Robot Structural Analysis: Making the Change

Ken Marsh – Marsh API LLC

Code SE6881-L

Learning Objectives

At the end of this class, you will be able to:

- Discover the overall capabilities of Robot Structural Analysis software
- Learn how to navigate the Robot Structural Analysis software interface easily and develop a simple model
- Understand the key concepts of performing analysis and design with Robot Structural Analysis software
- Begin your own exploration of the capabilities Robot Structural Analysis software

About the Speaker

After 7 years as a structural engineer, Ken Marsh joined Autodesk, Inc., as a quality assurance analyst working on the Revit software product line. Ken has recently started his own firm, which is dedicated to advancing Building Information Modeling (BIM)-based structural engineering through the Revit software API add-ons. Ken is also the author of Robot Structural Analysis Professional 2015—Essentials, and he loves to discuss Autodesk technology as it relates to the architecture, engineering, and construction industry.

ken@marshapi.com
www.marshapi.com

Marsh API
179 Albion Street • Unit 2
Somerville, MA 02144
RobotEssentials@marshapi.com

Contents

Introduction

Robot Structural Analysis Professional is an incredibly powerful tool for general purpose structural engineering, analysis and design. No engineer I know would switch analysis programs without a fairly thorough investigation of the capabilities and detailed operation of the software. We're responsible for life safety and few, if any of us take that responsibility lightly. This tutorial will get you started on the road to your own investigation of Robot Structural Analysis.

The Goal of this Tutorial

My goal in writing this tutorial is to give you a guided, step-by-step tour of the software. I want to give you enough experience exercising the various features and functionality and a guide through the major workflows in the software such that you gain enough confidence in the fundamentals that you can begin your own investigation of the tool and compare it with previous designs you may have done with other software to see where it's the same, and if different, to be able to dig into why it's different. We will cover the basics: interface, navigation, setup, display, simple modeling, basic loading, seismic, results exploration, and basic material design workflow. We want you to have a good map of the functionality that you can start your own investigation and dissect models to learn more about them.

Robot General Capabilities

- Structural Modeling for linear members, planar members, shell members, and solids.
- Thorough loading and boundary conditions including standard nodal, linear, and surface loads as well as advanced loading (acceleration, time varying loading, displacements, temperature, etc.)
- Advanced meshing capabilities for shell elements
- Full complement of analysis types including static, non-linear, non-linear/p-delta, modal, moving load, time-history, buckling, and pushover
- Full implementation of AISC Direct Analysis Method
- Over 70 different country codes for material design: 40 for steel and 30 for concrete
- Through documentation functionality for presentation of calculation results.
- Full featured application programming interface (API) for creating custom solutions or advanced functionality and customized design.

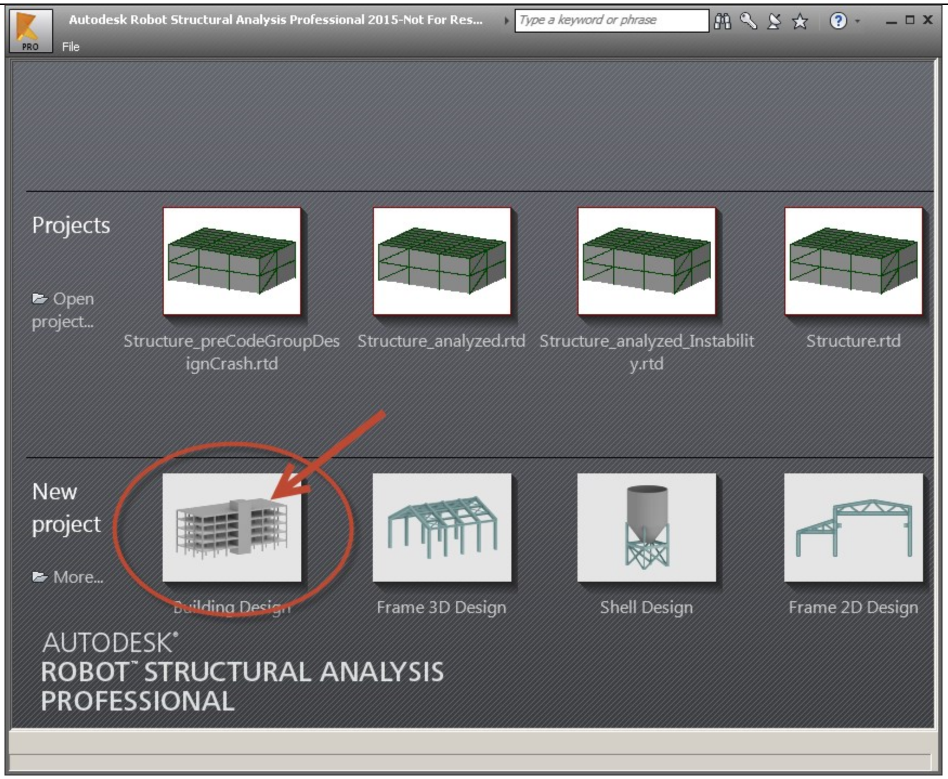
General Limitations

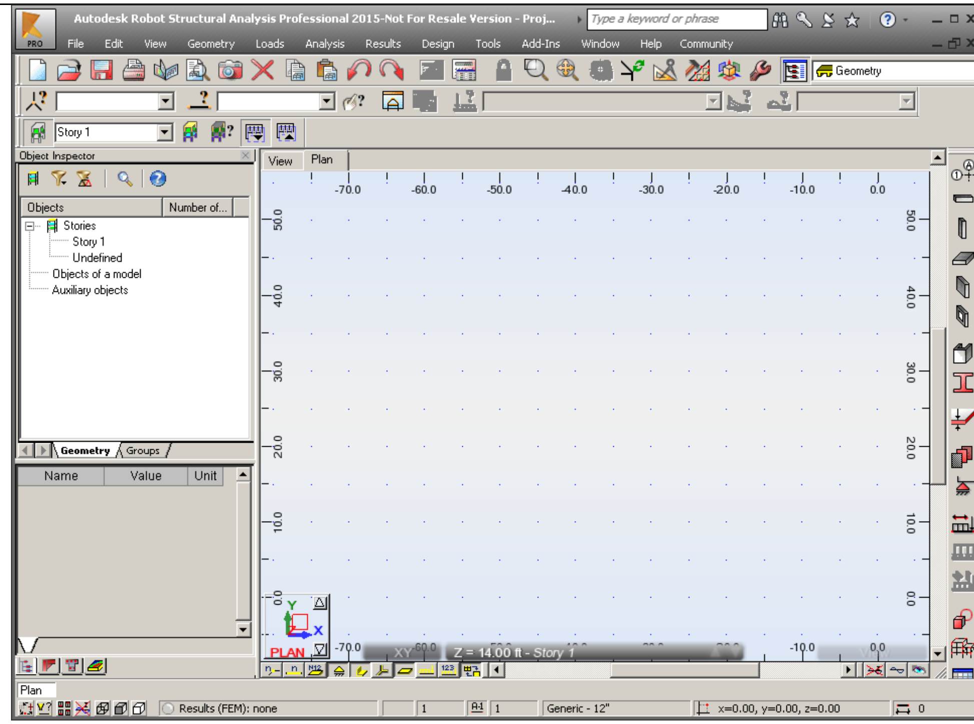
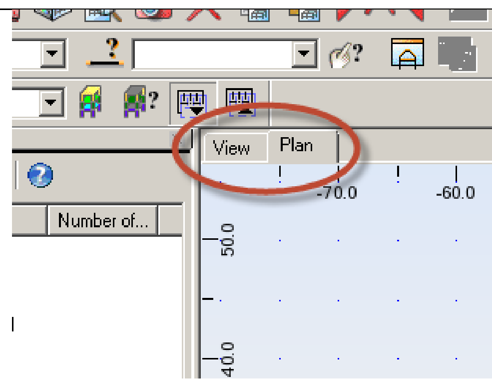
While there are a tremendous number of capabilities of the software, there are a few items which are not currently supported by Robot Structural Analysis:

- Composite design not supported
- Pre-stress/Post-tension not directly supported
- Steel strength and serviceability design are asynchronous.

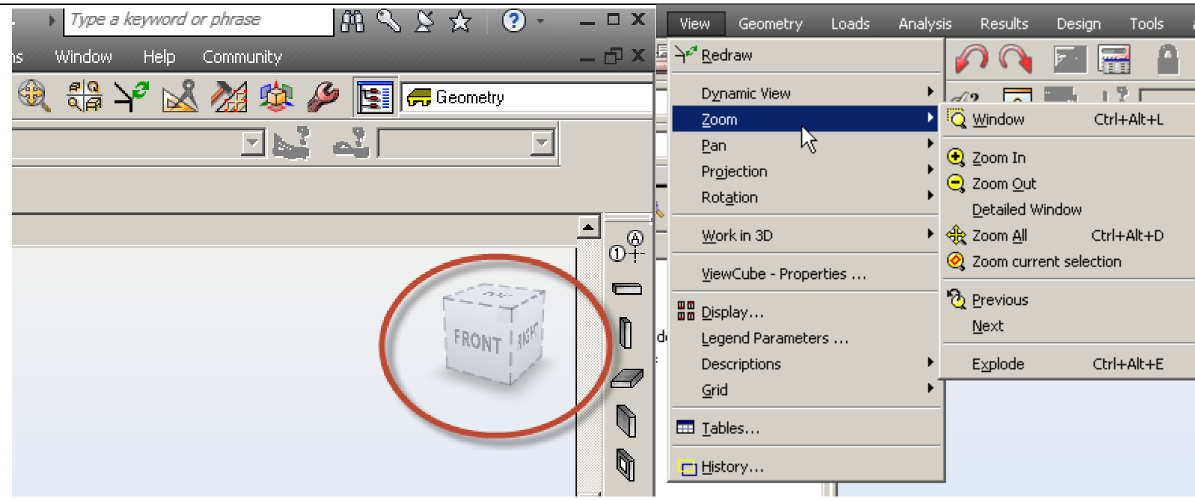
Section 1 - Getting Started

The Robot Interface

Open Autodesk Robot Strucutral Analysis and select "Building Design" under New Project. If you don't see building design, click "More…" and choose Building Design from the list of project templates	

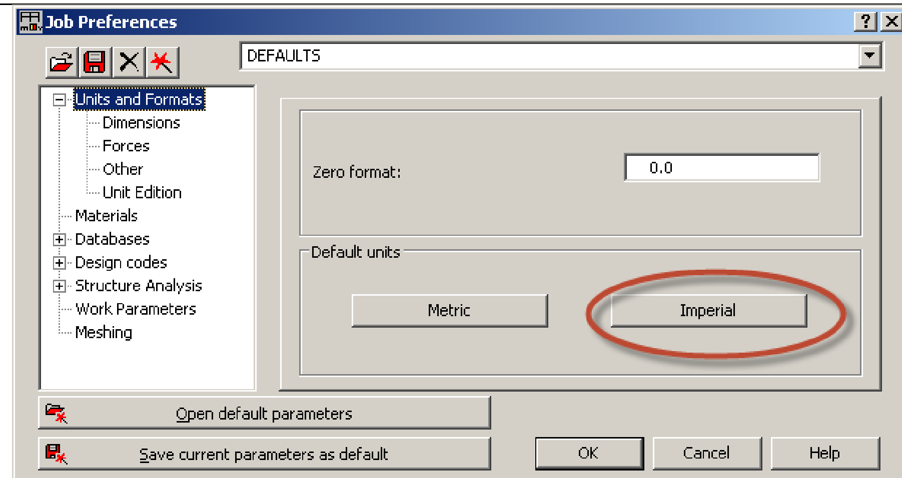
Robot will open the building design template. Take a look at the interface components:	
In particular notice the view tabs at the top. The building design template is the only Robot template with these two tabs. They are not view tabs like you might expect, they are tabs within this one view (the main view).	

When Robot first opens, the cursor will be in Zoom/Pan/Oribt mode so press escape to exit this mode. Notice that there is no view cube in the plan tab, but if you switch to the view tab you will see the familiar view cube: This works exactly like it does in AutoCAD or Revit. Other view controls which I use constantly are the middle mouse button to pan, middle mouse wheel to scroll and holding down the shift key and the middle mouse button to orbit a view. I find this completely intuitive but you are welcome to use the view cube, and/or the view commands from the view menu for zoom, pan, and orbit.

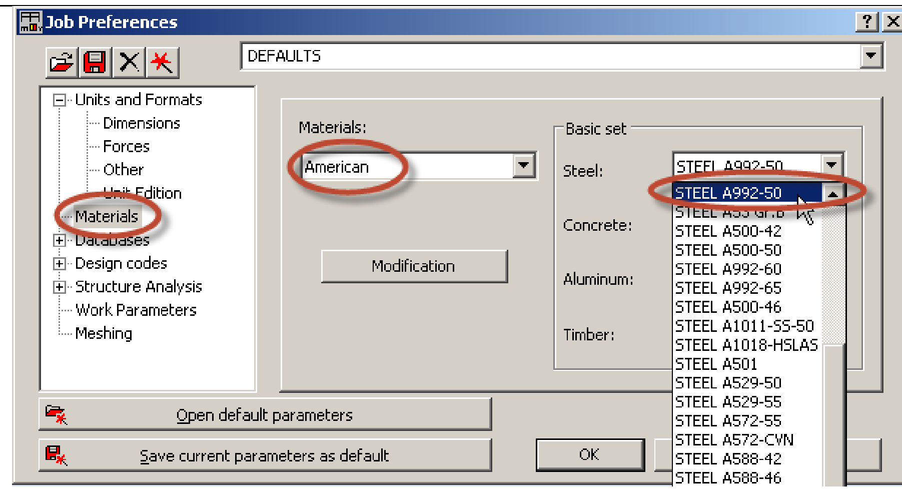


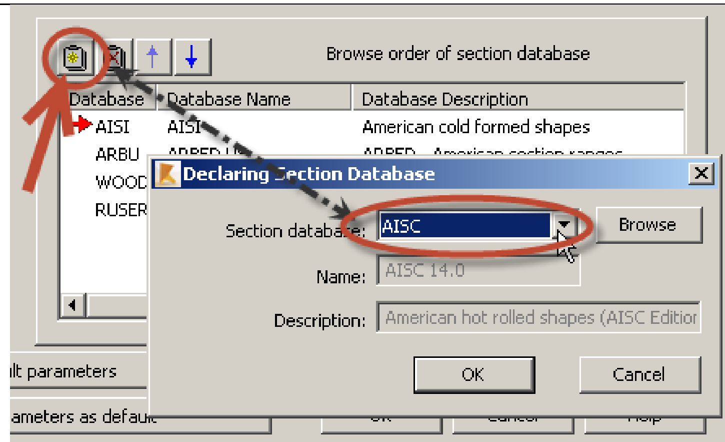
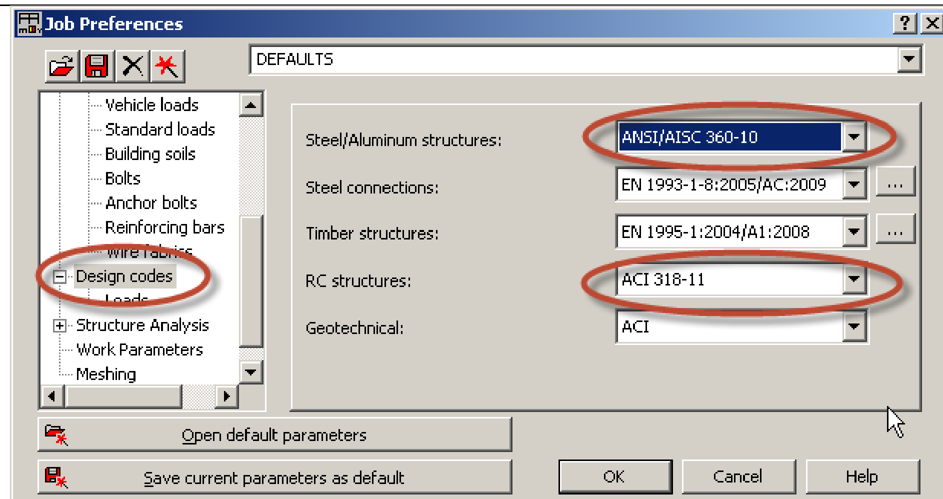
Project Setup (5min)

We'll start with some project setup. Open project settings from the tools menu: Tools>Job Preferences… then select "Imperial" for the default units.

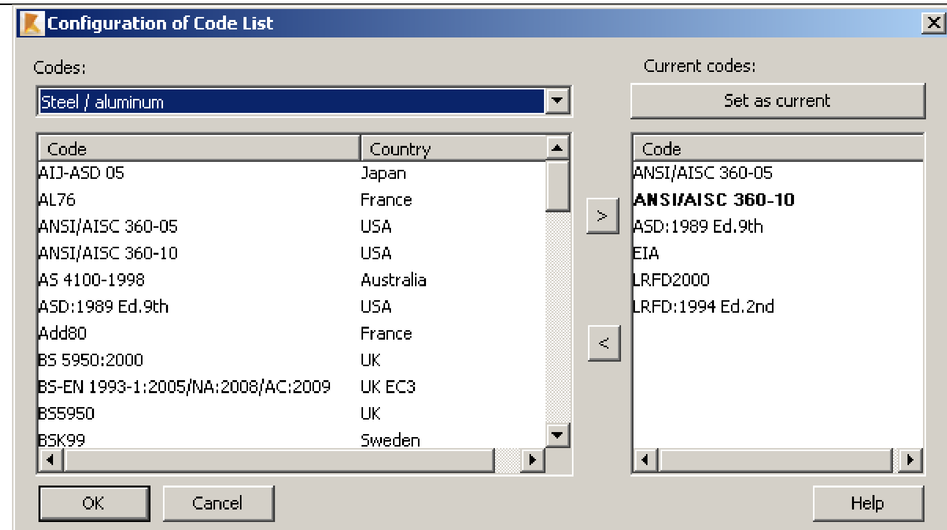


Next, select the Materials item on the left and select "American" for the materials. Then choose A992-50 for steel from the Steel dropdown menu.



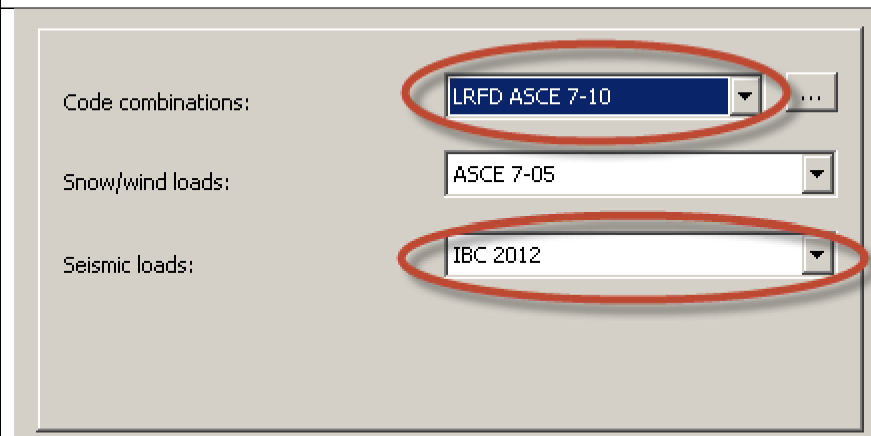
Next expand "Databases" and if AISC and SJI are not available, select the "Add a new database" button at the top and select AISC then repeat for SJI (note, SJI is at the bottom of the list) Use the "UP/DOWN" arrow buttons to move AISC to the top of the list for better organization later.	
Next, Select "Design Codes" and for "Steel/Aluminum structures" select "AISC 360-10" and for "RC Structures" select "ACI 318-11"	

Before we move on, in the steel dropdown, choose "more…" to see the full list of codes supported by Robot. The "Codes" dropdown on the left will allow you to browse the different code categories and the right arrow button (">") will allow you to add the code to the active list.



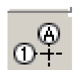
Next, expand "design codes" and choose "Loads". Under loads, select "LRFD ASCE7-10" for "Code Combinations", and "IBC2012" for "Seismic".

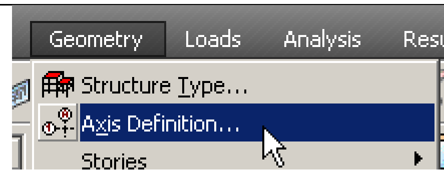
Once you've done that, press "Save current parameters as default" and then press "OK".



Section 2 – Structural Modeling

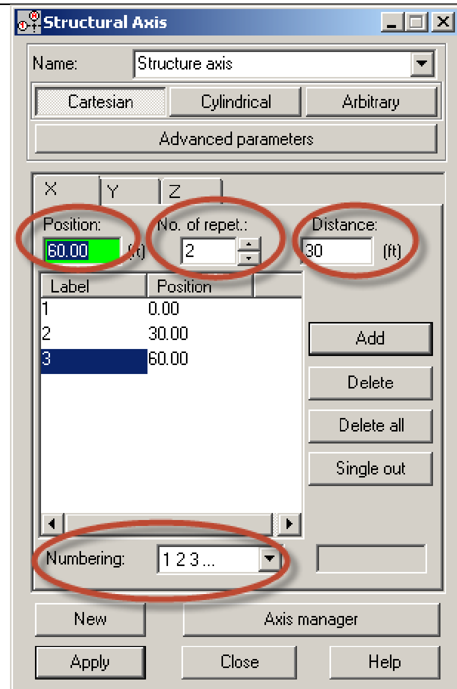
Modeling Structure Axes (5min)

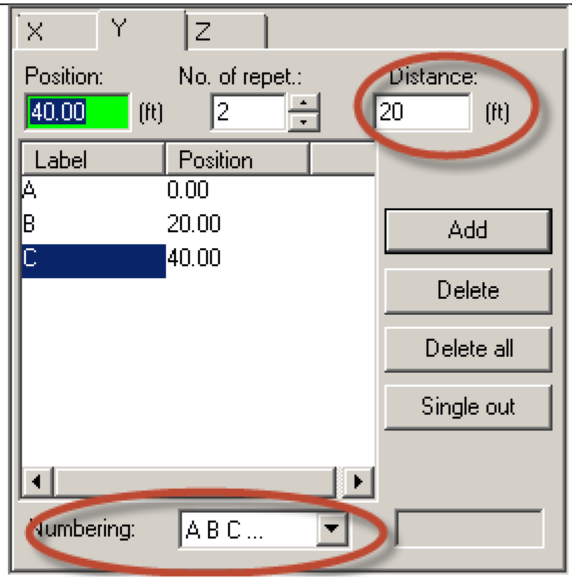
We're going to start by laying out some project axes to draw our structure. These are not required, you can simply get out there and start clicking members if you like. The grids are just helpful for reference and snapping points. Open the grids dialog from the Geometry menu: Geometry>Axis definition. Or use the axes button from the modeling toolbar on the right: 

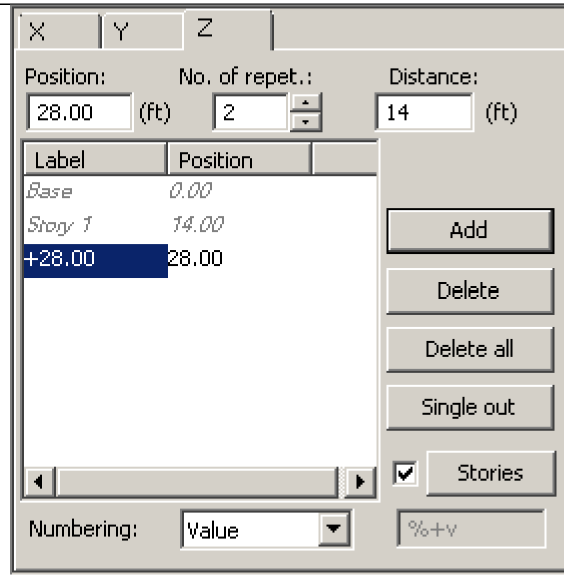


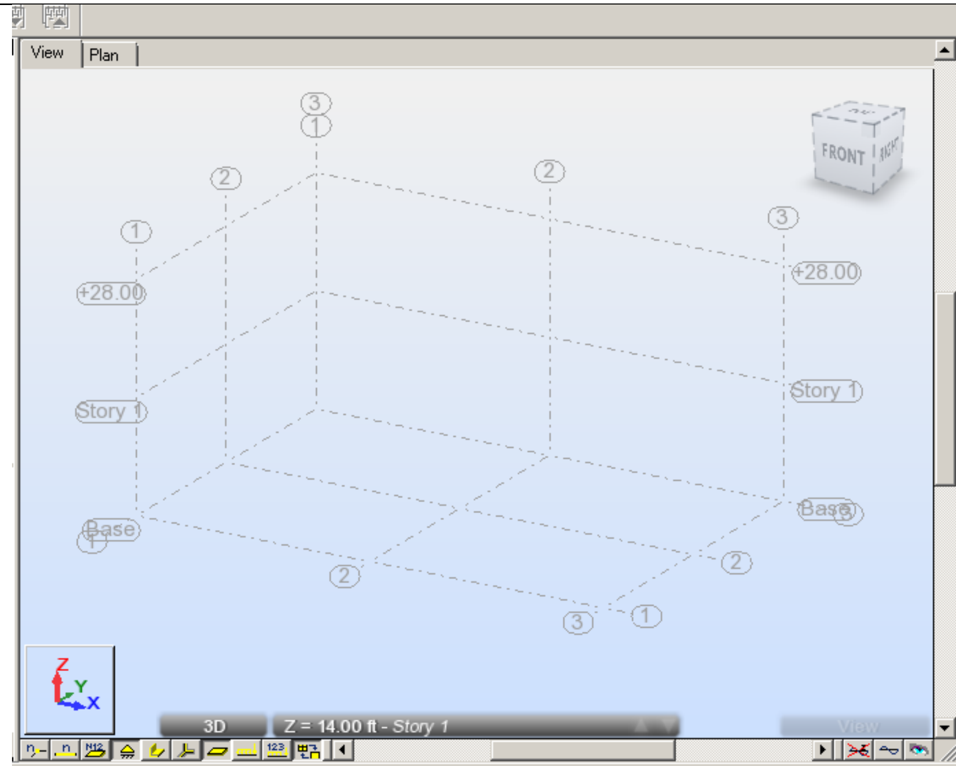
On the "X" tab, enter 0.0 for the position, then 2 for "No. of repet:" and 30.0 for Distance and select "1 2 3…" for Numbering at the bottom then press "Add".

You should now have 3 X axes 1, 2 and 3 at positions 0, 30, and 60 as shown here

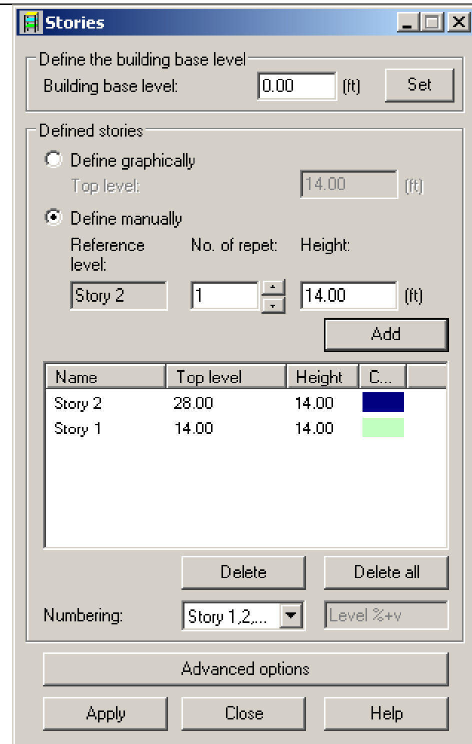


Now Switch to the "Y" tab and do the same except use 20 for the spacing which will give us a rectangular layout and choose "A B C…" for numbering.	

Next, give us Z grids of 0, 14, and 28 and use “value” for numbering	

Then click "Apply" and "Close" Your model should look like this:	

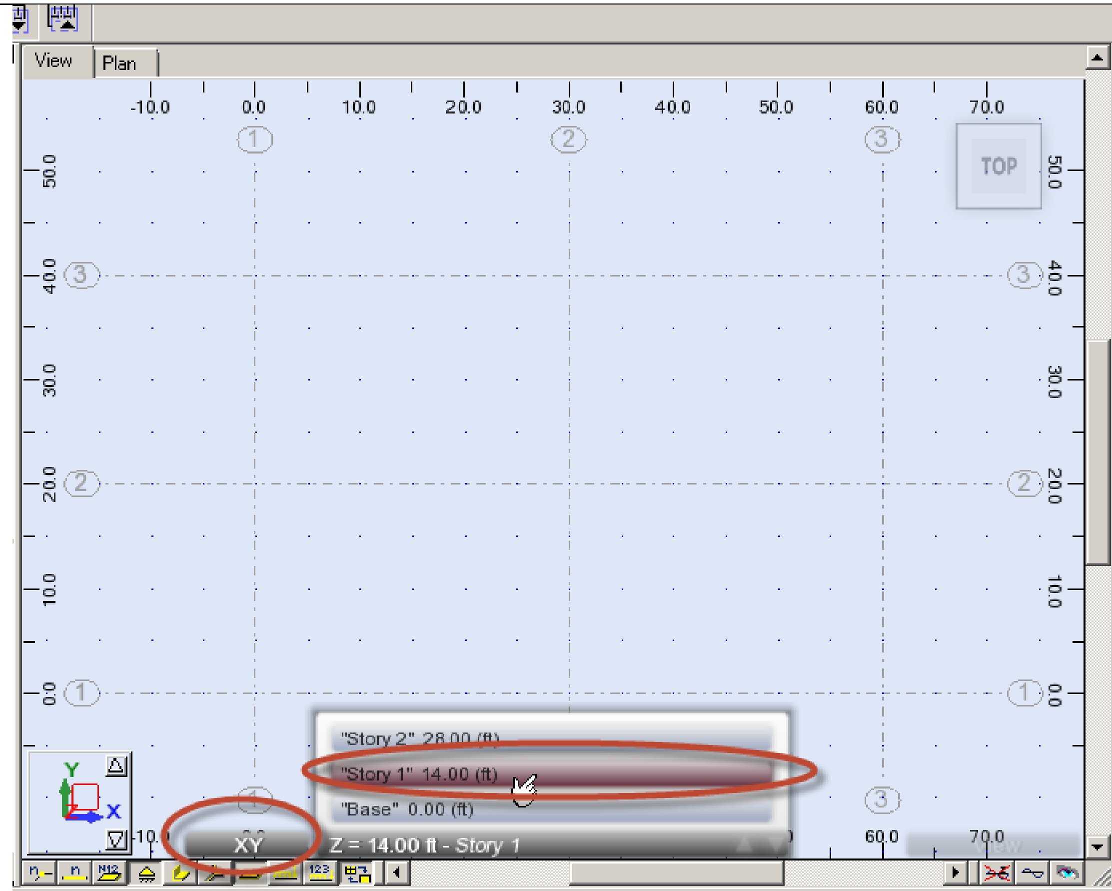
Now we'll set up our stories to match the levels we've added. Stories will be used in seismic analysis. Open Stories configuration from Geometry>Stories>Stories… First select "Define Manually" then enter "1" for "No.of Repet" and 14.0 for "Height" then press "Add". Your dialog should look like this:



Modeling Columns and Beams (15min)

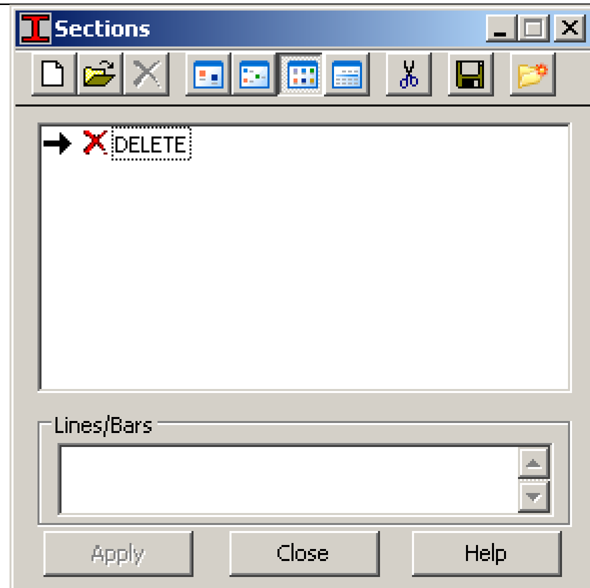
Now we'll move into laying out beams and columns for our model. You can continue with your current file, or you can open Dataset 1 – Setup and Grids.rtd to catch up.

We're going to start by orienting the project view to plan. Down at the bottom of the view, use the dynamic view controls to orient to "XY" in the first control and then select "Story 1 – 14.00ft" from the second control as shown here:

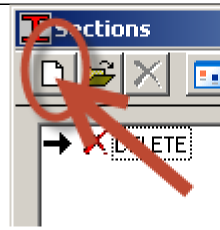


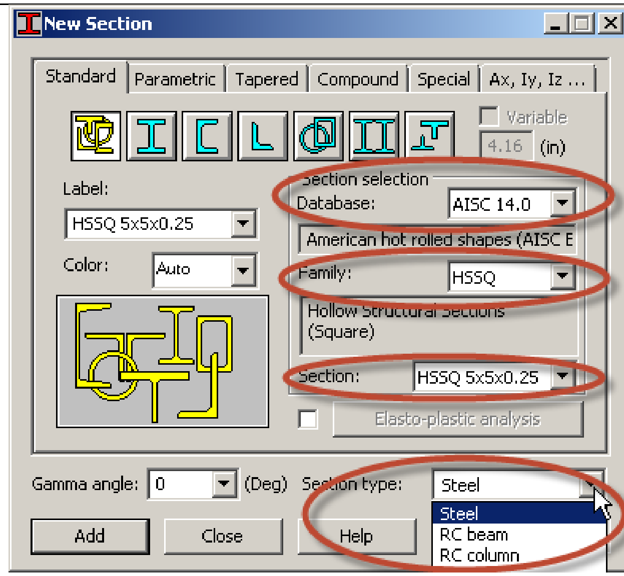
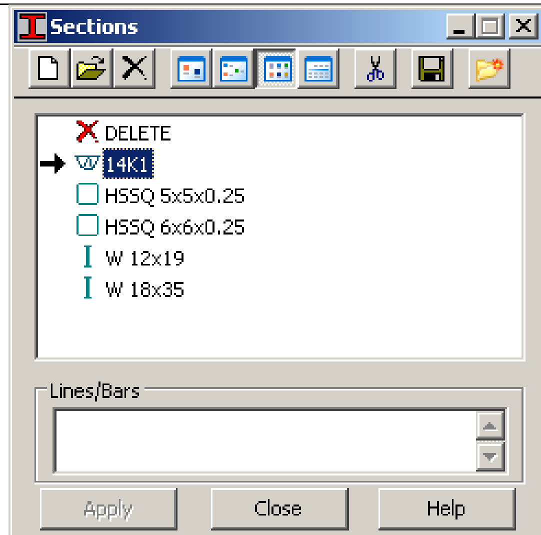
Next, we're going to configure some sections for us to use. From the geometry menu select Properties>Sections… or use the sections button from the modeling toolbar on the right:

This is the section labels dialog. It is used to configure and apply section labels to elements.



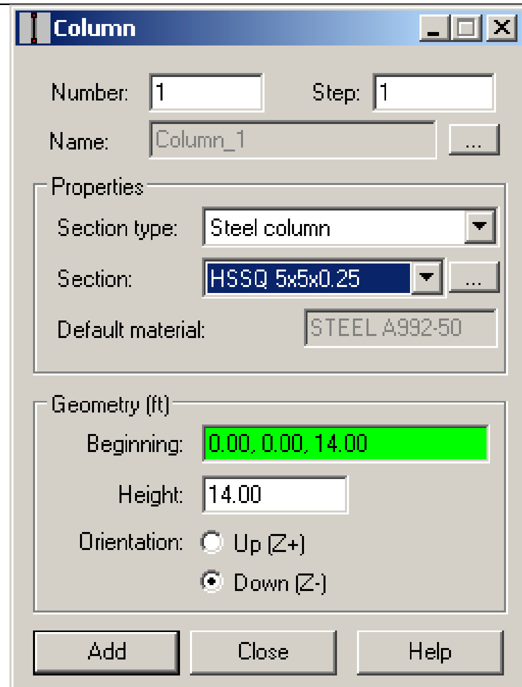
Let's configure a few sections to use. First click the new section button on the toolbar of this dialog:

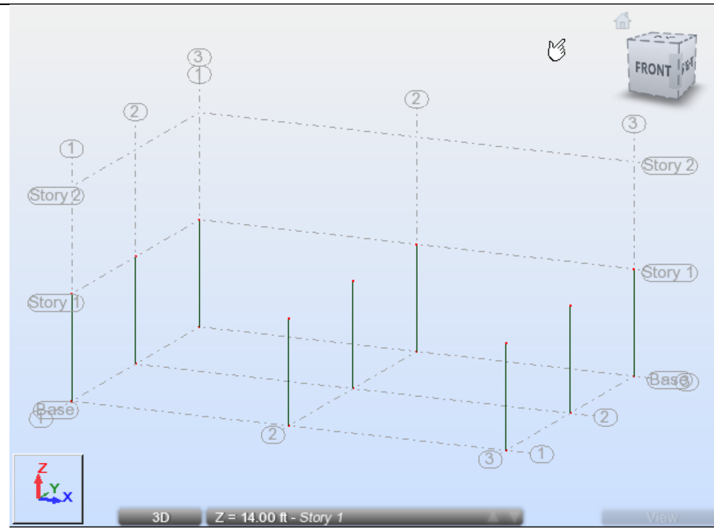


In the New Section dialog, make sure that the “Section Type” dropdown is on “Steel” (look at the other options there too). Then select “AISC 14” for the database, “HSSQ” for the Family and “HSSQ 5x5x0.25” for the Section. Then press “Add”.	
Repeat this for the following sections “W12x19”, “W18x35”, “HSSQ 6x6x0.25”, and see if you can get a 14k1 joist added to the labels list as well and the close the New Section dialog: Your Section labels dialog should now look like this:	

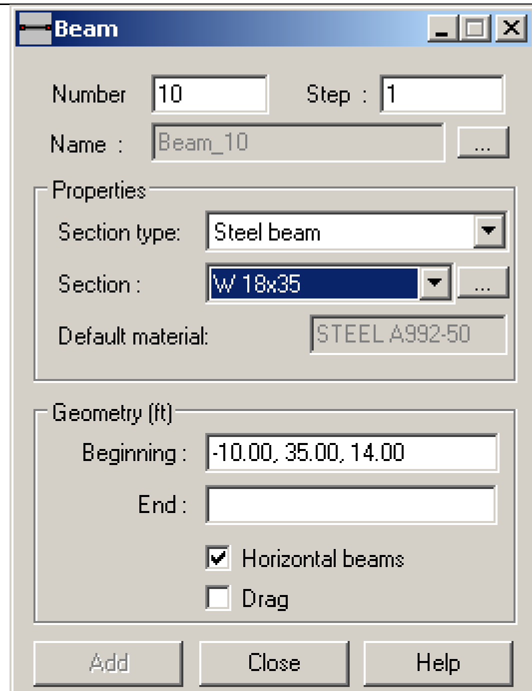
Next, start the column placement tool from the Geometry menu: Geometry>Columns…

Notice that, if you select "Steel Column" from the section type drop down, our newly configured sections will be available. Pick HSSQ 5x5x0.25 and make sure that "Height" is 14.0 and Orientation is "Down (Z-)".

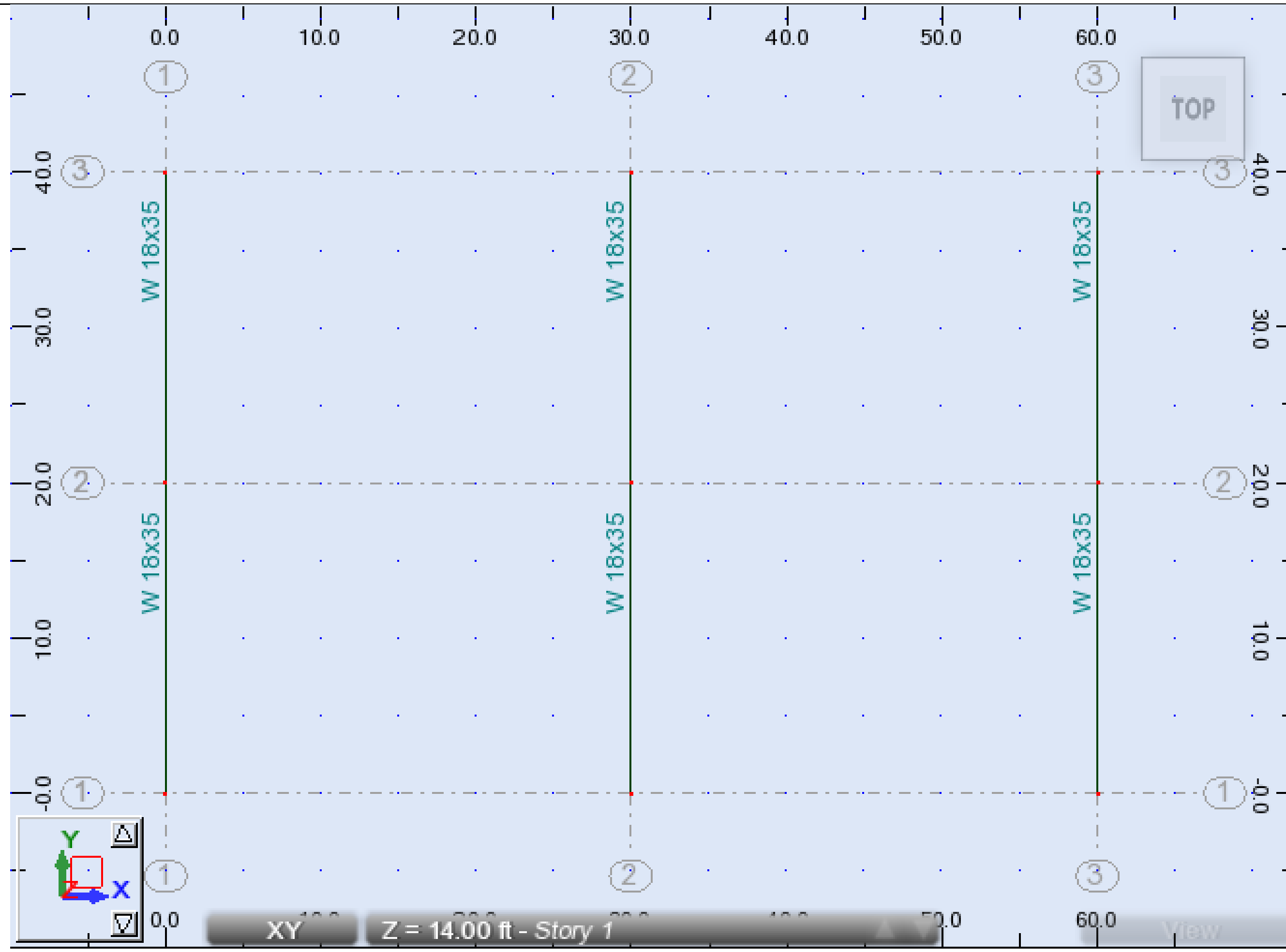


Next click to place columns at each grid intersection as shown here.	

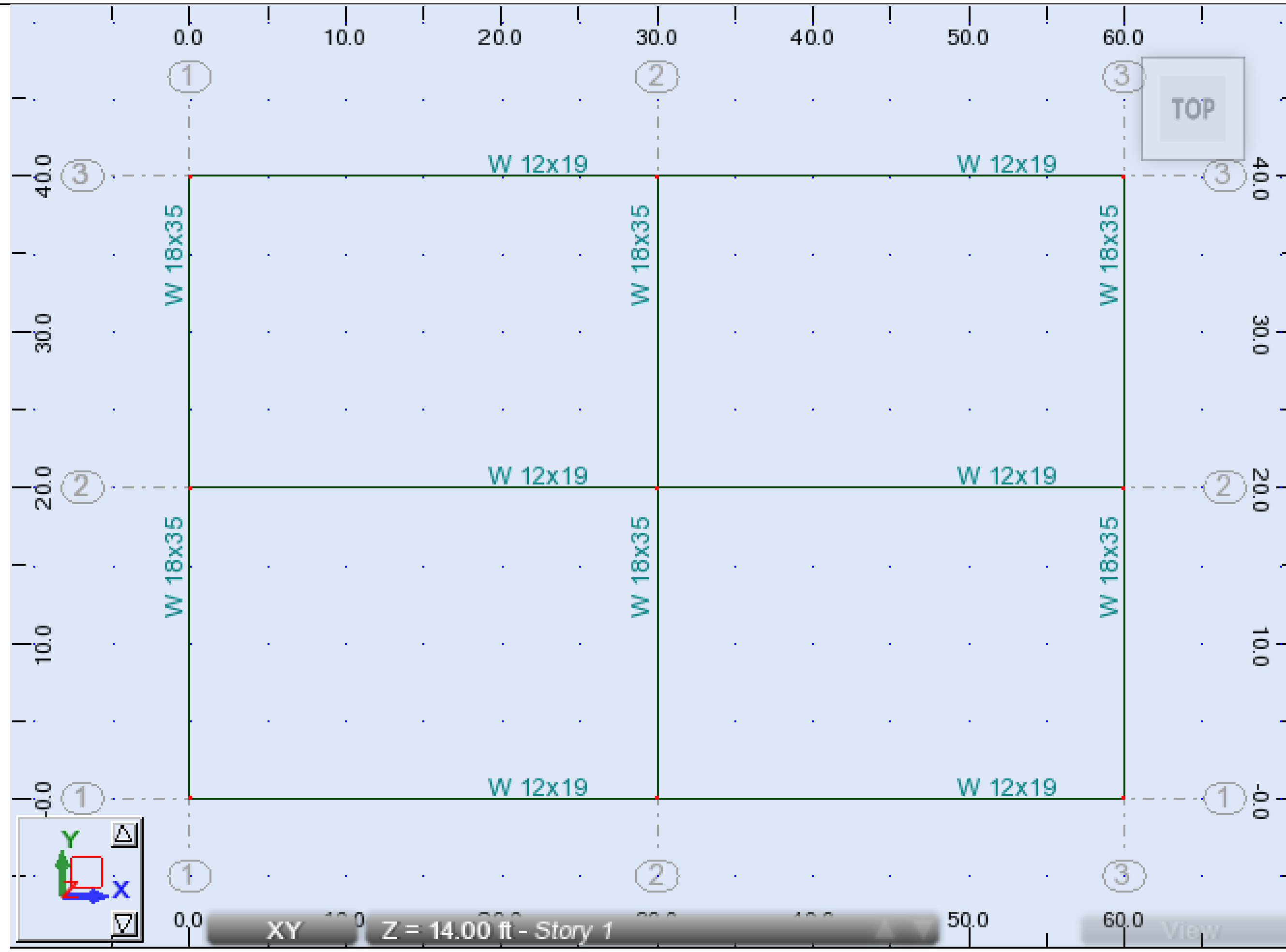
If you happened to orbit the view around to see the columns, bring it back with the dynamic view controls, then open the beams placement editor from Geomtery>Beams…Select "Steel Beam" for section type and "W18x35" for the Section.



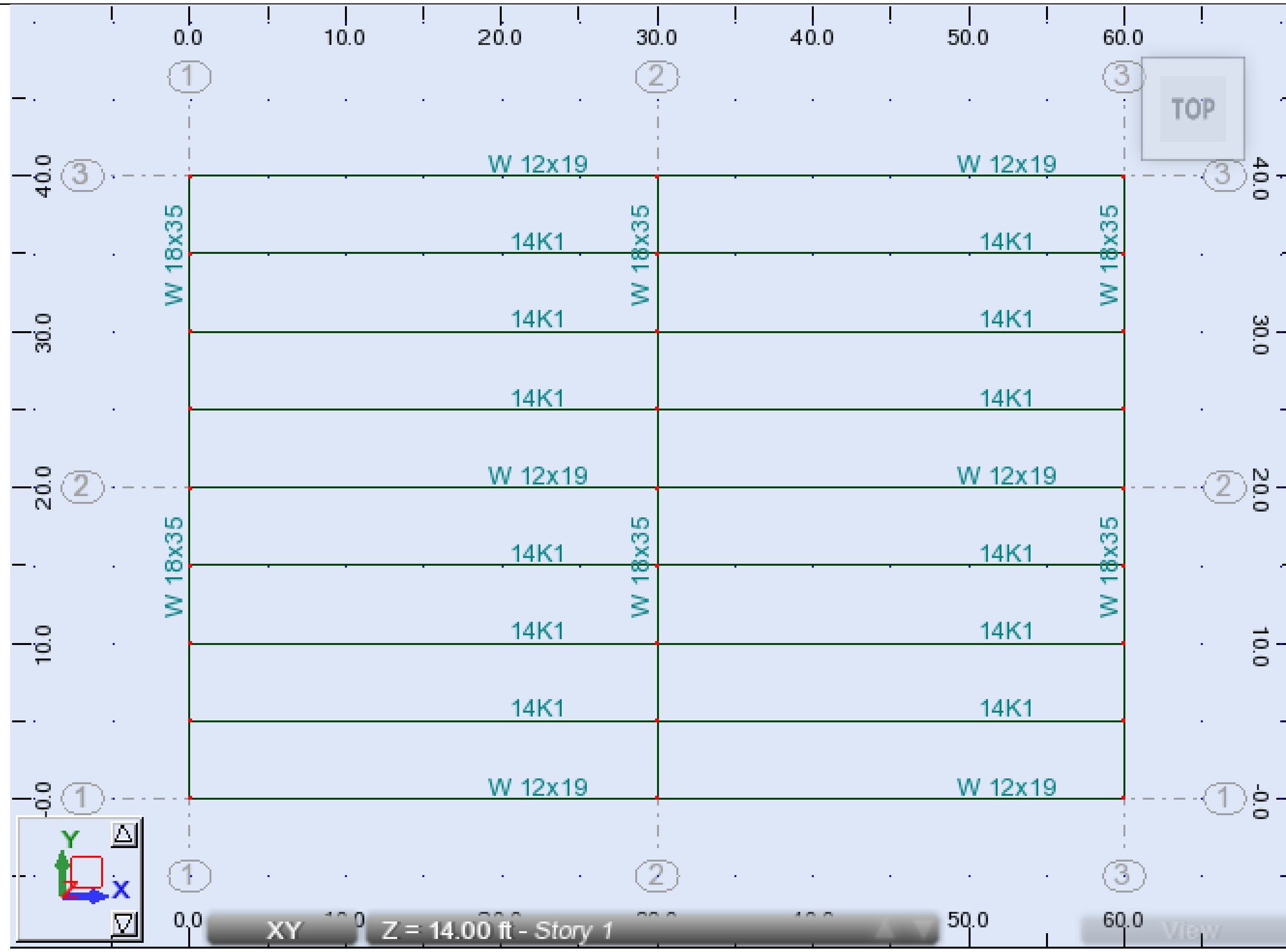
In the project click to place beams by clicking the start point and the end point of each beam. We want W18x35s on girds 1, 2, and 3. (right click and choose display>Bars>Bar descriptions>Section Names to see text descriptions)



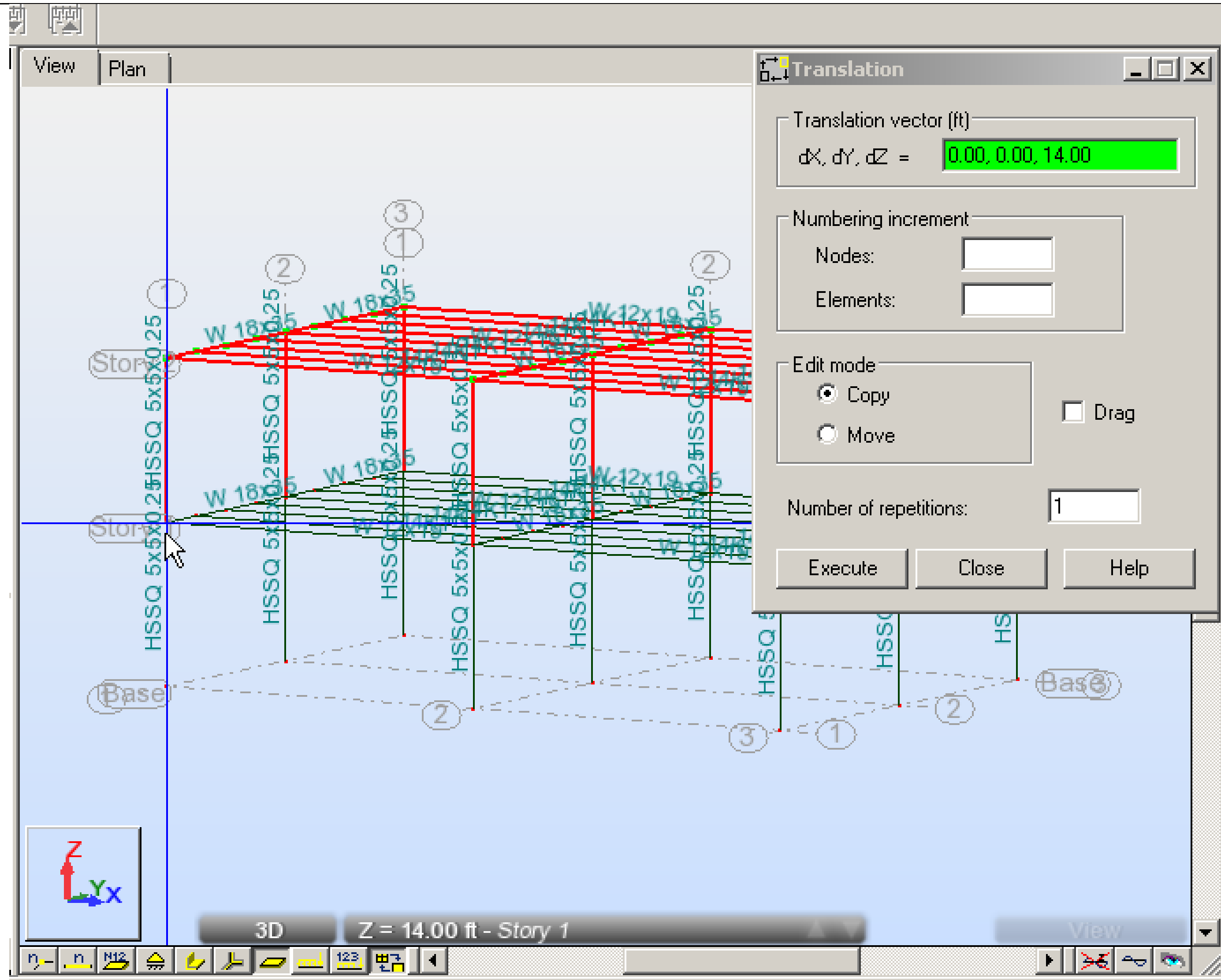
Do the same thing for W12x19 and put them on the column lines in the other direction:

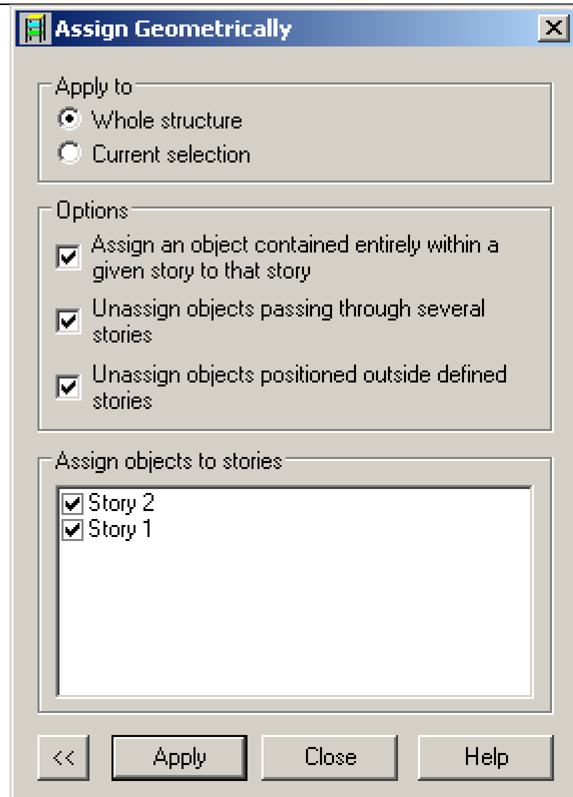


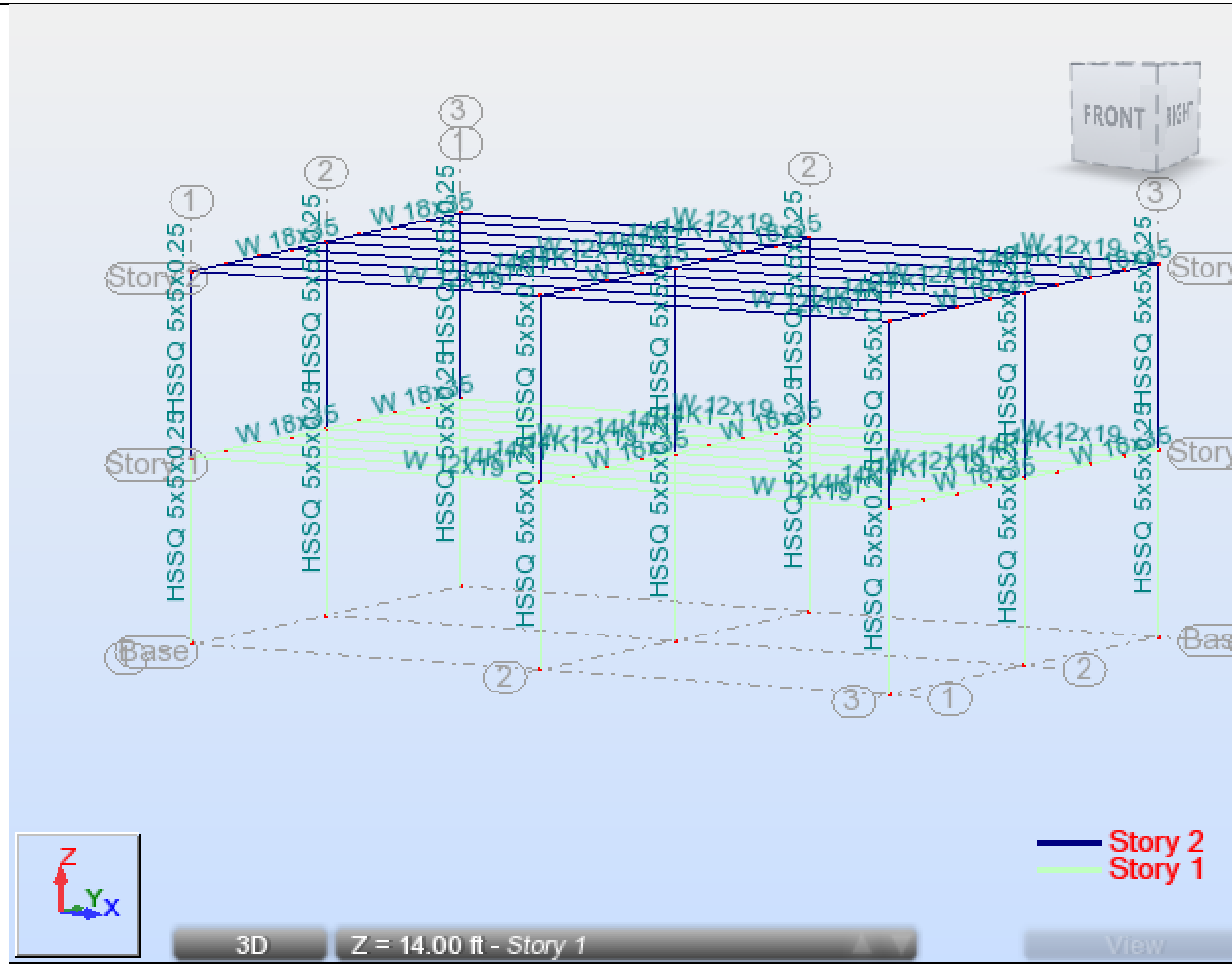
Next we'll put joists in for the floor. Just click and place each one, use coordinates, or you could divide members without splitting to add nodes, and use Edit>Edit>Move/Copy… to quickly lay them out.



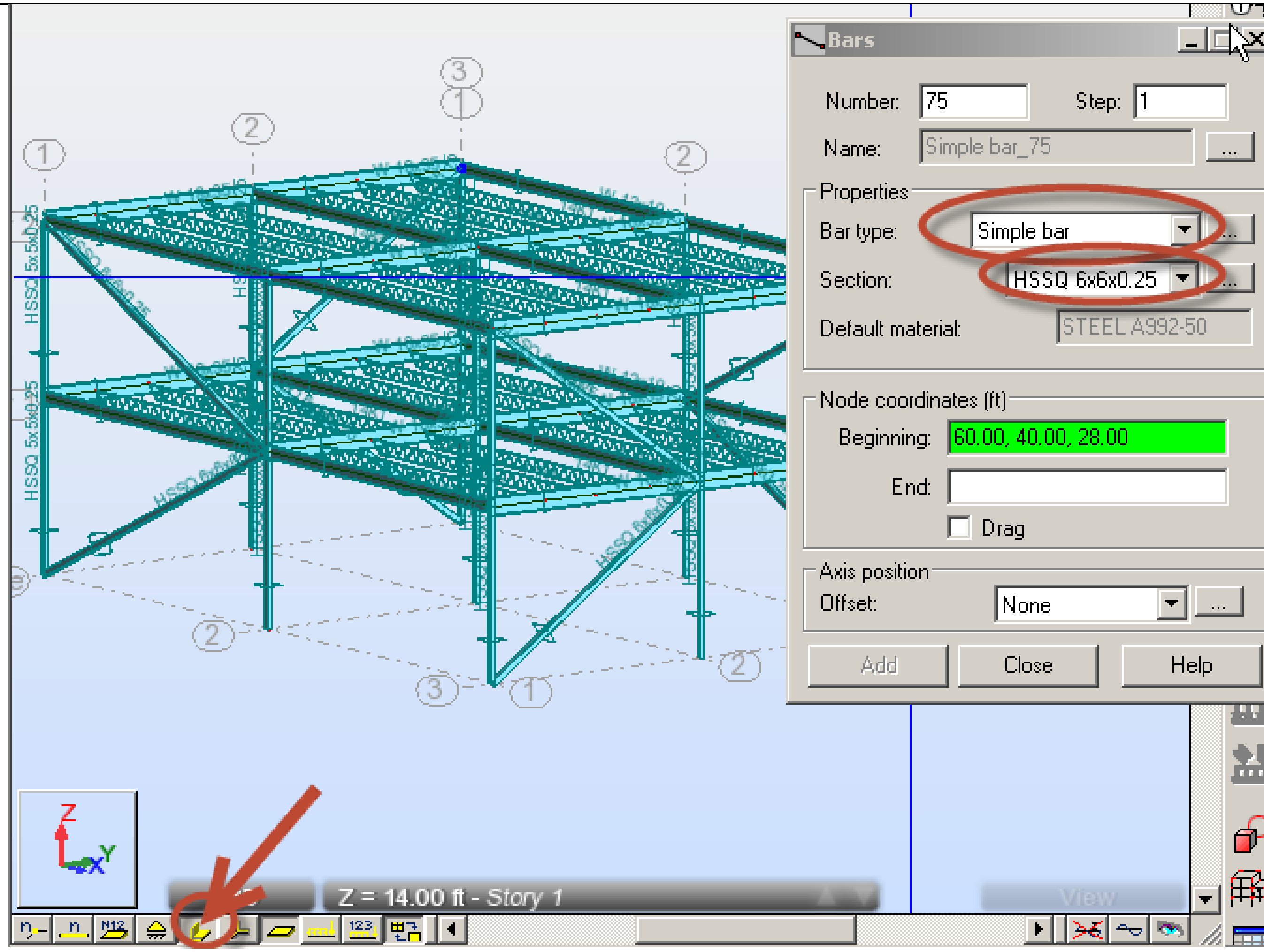
Orbit your model around to see your handiwork: Then use crossing selection to select all the elements then start the copy command from Edit>Edit>Move/Copy… then make sure "copy" is selected, and enter "1" for "Number of repetitions", then click once in the "Translation Vector" edit control then move into the project and click the bottom of a column and then the top to copy: (You can also type in 0,0,14 for the translation vector and then press "Execute")

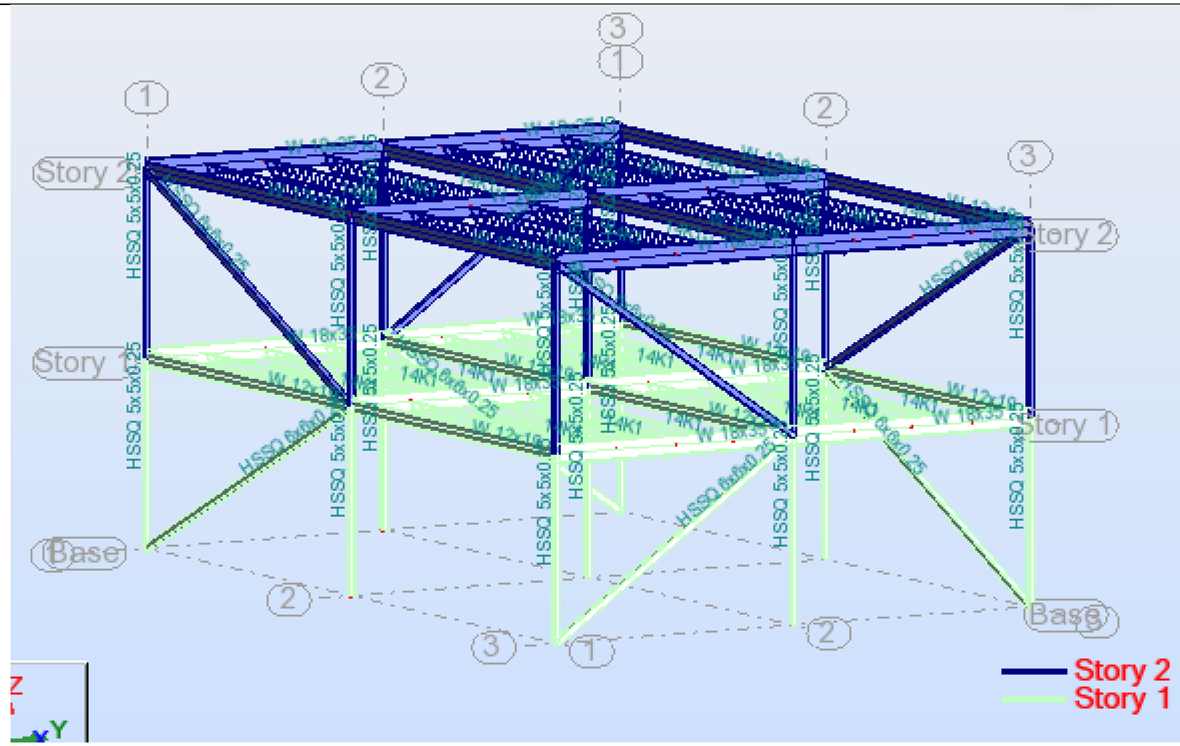


Now we'll make sure we have elements assigned to the correct stories by selecting Geometry>Stories>Assign Geometrically… Accept the defaults and press "Apply" and "Close"	

Check your stories by using Filter stories or View>Display>Mark with Colors>Stories – legend by colors to get visual feedback on proper story assignment.	

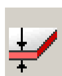
Let's add some braces with the Bars tool from Geometry>Bars… Set "Bar Type" to "Simple Bar", "Section" to "HSSQ 6x6x0.25" then click, snapping to member ends to place concentric braces on each of the four sides of the structure: (use the section shapes button at the bottom to see physical geometry)



Check our stories again. Fix any problems you notice…	

Floor and Roof Decks (5min)

You can continue with your current file, or you can open Robot Dataset 2 - Beams and Columns.rtd to catch up.

Let's get back to our XY view at story 2 level to facilitate placement of the second floor deck. First we'll configure our deck thicknesses and then create the deck elements. Open the thickness label dialog. Geometry>Properties>Thickness… or using the Thickness button from the modeling toolbar: 

Choose "orthotropic"

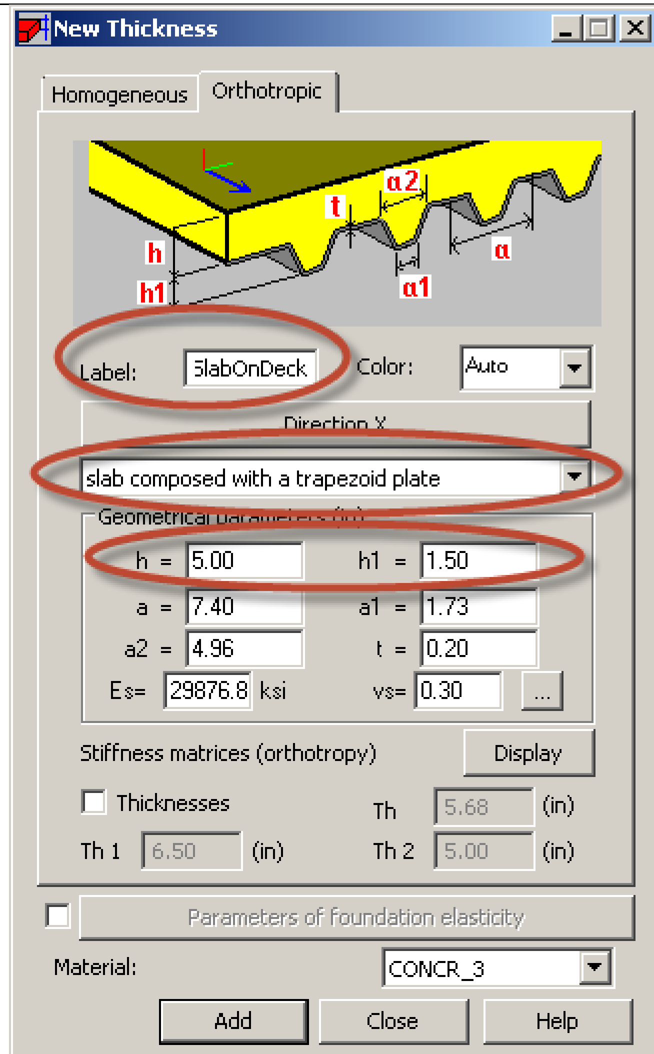
Label: SlabOnDeck

"Slab composed with trapezoidal plate"

h: 5.00

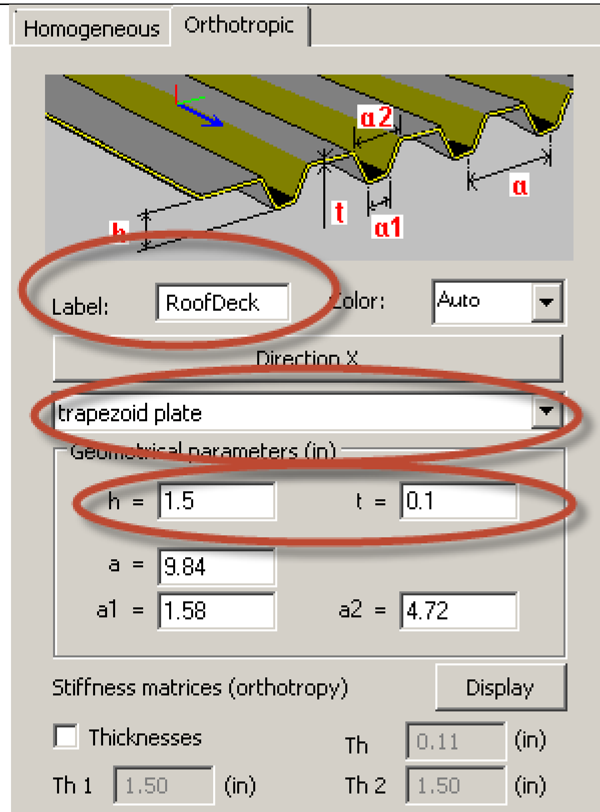
h1: 1.5

Then Press "Add"



Do the same for the roof deck, selecting "Trapezoidal plate" and 1.5" for "h". You can be more specific with the rib dimensions but it is not necessary for this tutorial.

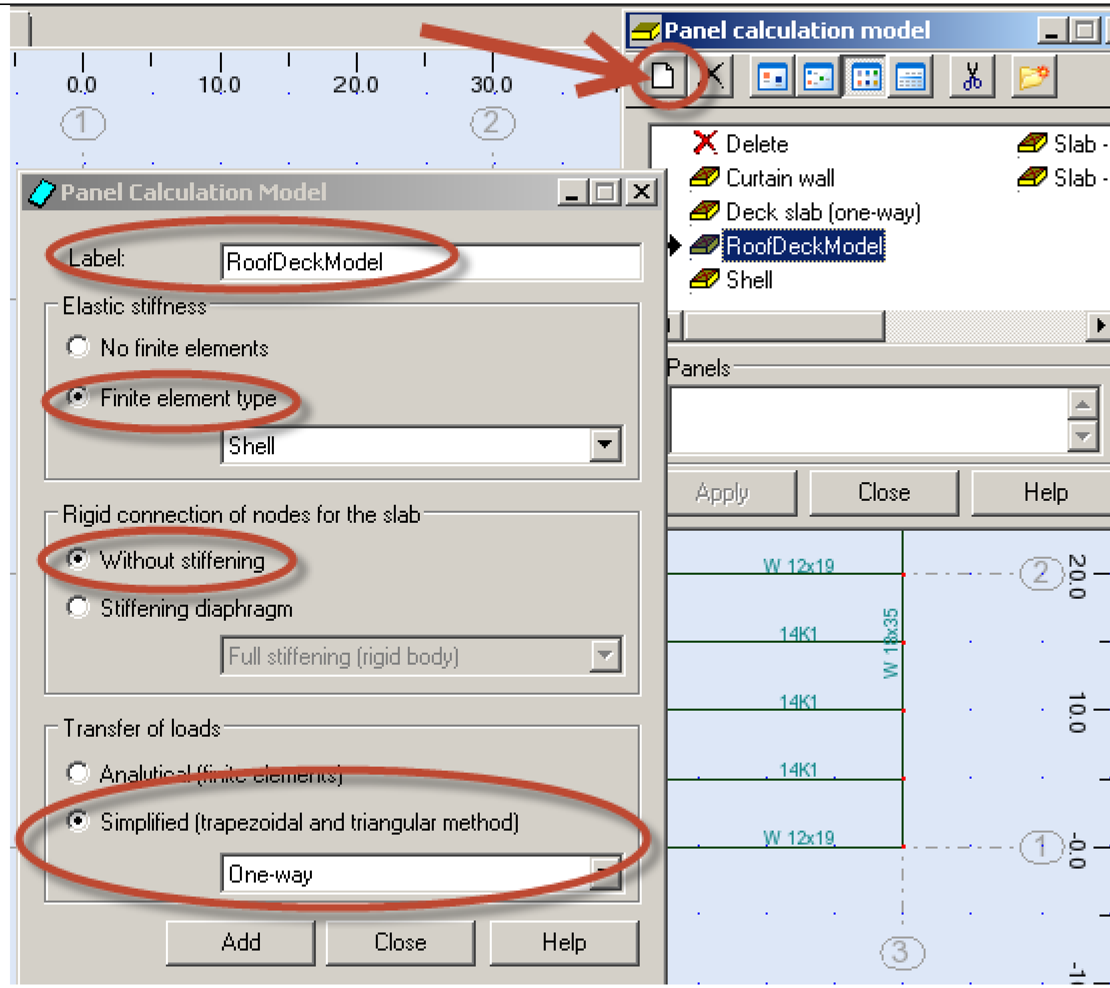
Add the roofdeck to our list of thicknesses and next we'll setup calculation models.



Now that we have thicnkesses, we'll configure our calculation models. Open Panel Calculation Models from the Geometry>Properties>Panel Calculation Model…

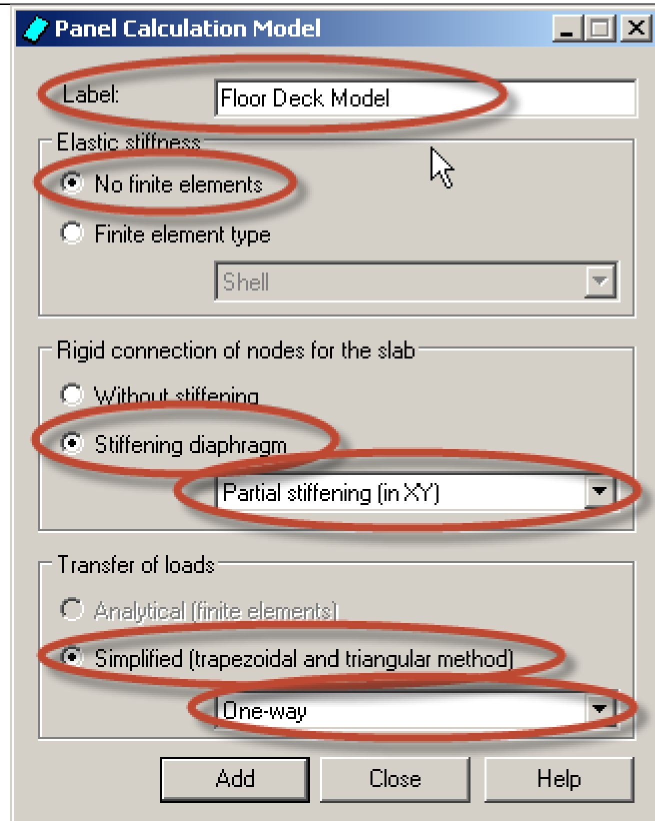
Click on the new button to add a new model and configure the parameters of RoofDeck as shown.

Make sure to "Add" it to the list of labels.



Do the same thing for the floor deck but use these settings:

Add it to the label list and then close the labels dialog.



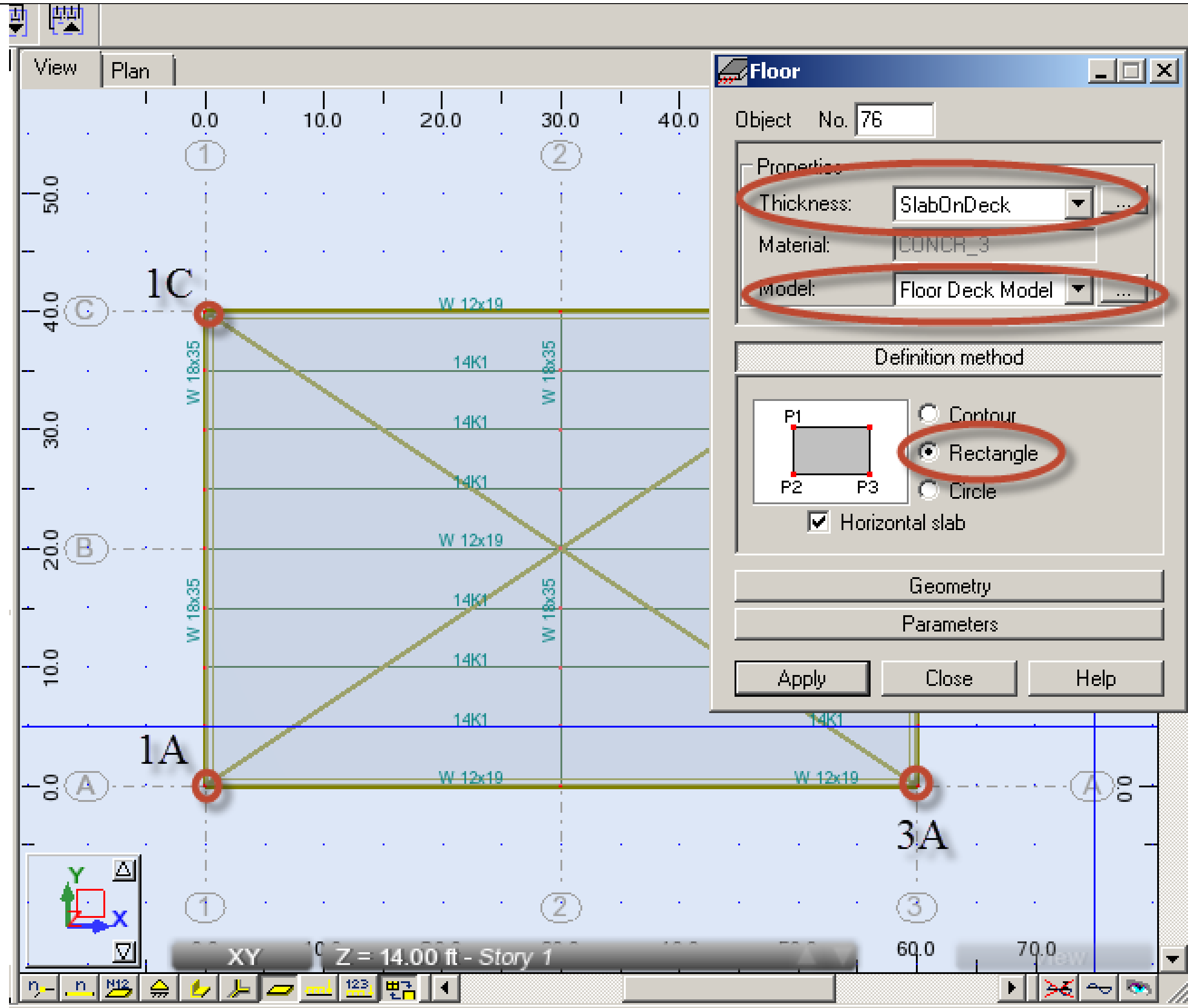
Start the floor tool from Geometry>Floors… or from the modeling tool bar with the Floors button: 

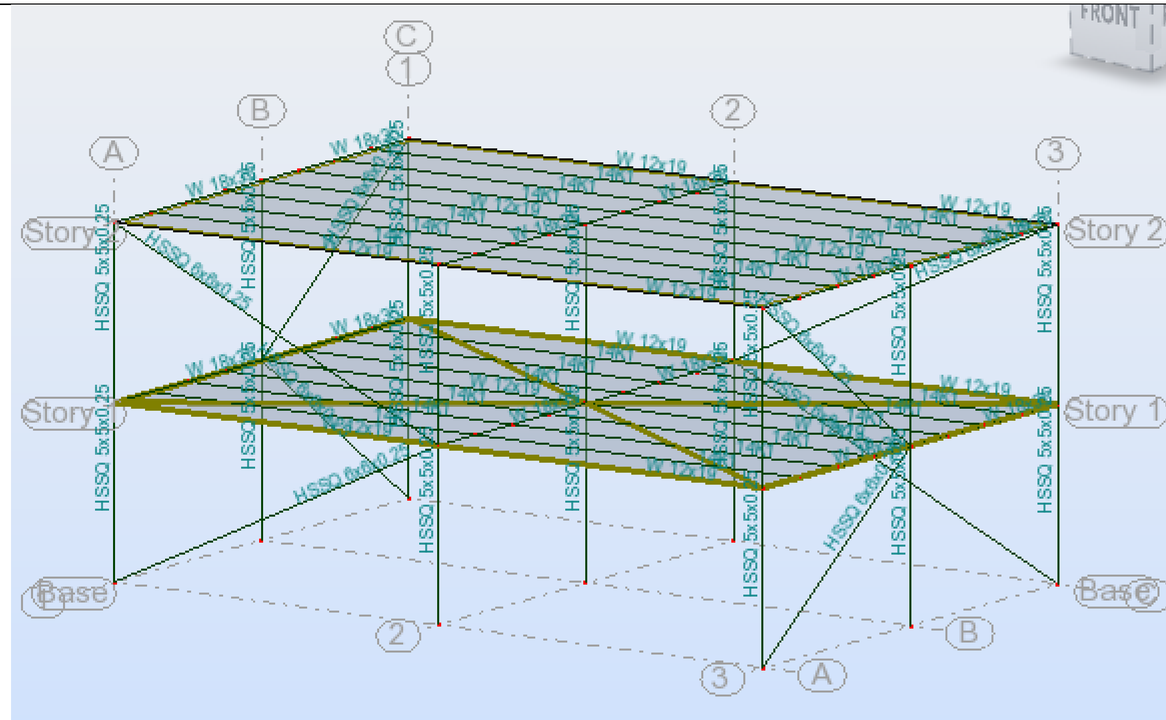
Select “SlabOnDeck” for the Thickenss

Select “Floor Deck Model” for the Model

Select “Rectangualr” for Definition method.

Next, click intersection of grid 1C, 1A, and 3A to define the three points of the rectangular floor.



Move up to the roof level and repeat to create the roof deck:	

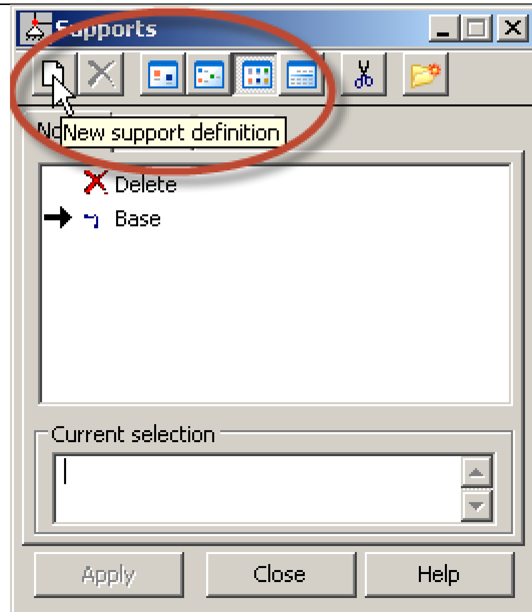
Section 3 – Loads and Calculations

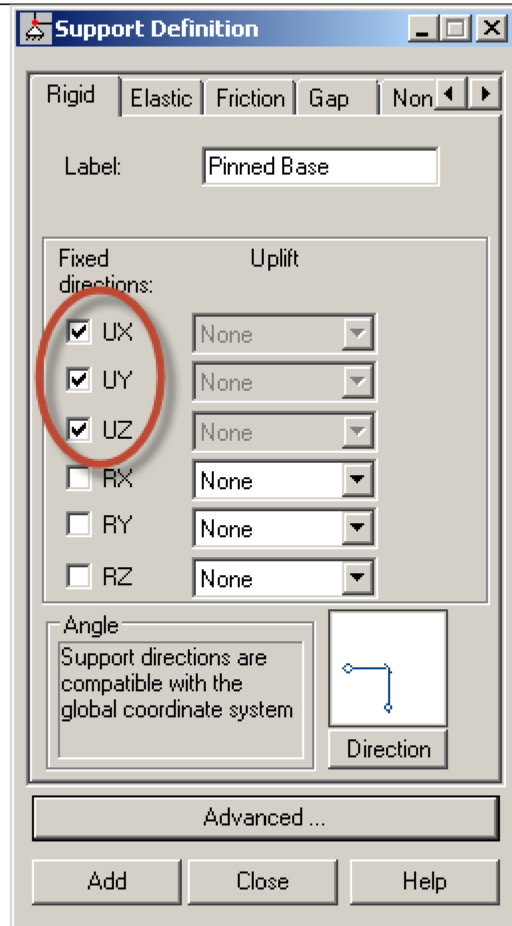
Loads and Boundary Conditions (10min)

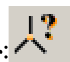
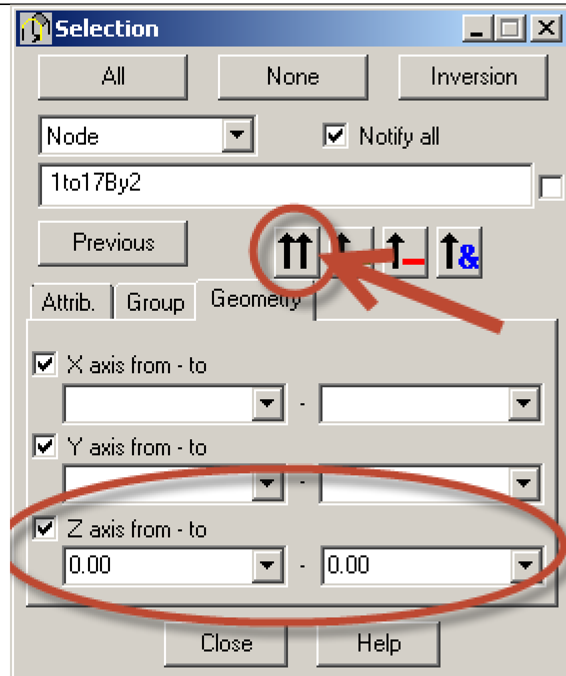
You can continue with your current file, or you can open Robot Dataset 3 - Floor and Roof Decks.rtd to catch up.

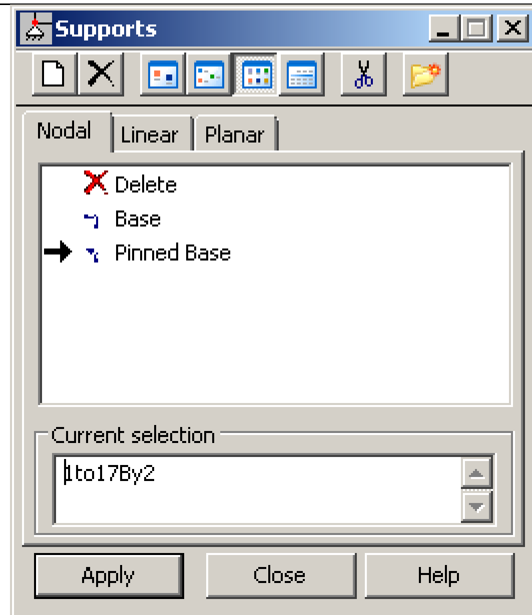
The first thing we'll do is set up our boundary conditions. To do this we'll open the "Supports" label dialog from Geometry>Supports…or the supports icon from the modeling toolbar: 

Click "new support definition"



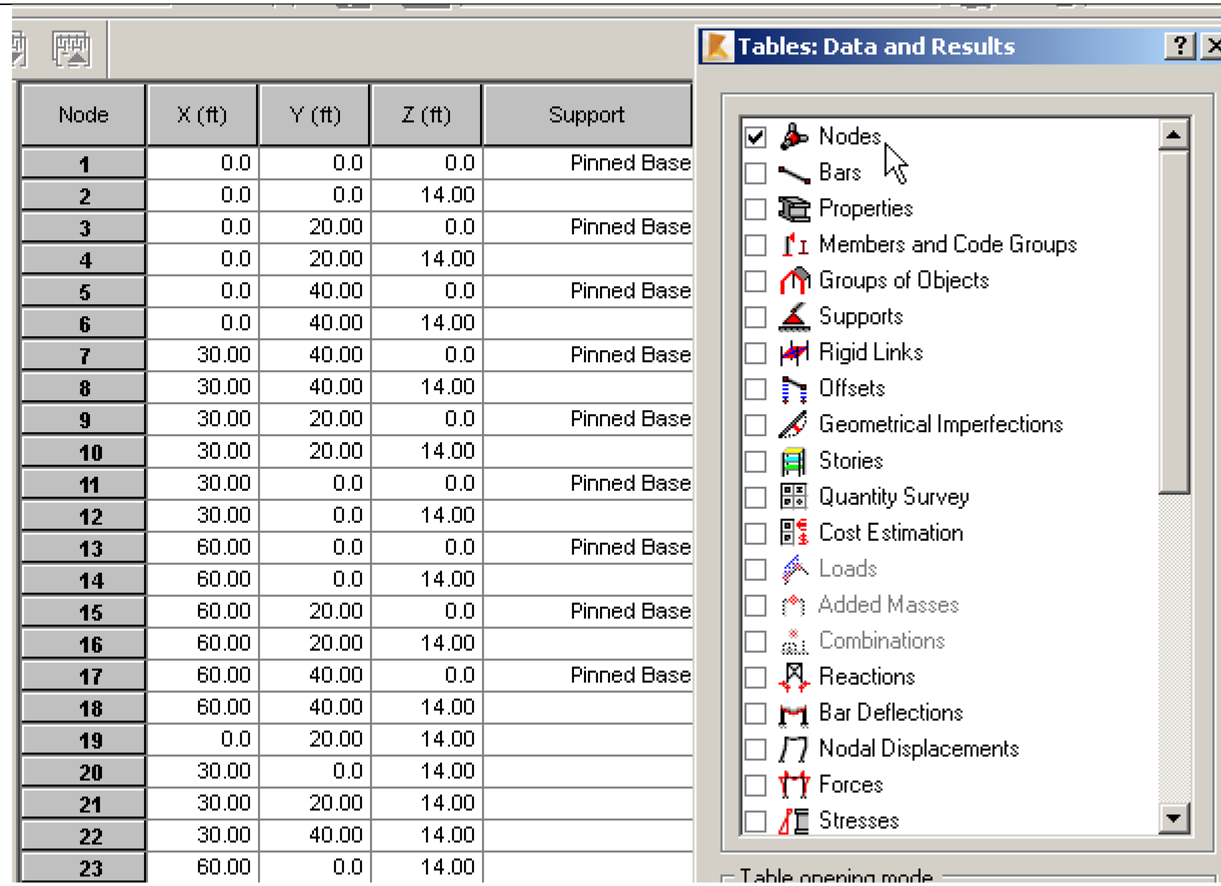
Give our support a name "Pinned Base" and tick the UX, UY, and UZ directions then press "Add".	

Now our supports lables dialog should have our new "Pinned Base" support definition. We need to apply it to the base nodes of our structure. We'll use the selection dialog to accomplish this task. First start the node selection dialog from the selection bar:  Use the "Geometry" Tab and select Zaxis from 0.00 to 0.00 then use the double up arrow to create the selection:	

Next, click into the "Current Selection" edit control and the selection will automatically populate. Then select the "Pinned Base" label and press the "Apply" button to apply the label to the nodes.	

Check the support in the Nodes Table. From View>Tables… select the “Nodes” table.

You can see that our “Pinned Base” support has been applied to the nodes we selected.



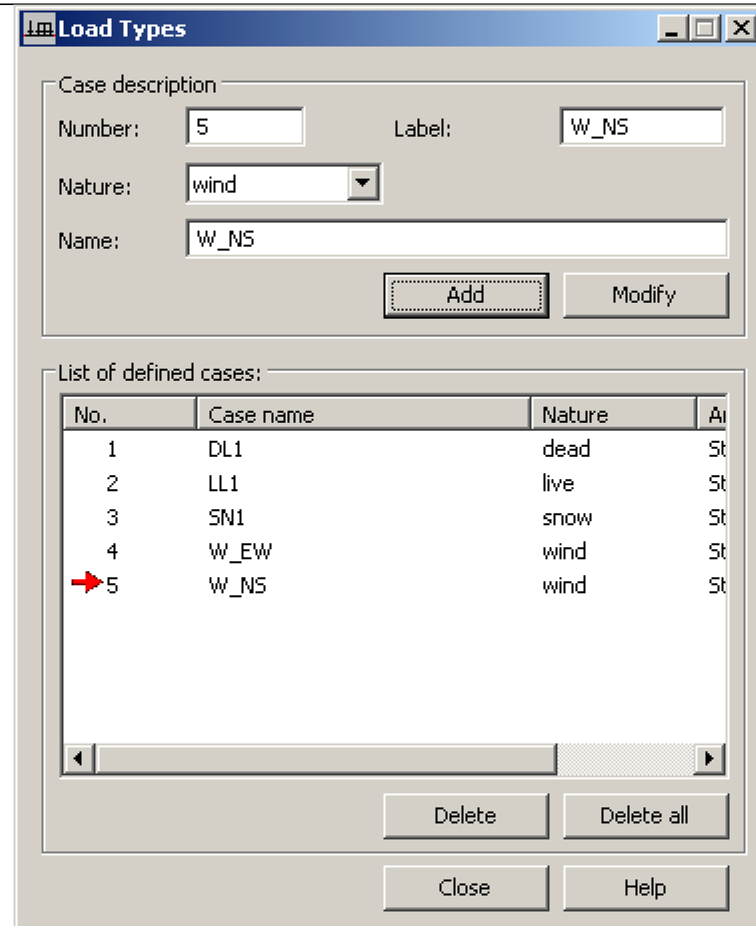
Now let’s create some load cases

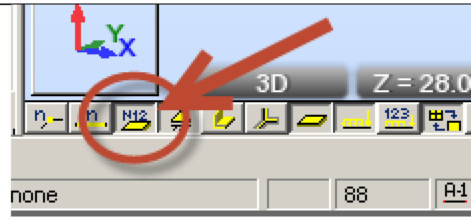
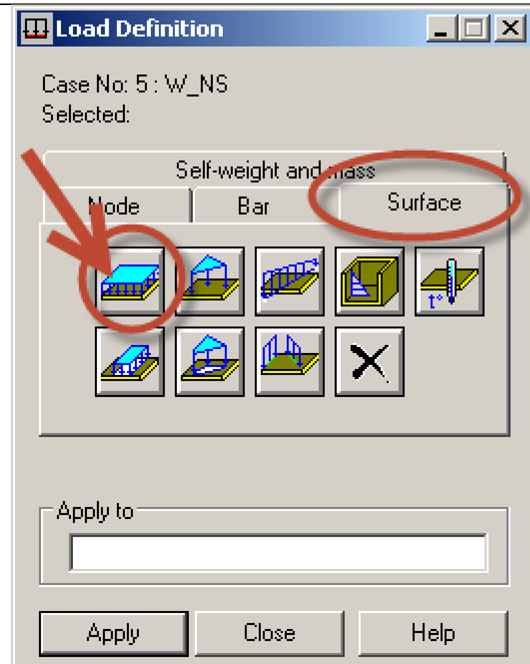
Open the Load Types Dialog from Loads>Load Types…Add the following Types.

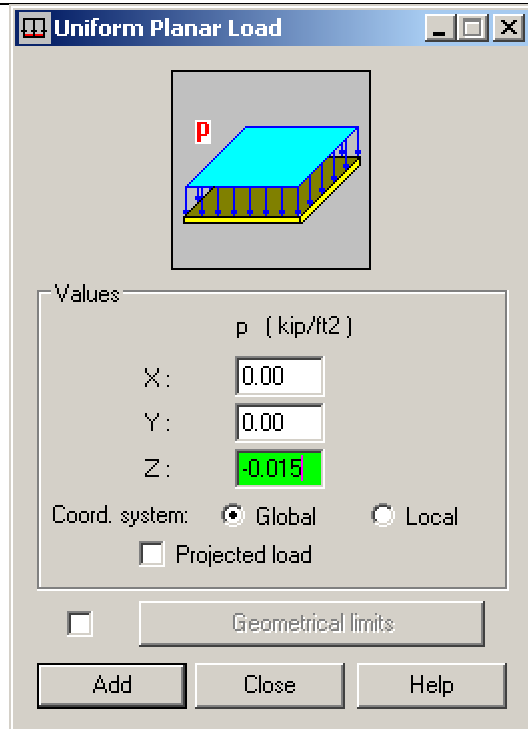
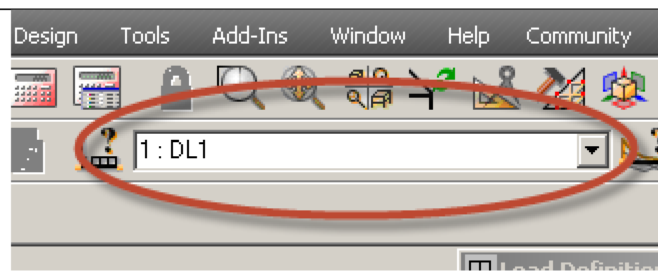
Number	Nature	Label	Name
1	Dead	DL1	DL1
2	Live	LL1	LL1

These are our basic load cases to which we will add our loads.

3	Snow	SN1	SN1
4	Wind	W_EW	W_EW
5	Wind	W_NS	W_NS

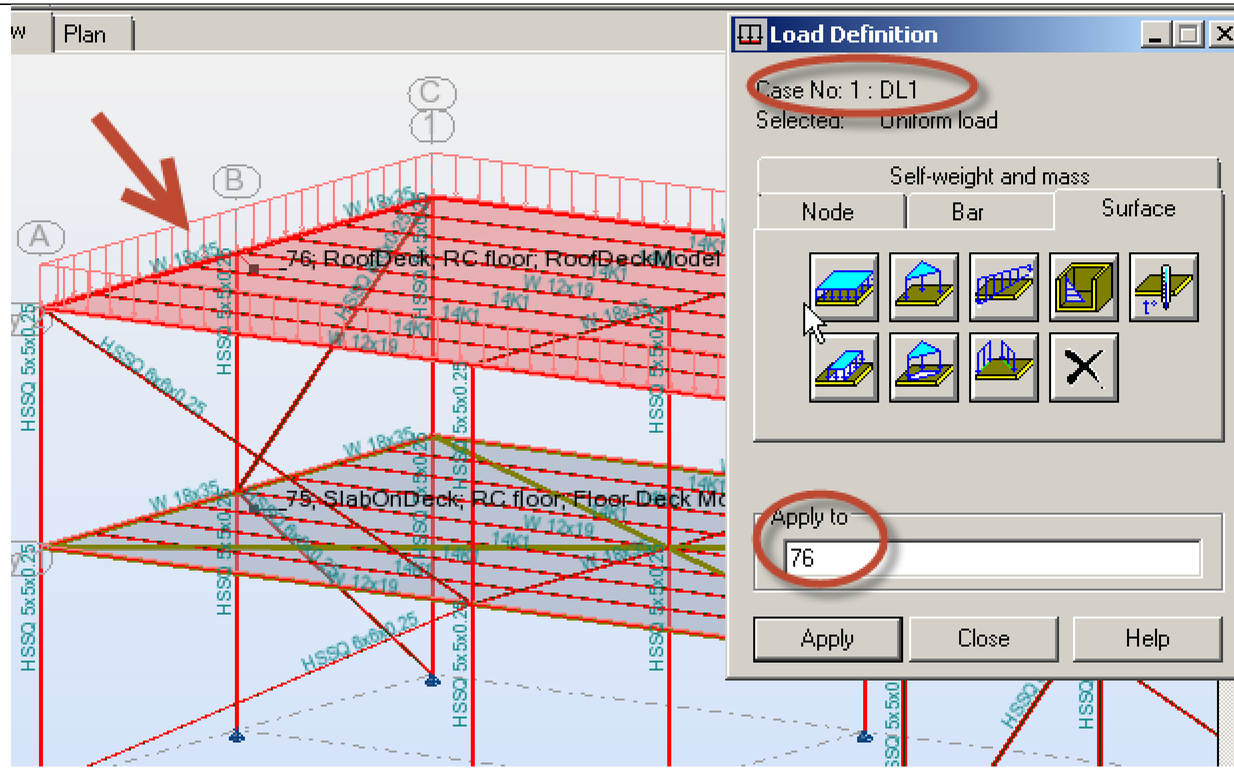


Close the Load Types dialog and before opening the load definition dialog, go ahead and turn on panel descriptions. Note the panel numbers for the floor and roof. I have 75 and 76 respectively, yours may differ.	
Start the Load Definition dialog from Loads>Load Definition… and select surface tab and click Uniform Planar Load.	

In the Uniform Planar Load Dialog enter -0.015 for the Z value and press “Add”	
Now you’re back in the Load Definition dialog, Choose 1:DL1 from the load case selector:	

Check to see that the Load Definition Dialog has the DL case shown then enter the roof deck element number in the "Apply to" edit control and press "Apply".

You will see your load created in the view as visual feedback of the application.



Do the same for Floor Dead, Floor Live, Snow, and Wind uplift on the roof

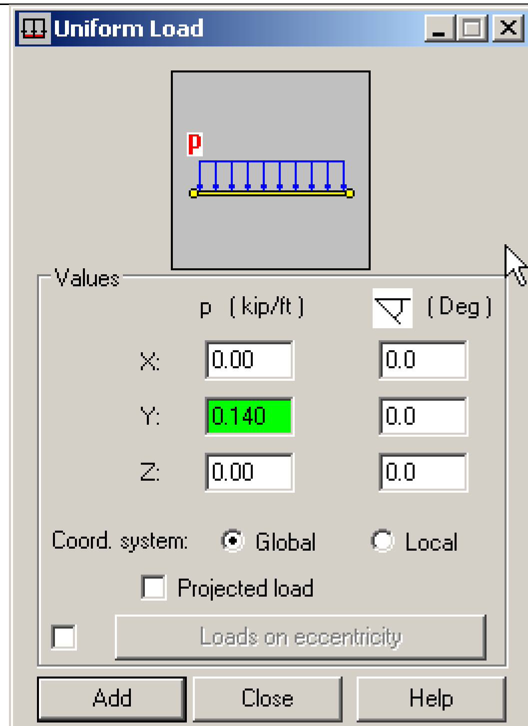
Floor Dead (DL1)	Z: -0.025
Floor Live (LL1)	Z: -0.050
Snow Load (SN1)	Z: -0.040
Wind Load (W_EW and W_NS)	Z: 0.02

Take a look at the loads table for verification: View>Tables>Loads

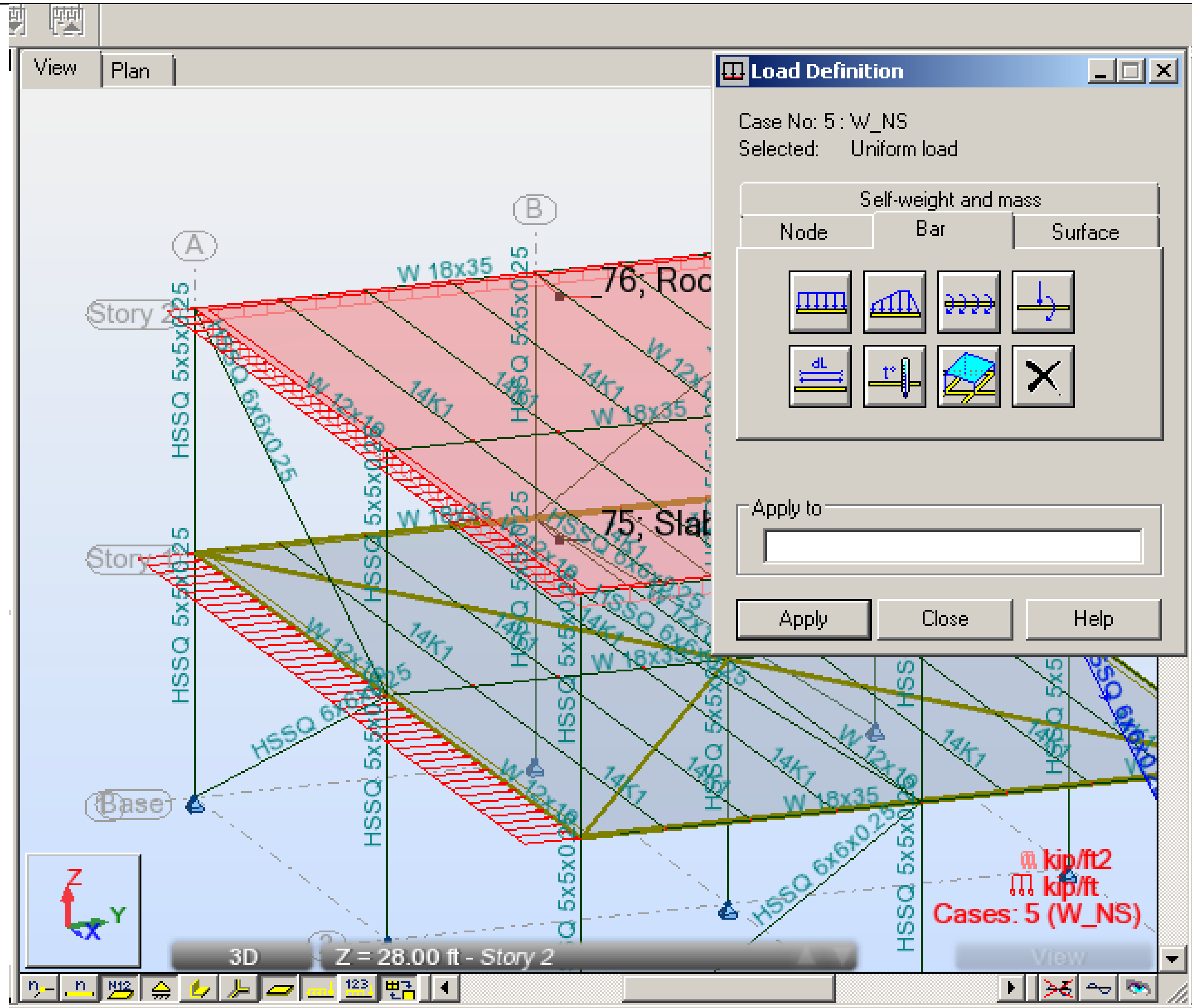
	Case	Load type	List					
	1:DL1	self-weight	1to76	Whole structu	-Z	Factor=1.00	MEMO:	
	1:DL1	(FE) uniform	76	PX=0.0	PY=0.0	PZ=-0.01	global	not project
	3:SN1	(FE) uniform	76	PX=0.0	PY=0.0	PZ=-0.04	global	not project
	1:DL1	(FE) uniform	75	PX=0.0	PY=0.0	PZ=-0.02	global	not project
	2:LL1	(FE) uniform	75	PX=0.0	PY=0.0	PZ=-0.05	global	not project
	4:W_EW	(FE) uniform	76	PX=0.0	PY=0.0	PZ=0.02	global	not project
	5:W_NS	(FE) uniform	76	PX=0.0	PY=0.0	PZ=0.02	global	not project
*								

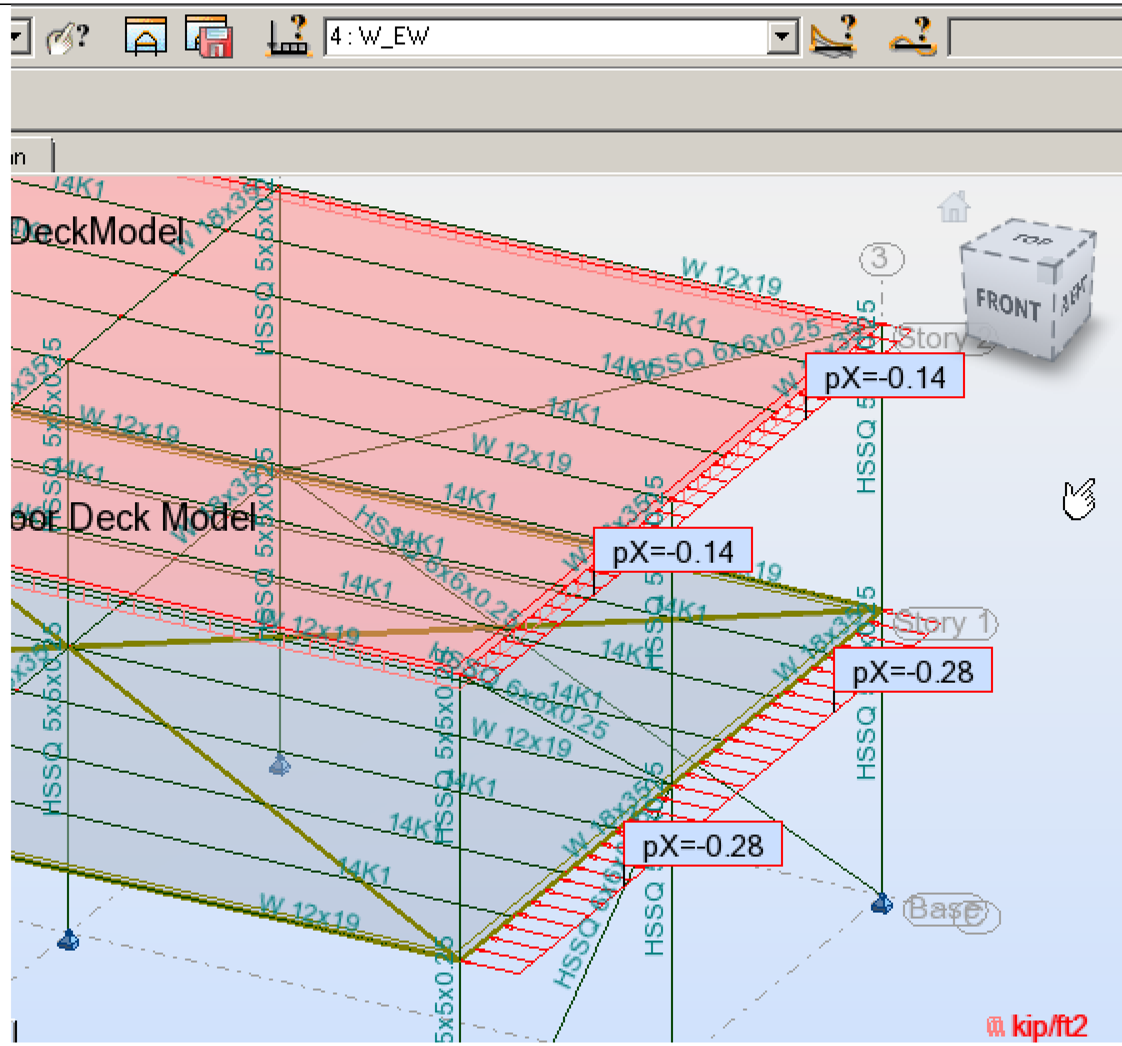
Next we'll apply wind loads for the side walls as linear loads at the levels as though there were vertical studs distributing the loads to the floors.

Back in the Load Definition Dialog, select the Bar tab and "Uniform Load" then in the Uniform load dialog, enter 0.140 for the Y value of load at the roof then Press "Add"



Make sure W_NS is selected in the case selector then use the click-to-apply method to apply the load to the windward beams on the south wall. Then do the same for the floor portion of Y: 0.280 kip/ft on the floor level beams



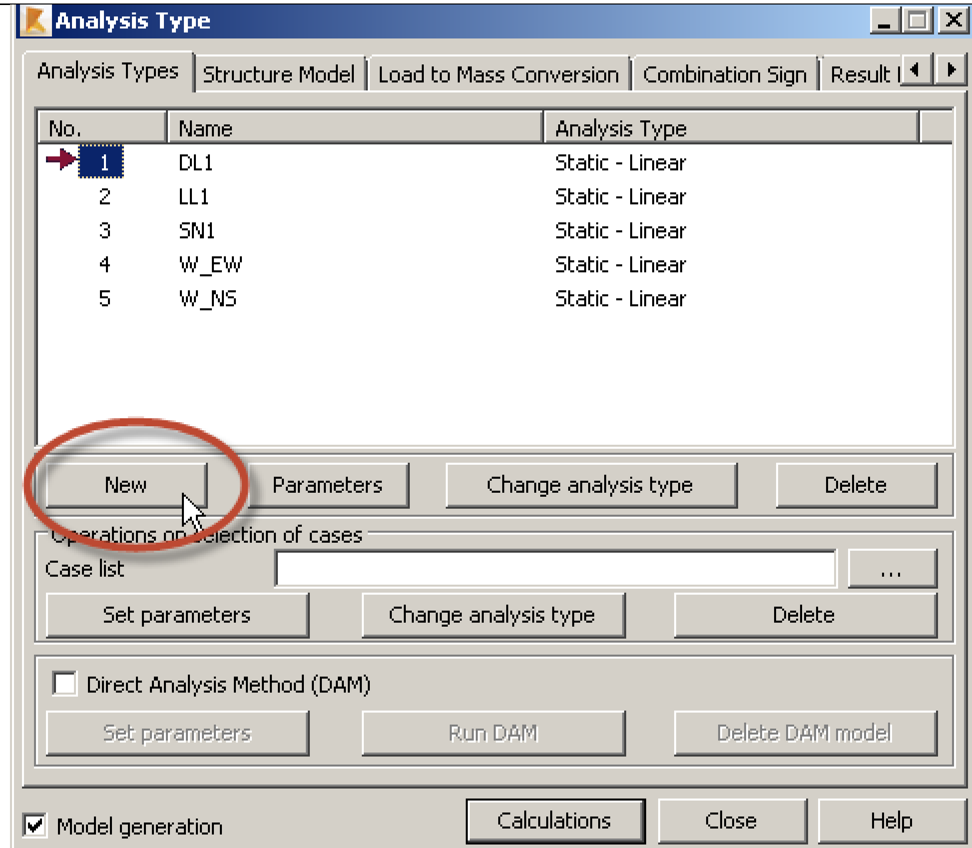
Do the same for W_EW with the same load magnitudes	

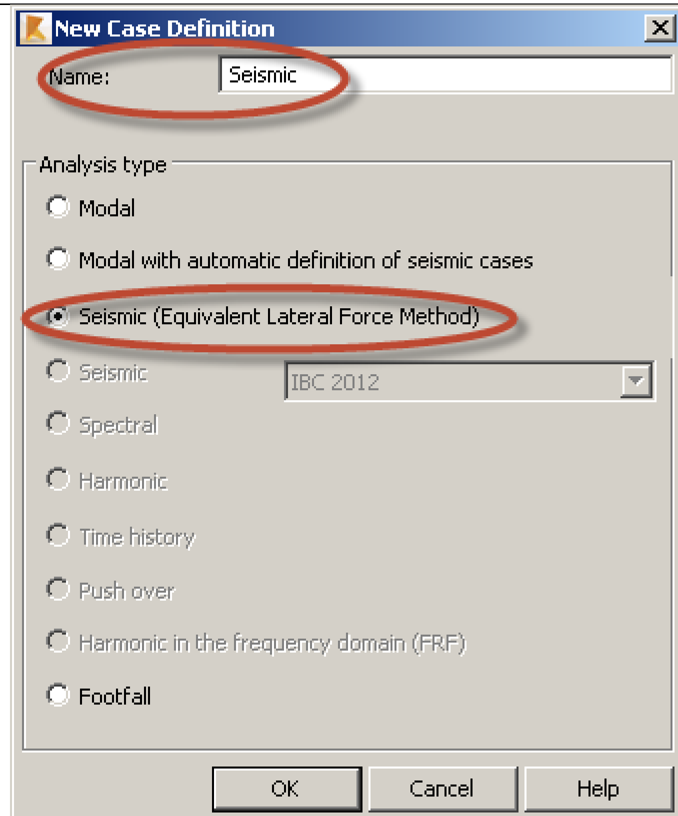
Configure Basic Seismic Load (10min)

You can continue with your current file, or you can open Robot Dataset 4 - Loads and Boundary Conditions.rtd to catch up.

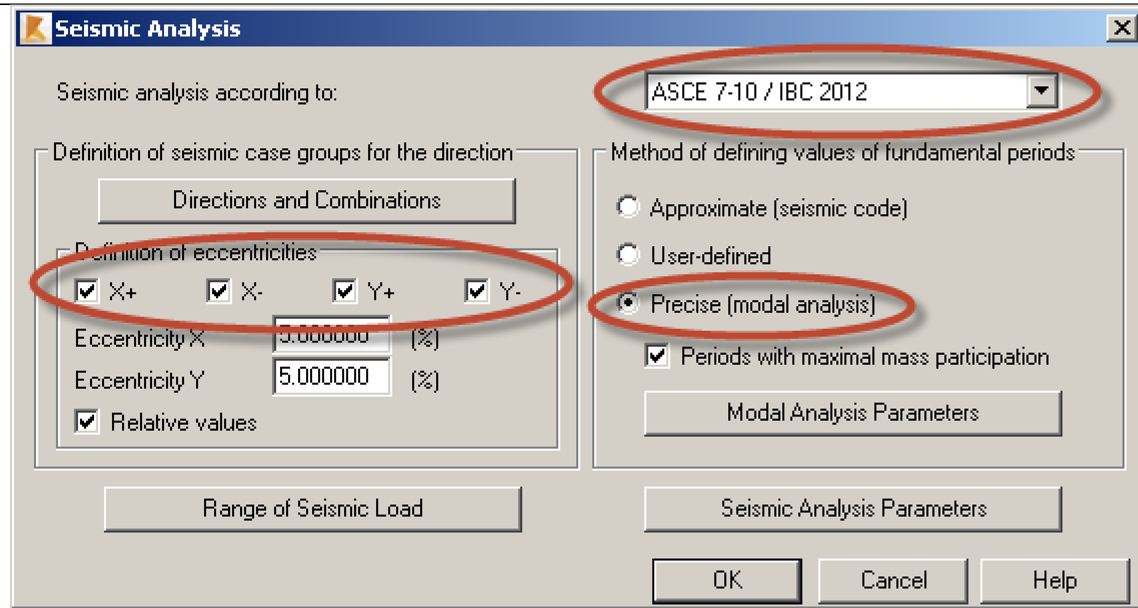
We'll configure an equivalent lateral load seismic load from the Analysis types menu. Analysis>Analysis Types…

Select "New"

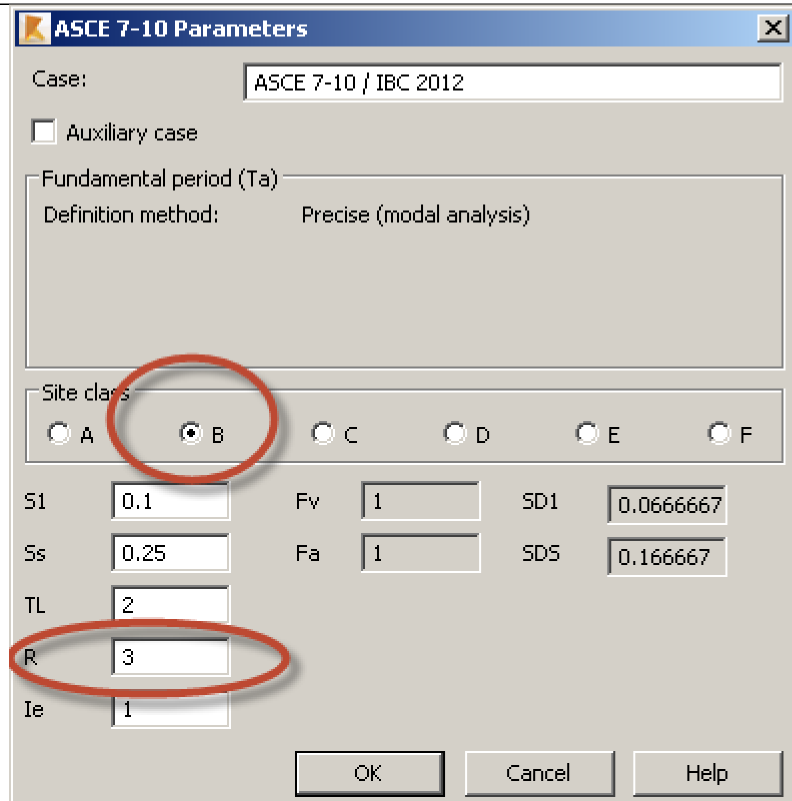


Give your new analysis a name like "Seismic" and select "Seismic (Equivalent Lateral Force Method)" then press "OK"	

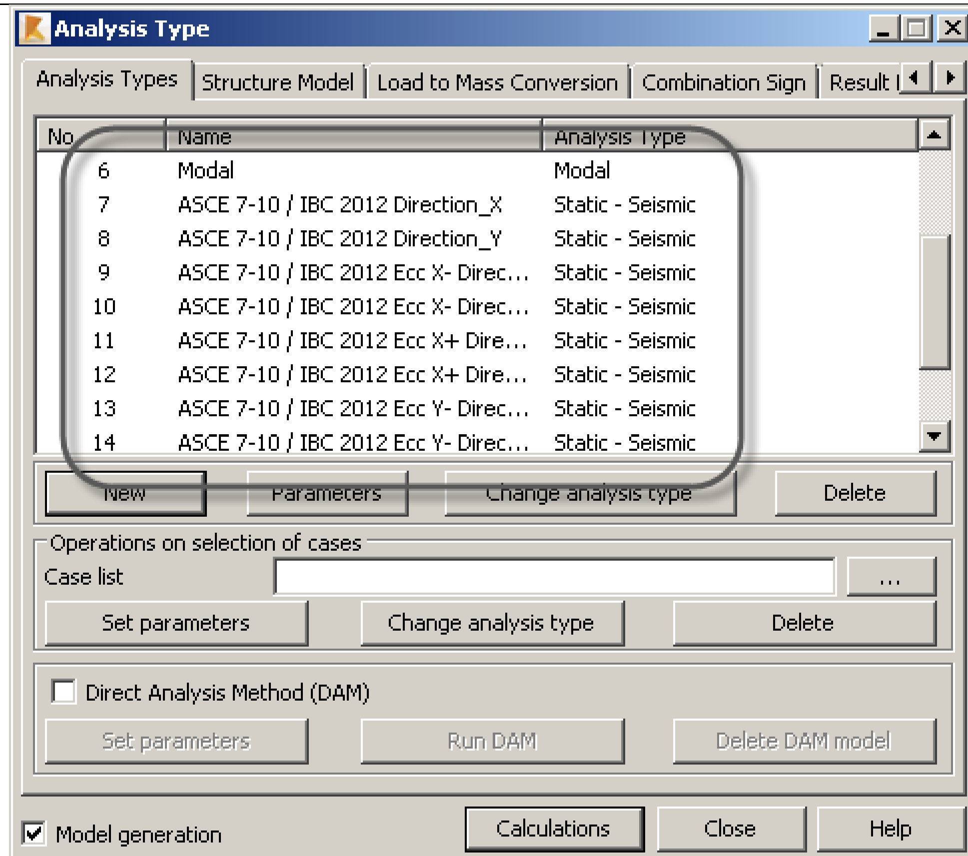
Now you will need to specify the parameters for the seismic loading. On the Seismic Analysis dialog, select all 4 eccentricities, ASCE7-10 for the Seismic analysis according to, and "Precise (Modal analysis) for the "Method of defining values of fundamental period" as shown here



The only other thing we'll need to configure here are the "Seismic Analysis Parameters" press this button to open the parameter settings for ASCE 7-10. We'll configure site class B with an R factor of 3. Press "OK" once you have configured these settings. Then press "OK" in the Seismic analysis dialog.

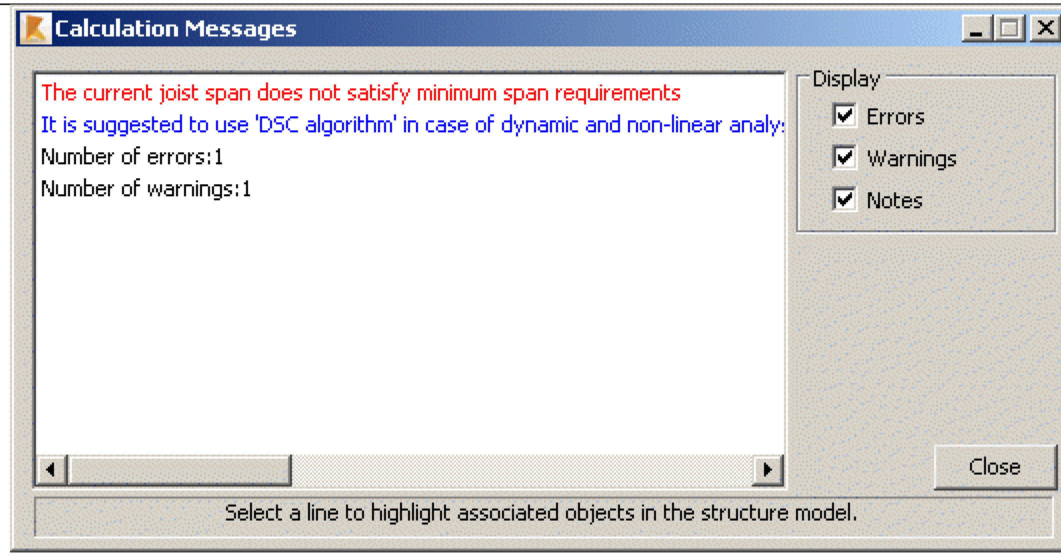


Now your Analysis types dialog will list your new modal analysis as well as the seismic load cases which use the modal analysis as input. Take a quick look at the list of load cases generated for you.



Let's go ahead and run calculations at this point to generate our seismic loads and let us look at the results

We may get a warning about the joist span, and we may get some type 3 instabilities (these are due to the type of deck we selected because the stiffness laterally is very very small and causes a big difference in the overall range of the stiffness matrix. We may also get a suggestion to use DSC algorithm for structures with releases; the joists are released for bending at their ends. This is an algorithm that facilitates dynamic and non-linear analysis of structures with member end releases, you can read more about it in the help files and it it enabled in job settings>structural analysis.)

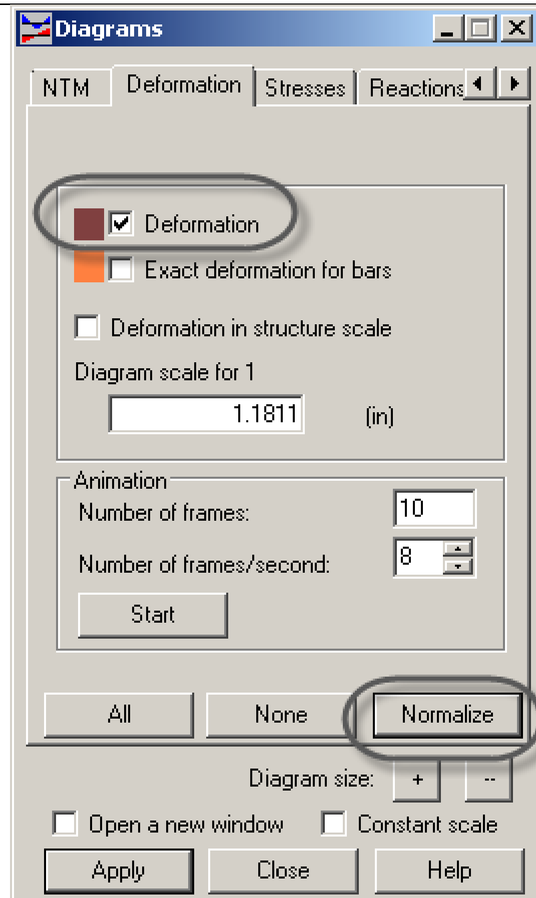


Now that calculations have been run, we can start looking at our seismic loading. We'll start by looking at our modal analysis. Open the modal analysis results from Resutls>Advanced>Modal Analysis.

Here you can view your structure natural frequencies and periods calculated by Robot.

Case/Mode	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Rel.mas.UZ
6/ 1	5.19	0.19	0.05	99.11	
6/ 2	5.27	0.19	99.64	99.17	
6/ 3	8.61	0.12	99.64	99.17	
6/ 4	14.60	0.07	99.69	99.90	
6/ 5	16.06	0.06	100.00	99.98	
6/ 6	22.08	0.05	100.00	99.98	
6/ 7	24.15	0.04	100.00	99.98	
6/ 8	26.00	0.04	100.00	99.98	
6/ 9	27.42	0.04	100.00	99.98	
6/ 10	28.61	0.03	100.00	99.99	

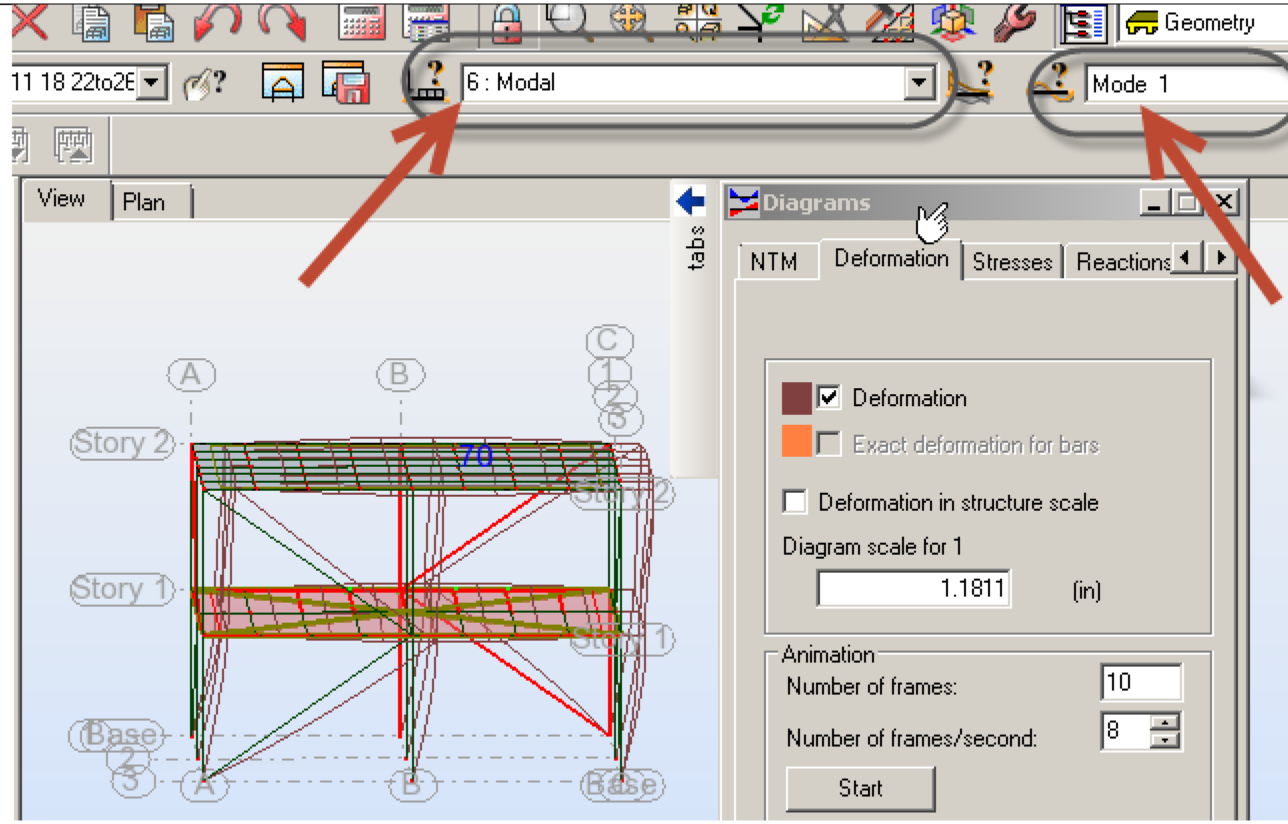
Next we'll look at the mode shapes for our building. Close the modal analysis table and go to Results>Diagrams for bars… which will bring up the Diagrams dialog. Go to the "Deformation" tab and check "deformation" Then press "Normalize"



Next use the load case selector to change to the modal analysis case and then look just to the right for the mode selector and pick mode 1

Now you will see the mode shape as applied to the model.

Take a moment to look through the other mode shapes for the building.



Next we'll look at the loading generated for the seismic cases by starting with the calculation notes.

From the analysis menu choose Analysis>Calculation notes>full note.

Scroll down the report and look for our first seismic case " Case 7: ASCE 7-10 / IBC 2012 Direction_X"

Note the fundamental period selected, the total seismic weight W of 200.81kips and the total base shear force.

You should be able to do some quick verification of the values calculated here.

Case 7 : ASCE 7-10 / IBC 2012 Direction_X

Analysis type: Static - Seismic

Excitation direction:

X = 1.000 Y = 0.000 Z = 0.000

Data:

Soil	:	B
S_1	:	0.100
S_S	:	0.250

Spectrum parameters:

F_a =	1.000	F_v =	1.000
S_{MS} =	0.250	S_{M1} =	0.100
S_{DS} =	0.167	S_{D1} =	0.067
T_o =	0.080	T_S =	0.400
T_L =	2.000		
I =	1.000	R =	3.000

Fundamental period:

Modal analysis T = 0.190 (s)

Structure range:

Top story Story 2

Bottom story Story 1

Effective height H_n = 28.00(ft)

Base shear

C_s = 0.056

$C_{s\ max}$ = 0.117

$C_{s\ min}$ = 0.010

Effective seismic weight W = 200.81(kip)

Shear force V = 11.16(kip)

Vertical distribution of seismic forces

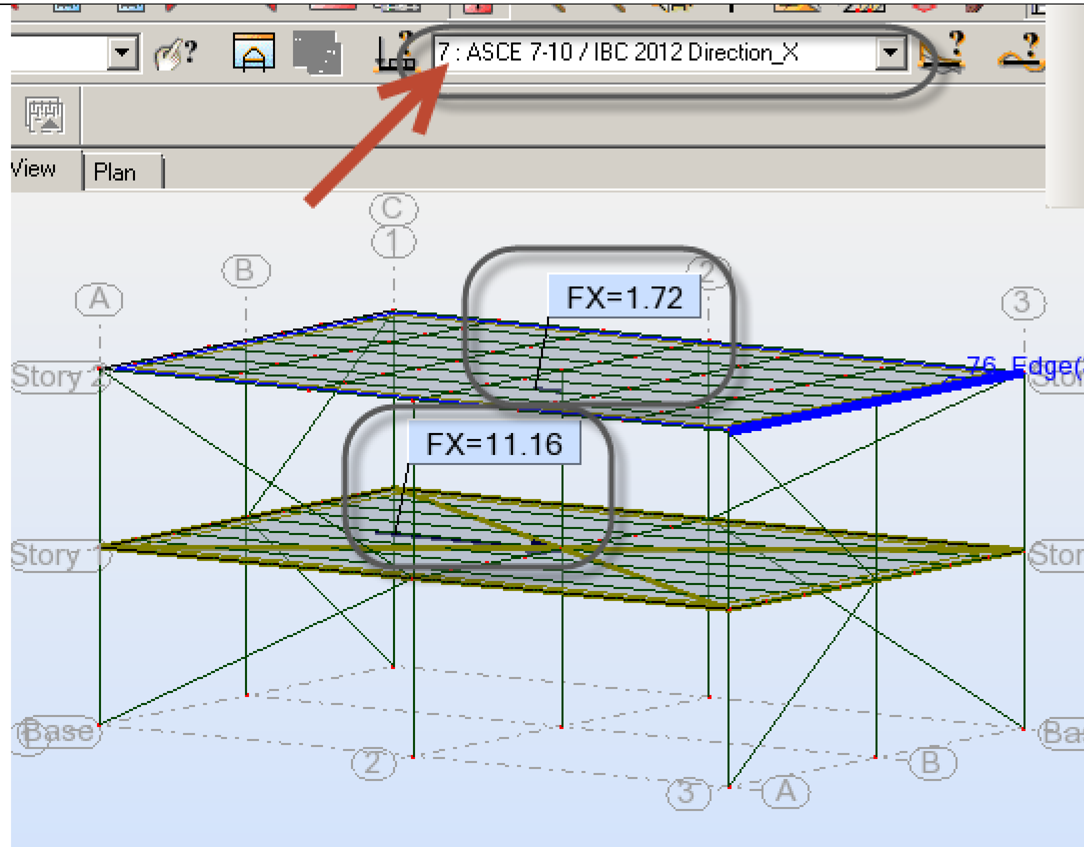
Story	Height (ft)	Weight (kip)	F(kip)	M(kip*ft)
Story 1	14.00	184.05	9.44	0.00
Story 2	14.00	16.76	1.72	0.00

The last thing we'll look at is the reduced forces for the building applied as a result of these seismic cases. Open the Building diagrams from the results menu: Results>Diagrams for Buildings.

In this dialog switch to the "Forces" tab and turn on "Reduced forces in G" for FX, and FY then tick the box by "Descriptions" and then press "Normalize".

Note that you will need to switch to one of the seismic load cases…

(You can also use this tool to look at total lateral load from other lateral cases like wind load.)

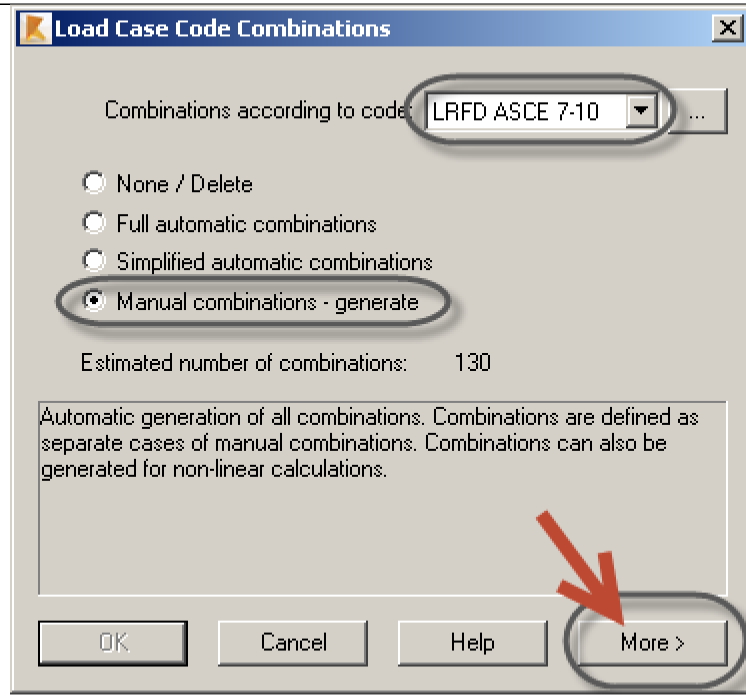


Load Combinations and Calculations

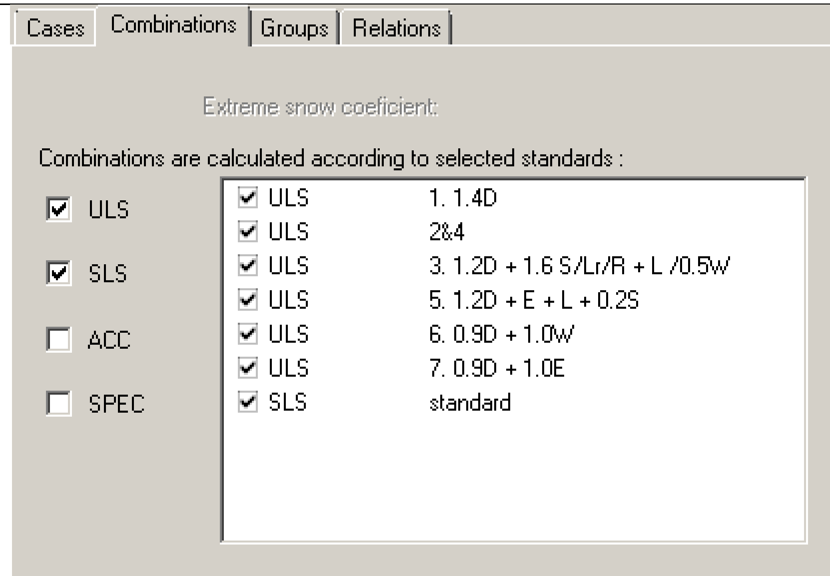
You can continue with your current file, or you can open Robot Dataset 5 - Seismic Load.rtd to catch up.

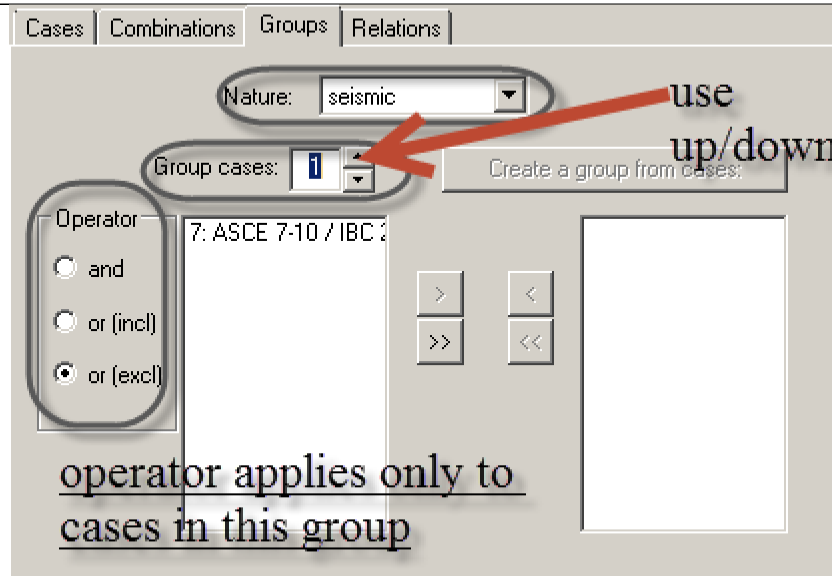
Now we want to generate load combinations to use in analysis. We'll start from the Loads menu Loads>Automatic Combinations…

In the Load Case Combinations Dialog, select "Manual Combinations – generate" then press "More"



Now we will configure the parameters that control which combinations are created. On the first page (Combinations) leave all USL and SLS checked:

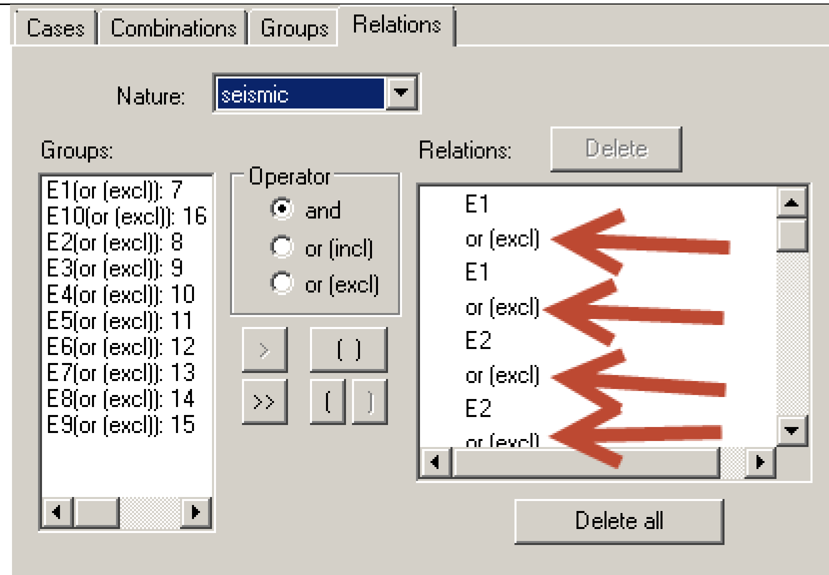


Next, switch to the Groups tab. Take a moment to look through the groups. They are arranged by nature. Note especially the two wind load cases have been grouped with an "Or(excl)" and that there are actually several groups created for the seismic load cases. Take a look at what operator has been applied to each group.	

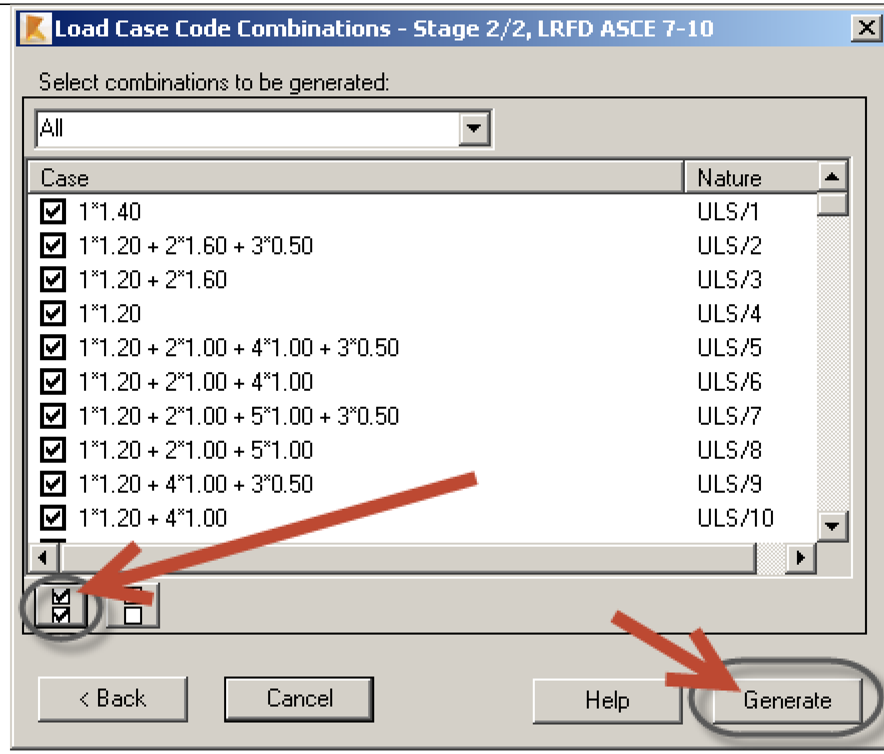
Next, switch to the relations tab and again review the relations which have been set up.

Notice that for seismic, where we had several different groups, each group has been added with a relationship to the other groups. What we have here is that each seismic load case group has been related to every other by an "or(excl)".

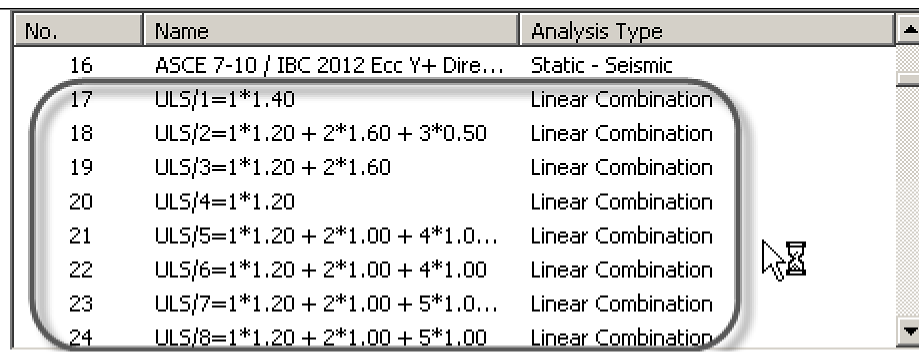
Groups and relations are REQUIRED for generation of load combinations. Make sure they are set up properly.



Press the "Next" button to be taken to the list of combinations that Robot plans to create for you. Check all that you want to use, or use the "check all" button at the bottom then press "generate"



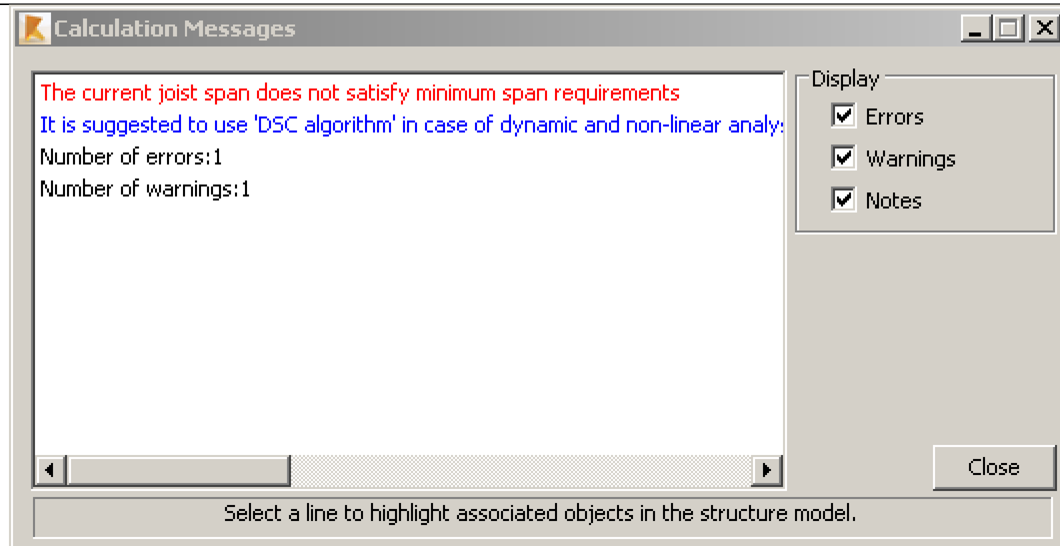
Now Take a look at your Analysis types dialog and you'll see your newly added load combinations right after your load cases:



At this point, we want to run calculations to start looking at results and getting ready for material design.

Press "Calculations" in the Analysis Types Dialog.

Pretty painless and we have a couple of errors/warnings but we'll ignore them for now. In this particular model, DSC algorithm doesn't seem to make any difference in the modal analysis and we know that a 14k1 won't go 30'-0" (no surprise there)



Section 3 - Results

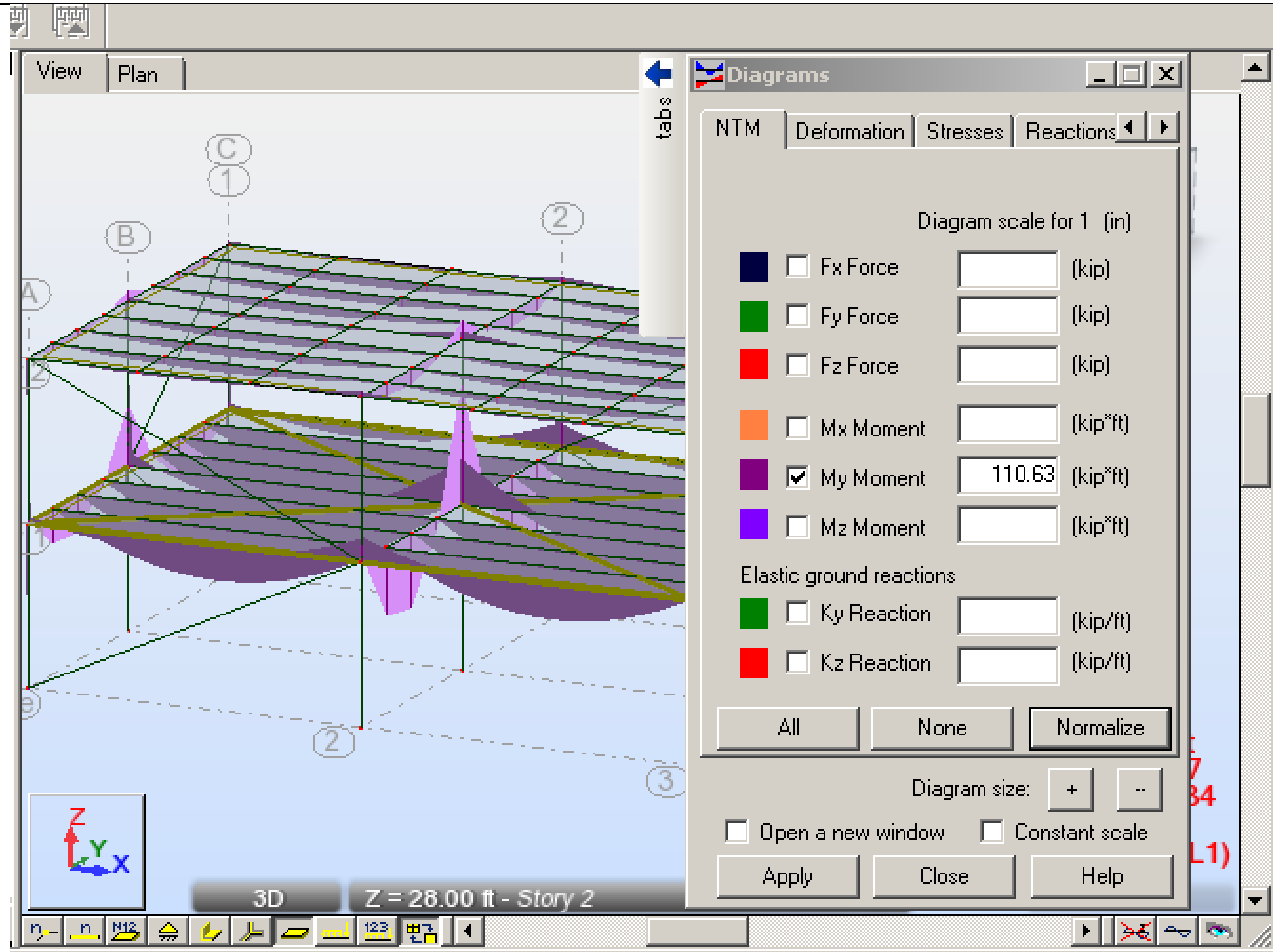
Exploring Results for Bars

You can continue with your current file, or you can open Robot Dataset 6 - Load Combinations.rtd to catch up.

Let's start by looking at results for bars (linear elements). We'll open the diagrams dialog again from Results>Diagrams on Bars…

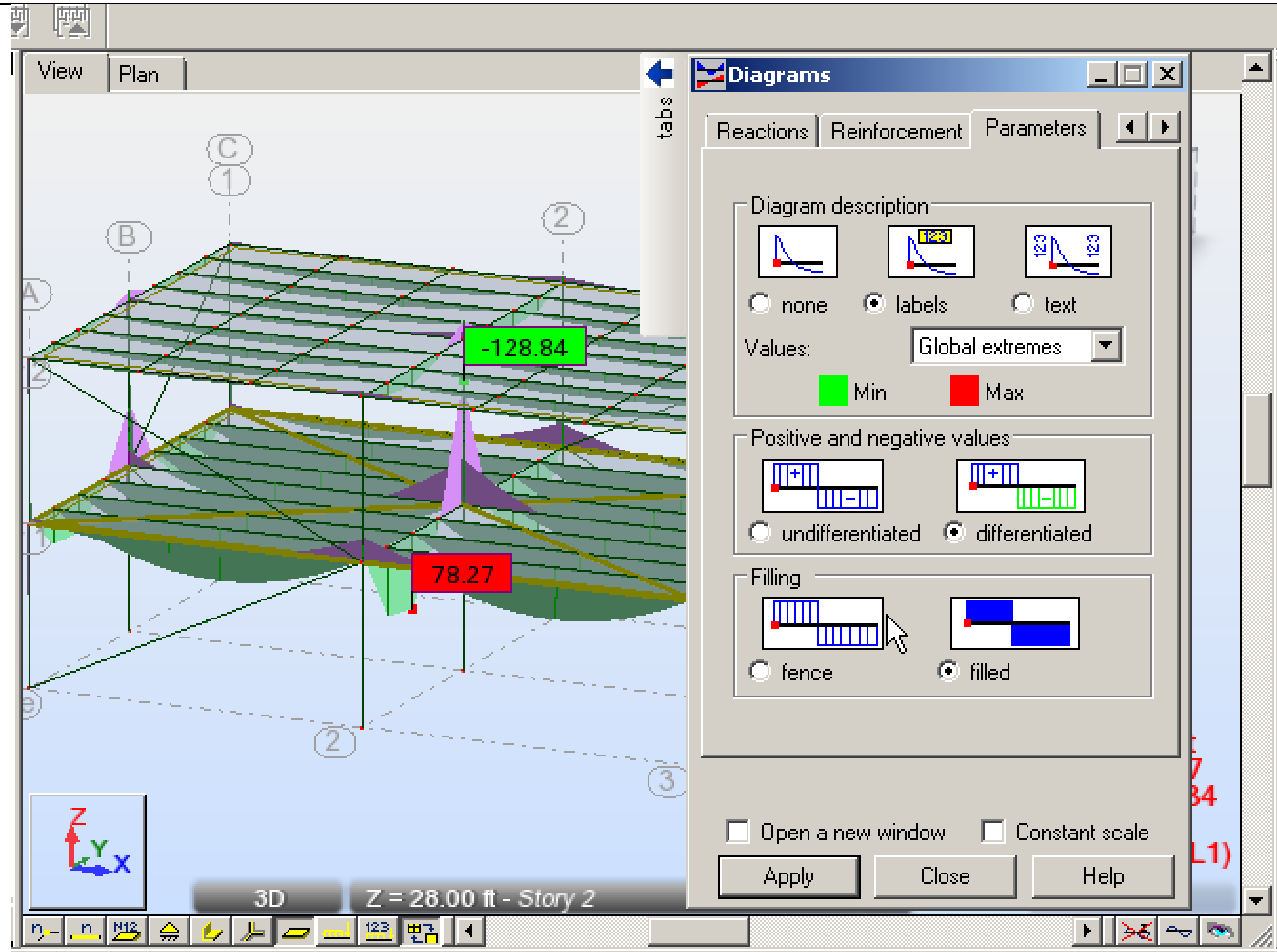
If not selected, switch to the NTM tab and tick My to look at major axis moments on the elements. Press "Normalize":

Use the Load Case selector to look at different load cases/combination or envelopes.



Try some of the options on the "Paramters" tab to adjust the display results to show result magnitudes, or change the display style of the diagrams.

Here we have differentiated positive and negative moment and added labels at the global extremes of the result values.



Next we'll take a quick look at the result tables for reaction, member forces, displacements.

Access each of these from Results>Reactions, Results>Forces, Results>Displacements

Not that results are arranged by Node/Case

Node/Case	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
1/ 1	-0.92	0.20	14.27	0.0	-0.00	0.00
1/ 2	-0.41	0.10	5.42	0.00	0.00	0.00
1/ 3	-0.41	0.01	4.26	0.00	-0.00	-0.00
1/ 4	8.58	-0.00	3.02	-0.00	0.00	0.00
1/ 5	0.22	-0.03	-2.41	0.00	0.00	-0.00
1/ 7	-5.56	-0.00	-2.98	0.00	-0.00	-0.00
1/ 8	-0.00	-0.01	-0.07	0.00	-0.00	-0.00
1/ 9	-5.56	-0.00	-2.98	0.00	-0.00	-0.00
1/ 10	0.26	-0.01	0.07	-0.00	0.00	0.00
1/ 11	-5.56	-0.00	-2.98	0.00	-0.00	-0.00
1/ 12	-0.26	-0.01	-0.20	-0.00	0.0	0.00

Of particular interest might be the Envelope tab at the bottom. This tab will show you summarized results for each node and each force component.

Here we see the max and min values for reaction FX at node 1 and the associated load case/combination which generated this max/min value.

Use the load case selector to remove the simple cases from the results to see only combination effects.

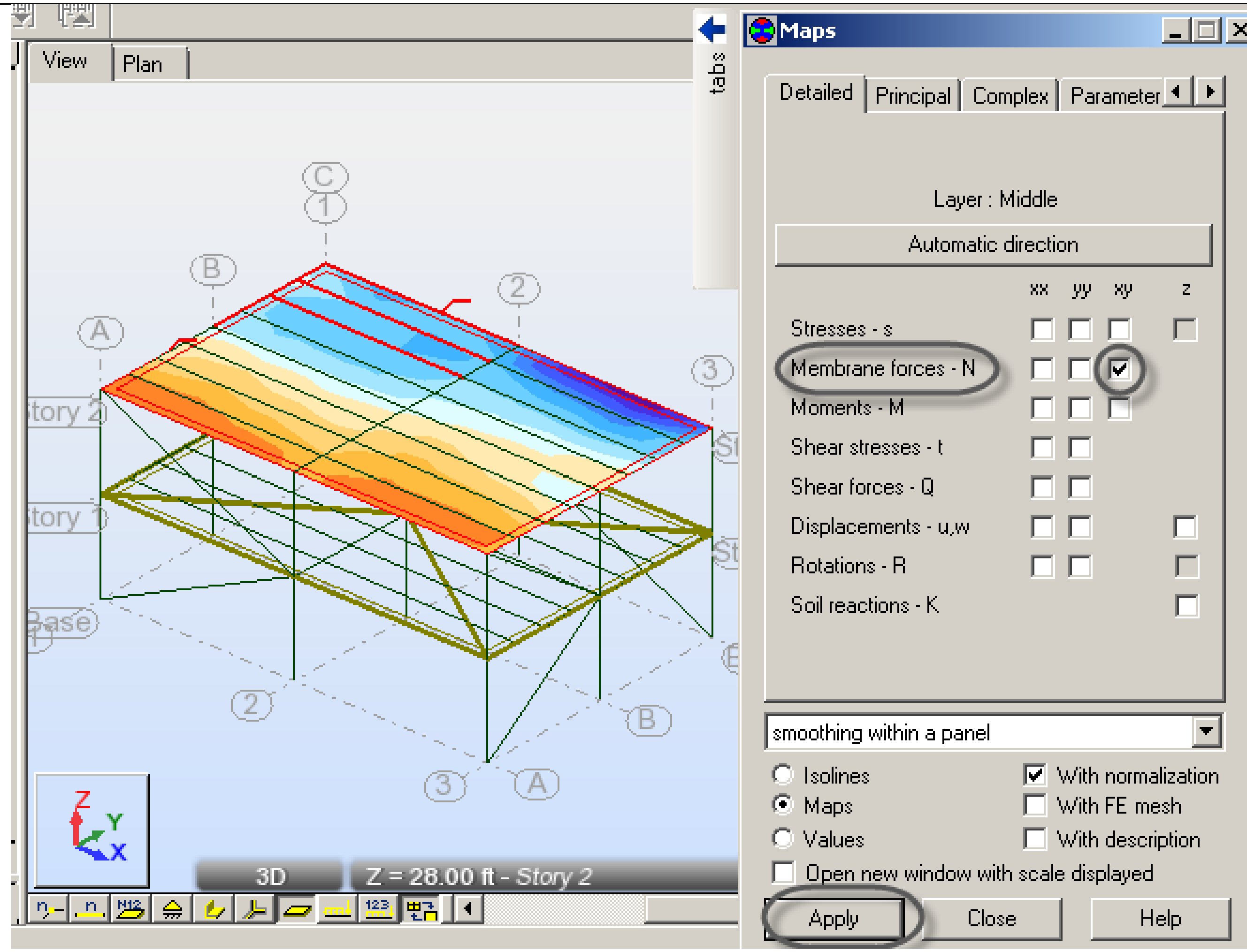
Node/Case	FX (kip)	FY (kip)	FZ (kip)	MX (kip-ft)	MY (kip-ft)	MZ (kip-ft)
1/ 4	**8.58>>**	-0.00	3.02	-0.00	0.00	0.00
1/ 60 (C) (CQC)	**-7.32<<**	0.34	20.32	0.00	-0.00	0.00
1/ 18 (C)	-1.95	**0.40>>**	27.94	0.00	-0.00	0.00
1/ 5	0.22	**-0.03<<**	-2.41	0.00	0.00	-0.00
1/ 29 (C)	-2.16	0.35	**29.37>>**	0.00	-0.00	0.00
1/ 13	-5.73	0.00	**-3.08<<**	0.00	-0.00	-0.00
1/ 40 (C) (CQC)	-1.59	0.33	23.33	**0.00>>**	-0.00	0.00
1/ 83 (C) (CQC)	-1.10	0.25	17.19	**-0.00<<**	-0.00	0.00
1/ 2	-0.41	0.10	5.42	0.00	**0.00>>**	0.00
1/ 62 (C) (CQC)	-6.91	0.24	14.90	0.00	**-0.00<<**	0.00
1/ 19 (C)	-1.75	0.40	25.80	0.00	-0.00	**0.00>>**
1/ 8	-0.00	-0.01	-0.07	0.00	-0.00	**-0.00<<**

Exploring Results on Surfaces

Now we want to take a look at results for surfaces, in particular our roof deck. From the results menu, select Results>Maps… to access the Maps dialog.

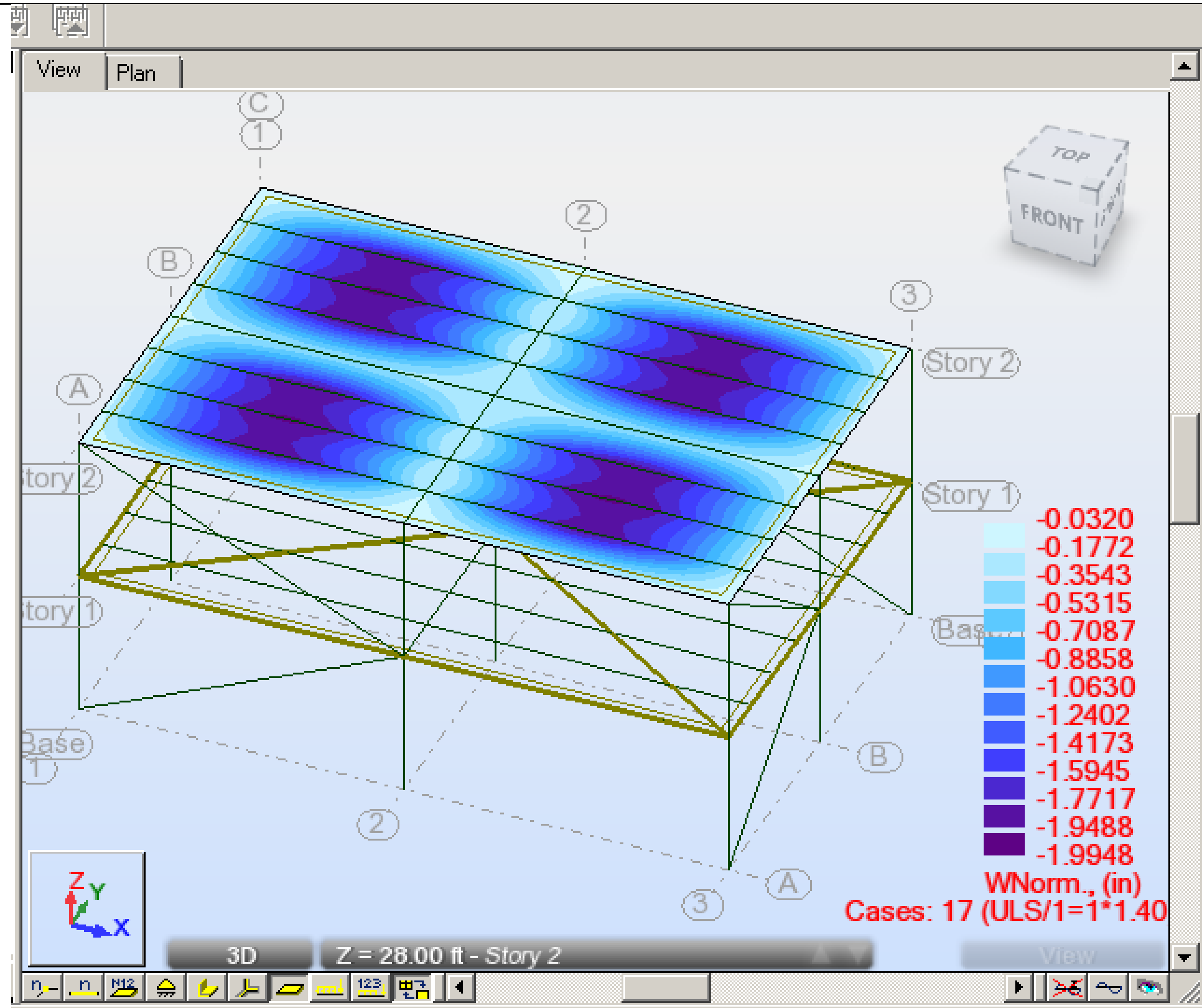
Let's look at the diaphragm shear in our roof diaphragm by ticking xy under Membrane forces – N as shown here and select case W_EW for the load case to display.

Take a few moments to try some of the different options here including the display options at the bottom of the dialog.



You can also get a quick look at the deflection on the surface to see what you're dealing with. Just tick the "z" box under "Displacements – u,w".

The legend on the lower right is helpful in getting your bearings:



Section 4 - Steel Design Workflow

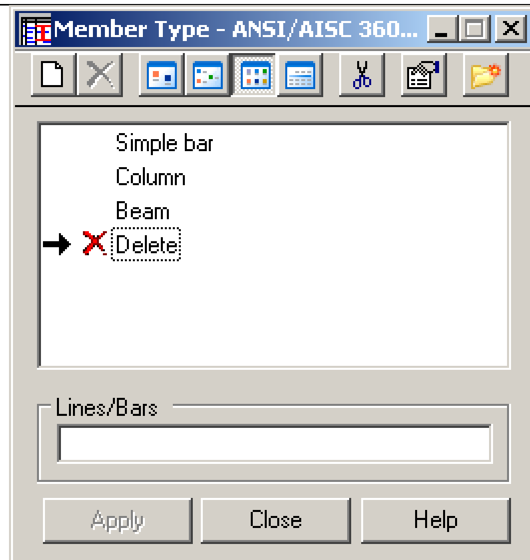
Configuring Member Types

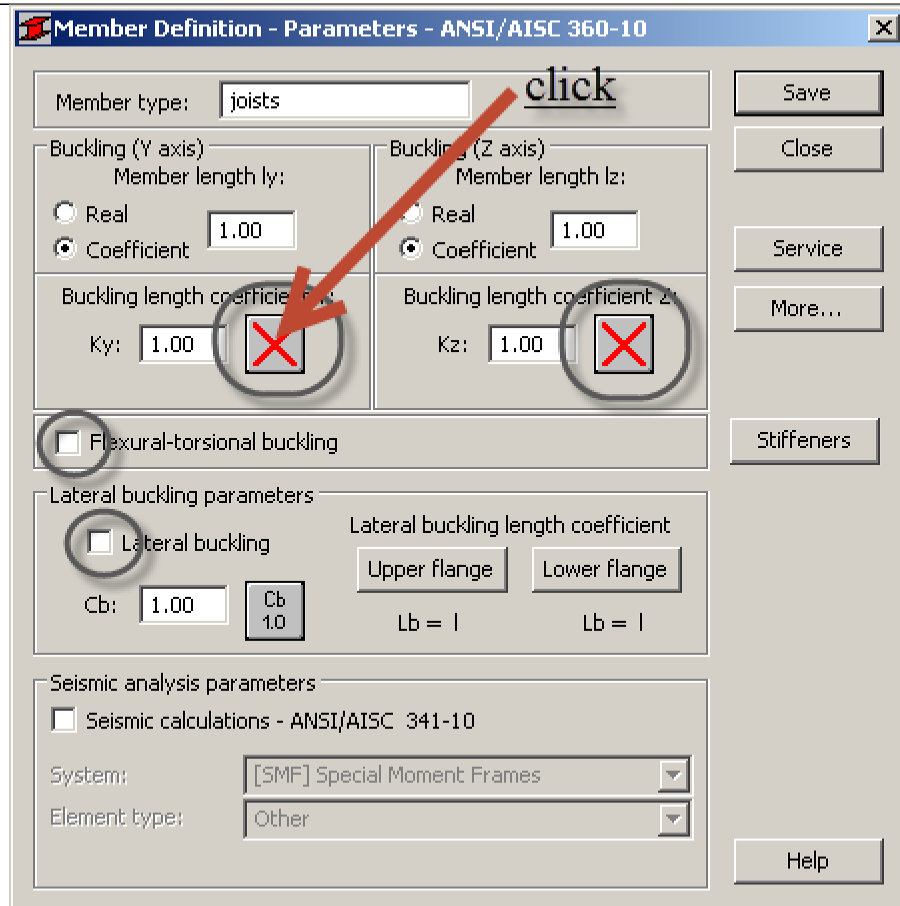
You can continue with your current file, or you can open Robot Dataset 7 - Results Exploration.rtd to catch up.

The first step of steel design is properly configuring the "member types" these are the code (AISC, and other) parameters assigned to the members which will govern the member design.

Open the code parameters dialog from the Design menu: Design>Steel Members Design – Options>Code Parameters…

This will bring up the "Member Type – AISC 360-10" dialog.

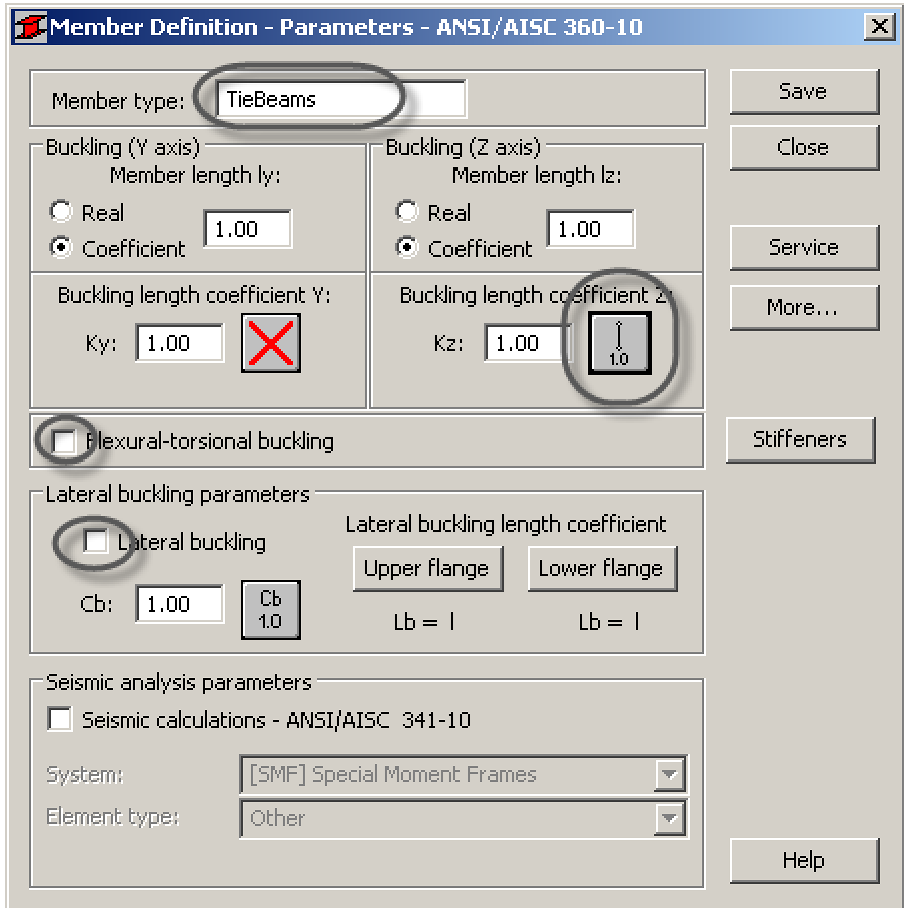


Let's add some types to represent our members. Start by clicking on the "new steel member type definition" button to bring up the member definition dialog. Member type will be Joist for our joists. Make other settings as shown here and then save it.	

Next we'll create the member type for our tie-beams which we assume will be fully laterally supported by the deck.

Member Type TieBeams:

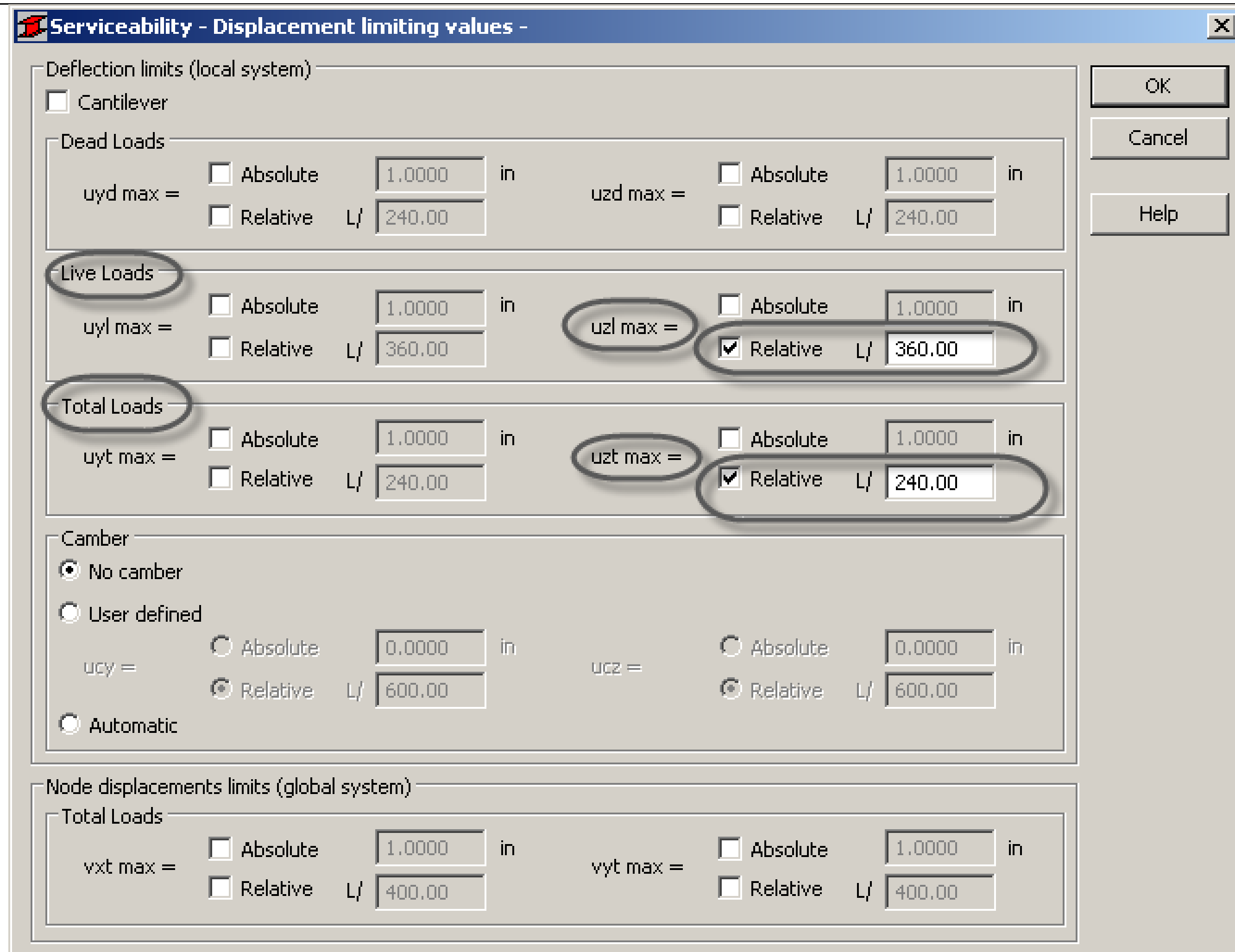
Buckling length coefficient for z axis will be 1.0, leave flexural torsional and lateral buckling alone as we do not expect either of these failure modes.

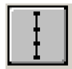


While we're still on TieBeams, let's also configure the deflection limits for this member. Press "Service" on the right hand side to access the "Serviceability – Displacement limiting values" dialog:

For live loads, check relative under uzl max with 360 and relative under uzt max and enter 240 as shown:

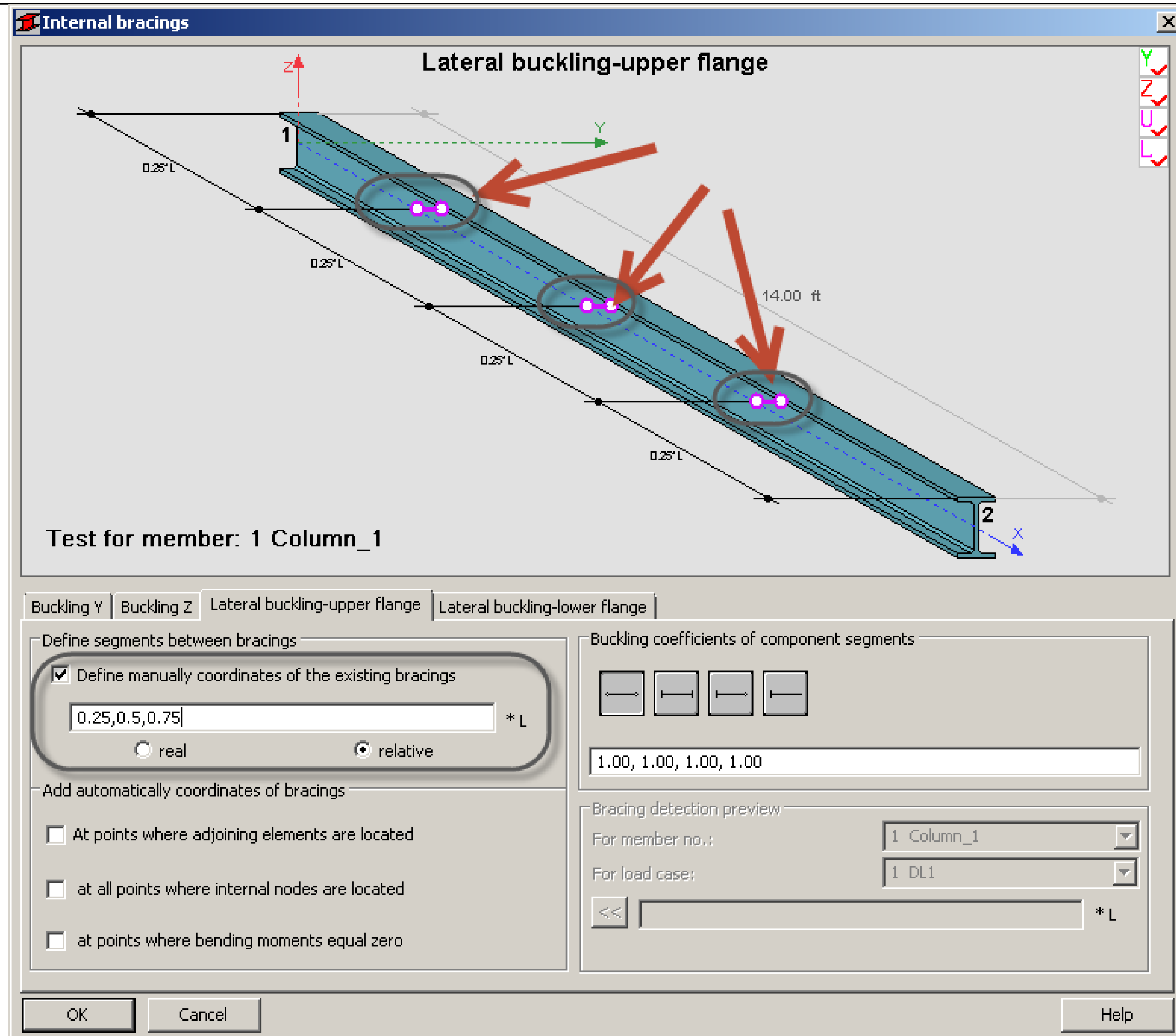
Then press "OK" and "Save" this TieBeam definition.



Next we'll configure the girders which we expect to only be laterally supported by the adjoining members, not the roof deck. Create a new member type called "Girders" and for the top flange press the  button to access the intermediate bracings dialog:

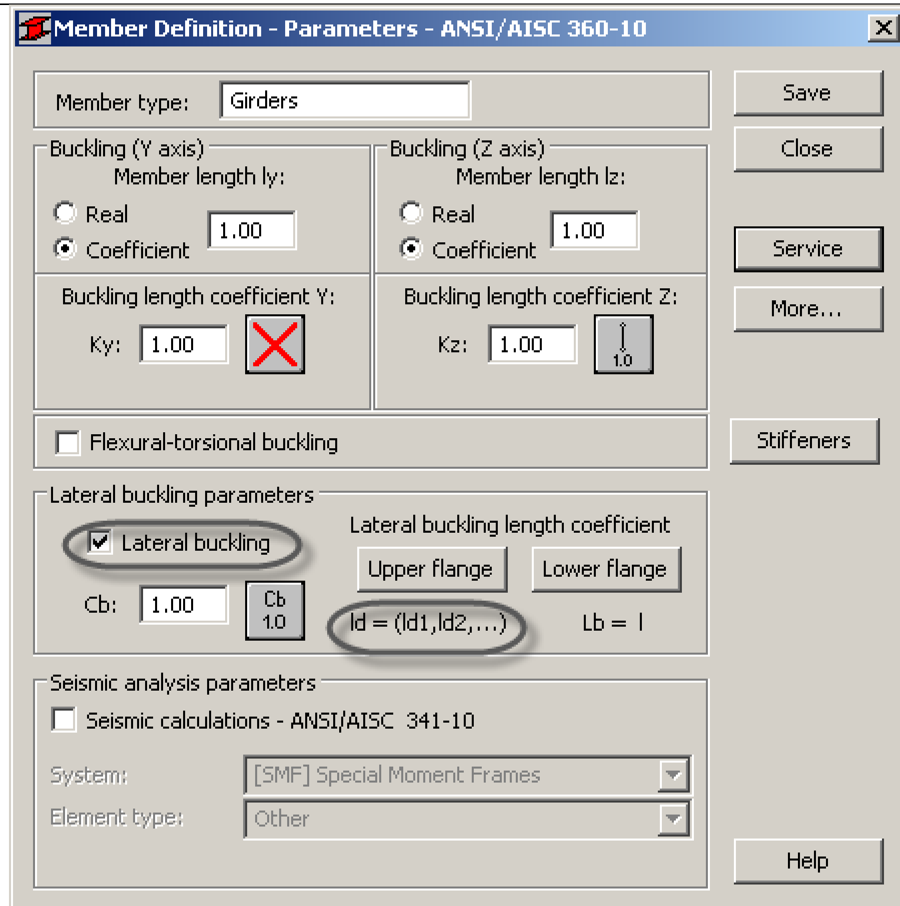
Tick the box next to "Define manually coordinates of existing bracings" and enter "0.25, 0.5, 0.75" and make sure "relative" is checked. This will define lateral bracing at the top flange at the quarter points.

The dialog graphic indicates locations of bracing to give you visual feedback about the locations of actual bracing.

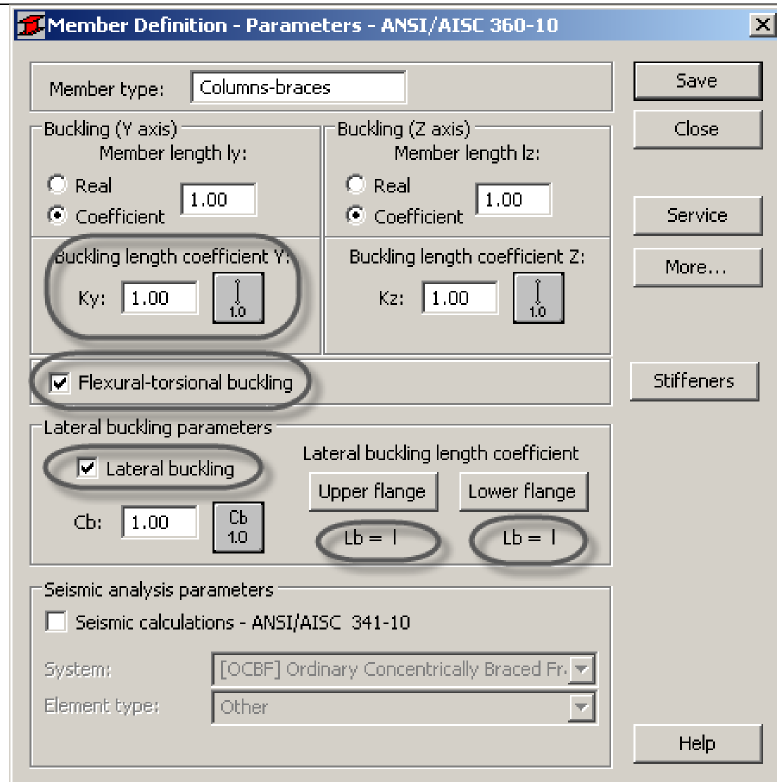


Don't forget to tick the box by "Lateral Buckling" so that Robot will consider lateral flange buckling in the member design checks. Your Girders should look like this:

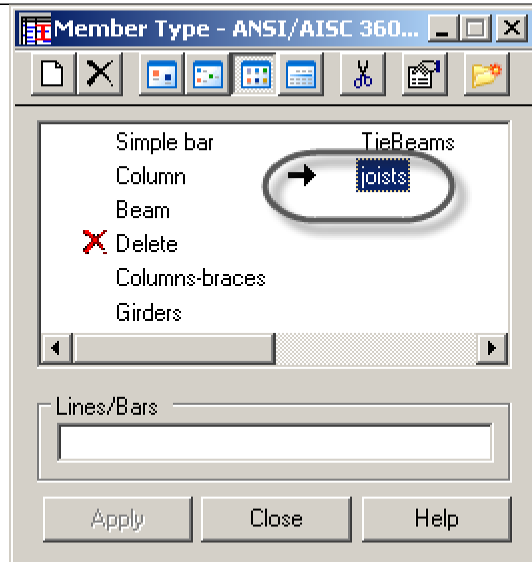
Note, unless you started with a new member, your service criteria will be the same as your tie-beams. Give it a quick check to see.



Now we'll configure columns and bracings using a buckling length coefficient of 1.0 and no lateral bracing along the length of the members.

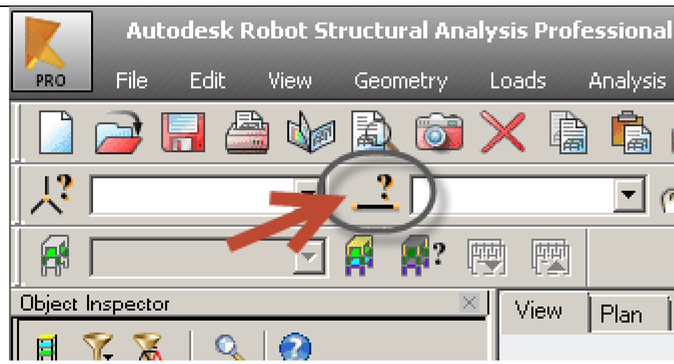


Close the Member definition dialog and we'll go ahead and apply these labels to our members. In the Member Type dialog, select joists and then click in the "Lines/Bars" field.



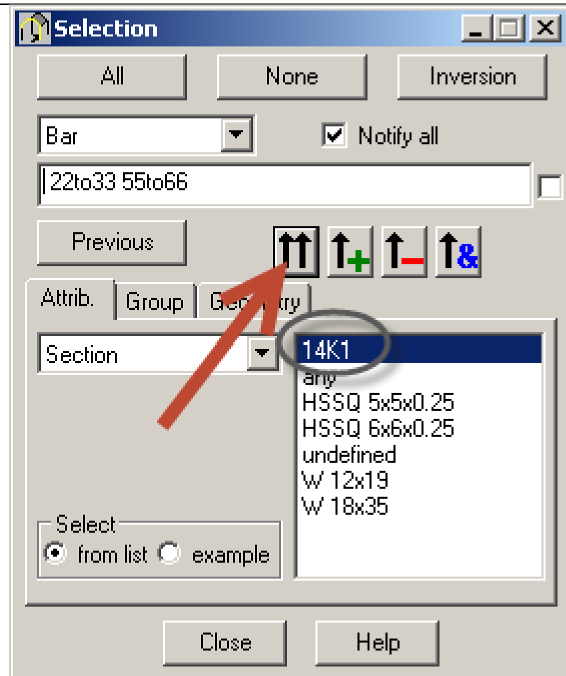
We now need to build a selection of members to which we will apply this member type label.

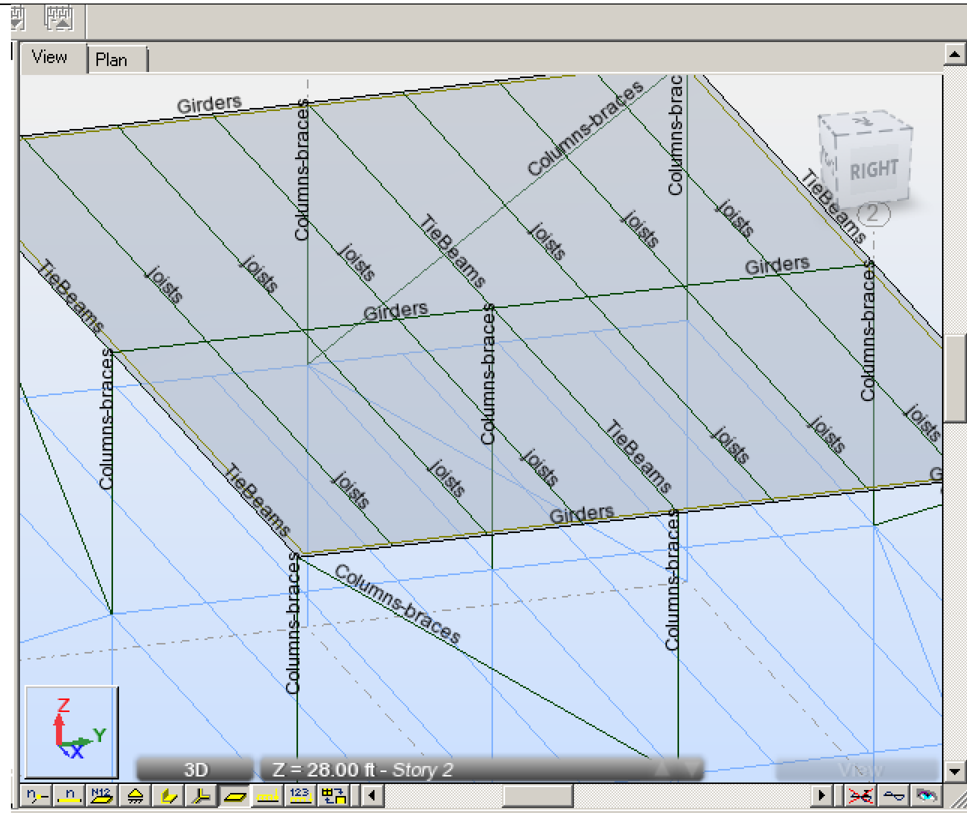
First click on the bar selection button on the selection toolbar:



In the Selection Dialog we can select the 14k1 attribute and press the double up arrow to add all 14k1s to the selection.

Copy this selection to the clipboard with Ctrl+C after selecting the selection set (in this example, "22to33 55to66"



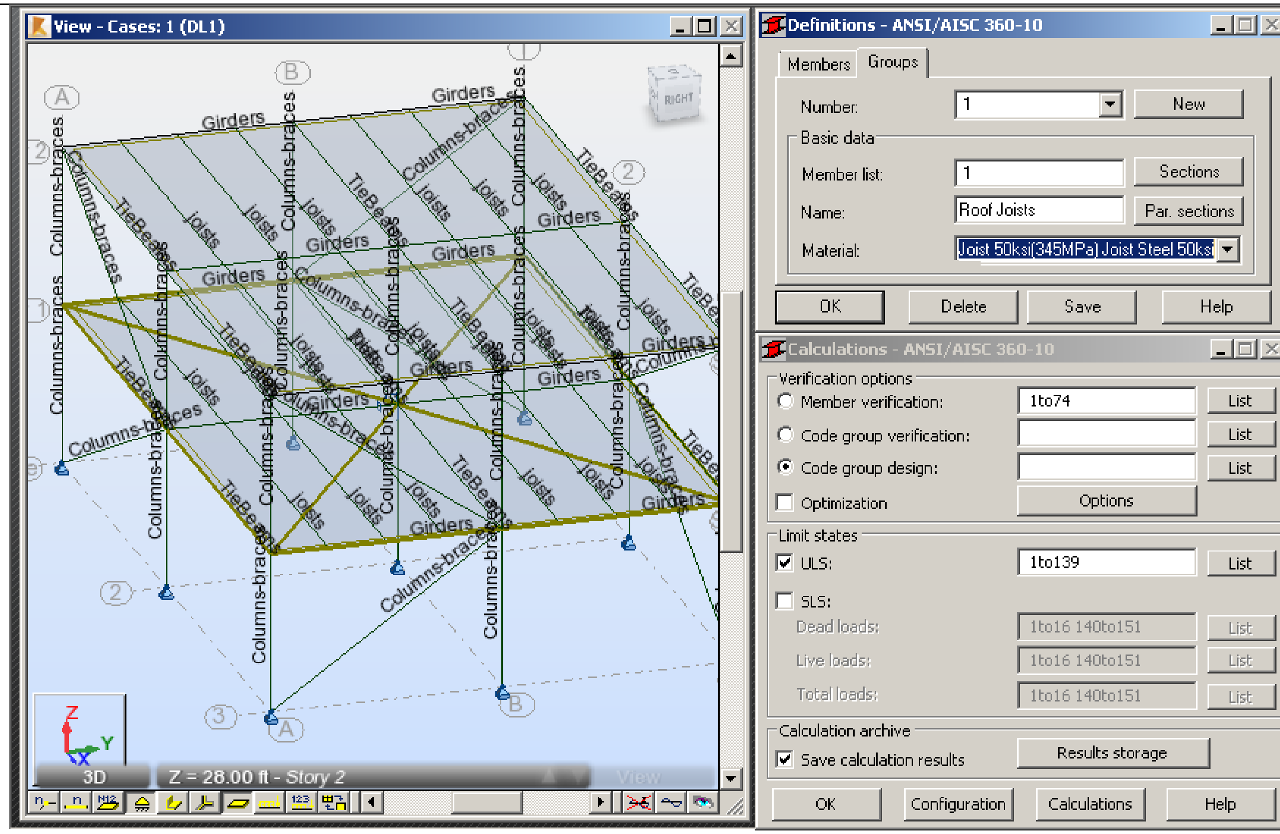
Paste this into the member type label dialog in the "Lines/Bars" field. (you may find that it automatically populates) Then press "Apply" Repeat this process for the girders (W18x35s), the tie-beams (W12x19s), and the columns (HSSQ5x5x0.25 and HSSQ6x6x0.25) then use View>Display>Bar descriptions to show the member types:	

Configure Design Groups

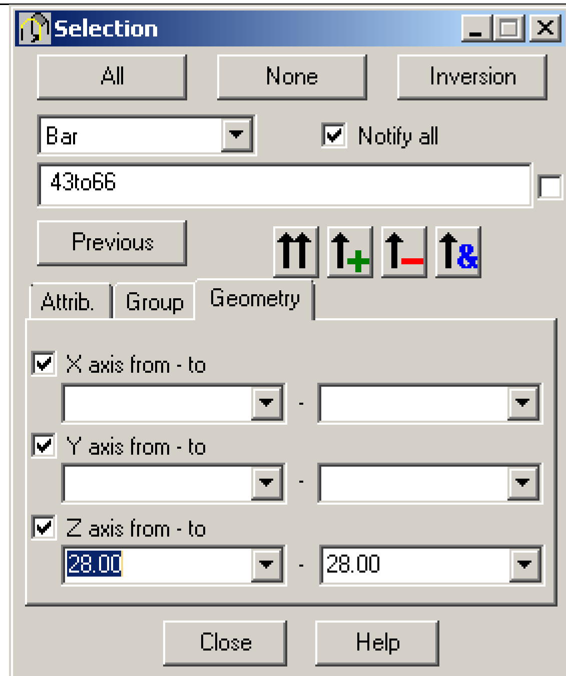
Now we need to configure design groups to control the member design. Open the steel design layout from the design menu: Design>Steel Members Design…

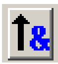
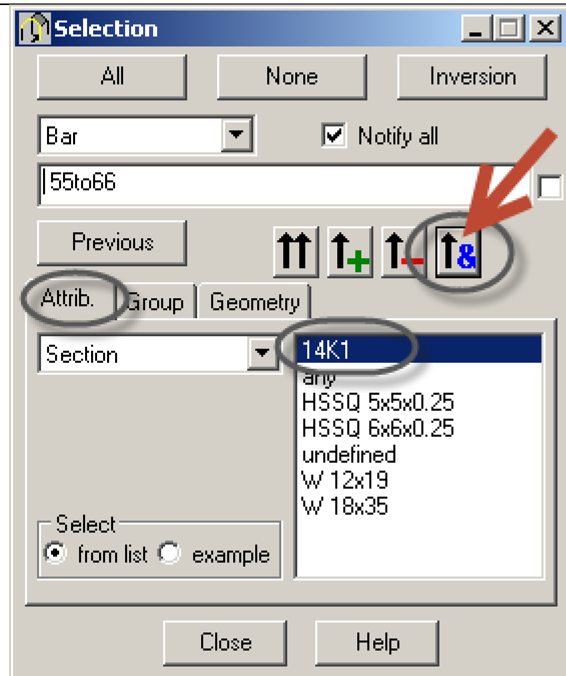
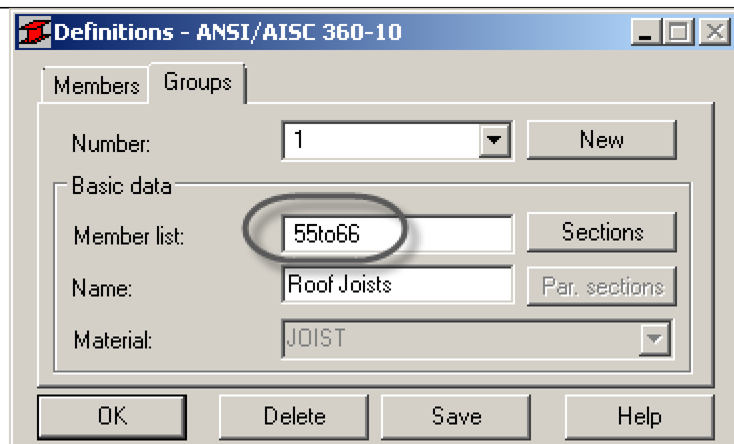
Click on the "groups" tab in the "Definitions" dialog:

Press the "new" button, enter "Roof Joists" for the Name.



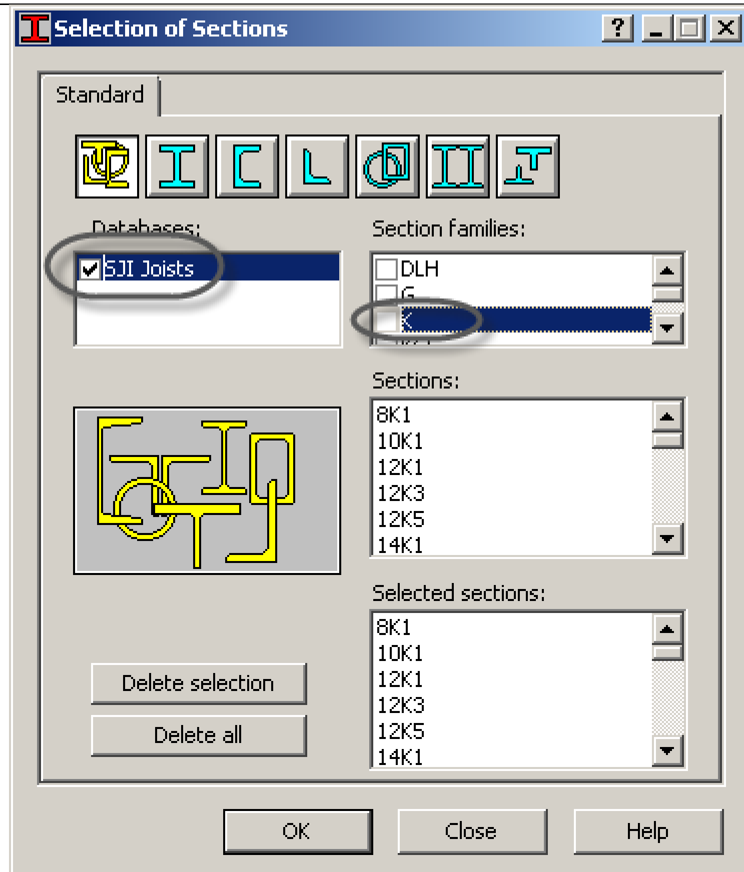
Next we'll configure a selection of roof joists only by combining two options in the bar selection dialog. Launch the bar selection dialog and first switch to the "Geometry" tab, select "Z axis from-to" and select "Structure Axis Story 2 – 28(ft)" for each dropdown then press the double up arrow:

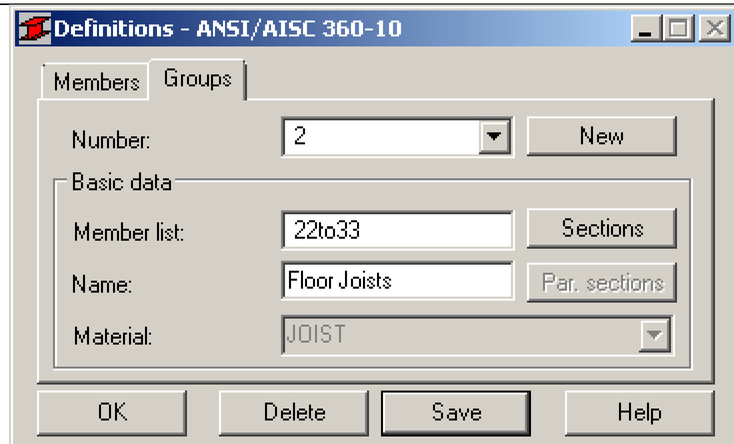


Next we'll pull only items which match the section type we're looking for. Switch to the "Attrib." tab and select 14k1 but this time press the "And" button: 	
Copy paste this selection ("55to66" in my example) to the "Member list" field of the groups definition.	

Next, Press the "Sections" button to configure allowable trial sections for design. Tick "SJI Joists" in the "Databases" field and then click "K" in the "Section Families" list to populate the "Selected sections" list as shown:

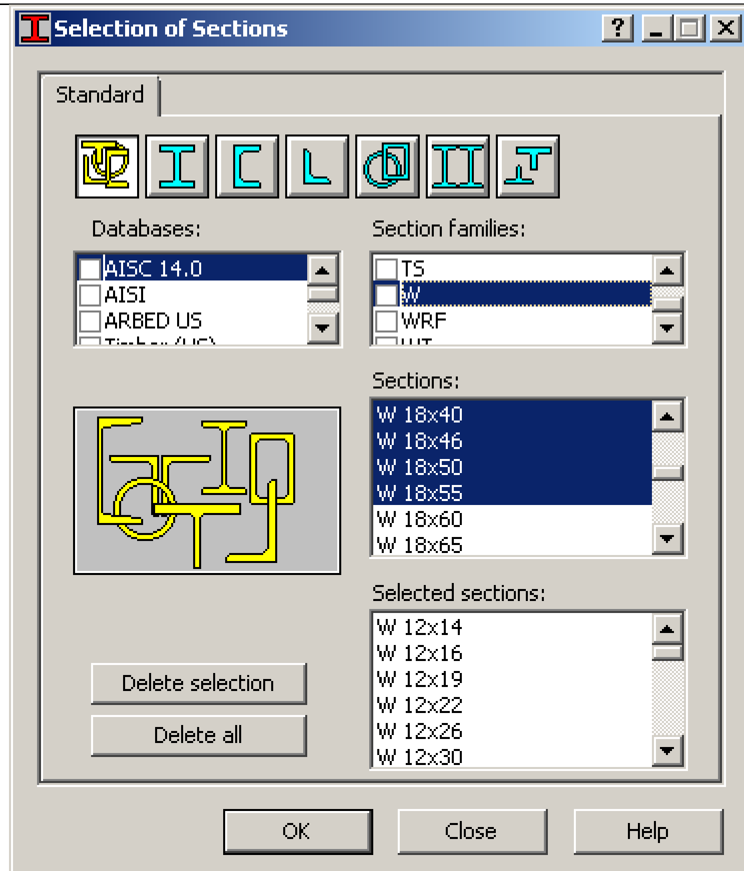
Next, click "OK" and then "Save" your first design group.



Repeat these steps to create a design group for the 2nd floor joists called "Floor Joists" which will also use the "K" joists as available sections for design.	

We'll create another group for roof girders (currently modeled as W18x35s). This time you will need to pick the design material (we'll use A992-50 steel).

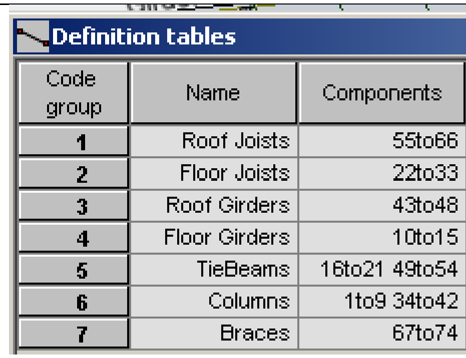
We can limit the number of sections available by opening the "Sections" dialog and clicking on the AISC database (do not tick the box), then click the "W" in the list of Section families (do not tick the box) Next, select individual sections and as you click or select them (ctrl, shift, and/or ctrl+shift type selections are allowed) they will be added to the "Selected Sections" list. Try W12s through W18s by selecting the first W12 and the then holding down shift while clicking W18x55.



Continue creating groups for Floor girders, one for tie beams, one for columns, and one for braces.

Name	Floor Girders	TieBeams	Columns	Braces
Material	A992 - 50	A992 - 50	A500-46	A500-46
Sections	AISC W sections (all)	AISC W sections (all)	AISC HSSQ section (all)	AISC HSSQ section (all)

Use the members/groups table to take a look at the groups you've created: Design>Steel Members Design – Options>Table of Members/Groups



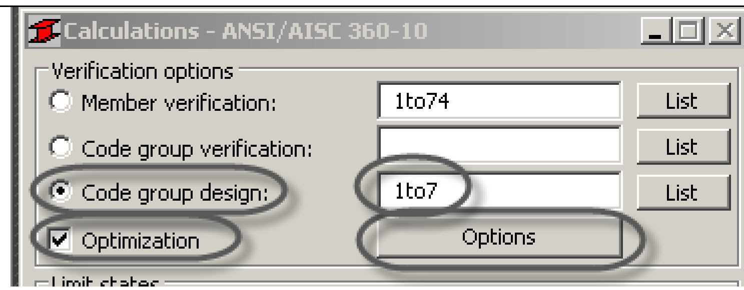
Definition tables

Code group	Name	Components
1	Roof Joists	55to66
2	Floor Joists	22to33
3	Roof Girders	43to48
4	Floor Girders	10to15
5	TieBeams	16to21 49to54
6	Columns	1to9 34to42
7	Braces	67to74

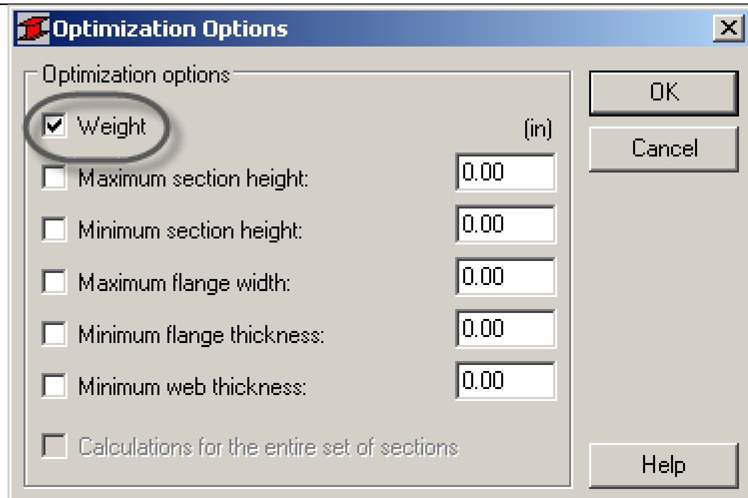
Group Design and Check

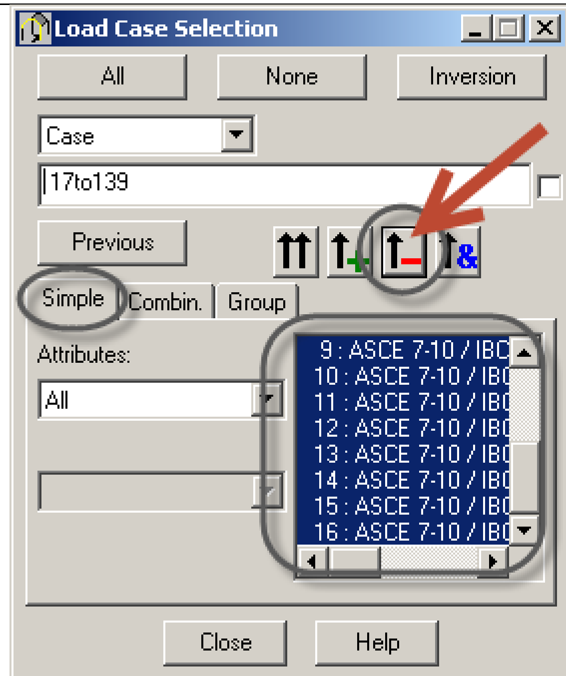
You can continue with your current file, or you can open Robot Dataset 8 - Steel Design - Setup.rtd to catch up.

Now we turn our attention to the Calculations dialog where we will want to select "Code group design". Use the "List" button to the right to add all of our design groups (there should be 7 of them).



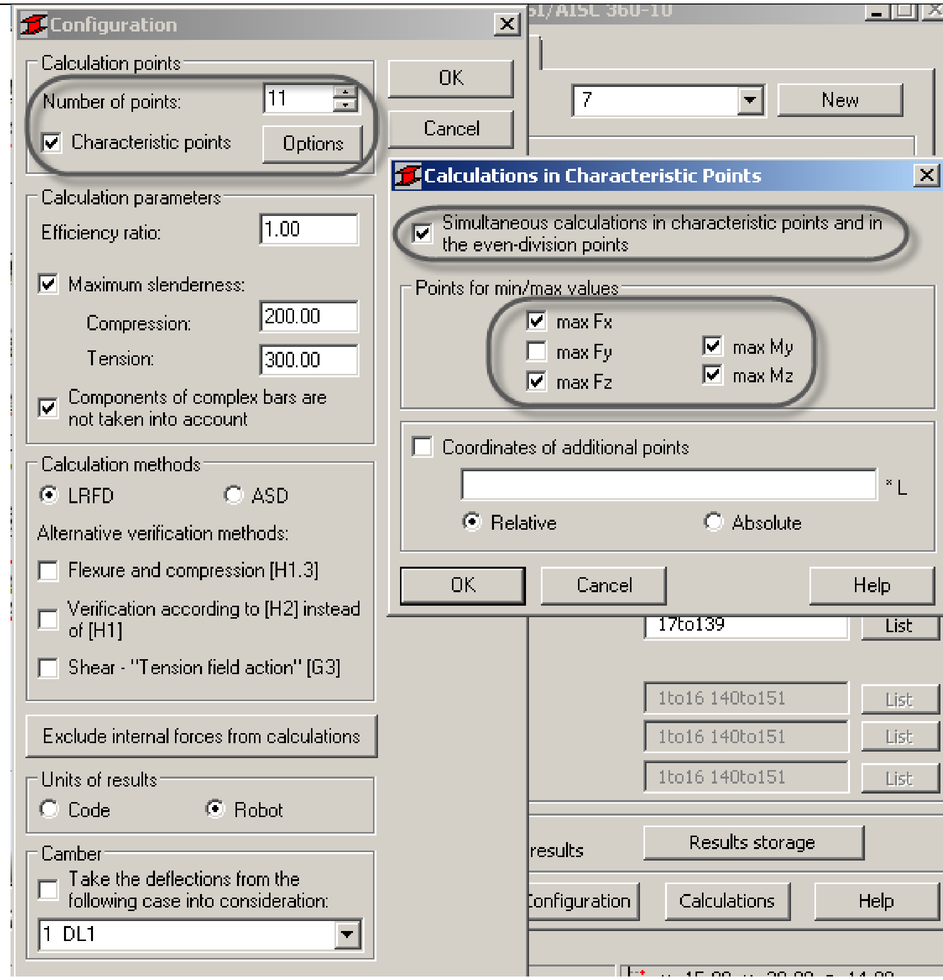
Next, tick "Optimization" and press the "Options" button then tick "Weight"



In the lower half of the Calculations dialog tick “ULS” for limit states and pick your ULS load combinations if they have not already been selected. You can use the List button to clear out basic load cases by selecting all the basic cases and using the subtraction button 	

Next, press the "Configuration" button to adjust the calculation points for the members.

Change number of points to 11 and tick "characteristic points" then press the "Options" button to allow you to check maximum forces.



Then close these dialogs and press "Calculations". Robot will begin checking each member and also attempting to optimize by weight for each code group. The results will look like this:

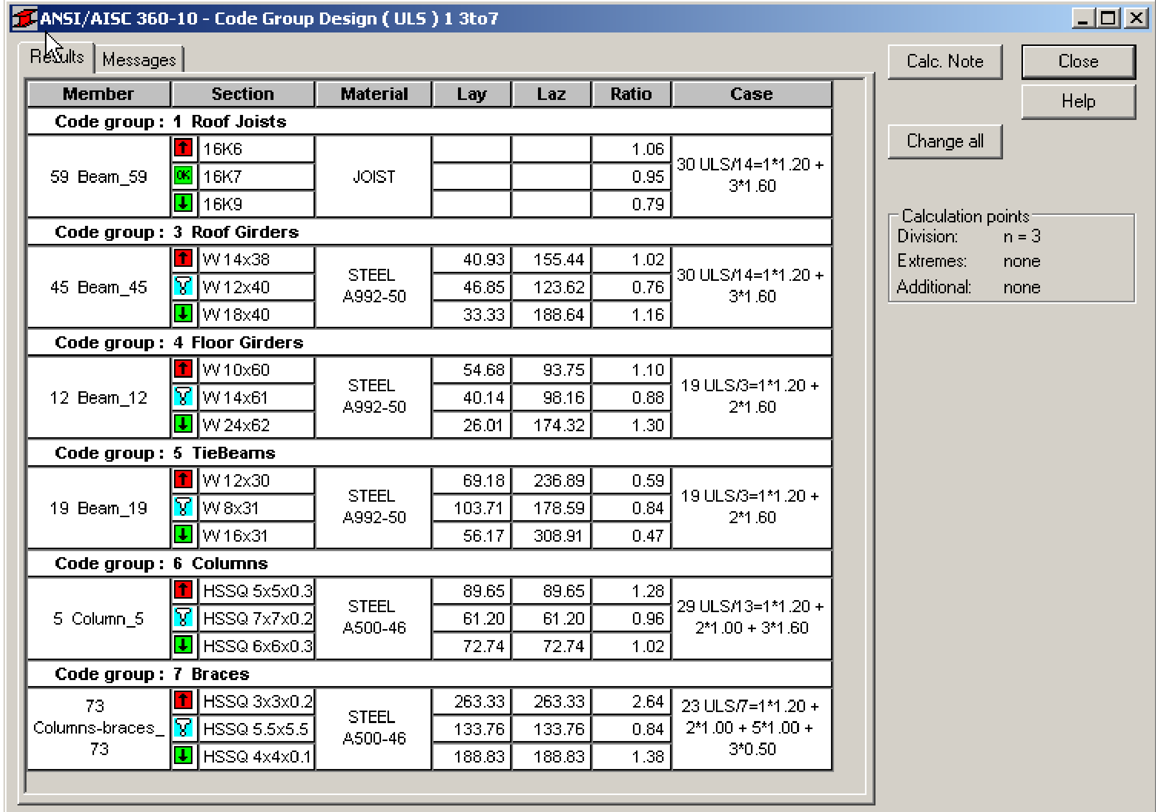
Notice that each group has a controlling member, and lists the trial sections around the optimal section.

: previous failed section

: Optimal section

: Next section after optimal section

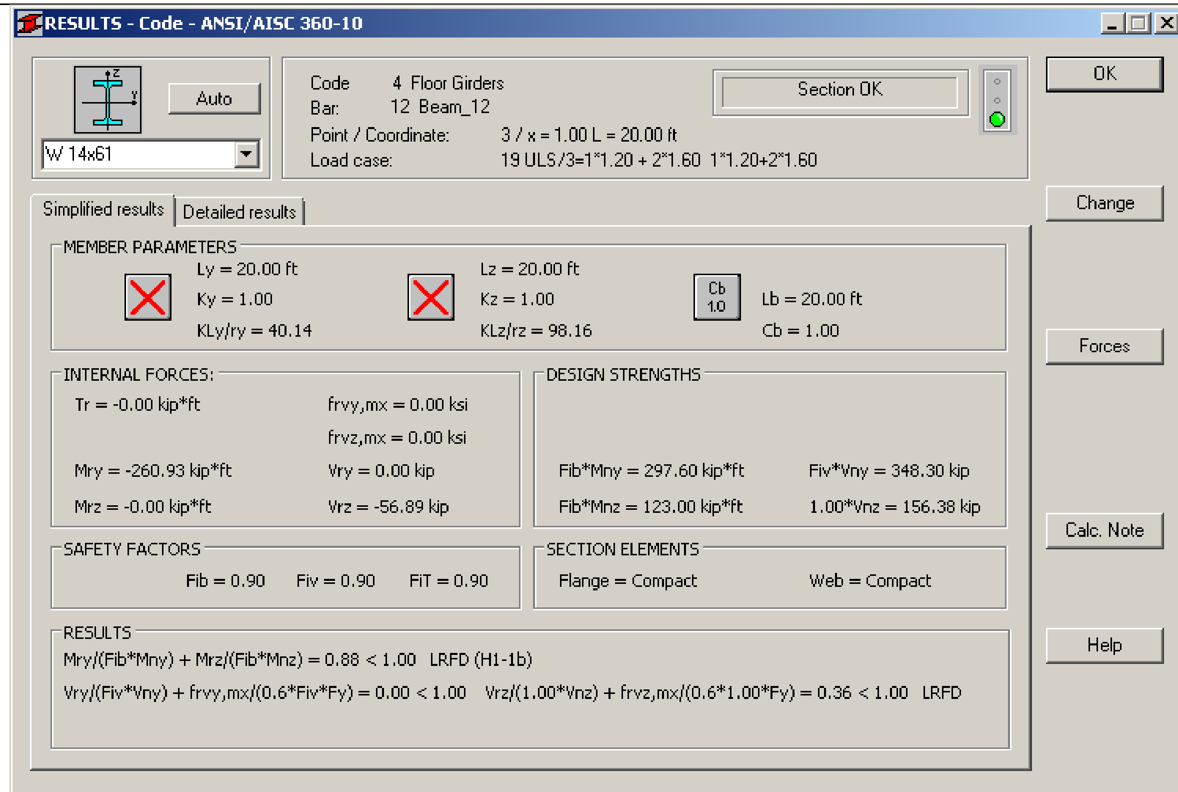
NOTE: Next section may not be acceptable. Check the Ratio to be sure if you want to use it.



Member	Section	Material	Lay	Laz	Ratio	Case
Code group : 1 Roof Joists						
59 Beam_59	16K6	JOIST			1.06	30 ULS/14=1*1.20 + 3*1.60
	16K7				0.95	
	16K9				0.79	
Code group : 3 Roof Girders						
45 Beam_45	W 14x38	STEEL A992-50	40.93	155.44	1.02	30 ULS/14=1*1.20 + 3*1.60
	W 12x40		46.85	123.62	0.76	
	W 18x40		33.33	188.64	1.16	
Code group : 4 Floor Girders						
12 Beam_12	W 10x60	STEEL A992-50	54.68	93.75	1.10	19 ULS/3=1*1.20 + 2*1.60
	W 14x61		40.14	98.16	0.88	
	W 24x62		26.01	174.32	1.30	
Code group : 5 TieBeams						
19 Beam_19	W 12x30	STEEL A992-50	69.18	236.89	0.59	19 ULS/3=1*1.20 + 2*1.60
	W 8x31		103.71	178.59	0.84	
	W 16x31		56.17	308.91	0.47	
Code group : 6 Columns						
5 Column_5	HSSQ 5x5x0.3	STEEL A500-46	89.65	89.65	1.28	29 ULS/13=1*1.20 + 2*1.00 + 3*1.60
	HSSQ 7x7x0.2		61.20	61.20	0.96	
	HSSQ 6x6x0.3		72.74	72.74	1.02	
Code group : 7 Braces						
73 Columns-braces_73	HSSQ 3x3x0.2	STEEL A500-46	263.33	263.33	2.64	23 ULS/7=1*1.20 + 2*1.00 + 5*1.00 + 3*0.50
	HSSQ 5.5x5.5		133.76	133.76	0.84	
	HSSQ 4x4x0.1		188.83	188.83	1.38	

You may not particularly like the sections that Robot has selected. The way to control this is to go back to your groups definitions and refine the list of choices. For instance, Robot has picked a 14x61 as the optimal section for group 4 floor girders… I know I would rather use a 16 or an 18 as 14x61 is not a common beam size in the US.

Clicking on any row will give you access to the detailed results for this calculation:



You can see details on the forces for design in the "Forces" button, review the calculation notes for the group in "Calc. Note" and replace all members of the group with the optimal section by using the "Change" button.

Now we can rerun-calculations and then do a member verification for ULS and SLS to check where we are on service criteria.

Select the "Member Verification" option in the calculations dialog and then also check ULS and SLS in the Limit States.

Use the "List" buttons to select service load cases for dead, live, and total respectively then run the calculations.

ANSI/AISC 360-10 - Member Verification (SLS ; ULS)

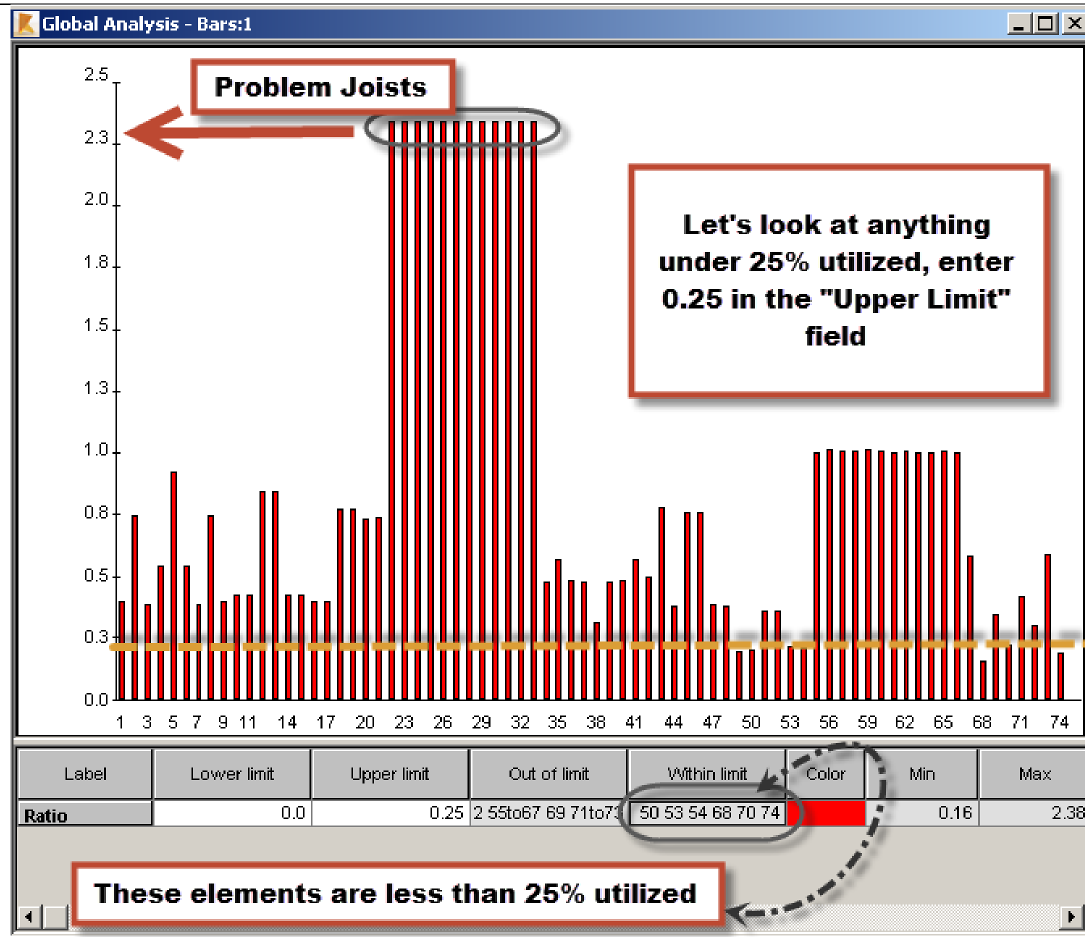
Results | Messages

Member	Section	Material	Lay	Laz	Ratio	Case	Ratio(uz	Case (uz)
8 Column_8	OK HSSQ 7x7x0.2	STEEL A500-4	61.20	61.20	0.76	18 ULS/2=1*1.20 + 2	0.35	146 SLS:STD/7=1*1.
9 Column_9	OK HSSQ 7x7x0.2	STEEL A500-4	61.20	61.20	0.41	18 ULS/2=1*1.20 + 2	0.17	146 SLS:STD/7=1*1.
10 Beam_10	OK W 14x61	STEEL A992-5	40.14	98.16	0.44	19 ULS/3=1*1.20 + 2	0.17	146 SLS:STD/7=1*1.
11 Beam_11	OK W 14x61	STEEL A992-5	40.14	98.16	0.44	19 ULS/3=1*1.20 + 2	0.17	145 SLS:STD/6=1*1.
12 Beam_12	OK W 14x61	STEEL A992-5	40.14	98.16	0.86	19 ULS/3=1*1.20 + 2	0.33	146 SLS:STD/7=1*1.
13 Beam_13	OK W 14x61	STEEL A992-5	40.14	98.16	0.86	19 ULS/3=1*1.20 + 2	0.33	145 SLS:STD/6=1*1.
14 Beam_14	OK W 14x61	STEEL A992-5	40.14	98.16	0.44	19 ULS/3=1*1.20 + 2	0.17	146 SLS:STD/7=1*1.
15 Beam_15	OK W 14x61	STEEL A992-5	40.14	98.16	0.44	19 ULS/3=1*1.20 + 2	0.17	145 SLS:STD/6=1*1.

Note: "OK" icon, section, material, Utilization Ratio, the Controlling load combations, and the Utilization Ratio for service "Ratio(uz)" and the controlling service load case. Any errors will be readily apprant here.

We can use the “Analysis” button to get a sense of how our members are performing, identify any members well over capacity and consider creating additional design groups to select some more useful members for these positions. You should be able to see under utilization in some of the girders in particular as the exterior girders have about half the load.

We can look at the most under-utilized elements by entering clipping value in the “upper limit” field of 0.25 for instance.



Now we can use "Edit in a new window" button to get a really quick look at the elements we can give their own design groups to improve utilization if we wish.

We quickly see that most of the upper story braces and the outside tie-beams could be re-designed for better utilization if desired.

You can continue this refinement process until you are satisfied with the balance between utilization and uniformity in your structure.