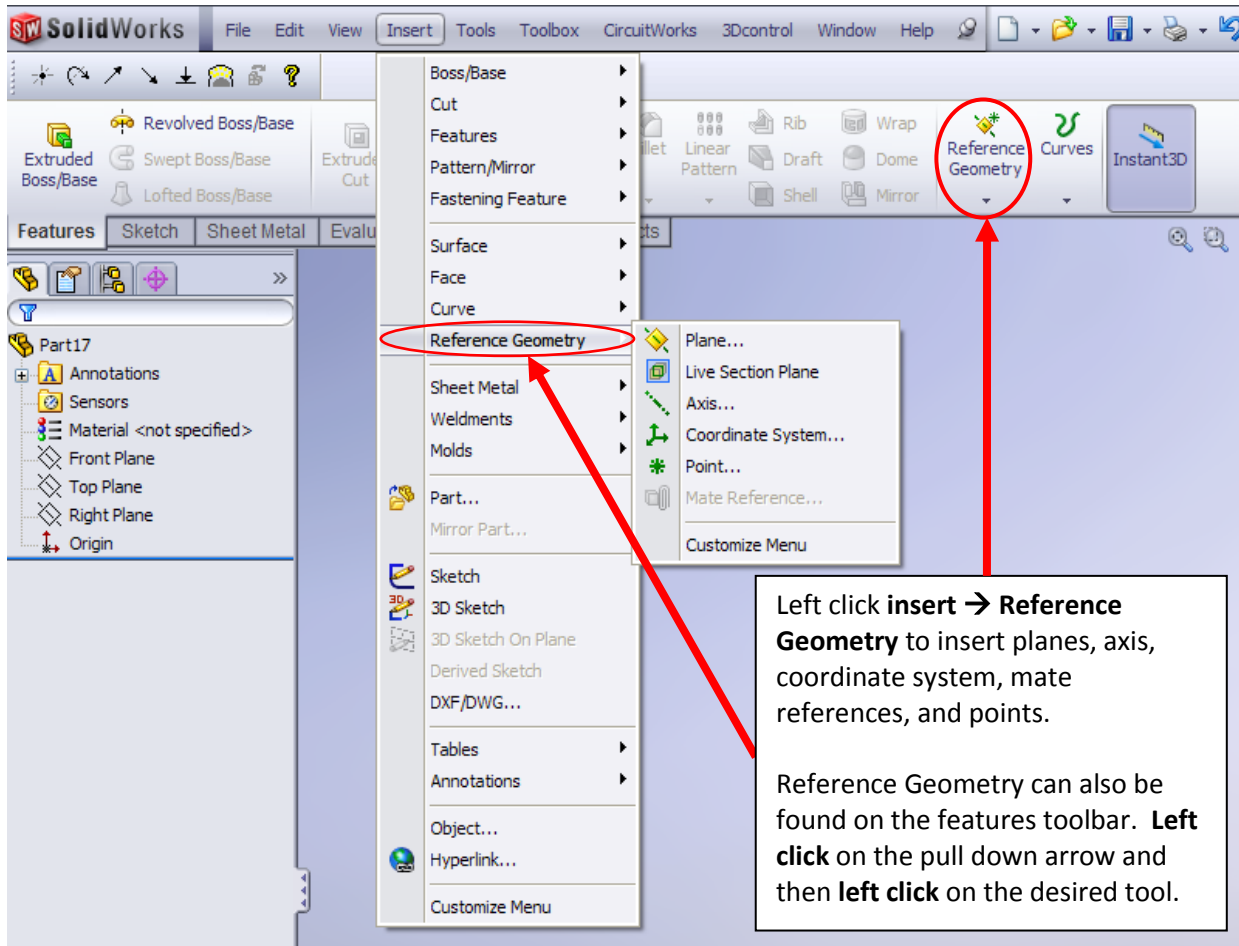


# Reference Geometry

*Description:* Planes, axes, coordinate systems, and 3d curves are all types of reference geometry. These references serve as aides to the design of parts.

*Note:* Reference geometry can be helpful in using features such as lofts, sweeps, patterns, drafts, and chamfers.



# Types of Reference Geometry

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This feature allows the user to create additional planes to sketch on other than the given right, front, and top plane. To create a reference plane **left click on plane** and select the desired option in the property manager for creating the plane. These options include:

Through lines/points: creates a plane through a point an edge, sketch line, three points, or axis.

Parallel plane at point: creates a plane through a point parallel to a plane or face.

At angle: create a plane through an axis, edge or sketch line at a specified angle from a face or plane.

Offset distance: create a plane parallel to an existing plane and specifying the desired distance.

Normal to curve: Create a plane through a specified point and makes it perpendicular to a desired curve or edge.

On surface: creates a plane on a non-planer face or angled surface.



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Allows the user to insert an axis in the part document. Axes are particularly useful in creating part features. To create a reference axis **left click on axis** and select the desired option in the property manager for creating the axis. These options include:

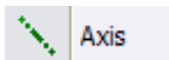
On edge: places an axis on a desired edge or sketch line of the part.

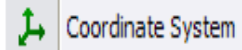
Two planes: create an axis at the intersection of two planes or planar faces.

Two points/vertices: create an axis through two selected points, vertices, or midpoints on a line, edge, or part corner.

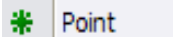
Cylindrical/conical faces: places an axis through the center of a circular, cylindrical, or conical face. This axis can also be seen by **right clicking on view** and then selecting **temporary axes**.

Point and a face: creates an axis perpendicular to a selected face or plane and through a selected point, midpoint, or vertex.





Create an additional coordinate system on the part by specifying the x, y, and z directions with the parts geometry. This can be helpful when integrating a part into an assembly as a reference for directional part interactions as well as setting up a zero point for machining and manufacturing the part.



Add a singular location on a part document that references geometries found on the modeled part. These can be placed at the center of an arc, the center of a face, an intersection, along a curve, or projected onto another entity. These can be helpful in making point clouds to help define and place three or two dimensional sketches. Most commonly used in complicated parts such as vehicle frames.

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