

Starting the application and setting preferences

Choose **File**→ **New**, and on the **Model** page, select the **NX Sheet Metal** template.

The file opens in NX Sheet Metal.



























In the application, you should see the **NX Sheet Metal** toolbar.



If you do not see it:

1. Choose **Tools**→**Customize**.
2. On the **Toolbars** page, select the **NX Sheet Metal** box, then click **Close**.

Some of the NX Sheet Metal features have been grouped together in drop-down lists. For example, the bending features have been grouped together, as have the punch-type features. If you do not see a feature that you were expecting to see on the toolbar, click the little black triangle next to a feature button. The following graphic table shows the lists and the features they contain. Click on the links to go to the help for that command.

Tab	Bend	Corner	Punch	Cut	Resize	Form	Convert	Flat Pattern
Tab 	Flange 	Closed Corner 	Dimple 	Extrude 	Resize Bend Radius 	Unbend 	Convert to Sheetmetal 	Flat Solid 
	Contour Flange 	Three Bend Corner 	Louver 	Normal Cutout 	Resize Bend Angle 	Rebend 	Edge Rip 	Flat Pattern 
	Lofted Flange 	Break Corner 	Drawn Cutout 	Bend Taper 	Resize Neutral Factor 			
	Hem Flange 			Bead 				
	Bend 			Solid Punch 				



Additional Modeling commands for creating sheet metal parts are available. If you do not see the following toolbars on your screen:



Feature



Feature Operation

1. Choose **Tools**→ **Customize**.
2. On the **Toolbars** page, select the **Feature** and **Feature Operation** boxes, then click **Close**.

NX Sheet Metal provides default values for typical sheet metal settings such as material thickness, bend allowance, bend radius, and neutral factor. You do not have to set them for each feature that you create. If you change them, they remain valid for all features in that part file. (You can override them in individual commands.)

To see what these settings are, choose **Preferences**→**NX Sheet Metal**. This brings up the **NX Sheet Metal Preferences** dialog, which includes those settings already mentioned, along with other bend options, and flat pattern settings. See [Changing the defaults](#) for details.

For instructions on how to change these default values permanently, see [NX Essentials](#)→[Customizing NX](#)→[Using Customer Defaults](#) .