Composite behavior FAQ

This page is devoted to frequently asked questions (FAQ) related to Composite behavior.

On this page:

Modeling

How do I model an object with different tensile and compressive stiffness relationships?

Extended Question: I am modeling a composite brace in which a standard steel shape is enclosed within concrete. I would like only the steel to carry tension, while both the concrete and the steel carry compression. How do I achieve this behavior during a linear-dynamic analysis?

Answer: Rather than using a frame object, you can simulate this behavior by drawing a two-joint multilinear elastic link. Within the link definition, tensile and compressive force-displacement relationships may be drawn differently, and may follow any monotonic pattern desired.

NOTE: Because a nonlinear material relationship is assigned, nonlinear analysis must be run to capture the effect of different tensile and compressive stiffness relationships.

See Also:
- CSI Analysis Reference Manual (Chapters 14 and 15)
- CSI Watch & Learn video

How are partially composite sections modeled?

Extended Question: For a concrete-deck over steel-girder section that is 85% composite, will the assignment of a shell-object membrane-stiffness modifier of 0.85 simulate 85% composite action?

Answer: Given this design, composite-section stiffness is a function of three sources, which include:

1. Stiffness of the girder about its own center of gravity.
2. Stiffness of the deck about its own center of gravity.
3. Additional girder and deck stiffness contributions about the center of gravity of the entire section.

Changing the deck membrane modifier to 0.85 would directly affect the 1st source of stiffness, and indirectly affect the 3rd source. However, 85% composite action would allow some slip between deck and girder, therefore only the 3rd source of stiffness should be affected.

In a detailed model that explicitly models the deck and the girder, the 3rd source of stiffness may be reduced using flexible links (rather than rigid or fixed) to connect the girder to the deck. The stiffness derivation for these flexible links would be based on model discretization and the prescribed 85% composite action.

Further, if a single frame object is used to model the composite section, the stiffness modifiers would need to be derived for the entire section.

Is there any difference between the modeling of composite sections in SAP2000 and ETABS?

Answer: Composite-section modeling and analysis is similar between SAP2000 and ETABS. One additional capability of ETABS is that the software can design the objects within a composite section.

Noncomposite action

How is noncomposite action modeled for frame-object girders and shell-object slabs?

Answer: When common joints connect a girder and a slab, composite action occurs. The slab may be offset from the girder using joint offsets and frame in section points, as described further in the Composite section tutorial. Composite action may also result when girders and slabs do not share common joints, but instead are connected through full-body constraints which connect girder joints to certain corresponding slab joints.

Noncomposite action results when the girder and slab both individually contribute to stiffness properties. Slip is allowed along the slab-girder interface. To model noncomposite action, the girder and slab should not share common joints, but they should be connected such that corresponding joints share the same vertical deflection. For example, a horizontal girder connected to a noncomposite slab may be modeled using the equal Z constraint.

A condition where the slab does not contribute to stiffness may be modeled using property modifiers where flexural and axial stiffness are reduced in the direction parallel to girders. Very small values on the order of 1e-3 are recommended to avoid the numerical problems which may result from zero-modifier application. To model the stiffness contribution from forms, property modifiers may also be applied in the direction perpendicular to girders.
**Composite response**

**Why is unexpected response generated for a composite section?**

**Extended Question:** I have two comparable models, described below, but the deflection of Model 1 is less than expected. Is there an explanation?

- **Model 1:**
  - Composite section with steel-girder frames and concrete-deck shells.
  - U1, U2, and U3 restraints are assigned to either end of the girders.

- **Model 2:**
  - A single composite-section frame object created in the Section Designer.

**Answer:** The unexpected results of Model 1 may be attributed to its boundary conditions. The longitudinal restraints at either end of the girder are not located at the cross-section centroid, causing longitudinal force to act along a moment arm extending from the neutral axis. This behavior induces an additional moment which influences flexural response and vertical displacement. While kinematically correct, this behavior is different from that of Model 2 because of these boundary conditions.

The resultant axial force is the net effect of longitudinal stresses acting along the composite section. When longitudinal restraints are higher in the cross section, the neutral axis moves upward as well. To maintain equilibrium with the tensile forces located below the neutral axis (now greater because a larger portion of cross section is below the neutral axis), compressive forces are introduced at the restraints. These axial forces may be released with the assignment of a roller support at either end. This would allow axial force to transfer into longitudinal displacement.

**What causes jumps in the moment diagram of a composite frame?**

**Answer:** The deck and girders of a composite section are connected through common joints. During composite action, forces are transferred from the deck to the girders through these joints. These forces, concentrated at the connection points, cause jumps in the moment diagram. Discontinuities may be reduced by refining the discretization of frame and shell objects such that girder and deck connection points have closer spacing.

**Design forces**

**How are design forces obtained for a composite section modeled using frame and shell objects?**

**Extended question:** I modeled a composite reinforced-concrete T-beam floor system using finite elements for the slab and frame objects for the girders. I found that member forces cannot be directly read for the design process. How are member forces obtained for design?

**Answer:** Design of a T-beam floor system is dependent upon the forces within both the girder and the tributary slab width. Design forces are derived as a combination of those within the frame and shell objects which compose the composite system.

Design forces in may be obtained using either of the following methods:

- Create a sequence of section cuts, as described on the Section cut FAQ page, to obtain design forces. Done manually, this process may require significant effort. A more practical approach may be to automate the process using the Application Programming Interface (API). Please note that discretization should be refined as necessary to adequately define the section cuts.

- Obtain design forces by replacing rectangular beams with T-Beam sections, then use property modifiers to modify adjacent shell objects such that they do not contribute the same stiffness and weight as the T-beams (not applicable to SAFE program as weight of slab/T-beam flange overlap is not double counted in SAFE). Frame forces in the T-beams would then directly correspond with composite-section design forces.

**Composite design**

**How are composite and noncomposite sections designed in the same model?**

**Extended Question:** I am using auto selection to design a steel structure with both composite and noncomposite sections. Composite design returns the correct results, but the noncomposite design for the gravity system does not generate results except for columns. Noncomposite member check is shown as composite design. How is this resolved?

**Answer:** The design procedure may be changed by selecting the appropriate members, then switching from Composite Beam Design to Steel Frame Design through Design > Overwrite Beam Design.