



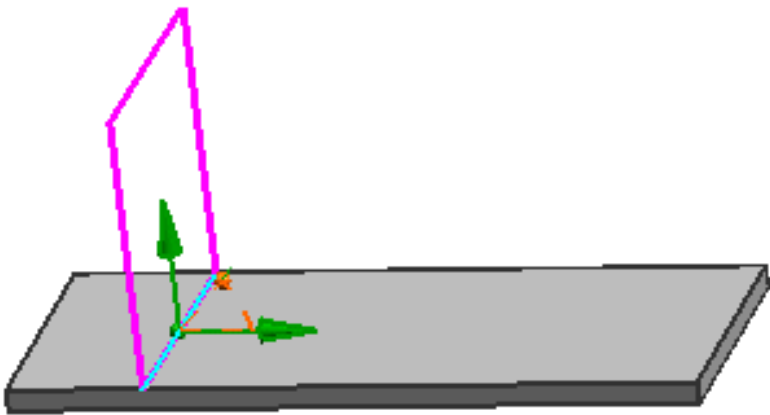
Bend overview

Overview

[How To](#)

[Options](#)

Use this command to create a bend in a planar area on a sheet metal part.



Where do I find it?

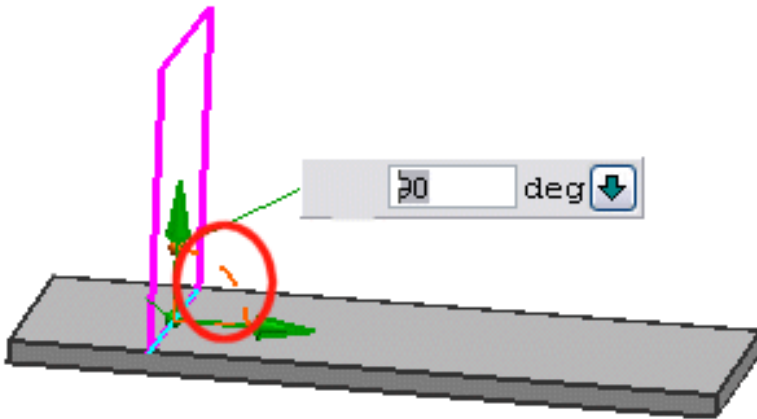
Application	NX Sheet Metal
Toolbar	NX Sheet Metal→Bend list  →Bend
Menu	Insert→Bend→Bend



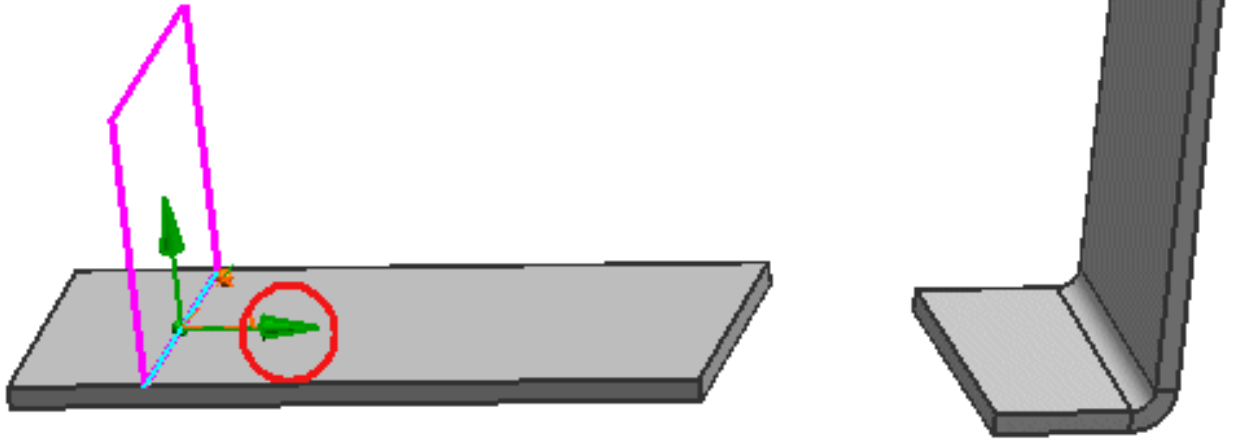
Create a Bend

[Overview](#)**How To**[Options](#)

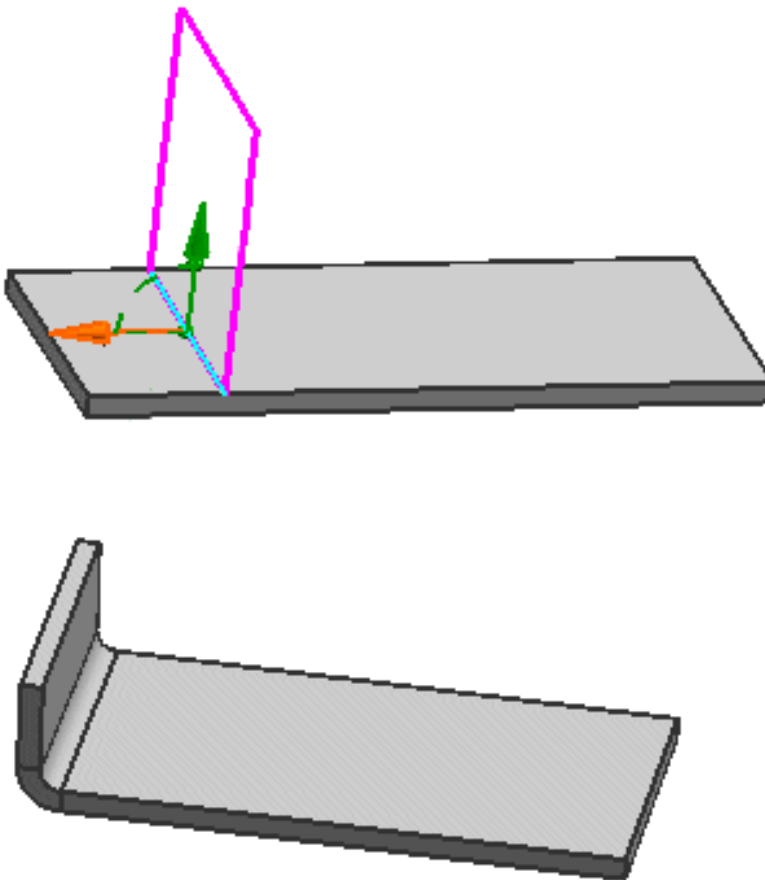
1. Click **Bend** .
2. Define the profile plane.
3. Draw a profile. The profile, which must be a single linear element, represents the approximate location of the bend.
4. Define the bend location with respect to the profile.
5. To change the default bend angle (90 degrees), drag the angle handle or enter a value in the input field.



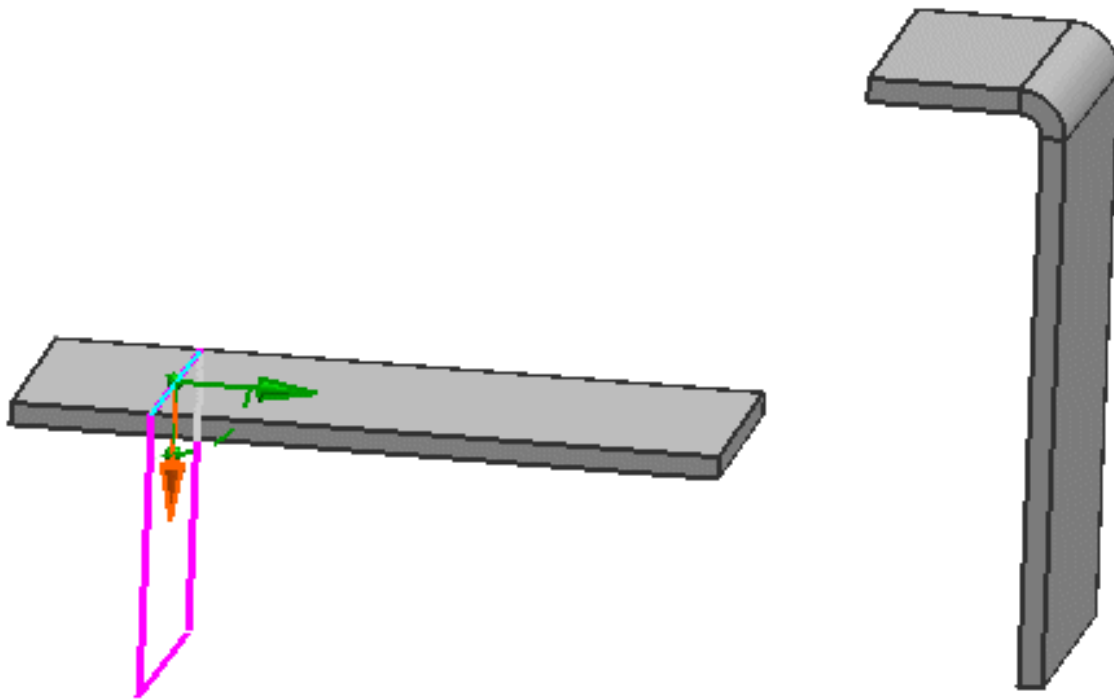
6. Specify the side of the part to move. The direction arrow points towards the portion that will move.



To change the direction, double-click the direction arrow.



7. Define the bend direction. To change the bend direction, double-click the direction arrow.



8. Click **OK** to finish the feature.



Bend options

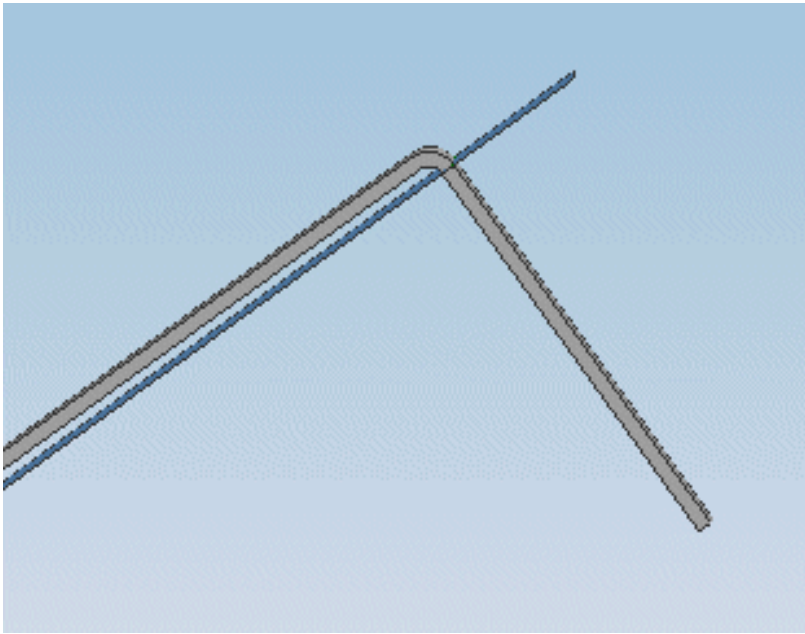
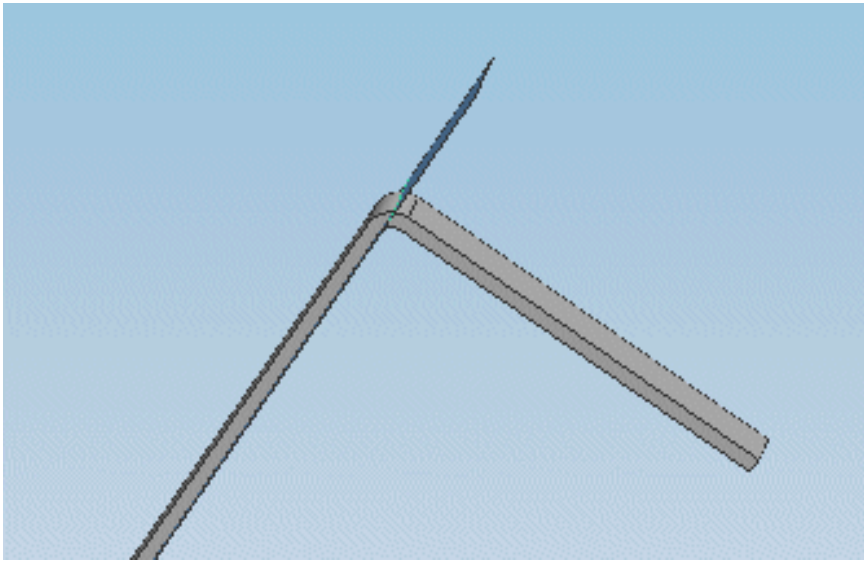
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- Options

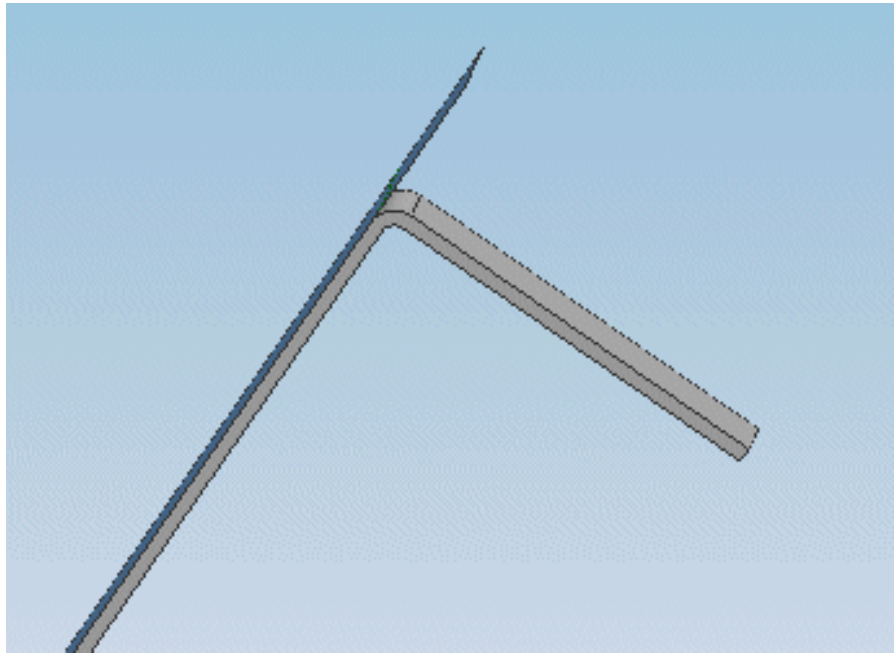
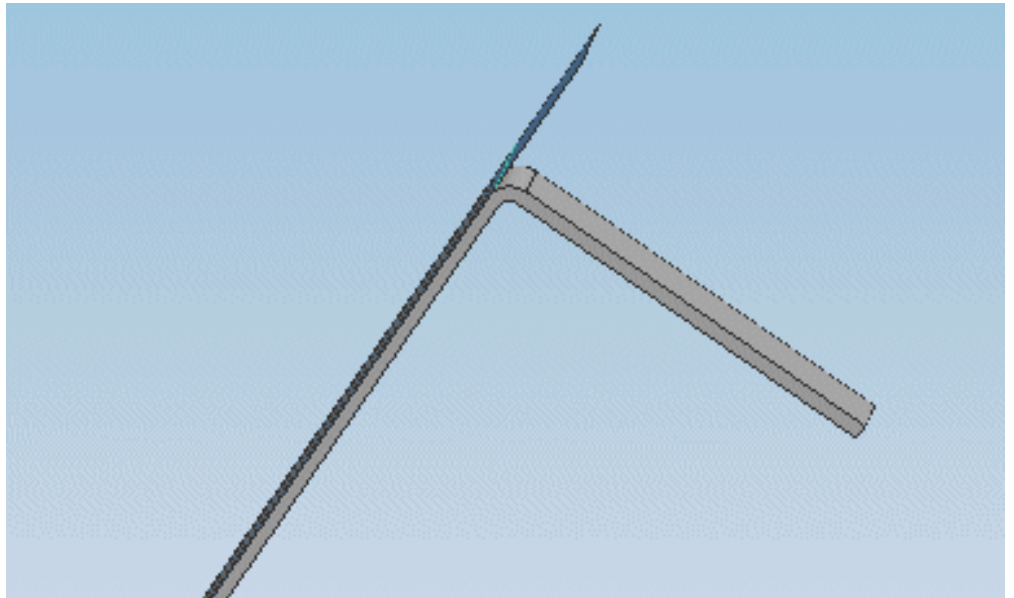
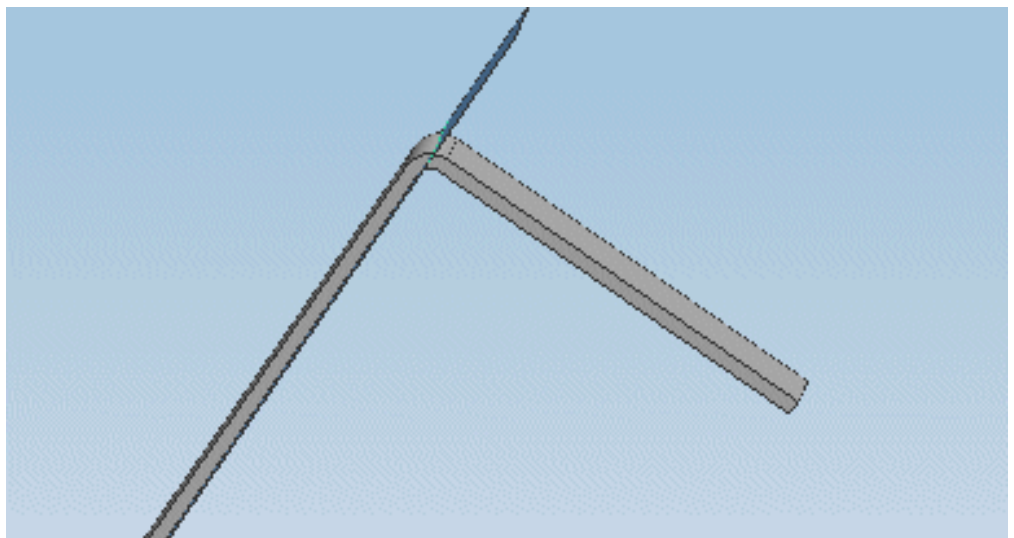
Bend Properties	
Angle	Specifies the angle of the bend.
Reverse Direction 	Changes the direction of the bend from up to down and vice versa.
Reverse Side 	Changes the side of the part that moves to create the bend.
Inset	Outer Mold Line Profile  specifies that the profile represents the line created by the junction between the planar stationary region and the cylindrical bend region in the flattened state.
	Bend Center Line Profile  specifies that the profile line represents the center line of the bend, and that the bend region will be distributed evenly on either side of the profile in the flattened state.
	Inner Mold Line Profile  specifies that the profile represents the line created by the junction of the planar web region and the cylindrical bend region in the flattened state.
	Material Inside  positions the portion of the feature that is perpendicular to the profile plane inside of the profile plane.
	Material Outside  positions the portion of the feature that's perpendicular to the profile plane outside of the profile plane.

Extend Section	Extends the linear profile you draw to the edges of the part.

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

The following figures show how the bend is positioned differently, depending on which bend location option you choose. The extended plane going through the part is a visual aid to highlight the bend location. Note: The **Bend Center Line**, **Inner Mold Line**, and **Outer Mold Line** profiles need to be shown in the flattened state because this is the state in which these measurements make sense.

Bend Location	Graphic
Outer Mold Line Profile	
Bend Center Line Profile	

Inner Mold Line Profile**Material Inside****Material Outside**

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