

NX CAD Basic Design

course code TR10053
user level Beginner
duration 2 days

Basic Design is designed to give an entry level user a high-level overview of NX modeling, assemblies and drafting topics. This one class, through professional instruction related to product design, assembly modeling, and master model concepts, allows the student to transfer classroom instruction to job productivity.

At the completion of the **Basic Design** class, the student will be able to develop basic solid and assembly models as well as drawings using the master model concept. These concepts can be applied in the real world of product development and collaboration. As with each course developed and taught by Siemens PLM Software professionals, this class reinforces our intimate knowledge of software's developments and instructs the students based on the underlying principles incorporated within the NX product suite. The NX Basic Design class was designed to increase the productivity of the student, teaching the necessary skills to accomplish the following:

- Open and examine NX models
- Create and modify basic assembly structures
- Create and edit parametric solid models
- Create and modify simple drawings

WHO SHOULD ATTEND	PRIMARY COURSE TOPICS
Designers, engineers, manufacturing engineers, application programmers, NC programmers, CAD/CAM managers, and system managers who need to manage and use NX	<ul style="list-style-type: none"> • Introduction and overview • NX part files • User interface • Sketching • Datum features • Swept features • Holes • Edge operations • Introduction to assemblies • Assembly constraints • Introduction to drafting
PREREQUISITES	
None	
PROVIDED COURSE MATERIALS	
<ul style="list-style-type: none"> • Student Guide • Activity Material 	