Heavier equipment with faster machine speeds coupled with higher costs have made rule-of-thumb approaches and "rigid structure" assumptions either unsafe or too conservative for the design of many structures with vibrating machinery. SAP2000 can easily compute natural frequencies, deformations and forces in the structure with consideration of structure mass and flexibility. Using less rigorous design approaches, these factors are either ignored, conservatively assumed, or handled in a simplified approximate fashion. Sizing and design of a skid or foundation supporting vibrating equipment is beyond the scope of this tutorial. However, here are a few considerations:

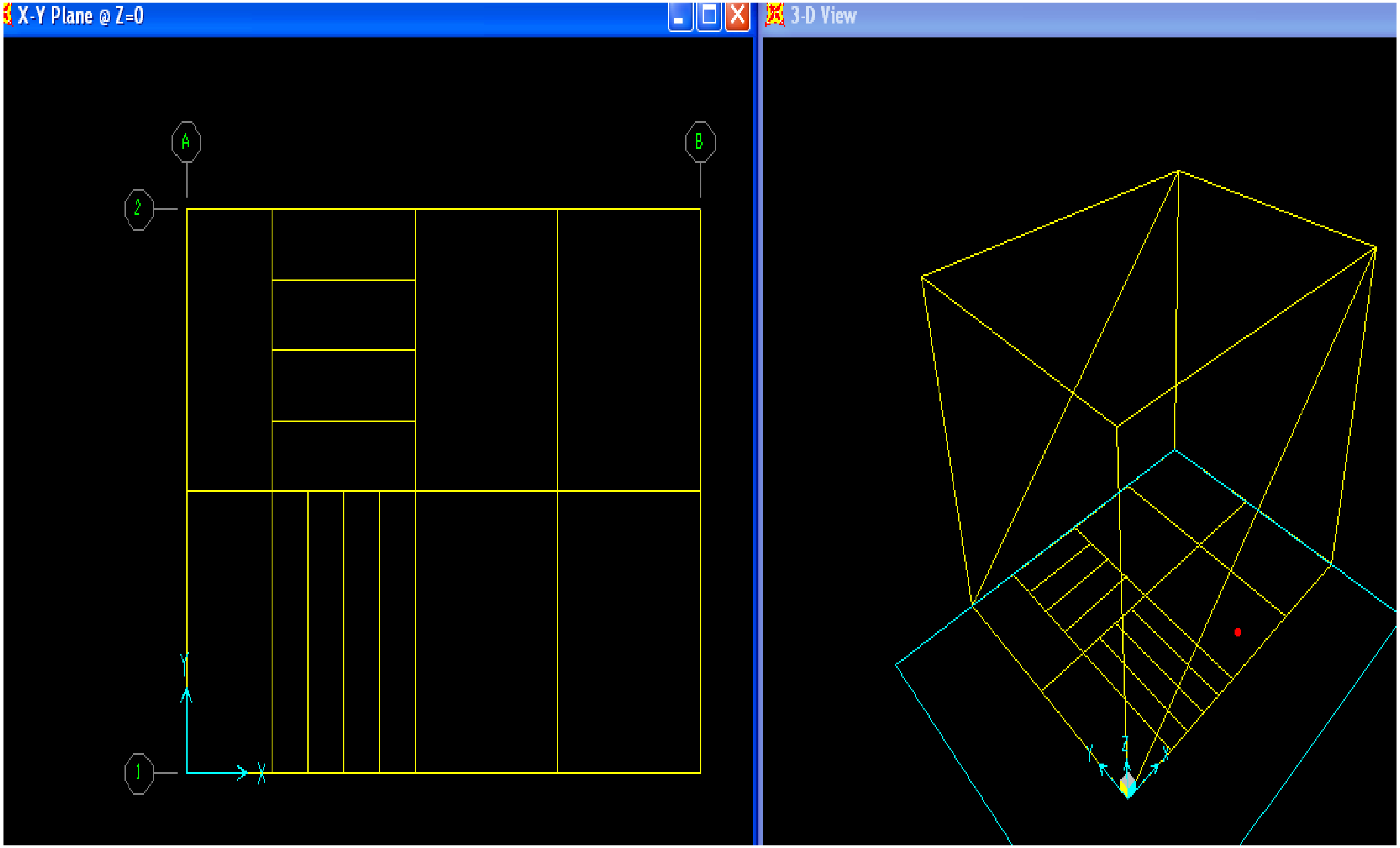
- Run multiple analyses which vary soil spring constants and damping ratio to account for uncertainties of soil data.

- Acting frequency of the vibrating machinery should not be close to the structure’s resonant frequency

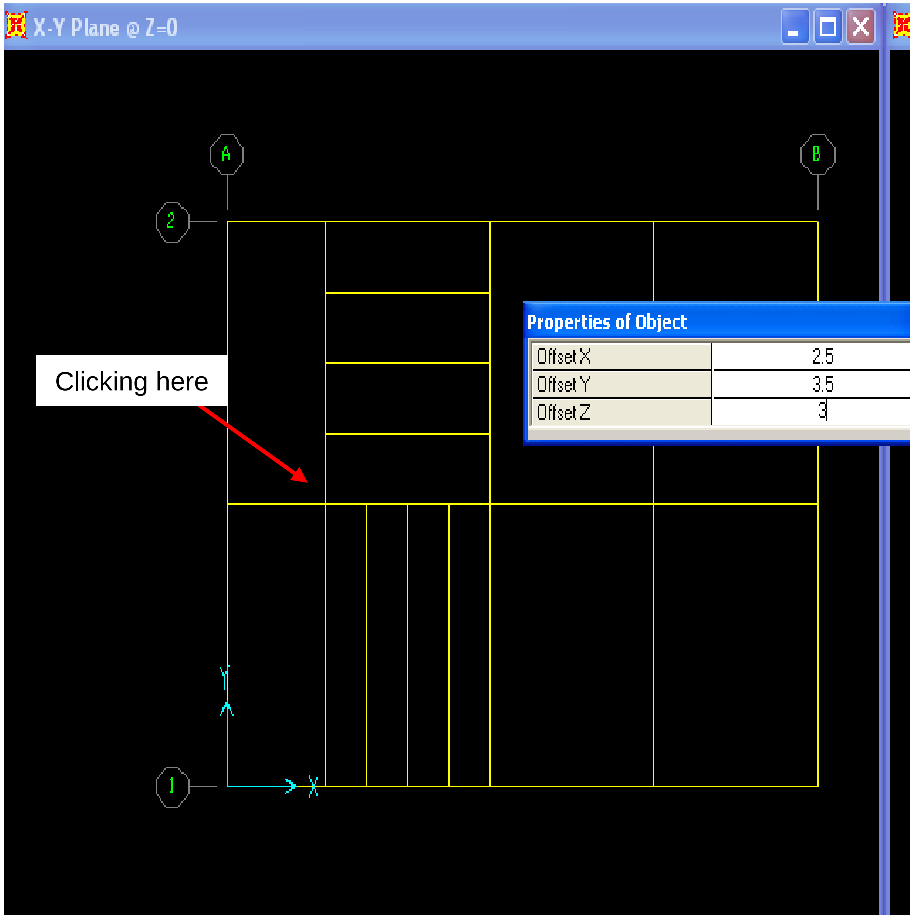
- Dynamic effects to and from adjoining structures and equipment. Modular skids may be designed in isolation for vibrating equipment. Yet these skids can be added to other parts of a larger structure which may change the initial design assumptions.

SAP2000

Use SAP2000's "Draw special joint" to add joints at the Center of Mass (CoM) for each piece of vibrating equipment and also at the centroid of unbalanced load locations. CoM can be referred to as Center of Gravity in this context. In many cases, the vendor will give separate CoM locations for major components (pump, gearbox, turbine, turbine rotor, generator, etc.), in which case you would add joints for each component of the equipment where you will assign force/weight which will be converted to mass. In addition, you need to draw a joint at each unbalanced load location. In this example, we'll take a simplified modeling approach by lumping each equipment CoM at just 1 joint location per machine.

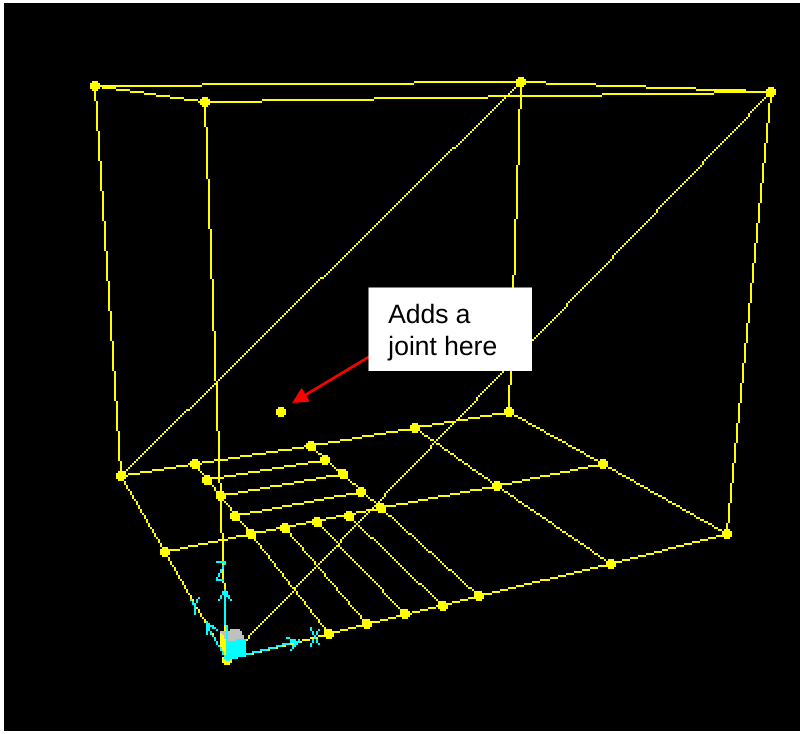


‘Draw special joint’ tool adds a joint offset in current units (units displayed bottom right portion of the SAP2000 screen) from wherever you mouse click in the model. In this example, we use ft. units modeling the CoM and unbalanced load location for two pieces of equipment. Use ‘Draw special joint’ button on the toolbar or find it in the Draw menu.



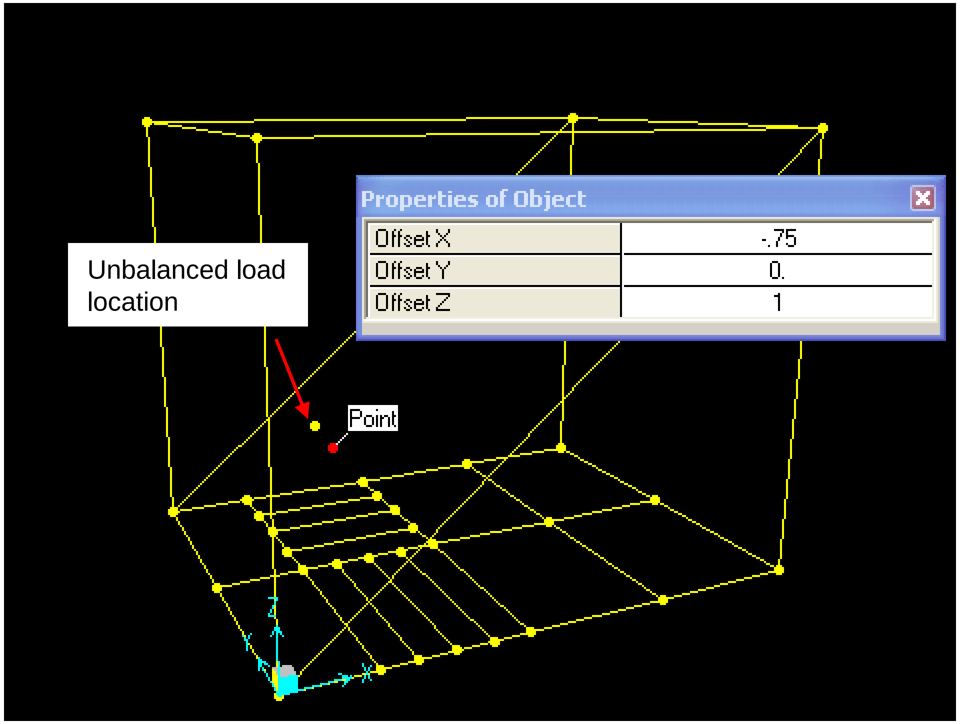
Offset X	2.5
Offset Y	3.5
Offset Z	3

Clicking here



Adds a joint here

Assuming the 1st special joint was the machine CoM, we'll draw another special joint to specify the centroid of unbalanced load by adjusting offsets and then clicking the CoM joint to specify the unbalanced load location, In this fictitious example, the unbalanced load is positioned 1 ft. above and -.75 ft Y from the CoM. Using the same Draw tool, we add 2 more joints on the other side based on vendor specified CoM and unbalanced load location. Dimensions for this example are not important, because each project will be different. At this point we have 4 joints offset from the structure which need to be connected to the structure.

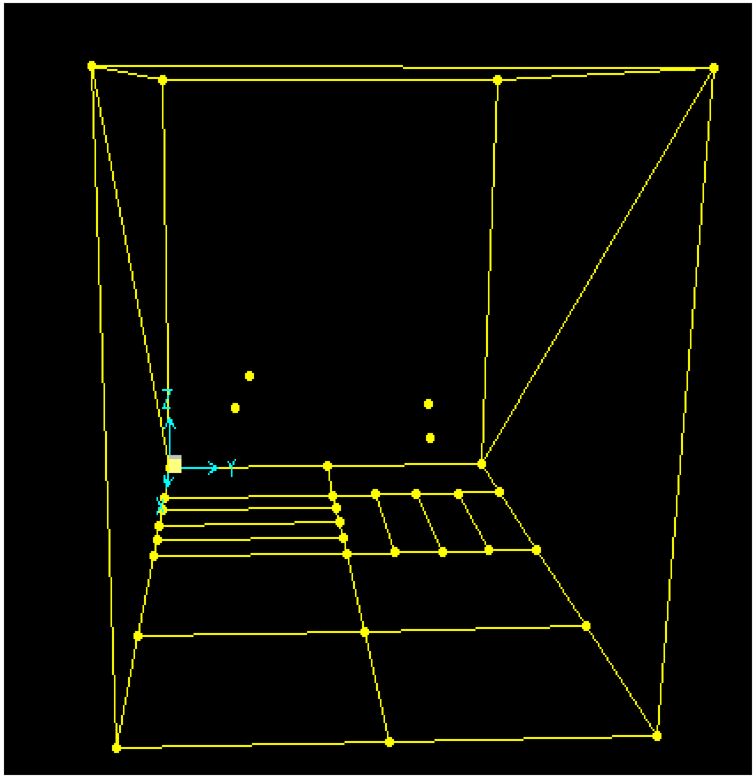


Unbalanced load location

Properties of Object

Offset X	-.75
Offset Y	0.
Offset Z	1

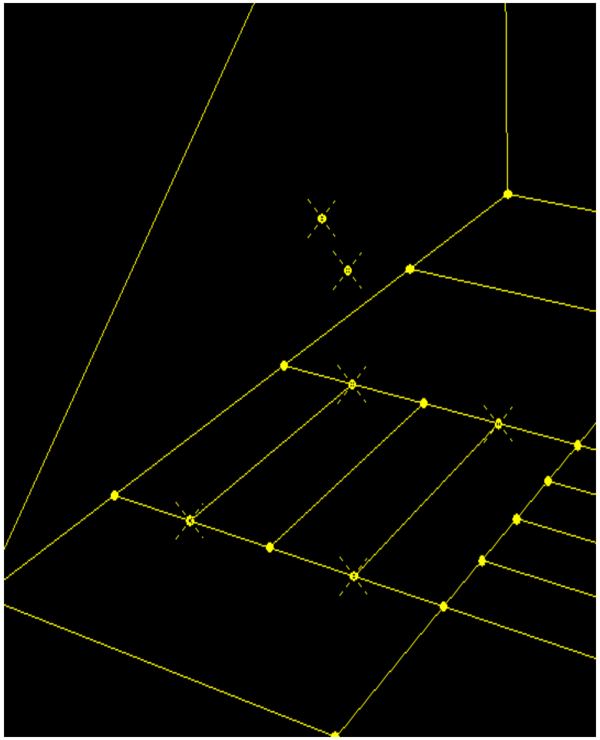
4 joints offset from structure

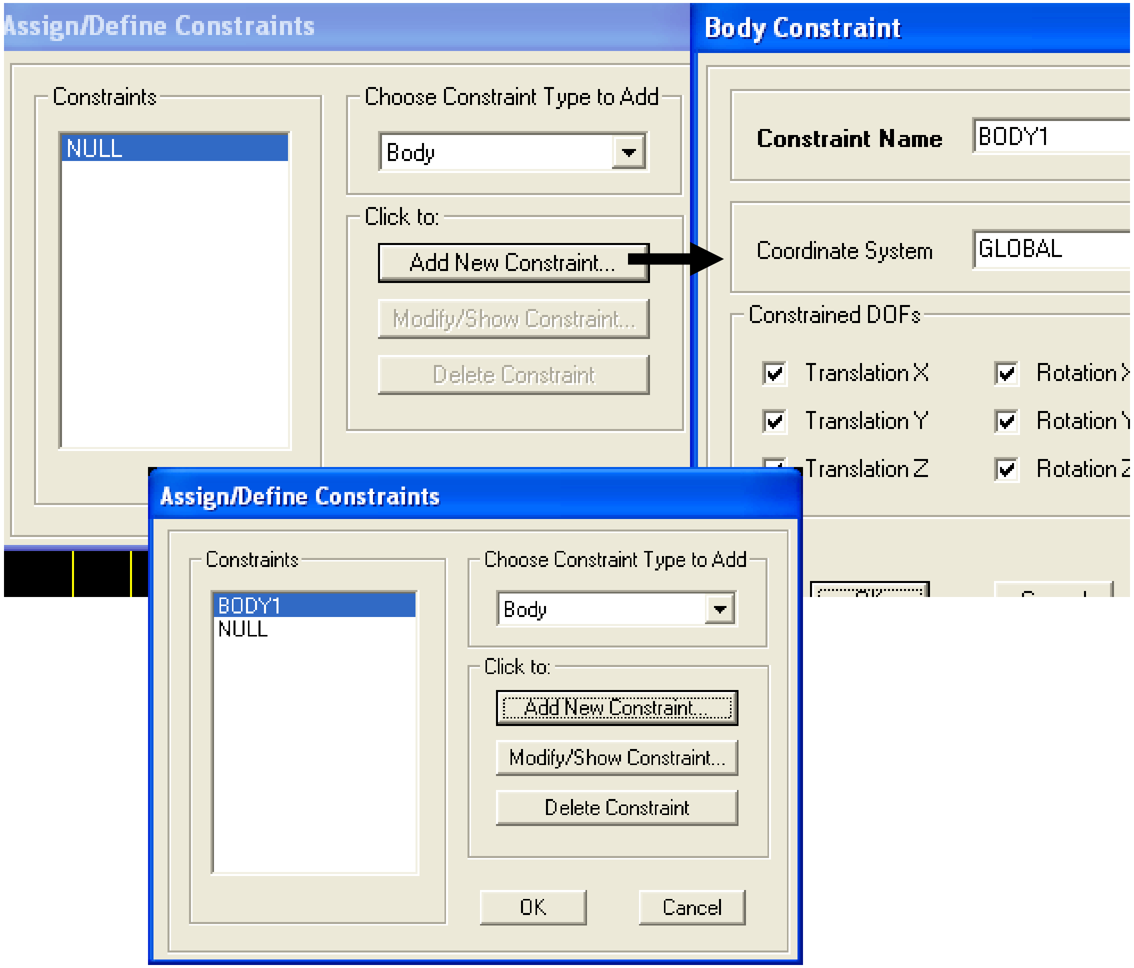


Connect the CoM and unbalanced load joint of each piece of equipment to the structure by selecting the CoM and unbalanced load joints of one machine along with connection points where the equipment is bolted or welded to the structure, then use Assign menu>Joint>Constraints using Body type of constraint as shown below. Body joint constraints rigidly link the selected joints together while considering the moment differentials from their offset distances. Joint Constraints are preferable to rigid “dummy” frame members because overly rigid elements relative to the stiffness of what they are connecting to can introduce analytical problems, particularly in dynamic analysis. Constraints avoid these potential numerical instabilities. If there were multiple CoM’s for a particular machine representing various components, you would select all CoM’s for the machine when assigning a constraint to the structure.

Repeat this procedure to assign a new, separate Body constraint, BODY2, to the equipment on the other side.

Here we have selected a CoM joint, the unbalanced load joint, and 4 joints below where the equipment is bolted to the structure.





Constraint Name: BODY1

Coordinate System: GLOBAL

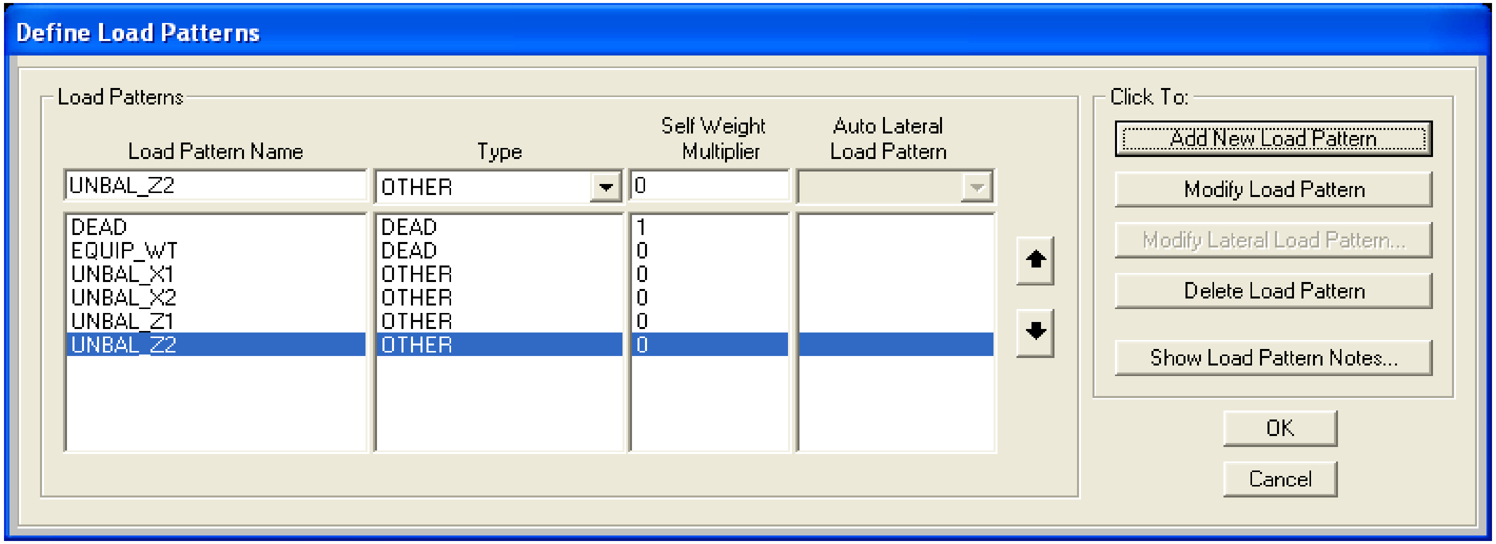
Constrained DOFs

☑ Translation X ☑ Rotation

☑ Translation Y ☑ Rotation

☑ Translation Z ☑ Rotation

Under Define>Load Patterns, add load patterns for equipment CoM and max unbalanced loads. You can assign the CoM (weight) of the equipment all to one load pattern, or alternatively create separate load patterns for each equipment weight. Since the unbalanced loads will vary in both magnitude and machine speed, you would define separate load patterns for each unbalanced load assignment. We will next assign max unbalanced load in the X and Z directions for each piece equipment, .7 Kip unbalanced joint load in each direction for one machine and 1.1 Kip unbalanced joint load for the other machine.

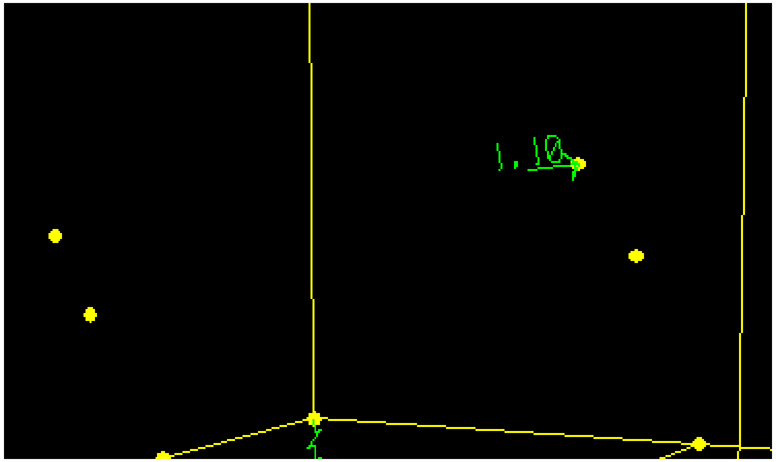


Define Load Patterns

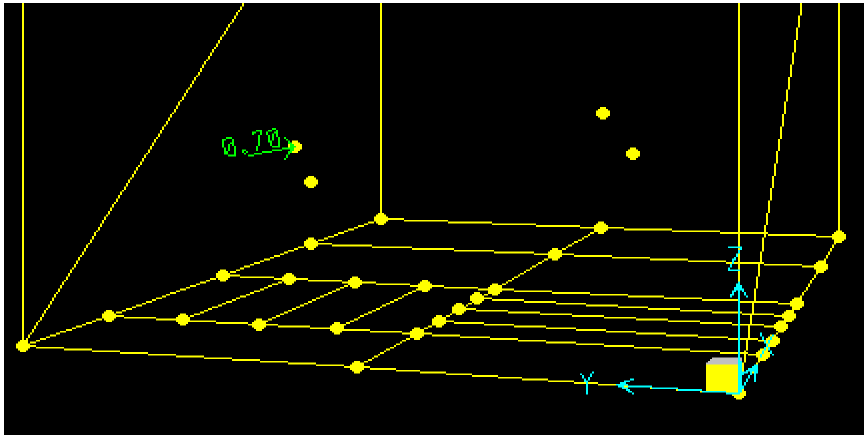
Load Patterns

Load Pattern Name	Type	Self Weight Multiplier	Auto Lateral Load Pattern
UNBAL_Z2	OTHER	0	
DEAD	DEAD	1	
EQUIP_WT	DEAD	0	
UNBAL_X1	OTHER	0	
UNBAL_X2	OTHER	0	
UNBAL_Z1	OTHER	0	
UNBAL_Z2	OTHER	0	

Select joints, one at a time, and assign X direction unbalanced load to their respective load patterns

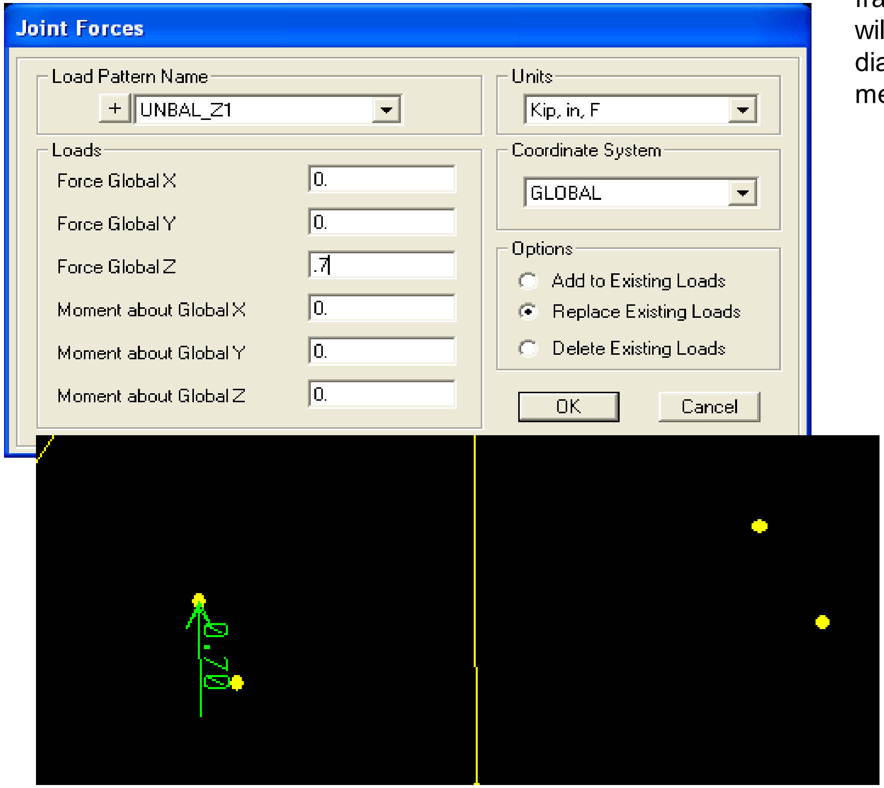


UNBAL_X2



UNBAL_X1

Next, assign unbalanced loads in Z direction



Load Pattern Name: UNBAL_Z1

Units: Kip, in, F

Loads

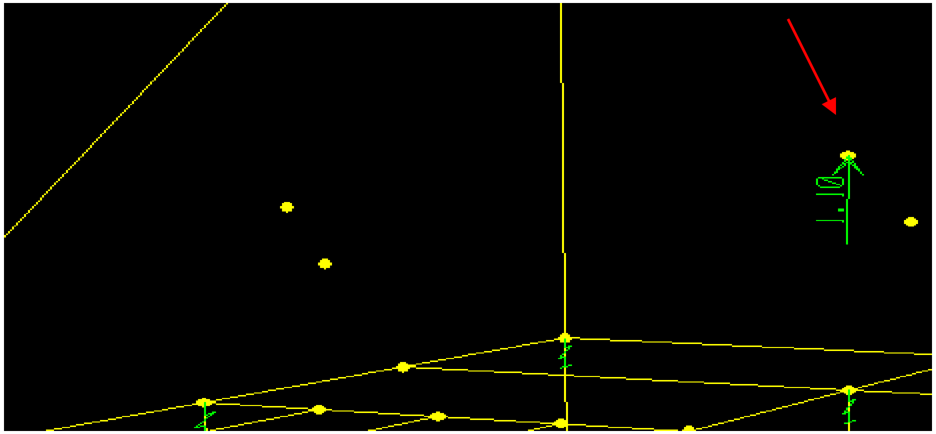
Force Global X: 0.
Force Global Y: 0.
Force Global Z: .7
Moment about Global X: 0.
Moment about Global Y: 0.
Moment about Global Z: 0.

Coordinate System: GLOBAL

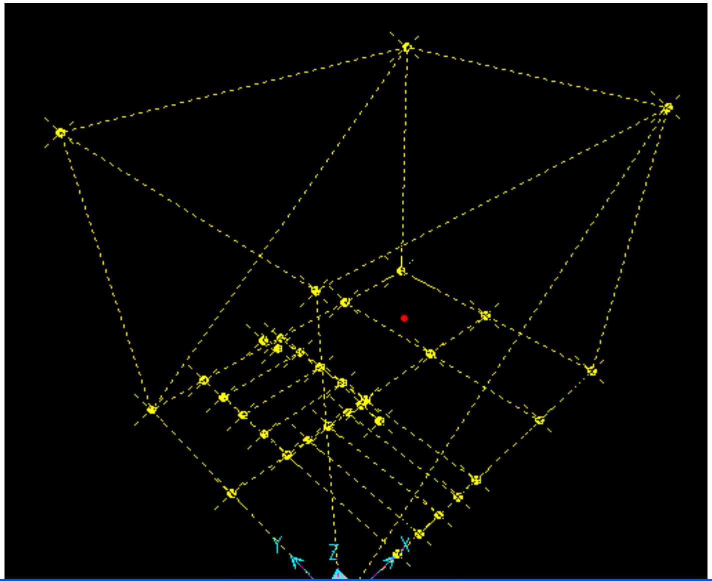
Options

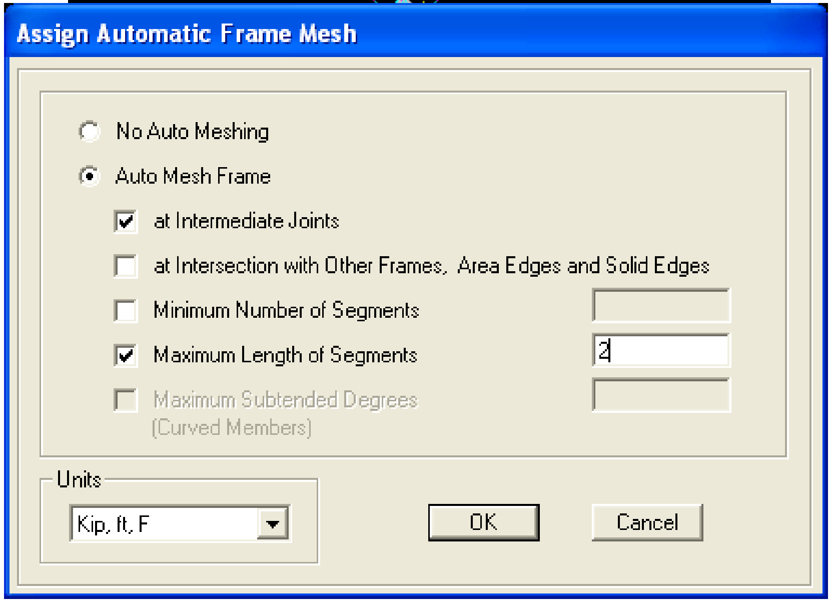
- ○ Add to Existing Loads
- ● Replace Existing Loads
- ○ Delete Existing Loads

UNBAL_Z2



Select all, then Assign menu>Frame>Automatic Frame mesh and specify that frames be automatically divided for internal analysis with a max length of 2 ft. This will only affect internal analysis. Each frame element will display 1 moment diagram and be designed using it’s physical length. For dynamic analysis, meshing can affect results. It’s up to the engineer to determine mesh sensitivity.



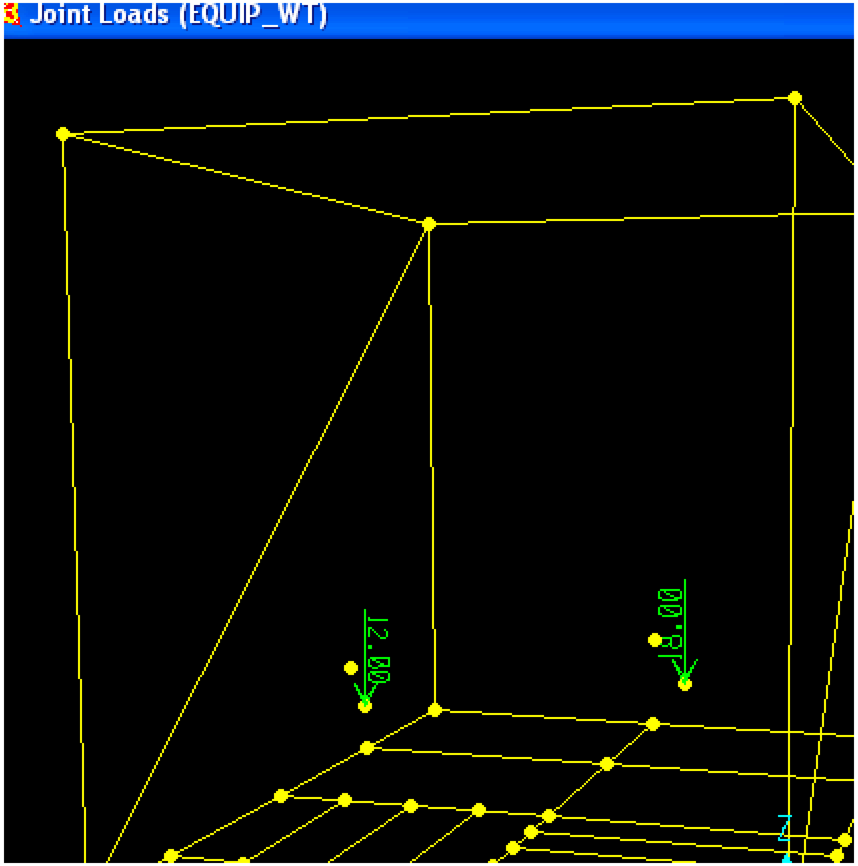


- ○ No Auto Meshing
- ● Auto Mesh Frame
  - ☑ at Intermediate Joints
  - ☐ at Intersection with Other Frames, Area Edges and Solid Edges
  - ☐ Minimum Number of Segments
  - ☑ Maximum Length of Segments: 2
  - ☐ Maximum Subtended Degrees (Curved Members)

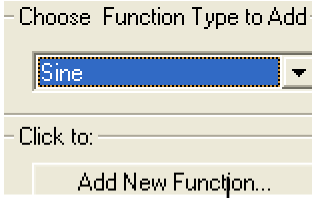
Units: Kip, ft, F

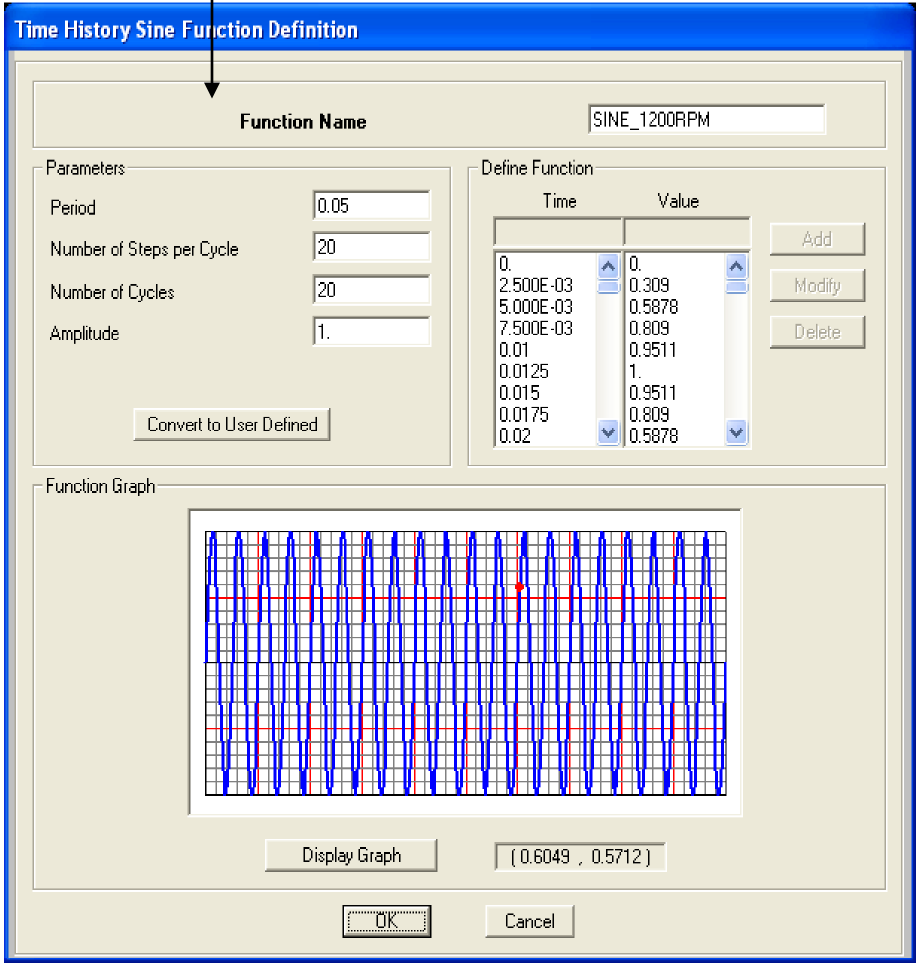
We assign CoM (weight) loads for each piece of equipment. In this example, a 12 Kip weight load for one piece of equipment and an 18 Kip weight load for the other equipment, assigning them both under the EQUIP_WT load pattern.

This is a simplified model which considers only the vibrating equipment. But if there were other stationary equipment or piping or anything else with significant mass that will be on this structure along with the vibrating equipment, those items would need to be modeled using loads, or modeled explicitly with elements.



Using Define menu>Functions>Time history, define sine and cosine functions based on machines speeds. In this example we'll analyze one machine running at 1200rpm while the other is running at 1800rpm. Use the Sine and Cosine function types to help you quickly define functions. Period of 1200 rpm is .05. Period of 1800 rpm is .0333. Because we are going to run a "periodic" time history dynamic analysis, we do not have to define many cycles in order to avoid transient startup "spike" in results. 20 cycles should be plenty.


Choose Function Type to Add: Sine


Time History Sine Function Definition

Function Name SINE_1200RPM

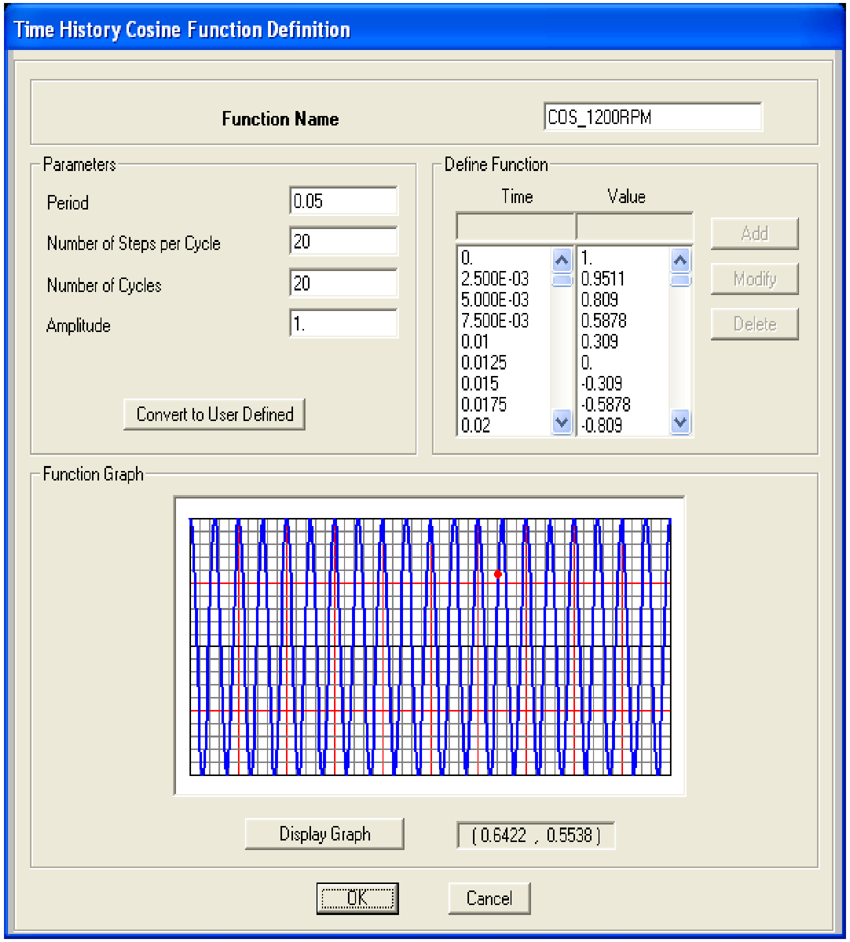
Parameters

Period	0.05
Number of Steps per Cycle	20
Number of Cycles	20
Amplitude	1.

Define Function

Time	Value
0.	0.
2.500E-03	0.309
5.000E-03	0.5878
7.500E-03	0.809
0.01	0.9511
0.0125	1.
0.015	0.9511
0.0175	0.809
0.02	0.5878

Cosine function


Time History Cosine Function Definition

Function Name COS_1200RPM

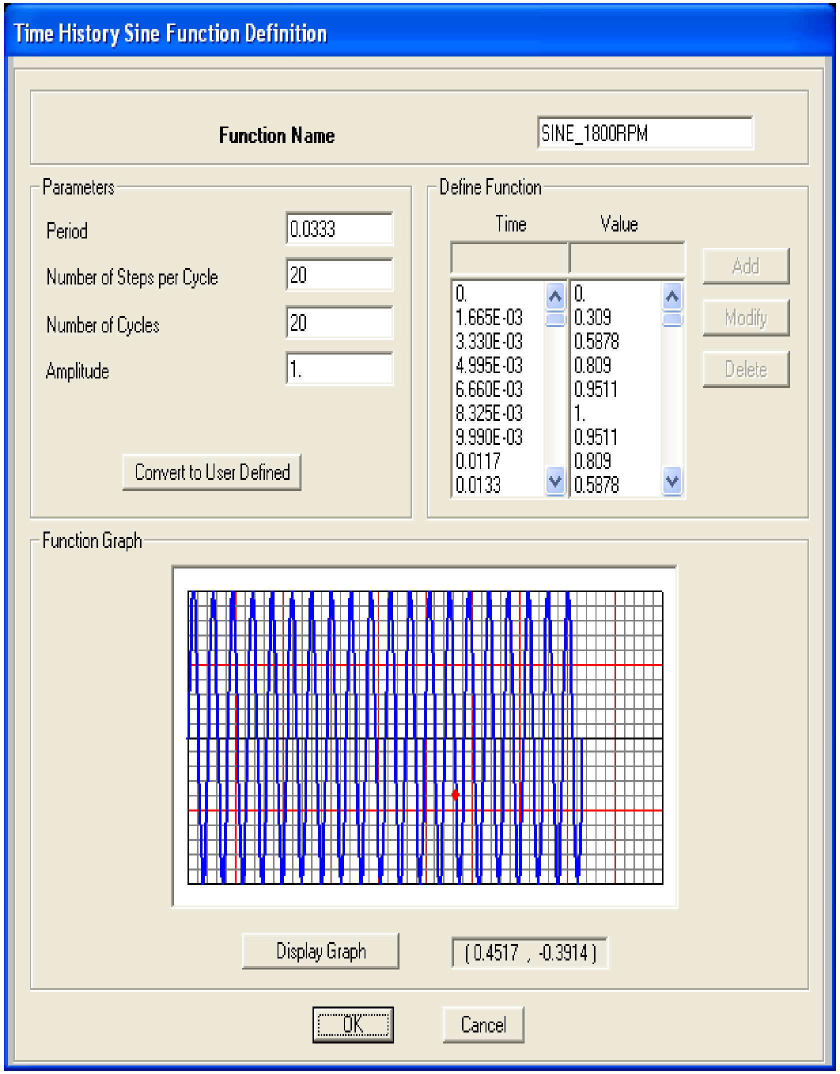
Parameters

Period	0.05
Number of Steps per Cycle	20
Number of Cycles	20
Amplitude	1.

Define Function

Time	Value
0.	1.
2.500E-03	0.9511
5.000E-03	0.809
7.500E-03	0.5878
0.01	0.309
0.0125	0.
0.015	-0.309
0.0175	-0.5878
0.02	-0.809

Sine and Cosine functions for 1800 rpm, period = .0333, 20 cycles each as shown



Time History Sine Function Definition

Function Name SINE_1800RPM

Parameters

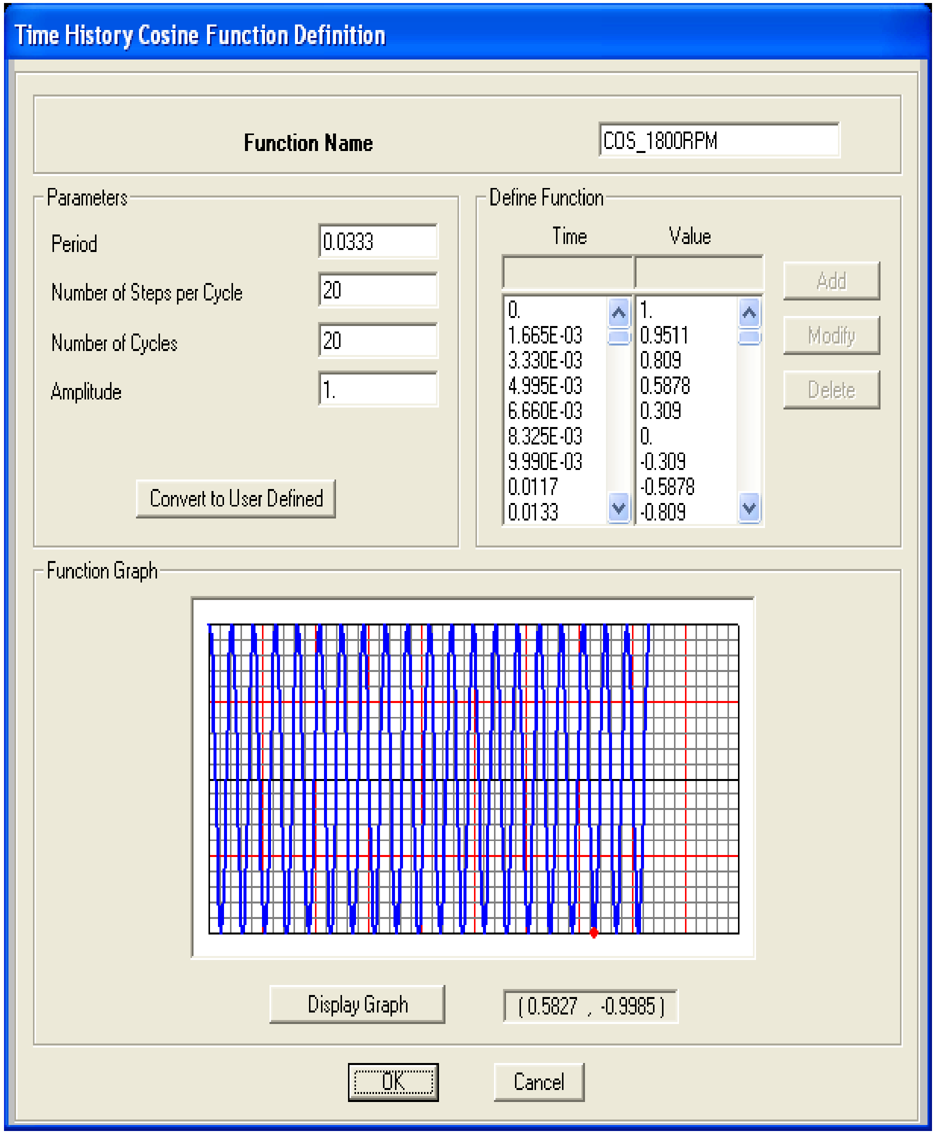
Period	0.0333
Number of Steps per Cycle	20
Number of Cycles	20
Amplitude	1.

Define Function

Time	Value
0.	0.
1.665E-03	0.309
3.330E-03	0.5878
4.995E-03	0.809
6.660E-03	0.9511
8.325E-03	1.
9.990E-03	0.9511
0.0117	0.809
0.0133	0.5878

Function Graph

(0.4517 , -0.3914)



Time History Cosine Function Definition

Function Name COS_1800RPM

Parameters

Period	0.0333
Number of Steps per Cycle	20
Number of Cycles	20
Amplitude	1.

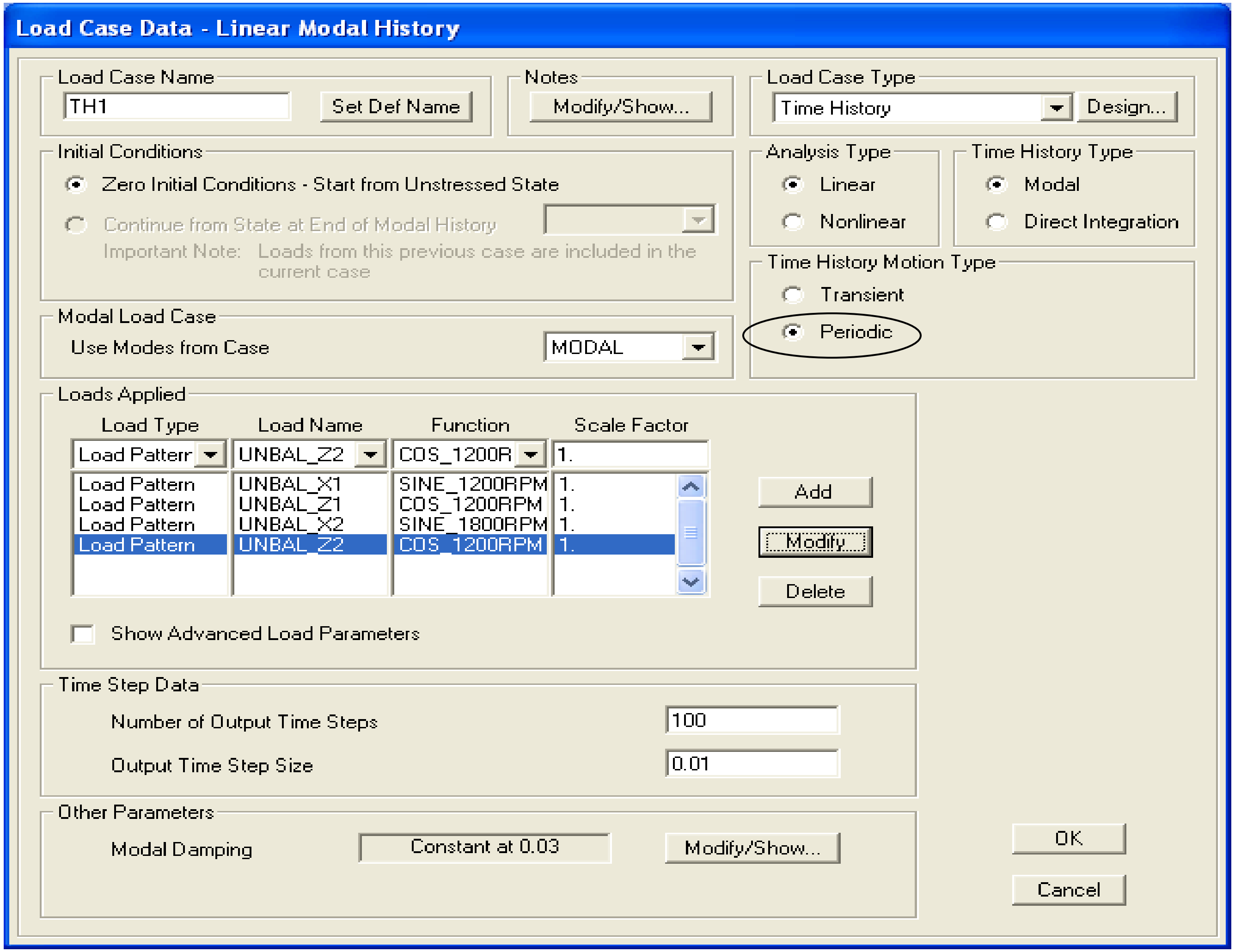
Define Function

Time	Value
0.	1.
1.665E-03	0.9511
3.330E-03	0.809
4.995E-03	0.5878
6.660E-03	0.309
8.325E-03	0.
9.990E-03	-0.309
0.0117	-0.5878
0.0133	-0.809

Function Graph

(0.5827 , -0.9985)

Define menu>Load cases, then click the ‘Add New Load Case’ button. Specify Load case type to be Time History as shown below, Linear “Periodic”. Here we add load patterns pairing them with their associated time history functions based on machine speed. X direction load patterns are paired with Sine functions and Z direction load patterns paired with Cosine functions. The number of output steps doesn’t have to exactly match the time history function when running a periodic TH, but for a transient TH analysis, the number of output time steps X time step size should be pretty close to the TH function period X number of cycles



Load Case Data - Linear Modal History

Load Case Name: TH1

Load Case Type: Time History

Initial Conditions
- ● Zero Initial Conditions - Start from Unstressed State
- ○ Continue from State at End of Modal History

Important Note: Loads from this previous case are included in the current case

Analysis Type
- ● Linear
- ○ Nonlinear

Time History Type
- ● Modal
- ○ Direct Integration

Time History Motion Type
- ○ Transient
- ● Periodic

Modal Load Case
Use Modes from Case: MODAL

Loads Applied

Load Type	Load Name	Function	Scale Factor
Load Patterr	UNBAL_Z2	COS_1200R	1.
Load Pattern	UNBAL_X1	SINE_1200RPM	1.
Load Pattern	UNBAL_Z1	COS_1200RPM	1.
Load Pattern	UNBAL_X2	SINE_1800RPM	1.
Load Pattern	UNBAL_Z2	COS_1200RPM	1.

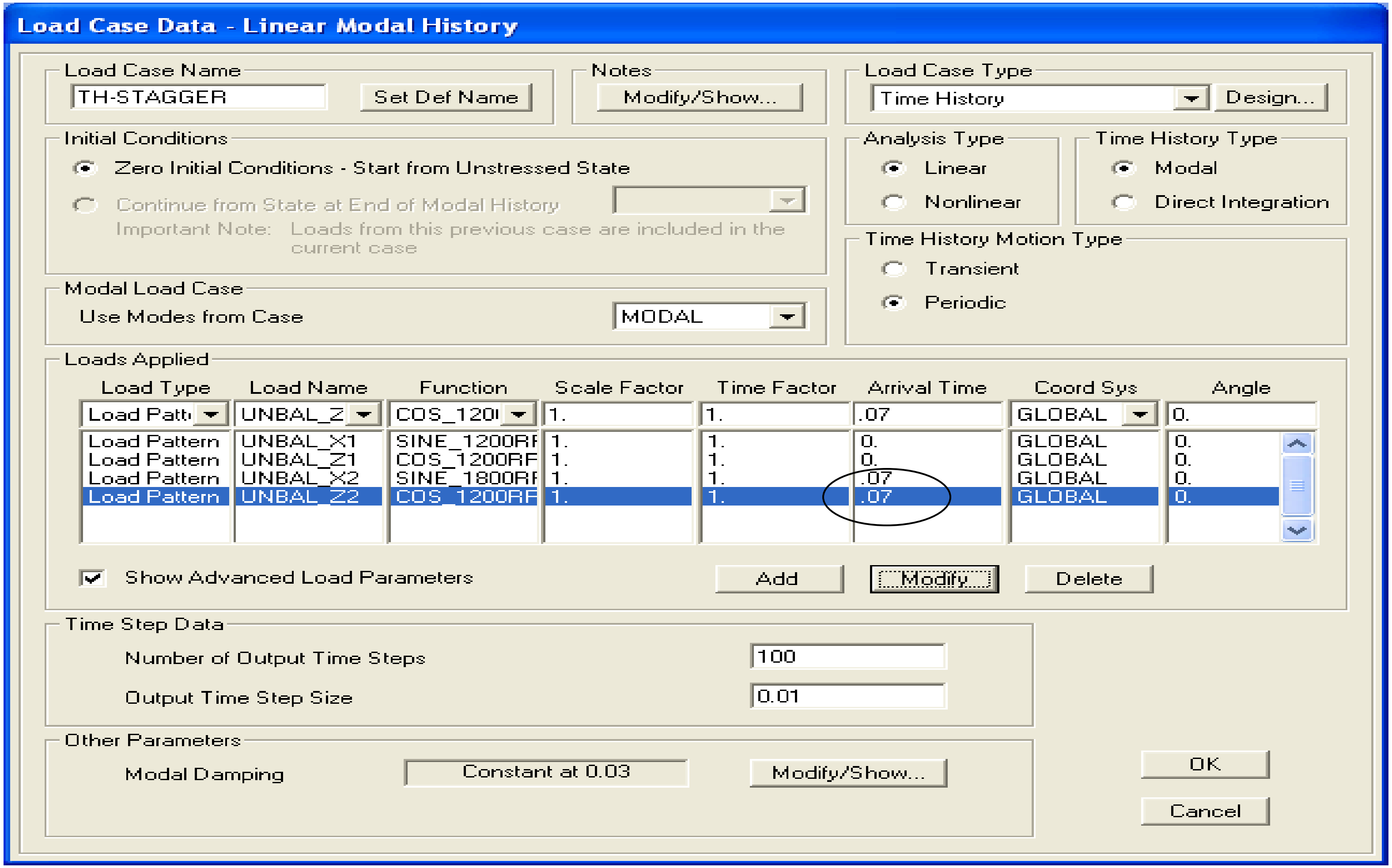
☐ Show Advanced Load Parameters

Time Step Data
Number of Output Time Steps: 100
Output Time Step Size: 0.01

Other Parameters
Modal Damping: Constant at 0.03

You can use the ‘Add copy of load case’ button to make a copy of the previous time history case and modify it to consider, for example, what would happen if one machine started out of synch from the other machine. In the case below, we use the “Advanced” load parameter to specify arrival times of loading.

There are no limits on the number of TH cases that you can run in 1 analysis. For now, we are basing our TH analyses on Eigen Modal analysis. There is an option to use Load Dependent Ritz vectors modes instead of Eigenvectors. In larger, more complicated structures, Ritz vectors are usually more accurate for TH or response spectrum analyses, and often require more than the default 12 modes. For a small skid, it usually doesn’t make much difference, so we’ll use the default Eigen Modal.



Load Case Data - Linear Modal History

Load Case Name: TH-STAGGER

Load Case Type: Time History

Initial Conditions:
● Zero Initial Conditions - Start from Unstressed State
○ Continue from State at End of Modal History
Important Note: Loads from this previous case are included in the current case

Analysis Type: ● Linear ○ Nonlinear

Time History Type: ● Modal ○ Direct Integration

Time History Motion Type: ○ Transient ● Periodic

Modal Load Case: Use Modes from Case: MODAL

Loads Applied

Load Type	Load Name	Function	Scale Factor	Time Factor	Arrival Time	Coord Sys	Angle
Load Patt	UNBAL_Z	COS_120	1.	1.	.07	GLOBAL	0.
Load Pattern	UNBAL_X1	SINE_1200RF	1.	1.	0.	GLOBAL	0.
Load Pattern	UNBAL_Z1	COS_1200RF	1.	1.	0.	GLOBAL	0.
Load Pattern	UNBAL_X2	SINE_1800RF	1.	1.	.07	GLOBAL	0.
Load Pattern	UNBAL_Z2	COS_1200RF	1.	1.	.07	GLOBAL	0.

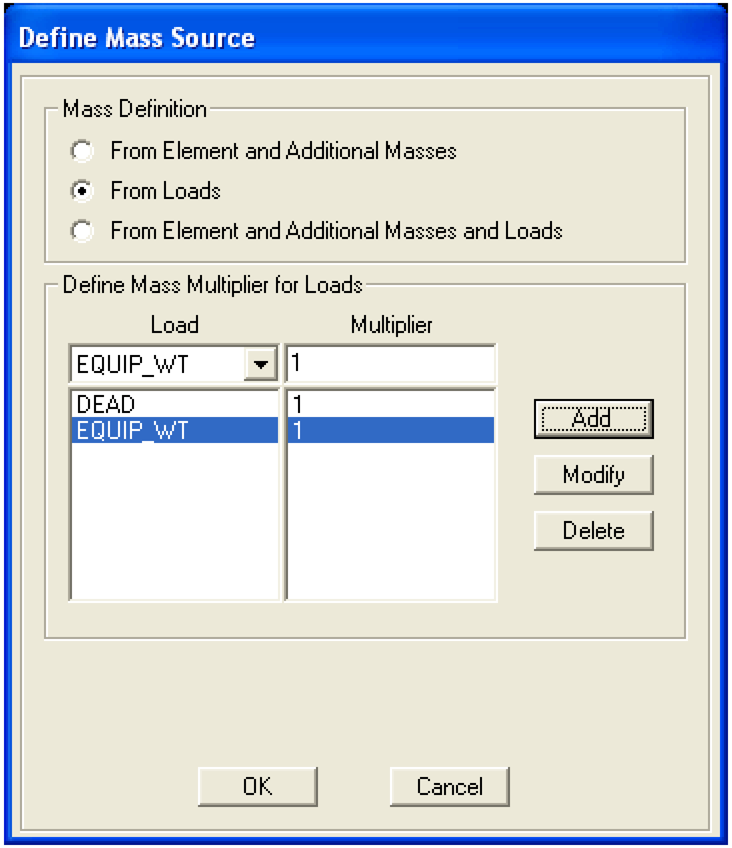
☑ Show Advanced Load Parameters

Time Step Data:
Number of Output Time Steps: 100
Output Time Step Size: 0.01

Other Parameters:
Modal Damping: Constant at 0.03

Since we assigned the vibrating machinery weight to a separate load pattern, in order to make sure it's included in the mass model use Define menu>Mass source and choose "From Loads" and add DEAD (element selfweight) and the Equipment load pattern as shown. If you had assigned other gravity loads (such as piping, cladding, stationary equipment, etc.), you would want to include them as well as part of the mass model.

Next, based on the soil report we will use Define>Section properties>Link/Support properties, choose "Linear type" to define soil damping (C) properties at selected joints in the U1, U2, and U3 local directions, which correspond to global X, Y, and Z directions. The values to be entered are the responsibility of the engineer based on soil data. Often, it's more efficient to assign the soil dampers at the pile/structure intersection joints using a link, then assign soil springs (K) using another method (Assign>Joint>Springs) at selected joints along the length of the pile.



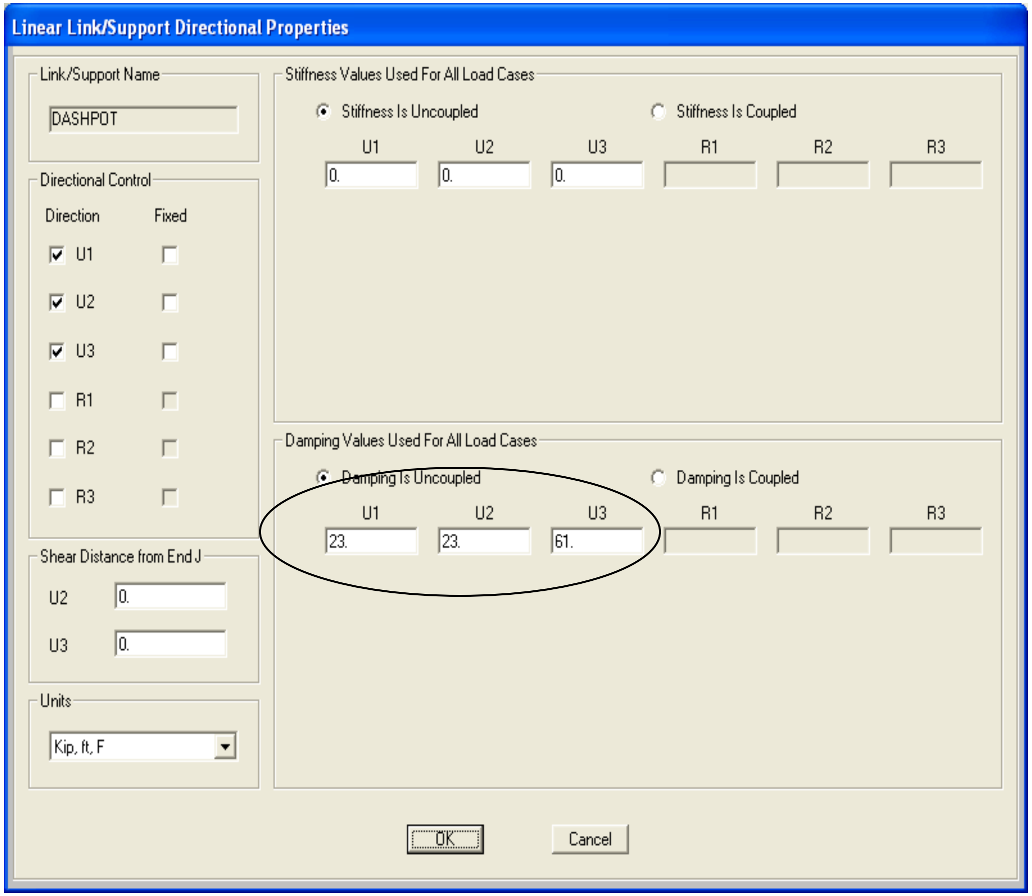
Define Mass Source

Mass Definition
○ From Element and Additional Masses
● From Loads
○ From Element and Additional Masses and Loads

Define Mass Multiplier for Loads

Load	Multiplier
EQUIP_WT	1
DEAD	1
EQUIP_WT	1

Damper link



Linear Link/Support Directional Properties

Link/Support Name: DASHPOT

Directional Control

Direction	Fixed
☑ U1	☐
☑ U2	☐
☑ U3	☐
☐ R1	☐
☐ R2	☐
☐ R3	☐

Shear Distance from End J
U2: 0.
U3: 0.

Units: Kip, ft, F

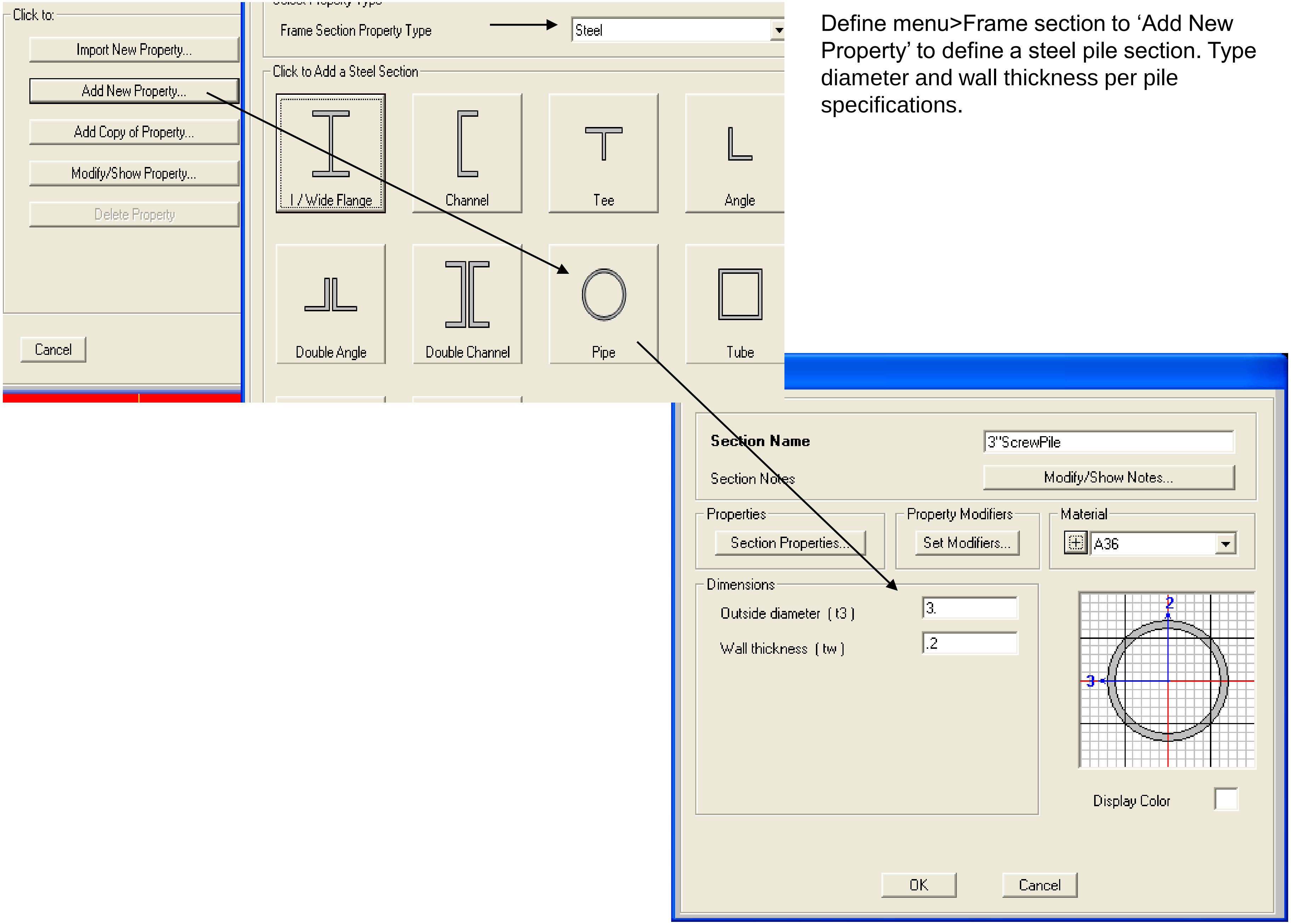
Stiffness Values Used For All Load Cases
● Stiffness Is Uncoupled ○ Stiffness Is Coupled

U1	U2	U3	R1	R2	R3
0.	0.	0.			

Damping Values Used For All Load Cases
● Damping Is Uncoupled ○ Damping Is Coupled

U1	U2	U3	R1	R2	R3
23.	23.	61.			

Define menu>Frame section to 'Add New Property' to define a steel pile section. Type diameter and wall thickness per pile specifications.



Frame Section Property Type: Steel

Section Name: 3"ScrewPile

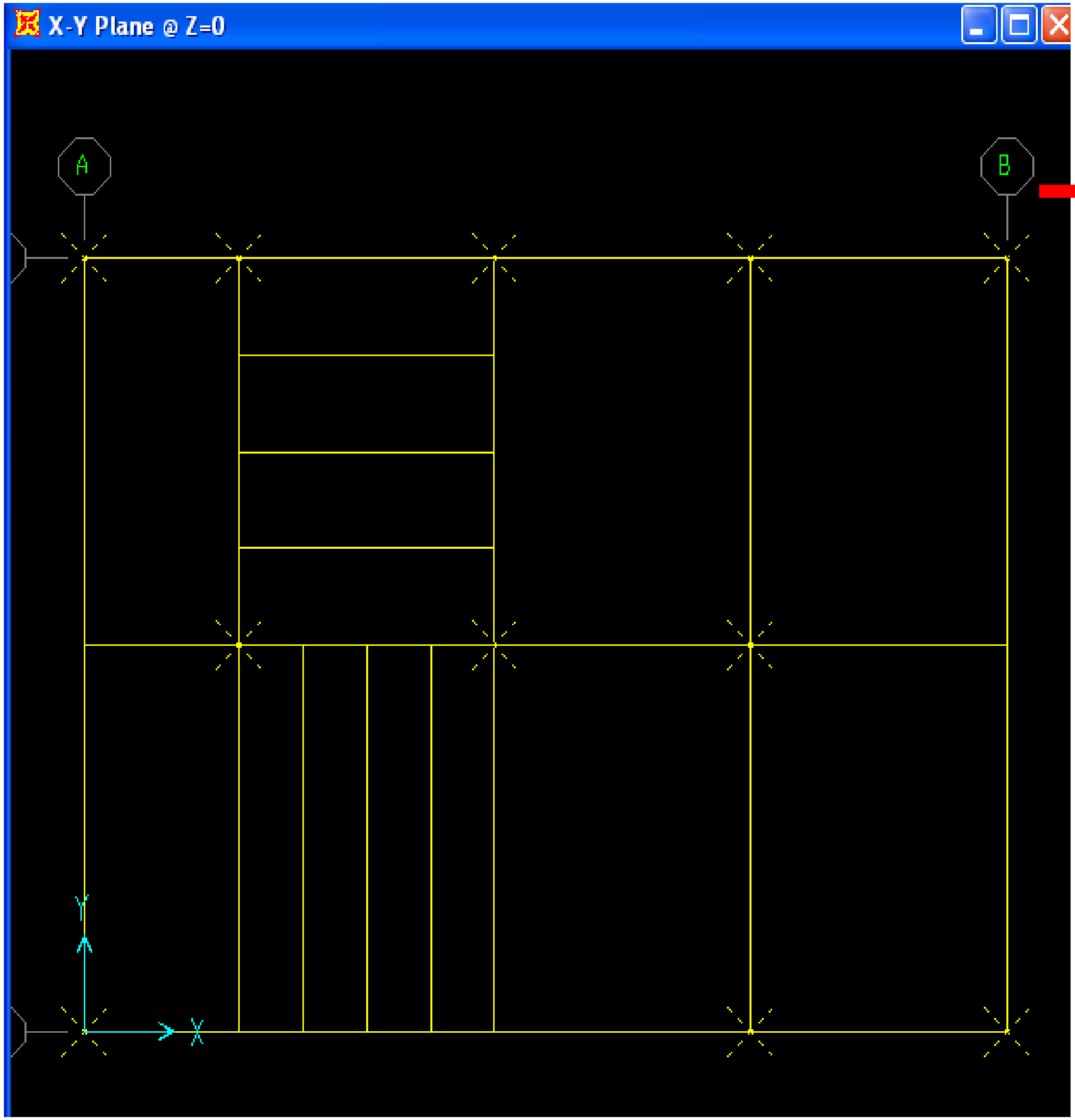
Material: A36

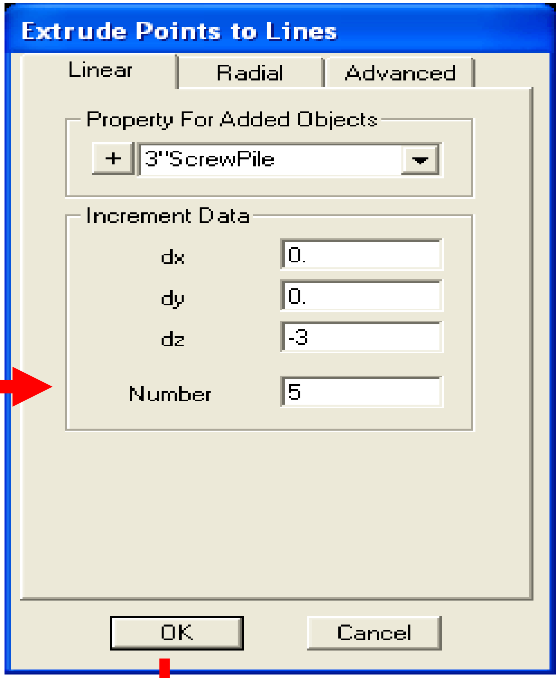
Dimensions

Outside diameter (t3): 3.

Wall thickness (tw): .2

In plan view select joints where piles are located. You may have to use the Draw special joint tool to add joints along the beams if there is not already a joint in place at a particular pile location. Next, Edit menu>Extrude>Extrude Points to frames using the Pile section as shown -3 Ft downward with 5 increments is a useful modeling technique. Later we will assign soil springs and dampers to the piles.




Extrude Points to Lines

Property For Added Objects: 3"ScrewPile

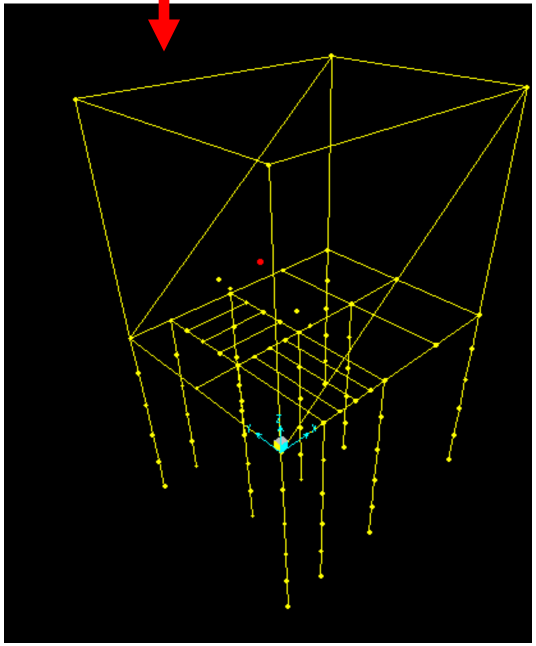
Increment Data

dx: 0.

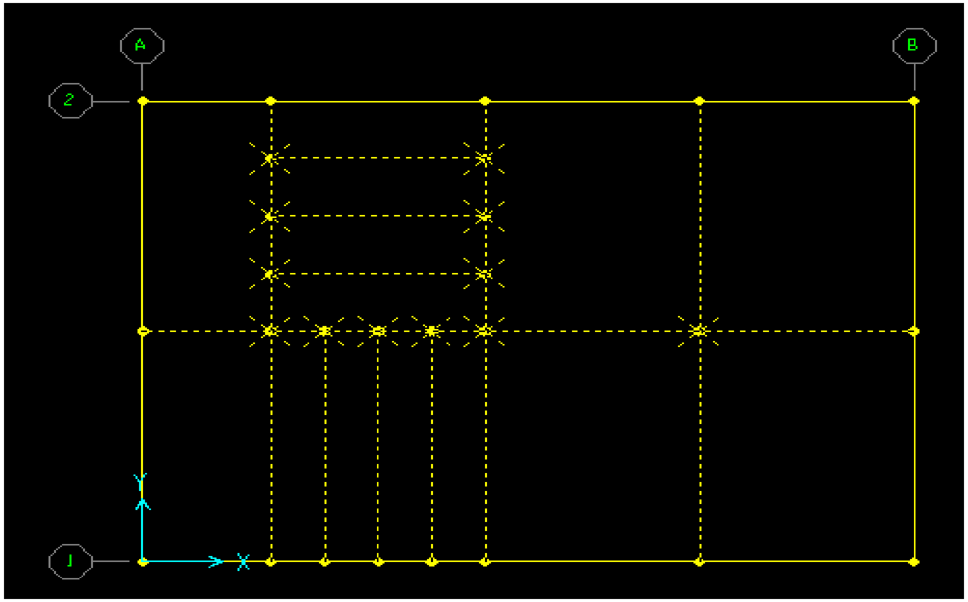
dy: 0.

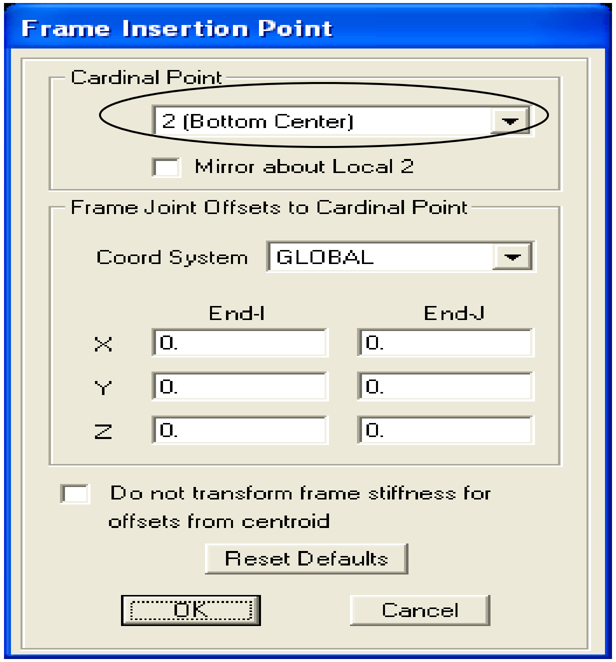
dz: -3

Number: 5



In some framing, the interior frames are connected on top of the main girders. Our default modeling assumed centerline connections, so select the interior beams and **Assign>Frame>Insertion point**. Here you can assign T.O.S and other cardinal point insertions. To the right are two options: Assign a cardinal point, or move the frame up 6", which is ½ the height of the girders. This will adjust moments based on the insertion point assignment.




Frame Insertion Point

Cardinal Point

2 (Bottom Center)

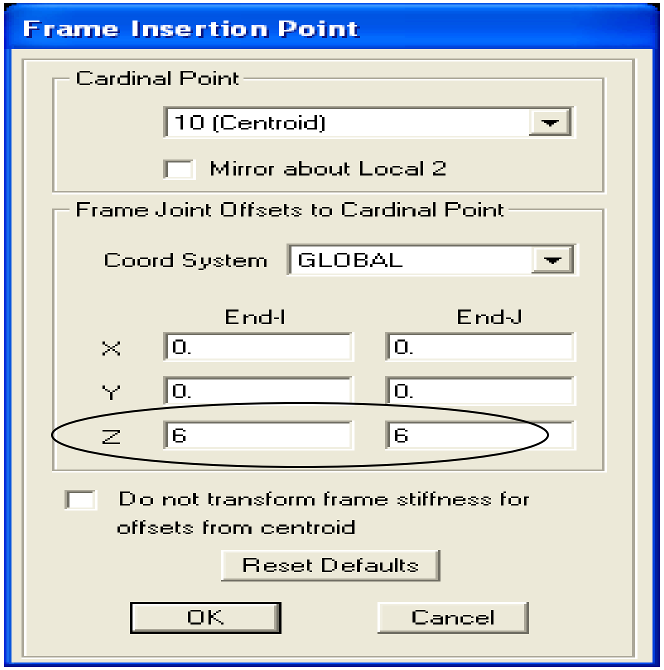
☐ Mirror about Local 2

Frame Joint Offsets to Cardinal Point

Coord System: GLOBAL

	End-I	End-J
X	0.	0.
Y	0.	0.
Z	0.	0.

☐ Do not transform frame stiffness for offsets from centroid


Frame Insertion Point

Cardinal Point

10 (Centroid)

☐ Mirror about Local 2

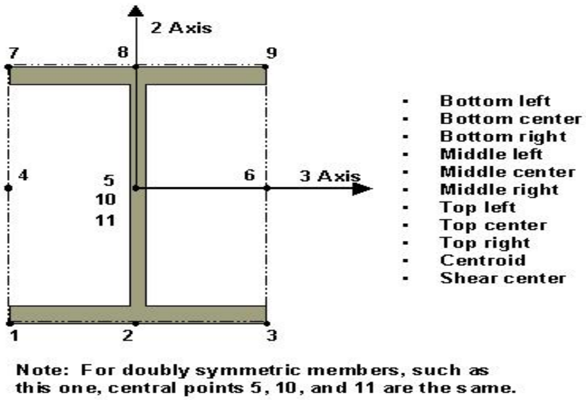
Frame Joint Offsets to Cardinal Point

Coord System: GLOBAL

	End-I	End-J
X	0.	0.
Y	0.	0.
Z	6	6

☐ Do not transform frame stiffness for offsets from centroid

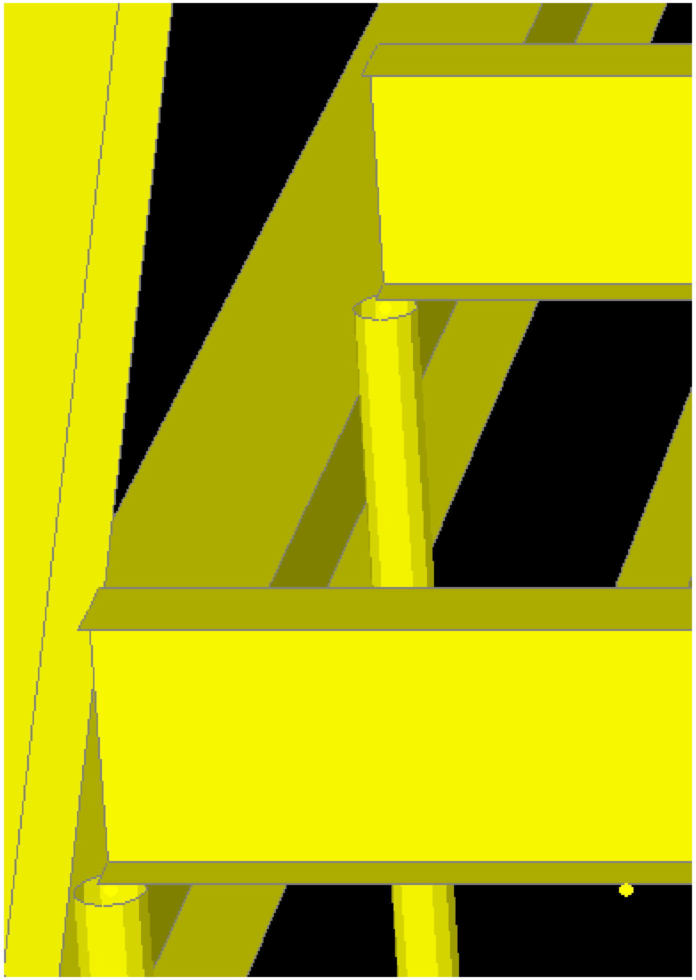
Press F1 for Help when you're in the Insertion point dialogue →

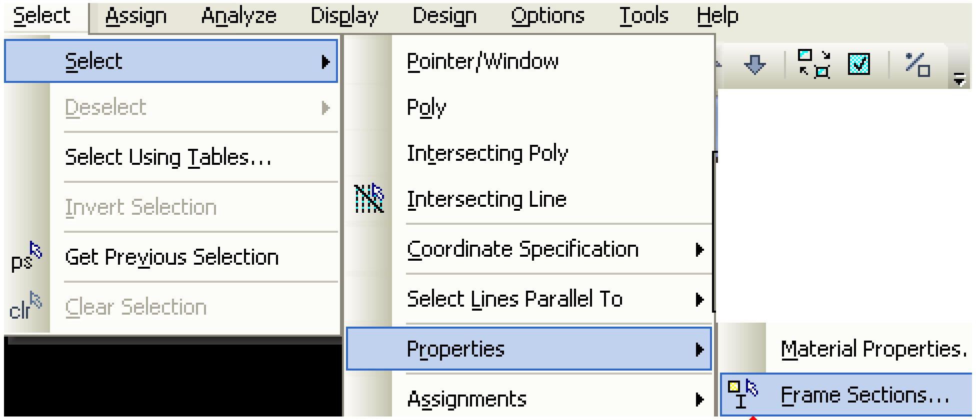


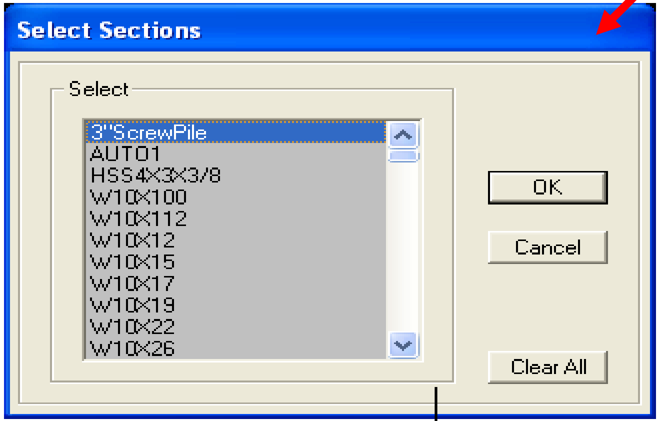
- Bottom left
- Bottom center
- Bottom right
- Middle left
- Middle center
- Middle right
- Top left
- Top center
- Top right
- Centroid
- Shear center

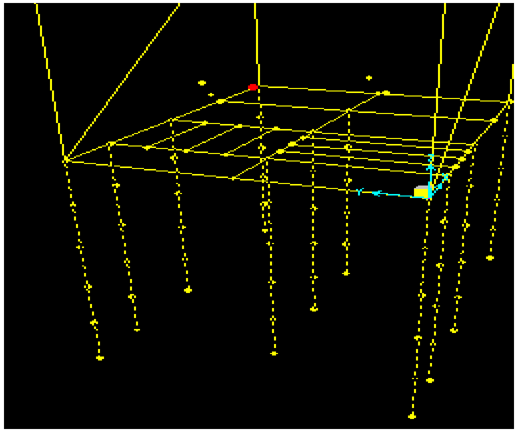
Note: For doubly symmetric members, such as this one, central points 5, 10, and 11 are the same.

Beams on top of the girders shown using Extruded view. The pile sections need to be adjusted next. You can use Select menu as shown right and select all the piles by properties>Frame sections as shown

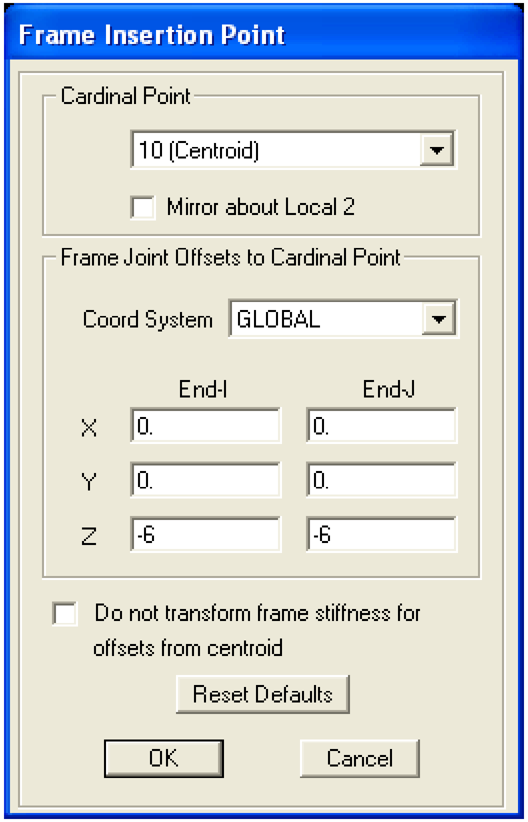








Assign>Frame>Insertion point as shown below to “move” the piles 6” in the –Z direction to connect at the bottom flange. Insertion point assignments move elements, but they are linked to the original joint location by an internal body constraint with is automatically added, and that constraint will account for moment differentials from the offsets


Frame Insertion Point

Cardinal Point

10 (Centroid)

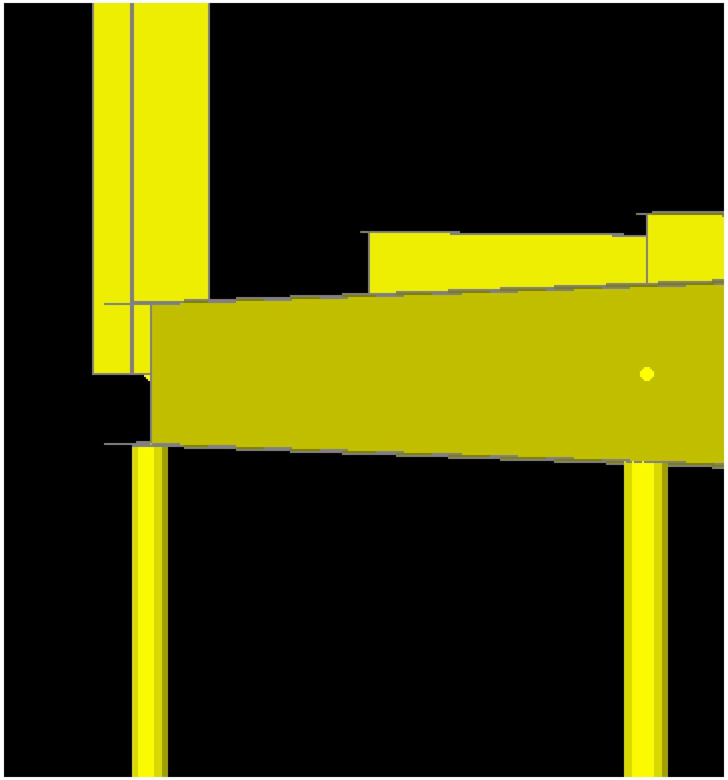
☐ Mirror about Local 2

Frame Joint Offsets to Cardinal Point

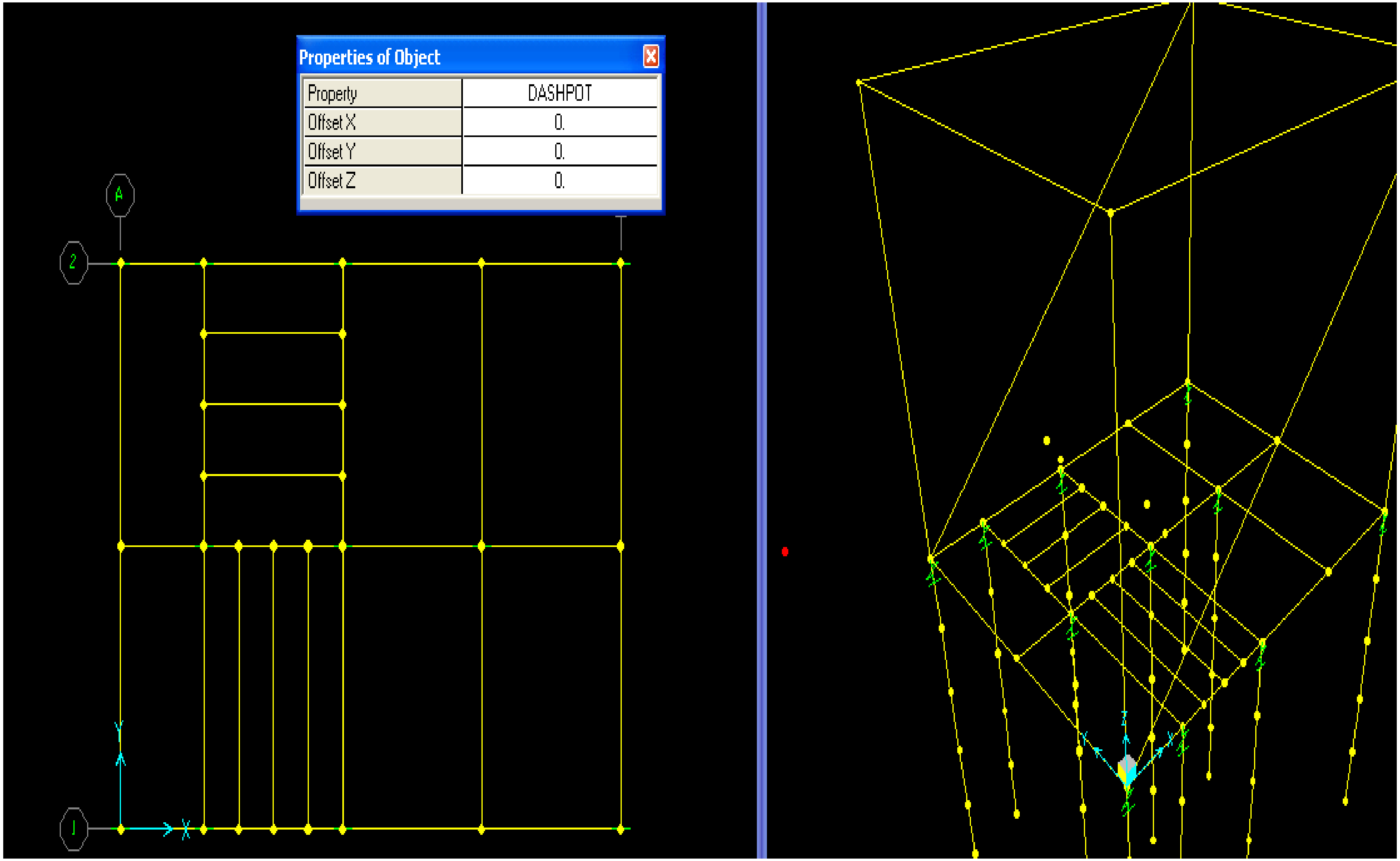
Coord System GLOBAL

	End-I	End-J
X	0.	0.
Y	0.	0.
Z	-6	-6

☐ Do not transform frame stiffness for offsets from centroid



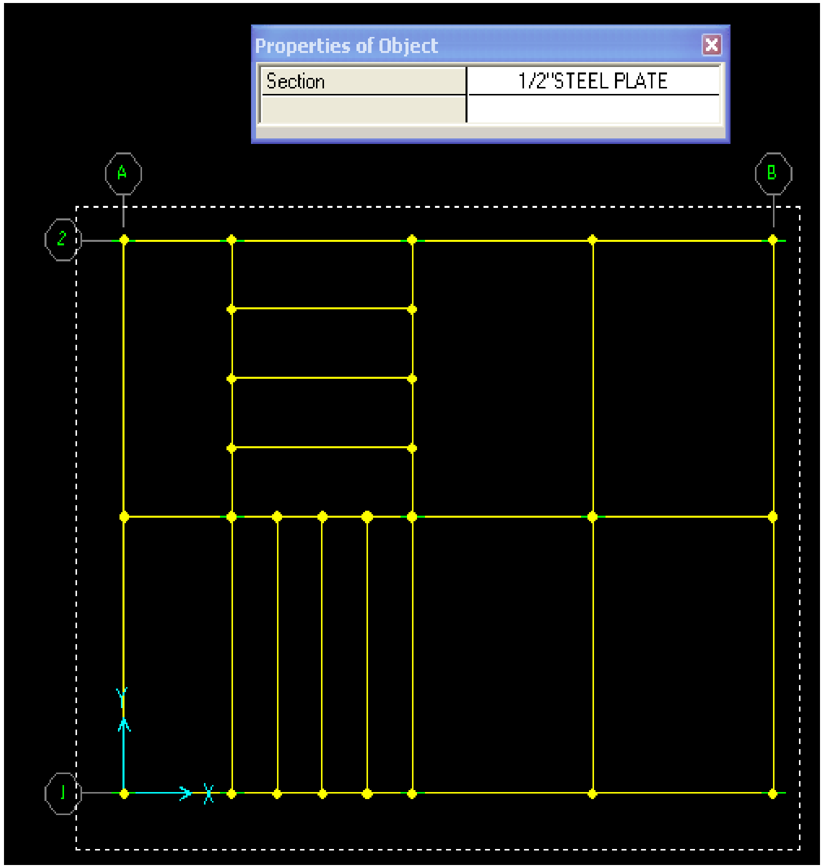
A common modeling approach is to lump the damping (C) at the joints where the pile intersects with the structure. Use Draw menu>Draw 1 joint links to add the Dashpot link. On the pile joints below, we will select them in a moment to assign lateral springs with a rotational spring component about the X.



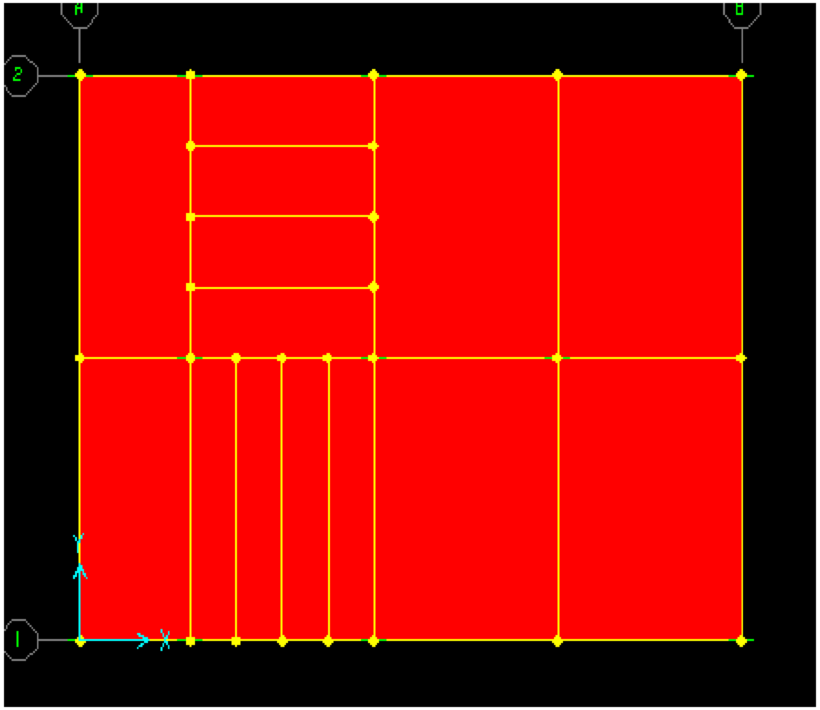
Properties of Object

Property	DASHPOT
Offset X	0.
Offset Y	0.
Offset Z	0.

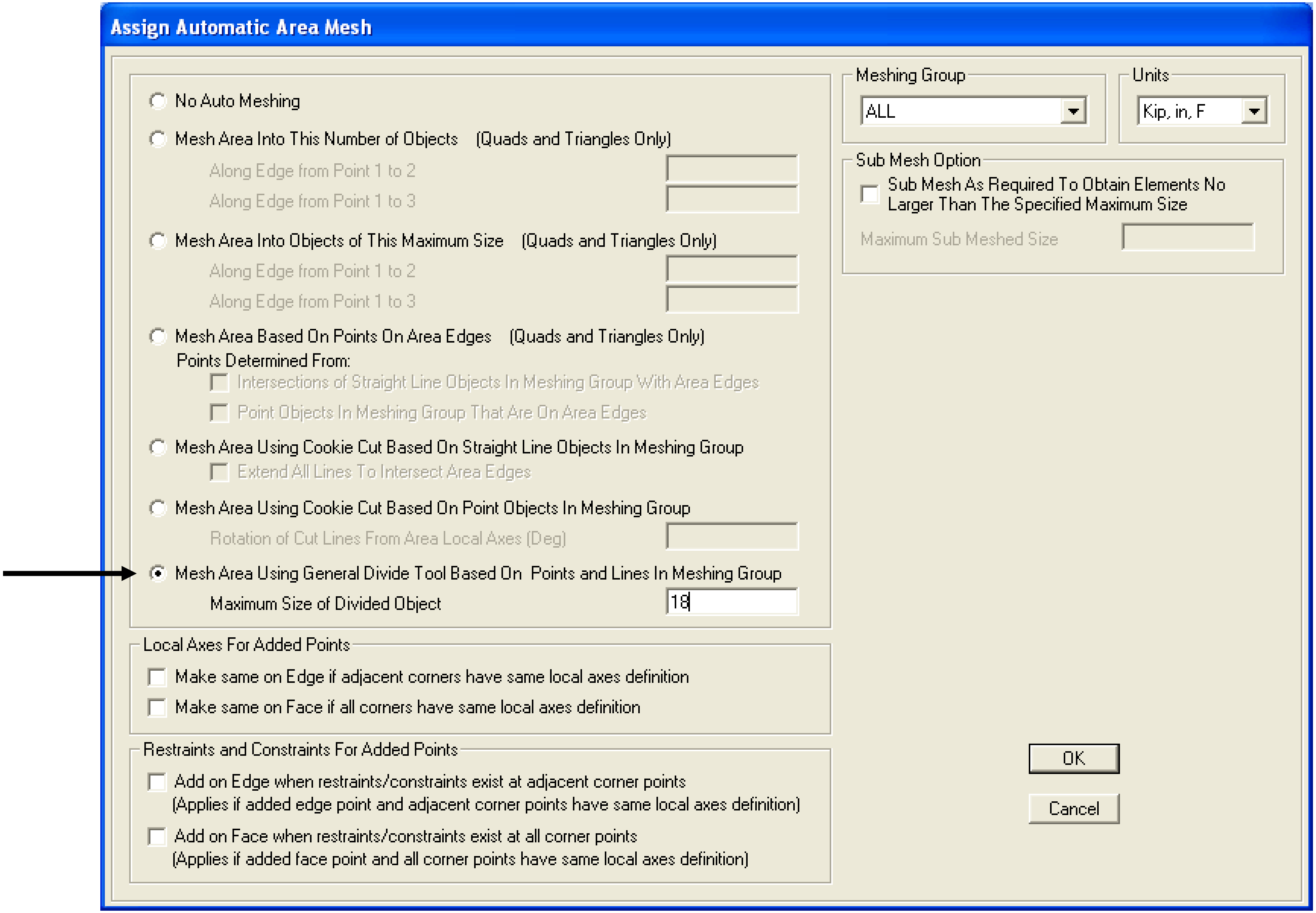
Before adding the soil springs to the piles, we will add a steel plate to the bottom. After having used Define menu>Section properties>Area sections to predefine a steel plate section (shell), in plan view, use ‘Quick draw area’ to window around the area to draw the plates using the steel plate section previously defined. This adds the steel plate sections, dividing the elements at visible grid intersections, which is a start, but it’s inadequate to connect to some of the interior beams, which is why we need to assign an area automesh. The nice thing about using the area automesh, is that if framing changes, the area mesh automatically changes, saving the user from having to rebuild and re-mesh sections of the model when design changes occur.



Section	1/2"STEEL PLATE



Select the steel plate/shell areas and Assign>Area>Automatic area mesh. Max size of divided element is 18” in this example. It’s engineering judgment as to area mesh sensitivity. Obviously a finer mesh can take longer to run. But this automesh tool makes it easy to run sensitivity checks with different max mesh sizes



Assign Automatic Area Mesh

○ No Auto Meshing

○ Mesh Area Into This Number of Objects (Quads and Triangles Only)
Along Edge from Point 1 to 2
Along Edge from Point 1 to 3

○ Mesh Area Into Objects of This Maximum Size (Quads and Triangles Only)
Along Edge from Point 1 to 2
Along Edge from Point 1 to 3

○ Mesh Area Based On Points On Area Edges (Quads and Triangles Only)
Points Determined From:
☐ Intersections of Straight Line Objects In Meshing Group With Area Edges
☐ Point Objects In Meshing Group That Are On Area Edges

○ Mesh Area Using Cookie Cut Based On Straight Line Objects In Meshing Group
☐ Extend All Lines To Intersect Area Edges

○ Mesh Area Using Cookie Cut Based On Point Objects In Meshing Group
Rotation of Cut Lines From Area Local Axes (Deg)

● Mesh Area Using General Divide Tool Based On Points and Lines In Meshing Group
Maximum Size of Divided Object: 18

Local Axes For Added Points
☐ Make same on Edge if adjacent corners have same local axes definition
☐ Make same on Face if all corners have same local axes definition

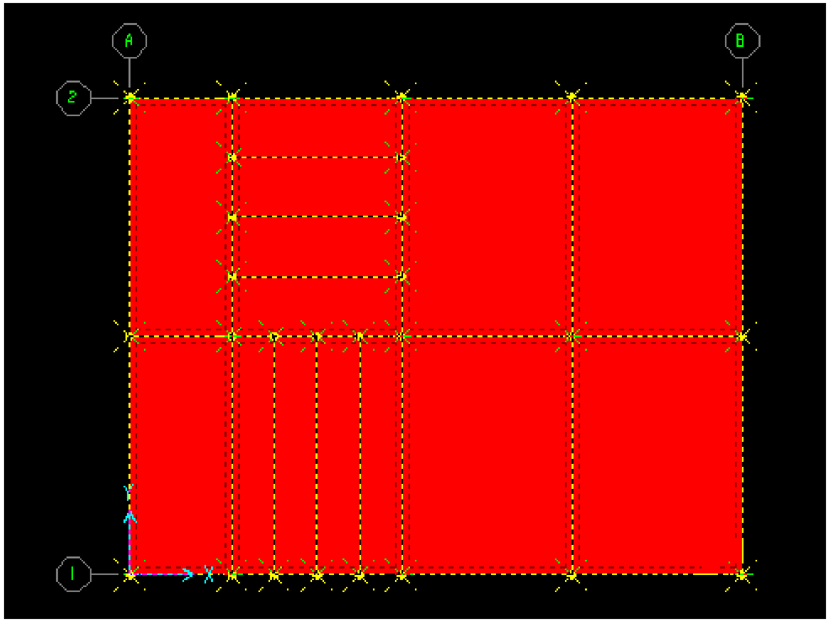
Restraints and Constraints For Added Points
☐ Add on Edge when restraints/constraints exist at adjacent corner points
(Applies if added edge point and adjacent corner points have same local axes definition)
☐ Add on Face when restraints/constraints exist at all corner points
(Applies if added face point and all corner points have same local axes definition)

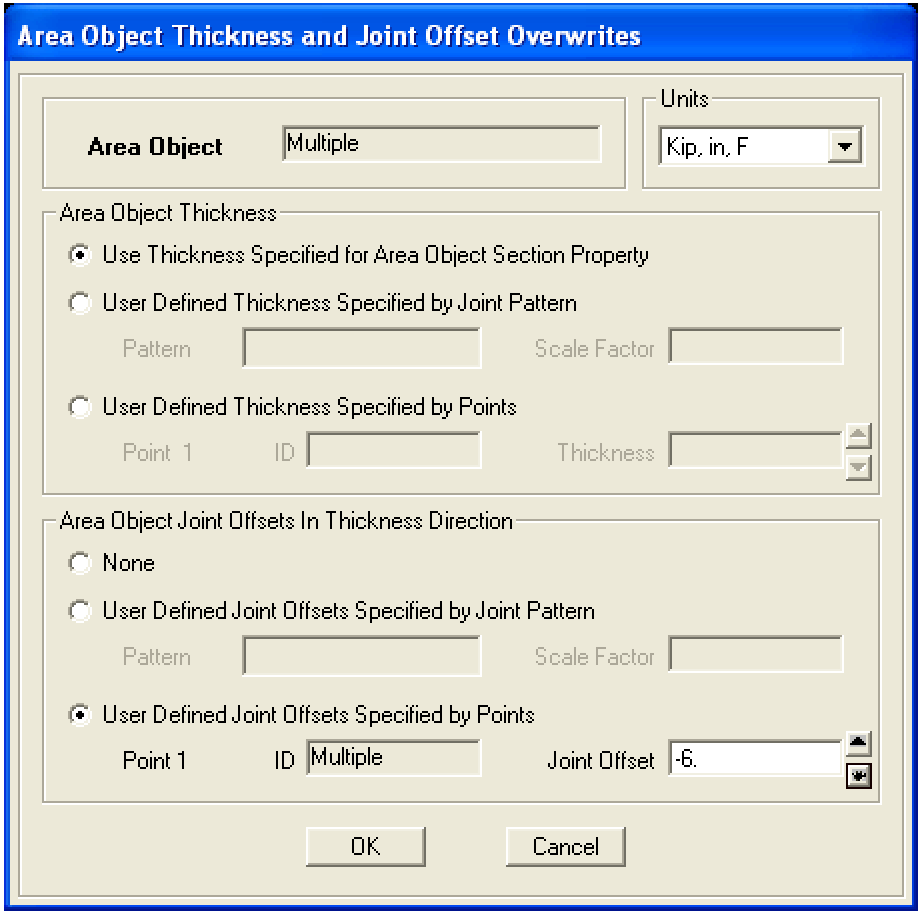
Meshing Group: ALL

Units: Kip, in, F

Sub Mesh Option
☐ Sub Mesh As Required To Obtain Elements No Larger Than The Specified Maximum Size
Maximum Sub Meshed Size

Since we modeled the steel plate at centerline, we'll need to "move" it to the bottom of the skid using a technique similar to the frame insertion point assignment. Select area elements then Assign>Area>Area thickness overwrites and specify, using current units (inches in this example) offsets in the local 3 normal direction, for each of 4 corners. In this example, -6 inches, ½ the depth of the girders. The screen below is Point 1. This area assignment must be repeated 3 more times for points 2, 3, and 4. See next slide.





Area Object Thickness and Joint Offset Overwrites

Area Object: Multiple

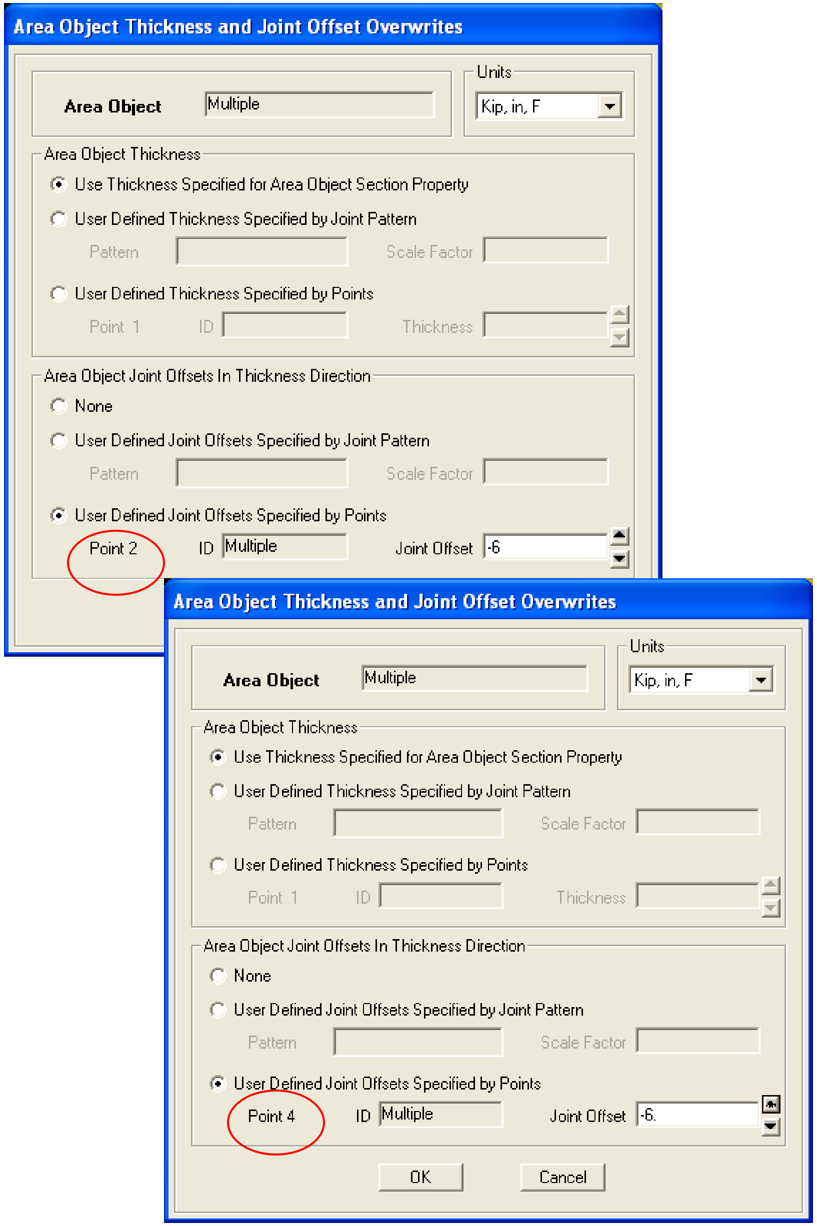
Units: Kip, in, F

Area Object Thickness

- ● Use Thickness Specified for Area Object Section Property
- ○ User Defined Thickness Specified by Joint Pattern
  - Pattern
  - Scale Factor
- ○ User Defined Thickness Specified by Points
  - Point 1 ID
  - Thickness

Area Object Joint Offsets In Thickness Direction

- ○ None
- ○ User Defined Joint Offsets Specified by Joint Pattern
  - Pattern
  - Scale Factor
- ● User Defined Joint Offsets Specified by Points
  - Point 1 ID: Multiple
  - Joint Offset: -6.



Area Object Thickness and Joint Offset Overwrites

Area Object Multiple

Units: Kip, in, F

Area Object Thickness

● Use Thickness Specified for Area Object Section Property

○ User Defined Thickness Specified by Joint Pattern

Pattern — Scale Factor

○ User Defined Thickness Specified by Points

Point 1 — ID — Thickness

Area Object Joint Offsets In Thickness Direction

○ None

○ User Defined Joint Offsets Specified by Joint Pattern

Pattern — Scale Factor

● User Defined Joint Offsets Specified by Points

Point 2 — ID: Multiple — Joint Offset: -6

Area Object Thickness and Joint Offset Overwrites

Area Object Multiple

Units: Kip, in, F

Area Object Thickness

● Use Thickness Specified for Area Object Section Property

○ User Defined Thickness Specified by Joint Pattern

Pattern — Scale Factor

○ User Defined Thickness Specified by Points

Point 1 — ID — Thickness

Area Object Joint Offsets In Thickness Direction

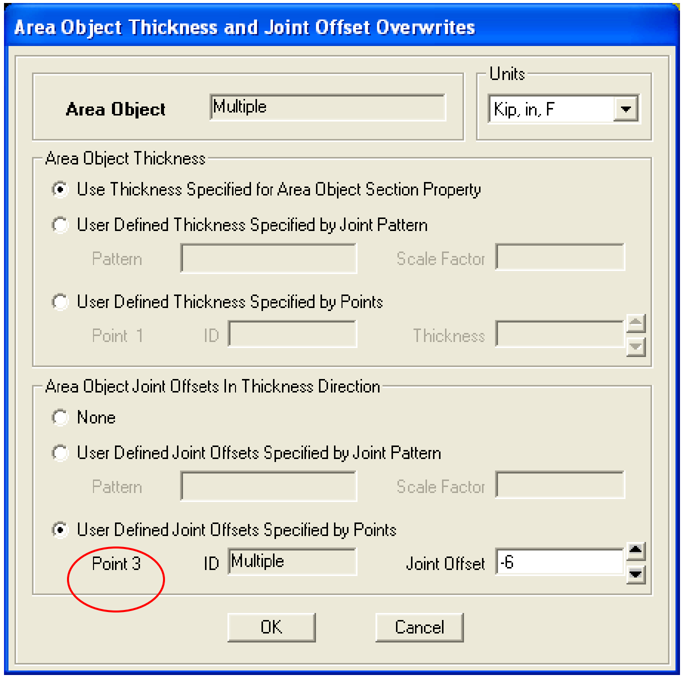
○ None

○ User Defined Joint Offsets Specified by Joint Pattern

Pattern — Scale Factor

● User Defined Joint Offsets Specified by Points

Point 4 — ID: Multiple — Joint Offset: -6.



Area Object Thickness and Joint Offset Overwrites

Area Object Multiple

Units: Kip, in, F

Area Object Thickness

● Use Thickness Specified for Area Object Section Property

○ User Defined Thickness Specified by Joint Pattern

Pattern — Scale Factor

○ User Defined Thickness Specified by Points

Point 1 — ID — Thickness

Area Object Joint Offsets In Thickness Direction

○ None

○ User Defined Joint Offsets Specified by Joint Pattern

Pattern — Scale Factor

● User Defined Joint Offsets Specified by Points

Point 3 — ID: Multiple — Joint Offset: -6

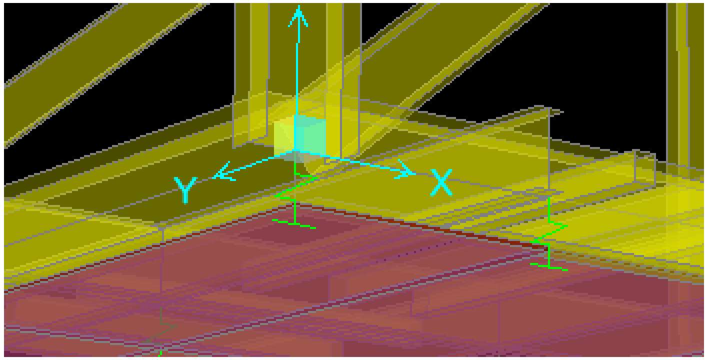
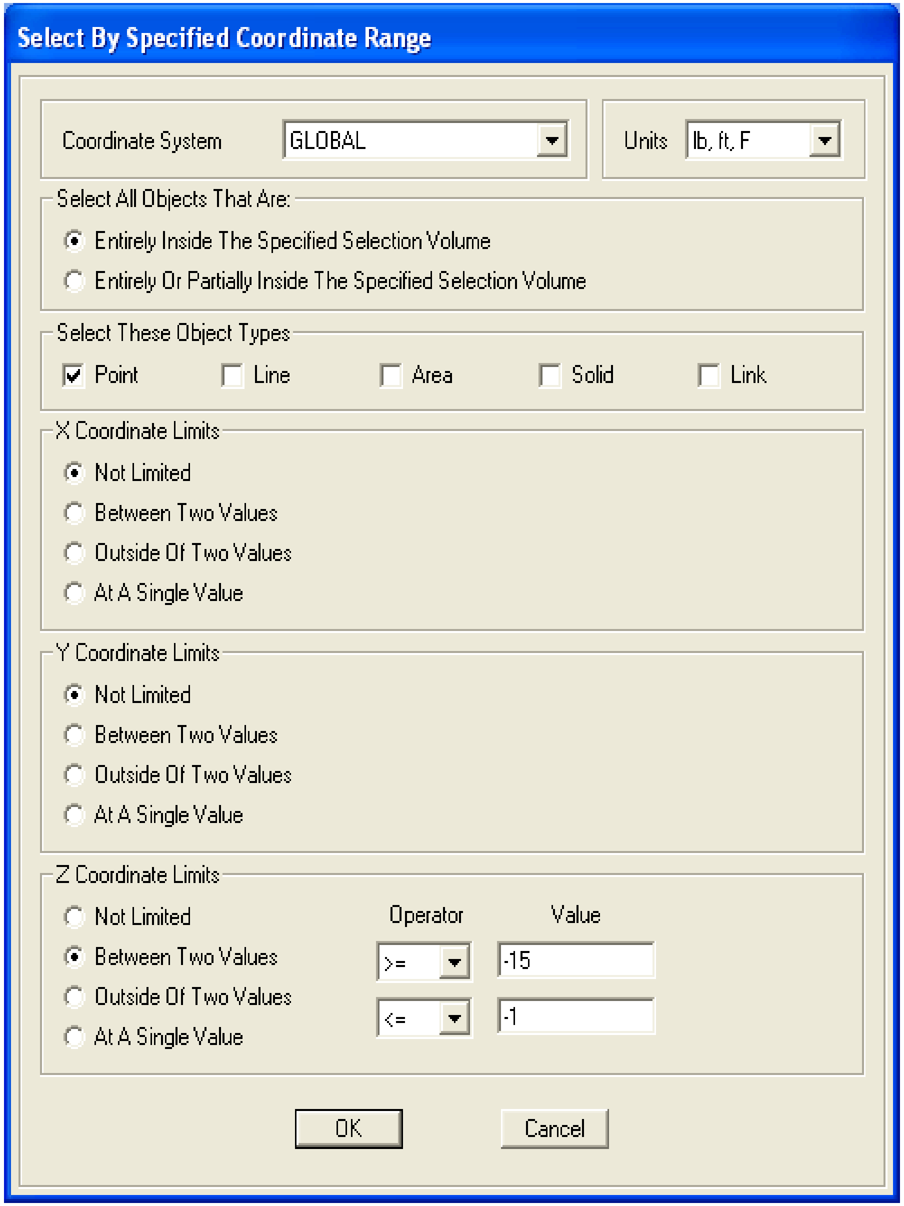


Plate at bottom of skid

Select menu>Select>Coordinate specifications all the joints below the pile/structure intersection in order to assign soil springs at pile joint locations based on soil data



Select By Specified Coordinate Range

Coordinate System: GLOBAL

Units: lb, ft, F

Select All Objects That Are:

- ● Entirely Inside The Specified Selection Volume
- ○ Entirely Or Partially Inside The Specified Selection Volume

Select These Object Types

☑ Point ☐ Line ☐ Area ☐ Solid ☐ Link

X Coordinate Limits

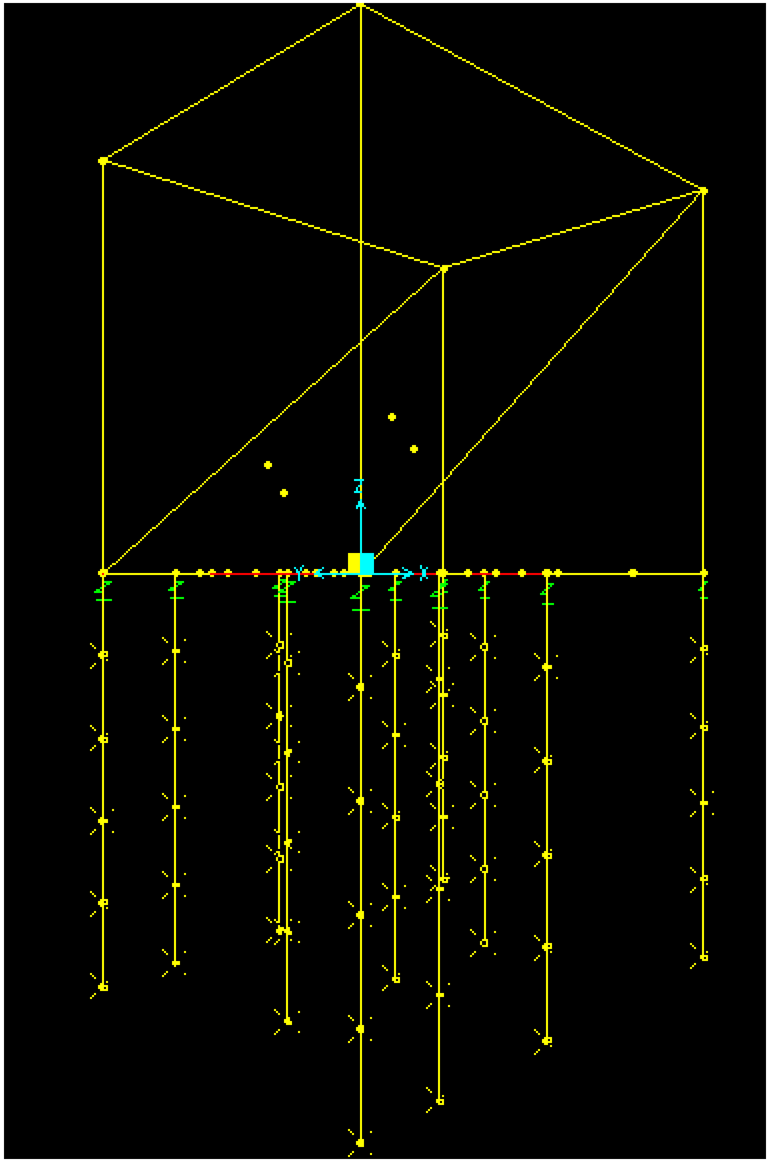
- ● Not Limited
- ○ Between Two Values
- ○ Outside Of Two Values
- ○ At A Single Value

Y Coordinate Limits

- ● Not Limited
- ○ Between Two Values
- ○ Outside Of Two Values
- ○ At A Single Value

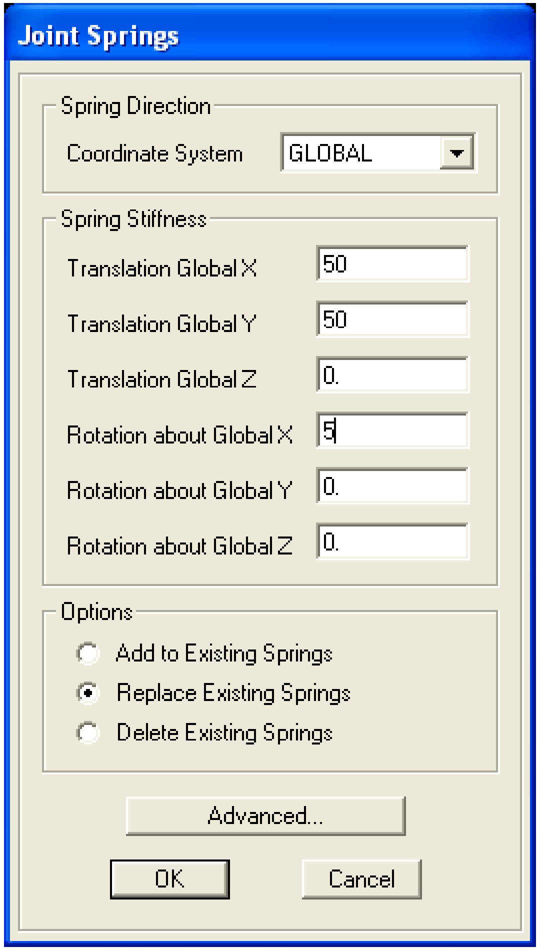
Z Coordinate Limits

	Operator	Value
○ Not Limited		
● Between Two Values	>=	-15
○ Outside Of Two Values	<=	-1
○ At A Single Value		



Next use Assign>Joint>Springs to assign soil joint springs along the piles. The values below are pulled out of thin air, but their ratios to each other are somewhat typical. Horizontal soil springs typically total 50% or less than vertical spring based on soil data. Often, engineers will lump the vertical spring at the bottom of the pile for dynamic analysis, while spreading the lateral soil springs at each meshed joint location along the length of the pile. A small rotational spring resisting rotations in the direction of the unbalanced loads are often assigned with the lateral springs along the length of the pile. I am not sure where engineers come up with the rotational spring values, possibly using recommendations from the book "*Design of Structures and Foundations for Vibrating Machines"* by Arya, O'Neill, and Pincus.

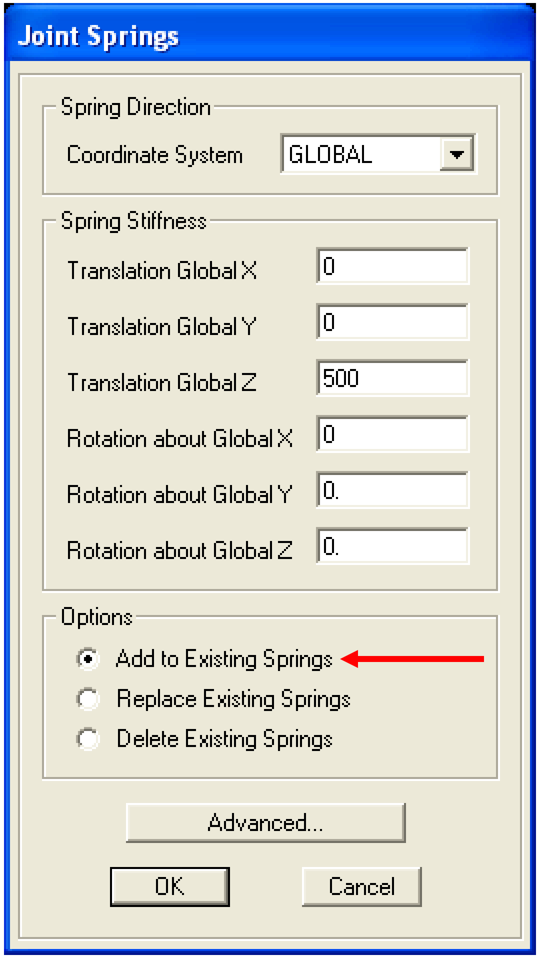
Lateral and rotational soil springs assigned to multiple joints along the length of each pile



Coordinate System	GLOBAL
Translation Global X	50
Translation Global Y	50
Translation Global Z	0.
Rotation about Global X	5
Rotation about Global Y	0.
Rotation about Global Z	0.

○ Add to Existing Springs
● Replace Existing Springs
○ Delete Existing Springs

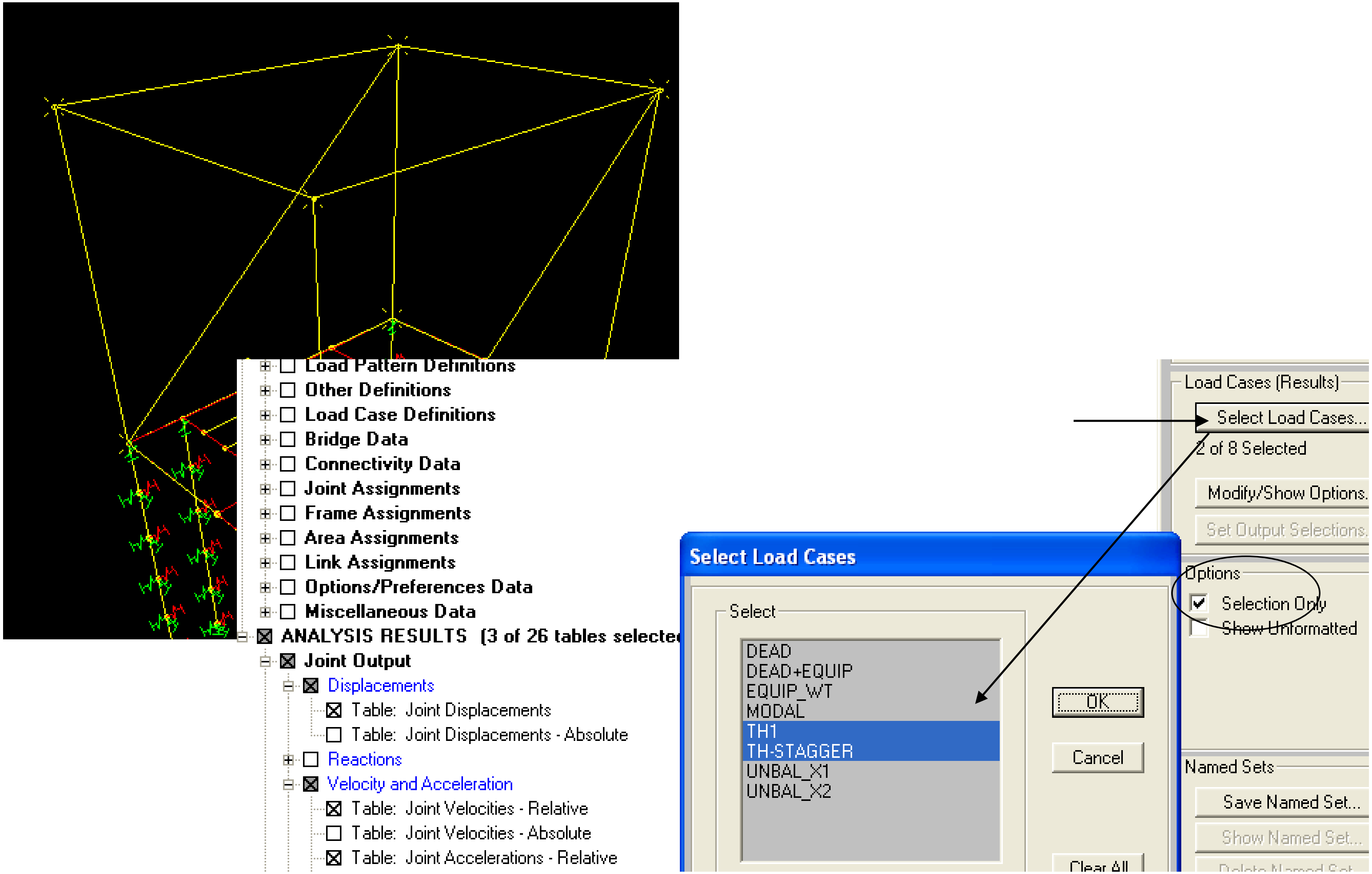
Vertical spring is often lumped at the bottom of each pile for dynamic analysis



Coordinate System	GLOBAL
Translation Global X	0
Translation Global Y	0
Translation Global Z	500
Rotation about Global X	0
Rotation about Global Y	0.
Rotation about Global Z	0.

● Add to Existing Springs
○ Replace Existing Springs
○ Delete Existing Springs

Analyze the model and after the analysis is complete, in order to minimize output for checking deflection amplitudes, velocities, and accelerations, click to select a few joints where the highest displacements are likely to occur in order to minimize output. By selecting joints, SAP2000 will only reported results for the selected joints. Use Display menu>Show tables, but click the 'Select load cases' to generate tables only for the TH cases as shown. Output selection options shown below left are typical.



- ☐ Load Pattern Definitions
- ☐ Other Definitions
- ☐ Load Case Definitions
- ☐ Bridge Data
- ☐ Connectivity Data
- ☐ Joint Assignments
- ☐ Frame Assignments
- ☐ Area Assignments
- ☐ Link Assignments
- ☐ Options/Preferences Data
- ☐ Miscellaneous Data
- ☒ ANALYSIS RESULTS (3 of 26 tables selecte
  - ☒ Joint Output
    - ☒ Displacements
      - ☒ Table: Joint Displacements
      - ☐ Table: Joint Displacements - Absolute
    - ☐ Reactions
    - ☒ Velocity and Acceleration
      - ☒ Table: Joint Velocities - Relative
      - ☐ Table: Joint Velocities - Absolute
      - ☒ Table: Joint Accelerations - Relative

Select Load Cases

Select

DEAD
DEAD+EQUIP
EQUIP_WT
MODAL
TH1
TH-STAGGER
UNBAL_X1
UNBAL_X2

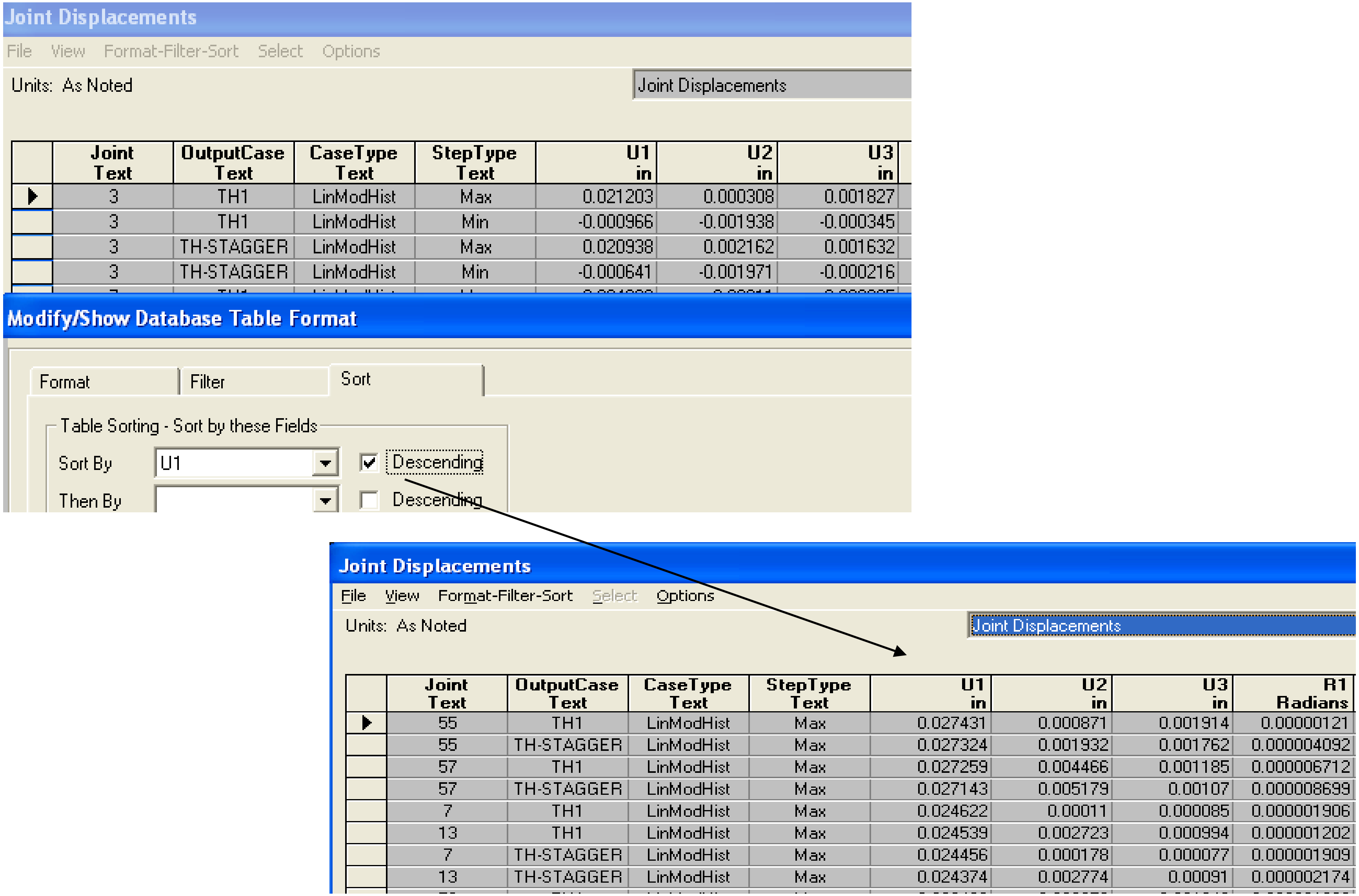
Load Cases (Results)

2 of 8 Selected

Options

☑ Selection Only
☐ Show Unformatted

Use the Format-Filter-Sort to sort for min and max values. In the example below, sorting for max U1 (X direction) deflections



Units: As Noted

Joint Text	OutputCase Text	CaseType Text	StepType Text	U1 in	U2 in	U3 in
3	TH1	LinModHist	Max	0.021203	0.000308	0.001827
3	TH1	LinModHist	Min	-0.000966	-0.001938	-0.000345
3	TH-STAGGER	LinModHist	Max	0.020938	0.002162	0.001632
3	TH-STAGGER	LinModHist	Min	-0.000641	-0.001971	-0.000216

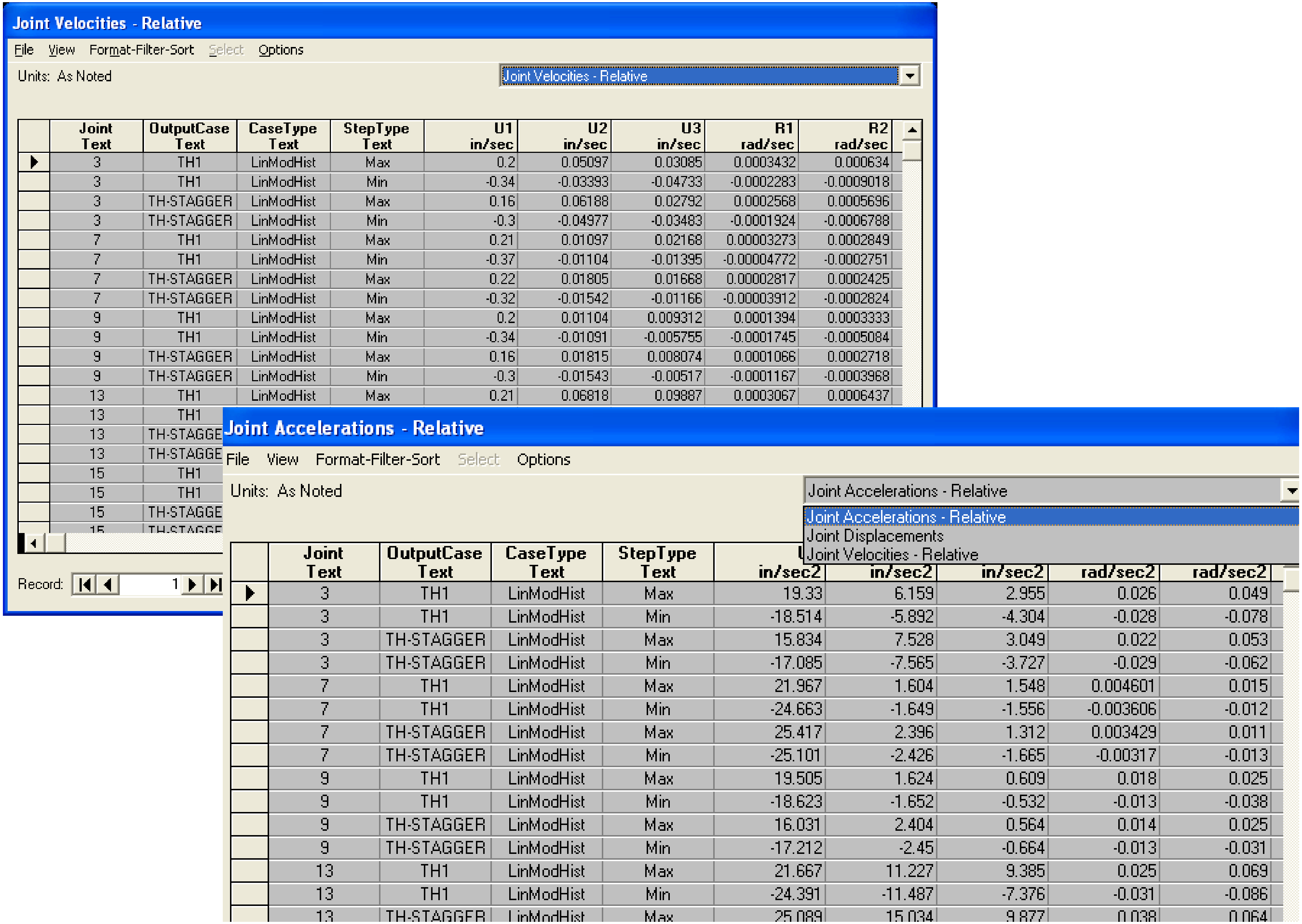
Table Sorting - Sort by these Fields

Sort By U1 ☑ Descending

Then By ☐ Descending

Units: As Noted

Joint Text	OutputCase Text	CaseType Text	StepType Text	U1 in	U2 in	U3 in	R1 Radians
55	TH1	LinModHist	Max	0.027431	0.000871	0.001914	0.00000121
55	TH-STAGGER	LinModHist	Max	0.027324	0.001932	0.001762	0.000004092
57	TH1	LinModHist	Max	0.027259	0.004466	0.001185	0.000006712
57	TH-STAGGER	LinModHist	Max	0.027143	0.005179	0.00107	0.000008699
7	TH1	LinModHist	Max	0.024622	0.00011	0.000085	0.000001906
13	TH1	LinModHist	Max	0.024539	0.002723	0.000994	0.000001202
7	TH-STAGGER	LinModHist	Max	0.024456	0.000178	0.000077	0.000001909
13	TH-STAGGER	LinModHist	Max	0.024374	0.002774	0.00091	0.000002174



Joint Velocities - Relative

Units: As Noted

Joint Velocities - Relative

	Joint Text	OutputCase Text	CaseType Text	StepType Text	U1 in/sec	U2 in/sec	U3 in/sec	R1 rad/sec	R2 rad/sec
▶	3	TH1	LinModHist	Max	0.2	0.05097	0.03085	0.0003432	0.000634
	3	TH1	LinModHist	Min	-0.34	-0.03393	-0.04733	-0.0002283	-0.0009018
	3	TH-STAGGER	LinModHist	Max	0.16	0.06188	0.02792	0.0002568	0.0005696
	3	TH-STAGGER	LinModHist	Min	-0.3	-0.04977	-0.03483	-0.0001924	-0.0006788
	7	TH1	LinModHist	Max	0.21	0.01097	0.02168	0.00003273	0.0002849
	7	TH1	LinModHist	Min	-0.37	-0.01104	-0.01395	-0.00004772	-0.0002751
	7	TH-STAGGER	LinModHist	Max	0.22	0.01805	0.01668	0.00002817	0.0002425
	7	TH-STAGGER	LinModHist	Min	-0.32	-0.01542	-0.01166	-0.00003912	-0.0002824
	9	TH1	LinModHist	Max	0.2	0.01104	0.009312	0.0001394	0.0003333
	9	TH1	LinModHist	Min	-0.34	-0.01091	-0.005755	-0.0001745	-0.0005084
	9	TH-STAGGER	LinModHist	Max	0.16	0.01815	0.008074	0.0001066	0.0002718
	9	TH-STAGGER	LinModHist	Min	-0.3	-0.01543	-0.00517	-0.0001167	-0.0003968
	13	TH1	LinModHist	Max	0.21	0.06818	0.09887	0.0003067	0.0006437
	13	TH1							
	13	TH-STAGGE							
	13	TH-STAGGE							
	15	TH1							
	15	TH1							
	15	TH-STAGGE							
	15	TH-STAGGE							

Joint Accelerations - Relative

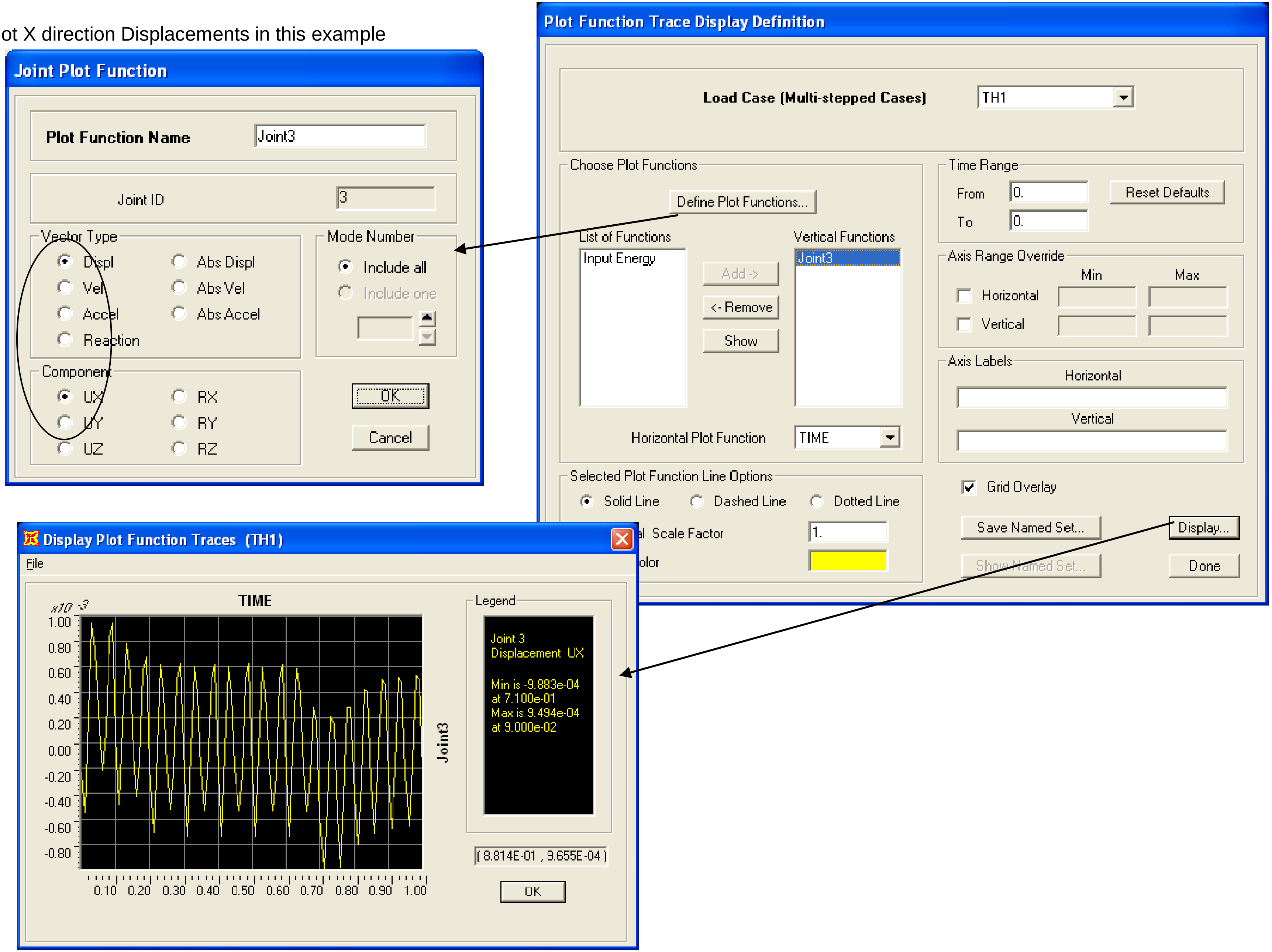
Units: As Noted

Joint Accelerations - Relative

	Joint Text	OutputCase Text	CaseType Text	StepType Text	U in/sec2	in/sec2	in/sec2	rad/sec2	rad/sec2
▶	3	TH1	LinModHist	Max	19.33	6.159	2.955	0.026	0.049
	3	TH1	LinModHist	Min	-18.514	-5.892	-4.304	-0.028	-0.078
	3	TH-STAGGER	LinModHist	Max	15.834	7.528	3.049	0.022	0.053
	3	TH-STAGGER	LinModHist	Min	-17.085	-7.565	-3.727	-0.029	-0.062
	7	TH1	LinModHist	Max	21.967	1.604	1.548	0.004601	0.015
	7	TH1	LinModHist	Min	-24.663	-1.649	-1.556	-0.003606	-0.012
	7	TH-STAGGER	LinModHist	Max	25.417	2.396	1.312	0.003429	0.011
	7	TH-STAGGER	LinModHist	Min	-25.101	-2.426	-1.665	-0.00317	-0.013
	9	TH1	LinModHist	Max	19.505	1.624	0.609	0.018	0.025
	9	TH1	LinModHist	Min	-18.623	-1.652	-0.532	-0.013	-0.038
	9	TH-STAGGER	LinModHist	Max	16.031	2.404	0.564	0.014	0.025
	9	TH-STAGGER	LinModHist	Min	-17.212	-2.45	-0.664	-0.013	-0.031
	13	TH1	LinModHist	Max	21.667	11.227	9.385	0.025	0.069
	13	TH1	LinModHist	Min	-24.391	-11.487	-7.376	-0.031	-0.086
	13	TH-STAGGER	LinModHist	Max	25.089	15.034	9.877	0.038	0.064

Click to select a joint of interest, then use Display menu>Show plot functions

Plot X direction Displacements in this example



Joint Plot Function

Plot Function Name: Joint3

Joint ID: 3

Vector Type:
- ● Displ
- ○ Vel
- ○ Accel
- ○ Reaction
- ○ Abs Displ
- ○ Abs Vel
- ○ Abs Accel

Mode Number:
- ● Include all
- ○ Include one

Component:
- ● UX
- ○ UY
- ○ UZ
- ○ RX
- ○ RY
- ○ RZ

Plot Function Trace Display Definition

Load Case (Multi-stepped Cases): TH1

Choose Plot Functions

List of Functions: Input Energy

Vertical Functions: Joint3

Horizontal Plot Function: TIME

Time Range: From 0. To 0.

Axis Range Override:
- ☐ Horizontal
- ☐ Vertical

Axis Labels: Horizontal, Vertical

Selected Plot Function Line Options:
- ● Solid Line
- ○ Dashed Line
- ○ Dotted Line

Scale Factor: 1.

☑ Grid Overlay

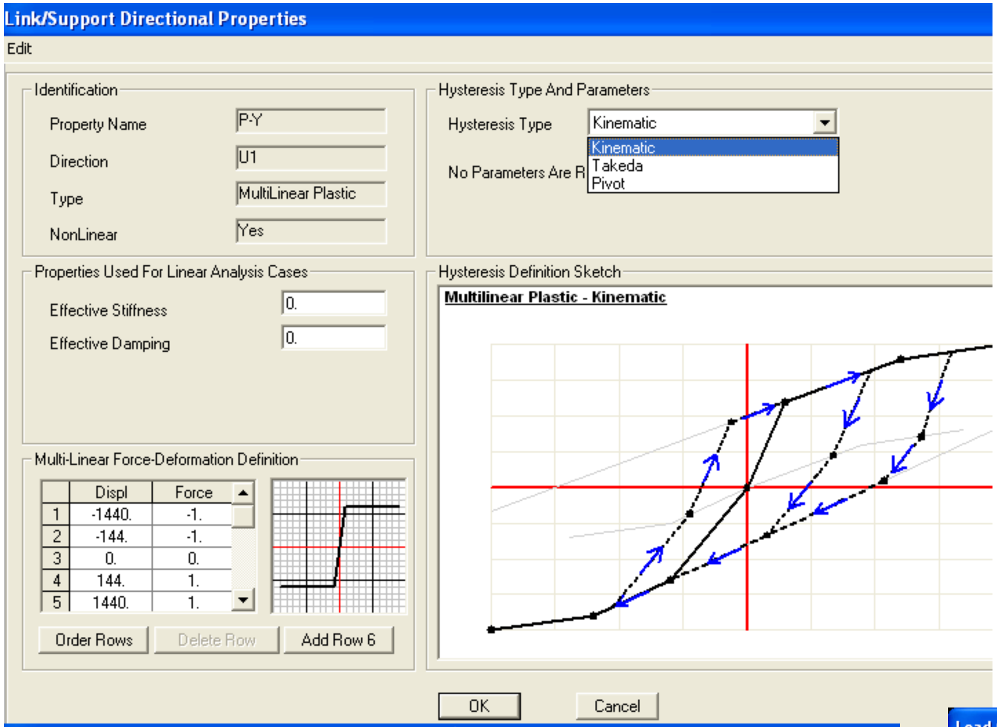
Display Plot Function Traces (TH1)

Joint 3
Displacement UX

Min is -9.883e-04
at 7.100e-01
Max is 9.494e-04
at 9.000e-02

(8.814E-01 , 9.655E-04)

Using SAP2000's multi-linear plastic link in lieu of linear springs, it's straightforward for engineers to consider nonlinear P-Y, T-Z soil data in a nonlinear TH analysis. A few changes would need to be made that are shown in the screenshots below. Ritz vectors would have to be defined and used in a NL modal TH case. Since nonlinear TH analyses do not offer the possibility of 'periodic', only a transient analysis could be performed. That's not a big deal, but the TH functions would have to be modified to increase the number of cycles. Using Ritz vectors with SAP2000's FNA nonlinear time history, run time will increase, but not by that much in many cases. I replaced 10 linear springs with plastic links in the pile/structure model, and was able to run a NL time history analysis in less than a minute.



Link/Support Directional Properties

Edit

Identification

Property Name: P-Y

Direction: U1

Type: MultiLinear Plastic

NonLinear: Yes

Hysteresis Type And Parameters

Hysteresis Type: Kinematic

No Parameters Are R

Properties Used For Linear Analysis Cases

Effective Stiffness: 0.

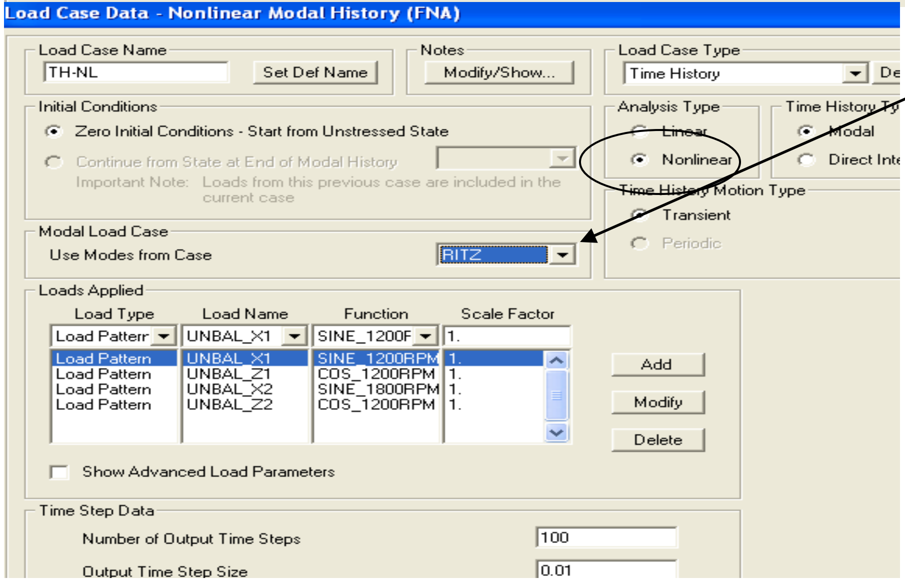
Effective Damping: 0.

Multi-Linear Force-Deformation Definition

	Displ	Force
1	-1440.	-1.
2	-144.	-1.
3	0.	0.
4	144.	1.
5	1440.	1.

Hysteresis Definition Sketch

Multilinear Plastic - Kinematic



Load Case Data - Nonlinear Modal History (FNA)

Load Case Name: TH-NL

Load Case Type: Time History

Initial Conditions

● Zero Initial Conditions - Start from Unstressed State

○ Continue from State at End of Modal History

Important Note: Loads from this previous case are included in the current case

Analysis Type

○ Linear

● Nonlinear

Time History Ty

● Modal

○ Direct Inte

Time History Motion Type

● Transient

○ Periodic

Modal Load Case

Use Modes from Case: RITZ

Loads Applied

Load Type	Load Name	Function	Scale Factor
Load Patterr	UNBAL_X1	SINE_1200F	1.
Load Pattern	UNBAL_X1	SINE_1200RPM	1.
Load Pattern	UNBAL_Z1	COS_1200RPM	1.
Load Pattern	UNBAL_X2	SINE_1800RPM	1.
Load Pattern	UNBAL_Z2	COS_1200RPM	1.

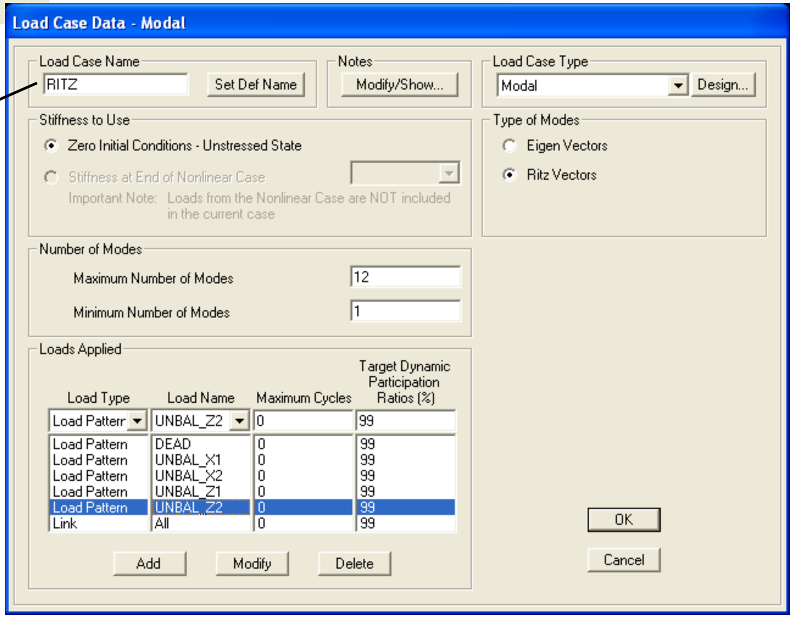
☐ Show Advanced Load Parameters

Time Step Data

Number of Output Time Steps: 100

Output Time Step Size: 0.01

Ritz modal includes all load patterns and Link "All" as shown



Load Case Data - Modal

Load Case Name: RITZ

Load Case Type: Modal

Stiffness to Use

● Zero Initial Conditions - Unstressed State

○ Stiffness at End of Nonlinear Case

Important Note: Loads from the Nonlinear Case are NOT included in the current case

Type of Modes

○ Eigen Vectors

● Ritz Vectors

Number of Modes

Maximum Number of Modes: 12

Minimum Number of Modes: 1

Loads Applied

Load Type	Load Name	Maximum Cycles	Target Dynamic Participation Ratios (%)
Load Patterr	UNBAL_Z2	0	99
Load Pattern	DEAD	0	99
Load Pattern	UNBAL_X1	0	99
Load Pattern	UNBAL_X2	0	99
Load Pattern	UNBAL_Z1	0	99
Load Pattern	UNBAL_Z2	0	99
Link	All	0	99