

Mastercam Design and Basic Milling - From Print to Part Program

This course is designed to introduce students to Mastercam X and enable them to create programs for CNC milling machines. The focus is “hands-on” learning that leads the student through Mastercam’s core features by using sample parts that require the use of progressively difficult Mastercam Design and Mill routines.

Scope of Class

Mastercam X interface:

- Software structure & framework
- Customizing your work environment & toolbars,
- Machine & CNC configuration,
- Key mapping

CAD Design topics – Create Geometry From Print:

- 2D and 3D geometry creation (point, line, arc, fillet and chamfer)
- Xform (translate, mirror, rotate, scale, offset and offset contour)
- Color management, level management, view management

CAM Milling topics –Generate Tool Path & NC Code:

- Toolpaths (face, contour, drill, pocket, transform, trim)
- Geometry selection (chaining)
- Tool selection/creation, tool library, cutting parameters
- Operations manager - toolpath management
- Posting the part program
- Working with the Mastercam editor
- DNC part program to machine

Working with Files:

- Importing and modification of non-Mastercam files.
e.g. Solidword, AutoDesk files.

Working with Work Coordinate System (WCS), Views, Planes:

- Setting of WCS and views for parts programs working off different faces of part.

Prerequisites:

- Perspective students must have general computer experience and a basic understanding of the machining process.