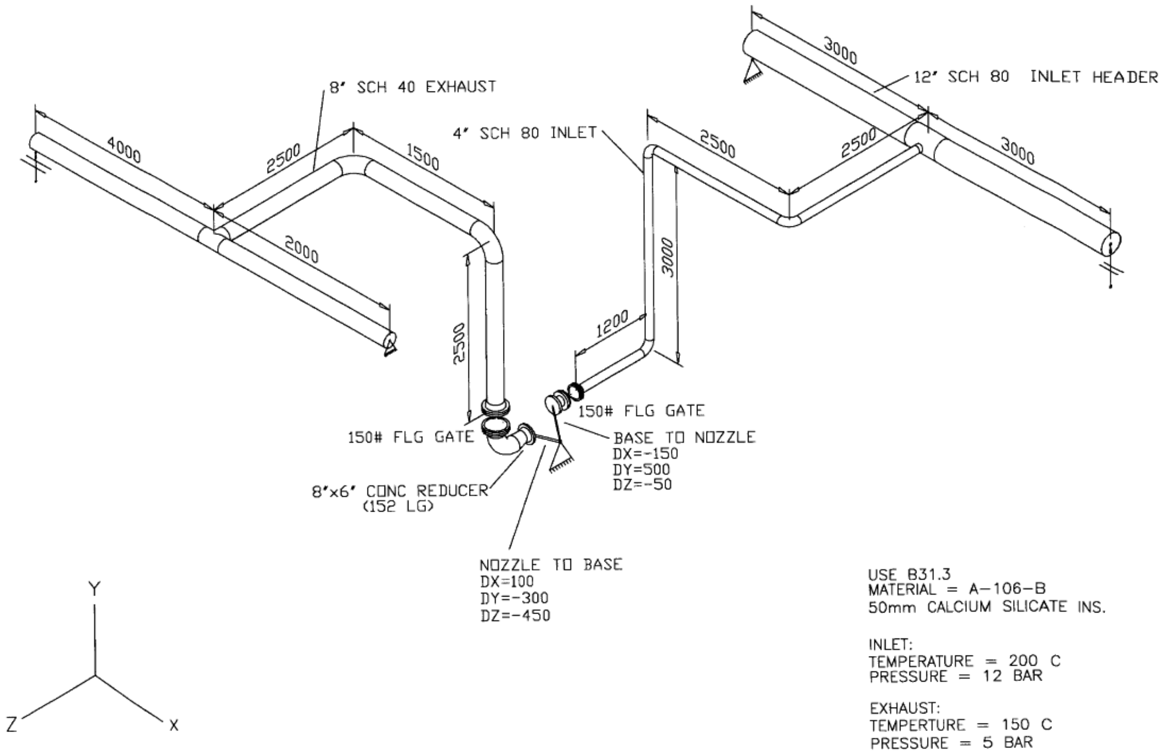
Turbo

This exercise will build upon knowledge we have gained so far, and introduce the following new features.

- Combining Pipe Models
- Rigid Construction Elements
- CNode Connections
- NEMA SM23 Analysis



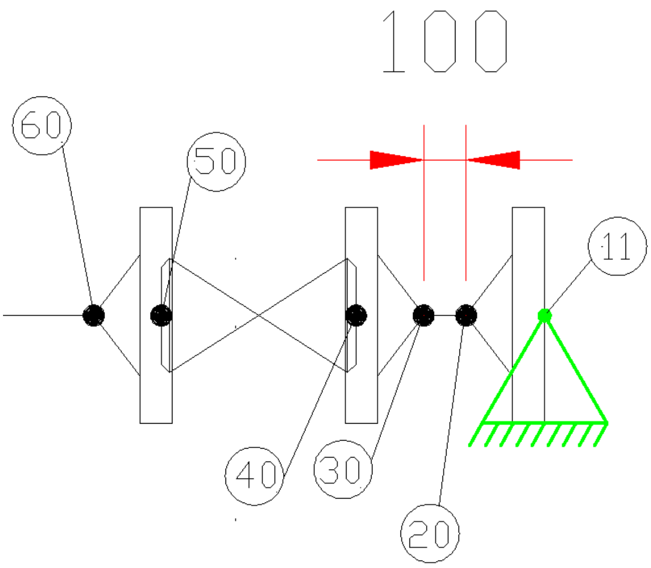
The model is shown above and is also supplied in the hand-out in a larger format. As can be seen, this is an inlet and exhaust on a turbine. The turbine itself is anchored to the floor at the anchor point specified.

To keep modelling simple, we can model the inlet and exhaust as two separate models. These two models can be combined into a single model later – there is a common location at the turbine anchor point.

We will model this common anchor point location as node number 5. However, we will come back and model this point last.

Model Inlet

Model the inlet pipework, on the first element; change 10 to 20 to 11 to 20. Begin at node 11, the flange connection to the turbine nozzle, as shown below. Note that node 11 also has an anchor attached.

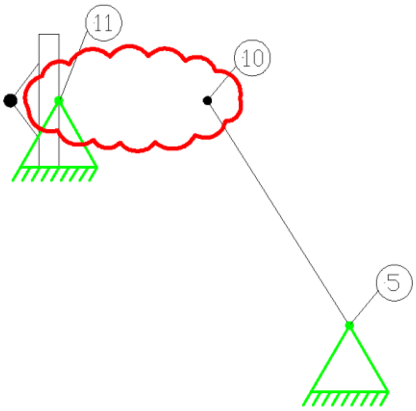


Once the inlet piping is complete, return to the first element, 11 to 20. We will now insert an element before 11 to 20, numbered 5 to 10. Node 5 will be the turbine anchor location. Node 10 will be connected to the anchor at node 11. We will connect these via CNode connections.

As there is an anchor at node 11, this node is fixed in all 6 degrees of freedom. However, the nozzle connection on the turbine will undergo some movement due to the thermal expansion of the turbine. Recall previously that we had a similar situation where displacements were specified to account for a similar situation in PIPE 1 for the vessel thermal expansion. In this case, we do not know what the thermal expansion is here so we cannot enter the displacements. To apply the displacements to the nozzle point we will add in an element from the nozzle to the turbine base. This will be a rigid element with zero weight, and it will have temperature applied (so it will undergo thermal expansion). This element will also be anchored at the turbine base.

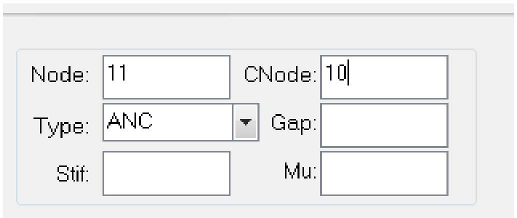
If however we specified this as 5-11 for example, nodes 5 and 11 are both anchored, so cannot move and all that will happen is we will obtain large forces at nodes 5 and 11. We need a way of connecting this rigid construction element to node 11 which will allow node 11 to move due to the expansion of the construction element, but still be fixed as far as the rest of the piping is concerned.

This can be achieved in CAESAR II this using CNodes. A CNode will allow the anchor at node 11 to be connected to another node "indirectly". Essentially, both these points will be at the same location, but CAESAR II will see these points as separate nodes. Therefore what will happen is that as element 5-10 expands, node 11 will act as an anchor, but as it is connected to node 10, node 11 will move to wherever node 10 moves to, while still acting as an anchor.

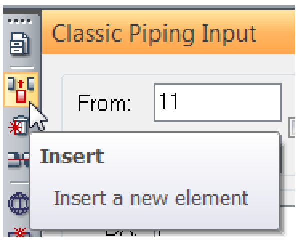


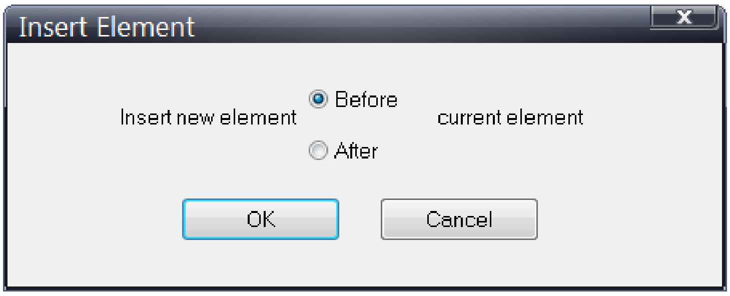
Nodes 10 and 11 are the same point in space.

Select element 11-20. First specify a CNode on the anchor at node 11; the CNode will be node 10.

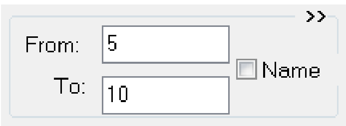


Next use the Insert function to insert an element **before** this element.

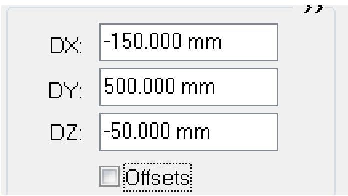




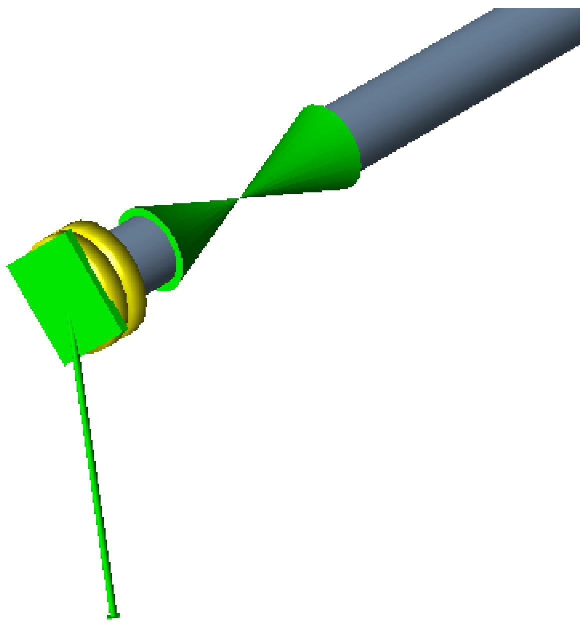
Change the node numbers to 5 to 10



Fill in the DX, DY, DZ fields as shown on the isometric sketch

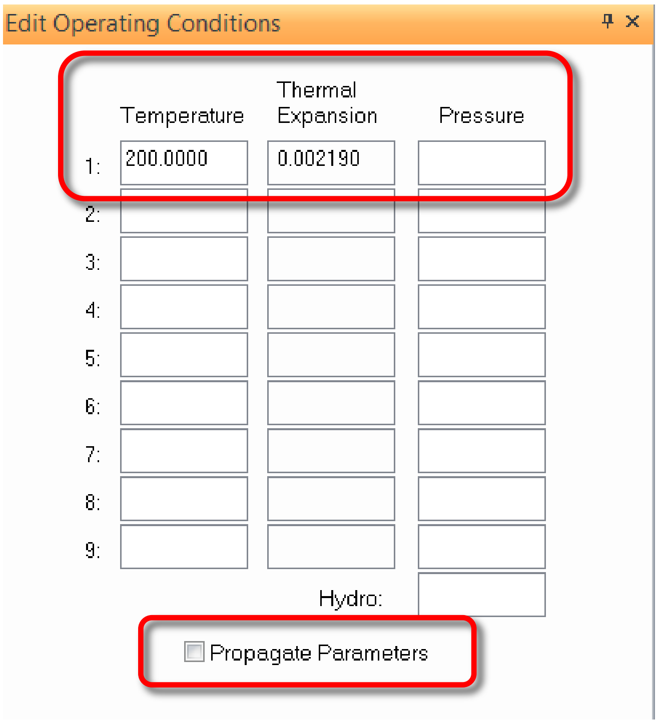


This is a rigid element, but leave the weight blank in this case. Also add an anchor at node 5.



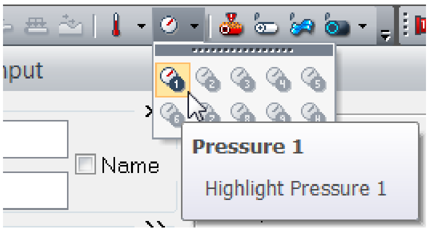
The zero weight rigid will be inserted and connected to node 11 as above. We now need to specify the temperature for the rigid, as this will undergo thermal expansion.

Select element 5-10 and double click the “>>” on the temperature/pressure section to “tear off” the Operating Conditions box. Uncheck the Propagate parameters box to ensure that the data entered here is only entered on this one element, and not the whole model. Enter 200°C for T1 and make sure that no pressure is specified.

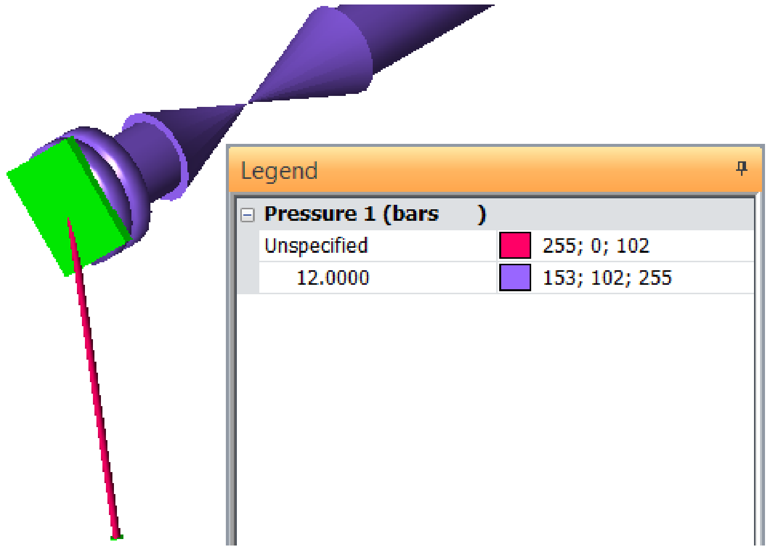


Check that this has been specified correctly, by using the Show Pressure graphical tool.

Display P1.



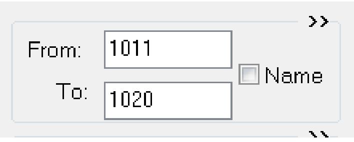
The rigid construction element should show as unspecified, and the rest of the model should show as 12 bars.



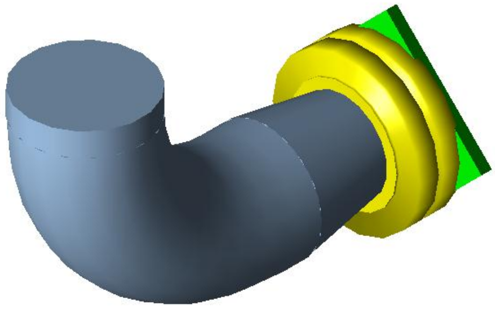
The inlet is now complete.

Save the file and using the same concept, create a new job file for the exhaust.

Start the exhaust using nodes 1011 to 1020 for the first element.



Continue on. After the reducer, there is a bend. Place a length of pipe 350mm long in the Z direction, adding a bend to this pipe. Place a corresponding pipe 350mm long in the Y direction afterwards to complete the bend.



Complete the rest of the exhaust.

Once finished, add one final element, inserted before 1011 to 1020 and numbered 5 to 1010. Set the pressure for this element to blank, and ensure that only this element has no pressure.

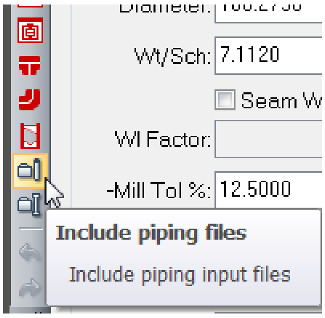
Connect via a CNode of 1010 at the anchor at node 1011.

The exhaust is now also complete. Notice that we have a node numbered 5 in both our inlet and exhaust models. This node is a common location in both files. As such we can combine the models and they will be linked at node 5.

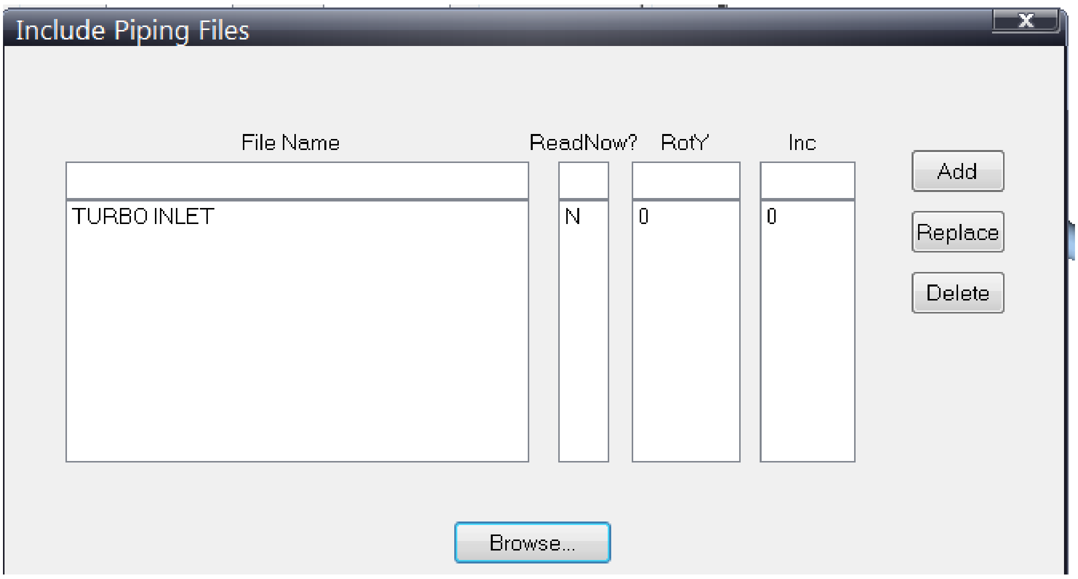
Combine Models

Save the Turbine exhaust as Turbo_Combined.c2

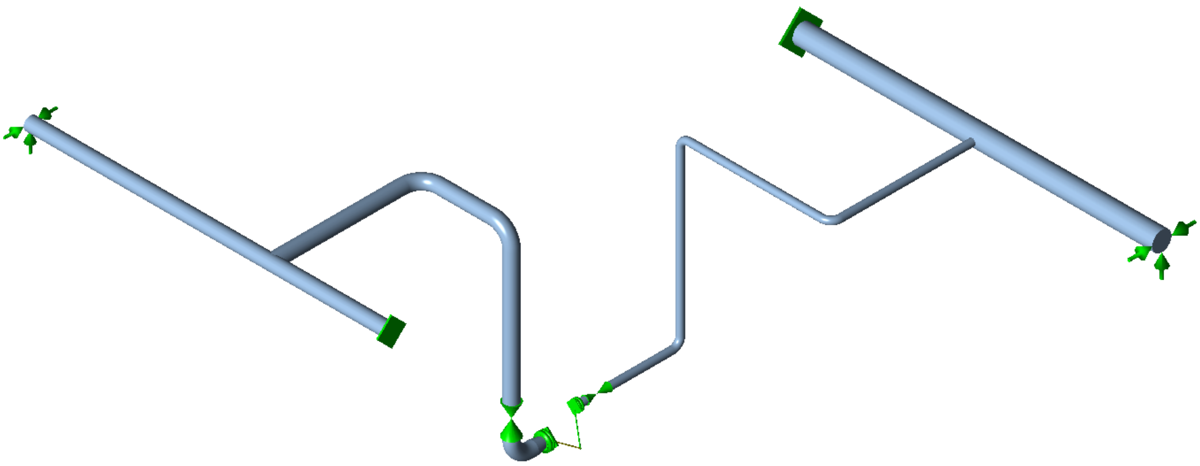
Select the Include Piping Files button to bring in the Inlet piping to create one single model.



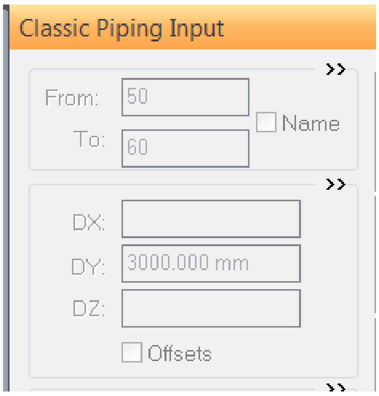
Browse for the inlet model and click OK.



The two models will be combined around node 5 – the common node in both files.



Select an element in the inlet. Notice that it can be selected, but the input screen has all fields disabled.



This is because currently the inlet is simply being "referenced" and not fully included. Return back to the combine pipe models screen. As well as the File name field, there are three other columns.

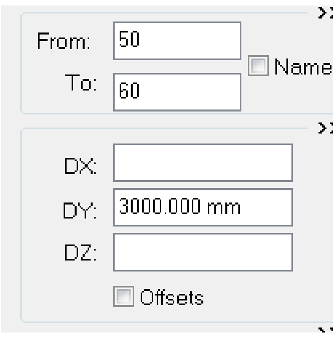
RotY – allows the included file to be rotated by a specified angle about the Y axis.

Inc – allows the node numbers in the new file to be incremented by a certain value. i.e. if the file included was 10-20, 20-30 etc., we could increment by 1000 so that the nodes would become 1010-1020, 1020-1030 etc.

The final column is Include Now? This column allows users to choosing whether to reference or actually permanently include the file. This can be useful if checking that the file is correct before including, so any changes required can be done in the original separate model.

If your model looks correct, change Include Now? to Y.

Selecting an element in the inlet side of the piping will now show all the fields to be enabled.



Error Check and analyse the model, use the recommended load cases.

The analysis results should show that the expansion and Sustained stresses are acceptable. Check the OPE case and note the loads on the nozzles at nodes 11 and 1011.

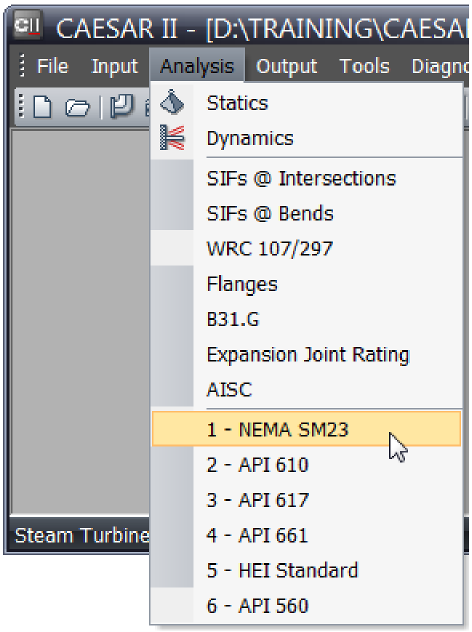
NODE	Load Case	FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.
11		Rigid ANC					
	1(OPE)	-457	-3363	508	-3309	406	141
90		Rigid ANC					
	1(OPE)	457	-7855	-298	460	536	-9602
100		Rigid GUI; Rigid +Y					
	1(OPE)	0	-4876	-210	0	0	0
1011		Rigid ANC					
	1(OPE)	905	-7288	-2021	1010	-736	-398

There is a reasonable force on these nozzles, as the anchor points are supporting some of the weight of the pipe. We will now take these forces and check them against NEMA SM23 to analyse the turbine to see if the loads on the nozzles are acceptable.

NEMA SM23

The National Electrical Manufacturers Association (NEMA) publishes and maintains a standard SM23 Steam Turbines for Mechanical Drive Service. This standard includes allowable loads that can be applied safely to our turbine nozzles. CAESAR II incorporates this standard and a separate module can be used to evaluate nozzles, using calculated loads from the piping analysis (as we have just done).

Return to the CAESAR II main window. Select the Analysis menu and select the NEMA SM23 option to load the module. You may need to use the to access the whole menu.

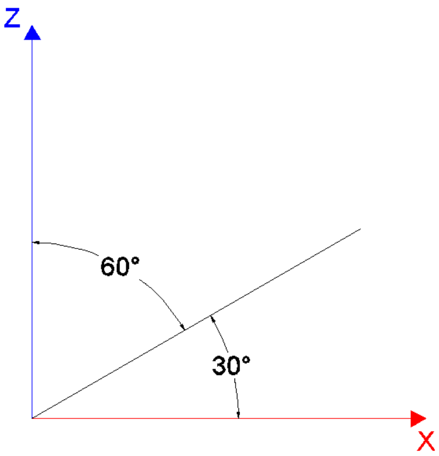


Create a new file called pass1.

If you wish, give the equipment a short description.

Select the NEMA Input Data tab.

The first data to enter is the direction of the equipment centreline. This must be specified as the direction Cosines.



The direction cosines are specified as the cosine of the angle of the centreline with the respective axis. For example, if the centreline was as shown above, 60° from the Z axis and 30° from the Y axis, the direction cosines would be:

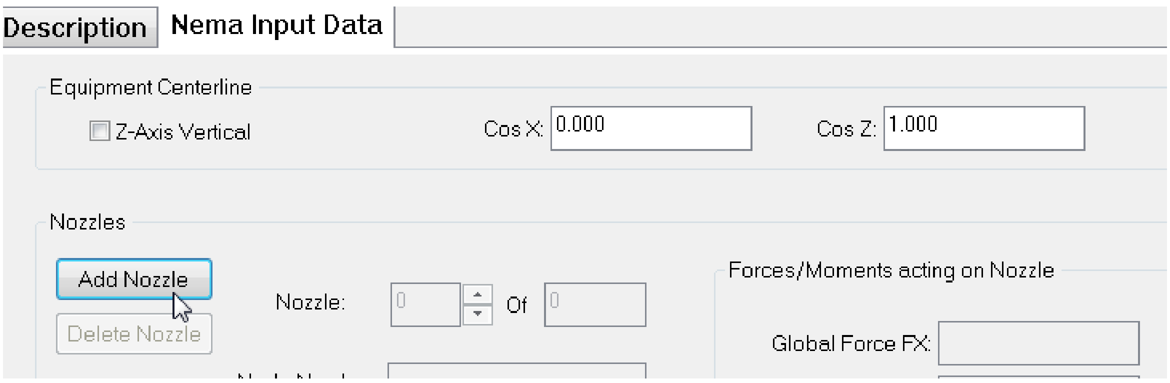
Z = Cos 60° = 0.5
X = Cos 30° = 0.866

Our equipment centreline is along the Z axis, so the angle between the centreline and the Z axis is 0° and the angle between the centreline and the X axis is 90°

Therefore the direction cosines are:

Z = Cos 0° = 1
X = Cos 90° = 0

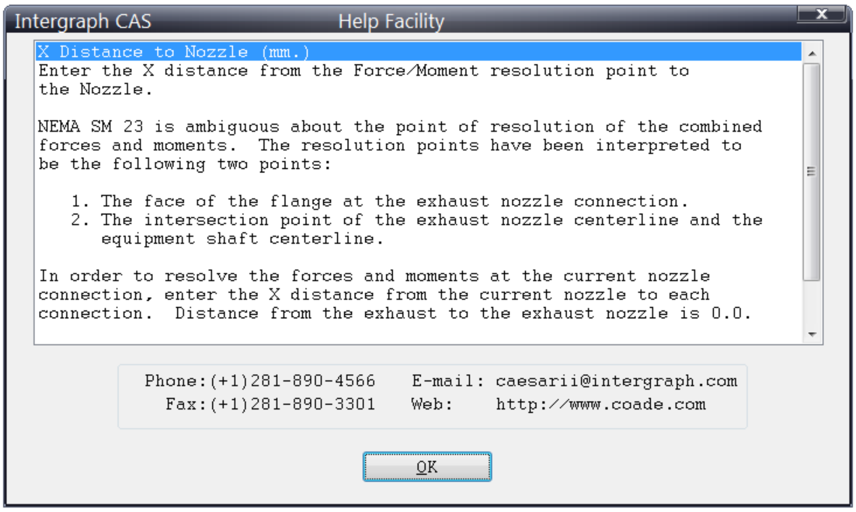
Now we can define the nozzles. Click the Add Nozzle button



The Node number is the corresponding node number which matches the node number in the CAESAR II model. Specify the Inlet first – so this node number is 11.

This is the Inlet nozzle and is 4" diameter

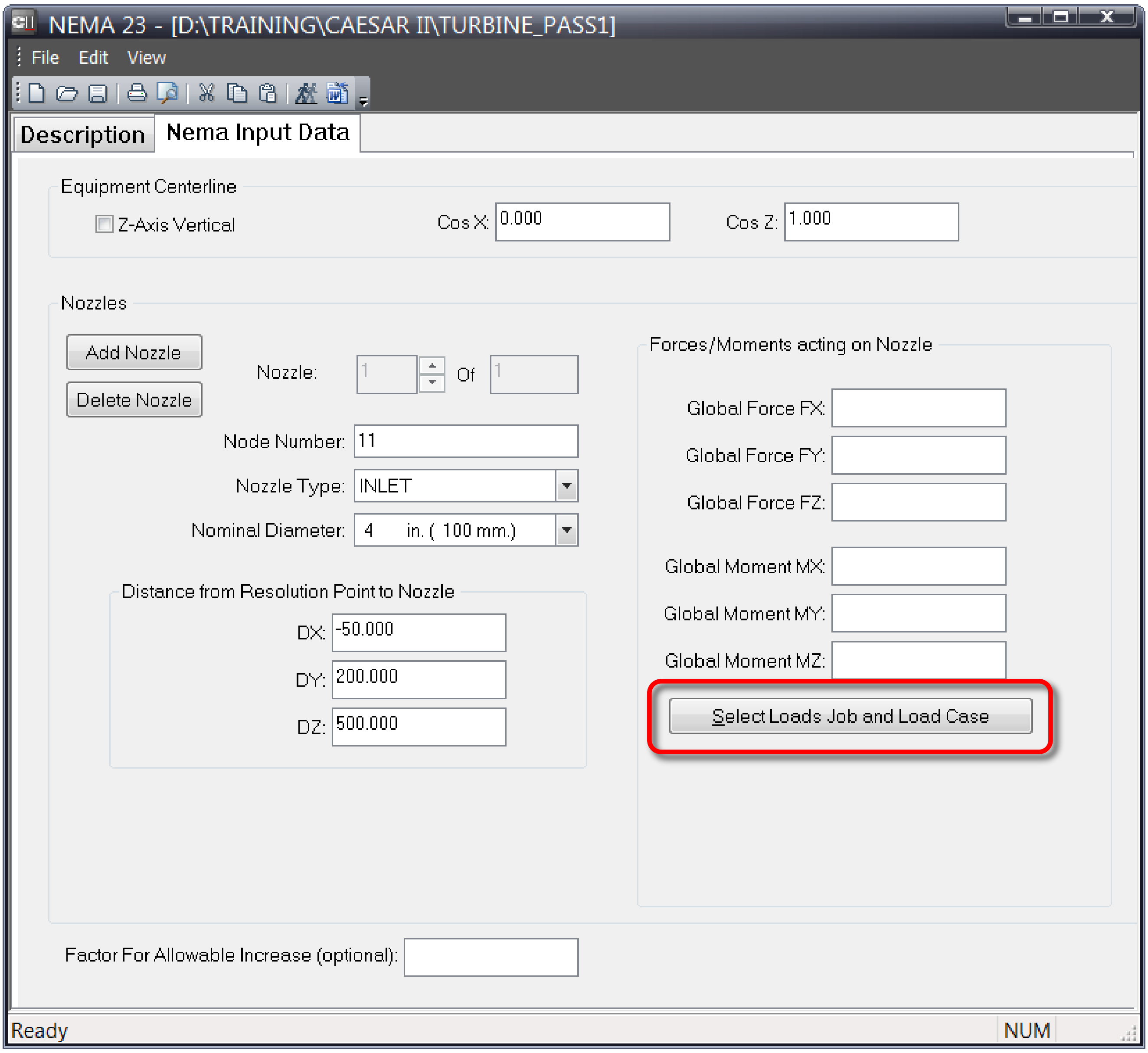
The next box is the Distance from Resolution Point to Nozzle. NEMA SM23 is ambiguous about the point of resolution of the combined forces and moments. Select one of the three fields and hit F1 to bring up the help.



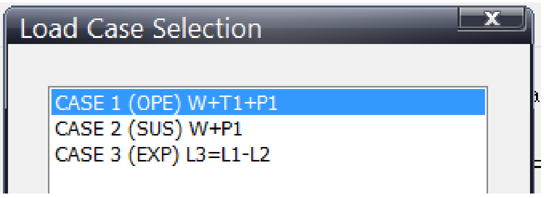
This point is the distance from the resolution point (face of the exhaust nozzle flange) to the nozzle at node 11. This is as follows:

DX = +100 -150 = **-50 mm**
DY = -300 + 500 = **+200 mm**
DZ = +450 + 50 = **500 mm**

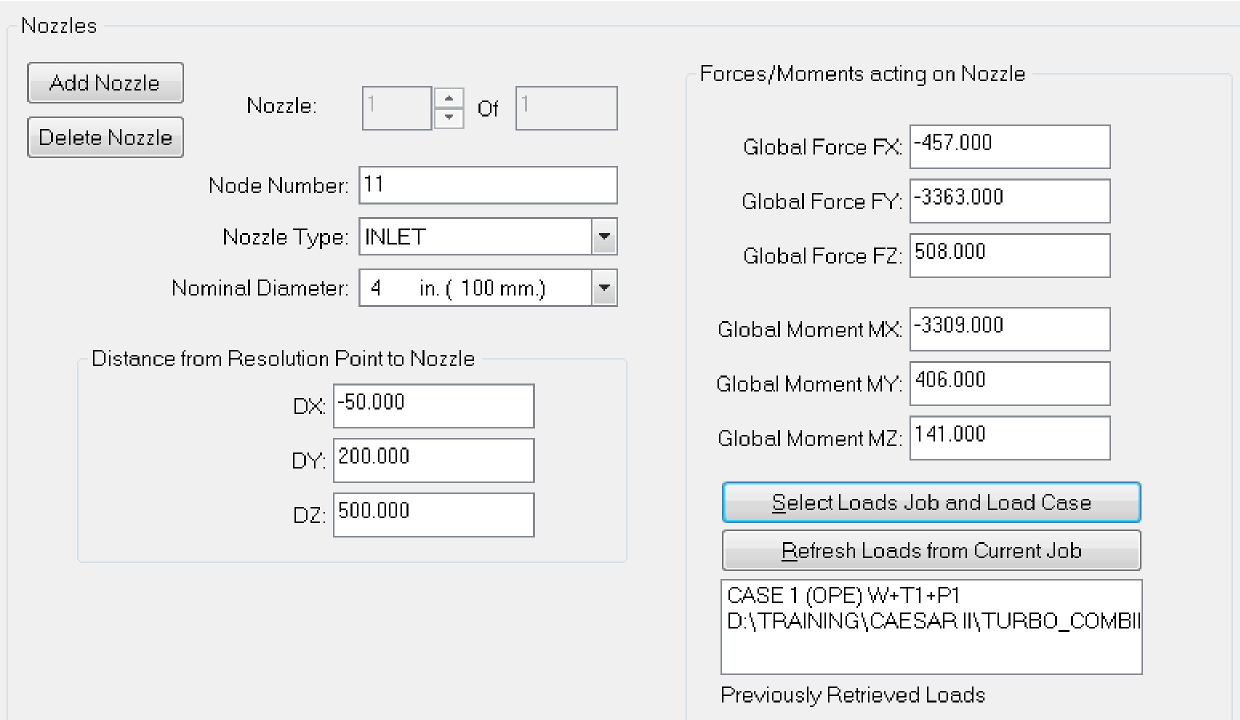
The basic nozzle information is input. The final thing to do for this nozzle is to apply the loads on this nozzle. We already have the loads in the CAESAR II job file that we have just analysed. So these loads can be imported straight into NEMA SM23 module. Click the Select Loads Job and Load Case button.



Browse to the combined turbine model and select the operating case.

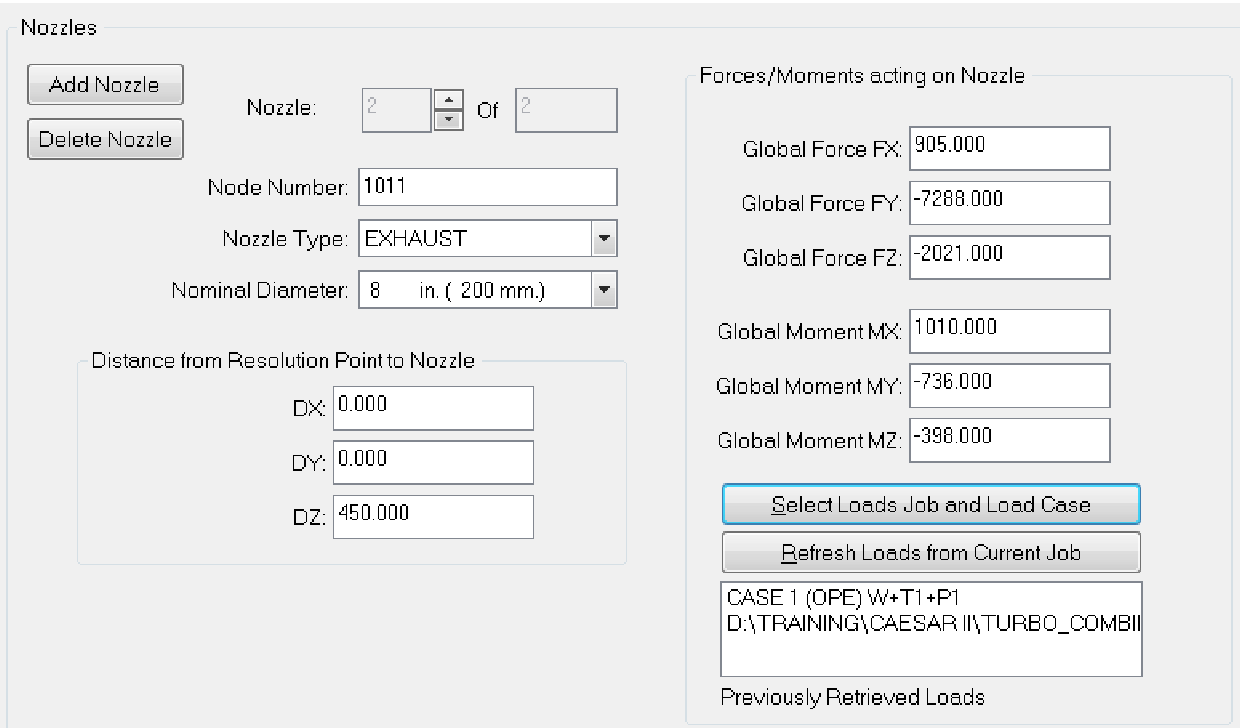


The loads will be imported from the job, these will be the loads at node 11 (as we specified the node number as 11).

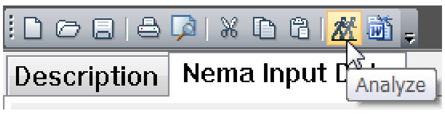


The inlet nozzle is now complete. Select to add a new nozzle and enter the Exhaust nozzle details.

As before, import the loads from the OPE case in the combined job file.



Once complete, analyse the model using the “Running Man” icon.



Another tab will now be present showing the results of the analysis. The analysis fails.

```
-----------------------------------
      Overall Status **FAILED**
-----------------------------------
```

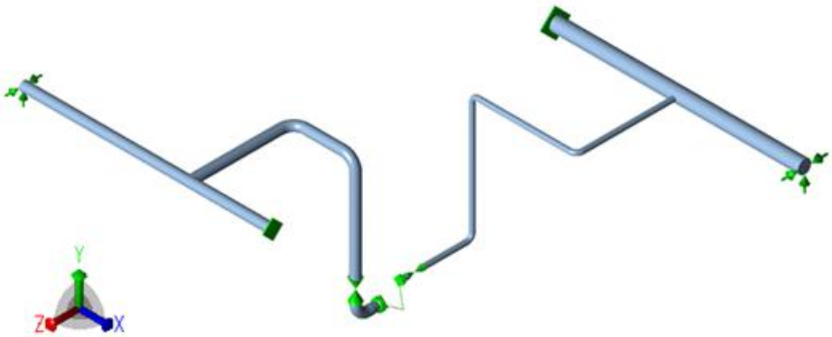
The Inlet and Exhaust both fail the analysis. A closer look at the results shows that the FY and MZ moments are by far the leading causes of the failure. The FY is 214% of the allowable, while the MZ is also 182%.

```
-----------------------------------------------------------------------------
SFX         =    -1513      -340     50*DC =      447     76.06
SFY         =   -10651     -2395    125*DC =     1118    214.18    **FAILED**
SFZ         =     -448      -101    100*DC =      894     11.26
FC(RSLT)    =    10767      2421

SMX         =        3         2    250*DC =     2236      0.08
SMY         =     -126       -93    125*DC =     1118      8.30
SMZ         =    -2764     -2039    125*DC =     1118    182.33    **FAILED**
MC(RSLT)    =     2767      2041

2FC + MC    =               6882    250*DC =     2236    307.77    **FAILED**
```

Save and Close the NEMA module and return to the piping input.

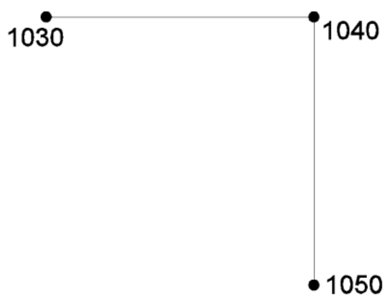


Looking at the input, it can be seen that the weight of the piping causing the FY will also clearly result in the MZ moment being excessive – there is very little supporting the weight of the piping other than the nozzles themselves. The only other supports are on the headers.

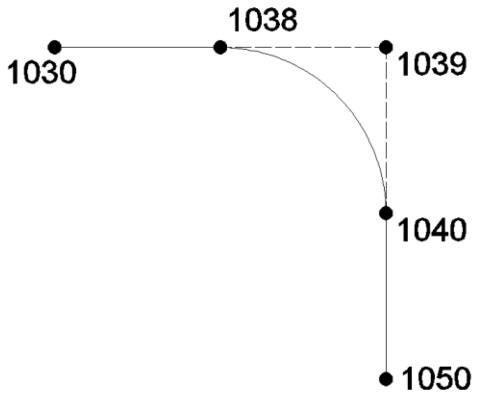
We need to support the weight of the pipe. Let us focus on the Exhaust first.

Let us support the weight of the pipe to reduce the FY component. Locate a Y support below the elbow. This will be node 1039. We have not defined node 1039, only 1030 and 1040; so why 1039?

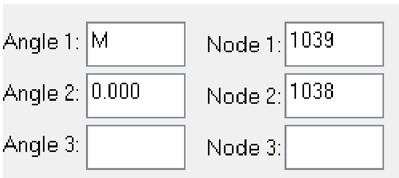
When building the model, it is built in the following way:



Once a bend is inserted, CAESAR II moves node 1040 to the end of the bend, and inserts two intermediate nodes:

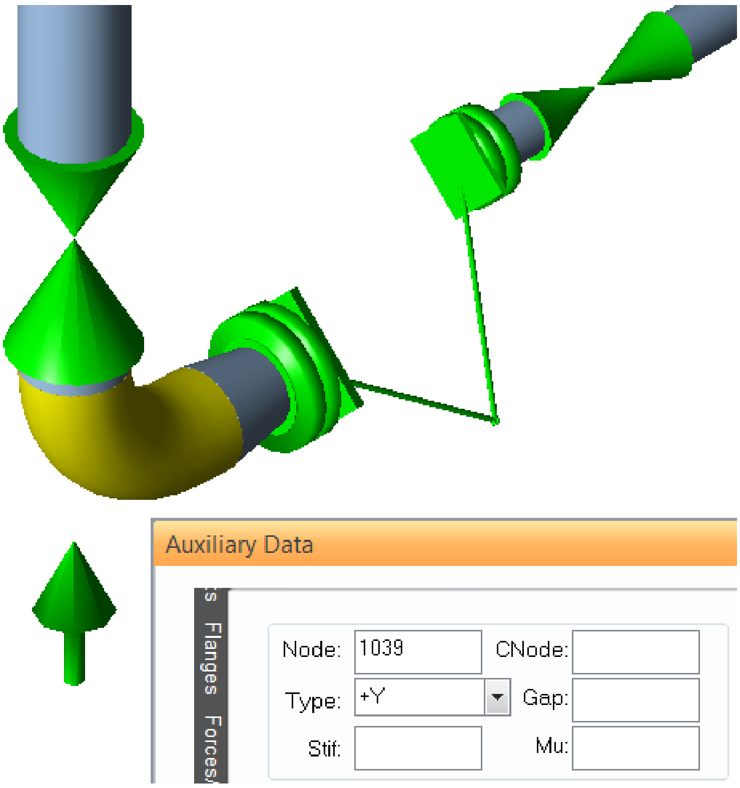


These intermediate nodes are added by default at 0° around the bend, and at the mid-point of the bend. This can be seen in the Bend auxiliary data (single click on the bend check box on element 1030 – 1040). M is used to designate the mid-point of the bend.



These nodes can be moved to the required location if needed for any reason by simply editing the angle. But we just wish to add a +Y support under the mid-point of the bend, at node 1039.

Add a +Y at 1039.

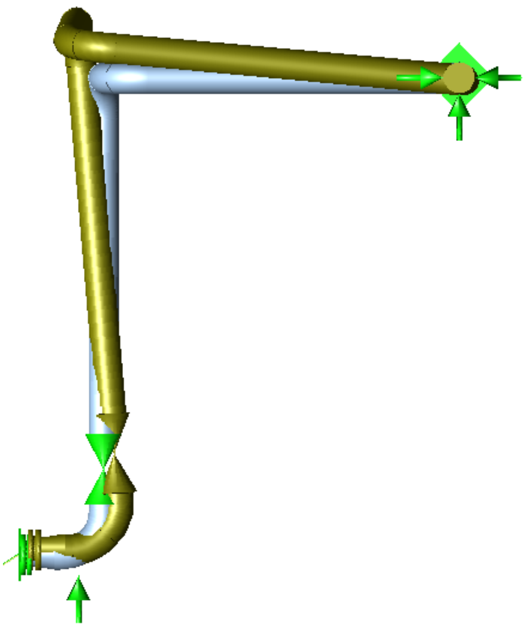


Re-run the analysis.
Check the SUS and expansion stresses, these should again be OK. Now review the OPE case restraint summary to check if the load on node 1011 has reduced.

The loads on node 1011 are the same. The loads on 1039 are 0. This restraint is not taking any load in the OPE case.

NODE	Load Case		FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.
1011		Rigid ANC						
	1(OPE)		905	-7288	-2021	1010	-736	-398
1039		Rigid +Y						
	1(OPE)		0	0	0	0	0	0

View the 3D plot to see the reason for this. View the deflected shape (you may need to increase the deflection scale). From a left view, it can be seen that the expansion is causing the pipe to lift off the support at node 1039.



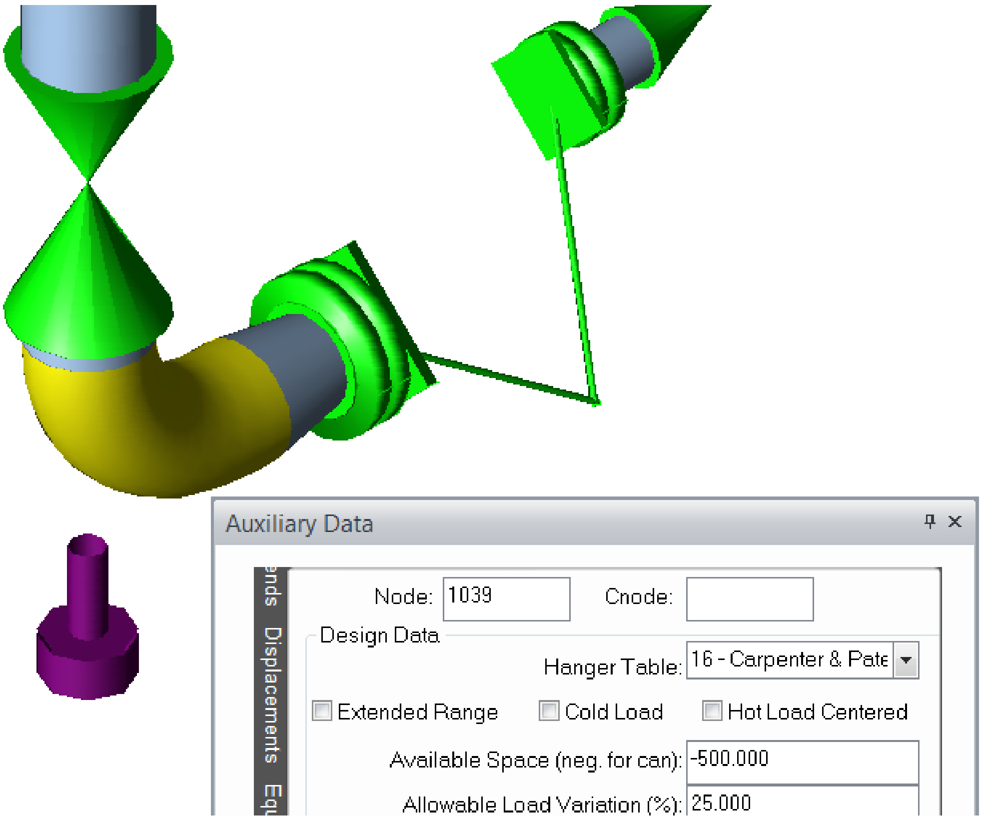
The Operating/Displacements report confirms this, node 1039 is experiencing 0.614mm displacement upwards – lift off.

NODE	DX mm.	DY mm.	DZ mm.	RX deg.	RY deg.	RZ deg.
1039	-0.196	0.614	1.440	-0.0289	-0.0165	0.0013

The load on node 1011 has not changed. As such there is no need to run another NEMA analysis – the results will be identical. Therefore, return to the input.

Change the design by replacing the Y restraint with a spring hanger.

Select a Carpenter and Paterson spring hanger, and enter -500 in the Available Space field. The can will be shown as below.



Re-run the error check and visit the Load cases window. We have added spring hanger into our model, so we require` the hanger cases to determine the hanger size. Use the recommended load cases again and analyse.

Cas...	Load Case	Description
1	W(HGR)	WEIGHT FOR HANGER LOADS
2	W+T1+P1(HGR)	OPERATING FOR HANGER TRAV..
3	W+T1+P1+H(OPE)	OPERATING CASE CONDITION 1
4	W+P1+H(SUS)	SUSTAINED CASE CONDITION 1
5	L3-L4(EXP)	EXPANSION CASE CONDITION 1

As before, the SUS and EXP cases are well within the code allowables. Check the restraint summary for the OPE case.

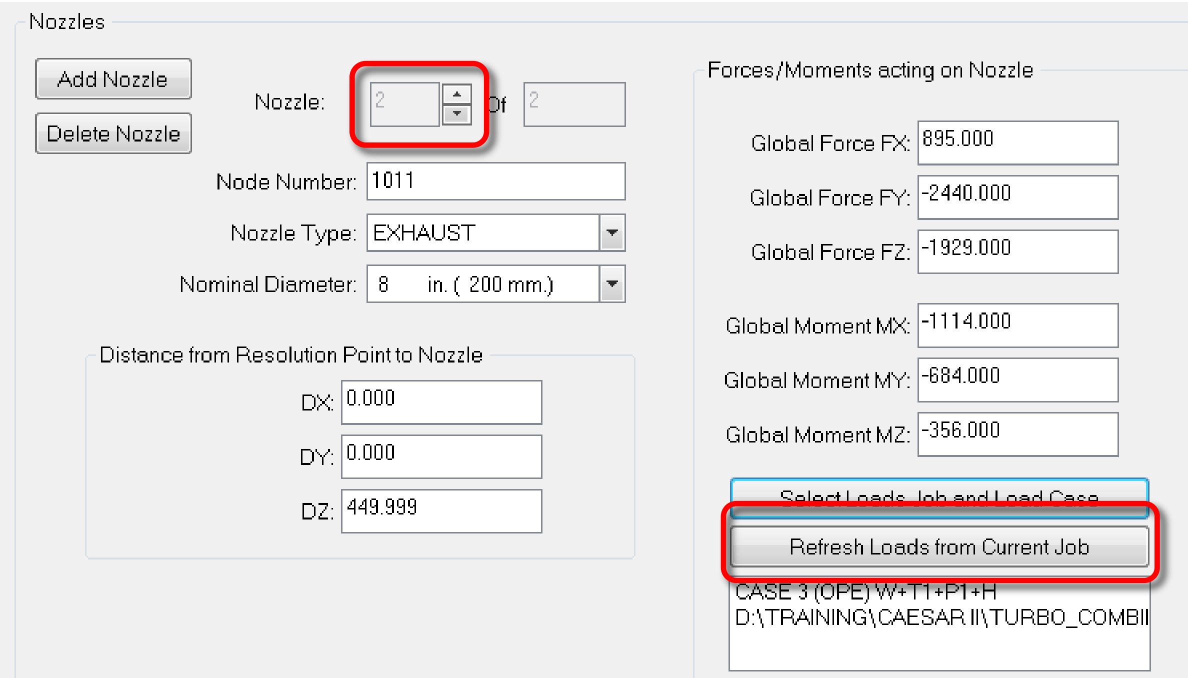
NODE	Load Case	FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.
1011		Rigid ANC					
	3(OPE)	895	-2440	-1929	-1114	-684	-356
1039		Prog Design VSH					
	3(OPE)	0	-4867	0	0	0	0

The spring hanger is taking some load now (4800N) and the load on the restraint at 1011 is much reduced.

Return to the NEMA module and we will re-analyse with our new loads.

Save As… on the existing NEMA file, and select Nozzle 2 of 2 – the exhaust. The Inlet has not changed so there is no need to re-import the inlet loads.

Refresh the loads from the current job to import the new changed data.



Once done, rerun the analysis. The exhaust now passes at 79% of the allowable.

```
EXHAUST        1011     FX=   -1929                     3F + M < 500*D(used)
                        FY=   -2440
                        FZ=    -895                     3F + M =      3182
                        Fr=    3237     F =      728
                                                   500*D(used) =      4000
                        MX=    -356
                        MY=    -684                % of ALLOW. =     79.56
                        MZ=    1114
                        Mr=    1355     M =      999

  Moments due to "Force Resolution"
                        MX=       0
                        MY=     403
                        MZ=   -1098
```

The Inlet of course still fails as we have not changed this.

Now let us reduce the load on the inlet. To do this, break element 40-50 with a new node number 42. This node should be 300m from node 40.

Locate a spring hanger at node 42. As before, select Carpenter & Paterson. Re-run the analysis and check the OPE restraint summary. The load on node 11 will be reduced.

NODE	Load Case	FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.
11	Rigid ANC						
	3(OPE)	-471	923	479	242	462	123
42	Prog Design VSH						
	3(OPE)	0	-4333	0	0	0	0

Return to the NEMA module and Save As… this file. Re-import the loads for the inlet nozzle and re-run the NEMA analysis.

The Inlet should now also pass.

```
INLET           11    FX=     479                      3F + M < 500*D(used)
                      FY=     923
                      FZ=     471                      3F + M =      1165
                      Fr=    1142     F =      257
                                               500*D(used)  =      2000
                      MX=     123
                      MY=     462                  % of ALLOW. =     58.26
                      MZ=    -242
                      Mr=     536     M =      395

  Moments due to "Force Resolution"
                      MX=      48
                      MY=    -212
                      MZ=     366
```

The whole turbine passes

```
 --------------------------------------------------------------------------
                         Overall Status **PASSED**
 --------------------------------------------------------------------------
```

The piping system is acceptable in within the code allowables, and the loads on the turbine are also within the allowables of NEMA SM23.