Oasys

Oasys GSA

Getting Started

Oasys

YOUR IDEAS BROUGHT TO LIFE

13 Fitzroy Street
London
W1T 4BQ
Telephone: +44 (0) 20 7755 3302
Facsimile: +44 (0) 20 7755 3720

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559
Facsimile: +44 (0) 191 238 7555

e-mail: oasys@arup.com
Website: oasys-software.com

Oasys GSA

Contents

Introduction

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

The analysis model consists of a simplified model of the structure. The design model represents the actual building,

The model(s) exists in two layers which share nodes. Other items in the layers may be mapped between the layers, or may exist wholly in one or another layer. The layers can be co-ordinated with one another (using the co-ordination tools) and there is some ability to specify mapping between 1D elements and members. The analysis model is also co-ordinated with the results from that model. 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 analysis layer, the design layer or both. Model creation is assisted by a wizard. 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.

Creating a simple model

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

Columns: UC 203x203x60

Beams: UB 406x 178x54
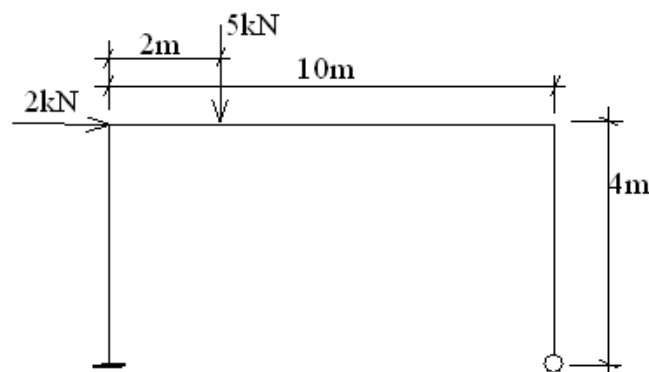
Setting up the model

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] and the [Design codes] as appropriate then click [Next].

The next screen allows you to:

1.   Add materials appropriate to your selected code. (This data can be used for both analysis and design of your model.)
2.   Add any catalogue sections that you wish to use in your model
3.   Generate some simple structures automatically.

This tutorial shows you how to add material grades and catalogue sections. You can then choose whether to generate the portal frame automatically or enter the analysis structure yourself.

Adding grades and catalogue sections

1. Select a standard steel grade. You can now add the sections required.

2. Check that [Steel catalogue] is selected for the Column sections and click [Add…]

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

3. Set the Name to "<d>" to use the section description as the section name in your model.
4. Select "British Steel" as the catalogue and "Universal Columns (UC)" as the type.
5. Select "203x203x60" as the section to add.
6. Click [OK] to add the section.
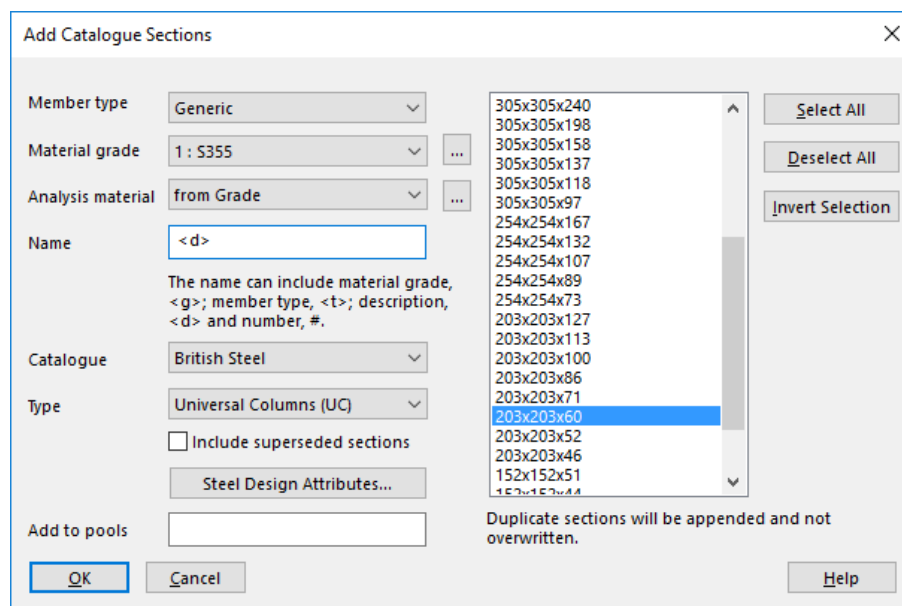
7. Check that [Steel catalogue] is selected for the Beam sections and click [Add…]

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

8. Set the Name to "<d>" to use the section description as the section name in your model.
9. Select "British Steel" as the catalogue and "Universal Beams (UB)" as the type.
10. Select "406x178x54" as the section to add.
11. Click [OK] to add the section.

If you wish to generate the model automatically click [Generate data for the selected structure type]. Otherwise, click [Finish] and go to the section 'Creating a model from scratch'.

Creating the model automatically

1. Clicking [Generate data for the selected structure type] opens a pane given you pictures of different structures. Select the portal frame on the top left and click [Next].
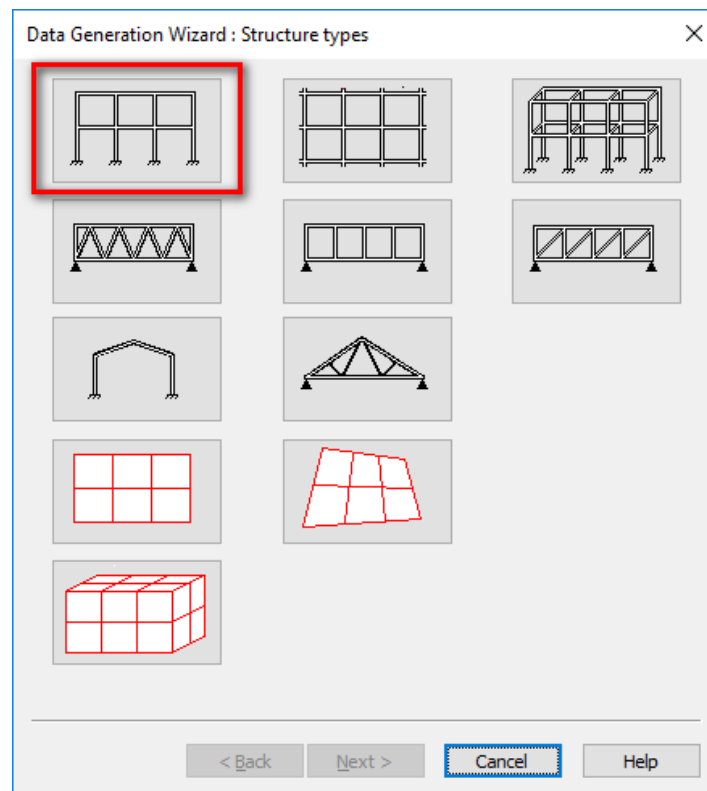
2. Confirm that the length units at the top are correct.
3. Set the Bay width to 10m and the Storey height to 4m.
4. Confirm that the Column and Beam sections assigned are correct.
5. Select "Include supports" to encastre the base nodes.
6. Select "Include self-weight" to create a simple gravity load.
7. Click [Next] then click [Finish].

A graphics window will open showing your portal frame.

Creating a model from scratch

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 grades and catalogue 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 ground nodes

1. Open the Nodes data table by double-clicking "Nodes" in the Data Explorer.
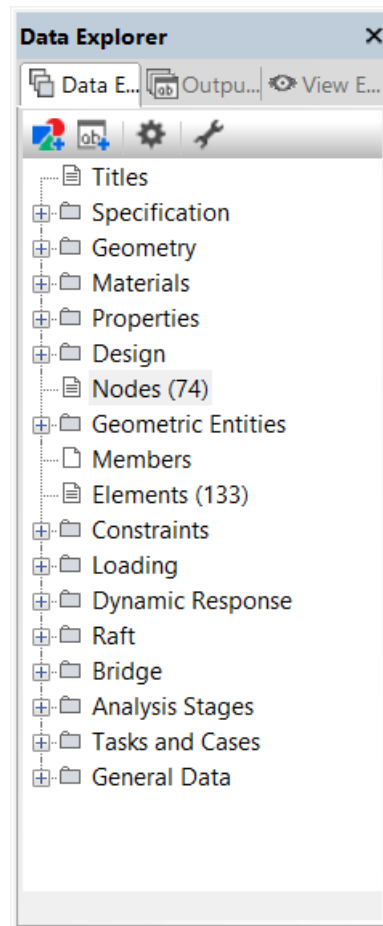
2. Type "==" (omitting the quotes) in the first cell and press [Tab] to copy all the default values in the line above to get the first node at 0,0,0.
3. Type 10 as the x coordinate and then type "==" in the next cell followed by [Tab] to copy all subsequent values in the line above.
4. Press the down arrow key or click in the next row to finish your entry.

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

## Setting the type of column you are going to create

1. Click the Defaults button ▣ to open the Data Defaults pane on the right.

2. Open the Elements section to show the properties of the item you are about to create.
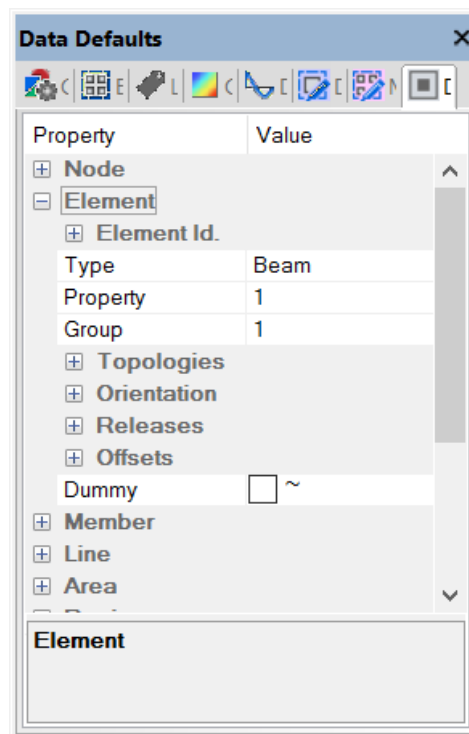

3. It has a Property set to "1". This means that the columns that you draw will use the first section defined in the section library.

4. Open the Sections Library to check which order your sections have been added to it. In the Data Explorer pane, select Properties and then double-click Section Library to open the section data.
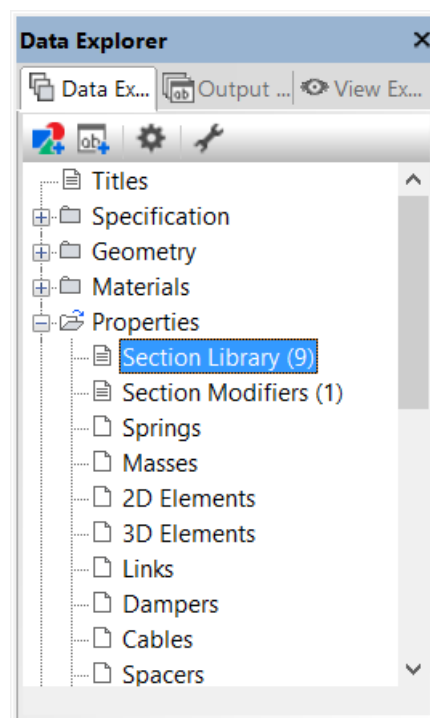
5. The Section Library data table opens. Confirm that the first entry is the UC 203x203x60 column. (If it is the second entry, you will need the change the Property option in the Data Defaults pane to 2.)

Creating the columns by extrusion

1. Change to the Graphics window by clicking 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 crosses 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 Beam elements along extrusion".
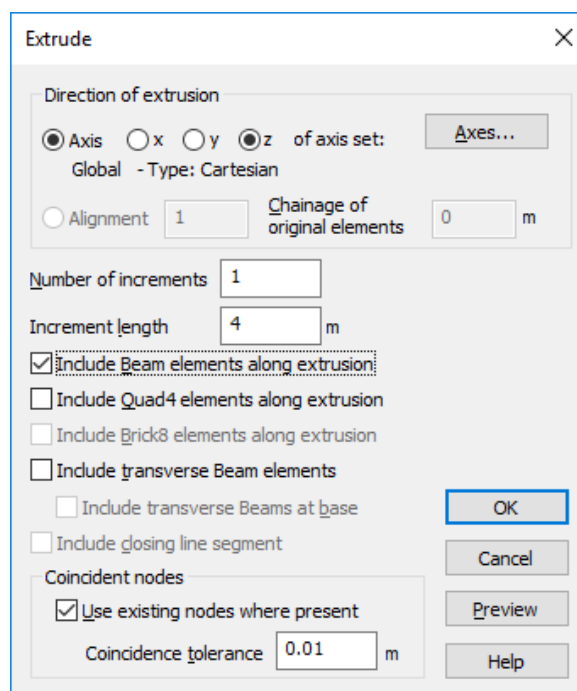
6. Click [Preview] to see what elements will be created.
7. Click [OK] when you are satisfied. The columns are shown in the Graphics window.

Inspecting elements in the Graphics window

1. Click the Rotate button ⟳ (or press R) and inspect your columns by dragging the view.
2. Click the Section Display button ⊥ to give a 3D view of your columns.
3. Click on a column to get a list of its properties displayed in the Message Pane at the bottom of the screen.
4. Click the Entity Definition tab at the top right of the screen to show the definition for the selected column.

Creating the beam by sculpting

1. Select the Add Elements button.

2. Go back to the Data Defaults pane. Change the section 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 and then select the Modify Entities pane ().

2. Go to the Constraints section and select the box next to Translation and Rotation to fully constrain the nodes.
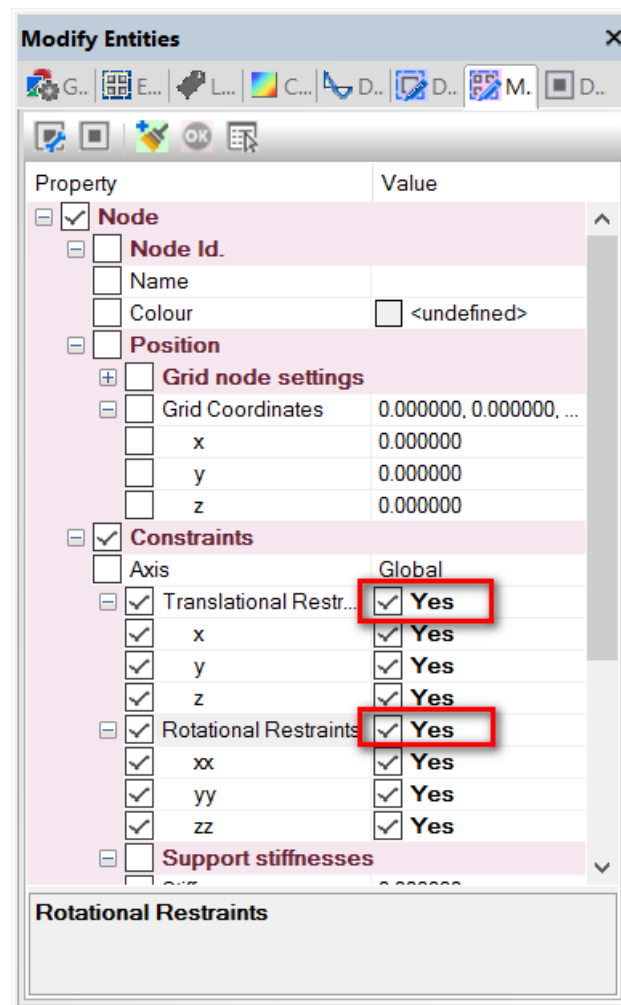
3. Click the OK button on the top bar to apply your changes
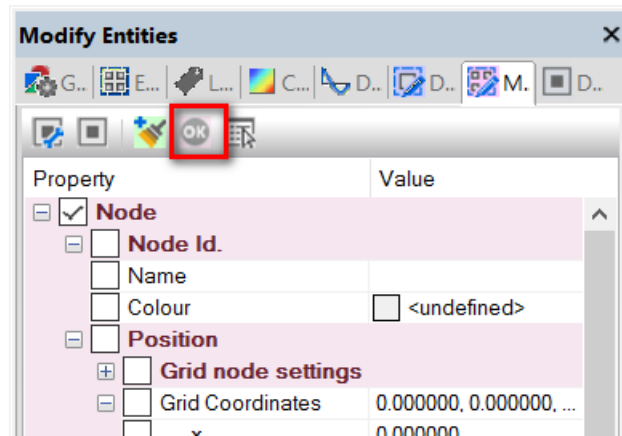
4. Click the Label restraints button ![icon] to show the restraints in the Graphics window.

Apply the node and beam loads

This explains how to apply the 2kN load to the top of the column and the 5kN load to the beam. It shows you how to:

1. Select items in the Graphics window and use the selection in a data table.
2. Change the Units using the option on the Status bar.
3. Find which end is which on a beam and apply a load at a specified distance from the end.
4. Display loads

Creating a lateral load on a node

1. Select the node at (0,0,4) and press [Ctrl]+C to copy it
2. Open the Node Loads data table from the Data Explorer (look in Loading).
3. Press [Ctrl]+V to paste the selected node into the Nodes cell.
4. Change the Direction of the load to x and the Value to 2kN. If your units are not in kN, you can quickly change them by clicking the unit display at the right hand end of the Status bar.
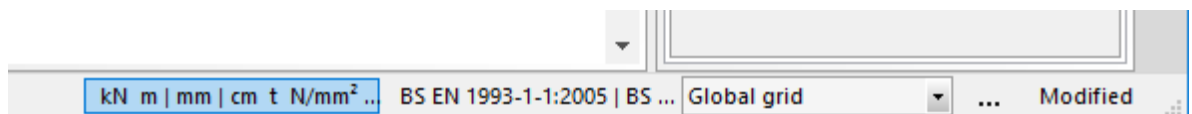

5. Move to the next row to enter the data and close the Node Loads data table.

Creating a vertical load on a beam

1. Change to the Element Select tool and select the cross-beam.
2. Open the Beam Loads data table from the Data Explorer (look in Loading).
3. Press [Ctrl]+V to paste the selected beam into the Beams cell.
4. Change the Type of the load to Point.
5. Change the Position of the load to 2m. This distance is measured in the +ve direction from End 1.
6. Change the Value of the load to -5kN in the Z direction.
7. 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. Click the Loads diagram button to display the loads. The node load will be displayed as a purple arrow and the beam load will be displayed as a brown arrow.

### 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 arrows 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 Report pane.
2. Change to the Graphics window and click to show the deformed shape.
3. Change to the Diagrams pane and select the Myy moment diagram
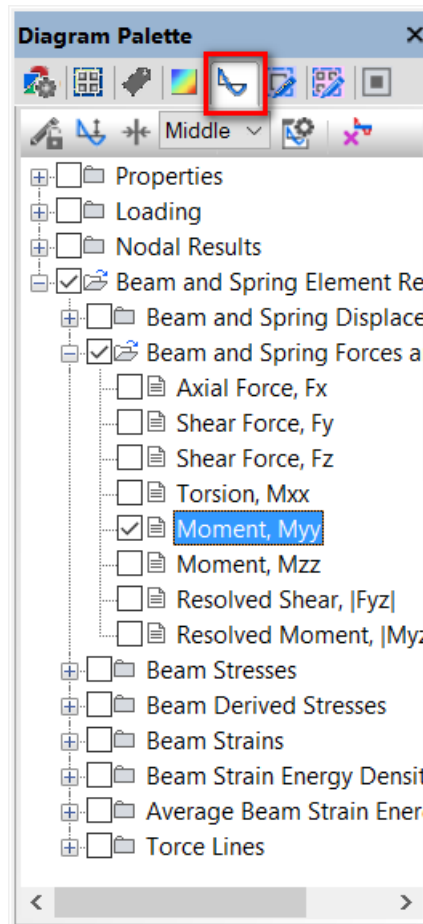
4. The bending moment will be drawn.
5. Select other diagrams that you wish to display (Note that shortcut buttons to common diagram views are available on the Diagram toolbar)

Displaying results as a data table

1. Select the Output Explorer tab on the left
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.

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 Explorer pane open the Tasks and Cases item and double-click on the Analysis Tasks.
2. Either form the Analysis menu 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.

Creating slabs

This section shows you how two ways to create a concrete slab. You can either create a single large 2d element and then split it to provide the equivalent of a simple mesh (which works well for simple regular shapes) or define a region and auto-mesh it (which works well for more complex shapes or shapes that include voids). 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 Section library from the Data Explorer.
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 [Control]+W.)
3. Change the material to concrete.
4. Select "Add code material" in the Grade field and choose an appropriate grade.
5. Leave the analysis field as "from Grade".
6. Click [Profile] and select Rectangular as the definition method.
7. Click [Next] and define the profile to be 600 x 600mm.
8. Click [Next] and then click [Finish].
9. 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.
10. Click [OK] to complete your definition.

Converting existing columns to concrete

Before you can make changes to your analysis model, you must delete any current analyses. . 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 𝚺x.
2. Click 🏠 to show a Y elevation in the Graphic window.
3. Change to Element select mode (E).
4. Click on the beam to select it and press [Del] (or right-click and select Delete Element from the context menu).
5. Drag from right to select across the columns to select them.
6. Select the Modify Entities pane and select the new Concrete section in the Property drop-down list.
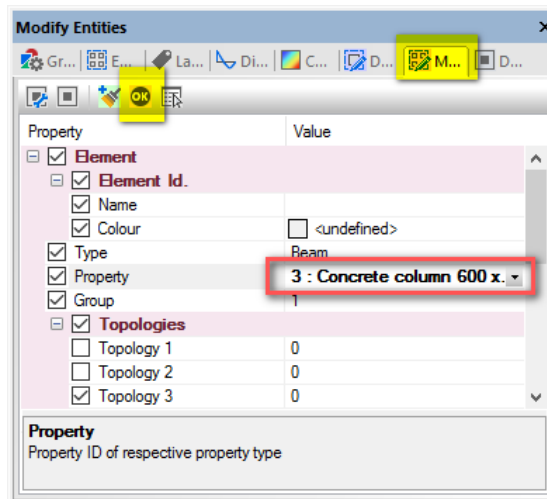
7. Click OK to apply your changes.

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 8m 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. Change to Rotate mode by clicking ⟳ or pressing R and inspect the columns.
6. Click ⊥ to give a 3D view of your columns.

Creating a simple slab using a Quad 4 element

This describes how to draw a single quad 4 element to form the slab at the top of the pillars.

1. Select the Add elements tool 🔲. The Data defaults pane becomes active. This defines what type of element you will add.
2. Go into the Data defaults pane and open the Element branch
3. Set the Type to Quad 4.
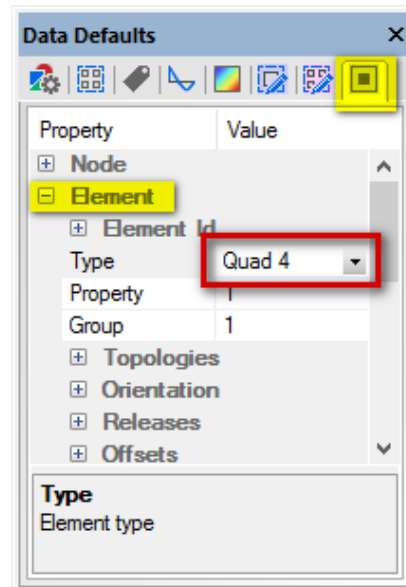
4. Click on the node at the top of one of the corner pillars. This marks the first corner of your Quad 4 element.
5. Click on the next three corner nodes in a clock-wise direction. When you have clicked all four nodes, the Quad 4 will appear in the window. The direction of node selection specifies the local axis direction of your slab.

Checking the properties of your slab

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

1. Open the Element data table to check your Quad 4 definition.

2. The Property cell is highlighted in red. This shows that you do not have a 2D property defined. (The blank line in the element table is where the deleted beam was stored. Elements in GSA keep their row numbering unless you change it.)

Setting the properties of your slab

This describes how to set the slab properties.

1. Open the 2D 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 250mm. The slab properties define the rebar used. Ignore them for now. Click [Next] and then click [Finish].
4. Close the 2D element 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 ⊥ to give a 3D view of your model.)
2. Click ⊥ to change to a line view of your model.
3. Go to the Labels pane and select the Element IDs.
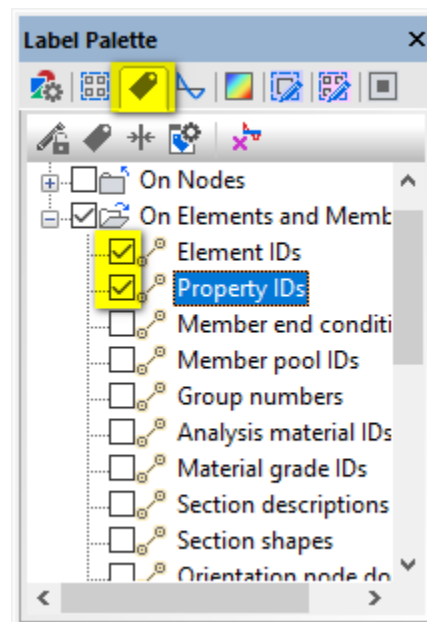
4. The Graphics view changes to add the labels. Add the Property IDs to show which associated property has been assigned to each element. The slab will have a property of pa1, showing that it has 2D (area) property 1.
   The shortcut button for Label Element IDs will show as selected.
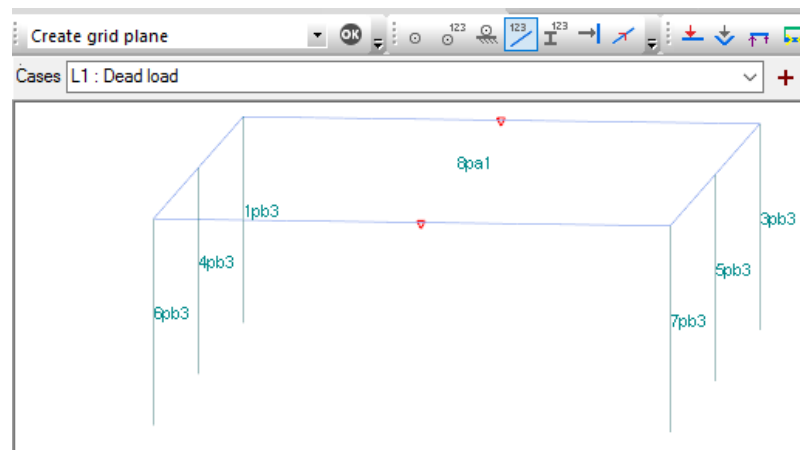
5. Select Analysis material IDs. If they are labelled as "m0" this shows that the analysis material is derived from the material grade. Check Material grade IDs as well to confirm your choices. Concrete material grades are prefixed with "c".
6. Select Element axes to confirm that the z-axis of your slab is pointing in the global direction. If not, you can use the Flip Elements command in the sculpt menu to correct it.

Add a face load 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 and select Face Loads in the 2D Element Loading section.
2. Start the Face Load wizard. Enter the element number of the Quad 4 (8 in the picture above).
3. Add a load of -15kN in the z direction and click [OK].
4. Change to the graphic window and click to display the load diagrams.
5. The face load will be displayed at the centre of the slab.

Manually meshing your slab

This shows you how to split your quad 4 element into multiple smaller elements.

1. Change to the Graphic window.
2. Change to Element select mode (E) and click your slab to select it.
3. Select the Sculpt menu, select 2D element operations and then select the Split Quad Elements command.
4. Enter 8 in each of the split fields and click Preview to display how the slab will be split.
5. Click [OK] to split the slab.
6. Switch on the load diagrams (if they are not still on) to show that the load has been split across all the new Quad 4 elements.

Meshing a complex shape in the Design layer

This section shows you how to create and mesh a region containing a void. This allows you to create and mesh complex slabs. It covers:

Creating a grid at a specified elevation

Defining two areas on the current grid

Specifying that one of the areas is a void

Converting the areas to a region containing a void.

Auto-meshing the region.

This section assumes that you have six concrete columns. If working through this guide, you will need to delete any existing analyses and Quad 4 elements before proceeding.

Create the grid to draw your slab on

1. Change to the Graphics window.
2. Change to the Design layer by pressing [Control]+[Alt]+[Delete]. This is where you can define a less abstract model based on construction codes.
   The analysis version of your model will be shown as a series of dashed lines connecting nodes together.
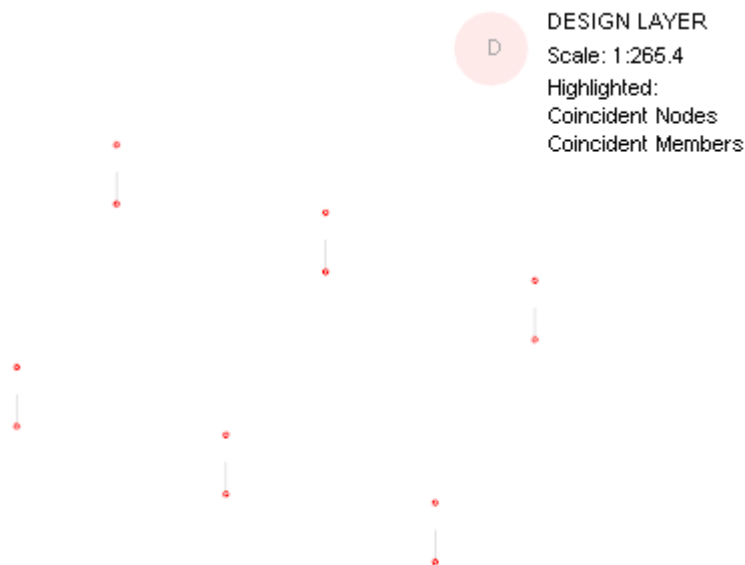

3. Click the Draw Grid button ⊞ to draw the current grid.
4. Press N to change to node select mode.
5. Right-click one of the nodes at the top of a column and select the option Set Current Grid to This from the context menu. You will be asked if you wish to create a new grid plane.
6. Click [Yes]. A new grid plane will be created at the top of the columns.
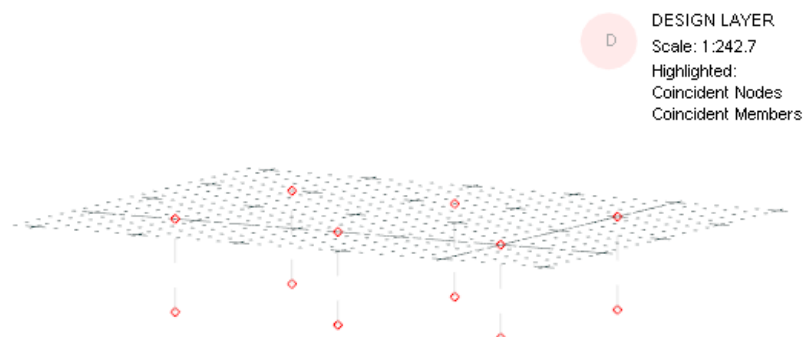
Draw a filled are to define the perimeter of your slab

1. You can now define the shape of your slab by drawing on the plane. Select the Add Lines tool.
2. Click on the first corner of the outside of your slab, then hold down the control key when you click on the next corner. This continues the line. Release the control key before you click to close the rectangle.
3. When the shape is closed, you will be asked if you wish to add areas. Click [Yes]. The area will be marked with a crossed circle.
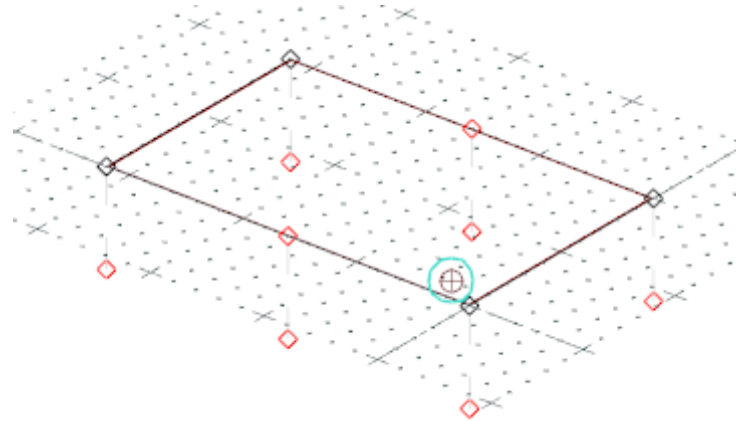
Define a void area to create a hole in your slab

1. Create an area to define the void in the same way that you created the perimeter. Choose the Area selection tool and select the void by clicking on the crossed circle.
2. Change to the Modify Entities pane and change the Area Type to Void, then click the OK button to apply the change.

Create a region from the two areas to define the slab

1. Using the Area selection tool , select both Areas.
2. Right click on the background of the Graphics window and select Add Region from the context menu.
3. The Add Region dialog will open. Confirm that Add One Region is selected and it will include areas 1 and 2.
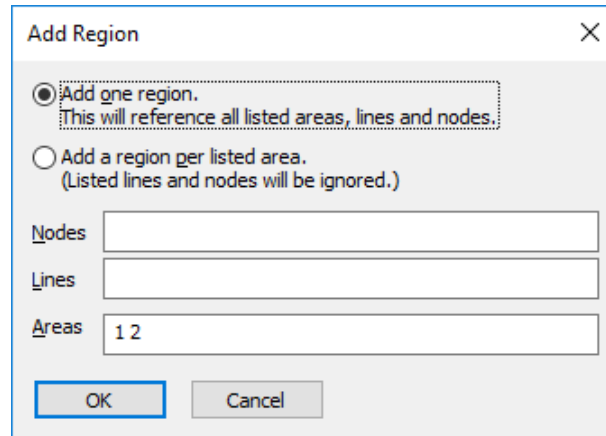
4. Click [OK]. You will be given the option to edit the region. Click [Yes] and give the region a name and click [OK].
5. A blue crossed circle appears to show the region's selection point.

Auto-meshing the slab

1. From the Model menu, select Mesh Generation and then select Check All Regions for Mesh Generation.
2. If the Region passes the check, select Model | Generate | 2D mesh for Regions. The Generate 2D Mesh for Regions dialog will be displayed with a list of regions to mesh. Click [OK].
3. The Region will be meshed into multiple Quad 8s. These will use the existing defined 2D property.

Applying a load to the slab

You can apply face loads to individual elements or you can apply a grid load across all the elements on the plane. This section describes how to apply a grid load via a grid surface. Grid surfaces are used to transmit a defined load to a list of elements on a grid plane.

Creating the grid surface

1. Open the Grid Surfaces data table in the Geometry section in the Data Explorer.
2. Double-click the first row to open the Grid Surface Definition wizard.
3. Set the Grid plane to reference to be 1. Grid plane 1 (the grid plane that you created at the top of the pillars)
4. Leave the element list as all to include all the elements on the plane.
5. Apply the surface to 2D elements.
6. Click [OK] to define the Grid Surface. This is where the load will be applied.

Defining the grid load

1. Open the Grid Areas Load data table in the Loading section in the Data Explorer.
2. Double-click the first row to open the Grid Load Definition wizard and define a load of -15kN in the z direction.

3. Click [OK] to create the load. Close the Data table and return to the graphics window.

4. Click the Loads diagram button to display the loads.

Displaying the slab displacement as a contour

1. Press the Analyse All button Σ to analyse the load.
2. Press P to change to a plan view in the Graphics window.
3. Go to the Contour palette pane and scroll to find the 2D element results.
4. Compare the element displacement results.

5.
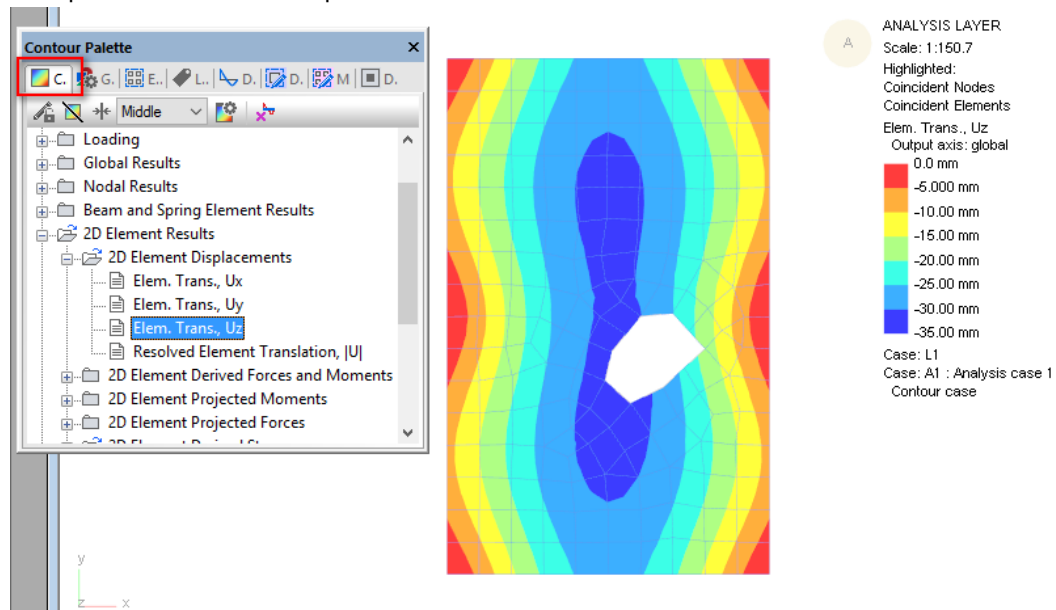
Meaning of symbols in the Graphics window

Symbol	Definition
Cyan circle	Current item
Magenta cross	Selected item
Black diamond	Node in analysis window
Red diamond	Unused node in design window / analysis window after analysis carried out?
Black triangle	Used node in the design window
Red cross	Nearest node
Red arrow point	Element axis
Grey line(s)	Element
Green line with squares	Closed polyline
Brown line with squares	Open polyline

Maroon circle with cross in polygon	Area
Green circle with cross in polygon	Region