Install and Start Fusion 360

Before the workshop begins, please do the following:
1. Log into the computer
2. If needed, create a free Autodesk Account, and download Autodesk Fusion 360 from Autodesk Education Community: [www.autodesk.com/workshop](http://www.autodesk.com/workshop)
3. Start Fusion
The Future of Making Begins Now

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Agenda

✓ Introductions
✓ Overview: The Future of Making Things
✓ Hands-on Fusion 360 workshop
  ▪ 3D Modeling
  ▪ Rendering
Introductions

- Dan Banach: Technical Manager NA MFG, Autodesk Education Programs
  - Using CAD products since 1988
  - 25 years teaching and consulting on CAD products
  - Authored 24 books on Autodesk 3D mechanical CAD software

- Your Experience
  - AutoCAD, Fusion 360, Inventor, SolidWorks, ProE, Others?
Enclosure inspired by Cy Keener:
https://www.autodesk.com/artist-in-residence/artists/cy-keener
Projects
- Preferences
  - Z up
  - Start a new file
- Data Panel
- Invite People
User Interface

1. Application bar
2. Profile & help
3. Toolbar
4. ViewCube
5. Browser
6. Canvas & marking menu
7. Timeline
8. Navigation bar & display settings bar
### Navigation

#### Mouse

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SCROLL</td>
<td>Scroll the middle mouse wheel to zoom in or zoom out.</td>
</tr>
<tr>
<td>HOLD</td>
<td>Click and hold the middle mouse button to pan the view.</td>
</tr>
<tr>
<td>SHIFT KEY +</td>
<td>Hold the SHIFT key and click and hold middle mouse button to orbit the view.</td>
</tr>
</tbody>
</table>

#### Mac Trackpad

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2 finger pinch</td>
<td>to zoom out.</td>
</tr>
<tr>
<td>2 finger spread</td>
<td>to zoom in.</td>
</tr>
<tr>
<td>2 finger swipe</td>
<td>to pan the view.</td>
</tr>
<tr>
<td>SHIFT +</td>
<td>Hold SHIFT and 2 finger swipe to orbit the view.</td>
</tr>
</tbody>
</table>
Create the Lens

1. If needed change the Units to Inch
2. Sketch a **2.75 inch** circle that is on the plane that is parallel to the Top plane
3. Extrude the circle **0.125 inches**
4. Change the material to Glass
5. If desired change the appearance
6. Save the file with the name **Lens**
Browser and Timeline

- **Browser**
  - Left side of screen
  - Top level data: components, joints, etc.

- **Timeline**
  - Bottom of screen
  - List of features
    - Edit features
    - Change order of a feature
    - Delete features
Create the Base

1. Start a new file +
2. Create new component with the name **Base** and ensure it’s active
3. Sketch, constrain, and dimension the profile as shown
   1. Create the sketch on the plane that is parallel to the Front plane
   2. For the outside profile draw a CV Spline with 4 points, add a horizontal & vertical constrain to the construction outside lines
4. Revolve the profile around the inside vertical line of the profile
5. To prevent the Base component from moving ground it by right-clicking on the **Base** in the browser and click Ground
Add Features to the Base

1. Create a tapped hole
   1. Place and dimension a Sketch Point on the top planar face
   2. Place a **#10-24 UNC x .5 inch** deep tapped hole
   3. Circular Pattern the tapped hole with a quantity of 4

2. Sketch, constrain and dimension the profile as shown
   1. Extrude the circle **0.25 inches**
   2. Add a **#3-48 UNC x .375 inch** deep tapped hole
   3. Circular pattern the extrude and hole feature with a quantity of 4

3. Save the file with the name **Enclosure Assembly**
Create the Cover

1. Make the top level Enclosure Assembly active
2. Create new component with the name **Cover** and ensure it’s active
3. Create a sketch on the top horizontal plane of the Base
4. Extrude the inside and outside profiles **0.25 inches**
Add features to the Cover

1. Add a 0.25 inch fillet to the top-outside edge
2. Add a 0.125 inch fillet to the top-inside edge
3. Make the top level assembly active
4. Save the file and a description “Added Cover”
Assembly Joints

- Before applying Joints, ground at least one Component that will NOT move
- If needed insert a model into the assembly
  - From the Data Panel right-click on file and click Insert into Current Design
  - Files must already exist in a Project you have access to
- Apply Assembly Joints
  - Joints can only be applied to components
  - Assembly > Joints
  - Joints are applied to points, axes and planes
  - Edit Joints via the Browser
Constrain the Base and Cover Components

1. Prove that the Cover can move by clicking and dragging on the Cover
2. Stop moving the Cover by pressing the Esc key (the Cover should return to its original position)
3. Since the Base and Cover components are in the correct location, constrain them with an As Built Joint with Rigid as the Type: Assemble > As-built Joint
4. Turn off the visibility of the Base
Place the Lens into the Assembly

1. From the Data Panel drag the **Lens** component into the Enclosure Assembly
2. Add a Rigid joint between the Lens and the Cover
   1. Rotate the viewpoint so you can see the bottom of the Cover
   2. Select the bottom circular face and then the center point of the Lens
   3. Select the bottom circular face and then the center point of the Cover

**Note:** Always select the component that will move first
Remove Material from the Cover

1. Use the Combine command to remove the area where the Lens interferes with the Cover: Modify > Combine
   1. Target = Cover
   2. Tool Bodies = Lens
   3. Operation = Cut

2. Remove additional material for the Lens to fit
   1. Turn off the visibility of the Lens
   2. Right-click and click the Press Pull command from the menu
   3. Select the inside circular face out **-0.01 inch**
   4. Turn the visibility of the Lens back on
Edit the Lens & Update the Assembly

1. Make the Lens file current
   1. Edit Sketch1 and change the diameter to 3 inches
   2. Save the file and add the description Changed diameter
2. Make the Enclosure Assembly file current
   1. You’ll be alerted that the Lens file is out of date, update to the latest version by right-clicking on the Lens entry in the browser and click Get Latest
3. Turn on the visibility of the Base component
4. Turn off the visibility of the Joints
5. Save the file and add the description Updated Lens
Create the Handle

1. At the top level assembly, create new component with the name **Handle**
2. Draw two **0.375 inch** diameter circles on the top planar face of the Cover
3. Extrude the outside circular profiles **0.375 inches**
4. Create a new Form, this starts the Form workspace
   1. Form = Organic Shapes
Create the Handle - continued

1. Draw a Fit Point Spline with 4 points that represents the center of the handle
   1. Use the origin plane that is parallel to the Front plane
2. Use the Pipe command to create a **0.25 inch** diameter handle about the spline
3. Smooth out the handle; from the Segments tab, drag the Density to the left
Selection Techniques & Tools

- **Window** (left to right): Selects only what’s 100% in the area
- **Crossing** (right to left): Selects what’s in the area & what the rectangle touches
- **Ctrl and Shift keys**: Adds or removes data
- **Double-click an edge**: Selects the entire loop
- **Freeform**: Define a closed area by dragging mouse
- **Paint**: Drag mouse
Edit Form

- Move, Rotate, Scale, Points, Edges, Faces
- Reorient Triad Direction
- Filter
- To add faces, hold Alt key while editing

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Create the Handle - continued

1. If needed Insert Edges & points: Modify > Insert Edge
2. Use Edit Form tool to alter shape
   1. Ensure the ends of the sculpt do NOT enter into the .151 inch hole
   2. Press ALT key to add faces
3. To mirror the sculpt body use the Symmetry > Mirror - Duplicate command
4. Delete the center faces that overlap
5. Fill the gap with the Modify > Bridge command
Create the Handle - continued

1. Finish Form
2. Turn the Handle Surface body and the two solid bodies into one body
   1. Start the Create > Boundary Fill command
   2. Select Tools = Two solid bodies and the surface body
   3. Select Cells = Check the three boxes
   4. Operation = New Body
   5. Turn off the visibility of the first three bodies
3. Change the Material to ABS Plastic
4. Make the top level assembly current, and save the file with the description
   Added a Handle

NOTE: Bodies vs. Components
✓ Bodies are modeling tools
✓ Components represent real-world parts
Insert Screws from McMaster-Carr

1. Insert a **#10-24 x .75”** & a **#10-24 x 1”** Round Head Screw
2. Change the file type to STEP
3. Add a Revolute Joint between the screws and the holes on the Handle and the Cover: Assemble > Joint
4. If desired insert O-Ring “1182N232” in the Base
Change the Material and Appearance

1. Change the Material of the components: Right-click > Material and drag onto a component
2. Change the Appearance of components: Right-click > Appearance and drag onto a component or a face
   - Appearance only changes how the component or face looks
3. Add a Decal: Insert > Decal
4. Save the file, and add the description **Added Decals**
Workspaces

- **Model**: Create solids with hard edges and flat faces
- **Patch**: Create open surfaces to stitch into solid bodies
- **Sheet Metal**: Create folded and flat sheet metal components
- **Render**: Set up the environment and create photo-realistic renderings
- **Animation**: Create exploded views, use for assembly instructions
- **Simulation**: Run an analysis on a component or assembly
- **CAM**: Create and simulate tool-paths then generate G code for subtractive manufacturing
- **Drawing**: Create 2D views or a component or assembly
Static Stress Simulation

1. Make the Simulation workspace current
2. In the Simply workspace Remove all the components and bodies except the Handle
3. For the Study Type, click Static Stress
4. Add the following Loads
   1. Constraints = two top faces of the bosses
   2. Loads = 10 lbs on the handle
5. Run the simulation
6. Review the results
Thermal Simulation

1. Make the Simulation workspace current
2. In the Simply workspace Remove the bolts, handle, and gasket
3. For the Study Type, click Thermal
4. Add the following Loads
   1. Radiation = 50 C on the top face of the PCB Board
   2. Convection = 5 outside faces
      1. Value = 50.00 W / (m^2 K)
      2. Ambient Temp. = 12 C
3. Contacts = Automatic Contacts
5. Run the simulation
6. Review the results
Animation
1. Make the Animation workspace current
2. Adjust the viewpoint
3. Transform components
4. Edit Storyboard
5. Add Trail Lines
6. Play animation
CAM

1. Open the file Enclosure Assembly CAM by clicking "<", then click Workshops & Events > Edu Design Now > University Workshops
2. Run the Simulation for Setup 1
3. Setup a new machine operation
   1. Stock:
Create the PCB Board

1. Make the top level assembly active
2. Create new component with the name PCB Board and ensure it’s active
3. Create a sketch on the top-horizontal plane on one of the inside extrusion of the Base
4. Project the other three holes on the inside extrusions of the Base
5. Draw a X inch circle
6. Extrude the profile using the Create > Create PCB > PCB Profile command
   1. Exclude the inside circular faces of the 4 holes
Next Steps & Learning Resources

 ✓ Download Fusion 360 from www.autodesk.com/workshop
 ✓ Download Autodesk software from www.autodesk.com/education
 ✓ Fusion 360 Learning Resources
   1. Fusion 360 Foundational Concepts
   2. Getting Started with Fusion 360 Tutorials
   3. Fusion 360 - Product Design Course Series on Design Academy
   4. Fusion How to Videos
   5. Fusion YouTube Channel
   6. Introduction to CAD and CAE using Autodesk Fusion 360 course
      1. Instructor curriculum
      2. Self-paced course

Request a Fusion Team Hub: http://autode.sk/FusionTeam [Case sensitive]