Sheet Metal Design: Synchronous or Ordered, the When, Why & More
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I am a GTAC Application Engineer with a total of over 40-years’ 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 24-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

Sheet Metal Design: Synchronous or Ordered, the When, Why & More

Yes…there are differences in choosing Synchronous, or Ordered methodology when it comes to working in Solid Edge Sheet Metal. We’ll examine what to consider when making that choice, or when you might choose a hybrid approach. We’ll also take a closer look at some of the newer commands in Sheet Metal and how they might best be put to use for your designs.
About this session

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!
Agenda

- Synchronous or Ordered
- Real world look at Synchronous/Ordered or Hybrid in Progressive Die Design
- A Design in Action
- Designing an Enclosure in Assembly
- A deeper look at Blank Body Command
- A deeper look at Part to Sheet Metal Command
- Questions
Synchronous or Ordered?

Sheet Metal Design

Synchronous

Ordered
Synchronous...Ordered...or Both?

Sheet Metal Design

- Synchronous
- Ordered
- Hybrid
## Synchronous/Ordered & Hybrid… the When & Why

### 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
- 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

### 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
  - Unbend/Rebend capability
  - Thin Part to SM – Ordered
  - More tolerant than Transform to Sync SM
  - Place stand alone bends (rolled parts)
  - Cuts and Holes on bends
  - Three Bend Corners

### Hybrid
- Provides user opportunity to get familiar with Synchronous techniques when applicable
- Integrated modelling (Using a combination of sync and ordered to incorporate strength of each)
- Start with Synchronous, Switch to Ordered (cannot go back to Synchronous after switching)
- 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
Synchronous/Ordered & Hybrid…the When & Why

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Hybrid Modeling Approach in Progressive Die Design

Hybrid method used in design of this Progressive Part Strip & Die

- Modify where needed after importing in Sync
- Convert to Ordered for use of Part modeling features like Replace Face and Unbend/Rebend in Ordered Sheet Metal
Progressive Die & Press – A Design in Action

Tooling and Parts Designed Exclusively in Solid Edge

- 4400 Ton Press
- Parts above generated on this press

http://www.summitsedge.com/
Design an Enclosure – Hybrid Approach in Assembly

Applying the Hybrid process to the development of a Sheet Metal Cover

Synchronous Only
or
Hybrid (Sync/Ordered)

Either can produce the same result

DEMO_2
A Deeper look into Blank Body Command

NOTE: Solid Edge ST8’s focus was not on new Sheet Metal functionality. Today we will delve deeper into the new “bigger bang for the buck” commands from ST7.
A Deeper look into Blank Body Command

Blank Body Command...can flatten most anything, as in die formed parts
A Deeper look into Blank Body Command

- For parts that are normally drawn (die-formed) sheet metal and generally do not consist of many, if any at all, linear or brake bends during manufacturing.

- Though this command is normally used for sheet metal parts, it is most useful for parts that we cannot create in Solid Edge sheet metal environment, because of the complex surfaces present in the model.

- Too much material in blank better than not enough…can always trim part after forming.
A Deeper look into Blank Body Command

Command ribbon shown in .PAR environment

Command ribbon shown in .PSM environment (in addition to Flatten)
A Deeper look into Blank Body Command

Also on Surfacing tab in .PAR
A Deeper look into Blank Body Command

Quick Bar

1. **Options Dialog** – Cut size, Formability, Alarm
2. **Blank Body Accuracy** – FEMAP mesh accuracy (process time +/-)
3. **Thickness of Blank** – only available in PAR, Mtl. Thk. used in PSM
4. **Offset Input faces** – in PAR, emulates neutral factor in PSM file
5. **Remove Loops** – inner holes cutouts removed in blank results
   (* = if loop intersects multiple faces it will not be removed, consider suppressing the features affected by this)
A Deeper look into Blank Body Command

Quick bar while in .PAR & .PSM

Unavailable in .PSM, uses Mtl. Thk.

Quick bar while in .PAR (Blank Surface)
A Deeper look into Blank Body Command

Blank Options dialog

- When in Sheet Metal PSM file
- Late entry, will eventually move to Material Table
A Deeper look into Blank Body Command

• Blank Body uses FEMAP for meshing and provides output to our solver

• If a material has not been assigned to the part prior to using the command, you’ll be prompted to so, before continuing in the command

• When using Blank Body in a .PAR file you must apply a material thickness

• When in a .PSM file, Blank Body will use the material thickness already assigned in the Gage/Material Table

• Linear bends will not have bend lines if Blank Body is used as in Flat Pattern

• Blank Body exits in Synchronous and Ordered environments
A Deeper look into Blank Body Command

Steps in .PAR & .PSM file:

- Select Blank Body command
- Assign Material, if not already applied (required)
- Review Options dialog
- Select Accuracy of mesh, course to fine
- Apply Thickness and Offset
  - Thickness not needed in .PSM, determined by Material Thickness, can apply an Offset (optional)
A Deeper look into Blank Body Command

Steps in .PAR & .PSM file (continued):

• Remove or leave internal loops
• Select Faces in model (note prompts)
• Preview, then Finish

Copy results to clipboard
*Can find blank dimensions in Variable Table

• If necessary, Edit Definition can be used after completion of the blank
A Deeper look into Blank Body Command

DEMO_3

DEMO_4

DEMO_5
A Deeper look into Blank Body Command

In a sheet metal file, it’s best to use Flatten whenever possible, unless Blank Body is needed as shown with die formed parts...why?
A Deeper look into Blank Body Command

In Part File - Surface

Formed

Blank
A Deeper look into Blank Body Command

In Draft

Use the same Flat Pattern option on the Drawing View Wizard in Draft for the Blank Body view. Blank does not contain bend lines.
A Deeper look into Part to Sheet Metal Command

Part to Sheet Metal Command

DEMO_6
A Deeper look into Part to Sheet Metal Command
A Deeper look into Part to Sheet Metal Command

- Mainly for parts modeled in .PAR initially, then desired to be sheet metal, or parts modeled in .PAR because they’re too difficult to model in sheet metal.

- Or for Part Bodies in Sheet Metal environment for converting to sheet metal.

- Quickly able to generate part/shape in .PAR file, then easily convert to sheet metal with options for changing bend radius and closed corners and more.

- When selecting edges you are either adding bends, or ripped edges, note prompting!

- To deselect edges that have been selected as bends, or ripped, SHIFT+CLICK on that edge.

- Rip option seldom needed as command capability has exceeded original expectations and practically eliminated need for it, so will often be done automatically where needed.
A Deeper look into Part to Sheet Metal Command

- Command can create non-manufacturable conditions, like overlapping flanges in the flat; but that is no different then in past versions

- Command will fail by design if the necessary bends, or ripped edges cannot make a solid, or generates disjoint condition

- If used in .PAR environment, part will maintain a .PAR extension on the file name

- Will not create bends on non-linear edges
A Deeper look into Part to Sheet Metal Command

Command ribbon shown in .PAR environment

- Only available in Ordered
A Deeper look into Part to Sheet Metal Command

Command ribbon shown in .PSM environment
• Only available in Ordered
A Deeper look into Part to Sheet Metal Command

Command bar in either .PAR, or .PSM

Part to Sheet Metal Options dialog
A Deeper look into Part to Sheet Metal Command

Part to Sheet Metal Options dialog

- **Keep body**: Retains the design body of the input model. If you “keep” the original body you can modify it, to edit the resulting sheet metal body.
- **Remove internal loops**: Removes interior loops from the selected faces.
A Deeper look into Part to Sheet Metal Command

Basic steps for using Part to Sheet Metal:

- Are you in Ordered model mode
- Select the Part to Sheet Metal command
- Review Options dialog
- Select edges for bends, or to be ripped
  - Can also deselect edges w/SHIFT+CLICK
- Define Corner options
  - Click red dots to override corner properties
- Right Mouse Click to Finish
A Deeper look into Part to Sheet Metal Command

Once Part to Sheet Metal is completed you can move the feature from Ordered to Synchronous
A Deeper look into Part to Sheet Metal Command

Edit individual bends on the fly…

Bend Edge

Ripped Edge

Material Thickness (global)

Flip material side

Corner Selection

Click to override Bend properties.

Bend radius: 0.038 in
Relief
- Square
- Round
Depth: 0.000 in
Width: 0.038 in
Neutral Factor: 0.330

None
- Open
- Closed
- Circular Cutout
- U-Shaped Cutout
- V-Shaped Cutout
- Square Cutout
- Miter
Thank you…Questions?

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