

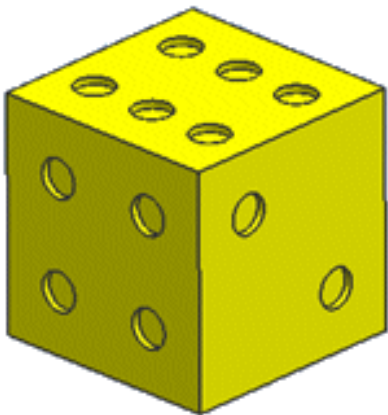


Sheet Metal from Solid overview

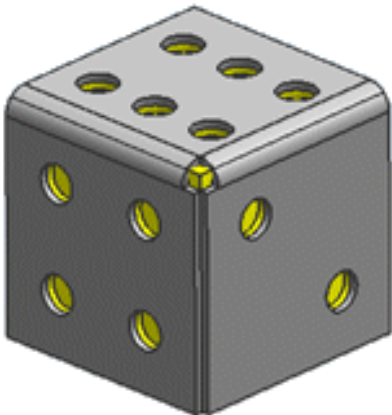
- Overview
- [How To](#)
- [Options](#)

Use the **Sheet Metal from Solid** command to create a sheet metal part using a solid to model it. The solid represents the inner void of an enclosure. You select the faces from the solid to be panel (web) faces, and edges between the faces to be bend regions. You specify bend properties, or use the ones you established in **NX Sheet Metal Preferences**. The software creates the model based on your inputs. You can then treat this part as you would any other sheet metal part.

Complex geometry can make creating individual flanges cumbersome. The **Sheet Metal from Solid** command greatly simplifies and streamlines this process. It also provides a type of workflow familiar to users of I-deas Sheet Metal.



Solid



Resulting sheet metal part

Where do I find it?

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



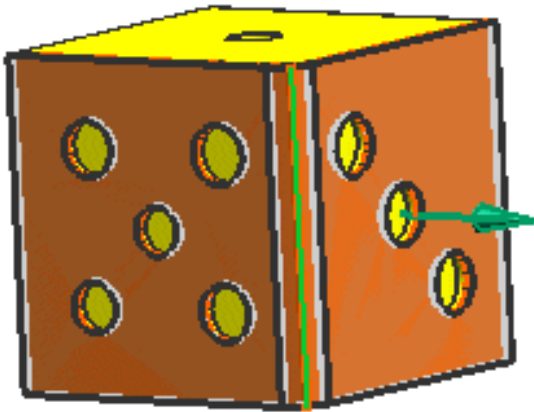
Create a sheet metal part from a solid part

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

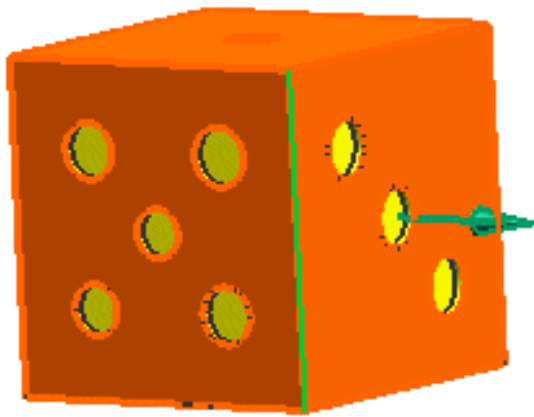
1. Create a solid part in Modeling. Surface features such as holes will be reflected in the sheet metal part.
2. Open NX Sheet Metal.

3. On the **NX Sheet Metal** toolbar, from the **Bend** list, select **Sheet Metal from Solid** , or choose **Insert**→**Bend**→**Sheet Metal from Solid**.
4. Select the web faces that you want to include in your sheet metal part. As long as there is only one logical bend option, you see a preview of the result. If you select faces where there is more than one choice of bends, you get the following warning: Ambiguous bend edges must be resolved. Select appropriate bend edges.

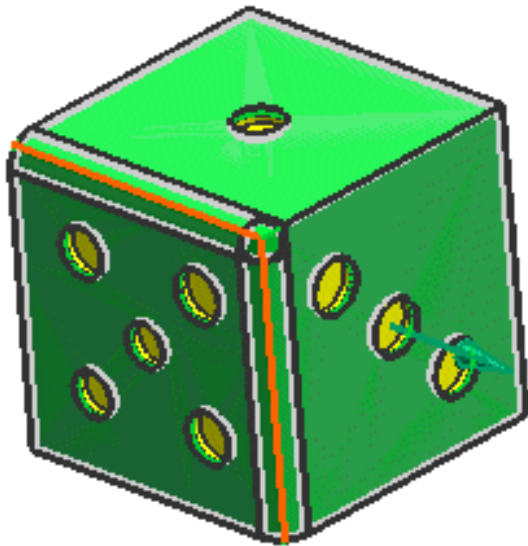
In this example, two faces are selected, and only one bend region is shared between them, so you see a preview.



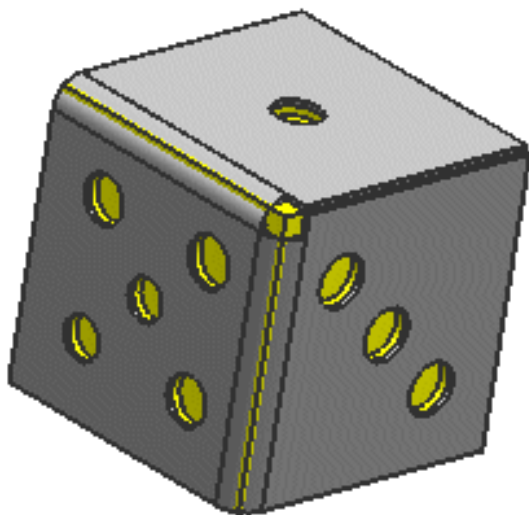
After the top face is selected, the preview disappears, and the warning message appears.



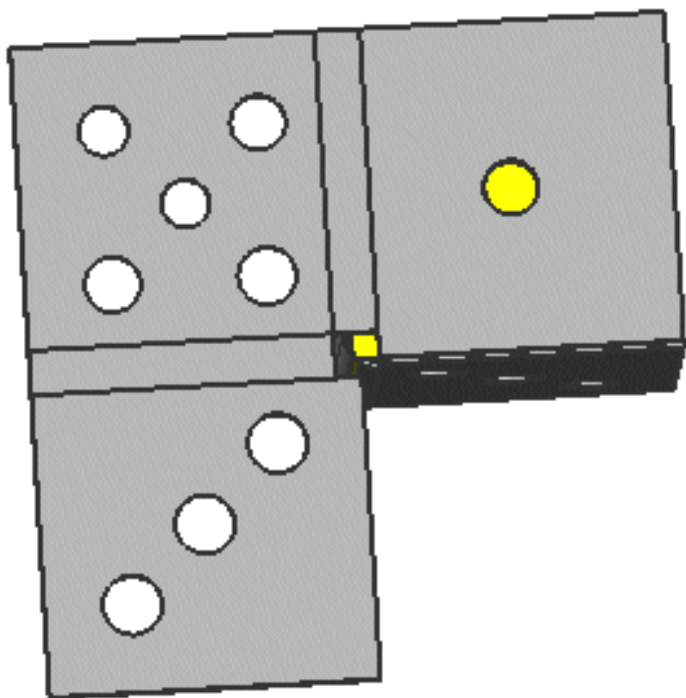
5. Select bend edges. If you select an appropriate combination of faces and edges (creating a part that can be flattened), the preview reappears.



6. By default, the sheet metal part is thickened on the outside of the solid model. Reverse the direction to thicken it in the opposite direction.
7. Click **OK** to create the sheet metal part.



Here is the flat solid view of the same part.





Sheet metal from solid options

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

Web Faces

Select Web Faces

Lets you select planar faces for your sheet metal part. The faces do not have to belong to the same body, but they must touch along linear edges. They can belong either to sheet bodies or to solid bodies.

Bend Edges

Select Bend Edges

Lets you select edges to be converted to bend regions. When possible, the software determines the bend edges automatically. If the location of bend edges is ambiguous, you must select the desired edges.

Thickness

Thickness

Specifies the thickness of the sheet metal part. To change it from the default, clear the **Use Global Value** check box.



Reverse Direction

Thickens the part in the opposite direction.

Preview

Show Result



Shows a preview of the result. Note that for performance reasons, the preview is an approximation only and may not exactly match the resulting part.

You can also modify some of the default [bend parameters](#). Some differences exist between **Sheet Metal from Solid** bend parameters and the default bend parameters:

- You cannot change the bend angle. It is inherited from the selected geometry.
- **Extend Relief** is not available.
- **Corner Relief** is not available.