



Sheet Metal Tips and Tricks



Sheetmetal Rules

Only two rules for
sheetmetal parts ! ! !

1. Parts must be of uniform thickness.
2. Sheetmetal parts must consist of **planes, cylinders or cones** in the “bend area” to be “valid” sheetmetal parts

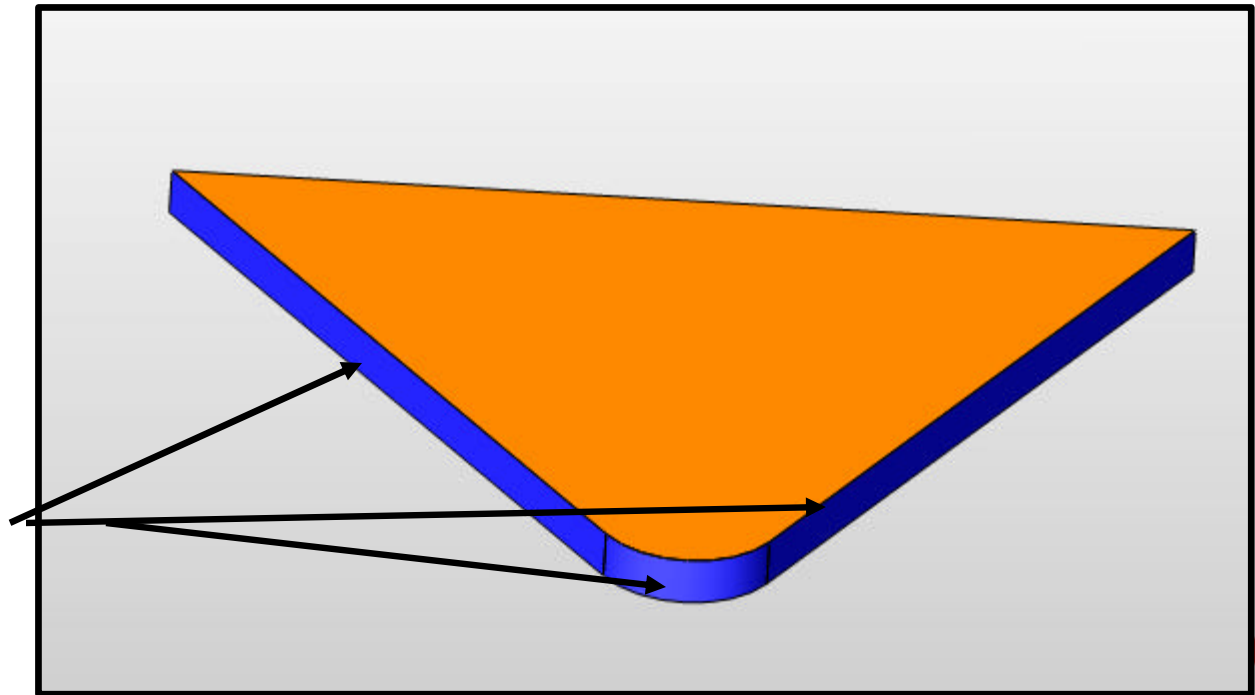
(other than form features)

Sheet Metal Rules

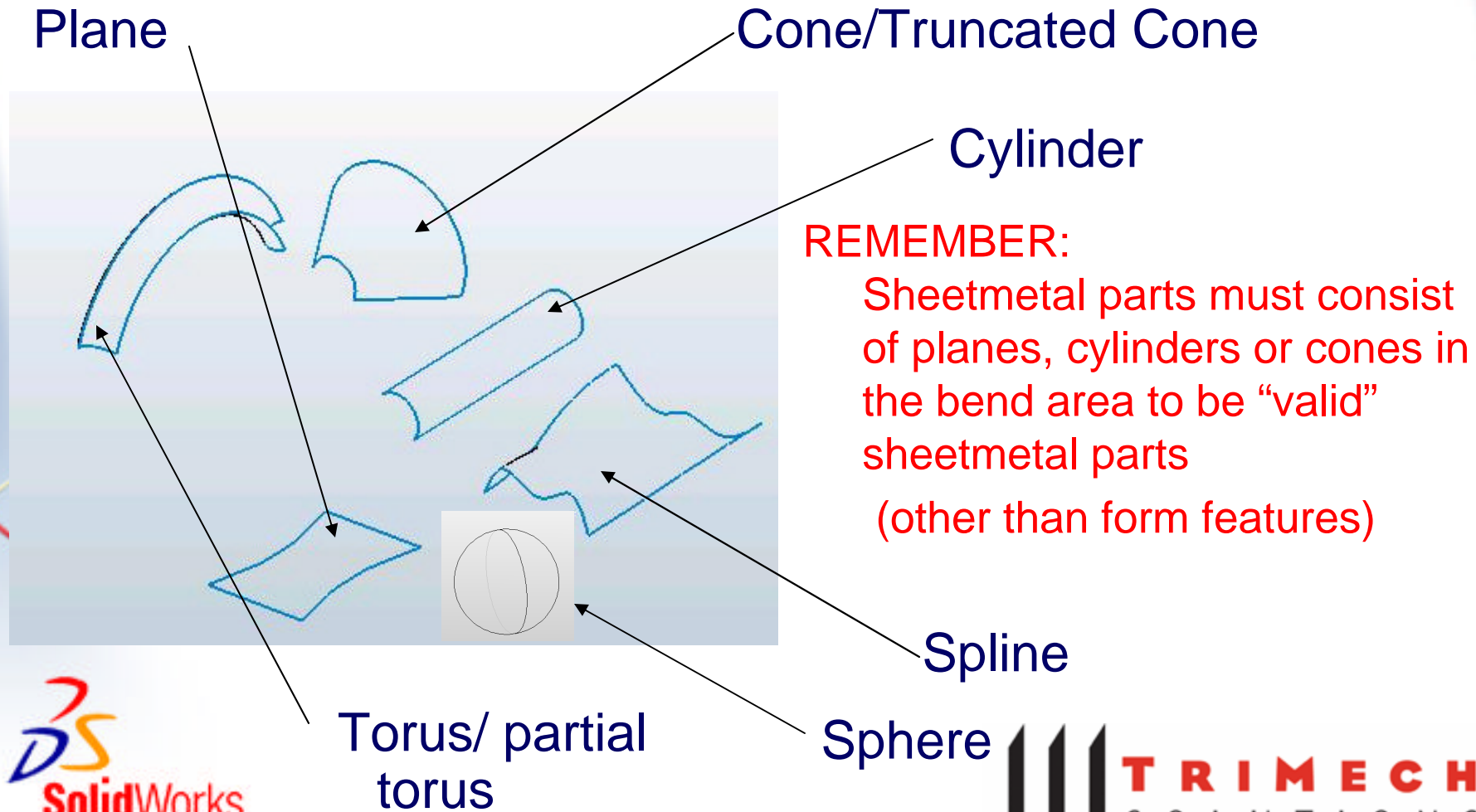
What is the “Bend Area” ? ?

Ans. = Any face that is not a side face, which defines the thickness of the sheetmetal part, such as the blue faces below

Side
Faces



Geometry types (faces)



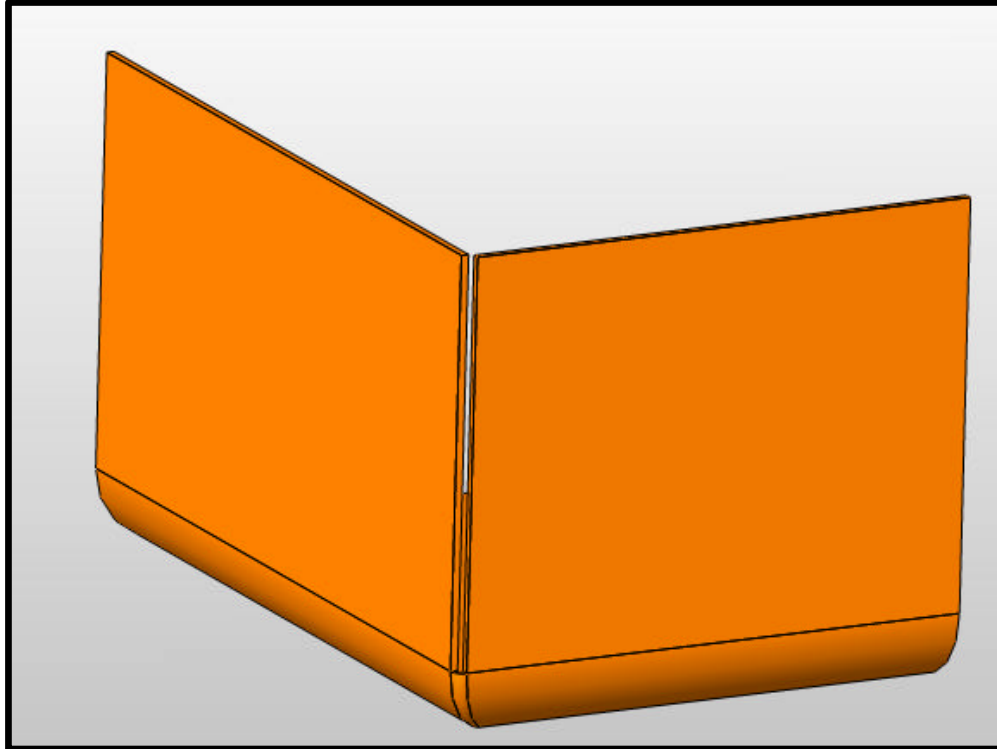
Contest

The next 6 slides show examples of Sheet Metal Parts
Determine whether or not the part can be flattened
with the reason why.

First correct answer wins a Prize.



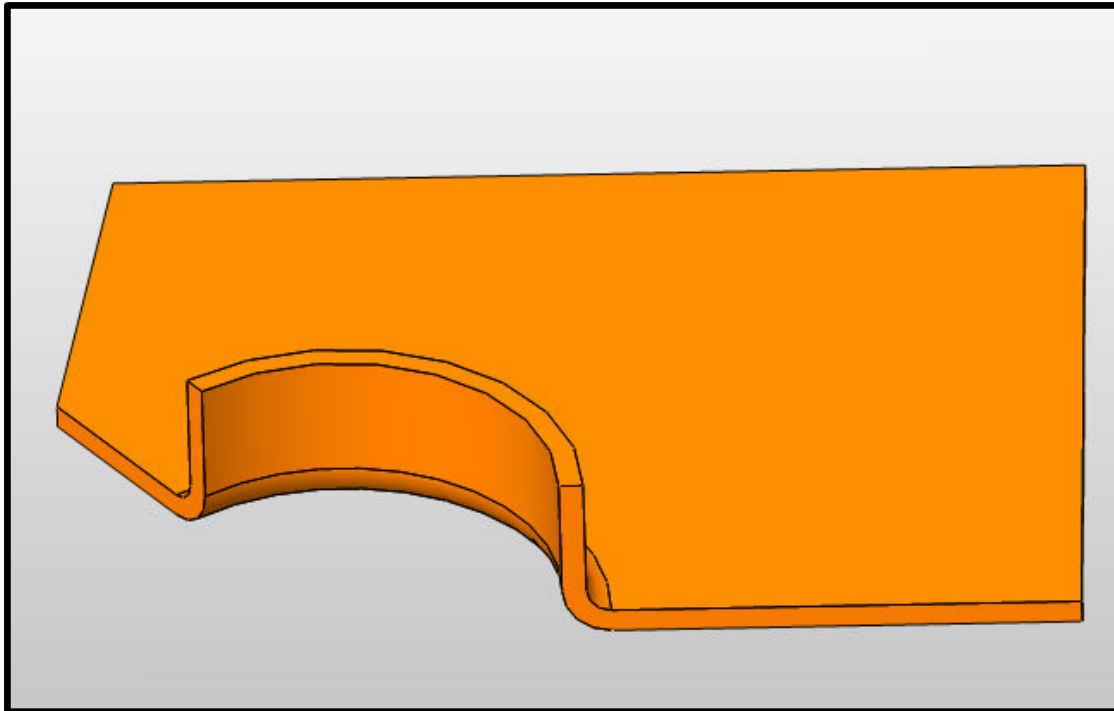
The Flatten Test #1



Can it be flattened: Yes or No?
Why?? Or Why not??

Ans. = No, Sphere

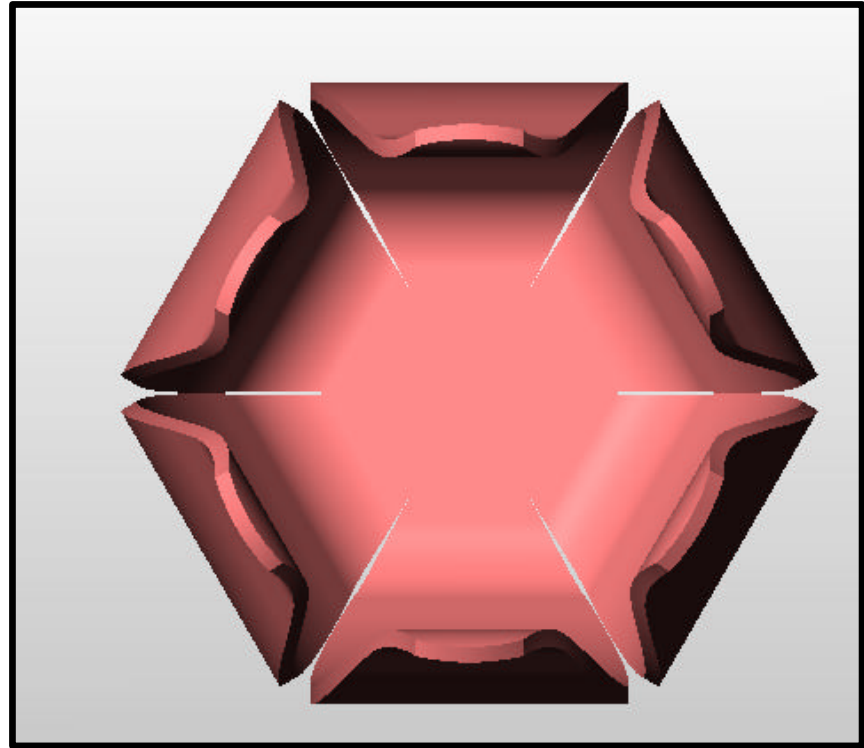
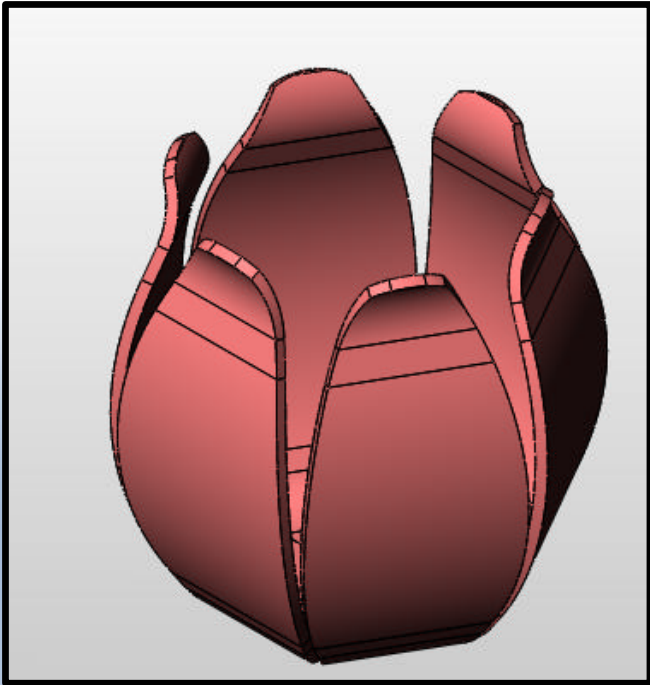
The Flatten Test #2



Can it be flattened: Yes or No?
Why?? Or Why not??

Ans. = No, Partial Torus

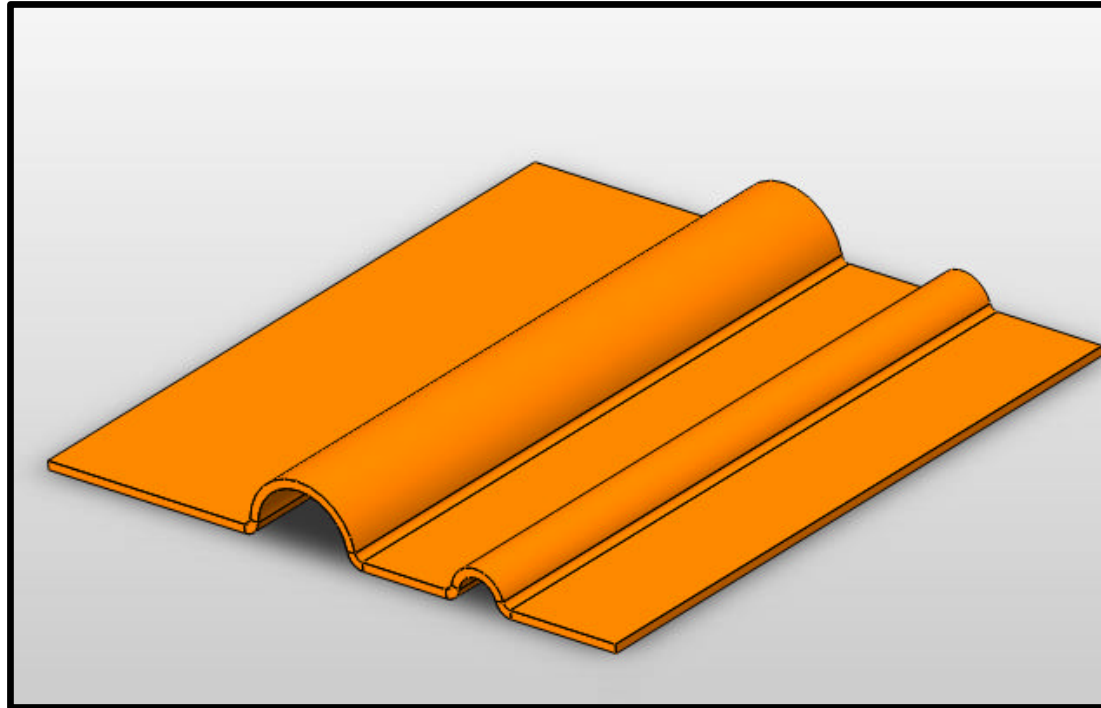
The Flatten Test 3#



Can it be flattened: Yes or No?
Why?? Or Why not??

Ans. = Yes, all planes and
cylinders

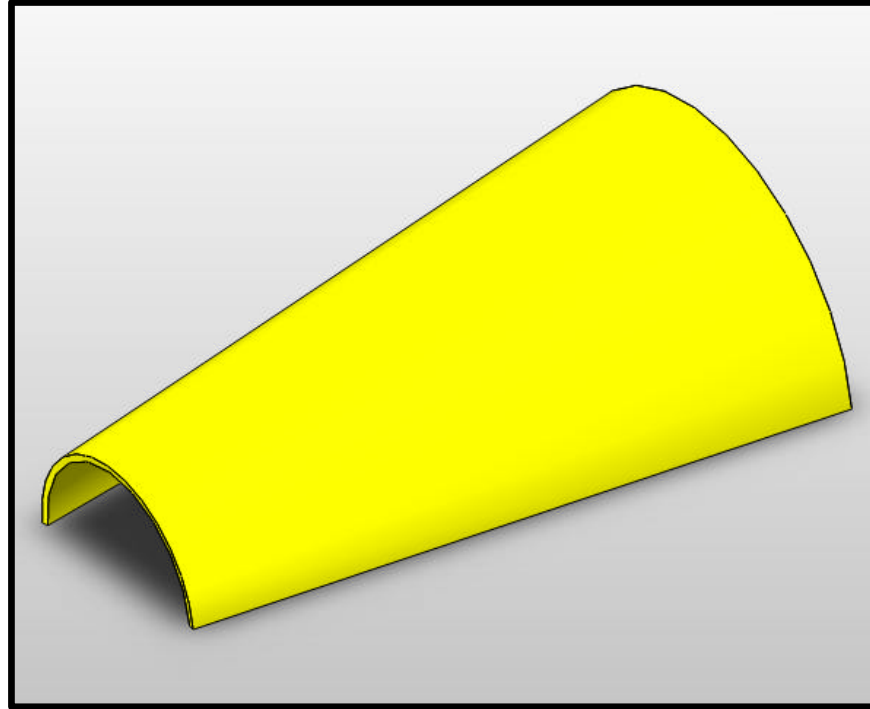
The Flatten Test #4



Can it be flattened: Yes or No?
Why?? Or Why not??

Ans. = Yes, all planes and
cylinders

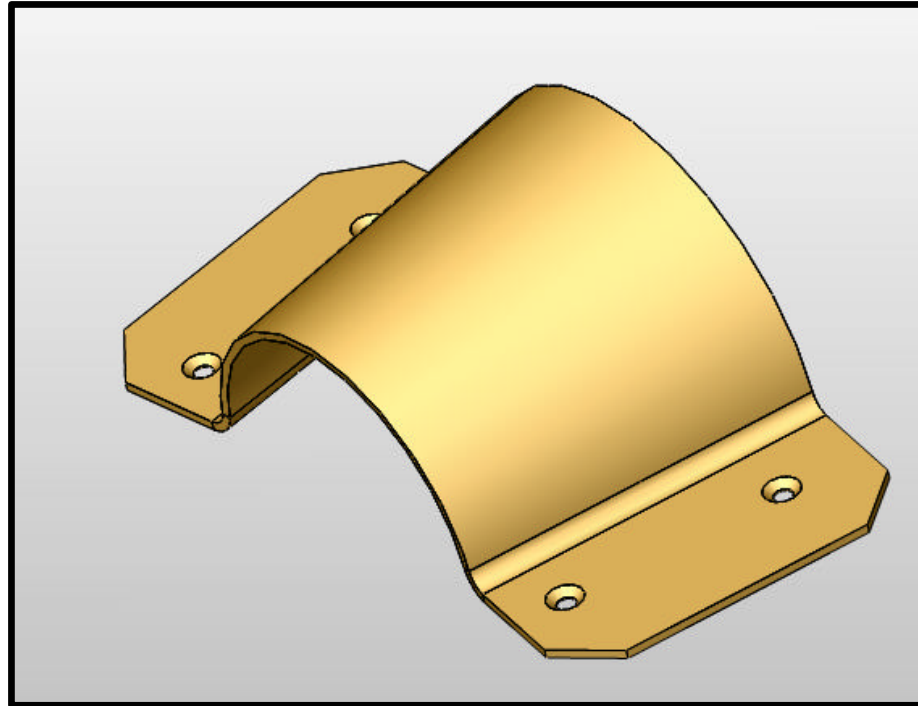
The Flatten Test #5



Can it be flattened: Yes or No?
Why?? Or Why not??

Ans. = Yes, All cones

The Flatten Test #6

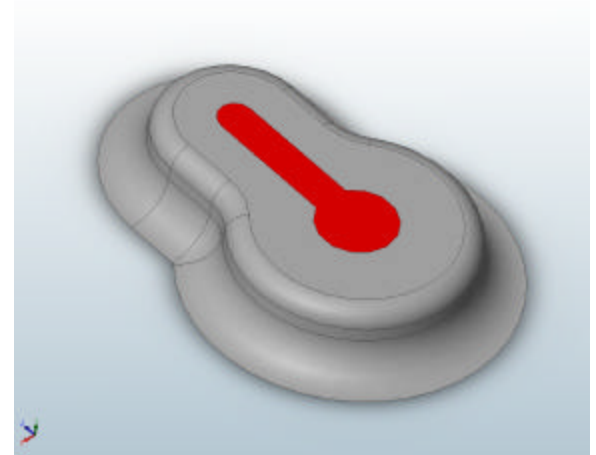


Can it be flattened: Yes or No?
Why?? Or Why not??

Ans. = No, Spline geometry.
Where??

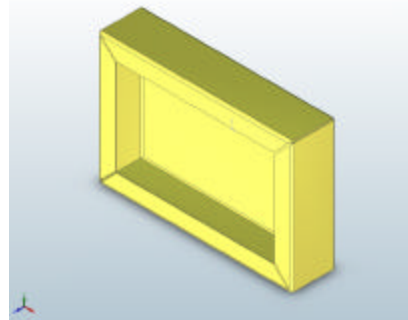
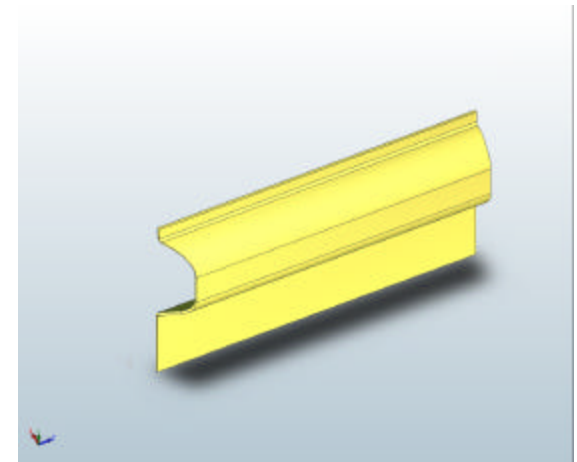
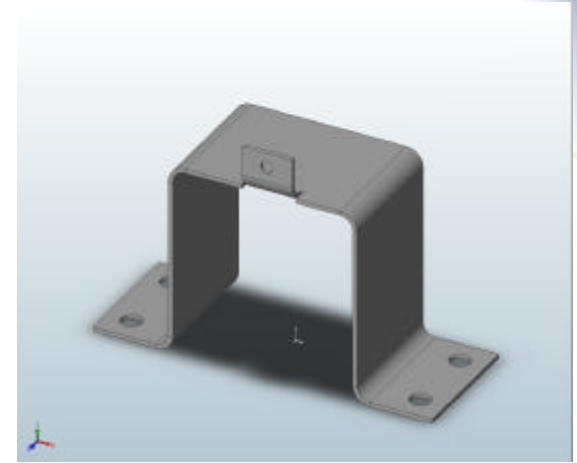
Sheet Metal Forming Tools

- Can Only be dropped on Sheet Metal Parts
- Acts as a die to bend, stretch or deform
- When creating the radius of curvature must be greater than the thickness
- Use the color **RED** to indicate the face that removes material



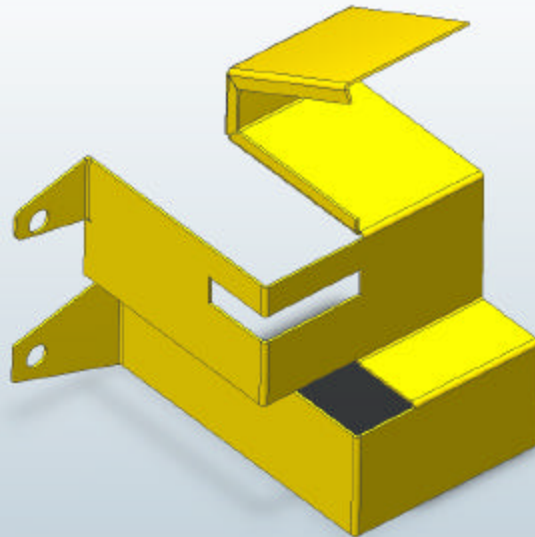
Flat Pattern Options

- Merge Faces: Faces that are planar & coincident in the flat are merged.
- Simplify Bends: Curves in the Bend Region are represented as Linear Edges in the Flat
- Add Corner Trim: Child of the Flat Pattern. Add Corner Treatment, Corner Trim (Square, Round or Bend Waist)



Using Symmetry

- Use it to simplify the design of brackets or parts that are symmetric.



Sheet Metal Resources

- www.sheetmetaldesign.com from Sean Adams
- www.engineersedge.com all sorts of Technical info
- www.sme.org Society of Manufacturing Engineers
- www.eng-tips.com Technical Moderated Groups
- www.sheetmetalworld.com Tons of Sheet Metal info
- www.autopol.com Solution Partner
- www.solidworks.com/swexpress/index.cfm
SolidWorks Express Newsletter



Parametrics, Design Tables and Equations

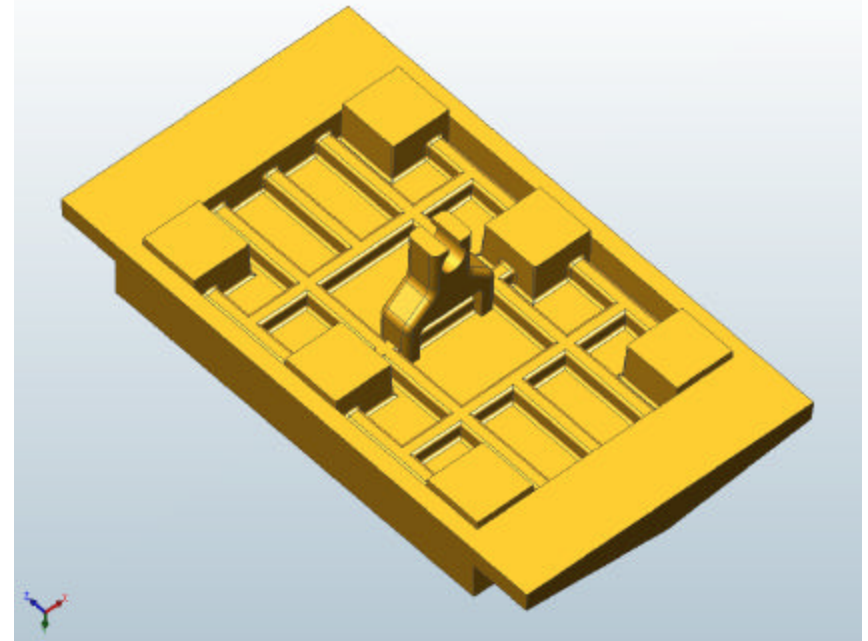
- Parametrics: The use parameters such as dimensions, geometric, in-context and assembly relations in order to capture design intent.
- Design Intent: Intelligence (Fit, Form and Function) captured within sketches, features, in-context and assembly relations in order to propagate change throughout the design. i.e. change the shaft O.D. and the bearing I.D. changes.

How are Parametrics Used

The most recognizable example of parametrics within a part are dimensions defining the size, location and relationship of features.

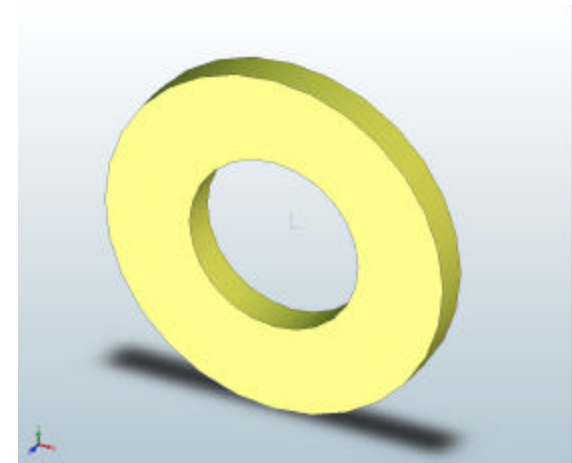
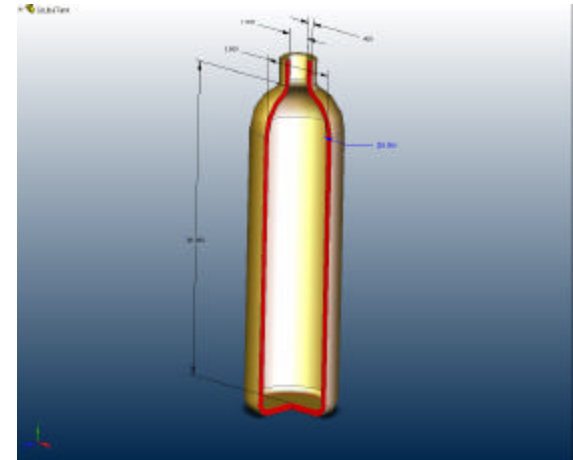
Dimensions can be linked. This will help define one dimension for a series of features such as fillets and rounds.

Sketches contain geometric relations that define how entities are controlled through the application of equal, parallel, horizontal, concentric, in-line, symmetry, etc.



Design Tables

- Used to create a family of parts or assemblies via configurations.
- For Parts, Design Tables can control Dimensions, Custom Properties, Suppression State, Comments and User Notes
- For Assemblies, Design Tables can control Dimensions, Suppression State, Custom Properties, Configuration and Visibility, BOM Expansion, Part Number, Comments, and User Notes
- Excel can be used to perform higher levels of design intent control and calculation.
- Design Tables can be Bi-directional.



Equations

- Used to create a relationship between parameters that cannot be achieved using geometric relations or modeling techniques.
- Rename dimensions that will be in the equation
- Equations in SolidWorks are solved in this order:
Dependant=Independent
if $A=B$ then A is solved when B is given
Before using an equation determine which is the Dependant variable and which is the Independent variable.

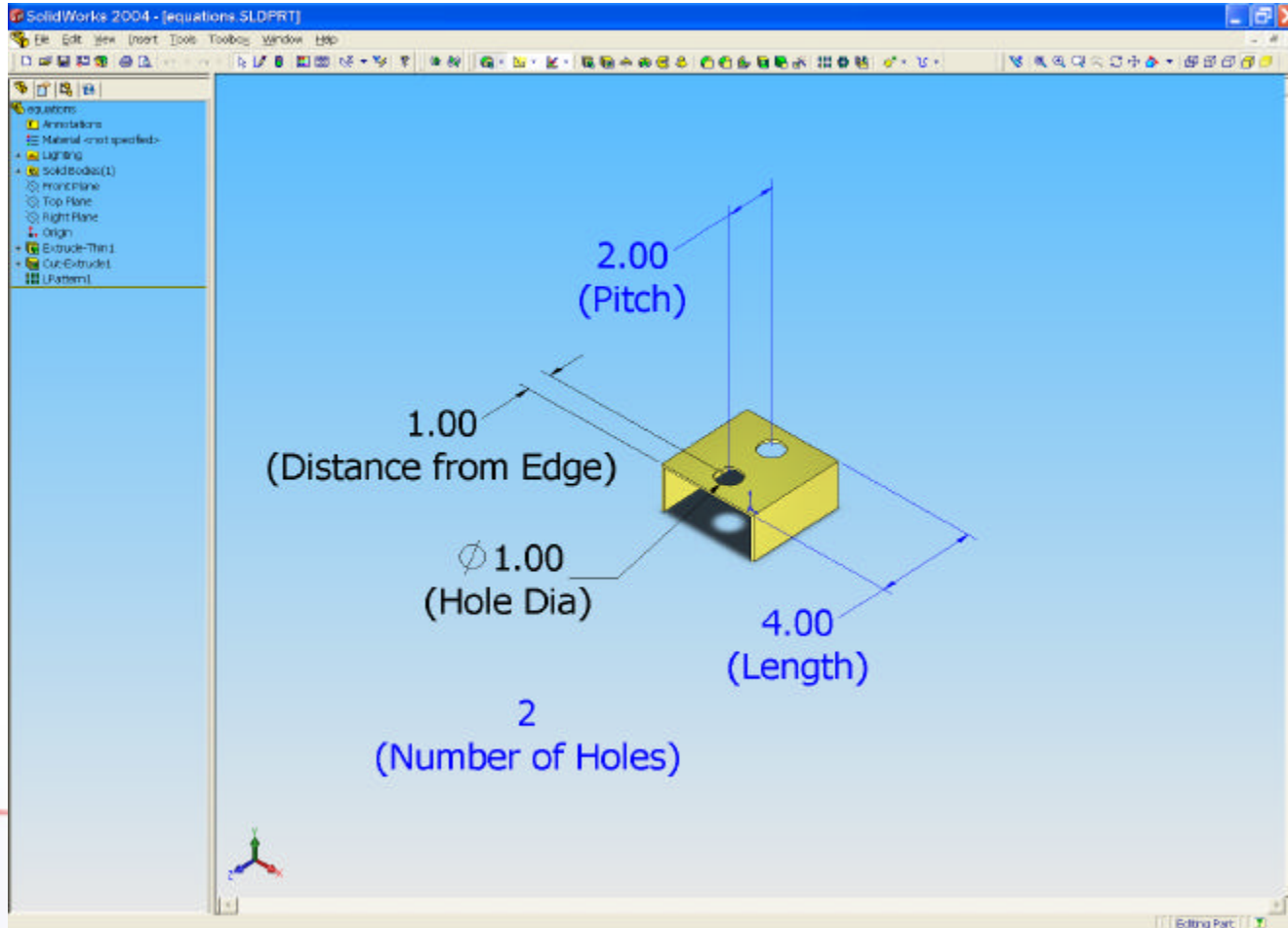
Equations

- Functions used in equations are:
Addition, Subtraction, Multiplication and Division. Trig Functions, Absolute Value, Integer, SQ Root and Exponential can be used and are subject to Microsoft VB rules.
- Dimensions can be added to equations by clicking on them.
- Once a dimensions is added to an equation it can't be changed by double clicking it.
- Watch out for circular references. $A=B+A$ is an example of a circular reference. Or when more then one equation controls 1 dimensions

[SolidWorks Express](#)



Example of Equations at Work





Thank You

