Altair S-FRAME 2021.1

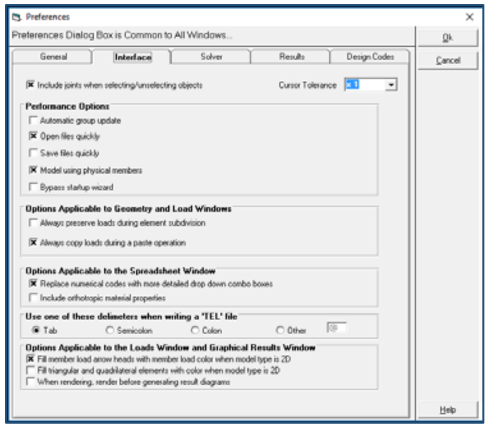
3D Structural Analysis
an Introduction

Disclaimer

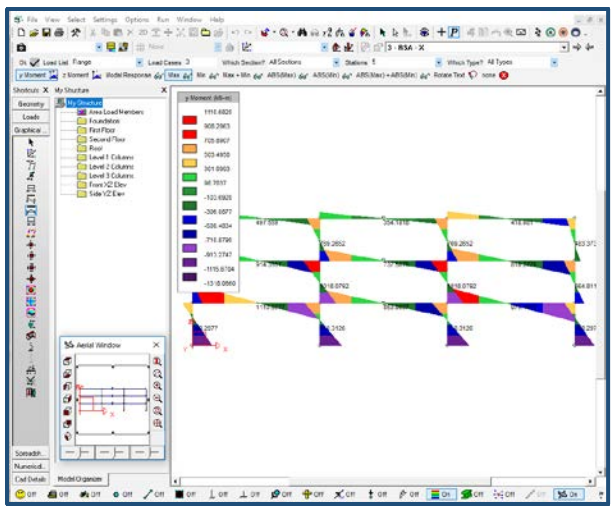
Considerable time, effort and expense have gone into the development and documentation of S-FRAME. It has been thoroughly tested. However, in using the product (including manuals), the user understands and accepts that no warranty on the accuracy or reliability of the product is expressed or implied 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.

Intellectual Property Rights Notice

This Intellectual Property Rights Notice is exemplary, and therefore not exhaustive, of intellectual property rights held by Altair Engineering Inc. or its affiliates. Software, other products, and materials of Altair Engineering Inc. or its affiliates are protected under laws of the United States and laws of other jurisdictions. In addition to intellectual property rights indicated herein, such software, other products, and materials of Altair Engineering Inc. or its affiliates may be further protected by patents, additional copyrights, additional trademarks, trade secrets, and additional other intellectual property rights. For avoidance of doubt, copyright notice does not imply publication. Copyrights in the below are held by Altair Engineering Inc. except where otherwise explicitly stated. Additionally, all non-Altair marks are the property of their respective owners.

This Intellectual Property Rights Notice does not give you any right to any product, such as software, or underlying intellectual property rights of Altair Engineering Inc. or its affiliates. Usage, for example, of software of Altair Engineering Inc. or its affiliates is governed by and dependent on a valid license agreement.

Altair Simulation Products

Altair® AcuConsole® ©2006-2021

Altair® AcuSolve® ©1997-2021

Altair Activate® ©1989-2021

Altair Compose® ©2007-2021

Altair® ConnectMe™ ©2014-2021

Altair® EDEM™ ©2005-2021 Altair Engineering Limited, ©2019-2021 Altair Engineering Inc.

Altair® ElectroFlo™ ©1992-2021

Altair Embed® ©1989-2021

Altair Embed® SE ©1989-2021

Altair Embed®/Digital Power Designer ©2012-2021

Altair Embed® Viewer ©1996-2021

Altair® ESAComp® ©1992-2021

Altair® Feko® ©1999-2021 Altair Development S.A. (Pty) Ltd., ©1999-2021 Altair Engineering Inc.

Altair® Flow Simulator™ ©2016-2021

Altair® Flux® ©1983-2021

Altair® FluxMotor® ©2017-2021

Altair® HyperCrash® ©2001-2021

Altair® HyperGraph® ©1995-2021

Altair® HyperLife® ©1990-2021

Altair® HyperMesh® ©1990-2021

Altair® S-STEEL™ ©1995-2021 S-Frame Software, Inc., ©2021 Altair Engineering Inc.

Altair® S-TIMBER™ ©1995-2021 S-Frame Software, Inc., ©2021 Altair Engineering Inc.

Altair® SEAM® ©1985-2019 Cambridge Collaborative, Inc., ©2019-2021 Altair Engineering Inc.

Altair® SimLab® ©2004-2021

Altair® SimSolid® ©2015-2021

Altair® ultraFluidX® ©2010-2018 FluiDyna GmbH, ©2018-2021 Altair Engineering Inc.

Altair® Virtual Wind Tunnel™ ©2012-2021

Altair® WinProp™ ©2000-2021

Altair® WRAP™ ©1998-2021 Altair Engineering AB

Altair Packaged Solution Offerings (PSOs)

Altair® Automated Reporting Director™ ©2008-2021

Altair® e-Motor Director™ ©2019-2021

Altair® Geomechanics Director™ ©2011-2021

Altair® Impact Simulation Director™ ©2010-2021

Altair® Model Mesher Director™ ©2010-2021

Altair® NVH Director™ ©2010-2021

Altair® Squeak and Rattle Director™ ©2012-2021

Altair® Virtual Gauge Director™ ©2012-2021

Altair® Weld Certification Director™ ©2014-2021

Altair® Multi-Disciplinary Optimization Director™ ©2012-2021

Altair HPC & Cloud Products

Altair® PBS Professional® ©1994-2021

Altair® Control™ ©2008-2021

Altair ®Access™ ©2008-2021

Altair® Accelerator™ ©1995-2021

Altair® Accelerator™ Plus©1995-2021

Altair® FlowTracer™ ©1995-2021

Altair® Allocator™ ©1995-2021

Altair® Monitor™ ©1995-2021

Altair® Hero™ ©1995-2021

Altair® Software Asset Optimization (SAO) ©2007-2021

Altair Mistral™ ©2021

Altair Drive ©2021

Contents

Tutorial 3D

The best way to learn any software application is to use it. This is why we suggested that you create your own simple model as you read through the Overview of Modeling procedure chapter, and why we provided a simple 2D example to get you up and running in the Tutorial (2D) chapter.

In this chapter, we shall use **S-FRAME** to create a simple 3D structure. Much of what we do in this chapter will extend what you read and did earlier. If you have not run through the two chapters mentioned above, then we would recommend that you do so now, **before** starting this 3D example. If you find that you do not know how to perform an action in this 3D example, then we suggest that you use the 2D example again to re-familiarize yourself with the basics.

Again the model shown in this tutorial is not intended to be realistic, nor is the method shown the best way to create it. Both model and method are intended to show you how to use the various features of **S-FRAME** when creating a model.

We shall create the model using **Metric** units and **American** sections to demonstrate the flexibility of **S-FRAME**.

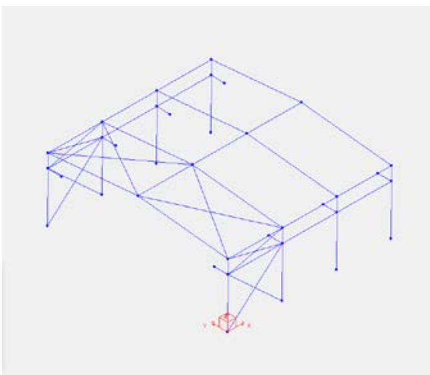


Figure 1. Isometric View

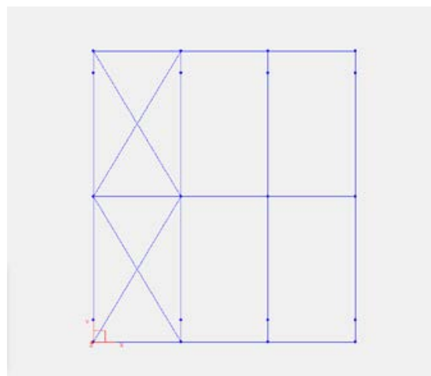


Figure 2. Top View

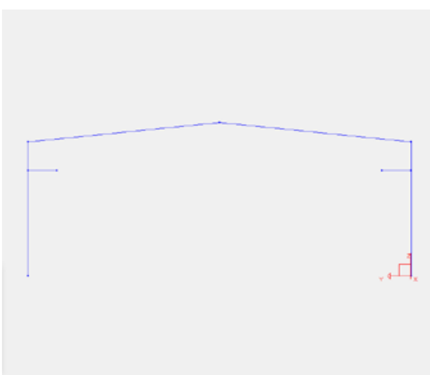


Figure 3. Left View (YZ View)

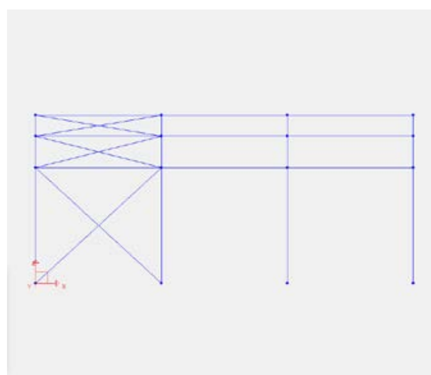


Figure 4. Front View (XZ View)

The view names shown above are those of the standard views which are relative to the current user coordinate system; they are not named using standard building terms where the **Side View** noted above would normally be referred to either as **End elevation** or **Gable**.

General 3D modeling advice

We would advise you to take much more care when you are modeling in 3D to ensure that the model you create is what you intend. In order to be careful, you have to be in complete control of the software. Generally for 3D modeling (but more specifically for large models) this means that you must be able to use some, if not all, of the more advanced tools described in the **S-FRAME Introduction manual - Detailed modeling hints and tips**. In particular, we would highlight the sections on:

- Rendering,
- Grouping / Folders,
- The User Coordinate System, and
- Views.

Starting the model

1. Start **S-FRAME**, create a new model, set the model type to 3D, and set the units to metric.
2. If the **Geometry** window does not have a **Groups** pane, then click View/ Groups to reinstate it.
3. Make sure that you can see the **Aerial** window (click **Window** > **Aerial Window** or click the Show/Hide Aerial window icon from the **Status Bar**) and in that window click the Zoom to grid extents icon.

Setting Preferences

Set your **Preference** settings to the initial set we suggested earlier.

For details **see "General Modeling Environment Settings"** in the **Introduction Manual – 2D Tutorial**

Setting display options – Geometry window

1. Ensure that the **Edit** toolbar, **Views and Grids** toolbar and the **Geometry Tools** toolbox are shown for the Geometry window.
2. Ensure that the **Grid**, **Title Bar**, **Legend**, **Zoom Extents**, **Local 'x' Axis**, **Joints, Redraw, Status Bar, Floating point format** and **Coordinate System** interface options are set.

Resizing and Repositioning a Structure

As you work through this example, and when you work on your own structures you will need to resize and/or reposition the view of the structure in the window in order to see the part on which you want to work.

For details see **"Resizing and Repositioning a Structure"** in the **Introduction Manual – 2D Tutorial**

Selecting Objects

You also need to be familiar with the different options available for selecting objects. In this example you will simply be told to select or deselect the required objects, it is up to you to use whichever means you prefer.

For details see **"Acquainting yourself with selection"** in the **Introduction Manual – 2D Tutorial**.

Defining Geometry - Overview

The following outline explains how we are going to create the basic frame geometry:

1. Build half the roof.
2. Move the roof into its final position.
3. Copy a part of the roof to create the foundation joints.
4. Add a column to the global origin.
5. Add one of the cantilever members to that column.
6. Copy and generate new columns and cantilevers along the **XZ**-face.
7. Use Member Links to generate beams parallel to the **X**-axis.
8. Add braces to the **ZX**-face.
9. Heighten the first story.
10. Reflect and copy the half-built structure.

We will also edit other geometrical attributes, supports, releases, properties and such like as we go along.

Defining the roof geometry

1. Use the **Grids Definitions** dialog to create a Basic Grid with grid lines along the X-axis at 0, 6, 12 and 18 m and along the Y-axis at 0 and 12 m. Having done this zoom to the extents of the grid.

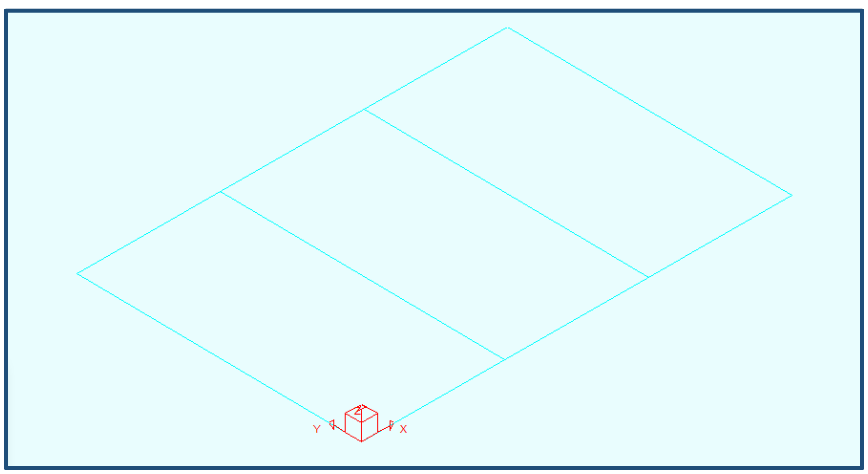


Figure 5.

2. Now we can proceed to define the members using this grid. We could do this one member at a time. However **S-FRAME** allows us to define ranges of members in a single operation. We shall use one of these now.
3. Since our sections are American, we should also use American steel for the material, therefore choose Def Steel (US) from the list in the **Views and Grids** toolbar.
4. Click the **Member Definition Tool** from the **Geometry Toolbox**.
5. Now click on the Top View and Zoom to grid extents icons from the **Aerial** window.

The **Data Bar** changes to include an extra list of options.

6. In the **Data Bar** choose the Rectangular option from this list.
7. Now drag a rubber rectangle round the entire grid. This will automatically create separate members between all the grid intersection points in both the **X**- and **Y**-direction and will also create the joints at their ends.

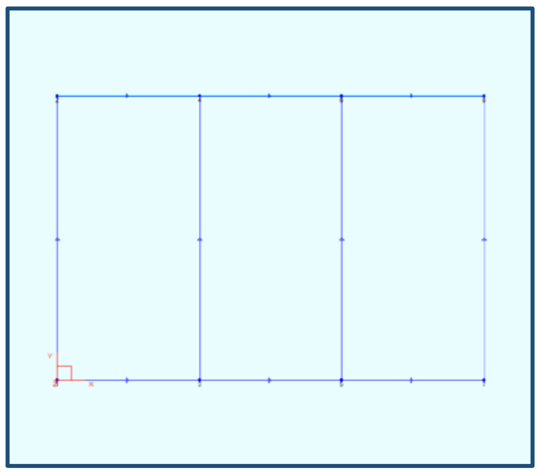


Figure 6.

Setting section properties

Now we shall define the properties that we want to use for all members of our frame, both for those we have defined already and for those we are yet to define.

1. Click the **Section Properties Tool** from the **Geometry Toolbox**.
2. Use this to include the following AISC sections in the **local database** for this model:
   - **W21X147** – roof beams (I section),
   - **HSS3.5X.216** – roof braces and side braces (hollow section),
   - **W10X49** – columns (I section),
   - **S6X17.25** – cantilevers (I section).

By thinking ahead in this way and listing all the sections, we need in the structure; we can save time by defining all the properties at once. Otherwise, we would have to access the **Section Properties Tool** dialog repeatedly to add new sections as we came to them.

Defining roof bracing

Now we want to add K bracing to the left-hand bay of our roof. In order to do this, we shall see another feature of **S-FRAME**, that to split an existing member in two when you create a member that intersects with it.

1. In order to use this feature, you need to switch the grid off (or lock it). For this example switch the grid off now.
2. Click the **Member Definition Tool** from the **Geometry Toolbox**.
3. Choose the **HSS3.5X.216** section size from the list in the **Views and Grids** toolbar.

Setting the section here means that this section will be used for new members we create. Again this saves time since we would otherwise have to change the section to the correct one after creating the members.

4. Click on joint 3 to start creating the bracing from that point, then click on the member which runs between joints 1 and 2. You will see the **Subdivide Connecting Member** dialog.

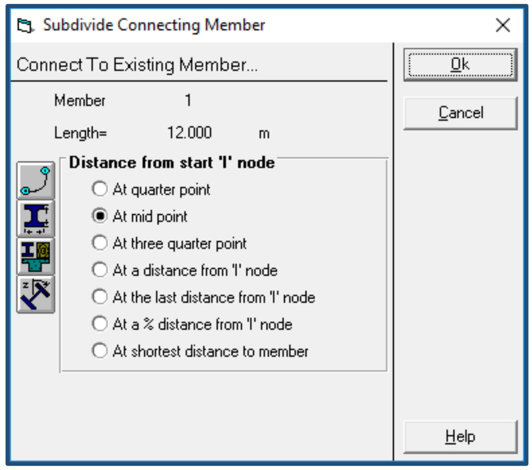


Figure 7.

This allows you to specify exactly where **S-FRAME** has to break the existing member.

5. Ensure that the **At mid-point** option is selected and then click **OK**.

The buttons down the left-hand side of the dialog relate to the member you are creating, and not to that which you are splitting. Each portion of the split member retains the details of the un-split member. **S-FRAME** will also adjust any loads to take account of the split if you have checked the **Always preserve loads during element subdivision** option in the **Preferences** sheet).

6. Create another bracing member running from joint 9 to joint 4. When you have done, you should have achieved the layout below.

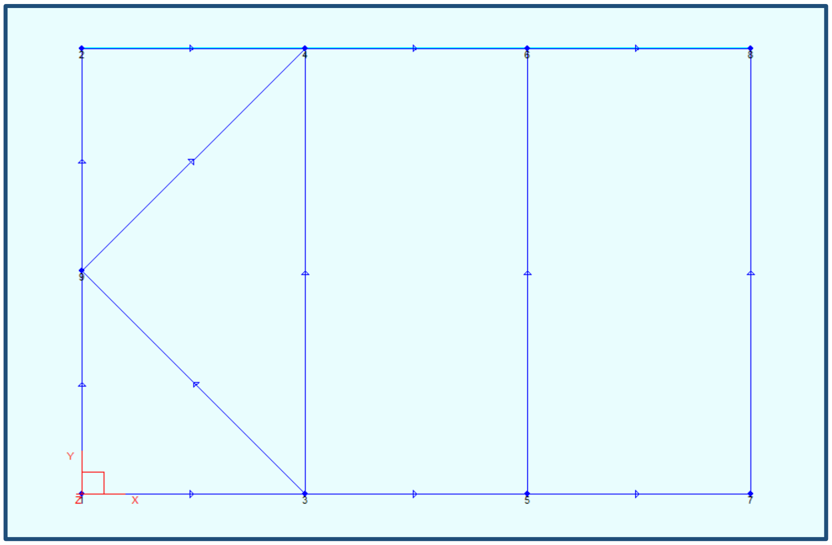


Figure 8.

Saving the Model

Now save the model as **Tutorial 2.tel**.

Moving the roof to its final position

We have now defined of one-half of our roof but at an elevation of zero meters. We now need to move the members and joints that we have defined up to their final level. First change back to the southwest isometric view of the structure.

In an actual model, you would create the roof at its finished level.

If you have followed the previous instructions exactly, everything in your structure should already be selected. If they are not, then click the Select All icon from the **Edit** toolbar.

1. Click the **Move** icon ( ) from the **Edit** toolbar.
2. Now translate the roof by **7 m** along the **Z**-axis.

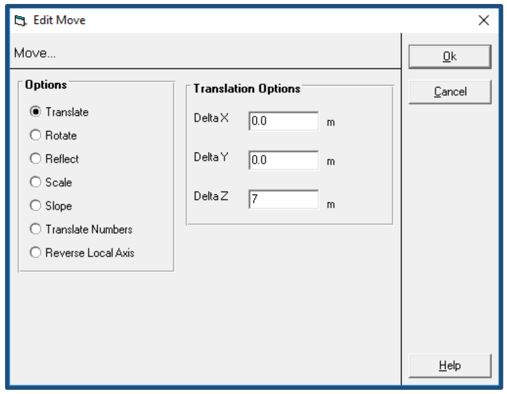


Figure 9.

Creating the columns

We have now defined the roof steelwork and can proceed to define the columns. To do this, we need to be able to see the existing roof joints and the grid.

1. Switch the grid on again and adjust the display to show the grid extents (this should allow you to see the grid and all the joints along the edge of the roof).
2. Make sure that the **Member Definition Tool** is selected.
3. Choose the **W10X49** section size from the list in the **Views and Grids** toolbar.
4. Now click on the grid intersection point immediately below joint 1 and create your column from this point to joint 1.

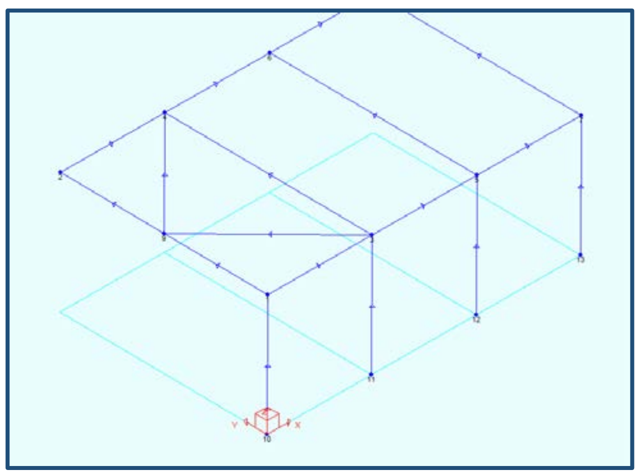


Figure 10.

Creating the columns

5. Repeat this process to create columns from the relevant grid intersection point up to joints 3, 5 and 7. When you have done this, change back to the **W21X147** section which we shall use for the side beams.
6. Click on the column which runs from joint 10 to joint 1. Again you will see the **Subdivide Connecting Member** dialog.

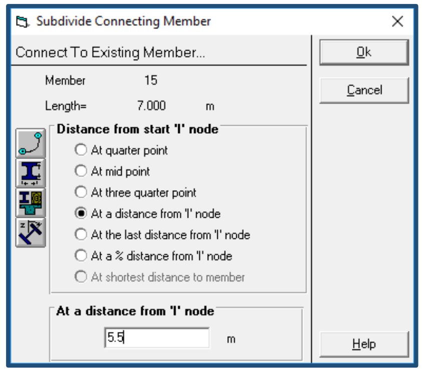


Figure 11.

7. This time choose the **At a distance from 'I' node** option and then enter **5.5** into the new box that appears. Click **OK** to split the column at 5.5 m from its base.
8. Repeat steps 5 and 6 for the column which runs between joints 13 and 7. This creates a single member which runs between the first and last columns of the model.

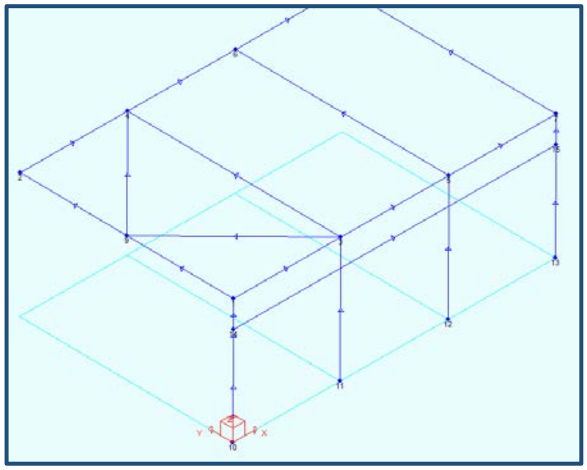


Figure 12.

9. However, we want a series of separate members which connect to each of the internal columns as well. Fortunately, **S-FRAME** can do this automatically. Simply click on the Connect Intersecting Members icon ( ) on the **Edit** toolbar and **S-FRAME** works out which of the selected members intersect and splits them appropriately.

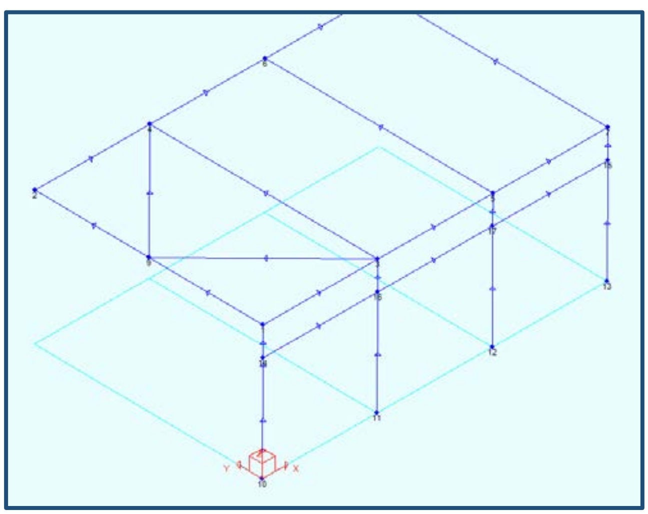


Figure 13.

Adding the cantilever

Now we shall add the cantilever to the joint part way up the column.

1. Click the Joints Tool from the **Geometry Toolbox** and then click on joint 14 – the **Data Bar** will show its coordinates.

If you were to click on a joint which had no members connected to it, then this would **delete the node**.

2. Now type **1.5** into the Y box in the **Data Bar** and click Accept Data. This creates a new joint 1.5 m in the **Y**-direction from joint 14.
3. Choose the **S6X17.25** section from the list in the **Views and Grids** toolbar.
4. Click the Member Definition Tool from the **Geometry Toolbox** and create a new member between joint 14 and the new joint (joint 18).

Copying cantilever along the XZ face

We now need to copy the cantilever to the other columns along the XZ-face. There are several ways of doing this. However, we shall use the **Clone Tool.**

1. **Unselect** everything apart from the cantilever element (including for the purposes of this example the joints at the cantilever's ends).
2. Now click the **Copy to clipboard** icon from the **Edit** toolbar to copy the details of the selected members to the clipboard.
3. Now click the Clone Tool from the **Geometry Toolbox** and then right click the tool to view its dialog.

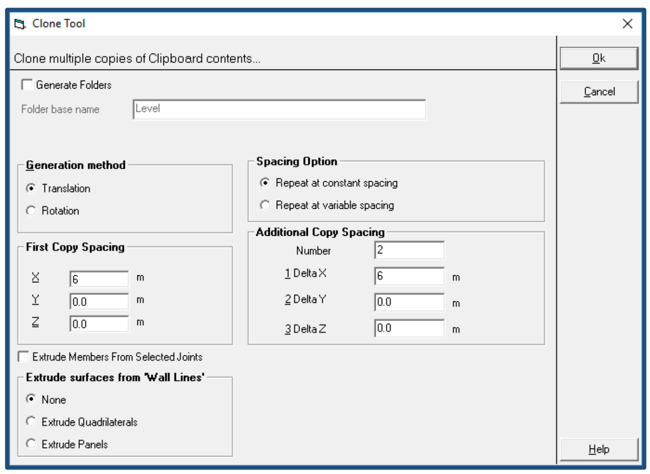


Figure 14.

We could just use the **Generate Tool** and click on the joints at the base of the columns we want to generate. We used this feature in the **Tutorial (2D)**. However, the tool dialog gives a more elegant solution.

4. Type **6** as the **Repeat at X value** – this will generate our first copy at 6 m along the **X**-axis from the current user coordinate system origin.

5. Type **2** as the **Repeat again at Number value** – this will generate two further copies at the Repeat again at **X**-, **Y**- and **Z**- values. For our purposes, we only need to type **6** as the Repeat again at 1 Delta X value.
6. Click **OK** to generate the copies.

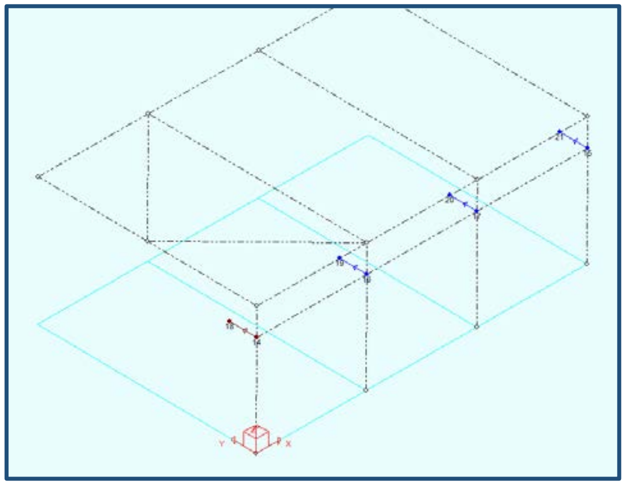


Figure 15.

This is still a trivial use of the power of this tool's dialog which can be used to great effect on towers, sloping foundations and soon.

Although we only copied the members, **S-FRAME** has automatically created the joints that these require. We can tell which joints have been created since they are selected.

Defining the wall bracing

Now we shall define the wall bracing.

1. Choose the **HSS3.5X.216** section from the list in the **Views and Grids** toolbar.
2. Click the **Member Definition Tool** and add the braces as shown below to the left-most bay of the **XZ**-face.

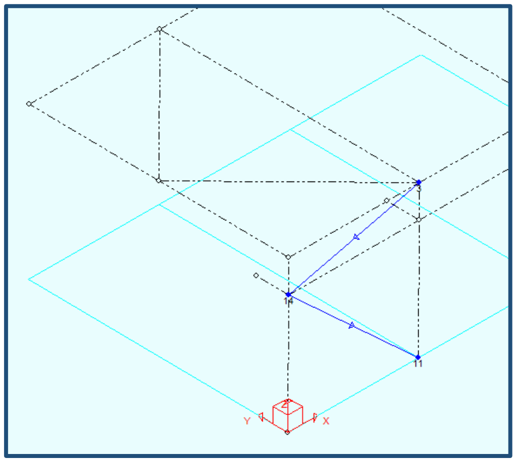


Figure 16.

Saving the structure

Save the structure at this point.

Defining the supports

Now we shall proceed to define the supports for the structure. Since all these lie on the **XZ**-plane and at a single level an alternate view of the structure would be appropriate.

1. Click the **Front View** icon in the **Aerial** window to view the structure from that direction

> With the **Options** we have specified the **Geometry** window automatically resizes the structure to fit the window.

2. Select all the nodes at the base level.
3. Click the **Supports Tool** from the **Geometry Toolbox**.
4. Now drag to select all four foundation nodes. This will create fully fixed supports at these locations.
5. We want the inner supports to be completely free to rotate and to have restraint against translation only in the **X**-axis and **Y**-axis directions (we shall add springs in the **Z**-axis in a minute). Make these settings for the **Support Tool**.

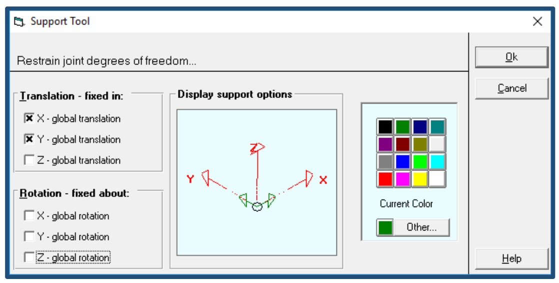


Figure 17.

6. Now identify the inner two supports again to replace the existing fully fixed supports with those with these settings. When you have done this select the entire structure and change back to the southwest isometric view to see the restraint that we have provided at each joint.

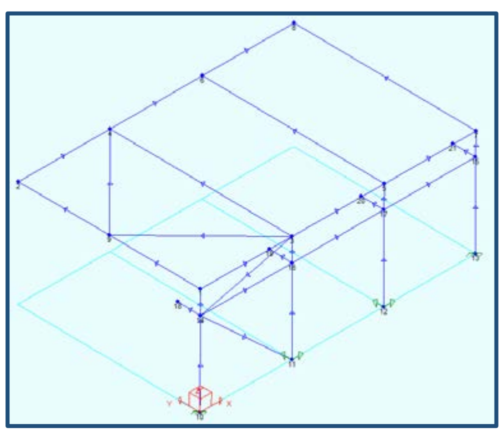


Figure 18.

Single-headed arrows indicate restraint against movement in that direction. **S-FRAME** uses double headed arrows to indicate restraint against torsion.

Defining springs

Now we can add the springs to provide support to the inner pair of columns.

1. Click the **Ground Spring Tool** from the **Geometry Toolbox**.
2. In the **Data Bar** ensure that the Z – Translation option is set and enter a stiffness of **42000**.
3. Now click the nodes at the bottom of the two inner columns to create springs with these properties at these points.

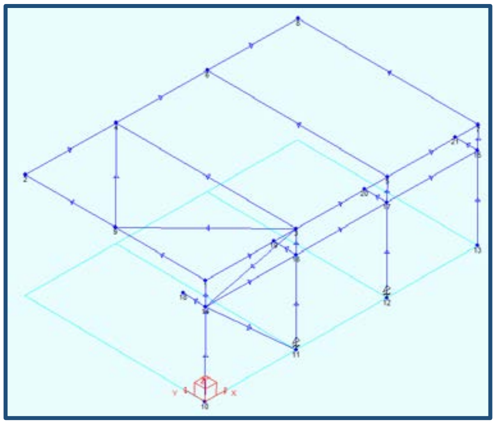


Figure 19.

Adding releases

At the moment all the members are fully restrained at each end. We want the wall and roof braces to be pinned at each end.

1. We first need to select these and deselect the other objects. **The special selection** offers one way of doing this. Click **Select > Special...**
2. We want to select all the braces. The easiest way to do this is to check **Section Numbers** and then type **2** into the Section range box and then click **OK**.

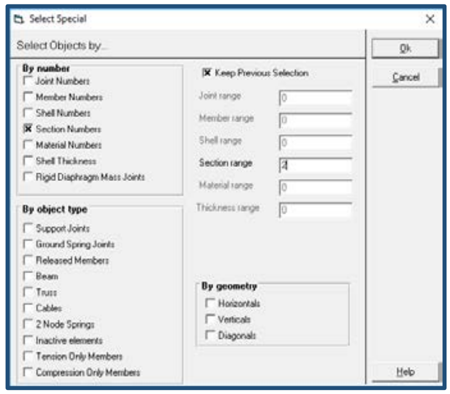


Figure 20.

This allows us to choose the members we want to select in many ways. We can select only the members that match the criteria we specify (clearing all other selections), or add them to the already selected objects.

If instead, we had checked the **By geometry** – **Diagonals** box, then this would have only selected the wall braces since the roof braces are horizontal at the moment.

3. Click the **Member Release Tool** from the **Geometry Toolbox** and then view its settings.

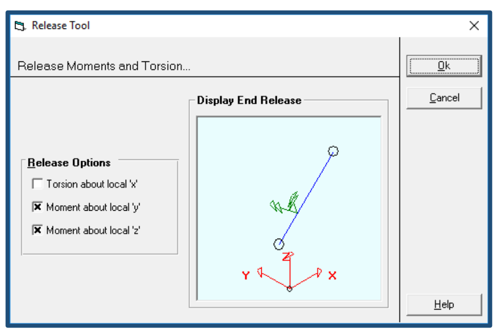


Figure 21.

You should never set both ends of a member to have the torsion component released. If you do, then you are making the member free to rotate about its axis and the analysis may fail or give spurious results.

4. These are the settings we want to apply so click **OK**.
5. Identify the selected members to apply these releases to them.

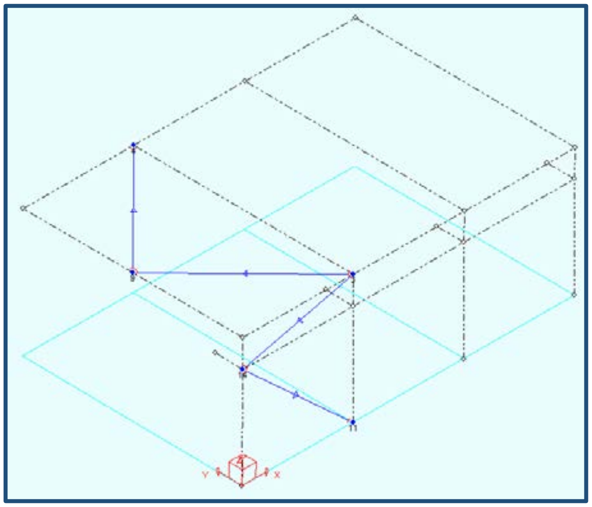


Figure 22.

The display as you see it above is indicative only. It simply tells you that releases of some description have been applied to these member ends. **S-FRAME** allows you to choose to see to which local axes releases have been applied.

6. To do this click **Options** > **Releases**. Double arrows indicate a release in that local axis direction at the end of the member nearest the arrows.

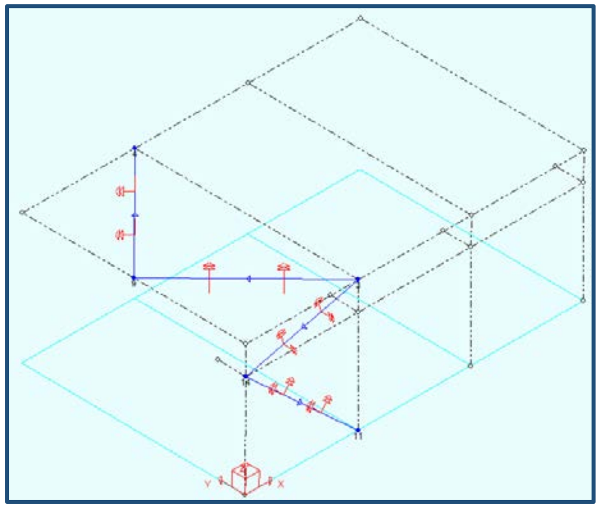


Figure 23.

Shearing the roof

We have now nearly completed one-half of our structure, apart from adding a 1 in 10 rise to the roof. The easiest way to do this is to use the **Move** dialog.

1. Select just four joints that will eventually lie in the center of the roof and the joint at the sharp end of the K-brace.

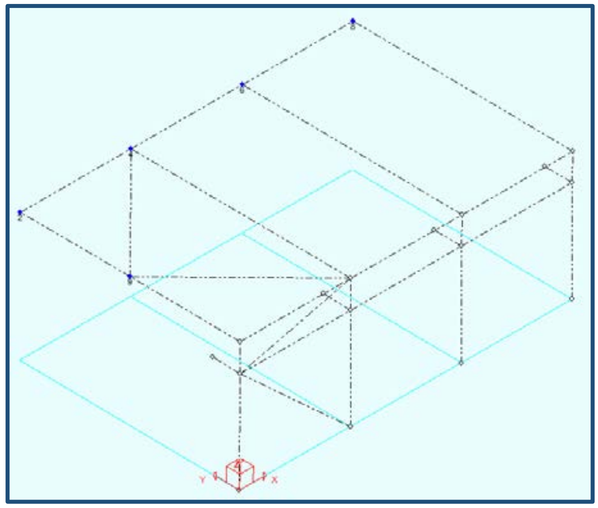


Figure 24.

2. Now click the **Move** icon from the **Edit** toolbar
3. Pick the Shear option, In the ZY Plane and enter **0.1** into the **Z'=Z + ……… x Y**. edit box. This will add 0.1 times the distance along the **Y**-axis to the **Z**-coordinate of all the selected nodes. Click **OK** and **S-FRAME** will perform the shear.

4. Now select the entire structure and zoom to see the structure extends.

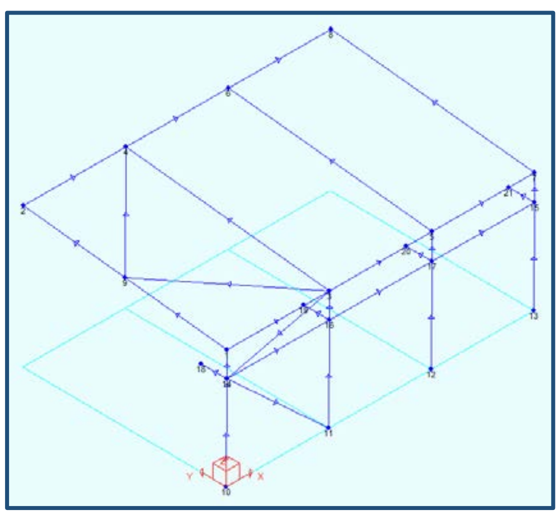


Figure 25.

Creating the rest of the structure

We have now defined all the details for one-half of our frame. We shall now create the remainder of the structure by copying the information we have already defined.

1. Click the **User Coordinate System Tool** from the **Geometry Toolbox**.
2. Click on joint 2 (at the end of the left-hand rafter) to move the user coordinate system to that joint.

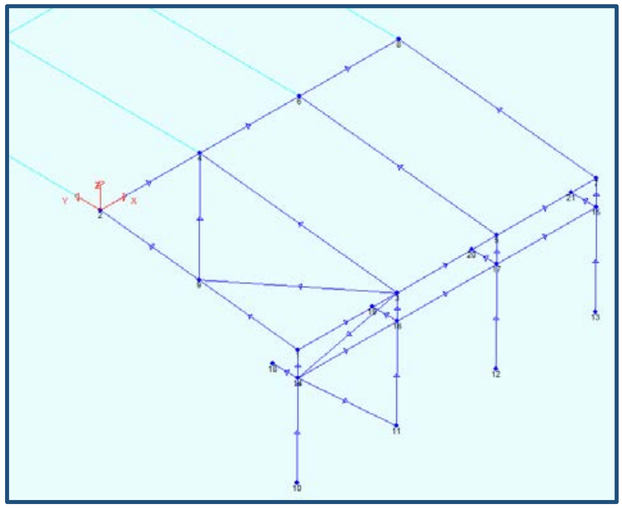


Figure 26.

When you move the user coordinate system to this new position, the grid moves with it.

3. Now click the **Move** icon from the **Edit** toolbar.
4. Choose the **Reflect** option, specify that this is **About the ZX Plane**, check the option to **Make a Copy** and then click **OK**.

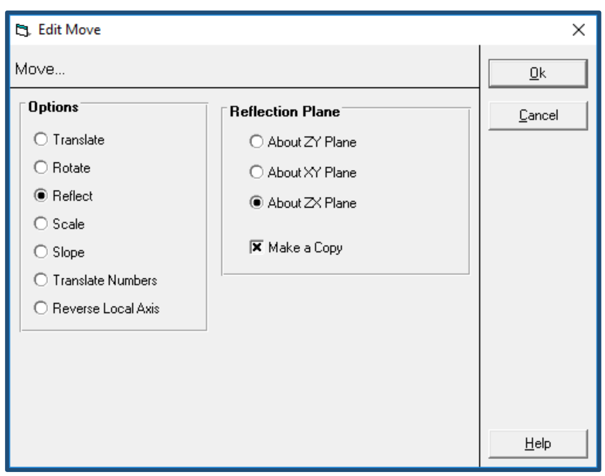


Figure 27.

5. Now click on the original user coordinate system origin joint (joint 10) to return the coordinate system to global.

This process has reflected all the joints and members; it has also reflected all the attributes that we have applied to the structure (releases, supports and so on) as well. You will find it best to use this powerful option as late in the day as possible so that it includes as much detail as possible.

6. Now zoom to the extents of the structure.

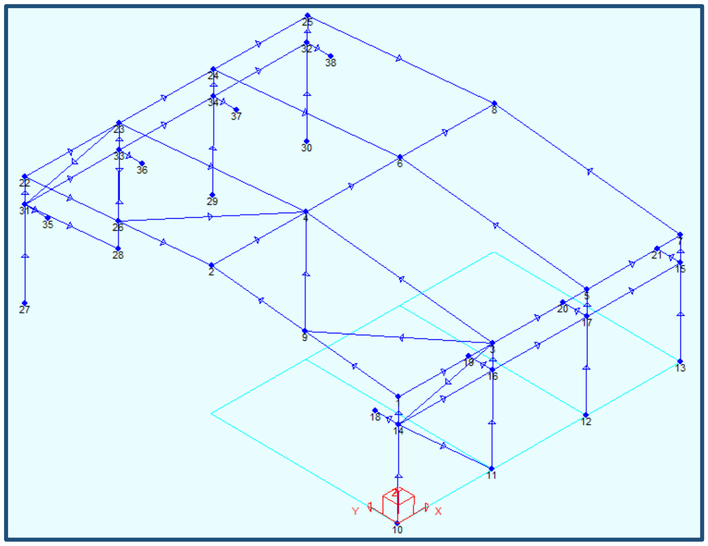


Figure 28.

7. Now that we have defined our model switch the grid and the display of joint numbers off.

Rendering the structure

Now that we have completed the definition of our structure, it is a good time to review it to ensure that the details we have defined are what we expect. A good way to review the structure is to render it, in this way we can see that the members look correct, and also that they have the correct orientation.

1. On the **Status Bar** below, switch the Render Model option to ON (On) or click **View** > **Render …**

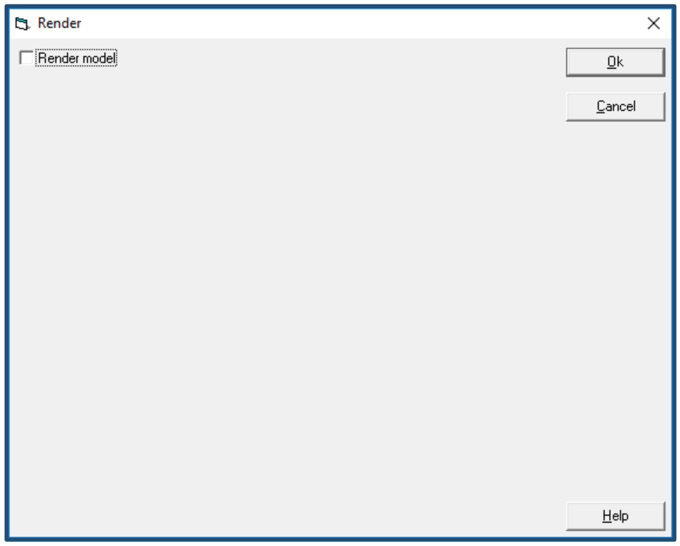


Figure 29.

2. Click the **Render model** option, change the **Length shrink factor** to 0.75, set the option to **Render Selected** members only.

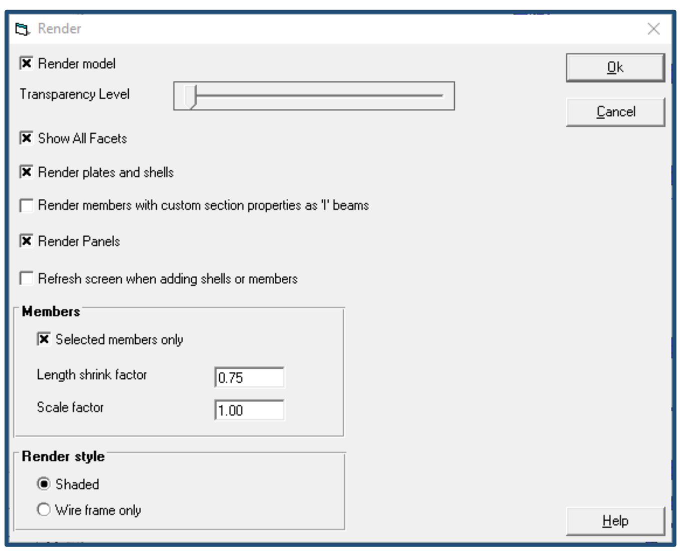


Figure 30.

3. Click **OK** to render the model with these settings.

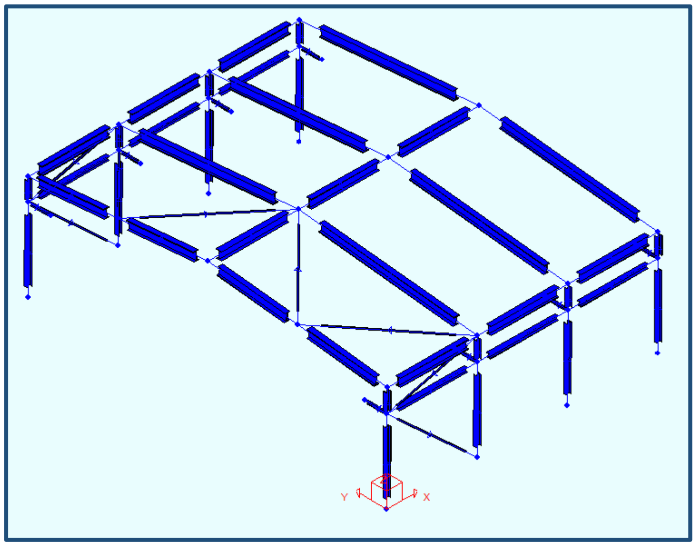


Figure 31.

We suggest that you always check **Selected members only**. Otherwise **S-FRAME** will render all sections irrespective of their selection state.

4. Zoom in to one of the front columns.

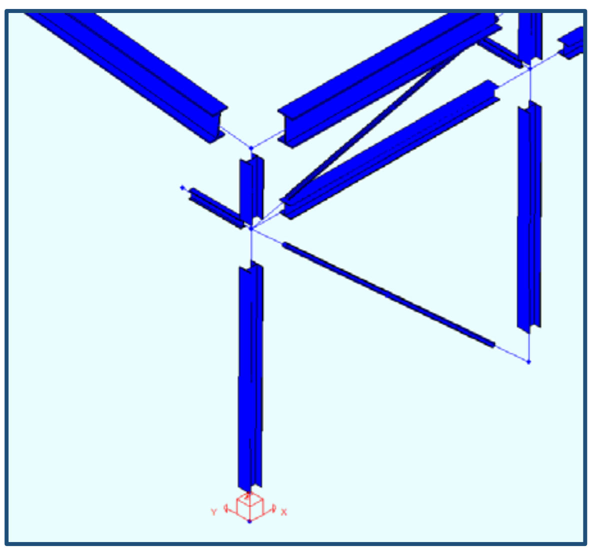


Figure 32.

As you can see these have the wrong orientation, they are oriented to take their bending to which they are obviously subject about their minor axis.

5. Click the **Member Axis Orientation Tool** ( ) from the **Geometry Toolbox** and then set the tool's properties.
6. Set the **Gamma angle** to 90° and click **OK.**

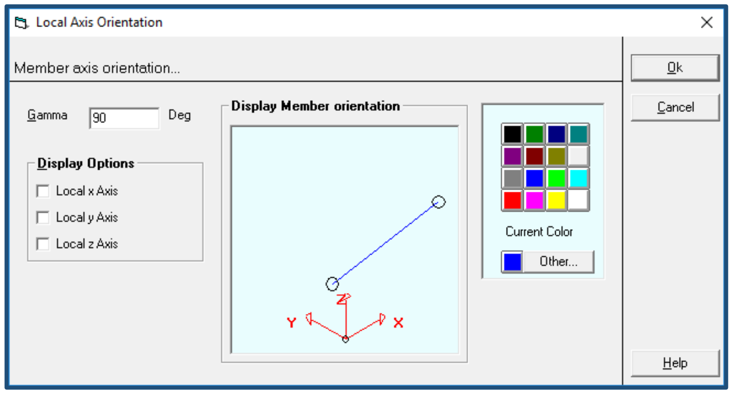


Figure 33.

7. Set this gamma angle for each column (top and bottom). If you zoom in to the same area as before, you will now see that the columns have the correct orientation.

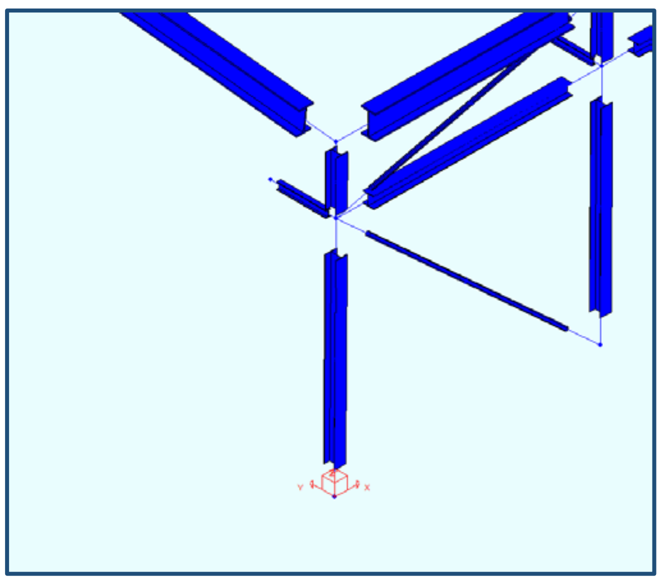


Figure 34.

8. The rendering is fast enough to allow you to work with it switched on at all times. However, you may find that this obscures the view of other information. Switch rendering off now.

Saving the structure

We have defined the entire structure, now is a good time to save it.

You can view and change the information we have defined graphically using the **Spreadsheet** window. The various pages show the details for various parts of the structure. You might want to investigate these at this point.

Structuring the model

Now that we have defined all the members in the model and their appropriate details we shall use the **Groups pane** to add some structure to our model. You can group a model in any way that you want, however, for this tutorial, we shall group the members of our model into the following categories:

- Rafters
- Columns
- Cantilevers
- Link beams
- Roof bracing
- Side bracing

Although you can create groups as-you-go, it is more efficient to define the group structure that you want to use, define the group folders and then add your members to these folders.

1. In the Groups pane, right-click on **My Structure** and click **New Group** from the context menu. Type in the name of the group – Rafters.
2. Repeat step 1 and define the following groups:
- Columns
- Cantilevers
- Link beams
- Bracings
- Side Bracing
3. In the **Groups pane**, right-click on Bracings and click **New Group** from the context menu. Type in the name of the group – Roof bracing. Your Groups pane should now look like this:

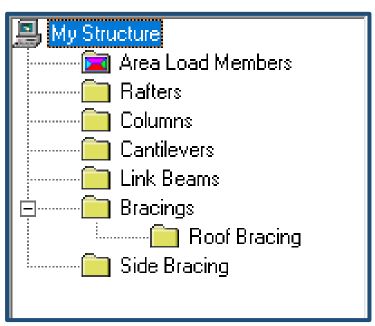


Figure 35.

4. In the **Status Bar** ensure that the **Automatically update open folder** icon (Off) is switched **Off.**
5. Click on the Rafters folder; you will see that the folder opens, and also that all the selections are cleared. This is because the Rafters folder does not yet contain any elements.

6. Select all the rafters and joints as shown below.

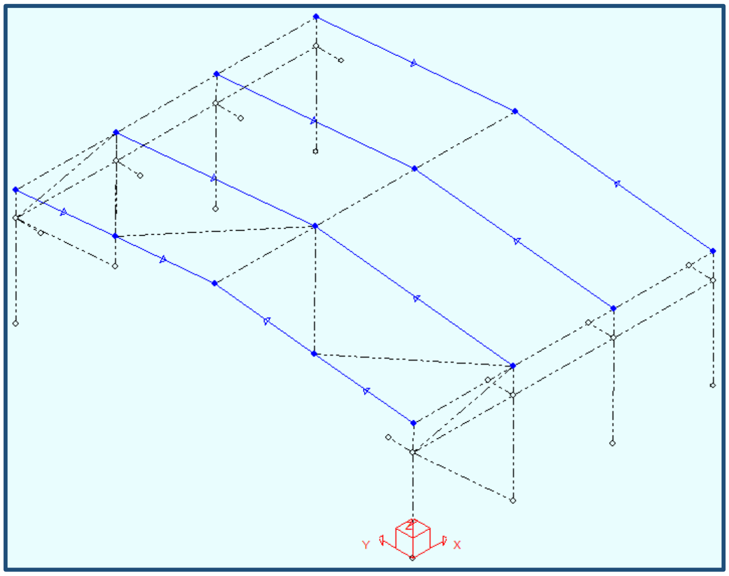


Figure 36.

For this group select the joints at the ends of the rafters for reasons that will become apparent later. You can select the joints automatically by checking the **Include joints when selecting/unselecting objects** in the **Preferences** dialog.

7. Now right click the Rafters folder and choose **Update Group** from the context menu. **S-FRAME** adds the selected members into this group, and its as easy as that.
8. If you now click on **My Structure,** you will see that everything is selected, and if you then click on Rafters you will return to the selection as shown above.

My Structure automatically contains every element in your model, and you cannot change this.

9. **S-FRAME** can automatically set the contents of a folder to match your current selection. We shall look at this now. In the **Status Bar** ensure that the Automatically update open folder icon (On) is switched **On.**

10. Click the Columns folder, and then select the columns as shown below.

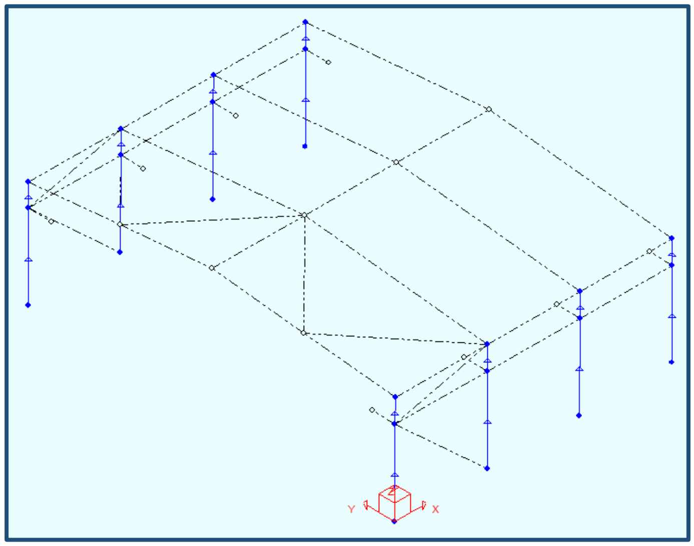


Figure 37.

The quickest way to do this is to do a **Select > Special** and then select by **Geometry – Verticals.**

And that's it. **S-FRAME** automatically sets the current folder to contain all the selected elements.

11. Use either of the methods above to create the following group contents:
    - Cantilevers

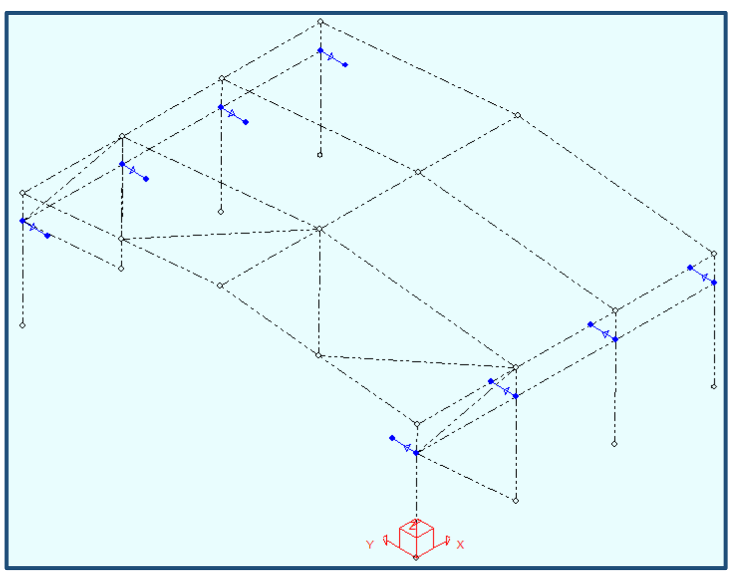


Figure 38.

For this group select the joints at the ends of the cantilevers, since we later need to add joint loads to these.

- Link Beams

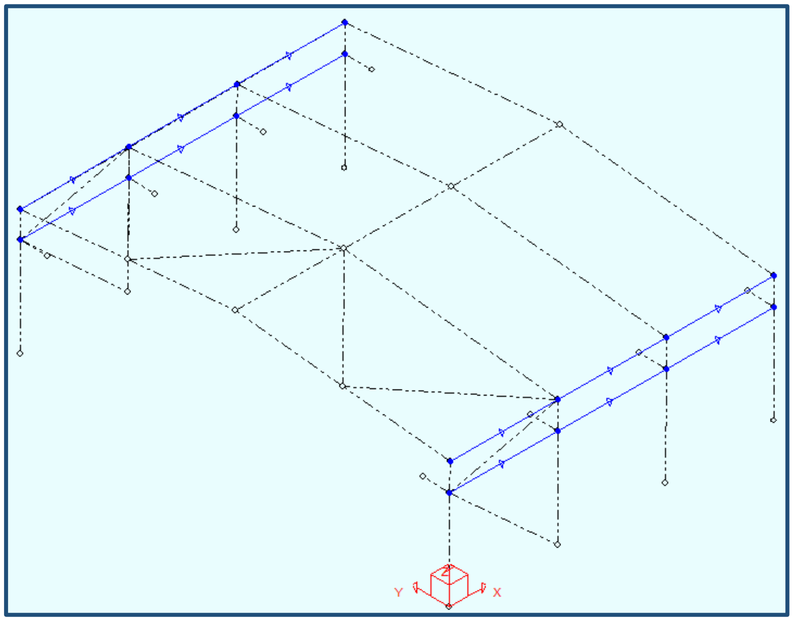


Figure 39.

- Roof Bracing

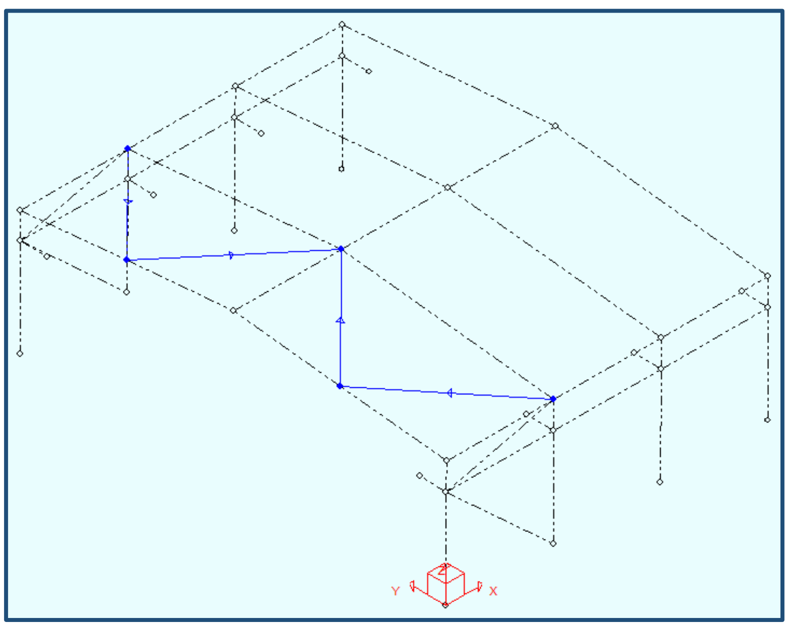


Figure 40

- Side Bracing

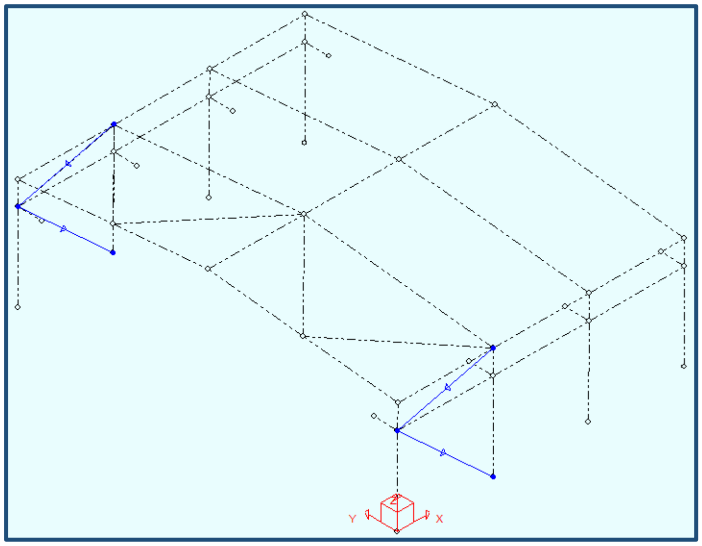


Figure 41.

12. Now that we have defined all our grouping let's take another look at the group structure. You might remember that it is:

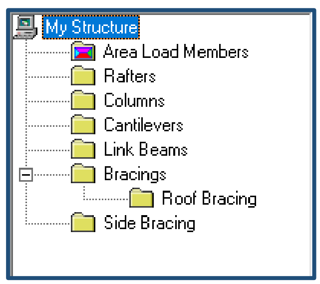


Figure 42.

Ideally, the Side bracing folder should be in the Bracings folder together with the Roof bracing, and it would be nice for the Bracings folder to contain all the bracing members, as well as the two sub-groups which allow you to look at a particular bracing type. This is easy as **S-FRAME** provides some sophisticated folder handling facilities.

13. Right-click on the Side bracing folder, and select **Move Group**. Select move to Bracings and click **OK**.

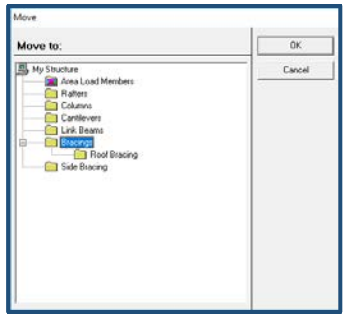


Figure 43

S-FRAME moves the Side bracing folder into the Bracings one.

14. If you click on the Bracings folder, you will see that no elements are selected. This is because folders do not automatically inherit the elements contained in any folders that they contain. However, it is easy to copy the elements from such folders up the folder tree.
15. Right-click on the Side bracing folder and select copy group. Select **Copy Group,** then the Bracings folder and click **OK**. If you select the Bracings folder, you will see that it now shows the side bracings. This time **S-FRAME** has copied the contents of the Side bracing folder into the Bracings one (you can check this by clicking on the Side bracing folder to see that it still contains the side bracings).
16. Repeat step 15, this time copying the contents of the Roof bracings folder into the Bracings one. The final group layout should be:

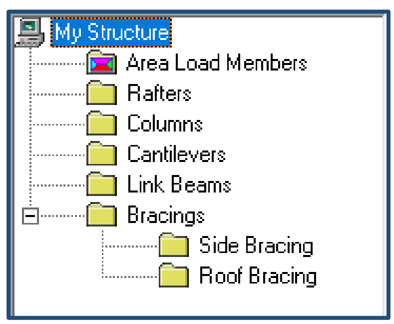


Figure 44.

Saving the structure

We have now defined the entire structure and the grouping that we want to apply to it. All that remains is to define the loading, so now is a good time to save the structure again.

Loading the structure

Now that we have defined our structure completely we can proceed to load it. To do this, we need to change to the Loads window.

We can use the groups that we have just defined to speed up the definition of the loading. Since we do not want to risk inadvertently changing the grouping that we have just done ensure that the **Automatically update open folder** icon is switched to **Off.**

Setting display options – Loads window

1. Ensure that the **Edit** toolbar, the **Load Tools** toolbox and the **Views and Grids** toolbar are shown for the Loads window.
2. Ensure that the **Title Bar, Zoom Extents, Local 'x' Axis Joints, Redraw, Status Bar, Floating point format** and **Coordinate System** interface options are set.

Creating the Self-weight load case

Define the self-weight load case for this model in exactly the same way as for the 2D model.

For a 3D model, you need to apply the self-weight loads in the Z-axis direction.

For further information **see "Defining self-weights"** in the **Introduction Manual – 2D Tutorial**

Creating the Roof Dead Load case

1. Create a new load case titled **Roof Dead Loads**.
2. Select only the roof members.

You can use the groups that we have just defined to do this.

3. Now apply **Full Uniform Global loads** (not global projected loads!) of value **-0.7** in the Z-global direction to the inner roof beams, and similar loads with a value of **-0.35** to the outer ones.

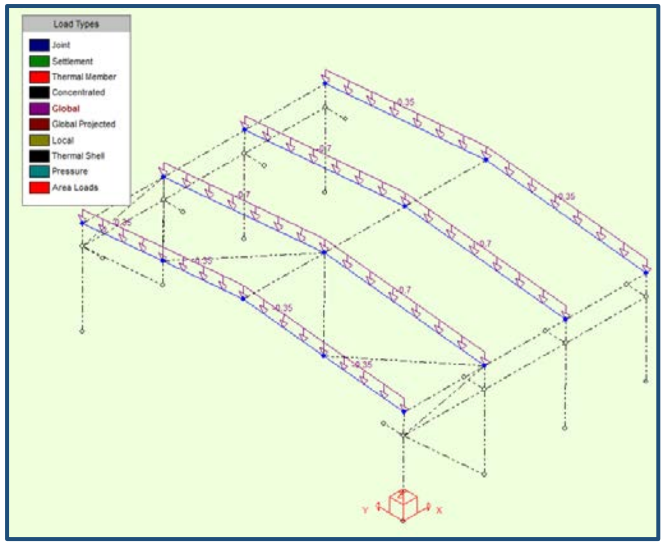


Figure 45.

The quickest way to do this is to apply the load of -0.35 to all the rafters, and then to apply the same load again to the inner ones. **S-FRAME** automatically adds the two loads together.

You might find it easier to apply the loads if you move to the standard **Top View** first.

Creating the Snow Load case

1. Create a new load case titled **Snow Loads**.
2. Now apply **Partial Varying Global Projected** loads with the settings a shown below to the inner roof beams at one side of the structure.

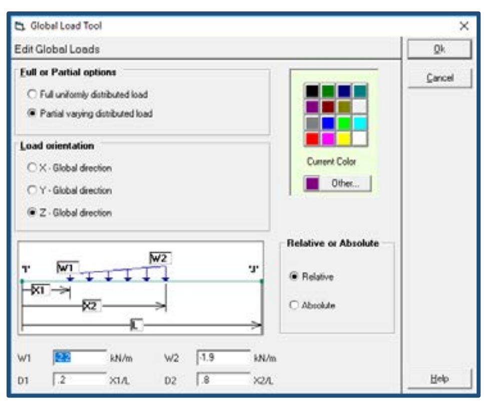


Figure 46.

Define further loads with the settings below, to the rear outer one.

To define the load on the front rafter (which is split by the K bracing), we could calculate the values of the load at the point where the bracing is split, and then apply two separate loads. Fortunately, **S-FRAME** provides a much more elegant way of doing this. With the **Data Bar** set as shown above click on joint 1, then move the pointer and click on joint 2. **S-FRAME** automatically applies the load over the entire length between these two joints (which is the same length as for the other rafters) and performs all the calculations for you in order to apply the correct loads to their respective members.

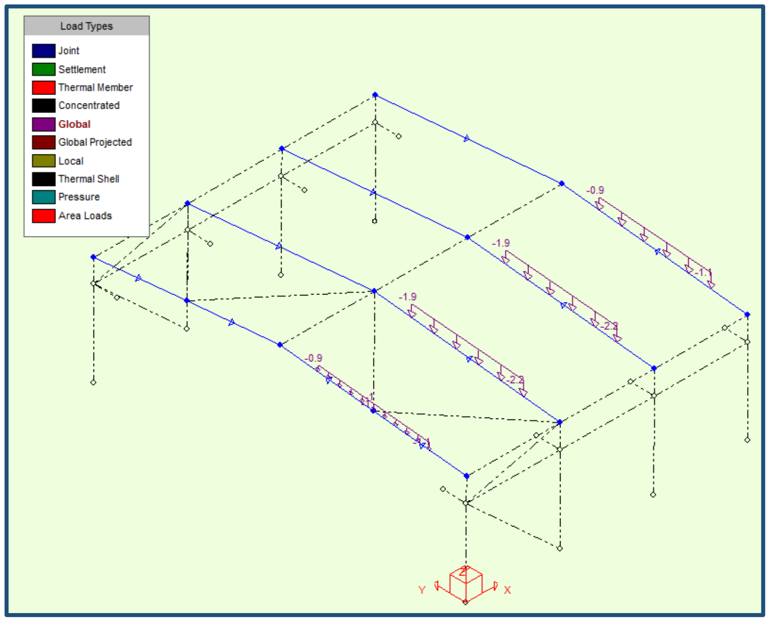


Figure 47.

Creating the Wind Load case

1. Create a new load case titled **Wind Loads**.
2. Click the **Local Load Tool** from the **Loads Toolbox**.
3. Now apply **Full Uniform Local** loads of value 0.7 in the direction shown below (using a local axis direction) to the back columns, and similar loads with a value of 1.25 to the front ones, ensure that the loads are applied in the direction shown.

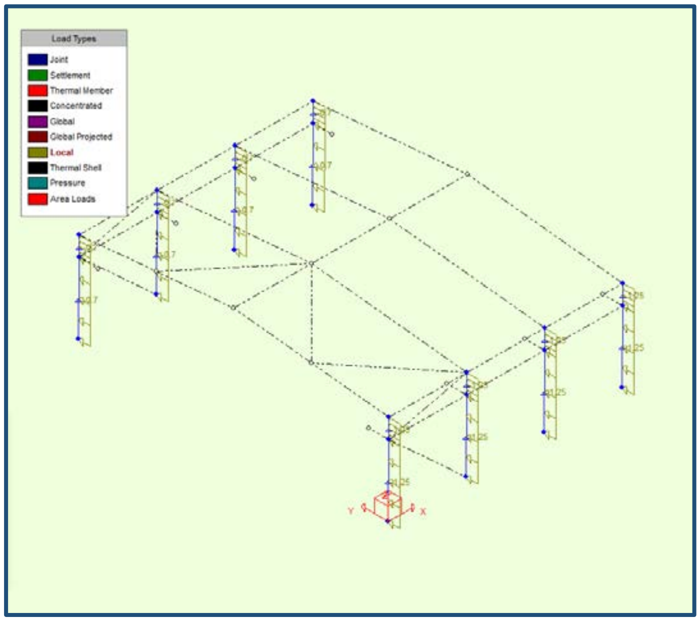


Figure 48.

In this case, it would be better to define the loads using the global axis system, however for this example use the member's local axis system. This shows:

- the difficulty of working with the local axis system,
- that the global and local axis system do not align.

You can switch back to the **Geometry** window and set the **Local Axis Orientation Tool** to show the local axes of the members to determine the appropriate direction and sign.

Creating the Crane Load case

1. Create a new load case titled **Crane Load**.
2. In the **Aerial** window click the Left View icon.
3. Now pick the group which has the nodes at the end of the cantilevers selected.
4. Click the **Joint Load Tool** choose a **Z**-Force of value **-11**.
5. Now draw a window around the nodes at the ends of the cantilevers.

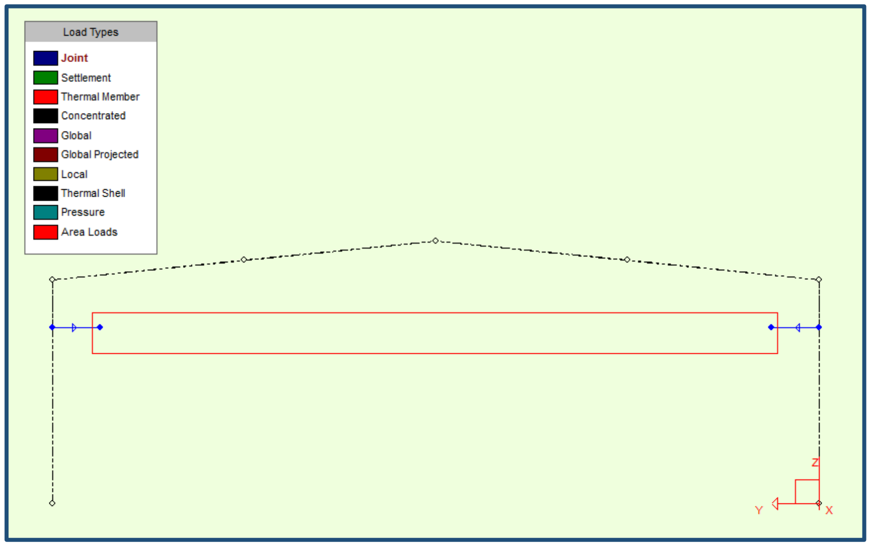


Figure 49.

6. In the **Data Bar** now choose an X-Force of value **3**. Again window the same area.

The display no longer shows the details of the force that we have just applied in the **Z**-direction. Don't worry, remember that the load is still there. **S-FRAME** normally only shows forces of the type you are applying, and in the direction you are applying them. This keeps screen clutter to a minimum. You can override this behavior and force the display to show the load types and directions that you require.

To view all the loads on your structure click the Display Joint Loads icon and the Display Member Loads icon from the **Status Bar**. Click these two buttons (switch them to **On**) now.

As you set **Display Joint Loads** to **On**, you will see that your **Z**-Force reappears.

The other loads that you have defined previously do not appear as they are in different load cases.

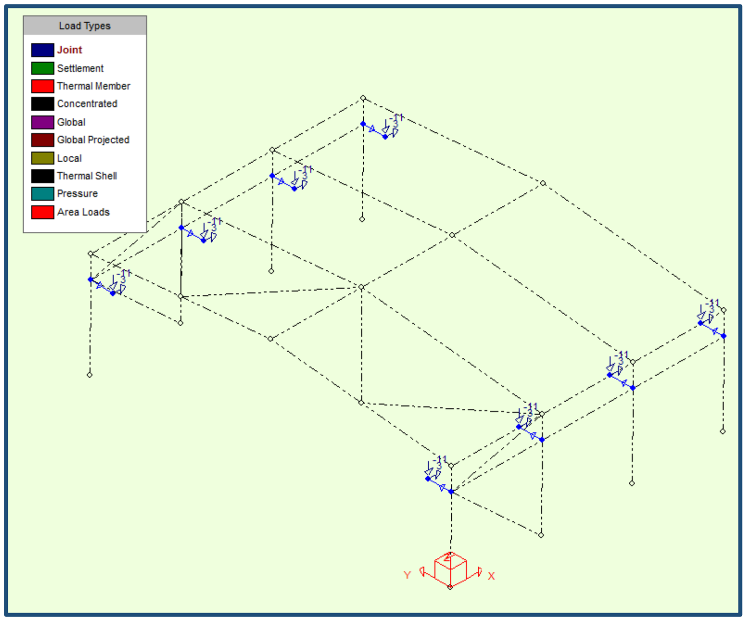


Figure 50.

Defining Combinations

Now that we have defined the load on our model, we shall create a series of combinations – these are the actual loads to which our model may be subjected.

Click **Edit > Load Combinations...** and then define the combinations shown below.

Comb No	Comb Description	Active Status 0=Yes,1=No	Notional X-Factor	Notional Y-Factor	Load Case	Comb Factor	Load Case	Comb Factor	Load Case	Comb Factor
1	SWT + Dead + Sno	Yes	0	0	Self-Weight	1.4	Roof Dead Lo	1.4	Snow Loads	1.6
2	SWT + Dead + Winc	Yes	0	0	Self-Weight	1.2	Roof Dead Lo	1.2	Wind Loads	1.2
3	SWT + Dead + Crar	Yes	0	0	Self-Weight	1.4	Roof Dead Lo	1.4	Crane Loads	1.6

Figure 51.

Saving the structure

We have loaded the structure and defined our combinations, now is a good time to save it. If you want to take a break from this tutorial, then now would be a good time to do so.

Analyze the structure

Now that we have defined our model, defined a structure for it and the views that we want to use, let's analyze it. Run a linear static analysis.

Setting Graphical Results window options

Before we continue to review the results we need to make sure that the configuration of the **Graphical Results** window is correct. If you have carried straight on with this example from the 2D one, then they should be. If not then make the same settings as for the 2D Tutorial.

If you cannot see the Structure pane down the left side of the **Graphical Results** window, click **View > Groups** to enable it. We shall use this pane in reviewing our results.

For further information see **"Setting Graphical Results window options"** in the **Introduction Manual – 2D Tutorial**

You would usually ensure that the grouping that you defined for loading purposes also included any additional groups that you need for output purposes. In reality, this usually turns out to be the same thing – similarly loaded members are the ones you will want to review as a group in the results and output.

Reviewing the results

Once you have defined your structure, loaded it, defined any combinations you require and analyzed it, then you will need to understand the results of that analysis. In doing this, you will find that the groups you have defined are invaluable and that they allow you to concentrate on particular members in your structure. All the features of a 2D analysis are available, and we shall not go through these again here. If you like, you can review the 2D example now and try out the same features on this 3D model.

In this section, we shall concentrate on the features that we believe you will find most valuable when working with a 3D model. Many of these only come truly into their own for models which are much larger than our example here, but we would advise you to get into good habits right from the start.

We also try to give you a few hints and tips on the best or most appropriate way to use the features we cover here.

Summarizing support reactions

For simple static analysis, probably the first check that you would want to make would be the fundamental one, that the Sum of applied loads = Sum of Reactions.

1. Click on **the Reactions Tool** from the **Result Tools** toolbox.
2. **S-FRAME** will display all the individual base reactions for the load case or combination you choose from the Views and Grids toolbar. The Legend displays the summary of all the loads in the appropriate directions.

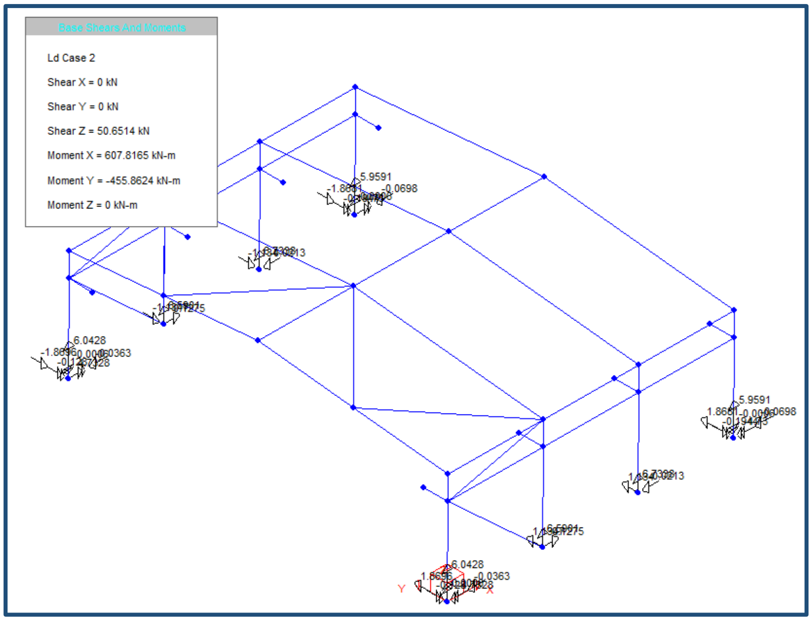


Figure 52.

Although the legend gives the overall details clearly, the screen display which shows all the members and textural values for the reaction in each direction is mainly illegible.

We can reduce the clutter somewhat by just selecting the support joints (using the **Select > Special** dialog), and we can choose to see the results in a specific direction by choosing that direction from the **Data Bar**.

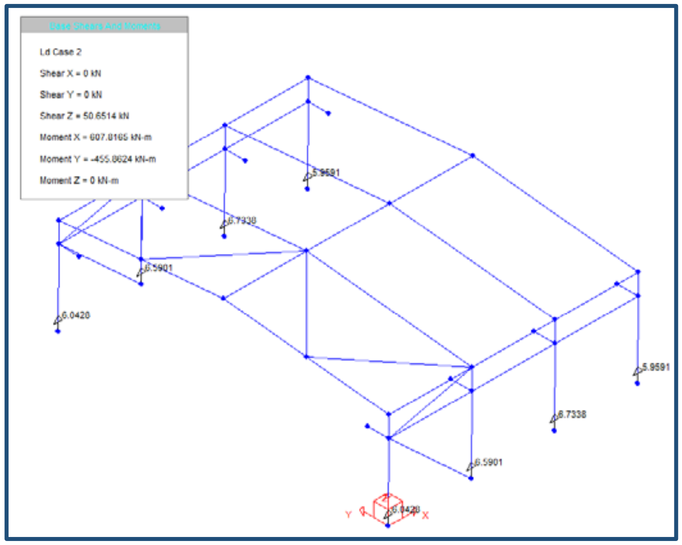


Figure 53.

You can also change the format of the numbers by clicking **Settings > Diagrams...** or clicking the **Diagram scale factors and styles** icon from the **Edit** toolbar to see the **Diagrams** dialog, then click **Format**, and reduce the number of decimal places to say 2.

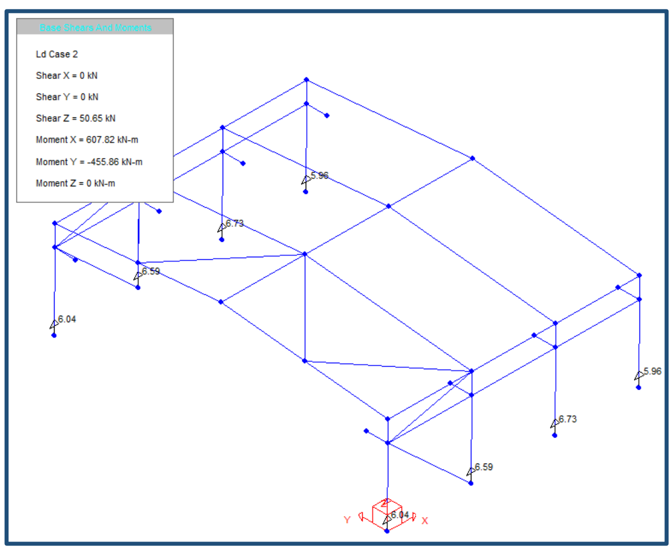


Figure 54.

You can now see the details clearly, and for a more complex structure you could zoom and pan to view particular areas of your structure.

If your support reactions are in the same plane, then you might also consider changing to 2D mode and viewing the details just for that plane.

Adding Numerical Values to Diagrams

You have the option to display various numerical values on diagrams. This is controlled from the **Diagrams** dialog.

1. Click **Settings > Diagrams...** or click the Diagram scale factors and styles icon from the **Edit** toolbar.

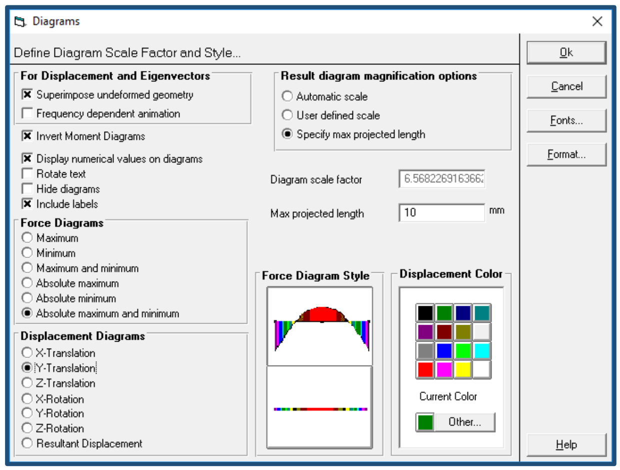


Figure 55.

2. Check the Display Numerical Values on Diagrams and the dialog is reconfigured with the additional options shown above. You can use these additional items to control the text that you would like to see on your force and deflection diagrams separately.

We would always recommend that you choose the option to **Specify maximum projected length**. A value between 5 and 10 mm (0.25 to 0.5 in) usually yields good results.

If you use these options carelessly you can easily obtain meaningless displays such as the one shown below.

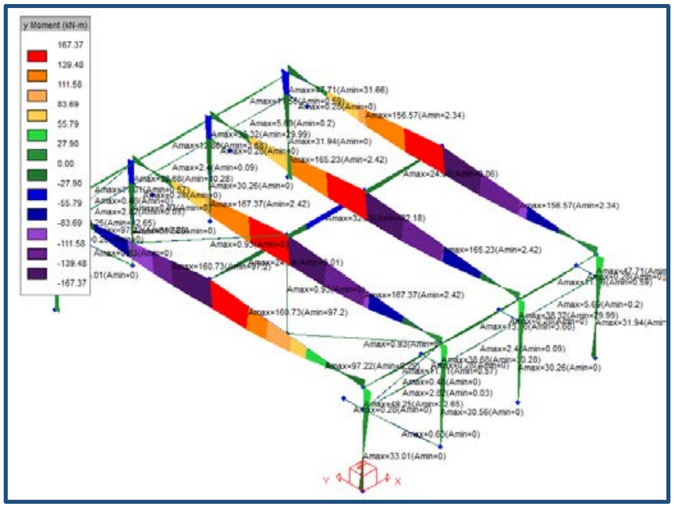


Figure 56.

3. You will probably find that adding text to diagrams is much more useful when you are working in a 2D view, however, with care, you can obtain useful information even in a 3D view.

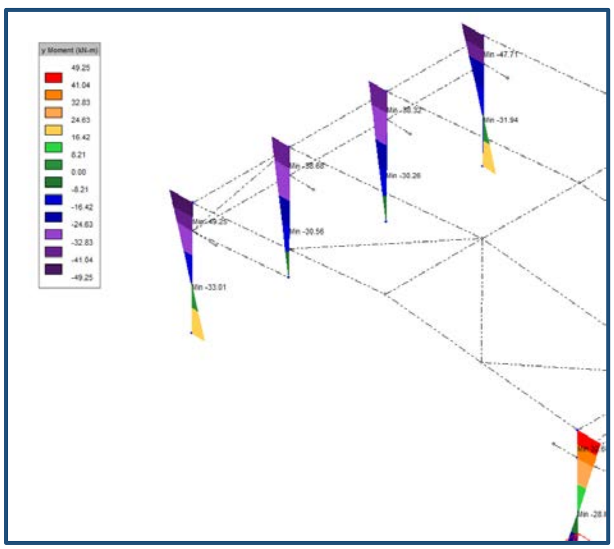


Figure 57.

4. If you define an appropriate user coordinate system, switch to a 2D view and switch the **Legend** off, then you can get clear, informative views such as shown below.

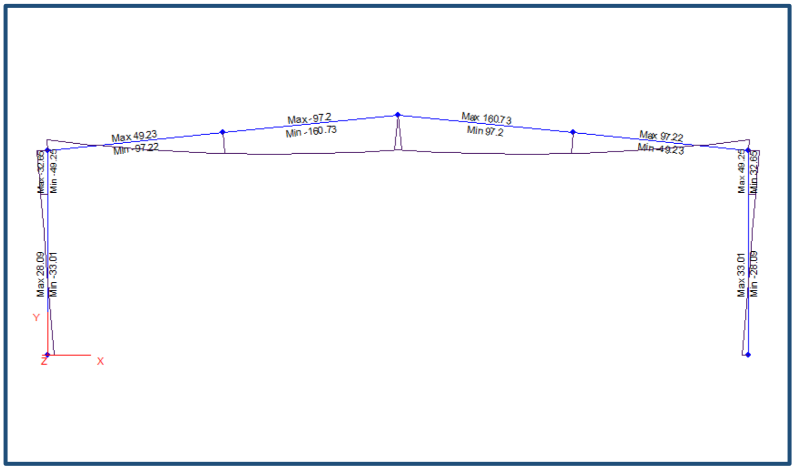


Figure 58.

You might consider creating another standard 2D view with these alternative settings – once again if you have set up a series of user coordinate system you can go into this 2D view and swap between a series of floor and elevation views.

5. Finally, the diagram below shows a 2D view with the envelopes option activated and all load combinations included in the envelope.

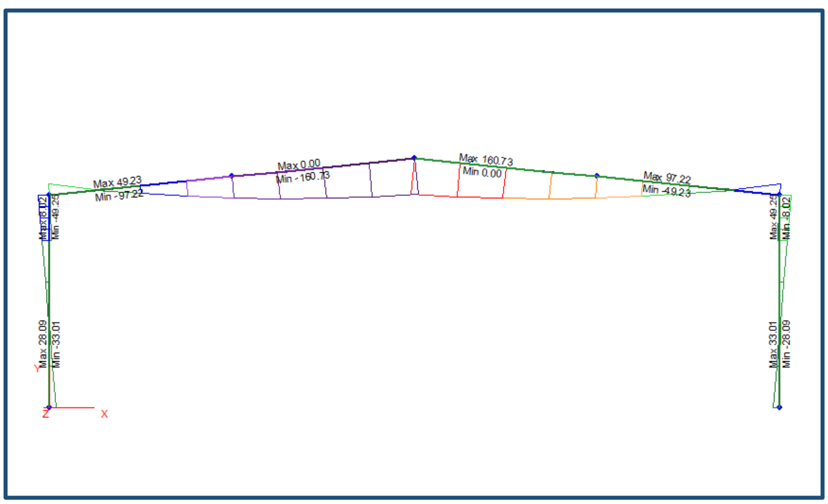


Figure 59.

This ends this **S-FRAME** (3D) tutorial. We hope that you have found it useful.