



Lofted Flange overview

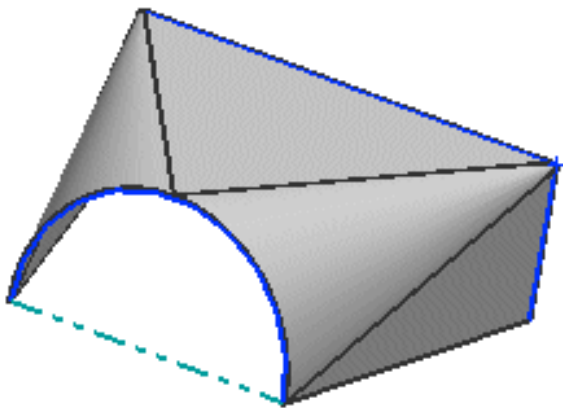
Overview

[How To](#)

[Options](#)

[Related Topics](#)

Lofted Flange is an extruded flange that uses start and end profiles to create the basic shape. The profiles must be open and must be on parallel reference planes. **Lofted Flange** is particularly useful for HVAC work, as it allows for round to square transitions. You can also create conical bend regions with **Lofted Flange**.



A **Lofted Flange** can be either a base or a secondary feature. The command automatically adds bends using the bend radius property. You do not have to draw an arc at each bend location.

Where do I find it?

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



Create a lofted flange

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

To create a lofted flange as a base feature:

1. Create parallel datum planes offset from each other. The offset distance should correspond to the length of the lofted flange you intend to create.

2. Click **Lofted Flange** . **Base Lofted Flange** is selected by default if there are no other bodies in the part file.

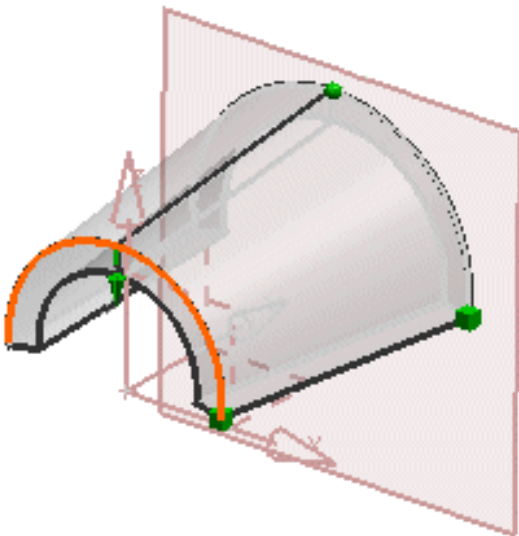
3. Choose a method for creating the first cross section: sketch a section on a profile plane



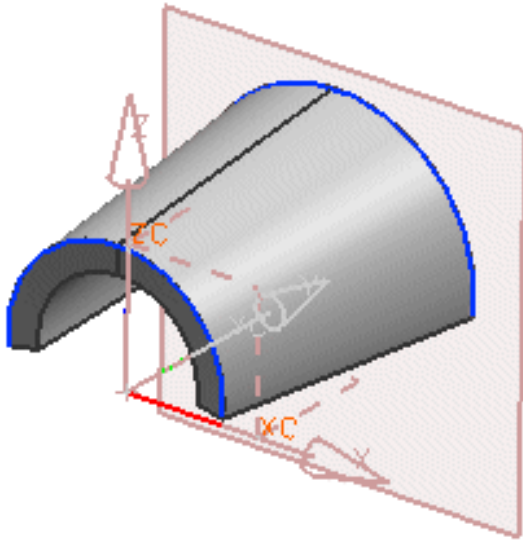
or select a section you've already created . Note that both cross sections must be open profiles. When prompted, select the section you sketched, then press Enter.

4. Select the start point for the first section  or construct one , then press Enter.

5. Repeat steps 3 and 4 for the second cross section on the second offset plane. The start points for the cross sections (the green cubes in the following illustration) should be in line with each other so as to avoid twisting the shape. (If it is twisted, it cannot be flattened).



6. Select a material side and a thickness value (or accept the defaults).
7. Click **OK** or **Apply** to complete the feature.





Lofted Flange options

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

Start Section is the curve you loft from.

Select Curve	Lets you select a start section. You can use a pre-existing sketch or you can create one. This is the curve you loft from. For more information on these sketch options, see Using Sketches in NX Sheet Metal .
Specify Point	Select a start point or create one on the curve you just selected.

End Section is the curve you loft to.

Select Curve	Select an end section, using a either a pre-existing sketch or creating one. It must lie on a plane parallel to the plane of the Start Section .
Specify Point	Select a start point on the End Section . It must correspond to (be on the same side as) the point on the Start Section . (If the loft is twisted, it will either fail, or at best will not flatten.

Bend Segments specifies whether to divide conical bend regions into multiple segments.

Use the slider to specify the **Number of Bend Segments**. You can divide a bend region into a maximum of 24 equal segments.

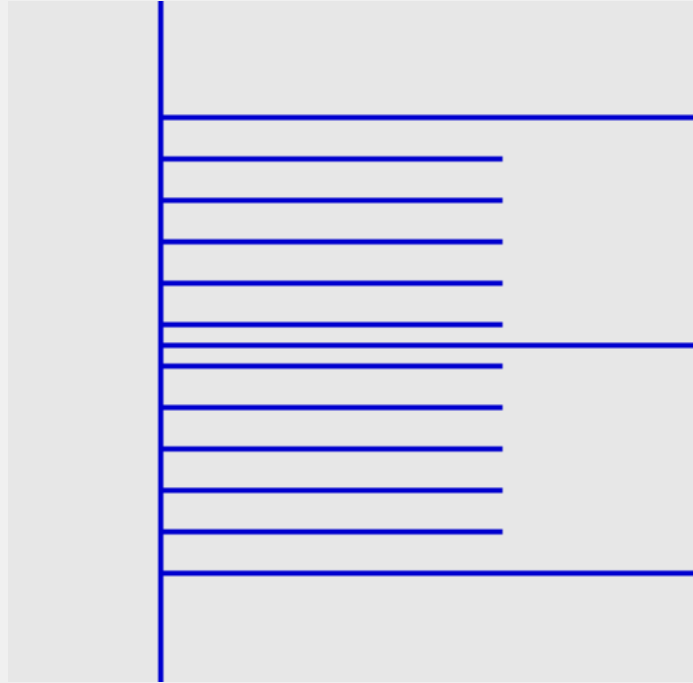
The following graphic shows a bend region divided into 12 segments.



Use Multi-segment bends

Index Mark Length lets you specify the length of short lines connected to the edge of the part along the bend. These lines help align the metal with the press brake, and they only appear on the flat pattern.

The following graphic shows the same bend region in flat pattern view, magnified to make the index marks clearly visible.



Lofted Flange has the standard bend and corner options. For more details, see [Bend parameters for sheet metal features](#).