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
- Standard Toolbar
- Utility Toolbar
- View Toolbar
- Selection Toolbar
- Message or Single-line Help
- Part Navigator
- Graphics Area
- Resource Bars
- Pull-down Menus

The pull-down menus at the top of the main window contain operations 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 an 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 is 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

- 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
(MB2+MB3) or [SHIFT]+MB2
Application Screen

- 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.

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

- Note the default sketch plane is aligned to the XC-YC plane of the displayed work coordinate system.
Sketching, Dimensioning and Modeling

1. Pick a location above the line to place the dimension.
2. Enter 2.5
3. Pick the top horizontal line as the object to dimension.
4. Enter 2.5 to adjust the length of the line.
Different Display Orientations

1. Select Front and change the view with the [Ctrl+Alt] command.

2. Select Top to orient the display windows and activate the Top command.

3. Select Right to change the view and activate the Right command.

4. Isometric view: Orients the work view to align with the isometric view (TFR-ISO).
CSG Tree
CSG Modeling and Parent-Child Relations
Adding a Base View

- In UGS NX Drafting mode, the first drawing view is the primary view in a drawing. Only one view. When creating a base view, UGS NX allows all views to be shown. By default, UGS NX will show the model view.

1. Click on the BaseView tab on the View dialog box to create a new drawing.

2. In the View dialog box, select the view to be added as a base view.

3. In the View Style dialog box, select the View Style options.

4. In the View Style dialog box, select the View Style options.

5. Choose the View Style options.

Drawing Generation

1. Click on the Start icon in the UGS NX Drafting Mode.

2. Select Drafting from the Options menu.

3. In the Sheet dialog box, UGS settings for the drawing sheet are displayed.

4. Select A-8.5x11 from the Sheet dialog box.

5. Confirm the Scale is full scale.

6. Note that Sheet 1 is the default sheet name that is displayed in the drawing area.

7. In the Settings area, set to Inches and then to 3rd Angle Projection.

8. Uncheck the Author View Command option.

9. In the View Style dialog box, select the View Style options.

10. Choose the View Style options.

11. In the View Style dialog box, select the View Style options.

12. Choose the View Style options.
Modeling – Extrusion and Revolving
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.

- **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 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 view 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.