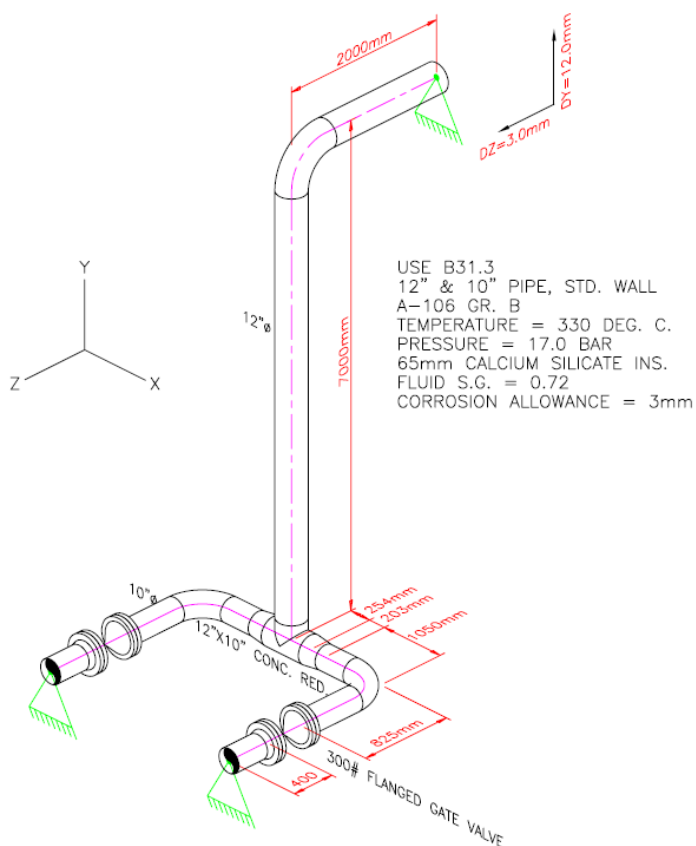


Exercise 01

This exercise will provide further practise with the piping input, and introduce alternative editing tools which may increase productivity in creating models. We will also investigate and review the results to see what to look for and see how the piping system is behaving, and how to correct any issues which may arise during the design.

The first stage of this exercise is to input the model. The model is below; you will also have the same isometric printed on a separate hand-out in a larger format.

As before with the cantilever example, the model will be input using the node numbering system. Each section between two nodes is called an element. i.e. node 10 to node 20 are linked together by an element, referred to by 'element 10 to 20'. Prior to entering geometry, it can be very useful and is a good idea to mark up the isometric drawing with the intended node number sequence.



We will use a slightly different method of inputting the data, which will allow us to maximise the graphics area during input. In the main "Classic Piping Input", on each area, notice the ">>" symbol in the top right corner:

Classic Piping Input

From: 10

To: 20

☐ Name

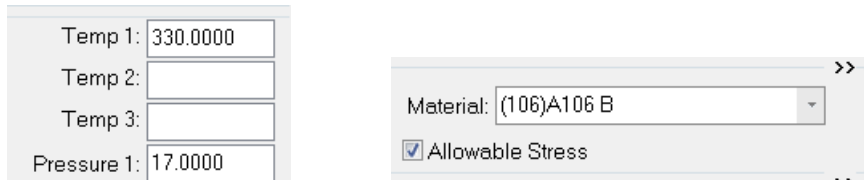
>>

Double click this symbol to “tear off” the particular section of the input spread sheet. This will allow the Classic window to be minimised for the most part thus maximising the graphics.

Tear off the Node Numbers, Dimension Deltas and Pipe Sizes areas. As the material temperatures and pressures do not change throughout the model we can enter these on the first element and then we will not need them again.

Input Model

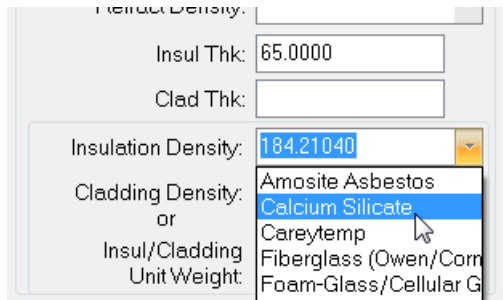
Enter A106-B as the material, 330°C as the temperature and 17 bars as the pressure



Temp 1: 330.0000
Temp 2:
Temp 3:
Pressure 1: 17.0000

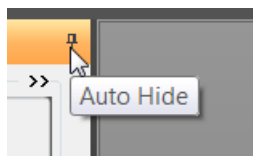
Material: (106)A106 B
☒ Allowable Stress

In this model we also require insulation; 65mm thick Calcium Silicate.



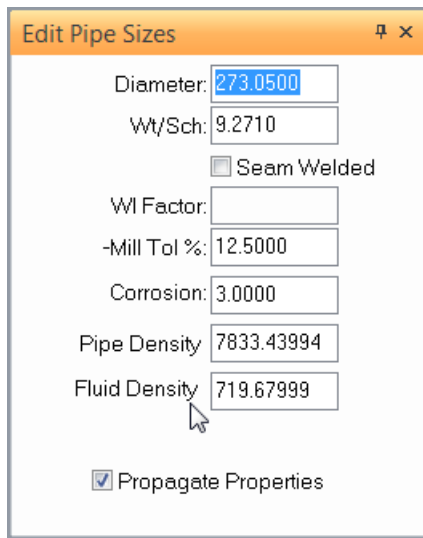
Insul Thk: 65.0000
Clad Thk:
Insulation Density: 184.21040
Cladding Density:
or
Insul/Cladding Unit Weight:
Amosite Asbestos
Calcium Silicate
Careytemp
Fiberglass (Owen/Corn
Foam-Glass/Cellular G

The rest of the information we will need to enter for our model can be done via the three windows we have “torn off”. Minimise the Classic piping input (of course this can always be maximised at any point if needed).



Finally we can enter the pipe size and schedule, along with the densities and corrosion allowance, as per the isometric.

The fluid density can be entered as 0.72SG and CAESAR II will convert this specific gravity to the correct units. As before the pipe size can be entered as 10 for 10” and S for STD schedule piping.



Edit Pipe Sizes

Diameter: 273.0500

Wt/Sch: 9.2710

☐ Seam Welded

WI Factor:

-Mill Tol %: 12.5000

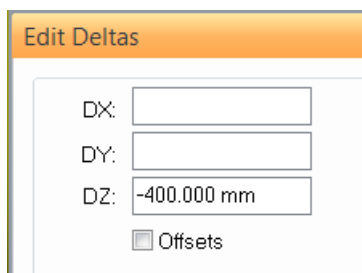
Corrosion: 3.0000

Pipe Density: 7833.43994

Fluid Density: 719.67999

☒ Propagate Properties

We will begin at the bottom “right” pipe where it is connected to a pump. This will be node 10. Note that this is an anchor, a fixed point in our system. Element 10 to 20 is 400mm in length, in the -Z direction. Enter DZ as -400mm



Edit Deltas

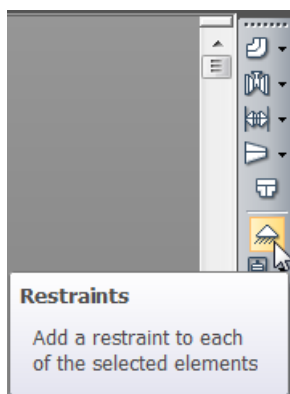
DX:

DY:

DZ: -400.000 mm

☐ Offsets

Node 10 is also fixed so we need to specify an anchor. Use the toolbar on the left hand side of the graphics window (default location) to specify a restraint.



The Auxiliary Data – Restraints window will appear. Specify that the anchor is at node 10. The auxiliary data window can now be closed.

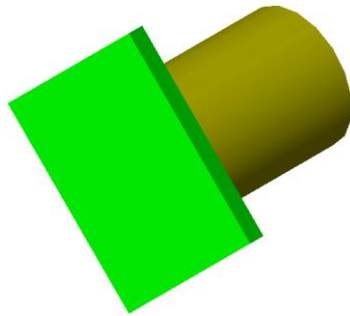
Auxiliary Data

Node: 10 CNod

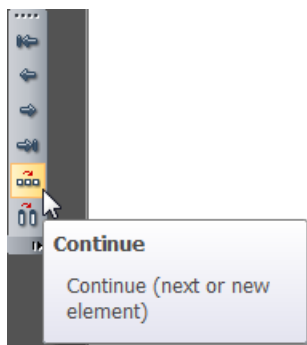
Type: ANC Ga

Stif: M

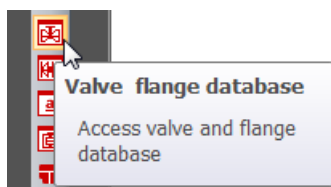
Our first element is complete, and should look like the one below:



Use the Continue button to create a new element:



This next element is a 300# flanged gate valve. We could enter this in a number of ways. The valve will be rigid relative to the surrounding piping, so must be specified as a “rigid element” with a weight. This can be done either as 3 separate elements (flange – valve – flange), or as one overall element with the total length and combined weight specified. This can be done manually or by using the valve flange database to obtain the length/weight automatically from CAESAR II’s catalogue, which we will do. Select the Valve flange database button and select a gate valve with flanged ends, class 300.



The Flange – Valve – Flange check box can be used to split the component into 3 elements if required.

Valve and Flange Database: CADWORX.DAT

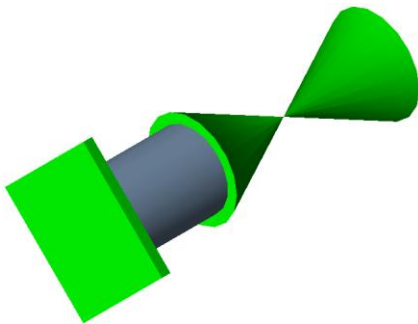
Rigid Type	End Type	Class
GATE	FLG	150
GLOBE	NOFLG	300
CHECK	THREAD	600
CONTRL	SOCKET	800
BALL		900
PLUG		1500
BTRFLY		2500
FLG.PR		
FLANGE		

☒ Whole element
 ☐ From end
 ☐ To end
 ☐ Both ends

☐ Flange - Valve - Flange

☐ Activate Flange Check

The element will appear in node 20 to 30.




The correct length will be inserted (and the element will continue in the same direction as the previous element). Also note that the Rigid check box is checked and the rigid weight has been entered with the relevant weight for a 300# gate valve and flanges. (Hover briefly over the Classic piping input where it is docked).

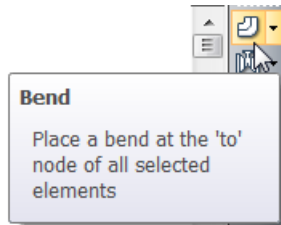
Classic Piping Input

<input type="checkbox"/> Bend	<input type="checkbox"/> Reducer
<input checked="" type="checkbox"/> Rigid	<input type="checkbox"/> SIFs & Tees
<input type="checkbox"/> Expansion Joint	

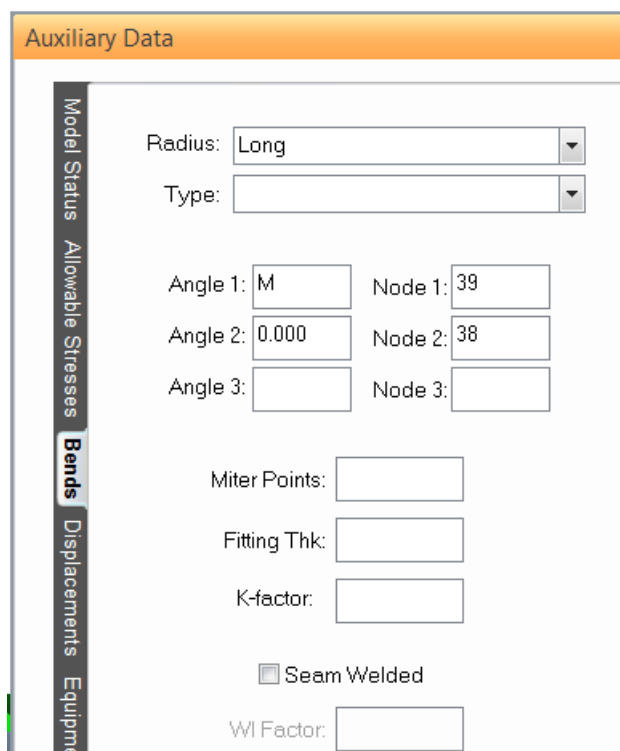
Rigid Weight: 4198.912

Continue to the next element .

Enter the DZ as -825mm. This element also leads into a bend, so press the Bend button on the right hand toolbar. If using the classic piping input we could check the bend check box to achieve the same result.

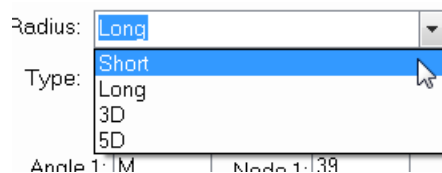


The bend auxiliary data window will appear.

The 'Auxiliary Data' window for a bend. It has a sidebar with tabs: 'Model Status', 'Allowable Stresses', 'Bends' (selected), 'Displacements', and 'Equipment'. The main area contains the following fields:


- Radius: Long (dropdown)
- Type: (dropdown)
- Angle 1: M Node 1: 39
- Angle 2: 0.000 Node 2: 38
- Angle 3: Node 3:
- Miter Points:
- Fitting Thk:
- K-factor:
- ☐ Seam Welded
- WI Factor:

The default bend type is a long radius (1.5D) bend This radius can be changed. Common bend radii are available in the drop down, alternatively any radius required can simply be typed in here.




In addition, further data can also be entered such as if the bend is flanged or mitred etc. Accept the default long radius bend.

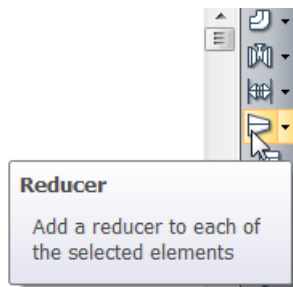
The graphics will not display the bend yet, as there is no following element.

Continue to the next element .


This time we are now continuing in the -X direction. DX is -1050. The bend will now be visible in the graphics.

Continue to the next element .

This element is a 10"x12" concentric reducer and is 203mm in length. Enter DX as -203mm and specify that this is a reducer.

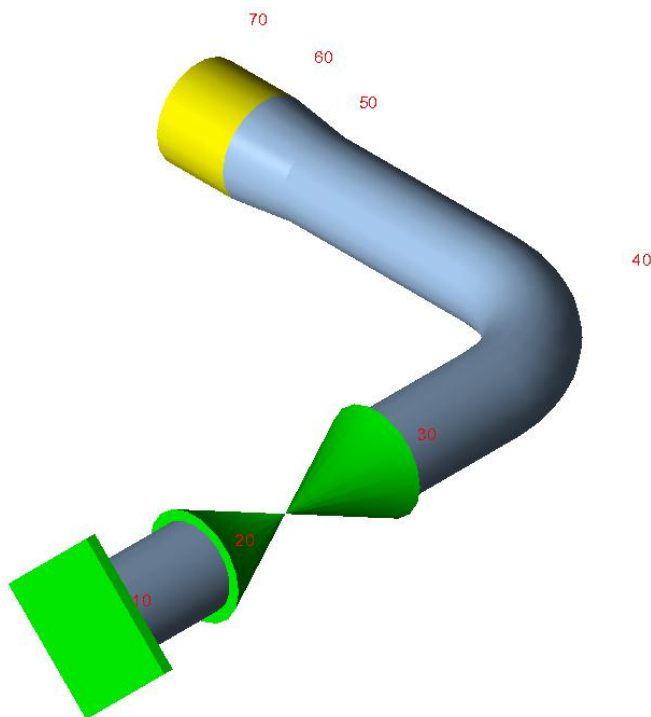


The Reducer Auxiliary will appear and we can specify further data, including the second end size. As before, entering a nominal size in here will be converted to the actual OD. Enter 12 in the diameter 2 and S in the thickness 2 fields, which will be converted to the actual values.

Continue to the next element .

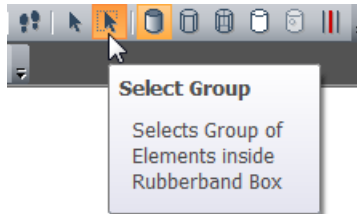
Finally continue from the end of the reducer to the centre of the tee, 254mm as shown on the isometric. DX is -254mm

The model at this point should now resemble the image below, note the node numbers in the image:



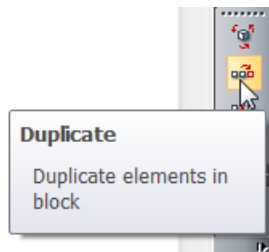
We can now take advantage of the fact that the model is symmetrical and use the functions in CAESAR II to mirror the piping to create the opposite leg.

Use the Select group function to activate the graphical selection mode and draw a window around the model.

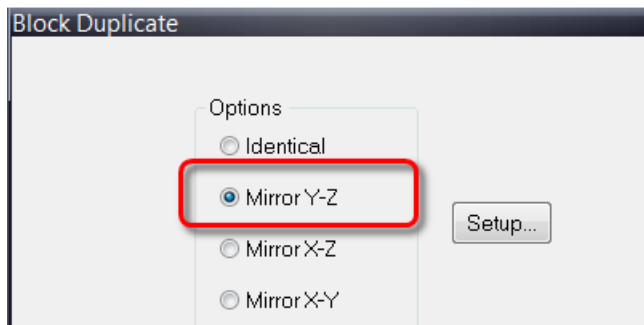


All elements will turn yellow to indicate that they are currently selected. Ensure all components are selected.

The Duplicate function can be used to copy, and mirror if required, selected elements.



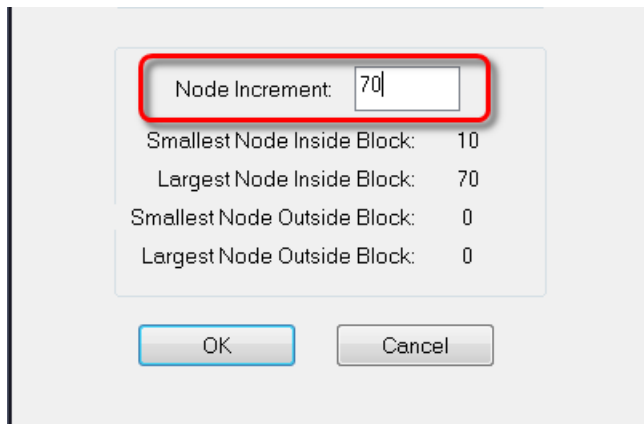
Duplicate the selected elements and choose to mirror about the Y-Z plane.



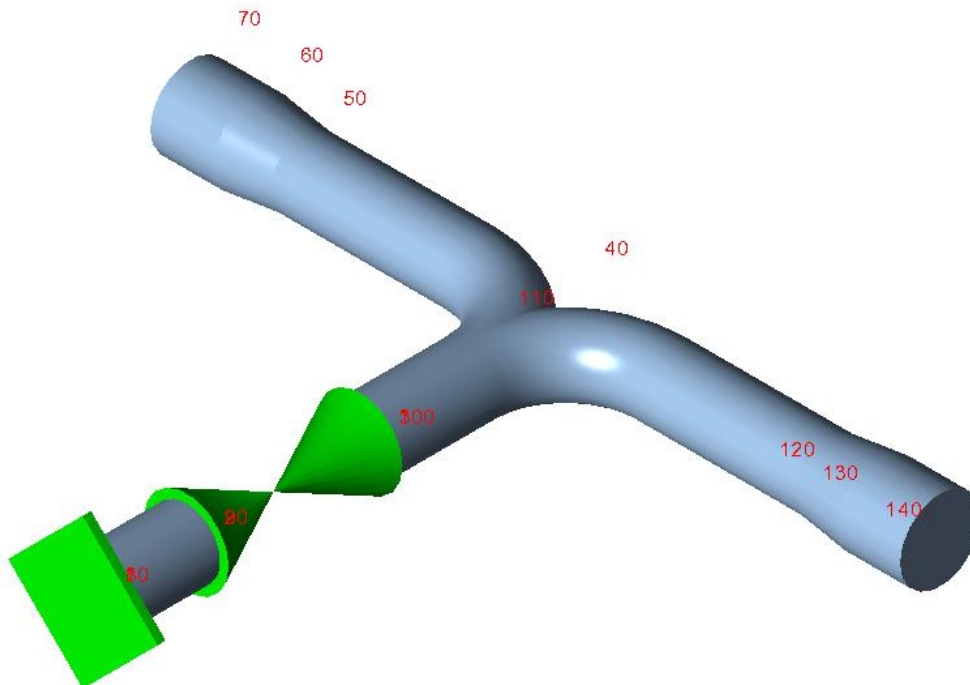
We also need to increment the node numbers so that we do not have duplicate nodes.

Currently our model goes from node 10 through to node 70.

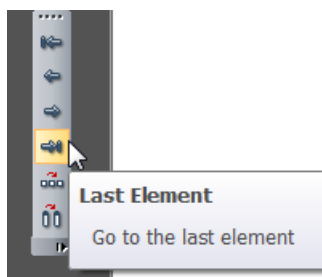
If we increase the node numbers by 70, node 10 will become node 80, 20 becomes 90 and so on. Therefore the second leg will be node 80 through to node 140. The only issue with this is that there are no common nodes, so the piping will not actually be connected. This can easily be fixed by changing node 140 (the centre of the tee on the second leg) to become node 70 (the node at the centre of the tee on the first leg). This will connect up the piping at the common node, 70 – the centre of the tee.



Click OK and the pipe will be duplicated, but as already stated there is no common node so CAESAR II does not know where to place the pipe. As such it locates it at the origin. The resulting model looks like the following.



All we need to do is connect element 130 – 140 to element 60 – 70. This can be done by changing 140 to become node 70. Select element 140. There are various ways of doing this – either double click in the graphics area, or use the navigation buttons to navigate to the correct element (as this is the last element the end button will quickly take you to the correct element).



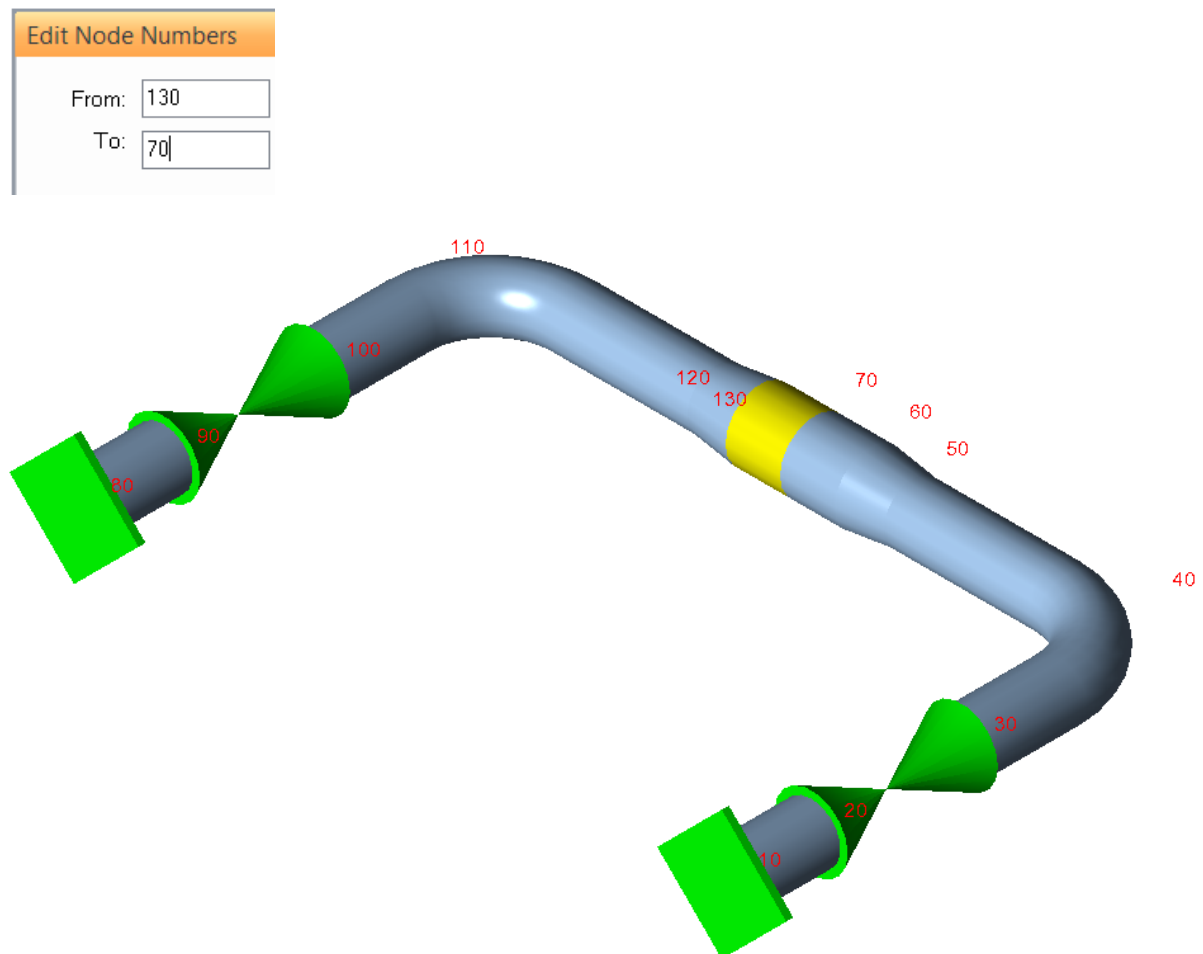
The Edit Node numbers window should now read 130 to 140 and the element will be highlighted in the model.

Edit Node Numbers

From:

To:

Simply change the “To” node from 140 to 70. The model will now be connected as should look like the one below:



We can now complete the model by adding the vertical leg and connection to the vessel.


Skip to the last element. This can be done by again using the Last Element navigation button or using the Ctrl + End buttons on the keyboard.

Click “Continue” to move to the next element . The node numbers will default to 70 to 80. We need to change this to 70 to 140.

Edit Node Numbers

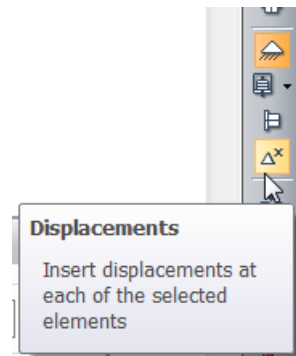
From:

To:

This element is the vertical leg, and is 7m in the Y direction. DY is therefore 7000. This also leads into a bend so select the Bend icon as well. .

Click “continue” and place the final element 140 to 150 in the –Z direction, 2000mm. The final element connects to the vessel, so we will place an anchor at this point. Click the restraint button and specify an anchor at node 150

Notice in the isometric that at the vessel connection, there are DY and DZ displacements. These are due to the thermal expansion of the vessel.



Select the Displacements button and enter in the required values 3mm in DZ and 12mm in DY.

Node 1: <input type="text" value="150"/>			
	Vector 1	Vector 2	Vector 3
DX			
DY	12.0000		
DZ	3.0000		
RX			
RY			
RZ			

Error Checking

The model is now complete, so run the error checker.

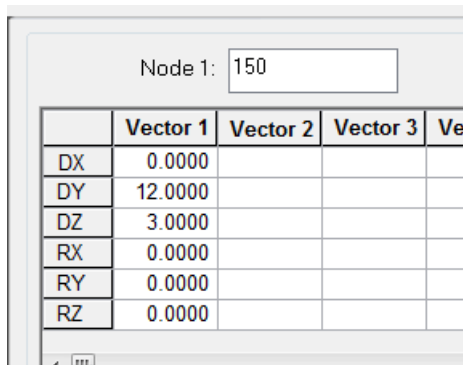


We will receive a fatal error and three warnings. We must correct the errors before we can analyse the model. The warnings may be acceptable but we should check to confirm that the input is as intended.

	Message Type	Message Number	Element/ Node Number	Message Text
1	ERROR	57E	140-150	At node 150 there is a DISPLACEMENT and a RESTRAINT specified. This results in a numerically overspecified boundary condition. The user should either give the displacements of the point, or restrain the point, but not both. If modelling initial thermal movements at a restraint, define some unique connecting node for the restraint, and specify the displacement for the connecting node.

So our error is mentioning that we have both an anchor and displacements specified at node 150. This cannot be possible as the anchor fixes the point, but the displacements move the same point. We cannot have both at the same time. Remove the anchor and edit the displacements.

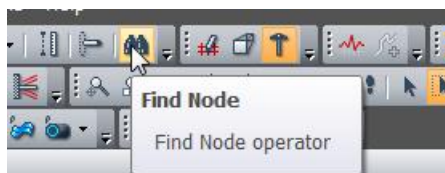
Double click the error message to go straight to the area of concern. Now click the restraints button to remove restraints. Click OK in the message which appears. Now edit the displacements and fill in 0 in all other field (DX,RX,RY,RZ). A displacement of zero will fix the node in that direction, so now our node is fixed in all directions, except for DY and DZ where the relevant displacements are applied.



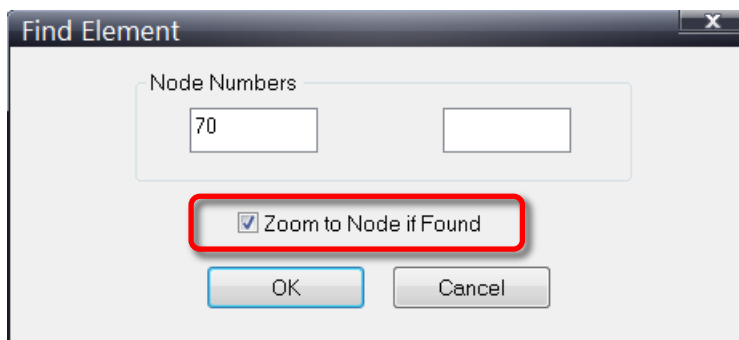
	Vector 1	Vector 2	Vector 3	Vector 4
DX	0.0000			
DY	12.0000			
DZ	3.0000			
RX	0.0000			
RY	0.0000			
RZ	0.0000			

Re run the error checker and investigate the warnings. The second two warnings are regarding the reducer alpha angle which is not specified. CAESAR II is therefore using a default computed value. This is acceptable here for us.

The first warning is stating that there is a geometric intersection at node 70 (the tee) but we have not specified a type of tee, and therefore a SIF. This can sometimes be correct but is most often the result of an oversight, as in this case. Return to the input and locate node 70. The Find tool can be used to do this:



The Zoom to Node if found check box will also zoom into that node/element if it is found, useful on larger models.



Find Element

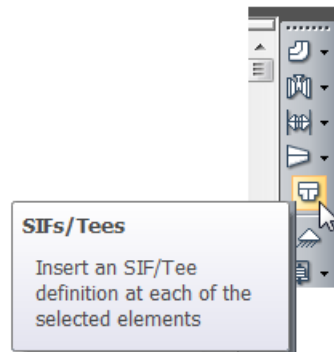
Node Numbers

70

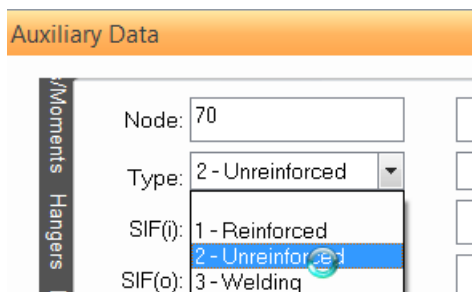
☒ Zoom to Node if Found

OK Cancel

On node 70 use the SIFs/Tees button to specify a SIF at this point. This only needs to be done on one of the elements connecting to node 70, it is not necessary to do this on all three.



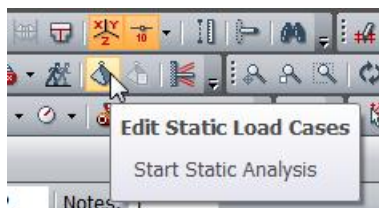
Select an unreinforced tee.



Re-run the error checker. All should now be OK, only the reducer alpha warnings will remain, plus the C of G report.

Review Load Cases

Access the load case editor



Recall from earlier the design code (we are using B31.3) addresses the stresses produced by the various loads. In our model we have the following loads applied:

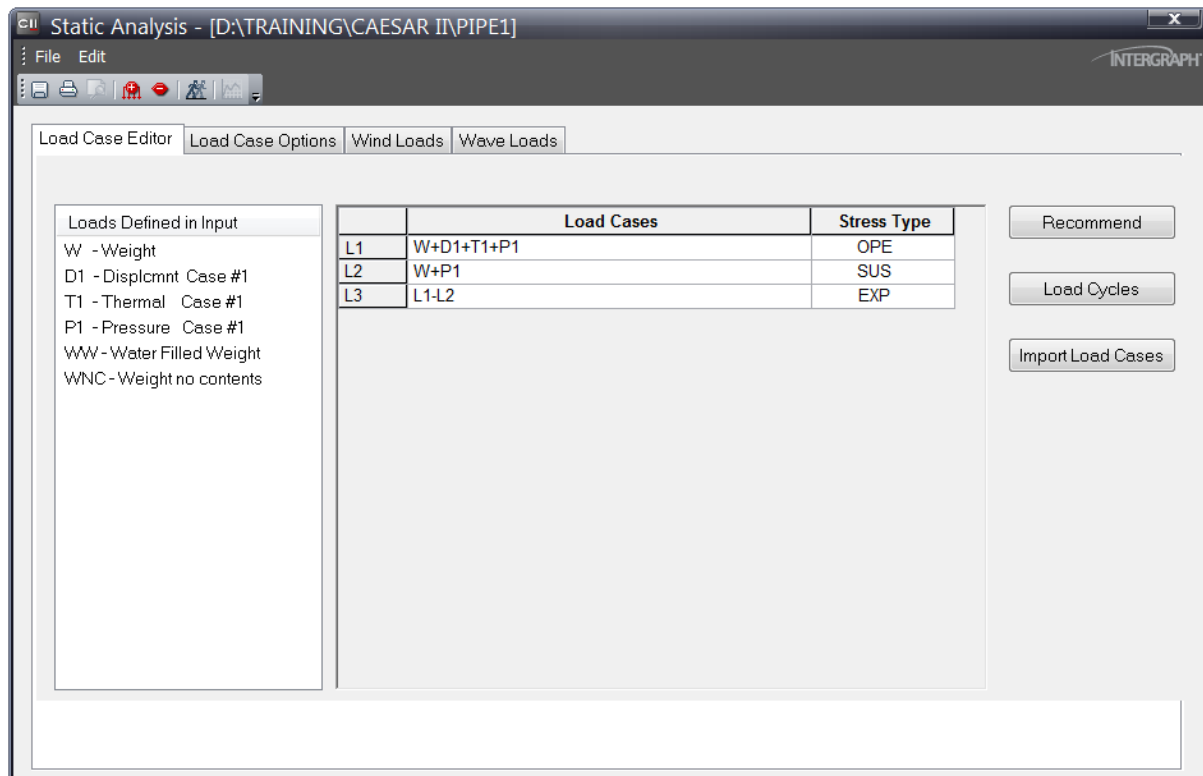
- Weight
- Pressure
- Temperature
- Displacement

B31.3 requires that two checks are performed – Sustained and Expansion

Sustained – Weight and Pressure

Expansion – Temperature and Displacement

These load cases are defined by CAESAR II as the default (recommended) load cases, shown in rows L2 and L3.



Row L1 is an operating case (OPE) and is the “Hot” case consisting of the ‘real world’ loads. This case is not required by B31.3 (although some codes do require this case also). However as this case is a “real world” scenario it is used to establish restraint loads and loads on equipment connections. In addition, it is used to derive the Expansion case. The expansion case is the algebraic difference between L1 and L2 ($L1 - L2$).

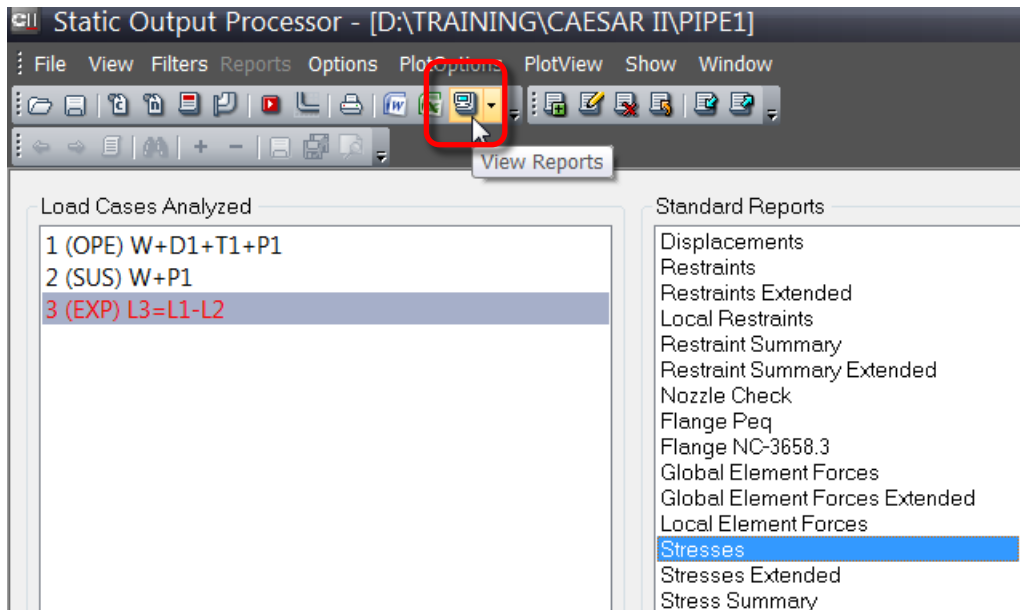
Accept these load cases and run the analysis by clicking the “Running Man” icon.

Review Results

After the analysis has run, the output processor will appear. The first thing to notice is that the EXP case is coloured red. This indicates that this case has failed to code stress check. That is, the computed stresses in the system at some point are greater than the allowables published in the code.

We need to fix this.

Select the Expansion case and view the results for the Stresses report.



The report shows that the code stress check failed and highlights in red where the check failed. Double clicking on any column will order the report by that column. Double click on the Code Stress column header to order by highest stress.

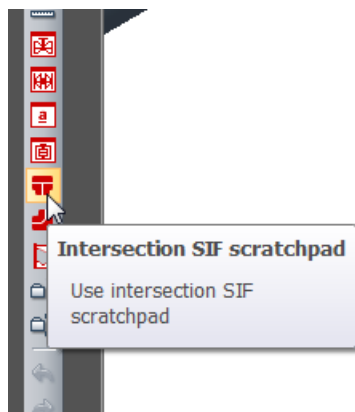
(3)Stresses									
CAESAR II 2011 SP1 Ver.5.30.01, (Build 110228) Date: JUN 3, 2011 Time: 13:53									
Job: D:\TRAINING\CAESAR II\PIPE1									
Licensed To: Seat -- ID #51									
STRESSES REPORT: Stresses on Elements									
CASE 3 (EXP) L3=L1-L2									
NODE	Bending Stress N./sq.mm.	Torsion Stress N./sq.mm.	SIF In Plane	SIF Out Plane	Code Stress N./sq.mm.	Allowable Stress N./sq.mm.	Ratio %	Piping Code	
Piping Code: B31.3 = B31.3 -2008, December 31, 2008									
CODE STRESS CHECK FAILED : LOADCASE 3 (EXP) L3=L1-L2									
Highest Stresses: (N./sq.mm.)									
CodeStress Ratio (%): 161.6 @Node 70									
Code Stress: 439.7 Allowable: 272.0									
Axial Stress: 12.5 @Node 138									
Bending Stress: 438.4 @Node 70									
Torsion Stress: 24.7 @Node 40									
Hoop Stress: 0.0 @Node 20									
3D Max Intensity: 634.0 @Node 70									
70	438.37	-16.85	4.625	5.833	439.66	272.04	161.61	B31.3	*
70	438.37	16.85	4.625	5.833	439.66	272.04	161.61	B31.3	*
70	393.11	0.00	4.625	5.833	393.11	281.83	139.49	B31.3	*
10	298.68	21.69	1.000	1.000	301.81	272.20	110.88	B31.3	*
80	298.68	-21.69	1.000	1.000	301.81	272.20	110.88	B31.3	*
150	257.31	-0.00	1.000	1.000	257.31	275.55	93.38	B31.3	
20	223.84	-21.69	1.000	1.000	228.01	282.78	80.63	B31.3	
90	223.84	21.69	1.000	1.000	228.01	282.78	80.63	B31.3	

The overstress points are at nodes 70, 10 and 80.

First we will check node 70. This is the tee. Look at the SIFs here. The in-plane SIF is 4.625 and the Out-Plane is 5.833. The stresses at this point are therefore being multiplied by 4 and 5 times. If we can reduce these SIFs then the stress will reduce and can easily be reduced below the code allowable.

Node	Bending Stress N./sq.mm.	Torsion Stress N./sq.mm.	SIF In Plane	SIF Out Plane	Code Stress N./sq.mm.	Allowable Stress N./sq.mm.	Ratio %	Piping Code	
70	438.37	16.85	4.625	5.833	439.66	272.04	161.61	B31.3	*
70	393.11	0.00	4.625	5.833	393.11	281.83	139.49	B31.3	*
10	298.68	21.69	1.000	1.000	301.81	272.20	110.88	B31.3	*
80	298.68	-21.69	1.000	1.000	301.81	272.20	110.88	B31.3	*
150	257.31	-0.00	1.000	1.000	257.31	275.55	93.38	B31.3	
20	223.84	-21.69	1.000	1.000	228.01	282.78	80.63	B31.3	

Return to the input and return back to node 70. Pick the Intersection SIF scratchpad and choose node 70.



Change the unreinforced tee to a reinforced tee. Specify a pad thickness of 10mm

Review Intersection SIF's

Node:

Type:

Pad Thk:

Head

Click the Recalculate button and notice the SIFs reduce dramatically. Now the stresses will be multiplied by 2.887 and 2.415 rather than 4 and 5.

Header SIF(i):

Header SIF(o):

Branch SIF(i):

Branch SIF(o):

Re-run the analysis using the batch run feature. The Expansion case is still shown in red, indicating the system is still overstressed. But check the Stress report for the Expansion case and notice that only nodes 10 and 80 are overstressed.

NODE	Bending Stress N./sq.mm.	Torsion Stress N./sq.mm.	SIF In Plane	SIF Out Plane	Code Stress N./sq.mm.	All Stress N./sq.mm.
10	298.68	21.69	1.000	1.000	301.81	
80	298.68	-21.69	1.000	1.000	301.81	
150	257.31	-0.00	1.000	1.000	257.31	
70	228.64	-16.85	2.415	2.887	231.11	
70	228.64	16.85	2.415	2.887	231.11	

So what is happening at nodes 10 and 80?

Nodes 10 and 80 are the initial anchor locations, so we need to find out what is causing the overstress. This is in the expansion case, so if we recall the code equation for the expansion case:

$$S_E = \frac{[(i_i M_i)^2 + (i_o M_o)^2 + 4M_T^2]^{1/2}}{Z}$$

From this equation it can be seen that the dominant factor in the code equation is the bending moment – it is the only factor in the expansion case. So which bending moment is this, M_i , M_o or M_t ?

M_t is torsion, M_z and M_i and M_o are inplane and outplane – so vary dependant on the location. What we can see from the results though, is which bending moment is the highest in terms of our axes.

View the Expansion case, Restraint Summary report.



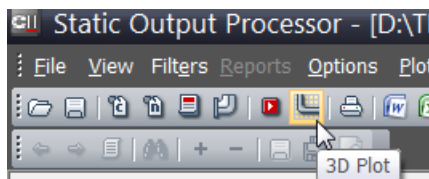
We can see from this report that at nodes 10 and 80, the highest bending moment is the MY moment, at 116 kN.m. The MX is also rather high at 88 kN.m.

(3)Restraint Summary							
CAESAR II 2011 SP1 Ver.5.30.01, (Build 110228) Date: JUN 3, 2011 Time: 14:12							
Job: D:\TRAINING\CAESAR II\PIPE1							
Licensed To: Seat -- ID #51							
RESTRAINT SUMMARY REPORT: Loads On Restraints							
CASE 3 (EXP) L3=L1-L2							
NODE	Load Case	FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.
LOAD CASE DEFINITION KEY							
CASE 3 (EXP) L3=L1-L2							
10	Rigid ANC						
	3 (EXP)	70590	-58673	7023	-88663	-116451	21262
80	Rigid ANC						
	3 (EXP)	-70590	-58673	7023	-88663	116451	-21262

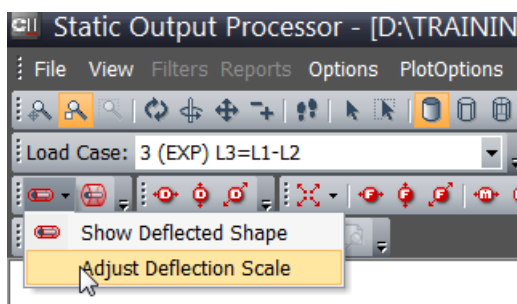
So we know what is causing the overstress, but how do we correct this and reduce the bending moment (and therefore the stress)?

Let us look at the 3D plot to view what is causing the bending moment.

Close the report and view the 3D plot

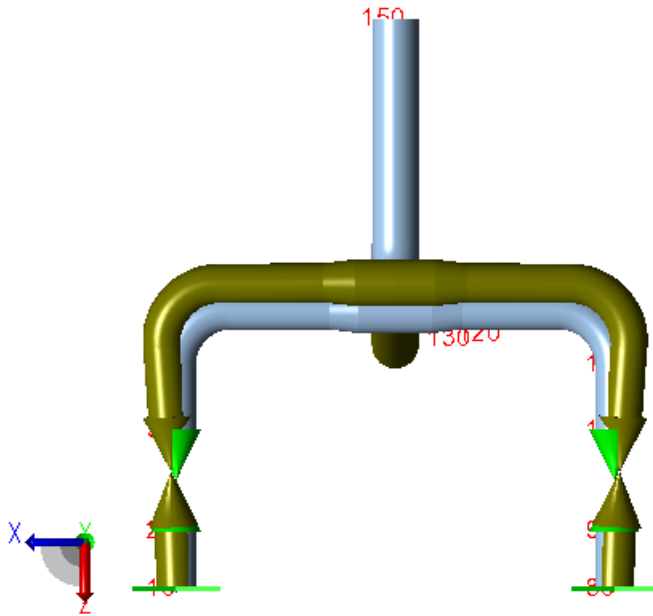


In the 3D plot window which appears, ensure that the Load case we are viewing is the expansion case and select to Show the deflected shape. You may need to Adjust the deflection scale to get a more exaggerated deflected shape.



View the pipe from the bottom, using the standard views available



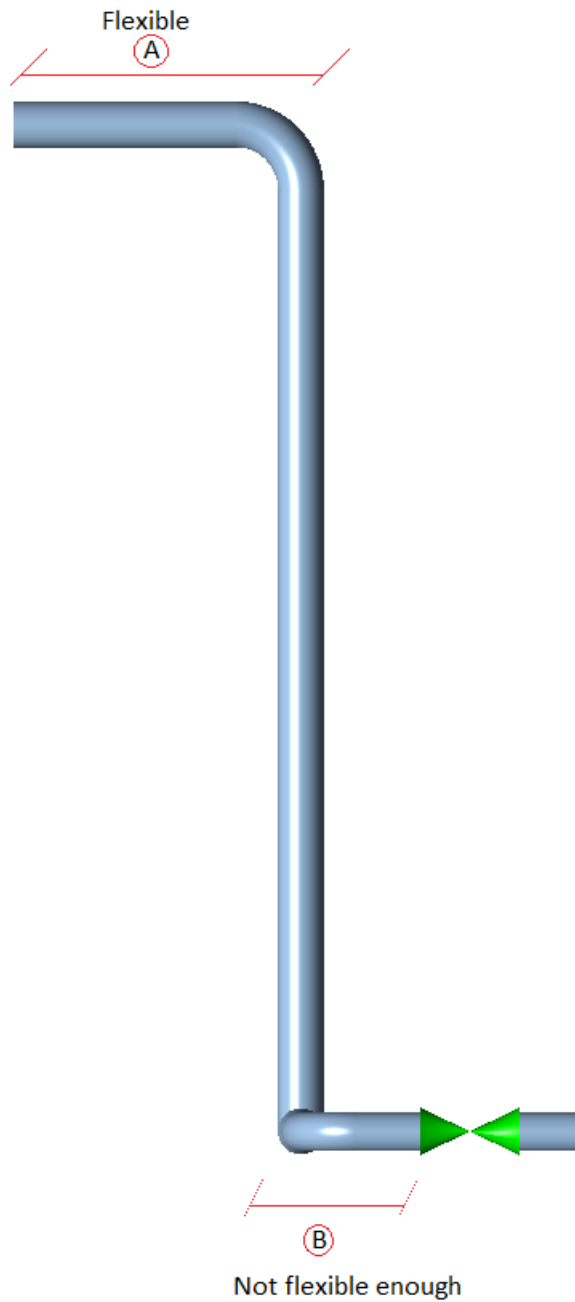


We are looking upwards at the pipe from below. The Y axis is pointing upwards (away from us). The pipe is undergoing thermal expansion and causing the pipe to bend at the anchor points.

Looking at the model from the side view will also explain the MX moment. As can be seen, the riser is expanding causing the MX bending moment.



If we could add some flexibility in to the area where the pipe is expanding we can absorb some of this expansion, and so reduce the bending moment. We have the top leg which is flexible, but the bottom leg is not flexible enough.



If we can transfer some of the flexibility in A to B then we may solve the problem.

To do this, increase the length of B by 1m and so consequently reduce the length of A by 1m. This should give us more flexibility at the bottom, hopefully without removing too much flexibility at the top.

Return to the input and select element 30 to 40. The DZ value here is 825mm. Edit this to 1825mm.

Edit Node Numbers

From: 30
To: 40

☐ Name

Edit Deltas

DX:
DY:
DZ: -1825.000 mm

☐ Offsets

Repeat for element 100 to 110.

We will have to also consequently reduce the length of 140 to 150 by the same amount (1m).
Change this from 2000mm to 1000mm.

Edit Node Numbers

From: 140
To: 150

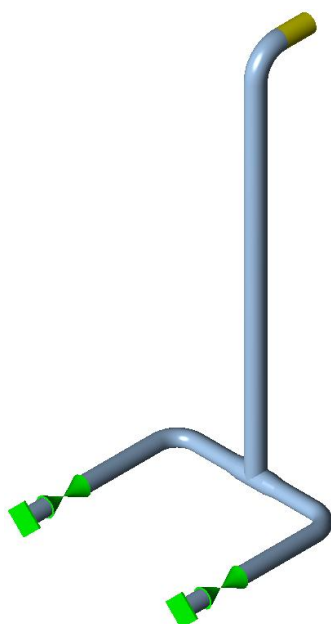
☐ Name

Edit Deltas

DX:
DY:
DZ: -1000.000 mm

☐ Offsets

The model should now look like the one below.



Re run the analysis and check the results. The expansion case is now no longer coloured red and the highest code stress is 91% of the allowable at node 10. We have successfully reduced the stresses and the model now passes the code stress checks.

Verify the sustained stress report that this is still acceptable – it should be around 37% of the allowable.

Our model is now compliant with B31.3.