



UNIGRAPHIC



About Course

Unigraphics NX (also known as Siemens nx) is an advanced High-end CAD/CAM/CAE software package originally developed by UGS Corporation, But since 2007 it is owned by Siemens PLM Software. It is used, among other tasks, for Design ,Engineering analysis ,Manufacturing finished design by using included machining modules.

UNIGRAPHICS

CURRICULUM

① INTRODUCTION

Introduction to NX
System Requirements
Getting Started with NX
Important Terms and Definitions
Understanding the Functions of the Mouse Buttons
Toolbars
Hot Keys
Color Scheme
Dialog Boxes in NX
Selecting Objects
Deselecting Objects
Selecting Objects Using the Quick Pick Dialog Box
Self-Evaluation Test

② DRAWING SKETCHES FOR SOLID MODELS

The Sketcher Environment
Starting NX
Starting a New Document in NX

Invoking Different NX

Environments

Creating Three Fixed Datum Planes (XC-YC, YC-ZC, XC-ZC)

**Displaying the WCS (Work
Coordinate System)**

Invoking the Sketcher Environment

Sketching Tools

Drawing Sketches Using the Profile Tool

Using Help Lines to Locate Points

Drawing Individual Lines

Drawing Arcs

Drawing Circles

Drawing Rectangles

Placing Points

**Drawing Ellipses or
Elliptical Arcs**

Drawing Conics

Drawing Studio Splines

Filleting Sketched Entities

The Drawing Display Tools

Fitting Entities in the Current Display

Zooming to an Area

Dynamic Zooming

Panning Drawings

Fitting View to Selection

Restoring the Original Orientation of the Sketching Plane
Setting Selection Filters in the Sketcher Environment
Selecting Objects
Deselecting Objects
Using Snap Points Options While Sketching
Deleting Sketched Entities
Exiting the Sketcher Environment

3 ADDING GEOMETRIC AND DIMENSIONS

Constraining Sketches
Concept of Constrained Sketches
Under-Constrain
Fully-Constrain
Over-Constrain
Degree of Freedom Arrows
Geometric Constraints
Applying Additional Constraints Individually
Applying Automatic Constraints to a Sketch
Controlling Inferred Constraints Settings
Showing All Constraints in a Sketch
Turning off the Display of All Constraints in a Sketch
Editing Sketched Entities by Dragging
Exiting the Sketcher Environment
Changing the View of the Sketch
Creating Base Features by Extruding

Extrude Dialog Box Options
Creating Solid Revolved Bodies
Hiding Entities
Showing Hidden Entities
Hiding All Entities Using a Single Tool
Rotating the View of a Model in 3D Space
Setting Display Modes

4

WORKING WITH DATUM PL

ANES, COORDINATE SYSTEMS
Additional Sketching and Reference Planes
Types of Datum Planes
Creating Three Fixed (Principle) Datum Planes
Creating Relative Datum Planes
Creating Datum Coordinate Systems
Creating Fixed and Relative Datum Axes
Other Extrusion Options
Specifying the Boolean Operation
Specifying Other Extrusion Termination Options
Projecting External Elements

5

ADVANCED MODELING TOOLS-I

Advanced Modeling Tools
Creating Simple Holes

Creating Counter bore Holes
Creating Countersink Holes
Creating Holes by using the Hole Tool
Creating General Holes
Creating Drill Size Hole
Creating Screw Clearance Hole
Creating Threaded Hole
Creating Hole Series
Creating Grooves
Creating Rectangular Grooves
Creating Ball End Grooves
Creating U Grooves
Creating Slots
Creating Rectangular Slots
Creating Ball-End Slots
Creating U-Slots
Creating T-Slots
Creating Dove-Tail Slots
Creating Chamfers
Creating a Chamfer Feature Using the Symmetric Method
Creating a Chamfer Feature Using the Asymmetric Method
Creating a Chamfer Feature Using the Offset and Angle Method
Creating an Edge Blend

6

ADVANCED MODELING TOOLS-II

Advanced Modeling Tools

Instance Feature Tool

Creating Rectangular Arrays Using
the Instance Feature Tool

Creating Circular Arrays Using
the Instance Feature Tool

Using the Pattern Face Option of
the Instance Feature Tool

Mirror Feature Tool

Mirror Body Tool

Sweeping Sketches along the Guide Curves

Creating Swept Features

Creating Tubes or Cables

Creating Threads

Creating Symbolic Threads

Creating Detailed Threads

Creating Shell Features

Shelling the Entire Solid Body

7

EDITING FEATURES AND

ADVANCED MODELING TOOLS-III

Editing Features

Editing a Hole Feature

Editing the Positioning of a Hole Feature

Editing the Positioning of a Groove Feature
Editing the Positioning of a Slot Feature
Editing the Parameters of Features
Editing the Parameters of Features with Rollback
Editing Sketches of the Sketch-based Features
Reordering Features
Advanced Modeling Tools
Creating Boss Features
Creating Pocket Features
Creating Pad Features
Creating Drafts

8 ASSEMBLY MODELING-I

The Assembly Environment
Invoking the Assembly Environment
Invoking the Assembly Environment Using the Assembly
Template from the New Dialog Box
Invoking the Assembly Environment
in the Current Part File
Types of Assembly Design Approaches
Creating Bottom-up Assemblies
Placing Components in the
Assembly Environment
Changing the Reference Set of a Component
Applying Assembly Constraints to Components

Points to Remember while Assembling Components
Creating a Component Array in an Assembly
Replacing a Component in an Assembly
Moving a Component in an Assembly
Repositioning a Component in an Assembly
Mirroring a Component in an Assembly
Modifying a Component in the Assembly File

9

ASSEMBLY MODELING-II

The Top-down Assembly Design Approach
Creating Components Using the Top-down Assembly Design Approach
Creating Subassemblies
Editing Assembly Constraints
Modifying the Assembly Constraints
Checking the Interference between the Components of an Assembly
Checking Interference and Clearance Using the Check Clearance Analysis
Checking Interference Using the Assembly Clearance Method
Checking Interference and Clearance, and Analyzing Cross-sections of Components Using the View Section Tool
Creating Exploded Views of an Assembly
Exploding Views Automatically
Exploding Views Manually

SURFACE MODELING

Introduction to Surface Modeling
Invoking the Sheet Modeling Environment
Creating an Extruded Surface
Creating a Revolved Surface
Creating a Ruled Surface
Creating a Surface Using the Through Curves Tool
Creating a Surface Using the Through Curve Mesh Tool
Creating a Surface Using the
Four Point Surface Tool
Creating a Swoop Surface
Creating the Planar Surfaces from 2D Sketches and
Edges of Solid or Surface
Creating a Transition Surface Using the Transition Tool
Creating an N-Sided Surface
Creating a Silhouette
Flange Surface
Extending a Surface Using the Law Extension Tool
Creating a Surface Offset Using the Offset Surface Tool
Trimming and Extending a Surface Using the
Trim and Extend Tool
Trimming a Sheet by Using the Trimmed Sheet Tool
Creating a Surface Using the Studio Surface Tool
Creating a Surface between Two Walls
Using the Styled Blend Tool

Creating Surfaces Using the Styled Sweep Tool
Sewing Individual Surfaces into a Single Surface
Adding Thickness to a Surface

11

ADVANCED SURFACE MODELING

Creating Curves from Bodies
Creating Intersection Curves
Creating Section Curves
Creating Extract Curves
Advanced Surface Modeling Tools
Creating Dart Features
Creating Emboss Sheet Features
Creating Face Blend Features
Creating Soft Blend Features
Creating Fillet Features
Creating Bridge Features

12

GENERATING, EDITING, AND DI

MENSIONING THE DRAWING VIEWS

The Drafting Environment
Invoking the Drafting Environment
Invoking the Drafting Environment Using the Drawing
Template from the New Dialog Box
Invoking the Drafting Environment in
the Current Part File

Editing the Drawing Sheet Parameters in the Drafting Environment

Invoking the Drafting Tools

Types of Drawing Views in NX

Base View

Projected View

Detail View

Section View

Auxiliary View

Half-Section View

Revolved Section View

Break-Out Section View

Broken View

Generating Drawing Views

Generating the Base View

Generating the Orthographic Drawing Views

Using the Projected View Tool

Generating the Detail View Using the Detail View Tool

Generating Section Views Using the Section View Tool

Generating the Half Section View Using the Half Section View Tool

Generating the Revolved Section View

Generating the Break-Out Section View

Generating the Broken View

Manipulating the Drawing View

Aligning the Drawing Views Using the Align View Tool View Boundary
Displaying the Model Using the Display Sheet Tool
Inserting a Drawing Sheet Using the New Sheet Tool
Modifying the Properties of a Generated Drawing View the Scale Value of the Drawing View
Adding Dimensions to the Drawing Views
Retrieving Dimensions from the Model
Adding Dimensions to the Drawing View
Generating Exploded Views of an Assembly
Creating Parts List and Associative Balloons
Creating a Parts List for an Assembly
Creating Associative Balloons
Creating a Tabular Note (Title Block)
Adding Multiline Text to a Drawing Sheet
Printing Tools
Print
Plot

13

SYNCHRONOUS MODELING

Introduction
Move Face
Pull Face
Offset Region
Replace Face

Resize Blend
Resize Face
Delete Face
Copy Face
Cut Face
Paste Face
Mirror Face
Pattern Face
Resize Chamfer
Label Chamfer
Make Coplanar
Make Coaxial
Make Tangent
Make Symmetric
Make Parallel
Make Perpendicular
Make Fixed
Show Related Face
Linear Dimension
Angular Dimension
Radial Dimension
Shell Body
Shell Face
Change Shell Thickness
Group Face
Cross Section



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