

NX CAM NX Manufacturing Fundamentals

course code TR11021
user level Beginner
duration 3 days

The **NX Manufacturing Fundamentals** course is designed to expedite the student learning experience in the generation of tool paths for 2 and 3-axis milling and drilling applications. As with each course developed and taught by Siemens PLM Software professionals, this class reinforces Siemens PLM's intimate knowledge of software's developments and instructs the students based on the underlying principles incorporated within the NX product suite.

WHO SHOULD ATTEND	PRIMARY COURSE TOPICS
<ul style="list-style-type: none"> Manufacturing engineers CAD/CAM managers NC/CNC programmer CAM Express users 	<ul style="list-style-type: none"> Introduction and Overview Part analysis for manufacturing User Interface Operation Navigator Machine Coordinate System Tooling Visualization/Verification Post Processing/Shop Documentation Planar/Cavity Milling Drilling Fixed Contour Area Milling Face Milling Text Engraving
PREREQUISITES	
Required courses: <ul style="list-style-type: none"> Basic Design (TR10053) CAM Express Users: You must first complete the self-paced tutorials supplied with CAM Express prior to attending the Manufacturing Fundamentals course. Essentials for NX Designers (TR10051) is for non CAM Express user 	
PROVIDED COURSE MATERIALS	
<ul style="list-style-type: none"> Student Guide Activity Material 	