

# Features

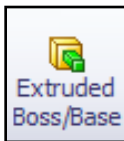
*Description:* An individual form that helps contribute to the formation of a part or assembly. Some features such as the extruded boss/base originates from a sketch whereas shells and fillets modify parts geometry.

*Note:* A user can select the desired feature before entering a sketch. **Left click** on the feature then select a plane or surface to sketch on. Upon exiting the sketch the desired feature property manager will appear on the left hand side of the screen.

## Features Toolbar At A Glance



## Feature Tools



### Extruded Boss/Base:

Projects a two dimensional sketch linearly to create a three dimensional part. Can be two directional.



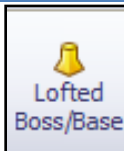
### Revolved Boss/Base:

Revolves a sketch entity about an axis creating a three dimensional part.



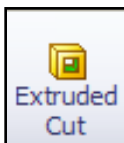
### Swept Boss/Base:

Moves a sketch entity, or profile, along a sketched or preexisting part path. *Note: these entities need to be separate sketches.*



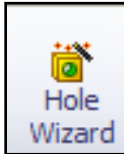
### Lofted Boss/Base:

Creates a transitional surface between two sketch entities. The sketch entities need to be located on two different planes or surfaces.



### Extruded Cut:

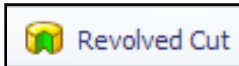
Cuts away a two dimensional sketch linearly in an existing part. Can be two directional.



Hole Wizard:

Create holes in a preexisting part that are the correct size for standard fasteners. The types of holes include: counterbore, countersink, tap, pipetap, simple holes, and legacy holes.

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Revolved Cut:

Similar to the revolve boss/base feature. Revolve cut takes a two dimensional sketch and revolves it about an axis to remove material from a preexisting part.

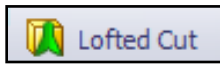
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Swept Cut:

Similar to the swept boss/base feature. Swept cut takes a sketch entity and extrudes it along a selected path. This feature removes material from a preexisting part.

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Lofted Cut:

Similar to lofted boss/base feature, lofted cut creates a transitional surface between two sketch entities and then removes the material from the part. The sketch entities need to be located on two different planes or surfaces.

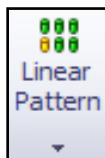
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Fillet:

Rounds off sharp corners, or adds volume to the material by rounding off an inner corner. The user inputs the radius of the fillet. This feature does not require a sketch to use; only a preexisting part is needed. Chamfer can be found under the pull down arrow. This feature performs similarly except that it breaks the edges rather than rounds them.

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Linear Pattern:

Patterns part features, such as holes, in a straight line. The user defines the amount of space between the patterns, quantity, whether it will be multi-directional, and if certain instances should be skipped.

Circular pattern can be found under the pull down arrow and performs similarly to linear pattern except that it patterns about a circular path. The user inputs the circular parameter, the angle, the number of instances and whether the patterned feature is equal in spacing to one another.

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Rib:

An extrude feature created from open or closed sketch contours. The user specifies the desired thickness and the directional path between the sketch contour and the part.

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Draft:

This feature allows the user to add an angle to part faces.

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Shell:

Hollow out the part by selecting a face to hollow out and specifying the thickness or thicknesses.

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Wrap:

Wrap a sketch around planar faces such as a cylindrical, conical, or extruded face. *Note: the sketch plane must be tangent to the desired wrap face. Also the sketch must contain only closed contours.*

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Dome:

Create domes on top of selected faces of the existing part. Typically used for aesthetic purposes.

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Mirror:

Create a reflected copy of a desired part feature about a plane or part face. If the parent or original part feature is updated the mirrored child part will also update.

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Flex:

Deforms models by bending, twisting, tapering, or stretching the part. This feature allows the user to manipulate the part by specifying the flex bounds and deforming it by specifying the movement with a triad. The flex feature property manager allows for multiple deforming options and inputs.

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Split:

Create multiple bodies from a single part. This allows the user to create multiple part files or an assembly from the newly made parts.

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Scale:

Resize or scale a part or surface about its centroid, origin, or coordinate system. This feature only scales the geometry, not the dimensions sketches or reference geometry.

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