

Sketch Relations

Description: Sketch relations help to define a sketch entity by relating them or constraining them to other sketch entities. Sketch relations can either be added or assumed by SolidWorks through built in features (the polygon tool).

Note: All relations can be seen and edited by using the display/delete relation icon. Selecting multiple entities will make the property manager window pop up and will display the relations between the selected entities (**ctrl + left click** selects multiple entities).



Sketch entities such as a line or point will remain horizontal with respect to the current coordinate axis.



Sketch entities such as a line or point will remain vertical with respect to the current coordinate axis.



Two or more sketch entities will remain parallel to eachother.



Sketch entities will remain 90 degrees from eachother.



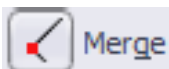
Two or more sketch entities will lie on the same line.



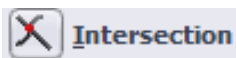
A point or vertex will lie on a line, arc, or ellipse.



A point or vertex will remain at the midpoint or center of a line.



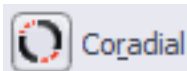
Combines two sketch points.



A point will remain at the intersection of two lines.



Two circles or arcs will share a common centerpoint.



Two circles or arcs share a common centerpoint and radius.



Two sketch entities will remain tangent to eachother. For example a line that

intersects an arc or circle at a single point.



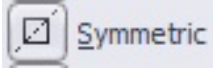
Equal

Set two entities to have the same length or radius. *Note: Equal entities will update when the major dimension is changed.*



Fix

The entities size and location are fixed in space.



Symmetric

Two sketch entities such as lines, arcs, or ellipses remain equidistant to a centerline.
