# Oasys GSA 'Step Through' Tutorial:

# 'General Use of GSA' training exercise

This tutorial guides you through the process of setting up a GSA model one step at a time. The tutorial is based on the main training exercise used in the 'General Use of GSA' training course, so a broad range of topics is covered.

Topics covered in this 'step through' tutorial (followed by the section in which the topic is covered):

- Grid lines (1, 3)
- Various sculpt operations (2 to 5)
- User axes (3)
- Mesh generation (4)
- Grid loading (5)
- Analysis stages (6)
- Raft analysis (7)

- P-delta analysis (8)
- Buckling analysis (8)
- Dynamic analysis (8)
- Footfall analysis (8)
- RC member design (9, 10)
- Steel member design (9, 10)

ANALYSIS LAYER Scale: 1:175.9

• RC slab design (10)

Each step of this tutorial is numbered, has a short "Step" description and "Actions" that describe the process. If you follow the actions then by the end of the tutorial you will have created the model shown below. This model has been saved at various stages in the process, with file names as indicated in the notes. These files are available in the <\Samples\Training\_General> folder beneath your GSA program folder, typically <C:\Program Files\Oasys\GSA 8.2\Samples\Training\_General>.



The Finished Model TrGen\_10.gwb

#	Step	Notes
Crea	te orthogonal f	frame
•	Set up grid l	
•	× •	orthogonal frame based on the grid lines.
1.1	Enable	'Tools   Preferences   Advanced Features' 'Enable All'
1.1	advanced	(Many advanced features will be used during this exercise.)
	features	(while advanced reactives will be used during this exercise.)
1.2	Open new	'File   New'
	model	
1.3	Structure	'New Model Wizard : Structure type   Type' : 'Space'
	type	(Allows mixed 1D and 2D elements.)
1.4	Grid lines	'New Model Wizard : Structure type   Grid Lines' to open 'Grid Line
		Definition' dialog; assign grid lines: 1 to 4, A to C:
		• Edit in line 1:
		o 1 Line 0 0 12 90
		• Add 3, copying line 1, incrementing in X by 5:
		o 2 Line 5 0 12 90
		o 3 Line 10 0 12 90
		o 4 Line 15 0 12 90
		• Edit line 4 X coordinate:
		o 4 Line 16 0 12 90
		• Edit in line A:
		• A Line 0 0 16 0
		• Add 2, copying line A, incrementing in Y by 6
		o B Line 0 6 16 0
		• C Line 0 12 16 0
		(Resulting in:
		$\circ$ 1 Line 0 0 12 90
		$\circ$ 2 Line 5 0 12 90
		○ 3 Line 10 0 12 90 ○ 4 Line 16 0 12 90
		$\begin{array}{cccccccccccccccccccccccccccccccccccc$
1.5	Generate	'New Model Wizard : Structure type   Generate data for the selected
1.0	beam	structure type' to open 'Data Generation Wizard : Orthogonal frame'
	elements	dialog
		Specify data for generation of orthogonal frame:
		• 'Generate elements'
		• 'Generate bays from grid lines'
		(Grid planes will be generated automatically.)
		• 3 storeys at 3.5m
		• Assign concrete sections to columns and x and y beams
		• 'Include supports'
		• 'Include self weight'
1.6	View the	'Graphic View'
	generated	• Label ( 🖾 ) on nodes: restraints
	data	
		<ul> <li>Label ( ) on elements: section shapes</li> <li>Drow grid</li> </ul>
17	Sava view	Draw grid     'Viow   Sava Graphic Viow'
1.7	Save view	'View   Save Graphic View'
	l	• Saved view name: "startup"



•

GSA Step Thru' Tutorial - 'General Use of GSA' training exercise

'Graphic View'

2.3

View the

modified

data

Do similar for longitudinal and transverse beams

'Graphic Display' toolbar; 'Section display'

'Graphic Display' toolbar; 'Shrink' 脑



### Create skew wing grid lines

- Create a user axis set at the axis of rotation.
- Add the skew grid lines, making use of the new user axis set in the 'Grid Line Definition' dialog.

	Definition d	nalog.
3.1	Reset	'Graphic View'
	display	• 'Graphic Display' toolbar; 'Reset display adornments' *
		(Resets the view to default view settings, as last specified by
		'View   Save Default View Settings')
3.2	Create axis	(We need an axis about which we can rotate grid lines.)
	set at node	'Graphic View'
	12	• Label ( 🗹 ) on nodes: node numbers
		• Label ( 🖾 ) axes: all user axes
		'Graphic View   Select nodes' cursor mode 🥵
		• Select nodes 9, 11, 12
		(Either click on nodes or right-click on the Graphic View
		background; 'Select List'; "9 11 12"; OK.)
		'Sculpt   Create User Axes'
		• Toggle the 'Axes definition' spin control till an axis set with
		origin at node 12 and vertical z (blue) axis is displayed.
		Create
3.3	Add skew	'Data   Grid Lines' to open 'Grid Line Definition' dialog; assign grid
	grid lines	lines: 5 to 8, A1 to C1:
		• Add 1, copying line 4, incrementing in X by zero: line 5
		• Add 3, copying line 5, incrementing in X by 5: lines 6,7,8
		• Rotate lines 5 to 8 about "axis 1" z by 30 deg.
		• Edit in line A.1:
		• A.1 Line 16 0 15 0
		• Add 2, copying line A.1, incrementing in Y by 6, to create
		lines B.1, C.1
		• Rotate A.1, B.1, C.1 about "axis 1" z by 30 deg.



# Create slab region

- Create area entities, made up of line entities, that define the outline of the slab and a hole in the slab.
- Create nodes at column positions.

•	Generate the	mesh.
4.1	Set current grid to "1st floor"	<ul> <li>Either: 'Data   Define Current Grid' to open 'Current Grid Definition' dialog</li> <li>Specify 'Grid plane' as "1st floor"</li> </ul>
	11001	Or: 'Graphic View' (any cursor mode)
		<ul> <li>Right-click on any node at 1st floor level; 'Set Current Grid to This'</li> </ul>
		(The grid lines are drawn on the current grid. Coordinates are reported with respect to current grid axes. Sculpt tools operate on the current grid surface.)
4.2	Form	'Graphic View' (any cursor mode)
	volume of	<ul> <li>'Orientation' toolbar; 'Y elevation' or <y></y></li> </ul>
	"1st floor"	'Graphic View   Volume' cursor mode 🙀
	only	• Drag left to right around 1 <sup>st</sup> floor
		(For clarity while sculpting.)
		'Graphic View' (any cursor mode)
		• 'Orientation' toolbar; 'Plan' or <p> 🖾</p>
4.3	Set Graphic	'Graphic View' (any cursor mode)
	View to	• Right-click on the Graphic View background; 'Switch layer'
	Design Lover	(Switches to 'Design Layer' - pink background.)
4.4	Layer Add area	(Members, lines, areas and regions exist on the Design Layer) Display grid, and snap to grid points
4.4	defining slab	Sculpt' toolbar; 'Grid'
	boundary	<ul> <li>Sculpt' toolbar; 'Snap to grid points' </li> </ul>
		'Graphic View   Add Lines Sculpt Tool' cursor mode
		Create a line by clicking 1:A and 2:A
		(Default line properties are assigned.)
		(Nodes are created as required.)
		• Right-click on new line; 'Edit Line'
		• In 'Line Definition' dialog, set 'Number of segments' = 4; OK
		(Mesh generation will aim to generate this many elements along line.)

4.5	Modify line to arc	<ul> <li>Right-click on new line; 'Set Line Defaults as This' (Now new lines will default to 'Number of segments' = 4.)</li> <li>Create new lines around slab in original portion of structure, connecting grid line intersections (Breaking lines at grid line intersections will result in nodes at column positions in the mesh.) (Ctrl+click causes second node of one line to be used as the first node of the next. i.e. quicker.) (<esc> quits the sculpt operation in progress.)</esc></li> <li>On closing the loop, accept the offer to add an area (Area 1 is created, referencing the loop of lines just closed.)</li> <li>Create areas 2 and 3 defining the slab extents for the skew wing and interconnecting segment, respectively</li> <li>Switch off snap to grid points</li> <li>'Sculpt' toolbar; 'Snap to grid points' I</li> <li>'Graphic View   Add Nodes Sculpt Tool' cursor mode 'Graphic View   Add Nodes Sculpt Tool' cursor mode 'Graphic View   Add Nodes Sculpt Tool' cursor mode 'Graphic View (any cursor mode)</li> <li>Right-click on line that is to be an arc; 'Edit Line'</li> <li>In 'Line Definition' dialog, set 'Arc defined by radius and point', set third node to 58, and 'Arc radius' = 12; OK (The arc will have a radius of 12m, lying in the plane of the three nodes defined for the line and pulled towards the third node.)</li> </ul>
4.6	Create hole in slab	<ul> <li>Ensure 'Snap to grid points' is off</li> <li>'Sculpt' toolbar; 'Snap to grid points'</li> <li>'Graphic View   Add Lines Sculpt Tool' cursor mode</li> <li>Create four lines bounding hole, vertices at approx.: <ul> <li>(X = 11.00m Y = 1.500m Z = 3.500m)</li> <li>(X = 14.50m Y = 4.000m Z = 3.500m)</li> <li>(X = 14.50m Y = 4.000m Z = 3.500m)</li> <li>(X = 11.00m Y = 4.000m Z = 3.500m)</li> <li>(X = 11.00m Y = 4.000m Z = 3.500m)</li> <li>(X = 11.00m Y = 4.000m Z = 3.500m)</li> </ul> </li> <li>Either: on closing the loop, accept the offer to add an area</li> <li>Or: select lines and 'Sculpt   Geometric Entity Operations   Add Area'</li> <li>'Graphic View' (any cursor mode)</li> <li>Right-click on area that is to be hole; 'Edit Area' (The selection handle for an area is the symbol inset from the first corner of the area.)</li> <li>In 'Area Definition' dialog, set 'Type' to 'Void'</li> </ul>
4.7	Add missing internal nodes	<ul> <li>Switch on snap to grid points</li> <li>'Sculpt' toolbar; 'Snap to grid points'</li> <li>'Graphic View   Add Nodes Sculpt Tool' cursor mode</li> <li>Create a nodes at 6:B.1 and 6:B.2 (These will be used to generate supporting columns and to identify to the region the column positions.) (The current grid is still set to "1st floor" so the new nodes will be created on that plane.)</li> </ul>
4.8	Add region	<ul> <li>'Graphic View   Select nodes' cursor mode <sup>∞</sup></li> <li>Select the internal nodes</li> <li>'Graphic View   Select areas' cursor mode <sup>∞</sup></li> <li>Ctrl+A to select all areas</li> </ul>

	l	• 'Sevent   Coompany's Entity Operations   Add Design'
		• 'Sculpt   Geometric Entity Operations   Add Region' (The region is a collection of nodes, lines and areas. A mesh
		generation operation generates a mesh for a region.)
4.9	Add 2D	(Generated 2D elements are assigned the property and group number
4.9	Properties	based on those set for the respective area. Areas created in sculpt
	Toperties	default to having the property and group set to the area number.)
		'Graphic View'
		• Label ( 2 ) on geometric entities: On areas: Property ref.s
		(The 2D properties can be entered before or after the mesh generation.)
		(For RC slab design we want the local axes to be aligned with the
		reinforcement direction, hence reference to user defined axes in the 2D
		Element Properties.)
		'Data   Geometry   Axes' (or from the Gateway) to open the 'Axes' table
		• Modify the type of axis 1 to 'Cylindrical' Create axis set aligned with skew wing
		<ul> <li>Select nodes 56, 63, 64</li> </ul>
		<ul> <li>Select houses 30, 03, 04</li> <li>(See section 3.2 above)</li> </ul>
		'Data   Properties   2D Element Properties' to open the '2D Element
		Properties' table.
		Enter three properties:
		• Axis: 'Global', axis 2, axis 1
		(For the original wing, skew wing and interconnecting
		segment, respectively.)
		• Type: 'Flat Shell'
		(i.e. in-plane and bending action.)
		• Material: 'Concrete long term'
		• Thickness: 0.3m
4.10	Save "1st	(This list will be used after the mesh has been generated. It's easier to
	floor column	select the list before the mesh is generated.)
	positions" node list	'Graphic View   Select nodes' cursor mode
	noue list	• Select all nodes at 1st floor level column positions
		• (All at 1st floor except those defining the void and third node of arc.)
		<ul> <li>Right-click on the Graphic View background; 'Save Selection</li> </ul>
		as List'
4.11	Generate	Either: 'Tools   Region Mesh Generation   Generate 2D Mesh for
	mesh	Regions'
		Or: right-click on the region; 'Generate 2D Mesh for Region'
		(The selection handle for a region is the symbol at the centre of the
		largest area included in the region.)
4.12	Check and	(It is preferable to adjust the condition of the mesh by adjusting the
	adjust the	parameters that define the mesh, rather than by adjusting the generated
	mesh	nodes and elements.)
		• 'Analysis   Check Data' reveals severe warnings on the shape
		of some the generated elements.
		• Setting the region 'Steps' setting to 'Linear' improves the
		condition of the mesh. (But less heed is taken of the specified
		number of segments on lines.)
		• Right-click on region; 'Edit Region'
		• Adjusting the number of segments on lines in the vicinity of the void improves the mesh.
	l	<ul> <li>○ Either: 'Graphic View', label ( <sup>∽</sup>) on geometric</li> </ul>

<ul> <li>entities: On lines: Dots along lines <ul> <li>Or: 'Display Favourites' toolbar; 'Label node dots'</li> <li>•</li> </ul> </li> <li>to get an idea.</li> </ul> <li>A quick fix is to set the step size consistently for all lines (instead of the number of elements along each line). <ul> <li>Right-click on a line; 'Edit Line'</li> <li>In 'Line Definition' dialog, set 'Mesh by step size' and set the 'Step size' = 1; OK</li> <li>Right-click on same line; 'Set Line Defaults as This'</li> <li>Select all lines (Ctrl+A)</li> <li>'Sculpt   Apply Defaults to Selection'</li> </ul> </li> <li>Further improvement can be achieved by adding 'construction lines' to the region to help guide the mesh generator. (See the lines connecting from the slab boundary to the void in TrGen_04.gwb)</li> <li>Following a mesh generation it is advisable to check the connectivity of the mesh by 'Graphics   Graphic Settings   Highlight Edges'</li>
TrGen_04.gwb (without the mesh)

#### Add skew wing columns, rigid diaphragm and loading

- Create columns using the sculpt 'extrude' option.
- Create rigid constraints.
- Define load case titles.
- Define face loading, grid loading and node loading.
- Define a combination case.

5.1	Set Graphic	'Graphic View' (any cursor mode)					
	View to	• Right-click on the Graphic View background; 'Switch layer'					
	Analysis	(Switches to 'Analysis Layer' - yellow background.)					
	Layer	(Elements exist on the Design Layer)					
5.2	View whole	(The view is currently of a defined volume.)					
	model	Either: 'Graphic View   Volume' cursor mode 🔗					
		• Click anywhere in the view					
		Or: Graphic View 'Lists' toolbar; 'Reset to All Entities' <b>all</b> (Not enabled when all entities are being displayed.)					

5.3	Extrude columns	<ul> <li>'Graphic View   Select nodes' cursor mode</li> <li>Select the nodes at which we want new columns to support the skew wing of the slab</li> <li>'Sculpt   Extrude Selection' <ul> <li>'Direction of extrusion': 'Global z</li> <li>'Number of increments': 1</li> <li>'Increment length': -3.5</li> <li>Set 'Include Beam elements along extrusion'</li> </ul> </li> <li>'Graphic View   Select elements' cursor mode <ul> <li>Select the new elements</li> <li>'Graphic View'</li> <li>Label (</li> </ul> </li> </ul>
		<ul> <li>(New columns are pointing downwards.)</li> <li>'Sculpt   Flip Elements'</li> <li>Label ( ) on elements: section shapes</li> </ul>
		(New columns need to be rotated.)
		<ul><li> 'Sculpt   Modify Selection'</li><li> 'Modify orientation angle' to 30 degrees</li></ul>
5.4	Assign rigid	'Graphic View   Select nodes' cursor mode
	constraint to	• Select the nodes on the 2nd floor
	2nd floor	Sculpt   Create Rigid Constraint'
		'Data   Constraints   Rigid Constraints' (or from the Gateway) to open
		the 'Rigid Constraints' table
		• For the new rigid constraint, set the 'Type of linkage' to 'XY Plane'
		(i.e. the nodes will behave rigidly in the xy plane and
		independently in the Z direction.)
5.5	Load case	'Data   Cases and Tasks   Load Case Titles' to open the 'Load Case
	titles	Titles' table, to add the following:
		• Case 1 "Dead load" (already entered by New Model Wizard)
		• Case 2 "First floor – face loads"
		• Case 3 "Second floor - GAL"
		• Case 4 "Roof – GAL" • Case 5 "Wind load"
5.6	Face load on	Case 5 "Wind load"     'Data   Loading   2D Element Loading   Face Loads' to open the '2D
5.0	slab	Element Loading   Face Loads' table, to add the following:
	Siuc	<ul> <li>Element list PA1 to PA*, load case 2, in the global Z direction,</li> </ul>
		value -10kN/m2
5.7	'Plane' grid	'Data   Loading   Grid Loading   Grid Area Loads' to open the 'Grid
	area loads	Loading   Grid Area Loads' table, to add the following:
		• Grid plane "2nd floor", Area type "Plane", load case 3, in the
		<ul> <li>global Z direction, value -10kN/m2</li> <li>Grid plane "3rd floor", Area type "Plane", load case 4, "= ="</li> </ul>
		• Ond plane Sid hoor, Alea type Plane, load case 4, $==$ (to copy the rest of the row from the row above)
5.8	Set current	(We're about to define a polyline on the roof. Polylines are defined on
	grid to "3rd	the current grid plane.)
	floor"	'Graphic View' (any cursor mode)
		<ul> <li>Right-click on any node at roof level; 'Set Current Grid to This'</li> </ul>
5.9	'Polyline'	Ensure grid is displayed, and 'snap to grid points' is switched off
2.7	grid area	<ul> <li>Sculpt' toolbar; 'Grid' ++</li> </ul>
	load	• 'Sculpt' toolbar; 'Snap to grid points' 🛨

		<ul> <li>'Graphic View   Polyline' cursor mode</li> <li>Click on roof to form vertices of polyline (The polyline will define the area that is to be loaded.) (In this context the polyline will be assume to be closed, so you don't need to click on the start point to close the polyline.)</li> <li>Right-click on the Graphic View background; 'Create Grid Loading   Create Grid Area Load'; enter:</li> <li>Load case 4, global Z direction, value -5kN</li> <li><esc> to clear the polyline</esc></li> </ul>
5.10	Wind load	<ul> <li>'Graphic View   Select nodes' cursor mode</li> <li>Either: Select the nodes at X=0 at the roof level <ul> <li>Or: right-click on the Graphic View background;</li> <li>'Select List' and enter 'Y37'; OK</li> </ul> </li> <li>Right-click on the Graphic View background; 'Create Nodal Loading   Create Node Load'; enter: <ul> <li>Load case 5, global X direction, value 20kN</li> </ul> </li> </ul>
5.11	Check the	'Graphic View'
	loading	• On the 'Display Favourites' toolbar; 'All load diagrams'
		<ul> <li>On the Graphic View 'Lists' toolbar; 'Next case'</li> <li>(To step through the load cases.)</li> </ul>
5.12	Combination case	<ul> <li>'Data   Cases and Tasks   Combination Cases' to open the</li> <li>'Combination Cases' table, to add the following: <ul> <li>"Design", "1.4A1 + 1.6A2 + 1.6A3"</li> </ul> </li> <li>(A combination case is a means of combining analysis results. The combining occurs post-analysis, at the time the results are required for display.)</li> </ul>
		a anop Scale: 1:63.9 Hojolopted: Edges Grid Area Loads, Fare: 5000. Nentpic.cm Case: L4 : Root - GAL
		TrGen_05.gwb
Anal	veic stages and	linear static analysis
Analy		al analysis stages.
•	-	rent restraint and constraint conditions per stage.
•	-	hear static analysis of an analysis stage.
6.1	Reset	'Graphic View (analysis layer)'
	display	• 'Graphic Display' toolbar; 'Reset display adornments' *
6.2	Analysis	'Data   Analysis Stages   Stage Definition' to open the 'Stage
	stages	Definition' table, to add the following:

		TrGen_09.gwb : Stag 💶 🛛
		Stage Stage Name Element List
		Defaults Stage # all
		1 skeletal model PB1 to PB*
		2 GsRaft model PB1 to PB* 3 buckling model PB1 to PB*
		3 buckling model PB1 to PB* 4 full model all
		5 footfall model PA1 to PA*
		The name used to identify this analysis stag
		The name asea to identify this analysis stag
6.3	Different	'Graphic View (analysis layer)'
0.5	restraints per	
	stage	<ul> <li>: 'Display Favourites' toolbar; 'Label restraints'</li> <li>'Graphic View   Select nodes' cursor mode</li> </ul>
	8-	Select all restrained nodes
		<ul> <li>Right-click on the Graphic View background; 'Modify</li> </ul>
		Selected Nodes'
		• Restraint type: 'Free'; OK
		<ul> <li><esc> to clear the selection</esc></li> </ul>
		'Data   Constraints   Generalised Restraints' to open the 'Generalised
		Restraints' table, to add the following:
		• "footfall model", "1st floor column positions": pinned restraint
		(i.e. restraint in x y z directions)
		• Other stages, "XY1": encastred restraint (i.e. restraint in x y z
		xx yy zz directions)
6.4	Different	'Graphic View (analysis layer)'
	rigid	<ul> <li>Label ( <sup>21</sup>) on nodes: Rigid constraints</li> </ul>
	constraints per stage	'Data   Constraints   Rigid Constraints' to open the 'Rigid Constraints'
	per stage	table:
		• Apply the rigid constraint that is currently applied to all stages,
		to stages "skeletal model", "GsRaft model" and "full model", thereby excluding the rigid constraint from other stages
		<ul> <li>Apply to stages "skeletal model" and "GsRaft model" rigid</li> </ul>
		constraints to "1st floor column positions" with 'XY plane'
		linkage
		(Choose any of the nodes in "1st floor column positions" as
		master.)
6.5	Inspect	'Graphic View (analysis layer)'
	stages	• On the Graphic View 'Lists' toolbar, 'Stages',
		Display Stage 🐱 1 : skeletal mode 🐱
		• 'Next display' +
		(To step through the stages.)
6.6	Linear static	'Analysis   New Analysis Task' to open the 'Analysis Wizard : Solver
	analysis of	Option' dialog:
	"full model"	• Task name: "Linear Static - full model"
		• Solver option: 'Linear Static'
		• Analysis stage: "full model"
		'Next' to open the 'Analysis Wizard : Static Analysis Cases' dialog:
		• 'Create default cases'
		'Next' to open the 'Analysis Wizard : Cases Set Up' dialog:
		• 'Analyse'



## Raft analysis

- Set up soil interaction data to specify the relationship between the soil (Vdisp data) and structure (GSA data).
- Set up the Vdisp data to define the soil model.
- Specify the Vdisp analysis parameters.

• Perform a raft analys	is.
-------------------------	-----

	I emonin a rai	tt analysis.						
7.1	Reset		view (analysis la					
_	display	• 'G	raphic Display'	toolbar; 'F	Reset disp	lay ador	mments	, <del>*</del>
7.2	Delete results Soil	<ul> <li>'GSA' toolbar; 'Erase all results' ≥</li> <li>(Results need to be erased to allow raft data to be added.)</li> <li>(Deleting the results does not delete the analysis tasks.)</li> <li>'Data   Raft   Soil Interaction' to open the 'Soil Interaction' table, to</li> </ul>						
1.5	interaction nodes	add the fol		, , , , , , , , , , , , , , , , , , ,		i interac	don ta	
			A	В	С	D	E	F -
					oundation		Pres	sure
		Record	Nodes	X dimension	Y dimension	Elevation	Minimum	Maximum =
				[m]	[m]	[m]	[N/m²]	[N/m²]
		Defaults none	1	1	1	0	0	1e+038
		1 XY1	~	2	2	-3	0	1e+038 🧹
		2						
		<						>
		The list of node:	s that are loaded by soil l					>
		The list of node:	s that are loaded by soil l					

			В	L C	D	E
	Layer		No of intermediat displacement	e Young	's modulus N/m²]	Poisson's
	Layer	[m]	levels	Тор	Bottom	ratio
	Defaults			1.00	Dottom	0.20
	1	0.00		) 1000000	0 20000000	0.20
	2	-7.00		) 2000000		
	3	1. C. II D		- 61		
			rofile 1 🖌 2: Soil Pr :art a new record	σπη		>
	- and:					
	TrGe	≘n_09.	gwb:Soil Profile	-		-0
		Α	В	C	D	E
	Layer	Level at top [m]	No of intermediat displacement levels		's modulus N/m²] Bottom	Poisson's ratio
	Defaults	[]	167613	Тор	DULLUIII	0.20
	1	0.00		800000	0 10000000	0.20
	2	5.00		) 1000000		0.20
	3					
		2: Soil P	rofile 2 🖌 Add Soil	Pro < 📃		>
			ofile 2] Cell [B][2]			
Soil zones			sp Data   Soil Zor	es' to ope	en the 'Soil Z	Zones' tab
Soil zones	'Data   Ra add the fo			es' to ope	en the 'Soil Z	Zones' tab
bil zones	add the fo	ollowing		•		Zones' tab
zones	add the fo	ollowing	g: en_09.gwb : Soil A B C	Zones D		
l zones	add the fo	ollowing	; en_09.gwb : Soil	Zones D	E Soil	
zones	add the fo	llowing TrGe Zone	g: A B C x coordinate y c [m] min max mi	Zones D pordinate [m] n max	E Soil profile	
zones	add the fo	llowing	g: A B C x coordinate y c [m] min max mi	Zones D pordinate [m] n max	E Soil profile Soil Profile 1	
zones	add the fo	llowing TrGe Zone	z: A B C x coordinate y c [m] min max mi -3.00 16.50 -3.1	Zones D pordinate [m] n max	E Soil profile Soil Profile 1 Soil Profile 1	
ones	add the fo	llowing TrGe Zone	g: A B C x coordinate y c [m] min max mi	Zones D pordinate [m] n max	E Soil profile Soil Profile 1	
ones	add the fo	llowing TrGe Zone	z: A B C x coordinate y c [m] min max mi -3.00 16.50 -3.1	Zones D pordinate [m] n max	E Soil profile Soil Profile 1 Soil Profile 1	
l zones	add the fo	Defaults	z: A B C x coordinate y c [m] min max mi -3.00 16.50 -3.1 16.50 38.00 -3.1	Zones D pordinate [m] n max 0 15.00 0 25.00	E Soil profile Soil Profile 1 Soil Profile 1	
zones	add the fo	Defaults	2: A B C x coordinate y c [m] min max mi -3.00 16.50 -3.0 16.50 38.00 -3.0	Zones D pordinate [m] n max 0 15.00 0 25.00	E Soil profile Soil Profile 1 Soil Profile 1	
il zones	add the fo	Defaults	2: an_09.gwb : Soil A B C x coordinate y c [m] min max mi -3.00 16.50 -3.1 16.50 38.00 -3.1 16.50 38.00 -3.1	Zones D pordinate [m] n max 00 15.00 00 25.00	E Soil profile Soil Profile 1 Soil Profile 2	
	add the fo	Defaults	g: A B C x coordinate y c [m] min max mi -3.00 16.50 -3.1 16.50 38.00 -3.1 MAB> to start a new re sp Data   Analysi	Zones D pordinate [m] n max 00 15.00 00 25.00	E Soil profile Soil Profile 1 Soil Profile 2	

		💹 TrGen_09.gwb : Analysis Parameters 🛛 🗔 🖂
		Analysis method: 💿 Mindlin 🔘 Boussinesq
		Global Poisson's Ratio: 0.2
		Maximum allowable ratio between values of E at adjacent displacement levels (>1):
		Rigid boundary level (level of
		Displacements at Modify above loads area centroids
		Apply Undo
7.7	Raft analysis	'Analysis   New Analysis Task' to open the 'Analysis Wizard : Solver
	of 'GsRaft	Option' dialog:
	model"	• Task name: "Raft – GsRaft model"
		• Solver option: 'Raft'
		• Analysis stage: "GsRaft model" 'Next' to open the 'Analysis Wizard : GsRaft Control' dialog:
		(Default settings.)
		'Next' to open the 'Analysis Wizard : Static Analysis Cases' dialog:
		• Add "L1 + L3 + L4"
		'Next' to open the 'Analysis Wizard : GsRaft Progress' dialog:
		(Default settings.)
		'Next' to open the 'Analysis Wizard : Cases Set Up' dialog:
		• 'Don't analyse now' 'Finish'
		'GSA' toolbar; 'Analyse all' Σ
		(To analyse all analysis tasks that are set up but not analysed.)
		stantup Analysis stage: 2: GeRuit model
		6 Guie - 1169 2 Labeir Boil Interaction No.s Boil Interaction Anna
		TrGen_07.gwb (without results)

#### **P-δ, buckling and dynamic analysis**

- Specify a static P-δ analysis task.
- Specify a buckling analysis task.
- Specify a modal analysis task.
- Perform analysis of the specified tasks.
- Set up data for response spectrum analysis.
- Specify and analyse a response spectrum analysis.
- Specify an SRSS combination case of basic responses.
- Specify and analyse a footfall analysis.

	opeeny und	
8.1	Static analysis of "buckling model" task	<ul> <li>'Data   Cases and Tasks   Analysis Tasks' to open the 'Analysis Tasks' view</li> <li>Right-click on background of Analysis Tasks View; 'New Analysis Task' (Alternative to 'Analysis   New Analysis Task'.)</li> <li>Task name: "Linear Static - buckling model"</li> <li>Solver option: 'Linear Static'</li> <li>Analysis stage: "buckling model"</li> <li>'Next' to open the 'Analysis Wizard : Static Analysis Cases' dialog:</li> <li>Add "L1", "L3", "L4" and "L5", as separate analysis cases (Double-click on the load cases to add them to the description.) (L2 is face loading applied to 2D elements, which are not included in the "buckling model" stage.)</li> <li>'Next' to open the 'Analysis Wizard : Cases Set Up' dialog: <ul> <li>'Don't analyse now'</li> </ul> </li> </ul>
		'Finish'
8.2	Static P-ð analysis of "buckling model" task	<ul> <li>'Data   Cases and Tasks   Analysis Tasks' to open the 'Analysis Tasks' view</li> <li>Right-click on background of Analysis Tasks View; 'New Analysis Task'</li> <li>Task name: "Static P-delta – buckling model"</li> <li>Solver option: 'Static P-delta'</li> <li>Analysis stage: "buckling model"</li> <li>'Next' to open the 'Analysis Wizard : GSS Static P-delta' dialog:</li> <li>Select 'P-delta analysis with each analysis case defining its own differential stiffness'</li> <li>'Next' to open the 'Analysis Wizard : Static Analysis Cases' dialog:</li> <li>Add 'Name': "P-delta"; 'Description': "L1 + L5"</li> <li>'Next' to open the 'Analysis Wizard : Cases Set Up' dialog:</li> <li>'Don't analyse now'</li> </ul>
8.3	Buckling analysis of "buckling model" task	<ul> <li>'Data   Cases and Tasks   Analysis Tasks' to open the 'Analysis Tasks' view</li> <li>Right-click on background of Analysis Tasks View; 'New Analysis Task'</li> <li>Task name: "Modal Buckling – buckling model"</li> <li>Solver option: 'Buckling   Modal'</li> <li>Analysis stage: "buckling model"</li> <li>'Next' to open the 'Analysis Wizard : GSS Buckling Parameters' dialog: <ul> <li>'Number of modes': 6</li> <li>'P-delta analysis case': "L1 + L5"</li> <li>'Maximum no. iterations': 128</li> </ul> </li> </ul>

		• '] 'Finish'	Don't analy	se now'					
8.4	Modal analysis of "full model" task	'Data   Ca view R A T S A 'Next' to Paramete ('] (') (') (') (') (') (') (') (') (') (')	ases and Tas Right-click of Analysis Tas Cask name: ' Colver option Analysis stag open the 'A rs' dialog: Number of n Maximum n Start mode' Additional r Mass option Mass derive open the 'A Don't analy	on backgro k' 'Modal– f c: 'Dynan ge: ''full n analysis W modes': 5 to. iteratic c: 1 estraint': '': 'Lump d from lo analysis W	ound of f full mod- nodel" /izard : f ons': 128 none mass at ads': "L	Analysis T el" dal' GSS Moda g nodes' 1", "Z"	asks Vie 1 Dynan	ew; 'Ne	
8.5	Analyse	'GSA' to	olbar; 'Anal s all the task	lyse all'	Σ ven't bee	en analysed	vet )		
8.6	Response spectrum data	'Data   D Spectra' t S S S S (1 'Data   D	ynamic Res	ponse   Ro is' toolbar izard' dia code 8' t paramet ponse   Ba	esponse ; 'Wizar log ers unch asic Res	Spectra' to rd' ( 🌋 ) t hanged) ponses' to	o open th	he 'Res	ponse
		TrGen	08.gwb : Basi	c Responses				ſ	
			A	В	С	D	E	F	
		Response	Name	Axis	Direction	Response	Mode Lis	Combina	
		Defaults 1 2 3 1 Control of the name use	Basic response # Basic response 1 Basic response 2 Basic response 3 d to identify the b	Global Global Global	x x y z ponse. Press	Spectrum 1 Spectrum 1 Spectrum 1 Spectrum 1 Spectrum 1	all all all all ted input.	* Metho CQC CQC CQC CQC	
8.7	Response spectrum analysis of "full model"	view • R A • T • S • A 'Next' to dialog:	ases and Tas Right-click o Analysis Tas Cask name: ' Colver option Analysis stag open the 'A Modal analy	on backgro k' 'Response n: 'Dynan ge: ''full n nalysis W	ound of Spectruic respondel" Vizard :	Analysis T um – full m onse   Resp Response S	asks Vie nodel" onse Spe Spectrum	ew; 'Ne ectrum' n Param	w neters'

		• 'Analyse' 'Finish'
8.8	SRSS	'Data   Cases and Tasks   Combination Cases' to open the
0.0	combination	'Combination Cases' table, to add the following:
	case of basic	<ul> <li>"SRSS of basic responses", "SRSS(A20, 0.35A21, 0.35A22)"</li> </ul>
	responses	(- or whatever the response spectrum analysis case turn out to
	responses	be!)
8.9	Footfall	'Data   Cases and Tasks   Analysis Tasks' to open the 'Analysis Tasks
	analysis of	view
	"footfall	• Modal analysis of "footfall model"
	model"	• Right-click on background of Analysis Tasks View; 'New
		Analysis Task'
		• Task name: "Footfall – footfall model"
		• Solver option: 'Dynamic response   Footfall'
		• Analysis stage: "footfall model"
		'Next' to open the 'Analysis Wizard : Response Spectrum Parameters'
		dialog:
		• 'Modal analysis task': (check that modal task is referenced)
		• (leave other settings as default)
		'Next' to open the 'Analysis Wizard : Cases Set Up' dialog:
		• 'Analyse'
		'Finish'
		Analysisatage: 5 : toofall model Scale: 1:174.7
		2
		TrGen_08.gwb
		(without results)
		(without results)
Irea	te Members for	r Design
•	Create RC m	
•	Create steel	members.
9.1	Create RC	'Graphic View (analysis layer)   Select elements' cursor mode 🔮
	members	<ul> <li>Right-click on the Graphic View background; 'Select List';</li> </ul>
		"PB1 to PB10"; OK
		(to select all beam elements that reference concrete sections)
		'Tools   Manipulate Model   Create members from elements'
		• 'Element list': " <current selection="">"</current>
		• Select '1 element = 1 member'
9.2	Create steel	'Graphic View (analysis layer)   Select elements' cursor mode
	members	<ul> <li>Select beam elements that reference steel sections, but not</li> </ul>
		Server seam erements that reference steer sections, but not
		transverse elements

		<ul> <li>'Tools   Manipulate Model   Create members from elements'</li> <li>'Element list': "<current selection="">"</current></li> <li>Select '1 element = 1 member'</li> <li>'Graphic View (analysis layer)   Select elements' cursor mode </li> <li>Select transverse beam elements that reference steel sections</li> <li>'Tools   Manipulate Model   Create members from elements'</li> <li>'Element list': "<current selection="">"</current></li> <li>Select 'Many elements = 1 member'</li> <li>Don't 'Divide members at intersections with columns'</li> </ul>
9.3	Inspect members	'Graphic View (design layer)'
	members	<ul> <li>'Graphic Display' toolbar; 'Shrink'</li> <li>'Graphic Display' toolbar; 'Saction display'</li> </ul>
		Graphic Display' toolbar; 'Section display'
		(without results)

#### **Design Properties**

- Specify design codes.
- Modify design property references and member restraint property references.
- Specify steel design properties.
- Specify steel member restraints.
- Inspect steel design results.
- Specify RC member design properties.
- Inspect RC member design results.
- Specify RC slab design properties
- Inspect RC slab design results.

	mspeernes	lub design results.
10.1	Design	'Data   Specification   Design Specification' to open the 'Design
	codes	Specification' dialog
		• Enter your preferred design codes for steel and concrete
10.2	Modify member design property references and member restraint property	<ul> <li>'Graphic View (design layer)'</li> <li>Label ( ) on elements: property references</li> <li>'Graphic View   Select members' cursor mode</li> <li>Right-click on the Graphic View background; 'Select List'; "PB2 PB3 PB12 PB13"; OK (to select all concrete and steel beam members)</li> <li>Modify the selected members</li> </ul>
	references	Modify selection 💌 ok

		•	'Modify	design	propert	v to' 2			
		• ]	Right-cli	ck on t	· ·	•	background;	'Select Lis	ť;
			"PB13";		-	a1 <b>b</b>	••••••••••••••		
			(to select Modify t				nembers)		
			Modify se			v ok			
		•	'Modify		nt prope				
10.3	Steel design				<u> </u>		Properties' to	open the '	Stee
	properties	Design F	Properties	s' table	e, to add	the follo	wing:		
				rGon '	10 owb	Steel De	sign Properties		
					A		B	<u> </u>	
			Pro	operty	Nan	ne	Grade	0	
			De 1		Design pro Steel colui	nn S27		Ni Ye	
			2		Steel bear	n S27	'5	N	
			The n	ame use	d to refer t	o this desigi:	n property. Press W	Vizard f	
		ties							
		.153	С	D	E	F	G	Н	
				Effecti	ve Lengl	h Lateral	Max Plastic:Elastic	Net:Gross	
			Override		zz	Torsiona		Area Ratio	=
				-	[m]/[%]	[ [m]/[%]			
			No	100%	% 100%	: 100%	1.2	! 1	
			Yes	1002			1.2	1	
			Yes No					1	~
			Yes No	90	\$ 90%		1.2	1	•
			Yes No	90	\$ 90%		1.2	1	•
		ss Wiza	Yes No rd for assist	90%	\$ 90%		1.2 1.2	1	
10.4	Steel	ss Wizar (Use the 'Data   P	Yes No rd for assist wizard	90 ted input to e	enter the gn   Ste	90% properticel Restrain	1.2 1.2 es.) nt Properties'	to open the	.::
10.4	restraint	ss Wizar (Use the 'Data   P	Yes No rd for assist wizard	90 ted input to e	enter the gn   Ste	90% properticel Restrain	1.2 1.2	to open the	.::
10.4		ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	903 ted input to e s   Desi Propert	≈nter the gn   Ste ies' tabl	90% propertie el Restrai e, to add	1.2 1.2 es.) nt Properties' the following:	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	903 ted input to e s   Desi Propert	≈nter the gn   Ste ies' tabl	90% propertie el Restrai e, to add	1.2 1.2 es.) nt Properties'	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	903 ted input to es s   Desi Propert <b>n_10.g</b> A Node /	enter the gn   Ste ies' tabl	90% propertid el Restrai e, to add el Restrai C	1.2 1.2 es.) nt Properties' the following: nt Properties	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	903 ted input to es l Desi Propert <b>0_10.g</b> A Node / Span	enter the gn   Ste ies' tabl	90% propertia el Restrai e, to add el Restrai C bad Height [m]	1.2 1.2 es.) nt Properties' the following: nt Properties D Restraint Desc	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	ed input ted input to es l Desi Propert <b>10.g</b> A Node / Span	enter the gn   Ste ies' tabl <b>wb : Ste</b> <b>B</b> <b>Ref. Lo</b>	90% propertia el Restrai e, to add el Restrai C pad Height [m]	1.2 1.2 es.) nt Properties' the following: nt Properties D Restraint Desc enc	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard properties estraint P	903 ted input to es l Desi Propert <b>0_10.g</b> A Node / Span	enter the gn   Ste ies' tabl	propertia el Restrai e, to add el Restrai C bad Height [m] 0 0	1.2 1.2 es.) nt Properties' the following: nt Properties D Restraint Desc	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	ed input to e to es l Desi Propert <b>10.g</b> A Node / Span Steel R	enter the gn   Ste ies' tabl wb : Ste B Ref. Lo 1 all all est Prop	90% propertie el Restrai e, to add el Restrai C pad Height [m] 0 0 0 0	1.2 1.2 1.2 es.) nt Properties' the following: nt Properties D Restraint Desc enc TP MAJV MINV F1L	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	ed input to e to es l Desi Propert <b>10.g</b> A Node / Span Steel R	enter the gn   Ste ies' tabl wb : Ste B Ref. Lo 1 all all est Prop	90% propertie el Restrai e, to add el Restrai C pad Height [m] 0 0 0 0	1.2 1.2 1.2 es.) nt Properties' the following: nt Properties D Restraint Desc enc TP MAJV MINV F1L	to open the	.::
10.4	restraint	ss Wiza (Use the 'Data   P 'Steel Ro	Yes No rd for assist wizard Properties estraint P	ed input to e to es l Desi Propert <b>10.g</b> A Node / Span Steel R	enter the gn   Ste ies' tabl wb : Ste B Ref. Lo 1 all all est Prop	90% propertie el Restrai e, to add el Restrai C pad Height [m] 0 0 0 0	1.2 1.2 1.2 es.) nt Properties' the following: nt Properties D Restraint Desc enc TP MAJV MINV F1L	to open the	.::

						teel Restraiı				
				A	В	С		D	~	
				Node / Span	Ref.	Load Heighl	Restrain	Descriptio	n	
			Defaults	Node	1	[m]	enc		=	
			1	Node ⊻	end1		TP MAJV N	4INV		
			2	Node	2		F12L TR M			
			3	Node	end2		TP MAJV N	AINV		
						op 2 🖌 Ad <			>	
				eor span	(elemer	nt) position to b	e restrained			
		(Use tl	ne wizard	to e	enter t	he propertie	es)			
10.5	Steel design					sults are av				
	results	•	-		-	gn layer)' co				
			0	'Steel U						
				•	'Ove	erall', 'Loca	l Combir	ned', 'Buc	kling	
					Com	bined',				
		•	'Outpu	t View':						
			0			n Synopsis'				
			0	'Steel U			10 11	11 (D	1 1.	
				•		erall', 'Loca	I Combir	ned <sup>7</sup> , 'Buc	kling	
			2	'Staal I		lbined', n Summary				
			0		•	n Calcs':				
			Ũ	•	-		f'			
0.6	RC member	'Data		٠	'Ver	bose', 'Brie		Properties'	to ope	n tł
0.6	RC member design		Properti	• es   Desi	'Ver	bose', 'Brie C Member	Design F			n tł
0.6			Properti	• es   Desi	'Ver	bose', 'Brie	Design F			n tl
10.6	design	'RC M	Properti Iember D	• es   Desi Design Pr	'Ver Ign   R ropert	bose', 'Brie C Member	Design F			n tł
10.6	design	'RC M	Properti Iember D	• es   Desi Design Pr	'Ver Ign   R ropert	bose', 'Brie C Member ies' table, to gn Properties B C	Design F add the	following	F	n tł
0.6	design	'RC M	Properti Iember D	• es   Desi Design Pr : RC Memb	'Ver ign   R copert	bose', 'Brie C Member ies' table, to gn Properties	Design F add the	following	F Link Diamete	
0.6	design		Properti Iember D en_10.gwb	• es   Desi Design Pr : RC Memb A	'Ver ign   R copert	bose', 'Brie C Member ies' table, to gn Properties B C ype Concrete Grade	Design F add the D Reinforce	following	F Link	r A
0.6	design		Properti Iember D en_10.gwb Its RC Memb RC Memb	es   Desi Design Pr : RC Memb A Name Der Design # Der Design 1	·Ver Ign   F ·opert Der Desi Bea Colu	bose', 'Brie C Member ies' table, to gn Properties B C gpe Concrete Grade m C35 mn C35	Design F add the Beinforce Main 460T 460T	following	F Link Diameter [m] 0.00 0.00	r A
0.6	design		Properti Iember D en_10.gwb Its RC Memb RC Memb	• es   Desi Design Pr : RC Memb A Name	'Ver ign   F copert per Desi Bea	bose', 'Brie C Member ies' table, to gn Properties B C gpe Concrete Grade m C35 mn C35	Design F add the Beinforce Main 460T	following	F Link Diameter [m]	r A
0.6	design	'RC M Prope Defaul 1 2 3 1	I Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt	• es   Desi Design Pr : RC Memb A Name Der Design # Der Design 1 Der Design 2	·Ver ign   R ·opert ·opert Bea Colu Bea	bose', 'Brie C Member ies' table, to gn Properties B C gpe Concrete Grade m C35 m C35 m C35	Design F add the Beinforce Main 460T 460T	following	F Link Diameter [m] 0.00 0.00	r A
0.6	design	'RC M Prope Defaul 1 2 3 1	I Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt	• es   Desi Design Pr : RC Memb A Name Der Design # Der Design 1 Der Design 2	·Ver ign   R ·opert ·opert Bea Colu Bea	bose', 'Brie C Member ies' table, to gn Properties B C gpe Concrete Grade m C35 mn C35	Design F add the Beinforce Main 460T 460T	following E ement Grade Link 460T 460T 460T	F Link Diameter [m] 0.00 0.00	r A
0.6	design	'RC M Prope Defaul 1 2 3 1	I Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt	• es   Desi Design Pr : RC Memb A Name Der Design # Der Design 1 Der Design 2	·Ver ign   R ·opert ·opert Bea Colu Bea	bose', 'Brie C Member ies' table, to gn Properties B C gpe Concrete Grade m C35 m C35 m C35	Design F add the Beinforce Main 460T 460T	following E ement Grade Link 460T 460T 460T	F Link Diameter [m] 0.00 0.00	r A
10.6	design	*RC M	I Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt	• es   Desi Design Pr : RC Memb A Name Der Design # Der Design 1 Der Design 2	·Ver ign   R copert oer Desi Bea Colu Bea Colu Bea	bose', 'Brie C Member ies' table, to gn Properties B C gn Properties B C Grade m C35 m C35 m C35 design property.	Design F add the Beinforce Main 460T 460T	following	F Link Diameter (m) 0.00 0.00 0.00	<b>r A</b>
0.6	design	*RC M	Properti Iember D en_10.gwb rty Its RC Memb RC Memb RC Memb e used to identi	• es   Desi besign Pr : RC Memb A Name Der Design # Der Design 1 Der Design 2 fy this concre	·Ver ign   R copert oer Desi Bea Colu Bea Colu Bea	bose', 'Brie C Member ies' table, to gn Properties B C gn Concrete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T	following E ement Grade Link 460T 460T 460T 460T M Bar Patter	F Link Diameter (m) 0.00 0.00 0.00	<b>r A</b>
0.6	design	*RC M Prope Defaul 1 2 3 1 The name F nk meter m]	I Properti Iember D en_10.gwb nty RC Memb RC MC	• es   Desi besign Pr : RC Memb A Name Der Design 1 ber Design 2 fy this concre H Top	·Ver gn   R oper Desi Per Desi Bea Colu Colu Col	bose', 'Brie C Member ies' table, to gn Properties B C gne Concrete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T 460T 460T	following	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	
0.6	design	*RC M Prope Defaul 1 2 3 1 The name F nk meter	I Properti Iember D en_10.gwb nty Its RC Ment RC Ment RC Ment e used to identi	• es   Desi besign Pr : RC Memb A Name Der Design # Der Design 1 Der Design 2 fy this concre H	·Ver ign   R copert ber Desi Bea Colu Colu	bose', 'Brie C Member ies' table, to gn Properties B C gn Correte gn Correte gn Correte gn Correte gn Correte Grade Correte gn Correte Grade Correte gn Correte Grade Correte gn Correte Grade Correte gn Correte gn Correte Grade Correte gn Correte Grade Correte Corre	Design F add the Reinforce Main 460T 460T 460T	following E ement Grade Link 460T 460T 460T 460T M Bar Patter	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	<b>r A</b>
0.6	design	<pre>'RC M</pre> <pre>'RC M</pre> <pre>Prope</pre> Defaul 1 <pre>2</pre> <pre>3</pre> 1< 3 1< The name F nk <pre>meter m] 0.008</pre>	I Properti Iember D en_10.gwb nty RC Memb RC MC	es   Desi pesign Pr : RC Memb A Name per Design 1 per Design 2 fy this concre H Top [m] 0.04	·Ver gn   R copert · · · · · · · · · · · · · · · · · · ·	bose', 'Brie C Member ies' table, to gn Properties B C gn Corcete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T 460T 460T 460T 460T	following E ment Grade Link 460T 400 40 400	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00	
10.6	design	<sup>c</sup> RC M <sup>m</sup> TrG <sup>r</sup>	Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt e used to identi G Aggregate [m] 0.025 0.025	es   Desi pesign Pr : RC Memb A Name per Design 1 per Design 2 fy this concre H Top [m] 0.04 0.04		bose', 'Brie C Member ies' table, to gn Properties B C gn Corcete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T 460T 460T 460T 460T 460T	following	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00	
10.6	design	<sup>c</sup> RC M <sup>m</sup> TrG <sup>r</sup>	Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt e used to identi G Aggregate [m] 0.025 0.025	es   Desi pesign Pr : RC Memb A Name per Design 1 per Design 2 fy this concre H Top [m] 0.04 0.04		bose', 'Brie C Member ies' table, to gn Properties B C gn Corcete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T 460T 460T 460T 460T 460T	following	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00	
10.6	design	<sup>c</sup> RC M <sup>m</sup> TrG <sup>r</sup>	Properti Iember D en_10.gwb Its RC Memt RC Memt RC Memt e used to identi G Aggregate [m] 0.025 0.025	es   Desi pesign Pr : RC Memb A Name per Design 1 per Design 2 fy this concre H Top [m] 0.04 0.04		bose', 'Brie C Member ies' table, to gn Properties B C gn Corcete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T 460T 460T 460T 460T 460T	following	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00	
10.6	design	• RC M Prope Defau 1         2         3         1         2         3         1         2         3         1 The name F         nk         meter         m]         0.008	Properti Iember D en_10.gwb rty RC Memt RC Memt RC Memt e used to identi G Aggregate [m] 0.025 0.025	es   Desi es   Desi perign Pr RC Memb A Name per Design 1 per Design 2 fy this concre H Top [m] 0.04 0.04 0.04 0.04	Ver     ign   R     copert     ter Desi     I     Bea     Colu     Bea     Colu     Bea     Colu     Bea     Colu     Bea     Colu	bose', 'Brie C Member ies' table, to gn Properties B C ype Concrete Grade m C35 m C35 m C35 design property.	Design F add the Reinforce Main 460T 460T 460T 460T 460T 460T 460T 460T	following  Ement Grade Link  460T  460T  460T  460T  460T  50  M Bar Patter  0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	F Link Diameter (m) 0.00 0.00 0.00 0.00 0.00	
10.6	design properties	<pre>'RC M 'RC M 'RC M ' 'RC M ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' '</pre>	Properti Iember D Iember D Iember D Its RC Memb RC Mem	• es   Desi besign Pr : RC Memb A Name Der Design 1 Der Design 1 per Design 2 fy this concre H Top [m] 0.04 0	·Ver ign   R oper Desi Per Desi Bea Colu Colu Bea Colu Bea Colu Col	bose', 'Brie C Member ies' table, to gn Properties B C ype Concrete Grade m C35 m C35 m C35 design property. Left Rig [m] [m] 0.04 (0) 0.04 (0	Design H add the Reinforca Main 460T	foÎlowing          E         ement Grade         Link         460T         460T         460T         460T         Bar Patter         D         0       0         0       0	F Link Diameter 0.00 0.00 0.00 0.00	
0.6	design properties	<pre>'RC M 'RC M 'RC M ' 'RC M ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' '</pre>	Properti Iember D Iember D Iember D Its RC Memb RC Mem	• es   Desi besign Pr : RC Memb A Name Der Design 1 Der Design 1 per Design 2 fy this concre H Top [m] 0.04 0	·Ver ign   R oper Desi Per Desi Bea Colu Colu Bea Colu Bea Colu Col	bose', 'Brie C Member ies' table, to gn Properties B C ype Concrete Grade m C35 m C35 m C35 design property.	Design H add the Reinforca Main 460T	foÎlowing          E         ement Grade         Link         460T         460T         460T         460T         Bar Patter         D         0       0         0       0	F Link Diameter 0.00 0.00 0.00 0.00	
	design properties	<pre>'RC M 'RC M 'RC M ' 'RC M ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' ' '</pre>	Properti Iember D Iember D Iember D Its RC Memb RC Mem	• es   Desi besign Pr : RC Memb A Name Der Design 1 Der Design 1 per Design 2 fy this concre H Top [m] 0.04 0	·Ver ign   R oper Desi Per Desi Bea Colu Colu Bea Colu Bea Colu Col	bose', 'Brie C Member ies' table, to gn Properties B C ype Concrete Grade m C35 m C35 m C35 design property. Left Rig [m] [m] 0.04 (0) 0.04 (0	Design H add the Reinforca Main 460T	foÎlowing          E         ement Grade         Link         460T         460T         460T         460T         Bar Patter         D         0       0         0       0	F Link Diameter 0.00 0.00 0.00 0.00	

## 10.8 RC slab design

'Data | Properties | Design | RC Slab Design Properties' to open the 'RC Slab Design Properties' table, to add the following:

		A	В	С	D	E	F	G	
			Reinford	-			to-Surface D	_	
			Direction		A	сшен аль	IC-JUNACE L		0
	Property	Name	Local		T	D_U	T	D_U	
			Α	В	Тор	Bottom	Тор	Bottom	<b>.</b>
			[*]	[*]	[m]	[m]	[m]	[m]	
	Defaults	RC Slab Prop #	0	90	0.035	0.035	0.055	0.055	
	1	RC Slab Prop 1	0	90 90	0.035	0.035	0.055	0.055	
	2	RC Slab Prop 2 RC Slab Prop 3	90	90 180	0.035	0.035	0.055 0.055	0.055	ö
	4	THE SIGDT TOP 5		100	0.035	0.033	0.033	0.033	140
	<								
	The parce i	ised to refer to th	oic property	. Use the	Winord to belo	coocifu dota	iled properties		
	The Halle C		iis propore,	, , oso alo	meana comoip	opeen y deca		•	
								_	
		н		1	J		ĸ		
				•	0			<u>^</u>	
		Overrid	-	Slab		. Bein	forcement		
		- Analysi	<sup>IS</sup>   ТЫ.	ckness	Concrete G	rade	Grade		
								Ξ.	
				[m]					
		155 No			User Defined		)efined		
		155 No			C60/75	B500E			
		155 No			C60/75	B500E			
		155 No		0.3	C60/75	B500E		~	
							>		
	(Use the	wizord X	to ont	or the	proportio	, )			
		wizard							
	(One RO	C slab desig	gn prop	erty pe	er 2D eler	nent proj			
	(One RO	C slab desig	gn prop	erty pe	er 2D eler	nent proj		axes, so	
	(One RO (Reinfor	C slab desig rcement dir	gn prop rections	erty pe s are w	er 2D eler ith respec	nent proj t to elen	nent local		
	(One RO (Reinfor inspect	C slab desig rcement dir 'element ax	gn prop rections	erty pe s are w	er 2D eler ith respec	nent proj t to elen	nent local		
	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element ax ns.)	gn prop rections kes' lab	erty person of the series are work of the series to be se	er 2D eler ith respec understan	nent pro t to elem d proper	nent local ty 3 reinf		
slab	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element ax	gn prop rections kes' lab	erty person of the series are work of the series to be se	er 2D eler ith respec understan	nent pro t to elem d proper	nent local ty 3 reinf		
	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element ax ns.) owing RC	gn prop ections kes' lab slab de	erty person of the service of the se	er 2D eler ith respec understan	nent prop t to elem d proper available	ty 3 reinf		
sign	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element av ns.) owing RC 'Graphic V	gn prop rections kes' lab slab de View (ar	erty pa s are w bels to sign re nalysis	er 2D eler ith respec understan esults are a s layer)' d	nent prop t to elem d proper available iagrams:	ty 3 reinf		
ign	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element av ns.) owing RC 'Graphic V	gn prop rections kes' lab slab de view (ar C Slab	erty person of the second seco	er 2D eler ith respec understan sults are a layer)' d prcement'	nent prop t to elem d proper available iagrams:	ty 3 reinf		
sign	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element av ns.) owing RC 'Graphic V	gn prop rections kes' lab slab de view (ar C Slab	erty person of the second seco	er 2D eler ith respec understan esults are a s layer)' d	nent prop t to elem d proper available iagrams:	ty 3 reinf		
sign	(One RO (Reinfor inspect direction	C slab desig recement dir 'element ax ns.) owing RC 'Graphic V o 'R	gn prop rections kes' lab slab de liew (an C Slab	erty person of the service of the se	er 2D eler ith respec understan esults are = 3 layer)' d prcement' Bottom'	nent prop t to elem d proper available iagrams:	ty 3 reinf		
C slab sign sults	(One RO (Reinfor inspect direction	C slab desig reement dir 'element ax ns.) owing RC 'Graphic V o 'R 'Graphic V	gn prop rections kes' lab slab de View (an C Slab	erty person sign re- sign re- nalysis Reinfo Top', ' nalysis	er 2D eler ith respec understan sults are a slayer)' d precement' Bottom' slayer)' c	nent prop t to elem d proper available iagrams: : ontours:	ty 3 reinf		
sign	(One RO (Reinfor inspect direction	C slab desig reement dir 'element ax ns.) owing RC 'Graphic V o 'R 'Graphic V	gn prop ections kes' lab slab de View (a) C Slab View (a) C Slab	s are w bels to sign re nalysis Reinfo Top', ' nalysis Reinfo	er 2D eler ith respec understan sults are i s layer)' d preement' Bottom' s layer)' c preement'	nent proj t to elem d proper available iagrams: : ontours: :	nent local ty 3 reinf	orcemen	
ign	(One RO (Reinfor inspect direction	C slab desig reement dir 'element ax ns.) owing RC 'Graphic V o 'R 'Graphic V	gn prop ections kes' lab slab de View (a) C Slab View (a) C Slab	s are w bels to sign re nalysis Reinfo Top', ' nalysis Reinfo	er 2D eler ith respec understan sults are a slayer)' d precement' Bottom' slayer)' c	nent proj t to elem d proper available iagrams: : ontours: :	nent local ty 3 reinf	orcemen	
gn	(One RO (Reinfor inspect direction	C slab desig rcement dir 'element ax ns.) owing RC 'Graphic V o 'R 'Graphic V o 'R	gn prop ections (estillations) (esti	s are w bels to sign re nalysis Reinfo Top', ' nalysis Reinfo	er 2D eler ith respec understan sults are i s layer)' d preement' Bottom' s layer)' c preement'	nent proj t to elem d proper available iagrams: : ontours: :	nent local ty 3 reinf	orcemen	
gn	(One RO (Reinfor inspect direction	C slab desig reement dir 'element ax ns.) owing RC 'Graphic V o 'R 'Graphic V o 'R 'Output Vi	gn prop rections kes' lab slab de View (ar C Slab View (ar C Slab ew':	erty pe s are w bels to sign re nalysis Reinfo Fop', ' nalysis Reinfo Fop', '	er 2D eler ith respec understan sults are i s layer)' d preement' Bottom' s layer)' c preement'	nent prop t to elem d proper available iagrams: : ontours: : direction	nent local ty 3 reinf	orcemen	



--- 000 ----

GSA Step Thru' Tutorial – 'General Use of GSA' training exercise