

Inventor Lecture #1

Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2009 by Randy Shih:
Chapters 1-2

Lecture Outline:

Getting Started

- Why use Inventor instead of AutoCAD?
- Computer requirements for Inventor
- Student software (180-day license)
- Program stability – save often and start new files often
- What is meant by *parametric modeling*?

The textbook lists the following steps for the **parametric part modeling process**:

1. Create a rough two-dimensional sketch of the basic shape of the base feature of the design.
2. Apply/modify constraints and dimensions to the two-dimensional sketch.
3. Extrude, revolve, or sweep the parametric two-dimensional sketch to create the base solid feature of the design.
4. Add additional parametric features by identifying feature relations and complete the design.
5. Perform analyses on the computer model and refine the design as needed.
6. Create the desired drawing views to document the design.

Starting Inventor

Use the Inventor icon on the desktop to launch Inventor (or use Start- Programs – Autodesk – Inventor)

Use File – Open (or the New icon)

select the English tab

select Standard.ipt (or Standard (in).ipt)

File types in Inventor:

- ipt (Inventor part) – will be begin the course by making single parts
- iam (Inventor assembly) – later we will combine several parts into a single assembly
- idw (Inventor drawing) – parts and assemblies can be brought into drawing files where multiple views and additional features can be added

Screen Layout and Menus

Pull-down menus

Standard toolbar

Sketch tools (such as the 2D Sketch Panel) – discuss how this menu changes

Part browser

Message and status bar

Graphic window (background color can be changed using *Tools – Application Options – Colors – Presentation* (for a white background))

Creating Rough Sketches

If you have used other CAD packages, the approach used in parametric modeling packages may seem unusual. However, you will soon find it to be a great way to create solid models!

The text lists some **general guidelines for creating rough sketches in Autodesk Inventor**:

- Create a sketch that is proportional to the desired shape.
- Keep the sketches simple.
- Exaggerate the geometric features of the desired shape (example: 85 degree angle).
- Draw the geometry so that it does not overlap.
- The sketched geometric entities should form a closed region.

LINE command

- Note how cursor changes
- Geometric constraint symbols give visual clues for when lines are vertical, horizontal, parallel, perpendicular, etc.
- Hold down the Ctrl key to prevent a constraint from occurring.
- Use Esc or Right mouse click – Done to end the line (and other commands)

PAN and ZOOM

- Using standard toolbar icons
- Using main pulldown menus
- Using the wheel on the mouse

Dimensions

- Dimensions are one type of constraint used in Inventor.
- Features may be subjected to several types of constraints in Inventor. For example, a line may have the following constraints:
 - It may be vertical
 - It may be perpendicular to another line
 - It may have a certain length (specified by a dimension)
- Select GENERAL DIMENSION from the 2D Sketch Panel (you may need to scroll down)
 - Pick the feature to be dimensioned
 - Pick the location to place the dimension
- Modifying dimensions
 - Click on the text of a dimension and change its value
 - Notice that the location or size of the feature changed!

More Examples:

Save your drawing and create a new part file.

Experiment with different features, including CIRCLES, and with various dimensions.

Finishing a Sketch

When you have completed a sketch (or modifications to a sketch), right click in the graphics window and select FINISH SKETCH.

Creating a solid feature

Note that after selecting FINISH SKETCH the **2D Sketch Panel** changed to **Part Features**. One of the simplest ways to create a solid is to use **extrusion** (discuss the use of extrusion in manufacturing).

EXTRUDE command

- Select EXTRUDE from the Part Features menu or use the shortcut, E.
- Select the Profile icon from the Extrude popup window and pick the closed surface to be extruded.
- Specify the thickness (also try changing the direction and the effect can be clearly seen on the model)
- Select OK
- The sketch has now been extruded to form a solid model, but it is hard to tell that it is solid while looking at it from a top view.

Changing the View

- Select View – Isometric View from the pulldown menu (or right-click on the Graphic Window and pick Isometric View)
- Experiment with PAN and ZOOM
- Experiment with ROTATE
 - Rotating about the horizontal axis
 - Rotating about the vertical axis
 - Free rotate
 - Click within the circle to center a point
 - Press the spacebar for Common Views
- Experiment with LOOK AT on the Standard Toolbar

Display Modes – Shaded Solid, Hidden-Edge Display, Wireframe (pulldown menu)

Orthographic versus Perspective Modes – (pulldown menu)

Switching between a solid model and a sketch

After creating a solid model, you may discover that you need to go back to the sketch to edit it or to add other features.

- Click on the + next to Extrusion 1 in the Parts Browser to show the related sketch
- Right-click on Sketch in the Parts Browser and select Edit Sketch

If you had been looking at an isometric view of the solid, you will notice that sketch is also shown as an isometric. You can edit the sketch from this view, but if you would prefer a standard true-shape view of the sketch, try the following:

- Right click on the sketch and select Finish Sketch (you should now see the solid again).
- Pick the **Look At** icon on the Standard Toolbar. Select the face that you would like to view (the face used to create the sketch).
- Right-click on Sketch in the Parts Browser and select Edit Sketch

More examples – try several examples if time allows

Printing

For the first couple of assignments, we will print sketches and models directly (using File – Print). This is not the preferred method of printing and may result in printouts that are not as clear as those we will create later by printing from a drawing file.

Annotating Sketches

In general, do not try to annotate sketches. For example, do not worry about centerlines, arrow sizes on dimensions, notes, etc. These types of features can be easily added later in drawing files. An important point is that *sketches are used to express features and constraints that are necessary to create the desired solid model*. Sketches are not used for annotation.