Altair S-FRAME 2021.1

Quick-Start Guide to S-FRAME

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.

Contents

This symbol indicates that video tutorial on the related topic exists in the S-FRAME help system.

Intellectual Property Rights Notice

Thank you

Thank you for your interest in S-FRAME Analysis. Rest assured; you have made the right decision by selecting not only a proven software solution but also 35+ years of structural analysis and design support expertise at your disposal. Our application engineers look forward to building a supportive and productive relationship with you.

We understand that like many powerful software products, S-FRAME Analysis can appear daunting at first to the new user. With so many features and options available. To help ease your transition to working within S-FRAME Analysis and ensure that you get the maximum benefit in minimum time we have developed this *Quick-Start* guide for new users. By following the steps in this guide, you should be able to create, analyse and review results for a simple model in approximately 1 hour.

Once you have completed the *Quick-Start* guide, bigger (more realistic) models may be to some extent just a matter of repetition. However, you probably want S-FRAME to do the repetitive work for you and there are numerous S-FRAME features designed to speed up your modeling and analysis workflow. To help you progress from an S-FRAME *"novice"* to an S-FRAME *"expert,"* you can find many more worked examples within the S-FRAME *Help* system and the detailed manuals supplied in electronic (.PDF) format.

Finally, we always encourage and welcome user comments and ideas about S-FRAME. Your input is important because client feedback helps to ensure that S-FRAME Analysis' ongoing development continues to meet your requirements. Many of S-FRAME's current features come from our direct interaction with customers like you. We are committed to maintaining and updating S-FRAME, and your contributions are vital to this.

How to Use the Quick-Start Guide

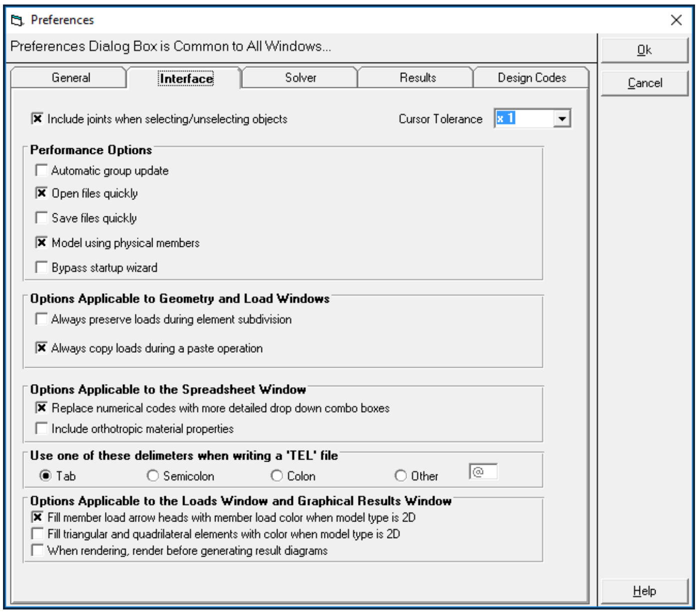
The S-FRAME Analysis *Quick-Start* guide is comprised of two parts.

Part I (Chapters 1 through 8) includes an overview of S-FRAME Analysis' basic tasks and features. Each chapter is further divided into smaller units, highlighting a specific area of study, method or concept. The reader may optionally practice these features while reading through, or simply read through Part1 and use the Example at the end for practice.

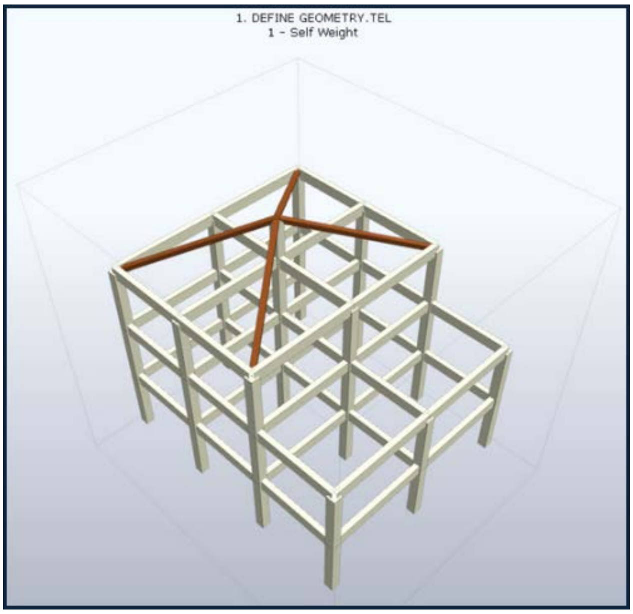
Part II (Chapter 9 and10) contain a complete step-by-step practice example designed to show the reader how to create, analyse and design a simple 3 story structure. By carefully following the provided steps you can ensure to take away as much as possible from this guide. The model file is included as part of the guide.

Chapter 10 provided information on how to get technical support and additional training.

Within the Guide 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:



Good to Know: Scattered throughout this guide in green boxes are short "Good to Know" facts.

Need to Know: Scattered throughout this guide in red boxes are short "Need to Know" facts.

Further Information: Scattered throughout this guide in blue boxes are additional resources for specific topics.

1.0 First Steps

Once S-FRAME Analysis is installed and licensed, opening the program displays the Product Home Page window. This window opens, by default, once a day for every S-FRAME execution. It includes the version history and any upcoming technical webinars and training classes.

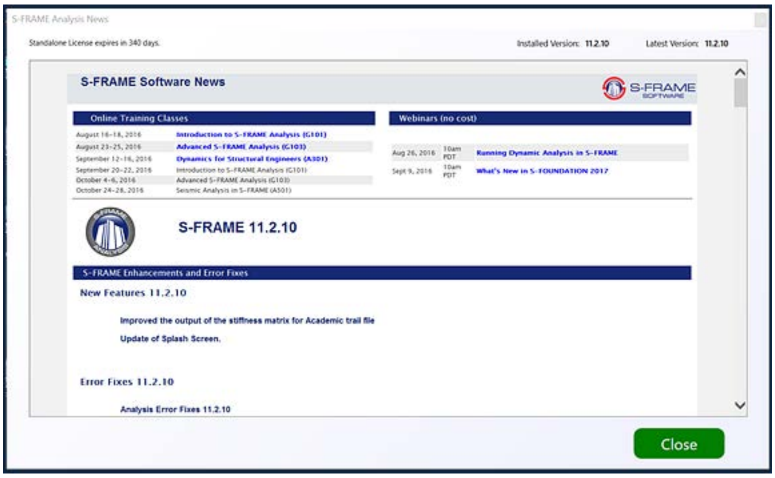


Figure 1. Product Home Page

Clicking on the green button, closes the product home page window and launches S-FRAME Analysis, but you can always re-open this home-page window it by selecting the S-FRAME drop-down menu item:

Help -> Show Home Page

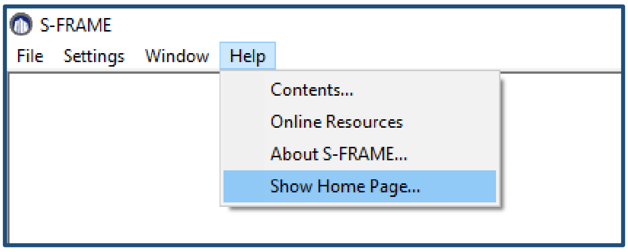


Figure 2. Displaying the Product Home Page

If you have not yet done so, click on the green close button to launch S-FRAME and open the **Open Structure** dialog window.

The Open Structure dialog is used to quickly access previously opened model files or to define a new model.

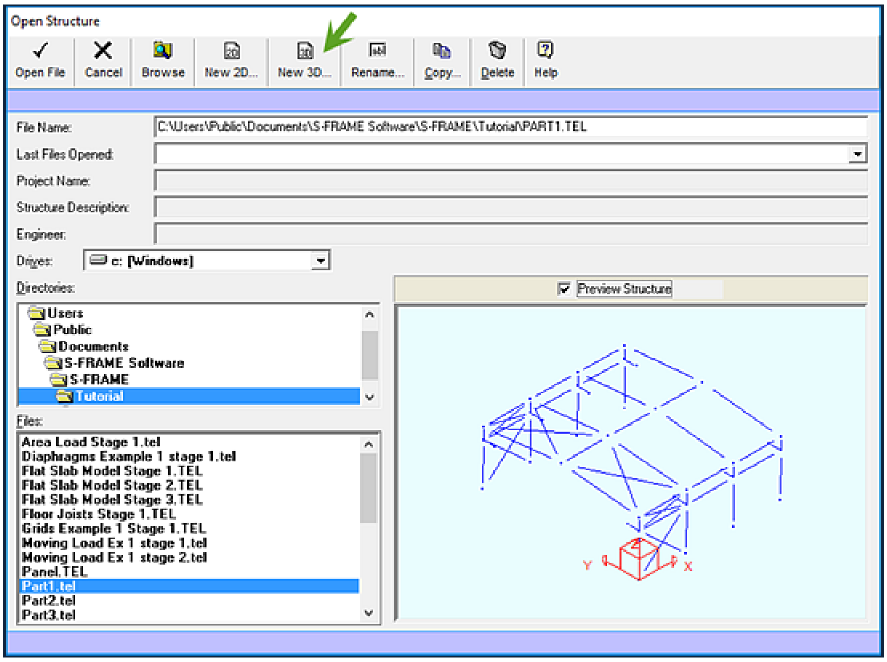


Figure 3. Open Structure Dialog

To create a new 3D structure, click the **New 3D button**, to open the **New Structure** dialog window.

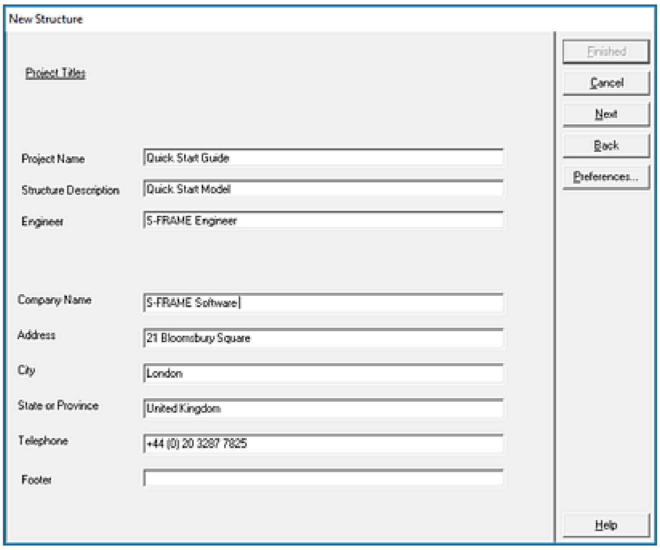


Figure 4. Project Titles

Here, the user would enter any of the Project title fields. This information appears on project reports generated from the structure. These fields can be edited later. Click **Next**.

In the next steps, the Input Units, the Results units, the Modeling Options and the Design Codes are defined.

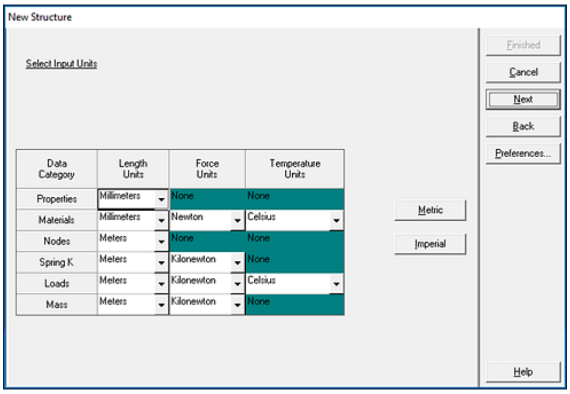


Figure 5. Input Units

Click **Metric** for the input units followed by **Next** to proceed to the Results Units. Note that it is possible to create a model in one unit system, but have the results displayed in a different unit system

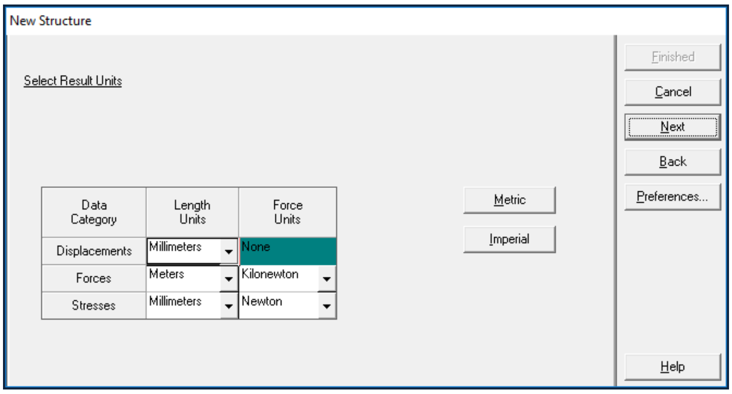


Figure 6. Results Units

Click **Metric** for the Results Units followed by **Next** to proceed to the Modeling Options.

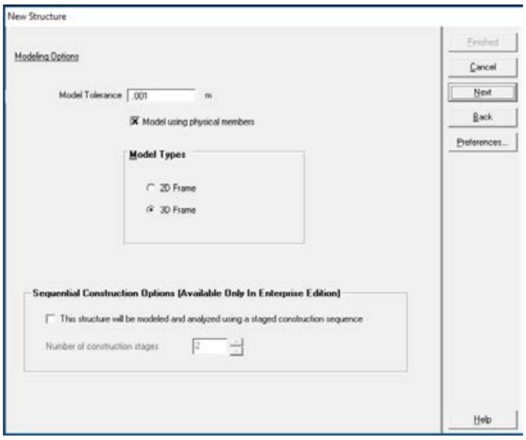


Figure 7. Modeling Options

Accept the defaults at this dialog and click **Next** to define the Design Codes.

i	You can find useful information for the Model Tolerance settings, in S-FRAME's Help System (Help → Contents → 'Search Tab') under the 'Integrity Checks' topic.

Specify the desired design codes for Concrete and Steel Design and click **Next**.

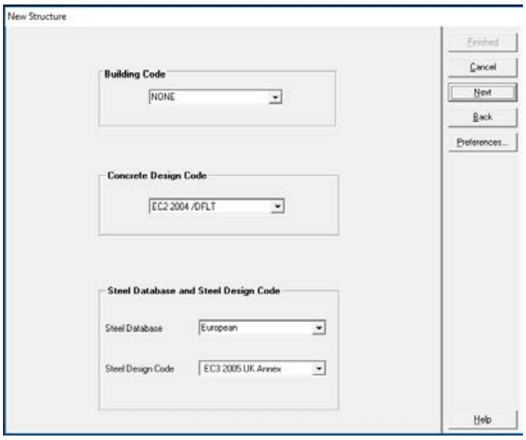


Figure 8. Design Codes

Once we have specified the modeling options, the Model Generation Options dialog open. Here you can invoke various wizards to generate different model types. To create our simple model from first principles, click **Start from Scratch - Open a blank model** and then click **Finished**.

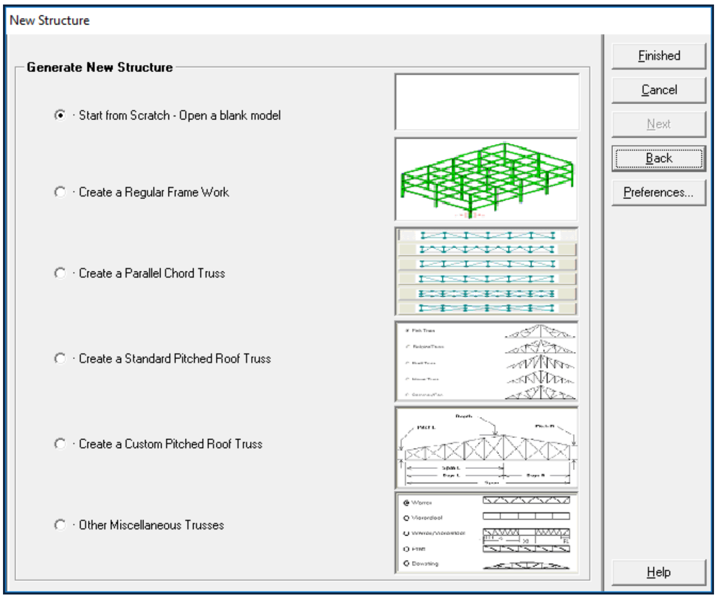


Figure 9. Model Generation Dialog Window

At this stage, the S-FRAME launch process has completed, and S-FRAME's Geometry window opens next for the user to begin defining the structure's geometry.

S-FRAME remembers your Input Units, Result Units, and Preference settings. In future, you will only need to access these dialogs to make changes or to ensure that no one else has made changes since you last used S-FRAME.

2.0 The S-FRAME Graphical User Interface

The Graphical User Interface's main components are described below. There are six views in the S-FRAME interface. The Geometry Window or *VIEW* is opened, by default.

- **GEOMETRY** – Graphical input of model objects and properties; joints, members, sections…etc.
- **LOADS** – Graphical input of loads; Joint Loads, Member Loads…etc.
- **GRAPHICAL RESULTS** – Post-analysis Graphical results; Deflection, Load Diagrams…etc.
- **SPREADSHEET** – Numerical definition of model objects, properties, and loads
- **NUMERICAL RESULTS** – Post-analysis Numerical results; Deflections, Member Forces…etc.
- **CAD DETAILS** – Graphical input of **non-structural** information – dimensions and offsets.

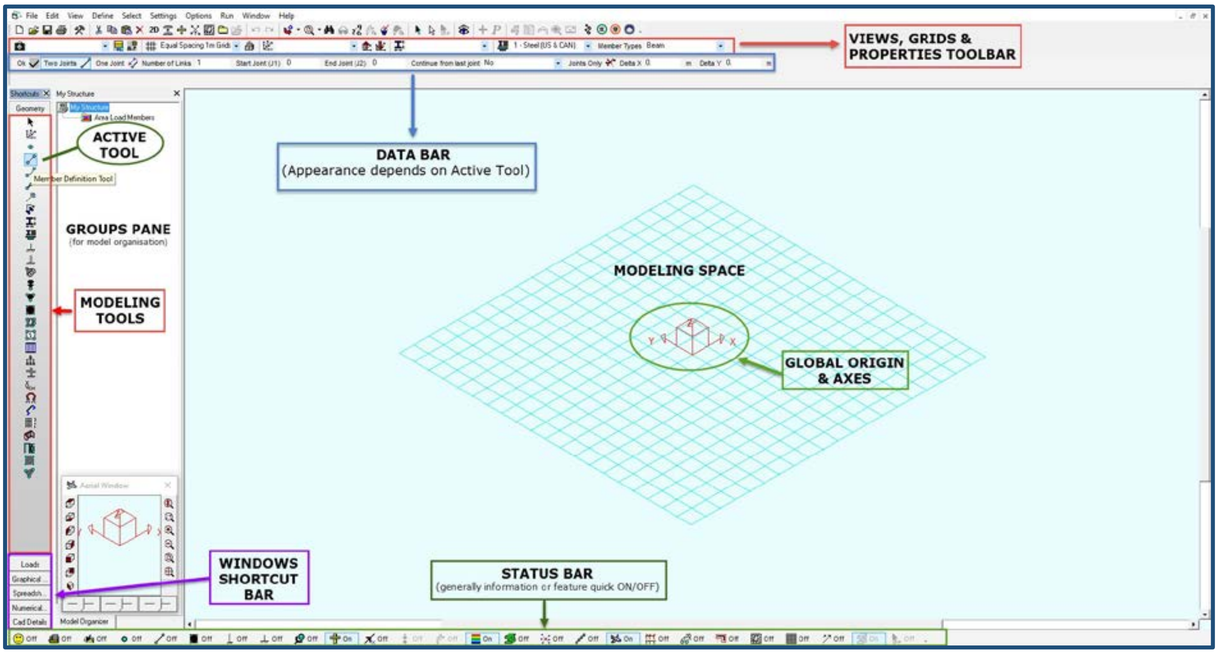


Figure 10. The Graphical User Interface

a. **The tools** on the left-hand side are used to create, edit and view the geometry, the loads, and the results.
b. Depending on the currently active tool, we have additional options in the **Data Bar** shown above.
c. Views, grids, coordinate systems, section and material properties can be edited in the **Properties Toolbar**.
d. The switches in the **Status Bar** below allow us to hide/unhide information of our model.

Navigating in the graphical Model Space

S-FRAME has numerous graphical navigation options, common to many modeling/drawing programs.

Mouse Navigation

The mouse can navigate quickly as follows:

- *ROTATE model* - hold down right mouse button and drag (+Shift to rotate about Y-axis)
- *ZOOM model* – scroll the mouse wheel
- *PAN model* – hold down mouse wheel and drag

The model used to demonstrate S-FRAME's navigation and selection tools is located in the default directory

C:/Users/Public/Documents/S-FRAME Software/S-FRAME/Tutorials/PART 1.TEL

If you want, you can open this model file to practice the navigation and selection options described below. Toggling the "Shrink Elements" tool allows for easier selection.

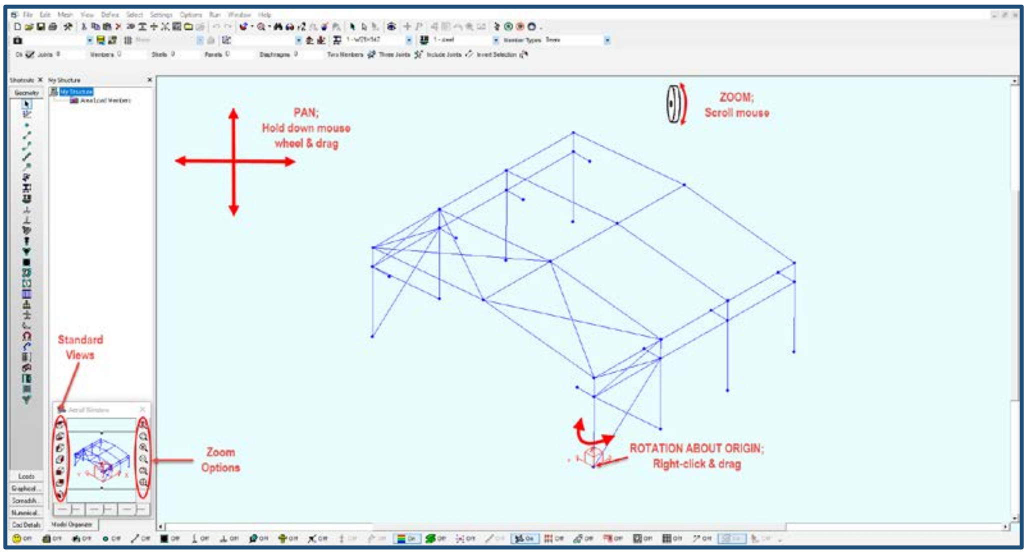


Figure 11. Mouse navigation

The Aerial Window provides quick access to many standard views and zoom/pan options and can be repositioned anywhere within the GUI.

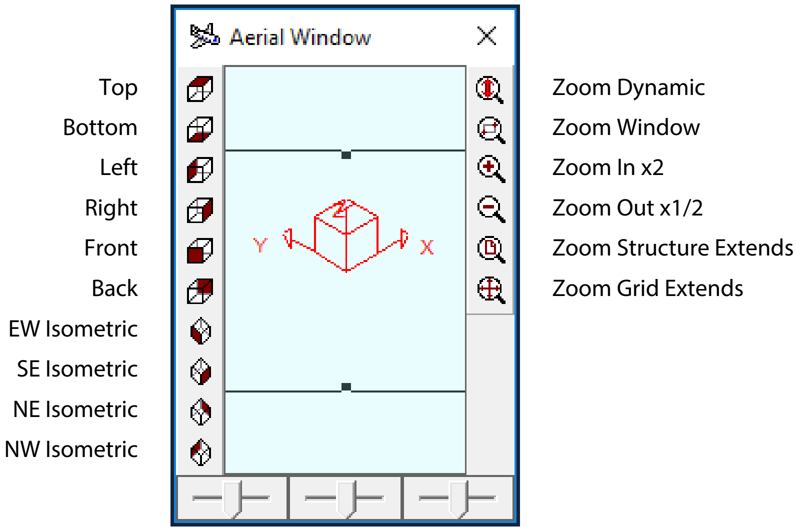


Figure 12. The Aerial Window

The Selection Tool

The Selection tool provides multiple ways to select or unselect objects in the viewport. This tool is common to all graphical windows. Note also the existence of the *Select Special* dialog, which you can access from any graphical window in S-FRAME.

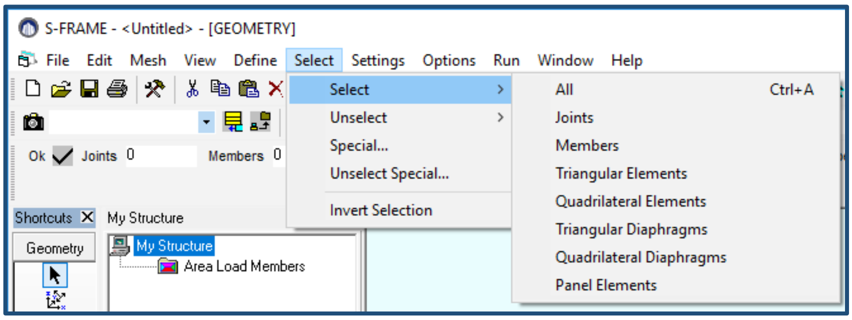


Figure 13. The selection Tool

- All ***OBJECTS*** such as Joints or Members are selected (active) by default and are indicated by solid lines and filled circles.
- Unselected (inactive) objects are indicated by dashed lines and empty circles.
- Unselected (inactive) objects can optionally be hidden to simplify the view.

The **SELECTION** Tool is Active by default. The selection tool is used to select/unselect objects.

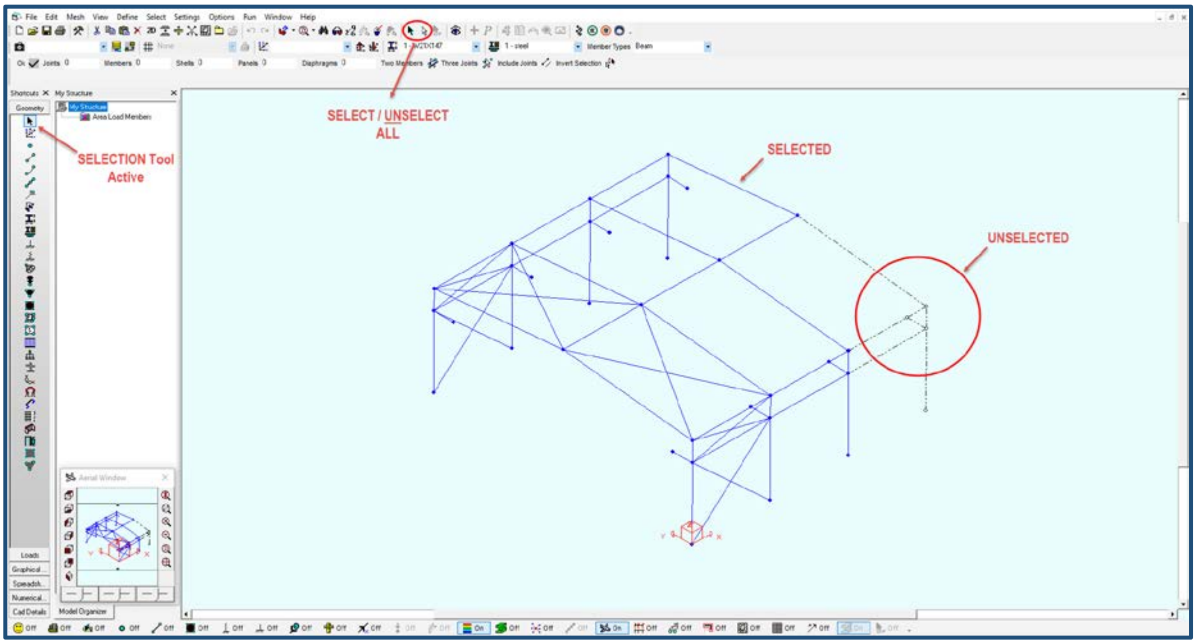


Figure 14. Selected – Unselected Objects

Actions – such as ***Delete*** and ***Move*** – can only be performed on **Selected** objects.

Standard Mouse Select Operations

S-FRAME provides numerous graphical ways to pick and unpick objects such as joints, members, dimensions, 2D elements, panels and diaphragms. The meaning of "pick" and "unpick" depends on the active tool.

There are five ways to pick (**or unpick**) as shown in Figures 13, 14, 15 and 16 below:

1. **Single Click**: While the Selection tool is active, click a joint, member or 2D object centroid to toggle the object's selection status. You can also toggle the selection status of an object by right-clicking it and choosing the appropriate command from the Viewport menu.
2. **All objects properly inside a box**: Drag the mouse from **right to left**.
3. **All objects inside a box, including objects that intersect the box**: Drag the mouse from **left to right.** **T**he application selects those objects that lie within or intersect the rectangle.
4. **All objects that intersect a line**. Hold the Shift key down while dragging the mouse across objectrs. The application selects those objects that lie intersect the line drawn.
5. **All objects properly within a polygon:** Click the mouse in empty space to define each vertex of a polygon—up to a maximum of 30. When you finish defining all of the vertices, click the first vertex again. The application selects only those objects that lie entirely within the polygon. Be sure the “Pick all Objects inside the Polygon” tool is selected.

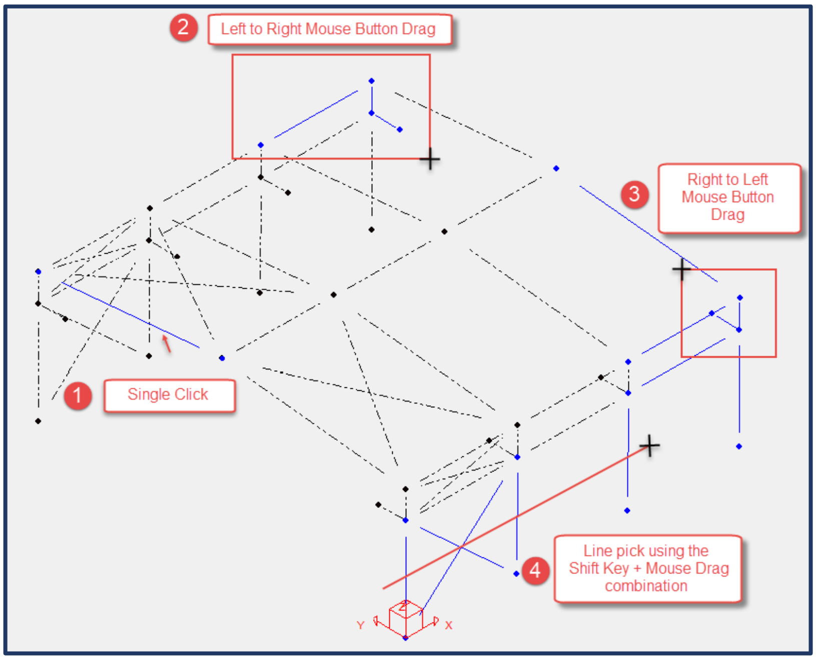


Figure 15. Select Objects

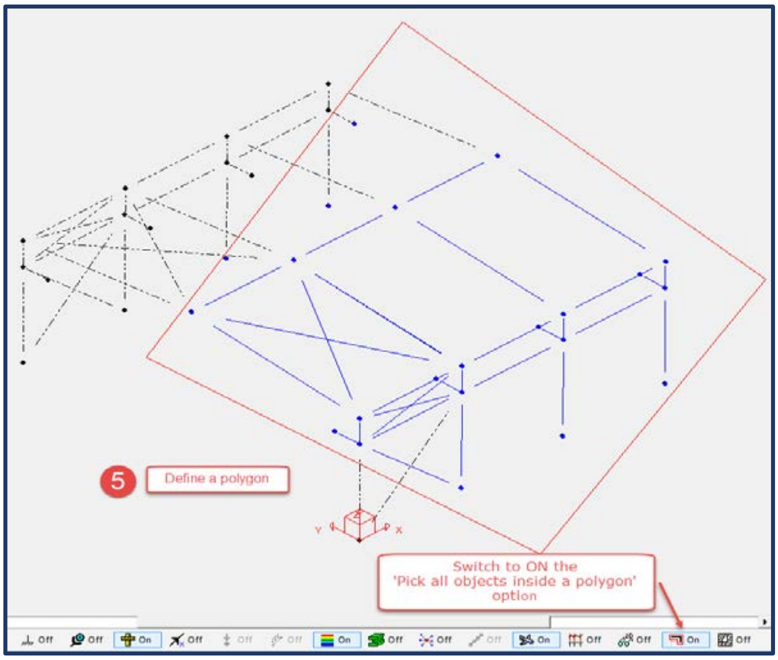


Figure 16. Select Objects inside a polygon

> **Depending on the tool selected, these operations can be used to ASSIGN attributes and loads.**

Holding down the **Ctrl key** instructs S-FRAME to **unpick** rather than pick.

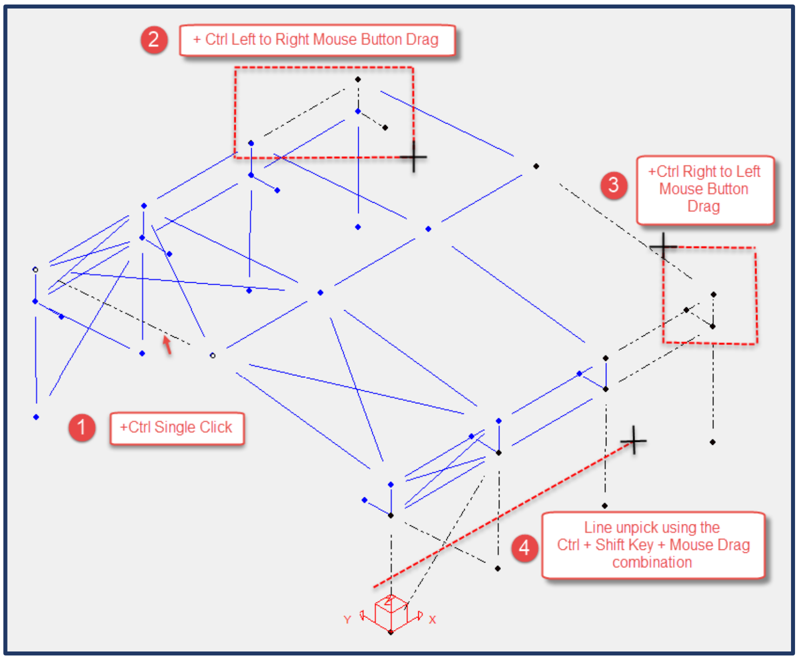


Figure 17. Unselect objects.

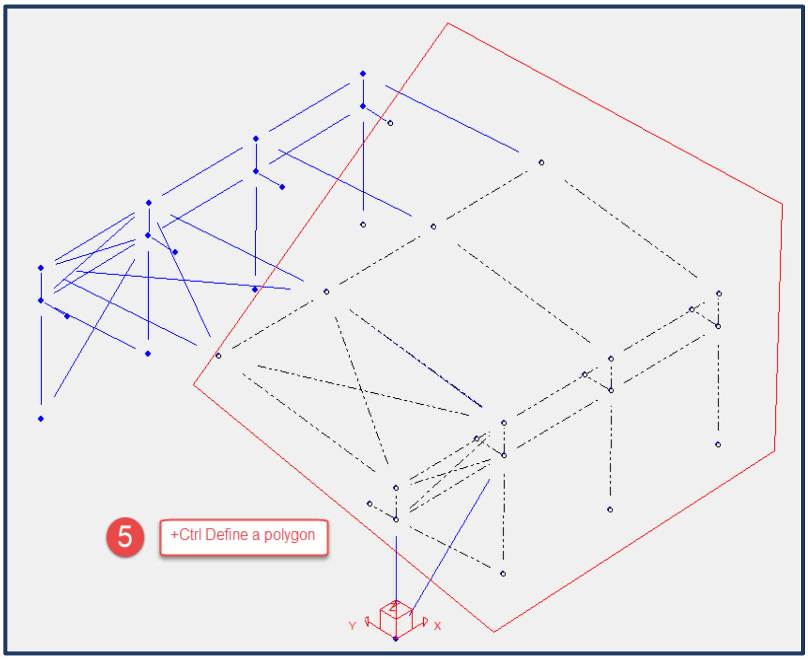


Figure 18. Unselect objects inside a polygon

The "Pick all Objects inside a Polygon" tool, located on the bottom toolbar, must be toggled on for the polygon selection/unselection method to work.

Depending on the tool selected, these operations can be used to ERASE attributes and loads.

A tutorial video demonstrating these features is available in our Online Resources
Help→Online Resources→S-FRAME Analysis Videos
The Model used in this demonstration is located in the default directory:
C:/Users/Public/Documents/S-FRAME Software/S-FRAME/Tutorials/PART 1.TEL.

3.0 Geometry Window - Define Geometry

There are several different ways to define a structure's geometry in S-FRAME Analysis

Regular Framework Command

You may have noticed that we can generate a Regular Framework (*Figure 9. Model Generation Options*). This framework generation can be performed when we launch the program or afterwards by going to Edit→Regular Framework

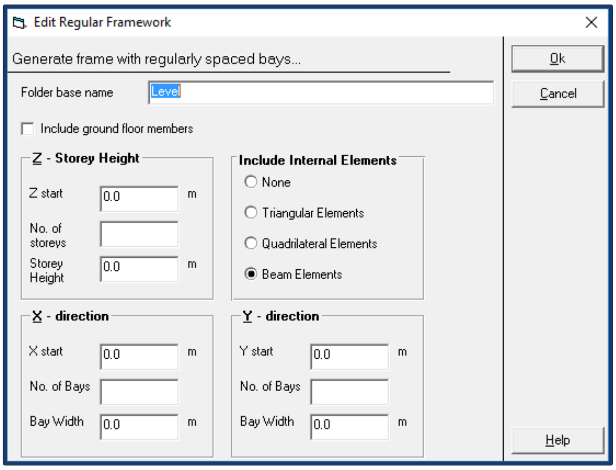


Figure 19. Edit Regular Framework dialog

You can find a tutorial video in the Online Resources
Help→Online Resources→S-FRAME Analysis Documents

Define geometry using the Member Definition Tool.

If you want to practice using this tool and have previously opened the Part1.tel file, close it now by exiting and re-opening S-FRAME Analysis, selecting the create New3D model option.

The Member Definition Tool is typically the most common way to define member geometry, but to define a member this way, we first need to define a **Grid** set or a set of **Joints**.

Grids

Grids are defined under the Define menu (Define → Grids) or by clicking the **Edit Grids** shortcut on the Views, Grids and UCS toolbar near the top of the interface.

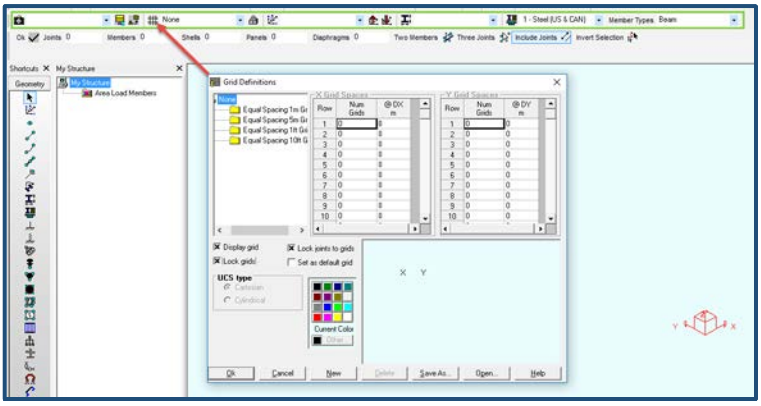


Figure 20. Open the Grid Definition dialog

S-FRAME comes with a set of default grids. If the default grids are not showing up in your dialogue, you can open the .DGD file by clicking the 'Open' button, and the '.dgd' file should be in the folder that appears. You can also create your grid by right-clicking on the **None** to select **New Grid Set**.

Once a Grid set is defined, members can be defined with the **Member definition tool** using the **Two Joint** or the **One Joint** method, and clicking on intersecting grid points. To practice in S-FRAME Analysis, select the "Equal Spacing 1m Grid", making sure the "Display Grid" and "Lock Grid" boxes are checked, then "OK".

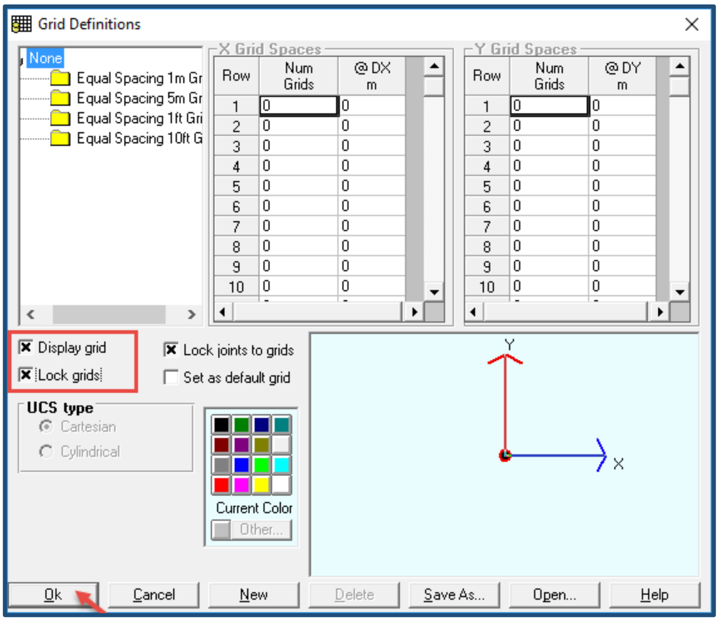


Figure 21. The Grid Definition dialog

Zoom out to see the grid, or click on the **'Zoom to grid extends'** in the Aerial Window.

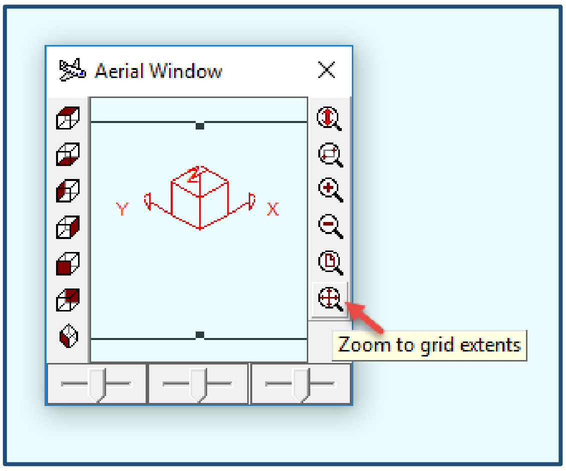


Figure 22. Zoom to grid extends

Two Joint Method for defining members:

1. Activate the **Member Definition Tool**
2. Set the **Two Joints** approach on the data bar above
3. Click **OK** to accept the settings
4. To add a member, click on two points where grid lines are intersecting
5. Locking the grid lines will prevent unintended changes to the geometry

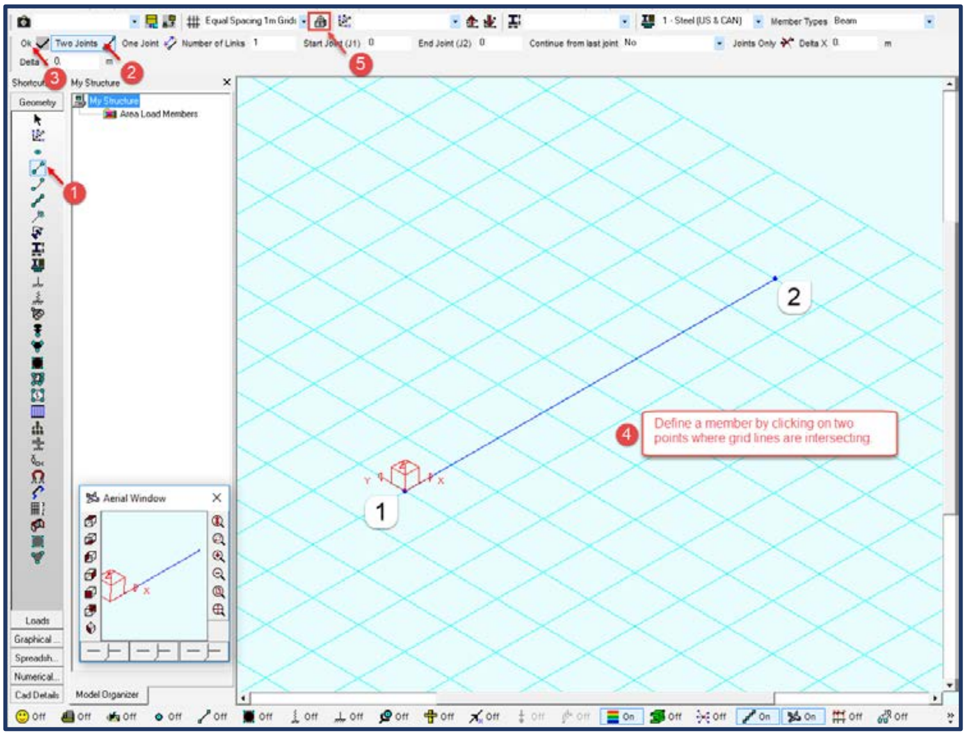


Figure 23. Define a member in +X by defining a grid and using the two joints approach.

One Joint Method for defining members:

Select the **One Joint** option and note that the contents of the data bar are automatically updated. We now have the choice to specify the direction and the extrusion method.

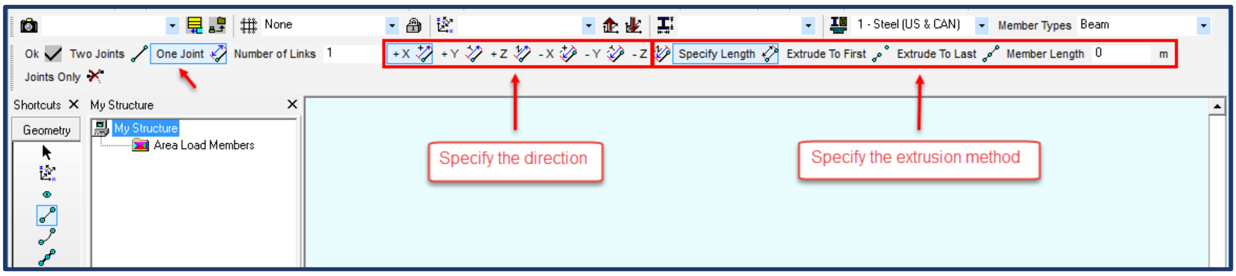


Figure 24. One joint options in the data bar.

1. Set the direction to **positive Z axis**, and
2. Select the option **Specify length** (if not already)
3. Specify the member length, and click on a point of the Grid set to define the member

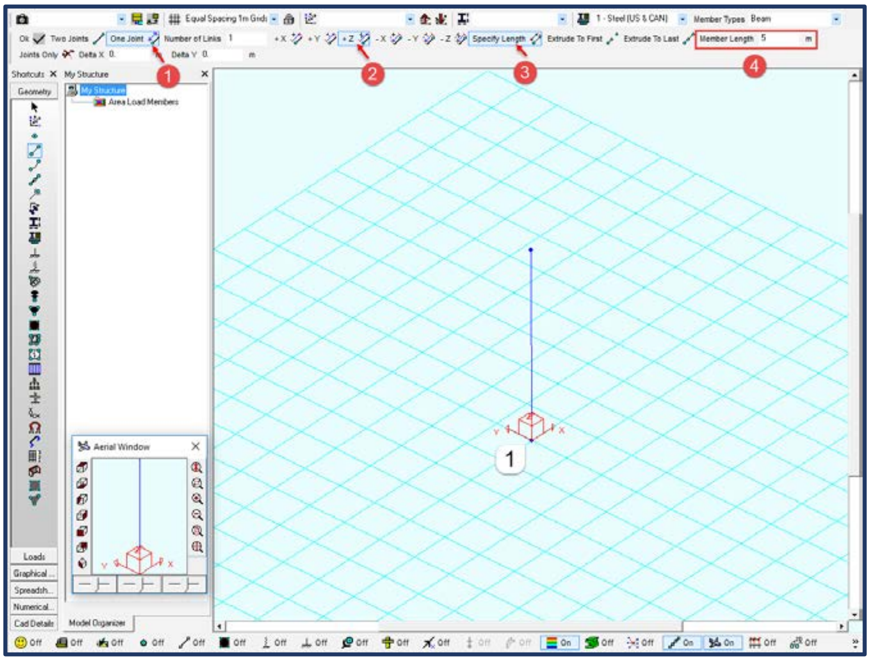


Figure 25. Define a member in +Z by clicking on one point.

In S-FRAME Grids can be used to define and **Edit** the structures' geometry. For more information visitHelp→Online Resources→S-FRAME Analysis Documents

Joints

Joints are defined using the Joints Tool, and entering Joint Coordinates in the data bar above. The process of adding joints, is as follows:

1. Select the **Joints Tool**
2. Enter the **Joint coordinates**
3. Click **OK** to add the joint

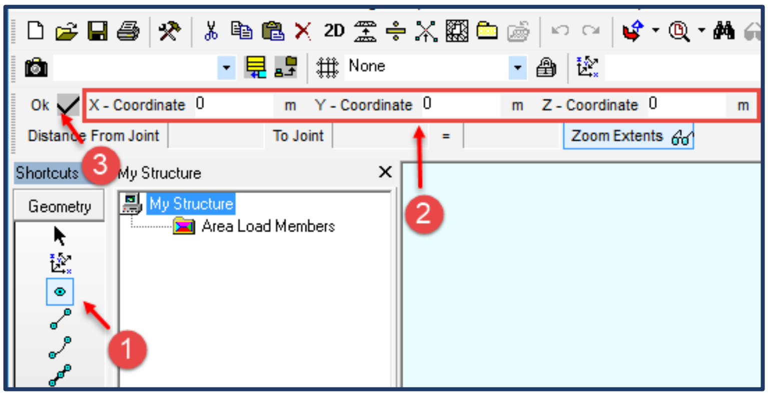


Figure 26. Add joints

Once a joint or a set of joints are defined, members can be added using the one or the two joints approach. Joints can also be added via the Joints Spreadsheet (imported from Excel^{Tm}). This topic is discussed in greater detail in the following chapters.

2D Elements

The Panel Element Tool acts similar to the Member Definition Tool and is used to add Diaphragms, Panels, and 2D elements. The panels can be of any shape and have up to 30 points.

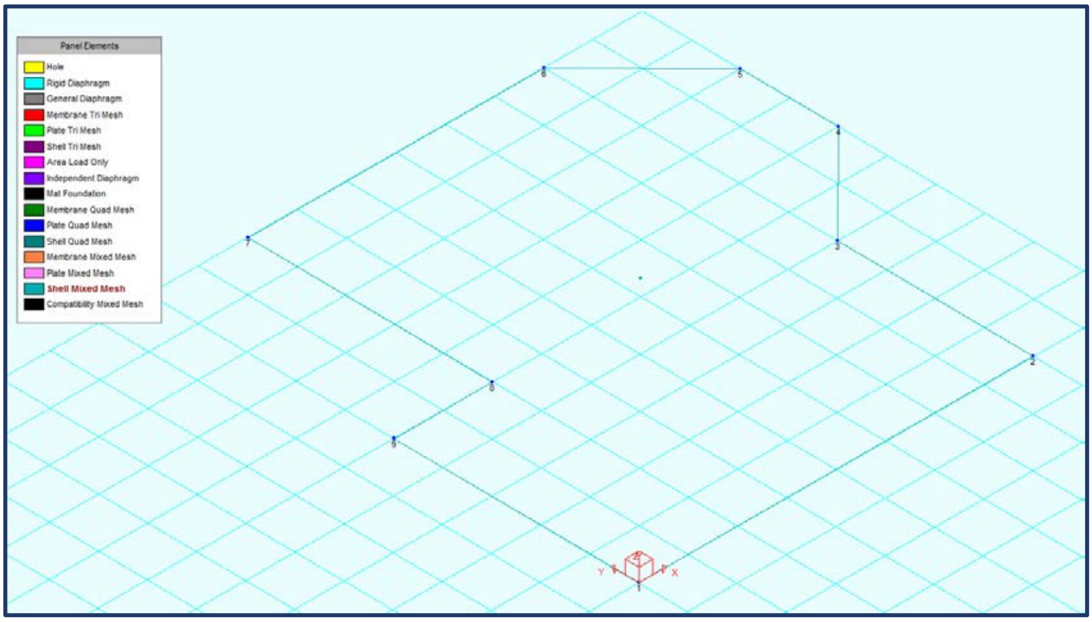


Figure 27. Panel Element

The available panel elements and their properties are listed below:

1. Hole: A hole panel within a larger panel instructs S-FRAME to ignore the hole area for diaphragm masses, mesh conversion and area loads.
2. Rigid Diaphragms: Used to assign constraints (X, Y translation and Z rotation) to a set of joints which fall in the same plane.
3. General Diaphragms: Used to assign constraints to all joints which fall in the same plane
4. Membranes (Triangular or Quadrilateral Mesh): Provide in plane Stiffness only.
5. Plates (Triangular or Quadrilateral Mesh): Provide out of plane stiffness only.
6. Shells (Triangular or Quadrilateral Mesh): Provide both in and out of plane stiffness.

   Membrane, Plate, and Shell elements are automatically meshed using S-FRAME's auto-meshing algorithm.

In addition, 2D elements can be defined using the Quadrilateral or the Triangular element tool.

Import DXF Files

It is possible to define geometry by importing an existing DXF file. Please review the DXF Import video, available in Help→Online Resources→S-FRAME Analysis Documents

Import a Revit™ Model

It is possible to define geometry by importing an existing Revit™ Model file. Please review the Revit™ Link, video available in Help→Online Resources→S-FRAME Analysis Documents

Import a Tekla™ Model

It is possible to define geometry by importing an existing Tekla™ Model file. Please review the Tekla™ Link video, available in Help→Online Resources→S-FRAME Analysis Documents

Defining Section Properties

Section Properties are defined using the **Section Properties Tool** (not the button on the top toolbar). Right-clicking to the tool brings up the **Section Properties Tool** dialog box.

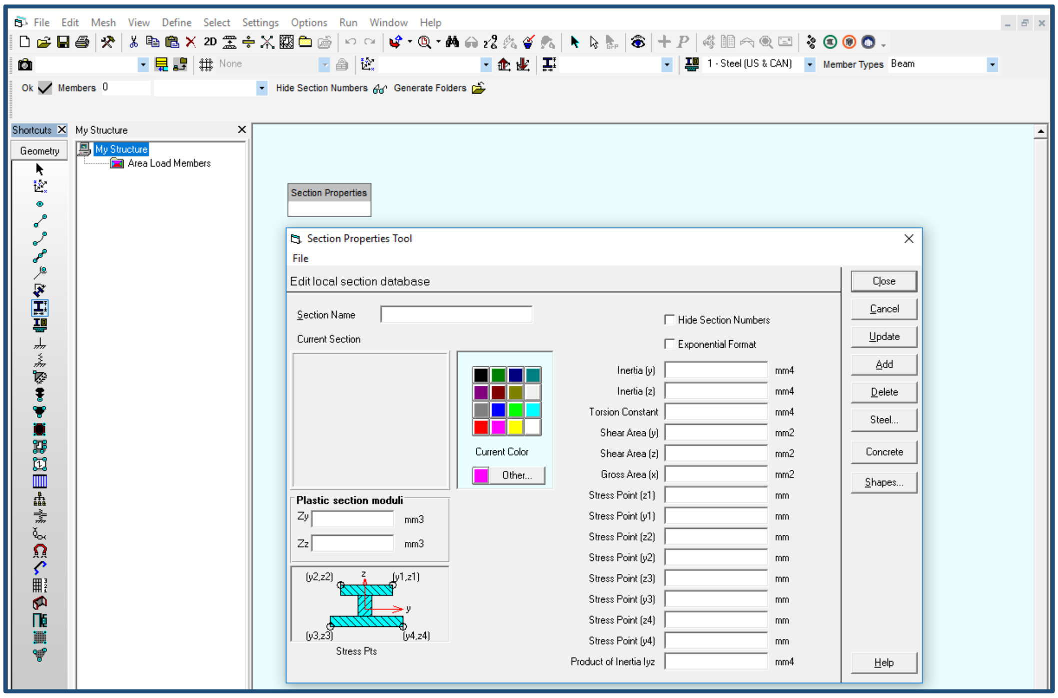


Figure 28. Section Properties Tool Dialog

Selecting the type of the section (Steel, Concrete or Shapes) brings up additional dialogs, where you can **Edit** the sections before they are added to your model.

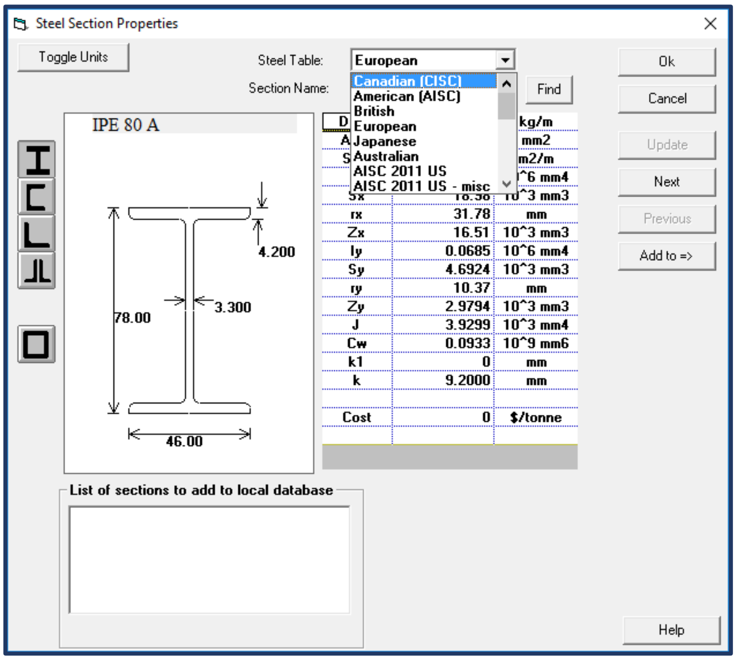


Figure 29. Steel Section Properties

When defining Steel sections, you have access to several international Steel database tables.

Concrete sections can be defined using the S-CONCRETE Visual Editor. Users need an S-CONCRETE license and the Integrated Concrete Design (ICD) feature enabled in order to define concrete sections. If enabled, you can code check and design your entire concrete structure according to your specified international concrete design code.

The example at the end of this guide includes concrete section definition and detailing.

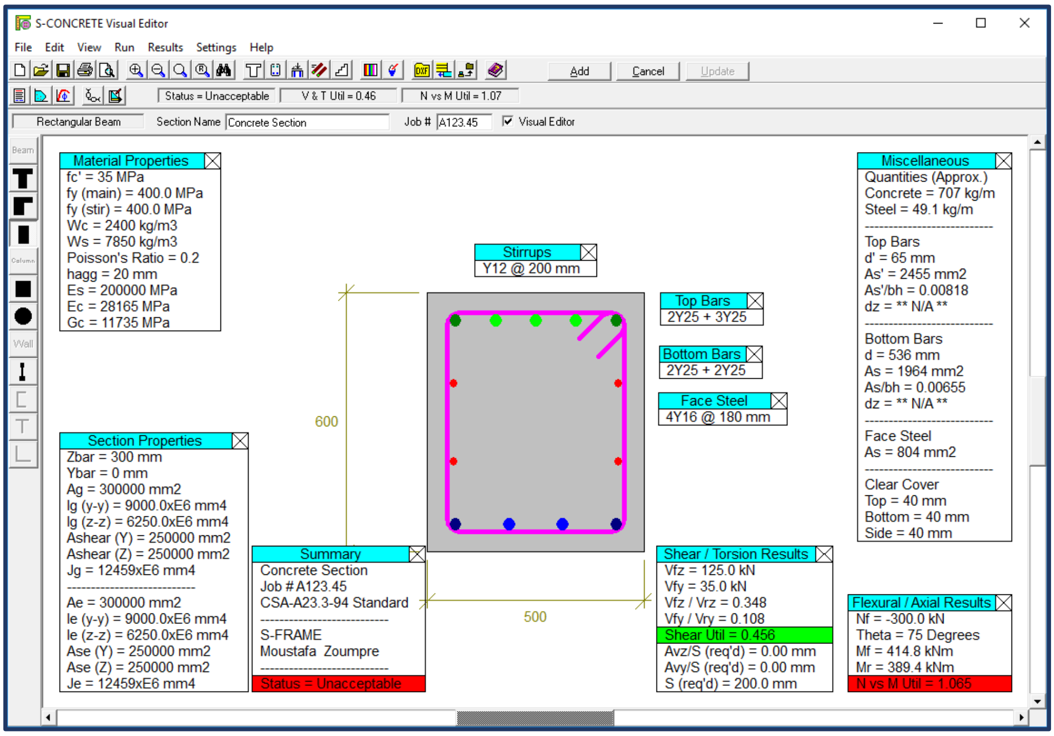


Figure 30. Concrete Section using S-CONCRETE's Visual Editor

Once concrete sections are defined, they are displayed in the Legend. The **Active** section is shown in **Bold Text.** Right-clicking on the on the colour box for each section changes the display colour. Left-click on the colour box sets the ACTIVE section.

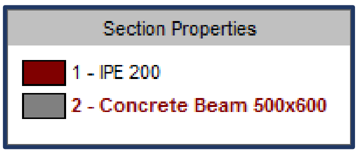


Figure 31. The Legend

Define Material Properties

Material properties are defined using the Material Properties Tool, located on the left-hand side Geometry View's modeling tool list. Left-clicking on the Material Properties Tool opens the Legend with the default material properties. Right-clicking this tool opens the Material Properties Tool dialog, shown below, where you can modify the existing materials and Add new.

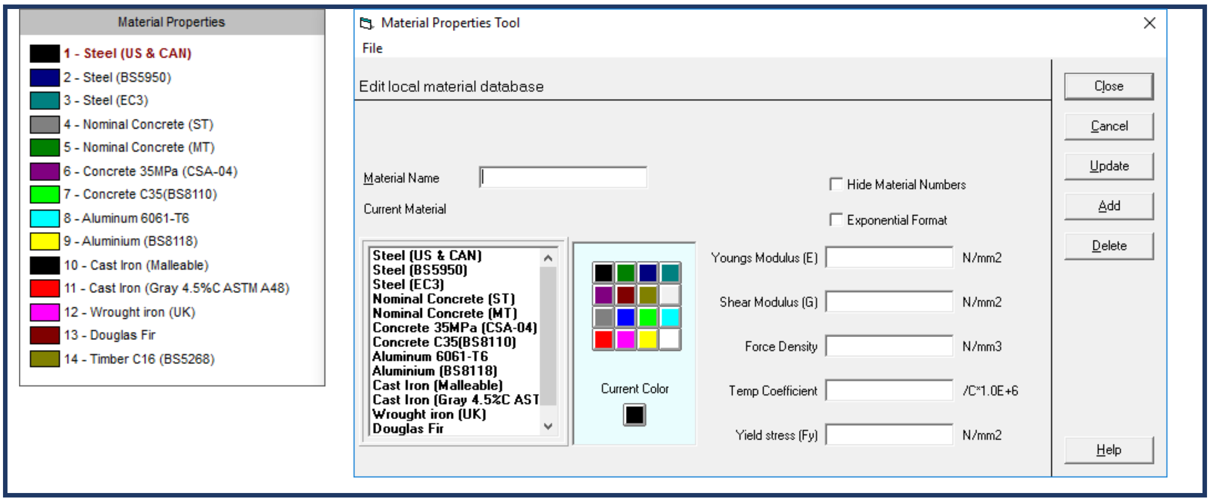
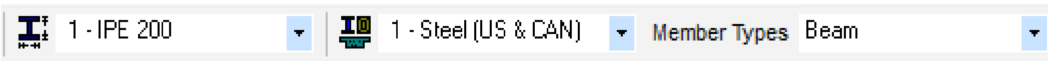


Figure 32. Material Properties

S-FRAME comes with a default material library. If this library is not showing up in your dialogue, you can open the .DMP file by clicking the 'Merge' button, and the '.dmp' file should be in the folder that appears.

Once the definition of the Section and Material Properties is complete, these properties are available to you in the combo boxes on the Views, Grids, and Properties Toolbar.



Supports Tool 

Supports are defined using the **Supports Tool**, located on the Geometry View's modeling tool list.

Left-clicking the **Support Tool** to edit the support conditions and assigning the constraints in the Data bar at the top of the GUI (figure 29).

Right-click on the tool to bring up the associated dialog (figure 30). These constraints can be then applied to the selected support joints, by simply re-selecting them using the standard mouse selection options.

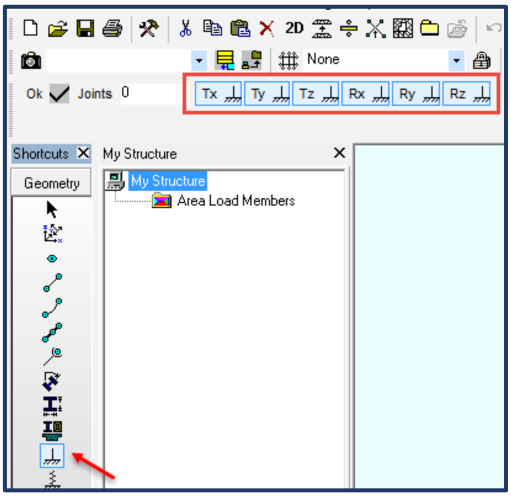


Figure 33. Assign constraints in the data bar

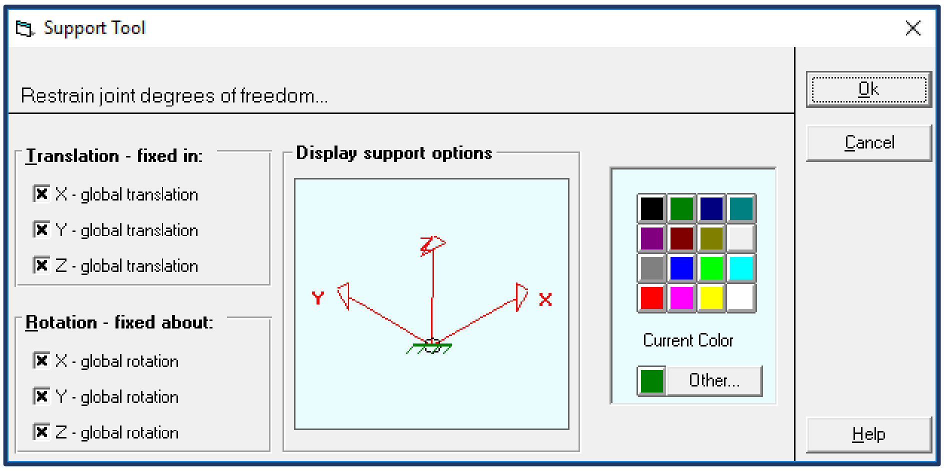


Figure 34. Assign constraints in the Support Tool dialog

We can also assign defaults constraints to all joints of a model, by going to Settings→Default Constraints. Assigning default constraints is useful if you want to simplify a 3D model into a 2D problem (2D constraints).

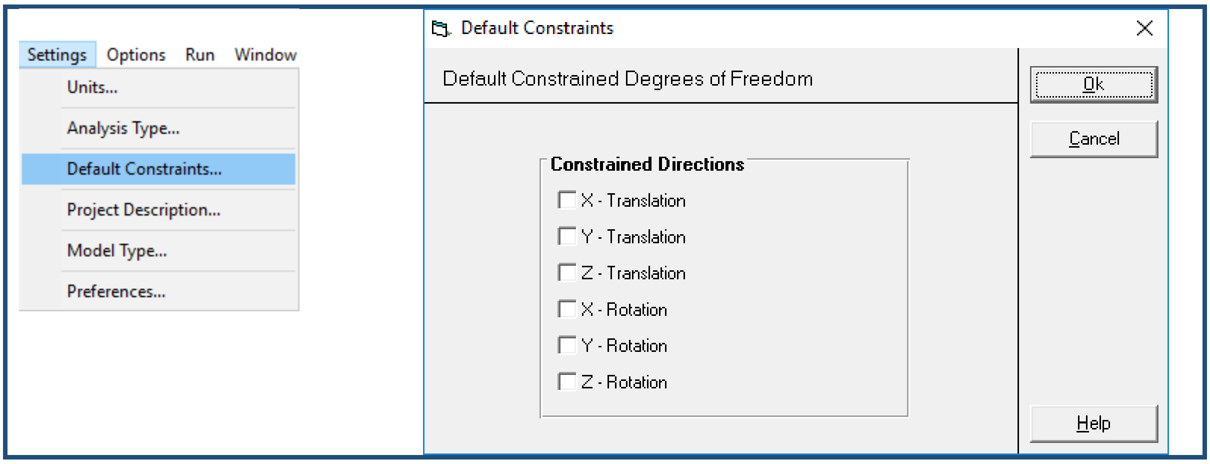


Figure 35. Assign Default Constraints to all Joints of a model

S-FRAME also supports **Linear springs**  and **Non-Linear springs** . These support conditions are edited and defined in a similar way to the above.

4.0 Loads Window - Define Loads

Loads are defined through the Loads Window. Click **Loads** on the Window shortcuts below the Geometry tools to switch to the Loads window. (be sure all open dialogs are closed first) All the modeling tools are now replaced with load-type tools.

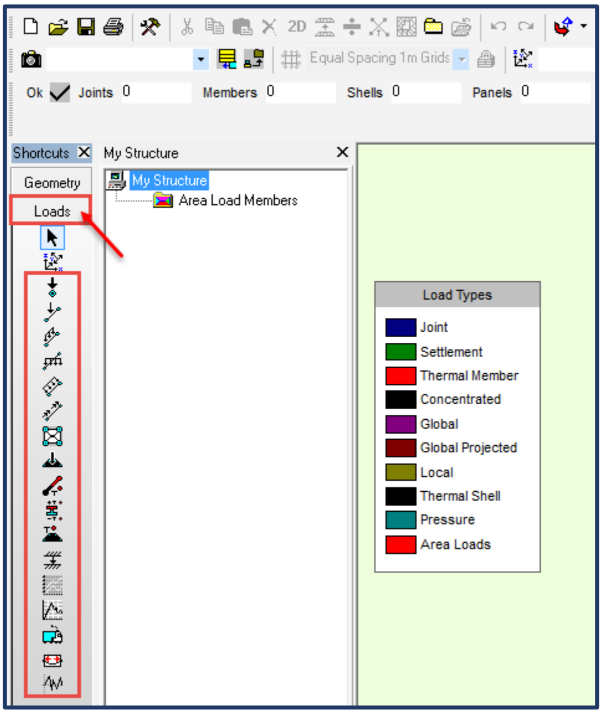


Figure 36. Load Types

In addition to the displayed Load Types in the Legend of the above figure, you can also define Response Spectrum Curves, Time History Loads, Moving Loads, Prestressed Tendon Loads, and Non-Linear Quasi Static Loads.

Similar to the Geometry Tools, Right-clicking on a Load tool symbol opens a dialog box to define the specific loading condition. Left-clicking on a Load tool allows the user to edit the loading condition through the data bar above.

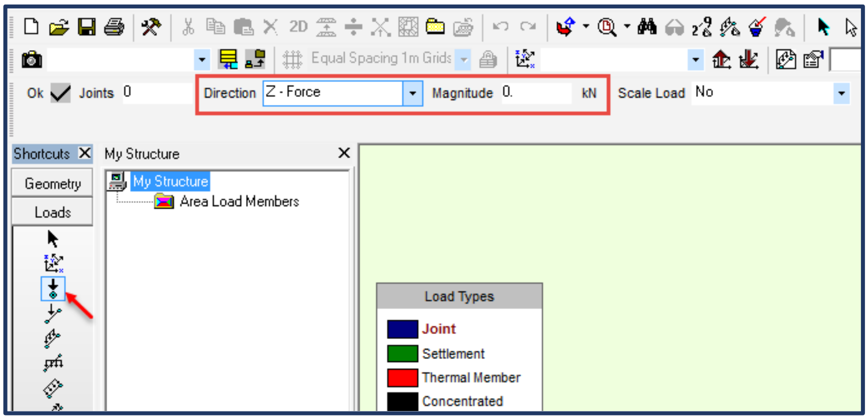


Figure 37. Define Joint Loads through the data bar

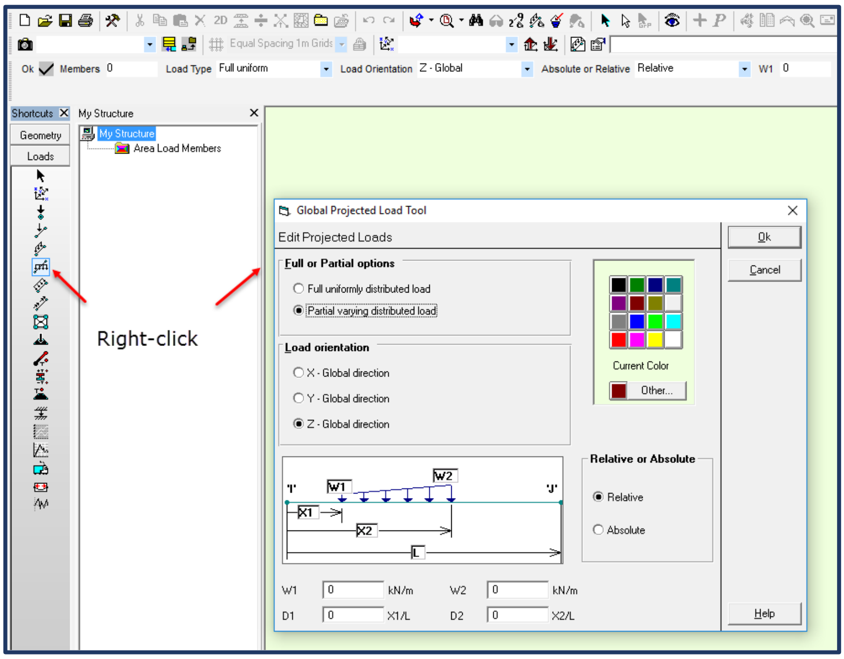


Figure 38. Define Global Projected Load through the Global Projected Load Tool dialog

Define a Load Case

A load case is a convenient way to organize a structure's loading. Each load case has a number and description. Load cases are defined through the Edit menu, or by clicking the New Load case shortcut in the View, Grids, Properties toolbar.

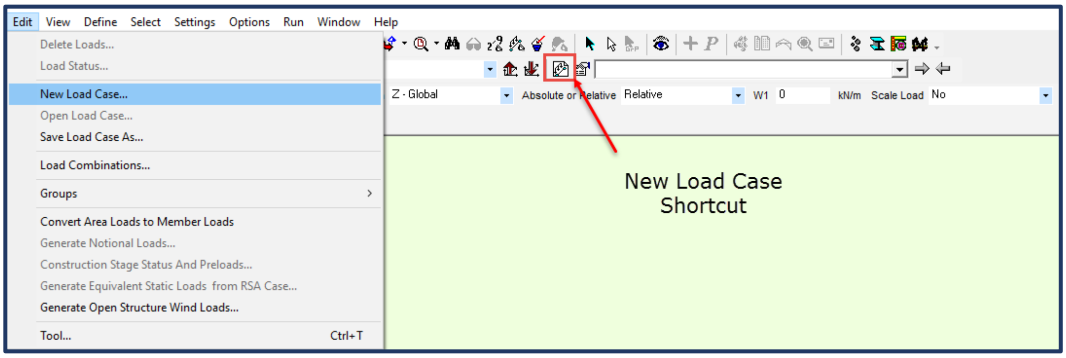


Figure 39. Define New Load Case

In the **New Load Case** dialog box, you define the name, the gravitational factors and the scale factor.

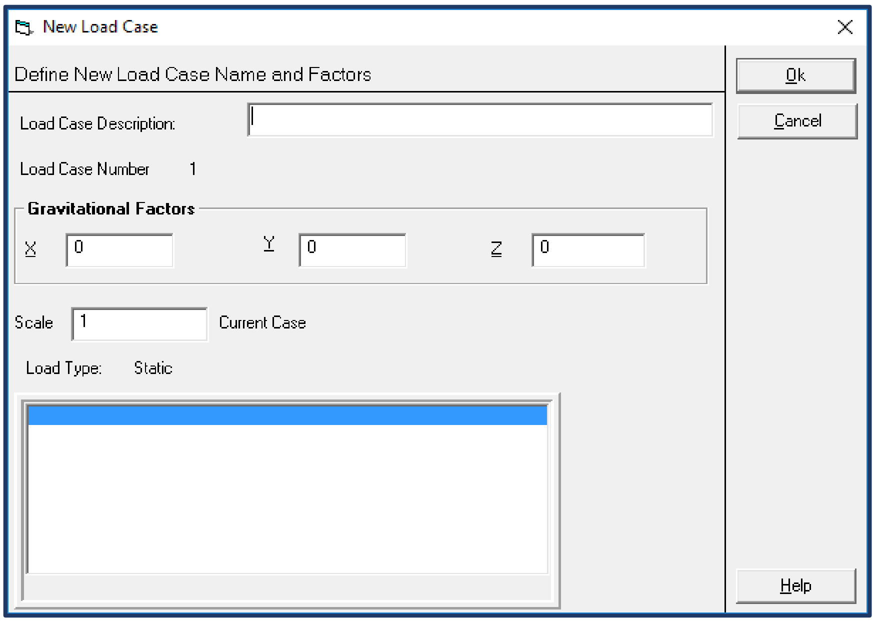


Figure 40. Edit New Load Case

Assigning a gravitational factor of '-1' in the Z direction instructs S-FRAME to calculate the self-weight of the structure.

Define a Load Combination

A load combination allows the user to factor and combine different load case results. A load combination typically uses factors recommended by an engineering code. Load combinations are defined through the Edit menu in the **Loads Window** or through the loads Spreadsheet in the **Spreadsheets Window** (the Spreadsheets Window gives more space than the Edit Menu option).

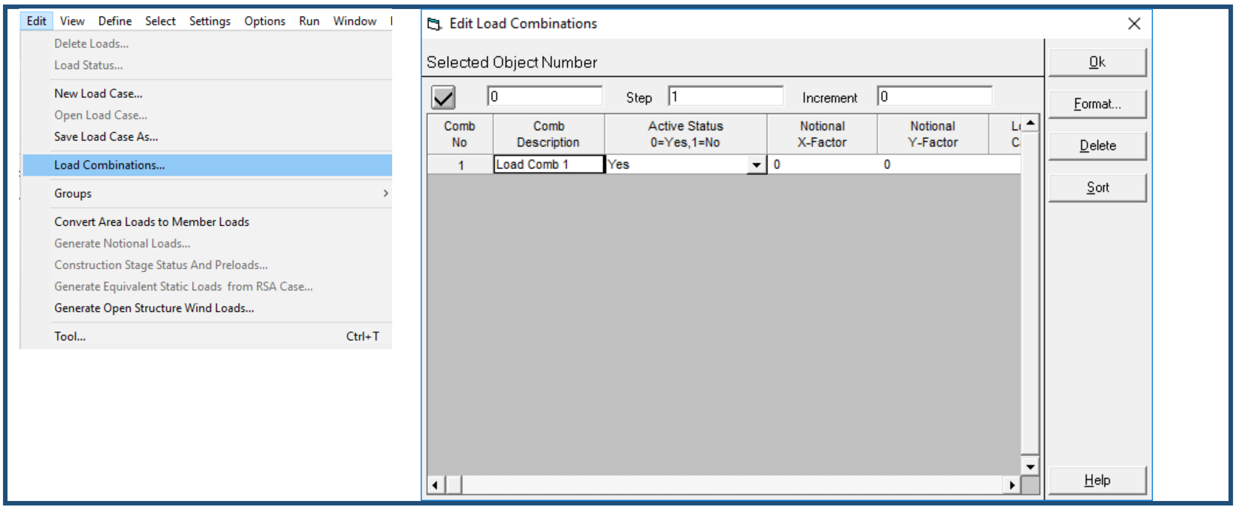


Figure 41. Edit Load Combinations within the Loads Window

5.0 Spreadsheets Window

An alternative way to edit the structure's Geometry and Loading is via the Spreadsheet View. Click **Spreadsheets** on the Window shortcuts to maximize the window.

Within this window, you can modify the structural attributes. Any changes applied here update automatically in your model. The spreadsheets function similar to Microsoft Excel™. You can copy and paste cells, change the size and/or export this information to an Excel spreadsheet, modify and import it back into S-FRAME.

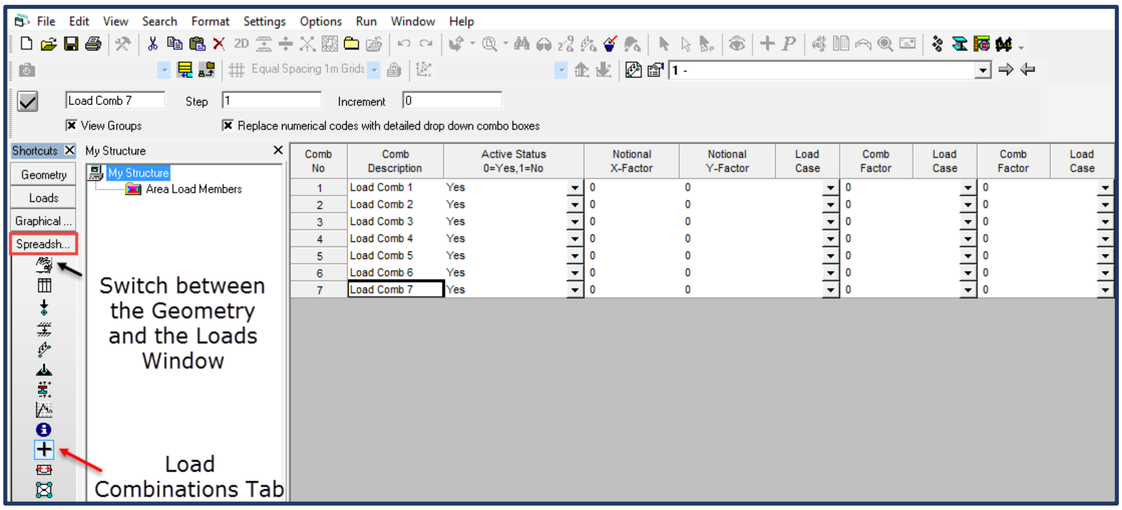
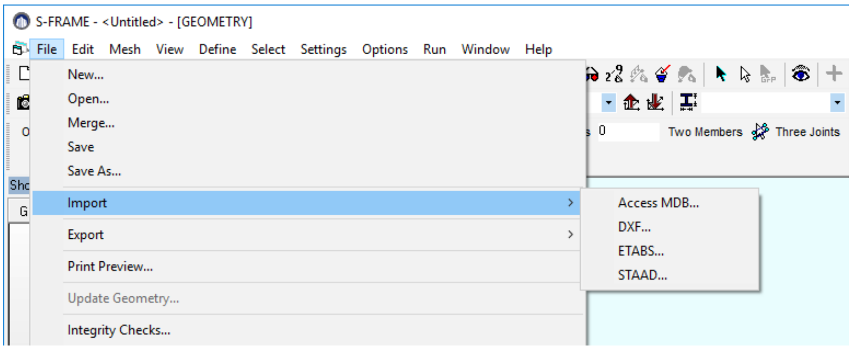


Figure 42. Edit Load Combinations within the Spreadsheets Window

Tutorial videos which demonstrate these features are available in the Online Resources
Help→Online Resources→S-FRAME Analysis Documents

In S-FRAME, you can also Import/Export data from/to a Microsoft Access™ Database. This data exchange is done by going to File→Import/Export→Access MDB in the **Geometry** Window.



6.0 Graphical Results Window

After an analysis is performed on the structure, the Graphical results window becomes active. Similar to the other windows, on the left-hand side we have a set of tools to view the results graphically.

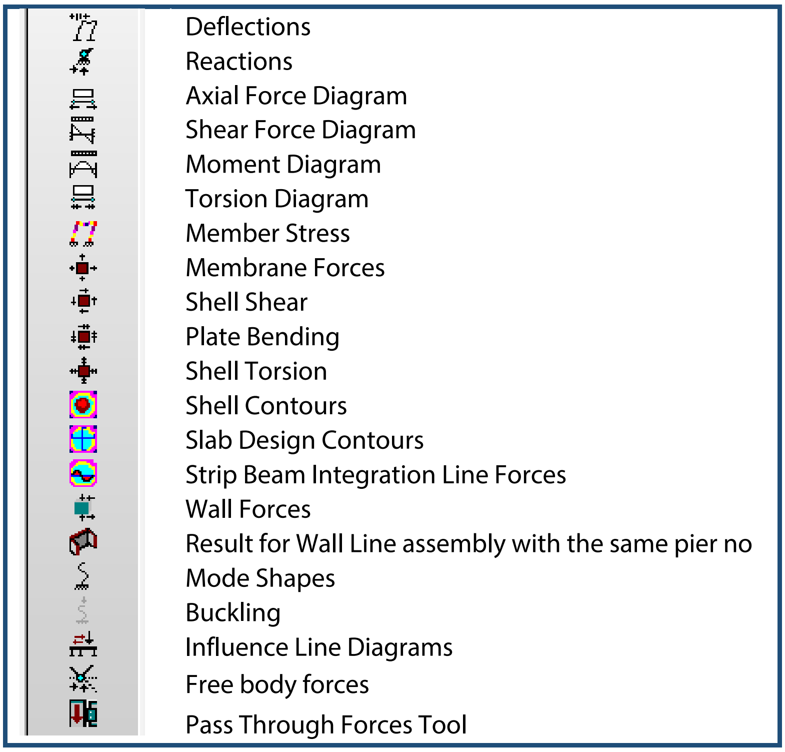


Figure 43. Tools in Graphical Results Window

Depending on which tool is active, we have additional options shown in the data bar above, to specify the results to be displayed.

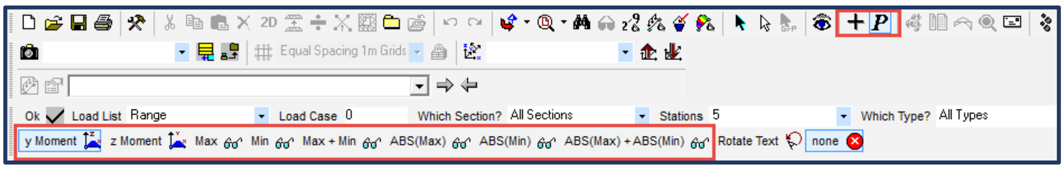
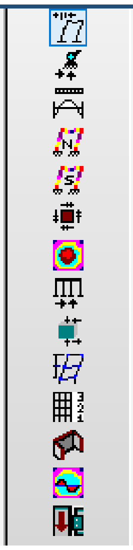


Figure 44.Additional Options to display results

A Tutorial video that demonstrating these features is available in the Online Resources
Help→Online Resources→S-FRAME Analysis Documents

7.0 Numerical Results Window

Results can also be reviewed in a Spreadsheet view, where we can create tables with specific results, and print or export these tables to Microsoft Word™. Click on Numerical Results on windows shortcuts to maximize the window. We can print tables for the following results:



Joint Displacements
Reactions
Member Forces
Member Normal Stress
Member Shear Stress
Shell Forces
Shell Stresses
Base shears, Overturning Moments and Center of Gravity
Wall Forces
Inter Story Joint drift
Floor forces, Storey Shears and Overturning moments
Result for Wall Line assembly with same pier no
Strip Beam Integration Line Forces
Results for Connection Design Pass Through Forces

Figure 45. Numerical Results

To generate reports click the Print shortcut or go to File→Print within the Numerical results window. Specify the properties and the results of interest in the Print Reports dialog, and print the desired results.

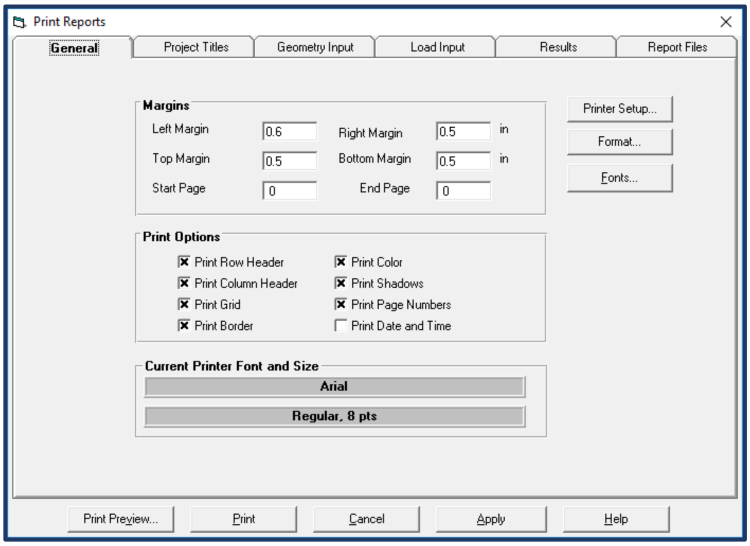


Figure 46. Print Reports dialog

A Tutorial video demonstrating these features is available in the Online Resources
Help→Online Resources→S-FRAME Analysis Documents

8.0 CAD Details

In S-FRAME Analysis you can add dimensions to your structure, and include them in the printed reports through the CAD details window. Once you add the desired dimensions, you can then switch to the Geometry, Loads or Graphical results window and display the defined dimensions by switching to **ON** the **Display Dimension Information** option in the status bar below.

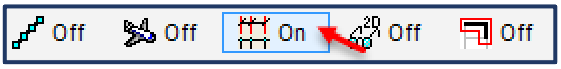


Figure 47. Dimension Information switch

Cardinal Offsets adjustments, which can be done from the CAD Details window, lets you specify non-structural offsets for members so that S-FRAME can render them more realistically. You can, for example, align the top of steel for floor beams—of varying depth—so that the top edges are coplanar.

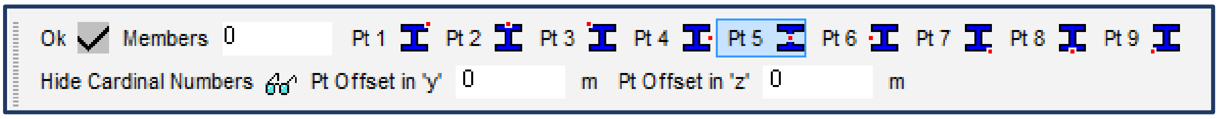


Figure 48. Cardinal Offsets

You can find useful information for the Cardinal Offsets, in S-FRAME's Help System (Help → Contents → 'Search Tab') under the 'Cardinal Points' topic.

9.0 Example

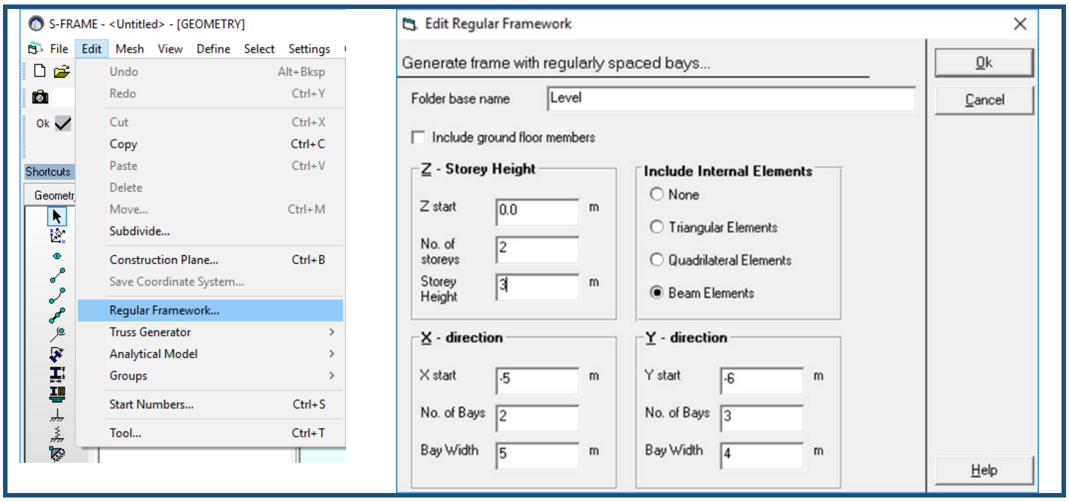
Below is a simple example designed to present many of the features discussed in the preceding pages. You may need to refer back to earlier sections in order to complete this example.

In this example we build a hypothetical 3 storey structure using the Regular Framework command and the Member definition tool. We then define section and material properties, support conditions and run an analysis.

Define Geometry.

1. Open S-FRAME Analysis if not already open (chapter 1.0 First Steps, page 6) and from the New Structure dialog select the second option: **Create a Regular Frame Work**.

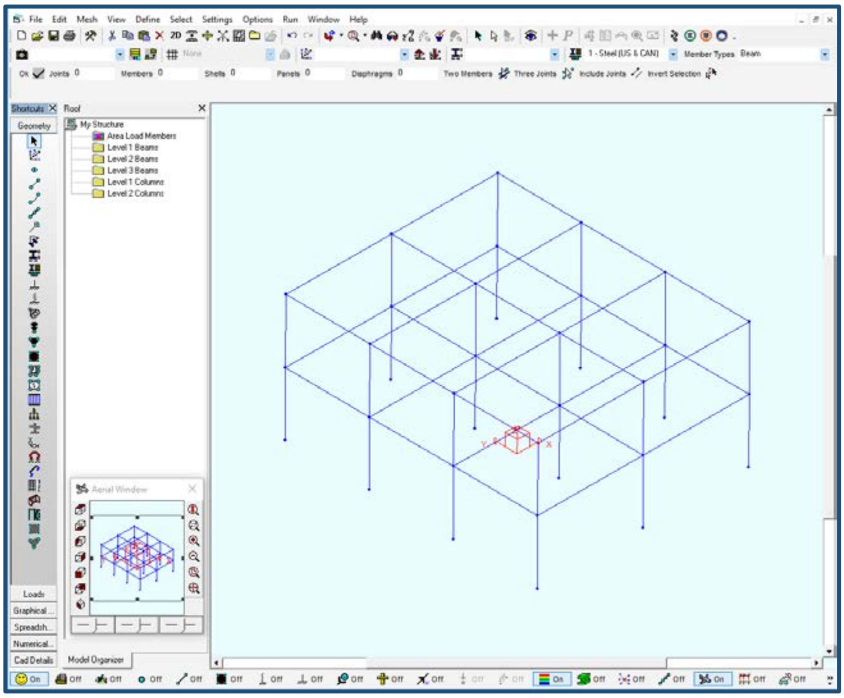
   If S-FRAME is already open, Go to **File→ Regular Framework**, to bring up the **Edit Regular Framework** dialog.
   Use the Regular Framework command to build the first two floors of our structure, where each floor is 3m high, with 2 bays of 5m in one direction (X), and 3 bays of 4m in the other direction (Y). We also want to keep the UCS at the centre of the structure, and this is done by entering the appropriate X and Y start coordinates. Once you enter the input as shown below, click **OK.**



Example Figure 1.

Any model generation or editing (translate, rotate, reflect, shear, scale...etc.) is always performed relative to the current UCS.

Based to the above information S-FRAME builds the model, and your structure should appear. Note the folders that are created for the structural objects of each level.

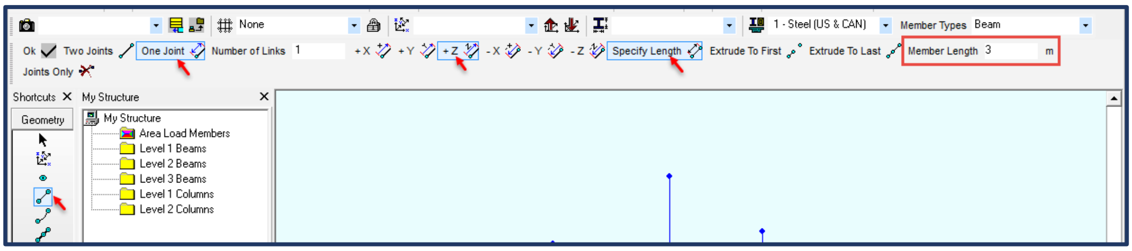


Example Figure 2.

2. Next, we define the 3rd storey using the **Member definition tool**. There are numerous approaches available for defining members using this tool, as we have already seen in Chapter 3 "*Define geometry using the Member Definition Tool*". First add additional members using the "one joint" approach:

Define a member using the one joint approach.

a. Select the **Member definition tool** from the tool bar on the left-hand side
b. Click on the **One Joint** option on the data bar
c. Specify the **Direction** of the member, in this case +Z
d. Select the **Specify Length** option for the extrusion method
e. Define the **Member Length**, in this case 3 m

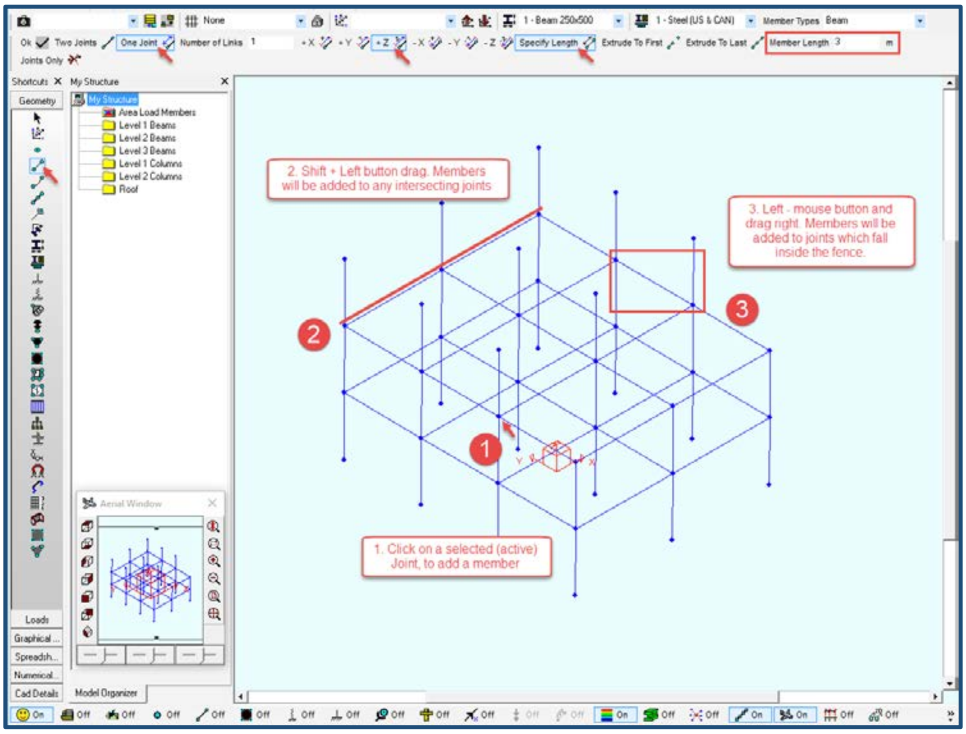


Example Figure 3.

Now that we have specified the member definition properties, we can add a 3m long members in the +Z direction by clicking selected (active) joints. We can select one joint at a time to add one member, or select multiple joints at a time to add multiple members.

The screenshot below demonstrates the single and the multi-select options we can use to pick Selected (active) joints and define the connected members.

Add nine additional 3 meter members (columns) to your structure, in the positive Z direction using the three different methods shown in the image below.



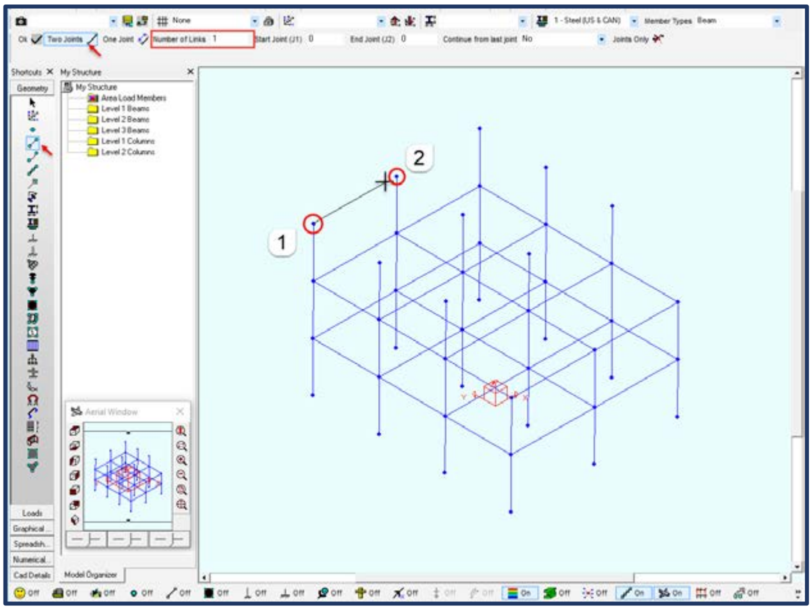
Example Figure 4.

3. After creating the nine additional members (columns) the next step is to create our model's third story by adding additional members (beams) in the X-Y plane, as shown in Example Figure 9

   Define a member using the two joints approach.

   a. Choose the **Member definition tool** from the toolbar on the left-hand side
   b. ii. Click on the **Two Joints** option on the data bar
   c. iii. Define a member by selecting two joints, as shown in *Example Figure 5*

   Note that the **Number of Links** is set to 1. This setting is used to introduce joints (nodes) along the length of a member. When the number of the links is 1, S-FRAME does not add additional nodes and the modelled member has only its **Start** and **End** joints.



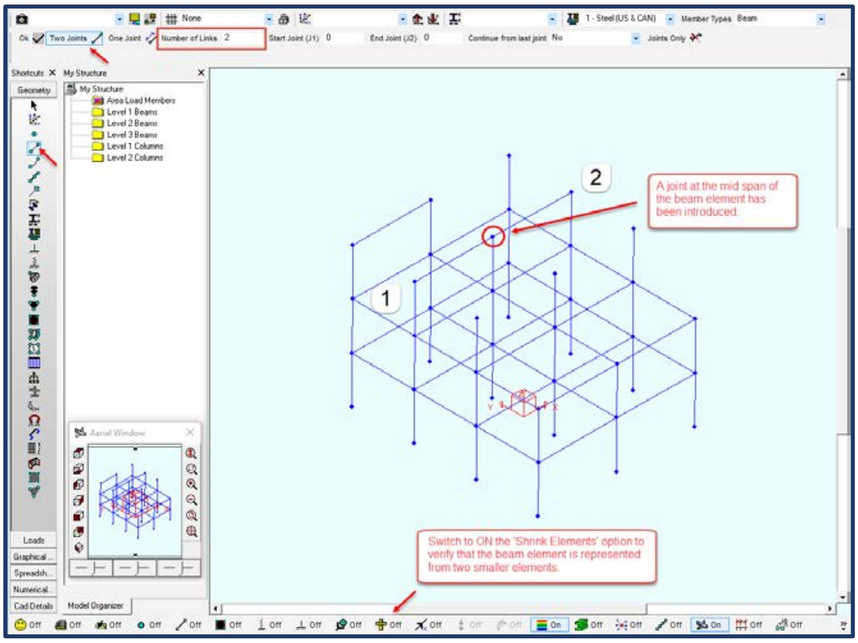
Example Figure 5.

Adding additional nodes along the length of a member is necessary in some cases, such as:

- Define a joint load on a member
- Get results at a specific point of a member
- Use smaller elements to represent a long single element
- Represent complex deflected shapes or mode shapes
- Connect additional members to a member.

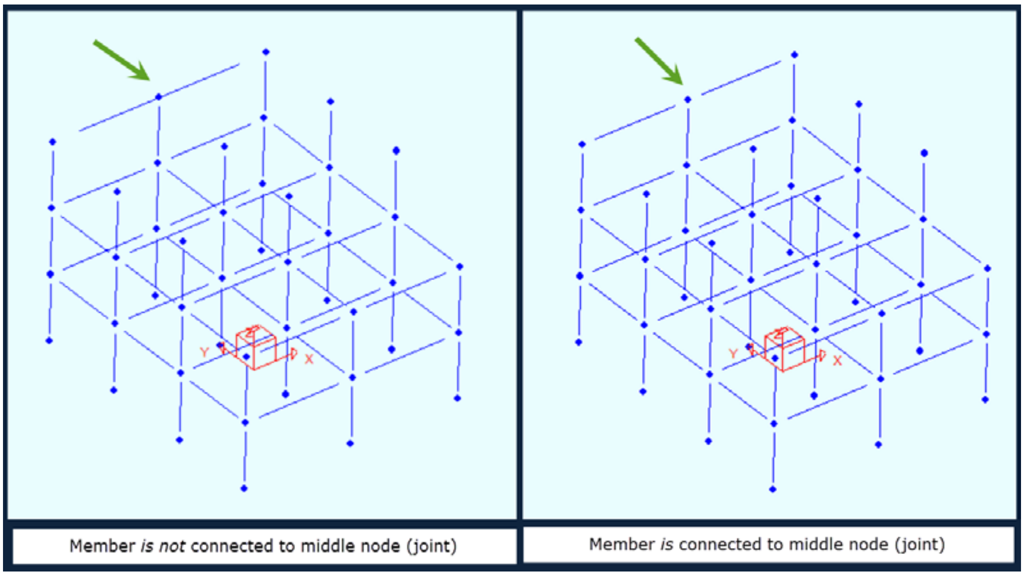
We will now add another member, ensuring that we connect additional members (columns) to the new member (beam)

Change the "Number of Links" setting to "2" and create the additional member by clicking on the points as shown below (1 and 2). S-FRAME builds a 10m long beam using two 5m long beams. This method ensures that the beam and the intersecting column are connected at the additional joint located at the beam elements' mid-span. **Please note that this is the case here because the column is located at the mid-span of the beam element we added.**



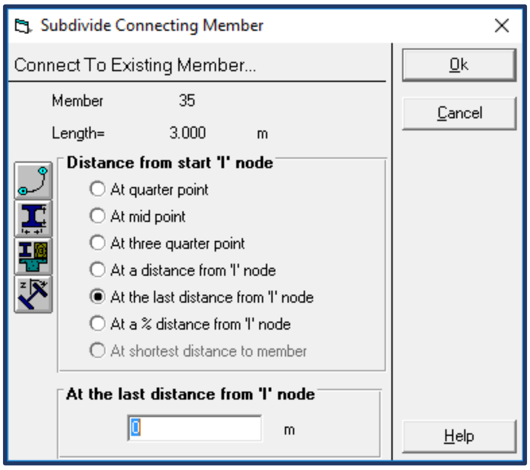
Example Figure 6.

Clicking the "Shrink Elements" icon will change the display to easily view which members are connected to which joints (nodes).



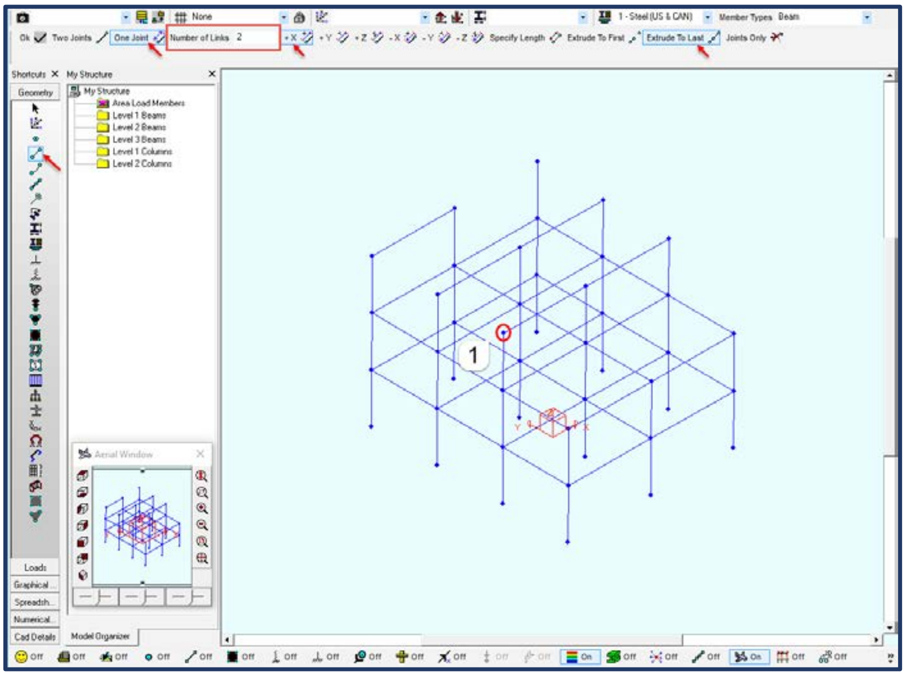
Example Figure 7.

To subdivide an existing member into two members, first select the Member Definition Tool and then click anywhere on the member where a join does not exist, to open the Subdivide Connecting Member dialog. Here you can specify the exact connecting location and the resulting new member.



The number of links is also available when using the one joint approach. Note the additional extrusion options and define a member as shown below.

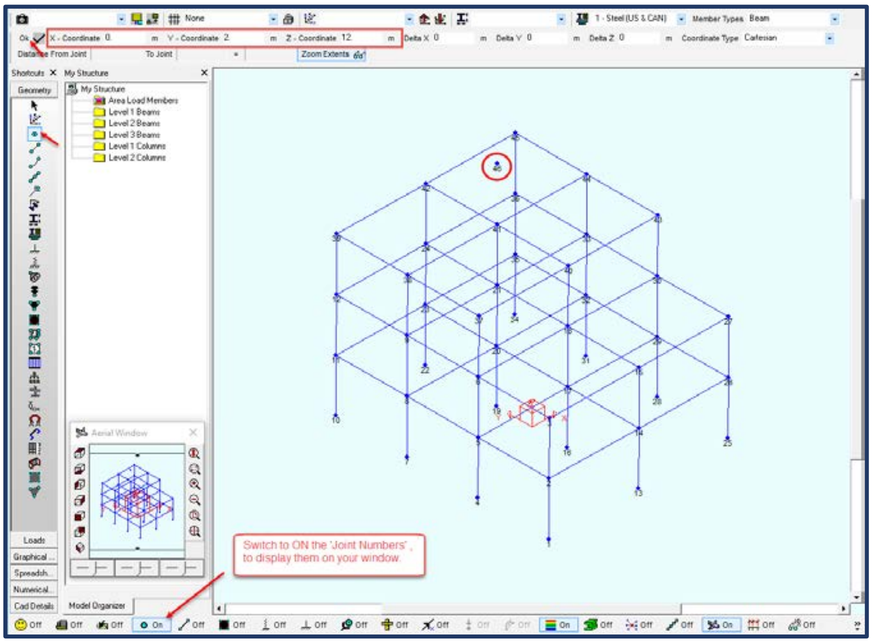
Another member creation option that utilizes the "Number of Links" option is to the use the one joint approach combined with the "Extrude to Last" option. Select the Member definition tool, set the number of links to 2, the direction +X and click on "Extrude to Last" as shown in Example Figure 8. Selecting the join labelled #1 will create members shown in example figure 8.



Example Figure 8.

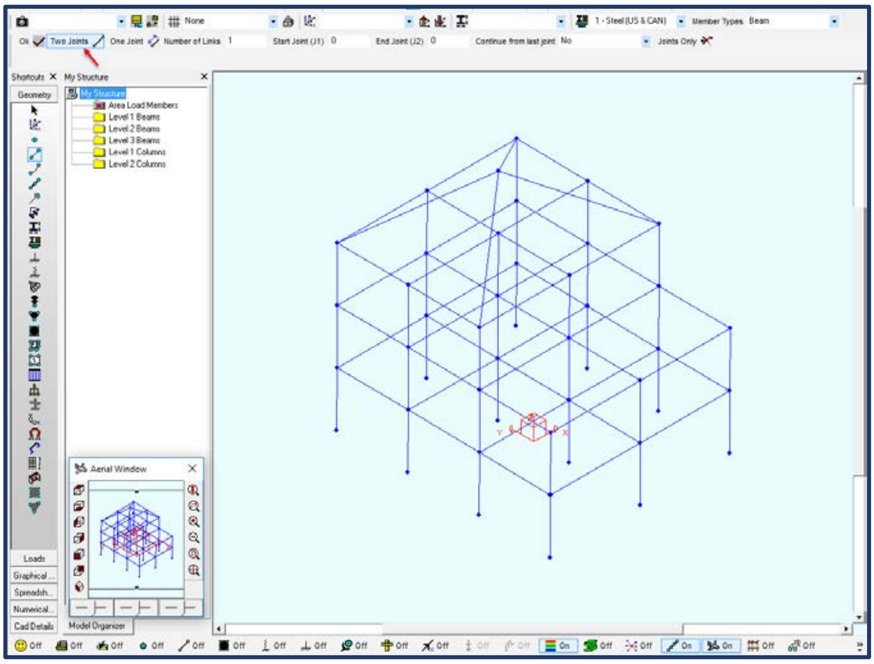
Finish connecting the 3rd-floor members (columns) using any of the above methods so that your model looks like Example Figure 9.

4. Once the 4th level members (beams) are added, the next step is to add a roof to our structure. We will first create a new joint (node) that defines the roof's peak.
   Select the **Joint Tool** and add a joint at location (0, 2, 12) by entering the desired coordinates in the data bar below and clicking OK.



Example Figure 9.

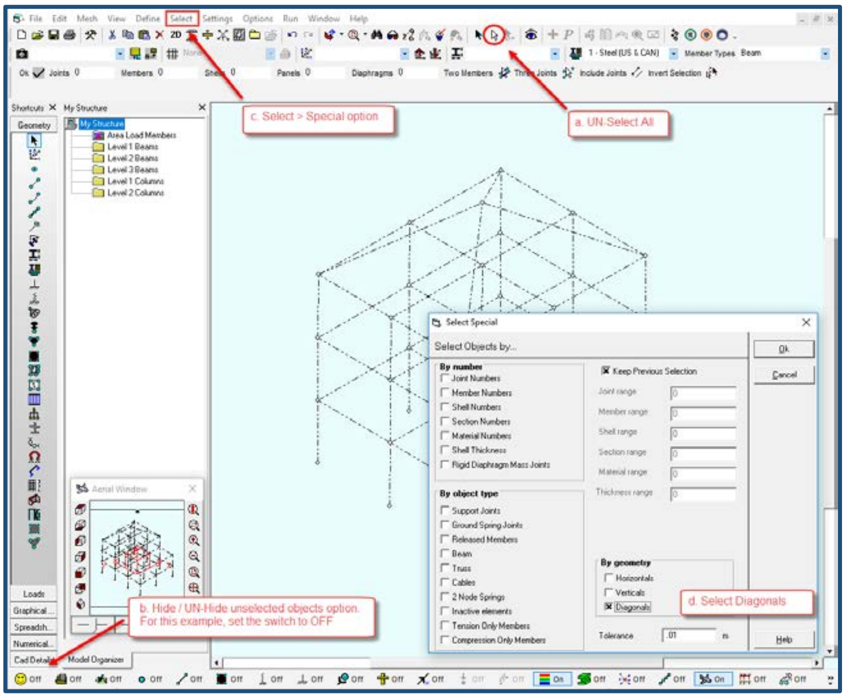
Using the Two joint approach define the roof beams as shown below.



Example Figure 10.

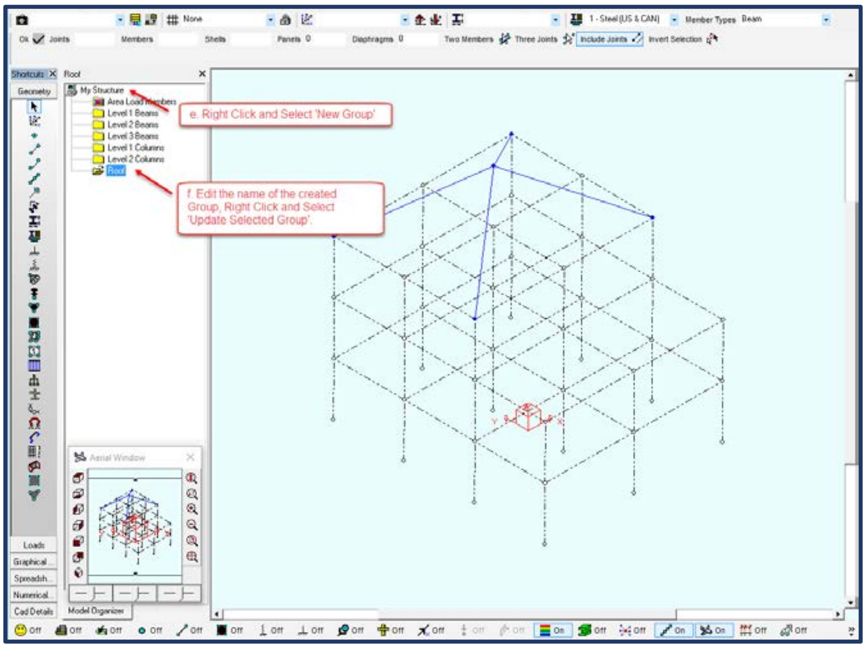
5. Next, we create a folder which includes our roof beams only.

a. Unselect all objects using the **Unselect All** shortcut
b. Make sure that the '**Hide Unselected Objects**' option is not set, by switching to **Off** the appropriate switch in the **Status Bar** below.
c. Go to **Select→Special**.
d. From the Select Special dialog, select the **Diagonals** option and click **OK**.



Example Figure 11.

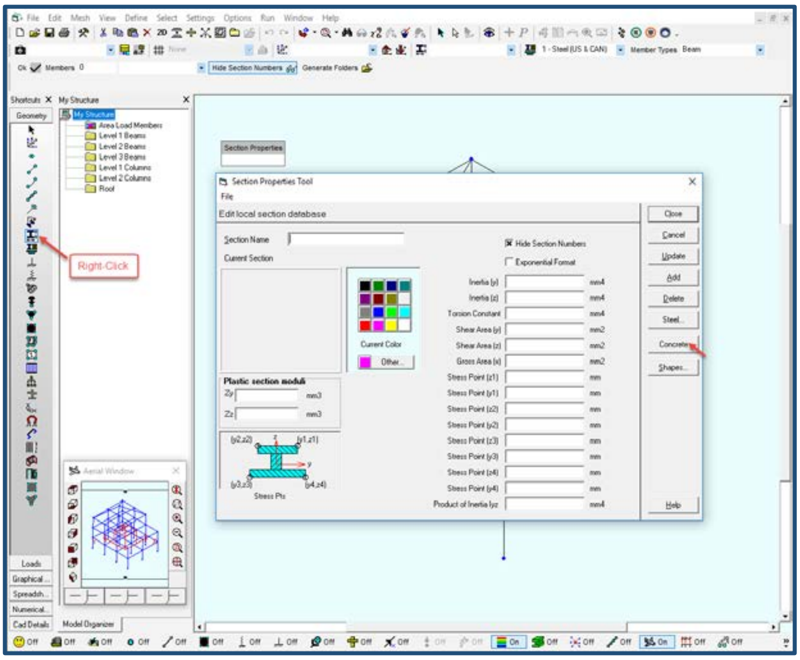
e. Right-click on '**My Structure**' and Select **New Group** from the pop-up menu.
f. A group is created. Edit its name if you wish, then right-click the group folder and select **'Update Selected Group'** from the pop-up menu.



Example Figure 12.

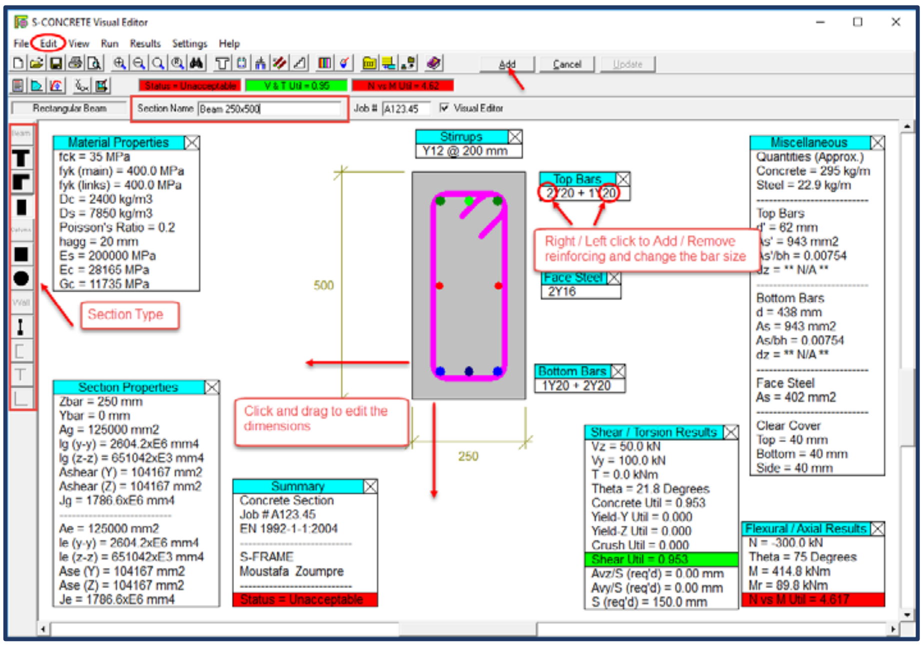
We have now created a group, which contains our roof beams. S-FRAME's folder technology is a powerful way to isolate objects and make editing objects easier as demonstrated next.

6. Next, we define **Section** Properties.
   a. Right-Click on the **Section Properties Tool** to bring up the associated dialog, and select **Concrete** (this can only be accessed if you have an S-CONCRETE and an ICD license). Alternatively, create a custom or tapered shape. The tapered shape shows up in the renderings.



Example Figure 13.

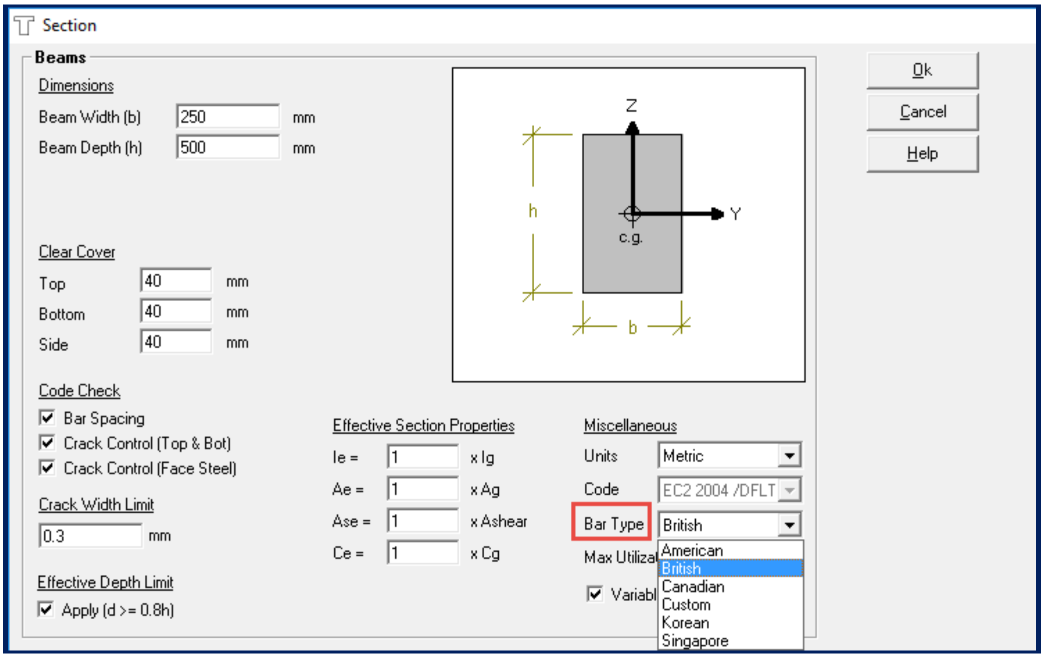
   b. In the **S-CONCRETE's Visual Editor** you can define and edit concrete sections. Section and reinforcing properties can be edited under the Edit menu, or graphically as shown below. Add a 250x500 beam column section by selecting the appropriate section type on the left-hand side, edit their dimensions, reinforcing, name and click the Add button.



Example Figure 14.

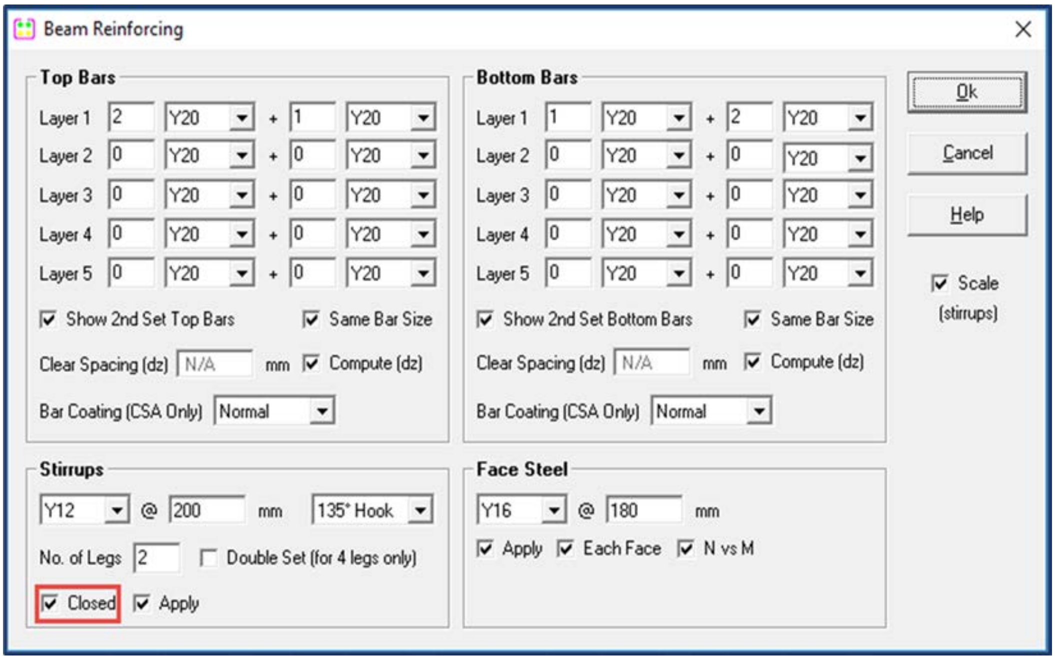
For this example, it is worth noting that S-FRAME Analysis only takes into account the Section Properties: the dimensions and the material. If your bar names and stirrups do not display as shown in Example Figure 14, you can set the following options:

Go to Edit→Section and select British for Bar Type



Example Figure 15.

Go to Edit→Reinforcing and select the closed option under the Stirrup data.



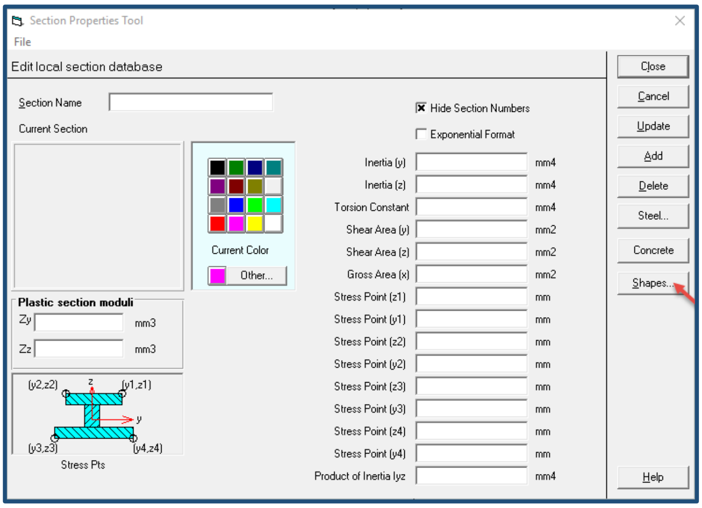
Example Figure 16.

c. Repeat this process to add a 500x500 concrete column section. Choose the "Rectangular Column" section type from the list of selection types on the left-hand side, change the dimensions to 500x500, provide a section name, and click the Add button.

> We could define sections using Custom or Tapered sections. However, when we define a section using S-CONCRETE's Visual Editor, the Editor automatically calculates the required section properties necessary for the analysis to run and creates additional files that include information needed to perform both a code check and a design of our structure.

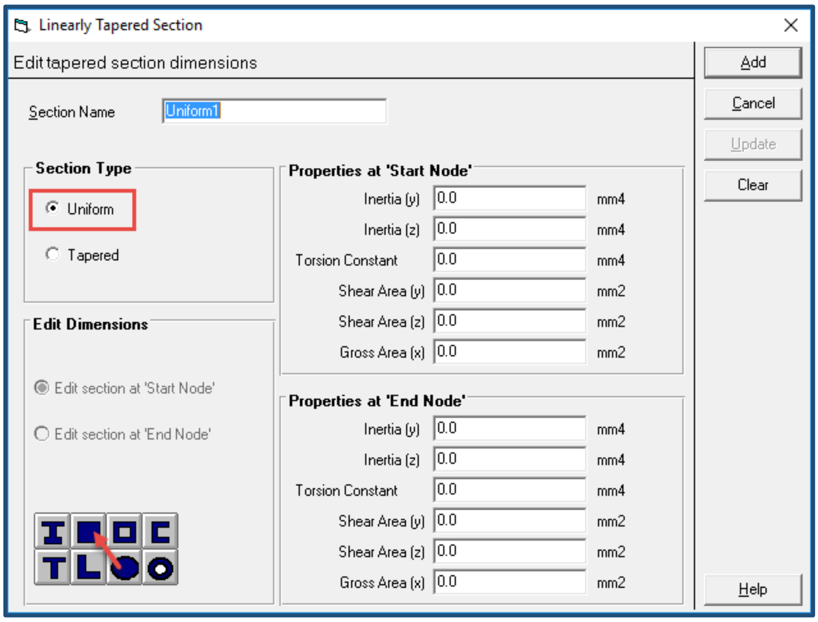
For those of you who do not have access to ICD (Integrated Concrete Design), the next steps describe how to use the **Custom** or **Tapered** option to define the concrete beam and column sections previously created using ICD.

We will define the sections using the **Shapes** option. Click on this option to bring up the associated dialog box.

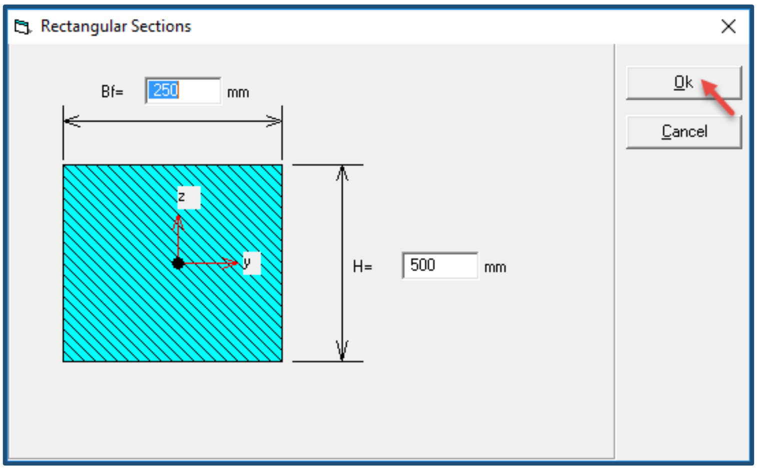


Example Figure 17.

Note that in the **Linearly Tapered Section** dialog box, you can specify whether the section will be **Uniform** or **Tapered**. Select the Uniform option, and click on the rectangular section below, to bring up the **Rectangular Sections** dialog box and define the dimensions of the section.

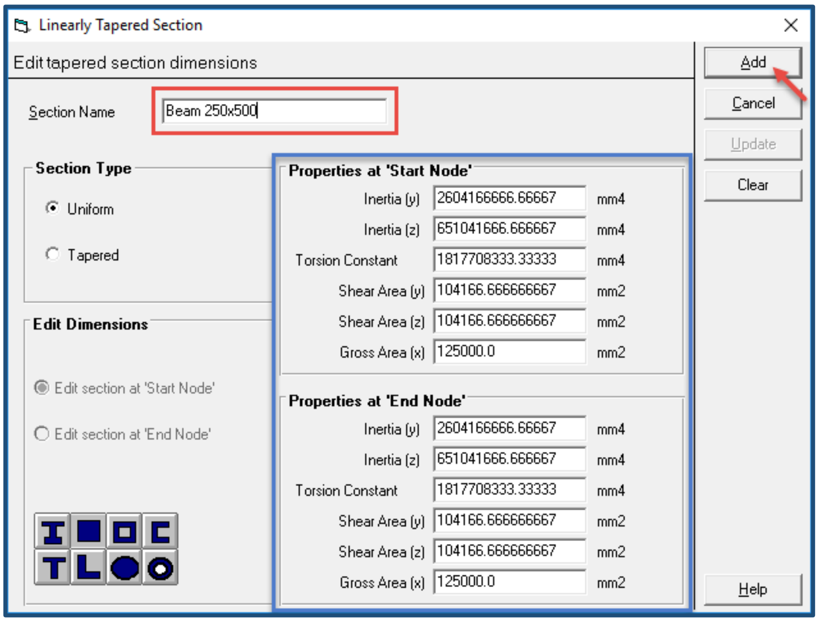


Example Figure 18.



Example Figure 19.

Once the dimensions have been defined, click **OK** and note the calculated section properties of the section. You can now edit the name, and click the **Add** button to add the new section to your sections library.



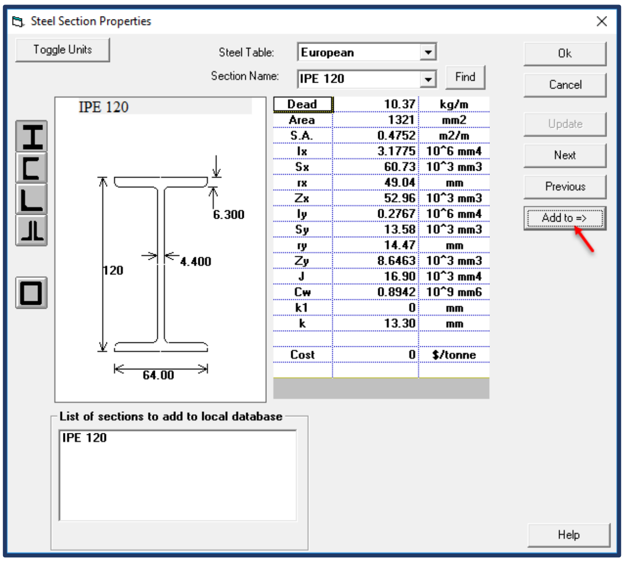
Example Figure 20.

Repeat the same process to add a 500x500 column section.

For compound section shapes not available in S-FRAME, you have a few options to define their properties and perform an analysis:

1. Calculate the section properties (moment of inertia, torsion constant, shear area and gross area) and add them manually by entering the values into the appropriate field
2. Use S-CALC™, a section property calculation tool, to calculate the section properties of any shape, and import them into S-FRAME.

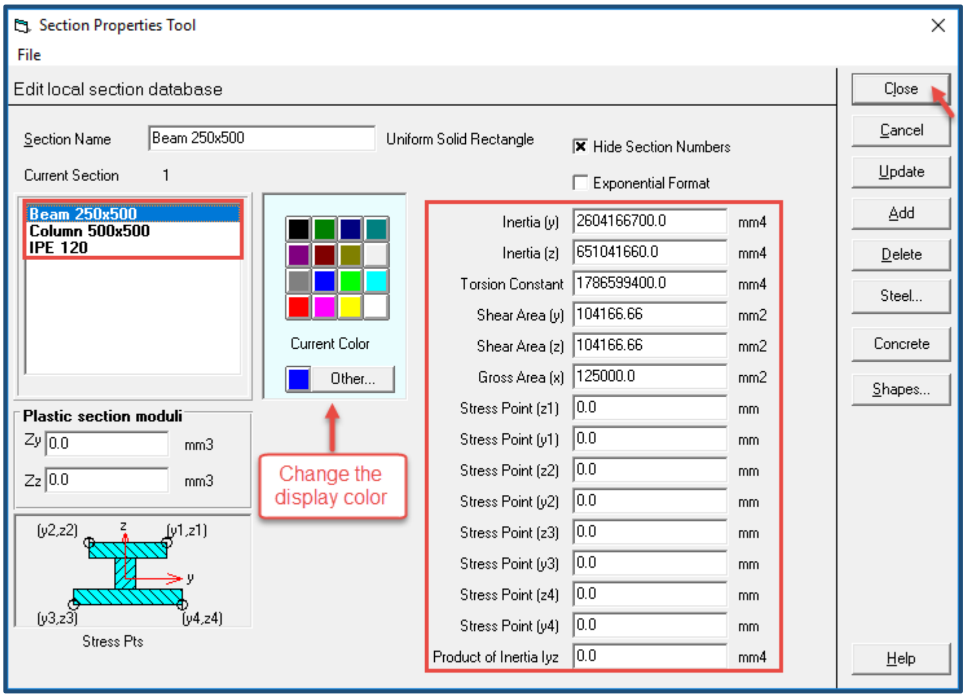
7. In addition to concrete sections, we will add a steel section next. Following the same process we add an I shape (IPE 120) steel beam, for our roof beams from the European steel table as shown below:



Example Figure 21.

To do this, right-click on the **Section Properties Tool** and this time select Steel to open the Steel Section Properties Dialog. Select the European Steel Table and the IPE 120 section from the steel database. Click "Add to => then Click **OK** to close the dialog.

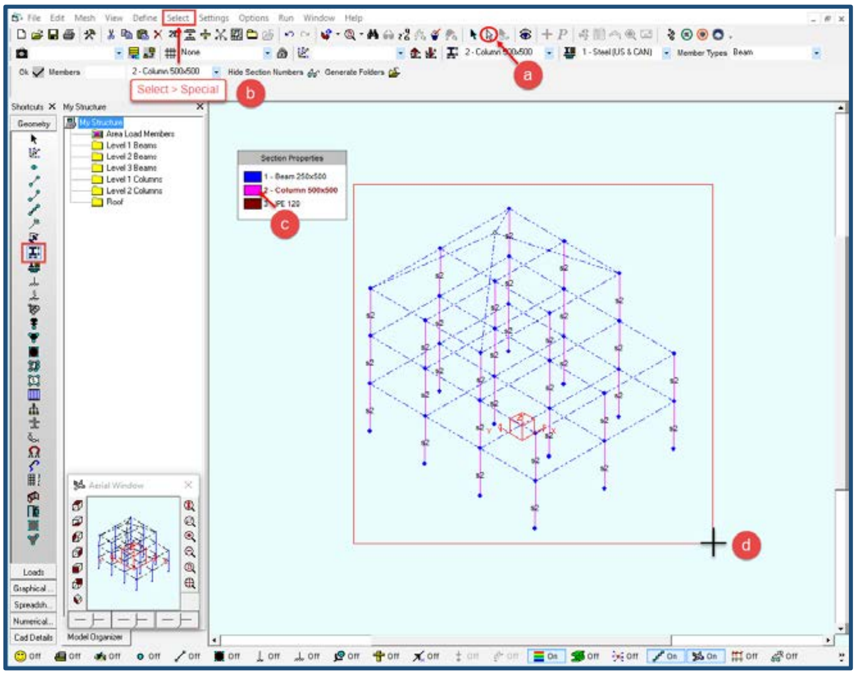
8. The added sections should appear on the left-hand side in the **Section Properties Tool** dialog. Within this dialog, you can change the display colour (select the section and select a colour), review the section properties, and more. Once done, click **Close** to close the dialog.



Example Figure 22.

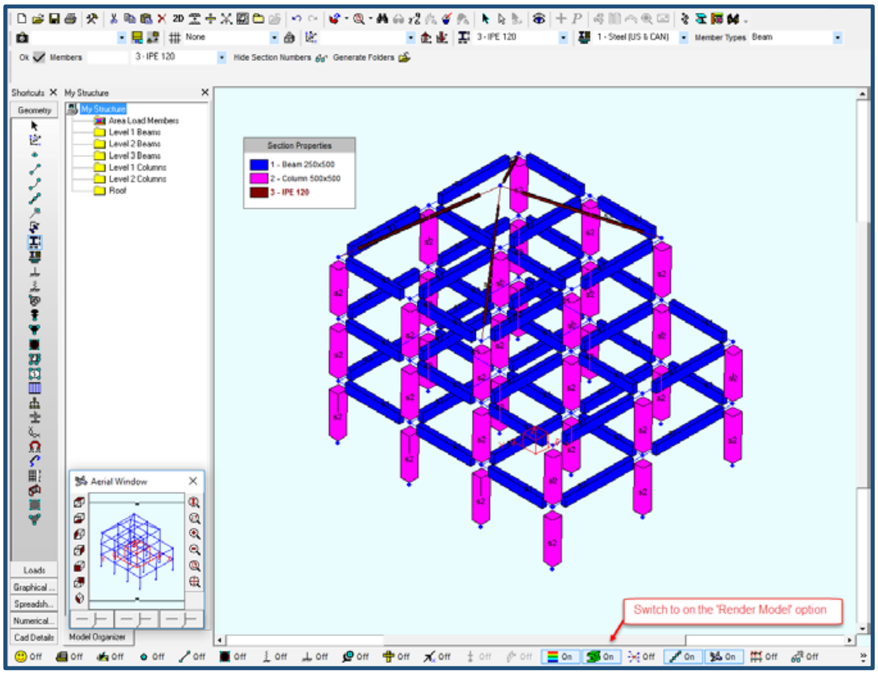
9. By default, S-FRAME assigns the first section to all of our members. We change the required elements' section properties using the Select Special option.

a. Unselect All
b. Go to Select → Special, and select Vertical elements only, then Click **OK**
c. While the Section Properties Tool is selected, click on the legend colour change the section (this can also be done through the combo bar above).
d. Once the appropriate section is selected, create a fence to the entire model to assign the section to your columns, as demonstrated below;



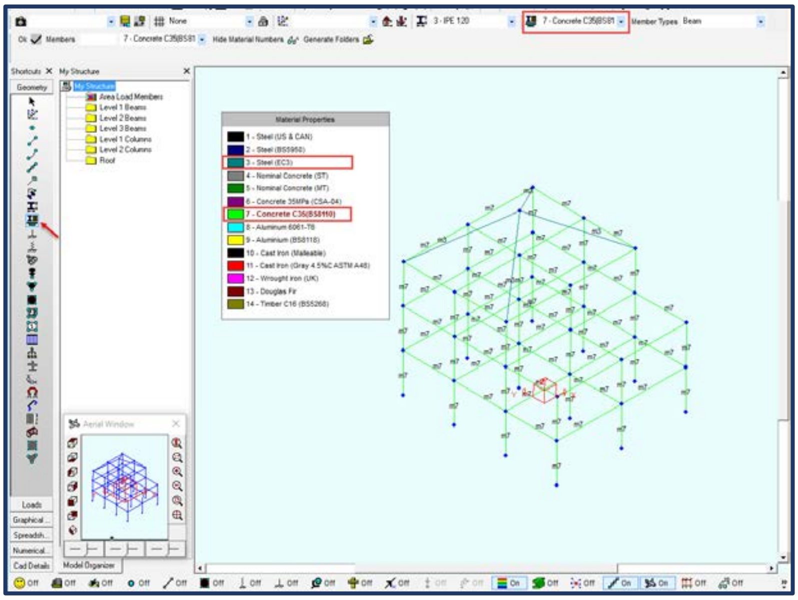
Example Figure 23.

10. Following the same steps, assign the I shape beam to your roof beams (Diagonals), and render the model to validate the assigned sections. Recall, the horizontal members were already assigned, by default, to the first defined section.



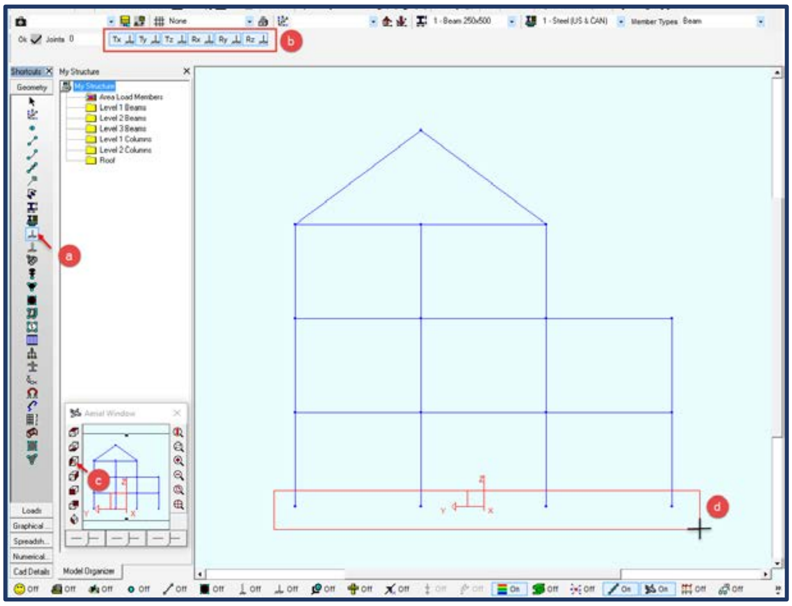
Example Figure 24.

11. The next step is to define Material Properties.
    a. Select the **Material Properties Tool**
    b. Select the **Concrete C35 (BS8110)** from the legend (clicking on its colour) or from the combo box above
    c. Create a fence to include the entire structure
    d. We have now assigned this material property to all members of our model. Open the previously created Roof folder by clicking on it. Select the **Steel EC3** material and create a fence which includes the entire structure, to define the appropriate material to our roof beams.



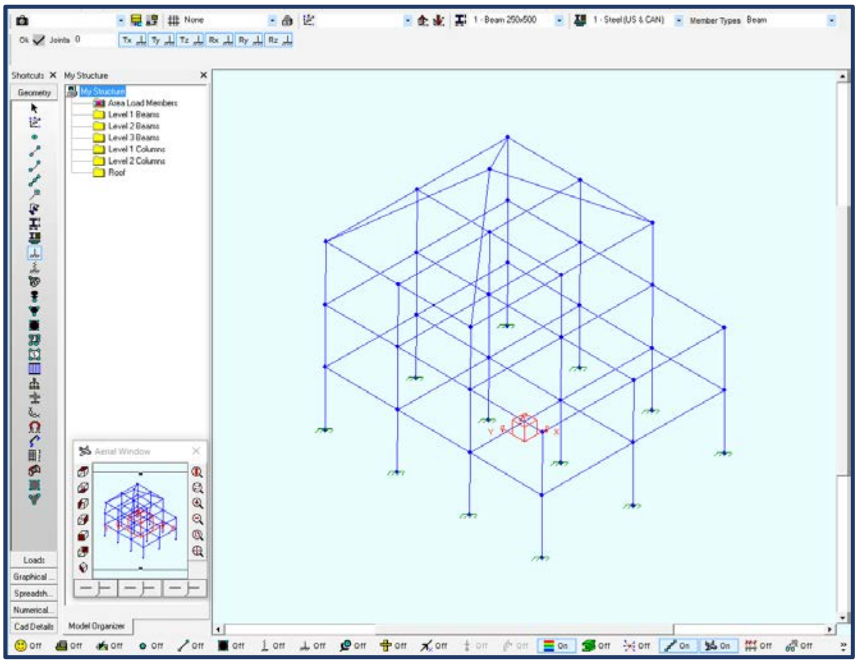
Example Figure 25.

12. Next, we define the support conditions of our structure.
    a. Select the **Supports Tool.**
    b. Note the constraints added in the **Data bar** above.
    c. Switch to the left view, using the Aerial Window.
    d. Create a fence which includes the joints of the basement of our structure.



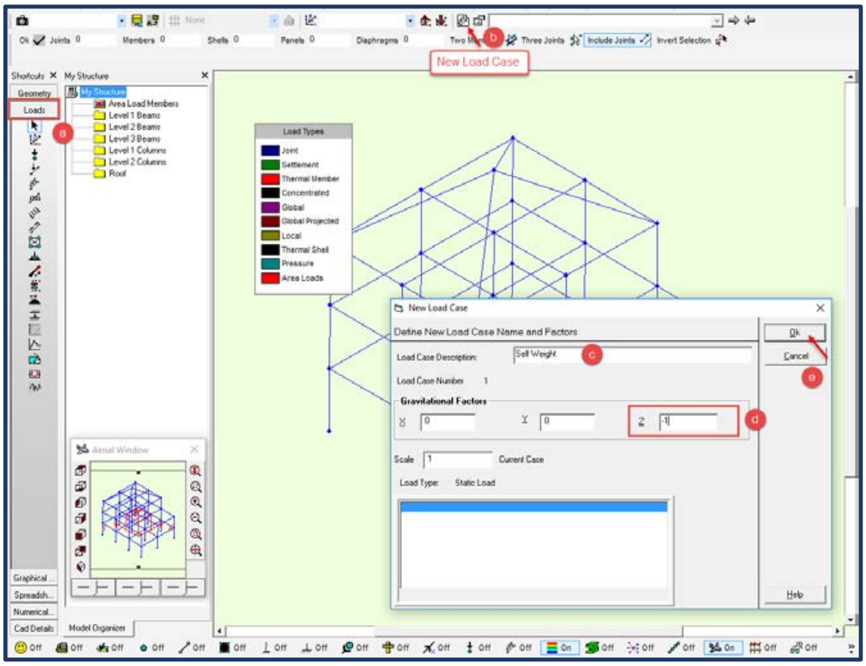
Example Figure 26.

e. Switch to an Isometric view and make sure that fully fixed supports have been defined for all basement joints.



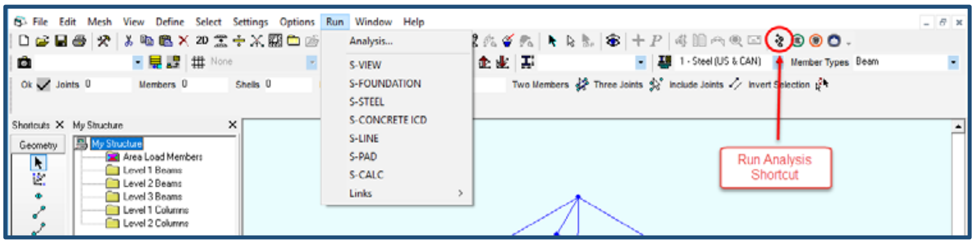
Example Figure 27.

13. The last step of this example is defining a self-weight load case.
    a. Switch to the **Loads window**
    b. Click the **New Load Case** shortcut
    c. Enter a Description for this load case, **Self-Weight**
    d. Enter a Gravitational factor **-1 about the Z axis**
    e. Click **OK** to complete the process



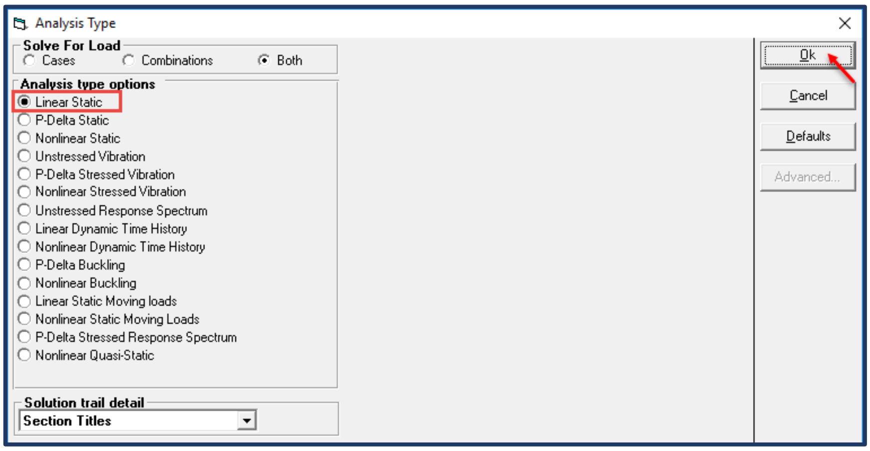
Example Figure 28.

We can now run an analysis. Go to **Run→Analysis**, or click on the **Run Analysis** Shortcut.



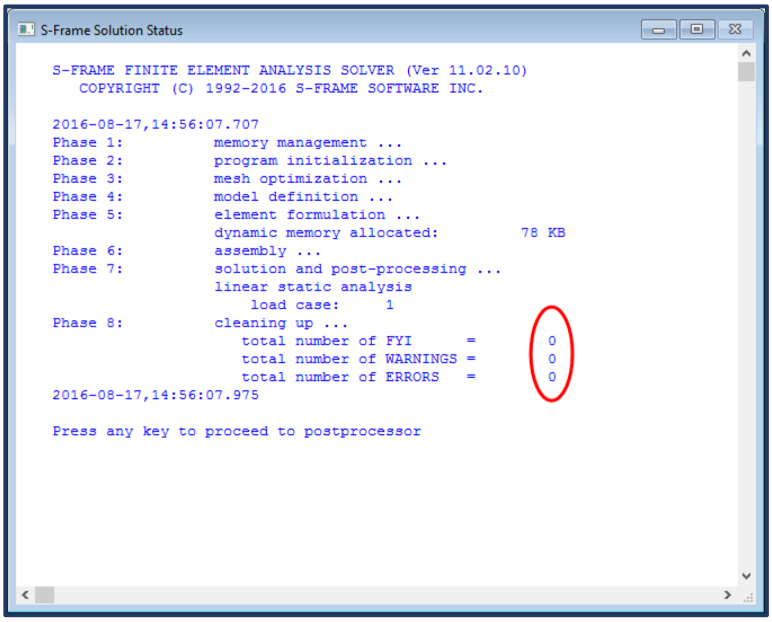
Example Figure 29.

Select **Linear Static** in the **Analysis Type** dialog, and click **OK** – **Save** the file if prompted.



Example Figure 30.

You may be asked first to save your model, click yes. Upon solving the model, S-FRAME opens a *Solution Summary* window. Always examine the summary and check that there are no FYI's Warnings or Errors.



Example Figure 31.

If the solver encounters any issues, they are reported here. Each issue has an ID and we refer to them as **Solver Diagnostic Messages** (SDM). All SDM are described greater detail in the document found in the Online Resources.

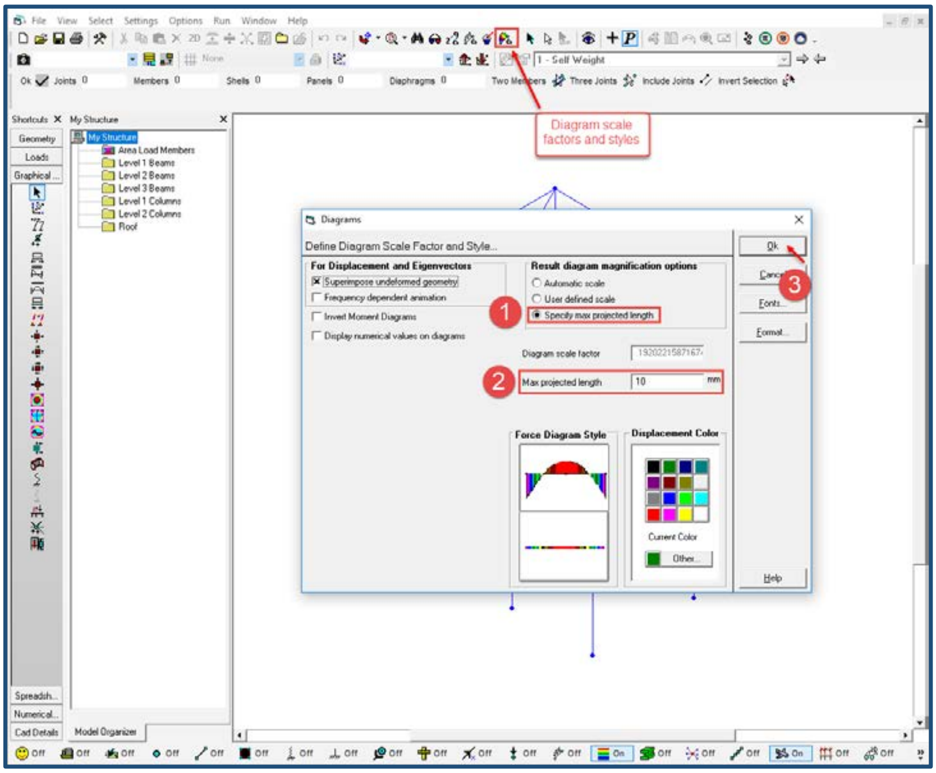
Help→Online Resources→S-FRAME Analysis Documents

Hit the **Enter** key on your keyboard when you are ready to proceed. After the analysis phase, S-FRAME launches the Graphical results window. In this window, you can view results of interest by selecting the appropriate tool to click on the additional options in the data bar above.

First, you may want to set up the scale factors so that the results are easy to view. Select the **Diagram scale factors and styles** shortcut to open the Diagrams dialog box shown below:

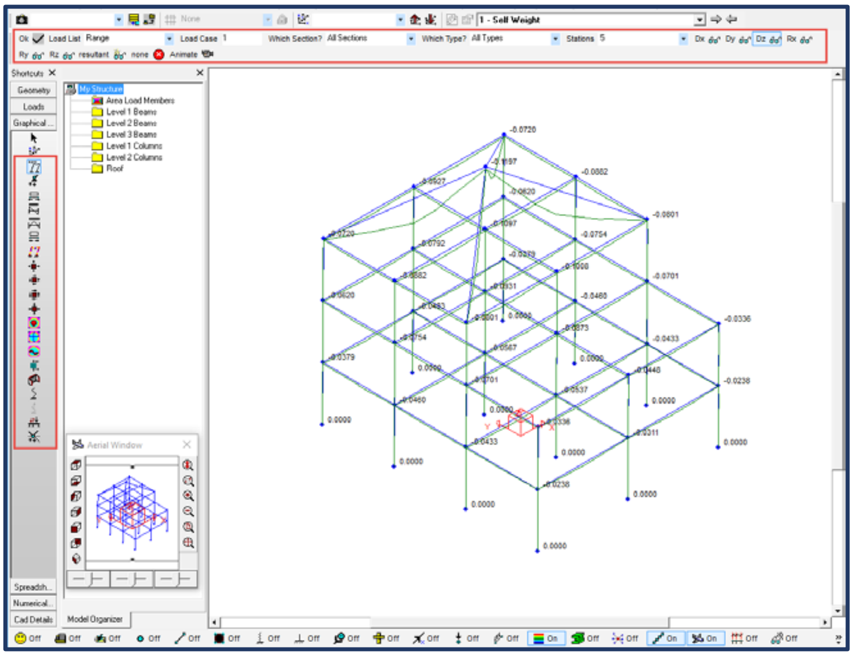
1. Select the **Specify max projected length** for **Result diagram magnification option**.
2. Set the **Max projected length** to 10mm
3. Click **OK** to accept the changes

Note the additional options we have available for plotting results.



Example Figure 32.

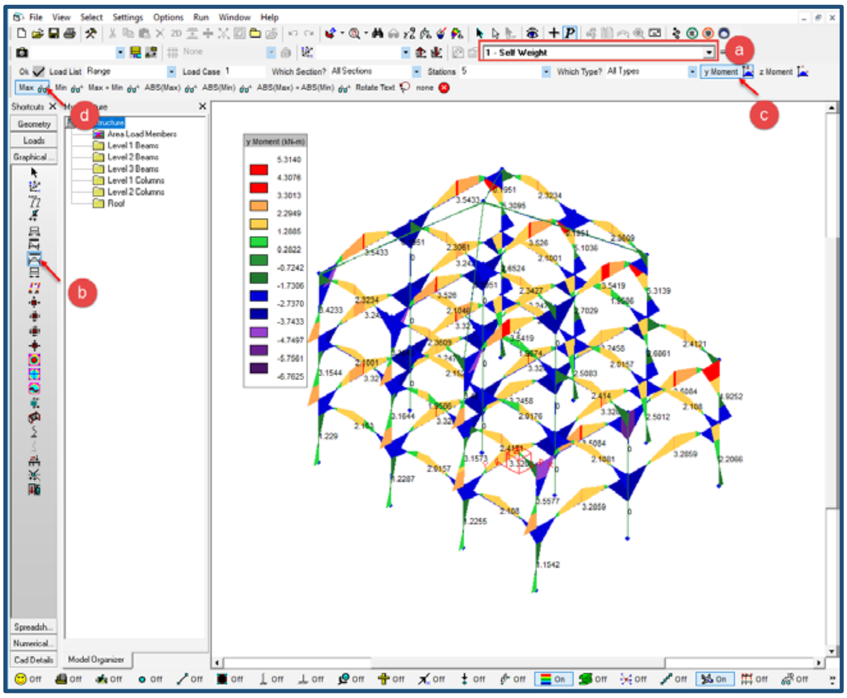
Once the scale factors and styles have been set up, select a tool and the results associated with the tool are displayed graphically.



Example Figure 33.

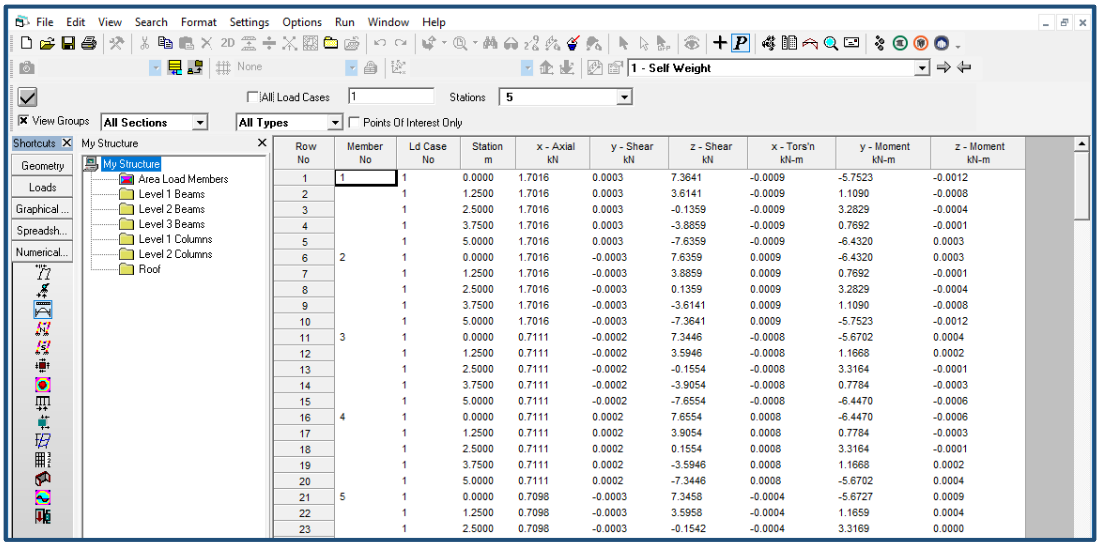
For instance, to view the moment diagrams about the Y axis for the Self-Weight Load case and display the minimum values;

a. Select the Load case of your interest
b. Select the Moment Diagram Tool
c. Select the moment diagram you want to plot
d. and the values you want to display



Example Figure 34.

Additionally, you can switch to the Numerical Results window, view results in a spreadsheet view and generate reports as demonstrated in our **Tutorial 09 – Numerical Results** video.

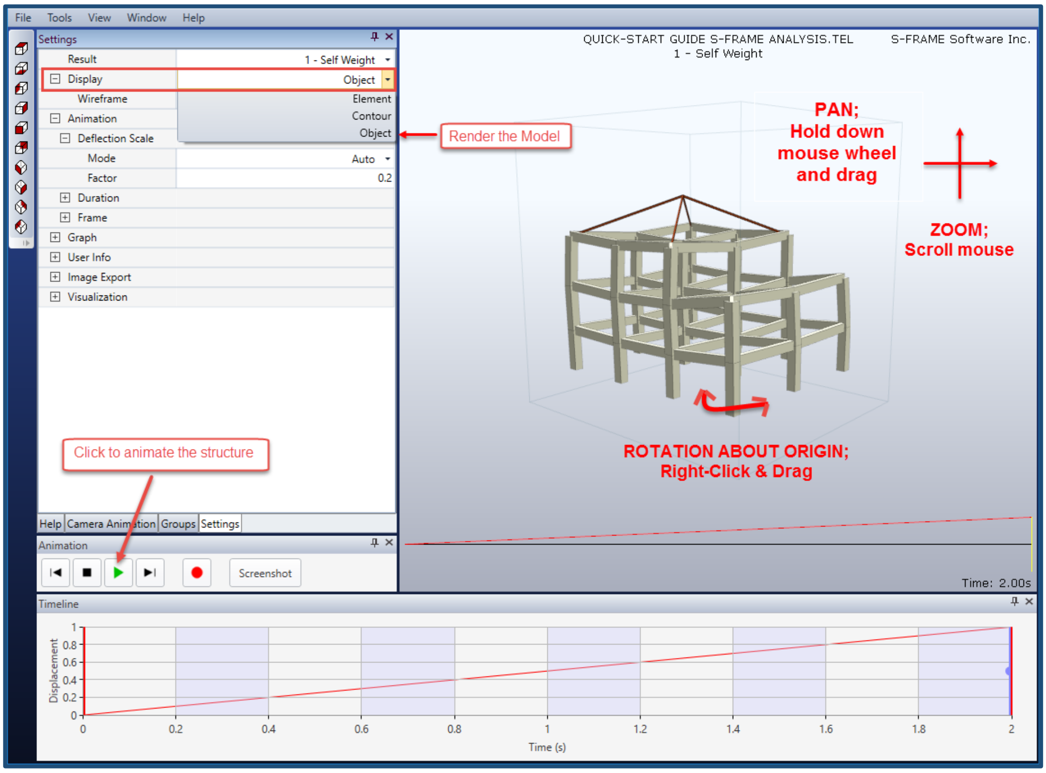


Example Figure 35.

10.0 Validation and Sharing with S-VIEW

S-FRAME R 11.1 and later are compatible with our newer generation products (S-VIEW, S-CALC and S-FOUNDATION).

S-VIEW is a free program that comes with S-FRAME. S-VIEW launches within S-FRAME by selecting from the top menu bar **Run→S-VIEW**. This opens the S-VIEW window to allow users to view their model and analysis results.



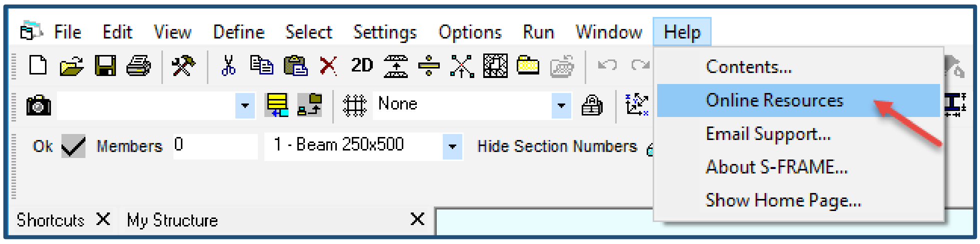
Example Figure 36.

S-VIEW is very effective for investigating and validating your post analysis structure by rendering and animating its behavior. It is also useful for sharing your models with colleagues - it is free to download and runs independently of S-FRAME Analysis. Users without S-FRAME Analysis only need the native .sview file to visually examine the model and results. Combined with S-FRAME design reports, S-VIEW presents a complete picture of your model.

11.0 Additional Resources

So now that you have built, analyzed, and designed your first structure, where do you go from here?

To better understand S-FRAME Analysis' capabilities and functionality, we recommend new users follow through a series of video tutorials.



Additional Online Resources.

In Addition, you can find much more worked examples within the S-FRAME Help system and the detailed manuals supplied in electronic (.PDF) format.

To help you progress from an S-FRAME "novice" to an S-FRAME "expert", we also offer short technical webinars on specific topics as well as in-depth online training classes. View the S-FRAME Analysis Product Home Page to find scheduled training classes and webinars or visit our Support page. Both webinar material and training classes qualify for Profession Development Hours (PDH).