

# NX CAD COURSE TRAINING OUTLINE



**Course Name :** NX CAD Basic Design & Freeform Modeling

**Course Duration :** 3 Days

**Course Outline :**

## **Modeling Fundamental**

- Opening and working with parts
- Getting to know the NX interface
- Creating parts with sketches
- Editing and manipulating sketches
- Sweeping geometry to create part features
- Creating and editing holes
- Creating and editing geometric relationships with formulas

## **Modeling Fundamental**

- Creating datum geometry to support design intent
- Examining the structure of a model
- Trimming a solid body
- Creating and manipulating shell features
- Blending and chamfering edges
- Modifying geometry of imported parts

## **Free form Modeling**

- Introduction to Mechanical Freeform
- Curves used for creating freeform features
- Edit Splines
- Curve Analysis

- Primary Sheets and Bodies
- Transitions sheet
- Face Analysis
- Working with sheets
- Sheets to solids

### **Drafting & Assembly**

- Adding and positioning parts in an assembly
- Top-down design and Interpart Link
- Assembly sequence
- Creating simple drawing
- Dimension and Annotation
- Section view