CAD Tutorial - 1

Purpose: Introduce you to solid modeling and basic solid modeling functions.

Importance: Solid Modeling is a way for scientists and engineers to geometrically model their work, check for fit and function. The 3D geometry created during Solid Modeling can be used to make blue prints, perform computer analysis (FEA or CFD), and create visualizations.

What to hand in: N/A

Goals: After this activity, you will be able to:

- Learn why Solid Modeling is important
- Learn the process for creating good Solid Models
- Learn the basic types of solid modeling 3D geometry creations commands
- Learn how to customize the Solid Modeling files for handing in

Software: Though you do not need to do this for lab, you may load SolidWorks® or Pro/ENGINEER® on your home/dorm computer. This might save trips to campus late at night.

SECTION 1: Introduction to Solid Modeling

The design and modeling processes are interdependent! It is difficult and irresponsible to design processes, machines, or other products without modeling how they will look and perform. Solid Modeling is one of the major steps in making machines and machine components.

1. Customer Modeling
2. Requirement Modeling
3. Analytic Modeling

4. Solid Modeling (Geometric Modeling)
5. Functional Modeling

Parametric Solid Modeling (PSM) is a useful tool, which will help you geometrically model your design(s). Solid modeling is not an easy skill to learn. You must know:

- How to run the software:
- How to use the software to get the results that you want (design intent)

Eventually, as you learn how to build models with the correct design intent, you will find that solid modeling will:

- Allow you to change your design with little effort
- Help illustrate your designs to others
- Make sure parts fit together
- Check for proper function
- Check for interference
- Make drawings / blue prints

The introduction of PSM does not mean the demise of sketching!!! Sketching is one of the most valuable skills a designer can have to quickly convey ideas and layout a design. As a matter of fact, it is often a GOOD idea to sketch out your designs before you proceed with PSM so that you can start to recognize the major shapes that make up your part. This is important as robust solid models are made of the fewest and least likely to change features one need. You will come to learn that the process of rough sketching is very similar to the process of solid modeling.

SECTION 2: PSM Process

Like the design process, PSM is best done when following a process. The general steps are listed below, you will learn them as you go through the tutorials.

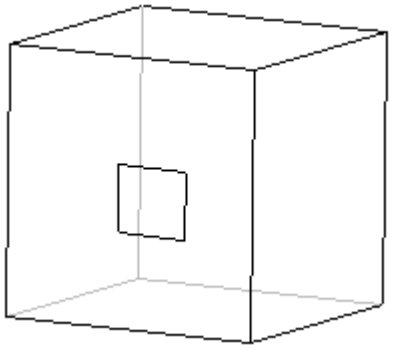
1. Determine what you want to do/make
2. Develop concepts for getting the job done (i.e. what are the best shapes to model this?)
3. Determine your design intent
4. Draw/input any supporting geometry or sketches needed
5. Draw a 2-D sketch
6. Extrude, Revolve, Sweep, or Loft the 2-D sketch to make a 3-D object
7. Combine these objects into assemblies
8. Make drawing of individual parts and assemblies with dimensions & tolerances

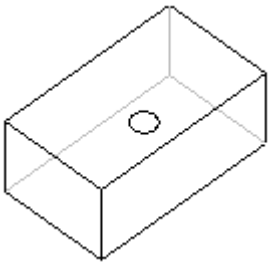
9. Make files for use with:

    - Computer aided manufacturing (CAM) software
    - FEA (finite element analysis) software
    - Dynamic modeling software
    - Photographic rendering software
    - Others

SECTION 3: Basic Functions in Parametric Solid Modeling

Before discussing PSM in more detail, the reader should be familiar with the basic operations which can be used to make a solid model of a part/product. Note that when applicable, the illustrations below start out with a 2D sketch, then they are transformed into 3-D geometry. The basic process may differ somewhat, depending on the CAD program you use.

Function To view, go to the Animations section	**What it's good for**	**Process**
 **Solid Extrusion**	Solid Extrusions are often used to: • Make the base part of a model • Adding bosses/features to a model	1. Pick/make a sketch plane 2. Draw & dimension a 2-D sketch 3. Issue "extrude boss" command 4. Define extrusion type (blind, midplane, etc....) 5. Enter extrusion depth if necessary
 **Cut/Slot Extrusion**	Subtracting portion of a model to form a feature like: • Holes • Keyways • Cavities	1. Pick/make a sketch plane 2. Draw & dimension a 2D sketch 3. Issue "extrude cut/slot" command 4. Define extrusion type 5. Enter extrusion depth if necessary

 **Revolved Extrusion**	Adding features to a model which can best be described in R-θ coordinates. These features include: • Base of a model • External pipes/ducts • Torous shaped features • Annular features/ribs	1. Pick/make a sketch plane 2. Draw & dimension a 2D sketch 3. Draw a centerline around which to revolve 4. Issue "revolve cut/slot" command 5. Enter revolution direction & depth (θ) if necessary
 **Revolved Cut**	Subtracting a portion of a model which can best be described in R-θ coordinates. These features include: • Internal pipes/ducts • External irregular shaped grooves	1. Pick/make a sketch plane 2. Draw & dimension a 2D sketch 3. Draw a centerline around which to revolve 4. Issue "revolve cut/slot" command 5. Enter revolution direction & depth (θ) if necessary
 **Loft [blend in Pro-E]**	Adding/subtracting features to a model where two ends of the feature are described by different x-sections (sketches) and the x-sections must transform into each other. • Base of a model • External features	1. Pick/make a sketch plane 2. Draw & dimension 1st sketch 3. Draw & dimension 2nd sketch 4. Issue "loft command"

 **Sweep**	Adding or subtracting parts to/from a model which can best be described by "sweeping" a x-section along a pre-described path: • Base of a model • External pipes/ducts • Springs • Torous shaped features • Annular features	1. Pick/make a sketch plane 2. Draw & dimension sweep path 3. Draw & dimension sweep x-section 4. Issue "sweep" command
 **Shell**	Making thin parts such as: • Cups • Bowls • Thin casings	1. Issue "shell" command 2. Pick a cut plane/face/ surface 3. Pick shell direction/ properties 4. Pick shell thickness