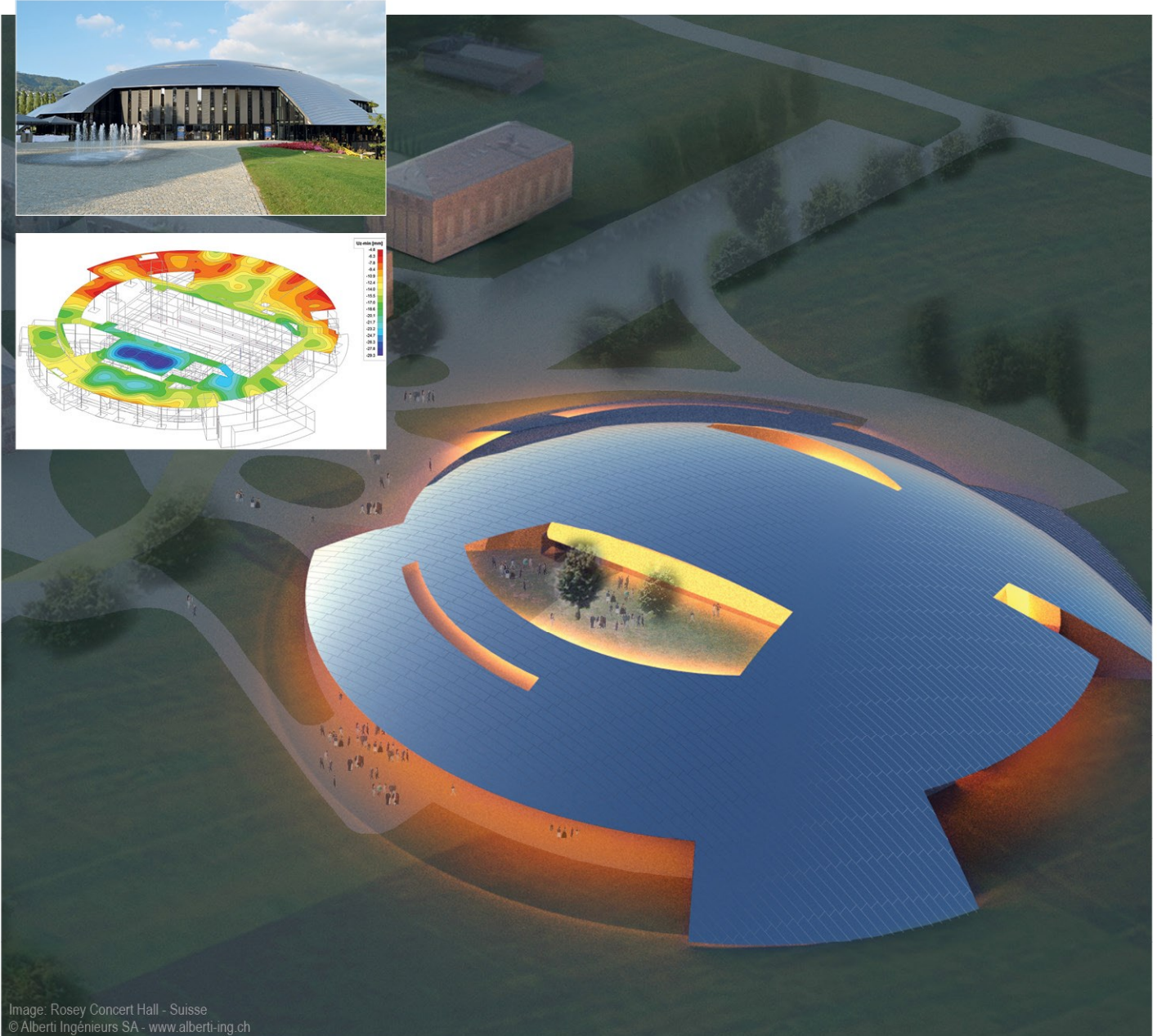


SCIAENGINEER



US Composites User Guide and Theoretical Background

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2016 SCIA nv. All rights reserved.

Table of contents

Table of contents	3
Quick Start Checklist	6
User Guide	7
Terminology	7
Composite Model Considerations	7
Setting Up Cross-Sections Groups in Model	7
1D Member Properties	7
2D Member Properties	8
Mesh Size.....	9
Loading Composite Members	9
Load Case Setup	9
Load Combination Setup.....	10
Load Class Setup.....	10
Composite Setup	11
Calculation approach	11
Design Options.....	12
Deflection and Camber Options	12
Running Composite Check and Design	13
Type of Load	13
Composite Check and Design.....	13
Updating Model with Design Parameters.....	13
Composite Results	13
Output Options	14
Theoretical Background	15
Overview	15
Design and Analysis Requirements	15
Design Codes.....	15
Units	15
Compatible Cross-Sections.....	15
Load Combinations	15
Computed Properties and Capacities	15
Effective width	15
Shear Stud Strength.....	15
Composite Flexural Capacity	16
Strength and Serviceability Checks	16
Construction Stage Strength Check.....	17
Final Stage Strength Check	17
Construction Serviceability Check.....	17
Final Live Load Serviceability Check	17
Final Total Load Serviceability Check	17
Detailing Checks	17
Moment Values	18
Displacement Values	18
Stud Design	18
Number of studs.....	19
Number of Rows of studs.....	19
Stud Layout: Uniformly Loaded Simple Span Members	19

Stud Layout: Segmented Members	19
Camber Design	20
Auto-Design	20
Procedure	20
Theoretical Background: Composite Analysis Model	23
General principles	23
Composite deck	23
Profiled steel sheeting	24
Bending components	24
Membrane components	24
Mean thickness (for calculation of self-weight)	25
Position of gravity center from bottom fiber (assumed for both directions)	25
Auxiliary variables	25
Concrete deck	25
Bending components	26
Membrane components	26
Mean thickness (for calculation of self-weight)	26
Position of gravity center from bottom fiber (assumed for both directions)	26
Auxiliary variables	26
Multi-layered orthotropy	27
Plate behavior components.....	27
Membrane behavior components.....	28
Layer eccentricities	28
One-way decks	28
Composite beam	30
Adjusted properties of beams “with standard composite action”	31
Composite action with partial connection.....	33
Construction stages for composite analysis	34
Creep.....	34
Propping	34

Quick Start Checklist

- For best optimization results, members should be grouped together with elements that have similar length, end releases and tributary areas.
- All cross sections intended for composite beam design must be wide flange (W) shapes (either metric or imperial naming convention)
- Correct setup of load cases, combinations and classes
 - Set up load cases and set stage (construction or final) within the case
 - Set up load combinations where every case in the combination is from the same stage. The combinations should use either the LRFD or ASD design method (other options are not supported). There should be at least four combinations:
 - 1 – Construction ULS (strength)
 - 2 – Construction SLS (serviceability)
 - 3 – Final ULS (strength)
 - 4 – Final SLS (serviceability)
 - Set up a load class which contains load combinations. The load combinations must all use the same design method.
- Set check and design parameters in the **Composite Setup**
 - If Calculation Approach is set to **Design**, the **design approach** can be set to:
 - 1 – **Minimize beam size** to design to the least weight section but higher stud count
 - 2 – **Minimize studs** to design for the least amount of studs but higher weight section
 - 3 – **Balanced condition** to design for a balanced condition in which both the beam size and number of studs is
 - 4 – **User defined** in order to design a cross-section according to a user defined target degree of composite action
- After the calculation is finished, select **Composite beam design (AISC 360-10)** from the Composite service and click **Refresh** in the **Properties** window under **Actions** to run the check
- Select Apply design proposals to model to update model with the optimized cross-section and target degree of composite action

Terminology

Construction stage: the state of the structure when the members have no composite action and the steel beams alone support loads. Also referred to as “pre-composite”

Final stage: the state of the structure when the steel cross-sections and the concrete slab exhibit composite action due to their connection using shear studs.

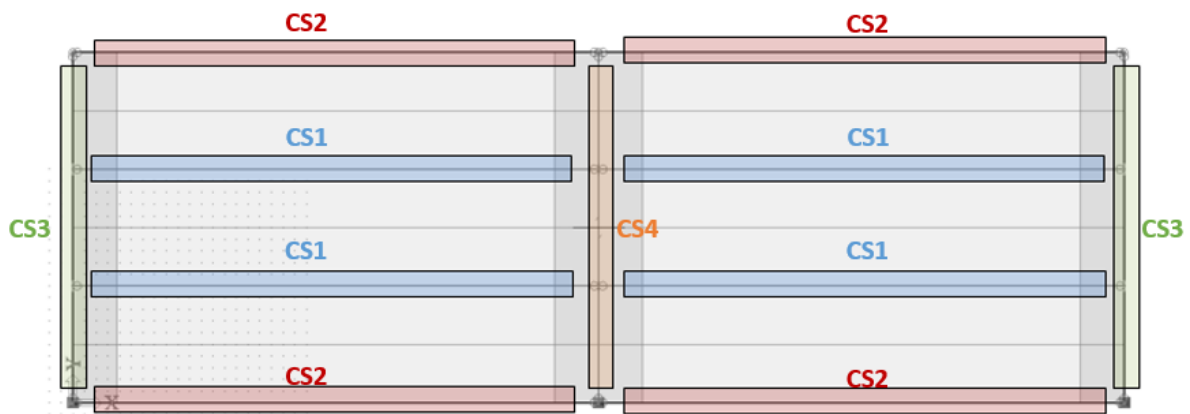
Target Degree of Composite Action: The degree of composite action or connection between the steel cross-section and concrete slab.

Composite Model Considerations


Setting Up Cross-Sections Groups in Model

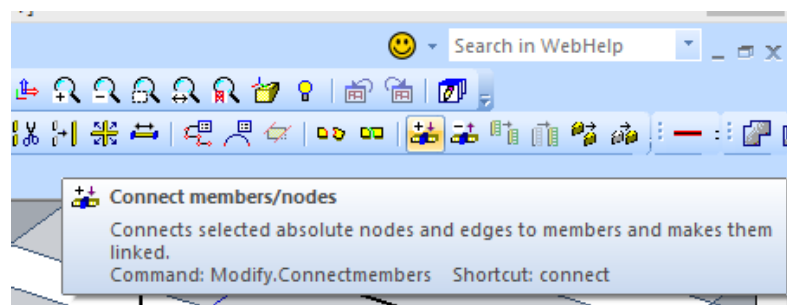
When building a model in SCIA that contains composite members, some consideration should be made to create cross-section groups that contain members with similar tributary areas. Because auto-design is performed on cross-section groups and all members in the group are designed based on the worst-case member in the group, grouping in this way will prevent some members from being considerably over-designed or under-designed.

The following diagram shows an example of how cross-sections should be grouped:



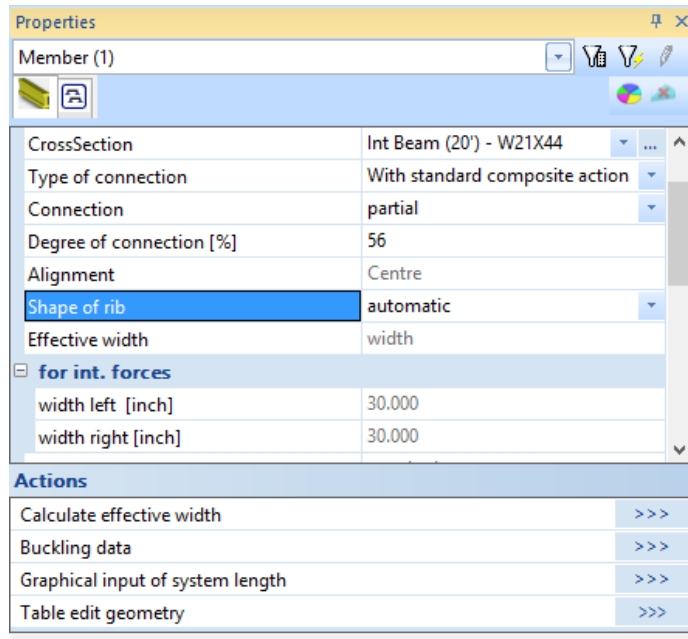
1D Member Properties

In SCIA, composite members are modelled as 1D members. Ribs can be added to a 2D member in two ways. First by direct input using the 1D member input functionality  Rib or by using the “Connect members/nodes” tool.




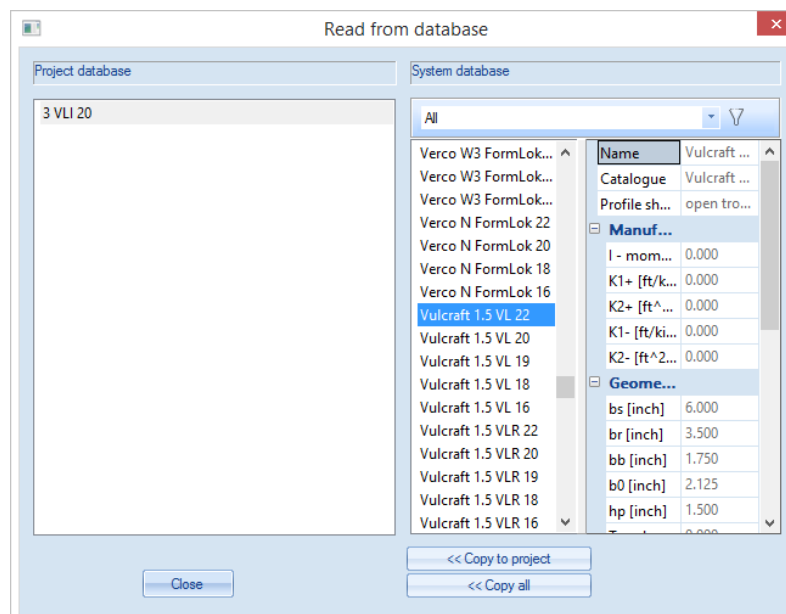
The following 1D member properties must be used when designing composite members:

- **Type of Connection** should be set to “With standard composite action” (default setting) to indicate that a connection exists between the steel member and the concrete slab. Alternatively, if there is no connection and the deck is supported by the beam, set **Type of Connection** to “Without composite action”. The option for “With advanced composite action” models the plate rib with its real eccentricity, but is not compatible with the composite beam design module.
- **Shape of rib** should be set to “automatic” in order for the effective width of the member to be automatically calculated. The effective width can be calculated by clicking “Calculate effective width” under **Actions**. Alternatively, the effective width can be defined manually by setting **Shape of rib** to T-symmetric, slab left, slab right, or asymmetric.

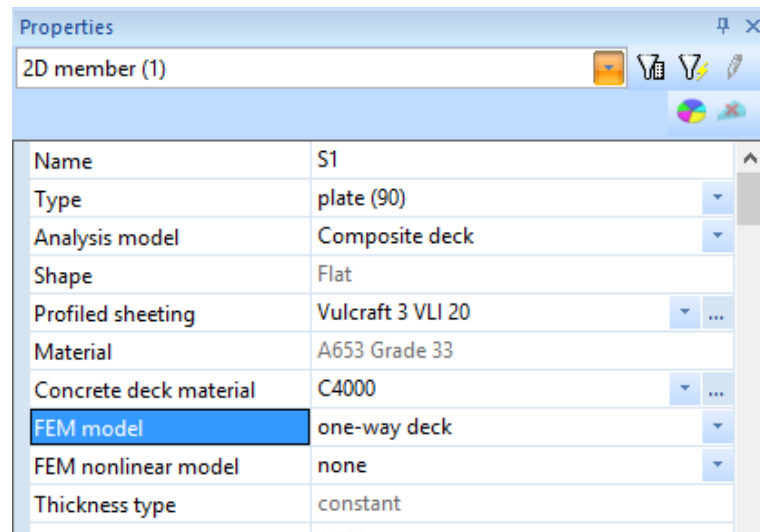


2D Member Properties

It is a requirement for the 2D Member properties that the **Analysis model** be set to “Composite deck”. When this is the case, the decking library becomes available. SCIA Engineer contains a full library of metal deck profiles. To access this library, click the blue folder icon  (“read from system database”) from within the **profile sheeting** dialog when adding a new 2D member or when updating the current profile. Select the desired profile and click “Copy to project” to make it available in the model.



Additionally, it is recommended that the FEM model be set to “one-way deck”. Setting **FEM model** to “one way” will remove the stiffness that exists in the deck in the direction perpendicular to the plate ribs, allowing the moment values at the hinges to be zero for members which are oriented perpendicular to the ribs.



Mesh Size

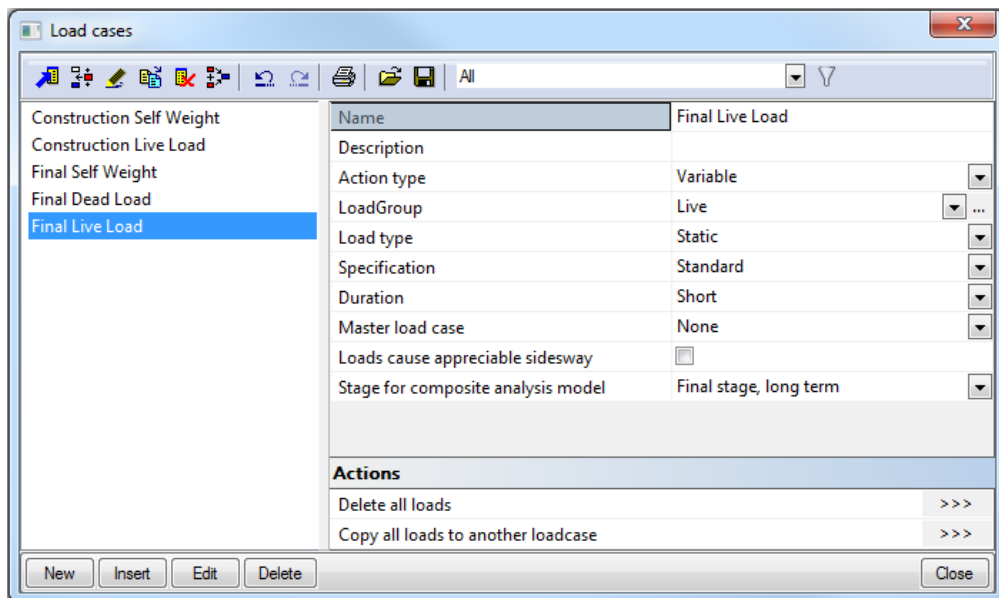
As with any finite element model, some consideration should be given to the size of the mesh before running an analysis. A trade-off between precision and speed exists as the size of the mesh is adjusted. It is generally recommended that the **Average size of 2d element/curved element** be set to a value approximately one-tenth of the average member length or one-tenth of the shortest member if needed. For most projects, a mesh size of 3ft is reasonable.

Loading Composite Members

Composite beam checks and design involve analysis of the members for both construction and final stages and for multiple limit states (strength and serviceability). For this reason, some consideration is required to properly set up load cases and combinations, as well as a load class. The load class is required so that all stages and limit states can be considered and checked simultaneously.

Load Case Setup

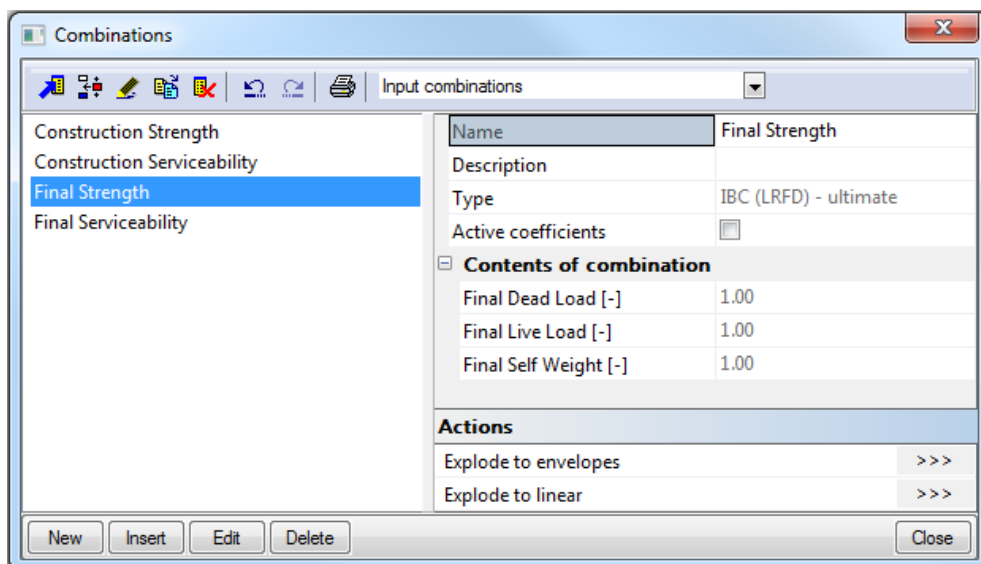
For each load case, the stage in which the load is intended to occur must be specified using the drop-down menu under **Stage for composite analysis model**. The options for the stages are “Construction stage”, “Final stage – long term”, and “Final stage – short term.” Putting the load at the intended stage will ensure the load is only included in the analysis for that stage. Any load case that is intended for both the construction and final stages, such as self-weight, must be entered twice.



Load Combination Setup

Once all load cases have been created, combinations containing these load cases should also be created. The load combinations can be given any name, but all load cases in the combination must be from the same stage - either construction or final (final long term and final short term loads are expected to be included in the same combination). This way the program can automatically determine for which stage the load combination is intended and can use the combination correctly for designing the member.

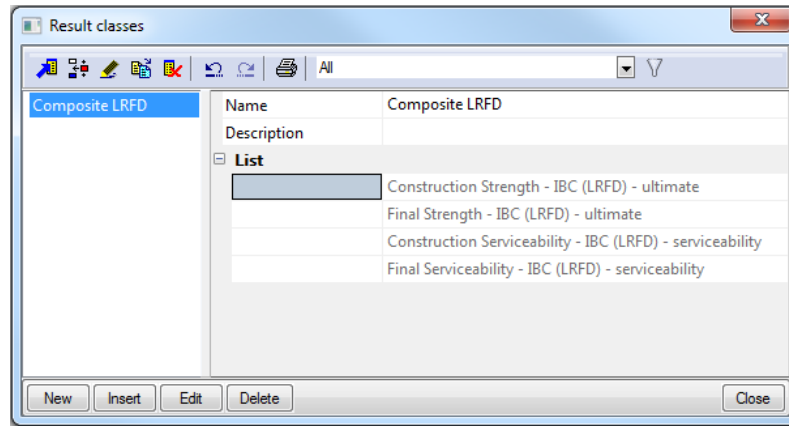
Additionally, the design methodology under **Type** must also be specified for the combination. The supported design methodologies for composite analysis are **ASD** and **LRFD**. At least four combinations of the same design method should be created: one for the construction stage strength (or ultimate) check, one for the construction stage serviceability check, one for the final stage strength (ultimate) check, and one for the final stage serviceability check.



Load Class Setup

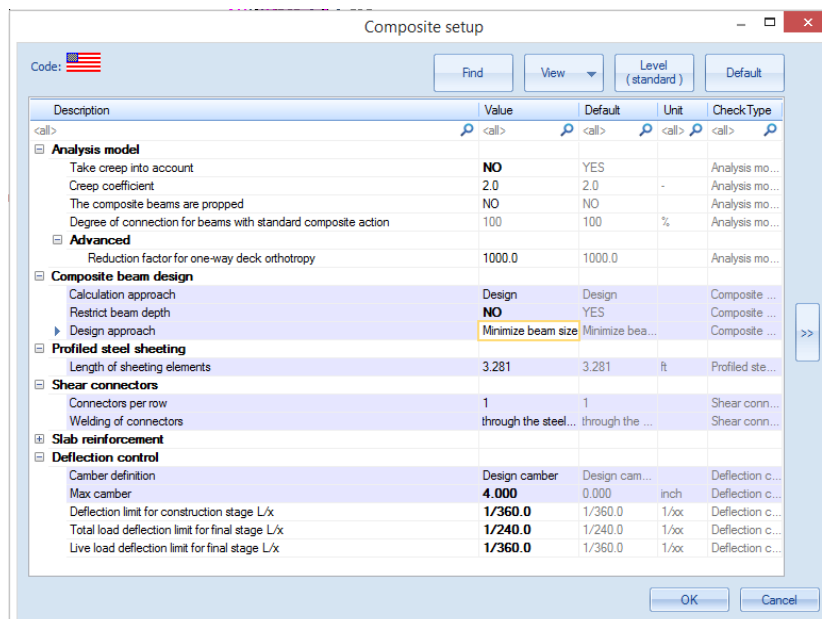
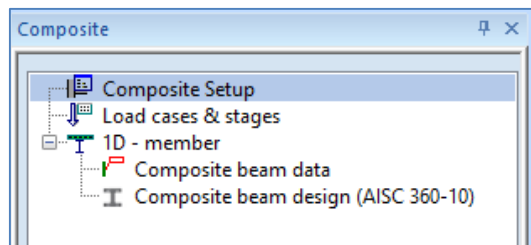
After load cases and combinations have been created as specified above, the load combinations must be put into a load class. It is important that the class contains the four required combinations and that these combinations use a consistent design method.

To add combinations to the class, simply click **New** and add the combinations:



Composite Setup

The options for composite beam analysis and design can be found in the **Composite Setup** window, as shown below:



Calculation approach

Composite members have the option to be checked and designed or just checked. This can be specified under the option of "Calculation approach." With the **check** option the program simply checks the members for strength and serviceability at construction and final stages. With the **design** option the members are not only checked, but an optimum cross-section and target degree of composite action will be calculated by the program for each cross-section group.

Design Options

If the calculation approach is set to design, a number of additional parameters can be set:

- **Restrict beam depth:**
 - Allows the maximum depth of the designed cross-section to be restricted
 - Select Options: Yes or No
- **Max beam depth:**
 - Visible if “Restrict beam depth” is set to “Yes”
 - A value that the maximum beam depth should not exceed can be entered
- **Design approach:**
 - Dictates how the beam cross-section and target degree of composite action should be designed
 - Options:
 - **Optimize beam:** Design based on member with the least weight. For this option, target degree of composite action will be set to 100% (full).
 - **Optimize studs:** Design based on fewest number of studs. For this option, target degree of composite action will be set to 30%.
 - **Balanced:** Design based on balanced approach. For this option, target degree of composite action will be set to 55%.
 - **User Defined:** Design will find section that works with target degree of composite action that the user inputs.
- **Target degree of composite action:**
 - Visible when “Design approach” is set to “User Defined”
 - A value for the target degree of composite action can be entered
- **Connectors per row**
 - Maximum number connectors/studs situated perpendicular to the length of the beam and attached to the beam flange permitted in the design (i.e. to allow only one stud, enter “1”, to allow one or two studs, enter “2”). Value should not exceed “2”. If “2” is entered, two rows will be used only if minimum spacing conditions require a second row and the width of the flange to which the studs are connected is wide enough to allow two rows.

Deflection and Camber Options

- Camber definition & Camber value/Max camber:
 - **Design camber:** During check of the current member in the model, the amount of camber needed will be determined based on deflection checks. It will not exceed the value entered in **Max camber**. Design of camber can be used whether “Calculation approach” is set to “Design” or “Check”
 - **No camber:** No camber will be taken into consideration for the deflection checks
 - **Input camber (absolute):** The absolute value (in inches) that is entered in “Camber value” will be used for the deflection checks of all members
 - **Input camber (relative):** The relative value (relative to member length, L/x) that is entered in “Camber value” will be used for the deflection checks of all members

- Deflection Limits:

The following deflection limits can be entered relative to member length, L/x :

- **Deflection limit for construction stage:** Deflection limit for load due to self-weight in construction stage. Default set to $1/360$

- **Total load deflection limit for final stage:** Deflection limit for total load in final stage. Default set to 1/240
- **Live load deflection limit for final stage:** Deflection limit for live load in final stage. Default set to 1/360

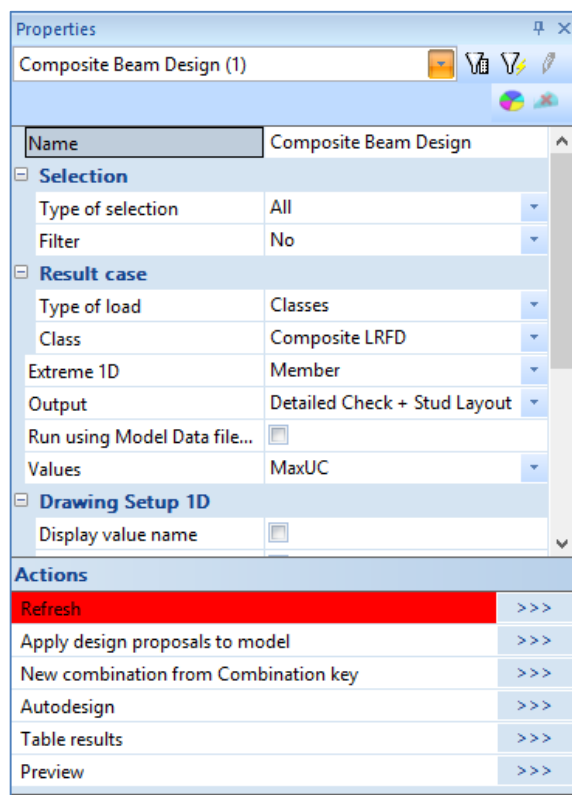
Running Composite Check and Design

Type of Load

The composite beam check is designed to work with a load class which contains multiple load combinations. The **Type of load** should be set to “Classes” and the load class for which the check is intended should be selected for **Class**.

Composite Check and Design

After a model has been created, preferences have been made in the setup, and the model has been analyzed, the check and design can be performed. To run the check select **Refresh** under **Actions** in the **Properties** window.



Setting **Values** to **MaxUC** will display the maximum unity check value for each member in the model. If the design approach is set to “design” in the composite setup, the program will automatically select a cross-section and/or a target degree of composite action that are optimized for each cross-section group.

Updating Model with Design Parameters

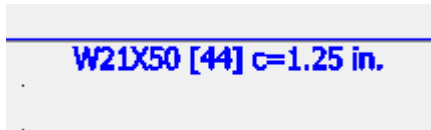
When design approach is set to “design” and new design parameters have been selected using the steps outlined in the previous section, these new parameters can be used to update the model by selecting **Apply design proposals to model** under **Actions** in the **Properties** window.

Composite Results

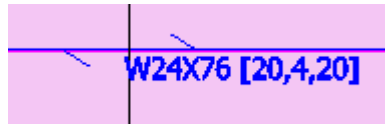
There are three result values that can be returned after running the composite check:

- **MaxUC:** The maximum unity check value from strength and serviceability checks for both the construction and final stage as well as the minimum spacing requirement check.

- **Uniform Result:** A string showing the steel cross-section, the number of uniformly spaced studs along the member, and the camber (if any). If the member is one which has segments (due to point loads), the uniform number of studs based on the minimum stud spacing will be displayed.



- **Segmented Result:** A string showing the steel cross-section, the number studs in each segment or sub-segment along the member, and the camber (if any). If the member has no segments (caused by the presence of point loads), the uniform number of studs will be displayed.



Output Options

The composite beam module has 4 levels of output which show results and calculations related to the check of each member:

- **Brief:** Only the three results (See Composite Results above) are shown for each member in a table
- **Summary:** A summary view which includes:
 - Internal force data
 - Steel, concrete, and composite member parameters
 - A diagram of the connection
 - Diagrams of the moment values and shear stud layout
 - Detailing check summary
 - Strength and serviceability check summary
- **Stud Layout:**
 - Diagrams of the moment values and shear stud layout only
- **Detailed + Stud Layout**
 - Includes all items in the Summary output plus detailed strength and serviceability checks

Theoretical Background

Overview

SCIA's Composite Beam Design module has been created to check and design composite beam members for strength and serviceability requirements under both construction (pre-composite) and final (composite) stages. Initially, the members in the model are checked by comparing internal forces and deflections to their capacities and to deflection limits. In addition to checking the members, the program is able to design the members based on optimization parameters that can be specified in the setup. The design suggests both a cross-section and target degree of composite action which have been calculated to meet the design requirements of the member and can easily replace existing members.

Design and Analysis Requirements

Design Codes

Composite beam checks are performed according to the AISC Steel Construction Manual (14th Edition). Both ASD and LRFD design methodologies are supported. The design method is determined automatically from the load combinations that exist in the load class used for the check.

Units

Units within the model, can be either Metric or Imperial.

Results for composite beam checks are shown in Imperial units.

Compatible Cross-Sections

The composite beam module only works for W (wide-flange) composite sections.

Load Combinations

The check requires that a load class be created and that load combinations for each stage of analysis are put into the load class according to the specifications in "Loading Composite Members" above. If the class is missing a combination or if combinations from different design methods (i.e. LRFD and ASD) are included in the same class, an error message, which explains what is missing or inconsistent in the load combination, will be displayed in the "Summary", "Stud Layout", or "Detailed" output preview. In addition, the label on the member will show a result of "XXX RLC" as an indication to review load combinations.

Computed Properties and Capacities

Effective width

The effective width of the composite member is a property of the member in the model which is calculated automatically. For more information on how effective width is calculated, see "[Automatic Calculation of the Effective Width of Plate Ribs](#)" from the online help.

Shear Stud Strength

The strength of a single shear stud is calculated using AISC Spec I8.2a

$$Q_n = 0.5A_{sc}\sqrt{f'_c E_c} \leq R_g R_p A_{sc} \sigma_u$$

Example calculation for 3/4 inch shear stud:

Shear Stud Strength - AISC 360-10 I8.2a

$$Q_n = \min\left(0.5 \cdot A_{sa} \cdot \sqrt{F_c \cdot E_c}; R_g \cdot R_p \cdot A_{sa} \cdot F_{u,stud}\right) \quad \text{Nominal strength of a single shear stud}$$
$$= \min\left(0.5 \cdot 0.442 \text{ in}^2 \cdot \sqrt{4 \text{ ksi} \cdot 4032 \text{ ksi}}; 1 \cdot 0.6 \cdot 0.442 \text{ in}^2 \cdot 65 \text{ ksi}\right) = 17.2 \text{ kip}$$

Composite Flexural Capacity

The positive and negative flexural strength of the composite member are determined from the plastic stress distribution on the composite section for the limit state of yielding as specified in Section I.4.2a and Section I.4.2b of the AISC Steel Specification.

The detailed calculation of capacity for any member can be viewed as part of the "Detailed Check + Stud Layout" output.

Example calculation for a W24x55 with a target composite action of 50%:

Positive Flexural Strength of Beam - AISC 360-10 I3.2a

$$P_y = F_y \cdot A_s = 50 \text{ ksi} \cdot 16.2 \text{ in}^2 = 810 \text{ kip} \quad \text{Steel section axial capacity}$$

$$C_{\text{Conc}} = A_c \cdot 0.85 \cdot F_c = 360 \text{ in}^2 \cdot 0.85 \cdot 4 \text{ ksi} = 1224 \text{ kip} \quad \text{Concrete slab axial capacity}$$

$$V_{\text{studs}} = N_{\text{studs}} \cdot Q_n = 24 \cdot 17.2 \text{ kip} = 414 \text{ kip} \quad \text{Shear studs capacity}$$

$$C = \min(P_y; C_{\text{Conc}}; V_{\text{studs}}) = \min(810; 1224; 414) = 414 \text{ kip} \quad \text{Compressive force in slab}$$

$$\begin{aligned} \text{CRatio} &= \min\left(1; \frac{V_{\text{studs}}}{\min(P_y; C_{\text{Conc}})}\right) \\ &= \min\left(1; \frac{414}{\min(810; 1224)}\right) = 0.511 \quad \text{Ratio of composite action} \end{aligned}$$

$$a = \frac{C}{0.85 \cdot F_c \cdot b_{\text{eff}}} = \frac{414 \text{ kip}}{0.85 \cdot 4 \text{ ksi} \cdot 120 \text{ in}} = 1.01 \text{ in} \quad \text{Depth of compression zone in concrete}$$

$$a = \min(a; h_s) = 1.01 \text{ in}$$

$$f_{cs} = \frac{P_y - C}{2} = \frac{810 \text{ kip} - 414 \text{ kip}}{2} = 198 \text{ kip} \quad \text{Compression in Steel Member}$$

PNA within web

$$a_s = 2.05 \text{ in} \quad \text{Depth of compression region in steel section}$$

$$d_1 = h_s + h_p - \frac{a}{2} = 3 \text{ in} + 3 \text{ in} - \frac{1.01 \text{ in}}{2} = 5.49 \text{ in} \quad \text{Distance from top of beam to center of slab}$$

$$\begin{aligned} d_2 &= \frac{C_{\text{flange}} \cdot \frac{t_f}{2} + C_k \cdot \left(t_f + \frac{a_{sk}}{2}\right) + C_{\text{web}} \cdot \left(\frac{a_{sweb}}{2} + k\right)}{f_{cs}} \quad \text{Distance from top of beam to centroid of compression region} \\ &= \frac{177 \text{ kip} \cdot \frac{0.505 \text{ in}}{2} + 4.92 \text{ kip} \cdot \left(0.505 \text{ in} + \frac{3.1 \text{ in}}{2}\right) + 16.3 \text{ kip} \cdot \left(\frac{0.826 \text{ in}}{2} + 1.22 \text{ in}\right)}{198 \text{ kip}} = 0.411 \text{ in} \end{aligned}$$

$$d_3 = \frac{d}{2} = \frac{23.6 \text{ in}}{2} = 11.8 \text{ in} \quad \text{Distance from T to the top of steel section}$$

$$\begin{aligned} M_{n+} &= C \cdot (d_1 + d_2) + P_y \cdot (d_3 - d_2) \quad \text{Positive Moment Capacity} \\ &= 414 \text{ kip} \cdot (5.49 \text{ in} + 0.411 \text{ in}) + 810 \text{ kip} \cdot (11.8 \text{ in} - 0.411 \text{ in}) = 972 \text{ kip}\cdot\text{ft} \end{aligned}$$

Strength and Serviceability Checks

The composite beam module checks members for strength and serviceability conditions at both the construction (or pre-composite) stage and the final (or composite) stage. All checks are performed together in one Design Form. For this reason load combinations for each check must be created and put into a load class according to "Loading Composite Members" section above.

The design method (ASD or LRFD) is automatically determined based on the design method specified in the load combinations.

Each check is performed on the whole member using the extreme values of internal force or deflection, rather than using values at each section along the member.

Each check returns a unity check (ratio of demand to capacity) value. The maximum unity check from all checks is called MaxUC and is available as a result which is visible on the member.

Construction Stage Strength Check

The construction stage strength investigates the capacity of the steel wide-flange cross-section (with no composite action) using the construction stage load combinations. It compares the flexural and shear capacity of the member, calculated according to AISC Specification Chapters F and G, respectively, to the maximum moment and shear forces.

If the plate ribs of the metal deck are perpendicular to the member (within 5 degrees), the member is considered to be continually braced.

The strength check does not support members that have a slender web.

The result returned for the construction stage strength check is the higher of two unity checks for bending and shear. Combined forces at each section are not considered in the check

Final Stage Strength Check

The final stage strength check investigates the capacity of the composite member using the final stage load combinations at the target degree of composite action.

The shear capacity of the composite member is taken available shear strength of the steel section alone as specified in Chapter G in accordance with Section I4.1(a) of the AISC Specification

The positive and negative flexural strength of the composite member are determined from the plastic stress distribution on the composite section for the limit state of yielding as specified in Section I4.2a and Section I4.2b of the AISC Steel Specification. If negative moment exists on the member requiring the negative flexural strength, the member is checked for compactness and a unity check value of 2.0 will be returned as well as a warning message if the member is non-compact. No non-compact members will be considered during the design selection process for members that have negative moment.

The result returned for the final stage strength check is the higher of two unity checks for bending and shear. Combined forces at each section are not considered in the check

Construction Serviceability Check

The construction serviceability check checks the deflection of the member due the self-weight of steel and concrete against the deflection limit set in the composite setup. The default construction deflection limit is 1/360. Camber is taken into consideration in the construction serviceability check. (For more information on how camber is designed when "Camber Definition" is set to "Design", see the subsequent section called "Camber Design".)

Final Live Load Serviceability Check

The final live load serviceability check checks the deflection of the member due to the live loads present in the final stage against the deflection limit set in the composite setup. The default construction deflection limit is 1/360. Camber is not considered in the final live load serviceability check.

Final Total Load Serviceability Check

The final total load serviceability check checks the deflection of the member due to all loads present in the final stage against the deflection limit set in the composite setup. The default construction deflection limit is 1/240. Camber is taken into consideration in the construction serviceability check. (For more information on how camber is designed when "Camber Definition" is set to "Design", see the subsequent section called "Camber Design".)

Detailing Checks

In addition to strength and serviceability checks listed above, checks for the detailing conditions are performed in the composite beam module according to AISC Specification Chapter I and shown in the "detailed" output. With the exception of the minimum stud spacing check, the detailing checks are not included in the "MaxUC" result value and are not considered in the auto-design of the member. The detailing checks shown in the detailed output are:

- Concrete compressive strength must be between 3000 psi and 10000 psi (AISC I1.1.3.1)

- Yield strength of steel reinforcement must be less than 75 ksi (AISC I1.1.3.2)
- Deck nominal rib height must not exceed 3 inches (AISC I3.2c1.1)
- Shear stud diameter must not exceed 0.75 inches (AISC I3.2c1.2)
- Studs must not extend less than 1.5 inches above the top of the steel deck (AISC I3.2c1.2) - Concrete cover must be at least 0.5 inches above top of studs (AISC I3.2c1.2)
- Concrete slab thickness must be at least 2 inches above the steel deck (AISC I3.2c1.3)
- Stud diameter must not be greater than 2.5 times the thickness of the metal to which it is welded (AISC I8.1)
- The stud height must be at least 4 times the stud diameter after installation (AISC I8.2)
- Minimum center-to-center spacing of studs along the longitudinal axis of the beam must not be less than six times the stud diameter except when the ribs of the metal deck are oriented perpendicular to the steel beam, the minimum center-to-center spacing of studs must not be less than four times the stud diameter. (AISC I8.2d)
- Maximum center-to-center spacing of studs along the longitudinal axis of the beam must not exceed 36 inches or 8 times the total slab thickness (AISC I8.2d)

Example of Detailing Checks output:

Detailing Checks

Check Description	Value	Reference	Pass/Fail
Concrete compressive strength is between 3000 psi $\leq F_c \leq$ 10000 psi	4000 psi	AISC I.1.1.3.1	Pass
Maximum reinforcement steel yield strength is 75 ksi	40 ksi	AISC I1.1.3.2	Pass
Deck nominal rib height does not exceed 3"	3 in	AISC I3.2c.1.1	Pass
Shear stud diameter does not exceed 0.75"	0.75 in	AISC I3.2c.1.2	Pass
Studs extend not less than 1.5" above top of steel deck	1.5 in	AISC I3.2c.1.2	Pass
Concrete cover c_c is minimum 0.5" above top of studs	1.5 in	AISC I3.2c.1.2	Pass
Concrete slab thickness is at least 2" above steel deck	3 in	AISC I3.2c.1.3	Pass
Stud diameter not greater than 2.5x thickness of metal to which it is welded (t_s)	0.75 in	AISC I8.1	Pass
The height of steel studs is $\geq 4d_s$ after installation	4.5 in	AISC I8.2	Pass
Minimum Spacing Requirements	4.19 in	AISC I8.2d	Pass
Maximum Spacing Requirements (8 x Total Slab Thickness or 36 inches)	4.19 in	AISC I8.2d	Pass

Moment Values

The moment values and diagrams used in the composite beam checks may vary from those displayed in the model. In the model, because the presence of the metal deck can cause the distribution of moment to be continuous at the support, it is possible that negative moment values are exhibited at pinned supports. While these values are generally small, in the composite beam design module, these small negative moments are removed and the moment values are shifted slightly to compensate for these values. Therefore, the strength design and the distribution of studs are based on an idealized moment diagram where moments at the pinned supports are zero.

Displacement Values

The displacement values used for serviceability checks are relative displacement values.

Stud Design

The design of the number of studs and layout of the studs along a member is prepared for the current cross-section that exists in the model at the time the composite beam design is executed.

Number of studs

Before any attempt to position the studs along the composite member, the program first determines the number of studs needed in the positive and negative moment regions based on the target degree of composite action. For the number of studs needed in the positive moment region, the minimum of the compressive strength of the concrete and the compressive strength of the steel section is multiplied by the target degree of composite action, and divided by the shear strength of a single stud. This number is then rounded up to determine the number of studs between zero and maximum positive moment.

$$N_{studs+} = RoundUp(\text{Min}(C_{steel}, C_{con}) * TargetCompositeAction / Q_n)$$

For the number of studs needed in the negative moment region, the minimum of the tensile strength of the steel reinforcement and the tensile strength of the steel section is multiplied by the target degree of composite action, and divided by the shear strength of a single stud. This number is then rounded up to determine the number of studs between zero and maximum negative moment.

$$N_{studs-} = RoundUp(\text{Min}(T_{steel}, T_{reinf}) * TargetCompositeAction / Q_n)$$

Number of Rows of studs

The program allows studs to be designed in either one or two rows placed on the flange, perpendicular to the length of the beam. To allow two rows of studs, set the value of "Number of Rows" in the setup to "2". If two rows are allowed, the program will first check to see if two rows are allowed on the section. Two rows are allowed if the spacing between the studs is greater than or equal to four times the stud diameter, with a distance to the edge of the flange of at least 1 inch or one times the stud diameter, whichever is greater. If two rows can be used, the program will design the studs with two rows only if needed based on the minimum spacing requirements.

Stud Layout: Uniformly Loaded Simple Span Members

For members which are uniformly loaded and have pinned ends, where the moment along the member is positive and the maximum moment occurs at or close to the mid-span, the number of studs needed between zero and maximum moment is doubled to attain the total number of studs. These studs are uniformly distributed along the length of the member. The number of studs will be increased to meet maximum spacing requirements if needed.

Stud Layout: Segmented Members

Members which support point loads, such as girders, or have both positive and negative moment, such as fixed-end members, may require an uneven distribution of studs in order to develop the required strength at different locations along the member. To do this, the program divides the member into segments and sub-segments. Any transition from negative to positive moment, or vice versa, initializes a new segment on the member. Any point load will initialize a new sub-segment within the segment in which it is found.

Members which have a segmented stud layout are also given a uniform stud layout. This uniform layout uses the minimum stud spacing of all segments and sub-segments and determines the number of studs needed along the entire beam assuming this spacing.

Members with Both Negative and Positive Moment

For members with both negative and positive moment, the number of studs in a positive region segment is based directly on the number of studs needed to develop the positive moment capacity from zero to maximum moment as determined by the target degree of composite action. Similarly, the number of studs in the segment for the negative region is based directly on the number of studs needed to develop the negative moment capacity from zero to maximum moment as determined by the target degree of composite action. If the segment is one in which the moment value increases from zero to a maximum moment and then decreases back to zero, the number of studs in the segment is equal to two times the number of studs required from zero to maximum moment.

Members with Point Loads

For members which support point loads, the number of studs needed in each sub-segments is determined by the following equation:

$$N_{seg} = N_{studs} * \frac{M - M_s}{M_c - M_s}$$

Where:

N_{seg} = Number of studs required in the sub-segment

N_{studs} = Number of studs required between zero and maximum moment

M = Difference between the maximum and minimum moment in the sub-segment

M_s = Moment resistance of steel beam alone

M_c = Moment resistance of the composite beam at the target degree of composite action

After the number of studs required in each sub-segment has been calculated, the percentage of studs required in each sub-segment relative to the total number of studs required in all sub-segments is used to distribute the studs according to the total number of studs in the segment as determined by the target degree of composite action on the member. This ensures that the total number of studs on the member matches the total number of studs expected according to the degree of composite action. In cases where the member is over-designed, the number of studs in each segment will be greater than required. In cases where the beam size and target degree of composite action have already been optimized, the number of studs in each segment will be equal to or close to the number required.

Camber Design

The design of camber (when "Camber Definition" is set to "Design") is executed for the current cross-section that exists in the model at the time the calculation is run.

1. Serviceability checks which consider camber are first performed by adding no camber. If these serviceability checks pass, the design ends, as no camber is needed for the member.
2. If any of the serviceability checks which consider camber fail, a minimum camber of 0.75 inches is added and the checks are performed again. If the checks fail again, camber is added in 0.25 inch intervals until the serviceability checks pass or until the maximum camber value is reached.

Auto-Design

If "Calculation Approach" is set to "Design" in the composite setup, an optimized cross-section and degree of composite action for each cross-section group can be determined. The auto-design procedure is executed based on the "Design Approach" also found in the setup. A maximum height restriction can also be placed on the designed member in the setup.

Procedure

The procedure used to select the optimum cross-section and degree of composite action is as follows:

1. The maximum internal forces and deflections for each stage are determined for the initially modelled member using the cross-section and target degree of composite action specified in the model. Deflection constants, which are used to predict deflections using other cross-sections and target degrees of composite action subsequently in the auto-design process, are also calculated. These deflection constants are calculated as follows:
 - a. For construction stage deflection:
 - i. Deflection Constant = (Deflection at construction stage) / (Moment of Inertia of Cross-Section)
 - b. For final stage deflections:
 - i. Deflection Constant = (Deflection at final stage) / (Lower Bound Moment of Inertia of Cross-Section)

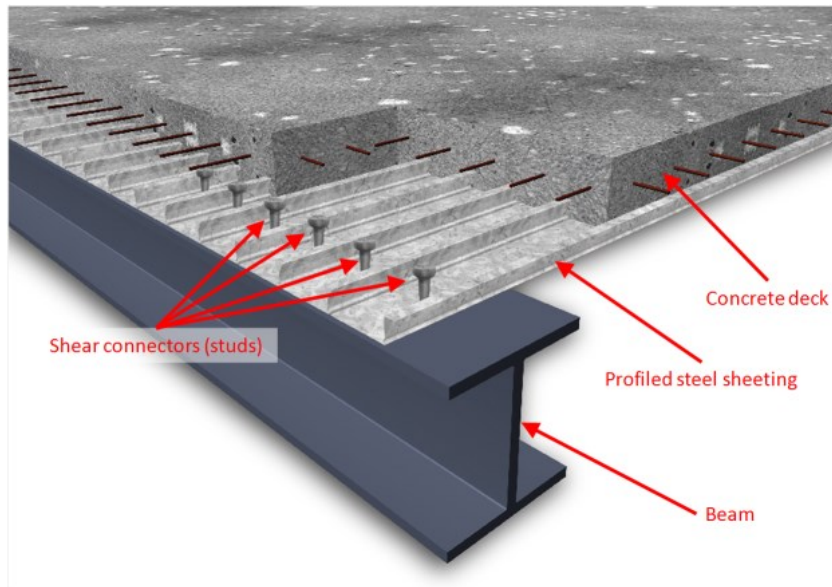
-
2. A table of cross-sections of the same type as the current member [e.g. W(Imp)] is created and sorted by weight. If a height restriction is imposed, only cross-sections with a height less than the maximum set by the user in the setup are included in this table.
 3. A target degree of composite action based on the design approach is set as follows:
 - a. For the "Optimize Beam" approach, a target degree of composite action of 100% is used.
 - b. For the "Optimize Studs" approach, a target degree of composite action of 30% is used.
 - c. For the "Balanced" approach, a target degree of composite action of 55% is used.
 - d. For the "User Defined" approach, the target degree of composite action entered by the user in setup is used.
 4. All checks (listed in the "Strength and Serviceability Checks" section) are performed on the initially modelled member (i.e. the one that exists in the model when the check/design is executed) using the target degree of composite action. For deflection checks, if camber is set to an absolute or relative value, the value of camber specified by the user is included in the check. If camber is set to "design", then the maximum camber value set by the user is included in the deflection check and the final camber value will be designed according to the procedure described in "Camber Design" section above.
 5. Optimization of Cross-Section: If the initially modelled member fails to pass any of the checks, the next strongest cross-section in the table (i.e. a cross-section which has a plastic section modulus higher than the current member) becomes the current member and all checks are performed on it in the same way the initially modelled member was checked (see Step 4). The program continues to iterate through stronger cross-sections until a cross-section is found which passes all checks at the target degree of composite action. If the initially modelled member passes all checks, the first cross-section in the table (i.e. the cross-section with the least weight) becomes the current member. If this member fails, the program iterates through stronger cross-sections until a cross-section is found which passes all checks. The end result is a cross-section which is optimized to meet the design requirements
 - a. Notes on the cross-section selection process:
 - i. The plastic section modulus is used to determine a stronger cross-section since the bending check is the one which generally governs the strength checks
 - ii. In cases where all strength and serviceability checks pass for the cross-section being considered, but the number of studs required at the target degree of composite action does not allow minimum spacing requirements to be met, the program will attempt to lower the target degree of composite action as low as possible before iterating to the next strongest cross-section. This ensures the minimum spacing requirement is met while preventing the cross-section from becoming over-designed. This case is most common when the design approach is set to "optimize beam," which uses a target degree of composite action of 100%.
 - iii. The deflections used in the serviceability checks are predicted by multiplying the deflection constants (See Step 1 above) by the moment of inertia and lower bound moment of inertia for the construction and final stages, respectively.
 6. Optimization of Degree of Composite Action: Once a cross-section has been found which meets all strength and serviceability requirements at the target degree of composite action determined by the design approach, the program then attempts to decrease the target degree composite action as low as possible using the bi-section method.
 7. If a new cross-section and degree of composite action can be found which meet design requirements, they can be used to update the model. If no cross-section or degree of composite action are found, a message is displayed in the detailed output. The most common reason a cross-section may not be found is if the height restriction is enabled, thus restricting the number of cross-sections that can be considered in the design. A second reason a section may not be found is due to minimum spacing requirements. In this case consider allowing 2 rows of studs on the member (see "Composite Setup").

Theoretical Background: Composite Analysis Model

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 modeled 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.



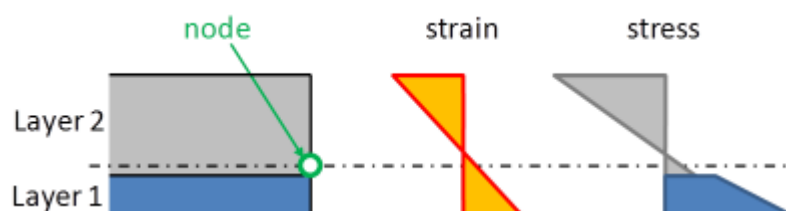
Composite deck

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 behavior 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, the profiled steel sheeting, and is mostly used for light-weight roofs.

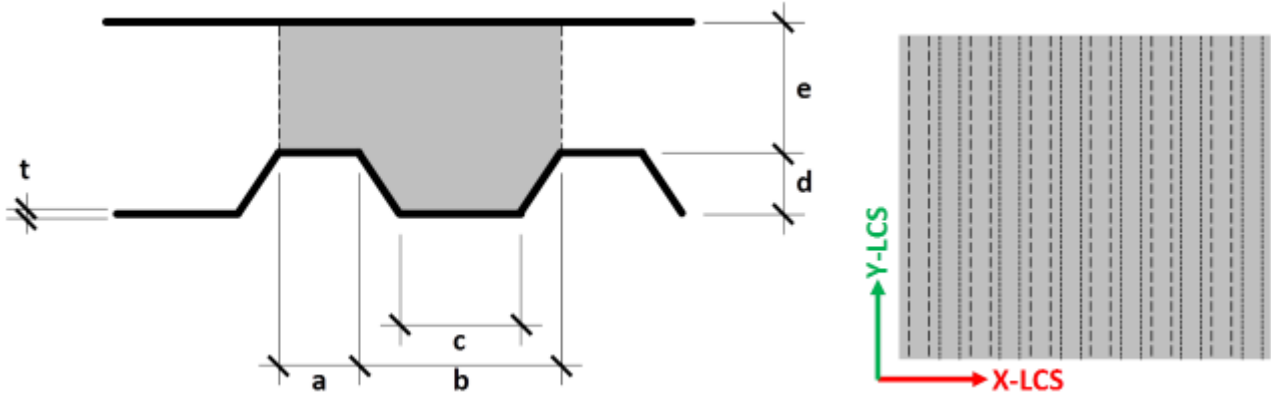
A composite deck is modeled as a multi-layered plate. Each layer has orthotropic properties and the eccentricity of each layer is taken into account.

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 reasonable for the longitudinal (i.e. the direction parallel to the corrugation) behavior of the composite deck. 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” behavior 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 behavior of the composite deck. That approximation is therefore acceptable.

Profiled steel sheeting



Where E_s , G_s , ν_s are Young's modulus, shear modulus and Poisson's ratio of steel, respectively.

Where 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

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

$$d'_{s,11} = \begin{cases} \text{if } d > 0 : E_s \cdot t \cdot \min \left[\frac{\left(\frac{t}{d}\right)^2}{3 \frac{1-k}{1-D/l} + 2 \cdot k}; 1 \right] \\ \text{if } d = 0 : E_s \cdot t \end{cases} \quad d'_{s,22} = E_s \cdot t \cdot k \quad d'_{s,12} = 0 \quad d'_{s,33} = \frac{E_s \cdot t}{2 \cdot k \cdot (1 + \nu_s)}$$

Mean thickness (for calculation of self-weight)

$$H_s = k \cdot t$$

Position of gravity center from bottom fiber (assumed for both directions)

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

Auxiliary variables

$$B = a + b$$

$$D = \frac{b-c}{2}$$

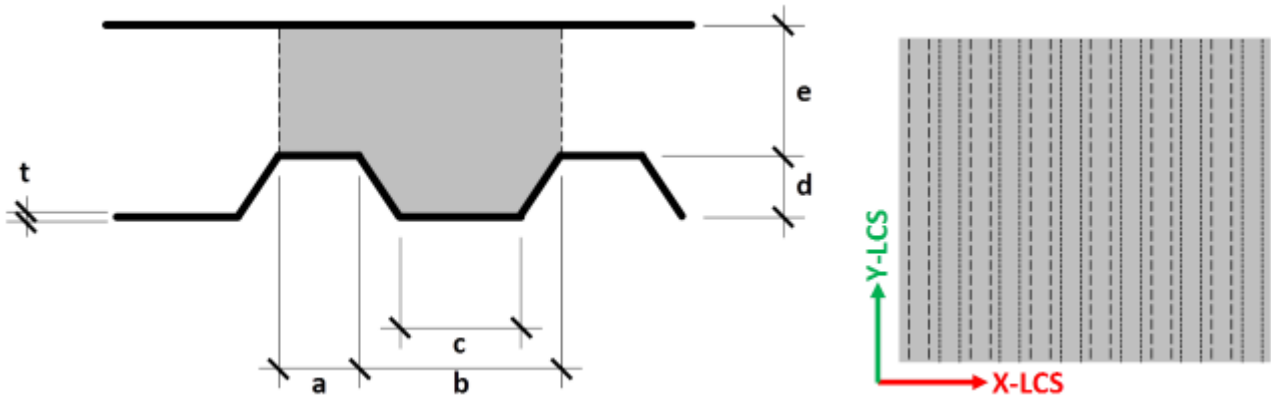
$$l = \sqrt{D^2 + d^2}$$

$$L = a + c + 2 \cdot l$$

$$k = \frac{L}{B}$$

$$I = t \cdot \left[a \cdot (d - z_{G,s})^2 + c \cdot z_{G,s}^2 + l \cdot \left(\left(\frac{d}{2} - z_{G,s} \right)^2 + \frac{d^2}{6} \right) \right]$$

Concrete deck



Where E_c , G_c , ν_c are Young's modulus, shear modulus and Poisson's ratio of concrete, respectively.

Where 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

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

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

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

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

Membrane components

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

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

Mean thickness (for calculation of self-weight)

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

Position of gravity center from bottom fiber (assumed for both directions)

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

Auxiliary variables

$$h = d + e$$

$$A_1 = B \cdot e$$

$$A_2 = C \cdot d$$

$$B = a + b$$

$$z_{G1} = d + \frac{e}{2}$$

$$z_{G2} = \frac{d}{3} \cdot \frac{2 \cdot b + c}{b + c}$$

$$C = \frac{b + c}{2}$$

$$I_1 = \frac{B \cdot e^3}{12}$$

$$I_2 = \frac{d^3}{36} \cdot \frac{b^2 + c^2 + 4 \cdot b \cdot c}{b + c}$$

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

Multi-layered orthotropy

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

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

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

$$R_3 = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix}$$

$$R_2 = \begin{bmatrix} c & s \\ -s & c \end{bmatrix}$$

Where

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

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

$$D_i = R_3^T \cdot D'_i \cdot R_3 \quad G_i = R_2^T \cdot G'_i \cdot R_2 \quad d_i = R_3^T \cdot d'_i \cdot R_3$$

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

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

Plate behavior components

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

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

Membrane behavior 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).

One-way decks

It is a common assumption to consider composite decks and metal decks as one-way plates. In the case of an orthotropic plate, implementing such a behavior also helps reducing "disturbances" due to the two-way behavior, as that typically produces results that are very different from hand calculation and are hence difficult to validate intuitively.

The one-way behavior will be introduced in the combined orthotropy matrix by reducing the bending stiffness components in the non-supporting direction, i.e. dividing the corresponding components by a reduction coefficient k_{one} .

It is however important to note, that the correction must be applied to the combined matrix - not to the individual layers. In order to do that, the relevant sub-matrix must be extracted before applying the correction. This is done in several steps, as follows:

1. Extract sub-matrices from the resulting orthotropy matrix

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

$$\begin{bmatrix} \overline{D}_{11} & \overline{D}_{12} & \overline{D}_{13} & & & \overline{S}_{11} & \overline{S}_{12} & \overline{S}_{13} \\ & \overline{D}_{22} & \overline{D}_{23} & & & \text{sym} & \overline{S}_{22} & \overline{S}_{23} \\ & & \overline{D}_{33} & & & \text{sym} & \text{sym} & \overline{S}_{33} \\ \hline & & & \overline{G}_{11} & \overline{G}_{12} & & & \\ & & & & \overline{G}_{22} & & & \\ \hline & \text{sym} & & & & \overline{d}_{11} & \overline{d}_{12} & \overline{d}_{13} \\ & & & & & & \overline{d}_{22} & \overline{d}_{23} \\ & & & & & & & \overline{d}_{33} \end{bmatrix} = \begin{bmatrix} \overline{D} & & \overline{S} \\ & \overline{G} & \\ \overline{S}^T & & \overline{d} \end{bmatrix}$$

2. Calculate the resulting eccentricity for each component of the matrix

$$\overline{z}_{ii} = \frac{\overline{S}_{ii}}{\overline{d}_{ii}} \quad i = 1, 2, 3$$

$$\overline{z}_{ij} = \sqrt{\overline{z}_{ii} \cdot \overline{z}_{jj}} \quad i, j = 1, 2, 3 \quad i \neq j$$

3. Calculate the bending stiffness due to the eccentricity of the membrane components

$$\overline{D}_{m,ij} = \overline{d}_{ij} \cdot \overline{z}_{ij}^2 \quad i, j = 1, 2, 3$$

4. Calculate the pure bending part of the bending components

$$\overline{D}_{b,ij} = \overline{D}_{ij} - \overline{D}_{m,ij} \quad i, j = 1, 2, 3$$

5. Backward rotation, back to the original axes of the steel sheeting

$$\widehat{D}_b = [\mathbf{R}_3^{-1}]^T \cdot \overline{D}_b \cdot \mathbf{R}_3^{-1}$$

$$\widehat{G} = [\mathbf{R}_2^{-1}]^T \cdot \overline{G} \cdot \mathbf{R}_2^{-1}$$

Note: the rotation matrices are the same as defined before in the Multi-layered orthotropy section.

6. Apply the stiffness reduction coefficient

$$\widehat{D}_{b,one} = \begin{bmatrix} \widehat{D}_{b,11} & \frac{\widehat{D}_{b,12}}{k_{one}} & \frac{\widehat{D}_{b,13}}{k_{one}} \\ & \frac{\widehat{D}_{b,22}}{k_{one}} & \frac{\widehat{D}_{b,23}}{k_{one}} \\ \text{sym} & & \frac{\widehat{D}_{b,33}}{k_{one}} \end{bmatrix}$$

$$\widehat{G}_{one} = \begin{bmatrix} \widehat{G}_{11} & \frac{\widehat{G}_{12}}{k_{one}} \\ \text{sym} & \frac{\widehat{G}_{22}}{k_{one}} \end{bmatrix}$$

7. Forward rotation, to the final orientation of the 2D member

$$\overline{D}_{b,one} = \mathbf{R}_3^T \cdot \widehat{D}_{b,one} \cdot \mathbf{R}_3$$

$$\overline{G}_{one} = R_2^T \cdot \widehat{G}_{one} \cdot R_2$$

Note: the rotation matrices are the same as defined before in the Multi-layered orthotropy section.

8. Re-inject the eccentric part of the bending stiffness (calculated in step 3)

$$\overline{D}_{one} = \overline{D}_{b,one} + \overline{D}_m$$

9. Re-combine the orthotropy matrix

$$D_{one} = \begin{bmatrix} \overline{D}_{one} & & \overline{S} \\ & \overline{G}_{one} & \\ \overline{S}^T & & \overline{d} \end{bmatrix}$$

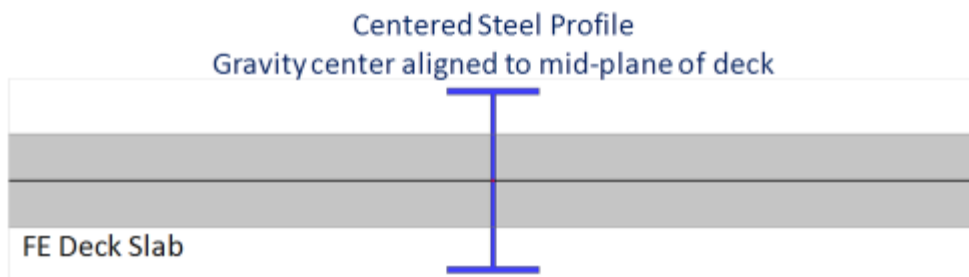
Important: some correction must also be applied in the calculation of the stiffness of the composite beam. Those will be taken into account in the following section.

Composite beam

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

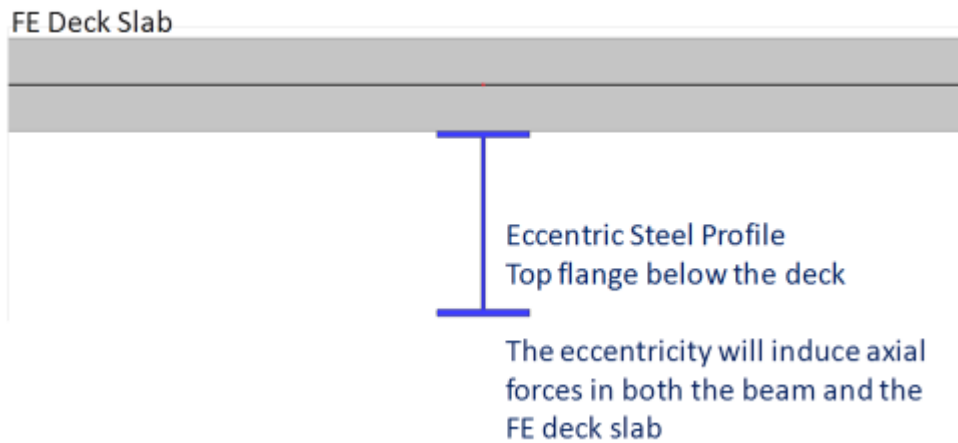
Without composite action

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 modelled by a plate rib without eccentricity



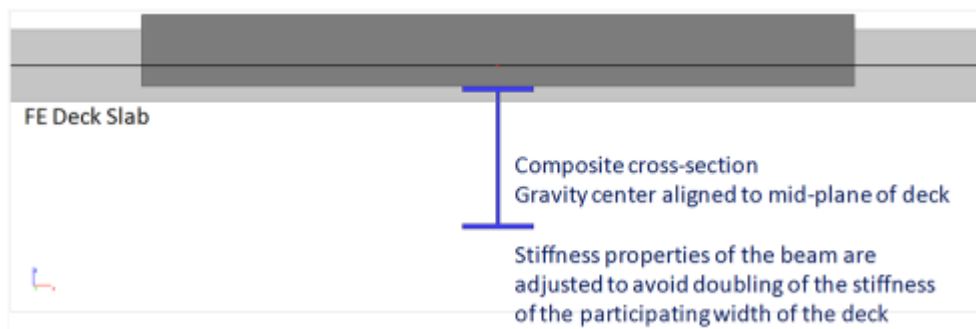
With advanced composite action

In this case, a perfect shear connection is assumed between the beam and the deck. The plate rib is modeled 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 modeled 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:

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

$$A_{eq,c} = \frac{d''_{c,11}}{E_b} \cdot b_d$$

$$A_{eq,s} = \frac{d''_{s,11}}{E_b} \cdot b_d$$

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

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

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

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

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

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

$$I_{y,adj} = I_{y,b} + I_{eq,c} \cdot \left(1 - \frac{1}{k_{onc}}\right) + A_b \cdot (z_b - z_G)^2 + A_{eq,c} \cdot (z_{G,c} - z_G)^2 + A_{eq,s} \cdot (z_{G,s} - z_G)^2$$

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 $d''_{c,33}=0$
$D''_{c,11}$	bending 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 $D''_{c,11}=0$
$d''_{s,11}$	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
$D''_{s,11}$	bending stiffness component of the orthotropy matrix of the profiled steel sheeting in the direction of the beam axis

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

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

$$d''_i = \begin{bmatrix} d''_{i,11} & d''_{i,12} & d''_{i,13} \\ \text{sym} & d''_{i,22} & d''_{i,23} \\ & & d''_{i,33} \end{bmatrix} = R_3^T \cdot d'_i \cdot R_3 = R_3^T \cdot \begin{bmatrix} d'_{i,11} & d'_{i,12} & 0 \\ \text{sym} & d'_{i,22} & 0 \\ & & d'_{i,33} \end{bmatrix} \cdot R_3$$

$$R_3 = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix}$$

$$c = \cos \theta$$

$$s = \sin \theta$$

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 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):

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

Notes:

- 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 negligible 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:

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

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

Definition of $I_{y,b}$, $I_{y,adj}$, $I_{eq,c}$, $I_{eq,s}$: 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 behaviors 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.

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

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).