ENERCALC 3D
Training Manual

ENERCALC, INC

ENERCALC 3D

Build 20

A product of

ENERCALC, INC.

ENERCALC 3D Training Manual
Build 20

Publisher

ENERCALC, INC.

Managing Editor

Michael D. Brooks, S.E., P.E.

ENERCALC, Inc.

P.O. Box 2208
Newport Beach, CA 92659
(949) 645-0151
(800) 424-2252

Sales: info@enercalc.com
Support : support@enercalc.com
Web : www.enercalc.com

Build 20 ENERCALC 3D Training Manual
January 2021

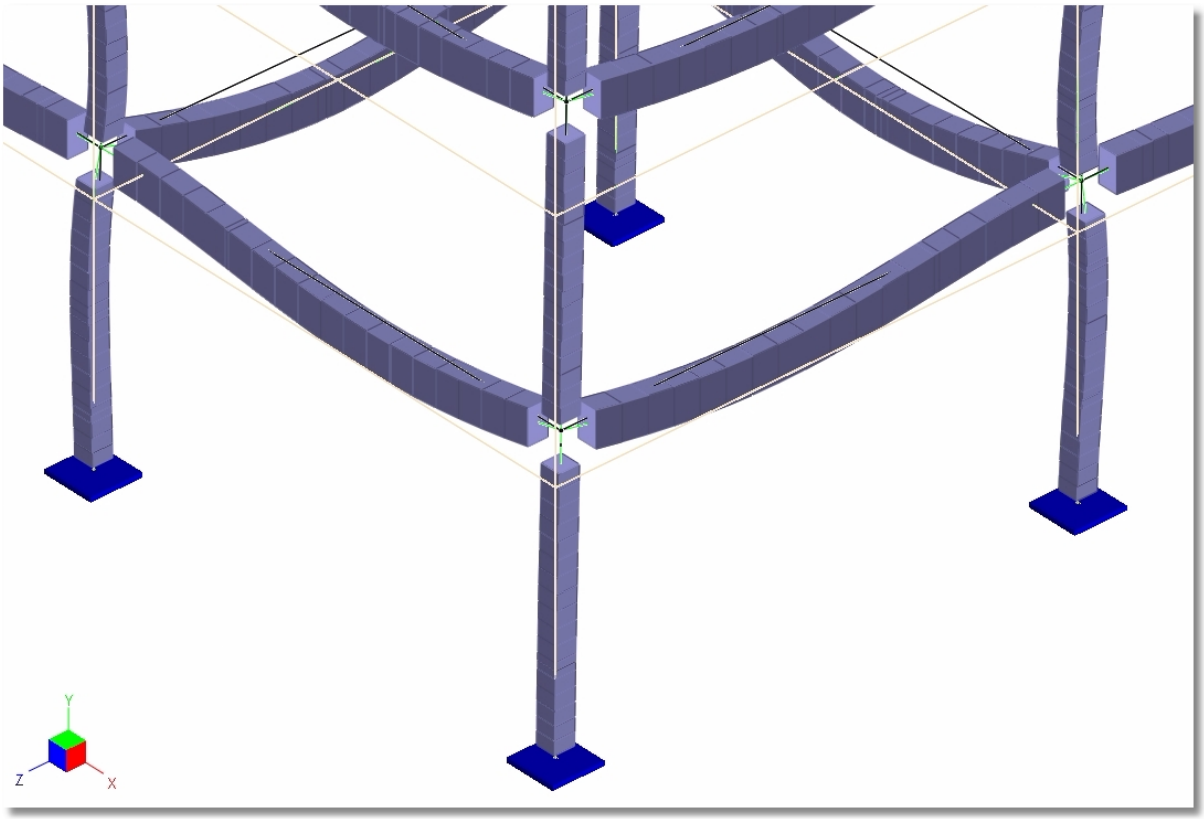
Table of Contents

Part I

1 Introduction

ENERCALC 3D Training Manual

Last Revised: 9 February 2021



1.1 Welcome to ENERCALC 3D Training

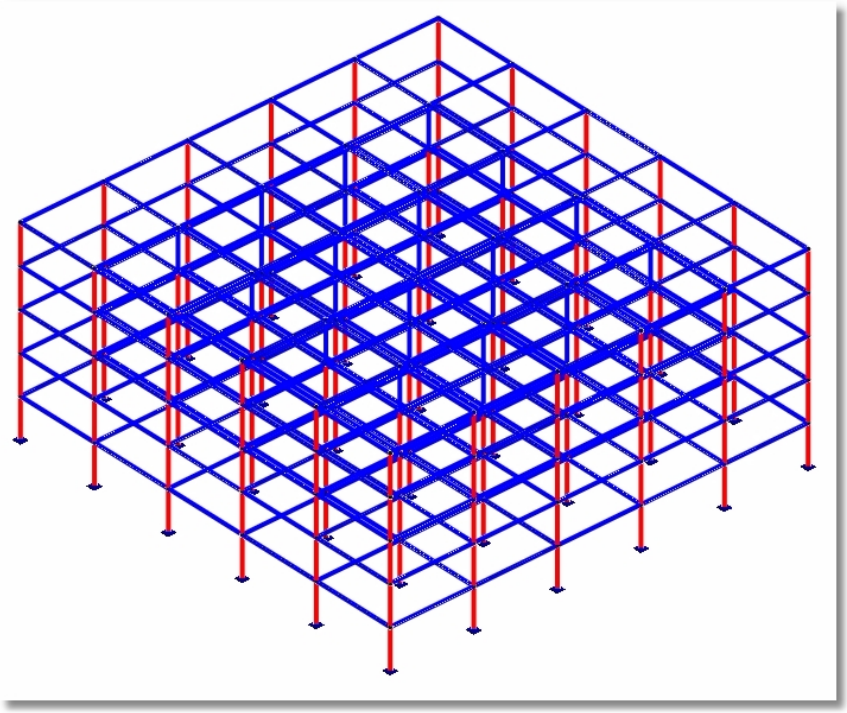
Welcome to ENERCALC 3D Training!

ENERCALC 3D is a powerful and versatile 3D structural analysis and design program with extensive capabilities. Its functionality is intuitive and easy to learn, and the purpose of this training is to make it even easier and faster to master. A quick glance down through the table of contents shows that the training will start out by introducing the graphical user interface. It will then cover some basic theory on modeling by introducing the entities that are available to model with in ENERCALC 3D.

Please refer to the ENERCALC 3D documentation for product specifications, system requirements, installation instructions, etc.

This training is intended for licensed practicing professional engineers and architects, or professionals in training. It assumes that the trainee has a working knowledge of structural analysis and design, and is generally familiar with the basic operation of Windows-based programs.

While this training will serve to familiarize you with the general operation of the software, it is not intended to replace the user's manual, nor is it intended to serve as a training in the field of structural analysis, structural design, code application, etc. All information offered is in support of ENERCALC software systems. No information shall be considered as professional consulting related to the provider’s professional registrations or affiliations.



1.2 A Tour of the Graphical User Interface

The graphical user interface can generally be broken into five major areas:

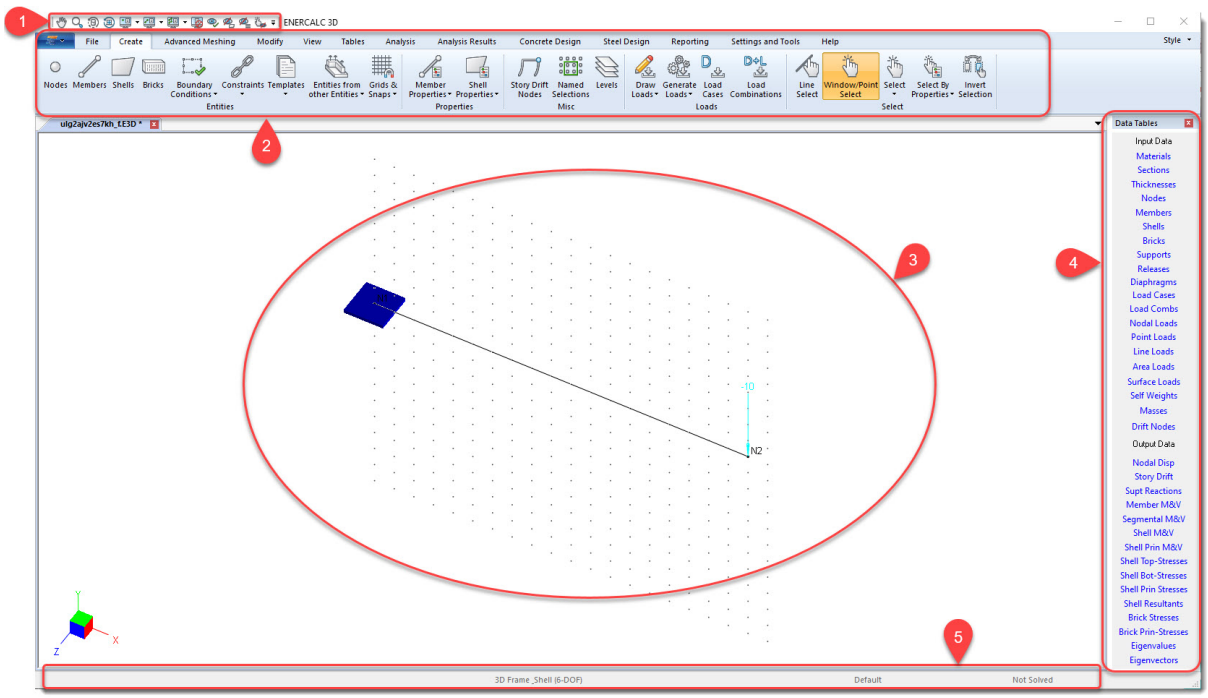
1. Quick Access Toolbar: Offers frequently used tools in plain sight with just a single click.

2. Ribbon bar: Organizes all available tools into logical categories.

3. Windows: Used to display the model and to issue graphical commands like adding, selecting, deleting and querying model geometry.

4. Data Tables Toolbar: Offers quick access to input and output data tables without navigating to the Tables tab in the ribbon.

5. Status bar: Used to provide feedback to the user on the status of the program and the model.



Parts of the graphical user interface are user configurable through commands such as:

Settings & Tools > Graphic Scales: Used to establish the scale of many items that are displayed on the screen.

Settings & Tools > Colors: Used to establish the color of the background and many items that are displayed on the screen.

Settings & Tools > Toolbar: Used to toggle the display of the Data Tables toolbar.

1.3 General Workflow

There is a logical process that should be used when working in ENERCALC 3D. The general workflow steps can be categorized as follows:

1. Construct an integral model that is stable internally as well as externally.

2. Apply properties to the model to represent proper material and section properties, connectivities or releases, non-linearities, etc.

3. Apply loads.

4. Perform analysis.

5. Evaluate analysis results like shears, moments, deflections.

6. (Optional) Perform code checking and design/optimization.

Part II

2 Model Geometry

2.1 Structural Entities

There are four different structural entities to model with in ENERCALC 3D:

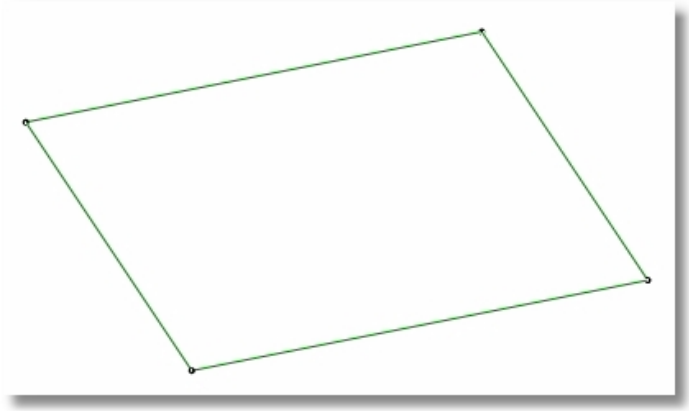
Nodes:

- Also known as "joints".
- Serve as points of connectivity and load transfer in a model.
- Think of a node as a beam to column connection, or a column base, or the point where chevron braces connect to a beam.

Members:

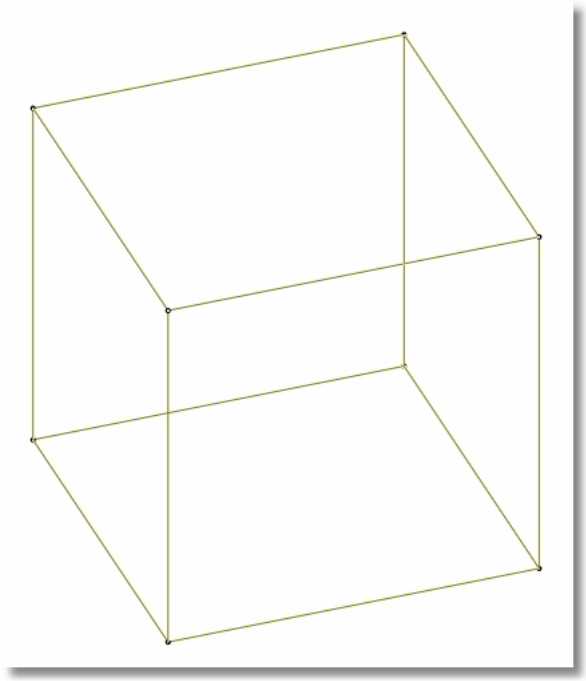
- Also known as "beams".
- One dimensional structural members that can have axial stiffness as well as biaxial flexural and shear stiffness.
- Members are useful for modeling beams, columns, braces, truss webs and chords, struts, hangers, brackets, outriggers, posts, links, etc.

Shells:



- Also known as "plates"
- Two-dimensional quadrilateral elements that have the ability to span in two directions and that have in-plane shear (racking) stiffness.
- Shells are useful for modeling masonry walls, concrete walls and floors, plywood walls, etc.

Bricks:

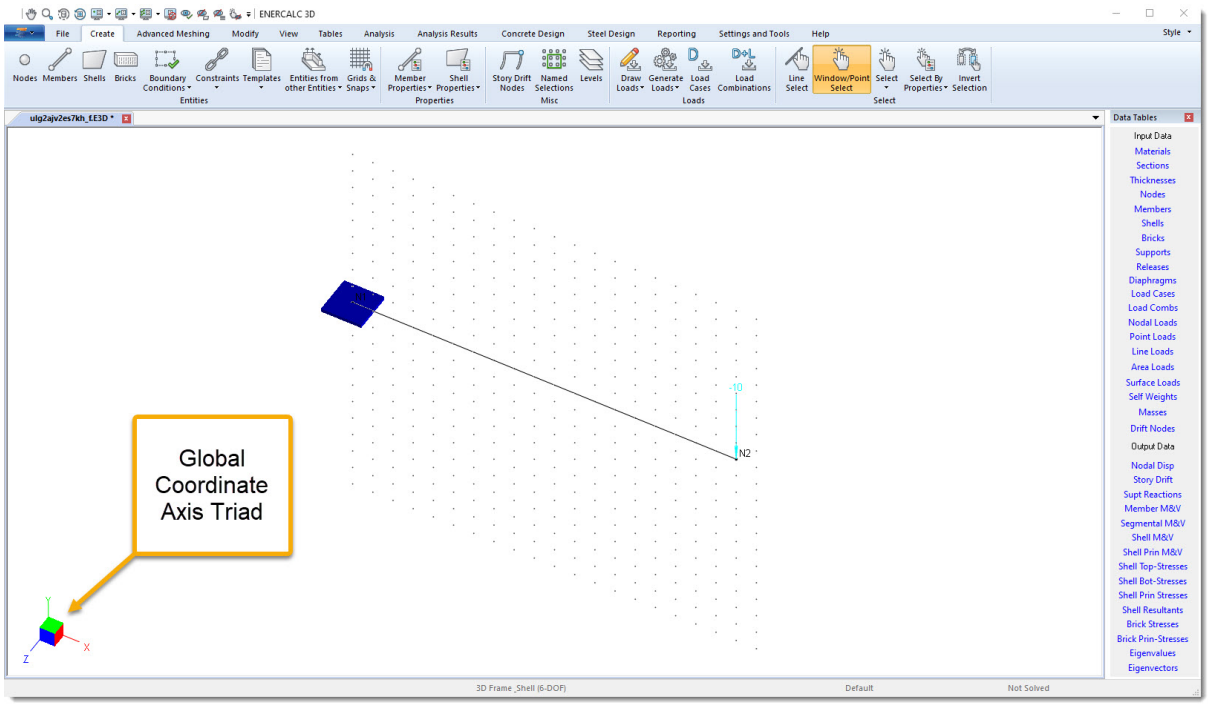


- Three dimensional solid elements defined by 8 nodes.
- Bricks are useful for modeling thick things like a massive combined footing or the face of a dam or stem of a retaining wall.
- More useful than shells in situations where the through-thickness effects have a significant role in the behavior of the element.

2.2 Global Coordinate System

The Global Coordinate System in ENERCALC 3D is a Cartesian system of X, Y and Z axes:

- Follows right-hand rule where Z cross X equals Y.
- Y axis is typically considered "up" in ENERCALC 3D ("Y to the sky").
- Axis triad is color coded to help with visualization, and coordinates with the view orientation buttons on the View toolbar:
  - X = Red
  - Y = Green
  - Z = Blue



- Axis triad shows positive directions of all three axes, and algebraic sign is important!
- Global coordinate axis will be useful for specifying model geometry, specifying loads, and interpreting results.

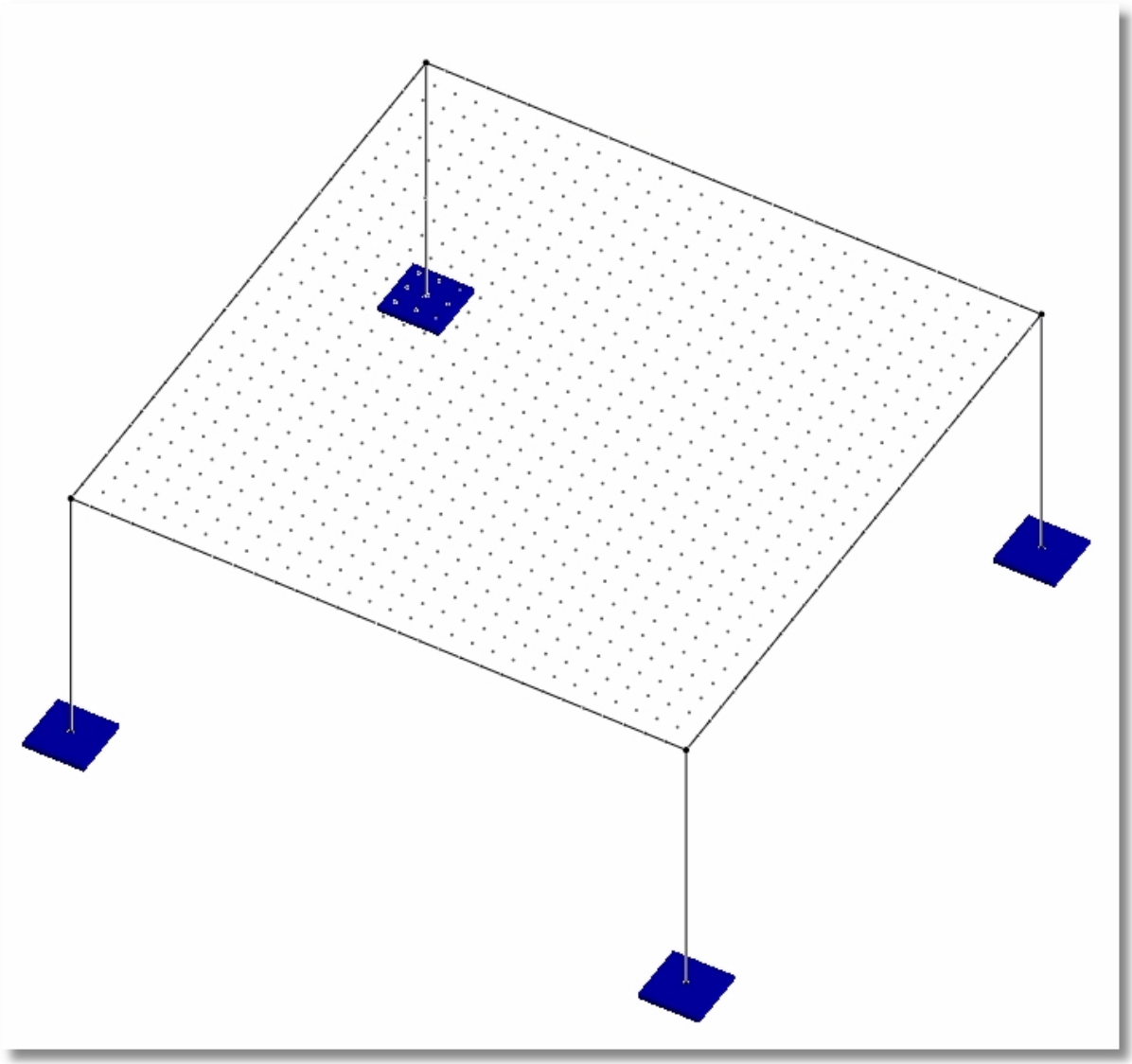
2.3 Grids

Grids are a graphical aid for model construction.

They are completely optional, and can be turned on and off by **View > Toggle Grid Display**, or by toggling the F7 key.

Grids can be useful for a few purposes:

- Provide a sense of orientation in the beginning, when very few members have been placed.
- Helpful for establishing a sense of scale in the early stages of modeling.
- Most important use is geometric control in the form of creating snap points to known coordinates when constructing model geometry.



Defining Grids:

1. Click **View > Drawing Grid Setup**.

2. Specify the desired spacings of dots in one, two or all three global axis input fields.
   - Specifying spacings in only one axis input box will create a line of dots along that axis.
   - Specifying spacings in two axes input boxes will generate a two-dimensional array of dots that lies in the plane of the two axes specified, and is probably the most commonly used.
   - Specifying spacings in three axes input boxes will generate a three-dimensional array of dots, which can sometimes be difficult to interpret if the dot spacing is too close.
   - Spacings can automatically be generated with syntax like 20@0.5.
   - Spacings can also be manually specified with comma delimited lists such as 30, 30, 28, 30, 30.

3. Specify the insertion point coordinates to move the defined grid around in the global coordinate system if necessary.

4. Specify a rotation angle and axis of rotation if necessary.
   - Can be useful for defining members or shells that lie in a sloping plane, such as a roof.

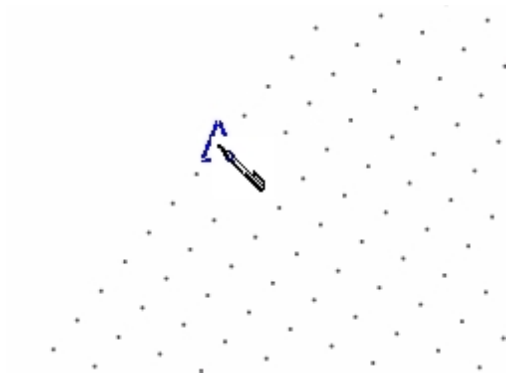
2.4 Constructing Model Geometry

2.4.1 Adding Nodes

Nodes can be added to a model in at least a few ways.

Adding Nodes Graphically:

With a grid displayed on the screen, click **Create > Nodes**. Notice that the cursor changes to a pen.



Hover over the grid and notice the triangle that tracks the pen movements and automatically snaps to grids. Note that the Status Bar displays the X, Y and Z coordinates of every location where the cursor snaps to the grid. This is useful for orientation.

To add a node, just click anywhere on the grid and then move the cursor to see that a dark, bold dot remains at the click location.

If node numbers are not currently displayed, they can be toggled on by clicking **View > Display Options > Node Display Options > Node Number**. Notice that there is also an icon on the Quick Access Toolbar that makes this process possible without leaving the Create tab: 

Click in a few more locations to add some additional nodes.

Click **View > Query**, and then click on any node to display the *Nodal Info* dialog. At this time, the dialog will list the coordinates of the selected node, but it won't have much additional info, because no loads have been assigned and no analysis results are available. Keep this Query function function in mind, because it can be an extremely useful way to get info about a model entity at any time.

Close the *Nodal Info* dialog if it is still open.

Adding Nodes with the Node Data Table:

Click **Tables > Nodes** to open the *Nodes* table.

Notice that the table lists all nodes that have been defined, and it displays their coordinates.

To add another node in the current model, use the New Rows button in the lower left corner of the *Nodes* table to add one new row. Then enter the desired coordinates for the new node, and then click OK.

Notice that the *Nodes* table does not constrain nodes to the grid increments, so it can be useful for entering nodes with coordinates that aren't nice and even. The table can also be a convenient way to move an existing node by editing its coordinates, and we will demonstrate that in the section on Editing Model Geometry.

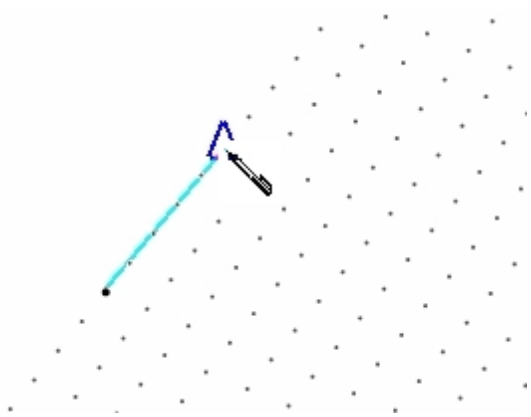
Note: The methods of adding nodes that we have discussed here can be thought of as the "explicit" methods. There are also methods of creating new nodes that can be thought of as the "implicit" methods, such as generating nodes in a pattern or automatically creating nodes at member intersections or splitting members or duplicating members. But we will cover all of those topics shortly.

2.4.2 Adding Members

As was the case with Nodes, Members can be added to a model in at least a few ways.

Adding Members Graphically:

With a grid displayed on the screen, click **Create > Members**. Notice that the cursor changes to a pen.



Hover over the grid and notice the triangle that tracks the pen movements and automatically snaps to grids. Note that the Status Bar displays the X, Y and Z coordinates of every location where the cursor snaps to the grid. This is useful for orientation.

- To add a member, just click anywhere on the grid and then move the cursor to see that a dark, bold dot remains at the click location and that a blue line stretches from the first click location to the cursor location.

- That dark, bold dot is the starting node of the member.
- Click a second time on a different grid location to specify the end of the beam, and you will have created the first member.
- Notice that the blue line continues to stretch from the last click, so the program can quickly add many beams connected with common nodes.
- To stop drawing, right-click the mouse.

If member numbers are not currently displayed, they can be toggled on by clicking **View > Display Options > Member Display Options > Member Number**. Notice that there is also an icon on the Quick Access Toolbar that makes this process possible without leaving the Create tab: 

Click **View > Query**, and then click on any Member to display the *Member Info* dialog. At this time, the dialog will list the starting and ending nodes of the selected member, along with some basic info about the member's properties. But it won't have much additional info, because no loads have been assigned and no analysis results are available. Keep this Query function function in mind, because it can be an extremely useful way to get info about a model entity at any time.

Close the *Member Info* dialog if it is still open.

Adding Members with the Member Data Table:

Make sure that the grid has at least two nodes displayed.

Click **Tables > Members** to open the *Members* table.

Notice that the table lists all members that have been defined (if any), and it displays their starting and ending nodes, along with some properties for the selected member.

To add another member in the current model, use the New Rows button in the lower left corner of the *Members* table to add one new row. Then enter the desired start and end nodes for the new member, and then click OK.

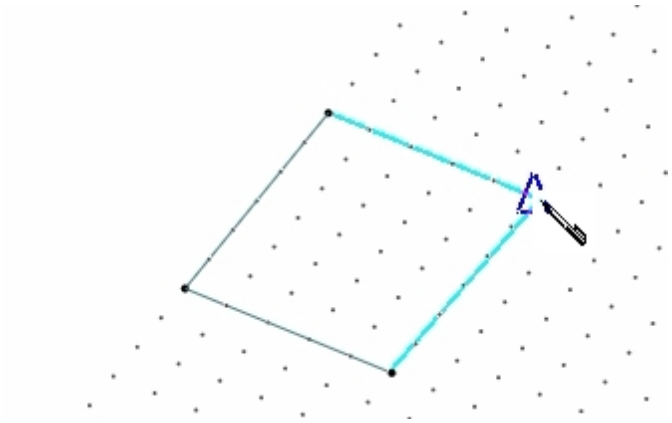
Note: The methods of adding members that we have discussed here can be thought of as the "explicit" methods. There are also methods of creating new members that can be thought of as the "implicit" methods, such as generating members in a pattern or splitting members or duplicating members. But we will cover all of those topics shortly.

2.4.3 Adding Shells

As was the case with Nodes and Members, Shells can be added to a model in at least a few ways.

Adding Shells Graphically:

With a grid displayed on the screen, click **Create > Shells**. Notice that the cursor changes to a pen.



Hover over the grid and notice the triangle that tracks the pen movements and automatically snaps to grids. Note that the Status Bar displays the X, Y and Z coordinates of every location where the cursor snaps to the grid. This is useful for orientation.

- To add a shell, just click anywhere on the grid and then move the cursor to see that a dark, bold dot remains at the click location and that a blue line stretches from the first click location to the cursor location.
- That dark, bold dot is the first node of the shell.
- Move in a counter-clockwise direction and click a second time on a different grid location to specify the second node of the shell.
- Continue in a counter-clockwise direction and click a third time, and finally click a fourth time, and you will have created the first shell.

Important: Always move in either a clockwise or a counter-clockwise direction when specifying the four nodes of a shell. Moving in a crisscross pattern will generate a warped shell, which looks like two triangles. It will not function properly!

If shell numbers are not currently displayed, they can be toggled on by clicking **View > Display Options > Shell Display Options > Shell Number**. Notice that there is also an icon on the Quick Access Toolbar that makes this process possible without leaving the Create tab: 

Click **View > Query**, and then click on the shell to display the *Shell Info* dialog. At this time, the dialog will list the nodes of the shell, along with some basic info about the shell's properties. But it won't have much additional info, because no loads have been assigned and no analysis results are available. Keep this Query function function in mind, because it can be an extremely useful way to get info about a model entity at any time.

Close the *Shell Info* dialog if it is still open.

Adding Shells with the Shell Data Table:

Make sure that the grid has at least four nodes displayed.

Click **Tables > Shells** to open the *Shells* table.

Notice that the table lists all shells that have been defined (if any), and it displays their nodes, along with some properties for the selected shell.

To add another shell in the current model, use the New Rows button in the lower left corner of the *Shells* table to add one new row. Then enter the desired nodes for the new shell, and then click OK.

Note: The methods of adding shells that we have discussed here can be thought of as the "explicit" methods. There are also methods of creating new shells that can be thought of as the "implicit" methods, such as generating shells in a pattern or meshing or duplicating shells. But we will cover all of those topics shortly.

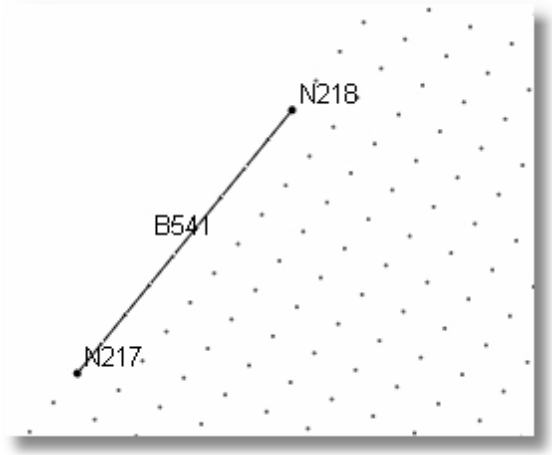
2.5 Understanding Model Geometry

2.5.1 Members

2.5.1.1 Starting and Ending Nodes

As we saw when we learned how to add a member, members are always defined by two nodes: a starting node and an ending node. The beam can be thought of as spanning from the starting node to the ending node.

The diagram below shows that beam 541 has nodes numbered 217 and 218:



The data in the table below indicate that beam 541 starts at node 217 and ends at node 218.

Member Data

	Member Id	Node-1	Node-2	Material	Section	Local Angle (deg)	Nonlinear	Status
465	525	195	201	2: Concrete40	2: Rect14x14	0	Linear	Frozen
466	526	196	202	2: Concrete40	2: Rect14x14	0	Linear	Frozen
467	527	197	203	2: Concrete40	2: Rect14x14	0	Linear	Frozen
468	528	198	204	2: Concrete40	2: Rect14x14	0	Linear	Frozen
469	529	199	205	2: Concrete40	2: Rect14x14	0	Linear	Frozen
470	530	200	206	2: Concrete40	2: Rect14x14	0	Linear	Frozen
471	531	201	207	2: Concrete40	2: Rect14x14	0	Linear	Frozen
472	532	202	208	2: Concrete40	2: Rect14x14	0	Linear	Frozen
473	533	203	209	2: Concrete40	2: Rect14x14	0	Linear	Frozen
474	534	204	210	2: Concrete40	2: Rect14x14	0	Linear	Frozen
475	535	205	211	2: Concrete40	2: Rect14x14	0	Linear	Frozen
476	536	206	212	2: Concrete40	2: Rect14x14	0	Linear	Frozen
477	537	207	213	2: Concrete40	2: Rect14x14	0	Linear	Frozen
478	538	208	214	2: Concrete40	2: Rect14x14	0	Linear	Frozen
479	539	209	215	2: Concrete40	2: Rect14x14	0	Linear	Frozen
480	540	210	216	2: Concrete40	2: Rect14x14	0	Linear	Frozen
481	541	217	218	2: Concrete40	3: Rect14x24	0	Linear	Normal

1 New Rows Print... OK Cancel

This establishes a directionality that can be very useful when modeling and when interpreting results. It also forms the basis for the Member Local Axis system covered in the next section.

Note: The direction and orientation of a member can be revised by some commands that we will cover in the section on Editing Model Geometry.

2.5.1.2 Member Local Axes

All members have a default local axis system. It is useful for a variety of things like defining member end releases, defining member end offsets, applying loads, defining member local angle, and interpreting results.

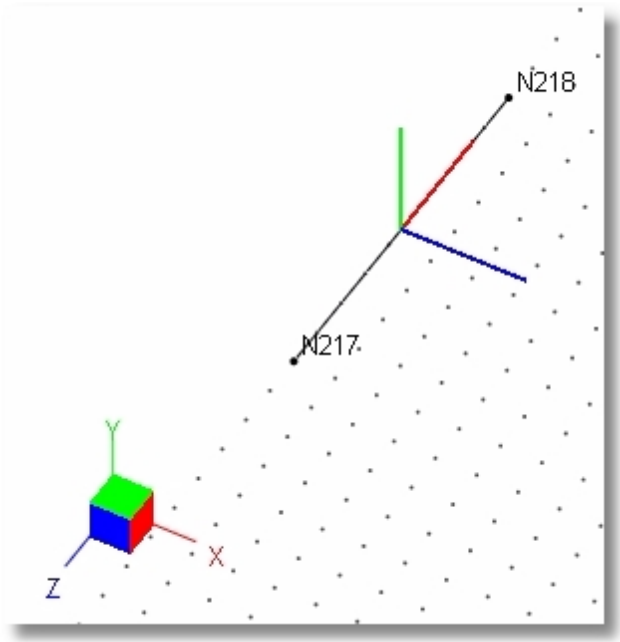
The default member local axis system is defined as follows:

1. The member local x (red) axis is defined as a vector pointing from the starting node to the ending node.

2. The default member local z (blue) axis is defined by the vector cross product of local x cross global Y. Think of this by pointing your right fingers in the direction of the local x axis, and then curl your fingers to envision swinging that local x axis into the global Y axis. The direction of your right thumb indicates the direction of the vector cross product, and therefore indicates the direction of the local z axis.

Note: The one condition where this rule cannot be applied is with vertical members, because it is not possible to calculate the vector cross product of two parallel vectors. So in that case, ENERCALC 3D adopts the convention that the local z axis will be oriented parallel to the global Z axis.

3. The default member local y (green) axis is defined by the rule that says in a Cartesian coordinate system, z cross x equals y. Think of this by pointing your right fingers in the direction of the local z axis, and then curl your fingers to envision swinging that local z axis into the local x axis. The direction of your right thumb indicates the direction of the vector cross product, and therefore indicates the direction of the local y axis.



A subsequent topic in the Editing Model Geometry section will discuss ways that the member local axis system can be revised to something other than this default orientation.

2.5.2 Shells

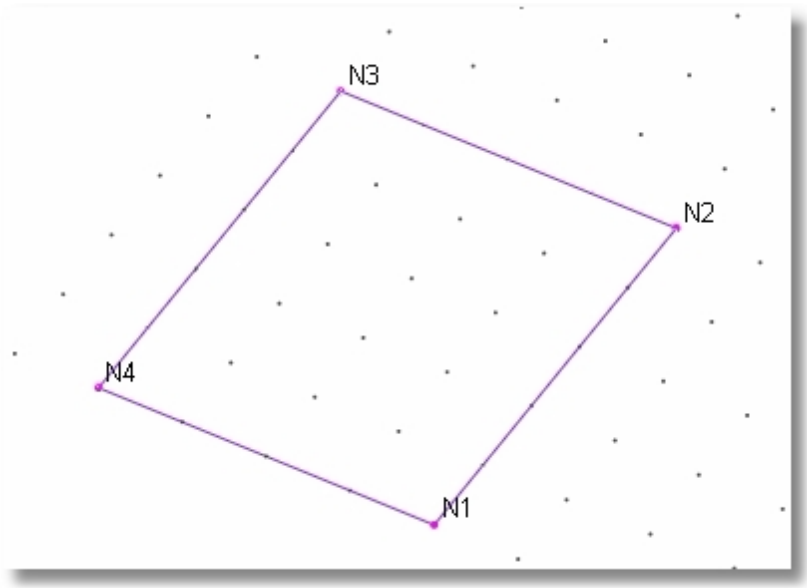
2.5.2.1 Order of Nodes

The order of the nodes defining a shell is important for a few reasons relating to numerical behavior, load application and interpretation of results.

To satisfy numerical behavior, it is important to define quad shells by either numbering their nodes sequentially clockwise or counter-clockwise. Specifying a quad shell by using a crisscross numbering order will result in a "warped" shell. These look like a bowtie, and they do not perform properly.

Shells have a top and a bottom which have meaning in terms of the output. To determine the orientation of a shell, use the fingers of your right hand to trace around the nodes of the shell in order from Node 1 to node 4. Your right thumb will naturally point in the direction of the "top"

of the plate. The shell below was drawn from node 1 to 2 to 3 to 4, so we are looking at its top.



Do not be too concerned with the orientation of shells during the modeling process, because there are some simple ways to orient groups of plates consistently once they have been modeled. These will be discussed in the section on Editing Model Geometry.

2.5.2.2 Shell Local Axes

As was the case with members, shells also have a local axis system that is important when applying loads and interpreting results.

Envision a quad shell where Node 1 = A, Node 2 = B, Node 3 = C and Node 4 = D.

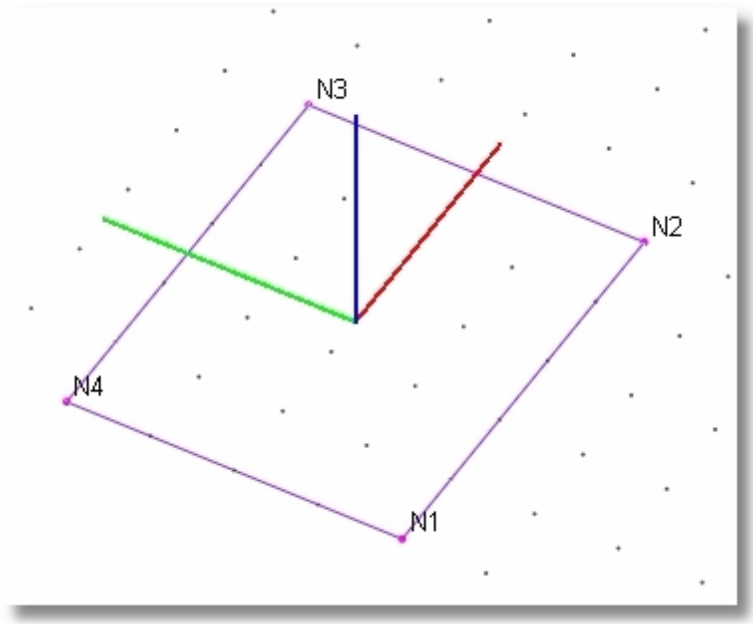
The default shell local axis system is displayed at the geometric center of a shell, and the axis orientation is defined as follows:

1. The default shell local z (blue) axis is perpendicular to the shell. It is defined by the cross product of vectors AB and AC. Think of this by pointing your right fingers in the direction of AB, and then curl your fingers to envision swinging AB into AC. The direction of your right thumb indicates the direction of the vector cross product, and therefore indicates the direction of the local z axis.

2. The shell local x (red) axis is defined as follows:
   - For horizontal shells that are parallel to global XZ plane, local x is parallel to global X.
   - For non-horizontal shells, local x is perpendicular to a plane formed by global Y and local z.

3. The default member local y (green) axis is defined by the rule that says in a Cartesian coordinate system, z cross x equals y. Think of this by pointing your right fingers in the direction of the local z axis, and then curl your fingers to envision swinging that local z axis

into the local x axis. The direction of your right thumb indicates the direction of the vector cross product, and therefore indicates the direction of the local y axis.

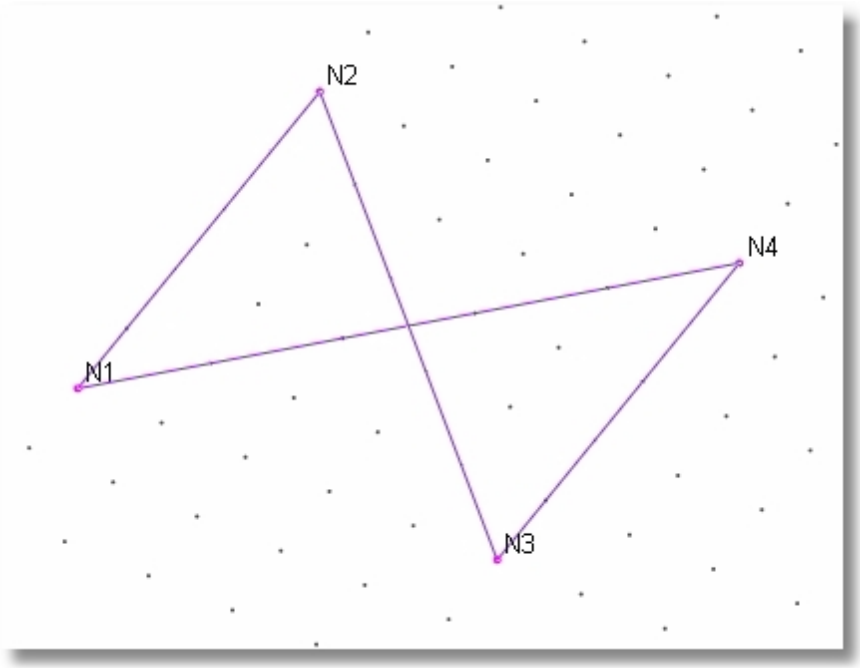


Note: local x and local y always fall in the plane of the shell, and local z is always perpendicular to the plane of the shell. The positive local z axis occurs on the "top" side of the shell.

2.5.2.3 Shell Quality

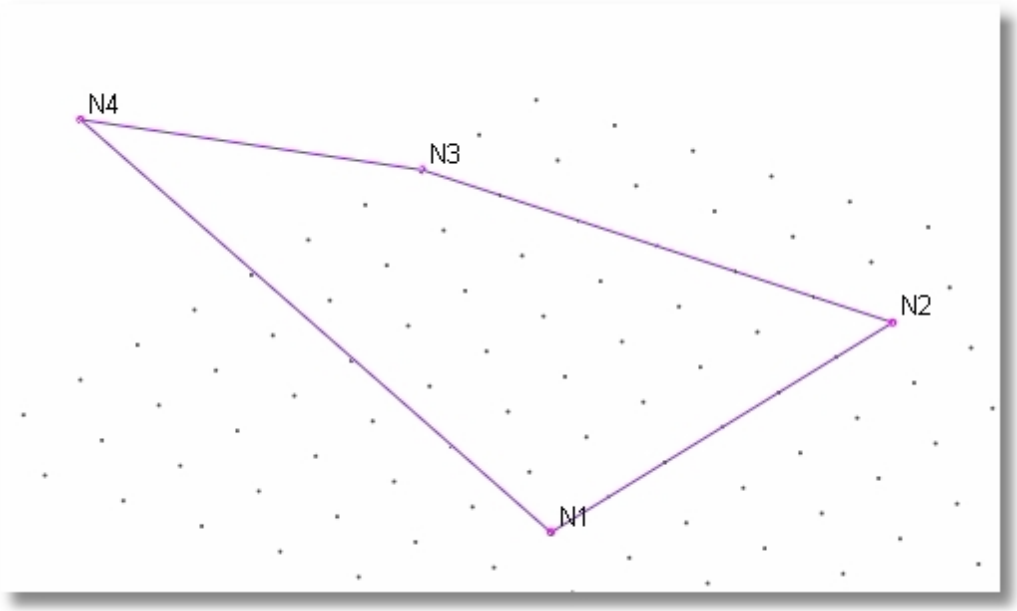
When performing a finite element analysis with quad shells, there are some geometric guidelines to keep in mind for the individual shells in order to ensure quality results.

Order of Node Selection: We have already seen that it is important to define shells in either a clockwise or counter-clockwise order to avoid warped plates which will produce invalid results.



NOT GOOD!

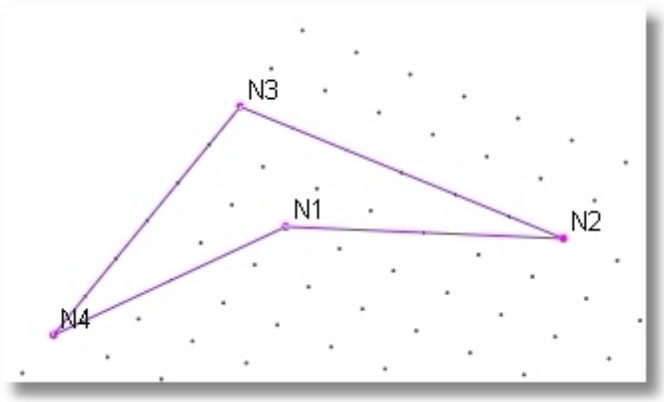
Planar Geometry: Along similar lines, it is important to ensure that all four nodes of a quad shell lie in one plane.



NOT GOOD!

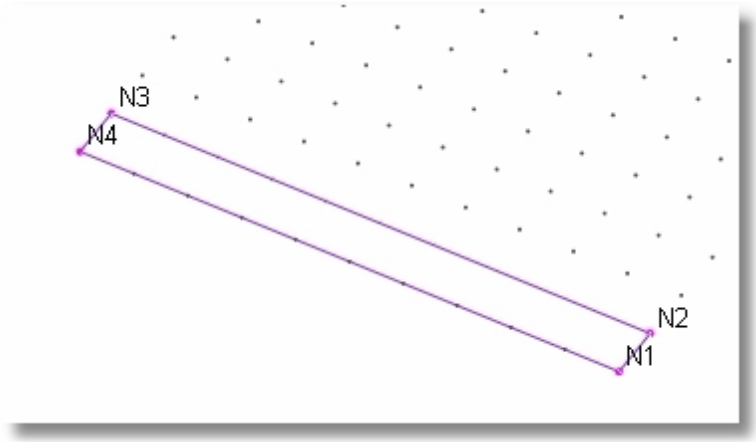
Shape: The ideal shape for a quad shell is a square. The more a quad shell deviates from a square, the greater the chances of introducing some error into the results. But in practical situations, it is not always possible to keep shells square, so here are some additional considerations to keep in mind with regard to shell quality:

Angles: Shells have some ability to tolerate angles other than 90 degrees, but try to avoid extreme skews. Interior angles should range between 60 and 120 degrees. Never create a shell that has an included angle that exceeds 180 degrees.



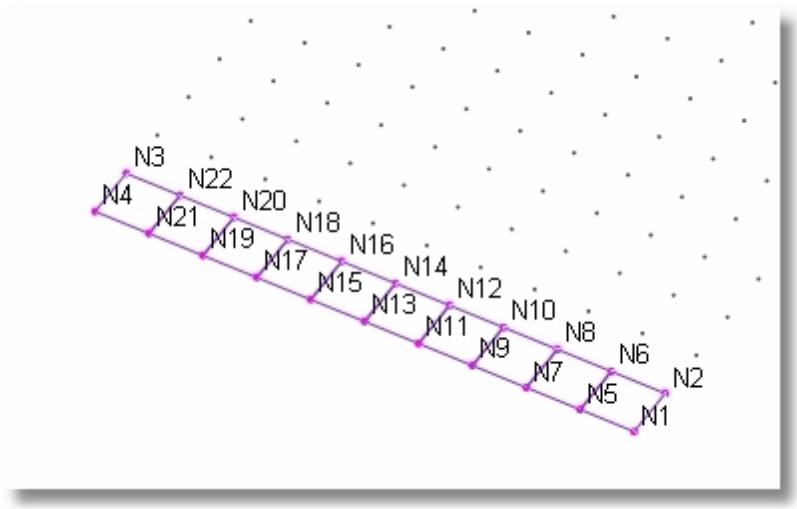
NOT GOOD!

Aspect Ratio: Shells have some ability to become rectangular, but try to avoid extreme aspect ratios. It is good to try to keep aspect ratios less than 2:1. Never create a quad shell with an aspect ratio that exceeds 4:1.



NOT GOOD!

Note that when it comes to improving shell quality, the solution is often to just break a poorly shaped shell into more shells that all have better quality.



MUCH BETTER!

This is a good segue into the next topic, which is mesh density.

Size Variation: We will cover the topic of mesh density in great detail in the section titled Mesh Density. But for the purposes of the discussion on shell quality, let it suffice to say that it is good practice to try to make the elements of a given mesh generally about the same size.

2.5.2.4 Mesh Density

Determining a suitable mesh density is more of an art than a science. There are no hard and fast rules to apply here, only general guidelines. The important thing to understand is that deflections are only reported at node locations, and shell stresses are only accurately reported at shell centers. So it is important to have nodes and shell centers relatively close to the locations of interest.

Mesh density becomes a tradeoff between having too few results locations and having too many results locations.

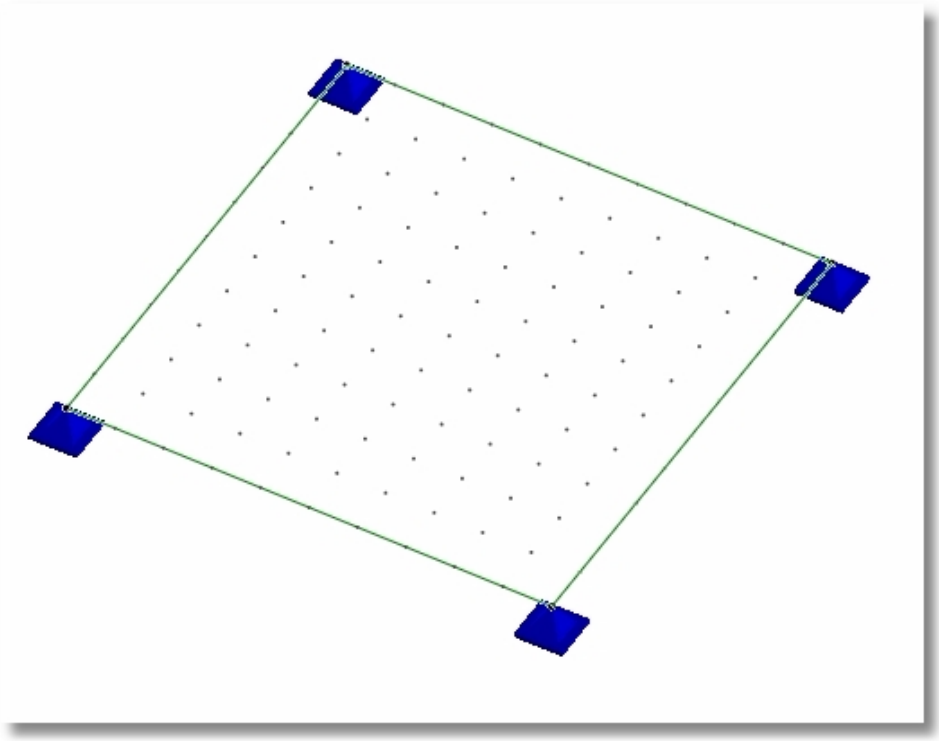
It helps to understand what happens at the extremes; that is when meshes are way too coarse and when meshes are way finer than they need to be.

Extremely coarse meshes will run more quickly than fine meshes, because of the significant reduction in the number of required calculations. But this can come at a cost of accuracy. For example, if a beam is known to have reverse curvature, we would never be able to sketch a reasonable guess for the deflected shape if we only had data for the endpoints and for one or two interior points. By sketching an assumed deflected shape and observing maxima, minima and points of inflection, you can start to establish a bare minimum number of data points that would be desirable for sketching a deflected shape. It stands to reason that you probably wouldn't want to create a mesh that is any more coarse than this, or it might not be useful in terms of quantifying the extremes.

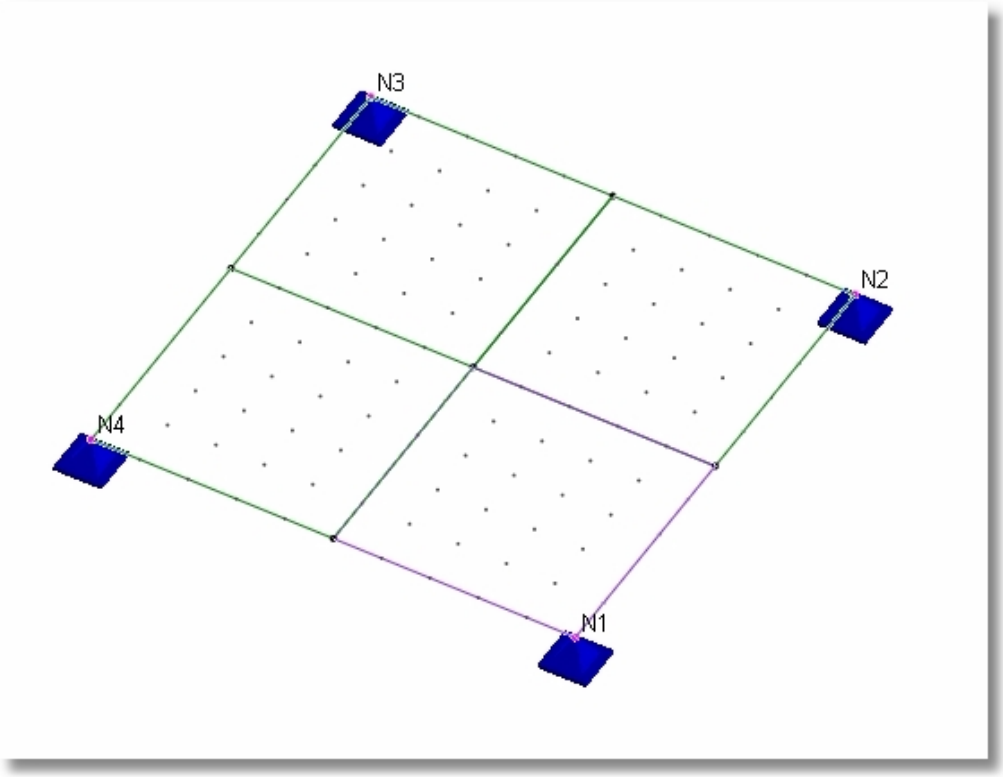
On the other hand, extremely fine meshes will result in long run times and will produce reams and reams of data that may be difficult to sort through. There is also a risk of inaccuracies when meshes are excessively fine.

So the key is to balance the need for accurate results with the time it takes to process and the complexity of the output. Fortunately modern computers are extremely powerful and current finite element formulations are very efficient, so run times for average models may not be a big problem. And thanks for the sorting and postprocessing features in ENERCALC 3D, the process of reviewing reams of data is often reduced to a trivial task through efficient reporting and onscreen display tools. So when determining an adequate mesh density, it is generally advisable to err to the side of making it finer than it needs to be, rather than using a mesh that is too coarse.

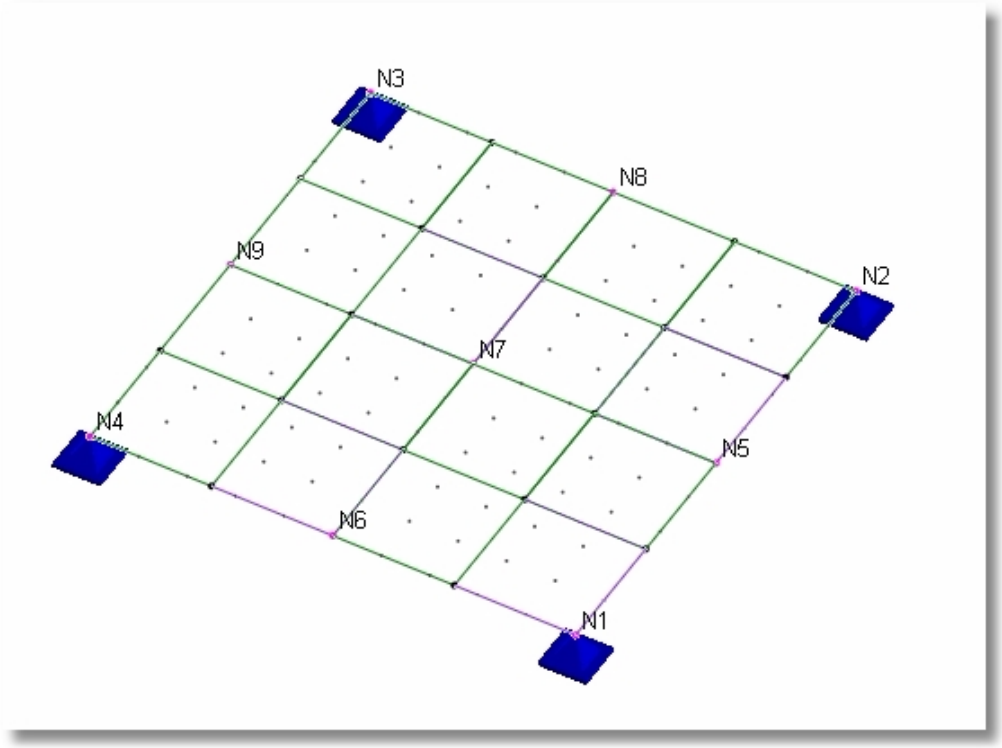
For those who are not yet comfortable with establishing a suitable mesh density, here is a suggested procedure for helping to calibrate your judgment. Create a mesh that is obviously too coarse, and apply realistic loads so that analysis results can be obtained. Make note of some result of interest, such as the deflection or moment at a particular location. Modify the mesh density by making it more dense, and observe the effect in the result of interest. Then keep repeating the process until a break point is reached where further subdivision of the mesh no longer produces a change in the value of interest. This will ultimately lead to a mesh density that is optimized for the current conditions.



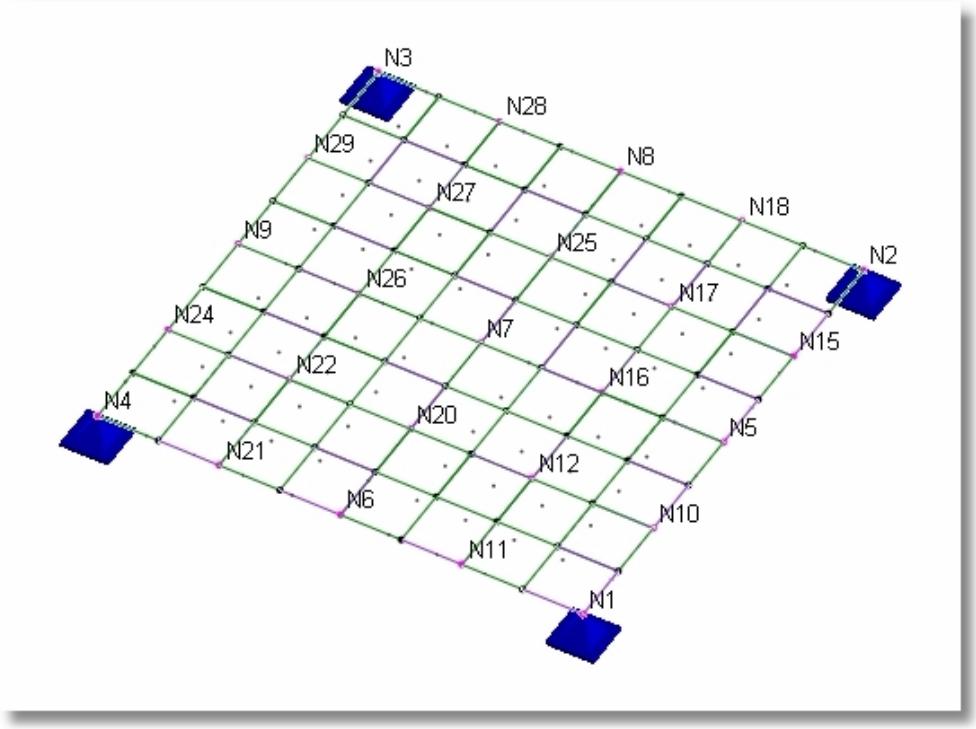
Mesh too coarse.



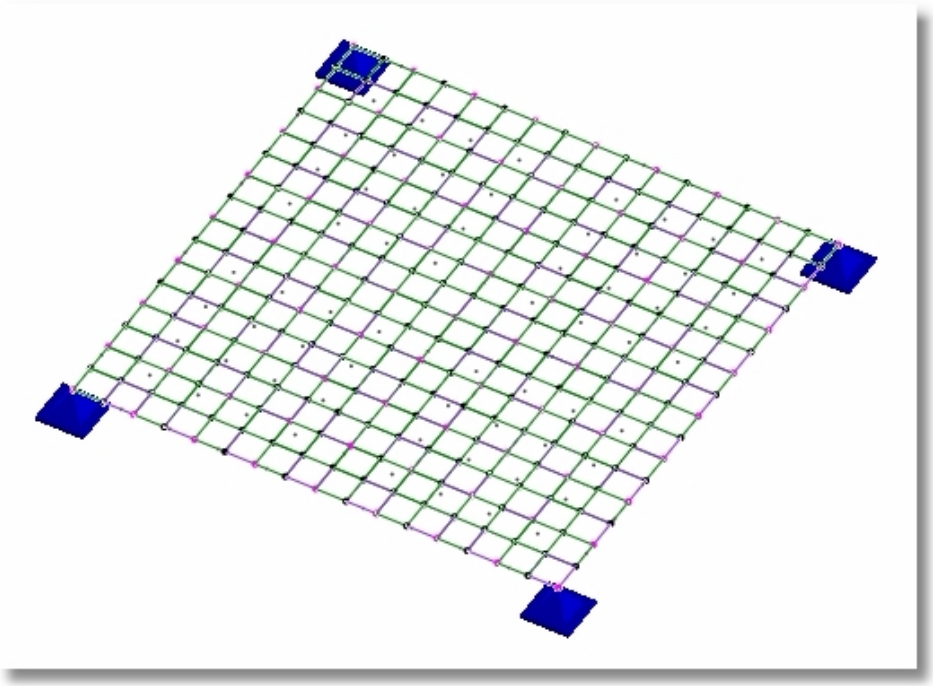
Mesh still too coarse.



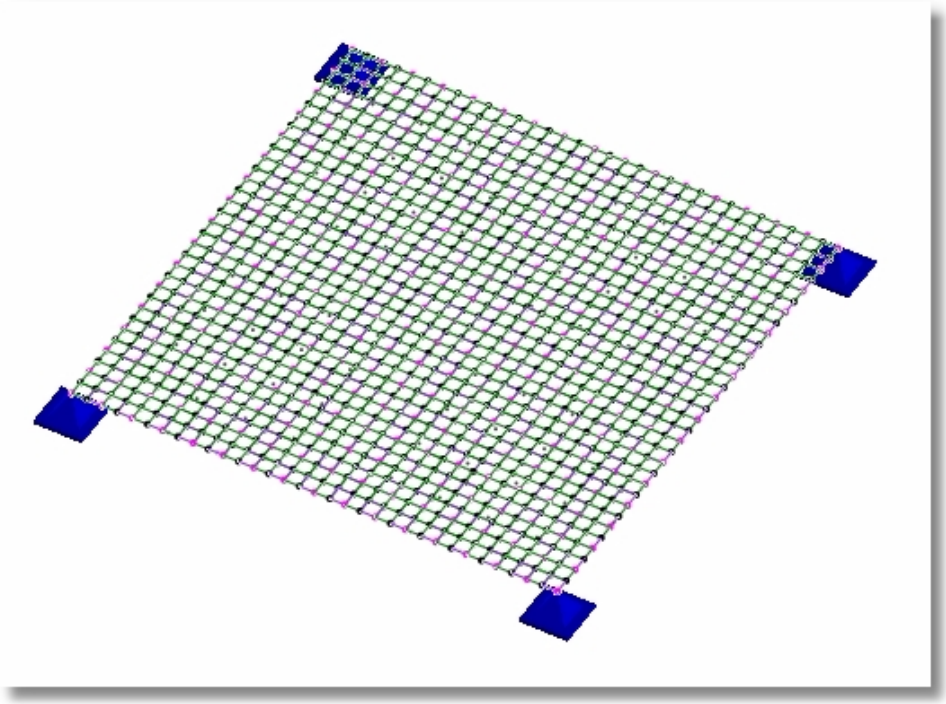
Mesh getting better.



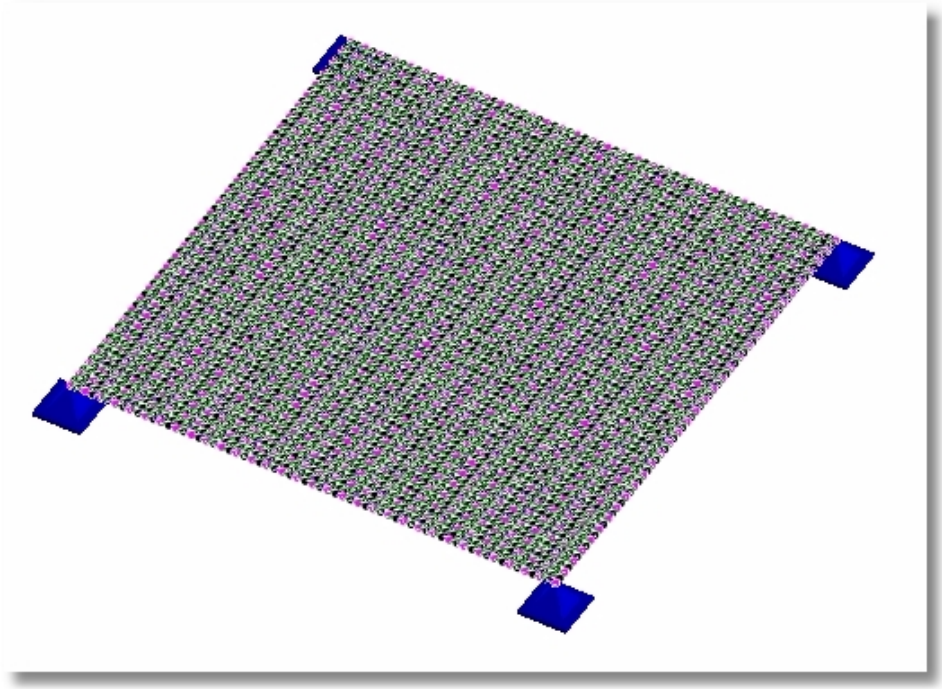
Mesh getting much better.



Could be ideal.



Probably finer mesh than necessary.



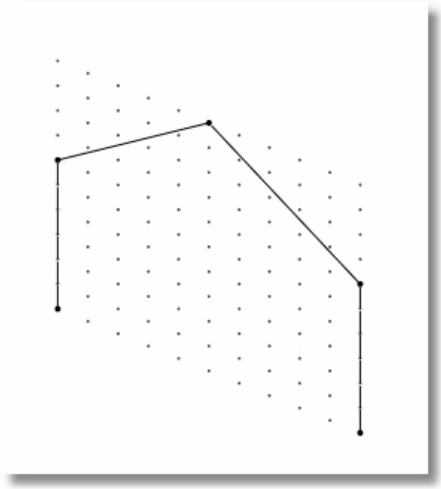
Mesh much too fine.

2.6 Editing Model Geometry

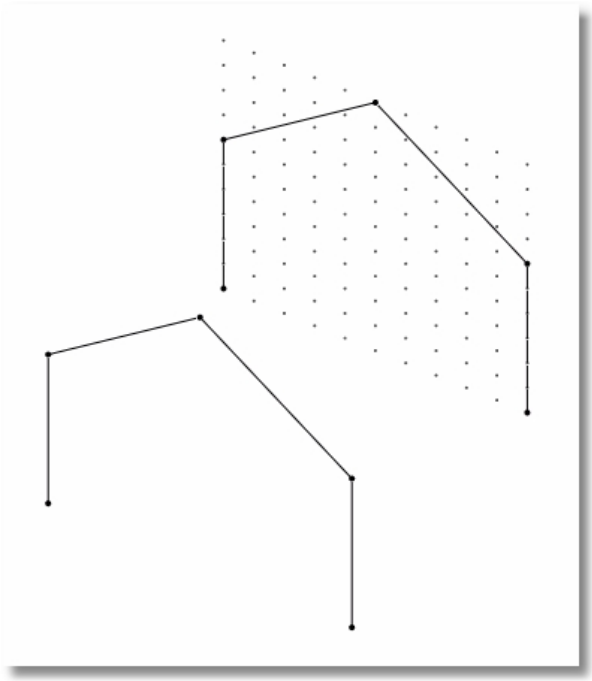
2.6.1 Duplicate

Modify > Copy creates a copy of the selected geometry and moves the copy by the specified X, Y, and Z delta values.

It has options that allow attached loads to be copied or not, and that allow nodes and elements to be merged. When the option is selected to Merge nodes and elements, if a copied node, member, or shell falls exactly on top of another node, member, or shell, the program will automatically eliminate the duplicate to avoid problems with model geometry.



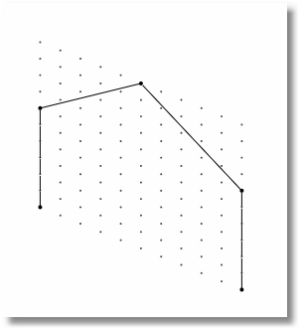
leads to



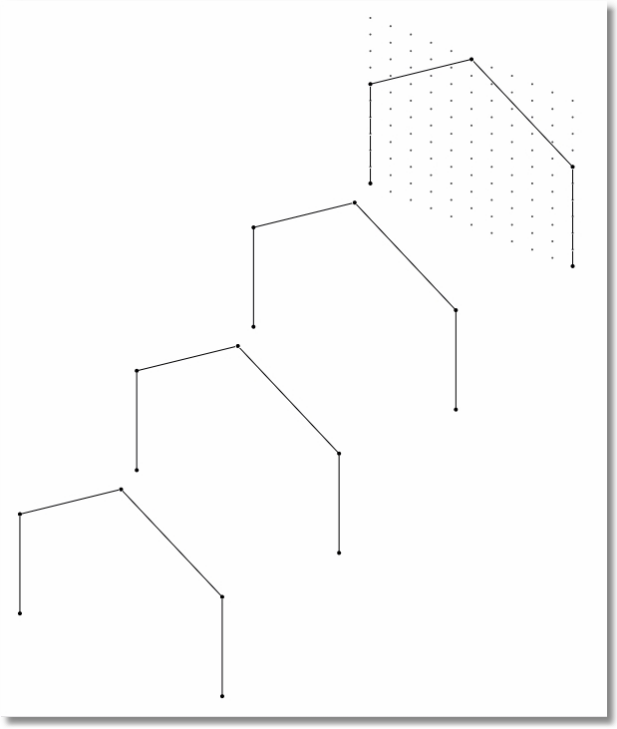
2.6.2 Array

Modify > Array can be thought of as an extension of the Copy command. It allows multiple copies of the selected geometry to be created, and each copy is separated by the specified X, Y, and Z delta step values.

It has options that allow assigned loads to be copied or not, and that allow nodes and elements to be merged. When the option is selected to Merge nodes and elements, if a copied node, member, or shell falls exactly on top of another node, member, or shell, the program will automatically eliminate the duplicate to avoid problems with model geometry.



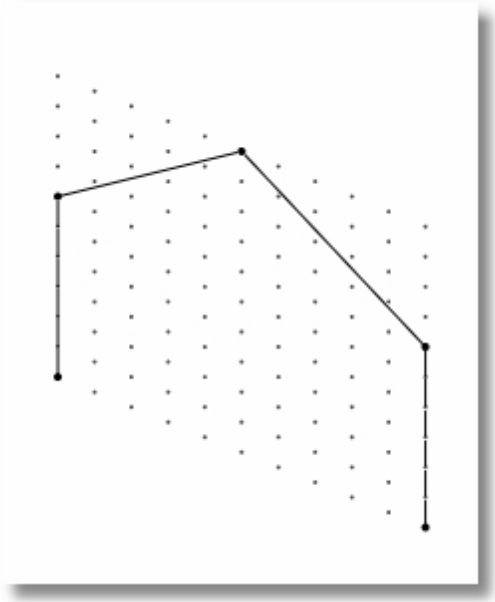
leads to



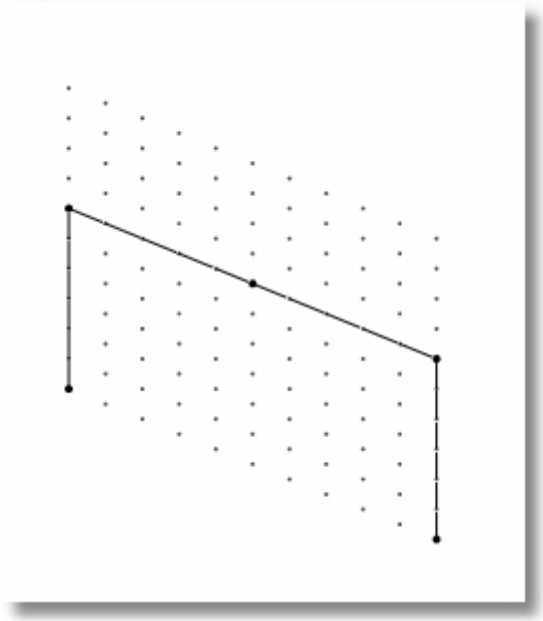
2.6.3 Move

Modify > Move acts on selected nodes and allows them and their connected members, shells and bricks to be moved by the specified X, Y, and Z distances.

It has an option to allow nodes and elements to be merged. When the option is selected to Merge nodes and elements, if a copied node, member, or shell falls exactly on top of another node, member, or shell, the program will automatically eliminate the duplicate to avoid problems with model geometry.



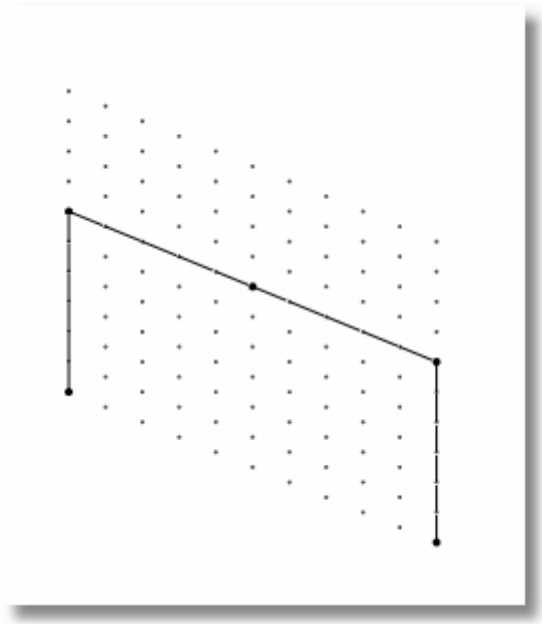
leads to



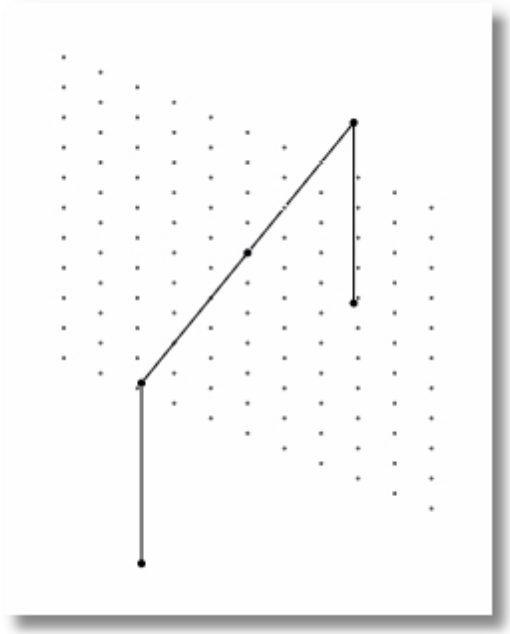
2.6.4 Rotate

Modify > Rotate acts on selected nodes and allows them and their connected members, shells and bricks to be rotated by the specified angle about an axis located by user-specified coordinates.

It has an option to allow nodes and elements to be merged. When the option is selected to Merge nodes and elements, if a copied node, member, or shell falls exactly on top of another node, member, or shell, the program will automatically eliminate the duplicate to avoid problems with model geometry.

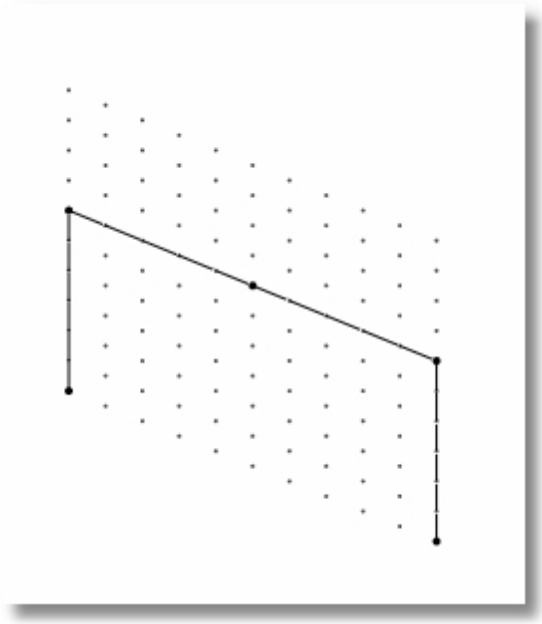


leads to

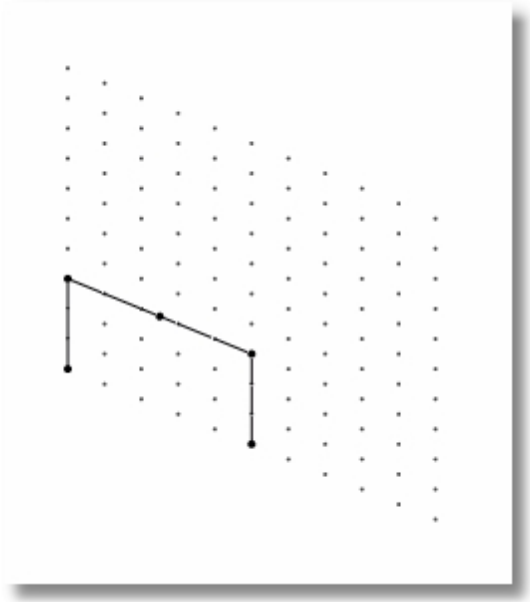


2.6.5 Scale

Modify > Scale acts on selected nodes and allows their connected members, shells and bricks to be scaled up or down by the specified scale factors for the X, Y, and Z directions. The scaling will act with respect to the defined base point.

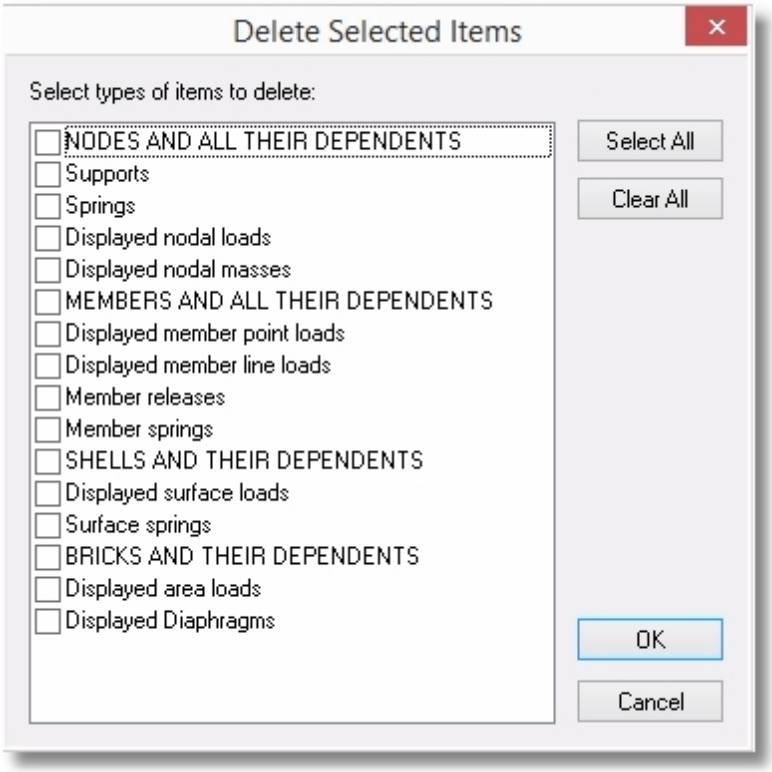


leads to



2.6.6 Delete

Modify > Delete acts on selected nodes, members, shells and bricks. It opens the *Delete Selected Items* dialog that allows the user to specify the types of items to delete.

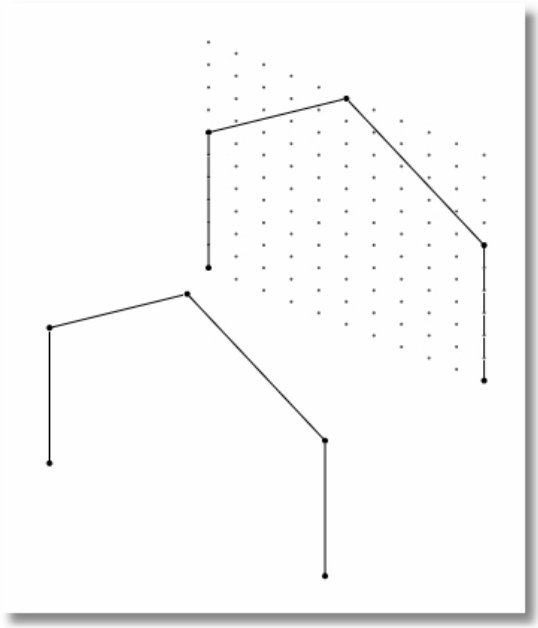


The options are organized into general categories of nodes, members, shells and bricks, and each category has a heading that automatically allows that type of entity and all of its dependents to be selected if desired. Otherwise, customize your desired selections with the appropriate checkboxes and then click the OK button to complete the deletion.

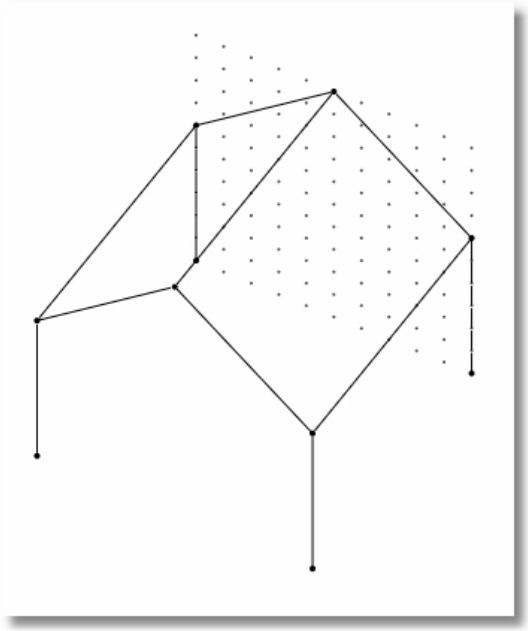
2.6.7 Extrude

Modify > Extrude acts on selected entities and allows them to be used to create new geometry.

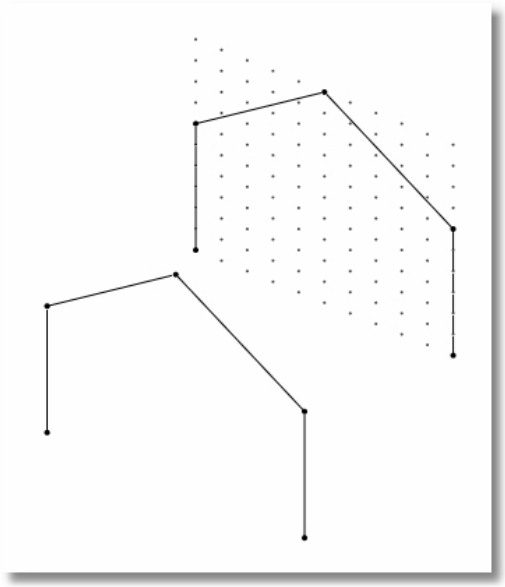
Extrude Nodes to Members acts on selected nodes. It collects a distance list and a direction. When executed, it produces one or more new members by extruding the selected node(s) by the specified distance in the chosen direction.



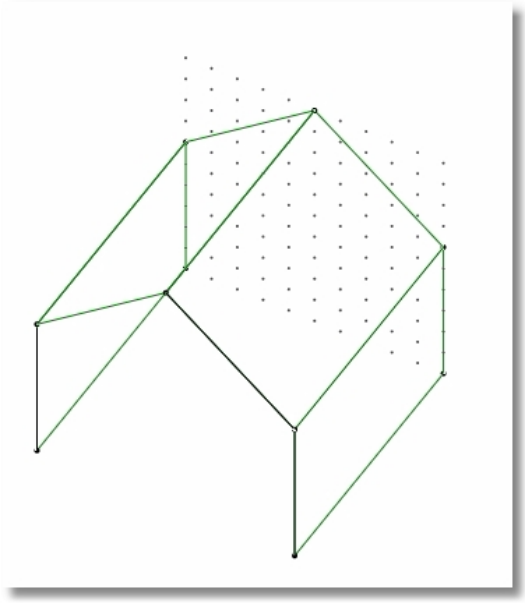
leads to



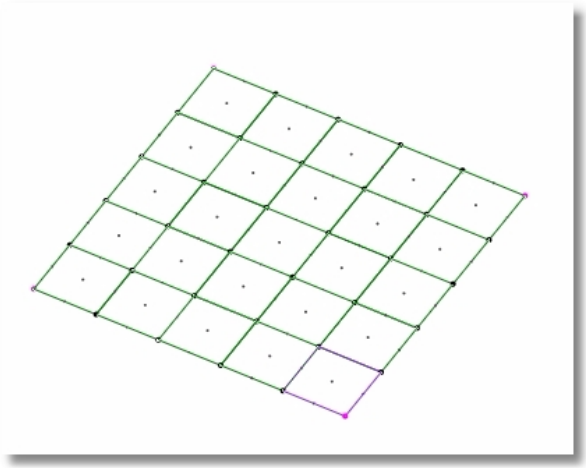
Extrude Members to Shells acts on selected members. It collects a distance list and a direction. When executed, it produces one or more new shells by extruding the selected member(s) by the specified distance in the chosen direction. This command offers the option to merge nodes and elements. When selected, if the extrusion results in a new node or member being placed directly on an existing node or member, the program will automatically merge (consolidate) the duplicate geometry to avoid modeling problems. This command also offers the option to delete the selected members after extrusion. This is useful if the member was only used as a construction tool to generate the shell and is not really needed in the model once the shell is generated.



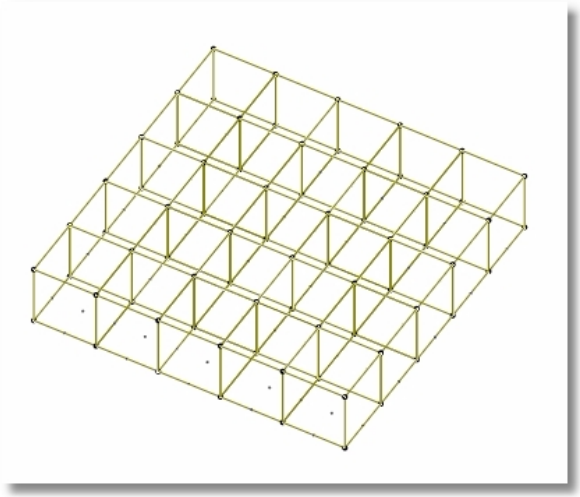
leads to



Extrude Shells to Bricks acts on selected shells. It collects a distance list and a direction. When executed, it produces one or more new bricks by extruding the selected shell(s) by the specified distance in the chosen direction. This command offers the option to merge nodes and elements. When selected, if the extrusion results in a new node or element being placed directly on an existing node or element, the program will automatically merge (consolidate) the duplicate geometry to avoid modeling problems. This command also offers the option to delete the selected shells after extrusion. This is useful if the shell was only used as a construction tool to generate the brick and is not really needed in the model once the brick is generated.



leads to



In all three versions of this command, note that the distance list accepts syntax such as:

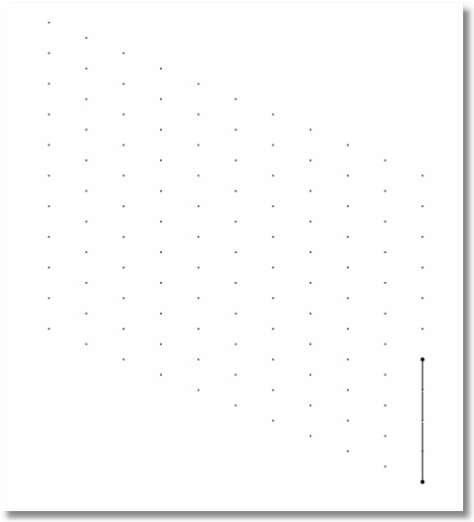
- 12 (to extrude once by 12 feet)
- 12, 24, 10 (to extrude by 12 feet, then by 24 feet, then by 10 feet)
- 3@20 (to extrude three times, each 20 feet long)

2.6.8 Revolve

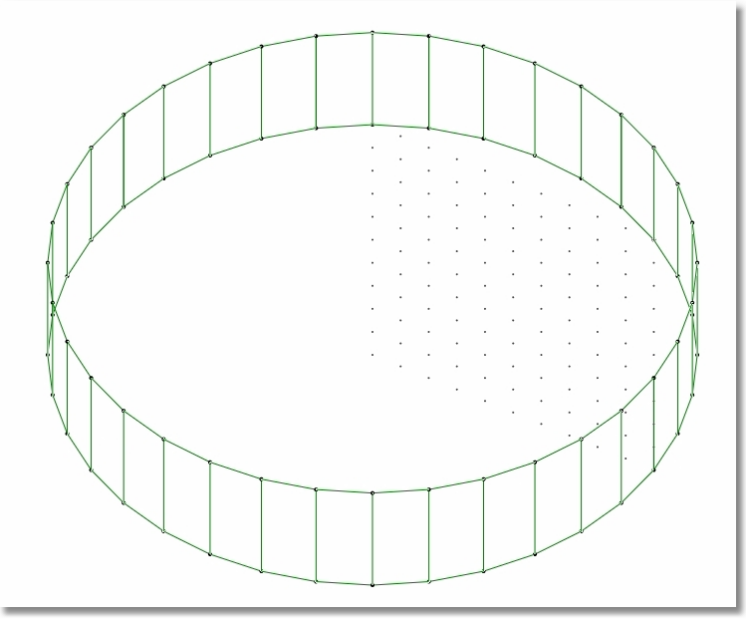
Modify > Revolve acts on selected entities and allows them to be used to create new geometry. It can be thought of as the rotational analogy to the Extrude commands.

Note: The initial members or shells should lie in the XY plane with Z=0.

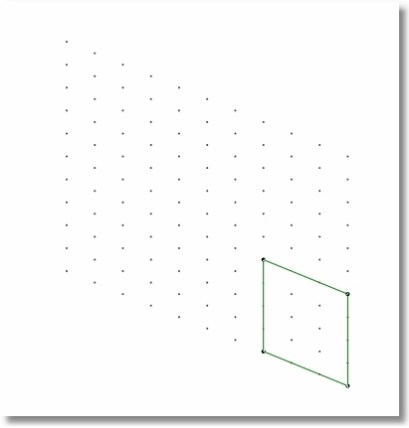
Revolve Members to Shells acts on selected members. It collects a number of segments, a start angle and an end angle. When executed, it produces one or more new shells by revolving the selected member(s) about the Y axis. This command automatically merges nodes if any duplicates are generated. It offers the option to delete the selected members after revolution. This is useful if the member was only used as a construction tool to generate the shell and is not really needed in the model once the shells are generated.



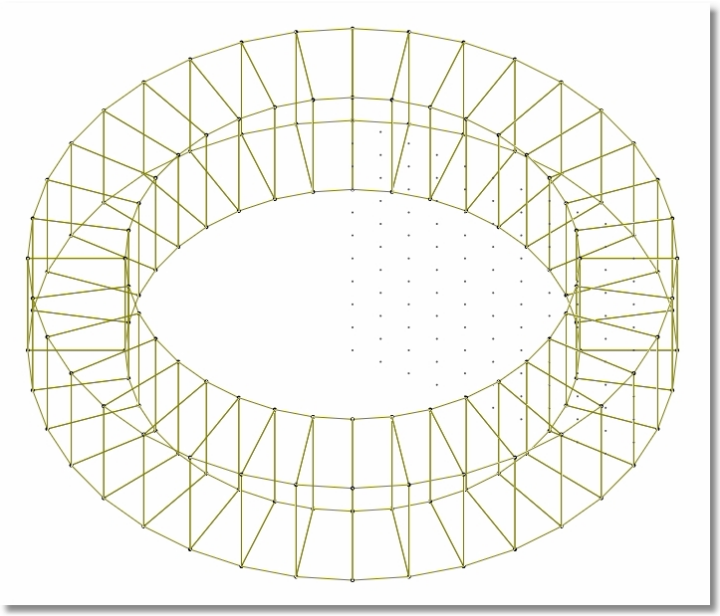
leads to



Revolve Shells to Bricks acts on selected shells. It collects a number of segments, a start angle and an end angle. When executed, it produces one or more new bricks by revolving the selected shell(s) about the Y axis. This command automatically merges nodes if any duplicates are generated. It offers the option to delete the selected shells after revolution. This is useful if the shell was only used as a construction tool to generate the bricks and is not really needed in the model once the bricks are generated.



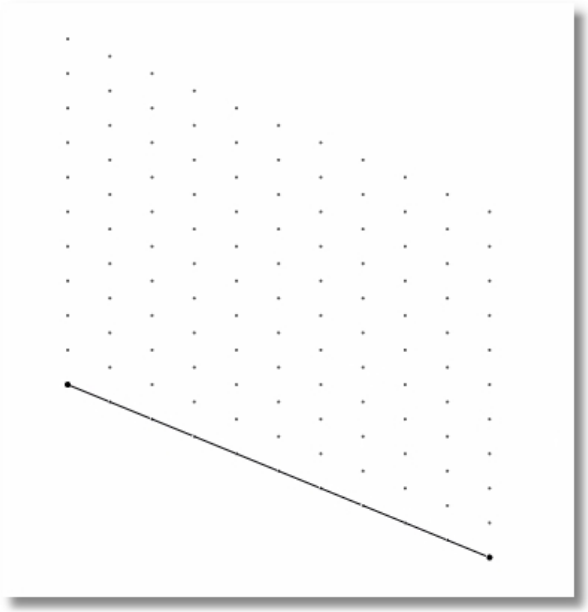
leads to



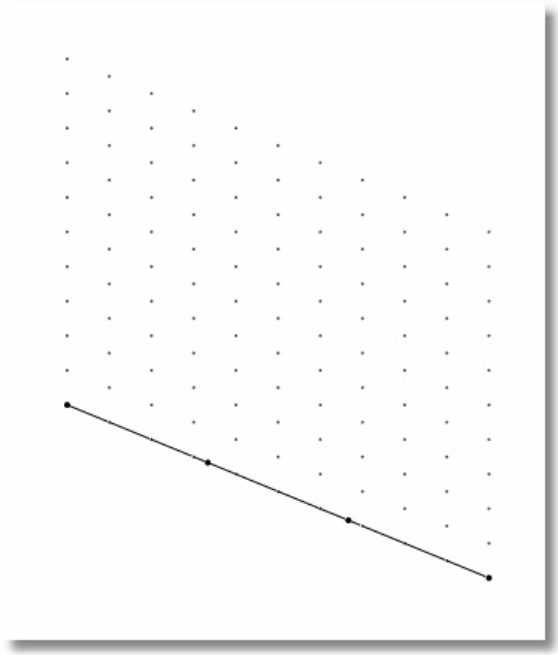
2.6.9 Split

Modify > Split provides a few different ways to subdivide the selected members.

Split > Divide selected members into segments of equal length allows the user to specify the number of segments. When executed, this option will subdivide all selected members into the specified number of members.

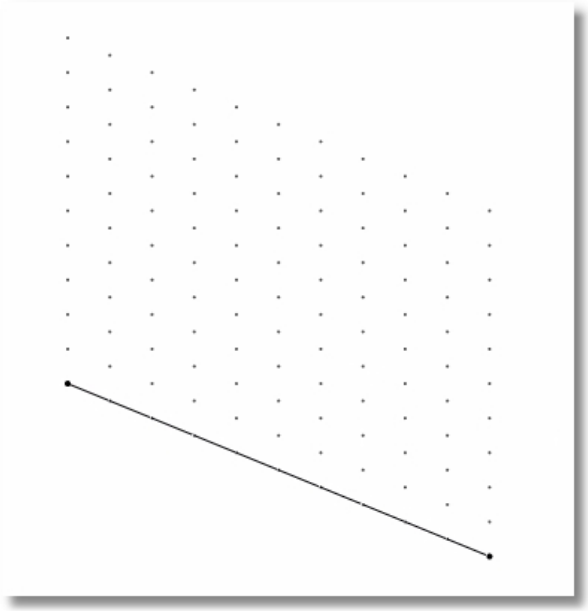


leads to

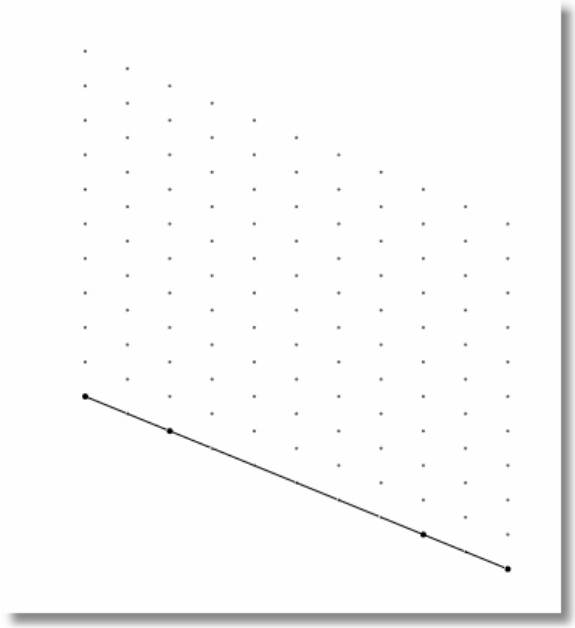


Split > Divide selected members by specifying a distance list allows the user to specify the locations at which the selected members are to be split. The distance list in this command accepts syntax such as:

- 12 (to split once at 12 feet)
- 12, 24, 10 (to split at 12 feet, then at 24 feet, then at 10 feet)
- 3@20 (to split three times, once at 20 feet, once at 40 feet and once at 60 feet)



leads to



Both options of this command offer the option to allow nodes and elements to be merged. When the option is selected to Merge nodes and elements, if the split results in a new node or

member that falls exactly on top of another node or member, the program will automatically eliminate the duplicate to avoid problems with model geometry.

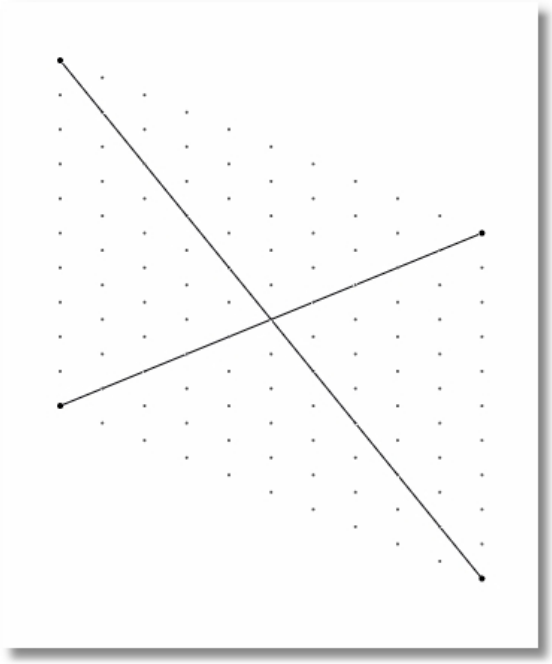
Note: After performing a Split command, it is advisable to renumber nodes by clicking Modify > Renumber > Auto Renumber All Nodes prior to running an analysis. This helps to conserve memory usage.

Split > Insert Nodes at Intersections of Selected Members:

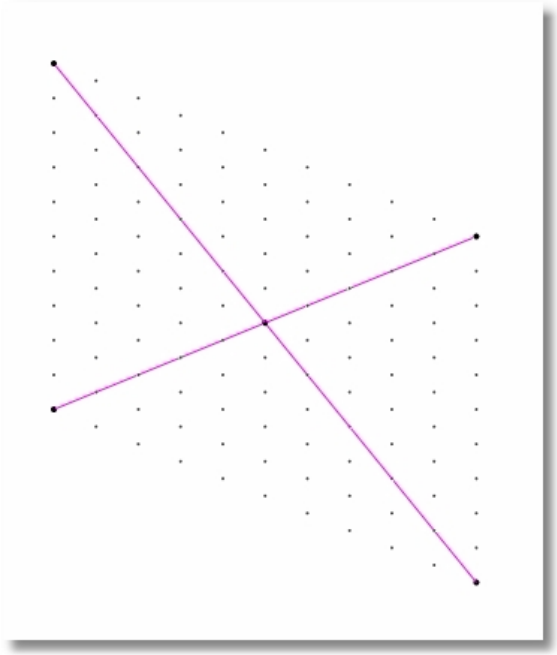
Just because lines cross each other in a plane in ENERCALC 3D does not mean that they are mathematically connected to each other at the intersection point. In fact, unless the members are all connected to a common node at the intersection point, there is no load transfer between the members. This can be thought of as a tension rod running right through a turnbuckle in the opposite diagonal. Each member can each carry load, but there is no connection at that intersection, so they don't share or transfer load at that point.

But now envision a cross brace consisting of HSS sections that connect to a common gusset plate at the intersection. This is a completely different situation that calls for a common node to be placed at the point of intersection.

The **Split > Insert Nodes at Intersections of Selected Members** command provides an easy way to add that node without having to calculate its coordinates. It acts on selected members, and it simply inserts a new node at all intersection locations of selected members.



leads to

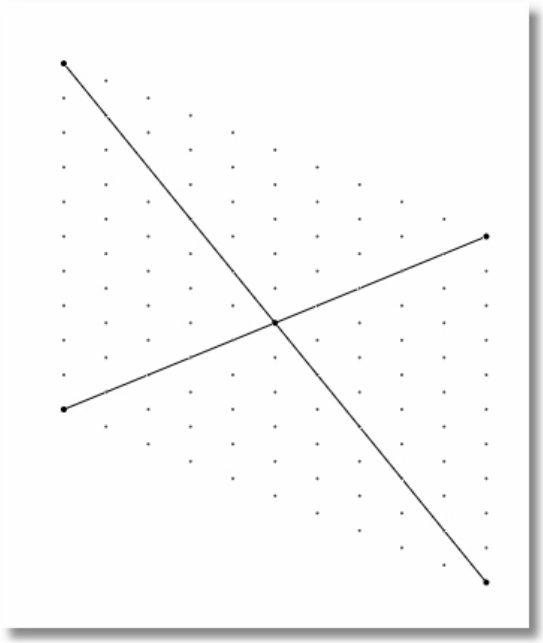


Note: This command ONLY inserts the node(s), but it does not automatically split the selected members so that they frame into that new node. A warning is displayed to this effect with a reminder that if the goal is to connect the members to that new node, the desired command is Split Selected Members at Nodes, which is covered next.

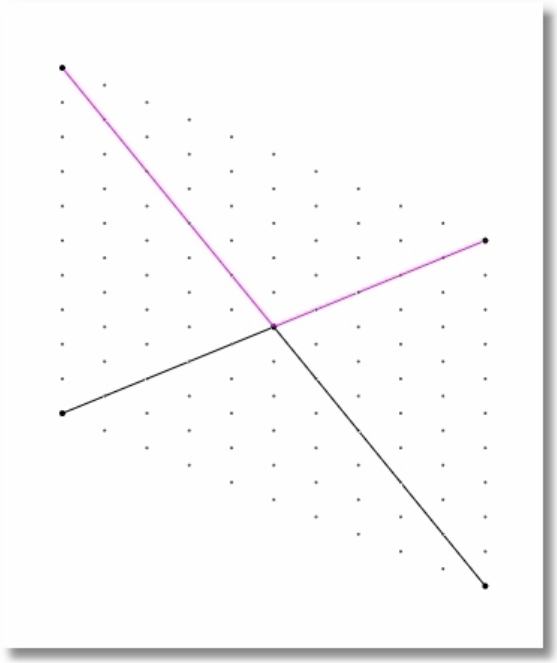
Split > Split Selected Members at Nodes acts on selected members and splits them at any locations where a node lies directly on the member.

It can be particularly useful after using the Split > Insert Nodes at Intersections of Selected Members command, or after placing infill beams that are intended to frame into a supporting girder.

After the command is executed, the member(s) will be split at node locations and all member incidences (connections to nodes) will be maintained and properly displayed in the *Members* table.

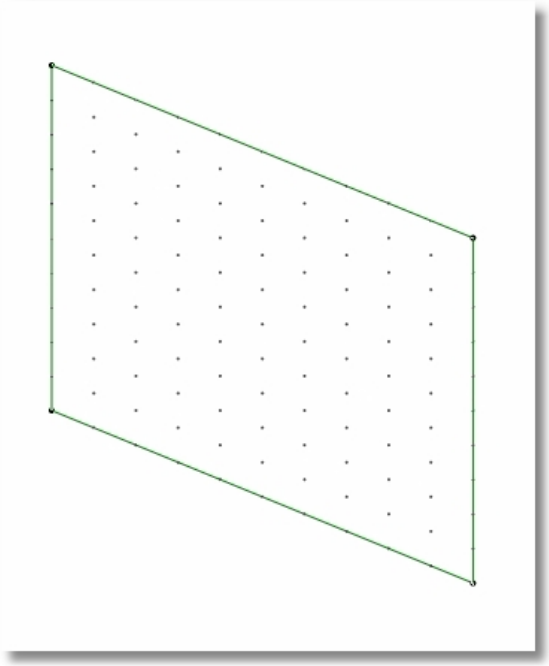


leads to

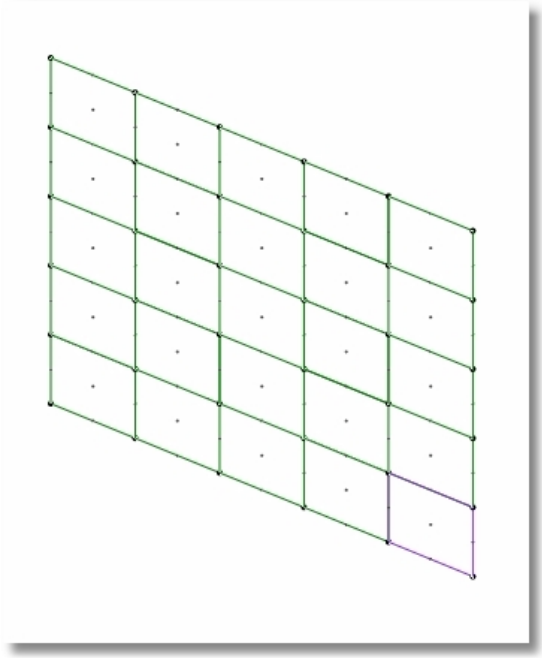


2.6.10 Sub-Mesh Shells

Modify > Sub-Mesh Shells subdivides the selected shell(s) by using the user-specified number of segments along both edges of the shell(s).



leads to



Remember that node numbers can be turned on for reference by clicking **View > Display Options > Shell Display Options > Shell Number**, or by using the  button on the Quick Access Toolbar.

To relate node numbers to the reference corners 1 through 4 of a selected shell, click **Tables > Shells**.

2.6.11 Renumber

Modify > Renumber offers a few ways to renumber entities in the model.

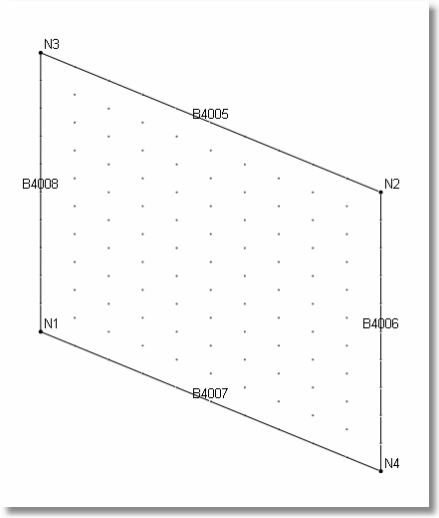
Auto Renumber All Nodes provides a quick way to automatically renumber all nodes in the current model. It provides control over how the renumbering is to progress with three dropdown list boxes that can be used to prioritize the process. This can be helpful because it allows the nodes to be renumbered, for example, in a way that generally progresses from top to bottom in the model, and then within a given level the renumbering will primarily flow from east to west and secondarily from north to south.

Renumber Selected Nodes provides two options:

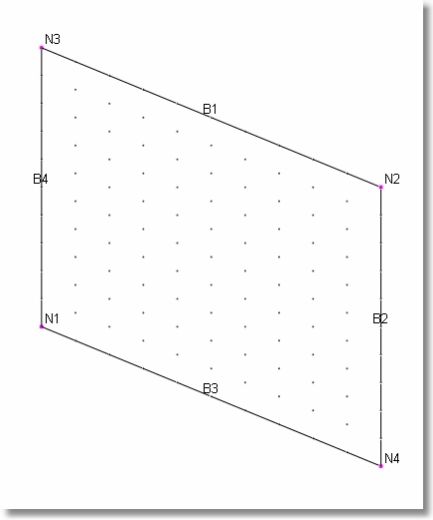
- *Increment each selected node number* allows node numbers to be incremented by a user-specified amount.
- *Renumber each selected node* allows nodes to be renumbered starting at a user-specified start number and incrementing by a user-specified step.

Renumber Selected Members provides two options:

- *Increment each selected member number* allows member numbers to be incremented by a user-specified amount.
- *Renumber each selected member* allows members to be renumbered starting at a user-specified start number and incrementing by a user-specified step.



leads to



Renumber Selected Shells provides two options:

- *Increment each selected shell number* allows shell numbers to be incremented by a user-specified amount.
- *Renumber each selected shell* allows shells to be renumbered starting at a user-specified start number and incrementing by a user-specified step.

Renumber Selected Bricks provides two options:

- *Increment each selected brick number* allows brick numbers to be incremented by a user-specified amount.
- *Renumber each selected brick* allows bricks to be renumbered starting at a user-specified start number and incrementing by a user-specified step.

In general, renumbering can be useful for the user's convenience. But it can also improve calculation performance by reducing the amount of memory required to analyze a given model. Naturally, it is more significant in larger models and in situations where large spans of unused node numbers have developed through the model generation process and through deletions.

2.6.12 Switch Coordinates

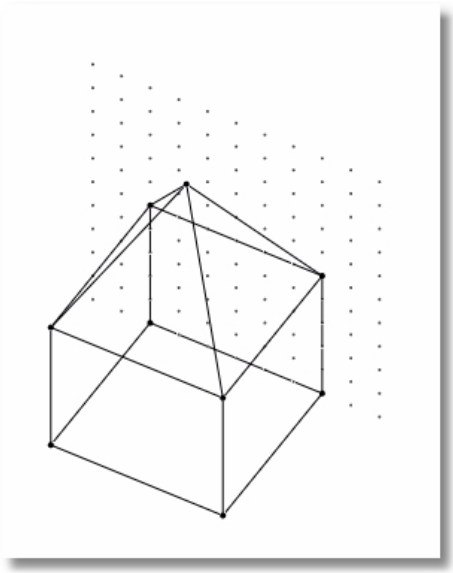
Modify > Switch Coordinate provides a few options for swapping the X, Y and Z coordinates of the selected geometry.

X --> Z & Z --> X has the effect of rotating the selected geometry about the Y axis.

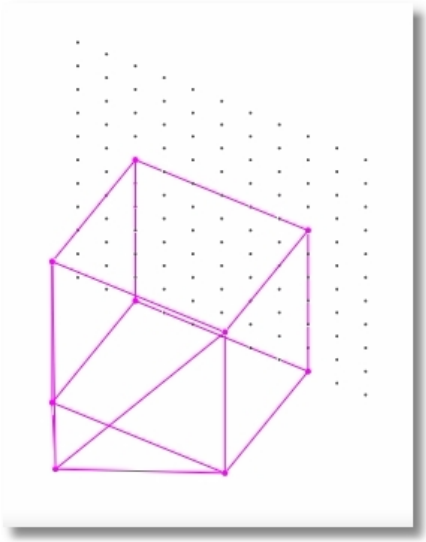
X --> -Z & Z --> X has the effect of rotating the selected geometry the other direction about the Y axis.

Y --> Z & Z --> Y has the effect of rotating the selected geometry about the X axis.

Y --> -Z & Z --> Y has the effect of rotating the selected geometry the other direction about the X axis.



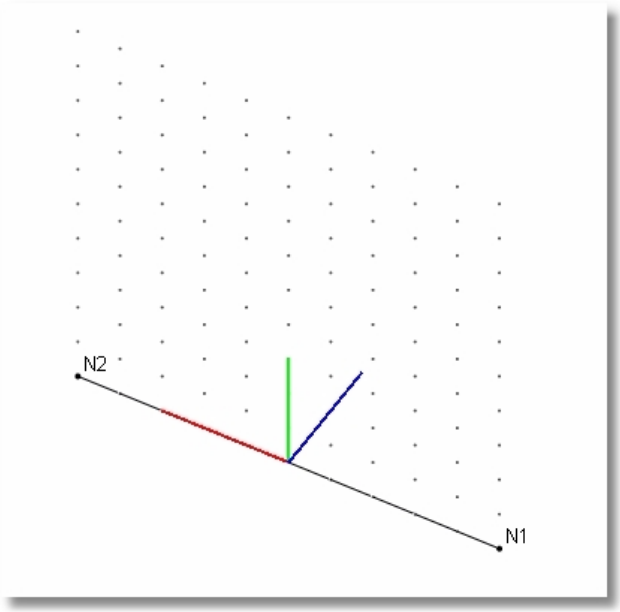
leads to



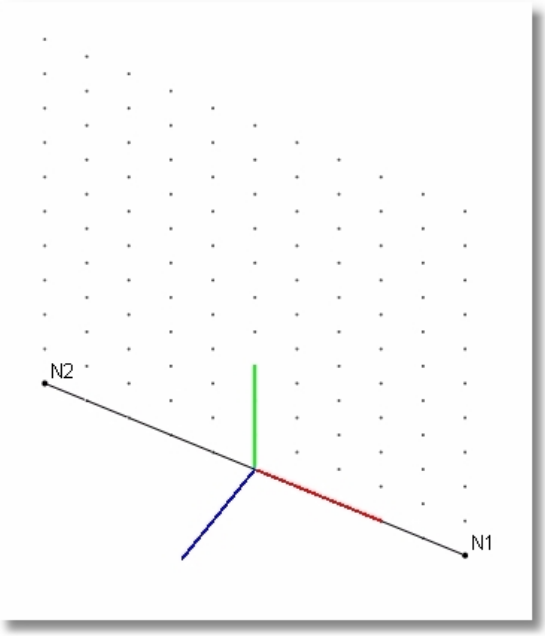
2.6.13 Reverse Node Order for Selected Elements

Modify > Reverse Node Order for Selected Elements has the following effects on selected elements:

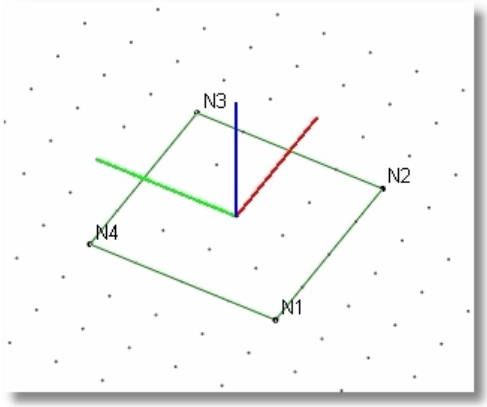
Members: The starting node becomes the ending node, so the local X axis points in the opposite direction.



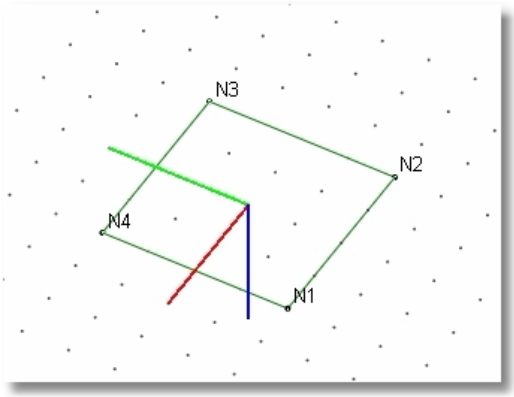
leads to



Shells: If the node order was originally 1-2-3-4, the order will be revised to 2-1-4-3. Notice that this has the effect of reversing the direction of the local z (normal) axis, as well as reversing the direction of the local x axis.



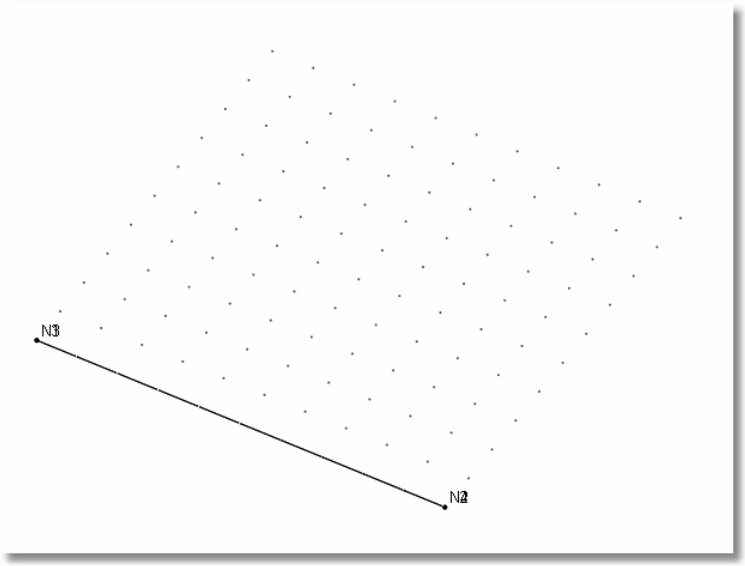
leads to



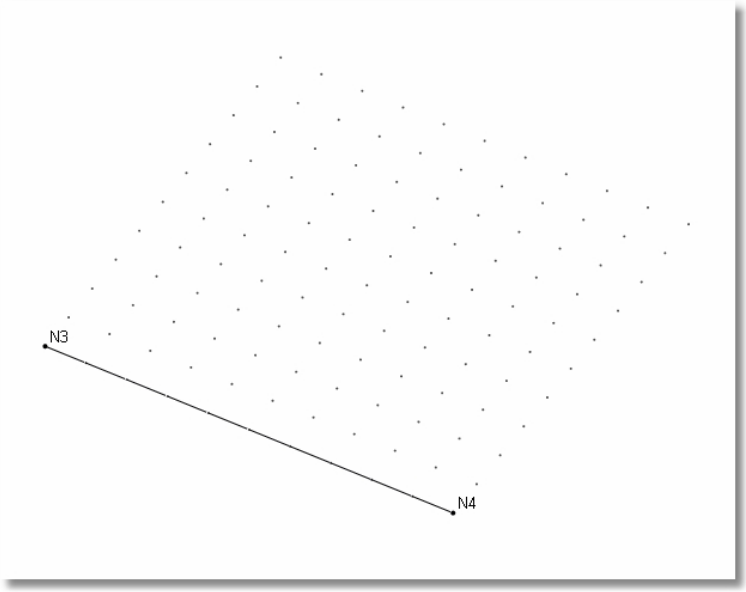
(Wondering why it didn't result in 4-3-2-1? Because, while that would have the effect of reversing the direction of the local z (normal) axis, it would also introduce the possibility of changing the orientation of the local x axis, rather than guaranteeing that it was simply reversed.)

2.6.14 Merge All Nodes & Elements

Modify > Merge All Nodes and Elements looks through the model to see if there are any duplicate nodes, members, shells or bricks. If it finds any, it consolidates them so there are no longer multiples in exactly the same spot, as that would have detrimental effects on model performance.

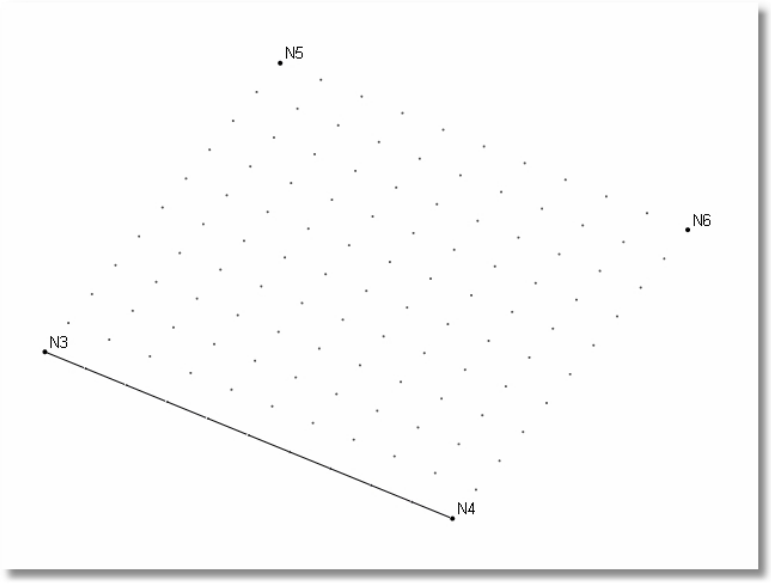


leads to

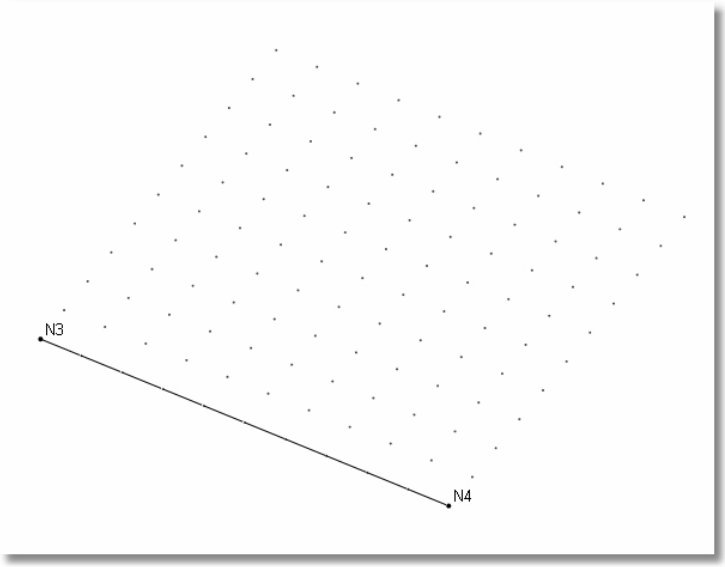


2.6.15 Remove All Orphaned Nodes

Modify > Remove All Orphaned Nodes looks through the model to see if there are any nodes that are disconnected from the any other entities. If it finds any, it deletes them, as they would have detrimental effects on the analysis of the model.



leads to

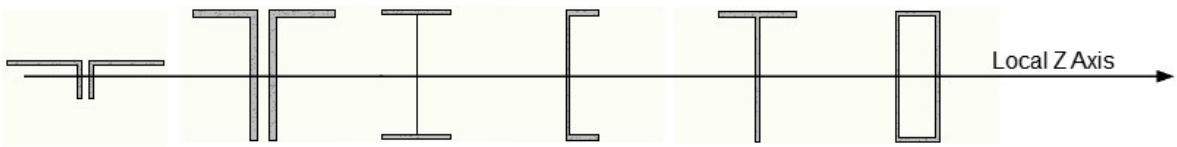


2.6.16 Element Local Angles

As discussed previously, members and shells have a local axis system that adopts a default orientation when the member or shell is first modeled.

For a refresher on the default member local axis system, refer to Understanding Model Geometry > Members > Member Local Axes.

The following diagram shows how typical steel sections are oriented on that default local axis system:

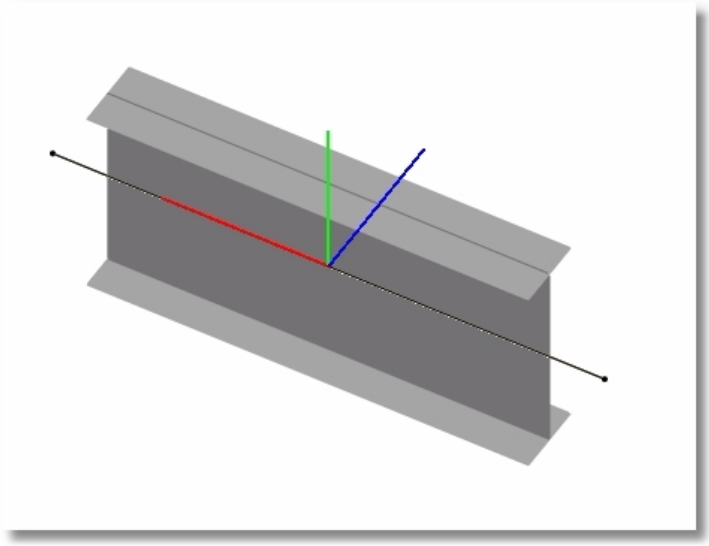


But there are times where a member needs to be rotated about its longitudinal (local x) axis to take on a different orientation, such as a channel with the flanges pointing downward or a WT with the stem pointing upward. This is the purpose of the **Modify > Member Properties > Element Local Angles** command.

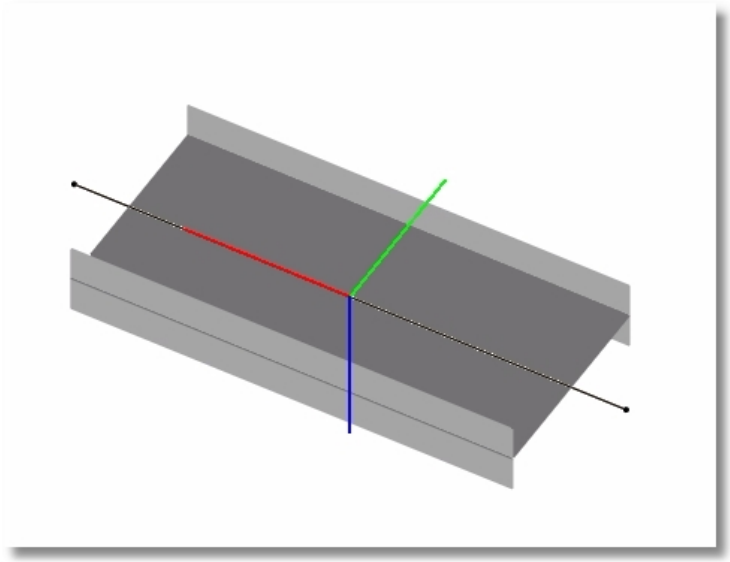
The default orientation represents an Element Local Angle of zero degrees. To understand how a positive Element Local Angle will affect a member, envision holding the member in your right hand, so that your right thumb points in the direction of the local x axis. The natural curl of your right fingers indicates the direction of rotation due to an Element Local Angle.

So when viewing the diagram above, assume the local x axis points into the plane of the diagram. To rotate the channel so the flanges point downward, you would apply an Element Local Angle of 90 degrees. And to rotate the WT so the stem points upward, you would apply an Element Local Angle of 180 degrees.

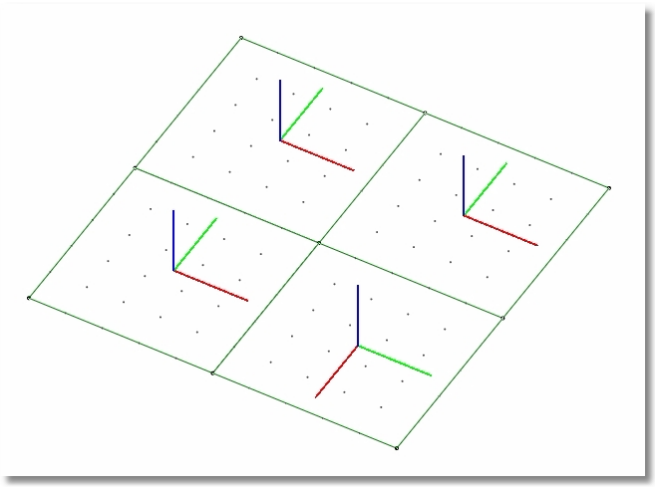
Note: Element Local Angles must always be specified as positive values. So if the goal is to apply a rotation of -90 degrees, use 270.



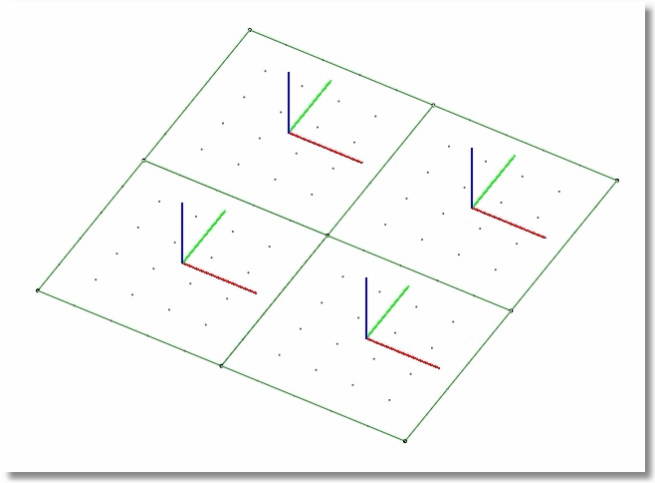
leads to



The concept of Shell Local Axes also applies. When modeling many plates, it is very useful to have them all oriented in the same way, to make it easier to interpret results. We have already seen a command called **Modify > Reverse Node Order for Selected Elements** that can be used to flip a plate over so that its top becomes the bottom and vice versa. But what if there is a need to turn a plate so that its local x axis corresponds to the local x axis of a neighboring plate? That is where **Modify > Member Properties > Element Local Angles** can be used, because is specifies a rotation of the shell local axes about the normal (local z) axis.



leads to



Part III

3 Assigning Properties to Modeling Entities

3.1 Properties for Nodes

3.1.1 Supports

Supports are boundary conditions applied to nodes. Supports can restrain movement or rotation for one or more global degrees of freedom, or they can enforce a prescribed movement or rotation for one or more global degrees of freedom (such as support settlement).

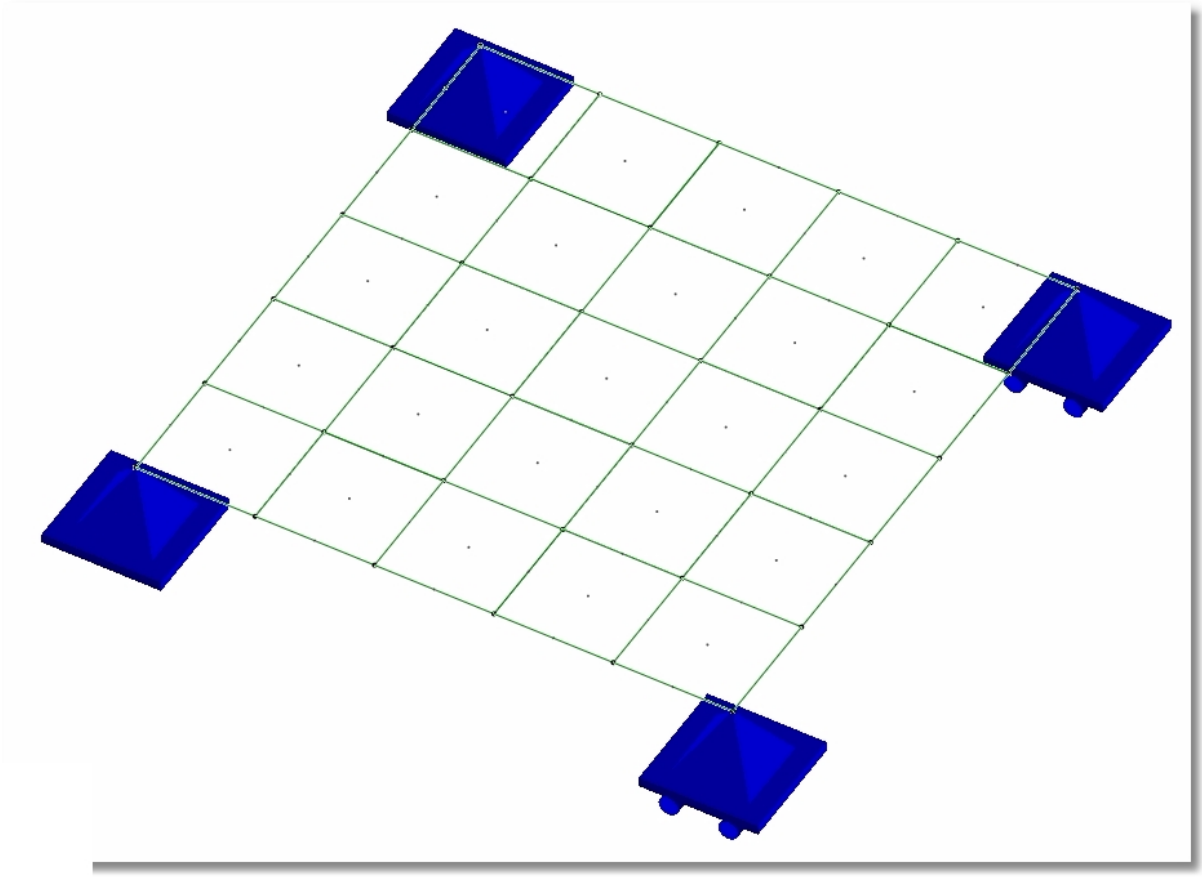
Note that the program refers to translation degrees of freedom as X, Y, and Z, which correspond to the global coordinate axes. And note that the program refers to rotational degrees of freedom as OX, OY, and OZ, representing rotation about global X, Y, and Z, respectively.

There are at least a couple different ways to define and assign supports.

Graphical Method

If you preselect the desired node or nodes, you can click **Create > Boundary Conditions > Support**. The *Support* dialog opens.

It allows you to define the desired support type and assign it to the selected nodes. In order to set a prescribed displacement, the corresponding degree of freedom must be set to restrained.



Tabular Method

Click **Tables > Supports** to open the *Supports* table. In this table you can edit existing support data or insert new rows and create new support assignments.

The column with the heading "6-DOFs" uses 1 for fixed and 0 for free in each of the 6 degrees of freedom listed in the following order: X, Y, Z, OX, OY, OZ. Using this syntax, you can create any type of support you want. In order to set a prescribed displacement, the corresponding degree of freedom must be set to restrained.

3.1.2 Springs

Springs, like Supports, are boundary conditions. But unlike Supports, Springs can be applied to nodes (as nodal springs), members (as line springs), and shells (as surface springs). Springs also have the ability to act as linear springs, compression-only springs, or tension-only springs.

Springs are defined by a stiffness (linear or rotational) associated with one or more global degrees of freedom.

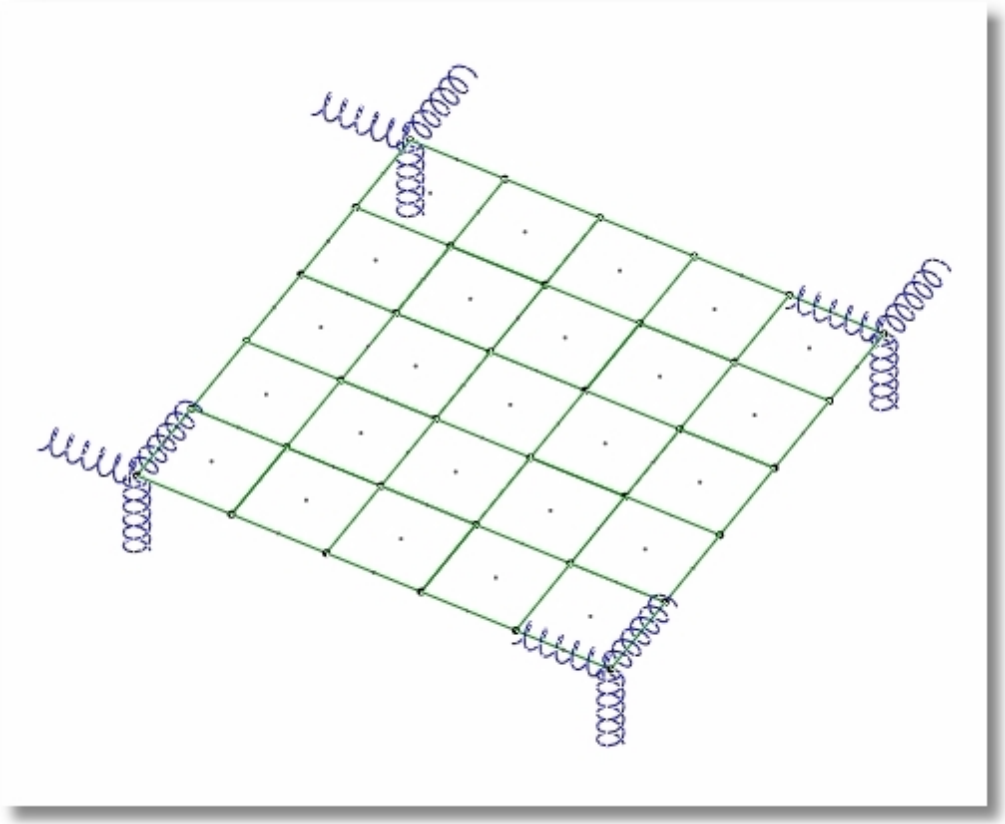
Note that the program refers to translation degrees of freedom as X, Y, and Z, which correspond to the global coordinate axes. And note that the program refers to rotational

degrees of freedom as OX, OY, and OZ, representing rotation about global X, Y, and Z, respectively.

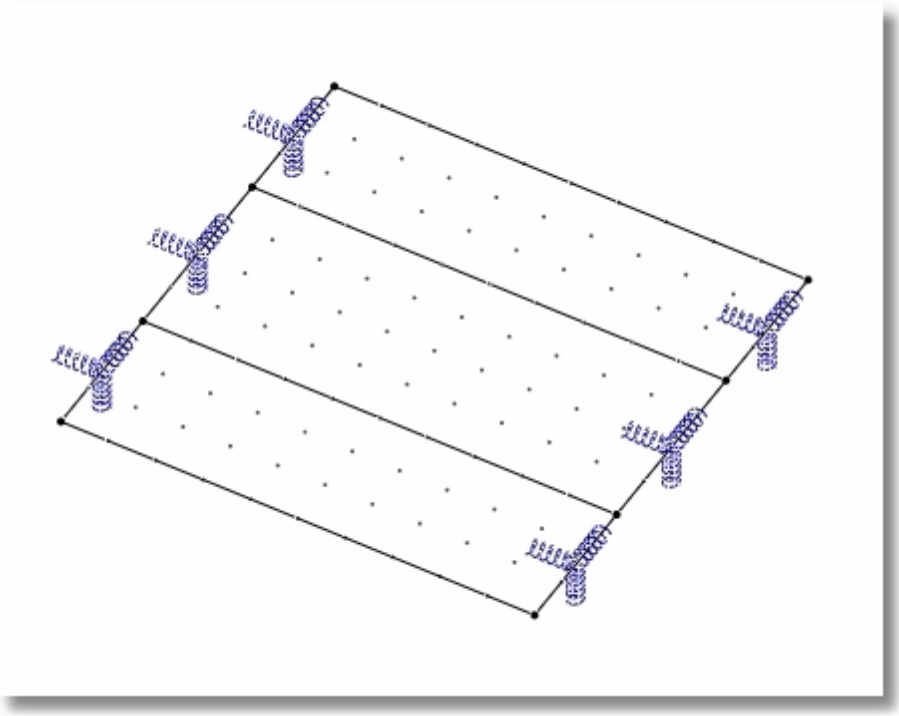
There are at least a couple different ways to define and assign springs.

Graphical Method

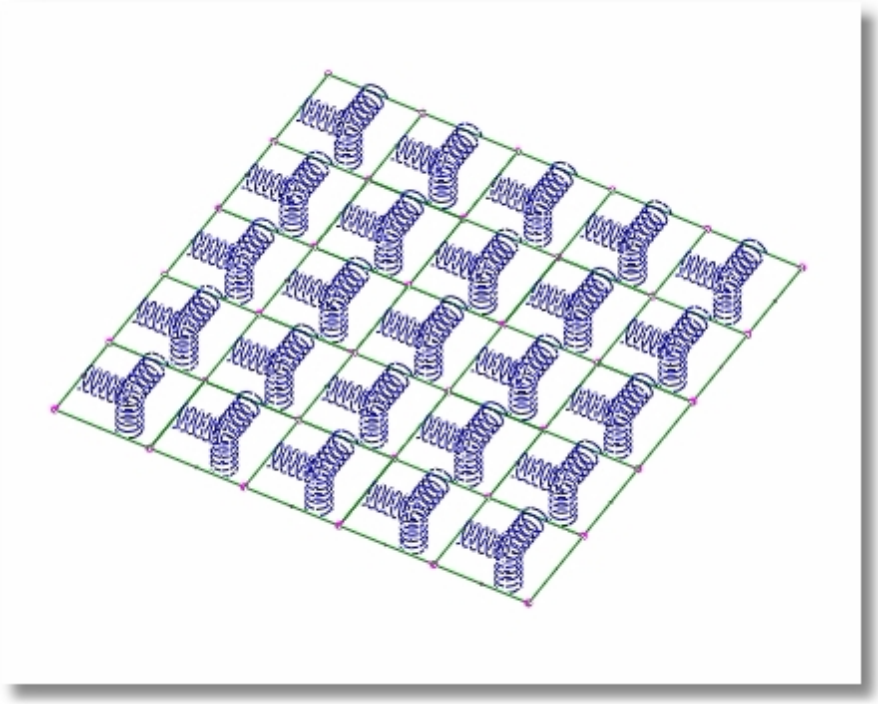
If you preselect the desired node or nodes, you can click **Create > Boundary Conditions > Springs**. Then you can select the desired spring type (nodal, line, or surface) and assign it to the selected nodes.



Nodal Springs



Line Springs



Surface Springs

Tabular Method

If you click **Tables > Springs**, you will be given the option to select **Nodal Springs**, **Line Springs** or **Surface Springs**. In all cases, you can edit existing spring data or insert new rows and create new spring assignments.

If you select Nodal Springs, the *Nodal Springs* screen displays. The column with the heading "6-DOFs" uses 0 for linear spring, 1 for compression-only spring and 2 for tension-only spring. All six degrees of freedom must use the same type of spring, and they listed in the following order: X, Y, Z, OX, OY, OZ. Using this syntax, you can create any type of spring you want. The columns to the right hold the stiffness values that you want to have applied in each degree of freedom.

If you select Line Springs, the *Line Springs* screen displays. The column with the heading "6-DOFs" uses 0 for linear spring, 1 for compression-only spring and 2 for tension-only spring. This type of spring only collects stiffnesses for the three translational degrees of freedom. All three degrees of freedom must use the same type of spring, and they listed in the following order: X, Y, Z. Using this syntax, you can create any type of spring you want. The columns to the right hold the stiffness values that you want to have applied in each degree of freedom.

If you select Surface Springs, the *Surface Springs* screen displays. The column with the heading "6-DOFs" uses 0 for linear spring, 1 for compression-only spring and 2 for tension-only spring. This type of spring only collects stiffnesses for the three translational degrees of freedom. All three degrees of freedom must use the same type of spring, and they listed in the following order: X, Y, Z. Using this syntax, you can create any type of spring you want. The columns to the right hold the stiffness values that you want to have applied in each degree of freedom.

3.1.3 Diaphragm

Diaphragms are a way to specify that a group of coplanar nodes must translate and rotate as though they were connected by a rigid plate.

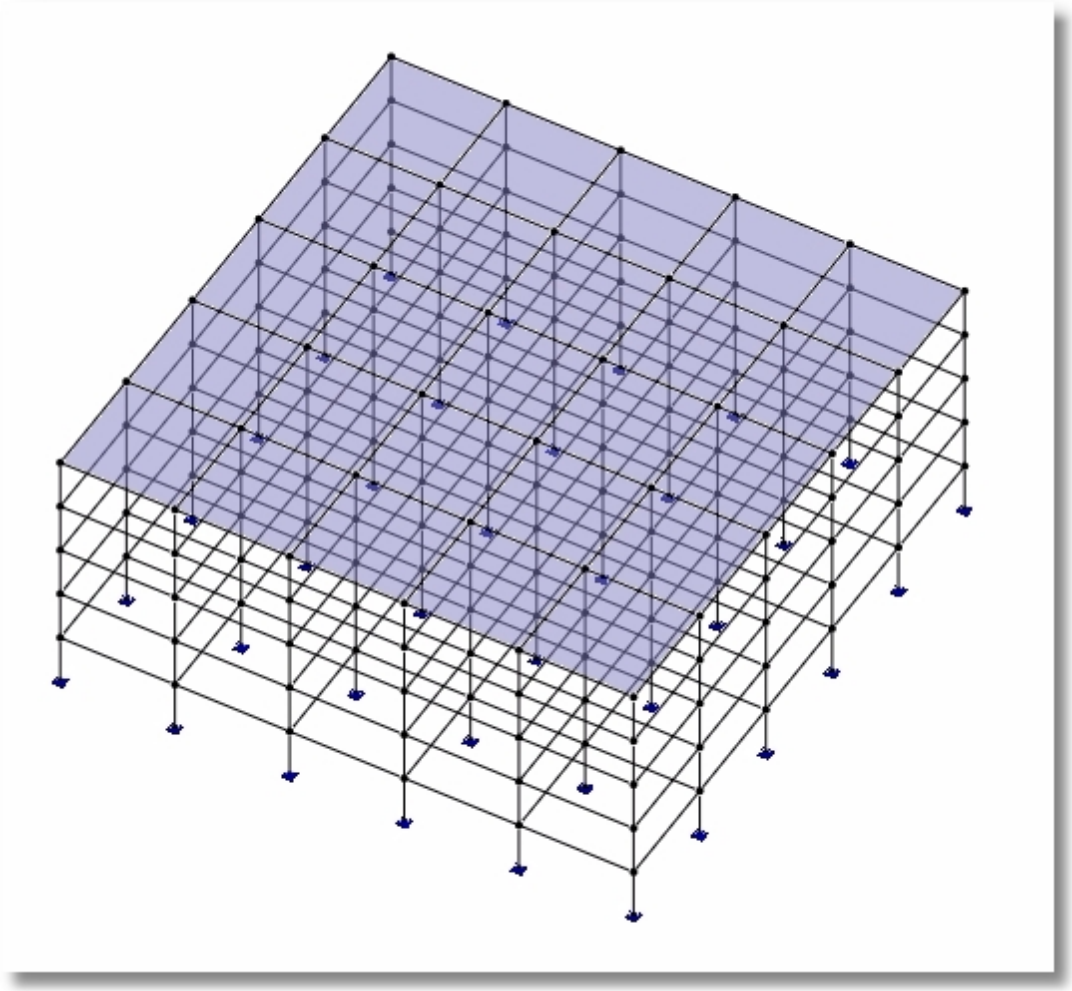
There are at least two different ways to define diaphragms: Graphically and in tabular format.

Graphical Method

If you click **Draw > Draw Diaphragms**, it will display the *Diaphragms* dialog. Note that it offers Generic and Regular diaphragms.

Generic diaphragms can be in any orientation, including sloping planes. They are defined by clicking on four nodes that define the perimeter of the area that is to be considered a diaphragm.

Regular diaphragms must be defined as falling in the XZ, YZ, or XY plane. They are defined by selecting a plane and one or more nodes. If more than one node is selected, the program will automatically create as many unique diaphragms as there are nodes selected.



Tabular Method

Click **Tables > Diaphragms Data** to open the *Diaphragms* table. This allows existing diaphragms to be edited or deleted, and allows new diaphragms to be defined.

The type can be set to Generic, XZ Plane, YZ Plane, or XY Plane. If the type is set to Generic, then four unique nodes must be entered in the four remaining columns in the table. If the type is set to one of the other options, then only one node number is required, and the same number should be entered into all four columns in the table.

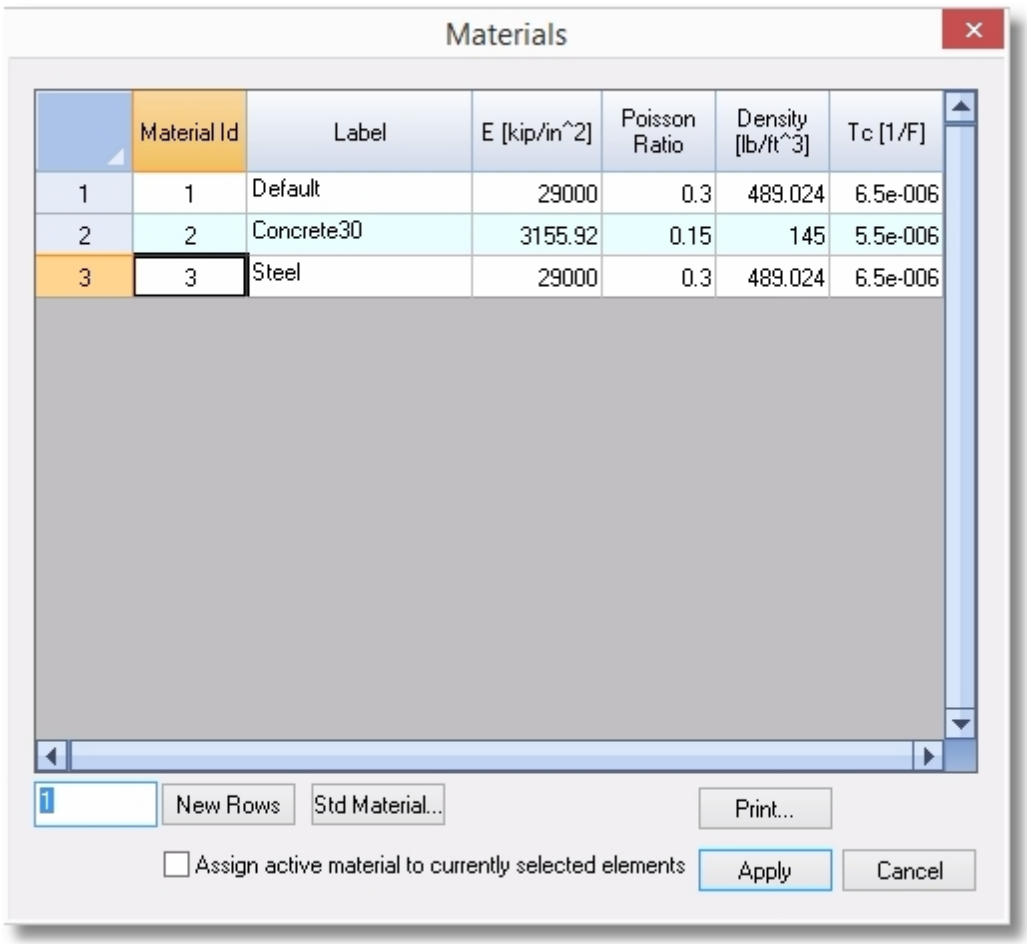
3.2 Properties for Members

3.2.1 Materials

Materials consist of the properties necessary for structural analysis: Young's Modulus, Poisson's Ratio, Density and Coefficient of Thermal Expansion or Contraction. Note that these are not material strengths, which are related to code checking. Those get entered in a different location.

Materials can be defined and assigned using the *Materials* table, which can be accessed by clicking:

- **Create (or Modify) > Member Properties > Materials**
- **Create (or Modify) > Shell Properties > Materials**
- or by clicking **Tables > Materials**.



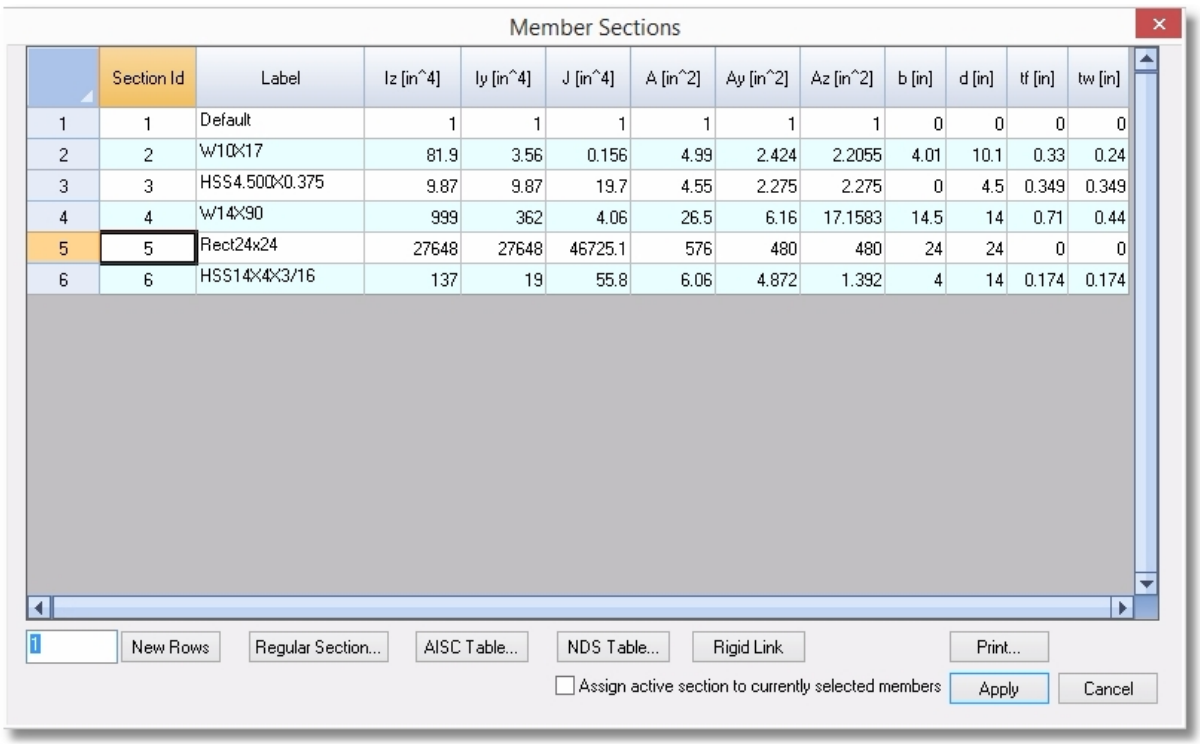
Regardless of how the table is accessed, it allows the user to create material definitions and to assign them to currently selected members.

Note that there is a Standard Material button on this table that allows for the quick definition of many standard steel and concrete materials.

3.2.2 Sections

Member Sections consist of the properties necessary for structural analysis, such as section properties and dimensions.

Member Sections can be defined and assigned using the *Sections* table, which can be accessed by clicking **Create (or Modify) > Member Properties > Sections**, or by clicking **Tables > Sections**.



	Section Id	Label	Iz [in^4]	Iy [in^4]	J [in^4]	A [in^2]	Ay [in^2]	Az [in^2]	b [in]	d [in]	tf [in]	tw [in]
1	1	Default	1	1	1	1	1	1	0	0	0	0
2	2	W10X17	81.9	3.56	0.156	4.99	2.424	2.2055	4.01	10.1	0.33	0.24
3	3	HSS4.500X0.375	9.87	9.87	19.7	4.55	2.275	2.275	0	4.5	0.349	0.349
4	4	W14X90	999	362	4.06	26.5	6.16	17.1583	14.5	14	0.71	0.44
5	5	Rect24x24	27648	27648	46725.1	576	480	480	24	24	0	0
6	6	HSS14X4X3/16	137	19	55.8	6.06	4.872	1.392	4	14	0.174	0.174

Regardless of how the table is accessed, it allows the user to create section definitions and to assign them to currently selected members.

Note that there are some convenience buttons at the bottom of the table:

Regular Section: allows for the quick definition of many standard section shapes, such as Rectangle, Round, Wide Flange and Tee.

AISC Table: opens a table of standard AISC steel sections.

NDS Table: opens a table of standard NDS wood sections including sawn lumber and glulam shapes.

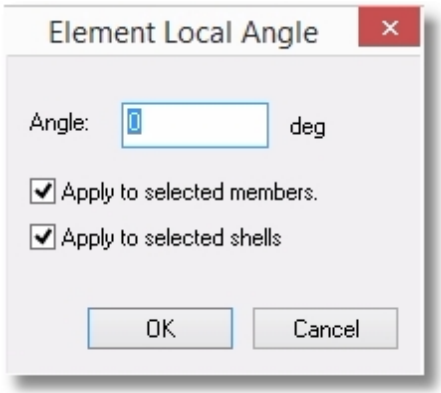
Rigid Link: allows for a way to create a "section property" definition that behaves as a completely rigid link when assigned to members in the model.

3.2.3 Local Angle

Local angle is a way to rotate a member about its longitudinal axis. It can be useful for situations such as when a wide flange steel member needs to be rotated from its default orientation of web-vertical to an alternate orientation, like web-horizontal when behaving as a girt.

Remember that each member has a local coordinate axis system. The orientation of the local x axis (from starting node to ending node) is particularly important. To visualize the way a positive local angle will affect a particular member, point your right thumb in the direction of the local x axis. The natural curl of your right fingers indicates the direction of rotation of the member when subjected to a positive local angle.

To assign a local angle to a member, select the member and then click **Create (or Modify) > Member Properties > Local Angles**. Either way, the *Element Local Angle* dialog opens. It allows for the specification of an angle to be applied to selected members. (It also can be applied to shells, which is covered in the section on Properties for Shells.)

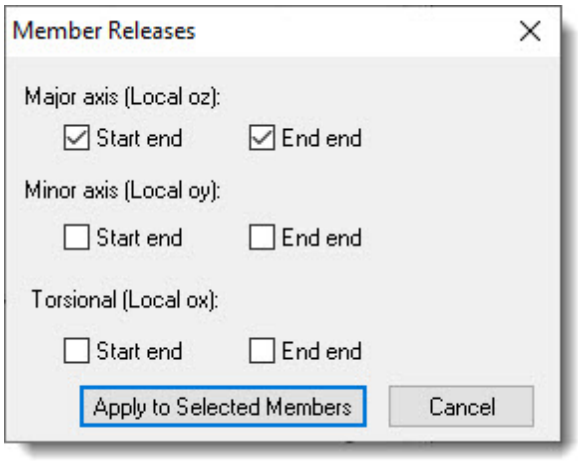


3.2.4 Moment Releases

Moment Releases provide a way to specify that a member has a hinge (no moment transfer capability). They can be applied to the start end, the ending end or both ends of a member, and they can be applied about the local y axis, the local z axis or both.

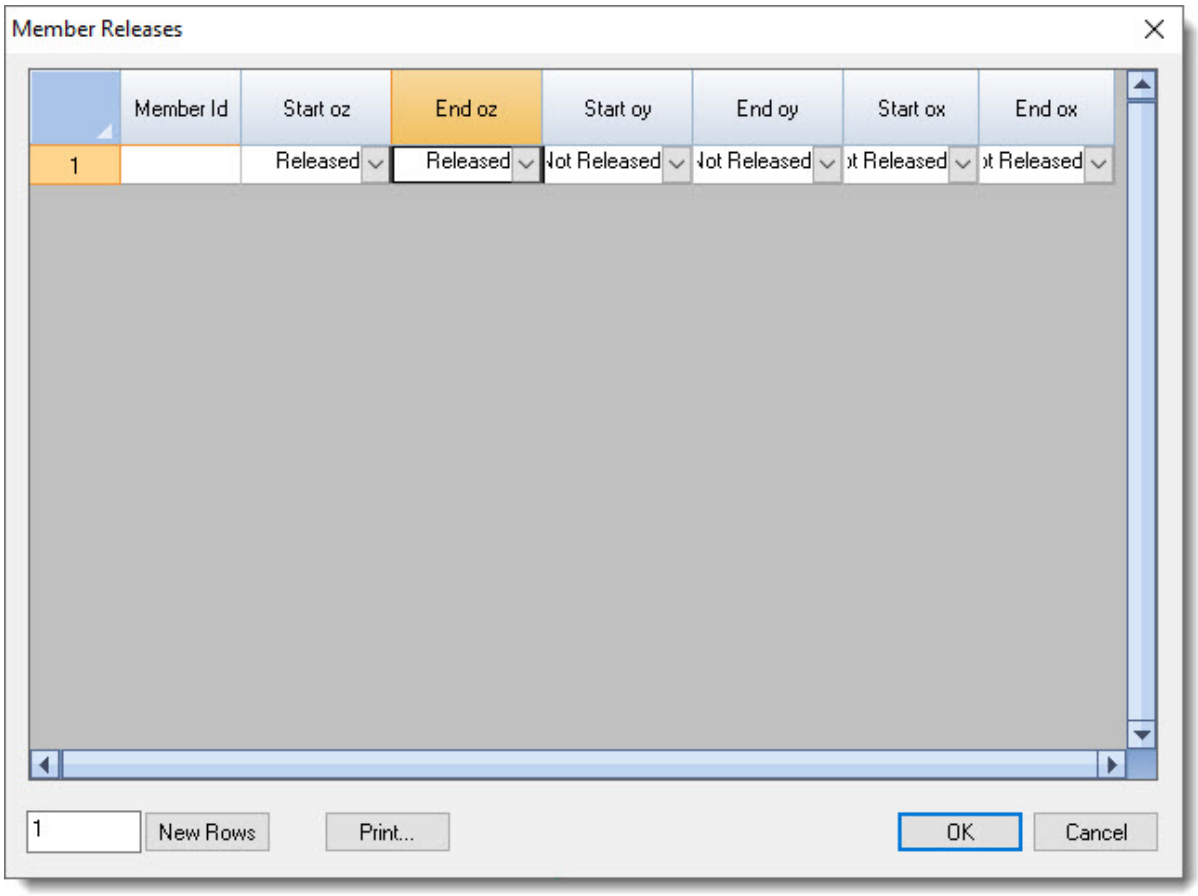
Graphical Method

Moment releases can be defined and assigned graphically by clicking **Create (or Modify) > Member Properties > Moment Releases**. This opens the *Member Releases* dialog, which allows a release to be defined and then assigned to selected members.



Tabular Method

Moment releases can be defined and assigned using the tabular method by clicking **Tables > Member Releases**. This opens the *Member Releases* table, which allows multiple member moment releases to be applied at once in the table.



Caution!

Use caution not to "over-release" when applying member releases, as this can lead to model instabilities. For example, picture two members connecting at a node. If the intent is to model a hinge at that location, then only ONE of the two members should have a release applied, because it only takes one release to prevent the transfer of moment. If BOTH members were released, it could cause a stability problem, because that common node is now disconnected (in a moment transfer sense) from ANY member in the model. So mathematically, it is able to spin, and this causes an instability in the mathematical solution of the model.

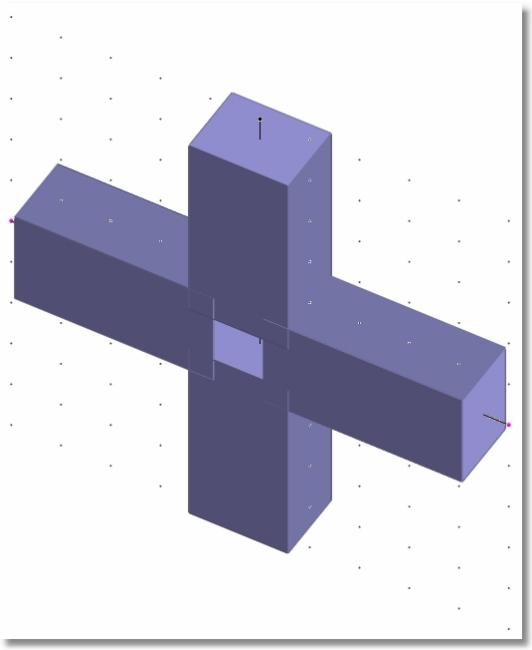
3.2.5 Rigid Offsets

Rigid Offsets provide a way to specify non-deformable end zones of a member. They come into play in concrete construction, where a beam frames into a relatively wide column.

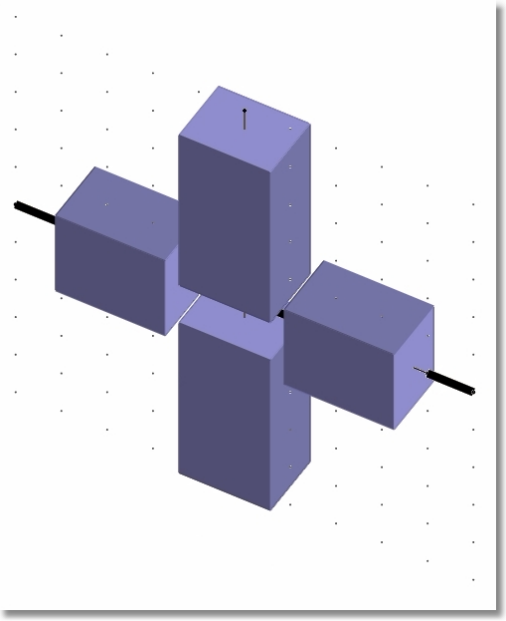
The analytical model consists of lines representing members. This means that a beam tends to span from column centerline to column centerline in a model. But in the physical structure where the columns have some real depth, the beam will behave much differently. It will actually behave more nearly like the span length is from face to face of the columns, rather than from center to center. This effect can have a significant impact of model behavior if spans are short and/or if columns are deep (wide).

The Rigid Offsets command provides a way to capture this effect mathematically.

To access this command click **Create (or Modify) > Member Properties > Rigid Offsets**. The *Member Rigid Offsets* dialog collects an offset dimension for the start and/or ending end of the member, and then allows that specification to be assigned to selected members.



leads to

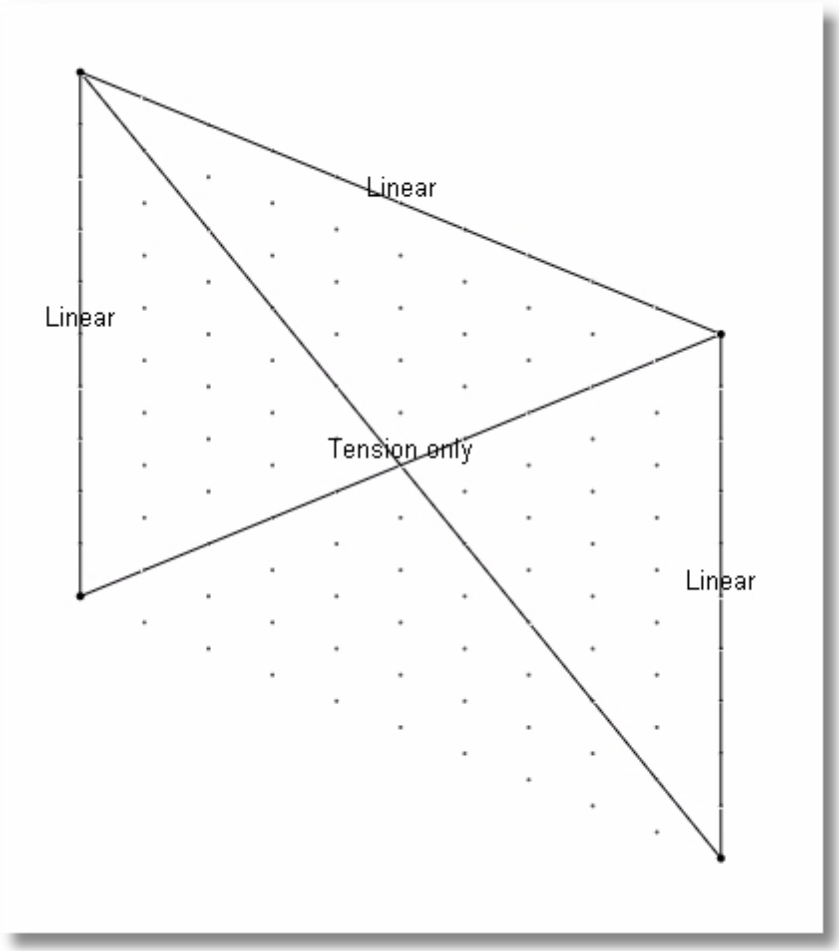


3.2.6 Tension/Compression Only

Sometimes it is desirable to specify that a particular member can only carry tension or can only carry compression forces. One example is a slender rod used as a hanger. Intuition might dictate that such a member would be so flexible that it would just tend to buckle and offer no resistance if it was ever put into compression. So these commands provide a way to specify that a particular member can only carry one type of force or the other.

If a specified member tends to experience the other type of force in any analysis, the program will simply disregard and stiffness or resistance from the specified member for that particular condition.

To specify that a member is tension-only or compression-only select the member and then click **Create (or Modify) > Member Properties > Tension/Compression Only**. The *Tension/Compression Only* dialog collects the specification and allows it to be assigned to selected members.

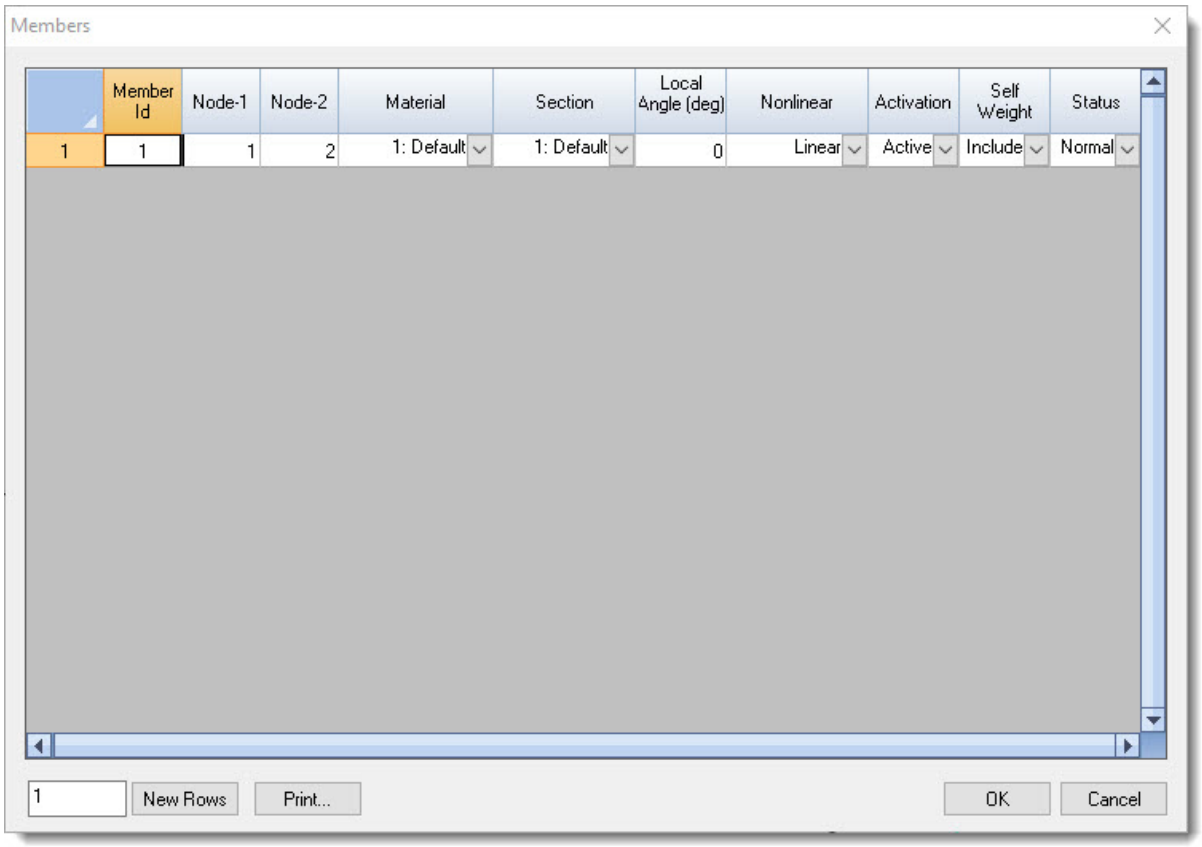


Tip: To remove a Tension/Compression Only specification, choose the Linear option and assign that to the member.

3.2.7 Members Table

Up to this point we have discussed creating and editing model geometry, and we have discussed defining and assigning properties to the members in a model. Many of these processes could be done either graphically or in tabular form.

The *Member Data* table is powerful because it brings many of these definitions and properties together in one concise table that can be reviewed and modified if needed. It can be opened by clicking **Tables > Members**.



The Members table includes:

- Member ID
- Starting and ending node numbers
- Material
- Section
- Local Angle
- Specification for Linear, Tension-Only or Compression-Only
- Status (options include Normal, Selected, and Frozen)

Other than the Member ID number, the remainder of these items can be revised in this table.

3.3 Properties for Shells

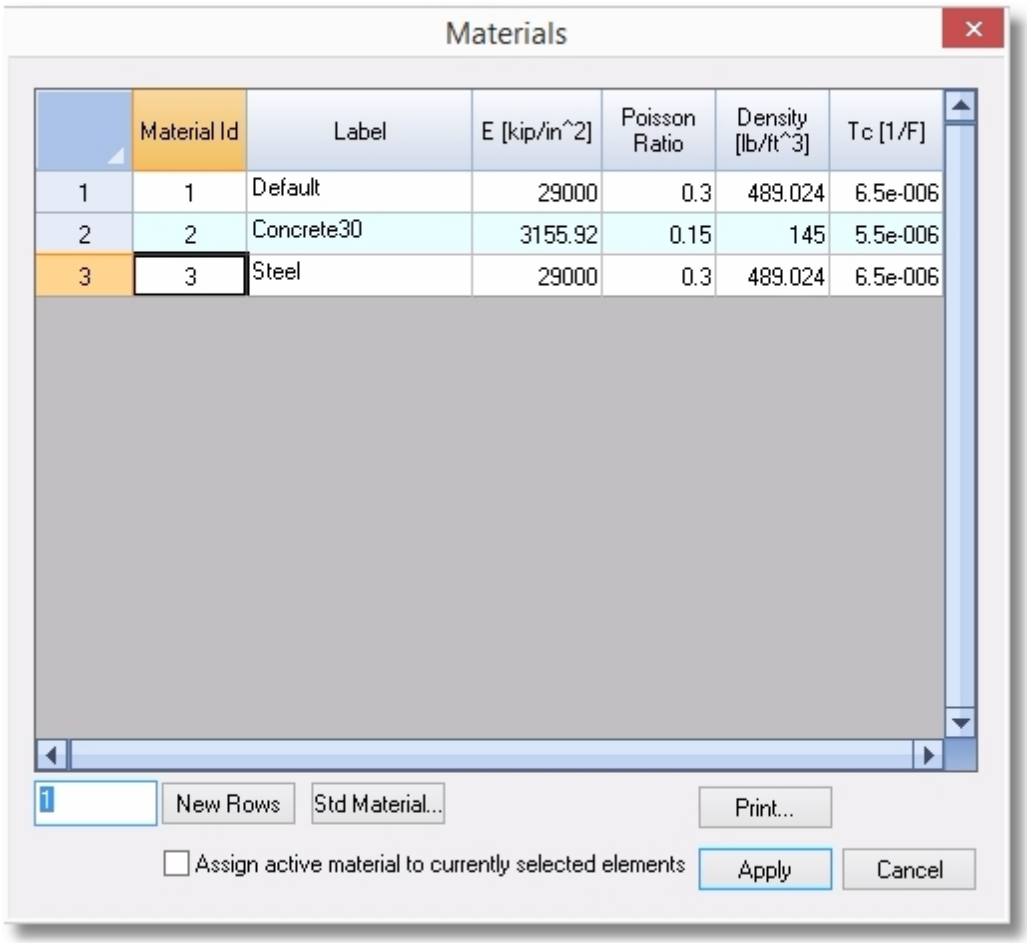
3.3.1 Material

The definition of materials for application to shells is identical to the process of defining materials for use with members. In fact the dialog is the same. The only difference is in some of the ways that the materials can be assigned.

Materials consist of the properties necessary for structural analysis: Young's Modulus, Poisson's Ratio, Density and Coefficient of Thermal Expansion or Contraction. Note that these are not material strengths, which are related to code checking. Those get entered in a different location.

Materials can be defined and assigned using the *Materials* table, which can be accessed by clicking:

- **Create (or Modify) > Member Properties > Materials**
- **Create (or Modify) > Shell Properties > Materials**
- or by clicking **Tables > Materials**.



	Material Id	Label	E [kip/in^2]	Poisson Ratio	Density [lb/ft^3]	Tc [1/F]
1	1	Default	29000	0.3	489.024	6.5e-006
2	2	Concrete30	3155.92	0.15	145	5.5e-006
3	3	Steel	29000	0.3	489.024	6.5e-006

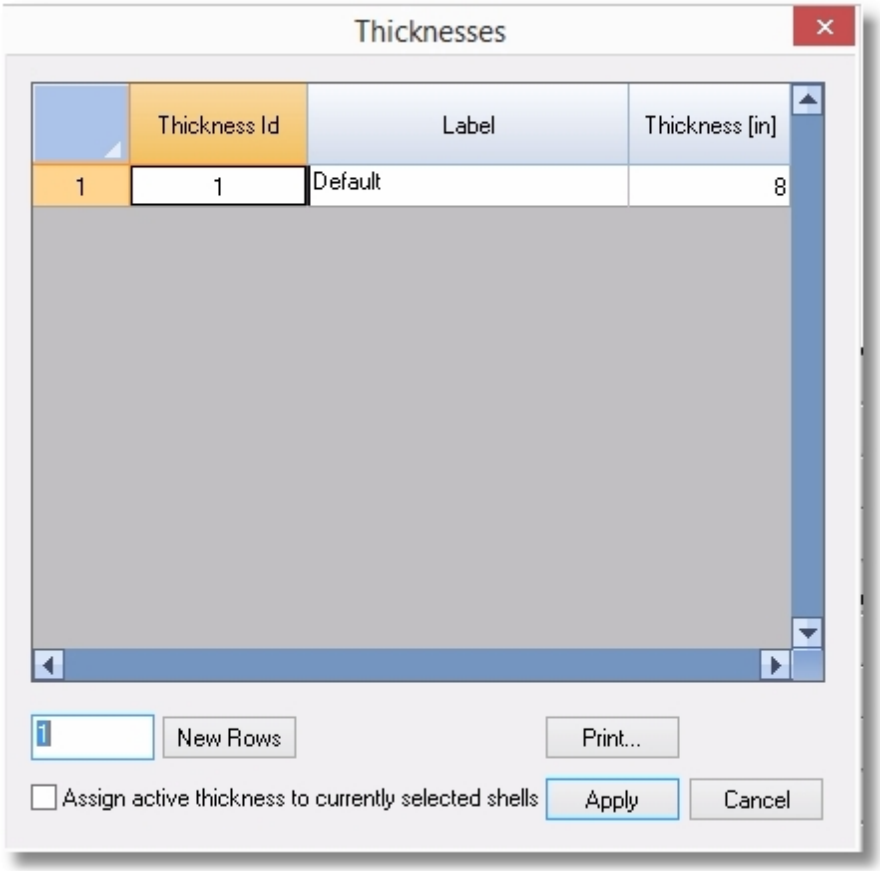
Regardless of how the table is accessed, it allows the user to create material definitions and to assign them to currently selected shells.

Note that there is a Standard Material button on this table that allows for the quick definition of many standard steel and concrete materials.

3.3.2 Thickness

The thickness is the only section property that is required for the analysis of a mesh of shells.

Thickness can be defined and assigned by clicking **Create (or Modify) > Shell Properties > Thicknesses**, or by clicking **Tables > Thicknesses**. Either method opens the *Thicknesses* dialog where shell thicknesses can be defined and assigned to the selected shells.



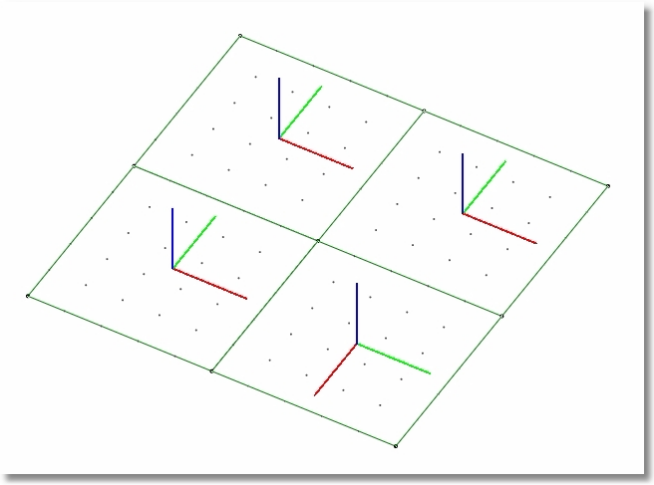
3.3.3 Local Angle

Local angle is a way to rotate the local coordinate system of a shell about its local z axis. It can be useful for situations such as a mesh has been created, but the orientations of some of the shells differ. Having a group of shells oriented the same way can facilitate the interpretation of output such as moment about the local y axis.

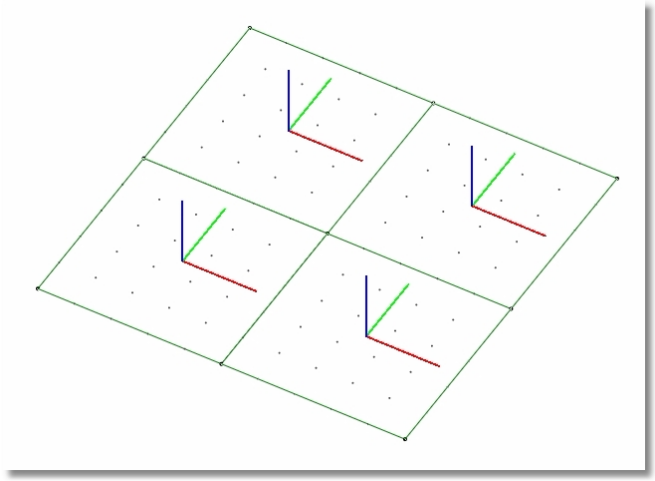
Remember that each shell has a local coordinate axis system. The local z axis is always perpendicular to the shell. To visualize the way a positive local angle will affect a particular shell, point your right thumb in the direction of the local z axis. The natural curl of your right

fingers indicates the direction of rotation of the shell's local coordinate system when subjected to a positive local angle.

To assign a local angle to a shell, select the shell and then click **Create (or Modify) > Member Properties > Local Angles**. Either way, the *Element Local Angle* dialog opens. It allows for the specification of an angle to be applied to selected shells. (It also can be applied to members, which is covered in the section on Properties for Members.)

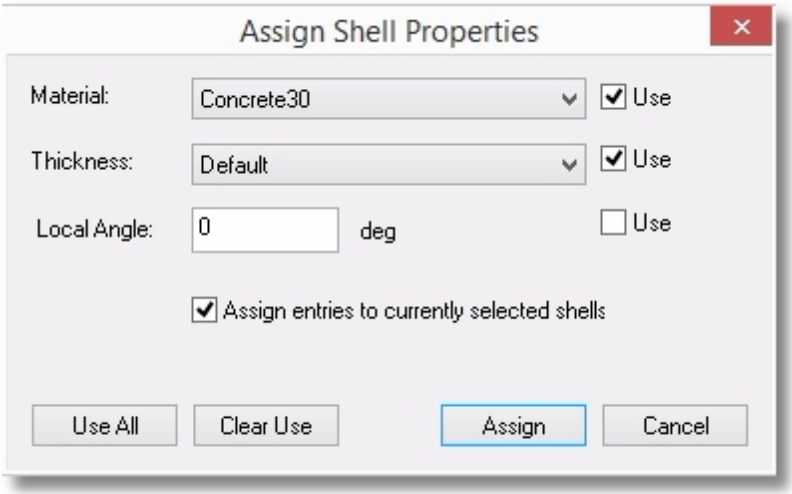


leads to



3.3.4 Shell Properties

We have already seen how to define Material, Thickness, and Local Angle and how to assign each of these to a shell. But sometimes it may be easier to define the necessary properties and then assign all of them to one or more shells all at once. This is where the **Create (or Modify) > Shell Properties > Shell Properties** command can be very useful. It opens the *Shell Properties* dialog, which allows you to select desired shells and then assign Material, Thickness, and Local Angle to the selected shells all in one step.



Part IV

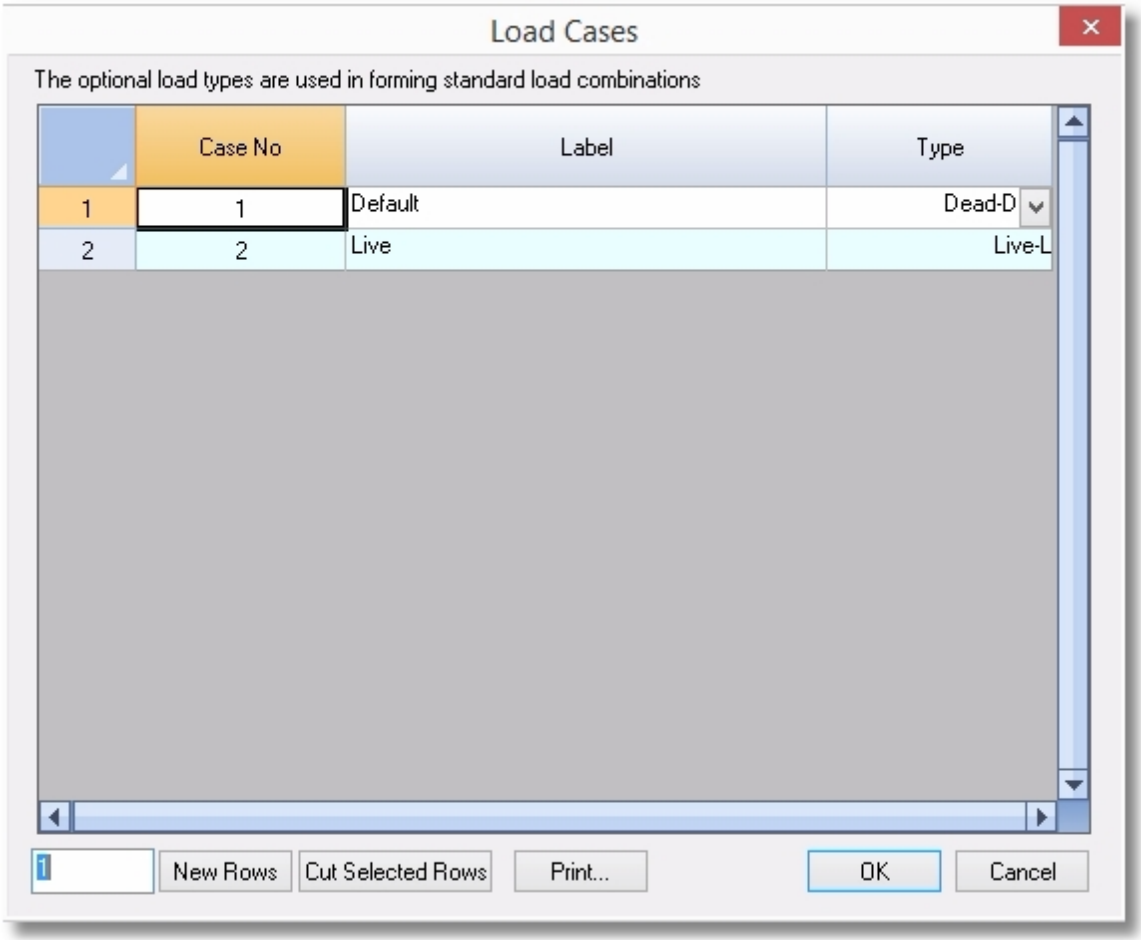
4 Working with Loads

4.1 Load Cases

Think of Load Cases as the individual load types that get combined in the load combinations.

So a typical model will probably have Dead, Live, Wind, Seismic, etc. Other available load types include Roof Live and Snow.

To define the Load Cases that will be used in a model click **Create > Load Cases**. The *Load Cases* dialog opens and is pre-populated with a load case named Default. You can use this case if you want, but you can't change the name "Default". If you prefer, just create all the load cases you want and completely ignore the presence of the Default load case.



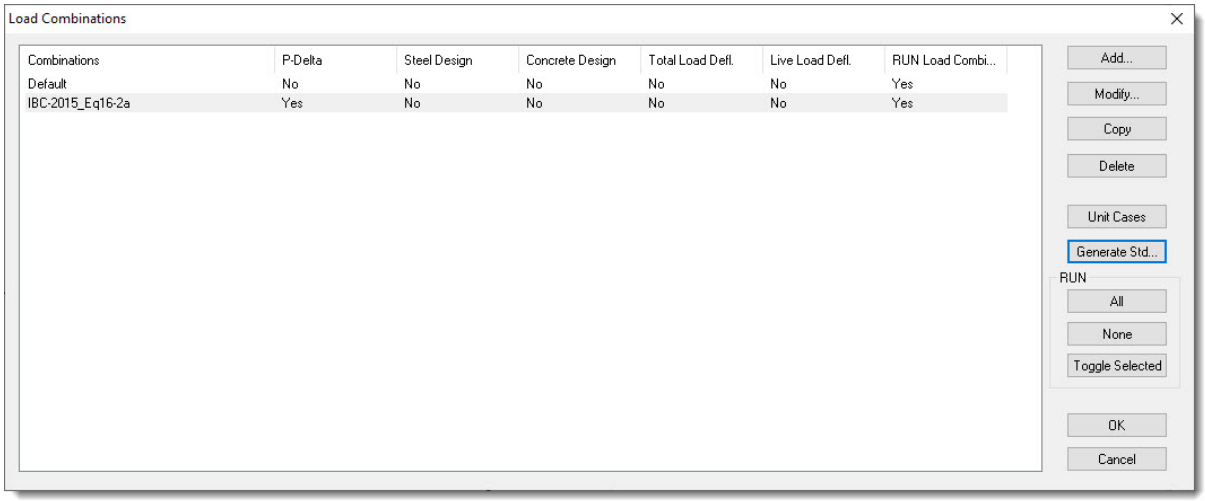
For each Load Case that you create, give it a recognizable name, and select the Type of load at the far right.

The Load Cases that you create here will be used when you start assigning load items in subsequent steps, because any load item that is added to a model *must* be assigned to one Load Case.

4.2 Load Combinations

Load Combinations specify the various ways in which Load Cases are to be summed and applied to the model.

To access the Load Combinations dialog click **Create > Load Combinations**. The *Load Combinations* dialog opens.



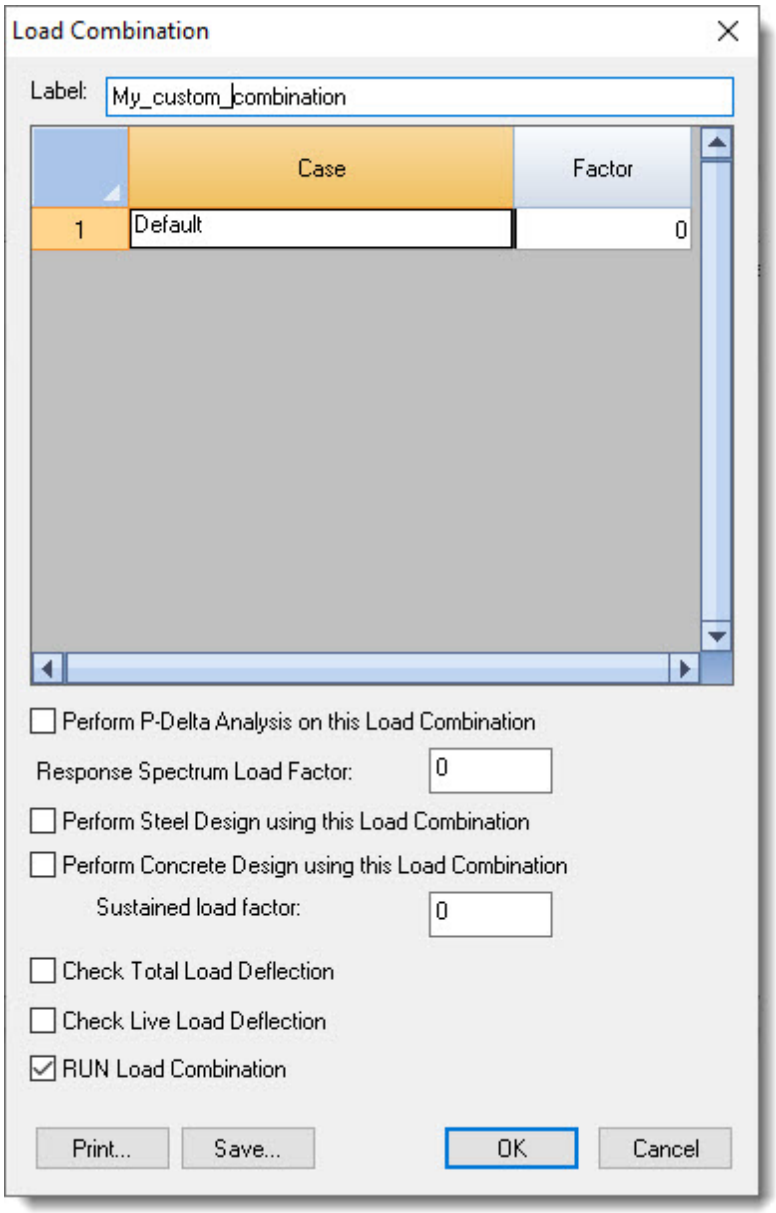
A quick glance at the dialog shows that it controls quite a few things. In addition to allowing combinations to be created (which we will see momentarily), it also controls whether a particular load combination is or is not to be run in and analysis, as well as whether or not the combination is to be considered for:

- P-Delta Analysis
- Steel Design
- Concrete Design
- Total Load Deflection check
- Live Load Deflection check.

Now let's explore how we can define a load combination and how we can set the various options that control what the load combination is used for.

To add a single load combination at a time:

Click the Add button and note that the Load Combination dialog opens.



It allows the label to be specified at the top of the window. Then it lists the individual Load Cases that have been created so far and allows the Factor to be specified for each. To exclude a Load Case from a particular Load Combination, just leave the corresponding factor set to zero.

Once the Factors have been set, move to the options in the bottom potion of the window. This is where you will indicate whether the Load Combination is to be run, as well as whether it is to be used for:

- P-Delta Analysis
- Steel Design
- Concrete Design
- Total Load Deflection check
- Live Load Deflection check.

Once you make your choices and click OK, the focus returns to the Load Combinations window, and the newly defined combination appears at the bottom of the list.

To modify an existing load combination:

Click on the desired Load Combination and then click the Modify button. The Load Combination dialog opens and allows the selected combination and associated settings to be modified and saved.

To copy an existing load combination:

Click on the desired Load Combination and then click the Copy button. The program will create a copy of the selected load combination and append "_Copy" to the end of the original label. This can be useful when creating similar load combinations where it is fastest to copy and edit, rather than starting each from scratch.

To delete an existing load combination:

Click on the desired Load Combination and then click the Delete button. The program will delete the selected load combination. To select multiple combinations hold down the CTRL key while clicking.

To create a load combination for each of the defined load cases:

Sometimes it is useful to have a load combination for one or more of the defined load cases. An example is when the value of Beta d needs to be calculated for concrete design, which requires the definition of a load combination of 1.0 x sustained loads. To automatically have the program create a load combination for each defined load case, simply click the Unit Cases button. Feel free to delete any unneeded combinations.

To generate standard load combinations based on ACI or AISC:

To generate standard load combinations based on ACI or AISC, click the Generate Standard button. It offers a dropdown list box to select the desired load combination set. Then individual combinations have checkboxes so you can pick or skip individual combinations.

To control which Load Combinations are and are NOT run:

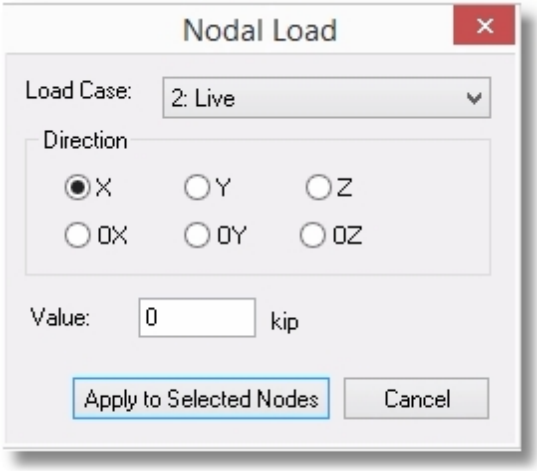
Use the commands in the RUN category to set which load combinations you do and do not want run in any particular analysis.

4.3 Nodal Loads

Nodal Loads are concentrated forces or moments that are applied to nodes.

Graphical Method

To apply nodal loads graphically, select the nodes that are to receive the load. Then click **Create > Draw Loads > Nodal Loads**. The *Nodal Load* dialog opens.



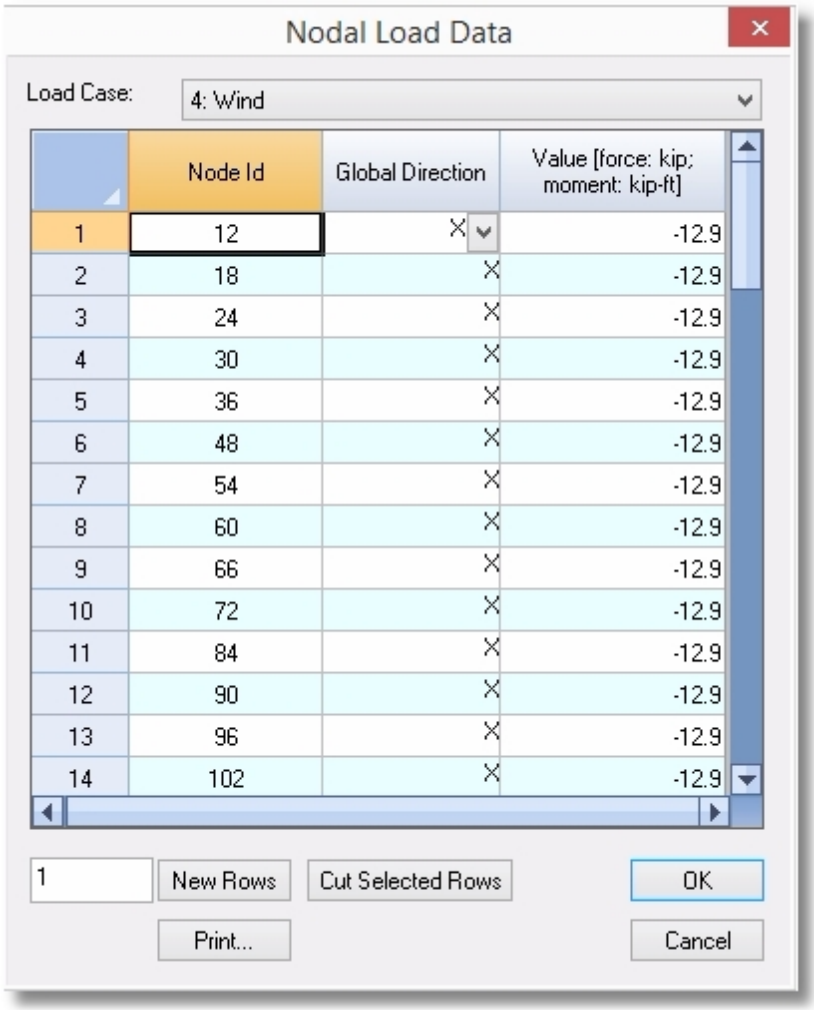
Select which Load Case with which the new load should be associated.

Select the direction, keeping in mind that these are with respect to the global coordinate axis system and the directions that start with "O" represent moments *about* the named global axis.

Specify the value and apply the load.

Tabular Method

Click **Tables > Nodal Loads** to open the *Nodal Loads* table.



Note that the Load Case dropdown list box is shown at the top of the table. This controls the load case for which loads will be shown.

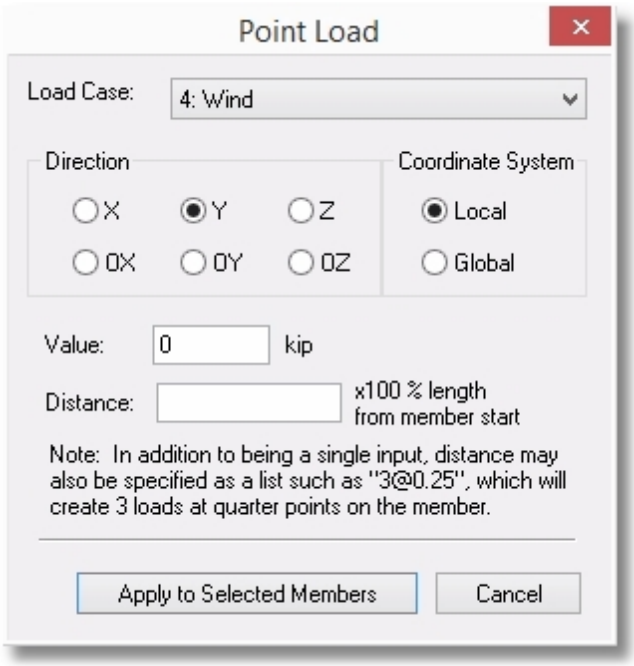
The table allows existing loads to be reviewed, edited and deleted. It also allows new loads to be defined.

4.4 Point Loads (on Members)

Point Loads are concentrated forces or moments that are applied to members.

Graphical Method

Select the members that are to receive the load. Click **Create > Draw Loads >Point Loads**. The *Point Load* dialog opens.



Specify the Load Case with which the new load will be associated.

Select the direction of the load and indicate whether the direction selection is to be interpreted as being in the local or global coordinate axis system.

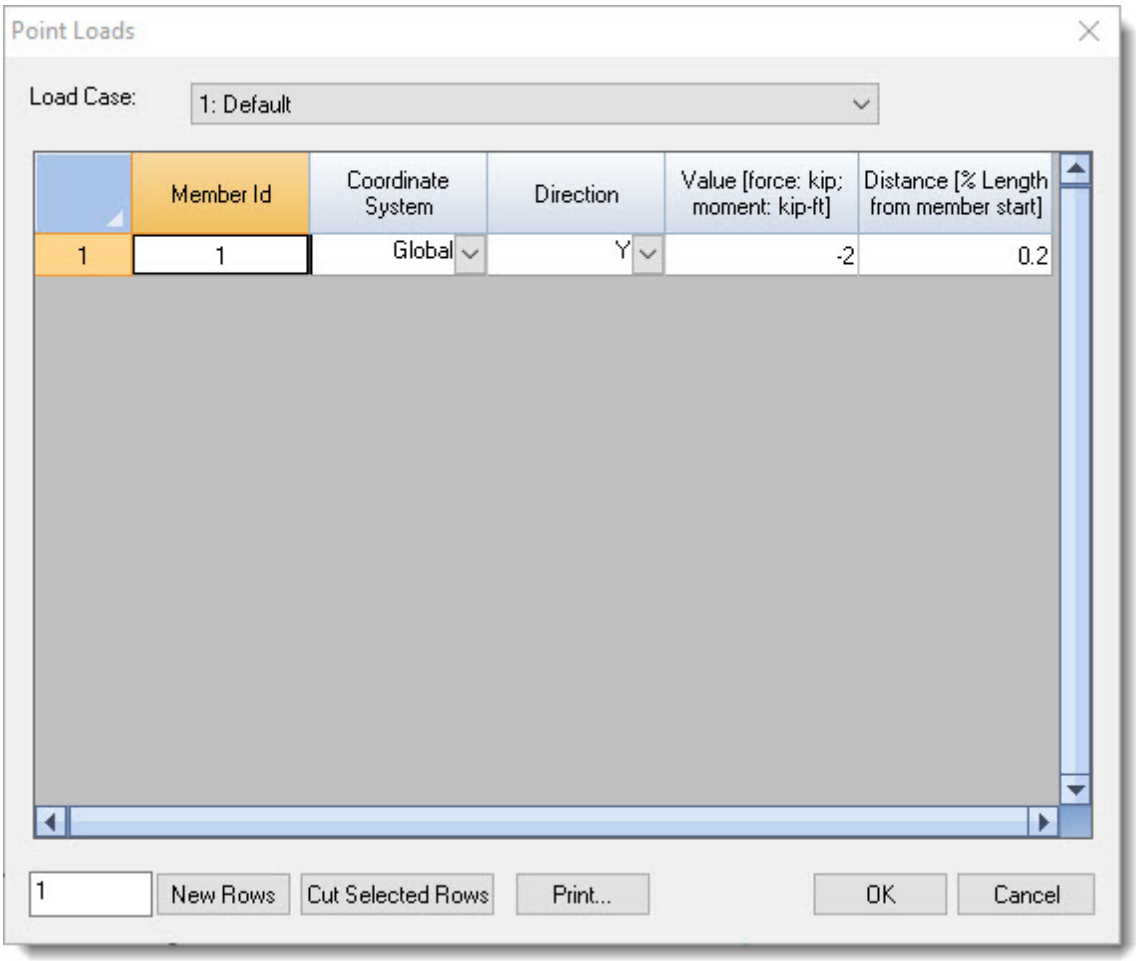
Specify the magnitude of the force.

Specify the distance in the form of 0 for the starting end, 0.5 for the midpoint, 1.0 for the ending end, etc.

Tip: Multiple Point Loads may be defined in a single use of this command by using the following syntax in the Distance input: "3@0.25", which would place loads on the quarter points of the assigned members.

Tabular Method

Click **Tables > Point Loads Data** to open the *Point Loads* table.



Note that the Load Case dropdown list box is shown at the top of the table. This controls the load case for which loads will be shown.

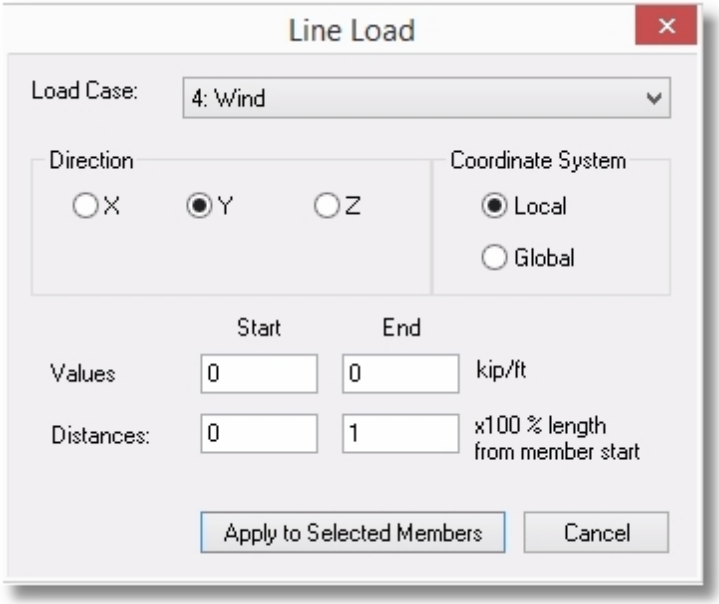
The table allows existing loads to be reviewed, edited and deleted. It also allows new loads to be defined.

4.5 Line Loads (on Members)

Line Loads are distributed forces that are applied to members.

Graphical Method

Select the members that are to receive the load. Click **Create > Draw Loads > Line Loads**. The *Line Load* dialog opens.



Specify the Load Case with which the new load will be associated.

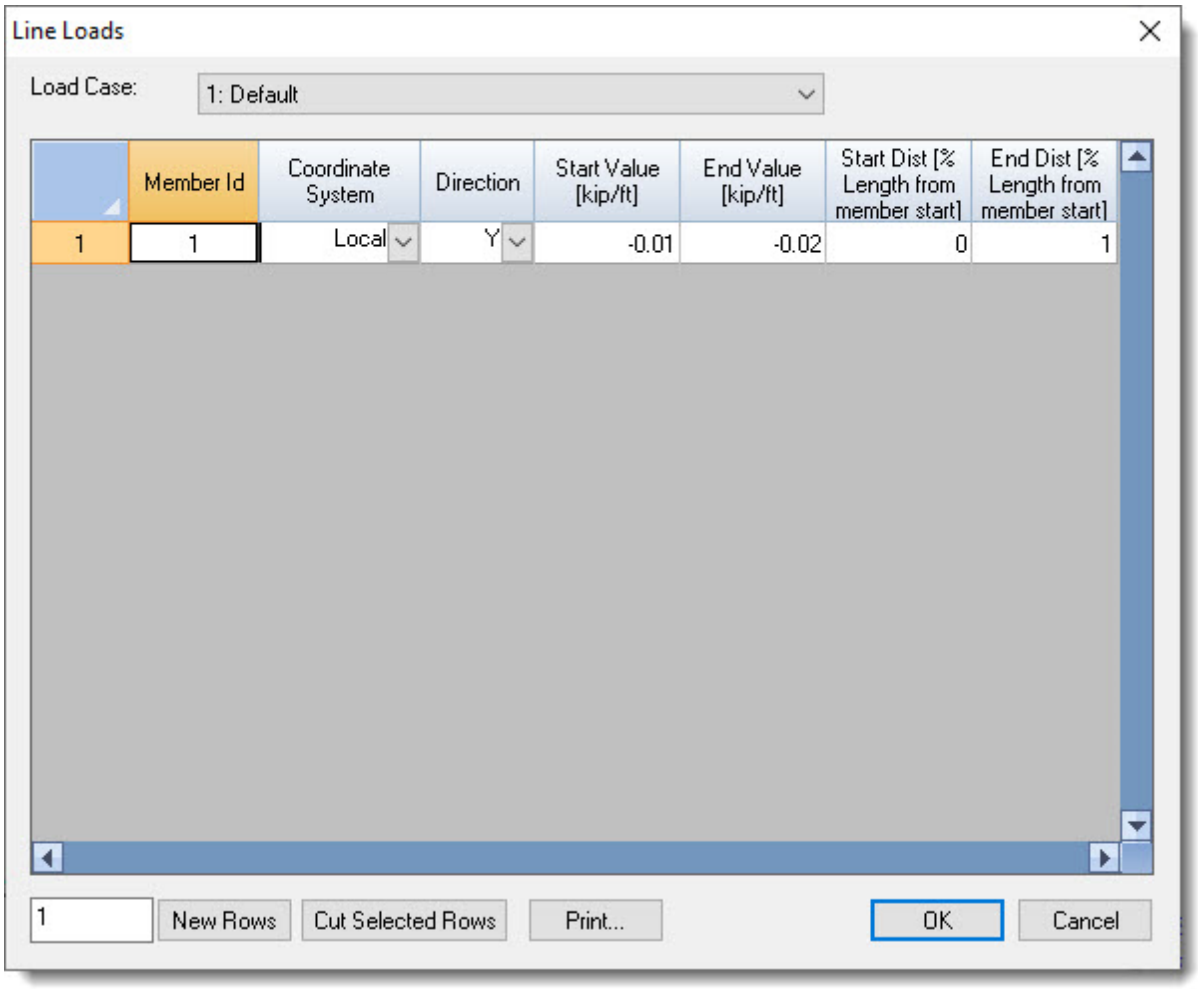
Select the direction of the load and indicate whether the direction selection is to be interpreted as being in the local or global coordinate axis system.

Specify the magnitude of the force at the start and at the end of the Line Load.

Specify the distances to the start and end of the force (referenced from the starting end of the member) in the form of 0 for the starting end, 0.5 for the midpoint, 1.0 for the ending end, etc.

Tabular Method

Click **Tables > Line Loads** to open the *Line Loads* table.



Note that the Load Case dropdown list box is shown at the top of the table. This controls the load case for which loads will be shown.

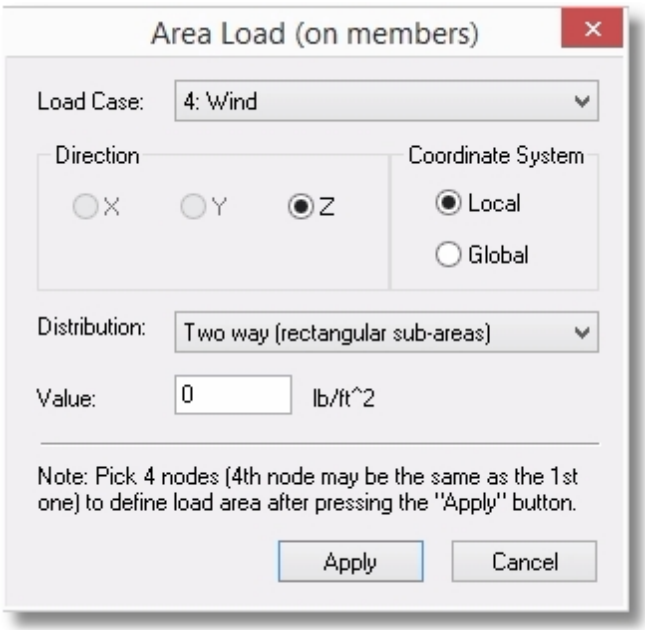
The table allows existing loads to be reviewed, edited and deleted. It also allows new loads to be defined.

4.6 Area Loads (on Members)

Area Load is a way to automatically apply load to members, and it can be convenient for loading many members in an area.

Graphical Method

Click **Create > Draw Loads > Area Loads** to open the *Area Load (on members)* dialog.

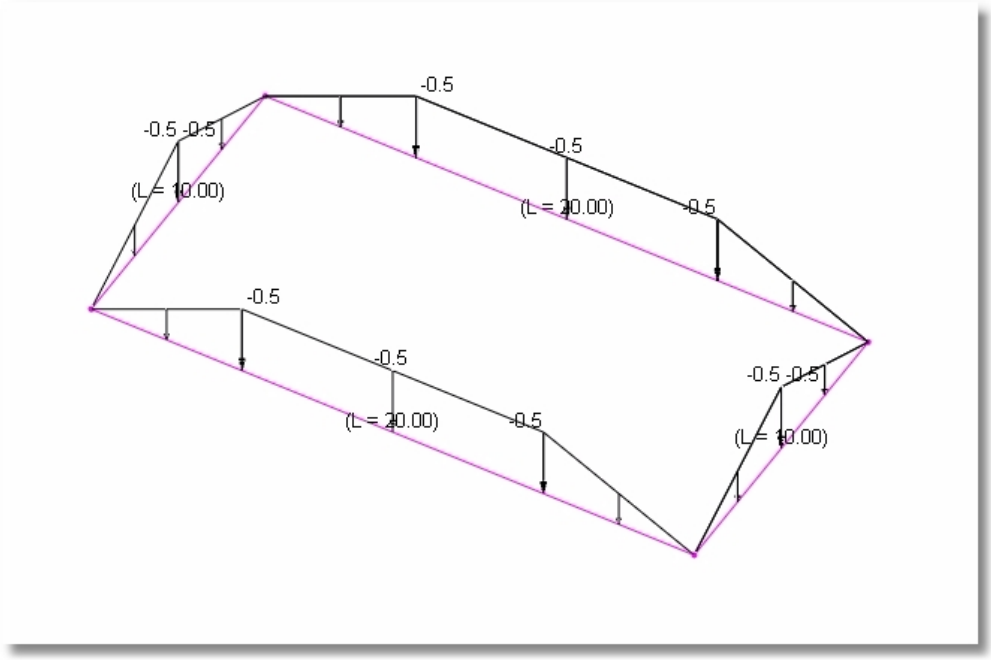


Specify the Load Case with which the new load will be associated. Select the direction of the load and indicate whether the direction selection is to be interpreted as being in the local or global coordinate axis system.

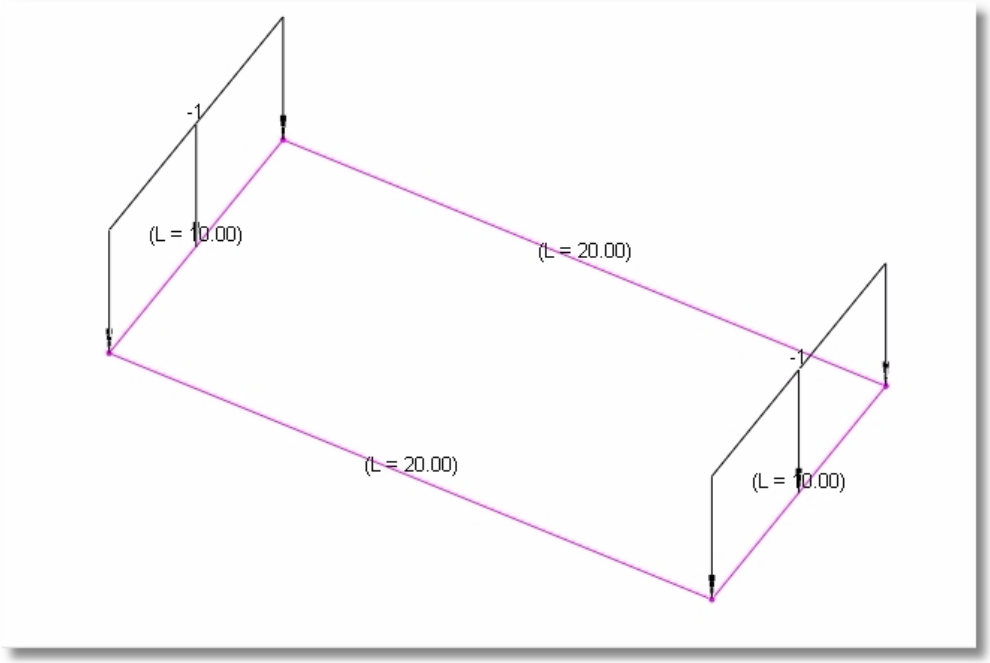
Tip: When the Coordinate System is set to Local, only the Z direction is available.

The Distribution dropdown provides many options. They are best explained graphically (below) by the application of 100 psf load to a rectangular bay with the dimensions shown:

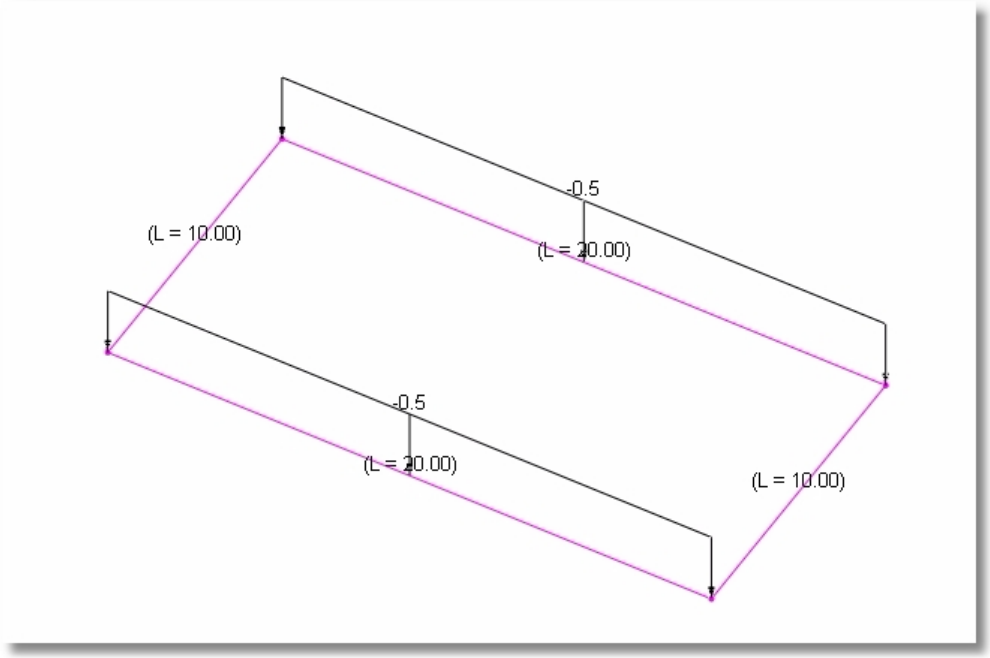
- Two-way (rectangular sub-areas):



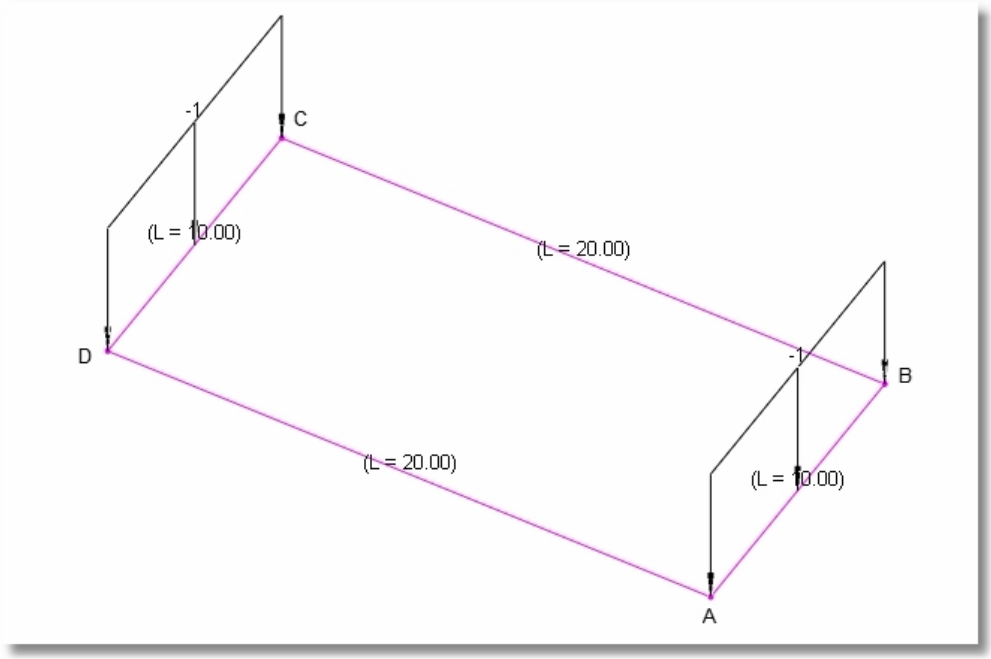
- Short sides (rectangular sub-areas):



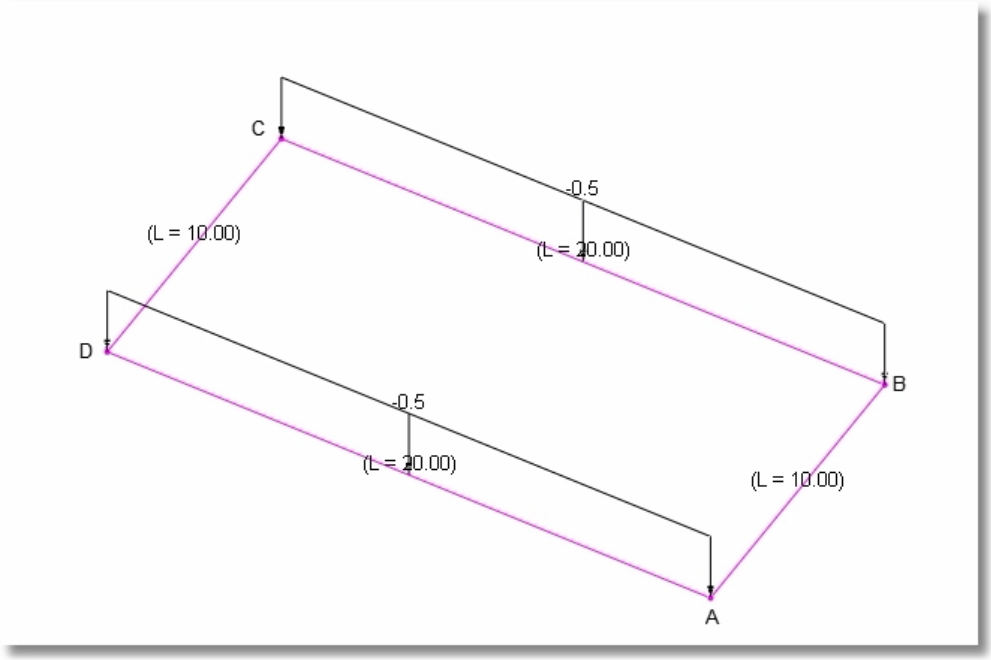
- Long sides (rectangular sub-areas):



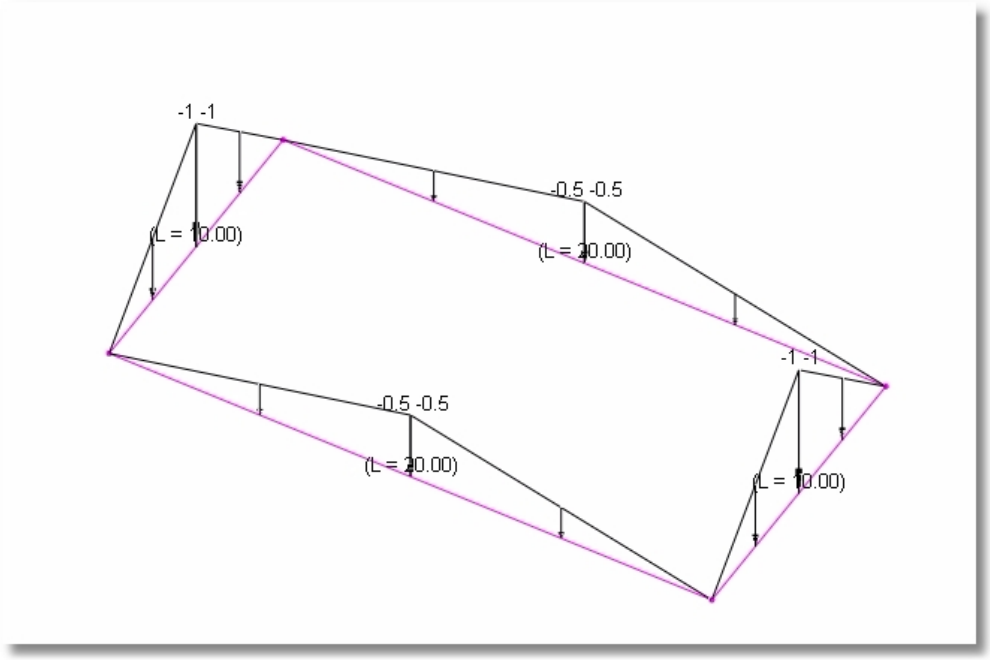
- AB and CD sides (rectangular sub-areas):



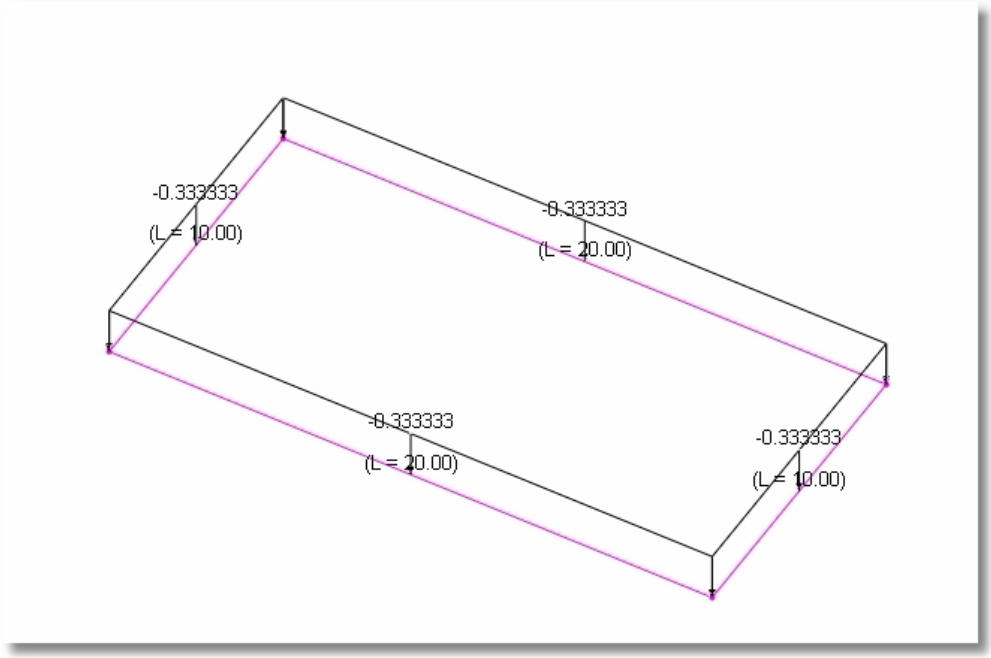
- BC and AD sides (rectangular sub-areas):



- Centroid-based:



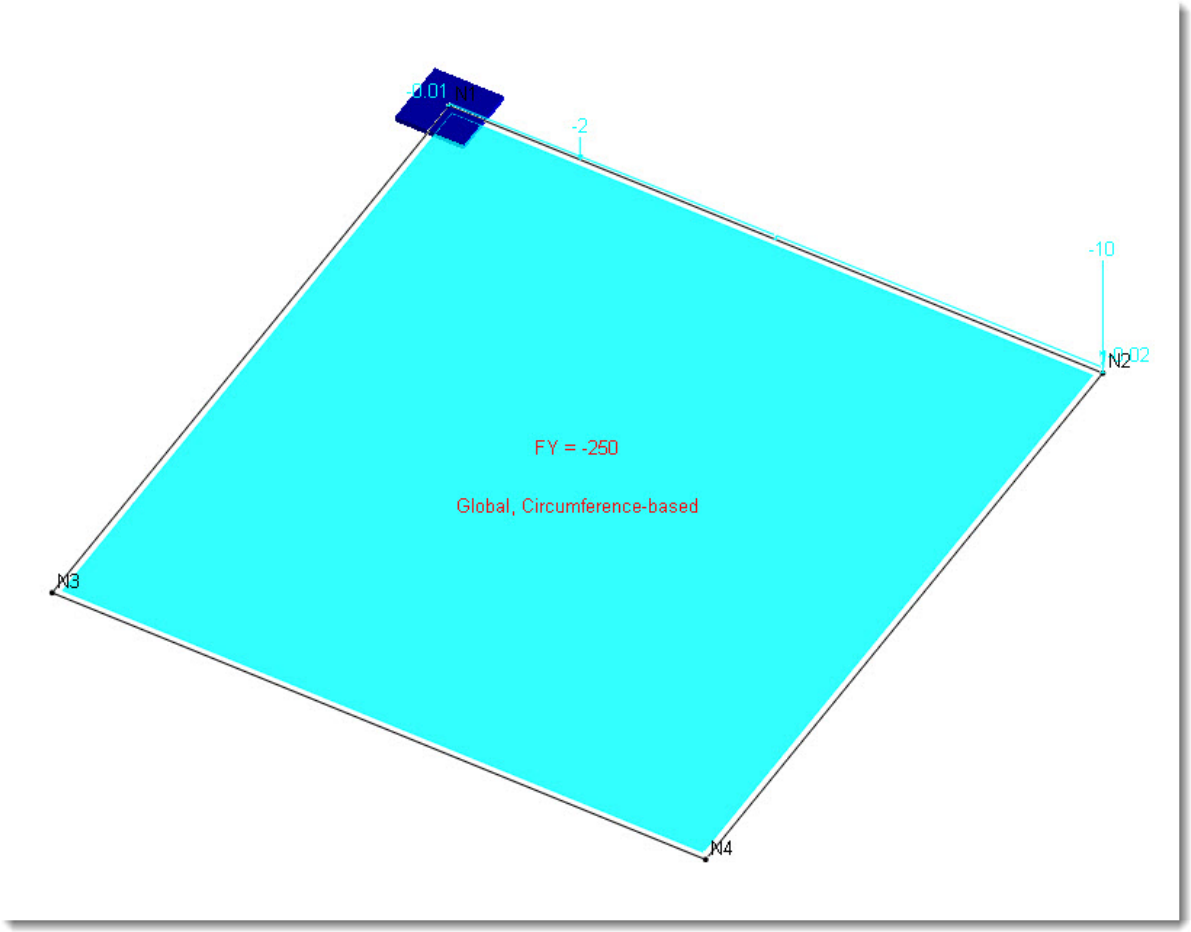
- Circumference-based: (Total load on bay is distributed uniformly along perimeter of bay.)



Enter the value and remember to use the proper algebraic sign for the direction of loading.

After clicking Apply, click on four nodes to define the perimeter of the loaded area. Keep in mind that for some of the load Distribution options, the order of selection of the corners establishes the directionality of the distribution.

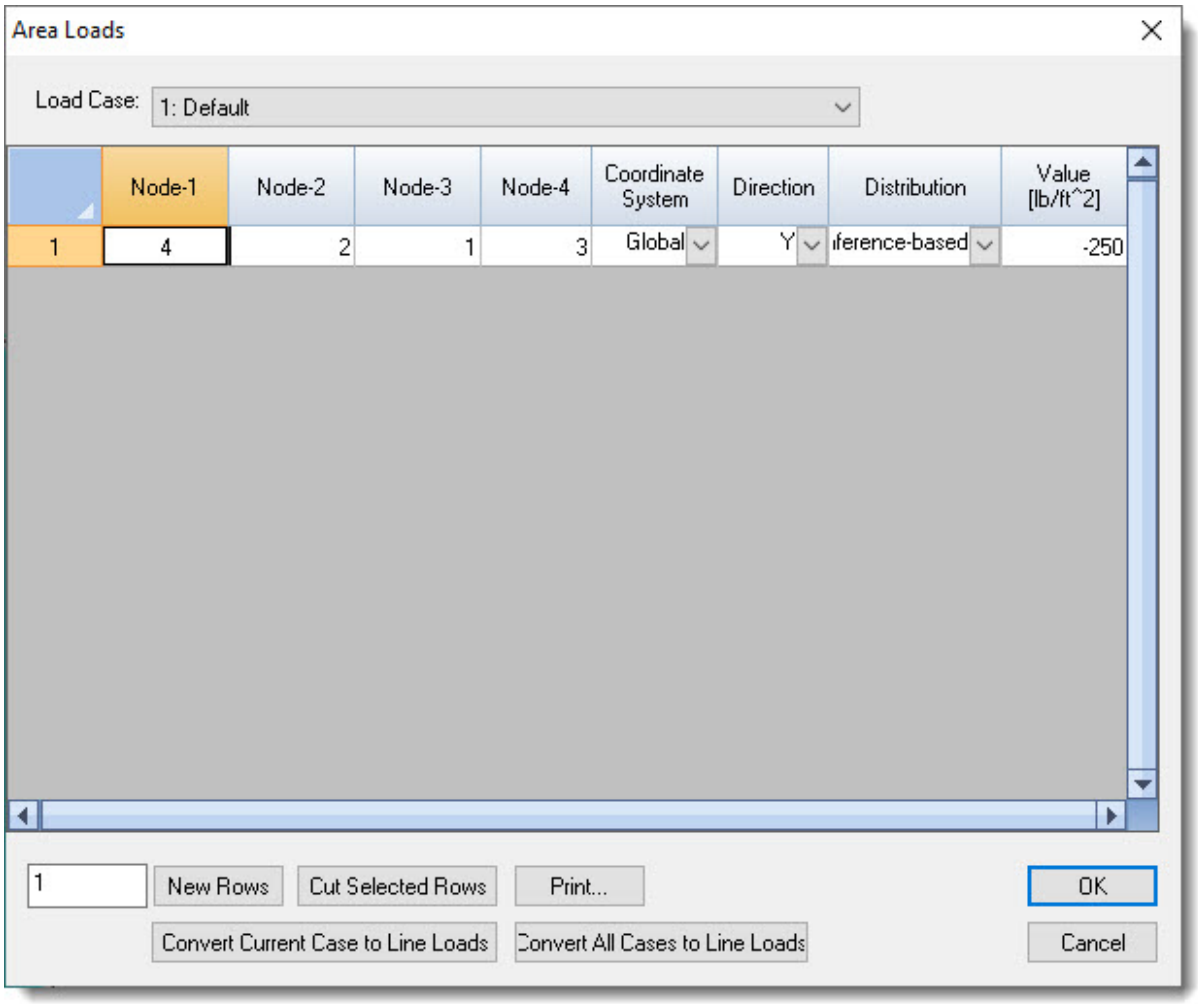
Here is what an area load will look like when it is first applied:



Tip: Keep in mind that an Area Load can be dissolved into a series of Line Loads on members by clicking **Create > Generate Loads > Convert Area Loads to Line Loads**. This may be helpful for visualizing loads like they have been shown in the screen captures above.

Tabular Method

Click **Tables > Area Loads** to open the *Area Loads* table.



Note that the table has a Load Case dropdown list box at the top.

This table can be useful for reviewing and editing or deleting previously defined Area Loads, but it can also be used to define new area loads by specifying the four perimeter nodes, the Coordinate System and the Direction, the Distribution method and the load intensity Value.

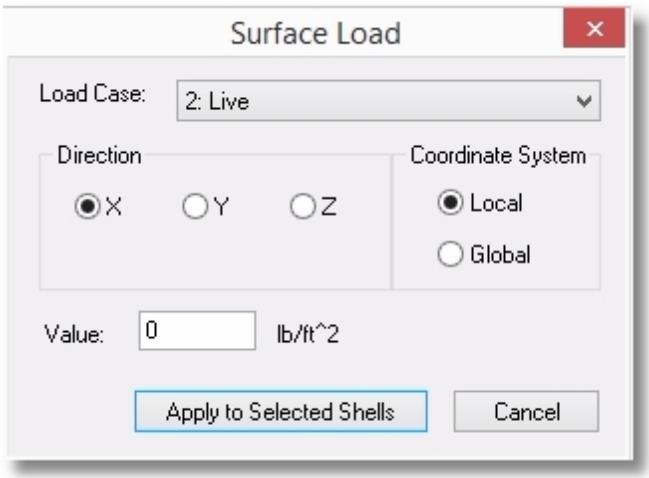
This table also has buttons near the bottom to Convert Current Case to Line Loads and to Convert All Cases to Line Loads. Area Loads don't *have* to be converted to Line Loads in order to have an effect on the model. But if they are converted to Line Loads, they are no longer visible in the *Area Loads* table, and will then be displayed in the *Line Loads* table.

4.7 Surface Loads (on Shells)

Surface Loads are uniformly distributed loads that are applied to shells.

Graphical Method

Select the shells to load. Then click **Create > Draw Loads > Surface Loads** to open the *Surface Load* dialog.



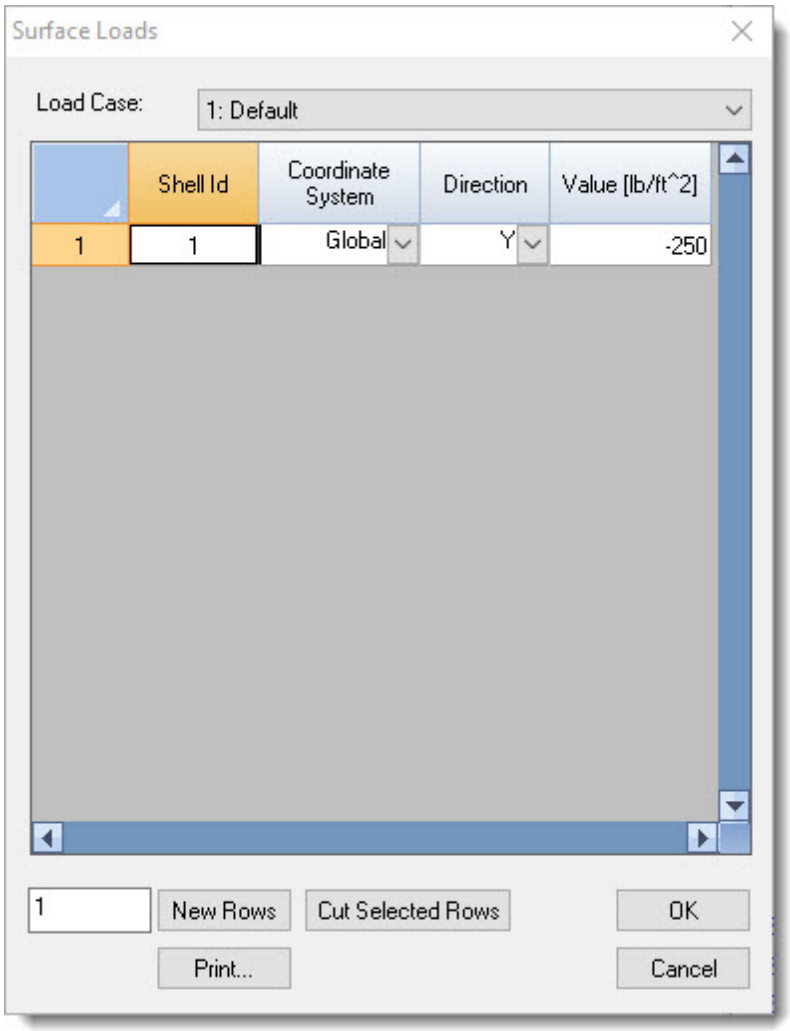
Specify the Load Case with which the new load will be associated.

Select the direction of the load and indicate whether the direction selection is to be interpreted as being in the local or global coordinate axis system.

Enter the value and remember to use the proper algebraic sign for the direction of loading.

Tabular Method

Click **Tables > Surface Loads** to open the *Surface Loads* table.



Note that the table has a Load Case dropdown list box at the top.

This table can be useful for reviewing and editing or deleting previously defined Surface Loads, but it can also be used to define new surface loads by specifying the shell ID, the Coordinate System and the Direction, and the load intensity Value.

4.8 Self Weight

Self Weight is a command that instructs the program to consider the self weights of entities in the model.

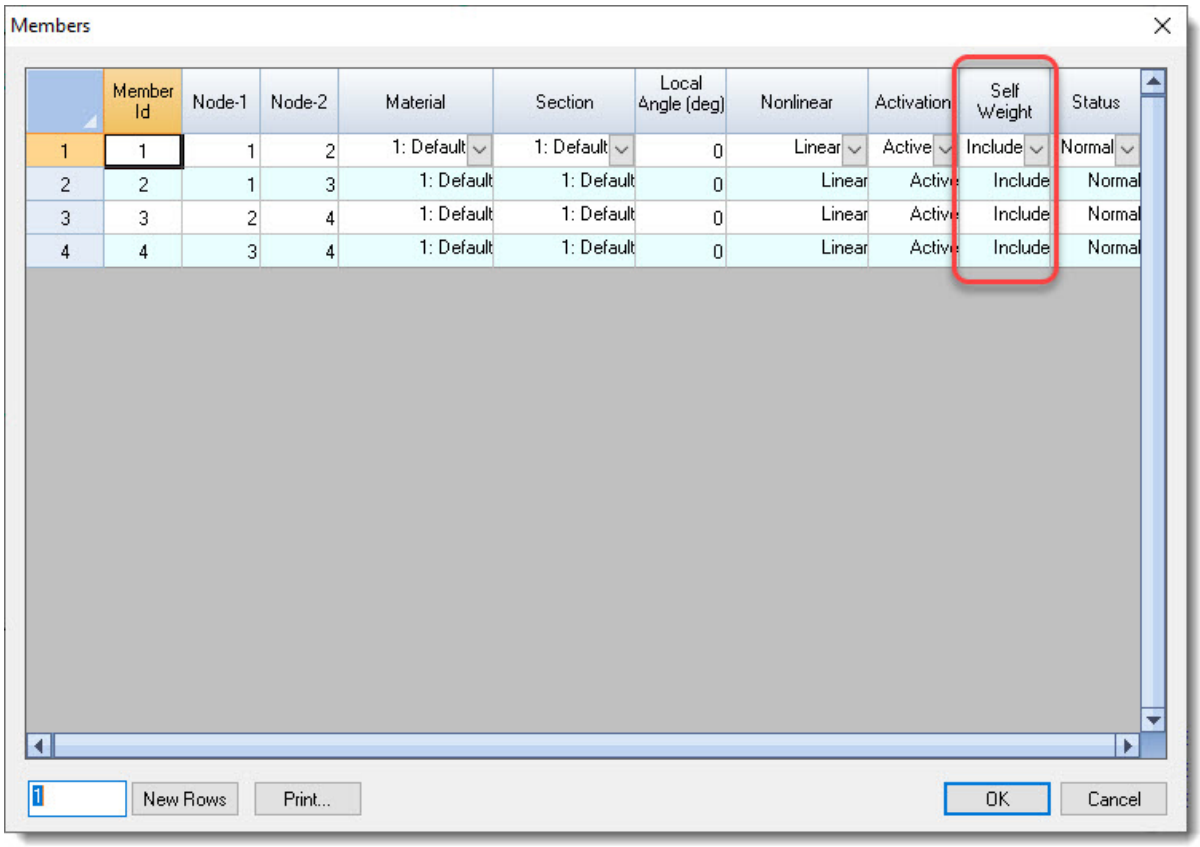
The command can be applied by clicking **Create > Draw Loads > Self Weights**, which opens the *Self Weights* dialog.

The assignment starts by selecting the load case with which the self weight load is to be associated. Select the global direction and then specify a multiplier.

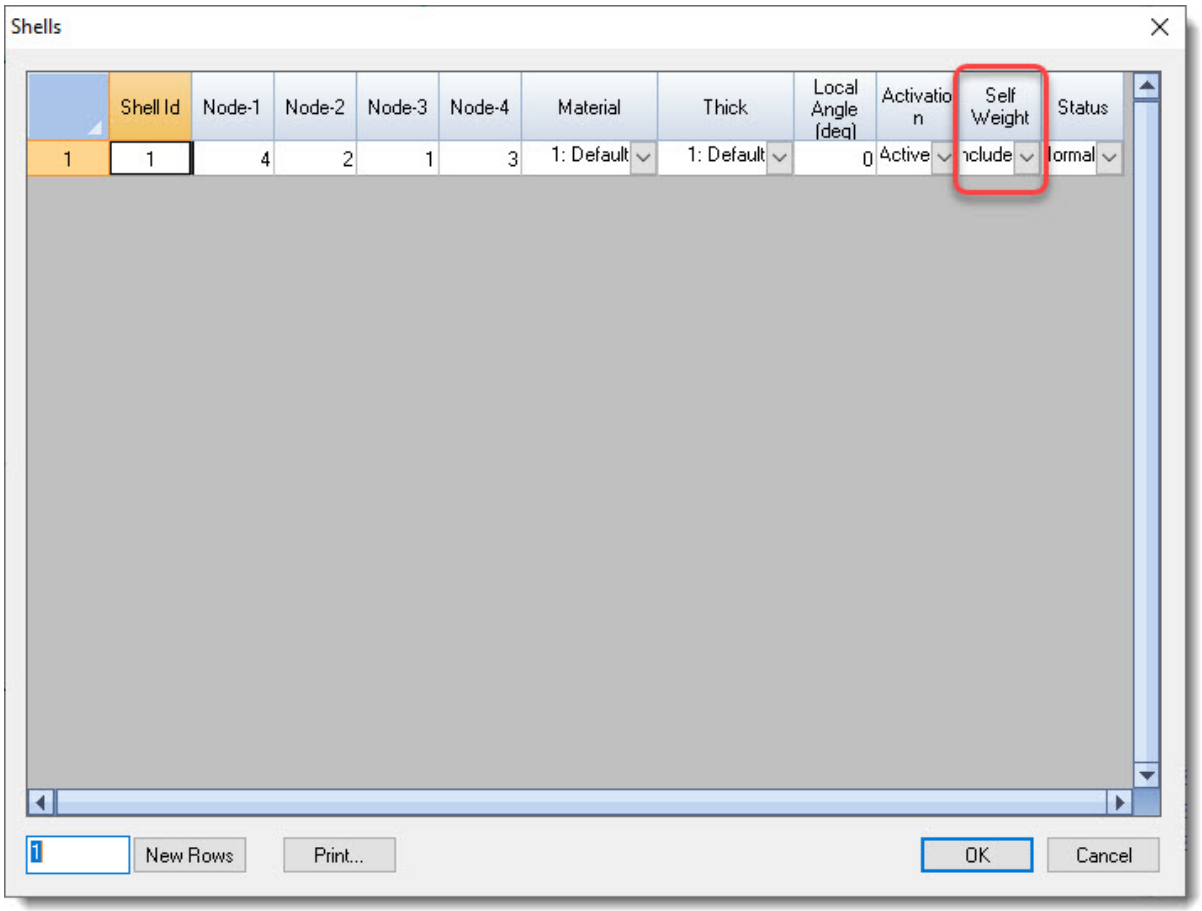
The most common use is Dead Load, Global Y, factor of -1.

Self Weight applies to all elements in the model that have their Self Weight control set to "Include":

To view the Self Weight control for members, open the Members table by clicking **Tables > Members**:

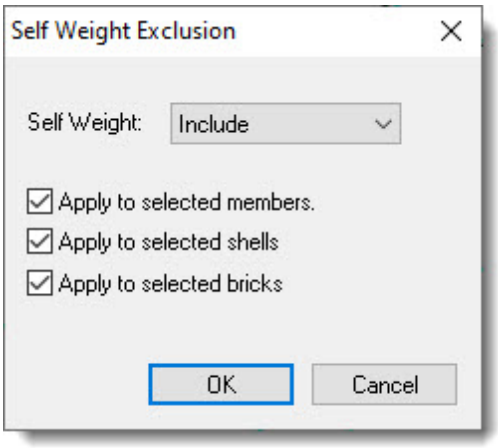


To view the Self Weight control for shells, open the Members table by clicking **Tables > Shells:**



There is an analogous column in the Bricks table, as well.

To remove a Self Weight assignment globally, one option is to use a factor of zero in the multiplier field in **Create > Draw Loads > Self Weights**. Another option is to use the **Create > Draw Loads > Self Weight Exclusion** command.



This method allows the self weight to be excluded for all or for a selected set of members, shells, and or bricks.

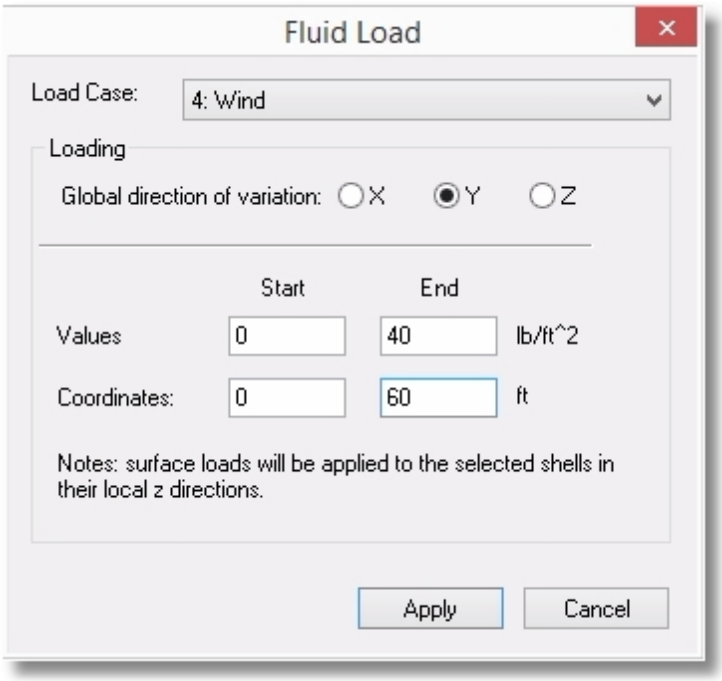
4.9 Fluid Loads (on Shells)

The Fluid Loads command provides a convenient way to apply a linear varying surface load on a selected mesh of shells. The load will be applied in the local z direction of all selected shells, that is, perpendicular to the shell and in the positive or negative local z direction based on the algebraic sign of the load value.

Note that although the load can vary in intensity, it will be discretized such that the load will be uniform on each individual shell.

Graphical Method

The command is accessed by clicking **Create > Generate Loads > Fluid Loads**, which opens the *Fluid Load* dialog.



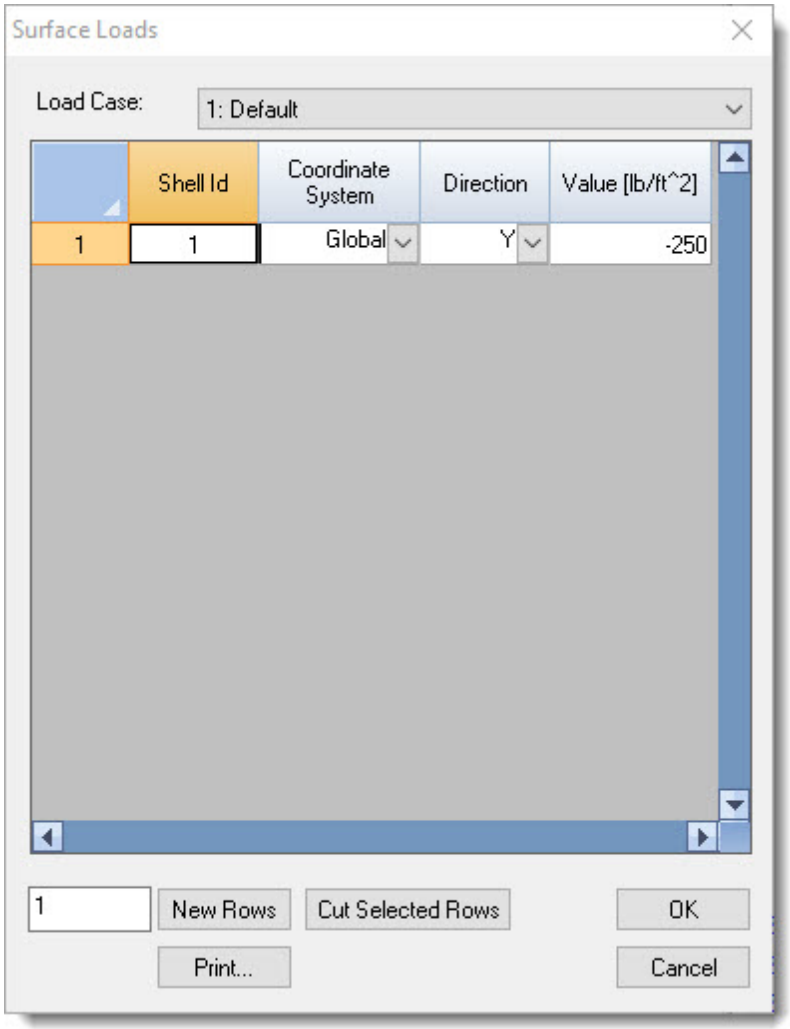
Select the load case with which the Fluid Load is to be associated.

The actual loading is specified as three components: a global direction of variation, a starting coordinate with load intensity and an ending coordinate with load intensity.

Tabular Method

The tables are useful for reviewing and editing the loads produced by the Fluid Loads command, because they get generated as surface loads on shells. But the tables don't have the power to *generate* Fluid Loads of varying magnitudes on multiple shells.

Clicking **Tables > Surface Loads** opens the *Surface Loads* table.



This table displays all loads applied to shells, including those generated by the Fluid Loads command. Any of the data can be edited in this table, and new data can be created by manually inserting new rows. But remember that the real utility of the Fluid Loads command is its ability to create varying loads on many shells automatically.

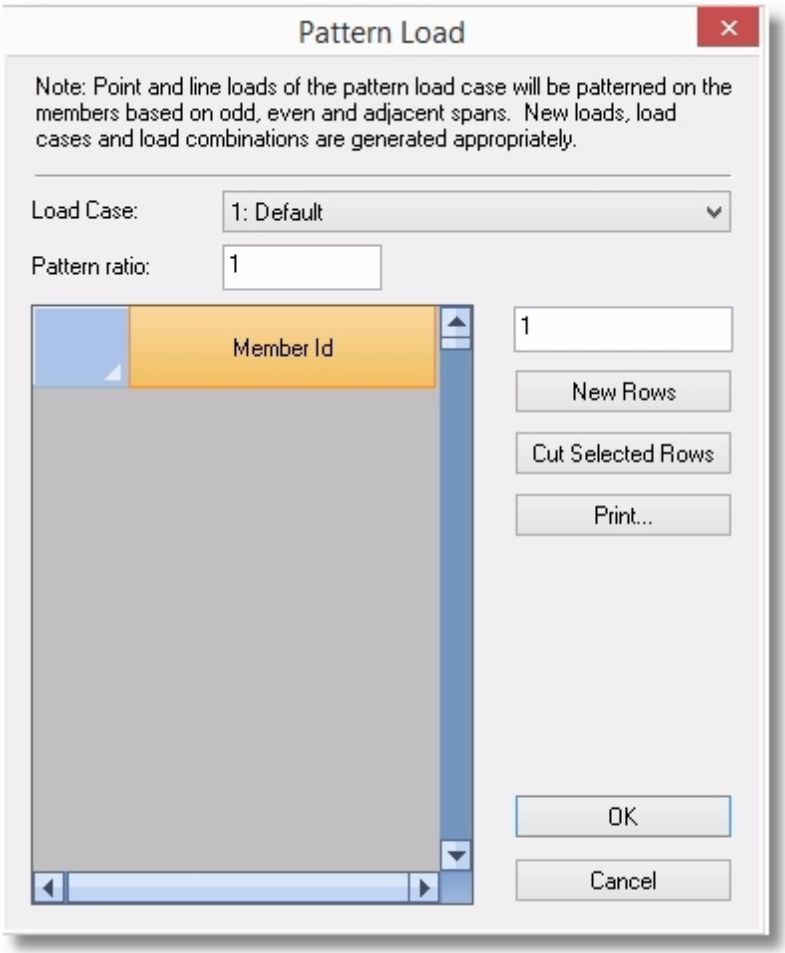
4.10 Pattern Loads

The Pattern Loads command is a way to specify that the loads of a particular load case should be applied in all possible patterns such as odd spans, even spans, and all other potential arrangements to generate maximum positive and negative moment in each span, maximum positive and negative moment at each support as well as maximum shear at each support.

Before applying the Pattern Loads command, be sure that the the load case to be patterned is already created. Also be sure that load combinations have already been defined.

Graphical Method

Select the members whose loads are to be patterned. Click **Create > Generate Loads > Pattern Loads** to open the *Pattern Load* dialog.



Select the load case whose loads are to be patterned. Enter the Pattern Ratio (ACI 318-05 specifies 0.75).

Click OK and note that the screen displays a graphic representation of the patterned loads.

Click **Create > Load Cases** and note that the table displays new load cases that represent the patterned loads.

Click **Create > Load Combinations** and note that the existing Load Combinations have been expanded to include the newly generated pattern loads.

4.11 Moving Loads

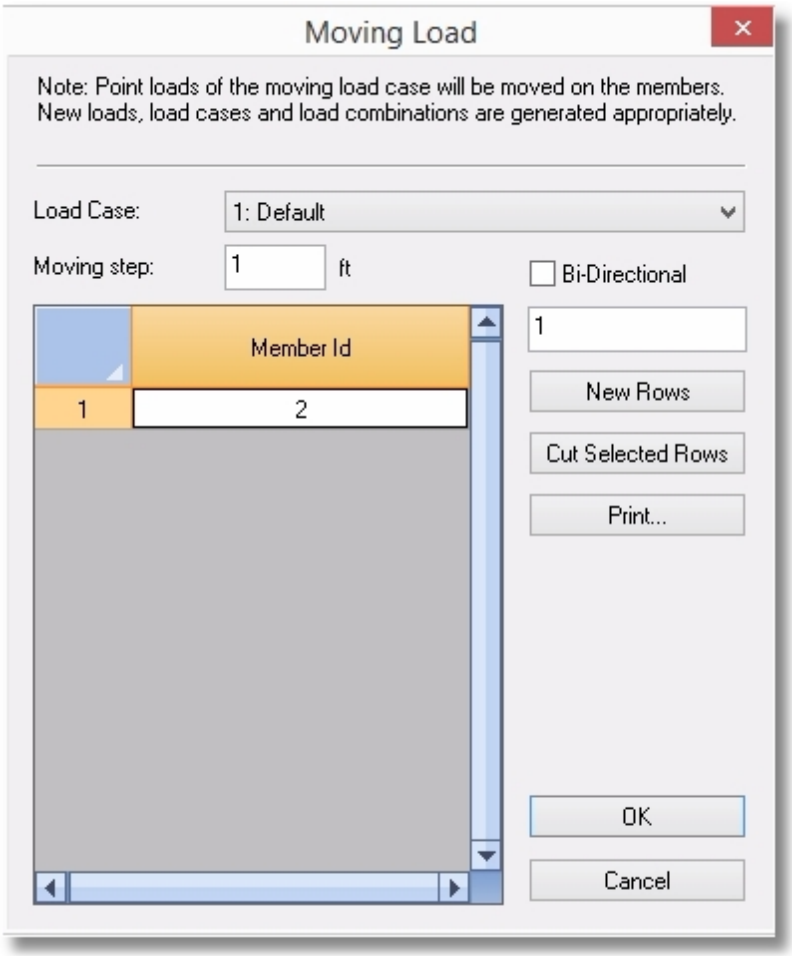
The Moving Loads command is a way to specify that a load (or a pattern of loads) on selected collinear members is to be moved incrementally and analyzed at each location.

Before applying the Moving Loads command, be sure that the load case to be moved is already created. Also be sure that load combinations have already been defined.

Graphical Method

Select a member that already has the desired arrangement of concentrated loads on it. Also select the connected collinear members that the load arrangement is to travel across.

Click **Create > Generate Loads > Moving Loads** to open the *Moving Load* dialog.

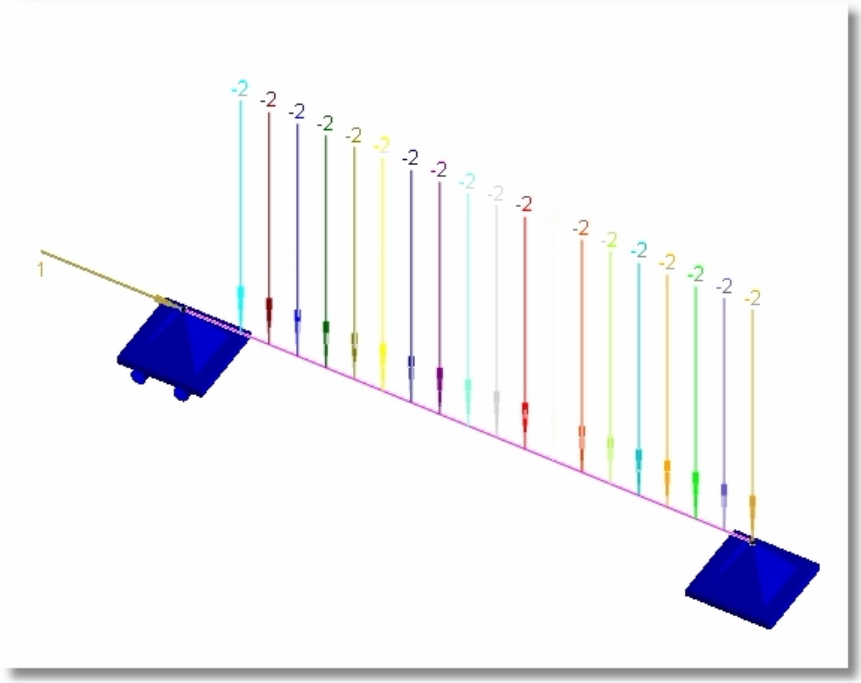


The selected members are listed in the table, and new members can be added if necessary.

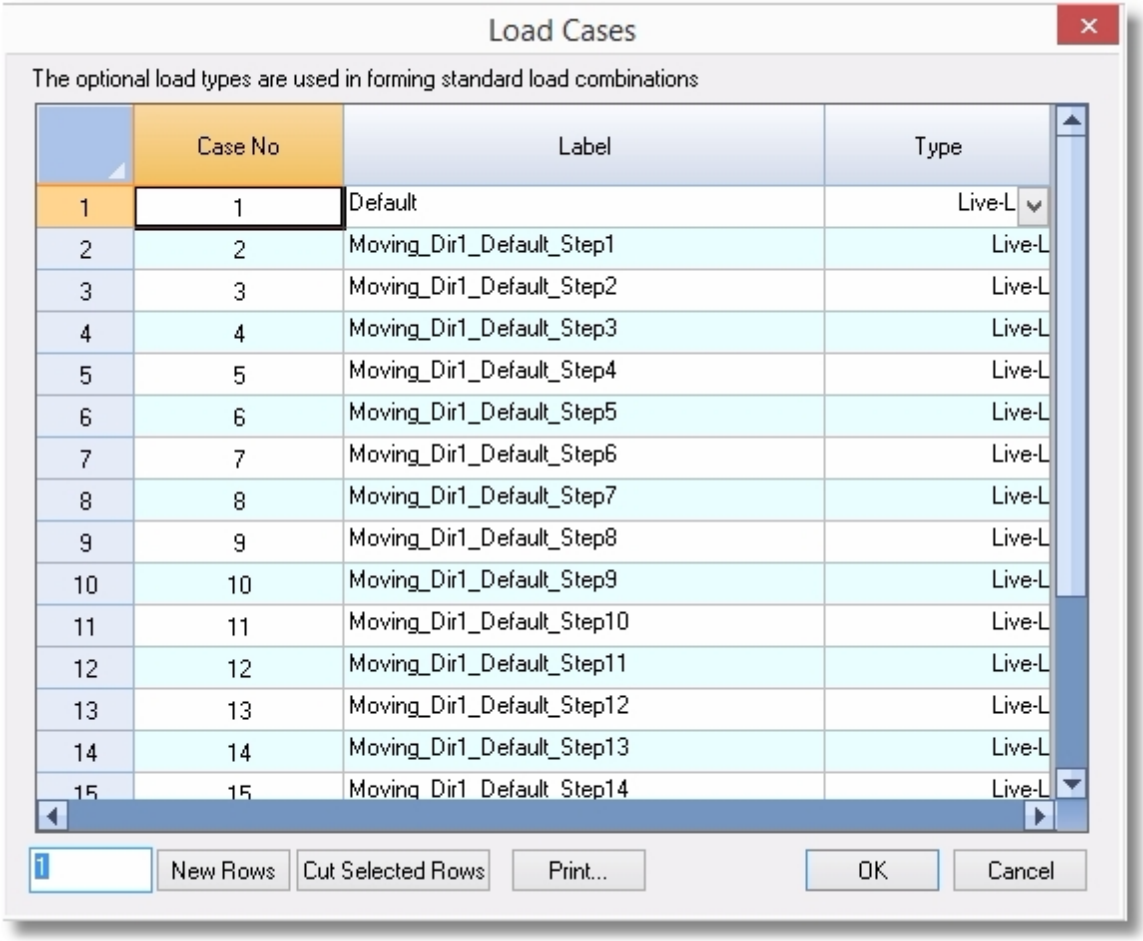
Select the Load Case that contains the load or loads to be moved.

Specify the Moving Step.

Click OK, and the screen will display the results of moving the concentrated load arrangement to the end of the string of selected members.

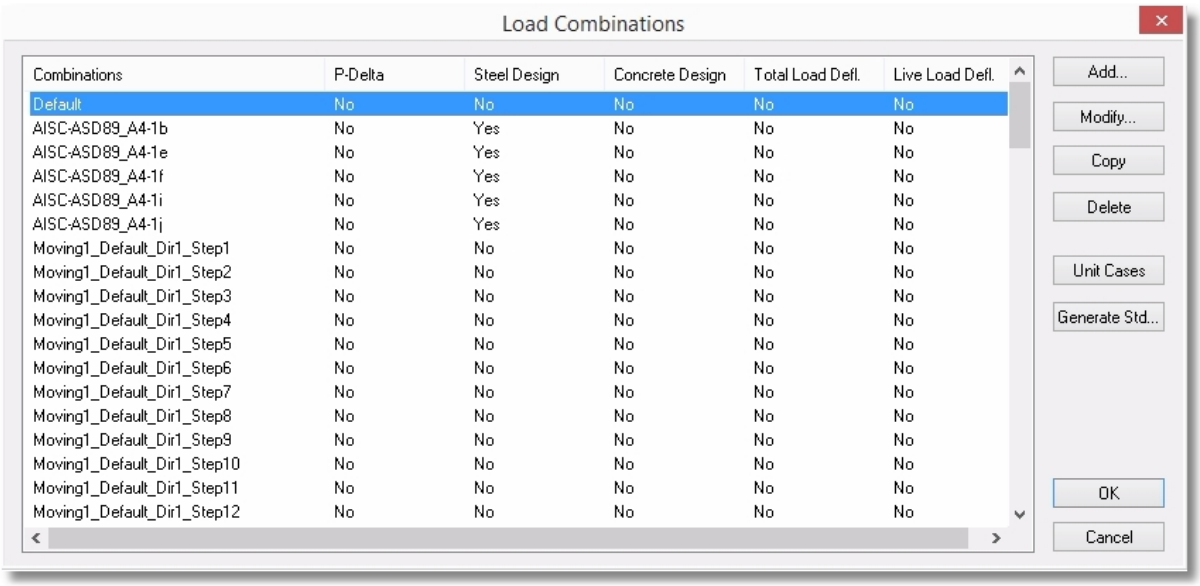


Click **Create > Load Cases** and note that the table displays new load cases that represent the moving loads.



	Case No	Label	Type
1	1	Default	Live-L
2	2	Moving_Dir1_Default_Step1	Live-L
3	3	Moving_Dir1_Default_Step2	Live-L
4	4	Moving_Dir1_Default_Step3	Live-L
5	5	Moving_Dir1_Default_Step4	Live-L
6	6	Moving_Dir1_Default_Step5	Live-L
7	7	Moving_Dir1_Default_Step6	Live-L
8	8	Moving_Dir1_Default_Step7	Live-L
9	9	Moving_Dir1_Default_Step8	Live-L
10	10	Moving_Dir1_Default_Step9	Live-L
11	11	Moving_Dir1_Default_Step10	Live-L
12	12	Moving_Dir1_Default_Step11	Live-L
13	13	Moving_Dir1_Default_Step12	Live-L
14	14	Moving_Dir1_Default_Step13	Live-L
15	15	Moving_Dir1_Default_Step14	Live-L

Click **Create > Load Combinations** and note that the existing Load Combinations have been expanded to include the newly generated moving loads.



Combinations	P-Delta	Steel Design	Concrete Design	Total Load Defl.	Live Load Defl.
Default	No	No	No	No	No
AISC-ASD89_A4-1b	No	Yes	No	No	No
AISC-ASD89_A4-1e	No	Yes	No	No	No
AISC-ASD89_A4-1f	No	Yes	No	No	No
AISC-ASD89_A4-1i	No	Yes	No	No	No
AISC-ASD89_A4-1j	No	Yes	No	No	No
Moving1_Default_Dir1_Step1	No	No	No	No	No
Moving1_Default_Dir1_Step2	No	No	No	No	No
Moving1_Default_Dir1_Step3	No	No	No	No	No
Moving1_Default_Dir1_Step4	No	No	No	No	No
Moving1_Default_Dir1_Step5	No	No	No	No	No
Moving1_Default_Dir1_Step6	No	No	No	No	No
Moving1_Default_Dir1_Step7	No	No	No	No	No
Moving1_Default_Dir1_Step8	No	No	No	No	No
Moving1_Default_Dir1_Step9	No	No	No	No	No
Moving1_Default_Dir1_Step10	No	No	No	No	No
Moving1_Default_Dir1_Step11	No	No	No	No	No
Moving1_Default_Dir1_Step12	No	No	No	No	No

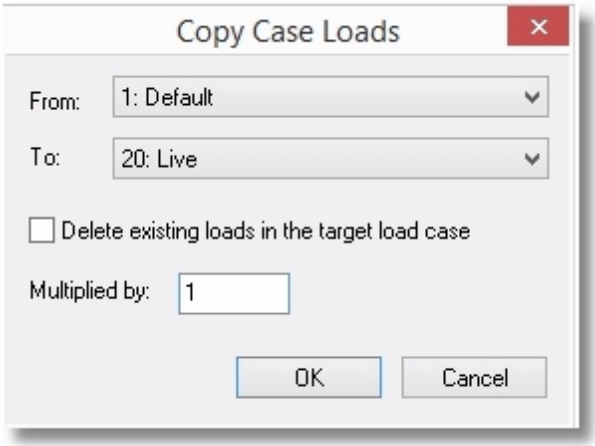
4.12 Copy Load Case

The Copy Load Case command allows loads from one load case to be copied to another load case.

It also provides the option to delete existing loads in the target load case, if desired.

This command can be useful if similar load cases are to be created with only minor differences.

Click **Create > Generate Loads > Copy Load Case** to display the *Copy Case Loads* dialog.



Select the From and To load cases.

Optionally select the checkbox to delete existing loads in the target load case.

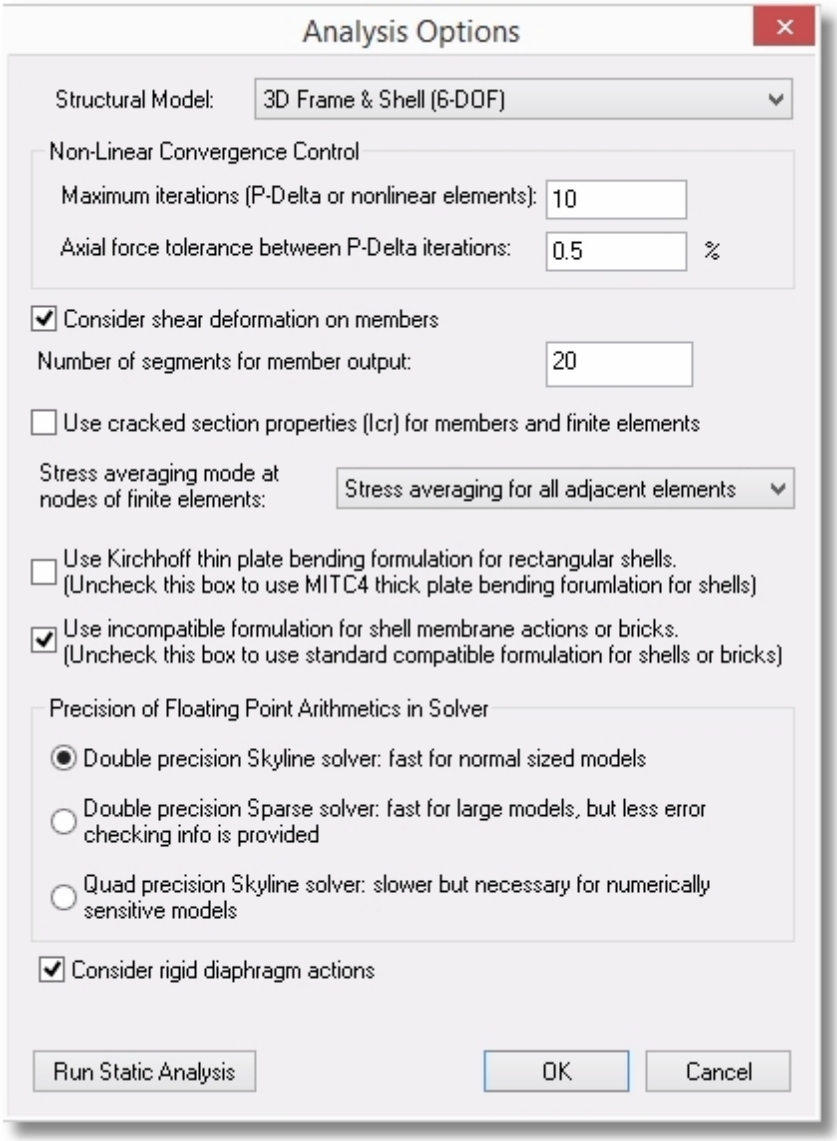
Part V

5 Specifying and Performing the Analysis

5.1 Analysis Options

The Analysis Options dialog offers many controls that affect the way certain aspects of the analysis are handled. The subtopics cover each option.

Note that there is a Run Static Analysis button at the bottom of the *Analysis Options* dialog. This allows an analysis to be initiated directly from the *Analysis Options* dialog if desired.



5.1.1 Specifying the Structural Model Type

The model type is specified in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

There are 8 different structure type options available, and their descriptions are offered in the User's Manual. Each allows a certain combination of degrees of freedom. By selecting a

model type with some limited combination of degrees of freedom, it simplifies the analysis and requires less resources to process the model. For the purposes of this training, the important thing to understand is that:

1. The most universal structural model type is 3D Frame & Shell, because all 6 degrees of freedom are available at all nodes, meaning it can be used for any model.

2. If there is ever a model that is so complex that it becomes necessary to conserve computing resources, there are other model types available that may help, and their descriptions are in the User's Manual.

5.1.2 Specifying Convergence Control

Convergence control is specified in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

When a model contains tension-only members or compression-only members, or when a P-Delta analysis is run, the analysis is an iterative process. Sometimes many iterations are required before convergence is achieved. Sometimes convergence requires too many iterations to be practical, or perhaps convergence is not even achievable. Without some control over the iteration process, this could lock up a processor in an infinite loop.

To avoid long delays and infinite loops, the program offers two controls on convergence: Maximum iterations and Axial force tolerance.

The Maximum iterations control allows the user to specify an overall maximum number of iterations that will be run when an analysis is performed. If convergence has been achieved before that number of iterations, the analysis stops when convergence has been reached. If convergence has not been achieved, the analysis stops when it reaches the specified maximum number of iterations.

The control named Axial force tolerance between P-Delta iterations sets a threshold for the program to determine if a P-Delta analysis has converged. By comparing the maximum amount of change in the axial force in any member with this specified value, the program is able to determine if the changes are small enough to consider the analysis to have converged.

5.1.3 Considering Shear Deformation

The decision as to whether to consider shear deformation is specified in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

5.1.4 Number of Segments for Member Output

The Number of segments to report in member output is specified in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

The number of segments for member segmental output may be set from 1 to 127. A value of 20 segments is recommended in most cases. More segments produce more accurate results, but require more usage of computer memory. The accuracy may be reflected in the smoothness of moment, shear and deflection diagrams.

Since member local deflection is computed based on the moment and shear diagrams, a value of more than 20 segments may be needed if very accurate local deflection is desired.

5.1.5 Use Cracked Section Properties

The option to use cracked section properties or gross section properties is specified in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

This applies to concrete beams, columns, and shells.

The cracking factors are specified in **Create (or Modify) > Member Properties > Cracking Factors** or in **Concrete Design > Cracking Factors**.

Cracking factors will not be applied until “Use cracked section properties (Icr) for members and finite elements” is checked here in the *Analysis Options* dialog.

5.1.6 Stress Averaging for Shells

The option to control stress averaging for shells is located in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

For each finite element (shell or brick), stresses are calculated at each node. If two elements share a node, the stresses at the node are different for each of these two elements. To make the results more accurate and the contours smoother, stresses can be averaged at nodes for adjacent shells.

This setting controls the way shell stresses are calculated at nodes. There are three options:

No stress averaging: Stresses at each node are output as calculated, with no averaging or smoothing.

Stress averaging for all adjacent elements: Stresses at each node are calculated by averaging nodal stresses of the elements that share the same node and have the local coordinate system parallel to the global coordinate system. With this option, if no elements have local axes coordinated with global axes, this option will result in no stress averaging.

Stress averaging based on local coordinate system: Stresses at each node are calculated by averaging nodal stresses of the elements that share the same node and have the same local coordinate system.

5.1.7 Thin Plate versus Thick Plate Option

The option to select the plate bending formulation for shells is located in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

By default, the program uses the MITC4 for shell bending formulation. The MITC4 is a thick plate formulation that accounts for the out-of-plane shear deformation.

However, if the shell elements are all rectangular, you may use the classical Kirchhoff plate bending formulation. The Kirchhoff plate element is a thin plate formulation that ignores out-of-plane shear deformation.

5.1.8 Compatible versus Incompatible Formulation for Shells and Bricks

The option to select the compatibility formulation for shells is located in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

For the membrane formulation of the shell element or brick element, incompatible modes may be added to the standard isoparametric (compatible) formulation.

The incompatible shell element models the in-plane bending more accurately than the standard compatible element.

The incompatible brick element, which produces much more accurate results than the compatible one, should almost always be preferred.

5.1.9 Precision of Floating Point Arithmetic in Solver

The option to select the precision of floating point arithmetic is located in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

Double precision solver:
- standard solver similar to many analysis programs in the market
- enough for most structures

Double precision Sparse solver:
- fastest, but lacks informative messages when something goes wrong during a solution
- can only be used for static analysis
- useful to solve extremely large structural models
- uses an out-of-core approach to minimize the requirement of computer memory

Quad precision Skyline solver:
- provides an invaluable alternative for some large or numerically sensitive models where the 64-bit floating point (double precision) solver may fail
- extremely stable and accurate, but relatively slow
- the recommended solver if the model contains rigid diaphragms to avoid numerical difficulties.

5.1.10 Rigid Diaphragm Action

The option to consider or disregard rigid diaphragm actions is located in the *Analysis Options* dialog, which can be opened by clicking **Analysis > Analysis Options**.

The option is provided mainly as a convenience to be able to run a model either way without having to delete the diaphragms in the model.

5.2 Running the Analysis

5.2.1 Static Analysis

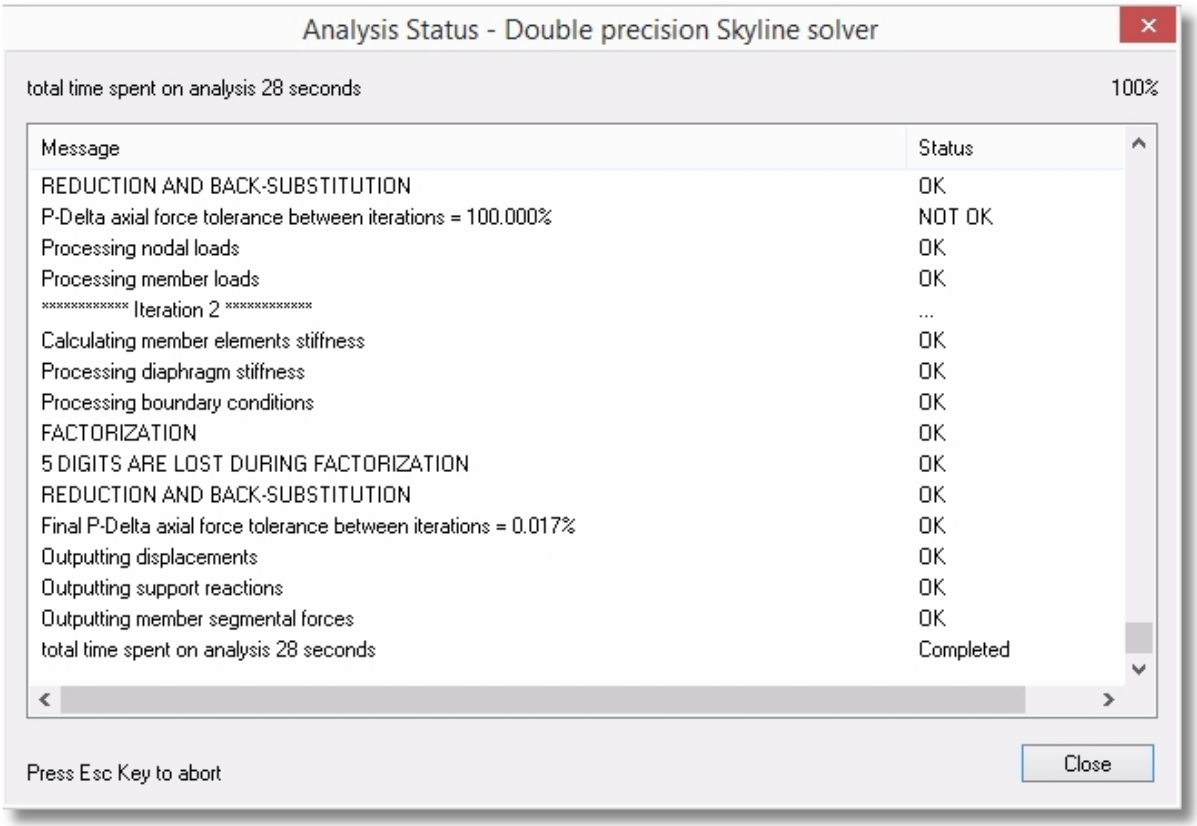
Analysis > Static Analysis performs the static analysis of the model.

You should set the appropriate analysis options before running this command. To do that, just click **Analyze > Analysis Options**.

For models with P-Delta load cases, it is recommended to first run them without a P-Delta analysis just to verify that the model is stable. Remember that P-Delta analysis is specified on a load combination by load combination basis. This can be set on the *Load Combinations* dialog by clicking **Create > Load Combinations**.

5.2.2 Analysis Status Window

After running an analysis the Analysis Status window will display information about the analysis.



If the analysis is successful, the window will include many lines of information about the various steps that were complete and the data that was being reported, including information about the number of iterations that were required to achieve convergence.

If the analysis was unsuccessful, the window may contain useful information on the cause of the failure, such as instabilities, as well as some suggestions, like using a different analysis type.

Part VI

6 Reviewing the Analysis Results

6.1 Query Function

One powerful way to review analysis results on a member by member basis is the Query function, which is available from **View > Query**.

This tool has the special "Query" cursor associated with it. When the Query cursor is displayed, clicking on any node, member or shell opens the Nodal Info, Member Info or Shell Info dialog that offers a great deal of info about the clicked node, member or shell. The info includes geometric, material and loading data. After a successful analysis the dialog also includes analysis results such as forces, moments, deflections, and rotations.

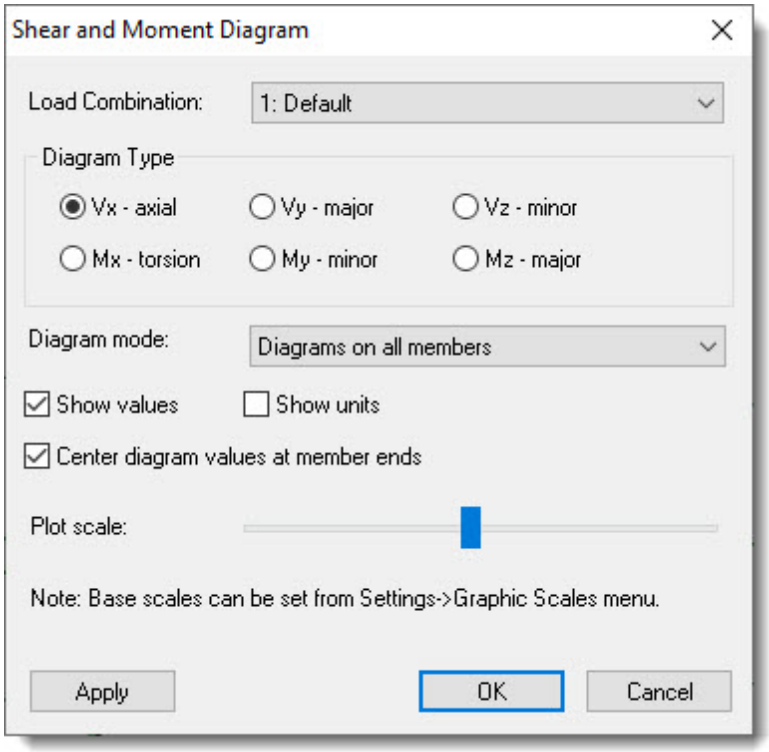
6.2 Result Diagrams

The Analysis Results tab offers a variety of ways to view the results of an analysis. Graphical results are often very telling in the early stages of a design. There is a whole category of graphical results available on the Analysis results ribbon. Look to the right and find the category named Results Diagrams. Each of these items is explained in its own subsection.

Tip: The graphical scales used to depict these diagrams can be adjusted by clicking **Settings and Tools > Graphic Scales**.

6.2.1 Shear and Moment Diagrams

Click **Analysis Results > Shear & Moment Diagram** to view shear or moment diagrams on the model itself This opens the *Shear and Moment Diagram* dialog.



Select the desired Load Combination. Select the desired Diagram Type, where options include:

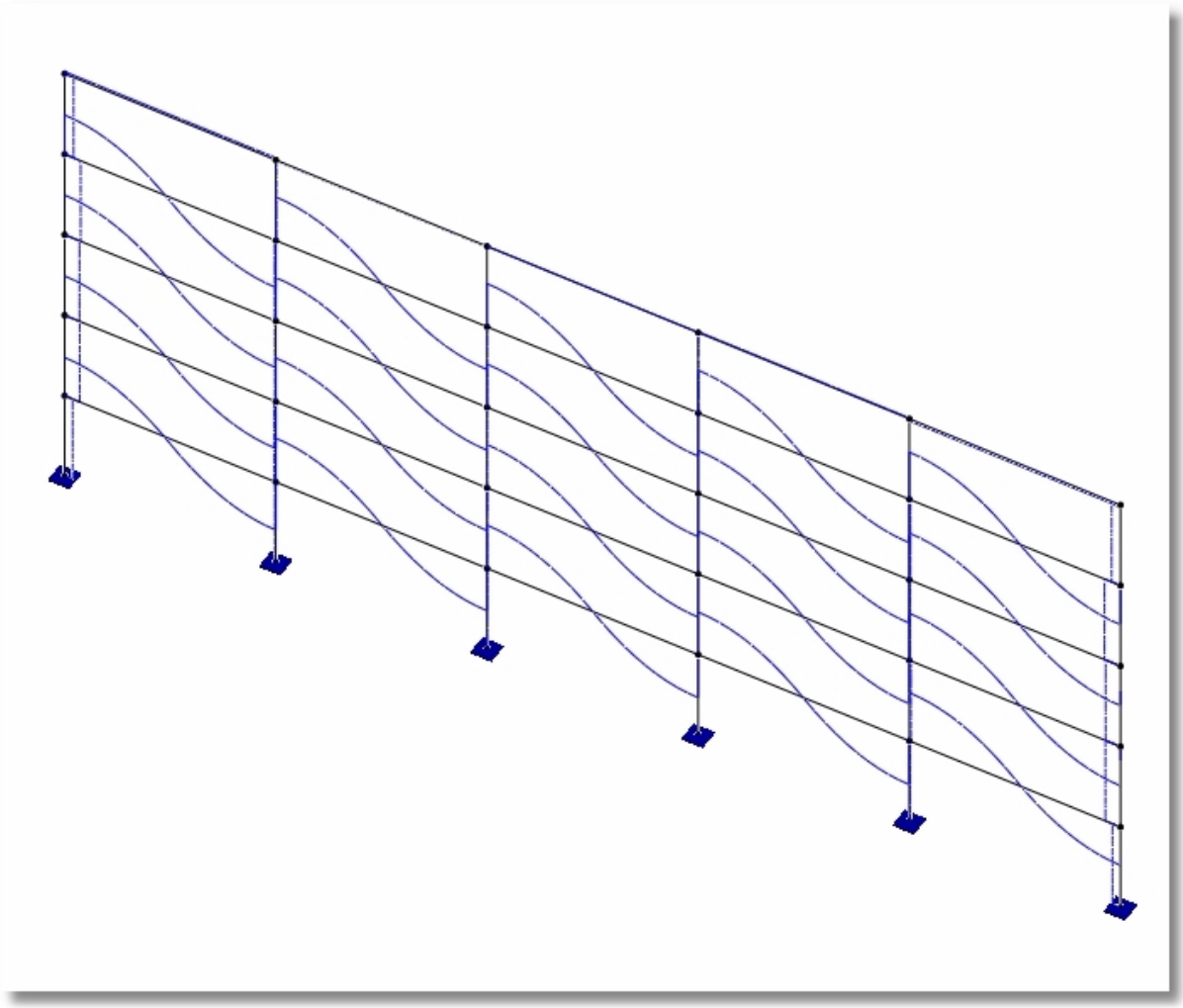
- Vx - axial force
- Vy - major shear
- Vz - minor shear
- Mx - torsional moment
- My - minor axis moment
- Mz - major axis moment

Select the Diagram mode, where options include:

- Diagrams on all members
- Diagrams on selected members
- Erase existing diagrams (This option is one way to remove diagrams from the model. Another way is to use the Restore Model icon.

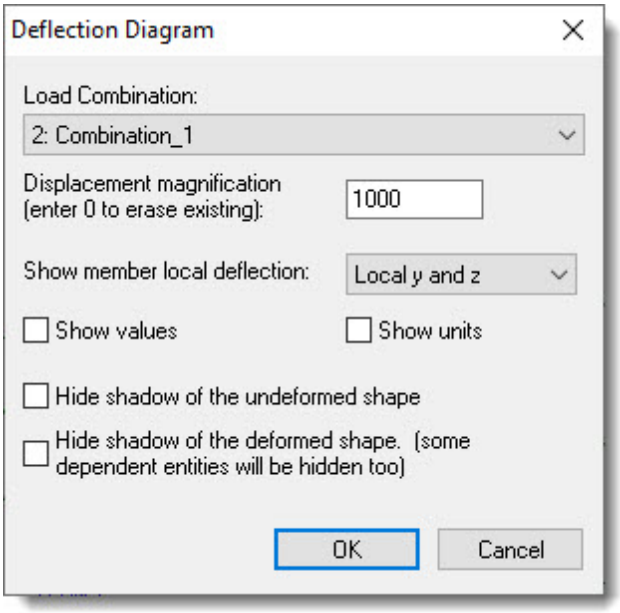
The final options include:

- Show values (If deselected, only diagrams are drawn.)
- Show units (Only has any meaning if Show values is selected.)
- Center diagram values at member ends (If selected, the labels will be centered on the member to clarify which label is associated with which member. If deselected, the labels will be offset to clarify the view of the member.)



6.2.2 Deflection Diagram

Click **Analysis Results > Deflection Diagram**, to view the deflection diagram on the model itself. This opens the *Deflection Diagram* dialog.



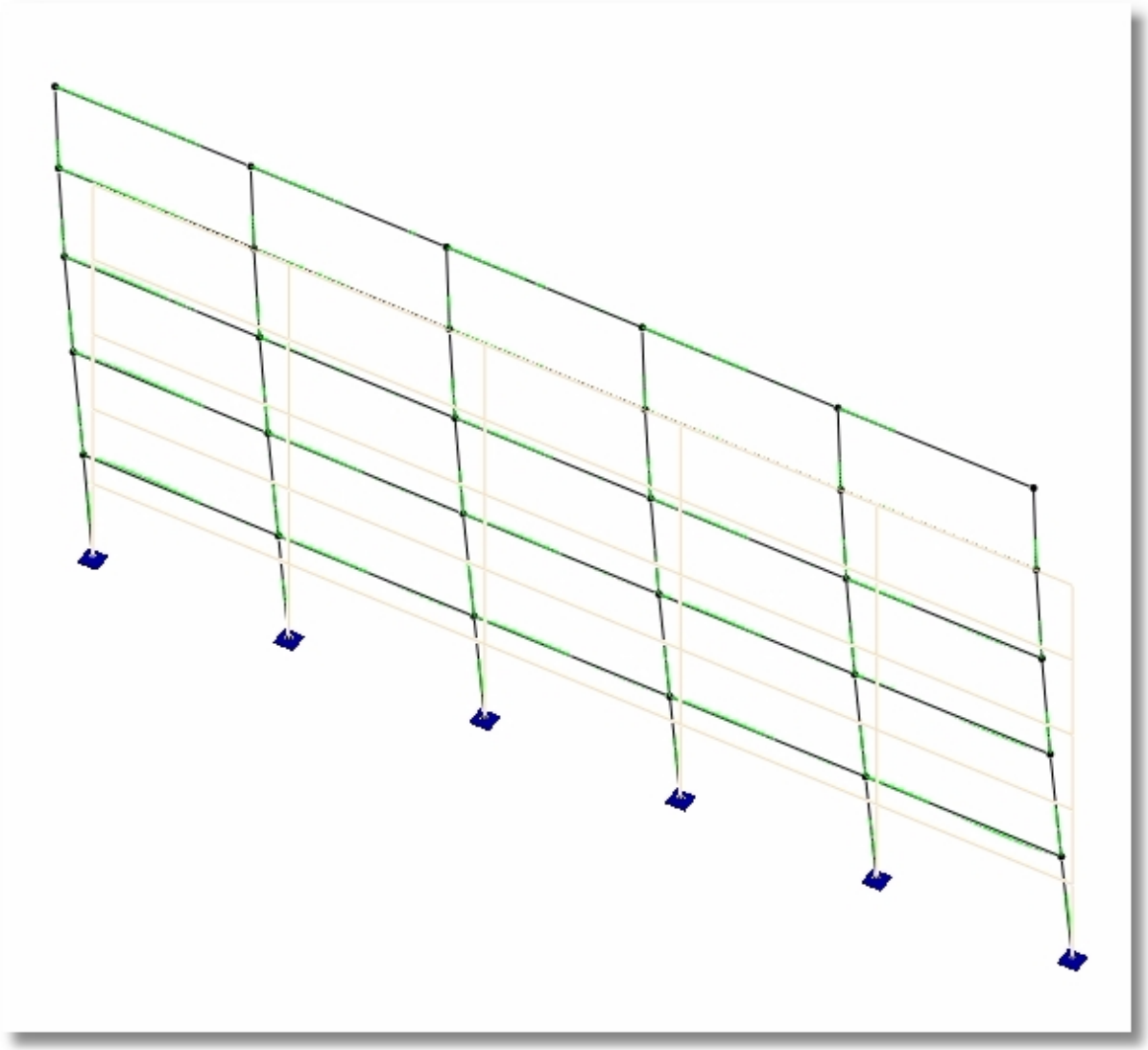
Select the desired Load Combination. Set the Displacement magnification.

Select the member local deflection mode, where options include:

- None
- Local y and z
- Local y only
- Local z only.

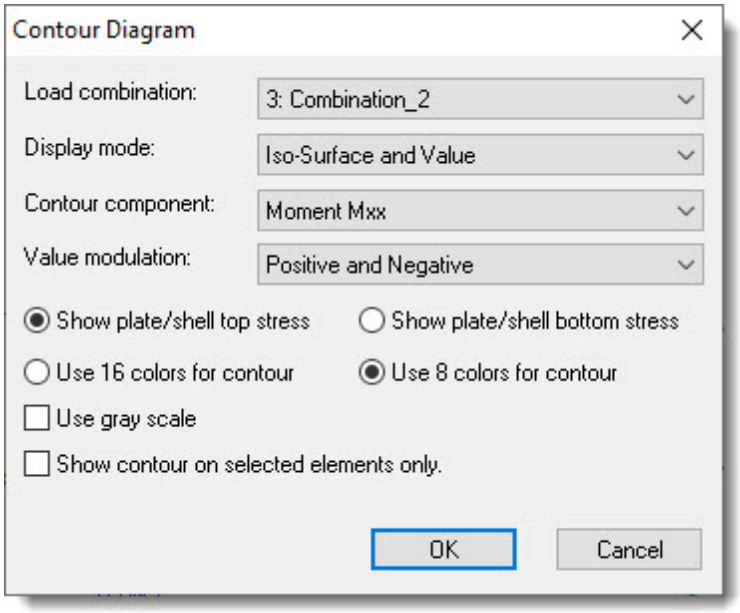
The final options include:

- Show values (If selected, values are shown.)
- Show units (Only has any meaning if Show values is selected.)
- Hide shadow of undeformed shape (If selected, the undeflected shape is hidden. If deselected, the undeflected shape is drawn.)
- Hide shadow of deformed shape (If deselected, the program superimposes the diagram of the deflected structure without local deflection. If selected, the diagram of the deflected structure without local deflection is hidden.)



6.2.3 Contour Diagram

Click **Analysis Results > Contour Diagram** to view the contour diagram on the model itself. This opens the *Contour Diagram* dialog.



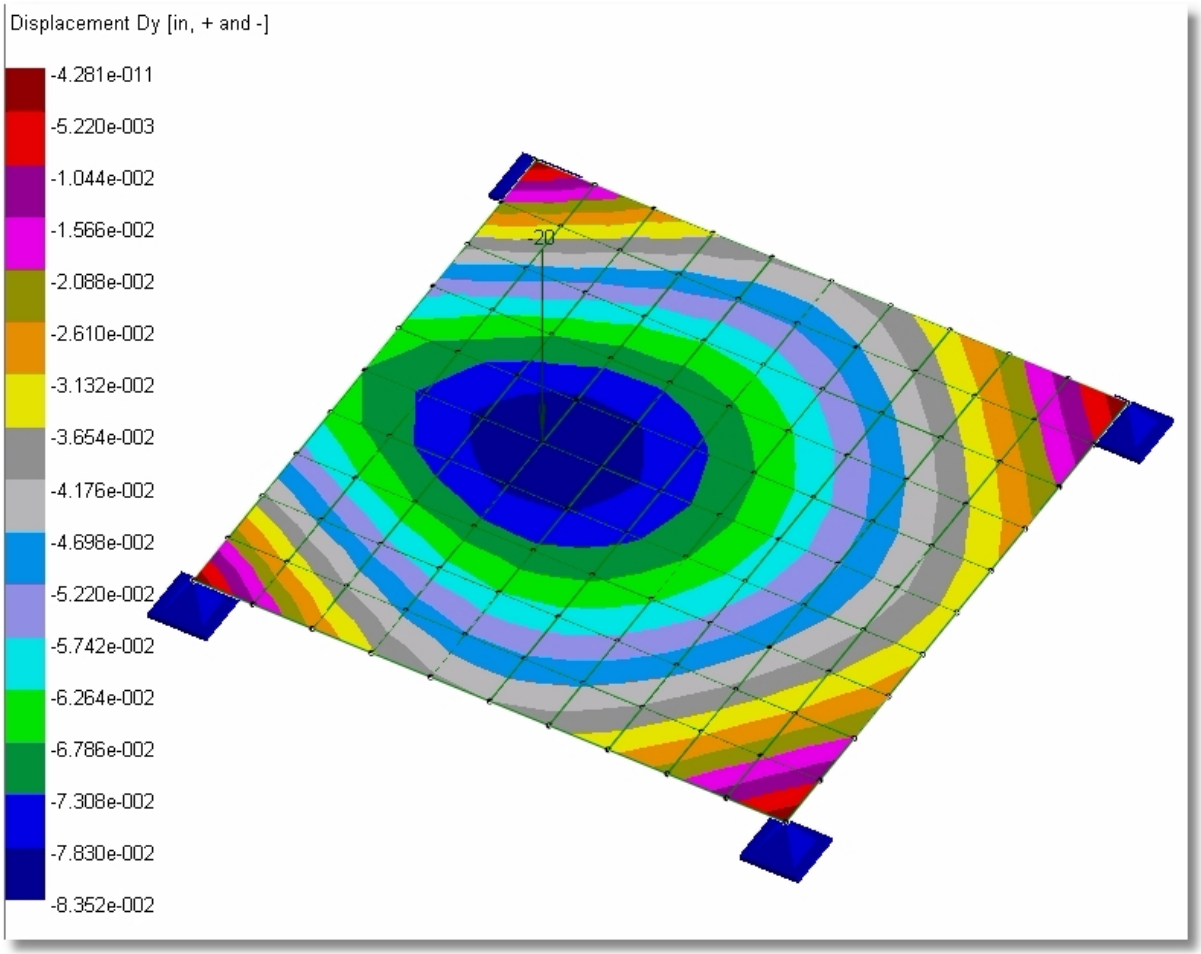
Select the desired Load Combination.

Select the display mode, where the options include diagrams with values, diagrams only, values only, or erase.

Select the Contour component, where the options include displacements, moments, bending shears, forces, and many types of plate stresses.

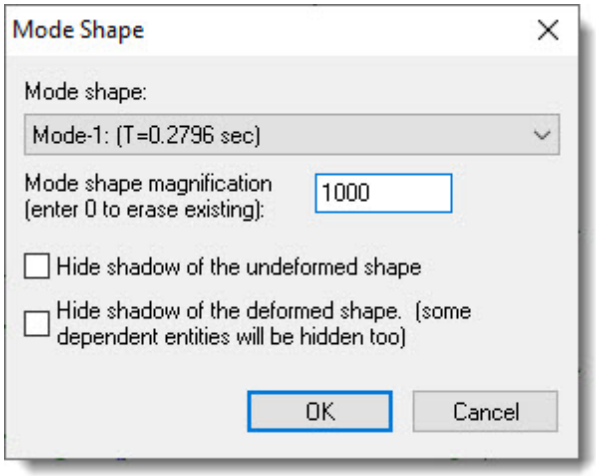
The final options include:

- Show plate/shell top stress or bottom stress
- Use 8 or 16 colors for contours
- Use gray scale or color
- Show contours on selected elements or on all elements



6.2.4 Mode Shape

Click **Analysis Results > Mode Shape** to view the contour diagram on the model itself. This opens the *Mode Shape* dialog.

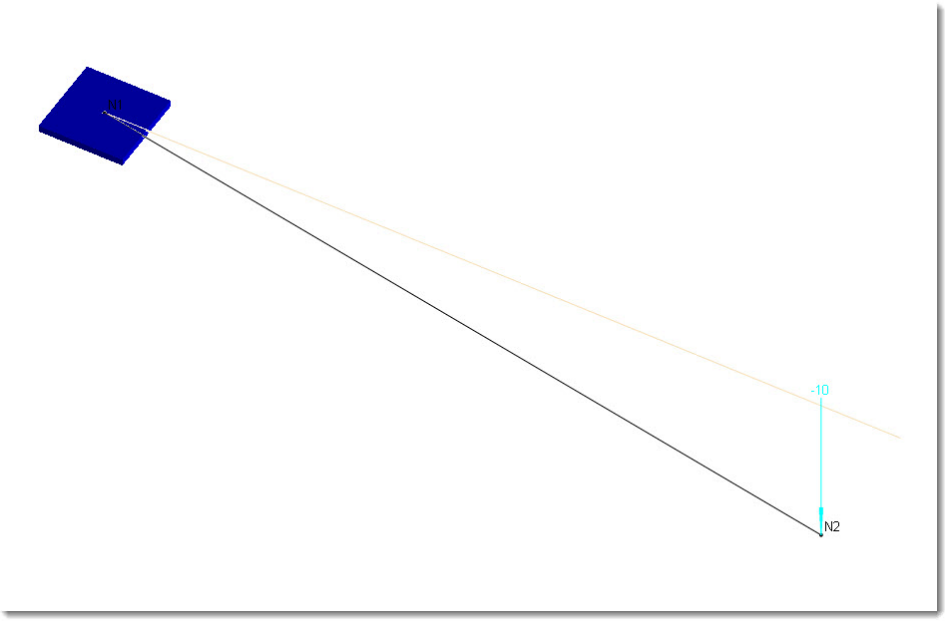


Select the desired mode.

Enter the magnification factor.

Select miscellaneous graphical choices and click OK.

The result is a display of the selected mode shape.



6.2.5 Response Animation

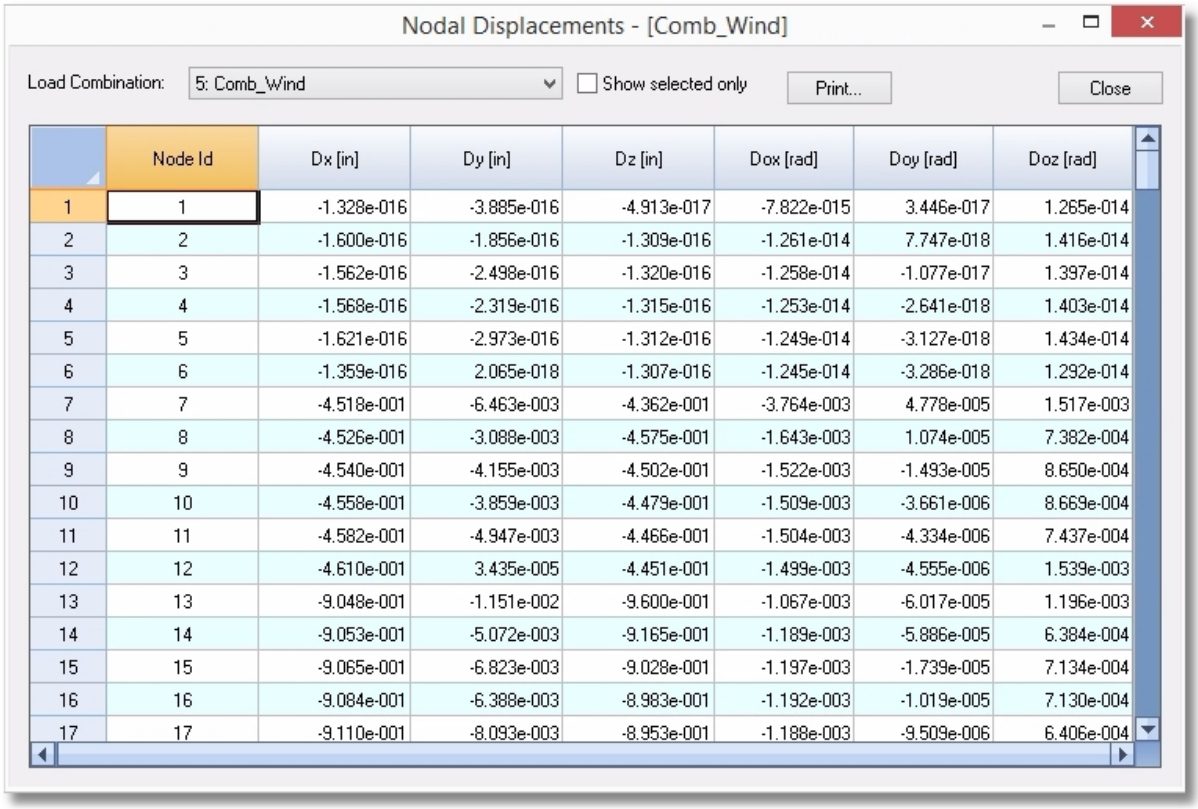
Response Animation is a visualization enhancement that allows any of the Result Diagrams to be animated.

Start by applying the necessary settings and scales to create the desired result diagram. With the diagram still in view on the screen, click **Analysis Results > Response Animation**.

To stop the animation, either click the Response Animation button again, or click the Restore Model button,  on the Quick Access Toolbar.

6.3 Nodal Displacements

Click **Analysis Results > Nodal Displacements** to see a table of nodal displacements. The *Nodal Displacements* table appears.



	Node Id	Dx [in]	Dy [in]	Dz [in]	Dox [rad]	Doy [rad]	Doz [rad]
1	1	-1.328e-016	-3.885e-016	-4.913e-017	-7.822e-015	3.446e-017	1.265e-014
2	2	-1.600e-016	-1.856e-016	-1.309e-016	-1.261e-014	7.747e-018	1.416e-014
3	3	-1.562e-016	-2.498e-016	-1.320e-016	-1.258e-014	-1.077e-017	1.397e-014
4	4	-1.568e-016	-2.319e-016	-1.315e-016	-1.253e-014	-2.641e-018	1.403e-014
5	5	-1.621e-016	-2.973e-016	-1.312e-016	-1.249e-014	-3.127e-018	1.434e-014
6	6	-1.359e-016	2.065e-018	-1.307e-016	-1.245e-014	-3.286e-018	1.292e-014
7	7	-4.518e-001	-6.463e-003	-4.362e-001	-3.764e-003	4.778e-005	1.517e-003
8	8	-4.526e-001	-3.088e-003	-4.575e-001	-1.643e-003	1.074e-005	7.382e-004
9	9	-4.540e-001	-4.155e-003	-4.502e-001	-1.522e-003	-1.493e-005	8.650e-004
10	10	-4.558e-001	-3.859e-003	-4.479e-001	-1.509e-003	-3.661e-006	8.669e-004
11	11	-4.582e-001	-4.947e-003	-4.466e-001	-1.504e-003	-4.334e-006	7.437e-004
12	12	-4.610e-001	3.435e-005	-4.451e-001	-1.499e-003	-4.555e-006	1.539e-003
13	13	-9.048e-001	-1.151e-002	-9.600e-001	-1.067e-003	-6.017e-005	1.196e-003
14	14	-9.053e-001	-5.072e-003	-9.165e-001	-1.189e-003	-5.886e-005	6.384e-004
15	15	-9.065e-001	-6.823e-003	-9.028e-001	-1.197e-003	-1.739e-005	7.134e-004
16	16	-9.084e-001	-6.388e-003	-8.983e-001	-1.192e-003	-1.019e-005	7.130e-004
17	17	-9.110e-001	-8.093e-003	-8.953e-001	-1.188e-003	-9.509e-006	6.406e-004

Select the desired Load Combination.

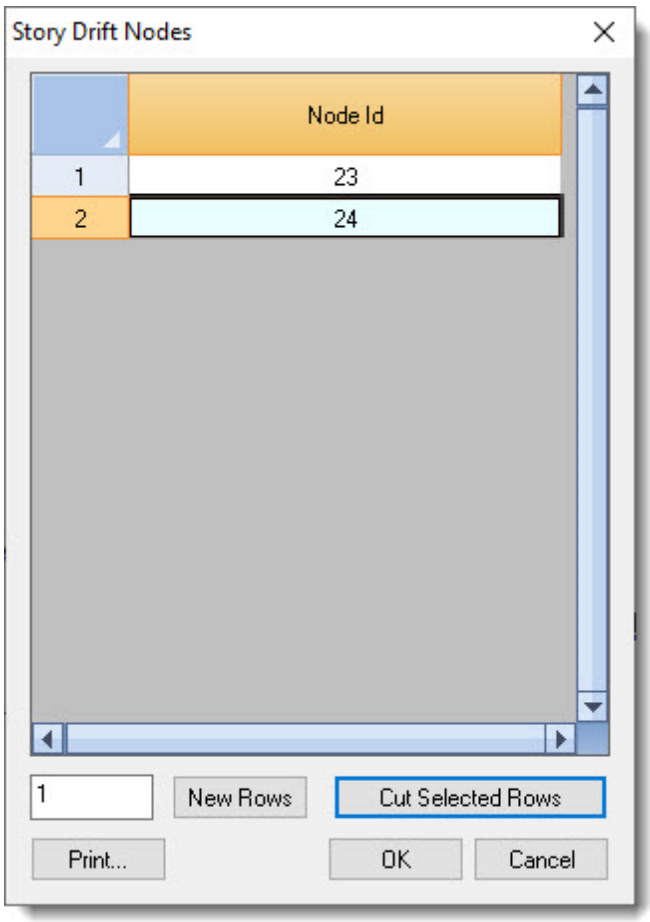
Choose whether the table should display results for the selected entities or for all entities.

The table reports displacements and rotations for the desired nodes.

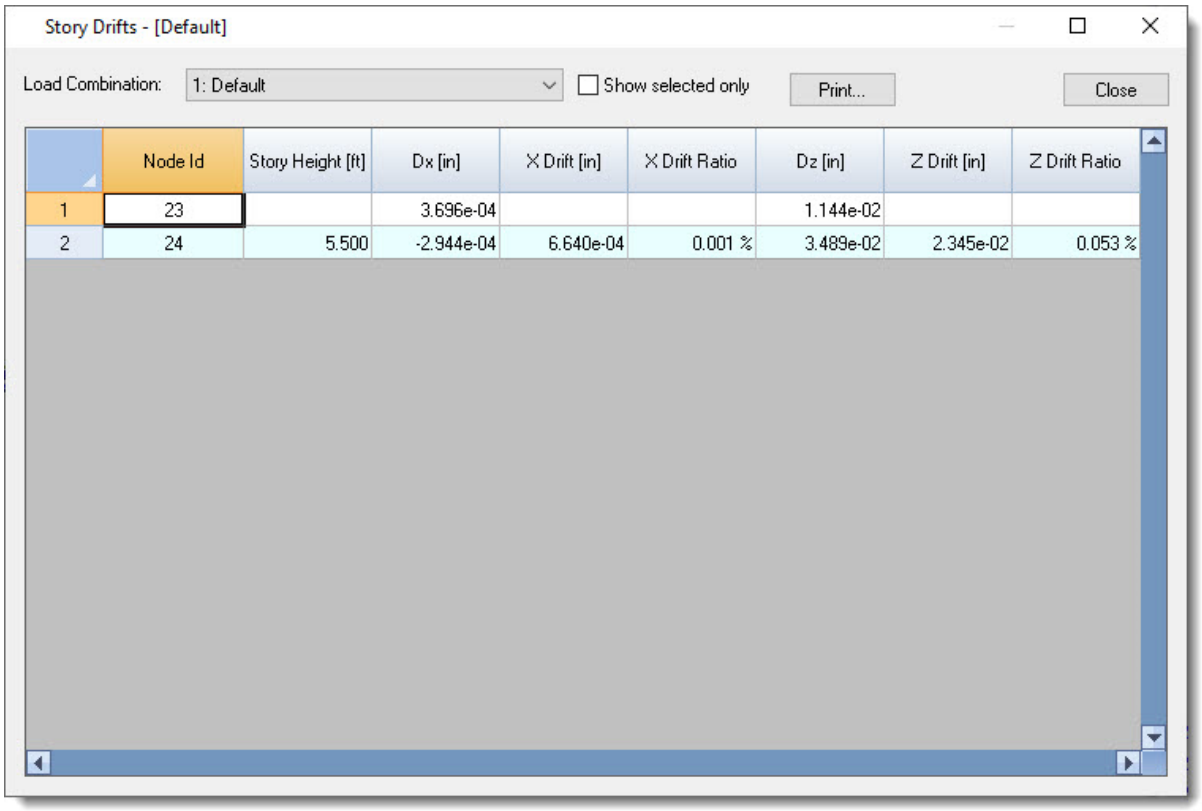
The table can be printed with the Print button.

6.4 Story Drifts

The first step in evaluating story drifts is to define Drift Nodes, which are the nodes at which drift will be evaluated. To do this, select the drift nodes and click **Create > Story Drift Nodes**. This will create a table of nodes (usually one at each level) that can be viewed by clicking **Tables > Story Drift Nodes.**



After selecting the drift nodes, click **Analysis Results > Story Drift**. The *Story Drifts* table opens.



	Node Id	Story Height [ft]	Dx [in]	X Drift [in]	X Drift Ratio	Dz [in]	Z Drift [in]	Z Drift Ratio
1	23		3.696e-04			1.144e-02		
2	24	5.500	-2.944e-04	6.640e-04	0.001 %	3.489e-02	2.345e-02	0.053 %

Select the desired Load Combination.

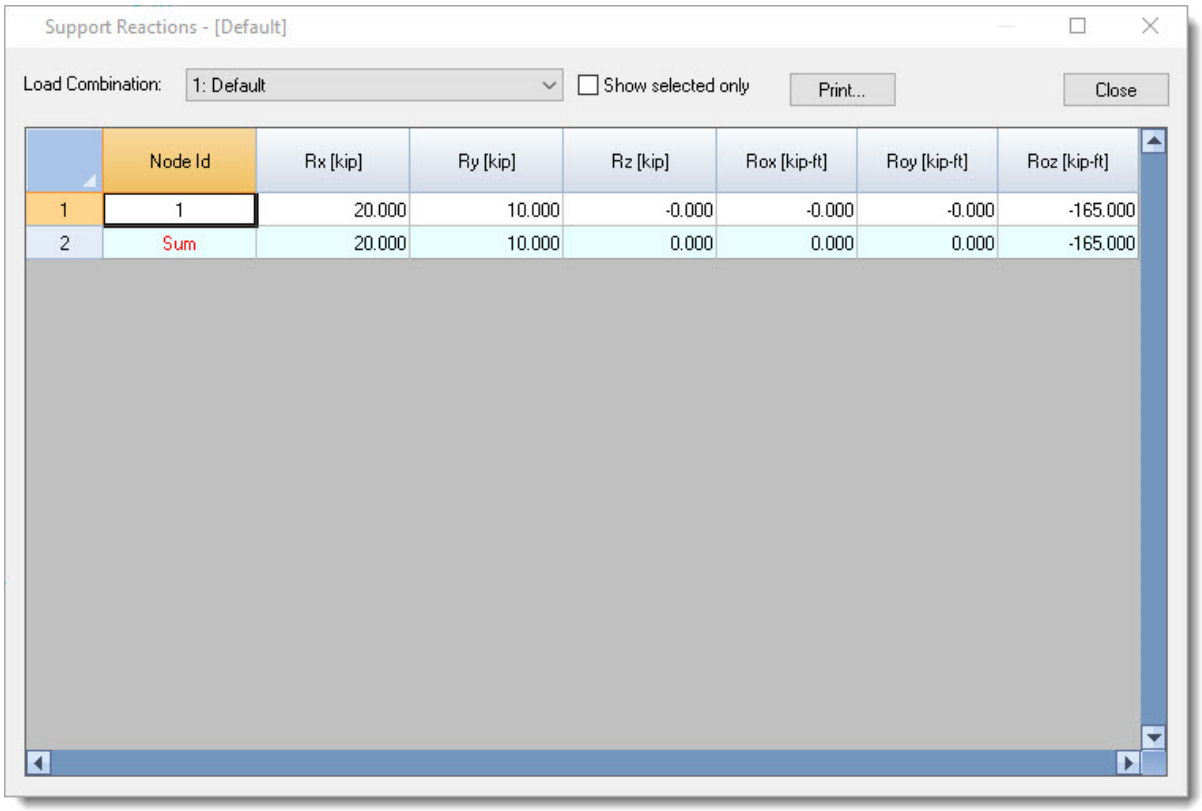
Since this table uses the predefined drift nodes, deselect the option to show selected entities only.

The table reports nodal displacement, inter-story drift, and the drift ratio in the X and Z direction for each story.

The table can be printed with the Print button.

6.5 Support Reactions

Click **Analysis Results > Support Reactions** to see a table of support reactions. The *Support Reactions* table appears.



	Node Id	Rx [kip]	Ry [kip]	Rz [kip]	Rox [kip-ft]	Roy [kip-ft]	Roz [kip-ft]
1	1	20.000	10.000	-0.000	-0.000	-0.000	-165.000
2	Sum	20.000	10.000	0.000	0.000	0.000	-165.000

Select the desired Load Combination.

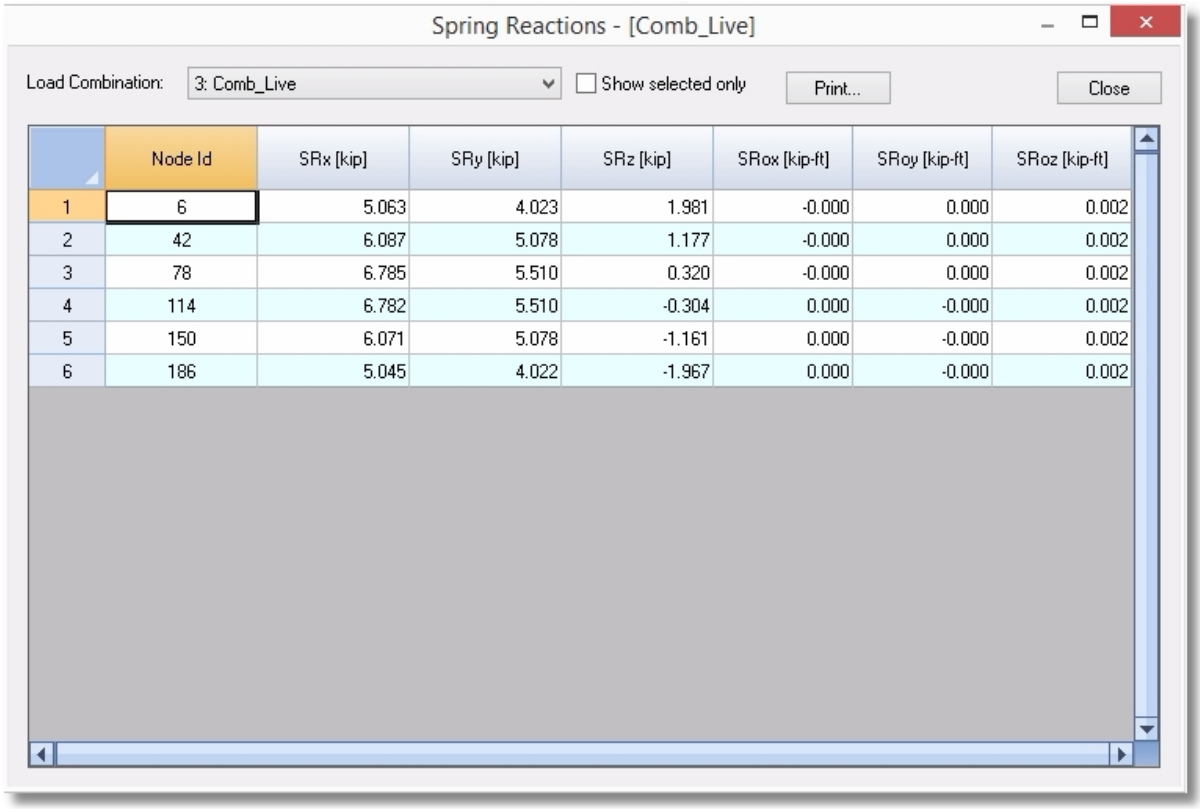
Choose whether the table should display results for the selected entities or for all entities.

The table reports reactions for the desired nodes.

The table can be printed with the Print button.

6.6 Spring Reactions

Click **Analysis Results > Nodal Spring Reactions [or Line Spring Reactions or Surface Spring Reactions]** to see a table of spring reactions. The *Spring Reactions* table appears.



	Node Id	SRx [kip]	SRy [kip]	SRz [kip]	SRox [kip-ft]	SRoy [kip-ft]	SRoz [kip-ft]
1	6	5.063	4.023	1.981	-0.000	0.000	0.002
2	42	6.087	5.078	1.177	-0.000	0.000	0.002
3	78	6.785	5.510	0.320	-0.000	0.000	0.002
4	114	6.782	5.510	-0.304	0.000	-0.000	0.002
5	150	6.071	5.078	-1.161	0.000	-0.000	0.002
6	186	5.045	4.022	-1.967	0.000	-0.000	0.002

Select the desired Load Combination.

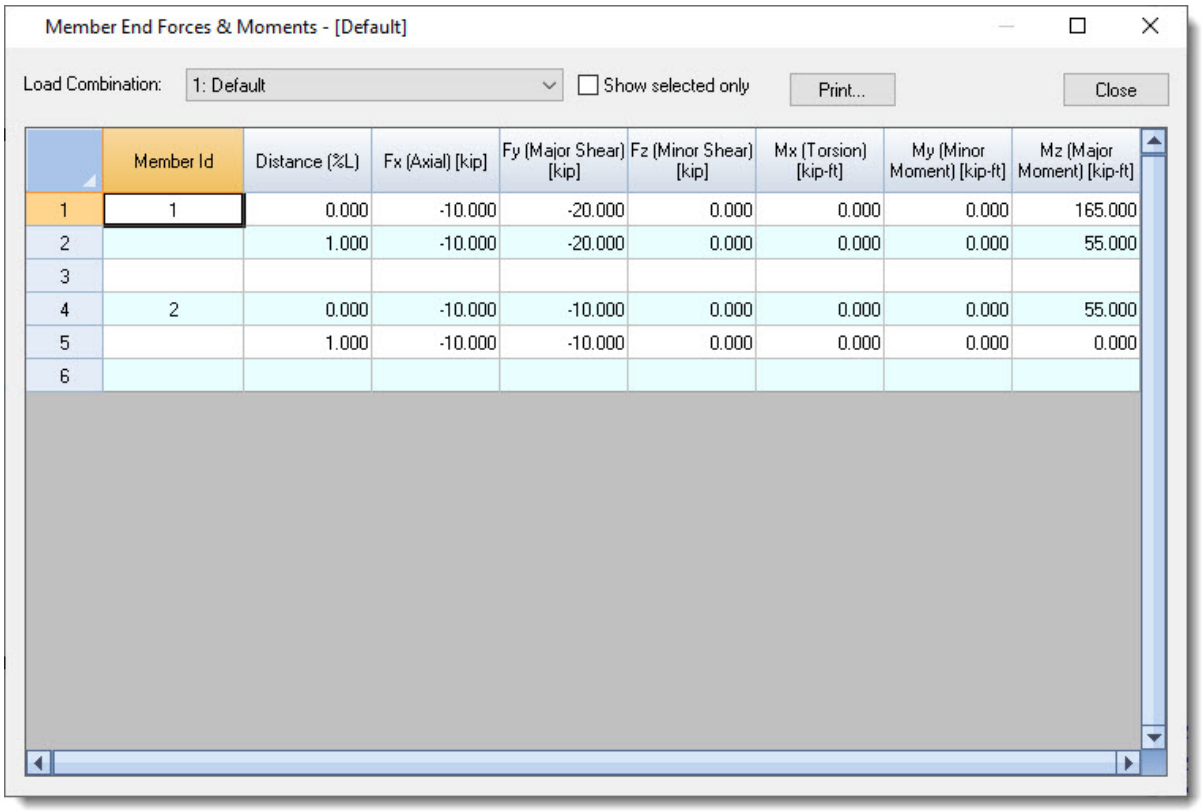
Choose whether the table should display results for the selected entities or for all entities.

The table reports reactions for the desired springs.

The table can be printed with the Print button.

6.7 Member End Forces & Moments

Click **Analysis Results > Member End Forces & Moments** to see a table of member end forces and moments. The *Member End Results* table appears.



	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-ft]	My (Minor Moment) [kip-ft]	Mz (Major Moment) [kip-ft]
1	1	0.000	-10.000	-20.000	0.000	0.000	0.000	165.000
2		1.000	-10.000	-20.000	0.000	0.000	0.000	55.000
3								
4	2	0.000	-10.000	-10.000	0.000	0.000	0.000	55.000
5		1.000	-10.000	-10.000	0.000	0.000	0.000	0.000
6								

Select the desired Load Combination.

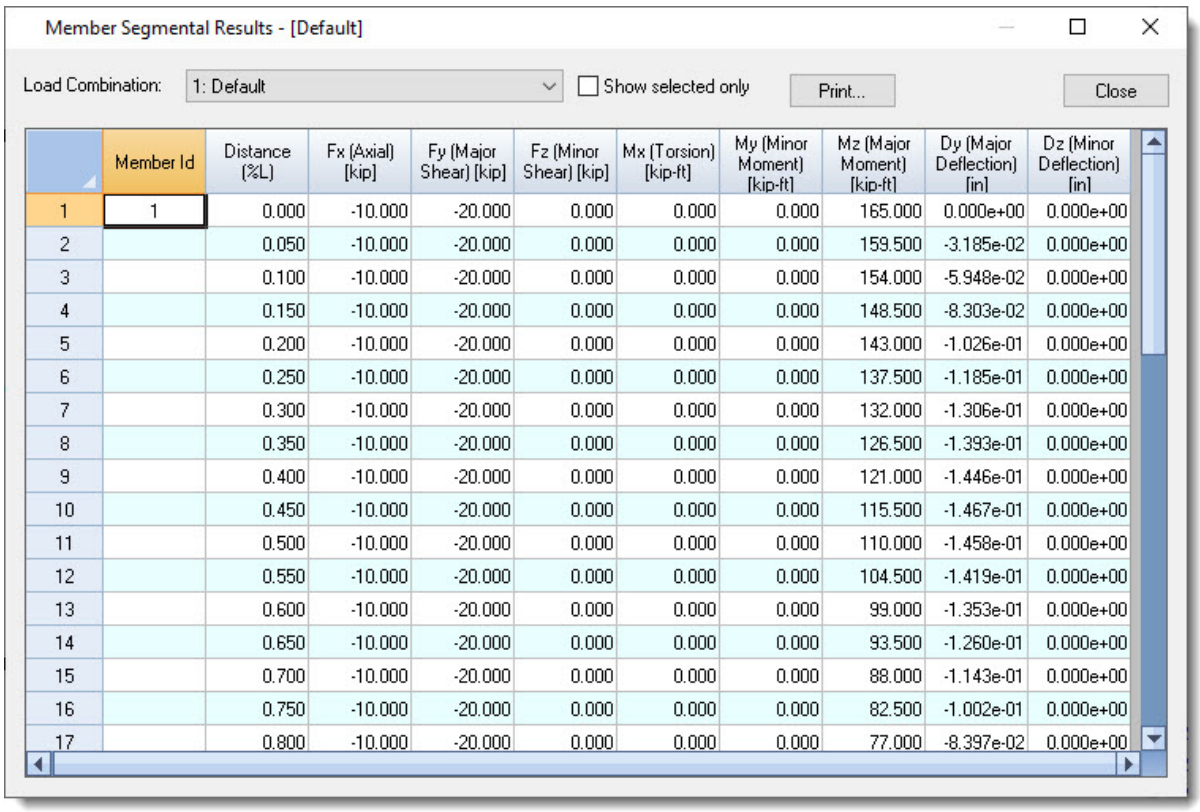
Choose whether the table should display results for the selected entities or for all entities.

The table reports shears, moments, torsion and axial force at the starting end and ending end for the desired members.

The table can be printed with the Print button.

6.8 Member Segmental Results

Click **Analysis Results > Member Segmental Results** to see a table of member forces and moments at small increments all along the length of the members. The *Member Segmental Results* table appears.



	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-ft]	My (Minor Moment) [kip-ft]	Mz (Major Moment) [kip-ft]	Dy (Major Deflection) [in]	Dz (Minor Deflection) [in]
1	1	0.000	-10.000	-20.000	0.000	0.000	0.000	165.000	0.000e+00	0.000e+00
2		0.050	-10.000	-20.000	0.000	0.000	0.000	159.500	-3.185e-02	0.000e+00
3		0.100	-10.000	-20.000	0.000	0.000	0.000	154.000	-5.948e-02	0.000e+00
4		0.150	-10.000	-20.000	0.000	0.000	0.000	148.500	-8.303e-02	0.000e+00
5		0.200	-10.000	-20.000	0.000	0.000	0.000	143.000	-1.026e-01	0.000e+00
6		0.250	-10.000	-20.000	0.000	0.000	0.000	137.500	-1.185e-01	0.000e+00
7		0.300	-10.000	-20.000	0.000	0.000	0.000	132.000	-1.306e-01	0.000e+00
8		0.350	-10.000	-20.000	0.000	0.000	0.000	126.500	-1.393e-01	0.000e+00
9		0.400	-10.000	-20.000	0.000	0.000	0.000	121.000	-1.446e-01	0.000e+00
10		0.450	-10.000	-20.000	0.000	0.000	0.000	115.500	-1.467e-01	0.000e+00
11		0.500	-10.000	-20.000	0.000	0.000	0.000	110.000	-1.458e-01	0.000e+00
12		0.550	-10.000	-20.000	0.000	0.000	0.000	104.500	-1.419e-01	0.000e+00
13		0.600	-10.000	-20.000	0.000	0.000	0.000	99.000	-1.353e-01	0.000e+00
14		0.650	-10.000	-20.000	0.000	0.000	0.000	93.500	-1.260e-01	0.000e+00
15		0.700	-10.000	-20.000	0.000	0.000	0.000	88.000	-1.143e-01	0.000e+00
16		0.750	-10.000	-20.000	0.000	0.000	0.000	82.500	-1.002e-01	0.000e+00
17		0.800	-10.000	-20.000	0.000	0.000	0.000	77.000	-8.397e-02	0.000e+00

Select the desired Load Combination.

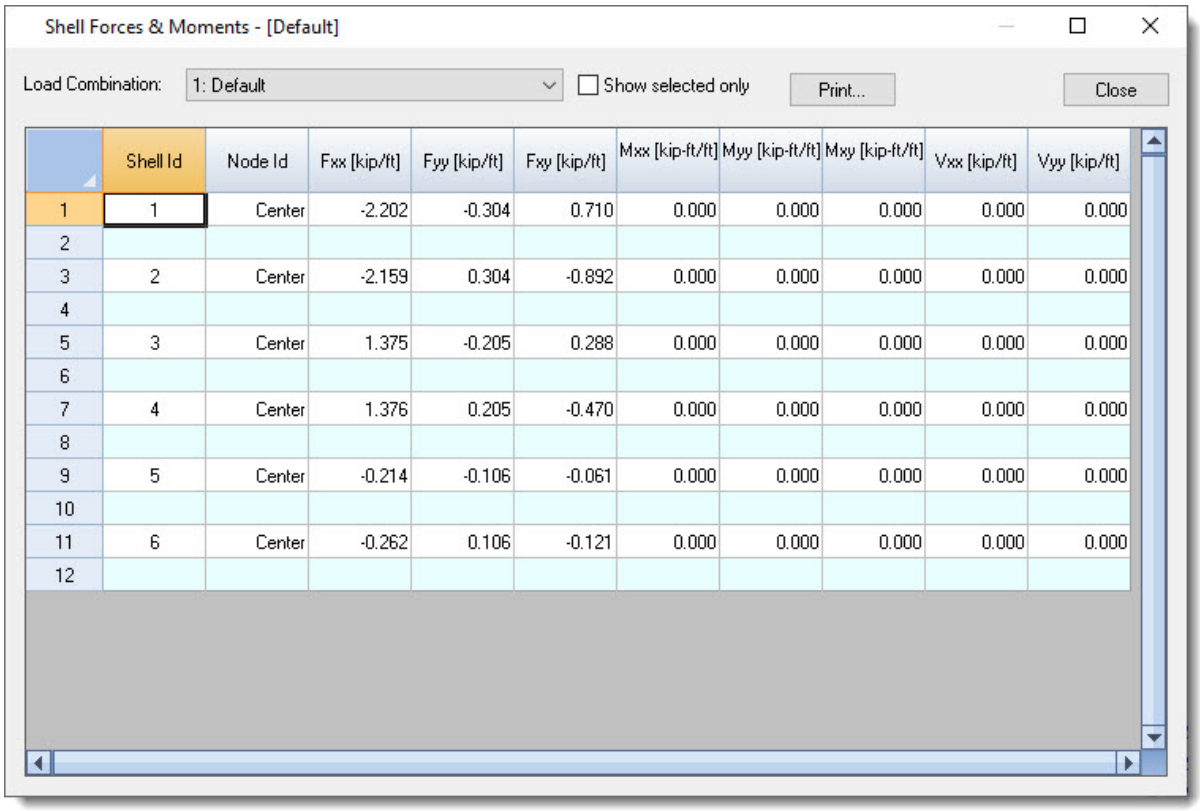
Choose whether the table should display results for the selected entities or for all entities.

The table reports shears, moments, torsion and axial force at many incremental locations along the length of the desired members.

The table can be printed with the Print button.

6.9 Shell Forces & Moments

Click **Analysis Results > Shell Forces, Moments & Stresses > Shell Forces & Moments** to see a table of shell forces and moments. The *Shell Forces and Moments* table appears.



	Shell Id	Node Id	Fxx [kip/ft]	Fyy [kip/ft]	Fxy [kip/ft]	Mxx [kip-ft/ft]	Myy [kip-ft/ft]	Mxy [kip-ft/ft]	Vxx [kip/ft]	Vyy [kip/ft]
1	1	Center	-2.202	-0.304	0.710	0.000	0.000	0.000	0.000	0.000
2										
3	2	Center	-2.159	0.304	-0.892	0.000	0.000	0.000	0.000	0.000
4										
5	3	Center	1.375	-0.205	0.288	0.000	0.000	0.000	0.000	0.000
6										
7	4	Center	1.376	0.205	-0.470	0.000	0.000	0.000	0.000	0.000
8										
9	5	Center	-0.214	-0.106	-0.061	0.000	0.000	0.000	0.000	0.000
10										
11	6	Center	-0.262	0.106	-0.121	0.000	0.000	0.000	0.000	0.000
12										

Select the desired Load Combination.

Choose whether the table should display results for the selected entities or for all entities.

Specify the force and moment locations to be at the nodes and/or the center of each shell by clicking **Settings and Tools > Data Options**.

The table reports shears and moments for the desired shells.

The shell forces and moments include:

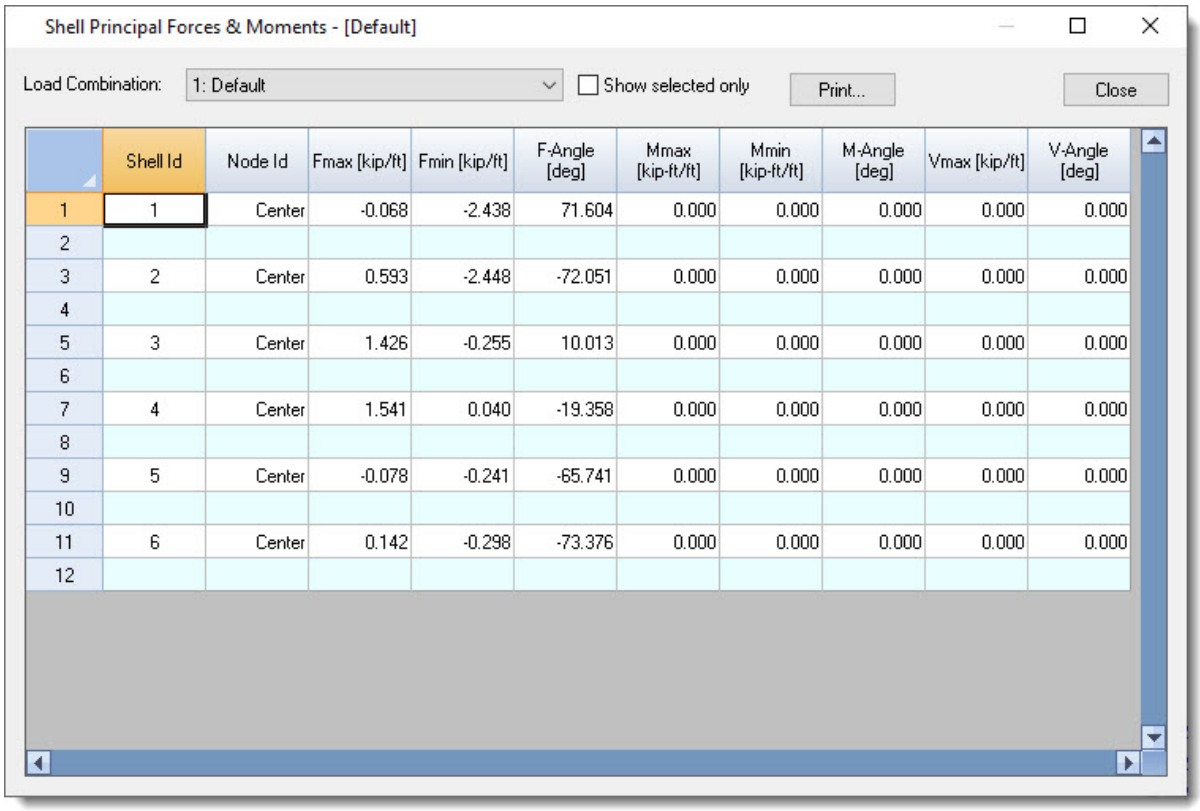
- Fxx: in-plane normal force in the local x direction
- Fyy: in-plane normal force in the local y direction
- Fxy: in-plane shear force
- Mxx: out-of-plane bending moment on the x face (about the local y axis)
- Myy: out-of-plane bending moment on the y face (about the local x axis)
- Mxy: out-of-plane torsional moment
- Vxx: out-of-plane shear force on the x face

- Vyy: out-of-plane shear force on the y face

The table can be printed with the Print button.

6.10 Shell Principal Forces & Moments

Click **Analysis Results > Shell Forces, Moments & Stresses > Shell Principal Forces & Moments** to see a table of shell principal forces and moments. The *Shell Principal Forces & Moments* table appears.



	Shell Id	Node Id	Fmax [kip/ft]	Fmin [kip/ft]	F-Angle [deg]	Mmax [kip-ft/ft]	Mmin [kip-ft/ft]	M-Angle [deg]	Vmax [kip/ft]	V-Angle [deg]
1	1	Center	-0.068	-2.438	71.604	0.000	0.000	0.000	0.000	0.000
2										
3	2	Center	0.593	-2.448	-72.051	0.000	0.000	0.000	0.000	0.000
4										
5	3	Center	1.426	-0.255	10.013	0.000	0.000	0.000	0.000	0.000
6										
7	4	Center	1.541	0.040	-19.358	0.000	0.000	0.000	0.000	0.000
8										
9	5	Center	-0.078	-0.241	-65.741	0.000	0.000	0.000	0.000	0.000
10										
11	6	Center	0.142	-0.298	-73.376	0.000	0.000	0.000	0.000	0.000
12										

Select the desired Load Combination.

Choose whether the table should display results for the selected entities or for all entities.

Specify the force and moment locations to be at the nodes and/or the center of each shell by clicking **Settings and Tools > Data Options**.

The table reports shears and moments for the desired shells.

The shell forces and moments include:

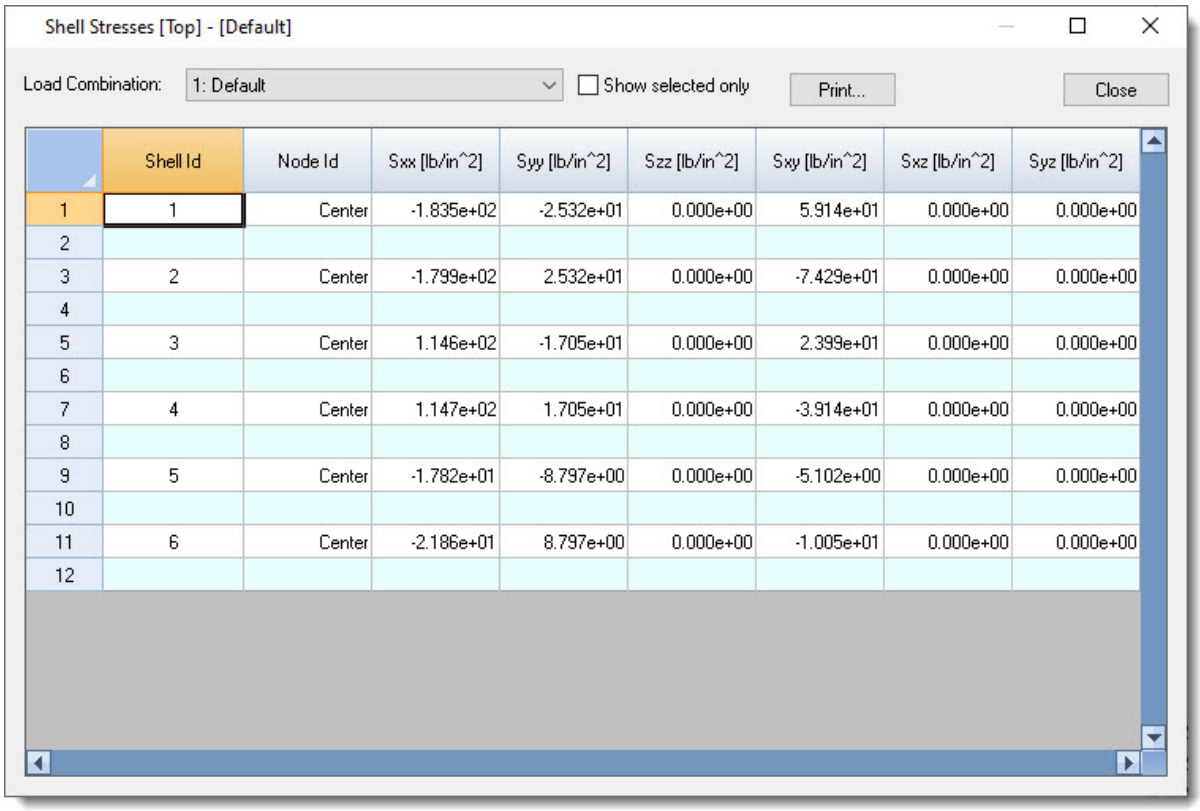
- Fmax: maximum normal force
- Fmin: minimum normal force
- F-Angle: angle at which extreme normal forces are achieved
- Mmax: maximum moment
- Mmin: minimum moment
- M-Angle: angle at which extreme moments are achieved

- Vmax: maximum shear force
- V-Angle: angle at which extreme shear forces are achieved

The table can be printed with the Print button.

6.11 Shell Stresses [Top or Bottom]

Click **Analysis Results > Shell Forces, Moments & Stresses > Shell Stresses [Top or Bottom]** to see a table of top or bottom shell stresses. The *Shell Stresses [Top or Bottom]* table appears.



	Shell Id	Node Id	Sxx [lb/in^2]	Syy [lb/in^2]	Szz [lb/in^2]	Sxy [lb/in^2]	Sxz [lb/in^2]	Syz [lb/in^2]
1	1	Center	-1.835e+02	-2.532e+01	0.000e+00	5.914e+01	0.000e+00	0.000e+00
2								
3	2	Center	-1.799e+02	2.532e+01	0.000e+00	-7.429e+01	0.000e+00	0.000e+00
4								
5	3	Center	1.146e+02	-1.705e+01	0.000e+00	2.399e+01	0.000e+00	0.000e+00
6								
7	4	Center	1.147e+02	1.705e+01	0.000e+00	-3.914e+01	0.000e+00	0.000e+00
8								
9	5	Center	-1.782e+01	-8.797e+00	0.000e+00	-5.102e+00	0.000e+00	0.000e+00
10								
11	6	Center	-2.186e+01	8.797e+00	0.000e+00	-1.005e+01	0.000e+00	0.000e+00
12								

Select the desired Load Combination.

Choose whether the table should display results for the selected entities or for all entities.

Specify the stress locations to be at the nodes and/or the center of each shell by clicking **Settings and Tools > Data Options**.

The table reports stresses for the desired shells.

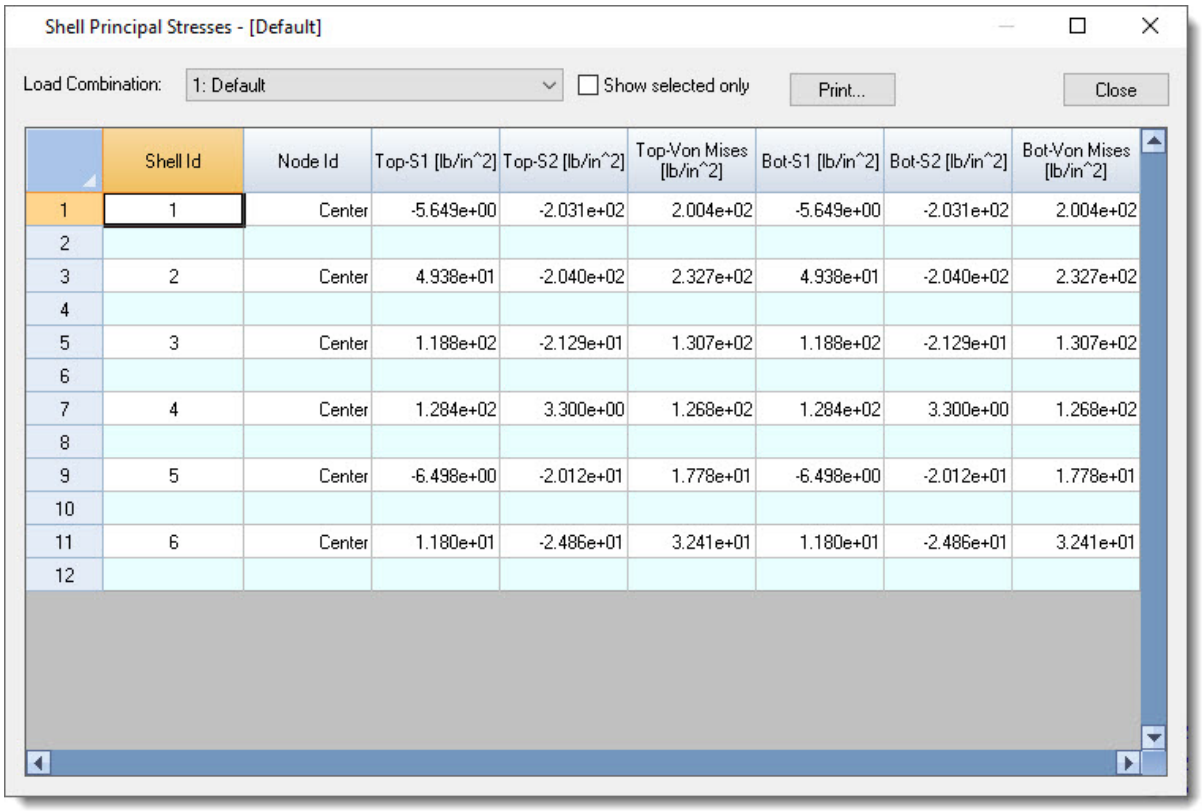
The shell stresses include:

- Sxx: in-plane normal axial stress in the local x direction
- Syy: in-plane normal axial stress in the local y direction
- Sxy: in-plane shear stress
- Sxz: out-of-plane shear on the x face
- Syz: out-of-plane shear on the y face

The table can be printed with the Print button.

6.12 Shell Principal Stresses

Click **Analysis Results > Shell Forces, Moments & Stresses > Shell Principal Stresses** to see a table of shell principal stresses. The *Shell Principal Stresses* table appears.



	Shell Id	Node Id	Top-S1 [lb/in^2]	Top-S2 [lb/in^2]	Top-Von Mises [lb/in^2]	Bot-S1 [lb/in^2]	Bot-S2 [lb/in^2]	Bot-Von Mises [lb/in^2]
1	1	Center	-5.649e+00	-2.031e+02	2.004e+02	-5.649e+00	-2.031e+02	2.004e+02
2								
3	2	Center	4.938e+01	-2.040e+02	2.327e+02	4.938e+01	-2.040e+02	2.327e+02
4								
5	3	Center	1.188e+02	-2.129e+01	1.307e+02	1.188e+02	-2.129e+01	1.307e+02
6								
7	4	Center	1.284e+02	3.300e+00	1.268e+02	1.284e+02	3.300e+00	1.268e+02
8								
9	5	Center	-6.498e+00	-2.012e+01	1.778e+01	-6.498e+00	-2.012e+01	1.778e+01
10								
11	6	Center	1.180e+01	-2.486e+01	3.241e+01	1.180e+01	-2.486e+01	3.241e+01
12								

Select the desired Load Combination.

Choose whether the table should display results for the selected entities or for all entities.

Specify the stress locations to be at the nodes and/or the center of each shell by clicking **Settings and Tools > Data Options**.

The table reports stresses for the desired shells.

The shell forces and moments include Top and Bottom values for:

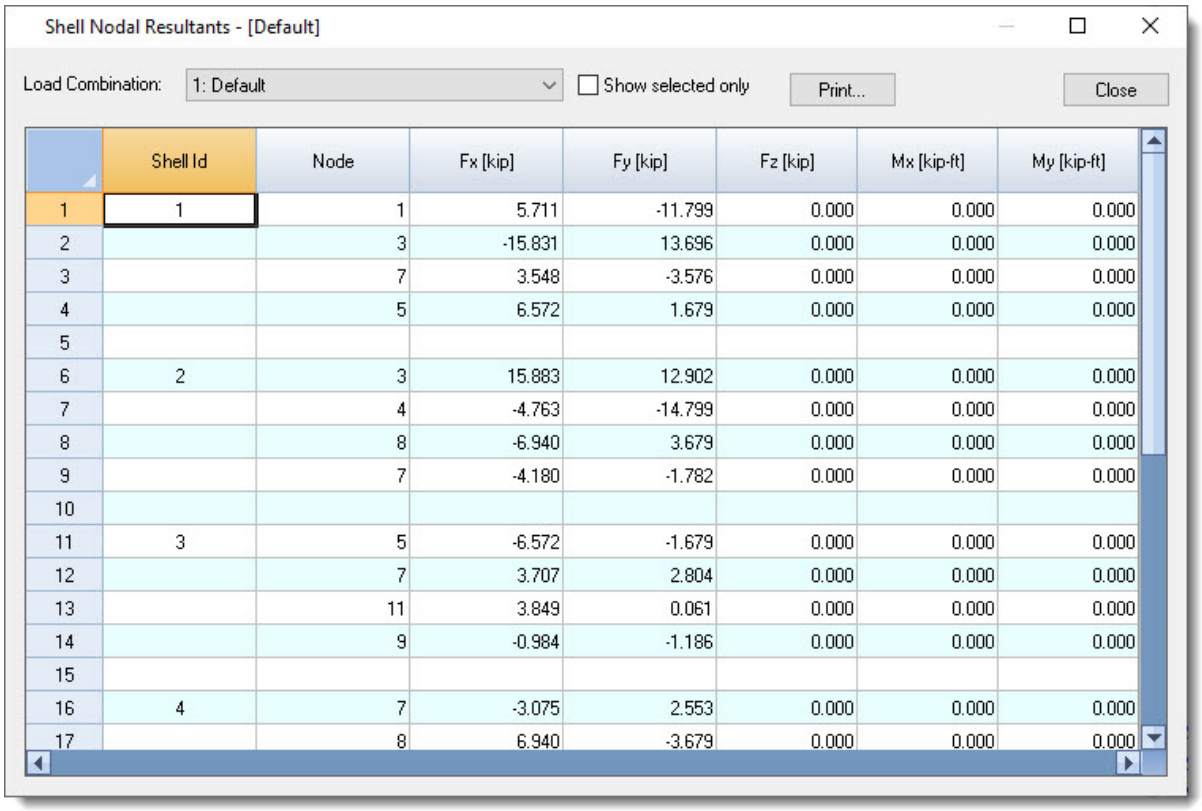
- S1: $$S_1 = \frac{\sigma_{xx} + \sigma_{yy}}{2} + \sqrt{\left(\frac{\sigma_{xx} - \sigma_{yy}}{2}\right)^2 + \sigma_{xy}^{\ 2}}$$

- S2: $$S_2 = \frac{\sigma_{xx} + \sigma_{yy}}{2} - \sqrt{(\frac{\sigma_{xx} - \sigma_{yy}}{2})^2 + \sigma_{xy}{}^2}$$
- S3: S3 = 0
- Von Mises: $$\sigma_{VonMises} = \sqrt{\frac{(S_1 - S_2)^2 + (S_1 - S_3)^2 + (S_2 - S_3)^2}{2}}$$

The table can be printed with the Print button.

6.13 Shell Nodal Resultants

Click **Analysis Results > Shell Forces, Moments & Stresses > Shell Nodal Resultants** to see a table of shell nodal resultants. The *Shell Nodal Resultants* table appears.



	Shell Id	Node	Fx [kip]	Fy [kip]	Fz [kip]	Mx [kip-ft]	My [kip-ft]
1	1	1	5.711	-11.799	0.000	0.000	0.000
2		3	-15.831	13.696	0.000	0.000	0.000
3		7	3.548	-3.576	0.000	0.000	0.000
4		5	6.572	1.679	0.000	0.000	0.000
5							
6	2	3	15.883	12.902	0.000	0.000	0.000
7		4	-4.763	-14.799	0.000	0.000	0.000
8		8	-6.940	3.679	0.000	0.000	0.000
9		7	-4.180	-1.782	0.000	0.000	0.000
10							
11	3	5	-6.572	-1.679	0.000	0.000	0.000
12		7	3.707	2.804	0.000	0.000	0.000
13		11	3.849	0.061	0.000	0.000	0.000
14		9	-0.984	-1.186	0.000	0.000	0.000
15							
16	4	7	-3.075	2.553	0.000	0.000	0.000
17		8	6.940	-3.679	0.000	0.000	0.000

Select the desired Load Combination.

Choose whether the table should display results for the selected entities or for all entities.

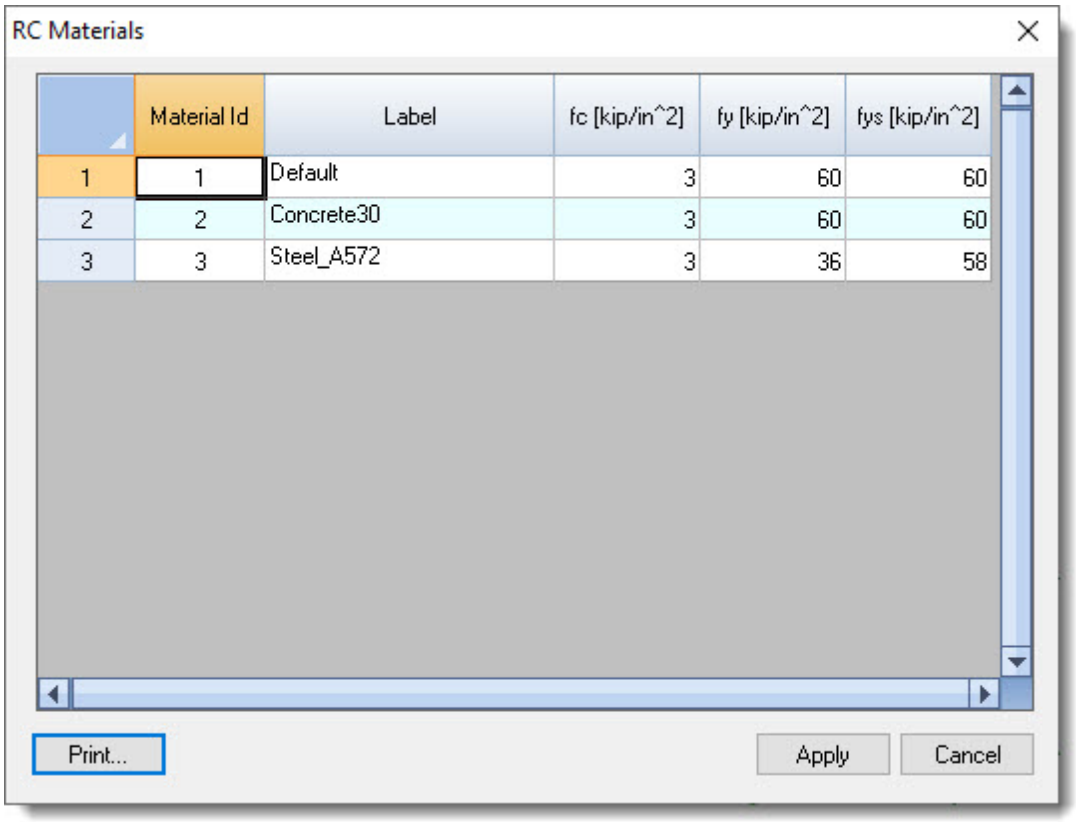
The table reports the forces and moments required to keep the shell in equilibrium. The values are expressed in the local coordinate system.

Part VII

7 Concrete Design

7.1 RC Materials

Click **Concrete Design > RC Materials** to view the *RC Materials* table. (This will list *any* materials that have been defined in the model, not just concrete materials.)



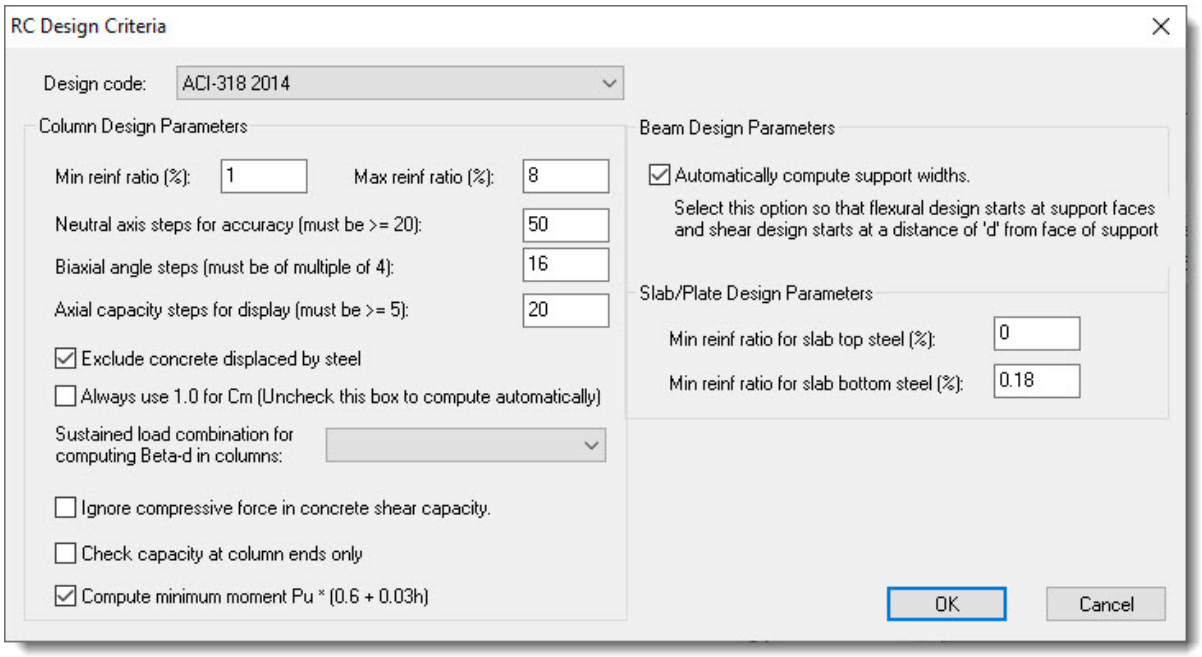
	Material Id	Label	fc [kip/in^2]	fy [kip/in^2]	fys [kip/in^2]
1	1	Default	3	60	60
2	2	Concrete30	3	60	60
3	3	Steel_A572	3	36	58

This table contains the materials that have been defined in **Tables > Materials**, but it lists their concrete design related properties as opposed to their analysis properties. The purpose of this table is to specify values for concrete strength, rebar yield strength and shear rebar yield strength for any concrete materials that are involved in a design.

Tip: The materials have probably already been assigned to the members in order to get an analysis. So at this point, there is no need to Assign anything specific to members. It is only necessary to specify the material strength values for any materials that are actually involved in a design.

7.2 RC Model Design Criteria

Click **Concrete Design > RC Model Design Criteria** to open the *RC Design Criteria* dialog.



This dialog controls many aspects of the concrete design process.

Design Code: Select the desired code.

<u>Column Design Parameters</u>

Minimum and Maximum reinforcing ratio: Enter values as whole number. Example: 1 for 1%, or 8 for 8%.

Neutral axis steps for accuracy: A control to balance accuracy with processing time. Do not use values less than 20.

Biaxial angle steps: Another control to balance accuracy with processing time. Must be a multiple of 4.

Axial capacity steps for display: A third control to balance accuracy with processing time. Must be 5 or more.

Exclude concrete displaced by steel: When selected, the program will exclude concrete displaced by rebar when calculating concrete capacity. This should be selected in practice. However, almost all examples in concrete textbooks include concrete displaced by rebar for calculation conveniences, so this can be useful if comparing program results to published examples.

Always use 1.0 for Cm: Uncheck this box to have the program compute Cm automatically.

Sustained load combination for computing Beta d in columns: Select the load combination that represents the sustained loads.

Ignore compressive force in concrete shear capacity: Offers the option to neglect the compressive force in a member when computing its shear capacity.

Check capacity at column ends only: If this is not selected, the concrete design procedure will check columns at every analysis segmental point along the member (more calculation time).

Compute minimum moment Pu*(0.6 + 0.03h): When selected, the program uses Pu*(0.6 + 0.03h) to calculate the minimum moment, and will consider both the minimum moment and the moment due to loading (whichever is larger). When this option is not selected, then the program will not compute the minimum moment and will use moment due to loading only.

Beam Design Parameters

Automatically compute support widths: Select this option so that flexural design starts at support faces and shear design starts at a distance of "d" from the face of support.

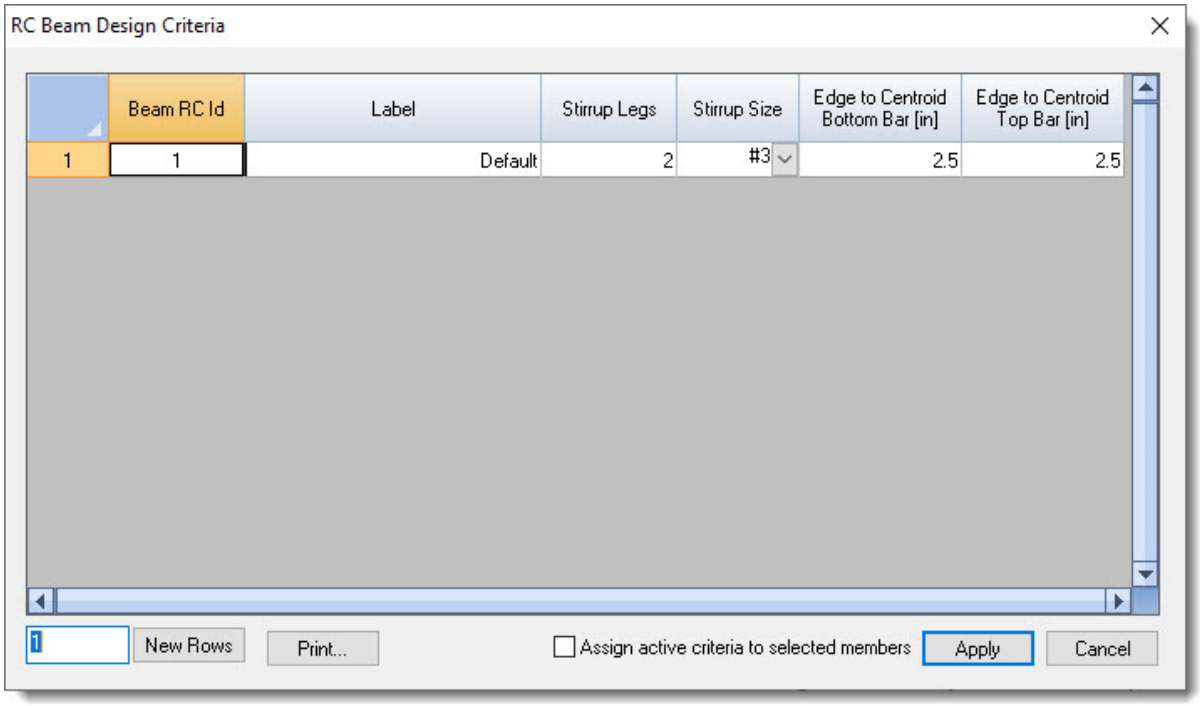
Slab/Plate Design Parameters

Minimum reinforcing ratio for slab top/bottom steel: The values entered will be treated as percentages. Example: 0.18 will be considered as 0.18% = 0.0018.

7.3 RC Design Criteria

7.3.1 RC Beam Design Criteria

Click **Concrete Design > RC Model Design Criteria > RC Beam Design Criteria** to open the *RC Beam Design Criteria* table.



This table collects design parameters for the concrete beam design process.

You can create as many RC Beam Design Criteria as you need to satisfy the requirements of your design. Once created, you can assign RC Beam Design Criteria to members graphically.

Label: A freeform text descriptor that has meaning to you.

Stirrup Legs: The number of legs per stirrup.

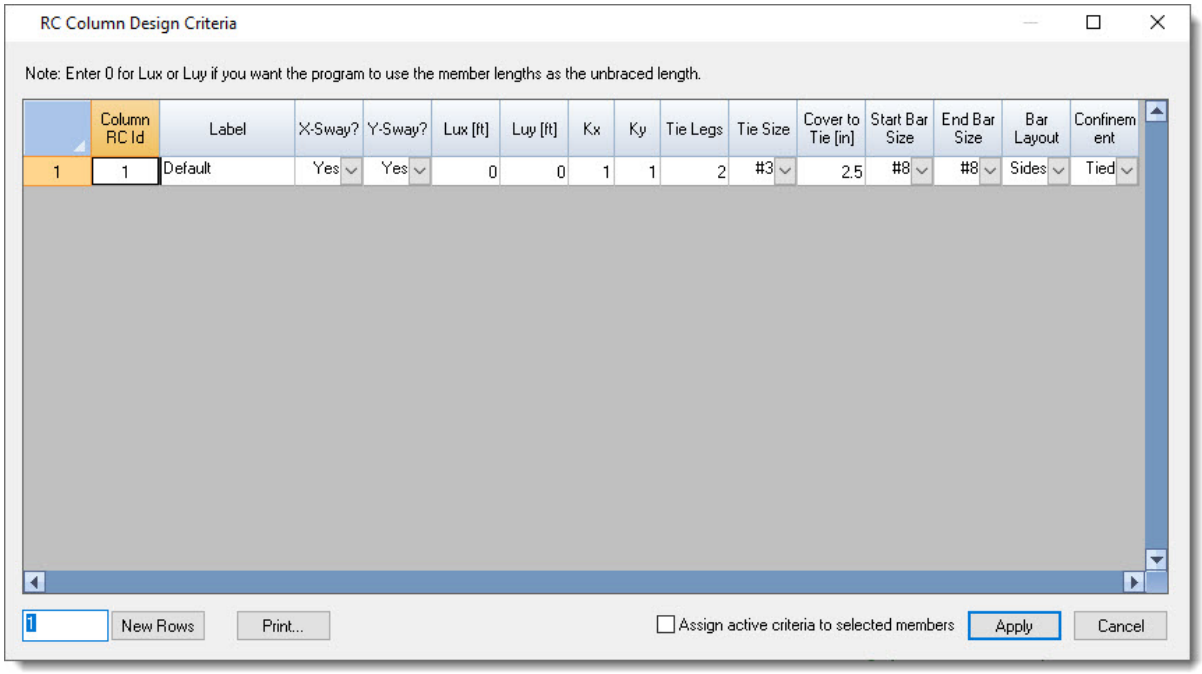
Stirrup Size: The rebar size used for stirrups.

Edge to Centroid Bottom Bar: The distance from the edge of concrete to the centroid of the bottom bars.

Edge to Centroid Top Bar: The distance from the edge of concrete to the centroid of the top bars.

7.3.2 RC Column Design Criteria

Click **Concrete Design > RC Model Design Criteria > RC Column Design Criteria** to open the *RC Column Design Criteria* table.



	Column RC Id	Label	X-Sway?	Y-Sway?	Lux [ft]	Luy [ft]	Kx	Ky	Tie Legs	Tie Size	Cover to Tie [in]	Start Bar Size	End Bar Size	Bar Layout	Confinement
1	1	Default	Yes	Yes	0	0	1	1	2	#3	2.5	#8	#8	Sides	Tied

This table collects design parameters for the concrete column design process.

You can create as many RC Column Design Criteria as you need to satisfy the requirements of your design. Once created, you can assign RC Column Design Criteria to members graphically.

Label: A freeform text descriptor that has meaning to you.

X-Sway & Y-Sway: X-sway and Y-sway flags designate if a member is a part of a sway frame in member strong/weak direction, respectively. These flags are only used for checking the validity of Kx and Ky values. For example, if X-sway flag is on and Kx is less than 1.0, the program will give a warning.

Lux: Unbraced length when considering buckling about the strong axis.

Luy: Unbraced length when considering buckling about the weak axis.

Kx: Effective length factor when considering buckling about the strong axis.

Ky: Effective length factor when considering buckling about the weak axis.

Tie Legs: Number of legs per tie.

Tie Size: Rebar size used for ties.

Cover to Tie: Specifies the clear cover to the tie or spiral reinforcing.

Start Bar Size: The bar size that the design process will start with when seeking a solution. Set this to the smaller size if different bar sizes are allowed in the solution.

End Bar Size: The bar size that the design process will end with when seeking a solution. Set this to the larger size if different bar sizes are allowed in the solution.

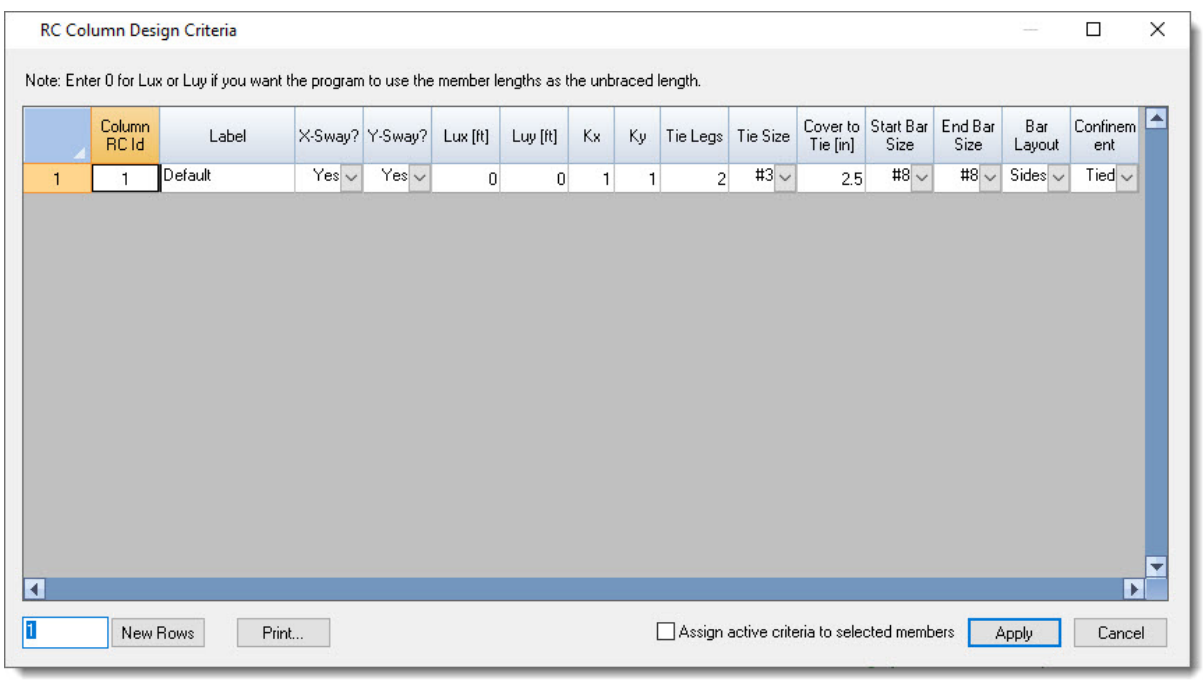
Bar Layout: See four options below:

- All Sides: Rebar will be placed on all sides
- Major Sides: Rebar will be placed on sides that are parallel to strong axis to resist Mz. No rebar on sides that are parallel to weak axis except at corners.
- Minor Sides: Rebar will be placed on sides that are parallel to weak axis to resist Mz. No rebars on sides that are parallel to strong axis except at corners.
- Equal Sides: Rebar placed in equal numbers on all sides.

Confinement: Used to specify whether the column is tied or spirally reinforced.

7.3.3 RC Plate Design Criteria

Click **Concrete Design > RC Model Design Criteria > RC Plate Design Criteria** to open the *RC Plate Design Criteria* table.



	Column RC Id	Label	X-Sway?	Y-Sway?	Lux [ft]	Luy [ft]	Kx	Ky	Tie Legs	Tie Size	Cover to Tie [in]	Start Bar Size	End Bar Size	Bar Layout	Confinement
1	1	Default	Yes	Yes	0	0	1	1	2	#3	2.5	#8	#8	Sides	Tied

This table collects design parameters for the concrete plate design process.

You can create as many RC Plate Design Criteria as you need to satisfy the requirements of your design. Once created, you can assign RC Plate Design Criteria to shells graphically.

Label: A freeform text descriptor that has meaning to you.

Edge to Centroid Bottom x Bar: Distance from the edge to the centroid of the bottom bars oriented parallel to the plate local x axis.

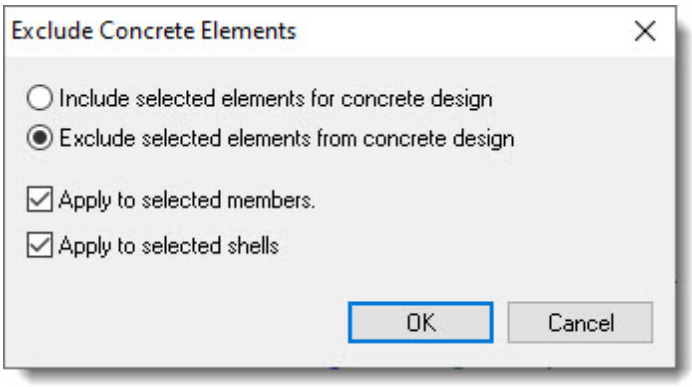
Edge to Centroid Bottom y Bar: Distance from the edge to the centroid of the bottom bars oriented parallel to the plate local y axis.

Edge to Centroid Top x Bar: Distance from the edge to the centroid of the top bars oriented parallel to the plate local x axis.

Edge to Centroid Top y Bar: Distance from the edge to the centroid of the top bars oriented parallel to the plate local y axis.

7.4 Exclude Concrete Elements

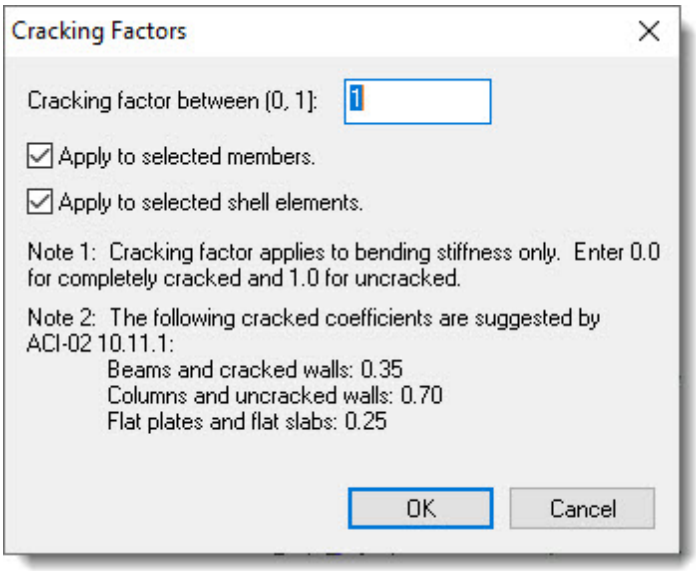
Click **Concrete Design > Exclude Concrete Elements** to open the *Exclude Elements* dialog.



This dialog provides a way to indicate that selected members and/or shells should be included or excluded from concrete design.

7.5 Cracking Factors

Click **Concrete Design > Cracking Factors** to open the *Cracking Factors* dialog.



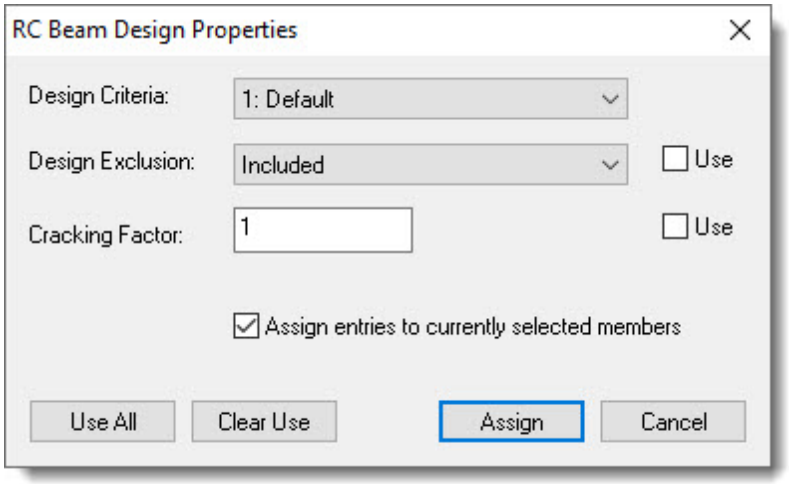
This dialog provides a way to specify cracking factors that will apply to the bending stiffness of selected members and/or shells. A value of 0 implies completely cracked. A value of 1.0 implies completely uncracked.

Tip: Regardless of how Cracking Factors have been set, they will only be considered if the option to "Use cracked section properties" has been selected in Analysis > Analysis Options.

7.6 RC Design Properties

7.6.1 Beam Design Properties

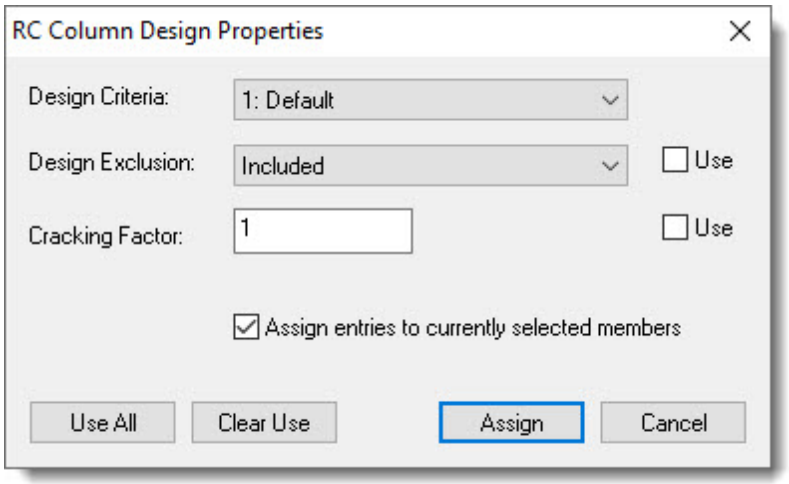
Click **Concrete Design > RC Design Properties > RC Beam Design Properties** to open the *RC Beam Design Properties* dialog.



This is a convenience dialog that allows the assignment of RC Beam Design Criteria and/or RC Beam Design Exclusion and/or Cracking Factor, all of which are explained in the sections above.

7.6.2 Column Design Properties

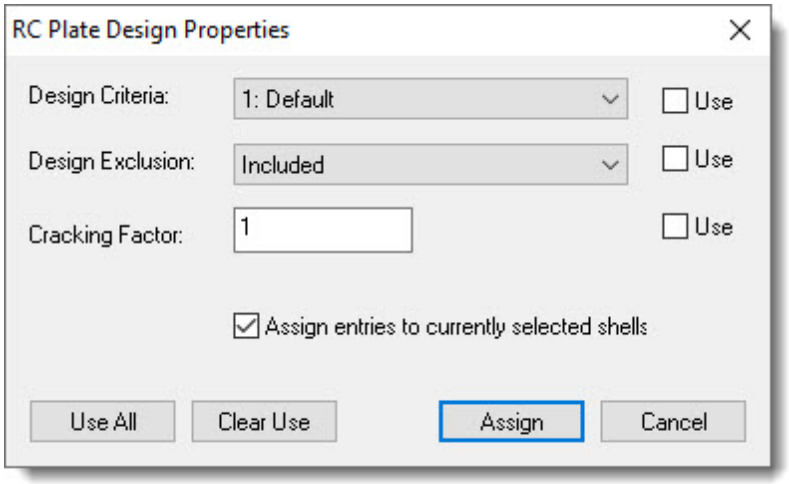
Click **Concrete Design > RC Design Properties > RC Column Design Properties** to open the *RC Column Design Properties* dialog.



This is a convenience dialog that allows the assignment of RC Column Design Criteria and/or RC Column Design Exclusion and/or Cracking Factor, all of which are explained in the sections above.

7.6.3 Plate Design Properties

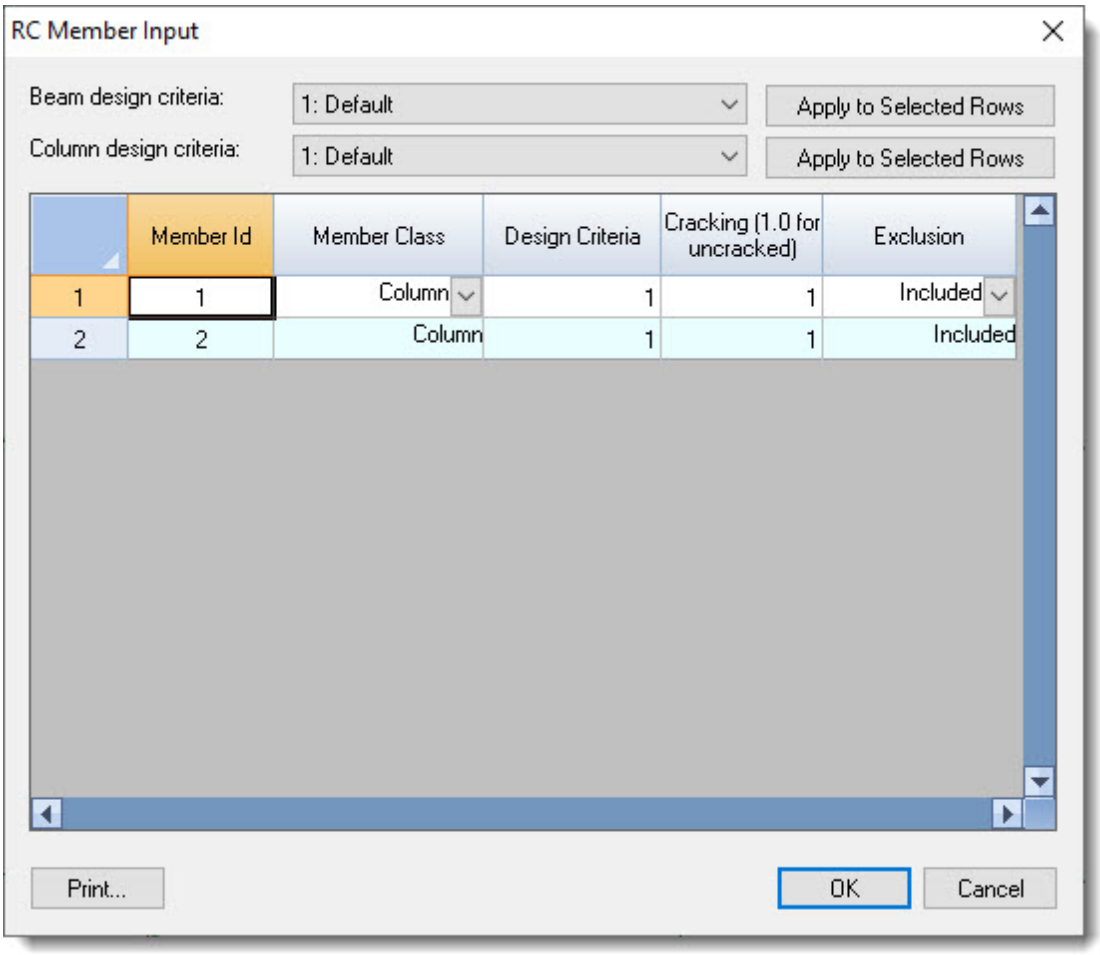
Click **Concrete Design > RC Design Properties > RC Plate Design Properties** to open the *RC Plate Design Properties* dialog.



This is a convenience dialog that allows the assignment of RC Plate Design Criteria and/or RC Plate Design Exclusion and/or Cracking Factor, all of which are explained in the sections above.

7.7 RC Member Input

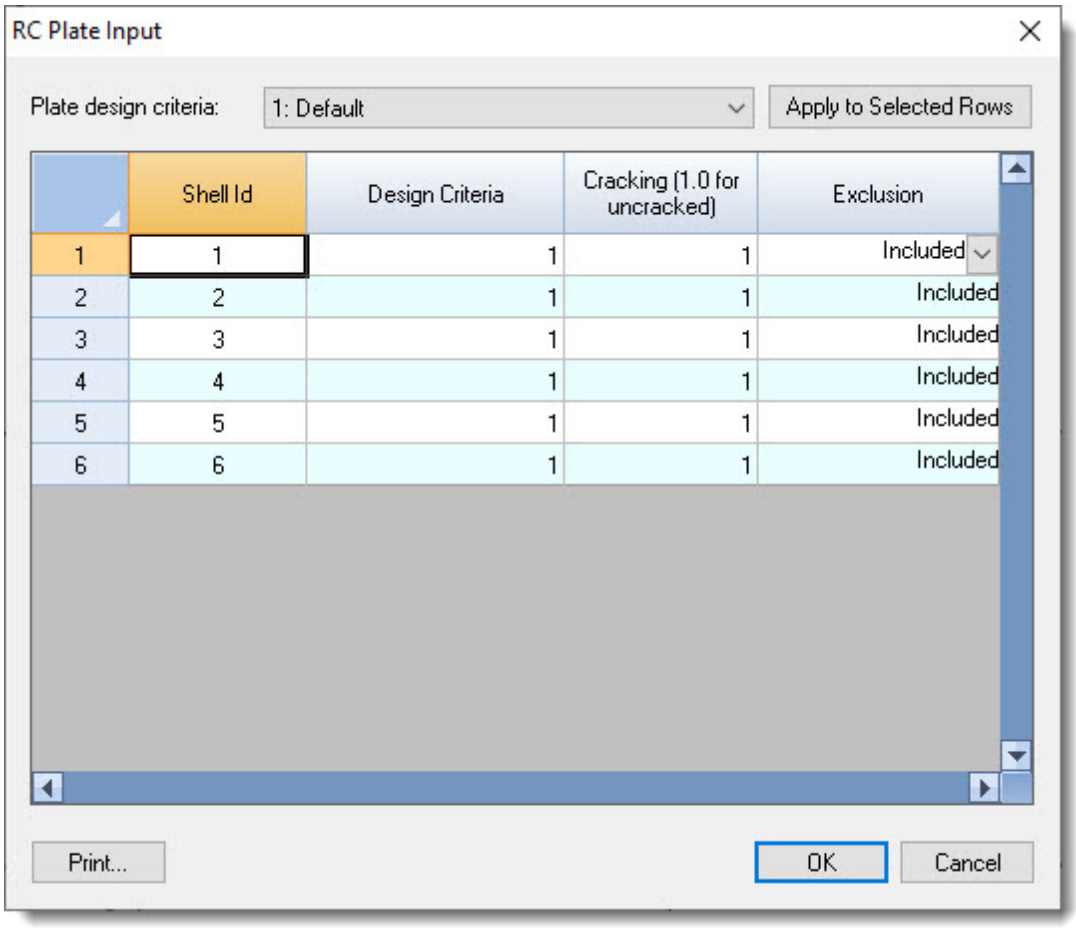
Click **Concrete Design > RC Member Input** to open the *RC Member Input* table.



This is a convenient table to review and change the assignments that have been made for RC Beam or RC Column Design Criteria, Cracking Factor, and the status of Inclusion/Exclusion.

7.8 RC Plate Input

Click **Concrete Design > RC Plate Input** to open the *RC Plate Input* table.



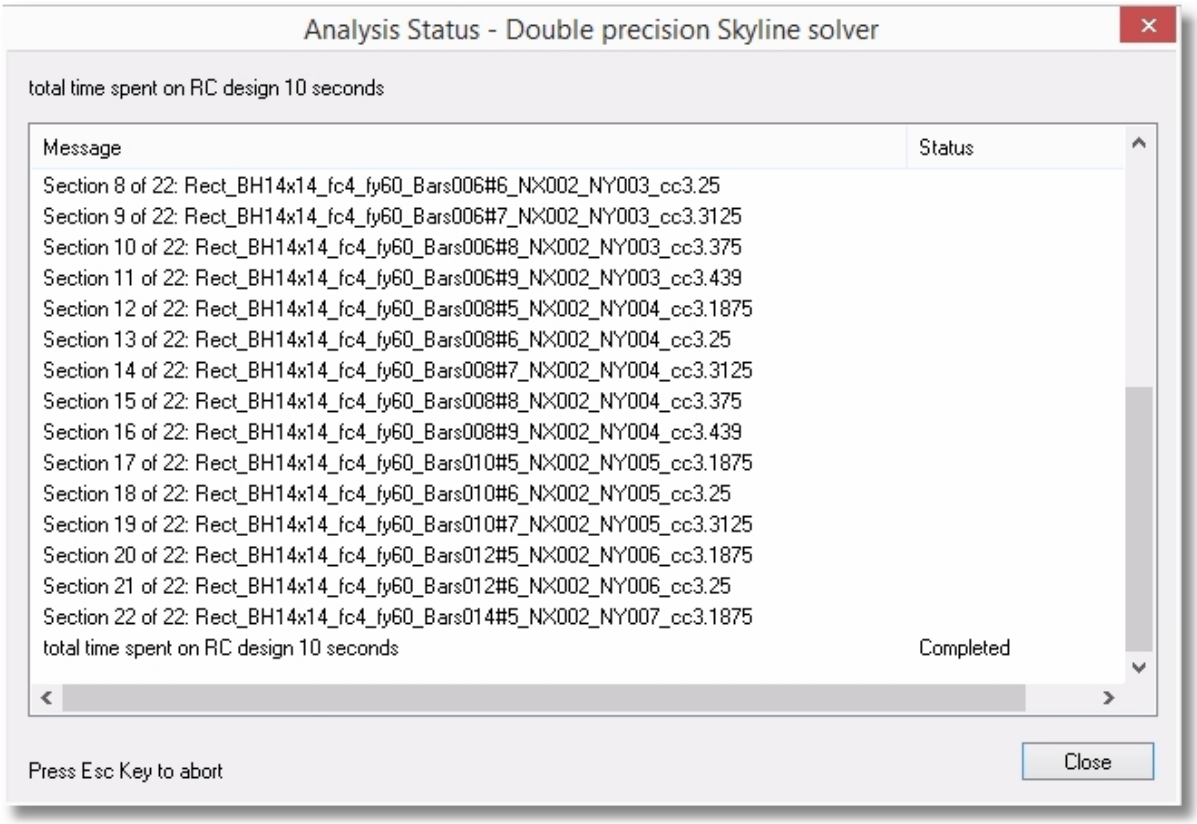
	Shell Id	Design Criteria	Cracking (1.0 for uncracked)	Exclusion
1	1	1	1	Included
2	2	1	1	Included
3	3	1	1	Included
4	4	1	1	Included
5	5	1	1	Included
6	6	1	1	Included

This is a convenient table to review and change the assignments that have been made for RC Plate Design Criteria, Cracking Factor, and the status of Inclusion/Exclusion.

7.9 Perform Concrete Design

Click **Concrete Design > Perform Concrete Design** to initiate a Concrete Design process. (Static analysis results must be current in order to run a concrete design.)

The message box will indicate when the design is complete.



7.10 Concrete Design Output

7.10.1 RC Analysis Envelope

Click **Concrete Design > Concrete Design Output > RC Analysis Envelope** to open the *RC Analysis Envelope* table.

RC Analysis Envelope

Load Combination: 1: Default ☐ Show selected only Print... Close

	Member Id	Distance (%L)	max Mz (Major Moment) [kip-ft]	min Mz (Major Moment) [kip-ft]	abs Fy (Major Shear) [kip]	max Mz Comb#	min Mz Comb#	abs Fy Comb#
1	1	0.000	0.000	-37.633	1.262	1	2	2
2		0.050	0.000	-37.288	1.250	1	2	2
3		0.100	0.000	-36.946	1.239	1	2	2
4		0.150	0.000	-36.607	1.227	1	2	2
5		0.200	0.000	-36.271	1.216	1	2	2
6		0.250	0.000	-35.938	1.204	1	2	2
7		0.300	0.000	-35.609	1.192	1	2	2
8		0.350	0.000	-35.282	1.181	1	2	2
9		0.400	0.000	-34.959	1.169	1	2	2
10		0.450	0.000	-34.639	1.158	1	2	2
11		0.500	0.000	-34.322	1.146	1	2	2
12		0.550	0.000	-34.009	1.135	1	2	2
13		0.600	0.000	-33.698	1.123	1	2	2
14		0.650	0.000	-33.391	1.111	1	2	2
15		0.700	0.000	-33.087	1.100	1	2	2
16		0.750	0.000	-32.786	1.088	1	2	2
17		0.800	0.000	-32.488	1.077	1	2	2

After selecting the desired load combination, this table reports forces and moments at small increments along the length of the member.

Max Mz: Maximum moment about the strong axis of the member.

Min Mz: Minimum moment about the strong axis of the member.

Abs Fy: Absolute value of the major axis shear in the member.

Max Mz Comb: Load Combination that produces the maximum moment about the strong axis of the member.

Min Mz Comb: Load Combination that produces the minimum moment about the strong axis of the member.

Abs Fy Comb: Load Combination that produces the absolute value of the major axis shear in the member.

7.10.2 RC Beam Results

Click **Concrete Design > Concrete Design Output > RC Beam Results** to open the *RC Beam Results* table.

RC Beam Results

Show selected only | Print... | Close

	Member Id	Distance (%L)	fc [kip/in^2]	fy [kip/in^2]	Bot-Mu [kip-ft]	Bot-As [in^2] (-1.0 means section too	Top-Mu [kip-ft]	Top-As [in^2] (-1.0 means section too
1	1							
2	Rect12x18	0.000	3.0	60.0	0.000	0.62	-96.647	1.53
3		0.050	3.0	60.0	0.000	0.62	-95.175	1.51
4		0.100	3.0	60.0	0.000	0.62	-93.727	1.48
5		0.150	3.0	60.0	0.000	0.62	-92.301	1.46
6		0.200	3.0	60.0	0.000	0.62	-90.898	1.43
7		0.250	3.0	60.0	0.000	0.62	-89.519	1.41
8		0.300	3.0	60.0	0.000	0.62	-88.162	1.39
9		0.350	3.0	60.0	0.000	0.62	-86.829	1.36
10		0.400	3.0	60.0	0.000	0.62	-85.518	1.34
11		0.450	3.0	60.0	0.000	0.62	-84.231	1.32
12		0.475	3.0	60.0	0.000	0.62	-83.596	1.31
13		0.500	3.0	60.0	0.000	0.62	-82.966	1.30
14		0.550	3.0	60.0	0.000	0.62	-81.725	1.27
15		0.600	3.0	60.0	0.000	0.62	-80.506	1.25
16		0.650	3.0	60.0	0.000	0.62	-79.311	1.23
17		0.700	3.0	60.0	0.000	0.62	-78.139	1.21

This table reports moments and required rebar areas at small increments along the length of the member.

fc and fy: Concrete strength and rebar yield strength.

Bot Mu: Moment producing tension on the bottom of the beam.

Bot As: Area of steel required to satisfy Bot Mu.

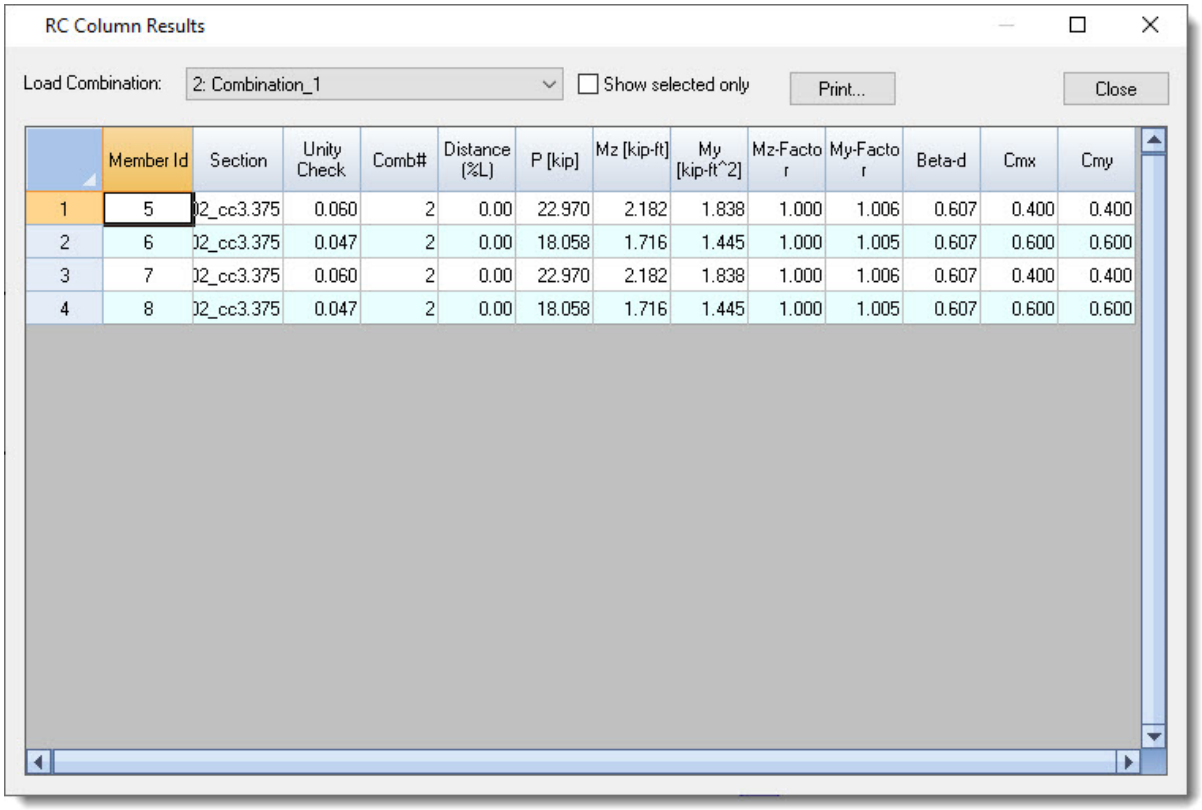
Top Mu: Moment producing tension on the top of the beam.

Top As: Area of steel required to satisfy Top Mu.

Tip: If Bot As or Top As indicate -1.0, that means that the section is too small for the loads.

7.10.3 RC Column Results

Click **Concrete Design > Concrete Design Output > RC Column Results** to open the *RC Column Results* table.



	Member Id	Section	Unity Check	Comb#	Distance (%L)	P [kip]	Mz [kip-ft]	My [kip-ft^2]	Mz-Factor	My-Factor	Beta-d	Cmx	Cmy
1	5	02_cc3.375	0.060	2	0.00	22.970	2.182	1.838	1.000	1.006	0.607	0.400	0.400
2	6	02_cc3.375	0.047	2	0.00	18.058	1.716	1.445	1.000	1.005	0.607	0.600	0.600
3	7	02_cc3.375	0.060	2	0.00	22.970	2.182	1.838	1.000	1.006	0.607	0.400	0.400
4	8	02_cc3.375	0.047	2	0.00	18.058	1.716	1.445	1.000	1.005	0.607	0.600	0.600

This table reports design results for each column member.

Tip: This table only reports results for the controlling load combination for each column. *The Load Combination selector is provided in this table only as a convenience to help the user relate the load combination number to the actual load combination itself*.

Section: A descriptor generated by the program that uniquely identifies the reinforcing pattern used.

Unity Check: The highest utilization ratio for all load combinations used in concrete design.

Comb: The load combination that produces the highest utilization ratio.

Distance: The location along the length of the member where the highest utilization ratio occurred. A value of zero indicates the starting end of the member. A value of 1.0 indicates the ending end. A value of 0.25 means 25% of the way from starting node to ending node.

P: The design axial load at the specified distance.

Mz: The design strong axis moment at the specified distance.

My: The design weak axis moment at the specified distance.

Mz Factor: The B1 moment magnification factor for Mz.

My Factor: The B1 moment magnification factor for My.

Beta d: The ratio of maximum factored axial sustained load to maximum factored axial total load. The factor Beta d accounts for the effects of creep. Generally:

$$\beta_d = \frac{Factored\ Dead\ Load}{Factored\ Dead\ Load + Factored\ Live\ Load}$$

Cmx and Cmy: The Equivalent Moment Factor:

Cm = 1. 0 if M1 = 0 or M2 = 0

Cm = 1. 0 if transverse load exists

If only end moments exist, then:

$$C_m = 0.6 + 0.4\frac{M_1}{M_2} \geq 0.4$$

7.10.4 Member Shear Results

Click **Concrete Design > Concrete Design Output > Member Shear Results** to open the *Member Shear Results* table.

Member Shear Results

Load Combination: 1: Default ☐ Show selected only Print... Close

	Member Id	Distance (%L)	fc [kip/in^2]	fys [kip/in^2]	Stirrup/tie-size	Stirrup/tie-legs	Shear [kip]	Axial [kip]	Stirrup/tie-spacing [in] (blank	phi-Vc [kip]	Comb#
1	1										
2	Rect12x18	0.000	3.0	60.0	#3	2	21.443	0.000	7.75	15.281	1
3		0.050	3.0	60.0	#3	2	21.443	0.000	7.75	15.281	1
4		0.100	3.0	60.0	#3	2	21.443	0.000	7.75	15.281	1
5		0.150	3.0	60.0	#3	2	21.443	0.000	7.75	15.281	1
6		0.200	3.0	60.0	#3	2	21.443	0.000	7.75	15.281	1
7		0.250	3.0	60.0	#3	2	21.334	0.000	7.75	15.281	3
8		0.300	3.0	60.0	#3	2	20.975	0.000	7.75	15.281	3
9		0.350	3.0	60.0	#3	2	20.616	0.000	7.75	15.281	3
10		0.400	3.0	60.0	#3	2	20.257	0.000	7.75	15.281	3
11		0.450	3.0	60.0	#3	2	19.898	0.000	7.75	15.281	3
12		0.475	3.0	60.0	#3	2	19.719	0.000	7.75	15.281	3
13		0.500	3.0	60.0	#3	2	19.539	0.000	7.75	15.281	3
14		0.550	3.0	60.0	#3	2	19.181	0.000	7.75	15.281	3
15		0.600	3.0	60.0	#3	2	18.822	0.000	7.75	15.281	3
16		0.650	3.0	60.0	#3	2	18.463	0.000	7.75	15.281	3
17		0.700	3.0	60.0	#3	2	18.104	0.000	7.75	15.281	3

This table reports shear design results for each member. Beam shear is based on the shear envelope, so results for specific load cases are not available.

Member ID: Lists the member number and indicates the section used for the member.

Distance: Reports incremental locations along the member, starting at the starting end and ending at the ending end.

fc: The compressive strength of concrete.

fys: The yield strength of the shear reinforcing.

Stirrup/Tie Size: The size of the stirrup or tie.

Stirrup/Tie Legs: The number of legs per stirrup or tie.

Shear: The design shear force to be resisted.

Axial: The design axial force.

Stirrup/Tie Spacing: The spacing of stirrups or ties.

phiVc: The shear strength of the concrete only.

7.10.5 Wood-Armer Moments

To see a table of Wood-Armer moments click **Concrete Design > Concrete Design Output > Wood-Armer Moments**. The *Wood-Armer Moments* table appears.

Wood-Armer Moments

Load Combination: 1: Default ☐ Show selected only Print... Close

	Shell Id	Node Id	Bot-Mux [kip-ft/ft]	Bot-Muy [kip-ft/ft]	Top-Mux [kip-ft/ft]	Top-Muy [kip-ft/ft]	Bot-Mux Comb#	Bot-Muy Comb#	Top-Mux Comb#	Top-Muy Comb#
1	1	Center	0.000	2.740	-54.699	-6.603	1	3	3	3
2										
3	2	Center	0.000	18.757	-40.245	-3.896	1	3	3	3
4										
5	3	Center	0.000	13.315	-11.388	0.000	1	3	3	1
6										
7	4	Center	0.000	18.061	-20.636	0.000	1	3	3	1
8										
9	5	Center	0.000	11.856	-51.062	-15.327	1	3	3	3
10										
11	6	Center	0.000	9.492	-18.269	0.000	1	3	3	1
12										
13	7	Center	0.000	3.578	-10.933	-4.339	1	3	3	3
14										
15	8	Center	0.000	1.054	-6.141	-2.621	1	3	3	3
16										
17	9	Center	0.000	1.553	-6.532	-1.214	1	3	3	3

Tip: This table only reports results for the controlling load combination for each moment location and direction. *The Load Combination selector is provided in this table only as a convenience to help the user relate the load combination number to the actual load combination itself.*

Choose whether the table should display results for the selected entities or for all entities.

Tip: Specify the force and moment locations to be at the nodes and/or the center of each shell by clicking Settings > Data Options.

The table reports top and bottom moments for the desired shells.

Tip: Be sure to have a thorough understanding of the meaning of the "top" and "bottom" of a plate before interpreting the results in this table!

Bot Mux: Wood-Armer design moment on the x edge of the plate (about the local y axis) for sizing bottom rebar parallel to the plate local x axis.

Bot Muy: Wood-Armer design moment on the y edge of the plate (about the local x axis) for sizing bottom rebar parallel to the plate local y axis.

Top Mux: Wood-Armer design moment on the x edge of the plate (about the local y axis) for sizing top rebar parallel to the plate local x axis.

Top Muy: Wood-Armer design moment on the y edge of the plate (about the local x axis) for sizing top rebar parallel to the plate local y axis.

Bot Mux Comb: The load combination responsible for producing Bot Mux.

Bot Muy Comb: The load combination responsible for producing Bot Muy.

Top Mux Comb: The load combination responsible for producing Top Mux.

Top Muy Comb: The load combination responsible for producing Top Muy.

7.10.6 RC Plate Results

Click **Concrete Design > Concrete Design Output > RC Plate Results** to display a table of plate design results for the controlling load combination. The *RC Plate Results* table appears.

RC Plate Results

Show selected only | Print... | Close

	Shell Id	Node Id	Design-H [in]	fc [kip/in^2]	fy [kip/in^2]	Bot-Mux [kip-ft/ft]	Bot-Muy [kip-ft/ft]	Top-Mux [kip-ft/ft]	Top-Muy [kip-ft/ft]	Bot-Asx [in^2]/ft	Bot-Asy [in^2]/ft	Top-Asx [in^2]/ft	Top-Asy [in^2]/ft
1	1	Center	12.00	3.0	60.0	0.000	2.740	-54.699	-6.603	0.259	0.259	1.411	0.149
2													
3	2	Center	12.00	3.0	60.0	0.000	18.757	-40.245	-3.896	0.259	0.435	0.991	0.087
4													
5	3	Center	12.00	3.0	60.0	0.000	13.315	-11.388	0.000	0.259	0.305	0.260	0.000
6													
7	4	Center	12.00	3.0	60.0	0.000	18.061	-20.636	0.000	0.259	0.419	0.481	0.000
8													
9	5	Center	12.00	3.0	60.0	0.000	11.856	-51.062	-15.327	0.259	0.271	1.301	0.353
10													
11	6	Center	12.00	3.0	60.0	0.000	9.492	-18.269	0.000	0.259	0.259	0.424	0.000
12													
13	7	Center	12.00	3.0	60.0	0.000	3.578	-10.933	-4.339	0.259	0.259	0.249	0.097
14													
15	8	Center	12.00	3.0	60.0	0.000	1.054	-6.141	-2.621	0.259	0.259	0.138	0.059
16													
17	9	Center	12.00	3.0	60.0	0.000	1.553	-6.532	-1.214	0.259	0.259	0.147	0.027

Choose whether the table should display results for the selected entities or for all entities.

Tip: Specify the force and moment locations to be at the nodes and/or the center of each shell by clicking Settings and Tools > Data Options.

The table reports top and bottom moments for the desired shells.

Tip: Be sure to have a thorough understanding of the meaning of the "top" and "bottom" of a plate before interpreting the results in this table!

Design H: The plate thickness.

fc: The compressive strength of concrete.

fy: Yield strength of rebar.

Bot Mux: Controlling design moment on the x edge of the plate (about the local y axis) for sizing bottom rebar parallel to the plate local x axis.

Bot Muy: Controlling design moment on the y edge of the plate (about the local x axis) for sizing bottom rebar parallel to the plate local y axis.

Top Mux: Controlling design moment on the x edge of the plate (about the local y axis) for sizing top rebar parallel to the plate local x axis.

Top Muy: Controlling design moment on the y edge of the plate (about the local x axis) for sizing top rebar parallel to the plate local y axis.

Bot Asx: Area of steel required to resist Bot Mux.

Bot Asy: Area of steel required to resist Bot Muy.

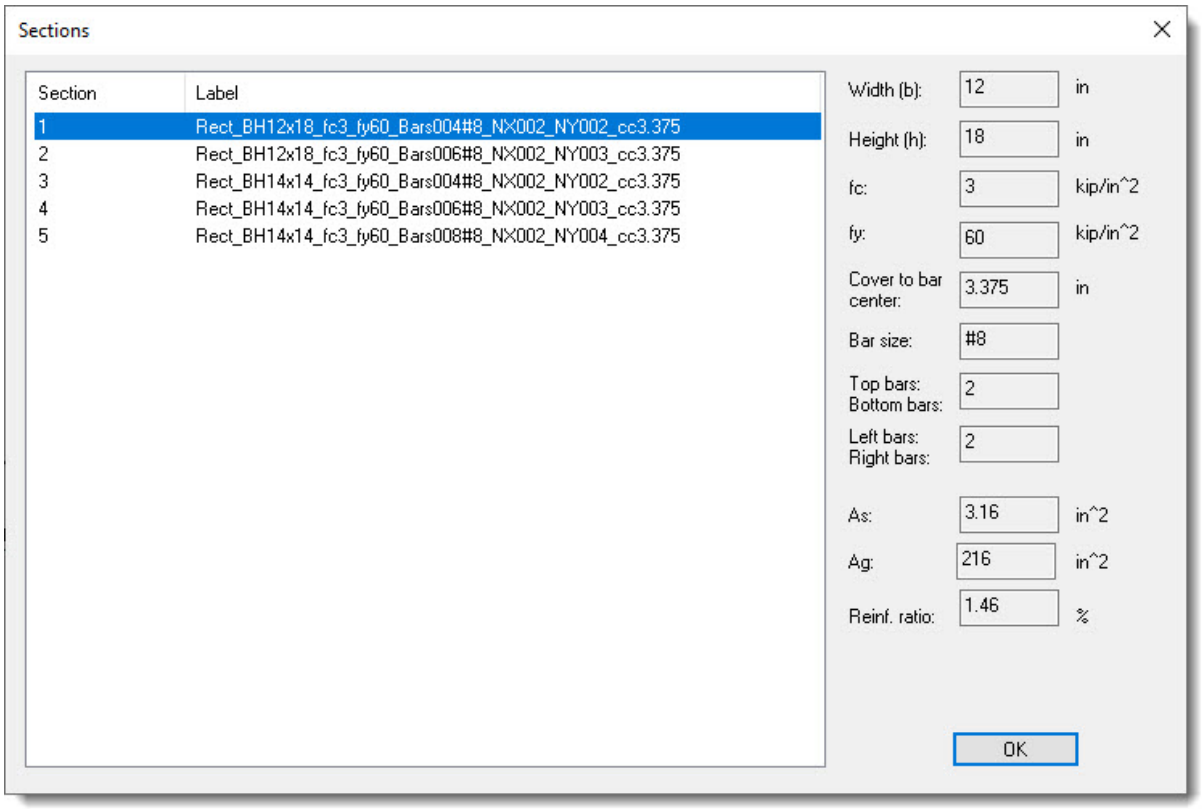
Top Asx: Area of steel required to resist Top Mux.

Top Asy: Area of steel required to resist Top Muy.

7.10.7 Flexural/Axial Interaction

7.10.7.1 Sections

Click **Concrete Design > Flexural/Axial Interaction > Sections** to open the *Sections* dialog.



This lists the various sections that met the specified RC Column Design Criteria and were tried as potential candidates for the solution to one or more members in the model.

Clicking on any section in the list will display all of the numerical data for that section in the labeled fields on the right side of the dialog.

7.10.7.2 P-Mx (+)

Click **Concrete Design > Flexural/Axial Interaction > P-Mx (+)** to open the table named *P-Mx (+), Compression on Top*.

P-Mx(+), Compression on Top

Section: 1: Rect_BH12x18_fc3_fy60_Bars004#8_NX002_NY002_cc3.375 Print... Close

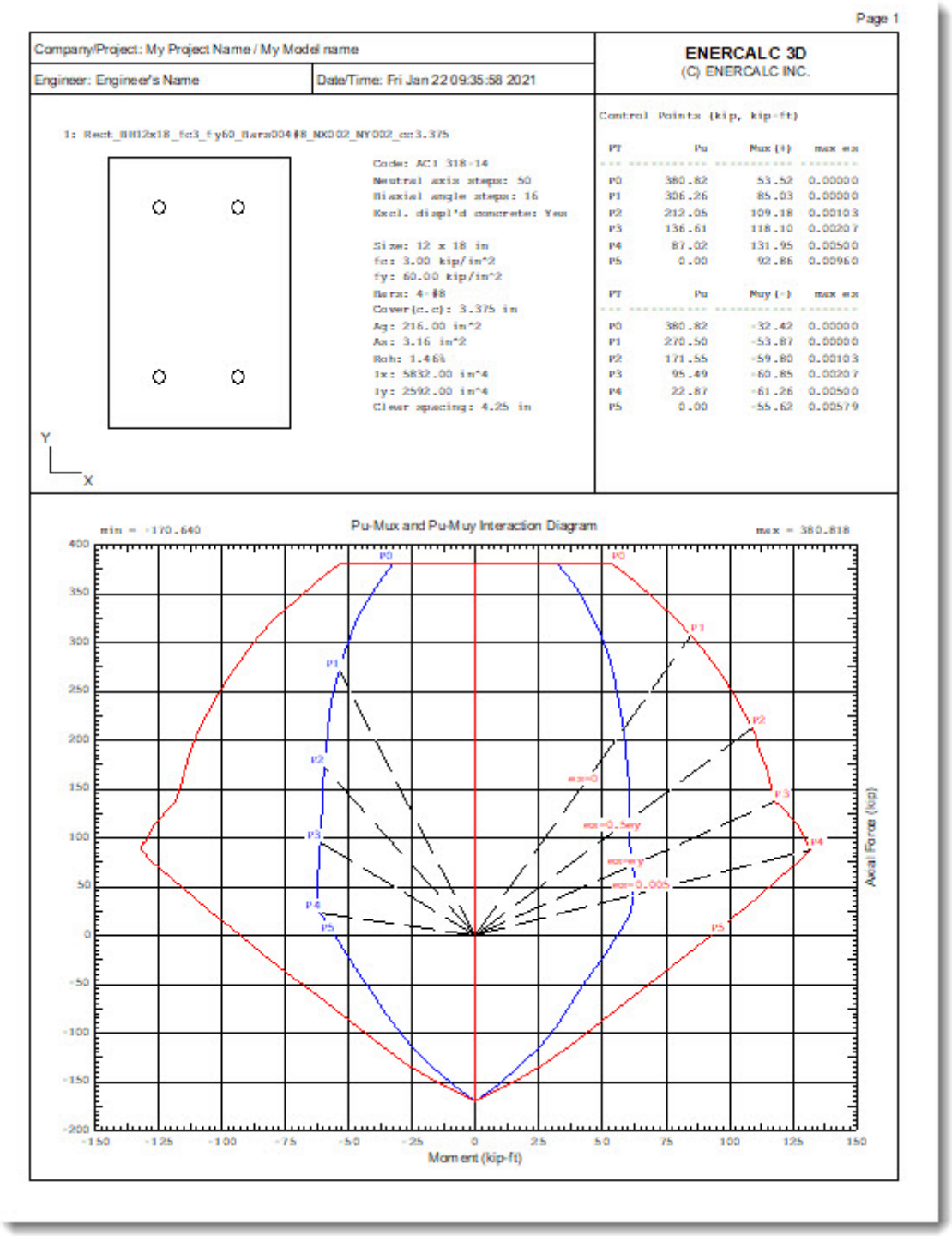
	Neutral Axis Depth [in]	phi * Pn [kip]	phi * Mnx [kip-ft]	Eccentricity [in]	Maximum Steel Tensile Strain	Phi
1	[Pure Compression]	380.818	0.000	0.00	0.00000	0.650
2	18.16	380.816	53.522	1.69	0.00000	0.650
3	17.25	361.777	62.985	2.09	0.00000	0.650
4	16.25	342.736	71.365	2.50	0.00000	0.650
5	15.39	323.695	78.862	2.92	0.00000	0.650
6	14.62	306.259	85.028	3.33	0.00000	0.650
7	14.56	304.654	85.562	3.37	0.00001	0.650
8	13.74	285.613	91.504	3.84	0.00019	0.650
9	13.12	270.495	95.755	4.25	0.00035	0.650
10	12.96	266.572	96.813	4.36	0.00039	0.650
11	12.47	254.492	99.877	4.71	0.00052	0.650
12	12.20	247.532	101.542	4.92	0.00060	0.650
13	11.48	228.491	105.804	5.56	0.00082	0.650
14	11.05	216.896	108.204	5.99	0.00097	0.650
15	10.88	212.049	109.178	6.18	0.00103	0.650
16	10.79	209.450	109.543	6.28	0.00107	0.650
17	10.19	190.409	112.051	7.06	0.00131	0.650

For the section selected in the dropdown list box at the top, this table presents interaction results for various neutral axis depths. The results include phiPn, phiMn, Eccentricity, Maximum Steel Tensile Strain, and the value of Phi.

Other selections are available to view P-Mx (-), P-My (+), P-My (-) and P-Mx-My.

7.10.7.3 Print Diagrams

Click **Concrete Design > Flexural/Axial Interaction > Print Diagrams** to allow diagrams to be printed for all sections considered in the current model.



The diagrams include a section diagram, section data, an interaction diagram, and data for the control points on the interaction diagram.

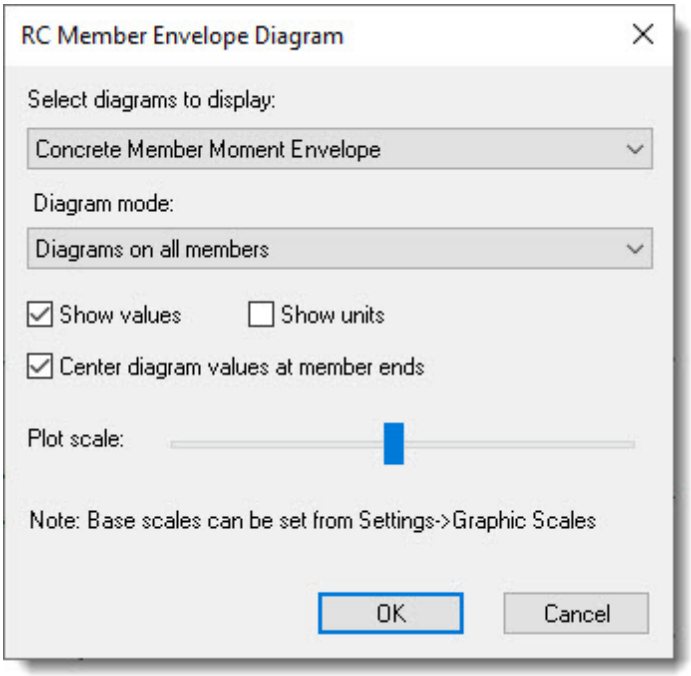
These diagrams can be zoomed and printed.

Click the Close button to return to the regular program display.

7.11 Concrete Design Diagrams

7.11.1 RC Member Envelope Diagram

To view member diagrams on the screen click **Concrete Design > Concrete Design Diagrams > RC Member Envelope Diagram**. The *RC Member Envelope Diagram* dialog opens.

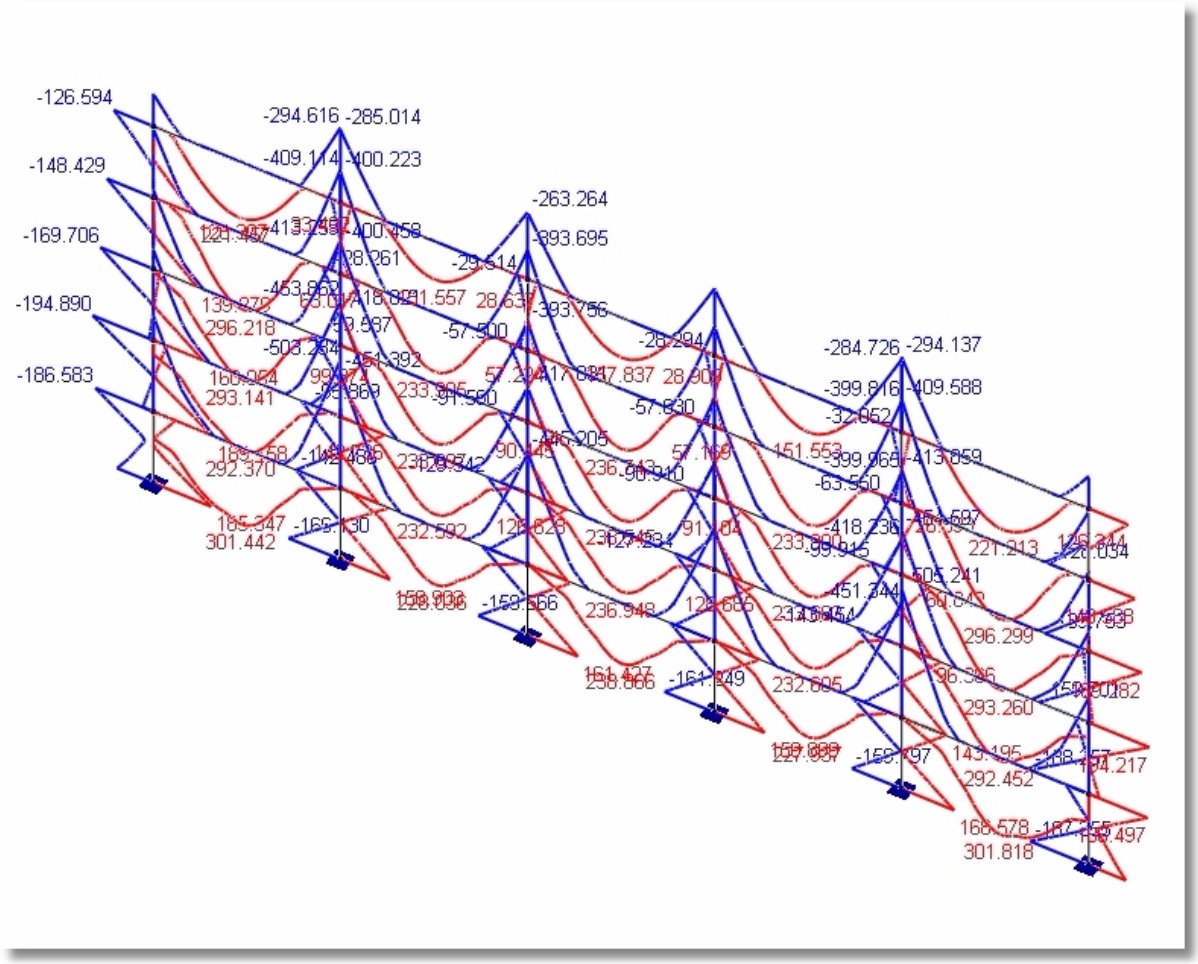


Remember that these are envelope diagrams. They are not specific to any one load combination.

Select Diagram to Display: Select from Concrete Member Moment Envelope, Concrete Member Shear Envelope, Required Flexural Reinforcement, or Required Shear Reinforcement.

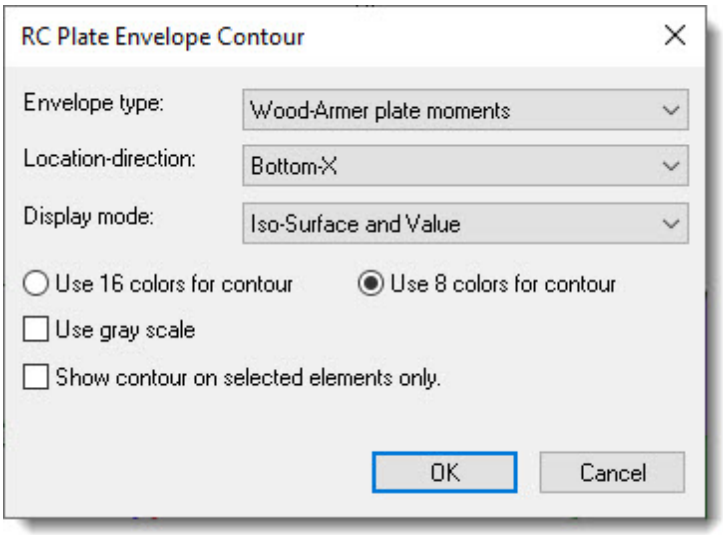
Diagram mode: Options include Diagrams on all members, Diagrams on selected members, or Erase existing diagrams.

Miscellaneous options include: Show values, Show units, and Center diagram values at member ends.



7.11.2 RC Plate Envelope Contour

To view plate contours on the screen click **Concrete Design > Concrete Design Diagrams > RC Plate Envelope Contour**. The *RC Plate Envelope Contour* dialog opens.



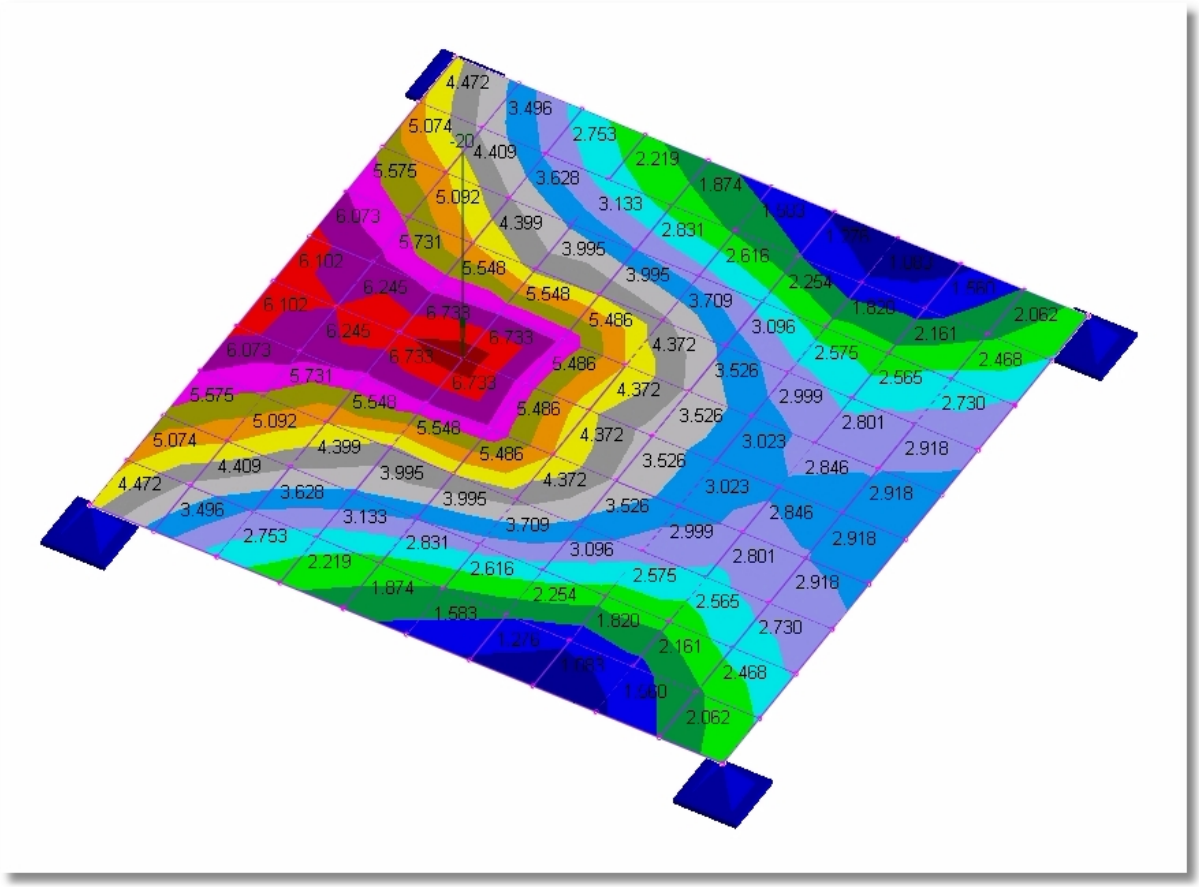
Envelope type: Select from Wood-Armer plate moments or Required plate reinforcement.

Location-direction: Select from Bottom-X, Bottom-Y, Top-X, or Top-Y.

Tip: Remember that these directions describe moment on the X or Y edge of the plate or reinforcement parallel to the plate local x or y direction.

Display mode: Options include Iso-Surface and Value, Iso-Surface only, Value only, or None/Erase.

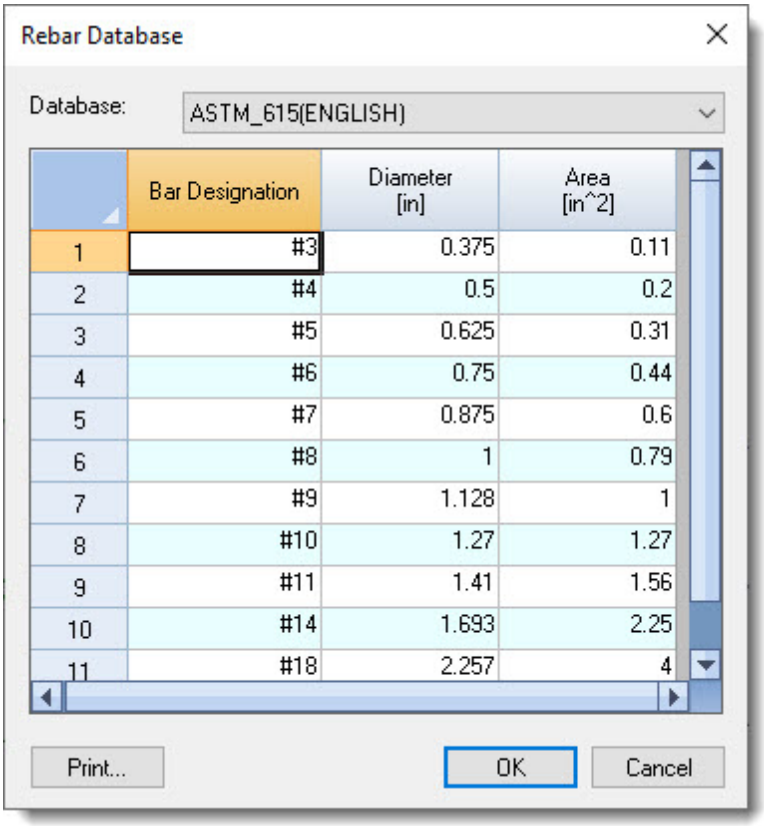
Miscellaneous options include: Use 8 or 16 colors, Use gray scale, Show contour on selected elements only.



7.12 Concrete Design Tools

7.12.1 Rebar Database

Click **Concrete Design > Concrete Design Tools > Rebar Database** to open the *Rebar Database* table.

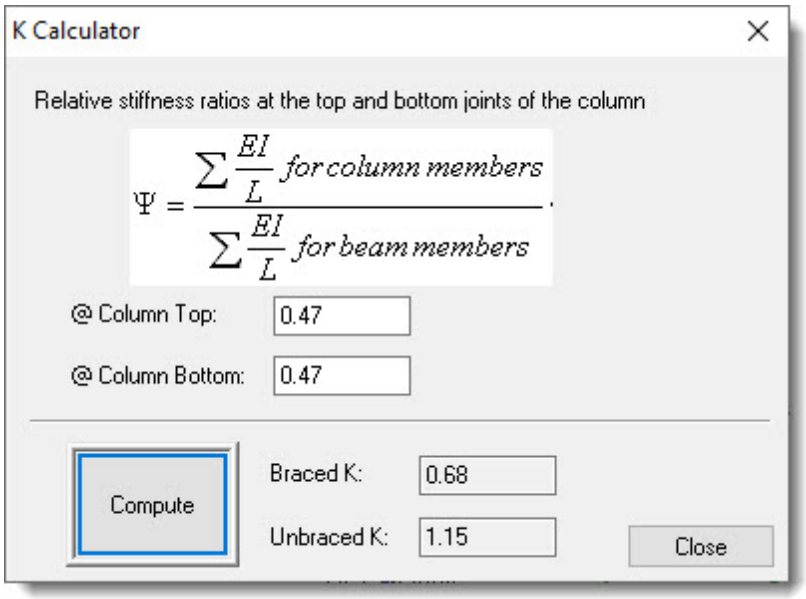


	Bar Designation	Diameter [in]	Area [in^2]
1	#3	0.375	0.11
2	#4	0.5	0.2
3	#5	0.625	0.31
4	#6	0.75	0.44
5	#7	0.875	0.6
6	#8	1	0.79
7	#9	1.128	1
8	#10	1.27	1.27
9	#11	1.41	1.56
10	#14	1.693	2.25
11	#18	2.257	4

Select the desired database to view bar designations, diameters, and areas.

7.12.2 K Calculator

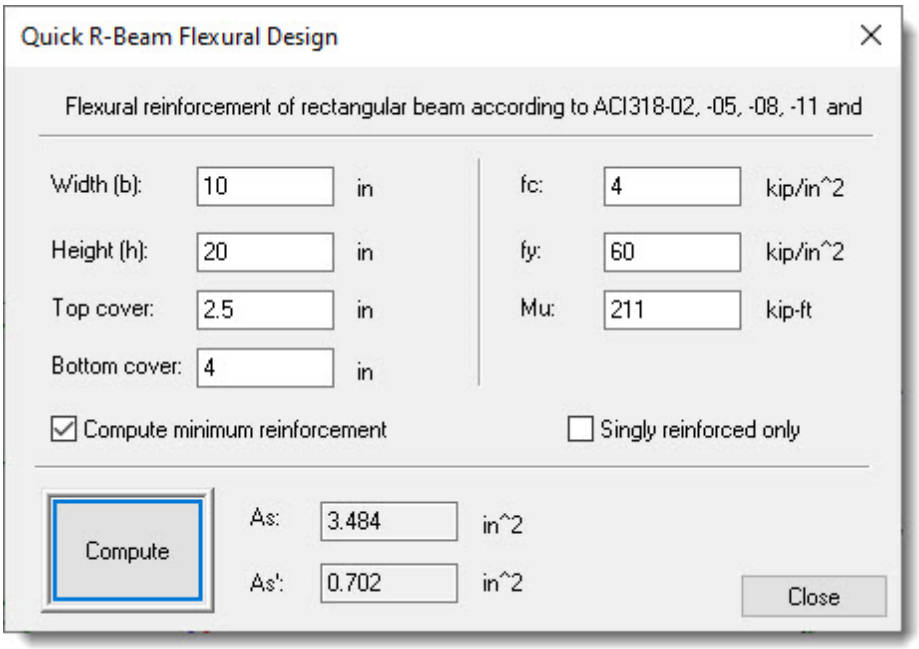
Click **Concrete Design > Concrete Design Tools > K Calculator** to open the *Compute Effective Length Factor K* dialog.



The dialog collects a value of Psi Top and Psi Bottom and then returns the calculated value of K for a braced and for an unbraced condition.

7.12.3 Quick Rectangular Beam Flexural Design

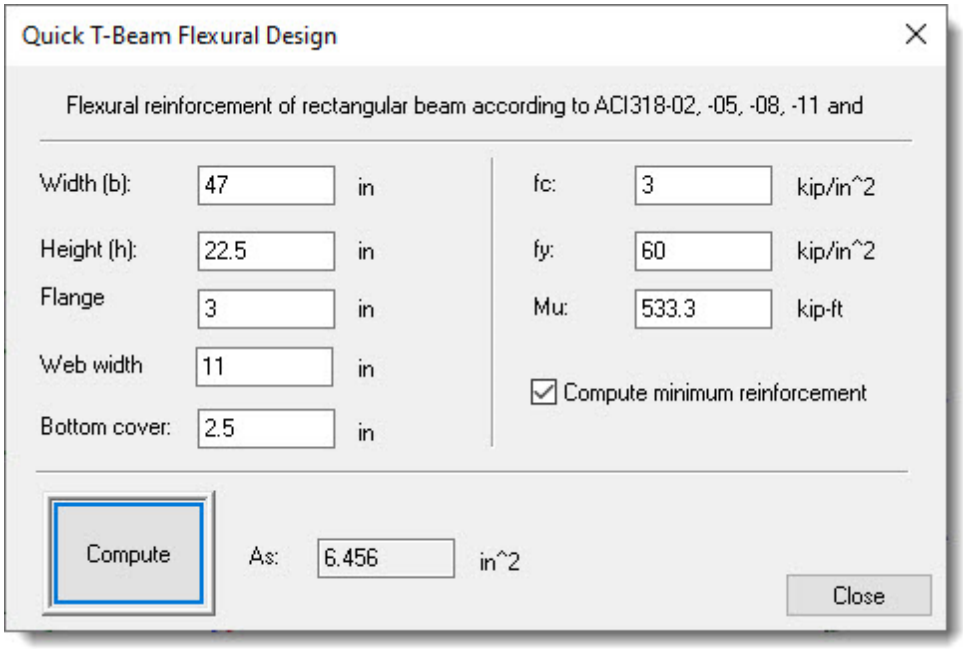
Click **Concrete Design > Concrete Design Tools > Quick R-Beam Flexural Design** to open the *Quick R-Beam Flexural Design* dialog.



This dialog collects the width, height, clear cover, fc, fy, and Mu, and then returns the required area of steel for either a singly reinforced or doubly reinforced beam.

7.12.4 Quick Tee Beam Flexural Design

Click **Concrete Design > Concrete Design Tools > Quick T-Beam Flexural Design** to open the *Quick T-Beam Flexural Design* dialog.



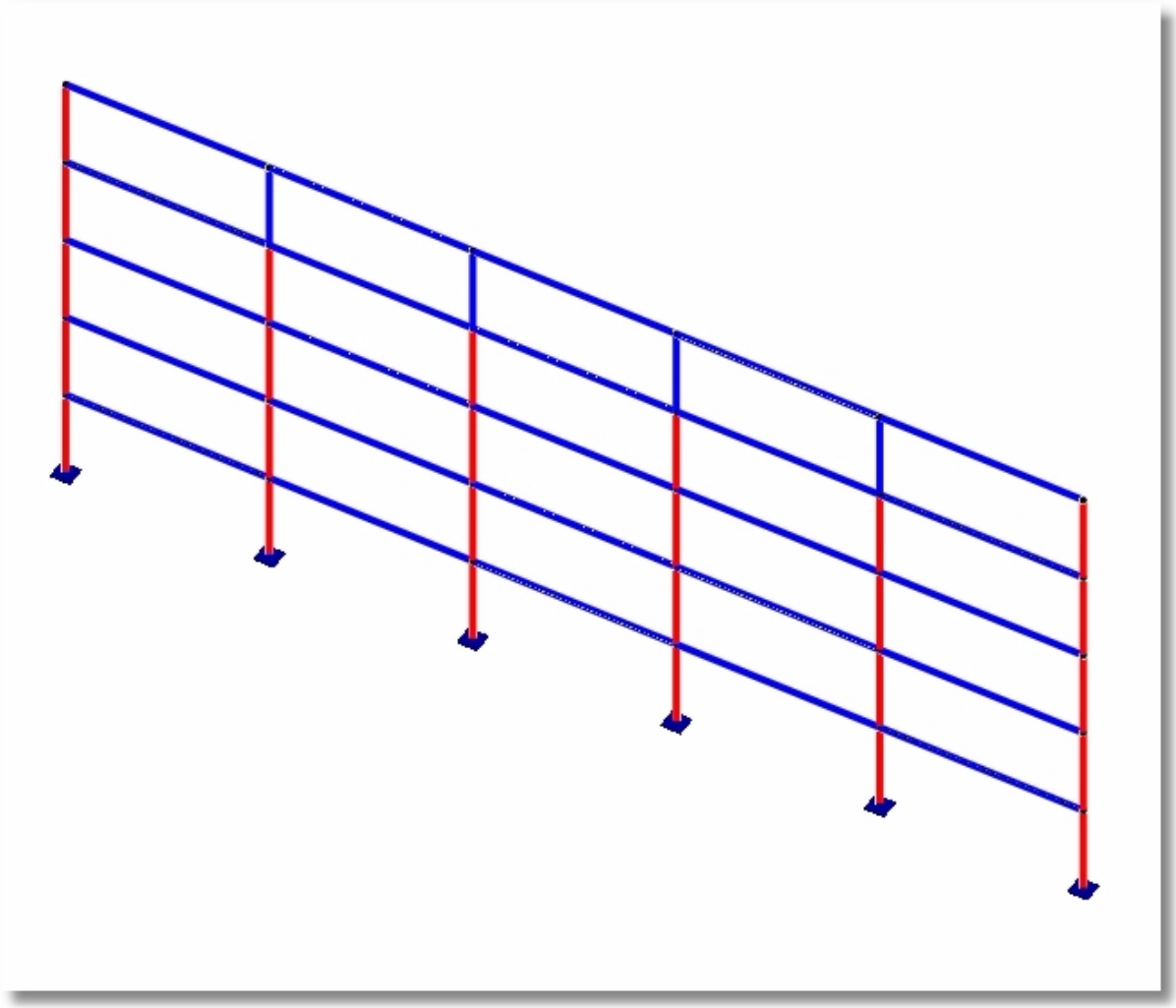
This dialog collects the width, height, flange thickness, web width, clear cover, fc, fy, and Mu, and then returns the required area of steel for a singly reinforced tee beam.

7.12.5 Unity Check

Click **Concrete Design > Unity Check** to view the Unity Check diagram on the model itself.

Tip: The Unity Check can only be displayed after steel or concrete design commands have been applied.

If steel or concrete design has been performed, the Unity Check diagram will show bold blue lines on members that have a unity ratio less than or equal to 1 (passing), and bold red lines on members that have a unity ratio greater than 1 (failing).

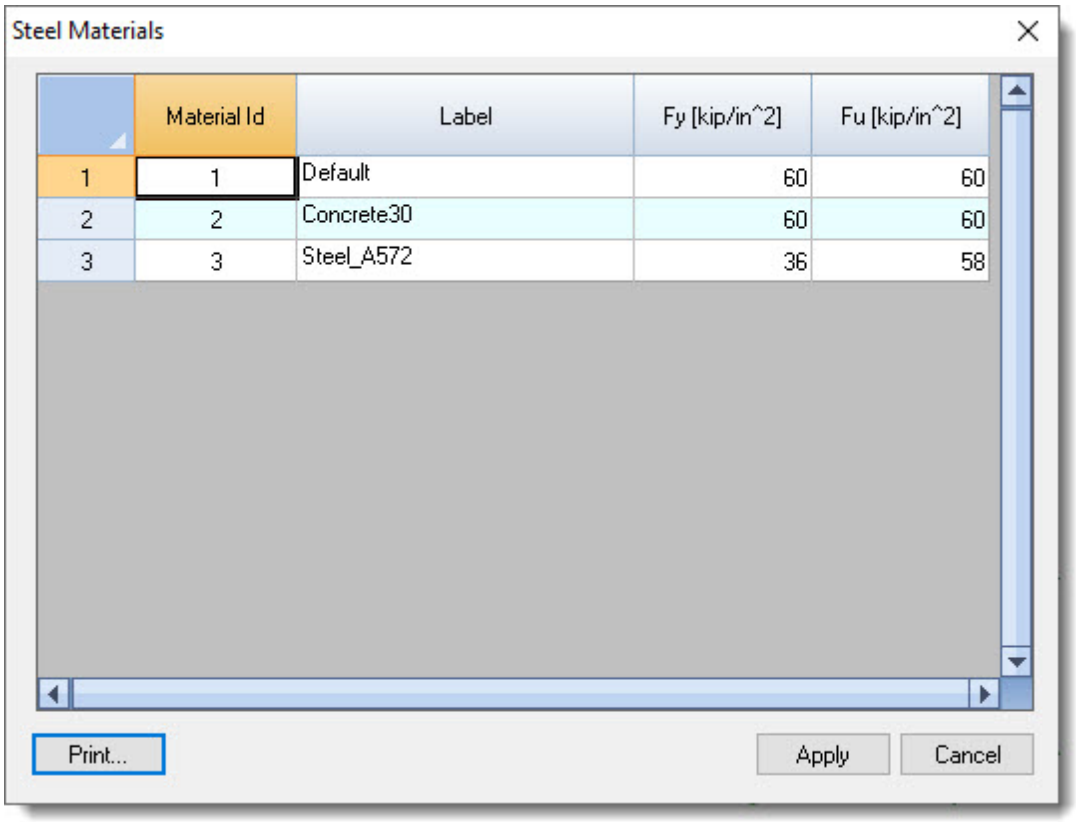


Part VIII

8 Steel Design

8.1 Steel Materials

Steel Design > Steel Materials opens the *Steel Materials* table. (This will list *any* materials that have been defined in the model, not just steel materials.)



	Material Id	Label	Fy [kip/in^2]	Fu [kip/in^2]
1	1	Default	60	60
2	2	Concrete30	60	60
3	3	Steel_A572	36	58

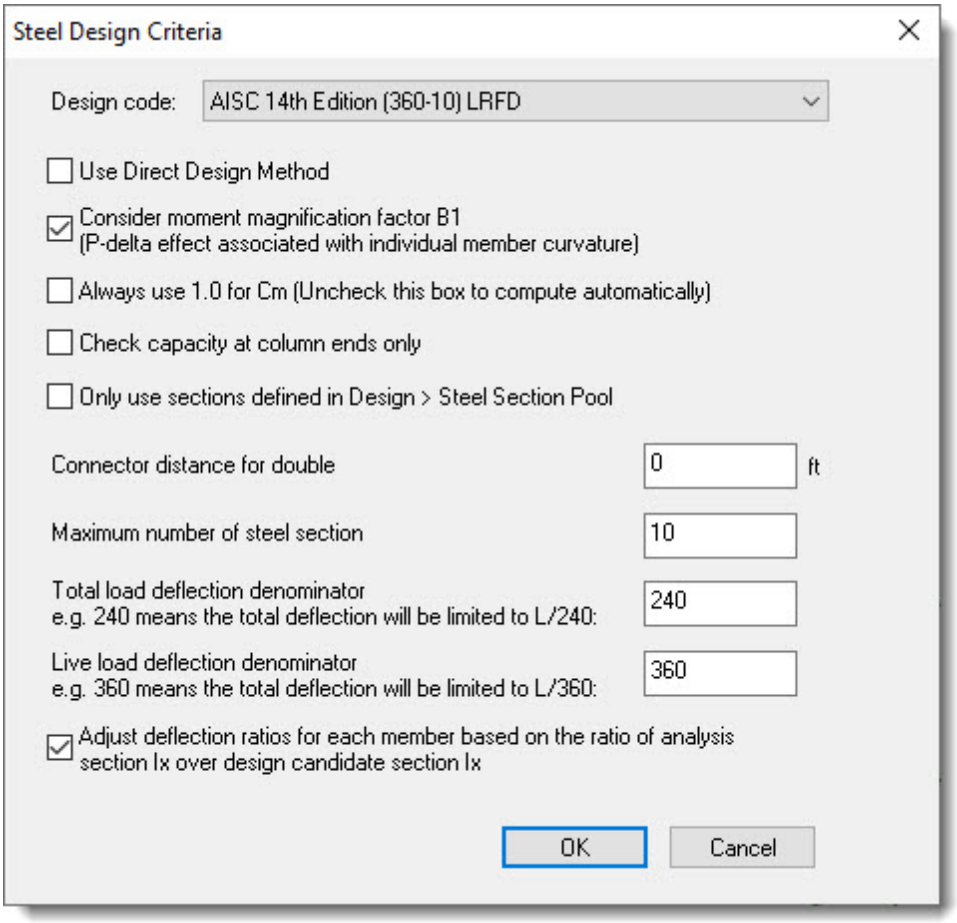
This table contains the materials that have been defined in **Tables > Materials**, but it lists their steel design related properties as opposed to their analysis properties. The purpose of this table is to specify values for yield strength and ultimate strength for any steel materials that are involved in a design.

Tip: The materials have probably already been assigned to the members in order to get an analysis. So at this point, there is no need to Assign anything specific to members. It is only necessary to specify the material strength values for any materials that are actually involved in a steel design.

8.2 Steel Design Criteria

8.2.1 Steel Design Criteria

Click **Steel Design > Steel Design Criteria > Steel Design Criteria** to open the *Steel Design Criteria* dialog. This dialog controls many aspects of the steel design process.



Design Code: Select the desired code.

Use Direct Design Method: Selecting this option affects the calculation of moment magnification factor B1 See AISC Eq. (A-8-3), Eq. (A-8-5), Eq. (C2-2a) and Eq. (C2-2b). It does not create notional loads.

Consider moment magnification factor B1: B1 is moment magnification factor for small p-delta along the member length. With this flag on, you do not need to break a member into multiple members in order to consider small p-delta effect.

Always use 1.0 for Cm: Forces Cm = 1 as opposed to using equation (A-8-4).

Check capacity at column ends only: If this is not selected, the steel design procedure will check columns at every analysis segmental point along the member (more calculation time).

Only use sections from the Section Pool: If this is not selected, the steel design procedure will not limit itself to sections in the Section Pool. Instead, it will stay within the family specified by the Section Prefix specified in Member Design Criteria.

Connector distance for double-angle members: Distance between intermediate connectors in double-angle members for use in Eq. (E6-2b).

Maximum number of steel sections to solve for: Tells the steel design routine how many passing sections to search for.

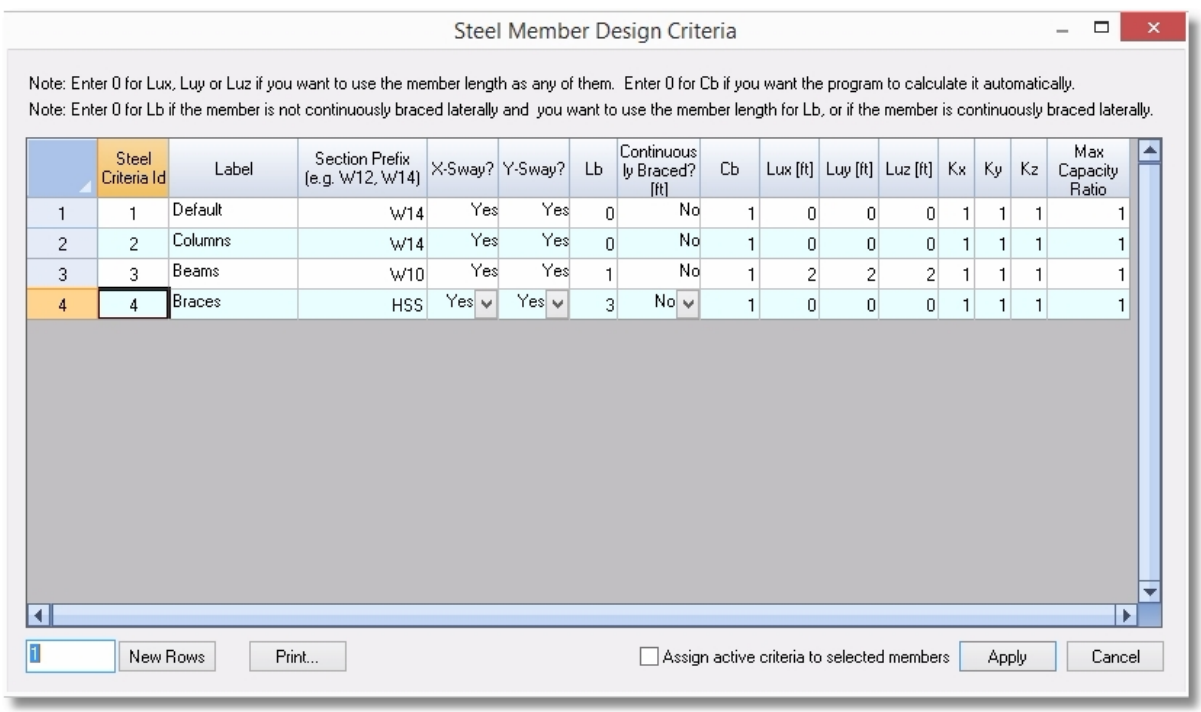
Total load deflection denominator: Total load deflection will be limited to span length/Total load deflection denominator.

Live load deflection denominator: Live load deflection will be limited to span length/Live load deflection denominator.

Adjust deflection ratios...: This item should be checked to prevent the situation where no candidate section is available when critical ratio in design for a member is caused by deflection limits.

8.2.2 Steel Member Design Criteria

Click **Steel Design > Steel Design Criteria > Steel Member Design Criteria** to open the *Steel Member Design Criteria* table. This table controls many aspects of the steel design process.



	Steel Criteria Id	Label	Section Prefix (e.g. W12, W14)	X-Sway?	Y-Sway?	Lb	Continuously Braced? [ft]	Cb	Lux [ft]	Luy [ft]	Luz [ft]	Kx	Ky	Kz	Max Capacity Ratio
1	1	Default	W14	Yes	Yes	0	No	1	0	0	0	1	1	1	1
2	2	Columns	W14	Yes	Yes	0	No	1	0	0	0	1	1	1	1
3	3	Beams	W10	Yes	Yes	1	No	1	2	2	2	1	1	1	1
4	4	Braces	HSS	Yes	Yes	3	No	1	0	0	0	1	1	1	1

You can create as many Steel Member Design Criteria as you need to satisfy the requirements of your design. Once created, you can assign Steel Member Design Criteria to members graphically.

Label: A freeform text descriptor that has meaning to you.

Section Prefix: Tells the program what family (and optionally what size class) to select from when designing.

X-Sway & Y-Sway: X-sway and Y-sway flags designate if a member is a part of a sway frame in member strong/weak direction, respectively. These flags are only used for checking the validity of Kx and Ky values. For example, if X-sway flag is on and Kx is less than 1.0, the program will give a warning.

Lb & Continuously Braced: These are controls for the unbraced length for bending. If Continuously Braced is set to Yes, then the beam is considered to be continuously braced against flexural buckling, and the value in the Lb column should be specified as zero, because the program is not going to do anything with the value.. If Continuously Braced is set to No, then the program will use the value of Lb as the unbraced length.

Tip: If Continuously Braced is set to No, and the value of Lb is set to zero, then the program will use the full member length as the unbraced length.

Cb: Establishes the value to use for Cb (Lateral-torsional buckling modification factor for nonuniform moment diagrams).

Lux, Luy, Luz, Kx, Ky, and Kz: Length and effective length factors for buckling capacity about all three axes.

Tip: The values with the x subscript refer to buckling about the strong axis of the member (the x axis in AISC nomenclature). The values with the y subscript refer to buckling about the weak axis of the member (the y axis in AISC nomenclature).

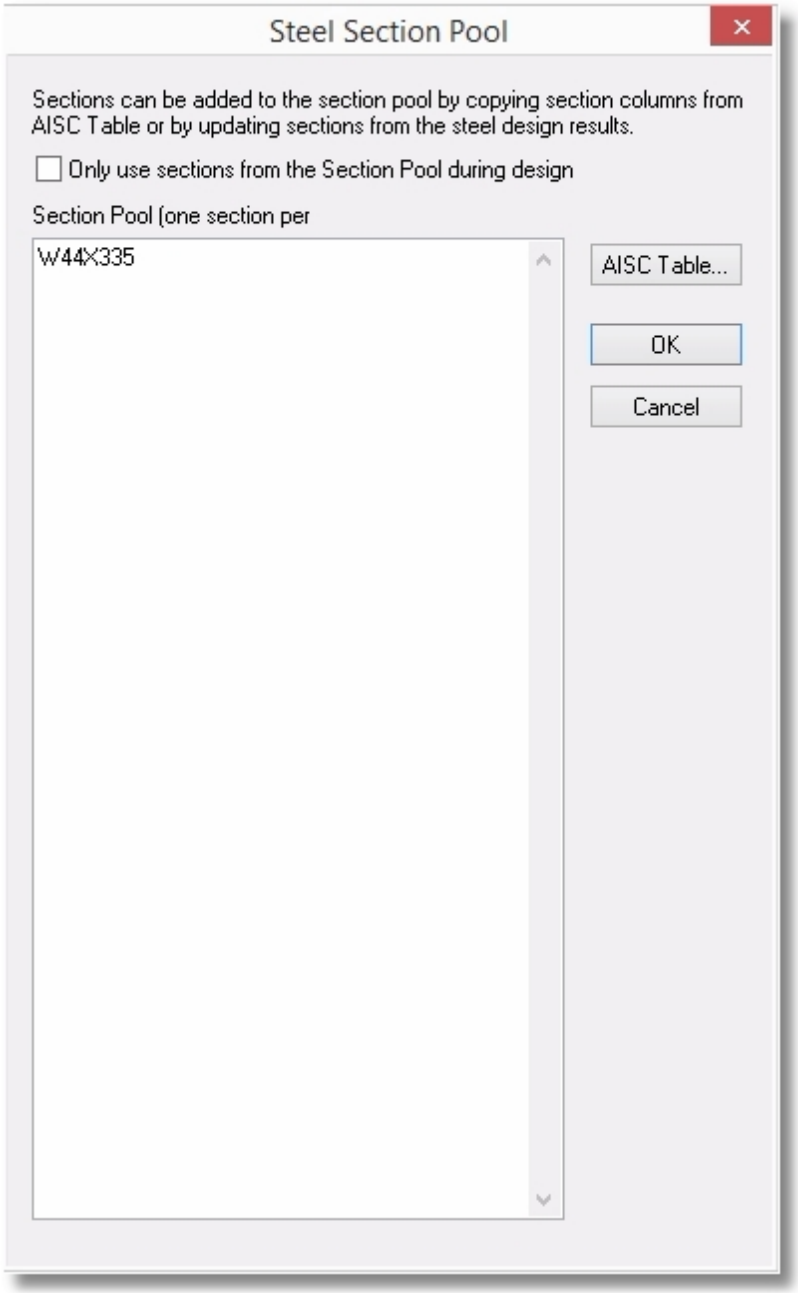
Max Capacity Ratio: Establishes the maximum capacity ratio that will be allowed when selecting passing members.

Tip: It is often helpful to use a value of Max Capacity Ratio somewhat less than one when starting to optimize a model, because loads will change when member stiffnesses change and the analysis is rerun. Limiting the Max Capacity Ratio is one way to "pad" the design a little, to allow for some inevitable stress redistribution.

Tip: Use View > Query to check a member's Steel Member Design Criteria.

8.2.3 Steel Section Pool

Click **Steel Design > Steel Design Criteria > Steel Section Pool** to open the *Steel Section Pool* dialog.



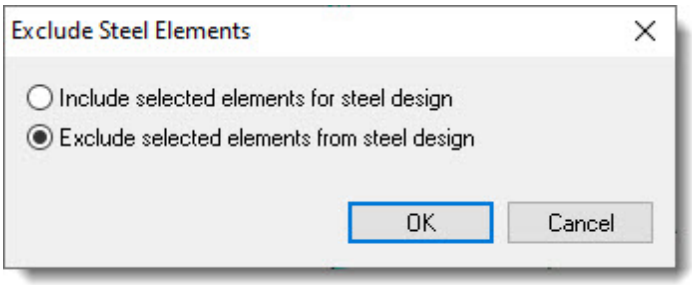
This dialog offers a way to create a list of preferred sections. There is also an option to ONLY select sections from the Steel Section Pool during a design.

Tip: The Steel Section Pool can be populated with preferred sections, but it will have no effect unless the option is selected to ONLY select sections from the Steel Section Pool during a design.

Tip: If a Steel Section Pool is used, be sure that it includes some sections from the family indicated in the Section Prefix column of the Steel Member Design Criteria table, or else the optimization routine will not be able to select any qualifying members.

8.2.4 Exclude Steel Elements

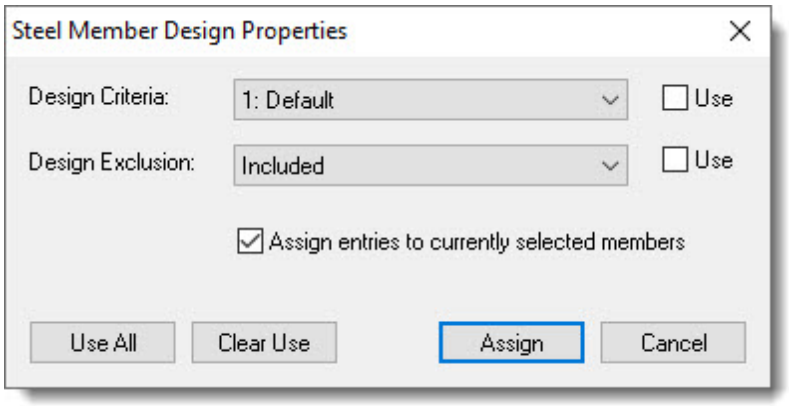
Click **Steel Design > Steel Design Criteria > Exclude Steel Elements** to open the *Exclude Steel Elements* dialog.



This dialog offers a way to explicitly exclude selected elements from any steel design. There is also an option to INCLUDE selected elements in a steel design.

8.3 Steel Member Design Properties

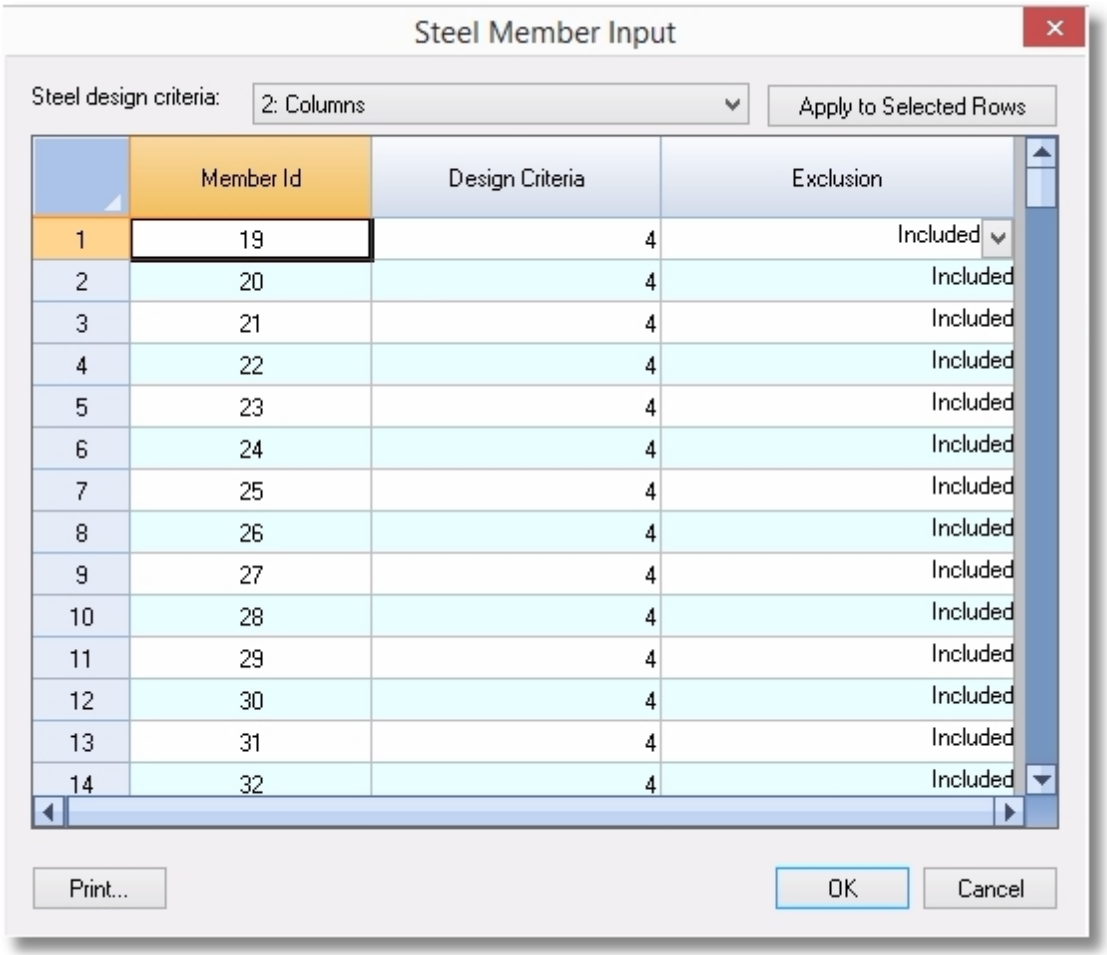
Click **Steel Design > Steel Member Design Properties** to open the *Assign Steel Member Design Properties* dialog.



This is a convenience dialog that allows the assignment of Steel Member Design Criteria and/or Steel Member Design Exclusion, both of which are explained in the section on Steel Design > Steel Design Criteria.

8.4 Steel Member Input

Click **Steel Design > Steel Member Input** to open the *Steel Member Input* table.



	Member Id	Design Criteria	Exclusion
1	19	4	Included
2	20	4	Included
3	21	4	Included
4	22	4	Included
5	23	4	Included
6	24	4	Included
7	25	4	Included
8	26	4	Included
9	27	4	Included
10	28	4	Included
11	29	4	Included
12	30	4	Included
13	31	4	Included
14	32	4	Included

This is a convenience table that lists Member ID, Steel Member Design Criteria, and the assignment of Steel Member Design Exclusion.

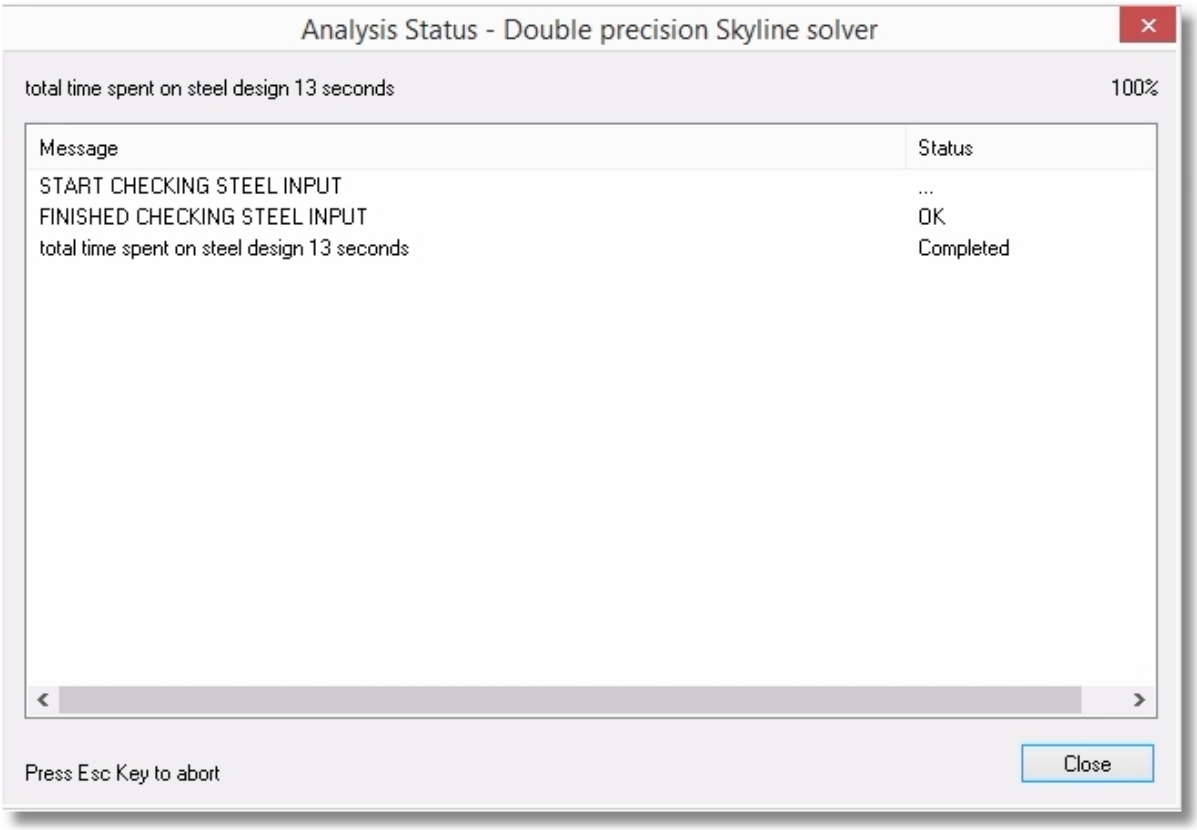
To change Steel Member Design Criteria, select one or more rows, choose the desired Steel Design Criteria from the dropdown list box at the top of the table, and click Apply to Selected Rows.

To change the Steel Member Design Exclusion, click in a cell and use the dropdown arrow to select the desired status.

8.5 Perform Steel Design

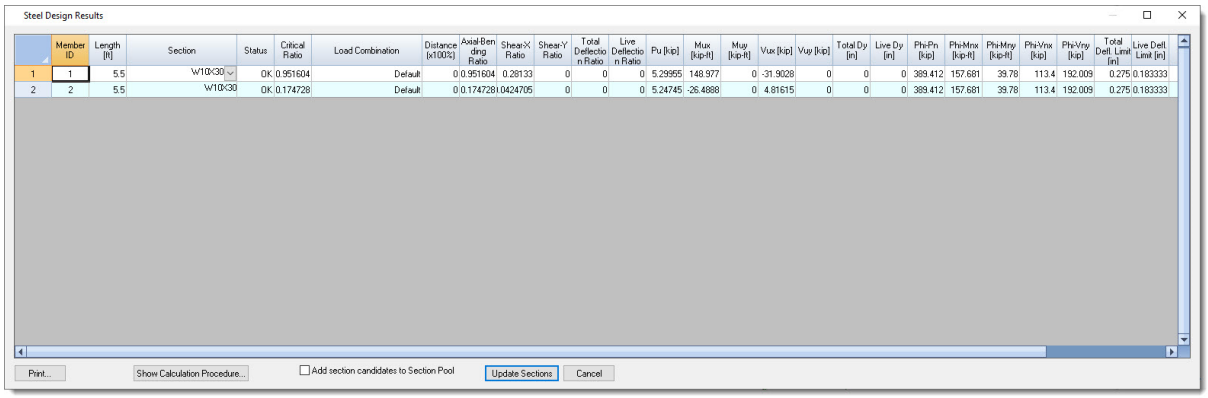
Click **Steel Design > Perform Steel Design** to initiate a Steel Design process. (Static analysis results must be current in order to run a steel design.)

The message box will indicate when the design is complete.



8.6 Steel Design Results

Click **Steel Design > Steel Design Results** to open the *Steel Design Results* table.



	Member ID	Length [ft]	Section	Status	Critical Ratio	Load Combination	Distance [x100%]	Axial-Bending Ratio	Shear-X Ratio	Shear-Y Ratio	Total Deflection Ratio	Live Deflection Ratio	Pu [kip]	Mux [kip-ft]	Muy [kip-ft]	Vux [kip]	Vuy [kip]	Total Dy [in]	Live Dy [in]	Phi-Pn [kip]	Phi-Mnx [kip-ft]	Phi-Mny [kip-ft]	Phi-Vnx [kip]	Phi-Vny [kip]	Total Defl. Limit [in]	Live Defl. Limit [in]
1	1	5.5	W10X30	OK	0.951604	Default	0	0.951604	0.28133	0	0	0	5.29955	148.977	0	-31.9028	0	0	0	389.412	157.681	39.78	113.4	192.009	0.275	0.183333
2	2	5.5	W10X30	OK	0.174728	Default	0	0.174728	0.0424705	0	0	0	5.24745	-26.4888	0	4.81615	0	0	0	389.412	157.681	39.78	113.4	192.009	0.275	0.183333

This table presents the results of the most recent Steel Design process. Each line represents a member in the model and presents the following data:

Member ID: The ID of the member.

Length: The actual member length, not the unbraced length for flexure or for Euler buckling.

Section: The AISC section for which design results are displayed.

Tip: Click the dropdown arrow in the Section cell to view a list of other sections selected by the Steel Design routine subject to the limitations of the Section Prefix and the Section Pool.

Tip: Selecting a different section will invalidate the analysis results due to changes in stiffness. So be sure to reanalyze the model and rerun the Steel Design after making any section changes.

Status: Indicates an OK or NG status to show whether the selected member satisfies code requirements or not.

Critical Ratio: Reports the highest ratio found after running all of the required steel code checks on the member considering Axial & Bending, Shear, and Deflection.

Load Combination: Indicates the Load Combination responsible for producing the Critical Ratio.

Distance: Reports the location where the critical design was found to occur. 0 indicates that the critical design location was at the starting end of the member. 1 indicates that the critical design location was at the ending end of the member. A decimal indicates a location somewhere in between starting and ending end.

Axial-Bending Ratio: Indicates the highest ratio that occurred on the member due to combined axial and flexural design when subjected to the load combination that produced the Critical Ratio.

Shear-X Ratio: Indicates the highest ratio that occurred on the member due to strong direction shear when subjected to the load combination that produced the Critical Ratio.

Shear-Y Ratio: Indicates the highest ratio that occurred on the member due to weak direction shear when subjected to the load combination that produced the Critical Ratio.

Total Deflection Ratio: Calculated as controlling Total Load Deflection/(Span/Total Load Deflection Denominator) when subjected to the load combination that produced the Critical Ratio.

Live Deflection Ratio: Calculated as controlling Live Load Deflection/(Span/Live Load Deflection Denominator) when subjected to the load combination that produced the Critical Ratio.

Pu: Axial load at the controlling distance when subjected to the load combination that produced the Critical Ratio.

Mux: Moment about the strong axis at the controlling distance when subjected to the load combination that produced the Critical Ratio.

Muy: Moment about the weak axis at the controlling distance when subjected to the load combination that produced the Critical Ratio.

Vux: Shear in the strong direction at the controlling distance when subjected to the load combination that produced the Critical Ratio.

Vuy: Shear in the weak direction at the controlling distance when subjected to the load combination that produced the Critical Ratio.

Total Dy: Total Load strong axis deflection.

Live Dy: Live Load strong axis deflection.

PhiPn: Axial capacity.

PhiMnx: Strong axis moment capacity.

PhiMny: Weak axis moment capacity.

PhiVnx: Strong direction shear capacity.

PhiVny: Weak direction shear capacity.

Total Deflection Limit: Span/Total Load Deflection Denominator.

Live Deflection Limit: Span/Live Load Deflection Denominator.

Cb: Lateral-torsional buckling modification factor for nonuniform moment diagrams.

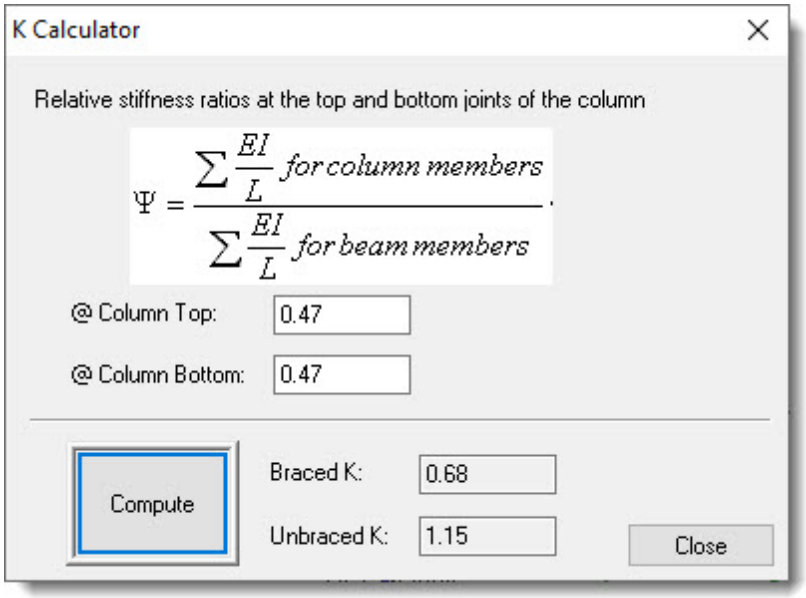
Cmx: Coefficient accounting for nonuniform moment about the strong axis.

Cmy: Coefficient accounting for nonuniform moment about the weak axis.

8.7 Steel Tools

8.7.1 K Calculator

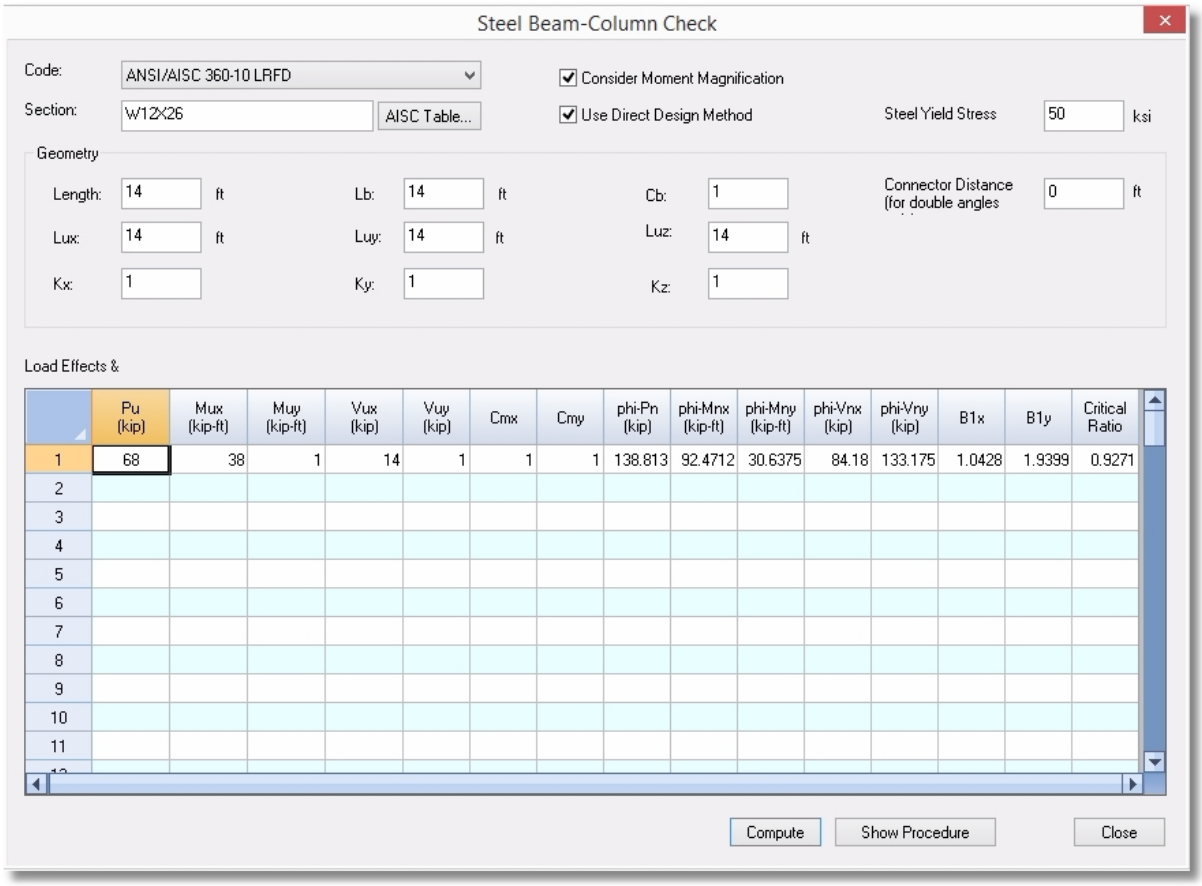
Click **Steel Design > Steel Tools > K Calculator** to open the *K Calculator*.



The dialog collects a value of Psi Top and Psi Bottom and then returns the calculated value of K for a braced and for an unbraced condition.

8.7.2 Section Check

Click **Steel Design > Steel Tools > Steel Section Check** to open the *Steel Section Check* dialog.



This tool allows the specification of member section, length, bracing, material properties, and loading and then performs a code check with those parameters.

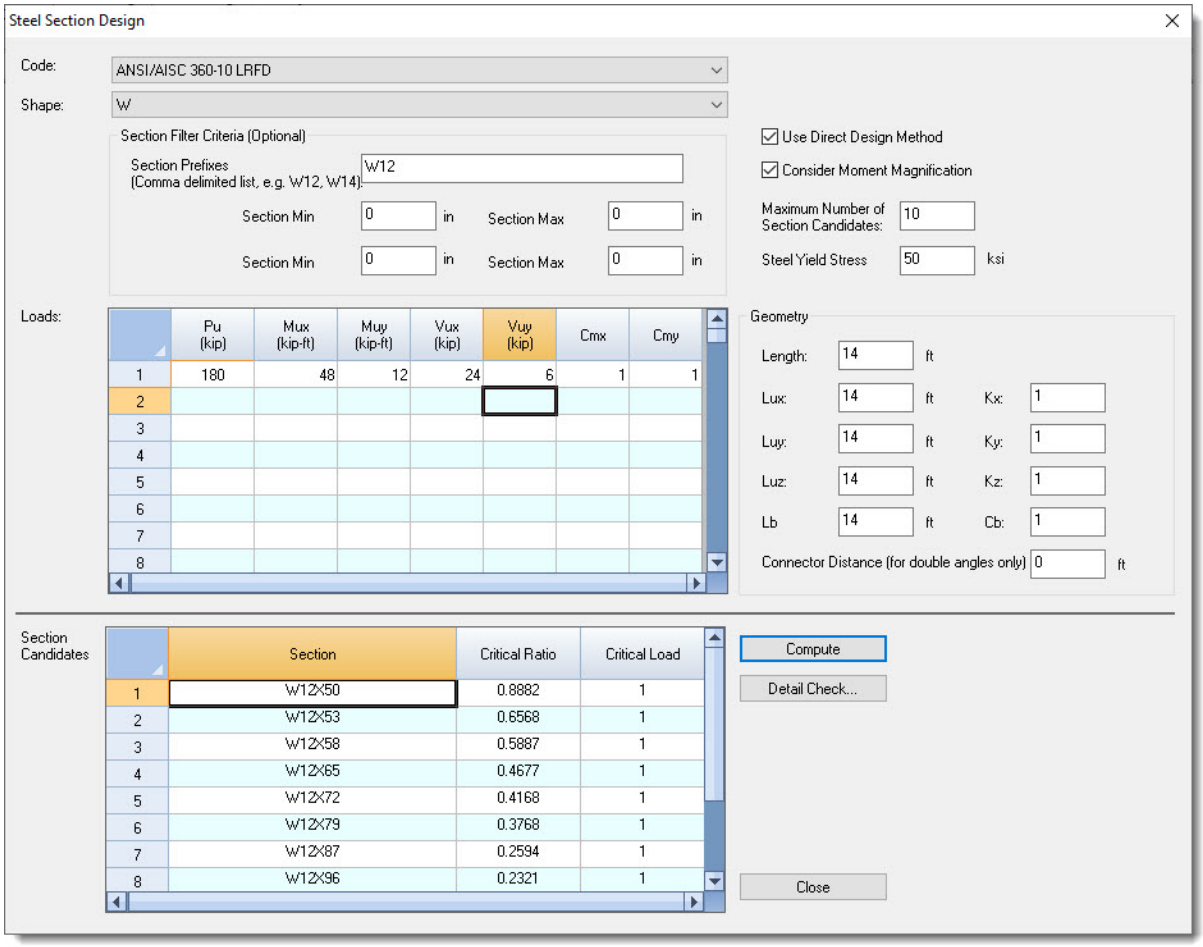
After filling in the parameters in the input fields at the top of the screen, populate the loading values in the columns for Pu, Mux, Muy, Vux, Vuy, Cmx and Cmy. (Each row can be thought of as the resulting factored loads for a load combination.)

Then click the Compute button and the program will complete the design and report the results in the remaining columns.

The Show Procedure button can be used to view and print the detailed calculations.

8.7.3 Section Design

Click **Steel Design > Steel Tools > Steel Section Design** to open the *Steel Section Design* dialog.



This tool allows the specification of member section prefix (like W12), length, bracing, material properties, and loading, and then performs a design with those parameters.

After filling in the parameters in the input fields at the top and right side of the screen, populate the loading values in the columns for Pu, Mux, Muy, Vux, Vuy, Cmx and Cmy. (Each row can be thought of as the resulting factored loads for a load combination.)

Then click the Compute button and the program will complete the design and report the Section Candidates and results in the remaining columns.

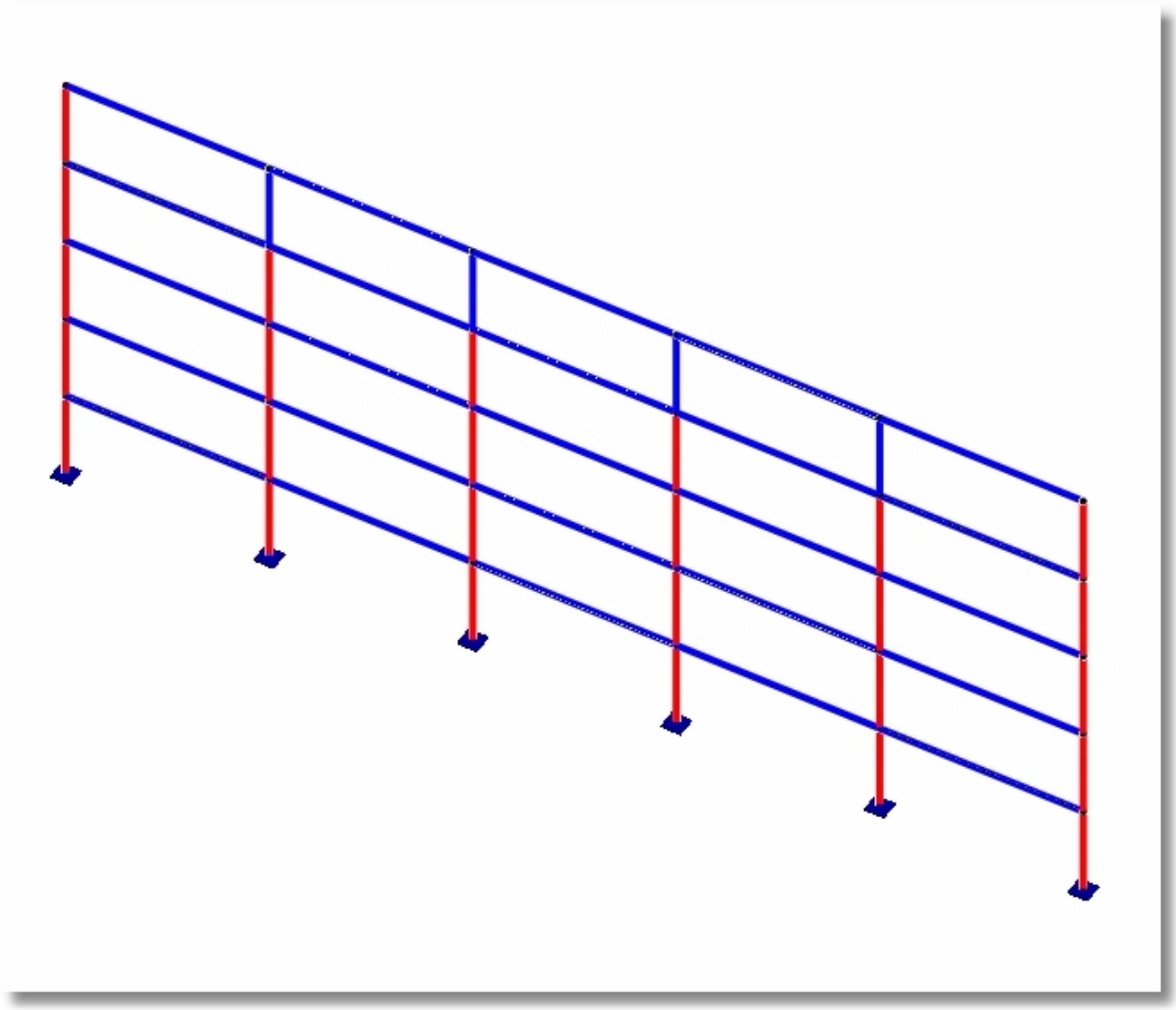
The Detail Check button can be used to open the *Steel Section Check* dialog, where individual load combinations can be investigated for intermediate values, and detailed calculations can be viewed and printed.

8.8 Unity Check

Click **Steel Design > Unity Check** to view the Unity Check diagram on the model itself.

Tip: The Unity Check can only be displayed after steel or concrete design commands have been applied.

If steel or concrete design has been performed, the Unity Check diagram will show bold blue lines on members that have a unity ratio less than or equal to 1 (passing), and bold red lines on members that have a unity ratio greater than 1 (failing).

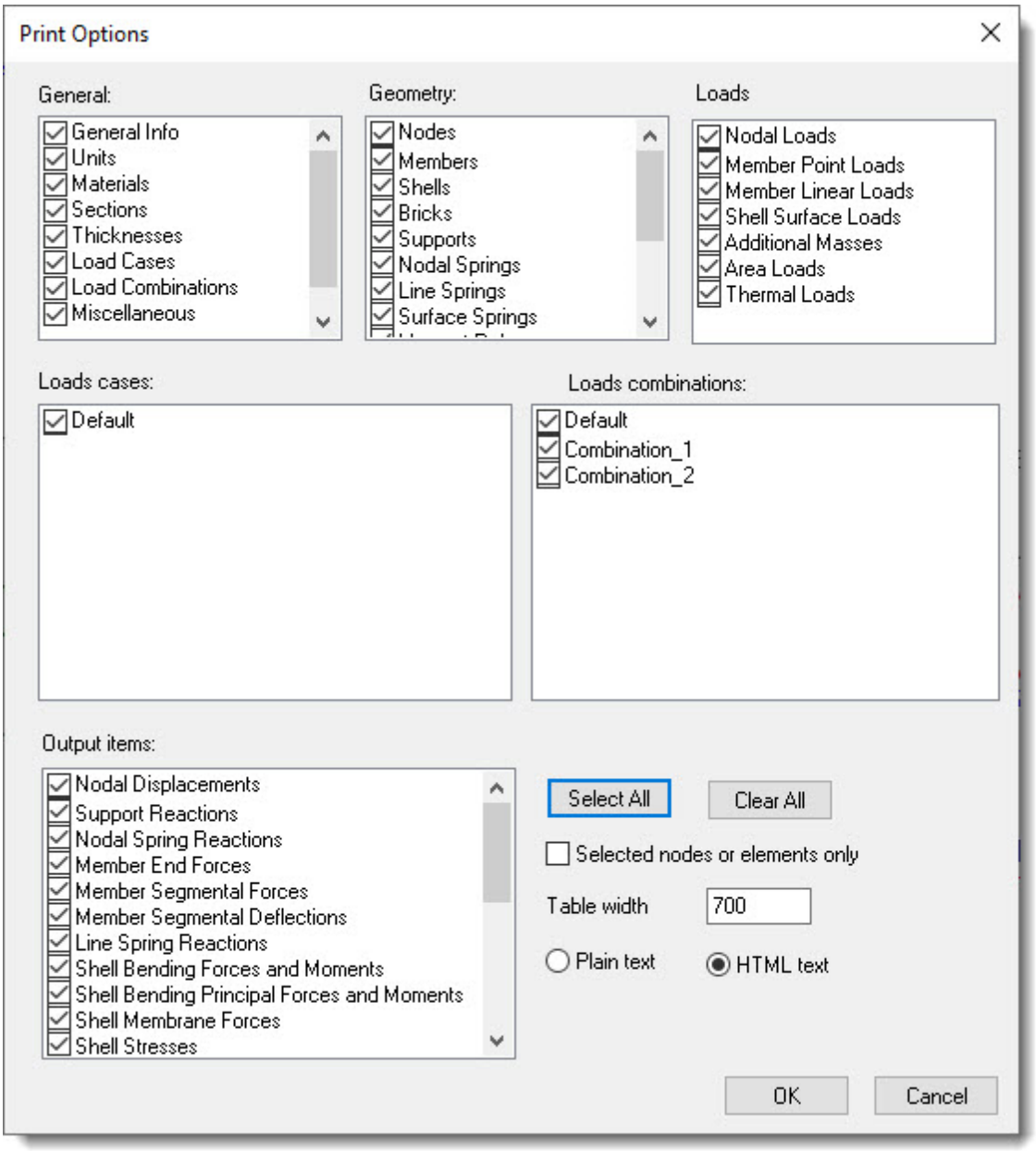


Part IX

9 Reporting

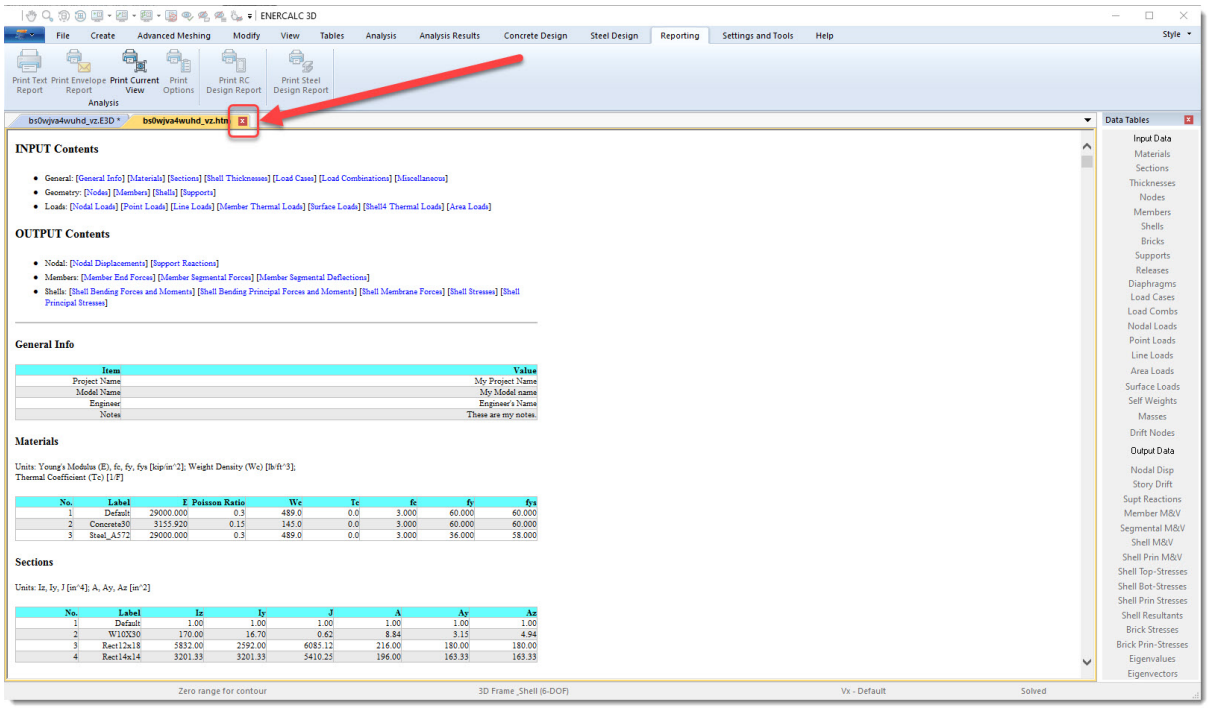
9.1 Prepare a Report

Click **Reporting > Print Options**. This will open the *Print Options* dialog.



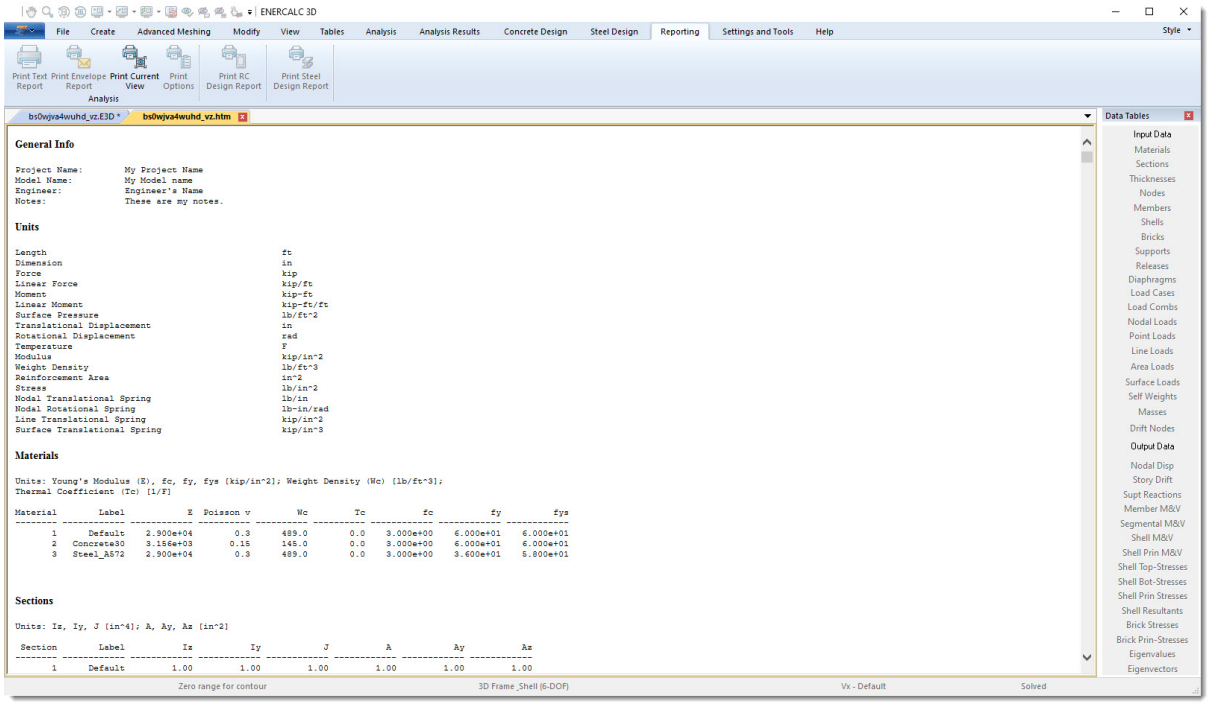
This dialog offers a variety of input and output options to choose from for inclusion in the report.

Once the desired options are selected click OK. The report will display. To dismiss the report preview, click the X at the right end of the tab:



9.2 Print a Report

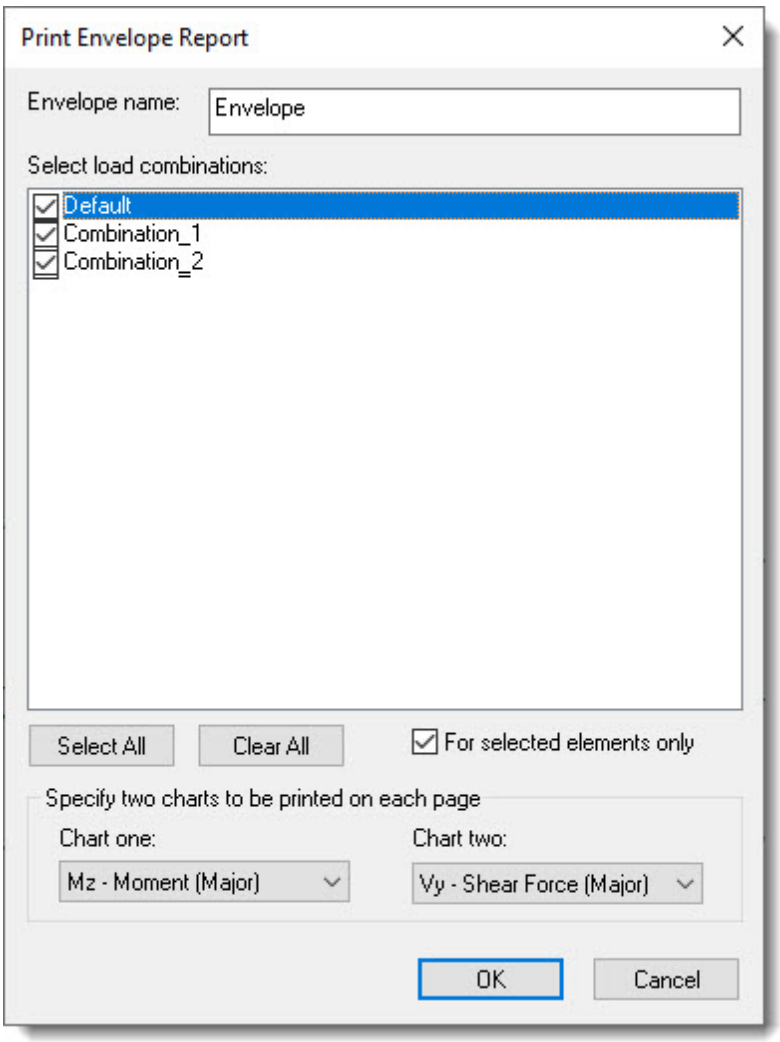
Click **Reporting > Print Text Report**. A preview of the report is displayed.



To print the report, right-click on the screen and click Print. To dismiss the report preview, click the X at the right end of the tab.

9.3 Print an Envelope Report

Reporting > Print Envelope Report provides a way to print graphs of the envelope of forces on members. The *Print Envelope Report* dialog offers the controls to set up what gets printed in the Envelope Report.

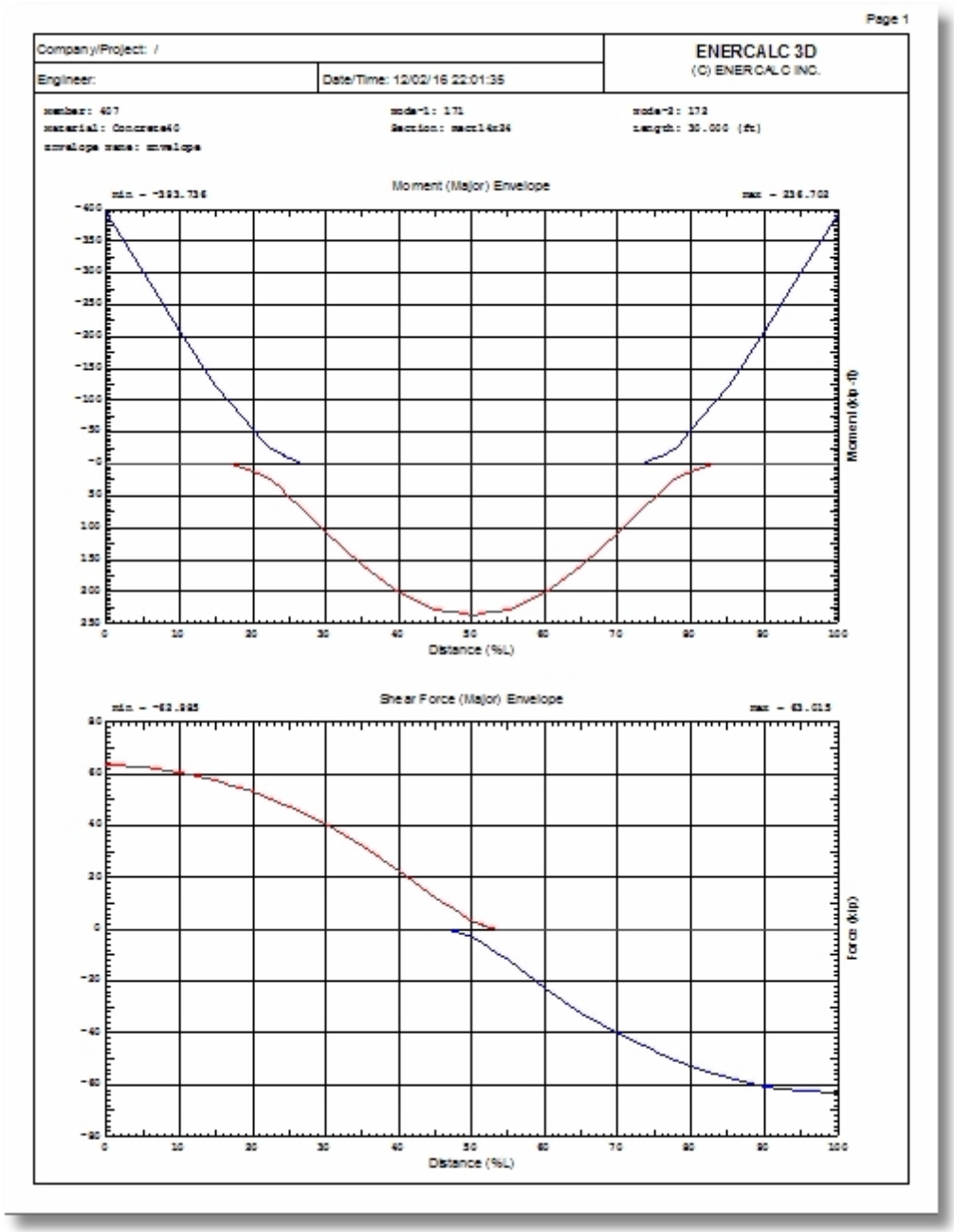


An Envelope name can be entered to help identify its purpose.

Any and all load combinations can be included.

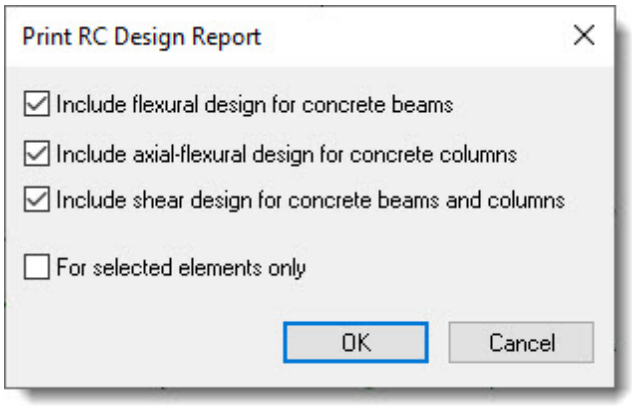
There is an option to report on selected elements only. If this is deselected, the Envelope Report will be created for all members in the model.

This report includes two charts on each page. Options include shears, moments, torsion and axial forces.



9.4 Print RC Design Report

Click **Reporting > Print RC Design Report** to open the *Print RC Design Report* dialog.

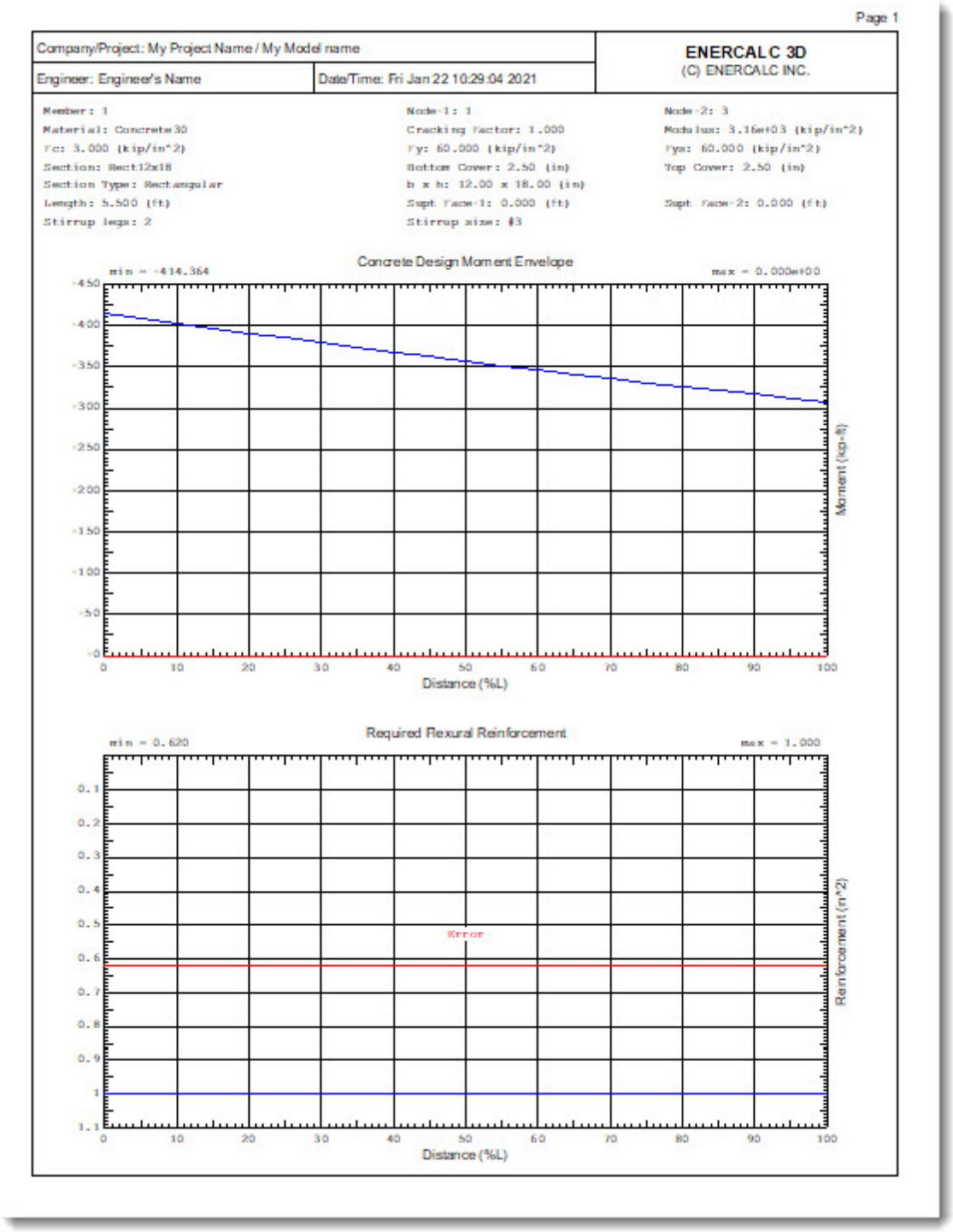


This dialog is used to generate a printed report for concrete beam and or column designs.

It starts by offering options:

- Include flexural design for concrete beams
- Include axial-flexural design for concrete columns
- Include shear design for concrete beams and columns
- Include designs for selected elements only.

When you click the OK button, the report is generated and shown in preview.

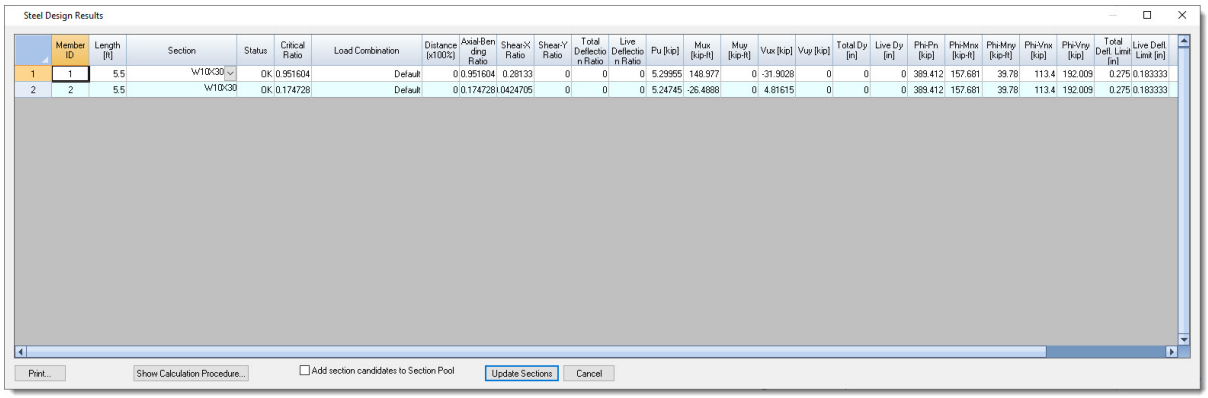


You can zoom, move through pages, and print if desired.

Click Close to return to the model view.

9.5 Print Steel Design Report

Click **Reporting > Print Steel Design Report** to open the *Steel Design Results* window.



	Member ID	Length [ft]	Section	Status	Critical Ratio	Load Combination	Distance [x100%]	Axial-Bending Ratio	Shear-X Ratio	Shear-Y Ratio	Total Deflection Ratio	Live Deflection Ratio	Pu [kip]	Mux [kip-ft]	Muy [kip-ft]	Vux [kip]	Vuy [kip]	Total Dy [in]	Live Dy [in]	Phi-Pn [kip]	Phi-Mnx [kip-ft]	Phi-Mny [kip-ft]	Phi-Vnx [kip]	Phi-Vny [kip]	Total Defl. Limit [in]	Live Defl. Limit [in]
1	1	5.5	W10X30	OK	0.951604	Default	0	0.951604	0.28133	0	0	0	5.29955	148.977	0	-31.9028	0	0	0	389.412	157.681	39.78	113.4	192.009	0.275	0.183333
2	2	5.5	W10X30	OK	0.174728	Default	0	0.174728	0.0424705	0	0	0	5.24745	-26.4888	0	4.81615	0	0	0	389.412	157.681	39.78	113.4	192.009	0.275	0.183333

To print concise results in tabular form, just click the Print button.

To print detailed calculations, click Show Calculation Procedure and then print from the resulting window.

Part X

10 Advanced Topics

- Modeling Accidental Eccentricity for seismic loads
- Modeling mat foundations
- Working with meshes of plates


ENERCALC
Structural Engineering Library
ENERCALC's industry leading software solutions to perform structural design and analysis for low to mid rise buildings.
ENERCALC