

Department of Mechanical Engineering

Title of the Program : Value Added Course on "CAD using UG - NX"

Date: 13.02.2024 to 19.02.2024

Participants : II year (2022 – 2026 Batch)

Academic Year: 2023 – 2024 EVEN

Conducted by : CIPET, Madurai

Venue : KCER-MECH-CAD Lab

Syllabus

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

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 - 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

EDITING, EXTRUDING, AND REVOLVING SKETCHES

Editing Sketches - Trimming Sketched Entities - Extending Sketched Entities - Creating a Corner between Sketched Entities - Mirroring Sketched Entities - Copying, Moving, and Rotating Sketched Entities - Transforming Sketched Entities - 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

MODELING TOOLS

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 Slots - Creating Dove-Tail Slots - Creating an Edge Blend - 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

ASSEMBLY MODELING

The Assembly Environment - 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 - Exploding Views Automatically & Manually

W. R. ...
Coordinators

J. ...
HoD/Mech

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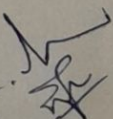
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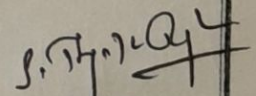
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Value Added Course Outcome

Course Outcome	Details
1	Understand industry standards and best practices for creating manufacturing-ready drawings.
2	Create 3D models of mechanical parts and assemblies using NX CAD's robust modeling tools.
3	Utilize simulation and analysis tools within NX CAD to evaluate the performance of designs.
4	Learn to manipulate surfaces, curves, and splines to achieve desired aesthetics and functionality.
5	Understand the importance of data integrity, security, and revision control in collaborative design projects.


Coordinators


HoD/Mech