

**Proposed**



**TIP**

**Wrap, Wrap, Wrap  
Wrap it Up**

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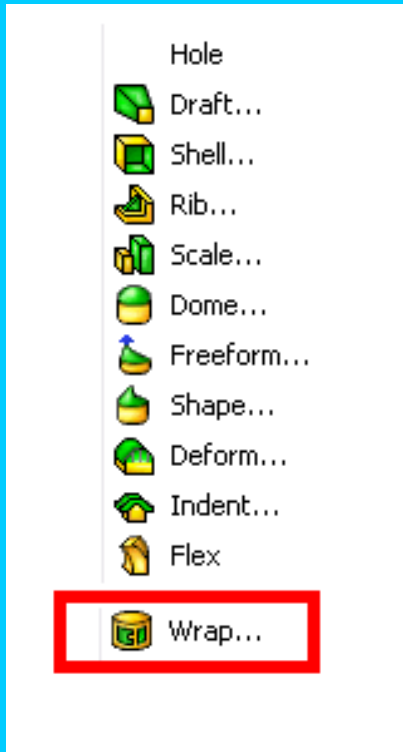
# Wrapped Around Your Finger

- Doesn't happen everyday but occasionally you want to add a feature on a non-planar surface.
- There's plenty of two step work-arounds but how about a one step super feature?

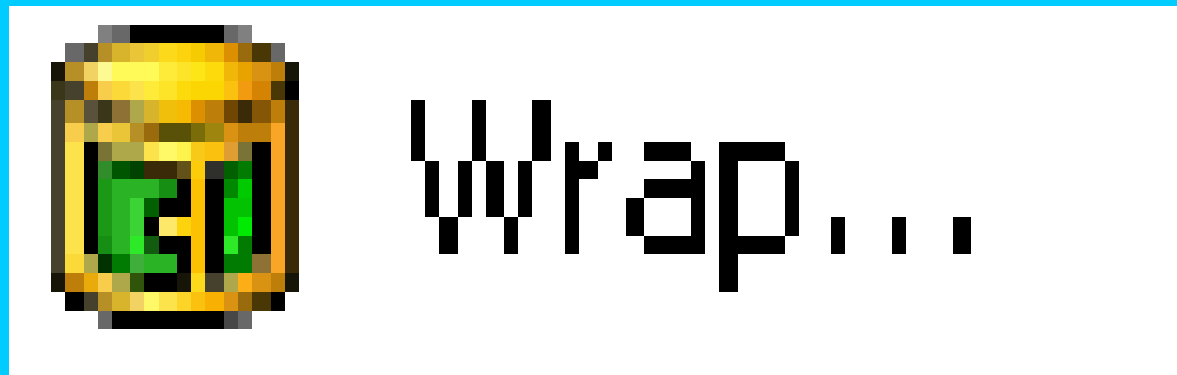
- ***The Wrap feature allows you to sketch on a plane and then “wrap” that sketch onto a non-planar surface.***



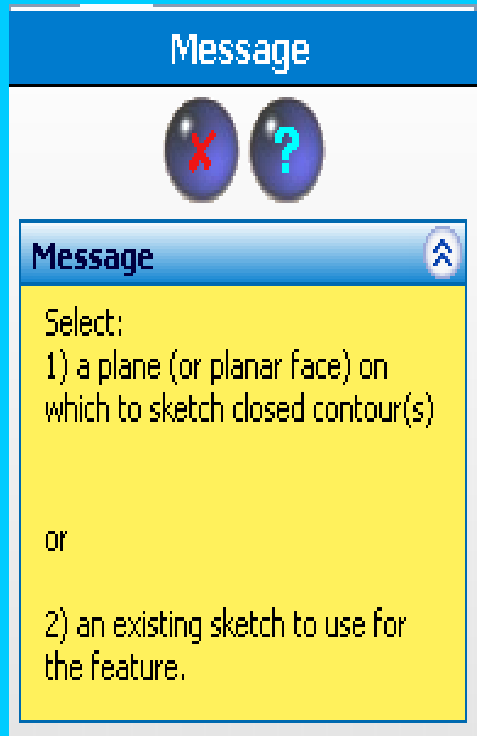
# Wrap Feature



- One of the little used super features built into SolidWorks is “Wrap”.
- The icon kind of says it all.



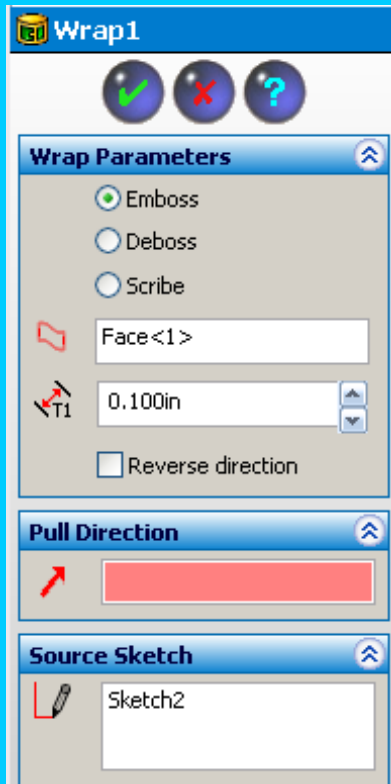
# Wrap Requires Sketch



- If you pre-select a sketch before you “wrap”, you don’t see this message.
- If you see this message, then either
  - Select a previously drawn sketch
  - Select a plane or planar surface to automatically start a new sketch.



# Emboss / Deboss



- Select a surface and then how much you want the new feature to extend beyond the surround surface (emboss) or engraved into the surrounding surface (deboss).
- Deboss should be really be called engraving.

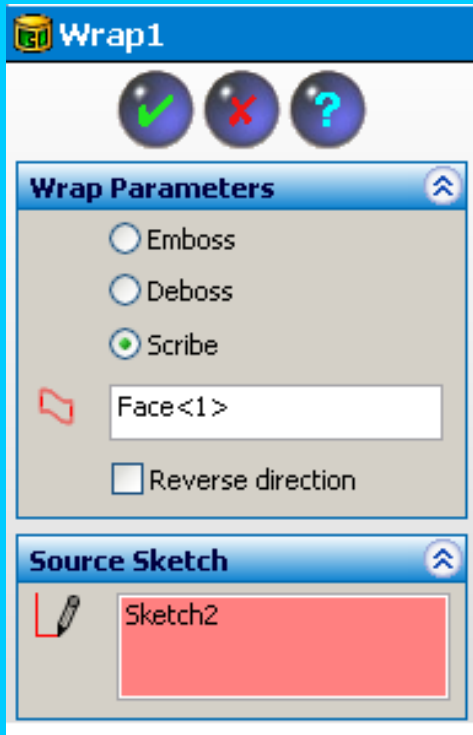


# Pull Direction Option

- “Pull direction” is a injection molding or die casting term that refers to the direction that the clamshell tooling moves.
- By default, the “pull direction” is normal to the sketch you created but you can select an edge or axis and change the pull direction.
- The feature always starts at the intersection of the sketch and the surface.



# Scribe



- Scribe creates a zero width line that represents the intersection of the sketch and the surface.
- Note, even for the scribe option the sketch must be a closed profile.



# Ideas...

- Remember the sketch tool to create text creates closed profiles so they are valid candidates for wrapping.
- Use the Wrap command to emboss or engrave custom sketched logos into non-planar surfaces.

- Well that about wraps it up...







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