

# Joint

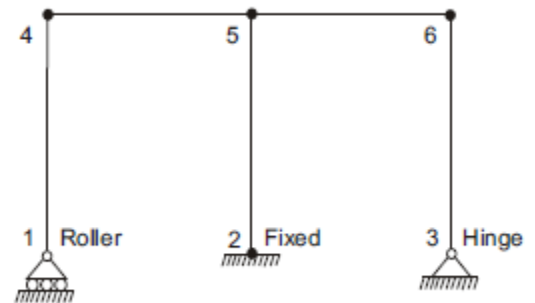
**Joints** represent points of connection between structural members. Joints are automatically created at the ends of line objects ([frames](#), [cables](#), [tendons](#), [links](#)) and at the corners of [area](#) and [solid](#) objects. Joints may also be added manually to capture localized behavior, or to simulate [restraint](#) supports. [Constraint](#) conditions are also applied to joints to establish correlation among their displacement DOF.

Joints are fundamental to numerical solution in that stresses and strains are resolved according to joint displacement. First, an object-based model is created by drawing structural objects which are connected by joints and subjected to loading.

CSI Software then generates a [finite-element](#) model according to structural geometry and individual-member properties. Users have additional control over [mesh](#) refinement and intersecting-object connections. The FEM model then provides for analysis, and results are reported in terms of the object-based model.

Element [mass](#) and distributed loading are automatically transferred to joint locations during analysis. Users may assign additional mass to joints. During dynamic analysis, users should also assign mass moment of inertia (MMI) to joints such that inertial forces correlate with joint [Acceleration](#). MMI formulation is presented in the

[CSI Analysis Reference Manual](#) (page 41), and guidelines for dynamic-analysis acceleration are presented in the [Time-history output-acceleration accuracy](#) article.



## Related Content

### Articles

### Tutorials

- [Complicated joint patterns](#)
- [Joint renumbering](#)
- [Joint-pattern first steps](#)
- [Modeling a pin connection between crossing members](#)
- [Radial point load](#)

### Test Problems

- [Frame to shell connections](#)
- [Modeling segmental-bridge joint openings](#)