

# Express: NX CAD basics

## Objective

The course serves as an introduction to the use of the NX functionality without a "modelling licence". The user learns the basics of the CAM Express interface and handling parts imported to NX. The focus is on dealing with CAD models in terms of the Master Model Concept for a process-based work method that is aligned with concurrent engineering. After the course, participants will be able to handle geometry models in assemblies and implement the potentially necessary changes to parts imported from external systems for NC machining.

## Content

- The NX program package (concept history)
- Basics of NX / user interface
- Creation of basic structures and reference elements
- Creation of curves (auxiliary curves for CAM)
- Rapid model change using "Synchronous Technology"
- Analysis and information of objects
- Introduction to the Master Model Concept
- Methods for assembling a module (assembly)
- Creation of associative copies (Wave Geometry Linker)

## Additional Requirements

PC knowledge. The course serves as a minimum requirement for participation in CAM seminars.

## Additional informations for the duration

Two additional days on-site are recommended in order to provide further explore the applications and increase efficiency. Additional days as required.



## Basic Course



## Duration

3 Days

## Language

German

## Documentation

German

## Participants

- Production engineers
- NC programmers
- production planning employees