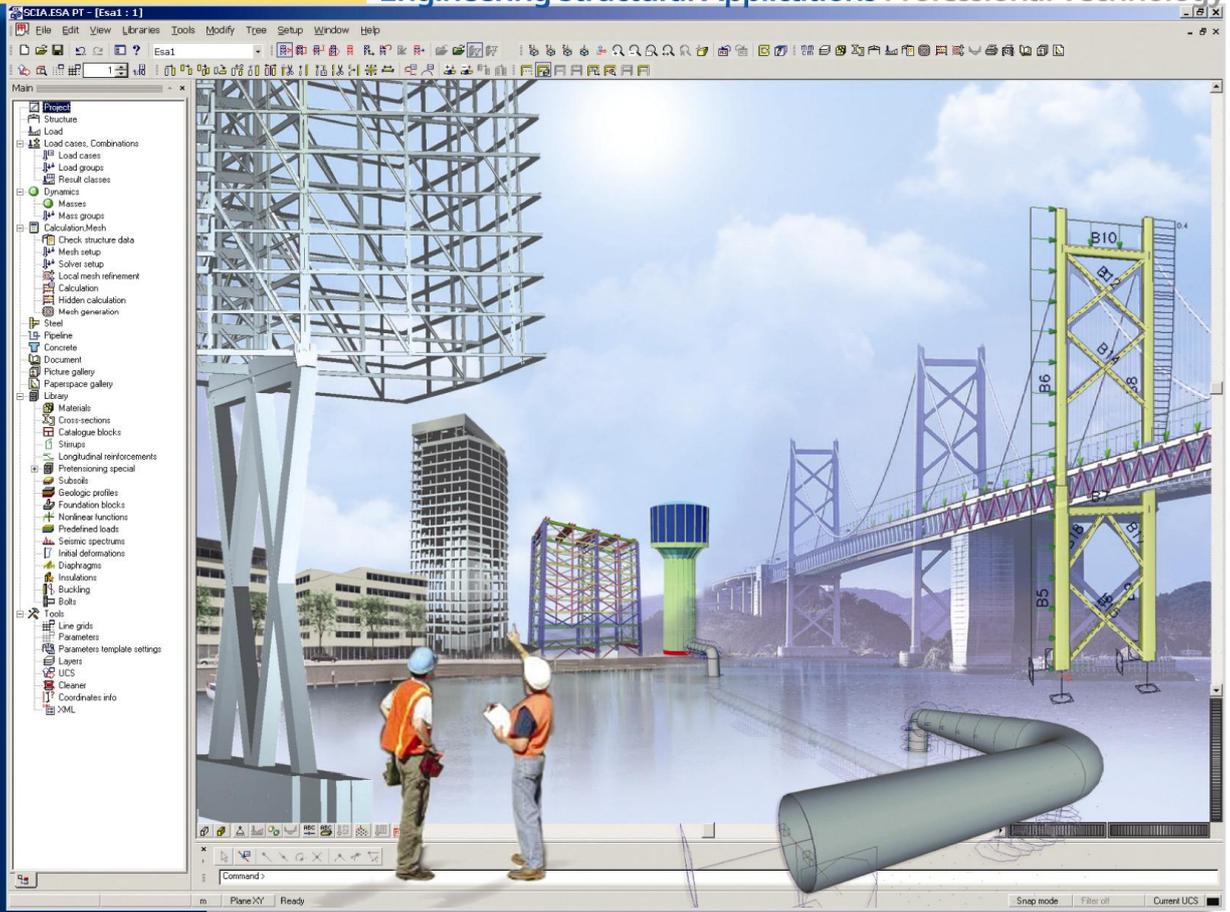


Enabling Innovation in Construction



## Manual Composite Beam Design



# **Composite Beam Design**

---



# Table Of Contents

<b>COMPOSITE BEAM IN SCIA ESA PT.....</b>	<b>1</b>
<b>About.....</b>	<b>1</b>
Introduction .....	1
<b>Composite beam design .....</b>	<b>1</b>
Overall features .....	1
Typical operation.....	1
Project data .....	2
<b>Composite beam library .....</b>	<b>3</b>
Composite beam library.....	3
Slab data and properties.....	3
Transverse and longitudinal reinforcement .....	10
Beam data .....	13
Fire insulation data .....	17
<b>Composite floor panel.....</b>	<b>20</b>
Composite floor panel .....	20
<b>Structural analysis .....</b>	<b>24</b>
Structural analysis .....	24
<b>Composite - 1D member Setup .....</b>	<b>25</b>
Material PSF .....	25
Span/deflection limits .....	25
Calculation parameters .....	26
Pre-camber.....	26
Fire resistance .....	27
<b>Composite - Member data .....</b>	<b>28</b>
Composite - Member data .....	28
<b>Composite - Check .....</b>	<b>28</b>
Composite - Check .....	28
ULS design check.....	30
SLS design check .....	34
Fire resistance check.....	37
<b>Document .....</b>	<b>41</b>
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/mm<sup>2</sup> for LWC or 30 N/mm<sup>2</sup> for NWC. Similarly, the maximum value of concrete strength shall not be taken as greater than 40 N/mm<sup>2</sup> for LWC or 50N/mm<sup>2</sup> for NWC for design.

Complying concrete grade	BS code	EC code
Normal concrete	C30 to C50	C20/25 to C60/75
Light weight concrete	C25 to C40	LC20/22 to LC60/65

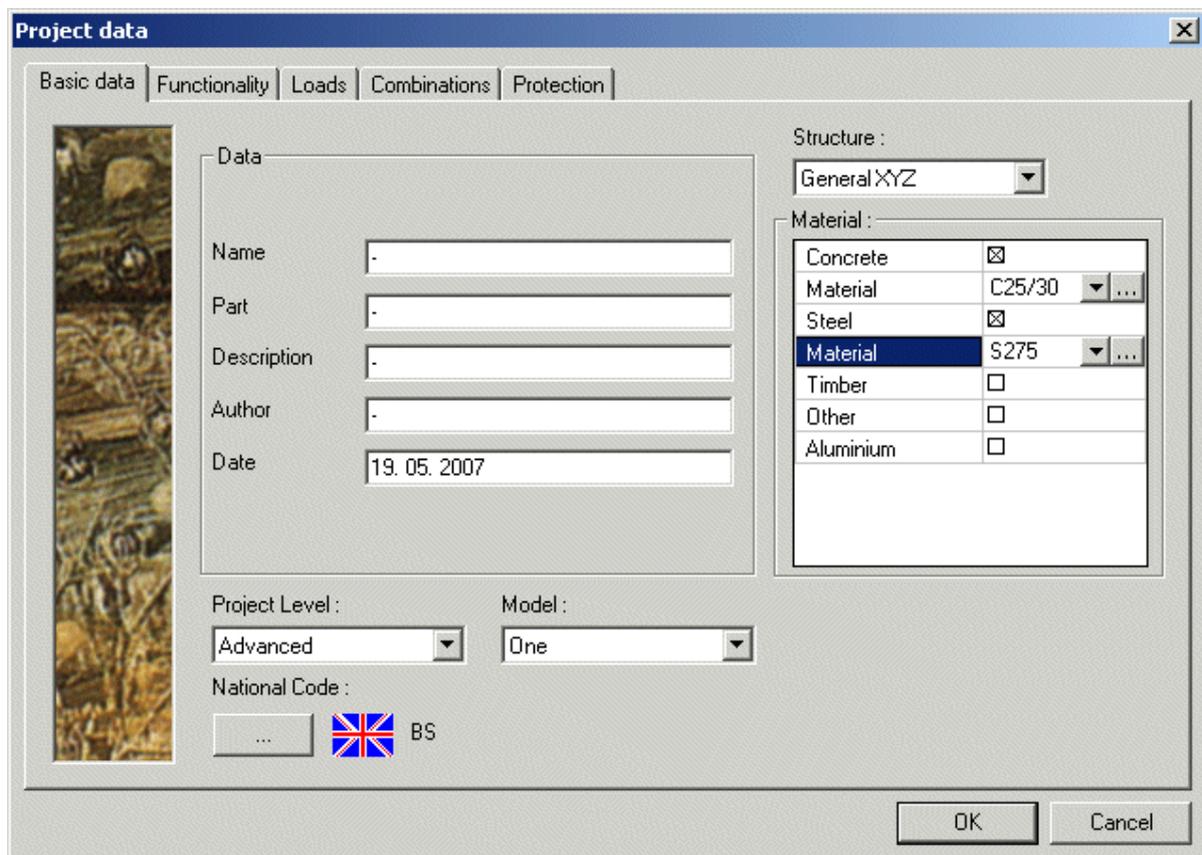


Figure 1: Project data – Basic data

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.

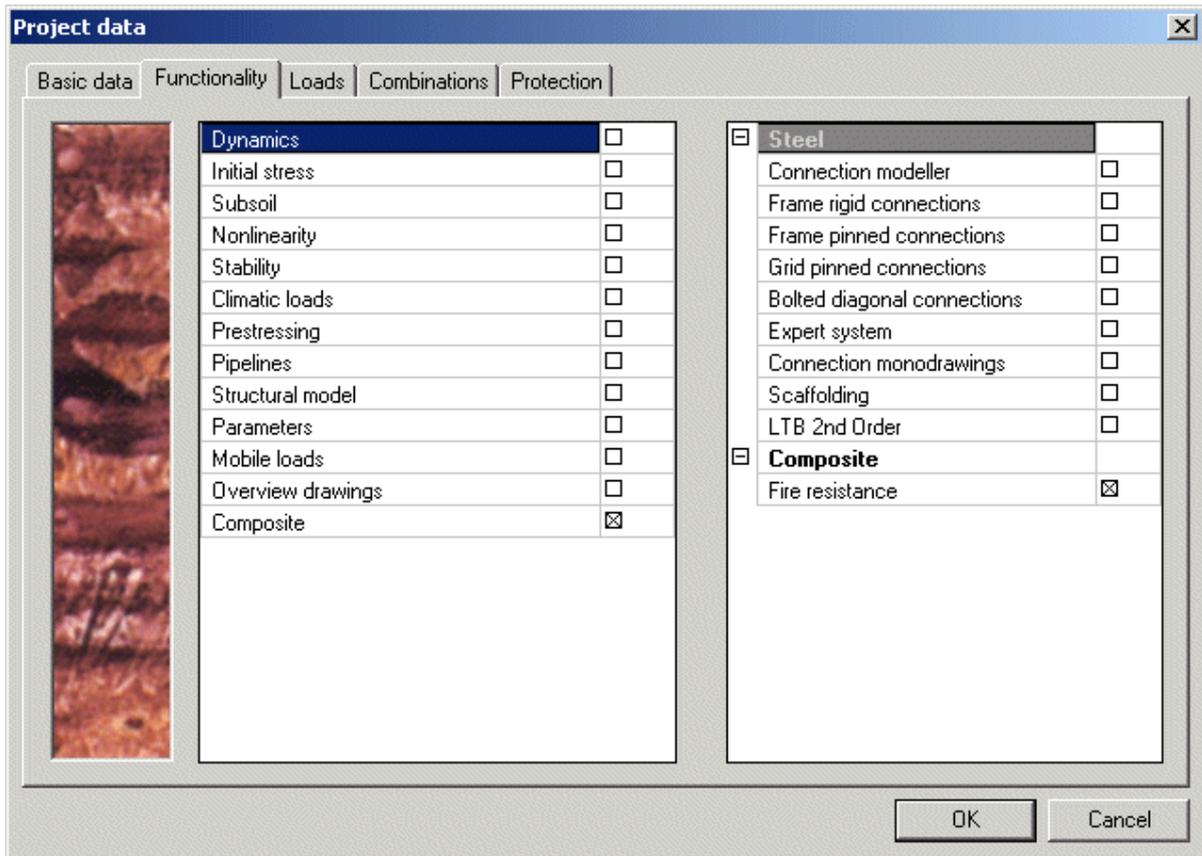


Figure 2: Project data - Functionality

## Composite beam library

### 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 "Pre-cast slab" option is selected, an additional dialog is activated to help select a profile deck / pre cast slab type from the program product libraries.

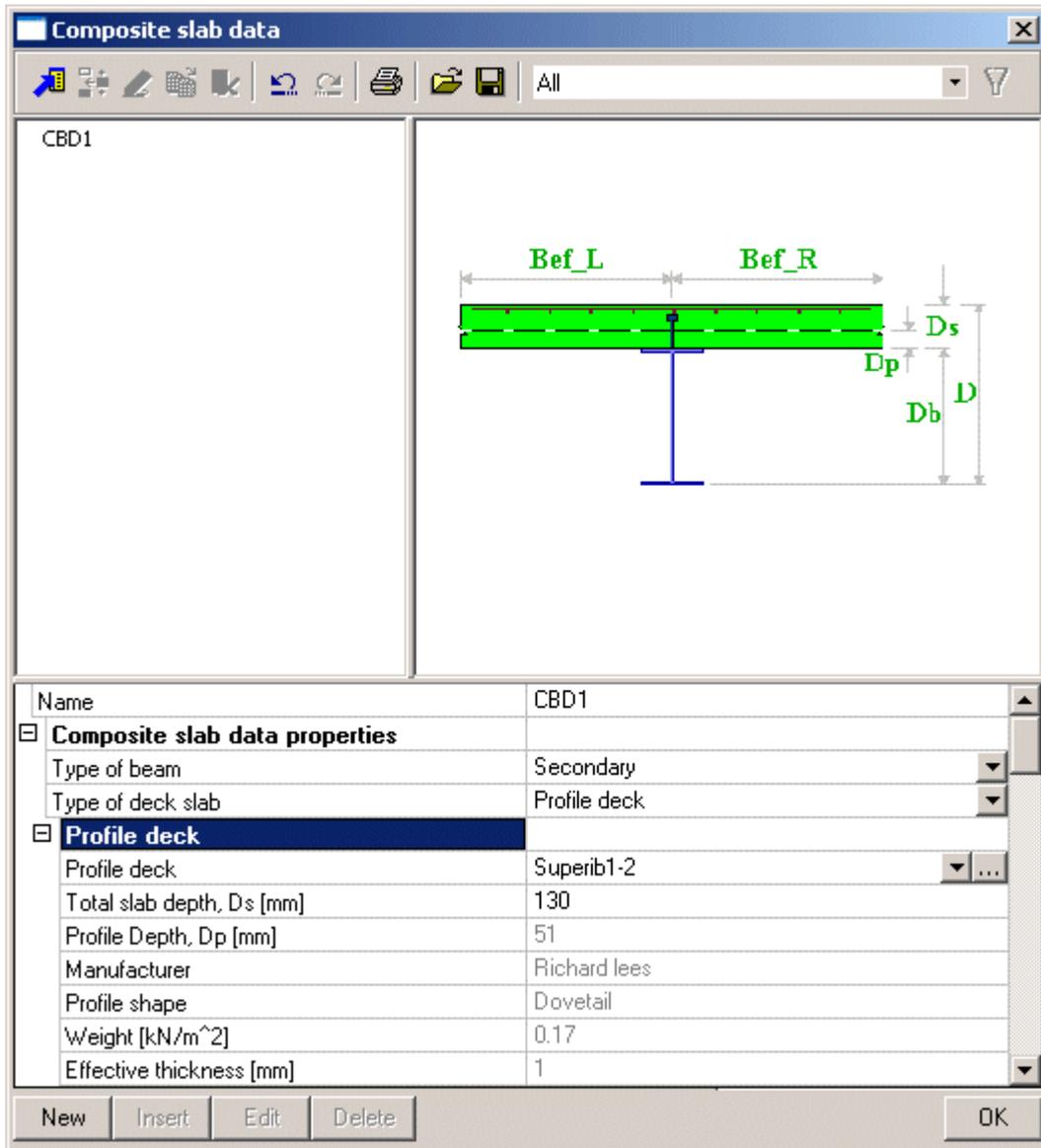


Figure 3: Composite slab data

**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.

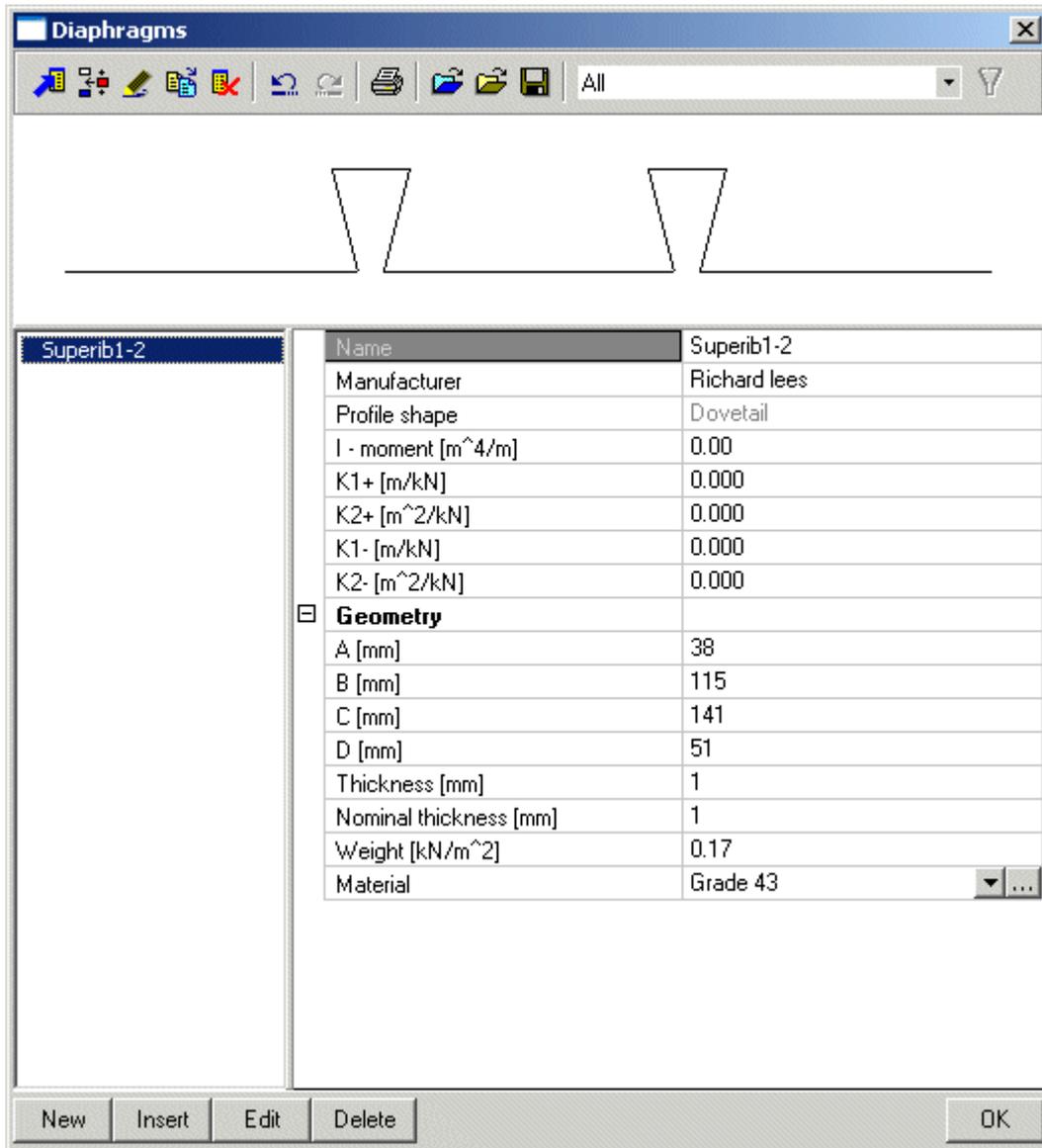


Figure 4: Profile deck library

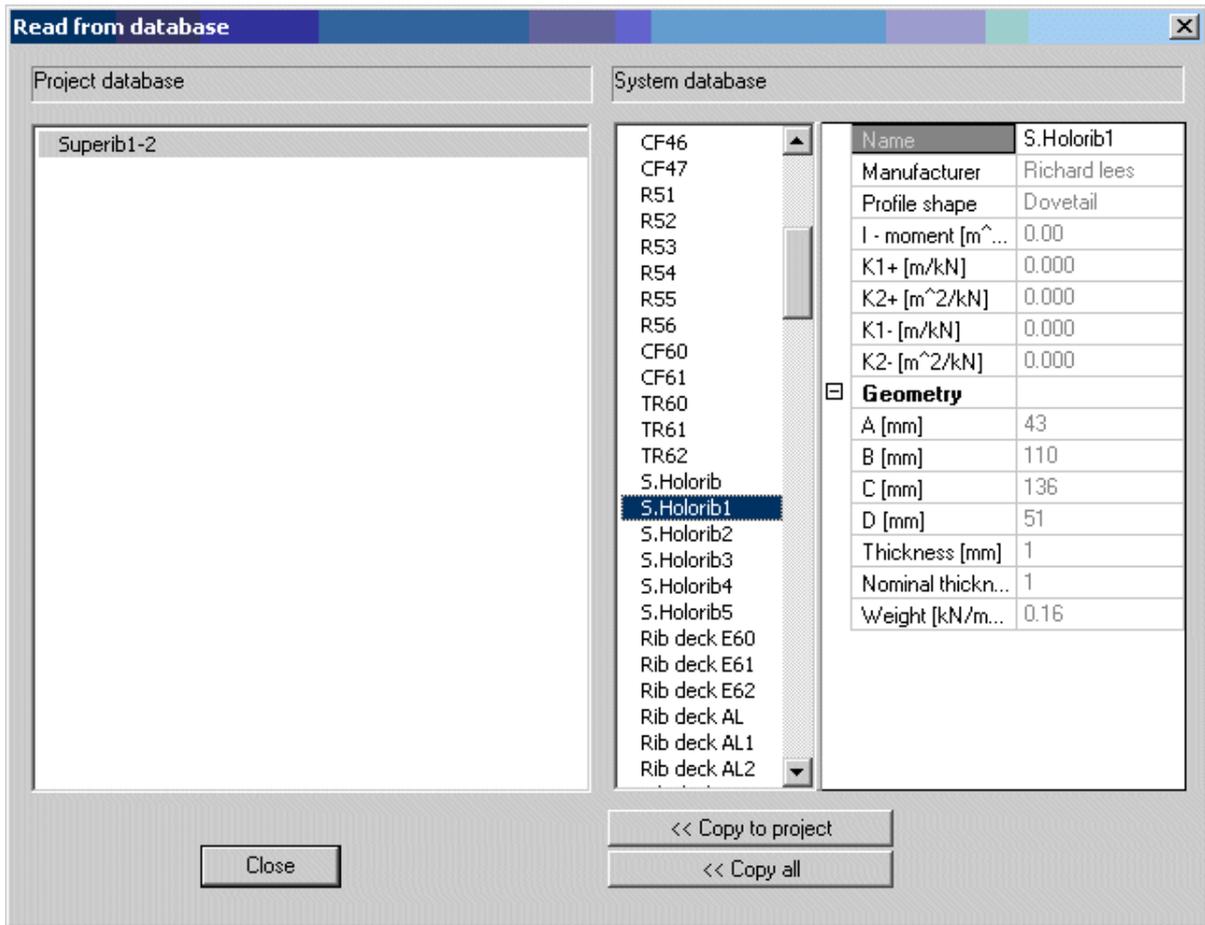


Figure 5: Profile deck system database

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.

<input type="checkbox"/> <b>Profile deck</b>	
Profile deck	Superib1-2
Total slab depth, $D_s$ [mm]	130
Profile Depth, $D_p$ [mm]	51
Manufacturer	Richard lees
Profile shape	Dovetail
Weight [ $\text{kN}/\text{m}^2$ ]	0.17
Effective thickness [mm]	1
Nominal thickness [mm]	1
Rib centers [mm]	153
Rib width [mm]	115
Yield strength [MPa]	275.0
Left side rib direction	Perpendicular
Right side rib direction	Perpendicular
Contribution to longitudinal shear resistance	<input type="checkbox"/> No
Sheet continuous	<input type="checkbox"/> No
Shear connector welded through profile sheeting	<input checked="" type="checkbox"/> Yes
Self weight of slab [ $\text{kg}/\text{m}^2$ ]	290.22

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.

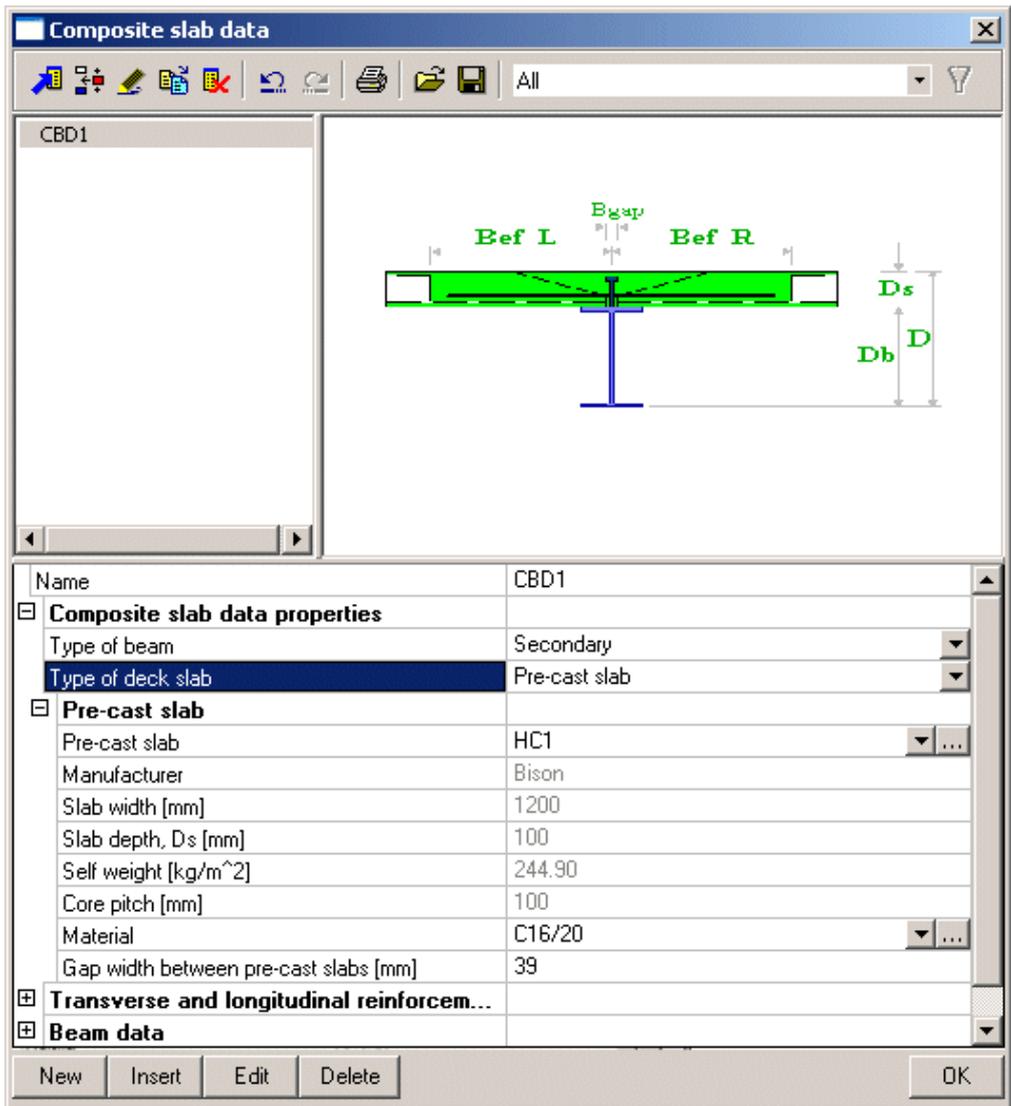


Figure 7: Slab data

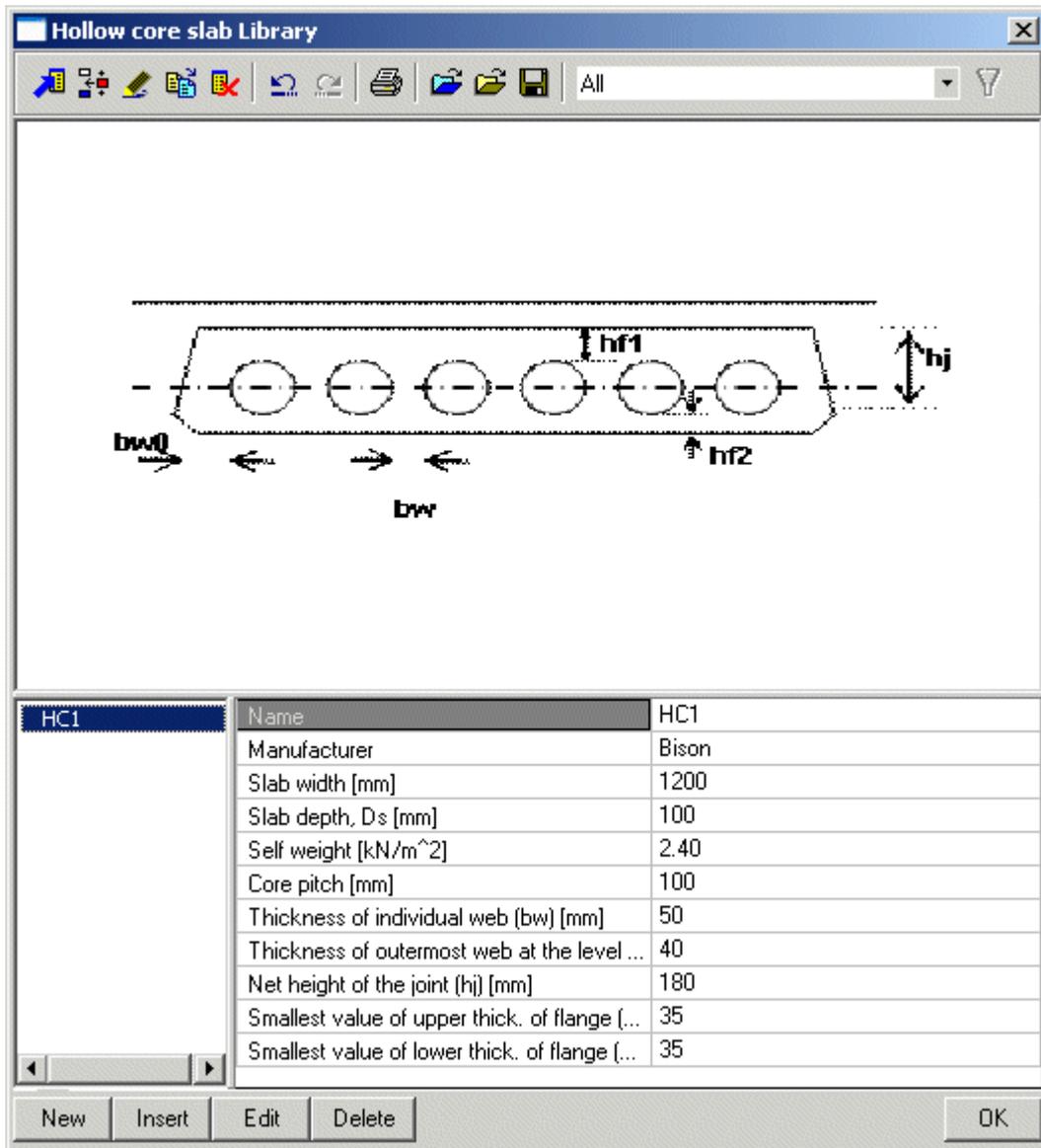


Figure 8: Pre-cast slab library

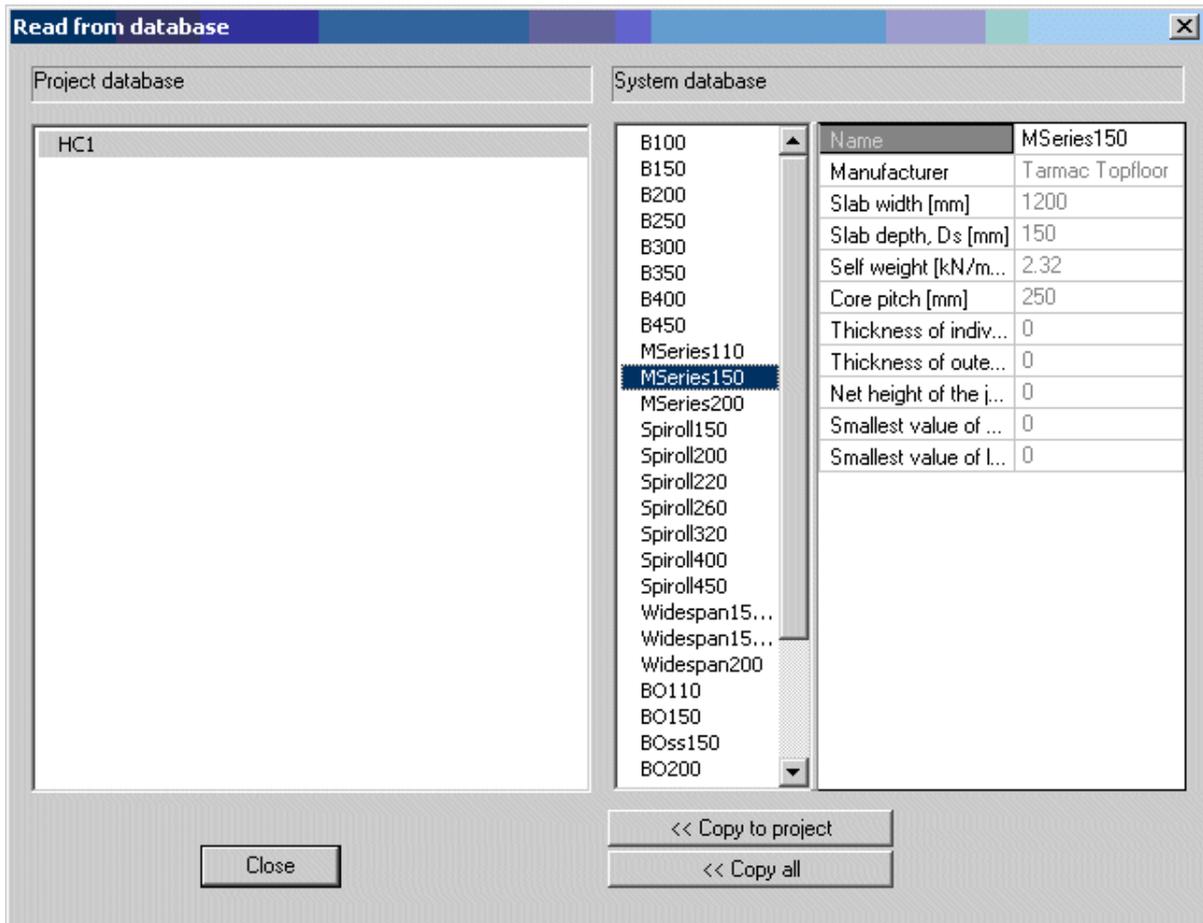


Figure 9: Pre-cast slab system database

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

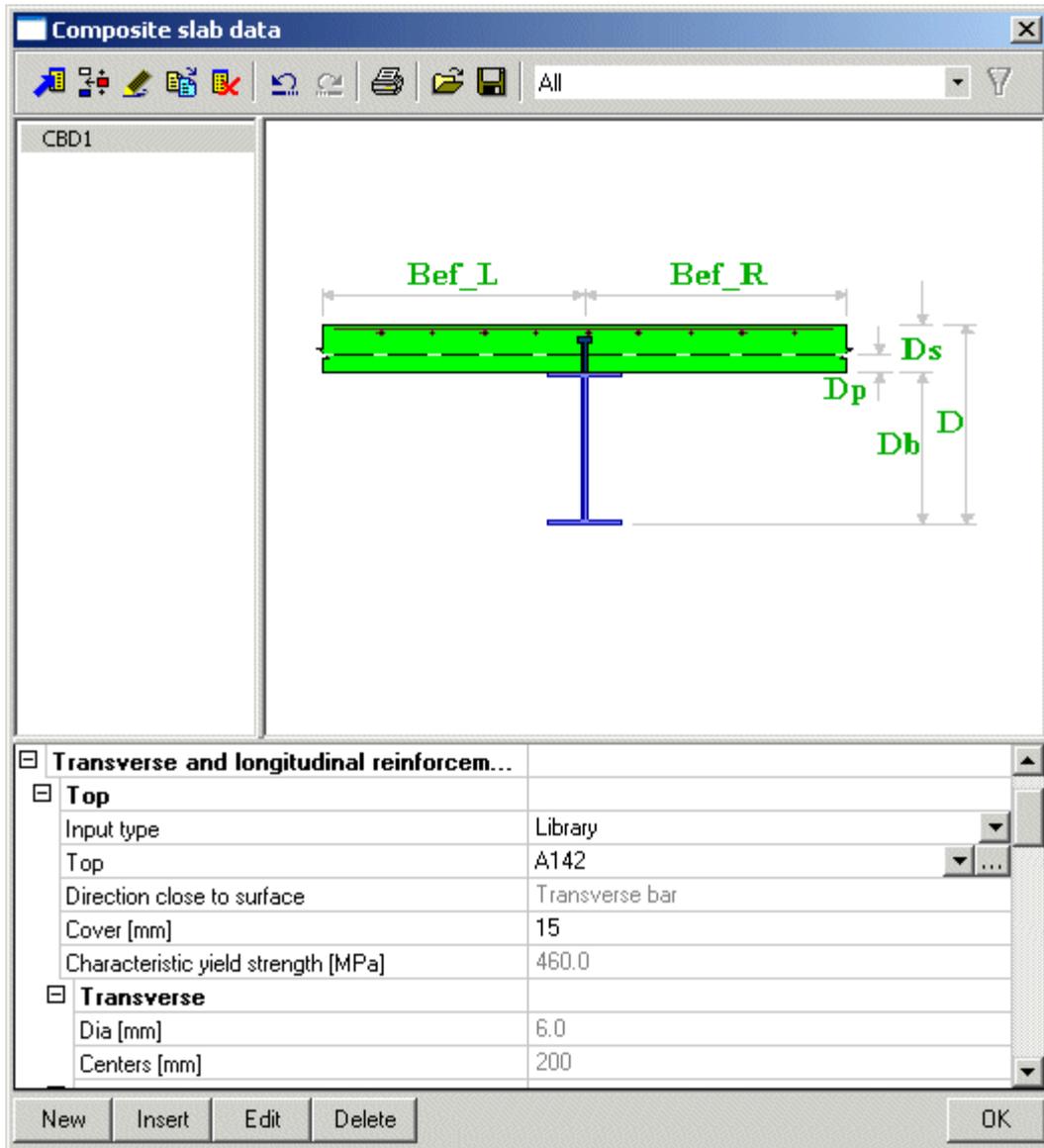


Figure 10: Reinforcement data

### Transverse 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.

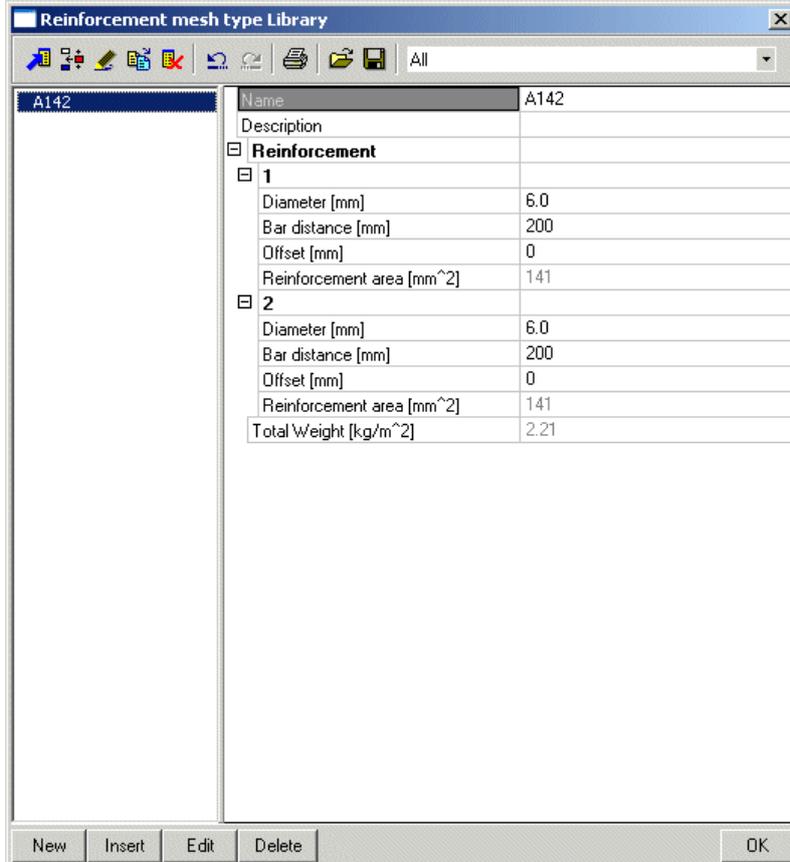
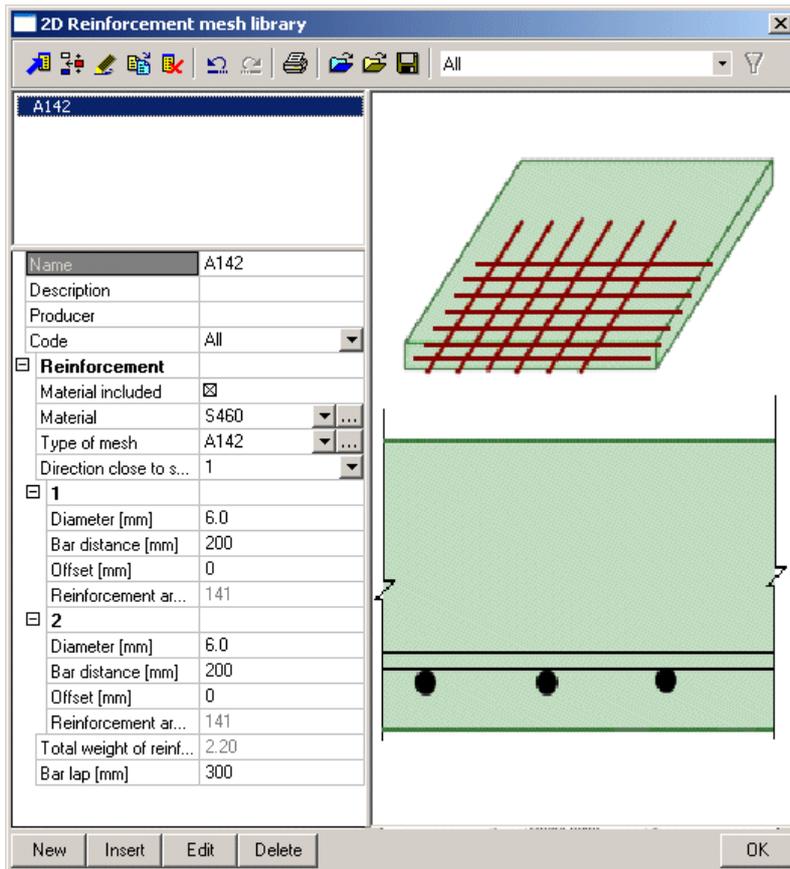


Figure 11: 2D reinforcement library Figure 12: Mesh type library

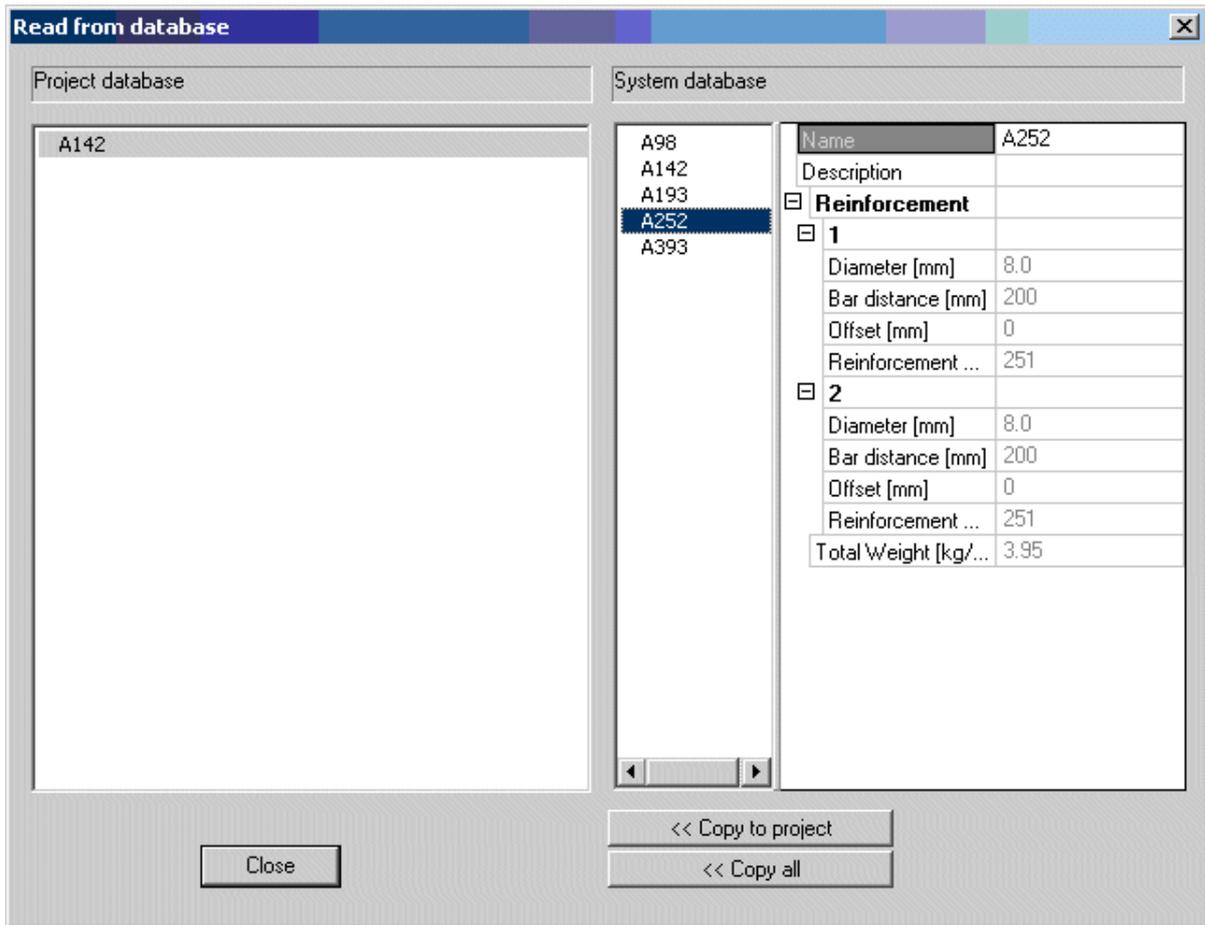


Figure 13: Reinforcement mesh system database

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

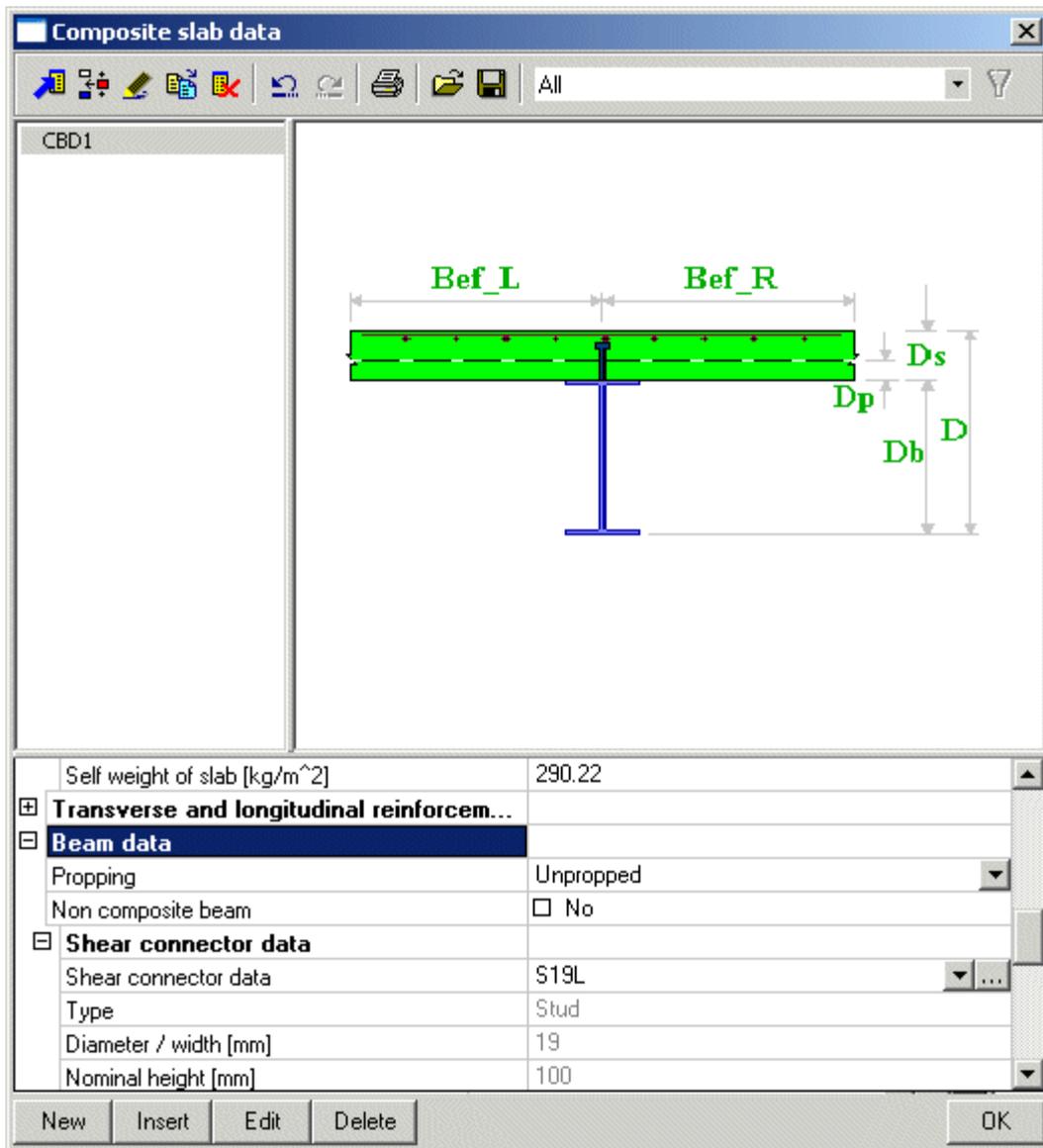


Figure 14: Beam data

### 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.

## Composite Beam Design

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.

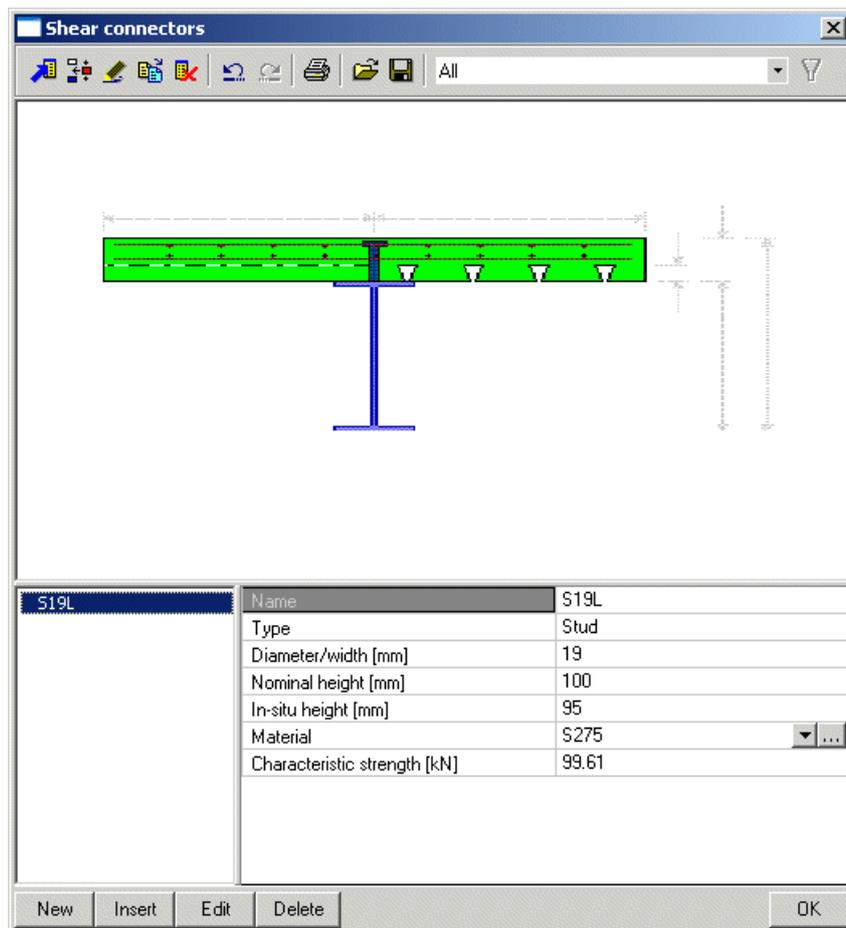


Figure 15: Shear connector library

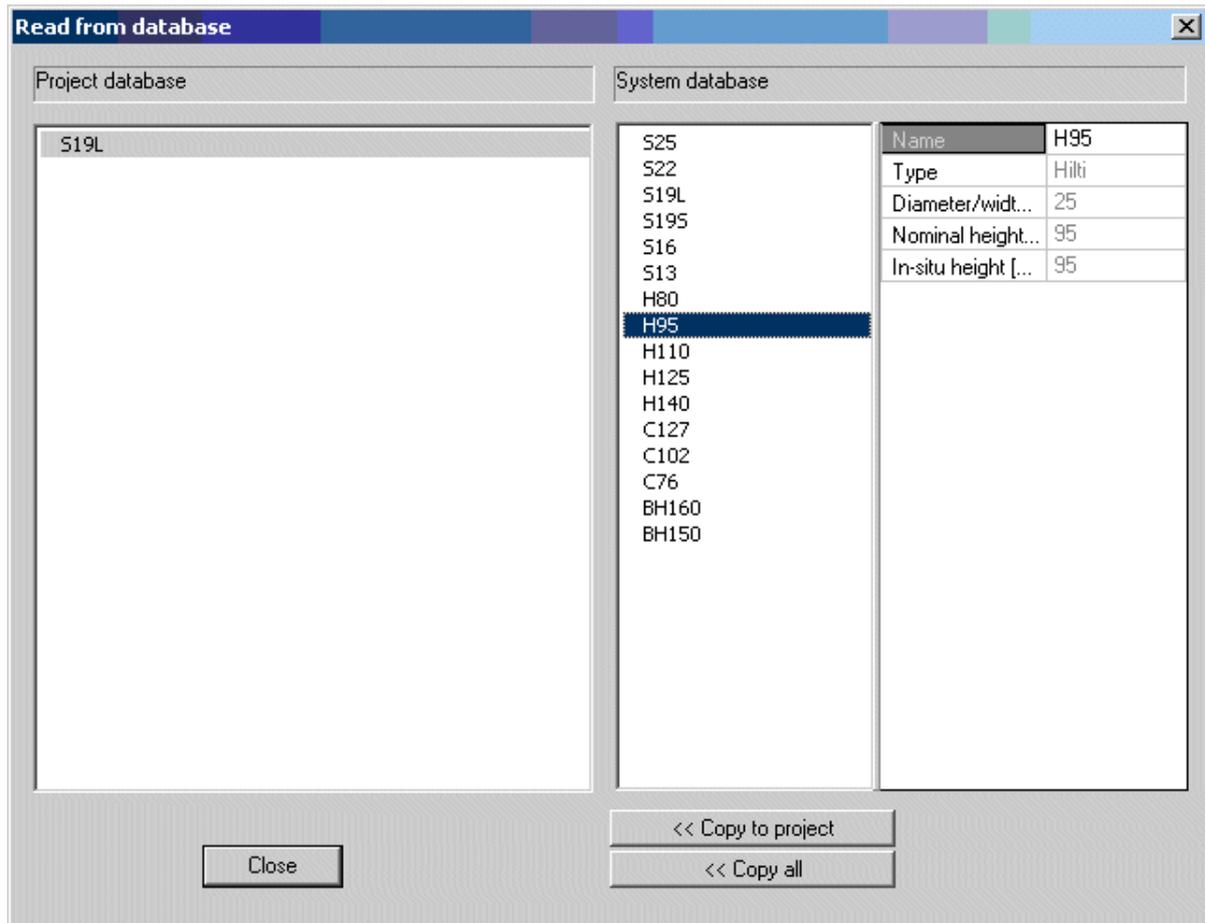


Figure 16: Shear connector system database

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.

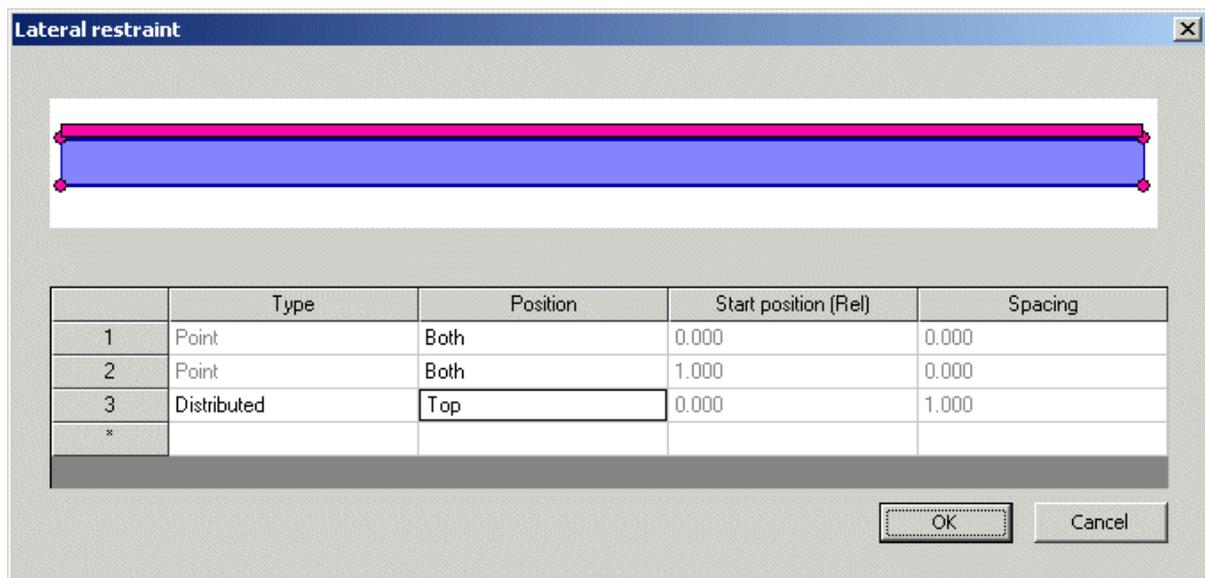


Figure 17: Lateral restraint dialog

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.

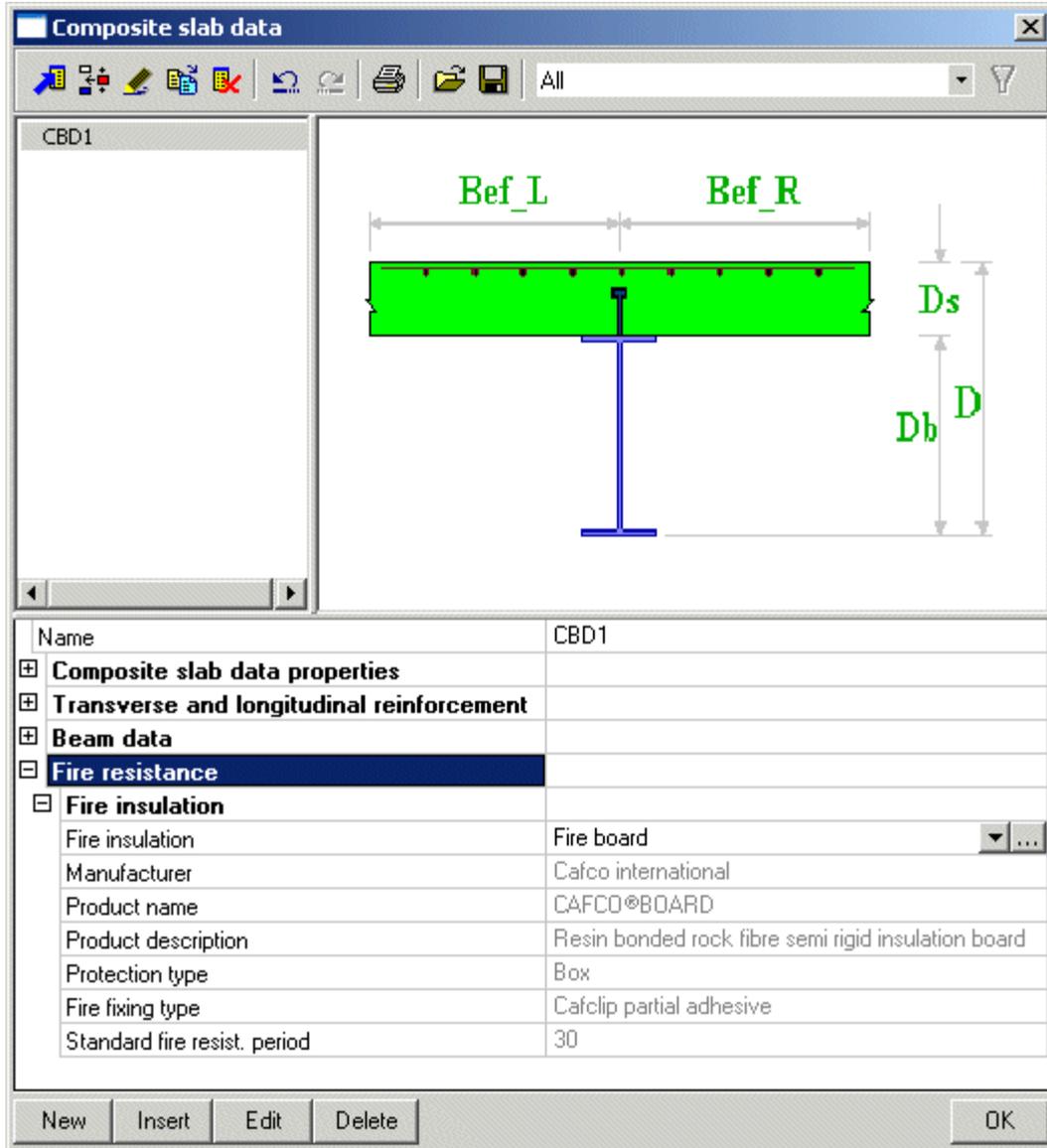


Figure 18: Fire resistance data

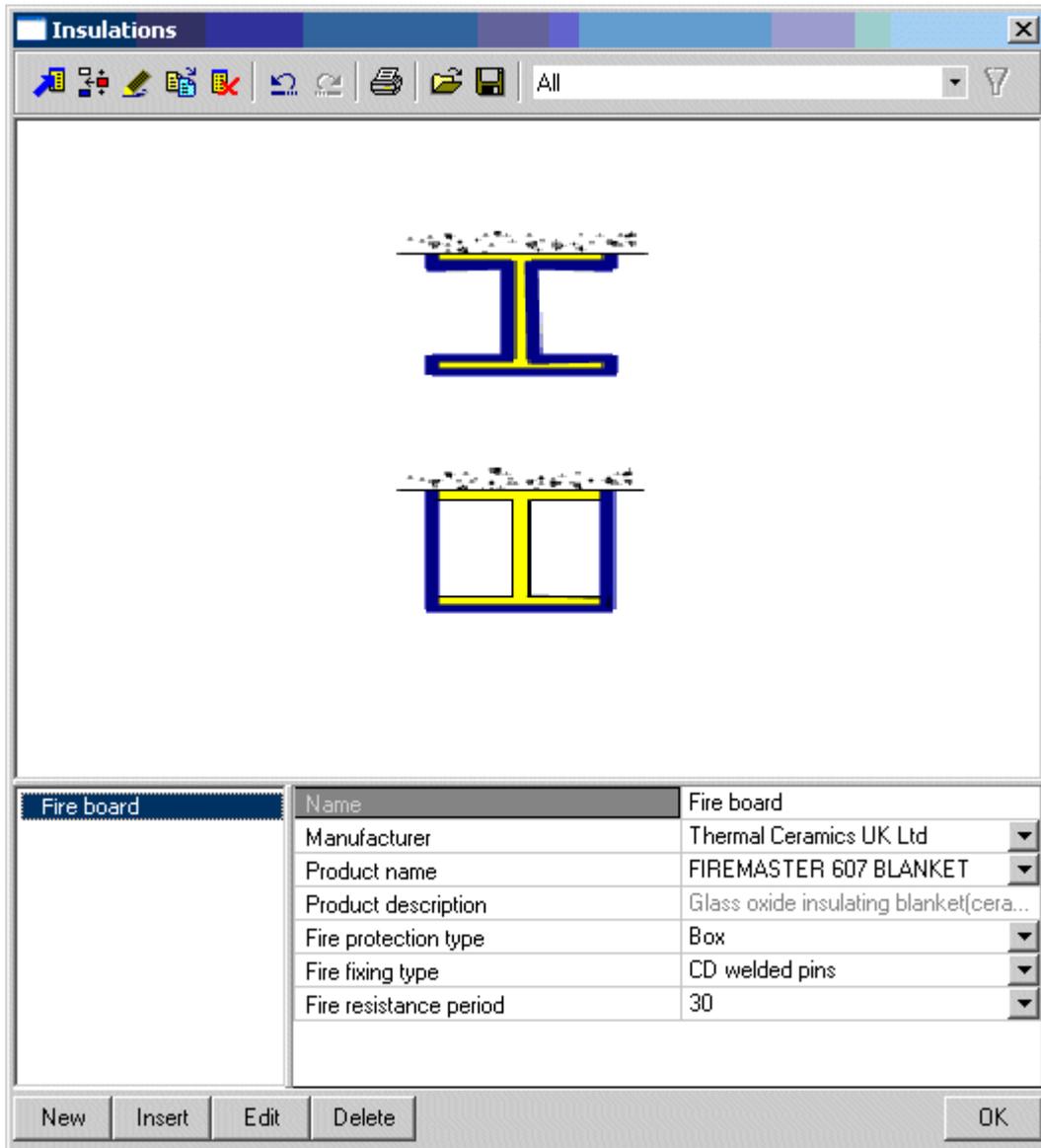


Figure 19: Fire insulation library

### 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 5950-3.1 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.

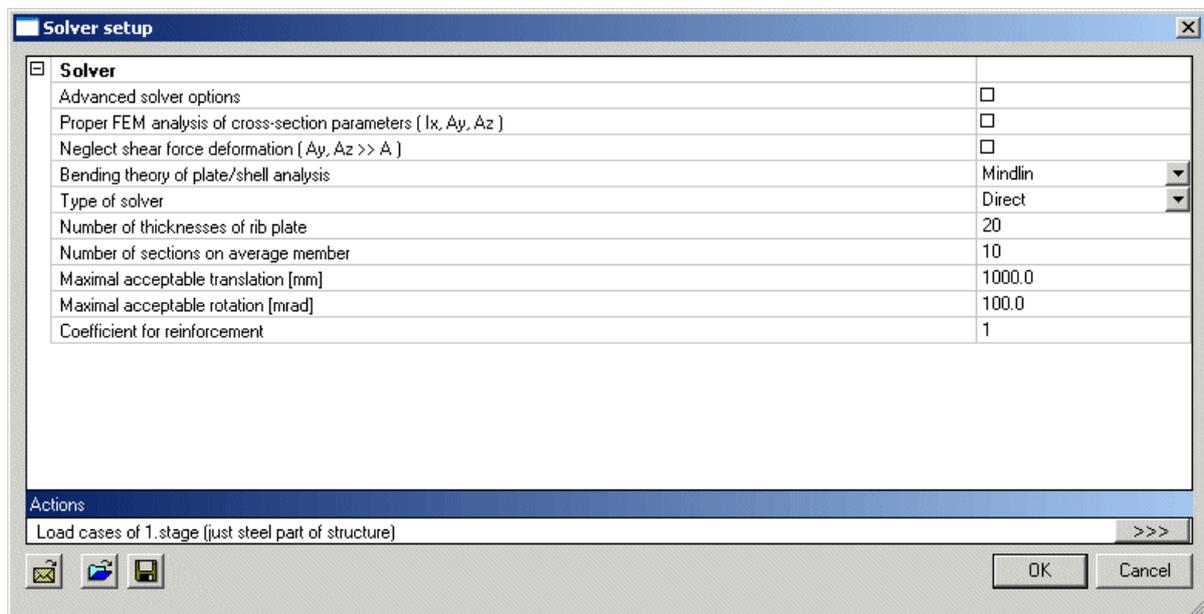


Figure 20: Solver setup

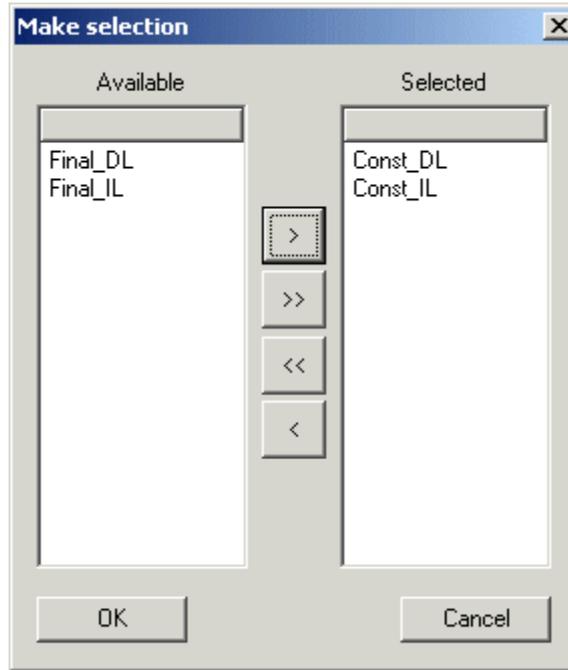


Figure 21: Construction stage load case selection

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 ■ESA 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

### 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.

**Project data** [X]

Basic data | Functionality | Loads | Combinations | Protection

Structure :  
General XYZ

Material :

Concrete	<input checked="" type="checkbox"/>
Material	C16/20
Steel	<input checked="" type="checkbox"/>
Material	S275
Timber	<input type="checkbox"/>
Other	<input type="checkbox"/>
Aluminium	<input type="checkbox"/>

Data

Name :  
Part :  
Description :  
Author :  
Date : 21. 05. 2007

Project Level : Standard  
Model : One

National Code :  
...  BS

OK Cancel

Figure 22: Project data – General XYZ

The basic frame (has shown below) is first constructed using the regular tools (Catalogue blocks) available in SCIA •ESA PT.

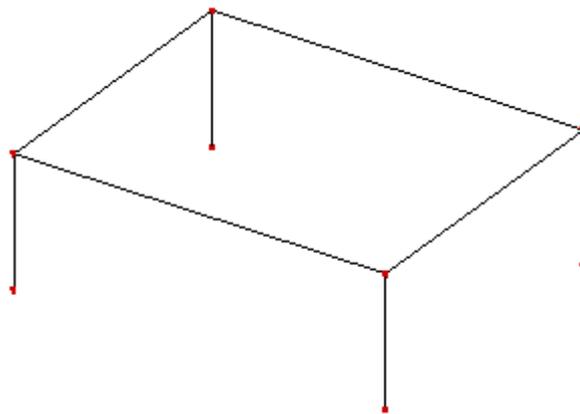


Figure 23: Simple frame

To add composite floor click on "Load/floor panel with beams"

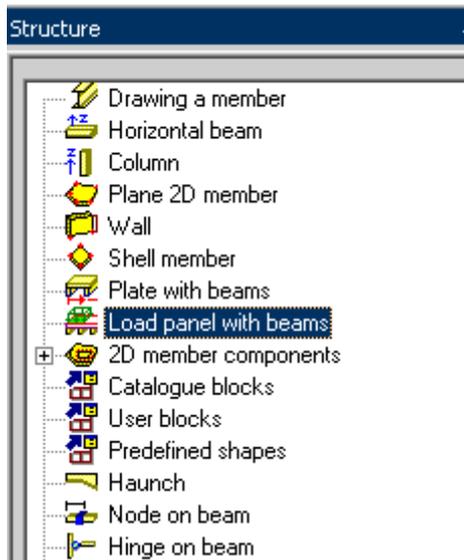


Figure 24: Structure tree

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.

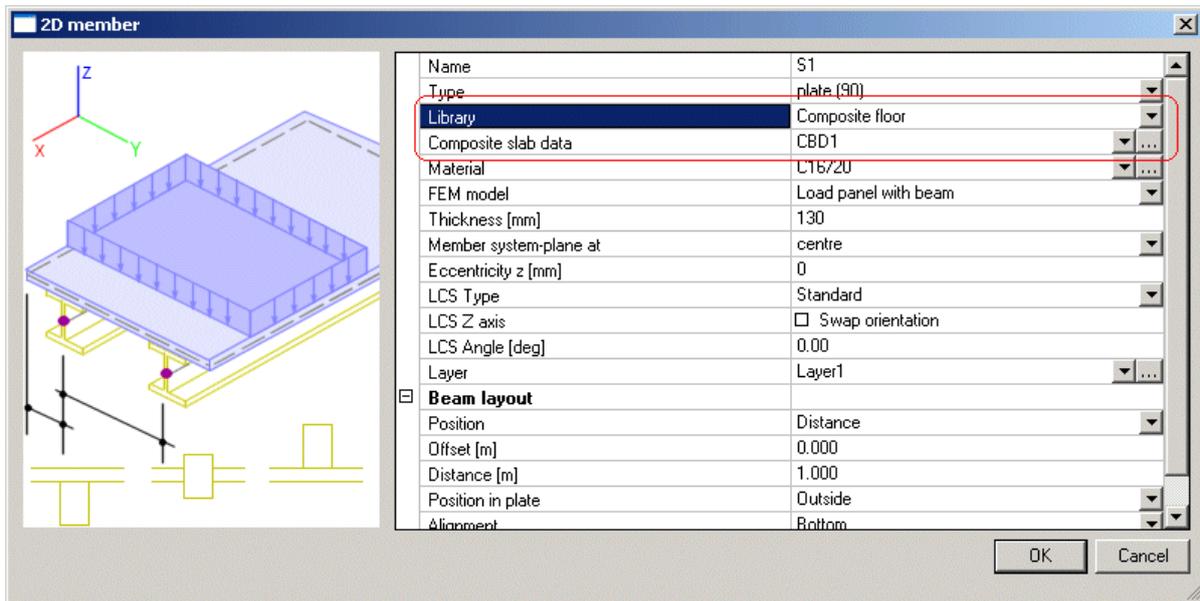


Figure 25: Composite floor property dialog

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).

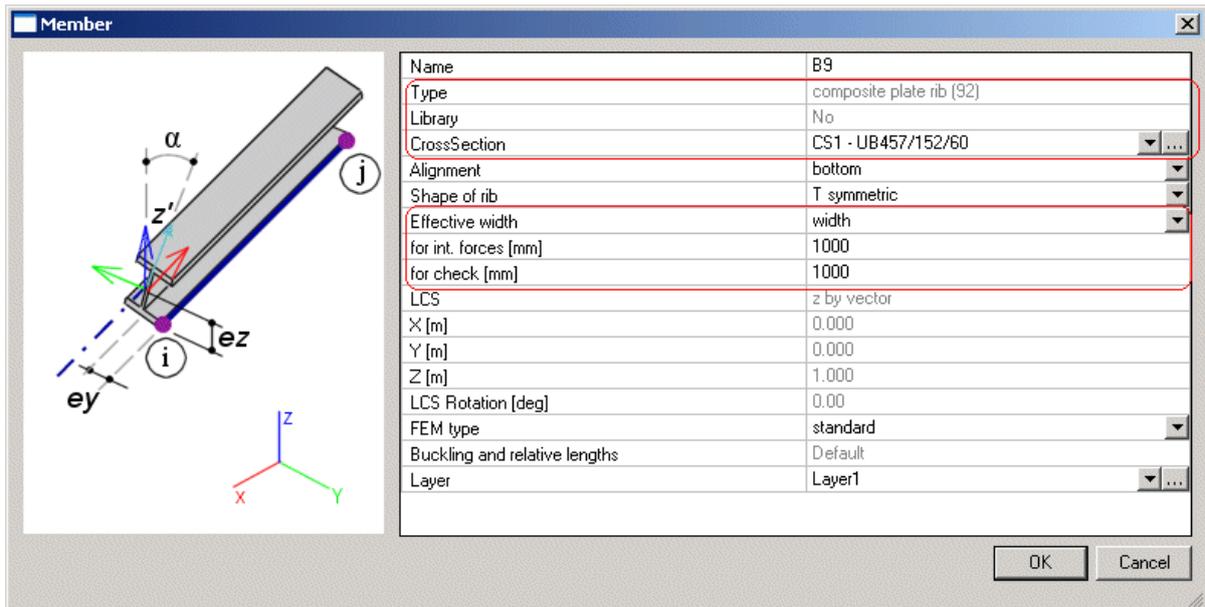


Figure 26: Secondary beam property dialog

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.

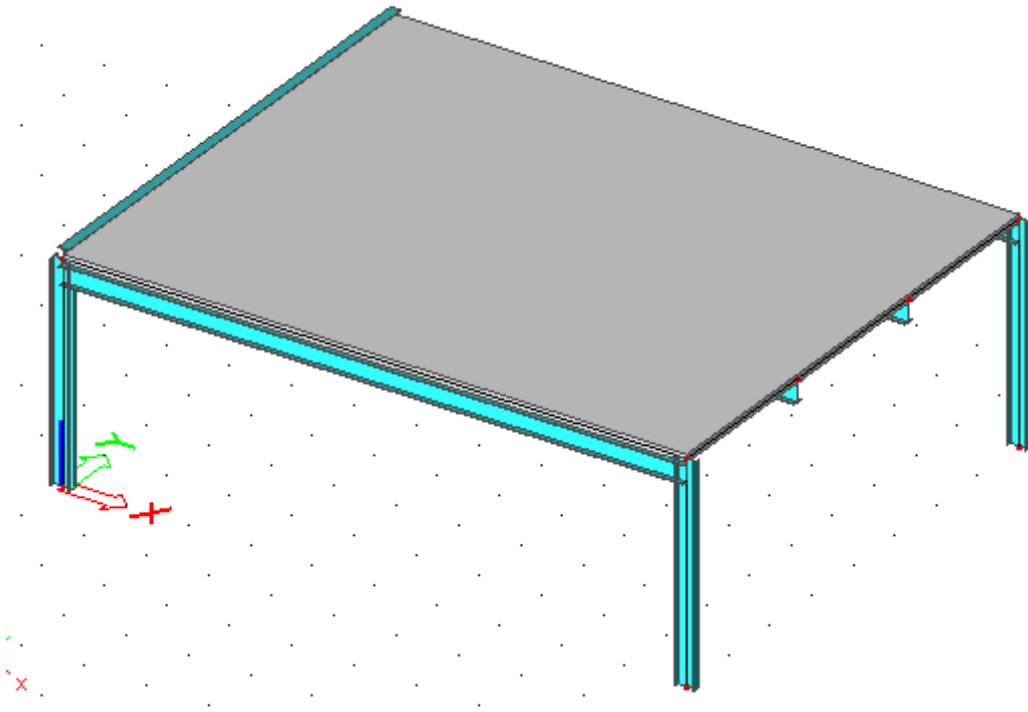


Figure 27: Composite floor panel with secondary &amp; primary beams

Surface loads applied to the plate are distributed automatically to the secondary beams during analysis. One way load distribution is assumed.

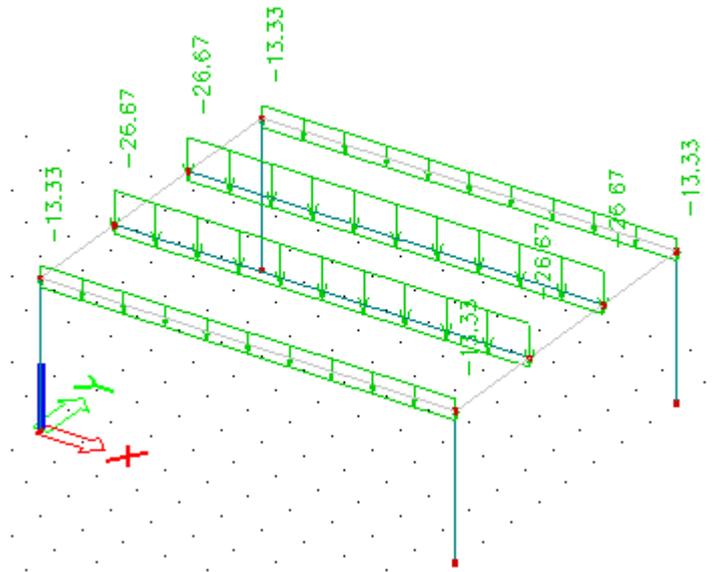
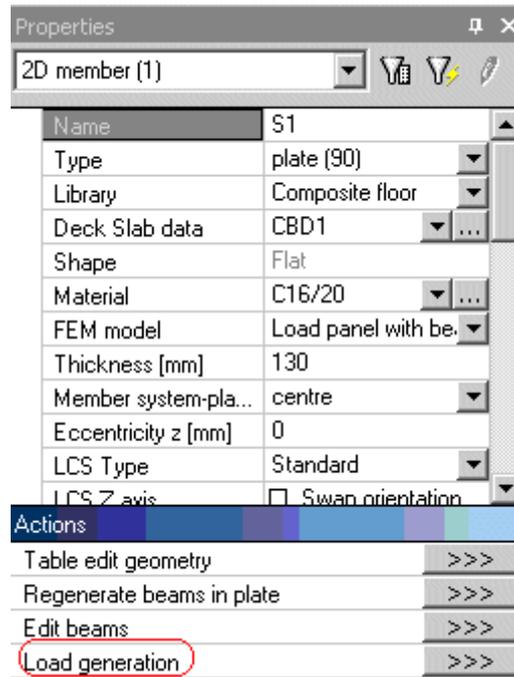


Figure 28: Member property showing load generation

Figure 29: Generated member loads from surface load

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 ( $A_y, A_z \gg A$ )" as the program does not consider composite cross section for shear area 'Ay' & 'Az' calculation.

# 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.

Material partial safety factors	BS 5950	EC4
Structural steelwork	1.00	-
Structural steelwork (cross section resistance)	-	1.00
Structural steelwork (buckling resistance)	-	1.00
Profiled sheet	1.00	1.00
Reinforcing steel	1.05	1.15
Shear connectors	1.25	1.25
Concrete in shear	1.25	1.50
Concrete in compression	1.50	1.50

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:

Deflection component	Span/deflection limit
Permanent load	240
Variable load	360
Combined load	500

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/m<sup>3</sup>. 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.

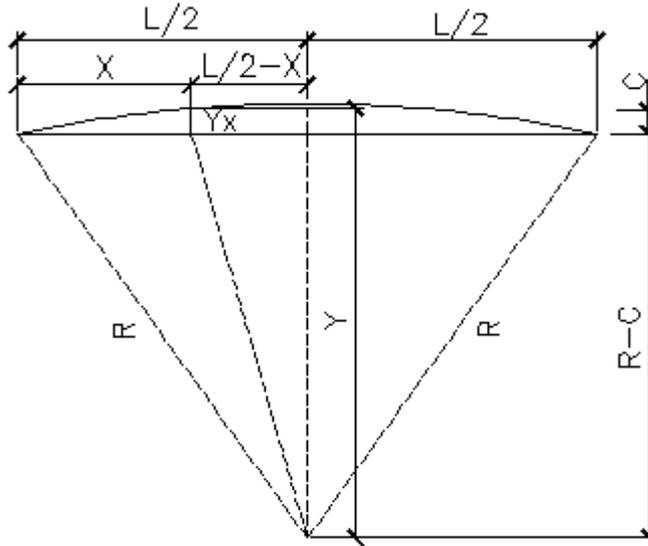
Pre-camber provided to the beam is assumed to be in a circular form. The radius of such a curve functions as the middle camber value [C] at the span [L] of the beam.

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.

$$Y = (R^2 - (L/2 - X)^2)^{1/2}$$

$$Y_x = Y - (R - C)$$



CALCULATION OF PRECAMBER

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

Material partial safety factors	BS 5950	EC4
Concrete in compression	1.5	1.00
Concrete in shear	1.25	1.00
Structural steelwork	1.00	1.00
Reinforcing steel	1.05	1.00
Shear connectors	1.25	1.00
Profile sheet steel	1.00	1.00

### 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/m<sup>2</sup> K
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 without 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

### 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.

Member data	
Name	CBD1
<b>Beam data</b>	
Propping	Unpropped
Non composite beam	<input type="checkbox"/> No
Stand alone beam	Yes
<b>Effective flange width</b>	
<b>Start zone</b>	
Left [mm]	1200
Right [mm]	1200
<b>End zone</b>	
Left [mm]	1200
Right [mm]	1200
<b>Shear connector data</b>	
Shear connector data	S19L
Type	Stud
Diameter / width [mm]	19
Nominal height [mm]	100
Insitu height [mm]	95
Number of rows	1

Figure 31: Composite member data

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 - 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.

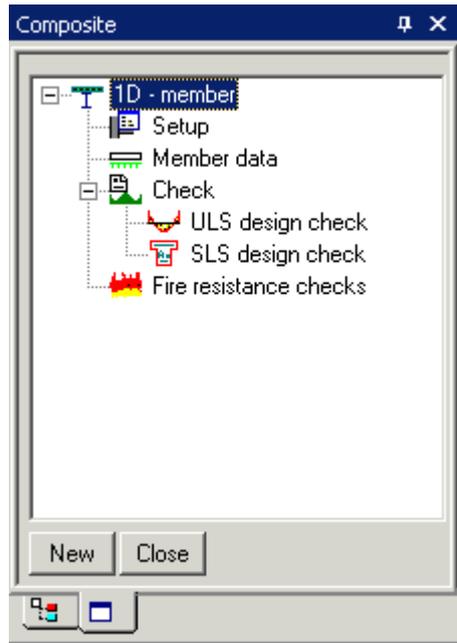


Figure 32: Composite check service

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.

## 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

ULS design check														
ULS checks - final stage														
Member	dx [m]	Comb/Case	My [kNm]	Ultimate moment capacity [kNm]	Utilisation - moment [.]	Vz [kN]	Shear capacity [kN]	Utilisation - shear [.]	Max. longitudinal shear force [kN/m]	Total long. resistance in flange [kN/m]	Utilisation - long. shear [.]	Equivalent uniform moment [kNm]	Utilisation - torsional buckling [.]	Utilisation [.]
B1	0.000	CO1/2	-90.00	132.23	0.68	90.00	340.53	0.26	0.00	92.65	0.00	-90.00	0.68	0.68

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

##### Bending moment

##### Distance from left support

- 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
  - 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.

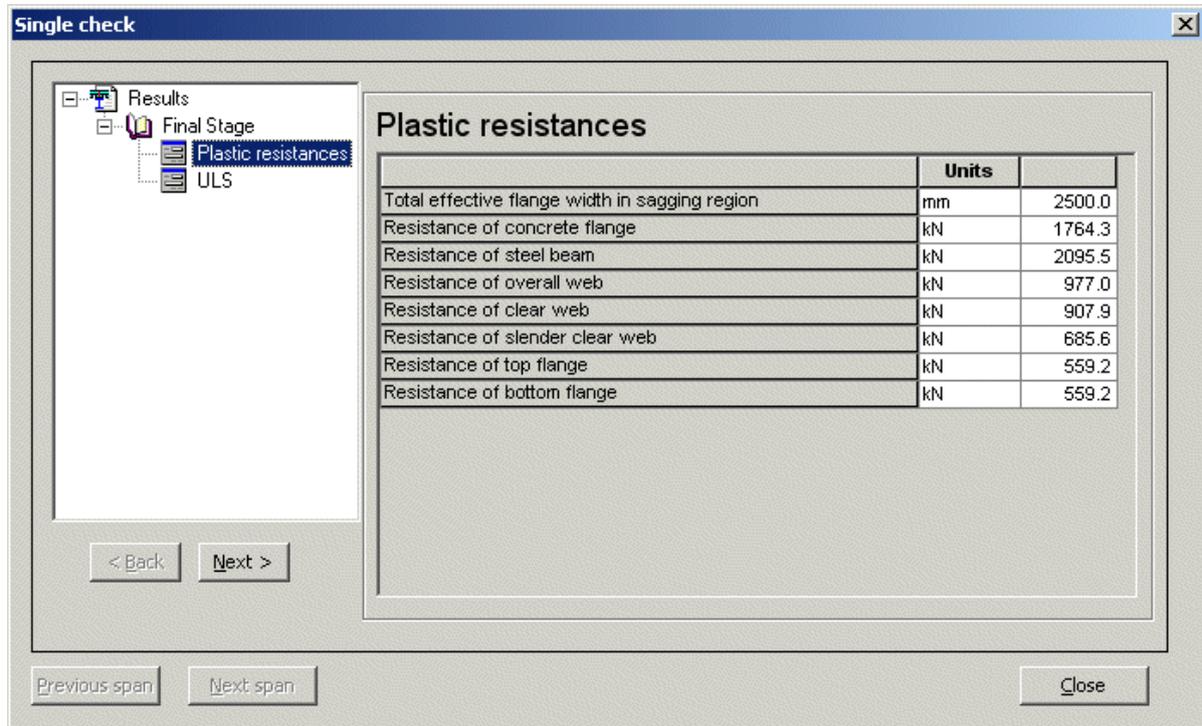


Figure 34: Final stage ULS single check output – Plastic resistances

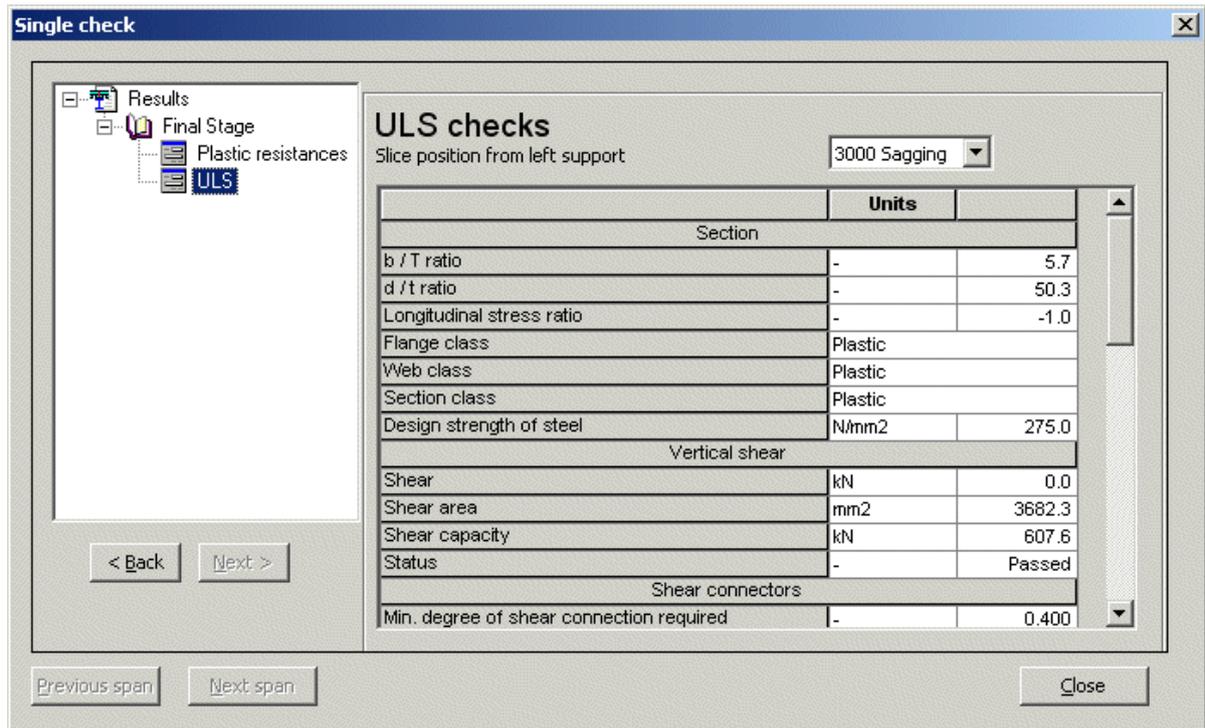


Figure 35: Final stage ULS single check output

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.

**ULS design check**

ULS design check

ULS checks - construction stage											
Member	dx [m]	Comb/Case	My [kNm]	Ultimate moment capacity [kNm]	Utilisation - moment [-]	Vz [kN]	Shear capacity [kN]	Utilisation - shear [-]	Equivalent uniform moment [kNm]	Utilisation - lateral buckling [-]	Utilisation [-]
B1	0.000	C03/1	-90.00	132.23	0.68	90.00	340.53	0.26	-90.00	0.68	0.68

Figure 36: Construction 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 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.

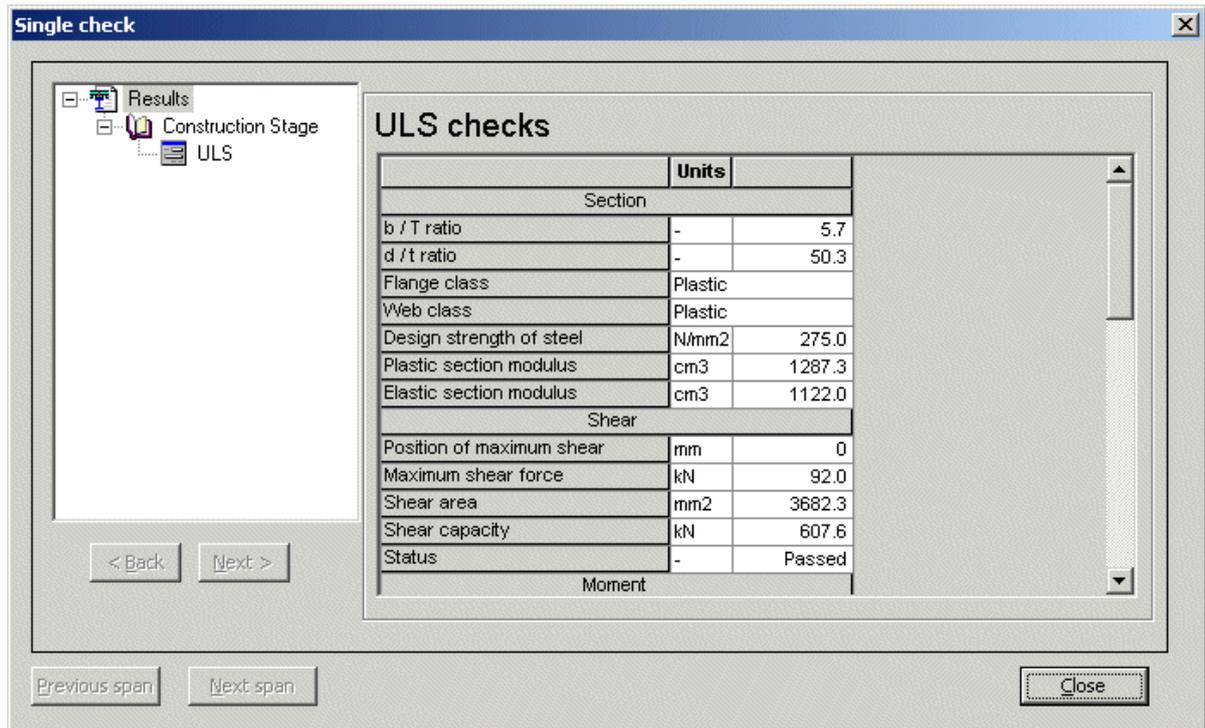


Figure 37: Construction stage ULS single check output

## 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.

#### SLS design check

SLS design check																		
Member	dx [m]	Case	Deflection due to permanent load [mm]	Limiting deflection [mm]	Unity check - permanent load deflection [ ]	Deflection due to variable load [mm]	Limiting deflection [mm]	Unity check - variable load deflection [ ]	Deflection due to combined load [mm]	Limiting deflection [mm]	Unity check - combined load deflection [ ]	Max steel stress [MPa]	Allowable steel stress [MPa]	Unity check - stress in steel [ ]	Tot stress in conc./reinf [MPa]	Allowable stress in conc./reinf [MPa]	Unity check - stress in conc / reinf [ ]	Unity check [ ]
B1	3.000	CO2/S	-1.7	-25.0	0.07	0.0	16.7	0.00	-1.7	-30.0	0.06	0.0	275.0	0.00	0.0	10.0	0.00	0.07

Figure 38: Final stage SLS brief output

**Detailed results:** The results reported are

#### Deflection check

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

Precamber

Actual deflection due to permanent load (calculated with final stage dead loads using second moment of area of steel beam only)

Allowable deflection due to permanent load

Utilisation for permanent load deflection

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

Allowable deflection due to variable load

Utilisation for variable load deflection

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

Allowable deflection due to combined load

Utilisation for combined load deflection

Natural frequency check

Actual natural frequency

Allowable natural frequency

Utilisation for natural frequency

Elastic stress check

Position of maximum moment

Moment

Second moment of area of final section

Depth of neutral axis from top of slab

Section modulus for concrete flange

Section modulus for steel flange

Actual maximum stress in concrete or reinforcement

Allowable stress in concrete or reinforcement

Utilisation for stress in concrete

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

Allowable maximum steel stress due to dead loads

Utilisation for maximum steel stress

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

Allowable maximum steel stress due to imposed loads

Utilisation for maximum steel stress

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

Allowable resultant maximum steel stress

Utilisation for resultant maximum steel stress

*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.

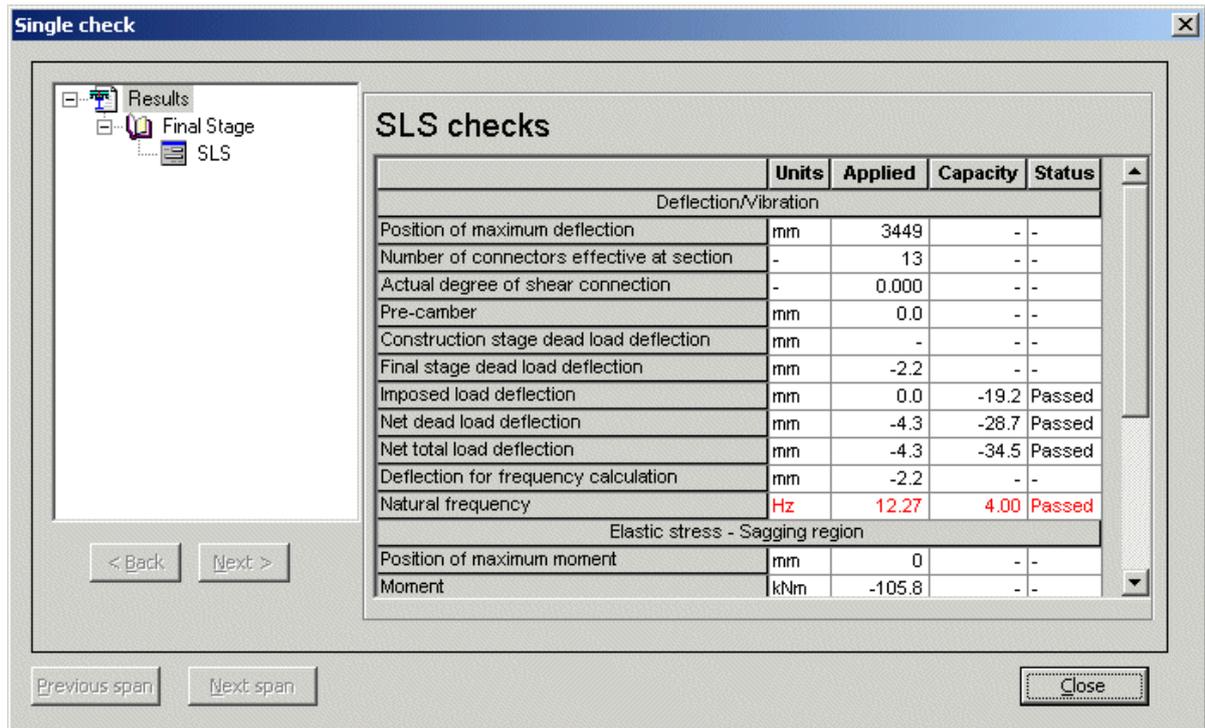


Figure 39: Final stage SLS single check output

**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.

**SLS design check**

SLS checks - construction stage															
Member name	dx [m]	Case	Deflection due to permanent load [mm]	Limiting deflection [mm]	Unity check - permanent load deflection [ ]	Deflection due to variable load [mm]	Limiting deflection [mm]	Unity check - variable load deflection [ ]	Deflection due to combined load [mm]	Limiting deflection [mm]	Unity check - combined load deflection [ ]	Max steel stress [MPa]	Allowable steel stress [MPa]	Unity check - stress in steel [ ]	Unity check [ ]
B1	3.000	C04/4	-5.7	-25.0	0.23	0.0	-16.7	1.00	-5.7	-30.0	0.19	30.0	275.0	0.11	1.00

Figure 40: Construction stage SLS brief output

*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
- Moment
- Actual steel stress
  - Allowable steel stress
  - Unity check

*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.

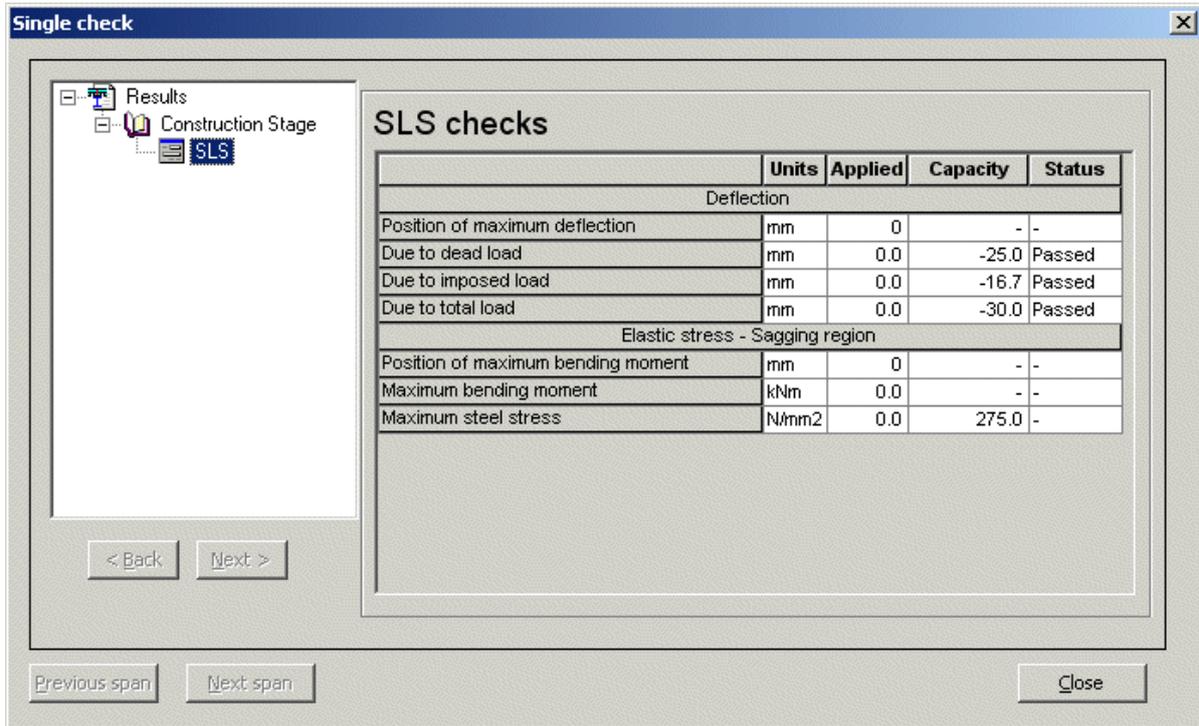


Figure 41: Construction stage SLS single check output

## 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

## Composite Beam Design

- Hollow encasement
- Contour encasement

## Insulation type

- Board protection
- Spray protection
- Intumescent protection

## **Fire insulation data for BS 5950-8:2003**

### Manufacturer

- Cafco international
- Lafarge plaster board limited
- Thermal Ceramics UK Ltd
- British Gypsum Limited
- Promat UK ITd
- Rockwool Ltd
- Cryothem insulation limited
- Bollom fire protection
- Firetherm intumescent & insulation supplies ltd
- Leigh's paints
- Tikkurila coatings Lts
- Nullifire limited
- Ameron international

### Product name

- Cafco Board
- Columnclad
- Onlit 150 Systems
- Conlit Tube
- Firecheck Board
- Firemaster 607 Blanket
- Promalit
- Promatect 250

### Fire protection type

- Profile
- Box
- Circular
- Solid

### Fire fixing type

- Cafclip partial adhesive fixing system or noggings and screw fixing systems
- Columnclad
- Glued Noggins, Dry Joints BEAMS & COLUMNS

**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:*

**Fire Resistance design check**

Fire res						
Member	dx [m]	Case	My [kNm]	Vz [kN]	Ultimate moment cap. [kNm]	Thickness of fire protection [m]
B1	0.000	C 01/2	-90.00	90.00	132.23	0.030

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

Single check:

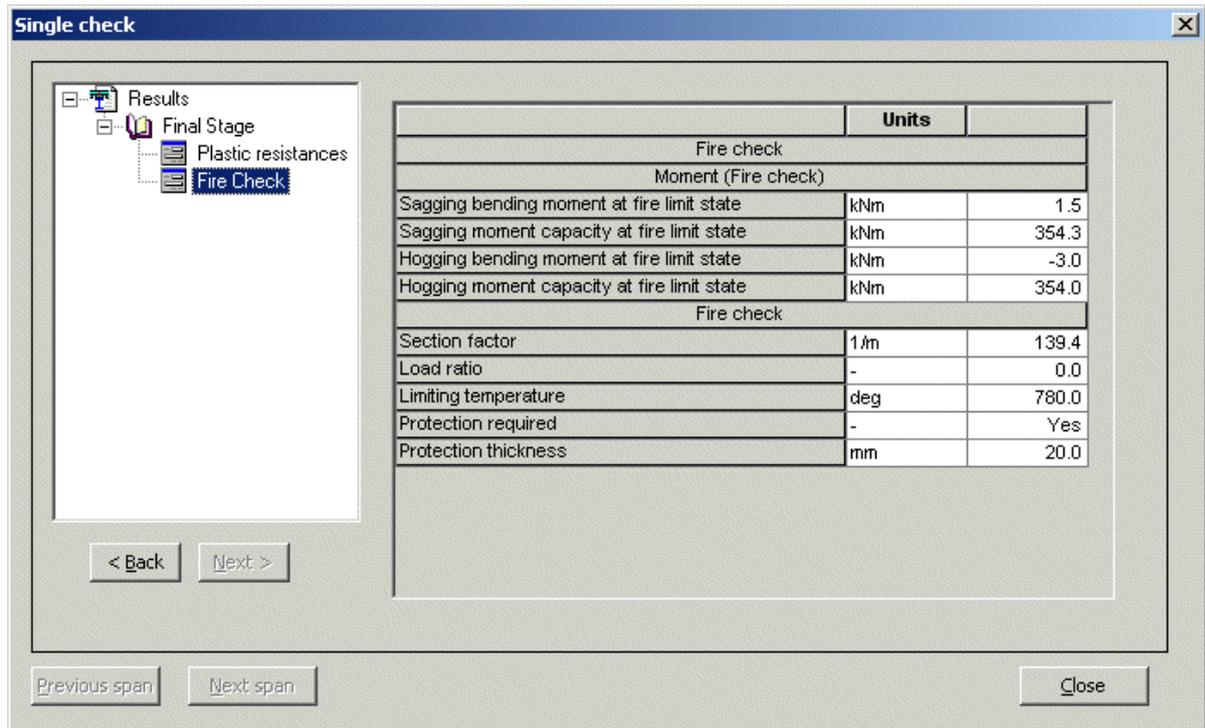


Figure 43: BS code fire check output dialog

**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**

Fire Resistance design check												
Member	dx [m]	Case	My [kNm]	Vz [kN]	Longitudinal shear force in sagging zone [kN]	Longitudinal shear resistance in sagging zone [kN]	Unity check - long. shear resistance sagg [-]	Ultimate moment cap. [kNm]	Unity check - moment [-]	Ultimate shear capacity [kN]	Unity check - shear [-]	Unity check [-]
B1	0.000	C01/1	-90.00	90.00	0.00	61.98	0.00	172.54	0.52	193.71	0.46	0.52

Figure 44: Euro code fire check brief output

Detailed results:

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:

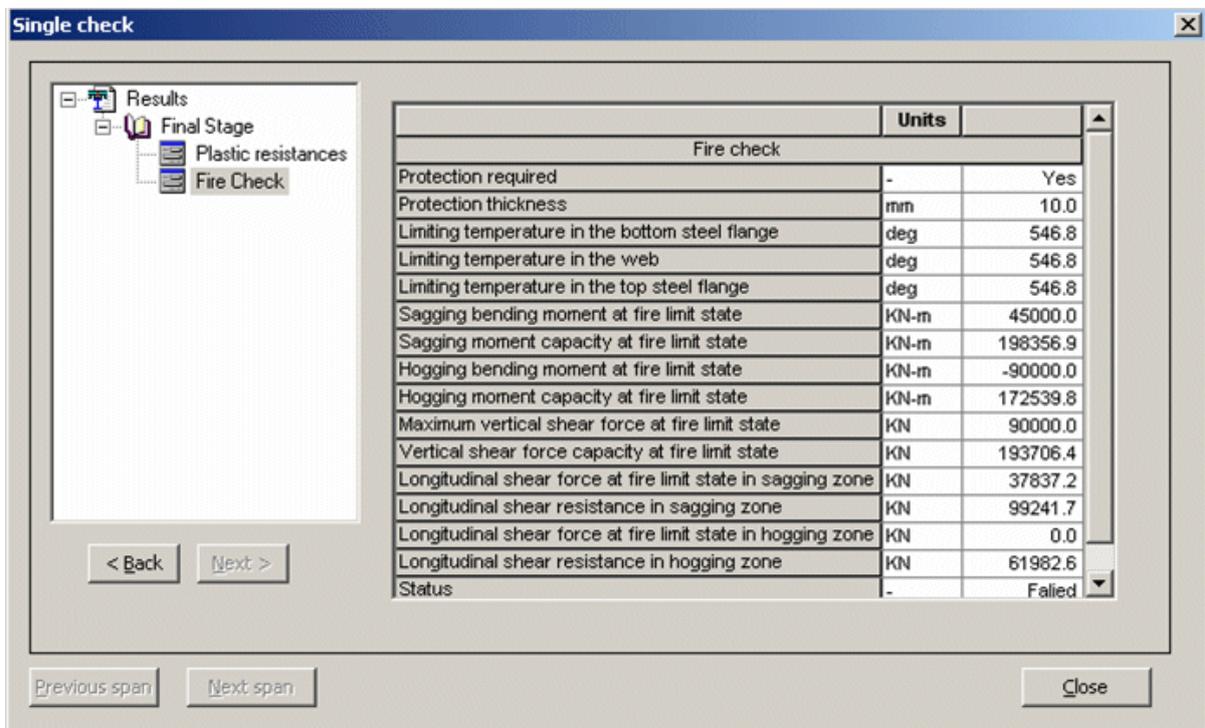


Figure 45: Euro code fire check output dialog

## 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.

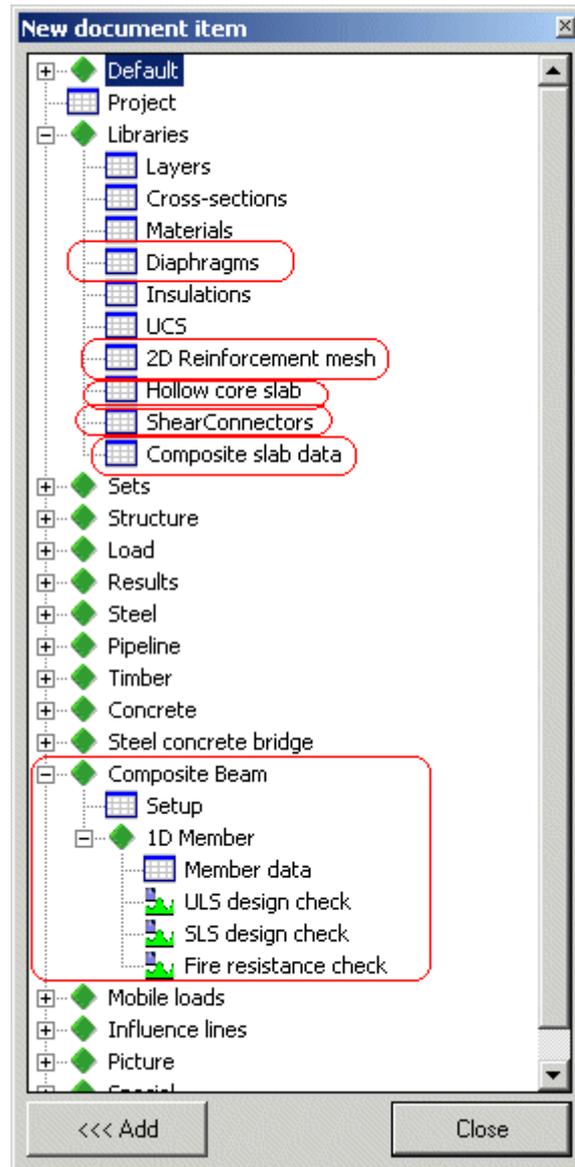


Figure 46: List of insertable items in Document

### Diaphragms (Profile deck data)

Type Name	Diaphragms	A [mm]	38
Name	Superib1-2	B [mm]	115
Manufacturer	Richard lees	C [mm]	141
Profile shape	Dovetail	D [mm]	51
I- moment [m <sup>4</sup> /m]	0.00	Thickness [mm]	1
K1+ [m/kN]	0.000	Nominal thickness [mm]	1
K2+ [m <sup>2</sup> /kN]	0.000	Weight [kN/m <sup>2</sup> ]	0.17
K1- [m/kN]	0.000	Material	S 275
K2- [m <sup>2</sup> /kN]	0.000		

Figure 47: Profile deck data output

## 2D Reinforcement mesh (data for transverse & longitudinal reinforcement)

Type Name	2D Reinforcement mesh
Name	A142
Code	All
Material included	✓
Type of mesh	A142
Direction close to surface	1
Diameter [mm]	6.0
Bar distance [mm]	200
Offset [mm]	0
Reinforcement area [mm <sup>2</sup> ]	141
Diameter [mm]	6.0
Bar distance [mm]	200
Offset [mm]	0
Reinforcement area [mm <sup>2</sup> ]	141
Total weight of reinforcement [kg/m <sup>3</sup> ]	2.20
Bar lap [mm]	300

Figure 48: Reinforcement data output

## Hollow core slab (Pre-cast slab data)

Type Name	Hollow core slab
Name	HC1
Manufacturer	Bison
Slab width [mm]	1200
Slab depth, Ds [mm]	100
Self weight [kN/m <sup>3</sup> ]	2.40
Core pitch [mm]	100
Thickness of individual web (bw) [mm]	50
Thickness of outermost web at the level of elastic gravity line (bwo) [mm]	40
Net height of the joint (hj) [mm]	180
Smallest value of upper thick. of flange (hf, 1) [mm]	35
Smallest value of lower thick. of flange (hf, 1) [mm]	35

Figure 49: Pre-cast slab data output

## Shear Connectors (Shear connector data)

Name	Type	Diameter/width [mm]	Nominal height [mm]	In-situ height [mm]	Material
S19L	Stud	19	100	95	S 275

Figure 50: Shear connector data output

### Composite slab data

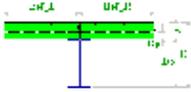
Deck slab data properties	Name		CBD1		
	Type of beam		Secondary		
	Type of deck slab		Profile deck		
	Profile deck		Profile deck	Superib1-2	
	Total slab depth, Ds [mm]		130		
	Profile Depth, Dp [mm]		51		
	Profile shape		Dovetail		
	Manufacturer		Richard lees		
	Weight [kN/m <sup>2</sup> ]		0.17		
	Effective thickness [mm]		1		
	Rib centers [mm]		153		
	Rib width [mm]		115		
	Yield strength [MPa]		275.0		
	Left side rib direction		Perpendicular		
	Right side rib direction		Perpendicular		
Contribution to longitudinal shear resistance		k			
Sheet continuous		k			
Shear connector welded through profile sheeting		✓			
Self weight of slab [kg/m <sup>2</sup> ]		301.94			
Type of deck slab					
Transverse and Longitudinal reinforcement	Top	Input type		Library	
		Top		A142	
		Direction close to surface		Transverse bar	
		Cover [mm]		15	
		Characteristic yield strength [MPa]		0.0	
		Transverse		Dia [mm]	6.0
				Centers [mm]	200
		Longitudinal		Left support zone	Dia [mm] 6.0 Centers [mm] 200
				Span zone	Dia [mm] 6.0 Centers [mm] 200
				Right support zone	Dia [mm] 6.0 Centers [mm] 200
Bottom		Input type		None	
Haunch		Input type		None	
Beam data	Propping		Unpropped		
	Non composite beam		k		
	Shear connector data		Shear connector data	S19L	
			Type	Stud	
			Diameter / width [mm]	19	
			Nominal height [mm]	100	
			Insitu height [mm]	95	
			Input type	Calculated	
			Characteristic strength [kN]	64.83	
			Number of rows	1	
		Transverse spacing [mm]	0		
		Longitudinal spacing	153.00		
Fire resistance	Temperature - Time curve		By setup		
	Standard fire resist. period		30		
	Protection		Yes		
	Fire insulation		Fire insulation	Fire board	
			Encasement type	Hollow encasement	
			Insulation type	Board protection	
			Unit mass [kg/m <sup>3</sup> ]	150.00	
			Thermal conductivity [W/mK]	2.0000e-001	
		Specific heat [J/gK]	1.2000e+000		
		Default value for thickness [mm]	10		

Figure 51: Composite slab data output

## Setup (Design parameters)

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

Material PSF-Structural steel work-resistance to cross-sections [-]	1.00
Material PSF-Structural steel work-resistance to buckling [-]	1.00
Material PSF-Reinforcing steel [-]	1.15
Material PSF-Concrete [-]	1.50
Material PSF-Design shear resistance of headed stud [-]	1.25
Material PSF-Profile steel sheet [-]	1.00
Calculation parameters-Type of concrete	Normal
Calculation parameters-Proportion of total load that is long term [-]	0.33
Calculation parameters-Perc. Of variable load accounted for calc. of natural frequency [-]	10.00
Calculation parameters-Perc. Of increase in inertia to allow for dynamic stiffness [-]	10.00
Calculation parameters-Natural frequency [-]	4.00
Calculation parameters-Value of factor C4 for LTB check (C4) [-]	6.20
Calculation parameters-Creep coefficient [-]	5.00
Calculation parameters-Creep multiplier [-]	0.55
Fire resistance-Concrete [-]	1.00
Fire resistance-Structural steel [-]	1.00
Fire resistance-Reinforcing steel [-]	1.00
Fire resistance-Design shear resistance of headed stud [-]	1.00
Fire resistance-Profiled sheet steel [-]	1.00
Fire resistance-Temperature curve	Standard temp-time curve
Fire resistance-Coefficient of heat transfer by convection [-]	25.00
Fire resistance-Emissivity related to fire compartment [-]	0.80
Fire resistance-Emissivity related to surface material [-]	0.63
Fire resistance-Analysis type	resistance domain
Span/def limits-Permanent load [-]	240.00
Span/def limits-Variable load [-]	360.00
Span/def limits-Combined load [-]	200.00
Pre-camber-Absolute value (mm) [-]	10.00

Figure 52: Design parameters output