



Oasys GSA 10.1

Getting Started



© Oasys 1985 – 2021

All rights reserved. No parts of this work may be reproduced in any form or by any means – graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems – without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

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	4
Starting a model	4
Setting up the model	4
Adding codes, grades, and sections	4
Editing the model	6
Add the support nodes	6
Setting the type of column you are going to create	7
Creating the columns by extrusion	7
Inspecting elements in the Graphics window	8
Creating the beam by sculpting	8
Setting restraints.	9
Apply the node and beam loads	9
Mesh the model	9
Adding self-weight loads	10
Creating a lateral load on a node	10
Creating a vertical load on a beam	10
Displaying loads in the Graphics window	11
Running a simple static analysis	11
Displaying results as a data table	12
Running a more complex analysis	13
Creating slabs	14
Defining your concrete sections	14
Converting existing columns to concrete	15
Creating a 2D Member in the Design layer	16
Checking the properties of your slab	18
Setting the properties of your slab	18
Applying a load to the slab	20
Displaying the slab displacement as a contour	20
Meaning of symbols in the Graphics window	21

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.

Add Catalogue Se	ctions		×
Member type	1D generic \checkmark	305x305x240 , 305x305x198	Select All
Material grade	1: S355 ~	305x305x158 305x305x137	Deselect All
Analysis material	from Grade \checkmark	305x305x118 305x305x97	Invert Selection
Name	<d></d>	254x254x167 254x254x132	Invert Selection
Nume	The name can include material grade	254x254x107 254x254x89	
	<g>; member type, <t>; description,</t></g>	254x254x73	
	<d> and number, #.</d>	203x203x127 203x203x113	
Catalogue	British Steel 🗸	203x203x100	
		203x203x86	
Туре	Universal Columns (UC) V	203x203x60	
	Include superseded sections	203x203x52	
	Steel Design Attributes	152x152x51	
		Duplicate sections will be apper	ded and not
Ασά το ρόοι	<u> </u>	overwritten.	
ОК	Cancel		Help

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.

Add Catalogue Se	ctions		×
Member type	1D generic 🗸	457x152x82	∧ Select All
		457x152x74	
Material grade	1: \$355 🗸	457x152x67	Decelect All
-		457x152x60	Deselect All
Analysis material	from Grade	457x152x52	
Analysis material	in on on one of the second sec	406x178x85	Invert Selection
		406x1/8x/4	
Name	<a>	406x1/8x6/	
		406x1/8x60	
	The name can include material grade,	406X178X54	
	<g>; member type, <t>; description,</t></g>	406x140x53	
	<d> and number, #.</d>	406x140x46	
Catalanus	Pritich Steel	400X140X39	
Catalogue	British Steel	350X1/1X0/	
		256v171v51	
Туре	Universal Beam(UB) 🗸 🗸	256v171v45	
		20Ev16EvE4	
	Include superseded sections	305x165x46	
		305x165x40	
	Steel Design Attributes	205-127-40	¥
		Duplicate costions will be app	and and and
Add to pool	0	overwritten	ended and not
		overwritten.	
OV	Cancel		Halp
UK	cancer		пер

- 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 [4] 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 [례] to copy all subsequent values in the line above.

GSA_getting_started 10.1.gwb : Nodes										×			
	Co	Coordinates		Constanting Auris	Spring Ma	Mass	Damper	Restraint			C -1		
Node	x	у	z	COnstraint Axis	Property	Property	Property	Tran.	Rotn.	Name	Colour		
	[m]	[m]	[m]										
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	v
\												>	

- 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

DESIGN LAYER Scale: 1:41.67 Isometric Scale: 1:51.04 ANALYSIS LAYER Scale: 1:41.67 Isometric Scale: 1:51.04

- 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 do 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.

Properties	4
💑 G 🔠 E 🛹 L	🗾 C 🌭 D 🕅 P
Member Prope	rties
Defaults	
Name	
(none)	
Colour	~
Member type	
1D generic	~
Property Group	Orientation
1 ~ 1	(none)
Orientation angle (°)	
0	
Releases x y z	xx yy zz
End 1	
End 2	
Release stiffness	
Offsets (m) x1 x2	y z
0 0	0 0
Dummy	
Automatic offsets	
1D element type	
2D analysis type	Ť
Linear elements	~
Mesh size (m)	
0	
Intersect with others	
V	
Free v	Free 🗸
Effective length	
Composite	

© Oasys Ltd 2021



Creating the columns by extrusion

- Change to the Graphics window by clicking the Isometric view button ⁽¹⁾ or the Perspective view button ⁽³⁾. Click the Nodes select button ⁽³⁾ (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".

Extrude	\times
Direction of extrusion Axis Global - Type: Cartesian X Oy Oz Axes Alignment 1 Chainage of original elements 0 m	
Number of increments 1	
Increment length 4 m	
Include 1D members along extrusion	
Include 2D members along extrusion	
Include transverse 1D members	
Coincident nodes Cancel	
Use existing nodes where present Preview	
Coincidence tolerance 1 mm Help	

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 \mathbf{I} to give a 3D view of your columns.
- 3. Click 4 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.

- 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 $\stackrel{<}{\leftrightarrow}$ 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:

Properti	ies				x
💦 G	88 E	₽ L	🗾 C	↓ D	🔀 P
Nod (2 sele	e Prope cted)	erties			
Name					
(non	e)				
Colour					
8	(none)				~
Positio	n (m)				
<var< td=""><td>ies></td><td>0</td><td></td><td>0</td><td></td></var<>	ies>	0		0	
Constra	aint axis				
Glob	al				~
Restrai Free x	nt Pin y z	Fix xx	уу	ZZ	

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:

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

Create Elements from Members	×
Member list 🗐 🗸 🗸	
Re-create elements for listed members that already contain elements	
OK Cancel Help	

- 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.

GSA_getting_started 10.1.gwb : Gravity Loading 📃 🔳 💌								
	Flamont List	Land Care	Grav	vity Fac	tors	Name		
Record	ciement List	LOau Case	x	у	z	Name		
			[9]	[9]	[g]			
Defaults	all	1	0	0	-1			
1	all 🔍	1	0	0	-1			
2							¥	
<						>		

Creating a lateral load on a node

- 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.
- 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 $\frac{1}{2}$; 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 \square to show the deformed shape.
- 4. Click by 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

Explorer

DATA

OUTPUT VIEWS

д 🗙

- 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_gett	ting_start	ed 10.1.gv	vb : Beam a	and Sprin	ig Forces a	nd Momer	nts Output				-
Cases all			~ + -	- 🗍 👘	Display	lements v	3		~	+ - a	II
Beam	and S	Spring	Forces	and	Mome	nts					
The force i	n an elem	ent at any	point is the	force real	uired to mai	intain					
equilibrium	n if the ele	ement is cu	t at that poir	nt and the	e end 2 part	of the					
element is	discarded	l. Thus: +ve	axial forces	are tensi	le '						
Forces and	moment	s are outpu	it in element	axis dire	ctions						
i.e. Fx: axia	l force; Fy	& Fz: shea	r forces; Mx	c torsion;	Myy & Mz	: moments					
Element av	or for our	ings are as	defined by t	the spring	property a	xis no.					
clement ax	es ior spr	ingo are ao									
Element lis	t: 3	ingo are as	,								
Element lis	t: 3	ingo are ao									
Element lis	t: 3 Case	Pos	Fx	Fy	Fz	Мхх	Муу	Mzz	Fyz	Myz	
Element lis	t: 3 Case	Pos	Fx [kN]	Fy [kN]	Fz [kN]	Mxx [kNm]	Myy [kNm]	Mzz [kNm]	Fyz [kN]	Myz [kNm]	
Element lis	t: 3 Case	Pos 3	Fx [kN] -0.9749	Fy [kN] 0.0	Fz [kN] -2.620	Мхх [kNm] 0.0	Myy [kNm] 2.652	Mzz [kNm] 0.0	 Fyz [kN] 2.620	 Myz [kNm] 2.652	
Element lis	Case	Pos 3 25.0%	Fx [kN] -0.9749 -0.9749	Fy [kN] 0.0 0.0	Fz [kN] -2.620 -1.310	Mxx [kNm] 0.0 0.0	Myy [kNm] 2.652 -2.260	Mzz [kNm] 0.0 0.0	 Fyz [kN] 2.620 1.310	[Myz] [kNm] 2.652 2.260	
Element lis	Case	Pos 3 25.0% 50.0%	Fx [kN] -0.9749 -0.9749 -0.9749	Fy [kN] 0.0 0.0 0.0	Fz [kN] -2.620 -1.310 0.0	Mxx [kNm] 0.0 0.0 0.0	Myy [kNm] 2.652 -2.260 -3.898	Mzz [kNm] 0.0 0.0 0.0	[Fyz] [kN] 2.620 1.310 0.0	[Myz] [kNm] 2.652 2.260 3.898	
Element lis	Case	Pos 3 25.0% 50.0% 75.0%	Fx [kN] -0.9749 -0.9749 -0.9749 -0.9749	Fy [kN] 0.0 0.0 0.0 0.0	Fz [kN] -2.620 -1.310 0.0 1.310	Mxx [kNm] 0.0 0.0 0.0 0.0	Myy [kNm] 2.652 -2.260 -3.898 -2.260	Mzz [kNm] 0.0 0.0 0.0 0.0	 Fyz [kN] 2.620 1.310 0.0 1.310	[Myz] [kNm] 2.652 2.260 3.898 2.260	
Element lis	Case	Pos 3 25.0% 50.0% 75.0% 4	Fx [kN] -0.9749 -0.9749 -0.9749 -0.9749 -0.9749	Fy [kN] 0.0 0.0 0.0 0.0 0.0	Fz [kN] -2.620 -1.310 0.0 1.310 2.620	Mxx [kNm] 0.0 0.0 0.0 0.0 0.0	Myy [kNm] 2.652 -2.260 -3.898 -2.260 2.652	Mzz [kNm] 0.0 0.0 0.0 0.0 0.0	 Fyz [kN] 2.620 1.310 0.0 1.310 2.620	[Myz] [kNm] 2.652 2.260 3.898 2.260 2.652	
Element lis	Case A1	Pos 3 25.0% 50.0% 75.0% 4 3	Fx [kN] -0.9749 -0.9749 -0.9749 -0.9749 -0.9749 -1.889	Fy [kN] 0.0 0.0 0.0 0.0 0.0 0.0	Fz [kN] -2.620 -1.310 0.0 1.310 2.620 -3.706	Mxx [kNm] 0.0 0.0 0.0 0.0 0.0 0.0	Myy [kNm] 2.652 -2.260 -3.898 -2.260 2.652 0.9532	Mzz [kNm] 0.0 0.0 0.0 0.0 0.0 0.0	 Fyz [kN] 2.620 1.310 0.0 1.310 2.620 3.706	[Myz] [kNm] 2.652 2.260 3.898 2.260 2.652 0.9532	
Element lis	A1	Pos 3 25.0% 50.0% 75.0% 4 3 20.0%	Fx [kN] -0.9749 -0.9749 -0.9749 -0.9749 -0.9749 -1.889 -1.889	Fy [kN] 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.	Fz [kN] -2.620 -1.310 0.0 1.310 2.620 -3.706 -3.706	Mxx [kNm] 0.0 0.0 0.0 0.0 0.0 0.0 0.0	Myy [kNm] 2.652 -2.260 -3.898 -2.260 2.652 0.9532 -6.459	Mzz [kNm] 0.0 0.0 0.0 0.0 0.0 0.0 0.0	 Fyz [kN] 2.620 1.310 0.0 1.310 2.620 3.706 3.706	[Myz] [kNm] 2.652 2.260 3.898 2.260 2.652 0.9532 6.459	
Element lis	A1	Pos 3 25.0% 50.0% 75.0% 4 3 20.0% 20.0%	F x [kN] -0.9749 -0.9749 -0.9749 -0.9749 -0.9749 -1.889 -1.889 -1.889	Fy [kN] 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.	Fz [kN] -2.620 -1.310 0.0 1.310 2.620 -3.706 -3.706 1.294	Mxx [kNm] 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0	Myy [kNm] 2.652 -2.260 -3.898 -2.260 2.652 0.9532 -6.459 -6.459	Mzz [kNm] 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0	Fyz [kN] 2.620 1.310 0.0 1.310 2.620 3.706 3.706 3.706 1.294	[Myz] [kNm] 2.652 2.260 3.898 2.260 2.652 0.9532 6.459 6.459	

Titles and Model Statistics
History
Environmental Impact Summary
Analysis Details
All Input Data

- Nodes
- Elements
- Members
- Materials
- Properties
- Loading
- Global Results
- Nodal Results
- Beam and Spring Element Results
 Beam and Spring Displacements
 Beam End Relative Displacements
 Beam and Spring Forces and Moments

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 \sum 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.

Analysis Wizard : Solver	Option	×	Analysis Wizard : Modal Dynamic Parameters
Task name	<solution></solution>		Task 2: Dynamic - Stage: Whole model
Static	⊖ Static O Static P-delta		Number of modes 4 Start mode 1
Dynamic	Nonlinear static Nonlinear static (explicit) Modal Modal Explicit time history		Max. no. iterations 128 ▲ Additional restraint none Frequency shift 0 Hz
Ritz analysis	◯ Ritz ◯ Ritz P-delta		Frequency cut-offs none V Lower 0 Upper 0 Hz
Dynamic response	OResponse spectrum OPseudo response spectrum OLinear time history OHarmonic		Mass option Mass derived from loads Output mass at nodes Only externally applied forces are included
Buckling	Footfall OPeriodic excitation Modal Component buckling Paff		O Lacuate mass from the case definition element shape functions [gnore element mass, except for mass elements <u>Direction</u> ○ X ○ X ◎ Z
Bridge	Bridge load optimisation		Scale factor 1 Scale factor 1
Miscellaneous Lightweight structures	Mass LS-DYNA time history Form finding		Calculation options Convergence tolerance 1E-12
Post-processing Diagnostic	Model stability Model stability P-delta		Calculate modal (material) damping
	Solver GSS/implicit		Stiffness proportion 1 Mass proportion 0
	Task number 2 Initial analysis case 1		
Imperfection	none V 0.5 % (height)		
Analysis stage	Whole model \vee		
	< Back Next > Cancel He	lp	< <u>B</u> ack <u>N</u> ext > Cancel Help

- 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 I to show the deformed shape. Click on a 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 automeshing 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 *k* 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].

© Oasys Ltd 2021

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.

Properties

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 \sum .
- 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 $\square \rightarrow \square$
- 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 rightclick 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 \rightleftharpoons 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.

Element Properties (2 selected)			
Name	e		
(no	ne)		
Colou	ur		
۵	(non	e)	~
Elem	ent type	2	
Bea	m		~
Prope	erty	Group	Orientation
3	^	1	(none)
(1	CAT	BS-UC 20	3x203x60
2	CAT	BS-UB 40	6x178x54
³ 600x600			

-b x

© Oasys Ltd 2021

6. Click \mathbb{I} 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.
- 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 *select* 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			×				
	Tura		Conver	^			
Element	туре	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	\sim
✓ ► \Elements (Offsets (Releases / < >							

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 doubleclick 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.
- 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.

Explore	r 🛛 🗖 🛛				
DATA	OUTPUT VIEWS				
Tit	les				
🕨 Sp	ecification				
🕨 Ge	Geometry				
► Ma	Materials				
💌 Pro	 Properties 				
	Section Library 3				
	Section Modifiers				
	Springs				
	Masses				
	2D properties				
	3D properties				

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 \mathbf{I} to give a 3D view of your model.)



2. Click \mathbf{I} to change to a line view of your model.

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

Labels 🎝 🗶
🕵 G. 🏢 E 🥜 L 🗾 C. 👆 D. 📝 P
🔏 🛷 🚸 😵 📩
⊞
🗄 🖳 🛅 On Nodes
– ✓ 🖉 On Elements and Members
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*.
- Start the 2D Element Face Load wizard. In the 2D element list type PA1

2D Element Face Load	X
Face load 1	
Name	
2D element pa1	
Load case 3 Case Specifiction	
Type	
Direction z v Axis Global V Projected	
Value 1 -5 2 -5 3 -5 4 -5 [kN/m²]
Position r 0 s 0	
OK Cancel	Help

- 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 \checkmark 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

- 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)