Manual

Composite Beam Design

© SCIA Group nv 2008
Composite Beam Design
# Table Of Contents

Composite beam in SCIA ESA PT ................................................................. 1

About ........................................................................................................ 1
  Introduction ............................................................................................. 1

Composite beam design ............................................................................. 1
  Overview features .................................................................................. 1
  Typical operation .................................................................................... 1
  Project data ............................................................................................ 2

Composite beam library ............................................................................. 3
  Composite beam library ....................................................................... 3
  Slab data and properties ....................................................................... 3
  Transverse and longitudinal reinforcement .......................................... 10
  Beam data ............................................................................................ 13
  Fire insulation data ................................................................................ 16

Composite floor panel ............................................................................... 20
  Composite floor panel .......................................................................... 20

Structural analysis .................................................................................. 24
  Structural analysis ................................................................................ 24

Composite - 1D member Setup ............................................................... 25
  Material PSF .......................................................................................... 25
  Span/deflection limits .......................................................................... 25
  Calculation parameters ......................................................................... 25
  Pre-camber ............................................................................................ 26
  Fire resistance ..................................................................................... 27

Composite - Member data ....................................................................... 28
  Composite - Member data .................................................................... 28

Composite - Check ................................................................................ 28
  Composite - Check .............................................................................. 28
  ULS design check ............................................................................... 30
  SLS design check ................................................................................ 34
  Fire resistance check .......................................................................... 37

Document ................................................................................................ 41
  Document .............................................................................................. 41
  Diaphragms (Profile deck data) ............................................................ 42
  2D Reinforcement mesh (data for transverse & longitudinal reinforcement) .................................................. 43
  Hollow core slab (Pre-cast slab data) .................................................... 43
  Shear Connectors (Shear connector data) ........................................... 43
  Composite slab data ............................................................................ 44
  Setup (Design parameters) ................................................................. 45
Composite beam in SCIA ESA PT

About

Introduction

This document explains the composite beam functionality within SCIA • ESA PT.

Modelling of both ‘standalone’ composite beams (defined individually) and ‘plate with beams’ (composite beams defined as part of a floor panel) are described here. The design checks that can be performed on the composite beam and their results are also presented in detail.

This document when read in conjunction with the theoretical background will be a useful reference for anyone who wishes to perform ‘design check’ for composite beams within SCIA • ESA PT.

Composite beam design

Overall features

SCIA • ESA PT supports single or multiple spanning beams and covers simply supported, continuous or cantilever construction.

The program covers:

- Final stage (composite) and construction stage (non-composite) design calculations for ULS, SLS and fire limit states
- Use of standard and user-defined profiled steel decking
- Use of solid slabs, haunched solid slabs and hollow core slabs
- Propped and unpropped construction of the beam
- Floor load generation for secondary beams
- Use of normal weight (NWC) and light weight concrete (LWC)
- Use of headed shear connectors, Hilti, channel and bar-hoop connectors

Typical operation

A typical operating sequence for a standalone composite beam might be as follows:

1. Define a project with ‘concrete’ & ‘steel’ as materials and choose ‘composite’ in the functionality.
2. Define the beam through ‘Horizontal beam’ or ‘Drawing a member’ by specifying the following properties.
   a) Library set to ‘Composite beam’
   b) Composite beam data which includes beam type, slab type, transverse and longitudinal reinforcement, shear connector type, lateral restraints for construction stage and fire insulation data
   c) Effective flange width to left and right of the steel cross section
   d) Steel cross section defined through ‘Profile library’
   e) Beam length or start and end points

Most of the data for composite beam are entered through specific libraries which can be used across projects.

3. Define the beam supports.
4. Define the permanent and variable load cases and corresponding loads.
5. Designate the load cases for construction stage using the solver options.
6. Run the analysis calculation and verify the results
7. Verify the parameters for code check in ‘Composite –> 1D member Setup’
8. Perform code check for ULS, SLS or Fire limit states

The above sequence also holds good for a beam part of floor panel except for the geometry definition.

The design check calculations are based on the chosen national code in project details. This version of SCIA • ESA PT supports composite beam checks in accordance with either BS5950: part 3: 1990 or BS EN 1994-1-1:2004.
Project data

Material - Concrete

The strength of concrete must meet the design code requirements for composite beams and shall not be less than 25N/mm² for LWC or 30 N/mm² for NWC. Similarly, the maximum value of concrete strength shall not be taken as greater than 40 N/mm² for LWC or 50N/mm² for NWC for design.

<table>
<thead>
<tr>
<th>Complying concrete grade</th>
<th>BS code</th>
<th>EC code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal concrete</td>
<td>C30 to C50</td>
<td>C20/25 to C60/75</td>
</tr>
<tr>
<td>Light weight concrete</td>
<td>C25 to C40</td>
<td>LC20/22 to LC60/65</td>
</tr>
</tbody>
</table>

The material grades can be chosen by opening the "Materials" library (click … next to the grade). There is a standard list of material grades available in the system database for various materials like steel, concrete, reinforcing steel etc. The required ones can be included in the project from the database. Note that Type of concrete” is input through ‘Composite > Setup > Calculation parameters’ and is not part of material library.

Functionality

'Composite' is enabled in the functionality only when both concrete and steel are chosen as project materials.

The user can choose ‘fire resistance’ (on selecting ‘composite’ in the functionality) in order to carry out fire resistance checks for the composite beam.
Composite beam library

**Beam type**

The beam type options are primary / secondary. This input is used for reference only.

The detailing of beams must be such that no significant torsion is transmitted to the beam. If this is not the case, additional calculations will need to be carried out.

**Slab data and properties**

The types of slab supported are

1. Solid
2. Haunched solid
3. Profiled deck
4. Pre-cast slab

When the "Profiled deck" or "Precast slab" option is selected, an additional dialog is activated to help select a profile deck / pre cast slab type from the program product libraries.
Figure 3: Composite slab data

**Composite slab data properties**

- **Type of beam**: Secondary
- **Type of deck slab**: Profile deck

**Profile deck**

- Superb 1.2
- Total slab depth, $D_s$ [mm]: 130
- Profile Depth, $D_p$ [mm]: 51
- Manufacturer: Richard lees
- Profile shape: Dovetail
- Weight [kN/m^2]: 0.17
- Effective thickness [mm]: 1

**Profiled sheet data**

Clicking on the ‘...’ next to the default deck opens the Diaphragm library. The user can view the data of standard and user defined decks by opening the system database from the Diaphragm library. The required data can be added to the project through the same.
Figure 4: Profile deck library

<table>
<thead>
<tr>
<th>Name</th>
<th>Superb1-2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manufacturer</td>
<td>Richard lees</td>
</tr>
<tr>
<td>Profile shape</td>
<td>Dovetail</td>
</tr>
<tr>
<td>I - moment [m^4/m]</td>
<td>0.00</td>
</tr>
<tr>
<td>K1+ [m/kN]</td>
<td>0.000</td>
</tr>
<tr>
<td>K2+ [m^2/kN]</td>
<td>0.000</td>
</tr>
<tr>
<td>K1- [m/kN]</td>
<td>0.000</td>
</tr>
<tr>
<td>K2- [m^2/kN]</td>
<td>0.000</td>
</tr>
</tbody>
</table>

**Geometry**

| A [mm] | 38 |
| B [mm] | 115|
| C [mm] | 141|
| D [mm] | 51 |
| Thickness [mm] | 1 |
| Nominal thickness [mm] | 1 |
| Weight [kN/m^2] | 0.17 |
| Material | Grade 43 |

Composite beam in SCIA ESA PT
Figure 5: Profile deck system database

Deck orientation: In the case of profiled deck slabs, it is necessary to define the orientation of the decking on either side of the beam. The rib direction is either Parallel or Perpendicular to the beam. The rib direction on either side of the beam may be different.

Figure 6: Profile deck properties

As shown in the above picture there are options to include the resistance offered by the profiled sheet in the calculation of the slab longitudinal shear resistance. The sheet continuous option is disabled by default whenever the orientation of profiled sheeting on any one side is parallel.
Please note that this software does NOT design the deck or the slab.

For primary beams, the deck direction is set parallel to the beam by default. User can modify the default left and right side rib directions. By default the beams generated by the composite load/floor panel are secondary beams with deck direction perpendicular to the beam. User can change this setting.

*Pre-cast concrete slabs*

In order to select a slab section or create a user-defined section click on the ‘...’ next to default slab. This will open the Hollow core slab library. The user can view the data of proprietary pre-cast concrete slab sections by opening the system database from the Hollow core library.

In this type of construction, the concrete flange of the composite beam is formed by infilling the voids in the hollow core slabs with in-situ concrete over a limited strip width. Typically this would be 500 to 600 mm on either side of the steel beam centre. The flange thickness is taken to be equal to the slab thickness.

The joints between the pre-cast slabs must also be solidly in-filled with concrete. The remaining hollow portions of the slab are not included in the effective flange of the composite beam. The reinforcement ‘dowel’ bars are placed in all / some of the in-situ in-filled cores to augment the longitudinal shear resistance. For solid pre-cast slabs, special pockets or cores are introduced at the ends of the slabs to accommodate the ‘dowel’ bars.

![Composite slab data](image)

**Figure 7: Slab data**
Figure 8: Pre-cast slab library
Total slab depth: The thickness of structural concrete must meet the requirements for strength and stiffness for its function as a suspended slab independently from the function as a composite beam flange. It should be noted, however, that this program does not design the slab. Concrete depth must meet the following conditions:

1. The overall slab depth must not be less than 100 mm, for a solid slab.
2. With steel decking, the concrete depth above the deck must be a minimum of 50 mm. The overall depth of the slab should also not be less than 105 mm.
3. A minimum cover of 15mm is required for all shear connectors.
Transverse and longitudinal reinforcement

In composite construction, the longitudinal shear resistance of the concrete slab must be consistent with the longitudinal shear transfer. Critical shear planes fall on the face of the beam on either side of the shear connectors (shear plane a-a) and around each shear connector or pair of connectors (shear surface b-b). See BS 5950 : Part 3 : Clause 5.6.5.
Figure 11: 2D reinforcement library  Figure 12: Mesh type library
The design longitudinal force is appropriately modified in the program, for the level of stress in the beam and for the number of shear connectors provided.

The components of concrete longitudinal shear resistance are:
1. Concrete shear strength, $V_{cr}$
2. Effectively anchored reinforcement crossing the shear surface.
3. Shear strength of the profiled steel decking, if present and deemed to be effective, $V_p$.

If the decking spans parallel to the beam on one or both sides, the program assumes that the decking is discontinuous at the critical shear plane. Any contribution of the decking is dependent on thorough — welding of alternate shear studs and is controlled by the tick box in the profiled sheet input page.

The maximum longitudinal shear force that can be transferred is limited by horizontal splitting of the concrete and is dependent on the slab depth and concrete strength only. This may limit the effectiveness of additional transverse reinforcement in thin slabs.

**Longitudinal reinforcement**

The longitudinal reinforcements are necessary to improve the moment capacity of the composite section in the hogging regions, where the concrete is assumed to be cracked and will not contribute on section capacity or in other words the composite model is with steel beam and the effective steel reinforcement. As per section 4.4 of BS 5950:Part3:section3.1, all welded mesh reinforcement and bar reinforcements which is less than 10mm are treated as nominal reinforcement and should not be included in the effective section.

**Mesh reinforcement:**

A layer of mesh reinforcement should be positioned close to the top surface of the slab with the minimum of concrete cover (refer to Section 3 of BS 8110: Part 1 : 1995. See also Section 6.9 of BS 5950: Part 4: 1994). Mesh reinforcement acts mainly as anti-crack steel but also as transverse reinforcement. Top face mesh reinforcement contributes to the longitudinal shear resistance of the slab on vertical planes type a-a, but will not contribute to the ‘tunnel shaped’ shear surface around the shear connectors. Bottom face mesh or bars are not usually provided in profiled deck slabs since the deck serves this purpose. However transverse bottom bars are frequently added locally to augment longitudinal shear.
transfer on section b-b especially in cases where the decking cannot be considered either continuous or adequately connected to the steel beam. Solid and haunched slab designs will always have bottom reinforcement and this can be taken into account for longitudinal shear resistance if it is adequately lapped or continuous across the beam.

Mesh sizes available with the program are the following: A98, A142, A198, A252 and A393. Bar diameter and centres can be specified in place of mesh information in the case of non-standard cases. If the user wants to place loose bars in troughs of profile decks or in haunched solid slab then the REBAR option should be used, since mesh is not practical in this case.

**Beam data**

**Figure 14: Beam data**

**Propping:**

Beam props may be introduced at the construction stage to reduce construction stage deflections of the beam or prevent the capacity of the steel beam exceeding before the concrete gains sufficient strength.

When the beam is specified as propped, this software assumes the beam is (theoretically) continuously propped along the whole length of the beam. All construction stage checks are omitted. In particular, steel stresses (before prop removal) are assumed to be zero.
Non-composite beam:

The user can design a section as non-composite (ordinary steel beam without composite action) by disabling the shear connector using the non-composite check box.

Shear connectors:

Composite construction requires the use of appropriately designed and detailed shear connectors to develop composite action between the steel beam and the concrete slab.

Shear connector data is provided in the form of a library. This can be accessed through Library > ShearConnectors on the Main tree.

The shear connectors dialog allows the user to specify the number of shear connectors in a row, transverse spacing, diameter, in situ height and longitudinal spacing. By default, the program assumes a single row of shear connectors. For high loads and/or thick slabs, the connectors may be arranged in pairs.

The longitudinal spacing of shear connectors can be input along the length of the beam. Same spacing is assumed for both hogging & sagging zones.

Figure 15: Shear connector library
Pressing the ‘...’ opens the library of shear connectors. This can be used to choose a standard shear connector or to create a user-defined connector. The list of shear connectors includes stud connectors with available diameters 13, 16, 19, 22 and 25 mm. These are available in standard lengths for which the characteristic resistance values given are taken from BS 5950-3.1. There are two standard lengths for 19 mm diameter studs namely 75mm and 100 mm. The 19 mm x 100 mm long stud is the program default as the most commonly used type. In accordance with the code, 5 mm is deducted from the nominal length of stud connectors to obtain the in-situ length to allow for loss of material in the fusion welding process. The capacity of studs less than standard height is calculated pro-rata to the insitu lengths but longer lengths do not increase capacity.

Other types of shear connectors include Hilti connectors, which are fixed by a mechanical process rather than by welding. Shear capacities are taken from Hilti product literature. Shear connectors may be formed from channels and bar-hoop fabrications as listed. These are profile welded to the top flange of the steel beam and have relatively high capacities proportional to their width. Details and capacities are taken from the former code of practice CP117 and the bridge design code BS 5400 part 4.

It should be noted that a minimum concrete cover of 15mm is required to the top of shear connectors. The number of shear connectors must meet the requirements for the composite strength of the beam and the code requirements for minimum degree of shear connection for the beam span.

Both the moment capacity and the shear connection degree are checked at the position of maximum moment. The moment capacity and the shear connection degree are also calculated at the position of any additional loads. However, only the bending moment capacity is checked at these points. In theory, for satisfactory performance under service loads, the distribution of the shear connectors should be reasonably close to the elastic distribution of longitudinal shear force. However, uniformly spaced connectors routinely resist uniform loads generating parabolic force distribution.

The quantity and distribution of shear connection must be such that they transfer the longitudinal force between the concrete and the steel to develop the required composite resistance at all points along the span.

The number of shear connectors provided can develop a longitudinal force $R_q$. The existence of full shear connection becomes evident if this is greater than the smaller of the axial capacity of the concrete slab $R_c$, and the axial (tensile) capacity of the steel beam $R_s$. If $R_q$ is less than the minimum of $R_c$ and $R_s$, then partial shear connection is developed.
The degree of shear connection is defined as the ratio of the number of shear connectors provided to the number required for full shear connection. Limits are placed on the minimum degree of shear connection in BS 5950: Part 3: Clause 5.5.2., as follows:

- For spans up to 10 metres, the degree of shear connection should not be less than 0.40.
- For spans between 10 and 16 metres, the degree of shear connection should exceed (Span-6)/10.
- For spans greater than 16 metres, partial shear connection is not allowed.

**Lateral restraints (construction stage):**

The lateral beam restraints are input by the user in this section. This is applicable only for construction stage checking. At the final stage the beam is considered as continuously restrained by the slab.

Restraints may be specified as Point, Distributed, Open point1 or Open point2. Open point allows a series of equally spaced point restraints to be defined on one line of input starting at either end 1 or end 2. The restraint position may be top, bottom, both or centre. A secondary beam of similar depth may be considered to provide restraint to both flanges of a primary beam if connected to the web by a normal endplate or fin plate connection. However, it is for the user to decide whether the actual beam depths and connections provide restraint to either flanges or merely centre or top restraint.

![Lateral restraint dialog](image)

Figure 17: Lateral restraint dialog

Restraints by default are taken at both ends of the beam and the user can define further intermediate restraints. If no intermediate restraints are specified, the beam is assumed to be completely unrestrained for lateral-torsional buckling (LTB) calculations at construction stage. For partial restraint conditions, the user must specify the number of restraints and their positions along the beam span. The program uses this information to divide the beam into a number of unrestrained segments. LTB checks are then carried out on all segments.

An internal composite beam supporting a solid slab may also be fully restrained at the construction stage by the formwork to the floor slab. The user however, must, decide whether this assumption is valid for the construction method in use. Similarly continuous lateral restraint may be provided by transversely spanning steel decking or pre cast concrete slabs, subject to appropriate details.

**Fire insulation data**

For fire resistance checks the user specifies a temperature-time curve, fire resistance period and requirement for protection. The temperature-time curve could be a standard temperature-time curve, external fire curve or hydro carbon curve. If the user wishes to have protection then appropriate fire insulation data has to be specified. The user can choose the following from a standard library – manufacturer name, encasement and insulation type. Other properties such as unit mass, thermal conductivity, specific heat and default value of thickness are displayed based on the chosen manufacturer data. These could be altered through the insulation library if required.
Figure 18: Fire resistance data
Cross section

The steel cross section within composite beam can be any one of the following

- Rolled I sections
- Welded I sections (Symmetric & Asymmetric)
- Rectangular Hollow Section
- Square Hollow Section

The above could be found from the following groups in SCIA-ESA PT

- Profile library
- Thin walled geometric sections
- Sheet welded

Steel grade

Complying steel grades for BS 5950-3 clause 3.1 are S275 & S355. Higher strengths are ignored.

The maximum allowable steel grade as per EN 1994-1-1, 3.3 (2) is S460

Design strength is picked from the relevant national code and suitably modified for the flange thickness. As the thickness of flange increases the strength decreases.
**Effective width of concrete flange**

Effective flange width to the left and right of the beam is specified as member property. This input is used in both analysis and design checks. For analysis the effective width is considered the same throughout the beam (no separate input for mid and end zones).

For the design check additional input for left and right zones can be specified through the ‘composite beam additional data’ (Member data). These inputs are necessary in order to cater for any type of span.

In case of simply supported beams the effective width entered in member property alone is sufficient for both analysis and design. For other types of beams where the nature of bending moment changes (like fixed beam) the user has to necessarily use the ‘composite beam additional data’ (Member data) and specify different effective widths for start and end zones to obtain better results.

Limitation: In this version of SCIA ESA PT, for beams with multiple spans the user cannot specify separate set of additional data for each individual span. The beam takes just a single set of data which is used for checking all the spans.

**Loading**

The loading on the composite beam can be categorised into

1. Final stage variable load
2. Final stage permanent load
3. Construction stage variable load
4. Construction stage permanent load

The user must create appropriate load cases representing the above through the standard load cases dialog. No distinction is made between final and construction stage load cases at this stage. This is done just before analysis when the user is required to specify the load cases for construction stage through solver options.

![Figure 20: Solver setup](image-url)
Loading under each of the above categories are calculated and input by the user for standalone composite beams. Please note that to avoid ambiguity between slab supported width (load sharing width) and slab flange width (effective width of concrete flange for design check), there is no automatic self weight calculation. All possible 1D member loading types in SCIA EES PT are applicable.

For composite beams created through the ‘load/floor panel with beams’ the floor load information can be input directly through ‘Surface load on 2D member’ load type.

Based on the load cases appropriate load combinations must be created by the user. A load combination must contain only final stage or only construction stage load cases. If a combination is found to have both it cannot be used for design checks.

During analysis the construction stage combinations are calculated with steel beam section properties and final stage combinations are calculated with composite section properties.

The program requires at least one permanent load case to be specified for final & construction stage load combination in order to proceed with the analysis.

Note: Although construction stage checks are not supported for EC code construction stage load case has to be created and specified in solver setup. The users can make use of these load cases to review the internal forces after analysis and later use ‘Steel’ service to perform design check.

**Composite floor panel**

To model ‘composite floor panel’ select structure type as ‘Frame XYZ or General XYZ’ in Project data. This will enable “Load/floor panel with beams” in the ‘Structure’ menu.
The basic frame (has shown below) is first constructed using the regular tools (Catalogue blocks) available in SCIA ESA PT.

To add composite floor click on “Load/floor panel with beams”
Clicking "Load/floor panel with beams" will open the 2D member properties dialog (see picture below). In the properties dialog set Library to 'Composite floor'. Once this is done composite slab data property will be enabled.

Secondary beam layout (within the panel) is then specified through the options at the bottom of the property dialog. Two options (number/distance) are available. Offsets can also be provided from the start.

Note: Beams are always placed along the local X axis of the plate. In order to modify this the property 'LCS Angle' has to be altered to the desired value.

The dialog is closed by clicking on OK. This will prompt the user to define a 2D polygon in main view. Once user completes defining the same the beam property dialog will open.

Note that in beam properties dialog (picture below) the type is set to 'composite plate rib' as we have specified the 2D member type as composite.

In the beam properties set the required cross section for the secondary beams and also the effective width to be used for analysis & design check (2 individual inputs available).
Figure 26: Secondary beam property dialog

Closing the dialog will create number of secondary composite beams attached to the plate. All the duplicate frame members coinciding with the secondary beams can be removed using 'Check structure data' option before analysis.

The secondary beams created through 'load/floor panel with beam' are similar to other beams i.e. rigidly connected by default. The user has to specify hinges for the beams to behave simply supported.

Figure 27: Composite floor panel with secondary & primary beams

Surface loads applied to the plate are distributed automatically to the secondary beams during analysis. One way load distribution is assumed.
The distributed member load can be viewed through Actions >> Load generation from the plate properties.

**Structural analysis**

**Structural analysis**

Only linear analysis is supported. Loads can be applied for both the stages - construction and final stages.

The final stage model will be analyzed with composite beam section properties (uncracked section properties).

The construction stage model will be analyzed with steel beam properties.

In the solver setup it is recommended to select "Neglect shear force deformation (Ay, Az >> A)" as the program does not consider composite cross section for shear area ‘Ay’ & ‘Az’ calculation.
Composite beam in SCIA ESA PT

Composite - 1D member Setup

Material PSF

The following partial safety factors are used in the program in accordance with BS 5950: part3: section 3.1 and BS EN 1994-1-1:2004.

<table>
<thead>
<tr>
<th>Material partial safety factors</th>
<th>BS 5950</th>
<th>EC4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Structural steelwork</td>
<td>1.00</td>
<td>-</td>
</tr>
<tr>
<td>Structural steelwork</td>
<td>-</td>
<td>1.00</td>
</tr>
<tr>
<td>(cross section resistance)</td>
<td>-</td>
<td>1.00</td>
</tr>
<tr>
<td>Structural steelwork</td>
<td>1.00</td>
<td>-</td>
</tr>
<tr>
<td>(buckling resistance)</td>
<td>-</td>
<td>1.00</td>
</tr>
<tr>
<td>Profilied sheet</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>Reinforcing steel</td>
<td>1.05</td>
<td>1.15</td>
</tr>
<tr>
<td>Shear connectors</td>
<td>1.25</td>
<td>1.25</td>
</tr>
<tr>
<td>Concrete in shear</td>
<td>1.25</td>
<td>1.50</td>
</tr>
<tr>
<td>Concrete in compression</td>
<td>1.50</td>
<td>1.50</td>
</tr>
</tbody>
</table>

Users can override the above values and recalculate design check when required.

Span/deflection limits

Acceptable deflection limits of the beam are defined by the user. The defaults in the program are as follows:

<table>
<thead>
<tr>
<th>Deflection component</th>
<th>Span/deflection limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Permanent load</td>
<td>240</td>
</tr>
<tr>
<td>Variable load</td>
<td>360</td>
</tr>
<tr>
<td>Combined load</td>
<td>500</td>
</tr>
</tbody>
</table>

Construction stage deflections of the beam are based on loads acting on the steel section without composite action.

At the final design stage, the program checks both the deflection under the variable load and the total deflection. The deflection under variable loads is used to assess the effect of the deflection of the beam for finishes, partitions etc.

The total deflection includes the deflection due to permanent and variable loads, and can be used to check the effect of the beam deflection on the overall appearance of the structure.

For un-propped construction, the component of permanent load deflection applied at construction stage is calculated using the properties of the steel section acting alone. The remainder of the total deflection is calculated using the properties of the composite section.

Calculated values of deflection are compared with the deflection limits entered by the user. In composite design, it is often found that serviceability criteria control the design of the beams. This is due to the composite beams being designed to be as shallow as possible for a given span. Serviceability performance is assessed using elastic section properties. Stress limits are placed on the concrete and steel in order to ensure that elastic conditions hold when calculating deflections.

Calculation parameters

The calculation parameters that can be configured are:
Concrete

Type of concrete: Normal Weight Concrete (NWC) or Light Weight Concrete (LWC) may be used for construction. Construction in LWC usually provides economic steel design but does not necessarily give the least cost overall. LWC concrete is worth considering for large buildings with long spans. This program issues a caution message when lightweight concrete dry density is less than 1750 kg/m³. Information on the properties of LWC may be found in the manufacturers’ literature.

Proportional of total load that is long term: This is used in effective modular ratio calculation and is based on short term and long term modular ratios. This calculated effective modular ratio is used in serviceability calculations (elastic section properties).

Vibration

Percentage of variable load accounted for natural frequency calculation: Normally it is 10% and this is set as default.

Percentage of increase in moment of inertia of the beam accounting for the dynamic stiffness: Normally it is 10% and this is set as default.

Natural frequency: Acceptable natural frequency limit of the beam is defined by the user to control floor vibration. The default in the program is 4.

Miscellaneous

Additional parameters required for EC code are grouped under this head.

1. Value of C4 for LTB check (C4): The default in the program is 6.2
2. Creep coefficient: The default in the program is 5.
3. Creep multiplier: The default in the program is 0.55

C4 is the factor for distribution of bending moment used in elastic critical buckling moment calculation. The value of C4 is represented as a curve for specific shape of bending moment in fig A.3 of Designer’s guide to EN 1994-1-1 (page no 207).

‘Creep coefficient’ and ‘Creep multiplier’ are used in effective modular ratio calculation.

Effective modular ratio is calculated as

\[ \alpha_E = \alpha_S + \rho_l (\alpha_l - \alpha_S) \]

Where:
- \( \alpha_E \): effective modular ratio
- \( \alpha_l \): long term modular ratio
- \( \alpha_S \): short term modular ratio
- \( \rho_l \): proportion of total load that is long term

Pre-camber

The pre-camber can be defined in different ways. It can be made equal to the proportion of span or an absolute value. The pre-camber value is deducted from the calculated deflection value to give the net deflection.

Pre-camber provided to the beam is assumed to be in a circular form. The radius of such a curve functions as the middle camber value \( C \) at the span \( L \) of the beam.

Radius of curvature \( R = 1/2C [(L/2)^2 + C^2] \)

The intermediate camber value at any given point at a distance "X" from the support is "Yx". It is calculated as follows.

\[ Y = (R^2 - (L/2 - X)^2)^{1/2} \]
\[ Yx = Y - (R - C) \]
Figure 30: Pre-camber calculation
Note: Pre-camber will be considered only if the downward deflection is maximum.

Fire resistance

*Material PSF*

The following partial safety factors are used for fire resistance checks

<table>
<thead>
<tr>
<th>Material partial safety factors</th>
<th>BS 5950</th>
<th>EC4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete in compression</td>
<td>1.5</td>
<td>1.00</td>
</tr>
<tr>
<td>Concrete in shear</td>
<td>1.25</td>
<td>1.00</td>
</tr>
<tr>
<td>Structural steelwork</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>Reinforcing steel</td>
<td>1.05</td>
<td>1.00</td>
</tr>
<tr>
<td>Shear connectors</td>
<td>1.25</td>
<td>1.00</td>
</tr>
<tr>
<td>Profile sheet steel</td>
<td>1.00</td>
<td>1.00</td>
</tr>
</tbody>
</table>

*Calculation parameters*

Additional calculation parameters for fire resistance check as per EC4 are given below.

1. Temperature curve: The user can choose from Standard temp-time curve or External fire curve or Hydro carbon curve
2. Co-efficient of heat transfer by convection: The default value is 25 W/m²K
3. Emissivity related to fire compartment: The default value is 0.8
4. Emissivity related to surface material: The default value is 0.625
5. Analysis type: Resistance domain

The fire check is performed as per the guidelines given in EC4. The resistance of the section are calculated with reduced material properties under fire. All the calculations are done assuming that the fire exposure is from three sides. The user can check the performance of a composite beam with or with out protection for the standard time by using various temperature curves. The unity checks, section capacities, limiting temperature etc are reported in the brief and detailed out puts.
Composite - Member data

This facility has been provided to override some of the data specified through ‘composite slab data’ library before analysis. The main advantage of specifying member data is that the user need not create a new library item for individual beam variations.

If the same input is modified through the member property dialog the user will have to create a new ‘composite slab data’ library item for each individual change. Otherwise modifying the existing library item will affect all the beams to which the data has been assigned.

Listed below is the data that may be overwritten through this dialog:

- Propping
- Design check as non-composite beam
- Effective flange width to left and right for start and end zone: The value entered in ‘composite slab data’ library is taken for the span (middle) zone. The left and right zones by default take the value from the span zone but may be overwritten here.
- Shear connector data such as type, diameter, nominal & insitu height, material grade, characteristic strength, number of rows and longitudinal spacing.
- Lateral restraints data for construction stage check of un-propped construction.
- Fire insulation data such as manufacturer, product name, protection type, fixing type and fire resistance period.

Composite - Check

Composite beam design in SCIA•ESA PT is primarily intended to carry out thorough checks on members for which the cross section is input by the user.
In broad sense the steps involved in checking a single composite beam are

- Construction stage check
- Final stage check

After entering the input data and completing the analysis the user selects 'Composite -> 1D member -> Check' to perform the design check.

The properties that need to be set prior to that are listed below.

- List of members for which the check has to be performed
- Type of load case or combination or result class
- The stage - construction/final
- Extreme value to be displayed (member/global/local/section)

The design checks are grouped into

1. ULS design check
2. SLS design check
3. Fire resistance check

Design check results are produced at supports, maximum positive bending moment location and point load locations on the beam.

The output from composite beam can be grouped under the following heads.

1. Graphical display of utilisation
2. Text output
   a. Brief
   b. Detailed

In addition the results for a single beam can be viewed through SCIA•ESA PT single check functionality.

Note: The calculations are generalized to handle sagging and hogging moments. The left zone is considered only if a hogging moment exists at the left support. The span zone is considered only if a sagging moment exists within the span. The right zone is considered only if a hogging moment exists at the right support.

Limitation: Construction stage ULS & SLS checks are not supported for EC code in this version of SCIA•ESA PT under Composite check. This has to be checked through the 'Steel' check service in SCIA•ESA PT.
Composite Beam Design

ULS design check

**Final stage**

For the final stage ULS check, utilisation is calculated for vertical shear capacity, bending moment capacity, longitudinal shear capacity and torsional buckling resistance. The maximum utilisation of the above is reported as final stage ULS u-ratio. The same value is displayed graphically in the main view.

**Brief results**: The brief output is displayed in a tabular form with results for a member in a single line. The users can use SCIA•ESA PT’s table composer to alter the table style and also rearrange the columns.

**ULS design check**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>E1</td>
<td>9000</td>
<td>1200</td>
<td>9000</td>
<td>5202</td>
<td>0.09</td>
<td>50.00</td>
<td>546.92</td>
<td>9.24</td>
<td>5.16</td>
<td>8.95</td>
<td>0.00</td>
<td>90.00</td>
<td>90.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 33: Final stage ULS brief output

**Detailed results**: The results listed are for the critical slice position. The various results listed under each heading are as below.

- **Vertical shear check**
  - Shear force
  - Shear capacity
  - Utilisation
  - Status

- **Moment capacity check**
  - Moment capacity
  - Shear status (High/Low)
  - No. of connectors effective at section
  - Reduction factor for connector capacity
  - Reduction factor as per clause 5.4.3
  - Reduction factor due to concrete type
  - Reduction factor due to slab profile
  - Capacity of single connector
  - Minimum degree of shear connection
  - Shear connector resistance
  - Actual degree of shear connection
  - Longitudinal stress ratio
  - Top flange class
  - Web class
  - Depth of concrete compression zone
  - Depth of plastic neutral axis
  - Neutral axis location
  - Plastic moment capacity
  - Elastic moment capacity
  - Ultimate moment capacity

- **Utilisation**
  - Status

- **Longitudinal shear check**
  - Maximum longitudinal shear force
  - Longitudinal shear resistance in flange
  - Longitudinal shear resistance due to concrete
  - Longitudinal shear resistance due to transverse reinforcement
  - Longitudinal shear resistance due to profile sheeting
Composite beam in SCIA ESA PT

Total shear resistance
Utilisation
Status
Torsional buckling check
Effective length
Slenderness correction
Slenderness factor
Equivalent slenderness
Buckling resistance moment
  Maximum bending moment
  Equivalent uniform moment factor
  Equivalent uniform moment
Utilisation
Status

Single check: Single check allows the user to view summary of results for a single span of a composite beam. The results presented for final stage ULS are as shown below.

![Single check interface](image)

Figure 34: Final stage ULS single check output – Plastic resistances
Results are presented at both hogging and sagging regions (only critical position). ‘Previous span’ & ‘Next span’ buttons can be used to view results for different spans of a beam.

**Construction stage**

For construction stage ULS check, utilisation is calculated for vertical shear, bending moment, lateral buckling and torsional buckling. The maximum utilisation of the above is reported as construction stage ULS u-ratio. The same value is displayed graphically in the main view.

**Brief results:** The brief output is displayed in a tabular form with results for a member in a single line. The users can use SCIA•ESA PT’s table composer to alter the table style and also rearrange the columns.

**Detailed results:** The results listed are for the critical slice position. The various results listed under each heading are as below.

- **Vertical shear capacity check**
  - Position of maximum shear force
  - Maximum shear force
  - Shear area
  - Shear capacity
  - Unity check
  - Status
Moment capacity check
- Position of maximum bending moment
- Maximum bending moment
- Moment capacity
- Unity check
- Status

Lateral buckling check
- Effective length
- Slenderness correction
- Slenderness factor
- Equivalent slenderness
- Buckling resistance moment
- Maximum bending moment
- Equivalent uniform moment factor
- Equivalent uniform moment
- Unity check
- Status

Torsional buckling check
- Effective length
- Slenderness correction
- Slenderness factor
- Equivalent slenderness
- Buckling resistance moment
- Maximum bending moment
- Equivalent uniform moment factor
- Equivalent uniform moment
- Unity check
- Status

Single check: Single check allows the user to view the summary of results for a single span of a composite beam. The results presented for construction stage ULS are as shown below.
SLS design check

Net deflection, natural frequency and elastic stresses are calculated for SLS load combinations.

Elastic stress is calculated only for sagging moments except in the case of cantilever spans for which the hogging moment at the support is considered for elastic stress check.

For the unpropped condition, stress due to permanent load is calculated based on properties of steel beam and stress due to variable load is calculated based on properties of composite beam. For propped beams, stress due to total load is calculated based on properties of composite beam.

Final stage

For final stage SLS check, utilisation is calculated for permanent load deflection, variable load deflection, combined load deflection and stress in steel beam. The maximum utilisation of the above is reported as final stage ULS u-ratio. The same value is displayed graphically in the main view.

Brief results: The brief output is displayed in a tabular form with results for a member in a single line. The users can use SCIA•ESA PT’s table composer to alter the table style and also rearrange the columns.

<table>
<thead>
<tr>
<th>Member</th>
<th>SLS design check</th>
<th>Deflection due to permanent load [mm]</th>
<th>Limiting deflection [mm]</th>
<th>Deflection due to variable load [mm]</th>
<th>Limiting deflection [mm]</th>
<th>Results</th>
<th>Net stresses</th>
<th>Stress due to variable load [MPa]</th>
<th>Stress due to permanent load [MPa]</th>
<th>Stress due to total load [MPa]</th>
<th>Stress due to combined load [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>0.000</td>
<td>1.7</td>
<td>0.7</td>
<td>0.0</td>
<td>0.7</td>
<td>0.6</td>
<td>0.0</td>
<td>-1.7</td>
<td>-0.5</td>
<td>0.3</td>
<td>0.6</td>
</tr>
</tbody>
</table>

Detailed results: The results reported are

Deflection check

- Position of maximum deflection
- Number of connectors effective at section
- Actual degree of shear connection
Precamber

<table>
<thead>
<tr>
<th>Actual deflection due to permanent load * (calculated with final stage dead loads using second moment of area of steel beam only)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable deflection due to permanent load</td>
</tr>
<tr>
<td>Utilisation for permanent load deflection</td>
</tr>
</tbody>
</table>

Actual deflection due to variable load * (calculated using second moment of area of composite beam)

<table>
<thead>
<tr>
<th>Allowable deflection due to variable load</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilisation for variable load deflection</td>
</tr>
</tbody>
</table>

Actual deflection due to combined load * (sum of "actual deflection due to permanent load" and "actual deflection due to variable load")

<table>
<thead>
<tr>
<th>Allowable deflection due to combined load</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilisation for combined load deflection</td>
</tr>
</tbody>
</table>

Natural frequency check

<table>
<thead>
<tr>
<th>Actual natural frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>Allowable natural frequency</td>
</tr>
<tr>
<td>Utilisation for natural frequency</td>
</tr>
</tbody>
</table>

Elastic stress check

<table>
<thead>
<tr>
<th>Position of maximum moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Moment</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Second moment of area of final section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Depth of neutral axis from top of slab</td>
</tr>
<tr>
<td>Section modulus for concrete flange</td>
</tr>
<tr>
<td>Section modulus for steel flange</td>
</tr>
</tbody>
</table>

Actual maximum stress in concrete or reinforcement

<table>
<thead>
<tr>
<th>Allowable stress in concrete or reinforcement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilisation for stress in concrete</td>
</tr>
</tbody>
</table>

Actual maximum steel stress due to dead loads * (calculated with final stage dead loads using section modulus of steel beam only)

<table>
<thead>
<tr>
<th>Allowable maximum steel stress due to dead loads</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilisation for maximum steel stress</td>
</tr>
</tbody>
</table>

Actual maximum steel stress due to imposed loads * (calculated using section modulus of composite beam)

<table>
<thead>
<tr>
<th>Allowable maximum steel stress due to imposed loads</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilisation for maximum steel stress</td>
</tr>
</tbody>
</table>

Actual resultant maximum steel stress * (sum of "actual maximum steel stress due to imposed loads" and "actual maximum steel stress due to dead loads")

<table>
<thead>
<tr>
<th>Allowable resultant maximum steel stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>Utilisation for resultant maximum steel stress</td>
</tr>
</tbody>
</table>

Single check: Single check allows the user to view the summary of results for a single span of a composite beam. The results presented for final stage SLS are as shown below.
**Composite Beam Design**

Figure 39: Final stage SLS single check output

**Construction stage**

For construction stage SLS check, utilisation is calculated for permanent load deflection, variable load deflection, combined load deflection, stress in steel beam. The maximum utilisation of the above is reported as construction stage SLS u-ratio. The same value is displayed graphically in the main view.

**Brief results:** The brief output is displayed in a tabular form with results for a member in single line. The users can use SCIA•ESA PT’s table composer to alter the table style and also rearrange the columns.

**Detailed results:** The results reported are

- **Deflection check**
  - Position of maximum deflection
  - Actual deflection due to permanent load
  - Allowable deflection due to permanent load
  - Utilisation for permanent load deflection
  - Actual deflection due to variable load
  - Allowable deflection due to variable load
  - Utilisation for variable load deflection
  - Actual deflection due to combined load
  - Allowable deflection due to combined load
  - Utilisation for combined load deflection

- **Elastic stress check**
  - Position of maximum moment
Single check: Single check allows the user to view the summary of results for a single span of a composite beam. The results presented for construction stage SLS are as shown below.

![Single check](image)

Figure 41: Construction stage SLS single check output

**Fire resistance check**

The fire resistance checks are optional as per the settings in the project functionality. Fire insulation material data for BS and EC are presented below. The user can choose these data from SCIA-ESA PT as part of the system database.

**Fire insulation data for EC4**

Product name
- Fire board
- Vermiculite and cement board
- Silicate / calcium silicate board
- Cement board
- Gypsum board
- Mineral fibre (low density)
- Vermiculite cement (low density)
- Perlite
- Vermiculite cement (high density)
- Vermiculite and gypsum (high density)

Encasement type
- Hollow encasement
- Contour encasement

Insulation type
\begin{itemize}
  \item Board protection
  \item Spray protection
  \item Intumescent protection
\end{itemize}

\textbf{Fire insulation data for BS 5950-8:2003}

\textbf{Manufacturer}
\begin{itemize}
  \item Cafco international
  \item Lafarge plaster board limited
  \item Thermal Ceramics UK Ltd
  \item British Gypsum Limited
  \item Promat UK ITd
  \item Rockwool Ltd
  \item Cryotherm insulation limited
  \item Boltom fire protection
  \item Firetherm intumescent & insulation supplies ltd
  \item Leigh’s paints
  \item Tikkurila coatings Lts
  \item Nullifire limited
  \item Ameron international
\end{itemize}

\textbf{Product name}
\begin{itemize}
  \item Cafco Board
  \item Columnclad
  \item Onlit 150 Systems
  \item Conlit Tube
  \item Firecheck Board
  \item Firemaster 607 Blanket
  \item Promalit
  \item Promatect 250
\end{itemize}

\textbf{Fire protection type}
\begin{itemize}
  \item Profile
  \item Box
  \item Circular
  \item Solid
\end{itemize}

\textbf{Fire fixing type}
\begin{itemize}
  \item Cafclip partial adhesive fixing system or noggings and screw fixing systems
  \item Columnclad
  \item Glued Noggins, Dry Joints BEAMS & COLUMNS
\end{itemize}
Fire resistance checks as per BS 5950-8:2003

The user must set up a load combination for the fire limit state as per table 2 of BS 5950-3. The evaluation of fire resistance is done by means of the section factor. The steps involved in the calculation are:

1. Calculation of load ratio 4.4.2.2
2. Calculation of limiting temperature as per table 5
3. Calculation of design temperature
4. Reduction in temperature for beams with low aspect ratio
5. Calculation of protection thickness as per Appendix D of BS 5950-8:2003

**Brief results:**

<table>
<thead>
<tr>
<th>Member</th>
<th>dx [m]</th>
<th>Case</th>
<th>My [kNm]</th>
<th>Vz [kN]</th>
<th>Ultimate moment cap. [kNm]</th>
<th>Thickness of fire protection [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>81</td>
<td>0.000</td>
<td>1</td>
<td>-30.00</td>
<td>90.00</td>
<td>132.23</td>
<td>0.030</td>
</tr>
</tbody>
</table>

Figure 42: BS code fire check brief output

**Detailed results:**

The results reported include:

- Hogging bending moment
- Hogging moment capacity
- Sagging bending moment
- Sagging moment capacity
- Section factor
- Load ratio
- Limiting temperature
- Protection required
- Thickness required
- Utilisation
Fire resistance checks as per EN 1994-1-2:2005

In case of EC4 by default it is assumed that fire protection is required and a thickness is obtained as input. SCIA ESA PT obtains the capacities under the impact of the fire with the provided thickness and compares it with the applied moments. Note that the capacity is proportional to the thickness of protection material. If the thickness of protection is more then the limiting temperature will come down which in turn increases the capacity.

The user can also assess the section capacity without considering protection. This is done by setting the property ‘Protection’ to ‘No’ under ‘Composite -> Member data -> Fire resistance’

Brief results:

**Fire Resistance design check**

The results reported include
- Input thickness
- Limiting temperature in the bottom steel flange
- Limiting temperature in the web
- Limiting temperature in the top steel flange
- Sagging bending moment
- Sagging moment capacity
- Hogging bending moment
- Hogging moment capacity
- Utilisation
- Status
Maximum vertical shear force
Vertical shear capacity
Utilisation
Status

Longitudinal shear force in sagging zone
Longitudinal shear resistance in sagging zone
Longitudinal shear force in hogging zone
Longitudinal shear resistance in hogging zone
Utilisation
Status

Single check:

Figure 45: Euro code fire check output dialog

Document

Document

All the results presented above are available through the ‘Document’ and can be directly printed or exported to a file in various forms (HTML/Text/RTF/PDF/XLS) with the standard output options available in SCIA-ESA PT.

In addition to the results other input data specific to composite beam are available for output from Libraries -> Diaphragms, 2D Reinforcement mesh, Hollow core slab, ShearConnectors, Composite slab data which are presented below.
Diaphragms (Profile deck data)

<table>
<thead>
<tr>
<th>Type</th>
<th>Name</th>
<th>Diaphragm</th>
<th>A [mm]</th>
<th>B [mm]</th>
<th>C [mm]</th>
<th>D [mm]</th>
<th>Thickness [mm]</th>
<th>Nominal thickness [mm]</th>
<th>Weight [kN/m²]</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 46: List of insertable items in Document

Figure 47: Profile deck data output
2D Reinforcement mesh (data for transverse & longitudinal reinforcement)

<table>
<thead>
<tr>
<th>Type Name</th>
<th>2D Reinforcement mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>A142</td>
</tr>
<tr>
<td>Code</td>
<td>All</td>
</tr>
<tr>
<td>Material included</td>
<td></td>
</tr>
<tr>
<td>Type of mesh</td>
<td>A142</td>
</tr>
<tr>
<td>Direction close to surface</td>
<td>1</td>
</tr>
<tr>
<td>Diameter [mm]</td>
<td>6.0</td>
</tr>
<tr>
<td>Bar distance [mm]</td>
<td>200</td>
</tr>
<tr>
<td>Offset [mm]</td>
<td>0</td>
</tr>
<tr>
<td>Reinforcement area [mm²]</td>
<td>141</td>
</tr>
<tr>
<td>Diameter [mm]</td>
<td>6.0</td>
</tr>
<tr>
<td>Bar distance [mm]</td>
<td>200</td>
</tr>
<tr>
<td>Offset [mm]</td>
<td>0</td>
</tr>
<tr>
<td>Reinforcement area [mm²]</td>
<td>141</td>
</tr>
<tr>
<td>Total weight of reinforcement [kg/m]</td>
<td>2.20</td>
</tr>
<tr>
<td>Bar:ap [mm]</td>
<td>3.00</td>
</tr>
</tbody>
</table>

Figure 48: Reinforcement data output

Hollow core slab (Pre-cast slab data)

<table>
<thead>
<tr>
<th>Type Name</th>
<th>Hollow core slab</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>HCl</td>
</tr>
<tr>
<td>Manufacturer</td>
<td>Dienen</td>
</tr>
<tr>
<td>Slab width [mm]</td>
<td>1200</td>
</tr>
<tr>
<td>Slab depth, Ds [mm]</td>
<td>100</td>
</tr>
<tr>
<td>Self weight [kN/m²]</td>
<td>2.40</td>
</tr>
<tr>
<td>Core pitch [mm]</td>
<td>100</td>
</tr>
<tr>
<td>Thickness of individual web (bw) [mm]</td>
<td>50</td>
</tr>
<tr>
<td>Thickness of outermost web at the level of elastic gravity line (bwo) [mm]</td>
<td>40</td>
</tr>
<tr>
<td>Net height of the joint (hj) [mm]</td>
<td>100</td>
</tr>
<tr>
<td>Smallest value of upper thick. of flange (hf, 1) [mm]</td>
<td>35</td>
</tr>
<tr>
<td>Smallest value of lower thick. of flange (hf, 1) [mm]</td>
<td>35</td>
</tr>
</tbody>
</table>

Figure 49: Pre-cast slab data output

Shear Connectors (Shear connector data)

<table>
<thead>
<tr>
<th>Name</th>
<th>Type</th>
<th>Diameter/width [mm]</th>
<th>Nominal height [mm]</th>
<th>In-situ height [mm]</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIGL</td>
<td>Stud</td>
<td>10</td>
<td>100</td>
<td>55</td>
<td>S275</td>
</tr>
</tbody>
</table>

Figure 50: Shear connector data output
### Composite slab data

#### Composite slab properties

<table>
<thead>
<tr>
<th>Name</th>
<th>CEB 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of slab</td>
<td>Secondary</td>
</tr>
<tr>
<td>Type of deck</td>
<td>Profile deck</td>
</tr>
<tr>
<td>Profile deck</td>
<td>Super 12</td>
</tr>
<tr>
<td>Total slab depth, $d$ [mm]</td>
<td>130</td>
</tr>
<tr>
<td>Profile depth, $b$ [mm]</td>
<td>51</td>
</tr>
<tr>
<td>Profile shape</td>
<td>Dowel tail</td>
</tr>
<tr>
<td>Manufacturer</td>
<td>Richard Less</td>
</tr>
<tr>
<td>Weight, $W$ [kN/m]$^2$</td>
<td>0.17</td>
</tr>
<tr>
<td>Effective thickness, $t$ [mm]</td>
<td>1</td>
</tr>
<tr>
<td>Rib centers</td>
<td>155</td>
</tr>
<tr>
<td>Rib width, $b$ [mm]</td>
<td>115</td>
</tr>
<tr>
<td>Yield strength, $f_y$ [MPa]</td>
<td>275</td>
</tr>
<tr>
<td>Left side rib direction</td>
<td>Perpendicular</td>
</tr>
<tr>
<td>Right side rib direction</td>
<td>Perpendicular</td>
</tr>
<tr>
<td>Contribution to longitudinal shear resistance, $k_y$</td>
<td>*</td>
</tr>
<tr>
<td>Shear connector</td>
<td>Continuous</td>
</tr>
<tr>
<td>Self weight of slab, $w$ [kg/m]$^2$</td>
<td>301.84</td>
</tr>
</tbody>
</table>

#### Transverse and Longitudinal reinforcement

<table>
<thead>
<tr>
<th>Top / Bottom / Beam data</th>
<th>Input type</th>
<th>Library</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top</td>
<td>Type</td>
<td>A142</td>
</tr>
<tr>
<td>Direction close to surface</td>
<td>Transverse</td>
<td></td>
</tr>
<tr>
<td>Cover, $d$ [mm]</td>
<td>15</td>
<td></td>
</tr>
<tr>
<td>Characteristic yield strength, $f_y$ [MPa]</td>
<td>0.9</td>
<td></td>
</tr>
<tr>
<td>Transverse</td>
<td>Dia [mm]</td>
<td>6.0</td>
</tr>
<tr>
<td>Centers [mm]</td>
<td>200</td>
<td></td>
</tr>
<tr>
<td>Longitudinal</td>
<td>Dia [mm]</td>
<td>6.0</td>
</tr>
<tr>
<td>Left support zone</td>
<td>200</td>
<td></td>
</tr>
<tr>
<td>Centers [mm]</td>
<td>200</td>
<td></td>
</tr>
<tr>
<td>Span zone</td>
<td>Dia [mm]</td>
<td>0.9</td>
</tr>
<tr>
<td>Centers [mm]</td>
<td>200</td>
<td></td>
</tr>
<tr>
<td>Right support zone</td>
<td>Dia [mm]</td>
<td>6.0</td>
</tr>
<tr>
<td>Centers [mm]</td>
<td>200</td>
<td></td>
</tr>
</tbody>
</table>

### Beam data

<table>
<thead>
<tr>
<th>Non composite beam</th>
<th>Input type</th>
<th>Library</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear connector data</td>
<td>Type</td>
<td>914E</td>
</tr>
<tr>
<td>Type</td>
<td>934</td>
<td></td>
</tr>
<tr>
<td>Diameter, $d$ [mm]</td>
<td>19</td>
<td></td>
</tr>
<tr>
<td>Nominal height, $h$ [mm]</td>
<td>130</td>
<td></td>
</tr>
<tr>
<td>Pitch height, $b$ [mm]</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>Input type</td>
<td>Calculated</td>
<td></td>
</tr>
<tr>
<td>Characteristic strength, $f_y$ [MPa]</td>
<td>84.99</td>
<td></td>
</tr>
<tr>
<td>Number of rows</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Transverse spacing, $s$ [mm]</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>Longitudinal spacing, $s$</td>
<td>130</td>
<td></td>
</tr>
</tbody>
</table>

### Fire resistance

<table>
<thead>
<tr>
<th>Temperature - Time curve</th>
<th>By setup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard fire rating, period</td>
<td>50</td>
</tr>
<tr>
<td>Protection</td>
<td>Yes</td>
</tr>
<tr>
<td>Fire insulation</td>
<td>Fire board</td>
</tr>
<tr>
<td>Enclosure type</td>
<td>Hollow core</td>
</tr>
<tr>
<td>Insulation type</td>
<td>Board protection</td>
</tr>
<tr>
<td>Unit mass, $m$ [kg/m]$^3$</td>
<td>150.00</td>
</tr>
<tr>
<td>Thermal conductivity, $k$ [W/mK]</td>
<td>20000.00</td>
</tr>
<tr>
<td>Specific heat, $q$ [W/mK]</td>
<td>12000.00</td>
</tr>
<tr>
<td>Default value for thickness, $t$ [mm]</td>
<td>10</td>
</tr>
</tbody>
</table>

Figure 51: Composite slab data output
Setup (Design parameters)

All the design parameters can be output through Composite Beam -> Setup

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material PSF Structural steel work resistance to cross sections [kN/m²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Material PSF Structural steel work resistance to bucking [kN/m²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Material PSF Reinforcing steel [kN/mm²]</td>
<td>1.15</td>
</tr>
<tr>
<td>Material PSF Concrete [kN/mm²]</td>
<td>1.50</td>
</tr>
<tr>
<td>Material PSF Design shear resistance of headed stud [kN/mm²]</td>
<td>1.25</td>
</tr>
<tr>
<td>Material PSF Profile steel sheet [kN/mm²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Calculation parameters: Type of concrete</td>
<td>Normal</td>
</tr>
<tr>
<td>Calculation parameters: Proportion of total load that is long term [%]</td>
<td>0.23</td>
</tr>
<tr>
<td>Calculation parameters: Percent of variable load accounted for calc. of natural frequency [%]</td>
<td>10.00</td>
</tr>
<tr>
<td>Calculation parameters: Percent of increase in inertia to allow for dynamic stiffness [%]</td>
<td>10.00</td>
</tr>
<tr>
<td>Calculation parameters: Natural frequency [Hz]</td>
<td>4.00</td>
</tr>
<tr>
<td>Calculation parameters: Value of factor C4 for LTB check [C4]</td>
<td>8.50</td>
</tr>
<tr>
<td>Calculation parameters: Deep coefficient [m]</td>
<td>6.00</td>
</tr>
<tr>
<td>Calculation parameters: Deep multiplier [m]</td>
<td>0.55</td>
</tr>
<tr>
<td>Fire resistance: Concrete [kN/mm²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Fire resistance: Structural steel [kN/mm²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Fire resistance: Reinforcing steel [kN/mm²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Fire resistance: Design shear resistance of headed stud [kN/mm²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Fire resistance: Profiled sheet steel [kN/mm²]</td>
<td>1.00</td>
</tr>
<tr>
<td>Fire resistance: Temperature curve</td>
<td>Standard temp-time curve</td>
</tr>
<tr>
<td>Fire resistance: Coefficient of heat transfer by convection [%]</td>
<td>25.00</td>
</tr>
<tr>
<td>Fire resistance: Emissivity related to fire compartment [%]</td>
<td>0.80</td>
</tr>
<tr>
<td>Fire resistance: Emissivity related to surface material [%]</td>
<td>0.63</td>
</tr>
<tr>
<td>Fire resistance: Analysis type</td>
<td>resistance or domain</td>
</tr>
<tr>
<td>Span/def limits: Permanent load [kN]</td>
<td>240.00</td>
</tr>
<tr>
<td>Span/def limits: Variable load [kN]</td>
<td>300.00</td>
</tr>
<tr>
<td>Span/def limits: Combined load [kN]</td>
<td>200.00</td>
</tr>
<tr>
<td>Pre-camber: Absolute value (mm)</td>
<td>10.00</td>
</tr>
</tbody>
</table>

Figure 52: Design parameters output