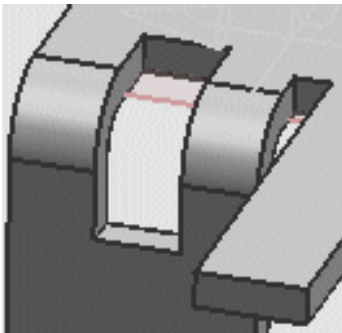




Normal Cutout overview

- Overview
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Use the **Normal Cutout** command to cut material by projecting a sketch onto the model and then cutting perpendicular to the faces intersected by the projection. You can create cutout profiles with planar sketches or 3D curves. The latter allow you to position the cutout accurately on non-planar faces of a part. A Normal Cutout feature has edges that remain perpendicular to the planar face of the part. The **Normal Cutout** command maintains sheet metal part properties by always cutting normal to the top and bottom faces of the model. It also conforms to all other global sheet metal part properties.

Normal cutouts are particularly useful when you want to make cuts across bend regions. The feature is also useful when you need to unbend and rebend sheet metal parts, because non-perpendicular thickness faces created with **Extrude- Subtract** may be deformed during **Unbend/ Rebend** operations. For an illustration of the different results produced by **Extrude** and **Normal Cutout**, see [Using Normal Cutout.](#))

This command can be used instead of the alternate workflow of unbending, punching, and then rebending the part.

Where do I find it?

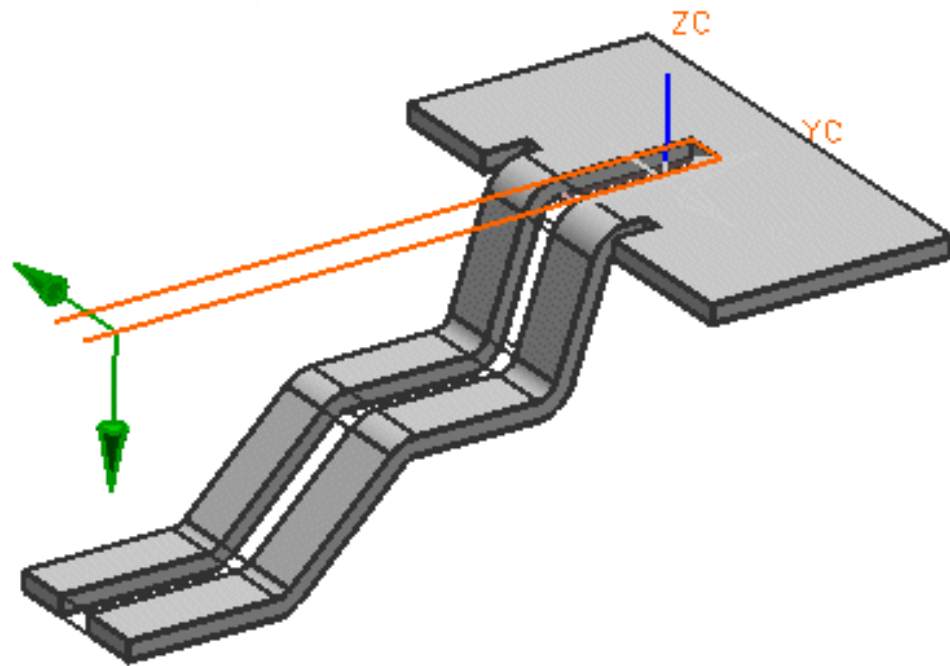
Application	NX Sheet Metal
Toolbar	NX Sheet Metal→Cut list  →Normal Cutout
Menu	Insert→Cut→Normal Cutout

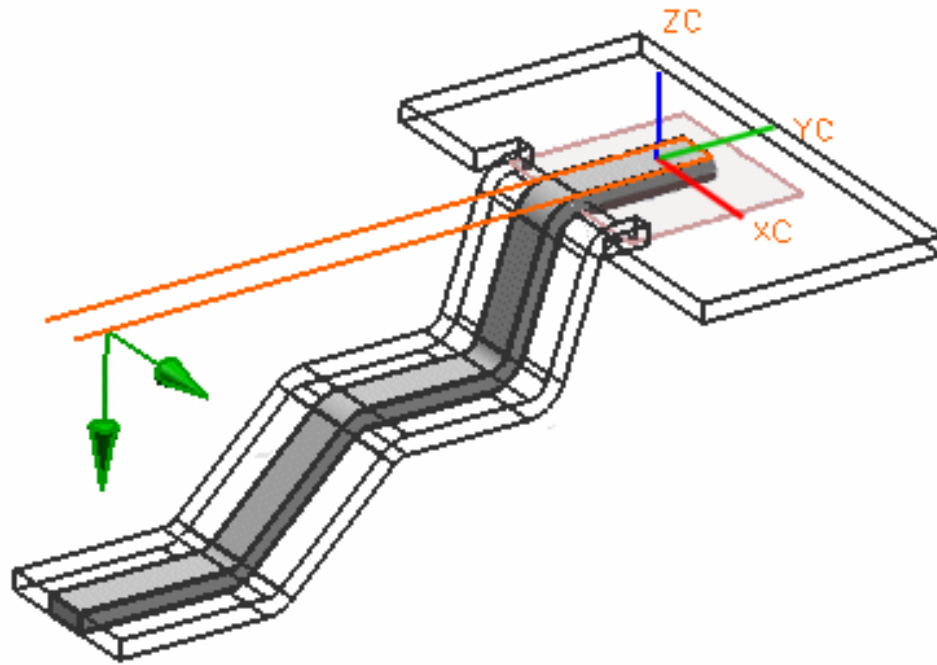


Create a Normal Cutout using a sketch

[Overview](#)[How To](#)[Options](#)[Related Topics](#)

1. On the **NX Sheet Metal** toolbar from the Cut list, select **Normal Cutout** , or choose **Insert**→**Cut**→**Normal Cutout**.
2. In the **Type** list, select **Sketch**.
3. Draw a profile, or select a section from an existing sketch.
 - A closed profile defines the cutout.
 - An open profile displays a direction handle that is perpendicular to the open profile and co-planar with the sketch. Use this handle to specify which side of the profile to cut. The following figure shows the effects of reversing the direction handle.





The curves of the open profile are extended tangentially until they intersect themselves to create a closed profile, or until they intersect the edge of the part.

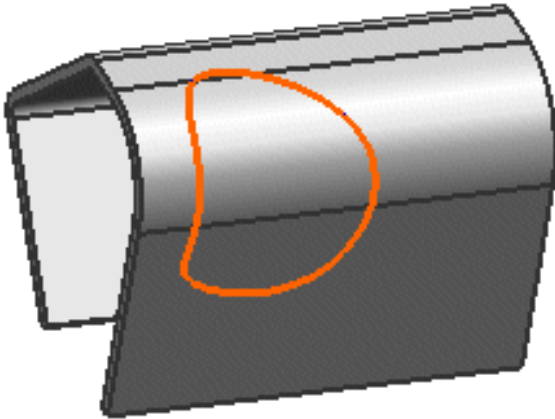
4. From the **Cut Method** list, select the type of cut you want to make: **Thickness** or **Mid-Plane**.
5. From the **Limits** list, specify the extent of the material you want to remove. If you select **Value**, you can select the **Symmetric Depth** check box to specify that you want to create a cut area symmetric in both directions from the sketch plane.
6. Click **OK** to create the **Normal Cutout** feature.



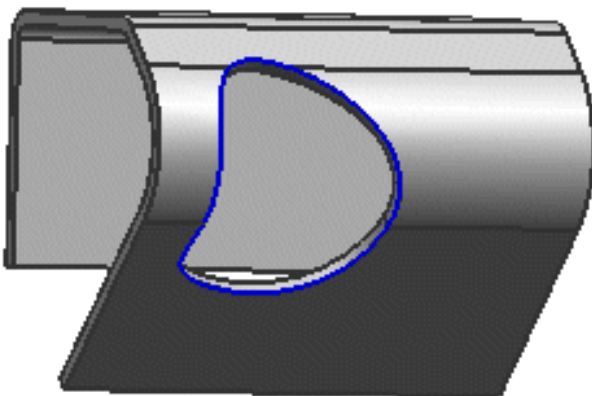
Create a Normal Cutout using a 3D curve

[Overview](#)[How To](#)[Options](#)[Related Topics](#)

1. On the **NX Sheet Metal** toolbar, from the Cut list, select **Normal Cutout** , or choose **Insert**→**Cut**→**Normal Cutout**..
2. In the **Type** list, select **3D Curves**.
3. Select the 3D curve that defines the cutout shape, as shown in the following figure.



4. Click **OK** to create the Normal Cutout feature.





Normal Cutout options

Overview	How To	Options	Related Topics
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Type specifies the way the profile is created.

3D Curves	Specifies a profile created from three dimensional curves that lie on a part face.
Sketch	Specifies a sketch that you select or create on a planar face of the part.

Section

Select Curve	Lets you select a sketch or create one when you select Sketch from the Type list. The sketch must lie on a planar face of the part. If you create an open profile, you can use the direction arrow to specify which portion is cut out, and which is retained. Lets you select 3D curves when you select 3D Curves from the Type list. The profile can be created using a variety of Modeling tools, such as Curve on Surface.
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Cutout Properties

Cut Method	Thickness  compensates for the material thickness of the part by projecting the profile onto both layer faces, inner and outer (top/bottom), then thickening the resulting surface into a solid tool, and removing the tool from the base part Mid-Plane  projects the profile onto the mid-plane of the sheet metal part, then thickens the resulting surface into a tool body. Usually the Thickness option consumes more material than the Mid-Plane option.
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Limits

- **Value**  lets you define the extent of the cutout manually, either by dragging the depth handle or by entering the distance in the **Depth** box.
- **Between**  lets you define the extent of the cutout by selecting two parallel planes or faces.
- **Until Next**  extends the cutout to the first edge encountered.
- **Through All**  extends the cutout through the whole part.
-  The **Symmetric Depth** check box is only available when you select **Value**. This option creates a cut area symmetric in both directions from the sketch plane.

