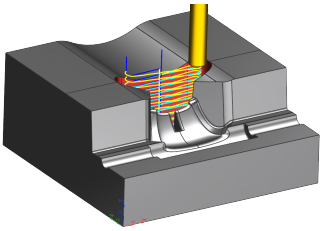
Manufacturing 2: NX CAM basics



Objective

This seminar covers the functionalities and opportunities provided by the NX CAM module in the areas of 3-axis rough machining, 2- and 3-axis finishing and simple drilling. The process-based philosophy and the effective use of the NX CAM module is taught using practical examples. The integration and consistency of NC machining on NX and in the customer's area of application will also be analysed and established.

Content

- The NX program package (concept, history)
- CAM process based on the Master Model Concept with Wave Technology
- Basic process from modelling through to production
- Working with the ONT Operation Navigation Tool (philosophy, tree structure)
- Basic machining of planar surfaces using the Floor Wall module
- Plane roughing complex geometries using the Cavity Milling module
- Basic drilling modules (drilling, sinking, reaming, tapping) using the Hole Making module
- Toolpaths check (verification)
- Workshop documentation (Shop Doc), machine control and post-processing
- 3-axis finishing (Z level, mill contour)

Additional Requirements

General principles of NC programming, participation in the "Manufacturing 1: NX CAD basics" seminar, or equivalent NX/CAD training and NX experience.

Additional informations for the duration

An additional three days of on-site practical application is recommended. Additional days as required.

Basic Course

Duration

5 Days

Language

German/English

Documentation

German/English

Participants

- Production engineers
- NC programmers
- production planning employees

Required Trainings

- Manufacturing 1: NX CAD basics