UGS NX 6

Course Webpage: Unigraphics NX Tutorials and Related Documents

- NX CAST 6 - A Comprehensive NX CAST 6 Tutorial
  - o Available on the PCs with UGS NX 6
  - o CD of NX CAST 6 Available
- NX5 for Engineering Design - A Complete Tutorial
  (M. C. Leu and A. Joshi, Missouri Univ of Sci. & Tech.)
  - o Introduction
  - o Getting Started
  - o Form Features
  - o Feature Operations
  - o Drafting
  - o Sketching
  - o Freeform Feature
  - o Assembly Modeling
  - o Manufacturing
  - o Finite Element Analysis
- Parametric Modeling with UGS NX 6 Book (by Randy H. Shih, from SDC)
  - o Table of Contents
  - o Sample Chapter 2 - Parametric Modeling Fundamentals
- NX6 Modeling Tutorial by John K. Layer (2008-8-26)
- Explanation of Buttons Used for Sketching in Unigraphics
- UGS NX Drafting Tutorial (Michigan Tech Univ)
- NX Advanced FEM (Version NX 5) (Siemens)
- Team center Visualization Concept Desktop Hands-on Tutorial (UGS)
- UGS Open and API Programming
- NX5 Review Article: Jeffrey Rowe, "CAD/CAM/CAE solution from Siemens PLM Software is primed for production," *Cadalyst*, Nov 1, 2007
- GM X Challenge UG Discussion (R. Salmon, GM Vehicle Modeling, Available to EcoCAR Members Controlled Page)
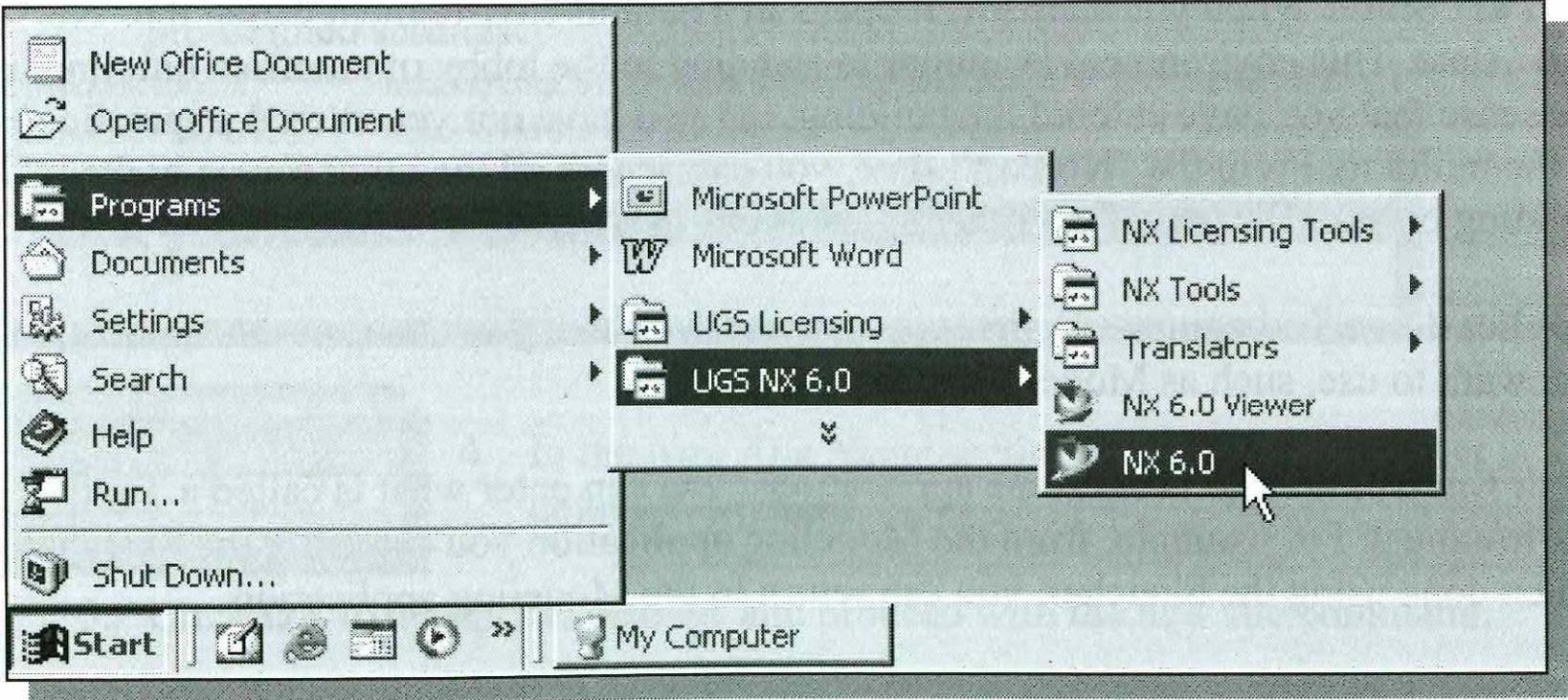
Parametric Modeling with UGS NX 6
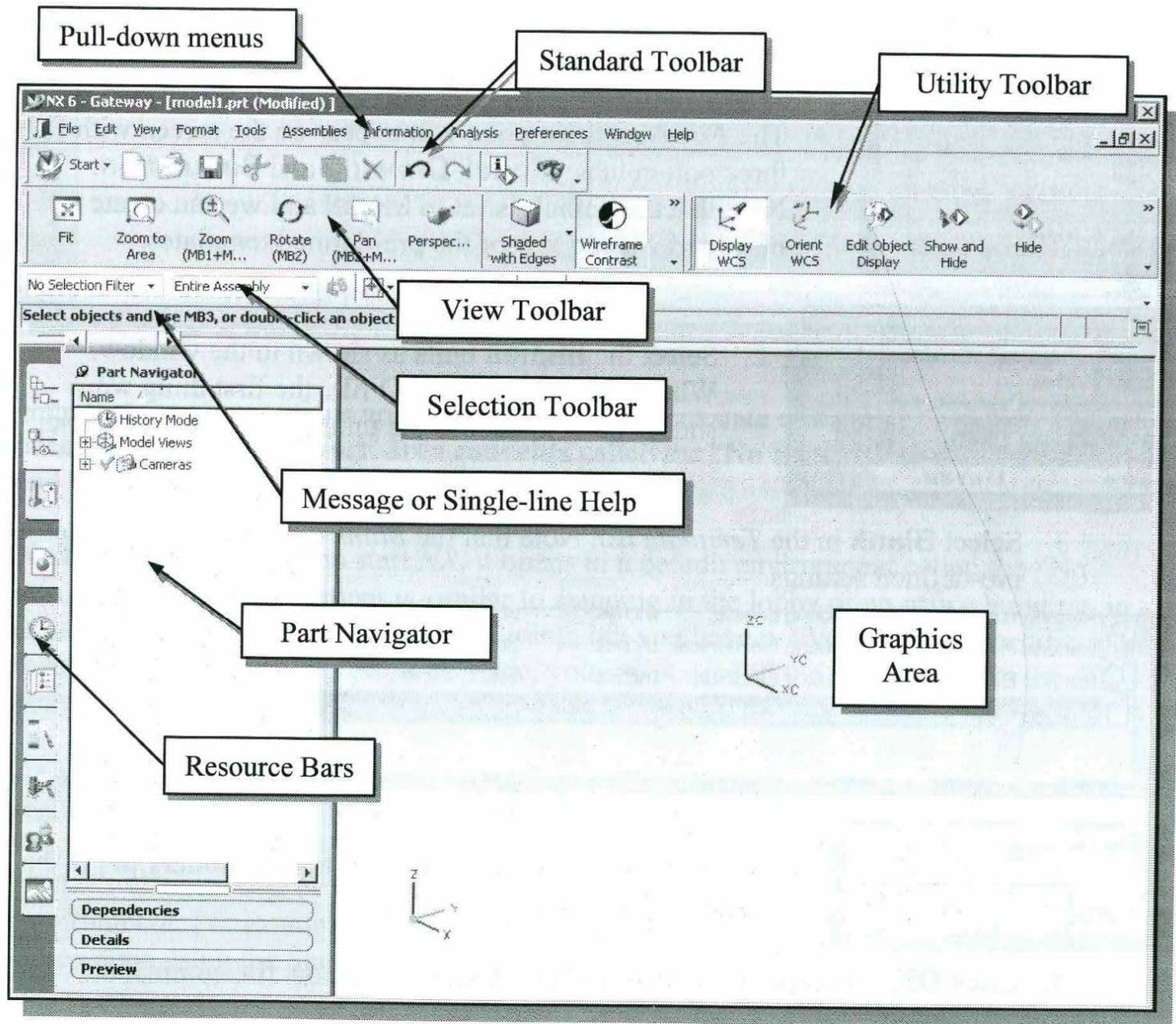
Randy H. Shih
Oregon Institute of Technology

SDC
PUBLICATIONS

Schroff Development Corporation
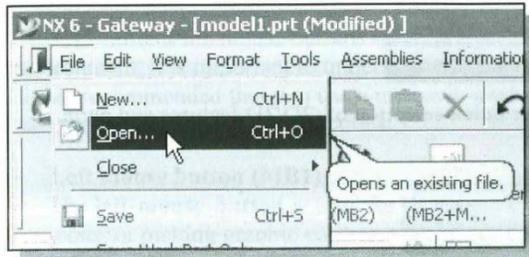www.schroff.com

Better Textbooks. Lower Prices.

Start UGS NX6

UGS NX6 Screen

Pull-down menus

Standard Toolbar

Utility Toolbar

NX 6 - Gateway - [model1.prt (Modified)]

File   Edit   View   Format   Tools   Assemblies   Information   Analysis   Preferences   Window   Help

Start

Fit | Zoom to Area | Zoom (MB1+M... | Rotate (MB2) | Pan (MB3+M... | Perspec... | Shaded with Edges | Wireframe Contrast | Display WCS | Orient WCS | Edit Object Display | Show and Hide | Hide

No Selection Filter | Entire Assembly

Select objects and use MB3, or double-click an object

View Toolbar

Selection Toolbar

Part Navigator

Name

History Mode
Model Views
Cameras

Message or Single-line Help

Part Navigator

Graphics Area

Resource Bars

Dependencies

Details

Preview

- **Pull-down Menus**

The *pull-down* menus at the top of the main window contain operation that you can use for all modes of the system.

- **Standard Toolbar**

The *Standard* toolbar at the top of the screen window allows us quick access to frequently used commands. For example, the file-related commands, such as Switching Applications, New Part, and Save.
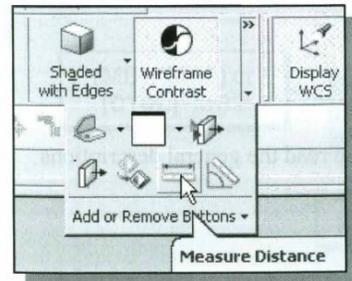
- **View Toolbar**
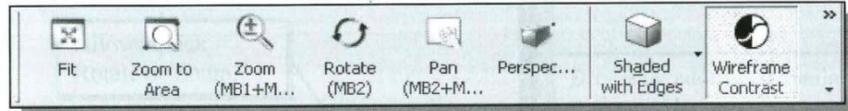
The *View* toolbar allows us quick access to frequently used view-related commands, such as Zoom, Rotate, Creating Section View and Shaded Solids.

- **Utility Toolbar**
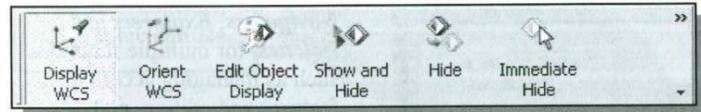
The *Utility* toolbar allows us quick access to *WCS manipulation*, such as WCS display, *Move WCS*, and control of the *Object display options*.

- **Additional Tools**

The double-arrow at the end of each toolbar can be used to access the additional tools that are available.

- **Selection Toolbar**
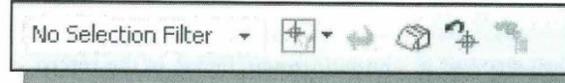
The *Selection* toolbar provides tools for quick selection of 2D/3D features and parts.
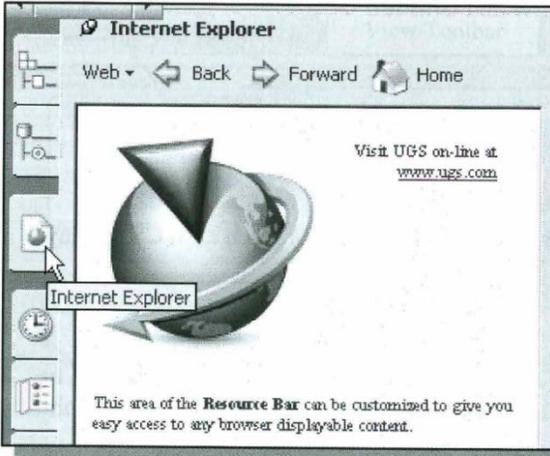
- **Message and Status Bar**
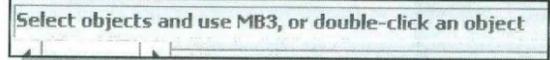
The *Message and Status Bar* area shows a single-line help when the cursor is on top of icon. This area also displays information pertinent to the active operation.

- **Part Navigator**
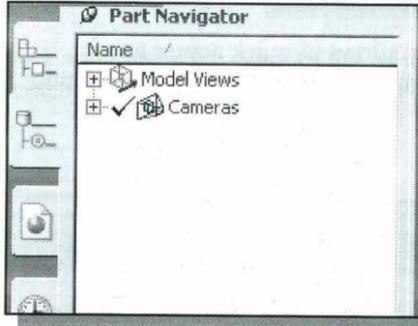
The *Part Navigator* area shows information regarding the current active model. This area i used by the *Resource Bars* options described below.
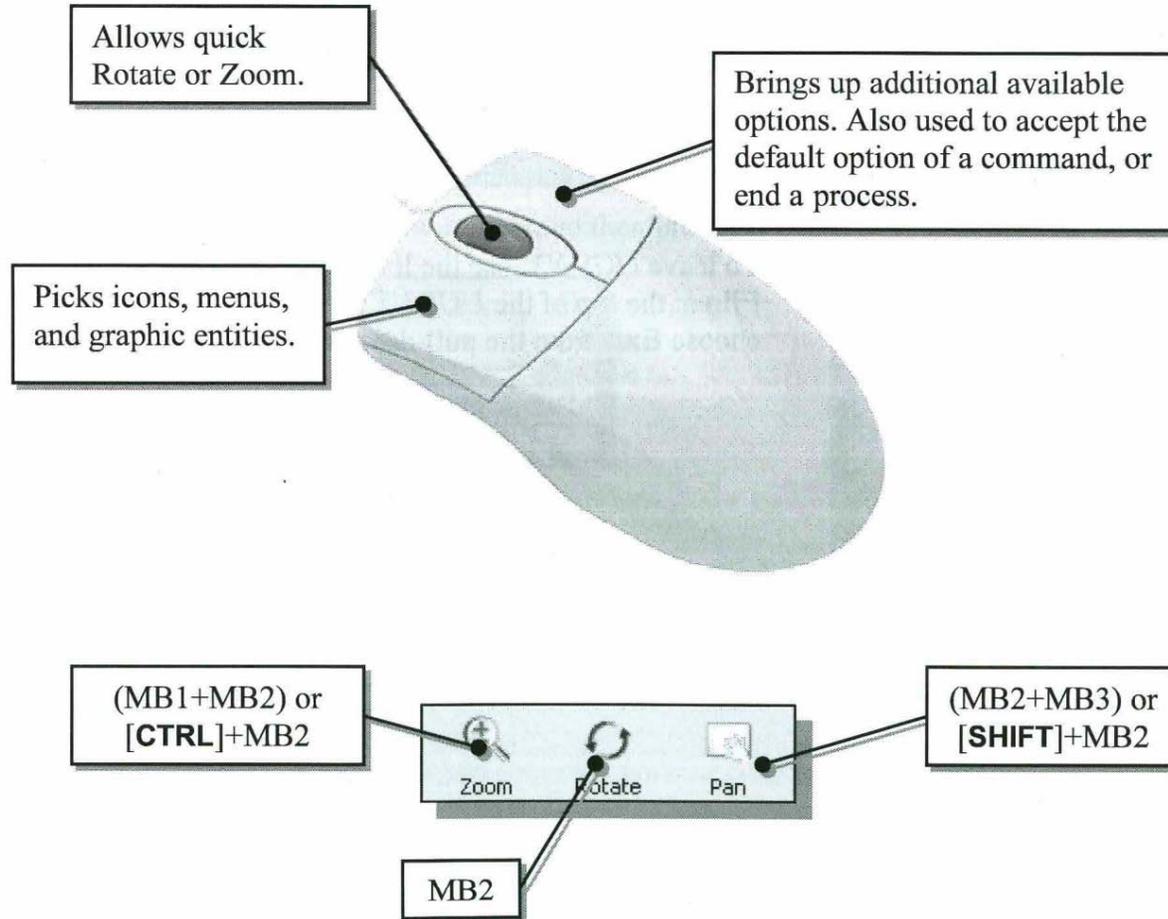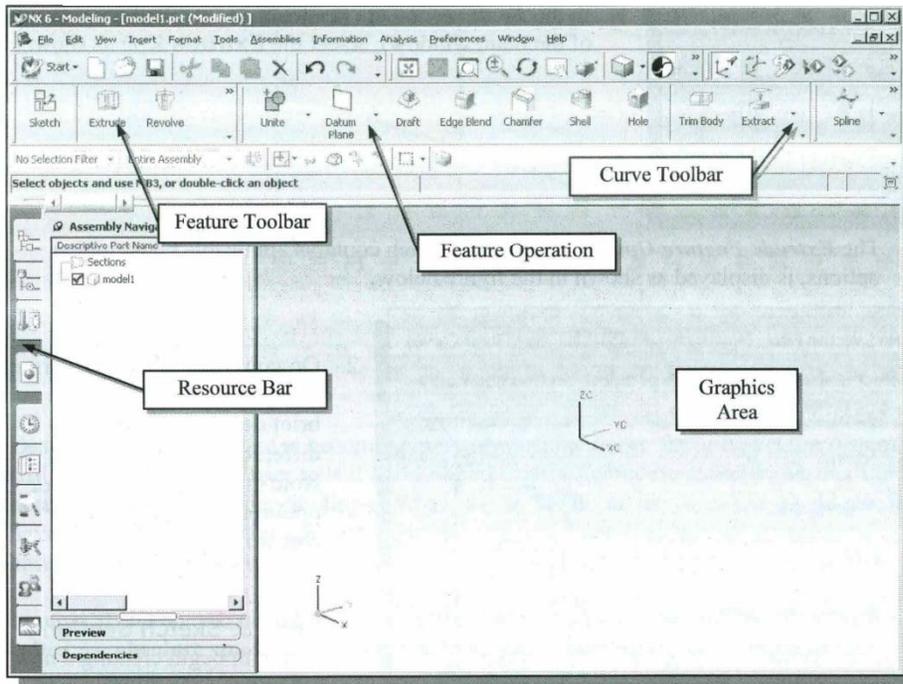
- **Resource Bars**

➢ The *Resource Bars* provide three groups of resources, *Navigators*, *Explorers* and *Palettes*, for multiple function such as managing access to features and editing, and providing alternate access to functions in the *Context* menu

Use of Mouse Buttons

Allows quick
Rotate or Zoom.

Brings up additional available
options. Also used to accept the
default option of a command, or
end a process.

Picks icons, menus,
and graphic entities.

(MB1+MB2) or
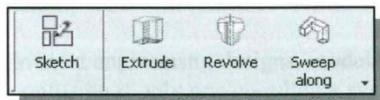[CTRL]+MB2

Zoom   Rotate   Pan
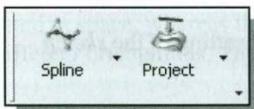
(MB2+MB3) or
[SHIFT]+MB2

MB2

Application Screen

Form Feature Toolbar

This toolbar contains tools that allow us to quickly create 2D sketches and form 3D features.
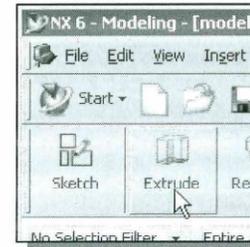
Feature Operation Toolbar

This toolbar contains tools that allow us to quickly create placed features.

Curve Toolbar

This toolbar contains tools that allow us to quickly access 2D curve options.

• For the *Adjuster* design, we will create an extruded solid as the first feature.

1. In the *Feature* toolbars (toolbars aligned to the right edge of the main window), select the **Extrude** icon as shown.

• The *Extrude Feature Options* dialog box, which contains applicable construction options, is displayed as shown in the figure below.
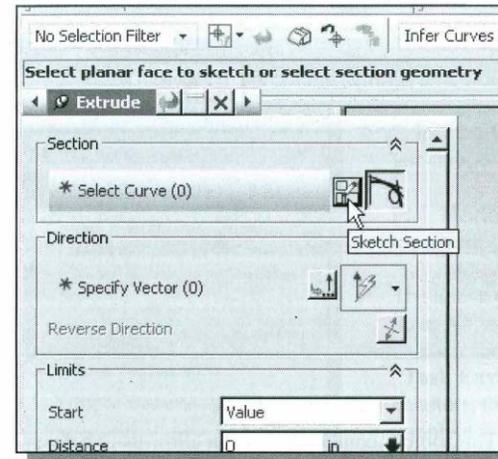
2. On your own, move the cursor over the icons and read the brief descriptions of the different options available. Note that the default *Extrude* option is set to **Select Section**.

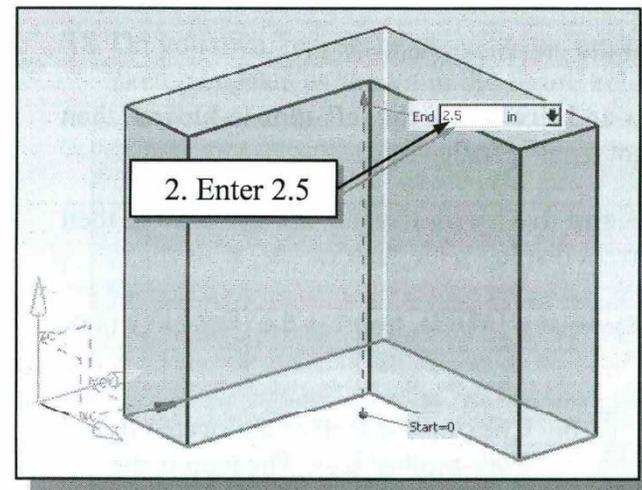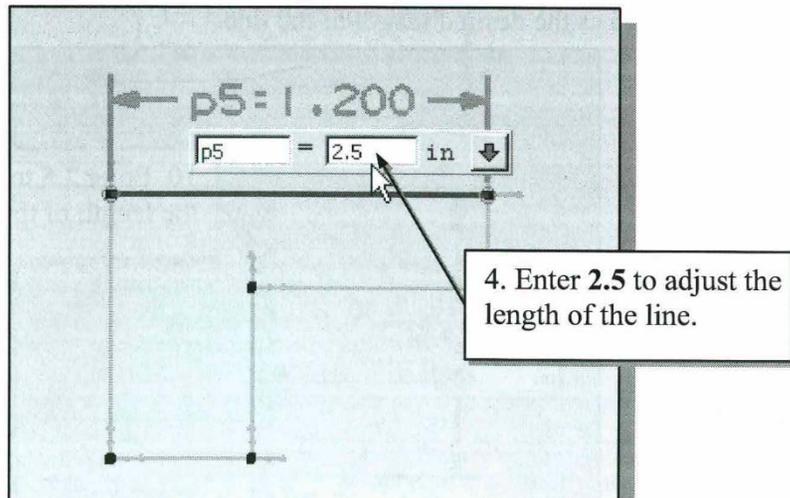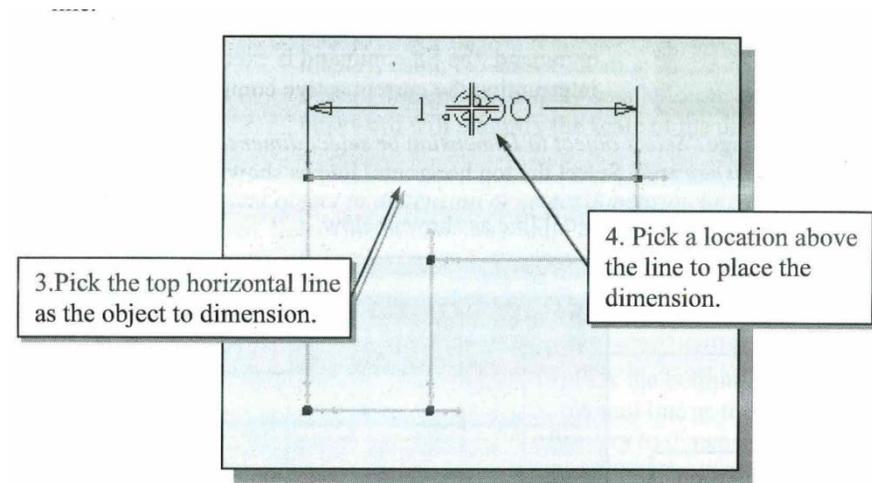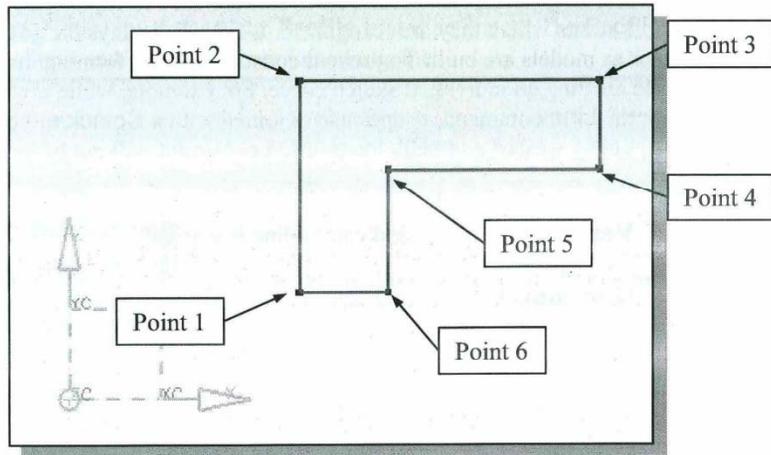3. Click the **Sketch Section** button to begin creating a new 2D sketch.
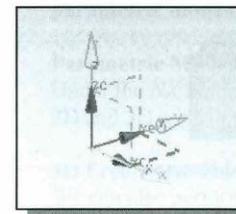
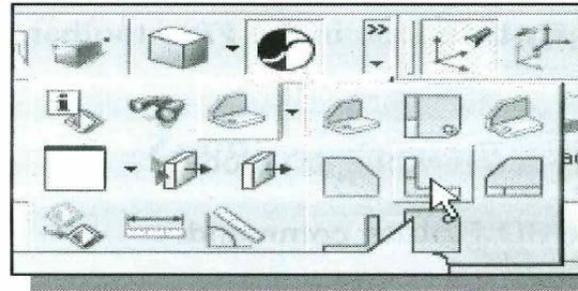❖ Note the default sketch plane is aligned to the *XC-YC* **plane** of the displayed *work coordinate system*.
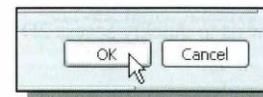
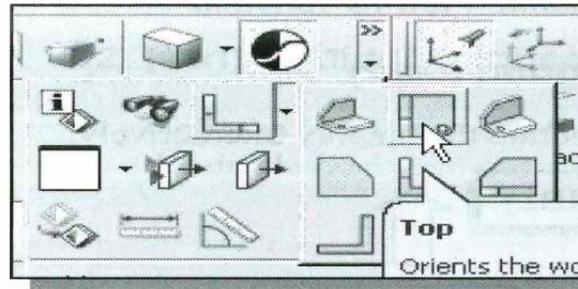4. Click **OK** to accept the default setting of the *sketch plane*.

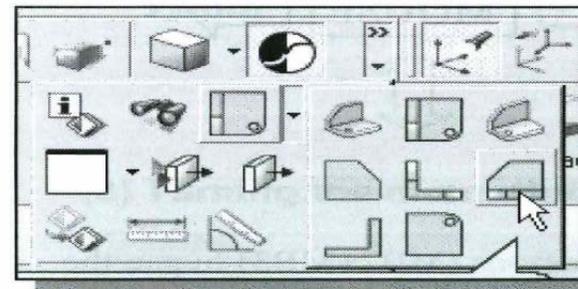# Sketching, Dimensioning and Modeling

Point 2

Point 3

Point 4

Point 5

Point 1

Point 6

1 0

3. Pick the top horizontal line as the object to dimension.

4. Pick a location above the line to place the dimension.

p5=1.200

p5 = 2.5 in

4. Enter **2.5** to adjust the length of the line.

End 2.5 in

2. Enter 2.5

Start=0

Different Display Orientations

1. Select **Fr**
   change th
   [Ctrl+Alt-
   command

2. Select **T**
   the displ
   [Ctrl+Al
   this com

3. Select **Ri**
   change th
   that the **[C**
   activate t
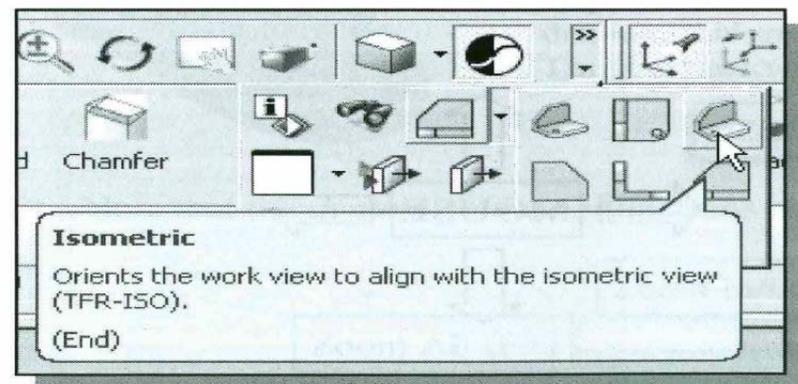
4.

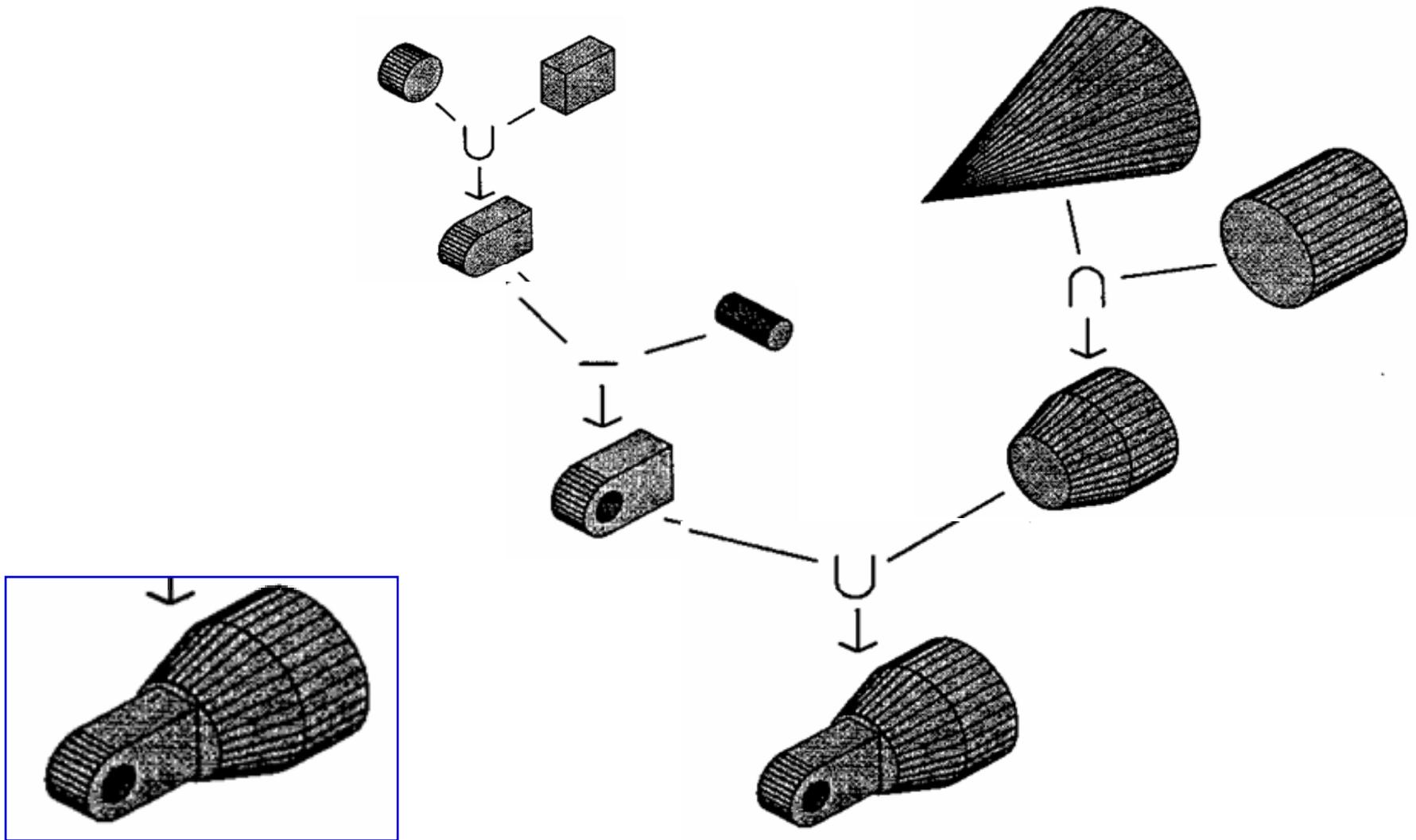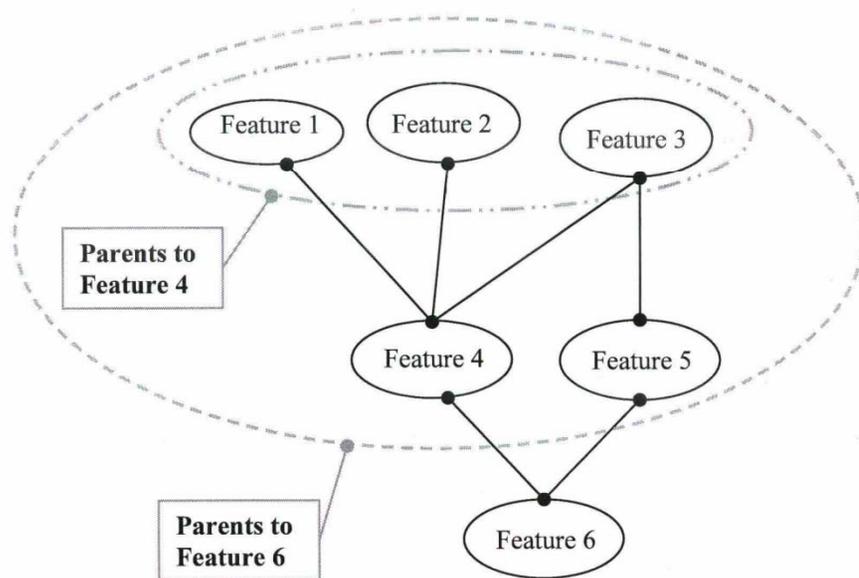**Isometric**

Orients the work view to align with the isometric view (TFR-ISO).

(End)

CSG Tree

CSG Modeling and Parent-Child Relations

Parents to Feature 4

Parents to Feature 6

Feature 1 Feature 2 Feature 3

Feature 4 Feature 5

Feature 6

Modeling Strategy - CSG Binary Tree
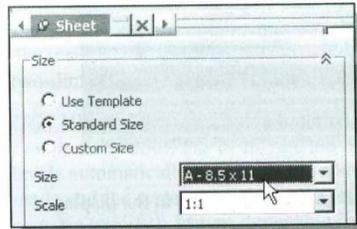
UNITE

SUBTRACT
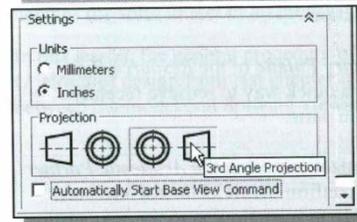
SUBTRACT

SUBTRACT
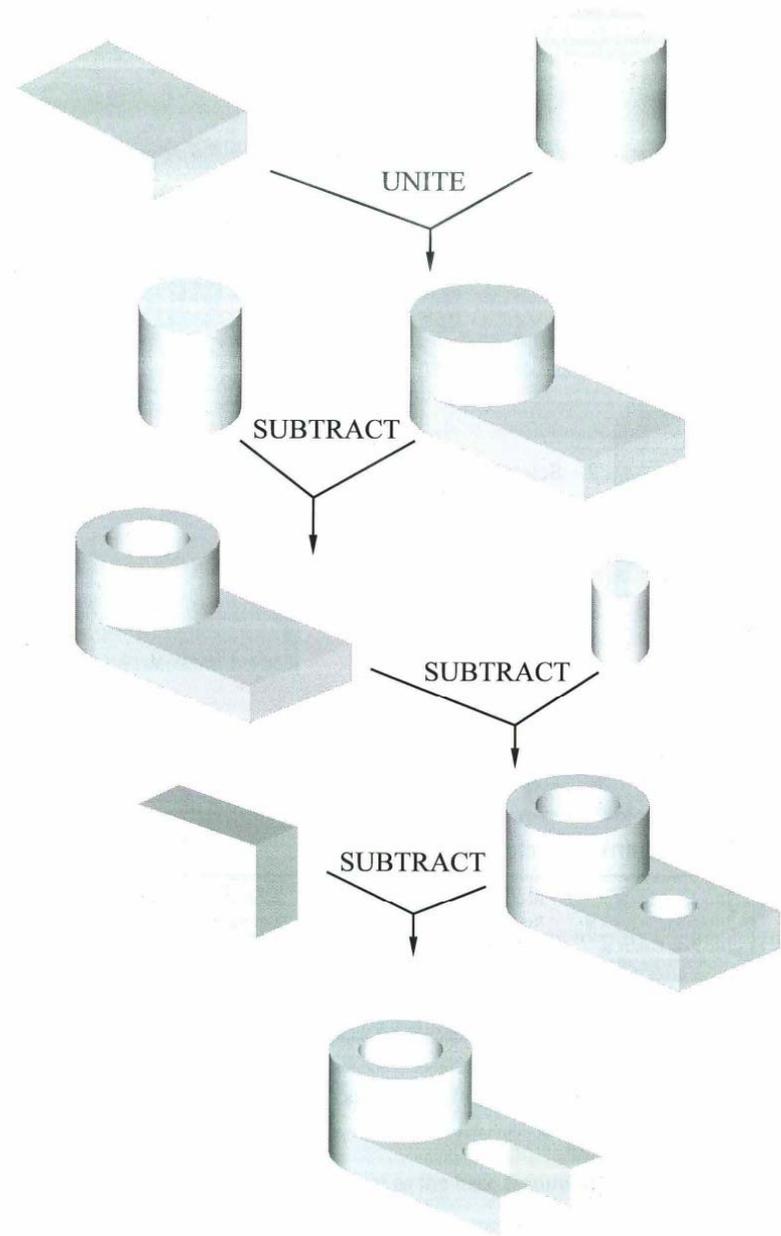
Drawing Generation

UGS NX Drafting Mode

1. Click on the **Start** icon in the
   to display the available option

2. Select **Drafting** from the opti

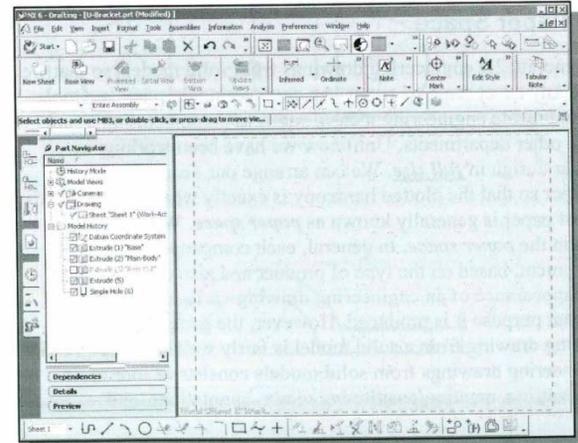❖ In the *Sheet* dialog box, UGS
   settings for the drawing sheet

3. Select **A-8.5x11** fro

4. Confirm the *Scale* is
   full scale.

➤ Note that **Sheet 1** is
   sheet name that is di
   sheet name area.

5. In the *Settings* area,
   set to **Inches** and th
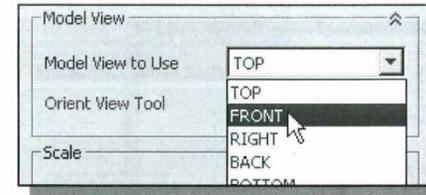   to **3rd Angle Proje**

6. **Uncheck** the Autom
   View Command o

Adding a Base View
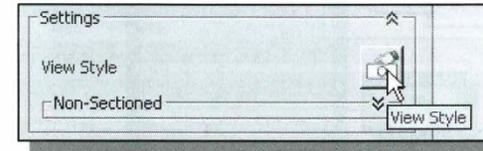
❖ In *UGS NX Drafting* mode, the first drawing vi
   base view is the primary view in a drawing; oth
   view. When creating a *base* view, UGS NX allo
   views to be shown. By default, UGS NX tr
   view of the solid model.

1. Click on the **Bas**
   toolbar to create a

2. In
   vie

4. In the *Vie*
   Lines ta

➤ The *View*
   settings f

5. Choose **D**
   hidden lin

Modeling – Extrusion and Revolving

Modeling Strategy

Sketch is fully constrained

p16=3.000
Rp20=0.250
p18=p16/2
p17=2.000
p19=p17/2
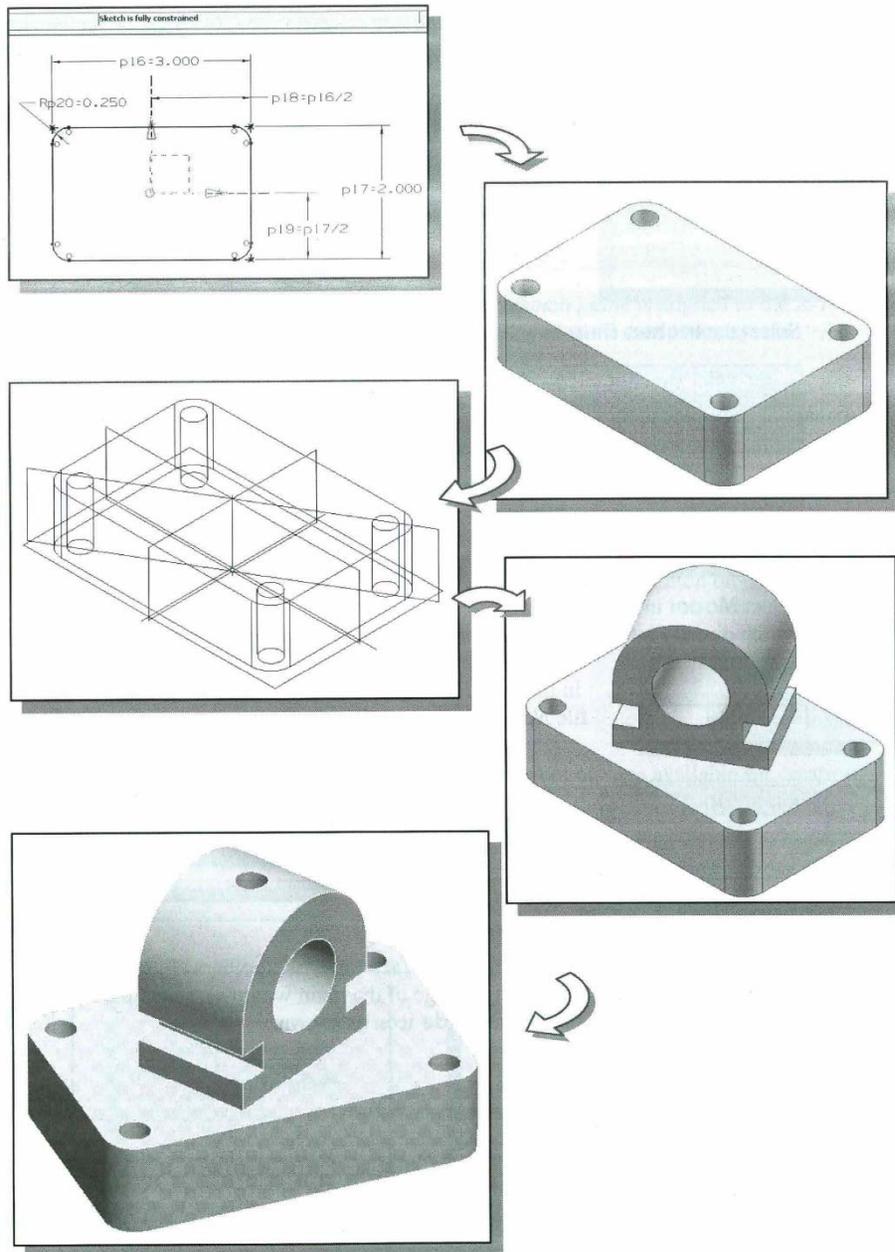
Modeling Strategy – A Revolved Design

YC
ZC   XC

Modeling Strategy


p4=0.000
R p7=2.000
p6=1.500
p5=0.000
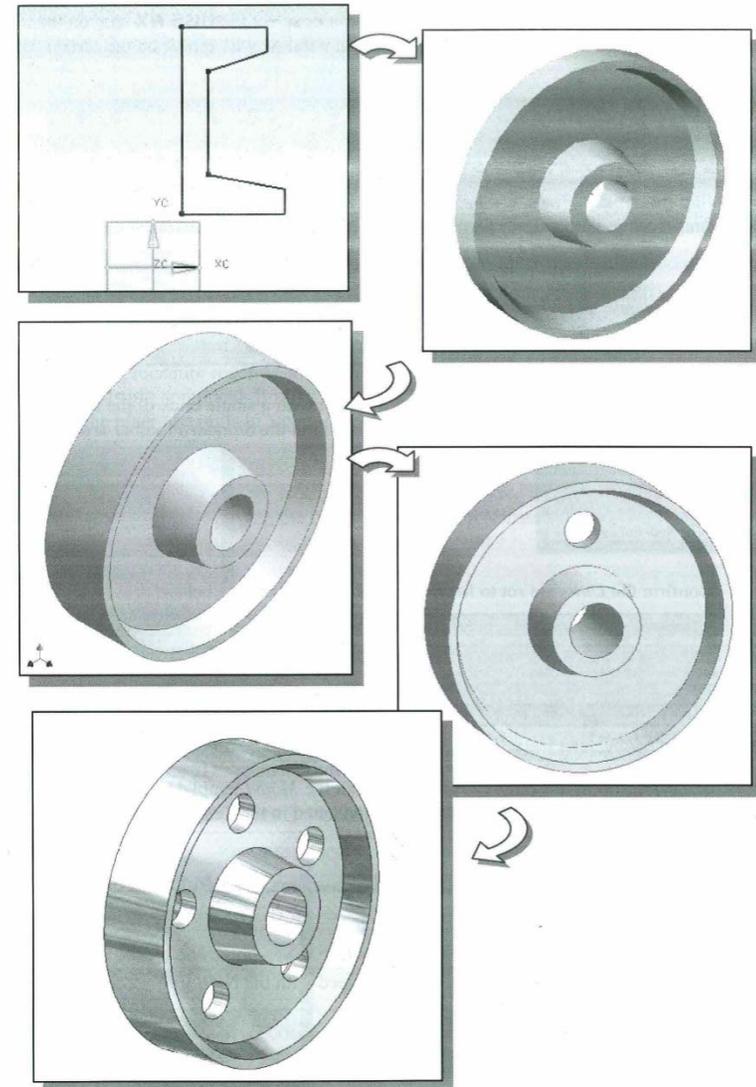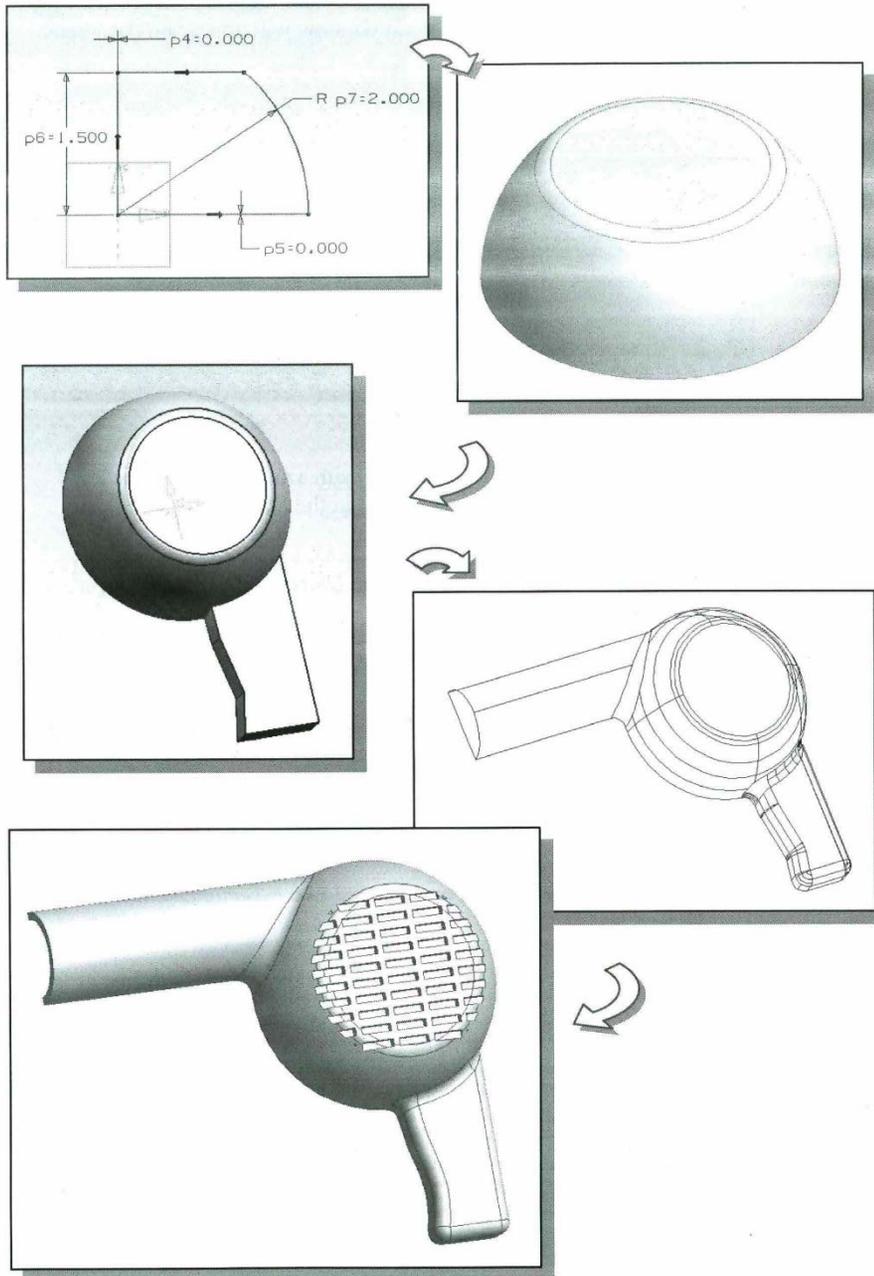
Advanced Modeling

Assembly Relations

➤ **Touch (Mate)** – Constraint positions components face-to-face, or adjacent to one another, with faces flush. Removes one degree of linear translation and two degrees of angular rotation between planar surfaces. Selected surfaces point in opposite directions. The Touch constraint positions selected faces normal to one another, with faces coincident.
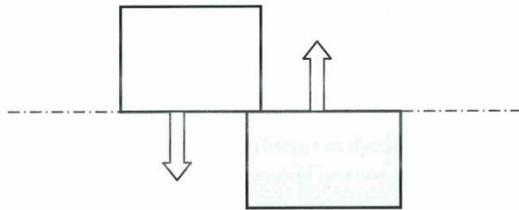
➤ **Align** – Makes two planes coplanar with their faces aligned in the same direction. Selected surfaces point in the same direction. The Align constraint aligns components adjacent to one another with faces aligned. For axisymmetric objects, Align can be used to align two circles or cylindrical surfaces, including their center axes and planes. Selected circular surfaces can become co-axial or tangent to each other.
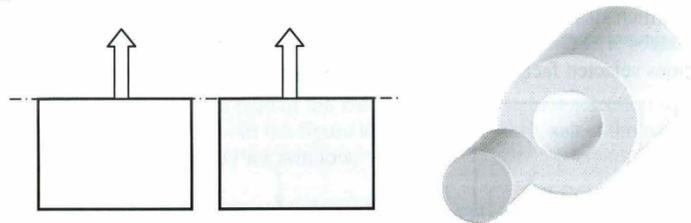
➤ **Angle** – Creates an angular assembly constraint between parts, subassemblies, or assemblies. Selected surfaces point in the direction specified by the angle.

Angle
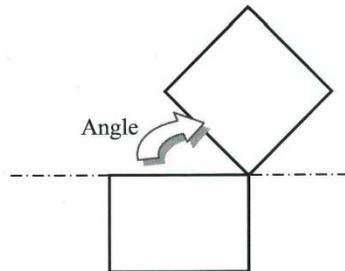
➤ **Parallel** – Constrains the surfaces or direction vectors of two selected objects as parallel to each other.

➤ **Perpendicular** – Constrains the surfaces or direction vectors of two objects as perpendicular to each other.

90°
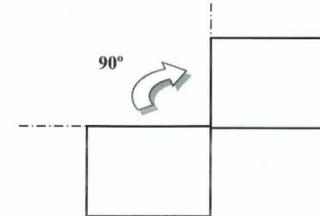
➤ **Concentric** – Lets you center one object to the center of the other of object.

➤ **Center** – Lets you center one object anywhere along the center of another object, or center one or two objects between a pair of objects.

➤ **Distance** – Constraint positions components face-to-face, and can be offset by a specified **distance**. Specifies the minimum 3D distance between two objects. You can control on which side of the surface the solution should be by using positive or negative values. The Distance constraint is similar to the Touch constraint, which also positions selected faces normal to one another, with faces coincident.

Distance
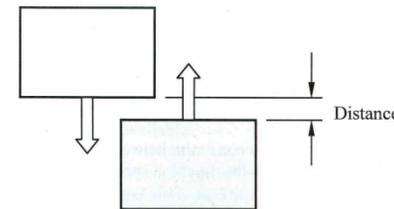
➤ **Align/Tangent** – Aligns selected faces, planes, cylinders, spheres, and cones to contact at the point of tangency. Tangency may be on the inside or outside of a curve, depending on the selection of the direction of the surface normal. A Tangent constraint removes one degree of translational freedom.
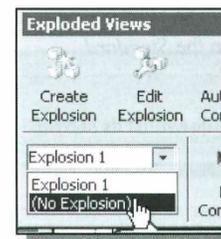
Tangent requires picking at least one curved surface.
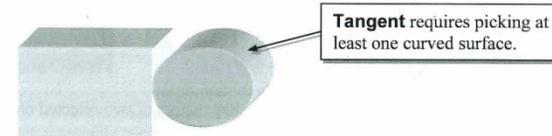
Assembly and Exploded View

The *Shaft-Support* Assembly
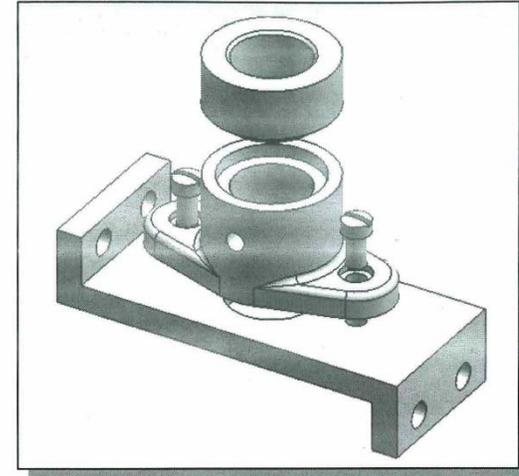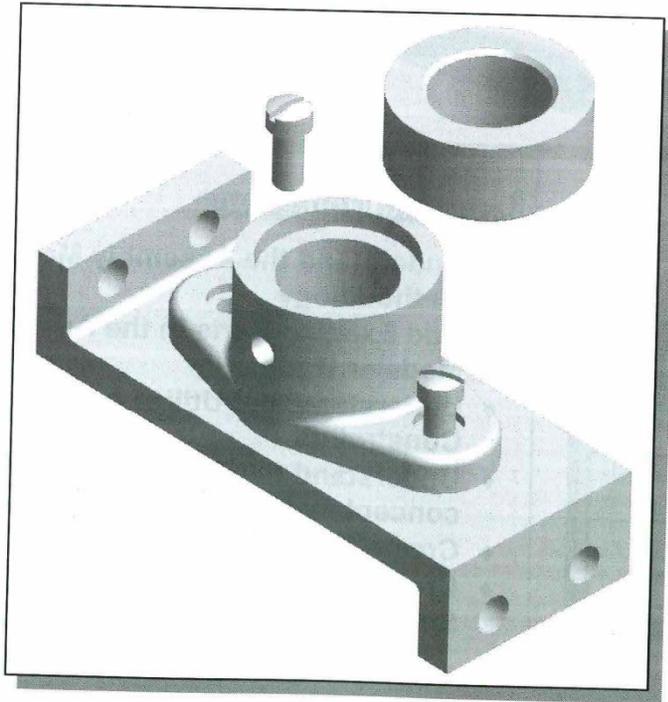
Switching the Exploded/Unexploded Views

- *UGS NX* allows us to store as many exploded views as desired and we can also sw
  to any of the exploded views by choosing the from the *Exploded Views* list box.

1. Inside the *Exploded Views* dialog box, select **(No Explosion)** to switch to the unexploded view of the assembly model.