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**Introduction** to CADDESK, Mechanical CAD/CAM/CAE Software, Basic Layout NXCAD and its uses, Advantages.

**Interface** of NX CAD, Mouse function, Basic commands (Save, Open, Import, Export.)

**Sketch** (Sketching, sketch in task environment, sketch curve) & **Sketch** (Constraint, Dimensions)

**Design feature** (Extrude, Revolve, datum plane, datum axis, datum CSYS, datum point.)  
Block, Cylinder, Cone, Sphere, hole, boss, pocket) (Pad, Emboss, Slot, Groove, rib, draft, thread)  
Pattern feature, mirror feature, unite, subtract, intersect, sew, un-sew, join face)

**Design feature** (Trim body, split body, trim sheet, extend sheet, un-trim, divide face, delete edge, delete body)

**Design feature** (shell, thicken, scale, offset, draft, chamfer, edge blend, face blend)

**Drafting** (Introduction, View creation wizard, base view, update view, section view)  
(Annotations, dimensions, tables.) (custom symbol, drawing format)

### 1.1 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

## 1.2 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

## 2. 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

## **2.2 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

## **3. SHEET METAL MODELING**

- Customer defaults,
- Tabs,
- Base features,
- Secondary features,
- Flanges, full width flange,
- Centered flange, at end flange,
- Offset flange, contour,
- Contour flange, lofted flange,
- Corner,
- Break corner,
- Closed corner,
- Three bend corner,
- Job, bend, unbend,

- Dimple,
- Louver, cutout, bead, solid punch,
- Edge rip, flat solid,
- Advanced sheet metal features.

+ PROJECTS

+ Industrial Projects

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