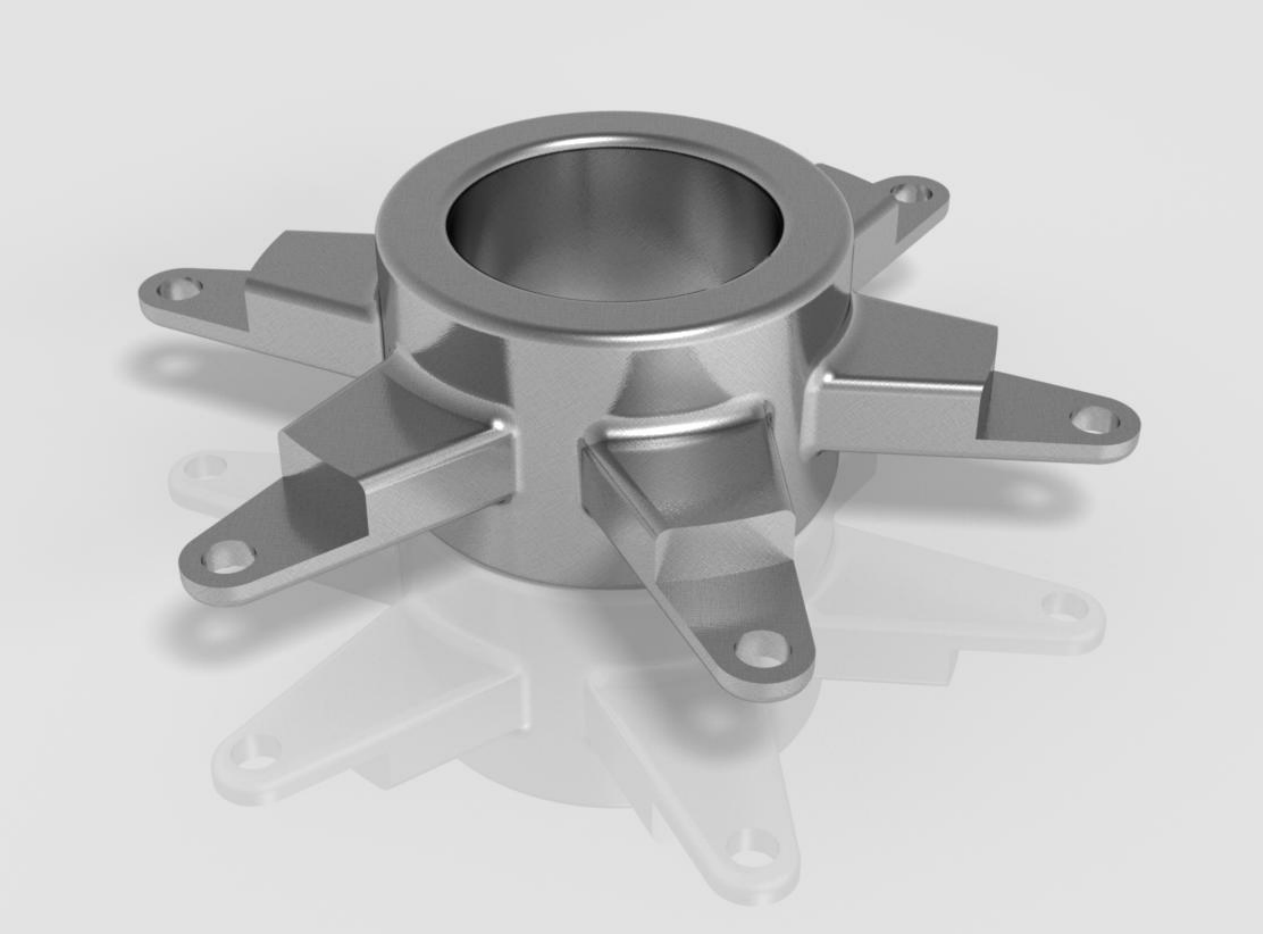
Autodesk Manufacturing Workshop

January 31st, 2013

iDESIGN solutions

Autodesk Manufacturing- Goal



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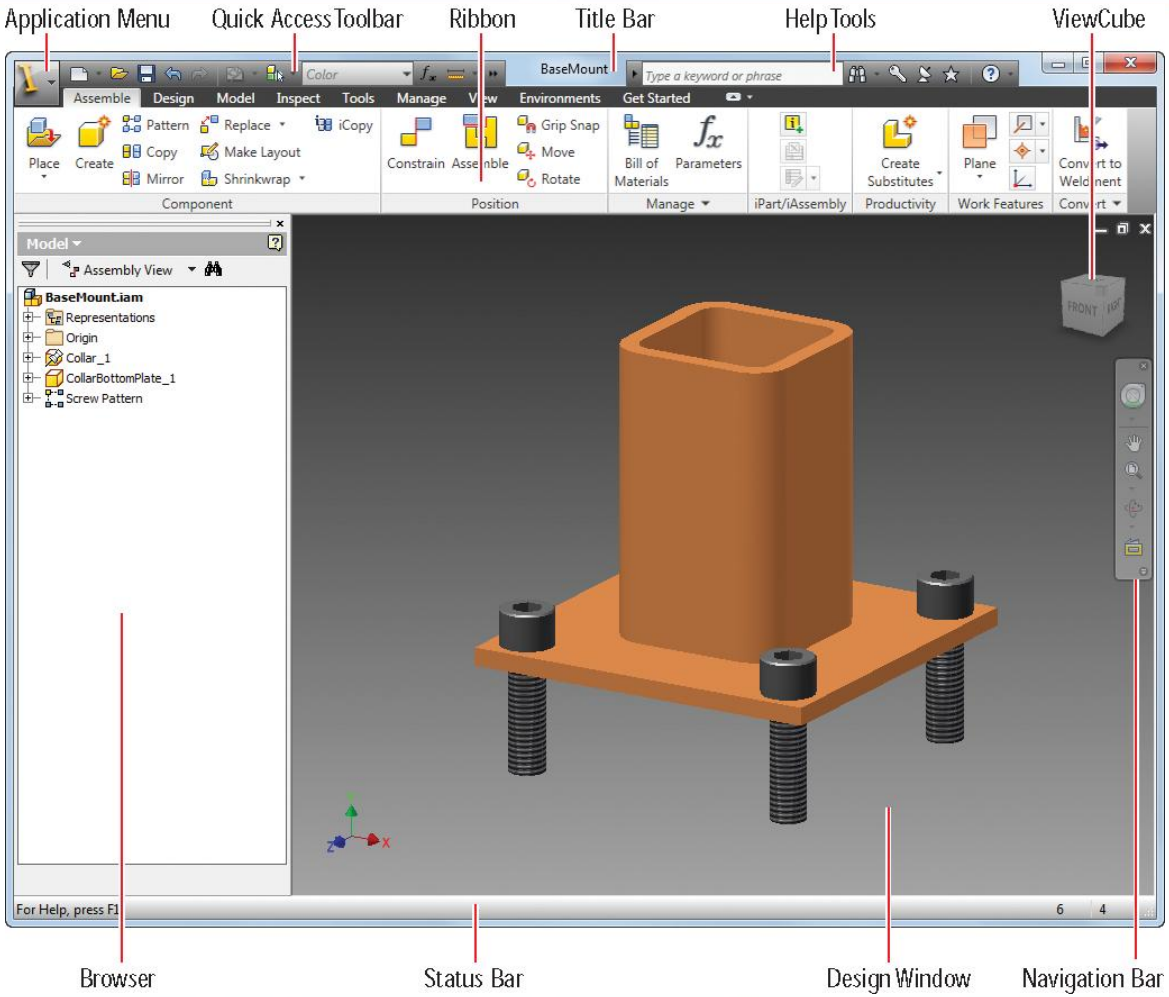
Autodesk Manufacturing- Objectives

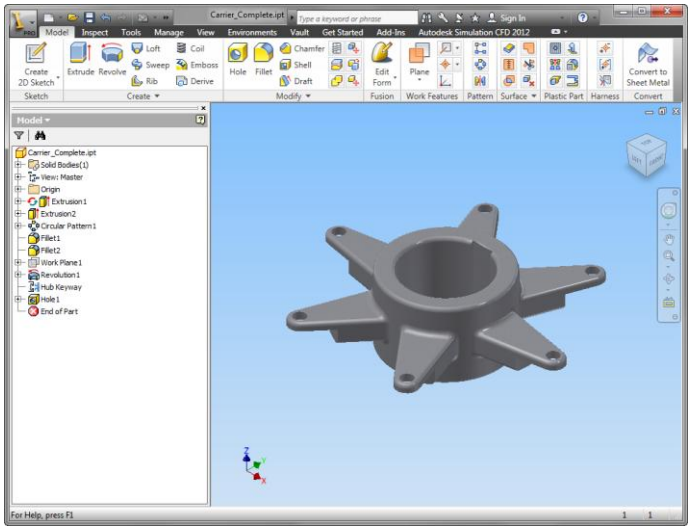
- Control where Inventor saves and accesses data
- Open an existing Autodesk Inventor file
- Review the primary elements of the Inventor user interface
- Develop a sketch controlled by dimensions and geometric constraints
- Build a 3D model of a Sprocket Carrier
- Create a 2D Production drawing of the Sprocket Carrier
- Update the drawing by modifying the of the Sprocket Carrier

Autodesk Inventor User Interface



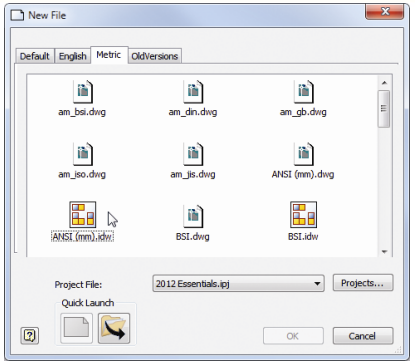
Autodesk Manufacturing - Instructions

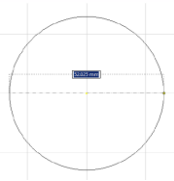
- Set your Project File to *c:\Shifter Cart\Shifter Cart.ipj*
- Open *c:\Shifter Cart\Carrier\Carrier Complete.ipt*
- Change active tabs in the Browser
- Select different faces on the ViewCube and restore the Home View
- Use F2, F3, and F4 to Pan, Zoom and Orbit the Model and Use Mouse wheel to control
- Change Graphics Window color using Application Options
- Hover over features in the browser to highlight part features
- Experiment with part color using pulldown in Quick Access toolbar

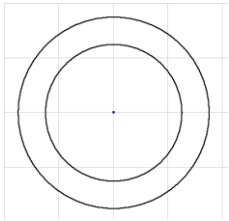


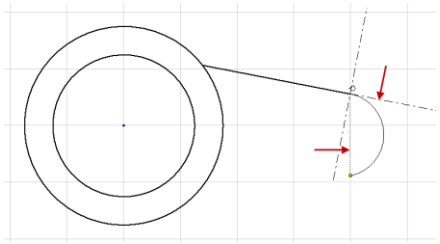
Autodesk Manufacturing - Instructions

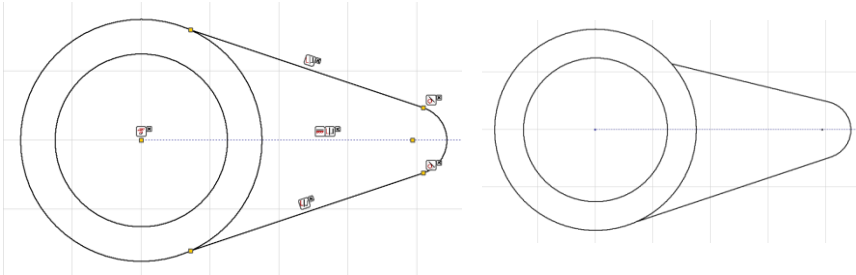
- Create new part using the *Standard (mm).ipt on the Metric tab*
- Draw Circle ~50mm in diameter
- Draw second, larger, concentric circle
- Create two lines and an arc using the line tool (remember: drag from endpoint to create arc)
- Add Horizontal Construction line
- Show sketch constraints using F8
- Constrain arc to Construction line.
- Add Symmetric constraint to lines about Construction Line











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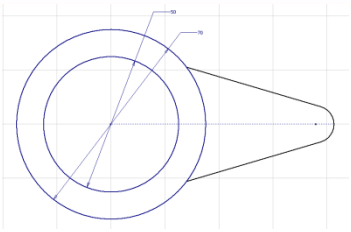
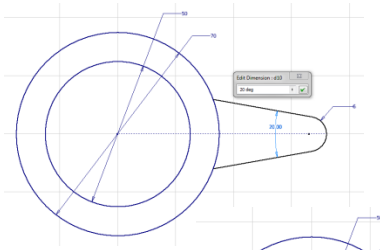
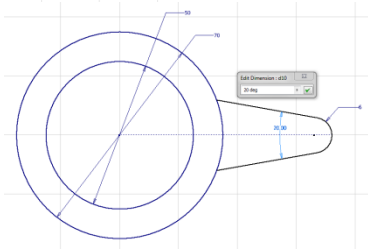
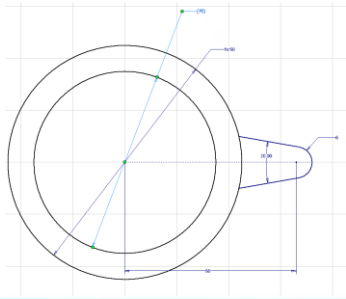
Autodesk Manufacturing - Instructions

- Continue using current .ipt or open *c:\Shifter Cart\Carrier\Sketch Phase 1.ipt and edit sketch*
- Add *50mm* and *70mm* diameter dimensions
- Add *6mm* radius and *20 deg* angle dimensions
- Set Spoke length to be *66mm* from center
- Set outer diameter to be equal to the *inner diameter + 20mm*
- Change inner diameter to *70mm* then make it a driven dimension
- Finish Sketch









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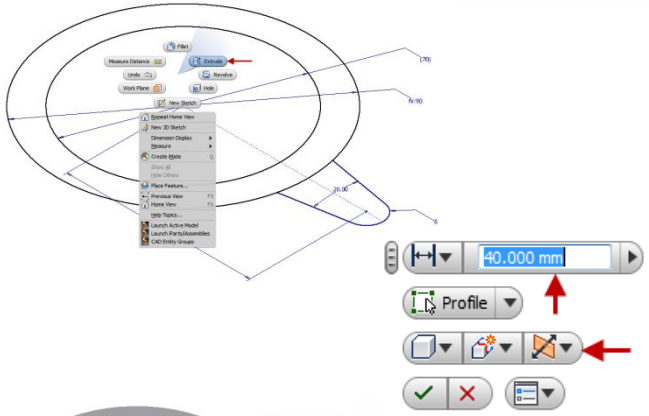
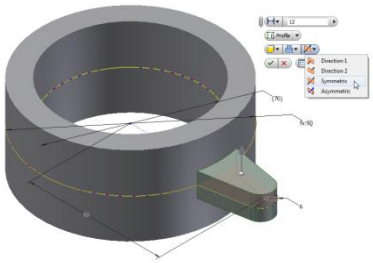
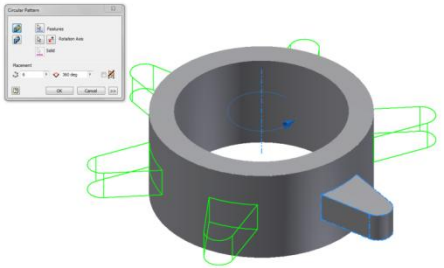
Autodesk Manufacturing - Instructions

- Continue using current .ipt or open *c:\Shifter Cart\Carrier\Sketch Phase 2.ipt*
- Extrude hub 40mm using Symmetric option
- Share sketch to reuse for second extrusion
- Extrude Spoke 12mm using Symmetric option
- Turn off shared sketch visibility
- Create Circular Pattern of six spokes about hub axis
- Save your part to the *c:\Inventor\Carrier* as *My Carrier.ipt*
- Close the file







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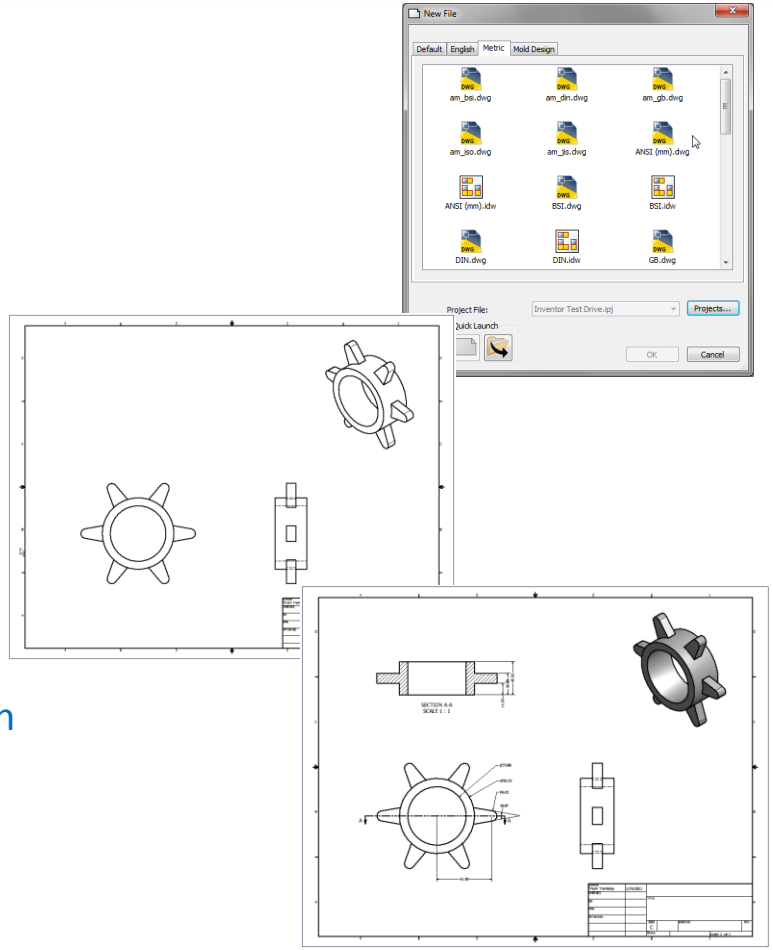
Autodesk Manufacturing - Instructions

- Open your *My Carrier.ipt* or open *c:\Shifter Cart\Carrier\Carrier Phase 1.ipt*
- Create new drawing based on the ANSI(mm).dwg template on the Metric tab
- Create a Base view of the Top of the Carrier and project a view to the right and an isometric view
- Double-click to edit the iso view and make it shaded
- Create a section view above the top view
- Right-click in the top view and retrieve model dimension
- Change Dimension Styles using Annotate Tab
- Add Baseline dimension set to the section view



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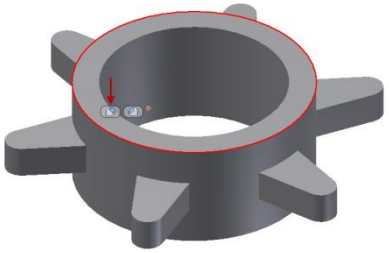
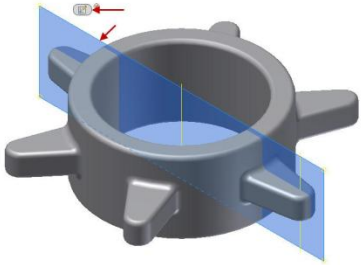
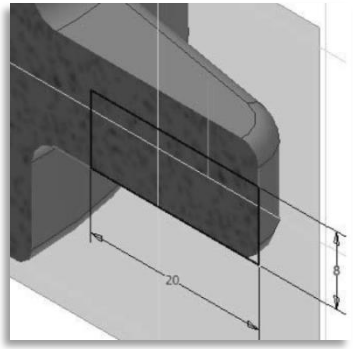
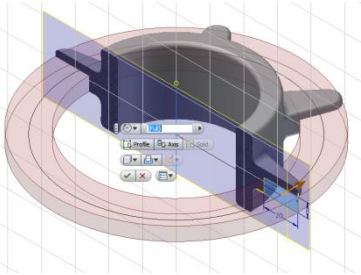
Autodesk Manufacturing - Instructions

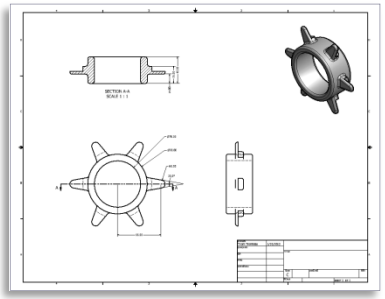
- Edit the part that you made your drawing views from or open c:\Shifter Cart\Carrier\Carrier Phase 2.ipt
- Add *2mm* fillet to hub edges and *All Rounds*
- Change *2mm* value to *3mm* and add *3mm* fillet to *All Fillets*
- Create a new sketch on the Origin XZ Plane and use F7 to Slice the Graphics of the part
- Project edges of Spoke and add 20mm x 8mm rectangle and Revolve making a cut
- Review Mass Properties (iProperties) from the browser
- Return to Drawing to review changes











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Autodesk Manufacturing - Questions

- Can you name two types of sketch constraint?
- What type of line is not used to create a 3D feature?
- How can you reuse a single sketch for multiple features?
- What is the name of the first drawing view placed?
- How can you reuse sketch dimensions in a 2D drawing view?
- What is the easiest way to update a 2D drawing view?

Autodesk Manufacturing - Goal

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Autodesk Manufacturing - Objectives

At the end of this workshop you will be able to:

- Review the use of construction surfaces in a solid model
- Develop a seat using surfaces to modify a solid
- Utilize symmetry to simplify modeling the entire seat

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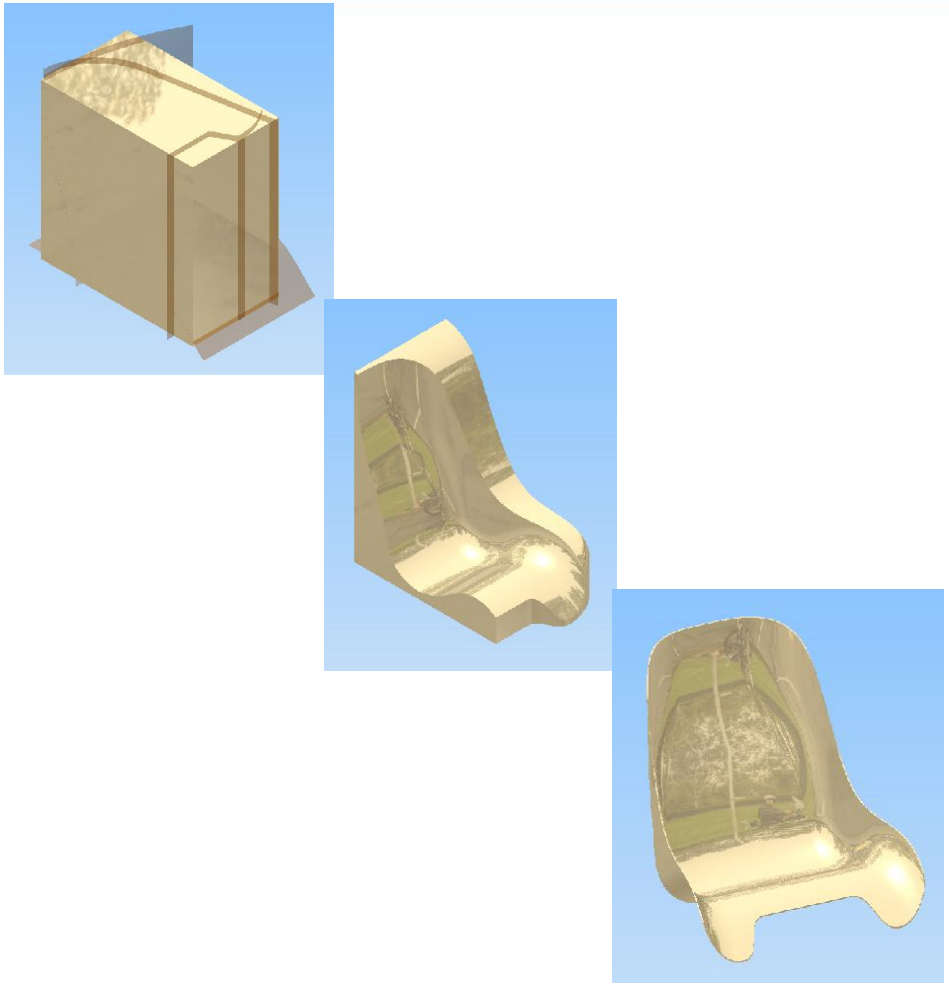
Autodesk Manufacturing - Hybrid

- Part modeling doesn't need to fall into surface or solid; it can be both
- Complex contours do not need to be created in one single step
- Traditional surface to solid workflows of forming "water tight" surfaces are not necessary
- The Sculpt tool can use combinations of solids, surfaces and work planes
- Look for opportunities to leverage symmetry (planar or axial)



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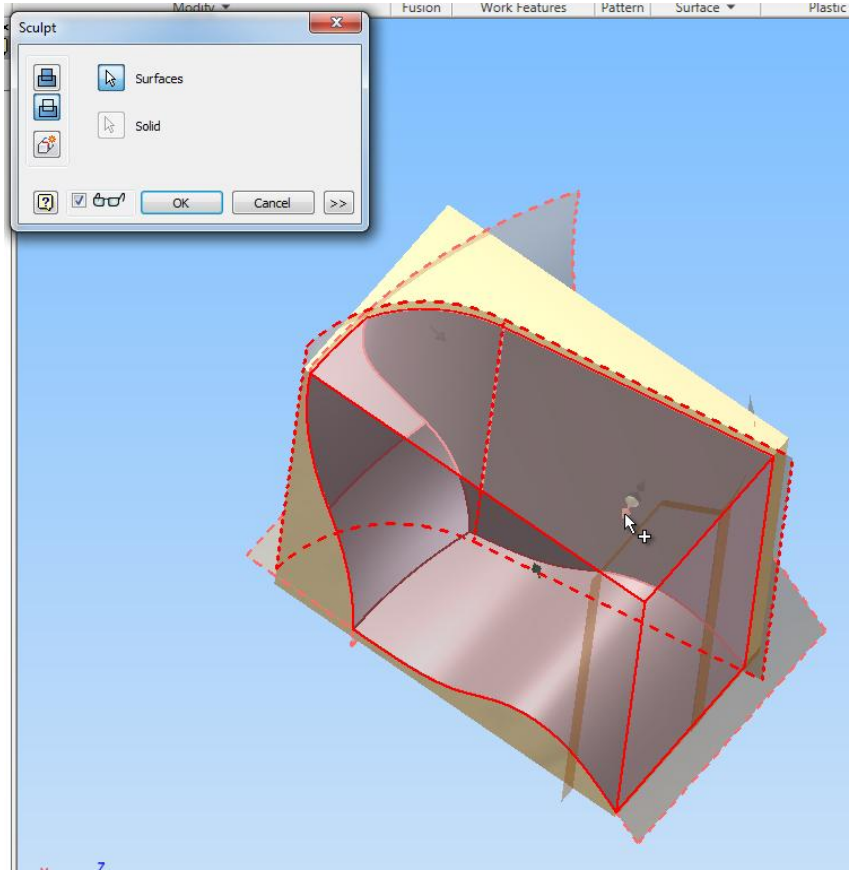
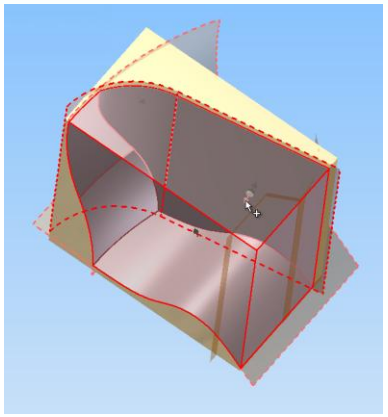
Autodesk Manufacturing - Hybrid

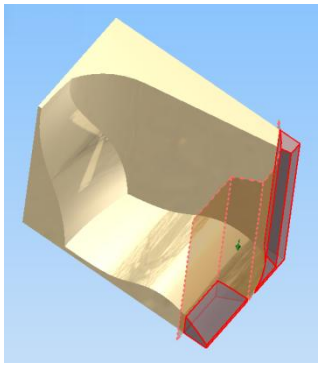
- The key is dissecting a finished shape into a series of sub-shapes
- Create a series of overlapping surfaces. This can include existing solids
- Select surfaces to fill in between or to remove between
- Use fillets to round edges rather than including them in loft profiles

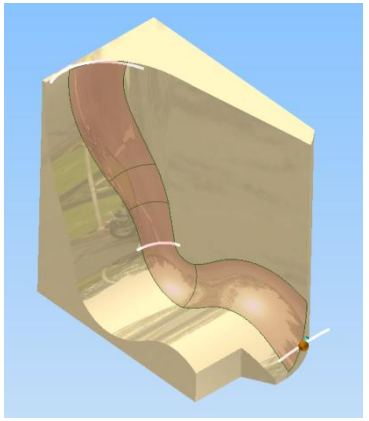


Autodesk Manufacturing - Instructions

- Open *c:\Shifter Cart\Workgroup\New Seat.ipt*
- Review Revolution, and Extrusion surfaces
- Start the Sculpt tool from the Surface panel of the Model tab
- Pick the first three surface and set the sculpt function to Cut.
- Reverse the direction of the side surface
- Use Sculpt to remove the front of the seat
- Add 80mm edge fillet to back of seat interior
- Add variable radius add a third radius point to the base of the back
- Set radius transition to 120 -> 50 -> 70
- Add 30 mm radius to sharp edges at front







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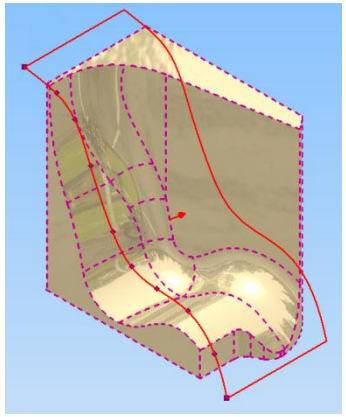
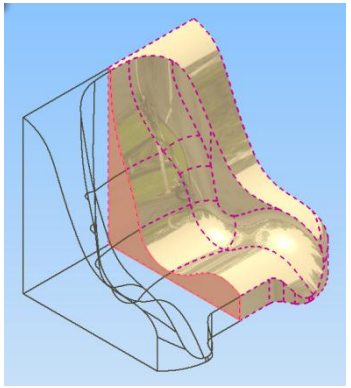
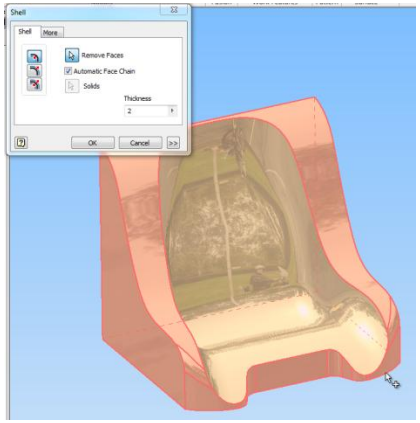
Autodesk Manufacturing - Instructions

- Move the End of Part marker to the bottom of the browser
- Start the Split tool from the Modify panel
- Select the sketch and set option to Trim Solid
- Add a 30mm fillet to the sharp edge created by the split
- Set the direction of the removal toward the top
- Start the Mirror tool from the Pattern panel and set the option to Mirror a Solid
- Select the side of the seat for the mirror plane
- Start the Shell tool in the Modify panel, set the thickness to 2mm and select all faces except the seat interior







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Autodesk Manufacturing - Questions

- What tool uses overlapping surfaces to create or edit a solid model?
- What type of fillet can place more than one radius on a single edge?
- What tool did you use to make the final body of the seat?

Autodesk Manufacturing

Thank you for time…

Please take a moment to fill out our feedback survey

http://www.surveymonkey.com/s/HK5FZXY

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