2015 COURSE DESCRIPTIONS

(Courses are offered year round; see the VU HTEC CNC Training Center website for schedule)

Vincennes University HTEC CNC Training Center

Website: www.vuhtec.org Phone: 812-888-4159

VU HTEC CNC Mill / Lathe Operator

Description:

This course is designed for those with a good mechanical aptitude that are seeking basic training for an entry level CNC operator position only. Even though repeated machine setups will be required to earn the credential, this course is not designed to certify a setup or a programming person. It is also designed for those currently employed as CNC operators wanting to enhance their knowledge and skills on Haas CNC machines. Programming is not covered in the hands-on portion of this class, but it is provided in the text book, the Haas Programming Workbook, and in the Immerse2Learn online training for reference and for personal study and skill enhancement.

Subjects covered in the textbook include: workplace skills, safety, measurement, math, quality, metals, maintenance, drawings including GD&T, layout, hand tools, saws, offhand grinding, drilling operations, CNC basics, CNC mill setup and operation. The skills covered in the online emulator training include CNC setup and operation.

The hands-on portion involves six 8-hour days over a two week period directly on Haas CNC machining centers and lathes with 1 person to a machine. One week is focused on the CNC mill and the other week is focused on the CNC lathe. Class size is limited to 5 people for the highest quality training possible. The hands-on training includes safety, machine pre-checks, coolant management, startup, tooling and workholding, setting work and tool offsets manually and with Renishaw probe routines, adjusting wear offsets, machining parts and inspecting results with precision measuring tools and probes.

The final day of the class each week is dedicated to completing the requirements for the NIMS Credentialing Achievement Record, for review, and for taking the theory test to complete the credential.

Textbook:

Precision Machining Technology; Hoffman, Hopewell, Janes, Sharp; Delmar-Cengage Learning
ISBN-13:978-1-4354-4767-7

Workbooks:

Haas Automation CNC Mill Programming Workbook (supplemental)
Haas Automation CNC Lathe Programming Workbook (supplemental)

Online Curriculum:

Immerse2Learn:
- Basic Haas CNC Milling Machine Setup
- Basic Haas CNC Lathe Setup
- Basic Haas CNC Milling Machine Programming (supplemental)
- Basic Haas CNC Lathe Programming (supplemental)

Summary:

30-40 hours textbook and online (begin at least 4 weeks prior to on-site training)
6 eight hour days over two week's on-site working on Haas CNC machining centers
Cost: $3,250

NIMS Credentials:

NIMS CNC Mill Operator Level 1
NIMS CNC Lathe Operator Level 1
NIMS Measurement, Materials and Safety

VU HTEC CNC Milling: Programming, Setup & Operations Level 1

Description:

This course is designed for those with some basic CNC mill G-code programming, setup and operations experience. Subjects covered in the textbook include: workplace skills, safety, measurement, math, quality, metals, maintenance, drawings including GD&T, layout, hand tools, saws, offhand grinding, drilling operations, CNC basics, CNC mill setup and operation, basic CNC mill G-code programming, and an introduction to CAD/CAM.

The skills covered in the online emulator training include CNC mill setup, programming and operation. The Haas CNC Mill Workbook also covers G&M-Code programming in much detail.

The hands-on portion involves three 8-hour days directly on Haas CNC machining centers with 1 person to a machine. Class size is limited to 5 people for the highest quality training possible. The hands-on training includes safety, machine pre-checks, coolant management, startup, tooling and workholding, setting work and tool offsets manually and with Renishaw probe routines, basic G-code programming, inputting and editing programs, adjusting wear offsets, machining parts and inspecting results with precision measuring tools and probes.

Projects:

The Mill level 1 work pieces are aluminum; carbide endmills and drills are high performance tools; parts are dimensioned in inch; machines are run in the inch system; cutter diameter compensation is used.

The final day of the class is dedicated to the NIMS Performance Test. The completed part is sent out to the NIMS Met-Tec Committee members for inspection. If the part is 100% within specification then VU will provide instructions for taking the theory test to complete the credential.

Textbook:

Precision Machining Technology; Hoffman, Hopewell, Janes, Sharp; Delmar-Cengage Learning
ISBN-13:978-1-4354-4767-7

Workbook:

Haas Automation CNC Mill Programming Workbook

Online Curriculum:

Immerse2Learn: Basic Haas CNC Milling Machine Setup
Basic Haas CNC Milling Machine Programming

Summary:

30-40 hours textbook and online (begin at least 4 weeks prior to on-site training)
3 eight hour days on-site working on Haas CNC machining centers
Cost: $1350, plus $75 for textbook, $40 NIMS registration (1 time), $28 for each NIMS test
½ off tuition, $675 Gene Haas Foundation scholarships are available for HTEC Network teachers
Note: There will be a $20.00 charge if the student wishes to have the NIMS project mailed to them.

NIMS Credentials:

NIMS CNC Milling: Programming Setup & Operations Level 1
NIMS Measurement, Materials and Safety (optional)

VU HTEC CNC Turning: Programming, Setup & Operations Level 1

Description:

This course is designed for those with some basic CNC lathe G-code programming, setup and operations experience. Subjects covered in the textbook include: workplace skills, safety, measurement, math, quality, metals, maintenance, drawings including GD&T, layout, hand tools, saws, offhand grinding, drilling operations, CNC basics, CNC lathe setup and operation, basic CNC mill G-code programming, and an introduction to CAD/CAM.

The skills covered in the online emulator training include CNC lathe setup, programming and operation. The Haas Lathe Programming Workbook also covers G&M-Code programming in much detail.

The hands-on portion involves three 8-hour days directly on Haas CNC lathes with 1 person to a machine. Class size is limited to 5 people for the highest quality training possible. The hands-on training includes safety, machine pre-checks, coolant management, startup, tooling and workholding, setting work and tool offsets manually and with Renishaw touch probe, basic G-code programming, inputting and editing programs, adjusting wear offsets, machining parts and inspecting results with precision measuring tools.

Projects:

The Lathe level 1 work pieces are aluminum; carbide inserted turning tools are used; parts are dimensioned in inch; machines are run in the inch system; tool nose compensation and type 1 turning cycles are used.

The final day of the class is dedicated to the NIMS Performance Test. The completed part is sent out to the NIMS Met-Tec Committee members for inspection. If the part is 100% within spec. then VU will provide instructions for taking the theory test to complete the credential.

Textbook:

Precision Machining Technology; Hoffman, Hopewell, Janes, Sharp; Delmar-Cengage Learning
ISBN-13:978-1-4354-4767-7

Workbook:

Haas Automation CNC Lathe Programming Workbook

Online Curriculum:

Immerse2Learn: Basic Haas CNC Lathe Setup
Basic Haas CNC Lathe Programming

Summary:

30-40 hours textbook and online (begin at least 4 weeks prior to on-site training)
3 eight hour days on-site working on Haas CNC machining centers
NIMS CNC Turning: Programming Setup & Operations Level 1 Credential
Cost: $1350, plus $75 for textbook, $40 NIMS registration (1 time), $28 for each NIMS test
½ off tuition, $675 Gene Haas Foundation scholarships are available for HTEC Network teachers
Note: There will be a $20.00 charge if the student wishes to have the NIMS project mailed to them.

NIMS Credentials:

NIMS CNC Turning: Programming Setup & Operations Level 1
NIMS Measurement, Materials and Safety (optional)

VU HTEC CNC Milling: Programming, Setup & Operations Level 2

Description:
This course is designed for those that have completed the VU HTEC CNC Milling: Programming, Setup & Operations Level 1 course and that have earned the NIMS CNC Milling: Programming Setup & Operations Level 1 Credential. (In some cases this prerequisite can be waived, contact the HTEC CNC Training Center.)

The hands-on portion involves three 8-hour days directly on Haas CNC machining centers with 1 person to a machine. Class size is limited to 5 people for the highest quality training possible. The hands-on training includes safety, machine pre-checks, coolant management, startup, tooling and workholding, setting work and tool offsets manually and with Renishaw probe routines, more advanced G-code programming, inputting and editing programs, adjusting wear offsets, machining parts and inspecting results with precision measuring tools and probes.

Projects:
The Mill level 2 work pieces are steel; the parts are dimensioned in metric and the machine is run in the metric system; the cutting tools are high performance tools and they are metric; tool diameter compensation is used; geometry and trig calculations are required; chamfer milling calculations and operations; drilling and tapping; and the use of sub-programming.

The final day of the class is dedicated to the NIMS Performance Test. The completed part is sent out to the NIMS Met-Tec Committee members for inspection. If the part is 100% within specification then VU will provide instructions for taking the theory test to complete the credential.

Textbook: (
Precision Machining Technology; Hoffman, Hopewell, Janes, Sharp; Delmar-Cengage Learning
ISBN-13:978-1-4354-4767-7

Workbook:
Haas Automation CNC Mill Programming Workbook

Online Curriculum:
Immerse2Learn: Basic Haas CNC Milling Machine Setup
Basic Haas CNC Milling Machine Programming

Summary:
30-40 hours textbook and online (begin at least 4 weeks prior to on-site training)
3 eight hour days on-site working on Haas CNC machining centers
Cost: $1350, plus $75 for textbook, $40 NIMS registration (1 time), $40 NIMS test
½ off tuition, $675 Gene Haas Foundation scholarships are available for HTEC Network teachers
Note: There will be a $20.00 charge if the student wishes to have the NIMS project mailed to them.

NIMS Credentials:
NIMS CNC Milling: Programming Setup & Operations Level 2

VU HTEC CNC Turning: Programming, Setup & Operations Level 2

Description:

This course is designed for those that have completed the VU HTEC CNC Turning: Programming, Setup & Operations Level 1 course and that have earned the NIMS CNC Turning: Programming Setup & Operations Level 1 Credential. (In some cases this prerequisite can be waived, contact the HTEC CNC Training Center.)

The hands-on portion involves three 8-hour days directly on Haas CNC lathes with 1 person to a machine. Class size is limited to 5 people for the highest quality training possible. The hands-on training includes safety, machine pre-checks, coolant management, startup, tooling and workholding, setting work and tool offsets manually and with Renishaw touch probes, more advanced G-code programming, inputting and editing programs, adjusting wear offsets, machining parts and inspecting results with precision measuring tools.

Projects:

The lathe level 2 work pieces are steel; carbide inserted turning tools are used; parts are dimensioned in inch; machines are run in the inch system; tool nose compensation and type 2 turning cycles are used; geometry and trig calculations are required.

The final day of the class is dedicated to the NIMS Performance Test. The completed part is sent out to the NIMS Met-Tec Committee members for inspection. If the part is 100% within spec. then VU will provide instructions for taking the theory test to complete the credential.

Textbook:

Precision Machining Technology; Hoffman, Hopewell, Janes, Sharp; Delmar-Cengage Learning
ISBN-13:978-1-4354-4767-7

Workbook:

Haas Automation CNC Lathe Programming Workbook

Online Curriculum:

Immerse2Learn: Basic Haas CNC Lathe Setup
Basic Haas CNC Lathe Programming

Summary:

30-40 hours textbook and online (begin at least 4 weeks prior to on-site training)
3 eight hour days on-site working on Haas CNC machining centers
NIMS CNC Turning: Programming Setup & Operations Level 1 Credential
Cost: $1350, plus $75 for textbook, $40 NIMS registration (1 time), $40 NIMS test
½ off tuition, $675 Gene Haas Foundation scholarships are available for HTEC Network teachers
Note: There will be a $20.00 charge if the student wishes to have the NIMS project mailed to them.

Credential:

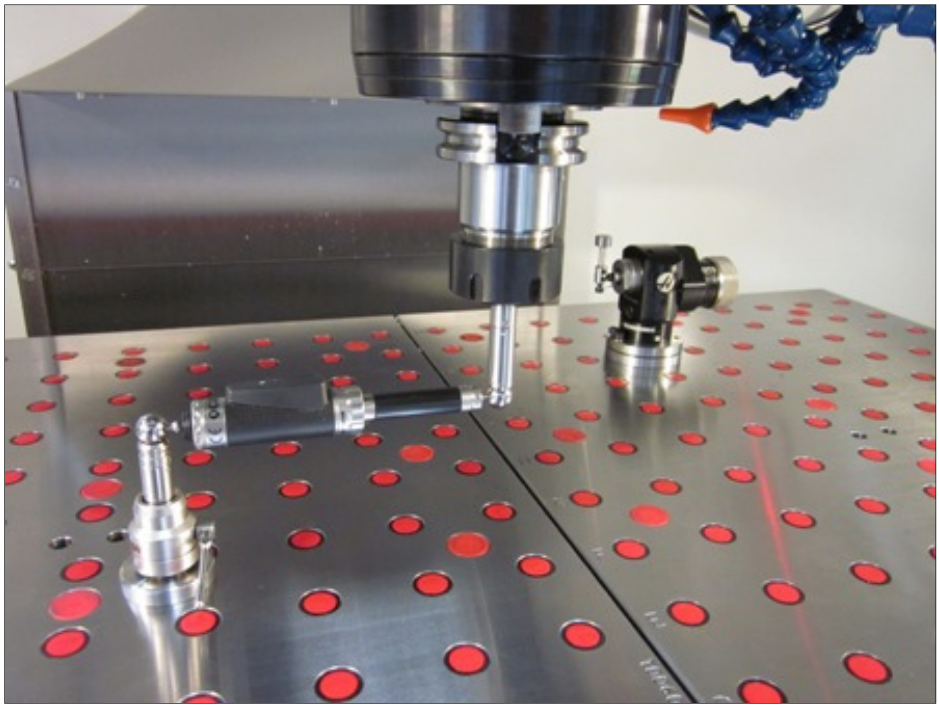
NIMS CNC Turning: Programming Setup & Operations Level 2 Credential

VU HTEC – Renishaw "Getting Started with Ballbar Testing" Course

Description:

This course is designed for new and potential Renishaw QC20-W ballbar users. It explains why ballbar testing is needed along with the effects of temperature on quality. It provides an overview of ballbar hardware and software, along with how to configure and calibrate the system. The course provides hands-on experience performing diagnostic tests on CNC machine tools. It includes an understanding of ballbar system components; planning and writing tests; creating test templates, part program generation: running tests, analyzing test data; performing volumetric ballbar tests; and lots of hands-on experience. The 21 degrees of freedom of a three axis machine tool will be explored. The tests performed will measure potential deviations and errors in the following areas:

- Test errors
- Straightness
- Backlash
- Cyclic error
- Lateral play
- Reversal spikes
- Scaling errors
- Servo mismatch
- Squareness
- Stick slip
- Vibration



Textbook: Renishaw QC20-W Ballbar training course manual (provided)
DVD: Getting Started with QC20-W Ballbar (provided)

Summary:

1 eight hour day onsite training to use a Renishaw QC20-W Ballbar to run diagnostic tests and analyze data
Cost: $495 per person
Registration is through Renishaw, Inc., contact Ann Molloy at (847) 286-9953 or ann.molloy@renishaw.com
Ballbar classes are offered once per month at Vincennes University

VU HTEC Mastercam Mill Design and Toolpaths for Certification

Description:

The Mastercam University® Mill Design & Toolpaths course takes Mastercam students from square one to successfully programming parts to be machined on a CNC mill. Each of the five modules builds on the last and offers a module review to help you advance. The first module introduces the Mastercam environment and basic general functions. In the second Design module, wireframe and solid modeling functions are taught and then applied through a series of specific applications-based exercises. The third and fourth modules focus on part setup and demonstrate fundamental Mastercam toolpaths, including 2D high speed machining. The final module introduces Feature Based Machining (FBM) used to automatically detect and generate all milling and drilling operations for solid model features.

The hands-on portion of this course is designed to work hand-in-hand with Mastercam University's Mill Design and Toolpaths online course. Subjects covered include 3D wireframe construction, levels, multiple construction planes and WCS's, solid modeling, stock setup and setup sheets, operations manager, speeds and feeds, print reading, machine definition, drilling, pocketing, slotting, engraving, and contour milling toolpaths, and post processing. The class reinforces the skills learned in the online training by using these techniques to design and program practical applications for the CNC machining centers. The programmed toolpaths are actually run on the CNC machines in the lab. **The final day of the class is dedicated to the Certification Test.**

Projects:

There are many excellent programming projects contained in the Mastercam University Mill Design & Toolpaths curriculum that will prepare the student for the on-site portion of the training and the certification test. Several of the programming operations generated in the classroom during the hands-on portion of the training will be taken out to the lab and run on the CNC machines to demonstrate how the Mastercam parameters affect the outcome of the program and the machining of the actual part. The test part is the six sided Mastercam Certification (Mill-CPgmM) project.

Textbook:

None required

Workbook:

None required

Online Curriculum:

Mastercam University Mill Design & Toolpaths

Summary:

60 hours of online Mastercam University training
3 eight hour days on-site hands-on training
Cost: $1350, plus $249 for the Mastercam University subscription
The online curriculum remains active for 12 weeks
½ off tuition Gene Haas Foundation scholarships are available for HTEC Network teachers

VU HTEC Mastercam Mill Design and Toolpaths

Description:

The Mastercam University® Mill Design & Toolpaths course takes Mastercam students from square one to successfully programming parts to be machined on a CNC mill. Each of the five modules builds on the last and offers a module review to help you advance. The first module introduces the Mastercam environment and basic general functions. In the second Design module, wireframe and solid modeling functions are taught and then applied through a series of specific applications-based exercises. The third and fourth modules focus on part setup and demonstrate fundamental Mastercam toolpaths, including 2D high speed machining. The final module introduces Feature Based Machining (FBM) used to automatically detect and generate all milling and drilling operations for solid model features.

The hands-on portion of this course is designed to work hand-in-hand with Mastercam University's Mill Design and Toolpaths online course. Subjects covered include 3D wireframe construction, levels, multiple construction planes and WCS's, solid modeling, stock setup and setup sheets, operations manager, speeds and feeds, print reading, machine definition, drilling, pocketing, slotting, engraving, and contour milling toolpaths, and post processing. The class reinforces the skills learned in the online training by using these techniques to design and program practical applications for the CNC machining centers. The programmed toolpaths are actually run on the CNC machines in the lab.

Projects:

There are many excellent programming projects contained in the Mastercam University Mill Design & Toolpaths curriculum that will prepare the student for the on-site portion of the training. Several of the programming operations generated in the classroom during the hands-on portion of the training will be taken out to the lab and run on the CNC machines to demonstrate how the Mastercam parameters affect the outcome of the program and the machining of the actual part.

Textbook:

None required

Workbook:

None required

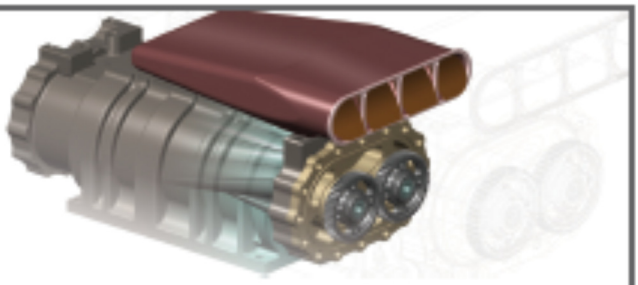
Online Curriculum:

Mastercam University Mill Design & Toolpaths

Summary:

40-60 hours of online Mastercam University training
3 eight hour days on-site hands-on training
Cost: $1350, plus $249 for the Mastercam University subscription
The online curriculum remains active for 12 weeks
½ off tuition Gene Haas Foundation scholarships are available for HTEC Network teachers

Mastercam U®



Mill Design and Toolpaths

Course Description

The Mastercam University® Mill Design & Toolpaths course takes Mastercam students from square one to successfully programming parts to be machined on a CNC mill. Each of the five modules build on the last and offer a module review to help you advance. The first module introduces the Mastercam environment and basic general functions. In the second Design module, wireframe and solid modeling functions are taught and then applied through a series of specific applications-based exercises. The third and fourth modules focus on part setup and demonstrate fundamental Mastercam toolpaths, including 2D high speed machining. The final module introduces Feature Based Machining (FBM) used to automatically detect and generate all milling and drilling operations for solid model features.

Course Objectives

Upon completion, you should be able to demonstrate:

Geometry
- Using levels, colors
- Create and apply multiple Work Coordinate Systems (WCS) and construction planes
- Using 2D and 3D construction methods
- 3D wireframe creation
- Geometry selection (pre/post selection)
- Modifying current geometry (Trim, Fillet, Color, Level, Group, and Attributes)
- Diameter/Radius part creation
- Geometry transformation (Scale, Rotate, etc.)

Solids
- Extrude, Extrude Cut, Solids Manager, and creation of curves on solids

Toolpath
- Efficient toolpath creation
- Example parts that reflect holding
- Define a new tool definition from scratch
- Depth of cut and feed rate based on tool and materials
- Appropriate toolpath operation for cutting application
- Saving custom tools to a level
- Setup sheets that communicate what the program is cutting, with order of cuts
- Set common and individual operation defaults
- Multiple Pocketing depths, Island, Face, Island facing
- Slot milling
- Contour toolpath, Ramp, and Multiple depth
- Importing and exporting operations
- Creating toolpath groups and machine groups
- Backplot, Quick Verify
- STL file creation and STL compare
- Stock setup
- Setting Machine and Control Definition
- Material library applied in Control Definition
- Drill toolpaths
- Posting G-Code
- Circle mill, Thread mill, and Slot mill toolpaths

2D High Speed Toolpaths (HST) and Feature Based Machining (FBM)
- Core mill, Peel mill, Blend mill, Area mill, Rest mill, and Dynamic mill techniques
- 2D HST applications
- FBM Mill and Drill
- Feature detection

Operations Manager
- Configuration
- Cut and paste geometry
- Display settings
- Tool offset registry settings
- Maximum RPM and feed rate
- Renumber tools

VU HTEC Advanced Mastercam Mill Design and Toolpaths

Description:

The Mastercam University® Advanced Mill Design & Toolpaths course covers many different ways to create and machine 3D geometry, including an extensive 4th axis module. The first module teaches many different surface creation methods fundamental to a well-rounded CAD/CAM background. The second module is an overview of 3D machining strategies. In this lesson you apply many different machining strategies to similar geometry, seeing firsthand the strengths as well as best application of each of the 3D high speed toolpaths. In the third module you put all of the previous lessons into practice by creating programs similar to parts in your shop, including fixturing. The fourth module covers 4-axis machining with lessons on axis substitution, axis positioning, and radial and rotary axis machining. This module prepares you for 5-axis machining. The onsite portion of the class reinforces the skills learned in the online training by using these techniques to design and program practical applications for the CNC mills. The programmed toolpaths are actually run on the CNC machines in the lab.

Projects:

There are many excellent programming projects contained in the Mastercam University Advanced Mill Design & Toolpaths curriculum that will prepare the student for the on-site portion of the training. Setting up a 4-axis rotary will be covered in detail, including finding the center of rotation. Several of the programming operations generated in the classroom during the hands-on portion of the training will be taken out to the lab and run on the CNC machines to demonstrate how the Mastercam parameters affect the outcome of the program and the machining of the actual part.

Textbook:

None required

Workbook:

None required

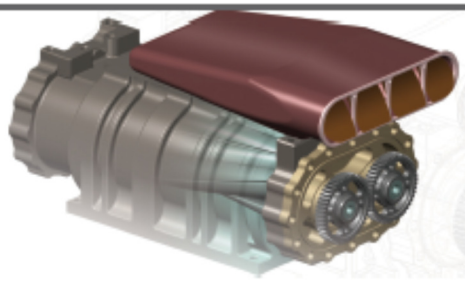
Online Curriculum:

Mastercam University Advanced Mill Design & Toolpaths

Summary:

40-60 hours of online Mastercam University training
3 eight hour days on-site hands-on training
Cost: $1350, plus $249 for the Mastercam University subscription
The online curriculum remains active for 12 weeks
½ off tuition Gene Haas Foundation scholarships are available for HTEC Network teachers

Mastercam U®



Advanced Mill Design and Toolpaths

Course Description

The Mastercam University® Advanced Mill Design & Toolpaths course covers many different ways to create and machine 3D geometry, including an extensive 4th axis module. The first module teaches many different surface creation methods fundamental to a well-rounded CAD/CAM background. The second module is an overview of 3D machining strategies. In this lesson you apply many different machining strategies to similar geometry, seeing first hand the strengths as well as best application of each of the 3D high speed toolpaths. In the third module you put all of the previous lessons into practice by creating programs similar to parts in your shop, including fixturing. The fourth module covers 4-axis machining with lessons on axis substitution, axis positioning, and radial and rotary axis machining. This module prepares you for 5-axis machining.

Course Objectives

Upon completion, you should be able to demonstrate skill in the following areas:

Advanced Wireframe Creation
- Creating tangent vectors relative to a surface edge or point on a surface
- 2D and 3D spline creation and manipulation
- Multiple construction planes
- User-defined construction planes
- 2D and 3D construction methods

Surface/Solid Modeling
- Surface Normal
- Surface geometry creation
- Application of all Mastercam surface creation functions
- Projecting geometry onto surfaces
- Advanced solids

Application of Mastercam Solid Creation
- Fillet/Chamfer
- Boolean operations
- Loft solid
- Swept solid
- Shell solid
- Thin Wall solids
- Trim a solid with a surface

Toolpath Overview
- Overview high speed toolpaths
- Core Roughing toolpath
- Setup for the core
- Horizontal Flat Boundary toolpath
- Rest Roughing toolpath
- Raster toolpath
- Scallop toolpath
- Waterline toolpath
- Spiral toolpath
- Radial toolpath
- Pencil toolpath
- Translate a Raster boundary toolpath
- Setup for the core
- Blend toolpath
- Rough Cavity toolpath

Toolpaths
- Efficient toolpath creation
- Scale the model for Shrink
- Creating core geometry, Boolean remove
- Set WCS, stock size, tool list
- Area Rough, Rest Rough
- Backplot, Quick Verify
- STL file creation and STL compare
- Stock setup
- Spiral finishing
- Selecting distance 2D vs. 3D

Toolpaths (continued)
- Waterline
- Pocket contain boundary
- Z depth/height control
- Toolpath transformation
- Ghosting operations
- Select core levels
- Determine stock boundary
- Core vs. Cavity machining strategies
- Selecting geometry by color
- Scallop toolpath
- Flowline toolpath
- Blend toolpath

4th Axis/Rotary
- Axis substitution
- Unwrapping the cylinder
- Defining the cylinder for stock
- Drilling the radial holes
- Axis Indexing
- Creating WCS planes form a solid
- Using the WCS offset for depth
- 4th axis machining
- Holding the part
- Creating solids for fixturing
- Multiaxis rotary 4th axis toolpath
- Radial cut
- Lead lag angle
- Roughing with an axial cut

VU HTEC Mastercam Multiaxis Toolpaths

Description:

The Mastercam University® Multiaxis Curve / Drill / Circle Mill course provides students with a strong foundation in 5-axis CNC programming. The class starts out with an extensive overview of axis combinations, rotary points, and zero positions as they relate to Mastercam. It then builds on how to control tool axis vectors using different geometric entities. The examples explain the many ways to control how the tool approaches and machines with a controlled 5-axis toolpath. This course also covers 5-axis positioning, drilling, trimming, circle milling, and machine simulation. The 4th Axis Rotary Machining section will serve as a review for those that have previously enrolled in the Advanced Mill Design and Toolpaths course. The onsite portion of the class reinforces the skills learned in the online training by using these techniques to design and program practical applications for the CNC mills. The programmed toolpaths are actually run on the CNC machines in the lab.

Projects:

There are many excellent programming projects contained in the Mastercam University® Multiaxis Curve / Drill / Circle Mill curriculum that will prepare the student for the on-site portion of the training. Setting up a 5-axis trunnion will be covered in detail, including finding the center of rotation. The hands-on portion will also cover basic full 5-axis simultaneous toolpaths. Several of the programming operations generated in the classroom during the hands-on portion of the training will be taken out to the lab and run on the CNC machines to demonstrate how the Mastercam parameters affect the outcome of the program and the machining of the actual part.

Textbook:

None required

Workbook:

None required

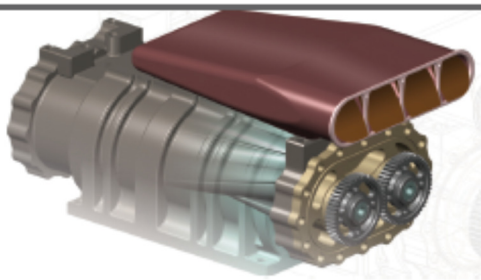
Online Curriculum:

Mastercam University® Multiaxis Curve / Drill / Circle Mill

Summary:

40-60 hours of online Mastercam University training
3 eight hour days on-site hands-on training
Cost: $1350, plus $249 for the Mastercam University subscription
The online curriculum remains active for 12 weeks
½ off tuition Gene Haas Foundation scholarships are available for HTEC Network teachers

Mastercam U®



Multiaxis (Curve / Drill / Circle Mill)

Course Description

The Mastercam University® Multiaxis Curve / Drill / Circle Mill course provides students with a strong foundation in 5-axis CNC programming. The class starts out with an extensive overview of axis combinations, rotary points, and zero positions as they relate to Mastercam. It then builds on how to control tool axis vectors using different geometric entities. The examples explain the many ways to control how the tool approaches and machines with a controlled 5-axis toolpath. This course also covers 5-axis positioning, drilling, trimming, circle milling, and machine simulation. The 4th Axis Rotary Machining section will serve as an review for those that have previously enrolled in the Advanced Mill Design and Toolpaths course.

Course Objectives

Upon completion, you should be able to demonstrate:

4th Axis Rotary Machining

- Axis substitution
- Unwrapping the cylinder
- Defining the cylinder for stock
- Drilling the radial holes
- Axis indexing
- Creating WCS planes from a solid
- Using the WCS offset for depth
- 4th axis machining
- Holding the part
- Creating solids for fixturing
- Creating a Multiaxis rotary 4th axis toolpath
- Radial cutting
- Defining lead / lag angle
- Roughing with an axial cut

5-Axis Toolpath

- Introduction to multiaxis machining concepts
- Overview of common 5-axis machines
- Zero position on a 5-axis machine
- Cut pattern
- Tool axis control

5-Axis Toolpath (continued)

- Geometry creation for tool axis control, including:
  - Points
  - Vectors (lines)
  - Chains
  - Planes
  - Surfaces
- Collision control, including Check Surfaces
- Machine simulation
- Mastercam Multiaxis toolpaths
  - Curve, trimming a part (face shield)
  - Drill, normal to a face, vector (lines), plane, and points
  - Circle Mill, geometry creation

Strongly Recommended Prerequisites

- Print reading with multiple views
- CNC machining experience
- CNC set-up experience
- Intermediate to advanced Mastercam experience

VU HTEC Mastercam Lathe Design and Toolpaths

Description:

The Mastercam University® Lathe Design & Toolpaths course takes Mastercam students from square one to successfully programming parts to be machined on a CNC lathe. As you program parts you learn to store your work and knowledge. This approach helps you to save time by building on proven techniques and toolpaths. The first module covers how to create wireframe and solid lathe specific geometry in Mastercam. The second module shows how to set up a lathe file from datum, to managing toolpath libraries, to verifying toolpaths. The third module provides extensive coverage of lathe toolpaths, including basic C & Y-axis toolpaths. The onsite portion of the class reinforces the skills learned in the online training by using these techniques to design and program practical applications for the CNC lathes. The programmed toolpaths are actually run on the CNC machines in the lab.

Projects:

There are many excellent programming projects contained in the Mastercam University® Lathe Design & Toolpaths curriculum that will prepare the student for the on-site portion of the training. Several of the programming operations generated in the classroom during the hands-on portion of the training will be taken out to the lab and run on the CNC machines to demonstrate how the Mastercam parameters affect the outcome of the program and the machining of the actual part.

Textbook:

None required

Workbook:

None required

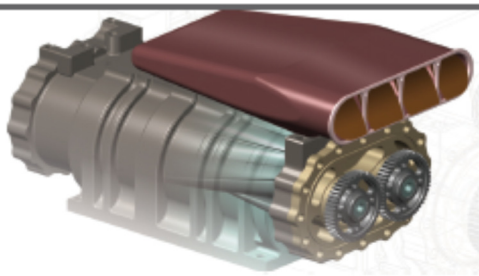
Online Curriculum:

Mastercam University® Lathe Design & Toolpaths

Summary:

40-60 hours of online Mastercam University training
3 eight hour days on-site hands-on training
Cost: $1350, plus $249 for the Mastercam University subscription
The online curriculum remains active for 12 weeks
½ off tuition Gene Haas Foundation scholarships are available for HTEC Network teachers

Mastercam U®



Lathe Design and Toolpaths

Course Description

The Mastercam University® Lathe Design & Toolpaths course takes Mastercam students from square one to successfully programming parts to be machined on a CNC lathe. As you program parts you learn to store your work and knowledge. This approach helps you to save time by building on proven techniques and toolpaths. The first module covers how to create wireframe and solid lathe specific geometry in Mastercam. The second module shows how to set up a lathe file from datum, to managing toolpath libraries, to verifying toolpaths. The third module provides extensive coverage of lathe toolpaths.

Course Objectives

Upon completion, you should be able to demonstrate:

Geometry
- Use of levels, colors, groups
- Lathe construction planes
- Use of 2D and 3D construction methods
- Modification of current geometry (Trim, Fillet, Color, Level, Group, and Attributes)
- Geometry selection (pre/post selection)
- Creation of curves on solids
- Diameter/Radius part creation
- Geometry transformation (Scale, Rotate, etc.)

Solids
- Extrude, Primitive, Fillet, Revolve

Toolpath
- Efficient toolpath creation
- Use of holding in example parts
- Create a tool definition from scratch
- Depth of cut and feed rate based on tool and materials
- Appropriate toolpath operation for cutting application
- Saving custom tools to a level
- Stock setup
- Setup sheet creation including documentation of cutting with order of cuts
- Internal and external operations for all Mastercam Lathe toolpaths
- Importing and exporting operations
- Toolpath group creation
- Machine group creation
- Backplot
- Quick Verify
- Verify section
- STL file creation and STL compare
- Plunge Turn toolpath

Live Toolpaths
- C-axis toolpath overview
- C-axis Face Mill
- C-axis Face Drill
- C-axis Cross Milling
- Y-axis Contour

Operations Manager
- Configuration
- Geometry cut and paste
- Use of display settings
- Use of tool offset registry settings
- Use of maximum RPM and feed rate
- Number of tools in the turret
- Understanding of RPM/FPR

Strongly Recommended Prerequisites
- Print reading with multiple views
- CNC machining experience
- CNC setup experience
- Algebra and trigonometry
- Mastercam experience