

limitstate  *slab*

automated yield-line analysis software

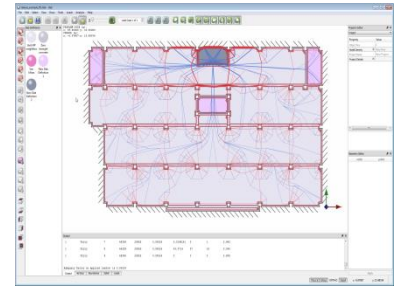
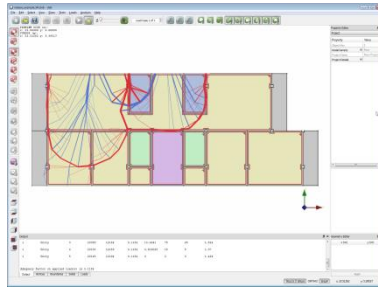
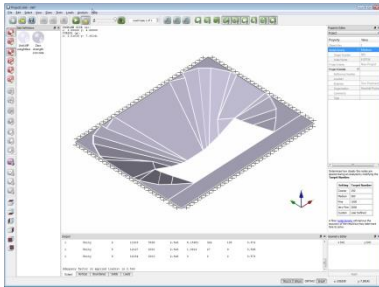
Product Description

Version 2.3.1 - November 2021

LimitState

The Innovation Centre
217 Portobello
Sheffield S1 4DP
United Kingdom
+44 (0) 114 224 2240
info@limitstate.com
limitstate.com

Summary



About LimitState:SLAB

LimitState:SLAB identifies in seconds yield-line failure mechanisms that cannot be identified using conventional ultimate limit state analysis software and which might take a lifetime to identify manually. To achieve this, the powerful and efficient numerical analysis procedure 'Discontinuity Layout Optimization' (DLO) is used to provide an automatic means of identifying very accurate limit analysis solutions:

- An easy to use and generally applicable ultimate limit analysis tool for yield-line analysis and design problems.
- No need to pre-determine the likely failure mechanism. Model the problem as it is and the DLO analysis procedure establishes the critical failure mechanism, meaning that it is suitable for a wide range of circumstances, including bridge and building slab analyses involving columns, holes, irregular geometries and boundary conditions.
- View and animate the failure mechanism on-screen. Query moments on solid bodies.
- Apply partial factors to loading and material strengths. Designed to be used with Eurocodes and other design codes of practice.

Underlying Technology

- LimitState:SLAB uses the Discontinuity Layout Optimization (DLO) procedure to obtain solutions.
- DLO automatically identifies the critical configuration of yield-lines at failure, which can then be visualized.
- The general formulation means that it can be applied to standard and non-standard problems alike.
- No need to independently consider different failure modes as all possible modes (anticipated or not anticipated by the engineer) are simultaneously considered, thus significantly reducing the time required to undertake a stability analysis.
- The DLO method is underpinned by rigorous theory, published in the leading peer reviewed journal, Proceedings of the Royal Society A.

Modelling Capabilities

- Wizards for quick generation of standard problem types.
- Intuitive GUI allows user defined problems to be rapidly constructed.
- Import / export geometries from and to CAD (AutoCAD DXF files).
- Quick and simple to modify geometry, boundary conditions, materials and loading.
- Multiple load cases can be handled easily, with the critical case always highlighted (no need to manually analyse different potential failure modes).
- Directly apply Partial Factors in the analysis, making the software straightforward to use with design codes.

Structural Definitions

- Define slabs using just moment resistance and self-weight values.

- Model orthotropic and isotropic slab strengths.

Partial Factors

- Specify 'Favourable', 'Neutral' or 'Unfavourable' loading.
- Categorise loads as 'Permanent', 'Accidental' or 'Variable'.
- Apply partial factors based on combinations of each of the above settings.
- Apply partial factors to material strength.
- Use the inbuilt 'Load Case Manager' to define and solve multiple scenarios in one go.

Output

- The solution is reported as a factor on load.
- Results are based on rigorous and proven limit analysis methods.
- Failure mechanisms are represented as yield-lines (a familiar concept to structural engineers).
- Animated displacements provide a highly useful visual interpretation of the failure mechanism.
- Moments and rotations along yield-lines are displayed both graphically and numerically.
- The analysis report output is easily tuned to user preferences.

Interactive Viewer

Define, alter and experiment with the model:

- Quickly define or modify a model by drawing the geometry directly in the viewer.
- Examine or edit object properties using the Property and Geometry editors.
- Add to / change structural definitions in a zone using drag and drop functionality.
- Switch between different load cases using the tabbed viewer window.
- Benefit from a comprehensive context sensitive help system.

Accessibility

Many features designed to help the user:

- Wizards to allow basic problem types to be rapidly defined and analysed.
- Customized reports of the analysis findings can be automatically generated, printed and saved.
- Integrated calculator with comprehensive unit conversion functionality.

Validation

- Underlying technology published in leading peer-reviewed science journal.
- Validated against a large, and ever-growing, number of standard test problems (limitstate.com/slab/verification).

Summary	1
About LimitState:SLAB	1
System Requirements	Error! Bookmark not defined.
Underlying Technology	1
Modelling Capabilities	1
Structural Definitions.....	1
Partial Factors	2
Output	2
Interactive Viewer	2
Accessibility.....	2
Validation.....	2
User Interface	5
Property Editor	6
Project	7
Structural Definitions.....	7
Flexural	7
Rigid.....	8
Vertices.....	8
Internal Boundaries	8
External Boundaries	8
Solids	9
Nodes	9
Yield-lines	9
Blocks	10
Geometry Editor	11
Load Case Manager	11
Partial Factor Sets.....	11
Calculator	12
Units.....	12
Functions	12
Explorers.....	13
Vertex, boundary, solid and load explorers.....	13
Structural definition explorer.....	13
Viewer Pane.....	14
Wizards.....	15
Sample Wizard – Rectangular Slab Project	16
Project Details.....	16
Geometry.....	16
Structural Definitions.....	17
Loads	18

Load Cases	18
Partial Factors.....	18
Loads.....	18
Analysis.....	19
Solve.....	20
Analysis Details.....	20
Analysis Engine.....	20
Results	20
Report Output.....	21
Report Contents	22
Menus	23
File.....	23
Edit	23
Select.....	24
View.....	24
View (contd...)	25
Draw	25
Tools.....	26
Analysis.....	26
Help	27
Dialogs.....	28
Selection Filter	28
Draw Settings	28
Create New Structural Definition	29
Project Details.....	30
Preferences.....	30
Preferences (contd...)	31
Load Case Manager	31
Report.....	32
Toolbars.....	33
File.....	33
Edit	33
Cursor	33
Draw	33
Draw (contd...)	34
Analysis.....	34
Animation	34
Zoom	34
Show.....	34
Show (contd...)	35
Select.....	35
Rotate 3D.....	35

View 3D 35

View 3D (contd...) 36

Loads 36

Help 36

Context Menus..... 37

 Toolbars and Property Editor 37

 Viewer Pane (General)..... 38

 Viewer Pane (Post-Solve Solid)..... 39

 Structural Definitions Explorer 39

 Other Explorers 39

 Geometry Editor..... 39

 Output Pane..... 39

User Interface

Figure 1 illustrates the main features of the LimitState:SLAB user interface:

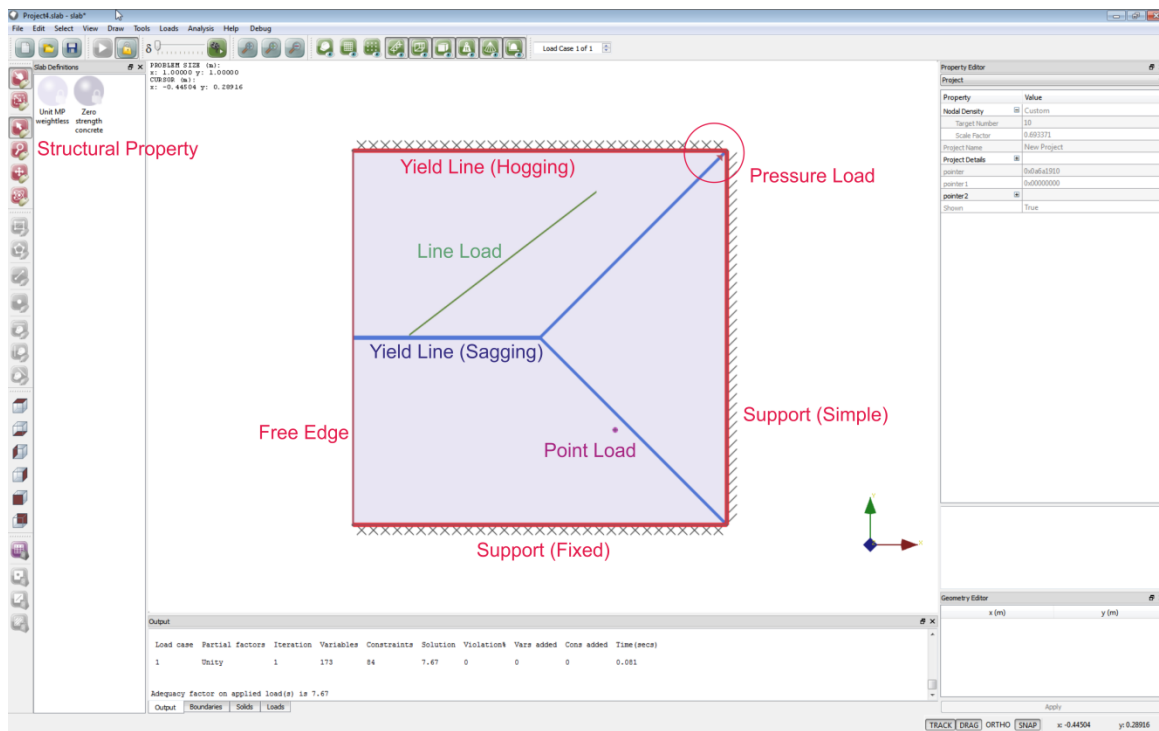


Figure 1 - The LimitState:SLAB User Interface

Property Editor

The Property Editor allows the user to quickly query and / or modify the attributes of one or more objects within the current project.

- Select objects on screen and modify their properties to better reflect the real-life circumstances of the model.
- Change a property for a single object or modify the common properties of multiple objects en masse.
- Access some of the more specialized functions.
- Changes made in the property editor can be undone / redone using 'undo' and 'redo'.



Property	Value
Object ID3	Solid S4
Baseline Nodal Spacing (m)	
.xspacing (m)	1
.yspacing (m)	1
Adequacy	True
Slab Definition	1 slab definition assi...
Unit MP weightless	
ID	Unit MP weightless
Color	
First: M_p^+	1
First: M_p^-	1
First: Angle	0
Second: M_p^+	1
Second: M_p^-	1
Second: Angle	90
Thickness	1
Unit Weight (kN/m ³)	0

The [baseline spacing](#) (m) between nodes.

Used when calculating the actual spacing between nodes to achieve the **Target Number**:

$$\text{Actual Nodal Spacing} = \text{Baseline Nodal Spacing} / \text{Scale Factor}$$

Figure 2 - The LimitState:SLAB Property Editor

Using the property editor it is possible to examine and / or modify the general properties of the **Project** and each of the **Structural definitions**. In addition, the following objects may be examined and / or modified:

- **Vertices** – points where two or more Boundary objects meet. A vertex may lie on a straight line. A Vertex is shared by the Boundary objects that meet at that vertex.
- **Boundaries** – straight lines that link two Vertices and defines the edge or boundary of a Solid. If a Boundary object separates two Solids then it is shared by those solids and forms an interface.
 - **Internal** – boundaries with a Solid to either side.
 - **External** – boundaries with a Solid to one side only. These can be assigned support conditions.
- **Solids** – 2D polygons defining a section of slab. Their extents are defined by the surrounding Boundary objects.
- **Nodes** – potential intersection-points of yield-lines.
- **Yield-lines** – lines coinciding with the direction of failure for regions of material stressed to the limiting or yield condition.
- **Point loads** – loading applied at specific points.
- **Line loads** – loading applied along specified lines.
- **Pressure loads** – loading applied to one or more Solids.

The following tables describe, in detail, the properties found in the Property Editor for each of the described features:

Project

Property	Description	Editable?
Nodal Density	Determines how closely nodes are spaced during an analysis by modifying the Target Number . Values can be coarse (250), medium (500), fine (1000), very fine (2000) or custom defined by the user.	✓
Target Number	The total number of nodes that the software will attempt to use when solving the problem. Altering this value will cause the Scale Factor on the Baseline Nodal Spacing to change, thus altering the Actual Nodal Spacing : Actual Nodal Spacing = Baseline Nodal Spacing / Scale Factor Note: The Actual Nodal Spacing is calculated by the software to determine the spacing required to achieve the Target Number of nodes and cannot be directly modified by the user.	✓
Scale Factor	A global factor used to scale the spacing between nodes in the project to achieve (or alter) the Target Number of nodes). e.g. doubling the Scale Factor causes the number of nodes used along boundaries to double and the number of nodes used in solids to quadruple.	✓
Project Name	A title for the project.	✓
Project Details	General information about the project.	
Reference Number	A reference number.	✓
Location	The physical location of the project.	✓
Engineer	Name of the engineer responsible.	✓
Organization	Name of the organization responsible.	✓
Comments	Any additional comments.	✓
Tags	Add tags to the file which can be used when searching outside of LimitState:SLAB	✓

Structural Definitions

Flexural

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Color	The color of a material as shown in the viewer.	✓
First: Mp*	Sagging moment capacity per unit length, in the direction specified by the angle β .	✓
First: Mp	Hogging moment capacity per unit length, in the direction specified by the angle β .	✓
First: Angle β	The angle (anticlockwise) described between the global x axis and the direction in which Mp is acting.	✓
Second: Mp*	Sagging moment capacity per unit length, in the direction specified by the angle β .	✓
Second: Mp	Hogging moment capacity per unit length, in the direction specified by the angle β .	✓
Second: Angle β	The angle (anticlockwise) described between the global x axis and the direction in which Mp is acting.	✓
Thickness	Thickness of the slab in a particular Solid area. Used in conjunction with the Unit Weight to determine the self-weight of the slab.	✓
Unit weight	Unit weight of the slab in a particular Solid area. Used in conjunction with the Thickness to determine the self-weight of the slab.	✓

Rigid

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Color	The color of a material as shown in the viewer.	✓
Thickness	Thickness of the slab in a particular Solid area. Used in conjunction with the Unit Weight to determine the self-weight of the slab.	✓
Unit weight	Unit weight of the slab in a particular Solid area. Used in conjunction with the Thickness to determine the self-weight of the slab.	✓

Vertices

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
x	The x coordinate of a vertex.	Use Geometry Editor
y	The y coordinate of a vertex.	Use Geometry Editor

Internal Boundaries

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Baseline Nodal Spacing	The baseline spacing (m) between nodes. Used when calculating the actual spacing between nodes to achieve the Target Number: Actual Nodal Spacing = Baseline Nodal Spacing / Scale Factor.	✓
Structural Definition	A list of the materials assigned for use in a boundary.	✓
Support Type	The type of support assigned to a Boundary	✓
Free	Boundary is able to move and rotate.	
Knife-edge	The boundary will not move unless 'lift-off' is permitted. The overlying slab is considered to be continuous, with moments over the support and rotation around the line of the axis being permitted. A yield-line will form if the hogging moment over the support reaches or surpasses the limiting moment resistance of the slab.	

External Boundaries

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Baseline Nodal Spacing	The baseline spacing (m) between nodes. Used when calculating the actual spacing between nodes to achieve the Target Number: Actual Nodal Spacing = Baseline Nodal Spacing / Scale Factor.	✓
Structural Definition	The Structural Definition assigned for use in a boundary. Selecting this field brings up a button that opens a dialog allowing a definition to be assigned / removed from a zone.	✓
Support Type	The type of support assigned to a Boundary	✓
Fixed	Boundary will not move or rotate.	
Partially Fixed	The boundary is fixed against displacements in all directions. Rotations of the slab around the axis of the boundary are only permitted as a result of yield-line formation at a moment equal to the specified ratio multiplied by the strength of the adjacent slab.	
Free	Boundary is able to move and rotate.	
Symmetry	Emulates symmetry boundary conditions.	

Simple	Boundary will rotate but not move (unless 'lift-off' is permitted, in which case vertical upwards movement can occur).	
--------	--	--

Solids

Property	Description	Editable?
Object ID	A unique identifier given to all objects in the project.	✓
Baseline Nodal Spacing	The baseline spacing (m) between nodes in a solid. Used when calculating the actual spacing between nodes to achieve the Target Number.	
x spacing	Baseline Nodal Spacing in the x direction.	✓
y spacing	Baseline Nodal Spacing in the y direction.	✓
Structural Definition	The Structural Definition assigned for use in a boundary. Selecting this field brings up a button that opens a dialog allowing a definition to be assigned / removed from a zone.	✓

Nodes

Property	Description	Editable?
x	The x coordinate of a node.	
y	The y coordinate of a node.	
z	The z coordinate of a node (zero)	

Yield-lines

Property	Description	Editable?
ID	A unique identifier given to all objects in the project.	
Start Node	The node at the start of a yield-line.	
x	The x coordinate of the point at the start of a yield-line.	
y	The y coordinate of the point at the start of a yield-line.	
z	The z coordinate of the point at the start of a yield-line (always zero).	
End Node	The node at the end of a yield-line.	
x	The x coordinate of the point at the end of a yield-line.	
y	The y coordinate of the point at the end of a yield-line.	
z	The z coordinate of the point at the start of a yield-line (always zero).	
Length	The Straight-line distance between 'Start Node' and 'End Node'.	
Moment	The total bending moment acting along the yield-line.	
Rotation	Instantaneous relative rotation.	

Blocks

Blocks can be selected once the problem is in the solved and in a displaced state.

Property	Description	Editable?
ID	A unique identifier given to all objects in the project.	✓
Point X		
ID	The identifier of the point.	✓
x	The x coordinate of the point.	
y	The y coordinate of the point.	
z	The z coordinate of the point.	

Geometry Editor

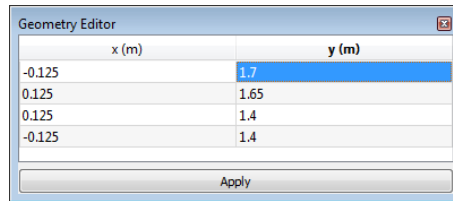


Figure 3 - The LimitState:SLAB Geometry editor

- Examine and / or modify the x and y coordinates of a vertex, boundary or solid
- Make rapid alterations to the problem geometry

Load Case Manager

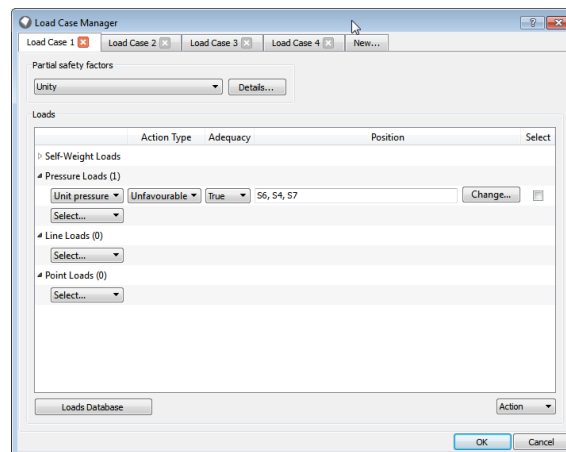


Figure 4 - The LimitState:SLAB Load Case Manager

Define specific combinations of partial factor and loads to be considered during analysis. If multiple load cases are defined, the critical case will be identified automatically during solve.

Partial Factor Sets

- Specify whether a single or multiple scenario analysis is carried out.
- Editable partial factors:
 - Neutral loads (permanent, accidental and variable).
 - Unfavourable loads (permanent, accidental and variable).
 - Favourable loads (permanent, accidental and variable).
 - Materials (M_p).
- Default available partial factor sets:
 - Unity.
- Import partial factor sets from .csv file.
- Export partial factor sets to .csv file.
- Define new partial factor sets.
- Delete a partial factor set from a multiple load case problem.

- Use the 'manage' dialog to rename or delete individual partial factor sets from the file.

Calculator

- Quickly calculate values for entry into any field that requires numerical input (simply click in the field and select the calculator button).
- Convert a wide variety of types into the default LimitState:SLAB units:

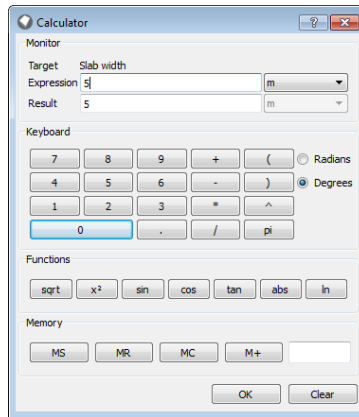


Figure 5 - The LimitState:SLAB Calculator

Units

Unit	Metric	Imperial	Converter Options
Force / Weight	kN	lbf	N, kN, MN, lbf
Pressure	kN/m ²	psf (lbf/ft ²)	kN/m ² , MN/m ² , lbf/ft ² , lbf/yd ²
Unit weight	kN/m ³	pcf (lbf/ft ³)	kN/m ³ , MN/m ³ , lbf/ft ³ , lbf/yd ³
Length	m	ft	mm, cm, m, ft, yd
Area	m ²	ft ²	mm ² , cm ² , m ² , ft ² , yd ²
Volume	m ³	ft ³	mm ³ , cm ³ , m ³ , ft ³ , yd ³

Functions

- +, -, *, / - add, subtract, multiply, divide
- **sqrt** - square root
- **x²** - square
- **sin, cos, tan** - trigonometric functions
- **abs** - absolute
- **ln** - natural logarithm
- **(x)** - parenthesis
- **^** - exponent
- **pi** - 3.14159
- **MS** – store in memory
- **MR** – recall memory
- **MC** – clear memory
- **M+** - add to number in memory

Explorers

Vertex, boundary, solid and load explorers

- When the project is unlocked, objects selected in the **Vertex**, **Boundary**, **Solid** and **Load** explorers are highlighted in the viewer pane (and *vice versa*).
- The **Vertex Explorer** displays the Object ID.
- The **Boundary Explorer** displays the Object ID and Support Type.
- The **Solid Explorer** displays the Object ID.
- The **Load Explorer** displays the Name, Pattern, Value, Units and Load Type.

Solids	
Object ID	
Project	
Solid S23	
Solid S50	
Solid S53	
Solid S59	
Solid S62	
Solid S65	

Figure 6 - The LimitState:SLAB Solid Explorer

Boundaries	
Object ID	Support Type
Project	
External Boundary E1	Simple
External Boundary E2	Fixed
External Boundary E3	Symmetry
External Boundary E4	Free

Figure 7 - The LimitState:SLAB Boundary Explorer

Structural definition explorer

- Drag and drop from the **Structural Definition Explorer** to assign 'materials' to boundaries and solids.
- Create user-defined structural definitions, with full control over properties for each type (see the Dialogs section).
- Import pre-defined structural definitions from file.
- Export user-defined structural definitions to file.



Figure 8 - The LimitState:SLAB Structural Definition Explorer

The Material Explorer contains a library of built-in structural definitions of different types (metric defaults shown):

Name	First: Mp+ (kNm/m)	First: Mp- (kNm/m)	First: Angle β	Second: Mp+ (kNm/m)	Second: Mp- (kNm/m)	Second: Angle β	Thickness (m)	Unit Weight (kN/m ³)
Unit MP weightness	1	1	0	1	1	90	0.25	0
Zero strength concrete	0	0	0	0	0	90	0.25	0
Unit MP concrete	1	1	0	1	1	90	0.25	24
Rigid weightless	-	-	-	-	-	-	0.25	0

Viewer Pane

- A fully interactive graphical modelling environment.
- Total freedom to define or modify problem geometries.
- View the model from any angle (including in 3D) and zoom into areas of interest.
- Select any part of the model with the mouse to view or modify its properties.

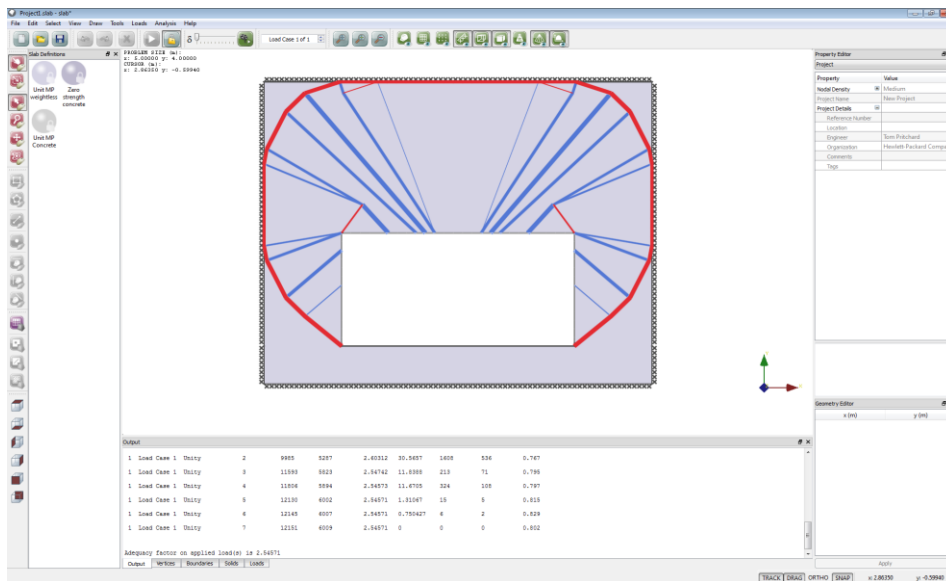


Figure 9 - The LimitState:SLAB Modelling Environment

Wizards

- Rapidly define a standard problem type using the built in wizards, which guide users through the process of defining the geometry, structural definitions, loading *etc.*

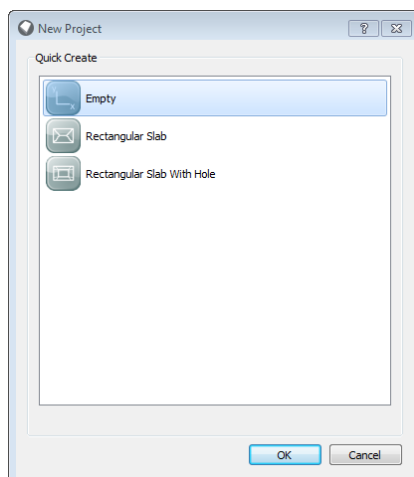


Figure 10 - The LimitState:SLAB New Project Dialog

- Choose from the following wizard types:
 - Empty
 - Rectangular Slab
 - Rectangular Slab With Hole
- Click 'Finish' at any point during the wizard process: LimitState:SLAB will automatically fill in any information that has not been explicitly supplied by assuming default values and using information already given.
- Move backwards and forwards through the various steps of the wizards and edit data as necessary; a very useful feature should any detail accidentally be omitted initially.

The dialogs found within most wizards are as follows:

- **Project** – defines the general properties of the project.
- **Geometry** – defines the geometric form of the slab to be analyzed.
- **Structural Definitions** – defines the properties of the different solid zones and interfaces.
- **Loads** – defines a standard loading configuration relevant to the particular problem.
- **Analysis** – define the nodal resolution of the problem.

Sample Wizard – Rectangular Slab Project

- Some of the dialogs shown in other wizards will be different to those shown here (*e.g.* those relating to geometry). This example wizard is however indicative of the different wizards available in LimitState:SLAB.
- In all wizard dialogs, default data is assumed unless otherwise specified by the user.

Project Details

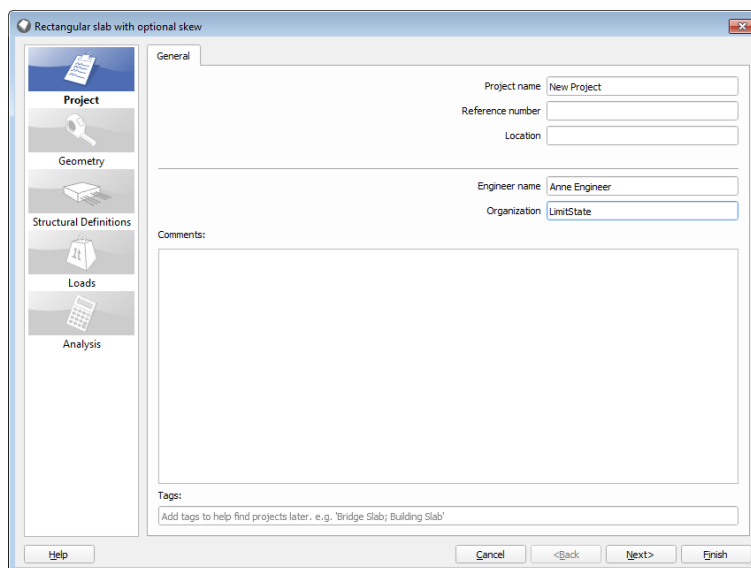


Figure 11 - The LimitState:SLAB Rectangular Slab Project 'Project Details' Dialog

- Deals with 'general' aspects of the model
- All of the details that can be specified here are optional:
 - Project Name
 - Reference number
 - Location
 - Engineer name
 - Organization
 - Comments
 - Tags

Geometry

Define the key dimensions of the footing (the figure in the dialog clearly indicates the meaning of the parameters to be entered):

- Slab dimensions
- Skew angle
- Boundary conditions

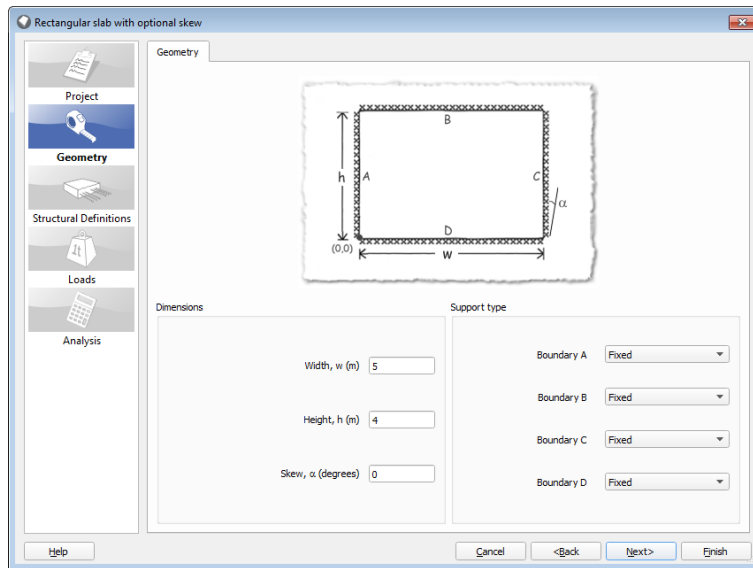


Figure 12 - The LimitState:SLAB Rectangular Slab Project 'Geometry' Dialog

Structural Definitions

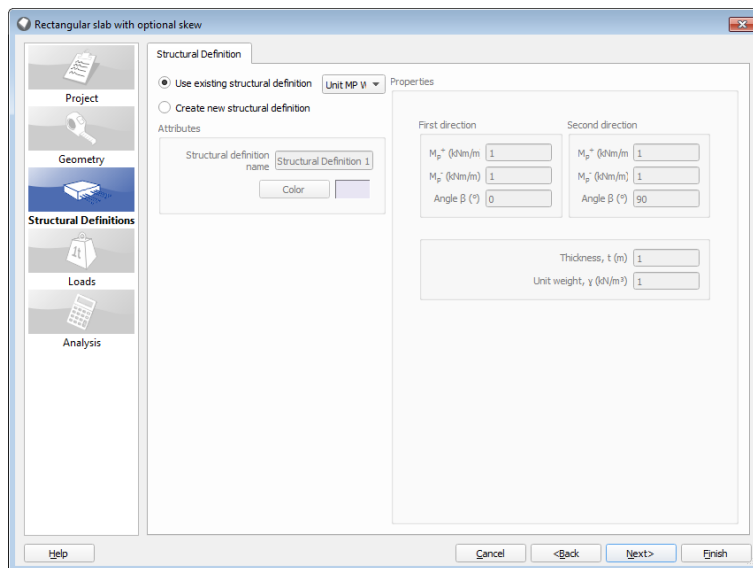


Figure 13 - The LimitState:SLAB Rectangular Slab Project 'Structural Definitions' Dialog

Define the 'material' properties of the slab:

- Create new structural definition:
 - Attributes – Name
 - Attributes – Color
 - Properties – First direction: M_p^+
 - Properties – First direction: M_p^-
 - Properties – First direction: Angle (degrees)
 - Properties – Second direction: M_p^+
 - Properties – Second direction: M_p^-
 - Properties – Second direction: Angle (degrees)
 - Properties – Thickness

- Properties – Unit weight
- Use existing structural definition

Loads

Define the vertical load applied to the slab:

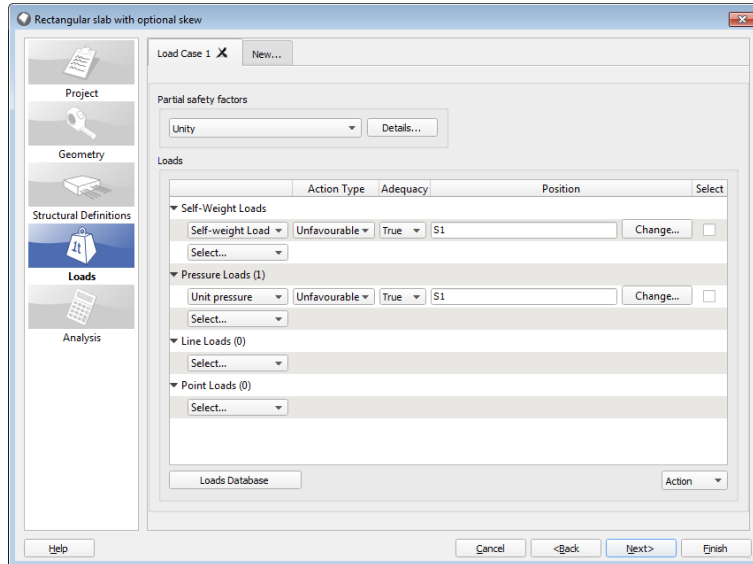


Figure 14 - The LimitState:SLAB Rectangular Slab Project 'Loads' Dialog

Load Cases

Define the different load cases to be considered during analysis:

Partial Factors

- Specify whether a single or multiple load case analysis is carried out.
- Editable partial factors:
 - Neutral loads (permanent, accidental and variable).
 - Unfavourable loads (permanent, accidental and variable).
 - Favourable loads (permanent, accidental and variable).
 - Materials (M_p).
- Default available partial factor sets:
 - Unity.
- Import partial factor sets from .csv file.
- Export partial factor sets to .csv file.
- Define new partial factor sets.
- Delete a partial factor set from a multiple load case problem.

Loads

- Define or delete loads in the Loads Database:
 - Self-weight
 - Point loads
 - Line loads
 - Pressure loads
 - All loads can be assigned Permanent, Accidental or Variable status.

- Assign loads:
 - To Solids (Pressure, Self-weight)
 - Between two Vertices (Line)
 - At a Vertex (Point)

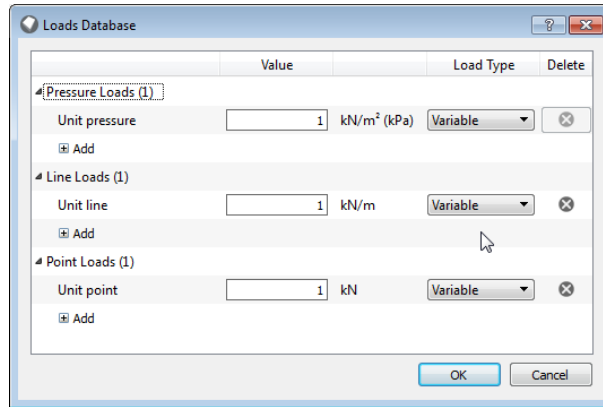


Figure 15 - The LimitState:SLAB Loads Database Dialog

Analysis

Define the nodal resolution and type of analysis to undertake.

- Specify the target nodal density.

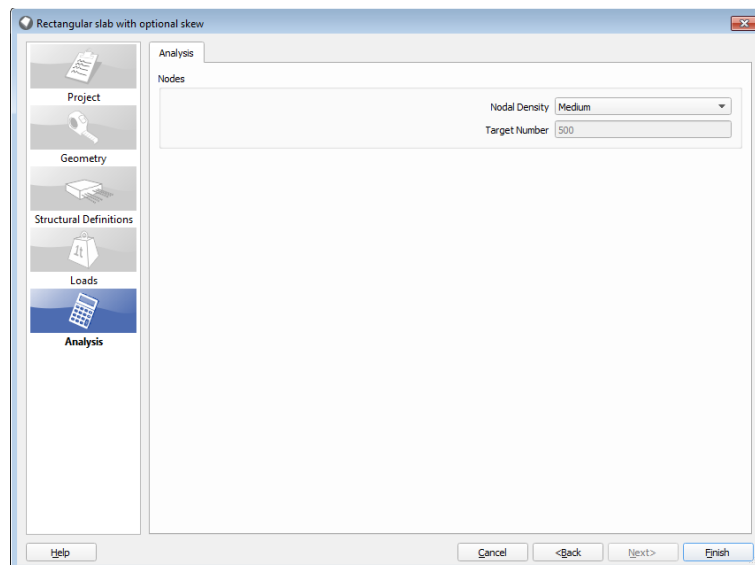


Figure 16 - The LimitState:SLAB Analysis Dialog

Solve

Analysis Details

- Performs a kinematic or 'mechanism' analysis based on plasticity theory, and assuming small displacements.
- Provides a solution in terms of an adequacy factor on loads.
- For a Project with a several load cases, the adequacy factor for each is computed and the lowest (most critical) is identified.

Analysis Engine

- Uses MOSEK, a powerful interior point linear programming solver (see www.mosek.com).
- For maximum efficiency all problem data is passed to the solver via memory.
- More detailed information on the progress towards a solution is available in the Preferences dialog.

Results

- Critical mechanism automatically identified and computed adequacy factor displayed in output window.
- If no solution can be found, or if model is unstable, output reports this and gives possible reasons for this behaviour.
- Critical mechanism can be animated for enhanced user understanding (instantaneous velocities magnified).
- Moments along yield-lines displayed in graphical and numerical formats when solids are selected after solve.

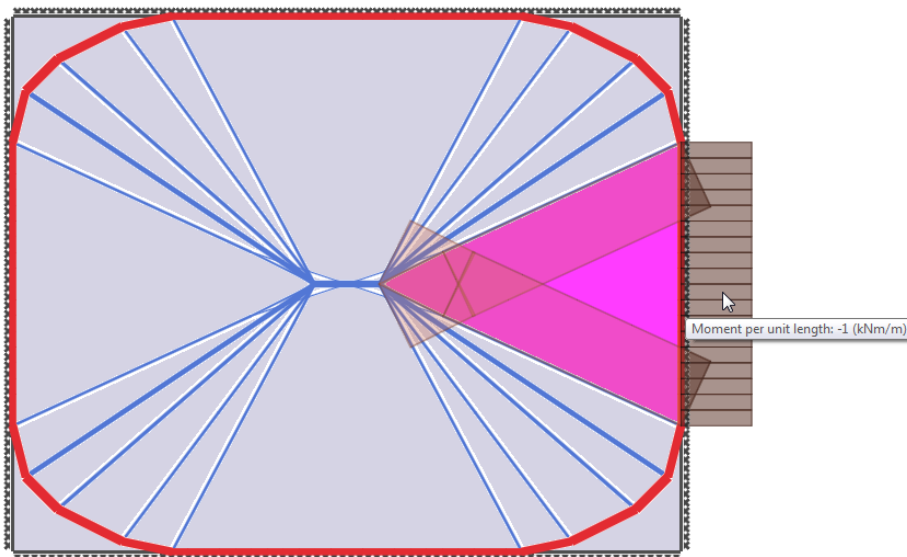


Figure 17 - Post-solve diagrams displayed by LimitState:SLAB (Moments)

Report Output

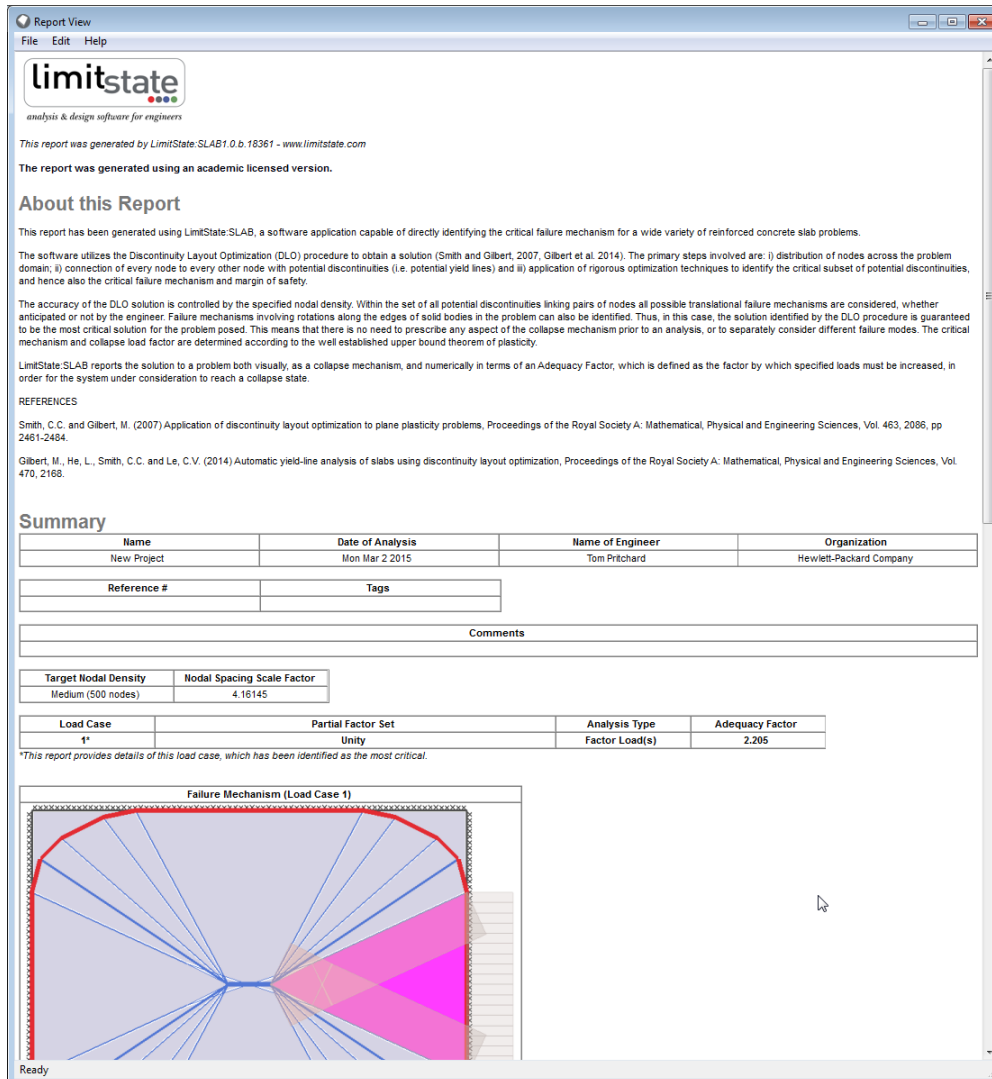


Figure 18 - The LimitState:SLAB Report View

- Chose the sections that are included in the report:
 - About
 - Summary
 - Geometry
 - Structural Definitions
 - Partial Factors
 - Loads
 - Load Cases
- Customize the document with user-defined headers and footers.
- Print the report directly from LimitState:SLAB or save the file in RTF format.

Report Contents

By default, each report contains all the sections mentioned above, containing the following details:

Section	Contents
About	<ul style="list-style-type: none"> A paragraph of text outlining the method of analysis (DLO) used by LimitState:SLAB
Summary	<ul style="list-style-type: none"> Name Reference Nº. Location Name of engineer Organization Date of analysis Comments Tags Target nodal density Nodal spacing scale factor Analysis result: <ul style="list-style-type: none"> Critical load case Partial factor set Analysis type Adequacy factor The critical load case is identified and a graphical representation of the critical failure mechanism is shown
Geometry	<ul style="list-style-type: none"> Global <ul style="list-style-type: none"> Number of vertices Number of boundaries Number of solids Boundary objects <ul style="list-style-type: none"> ID Start vertex ID End vertex ID Baseline nodal spacing Support type Structural Definition(s) Solid objects <ul style="list-style-type: none"> ID (any solids with loads applied are highlighted by an *) Vertex IDs Boundary IDs Baseline nodal spacing (x / y) Structural Definition(s)
Structural Definitions	<ul style="list-style-type: none"> All properties associated with the structural definitions used in the model: Flexural: <ul style="list-style-type: none"> Icon Name Thickness Unit weight First: M_p^+ First: M_p^- First: Angle Second: M_p^+ Second: M_p^- Second: Angle Rigid: <ul style="list-style-type: none"> Icon Name Thickness Unit weight
Partial factors	<ul style="list-style-type: none"> For each load case: <ul style="list-style-type: none"> Neutral: permanent Neutral: variable Neutral: accidental Unfavourable: permanent Unfavourable: variable Unfavourable: accidental Favourable: permanent Favourable: variable Favourable: accidental M_p The partial factor set used in the critical scenario is also identified.
Loads	<ul style="list-style-type: none"> Loaded object ID Value Units Type of load
Load Cases	<ul style="list-style-type: none"> For each load case: <ul style="list-style-type: none"> Name Action Type Position Adequacy (true/false)

Menus

File

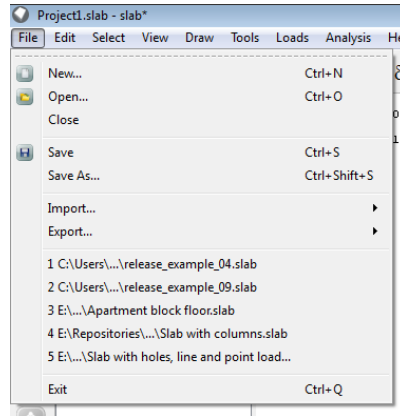


Figure 19 - The LimitState:SLAB File Menu

Action	Result
New...	Start a new project from scratch.
Open...	Open a previously saved project.
Close	Close the current project.
Save	Save the current project using the current filename.
Save As...	Save the current project under a different filename.
Import...	Import predefined Structural Definitions (.csv) or problem Geometry (.dxf).
Export...	Export problem Structural Definitions (.csv), Geometry (.dxf, .eps or .asy), an Image (.png, .jpg, .tiff, .eps or .ps) or Animation (.avi or .gif).
Recent Files	Open a recently saved project.
Exit	Exit LimitState:SLAB.

Edit

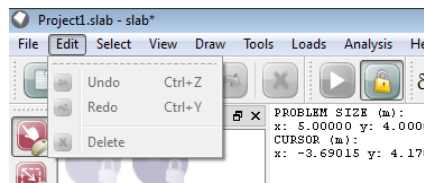


Figure 20 - The LimitState:SLAB Edit Menu

Action	Result
Undo	Undo the last action.
Redo	Redo the last undo.
Delete	Delete the currently selected object(s).

Select

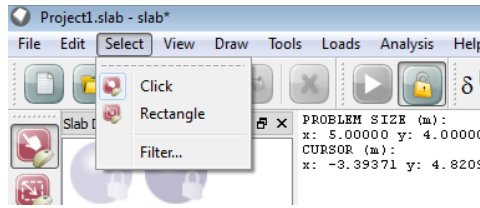


Figure 21 - The LimitState:SLAB Select Menu

Action	Result
Click	Select objects by clicking them with the mouse pointer.
Rectangle	Select objects by drawing a rectangle around them.
Filter...	Select only objects of a particular type (or types): <ul style="list-style-type: none"> ○ Vertex ○ Boundary ○ Solid ○ Yield-line ○ All

View

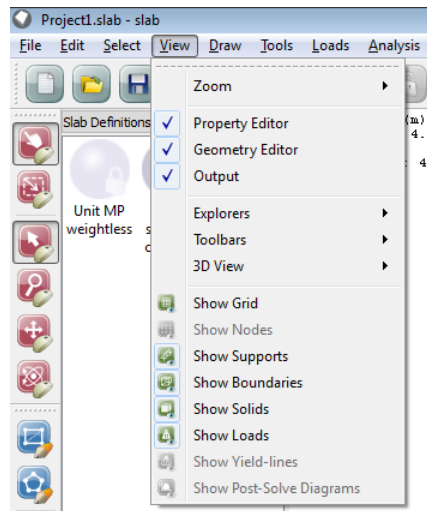


Figure 22 - The LimitState:SLAB View Menu

Action	Result
Zoom	Zoom towards or away from the model: <ul style="list-style-type: none"> ○ Zoom All (Display the entire model) ○ Zoom In (Zoom towards the centre of the model) ○ Zoom Out (Zoom away from the centre of the model)
Property Editor	Toggle the display of the Property Editor .
Geometry Editor	Toggle the display of the Geometry Editor .
Output	Toggle the display of the Output pane .

View (contd...)

Action	Result
Explorers	Toggle the display of the Explorers : <ul style="list-style-type: none"> ○ Vertex ○ Boundary ○ Solid ○ Structural Definitions ○ Loads
Toolbars	Toggle the display of the Toolbars : <ul style="list-style-type: none"> ○ File ○ Edit ○ Cursor ○ Draw ○ Loads ○ Rotate 3D ○ Analysis ○ Animation ○ Zoom ○ Show ○ View 3D ○ Help
3D View	View the model from various 3D viewpoints : <ul style="list-style-type: none"> ○ Top ○ Bottom ○ Left ○ Right ○ Front (Default) ○ Back ○ Show Global Axis ○ Toggle Perspective
Show Grid	Toggle the display of the Grid .
Show Nodes	Toggle the display of the Nodes .
Show Supports	Toggle the display of the Supports .
Show Boundaries	Toggle the display of the Boundaries .
Show Solids	Toggle the display of the Solid objects.
Show Loads	Toggle the display of the Loads .
Show Yield-lines	Toggle the display of the Yield-lines .
Show Post-Solve Diagrams	Toggle the display of post-solve 'on solve' diagrams.

Draw

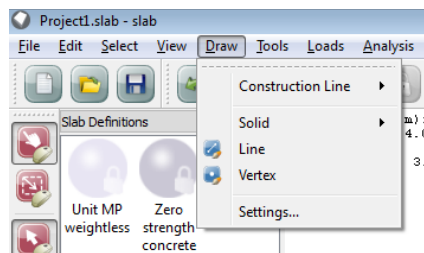


Figure 23 - The LimitState:SLAB Draw Menu

Action	Result
Construction Line	Draw a construction line on the viewer pane (these are to aid in the generation of the model and are not counted during analysis): <ul style="list-style-type: none"> o Vertical / Horizontal / Custom / Clear All
Solid	Draw a solid object: <ul style="list-style-type: none"> o Rectangle (a 4 sided solid defined by two corners) o Polygon (a multi-sided solid defined by all of its corners)
Line	Draw a line using two points.
Vertex	Add a vertex to a line.
Settings...	Open the draw settings dialog.

Tools

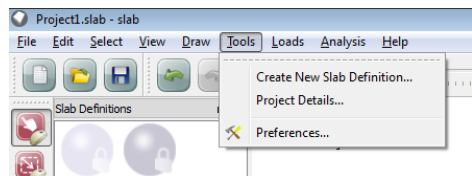


Figure 24 - The LimitState:SLAB Tools Menu

Action	Result
Create New Structural Definition...	Open the Create New Structural Definition dialog.
Project Details...	Open the Project Details dialog.
Preferences...	Open the Preferences dialog.

Analysis

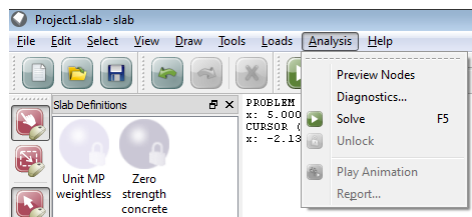


Figure 25 - The LimitState:SLAB Analysis Menu

Action	Result
Preview Nodes	Display the nodes that will be used during analysis (these are the potential end-points of yield-lines).
Diagnostics	Open the Diagnostics dialog to display information, warnings and errors associated with the problem.
Solve	Analyse the problem.
Unlock	Allow the problem to be edited following an analysis or previewing of the nodes.
Play Animation	Animate the failure mechanism for the selected load case.
Report...	Generate a report following analysis.

Help

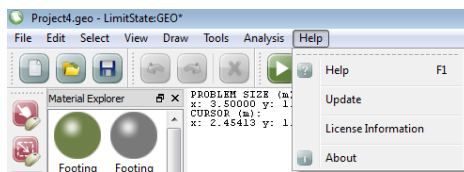


Figure 26 - The LimitState:SLAB Help Menu

Action	Result
Help	Open the LimitState:SLAB help system.
Update	Check for (and download) updates for LimitState:SLAB.
License Information	View information about the current license being used and swap to a different license if required.
About	Display information regarding the version of LimitState:SLAB currently being used.

Dialogs

Aside from standard Save / Open / Import / Export dialogs, the following dialogs are available in LimitState:SLAB:

Selection Filter

- Accessed from the Select menu.
- Set which object types can be selected in the viewer pane:

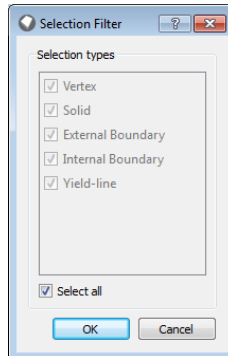


Figure 27 - The LimitState:SLAB Selection Filter Dialog

Draw Settings

- Accessed from the Draw menu.
- Specify grid dimensions and snap settings:

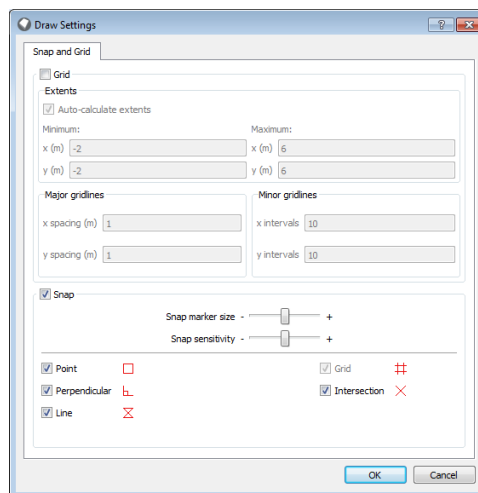


Figure 28 - The LimitState:SLAB Draw Settings Dialog

Grid / Snap	Option	Description
Grid	Extents : Auto calculate extents	Adaptively resize the drawing extents during modelling.
	Extents : Minimum x	Specify an initial minimum coordinate along the x axis.
	Extents : Minimum y	Specify an initial minimum extent along the y axis.
	Extents : Maximum x	Specify an initial maximum extent along the x axis.
	Extents : Maximum y	Specify an initial maximum extent along the y axis.
	Major gridlines: x spacing	Specify the spacing between major x axis gridlines.
	Major gridlines: y spacing	Specify the spacing between major y axis gridlines.
	Minor gridlines: x intervals	Specify the number of intervals between major x axis gridlines.
	Minor gridlines: y intervals	Specify the number of intervals between major y axis gridlines.
	Snap	Snap Marker Size
Snap Sensitivity		Increase or decrease the required distance between the cursor and a feature before a snap marker is displayed.
Point		Snap to Points.
Perpendicular		When drawing a line or polygon, snap to a point on an existing line that forms a right angle between that and the line being drawn.
Line		Snap to Line objects.
Grid		Snap to the Grid intersection points.
Intersection		Snap to Intersections between line objects.

Create New Structural Definition

- Accessed from the Tools menu.
- Define a new structural definition of any of the standard types:

Type	User-Defined Variables
Flexural	<ul style="list-style-type: none"> • Icon • Name • Thickness • Unit weight • First: M_p^+ • First: M_p^- • First: Angle • Second: M_p^+ • Second: M_p^- • Second: Angle
Rigid	<ul style="list-style-type: none"> • Icon • Name • Thickness • Unit weight

Project Details

- Accessed from the Tools menu.
- Specify general details about the current project:

Type	User-Defined Variables
Details	<ul style="list-style-type: none"> • Project Name • Reference N° • Location • Engineer Name • Organization • Comments and Tags

Preferences

- Accessed from the Tools menu.
- The Preferences dialog is the main place to alter the default behaviour of LimitState:SLAB. Within the dialog there are 6 tabs, each controlling a different aspect of the software:

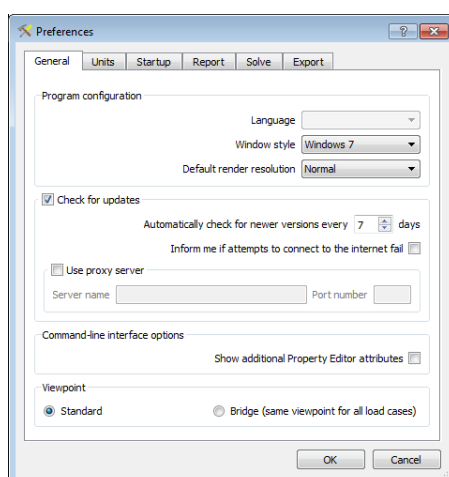


Figure 29 - The LimitState:SLAB Preferences Dialog

Tab	Setting	Description
General	Language	Change the interface language (generally fixed).
	Window Style	Change the appearance of LimitState:SLAB (7 styles available).
	Default display render resolution	Change the quality of the rendering in the viewer pane (low / normal / high).
	Check for updates	Cause LimitState:SLAB to automatically check the LimitState server for updates to the software (requires internet connection).
	Automatically check for new versions every <i>n</i> days	Determines how often the software checks for updates.
	Use Proxy Server	Use a proxy server to access the internet (requires Server Name and Port Number).
	Show additional Property Editor attributes	Display "Object key" information, which can be used in conjunction with the command line interface.
	Viewpoint	Whether to use the same viewpoint for all load cases (Bridge mode) or use a different viewpoint for each load case (Standard mode).

Preferences (contd...)

Tab	Setting	Description
Units	Metric	Use metric units (kN, m etc.) in the project.
	Imperial	Use imperial units (lb, ft etc.) in the project. Also option to use defaults tailored for Imperial Units (in Wizards and built-in materials).
Startup	Show 'Welcome' dialog next time	Display the welcome dialog the next time the software is started.
	Show analysis tip after wizards completed	Display the analysis tip each time a wizard is completed.
	Clear recent files list	Clear the list of recently accessed files.
Report	Header image	Change the default report header image for one of your own choice (requires png format image).
	Footer image	Change the default report footer image for one of your own choice (requires png format image).
Solve	Display detailed information in output window	Cause the output window to display detailed information about the analysis: <ul style="list-style-type: none"> • Load Case * • Partial factors * • Iteration • Variables • Constraints • Solution * • Violation% • Variables added • Constraints added • Time (sec) * Shown by default
	Number of significant figures to display in the answer	Increase or decrease the precision of the answer.
	Animate after solve for problems with less than 1000 nodes	Automatically animate the solution if the problem contains less than 1000 nodes.
	Animate if no more than X nodes	Automatically animate the solution after solve if the target number of nodes is less than X.
	Do not animate after solve	Do not animate the failure mechanism after a solution is found.
Export	Width (pixels)	Change the width of exported images and animations.

Load Case Manager

Define the loads and partial factor sets to be considered during analysis:

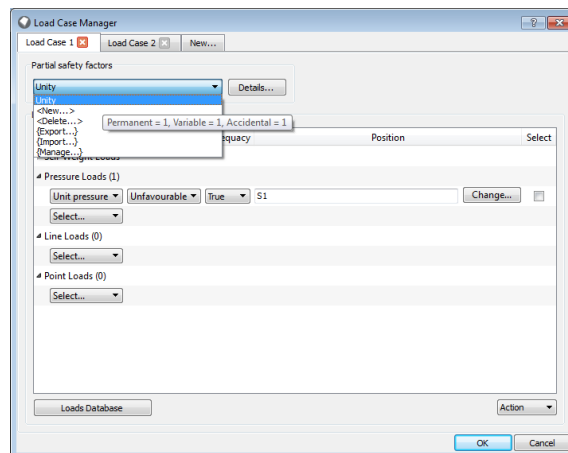


Figure 30 - The LimitState:SLAB Load Case Manager Dialog

Option	Option	Description
Editable partial factors	Unfavourable loads	Define partial factors (multipliers) for unfavourable permanent, accidental and variable loads.
	Favourable loads	Define partial factors (multipliers) for favourable permanent, accidental and variable loads.
	Neutral loads	Define partial factors (multipliers) for neutral permanent, accidental and variable loads.
	Materials	Define partial factors (divisors) for M_p
Predefined partial factors	Unity	All partial factors set to 1.0
Other options	{Import...}	Import partial factor sets from .csv file.
	{Export...}	Export partial factor sets to .csv file.
	<New...>	Define new partial factor sets.
	<Delete...>	Delete a partial factor set.

Report

Chose the sections that are included in the report:

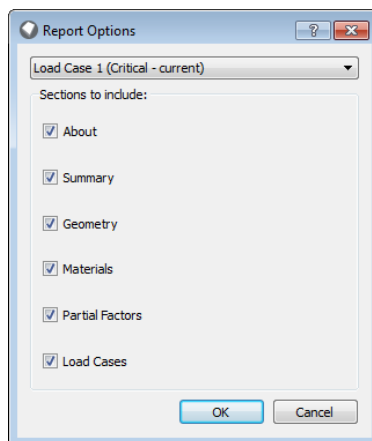


Figure 31 - The LimitState:SLAB Report Options Dialog




Toolbars

The following tables detail the buttons to be found on each of the toolbars within LimitState:SLAB:







File

Icon	Description
	Start a New project.
	Open an existing project.
	Save the current project.




Edit

Icon	Description
	Undo an action.
	Redo an action.
	Delete an object.





Cursor

Icon	Description
	Click Select .
	Rectangle Select .
	Select Cursor .
	Zoom Cursor .
	Pan .
	Rotate .



Draw

Icon	Description
	Draw Rectangle .
	Draw Polygon .
	Draw Line .


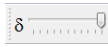
Draw (contd...)

Icon	Description
	Draw Vertex.
	Draw Horizontal construction line.
	Draw Vertical construction line.
	Draw Angled construction line.




Analysis

Icon	Description
	Solve.
<input type="text" value="Factor Load(s)"/>	Change Analysis type.
	Lock / Unlock.



Animation

Icon	Description
	Play animation.
	Magnify displacements.





Zoom

Icon	Description
	Zoom All.
	Zoom In.
	Zoom Out.



Show

Icon	Description
	Show Construction lines.
	Show the Grid.





Show (contd...)

Icon	Description
	Show Nodes .
	Show Supports .
	Show Boundaries .
	Show Solids .
	Show Loads .
	Show Yield-lines .
	Show Post-solve diagrams .
	View in 3D mode.




Select

Icon	Description
	Select a Single object .
	Select objects using a Rectangle .




Rotate 3D

Icon	Description
	Rotate around the x axis .
	Rotate around the y axis .
	Rotate around the z axis .
	Rotate 3D .





View 3D

Icon	Description
	View from the Top .
	View from the Bottom .
	View from the Left .



View 3D (contd...)

Icon	Description
	View from the Right .
	View from the Back .
	View from the Front .

Loads

Icon	Description
	Open the Load Case Manager dialog.
	Insert a Point Load .
	Insert a Line Load .
	Insert a Pressure Load .

Help

Icon	Description
	Help .
	Information .

Context Menus

The following details the context menus to be found when right-clicking the mouse in various areas of the LimitState:SLAB user interface:

Toolbars and Property Editor

Toggle the display of various GUI objects:

Option	Description
Vertex Explorer	Toggle the display of the Vertex explorer.
Boundary Explorer	Toggle the display of the Boundary explorer.
Solid Explorer	Toggle the display of the Solid explorer.
Structural Definitions Explorer	Toggle the display of the Structural Definitions explorer.
Property Editor	Toggle the display of the Property editor.
Geometry Editor	Toggle the display of the Geometry editor.
Output	Toggle the display of the Output pane.

Toggle the display of various toolbars:

Option	Description
File	Toggle the display of the File toolbar.
Edit	Toggle the display of the Edit toolbar.
Cursor	Toggle the display of the Cursor toolbar.

Option	Description
Draw	Toggle the display of the Draw toolbar.
Rotate 3D	Toggle the display of the Rotate 3D toolbar.
Analysis	Toggle the display of the Analysis toolbar.
Animation	Toggle the display of the Animation toolbar.
Zoom	Toggle the display of the Zoom toolbar.
Show	Toggle the display of the Show toolbar.
View 3D	Toggle the display of the View 3D toolbar.
Help	Toggle the display of the Help toolbar.

Viewer Pane (General)

Option	Description
Exit	Exit the context menu.
Select	Change the cursor to select objects.
Click	Select objects by clicking on them.
Rectangle	Select objects by drawing a rectangle around them.
Pan	Change the cursor to allow panning of the model in the viewer pane.
Rotate	Change the cursor to allow rotation of the model in the viewer pane.
Rotate 3D	Rotate the model around all 3 main Cartesian axes.
Rotate About x	Rotate the model around the x axis.
Rotate About y	Rotate the model around the y axis.
Rotate About z	Rotate the model around the z axis.
Zoom	Change the cursor to allow zooming into and out of the viewer pane.
Zoom In	Zoom into the image by 1 increment.
Zoom Out	Zoom out of the image by 1 increment.
Zoom All	Zoom out to the extents of the problem.
View	Change the view of the model.
Top	View the model from the top.
Bottom	View the model from the bottom.
Right	View the model from the right.
Left	View the model from the left.
Front	View the model from the front.
Back	View the model from the back.
View 3D	View the model in 3D.
Zoom All	Zoom out to the extents of the problem.
Show Global Axis	Toggle the display of the global axis.
Perspective	Toggle perspective view (needs to be in conjunction with 3D view).
Render	
High	View the model at high quality (maximum antialiasing).
High Resolution Wireframe	View the model as a high resolution wireframe.
Normal	View the model at normal quality (moderate antialiasing).
Wireframe	View the model as a wireframe.
Low Resolution	View the model in low resolution (minimal antialiasing).
Low Resolution Wireframe	View the model as a low resolution wireframe.
Save Image	Save the current view of the model as a png, jpg, tiff, eps or ps image.

Viewer Pane (Post-Solve Solid)

Option	Description
Moment	View moment diagrams for selected solids.

Structural Definitions Explorer

Option	Description
Delete User Defined Structural Definition	Delete the selected structural definition.
Duplicate Structural Definition	Copy all the properties of the selected structural definition to a new structural definition.
New Structural Definition...	Create a new structural definition from scratch.
Export Structural Definitions...	Save the current structural definitions library as a csv file.
Import Structural Definition...	Import a pre-saved csv structural definitions library into LimitState:SLAB.

Other Explorers

Option	Description
Copy Details	Copy all the properties of the selected object.
Paste Details	Paste all the properties of a previously copied object.
Copy	Copy the current cell.
Paste	Paste the properties of a previously copied cell.
Select All	Select all cells in the table.
Deselect All	Deselect all selected cells.

Geometry Editor

Option	Description
Copy	Copy the current cell.
Paste	Paste the properties of a previously copied cell.

Output Pane

Option	Description
Copy	Copy the currently highlighted text.
Select All	Select all text.
Clear All	Clear all text.

LimitState Ltd

The Innovation Centre
217 Portobello
Sheffield
S1 4DP
UK

+44 (0) 114 224 2240
info@limitstate.com
limitstate.com