Solid Edge Sheet Metal

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Agenda

- Who am I
- What will you learn
- Solid Edge Sheet Metal capabilities
- Demonstrations
- Questions
About: Ganapathy Kunda

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Gan is a senior member of the Solid Edge team with 18 years under his belt. Gan spent the first 15 years in Solid Edge software development until ST3 working on Part, Surfacing, Sheet Metal, Frames, Harness and Synchronous Technology. Gan transitioned to Planning in 2010 to help track Solid Edge technology requirements and coordinates with Open Tools teams, Parasolid and D-Cubed for specific technologies in addition to looking at Sheet Metal and Sketching areas. In addition Gan acts as a technology consultant for various internal teams. Gan holds a Masters in Mechanical Engineering from India and MBA from Vanderbilt University.
What will you learn

This session will show the fundamentals and capabilities of Sheet Metal modeling in Solid Edge along with ST7 new functionality. Attendees will learn the common concepts from basic Sheet Metal modeling to flat pattern creation. Techniques and workflows will be shown for creating Sheet Metal designs in part documents, part features in Sheet Metal documents, how to get comfortable with designing Sheet Metal models using Synchronous Technology and integrated modelling. Synchronous users that attend this class will learn about the powerful ordered features and when to use them. Traditional users will be exposed to the Synchronous environments so that they may understand when to convert their systems, models, or features to the new paradigm.
Topics

• Introduction
  • Solid Edge Sheet Metal
  • Sheet Metal Features
  • Flatten

• Sheet Metal Capabilities - Basic
  • Create Sheet Metal
  • Transform to Sheet Metal

• Options
  • Defining Gage
  • Defining Bend Properties
  • Defining Neutral Factor
  • Overriding Sheet Metal Properties

• Sheet Metal Capabilities - Advanced
  • Sheet Metal on Part Models (ST6)
  • Emboss (ST6)
  • Features on Bends (ST6 Ordered only)
  • Flat (ST6)
  • Create Blank (ST7)
  • Create Blank (Examples)

• Sync Sheet Metal – Why & When?
• Ordered Sheet Metal – Why & When?
• Integrated modelling
Solid Edge Sheet Metal

Dedicated document type (.psm)

Dedicated environments
• Modelling (Synchronous & Ordered)
  • Creation of Sheetmetal model with dedicated Sheet Metal commands

• Flatten
  • Create flat patterns for drawings with bend centerlines

• Simplify
  • Simplify Sheetmetal models for large assemblies
Sheet Metal Features

- Basic
  - Tab – basic tab
  - Flange - can be created on any linear edge including linear edges on bends
  - Bend – introduce bends into a flat
- Advanced
  - Contour Flange – for straight brake bends, rolled parts etc.
  - Lofted Flange – for transitional parts like arcs to lines
  - Hem
- Deform features
  - Dimple, Louver, Drawn cutout, Bead
  - Gusset
  - X-Brake
  - Etch
  - Emboss (ST6)
Sheet Metal Features

- Supporting features
  - Unbend
  - Re-bend
  - Jog
- Detailing features
  - Holes
  - Slot (ST5)
  - Cut
  - Normal Cutout
  - Break Corner
- Corners
  - 2 & 3 bend corners
- Transform
  - Part to Sheet Metal (ST7)
  - Thin Part to (Sync) Sheet Metal
  - Rip
- Switch To Part – to create part features
**Flatten**

- Flatten environment available in Sheet Metal & Part
  - Flat Pattern
    - Sheet Metal flatten with bend center lines and cut size
    - Generates bend tables for Draft
  - Blank (ST7)
    - To create the blank for Stamped Parts or for regular parts
    - Blank Body: Available in Flat environment only.
    - Blank Surface: Can be used to create flat representation of a set of faces
- Save As Flat as DXF
  - To save the flat as DXF
Create Sheet Metal

• Document Type
  • Create a Sheet Metal part in a Sheet Metal document
  • Create a Sheet Metal part in a Part document (ST5)
    • Typical usage
      • Change in design intent i.e., starts as regular part but it needs to be a Sheetmetal part now
    • Translated parts
  • Behaviour
    • Document type is unchanged but Sheet Metal properties added to the document
    • Sheetmetal environment activated
    • Sheetmetal parts can be created and edited like in native Sheet Metal documents

• Sheet Metal Part Creation
  • Create tabs and bends using Sheet Metal specific features
  • Transform regular parts into Sheet Metal parts with transform features
    • Part to Sheet Metal (ST7 Ordered Only)
    • Thin Part to Sheet Metal – Known as Transform to Sheet Metal pre-ST7
Create Sheet Metal

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Transform to Sheet Metal

- Part to Sheet Metal (ST7 – Ordered only)
  - Creates a Sheet Metal part from a regular part body
    - User selects edges to create bends
    - System automatically rips the model based on selected bend edges

- Thin Part to Sheet Metal (formerly known as Transform To SM)
  - Supported in Ordered & Sync
    - Automatically adds bends
    - Option to rip the model
    - In Sync creates native tabs and flanges

- Rip
  - Users can either use Rip using Sketch geometry or edges
    - or custom rip the part using part commands

- Note
  - Transform commands are enabled in Part documents from ST5
Thin Part to Sheet Metal

- Translated or thin part models can be transformed to SM using
- Thin Part to SM – Ordered Part to Ordered SM
- Thin Part to Sync SM – Ordered Part to Sync SM or Sync Part to Sync SM
- Transform Process
  - Introduces bends and create rips optionally
  - Users can rip the model using part features or Rip feature prior to transform
- 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 straight brake bends
- Thin Part to SM – Ordered
  - More tolerant than Transform to Sync SM
  - Handles stand alone bends (rolled parts), bends or
Part to Sheet Metal

- Only Ordered
- Creates a Sheet Metal part from a regular part body
  - User selects edges to create bends
- Rip
  - Automatically based on selected bend edges
  - Rip gap can be specified by the user
  - Both edges and faces will be ripped
  - Users can manually provide rip geometry
    - Sketch geometry
    - Point Pairs
- Option to either consume input body
- Individual bend radii can be overridden
- Individual corners can be overridden
- Useful to create a SM part from a set of faces or an occasional SM user or to create some odd shaped flanges
Transform

Non-SE Model (STP, IGS, XT, SW, Inventor etc.) → Import into SE → Uniform Thickness & full body → Yes → Thin Part to SM/Thin Part to Sync SM

Non-SE Model (STP, IGS, XT, SW, Inventor etc.) → Import into SE → Uniform Thickness & full body → No → Part to SM

Start as Part → Needs SM Part → Uniform Thickness & full body → Yes → Thin Part to SM/Thin Part to Sync SM

Start as Part → Needs SM Part → Uniform Thickness & full body → No → Part to SM
Sheet Metal on Part Models

- From ST6 SM features can be created on thin portions of a regular part - only Ordered
- Think stamped parts
- Sheet Metal features supported
  - Flanges, Contour Flanges
  - Deform features on thin portions
Emboss

• Emboss or Punch, a new feature in ST6
• Inputs
  • Target: A solid body
  • Tools: 1 or more solids
  • Thickness if target is NOT a SM part
  • Clearance value
Features on Bends

- Features on bends is supported
- Cylindrical bends only
- Features can span multiple bends
- Features can cross bends onto tabs
- Deform features
  - Dimples, Drawn Cuts, Beads
  - Louvers have to be internal to a bend
- Emboss supported
- Part features supported
- Flatten will account for features on bends
- Only features created in ST6 supported
Flat

- Flat pattern, unbend, rebend enhanced to support features on bends
- SM deform features supported on cylindrical bends in flat state
- Rounds, chamfers, holes
- Emboss
- Part features
- Flat features need to be created in ST6
  - Unbend, rebend and flat pattern
Create Blank

- ST7 introduces Blank feature
- Blank is “flatten” anything
- Useful for customers who have been using SM to get a blank
- Useful for deep drawn and stamped parts; can also be used for Sheet Metal parts
- Blank Body supported in Flat Pattern environment and will result in a solid
- Blank Surface supported in Sync and Ordered to create a flat surface
- Uses document material to determine the material deformation
- Inputs: A set of faces from any body, Draw Direction, formability properties
- Thickness inferred for Sheet Metal parts
Create Blank - Examples
Sync Sheet Metal – Why & When?

• The Why?
  • Easy creation, easier edits
  • Multi flange creation
  • Transforming translated SM parts to native SM parts
  • Top down design
  • Flange creation from Assembly
  • Individual bend control
  • Non-orthogonal plates-tabs – alternative to Lofted Flange
  • And a lot more……

• The When?
  • Model Types
    • Evaluate Sync functional capabilities to determine whether your parts can be created in Sync or not
      • E.g., Straight brake bends supported in Sync, translated parts that require thickness change, individual bend parameter control
  • Workflows
    • Evaluate your workflows to determine
      • Types of parts you create
      • Edits you make during design iteration- how often, edit types, down stream impact
  • Start simple
    • Start with Tabs, Flanges and holes
    • Graduate towards Contour Flanges, Patterns, Mirror
    • Editing translated SM parts
      • Thickness change, shape change, bend parameters etc.
    • Remember integrated or mixed modelling
Ordered Sheet Metal – Why & When?

- Stamped Parts
- Features across bends
- Rolled Parts
  - Lofted Flange
    - Arcs to lines and vice-versa
  - Contour Flange along arcs
- Unbend/Re-bend
- Smart Cuts (cuts across bends)
- 3 bend Corners
- Holes on bends
- Part features on Sheet Metal
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Integrated Modelling

- Integrated modelling: Using a combination of sync and ordered
- Typical workflow
  - Start with a tab in Sync
  - Switch to Ordered
  - Use Ordered features to build the model
- Advantages
  - Get familiarized with Sync concepts at your own pace
  - Overcome Sync SM limitations
    - Part features on Sheet Metal models
    - Cuts on bends
    - SM Features on bends
    - Part features on bends
    - Rolled parts
    - Stamped parts
  - Translated SM straight brake parts
    - Use Thin Part to Sync SM to convert the dumb model into a native SM, switch to Ordered and model. Allows thickness changes.
Contact Info

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