

SolidWorks: Sketch Relations

The following table describes the entities that you can select for a relation and the characteristics of the resulting relation.

Relation	Entities to select	Resulting relations
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the same infinite line.
Conradial	Two or more arcs.	The items share the same centerpoint and radius.
Perpendicular	Two lines.	The two items are perpendicular to each other.
Parallel	Two or more lines. A line and a plane (or a planar face) in a 3D sketch.	The items are parallel to each other. The line is parallel to the selected plane.
ParallelYZ	A line and a plane (or a planar face) in a 3D sketch.	The line is parallel to the YZ plane with respect to the selected plane.
ParallelZX	A line and a plane (or a planar face) in a 3D sketch.	The line is parallel to the ZX plane with respect to the selected plane.
AlongZ	A line and a plane (or a planar face) in a 3D sketch.	The line is normal to the face of the selected plane.
<p>Relations to the global axes are called AlongX, AlongY, and AlongZ. Relations that are local to a plane are called Horizontal, Vertical, and Normal.</p>		
Tangent	An arc, ellipse, or spline, and a line or arc.	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same centerpoint.
Midpoint	Two lines or a point and a line.	The point remains at the midpoint of the line.
Intersection	Two lines and one point.	The point remains at the intersection of the lines.
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.

Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.
Equal Curvature	Two splines	The radius of curvature and the vector (direction) matches between the two splines.
Symmetric	A centerline and two points, lines, arcs, or ellipses.	The items remain equidistant from the centerline, on a line perpendicular to the centerline.
Fix	Any entity.	The entity's size and location are fixed. However, the end points of a fixed line are free to move along the infinite line that underlies it. Also, the endpoints of an arc or elliptical segment are free to move along the underlying full circle or ellipse.
Pierce	A sketch point and an axis, edge, line, or spline.	The sketch point is coincident to where the axis, edge, or curve pierces the sketch plane. The pierce relation is used in Sweeps with Guide Curves.
Merge Points	Two sketch points or endpoints.	The two points are merged into a single point.

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- When you create a relation to a line, the relation is to the infinite line, not just the sketched line segment or the physical edge. As a result, some items may not touch when you expect them to.
 - When you create a relation to an arc segment or elliptical segment, the relation is actually to the full circle or ellipse.
 - If you create a relation to an item that does not lie on the sketch plane, the resulting relation applies to the projection of that item as it appears on the sketch plane.
 - Additional relations are created automatically when you use Offset Entities and Convert Entities.
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