

NX CAD
Introduction to NX for Experienced Users

course code TR13155
user level Beginner to Intermediate
duration 5 days

The **Introduction to NX for Experienced Users** course provides an introduction to NX for those students who will be using NX on a daily basis. Upon completion of this course, accomplished 3D Parametric CAD users will have the capability to create and modify parts, assemblies, and products in NX, leveraging their years of parametric modeling experience. The pace and topics of this course have been carefully planned specifically for the experienced 3D Parametric CAD user.

At the completion of this course the student will productively develop solid models, detail drawings, and product assemblies. This course introduces assembly modeling in the context of a real-life scenario that includes parts modeled by the student as well as part models that have already been created. As with each course developed and taught by Siemens PLM Software professionals, this course capitalizes on Siemens PLM Software's intimate knowledge of the software's development and instructs the students based on the underlying principles incorporated within the NX software.

WHO SHOULD ATTEND	PRIMARY COURSE TOPICS
Designers, engineers and CAD/CAM managers.	<ul style="list-style-type: none"> • Working with existing parts • NX user interface • Sketching • Datum features • Swept features • Trim Body • Hole features • Expressions • Coordinate systems • Part Navigator • Associative copies • Face and edge operations • Basic freeform • Creating and modifying assemblies • Assembly Constraints • Assembly Arrangements • Reference Sets • Interpart geometry and references • Component arrays • Reuse Library • Revise and replace components • Drafting
PREREQUISITES	
Working knowledge of parametric modeling	
PROVIDED COURSE MATERIALS	
<ul style="list-style-type: none"> • Student Guide • Activity Material 	