Converting a Solid Body to Sheet Metal
About Myself

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I am a GTAC Application Engineer with a total of over 41-year’s experience in engineering design & CAD. I spent 17-years in the mechanical design, drafting, and CAD administration fields, prior to joining Intergraph Corporation in 1991, now Siemens PLM Software. Over the past 25-years I have divided my time with Solid Edge classroom training, assisting in the planning and implementation of the Solid Edge Sheet Metal and Draft environments, initial implementation of the Solid Edge Reseller program, and the past several years providing customer support with the Solid Edge GTAC team. I am located in the Siemens PLM office, Huntsville, AL
About this Session

Converting a Solid Body to Solid Edge Sheet Metal

• Have you ever had a native Solid Edge or imported solid model that looks like a sheet metal part, but won’t convert to a sheet metal part? It can happen.

• In this session we’ll share examples of how to make these cases work! We’ll also combine a synchronous technology and ordered modeling approach, known as hybrid, to get the optimum and most efficient results.

• Also, as promised from my session last year, I’ll demonstrate how to implement Multi-Body functionality into your conceptual sheet metal and part design approaches. Multi-Body modeling provides the ability to design multiple part bodies in the same file and publish them later as individual part files and assemblies if desired.
Something to ponder…

• The “ultimate objective” of this presentation is to provide you with some forethought on how you might apply what is shown today, to your sheet metal design needs.

• “No method or approach to modeling is wrong, as long as you accomplish your modeling goals.”

• It is hoped that what we cover today will broaden your “toolbox” of modeling techniques and perhaps provide you with a more “efficient” approach to your future designs!
The Session Agenda

- Synchronous or Ordered? A review
- Importing a simple part that won’t convert to sheet metal and utilizing a basic approach to investigate why and make the required repairs, so it will work
  ✓ Importing a second part, a bit more complex and illustrating a second, perhaps more efficient approach to converting to sheet metal
- How to utilize the power of Synchronous features by converting cutouts to holes (a can’t do in Ordered)
- Design a mating sheet metal part within the context of the conceptual design file, with Add Body (Multi-Body)
  ✓ Use Hybrid approach of using Part to Sheet Metal
  ✓ Start with Sync, to Ordered and back to Sync again
  ✓ Publish the parts and create an assembly
Synchronous/Ordered & Hybrid…the When & Why

Sheet Metal Design

Synchronous

Ordered
Synchronous/Ordered & Hybrid…the When & Why

Sheet Metal Design

Synchronous

Ordered

Hybrid

Best of Both Worlds
Synchronous
• Sketches not needed for editing (restorable)
• Great for Imported parts/assemblies (editing)
• Quick Creation & Editing (push/pull/rotate)
• Can make Sync modifications from the assembly
• Thin Part to Sync SM (command)
  ✓ Transforms the model to native tabs and bends
  ✓ Model can be edited like native SM part
  ✓ Bend parameters can be modified
  ✓ Thickness can be modified
• Only supports Linear brake bends
• Recognize Holes

Hybrid
• Provides user opportunity to get familiar with Synchronous techniques when applicable
• Integrated modeling (Using a combination of sync and ordered to incorporate strength of each)
• Start with Synchronous, Switch to Ordered if needed
• Part to Sheet Metal (Ordered only)
• Unbend/Rebend capability
• Rolled Parts (Lofted Flange/Contour Flange along arcs, stand-alone bends)
• A method to use the convenience of Sync yet overcome Sync restrictions

NOTE: Move Ordered features to Synchronous, but not Synchronous features to Ordered (…no history/sketches for editing)

Ordered
• Sketches retained for editing
• Superb for stamped or die formed parts
  ✓ Part features can be used
• Part to Sheet Metal (Ordered only)
• Thin Part to Sheet Metal (as in Sync)
• Unbend/Rebend capability
• Place stand alone bends (rolled parts)
• Cuts and Holes on bends
• Three Bend Corners
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Importing a Simple Part – Method #1

Importing a STEP file from another system, or legacy data:

• Attempt converting to sheet metal
• If fails, interrogate model
  ✓ Geometry Inspector for geometry errors/Optimize maybe?
• Use Smart Measure to find inconsistencies in model looking for:
  ✓ Non-uniform wall thickness
  ✓ Non-concentric cylinders, causing non-uniform wall thickness
  ✓ Uniform wall thickness needed for Thin Part to Sheet Metal to work
• Edit the necessary cylinders to make them concentric
• Convert with Thin Part to Sheet Metal command
• Let’s do it…DEMO TIME
Importing a Simple Part – Method #1

Use **Thin Part to Sheet Metal** to convert Part Body to Sheet Metal

![Diagram showing the process of converting a part body to sheet metal](image)

**Warning:** The part contains cylinders that are non-concentric. These areas of the model will be defined as deformation features in sheet metal.

**Fails...Why?**
Importing a Simple Part – Method #1

Use **Smart Measure** to find inconsistencies & **Geometry Inspector/Optimize** to further Interrogate the Model.
Importing a Simple Part – Method #1

Use **Select** to Edit in Synchronous
Importing a Simple Part – Method #1

Use **Resize Rounds** to Edit in Ordered
Importing a Simple Part – Method #1

Use **Thin Part to Sheet Metal** to convert Part Body to Sheet Metal
Importing a Simple Part – Method #1

Use **Recognize Holes** to convert Cutouts to Holes with Hole Attributes.
Importing a Simple Part – Method #1

Importing a STEP file from another system, or legacy data:

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- If fails, interrogate model
  - **Geometry Inspector** for geometry errors/**Optimize** maybe?
- Use **Smart Measure** to find inconsistencies in model looking for:
  - Non-uniform wall thickness
  - Non-concentric cylinders, causing non-uniform wall thickness
  - Uniform wall thickness needed for **Thin Part to Sheet Metal** to work
- Edit the necessary cylinders to make them concentric
- Convert with **Thin Part to Sheet Metal** command
Importing a More Complex Part – Method #2

With Method #1, editing all inconsistencies previously shown, works well for small and simple models.

For models with more complexity we can apply a simpler method, perhaps making Method #2 faster and more efficient.

- The model interrogation is still required
  - Still need to know what is causing issues (if we can’t what?)
- With Method #2 you will focus on interrogating either all inside faces (rounds), or outside faces (rounds).
  - NOTE: You’ll have to determine whether you’ll work with all inside faces of the part, or outside faces, due to the method that will be used to repair the model for conversion to sheet metal.
- Let’s do it…DEMO TIME!
Importing a More Complex Part – Method #2

Use **Thin Part to Sheet Metal** to convert Part Body to Sheet Metal

Fails...Why?
Implementing an Alternate Approach to Repairing Model

• Due to error messaging received, may have to interrogate model in other ways, maybe Geometry Inspector?
  ✓ If still not successful we can attempt the following…
  ✓ Are the discrepancies with non-sheet metal faces? If yes edit:
  ✓ By finding, repairing, or removing those faces, or…
  ✓ …use the Copy Surface (Chain option) to copy the desired inside, or outside faces of part
Implementing an Alternate Approach to Repairing Model

- **Delete** the Part Body after faces are copied
- **Thicken** the copied faces
- Can now delete the copied faces, if desired
- Convert to sheet metal using *Thin Part to Sheet Metal*
- So…what happened?

Copied Faces

Thicken (to outside)
Importing a More Complex Part – Method #2

What was the problem with the Imported Part Body? Why wouldn’t it convert to sheet metal?

- There was a small chamfered face on the edge of a flange that was not detected
- An issue because it didn’t ruin the integrity of the model not being a valid solid (thus won’t show in Geometry Inspector)
- It did however, prevent the conversion to sheet model, mentioned in the messaging that was received when first trying to convert
Importing a More Complex Part – Method #2

When to use Method #1 in place of Method #2

- Generally due to complexity of the part to be converted
  - Consider number of rounds that will be converted to bends
  - Less part interrogating, only need to check inside faces, or outside faces of part, not both
- Of course can be used on smaller parts too
  - Must decide if it’s easier just to make quick measures and edits on smaller part vs. the additional steps in Method #2
Importing a More Complex Part – Method #2

Interpret the Messaging

Led to Method #1

- The part contains cylinders that are non-concentric. These areas of the model will be defined as deformation features in sheet metal.

Led to Method #2

- The input model contains non-sheet metal faces that must be removed before transforming.

Can be one, or the other, or a combination of the issues you’ve seen and perhaps even more.

- Interrogate
- Repair
- Convert
With Method #1, editing all inconsistencies previously shown, works well for small and simple models.

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  - With Method #2 you will focus on interrogating either all inside faces (rounds), or outside faces (rounds).
  - NOTE: You’ll have to determine whether you’ll work with all inside faces of the part, or outside faces, due to the method that will be used to repair the model for conversion to sheet metal.
Utilizing Multi-Body in Sheet Metal Design

Multi-Body is really the Add Body command that is available in Part & Sheet Metal environments and both Synchronous and Ordered

- Add additional solid bodies into a single part or sheet metal file
- Utilize Boolean commands between the bodies…union, subtract, intersect and split
- Publish to individual part files and/or assembly if desired
- Have a mix of part and sheet metal bodies in the same file
- GREAT conceptual design tool
- Only one part can be active at a time

A Word of Caution
- If you do not publish the parts and place the master file in an assembly, you’ll get one single item in the parts list, may not be what you expected
Utilizing Multi-Body in Sheet Metal Design

Use Add Body to add a new Sheet Metal part to existing file

- In this case we will add a Part body to the file?
- Going to quickly develop the envelope for the new part using Synchronous techniques in the Part environment (Switch to)
- Place a Synchronous feature with the Box command
- Quickly edit the sides of the box by making the sides planar to the inside faces of the existing sheet metal part
  - This provides part envelope
Utilizing Multi-Body in Sheet Metal Design

The new part envelope is now in place

- Up to this point no sketch elements have been drawn
- Place two lines to create a region to lower the center portion of the part
Utilizing Multi-Body in Sheet Metal Design

Convert the current part body to sheet metal
- Can **Switch To** sheet metal again, or do the same thing in part
- Use the **Part to Sheet Metal** command to convert the part body
  - Only available in Ordered
  - Can apply special case closed corners during the operation
  - Determine material side direction (this case will be inside)
- Now have a sheet metal part
Utilizing Multi-Body in Sheet Metal Design

New Part is complete
- Can now add holes, additional features at this time
- Can only have one Flat Pattern in a file at a time
  ✓ Another reason to Publish the parts
Utilizing Multi-Body in Sheet Metal Design

Multiple Bodies in Design File

- PathFinder now has multiple Design Bodies (can be renamed)
Utilizing Multi-Body in Sheet Metal Design

Multi-Body Publish

• Generates individual part files from the multiple bodies
Utilizing Multi-Body in Sheet Metal Design

Multi-Body Publish

- Specify what files to Save, or all and also create an Assembly

Select a new Path
Utilizing Multi-Body in Sheet Metal Design

Multi-Body Publish

- Assembly that was Published
- Parts are Grounded
- Assembly and Parts linked to Master File
Utilizing Multi-Body in Sheet Metal Design

In this case

- Used Add Body to add a Part (body) file
- Though the end result was sheet metal, started in part to utilize Synchronous geometric shape command (Box) to generate part envelope
- Sized the envelope to existing part
- Switched to Ordered to make use of Part to Sheet Metal command, converting the part body to sheet metal
- Add additional features at this time
- Published parts to individual files and to generate an assembly of the parts
Converting a Solid Body to Sheet Metal

Thank you and... *Enjoy SEU 16!*

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