Oasys GSA 'Step Through' Tutorial:

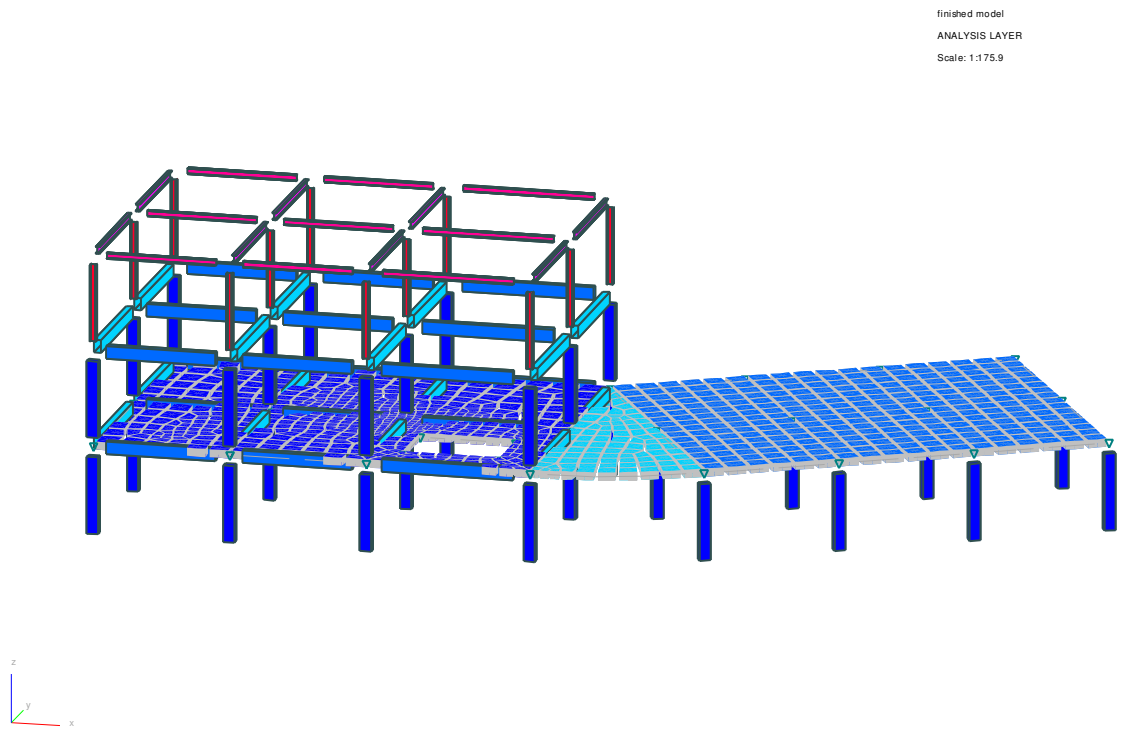
'General Use of GSA' training exercise

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)
- 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 >.



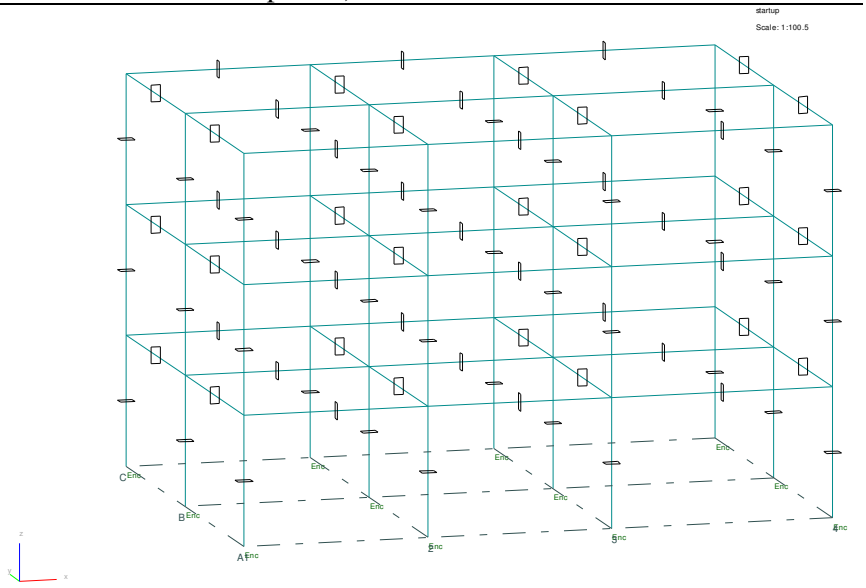
The Finished Model
TrGen_10.gwb

#	Step	Notes

Create orthogonal frame

- Set up grid lines.
- Generate an orthogonal frame based on the grid lines.

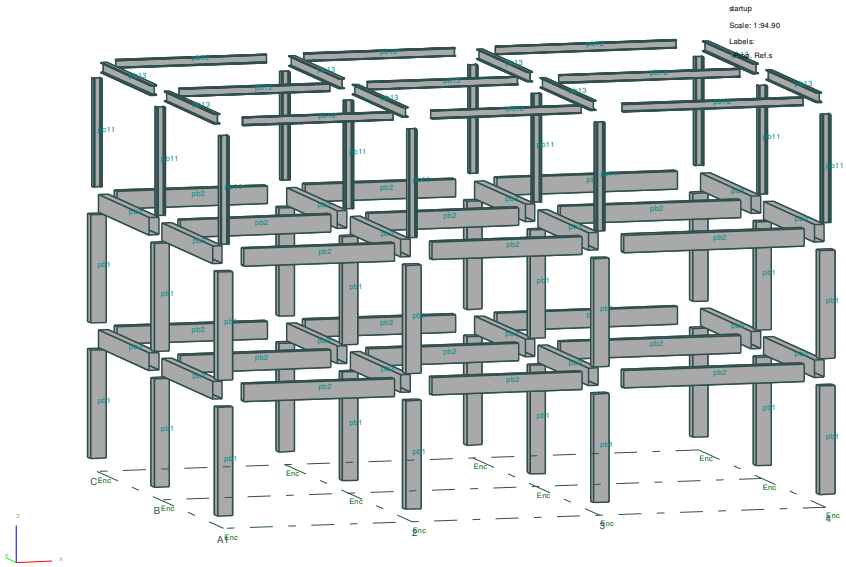
1.1	Enable advanced features	'Tools | Preferences | Advanced Features' 'Enable All' (Many advanced features will be used during this exercise.)
1.2	Open new model	'File | New'
1.3	Structure type	'New Model Wizard : Structure type | Type' : 'Space' (Allows mixed 1D and 2D elements.)
1.4	Grid lines	'New 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: o A Line 0 0 16 0 • Add 2, copying line A, incrementing in Y by 6 o B Line 0 6 16 0 o C Line 0 12 16 0 (Resulting in: o 1 Line 0 0 12 90 o 2 Line 5 0 12 90 o 3 Line 10 0 12 90 o 4 Line 16 0 12 90 o A Line 0 0 16 0 o B Line 0 6 16 0 o C Line 0 12 16 0)
1.5	Generate beam elements	'New Model Wizard : Structure type | Generate data for the selected structure type' to open 'Data Generation Wizard : Orthogonal frame' 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 generated data	'Graphic View' • Label ( ) on nodes: restraints • Label ( ) on elements: section shapes • Draw grid
1.7	Save view	'View | Save Graphic View' • Saved view name: "startup"

		(This view will be displayed in the initial Graphic View when the model is opened.)
		 TrGen_01.gwb

Make top floor a steel frame

- Modify the property references for the top floor elements.
- Specify new section properties.

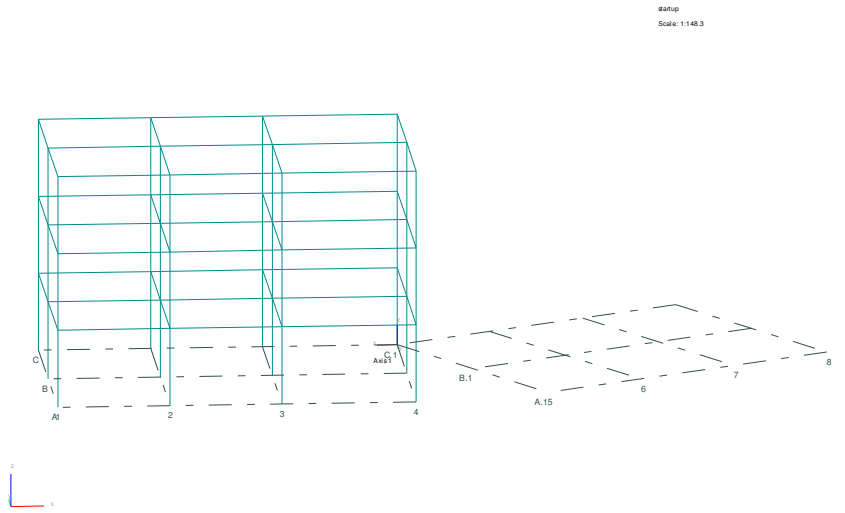
2.1	Modify top floor properties	'Graphic View' • Label ( ) on elements: property references 'Graphic View | Select elements' cursor mode • Select top level of columns (Drag from right to left to select elements even partly in drag rectangle.) • Right-click on the Graphic View background; 'Modify Selected Elements' • Modify columns to property 11 • Select longitudinal beams (Shft+select appends to selection.) • Modify longitudinal beams to property 12 (The 'Command' toolbar control Modify selection ok offers the last used sculpt command.) • Select transverse beams (Ctrl+select inverts selection.) • Modify transverse beams to property 13
2.2	Specify steel sections for top floor	'Graphic View', any cursor mode (May as well be 'Rotate' cursor mode:) • Right-click on any top floor column; 'Edit | Edit Property' • Specify steel column • Do similar for longitudinal and transverse beams
2.3	View the modified data	'Graphic View' • 'Graphic Display' toolbar; 'Shrink' • 'Graphic Display' toolbar; 'Section display'

		 TrGen_02.gwb

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.

3.1	Reset display	'Graphic View' • '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 set at node 12	(We need an axis about which we can rotate grid lines.) 'Graphic View' • 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 grid lines	'Data | Grid Lines' to open 'Grid Line Definition' dialog; assign grid 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.

		 TrGen_03.gwb

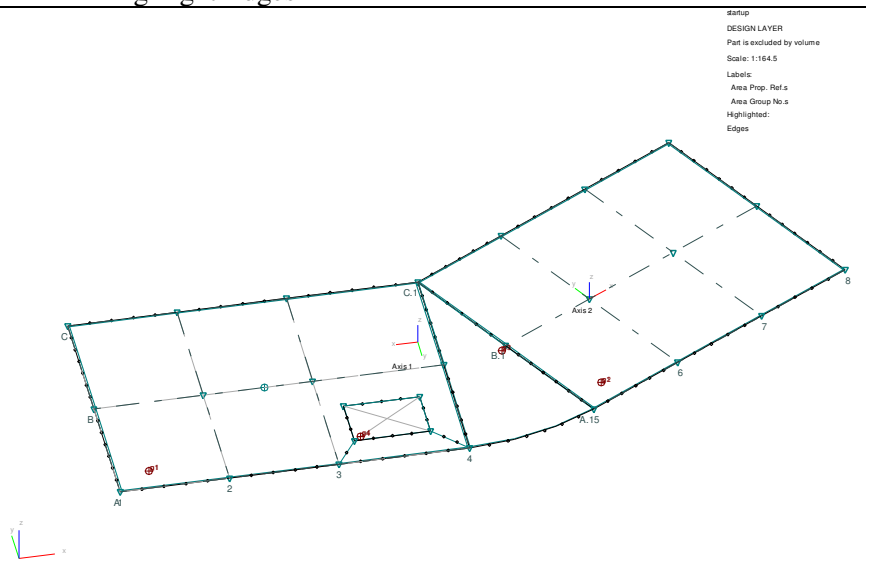
Create slab region

- 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"	Either: 'Data | Define Current Grid' to open 'Current Grid Definition' dialog • Specify 'Grid plane' as "1st floor" Or: 'Graphic View' (any cursor mode) • Right-click on any node at 1st floor level; 'Set Current Grid to This' (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 volume of "1st floor" only	'Graphic View' (any cursor mode) • 'Orientation' toolbar; 'Y elevation' or <Y> 'Graphic View | Volume' cursor mode • Drag left to right around 1st floor (For clarity while sculpting.) 'Graphic View' (any cursor mode) • 'Orientation' toolbar; 'Plan' or <P>
4.3	Set Graphic View to Design Layer	'Graphic View' (any cursor mode) • Right-click on the Graphic View background; 'Switch layer' (Switches to 'Design Layer' - pink background.) (Members, lines, areas and regions exist on the Design Layer)
4.4	Add area defining slab boundary	Display grid, and snap to grid points • 'Sculpt' toolbar; 'Grid' • 'Sculpt' toolbar; 'Snap to grid points' '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.)

		• Right-click on new line; 'Set Line Defaults as This' (Now new lines will default to 'Number of segments' = 4.) • 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.) • On closing the loop, accept the offer to add an area (Area 1 is created, referencing the loop of lines just closed.) • Create areas 2 and 3 defining the slab extents for the skew wing and interconnecting segment, respectively
4.5	Modify line to arc	Switch off snap to grid points • 'Sculpt' toolbar; 'Snap to grid points'  'Graphic View | Add Nodes Sculpt Tool' cursor mode  • Create a node 'outside' arc at approx. (20.0,-1.0,3.5) to create node 58 'Graphic View' (any cursor mode) • Right-click on line that is to be an arc; 'Edit Line' • 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.)
4.6	Create hole in slab	Ensure 'Snap to grid points' is off • 'Sculpt' toolbar; 'Snap to grid points'  'Graphic View | Add Lines Sculpt Tool' cursor mode  • Create four lines bounding hole, vertices at approx.: ○ (X = 11.00m Y = 1.500m Z = 3.500m) ○ (X = 14.50m Y = 1.500m Z = 3.500m) ○ (X = 14.50m Y = 4.000m Z = 3.500m) ○ (X = 11.00m Y = 4.000m Z = 3.500m) • Either: on closing the loop, accept the offer to add an area • Or: select lines and 'Sculpt | Geometric Entity Operations | Add Area' 'Graphic View' (any cursor mode) • 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.) • In 'Area Definition' dialog, set 'Type' to 'Void'
4.7	Add missing internal nodes	Switch on snap to grid points • 'Sculpt' toolbar; 'Snap to grid points'  'Graphic View | Add Nodes Sculpt Tool' cursor mode  • 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.)
4.8	Add region	'Graphic View | Select nodes' cursor mode  • Select the internal nodes 'Graphic View | Select areas' cursor mode  • Ctrl+A to select all areas

		• '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 Properties	(Generated 2D elements are assigned the property and group number based on those set for the respective area. Areas created in sculpt default to having the property and group set to the area number.) 'Graphic View' • Label () 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 • Select nodes 56, 63, 64 • (See section 3.2 above) 'Data | Properties | 2D Element Properties' to open the '2D Element Properties' table. • Enter three properties: o Axis: 'Global', axis 2, axis 1 (For the original wing, skew wing and interconnecting segment, respectively.) o Type: 'Flat Shell' (i.e. in-plane and bending action.) o Material: 'Concrete long term' o Thickness: 0.3m
4.10	Save "1st floor column positions" node list	(This list will be used after the mesh has been generated. It's easier to select the list before the mesh is generated.) 'Graphic View | Select nodes' cursor mode  • Select all nodes at 1st floor level column positions • (All at 1st floor except those defining the void and third node of arc.) • Right-click on the Graphic View background; 'Save Selection as List'
4.11	Generate mesh	Either: 'Tools | Region Mesh Generation | Generate 2D Mesh for 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 adjust the mesh	(It is preferable to adjust the condition of the mesh by adjusting the parameters that define the mesh, rather than by adjusting the generated 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.) o Right-click on region; 'Edit Region' • Adjusting the number of segments on lines in the vicinity of the void improves the mesh. o Either: 'Graphic View', label () on geometric

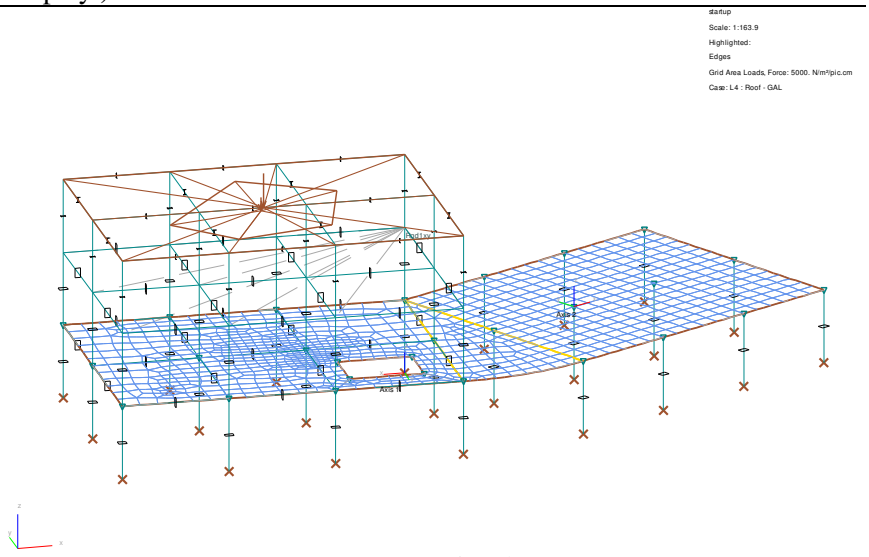
		entities: On lines: Dots along lines ○ Or: 'Display Favourites' toolbar; 'Label node dots'  - to get an idea. • A quick fix is to set the step size consistently for all lines (instead of the number of elements along each line). ○ Right-click on a line; 'Edit Line' ○ In 'Line Definition' dialog, set 'Mesh by step size' and set the 'Step size' = 1; OK ○ Right-click on same line; 'Set Line Defaults as This' ○ Select all lines (Ctrl+A) ○ 'Sculpt | Apply Defaults to Selection' • 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) • Following a mesh generation it is advisable to check the connectivity of the mesh by 'Graphics | Graphic Settings | Highlight Edges'
		 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 View to Analysis Layer	'Graphic View' (any cursor mode) • Right-click on the Graphic View background; 'Switch layer' (Switches to 'Analysis Layer' - yellow background.) (Elements exist on the Design Layer)
5.2	View whole model	(The view is currently of a defined volume.) Either: 'Graphic View | Volume' cursor mode  • Click anywhere in the view Or: Graphic View 'Lists' toolbar; 'Reset to All Entities'  (Not enabled when all entities are being displayed.)

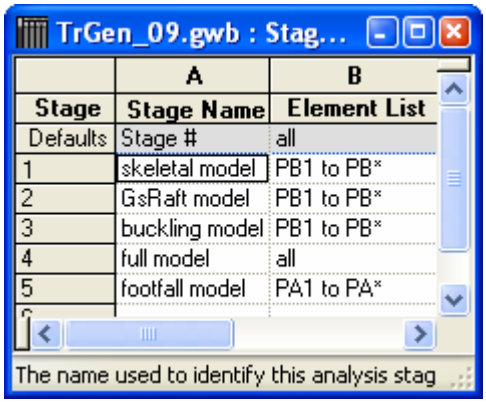
5.3	Extrude columns	'Graphic View | Select nodes' cursor mode • Select the nodes at which we want new columns to support the skew wing of the slab 'Sculpt | Extrude Selection' • 'Direction of extrusion': 'Global z • 'Number of increments': 1 • 'Increment length': -3.5 • Set 'Include Beam elements along extrusion' 'Graphic View | Select elements' cursor mode • Select the new elements 'Graphic View' • Label ( ) on elements: element x axis (New columns are pointing downwards.) • 'Sculpt | Flip Elements' • Label ( ) on elements: section shapes (New columns need to be rotated.) • 'Sculpt | Modify Selection' • 'Modify orientation angle' to 30 degrees
5.4	Assign rigid constraint to 2nd floor	'Graphic View | Select nodes' cursor mode • Select the nodes on the 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 titles	'Data | Cases and Tasks | Load Case Titles' to open the 'Load Case 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 slab	'Data | Loading | 2D Element Loading | Face Loads' to open the '2D Element Loading | Face Loads' table, to add the following: • Element list PA1 to PA*, load case 2, in the global Z direction, value -10kN/m2
5.7	'Plane' grid area loads	'Data | Loading | Grid Loading | Grid Area Loads' to open the 'Grid Loading | Grid Area Loads' table, to add the following: • Grid plane "2nd floor", Area type "Plane", load case 3, in the global Z direction, value -10kN/m2 • Grid plane "3rd floor", Area type "Plane", load case 4, "= =" (to copy the rest of the row from the row above)
5.8	Set current grid to "3rd floor"	(We're about to define a polyline on the roof. Polylines are defined on the current grid plane.) 'Graphic View' (any cursor mode) • Right-click on any node at roof level; 'Set Current Grid to This'
5.9	'Polyline' grid area load	Ensure grid is displayed, and 'snap to grid points' is switched off • 'Sculpt' toolbar; 'Grid' • 'Sculpt' toolbar; 'Snap to grid points'

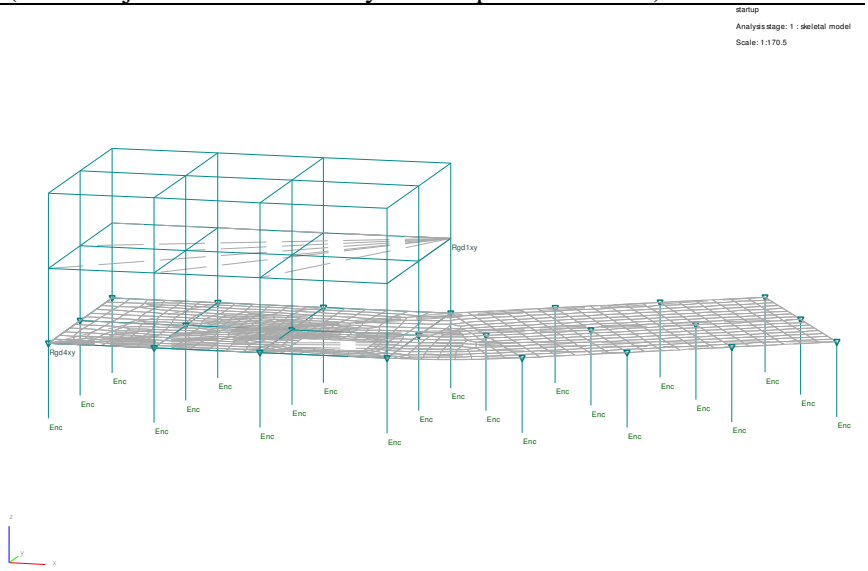
		‘Graphic View	Polyline’ cursor mode  • Click on roof to form vertices of polyline (The polyline will define the area that is to be loaded.) (In this context the polyline will be assume to be closed, so you don’t need to click on the start point to close the polyline.) • Right-click on the Graphic View background; ‘Create Grid Loading	Create Grid Area Load’; enter: o Load case 4, global Z direction, value -5kN • <Esc> to clear the polyline
5.10	Wind load	‘Graphic View	Select nodes’ cursor mode  • Either: Select the nodes at X=0 at the roof level o Or: right-click on the Graphic View background; ‘Select List’ and enter ‘Y37’; OK • Right-click on the Graphic View background; ‘Create Nodal Loading	Create Node Load’; enter: o Load case 5, global X direction, value 20kN
5.11	Check the loading	‘Graphic View’ • On the ‘Display Favourites’ toolbar; ‘All load diagrams’  • On the Graphic View ‘Lists’ toolbar; ‘Next case’  (To step through the load cases.)
5.12	Combination case	‘Data	Cases and Tasks	Combination Cases’ to open the ‘Combination Cases’ table, to add the following: • “Design”, “1.4A1 + 1.6A2 + 1.6A3” (A combination case is a means of combining analysis results. The combining occurs post-analysis, at the time the results are required for display.)
		 TrGen_05.gwb

Analysis stages and linear static analysis

- Set up several analysis stages.
- Assign different restraint and constraint conditions per stage.
- Perform a linear static analysis of an analysis stage.

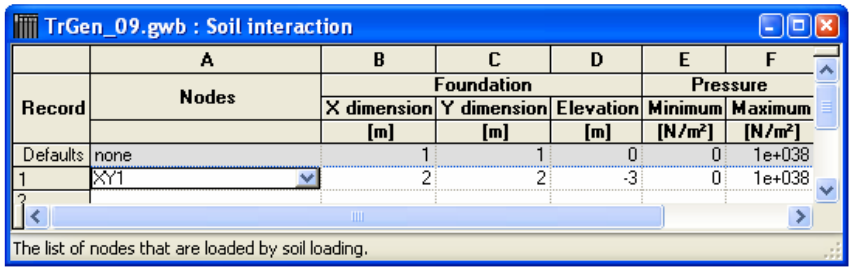
6.1	Reset display	‘Graphic View (analysis layer)’ • ‘Graphic Display’ toolbar; ‘Reset display adornments’ 
6.2	Analysis stages	‘Data	Analysis Stages	Stage Definition’ to open the ‘Stage Definition’ table, to add the following:

		
6.3	Different restraints per stage	'Graphic View (analysis layer)' • : 'Display Favourites' toolbar; 'Label restraints' 'Graphic View | Select nodes' cursor mode • Select all restrained nodes • Right-click on the Graphic View background; 'Modify Selected Nodes' • Restraint type: 'Free'; OK • <Esc> to clear the selection '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 rigid constraints per stage	'Graphic View (analysis layer)' • Label ( ) on nodes: Rigid constraints 'Data | Constraints | Rigid Constraints' to open the 'Rigid Constraints' 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 • Apply to stages "skeletal model" and "GsRaft model" rigid 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 stages	'Graphic View (analysis layer)' • On the Graphic View 'Lists' toolbar, 'Stages', Display Stage 1 : skeletal mode • 'Next display' (To step through the stages.)
6.6	Linear static analysis of "full model"	'Analysis | New Analysis Task' to open the 'Analysis Wizard : Solver Option' dialog: • 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'

		‘Finish’ (To initiate the analysis.) ‘Data	Cases and Tasks	Analysis Tasks’ to open the ‘Analysis Tasks’ view (The task just created and analysed is represented here.)
		 TrGen_06.gwb (without results)

Raft analysis

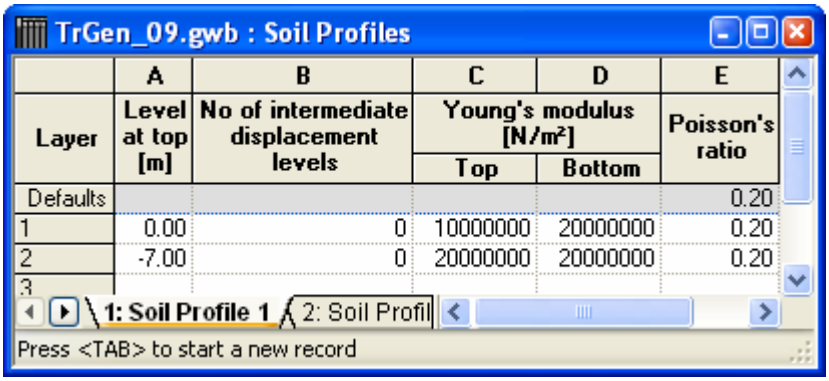
- Set up soil interaction data to specify the relationship between the soil (Vdisp data) and structure (GSA data).
- Set up the Vdisp data to define the soil model.
- Specify the Vdisp analysis parameters.
- Perform a raft analysis.

7.1	Reset display	‘Graphic View (analysis layer)’ • ‘Graphic Display’ toolbar; ‘Reset display adornments’
7.2	Delete results	‘GSA’ toolbar; ‘Erase all results’ (Results need to be erased to allow raft data to be added.) (Deleting the results does not delete the analysis tasks.)
7.3	Soil interaction nodes	‘Data	Raft	Soil Interaction’ to open the ‘Soil Interaction’ table, to add the following: 
7.4	Soil profiles	‘Data	Raft	Vdisp Data	Soil Profiles’ to open the ‘Soil Profiles’ table, to add the following:

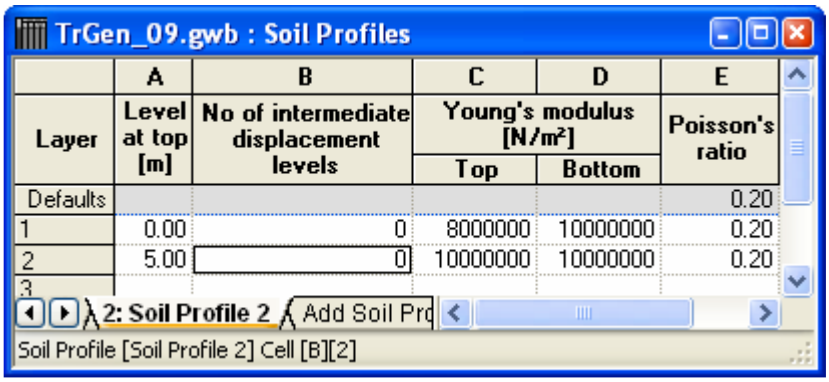
TrGen_09.gwb : Soil interaction

Record	A	B	C	D	E	F
	Nodes	Foundation			Pressure	
		X dimension	Y dimension	Elevation	Minimum	Maximum
		[m]	[m]	[m]	[N/m²]	[N/m²]
Defaults	none	1	1	0	0	1e+038
1	XY1	2	2	-3	0	1e+038
2						

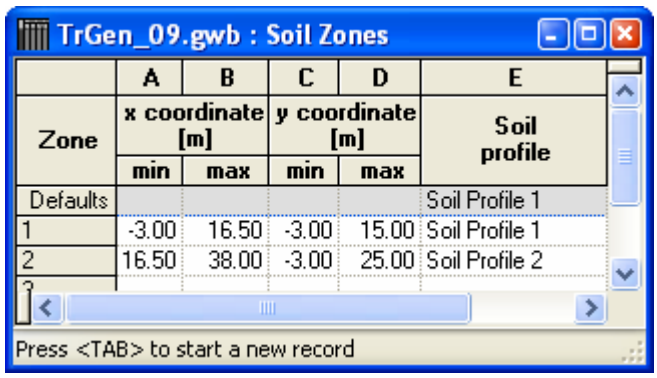
The list of nodes that are loaded by soil loading.



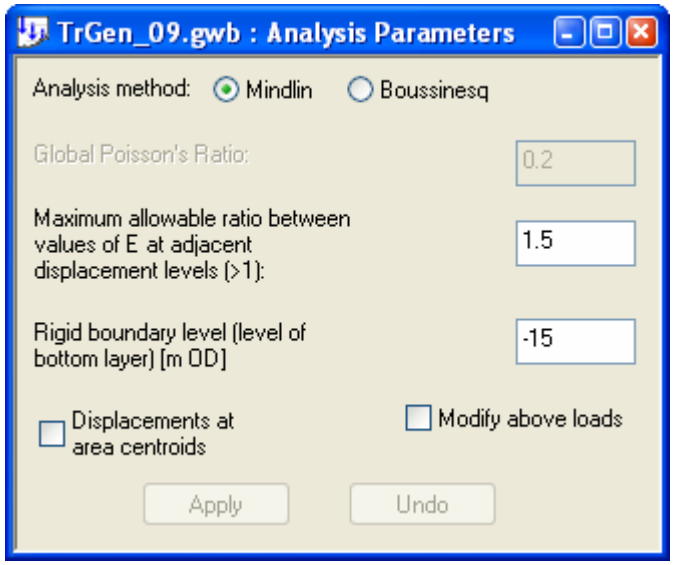
- and:

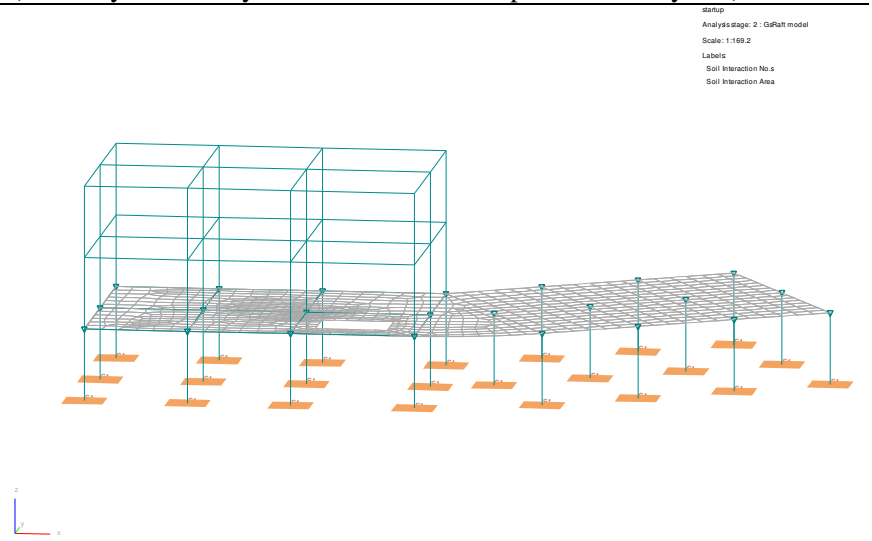


7.5	Soil zones	'Data | Raft | Vdisp Data | Soil Zones' to open the 'Soil Zones' table, to add the following:



7.6	Vdisp analysis parameters	'Data | Raft | Vdisp Data | Analysis Parameters' to open the 'Analysis Parameters' dialog:

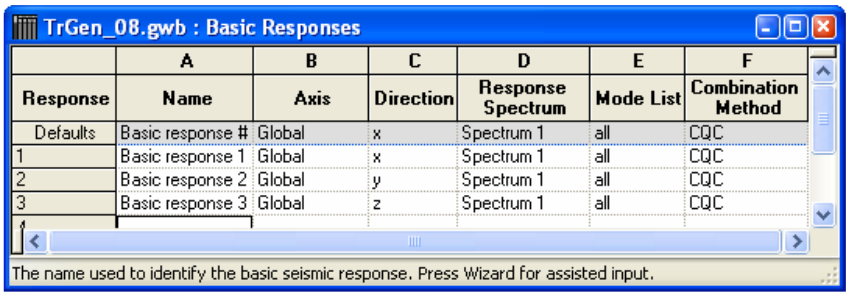


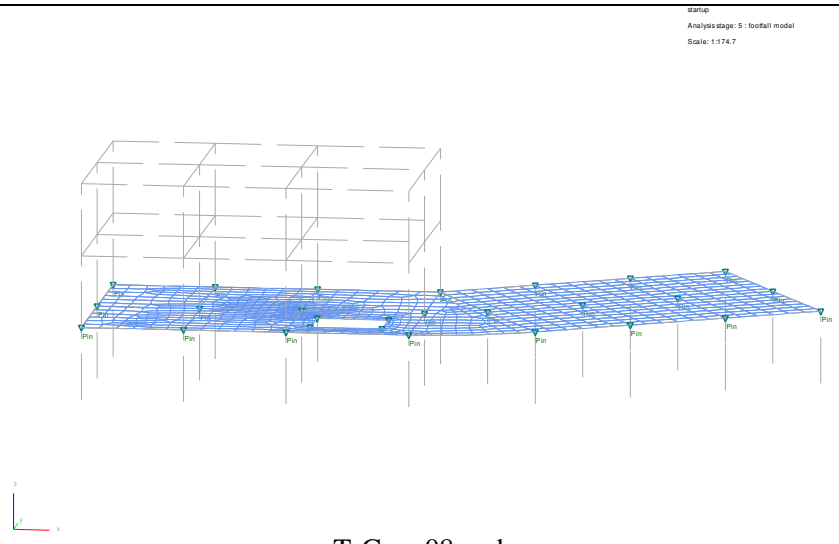
7.7	Raft analysis of 'GsRaft model"	'Analysis | New Analysis Task' to open the 'Analysis Wizard : Solver Option' dialog: • Task name: "Raft – GsRaft model" • Solver option: 'Raft' • Analysis stage: "GsRaft model" 'Next' to open the 'Analysis Wizard : GsRaft Control' dialog: (Default settings.) 'Next' 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.)
		 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.

8.1	Static analysis of "buckling model" task	'Data | Cases and Tasks | Analysis Tasks' to open the 'Analysis Tasks' view • Right-click on background of Analysis Tasks View; 'New Analysis Task' (Alternative to 'Analysis | New Analysis Task'.) • Task name: "Linear Static - buckling model" • Solver option: 'Linear Static' • Analysis stage: "buckling model" 'Next' to open the 'Analysis Wizard : Static Analysis Cases' dialog: • 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.) 'Next' to open the 'Analysis Wizard : Cases Set Up' dialog: • 'Don't analyse now' 'Finish'
8.2	Static P-δ analysis of "buckling model" task	'Data | Cases and Tasks | Analysis Tasks' to open the 'Analysis Tasks' view • Right-click on background of Analysis Tasks View; 'New Analysis Task' • Task name: "Static P-delta – buckling model" • Solver option: 'Static P-delta' • Analysis stage: "buckling model" 'Next' to open the 'Analysis Wizard : GSS Static P-delta' dialog: • Select 'P-delta analysis with each analysis case defining its own differential stiffness' 'Next' to open the 'Analysis Wizard : Static Analysis Cases' dialog: • Add 'Name': "P-delta"; 'Description': "L1 + L5" 'Next' to open the 'Analysis Wizard : Cases Set Up' dialog: • 'Don't analyse now' 'Finish'
8.3	Buckling analysis of "buckling model" task	'Data | Cases and Tasks | Analysis Tasks' to open the 'Analysis Tasks' view • Right-click on background of Analysis Tasks View; 'New Analysis Task' • Task name: "Modal Buckling – buckling model" • Solver option: 'Buckling | Modal' • Analysis stage: "buckling model" 'Next' to open the 'Analysis Wizard : GSS Buckling Parameters' dialog: • 'Number of modes': 6 • 'P-delta analysis case': "L1 + L5" • 'Maximum no. iterations': 128 'Next' to open the 'Analysis Wizard : Cases Set Up' dialog:

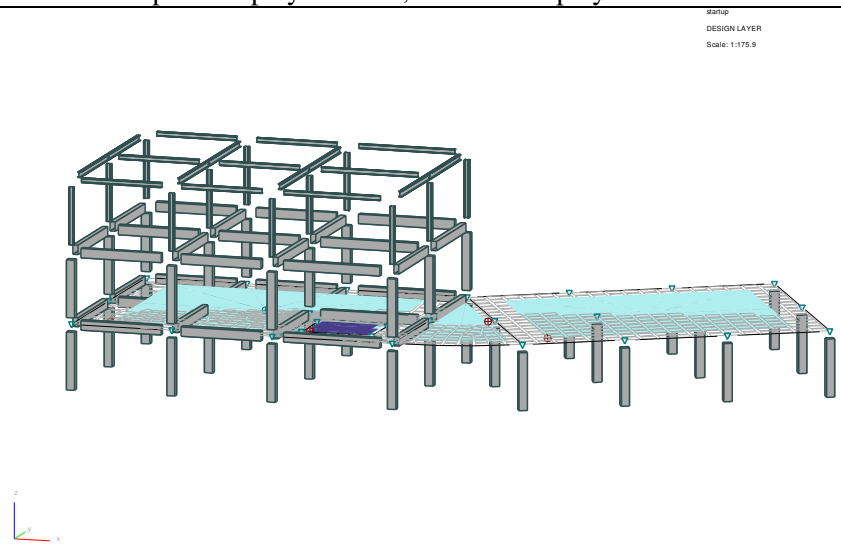
		• ‘Don’t analyse now’ ‘Finish’
8.4	Modal analysis of “full model” task	‘Data	Cases and Tasks	Analysis Tasks’ to open the ‘Analysis Tasks’ view • Right-click on background of Analysis Tasks View; ‘New Analysis Task’ • Task name: “Modal– full model” • Solver option: ‘Dynamic	Modal’ • Analysis stage: “full model” ‘Next’ to open the ‘Analysis Wizard : GSS Modal Dynamic Parameters’ dialog: • ‘Number of modes’: 5 • ‘Maximum no. iterations’: 128 • ‘Start mode’: 1 • ‘Additional restraint’: none • ‘Mass option’: ‘Lump mass at nodes’ • ‘Mass derived from loads’: “L1”, “Z” ‘Next’ to open the ‘Analysis Wizard : Cases Set Up’ dialog: • ‘Don’t analyse now’ ‘Finish’
8.5	Analyse	‘GSA’ toolbar; ‘Analyse all’  (Analyses all the tasks that haven’t been analysed yet.)
8.6	Response spectrum data	‘Data	Dynamic Response	Response Spectra’ to open the ‘Response Spectra’ table • ‘Data Options’ toolbar; ‘Wizard’ () to open the ‘Response Spectrum Wizard’ dialog • Select ‘Eurocode 8’ • (leave default parameters unchanged) ‘Data	Dynamic Response	Basic Responses’ to open the ‘Basic Responses’ table, to add the following: 
8.7	Response spectrum analysis of “full model”	‘Data	Cases and Tasks	Analysis Tasks’ to open the ‘Analysis Tasks’ view • Right-click on background of Analysis Tasks View; ‘New Analysis Task’ • Task name: “Response Spectrum – full model” • Solver option: ‘Dynamic response	Response Spectrum’ • Analysis stage: “full 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'
8.8	SRSS combination case of basic responses	'Data | Cases and Tasks | Combination Cases' to open the 'Combination Cases' table, to add the following: • "SRSS of basic responses", "SRSS(A20, 0.35A21, 0.35A22)" (- or whatever the response spectrum analysis case turn out to be!)
8.9	Footfall analysis of "footfall model"	'Data | Cases and Tasks | Analysis Tasks' to open the 'Analysis Tasks' view • Modal analysis of "footfall 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'
		 TrGen_08.gwb (without results)

Create Members for Design

- Create RC members.
- Create steel members.

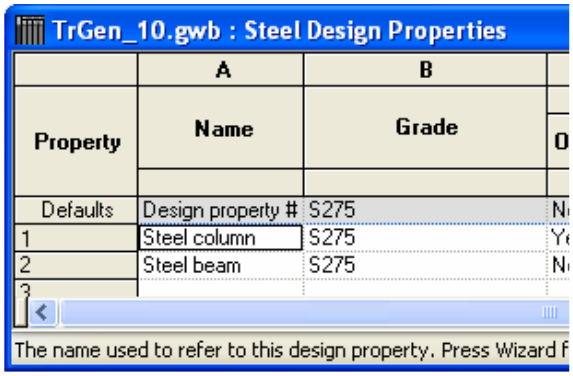
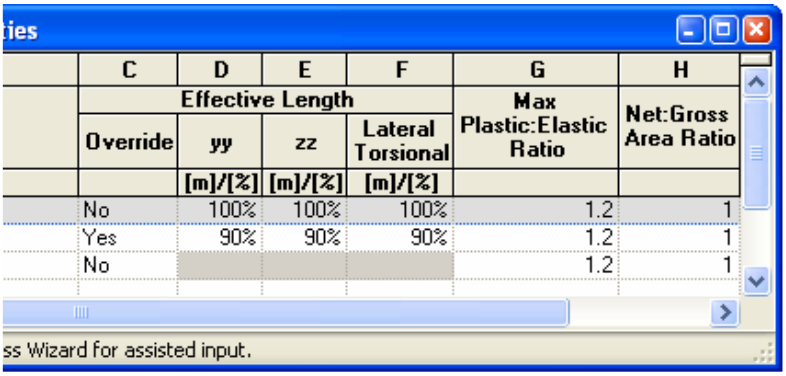
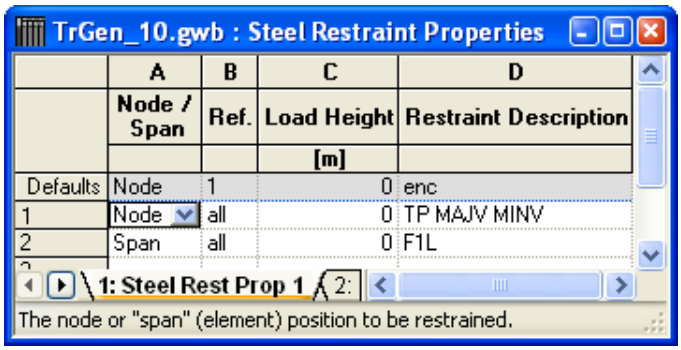
9.1	Create RC members	'Graphic View (analysis layer) | Select elements' cursor mode • Right-click on the Graphic View background; 'Select List'; "PB1 to PB10"; OK (to select all beam elements that reference concrete sections) 'Tools | Manipulate Model | Create members from elements' • 'Element list': "<current selection>" • Select '1 element = 1 member'
9.2	Create steel members	'Graphic View (analysis layer) | Select elements' cursor mode • Select beam elements that reference steel sections, but not transverse elements

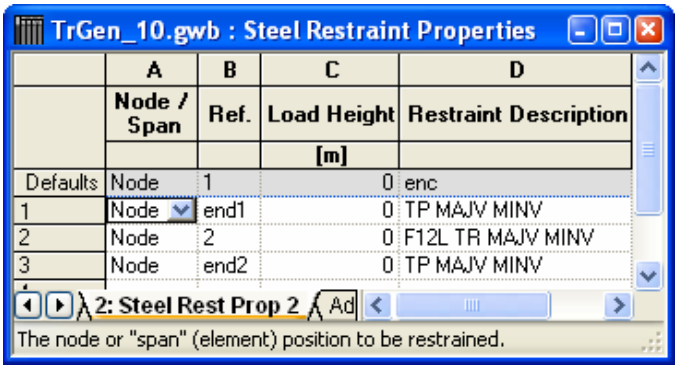
		'Tools | Manipulate Model | Create members from elements' • 'Element list': "<current selection>" • Select '1 element = 1 member' 'Graphic View (analysis layer) | Select elements' cursor mode • Select transverse beam elements that reference steel sections 'Tools | Manipulate Model | Create members from elements' • 'Element list': "<current selection>" • Select 'Many elements = 1 member' • Don't 'Divide members at intersections with columns'
9.3	Inspect members	'Graphic View (design layer)' • 'Graphic Display' toolbar; 'Shrink' • 'Graphic Display' toolbar; 'Section display'
		 TrGen_09.gwb (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.

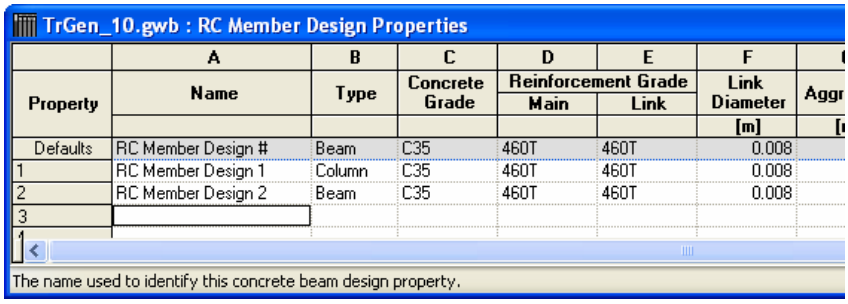
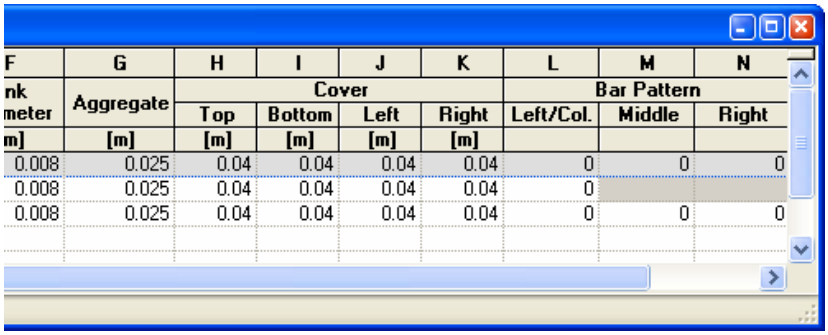
10.1	Design codes	'Data | Specification | Design Specification' to open the 'Design Specification' dialog • Enter your preferred design codes for steel and concrete
10.2	Modify member design property references and member restraint property references	'Graphic View (design layer)' • Label ( ) on elements: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

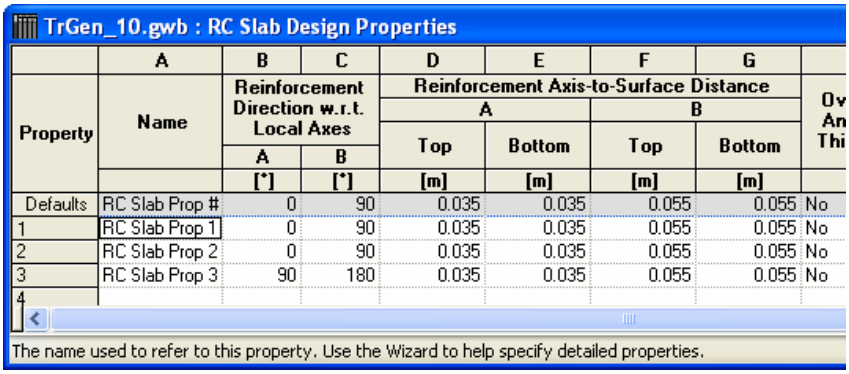
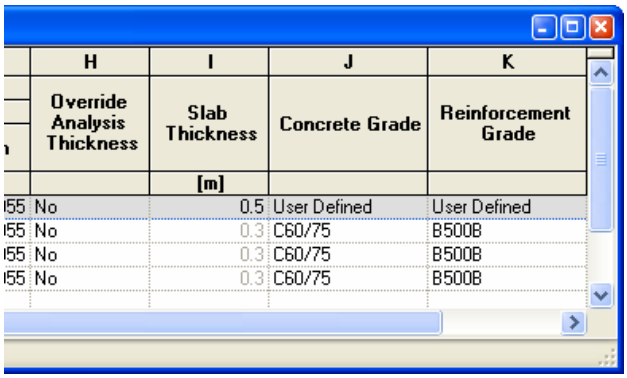
		• ‘Modify design property to’ 2 • Right-click on the Graphic View background; ‘Select List’; “PB13”; OK (to select transverse steel beam members) • Modify the selected members  • ‘Modify restraint property to’ 2
10.3	Steel design properties	‘Data	Properties	Design	Steel Design Properties’ to open the ‘Steel Design Properties’ table, to add the following:   (Use the wizard  to enter the properties.)
10.4	Steel restraint properties	‘Data	Properties	Design	Steel Restraint Properties’ to open the ‘Steel Restraint Properties’ table, to add the following:  - and:



(Use the wizard to enter the properties.)

10.5	Steel design results	The following steel design results are available:
		• 'Graphic View (design layer)' contours:
		  ○ 'Steel Utilisation':
		  • 'Overall', 'Local Combined', 'Buckling Combined', …
		• 'Output View':
		  ○ 'Steel Design Synopsis'
		  ○ 'Steel Utilisation':
		  • 'Overall', 'Local Combined', 'Buckling Combined', …
		  ○ 'Steel Design Summary'
		  ○ 'Steel Design Calcs':
		  • 'Verbose', 'Brief'

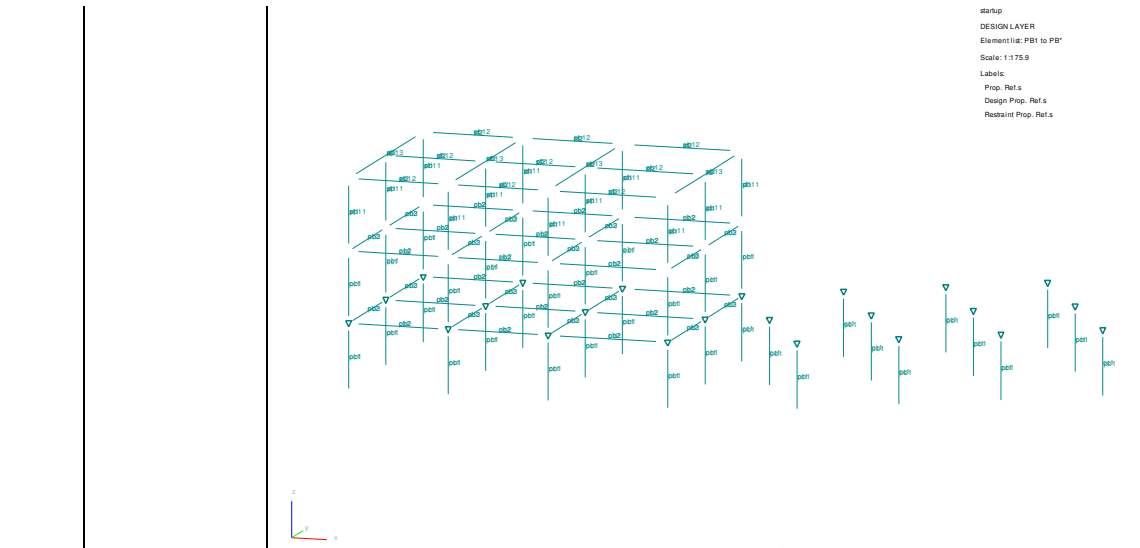
10.6	RC member design properties	'Data | Properties | Design | RC Member Design Properties' to open the 'RC Member Design Properties' table, to add the following:   (Use the wizard to enter the properties.)
10.7	RC member design results	'Design | RC Member Design' to open the 'RC Member Design' dialog

10.8	RC slab design properties	'Data | Properties | Design | RC Slab Design Properties' to open the 'RC Slab Design Properties' table, to add the following:  TrGen_10.gwb : RC Slab Design Properties The name used to refer to this property. Use the Wizard to help specify detailed properties.  (Use the wizard to enter the properties.) (One RC slab design property per 2D element property.) (Reinforcement directions are with respect to element local axes, so inspect 'element axes' labels to understand property 3 reinforcement directions.)

10.9	RC slab design results	The following RC slab design results are available:
		• 'Graphic View (analysis layer)' diagrams:
		  ○ 'RC Slab Reinforcement':
		  • 'Top', 'Bottom'
		• 'Graphic View (analysis layer)' contours:
		  ○ 'RC Slab Reinforcement':
		  • 'Top', 'Bottom', directions 'A' and 'B'
		• 'Output View':
		  ○ '2D Element Results | RC Slab Reinforcement'

Property	Name	Reinforcement Direction w.r.t. Local Axes A	Reinforcement Direction w.r.t. Local Axes B	Reinforcement Axis-to-Surface Distance A Top	Reinforcement Axis-to-Surface Distance A Bottom	Reinforcement Axis-to-Surface Distance B Top	Reinforcement Axis-to-Surface Distance B Bottom	Override Analysis Thickness
		[°]	[°]	[m]	[m]	[m]	[m]	
Defaults	RC Slab Prop #	0	90	0.035	0.035	0.055	0.055	No
1	RC Slab Prop 1	0	90	0.035	0.035	0.055	0.055	No
2	RC Slab Prop 2	0	90	0.035	0.035	0.055	0.055	No
3	RC Slab Prop 3	90	180	0.035	0.035	0.055	0.055	No
4								

Slab Thickness	Concrete Grade	Reinforcement Grade
[m]		
0.5	User Defined	User Defined
0.3	C60/75	B500B
0.3	C60/75	B500B
0.3	C60/75	B500B



TrGen_10.gwb
(without results)

--- oOo ---