

Proposed



TIP

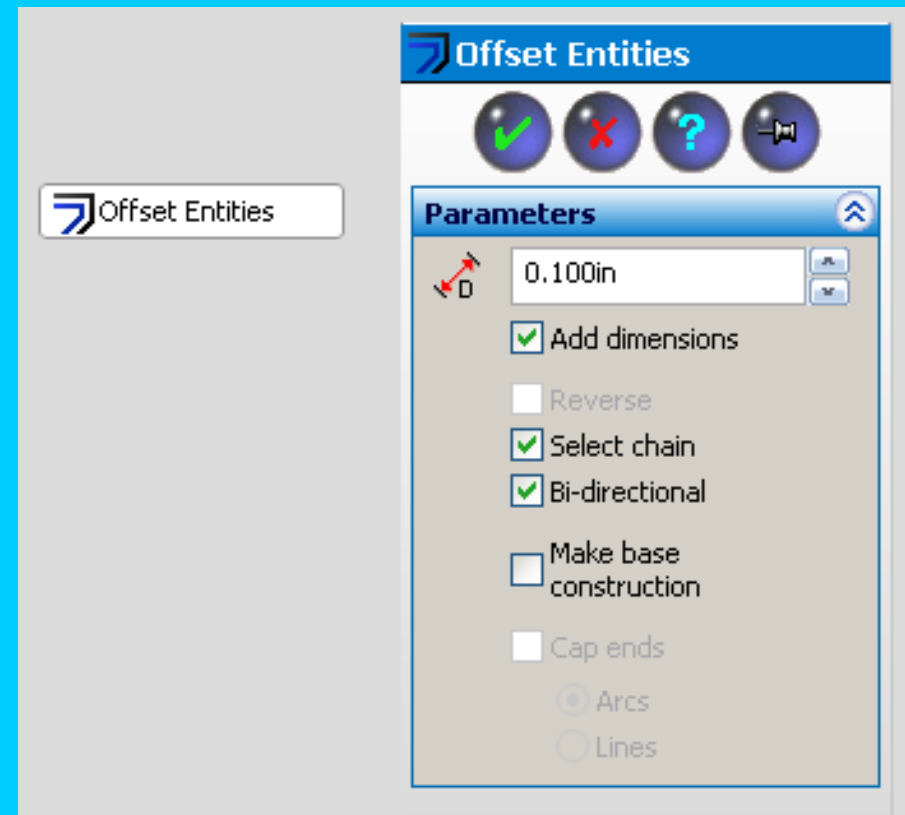
Thin Features

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Parallel Parking

- If you ever find yourself sketching something with a lot of parallel lines all at the same spacing - **STOP!!!**
- First off, haven't you ever heard of **Offset Entities**?
- Secondly, think **thin features!!!**



Thin Wall Pressure Vessels

- Thanks to over one hundred years of engineering pressure vessels, the term thin wall is indelibly linked to thin walled pressure vessels where the container wall thickness is less than a tenth of the vessel diameter.
 - ***But in SolidWorks, thin features can be anything!***



A Rose By Any Other Name

- Thin Features are a misnomer in SolidWorks.
- Thin Features should actually be called “Constant Width Features”.
 - Unlike thin walled pressure vessels, there is no requirement in SolidWorks for the thickness of the feature except that the resulting solid geometry cannot overlap.



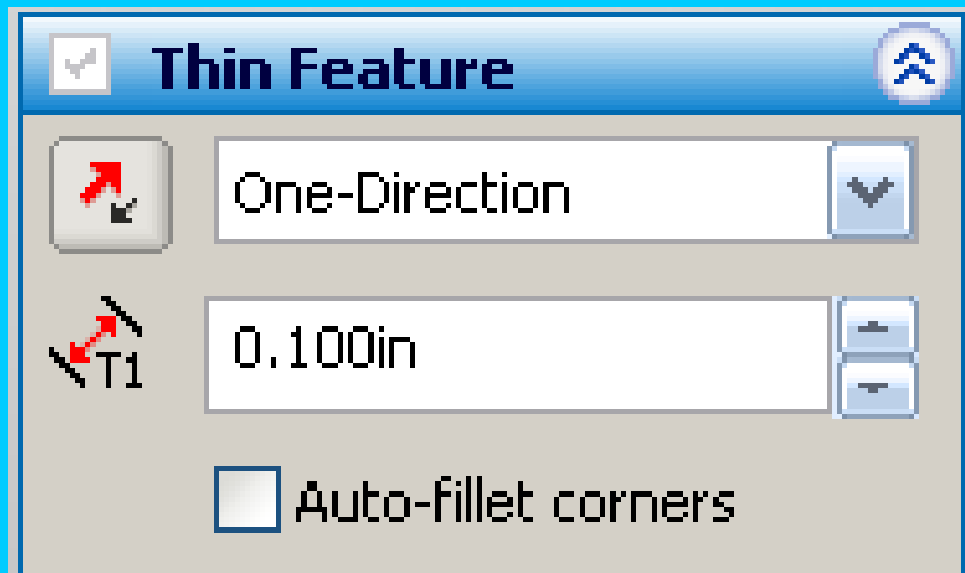
Thin is In!

- **Extrude** thin features to create walls very quickly.
- **Cut** thin features to create “lips” around the edge of parts.
 - Couple this with the **Convert Entities** command to make quick lips that Mick Jagger would envy.
 - Make O-Ring glands in a snap!
- **Revolve** a thin feature for, hey why not, a thin wall pressure vessel.
- **Sweep** a thin wall for a rubber accordion boot that you want to skip the shell command.



Thin Feature Option

- If your sketch is an **open profile**, the Thin Feature is automatically turned on.
- If your sketch is **closed**, you have to select the Thin Feature option.



1
min

TIP

Thin Feature Directions

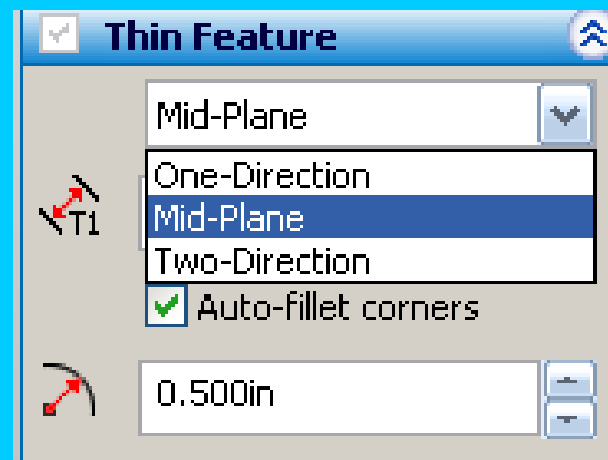
- **One-Direction:** You get to pick whether the feature is inside or outside your sketch.
- **Mid-Plane:** Just like other features, you enter the distance and the sketch is right in the middle.
 - This can be a little bit confusing since it is actually mathematically the median, not necessary a plane.
- **Two-Direction:** you enter two non-zero distances for how much the feature goes inside and outside your sketch.

– *Can you guess why non-zero?*



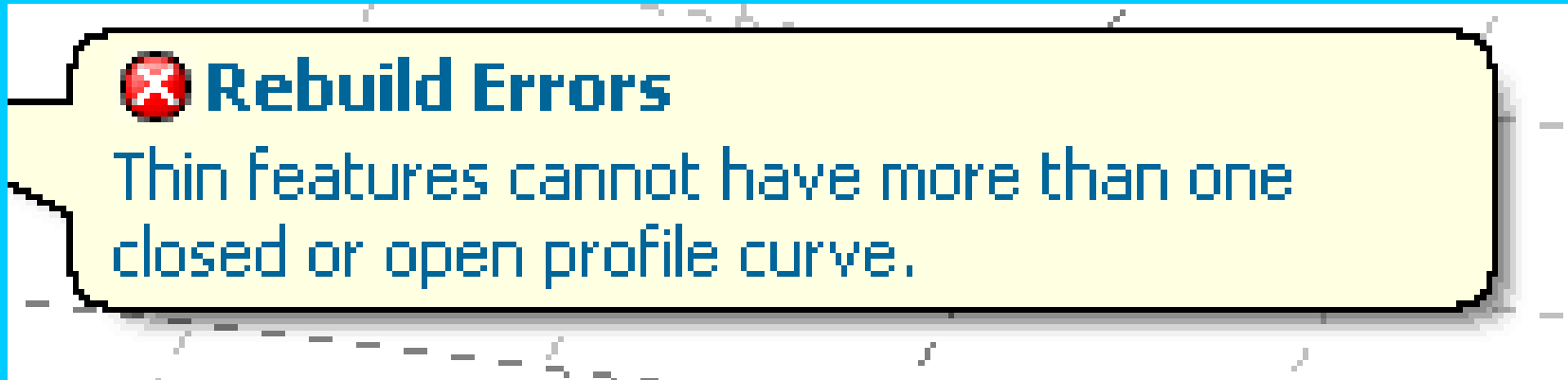
Smooth Move, Exlax

- Finally, you can have all the zero radius vertices in a sketch smoothed out by selecting the “Auto-fillet corners” and entering the fillet you wish.
 - By the way, this fillet is at the sketch not necessarily at either surface.



Thin Feature Errors

- Certain sketches can not support thin features and you get this message:



- You also get an error if the resulting solid geometry intersects itself.





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