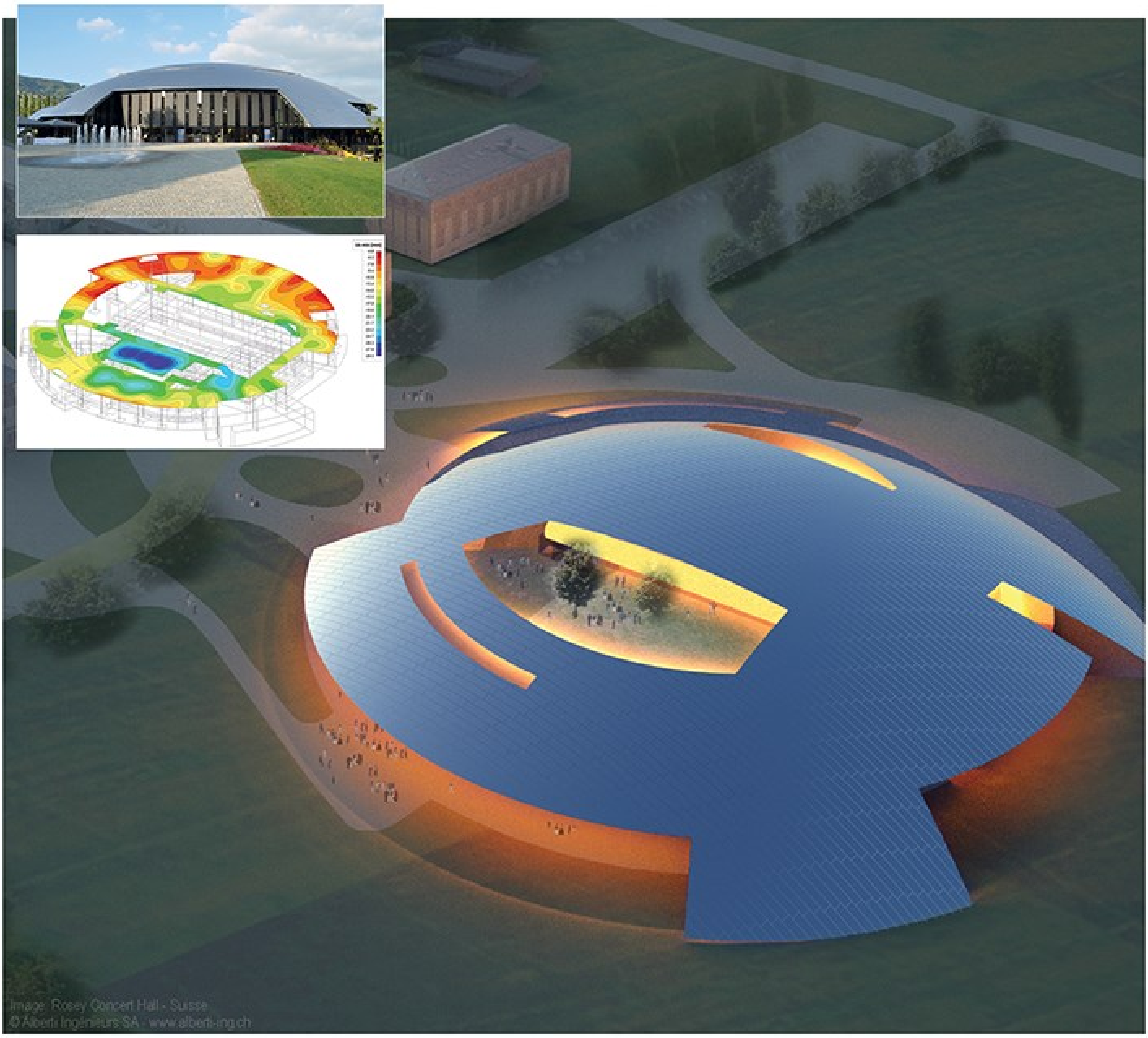
Scia Engineer



Composite Design

Steel-concrete composite structures

Introduction

This document describes the Composite Analysis Model (CAM) in Scia Engineer. CAM is used in Scia Engineer to analyse and design composite beams that comprise:

- a heavy gauge steel profile,
- a reinforced concrete deck,
- shear connectors between steel beam and deck,
- and corrugated steel sheeting that serves also as formwork for the wet concrete during the stages of construction.

The CAM is a generic modelization and numerical analysis method that aims to analyse accurately the behaviour of steel-concrete composite structures. As the CAM is based on standard 3D modelization tools of Scia Engineer, there are no restrictions regarding the geometry of the structure. At this time, any structure that contains composite floors may be analysed using it (buildings, bridges, industrial structures...). As results, the CAM provides deflections and internal forces that can be used in composite beam design based on design code checks.

Fundamentally, a composite deck with beams is modelized as a plate with eccentric ribs. The plate represents the composite deck, which is in itself a composite structural element made of a profiled steel sheeting with a reinforced concrete topping. The steel beams are represented by eccentric 1D members, i.e. plate ribs, connected to the plate. That presents several advantages in comparison to more traditional approaches where the supporting structure is modelized as a beam grid:

- no load panels are necessary to distribute loads to the beams; also, no simplified assumptions are needed regarding that load distribution
- complex geometries of the composite decks can be taken into account without simplifying assumptions
- the in-plane stiffness of the deck is automatically calculated and taken into account, as the deck is modelized as a structural member; thus no additional simplifying assumptions are needed in the case of horizontal loading of the structure

In a simple as straight-forward way, the CAM takes into account the three main construction phases important in the design of composite structures:

- the construction stage, during which the steel beams and sheeting alone carry the wet concrete and take up any applied loads; the self-weight of freshly cast concrete is calculated and directly taken into account as part of the self-weight of the structure
- the final stage for long term actions, where the composite effect is taken into account; the effect of creep is taken into account by a reduced stiffness of the concrete parts
- the final stage for short term actions, where the composite effect is taken into account with the nominal stiffness of concrete

The effect of creep may be optionally disabled. A setting also allows for all composite parts to be considered as propped during the stage of construction.

Version

The current version of this manual applies to Scia Engineer 15. In this release, the focus has been set on the analysis of composite decks with beams.

License

The functionality described in this manual requires one of the following license:

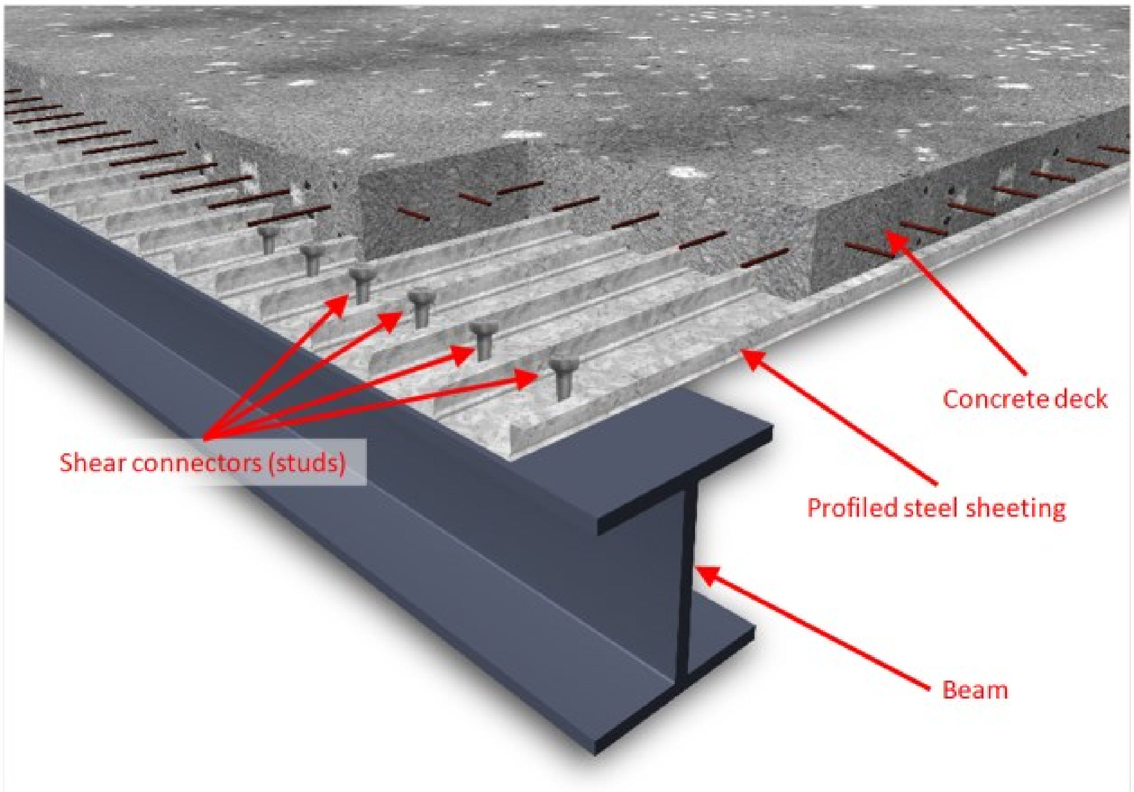
- esacbd.01.01 - Composite Beam Design - EN1994
- esacbd.01.05 - Composite Beam Design - AISC 360-10

Theoretical background

This chapter describes the theoretical background that is used in the composite analysis model (CAM) of Scia Engineer. Some aspects of it are general while some other are focussing only on the methods that are implemented in Scia Engineer. All principles that are presented here are code independent.

General principles

On the contrary to more traditional approaches, the CAM is based on a standard 3D modelization of the structure. Fundamentally, a composite deck with beams is modelized as a plate with eccentric ribs. The plate represents the composite deck, which is in itself a composite structural element made of a profiled steel sheeting with a concrete topping. The steel beams are represented by eccentric 1D members, i.e. plate ribs, connected to the plate.



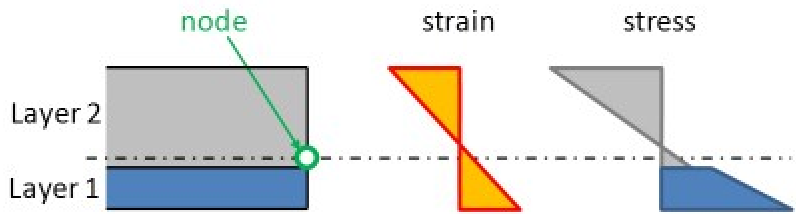
Composite deck

In the context of the CAM, the "deck" is the plate that carries the loads and transfers them to the beams. This chapter describes the principle of analysis of that plate only. The behaviour of composite beams will be discussed in another chapter.

There are composite decks and metal decks. A composite deck has two layers: a profiled steel sheeting and a concrete topping, reinforced or not. A metal deck has only one layer, i.e. the profiled steel sheeting, and is mostly used for light-weight roofs.

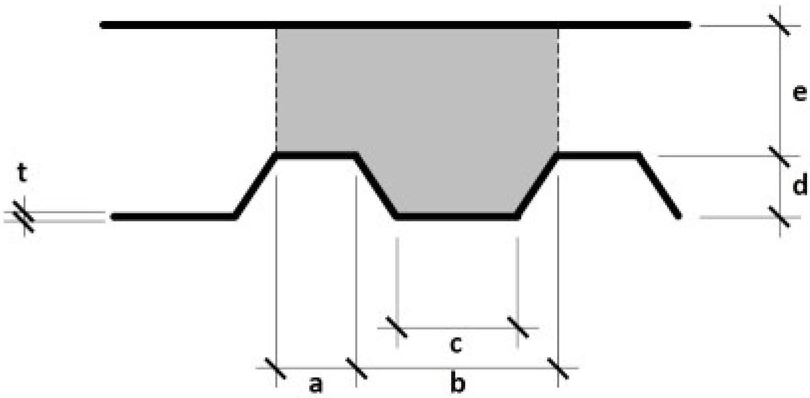
A composite deck is modelized as a multi-layered plate. Each layer has orthotropic properties and the eccentricity of each layer is taken into account.

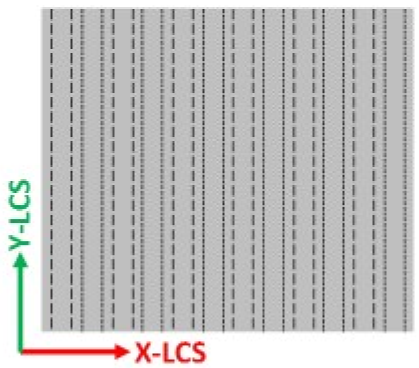
The interaction of the layers is considered as a perfect bond, i.e. without any slip between the layers (concrete and steel sheeting). The strains are determined from the displacements and rotations at the nodes of the finite element mesh.



The assumption of perfect bond is definitely reasonable for the longitudinal behaviour of the composite deck, i.e. in the direction parallel to the corrugation. In the direction perpendicular to the corrugation, this seems less obvious, since the profiled sheeting properties are first determined independently. In the composite deck, the "accordion" behaviour of the sheeting will be stabilized by the concrete in case the sheeting is in compression. However, the stiffness of the sheeting in that direction is very low and will hardly influence the behaviour of the composite deck. That approximation is therefore acceptable.

Profiled steel sheeting





E_s, G_s, ν_s = Young's modulus, shear modulus and Poisson's ratio of steel

a, b, c, d, t define the geometry of the profiled sheeting; e is the thickness of the concrete topping and is not used in this context.

The formulas below give the components of the equivalent orthotropic properties of a generic profiled steel sheeting as shown in the above picture. Formulas adapted from Samanta & Mukopadhyay [1, 2].

Bending components

$$D'_s = \begin{bmatrix} D'_{s,11} & D'_{s,12} & 0 \\ & D'_{s,22} & 0 \\ sym & & D'_{s,33} \end{bmatrix}$$

$$D'_{s,11} = \frac{E_s \cdot t^3}{12 \cdot k} \quad D'_{s,22} = \frac{E_s \cdot I}{B} \quad D'_{s,12} = \nu_s \cdot \sqrt{D'_{s,11} \cdot D'_{s,22}} \quad D'_{s,33} = k \cdot \frac{E_s \cdot t^3}{6 \cdot (1+\nu_s)}$$

$$G'_s = \begin{bmatrix} D'_{s,44} & 0 \\ 0 & D'_{s,55} \end{bmatrix}$$

$$D'_{s,44} = \frac{G_s \cdot t}{1.2} \quad D'_{s,55} = \frac{G_s}{B} \cdot t \cdot max\left[2 \cdot d; \frac{L}{1.2}\right]$$

Membrane components

$$d'_s = \begin{bmatrix} d'_{s,11} & d'_{s,12} & 0 \\ & d'_{s,22} & 0 \\ sym & & d'_{s,33} \end{bmatrix}$$

$$d'_{s,11} = \begin{cases} \text{if } d > 0: \ E_s \cdot t \cdot min\left[\frac{\left(\frac{t}{d}\right)^2}{3\frac{1-k \cdot D/l}{1-D/l} + 2 \cdot k}; 1\right] \\ \text{if } d = 0: \ E_s \cdot t \end{cases} \qquad d'_{s,22} = E_s \cdot t \cdot k \qquad d'_{s,12} = 0$$

$$d'_{s,33} = \frac{E_s \cdot t}{2 \cdot k \cdot (1+\nu_s)}$$

Mean thickness (for calculation of self weight)

$$H_s = k \cdot t$$

Position of gravity centre from bottom fibre (assumed for both directions)

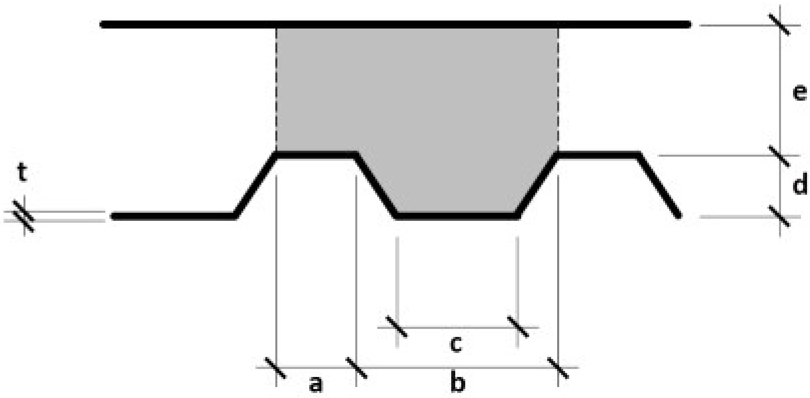
$$z_{G,s} = d \cdot \frac{a+l}{L}$$

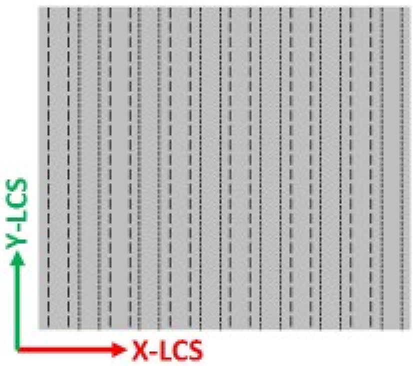
Auxiliary variables

$$B = a + b \qquad D = \frac{b-c}{2} \qquad l = \sqrt{D^2 + d^2}$$

$$L = a + c + 2 \cdot l \qquad k = \frac{L}{B} \qquad I = t \cdot \begin{bmatrix} a \cdot \left(d - z_{G,s}\right)^2 + c \cdot z_{G,s}{}^2 \\ +l \cdot \left(\left(\frac{d}{2} - z_{G,s}\right)^2 + \frac{d^2}{6}\right) \end{bmatrix}$$

Concrete deck





E_c, G_c, ν_c = Young's modulus, shear modulus and Poisson's ratio of concrete

a, b, c, d, e define the geometry of the concrete deck.

The formulas below give the components of the equivalent orthotropic properties of the concrete topping cast onto a generic profiled steel sheeting as shown in the above picture. Formulas adapted from Samanta & Mukopadhyay [1, 2].

Bending components

$$D'_c = \begin{bmatrix} D'_{c,11} & D'_{c,12} & 0 \\ & D'_{c,22} & 0 \\ sym & & D'_{c,33} \end{bmatrix}$$

$$D'_{c,11} = \frac{E_c \cdot B \cdot e^3}{12 \cdot \left(B - C + \left(\frac{e}{h}\right)^3 \cdot C\right)} \quad D'_{c,22} = \frac{E_c \cdot I}{B \cdot (1 - \nu_c^2)} \quad D'_{c,12} = \nu_c \cdot \sqrt{D'_{c,11} \cdot D'_{c,22}} \quad D'_{c,33} = \frac{E_c \cdot H_c^3}{24 \cdot (1 + \nu_c)}$$

$$G'_c = \begin{bmatrix} D'_{c,44} & 0 \\ 0 & D'_{c,55} \end{bmatrix}$$

$$D'_{c,44} = \frac{G_c \cdot H_c}{1.2} \quad D'_{c,55} = \frac{G_c \cdot e}{1.2}$$

Membrane components

$$d'_c = \begin{bmatrix} d'_{c,11} & d'_{c,12} & 0 \\ & d'_{c,22} & 0 \\ sym & & d'_{c,33} \end{bmatrix}$$

$$d'_{c,11} = d'_{c,22} = \frac{E_c \cdot H_c}{1-\nu_c^2} \quad d'_{c,12} = \nu_c \cdot d'_{c,11} \quad d'_{c,33} = G_c \cdot H_c$$

Mean thickness (for calculation of self weight)

$$H_c = \frac{e \cdot (B - C) + h \cdot C}{B}$$

Position of gravity centre from bottom fibre (assumed for both directions)

$$z_{G,c} = \frac{z_{G1} \cdot A_1 + z_{G2} \cdot A_2}{A_1 + A_2}$$

Auxiliary variables

$$h = d + e \qquad A_1 = B \cdot e \qquad A_2 = C \cdot d$$

$$B = a + b \qquad z_{G1} = d + \frac{e}{2} \qquad z_{G2} = \frac{d}{3} \cdot \frac{2 \cdot b + c}{b + c}$$

$$C = \frac{b + c}{2} \qquad I_1 = \frac{B \cdot e^3}{12} \qquad I_2 = \frac{d^3}{36} \cdot \frac{b^2 + c^2 + 4 \cdot b \cdot c}{b + c}$$

$$I = I_1 + I_2 + A_1 \cdot \left(z_{G1} - z_{G,c}\right)^2 + A_2 \cdot \left(z_{G2} - z_{G,c}\right)^2$$

Multi-layered orthotropy

The orthotropy sub-matrices are obtained from the formulas in the previous paragraphs. For each layer, there are:

$$\boldsymbol{D}'_i = \begin{bmatrix} D'_{i,11} & D'_{i,12} & 0 \\ & D'_{i,22} & 0 \\ sym & & D'_{i,33} \end{bmatrix} \quad \boldsymbol{G}'_i = \begin{bmatrix} D'_{i,44} & 0 \\ 0 & D'_{i,55} \end{bmatrix} \quad \boldsymbol{d}'_i = \begin{bmatrix} d'_{i,11} & d'_{i,12} & 0 \\ & d'_{i,22} & 0 \\ sym & & d'_{i,33} \end{bmatrix}$$

where i is the layer index; in the case of a composite deck, s (steel) or c (concrete).

Proper rotation of the matrices must be applied before combining the layers, in case the orientation of the steel sheeting does not correspond to the default coordinate system. The rotation matrices are

$$\boldsymbol{R}_3 = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix}$$

$$\boldsymbol{R}_2 = \begin{bmatrix} c & s \\ -s & c \end{bmatrix}$$

where

$$c = \cos\beta \qquad s = \sin\beta$$

β is the angle between the principal orthotropy direction Y' (corrugation of the steel sheeting) and the default (non rotated) local Y axis of the 2D member. In Scia Engineer, it is given by the LCS rotation angle in the 2D member properties.

The rotated orthotropy sub-matrices, for each layer, are

$$\boldsymbol{D}_i = \boldsymbol{R}_3^T \cdot \boldsymbol{D'}_i \cdot \boldsymbol{R}_3 \qquad \boldsymbol{G}_i = \boldsymbol{R}_2^T \cdot \boldsymbol{G'}_i \cdot \boldsymbol{R}_2 \qquad \boldsymbol{d}_i = \boldsymbol{R}_3^T \cdot \boldsymbol{d'}_i \cdot \boldsymbol{R}_3$$

Finally, the layers must be combined and the eccentricity terms added in the matrix. The final orthotropy matrix has the form

$$\boldsymbol{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & & & D_{16} & D_{17} & D_{18} \\ & D_{22} & D_{23} & & & sym & D_{27} & D_{28} \\ & & D_{33} & & & sym & sym & D_{38} \\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & & & & D_{66} & D_{67} & D_{68} \\ & sym & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix}$$

Plate behaviour components

$$D_{jk} = \sum_{i=1}^{n} D_{i,jk} + z_i^2 \cdot d_{i,jk} \qquad j,k = 1..3$$

$$D_{jk} = \sum_{i=1}^{n} D_{i,jk} \qquad j,k = 4..5$$

Membrane behaviour components

$$D_{66} = \sum_{i=1}^{n} d_{i,11} \quad D_{67} = \sum_{i=1}^{n} d_{i,12} \quad D_{68} = \sum_{i=1}^{n} d_{i,13}$$

$$D_{77} = \sum_{i=1}^{n} d_{i,22} \quad D_{78} = \sum_{i=1}^{n} d_{i,23}$$

$$D_{88} = \sum_{i=1}^{n} d_{i,33}$$

Layer eccentricities

$$D_{16} = \sum_{i=1}^{n} z_i \cdot d_{i,11} \quad D_{17} = \sum_{i=1}^{n} z_i \cdot d_{i,12} \quad D_{18} = \sum_{i=1}^{n} z_i \cdot d_{i,13}$$

$$D_{27} = \sum_{i=1}^{n} z_i \cdot d_{i,22} \quad D_{28} = \sum_{i=1}^{n} z_i \cdot d_{i,23}$$

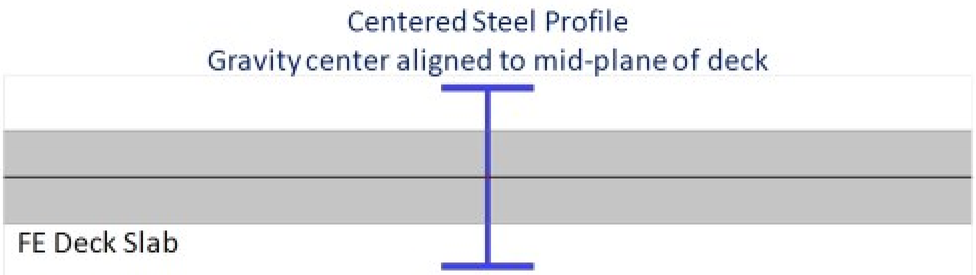
$$D_{38} = \sum_{i=1}^{n} z_i \cdot d_{i,33}$$

where z_i is the position of the gravity center of the i-th layer (profiled sheeting or concrete).

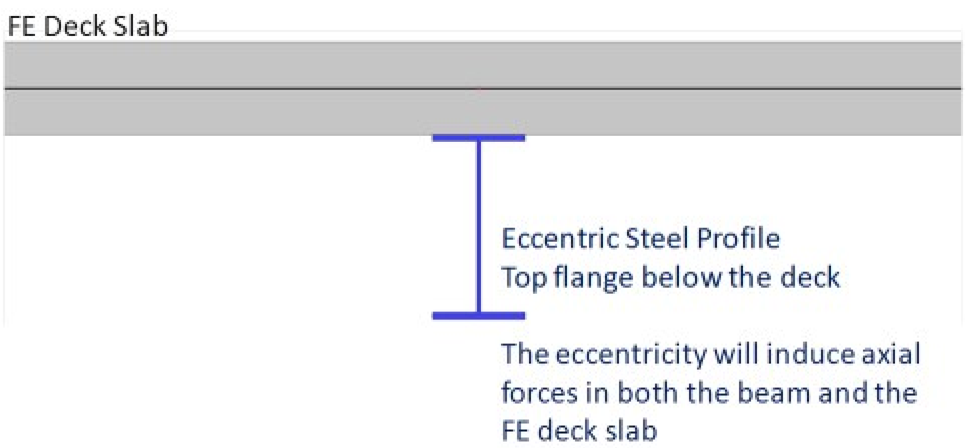
Composite beam

In the context of composite structures, there are currently 3 possible types of behaviour for plate ribs in Scia Engineer:

without composite action
this is meant for beams that are connected to the deck without shear connectors, i.e. the deck is just lying on the beam. There is actually no composite action in that setup. This is modelized by a plate rib without eccentricity

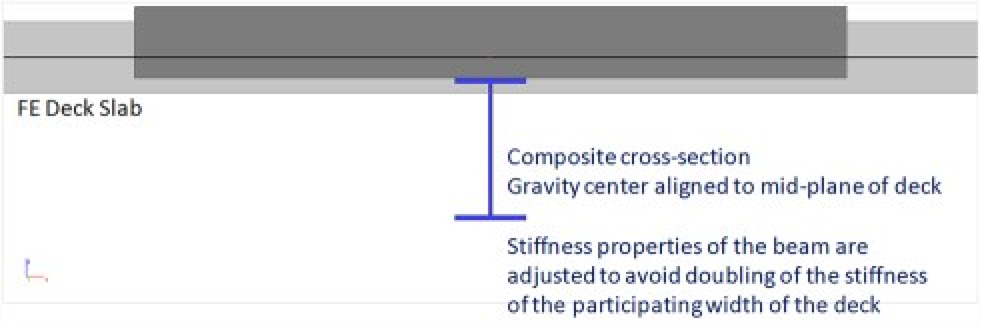


with advanced composite action
in this case, a perfect shear connection is assumed between the beam and the deck. The plate rib is modelized with its real eccentricity. In this configuration, an axial force will appear in the beam and membrane forces will appear in the deck. The diffusion of the membrane forces in the deck will be automatically calculated by the FE modelization of the deck. The internal forces for the composite checks will be obtained by integrating the obtained stresses in both the steel beam and the deck (within the participating width of the deck).



with standard composite action
in this case, by default, a perfect shear connection is assumed between the beam and the deck. The plate rib is modelized by a beam without eccentricity. In order to take the composite action into account, the stiffness of the

beam is adjusted to take into account the effect of the eccentricity and of the participating width. The adjustments of the cross-section properties are detailed below. In the same way as for the so-called advanced model (see above), the internal forces for the composite checks will be obtained by integrating the calculated stresses in both the steel beam and the deck. However, as there is no eccentricity between the plate rib and the plate, no axial nor membrane forces will appear in the model due to the composite action.



Adjusted properties of beams "with standard composite action"

In this case, the geometric properties of the cross-section of the beam will be adjusted to take into account the effects of the eccentricity and of the participating width of the deck.

Only some properties need adjusting, most of them remain unchanged because they are affected neither by the eccentricity nor by the participating width of the deck. The actual participation of the deck itself is already taken into account by the FE modelization of the deck and must therefore not be taken into account again in the properties of the beam.

A	area – from the steel beam
A_y	shear area y – from the steel beam
A_z	shear area z – from the steel beam
I_x	torsional inertia – **adjusted**, see below
I_y	bending inertia y-y – **adjusted**, see below
I_z	bending inertia z-z – from the steel beam

For **adjusted** properties, the following formulas will be used:

$$b_d = b_{d,left} + b_{d,right}$$

$$A_{eq,c} = \frac{d''_{c,11}}{E_b} \cdot b_d \qquad A_{eq,s} = \frac{d''_{s,11}}{E_b} \cdot b_d \qquad A_{y,eq,c} = \frac{d''_{c,33}}{G_b} \cdot b_d \qquad A_{y,eq,s} = \frac{d''_{s,33}}{G_b} \cdot b_d$$

$$z_G = \frac{z_b \cdot A_b + z_{G,c} \cdot A_{eq,c} + z_{G,s} \cdot A_{eq,s}}{A_b + A_{eq,c} + A_{eq,s}}$$

$$I_{x,adj} = I_{x,b} + A_{y,b} \cdot (z_b - z_G)^2 + A_{y,eq,c} \cdot \left(z_{G,c} - z_G\right)^2 + A_{y,eq,s} \cdot \left(z_{G,s} - z_G\right)^2$$

$$I_{y,adj} = I_{y,b} + A_b \cdot (z_b - z_G)^2 + A_{eq,c} \cdot \left(z_{G,c} - z_G\right)^2 + A_{eq,s} \cdot \left(z_{G,s} - z_G\right)^2$$

where the following variables are obtained from the steel beam cross-section properties:

$I_{x,b}$	torsional inertia of the steel beam
$A_{y,b}$	shear area of the steel beam in the y direction
$I_{y,b}$	bending inertia y-y of the steel beam
A_b	area of the steel beam
E_b	elasticity modulus of the steel beam
G_b	shear modulus of the steel beam
b_d	total effective width of the composite beam

The equivalent area values $A_{eq,c}$, $A_{eq,s}$, $A_{y,eq,c}$ and $A_{y,eq,s}$ are obtained from the orthotropic properties of the deck:

$d''_{c,11}$	axial membrane stiffness component of the orthotropy matrix of the concrete part of the deck in the direction of the beam axis; in case of a metal deck, use $d''_{c,11} = 0$
$d''_{c,33}$	shear membrane stiffness component of the orthotropy matrix of the concrete part of the deck in the direction of the beam axis; in case of a metal deck, use $d''_{c,33} = 0$
$d''_{s,11}$	axial membrane stiffness component of the orthotropy matrix of the profiled steel sheeting in the direction of the beam axis
$d''_{s,33}$	shear membrane stiffness component of the orthotropy matrix of the profiled steel sheeting in the direction of the beam axis

The components of the input matrices $\boldsymbol{d}'_i$ can be found in the previous chapters for concrete ($\boldsymbol{d}'_c$) and for the profiled steel sheeting ($\boldsymbol{d}'_s$).

The calculation of the orthotropy components is obtained by rotation of the orthotropy matrix of the deck parts:

$$\boldsymbol{d''}_i = \begin{bmatrix} d''_{i,11} & d''_{i,12} & d''_{i,13} \\ & d''_{i,22} & d''_{i,23} \\ sym & & d''_{i,33} \end{bmatrix} = \boldsymbol{R}_3^T \cdot \boldsymbol{d'}_i \cdot \boldsymbol{R}_3 = \boldsymbol{R}_3^T \cdot \begin{bmatrix} d'_{i,11} & d'_{i,12} & 0 \\ & d'_{i,22} & 0 \\ sym & & d'_{i,33} \end{bmatrix} \cdot \boldsymbol{R}_3$$

$$\boldsymbol{R}_3 = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix}$$

$$c = \cos\theta$$

$$s = \sin\theta$$

where θ is the angle between the LCS Y-axis of the deck and that of the beam. Please note, that this is the same rotation matrix $\boldsymbol{R}_3$ as used in the paragraph related to multi-layered orthotropy, just with a different angle.

For the required components in the current context, this leads to:

$$d''_{i,11} = c^4 \cdot d'_{i,11} + 2 \cdot c^2 \cdot s^2 \cdot d'_{i,12} + s^4 \cdot d'_{i,22} + 4 \cdot c^2 \cdot s^2 \cdot d'_{i,33}$$

$$d''_{i,33} = c^2 \cdot s^2 \cdot d'_{i,11} - 2 \cdot c^2 \cdot s^2 \cdot d'_{i,12} + c^2 \cdot s^2 \cdot d'_{i,22} + (c^2 - s^2)^2 \cdot d'_{i,33}$$

In a similar way, the bending stiffness of the slab must be calculated, in case of partial composite connection (see next paragraph):

$$D''_{i,11} = c^4 \cdot D'_{i,11} + 2 \cdot c^2 \cdot s^2 \cdot D'_{i,12} + s^4 \cdot D'_{i,22} + 4 \cdot c^2 \cdot s^2 \cdot D'_{i,33}$$

1) In theory, for the calculation of the torsional inertia ($I_{x,adj}$), the coordinate of the shear center should be used instead of that of the gravity center (z_G), but the inaccuracy is most probably neglectable in this case.

2) In theory A_z should be adjusted too, as the distribution of shear stresses in the composite section differs from that in individual parts. However, as the shear connection between steel and concrete is ensured only locally by studs, the reality is somewhere between those two limit cases and that simplification is acceptable.

Composite action with partial connection

The previous paragraph defines values for the case of a full composite connection. It is however common – and often economical – to use partial composite connection. Partial composite connection is taken into account according to the following simplified method, using a reduced bending stiffness for the composite beam. The following adjustments must be done:

$$I_{eq,c} = \frac{D''_{c,11}}{E_b} \cdot b_d \qquad I_{eq,s} = \frac{D''_{s,11}}{E_b} \cdot b_d$$

$$I_{y,adj,k} = \max \begin{cases} I_{y,b} - I_{eq,c} - I_{eq,s} + \sqrt{K} \cdot \left(I_{y,adj} - I_{y,b} + I_{eq,c} + I_{eq,s} \right) \\ I_{y,b} \end{cases}$$

$I_{y,adj,k}$ adjusted inertia of the composite beam, to be used in case of partial composite connection instead of $I_{y,adj}$

Definition of $I_{y,b}$, $I_{y,adj}$, E_b , b_d , $D''_{c,11}$ and $D''_{s,11}$: see previous paragraph.

K is a value between 0 and 1 that defines the degree of composite connection; 0 = no connection, 1 = full connection.

Construction stages for composite analysis

Construction stages must be taken into account in the analysis of composite structures mostly for two reasons:

- the profiled steel sheeting is used as a formwork for the concrete topping, hence it has to carry alone the weight of concrete
- the behaviours of steel and concrete are fundamentally different: stiffness, creep

In the general case, construction stages are taken into account in a simplified way, by calculating each load case in the stage corresponding to its assumptions. The results (displacements, internal forces…) can then be combined in load case combinations.

Construction stage
in this stage, only the steel of the composite decks is enabled. Concrete has no stiffness and its self weight is hence carried by the steel structure (profiled steel sheeting and steel beams). By default, only the self weight load case is assigned to this stage.

Final stage, long term
in this stage, the composite decks are enabled. The concrete stiffness is reduced to take into account the effect of creep under long term loads. By default, all permanent load cases, except self weight, are assigned to this stage.

Final stage, short term
in this stage, the composite decks are enabled. The nominal concrete stiffness is used, for use under short term loading. By default, all variable load cases are assigned to this stage.

In the standard composite analysis model, **3 construction stages are defined for the entire structure**. There is no such thing as stages for casting of concrete or staged building of the steel structure. It is however planned that this will be supported in a later version.

Creep

Creep is taken into account using a reduced value of the elasticity modulus for concrete in the final stage, long term. The creep coefficient is defined in the composite setup for the entire structure and applied to all composite decks.

During the calculation of the orthotropy matrices, adjusted values of E_c and G_c are used in each stage for concrete:

$$E_c = k_E \cdot E_{c0}$$

$$G_c = k_E \cdot G_{c0}$$

where E_{c0} (G_{c0}) is the E-modulus (G-modulus) of concrete from the material library.

$$k_E = \begin{cases} 0 & \textit{for construction stage} \\ 1/(1+\varphi) & \textit{for final stage, long term} \\ 1 & \textit{for final stage, short term} \end{cases}$$

where φ is the creep factor defined in composite setup.

Creep can be optionally disabled, in which case load cases from long term stage are moved to short term stage during the analysis.

Propping

It is assumed by default that the weight of concrete is carried solely by the steel structure. It can be optionally assumed, that the steel structure is entirely propped during the casting of concrete. The propping is then removed after the concrete has hardened.

This can be taken into account by moving all load cases from the construction stage to the final stage (long term or short term, depending on creep settings).

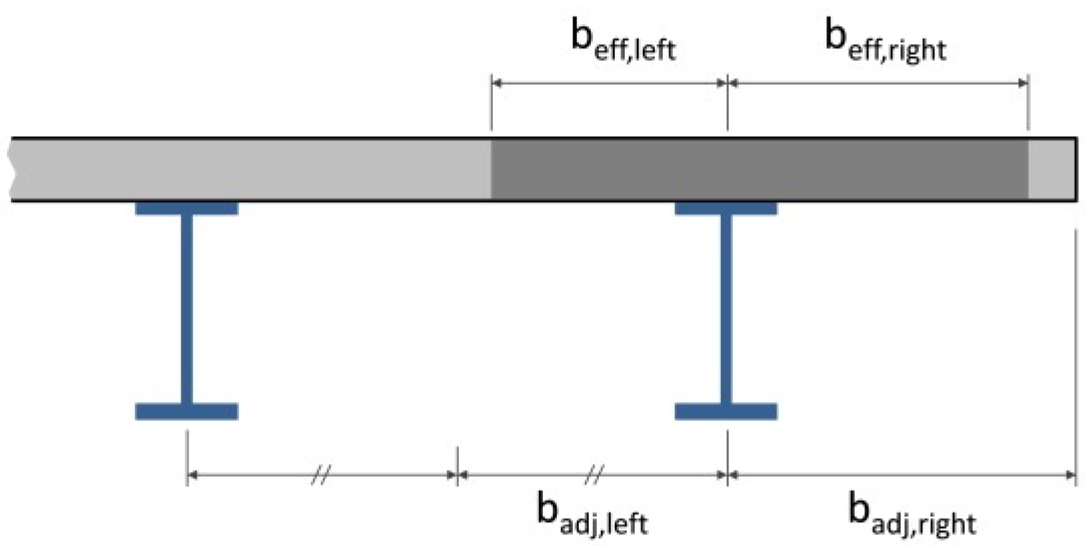
Automatic calculation of effective width

The effective width of the plate ribs can be defined manually by the user or it can be calculated automatically by the Scia Engineer.

In the case of automatic calculation, the effective width is determined according to the following geometric rules:

$$b_{eff,left} = \frac{L}{a} \leq b_{adj,left}$$

$$b_{eff,right} = \frac{L}{a} \leq b_{adj,right}$$



$b_{eff,left}$, $b_{eff,right}$ are the left and right hand side effective width of the rib

L is the span length of the considered beam; in Scia Engineer,

α is a configurable constant; it is taken = 8.0 by default, which corresponds to the specifications of the ASCI (IBC) code for the effective width in the analysis model.

$b_{adj,left}$, $b_{adj,right}$ are the maximum values of effective width on the left and right hand side due to adjacent entities. The effective width on a side may not exceed:

- The distance to an edge of the slab
- Half of the distance to the adjacent beam

Only entities that fulfil all the following conditions are taken into account. Those are plate ribs, outer edges or opening edges that are:

- parallel or nearly parallel to the considered beam; a tolerance angle is configurable (default=10°)
- linked to or contained by a plate that is
    - located in the same plane as the slab that contains the considered beam
    - connected by at least one edge to the containing slab of the considered beam (a single point connection is not sufficient)

The distance to the adjacent entity is calculated at mid-length of each member. The effective width is calculated as a uniform value per member. That assumption corresponds to the mentioned design codes.

A 1D member may not cover more than one span for the automatic calculation of effective width. If it does, the calculation of effective width will fail and the rib will switch back to manual input. Continuous beams should therefore be modelled as multiple 1D members, rigidly connected to each other.

It is however possible, in the case of nearly parallel beams, to take into account the variable spacing of the beams by splitting each span into several members.

The current implementation supports only the method described here. For special, unsupported use cases, it is recommended to use manual input of the effective width.

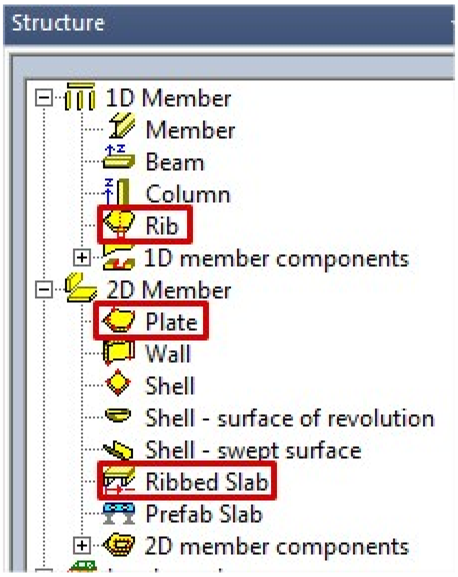
Composite Analysis Model in Scia Engineer

Principles

The Composite Analysis Model (CAM) has been kept as simple as possible. It uses standard modelization functionality of Scia Engineer

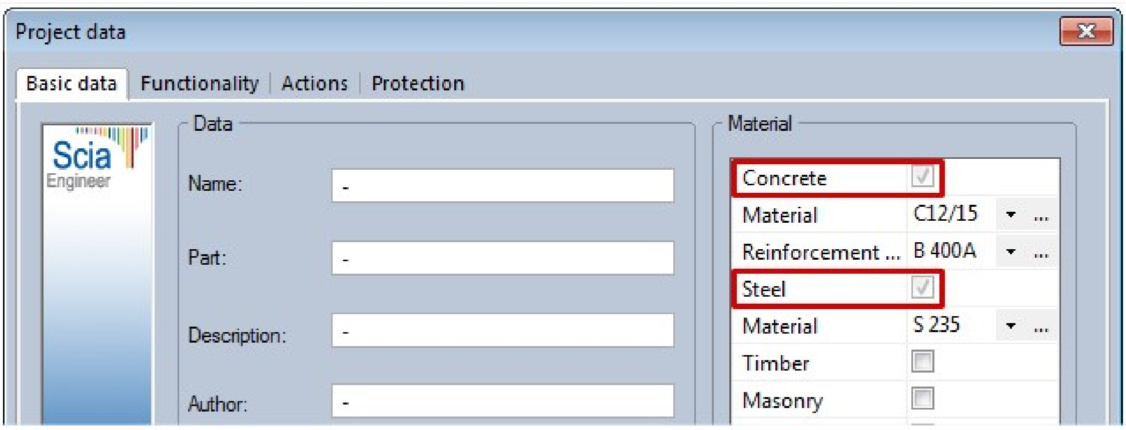
Fundamentally, a composite deck with steel beams is modelized using a standard plate with plate ribs. Only a limited number of properties needs to be configured, in the plate and in the beam properties, to make those structural parts behave as composite.

Using the standard menu items in Scia Engineer, the composite system may be defined as a plate, with ribs added afterwards, or directly using a ribbed slab.



Project settings

In the project settings, simply enabling both concrete and steel material libraries will enable the CAM.



The CAM functionality is code independent and thus may be used with any design code. However, the related composite checks are code dependent and are currently available for the EN1994 and AISC 360-10 codes.

Note: the *composite* feature located in the functionality tab is related to another (dated) composite functionality, which is incompatible with the CAM. **Enabling that functionality will disable the CAM.**

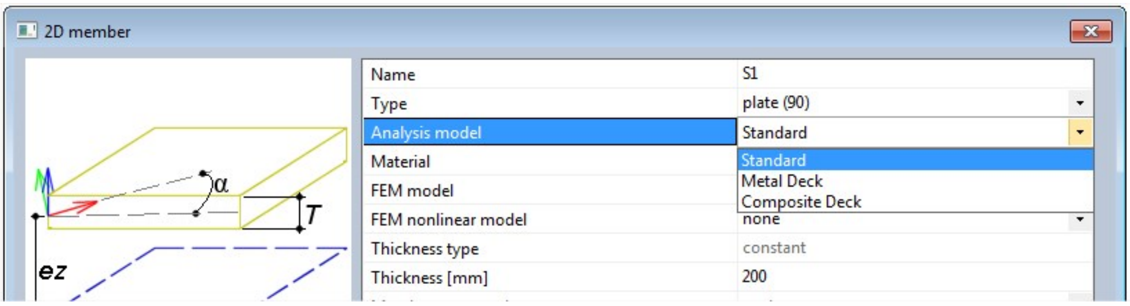
The composite checks linked to that functionality are intended to be eventually entirely replaced by the CAM and its related composite checks.

Definition of the deck

A deck (composite deck or metal deck) is essentially a plate in Scia Engineer. It may be input as any 2D member that accepts ribs. That includes plates and straight walls.

To define a composite deck, the user would usually define a standard plate, or a ribbed plate in the case that he would like to assign the ribs in the very same modelling operation.

A composite deck is created from a standard plate via the property *Analysis model*. This property defines whether the 2D member is a standard plate, a composite deck, or a metal deck.

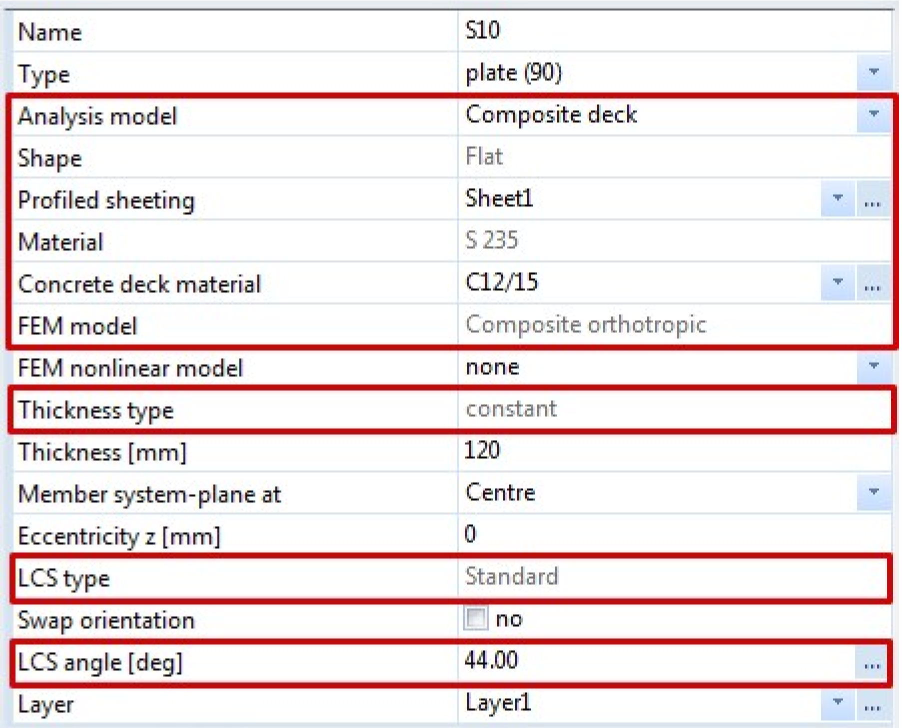


A composite deck is made of a profiled steel sheeting with a concrete topping.

A metal deck has only the profiled steel sheeting (usually intended for light-weight roofs).

A standard plate may be used in a composite analysis model, together with some composite deck or metal deck members, but it will not have any of the composite analysis features.

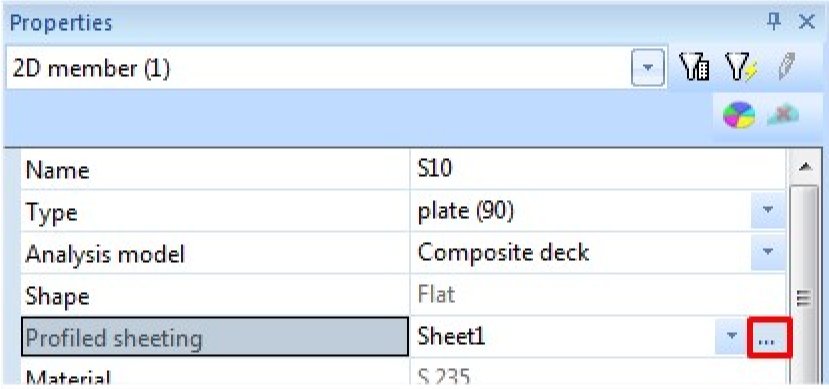
The specific properties for a composite/metal deck are:

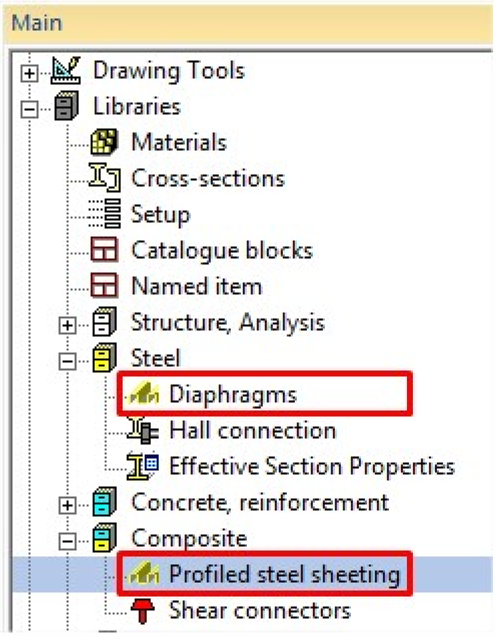


Analysis model	- standard: standard plate, with regular properties; a member with this setting will not be affected by the CAM - metal deck: profiled steel sheeting alone (1 layer) - composite deck: 2-layers member, profiled steel sheeting + concrete topping
Profiled Sheeting	selection of a profiled steel sheeting from the library (see below)
Material	material of the selected profiled steel sheeting; this property is read-only and its value is taken from the profiled sheeting library
Concrete deck material	material of the concrete topping (only for composite deck)
FEM model	read-only; composite orthotropic ("Theoretical background")
Thickness type	read-only; constant; this cannot be edited for composite/ metal decks
Thickness	for metal deck: read-only; height of the selected profiled sheeting for composite: editable; total height of the composite deck; may not be smaller than the height of the selected profiled sheeting
LCS Type	read-only; standard; only standard LCS type allowed for composite/metal decks
LCS Angle	same meaning as usual, but additionally defines the orientation of the corrugation of the profiled sheeting; the corrugation of the sheeting is always directed along the LCS Y axis

Profiled sheeting library

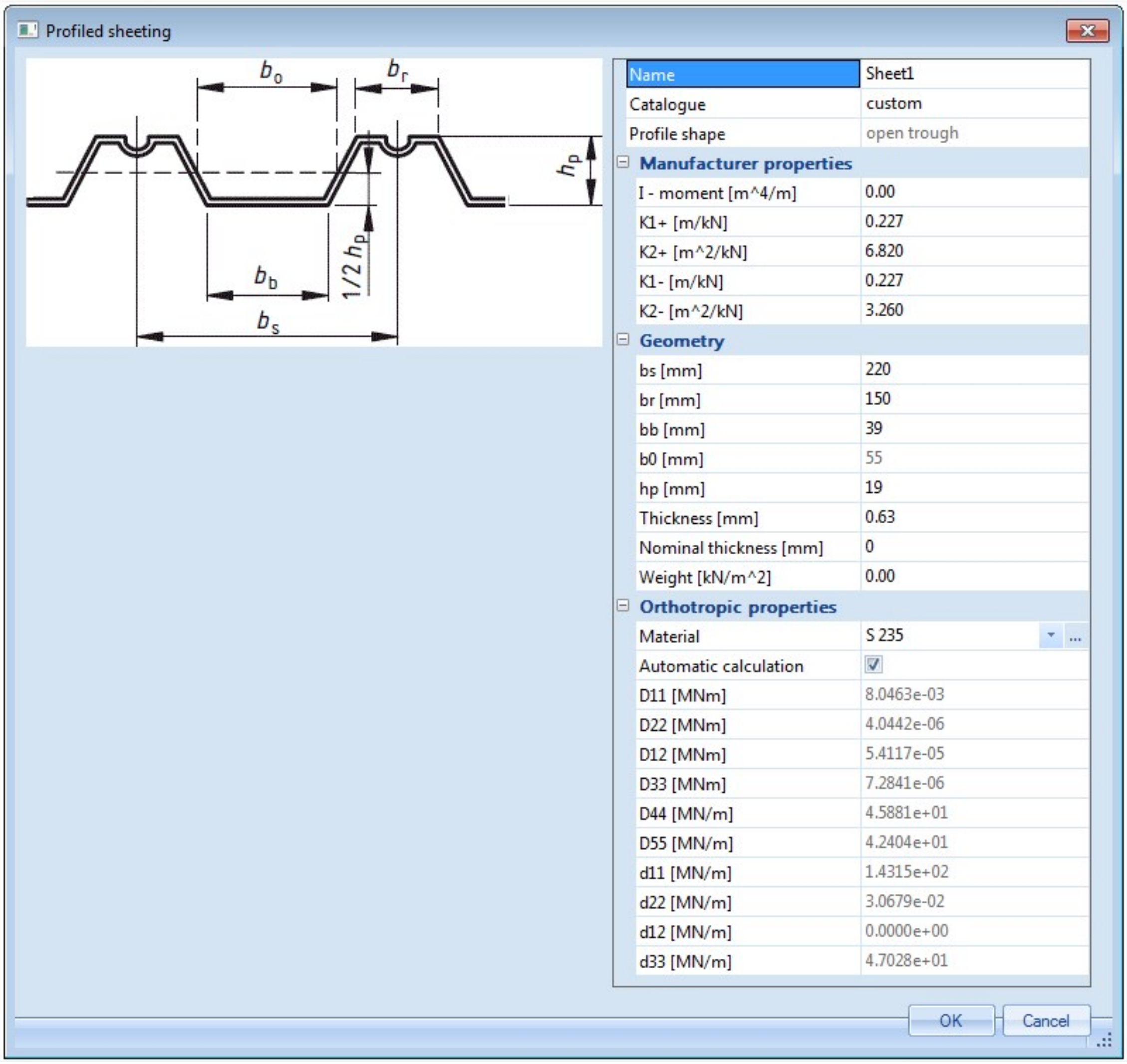
The profiled sheeting library can be accessed from the properties of a composite or metal deck or directly in the Libraries composite sub-tree. It is actually the same library that is used for diaphragm constraints in steel code checks for LTB stabilization.



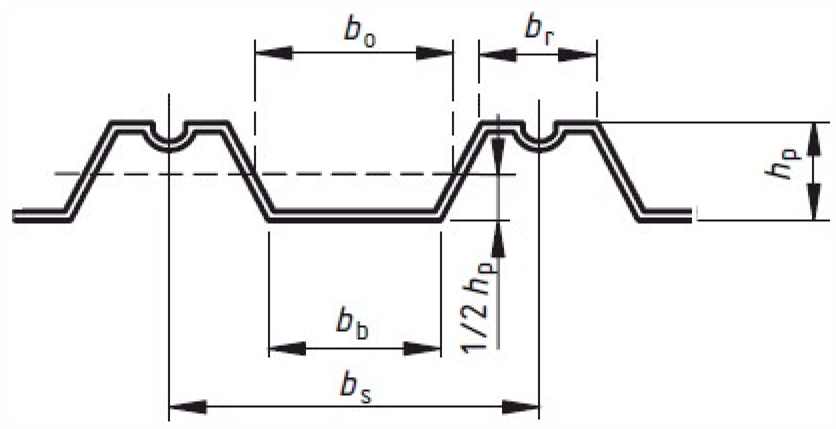


The properties of the profiled steel sheeting are divided in 4 groups.

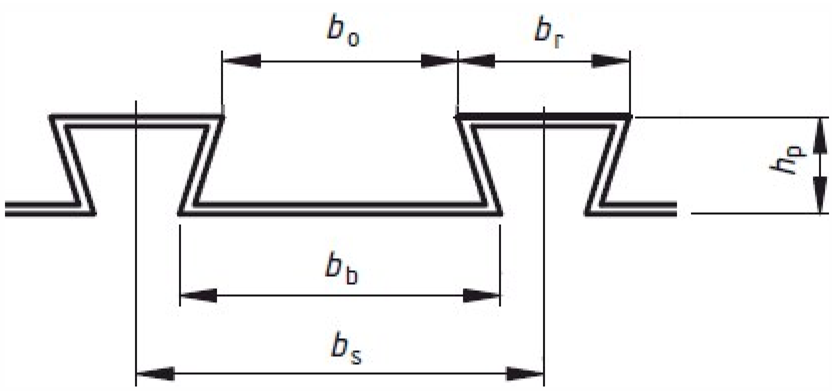
- some general properties (name, etc...)
- **Manufacturer properties**, which are used only in the case of diaphragm constraints in the steel code checks; those will not be described here
- **Geometry**, which defines the main dimensions of the profiled sheeting
- **Orthotropic properties**, which define the mechanical properties of the sheeting for the CAM



Name	name of the profiled sheeting library item
Catalogue	user defined field; may be used to define lists of profiled sheeting and filter the available profiled sheetings in the library; it can be, for instance, the manufacturer's name, some type of sheeting, etc...
Profile shape	type of profile shape; it is automatically determined from the geometry of the profile; it can be **open trough**



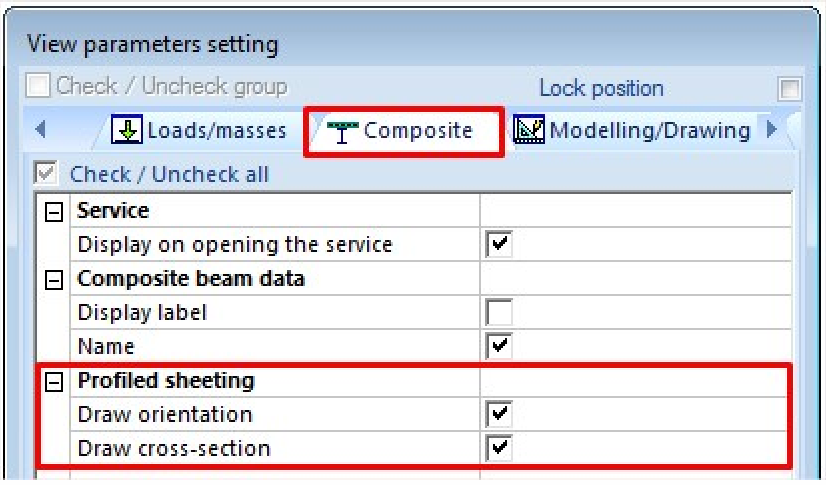
or **re-entrant trough**



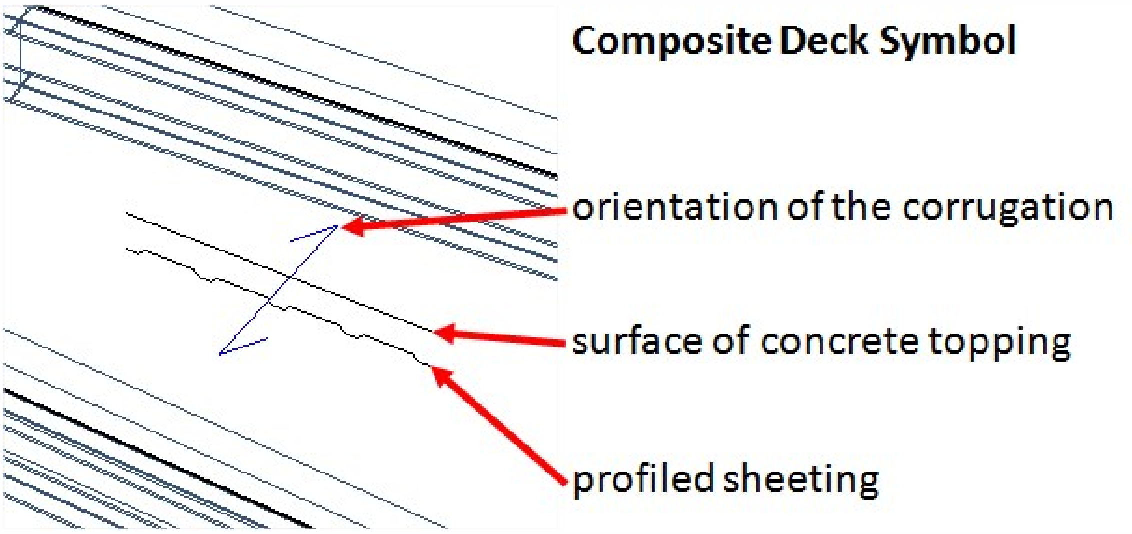
b_s, b_r, b_b, b_0, h_p	dimensions of the profiled sheeting, as shown in the pictures above; b_0 is read-only and calculated from the other dimensions
Thickness	thickness of the sheeting
Nominal thickness	nominal thickness; not used for the CAM
Weight	surface weight of the sheeting; not used for the CAM; the self-weight of the sheeting is actually calculated from it geometry
Material	steel material of the sheeting
Automatic calculation	when ON, the orthotropic properties of the profiled sheeting are calculated from the geometry ("Theoretical background"); when OFF, the components of the orthotropy matrix may be defined manually
D11 ... d33	components of the orthotropy matrix of the profiled sheeting

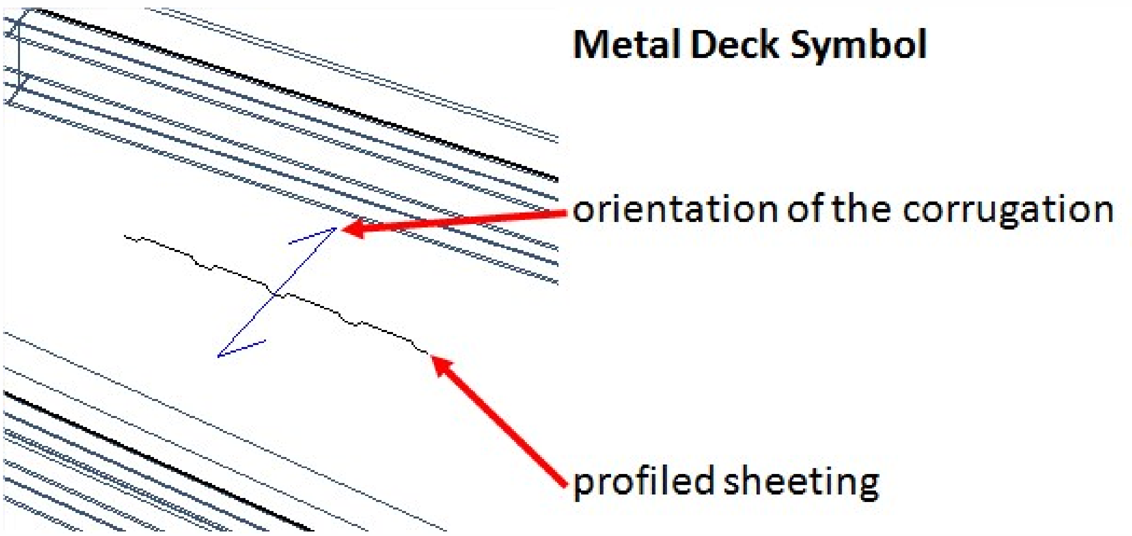
Display settings

For composite and metal decks, it is possible to display the shape and orientation of the profiled sheeting directly on the model. The corresponding view settings are located in the Composite tab of the general view settings:



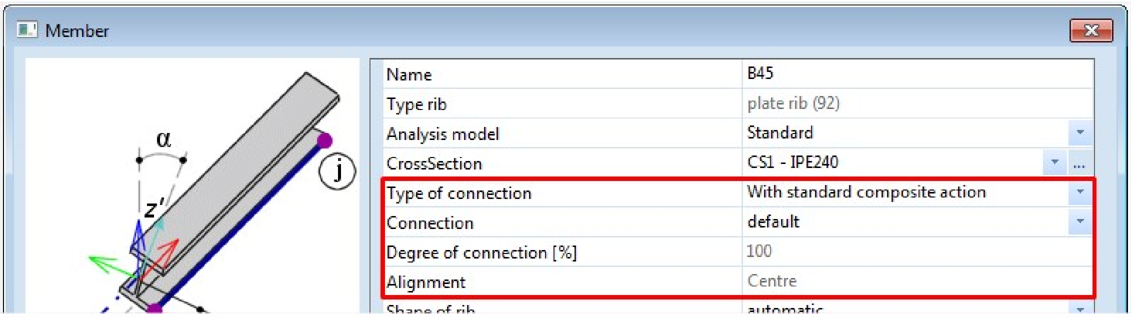
Those settings allow to display on the model symbols that show the main supporting direction (i.e. the direction of the profiled sheeting), the shape of the profiled sheeting and the thickness of the concrete topping.





Definition of the beams

Composite beams are to be defined as standard plate ribs. They may also be created using a ribbed plate 2D member, which includes the definition of the ribs. A few properties will then specify the beam as composite.



Type of connection	that property is visible only when the CAM is enabled. There are 4 possible values: **with standard composite action** applies a composite cross-section to the plate rib without eccentricity on the 1D member; this has the advantage that it does not create parasite axial forces in the model; for details "Theoretical background" **with advanced composite action** uses a standard eccentric plate rib, without adjustment of the cross-section; the composite effect is taken into account directly through the real eccentricity of the 1D member in the finite element model; this will, however, generate axial forces in both the beam and the deck **without composite action** assumes that there is no longitudinal shear connection between the beam and the deck; it is taken into account by means of a plate rib without eccentricity using the selected beam cross-section without adjustment **user-defined eccentricity** corresponds to a standard plate rib; it will not allow using composite functionalities on that beam
Connection	that property is visible only for a type of connection with standard composite action; it allows to define the degree of shear connection between the beam and the deck. The possible values are: **default**: the actual connection is defined in the composite setup **full**: this assumes a perfect shear connection between the beam and the deck **partial**: this assumes a partial connection between the beam and the deck
Degree of connection	that property is visible only for a type of connection *with standard composite action*; it defines the actual degree of connection between the beam and the deck. Only in case of a partial connection, the degree of connection may be defined here. Otherwise, this property is read-only. The default value of the degree of connection is defined in the composite setup.
Alignment	For all composite types of connection, the alignment is read-only and configured automatically according to the selected type of connection. It is *centered* for *standard* and *without composite action*; it is *bottom* for *advanced composite action*. For *user-defined eccentricity*, the alignment is defined by the user.

Effective Width

Effective width is a standard property of any plate rib. It is used for several purposes in a usual analysis model:

- *internal forces* effective width is used for integrating stresses from the plate during output of internal forces in the rib (in *1D internal forces* result service, when ticking the option *rib/integration strip*)
- *check* effective width defines the width of the plate part that will be taken into account in the checks to calculate the resistance of the cross-section (code-dependent)

It is important to note, that none of the above affects the stiffness of the system. Modifying the effective width does not modify the properties of the cross-sections in the analysis model.

There is however a difference when using the CAM, which uses the effective width in a third way:

- *internal forces* effective width is taken into account when calculating the stiffness of the composite cross-section of a beam defined *with standard composite action*; for details, see "Theoretical background"
- in case of beam *without composite action*, all effective width properties are set to zero

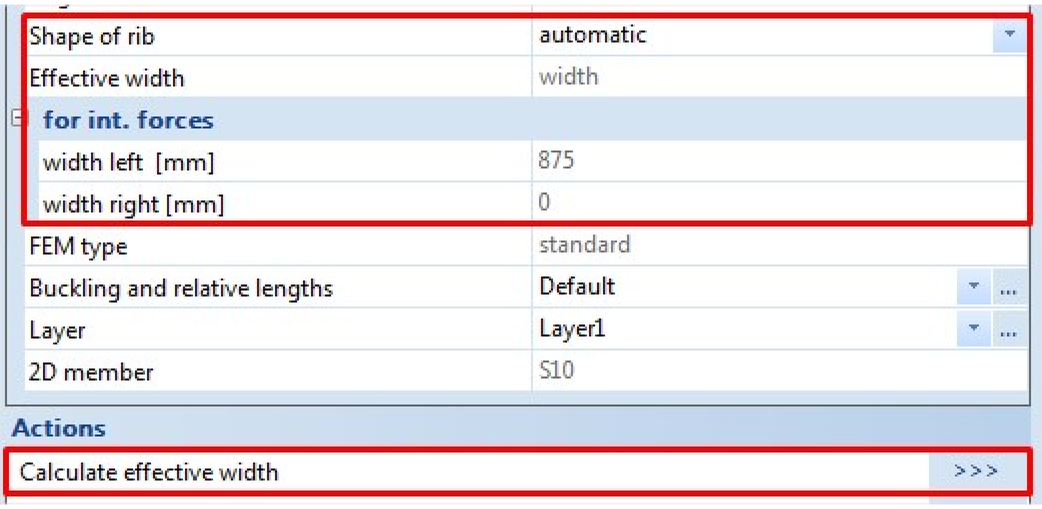
Important: unlike in all other cases, in a beam *with standard composite action*, the effective width affects the stiffness of the beam

Automatic calculation of the effective width

The effective width can be defined manually or calculated automatically.

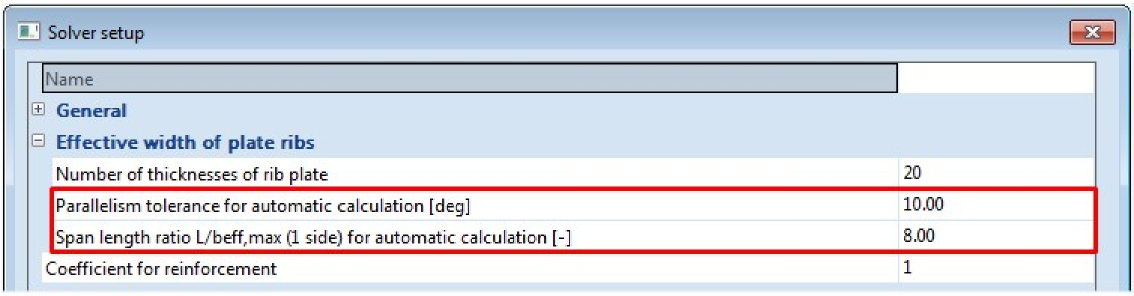
When the property *shape of rib* is set to *T-symmetric*, *slab left*, *slab right* or *asymmetric*, effective values are to be defined manually.

When *shape of rib* is set to *automatic*, the effective width is calculated automatically from the geometry of the structure. The calculation of the effective width is performed for the entire structure when running the analysis. It can also be triggered by clicking the action button *calculate effective width* at the bottom of the property sheet of the considered beam.



The elements that influence the value of the effective width are

- the span length of the beam
- adjacent entities (beams, slab edge, openings)
- effective width settings in the solver setup



Parallelism tolerance	maximum angle between a beam and its considered adjacent entity, for them to be considered parallel
Span length ratio L/b$_{eff,max}$ (1 side)	ratio of the span length used for the calculation of the effective width

> Those settings are located in the solver setup, because they influence the analysis model. They are not in the composite setup, because they are not directly related to the CAM and may be used for any plate rib, not only composite beams.

In the current implementation, the effective width, is calculated separately on each side (left and right) of the rib as the smallest of the three following values:

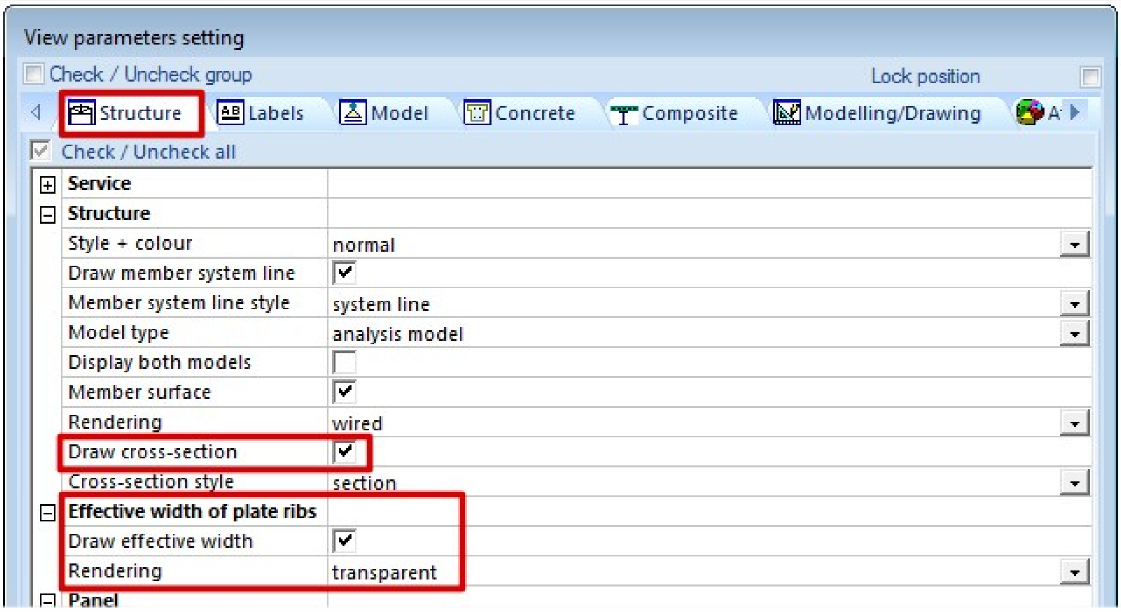
- ratio of the span length L/x, where x is defined in the solver setup
  The span length is determined from the buckling system of the beam and can hence be longer than the beam. It can also be influenced by modifying the buckling boundary conditions, using the *buckling and relative lengths* property of the beam or the *graphical input of system length* action button.
- half-distance to the closest parallel (or nearly parallel) beam
  The distance is measured in the middle of the considered beam.
- distance to the closest parallel (or nearly parallel) free edge
  The distance is measured in the middle of the considered beam.

> Scia Engineer stores the calculated effective width values for both the *internal forces* and *check* effective width. However, the checks might use or not use those stored values when determining the resistance of the cross-sections. Please refer to the detailed description of the considered checks for more information about this topic.

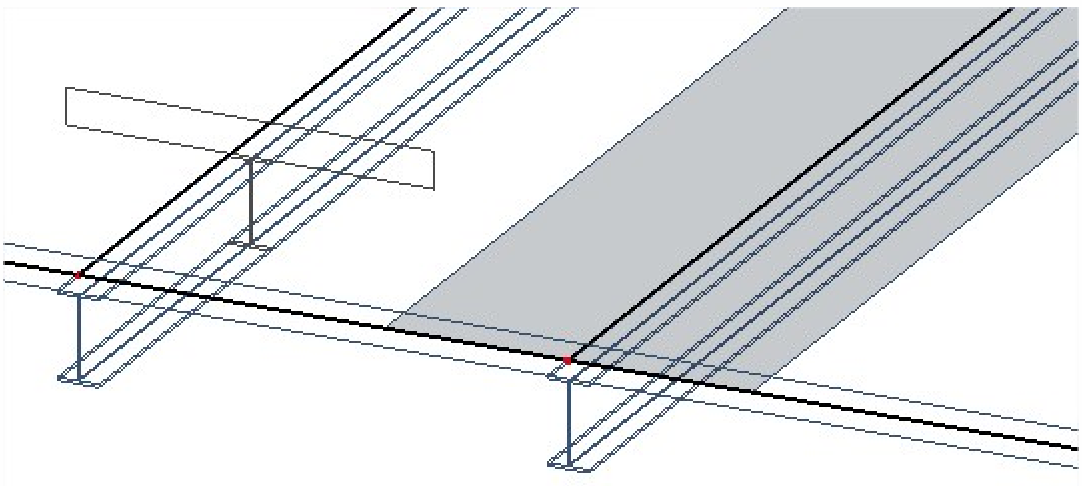
For more details "Theoretical background".

Display settings for effective width

The effective width for internal forces can be graphically displayed on the analysis model, in two different ways.



- The display setting **Draw cross-section** displays one cross-section on each beam, showing the actual cross-section of the rib and the portion of the slab corresponding to the effective width (see picture, left)
- The display setting ***Effective of plate ribs > Draw effective width*** shows the effective width in the plane of the slab along the entire beam (see picture, right); several rendering modes are available



Construction stages for the composite analysis model

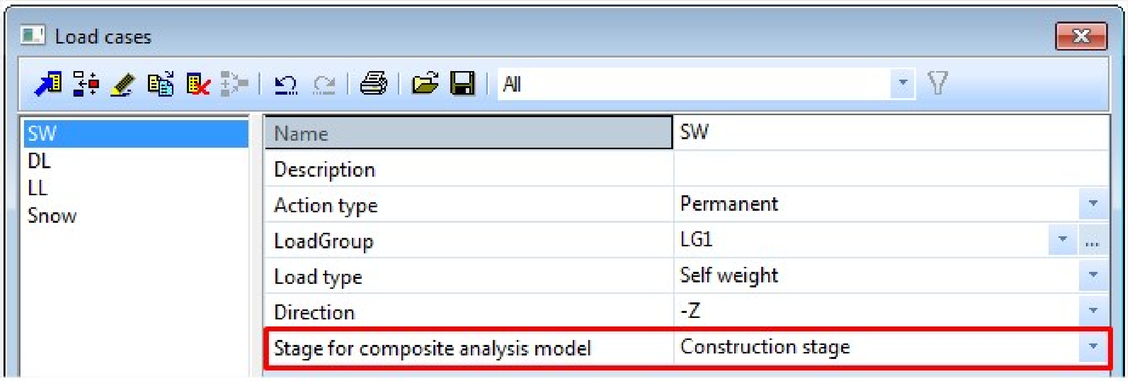
The construction stages of a composite structure are taken into account for the entire structure at once. There are three stages in the CAM:

- construction stage: only the steel structure carries the loads; the concrete is soft, its self-weight acts on the structure, but it has no stiffness
- final stage, long term: the composite structure carries the applied long term loads; the effect of creep is taken into account via a reduced apparent stiffness of the concrete
- final stage, short term: the composite structure carries the applied short term loads; there is no creep effect and the full stiffness of concrete is used

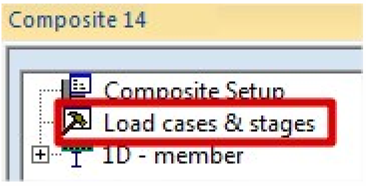
To account for those stages, only the stiffness of concrete is modified from one stage to the next one. Each load case is assigned to one of those three stages. By default, when creating the load cases:

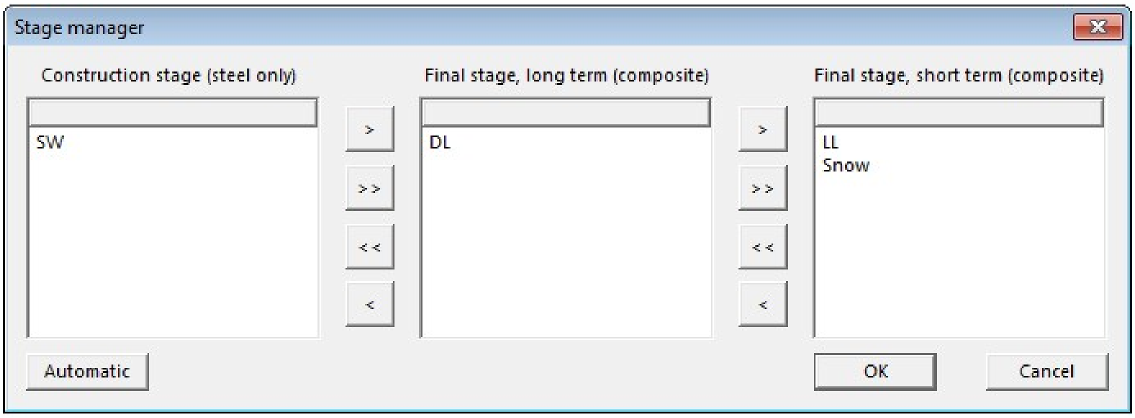
- self-weight load case is assigned to the construction stage
- permanent load cases are assigned to the final stage, long term
- variable load cases are assigned to the final stage, short term

The allocation of load cases to stages can be seen and edited in the load case properties:



It is also possible to see an overview of the load cases allocation via the *Load cases & stages* manager in the composite service:

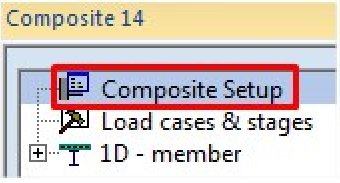


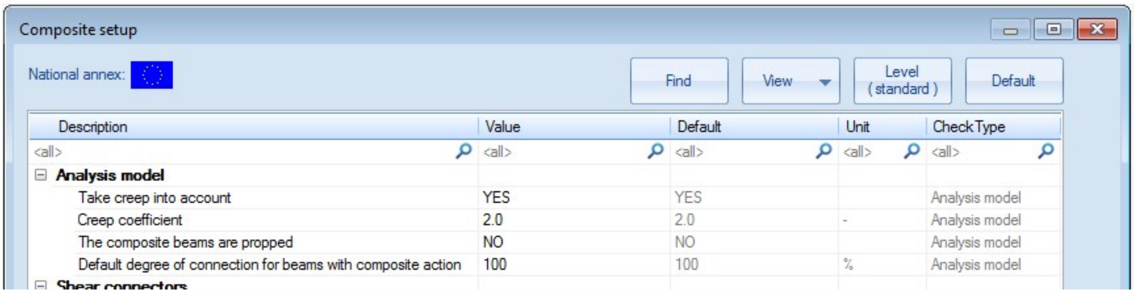


Load cases can be moved from one stage to the other. The *Automatic* button resets all assignments to default, as described above.

Composite setup

The composite setup can be accessed from the composite service tree. It contains a number of settings that are related to the CAM and to the composite checks. Only CAM related settings are detailed here. Those settings all affect the analysis model. Check related settings are presented later, along with composite checks ("Composite Checks").





Take creep into account	defines whether creep should be taken into account or not in the CAM; when this setting is disabled, *final stage, long term* is not used and any load case assigned to that stage are moved to *final stage, short term*
Creep coefficient	value of the creep coefficient for the calculation of the effect of creep in *final stage, long term*
The composite beams are propped	defines whether the composite beams should be considered propped or not during casting; when this setting is enabled, *construction stage* is not used and any load case assigned to that stage is moved to *final stage, long term* or *final stage, short term* depending on the *creep* setting
Default degree of connection for beams with composite action	default value for degree of connection for beams *with standard composite action*; this value is used for composite beams where the *connection* setting is set to *default*

About results...

Most standard results are available in a completely standard form when using the CAM (displacements, internal forces, support reactions...).

There are however some restrictions regarding the output of stresses as standard results.

- stresses in 1D members can be displayed only in the eccentric part (steel profile) of composite beams *with advanced composite action* and *without composite action*; output of stresses in beams *with standard composite action* is not supported at this time
- stresses in 2D members (composite decks and metal decks) are not supported because of their orthotropic behaviour, which would require some special processing that is not compatible with the standard 2D-stress output

Stresses can be, however, handled in detail in **composite checks**.

Example

Simple composite building modelized for IBC composite check: CAM_demo_IBC.esa

Composite Checks

Composite code checks are based on the Open Checks technology and Scia Design Forms. However, in order to avoid duplicate input of some of the data and to take most advantage of the CAM, all input needed for composite checks have been centralized in the composite service in the Composite Beam Data member attribute and in the Composite setup.

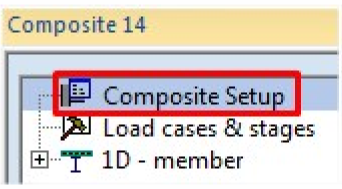
This chapter gives detailed information about the settings available in the Composite setup and in the Composite Beam Data which are related to the composite checks.

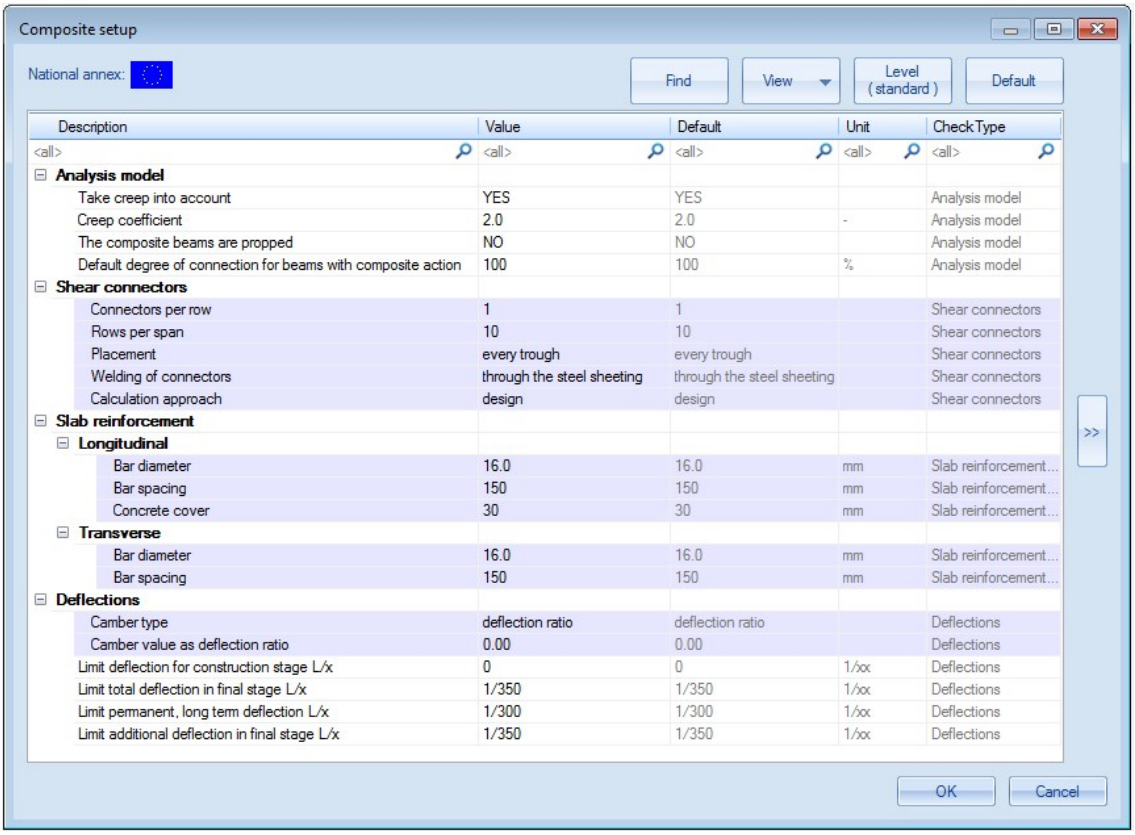
Some general information about how to use the checks is also provided. Theoretical background about the content of the checks is not provided here. References to the appropriate code articles are usually provided in the detailed output of the checks themselves.

For more information about Open Checks and Scia Design Forms, please refer to "Open Checks: Link with Scia Design Forms".

Composite Setup

The composite setup is accessible from the composite service tree.





Description	Value	Default	Unit	Check Type
<all>	<all>	<all>	<all>	<all>
Analysis model				
Take creep into account	YES	YES		Analysis model
Creep coefficient	2.0	2.0	-	Analysis model
The composite beams are propped	NO	NO		Analysis model
Default degree of connection for beams with composite action	100	100	%	Analysis model
Shear connectors				
Connectors per row	1	1		Shear connectors
Rows per span	10	10		Shear connectors
Placement	every trough	every trough		Shear connectors
Welding of connectors	through the steel sheeting	through the steel sheeting		Shear connectors
Calculation approach	design	design		Shear connectors
Slab reinforcement				
Longitudinal				
Bar diameter	16.0	16.0	mm	Slab reinforcement...
Bar spacing	150	150	mm	Slab reinforcement...
Concrete cover	30	30	mm	Slab reinforcement...
Transverse				
Bar diameter	16.0	16.0	mm	Slab reinforcement...
Bar spacing	150	150	mm	Slab reinforcement...
Deflections				
Camber type	deflection ratio	deflection ratio		Deflections
Camber value as deflection ratio	0.00	0.00		Deflections
Limit deflection for construction stage L/x	0	0	1/xx	Deflections
Limit total deflection in final stage L/x	1/350	1/350	1/xx	Deflections
Limit permanent, long term deflection L/x	1/300	1/300	1/xx	Deflections
Limit additional deflection in final stage L/x	1/350	1/350	1/xx	Deflections

The settings related to the analysis model (CAM) have been detailed already in the previous chapter "Composite Analysis Model in Scia Engineer". The rest of the settings are related to the composite checks. Most of them define default settings to be used for composite members without specific settings. Those settings can be overriden (marked *overridable* in the table below) by means of Composite Beam Data attributes (see next chapter "Composite Beam Data").

Shear connectors

Connectors per row

number of connectors per row (perpendicularly to the axis of the beam)

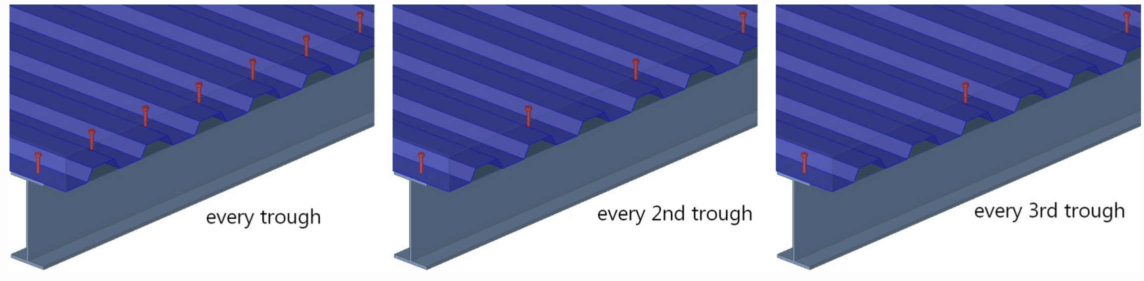
overridable

Rows per span

total number of rows of connectors in one span; used in case the corrugation of the sheeting is parallel to the beam

overridable, only EN1994

Placement



used in case the corrugation of the sheeting is perpendicular to the beam

overridable, only EN1994

Number of rows (between points of min & max moment)

number of rows of connectors between the points of min and max moments in one span

overridable, only AISC 360-10

Welding of connectors

method of welding of the shear connectors

- **through the steel sheeting**: the connectors are welded together with the sheeting to the beam
- **directly to the beam**: the connectors are placed in openings in the sheeting and welded directly to the beam

overridable

Calculation approach

approach for the calculation of the connectors

- **design**: the composite check returns the number of required shear connectors to fulfil the code requirements, based on the provided assumptions (placement or rows per span are then not used for the calculation)
- **check**: the composite check controls whether the provided shear connectors fulfil the code requirements

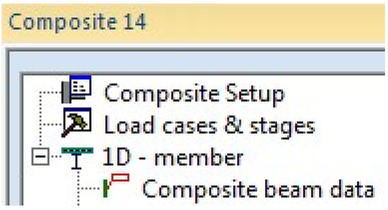
overridable

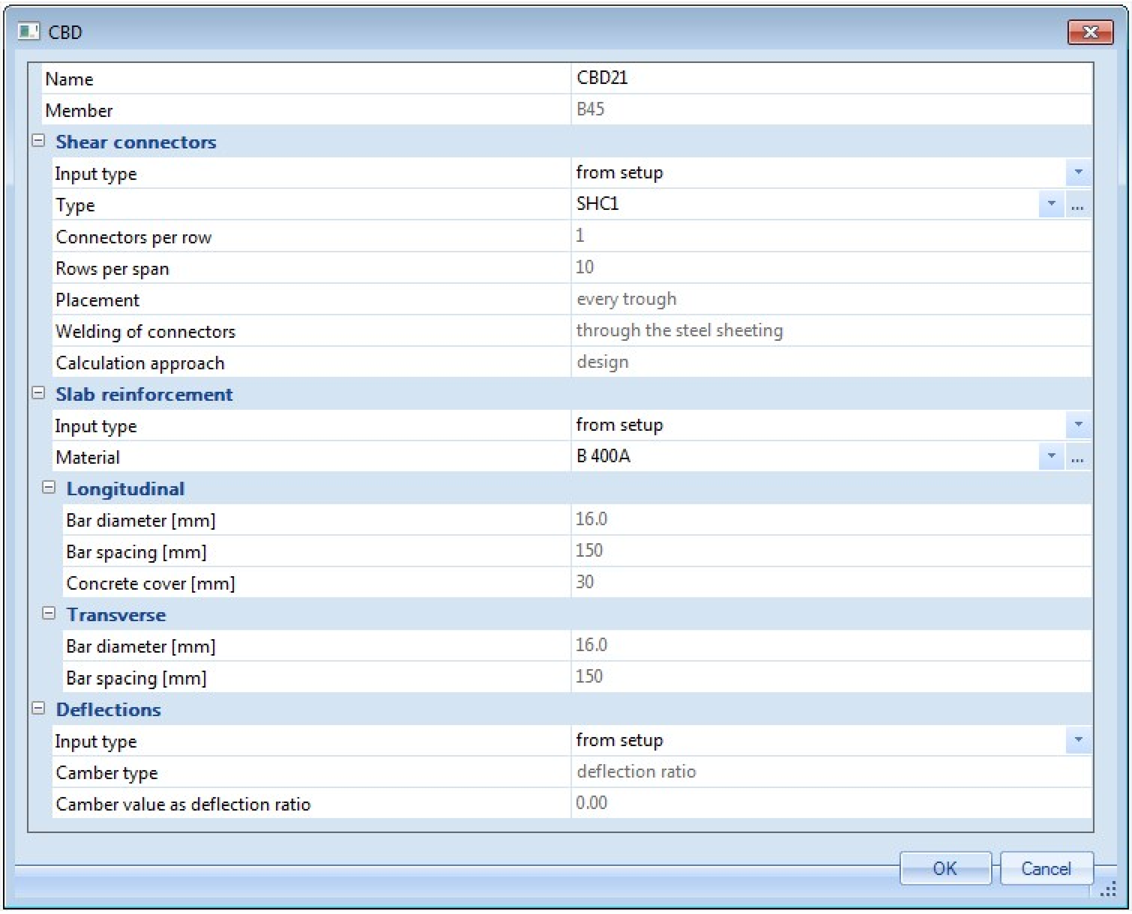
Slab reinforcement

Lon-gitudinal	Bar diameter, bar spacing and concrete cover of reinforcement bars in the slab parallel to the beam *overridable*
Transversal	Bar diameter and bar spacing of reinforcement bars in the slab perpendicular to the beam *overridable*
Deflections	
Camber type	type of definition of the camber value • **absolute**: the camber is defined as a fixed value (in the length units defined in the general settings of Scia Engineer) • **relative**: the camber is defined as a ratio of the span length, e.g. L/200 • **deflection ratio**: the camber is defined as a ratio of the cumulated deflection in construction stage and final stage, long term, i.e. usually all permanent loads *overridable*
Camber value	camber value for type *absolute*, defined as a fixed length *overridable*
Camber value L/x	camber value for type *relative*, defined as a ratio of the span length, e.g. L/200 *overridable*
Camber value as deflection ratio	camber value for type *deflection ratio*, defined as a ratio of the permanent deflection *overridable*
Limit deflection for construction stage	limit allowable deflection for deflection check in construction stage, defined as a ratio of the span length
Limit total deflection in final stage	limit allowable deflection for total deflection check in final stage, defined as a ratio of the span length
Limit permanent, long term deflection	limit allowable deflection for permanent deflection check in final stage, defined as a ratio of the span length
Limit additional deflection in final stage	limit allowable deflection for deflection check under variable actions in final stage, defined as a ratio of the span length *EN1994 only*

Composite Beam Data

The composite beam data is accessible from the composite service tree. It can be added to any composite beam.

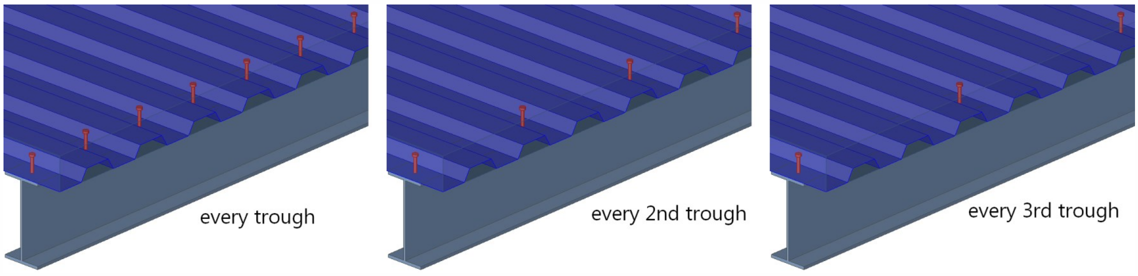




The composite beam data attribute allows to override for a specific composite beam the default settings defined in the composite setup. For each group of settings (e.g. shear connectors or slab reinforcement) it is possible to separately specify if the default settings must be used or rather some customized values.

Most of the composite beam data settings get their default value from the composite setup. A few exceptions exist, which are mentioned in the table below.

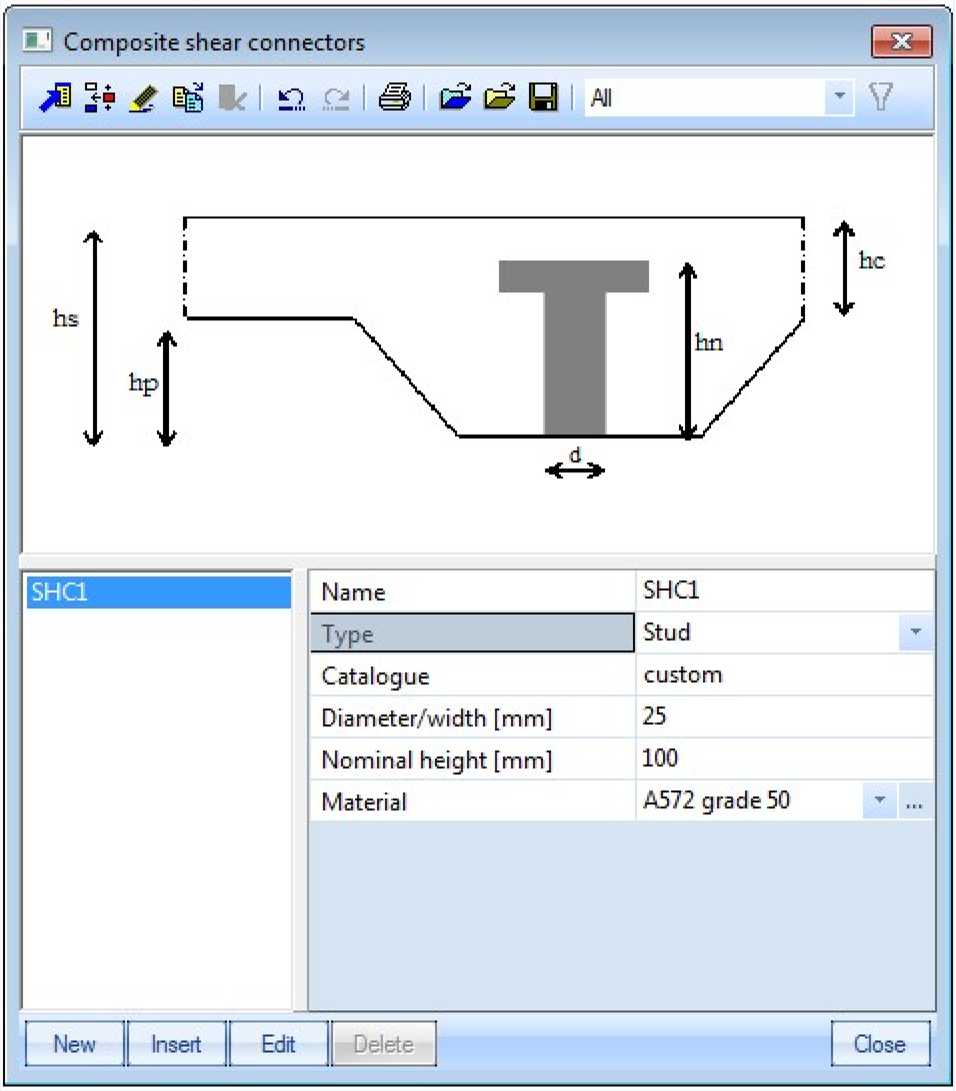
Name	name of the composite beam data attribute
Member	read-only; name of the related composite beam
Shear connectors	
Input type	override setting • **from setup**: use the values defined in the composite setup for shear connectors • **manually**: define specific values overriding the defaults
Type	type of shear connectors, selected from the shear connectors library (see below "Shear connectors lib-

	rary") *There is no default value for this setting in the composite setup. The default is taken as the first shear connector type available in the shear connectors library. This applies for composite beams without composite beam data.*
Connectors per row	number of connectors per row (perpendicularly to the axis of the beam)
Rows per span	total number of rows of connectors in one span; used in case the corrugation of the sheeting is parallel to the beam *only EN1994*
Placement	 used in case the corrugation of the sheeting is perpendicular to the beam *only EN1994*
Number of rows (between points of min & max moment)	number of rows of connectors between the points of min and max moments in one span *only AISC*
Welding of connectors	method of welding of the shear connectors • **through the steel sheeting**: the connectors are welded together with the sheeting to the beam • **directly to the beam**: the connectors are placed in openings in the sheeting and welded directly to the beam
Calculation approach	approach for the calculation of the connectors • **design**: the composite check returns the number of required shear connectors to fulfil the code requirements, based on the provided assumptions (placement or rows per span are then not used for the calculation) • **check**: the composite check controls whether the provided shear connectors fulfil the code requirements
Slab reinforcement	
Input type	override setting • **from setup**: use the values defined in the composite setup for slab reinforcement • **manually**: define specific values overriding defaults
Material	reinforcement steel material for the slab reinforcement

	There is no default value for this setting in the composite setup. The default is taken as the default reinforcement steel material from project settings (EC-EN) or as the first available reinforcement steel material in the library of materials. This applies for composite beams without composite beam data.
Longitudinal	Bar diameter, bar spacing and concrete cover of reinforcement bars in the slab parallel to the beam
Transversal	Bar diameter and bar spacing of reinforcement bars in the slab perpendicular to the beam
Deflections	
Input type	override setting • **from setup**: use the values defined in the composite setup for camber • **manually**: define specific values overriding defaults
Camber type	type of definition of the camber value • **absolute**: the camber is defined as a fixed value (in the length units defined in the general settings of Scia Engineer) • **relative**: the camber is defined as a ratio of the span length, e.g. L/200 • **deflection ratio**: the camber is defined as a ratio of the cumulated deflection in construction stage and final stage, long term, i.e. usually all permanent loads
Camber value	camber value for type *absolute*, defined as a fixed length
Camber value L/x	camber value for type *relative*, defined as a ratio of the span length, e.g. L/200
Camber value as deflection ratio	camber value for type *deflection ratio*, defined as a ratio of the permanent deflection

Shear connectors library

The composite shear connectors library is accessible as any standard library, from the Libraries menu (composite submenu), from the main tree view (Libraries>Composite branch) and from the composite beam data attribute, when assigning shear connectors to a composite beam (see above "Composite Beam Data"). A selection of pre-defined shear connectors is available in the system library ( ).

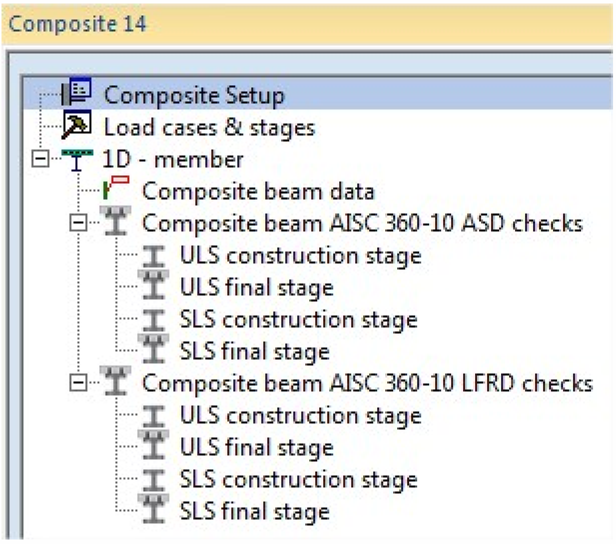


Name	name of the shear connector type
Type	generic type of connector; possible values are **Stud**, **Hilti**, **Channel** and **Bar hoop**
Catalogue	catalogue designation, can be any text, keyword... may be used for filtering the library (using the *catalogue* filter). Typical use would be the name of the manufacturer or the name of the product range
Diameter/width	cross-sectional dimension of the connector; typically diameter of a stud
Nominal height	nominal height of the connector for the calculation of the resistance
Material	steel material of the connector

All settings above are transmitted to the composite checks. They do not affect the analysis model.

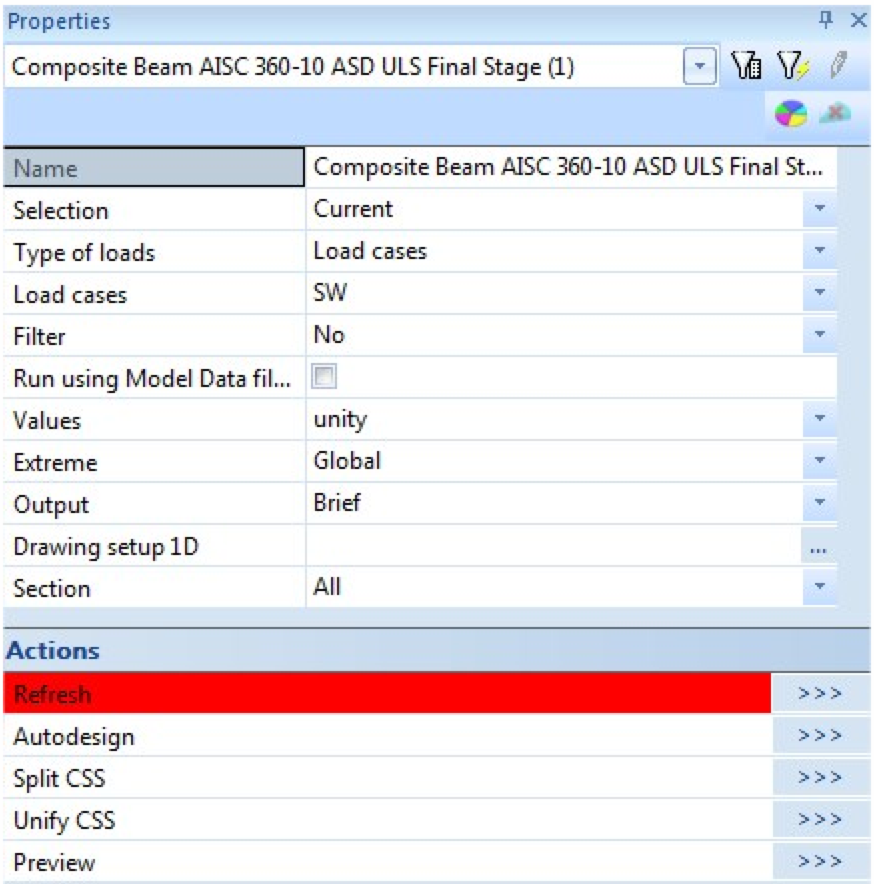
Composite Checks

All composite checks are accessible from the composite service tree. They are displayed after a successful analysis of the structure (analysis results available).



General use of the composite checks

All composite checks use the same standard settings as other result services in Scia Engineer. Results are available as text and graphical output. The text output can be obtained in a summary table output (only main results, one row per result) or as detailed output (full details of check, with intermediate results…).



Name name of the selected check

Selection	selection of entities on which the check will be performed (*all, current, advanced, named selection, design group*)
Type of loads	type of actions to be used for the check (*load case, combination, result class*)
Load case	selected load case for the check (in case *type of loads* = *load case*)
Combination	selected load case combination for the check (in case *type of loads* = *combination*)
Class	selected result class for the check (in case *type of loads* = *class*)

Combinator strategy	strategy used for handling envelopes
	• **Strain strategy**: so-called dangerous combinations, which are assumed to produce most critical results, are determined according to the following rules; example for internal forces in 1D members, the check is performed at most for each of the following 20 dangerous combinations (duplicates are eliminated):
	  • 12 combinations producing extreme values (min and max) of each of the 6 internal forces components
	  • 8 combinations producing extreme longitudinal strains (min and max) in each corner of an idealized rectangular cross-section having the same stiffness components than the considered cross-section
	• **All possible combinations**: all possible combinations from the selected envelope are processed; WARNING! this might lead to extremely long computation time!

Filter	standard filter on members (*wildcard, cross-section, material, layer*)
Print combination key	when enabled, print the combination key along with the results in the text output
Values	selection of the value(s) for graphical representation
Extreme	extreme selection mode (*section, local, member, interval, cross-section, global*)
Output	text output format • **brief**: table output, one row per result (depending on the selected *extreme* setting) • **detailed**: full, detailed output of the check
Drawing setup 1D	detailed configuration of graphical output
Section	sections for which the check must be performed on each selected member (*all, ends, inputted, inputted+ends*)
Refresh >>>	action button: perform the check
Autodesign >>>	action button: autodesign according to selected settings; filter must be *cross-section* (see more detailed explanation in Autodesign manual "AutoDesign - Global optimization")
Split CSS >>>	action button: split cross-section optimization according to unity check value of selected members
Unify CSS >>>	action button: assign the same cross-section to several members having different cross-sections
Preview >>>	action button: display the text output window

Available checks in Scia Engineer 15

- EN1994 Composite Beam Checks
  - ULS construction stage check
  - ULS final stage check

- AISC 360-10 ASD Composite Beam Checks
    - ULS construction stage check
    - ULS final stage check
    - SLS construction stage check (deflections)
    - SLS final stage check (deflections)
- AISC 360-10 LFRD Composite Beam Checks
    - ULS construction stage check
    - ULS final stage check
    - SLS construction stage check (deflections)
    - SLS final stage check (deflections)

References

1. A. Samanta and M. Mukhopadhyay, Finite element static and dynamic analyses of folded plates, Engineering Structures, vol. 21, no. 3, pp. 277–287, 1999

2. David Wennberg, Per Wennhage and Sebastian Stichel, Orthotropic Models of Corrugated Sheets in Finite Element Analysis, ISRN Mechanical Engineering, Volume 2011, Article ID 979532, 2011