Tips & Tricks: Editing in a MultiCAD Environment

OneSpace Modeling

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OneSpace Modeling's history-free, dynamic modeling allows you to reduce a model's complexity by adding or removing problem areas from a model. In this step-by-step example, a blend modification is completed once the imported model's blend faces are corrected and successfully re-inserted.

A Step-by-Step Process for Reducing Model Complexity

Design News magazine published this tips & tricks article on June 14th, 2005

Read the article: http://www.designnews.com/article/CA608269.html?industryid=22203

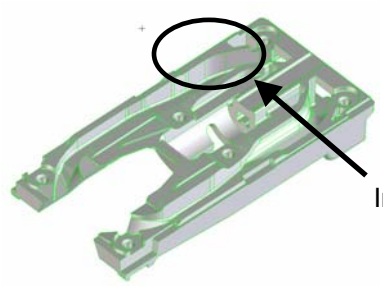
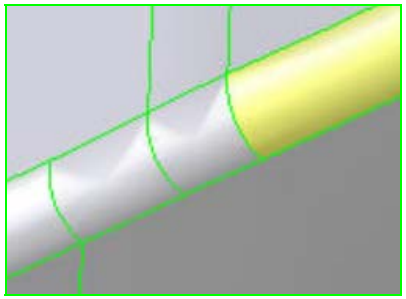
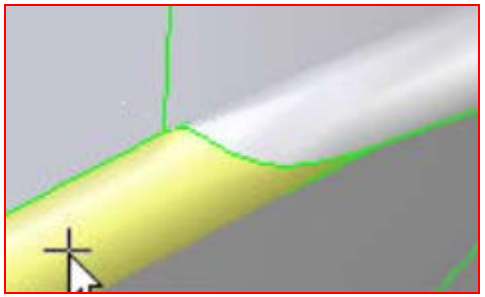


Figure 1:

After importing the model, fly-by-measure highlights the blend's radius at 0.95121 mm but the product design specifications dictate that the blends should be at 1 mm. Before attempting to modify the blends, we notice that the blend radius faces (tangency conditions) in this area have imported incorrectly (highlighted in red). See below.



Correct tangency condition (green circle)



Incorrect tangency condition (red circle)

Figure 2:

Even though we noticed the incorrect blend tangency conditions, try to increase the blend radius anyway to see if the modification will automatically fix the problem – Change the blend radius from 0.95121 mm to 1 mm.

As seen in figure 2, the blend modification fails. At this point, we will use the flexibility of OneSpace Modeling's history-free modeling to fix this imported problem so we can move on with our design modification.

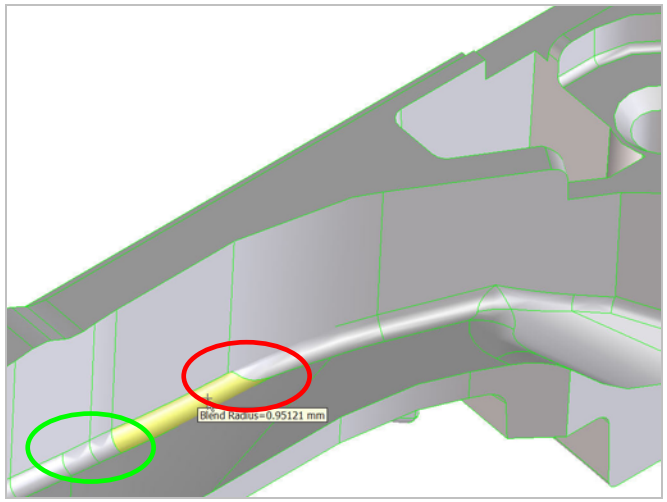


Figure 1 (Highlighting Blend Radius)

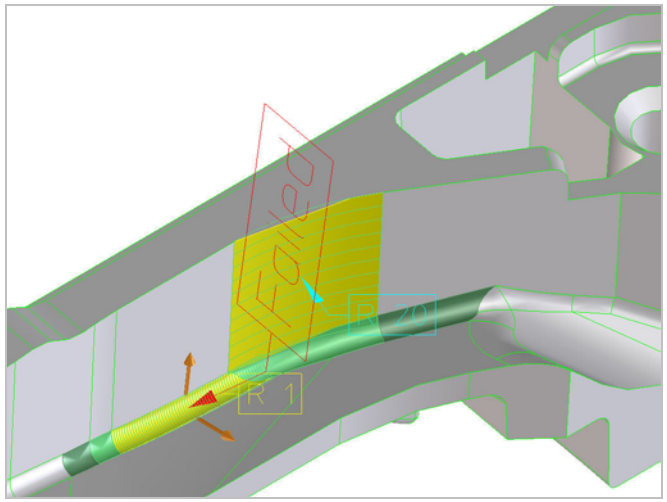


Figure 2 (Blend Modification Failed)

A Step-by-Step Process for Reducing Model Complexity

There's an old saying…

…among parametric or history-based CAD users that goes something like, "Never fill the hole that you've dug for yourself." However for OneSpace Modeling this concept is a valid modeling technique and it is how we will solve our import blend problem.

Modeling like it was clay:

By adding material on top of the problem or subtracting material from the problem area, the model will heal itself, which allows the OneSpace Modeling user to continue modeling and finish the design work.

The history-free concept:

Rather than digging a deeper hole at this point or starting over, OneSpace Modeling allows the designer to literally backfill extra material over the problem area. This in essence permits the surfaces to be recreated from the new geometry, which allows the modifications to be executed cleanly.

Figure 3:
Using a workplane, draw a closed 2D profile that completely surrounds the imported problem – See Figure 3. Instead of using a 2D profile for an Extrude or Mill command, another way to do this is to add or subtract basic 3D shape like a cube. The idea here is to cover or remove the imported problem area with a shape that is easily modifiable afterwards. The most common shape used by many OneSpace Modeling users is a cube, scaled in size when necessary.

Figure 4:
After creating the 2D profile, we use the Extrude command, with the option To Part. This adds material over the imported problem, which completely covers the incorrect blend tangency conditions.

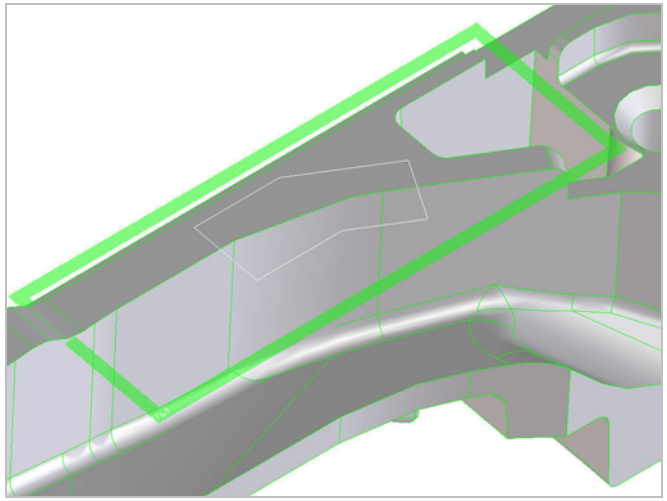


Figure 3 (Create a Workplane with 2D Geometry)

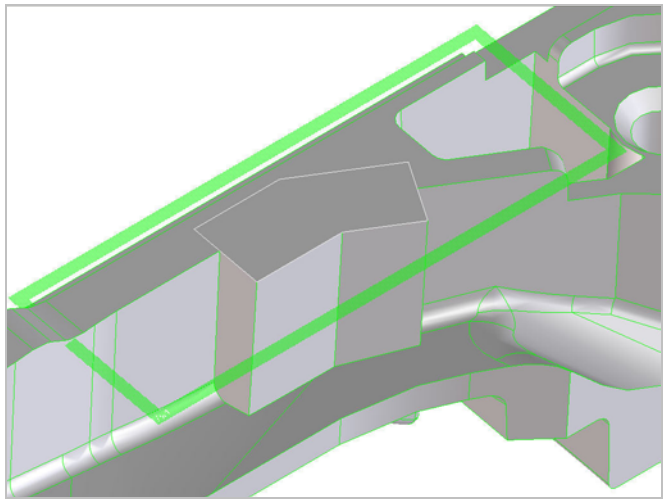


Figure 4 (Extrude 2D Geometry to Part)

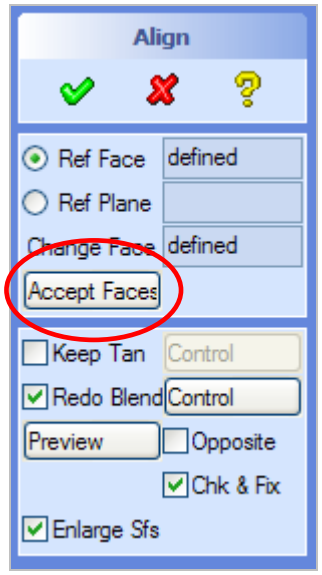
Figure 5:

A Step-by-Step Process for Reducing Model Complexity

Using the powerful Align command, select the four new faces (highlighted in yellow), which were created with the pervious Extrude operation, and align them to an adjacent face on the model (highlighted in green).

Notice that the tangential blend (highlighted in a dark green) will be recalculated because of the Align operation – it is at this point the OneSpace Modeling user knows that the imported problem has been resolved.

In a single Align operation, you can define multiple change faces (highlighted in yellow) to be aligned with a single reference face (highlighted in green) by using Accept Faces option.



Accept Faces – Example steps:

1. Select Change Face #1
2. Select Ref Face
3. Click **Accept Faces**
4. Select Change Face #2
5. Select Ref Face
6. Click **Accept Faces**
7. Select Change Face #3
8. Select Ref Face
9. Click **Accept Faces**
10. Select Change Face #4
11. Select Ref Face
12. Click **Accept Faces**
13. Click Preview
14. Click OK

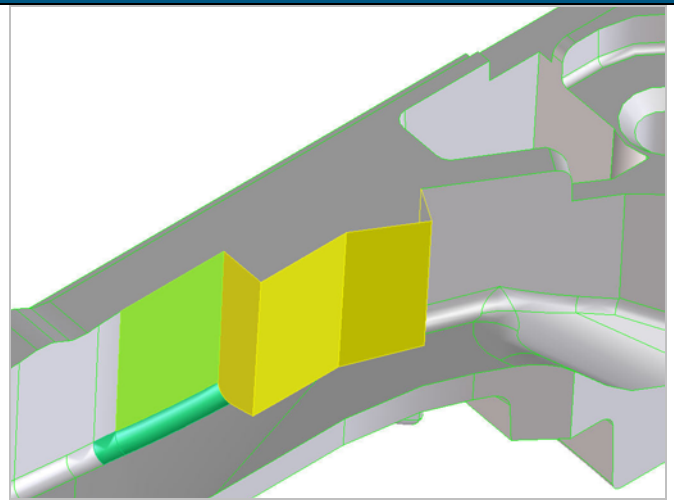


Figure 5 (Align the Faces back to Part)

Figure 6:
After the Align command removes the necessary added feature, which automatically healed the imported problem, all the blends are modified with the correct blend radius – See Figure 3 which shows the blend preview.

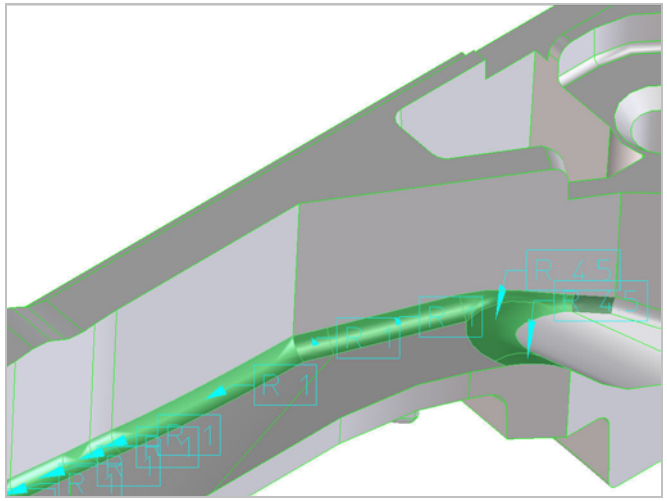
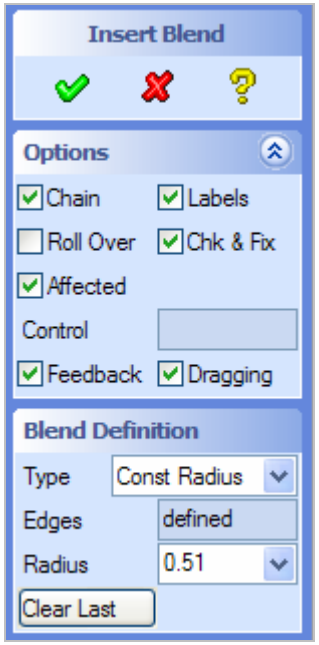


Figure 6 (Blend Chain Modification)

Figure 7:
Once the 1 mm blends are modified, the last step is to re-insert the 20 mm blend – See Figure 7.

A Step-by-Step Process for Reducing Model Complexity

This blend is created using the Insert Blend command.



What's the difference between Insert and Create Blend?

- Insert adds a new blend to surrounding blends (highlighted in green) recalculating the existing blends for a smooth transition
- Create produces a new blend without recalculating existing blends.

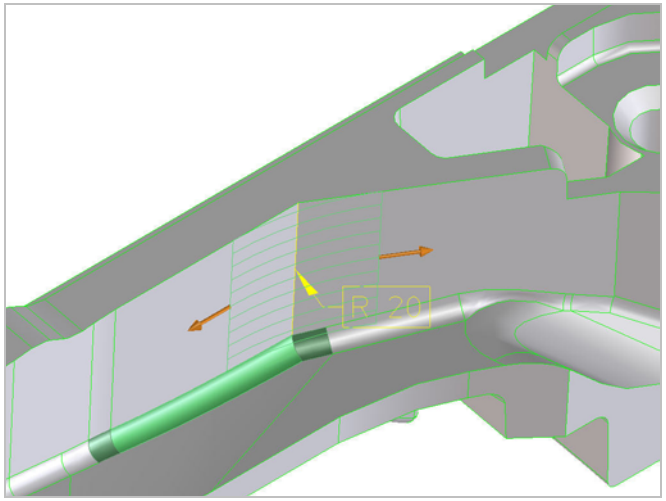
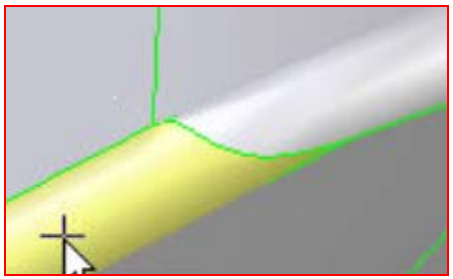


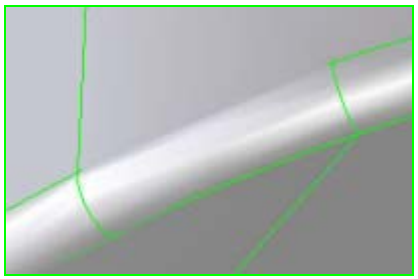
Figure 7 (Insert Blend)

Figure 8:

The blend modification was accomplished and all the blend tangency conditions are correct – See Figure 8



Before



After

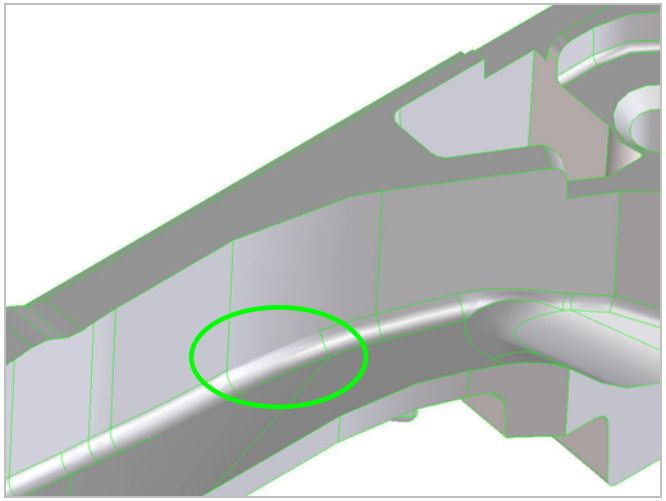


Figure 8 (Modification Complete)