Oasys

Oasys GSA 10.1

Getting Started

Oasys GSA

8 Fitzroy Street
London
W1T 4BJ
Telephone: +44 (0) 20 7755 4515

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559

email: oasys@arup.com
website: oasys-software.com

Contents

Introduction

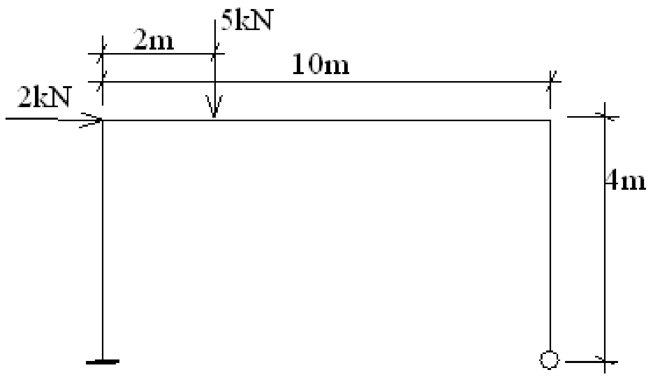
GSA allows engineers to create analytical and design models of a structure, then apply loads and actions to the model, analyse the behaviour, and view the analysis results as tabular data, charts, diagrams, or contour plots on the structure.

The model exists in two layers: the analysis layer holds a simplified model of the structure and the design layer represents the physical structure. Some items, such as nodes, may be mapped between the layers; others may exist only in one layer. The layers can be coordinated with one another using the coordination tools. You are unable to change the analysis data while results exist that depend on that data.

The model can either be imported from a 3D modelling program or created within GSA into the design layer, the analysis layer, or both. Wizards assist the initial model creation and subsequent editing. Once a model exists in GSA it can be edited in a graphical window or the base data can be edited directly via tabular views. The data required depends on the types of analyses being run.

Starting a model

This brief tutorial shows how to create a simple portal frame.



Setting up the model

1. Open GSA and select New Model at the top left of the splash screen). If that has been disabled, select New from the File menu, or click **Ctrl+N**

2. Complete the Titles fields as appropriate for your projects and click [**Next**]

3. Set up the [**Units**] as appropriate then click [**Next**]

4. The next screen allows you to specify design codes, material grades, and sections:

Adding codes, grades, and sections

1. Select a steel design code

2. Select a concrete design code

3. Select a standard steel grade

4. Select a standard concrete grade

5. Check that *Steel catalogue* is selected for the *Sections* and click *Add…*

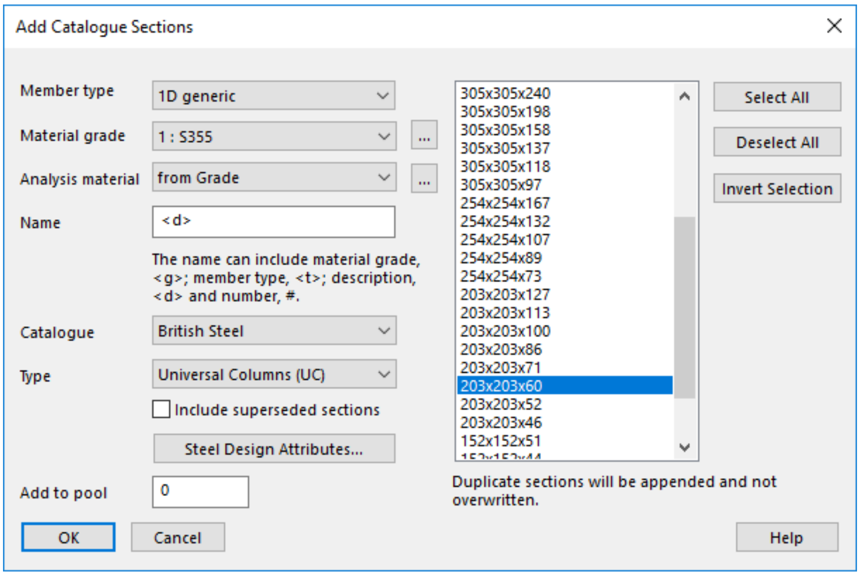
    a. The Add Catalogue Sections dialog appears. Your defined Material Grade will be selected automatically, and the Analysis material will be set to “From Grade”.

    b. Set the *Name* to “<d>” (if it is not already) to use the section description as the section name in your model.

    c. Select *British Steel* as the catalogue and *Universal Columns (UC)* as the type.

    d. Select *203x203x60* as the section to add. Note: hold the Ctrl key while selecting if you want to add multiple sections from the catalogue.

    e. Click *OK* to add the section.

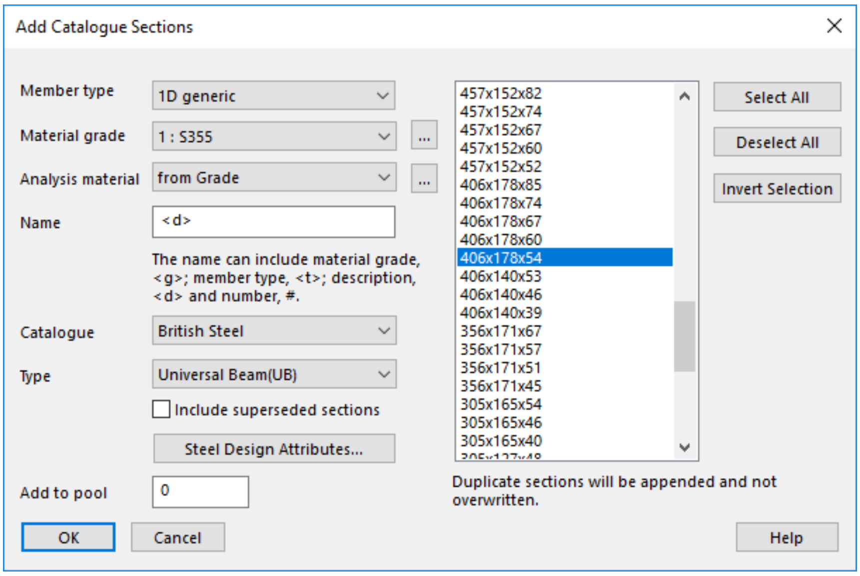


6. Click *Add…* again

a. Set the Name to “<d>” to use the section description as the section name in your model.

b. Select *British Steel* as the catalogue and *Universal Beam (UB)* as the type.

c. Select *406x178x54* as the section to add.



d. Click OK to add the section.

7. Click *Finish* to exit the wizard and start work on the new model

8. Save the model by clicking on the and give it a suitable name, changing the folder location as appropriate.

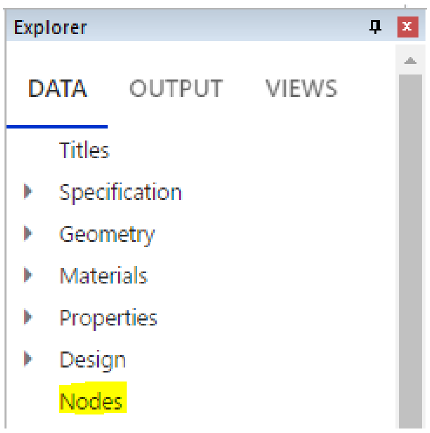
Editing the model

This assumes that you have added materials and catalogue sections to your new model using the options in the New Model wizard, as described in Adding codes, grades, and sections. This section describes how to add nodes to your model and create 1D elements by:

- extruding them from the nodes
- sketching them in the Graphics window

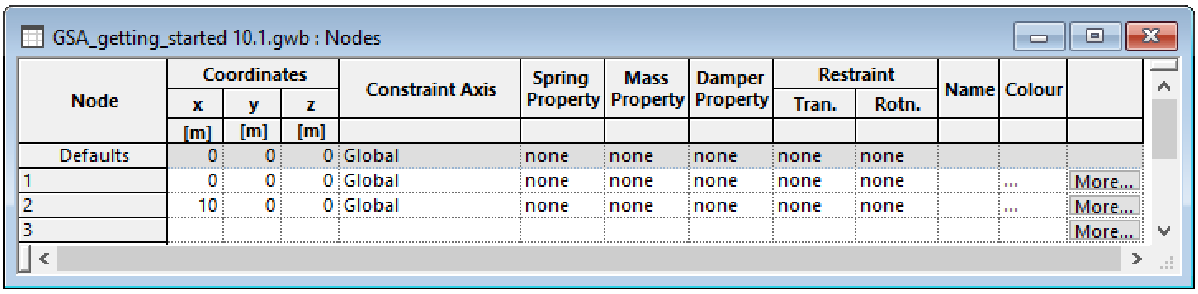
Add the support nodes

1. Open the Nodes data table by clicking *Nodes* in the *Data Explorer*.



2. Row 1: click in the first cell and press enter [⏎] to copy all the default values in the line above to get the first node at 0,0,0.

3. Row 2: Type 10 as the x coordinate followed by [⏎] to copy all subsequent values in the line above.


GSA_getting_started 10.1.gwb : Nodes

Node	Coordinates x [m]	y [m]	z [m]	Constraint Axis	Spring Property	Mass Property	Damper Property	Restraint Tran.	Restraint Rotn.	Name	Colour	
Defaults	0	0	0	Global	none	none	none	none	none			
1	0	0	0	Global	none	none	none	none	none		...	More...
2	10	0	0	Global	none	none	none	none	none		...	More...
3												More...

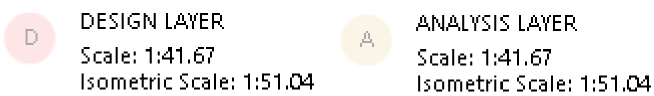
a. Tip: if you find that the text is too small in the table view, you can enlarge it by holding down the **[Ctrl]** keyboard button and rolling the mouse wheel

b. You can also change the text size for all the tables by clicking on the A font button on the Data Options toolbar

4. Click the Plan button (or press **P**) to change to a plan view in the graphics window. The nodes will appear as red dots.

Setting the type of column you are going to create

Note that If you are in the design layer then this will create Members if you are on the Design Layer (this is the recommended place to start) or Elements if you are on the Analysis Layer. You can tell from the legend in the top-right corner of the graphical window

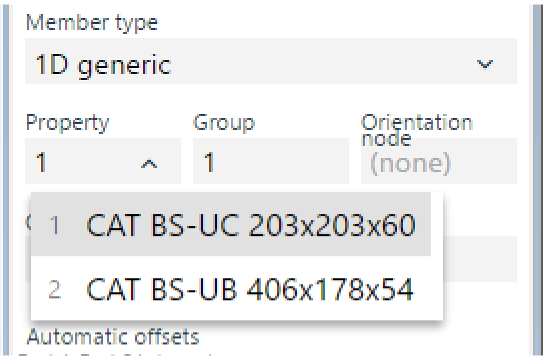


1. If you are on the Analysis layer, switch to the Design layer using the screen right click menu option.

2. Click the Add Entities button to open the Member Properties pane on the right.

3. Click the Property [**v**] to show the section of the item you are about to create.

4. It has a Property set to “1”. This means that the columns that you draw will use the first section defined in the section library.



Creating the columns by extrusion

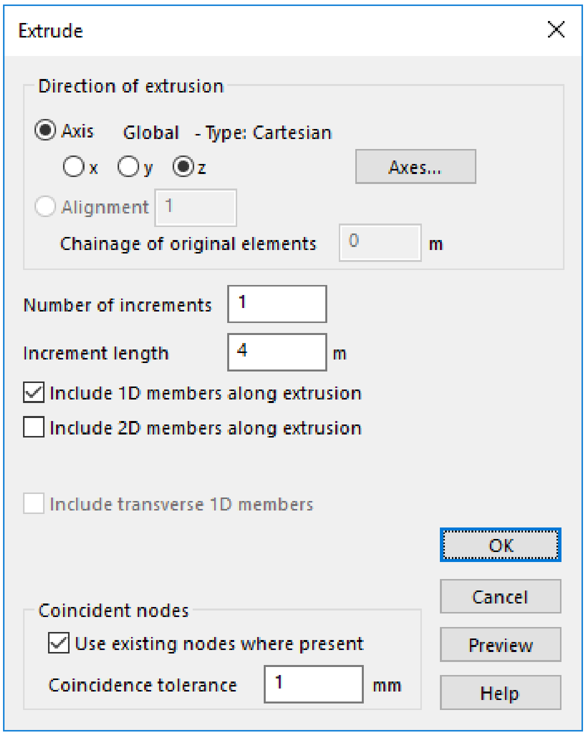
1. Change to the Graphics window by clicking the Isometric view button or the Perspective view button . Click the Nodes select button (or press **N**); you are now in node selection mode. Drag or click to select the two nodes you have created. Magenta dots will appear on the selected nodes.

2. Select "Extrude Selection" from the Sculpt menu to open the Extrude dialog.

3. Check that the extrusion axis is set to z.

4. Set the number of increments to 1 and the Increment length to 4.

5. Select "*Include 1D members along extrusion*".



6. Click [**Preview**] to see what elements will be created.

7. Click [**OK**] when you are satisfied. The columns are shown in the Graphics window.

Inspecting elements in the Graphics window

1. Click to resize the view and then inspect your columns by rotating the view: click and drag the right mouse button.

2. Click the Section Display button to give a 3D view of your columns.

3. Click and select a column to get a list of its properties displayed in the Messages pane at the bottom of the screen and in the Properties pane on the right-hand side.

Creating the beam by sculpting

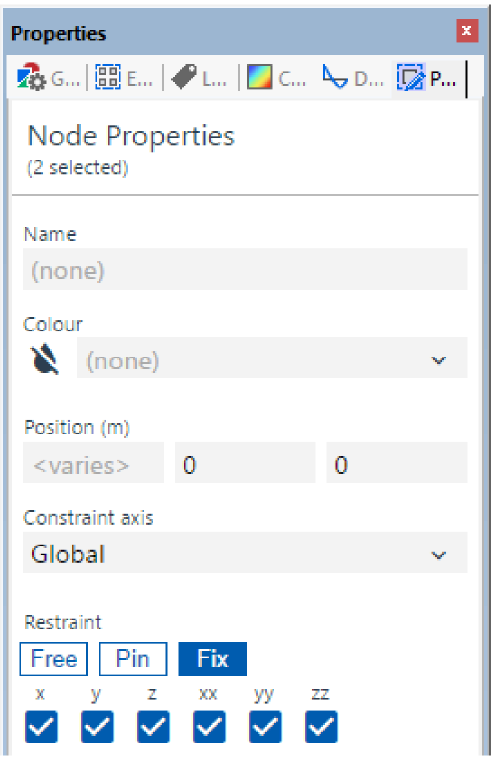
1. Select the Add Entities button.

2. Go back to the Member Properties pane. Change the property value to 2 to use the beam section.

3. Click on the node at the top of one of the columns. A line represents the beam that you are about to draw appears.

4. Click on the node at the top of the other column to complete the beam.

Setting restraints.



1. Click the Node Select button. Select the nodes at the bottom of the columns. This will display the Node Properties pane

2. Go to the Restraint section and select [**Fix**] to fully constrain the nodes.

3. Click the Label restraints button to show the restraints in the Graphics window.

Apply the node and beam loads

This section explains how to create the analysis elements and apply loads to the structure. It also shows you how to:

1. Mesh the model

2. Select items in the Graphics window and use the selection in a data table.

3. Change the Units using the option on the Status bar.

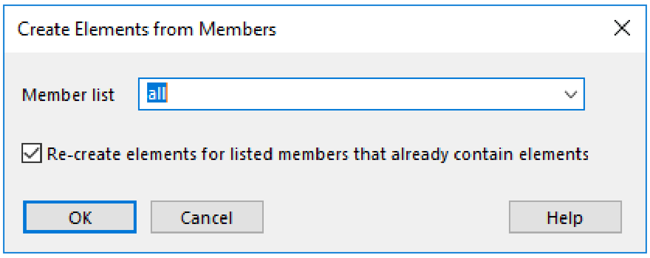
4. Find which end is which on a beam and apply a load at a specified distance from the end.

5. Display loads

Mesh the model

If you started in the Design layer, then create the Elements on the Analysis layer:

1. Use the menu command *Model* > *Coordination Tools* > *Create Elements from Members*, set the member list to *all*, and press [**OK**]



2. Switch to the Analysis layer by one of

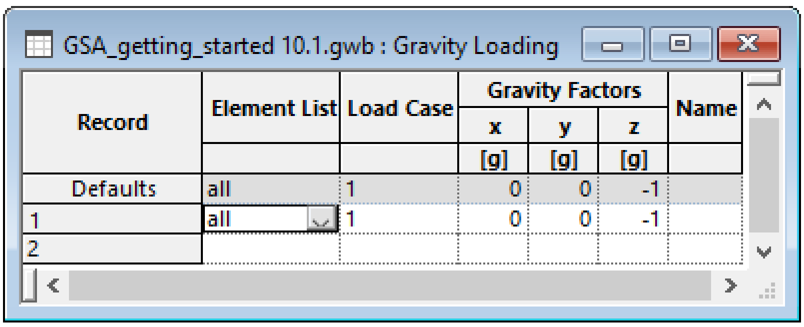
    a. Right clicking on the graphical window and selecting [**Switch Layer**]

    b. Use the keyboard shortcut **Ctrl+Alt+D**

Adding self-weight loads

1. In the *Data Explorer* open Loading, then Gravity Loading

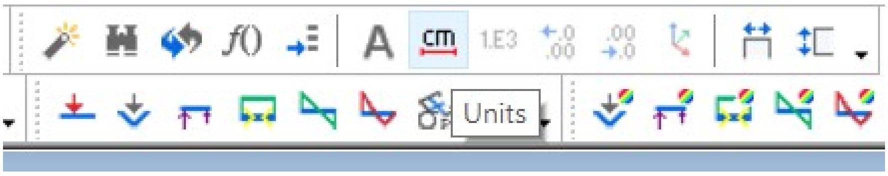
2. Click in the first cell of Row 1 and press **Enter** once to copy the default row. This will calculate the self-weight based on the material density and element volume.



Creating a lateral load on a node

1. Select the node at {0,0,4} (the node coordinates and other settings will appear in the Properties pane) and press **Ctrl+C** to copy it
2. Open the *Node Loads* data table from the *Data* pane (look in *Loading* > *Nodal loading*).
3. Select the Nodes cell on row 1 and press **Ctrl+V** to paste the selected node into the cell.
4. Set the load case to "2", change the Direction of the load to x, and the Value to 2 kN.

   If your units are not in kN, you can quickly change them by clicking the Units button on the Data Options toolbar.



5. Move to the next row to enter the data, then close the *Nodal Loads* data table.

Creating a vertical load on a beam

1. Change to the Element Select tool ; select the cross-beam and **Ctrl+C** to copy it.
2. Open the *Beam Loads* data table from the Data pane (look in Loading).
3. Press **Ctrl+V** to paste the selected beam into the row 1 Beams list cell.
4. Set the Load Case to **2**.
5. Change the Type of the load to Point.
6. Change the Position 1 of the load to 2 m. This distance is measured in the +ve direction from End 1.
7. Change the Value of the load to −5 kN in the Z direction.
8. Move to the next row to enter the data and close the Beam Loads data table.

Displaying loads in the Graphics window

1. Check you have the Graphics window selected.

2. Change the load case to L2

3. Click the Loads diagram button to display the loads. The node and the beam loads will be displayed as purple arrows.

Displaying element axes and flipping elements in the Graphics window

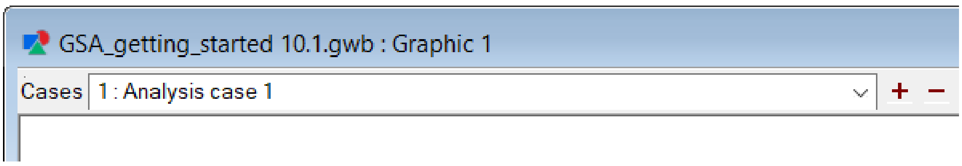
If you have the point load appearing towards the wrong end of the beam, you can either move it in the Beam load data table, or you can flip the beam, so its ends are the position you expect.

1. Click the Label element x-axis button to show the directions of the beams and columns. Small red cones are drawn on the elements.
2. Change to element select mode and select the beam.
3. Select *Flip Elements* in the *Sculpt* menu. The x-axis arrow changes direction and the point load moves to the other end of the beam.

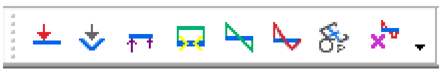
Running a simple static analysis

You have created a load case with a node load and a point load. You can run a static analysis immediately and view your results as a table or in the graphics window. This section shows you how to run the analysis and display your moment results.

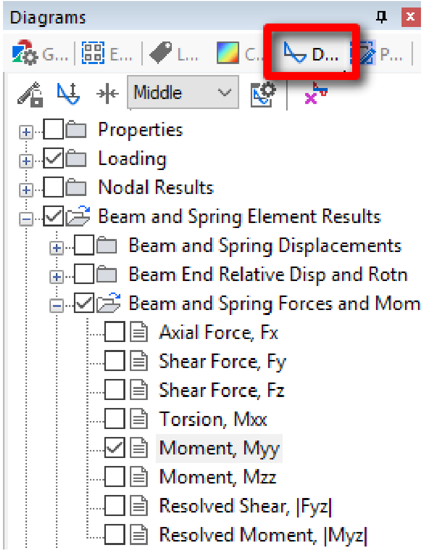
1. Press the Analyse All button Σ. A report window opens giving information about the analysis that you have just run. This information is also shown in the Reports pane.
2. Change to the Graphics window and set the case to Analysis case to give access to the results.



3. Click to show the deformed shape.
4. Click to show the Myy bending moments
   You can select other common results from the Diagram toolbar

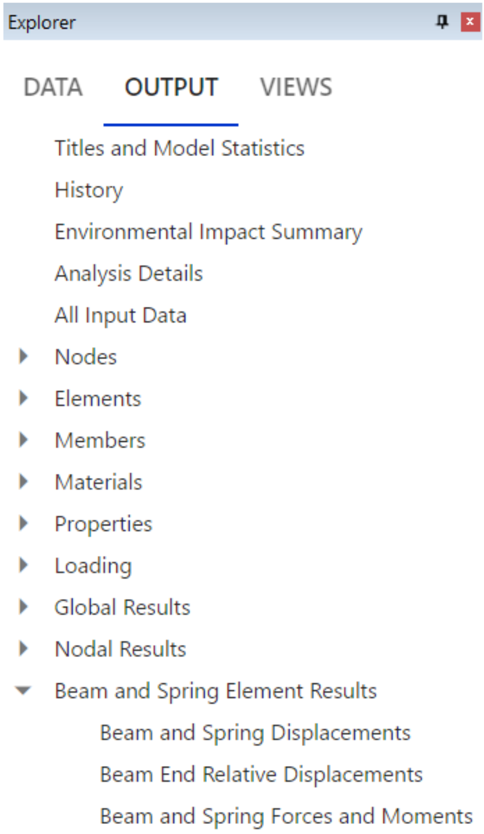


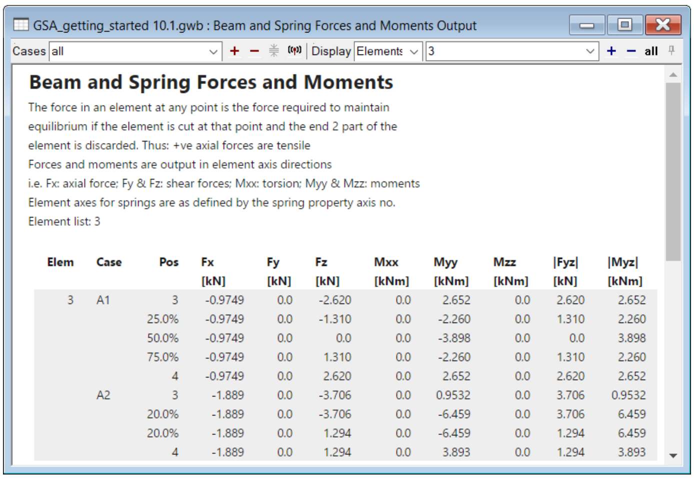
5. Go to the Diagrams pane to access all the available options.



Displaying results as a data table

1. Select the Output Explorer tab
2. Open the Beam and Spring Elements Results folder.
3. Select Beam and Spring Forces and Moments to see a table of results.
4. Select Elements in the drop-down list at the top of the window and type 3 to show the forces on the cross-beam.





GSA_getting_started 10.1.gwb : Beam and Spring Forces and Moments Output

Cases all | Display Elements | 3 | all

Beam and Spring Forces and Moments

The force in an element at any point is the force required to maintain equilibrium if the element is cut at that point and the end 2 part of the element is discarded. Thus: +ve axial forces are tensile
Forces and moments are output in element axis directions
i.e. Fx: axial force; Fy & Fz: shear forces; Mxx: torsion; Myy & Mzz: moments
Element axes for springs are as defined by the spring property axis no.
Element list: 3

Elem	Case	Pos	Fx [kN]	Fy [kN]	Fz [kN]	Mxx [kNm]	Myy [kNm]	Mzz [kNm]	|Fyz| [kN]	|Myz| [kNm]
3	A1	3	-0.9749	0.0	-2.620	0.0	2.652	0.0	2.620	2.652
		25.0%	-0.9749	0.0	-1.310	0.0	-2.260	0.0	1.310	2.260
		50.0%	-0.9749	0.0	0.0	0.0	-3.898	0.0	0.0	3.898
		75.0%	-0.9749	0.0	1.310	0.0	-2.260	0.0	1.310	2.260
		4	-0.9749	0.0	2.620	0.0	2.652	0.0	2.620	2.652
	A2	3	-1.889	0.0	-3.706	0.0	0.9532	0.0	3.706	0.9532
		20.0%	-1.889	0.0	-3.706	0.0	-6.459	0.0	3.706	6.459
		20.0%	-1.889	0.0	1.294	0.0	-6.459	0.0	1.294	6.459
		4	-1.889	0.0	1.294	0.0	3.893	0.0	1.294	3.893

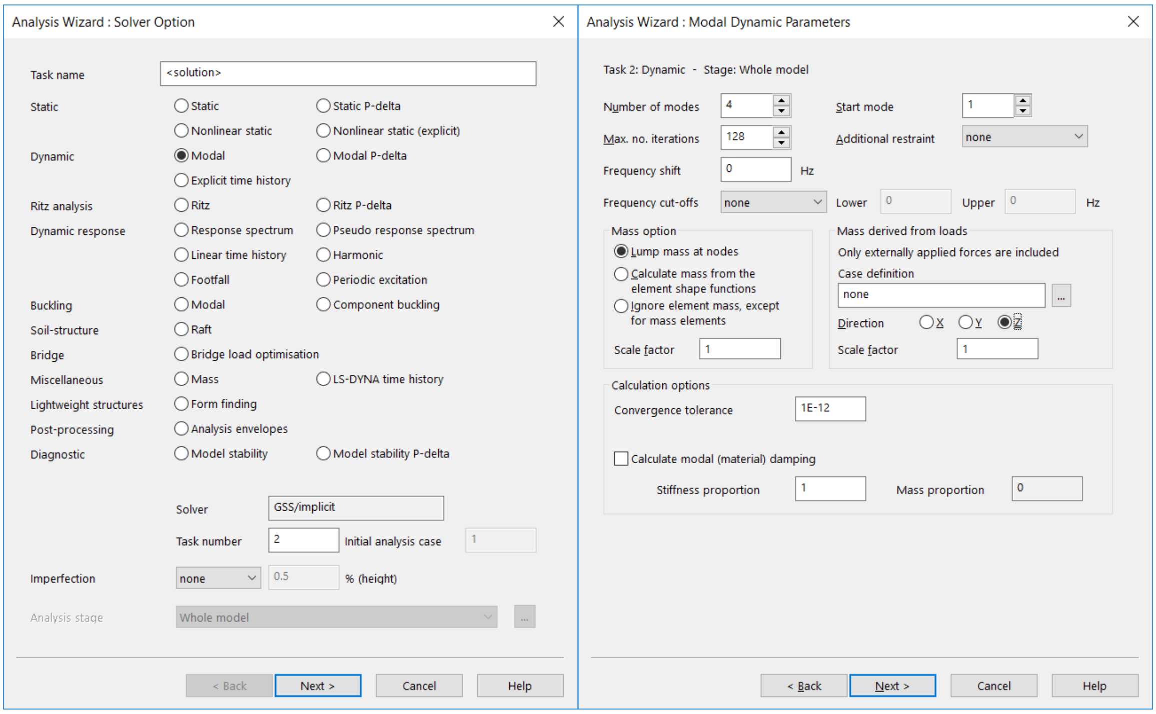
Running a more complex analysis

You have created a model and run a simple static analysis. To run a more complex analysis you need to set up the specific analysis parameters. This section shows you how to run a modal dynamic analysis and display your mode shapes.

1. In the Data pane open the Tasks and Cases item and click on the Analysis Tasks.

2. Either from the Analysis menu, the  button, or from the right-click menu select New Analysis Task. This will open the Analysis Wizard.

3. Select the Modal option to define the parameters that control the modal analysis, and then Next.



4. Set the number of modes to 4 but leave the other parameters unchanged, and then Next. You have now set up the modal analysis, so you can Finish, and GSA will now run the dynamic analysis.

5. Change to the Graphics window, select one of the analysis cases labelled “mode” and click  to show the deformed shape. Click on  to animate the mode shape.
Note that you might not see a deformed shape if you are looking square onto the structure and the mode direction is away from you. Try rotating the view or change to an Isometric or Perspective view direction  | .

Creating slabs

This section shows you how to create a concrete slab, by creating a 2D member and auto-meshing it. It covers:

- Creating concrete sections
- Changing your columns to concrete sections
- Copying entities
- Defining and checking 2D properties
- Defining a simple slab and meshing it manually
- Defining a slab with a void and using auto-meshing
- Assigning the concrete properties to the mesh.

Defining your concrete sections

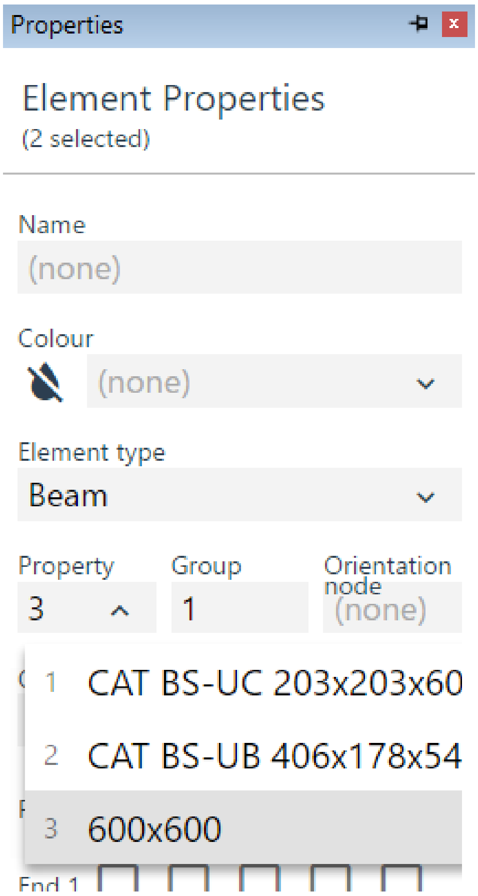
You can create different material properties for your analysis and design layers. If you have code materials for design, you can derive the analysis properties from them.

When defining the sections that you use for columns and beams, you can set which properties you use. This section shows how to create a 600 * 600 rectangular concrete section

1. Open the Properties > Section Library from the Data pane.
2. Click in the first cell of the next empty row in the table and click the wizard button to open the Section wizard (You can also double-click the next empty row, or press **Ctrl+W**)
3. In the Name field type “600x600”
4. Change the material to concrete.
5. In the Grade field select either the grade you defined earlier or click “Add code material” and create a new appropriate grade.
6. Leave the analysis field as “from Grade”.
7. Click [**Profile...**] and select Rectangular as the definition method.
8. Click [**Next**] and define the profile to be 600 x 600 mm.
9. Click [**Next**] and then click [**Finish**].

10. The profile definition “STD R 600 600” appears in the Profile field and a picture of the defined section appears on the right. It is possible to edit the definition directly.
    Click [**OK**] to complete your definition.

Converting existing columns to concrete



Before you can make changes to your analysis model, you must delete any existing analysis results. This shows how to delete the beam and change your existing column sections to the new concrete ones.

1. Delete your analysis results by clicking the Delete Analyses button.
2. In the graphical view change to the Design layer (right-click menu or **Ctrl+Alt+D**)
    a. If the structure appears grey, switch off the deformed shape →
3. Click to show a Y elevation in the Graphic window.
4. Change to Element select mode [**E**].
5. Click on the beam to select it and press [**delete**].
6. Click and drag around the columns to select them.
7. The Element properties pane should immediately display. If not select Modify in the right-click menu. Select the new Concrete section in the Property drop-down list.

Duplicating the concrete columns

1. With the columns selected, click the Move/Copy button to duplicate them. The Move or Copy Elements dialog opens.
2. Check that the option is set to copy.
3. Set the number of copies to 2.
4. Set the amount to shift to be 8 m in the y direction.
5. Click [**OK**]. You will be returned to the graphics window. There should now be six columns visible in the window. Rotate the view to inspect the columns by right-clicking and dragging on the view.

6. Click ⌶ to give a 3D view of your columns.

Creating a 2D Member in the Design layer

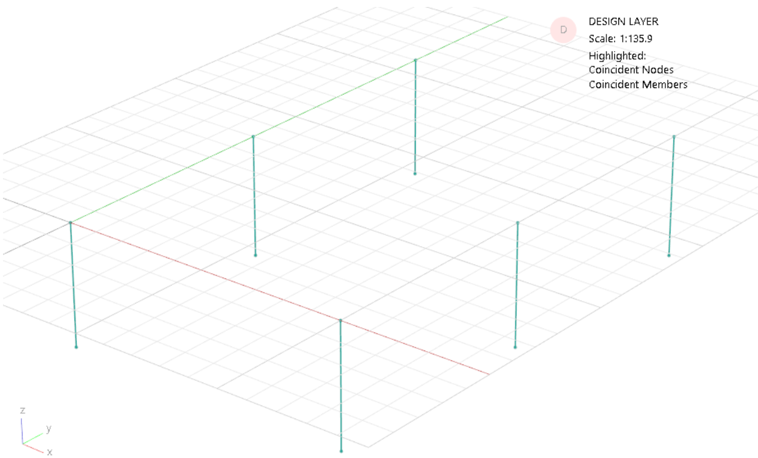
This section shows you how to create a slab with a void and mesh it using a 2D member. This allows you to create and mesh complex slabs and walls. It covers:

- Creating a grid at a specified elevation
- Defining two 2D members on the current grid
- Specifying that one of the members is a void
- Auto-meshing the slab.

This section assumes that you have six concrete columns. If working through this guide, you will need to delete any existing analysis results before proceeding.

Create the grid to draw your slab on

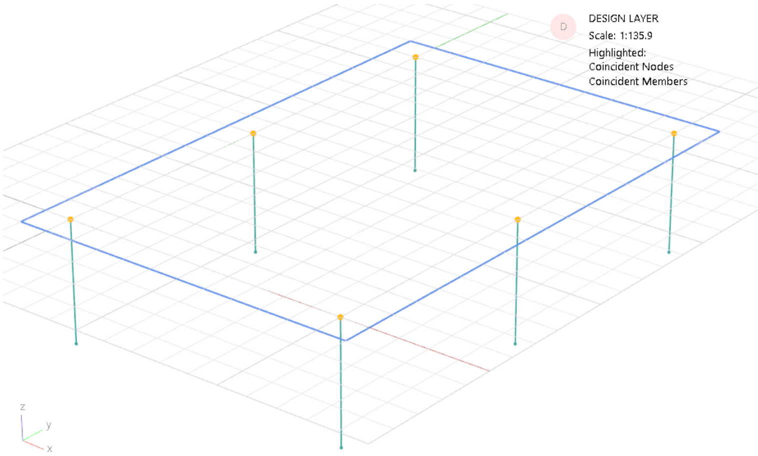
1. Change to the Graphics window.
2. Ensure that you are on the Design layer (press **Ctrl+Alt+Delete** to swap if necessary). This is where you can define a less abstract model based on construction codes.
   If you created the columns on the Analysis layer, then they will be shown as a series of dashed lines connecting nodes together.
3. If necessary, click the Draw Grid button ⊞ to draw the current grid.
4. Right-click one of the nodes at the top of a column and select the option *Set Current Grid to This* from the context menu. You will be asked if you wish to create a new grid plane.
5. Click [**Yes**]. A new grid plane will be created at the top of the columns.



Draw a 2D member to define the perimeter of your slab

1. You can now define the shape of your slab by drawing on the plane. Select the *Add Entities* tool and set the Member Properties to 2D Generic.

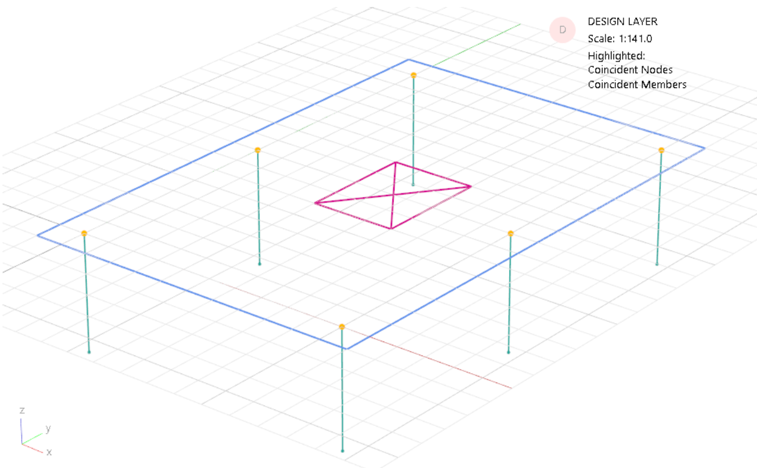
2. Click on the first corner of the outside of your slab, followed by the other corners in either the clockwise or anticlockwise direction. Finish the member by either clicking back on the first point or pressing Return on your keyboard.



Define a void cutter member to create a hole in your slab

1. Create a member to define the void in the same way that you created the perimeter, but change the Member Type to *2D Void Cutter*

2. If you forgot to change the member property type first, select the member and change the Member Type in the properties pane.



Auto-meshing the slab

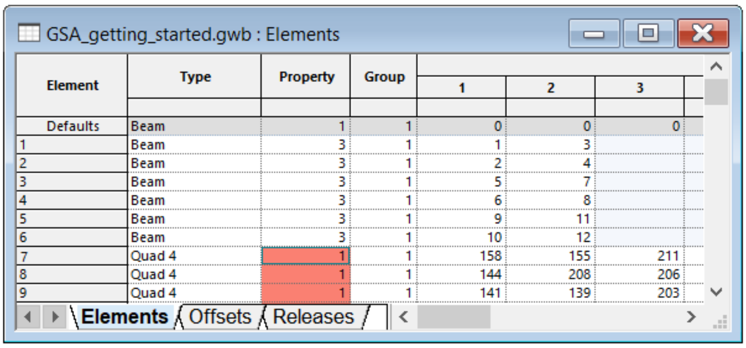
1. From the Model menu, select *Coordination Tools* and then select *Create Elements from Members*.

2. The Members will be meshed into multiple Quads and Triangles.

Checking the properties of your slab

This shows you how to check the entries in an Element data table.

1. Open the *Elements* data table to check your Quad definition.


GSA_getting_started.gwb : Elements

Element	Type	Property	Group	1	2	3
Defaults	Beam	1	1	0	0	0
1	Beam	3	1	1	3	
2	Beam	3	1	2	4	
3	Beam	3	1	5	7	
4	Beam	3	1	6	8	
5	Beam	3	1	9	11	
6	Beam	3	1	10	12	
7	Quad 4	1	1	158	155	211
8	Quad 4	1	1	144	208	206
9	Quad 4	1	1	141	139	203

2. The Property cell is highlighted in red. This shows that you do not have that 2D property defined.

Setting the properties of your slab

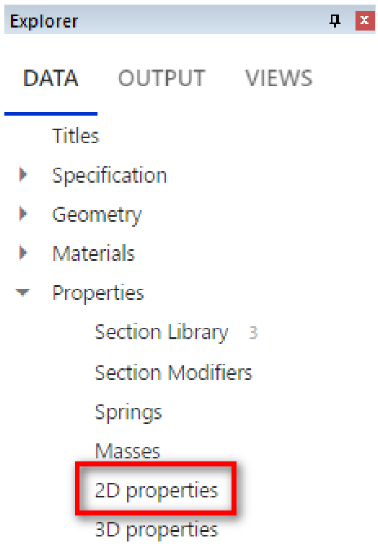
This describes how to set the slab properties.

1. Open the 2D Elements properties data table and double-click the first row to open the 2D Property wizard.

2. Confirm that the type is Shell, the Material is concrete, and the Grade and Analysis values are sensible.

3. Set the thickness of your slab to 250 mm. The slab properties define the rebar used; ignore them for now. Click [**Next**] and then click [**Finish**].

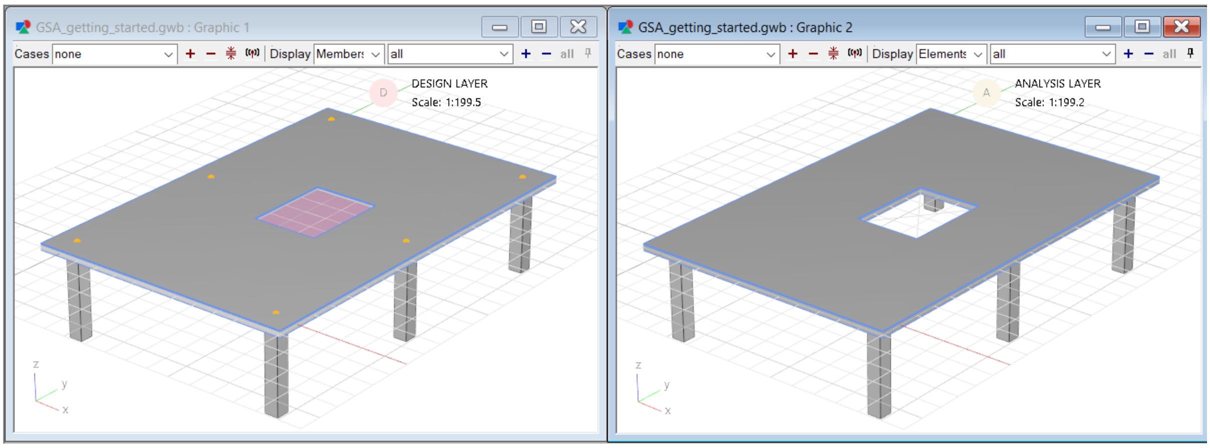
4. Close the 2D Properties data table. Confirm that the red highlighting has gone from the Elements table.



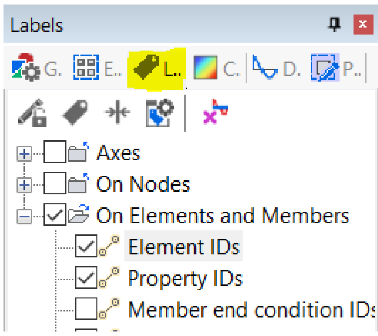
Check your element data in the Graphics window

1. Change to the graphics window to confirm that your slab is now thicker. (If the section display is not switched on, click 𝙸 to give a 3D view of your model.)

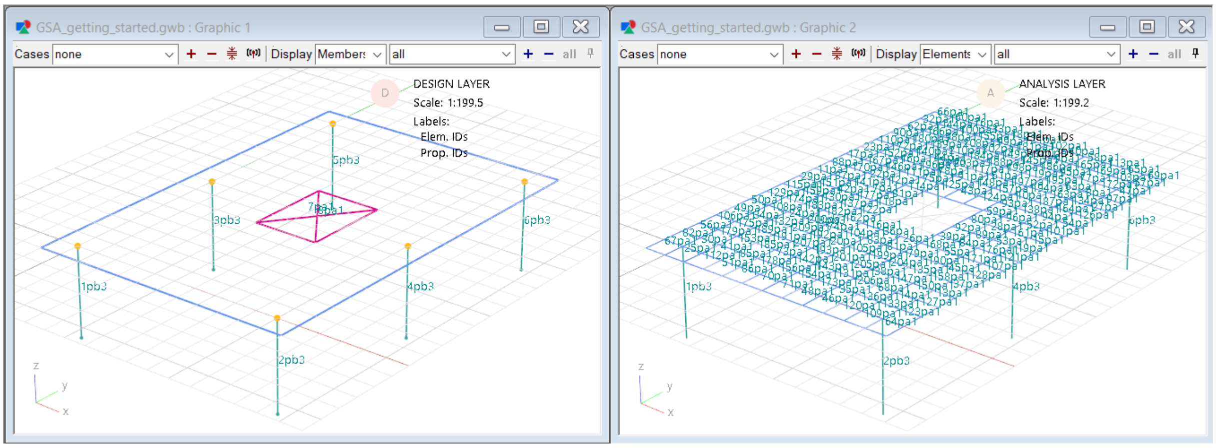
2. Click 𝙸 to change to a line view of your model.



3. Go to the Labels pane and select the Element IDs.



4. The Graphics view changes to add the labels. Add the *Property IDs* to show which associated property has been assigned to each element. The slab will have a property of PA1, showing that it has 2D (area) property 1, though you may need to zoom in to separate the text from the adjacent objects.



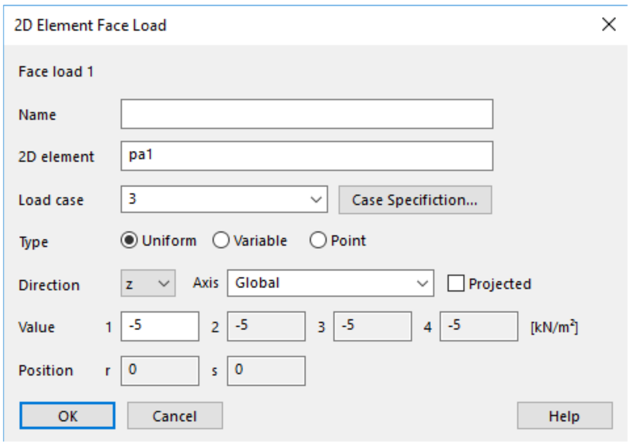
5. Select *Analysis material IDs*. If they are labelled as “m0” this shows that the analysis material is derived from the material grade. Check *Material grade IDs* as well to confirm your choices. Concrete material grades are prefixed with “c”.

6. Select *Element axes* to confirm that the z-axis (blue line) of your slab is pointing upwards. If not, you can use the *Flip Elements* command in the *Sculpt* menu to correct it.

Applying a load to the slab

You can apply loads to individual elements, to elements with certain properties, or you can apply a grid load across all the elements on the plane. This section describes how to apply loads using property references.

Add loads to your slab



This shows you how to apply a simple load across the whole slab

1. Open the *Loading* section of the *Data Explorer*, then *2D element loading*, and finally *Face Loads*.

2. Start the *2D Element Face Load* wizard. In the 2D element list type **PA1**

3. Set the load case to **3**

4. Add a load of **−5 kN/m²** in the **z global** direction and click [**OK**].

5. Change to the graphic window, ensure that you are on the Analysis layer, set the Case to L3 and click  to display the load diagrams.

6. The face load arrow will be displayed at the centre of each 2D element.

Displaying the slab displacement as a contour

1. In the *Data Explorer* right-click on *Tasks and Cases* > *Analysis Tasks* and select *Delete*. This step is not needed if you have not manually created an analysis task. The alternative is to open the Analysis Tasks window, double click on the Static analysis task, and use the wizard to add in the additional load cases.

2. Press the Analyse All button Σ to analyse the load. This will recreate the static analysis task and add in all the new load cases.

3. Change to the Graphics window and press **P** on your keyboard to change to a plan view.

4. Go to the Contour palette pane and scroll to find the 2D element results.

5. Display the element displacement results.

Meaning of symbols in the Graphics window

Symbol	Definition
Magenta circle	Selected item
Blue dot	Node
Red dot	Unused node
Yellow dot	Node where members intersect
Red square	Nearest node
Red arrow point	Element axis
Grey line(s)	Element or member present on the other layer but not on the current one Undeformed geometry (if Deformed Image is active)