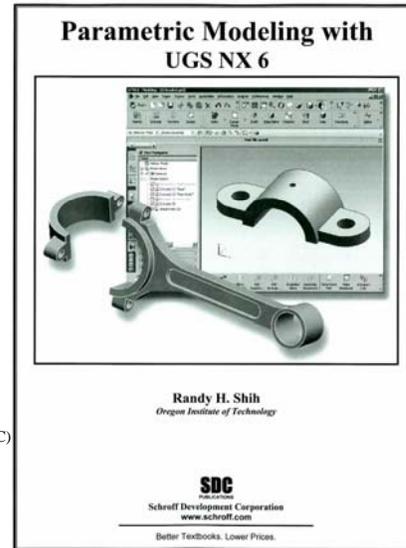


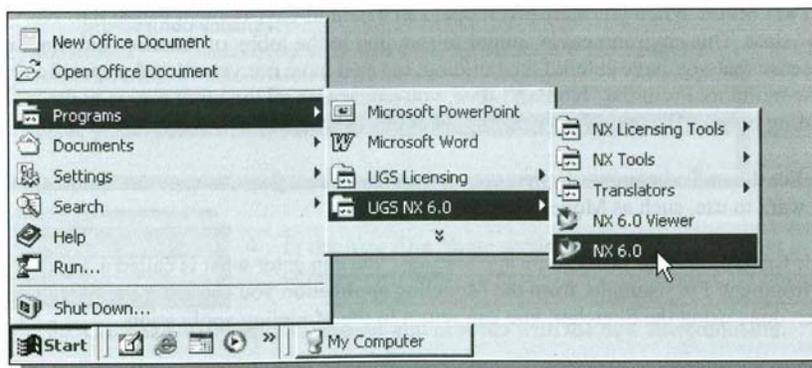
UGS NX 6

Course Webpage: [Unigraphics NX Tutorials and Related Documents](#)

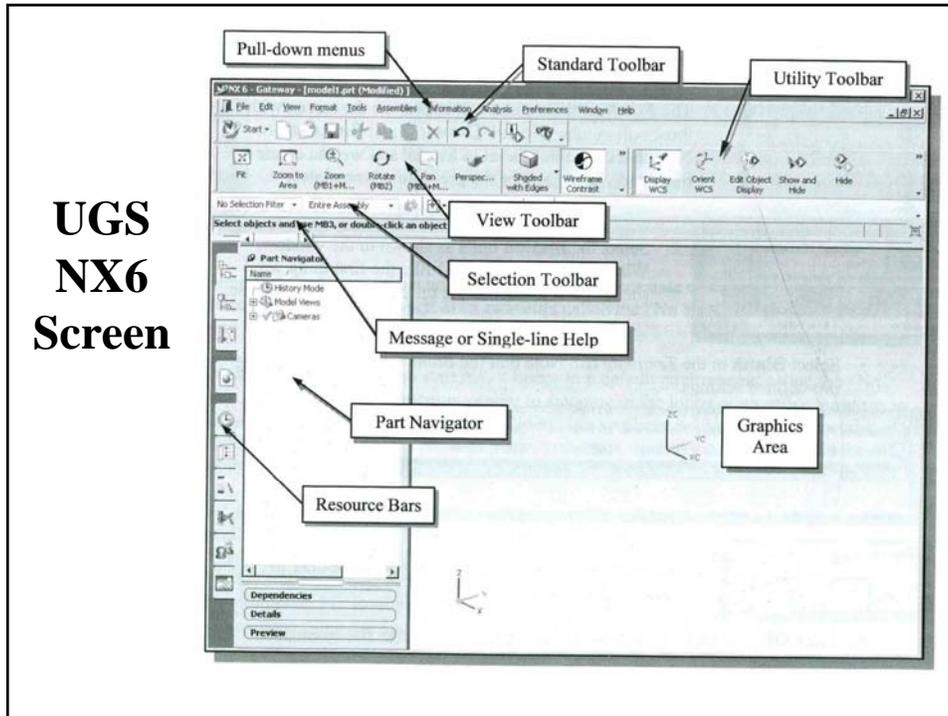
- **NX CAST 6 - A Comprehensive NX CAST 6 Tutorial**
 - Available on the PCs with UGS NX 6
 - CD of NX CAST 6 Available
- **NX5 for Engineering Design - A Complete Tutorial** (M. C. Leu and A. Joshi, Missouri Univ of Sci. & Tech.)
 - Introduction
 - Getting Started
 - Form Features
 - Feature Operations
 - Drafting
 - Sketching
 - Freeform Feature
 - Assembly Modeling
 - Manufacturing
 - Finite Element Analysis
- **Parametric Modeling with UGS NX 6 Book** (by Randy H. Shih, from SDC)
 - [Table of Contents](#)
 - 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)

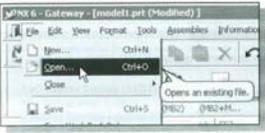


Start UGS NX6



UGS NX6 Screen



- Pull-down Menus**


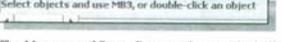
The pull-down menus at the top of the main window contain operation that you can use for all modes of the system.
- Standard Toolbar**

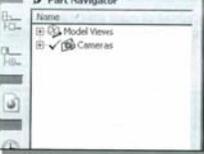

The Standard toolbar at the top of the screen window allows us quick access to frequently used commands. For example, the file-related commands, such as Switching Applications, New Part, and Save.
- View Toolbar**

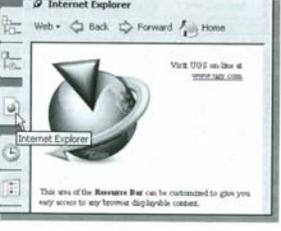

The View toolbar allows us quick access to frequently used view-related commands, such as Zoom, Rotate, Creating Section View and Shaded Solids.
- Utility Toolbar**


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**

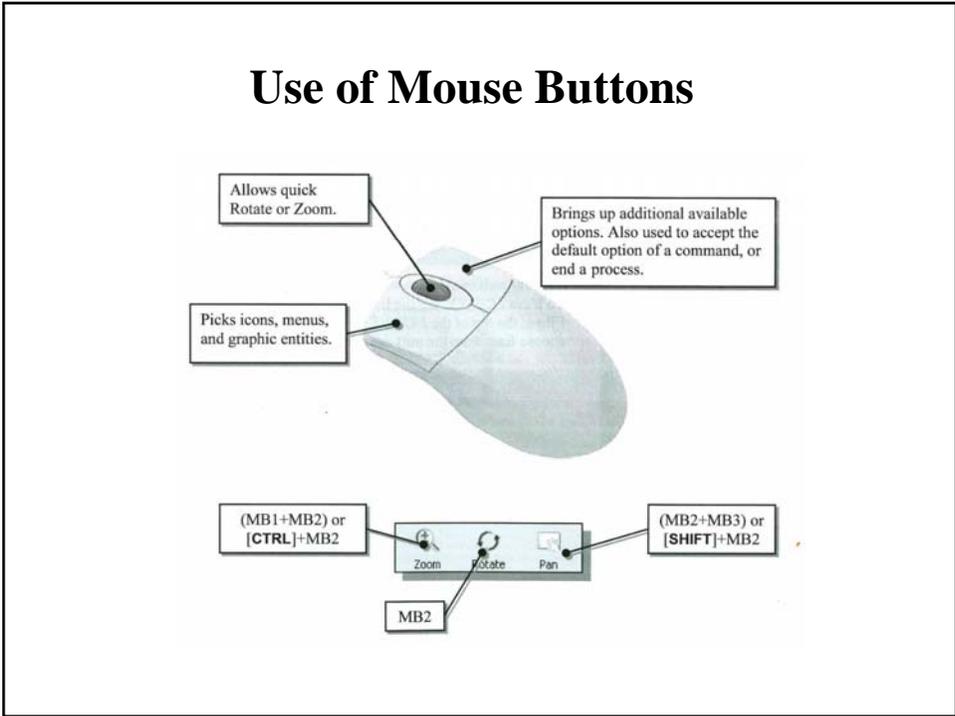

The Selection toolbar provides tools for quick selection of 2D/3D features and parts.
- Message and Status Bar**


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 used by the Resource Bars options described below.
- Resource Bars**


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

Use of Mouse Buttons



Application Screen

Form Feature Toolbar
 Sketch Extrude Revolve Sweep Along
 This toolbar contains tools that allow us to quickly create 2D sketches and form 3D features.

Feature Operation Toolbar
 Unite... Datum Plane... Draft Edge Blend Chamfer Shell Hide
 This toolbar contains tools that allow us to quickly create placed features.

Curve Toolbar
 Spline Project
 This toolbar contains tools that allow us to quickly access 2D curve options.

Resource Bar

Graphics Area

Curve Toolbar

- 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.

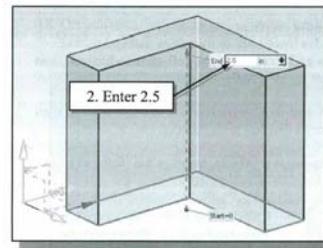
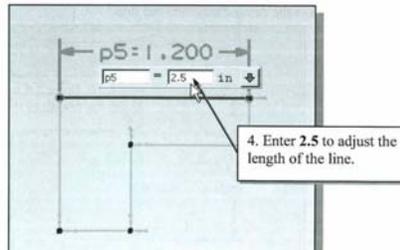
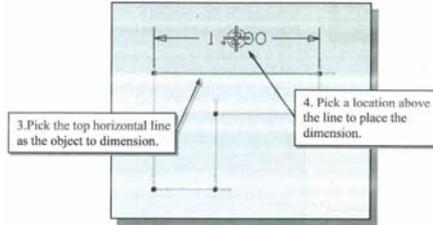
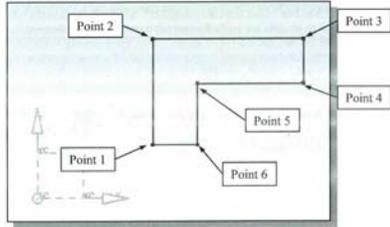
- The **Extrude Feature Options dialog box**, which contains applicable construction options, is displayed as shown in the figure below.

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.

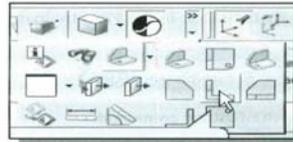
❖ Note the default sketch plane is aligned to the **XY-FC plane** of the displayed **work coordinate system**.

4. Click **OK** to accept the default setting of the **sketch plane**.

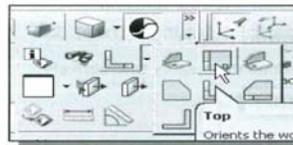
Sketching, Dimensioning and Modeling



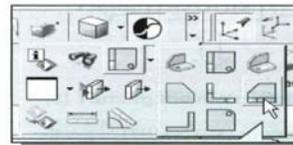
Different Display Orientations



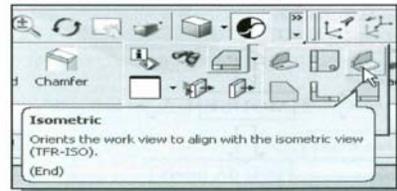
1. Select **Fr** change th
[Ctrl+Alt+command]



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

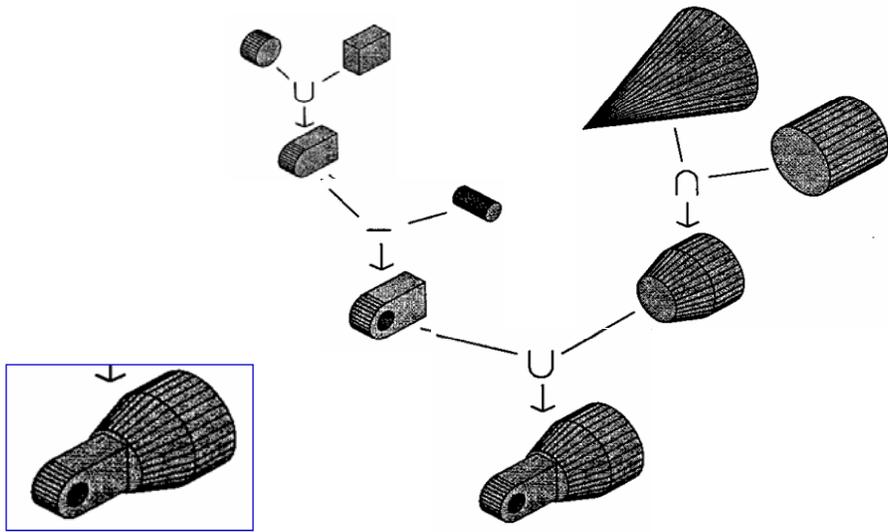


3. Select **Ri** change th
that the [C activate tl

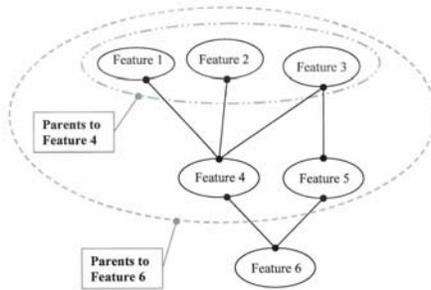


4

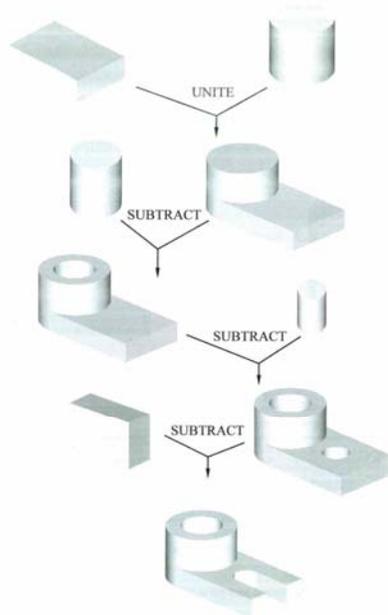
CSG Tree



CSG Modeling and Parent-Child Relations



Modeling Strategy - CSG Binary Tree



Drawing Generation

UGS NX Drafting Mode

1. Click on the **Start** icon in the toolbar to display the available option.
2. Select **Drafting** from the options.
3. In the **Sheet** dialog box, UGS settings for the drawing sheet.
 - 3. Select **A-8.5x11** for the sheet size.
 - 4. Confirm the **Scale** is **full scale**.
 - Note that **Sheet #** is in the **sheet name** area.
5. In the **Settings** area, set to **Inches** and to **3rd Angle Projection**.
6. **Uncheck** the **Automatically Start Base View Command**.

Adding a Base View

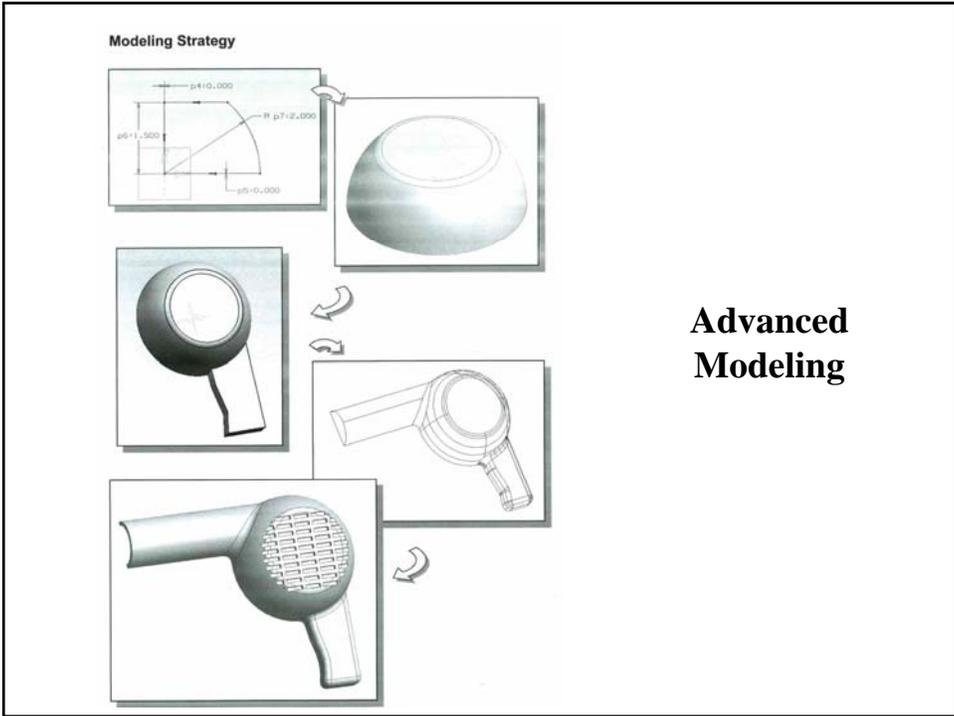
In UGS NX Drafting mode, the first drawing view is the primary view in a drawing; called a base view. When creating a base view, UGS NX will allow views to be shown. By default, UGS NX will show a view of the solid model.

1. Click on the **Base View** toolbar icon to create a base view.
2. In the **Model View** dialog, select **TOP** as the **Model View to Use**.
3. In the **Settings** dialog, set **View Style** to **Non-Sectioned**.
4. In the **View Style** dialog, set **Hidden Lines** to **Visible**.
5. Choose **D** for **hidden lines**.

Modeling – Extrusion and Revolving

Modeling Strategy

Modeling Strategy – A Revolved Design

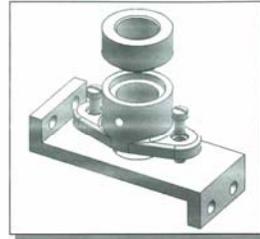


Assembly Relations

- **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.
- **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.
- **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.
- **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.

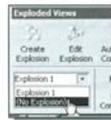
Assembly and Exploded View

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 switch 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.

