603 - Sheet Metal Modeling in Solid Edge: A Hands-on Experience

Steve Webb, Solid Edge Field Support, Siemens PLM Software
Agenda: 603 - Sheet Metal Modeling in Solid Edge: A Hands-on Experience

Who am I?
What you will learn
Hands On
Benefits of this topic
Enhancements
Sheet Metal Tips and Tricks
About: Steve Webb

Steve Webb
Solid Edge Field Support
Siemens PLM Software

Steve is an Applications Engineer in the Solid Edge Field Support group. He has over 25 years of experience in Mechanical Design Engineering and Mechanical CAD. He has held roles as an Application Engineer and Educator at Siemens PLM Software, PTC, Saratech Inc and Moderntech SI. Before moving into the CAD Industry, Steve applied his CAD knowledge in industries including hydraulic gear pumps, high speed packaging equipment, and textile equipment. He holds a two year drafting certification from the Williamston Career and Technology Center.
What you will learn

Does your company design sheet metal parts? Do you need a better understanding of how to model those parts using sheet metal design? If so, then this hands-on session is for you. We will walk through step-by-step modeling sheet metal using Solid Edge ST6. We will look at tools to help you create stamped parts, add deformed features across bends, place multiple deformed features at one time, and create sheet metal features on part models.
Punch (Emboss) in Sheet Metal & Part – Hands-On

- Open T-00001_A.asm
Punch (Emboss) in Sheet Metal & Part – Hands-On

- The Emboss command provides functionality for adding protruding or indented emboss feature to a sheet metal or part (uniformed thickness) body.

- Embossing is using Boolean operation for combining bodies with clearance and thickness.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Change display configuration to 01-Embossed. Use Radial menu.

- Apply and Close.
• Right mouse click T-00110-11_A.par and open file.

• Point out that this is a solid part model.

• Rotate model to show bottom side as a solid.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Add a multi-body and choose sheet metal body.
  - Type “Tool” for the Initial body name.
  - Type “Tab” for the New body name.

- OK
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Select the Tab command from the radial menu.
- Pick the bottom face of the part.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- In the Draw toolbar select Incline and Include with offset.

- OK

- In the command bar select Single Face and pick the main face.

- Right mouse click to accept.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Offset distance 19 to the outside.
- Pick Close sketch.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Thickness 3mm

- Add thickness going into the model.

- Finish and Cancel.
— Punch (Emboss) in Sheet Metal & Part - Hands-On —

- Under the Dimple command choose Emboss.
- Pick the Tab and then pick the Tool Body (red part).
- OK and ESC key.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Definitions

- Target Body – body to which the embossing feature is added to.
- Tool Body – body of the embossed shape that’s added to or subtracted from the target body.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Definitions

- Clearance - is offset applied to the tool body prior to modifying the target. Clearance body is subtracted from target body. This is a straight subtract.

Clearance – offset value in yellow
Punch (Emboss) in Sheet Metal & Part - Hands-On

• Definitions

• Thickness – is offset from the clearance value. The thickness adds solid material to the target.

![Diagram showing Clearance (yellow) and Thickness (blue)]
Punch (Emboss) in Sheet Metal & Part - Hands-On

• Emboss command for part environment is located under the “Thin” command for both Synchronous and Ordered.

• Emboss command for sheet metal environment is located under the “Dimple” command for both Synchronous and Ordered.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Option
- Target body
- Tool body
- Thicken
- Flip direction – only if Thicken is on
Punch (Emboss) in Sheet Metal & Part - Hands-On

Thicken - Off

Thicken - On
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Emboss Options
Punch (Emboss) in Sheet Metal & Part - Hands-On

Show/Hide Tool Bodies
- Show tools if they are Design bodies
- Show tools if they are Construction bodies
Punch (Emboss) in Sheet Metal & Part - Hands-On

Sheet Metal Rounding

- Include die-side rounding
  - Die: 2.00 mm
- Include punch and punch-side corner rounding
  - Radius: 2.00 mm
Punch (Emboss) in Sheet Metal & Part - Hands-On

Sheet Metal Rounding

- Include die-side rounding
- Die: 2.00 mm
- Include punch and punch-side corner rounding
- Radius: 2.00 mm

Sheet Metal Rounding

- Include die-side rounding
- Die: 2.00 mm
- Include punch and punch-side corner rounding
- Radius: 2.00 mm

Unrestricted © Siemens AG 2013 All rights reserved.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Go to the Tool body RMC and Toggle to Construction.
  - If this is not toggled to construction the embossed feature will not show up in the assembly file.

- Hide Tool body
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Ctrl+Tab to switch to assembly to show update.
  - Point out we need to fix the back corner of the part.

- Ctrl+Tab to switch to part and clean up end of part….one side only.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Under Hole command choose the Cut command.
- Pick the tabbed face.
- Incline these edges.
- Then add these lines to close.
Punch (Emboss) in Sheet Metal & Part - Hands-On

• Close sketch.

• Choose Through All and point down

• Finish and cancel

• Ctrl+Tab to switch to assembly to show update.
Punch (Emboss) in Sheet Metal & Part - Hands-On

- Deformed feature creation remain unchanged. To create deformed features across bends, user needs to unbend the desired bends and create deformed feature as usual in flatten state.

- Once deformed feature is created, user needs to rebend to see the feature go across the bend.

- For Ordered sheet metal modeling only.
• Go to this part T-00325_A.psm and in-place edit.

• Under Bend command select Unbend.

• Pick the big back face as the face to unbend about.

• Collect all bends and pick the part.

• Finish and Cancel
- Select the bead command and pick the same big face.

- Draw a line from the center of the hole down to the endpoint of the bottom cutout.

- Add 15mm dimension as shown.
Deform Sheet Metal Features Across Bends - Hands-On

- Mirror sketch line about the center of the cutout.

- Close sketch

- Pull to the outside

- Finish and cancel
Deform Sheet Metal Features Across Bends - Hands-On

- Go into the Rebend command.
- Collect all bends
- Ok, Finish and Cancel
- Close and return
• Rules for Beads, Dimples, Drawn Cutout

• Inside bend - the depth of the bead created or edited across the bend must be smaller than the associated bend radius.

• If the bead depth is greater than the bend radius then the feature will fail as it intersects in rebend.

• If the bead traverse across three bends; 6 mm, 10mm and 12 mm, then the bead depth should be less than the 6 mm.
Deform Sheet Metal Features Across Bends - Hands-On

• Louver traversing across bends will not be supported.

• Louver has to lie completely inside bend region.
Multiple Profile Dimples - Hands-On

- Multiple profiles can be created on the same planar face of the part.
- Only Dimples, Drawn cutouts and Beads are supported.

- All profiles for Dimples and Drawn cutouts need to be closed.

- Multiple Louvers may not be created at the same time.
Multiple Profile Dimples - Hands-On

- In place edit T00286_A.par

- Applications button and switch to sheet metal.

- Right mouse click in graphics window and Transition to Ordered.
Multiple Profile Dimples - Hands-On

- Go into Drawncut command
- Select from sketch and pick all 4 circle sketches.
- Enter a distance on 12mm
- Finish and Cancel
- Close and Return
Sheet Metal Features on Parts - Hands-On

- Deformed features supported
  - Dimple
  - Bead
  - Drawn cutout
  - Louver
  - Etch
  - X-Break
  - Gusset
Sheet Metal Features on Parts - Hands-On

- Change display configuration to 02-Fender.
- In-place edit T-00801_A.par which is the Fender.
- Point out that this is a part model and we want to add sheet metal features to the model.
Sheet Metal Features on Parts - Hands-On

- Sheet Metal Features Anywhere is only in ORDERED mode.

- Sheet Metal Features Anywhere allows sheet metal features to be placed on uniform thickness planar faces of regular Solid Edge parts.
• Applications button and Switch to sheet metal.

• Go into the Contour Flange command

• Pick this edge

• Place plane at this end.
• Create this sketch with these dimensions.

• Close sketch

• Go into option and change bend radius to 6mm.

• Choose contour flange chain and pick the arc.

• Finish and cancel
Flanges and Contour Flanges can be created off of any linear edge of the part.
Sheet Metal Features on Parts - Hands-On

- Rotate around to back of fender.

- Add a flange 60mm across bottom edge with bend outside.

- Close and return.

- Change Display Configuration to all parts.
Sheet Metal Features on Parts - Hands-On

- Other areas supporting sheet metal features on a part:
  - Jogs
  - Bend Reliefs
  - Hems
  - Tabs
  - Close Corners – on sheet metal features
  - Break Corner - on sheet metal features
  - Etching
  - X-Break
  - Normal Cutout
- No support for sheet metal features on a part:
  - Lofted Flange
  - Mid-Surface
Sheet Metal Features on Parts - Hands-On

- Other areas supporting sheet metal features on a part:
  - Feature Library
  - Cut/Copy/Paste
  - Save As Flat/DXF
  - Bend Tables
- No support for sheet metal features on a part:
  - Move to Sync
Sheet Metal Features on Parts - Hands-On

• Basic rules for parts with sheet metal features:
• Deformed features only support uniformed thickness and planar faces that are parallel to each other.
• Etch will work on any planar face.
• Gusset works on any cylindrical face which has a concentric counterpart.
• Users are NOT required to transform the part to SM either partially or fully.
• If uniformed thickness is not found then the feature will fail with appropriate error message.
Benefits of this topic

Embossed features – allows the user to create multi-body parts from parts, sheet metal or construction bodies. Reusable tool bodies allow for quick and easy reuse in other designs.

Deformed features across bends - common in stamped component design and ST6 easily handles beads, dimples and drawn cutouts across bends.

Multiple deformed features – creating multiple deformed features at one time has been a restriction in other releases of Solid Edge.

Sheet metal features anywhere – simplifies the design allow you to create simple features in the least number of steps by using tools like flange or contour flange commands.
Enhancements

- The following enhancements are for ORDERED only.
- Flanges on all linear edges excluding thickness edges.
  - Edge thickness example: edge used to break a corner.
  - No support for flanges on an arc.
Enhancements

- The following enhancements are for ORDERED only.
  - Contour Flange on a Contour Flange
  - Contour Flange on all plate/bend face edges.
Enhancements

- Flange on linear bend edge for deformed features and rolled part.
- Any linear edge of a deformed feature or rolled part can be used to create flanges.
- When creating a flange from these types of parts the only bend option will be Bend Outside.
- Only system defined reliefs will be supported.
Sheet Metal Tips and Tricks

• Appling Break Corners to a Synchronous part in Sheet Metal file.
  • After inserting the Sheet Metal file into another, or new Sheet Metal file with the Part Copy command, you will have a "Part Copy" in the feature tree. The part copy does not have sheet metal attributes until you "Transform to Sheet Metal". Once you Transform to Sheet Metal you will be able to apply the Break Corner to the required thickness face edges.

• Why is an inserted a sheet metal file as a Part Copy not showing the chain link in pathfinder?
  • You won't get this link if the file you inserted the Part Copy into is set to Synchronous. You will have to transition to Ordered first.
• Why are Etch Features not displayed in draft?
  • Etch features are considered Constructions, and are turned off by default in drawing views. To display an Etch in draft, right mouse click on the drawing view and select Properties -> Display. You should see 4 buttons between the tabs at the top of the dialog and the Parts List (tree). The 3rd button from the left is Parts List Options, click the Parts List Options and then select "List Constructions". You will then see an Etch feature in the Parts List. Select the Etch feature, check the Show checkbox, and then OK, update the drawing view.
• How can I show deformed Sheet Metal features in Draft?
  • When placing a flat pattern in Draft you can automatically show the deformed feature’s profile and origin. However, to enable this you must set the correct options in the model .psm file. In Solid Edge Options select Flat Pattern Treatments. Under the section “Formed feature display” you can select “As feature loop and feature origins”. Be sure to save the file.
Thank You

Steve Webb, Solid Edge Field Support, Siemens PLM Software