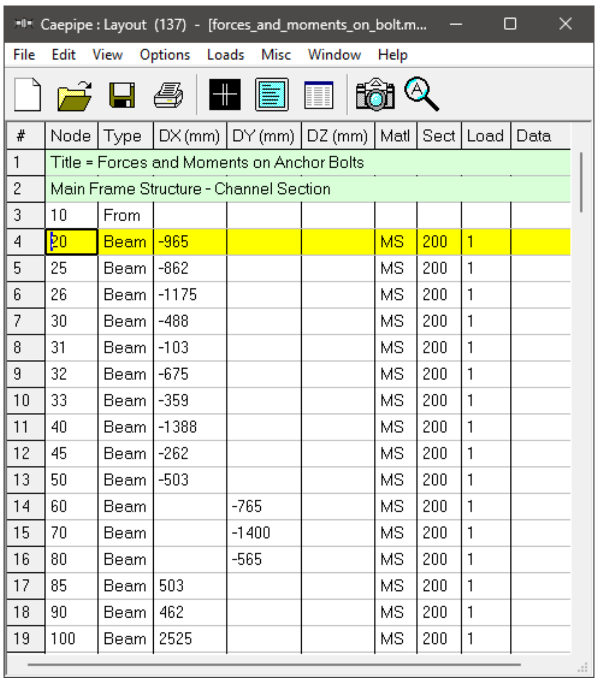
Tutorial on Structural Analysis using BEAM elements of CAEPIPE

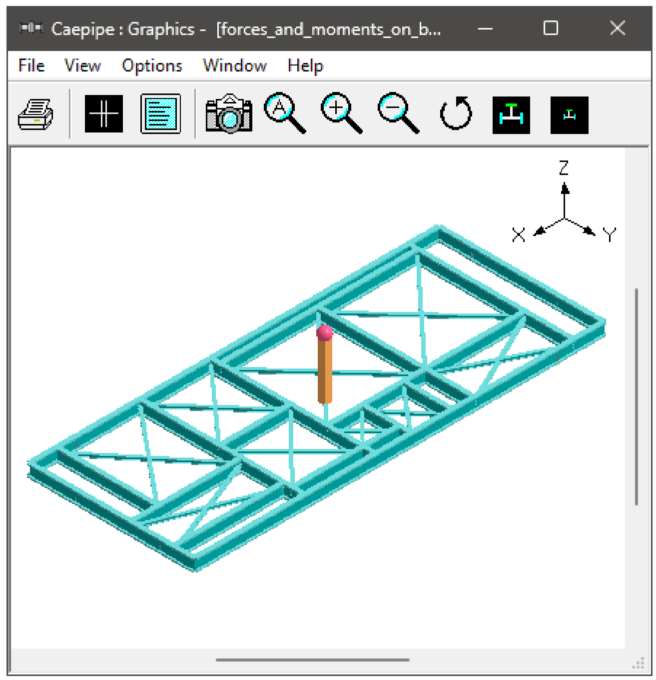
Tutorial

This tutorial explains how an equipment support frame is modeled and analyzed using BEAM elements of CAEPIPE. A similar procedure may be followed to include pipe support structures such as pipe rack along with the piping layout in pipe stress analysis, in case (a) the actual support structure stiffnesses are to be included in piping analysis, and (b) the forces and moments applied on the support structure members by the piping are to be determined to validate their designs.

Step 1:

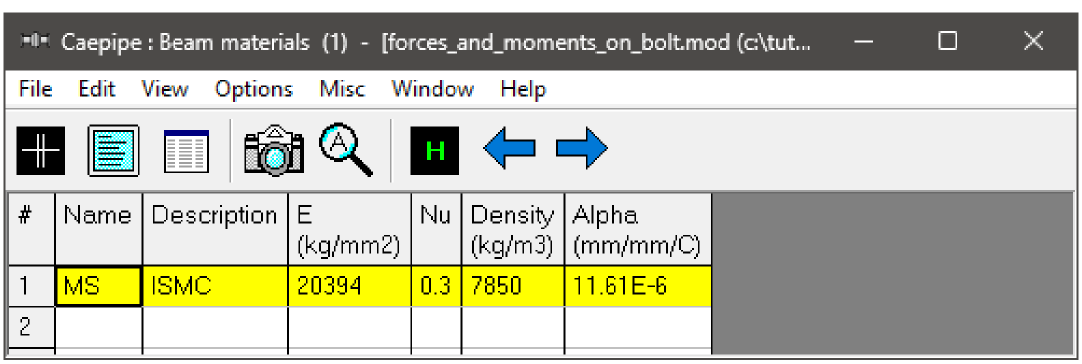
Attached is a sample CAEPIPE model developed for calculating Anchor Bolt Forces and Moments on equipment foundation of a Lubrication Oil Recirculation System due to Static Seismic G loading. The nodes 610 through 700 are Anchor Bolt Location nodes that are fixed inside the concrete pedestal.





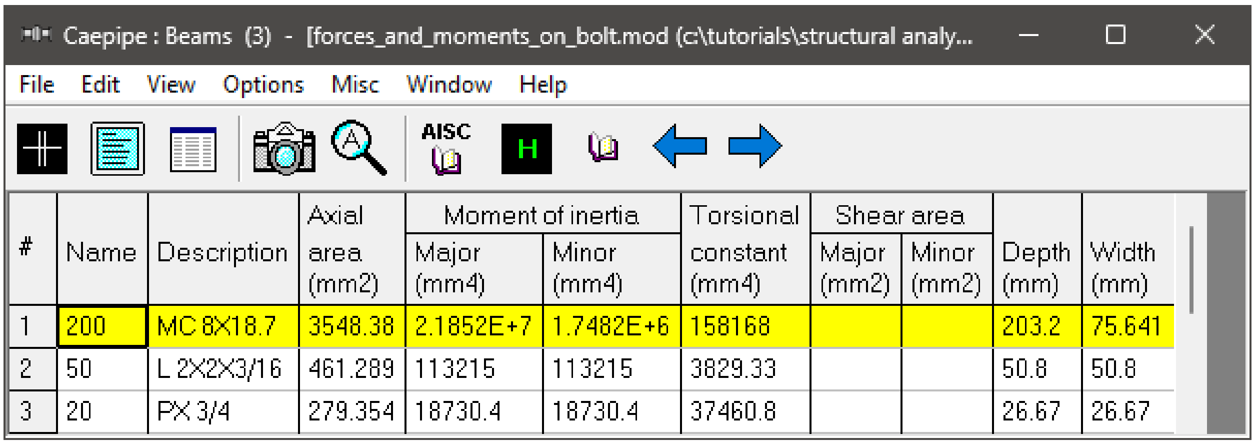
Step 2:

Beam Material required for this model is defined through Layout Window > Misc > Beam Material. See the figure shown below for details.



Step 3:

Beam Sections required for this model are defined through Layout Window > Misc > Beam Sections. See figure shown below for details. Properties shown for Channel (MC8x18.7), Angle (L2x2x3/16) and Circular Hollow Pipe (Px3/4) are selected from the AISC Library available in CAEPIPE. Since the value of Torsional inertia for Angle section L2x2x3/16 (same as Torsional Constant from AISC Standard) is not available within CAEPIPE at this time, it is manually entered by referring to AISC Standard. Also, the depth and width of this Angle section are manually entered.

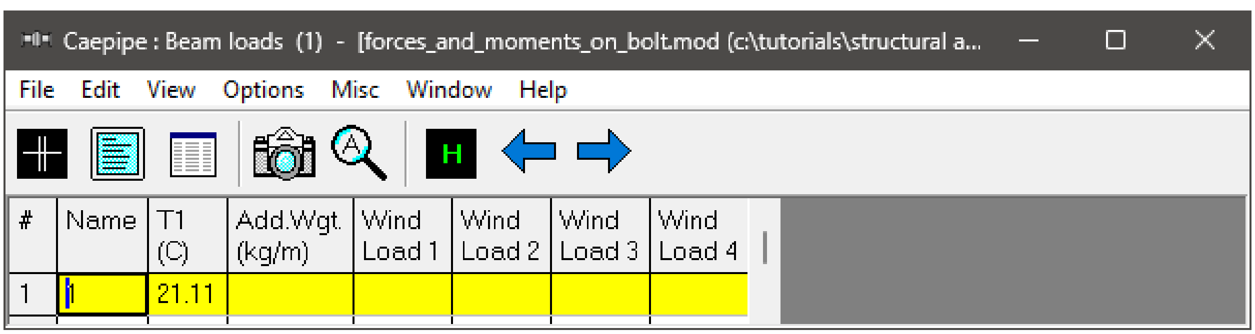


#	Name	Description	Axial area (mm2)	Moment of inertia Major (mm4)	Moment of inertia Minor (mm4)	Torsional constant (mm4)	Shear area Major (mm2)	Shear area Minor (mm2)	Depth (mm)	Width (mm)
1	200	MC 8X18.7	3548.38	2.1852E+7	1.7482E+6	158168			203.2	75.641
2	50	L 2X2X3/16	461.289	113215	113215	3829.33			50.8	50.8
3	20	PX 3/4	279.354	18730.4	18730.4	37460.8			26.67	26.67

The axial area, major and minor moments of inertia must be input. Input of torsional inertia is optional. If it is not input, it defaults to the sum of major and minor moments of inertia. Input of shear areas is optional. If shear areas are input, shear deflection effect is included in the analysis. Input of depth and width is optional. Presently, they are used only for rendered plots of the beam.

Step 4:

Similarly, Beam Load (Temperature) is defined through Layout Window > Misc > Beam Load. See figure shown below for details.



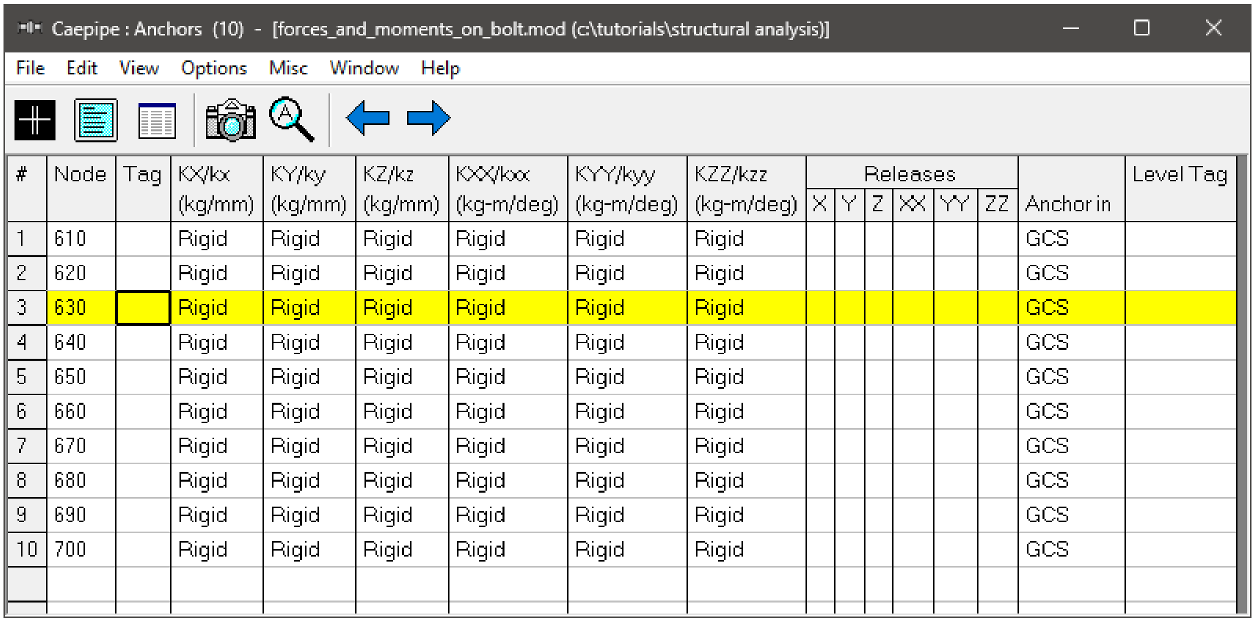
#	Name	T1 (C)	Add.Wgt. (kg/m)	Wind Load 1	Wind Load 2	Wind Load 3	Wind Load 4
1	1	21.11					

Step 5:

Equipment frame is then modeled using BEAM elements by defining the offsets DX, DY and DZ as shown in the attached model.

Step 6:

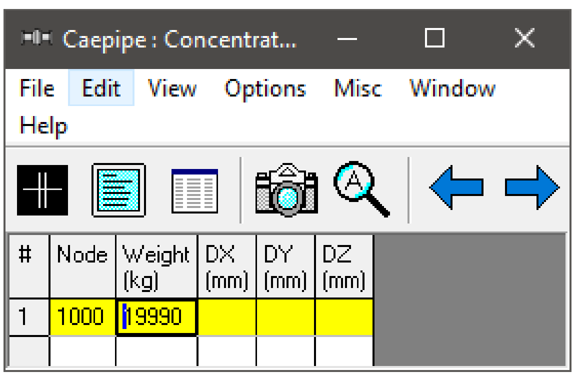
The Anchor Bolt Location nodes (610 through 700) that are fixed inside the concrete pedestal are then modeled as “Anchor” with their stiffnesses in three (3) translational directions and three (3) rotational directions as “Rigid”.



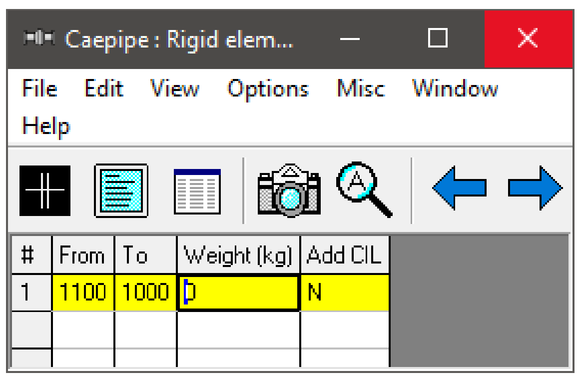
#	Node	Tag	KX/kx (kg/mm)	KY/ky (kg/mm)	KZ/kz (kg/mm)	KXX/kxx (kg-m/deg)	KYY/kyy (kg-m/deg)	KZZ/kzz (kg-m/deg)	Releases X	Y	Z	XX	YY	ZZ	Anchor in	Level Tag
1	610		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
2	620		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
3	630		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
4	640		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
5	650		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
6	660		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
7	670		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
8	680		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
9	690		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	
10	700		Rigid	Rigid	Rigid	Rigid	Rigid	Rigid							GCS	

Step 7:

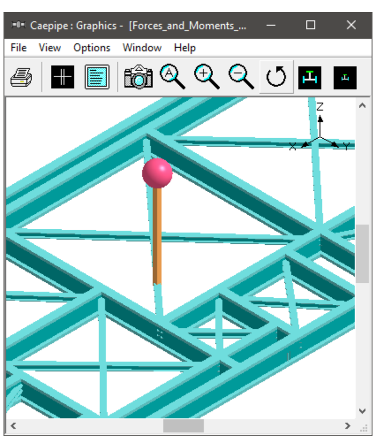
The weight of the equipment is then input using data type “CMASS” available in CAEPIPE. The weight thus defined is then applied on the equipment support frame by connecting a “mass-less” rigid element from the center of the support frame to the equipment Center of Gravity (COG).



#	Node	Weight (kg)	DX (mm)	DY (mm)	DZ (mm)
1	1000	19990			

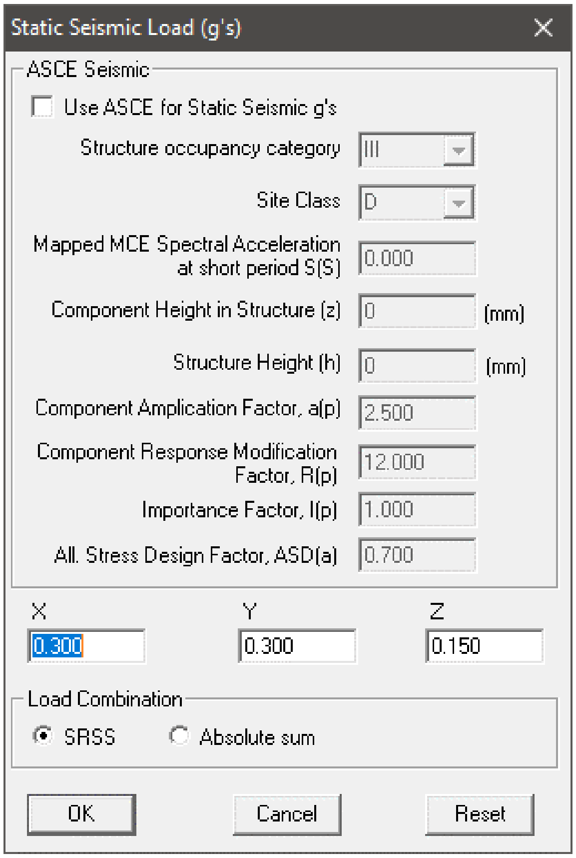


#	From	To	Weight (kg)	Add CIL
1	1100	1000	0	N



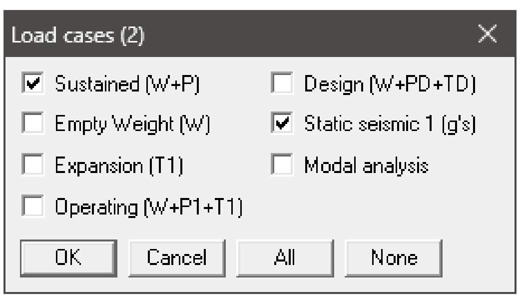
Step 9:

Defined Static Seismic Load (g’s) through Layout Window > Loads > Static Seismic.



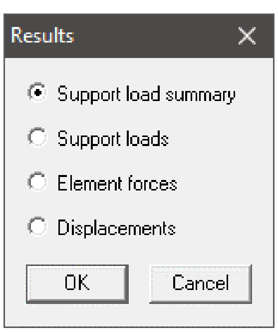
Step 10:

Select the load cases required for analysis through Layout Window > Loads > Load cases. Once done, save the model and perform the analysis through Layout window > File > Analyze.



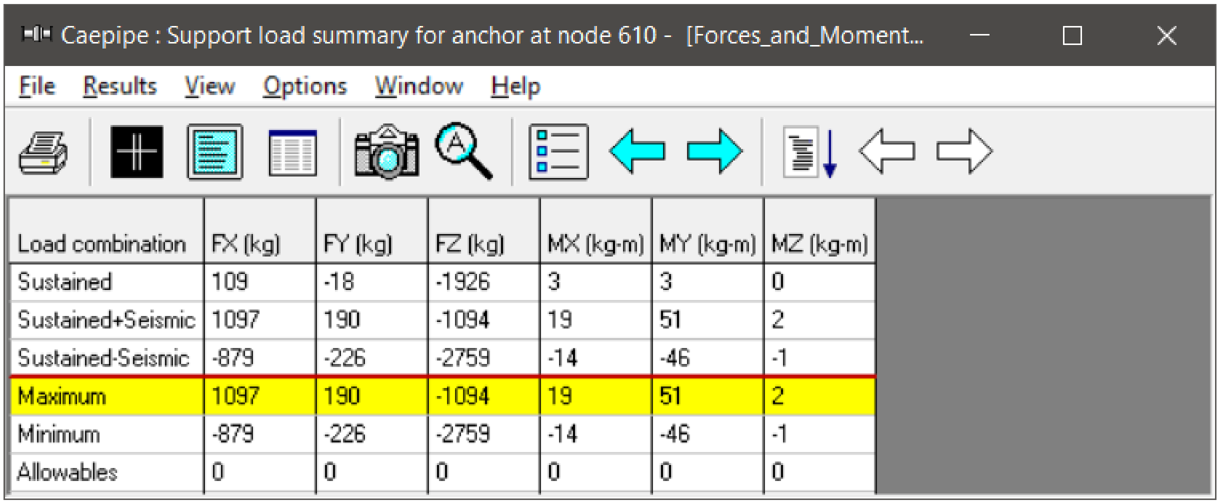
Step 11:

Upon successful analysis, CAEPIPE will show the following options under Results dialog.



Step 12:

Selecting the option “Support load summary” will show the Forces and Moments at each Anchor Bolt for different load cases and combinations selected for analysis.



Load combination	FX (kg)	FY (kg)	FZ (kg)	MX (kg-m)	MY (kg-m)	MZ (kg-m)
Sustained	109	-18	-1926	3	3	0
Sustained+Seismic	1097	190	-1094	19	51	2
Sustained-Seismic	-879	-226	-2759	-14	-46	-1
Maximum	1097	190	-1094	19	51	2
Minimum	-879	-226	-2759	-14	-46	-1
Allowables	0	0	0	0	0	0