

College of Engineering High School Outreach Program

Session # 3 Introduction to 3D CAD - Extrusions

Agenda

- Creating a 3D Extrusion from a 2D Sketch
- Solid extrusions and hollow extrusions

Reference

Inventor® R10, Chapter 2, pg 2-12 to 2-27

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3D CAD Terminology

- An Inventor **part** is the 3D “virtual” representation of a real object
- **Extrusions, cuts, holes and revolutions** of 2D-shapes create **features** of the part
- Multiple parts can fit together to create an **assembly**

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Extrusions

- An **extrusion** is created by dragging or “extruding” a 2D sketch/shape into the third direction i.e., by either adding **height or depth** to a sketch.
- Extrusions can create either:
 - **solid features** E.g.: cubes & cylinders or
 - **hollow features** E.g.: cuts and holes.
- Since each extruded feature depends on the base sketch, it can be **modified** by simply **editing** the base sketch.

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Going from Sketch to Model

After completing a sketch, right click to bring up menu –select **‘Finish Sketch’**

Go to Part Features – **‘Extrude’**

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Creating Extrusions

Part Features Menu

Open 2D L-shaped Sketch – Choose **‘Extrude’** to create a 3-D object

Extrude Dialog Box

The dialog box allows the designer to choose the **depth** and **direction** of extrusion.

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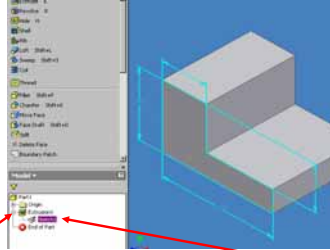
Identifying Extrusions

The L-block Extrusion is automatically named **‘Extrusion1’**

The **Hole** is another type of extrusion automatically named **‘Extrusion2’**

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Extrusions Are Based On Sketches



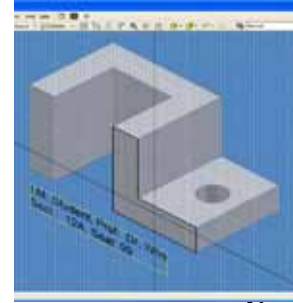
Each 'Feature' depends on a 'Sketch'
Click on '+' sign to show sketches

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Adding Your Name and Information

Do **NOT** create a new sketch plane on an already existing sketch plane.

Instead, return to the previously made L-shaped sketch plane and **'Edit'**.



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Today's Assignment:

- Start from the L-shaped Sketch of HW1.
- Make a 3D model of the "Adjuster" from Chapter 2.
- Refer to Inventor® R10, Chapter 2 pg 2-12 to 2-28.
- Add text information , save and print the model.

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