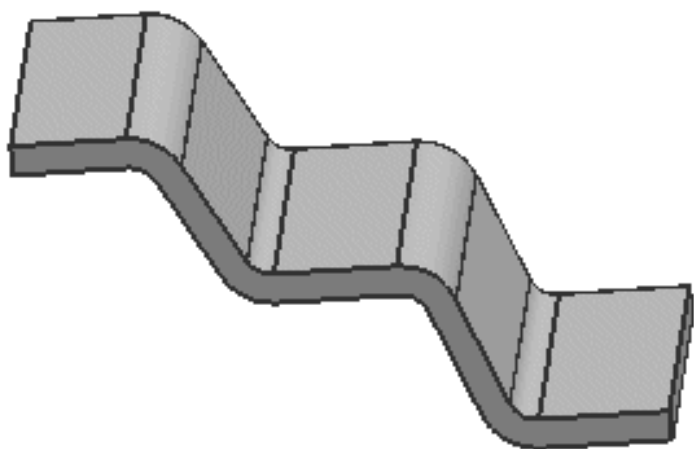
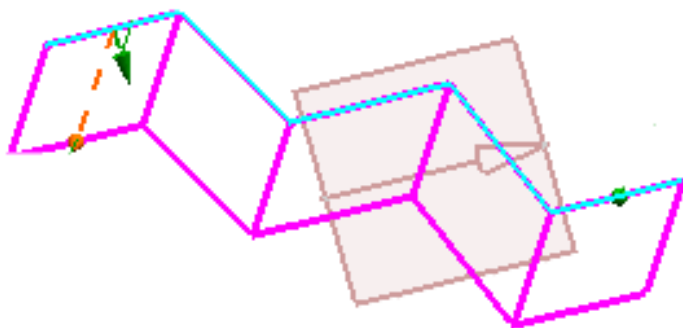




Contour Flange overview

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

Contour Flange constructs a flange with several contours by extruding a profile that represents its cross section.



You can use **Contour Flange** as the base feature of a new sheet metal part or to add a feature to an existing one. Using **Contour Flange**, you can construct one or more bends at any angle. With the **Chain Extent** option, the flange chains itself around multiple selected edges and has options for mitered corners between the different flange sections.

You don't have to draw the arcs at each bend location. Bends are added automatically using the default bend radius. You can see this in the part and profile shown above.

If you want to use a different bend radius value, draw arcs in the profile, or change the default bend radius.

You can also use **Contour Flange** to create [wrapped features](#).

Where do I find it?

Application	NX Sheet Metal
Toolbar	NX Sheet Metal→Bend list  →Contour Flange
Menu	Insert→Bend→Contour Flange



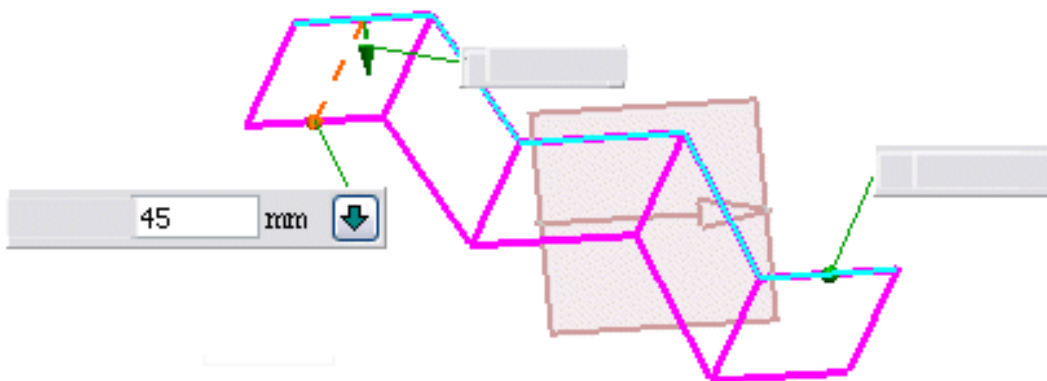
Create a contour flange

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

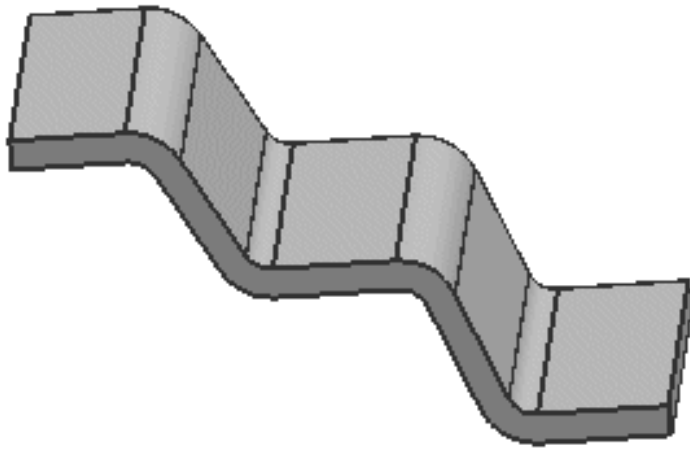
To create a **Base Contour Flange**:



1. Click **Contour Flange**. **Base Contour Flange** is selected by default if there are no sheet metal features in the part file. If there are, **Contour Flange** defaults to the secondary option.
2. Define the profile plane.
3. Draw an open profile, and select it when prompted to select section geometry.
4. Specify part thickness or accept the default.
5. Specify the width of the feature.



6. Choose options.
7. Click **OK** to complete the feature. The arcs at each bend location are added automatically when you finish the profile, unless you have sketched arcs in the bend areas.

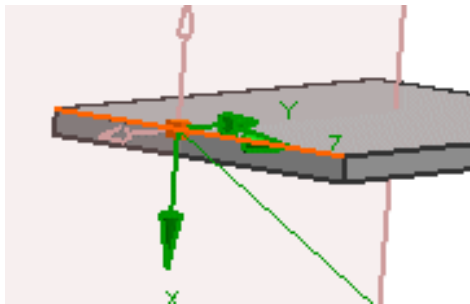


To add a contour flange to a sheet metal part (that is, create a **Secondary Contour Flange**):

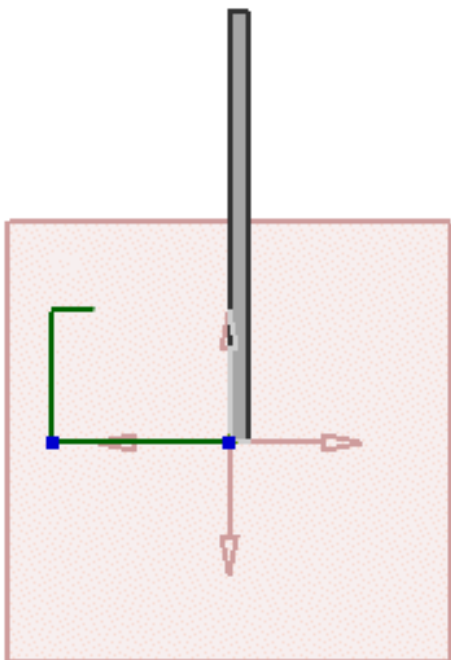
1. Click **Contour Flange** .

Secondary Contour Flange is selected by default because there is already a body in the part file.

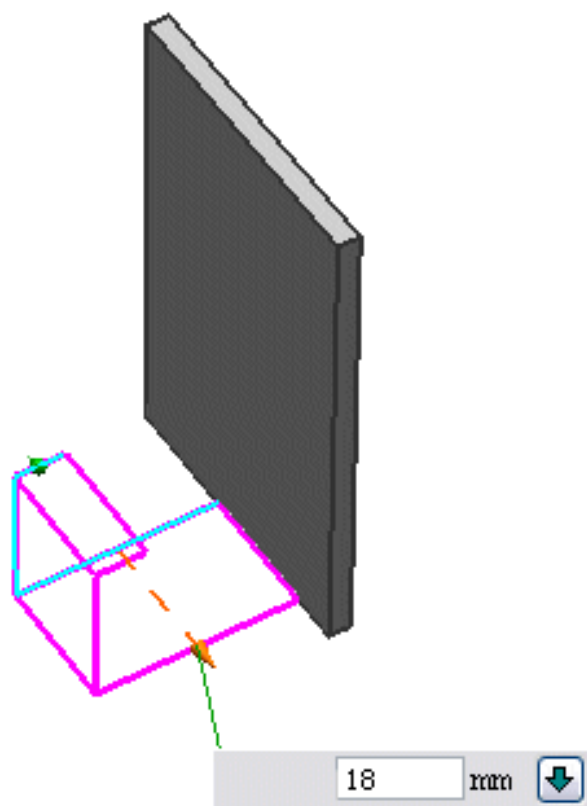
2. Select an existing section or click **Sketch Section**  and select a part edge. The plane on which you draw the profile must be normal to the part edge to which you are attaching the contour flange.



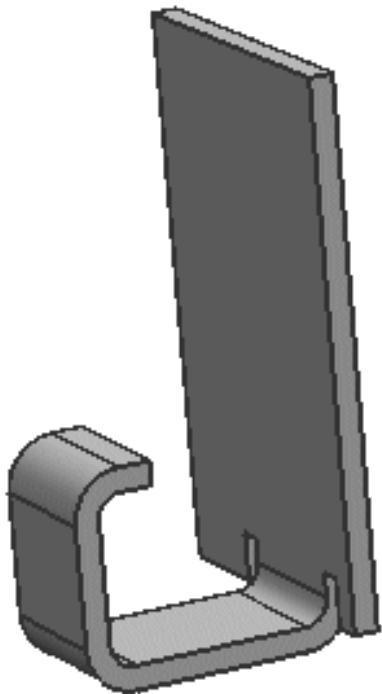
3. Draw an open profile. It must be coincident with a part edge.



4. Choose options in the dialog box, if any.
5. Specify the width of the flange.

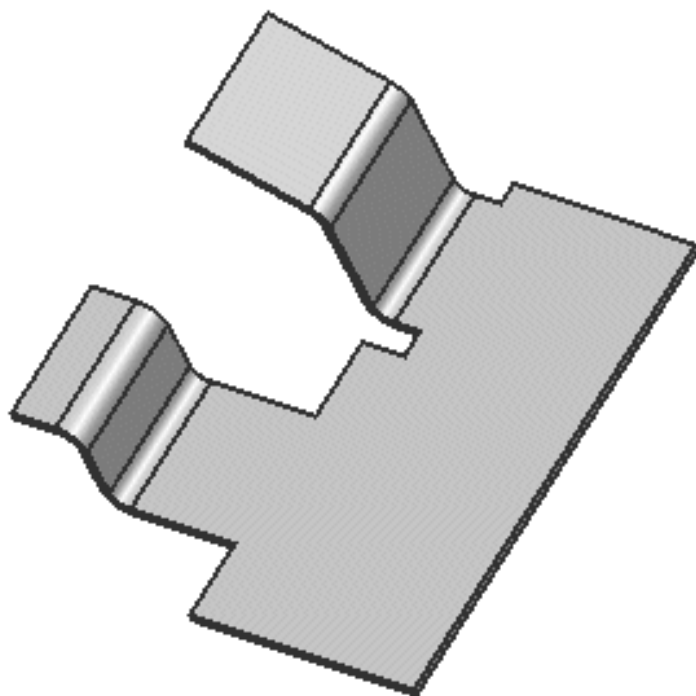


6. Click **OK** to complete the feature.



Tips and techniques for contour flanges

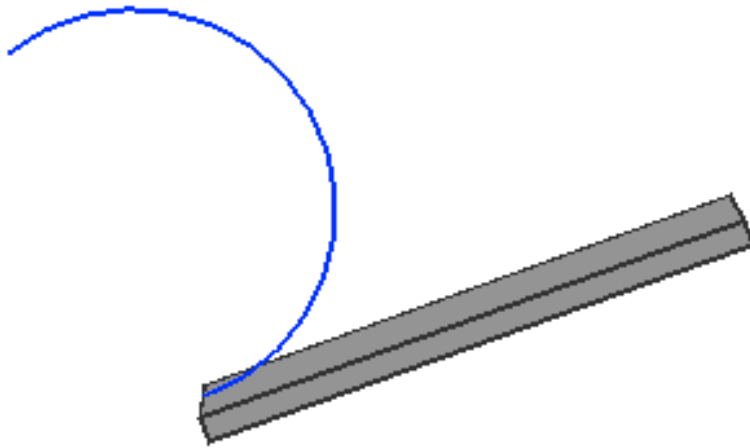
- You can only apply bend reliefs with **Contour Flange** when the bend section actually intersects the original edge on which you placed the flange. The figure below illustrates this. Bend relief 5 mm wide and 5 mm deep was selected for both contour flanges. Note that only the flange with the bend positioned at the edge has bend relief.



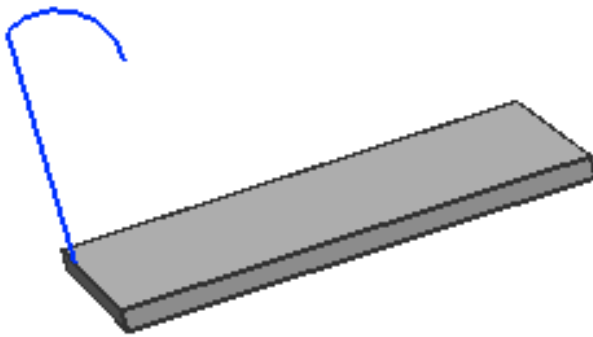
- If the underlying sketch for **Contour Flange** intersects the part with an arc section, the arc

must be tangent to the top or bottom face of the part where it intersects the placement edge.

- Any arcs that define the outer face of a bend region must have a radius greater than the part thickness.
- Certain contour flange profiles produce error conditions:
 - The following graphic shows an invalid three-point arc for a secondary contour flange. The problem is that the tangent line that is connected to the edge lies within the plane of the face of that edge



- The following profile, which consists of a combination of lines and arcs, does not maintain tangent continuity at the intersection of the arc and line.



- When creating contour flanges on curved edges, create the profile at the end of the curved edge for most predictable results.



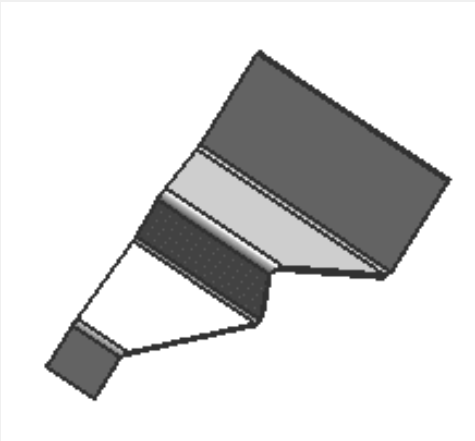
Contour Flange options

- Overview
- How To
- Options
- Related Topics

Width

Width Option	 • Finite Extent projects the profile a finite width to either side of the profile plane. You must supply a width. • Symmetric Extent  applies half the specified width to each side of the profile. You must supply a width. • To End Extent  projects the profile to the end of the selected edge. • Chain Extent  projects the contour flange along a series of edges that you select. • Width: Specify a width (Finite and Symmetric Extent only).
--------------	---

Miter: You can set the following options for both the **Start** and **Finish End** of the contour flange

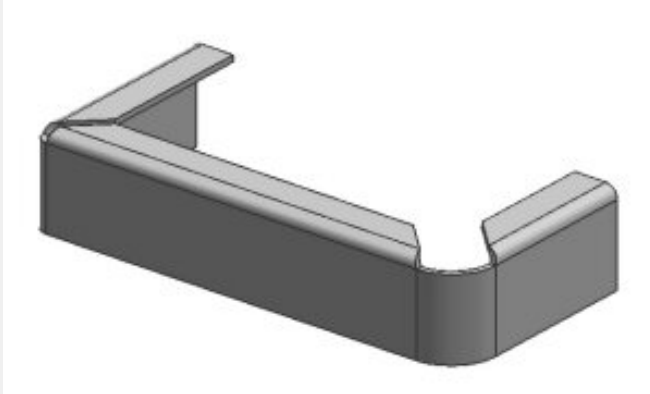
Miter Corners	Miters (trims diagonally) the end at the angle you specify. Here is a flange mitered at the standard -45 degree angle.  Cutout • Normal to Thickness Face miters the end of the contour flange perpendicular to the thickness face. • Normal to Source Face miters the end of the contour flange perpendicular to the source face. Angle sets the miter angle for the specified end of the contour flange. A negative value miters the flange
---------------	--

inward and typically removes material. A positive value miters the flange outward and typically adds material. With some complex contour flanges, material can be both added or removed when an end is mitered.

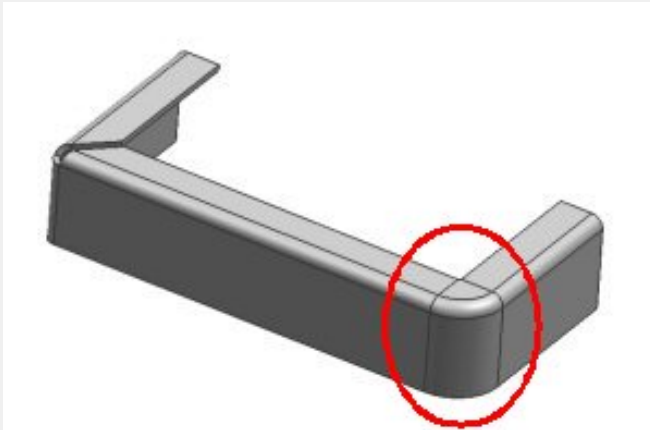
Miter Using Normal Cutout Method

Creates miters perpendicular to the thickness face at corners and ensures that you can unbend and rebend the contour flanges in your part. Note that this method removes more material at the corners, so if you want water-tight flanges at the corners and are not concerned about creating a flat pattern of this part, you may prefer not to use this option.

(**Chain Extent** only) Determines whether a miter should be applied, and only comes into effect when the chain edges of a contour flange span a bend region. This option for **Contour Flange** in NX 5 results from the new ability to flange around curved edges. To achieve behavior equivalent to NX 4, you must select only the tab edges and not the bend edges, as shown in the following figure.

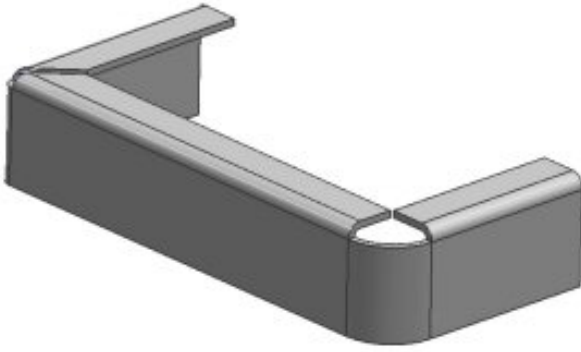


In NX 5, if you select the bend edges, the **Contour Flange** is built across those edges as well. If the radius of the bend region being spanned is large enough, the **Contour Flange** fills in this area with a rounded bend region, as shown below.

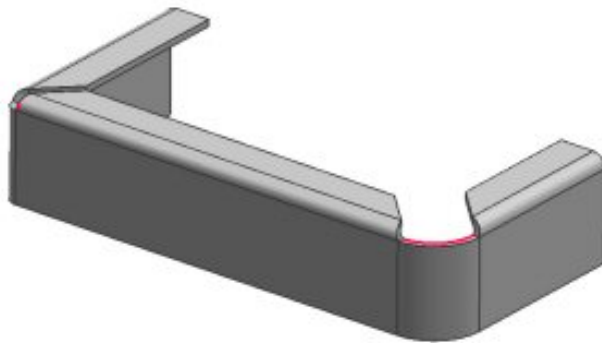


If you select **Miter Interior Corners When Necessary**, the software only miters the corners spanning bend regions if the corners would self-intersect. In the figure below, the bend region on the left is too small to accommodate the flange edges, so they are mitered. The bend on the right is large enough that it isn't necessary.

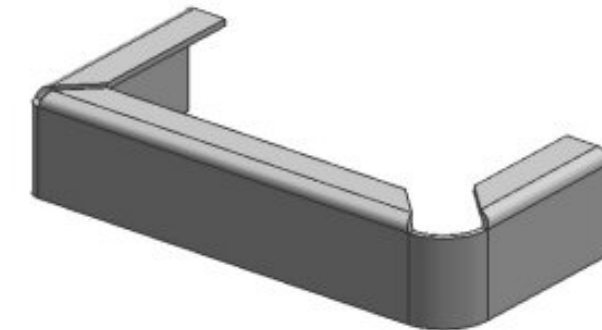
Miter Interior Corners When Necessary



The decision to miter or not is based on the length of the flange and the radius of the bend region being spanned. If the length of the flange is less than the radius of the bend region that the chain is traversing, NX Sheet Metal does not apply miters to that corner region. The radii in question are shown below highlighted in red.



If you do not select this option, NX Sheet Metal miters the sides of the flanges, regardless of the length of the flange or the radius of the bend region being spanned, as shown in the following figures.

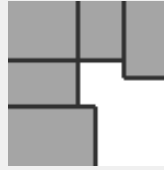


Corners

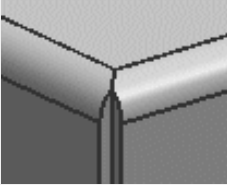
**Close
Corner**

Specifies that you want to close the interior corners. Bend relief is applied when you close three inside bend corners. The following treatment options are only available if you select **Close Corner**.

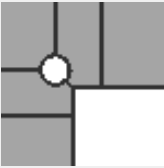
Open applies no treatment.



Close closes the bent faces of the flange until the edges intersect.



Circular Cutout creates a found cutout. If you select this option, you must specify a diameter for the circle.



You can also modify the default [bend parameters](#).