What’s New in v19?

- Better Manufacturing Documentation
  - Bend Tables in Part
  - Bend Table in Draft
  - Flat Pattern Cut Size
  - Divide Bend for Lofted Flanges (transitions)
  - Bend data to DXF
- Stiffer Models
  - Contour Flange on Curved Edges
  - Hem Command
  - Cross-Brake Feature
  - Gusset Command (Corner stiffener)
- Assembly Design
  - Flange Match Face
What’s New in v19?

- General Fixes
  - Deliver Stencil Fonts
  - Enable thickness key-in for initial Contour and Lofted Flange
  - Dimension Constraints for Flange Profile
  - Unbend option for “All Bends”
  - Nearest Cut Normal Cutout (Welding Cases)
  - Contour Flange: Show Selected Edge in
  - Contour Flange: Apply Side During Contour Flange Edit
  - Contour Flange: Correct Miters for Outside
  - Contour Flange: Correct Miters for Interior Contour Flanges
  - Contour Flange: Optional Miters for large bend
What’s New In Solid Edge v19

Better Tools for Documenting the Manufacturing process
A Bend Table option has been added to the Sheet Metal environment.

- User defined bend sequencing that is independent of the design tree.

- The Bend Table includes standard and custom data for each bend:
  - Radius, Direction
  - Machine Die No.

- The bend table can be accessed in a blank file so custom properties can be set in a template.
Bend Tables (Sheet Metal)

- For Family of Parts
  - The bend table is based on the current applied model
  - The Bend Table is copied to each member on create
  - Member Bend Tables are independent of the master file so they can be manipulated for that specific member.
- Established bend sequences will “survive” suppress and un-suppress states.
- Should the user delete a feature in the design tree, the bends are removed from the bend table and the sequence regenerated
- The “Direction” in the Bend Table is derived from the flat pattern model
Sequence a bend by clicking Up / Down Arrows on the dialog

Sequence a bend graphically by choosing the balloon and entering the desired sequence

Selected bends graphically highlight

Yes is the answer to your next question
Bend Tables are available in Draft

...in 5 clicks!

Automatic Call-outs

Automatic Bend table
Bend Table in Draft

- Automatically (optionally) add a table in Draft that lists all bends of a selected flat pattern
- Automatically (optionally) add bend callouts to the flat pattern
- Callout options for “balloons” or embedded
- Configurable callouts (“Bend Up 90”, “90 UP, R0.03”, etc.
- Associative to the 3D model
- Sequence of bends listed as defined in the 3D model
- Bend direction no lists Up / Down, +/- can still be used.
- The format of the table is editable
Bend Centerlines Transitions

- Users can easily document the manufacture process for transitions
- Bend centerlines are added to the 3D model (and into Draft)
- Bend index marks are added to the DXF file to assist bending using straight press brakes
- Stock Templates delivered for
  - Round to Round
  - Round to Square
  - Square to Round
Divide Bend for Lofted Flanges

- Users can set the number of bends
- Users can set the index mark length
- Index marks are placed on a separate layer in the DXF file
  - Index marks are used to align the metal on the press brake
Divide Bend for Lofted Flanges

- Bend centerlines are placed at interior places on a lofted flange and at any end connected to adjacent flanges
  - Arcs only: only add bends to the interior & not at the ends
  - Arc line cases: add bends to the intersection of the line/arc and interior to arcs
  - Line arc line cases: add bends to the intersection of both the line/arc and interior to arcs
Flat Pattern Cut Size

- This command will automatically alert you when the flat pattern exceeds a specified blank size.
- The cut size is dependent on the flat pattern so users have orientation control.
Flat Pattern Cut Size

- The blank size is a global setting listed in Tools | Options and can be established in a template file.
- The size of the blank can be uniquely set.
- The cut size dimensions are listed in the Variable Table (handy for expose variables).
- Users can quickly update the Flat pattern after a feature change and see the cut size.
Bend Data to DXF

- Automatic bending machines now have all bend information for an improved manufacturing process.
- Custom bend data such as bend direction, angle, radius, machine die#, etc. will be written to the DXF file for each bend.
- The data and format is specific to bend software—out of the box will work for Trumpf.
Quick Demo

- Bend Tables
- Flat Cut Size
- Transitions
- Bend Data To DXF
What’s New In Solid Edge v19

Better Tools for Creating Stronger Parts
A user can now create a Contour Flange on curved edges.

Typical uses are for creating weld flanges, safety edges, strengthening edges, etc.

The edge can be a curved edge on a planar face or around a cylinder.

Model the way you need, but manufacture using common rolling mills.
Users can also place a Contour Flange on curved edge of a tab.

The manufacturing process is typically some formed operation.

Yes is the answer to your next question…
Contour Flanges on curved edges will Flatten!

- The shapes for cutouts in the formed will be computed and be correct in the flat pattern and vice versa
- A square cut in the formed will become a pie shape in the flat
- A square in the flat will become a pie in the formed

Yes is the answer to your next question
Holes in the formed model can be **optionally** preserved in the flat pattern!!!(red)

- Pilot holes for mounting screws on a deformed flange will remain as holes in the flat.

- This option was added so you can easily manufacture the part, the hole on the rolled part will not be round, but in many cases that is acceptable

- See Tools | Options for the option
Flange on Curved Edges

- The dimension and geometry for relief on partial flanges will be preserved in the flat pattern.
- Relief cuts generated in the flat pattern will be punch-friendly.
- Bend relief sizes established in the formed state are preserved in the flat state.
- Thinness faces are always perpendicular in the flat pattern.
Flange on Curved Edges

- Locate / select / highlight behavior has changed, only highlighted geometry will be considered.
- Chain will automatically locate and select all edges except for a bend at the default bend radius.
  - Most cases, it’s undesired to flange those bends.
- Use Edge to select and deselect curved edges.
- Any edge(s) along the chain can get selected.
Hems

- Hems are now available for quickly and easily creating safety edges, hinge rolls, and other types of traditional edges.
- The Hem command supports 7 of the most common types:
  - Closed, Open, S-Flange, Curl, Open Loop, Closed Loop, Centered loop
  - Use Contour Flange for more complex shapes
- Hems are edge treatments so just click an edge. Simple!

Yes is the answer to your next question
Hems can be applied to curved edges
Hems

- Support for saved so frequent sizes can be reused
- Automatic mitering around corners
- Options for bend relief
- Full support for material inside, material outside and bend outside
- The bends resulting from a Hem will be listed in the Bend table
The new Cross-Brake command allows users to add crossing bends to create an "X" set of bends typically used for stiffening large sheet metal panels.

The feature will only add attributes to the faces and not actually deform the 3D model. Virtually fail-proof!
Cross-Brake Feature

- Each individual bend for the Cross Brake will participate in the Bend Table so you can document the exact order an “X” is to be bent.
- Cross Brake features can participate in FOP and Feature Library.
- The “bend” angle is listed in the variable table.
- Multiple cross-brake features are allowed, but linear sketch elements only.

Yes is the answer to your next question.
Cross Brake features will propagate to the flat pattern in and can be documented in Draft (and export to DXF).

- Traditional centerline in full support of annotations and dimensions
Gussets

- You can now add stiffening ribs across bends (Gussets)
- Standard straight stamp Gusset or user defined profile that supports custom shapes
- Pattern Gussets while in the command
- Saved settings is available for building a library of specific types
Gusset Command

Gusset Options

Yes is the answer to your next question
Gussets

Models with Gussets can be flattened

- Gussets are considered local deformation, thus the flat pattern will not be affected
- Gussets are post bend operations, so no information will be included in the flat pattern
- Users will document the locations of the Gussets on the formed model
Create Stiffer Parts

Quick Demo

- Flange on curved edges
- Hems
- Cross Brakes
- Gussets
Better Assembly Design Tools
Assembly Design

- Users now have easy methods to make “this flange” match “that face”
- Perfect for in-context design where parts must fit together
- Works in conjunction with Inter Part Copy for full associativity
- Supports offsets from the target face
- The target face can be at any angle.
Assembly Design

- Available in the Flange command during the extent step
- Simply choose the target face and specify an offset
- The profile and plane is transformed to the target face to support profile edits

Yes is the answer to your next question
Assembly Design

- Faces at any angle can be matched (within geometric reason)
- The initial dimension scheme (inside / outside) will be maintained
- Works with any of the flange types (full, partial, centered, etc.)
- Support for offsetting from a face to provide some clearance
Assembly Design

Face matching can be used on imported geometry or outside the flange command

- Also available as separate command for Direct Editing
- Ideal for making the vendor part fit my part
- Works on most flanges (outer flanges only)
Quick Demo

- Flange: Match Face
- Direct Edit: Match Face
What's New In Solid Edge v19

General Command Enhancements
Deliver Stencil Fonts

- Stencil fonts are being delivered and can be used for creating text on sheet metal parts
- Full support for size, italics, and bold
Base Feature Thickness Entry

- Contour Flange and Lofted Flange
  - During the side step, you can enter the default material thickness
  - Available for the base feature only
  - Available during edit
The constraints for flanges have been changed so miters can be quickly added by simply changing a dimension. No need to enter profile and modify the constraint system for creating miters.
Unbend All option

- A new selection option has been added to the Unbend command to choose all bends in the model.
Nearest Face Normal Cutout

- A new option for Normal cutouts to provide a cutout normal the face closest to the profile
- Ideal for welding pips to sheet metal parts

Thickness Cut  Mid-Plane Cut  Nearest Plane Cut
Contour Flange: Usability

- **Edge Highlight**
  - While in Profile for the Contour Flange, the selected edge remains highlighted for easy identification.
  - After a profile edit in Contour Flange, Solid Edge will automatically apply the correct material side.

- **Side Selection**
  - Users can still backup to that step and change the side.
Contour Flange: Smarter Mitering

- Option to set miters on flanges around bends when the bend is larger than the default bend radius.
- Option OFF by default in V19 so new files will get the correct miter by default.
- Pre v19 files will have the option on to preserve existing flanges—chance are you modeled the 90 degree miter.
Contour Flange: Smarter Mitering

- Contour flange around an outside edge will be mitered to produce a manufacture-ready part.
- There is no option to get the “old” method—we just do it right.
Contour Flange: Smarter Mitering

- Contour flange can now be placed around inside edges!!!!

- No special options, or sketches
Solid Edge Sheet Metal V19 Wrap Up

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- Assembly Design
  - Flange Match Face
- General Fixes
  - Deliver Stencil Fonts
  - Enable thickness key-in for initial Contour Flange and Lofted Flange
  - Dimensions for Flange constraints
  - Unbend all
  - Enhance Normal Cutout for welding case
  - Contour Flange: Usability
  - Contour Flange: Smart Mitering

Don’t forget to attend the hands-on