EGR 110
Engineering Graphics
File: N110IL2

Inventor Lecture #2

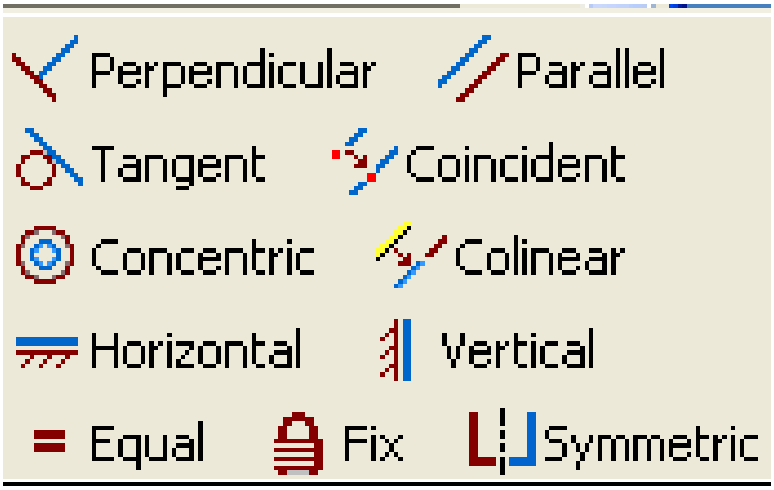
Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2009 by Randy Shih:
Chapter 2

Lecture Outline:

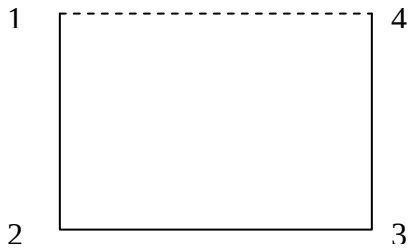
Geometric Constraints

Inventor will allow you to easily apply constraints, such as making features perpendicular, parallel, tangent, etc. Many of the features will be applied automatically, but can be turned off or changed as desired.
Inventor has 11 geometric constraints that you can apply to a sketch:
The icons for constraints are on the 2D Sketch Panel and are also shown below.

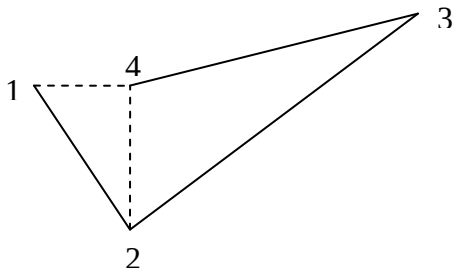


Perpendicular – The first line will stay in position and the second line will rotate until it forms a 90^{o} angle with the first line.
Parallel – The first line stays in position and the second line will move to become parallel to the first.
Tangent – An arc or circle and a line will become tangent to another arc or circle.
Coincident – A gap between two endpoints of arcs and/or lines will be closed.
Concentric – Arcs and/or circles will share the same center point.
Collinear – Two selected lines will line up along a single line. If the first line moves, so will the second. The two lines do not have to be touching.
Horizontal – Lines are positioned parallel to the x-axis. Can also be used to specify that center points of circles to share the same horizontal axis.
Vertical – Lines are positioned parallel to the y-axis. Can also be used to specify that center points of circles to share the same vertical axis.
Equal – Used to specify that two lines have the same length, two circles have the same radius or diameter. If one of the objects changes, so will the other.
Fix – Used to fix an endpoint or any other point to the permanent (world) coordinate system. This constraint overrides all other constraints.
Symmetric – Selected geometry will be symmetric about another line, centerline, or edge.

Inferred points – dashed lines are used to indicate a vertical or horizontal position related to another feature.



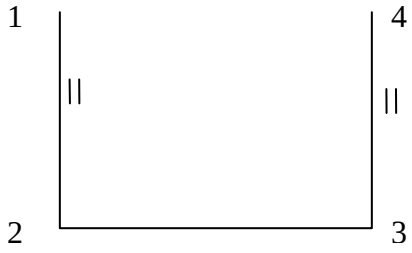
As line 3-4 is drawn, the dotted line indicates that point 4 will share the same y-coordinate as point 1. If the dotted line does not appear, briefly move the cursor over line 1-2.



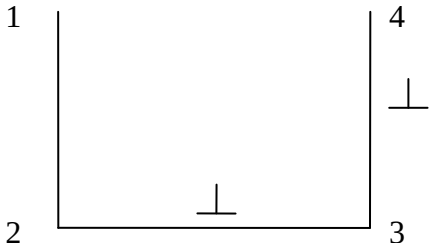
As line 3-4 is drawn, the dotted lines indicate that point 4 will share the same x-coordinate as point 2 and the same y-coordinate as point 1.

Automatic constraints & removing constraints – Some constraints will appear automatically as features are drawn. ***Hold down the CTRL key to avoid applying the constraint***.

Scrubbing (changing a constraint) – If one constraint appears and you want another constraint instead, try moving the cursor over the desired feature to which the new feature should be constrained.



As line 3-4 is drawn above, the Inventor seems to want to make lines 1-2 and 3-4 parallel.



Moving the cursor briefly over line 2-3 will let Inventor know that you prefer to apply the constraint that lines 2-3 and 3-4 be perpendicular.

Fully constrained objects – When an object is fully constrained, it can no longer be moved. Inventor will change the color of the object when it is fully constrained. This will generally not occur unless a point on the object has been fixed, since the object can otherwise be moved to a new location. Later in the course we will often seek to fully constrain sketches. Note that Inventor will generally not allow you to over constrain an object or to apply duplicate constraints.

Snaps – Another way to place features with a coincident constraint is to use ***snaps***. There are three types of snaps:

- Midpoint
- Center
- Intersection

To apply a snap, right-click in the graphics window and selected the desired snap.

Showing, adding, and deleting constraints

Pick the Show Constraints tool from the 2D Sketch Panel and pick a given feature to show its constraints.

Right-click on the Graphics window and select Show All Constraints to show constraints for all features. Similarly, right-click again and select Hide All Constraints.

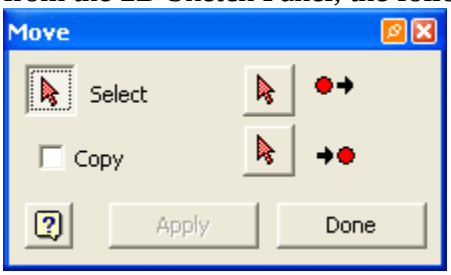
To add a constraint, select the desired constraint from the 2D sketch panel and then pick the features that it should be applied to. For example, pick the Perpendicular and then pick two lines that you wish to make perpendicular.

To delete a constraint, show the constraints, select the constraint with the mouse, and press the Delete key.

- Try deleting a constraint
- Try adding a constraint
- Move over a constraint (such as parallel or perpendicular) and the object that it is constrained to should be highlighted.
- Try dragging a feature that will not move. Remove a constraint that will allow you to drag it. Reconstrain the object.
- Try drawing two lines that are not perpendicular and then add a perpendicular constraint.

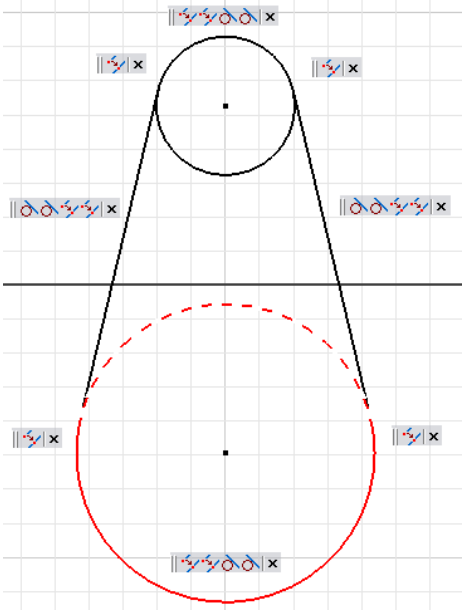
Editing Tools

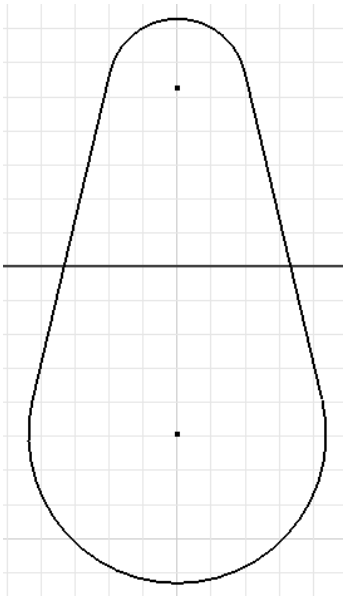
Move (Copy) – this tool can be used to move or copy selected features. When Move is selected from the 2D Sketch Panel, the following box appears:



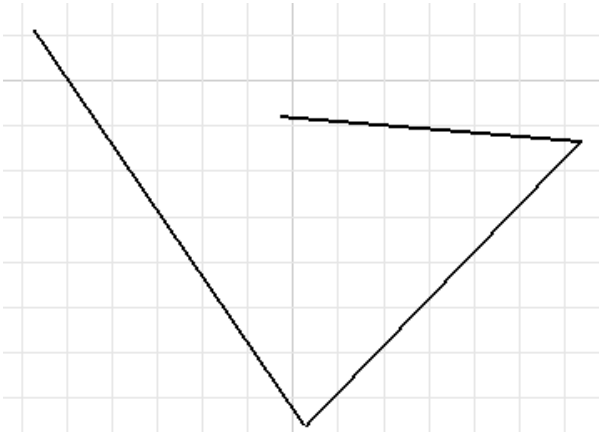
1) Check the Copy box if you wish to copy a feature instead of simply moving it.
2) Use the Select button to select the feature(s) to be moved or copied.
3) Select the top right arrow in the box for the Move (Copy) From Point.
4) Select the lower right arrow in the box for the Move (Copy) To Point

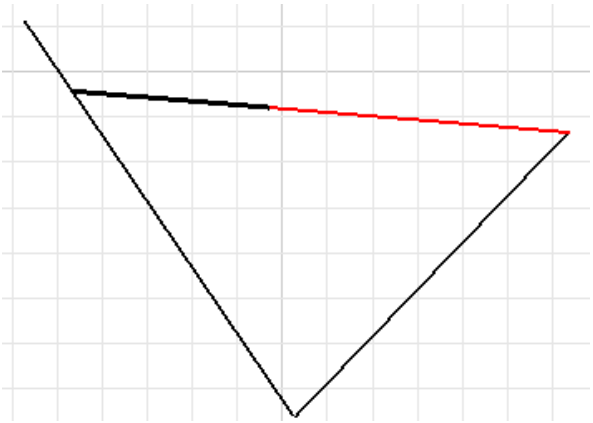
Trim – Select Trim from the 2D Sketch Panel and move the cursor over the desired feature to be trimmed. Options of parts of the feature to be trimmed will become dotted lines as you move the cursor over them.
Example: Sketch the object shown below. Add tangent constraints for the ends of each lines with the circles (4 constraints total). Trim the inside portion of the circles.





Extend – The command works similar to Trim for extending a feature (such as a line) until it intersects with another feature.
Example: Select Extend and move the cursor over the top line. It changes color and shows where the possible extension will go. Left click the mouse to accept the extension.





Sketch Planes

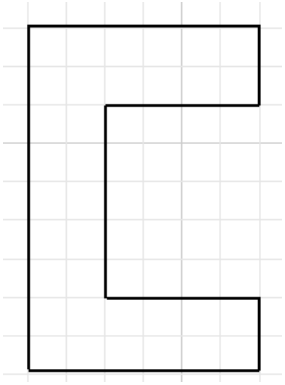
In earlier examples, a single default sketch plane was used to create a 2D profile that was then extruded to form a solid. In a similar manner, additional sketch planes can be added. These sketch planes can be located on:

- Planar faces of a part
- Work planes attached to part geometry
- XY, XZ, or YZ planes that are part of the World Coordinate System

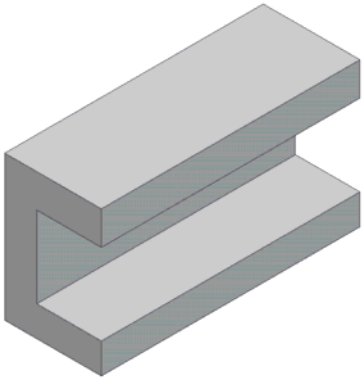
We will use the first option today (sketch planes on planar faces of a part).

Adding Multiple Extruded Features (Example)

- Open a new part file
- Create a sketch of some closed feature using lines (Sketch1)
- Extrude the sketch to form a solid and switch to an Isometric View (Extrusion1)



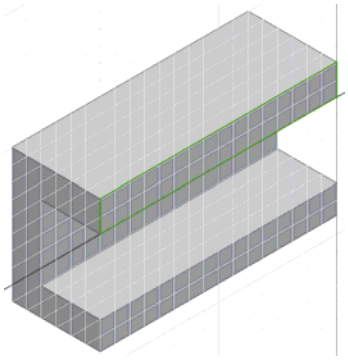
Sketch1



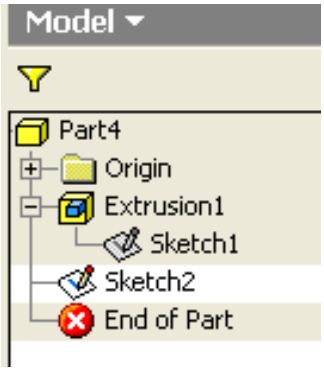
Extrusion1

Now add another extruded feature projecting from one of the faces of Extrusion1 as follows:

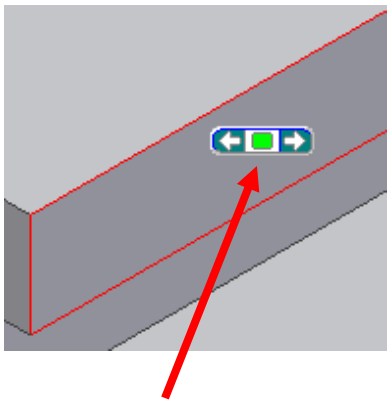
- Select the SKETCH icon on the Standard Toolbar. Note that the message bar at the bottom of the screen shows "Select plane to create sketch or an existing sketch to edit".
- Select the plane where the extruded feature is to be added (a sketch plane grid should now appear on the selected place)
- Sometimes it is difficult to select the desired feature. If multiple features can be selected, Inventor will show the "Feature Selection Tool". Use the arrows to cycle through available features and click on the center button when the desired feature is highlighted.



Sketch 2 Grid

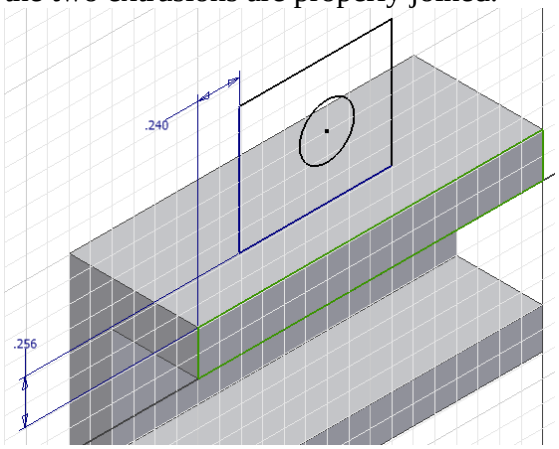


Browser

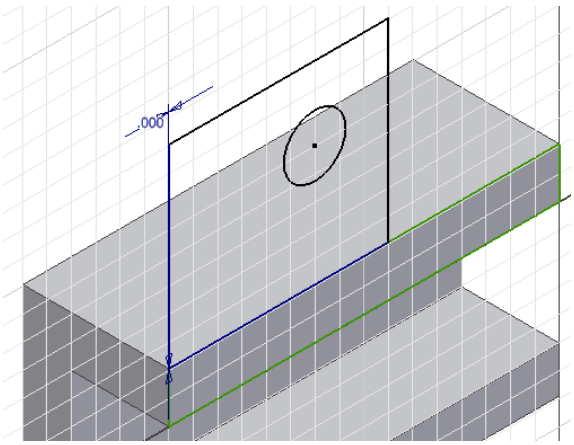


Feature Selection Tool

- Draw the 2D profile for the new feature to be extruded. Leave some space between the profile and the part and then add dimensions specifying the distance to be zero. This will insure that the two extrusions are properly joined.

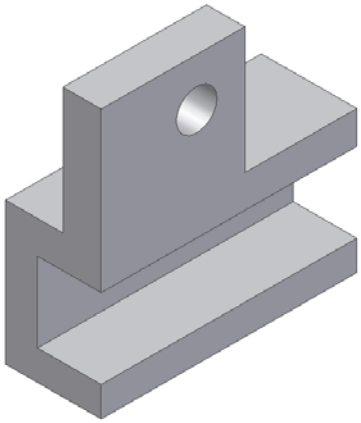


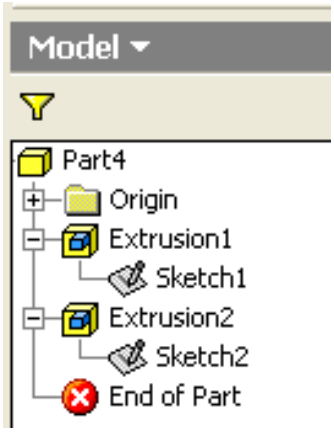
Sketch2 with dimensions for new feature



Dimensions set to zero to properly join the features

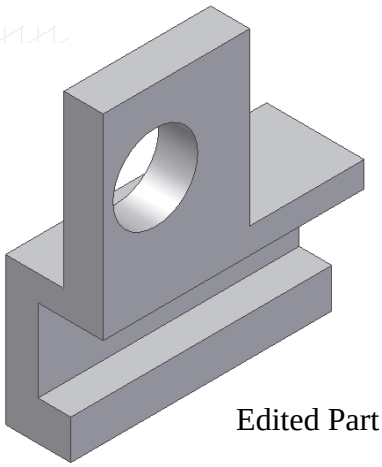
- Finish Sketch2 and Extrude it (try the extrusion direction icons to see the effect). Note the contents of the browser now.


Extrusion2 added



Browser shows all extrusions and related sketches

- Now try editing various features on the part.
  - Pick Sketch1 in the browser. Use LookAt to properly orient it. Add some dimensions and change their values. Right-click and pick Finish Sketch when done. Go back to the isometric view.
  - Repeat with Sketch2.

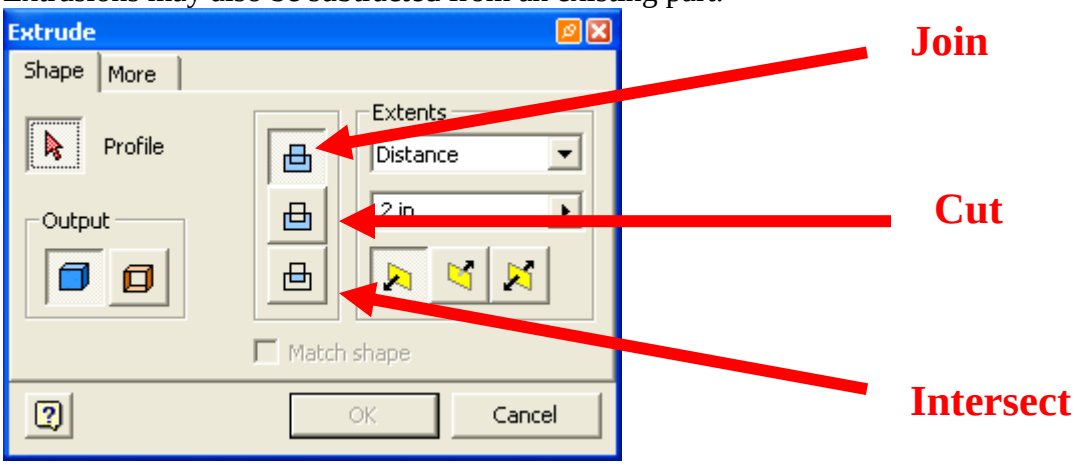

Edited Part

Other types of Extruded Features

So far we have seen how to add (or join) multiple extruded features. Extrusion options include:

- Join – used to add or union extruded features with an existing solid
- Cut – used to cut or subtract a new extruded feature from an existing solid
- Intersect – used to find the intersection a new extruded feature with an existing solid

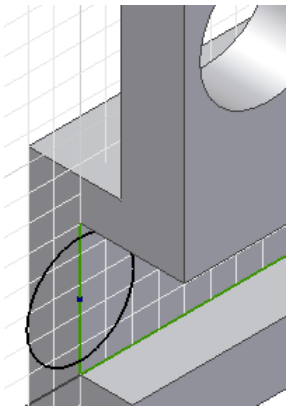
Extrusions may also be subtracted from an existing part.



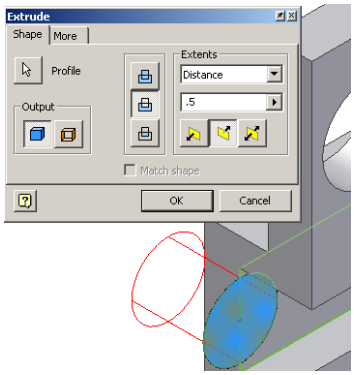
Example:

In the last example, or in a new example, try cutting a feature from an existing solid.

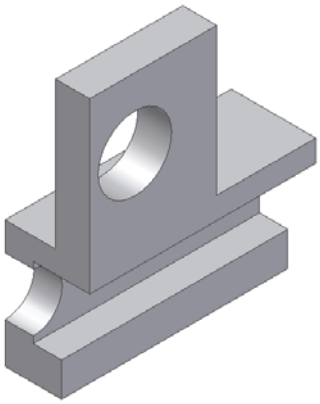
- Add Sketch3 and draw a circle
- Form Extrusion3 and cut it from the existing solid



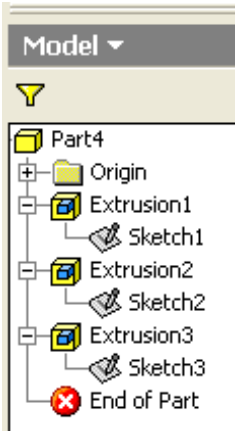
Sketch 3



Extrusion 3 using the CUT option



Final Part formed with 3 extrusions



Browser