

3D MODIFICATION TOOLS & COMMANDS

Vectorworks Tutorial by [Andy Broomell](#) © 2020.
Green text indicates advanced or supplementary notes.

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OVERVIEW

This tutorial provides guidelines for the following tools and commands which can be used to modify 3D objects:

2. Add Solids

3. Subtract Solids

4. Intersect Solids

5. Section Solids

6. Split Tool

7. Fillet Edge Tool

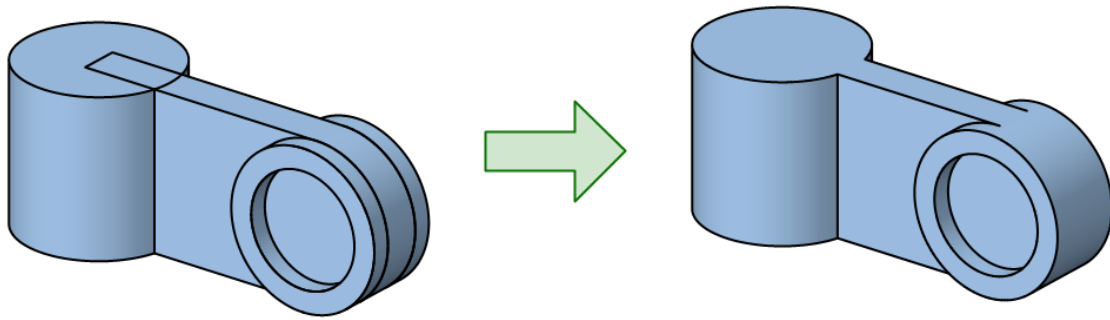
8. Chamfer Edge Tool

9. Shell Tool

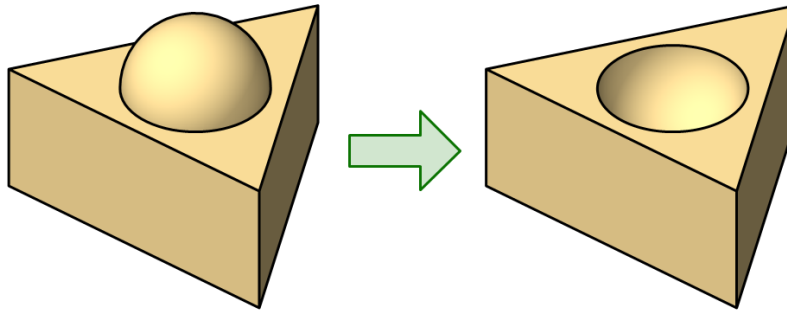
10. Modifying Fillets/Chamfers/Shells

11. Push/Pull Tool

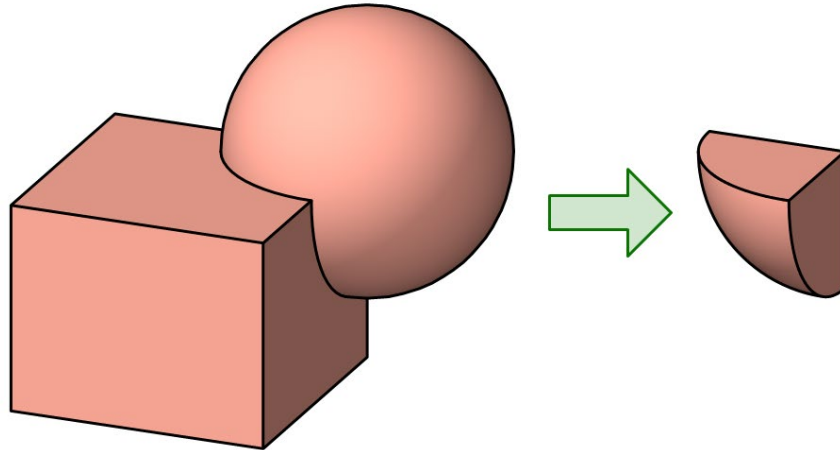
12. Taper Tool & Deform Tools



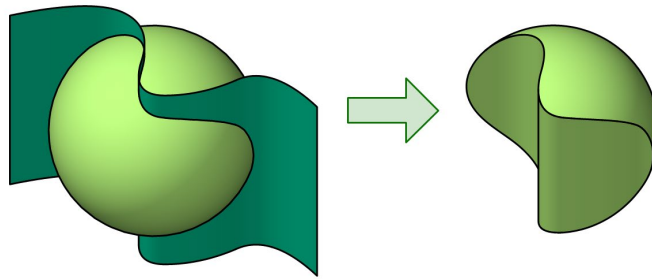
- The **Add Solids** command combines multiple 3D shapes into a single 3D object.
 - The shapes should usually be touching or overlapping.
 - This is analogous to the Add Surface command for 2D shapes.
- Select all the shapes you'd like to join together and go to **Model > Add Solids** (note the shortcut).
- The resulting object is called a **Solid Addition** in the OIP.
- You can **double click** the new 3D object to edit the original shapes involved in the operation, or to add additional 3D objects.
- To undo the operation and get back the original objects, select the Solid Addition and **Ungroup** it (Cmd+U / Ctrl+U).
- Use the Add Solids command when the need arises; it is not necessary to always add shapes together by default.



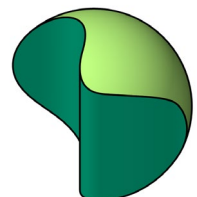
- The **Subtract Solids** command will subtract one or more 3D shapes from another 3D shape (the base shape).
 - The shapes should be overlapping.
 - This is analogous to the **Clip Surface** command for 2D shapes.
- Select all of the objects involved and go to **Model > Subtract Solids**.
- In the pop-up, use the arrows to select the base object (the one that remains). Decide whether the objects being subtracted should be deleted (default) or retained. Click OK.
- The resulting object is called a **Solid Subtraction** in the OIP.
- You can **double click** the new 3D object to edit the original shapes involved in the operation, or to add additional 3D shapes to be subtracted.
- To undo the operation and get back the original objects, select the Subtraction and **Ungroup** it (Cmd+U / Ctrl+U).



- As with the Add Solids and Subtract Solids operations, the **Intersect Solid** command requires two or more 3D objects. They need to be overlapping, because this command produces a new object based on the volume that **all original objects have in common**.
- Select the objects you'd like to intersect, then go to **Model > Intersect Solids**.
- The resulting object is called a **Solid Intersection** in the OIP, and can be **double clicked** to edit, or **Ungrouped** to get back the original shapes.




- The **Section Solids** command allows you to slice a 3D object with another 3D object. The slicing object should have no thickness, and should completely intersect the base object.
 - The slicing object in the example above is an extruded polyline.
- Select the shapes and go to **Model > Section Solids**.
- Use the arrows to select the **slicing object**. Click OK.
- The resulting object is called a **Solid Section** in the OIP.
- You can **double click** the new 3D object to edit the original shapes involved in the operation.
 - To undo the operation and get back the original objects, select the Solid Section and **Ungroup** it (Cmd+U / Ctrl+U).
- Note that in the OIP you reverse the direction of the cut.
- You can also check the “Use Section Color” checkbox in the OIP to make the sliced edge of the object use the fill color of the slicing object:



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SPLIT TOOL

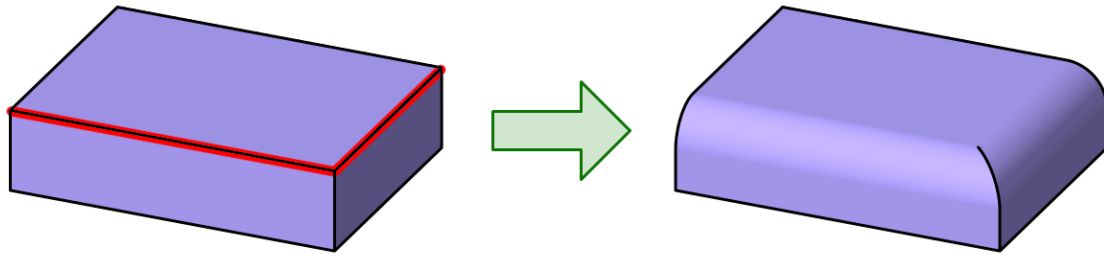
- The **Split** tool  in the Basic toolset (note the shortcut), is a tool that works on both 2D and 3D objects.
- When working with 3D objects, it essentially works like a knife slicing through the object, perpendicular to your computer screen.
 - You can use this tool from any view.
- Using the Split tool on a 3D object results in a **Solid Section** object, exactly the same as if you would have used the Section Solids command (previous page). It automatically creates the slicing plane along the line you draw.







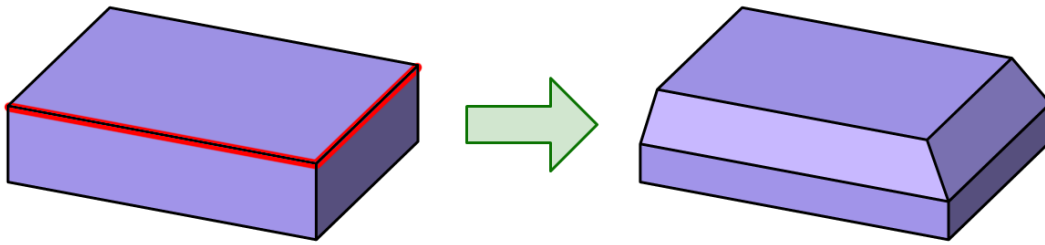
- It's important to note the **modes** of the Split tool, as the first mode doesn't work with 3D objects. The second mode retains both sides of the sliced object, while the third mode requires you to choose which side to retain. Typically you should use the "Currently Selected Objects" setting in the second mode group.





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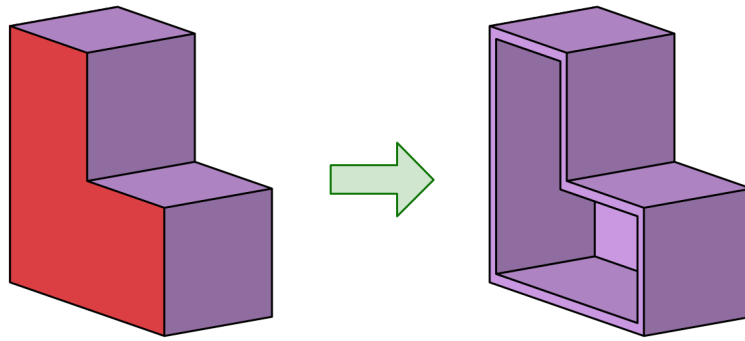
FILLET EDGE TOOL






- The **Fillet Edge** tool  , found in the 3D Modeling toolset  , allows you to create rounded edges on 3D shapes (analogous to the 2D Fillet tool).
 - If the rounded edge can be accomplished by editing the 2D shape inside an Extrude, do it there instead.
- After clicking on the tool, first click the **Preferences** button  in the mode bar. You'll typically use “**Constant Radius**” mode with nothing else checked (the other modes can be learned with experimentation or using VW Help). Also type in a **Radius**. Click OK.
- Select the edges of the object that you'd like to be filleted.
 - Hold down **Shift** to select multiple edges.
 - Hold down **Opt / Alt** to select obscured back edges.
- Click the **green checkmark**  in the mode bar, or press **Enter**.
- The resulting object is called a **Fillet** in the OIP, and the **radius** of the filleted edges can be adjusted at any point.
- If a Fillet fails to compute, the “Radius” is probably too large.



- The **Chamfer Edge** tool , located in the 3D Modeling toolset , works similarly to the Fillet Edge tool, except that it creates faceted corners instead of rounded corners.
- Enter a **Setback** value, then select the edges you wish to chamfer.
- Click the **green checkmark**  in the mode bar, or press **Enter**.
- Note that you can set the **Preferences**  for both the Chamfer Edge tool and Fillet Edge tool to select entire **faces** instead of **edges**.







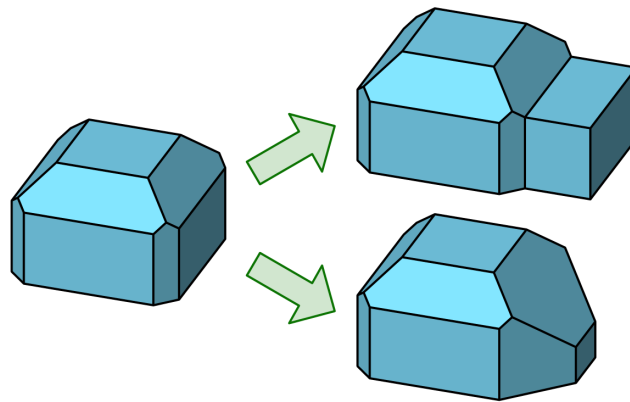
- The **Shell Solid** tool creates a hollow shell from a solid object.
 - It can also be used to give thickness to NURBS surfaces and planar objects.
- Click the Shell Solid tool , located in the 3D Modeling toolset , then type a **Thickness** into the mode bar.
- Select the face which you want to be “open”.
 - Hold down **Shift** to select multiple faces.
 - Hold down **Opt / Alt** to select obscured back faces.
- Click the **green checkmark**  in the mode bar, or press **Enter**.
- The resulting object is called a **Shell** in the OIP, and the **thickness** of the shell can be adjusted at any point.
 - You can also adjust whether the thickness is applied to the **Inside** or **Outside** of the original 3D shape.

- Vectorworks 2019 and older only: For 3D Fillets, Chamfers, and Shells, unfortunately there's no way to go back and modify which edges / faces are being used in the operation once the checkmark has been clicked, nor can you double click the resulting object to go back and modify the original shapes.
- However, you can **Ungroup** (Cmd+U / Ctrl+U) any of these objects to undo the modification and get your original shapes back. Then you can perform the operation again from scratch.
- As of VW2020, double clicking any of these object types will take you back to the original 3D object. Or, to modify which edges are affected, click the "Modify" button in the OIP.

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PUSH/PULL TOOL

- The **Push/Pull** tool , located in the 3D Modeling toolset , allows you to interactively reshape flat sides of a 3D object. You can pull outward to create protrusions, or push inward to remove volume.
- Unless you want to subdivide an existing face, you'll use either the first mode (Extrude Face ) or the second mode (Move Face ).



- When push/pulling the top or bottom face of a regular Extrude, this tool will simply alter the “Extr.” value, and the object will remain an Extrude. (Just like using the Reshape Tool covered previously).
- Push/pulling any other side of an object will create a Solid Addition or Solid Subtraction. (So using this tool extensively can create memory-hogging complex solids many ‘levels’ deep.)
- Some operations create Generic Solids which are un-editable.

- The **Taper** and **Deform** tools involve a bit more skilled clicking than the other tools in this tutorial, but both can create shapes that no other tools can create.
- Take a look at the help files within Vectorworks to read more about how to use these tools, including links to video tutorials on Vectorworks' YouTube channel.

- [Taper Tool Help](#)
- [Deform Tool Help](#)



- Note that these tools result in **Generic Solids**, a type of object with no history of how it was made. Modifications made with the Taper Face or Deform tools cannot be reversed in the future (except for immediate undoing).