0.0031
11	173.053	0.0058	0.4440	-0.2272	0.1155	0.0326	0.0085	0.0022
12	191.852	0.0052	0.2435	0.1143	-0.0572	0.0098	0.0022	0.0005

The frequencies range from mode 1 at 14.5 Hz to mode 12 at 192 Hz. In two instances, very similar horizontal and vertical modes appear in pairs, i.e. modes 3 & 4 and 7 & 8.

The relevant features of this system can be readily identified and understood, by reference to the dynamic-susceptibility table and the animated graphic display of mode shape. Results will be considered here in the order of decreasing susceptibility.

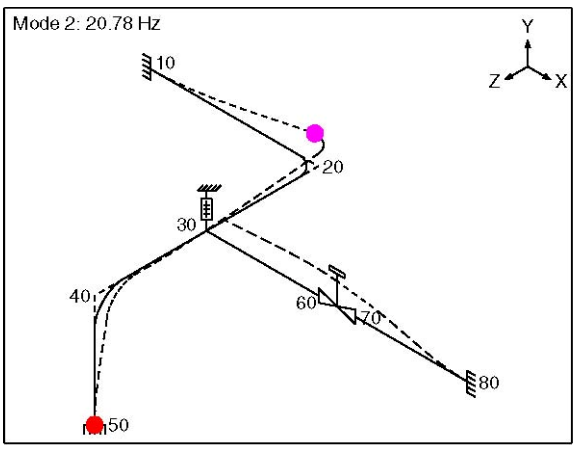

Caepipe : Dynamic Susceptibility - [Sample....

File Results View Options Window Help

#	Mode	Frequency (Hz)	Maxima Nodes Velocity	Maxima Nodes Stress	Susceptibility (psi / ips)
1	2	20.716	20A	50	651
2	4	31.172	70	80	526
3	3	27.703	60	80	522
4	1	14.467	20B	10	458
5	5	47.239	40A	50	384

5.1 Axial movement of long pipe run (large added mass in motion)

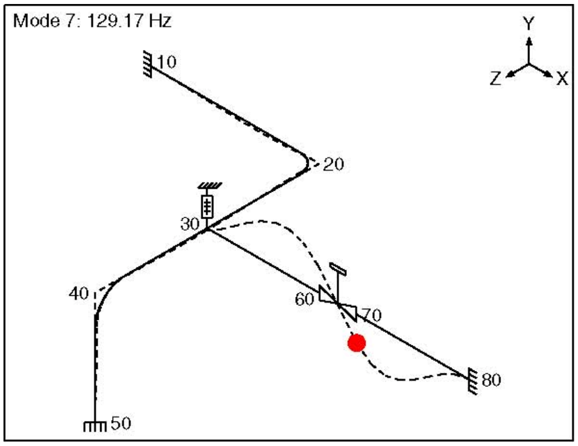
From the dynamic susceptibility table, the top of the list is mode 2 at 20.8 Hz, having a dynamic susceptibility of 651 psi / ips.

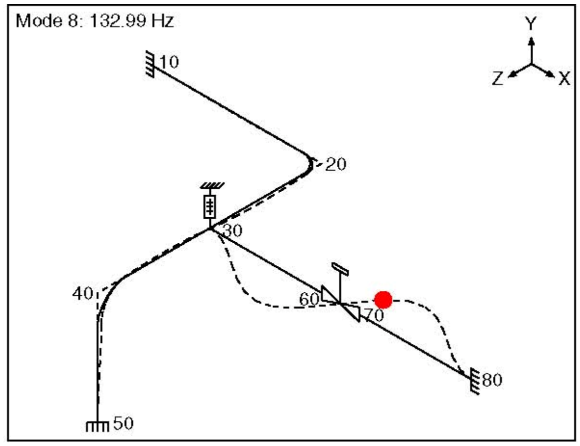


From the animated graphic display, note that the maximum dynamic bending stresses are at the anchored point, node 50. Note also that the dominant motion is a "Z" motion of the straight run between nodes 20 and 40 (i.e., in effect an axial motion of that run as a rigid body). The designer's interpretation here is that the vertical rise from node 50 to node 40 is effectively a cantilevered beam with an effective large added mass at the tip; that feature of layout accounts for the high susceptibility.

5.2 Effects associated with the valve (local rigidity to bending, and added mass)

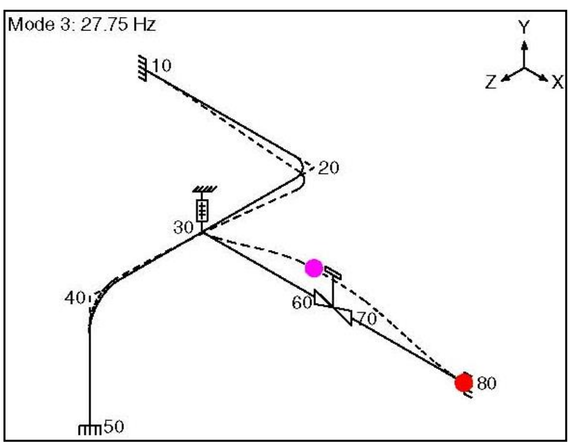
The next-highest values of susceptibility are for the two pairs of modes, modes 7 & 8 at 129 and 133 Hz, and modes 3 & 4 at 27.7 and 31.2 Hz. As will be shown here, these are associated with effects of the valve,

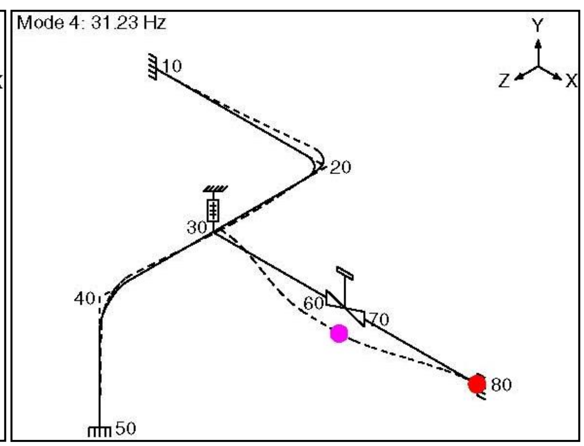




The susceptibility for modes 7 & 8, respectively 594 and 589 psi / ips, is attributable to the rigidity of the valve element within an otherwise flexible pipe run. This can be seen from a close look at the animated graphic. Notice that these relatively high frequency modes feature a reversal of bending curvature along the run between nodes 30 and 80. Notice also that there is a stronger localized curvature on approach to the valve body. The designer's interpretation here is that, since there cannot be any curvature of the rigid valve itself, there must be a more concentrated curvature of the adjacent pipe.

The dynamic susceptibility of modes 3 & 4, respectively 522 and 526 psi / ips, is associated with the more straightforward "concentrated mass" effect of the valve.

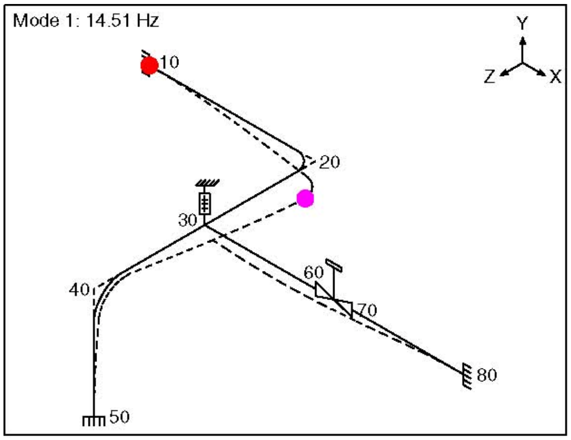


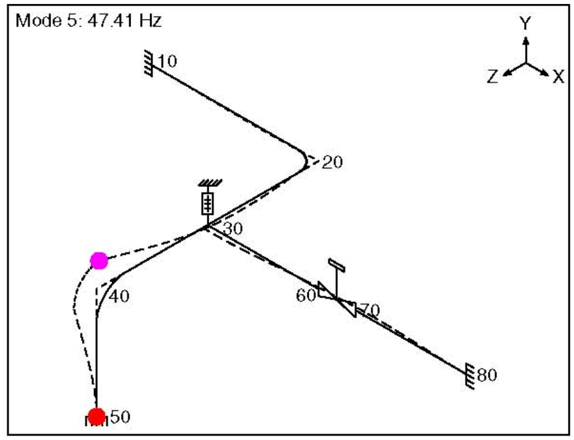


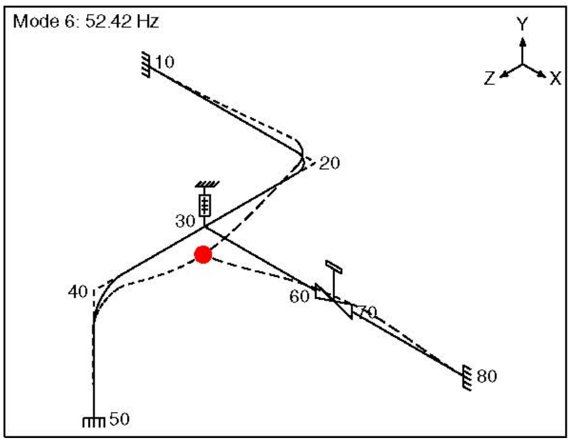
From the animated graphic, these modes feature a large amplitude vibration at the valve. The kinetic energy of this added mass must be stored as strain energy in the flexing (i.e., spring) element, resulting in elevated dynamic stresses.

5.3 Beam modes with "moderate" added mass effects of adjacent spans

Modes 1,5 and 6, with frequencies of 14.5, 47.4 and 52.4 Hz, show progressively decreasing "intermediate to low" values of susceptibility at respectively 458, 384 and 339 psi / ips.



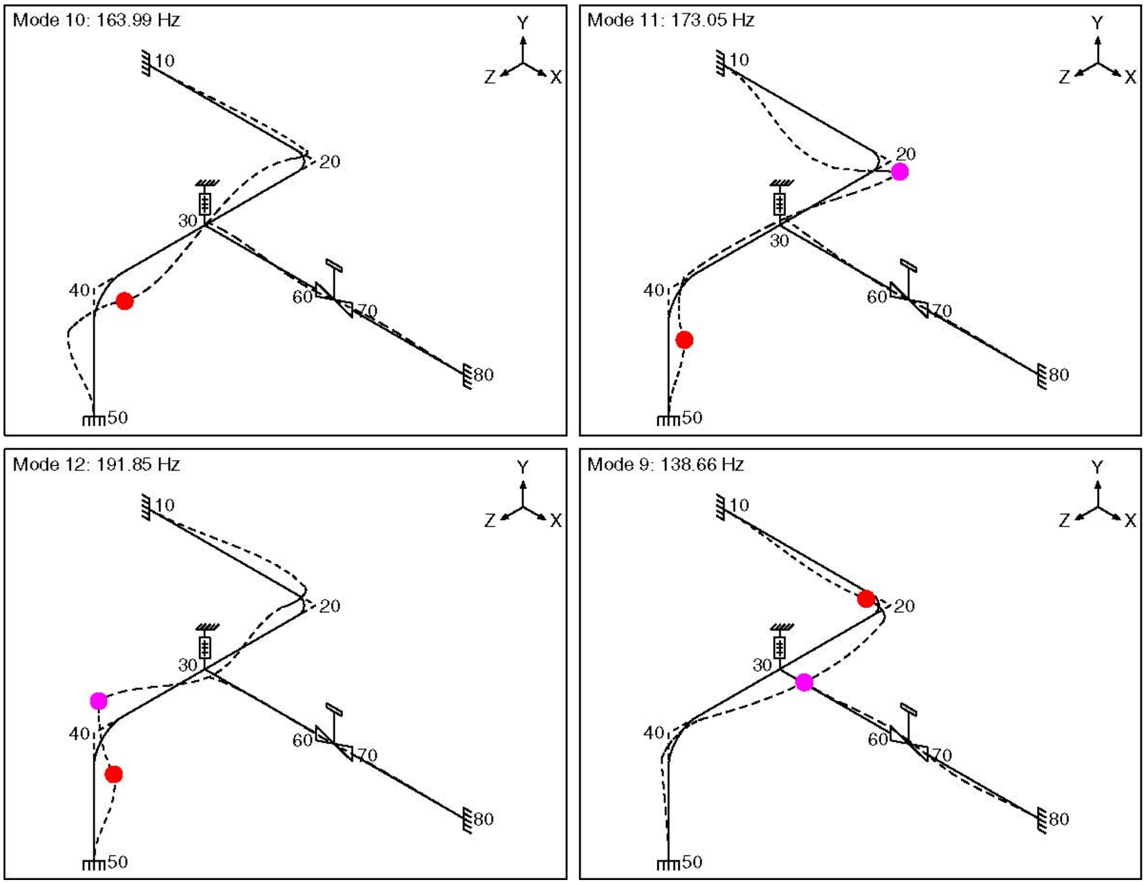




Reference to the animated graphics shows that these modes involve predominantly *transverse* vibration (as contrasted with the prominent *axial* movement of mode 2) and involve *little participation at the valve* (which accounted for the elevated susceptibility of modes7 & 8 and 3 & 4). Notice that these modes, 1, 5, and 6, involve varying degrees of the influence of effective added mass of adjacent spans, and of length of the cantilevered span contributing most to stiffness.

5.4 Modes approaching the "simple-beam baseline" behavior

Modes 10, 11 and 12 have significantly higher frequencies, 164 to 192 Hz, and correspondingly short wavelengths. Consequently, the vibration pattern tends to be transverse beam vibration "within the span," with little or no effect from connected spans or the valve. For these modes, the susceptibility ratios range from 256 to 272 psi / ips. These values are approaching the baseline values for uncomplicated mode shapes of the pipe section and pipe contents of this system.



NOTE: Mode 9, at 138 Hz, is clearly an exception, with a susceptibility of only 104 psi / ips, well below the baseline level. From the animated display, it can be seen that this is not really a "bending" mode; rather, the spring effect for this mode is an axial stretching of the run between nodes 80 and 30. Consequently, the bending stresses are low, as reflected in the abnormal susceptibility ratio. In effect, this mode lies outside the intended application of the dynamic susceptibility approach. Notice however, that the low susceptibility ratio has in effect "flagged" this mode as "not a bending mode"; that in itself provides the designer additional insight into system characteristics and behavior.

5.5 Summary Comment

As per paragraphs 5.1 to 5.4, the dynamic susceptibility method has incisively identified the key features of the Sample model, with respect to potentially large dynamic stresses. This of course is a relatively simple system. An experienced designer, with some appreciation of dynamics, might view the results as obvious. However, the method will do the same job, automatically and directly, on any larger or more complex system for which nothing is obvious!

6 Summary of "Dynamic Susceptibility" Analytical Capability

The stress / velocity method, implemented in CAEPIPE as the "Dynamic Susceptibility" feature, provides quantified insights into the stress versus vibration characteristics of the system layout per se.

In particular, the dynamic susceptibility table identifies specific modes that are susceptible to large dynamic stresses for a given level of vibration. The larger the stress / velocity ratio, the stronger the indication that some particular feature of layout, mass distribution, supports, stress raisers, etc., is causing susceptibility to large dynamic stresses.

The animated mode-shape display identifies, by the color-spot-markers, the locations of the respective maxima in dynamic stress and vibration velocity. Review of these animated plots will reveal the offending pattern of motion, and provide immediate insight into what features of the system are responsible for the large dynamic stresses.

Finally, the "dynamic stresses" table provides the distribution of dynamic stresses around the system, i.e., in effect, the mode shape of dynamic stresses to go along with the conventional mode shape of vibration. This information allows identification of other parts of the system, if any, with dynamic stresses comparable to the identified maximum.

7 Suggested Applications and Associated Benefits

7.1 At the Design Stage

At the design stage, the dynamic susceptibility feature allows the designer to quickly determine whether the system may be susceptible to very large dynamic stresses. This could be a broad look at all frequencies, or *could be focused on particular frequencies where excitation is likely to occur*. On identifying high susceptibility, the designer can then make changes to improve the design. It is important to note that this method is based upon the dynamic-stress versus vibration-velocity characteristics of the system per se. There is no need to specify a forcing function and perform a response calculation and stress / fatigue analysis. However, *where such analysis is a requirement,* the dynamic susceptibility module can assist the designer to *achieve a system layout that will meet the requirements and criteria*.

7.2 Commissioning, Acceptance Testing

The dynamic susceptibility feature can also contribute to planning acceptance testing and associated measurements where these are undertaken whether by formal requirement or by choice. Locations for measurement of vibration or dynamic strain can be selected based upon knowing the locations of the maxima and the distribution of vibration and dynamic stress. Reference to the dynamic susceptibility results can *help assure that the modes of most potential concern are well covered by the minimum set of practically-achievable measurements*. Furthermore, mode-specific acceptance criteria can be readily established to *avoid the restrictions of generally over-conservative guideline type criteria, while providing assurance that any highly-susceptible situations are identified and addressed*.

7.3 Troubleshooting and Correction

As mentioned earlier, when vibration and/or fatigue problems are recognized at start up or early operation, there is typically an ad hoc program of observation, measurement, assessment, diagnosis and correction. It is not uncommon for there to be some uncertainty about what to measure and what is acceptable. The dynamic susceptibility module can contribute very effectively in these situations.

Normally, the overall symptoms, approximate frequency and pattern of vibration are known to some extent from observation and/or a few measurements. After modeling the system, and obtaining the dynamic susceptibility results, the subsequent steps can be *highly focused on specific frequencies and locations,* the optimum measurements, and system-specific acceptance criteria.

Equally or more importantly, the *proposed solution options can be modeled and evaluated* to make sure they will achieve the required improvement.

7.4 General

The dynamic susceptibility module *does not apply directly to meeting code or other formal stress analysis requirements*. However, it is an incisive analytical tool to help the designer understand the stress / vibration relationship, assess the situation and to decide how to modify the design if necessary. It can be used for design, planning acceptance tests, and troubleshooting and correction.

8 Information for Reference

The Stress /Velocity method for screening piping system modes was developed and brought to the attention of SST Systems by Dr. R.T Hartlen of Plant Equipment Dynamics Inc.

The background material provided here is intended to provide only a concise summary of the underlying fundamentals, the universality for idealized systems, and the expected detail-dependent variations for real systems. The stress / velocity method, although not yet widely known and applied, is fundamentally theoretically sound. However, complete theoretical rigor is beyond the scope of this note.

For users who may wish to independently examine and validate the underlying theoretical fundamentals, a few key references are provided. References 1, 2 and 3 deal with fundamentals. References 5 and 6 deal with application to piping. The CEA research projects reported in References 3 and 4 were initiated and guided by Dr. Hartlen.

References

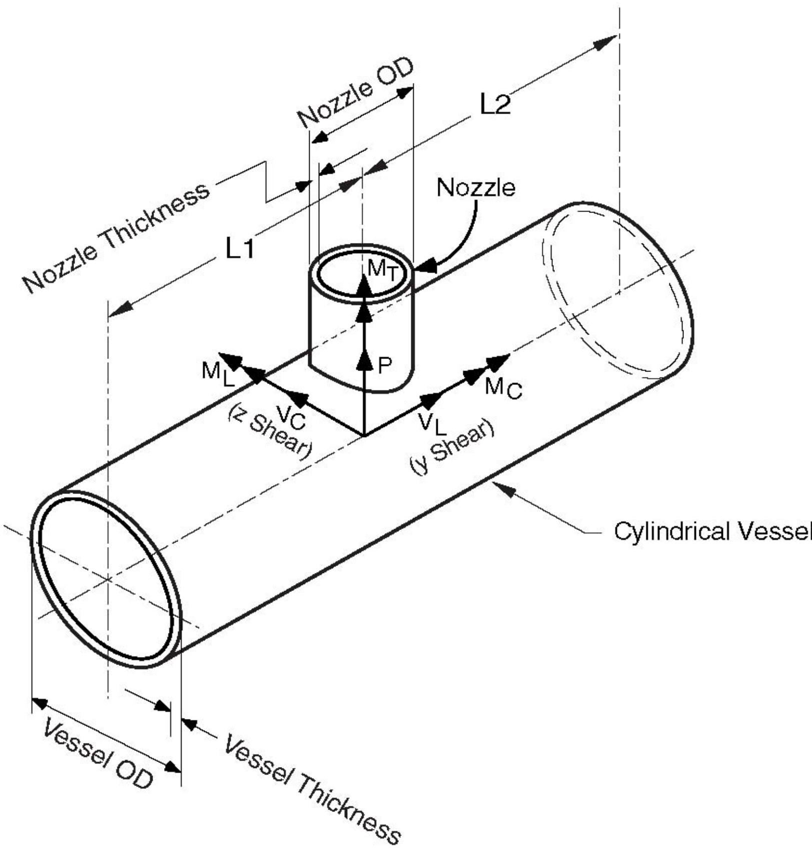
1. F.V. Hunt, Stress and Strain Limits on the Attainable Velocity in Mechanical Systems, JASA, 32(9) 1123-1128, 1960
2. E.E. Ungar, Maximum Stresses in Beams and Plates Vibrating at Resonance, ASME Journal of Engineering for Industry, v84, n1, pp149-155, 1962
3. R. Elmaraghy et al, Correlation of Vibratory Stress, Velocity and Sound, Canadian Electrical Association Project, G197, Feb 1982
4. J.D. Tulk, Correlation Between Dynamic Stress and Vibration Velocity in Complex Piping Systems, Canadian Electrical Association Project G521, March 1988
5. Michael P. Norton, Acoustically Induced Structural Vibration and Fatigue - A Review,
6. Third International Congress on Air-and Structure-borne Sound and Vibration, June 1994, Montreal, Canada

Annexure II

Nozzle Stiffness Calculations

Nozzle Stiffness Calculations (WRC 297)

Six stiffnesses are shown below at the nozzle-vessel interface — three are calculated and the other three are assumed rigid.



The coordinate system is as shown in the figure. The six components of the forces and moments at the nozzle-vessel interface are:

P = Radial load M_C = Circumferential moment
V_C = Circumferential load M_T = Torsional moment
V_L = Longitudinal load M_L = Longitudinal moment

Of the six components of stiffnesses, only three stiffnesses, axial (K_x), circumferential (K_{yy}), and longitudinal (K_{zz}), are calculated. The remaining three are assumed to be rigid.

Several graphs are given at the end of this annexure. The stiffness coefficients are obtained by interpolating logarithmically from these graphs.

The first two, Figures D-1 and D-2, are used to calculate nozzle stiffness coefficients for Nozzles on cylindrical vessels. Figure D-1 is used to calculate the axial stiffness coefficient and Figure D-2 is used to calculate circumferential and longitudinal stiffness coefficients.

Nomenclature

D = mean diameter of vessel
d = outside diameter of nozzle
T = thickness of vessel
t = thickness of nozzle
λ = $(d/D)\sqrt{D/T}$
Λ = $L/\sqrt{DT}$
L = unsupported length of cylinder
= $8L_1L_2/\left(\sqrt{L_1}+\sqrt{L_2}\right)^2$
$L1$ = distance from nozzle center line to vessel end
$L2$ = distance from nozzle center line to vessel end
E = modulus of elasticity of vessel material

Axial Stiffness(K_x)

$$K_x = \alpha \times \frac{4.95ET^2}{D\sqrt{\Lambda}} \tag{1}$$

where
α = stiffness coefficient read from Figure D-1

Circumferential Stiffness(K_{yy})

$$K_{yy} = \beta \times ET^3 \tag{2}$$

where
β =1stiffness coefficient read from Figure D-2.
The bottom three curves in Figure D-2, marked Circumferential moment $M_C$are used to findβ.

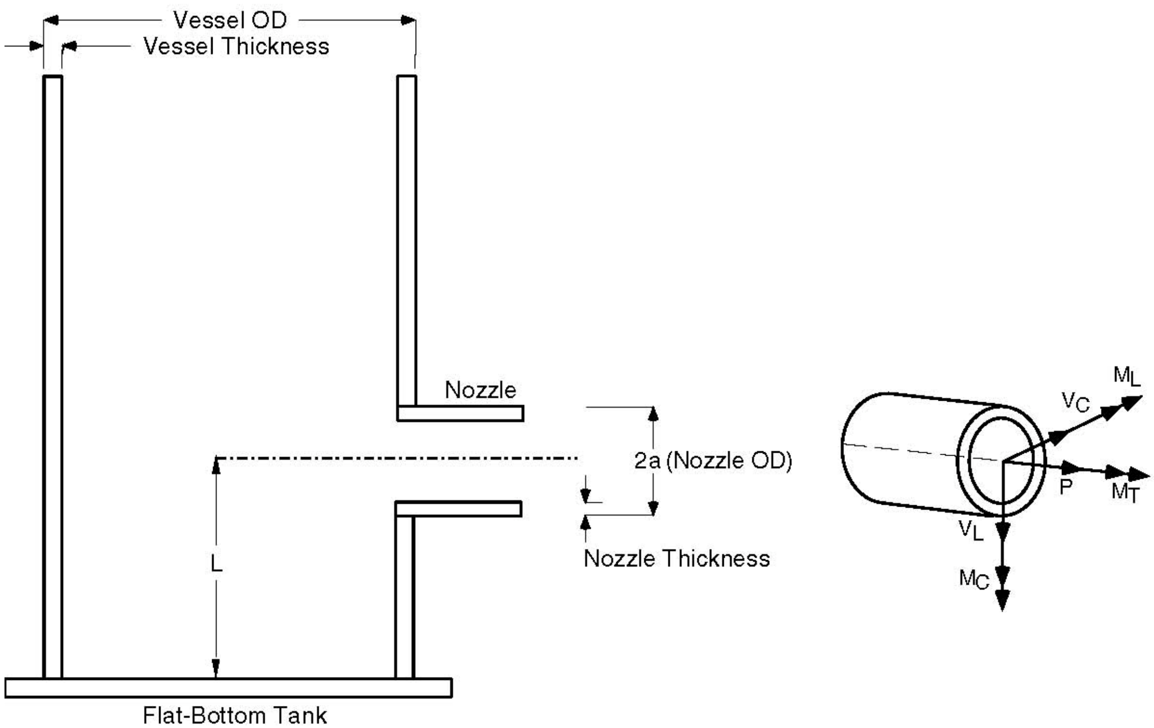
Longitudinal Stiffness(K_{zz})

$$K_{zz} = \gamma \times ET^3 \tag{3}$$

where
γ = stiffness coefficient read from Figure D-2.
The top two curves in Figure D-2, marked Longitudinal moment $M_L$are used to findγ.

Calculation of Nozzle stiffnesses for Nozzles on Flat-bottom tanks

This procedure is similar to the previous one.



As before, only three stiffnesses are calculated as the other three are assumed to be rigid. The ones that are calculated are axial(K_x), circumferential(K_{yy}), and longitudinal (K_{zz}).

For Nozzles on flat-bottom tanks, twelve graphs are given at the end of this annexure, Figures D-3 through D-14. Six are for "with reinforcing pad (on vessel)" with the other six for no reinforcing pad on the vessel. The stiffness coefficients are obtained using the appropriate graph.

Nomenclature

R = Mean radius of vessel
t = thickness of vessel
$2a$ = outside diameter of nozzle

Axial Stiffness(K_x)

$$K_x = K_R \times E \times (2a) \tag{4}$$

where
K_R = axial stiffness coefficient.

Circumferential Stiffness$\left(K_{yy}\right)$

$$K_{yy} = K_C \times E \times (2a)^3 \tag{5}$$

where
K_C = circumferential stiffness coefficient.

Longitudinal Stiffness(K_{zz})

$$K_{zz} = K_L \times E \times (2a)^3 \tag{6}$$

where
K_L =longitudinal stiffness coefficient.

The graphs for stiffness coefficients follow:

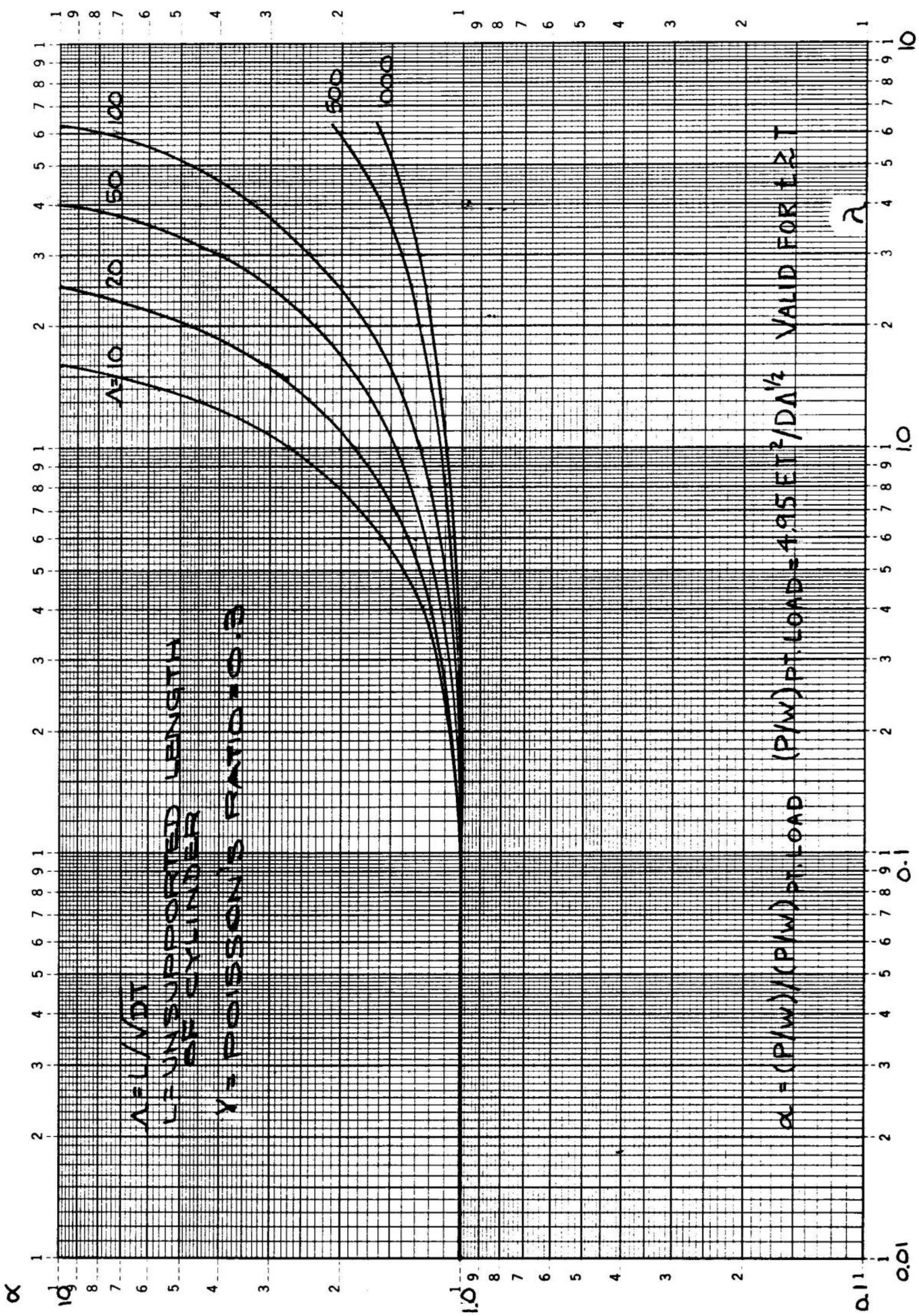


Figure D-1: Stiffness coefficient for axial load on nozzle

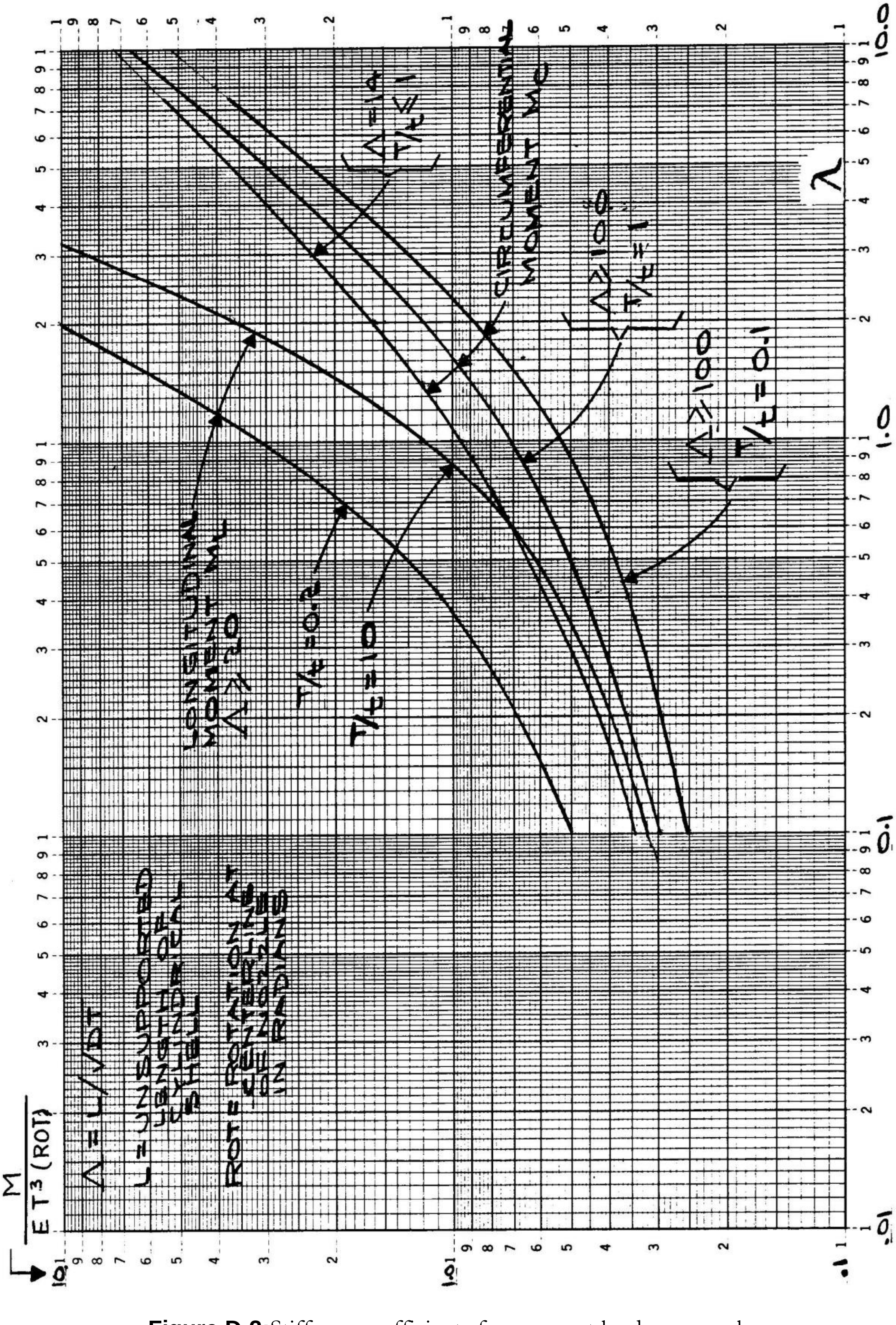


Figure D-2: Stiffness coefficients for moment loads on nozzle

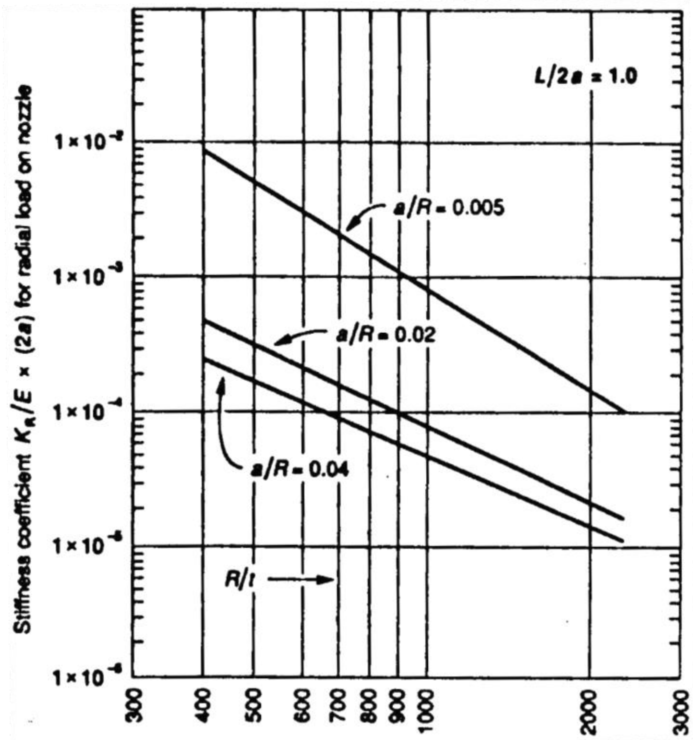


Figure D-3: Stiffness coefficient for axial load (with reinforcing pad)(L/2a = 1.0)

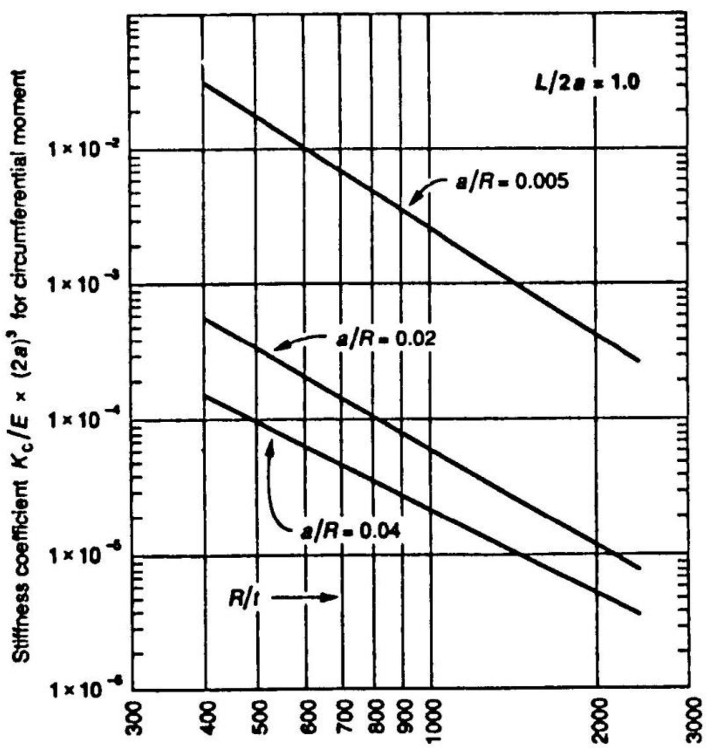


Figure D-4: Stiffness coefficient for circumferential moment (with reinforcing pad)(L/2a=1.0)

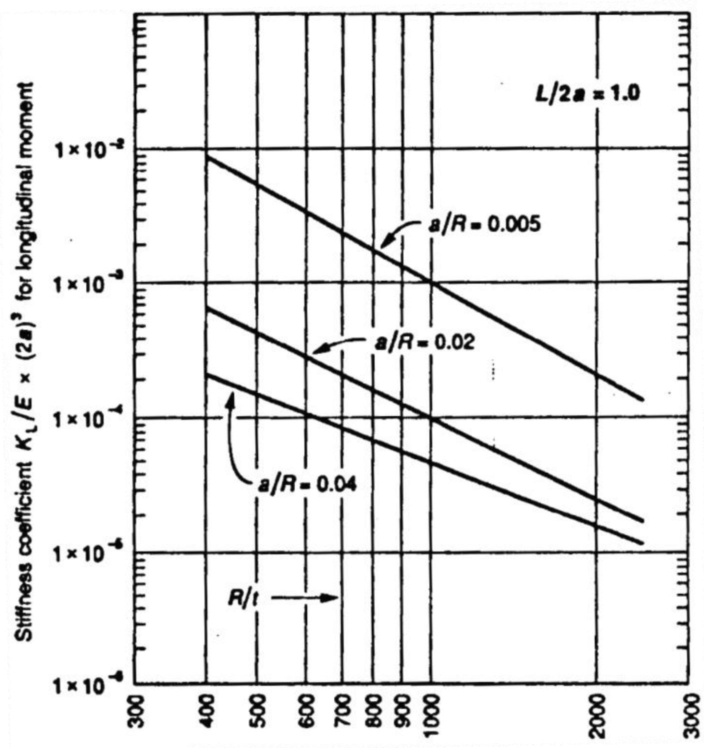


Figure D-5: Stiffness coefficient for longitudinal moment (with reinforcing pad)(L/2a = 1.0)

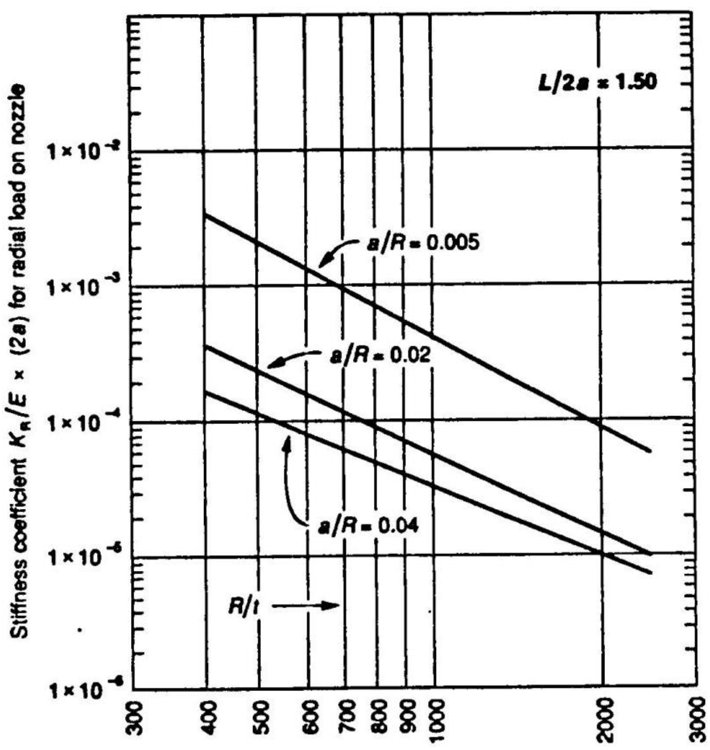


Figure D-6: Stiffness coefficient for axial load (with reinforcing pad)(L/2a = 1.5)

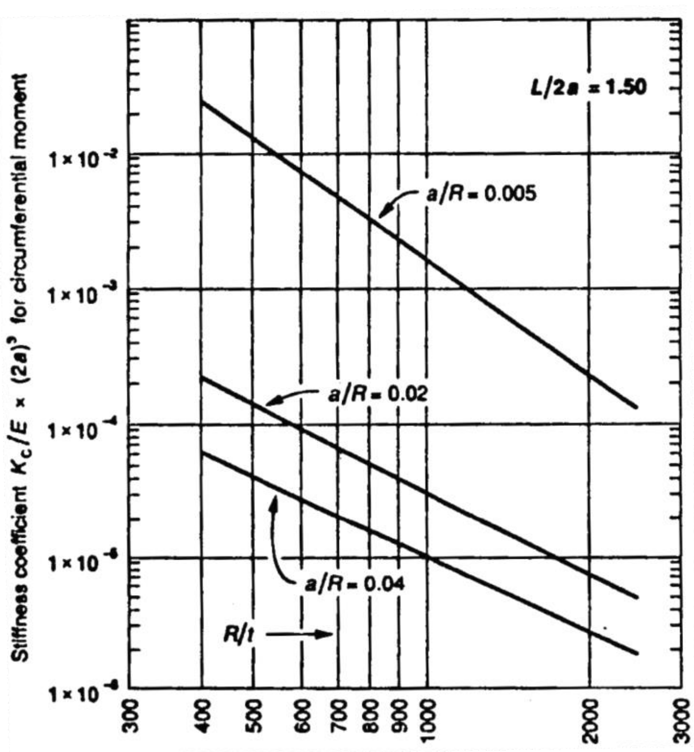


Figure D-7: Stiffness coefficient for circumferential moment (with reinforcing pad) (L/2a = 1.5)

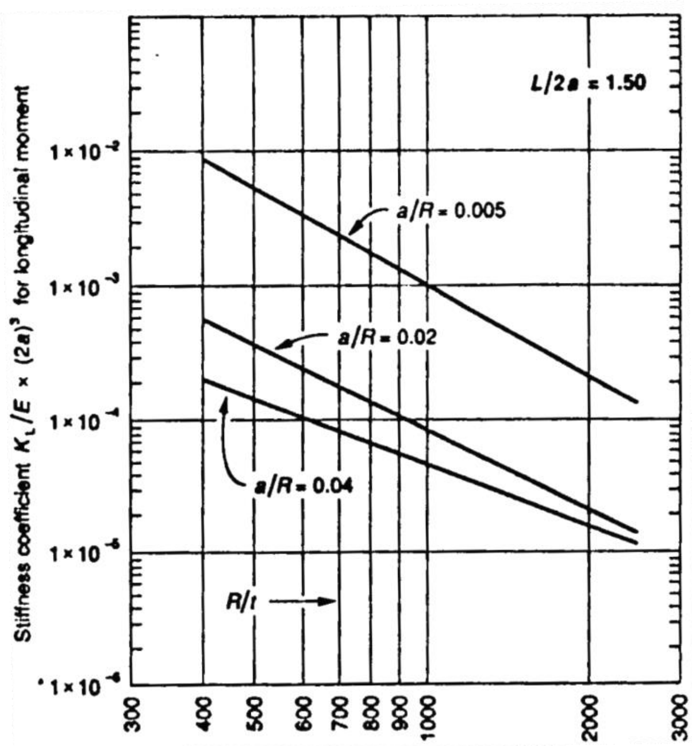


Figure D-8: Stiffness coefficient for longitudinal moment (with reinforcing pad)(L/2a=1.5)

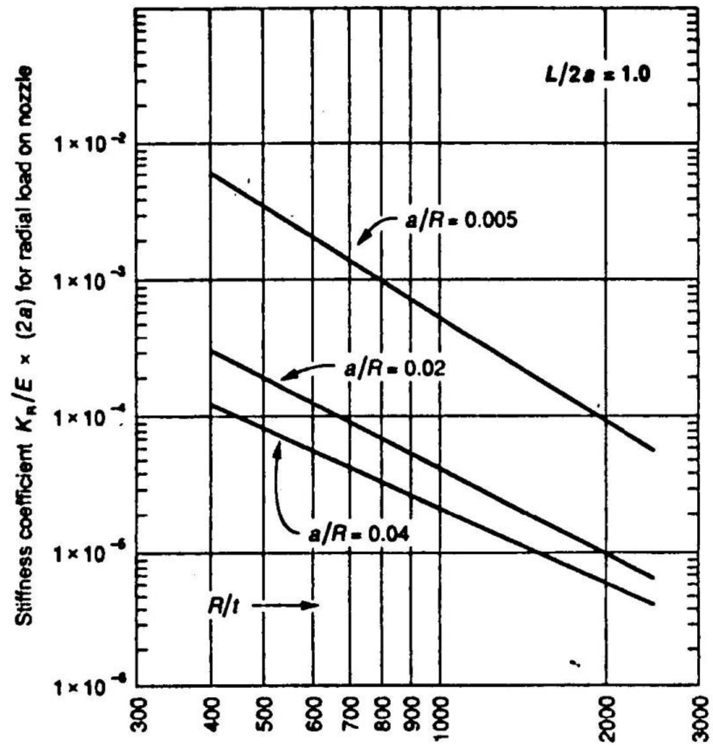


Figure D-9: Stiffness coefficient for axial load (no reinforcing pad)(L/2a = 1.0)

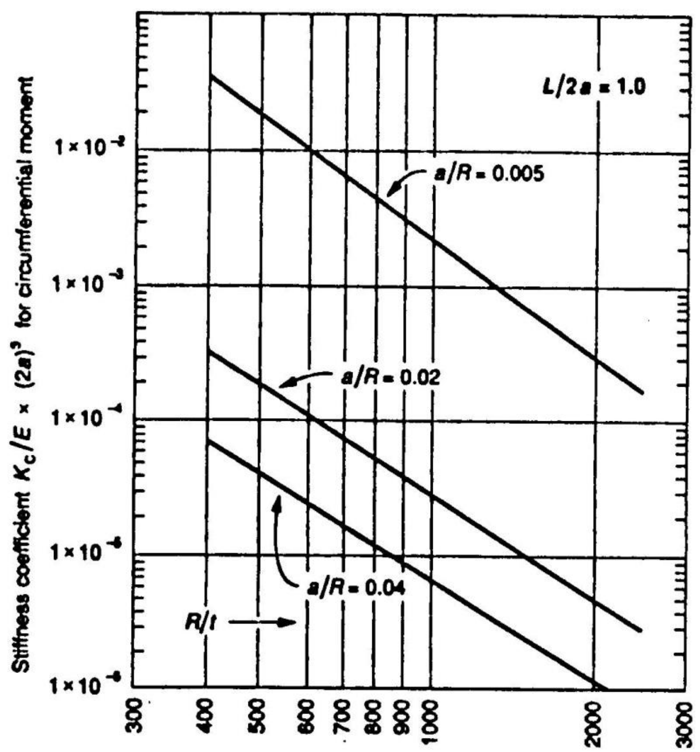


Figure D-10: Stiffness coefficient for circumferential moment (no reinforcing pad)(L/2a= 1.0)

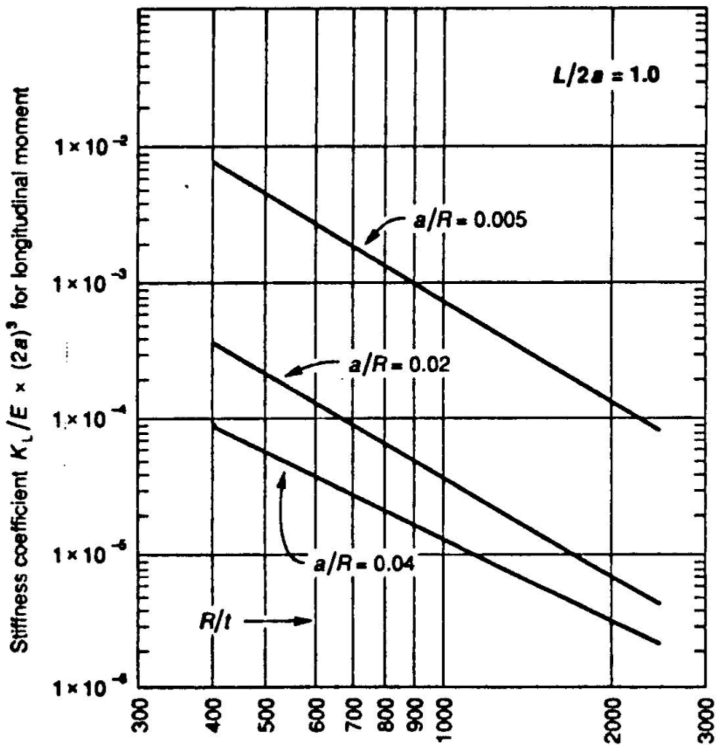


Figure D-11: Stiffness coefficient for longitudinal moment (no reinforcing pad)(L/2a = 1.0)

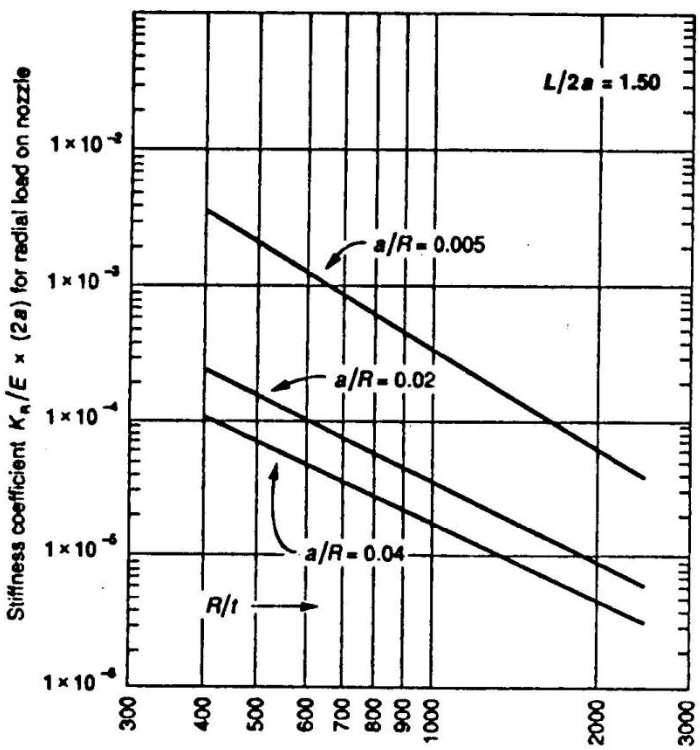


Figure D-12: Stiffness coefficient for axial load (no reinforcing pad)(L/2a = 1.5)

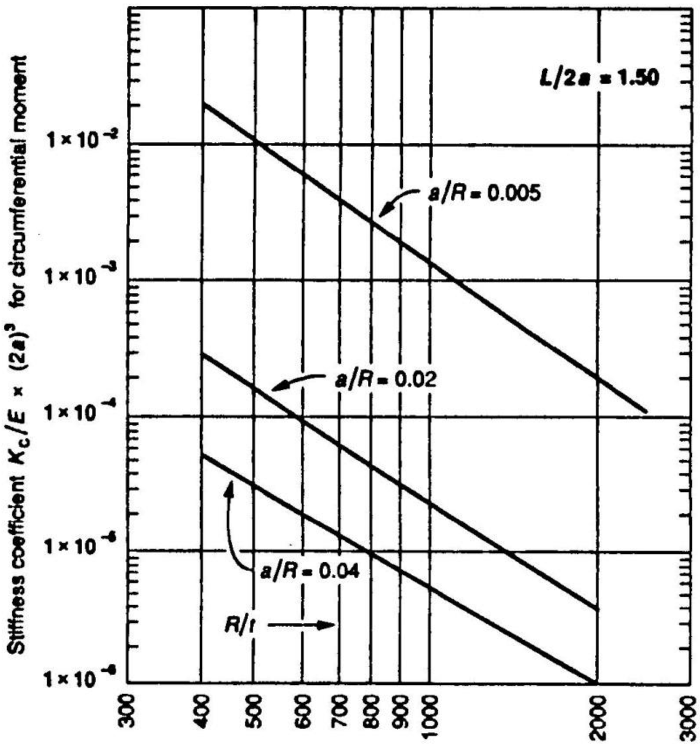


Figure D-13: Stiffness coefficient for circumferential moment (no reinforcing pad) (L/2a = 1.5)

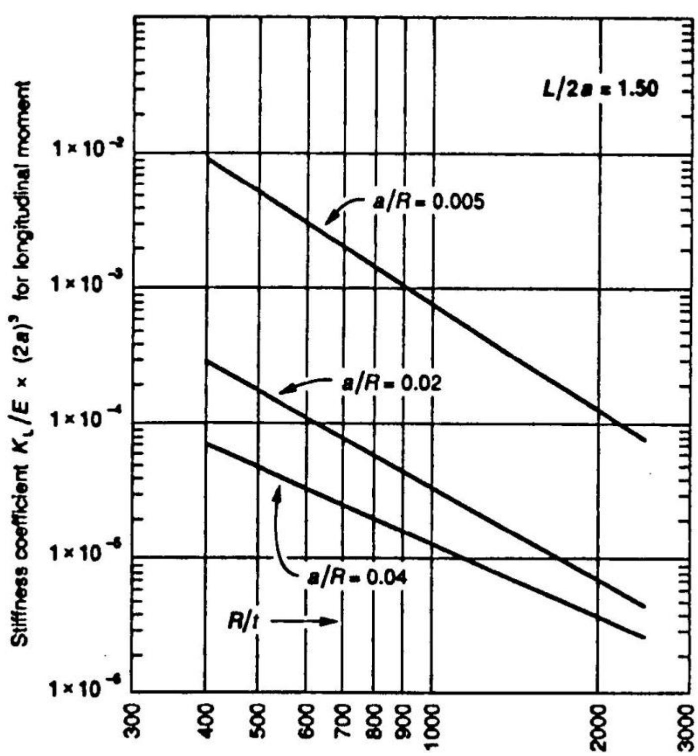


Figure D-14: Stiffness coefficient for longitudinal moment (no reinforcing pad)(L/2a=1.5)

Index

A

absolute sum, 82, 83
acceleration, 205, 320, 321
 spectral, 205
 vector, 78, 83, 84, 205
additional weight, 19, 172, 175, 297, 346, 347
anchor, 3
 displacements, 4
 releases for hanger selection, 4
 settlement, 5
 stiffness, 3
angle
 valve, 350
animate
 mode shape, 85
ANSI pipe sizes, 27, 302
API, 69, 256
API 610 report, 256, 263, 265
API 617, 256
API 617 report, 69
API 650, 221, 225
axes
 elastic element, 87
 local beam, 13
 local coordinate system, 176
axial force, 111
axial stiffness, 220

B

ball joint, 9
 friction, 217
 rotation limit, 10
 stiffness, 9
beam, 12
 additional weight, 19
 AISC library, 16
 beta angle, 13
 end releases, 13
 load, 18
 local coordinate system, 20
 material, 13
 orientation, 20
 section, 14
bellows, 24
 stiffness, 24
 tie rods, 328
bend, 26
 examples, 28
  180° bend, 31
  45° bend, 30
  90° bend, 29
  base supported, 39
  flanged, 32
  reducing, 35
  supported by a hanger, 37
 flexibility factor, 27
 intermediate nodes, 28
 long radius, 26, 27
 material, 27
 radius, 27
 short radius, 26, 27
 SIF, 28, 35
 tangent intersection point, 26
 thickness, 27
bottomed-out springs, 323
branch connection
 nonstandard, 345
branch line, 48, 135
branch SIF, 47
buried piping, 50
 general procedure, 50
 ground level, 53
 nomenclature, 54

C

circumferential
 joint factor, 196, 198, 202
 stiffness, 222, 375, 376, 377, 378
closely spaced modes, 83
cold load, 144
cold spring, 66
comment, 68
compressor, 69
concentrated mass, 72
cone angle, 266, 267
connected node, 164, 299
 hanger, 143
 skewed restraint, 306
constant support, 73
 user defined hanger, 332
coordinates, 135, 216
core pipe, 155, 159, 185, 314
core properties, 158
corrosion allowance, 301
cut pipe, 66

D

data types, 74
density, 202
design
 hanger, 142
 pressure, 174
 temperature, 174
DIN pipe sizes, 27, 302
direction, 75
displacement vector, 78, 82
displacements
 anchor, 4
 nozzle, 222
dynamic analysis, 78
 closely spaced modes, 83
 effective modal mass, 81

friction, 218, 220
modal analysis, 78
modal equations, 79
orthogonality, 79
participation factors, 80, 81
response spectrum, 82
support motion, 80
time history, 83
dynamic susceptibility, 85
environment variable, 85
method, 363

E

effective modal mass, 81
eigenvalues, 78
eigenvector, 78
normalized, 205
elastic element, 87
stiffness, 87
elbow, 26
element forces and moments
in local coordinate system, 178
element stresses
FRP, 103
environment variable
CPITER, 220
dynamic susceptibility, 85
HARTLEN, 85
exit, 220
expansion joints, 24, 88
bellows, 88, 90, 93
tied bellows, 88, 90, 93

F

fiber reinforced plastic piping (FRP), 103, 104
flexibility factor, 104
moduli, 104
SIF, 104
siffness matrix, 104
flange, 107
allowable pressure, 108
equivalent pressure, 108
flange and bolt stresses, 115
gasket diameter, 108
library, 110
rating, 108
flange and bolt stresses, 115
flange module, 115
flexibility factor
bend, 27
FRP, 104
miter bend, 209
force, 128
force spectrum, 129
convert time function, 132
load, 133
read from a text file, 132
frequencies
closely spaced, 83
frequency
circular, 81, 205
natural, 364
friction, 218
ball joint, 220
guide, 219
hinge joint, 219
in dynamic analysis, 218, 220
limit stop, 218
slip joint, 219
friction coefficient, 140, 164, 217, 323
friction force, 140, 162, 164, 218, 310
friction torque, 9, 148, 218, 219, 310
From node, 135

G

gasket diameter, 108
generic support, 137
global
coordinate system, 176
origin, 53
guide
friction, 219
friction coefficient, 140
gap, 141
local coordinate system, 141
stiffness, 141

H

hanger, 142
below, 143
catalog, 144, 322
cold load, 143
connected node, 143
constant support, 73
design procedure, 144
hot load, 144
load variation, 144
manufacturer, 144
number of, 143
report, 299
rod, 299
short range, 143
stiffness, 143
to be designed, 142
travel, 144, 218, 323
type, 143
user, 332
harmonic analysis, 84
hinge joint, 147
angular stiffness, 148
axis direction, 149
example, 150
friction, 219
friction torque, 148
rotational limit, 148
rotational stiffness, 148

Weight, 149
hydrotest
load, 153
load case, 154

I

input
new material, 198
insulation, 303
density, 303
thickness, 303
intermediate nodes, 28
internal nodes, 159

J

jacket end cap, 155
jacket pipe, 195
jacket properties, 158
jacketed bend, 157
internal nodes, 159
jacketed pipe, 156
internal nodes, 157
jacketed piping
jacketed bend, 157
jacketed reducer, 159
JIS pipe sizes, 27, 302
joint factor, 202
circumferential, 202
longitudinal, 198, 202

K

knuckles
reducer, 267

L

lateral tee, 49
left out force, 205
limit stop, 162
friction, 218
nonlinearities, 217
solution procedure, 217
lining, 303
load
additional weight, 175
Design pressure, 175
Design temperature, 175
pressure, 174
snow, 175
specific gravity, 175
static seismic-ANSI A58.1-1988, 320
static seismic-ASCE/SEI 7-16, 315
temperature, 174
wind, 175
load cases
cold spring, 66
force spectrum, 133
hydrotest, 154
loads
acceleration, 321
force spectrum, 130
local coordinate system, 176
element forces and moments, 178
for a beam, 20
for a bend, 178
for a guide, 141
for a pipe, 176
local forces, 178
location, 185
logarithmic interpolation, 82, 375
long radius, 26, 27
longitudinal
joint factor, 198, 202
pressure stress, 202
longitudinal stiffness, 222, 375, 376, 378

M

material
create library, 17, 202
define, 195
density, 202
description, 201
input, 198
name, 201
selectr from library, 200
type, 201
missing mass correction, 205
miter bend
closely spaced, 209
flexibility factor, 209
modeling procedure, 210
parameters, 209
widely spaced, 208, 209
modal analysis, 78
modal displacements, 79

N

node, 216
internal, 159
list coordinates, 216
specifying coordinates, 216
nominal diameter, 302
nonlinearities, 217
ball joint, 220
friction, 218
guide, 219
hinge joint, 219
limit stop, 218
misconvergence, 220
number of iterations, 220
slip joint, 219
types, 217
nonstandard
branch connection, 345
pipe, 302

nozzle, 221
axial stiffness, 222, 375, 376, 378
circumferential stiffness, 222, 375, 376, 378
coordinate system, 222, 375
displacements, 225
example
on a cylindrical vessel, 226, 227
on a flat-bottom storage tank, 228
longitudinal stiffness, 222, 375, 376, 378
on a cylindrical vessel, 221
reinforcing pad, 225
settlement, 226
stiffness calculations, 375
stiffness coefficients, 375
Nozzle Evaluation, 231
NRC Guide 1.92, 83
number of modes, 205
number of thermal loads, 19, 172

O

occasional load, 134

P

pad thickness, 47
participation factor, 80, 81, 205
period, 82
pipe skirt, 323
pipe slide assembly, 323
pipe slide/shoe assembly, 167
Poisson's ratio, 202
pressure
gauge, 174
negative, 174
pump, 256

R

reducer, 266
concentric, 266
example, 268
cone angle, 266, 267
delta, 268
eccentric, 266
example, 269
example
concentric, 268
eccentric, 269
jacketed, 271
knuckles, 267
SIF, 267
stresses, 268
weight, stiffness and stress calculation, 268
reducing tee, 48
releases for hanger selection, 3
relief valve, 350
report
flange, 113
response spectrum, 82, 226
restraint, 296
stiffness, 296
rigid body force, 205
rigid element, 297
stiffness, 297
weight, 297
rod hanger, 299
stiffness, 299
rotation limit
ball joint, 10
hinge joint, 148

S

section, 301
ANSI pipe sizes, 302
corrosion allowance, 302
DIN pipe sizes, 302
insulation, 303
JIS pipe sizes, 302
lining, 303
mill tolerance, 302
name, 302
nominal diameter, 302
nonstandard, 302
schedule, 302
section modulus, 302
settlement
anchor, 5
nozzle, 226
shear
area, 15
deflection, 15
shear deflection, 15
short radius, 26, 27
SIF
branch, 47
FRP, 104
reducer, 267
user, 345
weld, 351
sign conventions, 178
skewed restraint, 305
connected node, 307
direction, 306
example
sway brace, 307
rotational, 306
stiffness, 306
translational, 306
type, 306
slip joint, 310
friction, 219
friction force, 310
friction torque, 310
pressure thrust area, 310
snow load, 19, 172, 175
snubber, 313
stiffness, 313
specific gravity, 175

spider, 314
spring rate, 73, 144, 299, 332
SRSS, 82
stiffness
  anchor, 3
  ball joint, 10
  bellows, 24
  elastic element, 87
  guide, 141
  hanger, 143
  hinge joint, 148
  limit stop, 164
  nozzle, 221, 375
  reducer, 268
  restraint, 296
  rigid element, 297
  rod hanger, 299
  snubber, 313
  tie rod, 328
  user hanger, 332
  valve, 347
stiffness matrix, 78, 83, 84, 144, 217, 218
  FRP, 104
stresses
  FRP, 105
  occasional, 302
  reducer, 268
  sustained, 302
support motion, 80
supports
  anchor, 322
  generic, 137
  guide, 323
  hangers, 322
  limit stop, 323
  restraint, 322
  skewed restraint, 322
sway brace, 305, 307, 322
sway struts, 305, 322

T

tangent intersection point, 26, 178
tee
  SIF, 47
Tee, 325
tensile strength, 202
thermal loads, 4, 172, 226
tie rod, 328
  stiffness, 328
time history, 83
turbine, 329

U

u-bolt, 323
user hanger, 332
  constant support, 332
  load, 332
  spring rate, 332
  stiffness, 332
user SIF, 345
u-strap, 323

V

valve, 346
  additional weight, 347
  angle, 350
  calculation of moment of inertia, 350
  insulation weight X, 347
  length, 346
  library, 348
  relief, 350
  stiffness, 347
  thickness X, 347
  weight, 346
velocity vector, 83

W

weight multiplier, 350
weld, 351
wind, 352
wind load, 175
WRC 297, 221

Y

Y factor, 201

Z

zero length element, 9, 147