
Sheet Metal Design with NX



- 1. Sheet Metal workflow**
- 2. Sheet Metal application**
- 3. Establish basic part characteristics**
- 4. Using Sheet Metal preferences and customer defaults**
- 5. Define the basic shape of the part**
- 6. Constructing base features**
 - a. Tab
 - b. Flange
 - c. Contour Flange
 - d. Lofted Flange
 - e. Hem Flange
 - f. Jog
 - g. Bend
 - h. Unbend
 - i. Rebend
 - j. Sheet Metal from Solid
- 7. Sheet Metal corners**



8. Sheet Metal cutouts

- a. Normal Cutout
- b. Bend Taper

9. Sheet Metal deform features

- a. Dimple
- b. Louver
- c. Drawn Cutout
- d. Bead
- e. Solid Punch
- f. Gusset

10. Flat Solid and Flat Pattern

11. Advanced Sheet Metal commands

- a. Bridge Bend
- b. Advanced Flange
- c. Uniform
- d. Reform

12. Analyze Formability – One step



13. Working with non-sheet metal data

14. Resizing the parameters

- a. Resize Bend Radius dialog box
- b. Resize the bend radius of a flange
- c. Resize Bend Angle
- d. Resize Bend Angle dialog box
- e. Resize the bend angle
- f. Resize Neutral Factor
- g. Resize Neutral Factor dialog box