



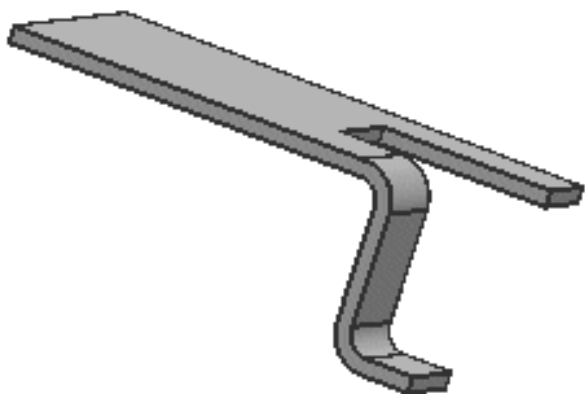
Jog overview

Overview

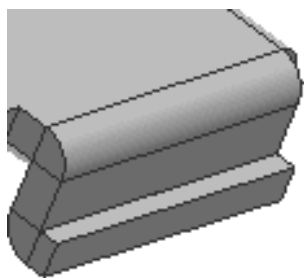
[How To](#)

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Jog constructs two 90-degree bends and adds material to them to jog to a planar face of a sheet metal part.



The profile for a **Jog** feature must be a single linear element and can only be constructed across a planar face. The jog can be minimal: for example, a slight offset or step to provide clearance or rigidity to a part.



Where do I find it?

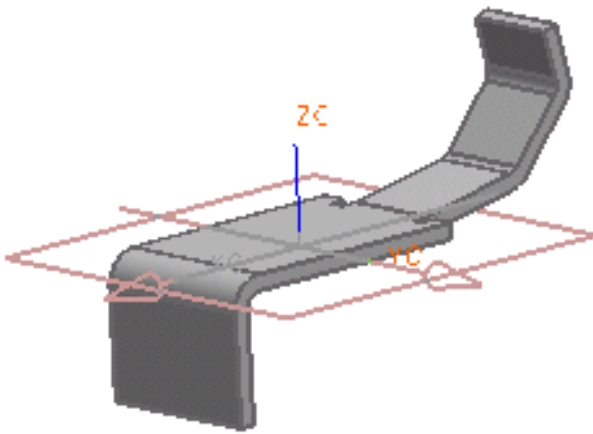
Application

NX Sheet Metal

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

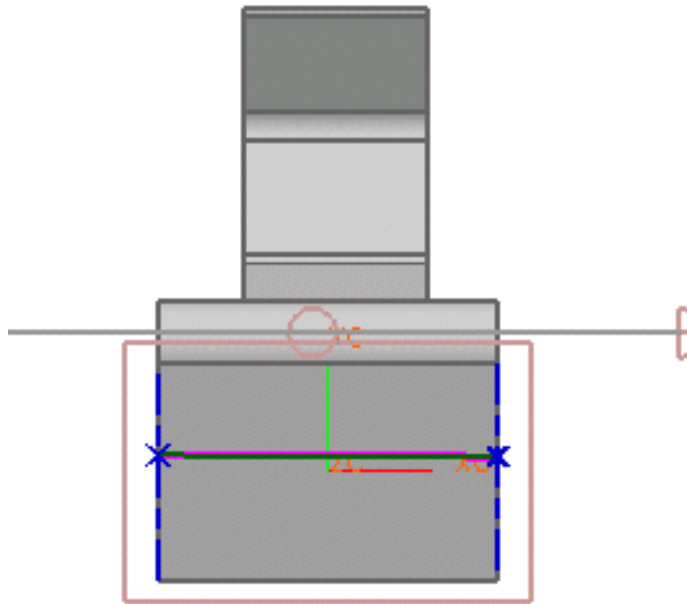


Create a Jog

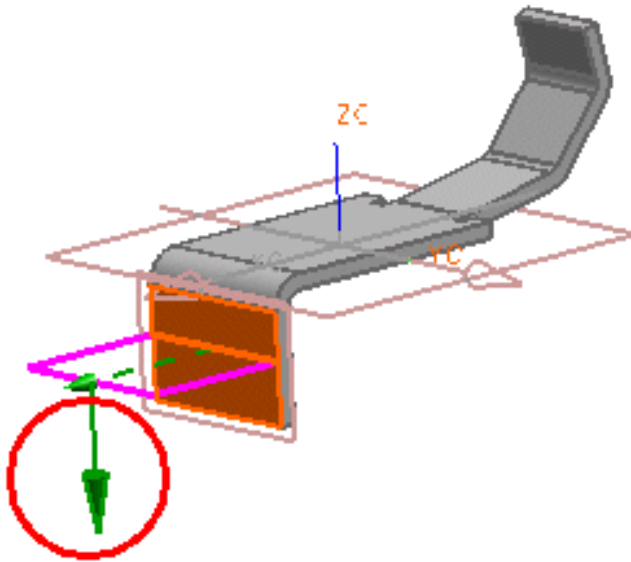
[Overview](#)[How To](#)[Options](#)

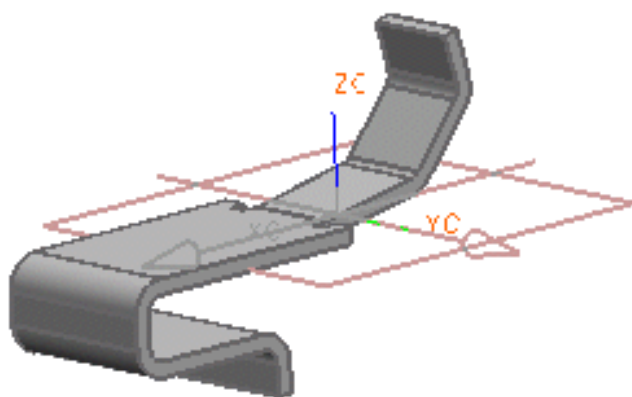
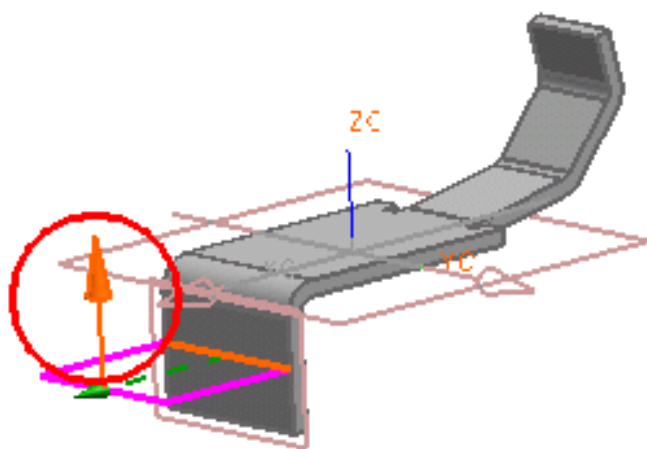
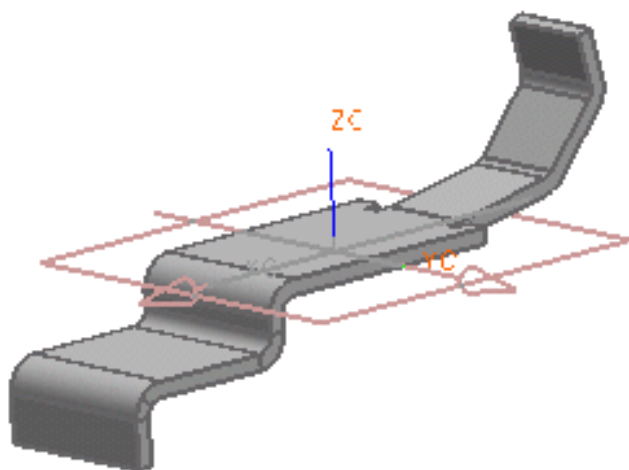
To create a **Jog** on a tab, or a planar region of a contour flange or lofted flange:

1. Click **Jog** .
2. Define the profile plane—select the face you want to jog on.
3. Draw a profile or pick one to serve as the **Jog Line**. The profile must be a single linear element.



4. If necessary, click on the drag handle to define the direction you want to jog. The drag handle should point toward the portion of the part you want to move. (The following figures illustrate the results produced by selecting different jog directions.)





5. Specify the height of the jog.
6. Click **OK** to finish the feature.



Jog options

[Overview](#)

[How To](#)

Options

Jog Properties

Height	Specifies the extent of the jog.
Reverse Direction 	Changes the direction of the jog from up to down and vice versa.
Reverse Side 	Changes the side of the part that moves to create the jog.
Height Reference	 • Inside applies the dimension from the selected face to the near side of the feature.  • Outside applies the dimension from the selected face to the far side of the feature.
Inset	 • 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.  • Bend Outside positions both the portion of the feature that's perpendicular to the profile plane and the bend 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](#).