TPP DESIGN SOFTWARE



USER GUIDE

Version 1.00

July 2026



Developed by The Steel Construction Institute (SCI)

Unit 2, The E Centre, Bracknell, RG12 1NF, UK
www.steel-sci.com

DISCLAIMER

Although care has been taken to ensure that the calculated values are correct, the user should analyse, verify and assess the software outputs to ensure that they are satisfied with them. The Steel Construction Institute and other parties associated with the Thomas Panels & Profiles Design Software assume no responsibility or liability for errors, misuse of this software or, loss or damage of any kind whatsoever, whether in contract, tort or otherwise, including consequential losses, incurred or suffered by the user or any other person due to use of the software. The software is provided on an “as is” basis and without warranty of any kind. The entire risk as to the quality and performance of the software is with the user. By using the software, the user agrees to this disclaimer and to be bound by the software’s Terms and Conditions, Privacy Policy and Cookie Policy available from the footer of the TPP Design Software.

TABLE OF CONTENTS

1 Getting Started 4
1.1 Introduction 4
1.2 Accessing the software 4
1.3 Login page 4
1.4 Account registration 5
1.5 Quick-start wizard 6
1.5.1 New or existing project 6
1.5.2 Creating a new project 6
1.5.3 Start your design 7
1.5.4 Project Explorer 7
1.5.5 On-screen help and user feedback 7
2 Software scope 8
2.1 Design standards and locale 8
2.2 Building types 8
2.2.1 Structural elements and system types 9
2.2.2 Design criteria 9
3 interface layout overview 10
4 Main Menu 11
5 Project Explorer 12
5.1 Project tree 12
5.1.1 Overview 12
5.1.2 Functionality 13
5.2 Toolbar 14
5.3 Building module 15
5.3.1 Overview of inputs 15
5.3.2 Building orientation 15
5.3.3 Site's snow loading 16
5.3.4 Wind loading 16
5.3.5 Building view 17
5.4 Standard Design sets 18
5.4.1 INPUTS tab 18
5.4.2 AUTOMATE tab 21
5.5 Drifted snow across centres purlin set 22
5.5.1 Overview 22
5.5.2 Additional details on available functionality 23
6 Reports 25

6.1 Project tree 25
6.2 Toolbar 26
6.3 Reports options 27
6.4 Creating the detailed project report 28
7 Automated calculation of wind loading 29
7.1 Overview of process 29
7.2 Performing building wind analysis 29
7.2.1 Performing site wind analysis via SCIPHYR interface 29
7.2.2 Performing building wind analysis via SCIPHYR+ 32
7.3 WIND LOADING WIZARD 32
7.3.1 Defining wind loading regions via the WIND LOADING WIZARD (Edit mode) 33
7.3.2 Assigning wind loading regions to structural elements via the WIND LOADING WIZARD (Read-only mode) 35
8 Wizards 37
8.1 STEPPED WIND LOADS WIZARD 37
8.2 SNOW DRIFT WIZARD 38
8.3 WATER WEIGHT WIZARD 40
9 Snapshots 41
9.1 Capturing snapshots 41
9.2 Snapshot management 41
10 On-screen help and User feedback 43
10.1 On-screen help 43
10.1.1 Tooltips 43
10.1.2 On-hover help 43
10.2 User feedback 43
10.2.1 Range checks 43
10.2.2 Error messages, Warning messages and Notes 44
11 Saving a project 46
11.1 Updating project details 46
11.2 Quick Save 46
11.3 Save project to file 46
12 Account password reset / update 47
12.1 Password reset 47
12.2 Password update 48
13 TROUBLESHOOTING 49

1 GETTING STARTED

1.1 Introduction

This software was developed by the Steel Construction Institute (SCI) for Thomas Panels & Profiles (TPP). It allows to design structural steelwork typically used on industrial buildings to support roofing and cladding, in accordance with the Eurocodes.

Specifically, the software enables the optimisation of cold-formed steel purlins, rails, eaves beams, cantilever purlins and mezzanine floor joists manufactured by Thomas Panels & Profiles, based on the required design loads.

One of the key advantages of the Thomas Panels & Profiles software is its integration with the SCIPHYR wind software, which allows for the automatic calculation of wind loads acting on the steelwork.

Additionally, the software offers extensive reporting capabilities, ensuring full traceability of design calculations for engineering projects.

This User Guide aims to provide users / design engineers with all the necessary information to design structural steelwork using the TPP Design software. It is divided into logical main sections, and it is recommended to read such sections in full to obtain a good understanding of the software's functionality.

1.2 Accessing the software

The TPP Design Software is a web-based application accessible via the following link:

tpp.software.steel-sci.com

To ensure full functionality, it is recommended to use the latest versions of the Microsoft Edge or Google Chrome internet browsers to access the software.

Additional information on how to use the TPP Design Software can be found on the Thomas Panels & Profiles Website on the following page:

www.panelsandprofiles.co.uk/tpp-design-software-user-guidance

1.3 Login page

Users are first directed to the login page shown in Figure 1.1 when accessing the software:

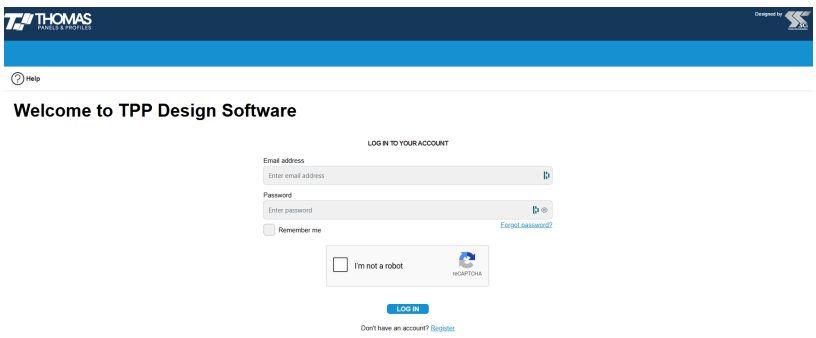


Figure 1.1: Login page

The software account registration and password reset pages are accessible from the login page (see red links in Figure 1.1).

1.4 Account registration

Each user must complete the account registration process via the registration page shown in Figure 1.2 to use the software.

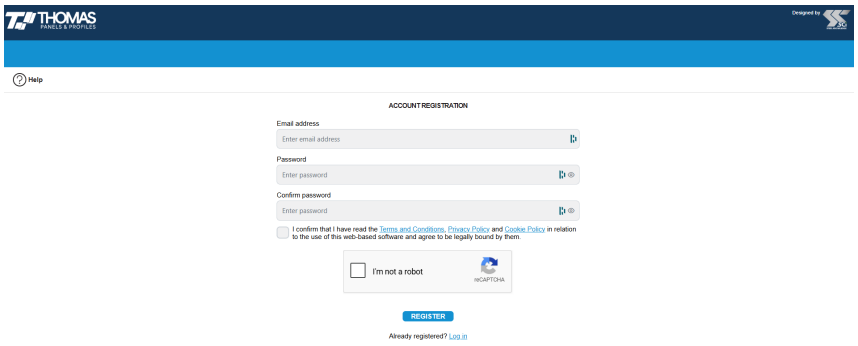


Figure 1.2: Registration page

After submitting the completed registration form, the on-screen message shown in Figure 1.3 is displayed. Users will then receive an automated verification email from SCI's licensing system as shown in Figure 1.4. Users ***must click the verification link*** to activate their account and be given access to the software.

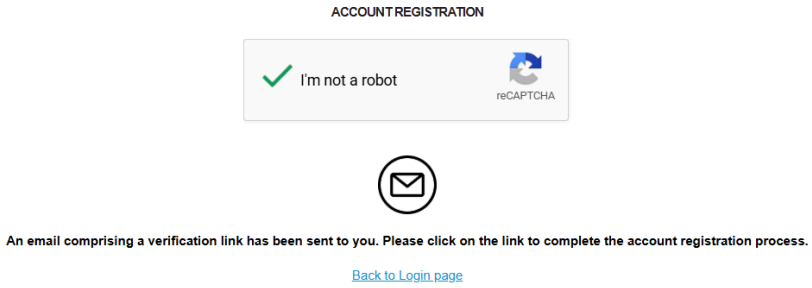


Figure 1.3: Form submission

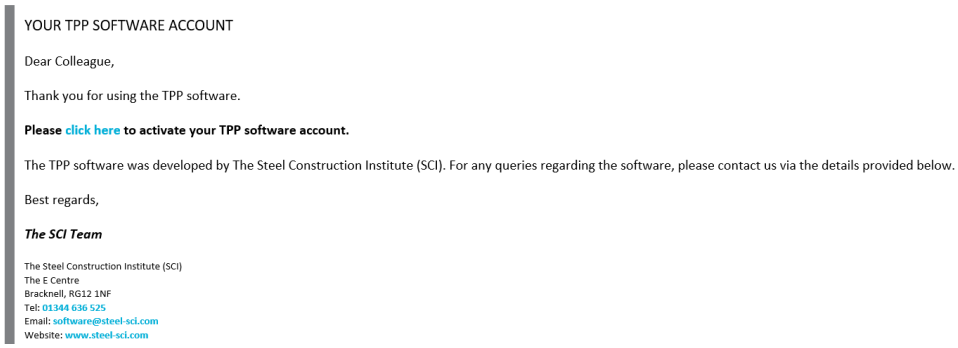


Figure 1.4: Verification email and link

The verification email will be sent from the noreply@steel-sci.com email address. Users should check their junk folders if the verification email does not land in their inbox within 5 min of completing the registration page. It is recommended that users whitelist the above email address.

Upon clicking on the verification link, the on-screen message shown in Figure 1.5 gets displayed, and users are then able to log in to the Thomas Panels & Profiles software using the password that they selected.

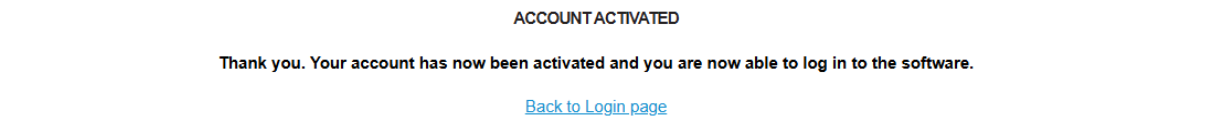


Figure 1.5: Successful account registration

1.5 Quick-start wizard

1.5.1 New or existing project

After login, users are directed to the “Quick-start wizard”, which guides them through the necessary steps to either create a new design project or open an existing project, as shown in Figure 1.6.

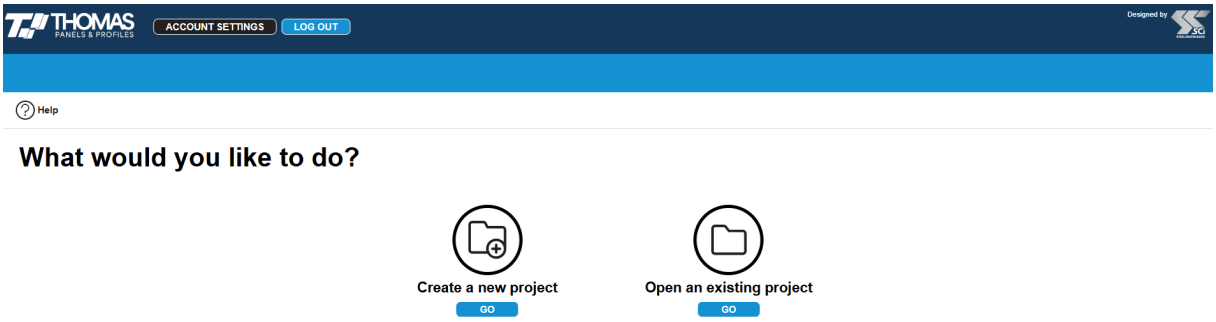


Figure 1.6: New / existing project

When selecting ‘Open an existing project’, users are able to select a previously saved project file via the file picker and are then be able to resume their design.

1.5.2 Creating a new project

When selecting ‘Create a new project’, users initially need to provide project details as shown in Figure 1.7 (inputs marked with * are required). Users must click on the SELECT FOLDER button to select a destination folder and enter the name for the project file in which the project will be saved, before clicking on the OK button. TPP software project files have ‘.scit’ extensions.

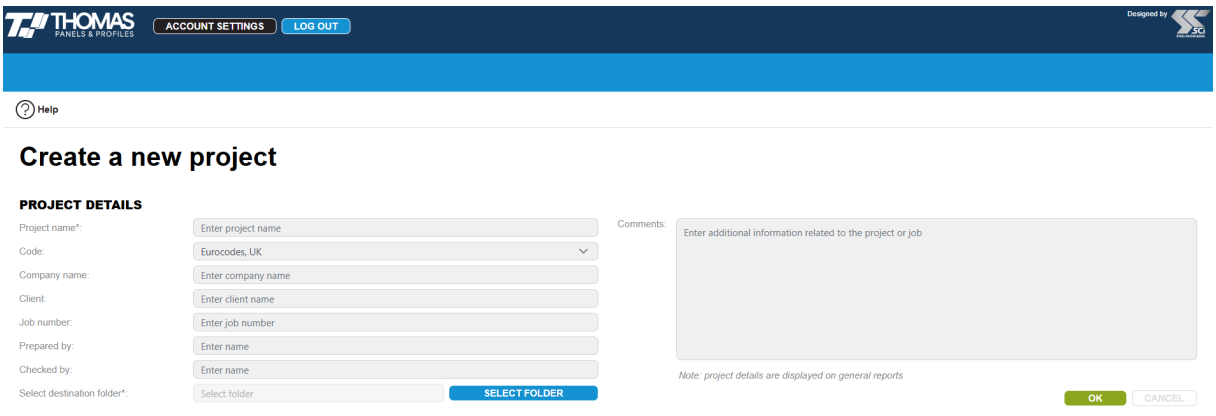


Figure 1.7: Project details

1.5.3 Start your design

The software then provides users with the range of possible options to start their design, as shown in Figure 1.8.

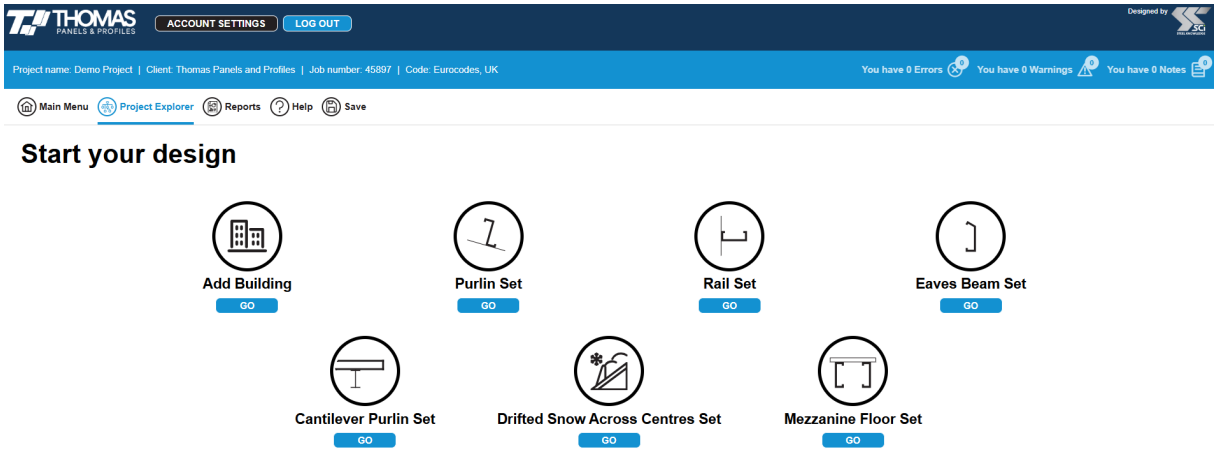


Figure 1.8: Start your design

1.5.4 Project Explorer

Once users have made their selection, they are directed to the 'Project Explorer' as shown in Figure 1.9. The functionality of the Project Explorer is described in more detail in Section 5 of this User Guide.

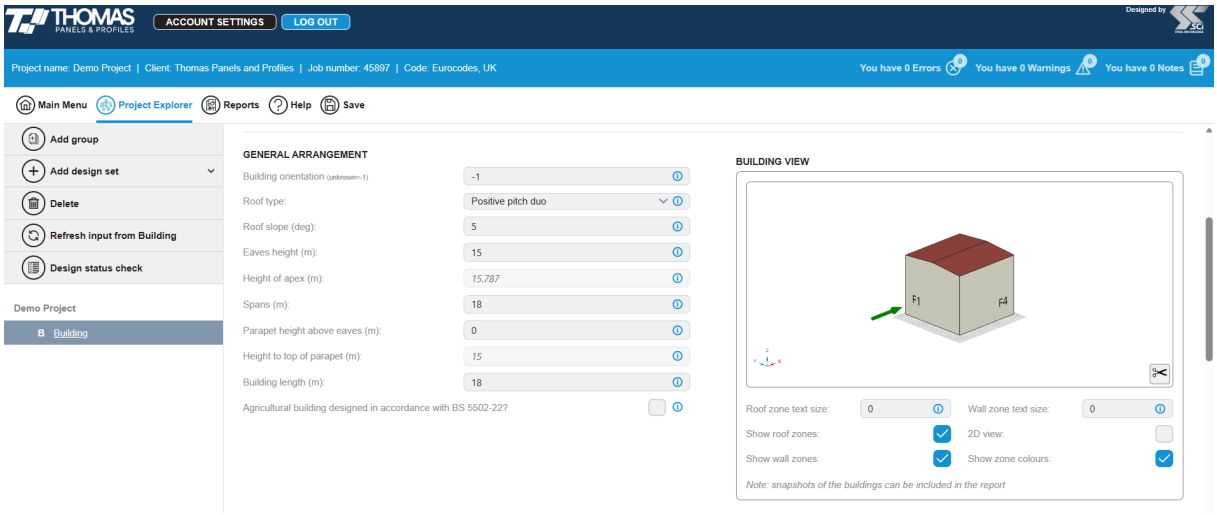


Figure 1.9: Project Explorer

1.5.5 On-screen help and user feedback

Please go to Section 9 of this User Guide for information on tooltips, input range checks and other user feedback provided by the software.

2 SOFTWARE SCOPE

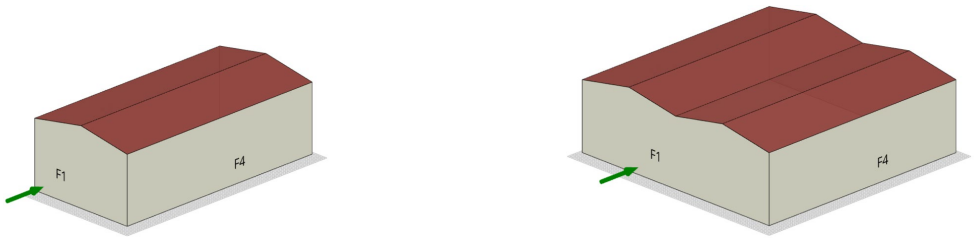
This section describes the scope of the Thomas Panels & Profiles software in terms of both engineering and functionality.

2.1 Design standards and locale

The engineering calculations performed by the software for the design of structural steelwork are in accordance with the *(1st Generation)* ***Eurocodes and relevant UK National Annexes***. Users can use the software to design structural steelwork for ***buildings located within the UK***.

2.2 Building types

The software allows users to define both single and multi-span buildings with rectangular footprints, as illustrated in Figure 2.1.



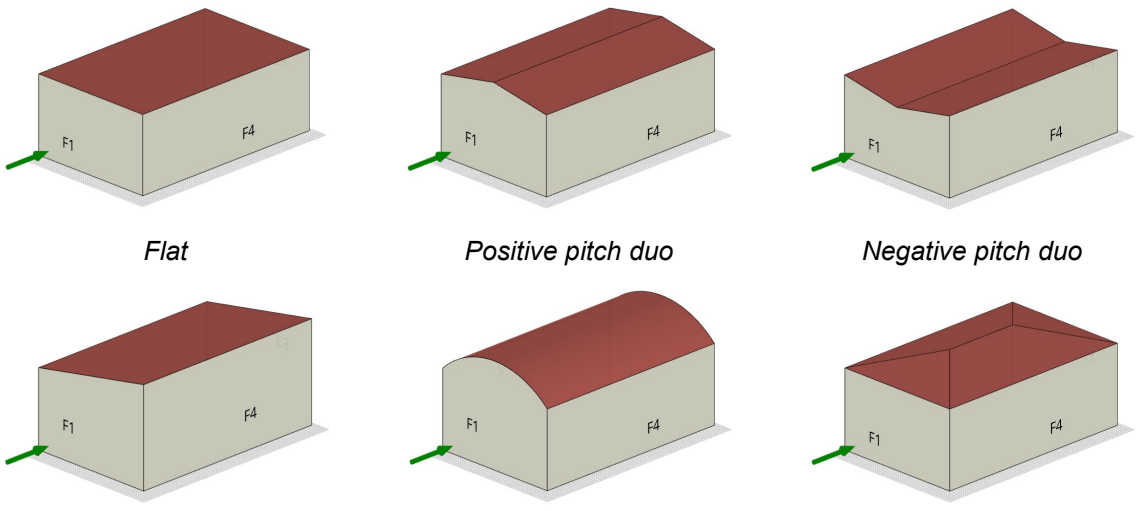
Single-span building *Multi-span building*

Figure 2.1: Building types within software scope

Notes on multi-span buildings:

- Users can specify up to 4 different spans.
- All spans will be the same length as defined for the first span.
- The apex height is defined by the first span and will be the same for all spans.

The software covers six different types of roofs, as illustrated in Figure 2.2.



Flat *Positive pitch duo* *Negative pitch duo*

Mono-pitch *Curved* *Hipped*

Figure 2.2: Building roof types within software scope

The software also allows users to design purlins and rails for Class 1 and Class 2 agricultural buildings (as defined by BS 5502-22) according to the relevant loading.

2.2.1 Structural elements and system types

The range of structural elements and system types covered by the software are indicated in Table 2.1.

System types	Purlins	Rails	Eaves beams	Cantilever purlins	Mezzanine floor joists
Butted Single Span	●	●	●	●	●
Butted Double Span	●	●			●
Sleeved Single Span	●	●		●	

Table 2.1: Structural elements & systems within software scope

2.2.2 Design criteria

The design criteria considered by the software for the various structural elements are indicated in Table 2.2.

Design criteria	Purlins	Rails	Eaves beams	Cantilever purlins	Mezzanine floor joists
Positive loading ULS	●	●	●	●	●
Negative loading ULS	●	●	●	●	
Drifted snow ULS	●		●	●	
Positive loading deflection SLS	●	●	●	●	●

Table 2.2: Design criteria considered by the software

3 INTERFACE LAYOUT OVERVIEW

This section provides an overview of the software interface layout as displayed after login once a project has been created / opened. The key elements of the interface are shown in Figure 3.1.

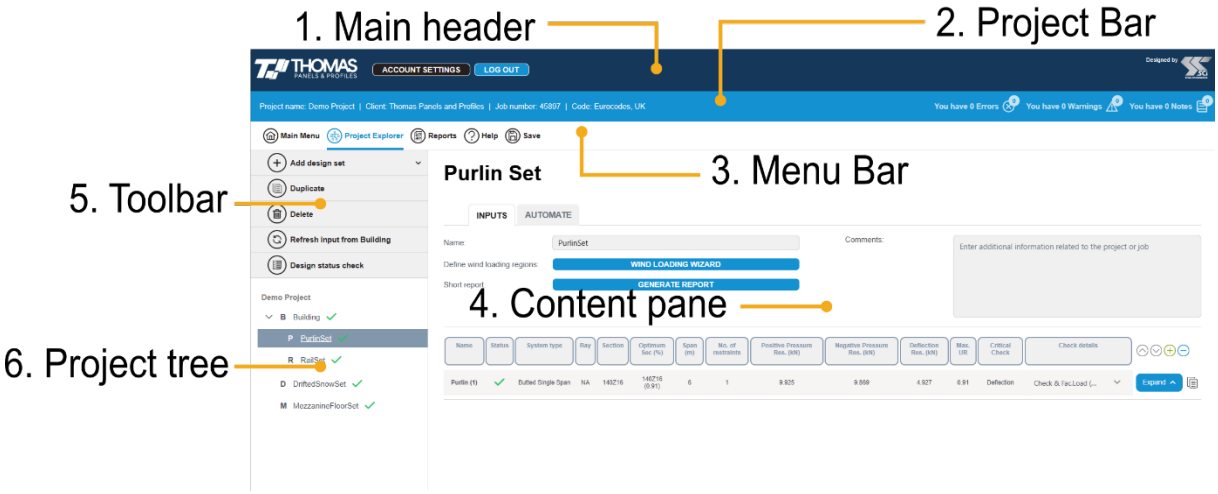


Figure 3.1: Interface layout overview

1. **Main header**: The Main header allows to access the Account settings or Log out.
2. **Project bar**: On the left-hand side, the Project bar provides key information on the currently opened project (as per the specified project details). The right-hand side of the Project bar provides access to the list of Error messages, Warning messages or Notes in relation to the currently opened project, if any.
3. **Menu bar**: The Menu bar provides users with quick access to key software functionality i.e. Main Menu, Project Explorer, Reports, Help and Save functionality.
4. **Content pane**: The contents displayed within the Content pane vary from one section of the software to another. Within the Project Explorer and Reports sections, the Content pane is offset to the right in order to make space for the Toolbar and Project tree on the left-hand side.
5. **Toolbar**: Within the Project Explorer and Reports sections, the Toolbar provides access to additional functionality. The tools displayed within the Toolbar depend on the Project tree selection (see Section 5.2 and Section 6.2 for more details).
6. **Project tree**: Within the Project Explorer and Reports sections, the Project tree allows users to manage Buildings, Design sets and Groups in a hierarchical manner (see Sections 5.1 and 6.1 for more details).

4 MAIN MENU

The Main Menu is always accessible via the software's Menu bar. The Main Menu provides access to all available software functionality as shown in Figure 4.1.

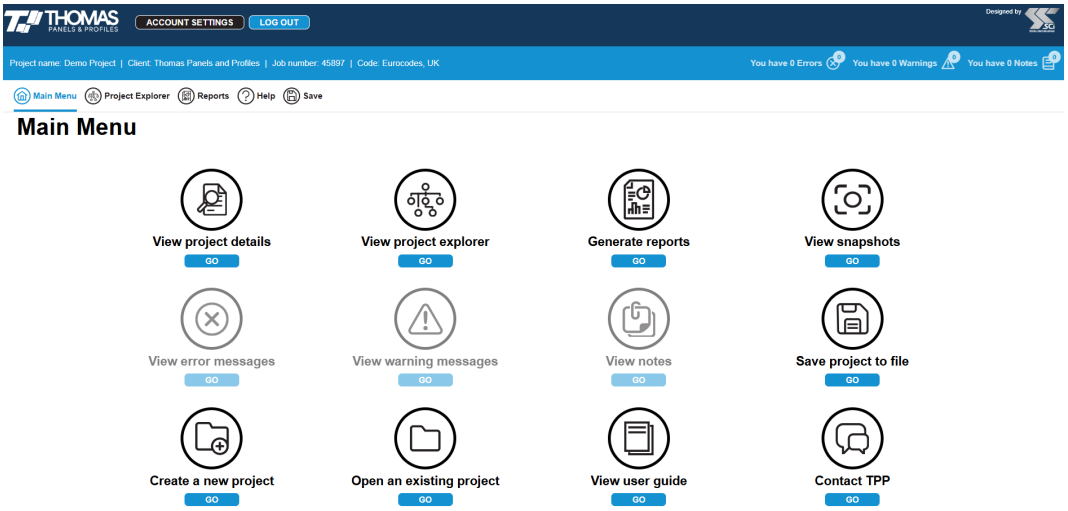


Figure 4.1: Main Menu

- **View project details**: Allows to edit the project details for the currently opened project.
- **View project explorer**: Opens the Project Explorer, the section of the software that allows users to carry out their design.
- **Generate reports**: Opens the Reports section, where users have the ability to customise the project reports.
- **View snapshots**: Allows to manage the snapshots taken by users for inclusion in the project reports.
- **View error messages**: Allows to view the list of Error messages, if any. These can also be accessed at any time via the Project bar.
- **View warning messages**: Allows to view the list of Warning messages, if any. These can also be accessed at any time via the Project bar.
- **View notes**: Allows to view the list of Notes, if any. These can also be accessed at any time via the Project bar.
- **Save project to file**: Opens the file picker to save the project to a new file. This is the equivalent to the 'Save a Copy' functionality within Microsoft Office products.
- **Create a new project**: Allows users to create a new project. Users should ensure that they have saved the currently opened project before proceeding with this option.
- **Open an existing project**: Allows users to open an existing project file. Users should ensure that they have saved the currently opened project before proceeding with this option.
- **View user guide**: Opens the Software User Guidance section of the Thomas Panels & Profiles Website in a new browser tab.
- **Contact Thomas Panels & Profiles**: Opens the Contact Us section of the Thomas Panels & Profiles Website in a new browser tab.

5 PROJECT EXPLORER

The Project Explorer is the main section of the Thomas Panels & Profiles software as it allows users to carry out their design. As indicated in Section 3, it comprises the following three components:

- **Project tree**: The Project tree is located in the bottom-left corner of the interface. Its function is to list all the Buildings, Design sets and Groups created by users within the currently opened project in a hierarchical manner.
- **Toolbar**: The Toolbar provides users with additional functionality. The tools available within the Toolbar depend on the current selection within the Project tree.
- **Content area**: Within the Project Explorer, the Content area displays the design inputs and outputs in relation to the current selection within the Project tree. The functionality available within the Content area depends on the type of item selected within the Project tree i.e. Building, Purlin set, Rail set, Eaves beam set, Cantilever purlin set, Mezzanine floor joist set or Drifted snow across centres purlin set.

5.1 Project tree

5.1.1 Overview

The Project tree lists all the Buildings, Design sets and Groups created by users within the currently opened project in a hierarchical manner. An example of Project tree is shown in Figure 5.1.

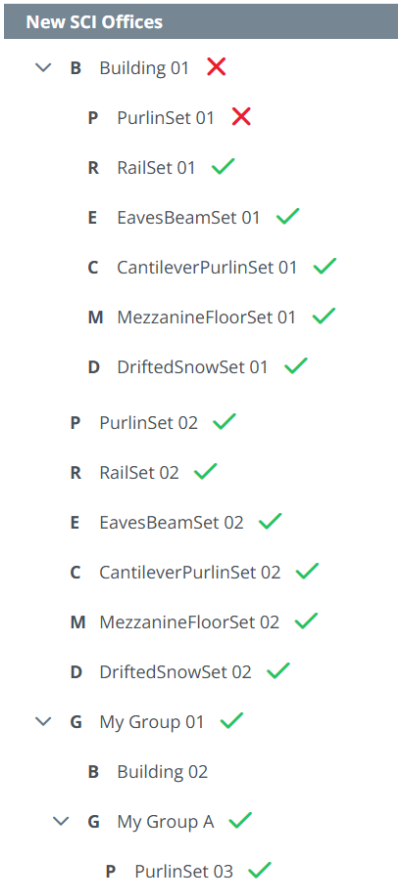


Figure 5.1: Project tree

In the example of Project tree shown in Figure 5.1:

- The currently selected item within the Project tree is the project itself, named ‘New SCI Offices’ (see grey highlight).
- The Design sets numbered ‘01’ (Purlin set, Rail set, Eaves beam set, Cantilever purlin set, Mezzanine floor joist set and Drifted snow across centres purlin set) were added within ‘Building 01’ and are all nested under ‘Building 01’ as a result.
- The Design sets numbered ‘02’ were all added directly under the project and are therefore independent i.e. not nested.
- ‘My Group 01’ was added directly under the project. ‘Building 02’ and ‘My Group A’ were added under ‘My Group 01’ and are therefore nested under ‘My Group 01’.
- ‘PurlinSet 03’ was added directly under ‘My Group A’ and is therefore nested under ‘My Group A’, which is itself nested under ‘My Group 01’.
- The green ticks displayed to the right of items indicate satisfactory designs.
- The red crosses displayed to the right of items indicate unsatisfactory designs.

5.1.2 Functionality

- **Item selection**: Click on an item to select it within the Project tree. You can select the project itself. The current selection gets highlighted in grey.
- **Toolbar**: After selecting an item, the available functionality in relation to that item is provided via the Toolbar. Please see Section 5.2 for more details.
- **Nesting**: To nest an item under another, click on the parent item and then add the child item (via the Toolbar). Click on the arrow to the left of a parent item to collapse/expand the group of nested items. The following rules apply to nesting:
    - A Building cannot be nested under another Building.
    - A Design set (i.e. Purlin set, Rail set, Eaves beam set, Cantilever purlin set, Mezzanine floor joist set or Drifted snow across centres purlin set) cannot be nested under another Design set.
    - A Group, which can essentially be considered as a folder, can only be nested under another Group or a Building.
- **Moving items**: It is possible to ‘click + drag’ to move items within the Project tree. You can nest and de-nest items by moving them within the Project tree. When doing so, the nesting rules above apply.
- **Green tick** ✓: A green tick next to a Design set indicates that all the structural elements within it pass the design criteria. A green tick next to a Group or Building indicates that all the structural elements within the Design sets nested under it pass the design criteria.
- **Red Cross** ✗: A red cross next to a Design set indicates that one or more of the structural elements within it fail the design criteria. A red cross next to a Group or Building indicates that for one or more Design sets nested under it, one or more of the structural elements within those Design sets fail the design criteria.

5.2 Toolbar

The Toolbar provides users with additional functionality as shown in Figure 5.2. The tools available within the Toolbar depend on the current selection within the Project tree, as indicated in Table 5.1.

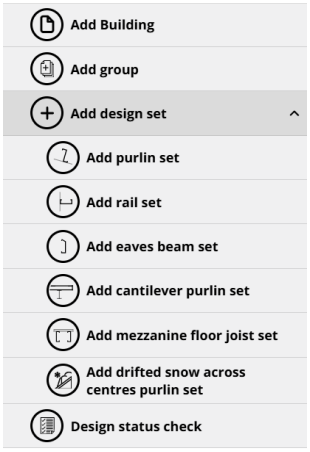


Figure 5.2: Project Explorer toolbar

Toolbar functionality	Project	Building	Group	Purlin set	Rail set	Eaves beams set	Cantilever purlin set	Mezzanine floor joists
Add Building	●		(1)					
Add Group	●	●	●					
Add purlin set	●	●	●	●	●	●	●	●
Add rail set	●	●	●	●	●	●	●	●
Add eaves beam set	●	●	●	●	●	●	●	●
Add cantilever purlin set	●	●	●	●	●	●	●	●
Add mezzanine floor joist set	●	●	●	●	●	●	●	●
Add drifted snow across centres purlin set	●	●	●	●	●	●	●	●
Duplicate				●	●	●	●	●
Rename			●					
Delete		●	●	●	●	●	●	●
Refresh input from Building		●		(2)	(2)	(2)		
Design status check	●	●	●	●	●	●	●	●

(1) Only if not nested within another Group
(2) Only if nested under Building

Table 5.1: Project Explorer toolbar functionality according to Project tree selection

The functionality is self-explanatory for most tools. However, the functionality of some of the tools is further clarified below:

- **Refresh input from Building**:
    - If a Building is selected, clicking on this tool updates the design inputs of the Design sets nested under it which are inherited from the Building (e.g. roof slope, etc.).
    - If a Design set is selected, clicking on this tool updates the design inputs inherited from the Building under which it is nested (e.g. roof slope, etc.).
- **Design status check**:
    - If a Building or Group is selected, clicking on this tool provides an overview of the design status of all Design sets nested under it via a pop-up window.
    - If a Design set is selected, clicking on this tool confirms its design status.

5.3 Building module

The main purpose of the Building module is to allow users to calculate the wind loading applied to the roof and faces of the Building automatically. This, in turn, allows to derive the wind loading to be applied to individual structural elements automatically. The objective of this section is to highlight the aspects of the Building module that are not self-explanatory. The process for the automated calculation of the wind loading via integration with the SCIPHYR and SCIPHYR+ software is described in more detail in Section 7.

5.3.1 Overview of inputs

The inputs listed within the Building module fall into the following categories:

- Inputs required for the calculation of wind loading.
- Inputs related to the calculation of other types of loading, to be inherited by any Design sets nested under the Building. These include:
    - Building type (agricultural building or not)
    - Building class (for agricultural buildings)
    - Roof slope
    - Imposed load
    - Service load
    - Snow load
    - Roof cladding self-weight
    - Wall cladding self-weight

5.3.2 Building orientation

The building orientation considered within the software is defined as the angle between the North-South axis and the axis along the length of the building, as shown in Figure 5.3. If the orientation of the building is unknown, users should set the input value to -1, in which case the most onerous wind loading for the site location will apply.

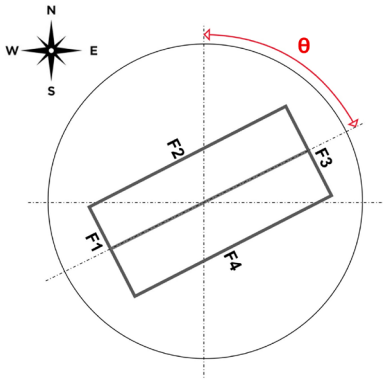


Figure 5.3: Building orientation

5.3.3 Site's snow loading

The snow loading for the site is calculated automatically based on the manually input 'Snow zone' and the site's 'Altitude'. Users can open the UK and ROI snow load map to select the correct 'Snow zone' for the site.

5.3.4 Wind loading

Users have two options for the wind load calculations:

- **Automatic**: Selecting this option allows users to calculate the wind loading automatically via the integration with the SCIPHYR and SCIPHYR+ wind software. To do so, users need to click on the OPEN SCIPHYR button. A link to the gridreferencefinder.com is provided on the interface to help users identify the correct location inputs for their site. When using the 'Automatic' option, the number of wind loading inputs to be provided manually is minimal. Instructions on the use of SCIPHYR are provided in Section 7.
- **Manual**: When selecting this option, users need to manually provide the wind loading inputs that would have otherwise been calculated by SCIPHYR automatically. The use of the manual mode will likely result in more onerous wind loading.

To perform the wind analysis, users must remember to systematically click the CALCULATE button as shown in Figure 5.4. This must be carried out every time inputs are updated.



Figure 5.4: CALCULATE button

The results of the wind loading calculations then become available via the WIND ANALYSIS tab located at the top of the Building module and shown in Figure 5.5. Wind loading is calculated for the four wind directions perpendicular to the building faces (F1, F2, F3, F4), and for each building span.



Figure 5.5: WIND ANALYSIS tab

5.3.5 Building view

Users can visualise the specified Building via the Building view provided on both the INPUTS and WIND ANALYSIS tabs, as shown in Figure 5.6. Once the wind analysis has been performed, the calculated wind zones can be visualised on the Building view, and hovering over a given zone displays the corresponding wind loading.

It should be noted that the Building view provided on the INPUTS tab is mainly aimed at visualising the geometry of the building. Users should go to the WIND ANALYSIS tab to review the wind loading, including via the Building view, where additional options are available e.g. for selection of the relevant wind direction or building span.

Users can move the 3D model as follows:

- **Pan**: Click left on the 3D model and move mouse.
- **Rotate**: Click right on the 3D model and move mouse.
- **Zoom in/out**: Place mouse pointer on the 3D model and scroll up/down.

Users can also take ***snapshots*** of the Building view via the following methods:

- Use the 'alt + s' keyboard shortcut.
- Click the scissors icon ✂ and snip to delimit the snapshot.

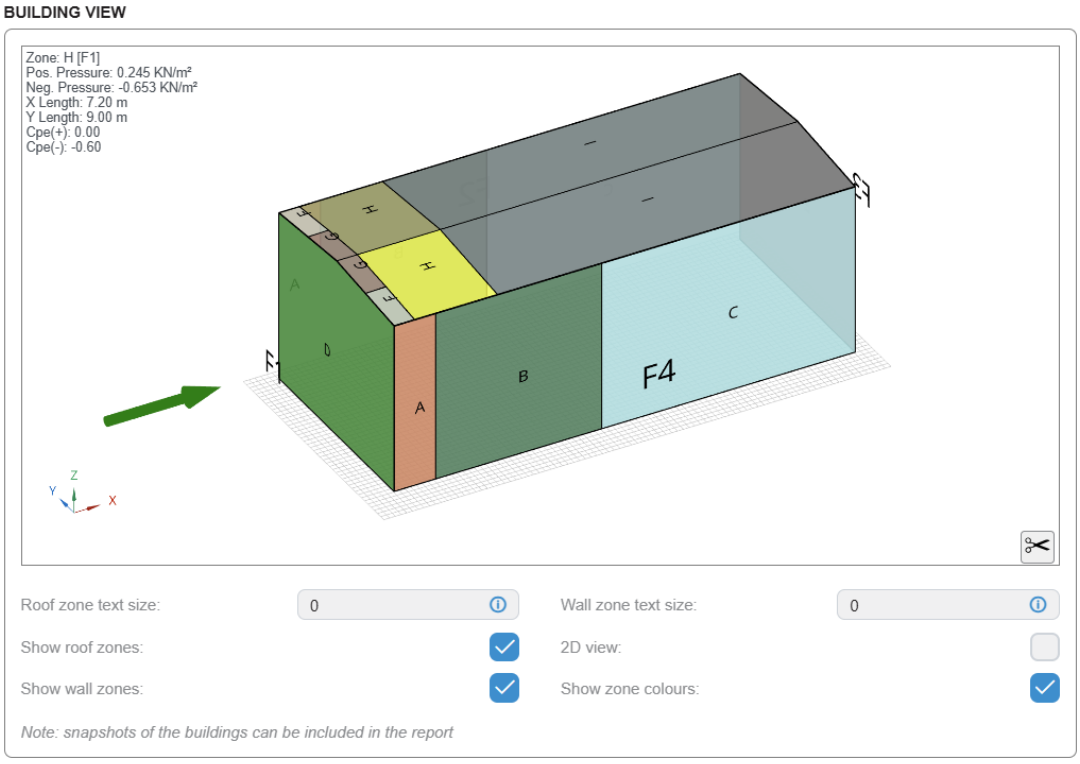


Figure 5.6: Building view

5.4 Standard Design sets

A Design set is a collection of individual structural elements of the same type. The Thomas Panels & Profiles software allows to design different types of structural elements, and there are therefore different types of Design sets. For most Design sets, the available functionality is the same. These are referred to as *Standard* Design sets in this User Guide, and are:

- Purlin set;
- Rail set;
- Eaves beam set;
- Cantilever purlin set;
- Mezzanine floor joist set.

5.4.1 INPUTS tab

Figure 5.7 shows the INPUTS tab for a Purlin set. When using the INPUTS tab, elements have to be designed one at a time. The functionality available from the Content area within the INPUTS tab for “Standard” Design sets is as follows:

- **General Design set inputs & functionality**: The top part of the INPUTS tab comprises the Name and Comments fields for the Design set, which can be edited by users. Users can also generate the “Short” report for the Design set from there, which opens in a new browser tab. For Purlin, Rail and Eaves beam sets nested under a Building, the WIND LOADING WIZARD, which allows to define regions on the roof and faces of the building and automatically calculate the wind loading for these, is also accessible from there. Instructions on the use of this wizard are provided in Section 7.3.
- **List of elements**: When a Design set is added, it initially comprises a single element. Users have access to the following functionality to view and manage the list:
  - **Overview**: Each row provides key inputs and outputs for the corresponding element.
  - **Select**: Click on a row to select an element. The current selection is highlighted in red.
  - **Duplicate**: Click on the ‘Duplicate’ button to duplicate the element. This will also copy the specified design inputs to the duplicated element.
  - **Add**: Click on the ‘Add’ button to add a new element, with the defaults inputs.
  - **Delete**: Click on the ‘Delete’ button to delete the selected element.
  - **Move up**: Click on the ‘Move up’ button to move the selected element upwards within the list.
  - **Move down**: Click on the ‘Move down’ button to move the selected element downwards within the list.
  - **Expand/Close** Expand: Click on the ‘Expand’ button to expand the row and view/edit the design inputs for the selected element. Once the row is expanded, the button changes to ‘Close’ to allow users to collapse the row.
- **Detailed inputs**: Expanding the row for a given element of the Design set displays the design inputs and related functionality for that element, as shown in Figure 5.8. Key available functionality is as follows:
  - **System selection**: When selecting the ‘System type’ from the drop-down menu, the image displayed on the right-hand side is updated to reflect the current system selection.

- **System type optimisation** ⚙: Clicking on the cog located to the right of the 'System type' drop-down menu automatically selects the system providing to the lightest solution. The number of inner and end bays specified in the dedicated input block will be taken into account for this purpose.
- **Section optimisation** ⚙: Clicking on the cog located to the right of the 'Section' drop-down menu automatically selects the lightest section that passes the design criteria. As such, the calculation routine is run as part of this process and users do not need to click the CALCULATE button afterwards.
- **Wind load source**: Users have different options to specify the wind loading applied to the structural element, as listed below:
  - *Uniform*: With this (default) option, users specify the wind loading manually within the input fields.
  - *Stepped*: When this option is selected, the EDIT button displayed under the wind loading input fields provides access to the STEPPED WIND LOADS WIZARD, which allows to calculate an equivalent UDL based on loading from up to 3 different zones overlapping with the element (see Section 8.1 for more details).
  - *Wind analysis*: This option is only available if the Design set is nested under a Building. If so, the EDIT button displayed under the wind loading input fields provides access to the WIND LOADING WIZARD, which allows to assign the element to a previously defined region (on a wall or roof) for which the wind loading has been calculated automatically (see Section 7.3 for more details).
- **Cleat specification**: For Mezzanine floor joist sets, when the selected 'Joist Type' is 'Secondary', users can specify the required 'Cleat Type' to be used and the software will specify the optimal cleat size for the selected 'Cleat Type'. If '*0 x 0 x 0*' is displayed for 'Cleat Dimensions' following calculation, there is no suitable cleat size for the combination of selected joist section and loading within the selected 'Cleat Type' range. This can be overcome by selecting a different joist section: increasing the joist gauge will increase the bearing capacity of the bolts on the joist, and increasing the joist depth will increase the capacity of the cleats.
- **Wizards**: Depending on the type of Design set selected, a range of wizards is available to users to calculate the loading to be applied on the selected element (see Section 8 for more details). These are:
  - *STEPPED WIND LOADS WIZARD*: As mentioned above (see Section 8.1 for more details).
  - *WIND LOADING WIZARD*: As mentioned above (see Section 7.3 for more details).
  - *WATER WEIGHT WIZARD*: This wizard is only available for Eaves beam sets and allows to calculate the self-weight of water, assuming a gutter fill level of 70% (see Section 8.3 for more details).
  - *SNOW DRIFT WIZARD*: This wizard is only available for Purlin and Eaves beam sets and allows to calculate an equivalent UDL for the loading from drifted snow for several scenarios (see Section 8.2 for more details). Note that for the Design sets considered in this section, this wizard considers loading from drifted snow along the span of the selected element. Design against loading from drifted snow in the direction perpendicular to the element's span i.e. "across centres" is covered via the 'Drifted snow across centres purlin set' described in Section 5.5.

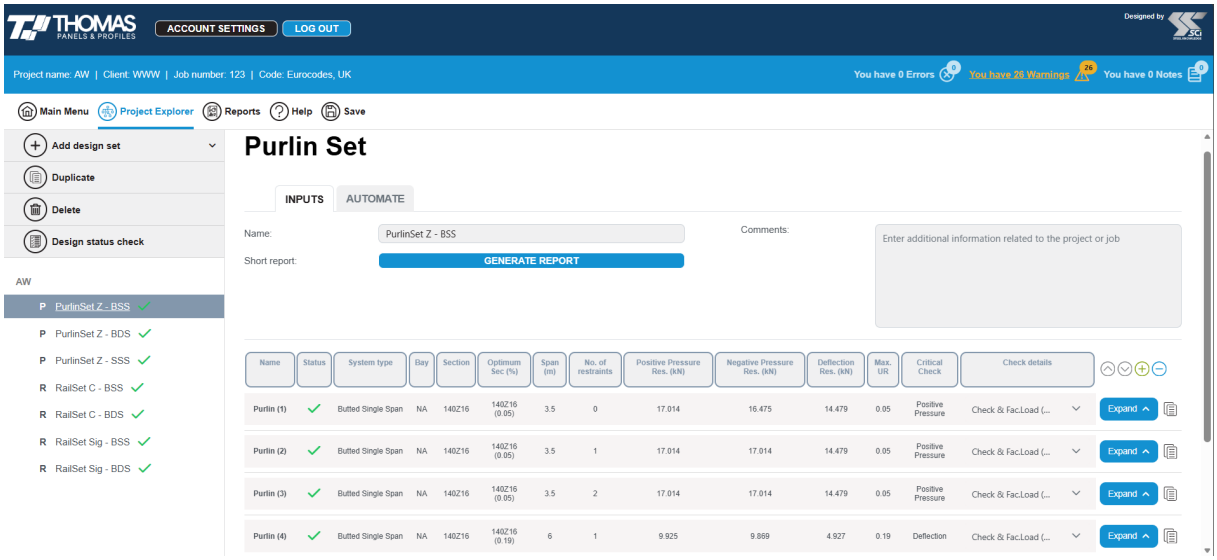


Figure 5.7: Purlin set - INPUTS tab

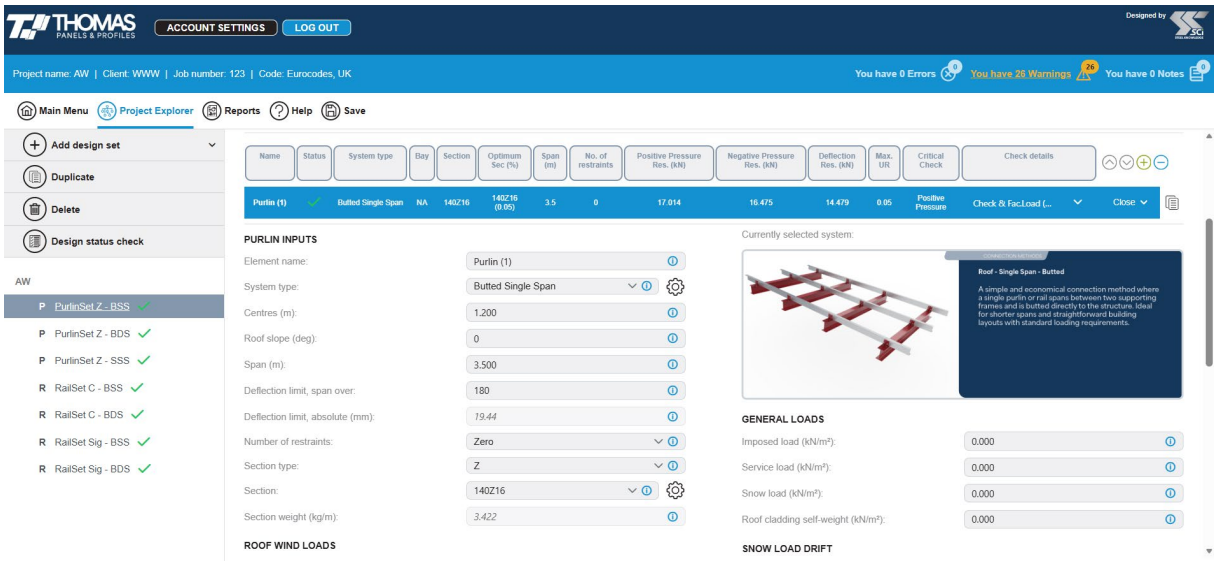


Figure 5.8: Expanded view of inputs for selected element

Users must remember to systematically click the CALCULATE button shown in Figure 5.9 to assess the selected structural element against the design criteria. This must be carried out every time inputs are updated.



Figure 5.9: CALCULATE button

If the element passes the design criteria, a green tick ✓ is displayed within the element's row, and a red cross ✗ is displayed otherwise.

5.4.2 AUTOMATE tab

The AUTOMATE tab runs through all possible solutions and allows users to optimise the section selection for all the elements within the Design set automatically. It is recommended to use the AUTOMATE functionality once the design inputs have been specified for all the elements within the Design set. Users have the following two AUTOMATE options, as shown in Figure 5.10:

- **Whole set**: Selecting this option returns the lightest solution (i.e. sections) whereby all the elements have the same section depth. The selected inputs for 'System type' and 'Section type' will remain unchanged for the elements.
- **Individual element**: Selecting this option returns the lightest solution (i.e. sections) whereby all the elements have been optimised individually i.e. section depths may differ from one element to another. The selected inputs for 'System type' and 'Section type' will remain unchanged for the elements.

After selecting one of the above options, users must click the AUTOMATE button to run the AUTOMATE routine.

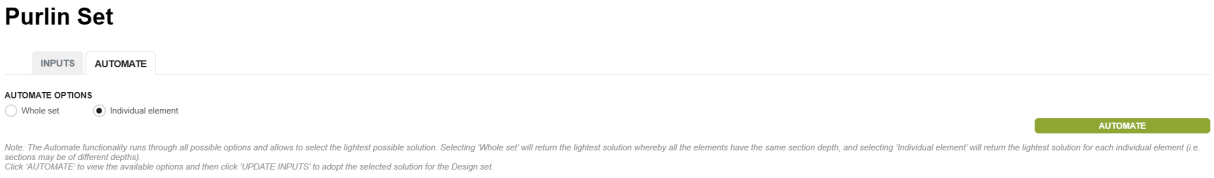


Figure 5.10: Purlin set - AUTOMATE tab initial screen

After clicking the AUTOMATE button, the AUTOMATE results are displayed, as shown in Figure 5.11. Users are then able to update the section selection for each element if needed, and clicking the UPDATE INPUTS button will assign the selected sections to the Design set elements defined within the INPUT tab.

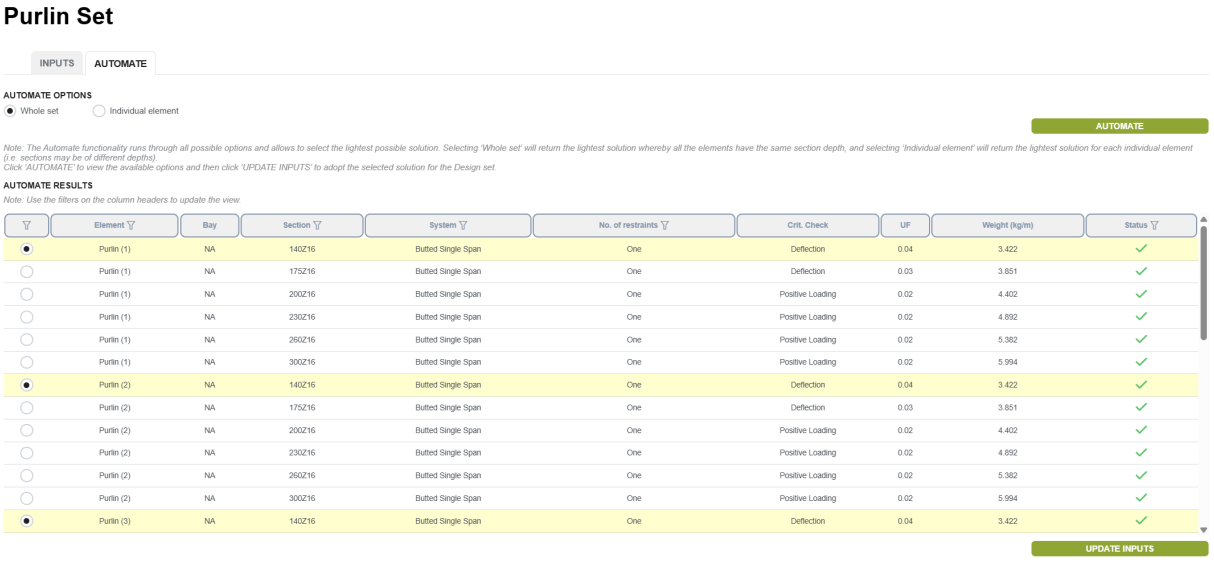


Figure 5.11: Purlin set - AUTOMATE tab results

5.5 Drifted snow across centres purlin set

The ‘Drifted snow across centres purlin set’ allows to optimise the design of purlins for the loading due to drifted snow in the direction perpendicular to the purlin span. The functionality provided by this Design set differs from that of the “Standard” Design sets described in Section 5.4. It is however recommended to read Section 5.4.1 before reading this section, as it provides an overview of the general functionality associated with Design set inputs. This section focusses on the functionality that is specific to the ‘Drifted snow across centres purlin set’.

5.5.1 Overview

- **Step 1**: As for the “Standard” Design sets, users must provide the Design set and purlin related inputs first (see Figure 5.12).
- **Step 2:** Users then provide the ‘Automate settings’ inputs and click on the AUTOMATE button. The solution returned by the optimisation routine is then displayed (see Figure 5.13).
- **Step 3**: Users are then able to edit the solution if needed and click on either the CALCULATE or RE-SPACE button.

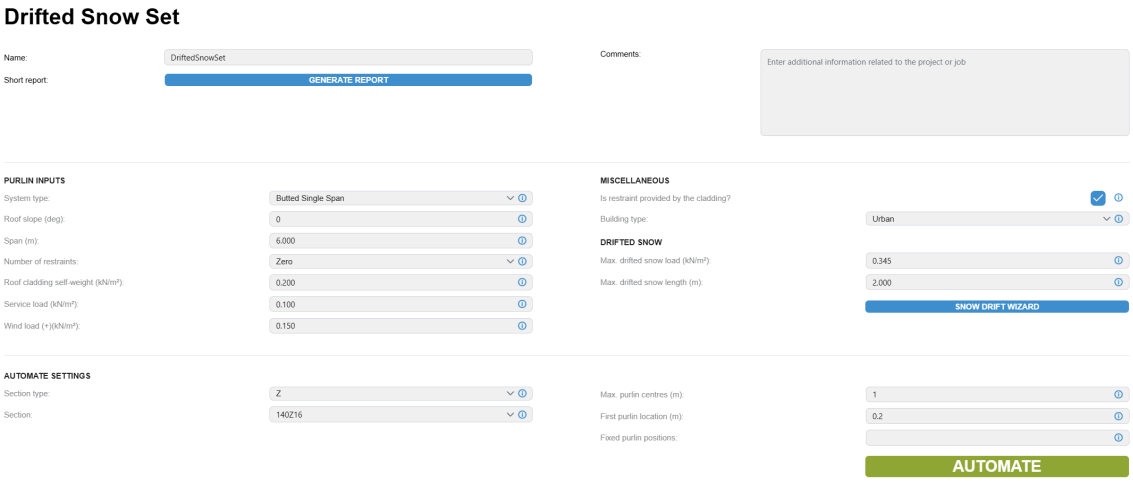


Figure 5.12: Drifted snow across centres purlin set - Initial inputs

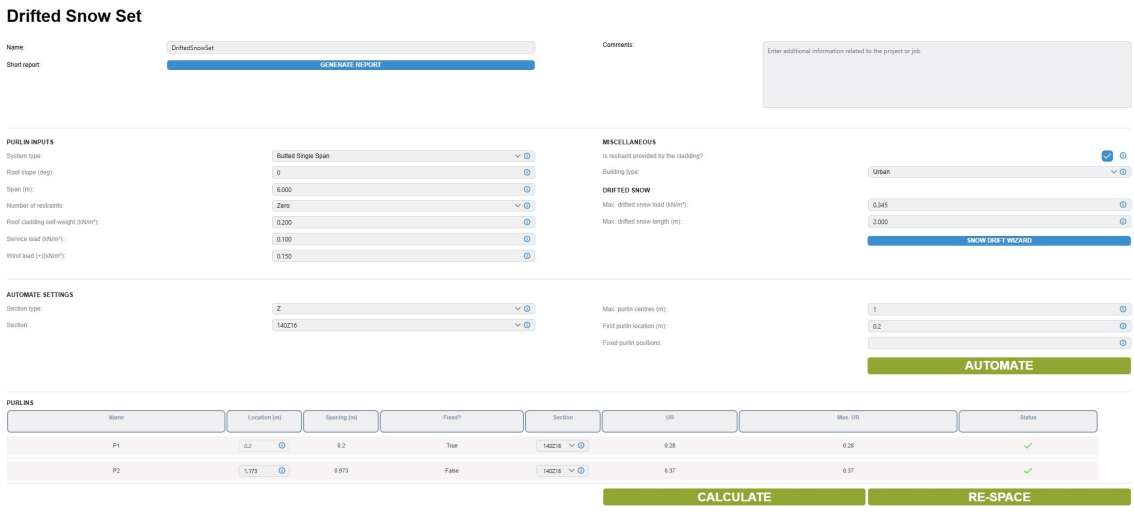


Figure 5.13: Drifted snow across centres purlin set - AUTOMATE results

5.5.2 Additional details on available functionality

- **Optimisation routine**: The optimisation routine returns the lightest solution that passes the design criteria based on the specified loading, section selection and 'Automate settings' inputs i.e. the calculated solution will initially comprise the minimum number of purlins required.
- **SNOW DRIFT WIZARD**: This wizard allows to calculate the maximum drifted snow load and length for several snow drift scenarios. Note that for the 'Snow drift across centres purlin set', this wizard only considers loading from drifted snow in the direction perpendicular to the purlin span i.e. "across centres" (see Section 8.2 for more details).
- **Purlin position:** Purlin position in this Design set is defined as the slope distance from the point of maximum drifted snow load (which is in most cases the end of the rafter at the lower end of the slope).
- **Fixed purlin positions**: It is possible to specify fixed purlin positions e.g. '1,2,3'. In that case, the solution will initially comprise fixed purlins at positions 1 m, 2 m and 3 m.
- **AUTOMATE**: Users can click the AUTOMATE button at any time. If so, the solution is re-calculated based on the current inputs.
- **Updating the solution**: Available functionality to update the solution initially calculated by the optimisation routine is as follows:
    - *Update purlin locations*: Users can update the position of each purlin via the input box provided within each row.
    - *Update purlin sections*: Users can update the section (but not the section type) of each purlin via the drop-down menu provided within each row.
    - *Select*: Click on a row to select a purlin. The current selection is highlighted in red.
    - *Add* ⊕: Click on the 'Add' button to add a purlin "above" the currently selected purlin.
    - *Delete* ⊖: Click on the 'Delete' button to delete the currently selected purlin.
    - *CALCULATE*: After updating the solution, clicking the CALCULATE button will assess the purlins against the design criteria. Note that either the CALCULATE or RE-SPACE button must be clicked after updating the solution.
    - *RE-SPACE*: Clicking the RE-SPACE button will run the optimisation routine, this time considering the specified purlin inputs, 'Automate settings' inputs and any updates made to the purlin sections within the solution. Purlins may be added or deleted automatically as a result. Note that either the CALCULATE or RE-SPACE button must be clicked after updating the solution.
- **Visualising the solution**: A diagram allowing users to visualise the solution is also provided. Additional details are provided in the top left corner when hovering over the loading or the purlins. It is also possible to take snapshots via the 'alt + s' keyboard shortcut or by clicking the scissors icon ✂ and snipping to delimit the snapshot.

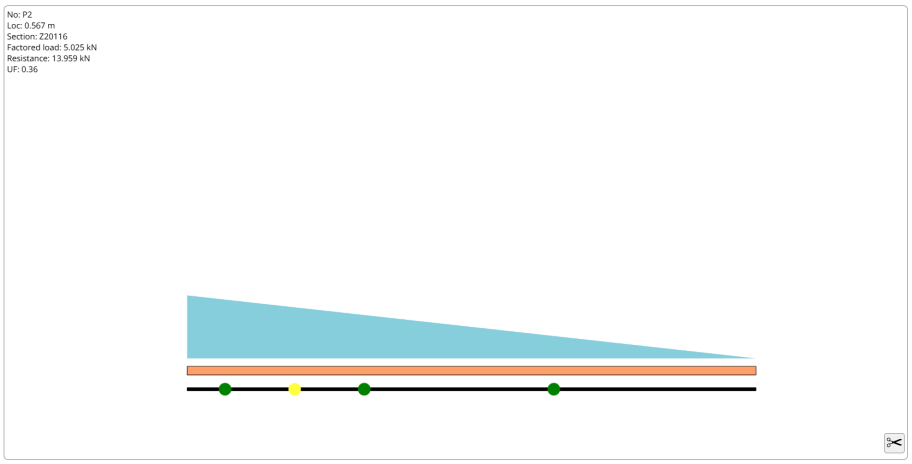


Figure 5.14: Drifted snow across centres purlin set - Visualisation diagram

6 REPORTS

The Reports section allows users to update options for both the short and detailed project reports, and to customise and generate the detailed project report. The layout of the Reports section is similar to that of the Project Explorer, and comprises the following components as shown in Figure 6.1:

- **Project tree**: As per the Project Explorer, the Project tree is located in the bottom-left corner of the interface. It lists all the Buildings and Design sets created by users via the Project Explorer, as well as any commentary and snapshots added within the Reports section. The following should however be noted:
    - Design sets nested under Buildings in the Project Explorer are also nested in the Reports section.
    - Groups are however not displayed in the Reports section. Buildings and Design sets nested under Groups in the Project Explorer are listed “independently” in the Reports section.
- **Toolbar**: The Toolbar provides users with additional functionality. The tools available within the Toolbar depend on the current selection within the Project tree.
- **Content area**: The top of the Content area allows users to update options for both the short and detailed project reports, and also comprises the GENERATE REPORT button. The remaining part of the Content area allows users to provide commentary and select snapshots for inclusion in the detailed project report.

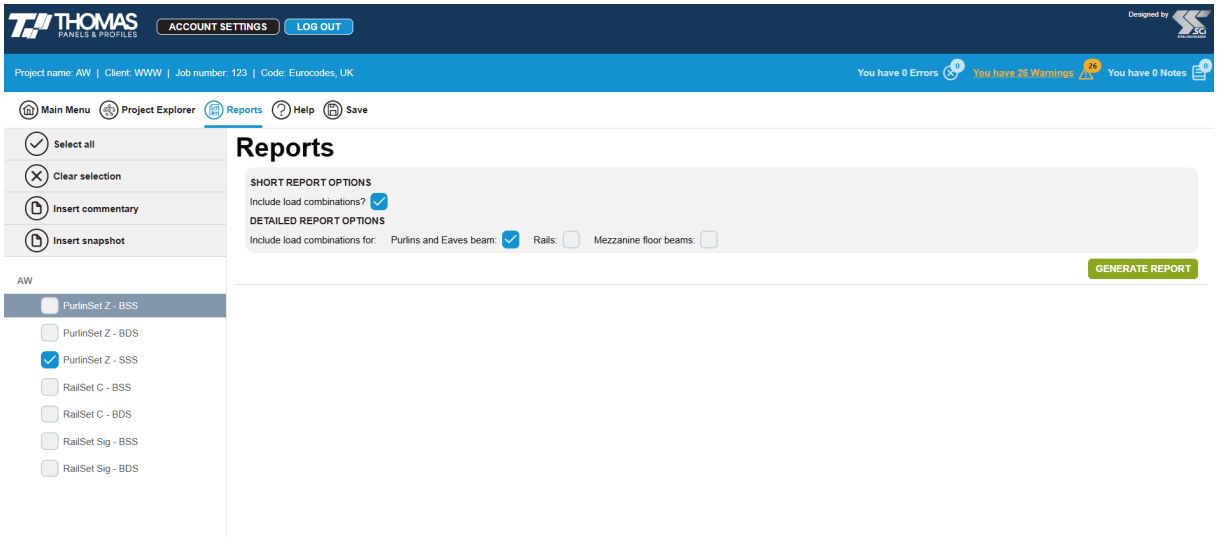


Figure 6.1: Reports section

6.1 Project tree

The functionality of the Project tree within the Reports section is as follows:

- **Inclusion in detailed report** : Tick the box in front of an item to include it within the detailed report.
- **Item selection**: Click on an item to select it within the Project tree. The current selection gets highlighted in grey.

- **Toolbar**: After selecting an item, the available functionality in relation to that item is provided via the Toolbar. Please see Section 6.2 for more details.
- **Nested items**: Click on the arrow to the left of a parent item to collapse/expand the group of nested items.
- **Moving items**: It is possible to 'click + drag' to move items within the Project tree to update the order in which the items will be listed within the detailed project report. The following should be noted:
    - Moving an item comprising nested items (i.e. the parent item) will also move the nested items.
    - You can de-nest an item by moving it elsewhere in the Project tree. It will however not be possible to move it back into the group of nested items.
    - It is not possible to move "independent" items into a group of nested items.

6.2 Toolbar

The Toolbar provides users with additional functionality as shown in Figure 6.2. The tools available within the Toolbar depend on the current selection within the Project tree, as indicated in Table 6.1.

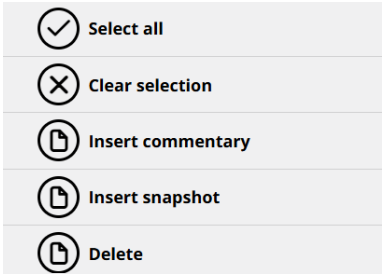


Figure 6.2: Reports section toolbar

Toolbar functionality	Building	Design set	Comment	Snapshot
Select all	•	•	•	•
Clear selection	•	•	•	•
Insert commentary	•	•	•	•
Insert snapshot	•	•	•	•
Delete			•	•

Table 6.1: Reports section toolbar functionality according to Project tree selection

The functionality is self-explanatory for most tools. However, the functionality available for inserting commentary and snapshots in the detailed report is further described below:

- **Insert commentary**: Clicking 'Insert commentary' displays the interface shown in Figure 6.3 within the Content area. Users must specify a 'Name' first, and a range of simple

formatting options are available for the commentary. Click the SAVE button to save and display the commentary within the Project tree, initially nested under the selected item. To update existing commentary, simply select it within the Project tree, update it as required, and click the REPLACE button.

- **Insert snapshot**: Clicking 'Insert snapshot' displays the interface shown in Figure 6.4 within the Content area. Users must select the relevant snapshot via the drop-down menu and click the SAVE button to save and display it within the Project tree, initially nested under the selected item. To update an existing snapshot, simply select it within the Project tree, update it as required, and click the REPLACE button.

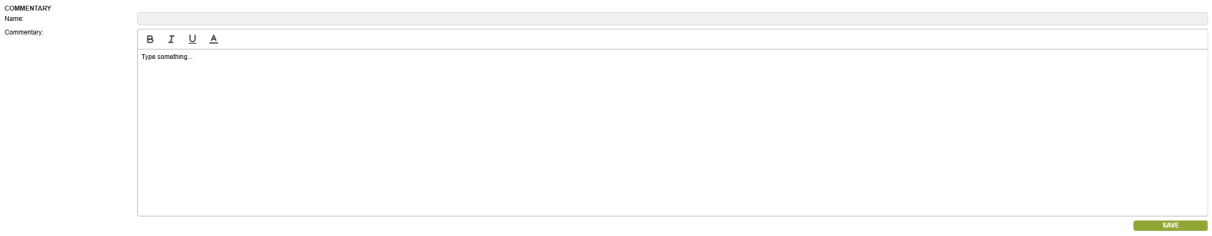


Figure 6.3: Commentary functionality

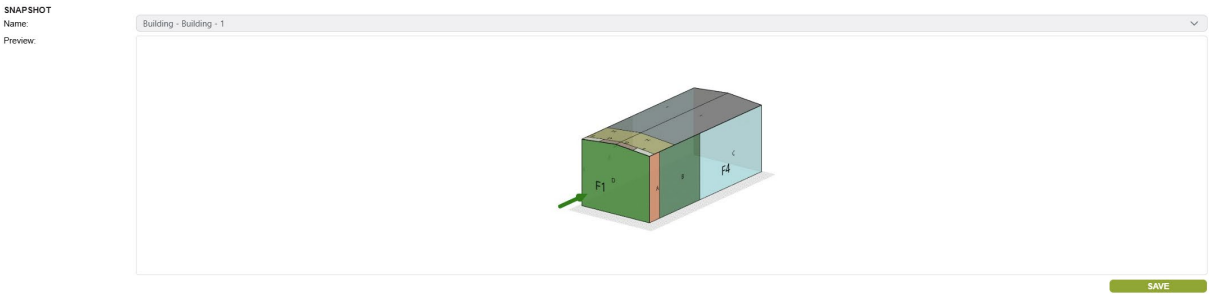


Figure 6.4: Snapshot functionality

6.3 Reports options

The top of the Content area allows users to update options for both the short and detailed project reports, as shown in Figure 6.5.

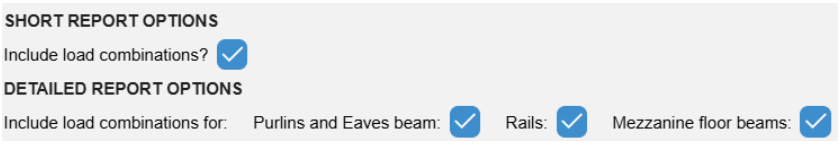


Figure 6.5: Reports options

- **Short report options**: Users have the option to include the load combinations within the short reports that can be generated for individual Design sets from the Project Explorer (see Section 5.4, and more particularly Section 5.4.1). When this option is selected and when viewing a given Design set, only the load combinations relevant to that Design set will be included in the short report.

- **Detailed report options**: Users can decide which load combinations to include within the detailed project report.

6.4 Creating the detailed project report

The steps to create the detailed project report are as follows:

1. Order the items within the Project tree according to the order in which these should be listed in the report.
2. Tick the items to be included in the report.
3. Add commentary and/or snapshots as required.
4. Update the options for inclusion of the load combinations in the report as required.
5. Click the GENERATE REPORT button. The PDF report should open in a new browser tab and can then be saved on the user's computer.

Note: The detailed report includes the project details initially provided by users when creating the project. These can be updated at any time via the Main Menu.

7 AUTOMATED CALCULATION OF WIND LOADING

7.1 Overview of process

This section describes the process for the automated calculation of the wind loading within the Thomas Panels & Profiles software via integration with the SCIPHYR and SCIPHYR+ wind software. The required steps are as follows:

1. Add a Building within the Project Explorer and perform the wind analysis via the SCIPHYR and SCIPHIR+ wind software.
2. Add Design sets under the Building within the Project Explorer.
3. For each Design set, open the WIND LOADING WIZARD (in Edit mode) to define wind loading regions on the roof or walls of the building.
4. For each Design set, assign each structural element to one of the defined wind loading regions for which the wind loading has been calculated automatically. This is also carried out via the WIND LOADING WIZARD, but this time in Read-only mode.

It should be noted that the automated calculation of wind loading is only available for Purlin, Rail and Eaves beam sets nested under a Building.

7.2 Performing building wind analysis

This section should be read in conjunction with Section 5.3 (and more particularly Section 5.3.4). It describes the necessary steps to perform a building wind analysis within the Thomas Panels & Profiles software via the integration with SCIPHYR and SCIPHYR+.

7.2.1 Performing site wind analysis via SCIPHYR interface

After adding a Building within the Project Explorer and viewing its related inputs within the Building module, users can perform the site wind analysis via the SCIPHYR interface shown in Figure 7.1 by clicking the OPEN SCIPHYR button.

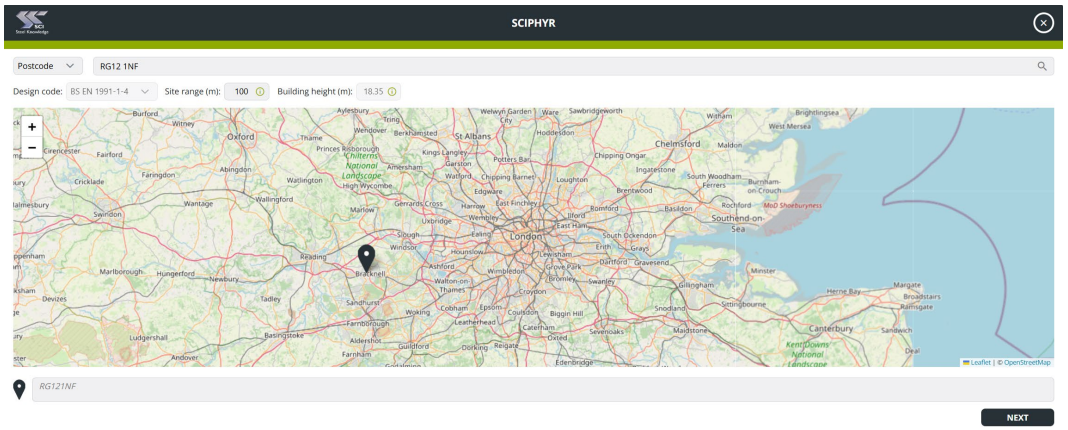


Figure 7.1: SCIPHYR interface: Location selection

STEP 1: Location selection

The first step is to select the correct building location via the interface. To do so, users have the following options:

- **Double click on the map**: The selected location is indicated on the map by the location pin, and the corresponding address gets displayed under the map. Users can zoom in and double click on the map again to refine the selected location if needed.
- **Manual input**: The location of the building can also be specified manually via the input box located above the map, and users have a range of options to specify the location e.g. Postcode, GPS coordinates, Ordnance survey coordinates, etc. After manually entering the location, users must press 'Enter' via their keyboard, and the location pin should then show on the map at the specified location, with the corresponding address displayed under the map. Users can zoom in and double click on the map to refine the selected location if needed.

As part of this first step, users must also input the building height and the site range. The site range allows users to estimate the wind loading when the exact location of the building within the site is not known yet. However, the larger the site range, the higher the conservatism, as SCIPHYR will return the most onerous wind loading within the specified site range. If the exact building location is known, the site range should be set to 0 to reduce conservatism.

Once the correct location and other inputs have been selected, users need to click on the NEXT button to calculate intermediate site wind analysis results.

STEP 2: Review and update intermediate site wind analysis results

After clicking on the NEXT button at the end of STEP 1, SCIPHYR calculates intermediate site wind analysis results based on the inputs provided in STEP 1, as shown in Figure 7.2.

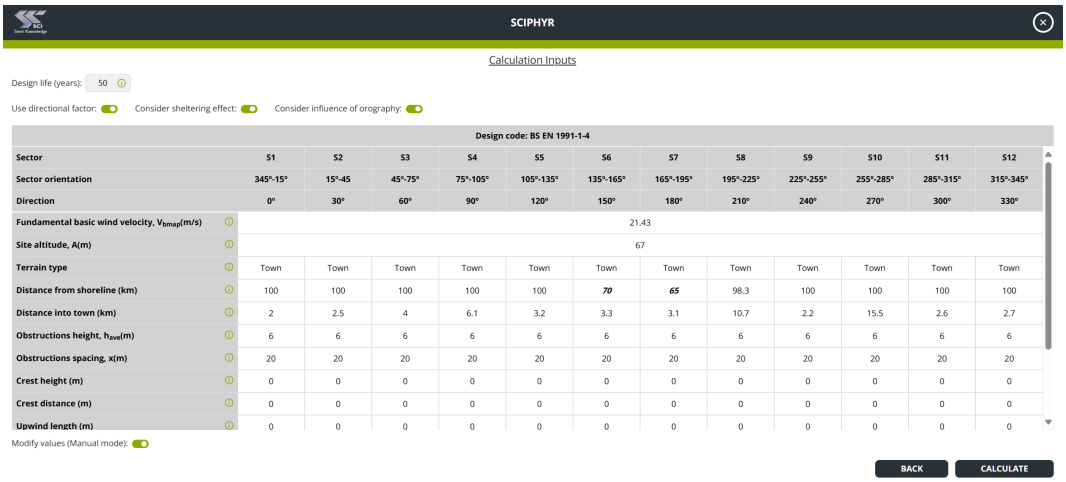


Figure 7.2: SCIPHYR interface: Intermediate results

On this screen, users are able to carry out the following:

- **Input building design life**: If set to 1, additional inputs related to the season factor are displayed.
- **Include** the **directional factor**, **sheltering effect** and/or **orography** in the wind analysis (or not) via the toggles located above the table.
- **Manually update calculated values**: For maximum flexibility, the SCIPHYR integration allows users to manually update the intermediate site wind analysis results calculated by

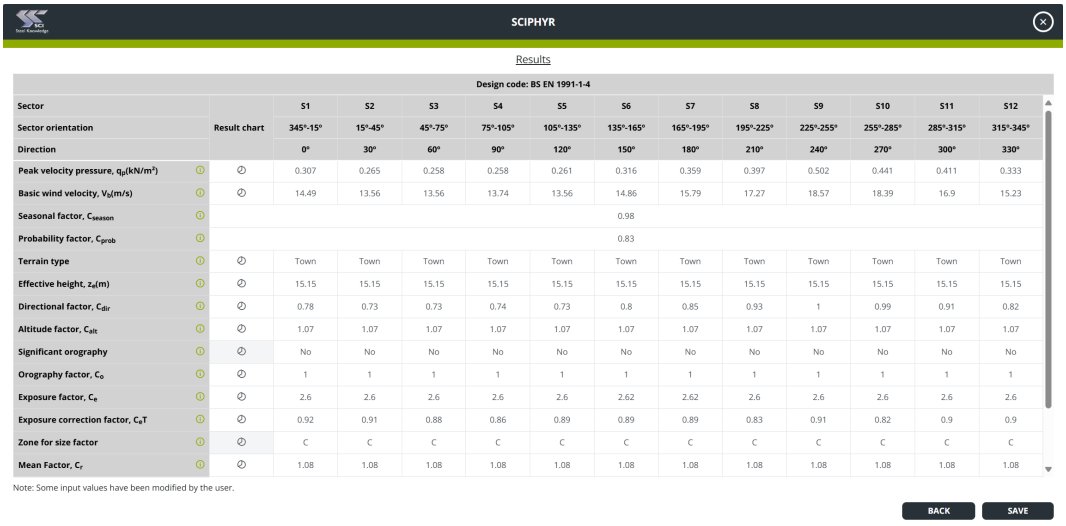
SCIPHYR. To do so, click the 'Modify values' toggle located under the table and update values directly within the table. Manually updated values show in bold italics. Then, click on the white area underneath the table to exit the Edit mode before clicking the CALCULATE button.

Once users are satisfied with the inputs and values from STEP 2, they need to click on the CALCULATE button to proceed with the calculation of the peak velocity pressures and other relevant outputs for the 12 sectors.

STEP 3: Review and export site wind analysis results

After clicking the CALCULATE button at the end of STEP 2, SCIPHYR calculates the peak velocity pressures and other relevant outputs for the 12 sectors based on the inputs provided in STEP 2, as shown in Figure 7.3. Users can view the results in graphical format via a range of charts (by clicking on the chart icons within the table), as shown in Figure 7.4.

After reviewing the site wind analysis results calculated by SCIPHYR, users need to click on the SAVE button to export these to the Building module in order to complete the wind analysis.



Sector		S1	S2	S3	S4	S5	S6	S7	S8	S9	S10	S11	S12
Sector orientation	Result chart	345°-15°	15°-45°	45°-75°	75°-105°	105°-135°	135°-165°	165°-195°	195°-225°	225°-255°	255°-285°	285°-315°	315°-345°
Direction		0°	30°	60°	90°	120°	150°	180°	210°	240°	270°	300°	330°
Peak velocity pressure, q_p(kN/m²)		0.307	0.265	0.258	0.258	0.261	0.316	0.359	0.397	0.502	0.441	0.411	0.333
Basic wind velocity, V_b(m/s)		14.49	13.56	13.56	13.74	13.56	14.86	15.79	17.27	18.57	18.39	16.9	15.23
Seasonal factor, C_{season}							0.98						
Probability factor, C_{prob}							0.83						
Terrain type		Town	Town	Town	Town	Town	Town	Town	Town	Town	Town	Town	Town
Effective height, z_e(m)		15.15	15.15	15.15	15.15	15.15	15.15	15.15	15.15	15.15	15.15	15.15	15.15
Directional factor, C_{dir}		0.78	0.73	0.73	0.74	0.73	0.8	0.85	0.93	1	0.99	0.91	0.82
Altitude factor, C_{alt}		1.07	1.07	1.07	1.07	1.07	1.07	1.07	1.07	1.07	1.07	1.07	1.07
Significant orography		No	No	No	No	No	No	No	No	No	No	No	No
Orography factor, C_o		1	1	1	1	1	1	1	1	1	1	1	1
Exposure factor, C_e		2.6	2.6	2.6	2.6	2.6	2.62	2.62	2.6	2.6	2.6	2.6	2.6
Exposure correction factor, C_eT		0.92	0.91	0.88	0.86	0.89	0.89	0.89	0.83	0.91	0.82	0.9	0.9
Zone for size factor		C	C	C	C	C	C	C	C	C	C	C	C
Mean Factor, C_r		1.08	1.08	1.08	1.08	1.08	1.08	1.08	1.08	1.08	1.08	1.08	1.08

Note: Some input values have been modified by the user.

BACK SAVE

Figure 7.3: SCIPHYR interface: Site wind analysis results

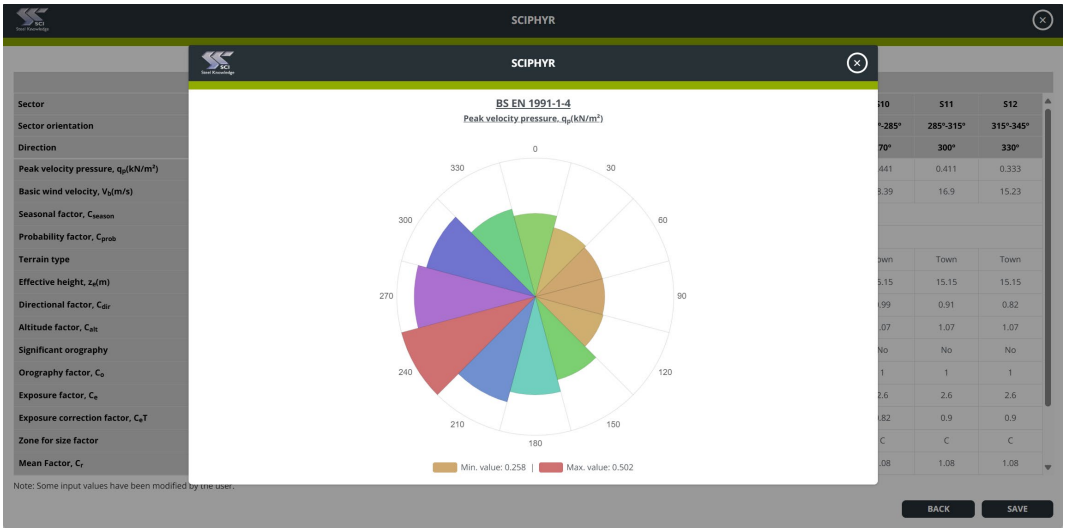


Figure 7.4: SCIPHYR interface: Example of chart

7.2.2 Performing building wind analysis via SCIPHYR+

After performing the site wind analysis via SCIPHYR as per Section 7.2.1, users simply need to double check the inputs specified within the Building module and then click on the CALCULATE button to proceed with the building wind analysis via SCIPHYR+.

The SCIPHYR outputs are used as inputs to the SCIPHYR+ calculations, and the full results of the wind analysis are provided via the WIND ANALYSIS tab of the Building module, as shown in Figure 7.5. The wind loading onto the Building's roof and walls is calculated for the four wind directions perpendicular to the building faces (F1, F2, F3, F4), and for each building span. The site wind analysis results for the 12 sectors calculated by SCIPHYR are also provided within the WIND ANALYSIS tab.

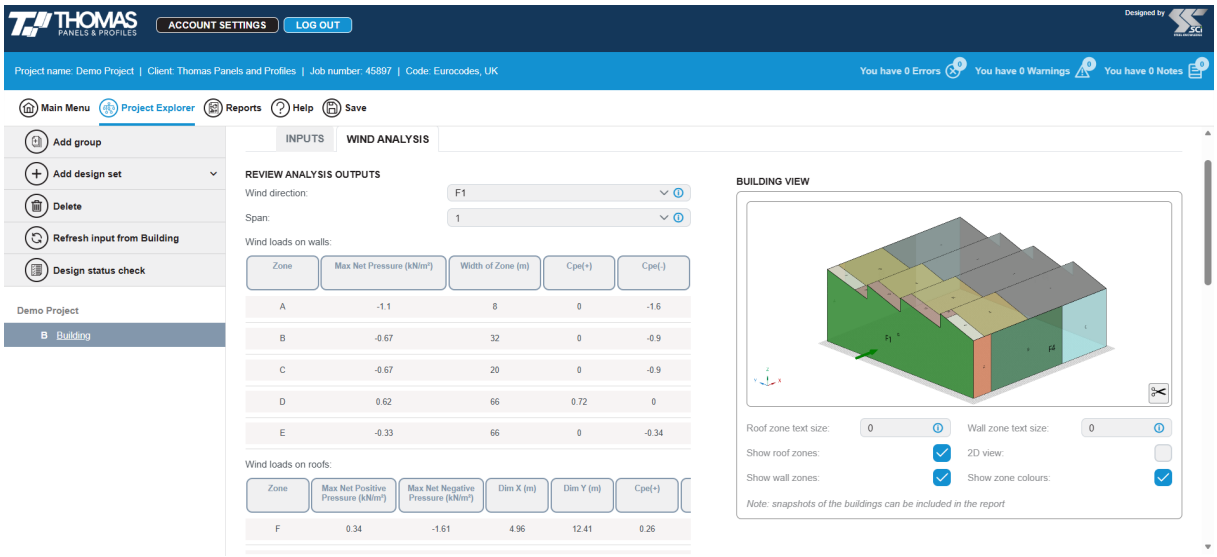


Figure 7.5: Wind analysis results within WIND ANALYSIS tab

7.3 WIND LOADING WIZARD

As per the process outlined in Section 7.1, users need to add a Building to the Project Explorer, add Design sets under that building, and perform the wind analysis for that building as per Section 7.2. Once this has been carried out, users need to do the following via the WIND LOADING WIZARD, which can be accessed via the INPUTS tab of a given Design set:

- Define wind loading regions on the roof or walls of the Building for each of the Design sets nested under the Building.
- For each Design set nested under the Building, assign each structural element to one of the defined wind loading regions for which the wind loading has been calculated automatically.

This section describes how to perform the above tasks.

7.3.1 Defining wind loading regions via the WIND LOADING WIZARD (Edit mode)

In order to define wind loading regions for a given Design set, click the WIND LOADING WIZARD button located at the top of the INPUTS tab. It should be noted that the WIND LOADING WIZARD is only available for Purlin, Rail and Eaves beam sets nested under a Building. In addition, the WIND LOADING WIZARD will only open if the building wind analysis has been carried out for the parent Building.

Overview of interface layout and functionality

The interface of the WIND LOADING WIZARD, as shown in Figure 7.6, varies slightly depending on whether it is opened from a Purlin, Rail or Eaves beam set.

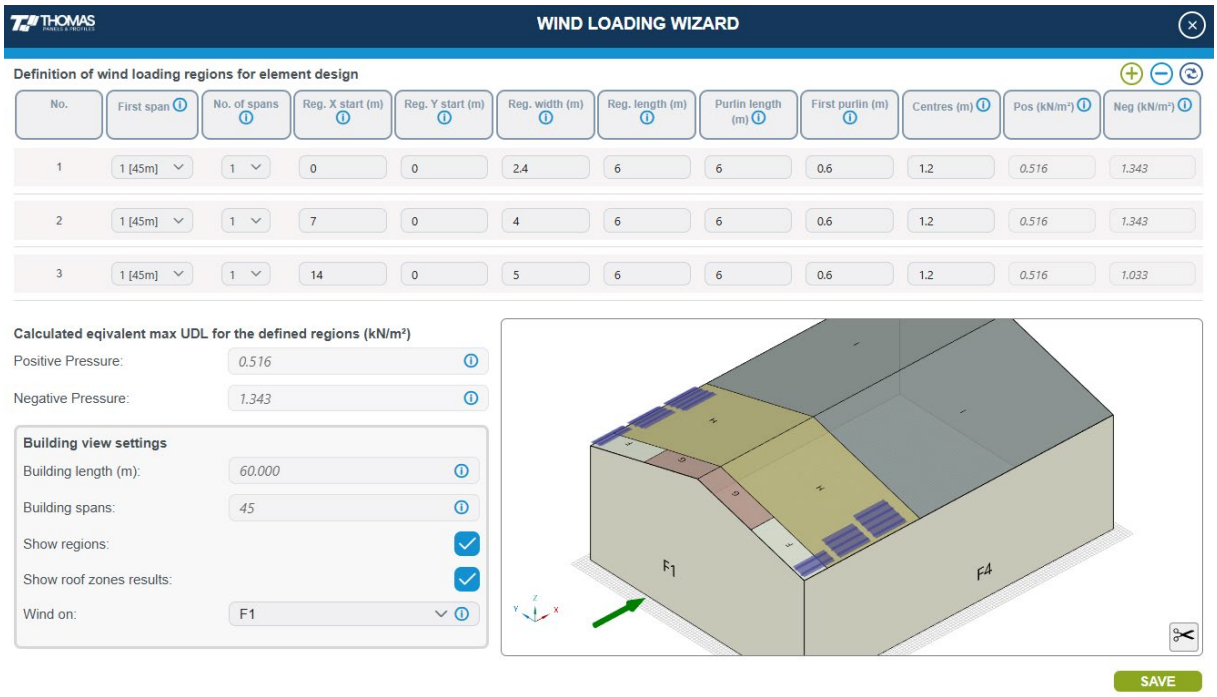


Figure 7.6: WIND LOADING WIZARD - Opened from Purlin set in Edit mode

Its general interface layout and functionality however remain the same from one Design set to another as described below:

- **Add** ⊕: Click on the 'Add' button located in the top-right corner of the interface to add a new region.
- **Delete** ⊖: Click on the 'Delete' button located in the top-right corner of the interface to delete the selected region.
- **Refresh calculations** ⟲: Click on the 'Refresh' button located in the top-right corner of the interface to refresh the wind loading calculations for all defined regions.
- **List of regions**: Once regions have been added, these are listed within the table located in the top half of the interface. Each table row comprises both the inputs defining one region and corresponding loading outputs.
- **Select region**: Click on a table row to select a region. The current selection is highlighted in red.

- **Loading case selection**: Only displayed when the 'Accidental dominant opening check' is carried out. If so, the WIND LOADING WIZARD calculates the wind loading for both the 'Normal' and 'Accidental' cases for the defined regions, and users can review the calculated wind loading for each loading case by selecting the relevant option and clicking on the 'Refresh' button.
- **Building view**: Users can visualise the building and defined wind loading regions via the Building view provided in the bottom-right corner of the interface. Hovering over a given region displays the corresponding wind loading in the top-left corner of the Building view. Please see Section 5.3.5 for more details on the Building view functionality.
- **Maximum wind loading for defined regions**: The maximum positive and negative loading (UDL) across all the defined regions are also calculated by the WIND LOADING WIZARD and displayed on the left of the Building view.
- **Building view settings**: This block, located in the bottom-left corner of the interface, provides users with the length and span(s) of the building as well as some viewing settings.

Overview of process for defining wind loading regions

The required steps are as follows:

1. Add one or more wind loading regions. Once added, these should be listed in the table.
2. Update table row inputs for each region. These define the location and size of each region.
3. Click on the 'Refresh' button to refresh the wind loading calculations for the defined regions.
4. Review the defined regions via the Building view.
5. Once satisfied with the definition of the regions, click on the SAVE button to save the wind loading regions to/for the Design set.

Additional notes and tips for the definition of wind loading regions

- **Definition of wind loading regions**: How to define wind loading regions via the WIND LOADING WIZARD (e.g. the number of regions, their locations, sizes, etc.) is ultimately the decision of the user / design engineer. Given that the objective of defining wind loading regions is to automatically calculate the wind loading to be applied to individual structural elements, it is however advised to define each wind loading region so that it coincides with the tributary areas of a group of structural elements to be designed. The number of wind loading regions to be defined can then be optimised based on the wind zones calculated by SCIPHYR+ (bearing in mind that these are calculated for 4 different wind directions). The wind loading calculated for a given region will depend on the wind zones (calculated for 4 different wind directions) that it intersects.
- **Speeding up the process**: Wind loading regions must be defined for each Design set. However, one way to speed up the process is to define wind loading regions for one Design set, and then duplicate that Design set, as the defined wind loading regions will also be copied (as opposed to creating several Design sets of the same type and then define wind loading regions for those separately).
- **Wind loading regions for Eaves beams**: For Eaves beam, the WIND LOADING WIZARD accounts for the wind loading on both the roof and wall adjacent to the defined region based on the specified distances to the first purlin and rail, respectively.

- **X, Y, Z axes**: For confirmation, the X axis is along the length axis of the Building, the Y axis is along the span axis and the Z axis is along the height axis.
- **Region length on roof**: The 'length' input for regions defined on building roofs is the sloped/curved length.
- **Region length / Element length**: The 'Region length' must be a multiple of the Element 'length'.
- **Region Z start (for Rails)**: For Rail sets, the 'Region Z start' input, which specifies the height at which a given region starts, is for illustration only (for the Building view) as it does not affect the wind loading on walls.
- **Element length**: For a given region, the Element 'length' input, which specifies the length of structural elements within that region, is for illustration only (for the Building view) i.e. this input does not affect the calculated wind loading for the defined region.
- **Centres**: For a given region, the 'Centres' input, which specifies the spacing between structural elements within that region, is for illustration only (for the Building view) i.e. this input does not affect the calculated wind loading for the defined region.

7.3.2 Assigning wind loading regions to structural elements via the WIND LOADING WIZARD (Read-only mode)

For a given Design set, once the wind loading regions have been defined as per Section 7.3.1, the next and final step is to assign each structural element within that Design set to a defined region, for which the wind loading has been calculated automatically.

As indicated in Section 5.4 (and more particularly Section 5.4.1), when viewing the detailed inputs for a given structural element within the Design set, the WIND LOADING WIZARD in Read-only mode can be accessed by selecting 'Wind analysis' for the 'Wind load source' input and clicking the EDIT button displayed under the wind loading input fields.

As a reminder, the WIND LOADING WIZARD is only available for Purlin, Rail and Eaves beam sets nested under a Building. In addition, wind loading regions must be defined as per Section 7.3.1 first, before assigning automatically calculated wind loading to structural element as described in this section.

The WIND LOADING WIZARD in Read-only mode is shown in Figure 7.7. In Read-only mode, users have the following two options to assign wind loading to the considered structural element:

- **Select relevant region + click APPLY SELECTED**: Users can select the relevant wind loading region by clicking the selection button within the relevant table row. Clicking the APPLY SELECTED button will export the automatically calculated wind loading for the selected region to the structural element's detailed inputs.
- **Click APPLY MAX.**: Clicking on the APPLY MAX. button will export the maximum calculated wind loading across all the defined regions to the structural element's detailed inputs.

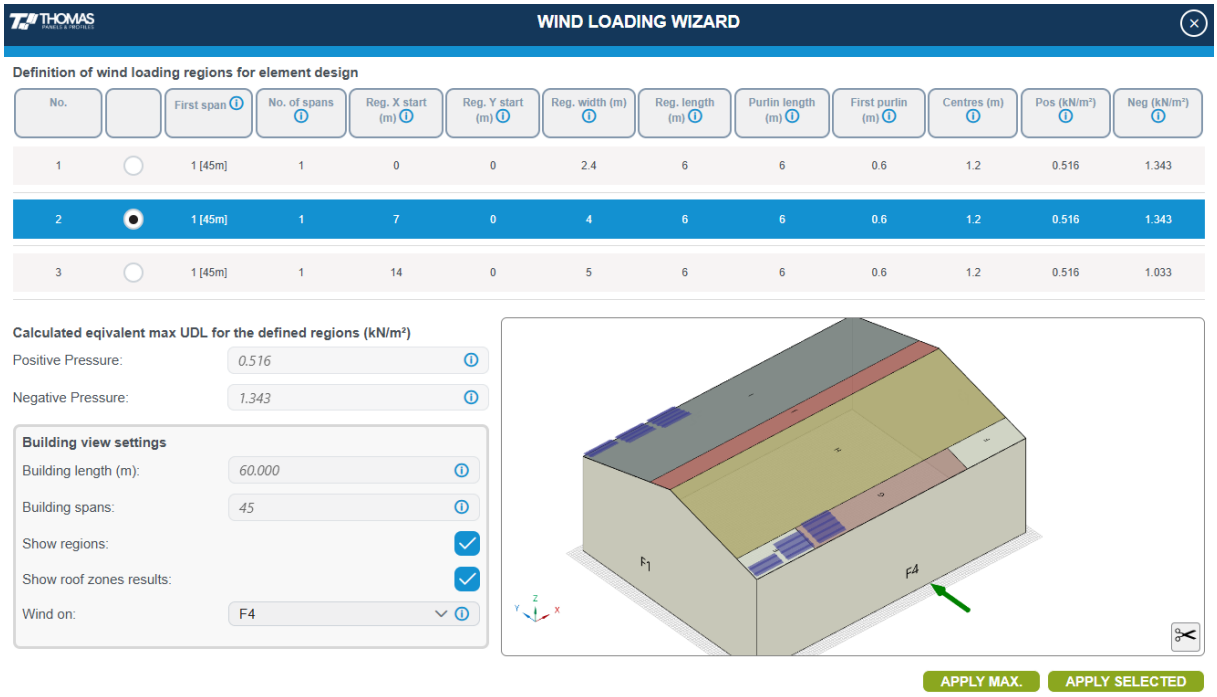


Figure 7.7: WIND LOADING WIZARD - Opened from Purlin set in Read-only mode

Additional notes and tips for assigning automatically calculated wind loading to structural elements

- **Speeding up the process**: Wind loading must be specified for each structural element. However, it may be possible to speed up the process by duplicating a structural element once all its inputs have been specified - including the wind loading - as all the defined inputs will also be copied.
- **Loading case**: The process for assigning automatically calculated wind loading is the same for the 'Normal' and 'Accidental' loading cases (when the 'Accidental dominant opening check' is carried out). When opening the WIND LOADING WIZARD in Read-only mode, the relevant loading case is selected by default.

8 WIZARDS

The Thomas Panels & Profiles software comprises a number of wizards to facilitate the input of the various types of loading acting on structural elements. This section provides an overview of those wizards, except the WIND LOADING WIZARD, which is described in detail in Section 7.3.

8.1 STEPPED WIND LOADS WIZARD

The STEPPED WIND LOADS WIZARD, as shown in Figure 8.1, is available for Purlin, Rail and Eaves beam sets. As indicated in Section 5.4 (and more particularly Section 5.4.1), when viewing the detailed inputs for a given structural element within a Design set, this wizard can be accessed by selecting 'Stepped' for the 'Wind load source' input and clicking the EDIT button displayed under the wind loading input fields.

The STEPPED WIND LOADS WIZARD allows to calculate an equivalent UDL based on loading from up to 3 different zones overlapping with the element.



Figure 8.1: STEPPED WIND LOADS WIZARD

How to use the wizard:

1. **Provide inputs for each zone**: For each of the three zones, enter the Negative pressure, Positive pressure and Zone length. Note that the sum of the zone lengths must be equal to either twice the span in the case of a single-span system, or the sum of the inner and end bay spans for a double-span system.

2. **Click CALCULATE**: Click on the CALCULATE button to calculate the equivalent UDL based on the provided inputs for the three zones.

3. **Review/visualise loading + take a snapshot**: Users can review the loading via the diagram displayed at the bottom of the wizard's interface. Hovering over a given loading displays relevant details in the top-left corner of the diagram. It is also possible to take snapshots of the diagram via the 'alt + s' keyboard shortcut, or by clicking the scissors icon ✂ and snipping to delimit the snapshot.

4. **Select relevant loading**: For single-span systems, users are given the option to select the calculated loading for either the left or right span.

5. **Click APPLY**: Click on the APPLY button to export the selected wind loading to the structural element's detailed inputs.

8.2 SNOW DRIFT WIZARD

The SNOW DRIFT WIZARD is available for Purlin, Eaves beam and Drifted snow across centres purlin sets. As indicated in Sections 5.4 and 5.5, this wizard can be accessed by clicking the SNOW DRIFT WIZARD button. The SNOW DRIFT WIZARD allows users to calculate the loading from drifted snow for a variety of cases.

It should be noted that the snow drift scenarios/cases covered by the SNOW DRIFT WIZARD depend on the Design set from which it is accessed:

- **Purlin and Eaves beam sets**: When accessed from a Purlin or Eaves beam set, the SNOW DRIFT WIZARD considers loading from drifted snow along the span of purlins / eaves beams and the following scenarios/cases are covered:
  - Parapet at gable;
  - Obstruction on roof (Left side);
  - Obstruction on roof (Right side);
  - Abrupt change of roof height.

  For the above scenarios/cases, the SNOW DRIFT WIZARD allows to calculate an equivalent UDL for the loading from drifted snow.
- **Drifted snow across centres purlin set**: When accessed from a Drifted snow across centres purlin set, the SNOW DRIFT WIZARD considers loading from drifted snow in the direction perpendicular to the purlin span and the following scenarios/cases are covered:
  - Multi-span drift, Valleys (Left)
  - Multi-span drift, Valleys (Right);
  - Parapet at eaves;
  - Significant obstruction on roof (Left);
  - Significant obstruction on roof (Right);
  - Abrupt change of roof height;
  - Snow on canopy.

  For the above scenarios/cases, the SNOW DRIFT WIZARD allows to calculate the maximum drifted snow load and length.

How to use the wizard:

1. **Click SNOW DRIFT WIZARD**: Click on the SNOW DRIFT WIZARD button to access the wizard.
2. **Select relevant snow drift scenario/case**: The SNOW DRIFT WIZARD initially provides users with a range of snow drift scenarios/cases, as shown in Figure 8.2. Select the relevant scenario/case. The wizard will then display a diagram of the selected scenario/case and relevant inputs.
3. **Provide relevant inputs**: Provide the relevant inputs for the selected scenario/case, as shown in Figure 8.3.
4. **Click CALCULATE**: Click on the CALCULATE button to calculate the drifted snow loading outputs.
5. **Take a snapshot**: If needed, it is possible to take a snapshot of the wizard by clicking the scissors icon ✂.
6. **Click APPLY**: Click on the APPLY button to export the calculated drifted snow loading outputs to the Design set / structural element.

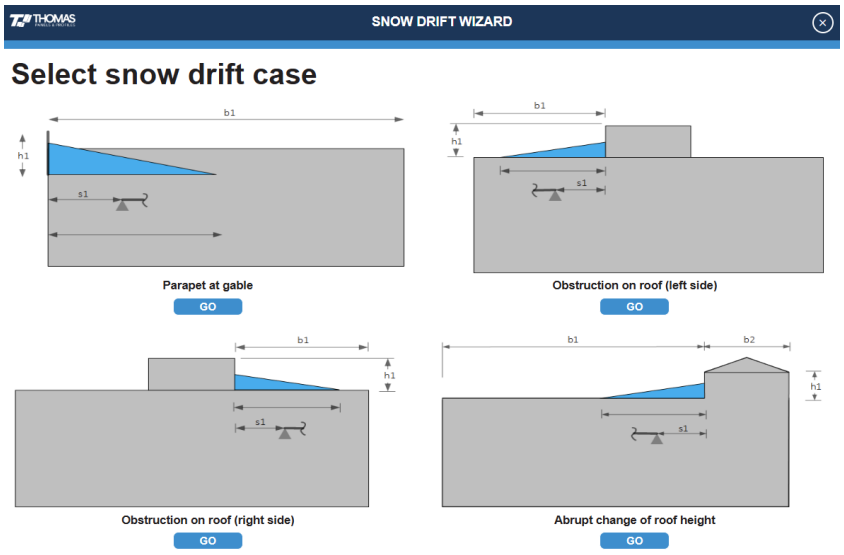


Figure 8.2: SNOW DRIFT WIZARD menu - Opened from Purlin set

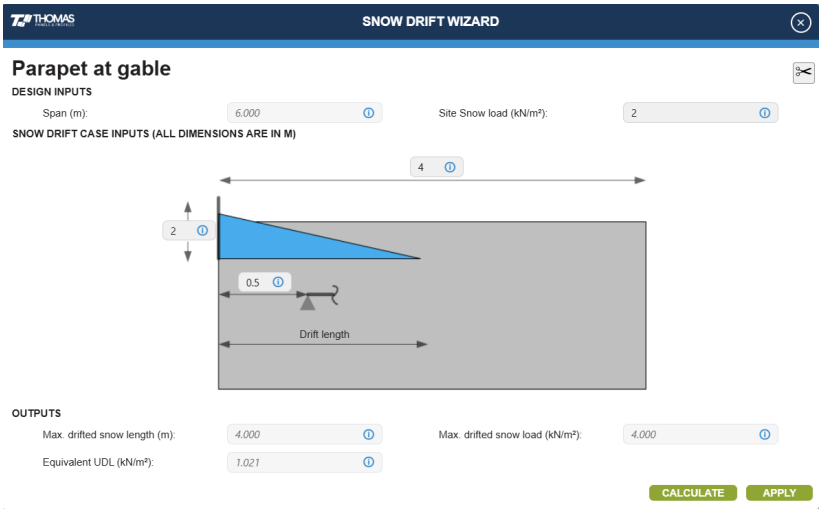


Figure 8.3: SNOW DRIFT WIZARD case - Opened from Purlin set

8.3 WATER WEIGHT WIZARD

The WATER WEIGHT WIZARD, as shown in Figure 8.4, is only available for Eaves beam sets. When viewing the detailed inputs for a given eaves beam within an Eaves beam set, this wizard can be accessed by clicking the WATER WEIGHT WIZARD button.

The WATER WEIGHT WIZARD allows to calculate the self-weight of water according to the gutter dimensions, assuming a gutter fill level of 70%.



Figure 8.4: WATER WEIGHT WIZARD

How to use the wizard:

1. **Provide gutter dimensions**: Enter the height and width of the gutter. The other dimensions displayed on the wizard are for information only.
2. **Click CALCULATE**: Click on the CALCULATE button to calculate the self-weight of water based on the provided gutter dimensions.
3. **Take a snapshot**: If needed, it is possible to take a snapshot of the wizard by clicking the scissors icon ✂.
4. **Click APPLY**: Click on the APPLY button to export the calculated 'Water self-weight' to the eaves beam's detailed inputs.

9 SNAPSHOTS

As described in previous sections of this user guide, users are able to take snapshots of various elements of the software interface for inclusion in the detailed project report. This section provides an overview of the snapshot related functionality available within the software, and particularly focusses on the available functionality to manage snapshots.

9.1 Capturing snapshots

Users are able to take snapshots of the following items:

- **Building view**: As described in Section 5.3.5, snapshots of the Building view can be taken by using the 'alt + s' keyboard shortcut or clicking the scissors icon ✂ and snipping to delimit the snapshot.
- **Drifted snow across centres purlin set diagram**: As described in Section 5.5.2, snapshots of the diagram within Drifted snow across centres purlin sets can be taken by using the 'alt + s' keyboard shortcut or clicking the scissors icon ✂ and snipping to delimit the snapshot.
- **STEPPED WIND LOADS WIZARD**: As described in Section 8.1, snapshots of the STEPPED WIND LOADS WIZARD can be taken by using the 'alt + s' keyboard shortcut or clicking the scissors icon ✂ and snipping to delimit the snapshot.
- **SNOW DRIFT WIZARD**: As described in Section 8.2, snapshots of the SNOW DRIFT WIZARD can be taken by clicking the scissors icon ✂.
- **WATER WEIGHT WIZARD**: As described in Section 8.3, snapshots of the WATER WEIGHT WIZARD can be taken by clicking the scissors icon ✂.

9.2 Snapshot management

Captured snapshots can be managed via the 'View snapshot' option of the Main Menu as shown in Figure 9.1. This directs users to the interface shown in Figure 9.2, which allows users to view, edit and delete snapshots.

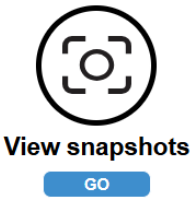


Figure 9.1: View snapshots

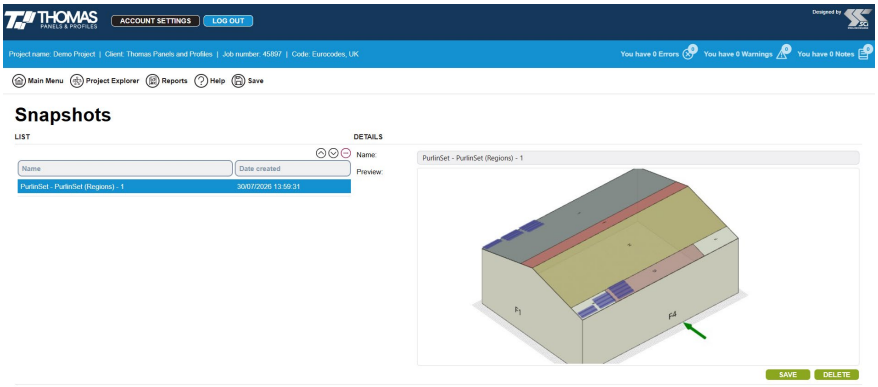


Figure 9.2: Snapshot management

Available functionality

- **List of snapshots**: The list of captured snapshots is provided within the table located on the left-hand side of the interface. Snapshot names are (initially) automatically generated.
- **Select**: Click on a table row to select a snapshot. Users can move the selection via the Up and Down buttons ⌃⌄ located above the list of snapshots. Users can select several snapshots at the same time by holding the 'Ctrl' key and clicking on rows (e.g. to delete them). The current selection is highlighted in red.
- **View**: When selecting one snapshot at a time, the selected snapshot is displayed within the Preview facility located on the right-hand side of the interface.
- **Delete** ⊖: Click the Delete button located above the list of snapshots to delete the currently selected snapshot(s). Alternatively, when selecting one snapshot at a time, click the DELETE button located below the snapshot Preview facility on the right-hand side of the interface.
- **Rename**: To rename a snapshot, select it and update its name within the input field located above the snapshot Preview facility on the right-hand side of the interface. Then, click the SAVE button to save the updated name.

10 ON-SCREEN HELP AND USER FEEDBACK

This section provides an overview of the variety of on-screen help and user feedback available within the Thomas Panels & Profiles software.

10.1 On-screen help

10.1.1 Tooltips

The software provides users with tooltips via information icons ⓘ for most input fields. Clicking on the information icons provides guidance on the expected input, as shown in Figure 10.1.

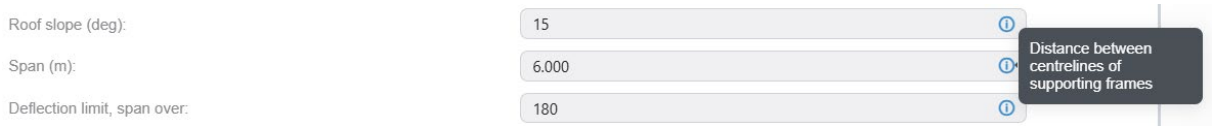


Figure 10.1: Tooltips

10.1.2 On-hover help

Where possible, the software also provides users with on-hover help for various elements of the interface, as shown in Figure 10.2.

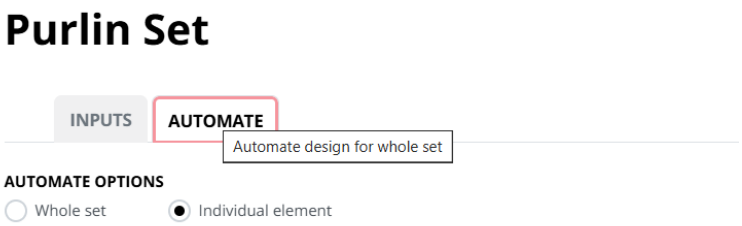


Figure 10.2: On-hover help

10.2 User feedback

10.2.1 Range checks

For input fields requiring a numerical value, range checks are carried out by the software. If an input is outside of the allowable range, the software flags this via an orange exclamation mark as shown in Figure 10.3, and hovering over the exclamation mark displays the allowable range.

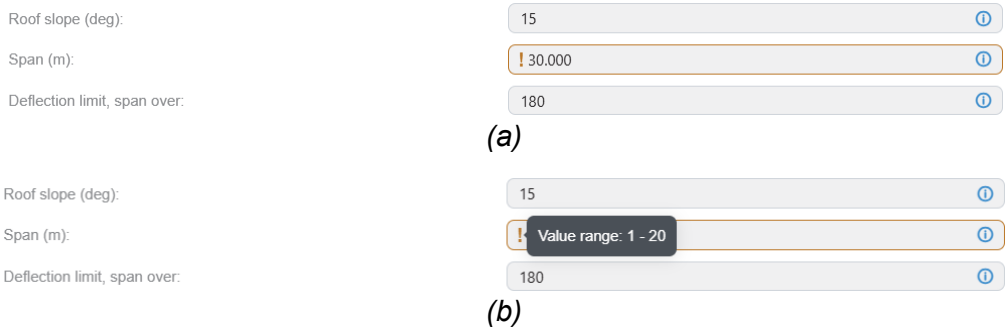


Figure 10.3: Input range check

It should be noted that the software will not proceed with the calculations if some of the inputs are outside of their allowable ranges.

10.2.2 Error messages, Warning messages and Notes

In addition to the input range checks carried out for separate input fields, the software carries out additional checks and notifies users of any issues via Error messages, Warning messages and Notes. Such checks often consider interactions between various inputs.

Users are notified of any Error messages, Warning messages or Notes via the right-hand side of the Project bar as shown in Figure 10.4.



Figure 10.4: Notification of Error messages, Warning messages and Notes

- **Error messages**: Error messages are generated for cases where inputs are incorrect from an engineering viewpoint. The software will not proceed with the calculations if an Error message has been generated for a given structural element.
- **Warning messages**: Warning messages are generated for cases where inputs should be checked. The software will still proceed with the calculations if a Warning message has been generated for a given structural element.
- **Notes**: Notes are generated to provide users with any additional relevant information. The software will still proceed with the calculations if a Note has been generated for a given structural element.

Functionality

The functionality provided for Error messages, Warning messages and Notes is similar. Upon clicking on the links provided within the Project bar to access any Error messages, Warning messages and Notes (see Figure 10.4), a pop-up window such as that shown in Figure 10.5 is displayed.

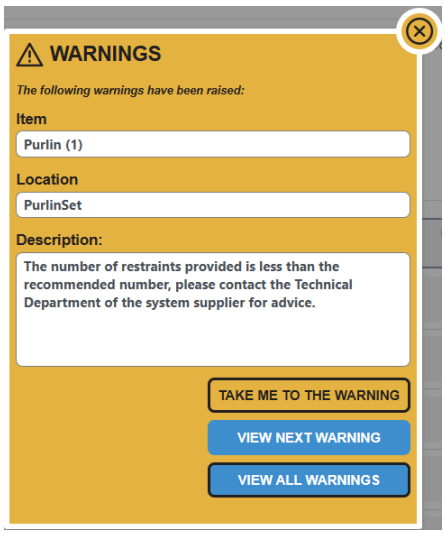


Figure 10.5: Warning message

The details of the affected structural element are provided within the pop-up window as well as a description of the issue. Using Warning messages as an example, users are able to:

- **Click TAKE ME THE WARNING**: Clicking on the TAKE ME TO THE WARNING button will direct users to the relevant Design set to allow them to update relevant inputs accordingly.
- **Click VIEW NEXT WARNING**: Clicking on the VIEW NEXT WARNING button will allow users to view the pop-up window for the next Error message, if any.
- **Click VIEW ALL WARNINGS**: Clicking on the VIEW ALL WARNINGS button will open a new pop-up listing all the Warning messages as shown in Figure 10.6. There, clicking on the TAKE ME TO IT button will direct users to the relevant Design set to allow them to update relevant inputs accordingly.

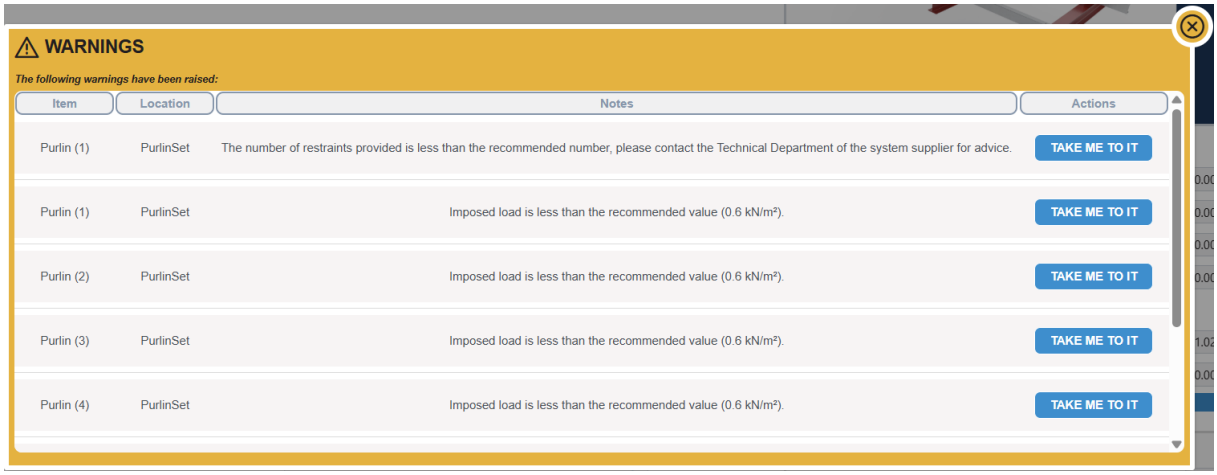


Figure 10.6: List of Warning messages

11 SAVING A PROJECT

Upon creating a new project as described in Section 1.5.2, users are required to select a destination folder and project file name, which defines the default location of the project file going forward.

11.1 Updating project details

The project details, including the destination folder and project file name, can be updated at any time via the 'View project details' option of the Main Menu as shown in Figure 11.1.

Several input fields are available to users in order to identify a project and potential design iterations, such as the Project name and Project file name.

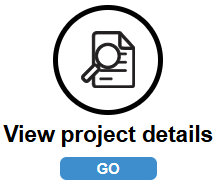


Figure 11.1: View project details

11.2 Quick Save

When working on a project, users can click on the 'Save' button located within the Menu bar as shown in Figure 11.2 at any time to save any updates to the project file (as defined within the project details as per Sections 1.5.2 and 11.1). When doing so, the confirmation message shown in Figure 11.3 is then displayed.



Figure 11.2: Quick Save

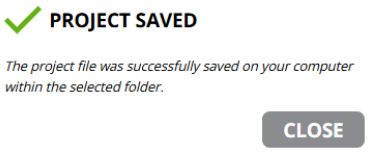


Figure 11.3: Project saved confirmation message

11.3 Save project to file

Users can also save the currently opened project to a different destination folder and project file name via the 'Save project to file' option of the Main Menu as shown in Figure 11.4. When doing so, the project details as described in Sections 1.5.2 and 11.1 are also updated.

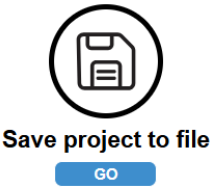


Figure 11.4: Save project to file

12 ACCOUNT PASSWORD RESET / UPDATE

12.1 Password reset

Users can reset their password via the password reset page show in Figure 12.1, which can be accessed via the red ‘Forgot password?’ link located on the login page below the ‘Password’ input field.

Users need to enter their Email address, complete the reCAPTCHA check and click on the GET EMAIL button.

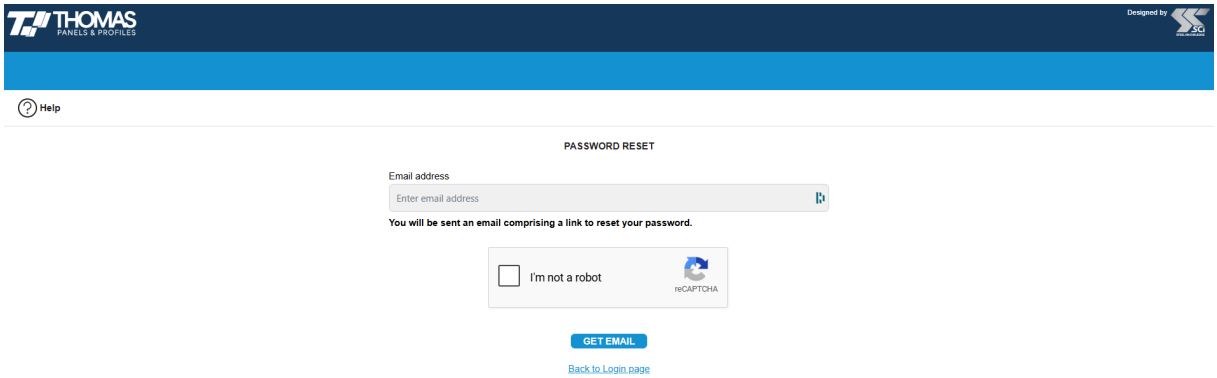


Figure 12.1: Password reset page (initial step)

After submitting the password reset form, the on-screen message shown in Figure 12.2 is displayed. Users will then receive an automated email from SCI’s licensing system as shown in Figure 12.3, and users need to click on the ‘Click here’ link within that email to be directed to the Password reset page shown in Figure 12.4. This automated email will be sent from the noreply@steel-sci.com email address. Users should check their junk folders if the email does not land in their inbox within 5 min of completing the registration page. It is recommended that users whitelist the above email address.

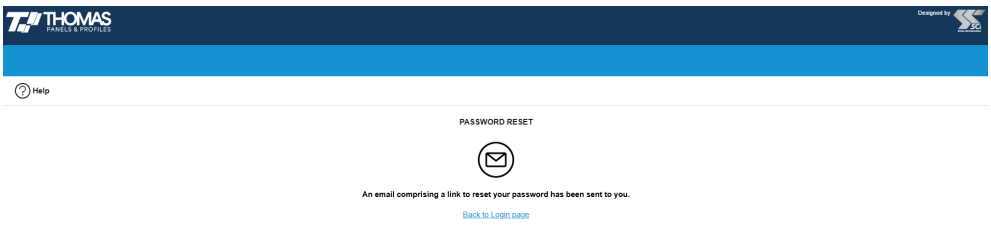


Figure 12.2: Password reset confirmation message

YOUR TPP SOFTWARE ACCOUNT

Dear Colleague,

Thank you for using the TPP software.

Please click here to reset your TPP software account password.

The TPP software was developed by The Steel Construction Institute (SCI). For any queries regarding the software, please contact us via the details provided below.

Best regards,

The SCI Team

The Steel Construction Institute (SCI)
The E Centre
Bracknell, RG12 1NF
Tel: 01344 636 525
Email: software@steel-sci.com
Website: www.steel-sci.com

Figure 12.3: Password reset email

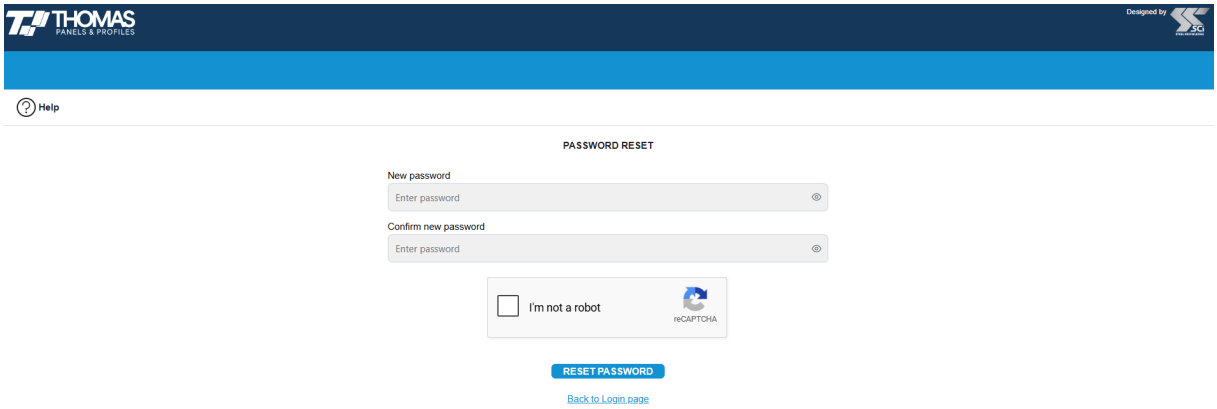


Figure 12.4: Password reset page (final step)

Finally, on the Password reset page shown above, users need to enter their New password, confirm their New password, complete the reCAPTCHA check and click on the RESET PASSWORD button to fully reset their password.

12.2 Password update

Once logged in to the software, users can update their account password at any time by clicking on the ACCOUNT SETTINGS button located within the software's Main header, and then clicking on the CHANGE PASSWORD button on the Account Settings page. The password update page is shown in Figure 12.5.

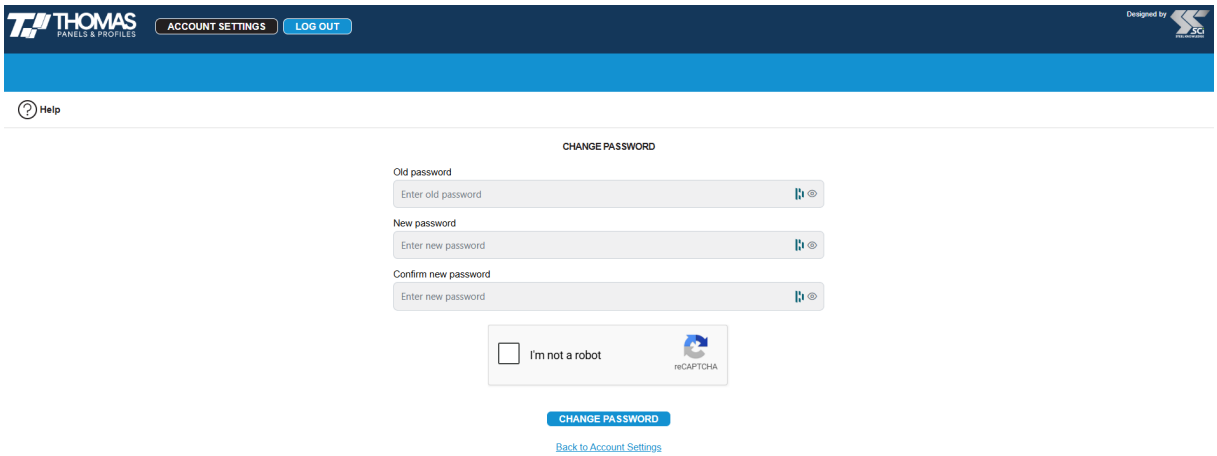


Figure 12.5: Password update page

Users need to provide their Old password, their New password, and also need to confirm their New password and complete the reCAPTCHA check. If the password update is successful, the confirmation message shown in Figure 12.6 is displayed after clicking on the CHANGE PASSWORD button.

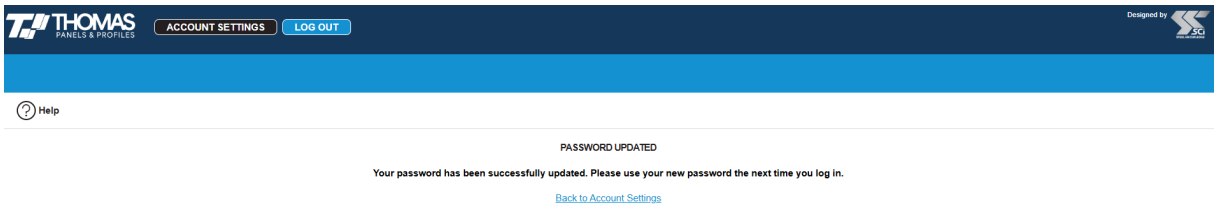


Figure 12.6: Password updated confirmation message

13 TROUBLESHOOTING

Before contacting Thomas Panels & Profiles, users experiencing issues with the TPP design software should carry out the following:

- **Access the software via recommended internet browser**: To ensure full functionality, it is recommended to use the latest versions of the Microsoft Edge or Google Chrome internet browsers to access the software. Most of the functionality may work with other internet browsers, but full functionality cannot be guaranteed when using another browser.
- **Clear 'Cookies and site data' as well as 'Cached images and files' from browser**: This can normally be carried out from the Settings menu of your internet browser, usually within the Privacy and security sub-menu.