Frame FAQ

This page is devoted to **frequently asked questions** (FAQ) related to frame elements.

On this page:

Is a truss object available for modeling?

Answer:

CSI Software seeks to use fewer objects which serve a greater variety of purposes. There is no feature specifically for truss systems, though users may model a truss by assigning end releases to frame objects. This, however, is not recommended.

NOTE: True truss behavior rarely occurs. During hand calculations, it is often convenient to idealize sufficiently slender objects, dominated by axial-force contribution, as truss members. However, at least some moment will occur due to member self-weight, which may lead to numerical problems if moment releases are assigned to both ends. Further, given the computational power of computer systems, simplification is not necessary.

How are nonprismatic frame sections created?

Answer: The process for creating nonprismatic frame sections is described in the Nonprismatic frame objects article. To summarize the process, users should select Define > Section > Properties > Frame Sections, then select Add New Property > Other for the frame-section property type. Select the Nonprismatic icon to open the Nonprismatic Section Definition menu in which multiple segments may be defined to characterize the taper of each member. Nonlinear sections may vary in a linear, parabolic, or cubic relation.

Can nonprismatic sections be created using the Section Designer?

Answer: Yes, sections created using the Section Designer may be nonprismatic. If auto meshing is assigned to the nonprismatic frame object, the section properties at intermediate locations are obtained through interpolation functions based on section properties at ends *i* and *j*.

Please note that the extruded view will accurately display the nonprismatic Section Designer section when the points at either end can be mapped one-to-one. Otherwise, the extruded view will be plotted using rectangular sections. Either way, analysis results will not be affected by display settings.

Why is the actual shape of the extruded member not shown?

Answer: Nonprismatic and extruded members created using the Section Designer are displayed as the rectangular bounding box which encloses the object.

How are nonprismatic precast-concrete sections modeled?

Answer: A nonprismatic precast sections may be modeled by subdividing a frame object into multiple segments, then assigning the properties of an equivalent I-girder to each segment.

How are tension-only and compression-only frame objects modeled?

Answer: For response, please see the Tension-only objects in ETABS article.

Can frame end forces be reported in an arbitrary coordinate system?

Extended Question: I am interested in recovering frame end forces which are resolved in an arbitrary coordinate system oriented at an angle from the member longitudinal axis. These internal forces would be displayed in a plane as though the object were cut at an angle intersecting its end. I understand that this can be done given member restraints, but how is this done without joint restraints?

Answer: Frame end forces may be recovered in an arbitrary coordinate system using the following process:

- At the end of the element for which forces are to be reported, modify the local joint coordinate system; then
- Select Display > Show Tables > Analysis Results > Element Output > Frame Output > Table: Element Joint Forces - Frames to recover forces oriented in the arbitrary coordinate system, which are then given as Element Joint Forces.

If, however, it is not desirable to modify the local joint coordinate system, it may be best to recover default frame forces and then post-process the results outside of SAP2000.

Is it possible to obtain coordinates for auto-meshed joints?

Answer: Yes, this information is available upon completion of analysis through Display > Show Tables > Analysis Results > Element Output > Objects and Elements > Table: Objects and Elements - Joints. In this table, joints generated through auto-meshing have a tilda (~) prefix. The point-object coordinates are presented, along with the name of the point object from which the joint element was created. If the joint object was not created from a point object, but instead was created as a result of meshing procedures internal to the software, then the corresponding JointObject entry is blank.

Why am I getting different forces at the object start joint and initial station?

Answer: The frame internal forces are computed at the centroid of the member, but the frame joint forces are reported at the joint, which is may be offset from the centroid based on the insertion point used. For example, when the insertion point is specified at cardinal point 8 (top center), the moments will differ by the axial force multiplied by the distance from the centroid to the top of the member. This is expected and correct. The joint forces are intended to represent the effect of the member on the rest of the structure through the joint. The internal forces are meant to represent the behavior in terms of the basic cross-sectional measures.