

**Course Name** : *Essentials for NX Designers*

**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:** There are no prerequisites for this class.
- **Course objectives**
  - After successfully completing this course, you should be able to:
    - Open and examine NX models.
    - Create and edit parametric solid models.
    - Create and modify basic assembly structures.
    - Create and modify simple drawings.
- **Course Contents**
  - Opening and Working with Parts
  - Getting to know the NX Interface
  - Impact of Coordinate Systems on Parts
  - Creating Parts with Sketches
  - Sweeping geometry to create part features
  - Creating and editing geometric relationships with formulas
  - Creating datum geometry to support design intent
  - Examining the structure of a model
  - Editing and manipulating sketches
  - Trimming a solid body
  - Creating swept features with offset and draft
  - Creating and editing holes
  - Creating and manipulating shell features
  - Copying and mirroring part segments
  - Blending and chamfering edges
  - Modifying geometry of imported parts
  - Loading and working with assemblies
  - Adding and positioning parts in an assembly
  - Creating simple drawings
  - Practice projects