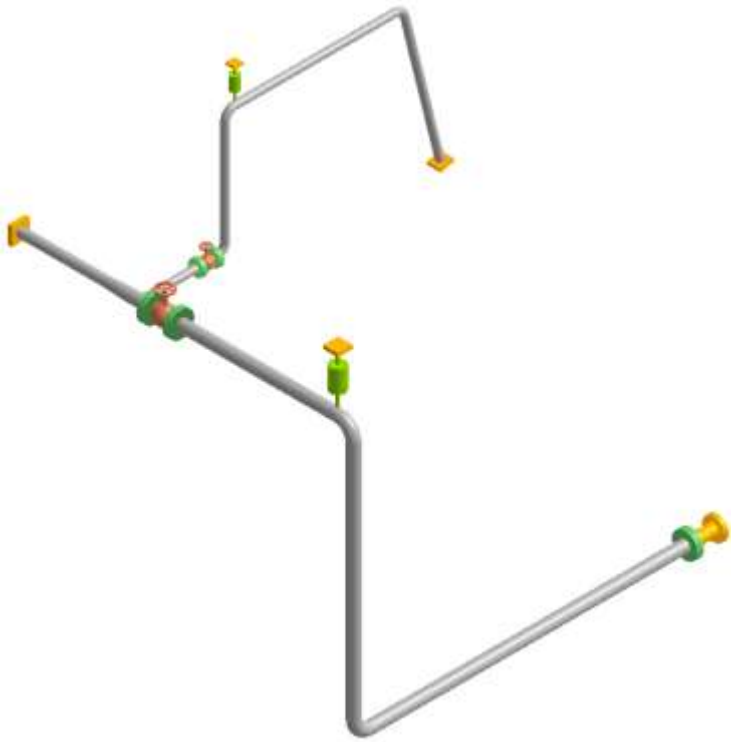
CAEPIPE™

Tutorial for Modeling and Results Review

Problem 2



SST SYSTEMS, INC.

The *FASTEST* Solutions for Piping Design and Analysis

CAEPIPE Tutorial, © 2026, SST Systems, Inc. All Rights Reserved.

Disclaimer

Please read the following carefully:

This tutorial has been developed and checked for correctness and accuracy by SST Systems, Inc. However, no warranty, expressed or implied, is made by the authors or by SST Systems, Inc., as to the accuracy, correctness and utilization of this tutorial. 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.

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

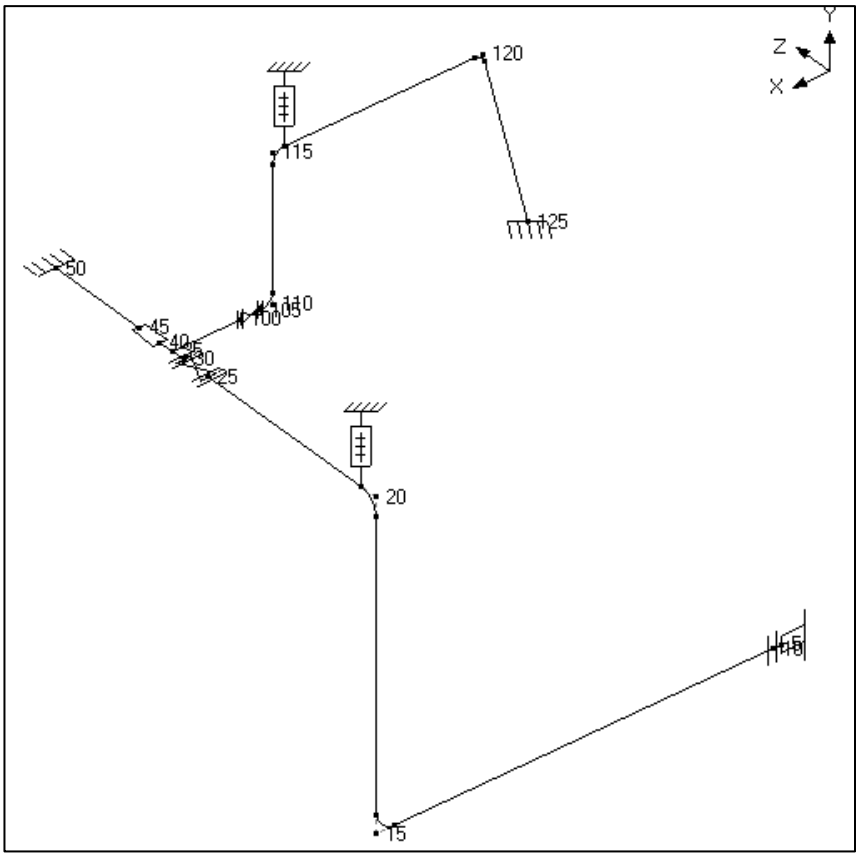
Tel: (408) 452 8111
Fax: (408) 452 8388

email: info@sstusa.com

www.sstusa.com

Modeling and Results Review – Problem 2

Let us model a slightly more advanced piping system now that you have familiarized yourself with the basic use of CAEPIPE via Tutorial 1. The details of the model (in SI units) are shown below:



You will learn how to:

1. Enter Title
2. Select Analysis options (piping code etc.)
3. Define Material, Section and Loads for the model
4. Input Model Layout (different loads for different segments)
5. Select Load Cases for Analysis
6. Analyze
7. View Results

Model Description

Details of the Layout, Material, Sections, Loads and Connection details are summarized for reference:

1. **Axes Chosen:** Global X = East, Global Y = Up and Global Z = South
2. **Piping Code:** ASME B31.1 (2020)
3. **Section Properties:**
   a. Main Line: 10" Schedule STD
   b. Branch Line: 6" Schedule STD

4. **Insulation throughout the Piping system**:
    a. **Density**: 176.2 kg/m3
    b. **Thickness**: 65 mm
5. **Material**: A 312 TP 316
6. **Temperature**:
    a. For Main Line and Branch Line up to Valve End Node 105:

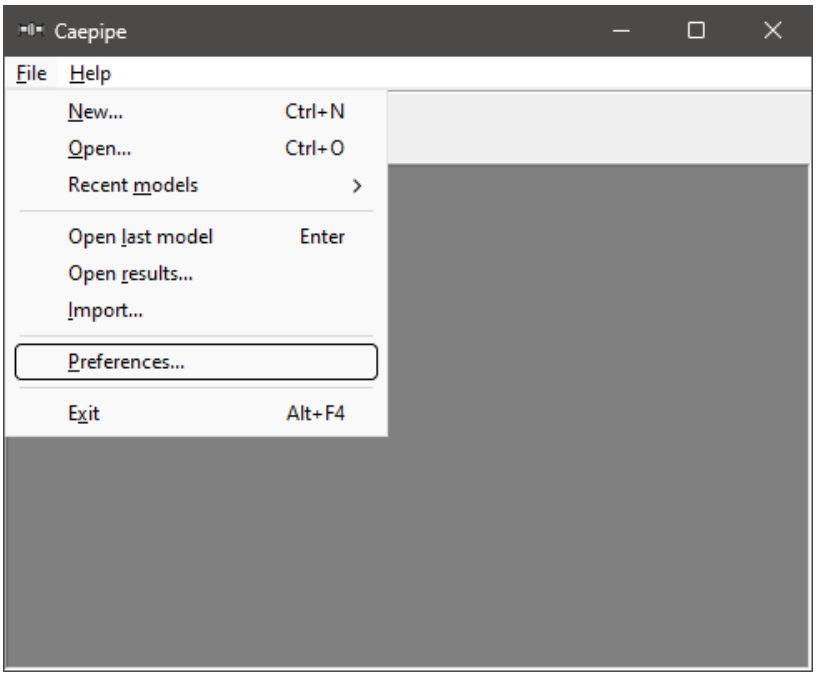
       Operating Temperature = 185 Deg. C and Peak Temperature = 230 Deg. C
    b. For Branch Line after Valve Node 105:

       Operating Temperature = 260 Deg. C and Peak Temperature = 300 Deg. C
7. **Pressure**:
    a. For Main Line and Branch Line up to Valve End Node 105:

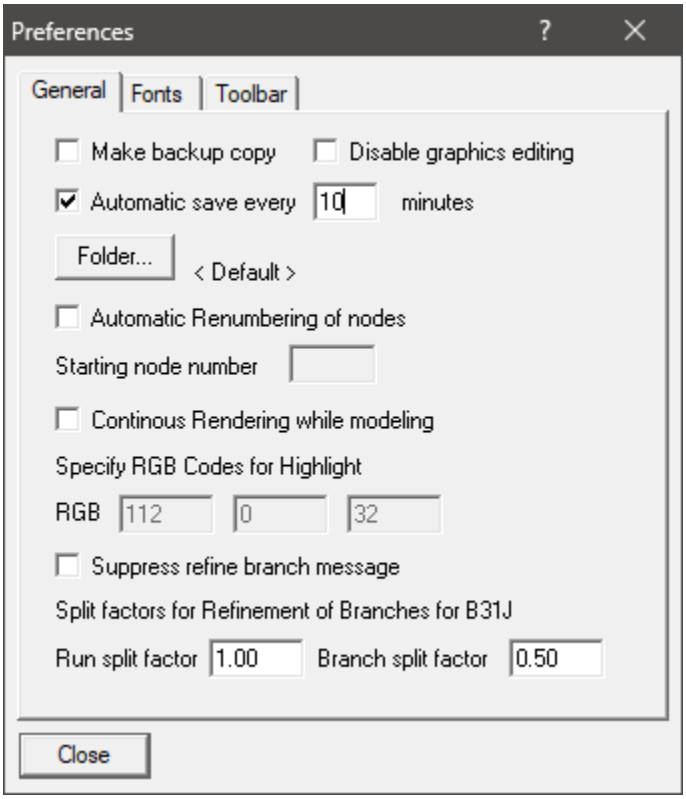
       Operating Pressure = 10 bar and Peak Pressure = 15 bar
    b. For Branch Line after Valve Node 105:

       Operating Pressure = 32 bar and Peak Pressure = 48 bar
8. **Operating Fluid and Specific Gravity**: Steam, 0.1
9. **Connection Details**:
    a. Node 5 connecting to Nozzle of a Cylindrical Vessel
    b. Node 50 connecting to Nozzle of a API 610 Horizontal Pump
10. **Wind Velocity**: 100 km/hr
11. **Static Seismic g's**: X=0.3, Y=0.2 and Z=0.3

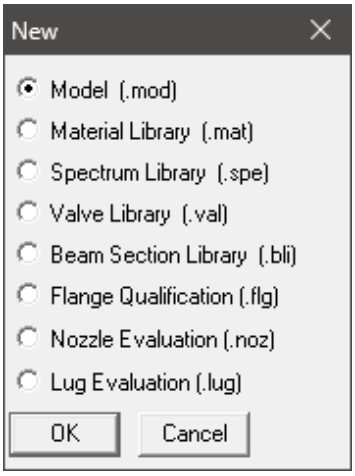
Start CAEPIPE. From the "File" pull down menu select "Preferences".



Make sure that the "Automatic save feature" is enabled and the "Automatic Renumbering of nodes" feature is disabled.

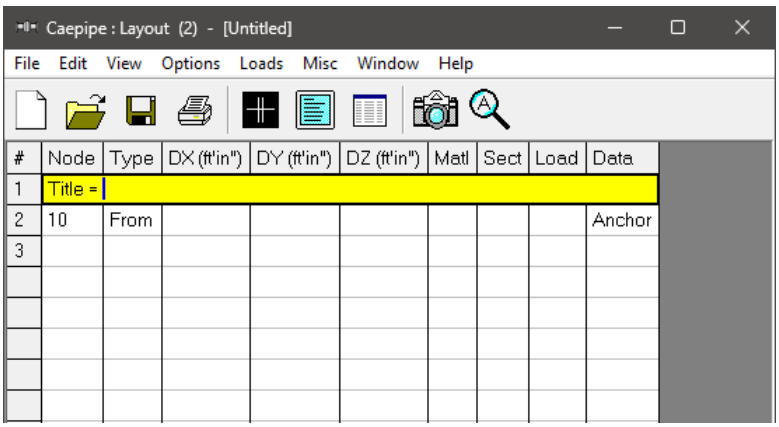


Start CAEPIPE. Then click on the "New" file button.

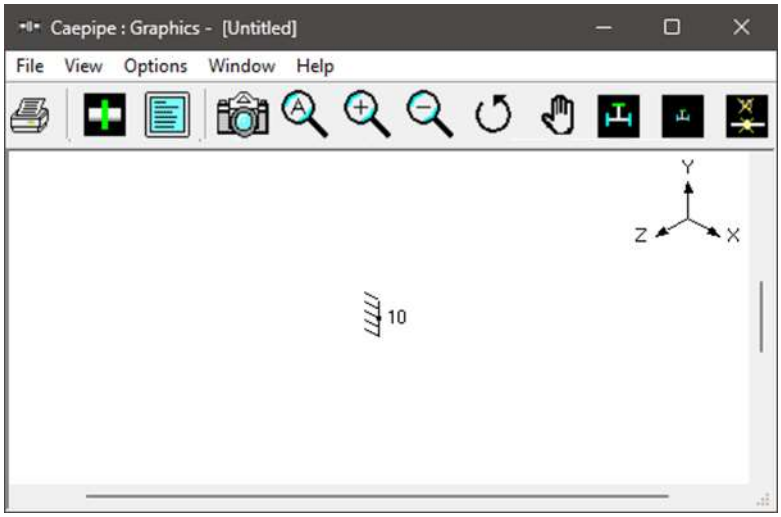


From the "New" file dialog, select the type of the new file as "Model (.mod)" file. This opens two independent windows: Layout and Graphics.

Layout window



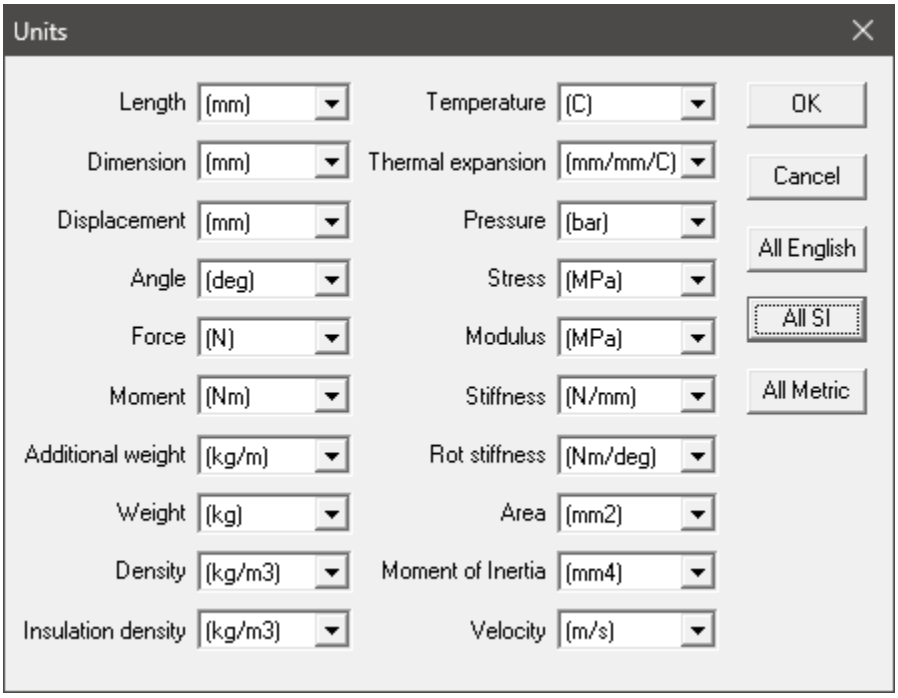
Graphics window



Adjust the size of the windows to fit your desktop such that you can view both comfortably at the same time.

Change Units

As this is a SI/Metric model, change the units appropriately. From the layout window, click on "Options menu > Units" (alternately, press the hotkey "Ctrl+U"). Click on "All SI" button followed by "OK". The layout window will show the offsets (DX/DY/DZ) in mm units.

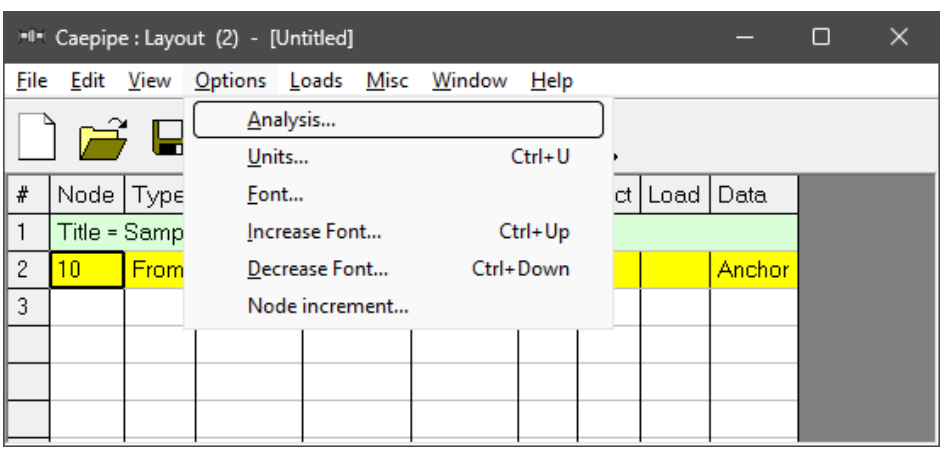


1. Enter Title

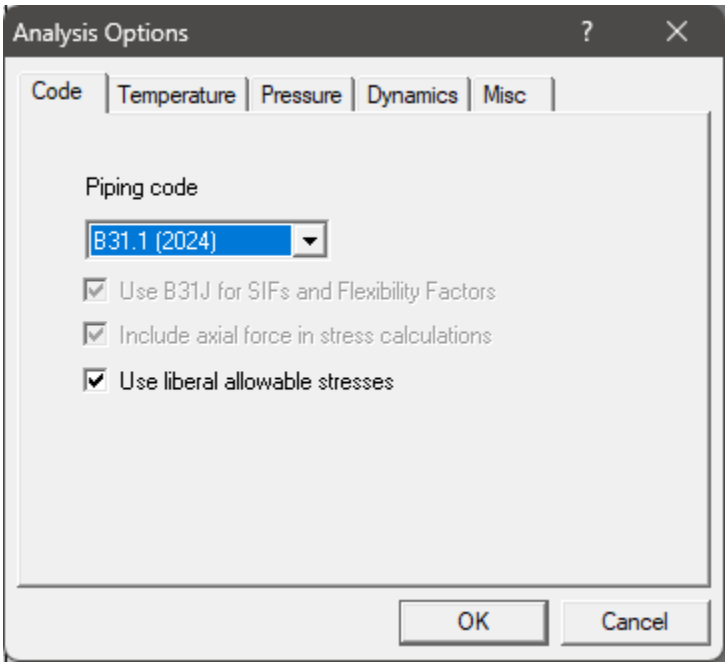
Type "Sample Problem 2" as the title in the first row that contains "Title =". Press Enter.

2. Select Analysis options (piping code etc.)

Click on the "Options" menu and then select "Analysis" (Options > Analysis) to specify options for analysis.



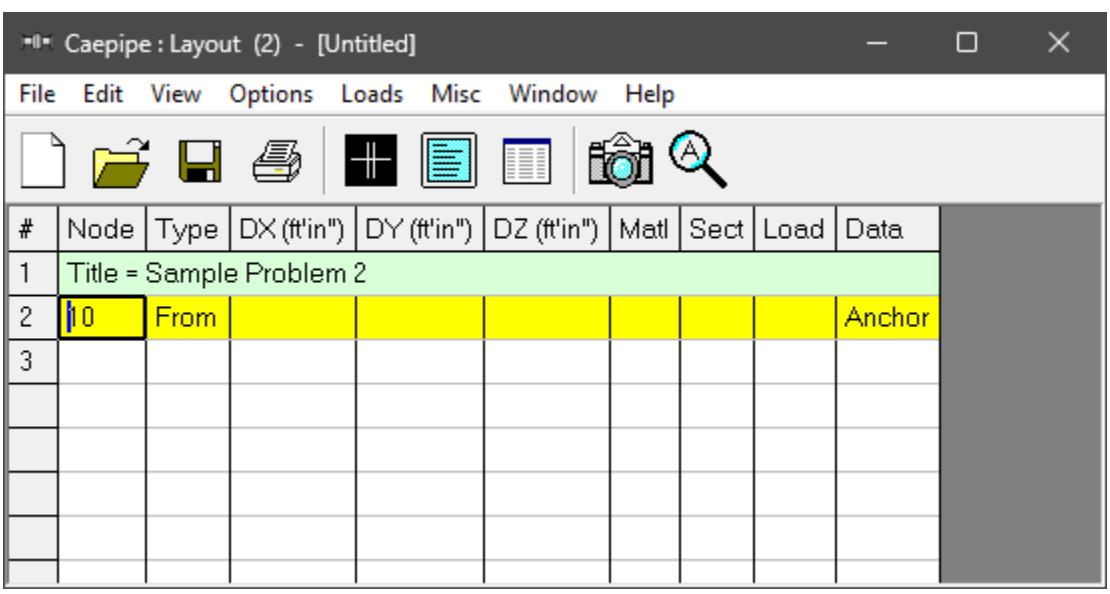
This opens the "Analysis Options" dialog.



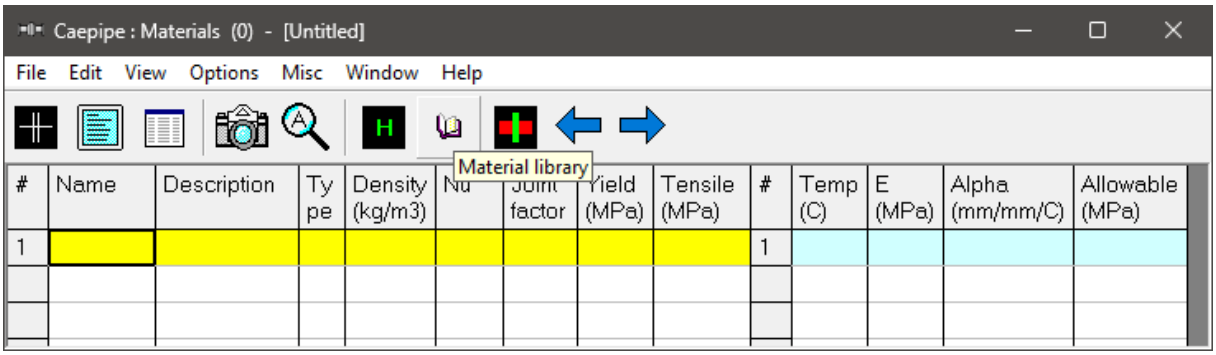
On the Code property page, select "B31.1 (2024)" for "Piping code". Turn ON the option "Use liberal allowable stresses". Then click on OK to close "Analysis Options" dialog.

3. Define Material, Sections and Loads

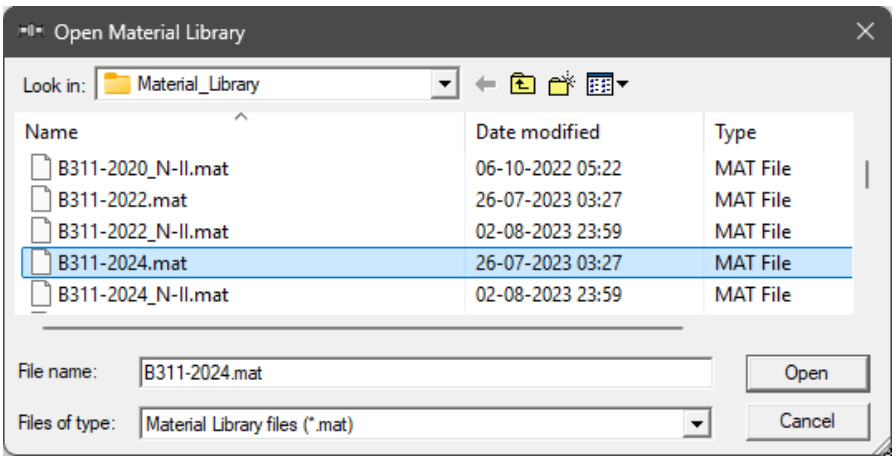
Click on "Matl" in the header in the Layout window (or press "Ctrl+Shift+M")



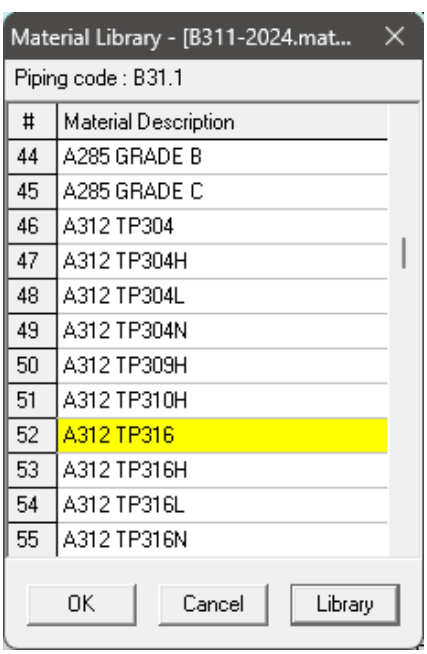
This opens up the "Materials" list in a separate List window. Position and resize the list window as you desire. Click on "Library" button on the Toolbar (or "choose File > Library").



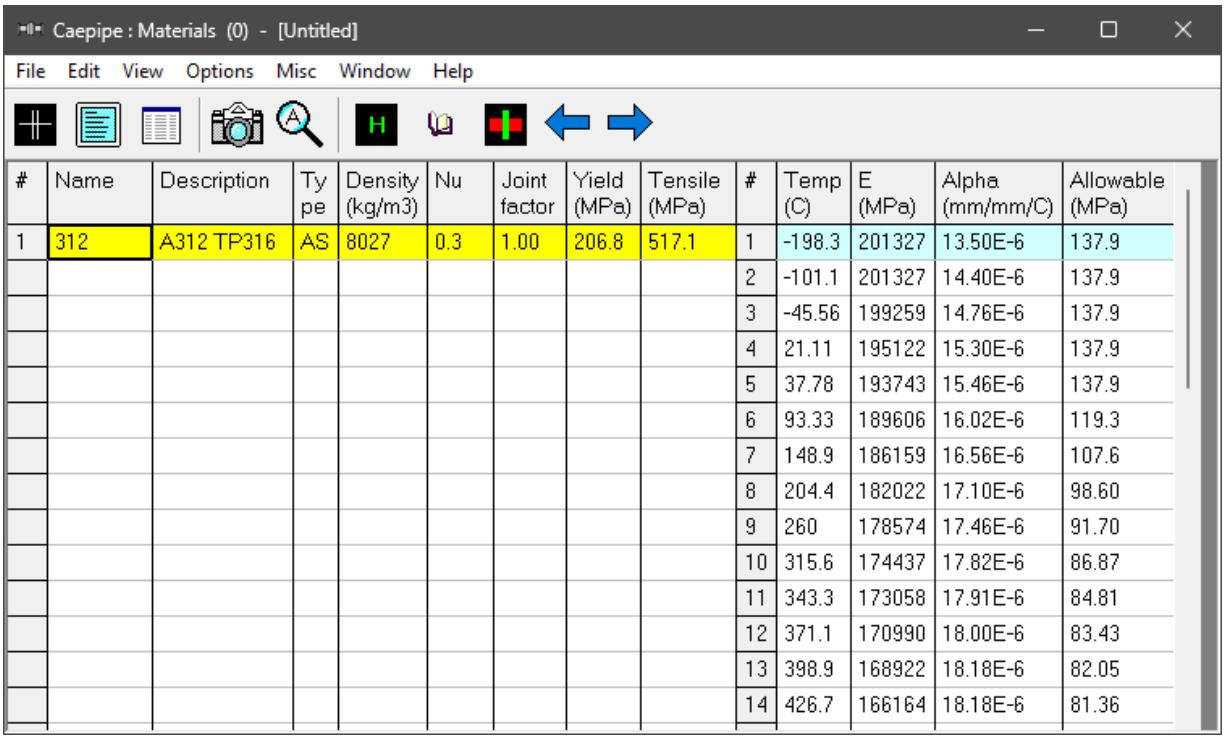
The Open Material Library dialog is shown. *If you don't see the folder shown below, then navigate to the Material Library folder under the CAEPIPE installed folder (usually C:\CAEPIPIE\xxxx, xxxx = version number).*



Select "B311-2022.mat" as the library file by double clicking on it. The available materials in the library are shown. Scroll down to "A312 TP 316". Double click on it or click on "OK" to select it.



The properties for this selected material are transferred to the material in the List window. Type "312" for material name and then **press "Enter"**.

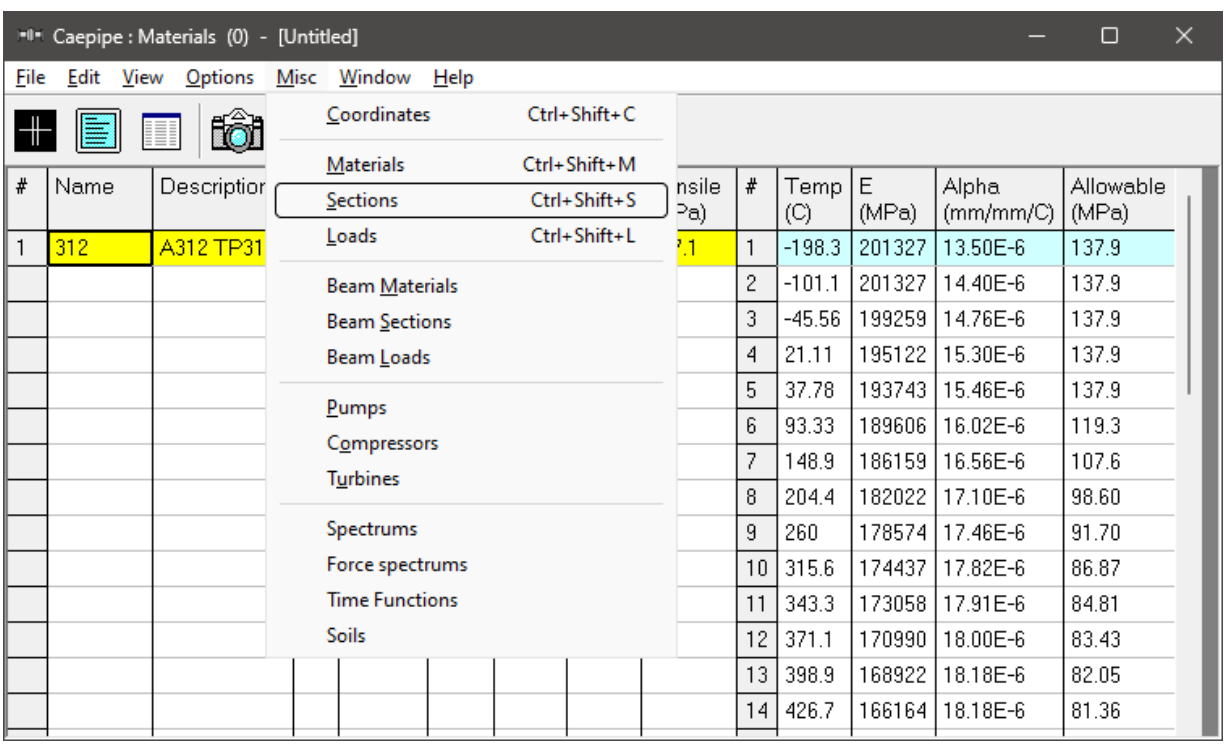


#	Name	Description	Type	Density (kg/m3)	Nu	Joint factor	Yield (MPa)	Tensile (MPa)
1	312	A312 TP316	AS	8027	0.3	1.00	206.8	517.1

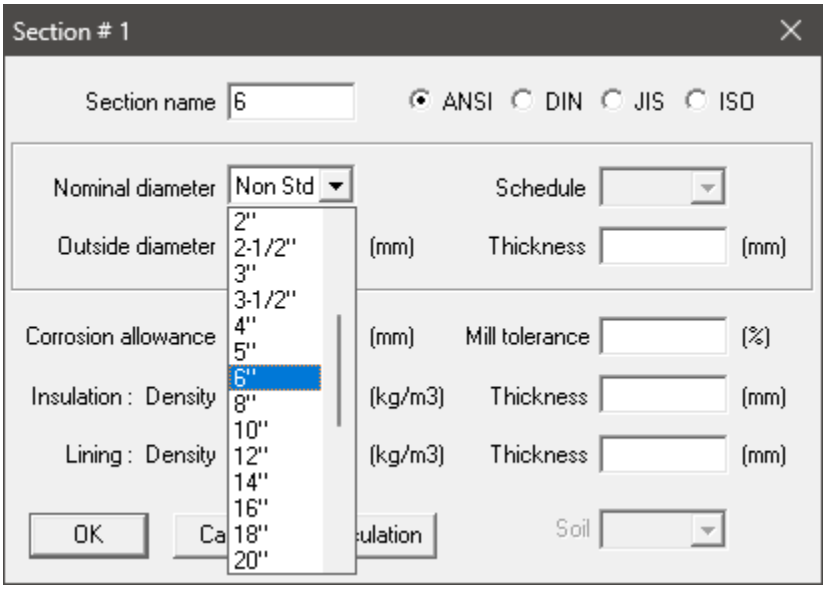
#	Temp (C)	E (MPa)	Alpha (mm/mm/C)	Allowable (MPa)
1	-198.3	201327	13.50E-6	137.9
2	-101.1	201327	14.40E-6	137.9
3	-45.56	199259	14.76E-6	137.9
4	21.11	195122	15.30E-6	137.9
5	37.78	193743	15.46E-6	137.9
6	93.33	189606	16.02E-6	119.3
7	148.9	186159	16.56E-6	107.6
8	204.4	182022	17.10E-6	98.60
9	260	178574	17.46E-6	91.70
10	315.6	174437	17.82E-6	86.87
11	343.3	173058	17.91E-6	84.81
12	371.1	170990	18.00E-6	83.43
13	398.9	168922	18.18E-6	82.05
14	426.7	166164	18.18E-6	81.36

Sections

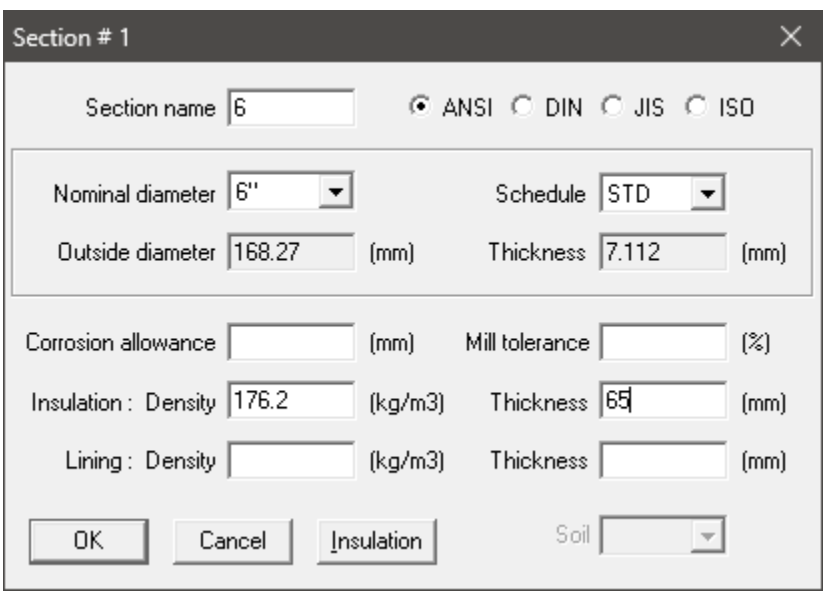
Select "Sections" from the "Misc" menu of the List window (or press "Ctrl+Shift+S").



A list of Sections is shown. This system has three sections: 6", 8" and 10". To enter the first section, type '6' for "Section name" and press "Enter". The "Section #1" properties dialog is shown with the section name 6.



Click on the down arrow of the dropdown combo box for "Nominal diameter" and select 6" for "Nominal diameter". Select/Enter other properties (STD thickness, Insulation density [Alt+I may be used for a list of insulation materials or you may enter your own density, in this case, 176.2 kg/cu.m] and thickness).

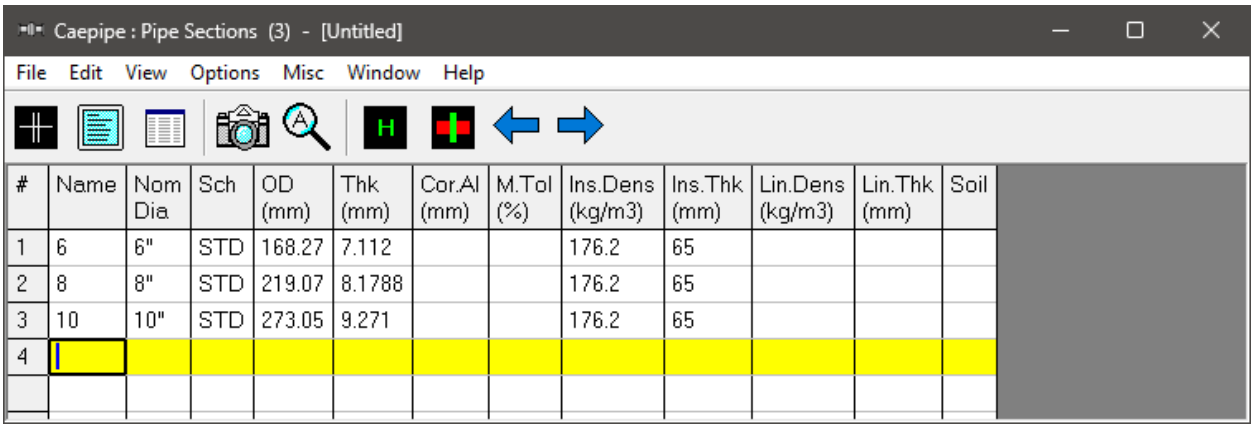


After entering all properties, press "Enter" or click on "OK" to enter the first section.

Now repeat the process for the 8" pipe section.

In row # 2, Type 8 for "Section name" and press "Enter". The "Section Properties" dialog is shown with the section name 8. Select 8" for "Nominal diameter", STD for "Schedule", and same insulation

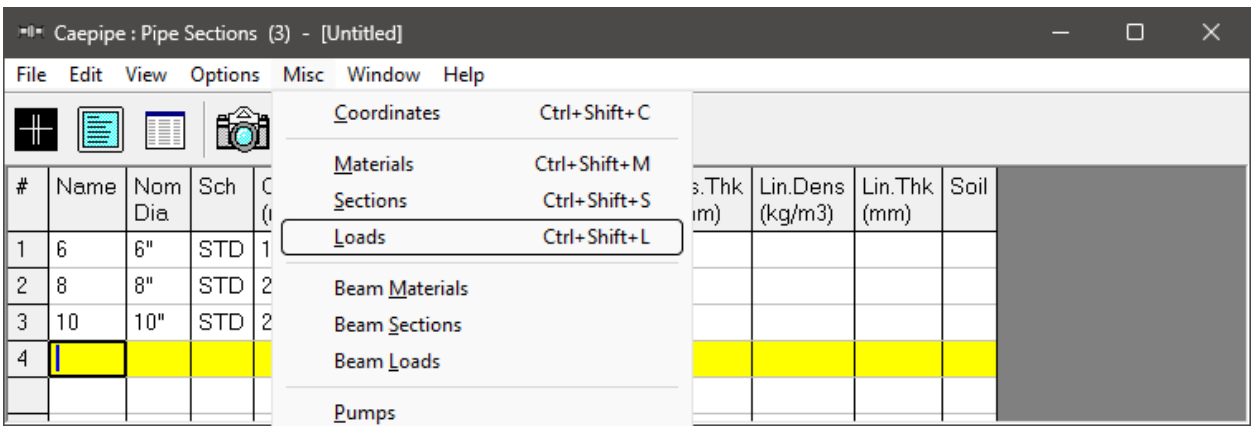
properties as before for "Insulation". Press "Enter" or click on "OK' to enter the second section. Do similarly for the 10" pipe section.



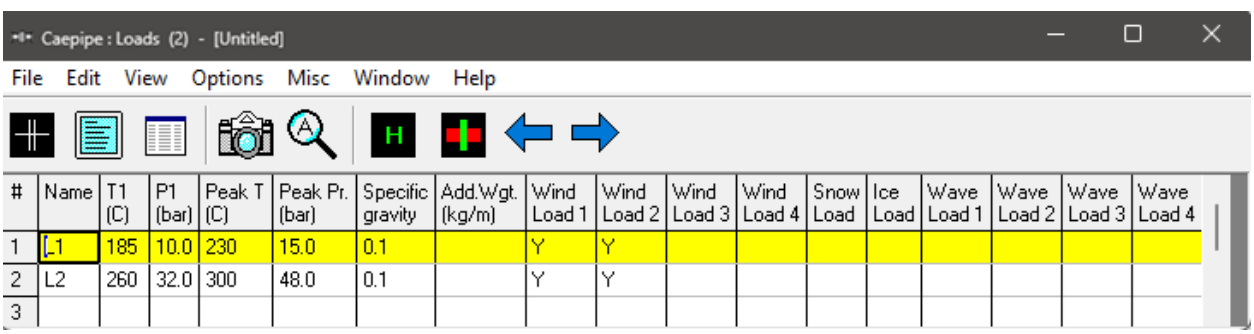
#	Name	Nom Dia	Sch	OD (mm)	Thk (mm)	Cor.Al (mm)	M.Tol (%)	Ins.Dens (kg/m3)	Ins.Thk (mm)	Lin.Dens (kg/m3)	Lin.Thk (mm)	Soil
1	6	6"	STD	168.27	7.112			176.2	65			
2	8	8"	STD	219.07	8.1788			176.2	65			
3	10	10"	STD	273.05	9.271			176.2	65			
4												

Load

Select "Loads" from the "Misc" menu (or press "Ctrl+Shift+L").



The Loads list is shown. To enter the first load, Type 'L1' for "Name", Tab to "T1" and type 185, Tab to "P1" and type 10 bar, tab to "Peak T" and type 230, Tab to "Peak Pr." and type 15 and Tab to "Specific gravity" and type 0.1. Then press "Enter". That is it! The load is entered. (Alternately, you could have pressed "Ctrl+E" on the first row and typed in the same information in a dialog box). Similarly, enter the second load set "L2" {260°C, 32 bar, 300°C, 48 bar and Sp. Gravity = 0.1}.



#	Name	T1 (C)	P1 (bar)	Peak T (C)	Peak Pr. (bar)	Specific gravity	Add.Wgt. (kg/m)	Wind Load 1	Wind Load 2	Wind Load 3	Wind Load 4	Snow Load	Ice Load	Wave Load 1	Wave Load 2	Wave Load 3	Wave Load 4
1	L1	185	10.0	230	15.0	0.1		Y	Y								
2	L2	260	32.0	300	48.0	0.1		Y	Y								
3																	

Click in the "Layout window" or press "F3" to move the focus to the "Layout window'.

4. Input Model Layout

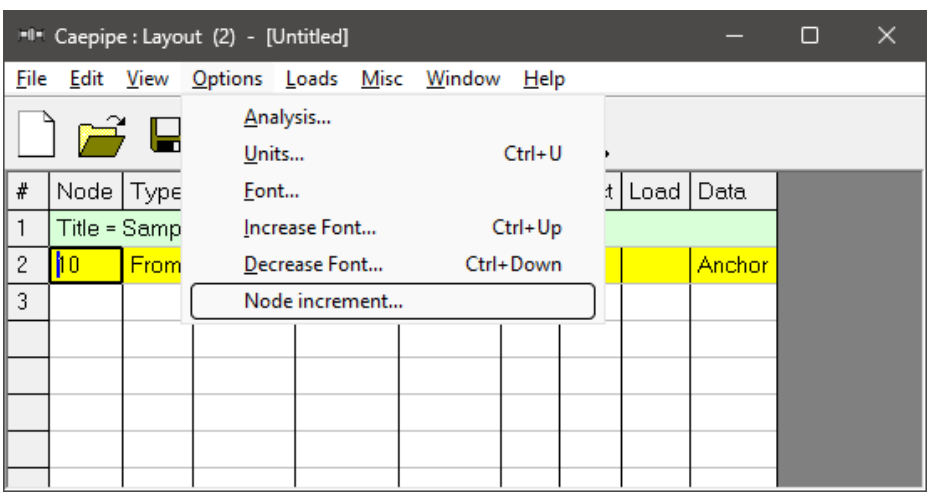
We are going to model the 10" main line first, followed by the 8" segment.

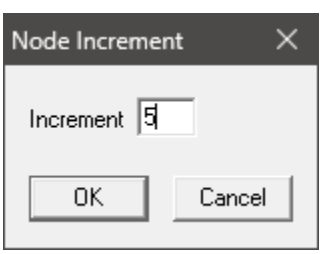
CONVENTIONS

- In the following text, the word 'type' should be distinguished from the words 'Type column' or simply 'Type' (upper case 'T'). The former ('type') will mean press the keys on the keyboard. The latter word 'Type' will refer to the "Type column" in the Layout spreadsheet. Of course, occurrence of Type at the beginning of a sentence will mean type the keys.
- Also, the instruction "type B for Bend" does not necessarily mean the upper case 'B'. The lower case 'b' can also be typed.
- For items in the "Data" column (such as "Anchor" or "Hanger"), the cursor needs to be in the "Data" column. To move the cursor quickly to that column, press "Ctrl+Shift+D" from any column or click in the "Data" column. Or press the Tab key repeatedly to reach the "Data" column.
- As the graphics window is simultaneously updated, you should position the graphics window in such a way that you can see it along with the input window. Simultaneous feedback is one of the chief design intents in CAEPIPE.
- For mouse clicks, when you read the word "click on xxx," this means left-click on your mouse. For the context menu, if referred to, right-click.

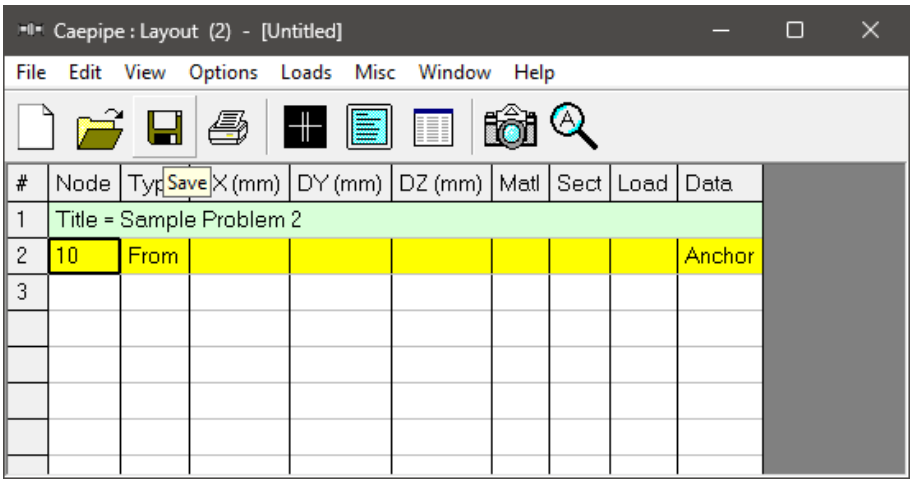
Change Node Increment

You might have noticed in the model drawing that the node numbering scheme has an increment of 5. CAEPIPE has a feature that allows you to specify a node increment. Select "Options menu > Node increment"…type 5 for value. Click on "OK".

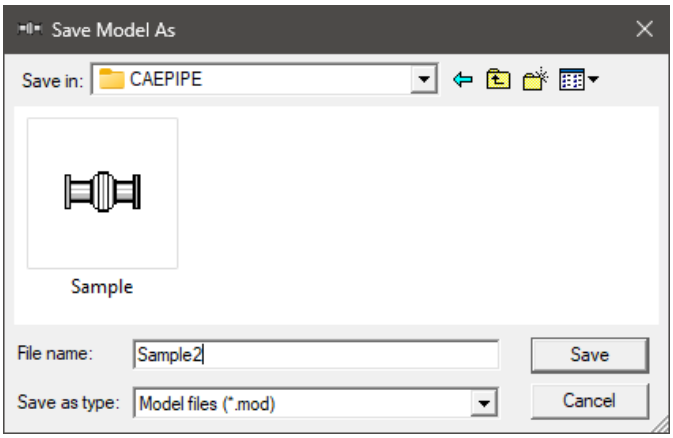




After defining the above parameters, Save the model by clicking on the Save button.



The "Save Model As" dialog is shown.



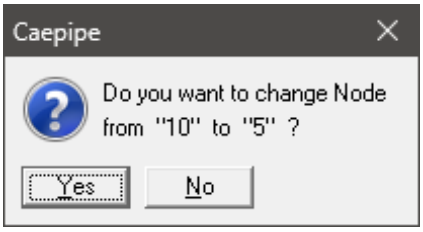
Type the File name as "Sample2" and press "Enter" to save the model.

First model the 10" Main line

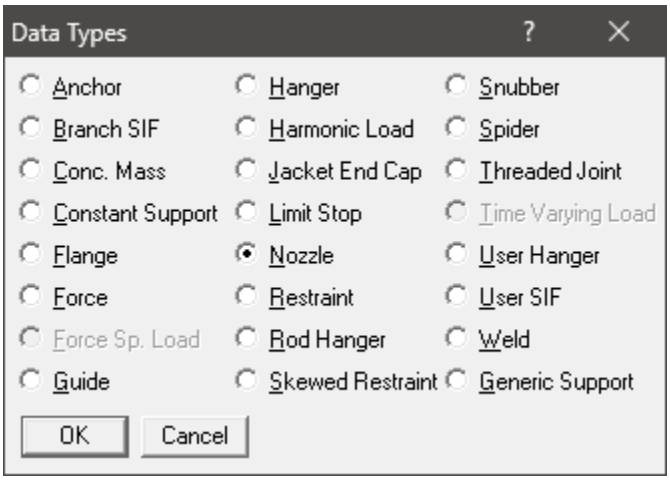
Following the "Title" at row #1, row #2 is already generated with Node 10 of Type "From" with an Anchor in the "Data" column.

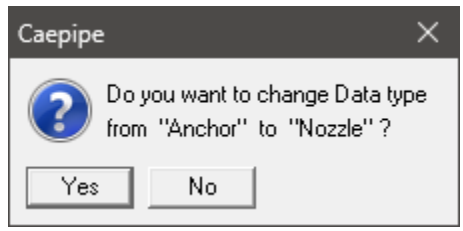
Model information shows that the piping is connecting to a Nozzle of a Cylindrical Vessel with node number as 5. So, to account for the stiffness of the Nozzle protruding out of the Cylindrical Vessel, the nozzle portion is modeled as a pipe in this model. The junction of this Pipe (Nozzle) and the Shell is modeled as "Nozzle".

To change the Node number and to replace "Anchor" with "Nozzle", click on 10, press Backspace to erase 10, type 5. Press Tab to advance. Confirm the node number change when asked (by clicking on Yes, or simply pressing the Spacebar key on the keyboard).



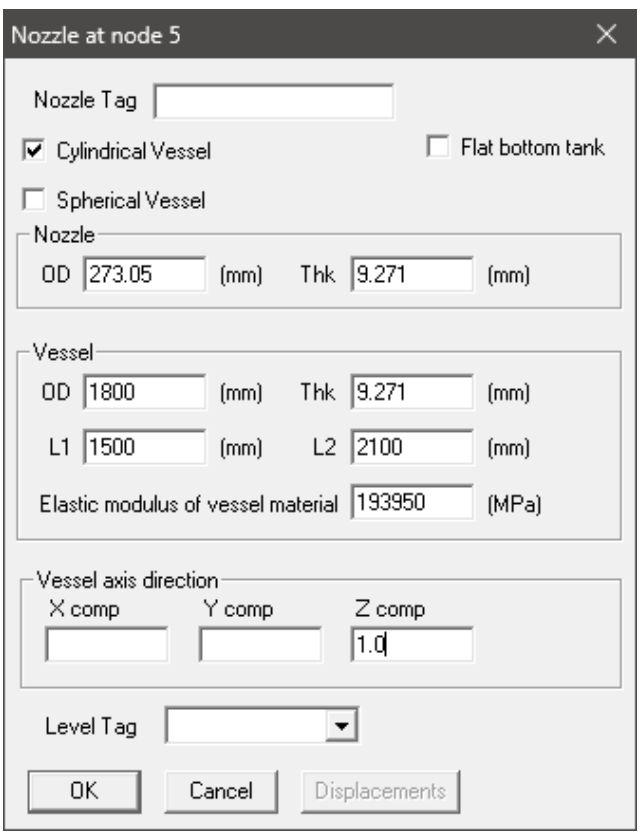
To replace the "Anchor" with "Nozzle", highlight the data type "Anchor" at row #2 using mouse left button and then click on "Data" in the header in the Layout window. From the "Data types" dialog box shown, select the new data type as "Nozzle".

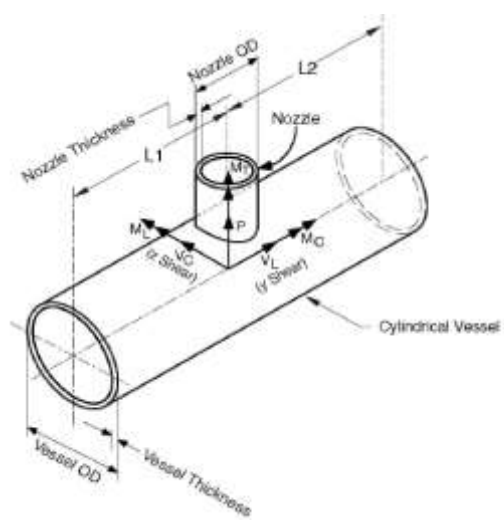




CAEPIPE will prompt as shown below. Press "Yes" to proceed.

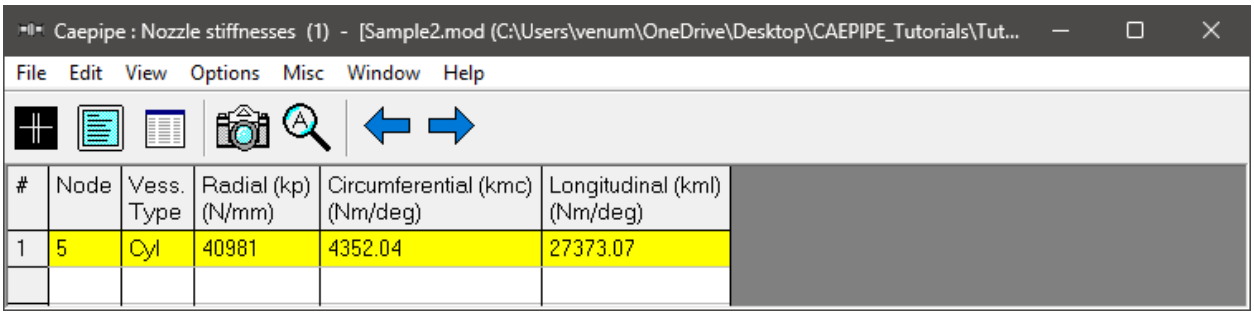
Enter the Nozzle and Vessel parameters as shown below and press "OK".





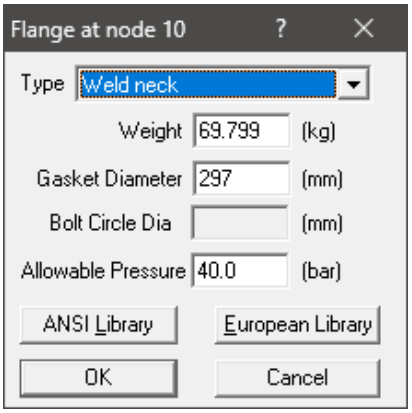
From the snap shots shown above, Lengths L1 and L2 on either side of the nozzle are the distances from the nozzle center line to the nearest location on vessel where the "ovalization deformation" of the vessel is stopped such as at a stiffener on the inner or outer surface of the vessel, or at the center of a

saddle support to the vessel or at the junction to the torispherical enclosure (also called the head) or at a tube sheet inside the vessel etc. Nozzle stiffness computed by CAEPIPE can be seen through Layout window > View > List > Nozzle Stiffnesses.

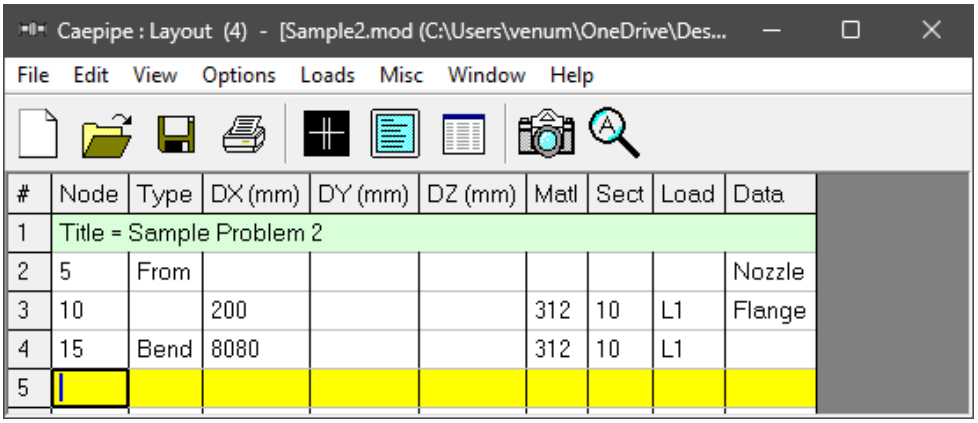


#	Node	Vess. Type	Radial (kp) (N/mm)	Circumferential (kmc) (Nm/deg)	Longitudinal (kml) (Nm/deg)
1	5	Cyl	40981	4352.04	27373.07

Now, press Enter to move the highlight to the next row (#3). Tab to the Type column. The next Node 10 is automatically assigned. Tab over to DX, type 200 (mm), Tab over to Material, press Enter to open the list of materials and select 312. Next Tab over to Section and press Enter. Select section 10 and press OK. Tab over to Load and press Enter, select L1 and click OK. Tab again to Data to input the flanges mating with the pipe and the equipment nozzle. Type "fl" to model flange and enter the data as shown below and press OK. CAEPIPE moves the highlight automatically to the next (new) row (#4).

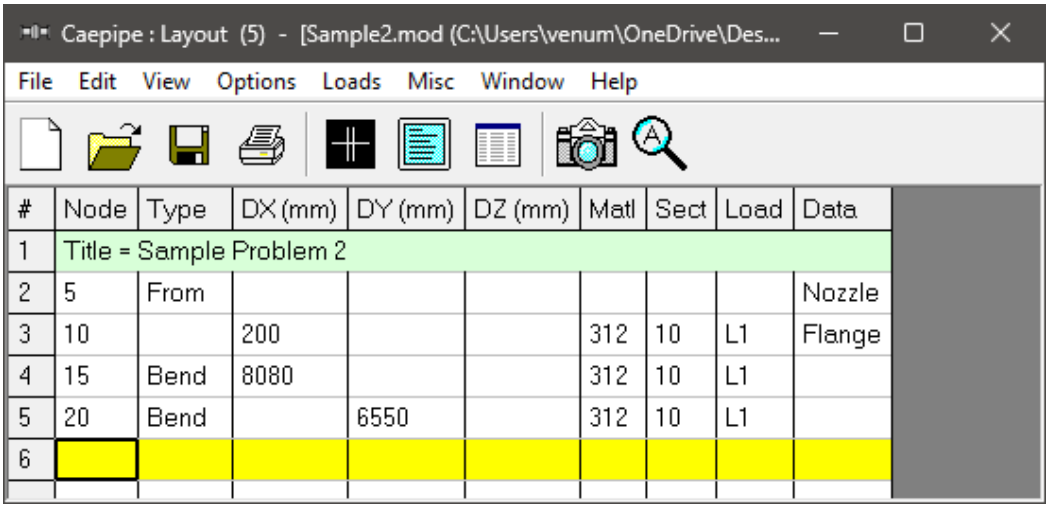


Tab to the type column. The next node 15 is automatically assigned. Node 15 has a LR (long radius) bend (in CAEPIPE, a bend node is defined always at the tangent intersection point, being such, this node does not exist on the physical bend). Tab to the Type column; type "ben" to insert a default LR bend. Tab to DX, type in 8080 (mm), press Enter. CAEPIPE automatically enters the material, section and load from the previous row and moves the highlight to the next new row.



#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
1	Title = Sample Problem 2								
2	5	From							Nozzle
3	10		200			312	10	L1	Flange
4	15	Bend	8080			312	10	L1	
5									

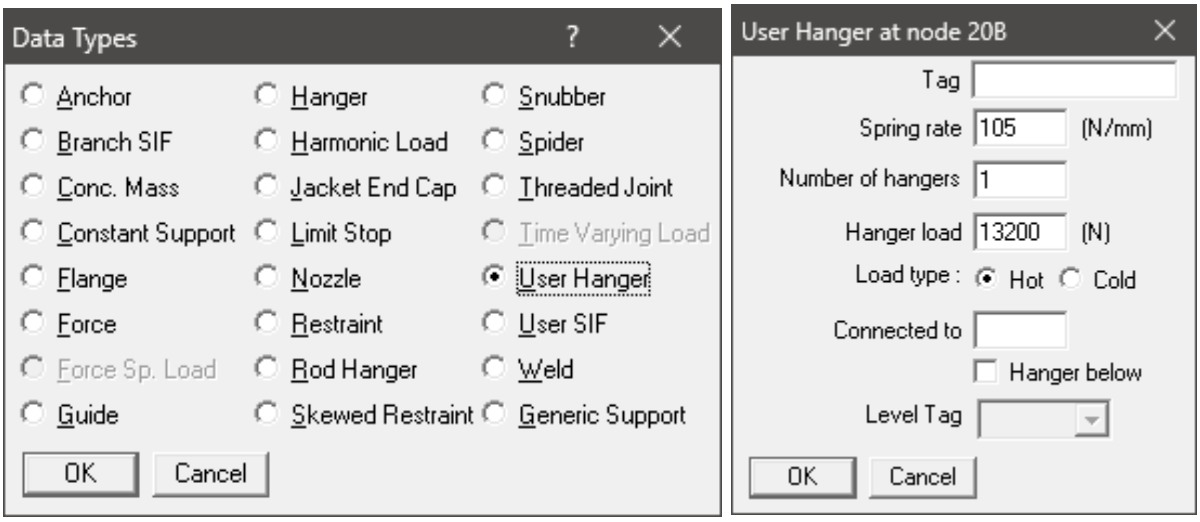
The following vertical bend (at node 20) can be modeled as before. Tab to Type (node 20 is automatically inserted), and type "ben" to insert a default LR bend, Tab again to DY, type 6550 (mm) and press Enter.



#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
1	Title = Sample Problem 2								
2	5	From							Nozzle
3	10		200			312	10	L1	Flange
4	15	Bend	8080			312	10	L1	
5	20	Bend		6550		312	10	L1	
6									

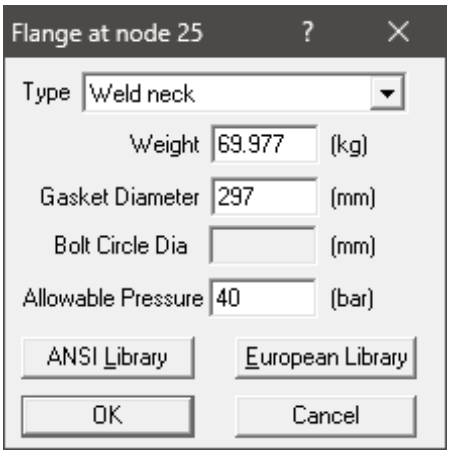
This bend has an already existing hanger (called "User Hanger" in CAEPIPE) at the far end, referred to as node 20B, an internally generated bend node.

So, on the next row, type 20B for Node, Tab to Type, press "L" for Location, which spawns the available data types you can insert at this node. Pick "User Hanger" from the dialog.

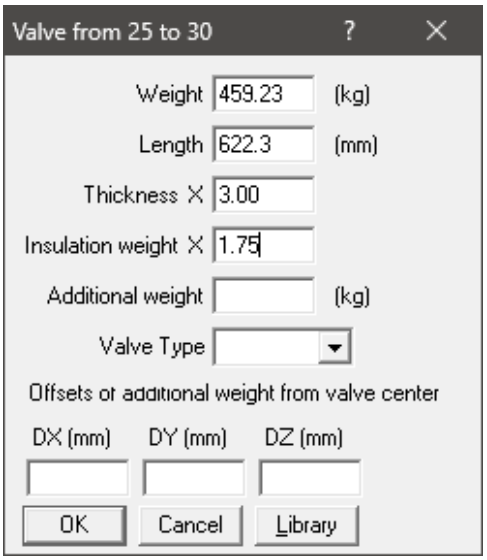


Enter its properties as shown. Click on OK.

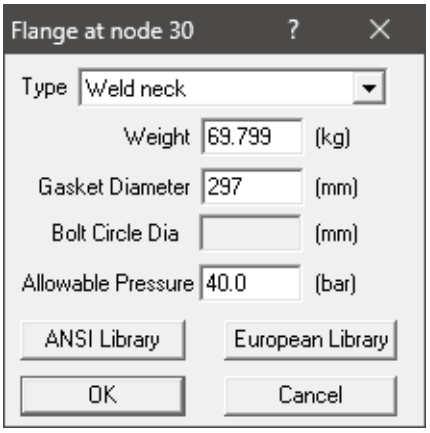
Next, the line moves in the Z direction to the flange node 25. Pressing Tab on the new row generates node 25 for you. Tab to DZ, type 4240, (click in Data column) or press Ctrl+Shift+D to move cursor to Data column. Type "fl" to open the Flange Data type dialog. Enter the details shown below and press OK.

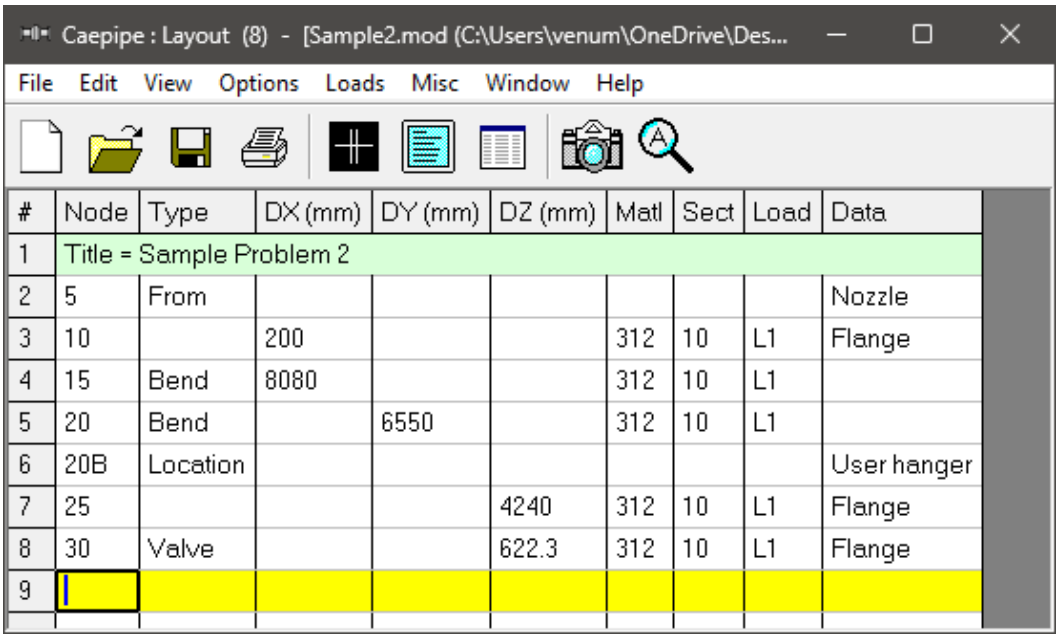


A valve is placed next from Node 25 to Node 30, where another mating flange is located. Pressing Tab on the new row generates node 30. Tab to the Type column; type "v" to insert a "Valve" and enter the data as shown below and press OK.



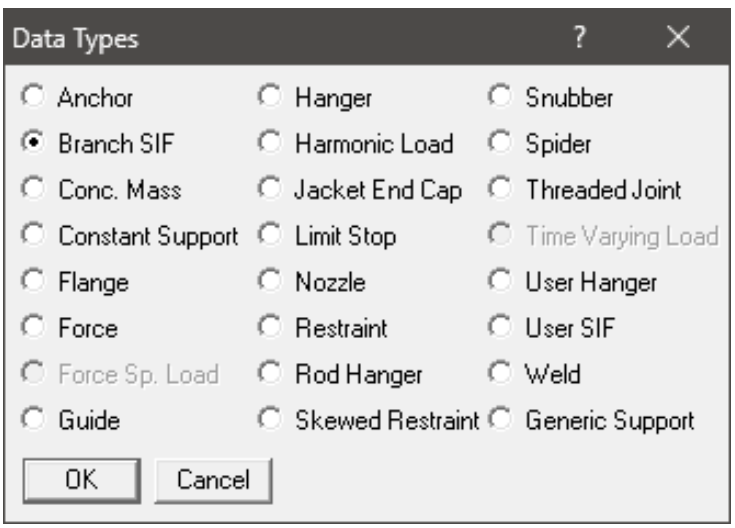
Tab to Data and type "fl" to enter a "flange". Type "fl" to open the Flange Data type dialog. Enter the details shown below and press OK.

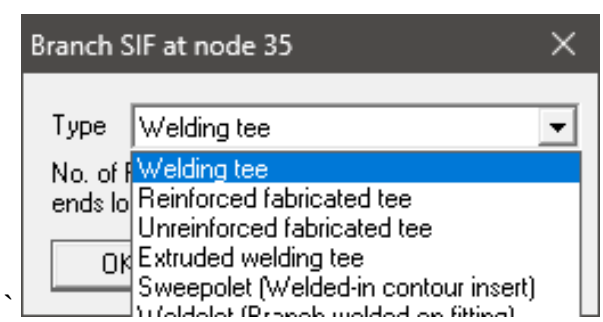


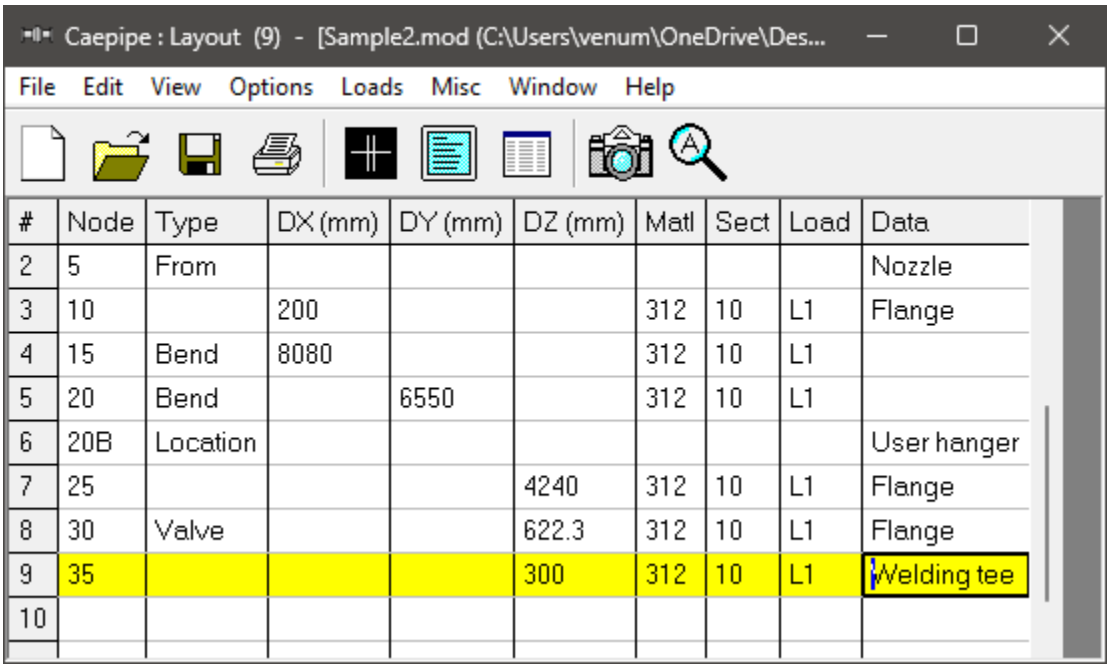


#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
1	Title = Sample Problem 2								
2	5	From							Nozzle
3	10		200			312	10	L1	Flange
4	15	Bend	8080			312	10	L1	
5	20	Bend		6550		312	10	L1	
6	20B	Location							User hanger
7	25				4240	312	10	L1	Flange
8	30	Valve			622.3	312	10	L1	Flange
9									

Next model a pipe element till node 35 (welding tee). Press Tab for node 35, Tab to DZ, type 300, (click in Data column) or press Ctrl+Shift+D to move cursor to Data column. Type "br" (or right-click in Data, select Branch SIF) to open the Tee types Data type dialog. Select Welding Tee from the dropdown box. Click on OK (or press Enter).



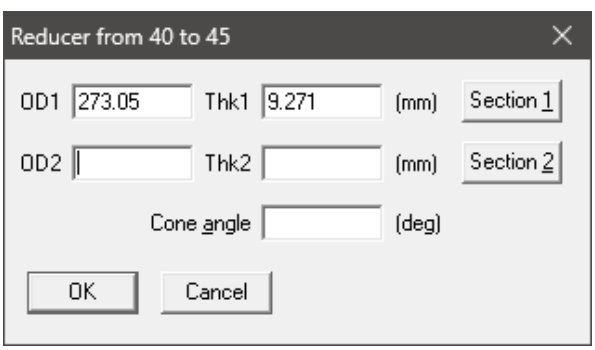




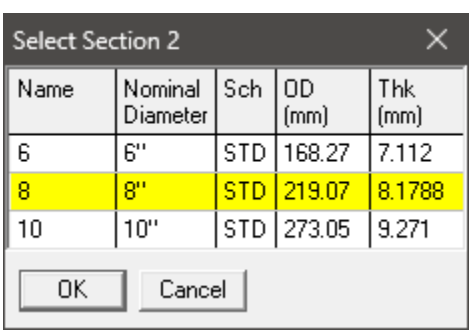
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
2	5	From							Nozzle
3	10		200			312	10	L1	Flange
4	15	Bend	8080			312	10	L1	
5	20	Bend		6550		312	10	L1	
6	20B	Location							User hanger
7	25				4240	312	10	L1	Flange
8	30	Valve			622.3	312	10	L1	Flange
9	35				300	312	10	L1	Welding tee
10									

Next model a pipe element till node 40. Press Tab for node 40, Tab to DZ, type 300 and press Enter.

The next element is a 10x8 concentric reducer. Here is how to model it. Tab for the next node # (45), type "red" for Reducer in the Type column. CAEPIPE displays the Reducer dialog with the current section properties.

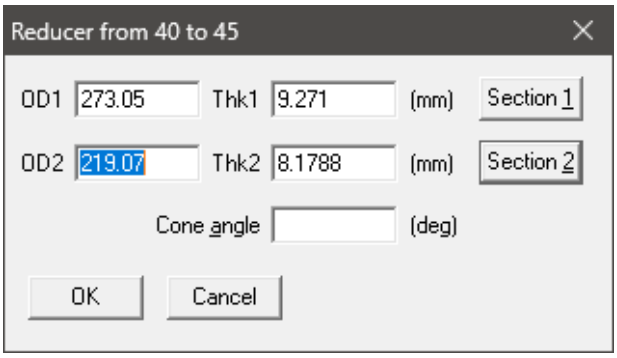


Click on "Section 2" button to select the following section, in this case, the 8" section. After placing the highlight on the 8" section, press Enter (or click on OK).

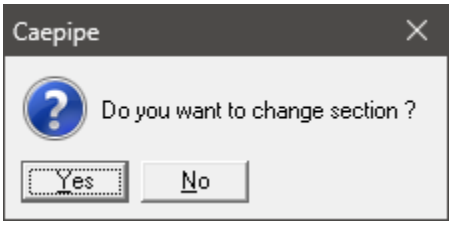


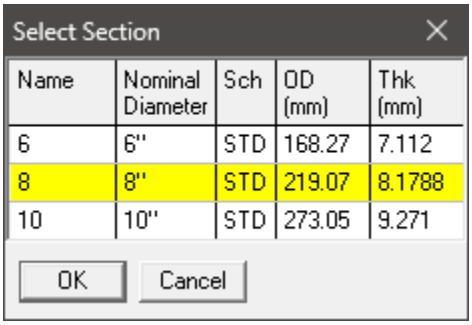
Name	Nominal Diameter	Sch	OD (mm)	Thk (mm)
6	6"	STD	168.27	7.112
8	8"	STD	219.07	8.1788
10	10"	STD	273.05	9.271

You are back at the Reducer dialog.



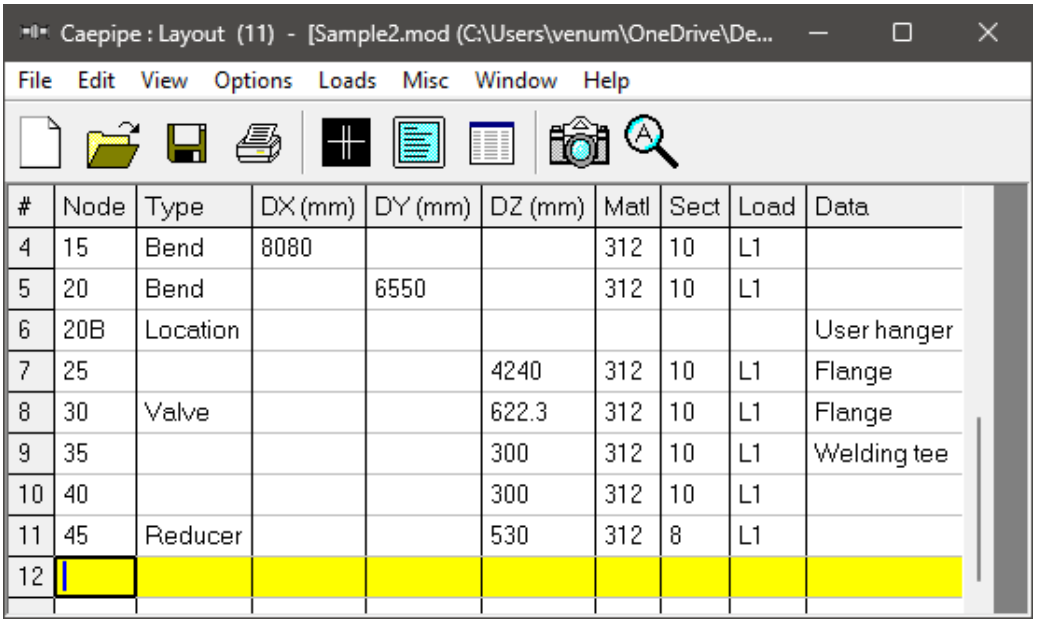
Click on OK to finish inserting the reducer. On the layout screen, type 530 for DZ and press Enter, at which point CAEPIPE wants you to confirm the section change. Click on Yes.





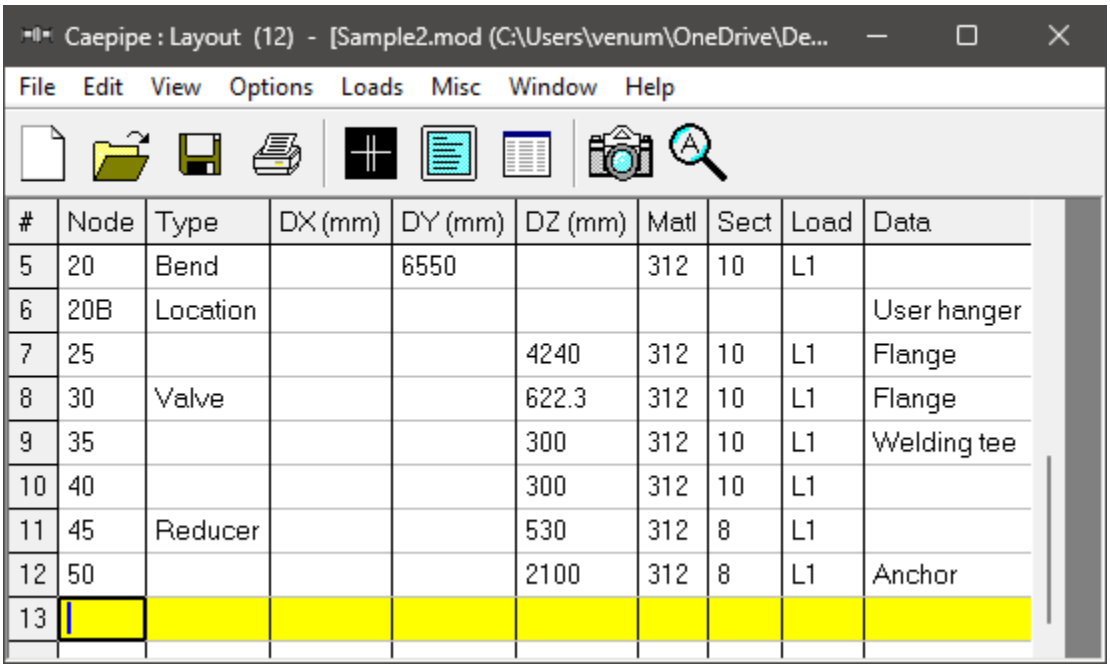
Name	Nominal Diameter	Sch	OD (mm)	Thk (mm)
6	6"	STD	168.27	7.112
8	8"	STD	219.07	8.1788
10	10"	STD	273.05	9.271

Then select 8 as the new section from here on. Press Enter to move to next row.



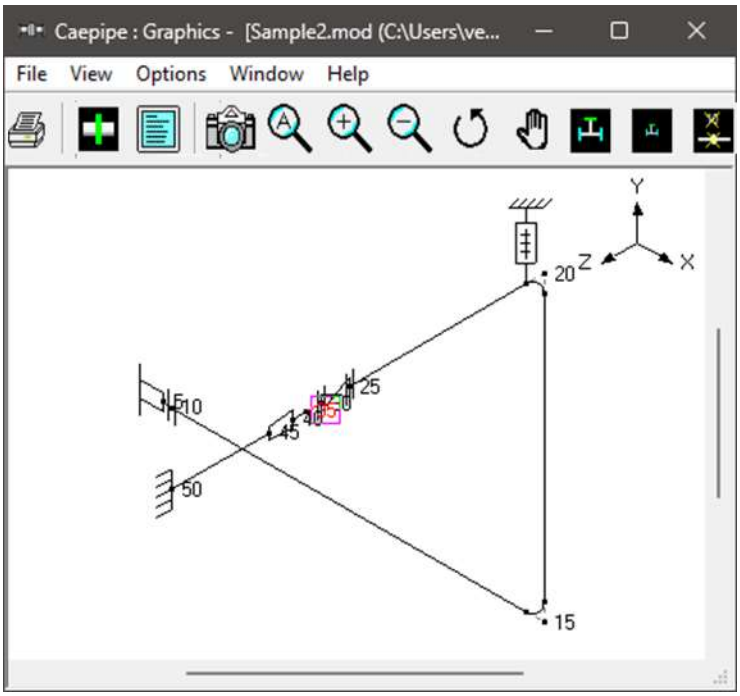
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
4	15	Bend	8080			312	10	L1	
5	20	Bend		6550		312	10	L1	
6	20B	Location							User hanger
7	25				4240	312	10	L1	Flange
8	30	Valve			622.3	312	10	L1	Flange
9	35				300	312	10	L1	Welding tee
10	40				300	312	10	L1	
11	45	Reducer			530	312	8	L1	
12									

The last element here is an 8" pipe that ends at node 50. As before, press Tab for Node 50, type 2100 for length in the same direction. Press Ctrl+Shift+D to go to Data and press A to insert a rigid anchor (note that CAEPIPE inserts the correct old material, new section and old load for this row).

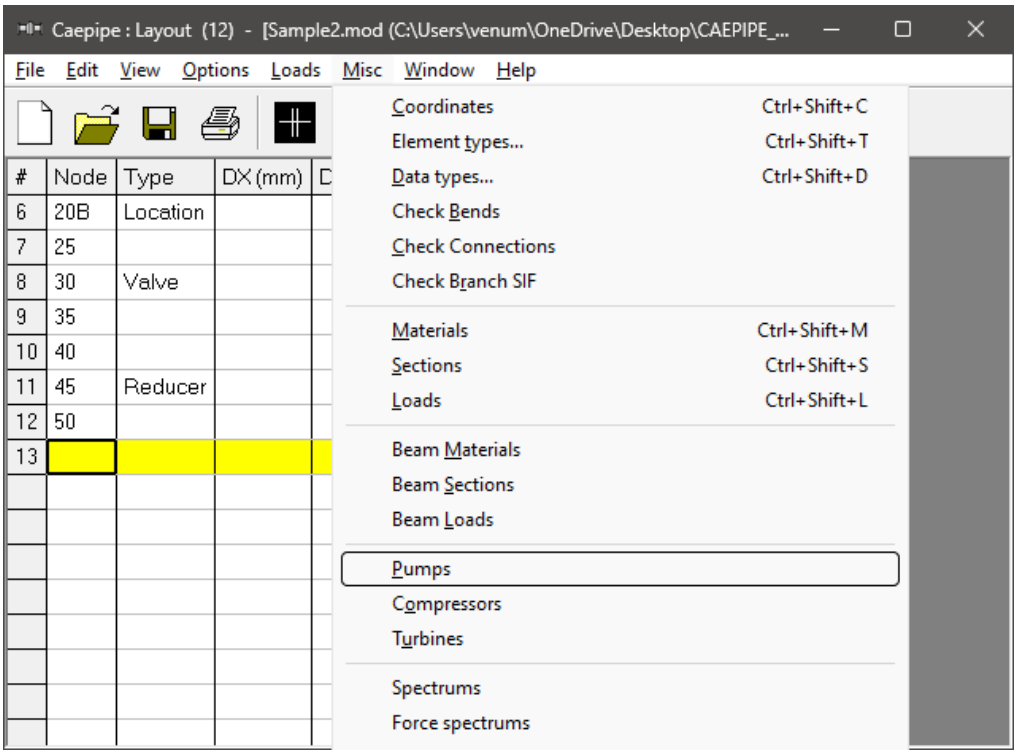


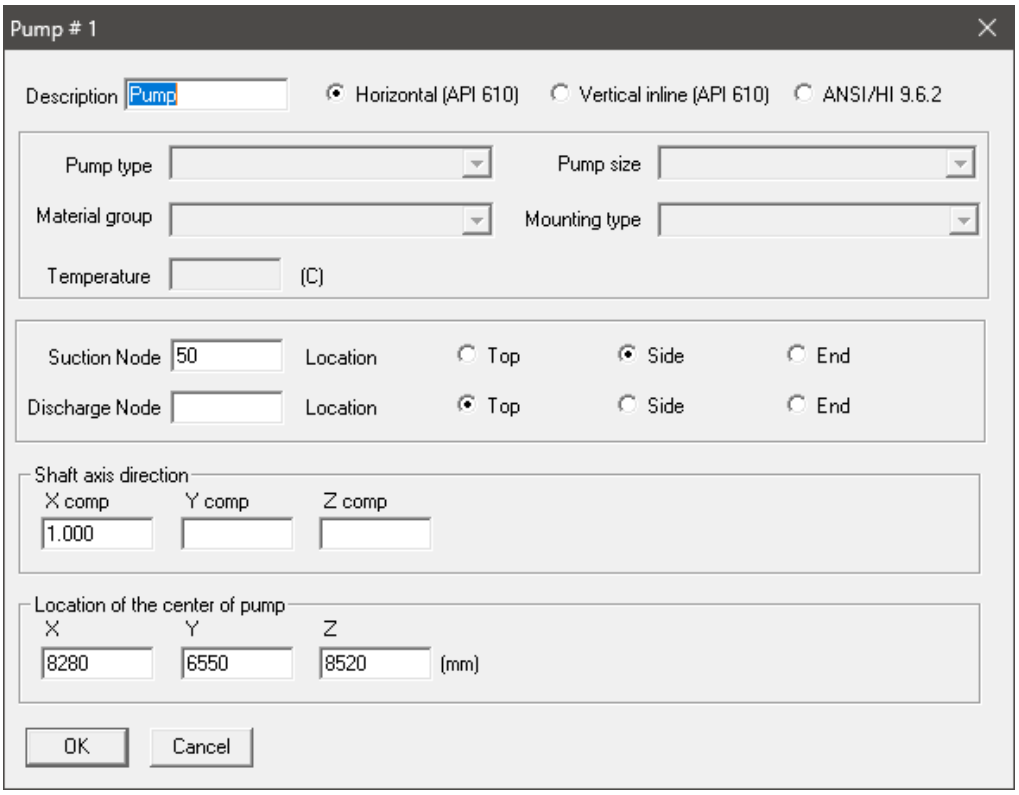
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
5	20	Bend		6550		312	10	L1	
6	20B	Location							User hanger
7	25				4240	312	10	L1	Flange
8	30	Valve			622.3	312	10	L1	Flange
9	35				300	312	10	L1	Welding tee
10	40				300	312	10	L1	
11	45	Reducer			530	312	8	L1	
12	50				2100	312	8	L1	Anchor
13									

Click on the Zoom All button (or press Ctrl+A) to view the header line fully in the graphics window.



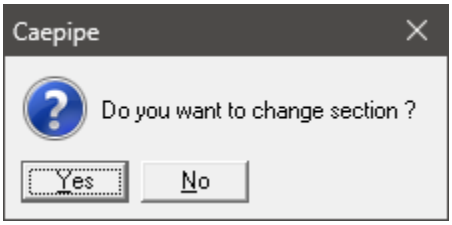
Node 50 is connecting to a Side Suction Nozzle of an API 610 Horizontal Pump. To model this, select the option "Pumps" through Layout Window > Misc. Double click on an empty row and enter the values as shown below. Once modeled, CAEPIPE will automatically perform the Pump Qualification and shows the report in Results.



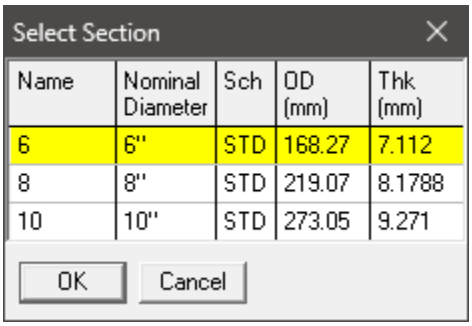


Now the 6" branch

On the next row (#13), type 35 for Node, Tab to the Type column, type 'f' (for "From", since we are beginning a new branch from an existing Node 35), press Enter. In the next row (#14), type "100" in the Node column to clearly identify the new branch. Tab to DX and enter –1400. CAEPIPE inserts the previous material, and automatically detects the new branch and asks if you want to change section.



Since we want to change the section to 6, click on Yes. This opens the Section selection dialog.

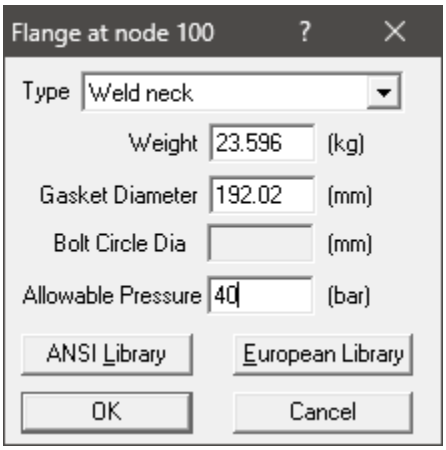


Select Section

Name	Nominal Diameter	Sch	OD (mm)	Thk (mm)
6	6"	STD	168.27	7.112
8	8"	STD	219.07	8.1788
10	10"	STD	273.05	9.271

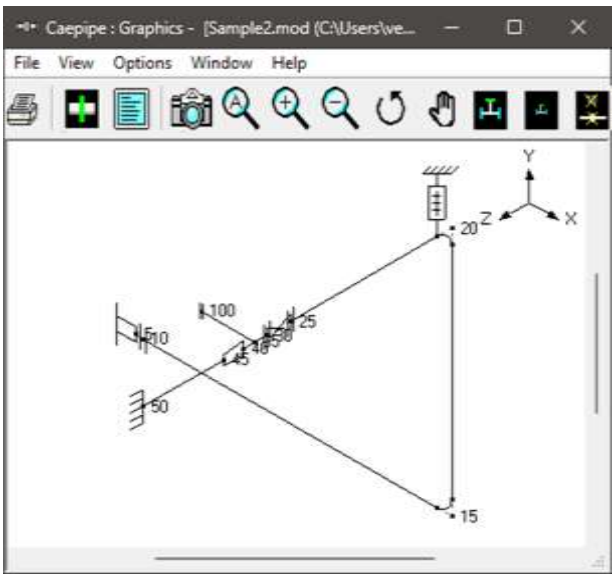
OK Cancel

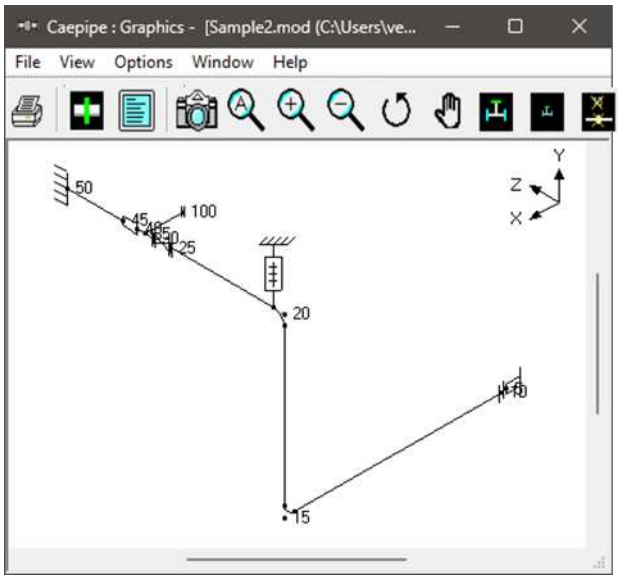
Select the 6" section by double clicking on it. The section (6) is entered in the Section column in the Layout window. The load is again automatically inserted from the previous load. Lastly, type "fl" in the Data column and hit enter to create a mating Flange. This will bring up the Flange type dialog box.

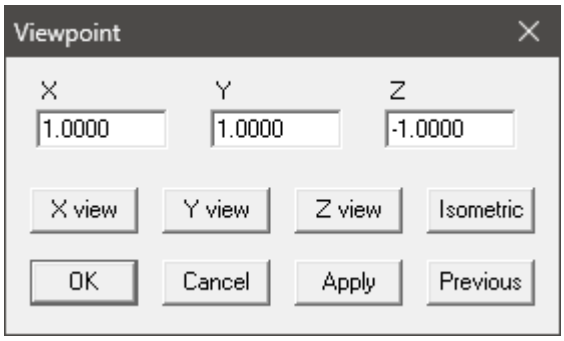


Type in 23.596 for Weight, 192.02 for Gasket Diameter, 40 for Allowable Pressure and click Ok.

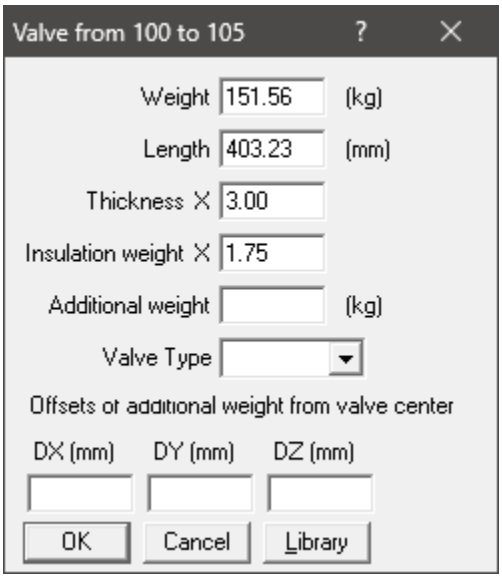
The graphics window will look like this. For better view, rotate the model by clicking the icon and scrolling the horizontal scroll bar towards left using the mouse left button or through keyboard left arrow key. Alternatively, you can specify the viewpoint as shown below by selecting the icon from the graphics frame.





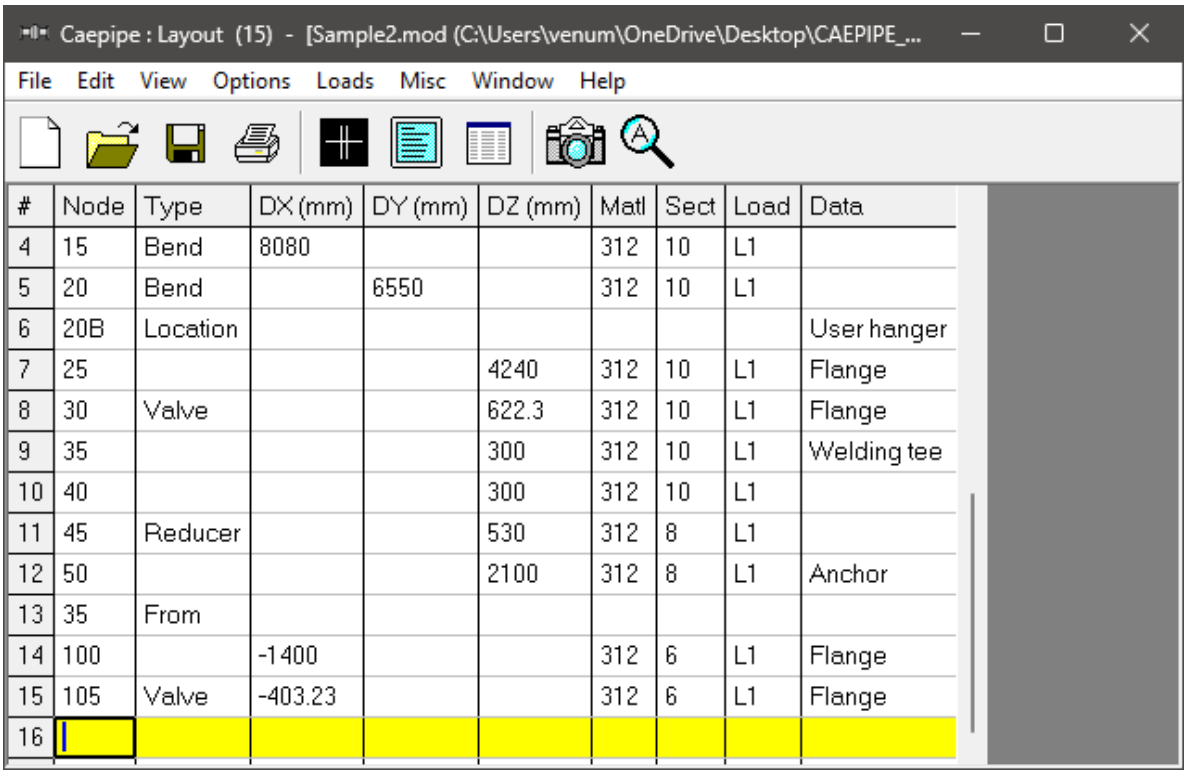


In the next row (#15), Tab to the Type column. The next Node 105 is automatically assigned. In the Type column, type 'v' (for Valve). This brings up the Valve dialog box. In the Valve dialog box, type 151.56 for Weight, 403.23 for Length, 3.00 for Thickness, and 1.75 for Insulation weight. Then press Enter or click on OK to input the valve. Press Enter again.



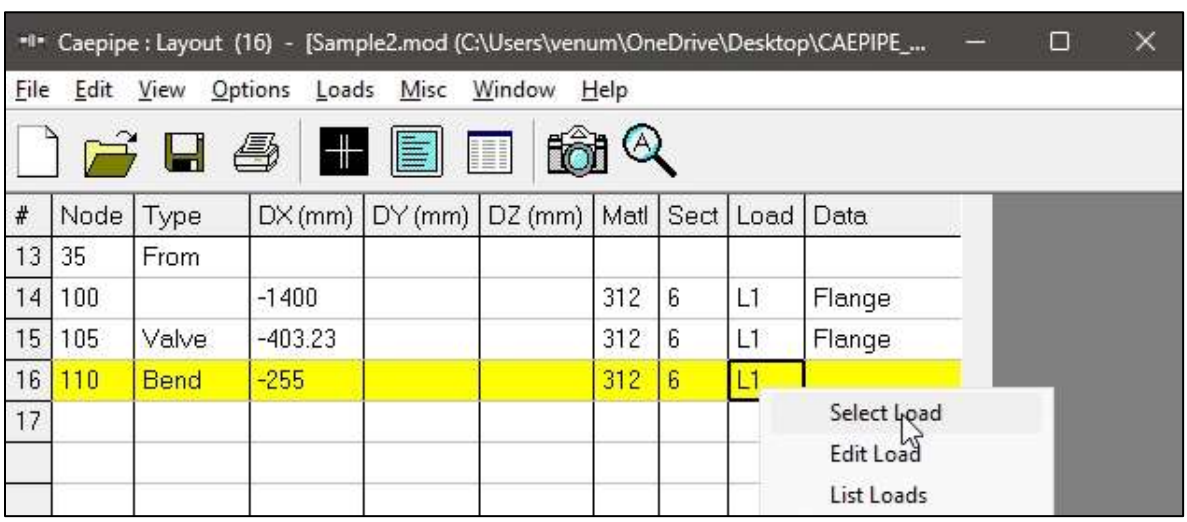
You will see that the DX, Material, Section and Load information is automatically input in the Layout window.

You can now copy the flange along with data from Node 100 and paste it at Node 105. To perform this, highlight row # 14 and press Ctrl+C. Then move the cursor to Data column of row #15 and press Ctrl+V to paste the flange. Press Enter to move to the next row.



#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
4	15	Bend	8080			312	10	L1	
5	20	Bend		6550		312	10	L1	
6	20B	Location							User hanger
7	25				4240	312	10	L1	Flange
8	30	Valve			622.3	312	10	L1	Flange
9	35				300	312	10	L1	Welding tee
10	40				300	312	10	L1	
11	45	Reducer			530	312	8	L1	
12	50				2100	312	8	L1	Anchor
13	35	From							
14	100		-1400			312	6	L1	Flange
15	105	Valve	-403.23			312	6	L1	Flange
16									

In the next row (#16), Tab to the Type column, type "ben" to create a Long Radius Bend and then Tab to the DX column. The default LR Bend is automatically input when you Tab over. In the DX column type –255 and hit Enter. The Material, Section and Load information and is automatically input. As the Temperature and Pressure is changing from this element, change the Load from L1 to L2 by right clicking on the "L1" in the Load field.



#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
13	35	From							
14	100		-1400			312	6	L1	Flange
15	105	Valve	-403.23			312	6	L1	Flange
16	110	Bend	-255			312	6	L1	
17									

This will bring up a small Context menu from which you will choose Select Load. This will bring up the Select Load window. Highlight L2 and click Ok. Press Enter to complete inputting Node 110 at row (#16).

Select Load

Name	T1 (C)	P1 (bar)	Specific gravity	Addialog.Wgt. (kg/m)	Wind Load
L1	185	10.0	0.1		
L2	260	32.0	0.1		

OK Cancel

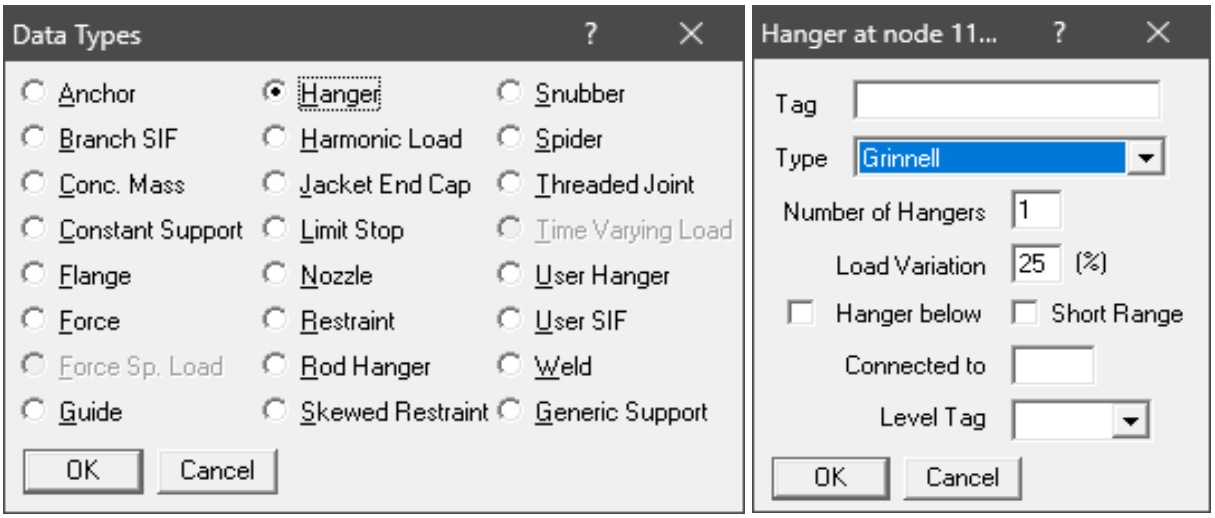
In the next row (#17), create another Long Radius Bend just like the one in row (#17), except change the DX -255 to DY 2950 and press Enter. Your layout window should look like this.

Caepipe : Layout (17) - [Sample2.mod (C:\Users\venum\OneDrive\Desktop\CAEPIPE_...

File Edit View Options Loads Misc Window Help

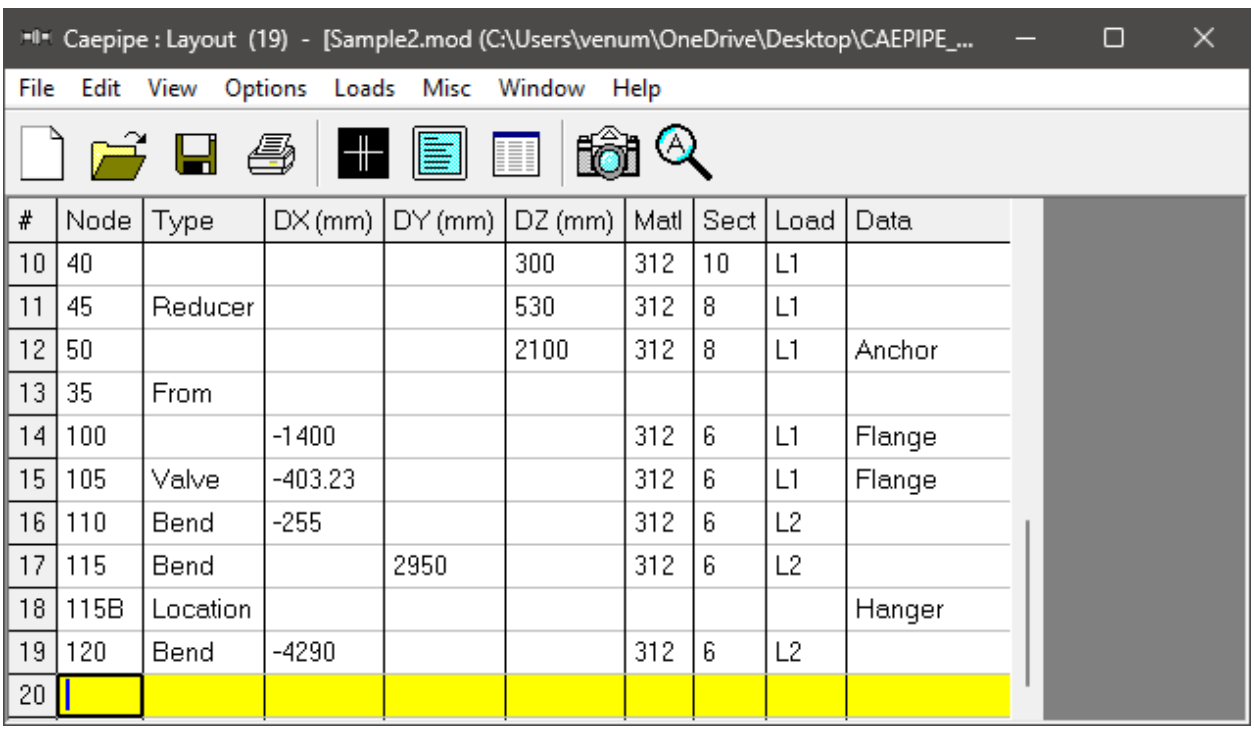
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
8	30	Valve			622.3	312	10	L1	Flange
9	35				300	312	10	L1	Welding tee
10	40				300	312	10	L1	
11	45	Reducer			530	312	8	L1	
12	50				2100	312	8	L1	Anchor
13	35	From							
14	100		-1400			312	6	L1	Flange
15	105	Valve	-403.23			312	6	L1	Flange
16	110	Bend	-255			312	6	L2	
17	115	Bend		2950		312	6	L2	
18									

Start the next row (#18) by typing 115B in the Node column. Tab to the Type column and type "L" to specify a Location type. This will automatically open the Data Types dialog box. Select Hanger.



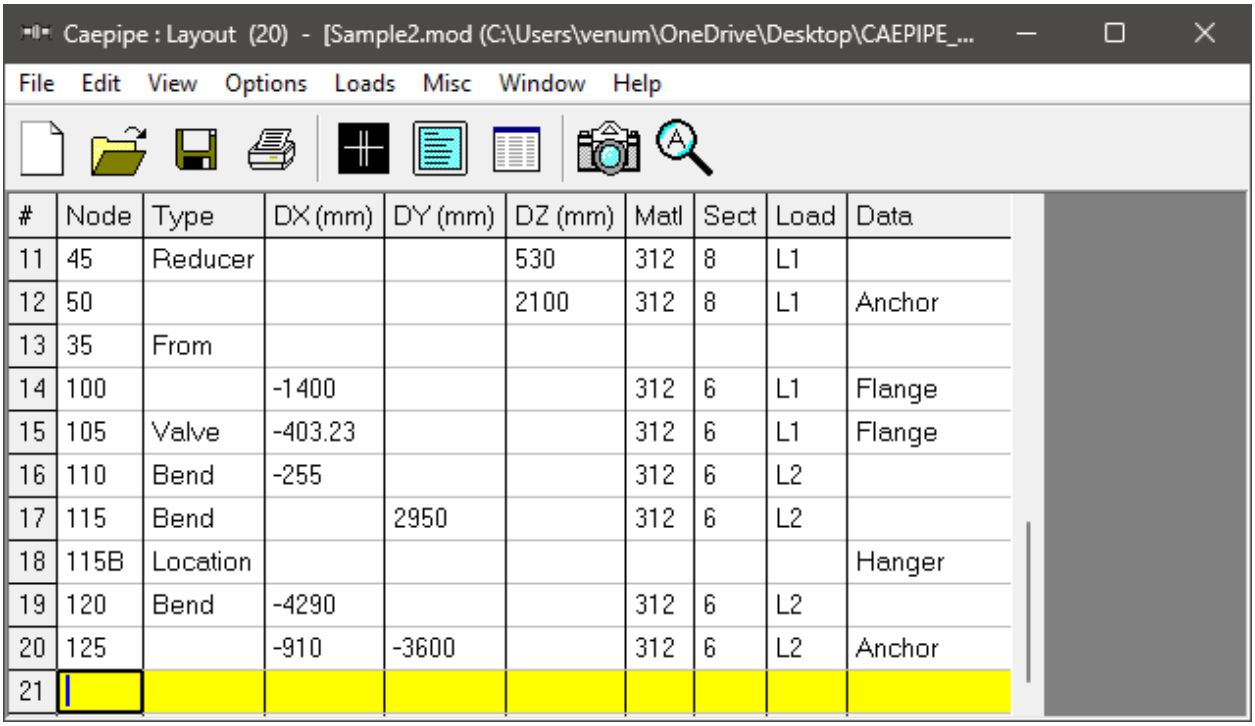
Another dialog box will appear with specific Hanger type input options. Keep the default settings and click OK.

At Node 120 on the next row (#19), Tab to the Type column and input a default LR Bend by typing "ben". Tab to the DX column and input –4290 and press Enter.



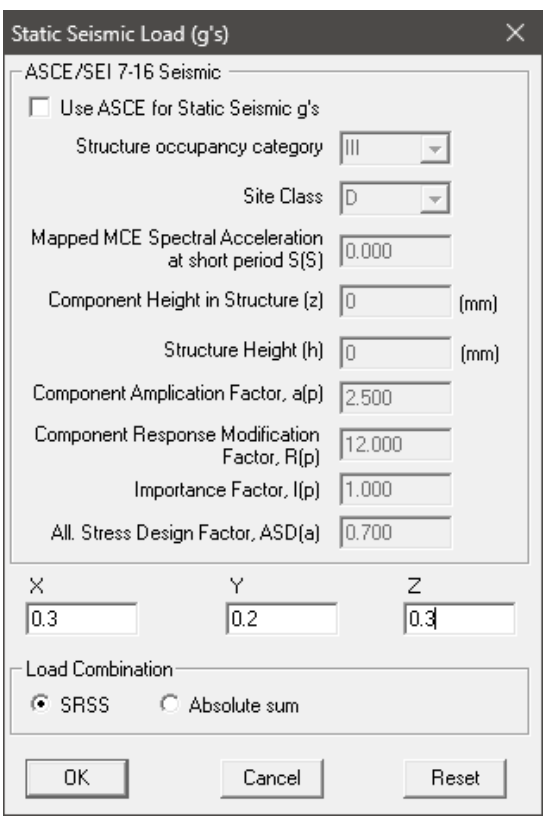
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
10	40				300	312	10	L1	
11	45	Reducer			530	312	8	L1	
12	50				2100	312	8	L1	Anchor
13	35	From							
14	100		-1400			312	6	L1	Flange
15	105	Valve	-403.23			312	6	L1	Flange
16	110	Bend	-255			312	6	L2	
17	115	Bend		2950		312	6	L2	
18	115B	Location							Hanger
19	120	Bend	-4290			312	6	L2	
20									

On the next row (#20), Tab over to the DX column and input -910, then in DY input –3660. Create an Anchor in the Data column by either pressing Ctrl+Shift+D or Tabbing to the Data column and typing "a". Press Enter and you are done with Layout window input.

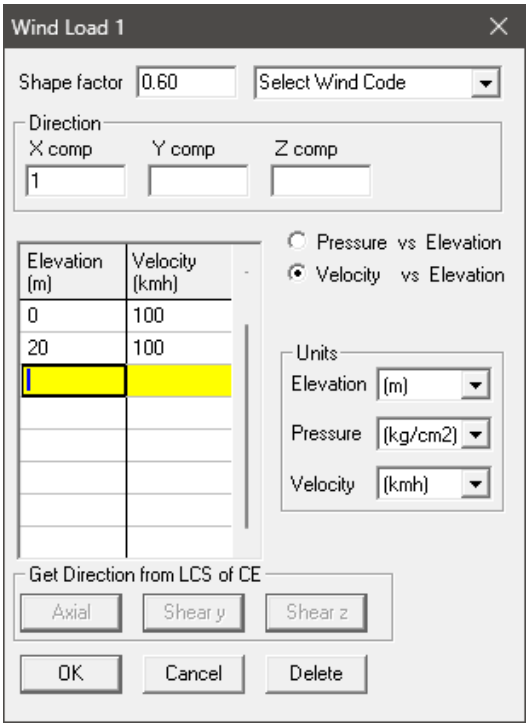


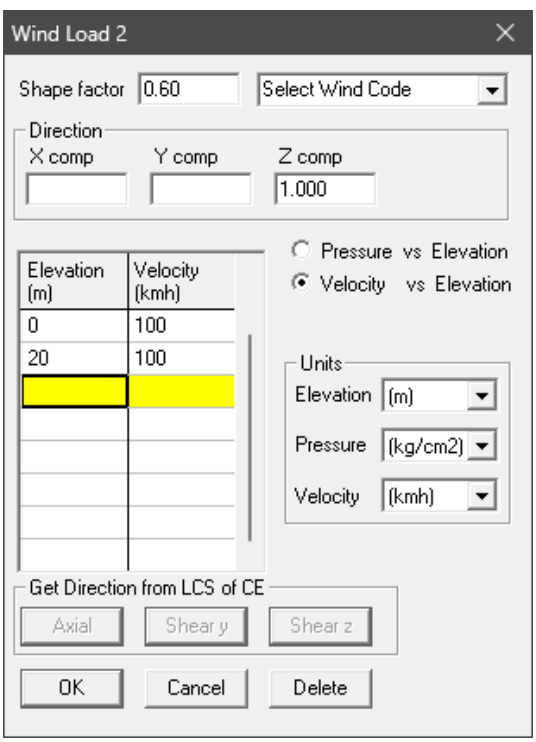
#	Node	Type	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
11	45	Reducer			530	312	8	L1	
12	50				2100	312	8	L1	Anchor
13	35	From							
14	100		-1400			312	6	L1	Flange
15	105	Valve	-403.23			312	6	L1	Flange
16	110	Bend	-255			312	6	L2	
17	115	Bend		2950		312	6	L2	
18	115B	Location							Hanger
19	120	Bend	-4290			312	6	L2	
20	125		-910	-3600		312	6	L2	Anchor
21									

Define "Static seismic" through Layout Window > Loads > Static Seismic. Enter the value as shown below.



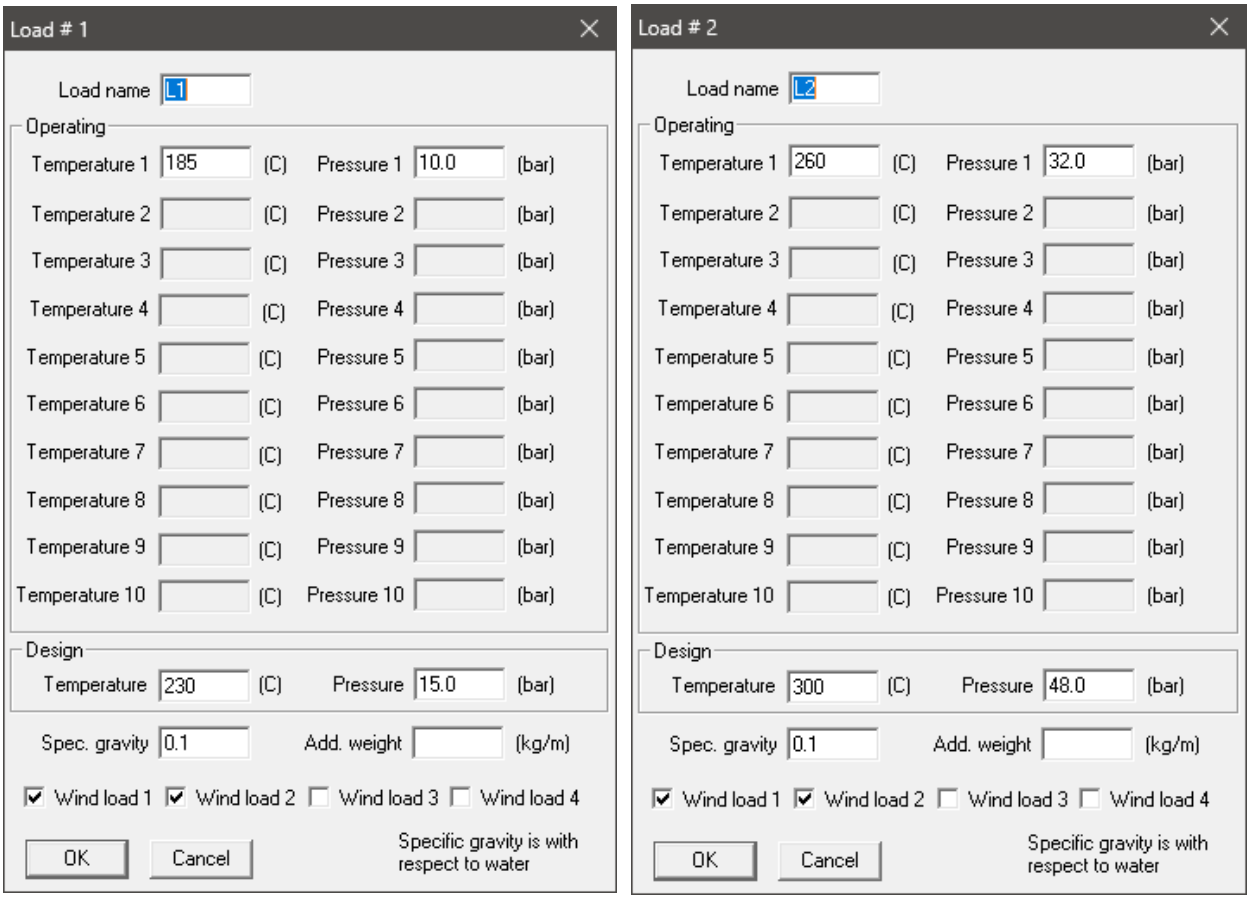
Let us define "Wind Load" profile in +X direction through Layout Window > Loads > Wind 1 and enter the data as shown below and press OK. The maximum elevation of 20m is chosen so that the entire piping system experiences wind load.





Similarly, define "Wind Load" profile in +Z direction through Layout Window > Loads > Wind 2 and enter the data as shown below and press OK.

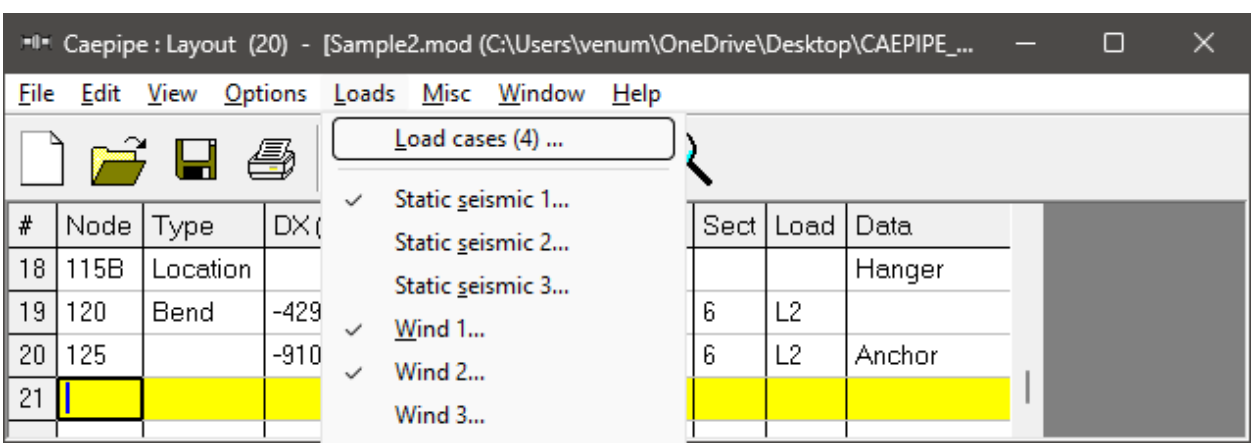
Assign the Wind Load defined above to the stress layout through Layout window > Misc > Loads and then double click on the Loads "L1" and select the check box "Wind load 1" and "Wind load 2" as shown below.



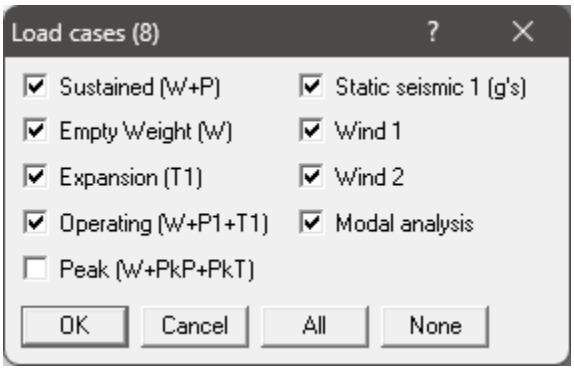
Similarly, select the check boxes "Wind load 1" and "Wind load 2" for "L2".

5. Select Load Cases for Analysis

Select Loads cases from the Loads menu.



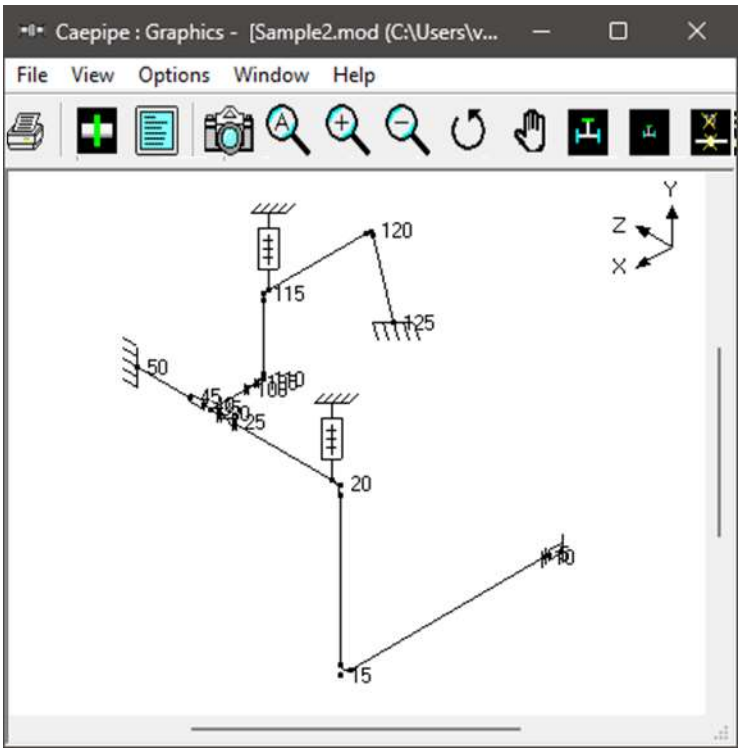
The Load cases dialog is shown.

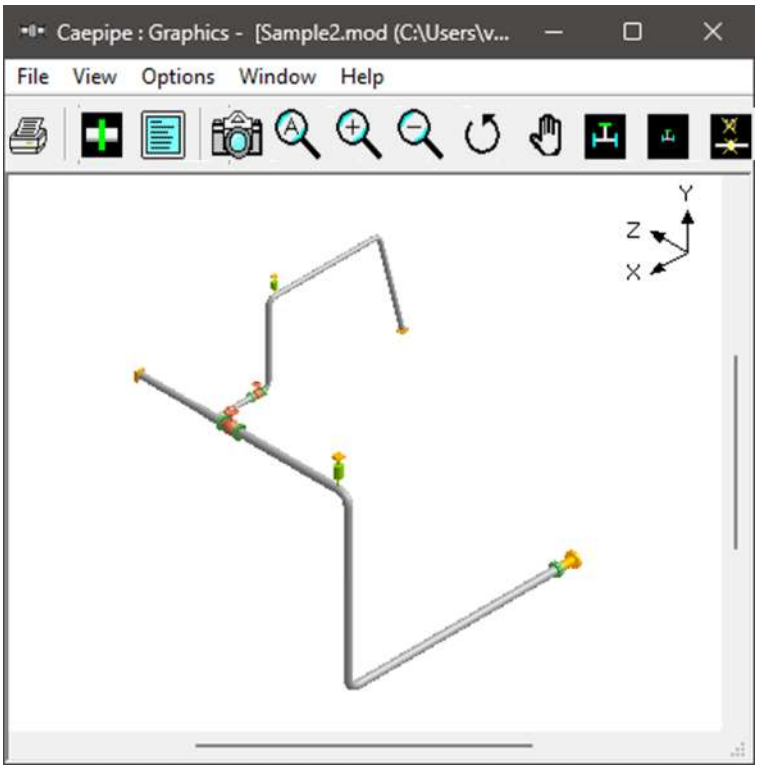


By default, Sustained (W+P), Empty Weight (W), Expansion (T1) and Operating (W+P1+T1) load cases are already selected. Add Static Seismic (g's), Wind 1, Wind 2, and the Modal analysis Load cases by clicking on the checkbox next to it. Peak (W+PkP+PkT) load cases when selected for the Analysis, CAEPIPE will compute and show results for Displacements, Element Forces & Moments, Support Loads and Support Load Summary. Peak load cases does not include Stress Calculations, Rotating Equipment Qualifications and Flange Equivalent Pressure Calculations. Press OK to return to the Layout window. The model input is now complete.

Click on the Zoom All button (or press Ctrl+A) to show the whole model in the graphics window.

To see a 3D rendered view of the model, click on the Render button (or press Ctrl+R) in the graphics window.





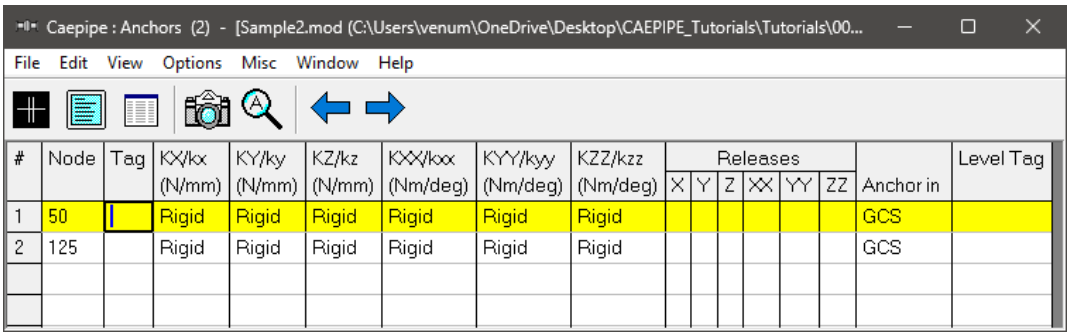
To return to the non-rendered view, click on the Do not render button (or press Ctrl+R).

List

One of the useful features of CAEPIPE is the ability to show a list of all like items such as anchors, bends etc. in a separate List window. Click on the List button (or press Ctrl+L) to show the list dialog.

Click on an item of interest to show the list for that item.

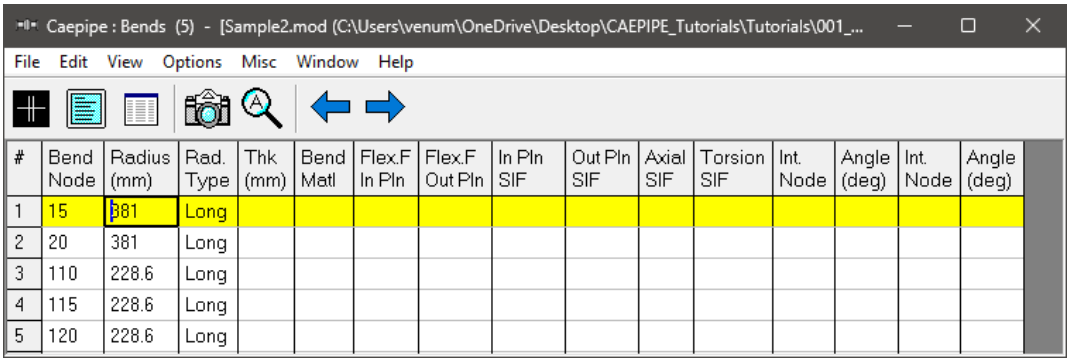
A list of all the anchors present in this sample model is shown below:



#	Node	Tag	KX/kx (N/mm)	KY/ky (N/mm)	KZ/kz (N/mm)	KXX/kxx (Nm/deg)	KYY/kyy (Nm/deg)	KZZ/kzz (Nm/deg)	Releases X	Y	Z	XX	YY	ZZ	Anchor in	Level Tag
1	50		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
2	125		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	

The highlighted item can be edited directly in the List window (in most cases) or in a dialog by pressing Ctrl+E. The items can be deleted by pressing Ctrl+X. The item is also highlighted in the graphics window by flashing and with a box around the node number.

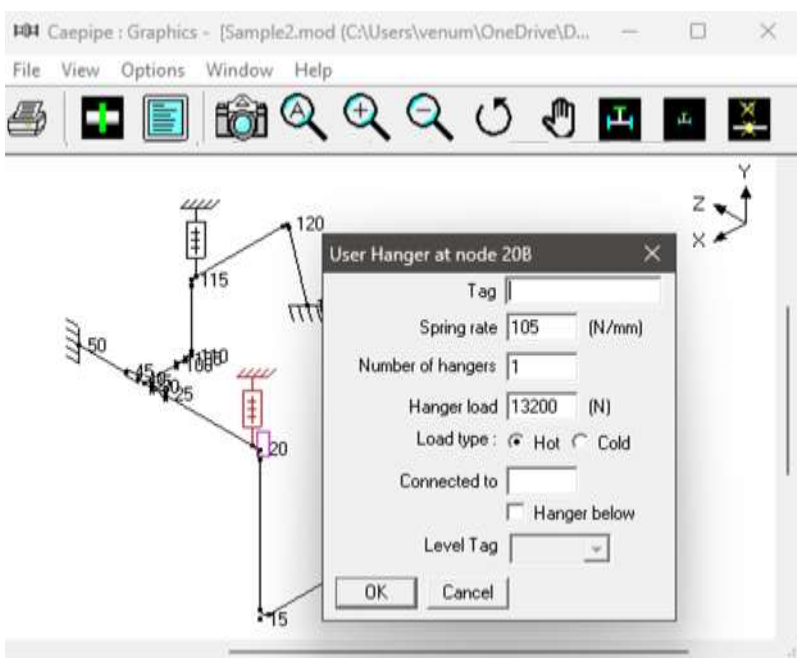
A list of all the bends in the sample model is shown below:



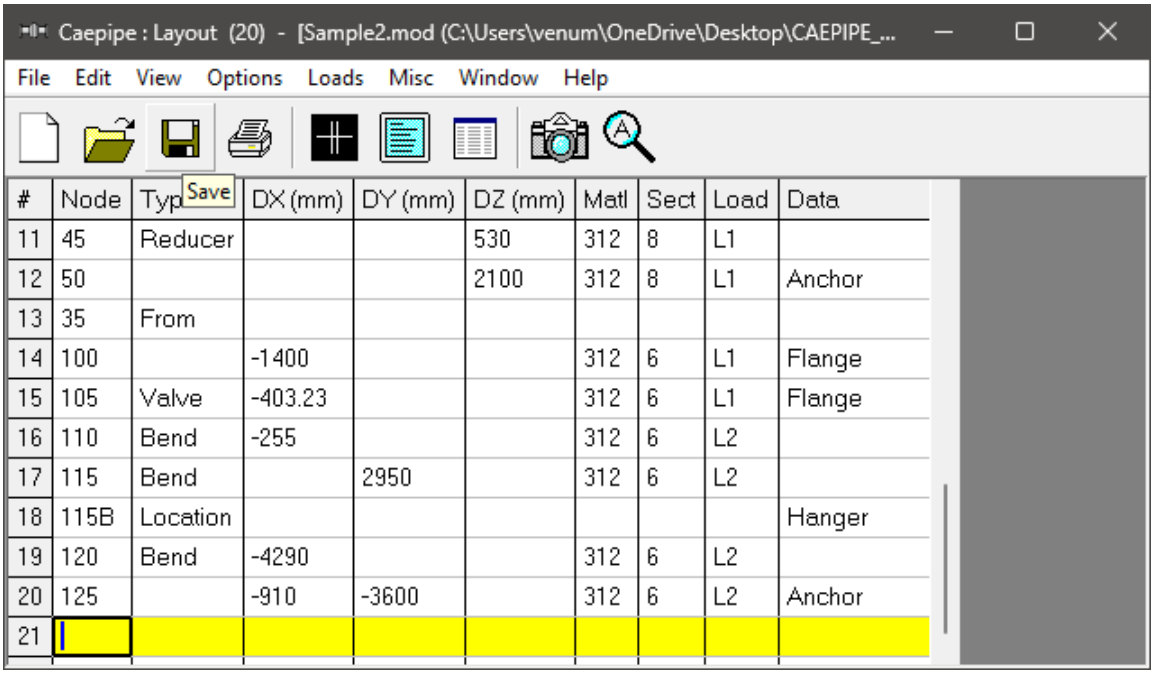
#	Bend Node	Radius (mm)	Rad. Type	Thk (mm)	Bend Matl	Flex.F In Pln	Flex.F Out Pln	In Pln SIF	Out Pln SIF	Axial SIF	Torsion SIF	Int. Node	Angle (deg)	Int. Node	Angle (deg)
1	15	381	Long												
2	20	381	Long												
3	110	228.6	Long												
4	115	228.6	Long												
5	120	228.6	Long												

Editing in the Graphics Window

Another useful feature is the ability to edit an item in the graphics window. When an item such as a Hanger is clicked in the graphics window, a dialog box for that item is opened, where it can be modified.



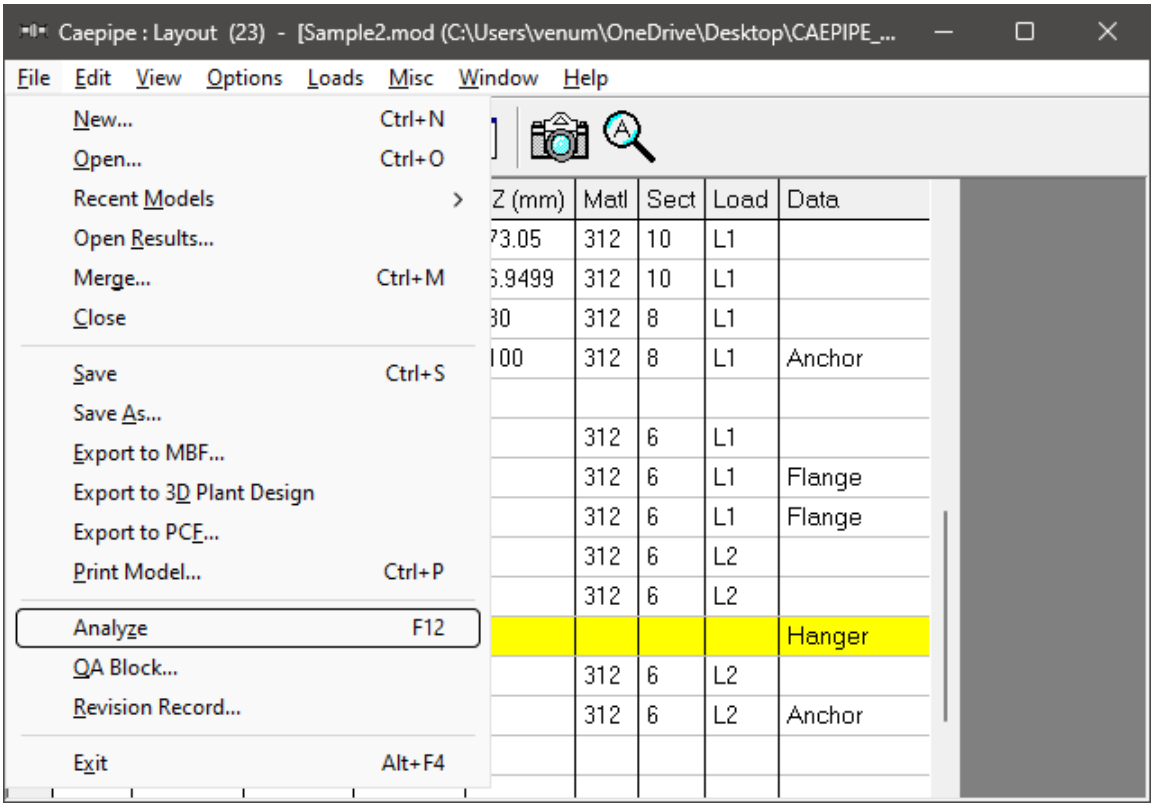
Save the model by clicking on the Save button.



#	Node	Typ	DX (mm)	DY (mm)	DZ (mm)	Matl	Sect	Load	Data
11	45	Reducer			530	312	8	L1	
12	50				2100	312	8	L1	Anchor
13	35	From							
14	100		-1400			312	6	L1	Flange
15	105	Valve	-403.23			312	6	L1	Flange
16	110	Bend	-255			312	6	L2	
17	115	Bend		2950		312	6	L2	
18	115B	Location							Hanger
19	120	Bend	-4290			312	6	L2	
20	125		-910	-3600		312	6	L2	Anchor
21									

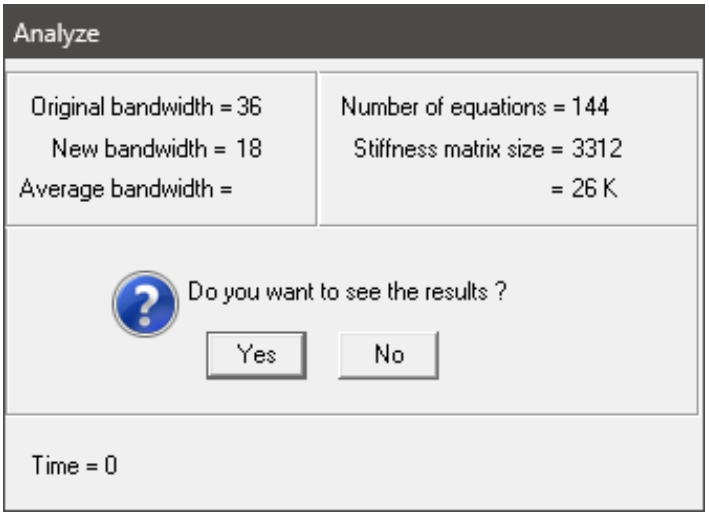
6. Analyze

Click on Analyze under the File menu.



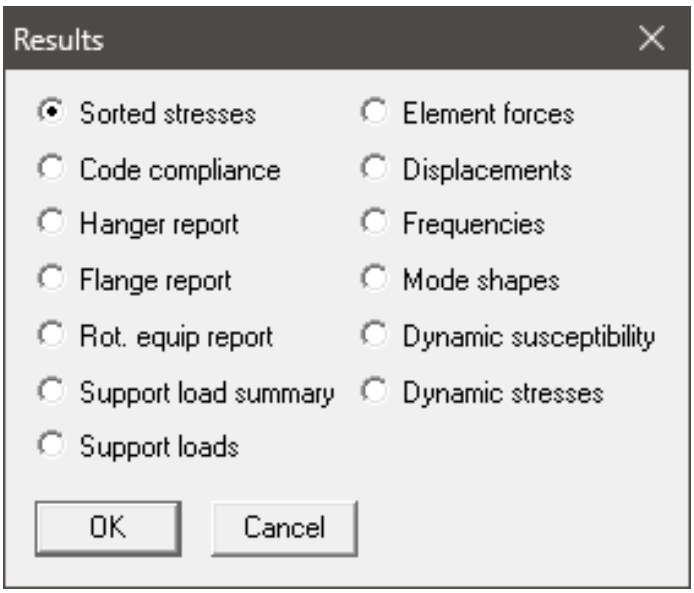
Z (mm)	Matl	Sect	Load	Data
73.05	312	10	L1	
6.9499	312	10	L1	
30	312	8	L1	
100	312	8	L1	Anchor
	312	6	L1	
	312	6	L1	Flange
	312	6	L1	Flange
	312	6	L2	
	312	6	L2	
				Hanger
	312	6	L2	
	312	6	L2	Anchor

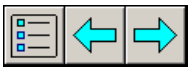
After the analysis, you are asked if you want to see the results. Select Yes.



7. View Results

After finishing the analysis and choosing to see the results or by opening the results file (.res), the results window is displayed. The Results dialog is opened automatically.





Select an item of interest by clicking on it. When you are viewing the results, use Tab (or Next Result button) to view the next result and Shift+Tab (or Previous Result button) to view the previous result. The Results dialog can be brought up by clicking on the Results icon (or press Ctrl+R).

While viewing the results, the model data can also be simultaneously viewed in separate Layout and List windows. These are now "read only" windows, i.e. the model data cannot be modified while viewing the results. Some of the results from the sample problem are shown below:

Sorted stresses

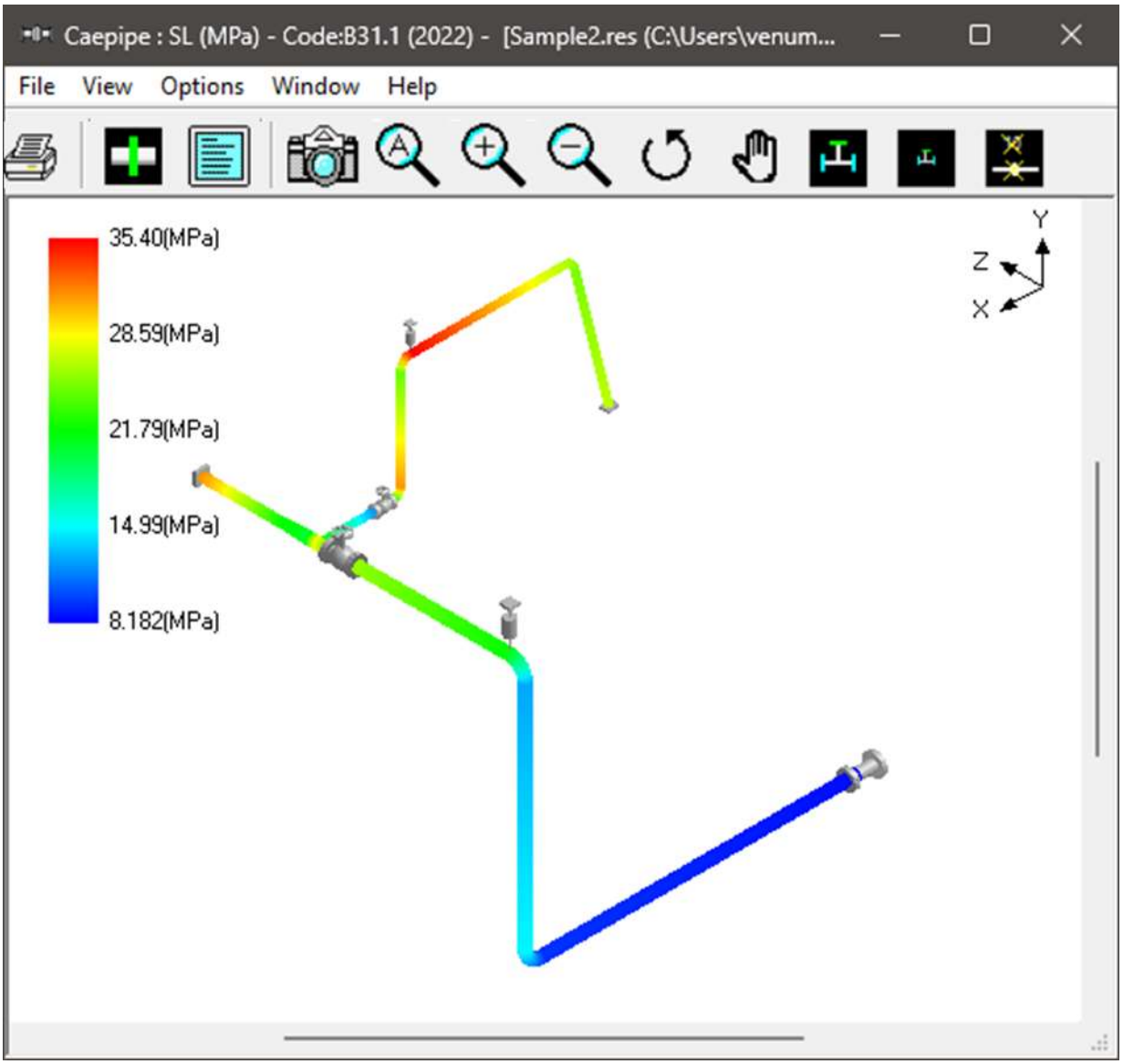
The computed stresses (sustained, expansion and occasional) are sorted in descending order by stress ratios.

Caepipe 3D+ : B31.1 (2024) Code compliance (Sorted stresse...

File Results View Options Window Help

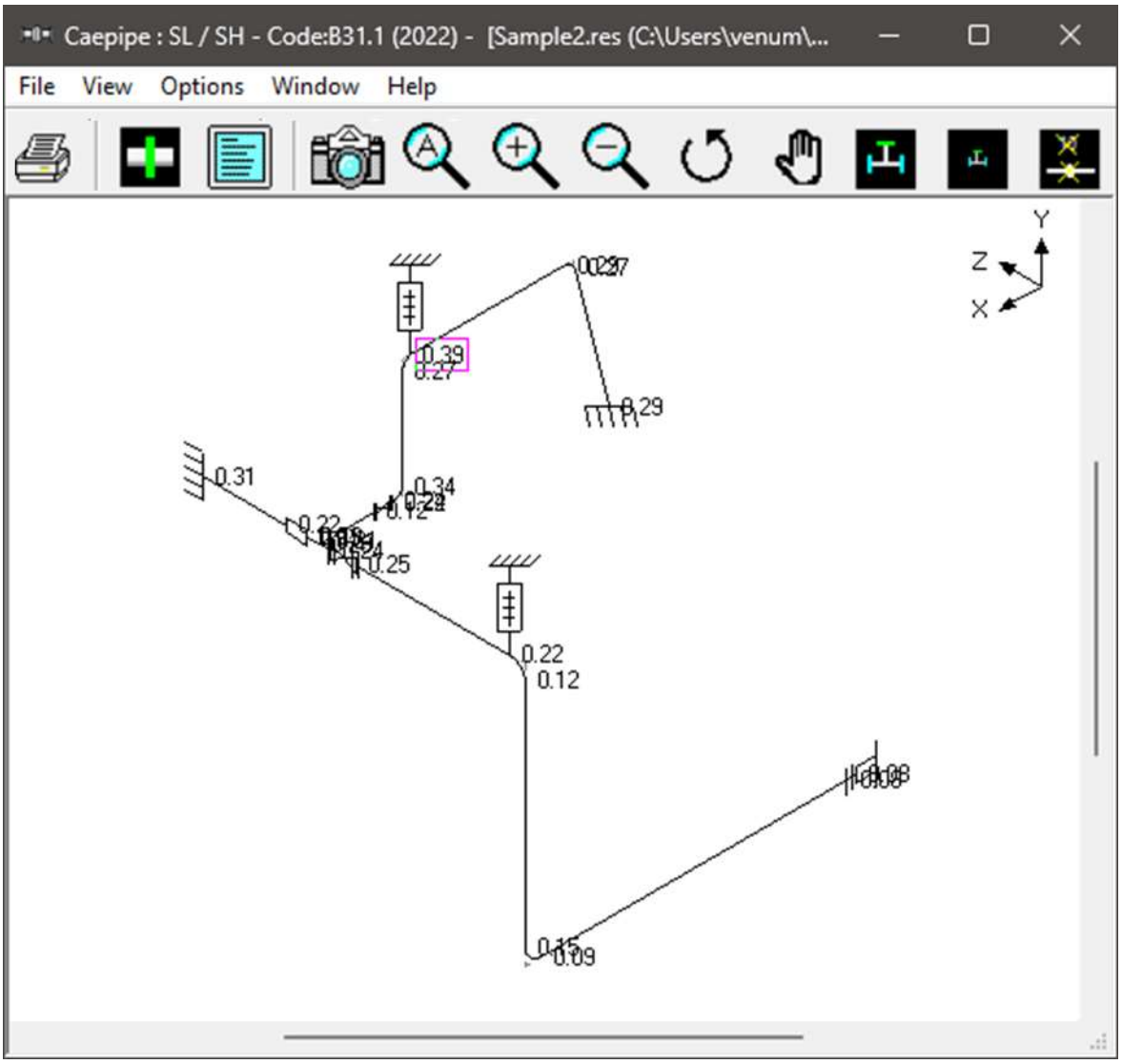
#	Sustained				Expansion				Occasional			
	Node	SL (MPa)	SH (MPa)	SL/SH	Node	SE (MPa)	SA (MPa)	SE/SA	Node	SO (MPa)	1.2SH (MPa)	SO/1.2SH
6	35	28.55	101.7	0.28	120A	33.61	260.5	0.13	120A	39.11	110.0	0.36
7	120B	25.12	91.70	0.27	50	34.29	267.7	0.13	20B	41.69	122.1	0.34
8	115A	24.64	91.70	0.27	35	29.54	271.0	0.11	120B	36.83	110.0	0.33
9	25	25.82	101.7	0.25	120B	27.11	261.9	0.10	45	40.32	122.1	0.33
10	110A	21.86	91.70	0.24	15A	29.96	290.0	0.10	110A	34.34	110.0	0.31
11	30	24.25	101.7	0.24	15B	26.86	284.7	0.09	65	35.03	122.1	0.29
12	55	24.02	101.7	0.24	105	24.17	266.8	0.09	115A	29.92	110.0	0.27
13	105	20.19	91.70	0.22	100	22.41	287.8	0.08	25	33.10	122.1	0.27
14	45	21.99	101.7	0.22	20B	20.32	277.6	0.07	105	29.12	110.0	0.26
15	20B	21.97	101.7	0.22	65	18.19	278.2	0.07	15B	32.20	122.1	0.26
16	65	21.37	101.7	0.21	45	16.86	277.5	0.06	30	31.59	122.1	0.26
17	40	20.90	101.7	0.21	20A	17.41	286.8	0.06	55	31.39	122.1	0.26
18	60	19.46	101.7	0.19	5	6.254	291.4	0.02	20A	30.61	122.1	0.25
19	15B	14.86	101.7	0.15	55	5.669	275.5	0.02	15A	28.51	122.1	0.23
20	20A	12.70	101.7	0.12	10	5.974	291.0	0.02	40	27.15	122.1	0.22
21	100	11.78	101.7	0.12	30	5.628	275.3	0.02	60	24.98	122.1	0.20
22	15A	9.538	101.7	0.09	25	4.866	273.7	0.02	100	20.96	122.1	0.17
23	10	8.529	101.7	0.08	40	4.933	278.6	0.02	5	17.36	122.1	0.14

Color coded stresses may be rendered in the graphics window by pressing the Show stresses button (or choose View > Show Stresses). The stresses in the highlighted columns (the bar highlights three columns simultaneously) are displayed in the graphics window. Use the left and right arrow keys to change the highlighted columns or click in a particular column.



The stress ratios may similarly be rendered by using the Show stress ratios button (or choose View > Show Stress Ratios).

Instead of rendering color coded stresses/stress ratios, the values of stresses/stress ratios may be plotted by using the menu: View > No color coding and pressing the icon S or S/A.



While plotting stresses or stress ratios, thresholds may be specified from the graphics window (choose View > Thresholds). Only those stresses or stress ratios exceeding the threshold are plotted.

Code compliance

Element stresses and stress ratios calculated according to the selected piping code are shown under Code compliance. Design pressure and CAEPIPE computed Allowable pressure are shown in 2nd column.

Caepipe 3D+ : B31.1 (2024) Code Compliance - [sample2.res...

File Results View Options Window Help

#	Node	Press. Allow. (bar)	Sustained SL (MPa)	Sustained SH (MPa)	Sustained SL/SH	Expansion SE (MPa)	Expansion SA (MPa)	Expansion SE/SA	Occasional SO (MPa)	Occasional 1.2SH (MPa)	Occasional SO/1.2SH
1	5	15.0	8.182	101.7	0.08	6.254	291.4	0.02	17.36	122.1	0.14
	10	66.6	8.529	101.7	0.08	5.974	291.0	0.02	16.74	122.1	0.14
2	10	15.0	8.529	101.7	0.08	5.974	291.0	0.02	16.71	122.1	0.14
	15A	66.6	8.493	101.7	0.08	12.66	291.0	0.04	19.28	122.1	0.16
3	15A	15.0	9.538	101.7	0.09	29.96	290.0	0.10	28.51	122.1	0.23
	15B	66.6	14.86	101.7	0.15	26.86	284.7	0.09	32.20	122.1	0.26
4	15B	15.0	11.44	101.7	0.11	12.51	288.1	0.04	21.06	122.1	0.17
	20A	66.6	10.98	101.7	0.11	9.751	288.6	0.03	20.84	122.1	0.17
5	20A	15.0	12.70	101.7	0.12	17.41	286.8	0.06	30.61	122.1	0.25
	20B	66.6	21.97	101.7	0.22	20.32	277.6	0.07	41.69	122.1	0.34
6	20B	15.0	14.96	101.7	0.15	9.512	284.6	0.03	25.96	122.1	0.21
	25	66.6	25.82	101.7	0.25	4.866	273.7	0.02	33.10	122.1	0.27
7	30	15.0	24.25	101.7	0.24	5.628	275.3	0.02	31.59	122.1	0.26
	55	66.6	24.02	101.7	0.24	5.669	275.5	0.02	31.39	122.1	0.26
8	55	15.0	24.02	101.7	0.24	5.669	275.5	0.02	31.39	122.1	0.26
	35	66.6	21.65	101.7	0.21	10.42	277.9	0.04	30.52	122.1	0.25
9	35	15.0	21.59	101.7	0.21	2.724	277.9	0.01	26.33	122.1	0.22
	60	66.6	19.46	101.7	0.19	3.196	280.1	0.01	24.98	122.1	0.20

Hanger report

The hanger report is shown below.

Caepipe 3D+ : Hanger Report - [sample2.res (c:\users\ven...

File Results View Options Window Help

#	Node	No of	Type	Figure No.	Size	Spring rate (N/mm)	Vert travel (mm)	Horz travel (mm)	Vert Angle (deg)	Hot load (N)	Cold load (N)	Var (%)
1	20B	1	User hanger			105	2.153	25.006	-	13200	13426	1
2	115B	1	Grinnell	B-268	10	45.533	16.882	15.847	-	5115	5884	15

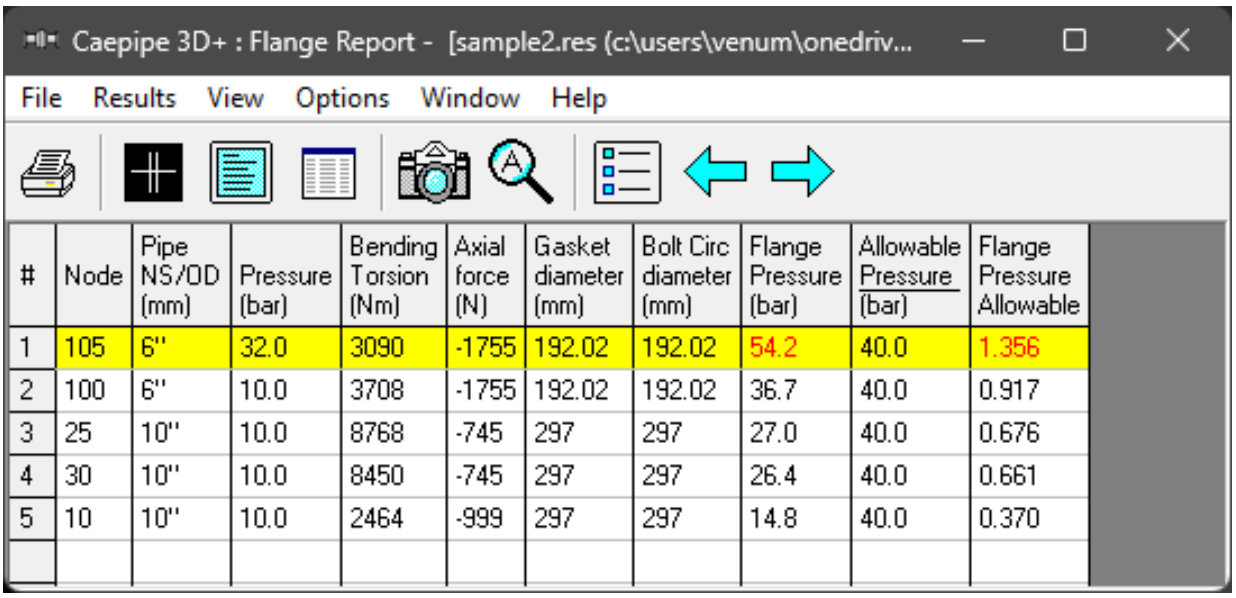
The "No of" field shows the number of hangers required at the indicated location. The Figure No. and Size refer to the manufacturer's catalog. The vertical travel (also referred to as "Hanger travel") is the vertical deflection at the hanger location for the first operating load case. Similarly, the horizontal travel is the resultant horizontal deflection at the hanger location for the first operating case. The hot load is the hanger load for the operating condition and the cold load is the hanger load at zero deflection.

Variability (%) = (Spring rate × Hanger travel / Hot load) × 100

Flange report

The Flange report in the CAEPIPE results window shows the loads at each flange location for the operating case (W+P1+T1).

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 for the first operating case (W+P1+T1).



#	Node	Pipe NS/OD (mm)	Pressure (bar)	Bending Torsion (Nm)	Axial force (N)	Gasket diameter (mm)	Bolt Circ diameter (mm)	Flange Pressure (bar)	Allowable Pressure (bar)	Flange Pressure Allowable
1	105	6"	32.0	3090	-1755	192.02	192.02	54.2	40.0	1.356
2	100	6"	10.0	3708	-1755	192.02	192.02	36.7	40.0	0.917
3	25	10"	10.0	8768	-745	297	297	27.0	40.0	0.676
4	30	10"	10.0	8450	-745	297	297	26.4	40.0	0.661
5	10	10"	10.0	2464	-999	297	297	14.8	40.0	0.370

The last column shows a ratio of this "equivalent" Flange Pressure to a user-input Allowable Pressure. This ratio is flagged in red when it exceeds 1.0.

Support load summary

Support load summary for each support is created by considering all the load cases and appropriate combinations and then showing the maximum and minimum loads.

Note: Allowable loads at an equipment nozzle can be calculated using the module "Nozzle Evaluation" available in CAEPIPE through Main Frame > New > Nozzle Evaluation.

The allowable loads thus calculated can then be entered as "User Allowables" in CAEPIPE Stress Model through Layout window > Misc. See the CAEPIPE tutorial titled "*Tutorial on Qualification of Nozzles to Equipment using CAEPIPE*" for more details.

Caepipe 3D+ : Support load summary for anchor at node 50 - [sample2.res (c:\users\venum\one...

File Results View Options Window Help

Load combination	FX (N)	FY (N)	FZ (N)	MX (Nm)	MY (Nm)	MZ (Nm)	FR (N)	MR (Nm)	Displacements (global) X (mm)	Y (mm)	Z (mm)
Empty W	-360	-4992	25	-4871	1556	-1057	5005	5222	0.000	0.000	0.000
Sustained	-330	-5482	7	-6729	1287	-1106	5491	6940	0.000	0.000	0.000
Operating1	2754	-5675	1078	-6457	-8087	-615	6399	10367	0.000	0.000	0.000
Sustained+Wind	878	-5602	151	-7396	-2594	-1786	5673	8039	0.000	0.000	0.000
Operating1+Wind	3962	-5795	1222	-7124	-11969	-1295	7126	13989	0.000	0.000	0.000
Sustained+Wind 2	-314	-5252	1816	-6761	1271	-1222	5566	6987	0.000	0.000	0.000
Operating1+Wind 2	2770	-5445	2887	-6489	-8104	-731	6757	10407	0.000	0.000	0.000
Sustained+Seismic 1	3544	-2665	6506	1726	12102	456	7873	12233	0.000	0.000	0.000
Sustained-Seismic 1	-4203	-8298	-6492	-15184	-9527	-2669	11343	18123	0.000	0.000	0.000
Operating1+Seismic 1	6628	-2858	7577	1998	2727	947	10464	3511	0.000	0.000	0.000
Operating1-Seismic 1	-1119	-8491	-5421	-14912	-18902	-2177	10136	24174	0.000	0.000	0.000
Maximum	6628	-2665	7577	1998	12102	947	11343	24174	0.000	0.000	0.000
Minimum	-4203	-8491	-6492	-15184	-18902	-2669	5005	3511	0.000	0.000	0.000
Allowables	0	0	0	0	0	0	0	0	0.000	0.000	0.000

Use the Other supports button (F6), Next support button (Ctrl+Right arrow) or Previous support button (Ctrl+Left arrow) to see loads on other supports (e.g. other anchors, hangers etc.).

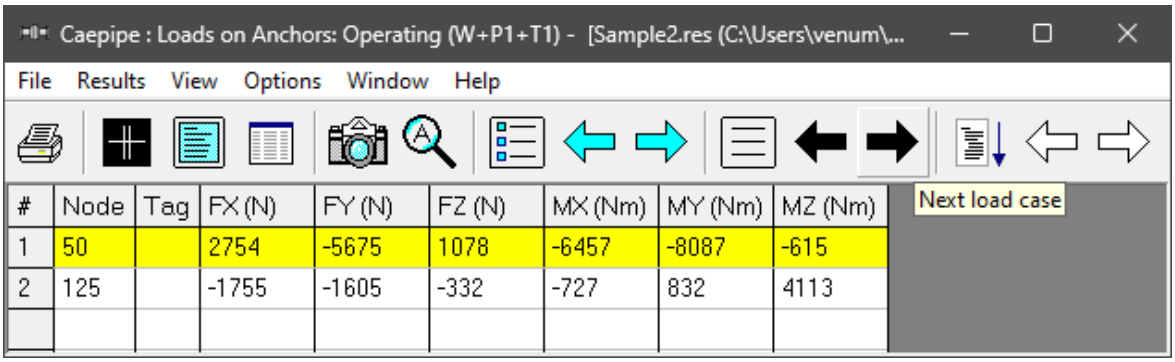
Support load summary

Node	Type
50	Anchor
125	Anchor
20B	User hanger
115B	Hanger
5	Nozzle

OK Cancel

Support loads

Support loads are the loads acting on all the supports of each support type for a specific loading case. The loads on anchors for the Operating load case are shown below.

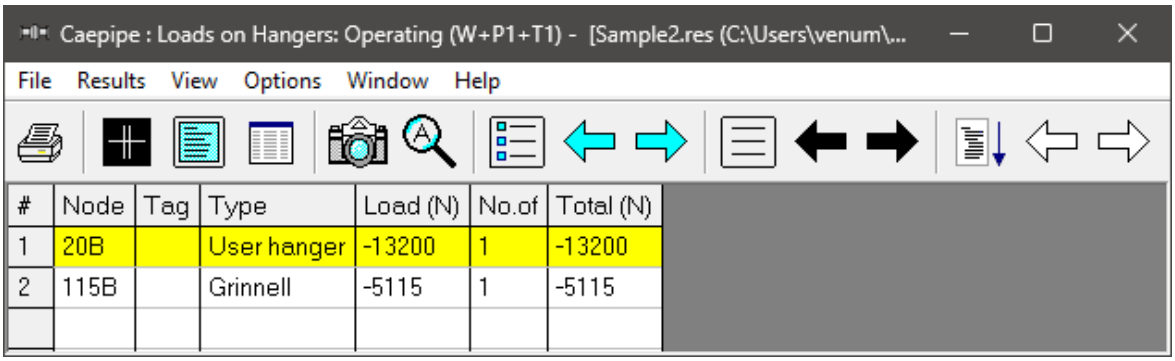


#	Node	Tag	FX (N)	FY (N)	FZ (N)	MX (Nm)	MY (Nm)	MZ (Nm)
1	50		2754	-5675	1078	-6457	-8087	-615
2	125		-1755	-1605	-332	-727	832	4113

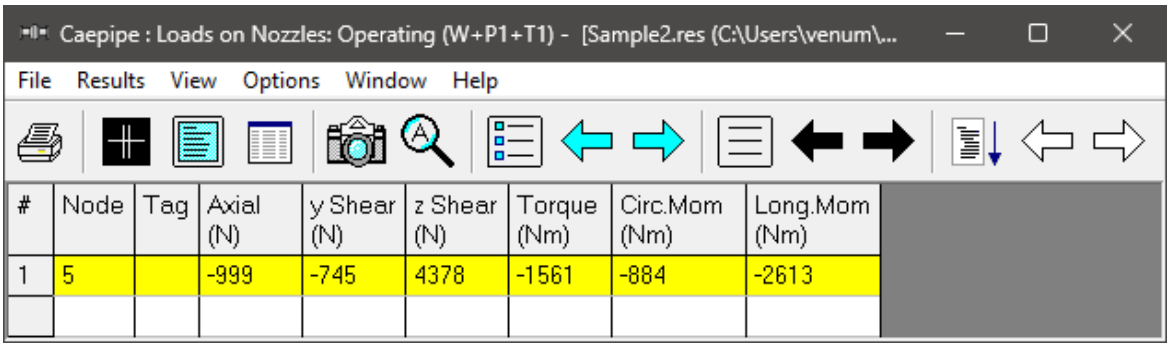
Use the Load cases button, Next load case button (Right arrow) or Previous load case button (Left arrow) to see loads for different load cases (e.g. Sustained, Expansion etc.).

Use the Other supports button (F6), Next support button (Ctrl+Right arrow) or Previous support button (Ctrl+Left arrow) to see loads on supports of different types (e.g. other anchors, hangers etc.).

The loads on hangers (i.e. the loads acting at the hanger locations imposed by the piping system) and the loads on the nozzle for the Operating case are shown below.



#	Node	Tag	Type	Load (N)	No.of	Total (N)
1	20B		User hanger	-13200	1	-13200
2	115B		Grinnell	-5115	1	-5115



#	Node	Tag	Axial (N)	y Shear (N)	z Shear (N)	Torque (Nm)	Circ.Mom (Nm)	Long.Mom (Nm)
1	5		-999	-745	4378	-1561	-884	-2613

Element forces

For pipe (also bend and reducer), element forces in local coordinates, Stress Intensification Factors (SIF) and stresses are shown by default for the selected load case.

Use the Local forces button (F7) to see the element forces in local coordinates again.

Caepipe : Pipe forces in local coordinates: Operating (W+P1+T1) - [Sample2.res (C:\Users\venum\OneDrive\Deskt...

File Results View Options Window Help

Global Forces

#	Node	Axial (N)	y Shear (N)	z Shear (N)	Torsion(Nm) Moment	Torsion(Nm) SIF	Inplane(Nm) Moment	Inplane(Nm) SIF	Outplane(Nm) Moment	Outplane(Nm) SIF	Flex. Factors FFi	Flex. Factors FFo	Flex. Factors FFt	Sopr (MPa)
1	5	-999	-4378	-745	-1561		-884		2613					13.70
	10	-999	-4224	-745	-1561	1.00	-24	1.00	2464	1.00				13.19
2	10	-999	-3539	-745	-1561	1.00	-24	1.00	2464	1.00				13.19
	15A	-999	2419	-745	-1561		4288		-3276					18.70
3	15A	-999	2419	-745	-1561	1.00	4288	2.61	-3276	2.17	8.13	8.13		34.44
	15B	2882	999	-745	-3560	1.00	2922	2.61	1277	2.17	8.13	8.13		25.79
4	15B	2882	999	-745	-3560		2922		1277					17.49
	20A	7362	999	-745	-3560		-2861		-3038					19.51
5	20A	7362	-745	-999	-3560	1.00	3038	2.61	-2861	2.17	8.13	8.13		30.10
	20B	-745	-7825	-999	-3241	1.00	6239	2.61	3179	2.17	8.13	8.13		43.90
6	20B	-745	-5375	999	-3241		-6239		-3179					23.01
	25	-745	-2389	999	-3241	1.00	8741	1.00	676	1.00				26.34
7	30	-745	3646	999	-3241	1.00	8350	1.00	1298	1.00				25.74
	55	-745	3667	999	-3241		8251		1325					25.56
8	55	-745	3667	999	-3241		8251		1325					25.56
	35	-745	3879	999	-3241	1.39	-1598	2.00	7221	1.00				25.82
9	35	-1078	3919	2754	615	1.39	-18	2.00	7732	1.00				23.10
	60	-1078	4130	2754	615		6633		770					20.91
10	60	-1078	4130	2754	615		6633		770					20.91
	40	-1078	4151	2754	615		6522		844					20.70
11	40	-1078	4151	2754	615	1.00	6522	1.52	844	1.52				27.70
	45	-1078	4502	2754	615	1.00	4228	1.52	2304	1.52				33.21
12	45	-1078	4502	2754	615		4228		2304					24.12
	50	-1078	5675	2754	615		-6457		8087					44.14
13	35	-1755	-40	-332	-511	1.07	1580	1.47	-3856	1.86		2.44		59.82
	65	-1755	12	-332	-511		3858		1535			2.44		35.48
14	65	-1755	12	-332	-511		3858		1535					35.48
	100	-1755	497	-332	-511	1.00	3537	1.00	1115	1.00				32.32
15	105	-1755	2512	-332	-511	1.00	2930	1.00	981	1.00				40.94
	110A	-1755	2522	-332	-511		2864		972					40.47

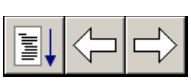
Use the Global forces button (F7) to see the element forces in global coordinates.

Caepipe : Pipe forces in global coordinates: Operating (W+P1+T1) - [Sample2.res (C:\Users\venum\OneDri...

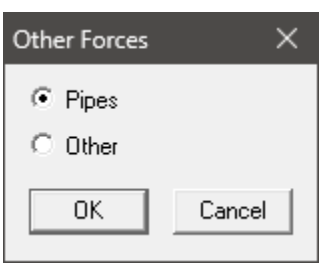
File Results View Options Window Help

Local Forces

#	Node	FX (N)	FY (N)	FZ (N)	MX (Nm)	MY (Nm)	MZ (Nm)
1	5 10	999 -999	4378 -4224	745 -745	1561 -1561	-2613 2464	884 -24
2	10 15A	999 -999	3539 2419	745 -745	1561 -1561	-2464 -3276	24 4288
3	15A 15B	999 -999	-2419 2882	745 -745	1561 -1277	3276 -3560	-4288 2922
4	15B 20A	999 -999	-2882 7362	745 -745	1277 3038	3560 -3560	-2922 -2861
5	20A 20B	999 -999	-7362 7825	745 -745	-3038 6239	3560 -3179	2861 -3241
6	20B 25	999 -999	5375 -2389	745 -745	-6239 -8741	3179 676	3241 -3241
7	30 55	999 -999	-3646 3667	745 -745	8350 -8251	-1298 1325	3241 -3241
8	55 35	999 -999	-3667 3879	745 -745	8251 -7221	-1325 1598	3241 -3241
9	35 60	2754 -2754	-3919 4130	1078 -1078	7732 -6633	-18 770	-615 615
10	60 40	2754 -2754	-4130 4151	1078 -1078	6633 -6522	-770 844	-615 615
11	40 45	2754 -2754	-4151 4502	1078 -1078	6522 -4228	-844 2304	-615 615
12	45 50	2754 -2754	-4502 5675	1078 -1078	4228 6457	-2304 8087	-615 615
13	35 65	-1755 1755	40 12	-332 332	-511 511	-1580 1535	3856 -3858
14	65 100	-1755 1755	-12 497	-332 332	-511 511	-1535 1115	3858 -3537
15	105 110A	-1755 1755	-2512 2522	-332 332	-511 511	-981 972	2930 -2864

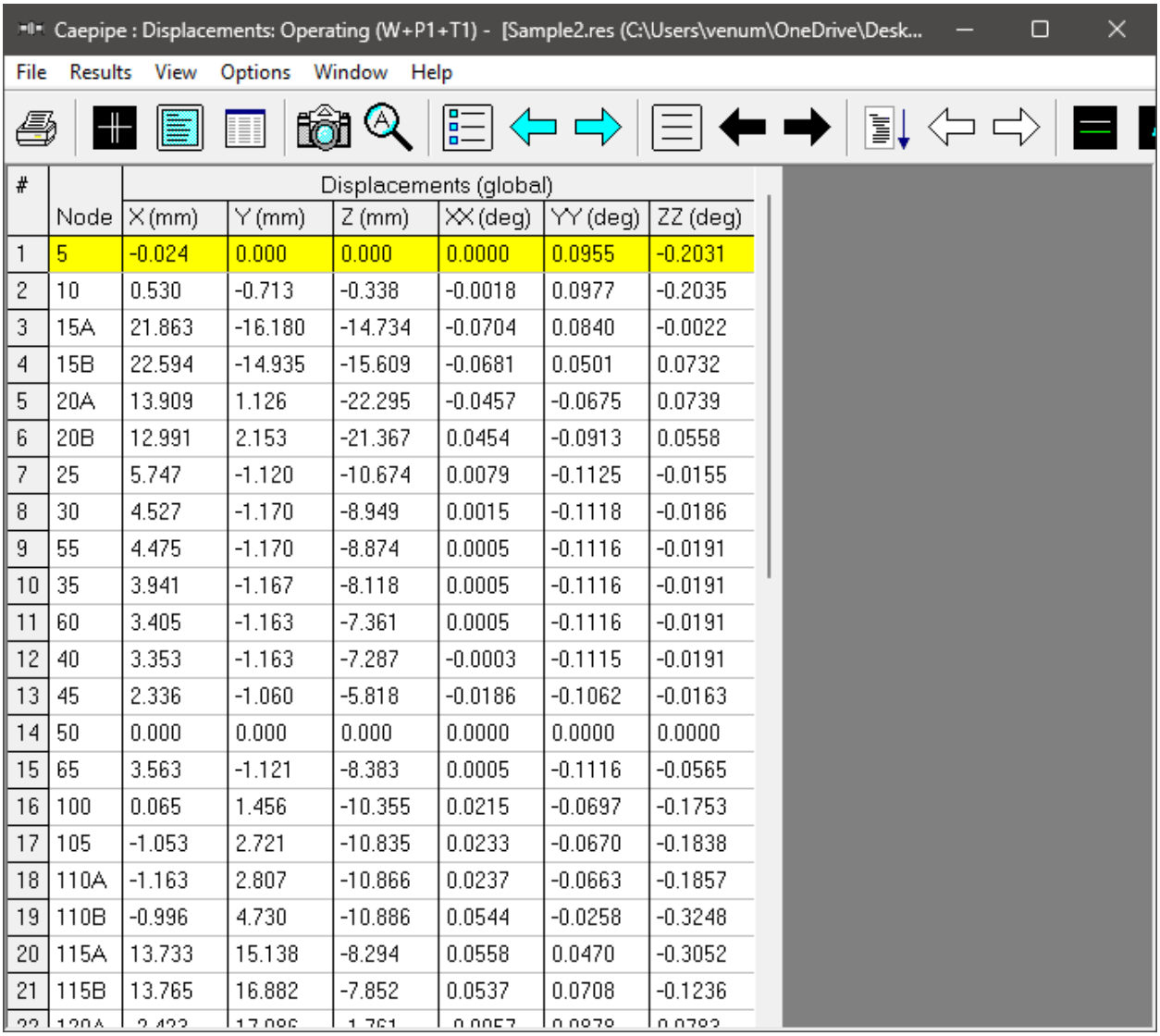


Use the Other forces button (F6), Next force button (Ctrl+Right arrow) or Previous force button (Ctrl+Left arrow) to see other element forces (e.g. valves, bellows etc.).



Displacements

The nodal displacements are shown.

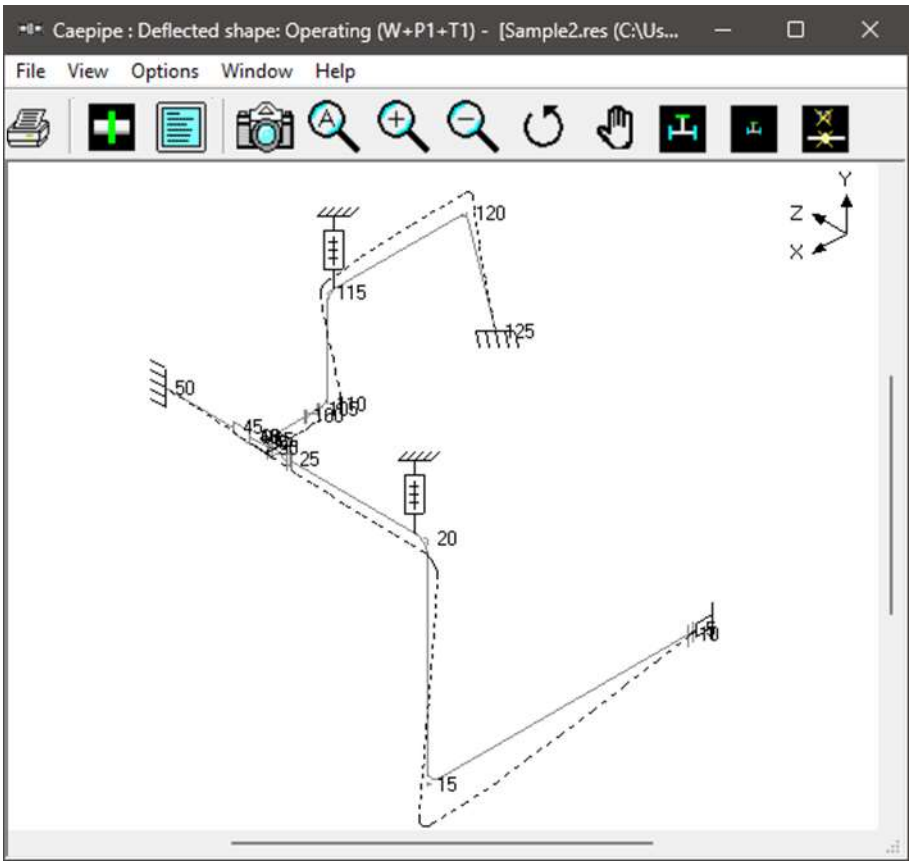


#	Node	Displacements (global)					
		X (mm)	Y (mm)	Z (mm)	XX (deg)	YY (deg)	ZZ (deg)
1	5	-0.024	0.000	0.000	0.0000	0.0955	-0.2031
2	10	0.530	-0.713	-0.338	-0.0018	0.0977	-0.2035
3	15A	21.863	-16.180	-14.734	-0.0704	0.0840	-0.0022
4	15B	22.594	-14.935	-15.609	-0.0681	0.0501	0.0732
5	20A	13.909	1.126	-22.295	-0.0457	-0.0675	0.0739
6	20B	12.991	2.153	-21.367	0.0454	-0.0913	0.0558
7	25	5.747	-1.120	-10.674	0.0079	-0.1125	-0.0155
8	30	4.527	-1.170	-8.949	0.0015	-0.1118	-0.0186
9	55	4.475	-1.170	-8.874	0.0005	-0.1116	-0.0191
10	35	3.941	-1.167	-8.118	0.0005	-0.1116	-0.0191
11	60	3.405	-1.163	-7.361	0.0005	-0.1116	-0.0191
12	40	3.353	-1.163	-7.287	-0.0003	-0.1115	-0.0191
13	45	2.336	-1.060	-5.818	-0.0186	-0.1062	-0.0163
14	50	0.000	0.000	0.000	0.0000	0.0000	0.0000
15	65	3.563	-1.121	-8.383	0.0005	-0.1116	-0.0565
16	100	0.065	1.456	-10.355	0.0215	-0.0697	-0.1753
17	105	-1.053	2.721	-10.835	0.0233	-0.0670	-0.1838
18	110A	-1.163	2.807	-10.866	0.0237	-0.0663	-0.1857
19	110B	-0.996	4.730	-10.886	0.0544	-0.0258	-0.3248
20	115A	13.733	15.138	-8.294	0.0558	0.0470	-0.3052
21	115B	13.765	16.882	-7.852	0.0537	0.0708	-0.1236
22	120A	2.423	17.086	1.761	0.0057	0.0878	0.0783

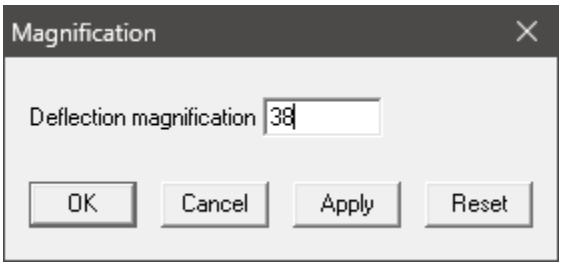
Use the Load cases button, Next load case button (Right arrow) or Previous load case button (Left arrow) to see loads for different load cases(e.g. Sustained, Expansion etc.).

Use the Deflected shape button (or View > Show deflected shape) to plot the deflected shape in the graphics window.

Use the Animated deflected shape button (or View > Show animated deflected shape) to plot the animated deflected shape in the graphics window.



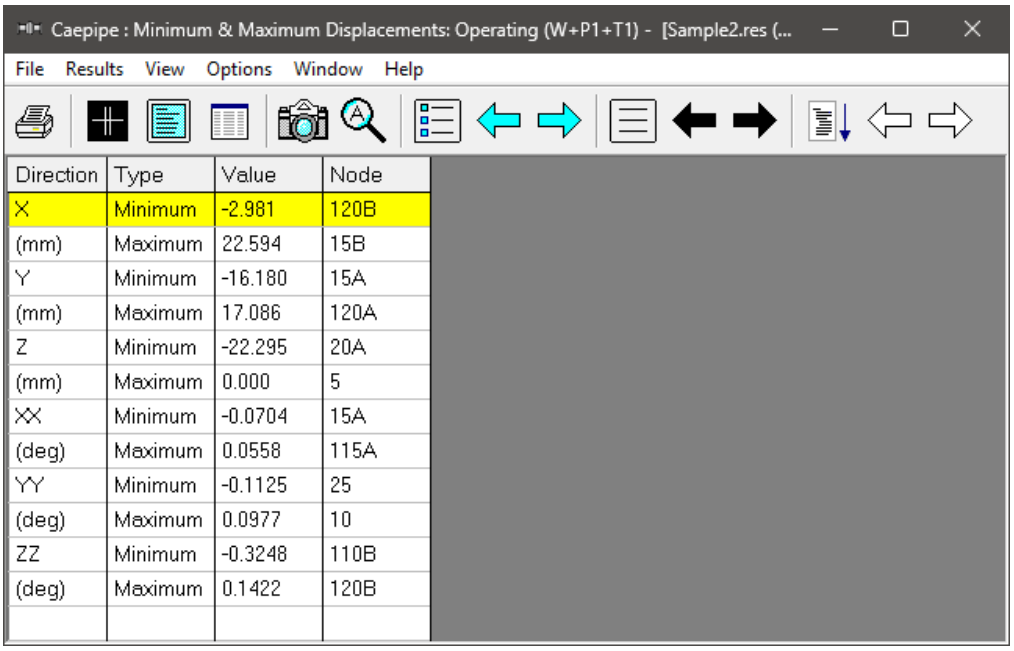
Choose View > Magnification to change the magnification of the deflected shape.



The reset button is used to calculate a default magnification factor which scales the maximum deflection to about 5% of the width of the graphics window.

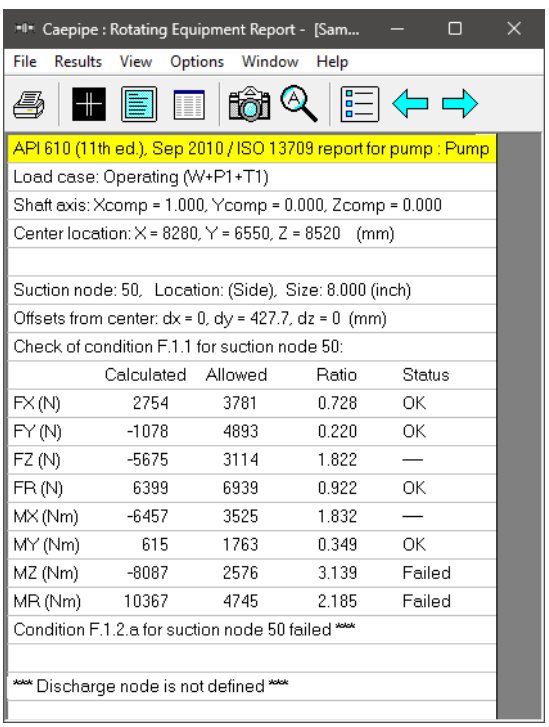
Use the Other displacements button (F6), Next displacement button (Ctrl+Right arrow) or Previous displacement button (Ctrl+Left arrow) to see other displacements (e.g. Min/Max, displacements at hangers, flex joints, limit stops etc.).

The minimum and maximum displacements for each of the directions and the corresponding nodes are shown below.



Direction	Type	Value	Node
X	Minimum	-2.981	120B
(mm)	Maximum	22.594	15B
Y	Minimum	-16.180	15A
(mm)	Maximum	17.086	120A
Z	Minimum	-22.295	20A
(mm)	Maximum	0.000	5
XX	Minimum	-0.0704	15A
(deg)	Maximum	0.0558	115A
YY	Minimum	-0.1125	25
(deg)	Maximum	0.0977	10
ZZ	Minimum	-0.3248	110B
(deg)	Maximum	0.1422	120B

The Pump qualification report (Rotating Equipment report) is shown below.



API 610 (11th ed.), Sep 2010 / ISO 13709 report for pump : Pump
Load case: Operating (W+P1+T1)
Shaft axis: Xcomp = 1.000, Ycomp = 0.000, Zcomp = 0.000
Center location: X = 8280, Y = 6550, Z = 8520 (mm)

Suction node: 50, Location: (Side), Size: 8.000 (inch)
Offsets from center: dx = 0, dy = 427.7, dz = 0 (mm)
Check of condition F.1.1 for suction node 50:

	Calculated	Allowed	Ratio	Status
FX (N)	2754	3781	0.728	OK
FY (N)	-1078	4893	0.220	OK
FZ (N)	-5675	3114	1.822	—
FR (N)	6399	6939	0.922	OK
MX (Nm)	-6457	3525	1.832	—
MY (Nm)	615	1763	0.349	OK
MZ (Nm)	-8087	2576	3.139	Failed
MR (Nm)	10367	4745	2.185	Failed

Condition F.1.2.a for suction node 50 failed ***

*** Discharge node is not defined ***

Frequencies and Mode shapes

A list of natural frequencies, periods, modal participation factors and modal mass fractions is shown next. You can show each frequency's mode shape graphically or animate it by clicking on Show mode shape or Show animated mode shape button in the toolbar.

Caepipe : Frequencies - [Sample2.res (C:\Users\venum\OneDrive\Des...

File Results View Options Window Help

#	Frequency (Hz)	Period (second)	Participation factors			Modal mass / Total mass		
			X	Y	Z	X	Y	Z
1	1.703	0.5872	0.7395	-1.9310	1.2246	0.0314	0.2138	0.0860
2	2.458	0.4069	0.8526	-2.2220	-1.5016	0.0417	0.2831	0.1293
3	2.675	0.3739	-2.4950	-1.3661	0.2625	0.3570	0.1070	0.0040
4	4.158	0.2405	-0.3444	0.3579	-1.2489	0.0068	0.0073	0.0894
5	6.661	0.1501	0.8111	-1.2509	0.1106	0.0377	0.0897	0.0007
6	6.761	0.1479	0.9343	0.6600	-0.0140	0.0501	0.0250	0.0000
7	7.456	0.1341	0.0803	-0.1084	-0.1687	0.0004	0.0007	0.0016
8	9.959	0.1004	1.2280	0.1913	-0.1710	0.0865	0.0021	0.0017
9	11.629	0.0860	0.0727	-0.9579	0.0383	0.0003	0.0526	0.0001
10	13.281	0.0753	-0.6669	-0.0590	-1.1835	0.0255	0.0002	0.0803
11	30.952	0.0323	-2.2993	-0.0525	0.0055	0.3032	0.0002	0.0000
12	53.862	0.0186	-0.1820	0.1712	-0.5697	0.0019	0.0017	0.0186
13					Total	0.9423	0.7834	0.4117

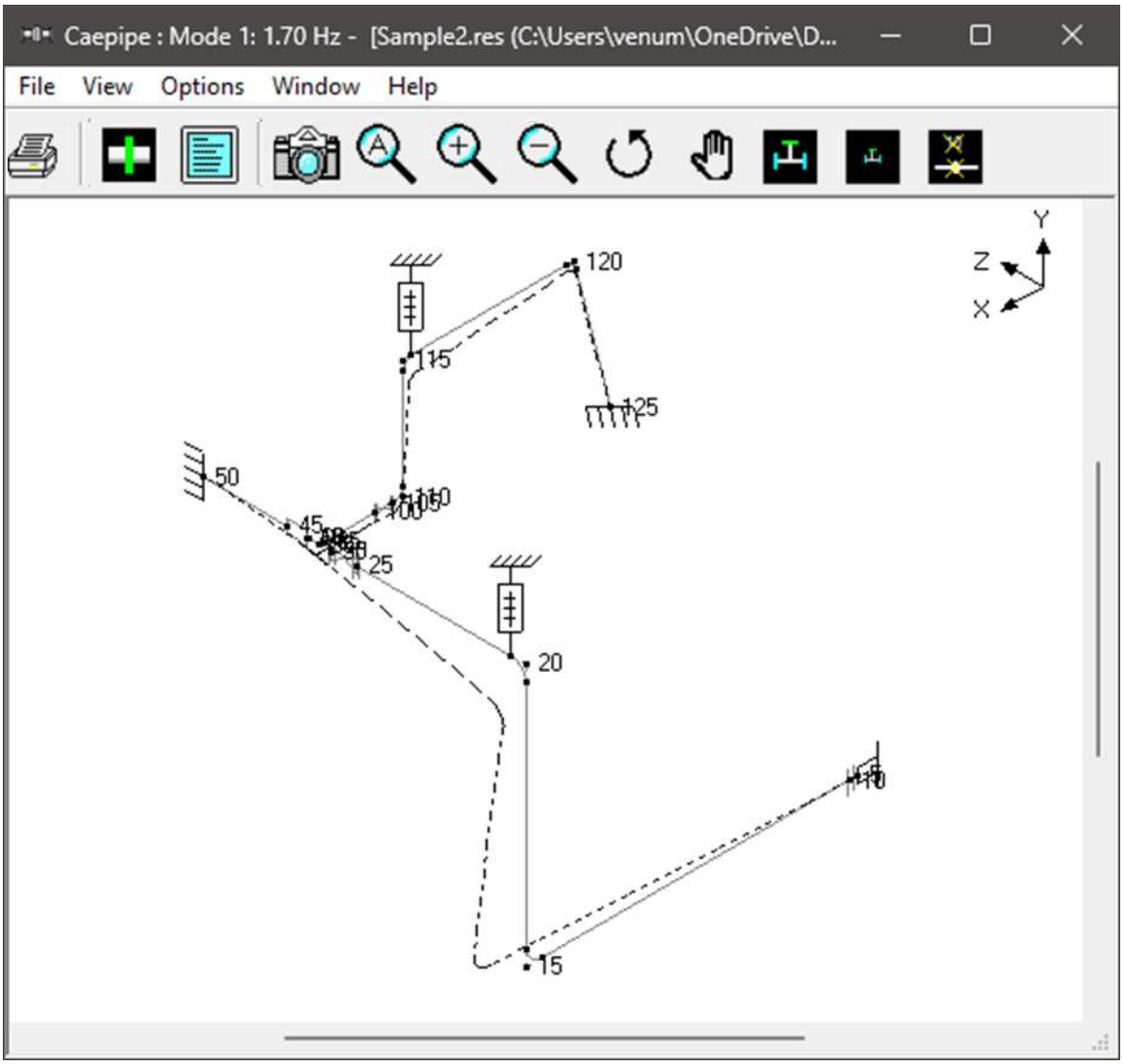
Each frequency's mode shape detail is shown in the next window. As in the earlier window, you can show graphically the mode shape or animate it by clicking on the appropriate button.

Caepipe : Mode 1: 1.70 Hz - [Sample2.res (C:\Users\venum\OneDriv...

File Results View Options Window Help

#	Node	X (mm)	Y (mm)	Z (mm)	XX (deg)	YY (deg)	ZZ (deg)
1	5	0.000	0.000	0.000	0.0000	-0.0583	-0.0444
2	10	0.000	-0.155	0.206	-0.0017	-0.0597	-0.0446
3	15A	0.000	-6.256	10.160	-0.0657	-0.0766	-0.0440
4	15B	0.278	-6.539	10.121	-0.0864	-0.0619	-0.0402
5	20A	3.941	-6.537	0.568	-0.0928	-0.0455	-0.0306
6	20B	3.830	-6.013	0.000	-0.0695	-0.0424	-0.0265
7	25	1.357	-2.102	0.000	-0.0461	-0.0297	-0.0142
8	30	1.039	-1.607	0.000	-0.0450	-0.0289	-0.0137
9	55	1.025	-1.586	0.000	-0.0448	-0.0288	-0.0136
10	35	0.887	-1.372	0.000	-0.0448	-0.0288	-0.0136
11	60	0.750	-1.158	0.000	-0.0448	-0.0288	-0.0136
12	40	0.736	-1.137	0.000	-0.0446	-0.0287	-0.0135

The graphic window will show the mode shape as below.

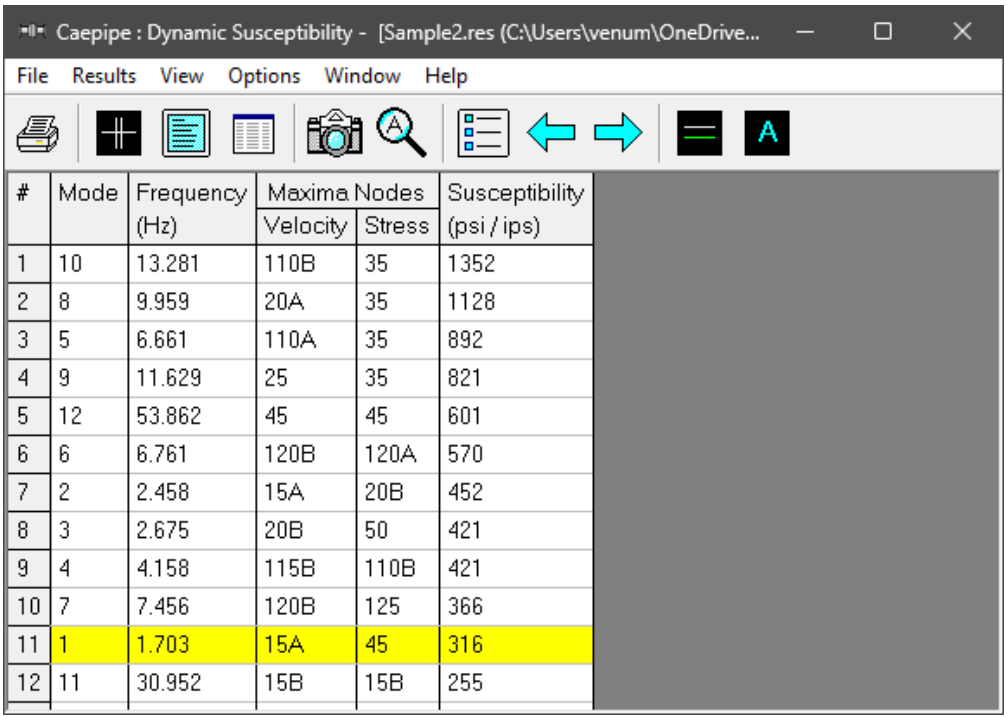


Use the black arrow buttons to cycle through the different Modes.

Dynamic Susceptibility

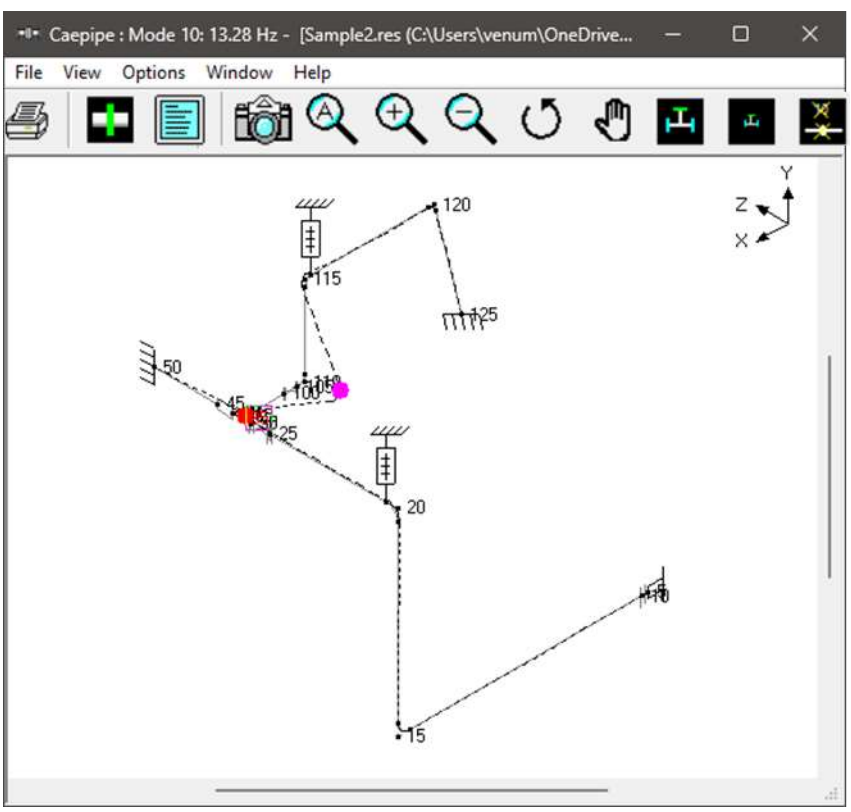
Note: *Dynamic Susceptibility is NOT available for Evaluation Version of CAEPIPE. For Full Version of CAEPIPE, this feature can be turned ON by setting an environment variable "HARTLEN" that needs to be declared under My Computer or This PC Icon > Mouse Right Click > Properties > Advanced System Settings > Environmental Variable with its Value set to (YES). Refer to CAEPIPE User's Manual for more details.*

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.

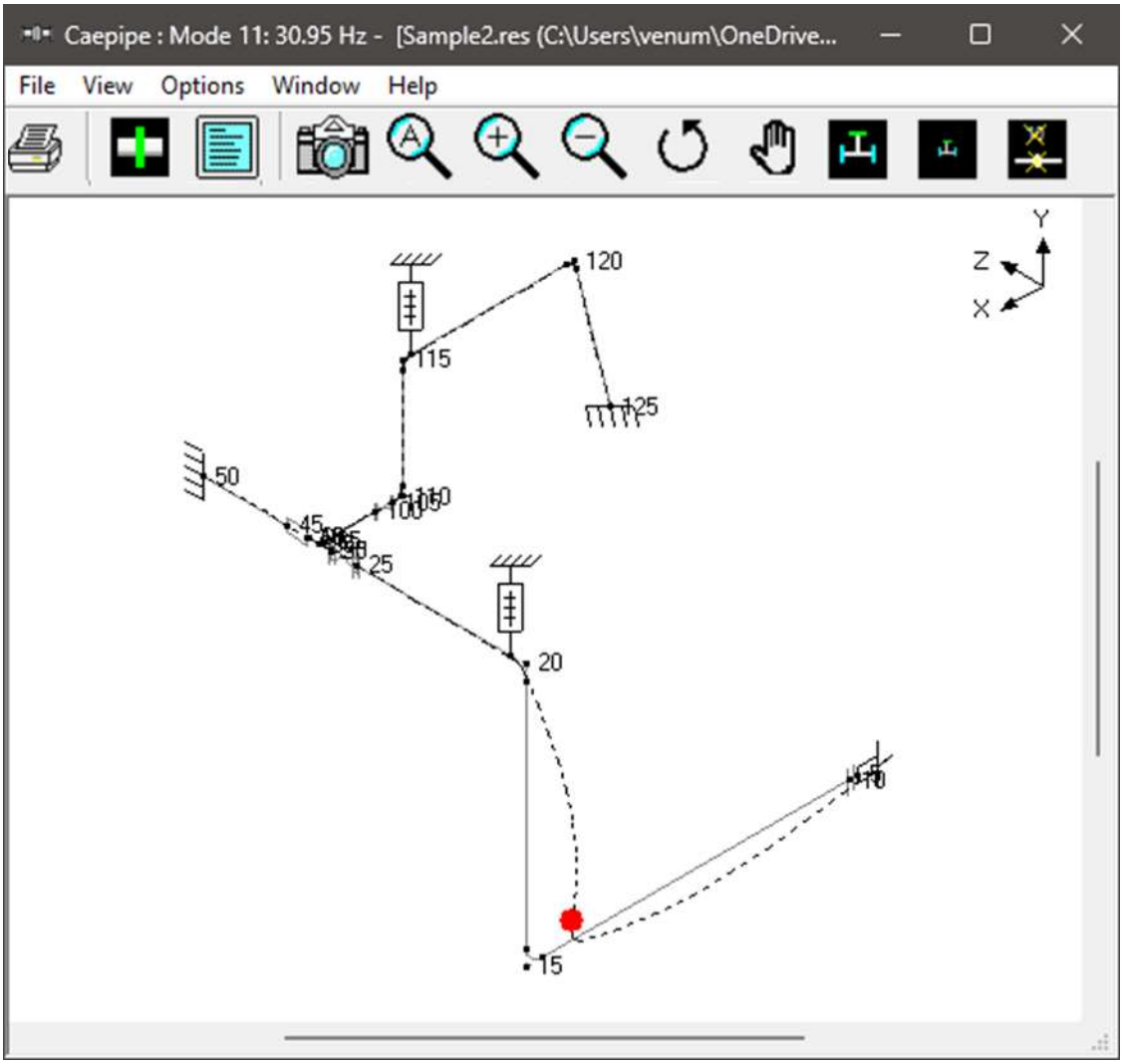


#	Mode	Frequency (Hz)	Maxima Nodes Velocity	Stress	Susceptibility (psi / ips)
1	10	13.281	110B	35	1352
2	8	9.959	20A	35	1128
3	5	6.661	110A	35	892
4	9	11.629	25	35	821
5	12	53.862	45	45	601
6	6	6.761	120B	120A	570
7	2	2.458	15A	20B	452
8	3	2.675	20B	50	421
9	4	4.158	115B	110B	421
10	7	7.456	120B	125	366
11	1	1.703	15A	45	316
12	11	30.952	15B	15B	255

Pressing the Animated mode shape button (or View > Show animated mode shape) for Mode 10, for example, shows the maximum dynamic bending stress at the Bend Node 35 (RED dot) and the maximum velocity at the Bend Node 110B (PINK dot).



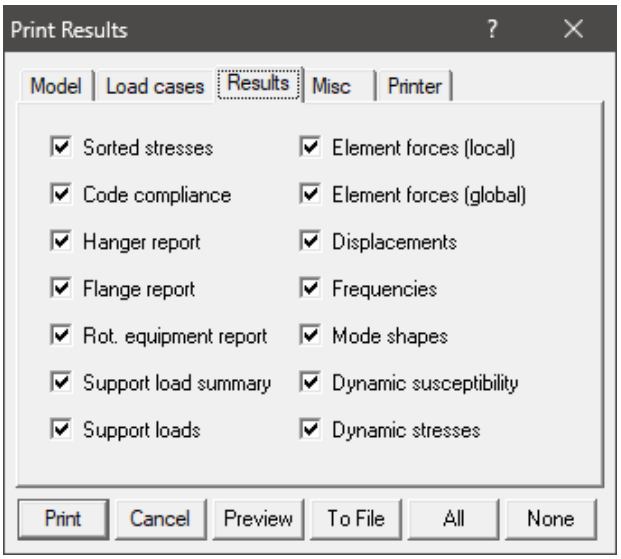
In case the maximum dynamic bending stress and the maximum velocity occur at the same node for a specific mode, then the RED and PINK dots overlap with each other and only the RED dot is seen for that mode. See the Animated mode shape shown below for mode 11 as an example.



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 to possibly reduce the susceptibility to vibration. It can be used for design, planning acceptance tests, troubleshooting and correction.

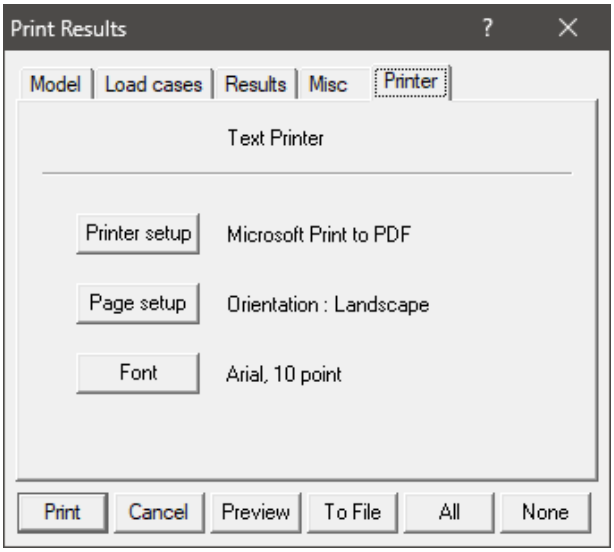
Print

To print results and model data, click on the Print button (or press Ctrl+P). In the Print Results dialog, the items to print can be selected.



You can also print to a text file by using the To File button. A preview of the printed output can be seen by using the Preview button.

The printing options such as choice of printer, margins, portrait or landscape and font can be set on the Printer tab.



This is the end of the tutorial. If you have questions or comments, please email them to: **support@sstusa.com**