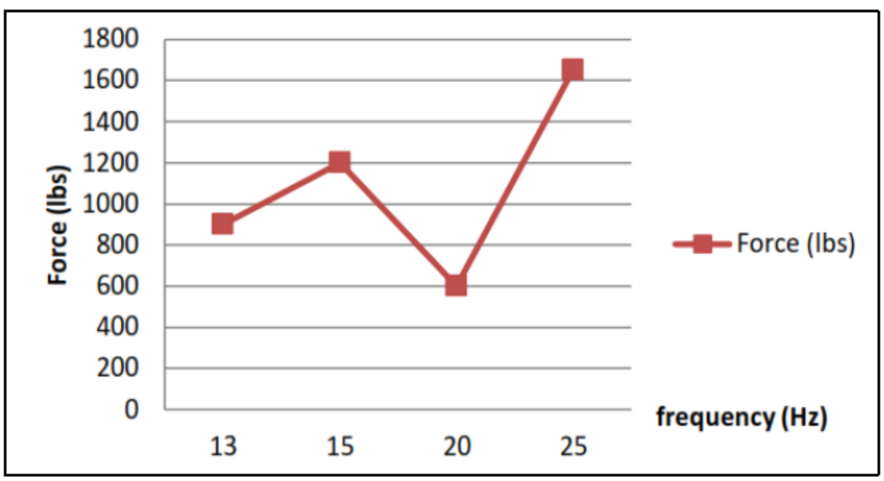
Tutorial for Force Spectrum Analysis using CAEPIPE

The following are the Steps for performing the Force Spectrum Analysis using CAEPIPE.

General

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

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



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

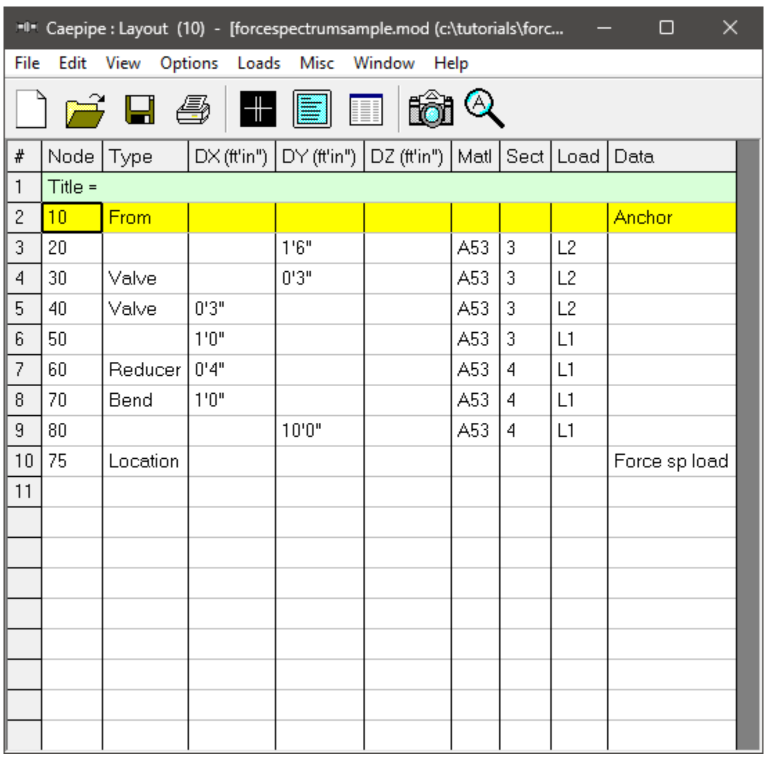
The results of the modal analysis are used with force spectrum loads to calculate the response (displacements, support loads and stresses) of the piping system. It is often used in place of a time-history analysis to determine the response of the piping system to sudden impulsive loads such as water hammer, relief valve and slug flow.

The force spectrum is a table of force spectral values versus frequencies that captures the intensity and frequency content of the time-history loads. It is a table of Dynamic Load Factors (DLF) versus natural frequencies. DLF is the ratio of the maximum dynamic displacement divided by the maximum static displacement. Note that Force spectrum is a non-dimensional function (since it is a ratio) defining a curve representing force versus frequency. The actual force spectrum load at a node is defined using this force spectrum in addition to the direction (FX, FY, FZ, MX, MY, MZ), units (lb, N, kg, ft-lb, in-lb, Nm, kg-m) and a scale factor.

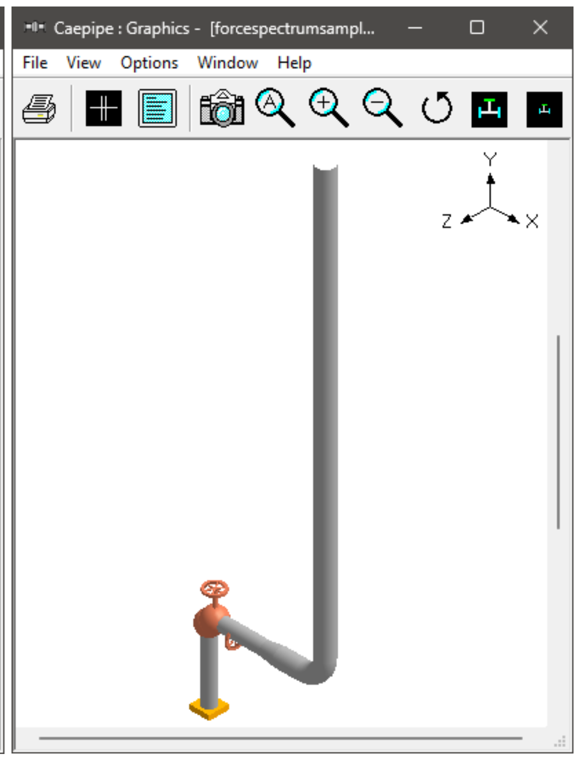
Tutorial

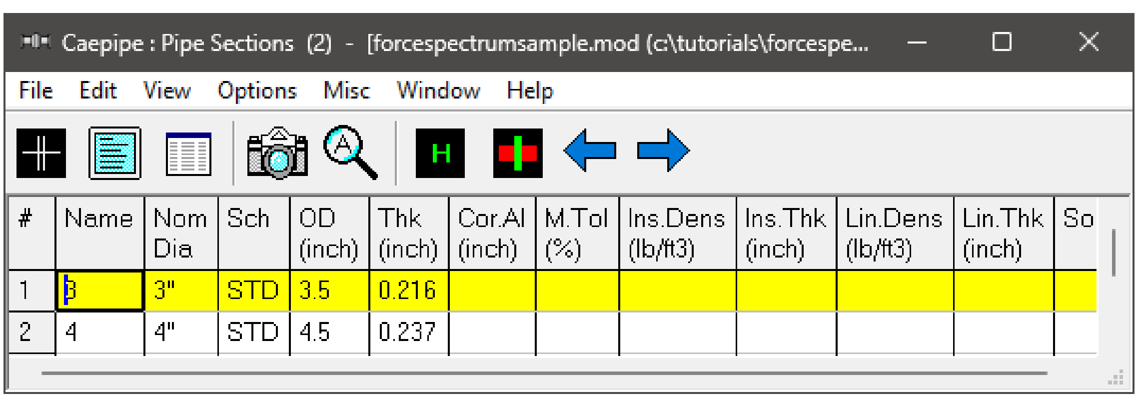
Step 1:

Attached is a sample CAEPIPE model with Force Spectrum. The steps followed in generating the model are shown in the snap shot below.

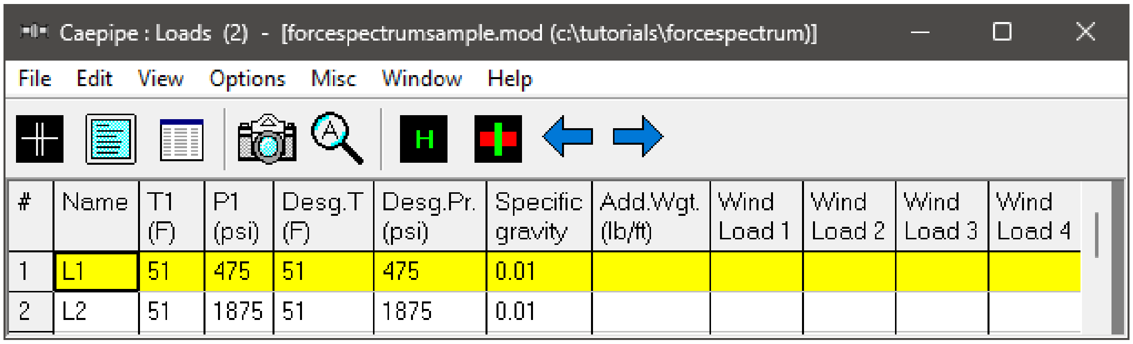


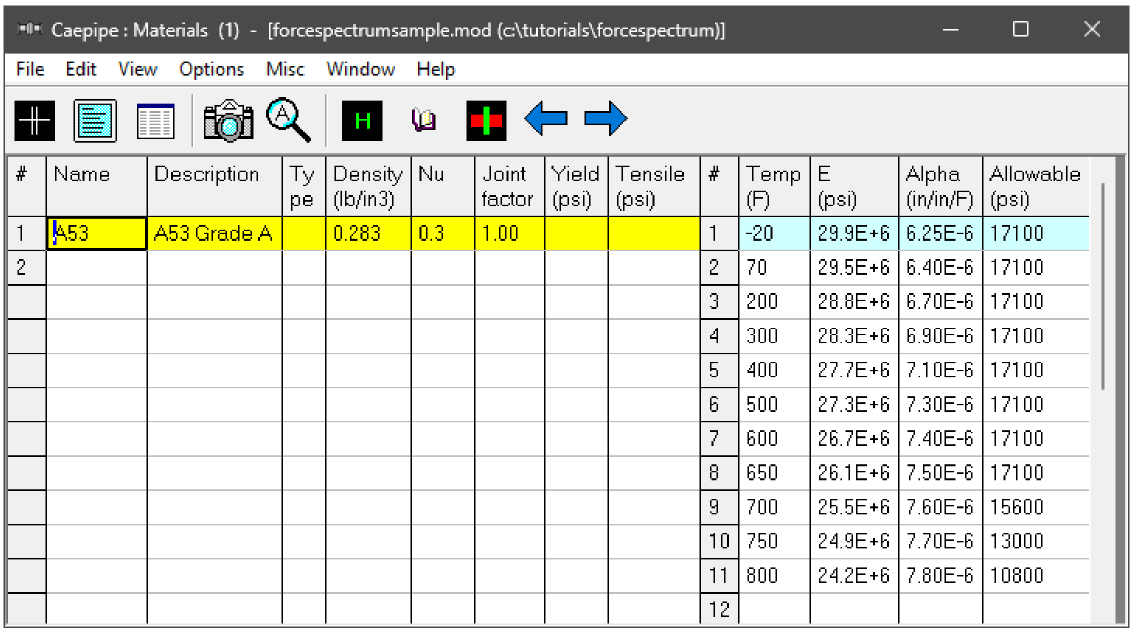
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title =								
2	10	From							Anchor
3	20			1'6"		A53	3	L2	
4	30	Valve		0'3"		A53	3	L2	
5	40	Valve	0'3"			A53	3	L2	
6	50		1'0"			A53	3	L1	
7	60	Reducer	0'4"			A53	4	L1	
8	70	Bend	1'0"			A53	4	L1	
9	80			10'0"		A53	4	L1	
10	75	Location							Force sp load
11									

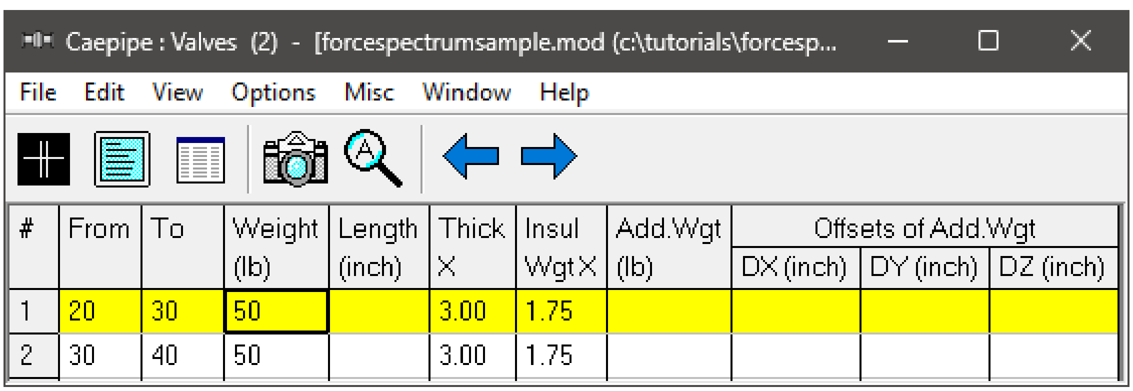




#	Name	Nom Dia	Sch	OD (inch)	Thk (inch)	Cor.Al (inch)	M.Tol (%)	Ins.Dens (lb/ft3)	Ins.Thk (inch)	Lin.Dens (lb/ft3)	Lin.Thk (inch)	So
1	3	3"	STD	3.5	0.216							
2	4	4"	STD	4.5	0.237							

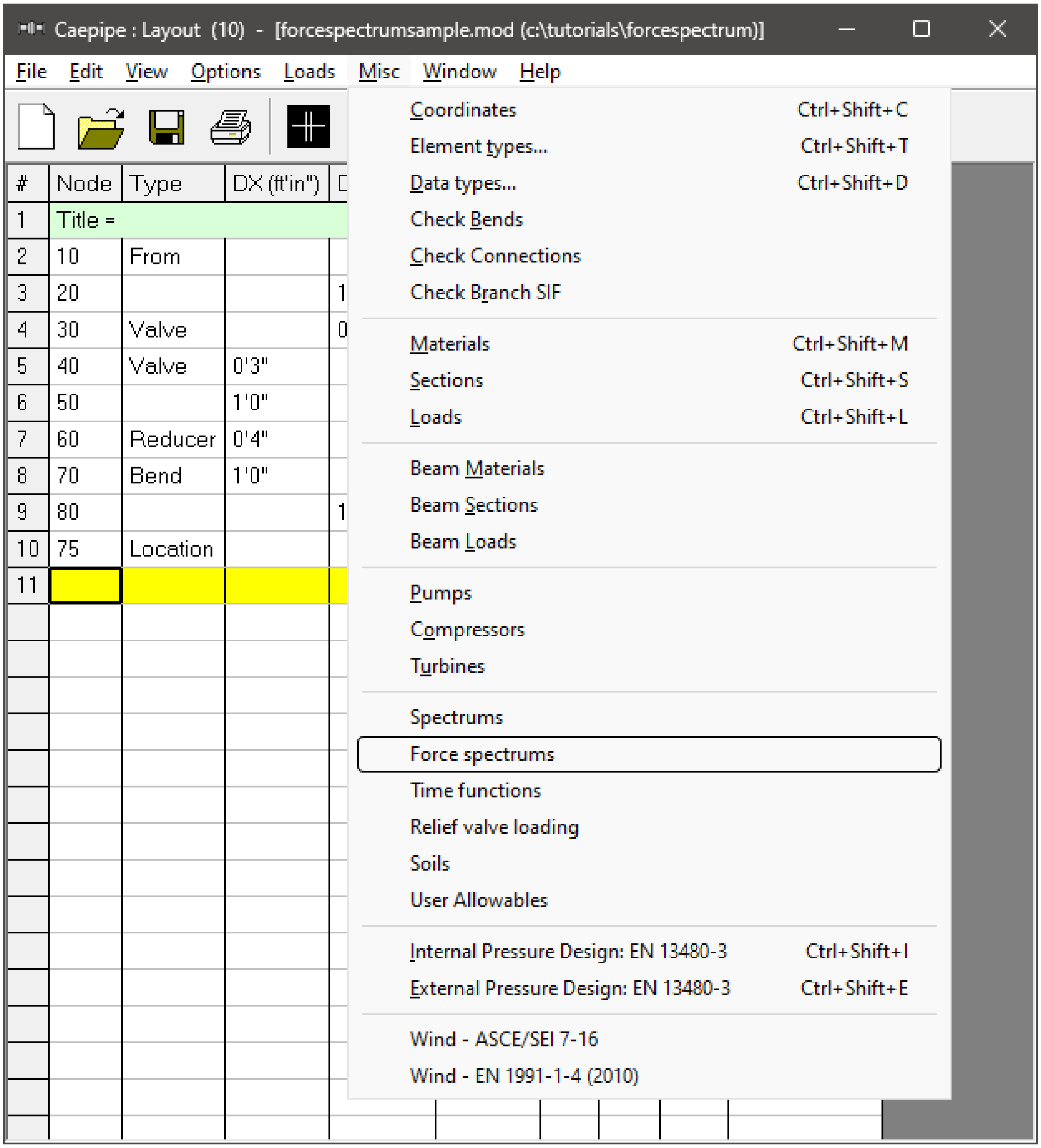






Step 2:

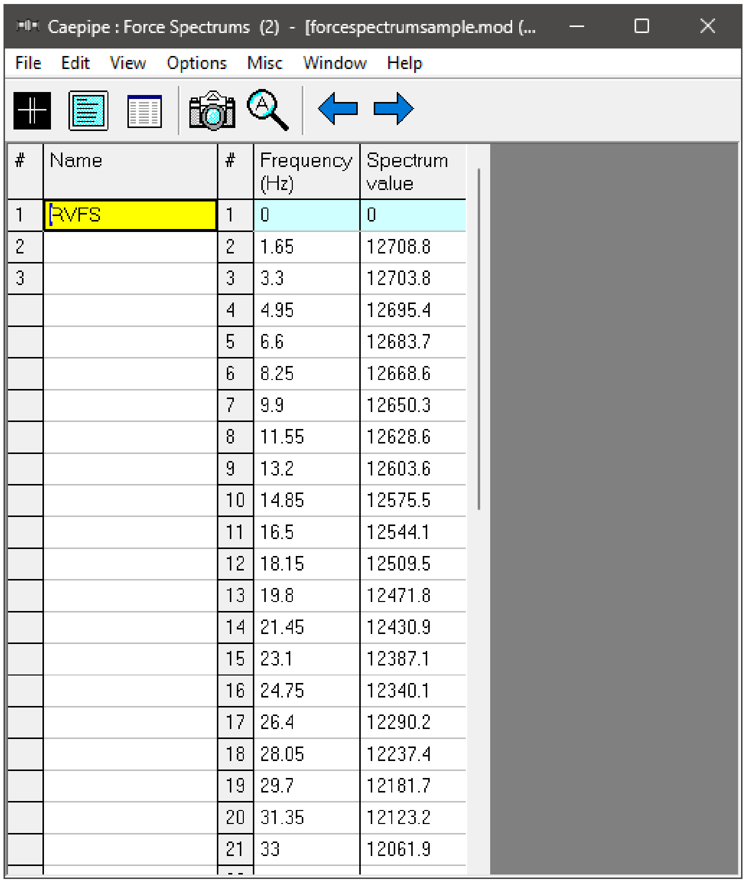
After creating your piping model (with node 75 being the center node of the discharge bend where the Force Spectrum will be applied), input Force spectrums through Layout or List menu: Misc > Force spectrums.



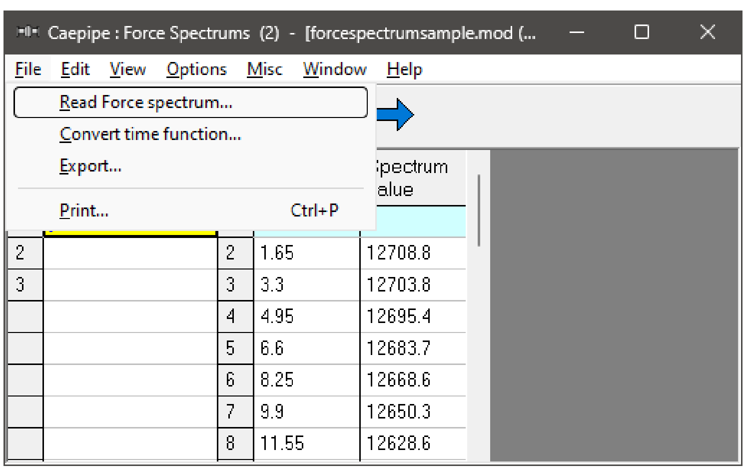
Step 3:

The Force spectrum list appears.

Enter a name for the force spectrum and spectrum values versus frequencies table.



In addition to inputting the force spectrum directly, it can also be read from a text file. This can be done through List menu: File > Read force spectrum.



The text file should be in the following format:

Name (up to 31 characters)

Frequency (Hz) Spectrum value

Frequency (Hz) Spectrum value

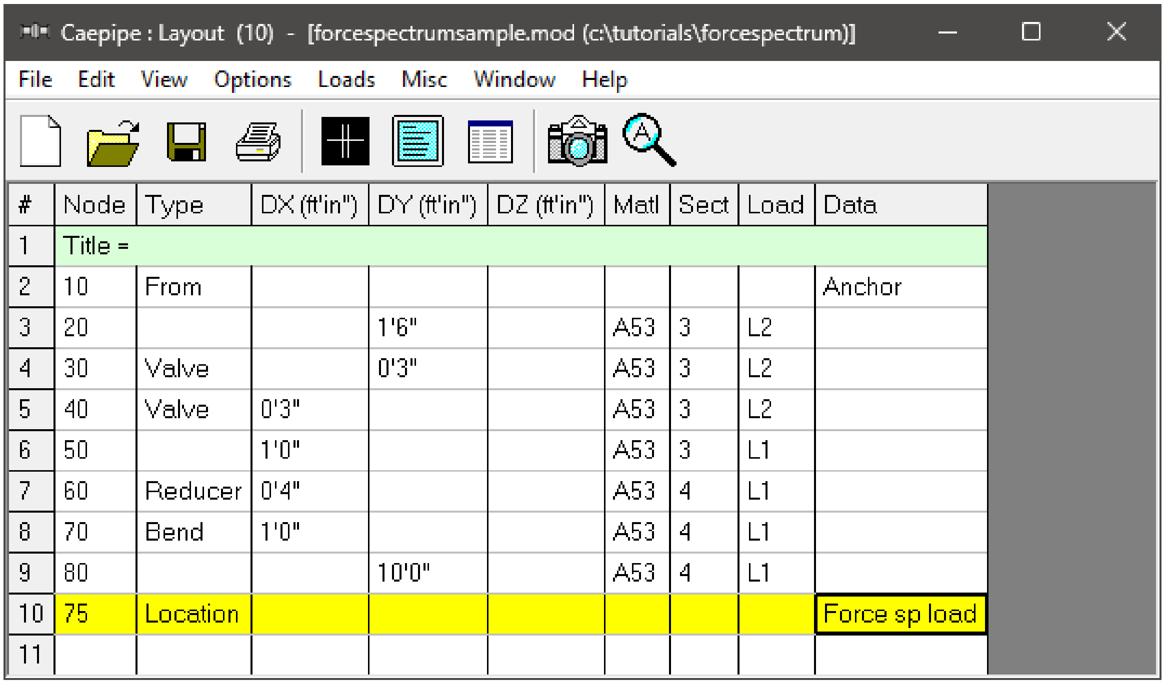
Frequency (Hz) Spectrum value

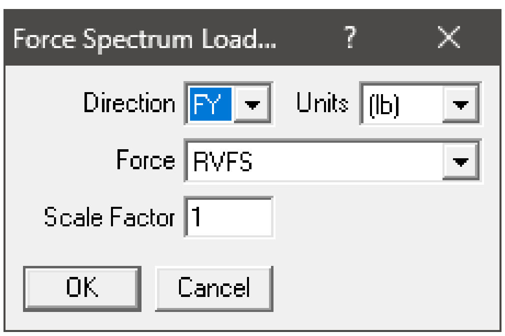
. .

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

Step 4:

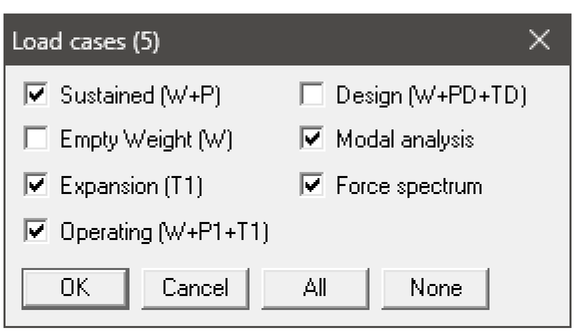
Apply the Force Spectrum Load thus generated at the bend center node 75 after the relief valve in vertical direction (FY) as shown below.





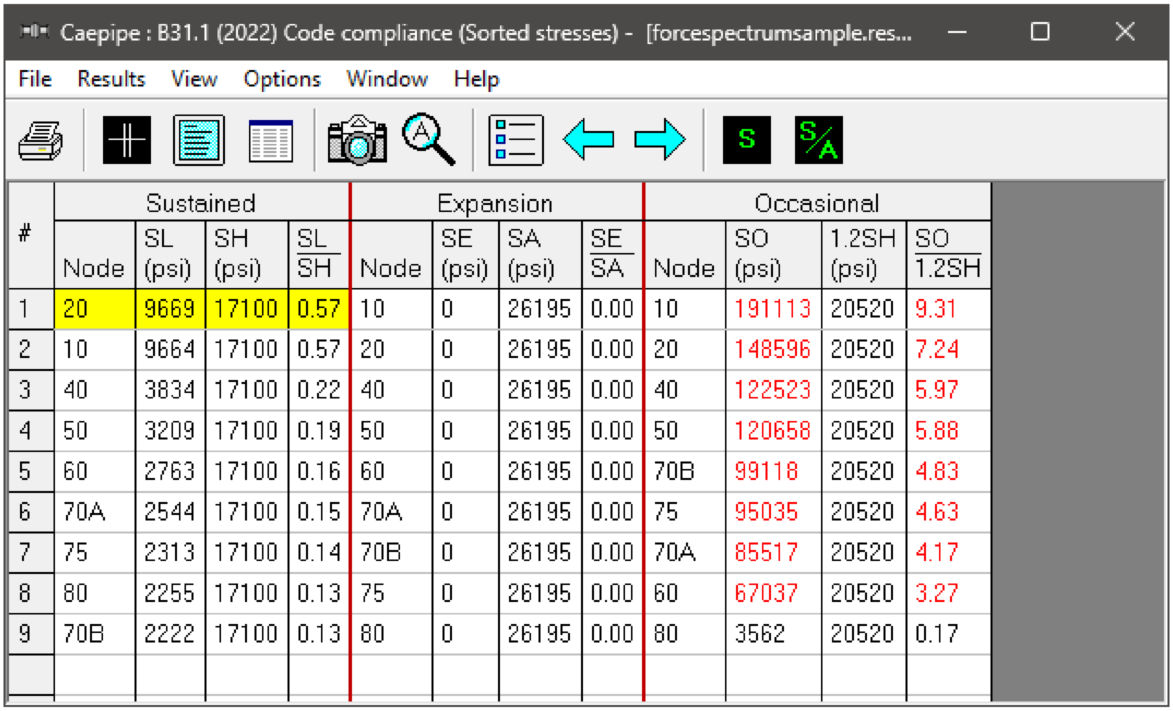
Step 5:

Check “Force Spectrum” for analysis through Layout window > Load cases. Click on OK.



Step 6:

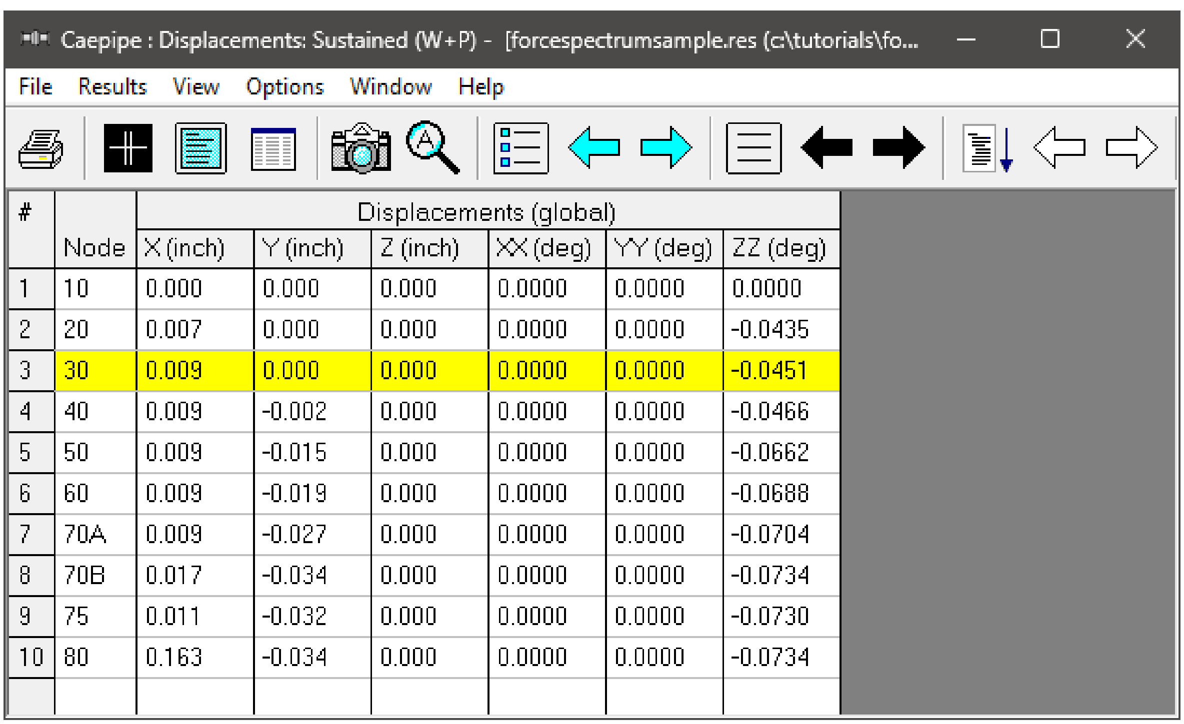
Save and Analyze the model. After analysis, CAEPIPE displays Occasional stresses which include the effects of the Force Spectrum load.



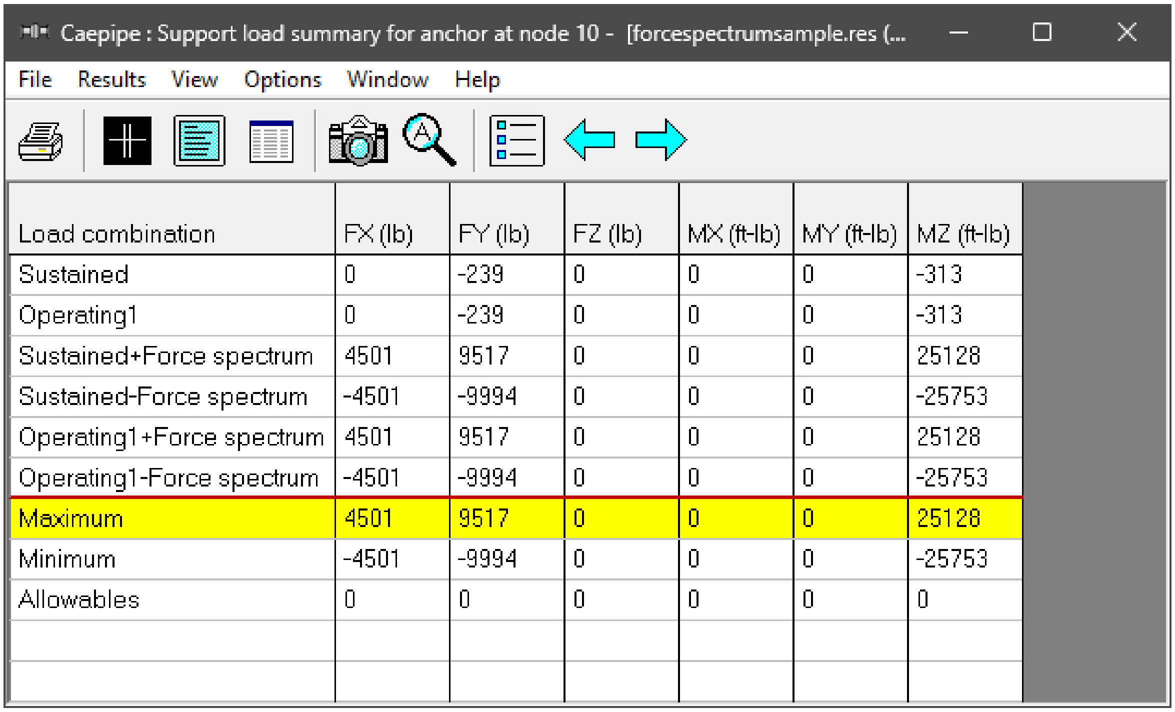
#	Sustained				Expansion				Occasional			
	Node	SL (psi)	SH (psi)	SL/SH	Node	SE (psi)	SA (psi)	SE/SA	Node	SO (psi)	1.2SH (psi)	SO/1.2SH
1	20	9669	17100	0.57	10	0	26195	0.00	10	191113	20520	9.31
2	10	9664	17100	0.57	20	0	26195	0.00	20	148596	20520	7.24
3	40	3834	17100	0.22	40	0	26195	0.00	40	122523	20520	5.97
4	50	3209	17100	0.19	50	0	26195	0.00	50	120658	20520	5.88
5	60	2763	17100	0.16	60	0	26195	0.00	70B	99118	20520	4.83
6	70A	2544	17100	0.15	70A	0	26195	0.00	75	95035	20520	4.63
7	75	2313	17100	0.14	70B	0	26195	0.00	70A	85517	20520	4.17
8	80	2255	17100	0.13	75	0	26195	0.00	60	67037	20520	3.27
9	70B	2222	17100	0.13	80	0	26195	0.00	80	3562	20520	0.17

Step 7:

Another load case called "Force Spectrum" will be available for which you can study displacements, support loads, support load summary (for sizing supports), etc.



#	Displacements (global)						
	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	20	0.007	0.000	0.000	0.0000	0.0000	-0.0435
3	30	0.009	0.000	0.000	0.0000	0.0000	-0.0451
4	40	0.009	-0.002	0.000	0.0000	0.0000	-0.0466
5	50	0.009	-0.015	0.000	0.0000	0.0000	-0.0662
6	60	0.009	-0.019	0.000	0.0000	0.0000	-0.0688
7	70A	0.009	-0.027	0.000	0.0000	0.0000	-0.0704
8	70B	0.017	-0.034	0.000	0.0000	0.0000	-0.0734
9	75	0.011	-0.032	0.000	0.0000	0.0000	-0.0730
10	80	0.163	-0.034	0.000	0.0000	0.0000	-0.0734



Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
Sustained	0	-239	0	0	0	-313
Operating1	0	-239	0	0	0	-313
Sustained+Force spectrum	4501	9517	0	0	0	25128
Sustained-Force spectrum	-4501	-9994	0	0	0	-25753
Operating1+Force spectrum	4501	9517	0	0	0	25128
Operating1-Force spectrum	-4501	-9994	0	0	0	-25753
Maximum	4501	9517	0	0	0	25128
Minimum	-4501	-9994	0	0	0	-25753
Allowables	0	0	0	0	0	0