

**Course Name** : ***NX CAD – Advanced (Synchronous Modeling, Surface Modeling, Sheet Metal Modeling, PMI, GD&T, Render Studio)***

**Course Duration** : 40 Hrs.

#### **Course overview**

- **Intended audience**
  - This course is suited for designers, engineers, manufacturing engineers, application programmers, NC programmers, CAD/CAM managers, and system managers who need to manage and use NX.
- **Prerequisites**
  - **Education:** Diploma completed or Degree 2<sup>nd</sup> year completed in any one of following Streams.
    - Aeronautical, Automobile, Civil, Industrial, Marine, Mechanical, Mechatronics, Metallurgy, Production and Manufacturing Engineering.
  - **Software:** **Siemens NX**
    - Working knowledge of the following:
      - NX interface.
      - Sketching and constraining techniques.
      - Assemblies & constraining assembly components.
      - Feature-based solid modeling.
      - Using the Part Navigator.
- **Course objectives**
  - After successfully completing this course, you should be able to:
    - Import and Export the CAD file to various formats.
    - Set up default standards and creation preferences.
    - Grouping the history in part navigator
    - Create and edit parametric solid models.
    - Utilize additional sketch techniques.
    - Perform unplanned design changes using synchronous modeling.
    - Create and edit splines and other curves.
    - Create and edit primary and transition surfaces using freeform curves and faces
    - Use Sheet Metal tools to create base features and add more advanced features to them.
    - Use the Sketch Task Environment to create and edit profiles for sheet metal parts.
    - Create Dimensions & Tolerances to part model (2D & 3D)
    - Visual Rendering of part model
- **Course Contents**
  - Customize User preferences
  - Import and Export the CAD file to various formats

- Part Navigator history grouping
- Synchronous sketching Synchronous Modeling
- Modify Face
- Detail Feature
- Delete Face
- Reuse commands
- Synchronous Modeling relationships
- Dimension commands
- Adaptive Shell
- Edit Cross Section and Edit Section
- Sweeping geometry to create part features
- Through Curve surfaces
- Through Curve Mesh surfaces
- Swept surfaces and bodies
- Section surfaces and bodies
- Bridge surfaces
- Face and Soft Blends
- N-Sided surfaces
- Analyze curves and faces.
- Sheet Metal workflow
- Establish basic part characteristics
- Define the basic shape of the part
- Constructing base features
- Sheet Metal corners
- Sheet Metal cut-outs
- Sheet Metal deform features
- Flat Solid and Flat Pattern
- Analyze Formability – One step
- PMI to 3D Models
- Insert Table
- Dimension to Drafting views
- Annotations to Drafting views
- Setup Visual Materials
- Add text, Image to 3D models