Fusion 360

Document Outline Layers

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Overview

Fusion 360 is a powerful tool that allows you to create in 3D. Like many fully-featured applications, the program can be very complex, so we will focus this program on the basics of 3D modelling for the purpose of 3D printing.

Workspace

Basics of the toolbar

1) Menus are context dependent. If you go to the different tabs, you will see different options, and if you go to different workspaces you will see different options.
2) Workspace menu
3) Timeline

Simple sketch, navigation, extrusion

We are going to start by modelling a coat hook.

1. To begin modelling, we need to need to click the create a sketch button
2. I like to think of planes visually rather than thinking about X,Y,Z. We can always rotate the object before printing. I almost always choose the bottom plane
3. Sketching
   a. Line tool
       i. Click to start at the origin
       ii. Type in the dimension and press enter
       iii. Continue the line, enter the dimension, then press tab to set the angle. Press enter one more time to complete the top, then press ESC.
       iv. Do the same with the complementary angle at the bottom.
   b. Arc tool

        i. Click the tangent arc tool at the completion of the bottom line

        ii. Enter the dimension and press enter

    c. Modify > Offset

        i. We will now select all of the sketch and click Offset in the Modify tab

        ii. Offset the sketch by the dimension of your choosing then press enter

    d. Connect the open segments with the line tools

4. Extrusion

    a. Click on the face of the sketch that was just created, then click extrude

    b. Raise the arrow to the thickness you would like and click OK

5. **Navigation:** For the easiest navigation, I would recommend you have a mouse that has a left click, right click, and a scroll wheel, if possible.

    a. Right click: revolves in 360 degrees

    b. Scroll wheel

        i. Hold: pans

        ii. In/Out: Zoom

    c. View cube

        i. Click on a pre-defined view

        ii. Click and hold and drag to view from the perspective you choose

    d. Bottom menu

        i. Orbit

        ii. Look at

        iii. Pan

        iv. Zoom in/Out

Going Further

- We can add additional sketches to the faces and add holes and threads for screws
  - Create a sketch
  - Add two cylinders in the middle of the coat hook's face
  - Highlight both cylinders and click extrude. This time we will extrude in the negative direction to create a hole
  - Even though it doesn't appear that the holes are there, they are still shapes, and you can highlight them. Do so, then click create > thread. Choose the measurements you would like to add
  - You can add features like a chamfer or other additional styles in the Modify tab
- When you have finished click Save, then go to File > Export
- Name your project and export as a .stl
- You can now [submit it to the Library](#) or another service for manufacturing
- To learn more, a good course available for free through the Library's LinkedIn Learning subscription is [Fusion 360 Essential Training](#)