

Course Name : *NX Basic Design*

Course Duration : 16 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 2nd year completed in 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
 - Creating parts with sketches
 - Creating datum geometry to support design intent
 - Sweeping geometry to create part features
 - Creating and editing holes
 - Blending and chamfering edges
 - Loading and working with assemblies
 - Adding and positioning parts in an assembly
 - Creating simple drawings