Altair S-FRAME 2021.1

Modeling
Hints and Tips

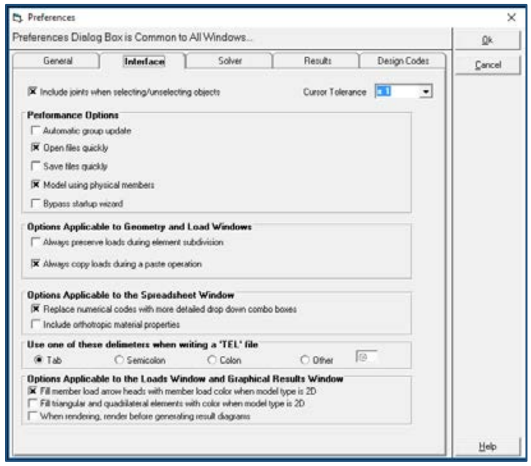
Intellectual Property Rights Notice

Disclaimer

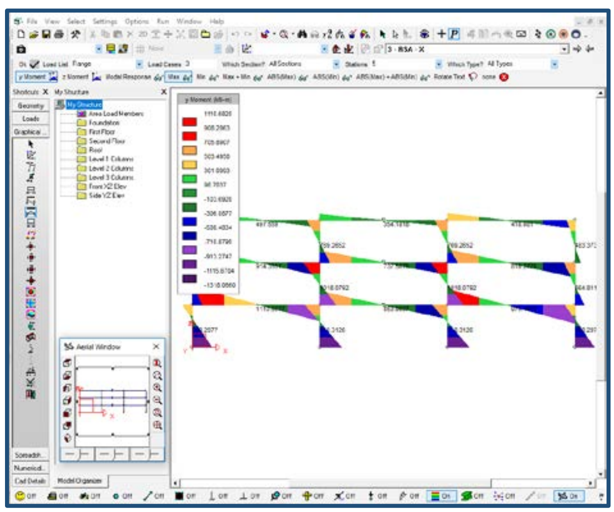
Considerable time, effort and expense have gone into the development and documentation of S-FRAME. It has been thoroughly tested. However, in using the product (including manuals), the user understands and accepts that no warranty on the accuracy or reliability of the product is expressed or implied by the developers or distributors. Users must understand the assumptions used in the product, know its limitations, and verify their own results.

Manual Setup and Conventions

Within the Manual there are screenshots and diagrams, emphasizing a certain step or results. In some cases, they indicate the correct data entry for a specific dialog:



In other cases, they may show the expected output of a process:



Notes: This symbol is used to highlight "Notes" on specific topics throughout this manual.

Good to Know: This symbol is used to highlight "Good to Know" facts throughout this manual.

Need to Know: This symbol is used to highlight "Need to Know" facts throughout this manual

Further Information: This symbol is used to highlight additional resources for specific topics.

Contents

Detailed Modeling Hints and Tips

In this manual we summarize some of **S-FRAME**'s more important features, we also give you hints and tips that we think may:

- help you to use **S-FRAME** more productively,
- help you to avoid some of the difficulties which other users have encountered.

Reviewing your model

During and after model creation there are many **View** settings that you can use to give you better feedback on the status and details of your model. You switch the most common options on and off by clicking on the appropriate status line buttons. Another particularly important feature when you are working with a 3D model is the facility to render it. This is described below.

There is no one fixed set of options that will suit everybody and every type of model. You can easily switch components of the display on and off. If you have worked through the tutorials, you will be comfortable with this.

Throughout this hints and tips section, you may find that the screenshots show more information than those you see by default as you use **S-FRAME**. We have not used anything other than standard **S-FRAME** options to create these screenshots, so anything you see you can repeat by making the appropriate settings. Furthermore, during the examples, we show in this chapter you will see that the details differ between screen shots. This is deliberate and done to show the pertinent information with maximum clarity. If you are following an example, then your screen may contain different content to the ones we show. This will not affect the feature we are demonstrating.

Rendering

S-FRAME includes options which allow you to render any section as well as triangular and quadrilateral elements.

S-FRAME uses the colors which you have defined for the current tool when rendering the members of your model; this enables you to see any anomalies in your structure very easily. The capture below illustrates the model when the Section Properties Tool is active.

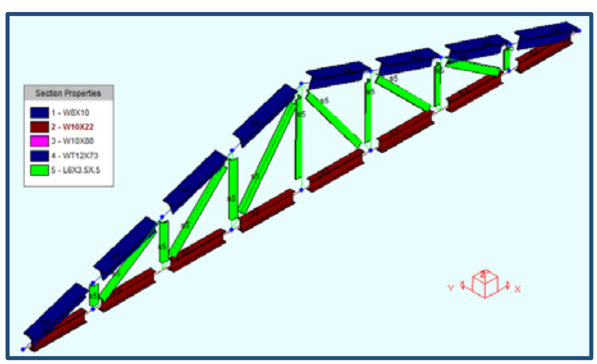


Figure 1.

You can clearly see that one of the infill members has the properties of the top boom.

While the capture below shows the same model when the Member Type Tool is active.

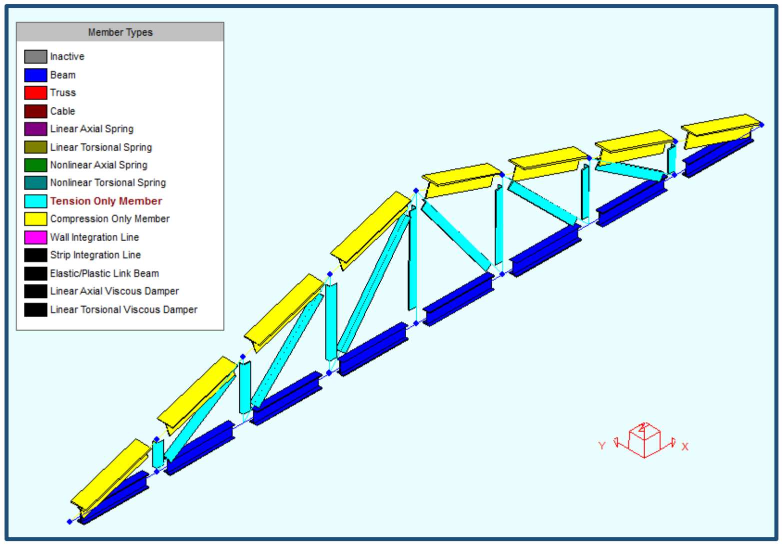


Figure 2.

Provided that you take the time to set the colors you can get detailed information about your model quickly.

You access the rendering options from the View menu.

If you only check the Render option (and none of the other options), then **S-FRAME** will only render those steel sections whose details come from the standard steel database tables.

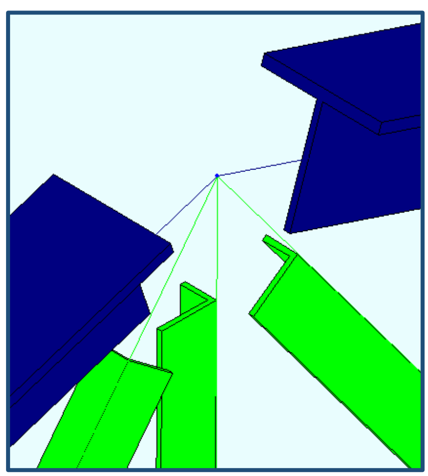


Figure 3.

The rendering is detailed and can be a most useful way to view member orientations in 3D structures. In the above example we corrected the previous anomaly, and then we set the Length Shrink Factor to 0.75. You can also use the Scale Factor to exaggerate the cross section dimensions if necessary.

You can also choose to render members whose properties do not come from the section database. In this case, **S-FRAME** renders the sections from the database as indicated above; **S-FRAME** renders all other sections using a large green I shape. This is to help you to visualize and confirm member orientation – the primary reason for the inclusion of this feature.

Using Folders

The screen shot below shows a complex structure where some effort has been made to sort the elements into logical groups. These can be selected/ manipulated/viewed using **Folder Technology**.

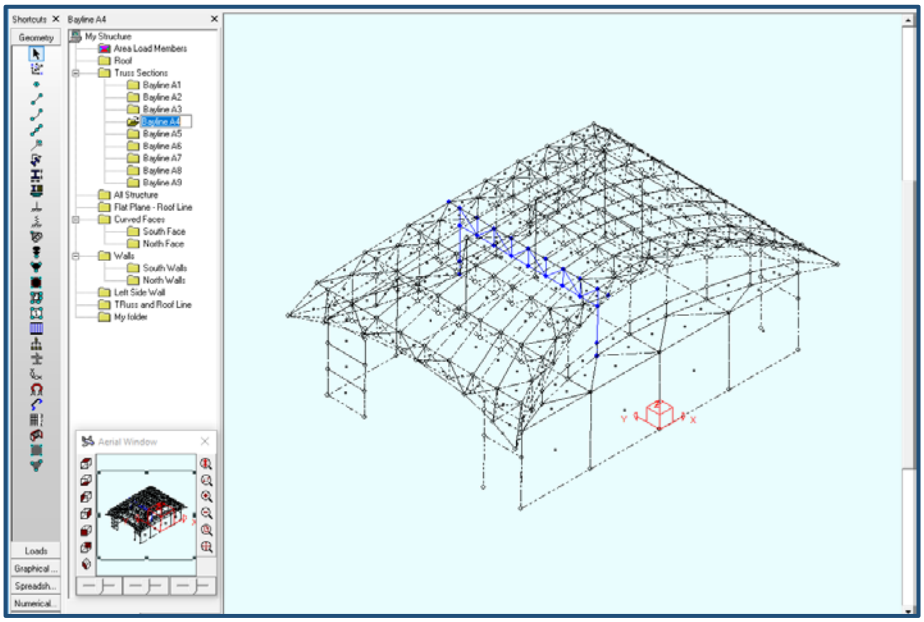


Figure 4.

Why Use Folders?

Anyone who has tried to work with large models in analysis packages will immediately see the advantages of folder grouping. Once you have groups then all of the following become easy, even in the largest of models, on a group by group basis:

- change section or material properties (or any other physical attribute),
- add loads (for instance to the All Top Chords group in the above model),
- review results graphically (for example examine the axial loads in one truss at a time),
- print results (for instance for the Diagonals group of one truss in the above example).

Once you start using folder grouping, you will wonder how you ever managed without it.

How do you create folders?

Folder technology is completely flexible. You create and name the folders in the hierarchy you want. Then you select which elements go in each folder. It is very much like working with Windows Explorer.

You can create or edit your folder hierarchy at any stage, both during model creation and in review after analysis.

You can include any member or joint in up to 30 different folders – you are not restricted to one folder per member. Your model can contain up to 2000 folders.

The creation of folders is well covered in the tutorials and the detailed help system.

An important part of the process of defining folder groups is the selection of the elements that go into each folder. Hence it is important that you have a good appreciation of the selection techniques available in **S-FRAME** – these are covered in the next topic.

Making and clearing selections

Understanding and using **S-FRAME's** selection options to pick specific groups or types of members quickly is perhaps the most important skill that any S-FRAME user needs to master. It is not difficult, but we suspect that most users seldom employ the more powerful options to their maximum potential.

Graphical Selection Options

These are covered in detail within the help system and are summarized in the tutorials.

For further information please see the **"Acquainting yourself with selection"** topic in the **Introduction Manual – 2D Tutorial**. If you have any doubts about these selection basics, we suggest you work through this section again.

If you want to use graphical selection in an advanced manner, then we strongly advise you to try to think 3 dimensionally. When you drag your 2D box across the screen, you are actually identifying a 3D slice through your structure. In practice, this means that you will often find it advantageous to adjust your 3D view's orientation to make selection easier.

To select all the members in an elevation, swap to a plan view first and then a single box selection does the job.

To select all the members on a floor, swap to an elevation view first and then a single box selection does the job.

Menu Selection Options

You will find a series of options directly on the Select menu which allows you to select or unselect all items of a given type simply.

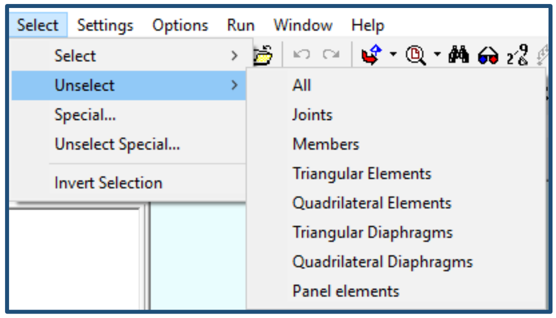


Figure 5.

The menu also gives access to **Select Special...** and **Unselect Special...** options that are described in detail within the help system.

We strongly recommend you take a little time to experiment with the various selection options.

Folder Selection

Essentially this is a way of repeating a previous selection of elements instantly. All you need to do is to create a folder, make your selection (using any method), and update your folder with your selection. Then you can select the same members again simply by clicking the folder.

On the **Status Bar**, you will find an **Automatically update open folder** setting. While you are creating folders, you will find that this is extremely useful. However, we strongly recommend that you switch this off when you are working on other aspects of your model (and hence not focused on re-grouping) as otherwise, you may inadvertently change the grouping you have painstakingly created.

As you build a model, you will generally assign similar section properties to members that will ultimately be loaded in the same way and then designed as a group. Therefore, you may want to use the **Section Numbers** (special) selection option to pick all members with the same section property in your model and then create a folder for that section. You can then load the members and review their results as a group easily.

Using user coordinate systems

This is an extremely powerful (although not new) feature which we think is underutilized and therefore we would like to take a little time to emphasize some of its uses.

Creating and saving user coordinate systems (UCS)

Moving the user coordinate system around lets you input model geometry relative to the most convenient origin. For all sorts of editing operations (moving, copying, generating, reflecting, etc.) it is invaluable.

Example

For example, the simple frame below shows one wing of an office building. The second wing is angled at 45° and offset by 4 m from the extreme right-hand column of this view.

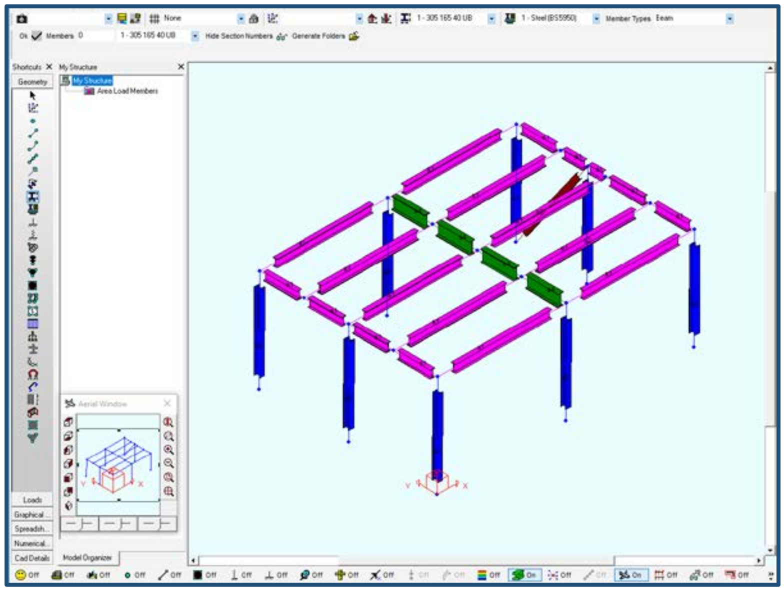


Figure 6.

If you want to work through this example with us, then load the model **UCS Example Stage 1**.TEL.

You can move the user coordinate system by picking the **UCS Tool** and then clicking on any node. This moves the user coordinate system to that node but does not rotate it.

To translate the user coordinate system, rotate it and create the new wing takes a few simple steps:

1. Left click on the **UCS Tool** and then click on the column base nearest to the required position. (Node 9)
2. Right click on the tool and specify a rotation of –22.5° about the **Z**-axis then choose OK.
3. Right click on the tool and specify a 2 m offset along **X** then choose OK. The coordinate system should now be in the position shown below, and we are ready to mirror the structure.

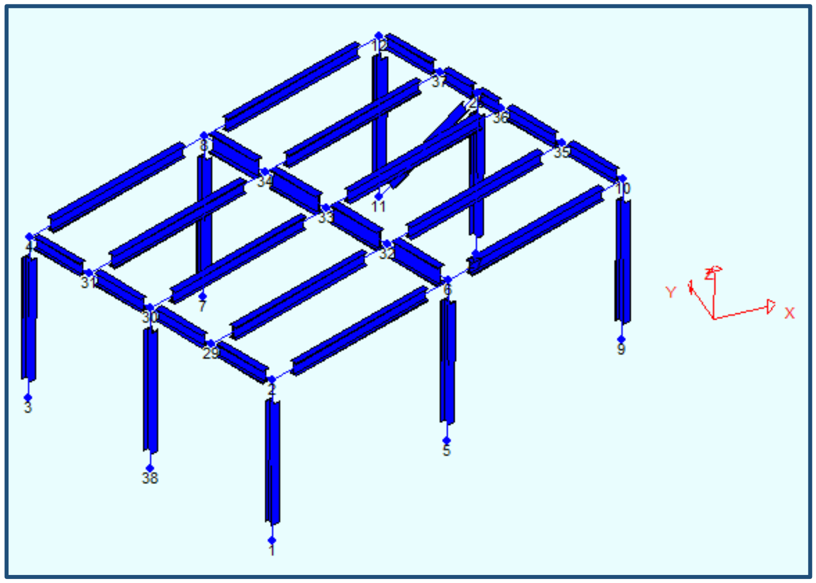


Figure 7.

4. Select the Move option (either from the Edit menu or the buttons along the top). Specify reflection in the **ZY**-plane, making a copy.

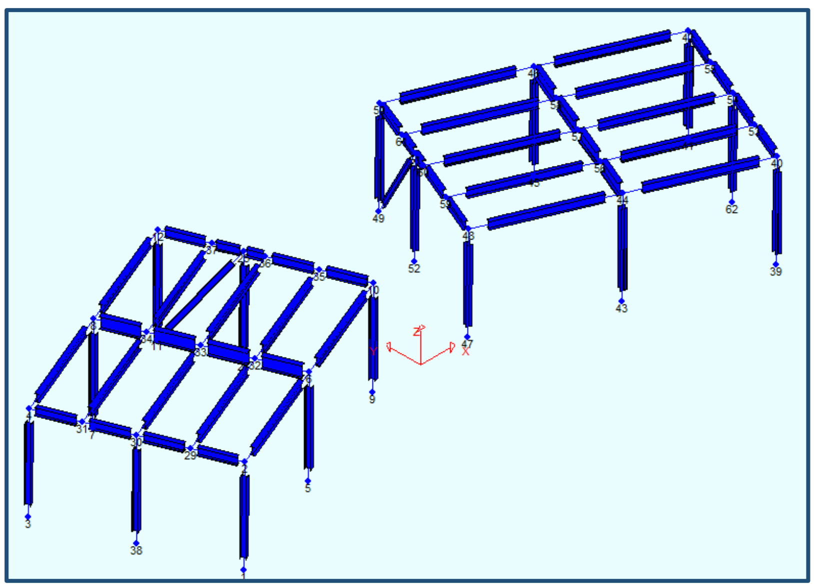


Figure 8.

Remember – always take the time to ensure that all the details of the members you are going to duplicate are correct before you use copy, reflect, generate, etc. If any details are incorrect, then you will have to edit the original and any copies later. This will take you at least twice as long and potentially much longer if you have made many copies.

At any stage of this process, you can save the user coordinate system with any name you like. In this example the user coordinate system used for the reflection was saved as Link Corridor and a further user coordinate system (shown above, and relevant to the second wing) was saved as Wing B.

There is a drop down list to allow you to choose a saved user coordinate system quickly.

S-FRAME allows you to apply any saved user coordinate system as a preferred joint displacement direction to any group of nodes so that you can apply loading and review the results in a more meaningful manner. The capture below shows where the Wing B user coordinate system has been applied for this purpose.

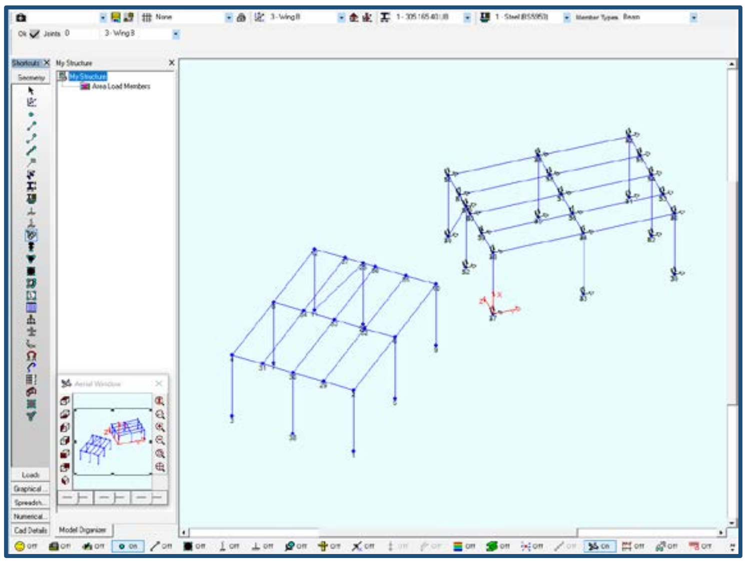


Figure 9.

This means that you can easily add Joint Loads which are relative to the wing, rather than having to resolve and apply these in the global axis system. You will also find that the base reactions for wing B are appropriate to the orientation of the columns.

If you want to review the model at this stage, then load the file **UCS Example Stage 2.TEL**. This model will be used as a basis to exemplify other features.

The above only serves as a very quick introduction to the flexibility and power of the user coordinate system. This is a very powerful feature indeed and has a host of other applications.

You could mix and match between Cartesian, cylindrical and spherical user coordinate systems within one model.

Control of the user coordinate system is also an important partner to many of the options discussed in this section of the documentation, including:

- Creating and saving views,
- Grids,
- Viewing results for shell elements.

Using Views

A **View** is a quick way of controlling a specific depiction of the whole or a part of your structure so that you can return to it easily.

You can name and save different views of your model from any **Graphical** window. The view then becomes available in all **Graphical** windows. When you save a view it takes in all sorts of details including:

- the current viewport scale,
- the currently selected folder,
- the current geometric label settings,
- the current option settings,
- the current grid settings,
- the current user coordinate system,
- the current projection type,
- the window size and location relative to the overall **S-FRAME** window.

When you create (or update) a view, then the current settings of the **Options** menu are stored and applied to all 3 **Graphical** windows. The settings of the **View** menu are stored for each **Graphical** window individually. Since the status line buttons are shortcuts to commands on both the **Option** and the **View** menus some **Status Line** settings update all the **Graphical** windows while others only update the current one.

As mentioned above, when you save a view, **S-FRAME** automatically saves the current **View** menu settings individually for all 3 **Graphical** windows. This means that you can save a view while working in the **Geometry** window and then use the same view in the **Graphical Results** window. However, you may need to revise and update the **View** menu settings when you first use the view in the **Graphical Results** window.

Why use Views?

With the advent of **Folder Technology**, the new active **Status Line** and the **Views and Grids** toolbar which allow you to customize each **Graphical** window more quickly views have lost some of their importance. However the ability to create views still remains a very useful feature when used carefully as the examples below illustrate.

Since the details stored for a view also include the active folder, it is easy to think of a view in terms of what you are viewing. If instead, you think of the view in terms of how you want the information to appear, then you will find that views are a much more powerful feature. You choose the appropriate view and then control what you want to show using the folders you have already created.

Example 1

If you want to work through this example with us, then load the model **Views Example 1 Stage 1.TEL**.

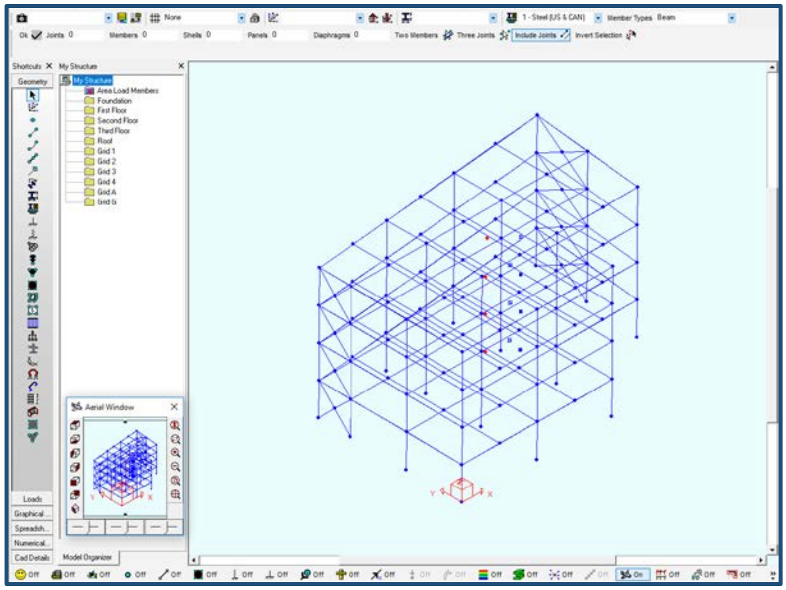


Figure 10.

This 4 story building is not a huge model. However, it is big enough to make a visual inspection of input and results difficult in some instances. With the following steps we create and save views to make our review process easier:

1. First, we create folders that group together obvious subsets of our model. We have already done this, but if you like, you can add further groups.
2. Select and maximize the **Geometry** window. Now ensure that the following status line options are switched on – Supports; Releases; Shrink Elements and Local X axis.
3. Either click the View icon ( ) from the **Views and Grids** toolbar or right click over the graphics area and select Views from the context menu.

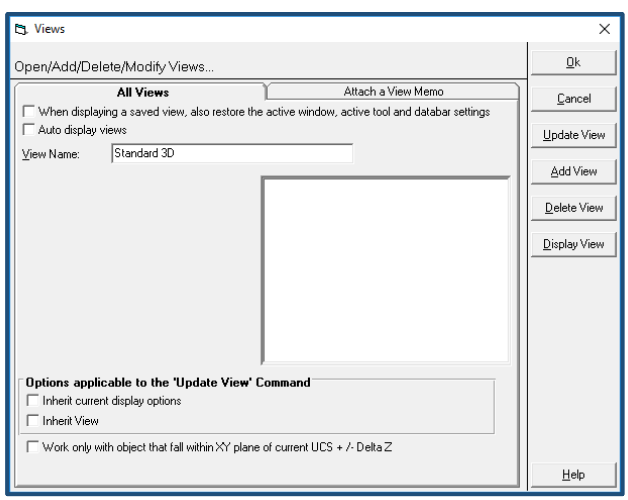


Figure 11.

4. Enter the view name **Standard 3D** and then click Add View.
5. Now click **View > Render...** and switch rendering on. Be careful to check the option to render selected members only
6. With the entire model rendered, repeat steps 3 and 4 to save a second view called **Rendered 3D**.
7. Now switch to the **Loads window**, and you will find you can switch between these two view types by picking them from the list in the **Views and Grids** toolbar. Choose the Rendered 3D view and then click different folder groups and you will see that the members of those groups are rendered.
8. Change back to the Standard 3D view and use the **Status Line** to switch on the options to hide unselected objects, view joint loads and view member loads. The view will become quite cluttered, but let's repeat steps 3 and 4 to save this view as **3D plus loads**.
9. With this third view active, start selecting floors from the folder list, you should see views like the one below.

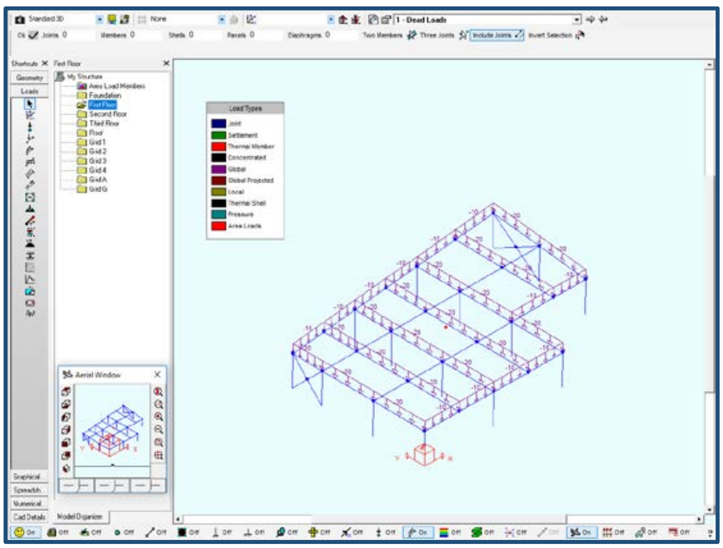


Figure 12.

Example 2

If you want, you can start working through the example at this point. If you want to do so, then load the model **Views Example 1 Stage 2.**TEL.

1. Run a linear elastic analysis and then when the **Results** window is displayed, pick the Standard 3D view. Then open the diagrams and scale factors dialog and make the settings shown below.

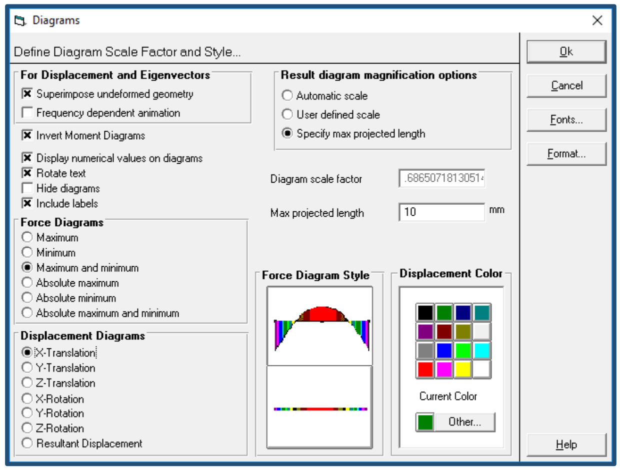


Figure 13.

2. Save these settings and then click the Moment Diagrams Tool – you will see a very cluttered display.
3. Increase the number of stations to 9 (don't forget to click Accept Data ( )).
4. Now use the **Status Line** to switch Hide Unselected Sections to **On**, and Display Legend to **Off**. The display will still be extremely cluttered but save this view as **Detailed Results**.
5. With this view active, click on different folders and try swapping to different load cases or combinations. By adjusting the view direction, you can see detailed views such as that shown below.

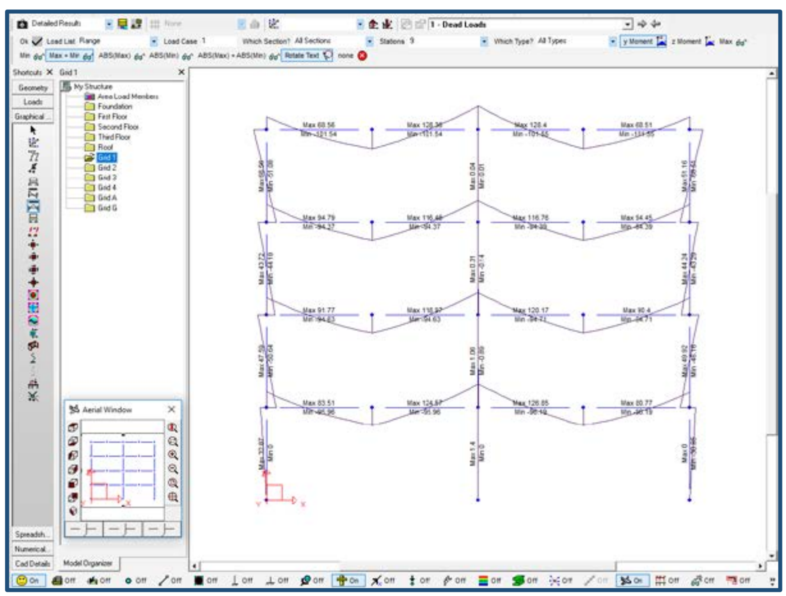


Figure 14.

Since it is relatively easy to customize views as you go along, it is up to you to decide whether it is worth saving many different views.

If you find that you are forever setting up a series of standard view styles which are the same in many models, then you could save these in a start-up (or template) model (choose **File / Save As** and give a descriptive name such as **TEMPLATE.TEL**). You should ensure that this start-up model does not contain joints, members, irrelevant user coordinate system information and such like. Each time you want to create a new model you simply load the appropriate template model and hence the views it contains.

Using grids to create your model

You can define grid systems before you create a single joint or member. You can then use the grid systems to place joints and/or members by clicking on the appropriate grid intersection points.

The process of creating and using grids has already been covered in some detail within the overview on creating a 2D structure, the 2D tutorial, and the 3D tutorial.

Using grids to edit your model

Irrespective of whether you have used grids to build your model, you can use them to great effect if you need to modify it.

In the following example we will:

- load a model where no grids have been defined,
- automatically generate grids from this existing model,
- use these automatically created grids to stretch the model.

Example

If you want to work through this example with us load the model **Grids Example 1 Stage 1**.TEL.

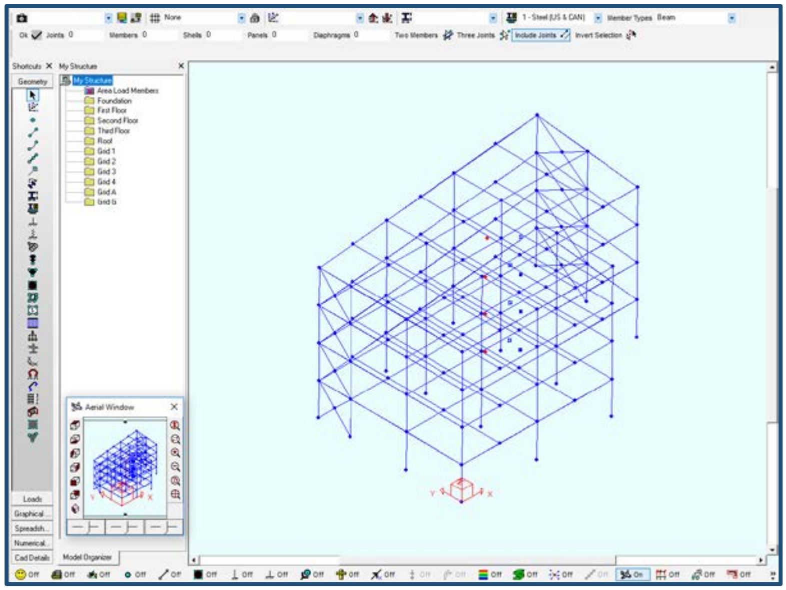


Figure 15.

1. Click the Edit Grids icon from the **Views and Grids** toolbar
2. Right click on the folder None and choose New grid set from the context menu that appears.
3. Change the name of the new grid set to **Foundations** and then click on this grid set to be sure it is open.

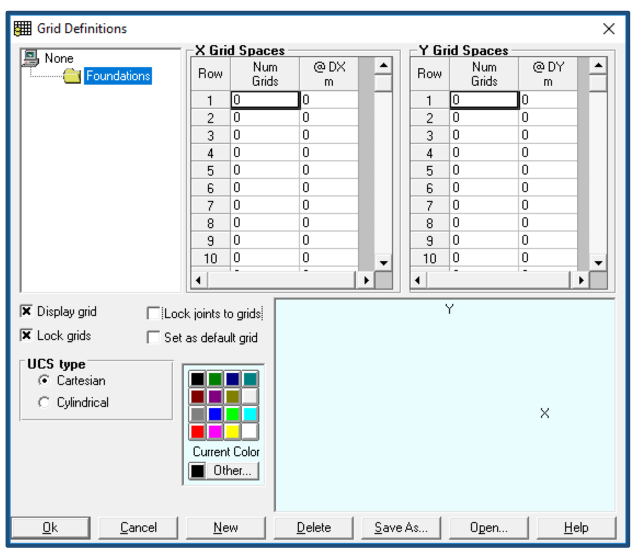


Figure 16.

4. Now right click on Foundations and choose **Auto Generate Grid Set** from the context menu.

5. Check the **Display Grid** option, and the **Grid Definitions** dialog should now look like that below.

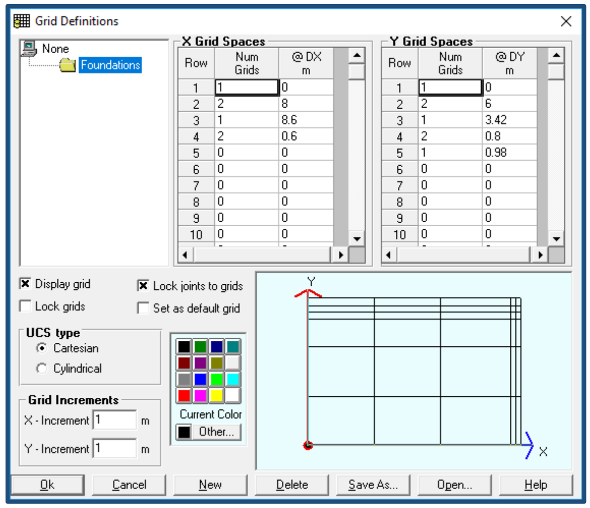


Figure 17.

When we clicked Auto Generate Grid Set **S-FRAME** automatically located every joint in the current **XY**-plane and ensured that a pair of grid lines passed through that joint position.

6. Now check Lock joints to grids and uncheck **Lock grids**. Set the **X**- and **Y**-grid-increments to 1 m — this is the increment by which we will stretch this structure.
7. Finally, click **OK** to close the dialog. You will see grid lines in drawn on the **XY**-plane.

8. Select the grid that runs along the **X**-axis, and drag it (by holding the mouse button down) as shown below.

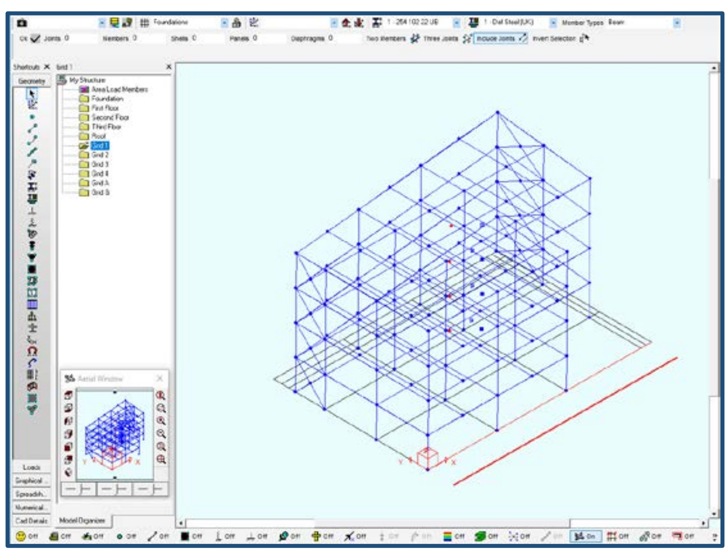


Figure 18.

9. Now release the mouse button and **S-FRAME** identifies and moves all of the joints that lie on a plane at 90° to the current XY-plane.

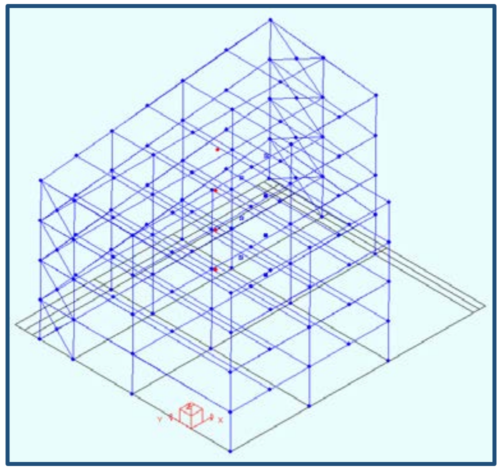


Figure 19.

10. Click the Undo icon from the **Edit** toolbar and **S-FRAME** returns all joints, members and grids back to their previous locations.
11. If you want you can try moving the other grids around to further edit the structure at this point.

By adjusting the increments (and zooming in if you are using small increments), you can make very precise adjustments to your model's geometry in this way.

You will find it difficult to move inclined elevations without distorting them. To do this successfully, you need to adjust your user coordinate system (the example below should help clarify this).

But what if we want to change our model's floor levels rather its plan geometry. The current grid in the current **XY**-plane is no use to us. We need to change the plane and then create a new grid system appropriate to the floor levels.

12. First, we choose the **UCS Tool** and access its dialog to rotate the coordinate system by 90° about the **X**-axis. The current grid system is immediately displayed in the new **XY**-plane, as shown below.

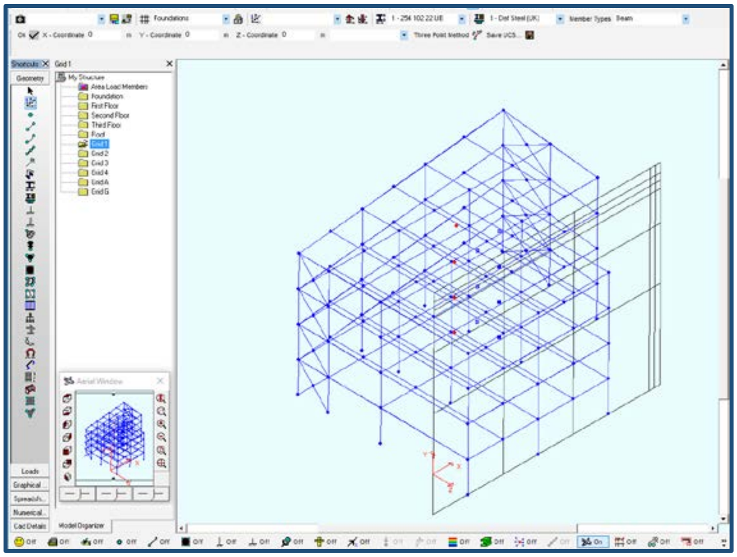


Figure 20.

13. Now we repeat steps 1 to 7 to access the **Grids** dialog, create the new grid set **Elevation**, and automatically generate its grid lines (which relate to the current **XY**-plane and close the **Grids** dialog).
14. You will now have grid lines at each floor level, and you can drag these around as the exaggerated capture below shows.

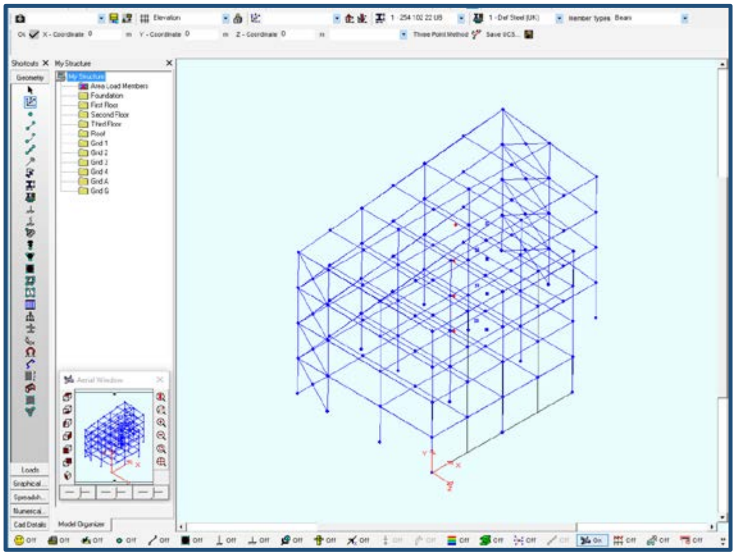


Figure 21.

The above example also shows that the cross bracing becomes distorted as the grids move. Editing your structure by dragging grid lines is clearly a very powerful feature, but you need to be careful to watch out for side effects such as this.

Generating Arrays of Equally Spaced Floor Joists

S-FRAME now includes facilities to allow you to create a series of infill joists quickly and easily. In many cases, you will be able to generate such joists across the entire width of a building if you need to.

Example

If you want to work through this example with us, load the model **Floor Joists Stage 1.TEL**.

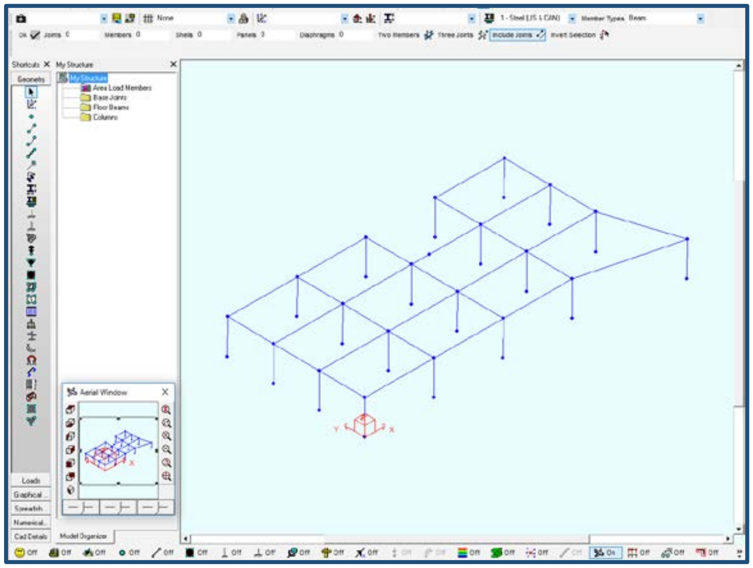


Figure 22.

The standard bay centers are 6 m in the X-direction and 4m in the Y - direction.

We shall add joists across this building using various options.

To make the example as easy to follow as possible, we shall work in 2D mode and use a plane which only contains the floor (at an elevation of 3 m).

1. Click the **2D** icon (2D) on the Edit toolbar, make the following settings, click **OK**.

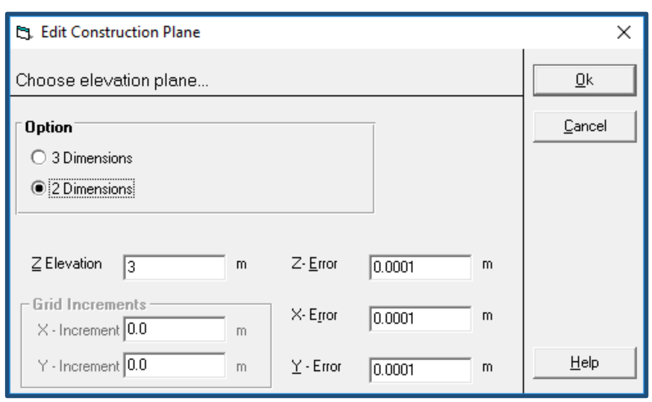


Figure 23.

2. Now we can zoom to the full extent of the structure.

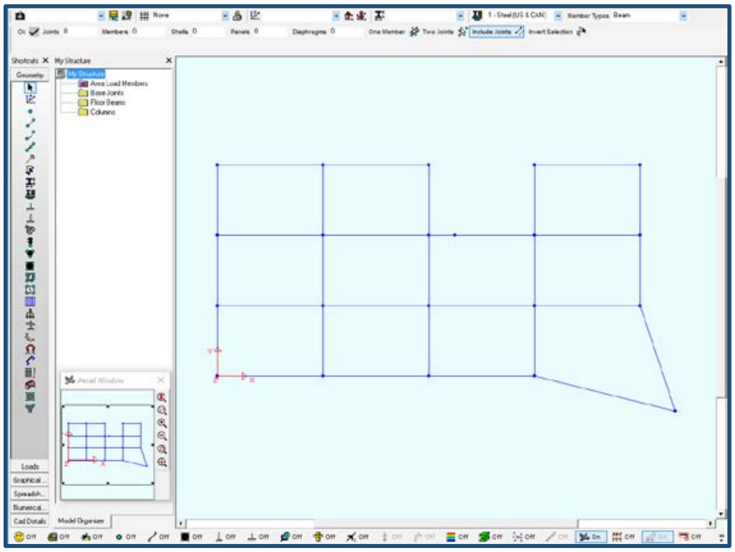


Figure 24.

3. Now to create the joists. Click the Member Definition Tool and then proceed as explained below.

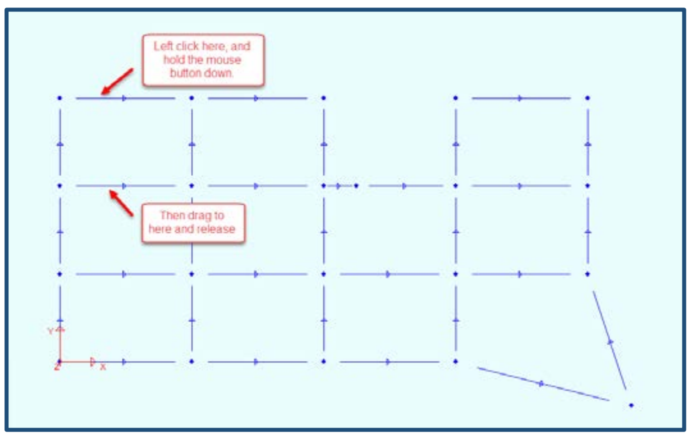


Figure 25.

4. You will see the **Add Floor Joists** dialog. Make the settings shown below and then click Ok.

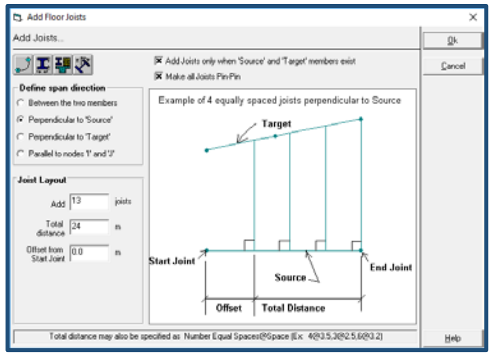


Figure 26.

5. This creates the joist layout below.

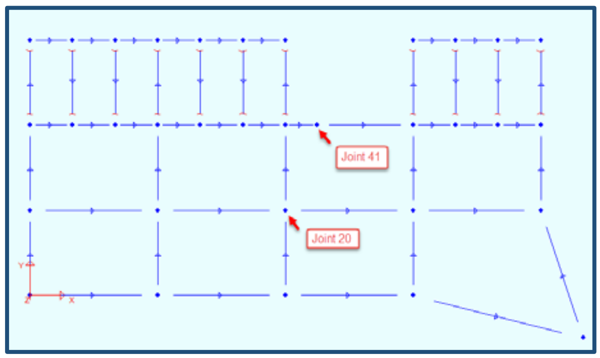


Figure 27.

The joist layout includes the existing beams. **S-FRAME** recognizes this automatically and retains the existing members, rather than creating new ones. Also, because of the settings we made, **S-FRAME** does not create members in the bay where there is no beam in the '**Source beam**' position.

6. We will now create a new series of joists which span parallel to the two nodes indicated above. If you switch on the display of the joint numbers, you will see that these are joints 20 and 41, we shall leave these off here since they clutter the display. Now click and drag as shown below.

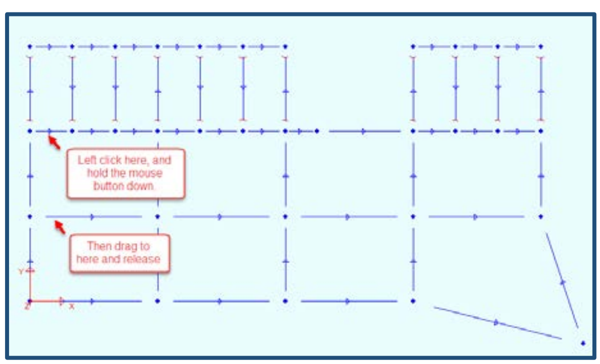


Figure 28.

7. You will see the **Add Floor Joists** dialog. Make the settings shown below and then click Ok.

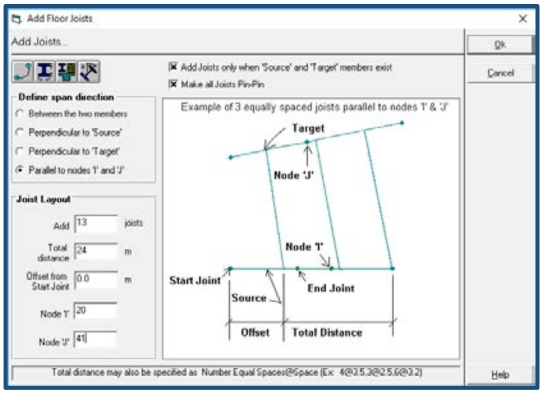


Figure 29.

8. You will see that this time the new joists are parallel to the line through the two nodes that we specified.

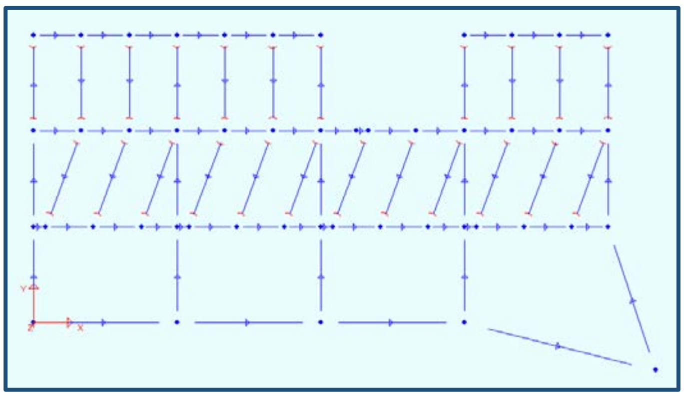


Figure 30.

9. Now to create the remaining floor joists, click and drag as shown below.

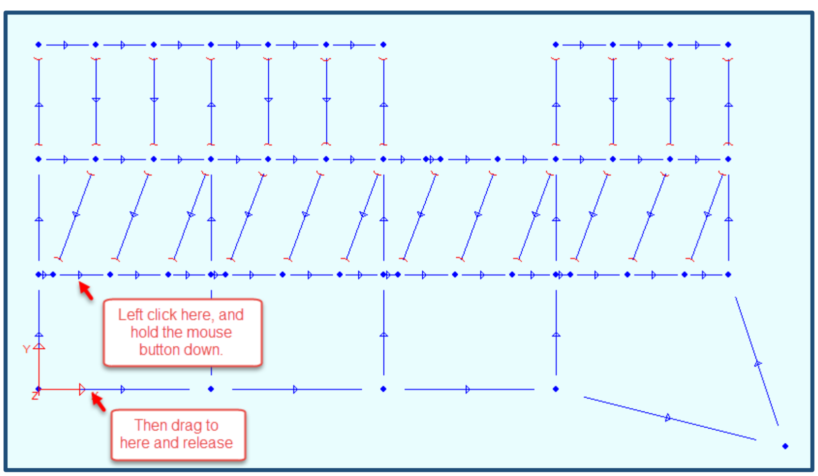


Figure 31.

10. You will see the **Add Floor Joists** dialog. Make the settings shown below and then click Ok.

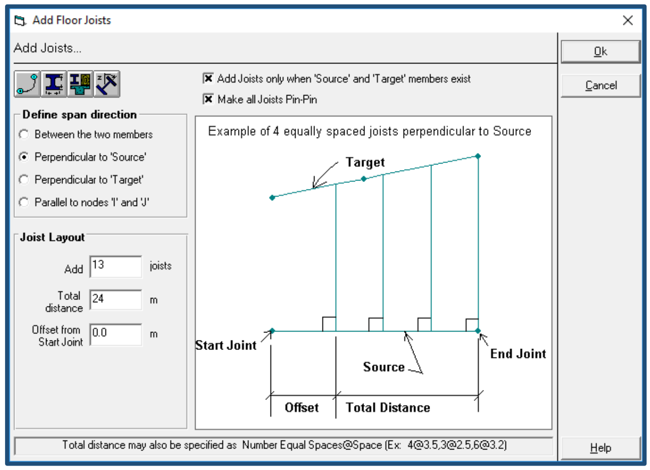


Figure 32.

11. You will see the layout of joists as shown below.

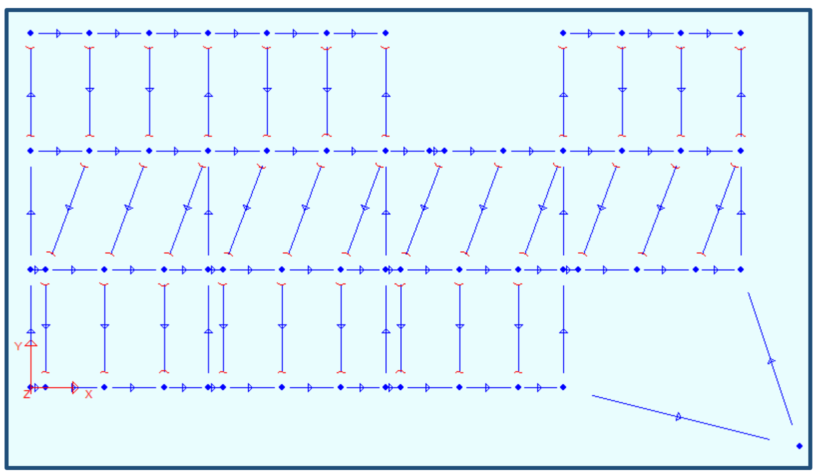


Figure 33.

But this is not the layout that we expected at all! So, what's gone wrong? Well, we told **S-FRAME** to create 13 joists over a length of 24 m. However, **S-FRAME** has not been able to create the joists over the full 24 m since what would be the Target member in the last bay is not in line with the initial Target member we selected. **S-FRAME** has recognized this and has created 13 joists over the reduced length. Furthermore, it has recognized the presence of the existing nodes in the Source members and has tied the new joists into these.

12. Click **Undo** twice, to completely clear the generation of this incorrect layout. Now click and drag again as shown below, note the change in direction.

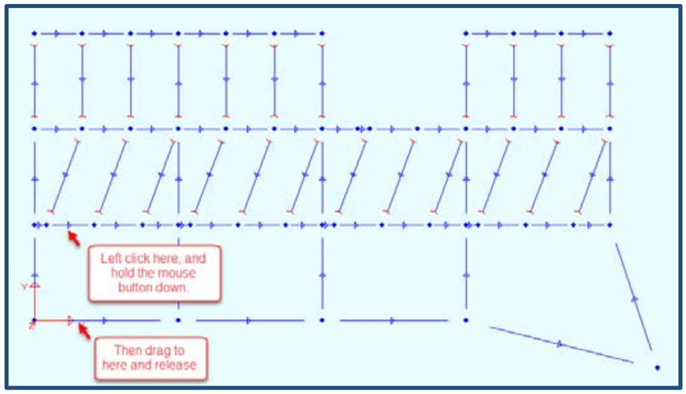


Figure 34.

13. Complete the **Add Floor Joists** dialog as shown below and then click Ok.

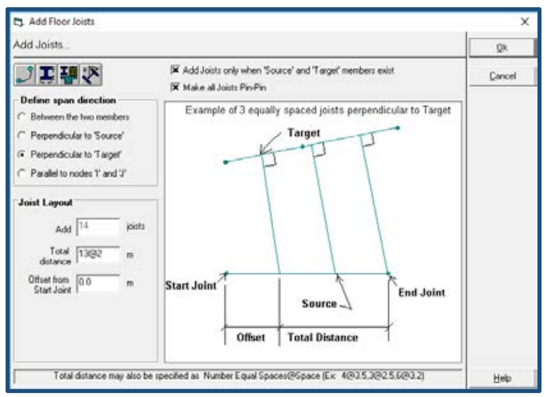


Figure 35.

14. This layout of floor joists is more what we might expect, **S-FRAME** has created the joists as required but has still not created the joists in the final (skew) bay.

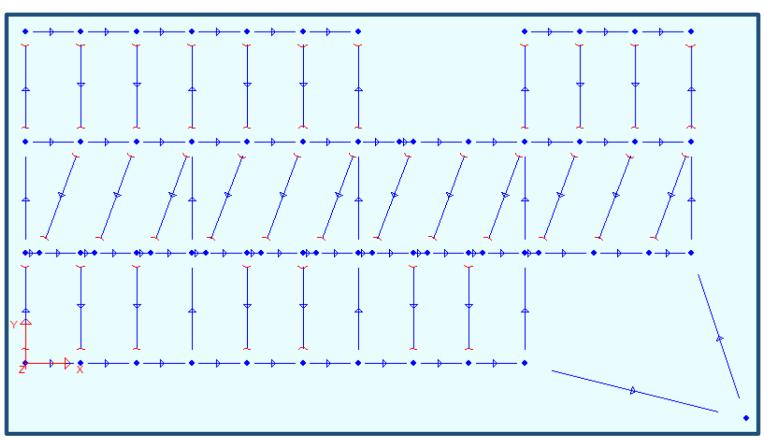


Figure 36.

15. So let's create an arrangement in this bay. We don't have to choose two beams along opposite sides of the bay; we can choose any two beams. So click as shown below.

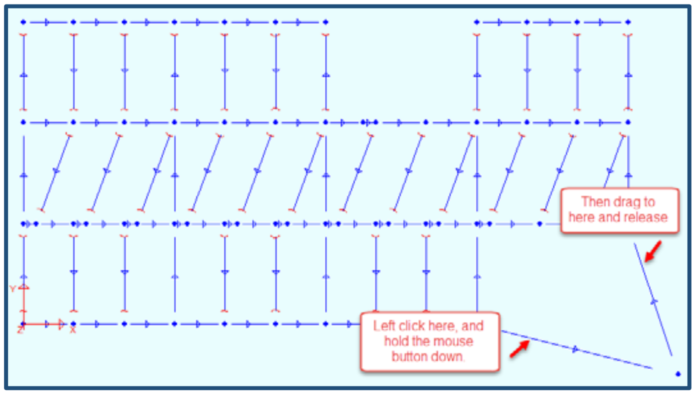


Figure 37.

16. Complete the **Add Floor Joists** dialog as shown below and then click Ok.

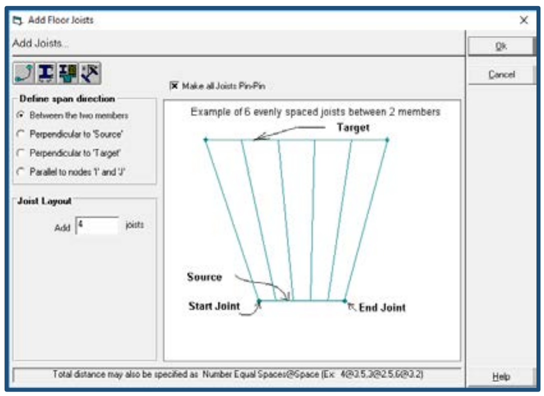


Figure 38.

17. Now we get the arrangement of joists shown below.

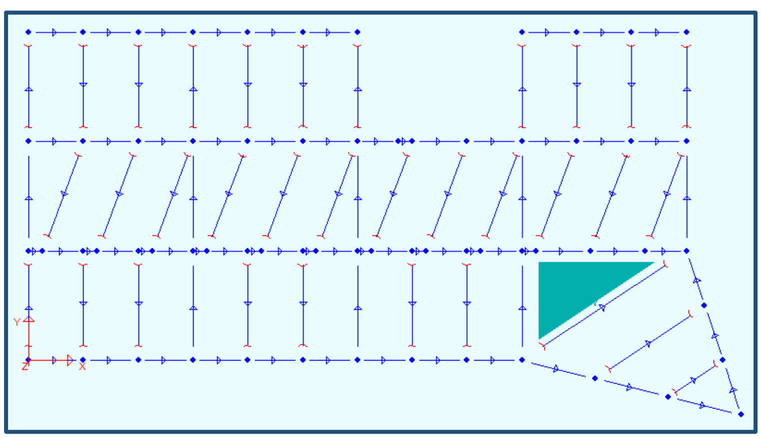


Figure 39.

18. If you want, you can continue and define joists in the shaded area above.

The Clone Tool

The **Clone Tool** dialog has been continually enhanced during the development of **S-FRAME**. It is now possible to extrude members (and shells or panels) between each generated set of nodes. Additionally, extruded members can be evenly subdivided by specifying more than 1 link.

In the example below the truss was created using the truss generator options and then rotating and reflecting. The members were copied to the clipboard, and the **Clone Tool** was accessed with a left click then a right click.

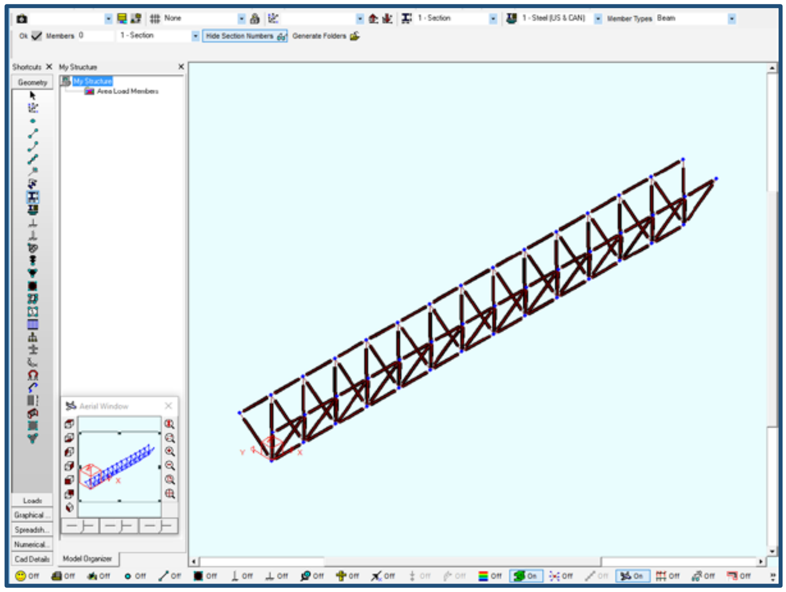


Figure 40.

The dialog was completed as shown.

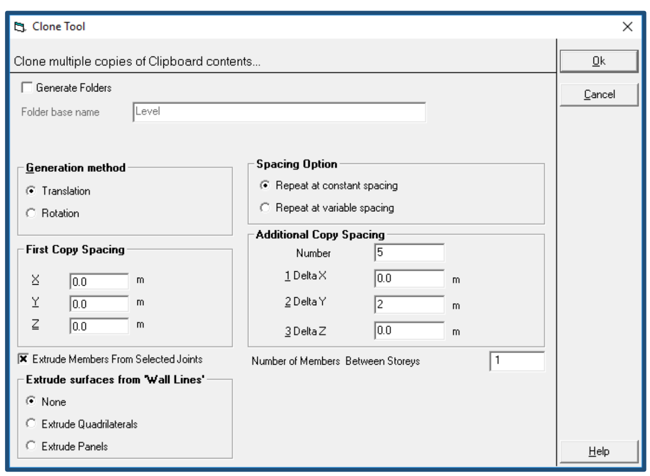


Figure 41.

With these settings the structure was generated as shown below.

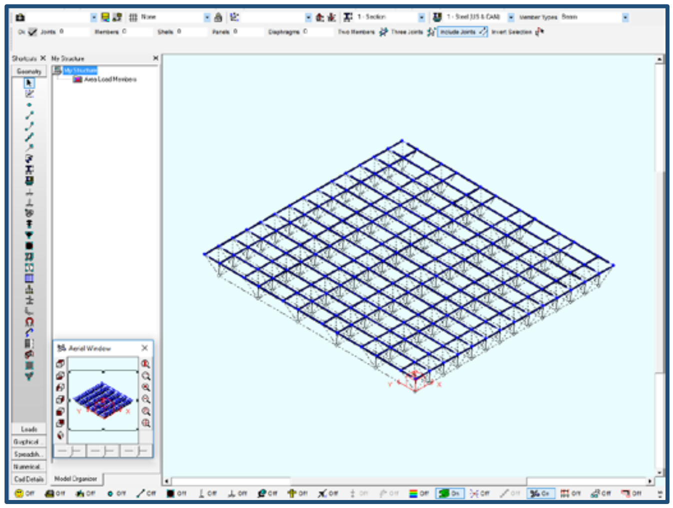


Figure 42.

No offset needs to be specified for the first repeat. Duplicate members will not be created in this instance.

In the view above, only the top chords are selected and rendered. The extruded members are given the current default section and material properties.

Other generation options

This section is simply a reminder of the options that **S-FRAME** provides to enable you to generate all or part(s) of your structure quickly. You can combine many options within a single model, and careful use will significantly speed up the creation of your model.

You access all these generation options from the Edit menu, and they are covered in some detail within the help system. You can use the links below if you want to find out more.

Create regular frameworks

This option allows you to create a regular, basic beam and column (for example an office block), structure.

Generate trusses

The **Truss Generator** options allow you to create:

- parallel chord trusses,
- pitched chord trusses,
- miscellaneous trusses.

Generate meshes

The **Mesh Generator** options allow you to:

- mesh curved surfaces (creating shell elements),
- mesh circular cut-outs (creating shell elements),
- mesh quadrilateral cut-outs (creating shell elements),
- mesh panel elements (creating shell, plate, or membrane elements),
- convert beam elements to meshable panels.

Generate physical/ analytical models

The **Analytical Model** options allow you to:

- generate rigid diaphragm mass

This does not produce any elements. Instead, it exposes a center of mass node to which you assign a lump mass and whose position you can edit,

- connect overlapping joints, which allows you to merge separate structures and in effect **meld** them together,
- connect intersecting members, which automatically splits members that cross to create members that intersect at the crossing point. You would perhaps use this option before meshing your structure.

Subdividing (selected) elements

This option is particularly important when you automatically mesh panels as described later. However, you can choose to subdivide any or all selected elements irrespective of the presence of panels.

The procedure for subdivision is simple:

- select the elements you want to subdivide,
- click on the sub-division toolbar button,
- complete the dialog, specifying into how many segments you want **S-FRAME** to divide each type of divisible element.

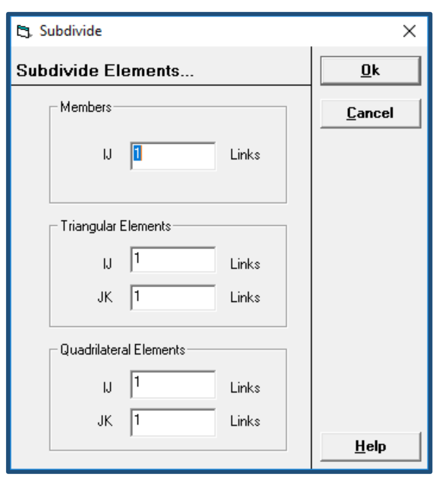


Figure 43.

If you have applied loads to members or shells that are subdivided, then **S-FRAME** automatically recalculates the equivalent loads on the subdivided elements, if you have checked the **Always preserve loads during element subdivision** option in **Preferences**.

We would advise you to exercise caution if you use this (much requested) feature. It makes it very easy to **over model** any structure (create a vast, slow and unmanageable model that does not increase the accuracy of the results but does greatly increase the amount of paper needed to print them). Before you use this tool, it is essential that you think carefully about your objectives for a particular analysis. As a general rule, we would recommend that you keep your model as simple as is possible while remaining consistent with these objectives.

The model below is a simple example of **Bad Modeling**.

Shells have been defined between the beams, and then both beams and shells have been subdivided to create meshed slab panels.

This simple looking model (investigated by many users) presents many difficulties, for example:

The loads will be dispersed logically, but this logical, analytical, dispersion will not always be what you expect For instance as the beams deflect the slab will start to carry load directly to the columns. As a consequence, the beam bending moments may be less than those anticipated in normal engineering practice.

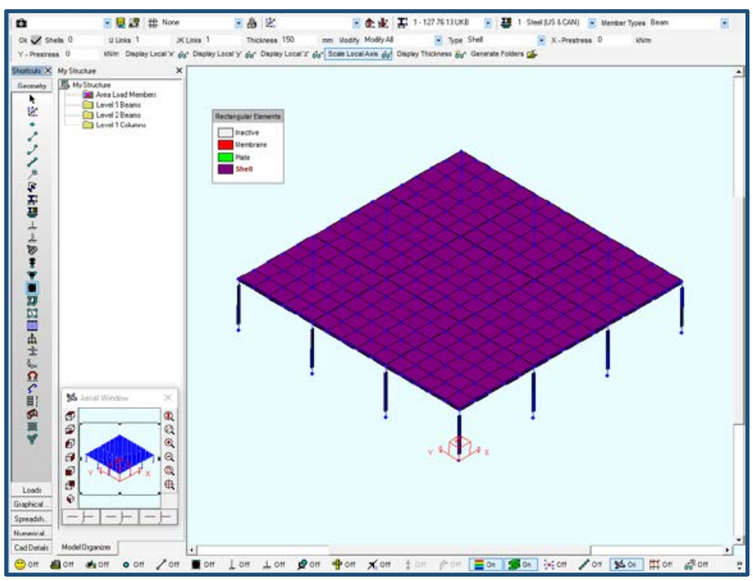


Figure 44.

Using rigid diaphragms

There are several ways to model diaphragm action in S-FRAME:

- using shell elements,
- using slaving (linked degrees of freedom),
- using equivalent (or fictitious) bracing elements.

S-FRAME allows you to automate the third technique in various ways.

Example

If you want to work through this example with us, load the model **Diaphragms Example 1 Stage 1.TEL**.

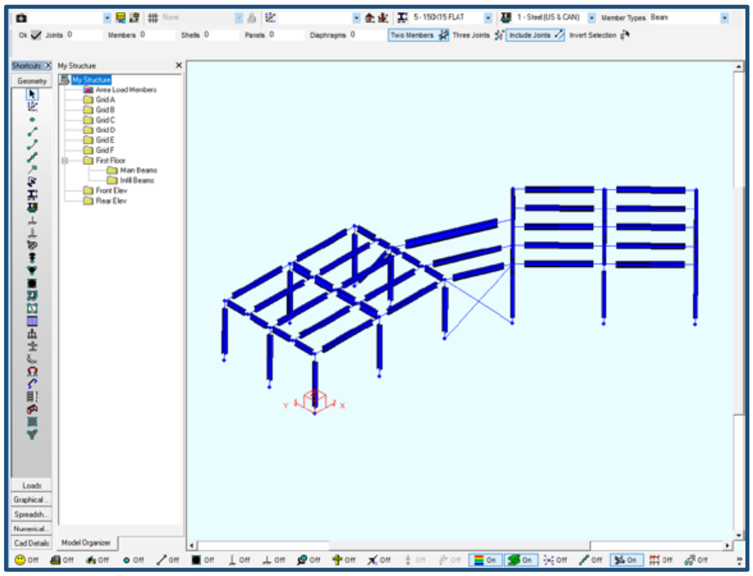


Figure 45.

This model is based on the example we started earlier, see **"To create and save user coordinate systems (UCS)"**

To see the view above choose the Rendered 3D view from the list.

Since we last saw the model some changes have been made namely:

- additional steel across the link,
- a simple grouping of members.
- a single load case has been defined. This imaginary case (approximating to wind blowing on to the grid A) would cause translation and rotation of the floor plate.

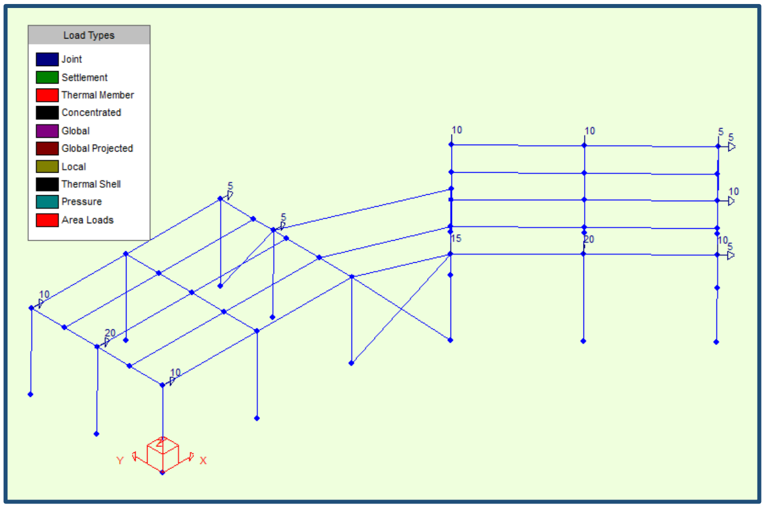


Figure 46.

The cross bracing is not rendered because it is formed from non-database sections, namely flats.

The loads applied to the right-hand wing are rotated to a different (and more relevant) user coordinate system using the **Joint Displacement Tool**.

If you review the structure, you will find that the extreme left and right-hand gables (grids A & F) are moment frames. All other connections are pinned.

Now perform a linear static analysis of the structure. On reviewing the results, the best indication that something is wrong comes from examining the deflection diagrams – we would not expect to see a deflection of 39 m!

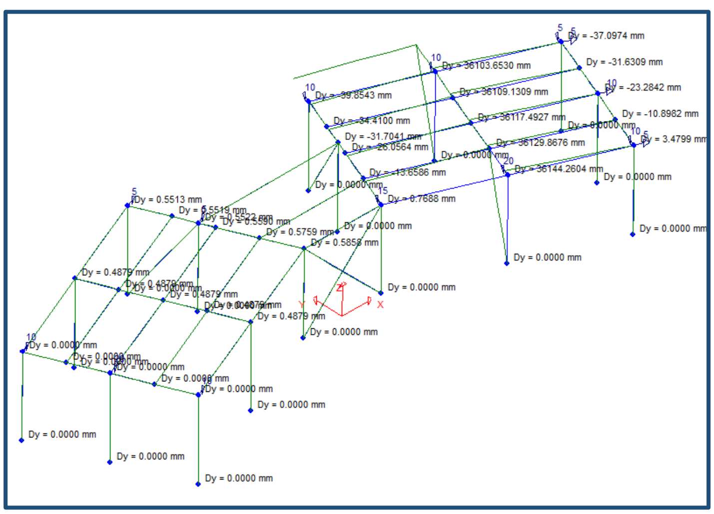


Figure 47.

In some cases, **S-FRAME** will warn you that a structure seems to be unstable. Regardless of whether or not you see such warnings, it is always advisable to examine deflections for excessive movement.

(**Almost invariably true**) If your results don't make sense, spend some time looking at the deflections — are they going in the direction you expect? — are they of the expected magnitude? If the answer to either of these is **No**, then you need to check your modeling.

S-FRAME allows you to add diaphragms to your model in several ways, but there are two basic types of diaphragm:

General Diaphragm - If you use this option you pick a series of nodes to define a plane (the number of nodes depends on the tool you are using). **S-FRAME** automatically finds all the nodes in that plane (and which do not lie in a region defined as a hole) and connects them all to a single node using a hidden system of rigid bracing.

Whichever tool you use to define a general diaphragm, it is best to use nodes that are placed reasonably centrally within the area covered by the diaphragm.

For new models, we would recommend that you use the **Panel Element Tool** to create a **Rigid** diaphragm as shown in the following example.The **General** diaphragm options were developed as a way of dealing with awkward geometries before **S-FRAME** could handle panels.

Rigid Diaphragm - If you use this option you pick a series of nodes that define an area within a plane, you can repeat this procedure to define as many separate areas as you like. **S-FRAME** automatically finds all the nodes in all the areas defined in the plane (and which do not lie in a region defined as a hole) and connects all the nodes within them to a single central node using a hidden system of rigid bracing.

S-FRAME automatically calculates the position of the central node, the **center of mass joint** and adds this to the model. For dynamic analysis purposes, **S-FRAME** also calculates the lumped masses **Fx**, **Fy,** and **Mz** relative to the diaphragm plane and applies these to this joint.

For detailed coverage of **Mass modeling** for vibration and response spectrum analysis **see Vibration Analysis Options** in **Chapter 4** of the **S-FRAME Verification Manual** and **Analysing and Designing for Earthquakes** in **Chapter 5** of the **S-FRAME Verification Manual**.

Creating a diaphragm

You can add both rigid and general diaphragms to your model using the Triangular and Quadrilateral diaphragm tools, or alternatively you can use the Panel diaphragm options. We anticipate that this latter option will be the most commonly used, and so we will use it in this example.

1. Select the Panel Element Tool, and then pick Rigid Diaphragm from the list.

   For linear elastic analysis, the diaphragm thickness and its material are irrelevant. You can set these to 250 mm and Concrete as shown above if you wish.

2. Click on the joints that define the outer perimeter of the first-floor area (2, 10, 48, 40, 42, and so on) to create a panel the same shape as the floor.

3. Switch Hidden Surface Removal On (under the options menu), Shrink Elements Off, and adjust the diaphragm color (by right clicking on the Diaphragm Tool) to see the diaphragm highlighted as shown below.

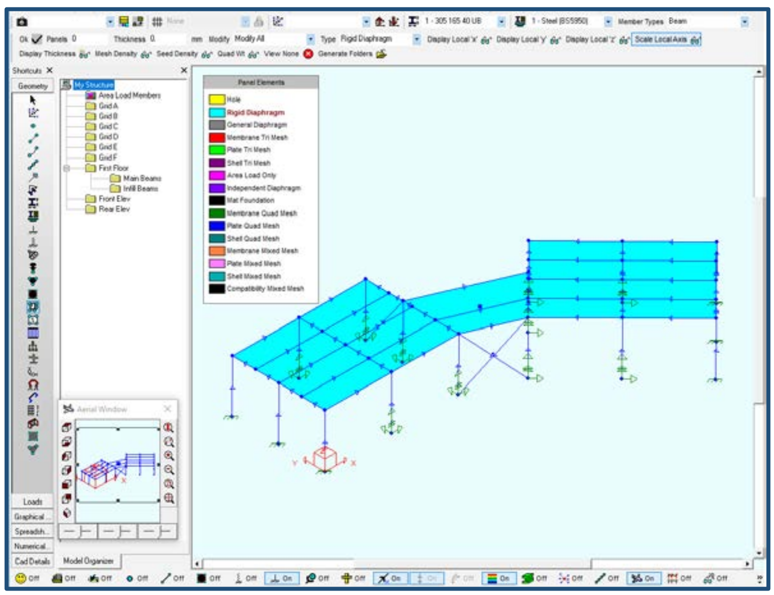


Figure 48.

4. Now analyze the structure again, and a plan view of the deflected shape clearly shows translation and a degree of rotation which are more in keeping with our expectations.

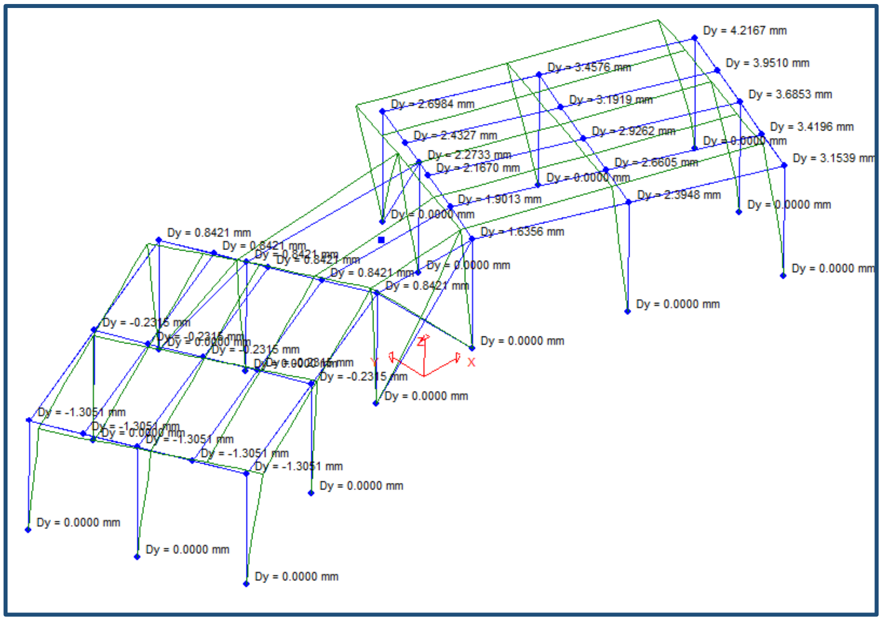


Figure 49.

The displacements for the right-hand wing are reported using the same user coordinate system as that used for the application of the loads. It is clear that the left-hand gable is not swaying as significantly as the right.

5. If you view the bending moments, you will see that the higher forces are in the right-hand gable, as we would expect for this load condition.

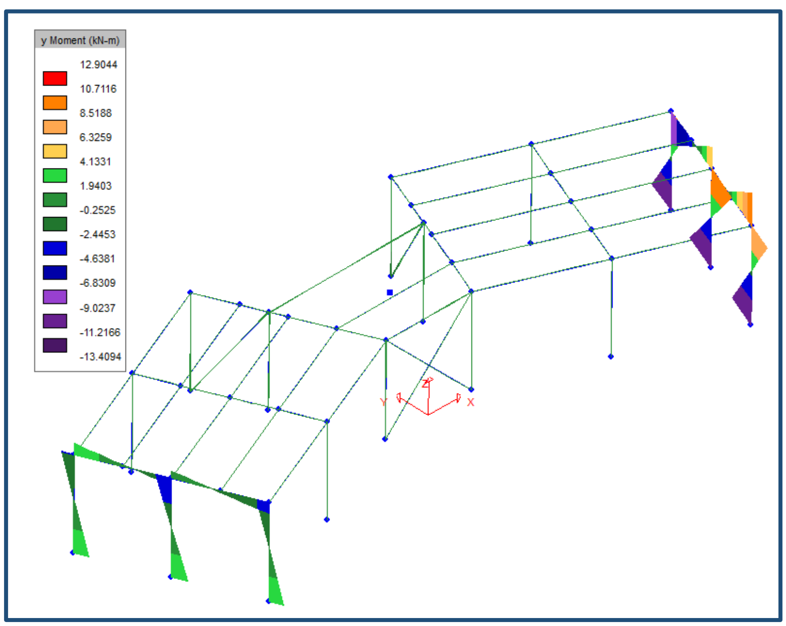


Figure 50.

Using panels

Panel options give you several ways of doing two basic things:

- they allow you to define areas where diaphragm action applies,
- they allow you to define panels for area loads
- they allow you to define areas where you require automatic generation of meshes of plates, shells or membranes.

Defining area loads

New tools have been added to the 3D Geometry and 3D Loads tool boxes and a new panel type Area Load Only has been added to the 3D Panel tool. Working together these options allow you to create loads over large areas of your structure quickly and easily. If you want to add area loads, then you must follow a simple 4 stage procedure:

In the Geometry window:

- ensure that the members you want to load are in the special Area Load Members folder,
- define panels which cover the parts of your structure to which your area loads apply,
- define the direction of the span for each panel you created above,

In the Loads window:

- apply the area loads to the structure.

Example

If you want to work through this example with us, load the model **Area Load Stage 1.TEL**.

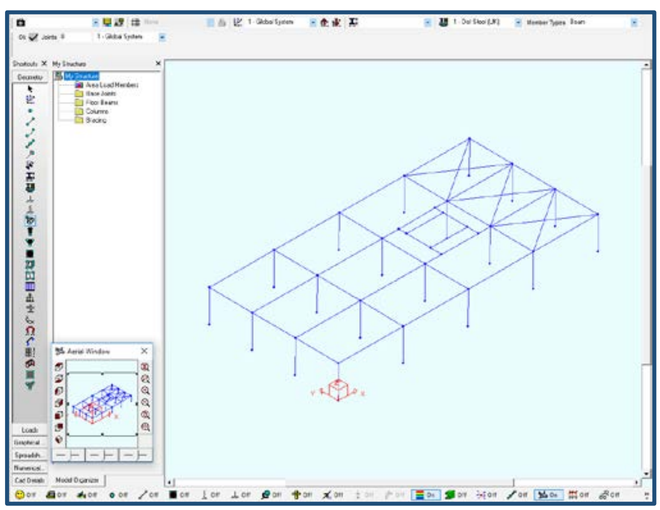


Figure 51.

We shall add a vertical area load to the floor, excluding an area where there is a hole.

To make the example as easy to follow as possible, we shall work in 2D mode and use a plane which only contains the top floor (at an elevation of 3 m) until we come to load our floor.

1. Click the **2D** icon on the **Edit** toolbar, make the following settings, and click OK.

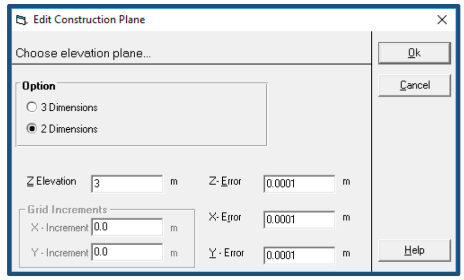


Figure 52.

2. Now we can zoom to the full extent of the structure.

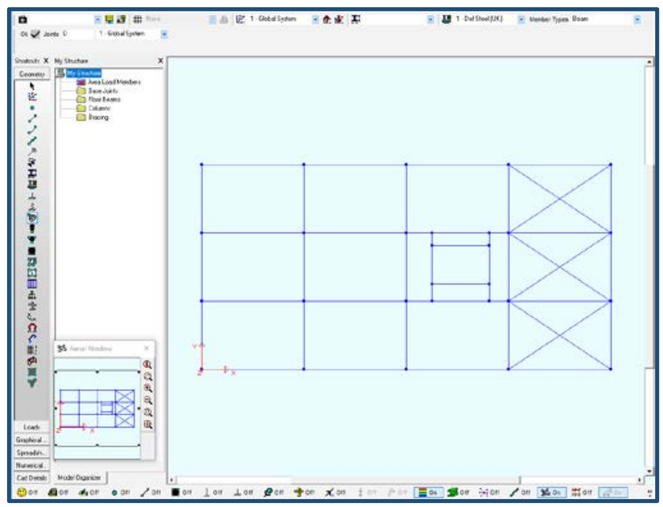


Figure 53.

3. Click the various group folders, and you will see that the Area Load Members folder contains no members, and the Floor Beams and Bracing folders contain the members you would expect.
4. We shall now add the members which are to carry our area loads (that is the Floor Beams) into the Area Load Members folder. Click the Floor Beams folder to open it, then right-click it and choose Copy from the context menu that appears.
5. In the **Copy** dialog, click the Area Load Members folder and then click Ok. This copies all the beams in the Floor Beams folder into the Area Load Members folder. Only members which are in this special folder can carry area loads. This means that the bracings (which are not in the Area Load Members folder) will not take any area load.
6. Select the Panel Element Tool and access its settings dialog using any method.

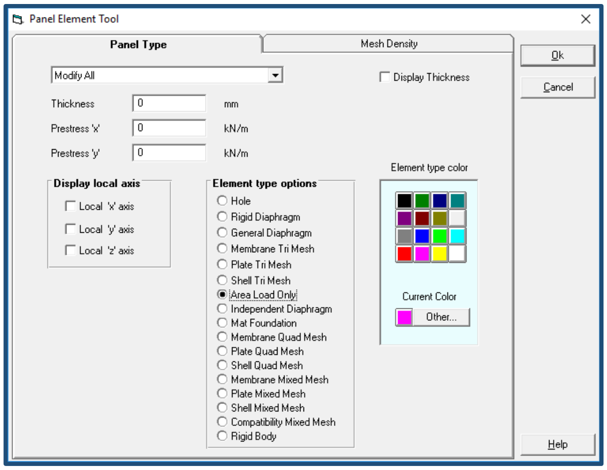


Figure 54.

7. We can specify area loads on any panel type (other than Hole), however, if we choose a panel type other than Area Load Only, this will add stiffness to our structure, and the panel will thus affect the analysis. For our example, we only want to use the panel to apply area loads to our structure. Therefore we select Area Load Only and click Ok.
8. Now we define the panel over the entire floor area by clicking the joints indicated below.

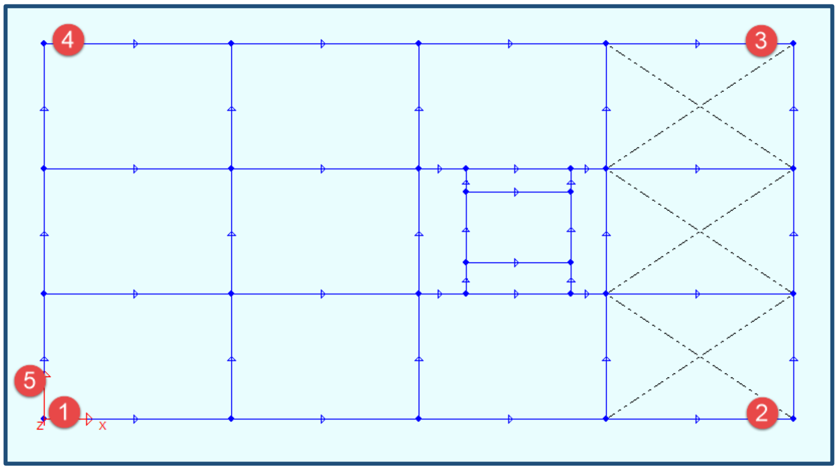


Figure 55.

9. This creates the panel shown below (Shrink Elements is switched On, and so the box denoting the panel does not obscure the beams).

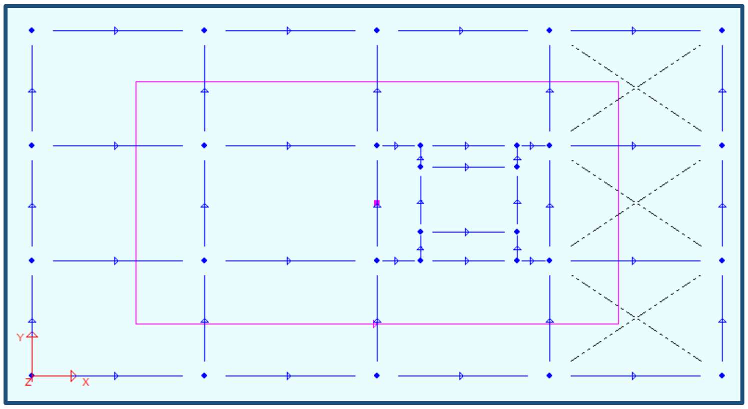


Figure 56.

10. Now we can define the hole, In the **Data Bar** choose Hole from the drop list, and then create the hole in our **Area Load Only** panel, by clicking as shown below.

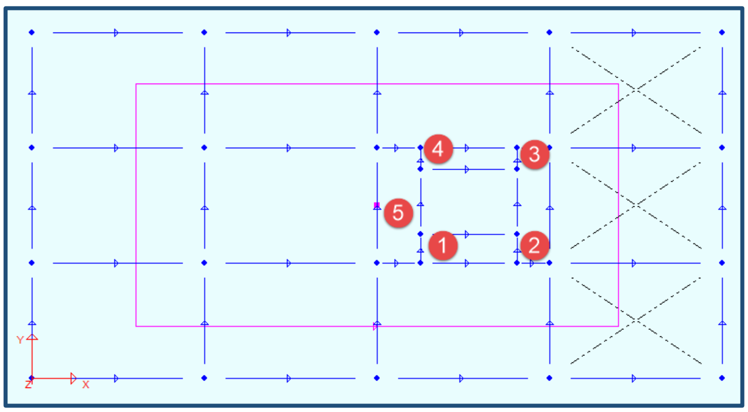


Figure 57.

11. We now have a panel with a hole in it.

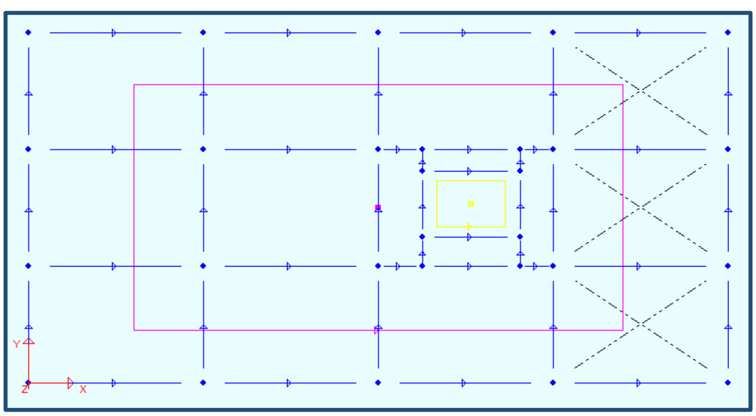


Figure 58.

12. Now we need to choose the way in which we want the load to span. Select the Span Direction Tool and access its settings dialog using any method.

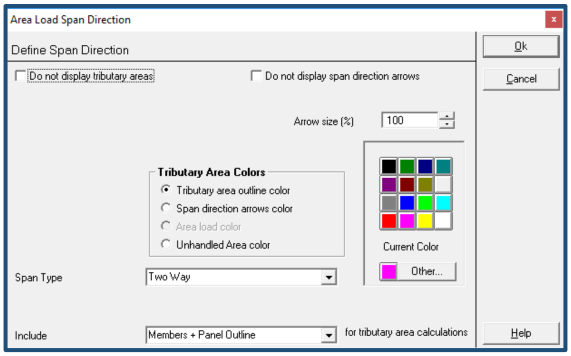


Figure 59.

13. Choose the **Tributary Area Colors** that will enable you to see these clearly and then click **OK** to close the dialog.
14. Now click on the center (square) node for the area load only panel that we created and **S-FRAME** instantly calculates the loading break down pattern for us and automatically takes account of the hole.

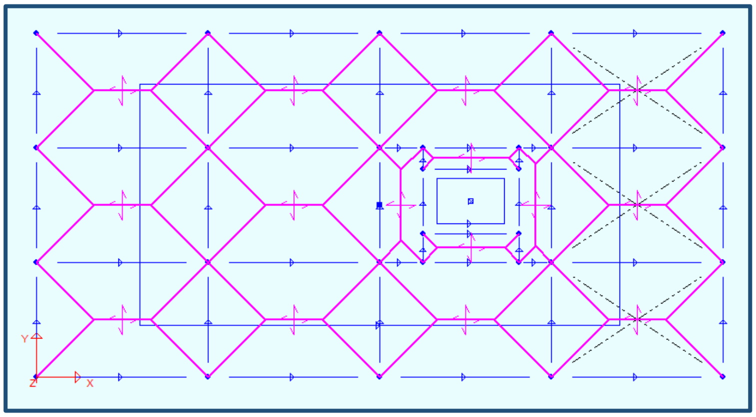


Figure 60.

At this point, you might want to investigate the distribution patterns that you can achieve with the other span direction settings. Simply choose the setting in the **Data Bar**, then click on the center node of the panel twice (the first click removes the existing pattern and the second one creates the new panel). When you have finished, please apply the **Two Way** spanning slab.

15. Switch to the **Loads** window view the structure in an isometric view, switch Shrink Elements On, switch **Display member loads** to **On**, and finally create a new load case titled **Area Load 1**.
16. Use the Selection Tool to ensure that the area load only panel is selected.
17. Now pick the Area Load Tool and define a load of value -5 in the Global Z direction using the Data Bar.

18. Now click the center node of the area load only panel that we created, and you will see that S-FRAME adds the load, shows it at this node and also shows its distribution pattern.

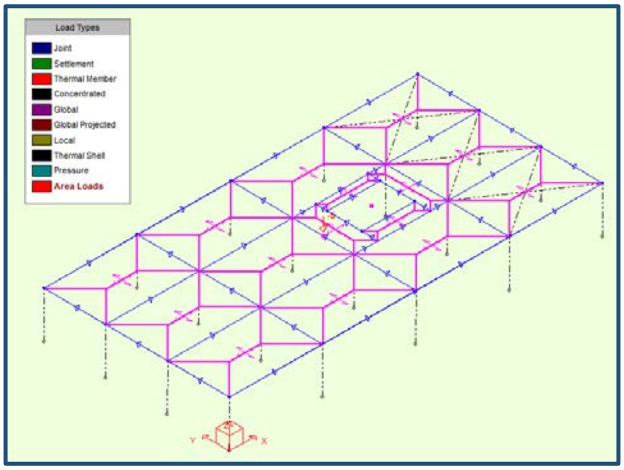


Figure 61.

19. Select the Area Load Tool and access its settings dialog using any method.
20. Check the **Do not display tributary loads**, **Do not display span direction** arrows and **Convert area load to member loads** boxes and then click **OK** to close the dialog.
21. Now click on the center node for the area load only panel again twice. The first time removes the existing area load, and the second click reinstates it. This time, however, you will see that instead of showing the area load and distribution panel S-FRAME instantly calculates the loading break down and applies this to a series of member loads.

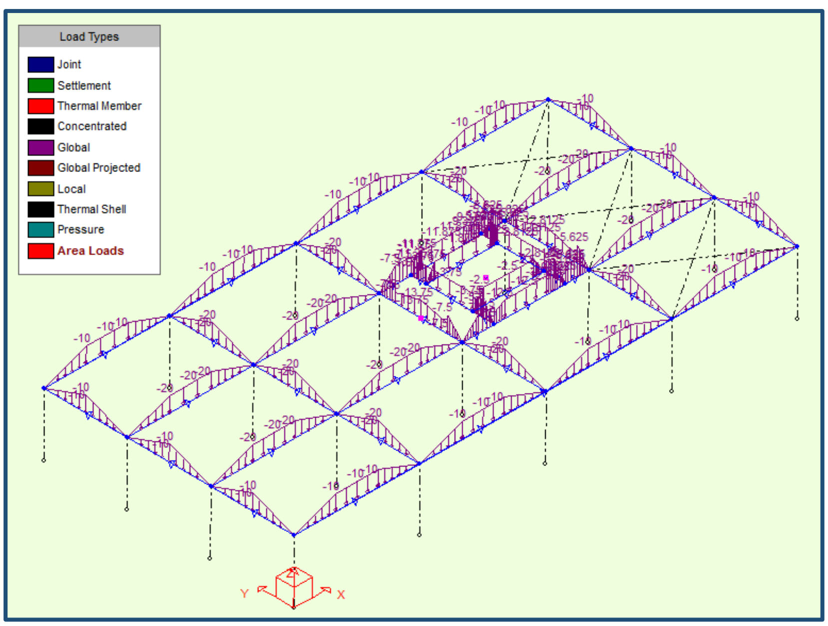


Figure 62.

22. If you zoom into the structure, focusing on the area around the hole, then you will see that the member loads also take full account of the hole.

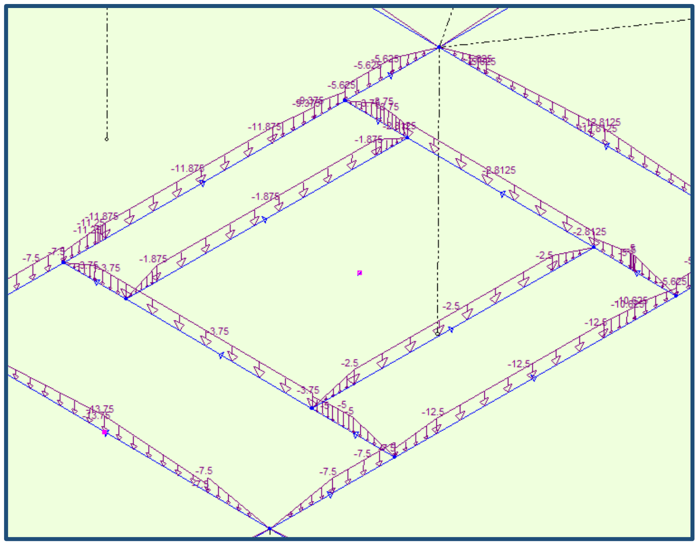


Figure 63.

Connecting overlapping joints (merging structures)

A new command (Connect Overlapping Joints) has been added to the Edit menu when you are in the Geometry window. This command allows you to fuse / merge overlapping joints and members together to create one joint and/or member.

In the simple example below, we move the two structures together.

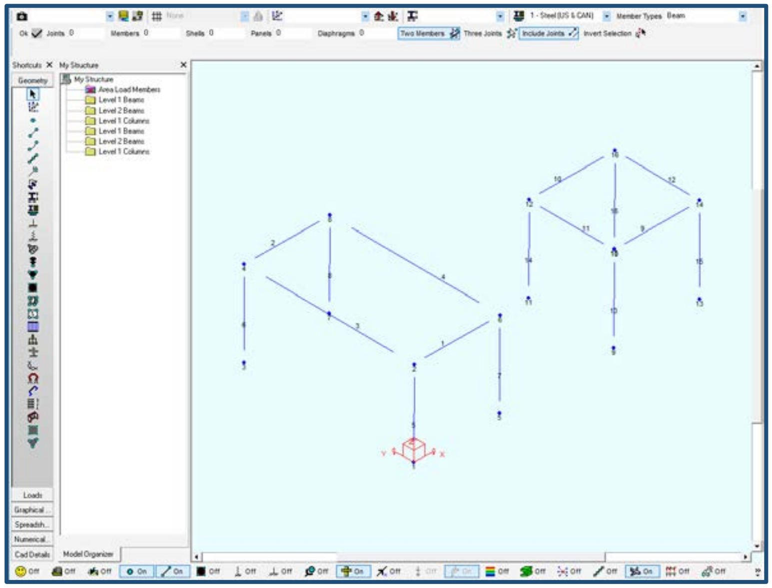


Figure 64.

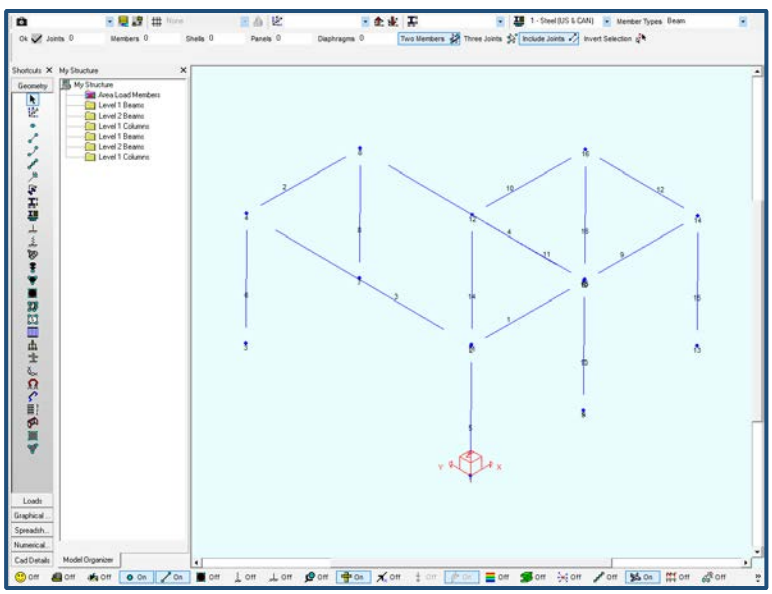


Figure 65.

Undo / Redo is fully supported during this operation.

Click **Edit** > **Analytical Model** > **Connect Overlapping Joints** and **S-FRAME** automatically:

- merges joints 5 and 9 as joint 9,
- merges joints 6 and 10 as joint 6,
- reconnects members 9 and 11 to the merged joint 6 (and renumbers them as members 17 and 18 in the process).
- reconnects member 13 to the merged joints 5 and 6,
- deletes member 13 as it duplicates member 7,
- Leaves members 4 and 18 as they are since they are not duplicates.

These members overlap, but **S-FRAME** does nothing with them. It is up to you to decide the appropriate action to take

This operation only applies to selected objects which you pick using any of **S-FRAME's** selection methods.

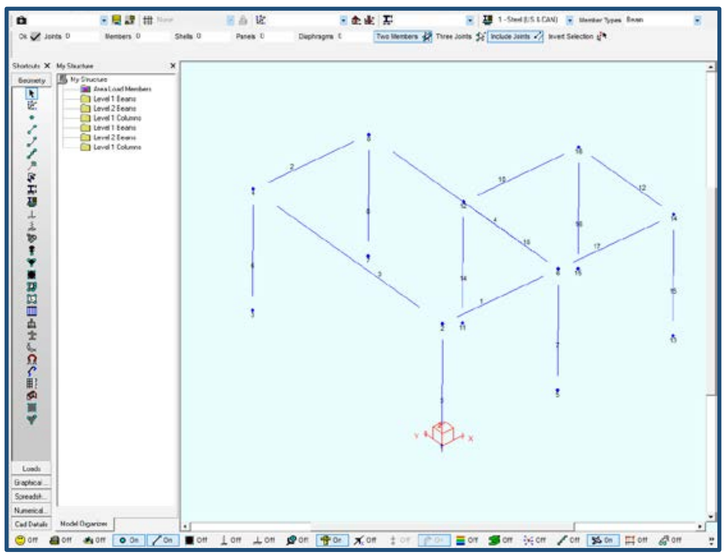


Figure 66.

You can check for overlapping joints and members through **File > Integrity Checks...** (you can specify the tolerance you want the checks to use). You can use the **Integrity Check** options to delete overlapping elements. However, this is not a replacement for the merge option since if you choose to delete overlapping joints **S-FRAME** also deletes any members which are attached to a deleted joint as shown below.

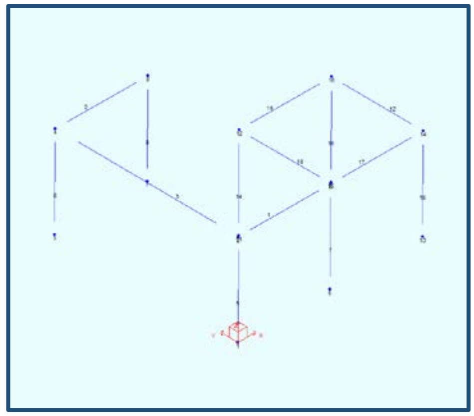


Figure 67.

The key feature of **Merge/Fuse** is that it reconnects members to the retained joint or joints that were merged before identifying and deleting any duplicate members (those that run between the same pair of nodes).

The main limitation of this feature is that loads are not merged. S-FRAME will delete any loads on deleted joints and members. You should always check your model's loading after you use this feature.

Connecting intersecting members

As well as connecting overlapping joints **S-FRAME** also allows you to determine where two members intersect. It will split the members at these points, introduce new nodes as necessary and connect the split members to these newly created nodes.

Undo / Redo is fully supported during this operation.

If you want **S-FRAME** to preserve the loads during element sub-division, then you need to ensure that the **Always preserve loads during element sub-division** box of the **Preferences** dialog is checked. But see the important comment below.

To continue the simple example in the previous section, click **Edit** > **Analytical Model** > **Connect Intersecting Members** or the **Connect Intersecting Members icon** from the **Edit** toolbar. S-FRAME automatically splits member 18 at node 12 – creating member 19, and then removes the duplicate member 4.

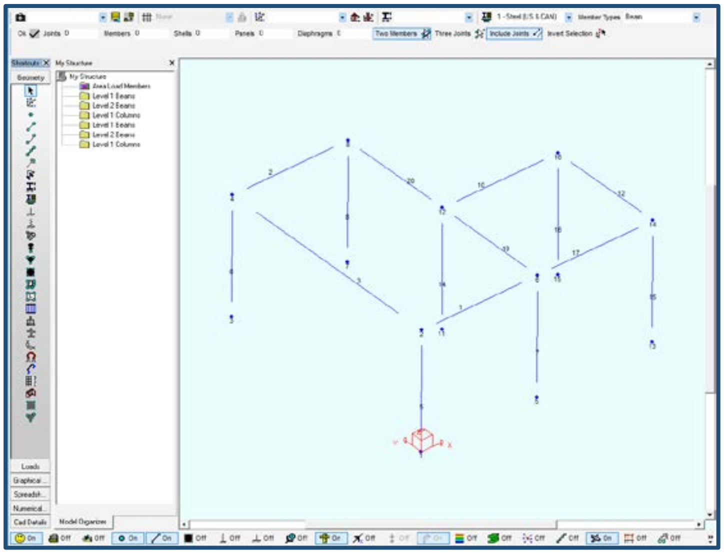


Figure 68.

With the correct **Preferences** settings **S-FRAME** splits the loads on member 18 and applies these to members 18 and 19. However, in this unusual example, the splitting of member 18 also means that it and member 4 are duplicate members. **S-FRAME** recognizes this and automatically deletes member 4, and therefore, any loads it carries will be lost.

S-FRAME may take some time to determine member intersections on a large 3D structure. However, you can cancel the process at any time.

If you are working with a large model, you can restrict the search for intersecting members to a 2-dimensional plane or to a selected subset of members. You can improve performance greatly if you restrict the feature in this way.

Defining Moving Loads

S-FRAME Professional edition allows you to run linear moving load analysis; that is moving load analysis on structures for which a linear static analysis is appropriate.

In **S-FRAME Enterprise** edition you can also run Nonlinear moving load analysis, that is moving load analysis on structures for which a nonlinear static analysis is appropriate, for example, a cable-supported structure.

In either case, you can combine the moving loads with any other static load cases or combinations.

You can use multiple lanes for moving load analysis with different vehicles on each lane. You can configure the vehicles so that they move simultaneously (but at different speeds if you wish) along the different lanes.

The following simplified example (a concrete deck with upstand edges) demonstrates the use of lanes in moving load analysis.

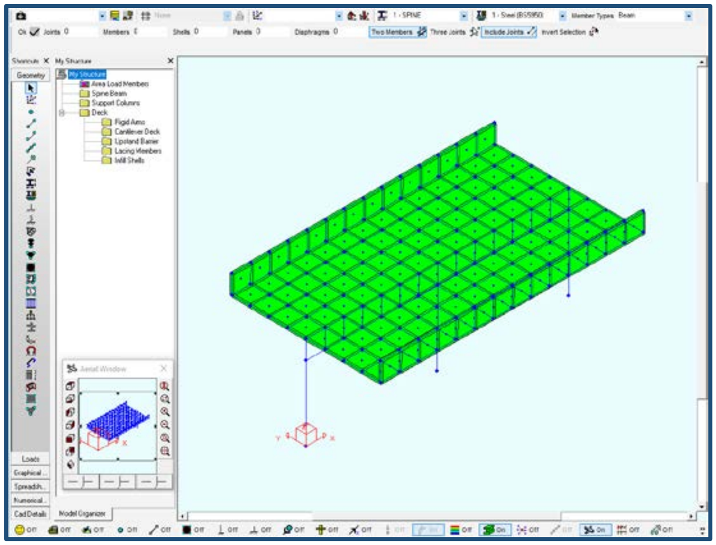


Figure 69.

We have used rendered shells to give a better graphical representation but have used beam elements for detailed modeling purposes. (The shells have very low material properties so that they don't attract load or affect the analysis).

Example

If you want to work through this example with us, then load the model **Moving Load Ex 1 Stage 1**.TEL.

The cross section below shows the basic model.

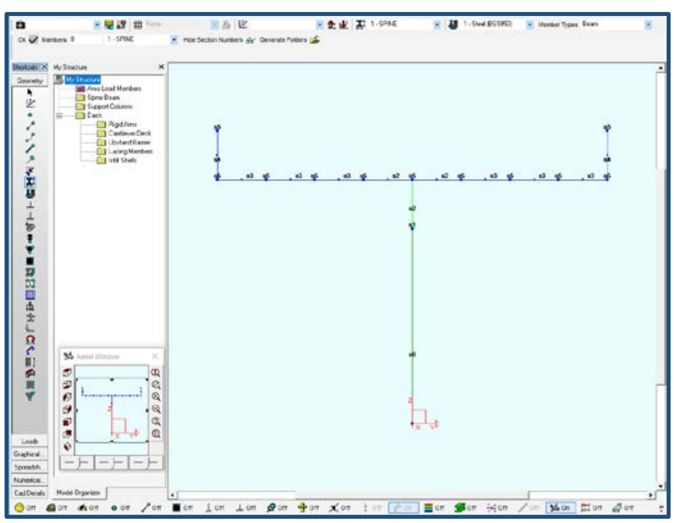


Figure 70.

The model consists of:

- a large longitudinal spine element (**Section 1 – SPINE**) centered on the origin which supports the cross section.
- **Section 2 – RIGID ARMS** is a rigid element which extends to the true physical boundary of the spine.
- **Section 3 – 1M WIDE DECK** is a 1 m width of the cantilever deck, and
- **Section 4 – 1M WIDE BARRIER** is a 1 m width of the upstanding barrier,
- **Section 5 – LACER BEAM** is a beam which allows the load to be dispersed and shared between the 1 m wide slab cantilever elements,
- **Section 6 – SUPP COL** is the columns which support the spine elements.

At this point, we would suggest that you spend a little time familiarizing yourself with the structure and its grouping. When you have done this, switch to the Loads window, and you will find that the model contains a Dead and a Live load case and a single combination of these.

With all this information already defined, we can proceed to define our moving load case.

1. Create a new load case, giving it the name **Moving Load**.

2. Now click the **Moving Load Tool** and then right click it to open its dialog as shown below.

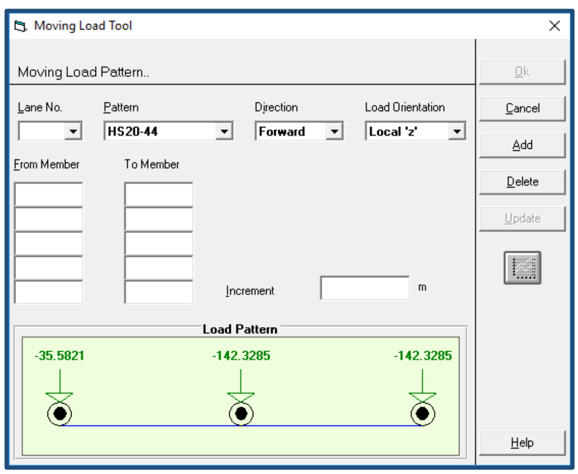


Figure 71.

3. Open the wheel load pattern file **Movedata.DML**. (C:\ProgramData\S-FRAME Software\Data Bases\Vehicle)

If you are not sure how to do this refer to the **Help System.**

You might wish to take this opportunity to review the moving load features using the help system.

4. Now we define our first moving load in lane 1 (running along lacing members 208 to 221).

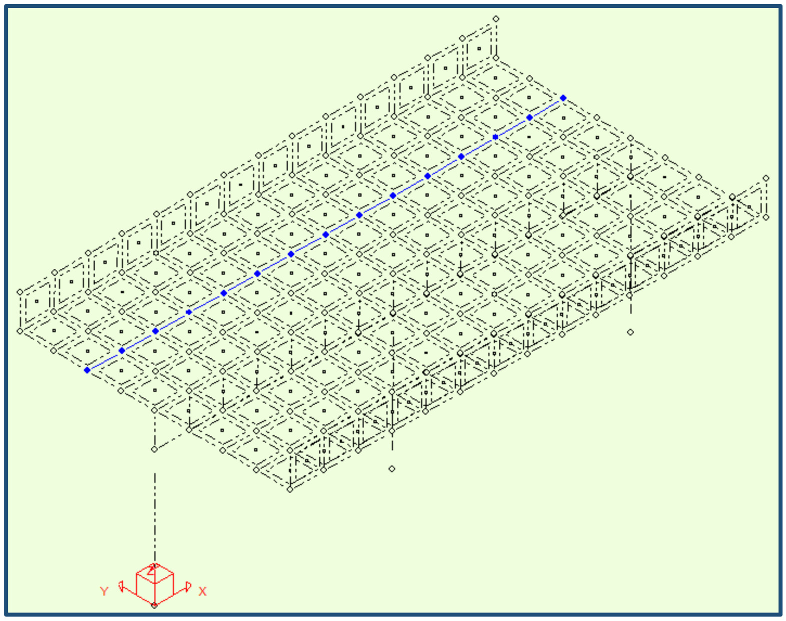


Figure 72.

We constructed the model carefully so that the lacing members are numbered sequentially.

5. Once you have selected and completed all details in the dialog exactly as shown below, click **Add** – you have now defined the loads on lane 1.

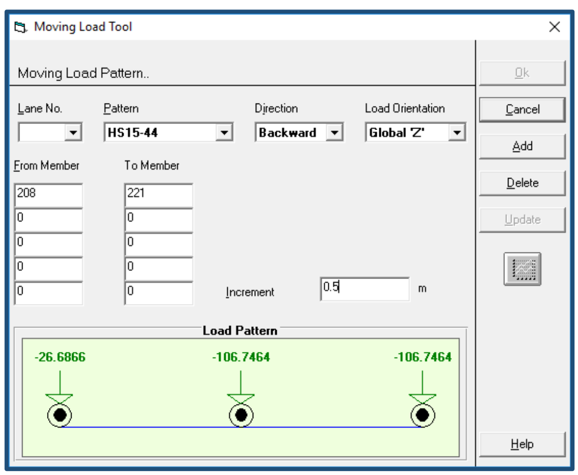


Figure 73.

The Increment defines the steps in which the leading axle moves across the members. If you want to achieve simultaneous lane loading, then you must set the same increment for each lane

6. The moving load in lane 2 runs from lacing member 264 to 277. This is a different load pattern (heavier loads and more axles). Complete all details as shown below and then click **Add** again.

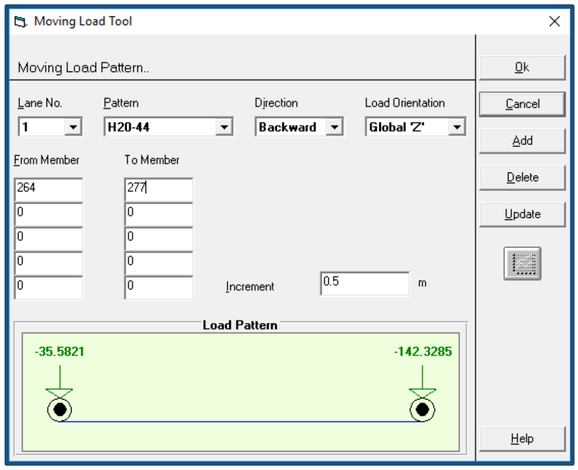


Figure 74.

The dialog now shows the loads in lane 2.

You can use the Lane No. list to select and view the loads in either lane. To edit the loads in either lane, simply select the lane from the list, make the edits and then click Update.

7. Click OK to close the **Moving Load Tool** dialog.
8. Analyze the model by clicking the Analyse icon on the **Edit** toolbar.

9. Make the settings shown below and then click Ok.

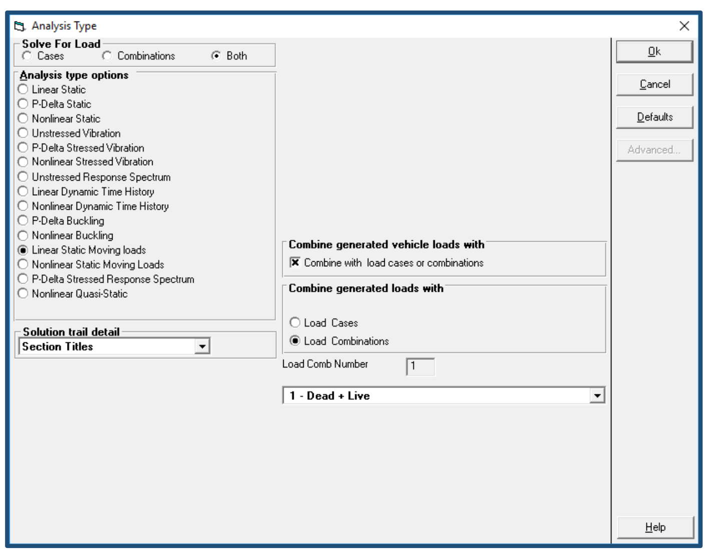


Figure 75.

During the analysis, you will see that S-FRAME analyzes a whole series of additional load cases. These are for each increment of the moving load.

Reviewing moving load results

If you want to work through the remainder of this example with us, then we would suggest that you load the model Moving Load Ex 1 Stage 2.TEL. This should be exactly the same as the one you have just created, but in case you made any other changes inadvertently we suggest that you open the Stage 2 model and work with it.

After analysis, you can step through all the cases generated as the loads move along their defined paths. When viewing load cases a separate case is created for each load step. The picture below shows the fourth step. At this stage, only the leading axles have started along their paths.

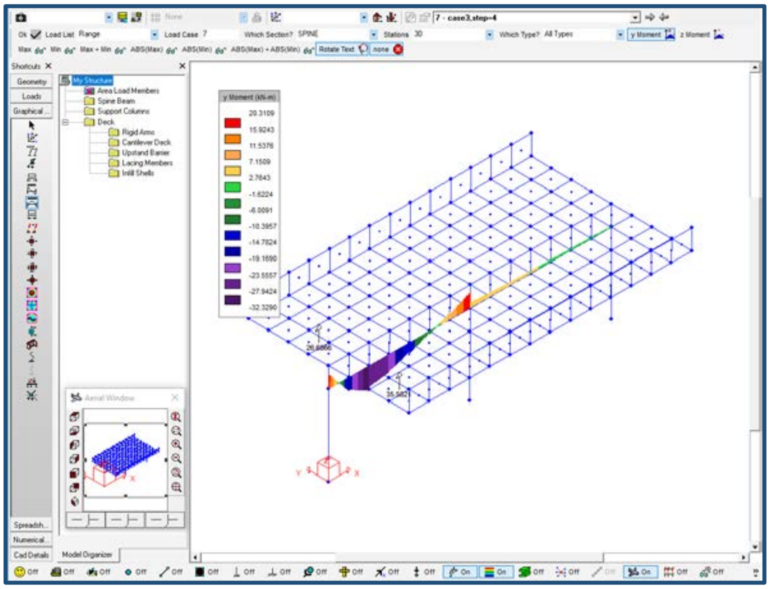


Figure 76.

By picking SPINE from the drop down list, we tell **S-FRAME** that we only want to see these results.

Later, at step 26, all axles of both moving loads are on the deck.

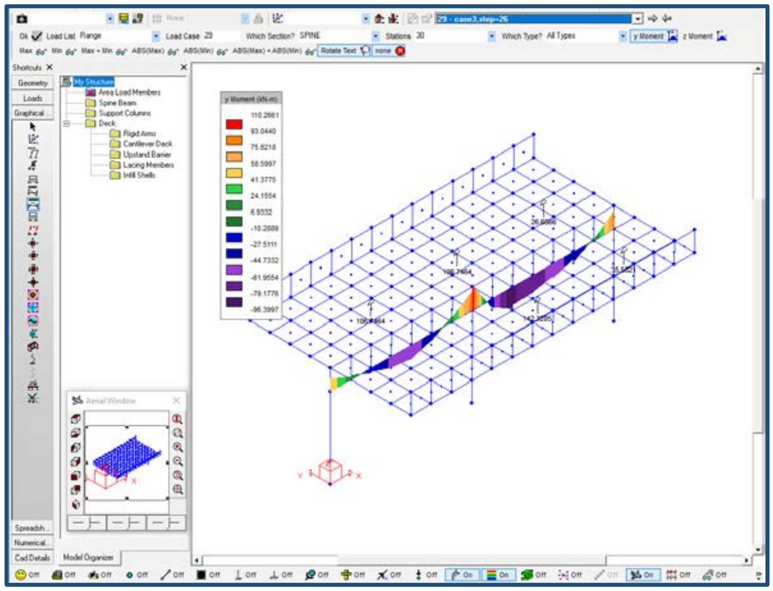


Figure 77.

When viewing load combinations, a separate combination is created for each load step. The picture below shows the 26th step again.

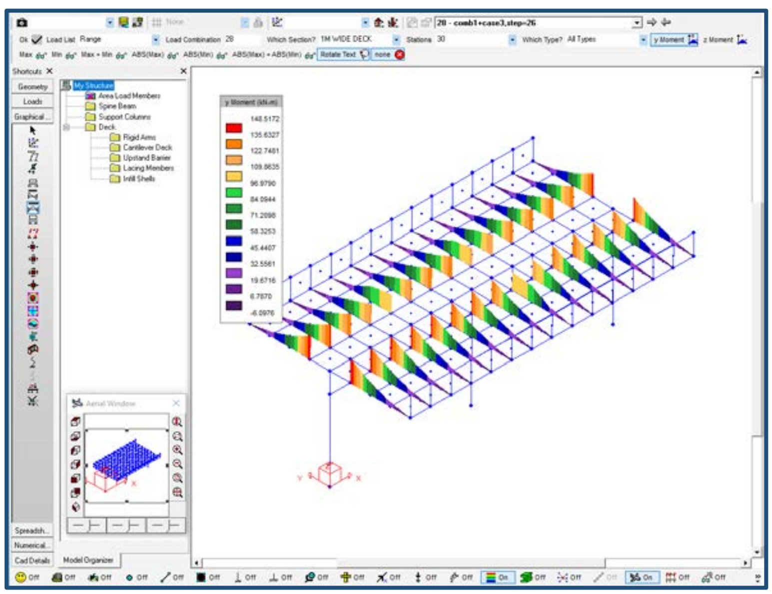


Figure 78.

This time we pick the cantilever deck elements, note how the moment peaks adjacent to the wheel loads.

Although there is no analytical reason for defining shell elements, they provide a visually informative means of viewing the deflections at each step.

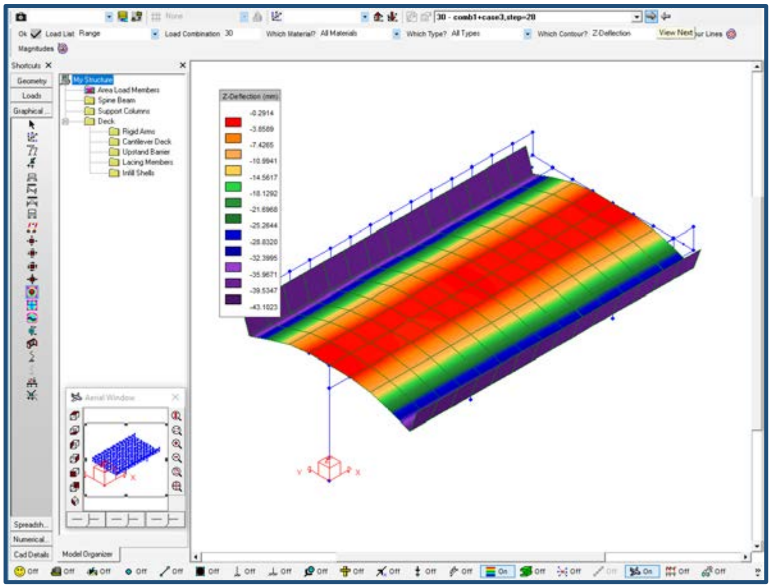


Figure 79.

You can use **S-FRAME** to generate envelope diagrams covering a selected range or all of the moving load steps. The diagram below shows a sectional view of the spine beam has been generated with an envelope diagram and the optional max/ min text.

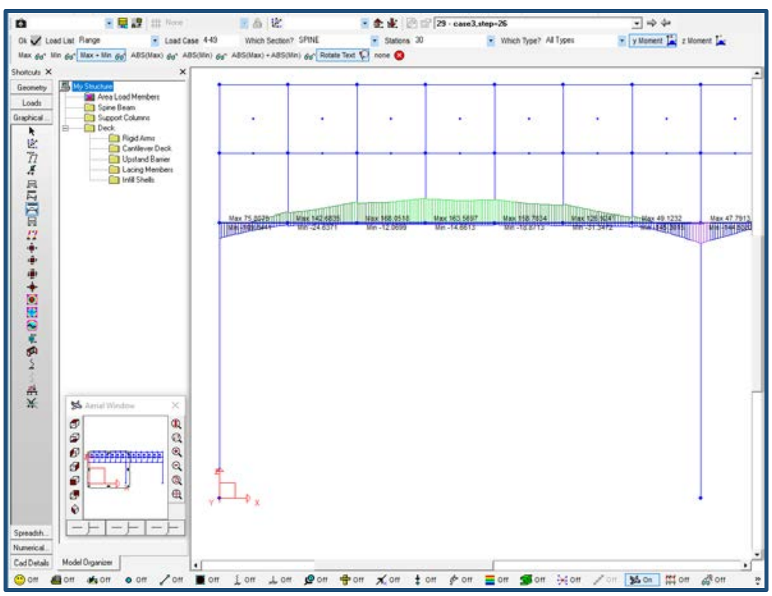


Figure 80.

As noted at the start of this section, **S-FRAME** can also perform a nonlinear analysis of moving loads. **S-FRAME** can combine the results from this analysis with those of other load cases or combinations.

The capture below shows the settings for a Nonlinear Static Moving Loads analysis (which uses the Full Newton-Raphson method).

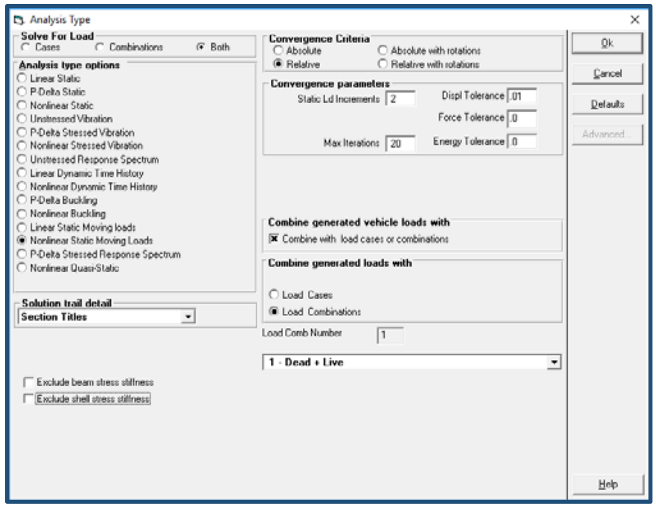


Figure 81.

You must specify various increment and convergence criteria which are appropriate to this analysis type.

The other settings tell S-FRAME to combine the results of the analysis with those of the Dead + Live combination. To achieve this analysis S-FRAME adds the loads for the current step in the unfactored rolling load case to the factored loads from each load case in the combination and then does the analysis for this step.

If you want to factor the rolling load, then you can do this by adjusting the **Load case Scale Factor** in the **Edit Load Status** dialog available from the Edit menu or by using the **Edit Load Status** icon from the **Views and Grids** toolbar. Both of these options are only available in the **Loads** window.

After analysis, you can view the results for load cases or for combinations by picking the appropriate icon (and respectively) from the **Edit** toolbar, and choosing the particular details from the list in the **Views and Grids** toolbar.

This means that design in **S-Steel** and the links to the concrete design programs all now support design for rolling load combinations.

For nonlinear analysis, **S-FRAME** has to solve every load case and combination separately since the principle of superposition does not apply. This means that a nonlinear analysis will take longer than a linear one.

Finally, you can generate influence line diagrams from any unit (single wheel) rolling load case.

If you loaded the model **Moving Load Ex 1 Stage 2**.TEL, then this already contains such a load case called **Point Load**. This load case is inactivated. You can reactivate it using the **Edit Load Status** dialog.

This load case contains a single wheel load which moves along members 208 to 221 (see earlier diagrams) in 0.5 m steps.

The wheel load can be any value you like, 1 kN, 100 kN, 6.357 kN and so on. The results **S-FRAME** calculates relate to this load value; that is they are **not** limited to a load value of 1.

Reviewing and Printing Results for Beam Elements

In the **Graphical Results** window **S-FRAME** can show much detailed information for your structures, be they large or small.

In general, you will find that the screen clutter increases directly proportionally to the amount of information you ask S-FRAME to include. As a guiding principle, we would recommend that you reduce the number of members whose results you require as you request more information for those members. You will find that this principle is covered in some detail in the Tutorial and elsewhere in this Hints and Tips section.

The combination of View and Diagram options and settings gives you incredible flexibility together with the ability to save your favorite settings and to swap and change between them.

Using Shell Contours and Slab Design Contours

When you start working with models containing shell elements, you will soon find it is essential to organize and control your views of the results. The options available allow you to see results for these element types in a meaningful and informative manner.

Using Contours for a Flat Slab

In this example, we introduce guidance on modeling a regular slab, in order to achieve a finite element solution which correlates with an empirical code design solution. While this guidance relates to a specific code and a particular simple regular grid layout, there is no reason why a similar approach would not work equally well for irregular slab layouts, and for different design codes. The particular regular floor layout is shown below.

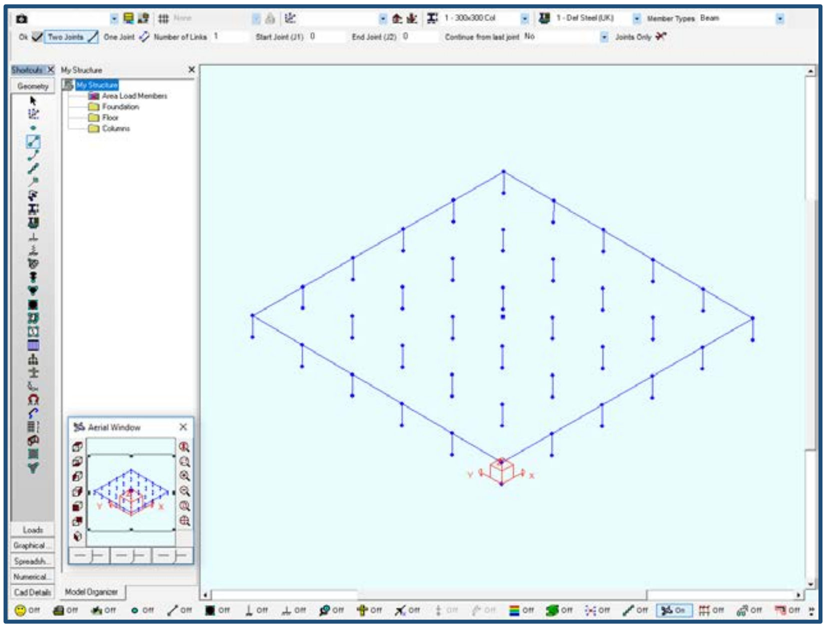


Figure 82.

In industry we see more and more emphasis being placed on modeling flat slabs using a finite element analysis. The reasons for this are many, including:

- the need to provide reduced structural zones,
- the need to incorporate the potential that flat slabs provide for fast formwork erection and construction,
- the need to provide unobstructed service zones,
- the need (perhaps) to be able to handle irregular column layouts.

BS 8110 and CP 65 provide guidance allowing you to idealize and analyze an orthogonal flat slab structure. If you are going to use this guidance you must:

1. be able to identify **column strips** and **middle strips**,
2. create and analyze a sub-frame (as you would for any other beam and column sub-frame model),
3. cater for pattern loads directly or apply moment re-distribution to the results based on the analysis of that load case where all the spans are fully loaded,
4. proportion the resulting forces between the **column strips** and **middle strips** which you identified earlier in an empirical fashion.

For an orthogonal structure, such as that shown below, it is easy to visualize an arrangement of the two strip types.

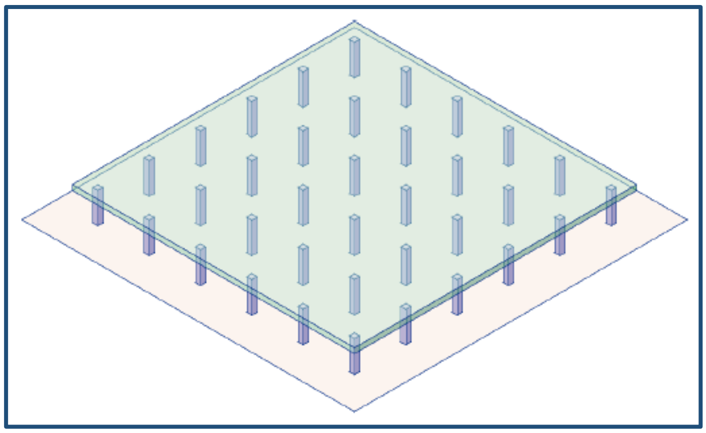


Figure 83.

But for an irregular structure, this is far less straightforward! In the example below there is no obvious orthogonal system of strips.

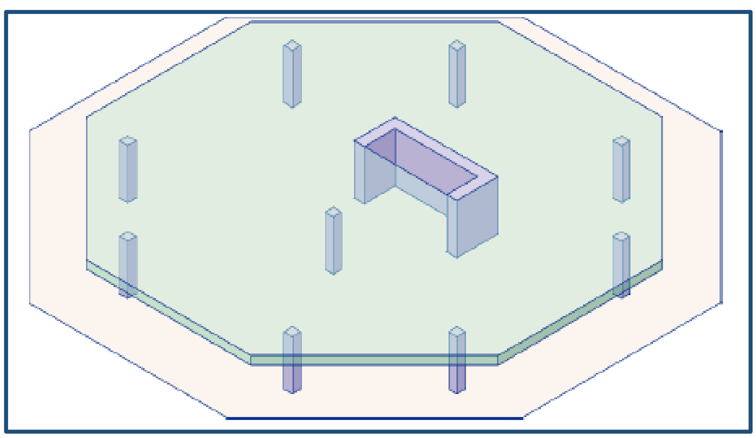


Figure 84.

In fact, for this structure, a system of triangular zones radiating from a core that combined the walls and the column would be more rational. Codes give no way to deal with unusual arrangements such as this in a simplified fashion.

These are the buildings about which we are asked regularly. To generalize:

- virtually no-one shows us a nice regular layout,
- the engineer has usually tried and failed to apply the code's strip idealization in a way with which he is comfortable,
- the engineer, therefore, wants to turn to a finite element package in order to perform a more sophisticated analysis.

However, although a finite element analysis is more sophisticated, it is not a design methodology.

Any finite element analysis will yield results that need to be used carefully, but there appears to be little authoritative guidance on this, and so the engineers we speak to are hoping that we can provide this. We can only offer as guidance the following summary of important aspects of which you need to be aware.

So, where should we start? - Many of the questions we face relate to highly irregular layouts where the results are not as expected – typically the results show:

- high hogging moments, and/or
- small sagging moments.

However, in such cases, there is no strong basis for suggesting what the correct results should be.

So the best place to start has to be with a regular structure where we can compare the expected results based on long-standing and proven code idealizations with the results from our finite element analysis. We shall concentrate on the model shown below:

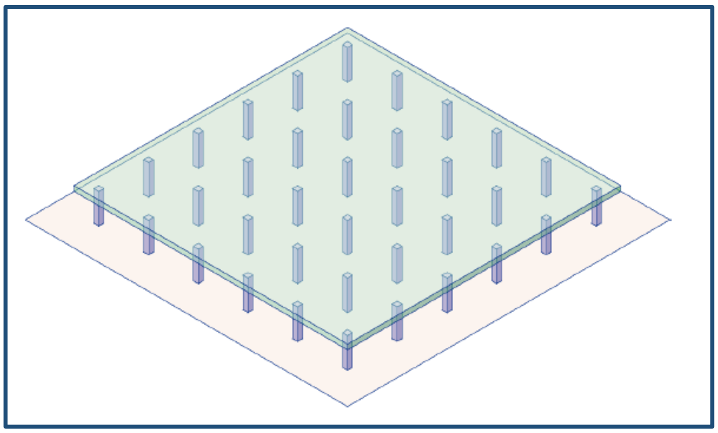


Figure 85.

The details for this regular structure are:

- it is a 5 bay by 5 bay structure
- it has 8 m bays in both directions,
- it is a top floor and thus has columns only below it,
- the column length is 3 m,
- these columns are of a 300 mm square section,

- the slab is 300 mm thick,
- the Dead load (**G**) is = 8.7 kN/m2,
- the Imposed load (**Q**) is 5.0 kN/m2,
- The ultimate load is thus 20.18 kN/m2 (**1.4xG + 1.6xQ**)

The BS 8110's strip method - For a structure as simple as this the code provides alternative simplified methods.

The very simplest method is outlined in clause 3.7.2.7 and allows strip design moments to be determined using the coefficients in table 3.12.

The more complex option requires an idealized frame analysis that is more tailored to the geometry of each specific building. This is the approach we will examine here.

The Analysis Model - Clause 3.7.2.2 indicates that the idealized 2D frame analysis should be based on the gross section properties of the columns and the slab strips (where the strip width is equal to the full panel width – 8 m in this case, 300 mm thick). The strips are loaded with the full slab load.

We can create and analyze this simple frame in **S-FRAME**.

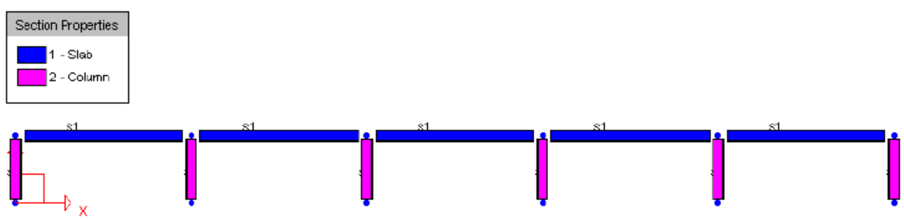


Figure 86.

Basic Analysis Results - The results of a linear elastic analysis for the case with all spans fully loaded are shown below.

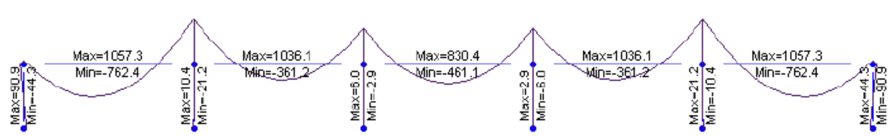


Figure 87.

Since the results are those for a linear elastic analysis clause 3.7.2.1 (which itself refers on to clause 3.5.2.3) applies and you need to make a 20% redistribution of moments from the supports into the spans.

This redistribution is required to account for the pattern loading effects.

The effects of this redistribution are detailed below:

	Support 1	Span 1	Support 2	Span 2	Support 3	Span 3
Moments before redistribution	91	-763	1057	-363	830	-461
Moments after redistribution	73	-877	846	-550	664	-627
Change	-20%	+15%	-20%	+52%	-20	+36

Table 1.

Proportioning of Design Moments in Strips - BS 8110 Table 3.12 indicates how the design moments should be proportioned between column strips and middle strips.

Clause 3.7.3.1 also requires that the central half of the column strip be designed for 2/3 of the hogging moment.

The distribution of design forces in each of the 4 m wide strips is therefore as tabulated below:

	Column Strip	Middle Strip
Maximum Hogging (negative)	50% in 2 m wide central section of strip	25% over full 4 m width of strip.
Reduced Hogging	12.5% in 1 m wide edge sections of strip	
Maximum Sagging (positive)	55% over full 4 m width of strip	45% over full 4 m width of strip

Table 2.

or alternatively representing this graphically.

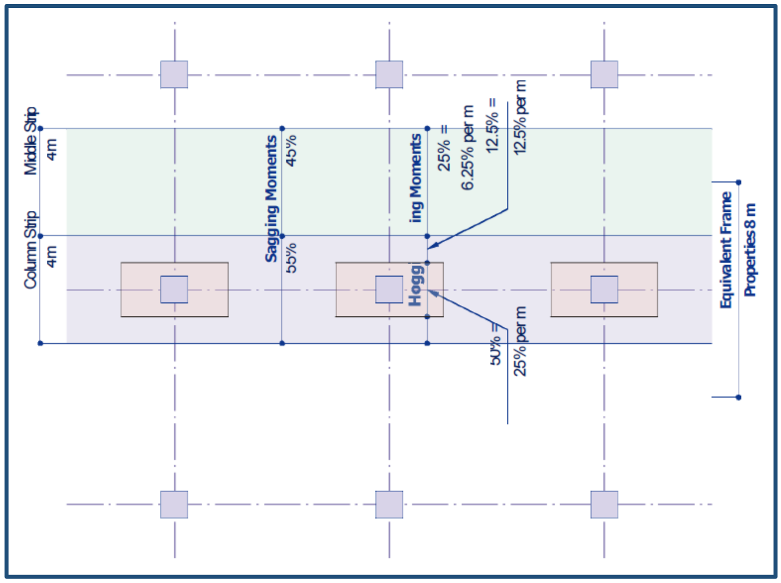


Figure 88.

Proportioning the moments to the strip regions thus gives:

	Support 1	Span 1	Support 2	Span 2	Support 3	Span 3
Moments after redistribution	73 kNm	877 kNm	846 kNm	550 kNm	664 kNm	627 kNm
Column strip centre zone (2 m wide)	Special provisions apply	55% 120 kNm/m	50% 212 kNm/m	55% 76 kNm/m	50% 166 kNm/m	55% 86 kNm/m
Column strip edge zones (2x1 m wide)	0	ditto	25% 106 kNm/m	ditto	25% 83 kNm/m	ditto
Middle strip (4 m wide)	0	45% 99 kNm/m	25% 53 kNm/m	45% 62 kNm/m	25% 42 kNm/m	45% 71 kNm/m

Table 3.

Review - after performing the moment redistribution and proportioning the relative magnitudes of design moments it may be worth recognizing the following relationships.

Sagging Moments - the sagging moments vary from span to span with the highest moments occurring in the end spans.

- For a regular layout like this internal span, moments are restricted to around 70% of the end span moments.
- The moments only vary in the proportion of 45/55 (a difference of 20%) in any given span, (comparing the column strip to the middle strip).
- A possible reinforcing strategy to cater for these sagging moments could, therefore, be to provide something like 70 to 80% of the peak sagging

moment across the entire slab as a general provision. This general provision should be adequate almost everywhere and can be increased locally where necessary.

Hogging Moments - the hogging moments intensify rapidly as you approach the support.

- In the middle strip, the design hogging moment (53 kNm/m) is less than half of the peak sagging moment (120 kNm/m).
- The peak design hogging moment over the column (212 kNm/m) is a little less than double the peak sagging moment (120 kNm/m).
- A possible reinforcing strategy to cater for these hogging moments could be to provide reinforcement for a hogging moment of around half the peak sagging moment across the entire slab as a general provision. Additional reinforcement can then be placed over the columns as necessary.

A finite element approach - In our finite element analysis we are going to build larger, more complex models. We, therefore, need to think about the properties which we are going to use in these.

We need to use appropriate section properties for our members. BS 8110 suggests that you should use consistent properties. However, this may not be appropriate for checking deflections. However, the choice of section properties must be your decision. We would suggest that if you are concerned about deflections, then you should consider this from the beginning. If you model in such a way as to give the slab deflections, then the self-same model should also give the design moments as well.

Let's consider the deflection in more detail. In general, deflection results for flat slabs (or any concrete structures) should be regarded as estimates no matter what type of analysis you use. The unknowns associated with the accuracy of the material properties dictate this. For example BS 8110 indicates that for C40 grade

concrete, E (short term) might lie somewhere in the range 22 kN/mm2 to 34 kN/

mm2.

The stiffness of your members also needs to be modified to allow for creep and cracking. CIRIA Report 110, Design of Reinforced Concrete Flat Slabs to BS 8110 gives some guidance on adjustments that may be appropriate to allow for both creep and cracking. In simple terms, this suggests reducing E (short term) by half to two-thirds to allow for creep and also reducing I by half to allow for cracking. This means that the stiffness (EI) of your members should be reduced by a factor of between 4 and 6. Thus an overall stiffness adjustment factor of between 0.16 and 0.25 would be applicable.

We can adjust the stiffness of our members by only adjusting the value of E, so for

C40 grade concrete an E-value anywhere in the range of (0.16x 22kN/mm2) =3.5 kN/mm2 to (.25x34 kN/mm2) =8.2 kN/mm2 might be appropriate.

Some companies have internal guidelines that their engineers adhere to in this regard.

If you want to review this subject further, you will find that the reports on the results of the Cardington Institute Concrete Building are of interest. We would particularly recommend the papers by Messrs Hossain and Vollum, in particular, the paper Prediction of Slab Deflections and validation against Cardington.

These studies indicate that propping and in particular the timing of prop removal have an important influence on slab deflections.

In the absence of better data, this work appears to conclude that long-term deflections can be estimated on the basis of a linear static analysis with an adjusted **EI** value. Suggested adjustments are towards the bottom end of the **0.16** to **0.25** range. If a slower pace of construction was anticipated, or if construction were more controlled it is feasible that higher multipliers could be justified.

The deflections that you review should be for the combined dead and imposed loading case under serviceability loading.

To do this, you could create a separate un-factored combination, or, you could review deflections based on the factored combination. If you adopt this second approach, then you need to account for the load factors in this combination. To do this, you need to increase the stiffness adjustment factors by a factor of between 1.4 and 1.6.

Hence in the following example, we will use an adjusted **E** value of **7 kN/mm^2** for the slab. This value is on the low side of the figures discussed above.

For the columns, we will use an uncracked long-term modulus of **14 kN/mm^2**.

Example

If you want to work through this example with us, then load the model **Flat Slab Model Stage 1.**TEL.

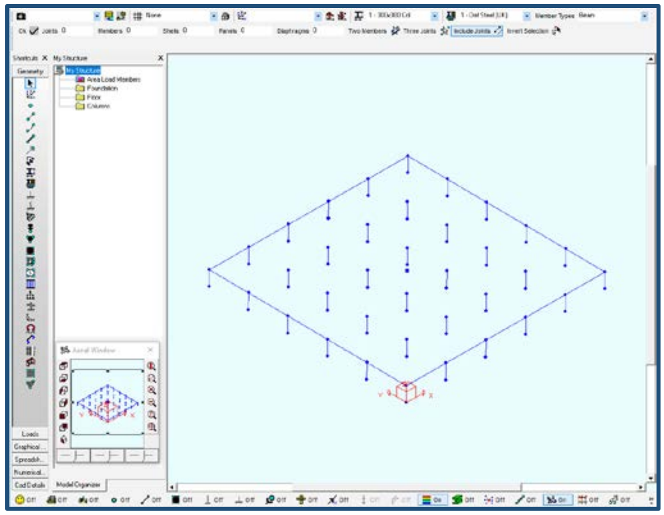


Figure 89.

At this stage, the columns have been defined, as has the flat slab panel. To create our shells we need to mesh this model.

Meshing - By creating a panel sub-dividing the meshed shells we can create a fine mesh very quickly. Generally, we would advise you to try to ensure that there are at least 6 segments between adjacent column/support positions. If you are in doubt, we would always advise that you mesh a little more finely.

1. Click **Mesh** > **Mesh Panel Elements** or click the **Mesh Panel Elements Tool** ( ). **S-FRAME** meshes the panel as shown below.

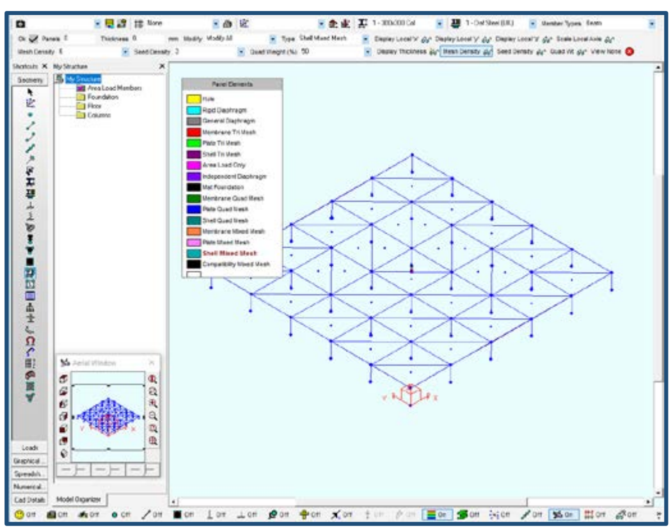


Figure 90.

This mesh is far too coarse; we, therefore, need to subdivide the shells to create a mesh which is finer.

2. Click **Edit > Subdivide...** or click the **Subdivide Selected Elements Tool** ( ).Make the settings shown below and then click OK.

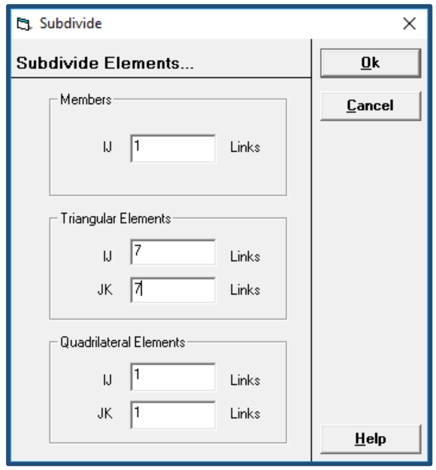


Figure 91.

The resulting fine mesh (shown in a zoomed view below and with Shrink Elements switched On and with Display local 'x' axis switched Off) is now suitable.

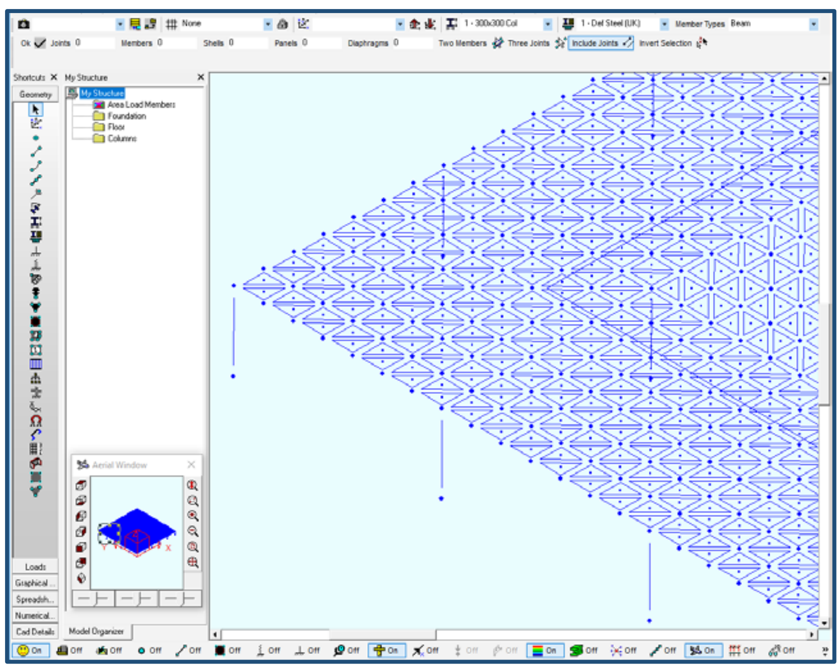


Figure 92.

Now we can proceed to load our structure. Access the **Loads** window and apply a pressure load of **-8.7 kN/m2** in the Global Z direction into the Dead load case and a similar load of value **-5.0 kN/m2** into the Live load case.

If you want to pick up the model at this stage, then load the model Flat Slab Model Stage 2.TEL

3. Click the **Analyze** () icon to perform a linear static analysis on this model.

Once the analysis is complete and you have closed the **Solution Summary** dialog you will see the **Graphical Results** window.

4. Since we are interested in the shells, the most intuitive way to view the results is to use the **Shell Contours Tool** (), pick **Z-Deflection** from the list of available contours and click **Accept Data** to see a contour map of the Z-deflections.

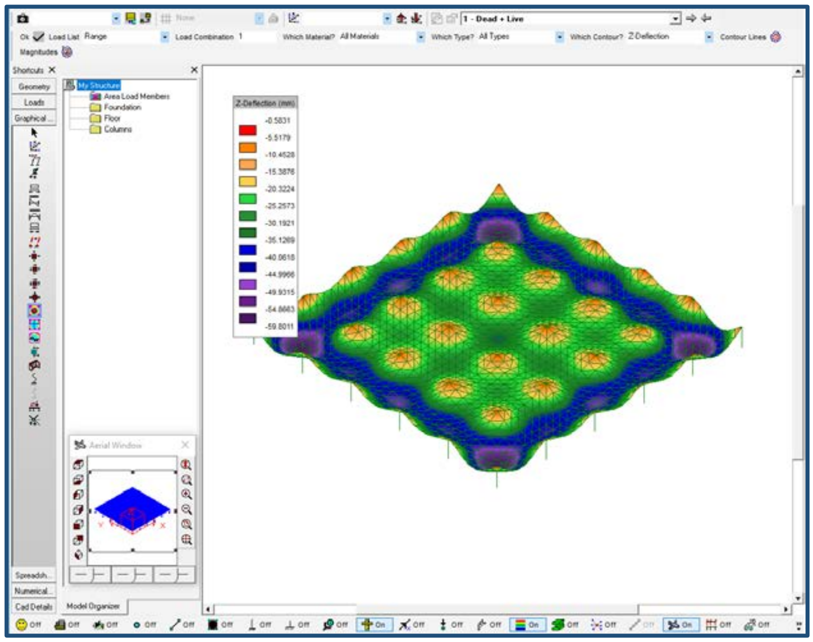


Figure 93.

This analysis' results give us an **estimated** long-term deflection **around** 60 mm, but remember that adjusting **E** for the slab concrete will have an almost directly proportional effect on this result. In addition, creating a more detailed column-head model could also have an impact on deflections.

If your display does not look like this, then click **Settings >Diagrams...** and make the settings shown below before clicking OK.

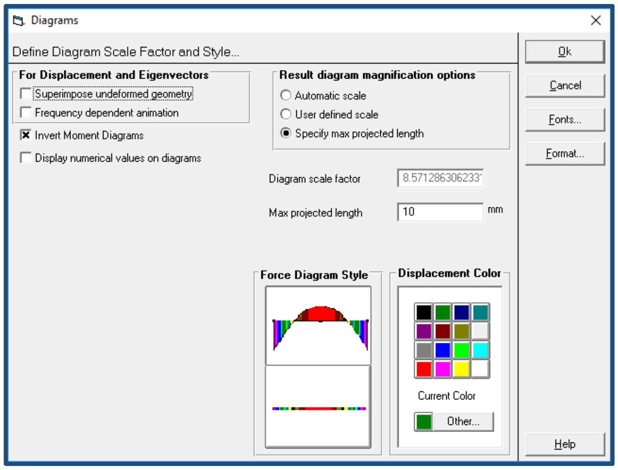


Figure 94.

Next right click the title of the **Legend** and then pick **Properties...** from the context menu. Make the following settings in the **Contour Diagram Factors** dialog.

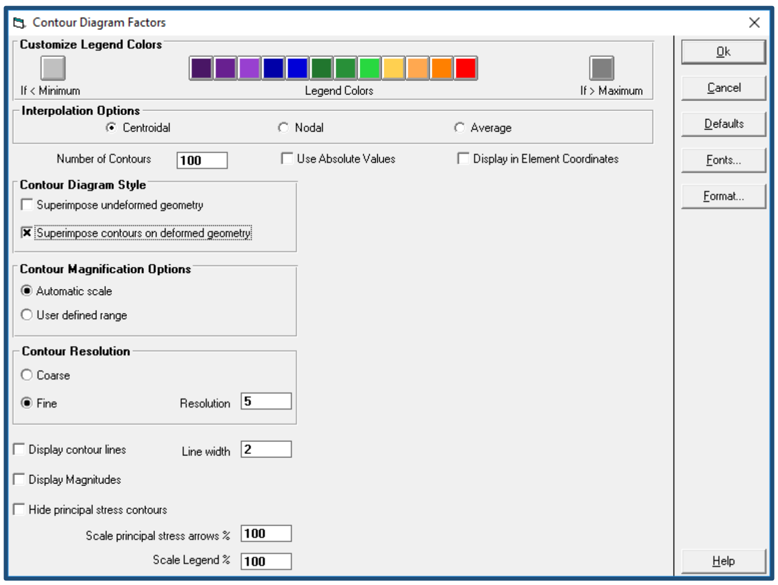


Figure 95.

For the remainder of this example check the Superimpose undeformed geometry option and uncheck the Superimpose contours on deformed geometry options in the **Contour Diagram Factors** dialog. Your screen should now look like that below.

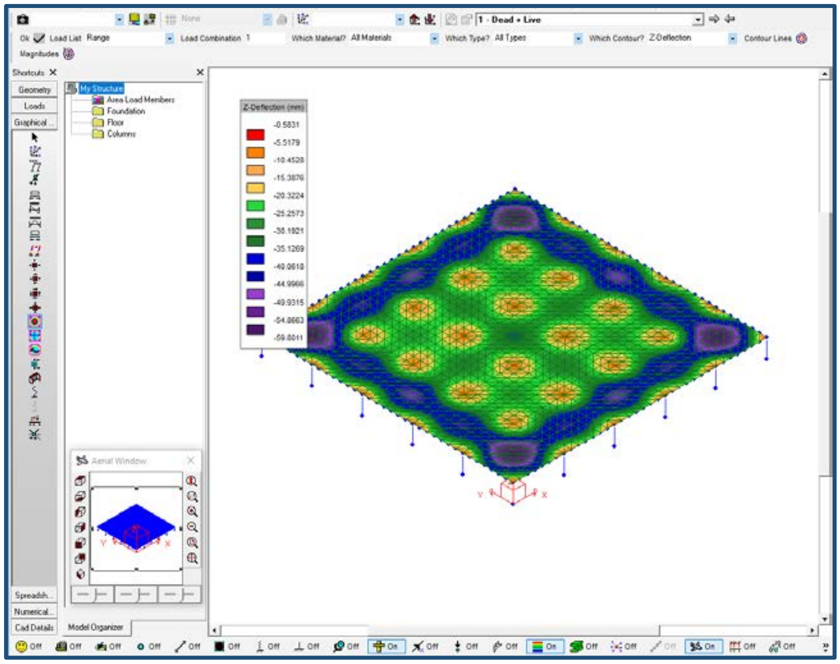


Figure 96.

5. Now pick the option to display Mx contours from the list of available contours, and then view your structure from the top. You should now see the display below.

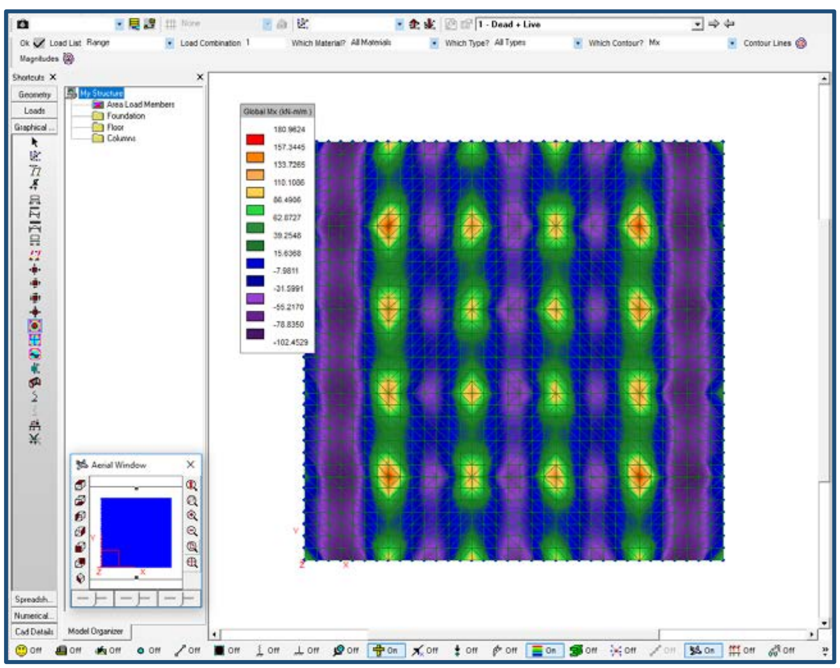


Figure 97.

The maximum (centroidal) Mx moment is +181 kNm and the minimum Mx moment -102 kNm.

The maximum (nodal) Mx moment is +424 kNm and the minimum Mx moment -109 kNm.

Interpretation Options - 3 of these options are provided in the Contour Diagram Factors dialog (see the previous capture) and Concrete Steel Area Contours Design Parameters.

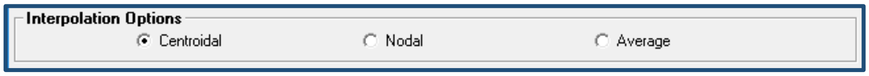


Figure 98.

In this model the peak design hogging moments vary as follows:

- Centroidal — **181 kNm**
- Average — **303 kNm**
- Nodal — **424 kNm**

This range relates to the variation of nodal forces that you see for each shell. Typically shells attached to a column head will have a wide variation (but this is dependent on the mesh density at that column head).

Row No	Shell No	Ld Comb No	Joint No	Fx kN	Fy kN	Fz kN	Mx kN-m	My kN-m	Mz kN-m
2549	638	1	454	-9.0536	2.3983	-193.7325	-30.8243	6.7520	0.0000
2550			52	11.4519	-11.4082	400.5629	-202.2025	-202.2520	0.0000
2551			203	-2.3983	9.0099	-193.6515	6.6902	-30.9291	0.0000
2552				0.0000	0.0000	0.0000	0.0000	0.0000	0.0000

Figure 99.

For the **Centroidal** option, the contouring is based on the centroidal forces for all shells around the column head node.

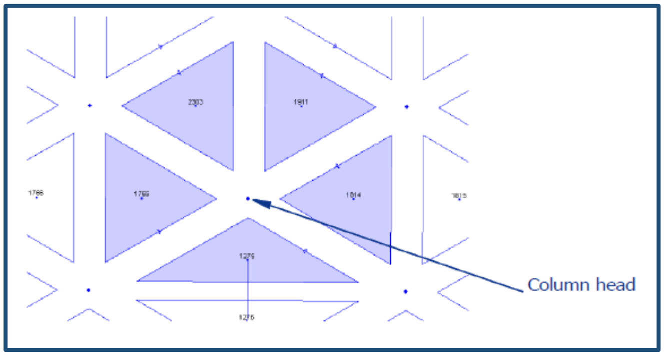


Figure 100.

For the **Nodal** option, the contouring is based on the nodal force at the column head node for all shells around the column head node.

The Average option is the sum of the centroidal and nodal values divided by two.

As well as these direct deflections, forces and moments **S-FRAME** can also produce contours of information appropriate for the concrete design. Click the Slab Design Contours Tool ( ). You will then see a display of the contours for the Mx Top Wood-Armer.

The maximum Mx moment +187 kNm (centroidal), 432 kNm (nodal).

You will see that there are options for contouring Mx Wood-Armer Bottom, Mx Wood-Armer Top, My Wood-Armer Bottom, X-Steel Top, X-Steel Bottom, Y-Steel Top and Y-Steel Bottom.

Wood and Armer Moment Adjustments - In this example (and for most reasonably regular models) the reinforcement will tend to be aligned with the principle stress directions and Mxy twisting moments tend to be low where the peak hogging and sagging occurs. In such cases, Wood and Armer adjustments are not very significant. However, for irregular models, it is important that you take account of these adjustments.

If you switch back to the **Shell Contours Tool**, pick one of the **S/Sxy Max/Min Top/Mid/Btm** options, then access the **Contour Diagram Options** dialog and check the option to **Hide principle stress contours**, then you can see arrows representing the directions in which these stresses run for each shell. In some complex models, this may help you to decide on the best reinforcement orientation.

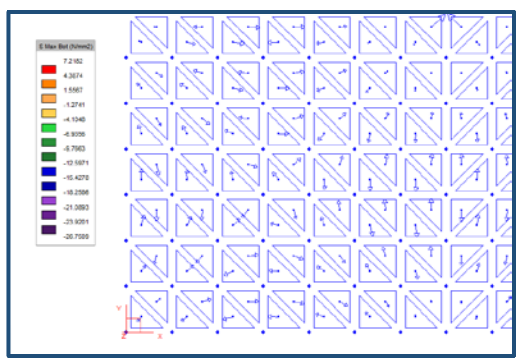


Figure 101.

If you have changed to the **Shell Contours Tool**, then don't forget to return to the **Slab Design Contours Tool** and choose the option to display **Mx Bot Wood Armer** moments before proceeding.

Sagging Moments – Comparison & Adjustment- The code approach for this model suggests a peak sagging moment of **120 kNm/m** after re-distribution.

The moment before re-distribution would have been.

$$\frac{763}{877}x120 = 104\ kNm/m$$

Finite element analysis shows results ranging from **105 kNm/m** (centroidal) to **115 kNm/m** (nodal) before any accounting for re-distribution. Therefore we see an excellent correlation between finite element and traditional idealized strip analysis. This leaves the question "do you still need to do something to cater for pattern loading." Some would suggest that this is not necessary. However, according to the code we probably do need to cater for this (especially if you are going to design for the peak nodal results which expose local peaks without averaging them into strips). Moment redistribution is not an option, so we suggest that you consider a simpler approach and perhaps simply amplify the design sagging moments by something between 10 and 20%.

Hogging Moments – Comparison & Adjustment- The code approach for this model suggests a peak hogging moment of **212 kNm/m** after re-distribution.

The moment before re-distribution would have been.

$$\frac{212}{0.8} = 265\ kNm/m$$

The code requires you to reinforce for peak moments as though they are constant over quite significant strip widths. Any finite element analysis will expose more intense peaks that occur over shorter distances. **S-FRAME** provides options which allow you to interpolate across a width of the slab, and thus iron out these peaks. This is in effect an averaging option but the zones over which the results are averaged relate to the mesh density. In this example the three options show peak design hogging moments varying from:

- Centroidal — 187 kNm/m
- Average — 310 kNm/m
- Nodal — 432 kNm/m

You need to change to the display of **Mx Top Wood Armer** moments to expose these values.

If you were able to take the time to sub-divide all the shells in this model so that there a much finer mesh (14 segments between column heads) then the results change as follows:

- Centroidal — 291 kNm/m
- Average — 413 kNm/m
- Nodal — 535 kNm/m

The more finely you mesh, the higher are the peak values that are exposed.

- With 7 nodes between columns, the peak nodal moment is effectively an average over a length of: $\frac{1}{7}\text{x}\ 8 = 1.14\ m$
- With 14 nodes between columns, the peak nodal moment is effectively an average over a length of: $\frac{1}{14}\text{x}\ 8 = 0.57\ m$

Since these relate to the mesh density (node spacing), they become ever more irrelevant in terms of the code's advice that you provide uniform reinforcement over a central strip width equal to 1/4 of the column spacing (in this case a 2 m wide strip). What you really need to be able to do is define a strip across a column head and have the average results reported to you on the basis of the strip width you choose. **S-FRAME** allows you to do just this.

When viewing any contour diagram you can cut a strip simply by clicking on two nodes to define the start and end of the line, and then a third node to set the plane of interest.

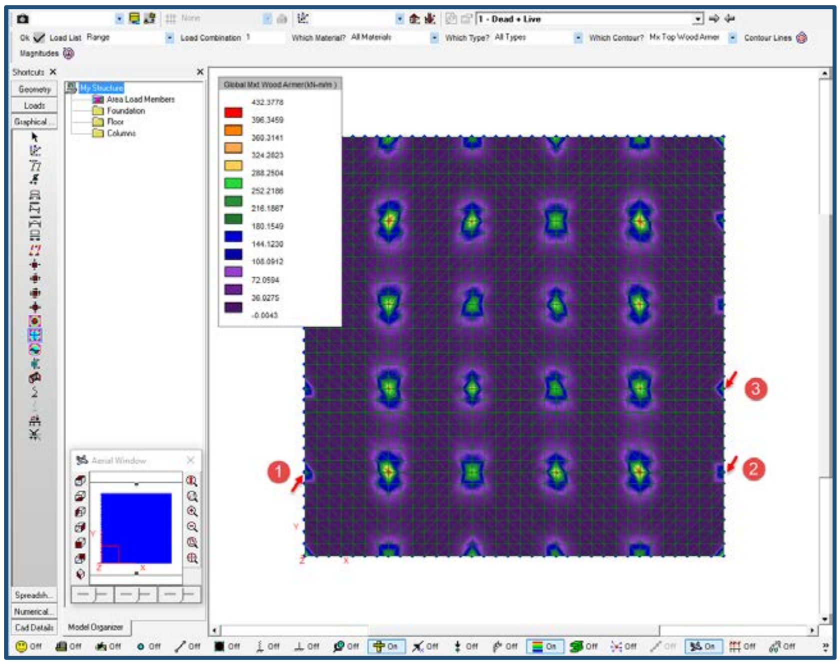


Figure 102.

When you are using this option, we would recommend that you work on a 2D plane. To do this move the user coordinate system so that its XY plane lies along the plane in which you are interested. Click the **Views** icon ( ) and check the **Work only with objects that fall within XY plane of current UCS +/- Delta Z** option. If you do not do this, then you will find it impossible to define the strip if the nodes you need to pick to define the strip are coincident with other out-of-plane nodes in your current view.

Below are the results for the above strip. This strip runs across the column heads and has the potential to expose the peak hogging moments.

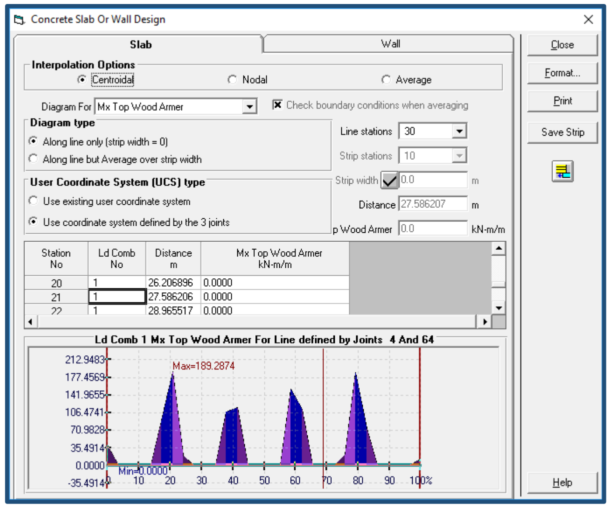


Figure 103.

The **Diagram Type** is set to **Along Line Only** which means that the values shown are those directly along the line which you identified by your first two points.

Furthermore, the number of **Line Stations** is taken along the entire line. **S-FRAME** only determines the moments at these stations, and not at any intermediate points. If you want greater accuracy, then you might consider increasing the number of stations and dealing with your strip in smaller lengths (for example between adjacent column heads).

The capture below shows the difference which specifying a different number of Line Sections makes.

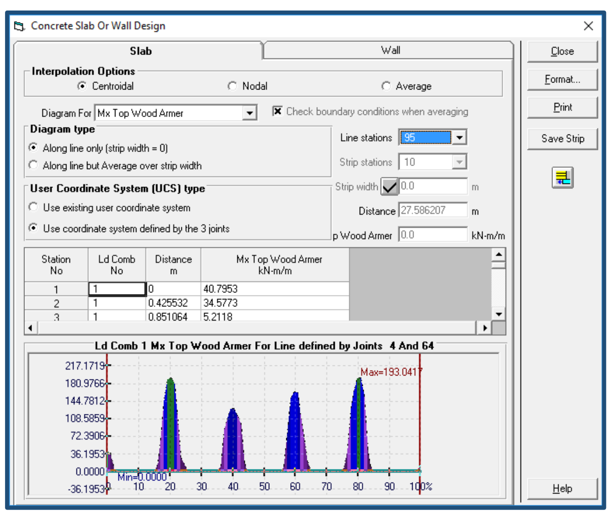


Figure 104.

In practice, if you are going to look at long strips you should take the time to ensure that the number of **Line Stations** which you specify does pick up the worst points along your line.

If you set the diagram type to Along Line but integrate over strip then you can enter the width of the strip in which you are interested and **S-FRAME** reports the total (integrated) moment over the specified width – in the capture below this is 2 m.

Hence the design moment based on this strip width is **319 kNm/m**. This still seems somewhat high, and we need to ask why? The main reason is probably related as much to the modeling as to the methods of exposing and interpreting the results.

It would make sense to create a more sophisticated model with rigid zones extending from the idealized column centre-line to their faces. Once you have done this, you can use panel meshing as before.

If you want to pick up the model at this stage, then load the model Flat Slab Model Stage 3.TEL. This model has been created using the solution proposed above and with extra meshing to cater for these rigid arms.

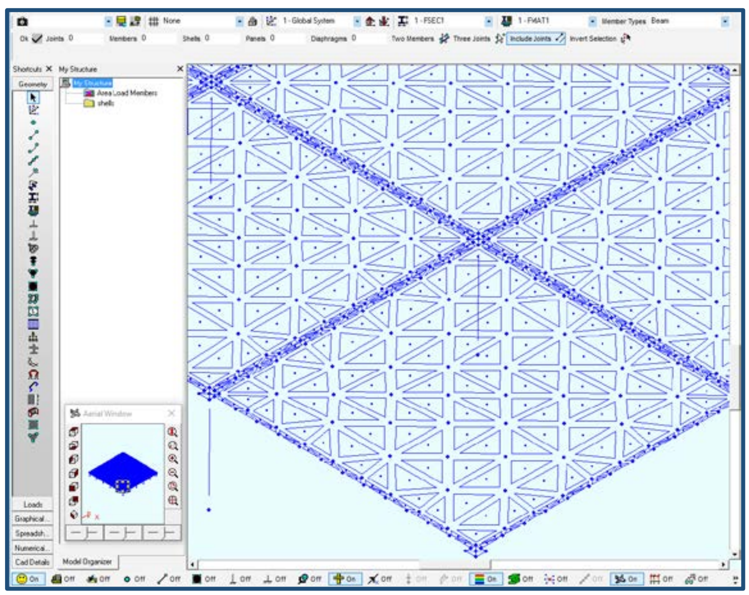


Figure 105.

Using the Nodal contouring option, the peak hogging moment is significantly reduced from **432** to **331 kNm/m**.

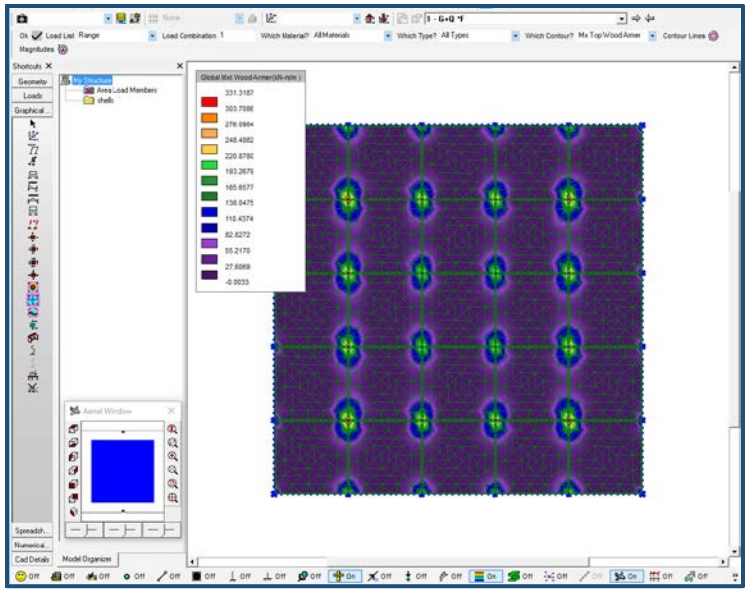


Figure 106.

If we take the results for a 2 m wide strip running from one column head to the next, then the results are as shown below:

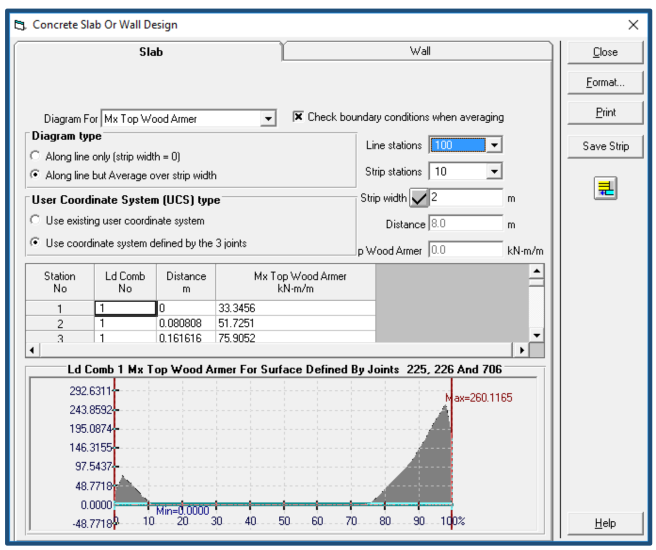


Figure 107.

The peak moment is now **260 kNm/m** which looks very reasonable when compared with the estimated **265 kNm/m** (before redistribution) using the code's equivalent frame approach.

You can also look at the sagging design moment on this section for a 4 m wide strip.

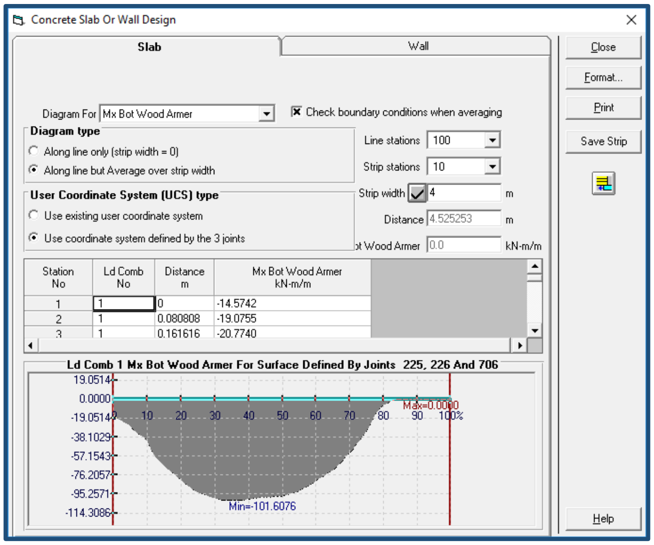


Figure 108.

This gives a result of **102 kNm/m**.

Design Moments – Comparison / Discussion - We have shown that contouring and strip results are all dependent on mesh densities, the modeling sophistication, and the methods used to include or average out analytical peaks.

We have also shown that by using a combination of a more sophisticated model and strip averaging options some very reasonable results can be achieved.

The design sagging moment has been established in different ways. For this alternative model the results are as follows:

- by contouring based on nodal forces – 130 kNm/m
- by contouring based on centroidal forces – 115 kNm/m
- by cutting a strip and averaging across a 4 m width – 102 kNm/m

As noted previously, the code indicates that pattern loading should be considered, You have the option to manually pattern the loads, but you might also consider making some sort of adjustment to the sagging design moments. For the averaged strip moments 15% to 20% adjustment is necessary to achieve a design moment in the region of the 120 kNm/m suggested by the code. However, if the design is to be based on the local peaks exposed by contouring based on nodal forces, then little or no adjustment seems necessary in this example. Using this model, it seems that you could conclude that the design sagging moment should be somewhere between

1.2 x 102 = 122 kNm/m and 130 kNm/m.

Bearing in mind these relationships established by looking at the code approach the peak design hogging moment over the column is expected to be a little less than double the peak sagging moment – 130 x 2 = 260 kNm/m. For this model, a result of 265 kNm/m has been shown. This is still higher than the peak value suggested by the code, but that applies after redistribution.

If the contour/strip results are not looking close to this sort of relationship, then closer examination/checking is appropriate.

Summary

Codes do not offer guidance for complex flat slab models. (only more simple regular models), therefore you can only make a comparison between the code and **S-FRAME** for these regular models.

For sagging moments finite element and code results are easily and well matched.

Using strip options (and more detailed modeling where necessary) **S-FRAME** also gives similar results to code for the hogging moments.

The above principles are equally applicable to and can be used on, more complex models.

However, when using a finite element analysis for flat slab design, remember:

- to consider the need to amplify the sagging moments to allow for pattern loading,
- to adjust the stiffness value **EI** to allow the analysis to estimate long-term cracked section deflections.

Using contours for a Wall - Example

If you want to work through this example with us, then load the model **Shells Example 1 Stage 1**.TEL. In this example, a shear wall and frame are subjected to a total lateral loading of 250 kN (50 kN per floor).

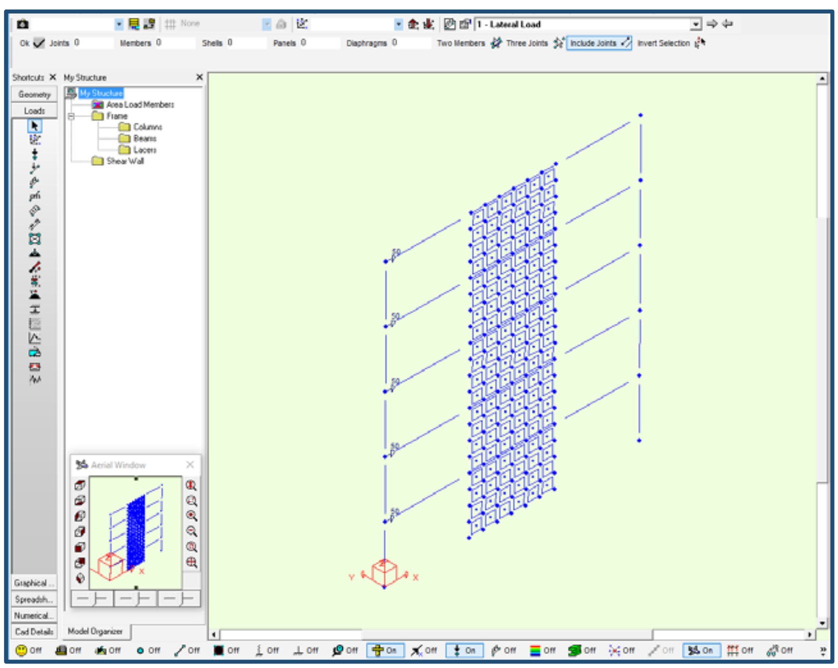


Figure 109.

This model has a number of notable points:

- it is a 2D frame modeled in 3D. Therefore default out of plane constraints have been applied,
- where beams meet shells in the same plane the two element types will not, by default, interact in such a way as to generate a moment connection. This is a common feature of analysis models. In order to generate a moment connection between the beams and the wall lacer elements have been threaded through the wall. (Another commonly accepted model that generates the fixity is to create a fan of elements from the beam end to all the adjacent nodes in the wall),
- you can see the local x-axis of all the shells. In general, these have all been aligned, with one highlighted exception.

Analyze the model and then select the Standard 3D view in the Graphical Results window.

Select the **Membrane Forces Tool** and **S-FRAME** displays color coded element center forces.

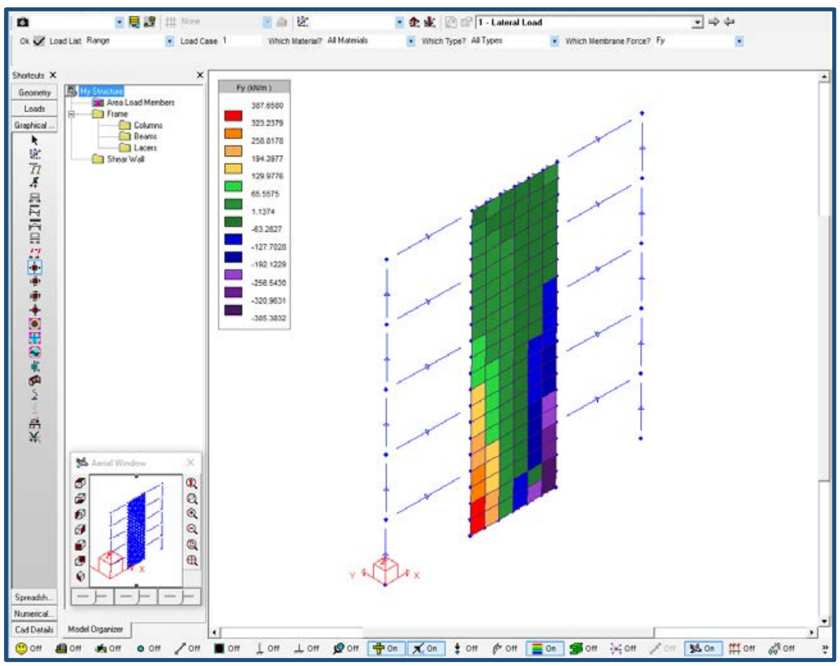


Figure 110.

In this example, the local **y** for all the elements (except the one highlighted previously – the second-to-right, second-to-bottom shell) are aligned with the Global **Z** axis as shown. The exception's local **x**-axis has been rotated by 90° and is, therefore, parallel to the global **Z** axis.

If you examine the results closely, you will see that the results for this shell don't fit into the expected pattern of stressing – which should run from high tension at the left to high compression at the right. This is because the force for this element is actually that at 90° to the forces for all the other elements.

Using contouring, we can see all sorts of results in relation to one consistent coordinate system. However, if you simply select the Shell Contours Tool and try to view results for the vertical loading in this wall (**Fz**) you will find that there is no such option.

You can only view contours for **Fx** and **Fy**. (While in this particular case the additional **Fz** option might appear to make sense, this would be no good if the wall were at some arbitrary angle.) Generally in order to see sensible contour results (for both forces and stresses) you need to use an adjusted user coordinate system with the **X**- and **Y**-axis in the plane of the shells under consideration.

For details of the sign conventions relating to shell elements, **see Coordinate Systems and Local Axes** in **Chapter 2** of the **S-FRAME Verification Manual**.

For further information on user coordinate systems see **"Using user coordinate systems"** on the **Introduction Manual – 3D Tutorial.**

Select the **Wall 1** user coordinate system for this model, and you will start to see meaningful contouring for **Fy** forces. The view below shows a top view which always looks down onto the current **XY**-plane.

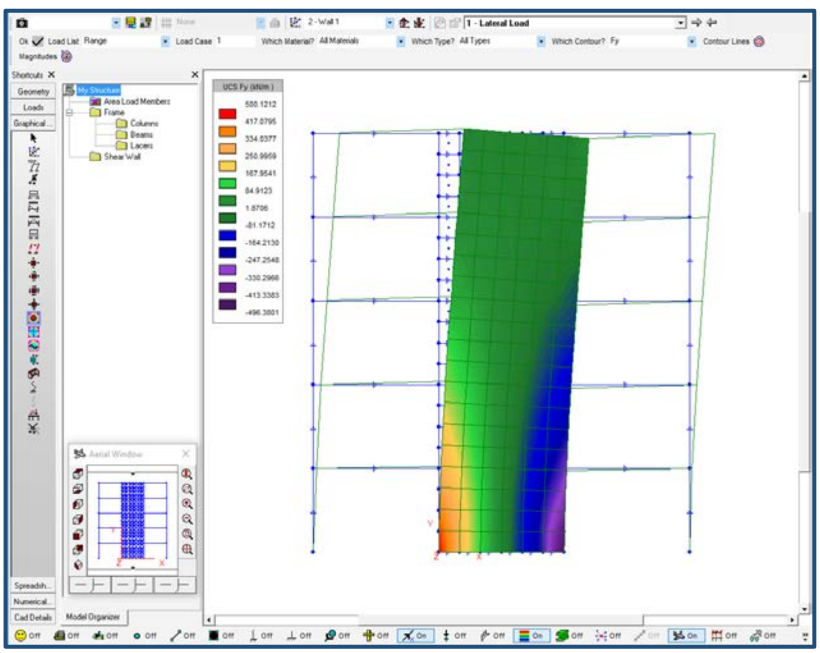


Figure 111.

You may find it useful to save a 2D view of each wall or panel for a model with numerous panels in numerous different planes. The next example (a "box" model) illustrates this point nicely.

Finally, you may want to see wall panel design forces. Once again you can achieve this by cutting a strip through the wall. Simply click on two nodes to define the start and end of the line, and then a third node to set the plane of interest.

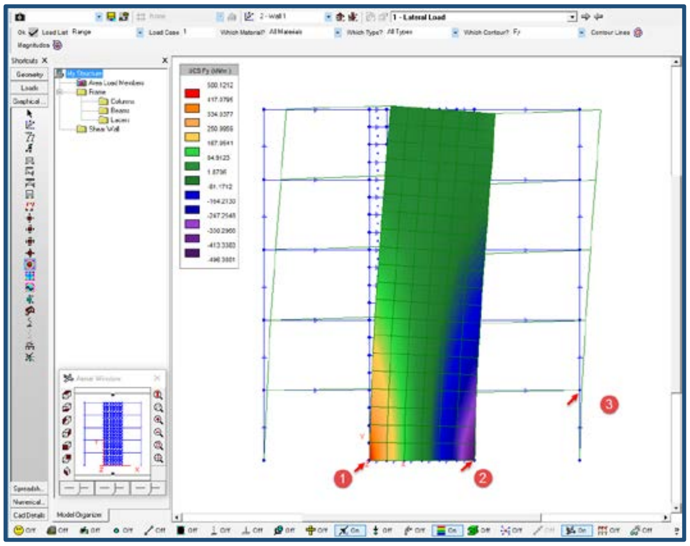


Figure 112.

It is easier to click the 3rd node as shown above since this node is not masked by the contouring. The actual location of the node is immaterial since it is only used to set the direction of the plane.

In this case, there is no need to work on a 2D plane since this is a 2D model which is modeled in 3D.

You will see the **Concrete Slab or Wall Design** dialog. Since we are dealing with a wall, click the **Wall** tab.

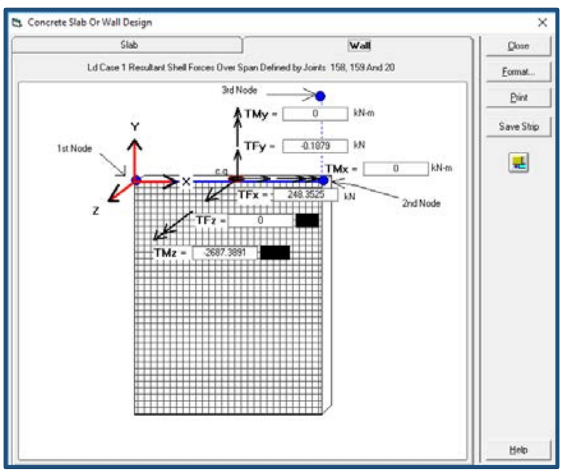


Figure 113.

This shows you the forces in the wall along the plane which you picked. These results can be used as panel force input for wall design in **W-Sect**.

Using Contours for a Box - Example

If you want to work through this example with us, then load the model Shells Example 2 Stage 1.TEL.

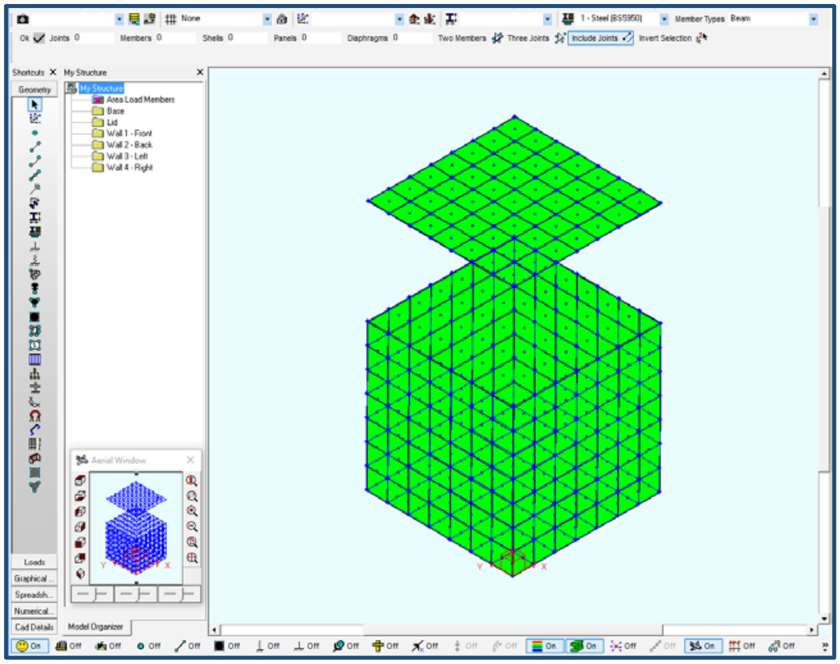


Figure 114.

A common question you ask is how to go about modeling

things such as:

- tank lids that bear on and laterally restrain tank walls, but with no transfer of moment across the joint,
- continuous slabs that bear on steel beams but which can rotate on the top flange,
- slab discontinuities.

In all cases, you can separate the discontinuous panels within the model and then slave the elements in order to model the correct interaction.

In this example, we raise the Lid above the tank walls and slave the perimeter joints at the top of the walls and the corresponding joints at the edge of the lid.

If you use the **Edit** > **Start Numbers** menu option to preserve number sequences during copy/paste operations, you can slave the joints very quickly using the **Spreadsheet** window.

If you limit slaving to two opposite edges, you can achieve a one-way-spanning load distribution.

The model has two load cases, the first is pressure on the lid (shown below), the second is an outwards pressure on each of the four walls.

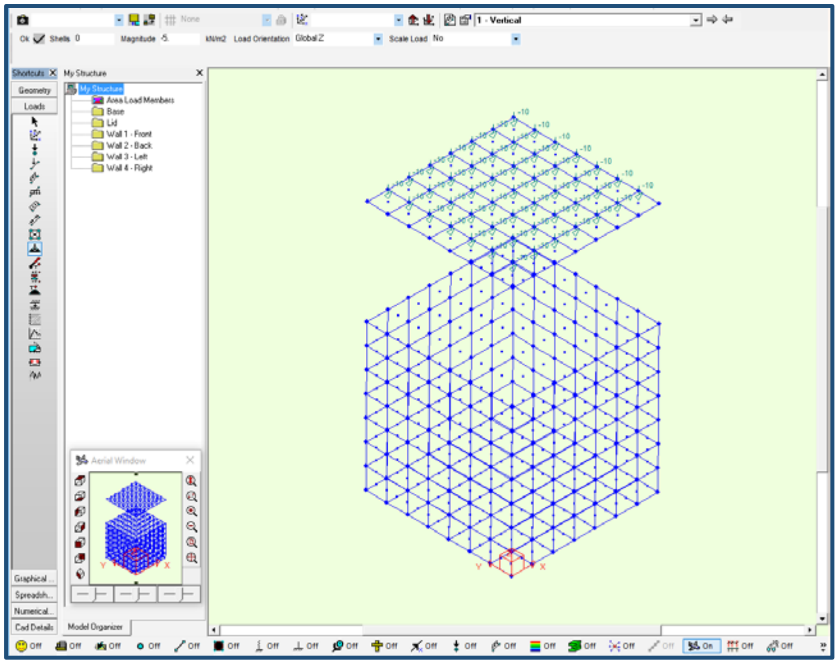


Figure 115.

Run a linear static analysis on this model, select the Shell Contours Tool, the Mx contours option for the lateral load case and you should see the result below.

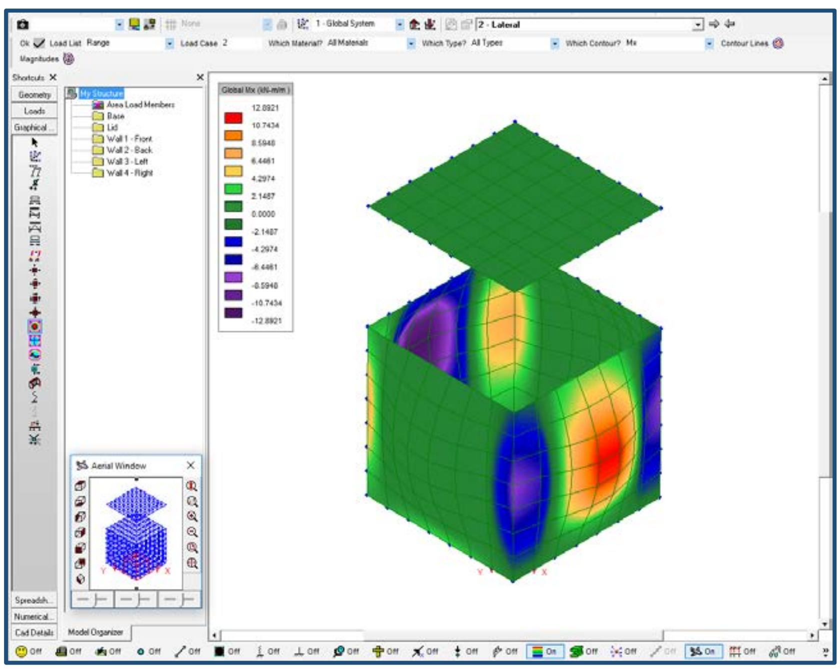


Figure 116.

The analysis will issue FYI (information) messages, in this case, these indicate that no Mz restraints have been applied to the lid, in practice, the slaving arrangement means that there is no instability and you can ignore this message.

The current coordinate system has it's XY-plane in the plane of the tank base. Although the details for the front and back walls look tantalizingly correct they are not. Contouring the walls relative to this user coordinate system is, to put it bluntly, nonsense.

You can customize the view by right-clicking on the Shell Contours Tool to display the Contour Diagram Factors dialog. If your view is not the same as that above, adjust your view settings to those shown below.

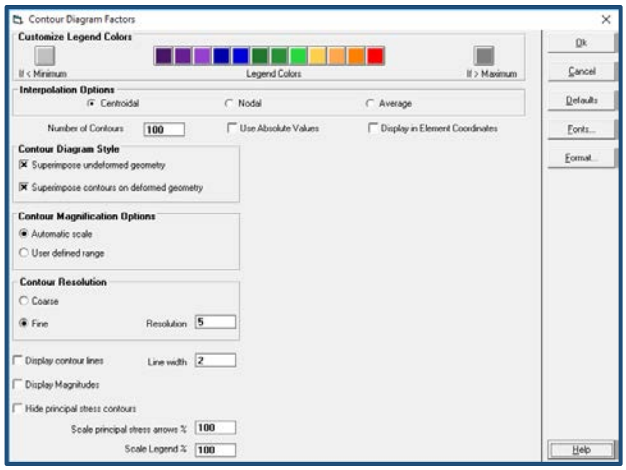


Figure 117.

Select the Front Wall user coordinate system that has been set up in this model, and you can move towards seeing meaningful contouring for the Mx forces on the front wall.

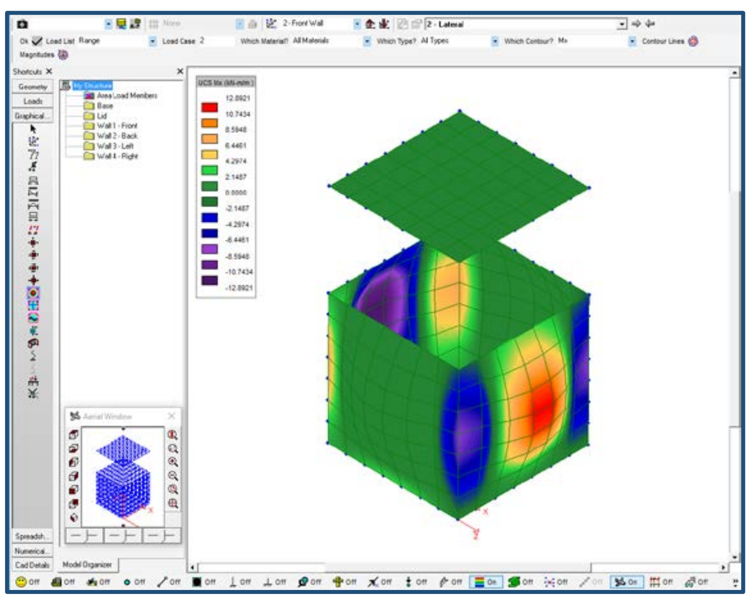


Figure 118.

Bear in mind that this is a symmetrical model, symmetrically loaded. It is obvious that the contouring above is not symmetrical. This indicates the importance of contouring relative to an appropriate coordinate system.

Mx moments are those that act on (not around) the **X**-axis – so, in this case, they are the moments for the horizontal span of the wall.

The range of forces reported for this front wall as shown in the legend are in the range **±12.9 kNm / m**.

Click on the Wall 1 – Front group to see the results for only the front wall.

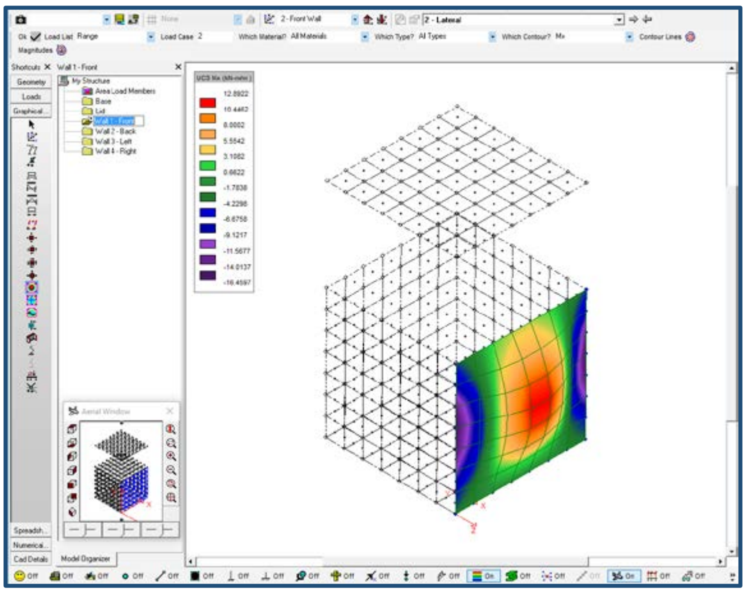


Figure 119.

You will now see that the range of forces reported for the front wall are in the range **+12.9** to **-16.5 kNm/m**. We need to ask ourselves "Why is there this difference?"

The answer is that **S-FRAME** always interpolates between adjacent selected shells when it is producing contours. Click on the **Wall 3– Left** group, and you will see the results for the left wall.

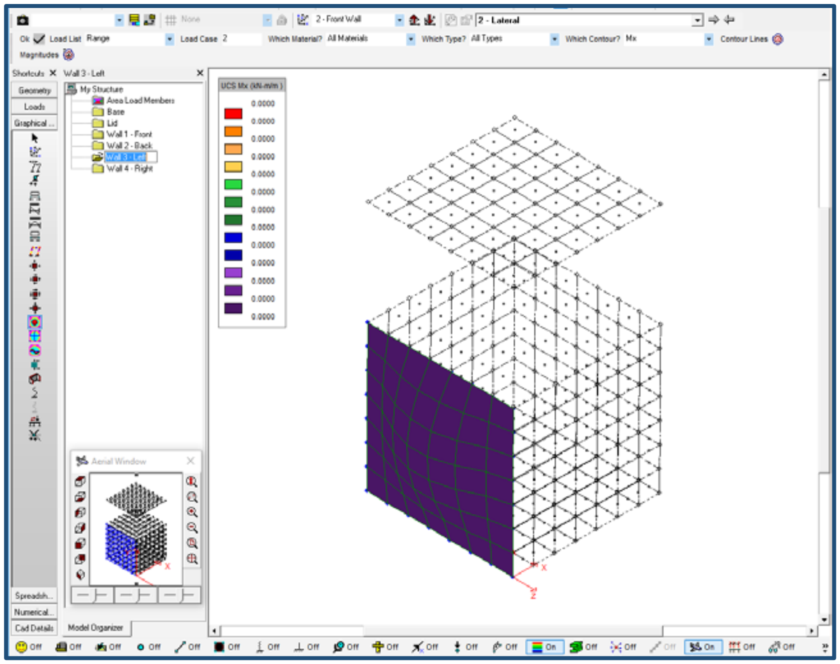


Figure 120.

With the current coordinate system, there are no forces in the Mx direction in the left wall. Thus when both the front wall and side walls were selected **S-FRAME** adjusted the front wall contouring to take account of the zero values in the side walls. This illustrates the importance of ensuring that you only select the shells in which you are interested, and for which the current user coordinate system is valid.

It should be becoming obvious that you need to set up several coordinate systems to be able to see meaningful results for each wall. Even if you do this, you might find that your view of one wall's results gets obscured by the others. It is here that the **Views** feature becomes particularly important for models with shells.

For further information **see "Using Views"** on the **Introduction Manual – 3D Tutorial.**

A view includes details of the current coordinate system, the current folder, and can also be marked as a 2D view (that is it will only show elements that exist in the current **XY**-plane). Therefore, although it takes a little time to set up, we think that it is well worth saving a view for each panel in a model such as this. You will find all the views you need in this model.

Select the **Wall 3 – Left** view and you will see the Mx forces on that wall.

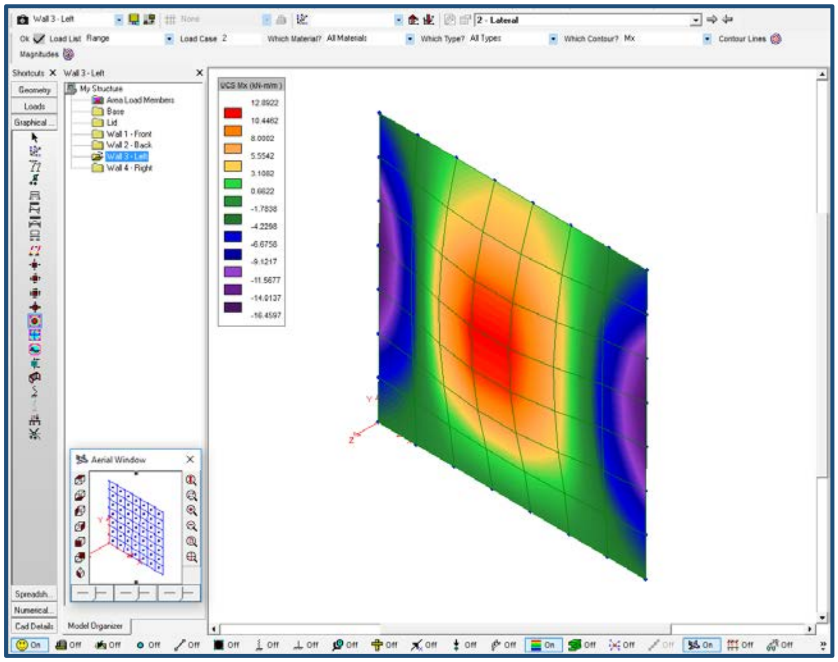


Figure 121.

As expected, the Mx moments in this wall are the same as for the front wall.

Now right click on the Shell Contours Tool to display the Contour Diagram Factors dialog. Choose the Nodal interpolation option as shown below and click Ok.

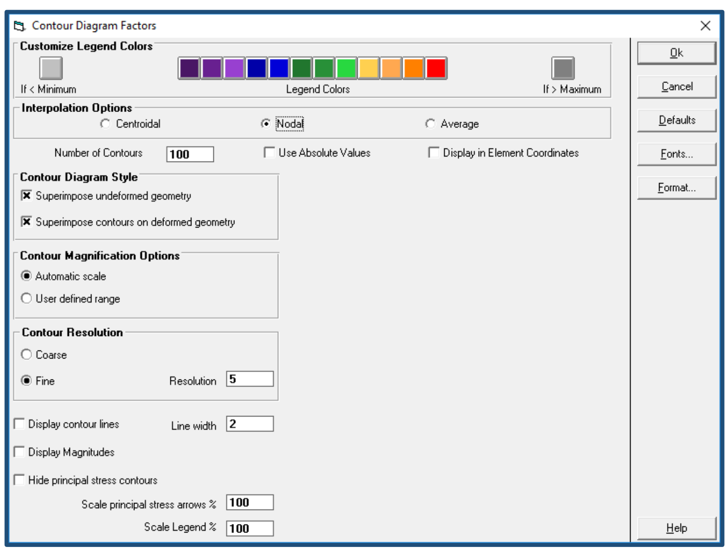


Figure 122.

You will now see that the range of forces changes in the contoured diagram.

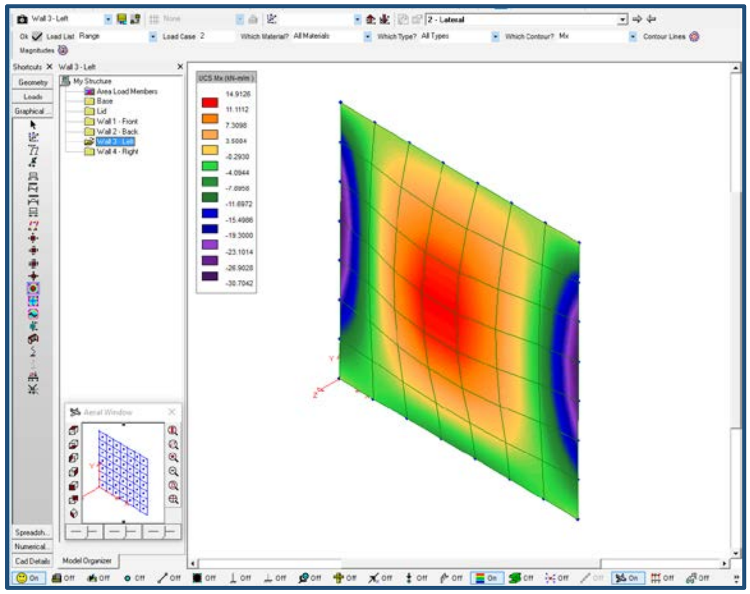


Figure 123.

You will now see that the range of forces reported for the left wall are in the range **+14.9** to **-30.7 kNm/m**. Again we need to understand why there is this difference. It is in the way that **S-FRAME** determines the forces in each node for contouring purposes. This has been covered previously.

For further information, please see "**To make contour diagram settings"** topic in the **S-FRAME Help System.**

Right click on the Shell Contours Tool to display the **Contour Diagram Factors** dialog. Choose the Centroidal interpolation option and click Ok.

Finally, let's look at Von Mises Stresses for the entire model. Select Von Mises Top from the list of contouring options and the Standard 3D view.

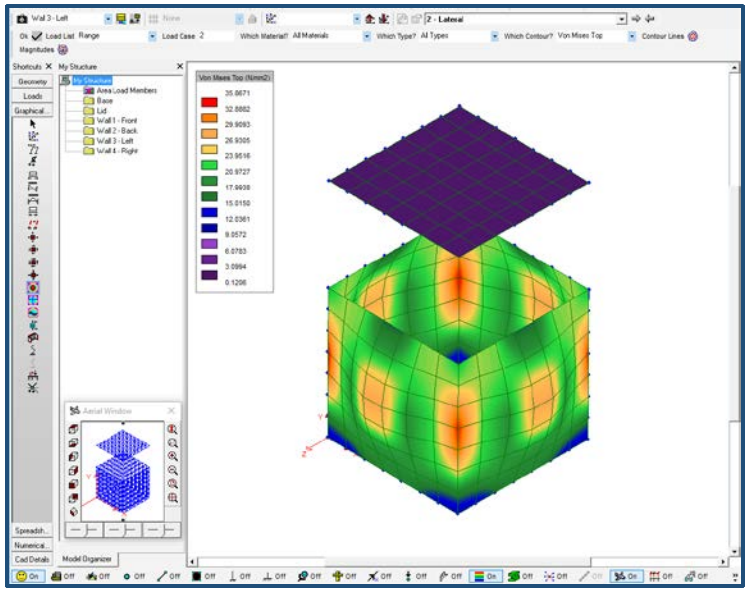


Figure 124.

Here we do see a degree of symmetry starting to arise. However contouring relative to a common UCS is not supported for Von Mises, Smax, Smin, and Sxymax contouring options. When these options are selected, the local Z is used to define the **top / middle / bottom** surface. Hence you will find that checking the# option to Display in element coordinates makes no difference for these contouring options because they are always displayed in element coordinates, the local **z** direction being the only important aspect of the coordinate system for any of these options. If you intend to use these options, it is, therefore, important that you define your elements with a common local **Z**-direction. This can often be achieved by defining your elements in a consistent clockwise or counter-clockwise manner. However, in some cases you will need to deviate from this - containers for example (see the documented box model) in such cases you might think of **outer** and **inner** surfaces and so you need to reverse your convention on opposite walls. so if we were to reverse the local **z**-axes on the back wall of this box so that all local **z**-axes either point out (or in) then the Von Mises contouring would be symmetrical.

For more detail on the derivation of Von Mises stresses, refer to the next section.This appears to show virtually no stresses in the walls, but again this is a function of the current user coordinate system, on which the contouring is based. We have chosen to see the Von Mises Top stresses, but Top means the face on the positive **Z**-side of the shells, and since the **Z**-axis is in the plane of the walls this again is inappropriate.

For a further example looking at the influence of coordinate systems and interpolation options when contouring, **see Example 22 - Cylindrical roof static analysis** in **Chapter 3** of the **S-FRAME Verification Manual**.

Overview of Maximum Shear Criterion

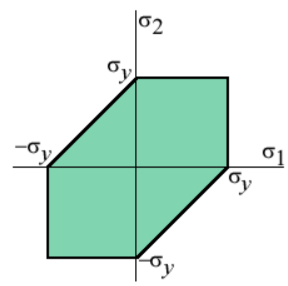
The information below is extracted from the efunda (engineering fundamentals) website; you can find the full version (and much more of interest) at:

http://www.efunda.com/formulae/solid_mechanics/failure_criteria/failure_criteria_ductile.cfm

The maximum shear stress criterion, also known as Tresca's or Guest's criterion, is often used to predict the yielding of ductile materials. Yield in ductile materials is usually caused by the slippage of crystal planes along the maximum shear stress surface. Therefore, a given point in the body is considered safe as long as the maximum shear stress at that point is under the yield shear stress obtained from a uniaxial tensile test. With respect to 2D stress, the maximum shear stress is related to the difference in the two **principal stresses**. Therefore, the criterion requires the principal stress difference, along with the principal stresses themselves, to be less than the yield shear stress,

$$|\sigma_1| \le \sigma_y, \ |\sigma_2| \le \sigma_y, \text{ and } |\sigma_1 - \sigma_2| \le \sigma_y$$

Graphically, the maximum shear stress criterion requires that the two principal stresses be within the green zone indicated below.



Von Mises Criterion

The von Mises Criterion (1913), also known as the maximum distortion energy criterion, octahedral shear stress theory, or Maxwell-Huber-Hencky-von Mises theory, is often used to estimate the yield of ductile materials.

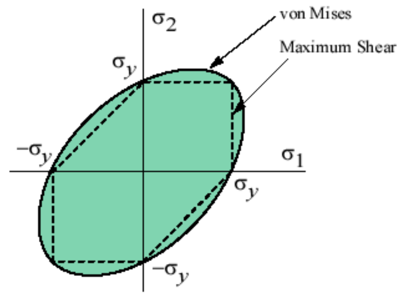
The von Mises criterion states that failure occurs when the energy of distortion reaches the same energy for yield/failure in uniaxial tension. Mathematically, this is expressed as,

$$\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right] \le \sigma_y{}^2$$

In the cases of plane stress, $\sigma_3 = 0$. The von Mises criterion reduces to,

$$\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2 \le \sigma_y^2$$

This equation represents a principal stress ellipse as illustrated in the following figure,



Also shown on the figure is the **maximum shear stress criterion** (dashed line). This theory is more conservative than the von Mises criterion since it lies inside the von Mises ellipse.

In addition to bounding the principal stresses to prevent ductile failure, the von Mises criterion also gives a reasonable estimation of fatigue failure, especially in cases of repeated tensile and tensile-shear loading.

Principal Directions, Principal Stress

The normal stresses (σ_x'and σ_y') and the shear stress ($\tau_{x'y'}$) vary smoothly with respect to the rotation angle , in accordance with the coordinate transformation equations. There exist a couple of particular angles where the stresses take on special values.

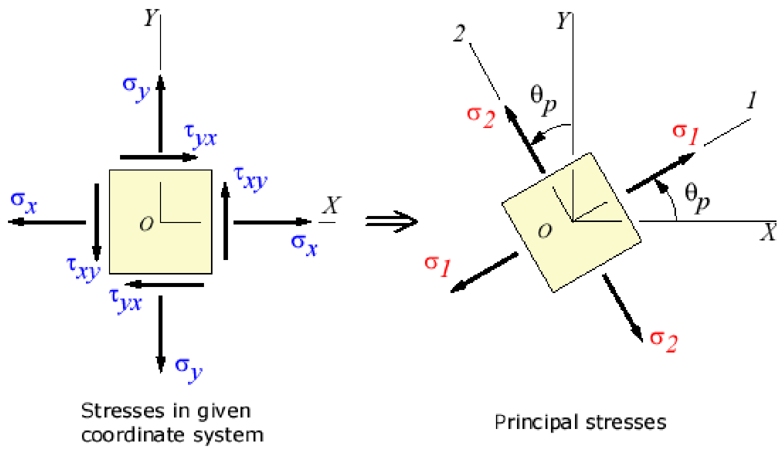
First, there exists an angle where the shear stress becomes zero. That angle is found by setting τ $\tau_{x'y'}$ to zero in the above shear transformation equation and solving for q (set equal to θ_p). The result is,

$$\tan 2\theta_p = \frac{2\tau_{xy}}{\sigma_x - \sigma_y}$$

The angle θ_p defines the principal directions where the only stresses are normal stresses. These stresses are called principal stresses and are found from the original stresses (expressed in the x,y,z directions) via,

$$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}{}^2}$$

The transformation to the principal directions can be illustrated as:



Maximum Shear Stress Direction

Another important angle, θ_s, is where the maximum shear stress occurs. This is found by finding the maximum of the shear stress transformation equation and solving for θ. The result is,

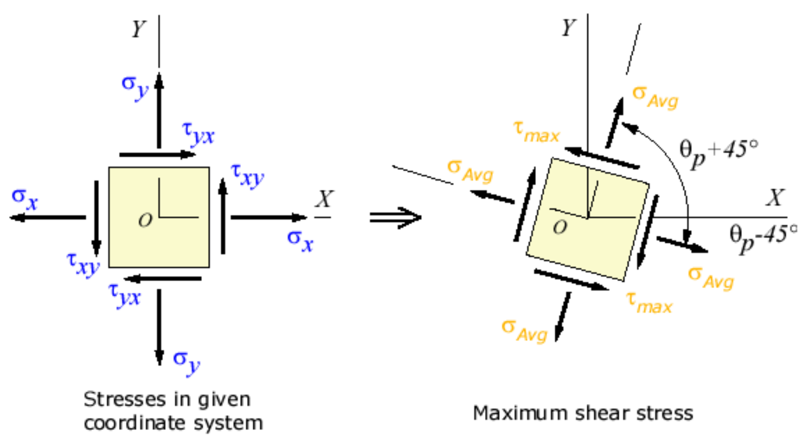
$$\tan 2\theta_s = -\frac{\sigma_x - \sigma_y}{2\tau_{xy}}$$

$$\Rightarrow \theta_s = \theta_p \pm 45^\circ$$

The maximum shear stress is equal to one-half the difference between the two principal stresses,

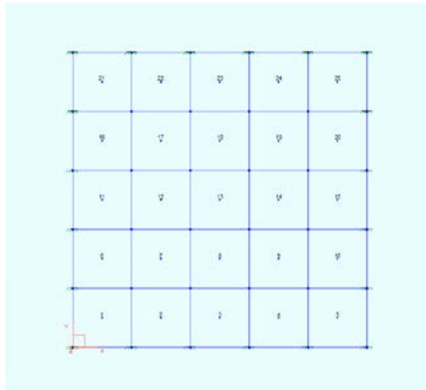
$$\tau_{\max} = \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}{}^2} = \frac{\sigma_1 - \sigma_2}{2}$$

The transformation to the maximum shear stress direction can be illustrated as:

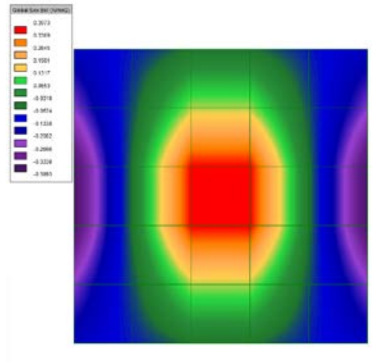


Example

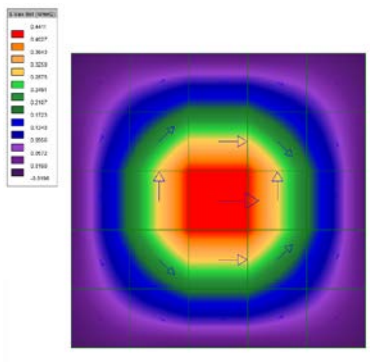
This 5 m square, 250 mm thick slab has a uniform area load of 10 kN/m2.



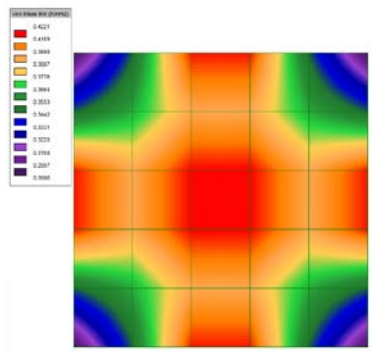
Model.



Normal Stress - Sxx



Principal Stress - Smax



Von Misses Stress

Consider Shell Number 9

Normal Stress, S_{xx}	σ_x = **0.268** N/mm2	calculated by S-Frame
Normal Stress, S_{yy}	σ_y = **0.268** N/mm2	calculated by S-Frame
Shear Stress, S_{xy}	σ_{xy} = **0.143** N/mm2	calculated by S-Frame

Principal Stresses

Max Principal Stress, Smax	$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + {\tau_{xy}}^2}$	= **0.411** N/mm2
Min Principal Stress, Smin	$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + {\tau_{xy}}^2}$	= **0.125** N/mm2
Angle	$\theta_p = \frac{1}{2}\, atan[\frac{2\tau_{xy}}{\sigma_x - \sigma_y}]$	= 45°

Von Mises Stress

Von Mises Stress	$\sigma_y = \sqrt{\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2}$	= 0.365 N/mm^2

S-FRAME Results

The results from the **S-FRAME** analysis are shown in the following table:

Shell	Location	Sxx N/mm2	Syy N/mm2	Sxy N/mm2	Smax N/mm2	Smin N/mm2	Angle Deg	Von Mises N/mm2
1	Top	0.1588	0.1588	0.1391	0.2979	0.0196	45	0.2886
	Middle	0	0	0	0	0	0	0
	Bottom	-0.1588	-0.1588	-0.1391	-0.0196	-0.2979	-45	0.2886
2	Top	0.0394	0.3443	0.138	0.3975	-0.0138	68.9203	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.0394	-0.3443	-0.138	0.0138	-0.3975	-21.0797	0.4046
3	Top	0.0399	0.4543	0	0.4543	0.0399	-90	0.4358
	Middle	0	0	0	0	0	0	0
	Bottom	-0.0399	-0.4543	0	-0.0399	-0.4543	0	0.4358
4	Top	0.0394	0.3443	-0.138	0.3975	-0.0138	-68.9203	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.0394	-0.3443	0.138	0.0138	-0.3975	21.0797	0.4046
5	Top	0.1588	0.1588	-0.1391	0.2979	0.0196	-45	0.2886
	Middle	0	0	0	0	0	0	0
	Bottom	-0.1588	-0.1588	0.1391	-0.0196	-0.2979	45	0.2886
6	Top	0.3443	0.0394	0.138	0.3975	-0.0138	21.0797	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.3443	-0.0394	-0.138	0.0138	-0.3975	-68.9203	0.4046
7	Top	-0.2673	-0.2673	0.1431	-0.1243	-0.4104	45	0.3645
	Middle	0	0	0	0	0	0	0
	Bottom	0.2673	0.2673	-0.1431	0.4104	0.1243	-45	0.3645
8	Top	-0.3999	-0.3676	0	-0.3676	-0.3999	-90	0.3848
	Middle	0	0	0	0	0	0	0
	Bottom	0.3999	0.3676	0	0.3999	0.3676	0	0.3848
9	Top	-0.2673	-0.2673	-0.1431	-0.1243	-0.4104	-45	0.3645
	Middle	0	0	0	0	0	0	0
	Bottom	0.2673	0.2673	0.1431	0.4104	0.1243	45	0.3645
10	Top	0.3443	0.0394	-0.138	0.3975	-0.0138	-21.0797	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.3443	-0.0394	0.138	0.0138	-0.3975	68.9203	0.4046
11	Top	0.4543	0.0399	0	0.4543	0.0399	0	0.4358
	Middle	0	0	0	0	0	0	0
	Bottom	-0.4543	-0.0399	0	-0.0399	-0.4543	90	0.4358
12	Top	-0.3676	-0.3999	0	-0.3676	-0.3999	0	0.3848
	Middle	0	0	0	0	0	0	0
	Bottom	0.3676	0.3999	0	0.3999	0.3676	90	0.3848
13	Top	-0.5542	-0.5542	0	-0.5542	-0.5542	0	0.5542
	Middle	0	0	0	0	0	0	0

	Bottom	0.5542	0.5542	0	0.5542	0.5542	0	0.5542
14	Top	-0.3676	-0.3999	0	-0.3676	-0.3999	0	0.3848
	Middle	0	0	0	0	0	0	0
	Bottom	0.3676	0.3999	0	0.3999	0.3676	-90	0.3848
15	Top	0.4543	0.0399	0	0.4543	0.0399	0	0.4358
	Middle	0	0	0	0	0	0	0
	Bottom	-0.4543	-0.0399	0	-0.0399	-0.4543	-90	0.4358
16	Top	0.3443	0.0394	-0.138	0.3975	-0.0138	-21.0797	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.3443	-0.0394	0.138	0.0138	-0.3975	68.9203	0.4046
17	Top	-0.2673	-0.2673	-0.1431	-0.1243	-0.4104	-45	0.3645
	Middle	0	0	0	0	0	0	0
	Bottom	0.2673	0.2673	0.1431	0.4104	0.1243	45	0.3645
18	Top	-0.3999	-0.3676	0	-0.3676	-0.3999	-90	0.3848
	Middle	0	0	0	0	0	0	0
	Bottom	0.3999	0.3676	0	0.3999	0.3676	0	0.3848
19	Top	-0.2673	-0.2673	0.1431	-0.1243	-0.4104	45	0.3645
	Middle	0	0	0	0	0	0	0
	Bottom	0.2673	0.2673	-0.1431	0.4104	0.1243	-45	0.3645
20	Top	0.3443	0.0394	0.138	0.3975	-0.0138	21.0797	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.3443	-0.0394	-0.138	0.0138	-0.3975	-68.9203	0.4046
21	Top	0.1588	0.1588	-0.1391	0.2979	0.0196	-45	0.2886
	Middle	0	0	0	0	0	0	0
	Bottom	-0.1588	-0.1588	0.1391	-0.0196	-0.2979	45	0.2886
22	Top	0.0394	0.3443	-0.138	0.3975	-0.0138	-68.9203	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.0394	-0.3443	0.138	0.0138	-0.3975	21.0797	0.4046
23	Top	0.0399	0.4543	0	0.4543	0.0399	90	0.4358
	Middle	0	0	0	0	0	0	0
	Bottom	-0.0399	-0.4543	0	-0.0399	-0.4543	0	0.4358
24	Top	0.0394	0.3443	0.138	0.3975	-0.0138	68.9203	0.4046
	Middle	0	0	0	0	0	0	0
	Bottom	-0.0394	-0.3443	-0.138	0.0138	-0.3975	-21.0797	0.4046
25	Top	0.1588	0.1588	0.1391	0.2979	0.0196	45	0.2886
	Middle	0	0	0	0	0	0	0
	Bottom	-0.1588	-0.1588	-0.1391	-0.0196	-0.2979	-45	0.2886

Table 4.

Creating Curved Members

Curved sections are increasingly used in constructional steelwork. They combine design efficiency with strong aesthetic appeal; providing excellent value for money.

Whilst not covered specifically by many design codes, designers cope well with special considerations required for such sections both in terms of analysis and design.

Whenever we are asked if we support curved analysis members, we advise that you model the curve as series of straight members. Since this is a common issue, we give guidance below on the best way of building these sections into a 3D analysis model.

The process

The following pages detail how you can create curved members within S-FRAME.

Before you generate the curved member you need to know:

- its rise, r
- its span, L
- the number of nodes that you want to use to define the curved member, n.

These are indicated in the diagram below.

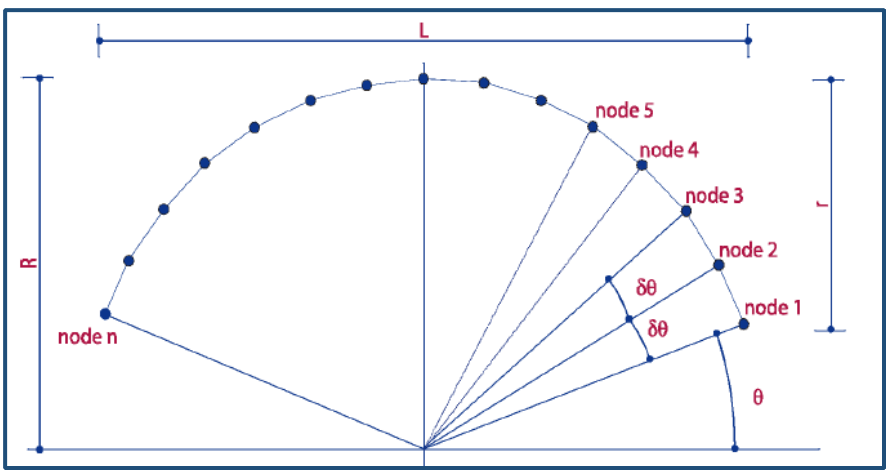


Figure 125.

These terms can be used to calculate:

- the radius of the curved member, where $R = \frac{\frac{L^2}{4}+r^2}{2r}$
- the angle through which the x-axis would rotate to cut through the first node, θ, where $\theta = \mathrm{acos}(\frac{L}{2R})$
- the angle through which the curved member will sweep between two nodes, δθ, where $\delta\theta = 2\frac{(90-\theta)}{(n-1)}$

For maximum accuracy, the angles and should be recorded to 6 decimal places.

Once these variables have been calculated, they can be used to specify the curved member in **S-FRAME** or **P-FRAME**.

Moving the user coordinate system

1. From the **Toolbox** select the User Coordinate System Tool **(UCS)** with a click of the left mouse button. A click of the right mouse button will then open the dialog.
2. Move the user coordinate system to where the curved member is to begin. This is where the first node will be placed. To move the user coordinate system enter its new coordinates into the **Data Bar**. Alternatively, if there is a node already at the required position click on that node.

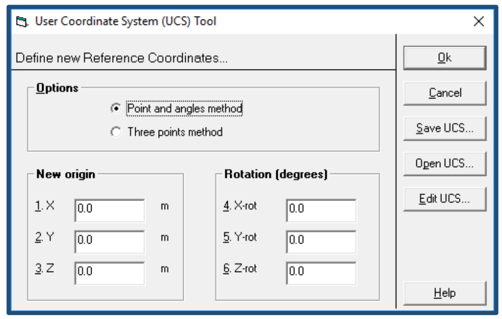


Figure 126.

3. Now place the user coordinate system at the center of the circle forming your curved member. Move the user coordinate system a distance **L** /**2** along the X-axis and (**r** – **R**) along the Y-axis.
4. Select Save UCS. Give the coordinate system a name and set the UCS type to Cylindrical. Click on OK.

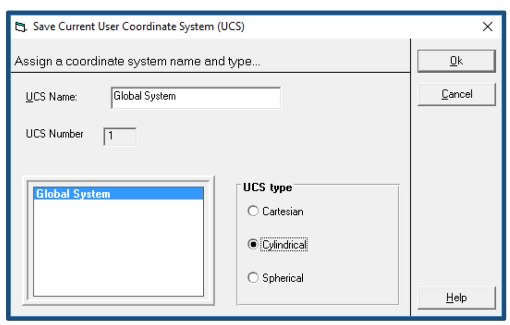


Figure 127.

Before going any further, ensure that no items are currently selected using **Unselect All** in the menu bar.

Define the Joints

1. Select Joint Tool from the **Toolbox** and then click on the right mouse button to display the **Joint Tool** dialog.
2. Set the Coordinate System to Cylindrical (Polar for P-FRAME).

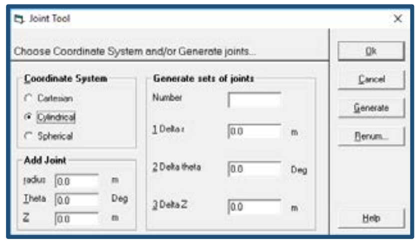


Figure 128.

3. Set the following values:
   - radius = R ,
   - Theta = θ ,
   - Delta theta = δθ,
   - Number = n – 1 ,
   - Delta r = 0

   For **S-FRAME** you will also need to set the values:
   - Z =0 ,
   - Delta Z = 0
4. To create these points select Generate and then OK.

If you choose **OK** but do not generate the points first, then no joints will appear.

In the **Geometry** window, the specified points will now have been drawn.

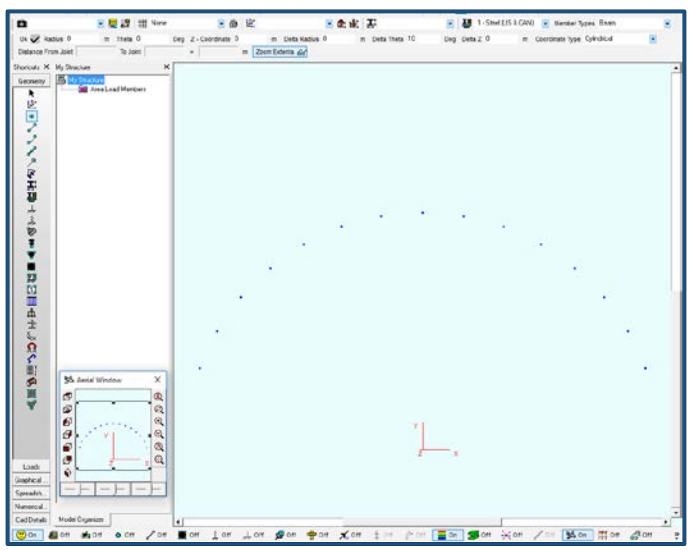


Figure 129.

Define the Members

In a curved member containing a large number of nodes, it is best to draw one of the members and then copy this to the other locations.

1. Ensure that only the two right-hand nodes are selected.
2. Select Member Definition with a click of the left mouse button.
3. Draw a member between these two nodes by a click of the left mouse button on each of the joints in turn.

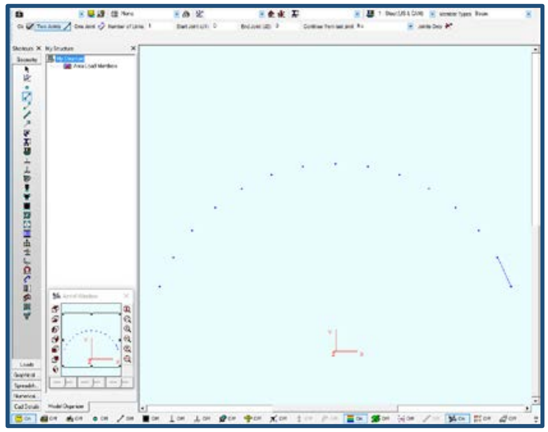


Figure 130.

4. Choose Edit > Copy. From the **Toolbox** select the Clone Tool and then click on the right mouse button to display the **Clone Tool** dialog. Choose Rotation as the Generation method.
5. The only fields that need to be amended are *Z rot*, *Number,* and *Delta Z rot*. Set these as follows:
    - Z rot = δθ,
    - Delta Z rot = δθ,
    - Number = (n-3).

   All others can be left as zero.

6. Choose *OK*. In the Geometry window, the remaining members will now have been drawn.

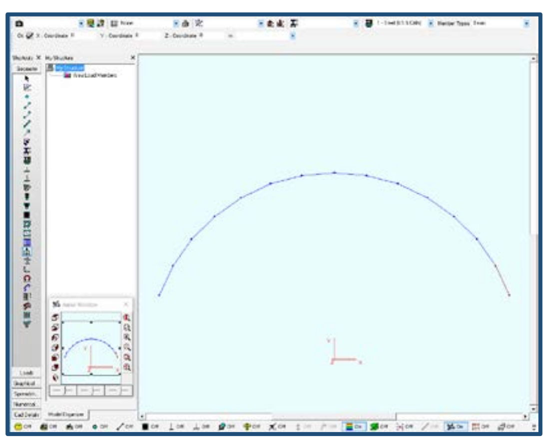


Figure 131.

Rotating a Curved Member in S-FRAME

In **S-FRAME** it is necessary to move the curved member from the horizontal **XY**-plane in which it has been generated into a vertical plane, either **YZ** or **XZ**.

To move it into the **YZ**-plane go to the Edit menu and select Move.

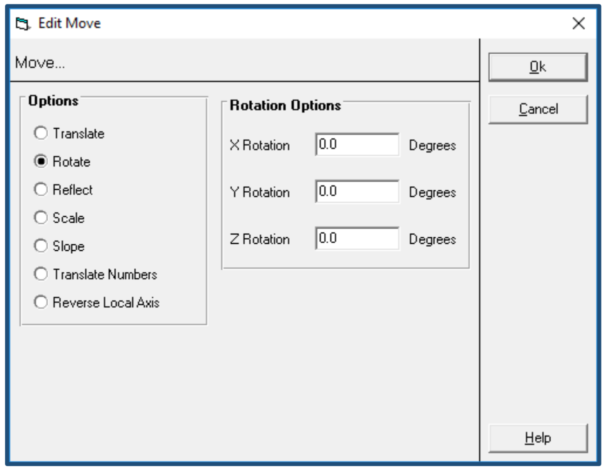


Figure 132.

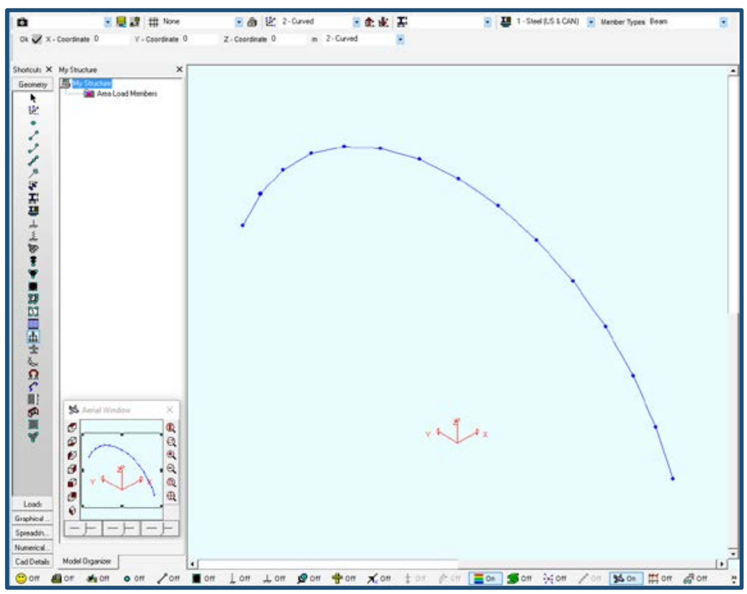


Figure 133.

To move your curved member into the **XZ**- rather than the **YZ**-plane, enter an **X Rotation** value of 90 degrees.

If the ends of the curved member overlap existing joints, the program will detect this and delete the duplicate joints. If this occurs, you may need to redefine the deleted members.