

# Nx For Beginners: Part Extrude and Revolve Features, Placed Features and...

Nx is a powerful CAD/CAM/CAE software suite that is used by engineers and designers all over the world. It is known for its ease of use, powerful features, and scalability. In this article, we will discuss the Extrude and Revolve features in Nx, as well as how to use Placed Features and...

The Extrude feature is used to create solid or surface features by extruding a sketch or face in a specified direction. The Extrude feature can be used to create a variety of features, such as bosses, ribs, and slots.

To create an Extrude feature, first select the sketch or face that you want to extrude. Then, click on the Extrude icon in the Features toolbar. The Extrude dialog box will appear.



## NX 9 for Beginners – Part 2 (Extrude and Revolve Features, Placed Features, and Patterned Geometry)

by Pauline Irene Stacey

★★★★☆ 4 out of 5

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In the Extrude dialog box, you can specify the following options:

- **Direction:** The direction in which you want to extrude the sketch or face.
- **Distance:** The distance that you want to extrude the sketch or face.
- **Taper Angle:** The taper angle of the extruded feature.
- **End Condition:** The end condition of the extruded feature.

Once you have specified the desired options, click on the OK button to create the Extrude feature.

The Revolve feature is used to create solid or surface features by revolving a sketch or face around a specified axis. The Revolve feature can be used to create a variety of features, such as cylinders, cones, and spheres.

To create a Revolve feature, first select the sketch or face that you want to revolve. Then, click on the Revolve icon in the Features toolbar. The Revolve dialog box will appear.

In the Revolve dialog box, you can specify the following options:

- **Axis:** The axis around which you want to revolve the sketch or face.
- **Angle:** The angle through which you want to revolve the sketch or face.
- **End Condition:** The end condition of the revolved feature.

Once you have specified the desired options, click on the OK button to create the Revolve feature.

Placed Features are features that can be placed on a part or assembly. Placed Features include features such as holes, threads, and chamfers.

To place a Placed Feature, first select the feature that you want to place. Then, click on the Place Feature icon in the Features toolbar. The Place Feature dialog box will appear.

In the Place Feature dialog box, you can specify the following options:

- **Placement:** The location where you want to place the feature.
- **Orientation:** The orientation of the feature.
- **Parameters:** The parameters of the feature.

Once you have specified the desired options, click on the OK button to place the Placed Feature.

The Extrude, Revolve, and Placed Features features are powerful tools that can be used to create a variety of features in Nx. By understanding how to use these features, you can improve your productivity and create more complex designs.



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