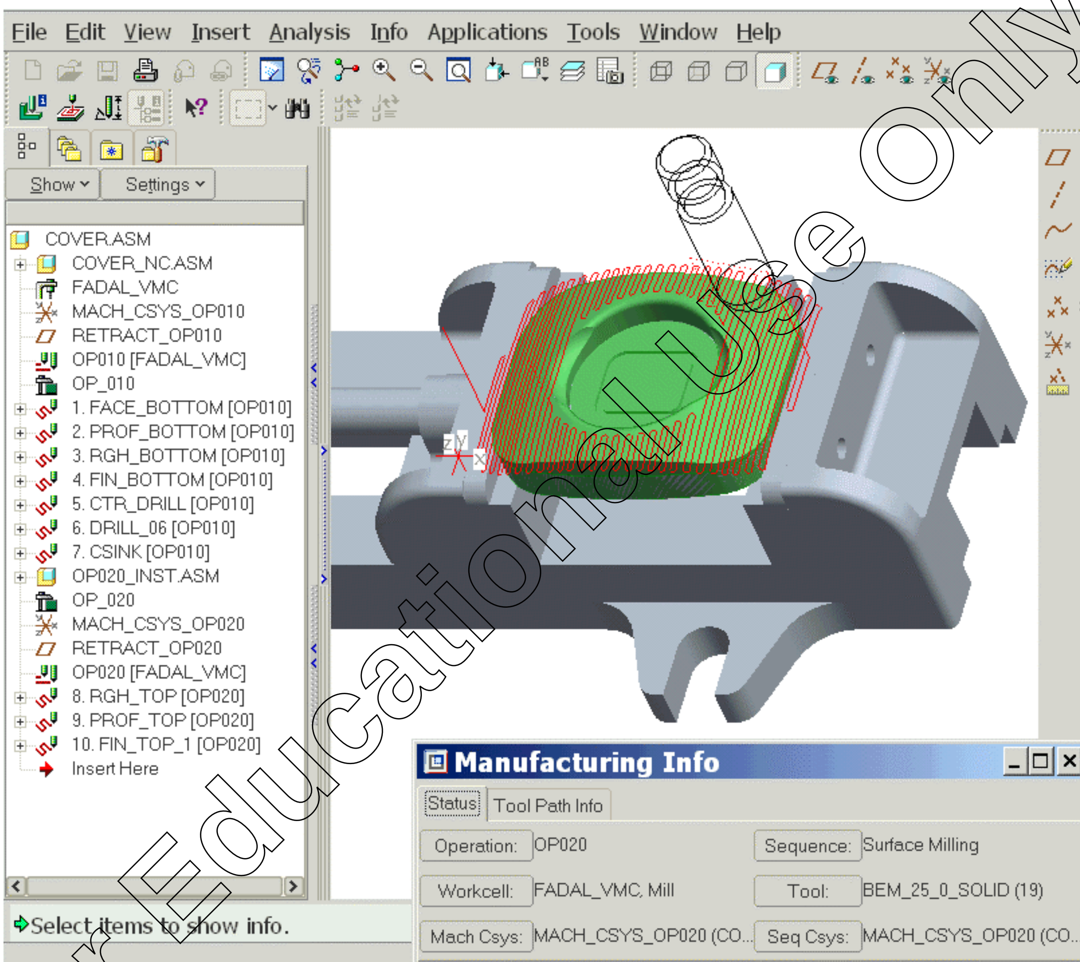
PTC

PTC Global Services



Creating Milling Sequences with Pro/ENGINEER Wildfire

T979-330-02

Copyright

Creating Milling Sequences with Pro/ENGINEER Wildfire

Copyright

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

PRINTING HISTORY

Document No.	Date	Description
T979-330-01	02/05/03	First Printing of **Creating Milling Sequences with Pro/ENGINEER Wildfire**.
T979-330-02	05/12/03	Second Printing of **Creating Milling Sequences with Pro/ENGINEER Wildfire**.

Order Number DT-979-330-EN
Printed in the U.S.A

PTC Telephone and Fax Numbers

Education Services Registration in North America

Tel: (888) 782-3773

Fax: (781) 370-5553

Technical Support (Monday - Friday)

Tel: (800) 477-6435 (U.S.)

(781) 370-5332 or (781) 370-5523 (outside U.S.)

Fax: (781) 370-5650

License Management

Tel: (800) 216-8945 (U.S.)

(781) 370-5559 (outside U.S.)

Fax: (781) 370-5795

Contracts

Tel: (800) 791-9966 (U.S.)

(781) 370-5700 (outside U.S.)

In addition, you can access the PTC Web site at ***www.ptc.com***. Our Web site contains the latest training schedules, registration information, directions to training facilities, and course descriptions. You can also find general information about PTC, Pro/ENGINEER, Consulting Services, Customer Support, and Pro/PARTNERS.

Precision Learning

THE PRECISION LEARNING METHODOLOGY

PTC Global Services is dedicated to continually providing the student with an effective, comprehensive learning experience. Toward this goal, PTC developed Precision Learning, which matches the *right* training to the *right* people at the *right* time using the *right* method.

Precision Learning is based on a three-stage, **Learn—Assess—Improve** methodology.

Stage 1: LEARN

The student attends a PTC training course, including any:

- Instructor-led training course (ILT) at a PTC training center.
- On-site training course.
- Customized training course.
- Web-based training (WBT) course.

Stage 2: ASSESS

The impact of a training course is assessed using Pro/FICIENCY. Pro/FICIENCY is a Web-based skills assessment and development-planning tool. It is designed to deliver information that will help improve the skills and productivity of the student.

Stage 3: IMPROVE

Pro/FICIENCY enables customers to identify areas for improvement. The training wizard will direct customers to the appropriate class based on their job responsibilities.

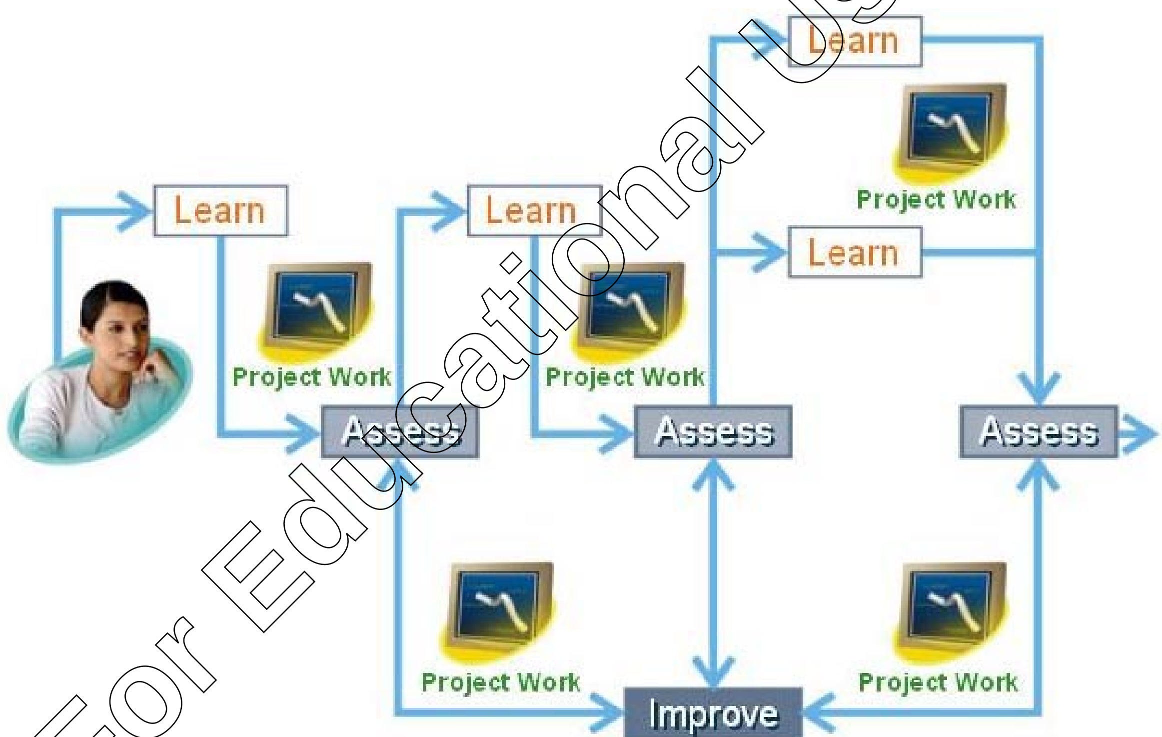
Precision Learning

Customers have access to a range of resources that include:

- Internal and external user groups.
- PTC technical support resources.
- Web-based courses and lessons.

CONTINUOUS IMPROVEMENT

The Precision Learning methodology provides a continuous cycle of knowledge expansion and improvement.



Precision Learning

PRECISION LEARNING IN THE CLASSROOM

The **Learn—Assess—Improve** Precision Learning methodology is also implemented in selected PTC instructor-led courses. Throughout the class, students will take Pro/FICIENCY assessments to evaluate their own comprehension. The group results are also used to identify areas for the instructor to review with the class as a whole. At the end of the class, each student will complete an Education Circuit form. This Education Circuit is the student's *action plan*, identifying topics for improvement, as well as the steps to take in order to enhance the skills in those areas.

The following pages provide a sample Education Circuit action plan, and a blank action plan. Instructions for using the Education Circuit action plan will be discussed in the course.

Precision Learning

EDUCATION CIRCUIT EXAMPLE

The following is an example of a student's Education Circuit at the end of the **Designing Products with Pro/ENGINEER Wildfire** training class.

Pro/FICIENCY Assessment Results

After reviewing assessment results for this course, the following lists the questions I answered incorrectly and need to research further:

Question	Improve Action
Weak and strong dimensions	Practice creating simple features with the desired dimensioning scheme. Web Lesson *Dimensioning Scheme*
Draft Features	See colleague at work for advice and product examples.
Configuration file options	Consult company user group for guidelines.

Class Evaluation Form Topics

After reviewing the questions on the class Evaluation form, the following lists the topics I need to research further:

Objective	Improve Action
Setting up the default view of a part	Practice on simple parts using different sketching planes and reference planes.
Creating sweeps	Web Lesson *Swept Forms*
Resolve Mode	Create some simple models and make them fail.
Resolve Mode	Web lesson *Resolve Mode*

Future Courses

After reviewing the Role Based Training guidelines, the following lists the courses recommended to improve my skills and enhance my job performance:

Next Courses	Next Courses
Advanced Assembly Management with Pro/ENGINEER Wildfire	
Advance Surface Modeling with Pro/ENGINEER Wildfire	

Precision Learning

Pro/FICIENCY Assessment Results

After reviewing assessment results for this course, the following lists the questions I answered incorrectly and need to research further:

Question	Improve Action

Class Evaluation Form Topics

After reviewing the questions on the class Evaluation form, the following lists the topics I need to research further:

Objective	Improve Action

Future Courses

After reviewing the Role Based Training guidelines, the following lists the courses recommended to improve my skills and enhance my job performance:

Next Courses	Next Courses

Courseware Development Software

Creating Milling Sequences with Pro/ENGINEER Wildfire

The following software versions were used in developing this course:

Build Code(s)

Title/Version	Build
Pro/ENGINEER Wildfire for Windows NT 4.0 2000 and XP for Intel based systems	2002380

Third Party Software

Title/ Version	Build
Vericut 5.2	

Trail File(s)

Title/ Link

Training Agenda

Creating Milling Sequences with Pro/ENGINEER Wildfire

Day 1

Introduction

Creating Workpieces

Creating Manufacturing Models

Creating Manufacturing Operations

Day 2

Creating Tools

Using Manufacturing Parameters

Creating Face Milling Sequences

Creating Profile Milling Sequences

Day 3

Creating Volume Milling Sequences

Creating Local Milling Sequences

Creating Surface Milling Sequences

Creating Trajectory Milling Sequences

Day 4

Creating Holemaking Sequences

Creating Roughing and Reroughing Sequences

Post-Processing CL Data

Correcting Toolpath Failure

Day 5

Configuring the Manufacturing Environment

Machining Project

Using Index Tables and Duplicating NC Sequences

Automating Workflow with Mapkeys

Table of Contents

Creating Milling Sequences with Pro/ENGINEER Wildfire

PTC

Module

1

Introduction

Introduction

Prior to using Pro/ENGINEER Wildfire to machine components, it is necessary to understand the overall manufacturing process, and the steps involved in this process. It is also important to understand and learn how to use the basic elements and tools that make up the Pro/ENGINEER Wildfire interface. You can achieve this by opening and reviewing elements within completed manufacturing models. This can include reviewing reference models, machine types, tools, fixtures and completed tool paths.

Note:
This training course has been developed using metric units. The English (or Imperial) unit equivalent is shown in parenthesis after metric dimensions in order to put the dimension into context for students that do not use metric units.

Objectives

After completing this module, you will be able to:

- Describe the Pro/ENGINEER Wildfire manufacturing process.
- Describe the elements that make up completed manufacturing models.
- Navigate the Pro/ENGINEER Wildfire interface.
- Review the elements that make up completed manufacturing models.
- Review completed NC sequences.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

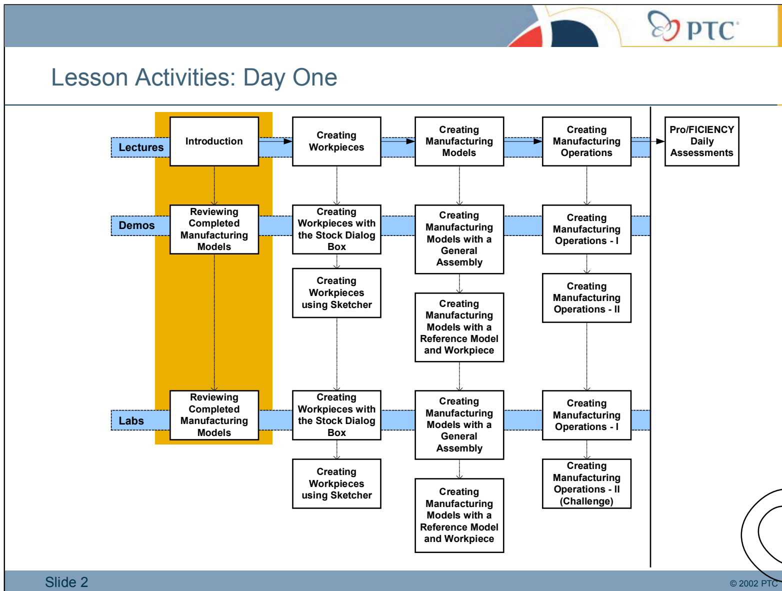
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

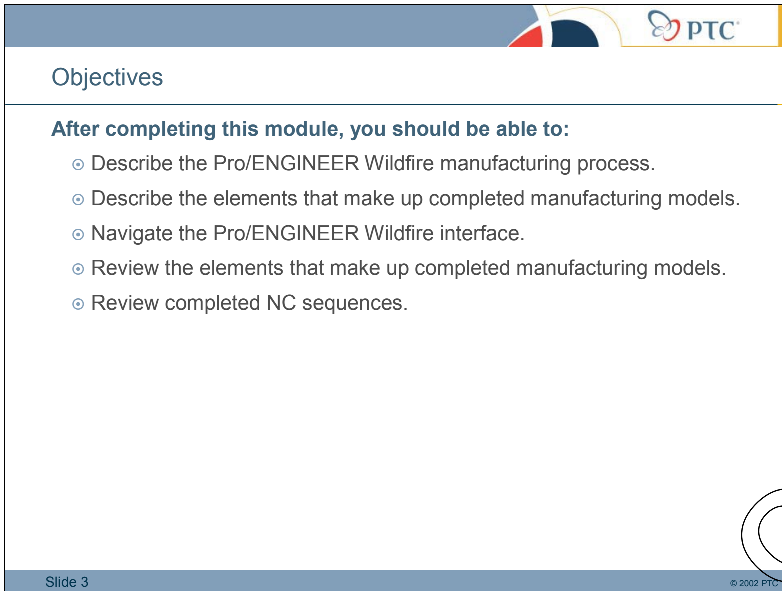
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 25 mins
- Labs : 50 mins
- Total: 1 hr 30 mins

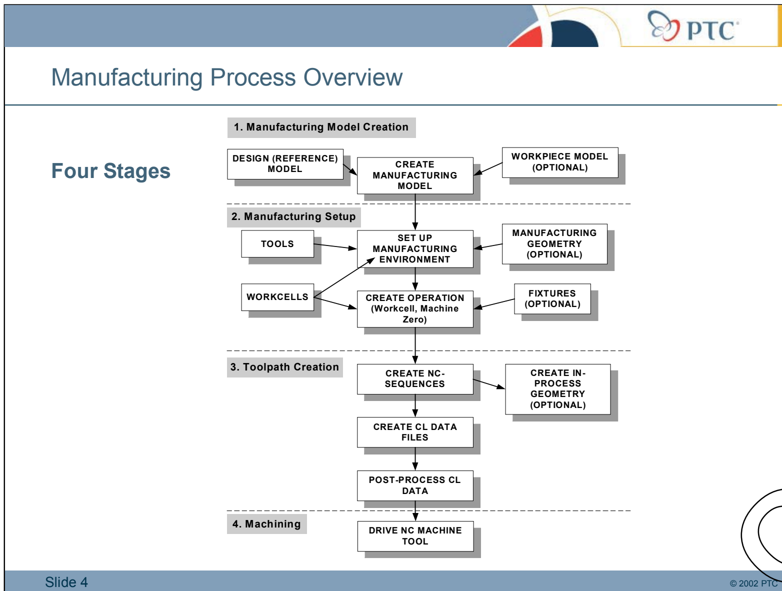


Overview

Prior to using Pro/ENGINEER Wildfire to machine components, it is important to understand the overall manufacturing process, and the steps involved in this process. You can achieve this by opening and reviewing elements within completed manufacturing models. This can include reviewing reference models, machine types, tools, fixtures and completed tool paths.

After successfully completing this module, you will be able to:

- Describe the Pro/ENGINEER Wildfire manufacturing process.
- Describe the elements that make up completed manufacturing models.
- Navigate the Pro/ENGINEER Wildfire interface.
- Review the elements that make up completed manufacturing models.
- Review completed NC sequences.

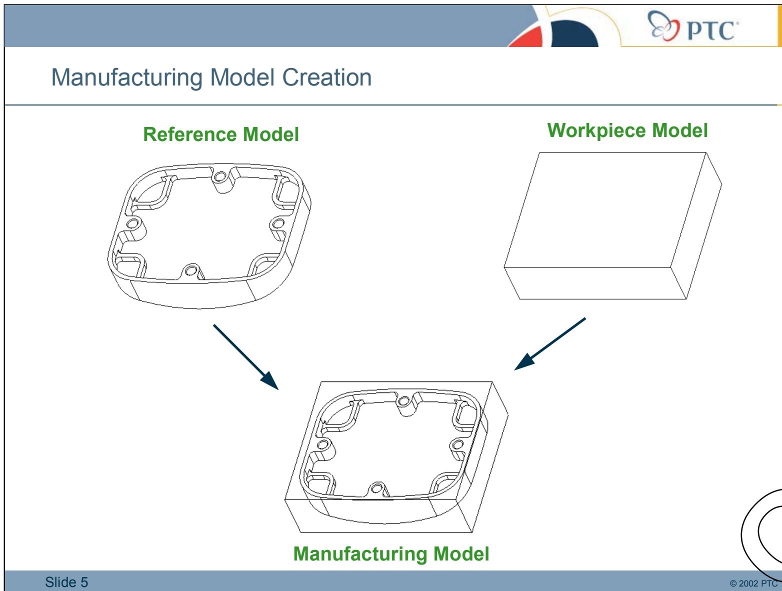


Manufacturing Process Overview

The Pro/ENGINEER Wildfire manufacturing process can be broken into four stages:

1. Stage 1 involves creating manufacturing models using a design model (also known as a reference model) and a workpiece.
2. Stage 2 involves setting up your manufacturing environment this can involve configuring tools, fixtures, machines (known as workcells), and creating manufacturing reference geometry.
3. The next stage involves creating NC sequences (NC programs). Creating NC Sequences involves selecting a cutting tool, selecting geometry on the reference model to machine, (e.g. surfaces for milling holes for drilling), and specifying machining parameters to control how the toolpath is generated. Once NC sequences are finalized you can create Cutter Location (CL) data files, these are then post-processed to produce machine code data (MCD files).
4. The final stage involves using the MCD files to drive machine tools.

This course will cover stages 1 to 3 in this process, finishing with creating machine code data. Stage 4 involves driving machine tools and is therefore beyond the scope of this course.



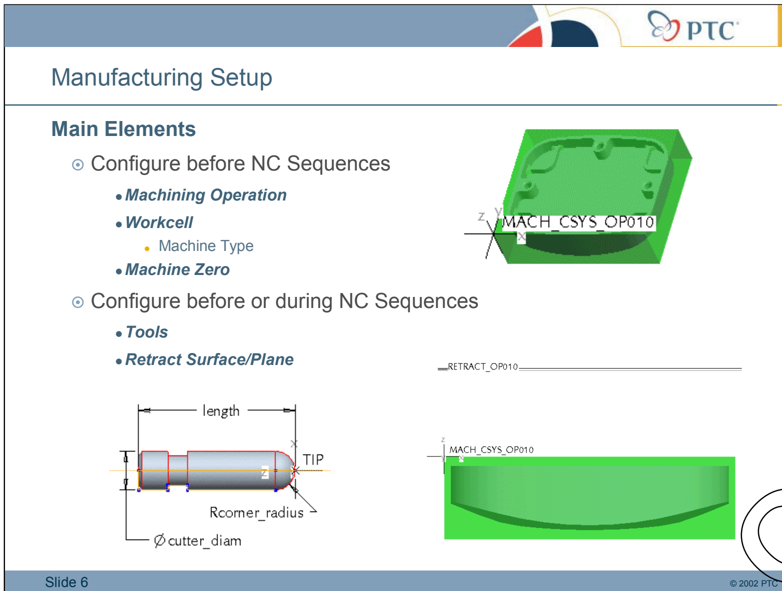
Manufacturing Model Creation

Manufacturing models consist of a reference model (representing the final designed component) and a workpiece assembled together. The workpiece represents the un-machined stock material.

Note to Instructor: Ensure the meaning of all key terminology is noted by students in this slide.

Key Terminology:

- Manufacturing Model
- Reference Model
- Workpiece Model



Manufacturing Setup

The manufacturing environment is made up of a number of elements. Some elements must be configured before creating NC sequences, other elements can be configured either before or during the creation of NC sequences, and some elements are optional.

Here are the main elements:

Configured before creating NC Sequences

- Machining Operations are defined as a series of NC sequences performed at a particular workcell (machine) and referencing a particular coordinate system. For example, configuring a machining operation to machine one side of a component by specifying the machine tool (workcell) to use and the program zero position. Any subsequent NC sequences are then associated with the configured operation.
- Workcells specify machine tools in terms of the type of machine tool and numerous machine tool parameters.
  - For example you could specify a workcell as a 3-axis milling machine with various machine tool parameters such as feed units, maximum spindle speed, and travel limits in the X, Y and Z directions.
- Machine Zero specifies the program zero position in x,y and z on the machine tool.

Configured either before or during creating NC Sequences

- Tools: a tool must be configured for each NC sequence you create.
- Retract Surface (or Plane) specifies the clearance level to which the tool is retracted after completing an NC sequence.

Optional

- Fixtures are parts or assemblies that help orient and hold the workpiece during a manufacturing operation.
- Manufacturing Geometry is made up of features that assist you in creating NC sequences, for example a mill surface to assist in face milling.

Other optional elements such as parameter files and PPRINT tables will be discussed later.

Note to Instructor: Ensure the meaning of all key terminology is noted by students in this slide.

Key Terminology:

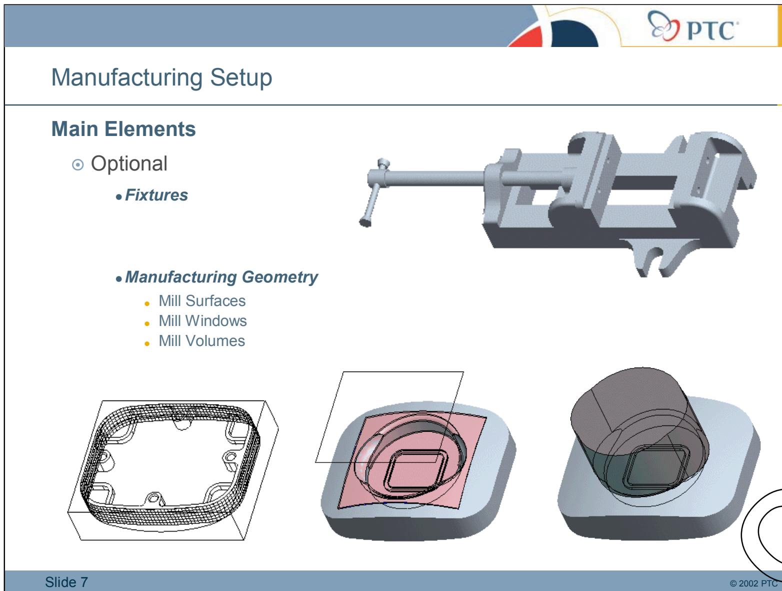
•Machining Operation

•Workcell

•Machine Zero

•Tools

•Retract Surface or Plane



Manufacturing Setup

Optional

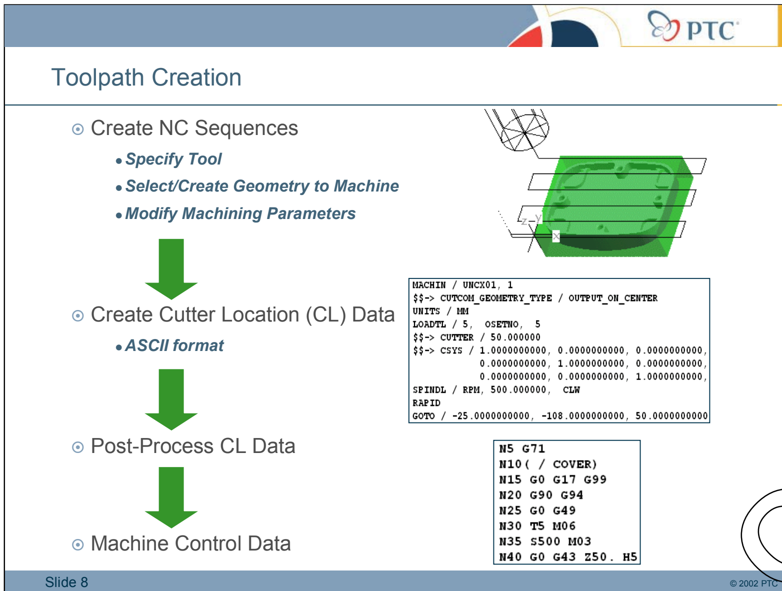
- Fixtures are parts or assemblies that help orient and hold the workpiece during a manufacturing operation.
- Manufacturing Geometry is made up of geometry features that assist you in creating NC sequences:
  - Mill surfaces are surfaces that can be used as references in many NC sequences, for example a mill surface can be configured as the surface for machining in a profile or face milling NC sequence.
  - Mill Windows are closed 2-D outlines that are typically used to specify the volume of material to be machined when volume milling.
  - Mill Volumes are closed 3-D volumes that can also be used to specify the volume of material to be machined when volume milling.

Other optional elements such as parameter files and PPRINT tables will be discussed later.

Note to Instructor: Ensure the meaning of all key terminology is noted by students in this slide.

Key Terminology:

- Fixtures
- Manufacturing Geometry



Toolpath Creation

The next step in this process is to create NC Sequences, this involves:

- Specifying a tool.
- Selecting or creating geometry to machine, (e.g. surface to machine or holes to be drilled).
- Specifying **how** the tool machines the selected geometry by modifying machining parameters, for example specifying cutting feed rate and spindle speed.

Each NC sequence is made up of a series of tool motions, (with the addition of specific post-processor words that are not motion-related but required for the correct NC output).

The next step is to create Cutter Location (CL) data files, these are generated from the cutter paths specified within NC sequences.

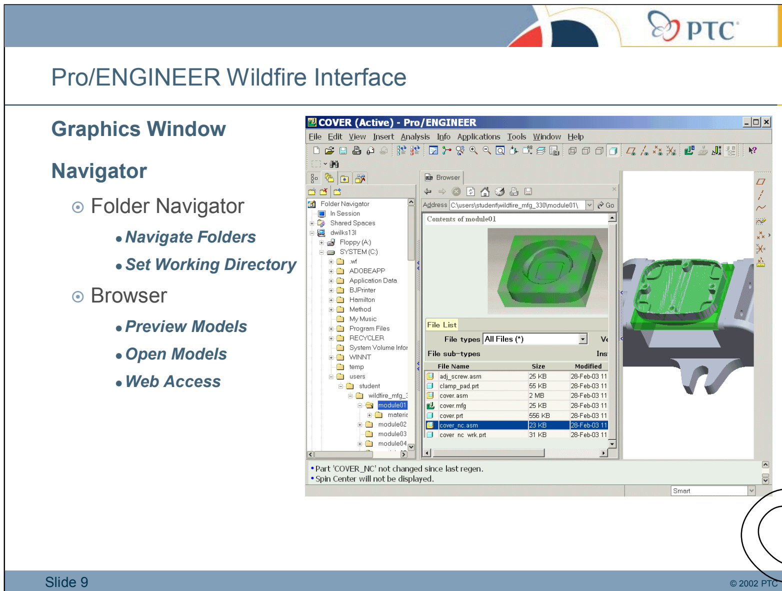
The CL data files can then be post-processed using machine-specific or generic post-processors to create Machine Control Data (MCD) files.

Finally, MCD files are used to drive machine tools.

Note to Instructor: Ensure the meaning of all key terminology is noted by students in this slide.

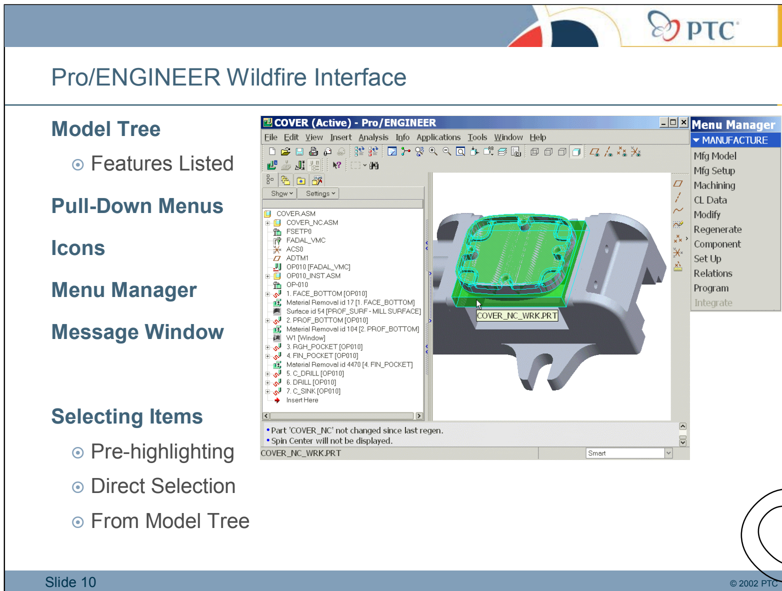
Key Terminology:

- NC Sequences
- Cutter location data (CL data)
- Post-Processing
- Machine Control Data (MCD)



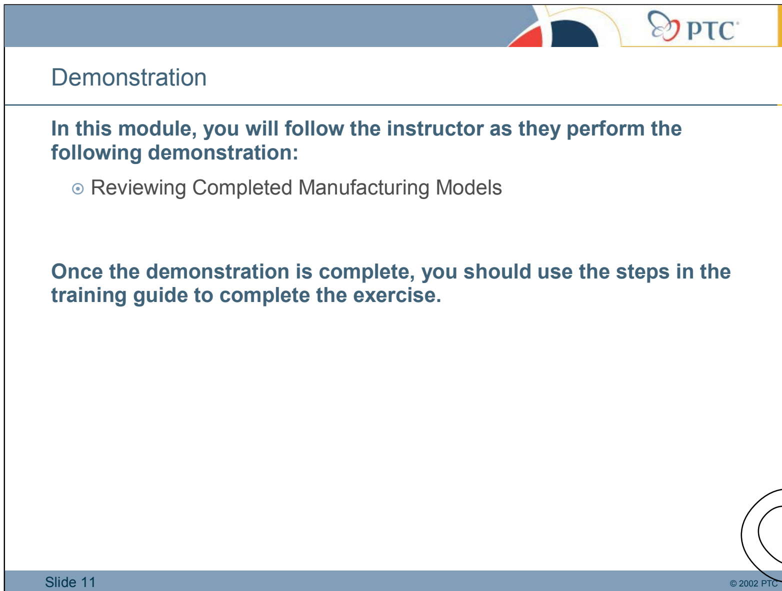
Pro/ENGINEER Wildfire Interface

- The Pro/ENGINEER Wildfire interface consists of a main graphics window, where you can create manufacturing models and NC sequences.
- On the left side of the graphics window is a folder navigator that allows navigation of the folder structure, it is also used to set the working directory.
- The integrated web browser enables selection and previewing of models within a folder, once a desired model is located it can be opened directly from within the browser.
- The web browser is also used to access web site information such as supplier and contactors' web sites.



Pro/ENGINEER Wildfire Interface

- When a manufacturing model is open a **model tree** appears to the left of the graphics window. The model tree lists all the models and features that make up the manufacturing model, this can include reference models, fixtures, machine operations, and NC sequences.
- Fully integrated **pull-down menus** are located at the top of the graphics window.
- **Toolbar icons** are by default located at the top, and to the right of the graphics window, these provide access to commonly used functions.
- The **menu manager** is located by default to the right of the graphics window and provides access to most of the manufacturing tools, such as setting up the manufacturing environment, and creating NC sequences.
- The **message window** which by default is located at the bottom of the graphics window provides feedback on recent selections and operations.
- Moving the cursor over a model in the graphics window causes different pieces of geometry and components to highlight automatically, this is know as **pre-highlighting**, and enables easy selection of objects within the graphics window.
- Items such as features or models can be selected **directly** from the model as they are highlighted. Alternatively items can be selected from within the **model tree** as required.



Demonstration: Reviewing Completed Manufacturing Models

Scenario

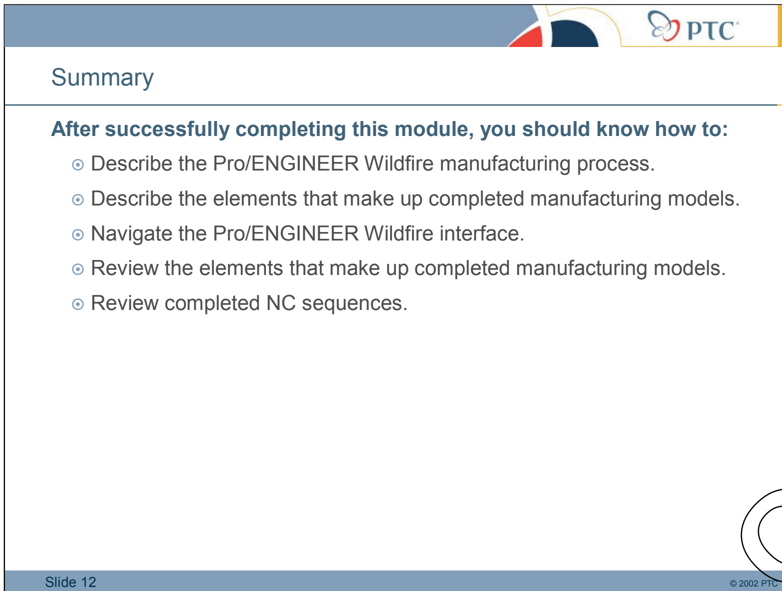
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the Pro/ENGINEER Wildfire manufacturing process.
- Describe the elements that make up completed manufacturing models.
- Navigate the Pro/ENGINEER Wildfire interface.
- Review the elements that make up completed manufacturing models.
- Review completed NC sequences.

Module 1 Lab Exercises

Exercise 1: Reviewing Completed Manufacturing Models

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you review the elements that make up completed manufacturing models. This involves opening a completed manufacturing model and identifying the reference model and the workpiece. You also review other selected elements such as the manufacturing operation, the workcell, the machine zero, and the retract plane. You will also review and change some of the NC sequences and observe how the tool paths update. The ability to review existing manufacturing models and tool paths is an important skill, and also provides you with an excellent overview of the manufacturing process.

Business Scenario Background

Precision Machining Inc. (PMI) is a supplier of electromechanical components, and tooling to many industries. The manufacturing department needs to generate a number of new milling NC programs to meet customer requests. As a manufacturing engineer, you are tasked with creating NC programs for a machined aluminum cover, and for a mold cavity. Other tasks may be assigned to you as the training scenario develops.

Objectives

After successfully completing this exercise, you will know how to:

- Review the elements that make up completed manufacturing models.
- Review completed NC sequences.

Scenario

In this exercise, you will review a completed manufacturing operation for the aluminum cover, including the review of existing NC-sequences. This will give you an opportunity to see a machined version of the aluminum cover, which you will then create stage-by-stage through the remaining exercises.

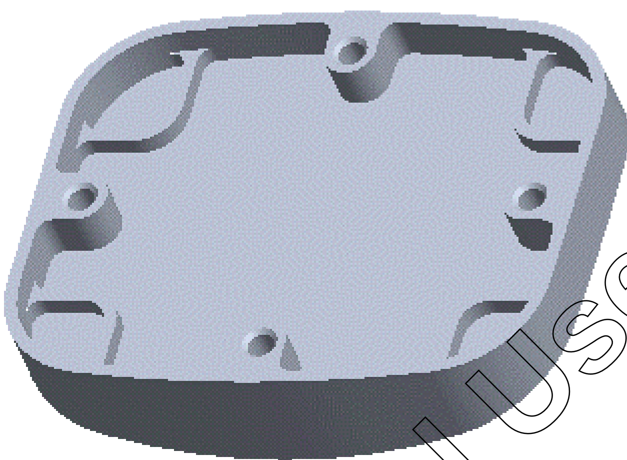


Figure 1: The underside view of the aluminum cover.

Step 1. Open the completed manufacturing model for the aluminum cover; identify the reference model and the workpiece.

Before reviewing the manufacturing model for the aluminum cover, we will preview the Pro/ENGINEER Wildfire interface so that we can become more comfortable with it. The benefit in this task is that becoming familiar with the Pro/ENGINEER Wildfire interface will enable us to focus more on creating NC programs.

Note the following items on the Wildfire Interface:

- Fully integrated pull down menus, which benefits users that do not prefer icon driven applications.
- Toolbar functionality with logically arranged icons.
- Note the tabs on the navigator. These tabs allow us to quickly change between the folders on our computer folder structure, to our favorites directory, to our different Internet connections, and to our model tree when we have a model.
- Navigator Browser that allows us to see outside of Pro/ENGINEER into supplier websites, contractors' websites, and PDM systems.

In order to review the manufacturing model, we first need to locate it and open it using the folder navigator. Once we have located the folder, we can set this as our working directory, and open *the manufacturing model of the cover. Notice how the model tree appears on the left of the screen. The model tree lists components within the manufacturing model, and manufacturing features such as operations, workcells, and NC sequences. We can identify the reference model and the workpiece in the model tree and on-screen.*

1. Start Pro/ENGINEER Wildfire if it has not already been started for you.
2. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module01**, right-click, and select **Make Working Directory**.
3. In the browser, select COVER.MFG to preview it, as shown in the following figure.
4. Drag the COVER.MFG manufacturing model into the graphics window to open it.

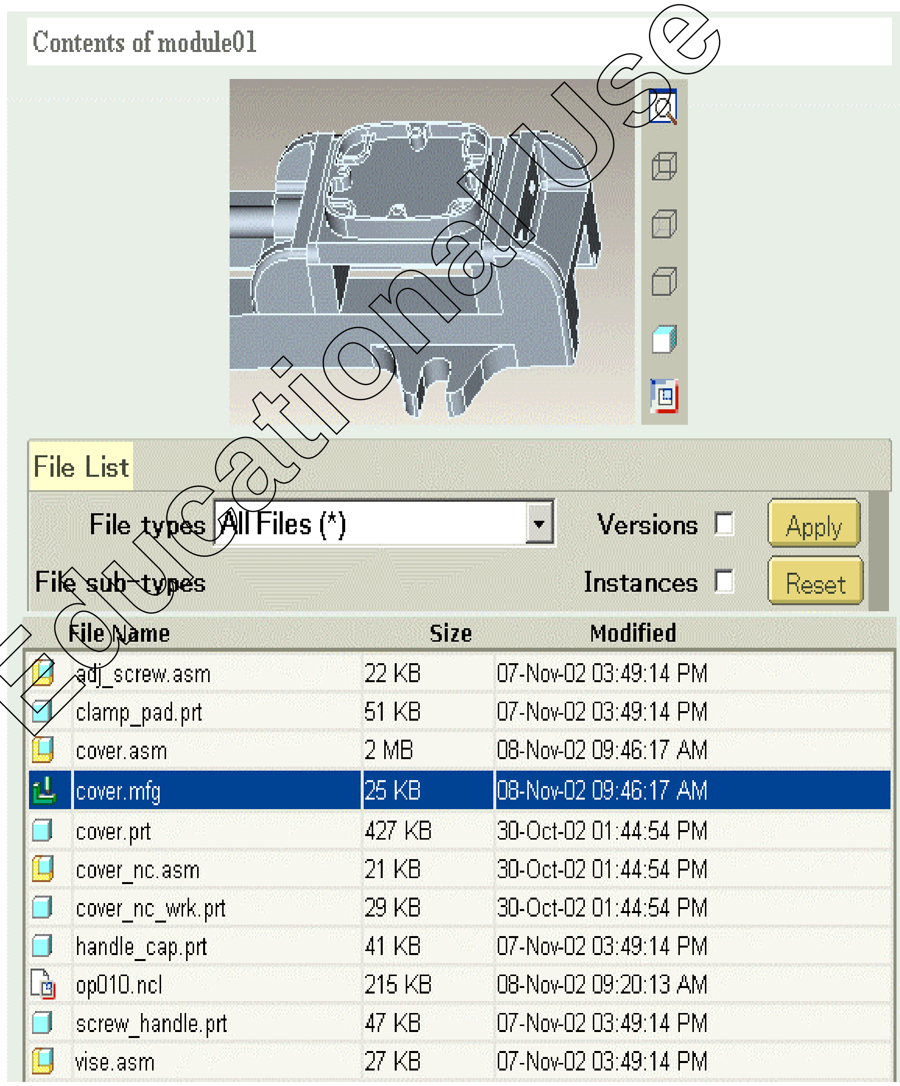


Figure 2: Selecting the manufacturing model for the cover.

> **Note:**
>
> Notice after the manufacturing model appears, that the model tree replaces the navigator on the left of the screen. The model tree lists all components within a manufacturing model, and manufacturing features such as the operations, workcells, and NC sequences.

5. Using the toolbar icons at the top of the graphics window, turn off the display of datum planes, axis, co-ordinate systems, and points.

6. Click on the saved view list icon and set the view to the named view **OP010**.

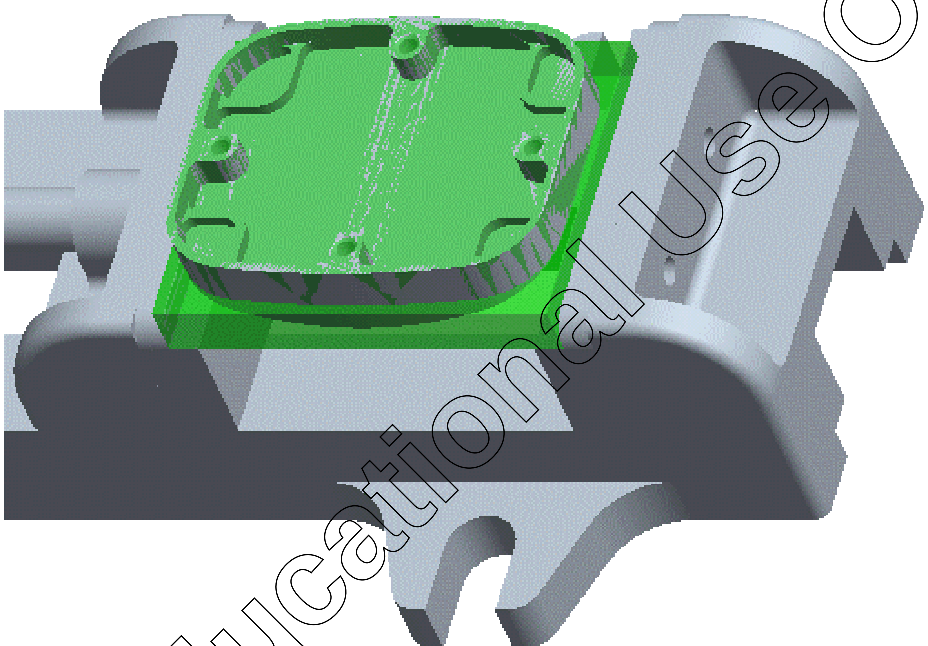


Figure 3: The view set to the named view OP010.

7. Expand COVER_NC.ASM in the model tree to see the models.

8. Select the reference model (COVER.PRT) in the model tree.

9. Select the workpiece model (COVER_NC_WRK.PRT) in the model tree. De-select the highlighted model by clicking in the background of the graphics window.

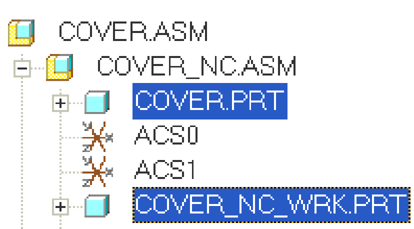


Figure 4: View of the expanded model tree.

> **Hint:**
> Notice how selecting models in the model tree highlights them on the screen. You can also highlight models by selecting them in the graphics window. To de-select a model you must click in the background of the graphics window.

Step 2. Review the workcell, the manufacturing operation, identify the machine zero, and retract surface.

The workcell dialog box, accessible from the Mfg Setup menu provides us with more information about the workcell.

Similarly the operation dialog box, also accessible from the Mfg Setup menu, provides us with more information about the operation. Here, we can preview the machine zero position and the retract surface.

1. Using the Menu Manager, select **Mfg Setup > Workcell** and examine the contents of the Workcell Dialog box. Close the Workcell Dialog box.

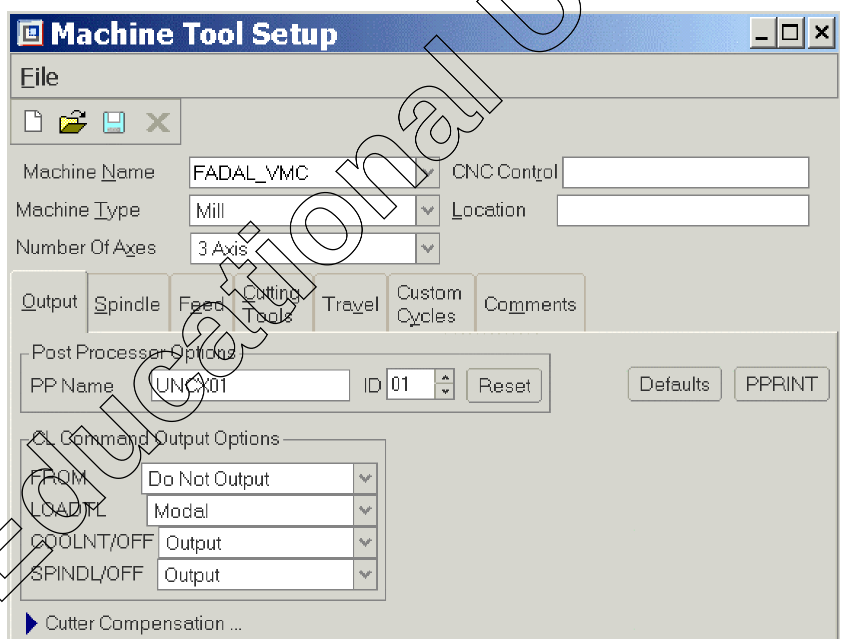


Figure 5: The workcell dialog box.

2. Using the Menu Manager, select **Mfg Setup > Operation**, and review the Operation dialog box.

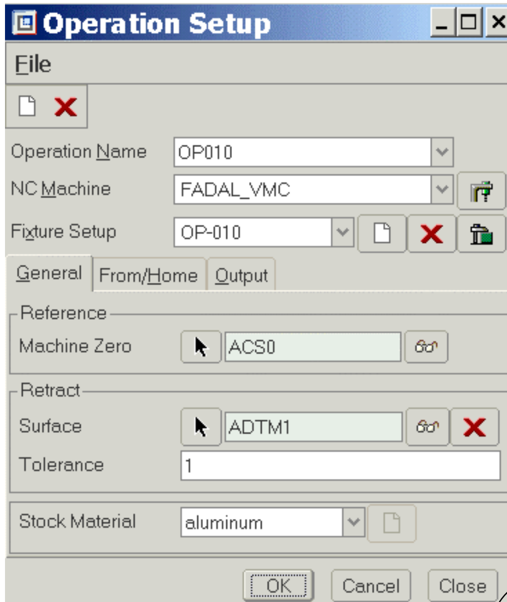


Figure 6: The operation dialog box.

3. Click on the preview icon for the machine zero, this highlights the machine zero in the graphics window. Click on the preview icon for the retract surface, this highlights the retract surface in the graphics window. Close the Operation dialog box.

4. Turn off the display of datum planes and datum co-ordinate systems that were turned on when previewing the machine zero and retract surface.

Step 3. Identify and review tools used in the machining process.

The Tool dialog box is also accessible via the Mfg Setup menu. Here we can review the various tools stored with this workcell.

1. In the Mfg Setup menu, select **Tooling > FADAL_VMC**, and examine the tools for the FADAL_VMC workcell. Click on each tool name to review the tool parameters.

2. When finished close the tool dialog box, and return to the Manufacture menu.

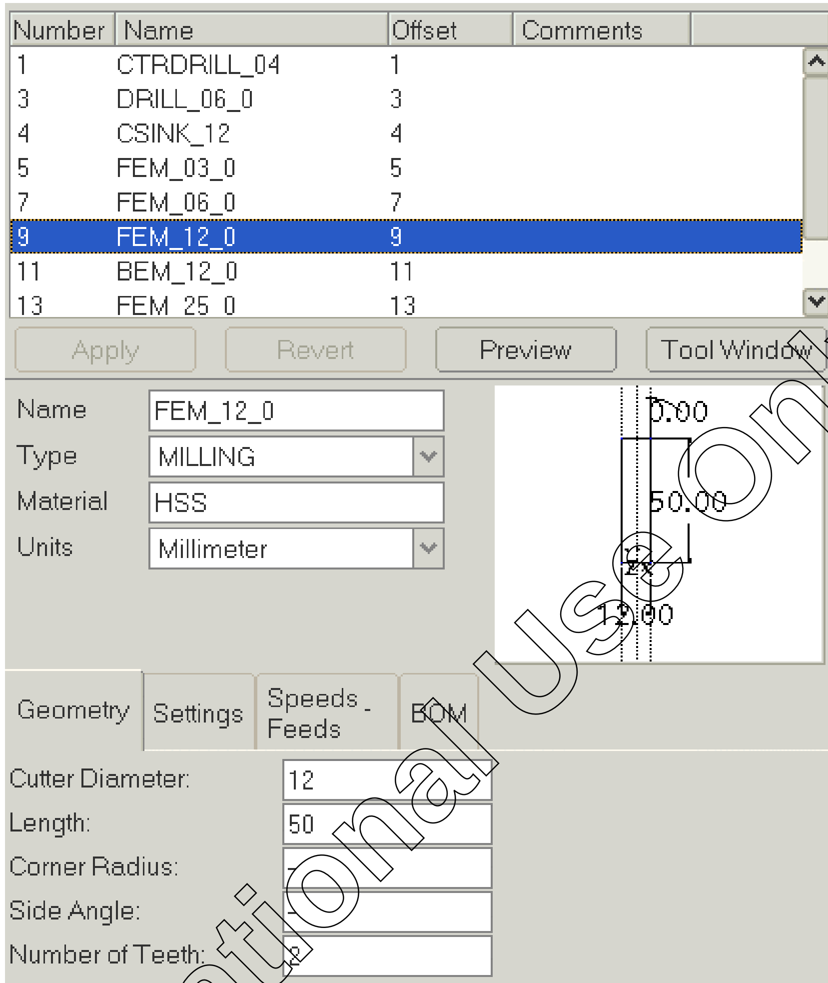


Figure 7: The view of the tool dialog box.

Step 4. Identify and review the fixture used in the machining process.

Fixture setups are configured within the Operation dialog box where we can identify the fixture setup used in this machining operation. We can also identify the fixture assembly in the model tree. Notice the fixture assembly highlights on screen when selected in the model tree.

Fixture setups are useful because once configured they can be easily activated and de-activated for different operations.

1. Using the Menu Manager, select **Mfg Setup > Operation**, and click on the Fixture Setup icon, this opens the Fixture Setup dialog box.

2. In the Components tab, select the fixture component OP010_INST. Then close the Fixture Setup dialog box, close the Operation dialog box, and return to the Manufacture menu.

3. In the model tree, select the fixture assembly (OP010_INST.ASM). De-select the highlighted model by clicking in the background of the graphics window.

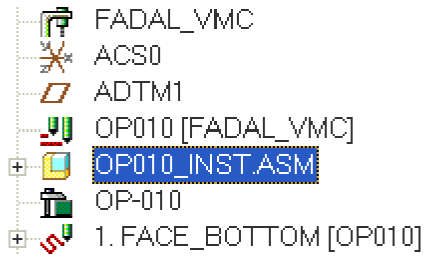


Figure 8: Fixture assembly selected in model tree.

> **Note:**
> Notice how the fixture assembly highlights in the graphics window when selected in the Components tab of the Fixture dialog box or when selected in the model tree.

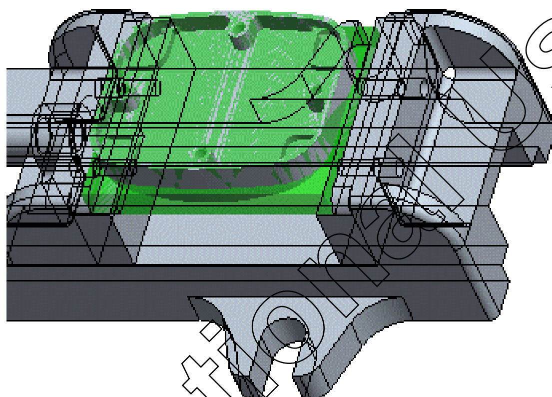


Figure 9: Fixture assembly highlighted in the graphics window.

Step 5. Open and review a site manufacturing parameter file.

We can review a manufacturing parameter site file by showing the site file used in this operation.

Manufacturing site files are useful because we can configure and store default sets of manufacturing parameter values that can be loaded at the beginning of each operation. In addition, we can use them to control parameter visibility and specify maximum and minimum parameter values.

1. Using the Menu Manager, select **Mfg Setup > Param Setup > Site,** and **Show**.
2. Select the **RGH_3_AXIS_MIL** site file, and review the contents of the information window.

> **Note:**
> Notice the parameter column lists all the available parameters for a milling site file, the default column shows the current default values.

3. Close the information window, and return to the Manufacture menu.

Step 6. Examine the CL data file for the first operation using Vericut.

The first operation is made up of several NC sequences. We can review the cutter location (CL) data file for the first operation. CL data files are created when an operation is complete and provide the input to post-processors to enable creation of machine code data (MCD) files.

We already have a CL data file ready to review, so we can review the CL data file using Vericut, enabling us to review the toolpath and view the machining simulation. Start Vericut and open the CL file OP010.NCL.

Note:

Selecting the NC Check option in the CL Data menu starts Vericut. This will open a separate Vericut window to perform the machining simulation.

1. Using the Menu Manager, select **CL Data > NC Check > CL File**, and **Open** the op010.ncl file, then select **Done** to start Vericut. (You may need to wait a few moments for Vericut to start).

2. In the Vericut window, click on the Setup Motion icon, and in the Motion dialog box drag the animation speed slider to the left (this will reduce the speed of the animation). Then close the Motion dialog box.

3. Click on the View Toolpath File icon to see the CL data file during the simulation. Then click on the Play to End icon to view the machining simulation.

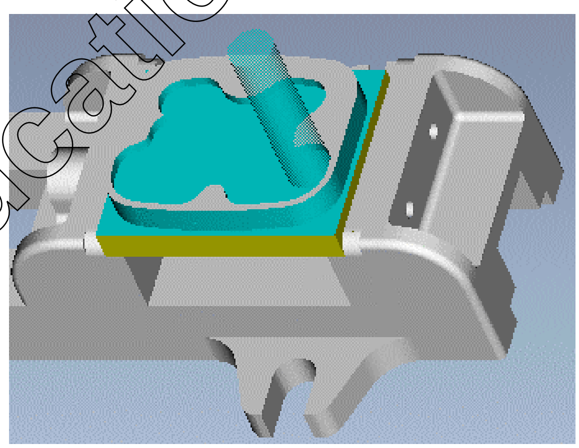


Figure 10: Playing the CL data file in Vericut.

```
MACHIN / UNCX01, 1
$$-> CUTCOM_GEOMETRY_TYPE / OUTPUT_ON_CENTER
UNITS / MM
LOADTL / 15,  OSETNO,  15
$$-> CUTTER / 50.000000
$$-> CSYS / 1.0000000000, 0.0000000000, 0.0000000000, 0.0000000000,
            0.0000000000, 1.0000000000, 0.0000000000, 0.0000000000,
            0.0000000000, 0.0000000000, 1.0000000000, 0.0000000000
SPINDL / RPM, 1165.014183,  CLW
COOLNT / ON
RAPID
GOTO / -25.0000000000, 0.0000000000, 40.0000000000
RAPID
GOTO / -25.0000000000, 0.0000000000, 3.0000000000
FEDRAT / 302.903688,  MMPM
GOTO / -25.0000000000, 0.0000000000, -2.0000000000
GOTO / 158.0000006939, 0.0000000000, -2.0000000000
```

Figure 11: Viewing the CL data file.

4. Close Vericut when finished, and return to the Manufacture menu.

Step 7. Review and adjust the face milling NC sequence.

Reviewing and adjusting NC sequences is important because it allows us to verify and optimize existing tool paths. You will repeat the following method every time you want to adjust a tool path.

There are several ways we can review NC sequences. In this case, we will access the face milling sequence through the Machining menu. All the sequences for the first operation appear in a list, and we can select the sequence we want to review. We will play the toolpath on screen. Note that we can display the CL data at the same time if desired. We can now adjust some manufacturing parameters, and review the updated tool path. We can also view the machining simulation using Vericut.

1. Using the Menu Manager, select **Machining > NC Sequence** and select the **FACE_BOTTOM** NC sequence for review. When prompted **Suspend All** child features.

1: FACE_BOTTOM, Operation: OP010
2: PROF_BOTTOM, Operation: OP010
3: RGH_POCKET, Operation: OP010
4: FIN_POCKET, Operation: OP010
5: C_DRILL, Operation: OP010
6: DRILL, Operation: OP010
7: C_SINK, Operation: OP010

Figure 12: Selecting the sequence from the NC Sequence list.

2. Review the toolpath by selecting **Play Path > Screen Play**.

3. In the Play Path dialog box, click on the blue CL data bar, this will make the CL data visible in the dialog box. Click on the Play Forward button to review the toolpath. To slow down the toolpath simulation, click and drag the display speed slider to the left.

> **Note:**
> When the toolpath simulation is finished you can replay the toolpath by clicking the Rewind button and then clicking the Play Forward button again.

4. Close the Play Path dialog box when finished.

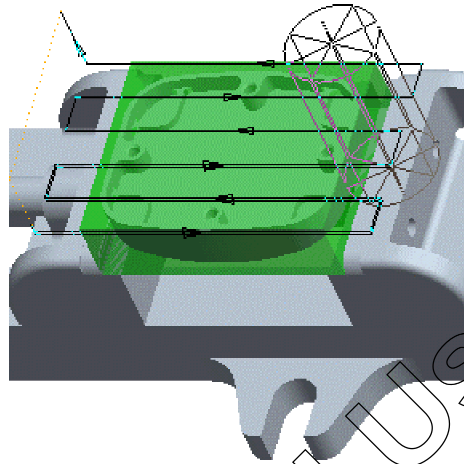


Figure 13: Playing the face milling tool path.

5. To modify the manufacturing parameters click on the Manufacturing Parameter Tree icon at the top of the graphics window. This will make the parameter window appear.

6. In the Parameter Tree window, modify the CUT_ANGLE value from 0 to [**90**], then close the Parameter Tree window.

7. Review the toolpath again by selecting **Play Path > Screen Play**. Then click the **Play Forward** button to review the toolpath.

> **Note:**
> Notice when reviewing the toolpath the tool cutting angle has changed by ninety degrees.

8. Click on the Manufacturing Parameter Tree icon again, and in the Parameter Tree window modify the SCAN_TYPE parameter value to **TYPE_ONE_DIR**, then close the parameter tree window.

9. In the Play Path menu, select **NC Check**. (You may need to wait a few moments for Vericut to start). Click on the Play to End icon to view the machining simulation.

> **Note:**
> To slow down the simulation you can click on the Setup Motion icon, and in the motion dialog box drag the animation speed slider to the left.

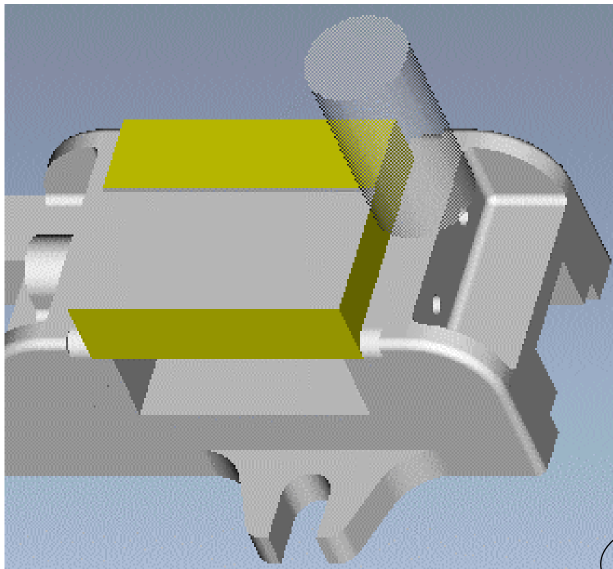


Figure 14: The face milling sequence simulation in Vericut.

10. When finished close Vericut, then complete the sequence, by selecting **Done Seq**, and return to the Manufacture menu.

Step 8. Review and adjust the profile milling NC sequence.

We can repeat the process and adjust the profiling sequence.

1. Using the Menu Manager, select **Machining > NC Sequence** and select the **PROF_BOTTOM NC** sequence for review. When prompted **Suspend All** child features.

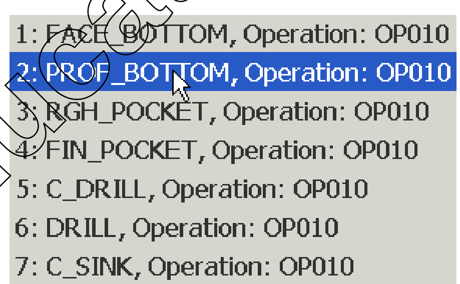


Figure 15: Selecting the sequence from the NC Sequence list.

2. Review the toolpath by selecting **Play Path > Screen Play**.

3. Click on the Play Forward button to review the toolpath. To slow down the toolpath simulation, click and drag the display speed slider to the left.

4. Close the Play Path dialog box when finished.

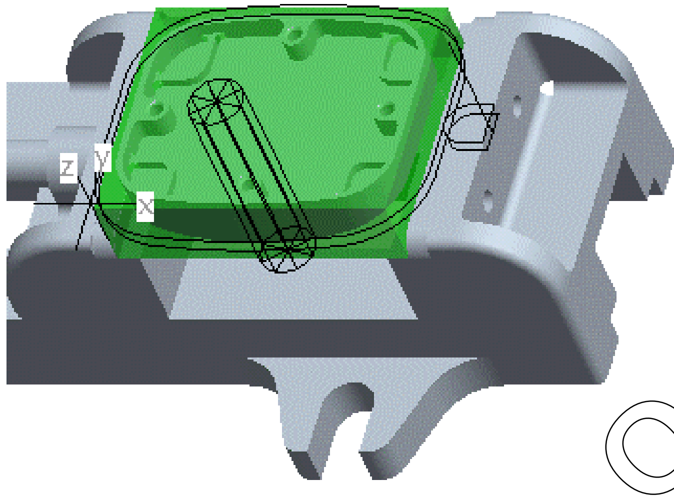


Figure 16: Playing the profile milling tool path.

5. To modify the manufacturing parameters, click on the Manufacturing Parameter Tree icon at the top of the graphics window. This will make the parameter window appear.

6. In the Parameter Tree window, modify the STEP_DEPTH parameter value to [**6**] *(0.24 in.).*

7. Review the toolpath again by selecting **Play Path > Screen Play**. Then click on the Play Forward button to review the toolpath.

Note:
Notice how the tool now makes three profile passes around the reference model.

8. View the machining simulation using Vericut, by selecting **NC Check** in the Play Path menu. (You may need to wait a few moments for Vericut to start). Click on the Play to End icon to view the machining simulation.

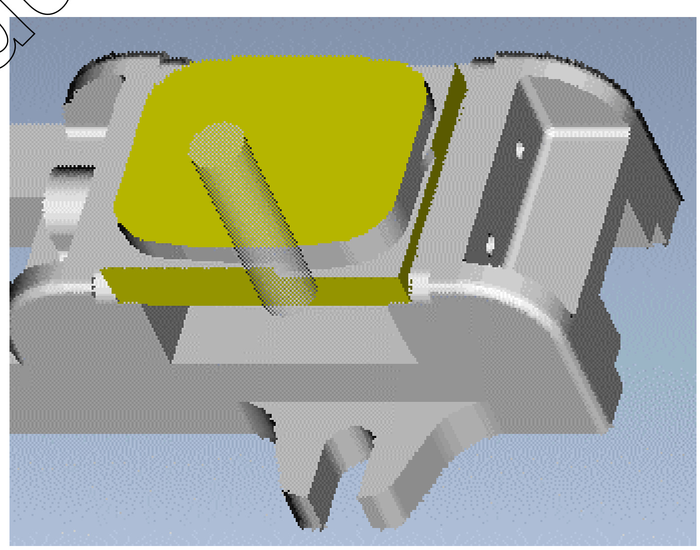


Figure 17: The profile milling sequence simulation in Vericut.

9. When finished, close Vericut, then complete the sequence by selecting **Done Seq**, and return to the Manufacture menu.

10. To save the manufacturing model, click **File > Save**, and press **ENTER** on the keyboard.

11. Click **Window > Close** to close all windows

12. To erase all components from memory, click **File > Erase > Not Displayed**. Then click **OK** in the erase not displayed dialog box.

> **Note:**
> Closing windows does not remove components from the current session of Pro/ENGINEER Wildfire. This is only achieved by erasing components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the Pro/ENGINEER Wildfire manufacturing process.
- Describe the elements that make up completed manufacturing models.
- Navigate the Pro/ENGINEER Wildfire interface.
- Review the elements that make up completed manufacturing models.
- Review completed NC sequences.

Module

2

Creating Workpieces

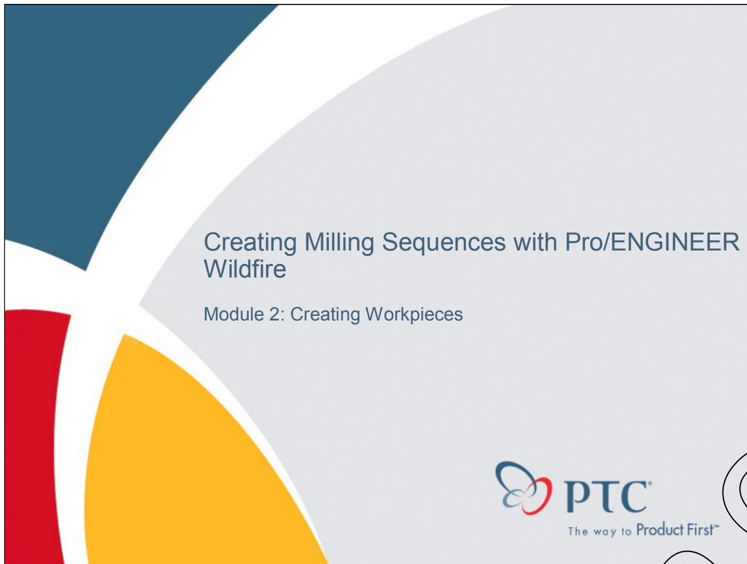
Introduction

The first step in the manufacturing process typically involves creating the stock material (known as the workpiece) for machining. During this process, the size of the as-designed component is used to determine a suitable workpiece size. In this module, you will learn two different methods for creating workpieces, providing you with a flexible approach to workpiece creation. Using workpieces enables you to create an as-machined version of the workpiece after the completion of each NC sequence. When reviewing NC sequences you can also view the cutting tool machining the workpiece. When creating workpieces it is important to know how to manipulate workpiece models and how to select and modify workpiece geometry.

Objectives

After completing this module, you will be able to:

- Describe the purpose of workpieces.
- Create NC model assemblies consisting of reference models and workpieces.
- Create workpieces using the Create Stock dialog box.
- Create workpieces using template parts.
- Reorient models and select model geometry.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

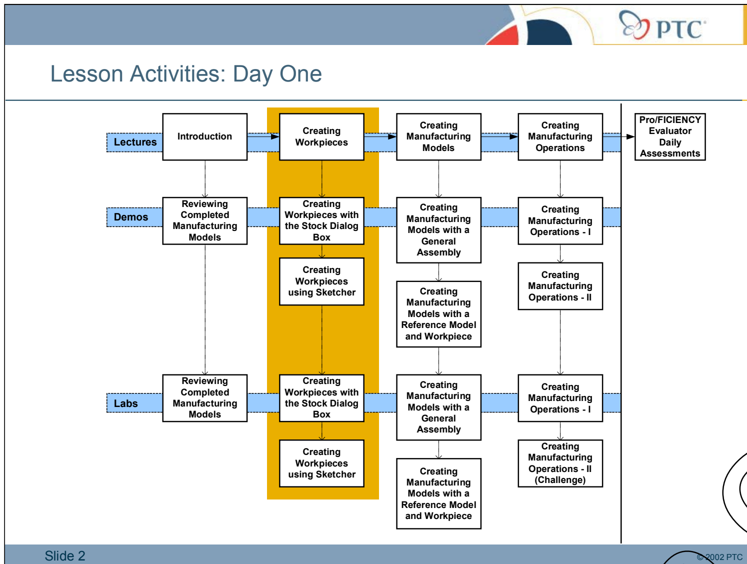
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

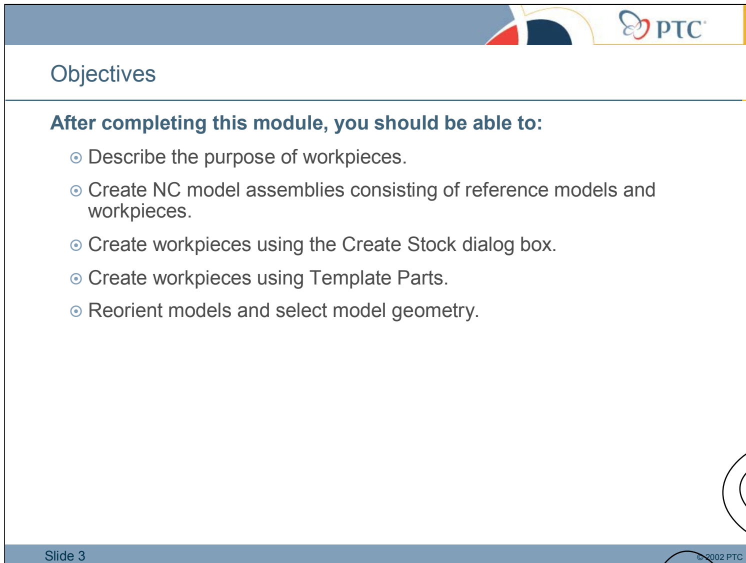
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 20 mins
- Labs : 40 mins
- Total: 1 hr 15 mins

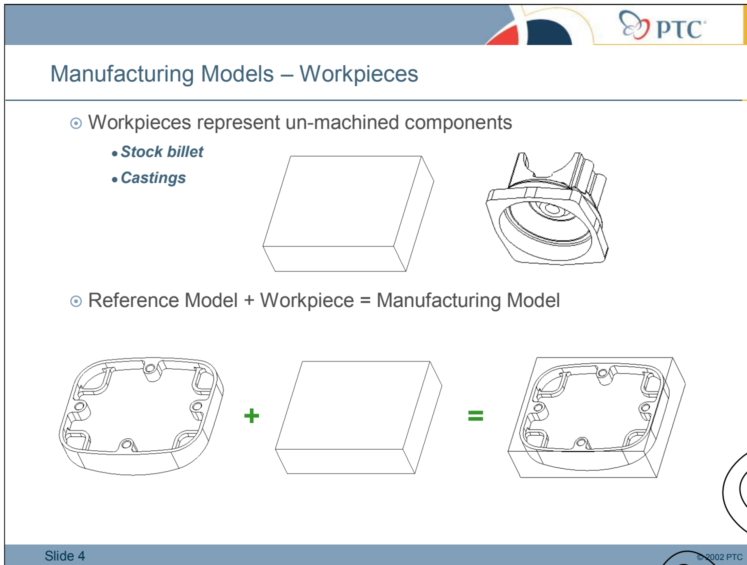


Overview

The first step in the manufacturing process typically involves creating the stock material (known as the workpiece) for machining. During this process the size of the as-designed component is used to determine a suitable workpiece size. In this module, you will learn two different methods for creating workpieces, providing you with a flexible approach to workpiece creation. Using workpieces enables you to create an as-machined version of the workpiece after the completion of each NC sequence. When reviewing NC sequences you can also view the cutting tool machining the workpiece. When creating workpieces it is important know how to manipulate workpiece models and how to select and modify workpiece geometry.

After successfully completing this module, you will be able to:

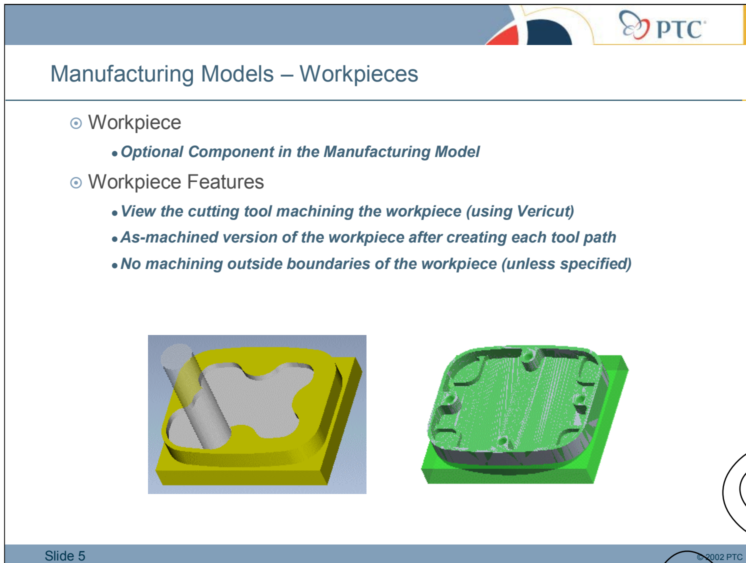
- Describe the purpose of workpieces.
- Create NC model assemblies consisting of reference models and workpieces.
- Create workpieces using the create-stock dialog box.
- Create workpieces using template parts.
- Reorient models and select geometry.



Manufacturing Models – Workpieces

The workpiece represents the stock material that is going to be machined when creating NC sequences. The workpiece can be a standard stock billet or it can represent a casting.

As stated previously, the reference model and workpiece are assembled together to form the manufacturing model.



Manufacturing Models – Workpieces

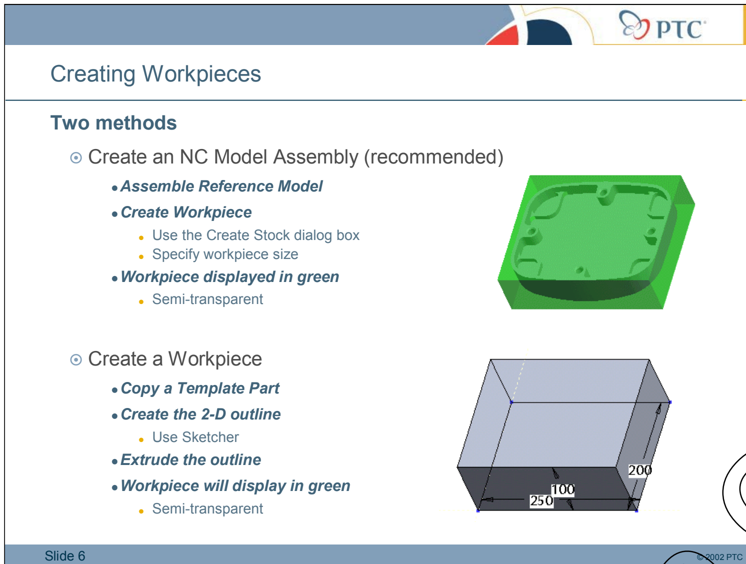
The workpiece is an optional component within the manufacturing model.

If you are not concerned with simulating the machining of the workpiece or creating an as-machined version of the workpiece, then you do not need to use a workpiece. (Therefore, the minimum configuration of a manufacturing assembly is one reference model.)

Workpiece features include:

- Viewing the cutting tool machining the workpiece (using Vericut).
- After creating each tool path you can update the workpiece to show an as-machined version of the workpiece.
- No machining outside boundaries of the workpiece.
  - Unless specified

If a workpiece is not used, NC sequences can still be created using the reference model. Viewing a simulation of the machining process using Vericut is still possible, but no workpiece is shown.



Creating Workpieces

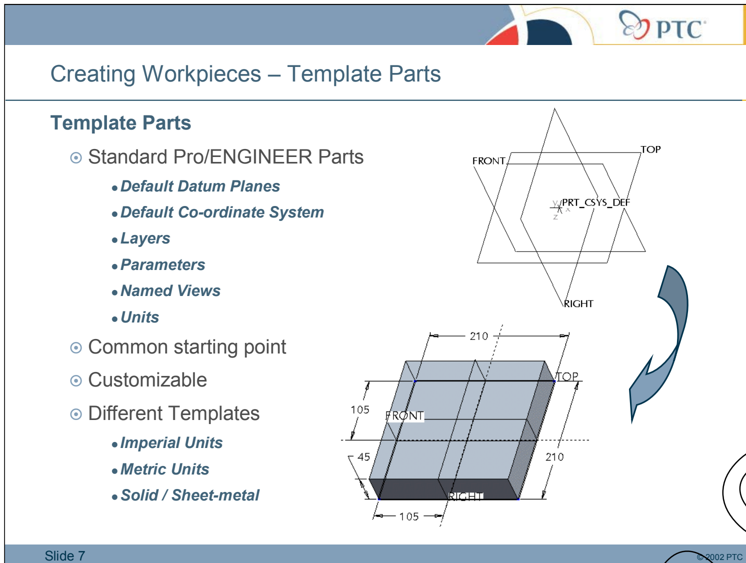
There are two methods for creating workpieces:

- The recommended method is to create an NC Model Assembly:
  - The reference model is assembled first.
  - The workpiece is created by selecting a default billet size and specifying stock allowances with respect to the reference model.
- The resultant workpiece is created and becomes part of the NC model assembly.
- The NC model assembly can then be used directly when creating the manufacturing model.
- When you assemble or create a workpiece in a manufacturing model, it is displayed in green. This helps to see the difference between the reference part and the workpiece geometry.
- The reference part is usually assembled inside the workpiece. If your Display Style environment is set to Shading, the workpiece would obscure the reference part. To prevent this, the workpiece in manufacturing mode is displayed as semi-transparent.

- An alternative method is to create the workpiece using a template part model.
  - First determine the size of the reference model.
  - Create a workpiece by copying a template part.
  - Then create the workpiece geometry by sketching a 2-D outline and extruding an outline based on the reference model dimensions.
  - A workpiece created using this method will display in green when it is assembled into a manufacturing model.

Advantages of the first method:

- The workpiece size can be determined with direct respect to the reference model
- Changes to the workpiece dimensions update immediately on screen
- The workpiece can be created without knowledge of the Sketcher interface



Creating Workpieces – Template Parts

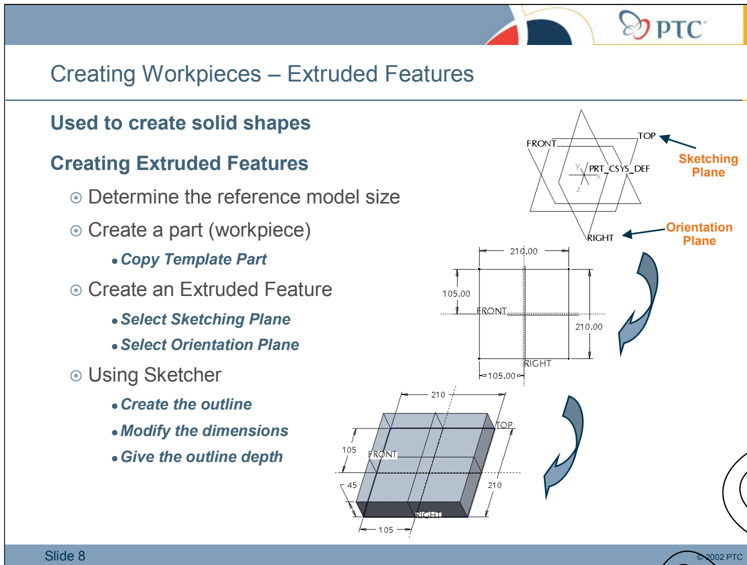
Template parts are standard Pro/ENGINEER parts that can be copied and used as common starting point for all parts including workpieces. By default they are made up of:

- Three default datum planes
- Default co-ordinate system
- Layers for features
- Parameters, such as description and drawn_by.
- A series of named views.
- Pre-defined units, such as inches or millimetres.

•They provide a common starting point for all designs

•They can be customized to meet customers requirements.

•By default different template parts are available, for example templates with imperial or metric units and templates for solid geometry or sheet-metal parts.



Creating Workpieces – Extruded Features

Extruded features are used to create solid extruded geometry, and involve sketching and dimensioning an outline and then extruding the outline to create solid geometry.

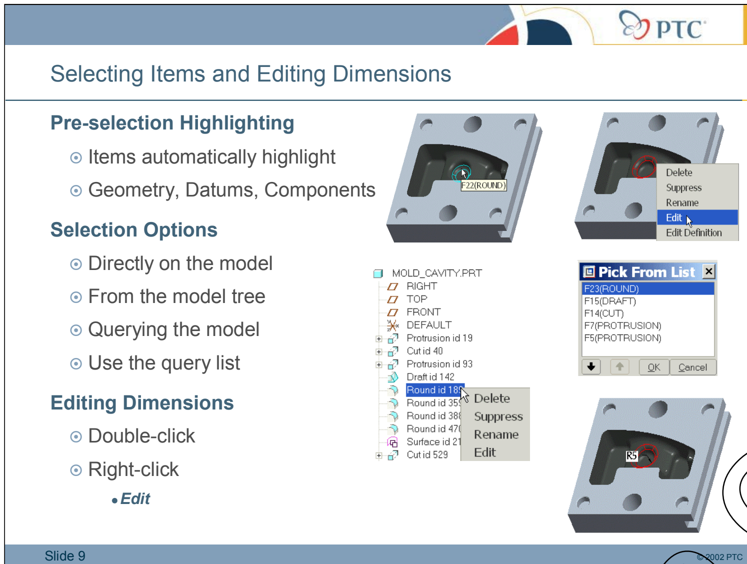
Looking at this method in more detail:

- First determine the reference model size.
- Then create the workpiece by sketching an outline based on the reference model dimensions.

This involves:

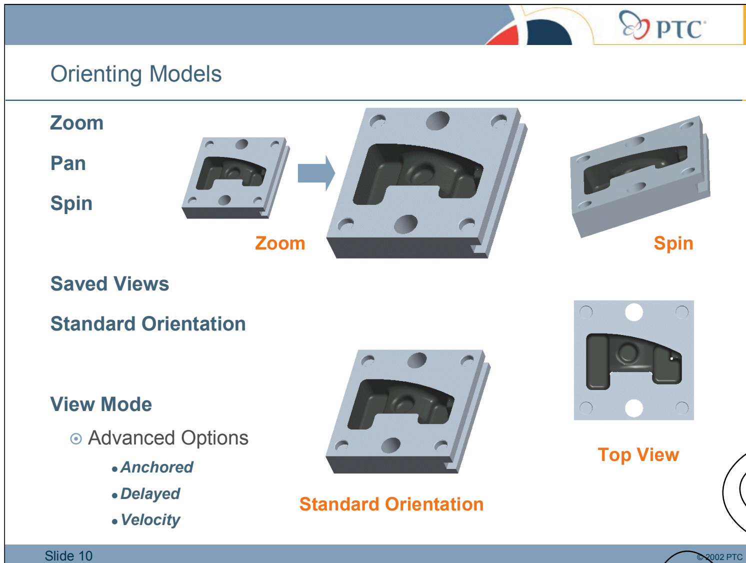
- Creating a part, (workpiece), usually by copying a template part.
- Creating an extruded feature within the part.
- An extruded feature is a sketch that extrudes a section perpendicular from a sketching plane to create a protrusion.
- Here is the process of creating the extrude feature:
  - Select a sketching plane and a plane to orient the sketch.
  - Draw the outline in Sketcher.
    - Dimensions will be added automatically.
  - Modify the dimensions and complete the sketch.
  - Determine the depth, and complete the feature.

You use extruded features when you want to create solid shapes within parts.



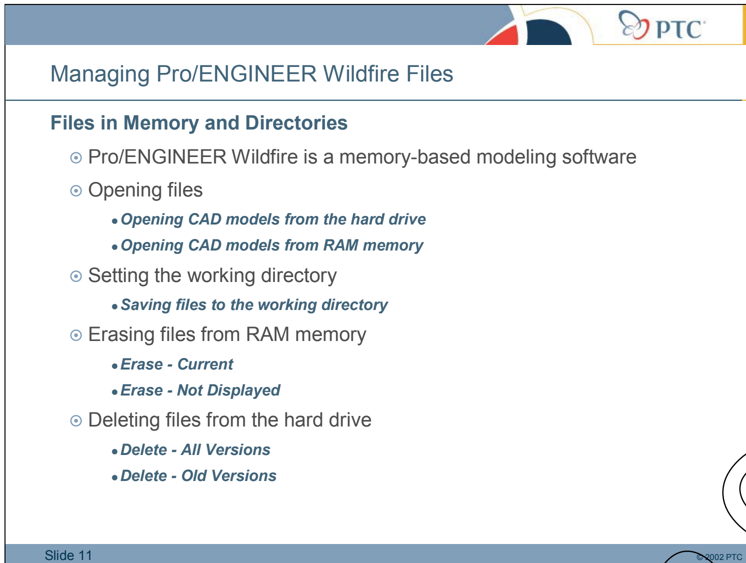
Selecting and Editing Geometry

- **Pre-selection highlighting** allows you to visually confirm the item that will be selected before you select it. As you drag the cursor over the graphics window, selectable items such as geometry features, datum features, or components are highlighted in cyan. If you leave the cursor over the item, a ToolTip displays the name of the highlighted item and it is also displayed in the Status Bar.
- Items can be selected **directly** on the model when highlighted, or they can be selected from the **model tree**. Alternatively items that cannot be selected directly I.e. hidden items can be selected by **querying the model**, this involves moving the cursor over the required item and right-clicking the mouse until the desired item is highlighted. A final alternative is to activate the **query list** which provides a list of all selectable items.
- **Editing** items dimensions is possible by **double-clicking** the item when highlighted.
- Alternatively when an item is selected it turns red by default, any dimensions associated with the item can then be seen, by **right-clicking** and selecting edit from the drop down list.



Orienting Models

- To set the model at the desired orientation, you can spin and pan the model around the screen, and zoom in and out using a combination of keyboard and mouse functions.
- You can also save a series of set orientations such as front, top, and side for later use.
- You can return the model to its default or standard orientation at any time.
- View Mode allows spin/pan/zoom operations without pre-highlighting or selection occurring.
- View Mode also has options such as Anchored, Delayed, and Velocity:
  - Anchored: Alternate spin algorithm utilizing on-screen spin axis.
  - Delayed: There is a time delay between when the spin, pan or zoom is done and the model orientation updates on the screen, which is useful when working with large assembly models.
  - Velocity: You can change the orientation of the model by spinning, panning or zooming at a controlled speed on the screen.
  - You can also orient about reference geometry such as an edges or surfaces with View Mode.



Managing Pro/ENGINEER Wildfire Files

- Pro/ENGINEER Wildfire is a memory-based modeling software, which means that files you create and edit are stored within RAM memory while you are working on them.

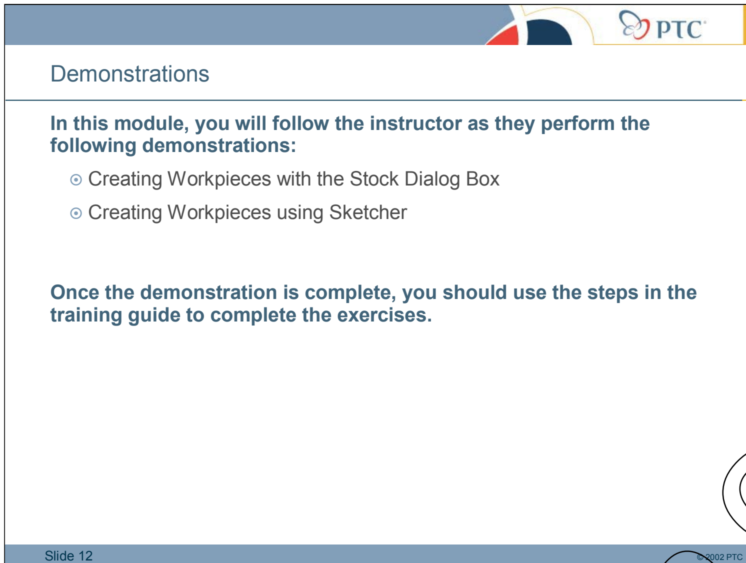
- Initially, all files are opened from the computer hard drive or network storage area.
- The files are now stored within RAM memory and continue to be stored in RAM memory as you create/edit them, and re-open them.

- Setting your working directory is important because all Pro/ENGINEER Wildfire files are saved to that location on your hard drive or network storage area.

- The files are now stored within RAM memory and continue to be stored in RAM memory while you create/edit them.
- Files are not saved to your hard drive or network storage area until you explicitly save them.

- Erasing models from memory is important because they are stored in RAM memory until you either erase them or exit Pro/ENGINEER Wildfire. This is especially important if you are working on files that have the same name but are in various stages of completion such as in this course. Erasing models does not delete them from the hard drive or network storage area.

- Deleting files permanently removes them from the working directory on your hard drive or network storage area. Be careful when deleting files because you cannot undo deleted files.



Demonstrations:

- **Creating Workpieces with the Stock Dialog Box**
- **Creating Workpieces using Sketcher**

Scenario

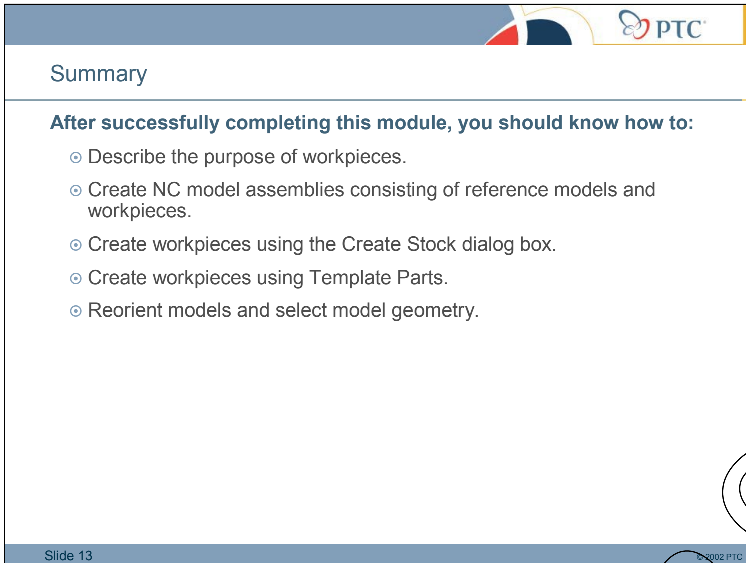
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the purpose of workpieces.
- Create NC model assemblies consisting of reference models and workpieces.
- Create workpieces using the create-stock dialog box.
- Create workpieces using template parts.
- Reorient models and select geometry.

Module 2 Lab Exercises

Exercise 1: Creating Workpieces using the Stock Dialog Box

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you will show the recommended method for creating workpieces. This involves creating an NC model assembly and assembling a reference model, then creating a workpiece using the Stock dialog box. An alternative method will be demonstrated in the next exercise. The ability to create workpieces using different methods provides you with a flexible approach to workpiece creation.

Using workpieces enables you to create an as-machined version of the workpiece after the completion of each NC sequence. When reviewing NC sequences, you can also view the cutting tool machining the workpiece.

Objectives

After successfully completing this exercise, you will know how to:

- Create NC model assemblies consisting of reference models and workpieces.
- Create workpieces using the Create Stock dialog box.

Scenario

You need to create the workpiece before you can machine the aluminum cover; this is typically the first step in the manufacturing process. The recommended method for creating workpieces involves creating an NC model assembly, assembling the reference model and creating the workpiece using the Stock dialog box.

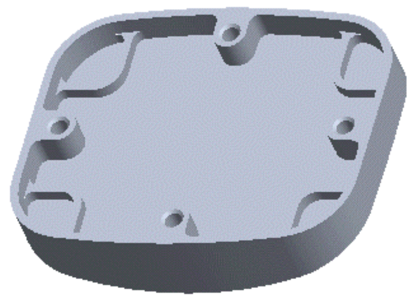


Figure 1: The underside view of the aluminum cover.

Step 1. Create a new NC assembly for the aluminum cover; assemble in the aluminum cover as the reference model.

We need to create the workpiece before we can machine the aluminum cover; this is typically the first step in the manufacturing process. We use the recommended method for creating workpieces, this involves creating an NC model assembly, assembling the reference model and creating the workpiece using the Create Stock dialog box. Before we create any models we should load a configuration file to ensure that the workpiece is created with metric units, as all our models in the exercises have metric units.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module02**, right-click, and select **Make Working Directory**.
2. Before creating any models load the configuration file, config.pro. Click **Tools > Options**, use the Open a Configuration File icon and select the **config.pro** file from the module02 folder. Then **Apply** and, **Close** the Options dialog box.

> **Note:**
> By default Pro/ENGINEER creates workpieces with imperial units, loading the configuration file ensures workpieces are created with metric units.

3. Create a new NC model assembly, click **File > New**, select the **Assembly** and **NC Model** options, and name the assembly [**cover_nc**]. When prompted select the **cover.prt** model from the list, (this assembles the reference model).

Step 2. Create the workpiece using the Create Stock dialog box.

The Stock dialog box allows us to create bar stock and billet stock. We can create stock using standard size billets, or we can create our own custom size, alternatively we can add material allowances to the size of the reference model.

In this case, we create our own custom size; notice the default values for length, width, and thickness are the dimensions of the reference model, so it is easy to calculate suitable stock *dimensions. Notice also that the workpiece is green and semi-transparent. This helps to see the difference between the reference model and the workpiece geometry.*

1. Using the Menu Manager, select **Create Stock**. In the Setup Stock dialog box, create the stock using the **Custom** stock instance option and enter the following dimensions, length [**133.0**] *(5.24 in.)*, width [**38.0**] *(1.50 in.)*, and thickness [**108.0**] *(4.25 in.)*. Then close the dialog box.

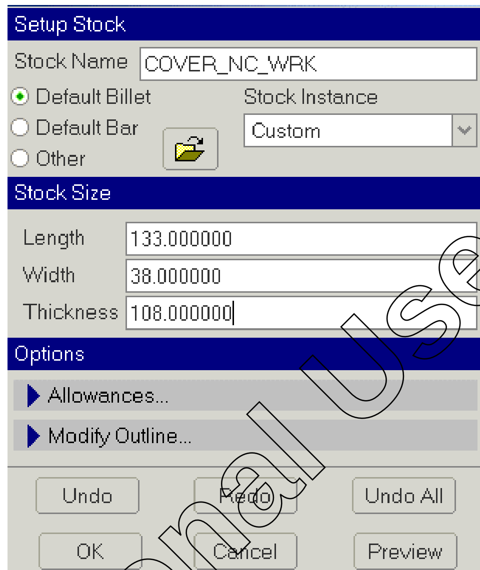


Figure 2: Create Stock Dialog Box

2. Click **File > Save** to save the NC model assembly.

3. Click **Window > Close** to close all windows.

4. Click **File > Erase > Not Displayed > OK** to erase all components from memory.

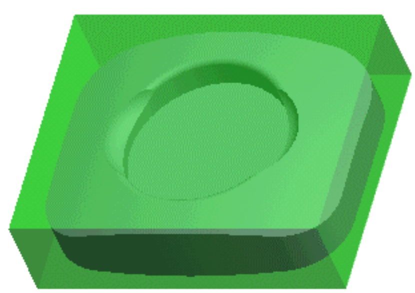


Figure 3: Cover NC Model Assembly

> **Note:**
> When you assemble or create workpieces in manufacturing mode, they are displayed in green and shown as semi-transparent. This helps to see the difference between the reference model and the workpiece geometry.

Exercise 2: Creating Workpieces using Sketcher

Introduction

In this demonstration, you will demonstrate an alternative method for creating workpieces. This involves creating a workpiece by sketching an outline based on reference model dimensions. The ability to create workpieces using different methods provides you with a flexible approach to workpiece creation.

Objectives

After successfully completing this exercise, you will know how to:

- Create workpieces using Sketcher.

Scenario

You need to create a workpiece for the mold cavity. You will use an alternative method that involves creating a part and sketching an outline based on the dimensions of the reference model, in this case the mold cavity.

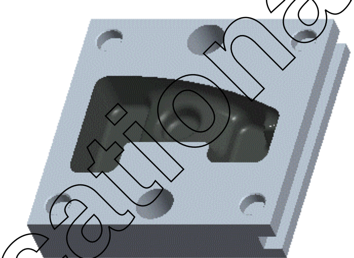


Figure 4: Mold Cavity

Step 1. Open the mold cavity and determine its size.

We need to create a workpiece for the mold cavity. We use an alternative method that involves creating a part and sketching an outline based on the dimensions of the reference model, in this case, the mold cavity.

We must first open the mold cavity part.

Determining the dimensions of this model will provide us with an excellent opportunity to learn how to select geometry on models and manipulate and orient models on screen. These are essential skills when working with models.

First, we learn how to select geometry.

Notice how different pieces of geometry pre-highlight as we move the cursor over the model. We can move the cursor over the bottom of the model until the main protrusion highlights in pale blue.

We can double-click the geometry to see the dimensions. Having noted the size of the mold cavity, we can select the background to de-select the highlighted geometry.

*An alternative method is to select the geometry from the model tree. This can be achieved by selecting the first protrusion in the model tree; notice how the same geometry highlights on screen. We can right-click and select **Edit** from the drop down list; this will make the dimensions of the selected piece of geometry display.*

*A third alternative is to move the cursor over the model again and select the geometry; notice how the selected geometry changes color to red, and the protrusion is highlighted in the model tree. We can right-click and select **Edit** from the drop down list, this will make the dimensions of the selected piece of geometry display.*

You should practice selecting geometry on the model during this exercise.

1. Click **File > Open** to open the MOLD_CAVITY.PRT model.
2. Move the cursor over different pieces of geometry in the model move the cursor to the bottom of the model and double-click on the (F5) PROTRUSION to display the dimensions as shown in the following figure. Make note of the dimensions for future reference.

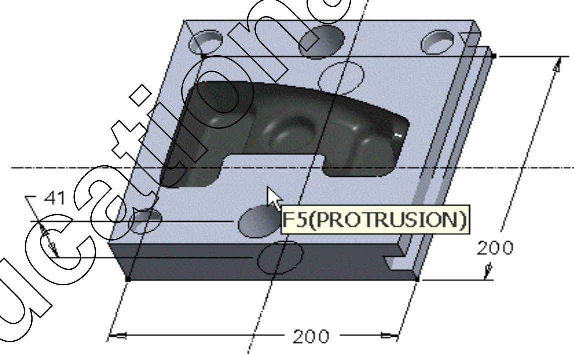


Figure 5: Mold Cavity Dimensions

> **Note:**
> Notice how different pieces of geometry highlight as you move the cursor over the model. Each piece of geometry corresponds to a different feature in the model.

3. De-select the highlighted geometry by clicking in the background of the graphics window.
4. Practice showing dimensions of other features by double-clicking on other pieces of geometry in the model. De-select each piece of highlighted geometry by clicking in the background of the graphics window.
5. Select the first protrusion listed in the model tree, then right-click and select **Edit** from the list. Notice how the protrusion geometry highlights on the model and the protrusion dimensions are displayed.

6. De-select the highlighted geometry by clicking in the background of the graphics window.

7. Practice this technique by selecting other features in the model tree. De-select each piece of highlighted geometry by clicking in the background of the graphics window.

8. Move the cursor over the model until the F5 (PROTRUSION) highlights, and then select the feature. Without moving the cursor, right-click and select **Edit** from the list. Notice how the protrusion geometry highlights on the model and the protrusion dimensions are displayed.

9. De-select the highlighted geometry by clicking in the background of the graphics window.

> **Note:**
> You have now learned three techniques for selecting geometry and showing model dimensions. You can select geometry on the model, then double-click, or right-click and edit to see dimensions, or you can select features in the model tree and right-click and edit to see dimensions.

Step 2. Learn how to rotate, zoom, and pan a model in the graphics window.

All view manipulation and orientation can be accomplished by using the middle mouse button.

To rotate the model, middle-click and drag the model. The model will update as we move the mouse.

To zoom the model, press CTRL, middle-click and drag the model up and down.

To turn about an axis normal to the graphics window, press CTRL + middle-drag left or right.

To pan the model, press SHIFT, middle-click and drag the model.

The benefit of this functionality is that we can now quickly orient models using the middle mouse button. To return the model to its starting orientation we set the view to ***default*** *using the drop-down saved view icon at the top of the screen.*

You should practice orientating the model on the screen during this exercise.

You should refer to the quick reference guide for information on selecting geometry and orientating models.

1. To rotate, move the cursor over the model and middle-drag.

2. To zoom, move the cursor over the model, press CTRL + middle-drag up or down.

3. To turn about an axis normal to the graphics window, move the cursor over the model, press CTRL + middle-drag left or right.

4. To pan, move the cursor over the model, press SHIFT + middle-drag.

5. To return to the default view click on the Saved View List icon in the toolbar and set the view to **Default**.

6. Practice the rotate, zoom, and pan techniques until you are comfortable with them.

> **Note:**
> You can always refer to the printed Quick Reference guide to remind you of the rotate, zoom and pan techniques.

7. Click **Window > Close** to close the window when finished.

Step 3. Create a new workpiece part for the mold cavity, by creating an extruded block to represent the stock material.

We are ready to create the workpiece by creating a new part; this provides us with an excellent opportunity to learn how to sketch shapes using the Sketcher tool. This important skill can be used when creating workpieces and also when creating other types of geometry such as mill windows and mill surfaces.

We start by creating a new part. It's a good idea to relate the part name to the reference model; in this case we name it ***`mold_cavity_wp`.*** *We do not use the default template because this creates a part with imperial dimensions; instead we choose to create a part with metric units, achieved by selecting* ***mmns_part_solid*** *as our template part.*

Template parts are useful because they allow us to customize the content of new parts with items such as specified units, layers, and named views.

Notice how the part appears on screen with datum planes and a co-ordinate system already created. We can now sketch our extruded block by using the extrude tool. We decide to sketch our block outline on the front datum plane.

Working in the Sketcher environment enables us to sketch the shape we want to extrude. We can sketch a rectangle positioned symmetrically about the right and front datum planes. At this point we can modify the size and position of our rectangle. We need to make the overall width and length equal to 210, and we can also add dimensions to ensure the rectangle remains symmetrically positioned about the datum planes. We can complete the sketch and enter a value of 45.0 for the depth of the block. When satisfied with the modifications, we can complete the extruded block.

1. Click **File > New** and create a new part called [`mold_cavity_wp`]. Clear the Use Default Template check box, click **OK**, and use the **mmns_part_solid** template.

> **Note:**
> Using the **mmns_part_solid** template creates a part based on metric units.

2. Turn on the display of datum planes by toggling the Datum Planes On/Off icon.

3. Create an extruded block larger than the mold cavity. Click the Extrude Tool icon on the right side of the graphics window. In the dashboard, click the Activate Sketch icon to start the Sketcher tool. Select the TOP plane as the sketching plane.

> **Note:**
> Notice the RIGHT default datum plane is automatically selected as the right orientation reference. There is no need to select an orientation plane for the sketch.

4. Click the Sketch button to begin sketching.

5. **Close** the References dialog box and accept that not enough references have been specified.

6. Create a rectangular outline approximately symmetrical about the datum planes as shown. Modify the dimensions to the values shown in following figure, and complete the sketch.

7. Click the Saved View List icon at the top of the graphics window and select the **Default** view.

8. In the dashboard, make the depth of the block [**45.0**] *(1.77 in.)*, and complete the feature.

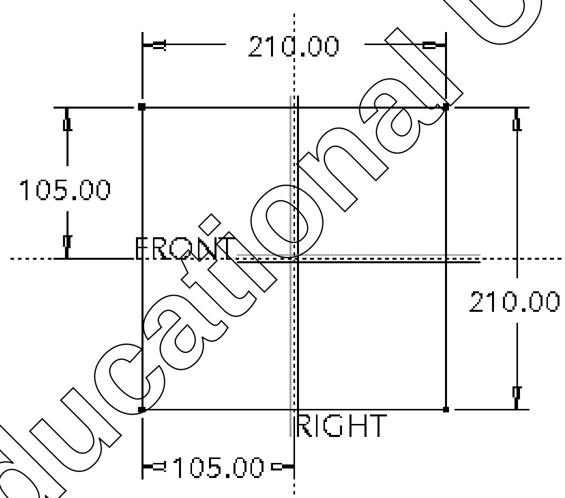


Figure 6: Sketched outline with dimensions.

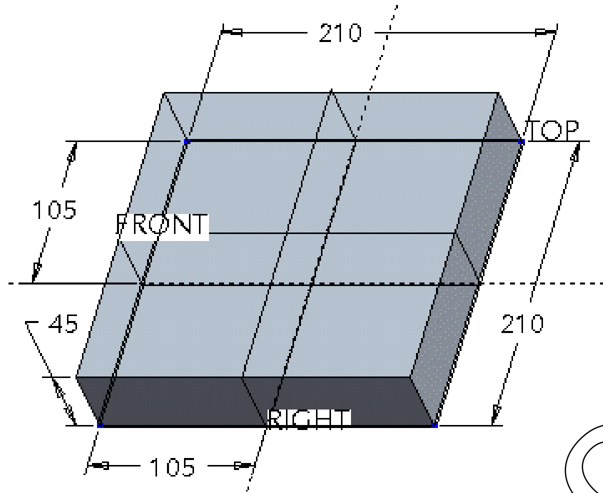


Figure 7: Extruded block to a depth of 45.0 *(1.77 in.)*.

9. Click **File > Save** to save the model.

10. Click **Window > Close** to close all windows

11. Click **File > Erase > Not Displayed > OK** to erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the purpose of workpieces.
- Create NC model assemblies consisting of reference models and workpieces.
- Create workpieces using the Create Stock dialog box.
- Create workpieces using template parts.
- Reorient models and select model geometry.

Module

3

Creating Manufacturing Models

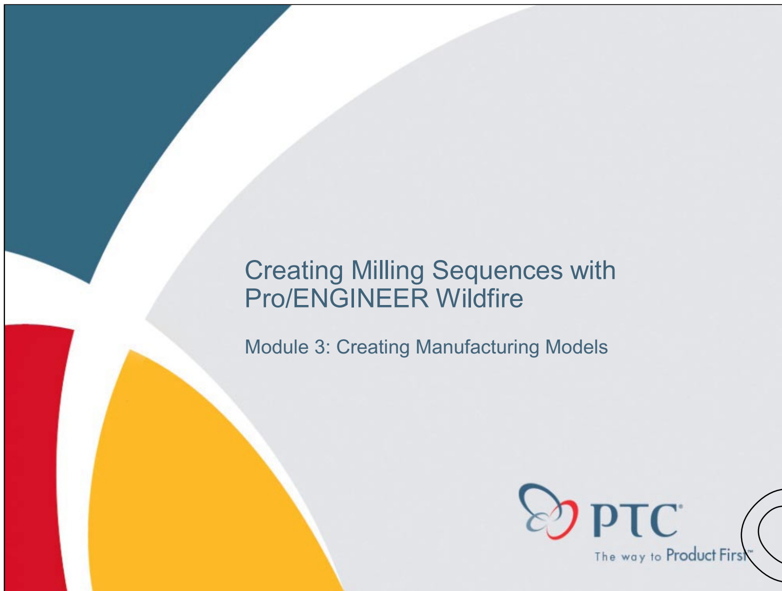
Introduction

The next step in the manufacturing process involves creating manufacturing models. Manufacturing models typically consist of a reference model, and a workpiece assembled together. You can create manufacturing models using two different methods, depending on your requirements. Manufacturing models are essential to the manufacturing process; containing reference models, workpieces, and all the other elements that make up the manufacturing process such as fixtures, operations, and NC sequences.

Objectives

After completing this module, you will be able to:

- Describe the components that make up manufacturing models.
- Create manufacturing models by assembling an NC model assembly.
- Create manufacturing models by assembling a reference model and workpiece.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

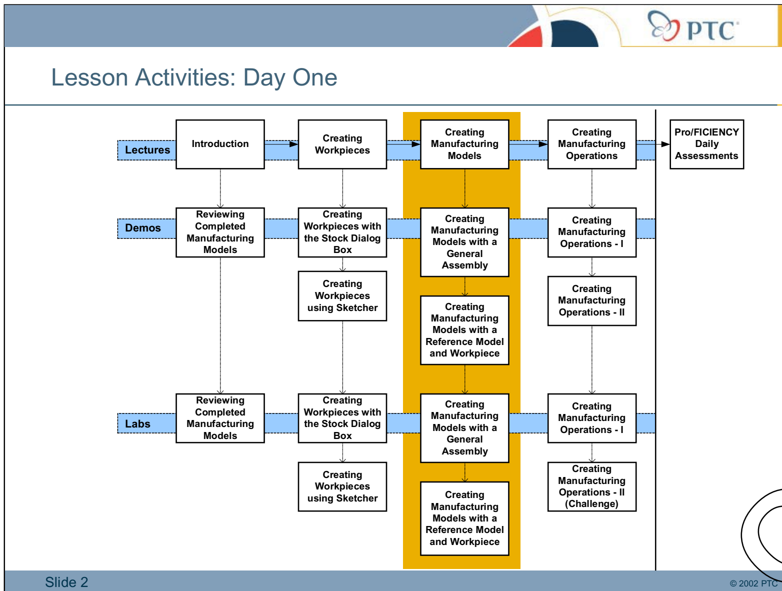
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

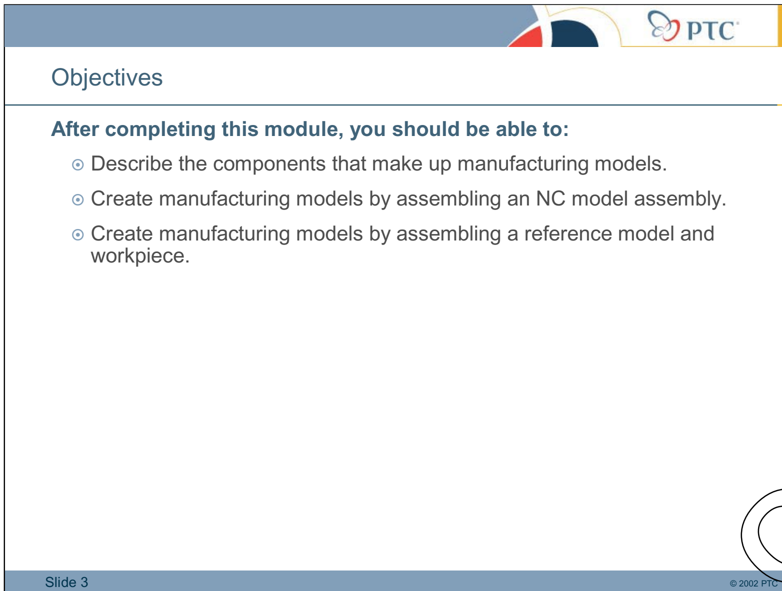
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 20 mins
- Demos : 20 mins
- Labs : 30 mins
- Total: 1 hr 15 mins

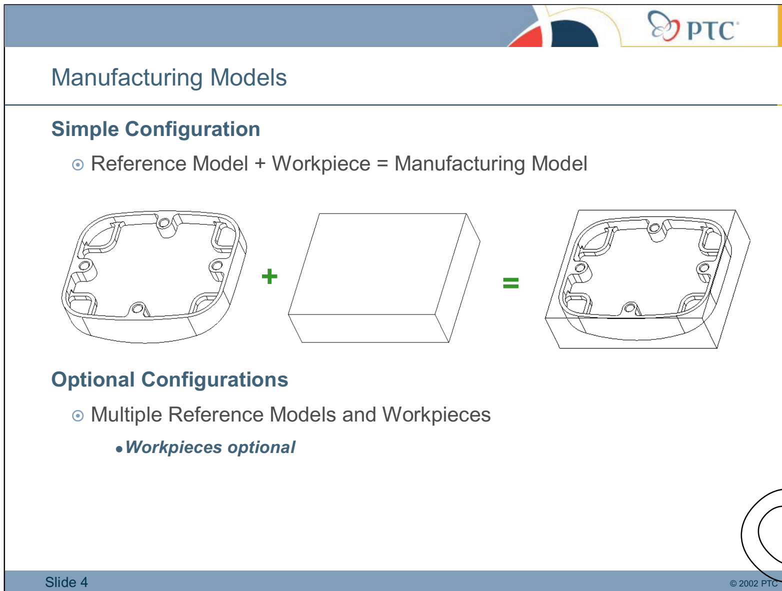


Overview

The next step in the manufacturing process involves creating manufacturing models. Manufacturing models typically consist of a reference model, and a workpiece assembled together. You can create manufacturing models using two different methods depending on your requirements. Manufacturing models are essential to the manufacturing process, containing reference models, workpieces, and all the other elements that make up the manufacturing process such as fixtures, operations, and NC sequences.

After successfully completing this module, you will be able to:

- Describe the components that make up manufacturing models.
- Create manufacturing models by assembling an NC model assembly.
- Create manufacturing models by assembling a reference model and workpiece.



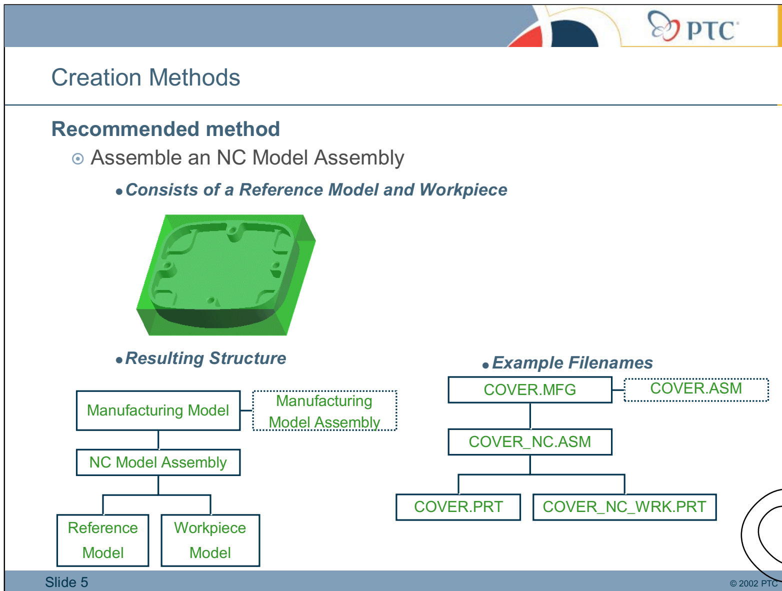
Manufacturing Models

Manufacturing models typically consist of a reference model (representing the final designed component) and a workpiece assembled together. The workpiece represents the un-machined stock material.

If you are not concerned with simulating the machining of the workpiece or creating an as-machined version of the workpiece, then you do not need to use a workpiece. Therefore, the minimum configuration of a manufacturing assembly is one reference part.

Depending on your machining needs, the manufacturing model can be an assembly of any level of complexity, and can contain any number of independent reference models and workpieces. It can also contain other components that may be part of the manufacturing assembly, but have no direct effect on the actual machining process (for example, fixtures or clamps).

Manufacturing models are essential to the manufacturing process, containing reference models, workpieces and all the other elements that make up a manufacturing process such as fixtures, operations, and NC sequences.



Creating Manufacturing Models

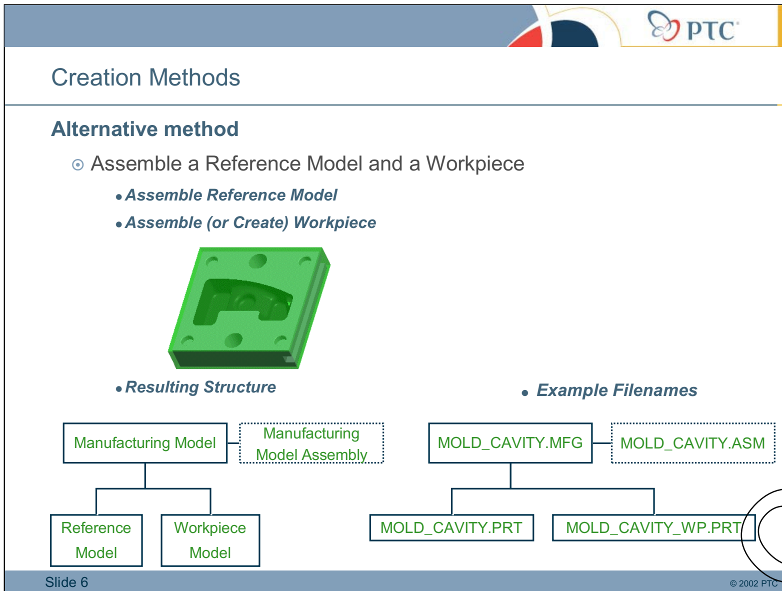
There are two methods for creating manufacturing models.

The recommended method is to assemble a previously created NC model assembly. The NC model assembly can be retrieved directly into the manufacturing model, there is no need to position components within the assembly, and the models are automatically classified correctly as a reference model and a workpiece model.

When a manufacturing model is created in this way, it consists of the following separate files:

- The manufacturing model, often called the process file, contains all the manufacturing process information such as operation, workcell, and NC sequence information. It has the format **"manufacturing model name".mfg**
- The manufacturing assembly, this file is created automatically and stores the assembly information for the manufacturing model. It has the format **"manufacturing model name".asm**
- The NC model assembly, this file is already created during the creation of the NC model assembly, and contains the reference model and the workpiece. It has the format **"NC model name".asm.**
- The design model, it has the format **"reference model name".prt.**
- The workpiece it has the format **"workpiece model name".prt.**

The diagrams shows the resulting structure, and example file names.



Creating Manufacturing Models

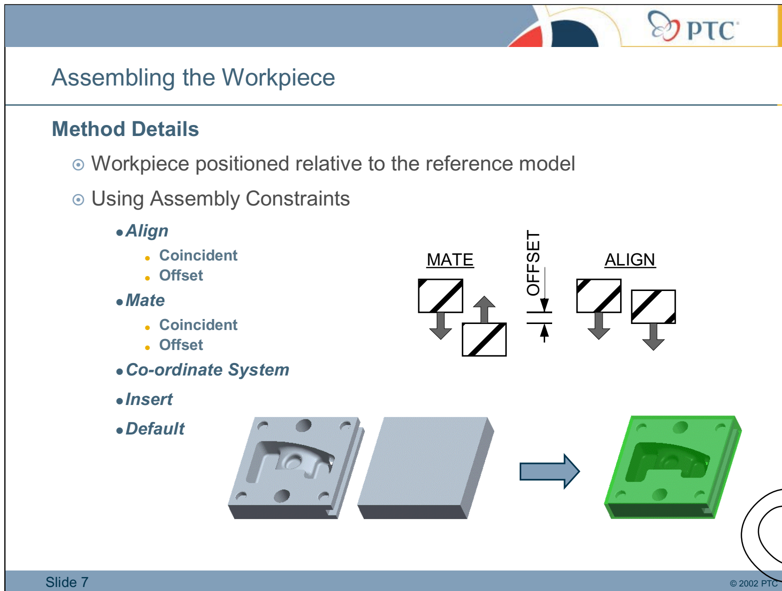
An alternative method is to create the manufacturing model by assembling a reference model, and then assembling (or creating) a workpiece.

The reference model is retrieved first and the workpiece model is positioned relative to the reference model.

The diagrams shows the resulting structure, and example file names.

When a manufacturing model is created in this way, it consists of the following separate files:

- The manufacturing model - often called the process file, contains all the manufacturing process information such as operation, workcell, and NC sequence information. It has the format **”manufacturing model name”.mfg**
- The manufacturing assembly - created automatically and stores the assembly information for the manufacturing model. It has the format **“manufacturing model name”.asm**
- The design model - it has the format **“reference model name”.prt.**
- The workpiece - it has the format **“workpiece model name”.prt.**

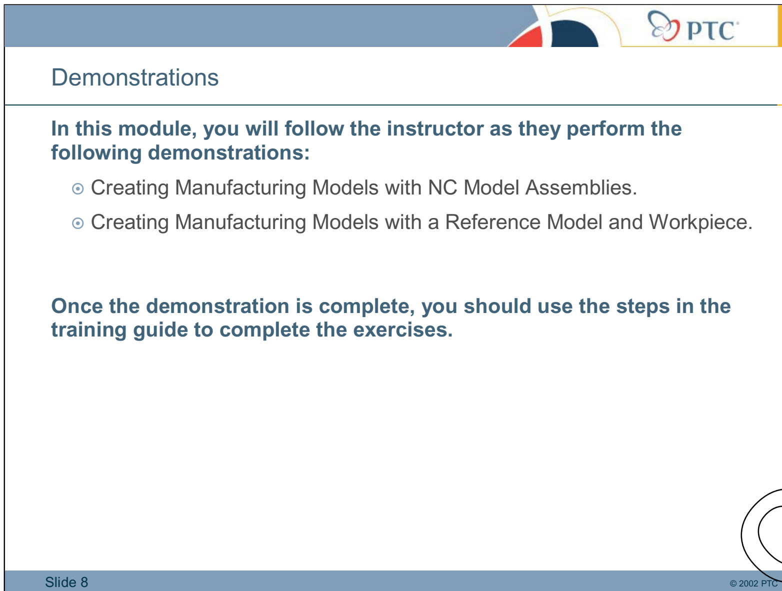


Assembling the Workpiece

To assemble the workpiece you need to uniquely configure the position of the workpiece relative to the reference model.

This is achieved by using a series of assembly constraints.

- Align: Used to make two selected surfaces **face in the same direction**. The surfaces can be coincident or offset from each other by a distance.
- Mate: Used to make two selected surfaces **face in the opposite direction**. The surfaces can be coincident or offset from each other by a distance.
- Co-ordinate system: make two selected co-ordinate systems coincident.
- Insert: used for shafts and holes, you can insert a shaft into a hole. The hole and shaft axis become co-axial.
- Default: place the workpiece at the center (default position) of the manufacturing model.



Demonstrations:

- Creating Manufacturing Models with NC Model Assemblies.
- Creating Manufacturing Models with a Reference Model and Workpiece.

Scenario

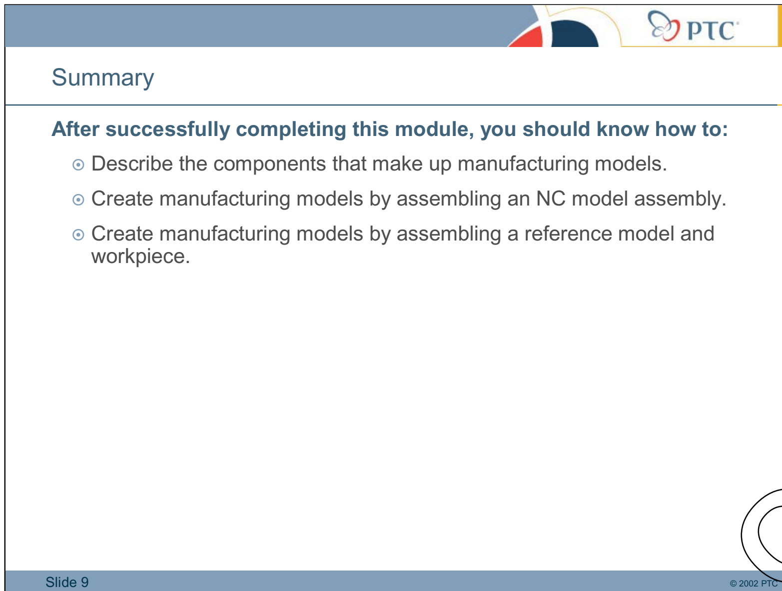
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the components that make up manufacturing models.
- Create manufacturing models by assembling an NC model assembly.
- Create manufacturing models by assembling a reference model and workpiece.

Module 3 Lab Exercises

Exercise 1: Creating Manufacturing Models with NC Model Assemblies

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you will show the recommended method for creating manufacturing models. This involves creating a manufacturing model and assembling a previously created NC model assembly consisting of a reference model and a workpiece. As the assembly is retrieved into the manufacturing model, the reference model and workpiece are automatically classified correctly. An alternative method will be demonstrated in the next exercise. The ability to create manufacturing models using different methods provides you with a flexible approach to machining.

Manufacturing models are essential to the manufacturing process, containing reference models, workpieces and all the other elements that make up a manufacturing process such as fixtures, operations, and NC sequences.

Objectives

After successfully completing this exercise, you will know how to:

- Create manufacturing models by assembling NC model assemblies.

Scenario

The next step in the manufacturing process typically involves creating manufacturing models. In this exercise, you create a manufacturing model for the aluminum cover. You use the recommended method for creating manufacturing models; this involves creating a manufacturing model and assembling a previously created NC model assembly consisting of a reference model and a workpiece.

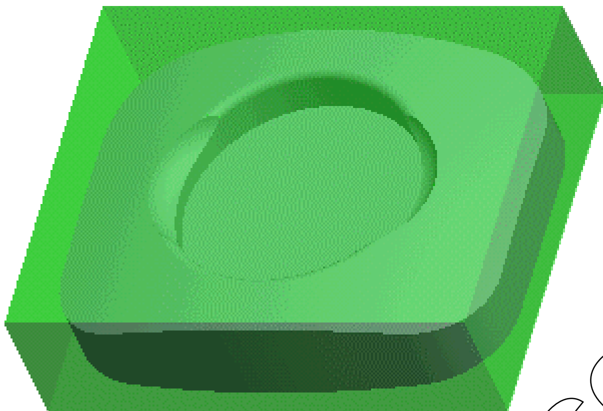


Figure 1: The default view of the NC model assembly for aluminum cover.

Step 1. Create a new manufacturing model for the aluminum cover.

*We can now create the manufacturing model for the aluminum cover. We use the recommended method for creating manufacturing models. We start by creating a manufacturing NC assembly for the aluminum cover; a sensible naming convention should be used, so name the manufacturing model **cover**.*

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module03**, right-click, and select **Make Working Directory.**
2. Create a new manufacturing model for the aluminum cover, click **File > New**, select the **Manufacturing** and **NC Assembly** options, and name it [**cover**].

Step 2. Assemble in the aluminum cover NC model assembly as a general assembly.

*We can now assemble the previously created NC model assembly containing the aluminum cover and the workpiece for the cover. Using the Menu Manager, we can assemble a general assembly, from the list of files select **cover_nc.asm.** Notice when the assembly is retrieved into the manufacturing model; the reference model and workpiece are automatically classified correctly.*

1. Using the Menu Manager, select **Mfg Model > Assemble > Gen Assem**, and assemble COVER_NC.ASM into the manufacturing model.

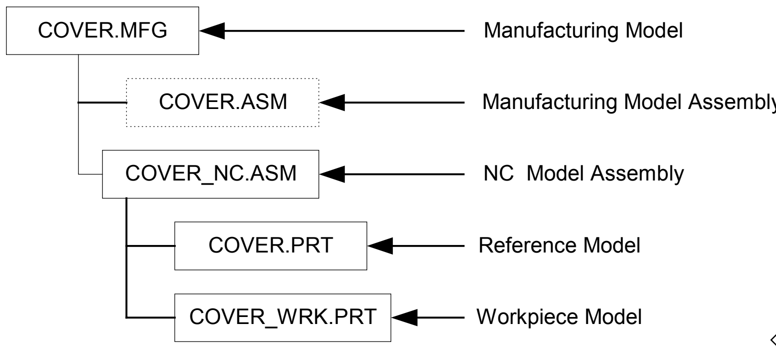


Figure 2: Resulting model structure and filenames.

2. Save the manufacturing model.

3. Close all windows and erase all components from memory.

> **Hint:**
> Click **File > Erase > Not Displayed > OK**, to erase all components from memory.

Exercise 2: Creating Manufacturing Models with a Reference Model and Workpiece

Introduction

In this demonstration, you will use an alternative method for creating manufacturing models. This involves creating a manufacturing model by assembling a reference model, then assembling a workpiece. The ability to create workpieces using different methods provides you with a flexible approach to workpiece creation.

Objectives

After successfully completing this exercise, you will know how to:

- Create manufacturing models with a reference model and a workpiece.

Scenario

You need to create a manufacturing model for the mold cavity. You will use an alternative method that involves creating a manufacturing model by assembling a reference model (the mold cavity), then assembling a workpiece.

Step 1. Create a manufacturing model for the mold cavity.

We can create the manufacturing model for the mold cavity using an alternative method. We start by creating a manufacturing NC assembly for the mold cavity; a sensible naming convention should be used so name the manufacturing model **mold_cavity**.

Using the Menu Manager we can assemble a reference model, from the list of files select ***mold_cavity.prt***.

We can now assemble the workpiece; from the list of files select ***mold_cavity_wp.prt.*** *Notice how the workpiece appears next to the reference model in the graphics window.*

To assemble the workpiece we need to uniquely configure the position of the workpiece relative to the reference model. This is achieved by creating positioning constraints.

We make sure co-ordinate systems are visible before configuring any constraints.

To create our first constraint we select the DEFAULT co-ordinate system on the reference model and the PRT_DEF_CSYS co-ordinate system on the workpiece. Pro/ENGINEER Wildfire automatically assumes these co-ordinate systems will be coincident. This is the only constraint we need. The workpiece has been positioned correctly and we can complete the assembly process.

1. Create a new manufacturing model for the mold cavity, click **File > New**, select the **Manufacturing** and **NC Assembly** options, and name it [`mold_cavity`].

2. Turn on the display of datum co-ordinate systems, if not already visible in the graphics window.

3. Using the Menu Manager, select **Mfg Model > Assemble > Ref Model** and assemble the MOLD_CAVITY.PRT as the reference model.

4. In the Mfg Model menu, select **Assemble > Workpiece** and select the MOLD_CAVITY_WP.PRT as the workpiece.

5. In the Component Placement dialog box, change the constraint type from Automatic to **Coord Sys**, then select the co-ordinate system on the workpiece and the co-ordinate system on the reference model, as shown in the figure below.

6. Complete the component placement, and return to the Manufacture menu.

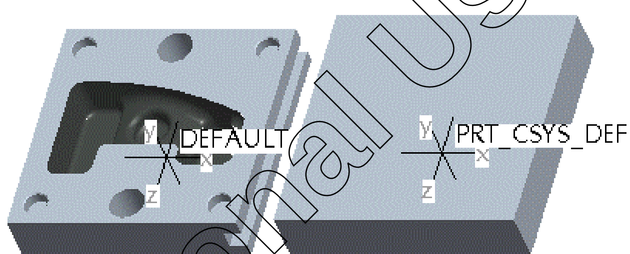


Figure 3: Reference model and workpiece showing co-ordinate systems selected.

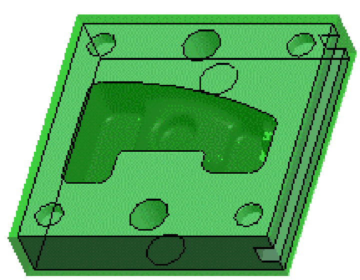


Figure 4: Reference model and workpiece after assembly.

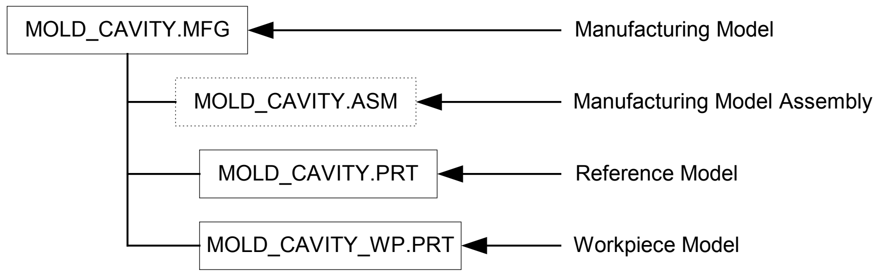


Figure 5: Resulting model structure and filenames.

7. Save the manufacturing model

8. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the components that make up manufacturing models.
- Create manufacturing models by assembling an NC model assembly.
- Create manufacturing models by assembling a reference model and workpiece.

Module

4

Creating Manufacturing Operations

Introduction

The next step in the manufacturing process involves configuring manufacturing operations. Manufacturing operations are created by configuring a number of required elements such as the type of machine tool (workcell), and the machine zero position. In addition, a series of optional elements can be configured, such as the retract surface, workpiece material, and fixtures. Once an operation is configured, NC sequences can be created that reference the operation's specific machine tool (workcell) and machine zero position. It is important to understand why each element in an operation is required and when it is appropriate to configure optional elements.

Objectives

After completing this module, you will be able to:

- Describe the elements that make up manufacturing operations.
- Configure selected elements of manufacturing operations.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

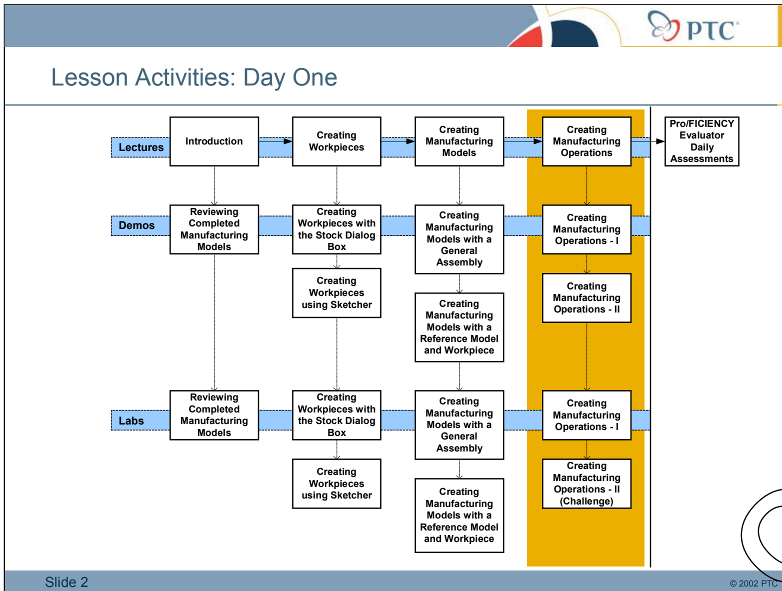
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

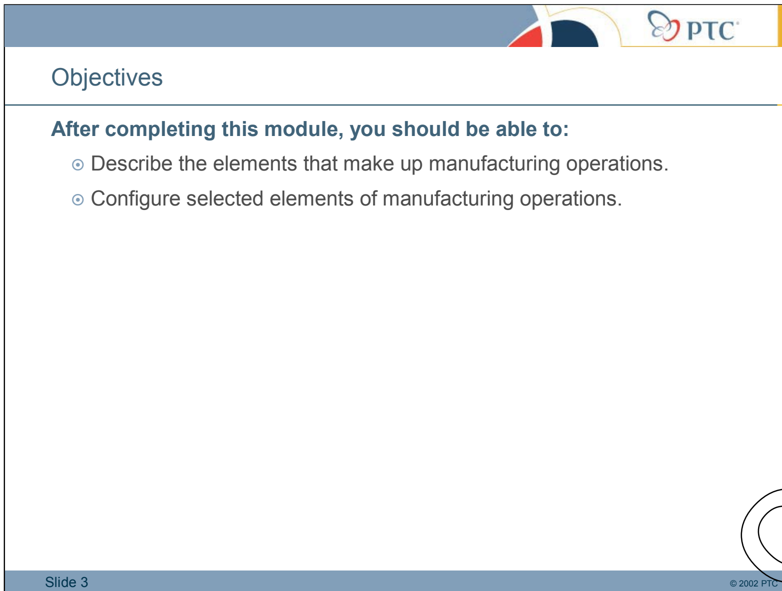
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 25 mins
- Labs : 50 mins
- Total: 1 hr 30 mins

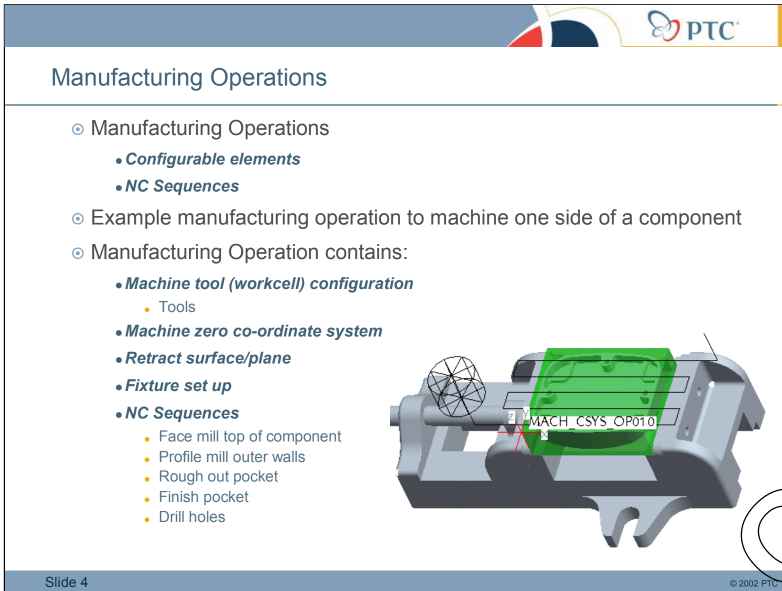


Overview

The next step in the manufacturing process involves configuring manufacturing operations. Manufacturing operations are created by configuring a number of required elements such as the type of machine tool (workcell), and the machine zero position. In addition, a series of optional elements can be configured, such as the retract surface, workpiece material, and fixtures. Once an operation is configured, NC sequences can be created that reference the operation's specific machine tool (workcell) and machine zero position. It is important to understand why each element in an operation is required and when it is appropriate to configure optional elements.

After successfully completing this module, you will be able to:

- Describe the elements that make up manufacturing operations.
- Configure selected elements of manufacturing operations.



Manufacturing Operations

Manufacturing operations are made up of a number of configurable elements such as machine tools, machine zero positions, and fixtures.

They also contain NC sequences that reference the configured machine tool (workcell) and the defined machine zero position.

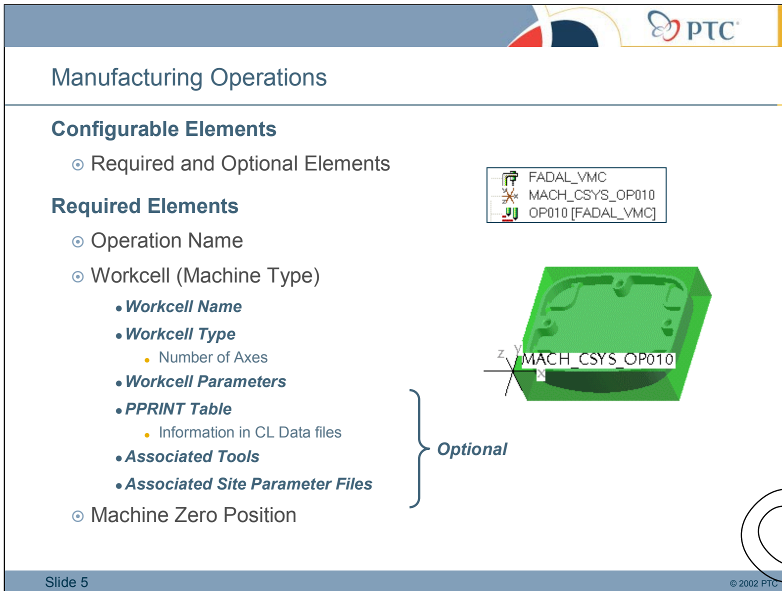
For example, you can create an operation to machine one side of a component.

The operation consists of a number of configured elements such as:

- The machine tool (workcell) configuration.
- The machine zero position (configured with a co-ordinate system) - this also defines the direction of the x,y and z axis on the machine tool.
- A retract surface which the tool retracts to between NC sequences.
- A fixture set up - in this case a vice assembly.

NC sequences can then be created that machine the component. These sequences all reference the machine tool and the machine zero position. NC sequences cannot be created until the manufacturing operation is configured.

In summary, manufacturing operations are defined as a series of NC sequences performed by a specific machine tool (workcell) and referencing a specific machine zero position.

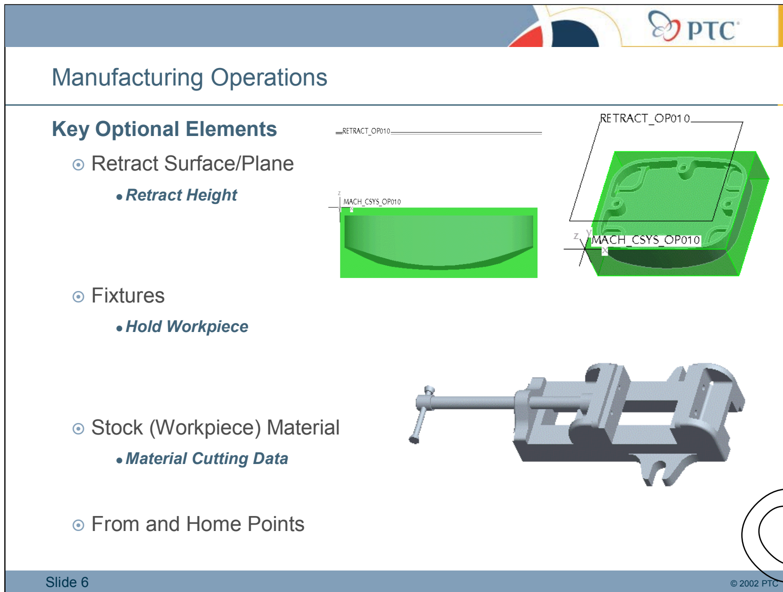


Manufacturing Operations

The configurable elements within a manufacturing operation, can be divided into required and optional.

Required Elements

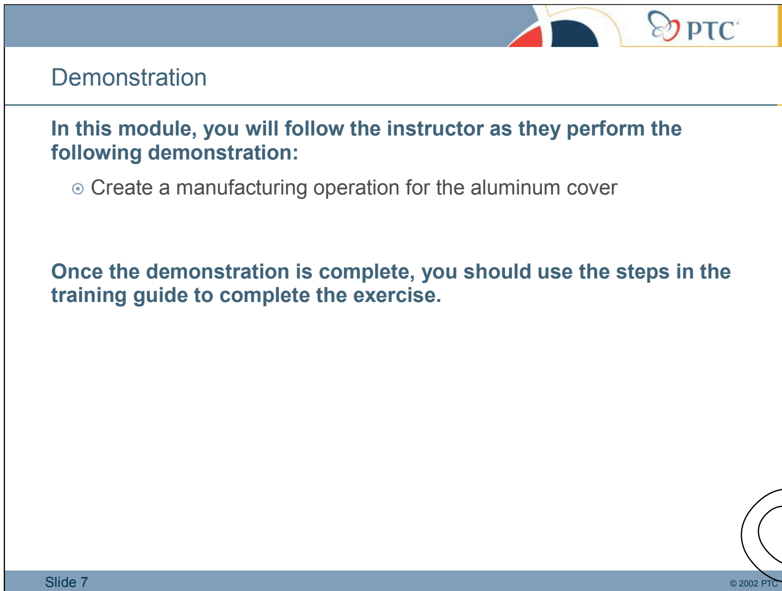
- **Operation name** - when configuring an operation you must provide a name for identification.
- **Workcell (Machine Tool) -** You must configure the workcell to be used, there are many options, that can be configured within a workcell
  - **Workcell Name**
  - **Workcell Type** - this can be Mill, Lathe, Mill/Turn, or Wire EDM.
    - Depending on the workcell type the number of axes can be specified
    - For Mill—3 Axis (default), 4 Axis, or 5 Axis.
    - For Lathe—1 Turret (default) or 2 Turrets.
    - For Mill/Turn—2 Axis, 3 Axis, 4 Axis, or 5 Axis (default).
    - For Wire EDM—2 Axis (default) or 4 Axis.
  - **Optional Workcell Items**
  - **Workcell Parameters** - there are numerous parameters such as
    - Tool travel limits
    - Maximum spindle speed
    - The name of the default post-processor associated with the machine.
  - The **PPRINT table** - enables machining information such as 'operation name' to be output into CL data files. The table uses yes/no flags to determine if the information is output.
  - Any **tools** created can be associated with a specific workcell (machine tool)
  - If a global **site parameter file** has been created it can be associated with the workcell. Site parameter files control the default manufacturing parameter values used in NC sequences. For example, you could set the default spindle speed.
- The **machine zero position** is configured by creating or selecting a co-ordinate system. It acts as the origin (0,0,0), for CL data output, and defines the direction of the x,y and z axes.



Key Optional Elements

Here are some of the more important optional elements:

- The **retract surface (or plane)** specifies the level to which the tool retracts to between machining. It can be created during the configuration of the operation or later when creating an NC sequence. It is created by creating an infinite datum plane in the manufacturing model.
- **Fixtures** are parts or assemblies that help orient and hold the workpiece during a manufacturing operation. They are useful because they can be easily activated and de-activated.
- If you specify the **stock (workpiece) material**, you can automatically load in cutting data that will set default values for spindle speed, cutting feed, cutting depth, and tool step over.
- If configured, **From and Home points** serve as start and finish positions for NC sequences.



Demonstration:

- Create a manufacturing operation for the aluminum cover.

Scenario

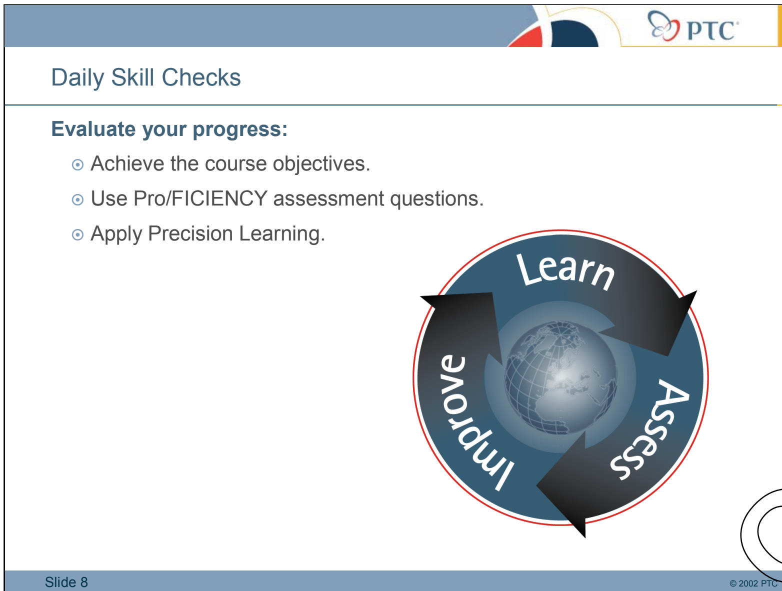
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY.
- **Improve:** The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Getting Started

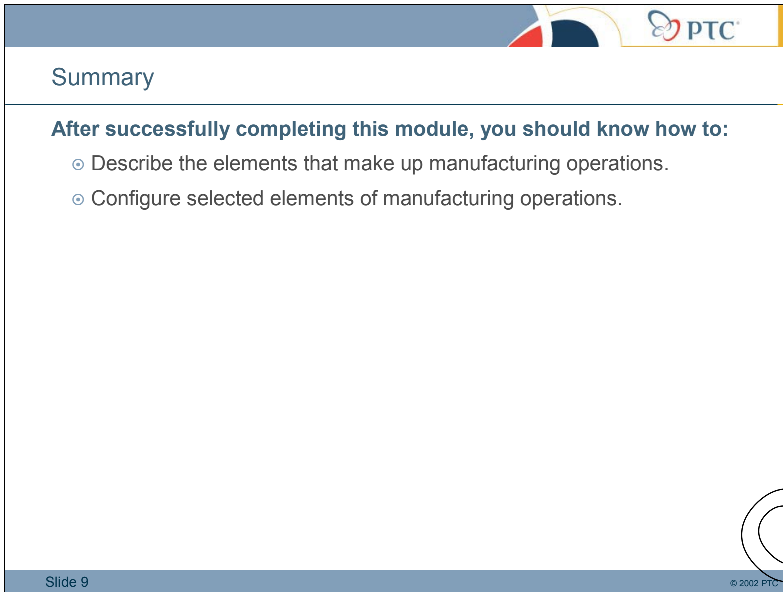
- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Tests

Description: For each course, 5 new 10 question exams based upon the topics covered each day.

How to use it:

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After successfully completing this module, you should know how to:

- Describe the elements that make up manufacturing operations.
- Configure selected elements of manufacturing operations.

Module 4 Lab Exercises

Exercise 1: Creating a Manufacturing Operation for the Aluminum Cover

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you configure selected elements of a manufacturing operation to enable machining of the aluminum cover. This involves creating a workcell, a machine zero, a retract surface, specifying the workpiece material, creating a PPRINT table, and setting up a fixture. An additional challenge exercise is available for students to configure a manufacturing operation for the mold cavity.

Manufacturing operations are essential to the manufacturing process. They contain many elements that make up a manufacturing process such as workcells, machine zeros, and fixtures. NC sequences cannot be created until the manufacturing operation is configured.

Objectives

After successfully completing this exercise, you will know how to:

- Configure selected elements of manufacturing operations.

Scenario

The next step in the manufacturing process typically involves creating manufacturing operations. To enable machining of the aluminum cover, you set-up a manufacturing operation in the previously created manufacturing model of the aluminum cover. This involves creating a workcell, a machine zero, a retract surface, specifying the workpiece material, creating a PPRINT table, and setting up a fixture. To assist in viewing the manufacturing model in different positions you also create a series of named views.

Step 1. Open the manufacturing model for the aluminum cover

We can locate the folder for this exercise and set this as our working director. We can then open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module04**, right-click, and select **Make Working Directory.**

2. Click **File > Open** to open the manufacturing model for the aluminum cover: COVER.MFG.

Step 2. Create a named view, then create an operation, configure a 3-axis milling workcell, create a PPRINT table, and configure the stock material.

We can re-orient the model to view the underside; this is the side we intend to eventually machine. We can also save this view for future reference. Using the Menu Manager, we can open the Operation dialog box, at this point we can type the name of our machine tool, and create a PPRINT table. PPRINT tables are important because we can output useful manufacturing information to CL data files by changing the option flags to yes.

At this point, we can specify the workpiece material for the operation, in this case aluminum. Configuring the workpiece material at this stage will enable us to set default tool feeds, tool speeds, and cutting depths when creating NC sequences. These default values are based on workpiece material and roughing or finishing cutting conditions.

1. To see the underside of the model, rotate the model by 180° around the x-axis of the spin center, click **View > Orientation > Reorient**. In the Orientation dialog box, change the Type drop-down list from Orient by Reference to **Dynamic Orient**. Then click the Spin Center Axis icon, and type [**180**] as the x-axis rotation angle.

2. Expand the Saved Views interface, type [**bottom-tri**] in the Name text box, and click the **Save** button. Close the Orientation dialog box.

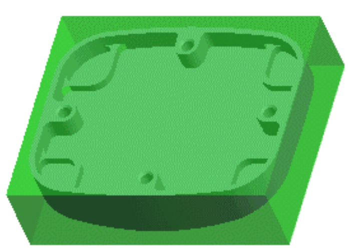


Figure 1: Model rotated by 180° around the x-axis of the spin center.

3. Using the Menu Manager, select **Mfg Setup**, and open the Operation Setup dialog box.

4. Click the Machine Tool Dialog icon, and open the NC Machine Tool Setup dialog box, type [**fadal_vmc**] in the Machine Name text box.

5. Create a PPRINT table; click the PPRINT option, then select **Create** to open the PPRINT window. Click the **DATE_TIME** item, and click the **Yes** option at the bottom of the window. Change the following items to yes.

Table 1: PPRINT Table Options

Item	Option
DATE_TIME	Yes
OPERATION_COMMENTS	Yes
TOOL_TABLE	Yes
ONLY_OUTPUT_USED_TOOLS	Yes
NC_SEQUENCE_NAME	Yes
NC_SEQUENCE_COMMENTS	Yes

6. Close the PPRINT window, then **Save** the PPRINT table (accept the default name), select **Done/Return**, and close the Machine Tool Setup dialog box.

7. In the Operation Setup dialog box, select **aluminum** in the Stock Material drop-down list.

Step 3. Create the machine zero co-ordinate system and the retract surface.

We can create the machine zero position, ensuring that we create the co-ordinate system in the cover assembly by selecting it in the model tree. Selecting three surfaces on the workpiece, we can position the co-ordinate system in the front-left-top corner of the workpiece. We can set the direction of the axes to match our requirements, and rename the co-ordinate system to make it easier to identify. We can also create a retract surface and position it relative to our machine zero position. When the operation definition is complete, we can rename the retract surface in the model tree to make it easier to identify.

The machine zero position acts as the origin for CL data output, and defines the direction of the x, y and z axes. The retract surface specifies the level to which the tool retracts to between cutting.

Note:
Turn off the display of the blended background in the graphics window to make viewing of co-ordinate system axes easier.

1. Click **View > Display Settings > System Colors**, clear the Blended Background check box, and click **OK** to complete the configuration.

2. Turn on the display of co-ordinate systems, if not already visible in the graphics window.

3. Create a machine zero co-ordinate system positioned at the front-left-top corner of the workpiece. In the Operation Setup dialog box, click the Machine Zero icon, then select **Create** in the MACH CSYS menu, when prompted to pick a model to create the coordinate system in, select the COVER.ASM assembly in the model tree.

4. To refresh the screen, click **View > Repaint**.

Note:

To select multiple surfaces you must press and hold the CTRL key when selecting surfaces. To select hidden surfaces you must move the cursor over the hidden surface and right-click to highlight the hidden surface.

5. Select surfaces to configure the co-ordinate system origin. Press and hold CTRL, and select the front and top surfaces on the workpiece as shown in the following figure.

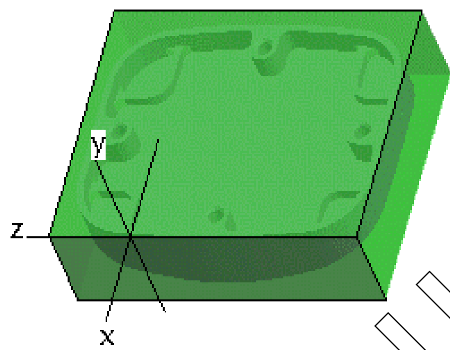


Figure 2: Showing selected front and top surfaces.

6. To select the hidden left surface, move the cursor over the model as shown in following figure, then press CTRL + right-click, when the hidden left surface highlights select it.

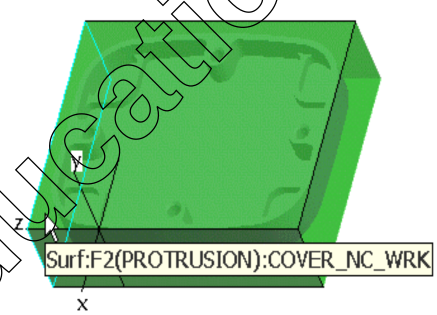


Figure 3: Showing selected left surface highlighted.

Hint:

The co-ordinate system origin will move to the front-top-left corner of the workpiece when all three surfaces have been selected correctly. (As shown in the following figure).

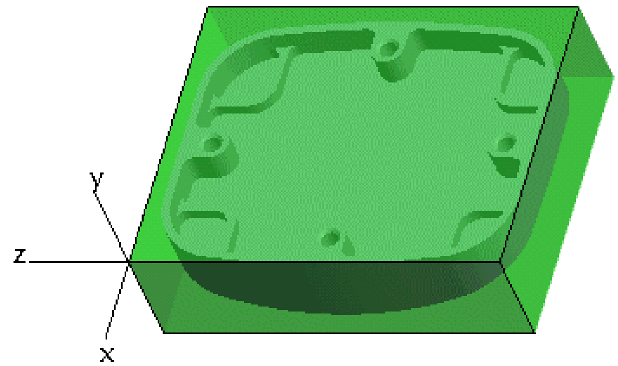


Figure 4: Showing position of co-ordinate system after all three surfaces have been selected.

7. In the Coordinate System dialog box, select the Orientation tab. Use the drop-down axis options and the Flip button to modify the orientation of the axes as shown in the following figure.

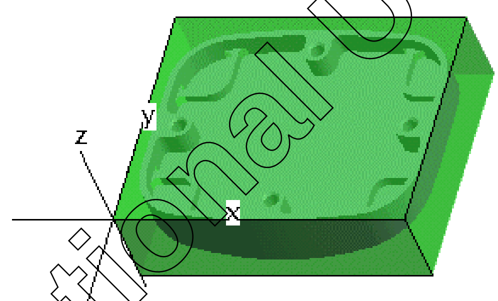


Figure 5: Showing re-oriented X, Y and Z axes.

8. In the Coordinate System dialog box, select the Properties tab, and rename the co-ordinate system to [`mach_csys_op010`]. Close the Coordinate System dialog box when finished.

9. In the Operation Setup dialog box, click the Retract Surface icon, select the Along Z Axis option and type [**40**] *(1.57 in.)* as the z distance. Close the Retract Selection dialog box.

10. Close the Operation Setup dialog box.

11. Select the retract surface in the model tree (ADTM1), then right-click and rename it [`retract_op010`].

12. Turn on the blended background again. Click **View > Display Settings > System Colors**, select the Blended Background check box, and click **OK** to complete the configuration.

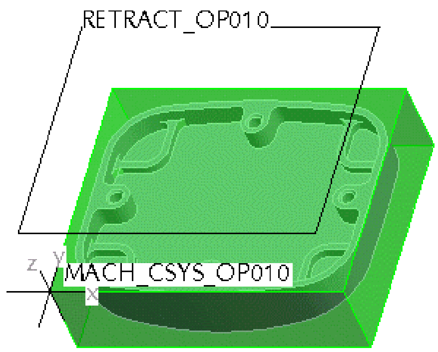


Figure 6: Machine zero co-ordinate system and retract surface.

Step 4. Create the fixture setup and assemble the vise assembly.

To clarify the view of the model, we turn off the display of all datum features. We can open the Fixture dialog box and name the fixture setup op_010. We can now add a component to the fixture setup; in this case we can select the VISE.ASM from the list. To ensure the vise assembly jaws are the correct distance apart, we select the OP010_INST from the list of instances. We can position the vise assembly using two mate constraints and an align constraint; we can also display the vise assembly in a separate window to make the assembly process easier to complete. Alternatively, we can spin the vise assembly by pressing CTRL + ALT and middle-click the mouse and drag the model.

Once assembled, we can close the fixture setup and Operation dialog boxes.

Fixtures are useful because they show how machined components are held during machining and they can be easily activated and de-activated.

1. Turn off the display of datum planes, datum axes and datum co-ordinate systems.
2. Using the Menu Manager, select **Mfg Setup > Operation** to open the Operation Setup dialog box.
3. Click the Fixture Setup icon to open the dialog box, type [**op_010**] in the Fixture Setup Name box.
4. Click the Add Component icon to add a component to the fixture setup; select **VISE.ASM** from the list of models, then select the **op010_inst** from the list of instances.

> **Tip:**
> The assembly process is easier if you do not display the vise assembly in the main graphics window when specifying assembly constraints; instead show the vise assembly in a separate window.

5. In the Component Placement dialog box, click the Show Component in a Separate Window icon, then remove the vise assembly from displaying in the graphics window by de-selecting

the Show Component in Assembly Window icon. Ensure both the manufacturing model and the vise assembly are clearly visible.

> **Note:**
> You need to create two mate constraints and an align constraint to correctly position the vise assembly.

6. Assemble the vise assembly, as shown in the following figure. In the Component Placement dialog box, change the first constraint type from Automatic to **Mate**, then select the hidden left surface on the workpiece and the small vertical surface on the clamp pad part.

7. Change the second constraint type from Automatic to **Mate**, then select the hidden bottom surface on the workpiece and the small horizontal surface on the clamp pad part.

8. Change the third constraint type from Automatic to **Align**, then select the front surface on the workpiece and the vertical front surface on the clamp pad part.

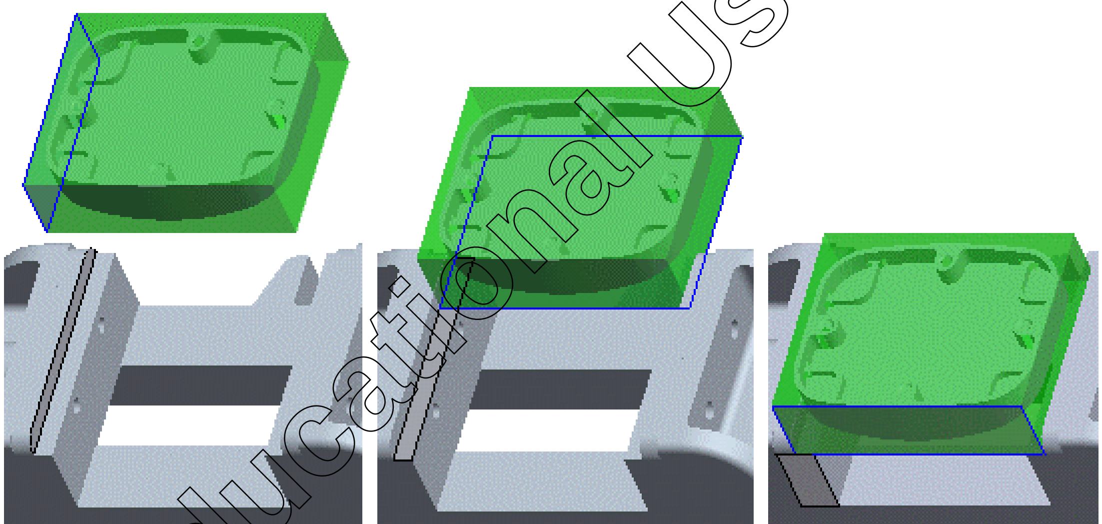


Figure 7: Showing the two mate and one align constraints; selected pairs of surfaces are highlighted.

9. When the three constraints have been created, click the Preview option in the Component Placement dialog box to check the vise has been positioned correctly. Complete the component placement when satisfied.

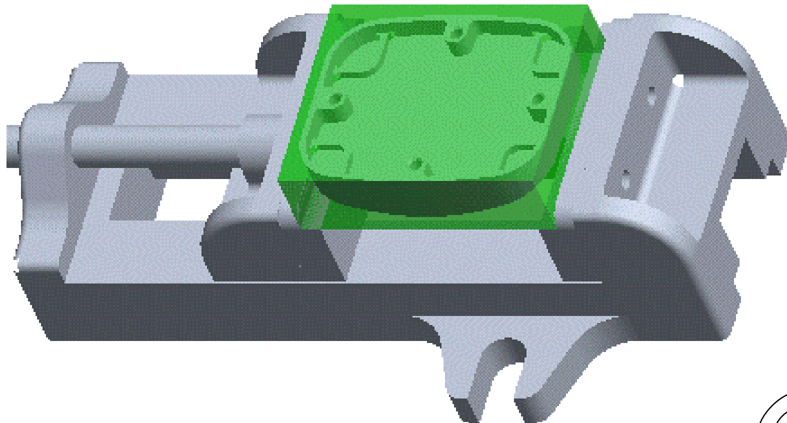


Figure 8: Final assembled position of the vise assembly

10. Close the Fixture Setup dialog box.

11. Close the Operation dialog box, and return to the Manufacture menu.

Step 5. Create a series of named views.

We can create a series of named views to assist in viewing the manufacturing model in different positions. Starting with our previously defined **bottom-tri** *named view, we can zoom in on the model and create a named view call* **`op010-tri`**. *We can also re-orient the model to create a top view called* **`op010-top`**, *and a front view called* **`op010-front`**.

1. Using the Zoom In icon in the toolbar at the top of the graphics window, zoom into the manufacturing model, as shown in the following figure.

2. To create a named view click **View > Orientation > Reorient**, in the Orientation dialog box type [**`op010-tri`**] in the Name box, and save the view.

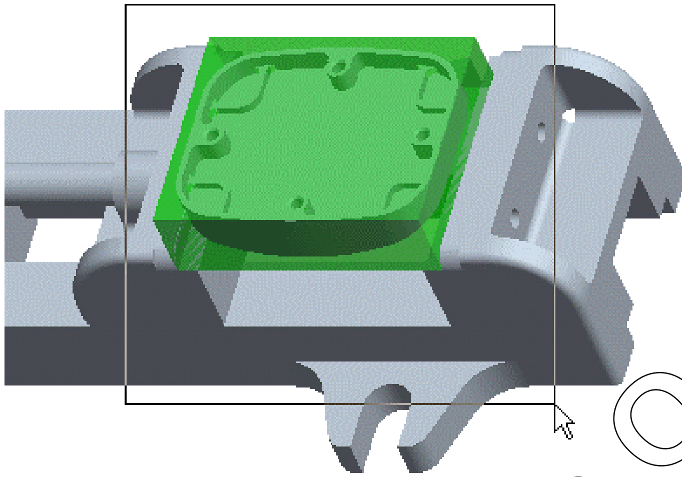


Figure 9: Showing suggested zoom window for named view op010-tri.

3. To create a top view, select the top surface of the workpiece for Reference 1, then change the Reference 2 drop-down option to **Bottom** and select the front surface of the workpiece for Reference 2, this reorients the model as shown in the following figure.

4. Using the Zoom icon, zoom into the model; type [**op010-top**] in the Name box, and save the view.

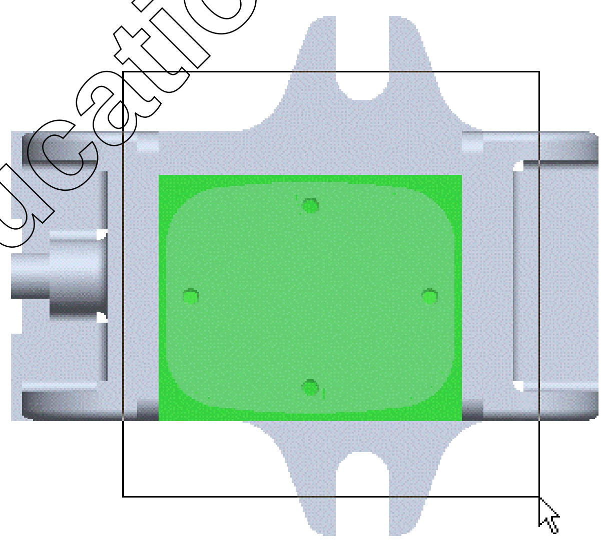


Figure 10: Showing suggested zoom window for named view op010-top.

5. In the Orientation dialog box, select **op010-tri** from the list of saved views and click Set.

6. To create a front view, click the Select Reference Icon for Reference 1, and select the front surface of the workpiece. Then change the Reference 2 drop-down option to **Right** and select the right surface of the workpiece for Reference 2.

7. Using the Zoom icon, zoom into the model; type [**`op010-front`**] in the Name box, and save the view.

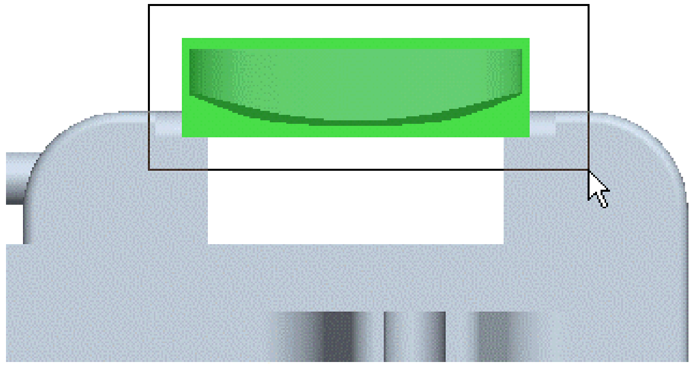


Figure 11: Showing suggested zoom window for named view op010-front.

8. Close the Orientation dialog box. Click the Saved View List icon; select the named view **`op010-tri`** from the list of views.

9. Save the manufacturing model.

10. Close all windows and erase all components from memory.

Exercise 2: Creating a Manufacturing Operation for the Mold Cavity (Challenge)

Objectives

After successfully completing this exercise, you will know how to:

- Create Manufacturing Models with a Reference Model and a Workpiece.

Scenario

To enable machining of the mold cavity, you set up a manufacturing operation in the manufacturing model for the mold cavity. You do not need to configure the same operation elements as you did for the aluminum cover. For the mold cavity, you create a workcell, a machine zero, and a retract surface (additional items will be configured during the creation of NC sequences).

Step 1. Open the manufacturing model for the mold cavity. Create an operation, and configure a 3-axis milling workcell.

1. Open the manufacturing model for the aluminum cover: MOLD_CAVITY.MFG.

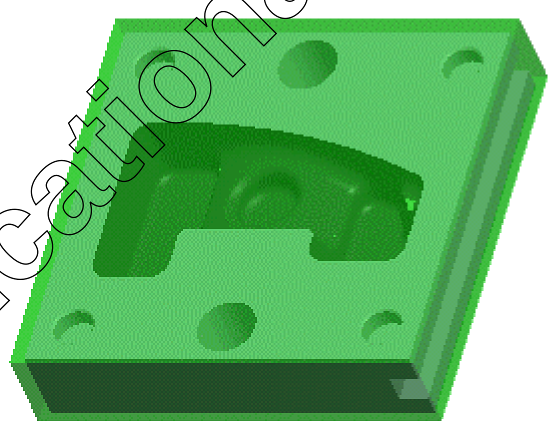


Figure 12: Mold cavity manufacturing model.

2. Using the Menu Manager, open the Operation Setup dialog box.
3. Open the Machine Tool Setup dialog box, retrieve the mazak workcell by selecting **MAZAK.GPH** from the list.

> **Note:**
> Retrieving the mazak workcell also retrieves and activates a site parameter file.

4. Close the Machine Tool Setup dialog box.
5. Select **steel-300-bhn** in the Stock Material drop-down list.

6. Turn on the display of co-ordinate systems, if not already visible in the graphics window.

7. Create a machine zero co-ordinate system positioned at the front-left-top corner of the workpiece. Set the orientation of the axes as shown in the following figure. Rename the co-ordinate system to [**`mach_csys`**].

> **Note:**
> When prompted, create the co-ordinate system in the MOLD_CAVITY.ASM assembly, and select this assembly from the model tree.

8. Create a retract surface along the Z-axis of the machine zero co-ordinate system, type [**5**] *(0.20 in.)* as the z distance.

9. Close the Operation dialog box.

10. Rename the retract surface to [**`retract_op010`**] in the model tree.

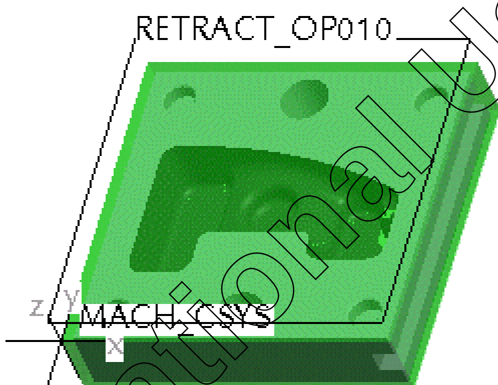


Figure 13: Machine zero co-ordinate system and retract surface.

11. Save the manufacturing model and close the window.

Summary

After successfully completing this module, you should know how to:

- Describe the elements that make up manufacturing operations.
- Configure selected elements of manufacturing operations.

Module

5

Creating Tools

Introduction

Creating tools is an essential step in the manufacturing process, as a tool must be configured for each NC sequence you create. Tools can be created when the workcell is created or they can be configured as needed for each NC sequence. Once a tool is configured the information can be stored and used again.

There are three different tool types: edited, solid, and sketched. Each type of tool is created in a different way and is designed for a specific purpose. It is important to understand the differences between each type of tool and when they should be used.

Objectives

After completing this module, you will be able to:

- Describe the differences between the different tool types: edited, solid and sketched.
- Describe how to specify tool feeds and speeds by configuring stock material.
- Create different types of edited tools through the configuration of tool parameters.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

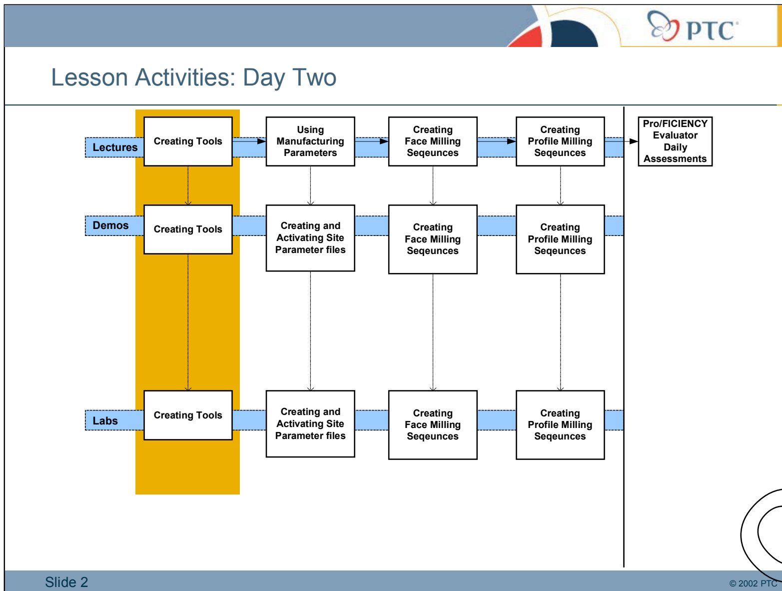
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

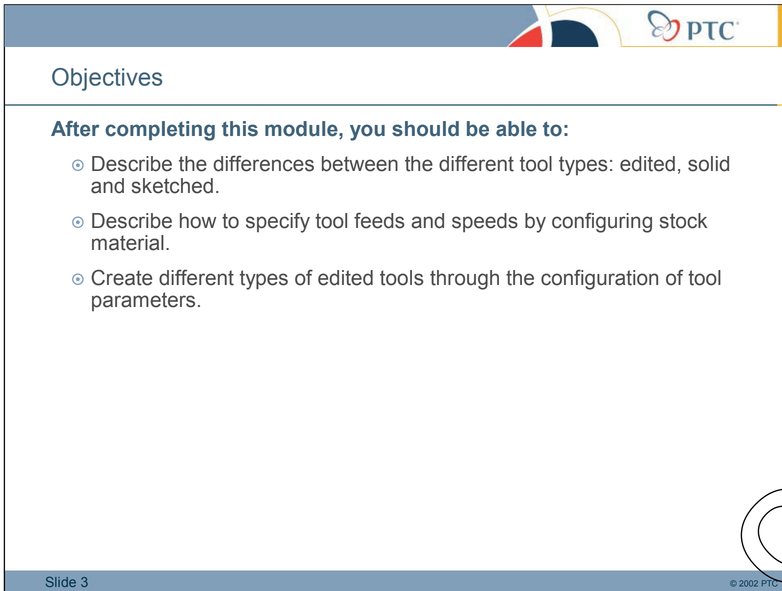
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 20 mins
- Labs : 40 mins
- Total: 1 hr 15 mins



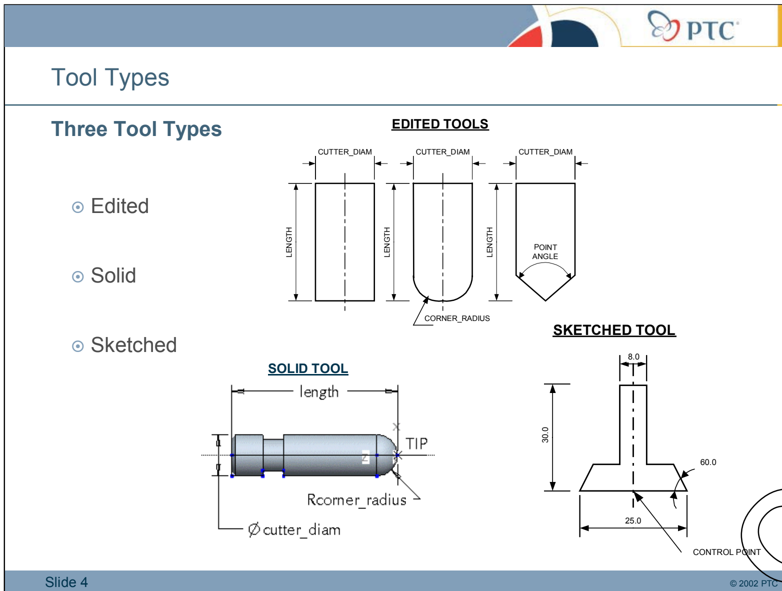
Overview

Creating tools is an essential step in the manufacturing process as a tool must be configured for each NC sequence you create. Tools can be created when the workcell is created or they can be configured as needed for each NC sequence. Once a tool is configured the information can be stored and used again.

There are three different tool types: edited, solid, and sketched. Each type of tool is created in a different way and is designed for a specific purpose. It is important to understand the differences between each type of tool and when they should be used.

After successfully completing this module, you will be able to:

- Describe the differences between the different tool types: edited, solid and sketched.
- Describe how to specify tool feeds and speeds by configuring stock material.
- Create different types of edited tools through the configuration of tool parameters.



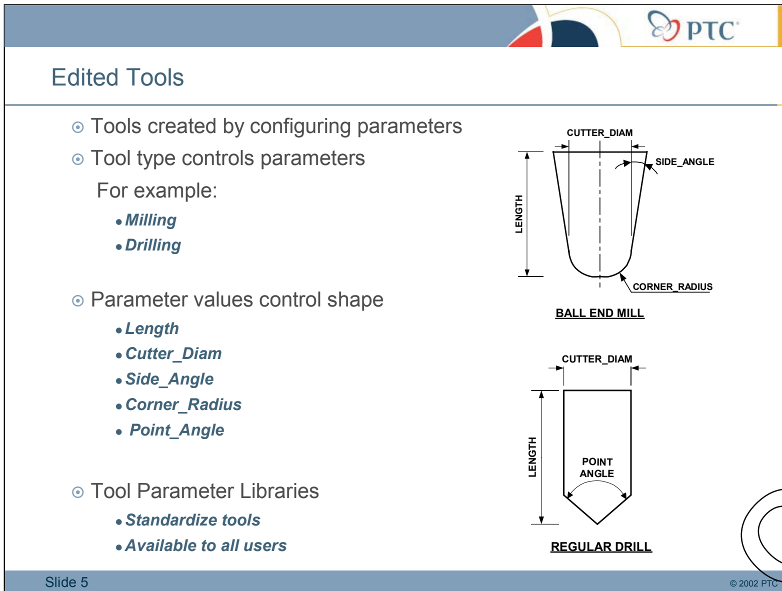
Tool Types

Tools can be created when the workcell is created or they can be configured as needed for each NC sequence. Once a tool is configured the information can be stored and used again.

There are three different tool types: edited, solid, and sketched. Each type of tool is created in a different way and is designed for a specific purpose.

- Edited tools are used when no special tool representation is required.
- Solid tools are used when you need to enhance the tool display and check for interference between the tool and workpiece when reviewing tool paths.
- Sketched tools are typically used when a non-standard tool shape or an alternative tool control point is required; for example during trajectory milling you can create a sketched tool for machining slots.

We will review each type in turn.



Edited Tools

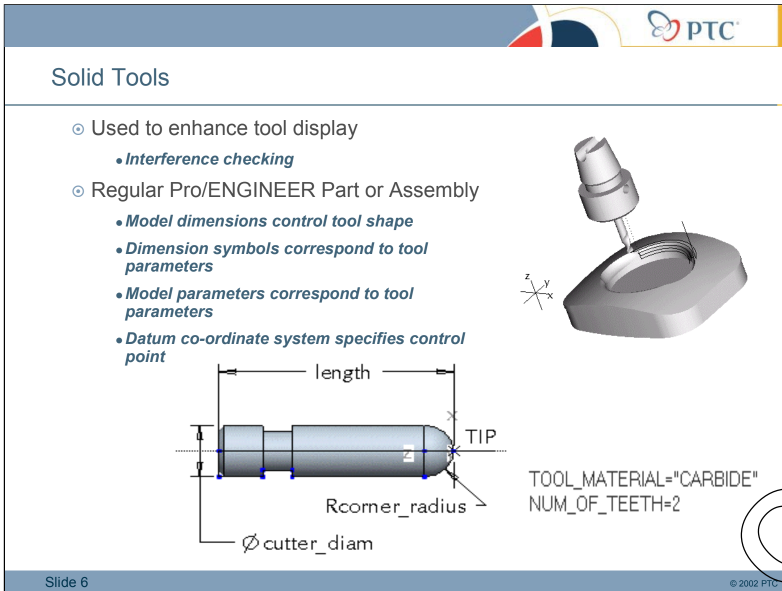
Edited tools are created by configuring tool parameters.

The available tool parameters are based on the type of tool being configured for example milling or drilling.

The tool parameter values control the tool shape, a graphic image of the tool is displayed during configuration.

You can create your own tool library, where all the tool parameter files are stored. This enables access to a standard set of tools for retrieval into manufacturing processes, for modifying, or for setting up new tools.

When playing a toolpath the tool is displayed based on parameter values.



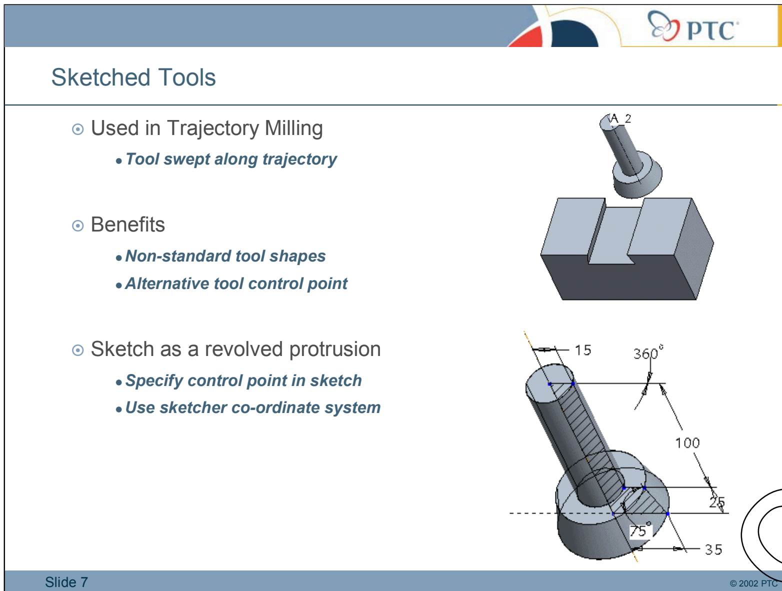
Solid Tools

You can enhance the CL data display when checking tool paths, and check for interference by showing a "real" tool.

Solid tools provide the option to display the default tool section or the real tool model.

The tool is created as a regular Pro/ENGINEER model (part or assembly), a number of items need to be configured:

- The model's dimensions are related to tool parameters, by changing dimension symbols to correspond to tool parameter names.
  - E.g.: length, corner_radius, cutter_diam.
- Model parameters are set to correspond to tool parameters, e.g. tool_material, num_of_teeth.
- The solid model must also have a datum co-ordinate system named TIP to represent the tool origin (or control point), and for milling and holemaking tools the z-axis of the co-ordinate system must point into the tool.



Sketched Tools

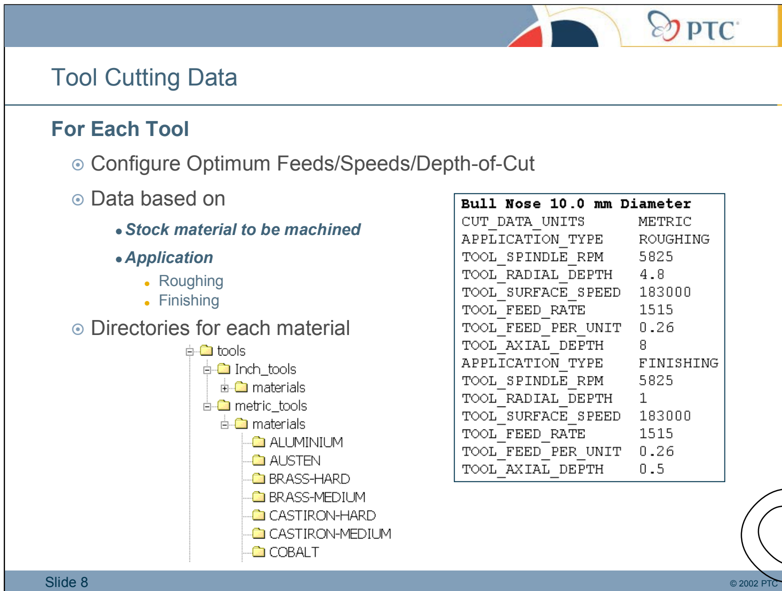
Sketched tools are used in trajectory milling where the tool is swept along a defined trajectory; for example when machining slots.

Sketched tools are typically used when a non-standard tool shape or an alternative tool control point is required.

The tool is sketched as a revolved protrusion:

- The sketch represents half of the tool cross-section. The whole sketch must lie on one side of the axis of symmetry. The axis of symmetry must be vertical, with the sketch lying on the right.
- The section must be closed.

For a sketched tool, you can specify a control point other than its tip by adding a coordinate system to the tool section sketch. The tool will then be swept so that its control point follows the specified trajectory.



Tool Cutting Data

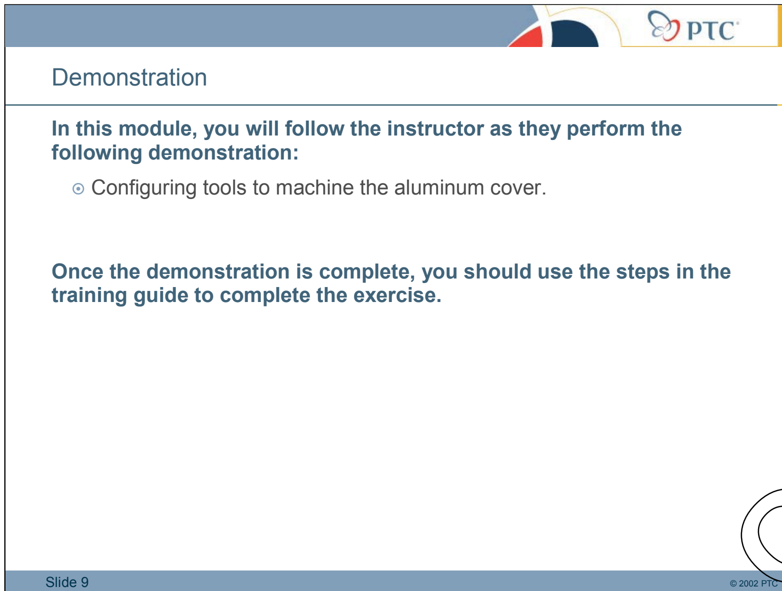
Tool cutting data can be created for tools, enabling you to configure optimum feed rates, spindle speeds, and depth-of-cut.

The data is based on the material to be machined and the application; either roughing or finishing.

Cutting data files can be created for each tool and for each workpiece material. This information is stored in a directory structure with different directories for each material.

The feed and speed data can be passed into the manufacturing parameters of an NC sequence.

- This can be done automatically through relations in the manufacturing parameters.
- Or manually by choosing to copy the data when configuring manufacturing parameters.



Demonstration:

- Configuring tools to machine the aluminum cover.

Scenario

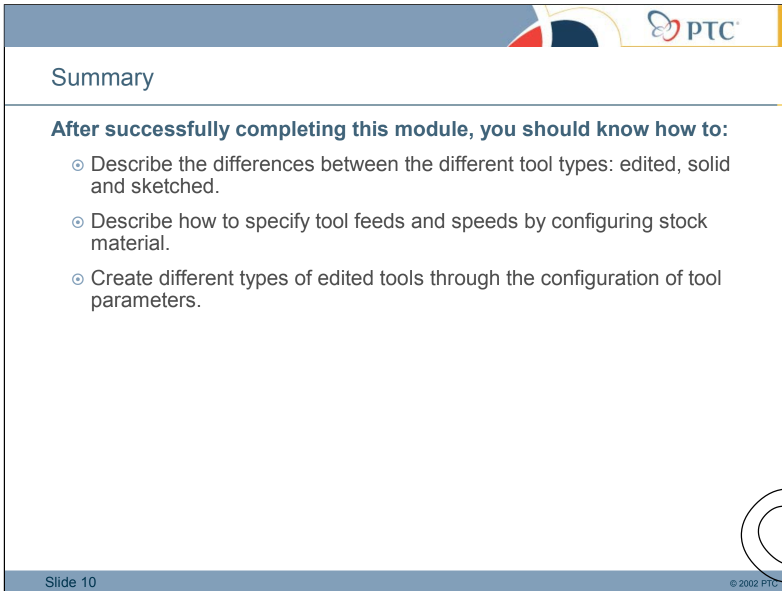
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the differences between the different tool types: edited, solid and sketched.
- Describe how to specify tool feeds and speeds by configuring stock material.
- Create different types of edited tools through the configuration of tool parameters.

Module 5 Lab Exercises

Exercise 1: Creating Edited Tools

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you configure a number of tools to enable machining of the aluminum cover. For each tool you specify the tool type, and configure tool parameters to control the tool shape. All the tools will be configured within the FADAL_VMC workcell. You also configure feed and speed data for each tool by reading-in optimum tool feeds, spindle speeds and depth of cut data.

Creating tools is an essential step in the manufacturing process, as a tool must be configured for each NC sequence you create. Tools can be created when the workcell is created or they can be configured as needed for each NC sequence. Once a tool is configured, the information can be stored and used again.

Objectives

After successfully completing this exercise, you will know how to:

- Create different types of edited tools through the configuration of tool parameters.

Scenario

The next step in the manufacturing process typically involves creating tools. To enable machining of the aluminum cover, you create a number of different types of edited tools. This is achieved through the configuration of tool parameters. You also learn how to specify optimum tool feeds and speeds by configuring stock materials.

Step 1. Open the manufacturing model for the aluminum cover.

To ensure we are using the correct version of the manufacturing model, we close all windows and erase all components from memory. We can then open the manufacturing model for the cover from the module05 directory.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module05**, right-click, and select **Make Working Directory.**
2. Open COVER.MFG.

Step 2. Configure three drilling tools for the FADAL_VMC workcell.

All the tools configured in this exercise are associated with the FADAL_VMC workcell. After the tools are saved it is possible to use them again with other workcells.

We can start by configuring a drilling tool. The tool name is a means of identifying and describing a tool as each tool is given a unique name during configuration. The tool type controls the tool geometry parameters available for configuration. It is important to specify the correct tool type to ensure proper parameters are available for each tool.

The tool geometry parameters control the shape of the tool. The tool number specifies the tool pocket number within the workcell, and the tool-offset number can be used to specify an offset value in the gauge length register. Each tool is saved to enable it to be used again as required.

1. Using the Menu Manager, select **Mfg Setup > Tooling > FADAL_VMC** to open the Tools Setup dialog box.
2. In the Tools Setup dialog box, create a center-drill tool. Using the Geometry and Settings tabs, configure the tool parameters as shown in the table below.

Table 1: Tool Parameters

Tool Parameter	Parameter Value
Name	CTRDRILL_04
Type	CENTER-DRILLING
Material	-
Units	Millimeter
Cutter Diameter	8 *(0.31 in.)*
Length	54 *(2.13 in.)*
Point Angle	118
Csink Angle	60
Drilling Diameter	4.0 *(0.16 in.)*
Drill Length	4.0 *(0.16 in.)*
Tool Number	1
Offset Number	1

3. **Apply** the changes to the tool definition and save the tool.

4. Create two more drilling tools using the following tables. Save each tool after configuring the parameters.

> **Note:**
> Remember to change the tool type to see the correct tool parameters.

Table 2: 6.0 mm drill

Tool Parameter	Parameter Value
Name	DRILL_06_0
Type	DRILLING
Material	HSS
Units	Millimeter
Cutter Diameter	6 *(0.24 in.)*
Length	70 *(2.76 in.)*
Point Angle	118
Point Diameter	-
Tip Offset	-
Tool Number	3
Offset Number	3

Table 3: Countersink tool

Tool Parameter	Parameter Value
Name	CSINK_12
Type	COUNTERSINK
Material	-
Units	Millimeter
Cutter Diameter	12 *(0.47 in.)*
Length	50 *(1.97 in.)*
Point Diameter	-
Point Angle	90
Gauge Offset	-
Tool Number	5
Offset Number	5

Step 3. Configure six milling tools for the FADAL_VMC workcell.

We can configure the milling tools we need to machine the aluminum cover now or later, when creating NC sequences. It is important to configure the correct number of teeth as this information is used later when calculating optimum tool feeds and spindle speeds.

1. Create a flat end mill tool with a diameter of three millimeters using the tool parameters as shown in the table below.

Tip:
Change the tool type to milling before starting to create the tool.

Table 4: Tool Parameters

Tool Parameter	Parameter Value
Name	FEM_03_0
Type	MILLING
Material	HSS
Units	Millimeter
Cutter Diameter	3 *(0.19 in.)*
Length	25 *(0.98 in.)*
Corner Radius	-
Side Angle	-
Number of Teeth	2
Tool Number	7
Offset Number	7

2. **Apply** the changes to the tool definition and save the tool.

3. Create five more milling tools using the following tables. After configuring the parameters apply the changes and save each tool.

Table 5: 6.0 mm flat end mill

Tool Parameter	Parameter Value
Name	FEM_06_0
Type	MILLING
Material	HSS
Units	Millimeter
Cutter Diameter	6 *(0.24 in.)*
Length	30 *(1.18 in.)*
Corner Radius	-
Side Angle	-
Number of Teeth	4
Tool Number	9
Offset Number	9

Table 6: 12.0 mm flat end mill

Tool Parameter	Parameter Value
Name	FEM_12_0
Type	MILLING
Material	CARBIDE
Units	Millimeter
Cutter Diameter	12 *(0.47 in.)*
Length	50 *(1.97 in.)*
Corner Radius	-
Side Angle	-
Number of Teeth	4
Tool Number	11
Offset Number	11

Table 7: 12.0 mm ball nosed end mill

Tool Parameter	Parameter Value
Name	BEM_12_0
Type	MILLING
Material	CARBIDE
Units	Millimeter
Cutter Diameter	12 *(0.47 in.)*
Length	50 *(1.97 in.)*
Corner Radius	6 *(0.24 in.)*
Side Angle	-
Number of Teeth	2
Tool Number	13
Offset Number	13

Table 8: 25.0 mm flat end mill

Tool Parameter	Parameter Value
Name	FEM_25_0
Type	MILLING
Material	CARBIDE
Units	Millimeter
Cutter Diameter	25 *(0.98 in.)*
Length	100 *(3.94 in.)*
Corner Radius	-
Side Angle	-
Number of Teeth	4
Tool Number	15
Offset Number	15

Table 9: 50.0 mm shell end mill

Tool Parameter	Parameter Value
Name	SH_MILL_50_0
Type	MILLING
Material	HSS
Units	Millimeter
Cutter Diameter	50 *(1.97 in.)*
Length	35 *(1.38 in.)*
Corner Radius	-
Side Angle	-
Number of Teeth	10
Tool Number	17
Offset Number	17

Step 4. Configure the optimum speed and feed data for the milling tools based on the stock material.

For the milling tools, we can configure optimum tool feed, spindle speed and depth of cut data based on the workpiece material. To save time, the data has already been created and stored in a material folder. We can make use of this data by specifying the stock material and reading-in the feed and speed information for each tool. When creating NC sequences, this data can then be passed into the manufacturing parameters of an NC sequence.

1. Select the FEM_03_0 tool, then click the Speeds-Feeds tab, and specify the stock material as **aluminum** from the Stock Material drop down list. Click the Read DB button to read-in the optimum speed and feed data.

> **Note:**
> Clicking the Read DB button reads in the cutting data for both Roughing and Finishing applications.

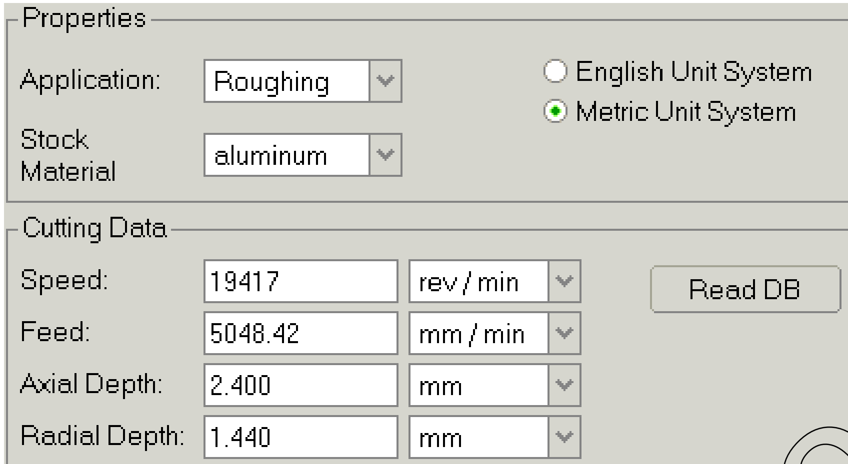


Figure 1: Speed and feed data for tool FEM_03_0.

2. Then **Apply** the changes to the tool definition, and confirm the tool description will be changed.
3. Repeat this process by specifying aluminum as the stock material and reading-in the optimum speed and feed data for the following milling tools.

Table 10: Tool names

Tool Name
FEM_06_0
FEM_12_0
BEM_12_0
FEM_25_0
SH_MILL_50_0

4. Close the Tools Setup dialog box.
5. Save the manufacturing model.
6. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the differences between the different tool types; edited, solid and sketched.
- Describe how to specify tool feeds and speeds by configuring stock material.
- Create different types of edited tools through the configuration of tool parameters.

Module

6

Using Manufacturing Parameters

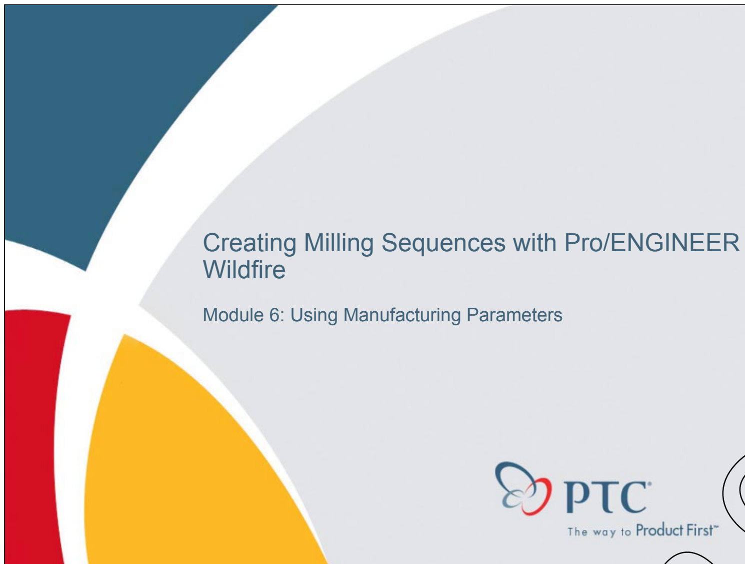
Introduction

Manufacturing parameters are the primary means for controlling how an NC sequence is generated. You can specify and edit parameter values as needed. Manufacturing parameters can be configured and stored in global site files, or in NC specific parameter files. These files can be retrieved, and enable you to quickly and easily set suitable parameter values when creating NC sequences.

Objectives

After completing this module, you will be able to:

- Describe how manufacturing parameters control NC sequences.
- Describe the different methods used to set and edit parameter values.
- Edit and activate site parameter files.



Instructor Preparation

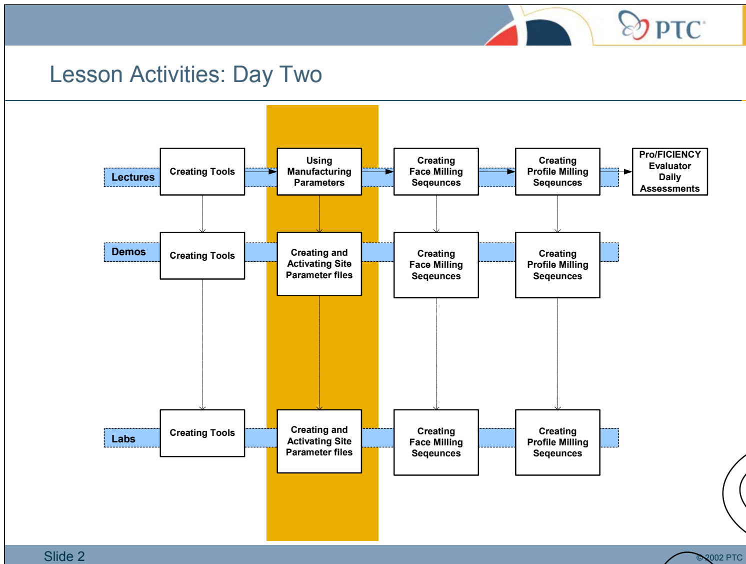
Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in /GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

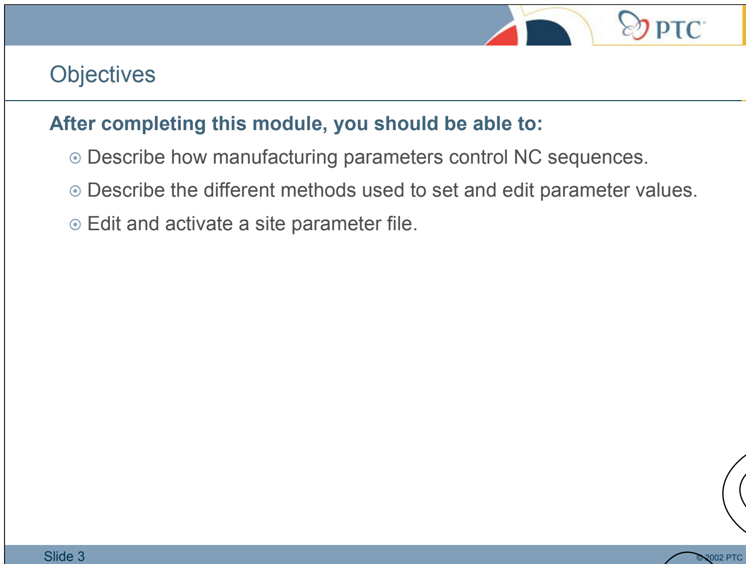
- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in /GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 15 mins
- Labs : 30 mins
- Total: 1 hr 00 mins

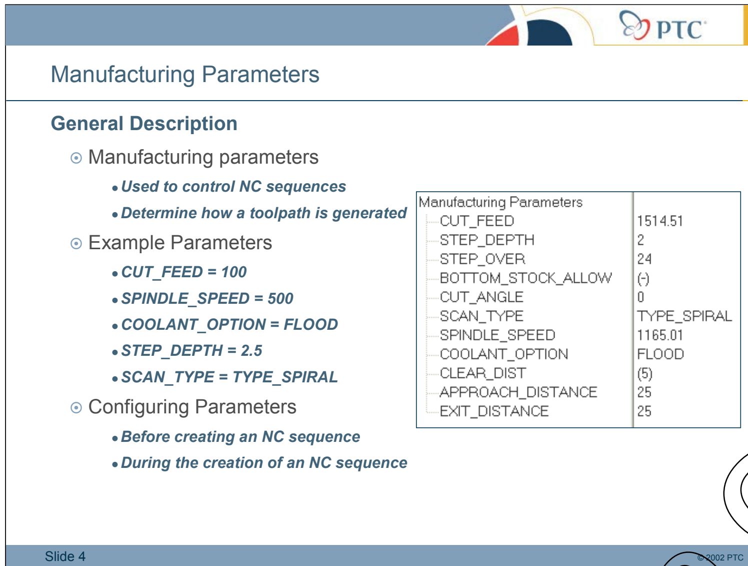


Overview

- Manufacturing parameters are the primary means for controlling how NC sequences are generated. It is important that you can specify the correct parameters and edit parameter values as needed. Manufacturing parameters can be configured and stored in global site files, or in NC specific parameter files. These files can be retrieved, and enable you to quickly and easily set suitable parameter values when creating NC sequences.

After successfully completing this module, you will be able to:

- Describe how manufacturing parameters control NC sequences.
- Describe the different methods used to set and edit parameter values.
- Edit and activate a site parameter file. This is a means of setting default parameter values for future NC sequences.



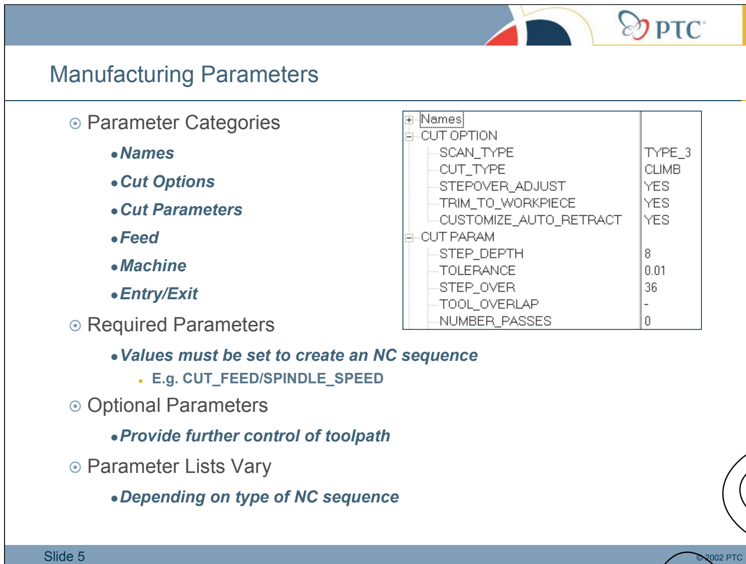
Manufacturing Parameters

Creating NC sequences involves, selecting or creating geometry to machine, then determining how the toolpath is generated by modifying manufacturing parameters.

Manufacturing parameters enable toolpath control when creating NC Sequences.

- **Example Parameters:**
  - CUT_FEED –controls the feed rate of NC Sequences during cutting motions.
  - SPINDLE_SPEED –controls spindle speed in NC Sequences.
  - COOLANT_OPTION –controls if coolant is used and the specific coolant option to be output (ON, OFF, MIST, FLOOD or THRU)
  - STEP_DEPTH – controls the incremental depth of each pass when cutting.
  - SCAN_TYPE – controls the method of scanning the machined area.

- **Configuring Parameters:**
  - You can set up default parameters before creating NC sequences.
    - You can create site parameter files to control the default values for all parameters.
    - You can also save parameters from NC Sequences to create a library and retrieve them when required.
  - When creating NC Sequences you have several options for reusing previously saved parameters or you can edit the parameters manually.



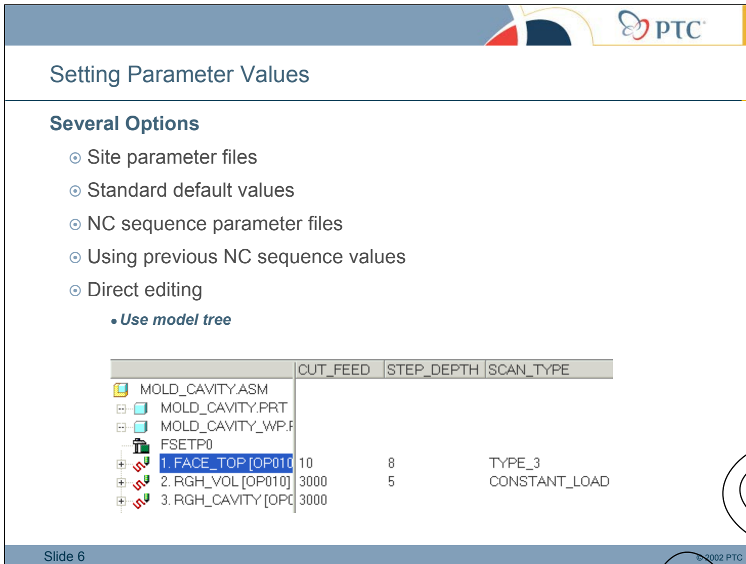
Manufacturing Parameters

- Parameter Categories*:*
  - Parameters are grouped into six logical categories, allowing you to quickly locate the relevant parameters to set.

- Required Parameters:
  - These are parameters that you must set in order to create an NC sequence.
    - Examples are STEP_DEPTH, CUT_FEED and SPINDLE_SPEED.

- Optional Parameters:
- Some parameters are optional and can be set if required and provide additional control of the toolpath.
- They are either not used or they can take their values from other parameters.
  - One example is TOOL_OVERLAP which can be used instead of STEP_OVER.

- Parameters vary depending on the type of NC sequence being created.
  - Some parameters such as feed rate and spindle speed will be present in all of the sequence types.
  - Some parameters are specific to certain types of sequences. When creating NC sequences, only the relevant parameters are available.

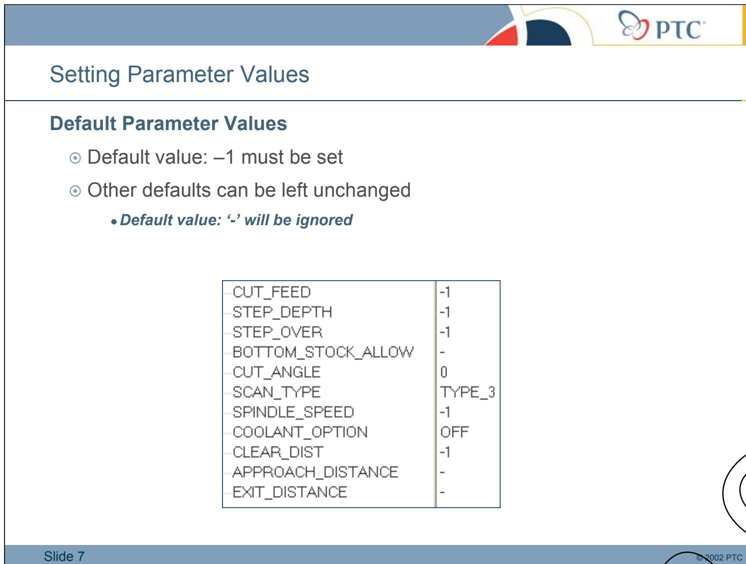


Setting Parameter Values

Several Options

There are several ways to set the parameters for NC Sequences:

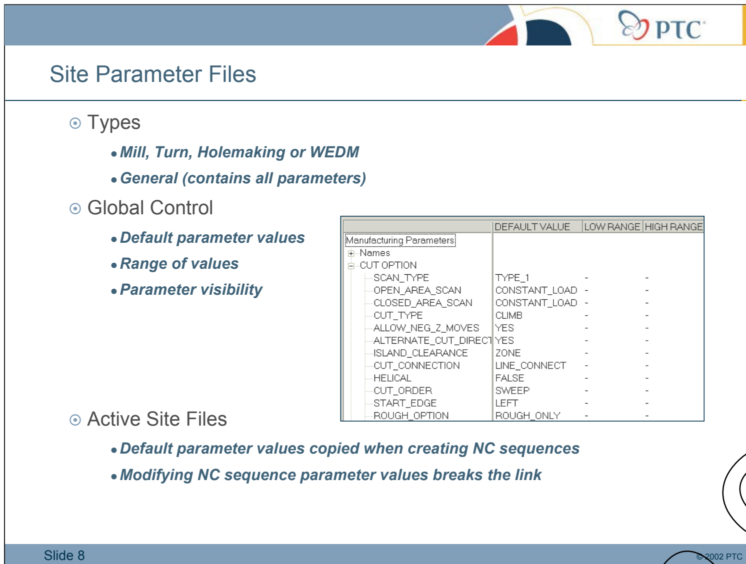
- Site parameter files allow you to set the default values for all NC sequence parameters. When you create an NC Sequence, these default values are read in as the initial parameter values.
- If you do not use a site file, standard default values for all parameters are automatically set.
- When creating NC sequences, you also have the option to read in parameter values from a file, or you can copy the parameter values from a previous NC sequence in your model.
- You can also edit parameter values in NC sequences manually.
  - You can also configure the model tree to include NC sequence parameters and set the parameters directly in the model tree.



Setting Parameter Values

Default Parameter Values

- When creating NC sequences, parameters will have either the default values or if a site file is used, they will have the site file values.
  - If a parameter has an initial value of –1, it is a required parameter. A value must be entered before continuing.
  - If a parameter has an initial value of '-', it is an optional parameter. No value is required.



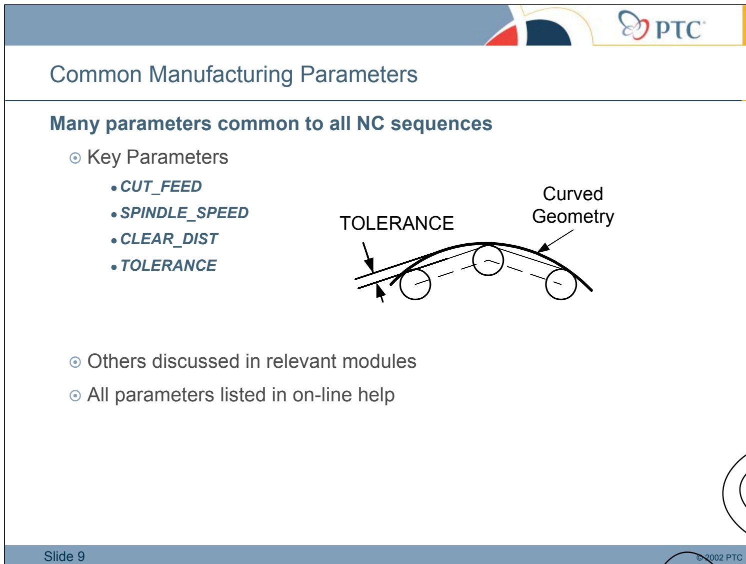
Site Parameter Files

- Types: several types of site parameter files can be configured: Mill, Turn, Holemaking, or Wire EDM. Each site type contains the parameters relevant to its NC sequence types. A general site file contains all of the parameters in Pro/NC.

- Global Control: site files control the default values used in NC sequences. They also control the range of values and the visibility of the parameters. NC sequence parameters get their values from site values, so if a change is made to a site file, the change will effect all of the sequences that reference it within the Pro/NC model.

- Active Site Files: a site file must be activated for it to be used automatically when creating new NC sequences. A site file can be activated manually (or a different site file selected) when defining NC sequence parameters.
  - Editing parameter values breaks the link to the site file for that parameter. Changes made to that parameter in the site file will no longer be passed to the NC sequence.
  - Site files can be de-activated in NC sequences. All of the parameter values will be retained, but parameters will no longer inherit their values from the site file.

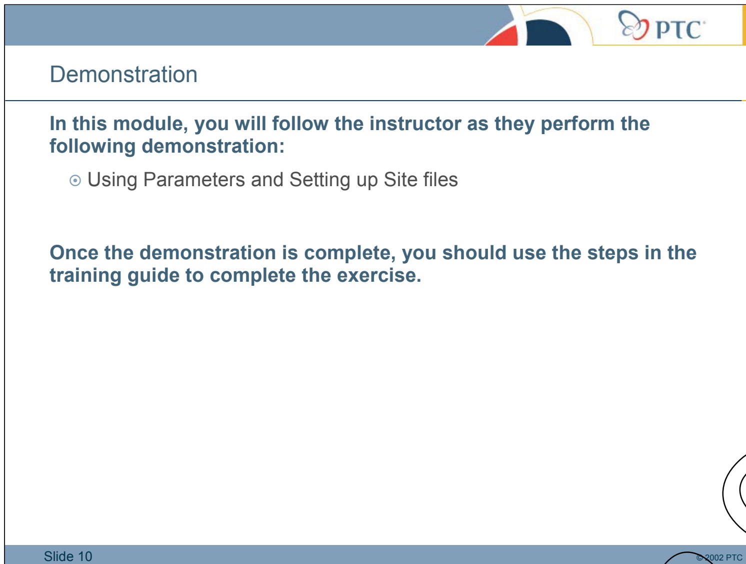
- Workcells & Site files
  - One of each type of site file can be assigned to a workcell, or a single general site file can be assigned. Pro/NC will automatically use the site file of the correct type when creating new NC sequences.



Common Manufacturing Parameters

There are a number of manufacturing parameters that are common to all NC sequences. Some of the key common parameters are:

- CUT_FEED: the feed rate used for cutting motion.
- SPINDLE_SPEED: specifies rotation rate for spindle.
- CLEAR_DIST: specifies the distance above the surface to be machined at which rapid feed ends and cut feed begins.
- TOLERANCE: tool paths approximate curved geometry by moving in small straight line increments. The maximum distance that the straight line path deviates from the curved geometry is set by TOLERANCE.
- Other key parameters will be discussed in the relevant modules.
- All parameters are listed in the on-line help, together with explanations for all parameter values.



Demonstration: Using Parameters and Setting up Site Files

Scenario

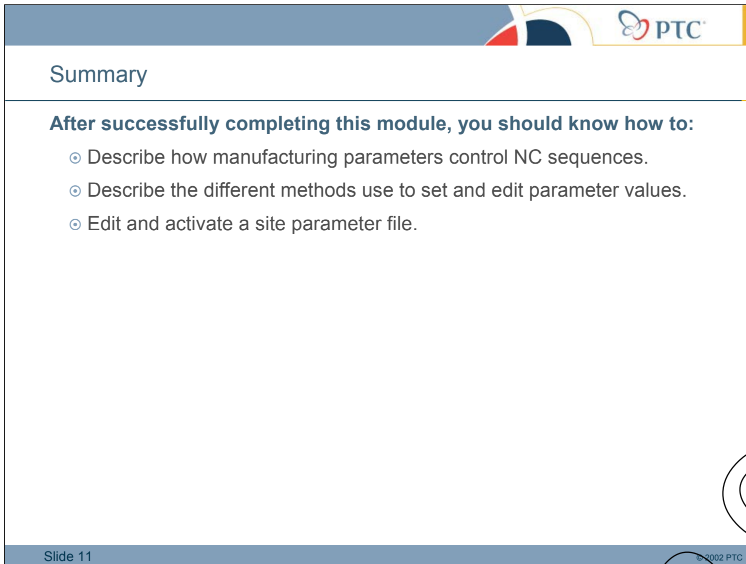
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe how manufacturing parameters control NC sequences.
- Describe the different methods use to set and edit parameter values.
- Edit and activate a site parameter file.

Module 6 Lab Exercises

Exercise 1: Creating and Activating Site Parameter Files

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create a site file. Site files provide default values for the NC sequences. We will discuss the various values you can enter for parameters and what some of the parameters control. After creating the site file, we will activate it within the workcell.

Objectives

After successfully completing this exercise, you will know how to:

- Edit and activate a site parameter file.

Scenario

In order to speed up the creation of NC sequences and to ensure that company standard parameter values are used, you will edit and activate a milling site parameter file that will set default parameter values for milling workcells.

Parameters

The following are site parameters modified in this exercise.

Table 1: Parameter Descriptions

Parameter	Description
LACE_OPTION	For surface milling, specifies the shape of connection between the endpoints of neighboring cuts
STEP_DEPTH	The incremental depth of each pass when cutting.

Parameter	Description
STEP_OVER	Distance between passes, within a slice
ROUGH_STOCK_ALLOW	Remaining stock after roughing passes.
CUT_FEED	Feed rate when cutting material.
RETRACT_FEED	Feed rate when the tool moves away from the workpiece.
FREE_FEED	Feed rate used for rapid traverse, default is RAPID.
SPINDLE_SPEED	Specifies rotation rate for spindle
COOLANT	Determines if coolant is used.
CUTCOM	Controls if cutter compensation is on or off.
CLEAR_DIST	Distance above surface to be machined at which rapid feed ends and cut feed begins.

Step 1. Locate and open the cover manufacturing model.

Before creating the parameter site file, we first need to locate the manufacturing model and open it using the folder navigator. Once we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module06**, right-click, and select **Make Working Directory.**

2. Open COVER.MFG.

Step 2. Create a site file.

We can now create a milling site parameter file. Site parameter files are useful because they allow us to set defaults for all NC sequence parameters. We can also enter relations in site files to automatically transfer parameter values from our tool feed and speed data.

Notice the headings at the top of the mill site window. The various columns allow us to set the default value, the upper and lower range, the visibility in the NC Sequence Set Up, the visibility in the Information output, and the inheritance.

Notice the tree structure on the left side, we can expand and collapse the categories as required.

Notice the various entries in the Default values column. The –1 entries indicate that the parameter is required and therefore must be specified when creating NC Sequences. The dash values indicate an optional item, which can be configured if required.

We can now set some new default values

Changing the default LACE_OPTION to LINE_CONNECT specifies that for surface milling sequences, cut motions will be connected instead of retracting and repositioning between each pass.

The *STEP_OVER* parameter controls the tool step over between passes. Setting it to *TOOL_ROUGH_RADIAL_DEPTH* will read in the value stored with the tool. The ROUGH in the parameter name indicates that it will read in the tool information for roughing the current stock material. We could change the ROUGH to FINISH and read in the corresponding finish parameter.

The *STEP_DEPTH* parameter controls the depth in Z of each pass. Setting it to *TOOL_ROUGH_AXIAL_DEPTH* will read in the value stored with the tool.

Setting the *ROUGH_STOCK* value to 0.75 will give us a standard amount of stock remaining for roughing sequences.

Setting the *CUT_FEED* to *TOOL_ROUGH_FEED_RATE* will automatically read in the feed data value stored with the tool.

Setting the RETRACT and FREE_FEED rates to zero will output RAPID feed rates whenever they are used.

Setting the SPINDLE_SPEED to TOOL_ROUGH_SPINDLE_RPM will automatically read in the spindle speed stored with the tool

We can set the default values for COOLANT and cutter compensation to on. The CUTCOM will only be applied when machining with the side of the tool.

Lastly, we set the default value for CLEAR_DIST, this controls the distance above the machined surface, at which the RAPID motion ends, and the PLUNGE_FEED begins.

1. Using the Menu Manager, select **Mfg setup > Param Setup > Site > Create**, name the site file [**RGH_3_AXIS**], and select the **Mill** option.
2. Review the contents of the site parameter file window.
3. In the CUT OPTION category, enter the following values in the DEFAULT VALUE column:

Table 2: Cut Option Default Values

Manufacturing Parameter	Default Value
LACE_OPTION	LINE_CONNECT

Note:
LINE_CONNECT specifies that for surface milling sequences, cut motions will be connected instead of retracting and repositioning between each pass.

4. In the CUT PARAM category enter the following values in the DEFAULT VALUE column:

Table 3: Cut Parameter Default Values

Manufacturing Parameter	Default Value
STEP_DEPTH	TOOL_ROUGH_AXIAL_DEPTH
STEP_OVER	TOOL_ROUGH_RADIAL_DEPTH
ROUGH_STOCK_ALLOW	0.75 *(0.03 in.)*

Note:
TOOL_ROUGH_AXIAL_DEPTH and TOOL_ROUGH_RADIAL_DEPTH will use calculated values based on workpiece material and tool diameter.

5. In the FEED category enter the following values in the DEFAULT VALUE column:

Table 4:Feed Option Default Values

Manufacturing Parameter	Default Value
CUT_FEED	TOOL_ROUGH_FEED_RATE
RETRACT_FEED	0
FREE_FEED	0

Note:
TOOL_ROUGH_FEED_RATE will use a calculated value based on workpiece material and tool diameter. Entering zero for a feed rate will output a RAPID command for that feed rate.

6. In the MACHINE category enter the following values in the DEFAULT VALUE column:

Table 5: Machine Option Default Values

Manufacturing Parameter	Default Value
SPINDLE_SPEED	TOOL_ROUGH_SPINDLE_RPM
COOLANT_OPTION	ON
CUTCOM	ON

Tips & Techniques
TOOL_ROUGH_SPINDLE_RPM will use a calculated value based on workpiece material and tool diameter.

7. In the ENTRY/EXIT category enter the following values in the DEFAULT VALUE column:

Table 6: Entry and Exit Default Values

Manufacturing Parameter	Default Value
CLEAR_DIST	5 *(0.20 in.)*

8. In the site tree window, click **File > Exit**, to close the window.

> **Note:**
> Do not select Quit, you will lose all your work, you must select Exit.

9. **Save** the site parameter file, select **RGH_3_AXIS_MIL** from the list.

> **Note:**
> Notice the site file name is appended with _MIL, to show it is a mill site file.

10. Close all of the open menus.

Step 3. Activate the site file within the workcell.

Finally, we can activate the site file within the workcell so that it will be used when we create new NC sequences. Site files are set using the defaults button in the workcell dialog box. You must activate the site file by selecting the check box next to its name. You can only associate a single site file of each type to your workcell. This means that you can have separate site files for milling and holemaking active in your workcell. If you use a general site file, you can only associate that one site file to the workcell.

When you are creating an NC Sequence, you have the option to deactivate this site file, or to pick another site file.

If we save this Workcell after we have activated our site file, the site file will be stored with the workcell.

1. Using the Menu Manager, select **Mfg Setup > Workcell** to open the Workcell dialog box. Click the Defaults button, and select the **RGH_3_AXIS_MIL** check box. This activates the site file.

2. Complete the process and close the Workcell dialog box.

> Note:
> You must activate the site file by selecting the check box next to its name. You can only associate a single site file of each type to a workcell.

3. Save the manufacturing model.

4. Close all windows and erase all models from memory.

Summary

After successfully completing this module, you should know how to:

- Describe how manufacturing parameters control NC sequences.
- Describe the different methods used to set and edit parameter values.
- Edit and activate site parameter files.

Module

7

Creating Face Milling Sequences

Introduction

When milling components, one of the first tasks is often to face down the workpiece. This enables you to define the top surface of the job and a reference surface for other NC sequences. When creating face milling sequences it is important that you use the correct manufacturing parameters to ensure the toolpath is generated efficiently. You can also create mill surfaces and use them as alternate references when model surfaces are inappropriate, as when multiple model surfaces need to be machined as one surface. When a face milling sequence is completed, you can update the in-process workpiece geometry. This is achieved by creating a material removal feature that removes the machined volume from the workpiece.

Objectives

After completing this module, you will be able to:

- Describe the face milling process.
- Describe the key face milling manufacturing parameters.
- Create face milling sequences.
- Create mill surfaces relevant to face milling.
- Create material removal features specific to face milling.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

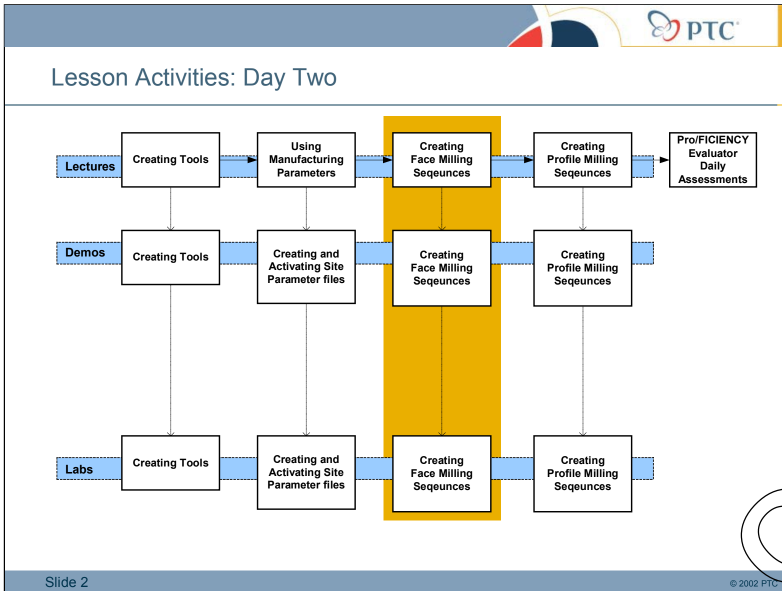
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

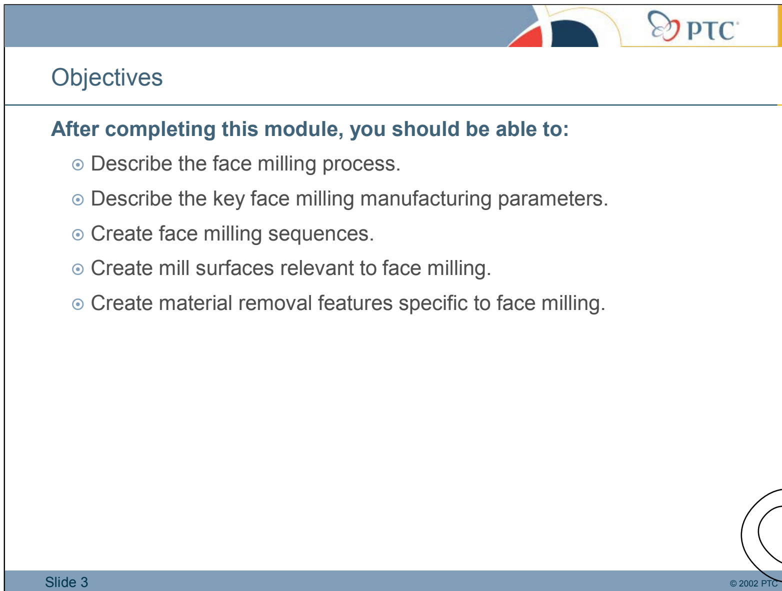
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos (1): 25 mins
- Labs (1): 50 mins
- Total: 1 hr 30 mins

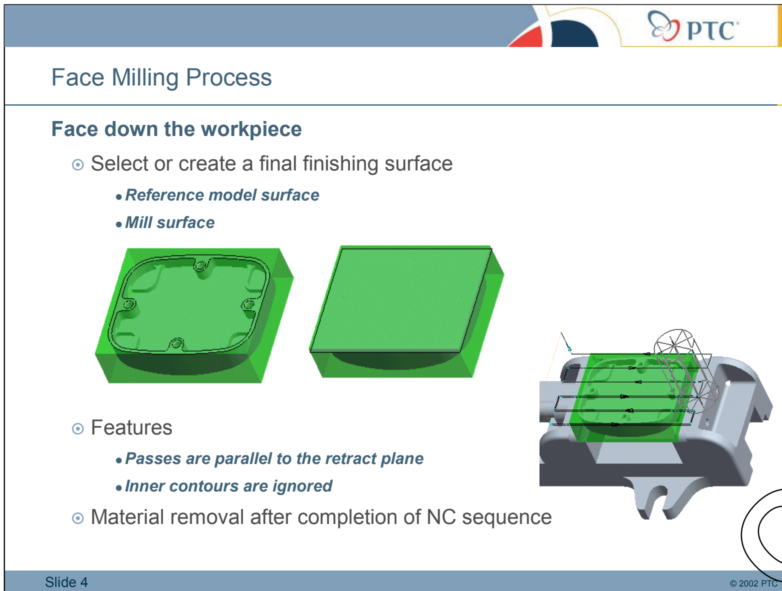


Overview

- When milling components one of the first tasks is often to face down the workpiece. This enables you to define the top surface of the job and a reference surface for other NC sequences. When creating face milling sequences it is important that you use the correct manufacturing parameters to ensure the toolpath is generated efficiently. You can also create mill surfaces and use them as alternate references when model surfaces are inappropriate, as when multiple model surfaces need to be machined as one surface. Once a face milling sequence is completed you can update the in-process workpiece geometry. This is achieved by creating a material removal feature that removes the machined volume from the workpiece.

After successfully completing this module, you will be able to:

- Describe the face milling process.
- Describe the key face milling manufacturing parameters.
- Create face milling sequences.
- Create mill surfaces relevant to face milling.
- Create material removal features specific to face milling.

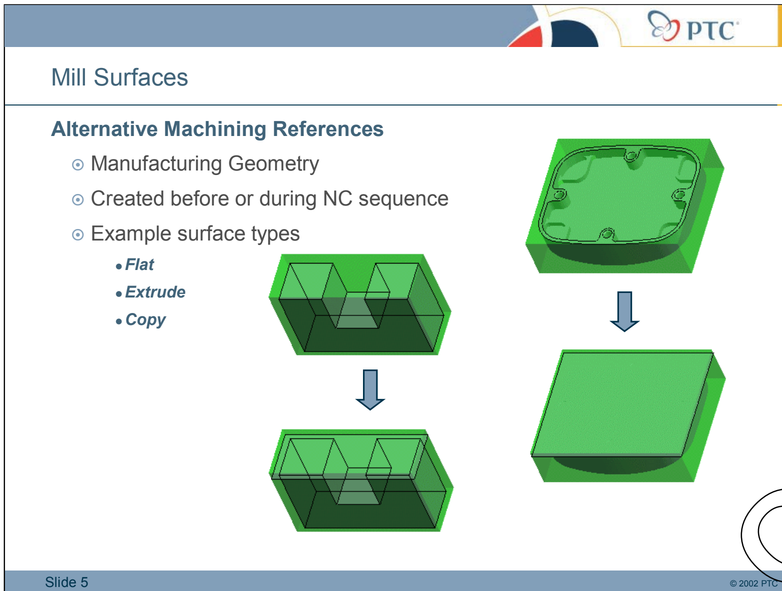


Face Milling Process

A face milling sequence allows you to face down the workpiece with a flat or radius end mill. The final finishing surface is configured by selecting or creating a planar surface parallel to the retract plane.

- All machining movements are parallel to the retract plane.
- All inner contours in the selected surfaces (holes, slots) will be automatically excluded.
- The toolpath will by default completely machine the selected surface.

Once the toolpath is completed it is possible to remove the machined volume from the workpiece by creating a material removal feature.



Mill Surfaces

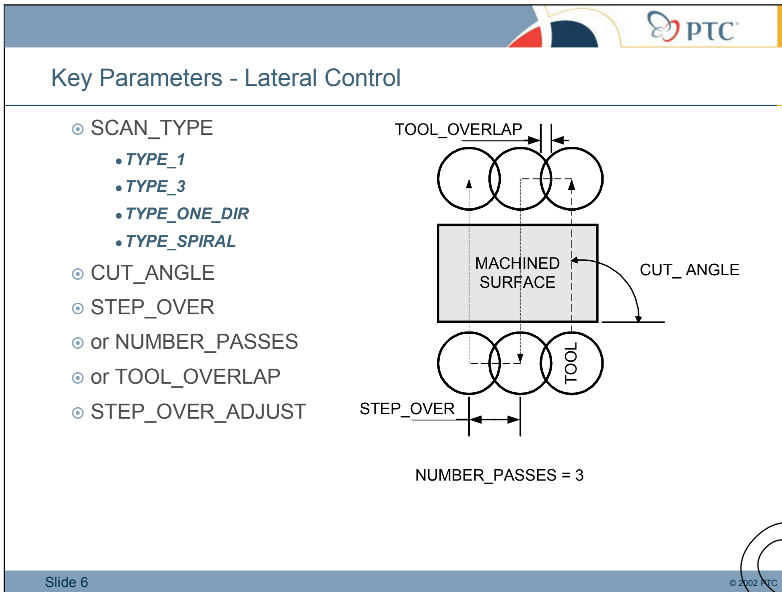
Mill surfaces can be used as alternative machining references, when model surfaces are not appropriate, for example when face milling multiple model surfaces.

•Mill surfaces are known as manufacturing geometry.

•They can be created before an NC sequence, or during the creation of an NC sequence.

•The most common surface types used are:

- Flat: create a flat surface by sketching an outline on a sketching plane.
- Extrude: create an extruded surface by sketching an outline on a sketching plane.
- Copy: copy existing model surfaces to form a new mill surface.



Key Parameters - Lateral Control

SCAN_TYPE:

- TYPE_1 moves back and forth across the surface.
- TYPE_3 if there are separate zones, machines each zone separately. Otherwise the same as TYPE_1.
- TYPE_ONE_DIR machines in one direction and retracts between passes.
- TYPE_SPIRAL will create a spiral toolpath.

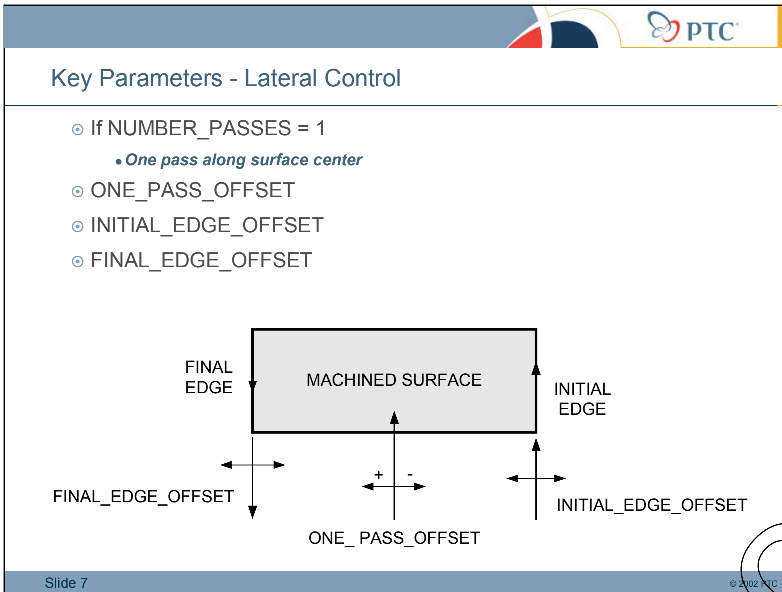
CUT_ANGLE: The angle between the cut direction and the X-axis of the NC Sequence coordinate system.

These three parameters control the step-over distance, the final toolpath will use whichever parameter produces the smallest calculated step-over.

- STEP_OVER controls the step over within a slice.
- Or NUMBER_PASSES can be used to explicitly set the number of passes to take in each slice.
- Or TOOL_OVERLAP can be used as an alternative method to control the step-over based on the tool overlap.

If NUMBER_PASSES is set to 1 however, step-over is ignored and one pass is created at the center of the machined surface for each slice. (see next slide)

The STEP_OVER_ADJUST parameter adjusts the passes in the slice to start and finish near the edges of the surface you are machining. It only reduces the step over distance, and adds an extra pass if needed.



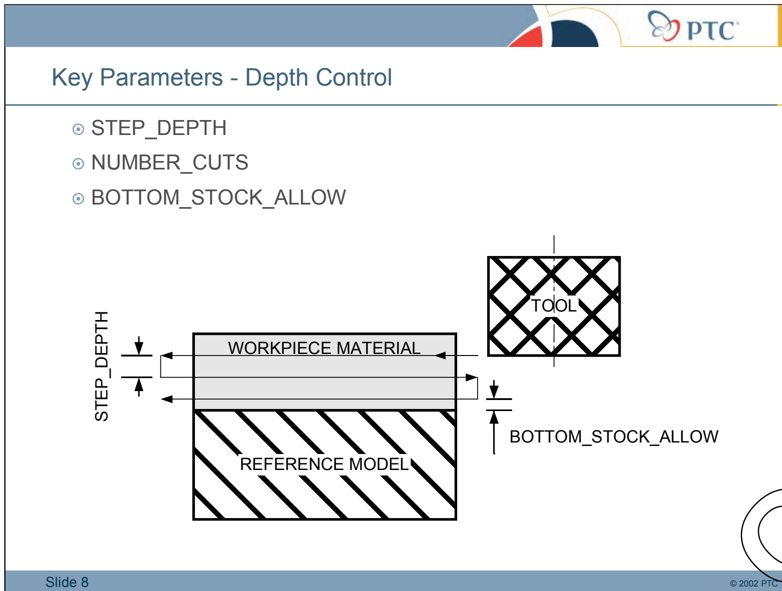
Key Parameters - Lateral Control

If NUMBER_PASSES is set to 1, step-over is ignored and one pass is created at the center of the machined surface for each slice.

The ONE_PASS_OFFSET controls the distance away from the centerline the single pass is made.

INITIAL_EDGE_OFFSET allows you to offset the first pass with respect to the edge of the surface being milled.

FINAL_EDGE_OFFSET allows you to offset the last pass with respect to the edge of the surface being milled.



Key Parameters - Depth Control

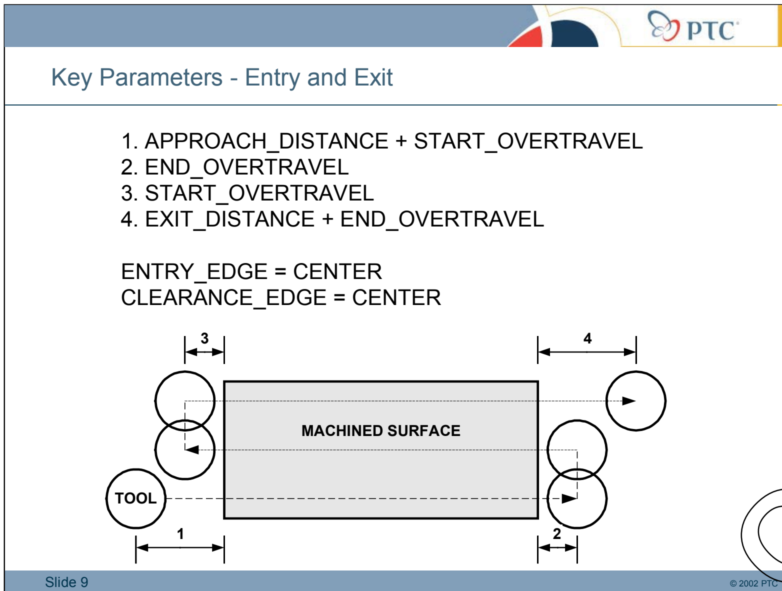
When using a workpiece, the starting height for the cuts will be determined by the top of the workpiece.

The STEP_DEPTH parameter can be used to specify the depth between each slice or the NUMBER_CUTS parameter can be used to determine the number of slices.

The parameter that creates the most slices will be used.

If the NUMBER_CUTS = 1 and the STEP_DEPTH is greater than the depth to be machined, then one pass is taken at full depth.

The BOTTOM_STOCK_ALLOW parameter determines how much stock is left on the machined surface. The default dash (-) value will leave zero stock.



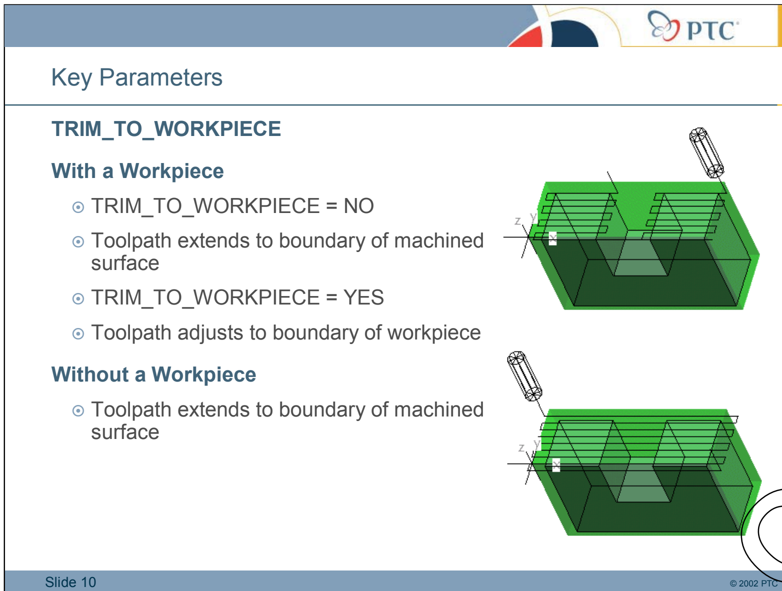
Key Parameters

The START_OVERTRAVEL adds an offset to the beginning of each pass in a slice. The END_OVERTRAVEL adds an offset to the end of each pass in a slice.

The APPROACH_DISTANCE parameter will add an extra approach distance to the first pass of each slice. The EXIT_DISTANCE parameters will add an extra distance to the last pass of each slice.

The ENTRY_EDGE parameter can be set to LEADING_EDGE (the default), CENTER or HEEL. This parameter controls which point of the tool is used for measuring the approach and over travel motions when the tool approach the material for each pass in a slice.

The CLEARANCE_EDGE parameter can be set to HEEL (the default), CENTER or LEADING_EDGE. This parameter controls which point of the tool is used for measuring the exit and over travel motions when the tool leaves the material for each pass in a slice.



Key Parameters

TRIM_TO_WORKPIECE

Face milling sequences behave differently depending on if a workpiece is in the model.

With a workpiece, the starting height for the cuts will always be one cut depth below the top of the workpiece.

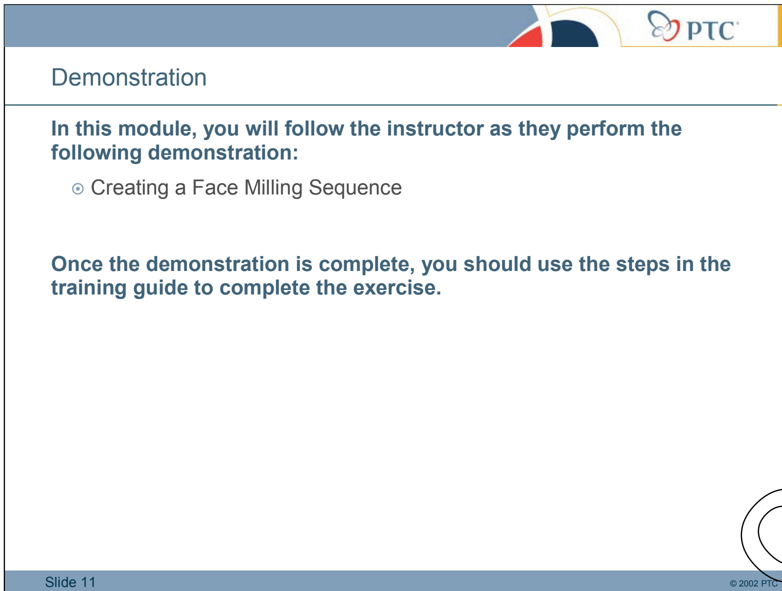
If TRIM_TO_WORKPIECE is set to NO, (top figure):

- The toolpath machines the selected surface without regard to the workpiece outline.

If TRIM_TO_WORKPIECE is set to YES, (bottom figure):

- The toolpath extends or trims to the workpiece cross-section at the depth of the surface you are machining.

Without a workpiece, the starting height for the cuts will be determined exclusively by the sequence parameters and the toolpath completely machines the selected surface.



Demonstration: Creating Face Milling Sequences

Scenario

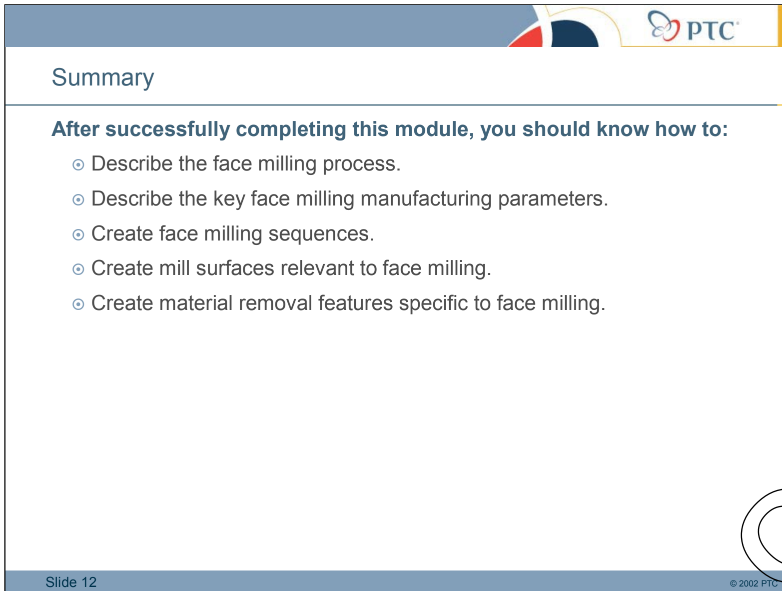
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the face milling process.
- Describe the key face milling manufacturing parameters.
- Create face milling sequences.
- Create mill surfaces relevant to face milling.
- Create material removal features specific to face milling.

Module 7 Lab Exercises

Exercise 1: Creating Face Milling Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create a mill surface for the facing sequence to create a modified toolpath that reflects the workpiece shape. You then create a facing sequence by using the reference model surface to specify the surface to machine. You then reference the mill surface for the facing sequence to create a modified toolpath that reflects the workpiece shape. Finally, you adjust the parameters to create the finished toolpath.

Objectives

After successfully completing this exercise, you will know how to:

- Create face milling sequences.
- Create mill surfaces relevant to face milling.
- Create material removal features specific to face milling.

Scenario

You are now ready to begin machining the aluminum cover; you start by creating a face milling sequence on one side of the component. During the creation of this sequence you adjust sequence parameters and references to create a more efficient toolpath.

Parameters

The following are key face milling parameters.

Table 1: Parameter Descriptions

Parameter	Description
SCAN_TYPE	The method of scanning the machined area.
CUT_ANGLE	The angle of cuts relative to NC sequence coordinate system.
STEP_OVER	Distance between passes.
NUMBER_PASSES	Number of passes per slice.
TOOL_OVERLAP	Tool step over calculated by overlap distance.
STEP_OVER_ADJUST	Adjust STEP_OVER to provide complete coverage of machined area.
ONE_PASS_OFFSET	For one pass milling offset pass to left or right.
INITIAL_EDGE_OFFSET	Offset the center of the tool from the initial edge.
FINAL_EDGE_OFFSET	Offset the center of the tool from the final edge.
STEP_DEPTH	The incremental depth of each cut.
NUMBER_CUTS	Calculate the step depth using number of cuts.
BOTTOM_STOCK_ALLOW	Amount of stock left on final machined surface.
APPROACH_DISTANCE	Extra approach distance for the first pass of each slice.
EXIT_DISTANCE	Extra exit distance for the last pass of each slice.
START_OVERTRAVEL	Offset distance at the beginning of each pass.
END_OVERTRAVEL	Offset distance at the end of each pass.
TRIM_TO_WORKPIECE	Specifies if the tool is restricted to machining inside the workpiece.

Step 1. Locate and open the cover manufacturing model.

Before creating the face milling sequence, we need to locate the manufacturing model and open it using the folder navigator. Once we have located the folder for this exercise we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module07**, right-click, and select **Make Working Directory.**

2. Open COVER.MFG, and turn off any displayed datum features.

Step 2. Create a mill surface as an alternative reference for the facing sequence.

We can create a mill surface to control the extent of the face milling sequence. We can do this if we are not using a workpiece or if we need to have more control over the toolpath.

Mill surfaces are useful because they can be used as alternative references when model references are not appropriate.

We need to create a flat rectangular surface that extends to the edges of the workpiece and is positioned on the top surface of the reference model.

To do this, we create a mill surface using the ***Flat*** *option. We select the top of the reference model as our sketching plane. In the sketcher environment, we click the Use Edge option and Loop check box and select the top surface of the workpiece to loop.*

When the surface is completed we can select the surface in the model tree, right-click and select ***Hide*** *from the drop down list to blank the surface.*

1. Using the Menu Manager, select **Mfg Setup > Mfg Geometry > Mill Surf > Create**, and name the surface [**FACE_SURF**].
2. Select **Add > Flat > Done** to create a flat mill surface.
3. When prompted to, select a sketching plane move the cursor over the hidden top surface of the reference model, right-click until the reference model surface highlights, then select it as the sketching plane, as shown in the following figure.
4. Accept the arrow pointing down into the workpiece as the direction for viewing the sketching plane.

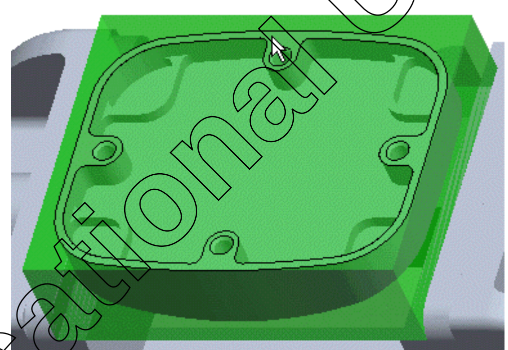


Figure 1: Showing cursor position and selected sketching surface.

5. When prompted to select a sketching reference, select the front surface of the workpiece as the bottom sketch view reference, as shown in the following figure.

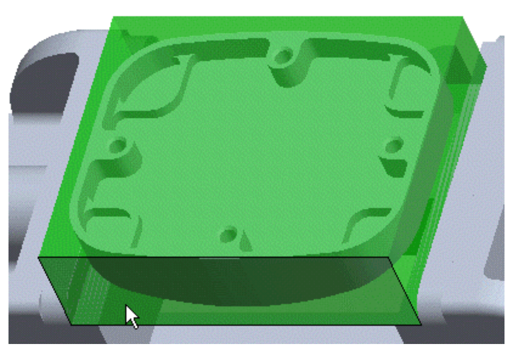


Figure 2: Showing cursor position and selected bottom sketching reference.

PTC

> **Note:**
> Selecting the sketching reference, enables the sketch to be orientated correctly when entering the sketcher tool.

6. **Close** the References dialog box and accept that not enough references have been specified.
7. Change your view to the named view **OP010-TRI**.
8. Click the Create an Entity from an Edge icon and select the four edges of the workpiece to create the sketch outline.

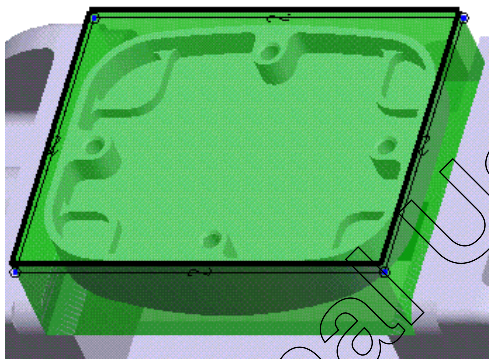


Figure 3: Showing selected edges of the workpiece.

9. Complete the sketch and close the Surface dialog box.
10. Return to the Manufacture menu.
11. Select the surface in the model tree, right-click and select **Hide** from the drop-down list.

> **Tip:**
> Using the model tree you can hide and unhide reference geometry as required.

Step 3. Create a face milling NC sequence.

We can now create a face milling sequence. We enter a name for the sequence to make it easy to identify. We can select the SH_MILL_50_0 tool. Notice how the site parameter file sets all the required manufacturing parameters; this includes reading in the speed, feed and depth-of-cut data for this tool. Select the top surface of the part to machine. Play the toolpath.

Notice how the toolpath is not square, but follows the outline of the surface we are machining.

We can change the TRIM_TO_WORKPIECE parameter to YES to see the effect on the toolpath. When we play the toolpath, notice how the toolpath now follows the workpiece shape instead of the surface we are machining.

1. Using the Menu Manager, select **Machining > NC Sequence > Face > Done** to create a face milling sequence.
2. In the Seq Setup menu, select the **Name** check box, and complete the selection of options.

> **Note:**
> The Seq Setup menu provides a list of options that can be configured for each NC Sequence. Some options are required, others are optional. You can reconfigure options at any time during the creation of an NC sequence. Notice the Tool, Parameters and Surfaces options were automatically selected as required.

3. Name the sequence [**FACE_BOTTOM**].
4. In the Tools Setup dialog box, select the **SH_MILL_50_0** tool, then right-click and select **Select Tool** and **Return**.
5. Change your view to the named view **OP010-TRI**.
6. In the Mfg Params menu, select **Done** to accept the default parameters.

> **Note:**
> Notice that you are not required to enter any parameters, the site parameter file configures all required parameters.

7. In the Surf Pick menu, accept the **Model** option to choose surfaces from the reference model.
8. Move the cursor over the hidden top surface of the reference model until the surface highlights, select it as the surface to machine, as shown in the following figure. Then complete the surface selection process.

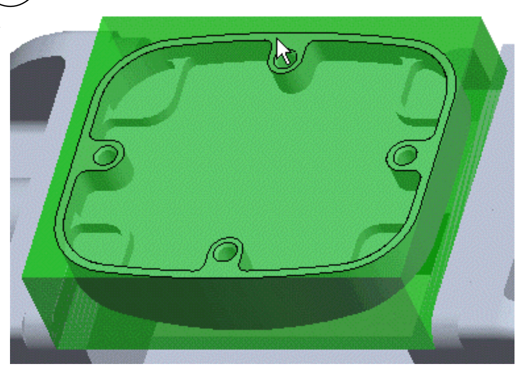


Figure 4: Showing cursor position and selected reference model surface.

9. Review the toolpath by selecting **Play Path > Screen Play**.
10. In the Play Path dialog box, click on the blue CL data bar, this will make the CL data visible in the dialog box. Click on the Play Forward button to review the toolpath. To slow down the toolpath simulation, click and drag the display speed slider to the left.

> **Note:**
> Notice that the corners of the toolpath are not straight. The toolpath uses the reference model surface being machined to determine the limits of the tool travel.

11. Close the Play Path dialog box when finished.

12. To modify the manufacturing parameters, click the Manufacturing Parameter Tree icon at the top of the graphics window. This will make the parameter window appear.

13. In the Parameter Tree window, click the Advanced button to see all the manufacturing parameters.

14. Modify the TRIM_TO_WORKPIECE parameter to **YES** then close the parameter tree window.

15. Play the toolpath again by selecting **Play Path > Screen Play**. Then click on the Play Forward button to review the toolpath. Notice how the toolpath adjusts to the shape of the workpiece.

> **Tip:**
> To see the CL data when playing the toolpath, click on the CL Data arrow. Notice the PPRINT entries at the beginning of the CL data file.

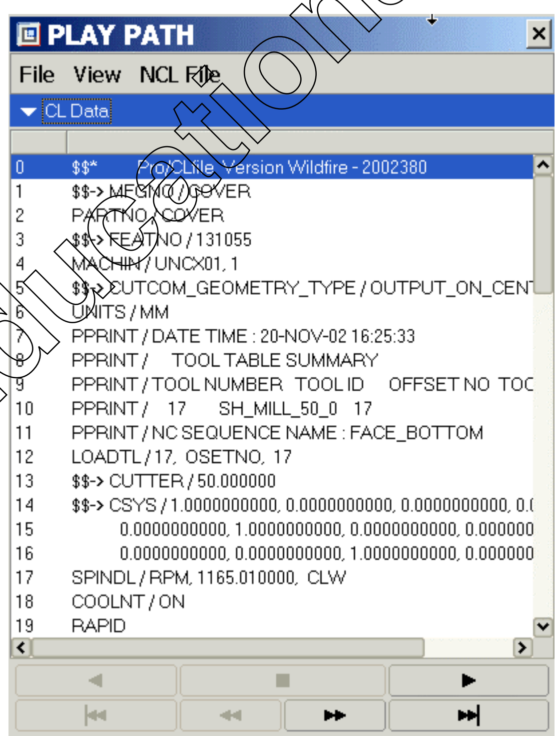


Figure 5: Showing PPRINT statements at beginning of CL data file.

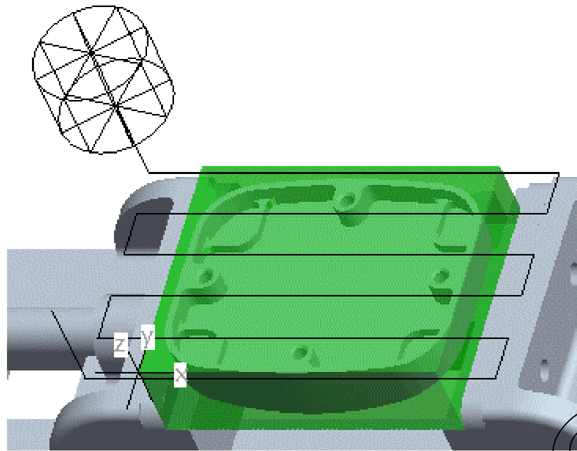


Figure 6: Toolpath with TRIM_TO_WORKPIECE set to YES

16. Close the Play Path dialog box when finished.

17. Open the Parameter Tree Window again and set the TRIM_TO_WORKPIECE parameter back to **NO**, close the parameter tree window to complete the modification.

Step 4. Use the mill surface as an alternative machining reference.

*We can use the previously created mill surface as an alternative reference. Once we select the surface we can confirm to use the upper side of the surface for machining, and also click to **Select All** of this surface for machining. This last option lets you select specific patches of surface quilts if required.*

We can play the toolpath and notice that the toolpath is square and is relative to the mill surface, not the top face of the reference model.

1. Using the Menu Manager, select **Seq Setup**, and select the **Surfaces** check box. Then select **Mill Surface > Done > Select Srf**.

2. In the Search Tool dialog box notice the surface named FACE_SURF is selected, and click **OK**.

3. When prompted, accept the upward facing side of the surface for machining, and use the **Select All** option to complete the surface configuration.

4. Play the toolpath again by selecting **Play Path >Screen Play**. Notice how the toolpath adjusts to the outline of the mill surface.

5. Close the Play Path dialog box when finished.

Step 5. Modify the sequence parameters to adjust the approach and exit moves, the step depth and the cut angle.

We can modify several of the toolpath parameters to see their effect on the toolpath.

Notice from the top view that the tool plunges close to the workpiece. We can specify an approach and exit move by setting APPROACH_DIST and EXIT_DIST parameters. We can also create a relation to the tool diameter, which updates if we change tools.

We can set the STEP_DEPTH to 2 to get several passes. We can also change the CUT_ANGLE to 90. The CUT_ANGLE is measured relative to the x-axis of the sequence co-ordinate system.

We can set the NUMBER_CUTS parameter to 3. This computes a step depth less than the 2 mm we entered previously and it will therefore override the STEP_DEPTH parameter and create 3 evenly spaced slices. We can also change the CUT_ANGLE back to 0.

1. Open the Parameter Tree window and set the following parameters in the NC sequence:

Tip:
You can adjust parameters by selecting the Parameter Tree icon at the top of the screen.

Table 2: Changed Parameters

Manufacturing Parameter	Value	Section
APPROACH_DISTANCE	CUTTER_DIAM/2	ENTRY/EXIT
EXIT_DISTANCE	CUTTER_DIAM/2	ENTRY/EXIT
STEP_DEPTH	2 *(0.08 in.)*	CUT PARAM
CUT_ANGLE	90	CUT PARAM

NOTE:
Notice some of the parameters, have brackets around the values. This means the parameter is taking its value from the site parameter file. You can type in values to break the links to the site parameter file. You can always reestablish the link by selecting the original value from drop down list.

2. Play the toolpath again by selecting **Play Path > Screen Play**.

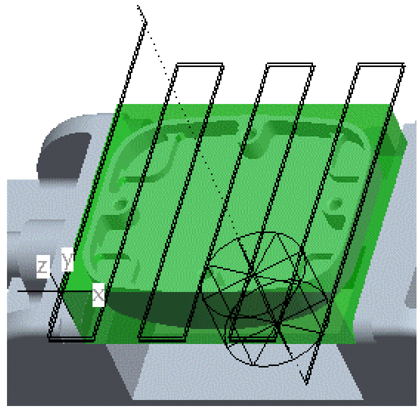


Figure 7: Showing the resultant toolpath.

Tool Path Review:

By setting an APPROACH_DISTANCE and EXIT distance, the tool clears the workpiece at the beginning and end of the toolpath. By setting a STEP_DEPTH of 2 *(0.08 in.)*, we have 3 passes (our stock is 4.4 mm) *(0.17 in.)*, notice the third pass is much smaller than the other two. The CUT_ANGLE makes the toolpath rotate 90 degrees relative to the X axis.

3. Open the Parameter Tree window and set the following parameters in the sequence:

Table 3: Changed Cut Parameters

Manufacturing Parameter	Value	Section
NUMBER_CUTS	3	CUT PARAM
CUT_ANGLE	0	CUT PARAM

4. Play the toolpath again by selecting **Play Path > Screen Play**.

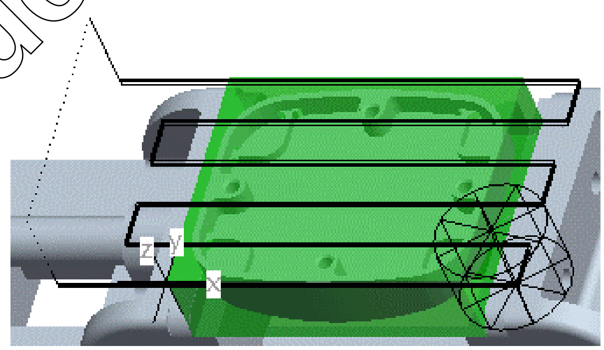


Figure 8: Showing the resultant toolpath.

Tool Path Review:

Setting the NUMBER_CUTS to 3 will compute a smaller step depth than 2 *(0.08 in.)* so NUMBER_CUTS will override the STEP_DEPTH parameter and we get 3 evenly spaced passes.

Step 6. Modify the sequence parameters to control the over travel on each pass. Check the final toolpath in Vericut.

We can modify the START_OVERTRAVEL and END_OVERTRAVEL to make the tool cut past the machined surface at the beginning and end of each pass.

We can also set the ENTRY_EDGE and CLEARANCE_EDGE to use the center of the tool as our reference point for determining the over travel distances.

We can set the EDGE_OFFSET parameters to move the toolpath toward or away from the initial and final edges (passes). A positive value moves the toolpath into the machined surface, whereas a negative value moves the toolpath away from the machined surface.

We can verify the toolpath in Vericut. We use NC Check to launch Vericut. When playing the toolpath, notice there are no collisions or gouging and we have machined down to our finish surface.

1. Open the Parameter Tree window and set the following parameters in the sequence:

Table 4: Changed Parameters

Manufacturing Parameter	Value	Section
START_OVERTRAVEL	25 *(0.98 in.)*	CUT PARAM
END_OVERTRAVEL	25 *(0.98 in.)*	CUT PARAM
ENTRY_EDGE	CENTER	ENTRY/EXIT
CLEARANCE_EDGE	CENTER	ENTRY/EXIT

2. Play the toolpath again using the **Play Path > Screen Play** options.

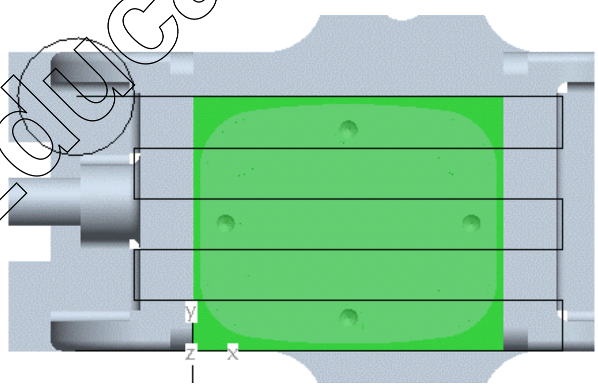


Figure 9: Showing the resultant toolpath.

Tool Path Review:

Changing the ENTRY_EDGE and CLEARANCE_EDGE to CENTER, we specify which part of the tool to measure the over travel from on each pass. Setting START_OVERTRAVEL and END_OVERTRAVEL to 25 *(0.98 in.)*, the center of the tool moves 25 mm *(0.98 in.)* past the machined surface for each approach and 25 mm *(0.98 in.)* past the machined surface for each exit.

3. Open the Parameter Tree window and set the following parameters in the sequence:

Table 5: Changed Offset Parameters

Manufacturing Parameter	Value	Section
INITIAL_EDGE_OFFSET	CUTTER_DIAM/4	CUT PARAM
FINAL_EDGE_OFFSET	CUTTER_DIAM/4	CUT PARAM

4. Play the toolpath again using the **Play Path > Screen Play** options.

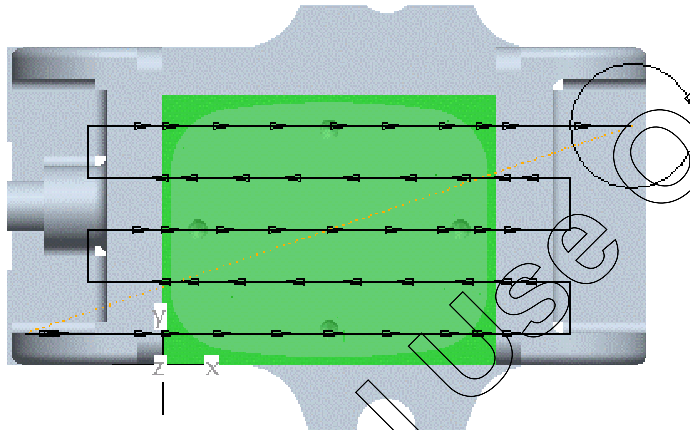


Figure 10: Showing the resultant toolpath.

Tool Path Review:

We can set the EDGE_OFFSET parameters to move the toolpath toward or away from the initial and final edges (passes). A positive value moves the toolpath into the machined surface, a negative value moves the away from the machined surface.

5. In the Play Path menu, select **NC Check** to start Vericut. (You may need to wait a few moments for Vericut to start). Click on the Play to End icon to view the machining simulation.

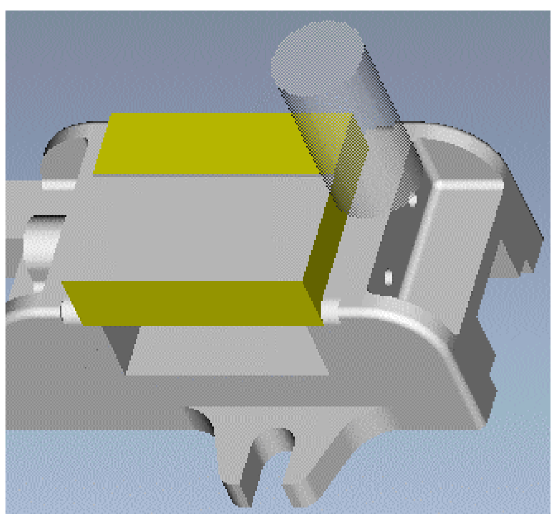


Figure 11: Showing the resultant toolpath in Vericut.

> **Tip:**
> To slow down the simulation, you can click on the Setup Motion icon, and in the Motion dialog box drag the animation speed slider to the left. When playing the toolpath, notice there are no collisions or gouging and we have machined down to the finish surface.

6. Close Vericut and complete the sequence, and return to the Manufacture menu.

Step 7. Create a material removal feature to cut away the machined volume from the workpiece.

We can create a material removal feature to cut away the machined volume from the workpiece. Using the ***Automatic*** *option means the system automatically determines the material to remove from the workpiece. You also have the option to create the cut manually.*

We can select the ***AutoAdd*** *option to pick the workpiece automatically. By default, the cut will only be visible in the assembly, you will not see the cut if you open the workpiece on its own.*

1. Using the Menu Manager, select **Machining > Matrl Removal > FACE_BOTTOM** to create a material removal feature for the face milling sequence.
2. Accept the **Automatic** option, click the **Auto Add** button to select the workpiece to intersect, and complete the feature.

> **Note:**
> Notice the automatic cut created in the workpiece.

Step 8. Modify the face milling sequence and review the results.

We can change the SCAN_TYPE parameter to TYPE_ONE_DIR, this makes the tool cut in one direction only, retracting between passes.

1. Using the Menu Manager, select **Machining > NC Sequence > FACE_BOTTOM**, to redefine the face milling sequence.
2. When prompted **Suspend All** child features.

> **Note:**
> In this case, suspending child features temporarily removes the material removal cut feature from the workpiece.

3. Open the Parameter Tree window and set the following parameter in the sequence:

Table 6: New Scan Type Parameter

Manufacturing Parameter	Value	Section
SCAN_TYPE	TYPE_ONE_DIR	CUT OPTION

4. Play the toolpath again using the **Play Path > Screen Play** options.

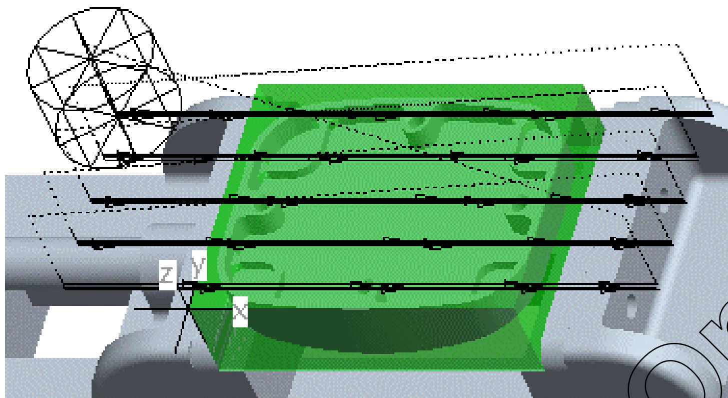


Figure 12: Showing the resultant toolpath.

> **Tool Path Review:**
> The TYPE_ONE_DIR setting will make the tool cut in only one direction, retracting between passes.

5. If time permits, set other SCAN_TYPE values and review the results. Return the SCAN_TYPE to TYPE_1 before completing the NC sequence.

6. Complete the NC sequence.

7. Save the manufacturing model.

8. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the face milling process.
- Describe the key face milling manufacturing parameters.
- Create face milling sequences.
- Create mill surfaces relevant to face milling.
- Create material removal features specific to face milling.

Module

8

Creating Profile Milling Sequences

Introduction

Profile milling sequences enable you to rough and finish mill both vertical and slanted surfaces. This is useful for machining the sidewalls of pockets and for machining outside profiles of components. When creating profile milling sequences, you can create mill surfaces and use them as alternate references when model surfaces are inappropriate, as when the model surface being machined has a variable depth. Another important skill with profile milling is the ability to refine the entry and exit motions of the toolpath to enable proper application of cutter compensation, and to eliminate witness lines from the machined surfaces. When a profile milling sequence is completed, you can update the in-process workpiece geometry. This is achieved by creating a material removal feature that removes the machined volume from the workpiece.

Objectives

After completing this module, you will be able to:

- Describe the profile milling process.
- Describe the key profile milling manufacturing parameters.
- Create profile milling sequences.
- Create mill surfaces relevant to profile milling.
- Create material removal features specific to profile milling.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

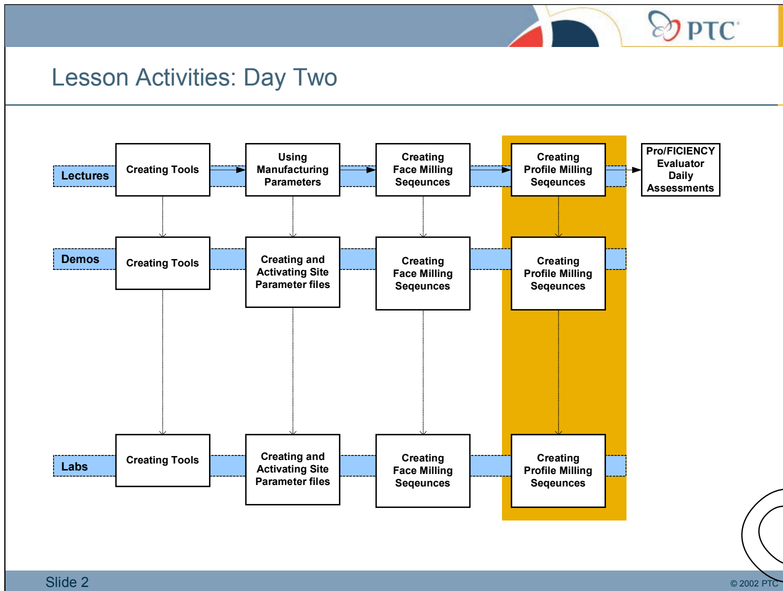
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

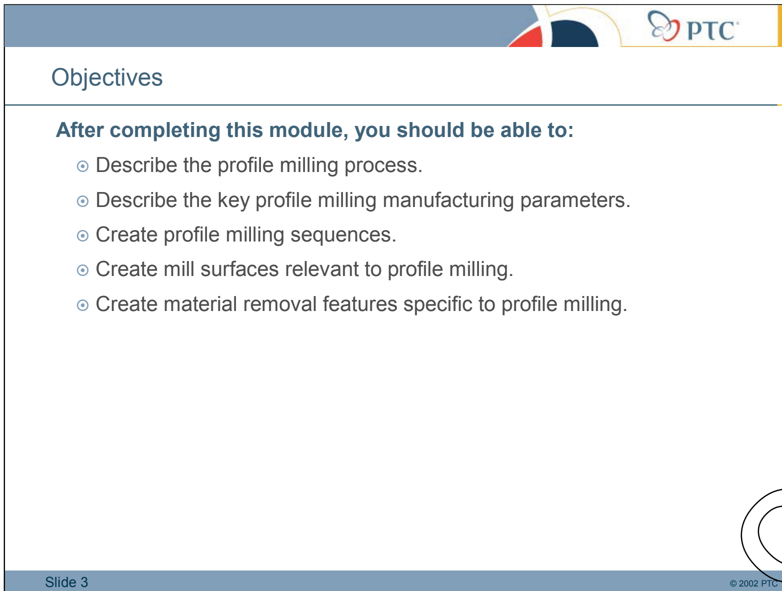
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos (1): 25 mins
- Labs (1): 50 mins
- Total: 1 hr 30 mins

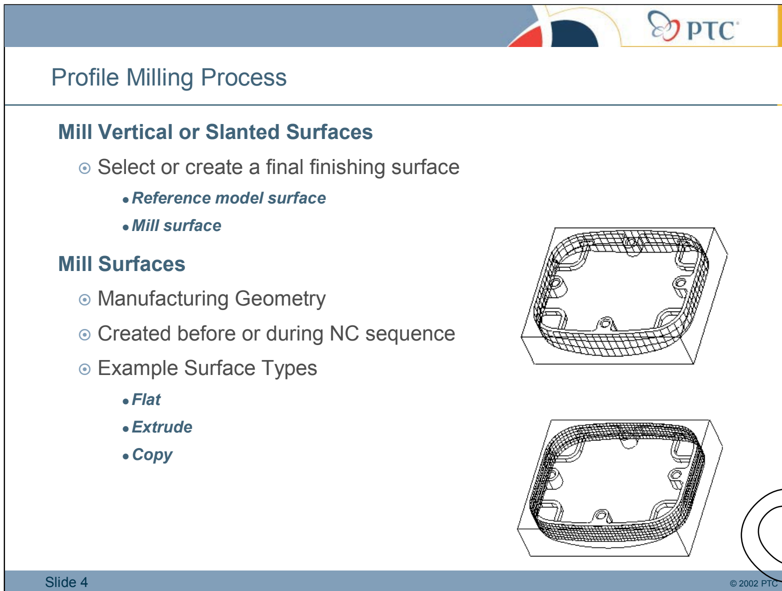


Overview

- Profile milling sequences enable you to rough and finish mill both vertical and slanted surfaces. This is useful for machining the sidewalls of pockets and for machining outside profiles of components. When creating profile milling sequences you can also create mill surfaces and use them as alternate references when model surfaces are inappropriate, for example when the model surface being machined has a variable depth. Another important skill with profile milling is the ability to refine the entry and exit motions of the toolpath to enable proper application of cutter compensation, and to eliminate witness lines from the machined surfaces. Once a profile milling sequence is completed, you can update the in-process workpiece geometry, this is achieved by creating a material removal feature that removes the machined volume from the workpiece.

After successfully completing this module, you will be able to:

- Describe the profile milling process.
- Describe the key profile milling manufacturing parameters.
- Create profile milling sequences.
- Create mill surfaces relevant to profile milling.
- Create material removal features specific to profile milling.



Profile Milling Process

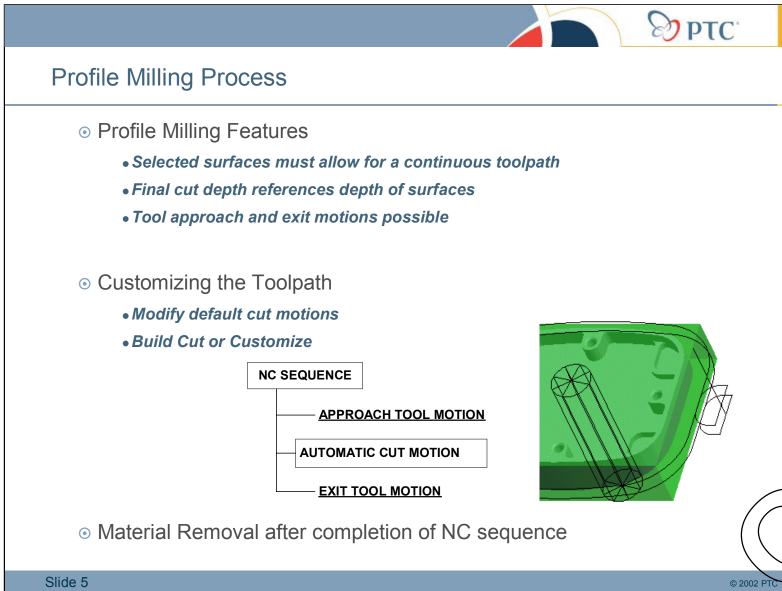
Profile milling sequences are used to rough or finish mill both vertical or slanted surfaces. The final finishing surface is configured by selecting reference model surfaces or creating a mill surface.

•Mill surfaces are known as manufacturing geometry.

•They can be created before an NC sequence, or during the creation of an NC sequence.

•The most common surface types used are:

- Flat: create a flat surface by sketching an outline on a sketching plane.
- Extrude: create an extruded surface by sketching an outline on a sketching plane.
- Copy: copy existing model surfaces to form a new mill surface.

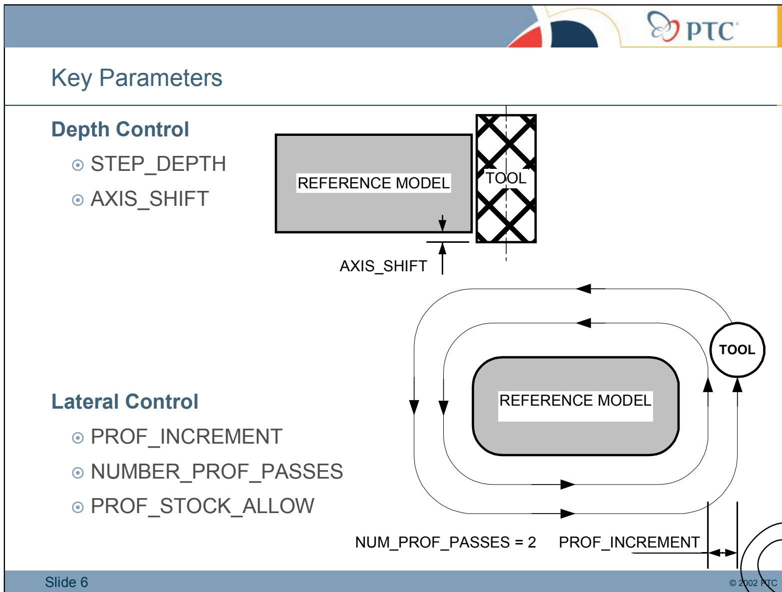


Profile Milling Features

- Selected surfaces must allow for a continuous toolpath.
- Final cut depth references depth of surfaces.
- Tool approach and exit motions are possible.

- By default tool-paths are generated based on the selected reference geometry and the manufacturing parameters.
  - It is possible to modify these default cut-motions by adding alternative approach and exit moves.
  - Two tools are available to do this:
    - Build Cut: the Build Cut functionality lets you add or remove slices or cutting passes, and specify approach and exit paths.
    - Customize: allows modification of the default cut motions, this includes removing default cut-motions and adding your own cut-motions including approach and exit tool-motions.
    - Both tools provide essentially the same functionality, but use different interfaces.

Once the toolpath is completed it is possible to remove the machined volume from the workpiece by creating a material removal feature.



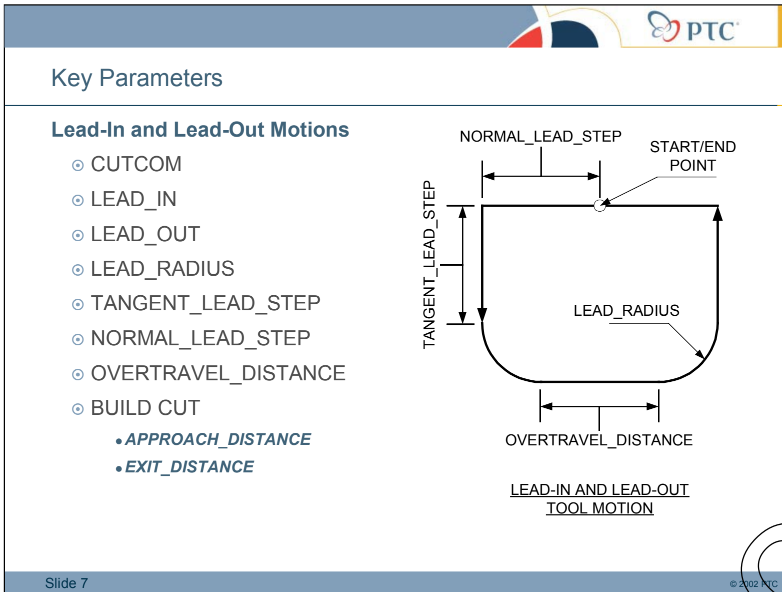
Key Parameters

Depth Control

- STEP_DEPTH: the incremental depth of each cut for each profile pass.
- AXIS_SHIFT: moves the CL data up or down along the tool axis.

Lateral Control

- PROF_INCREMENT: horizontal distance between passes.
- NUM_PROF_PASSES: number of profile passes at each depth.
- PROF_STOCK_ALLOW: remaining stock material left on profile milled surfaces.



Key Parameters – Lead-In and Lead-Out Motions

CUTCOM: turns cutter compensation on or off in the CL data file. When configuring a workcell you can specify if the cutter compensation is to the tool center (default) or tool edge.

LEAD_IN/LEAD_OUT: set to YES, tool enters and exits the workpiece along a tangent arc.

LEAD_RADIUS: radius value of the tangent circle move for lead-in or lead-out.

TANGENT_LEAD_STEP: linear distance of the tangent section of a lead in or lead out motion.

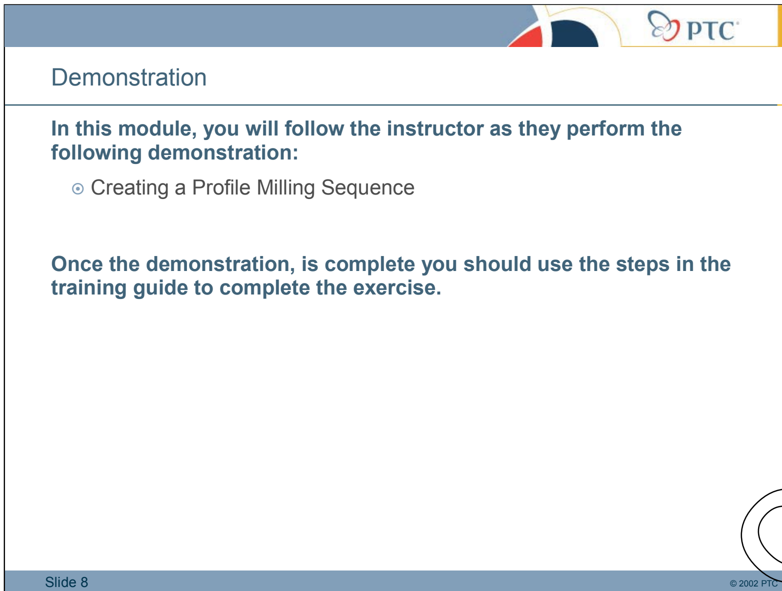
NORMAL_LEAD_STEP: linear distance of the normal section of a lead in or lead out motion.

OVERTRAVEL_DISTANCE: for profile milling the distance the tool travels past the start of a profile pass.

BUILD CUT: for profile milling, build cut is an optional element. It enables creation of approach and exit moves, these can be used as an alternative to lead-in and lead-out motions.

APPROACH_DISTANCE: used when creating cut motions or using build cut. The distance of an approach move for a tangent or normal approach.

EXIT_DISTANCE: used when creating cut motions or using build cut. The distance of an exit move for a tangent or normal exit.



Demonstration: Creating Profile Milling Sequences

Scenario

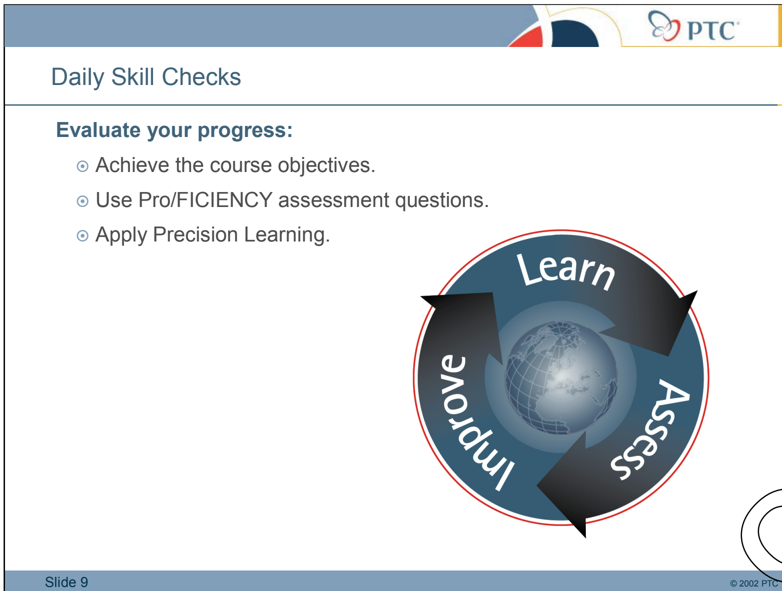
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY.
- **Improve:** The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Getting Started

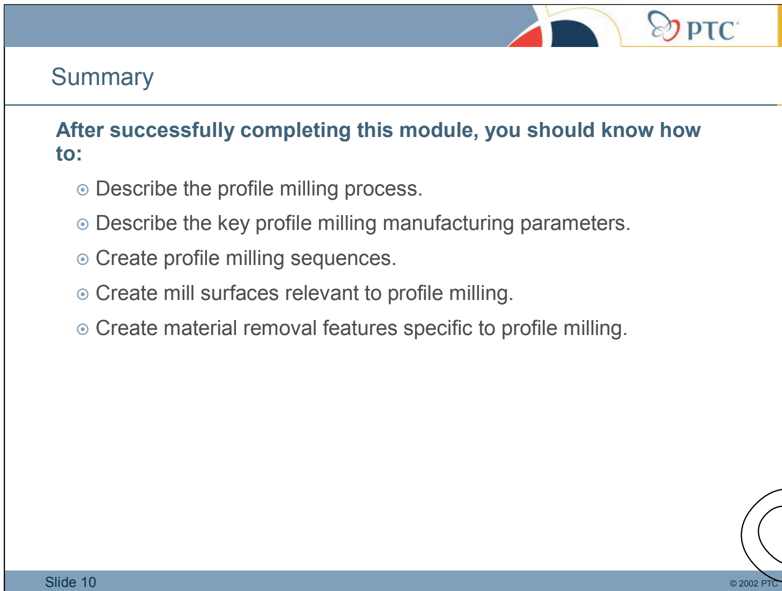
- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Tests

Description: For each course, 5 new 10 question exams based upon the topics covered each day.

How to use it:

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After successfully completing this module, you should know how to:

- Describe the profile milling process
- Describe the key profile milling manufacturing parameters
- Create profile milling sequences
- Create mill surfaces relevant to profile milling
- Create material removal features specific to profile milling

Module 8 Lab Exercises

Exercise 1: Creating Profile Milling Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** [dw1]as described in the classroom setup notes.

Introduction

In this demonstration, you create a profile milling sequence. You initially use a reference model surface to specify the surface to machine. You then use a mill surface for the profile milling sequence to create a modified toolpath that ensures the final pass will be at a constant z-depth. You also adjust the parameters to create approach and exit motions to eliminate witness lines from the machined surfaces.

Objectives

After successfully completing this exercise, you will know how to:

- Create profile milling sequences.
- Create mill surfaces relevant to profile milling.
- Create material removal features specific to profile milling.

Scenario

The next step as you continue machining the aluminum cover is to create a profile milling sequence to machine the material around the outside of the component. During the creation of this sequence, you adjust sequence parameters to create a more efficient toolpath. You then redefine the sequence to reference a mill surface that ensures the final pass will be at a constant z-depth. You also create lead-in and lead-out motions to eliminate witness lines from the machined surfaces.

Parameters

The following are key profile milling parameters.

Table 1: Parameter Descriptions

Parameter	Description
STEP_DEPTH	The incremental depth of each cut.
AXIS_SHIFT	Moves the CL data up or down along the tool axis.
NUM_PROF_PASSES	Number of profile passes at each depth.
PROF_INCREMENT	Horizontal distance between passes.
PROF_STOCK_ALLOW	Remaining stock material left on profile milled surfaces.
CUTCOM	Turns cutter compensation on or off in the CL data file.
LEAD_IN	Set to YES, tool enters workpiece along a tangent arc.
LEAD_OUT	Set to YES, tool exits workpiece along a tangent arc.
LEAD_RADIUS	Radius value of the tangent circle move for lead-in or lead-out.
NORMAL_LEAD_STEP	Linear distance of the normal section of a lead in or lead out motion.
TANGENT_LEAD_STEP	Linear distance of the tangent section of a lead in or lead out motion.
OVERTRAVEL_DISTANCE	For profile milling the distance the tool travels past the start of a profile pass.
APPROACH_DISTANCE	Used when creating cut motions or using build cut. The distance of an approach move for a tangent or normal approach.
EXIT_DISTANCE	Used when creating cut motions or using build cut. The distance of an exit move for a tangent or normal exit.

Step 1. Locate and open the cover manufacturing model.

Before creating the profile milling sequence, we need to locate the manufacturing model and open it using the folder navigator. When we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module08**, right-click, and select **Make Working Directory.**

2. Open COVER.MFG. Turn off any displayed datum features.

Step 2. Create a mill surface as an alternative reference for the profile milling sequence.

As an alternative to machining the reference model surfaces, we can create a mill surface to control the depth of the final profiling pass.

Mill surfaces are useful because they can be used as alternative references when model references are not appropriate.

We need to create an extruded surface from the top of the reference model and with the same outline as the outer profile of the reference model. The depth should be deep enough to machine the profile without gouging the vise.

*To do this we create a mill surface using the **Extrude** option. We select the top of the reference model as our sketching plane. In the sketcher environment, we click the Use Edge option and Loop check box and select the top surface of the reference model to get the required edges.*

*When the surface is completed we can select the surface in the model tree, right-click and select **Hide** from the drop down list to blank the surface.*

1. Using the Menu Manager, select **Mfg Setup > Mfg Geometry > Mill Surf > Create**, and name the surface [**PROF_SURF**].
2. Select **Add > Extruded > Done**. Accept the attributes of **One Side** and **Open Ends**.
3. When prompted to select a sketching plane move the cursor over the hidden top surface of the reference model, right-click until the reference model surface highlights, then select it as the sketching plane, as shown in the following figure.

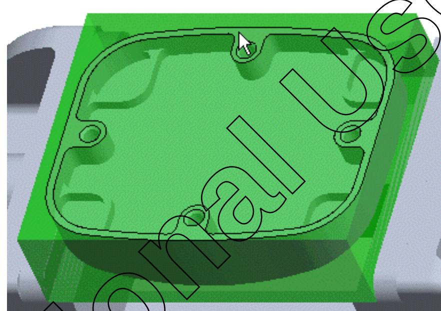


Figure 1: Showing cursor position and selected sketching surface.

4. Accept the arrow pointing down into the workpiece as the direction for viewing the sketching plane.
5. When prompted to select a sketching reference, select the front surface of the workpiece as the bottom sketch view reference, as shown in the following figure.

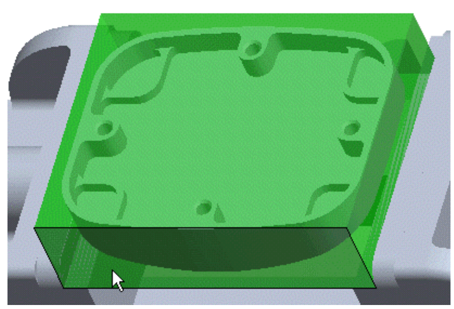


Figure 2: Showing cursor position and selected bottom sketching reference.

6. **Close** the references dialog box and accept that not enough references have been specified.
7. Change the view to the named view **OP010-TRI**.

Note:
Using the named view makes selecting the edges easier.

8. Click the Create an Entity from an Edge icon and using the Loop option, select the top surface of the reference model as shown in the following figure. **Accept** the outer loop of edges.

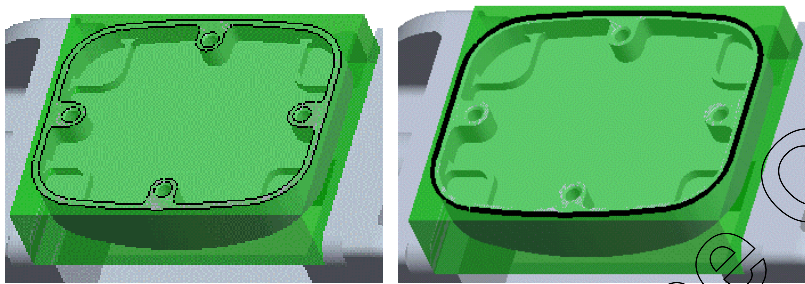


Figure 3: Showing selected loop surface and highlighted outer loop of edges

9. Complete the sketch.
10. Specify a **Blind** surface depth of [**16**] *(0.63 in.)*.
11. **Preview** and complete the extruded surface.
12. Return to the Manufacture menu.
13. Select the surface in the model tree, right-click and select **Hide** from the drop-down list.

Tip:
Using the model tree you can hide and unhide reference geometry as required.

Step 3. Create a profile milling NC sequence.

We can create a profile milling sequence. We enter a name for the sequence to make it easy to identify. We can select the FEM_25_0 tool. Notice how the site parameter file sets all the required manufacturing parameters; this includes reading in the speed, feed and depth-of-cut data for this tool. However, we can set the STEP_DEPTH to 10 before proceeding. We need to select the sidewalls of the reference model for machining. We can do this by choosing the loop option, selecting the top surface on the reference model, and selecting an outer edge to configure the surfaces to be machined. When playing the toolpath, notice the tool only makes one complete pass around the part. In the second cut, the tool cannot make a complete pass because the model surface is not continuous at that depth.

1. Using the Menu Manager, select **Machining > NC Sequence > New Sequence > Profile > Done**, to create a profile milling sequence.

2. In the Seq Setup menu, select the **Name**, **Tool** and **Parameters** check boxes, and complete the selection of options.

3. Name the sequence [**PROF_BOTTOM**].

4. Select the **FEM_25_0** tool for the sequence.

5. Change the view to the named view **OP010-TRI**.

> **Note:**
> Notice that the site parameter file configures values for all required parameters.

6. In the Mfg Params menu, select **Set**. In the Parameter Tree window, click **Advanced** to see all the manufacturing parameters, and modify the STEP_DEPTH parameter to [**10**] *(0.39 in.)*

7. Accept the **Model** option to choose surfaces from the reference model. When prompted to select surfaces to be machined use the **Loop** option.

8. Move the cursor over the hidden top surface of the reference model, right-click until the reference model surface highlights, then select it as the loop surface, as shown in the following figure.

9. When prompted select the edge as shown in the following figure, and complete the surface selection process.

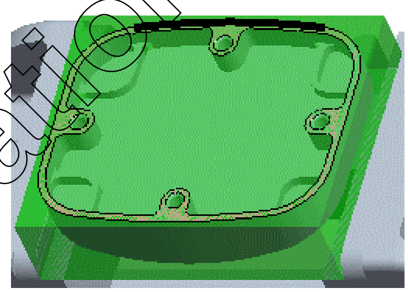


Figure 4: Showing selected reference model loop surface and selected edge.

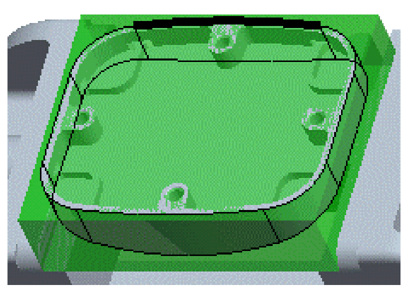


Figure 5: Showing highlighted edge and resulting vertical reference model surfaces.

10. Play the tool path using the **Play Path > Screen Play** options.

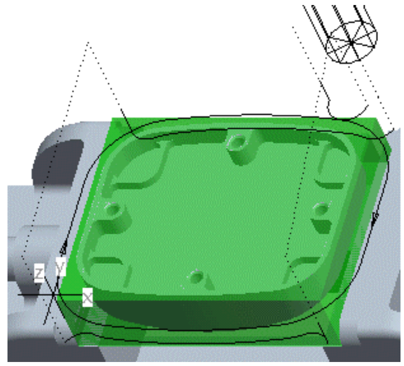


Figure 6: Showing the resultant toolpath.

> **Note:**
> Notice the tool only makes one complete pass around the model. In the second cut, the tool cannot make a complete pass because the model surface is not continuous at that depth.

Step 4. Use the extruded mill surface as an alternative machining reference.

We can use the previously created mill surface as an alternative reference. Once we select the surface, we can confirm to use the outward pointing surfaces for machining, and also click to ***Select All*** *of these surfaces for machining. This last option lets you select specific patches of surface quilts if required.*

We can play the toolpath and notice that the toolpath now makes two complete passes around the model and adjusts the depth of the final profile pass to the depth of the mill surface.

1. Using the Menu Manager, select **Seq Setup**, and select the **Surfaces** check box. Then select **Mill Surface > Done > Select Srfs**.
2. In the Search Tool dialog box, select the surface named PROF_SURF, and click **OK**.
3. When prompted, accept the outward facing side of the surface for machining, and use the **Select All** option to complete the surface configuration.

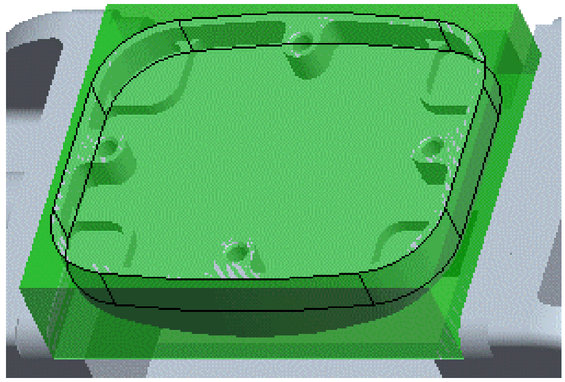


Figure 7: Showing the mill surface.

4. Play the toolpath again by selecting **Play Path >Screen Play**. Notice how the toolpath adjusts to the outline of the mill surface.

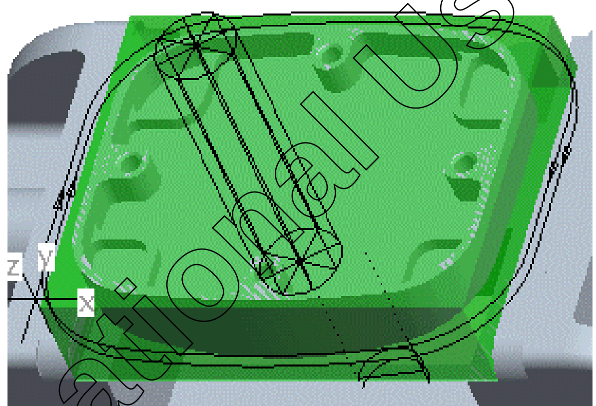


Figure 8: Showing the resultant toolpath.

5. Close the Play Path dialog box when finished.

Step 5. Modify the sequence parameters to adjust the entry and exit moves.

Notice how lead-in and lead-out moves already exist, this is because some of the site parameter file values have partially configured lead-in and lead-out moves.

We can change several of the toolpath parameters to modify the lead-in and lead-out moves on the toolpath.

We can specify a NORMAL_LEAD_STEP and TANGENT_LEAD_STEP of 10. This provides us with lead-in and lead-out moves where the cutter compensation is applied and removed.

We can also specify the OVERTRAVEL_DISTANCE as 5. This creates an over travel at the end of each pass ensuring that witness lines are removed.

1. Open the Parameter Tree window and check the values of the following parameters; notice these values have been set from the site parameter file.

Table 2: Entry/Exit parameters set by the site parameter file.

Manufacturing Parameter	Site Parameter Value
CUTCOM	ON
LEAD_IN	YES
LEAD_OUT	YES
LEAD_RADIUS	10 *(0.39 in.)*

2. Set the following parameters in the NC Sequence:

Table 3: Changed Parameters

Manufacturing Parameter	Value	Section
TANGENT_LEAD_STEP	10 *(0.39 in.)*	ENTRY/EXIT
NORMAL_LEAD_STEP	10 *(0.39 in.)*	ENTRY/EXIT
OVERTRAVEL_DISTANCE	5 *(0.20 in.)*	ENTRY/EXIT

3. Play the toolpath again using the **Play Path > Screen Play** options.

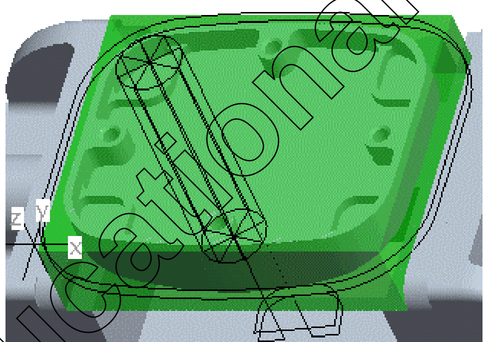


Figure 9: Showing the resultant toolpath.

Tool Path Review:

Notice during the lead-in and lead-out moves the cutter compensation is applied. Also the over travel distance on each pass removes any witness lines from the machined surfaces, however, we want to create a specific approach and exit position.

Step 6. Create a specific approach and exit position using build cut functionality.

We can create a specific approach and exit position by building a cut motion and creating a user defined approach and exit axis.

We can create a cut motion using the build cut option by configuring an approach point and creating an axis positioned 24.5 from the right surface of the workpiece, and 54 from the front surface of the workpiece. We can use the same axis to configure an exit point.

We can set the NORMAL_LEAD_STEP to zero as this move is now controlled by the position of the axis. We can also set the TANGENT_LEAD_STEP to 5.

We can verify the toolpath in Vericut. We use NC Check to launch Vericut. When playing the toolpath, notice there are no collisions or gouging and we have machined down to our finish surface.

1. Turn on the display of datum axis.
2. Using the Menu Manager, select **Seq Setup > Build Cut > Done** to use the build cut option to create approach and exit moves.
3. Using the **Approach**, **Point** and **Each Slice** options, **Create** an axis to specify the start point of the approach move.
4. When the Datum Axis dialog box appears, select the top surface of the workpiece to position the datum axis as shown in the following figure. Click and drag the positioning handles to snap onto the front and right edges of the workpiece as shown.

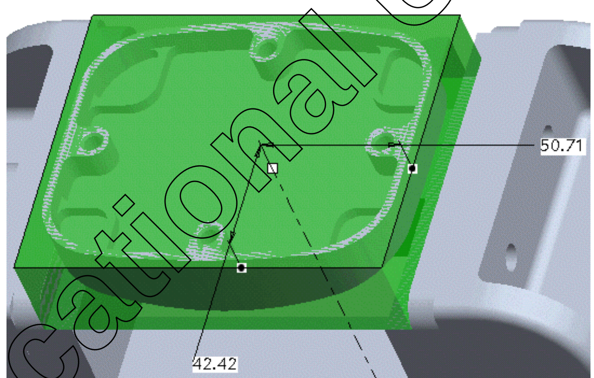


Figure 10: Showing initial axis position

5. Modify the dimensions so that the axis is positioned 24.5 *(0.96 in.)* from the right edge of the workpiece, and 54 *(2.13 in.)* from the front edge of the workpiece, as shown below. Complete the datum axis configuration.

Note:
You may need to enter negative values for the dimensions to get the axis to move to the right of the workpiece.

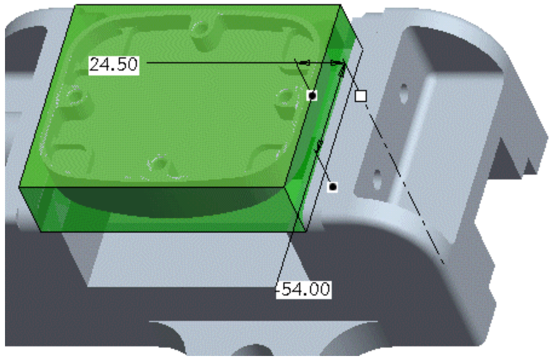


Figure 11: Showing axis position.

6. Create an **Exit** position and **Select** the same axis to configure the exit point, then complete the build cut.

7. Set the following parameters in the sequence:

Table 4: Changed Parameters

Manufacturing Parameter	Value	Section
TANGENT_LEAD_STEP	5 *(0.20 in.)*	ENTRY/EXIT
NORMAL_LEAD_STEP	0	ENTRY/EXIT

8. Play the toolpath again using the **Play Path > Screen Play** options.

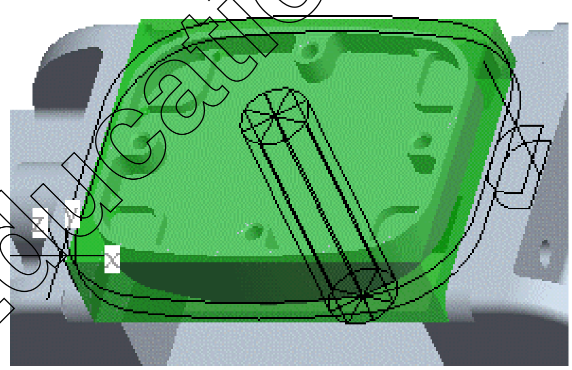


Figure 12: Showing the resultant toolpath.

Tool Path Review:
You can control the approach and exit position on the profile. Notice the NORMAL_LEAD_STEP is zero as this move is controlled by the position of the axis.

Step 7. Create an additional profiling pass to reduce the lateral depth of cut.

We can add an additional profiling pass for each slice of the profile; this reduces the lateral depth of cut. We change the NUM_PROF_PASSES to 2 and the PROF_INCREMENT to 2.5.

Notice how the toolpath now makes two passes at each depth.

1. Set the following parameters in the sequence:

Table 5: Changed Parameters

Manufacturing Parameter	Value	Section
NUM_PROF_PASSES	2	CUT PARAM
PROF_INCREMENT	2.5 *(0.10 in.)*	CUT PARAM

2. Play the toolpath again using the **Play Path > Screen Play** options.

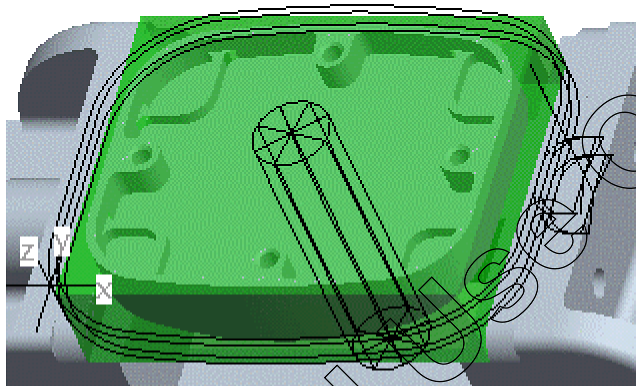


Figure 13: Showing the resultant toolpath.

Tool Path Review:

We can add an additional profiling passes for each slice of the profile; this reduces the lateral depth of cut. We change the NUM_PROF_PASSES to 2 and the PROF_INCREMENT to 2.5 *(0.10 in.)*. Notice how the toolpath now makes two passes at each depth.

3. In the Play Path menu, select **NC Check** to start Vericut.

4. Click on the Play to End icon to view the machining simulation.

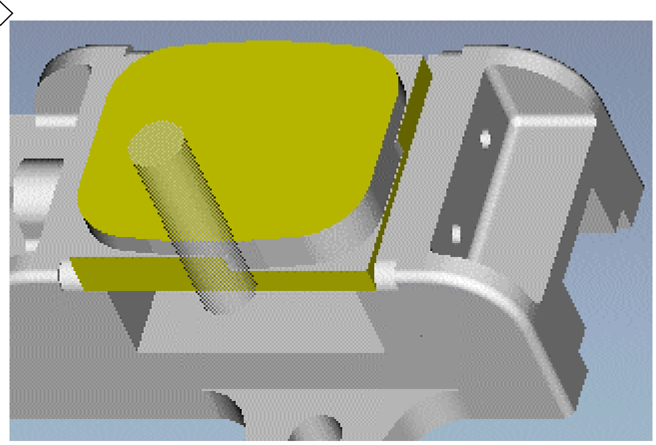


Figure 14: Showing the resultant toolpath in Vericut.

> **Tip:**
> To slow down the simulation you can click on the Setup Motion icon, and in the motion dialog box drag the animation speed slider to the left. When playing the toolpath, notice there are no collisions or gouging and we have machined down to the finish profile surface.

5. Close Vericut, complete the sequence, and return to the manufacture menu.

Step 8. Create a material removal feature to cut away the machined volume from the workpiece.

We can create a material removal feature to cut away the machined volume from the workpiece. Using the ***Automatic*** *option means the system automatically determines the material to remove from the workpiece. You also have the option to create the cut manually.*

We can select the ***AutoAdd*** *option to pick the workpiece automatically. By default, the cut will only be visible in the assembly, you will not see the cut if you open the workpiece on its own.*

1. Using the Menu Manager, select **Machining > Matrl Removal > PROF_BOTTOM** to create a material removal feature for the profile milling sequence.
2. Accept the **Automatic** option, click the **Auto Add** button to select the workpiece to intersect, and complete the feature.

> **Note:**
> Notice the automatic cut created in the workpiece.

3. Complete the NC sequence.
4. Save the manufacturing model.
5. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the profile milling process.
- Describe the key profile milling manufacturing parameters.
- Create profile milling sequences.
- Create mill surfaces relevant to profile milling.
- Create material removal features specific to profile milling.

Module

9

Creating Volume Milling Sequences

Introduction

The ability to remove large amounts of workpiece material is an important machining skill. Volume milling sequences are typically used for this purpose. Examples include machining mold cavities, machining pockets and slots, and facing down a workpiece. The volume milling process involves specifying the volume of material to be removed; this is achieved by creating either mill windows or mill volumes. When a volume milling sequence is completed you can update the in-process workpiece geometry. This is achieved by creating a material removal feature that removes the machined volume from the workpiece.

Objectives

After completing this module, you will be able to:

- Describe the volume milling process.
- Describe mill windows and mill volumes.
- Describe the key volume milling manufacturing parameters.
- Create volume milling sequences.
- Create mill windows.
- Create material removal features specific to volume milling.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

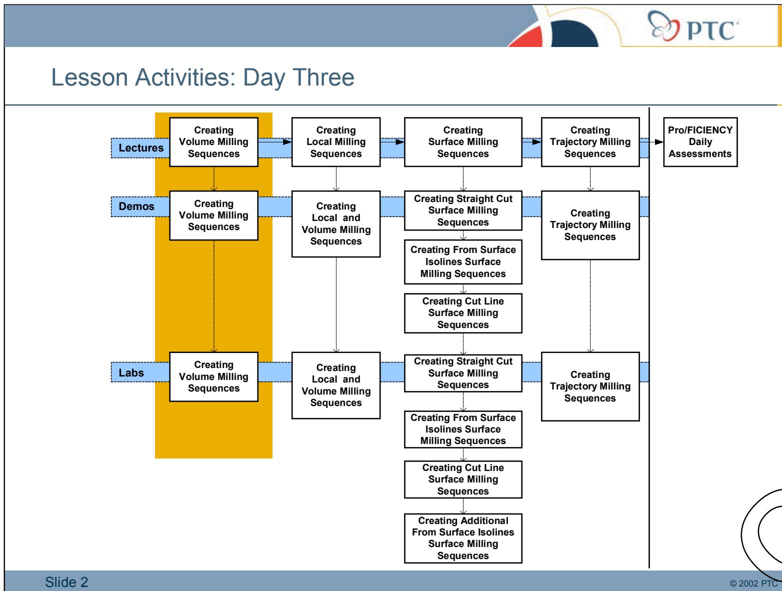
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

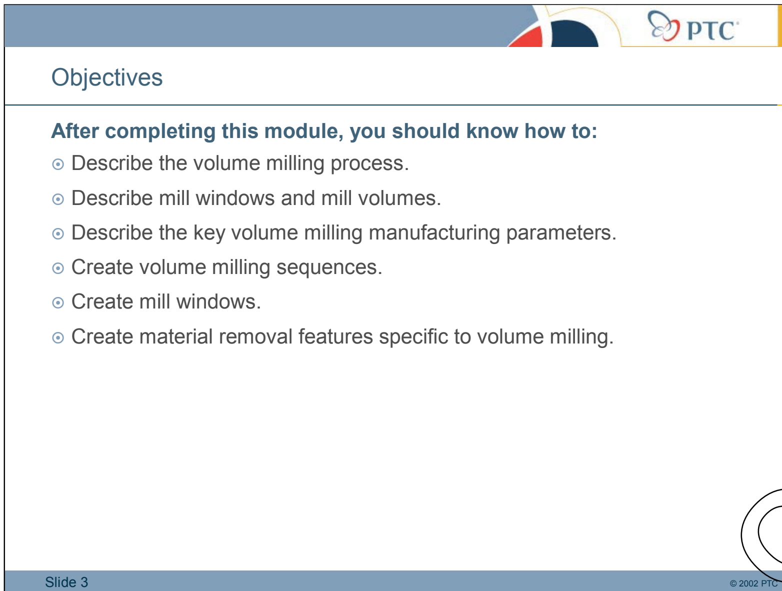
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 20 mins
- Demos : 25 mins
- Labs : 45 mins
- Total: 1 hr 30mins

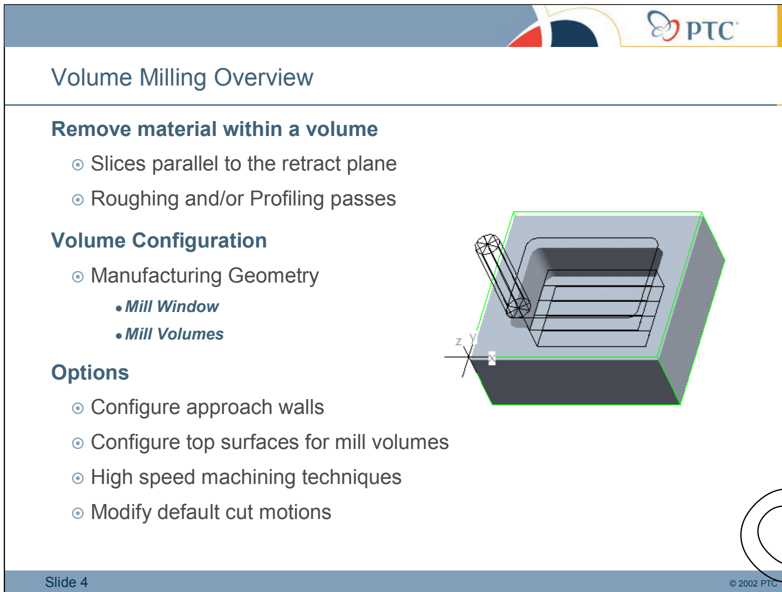


Overview

The ability to remove large amounts of workpiece material is an important machining skill. Volume milling sequences are typically used for this purpose. Examples include machining mold cavities, machining pockets and slots, and facing down a workpiece. The volume milling process involves specifying the volume of material to be removed; this is achieved by creating either mill windows or mill volumes. Once you have completed your NC sequence, you can represent the as-machined state of your workpiece by creating a material removal feature that removes the machined volume from the workpiece.

After completing this module, you will be able to:

- Describe the volume milling process.
- Describe mill windows and mill volumes.
- Describe the key volume milling manufacturing parameters.
- Create volume milling sequences.
- Create mill windows.
- Create material removal features specific to volume milling.



Volume Milling Process

Volume milling sequences are typically used to remove large amounts of workpiece material. Examples include machining mold cavities, and machining pockets and slots.

Volume milling sequences remove the material inside a specified volume slice-by-slice. All slices are parallel to the retract plane. Both roughing and profiling passes can be created within the sequence.

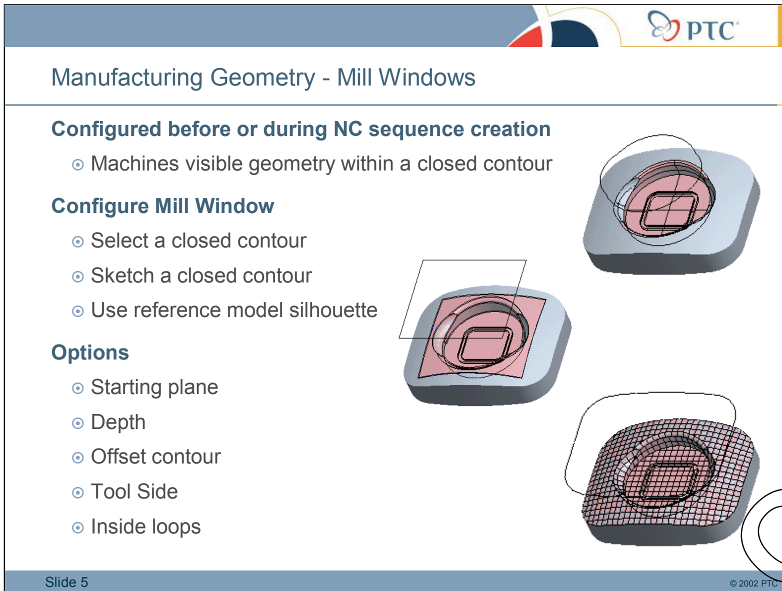
Volume Definition

Manufacturing geometry is used to configure the volume of material to machine, a Mill Window or Mill Volume can be created to represent the volume of material to be removed.

Options

By default the tool will not machine outside the specified volume.

- The tool can violate surfaces of a volume if they are specifically selected as approach walls.
- Top surfaces are surfaces of a mill volume that can also be penetrated by the tool when creating the tool path. This option has to be used only if some of the top surfaces of the volume are not parallel to the retract plane. If Mill Window is used, this option is not available. The window start plane will be used as the top surface.
- Various machining parameters are available that enable the creation of tool paths based on high speed machining principles.
- The default cut-motions can be modified by using the build-cut or customize functionality. Options include:
  - Automatic (default): use all the slices generated by the system.
  - Up to Depth: use slices up to a certain depth only.
  - From to depth: use slices in a certain range of depths.
  - Slice-by-Slice: generate Automatic Cut motions by specifying depth of each slice.
  - Approach and Exit motions can also be configured.



Mill Windows

Mill Windows are manufacturing geometry features used in creating volume milling sequences. They can be created before or during the creation of an NC sequence.

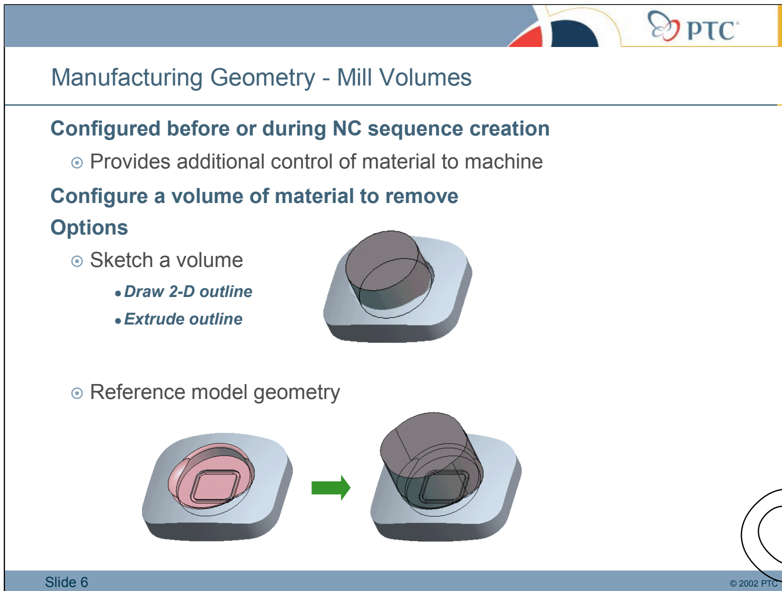
You create a Mill Window by:

- sketching or selecting a closed contour in an appropriate plane.
- projecting the silhouette of the reference part on the Mill Window start plane.

All reference model geometry visible within the window is machined.

Optional Elements

- Starting plane: defaults to retract plane.
- Depth: defaults to reference model geometry.
- Offset contour: enlarge or reduce contour by a constant distance.
- Tool Side: To/On/Past the mill window outline.
- Inside loops: if a reference part used for creating the silhouette contains through cuts or holes, you can specify if you want to keep or remove these loops by using the Inside Loops.

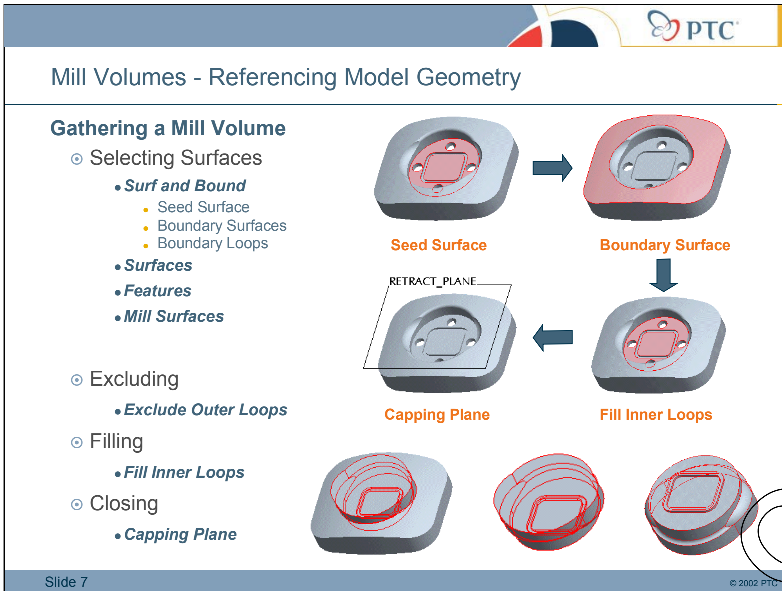


Mill Volumes

Mill Volumes are manufacturing geometry features used in creating volume milling sequences. They can be created before or during the creation of an NC sequence.

They are created by configuring a volume of material to remove. This can be done by:

- Sketching and extruding a volume.
- Referencing model geometry, then extruding the resulting outline.

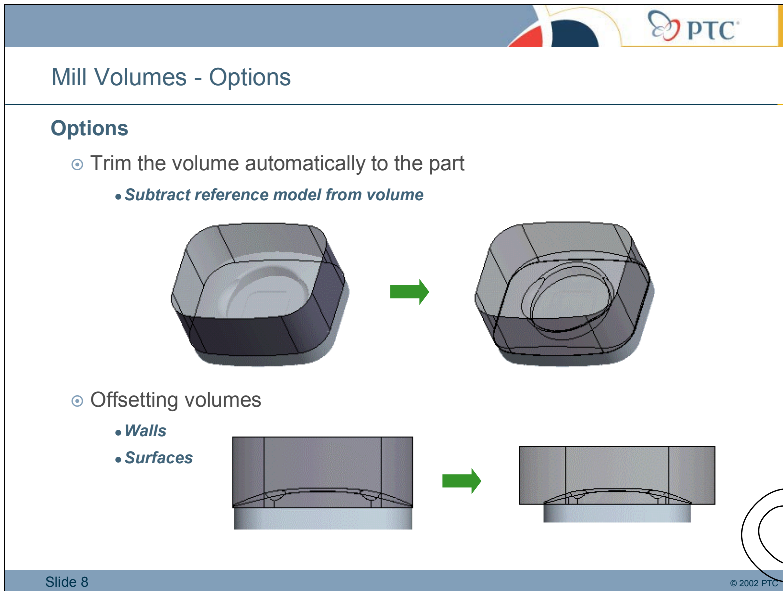


Mill Volumes – Referencing Model Geometry

The **gathering** technique allows you to create a mill volume by referencing model geometry including surfaces and edges.

The **gathering process** involves several steps:

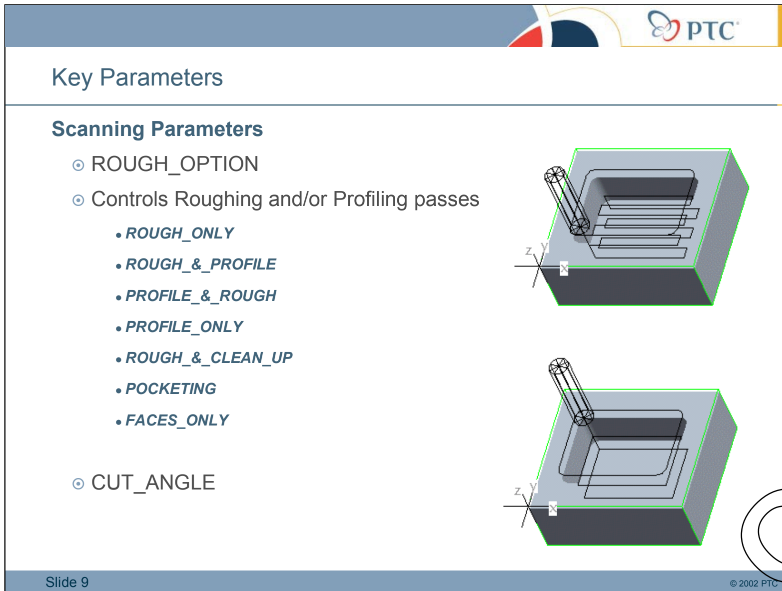
- **Selecting surfaces** to be machined; there are several methods available, with each method the selected surfaces are sewn together to form a single quilt which is by default extruded up to the retract plane to form a volume alternatively the quilt can be extruded up to some user defined plane if required.
  - **Surf and Bound**: This involves selecting one of the surfaces for machining (the **seed surface**), and then selecting **bounding surfaces**. The seed surface and all neighbouring surfaces up to the boundary surfaces will be sewn together into a single quilt. **Boundary loops** enables adding outer loops of edges to the boundary.
  - **Surfaces**: Select continuous surfaces to be machined.
  - **Features**: Select features to be machined. All the surfaces of selected features will be included.
  - **Mill Surfaces**: Select pre-defined Mill Surfaces.
- **Exclude**: The **Exclude** option is available only if you gather using an option other than **Surf & Bnd** (for example, **Surfaces**). It allows you to:
  - **Surfaces**—Exclude some of the chosen surfaces by selecting each of them individually. This is especially convenient when gathering using **Features** or **Mill Surf**.
  - **Loops**—Exclude outer loops. Use this option to delete unwanted portions of surfaces selected for gathering.
- **Fill**: When you fill an inner loop of edges on a surface selected for gathering it is equivalent to "patching" the base quilt of the Mill Volume. The volume will be built as if there was a smooth surface with no perforations. Options:
  - **All**—Fill all loops on a selected surface. Select a surface. All inner loops on this surface will be filled, whether they belong to bounding surfaces or not.
  - **Loops**—Select loops to be filled. For each loop to be filled, you have to select only one edge. If you gather using **Surf & Bnd**, the edges must lie on the bounding surfaces. Select additional bounding surfaces if necessary.
- **Close**: The system will close the volume automatically, by extruding the boundaries of the selected surface quilt vertically up to the retract plane. If you want to specify a different way to close the volume, select **Close**, and select an alternative capping plane.



Mill Volumes - Options

Options

- Trim the volume automatically to the part, this involves subtracting the reference model material from the mill volume. Usually applied to a sketched volume.
- It is possible to off-set selected surfaces or side walls of the volume by a distance.



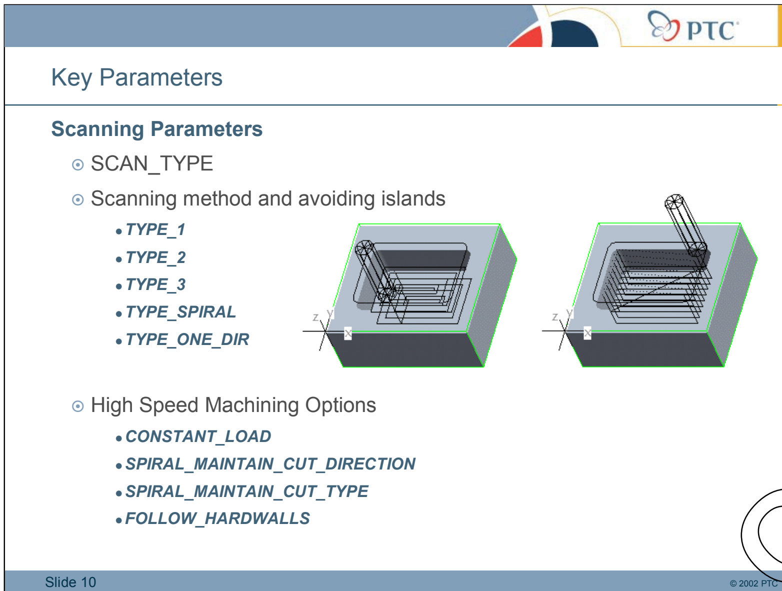
Scanning Parameters

ROUGH_OPTION

Controls if a profiling and/or roughing pass is created during volume milling

- ROUGH_ONLY—no profiling.
- ROUGH_&_PROF—rough then profile.
- PROF_&_ROUGH—profile then rough.
- PROF_ONLY—only profiling.
- ROUGH_&_CLEAN_UP—cleans up the walls of the volume without creating a profiling pass.
- POCKETING—profiles the walls of the volume and finish mills all the planar surfaces inside the volume that are parallel to the retract plane (island tops and bottom of the volume).
- FACES_ONLY—finish mills only the planar surfaces inside the volume that are parallel to the retract plane (island tops and bottom of the volume).

- CUT_ANGLE: the angle between the cut direction and the X-axis of the NC Sequence coordinate system.



Scanning Parameters

- **SCAN_TYPE**: For volume milling, refers to the way a milling tool scans the horizontal cross-section of a milling volume and avoids islands.

(for reference, describe as required)

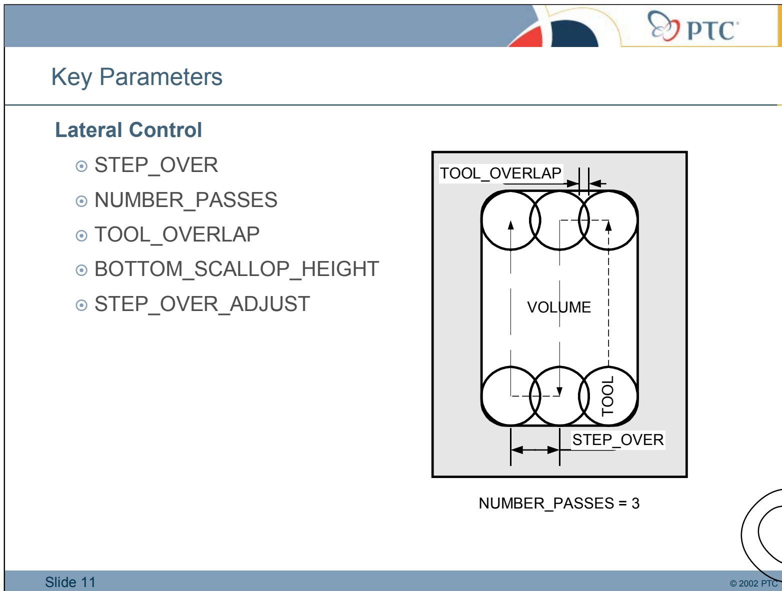
The options are:

- TYPE_1—continuously machines the volume, retracts upon encountering islands.
- TYPE_2—continuously machines the volume without retract, moving around the islands.
- TYPE_3—the tool removes material from continuous zones defined by the island geometry, machining them in turn and moving around the islands.
- TYPE_SPIRAL—generates a spiral cutter path.
- TYPE_ONE_DIR—the tool cuts in one direction only.
- TYPE_1_CONNECT—the tool cuts in one direction only. If there is a wall at the start of the cutting passes, the connection motion follows the profile of the wall to avoid gouging.

High Speed Machining Options (for reference only will be covered in a later module)

CONSTANT_LOAD—perform high speed roughing (with ROUGH_OPTION set to ROUGH_ONLY) or profiling (with ROUGH_OPTION set to PROF_ONLY).

- SPIRAL_MAINTAIN_CUT_TYPE—generates a spiral cutter path with reverse arc connections between cuts. This is a high speed machining option, which minimizes retracts.
- SPIRAL_MAINTAIN_CUT_DIRECTION—generates a spiral cutter path with S-shape connections between cuts. This is a high speed machining option, which minimizes retracts.
- FOLLOW_HARDWALLS—the shape of each cut follows the shape of the walls of the volume, maintaining fixed offset between the respective points of two successive cuts. If the cuts are closed, there are S-shape connections between the cuts.

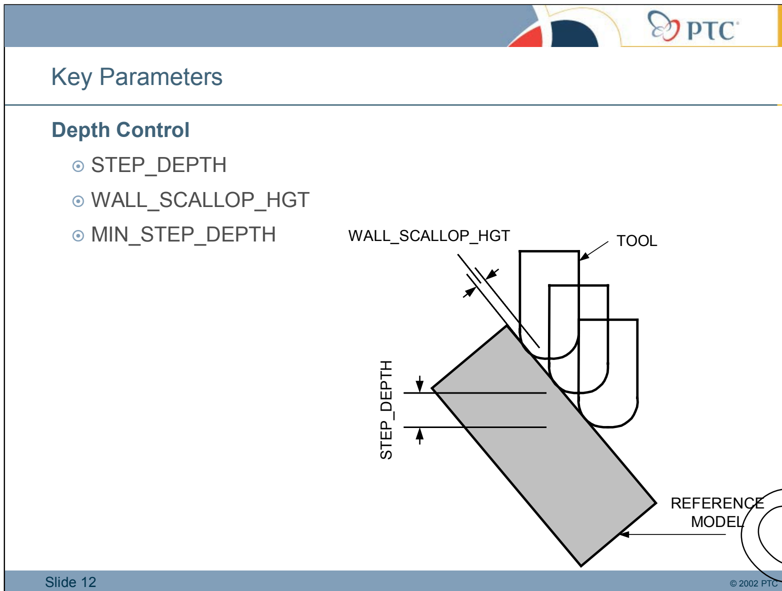


Lateral Control

These four parameters control the step-over distance, the final toolpath will use whichever parameter produces the smallest calculated step-over.

- STEP_OVER default parameter for controlling the step over within a slice.
- Or NUMBER_PASSES can be used to explicitly set the number of passes to take in each slice.
- Or TOOL_OVERLAP can be used as an alternative method to control the step-over based on the tool overlap.
- BOTTOM_SCALLOP_HEIGHT must be less than or equal to cutter radius, can also be used to calculate step-over.

- The STEP_OVER_ADJUST parameter adjusts the passes in the slice to start and finish near the edges of the volume you are machining. It only reduces the step over distance, and adds an extra pass if needed.



Depth Control

The STEP_DEPTH parameter can be used to specify the depth between each slice.

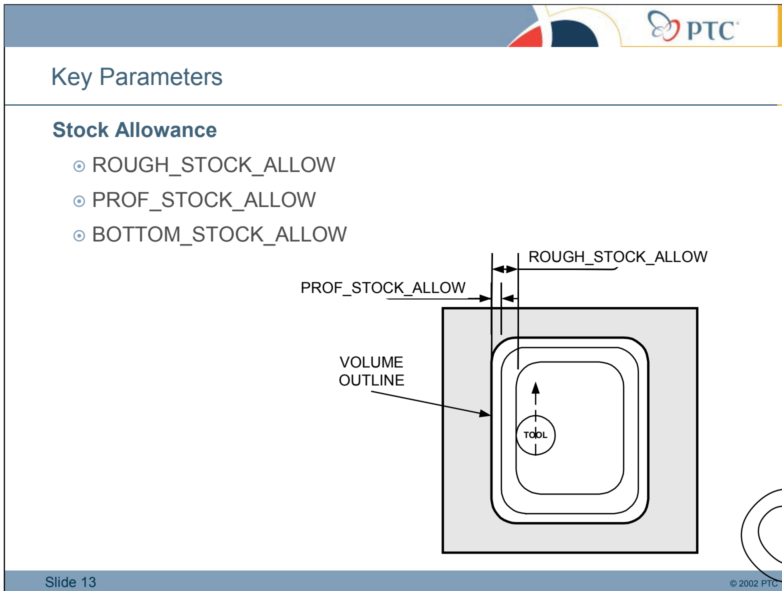
WALL_SCALLOP_HGT: controls the step depth for Volume milling.

WALL_SCALLOP_HGT (wsh) must be less than or equal to the cutter radius.

The default is 0.

- If WALL_SCALLOP_HGT is zero (*wsh* = 0), a scallop height is calculated using STEP_DEPTH.
- If you specify *wsh* > 0, a step depth is calculated using *wsh*. This calculated value is compared to the STEP_DEPTH, and the smallest calculated step-depth is used.

MIN_STEP_DEPTH: specifies the minimum allowable distance between slices.

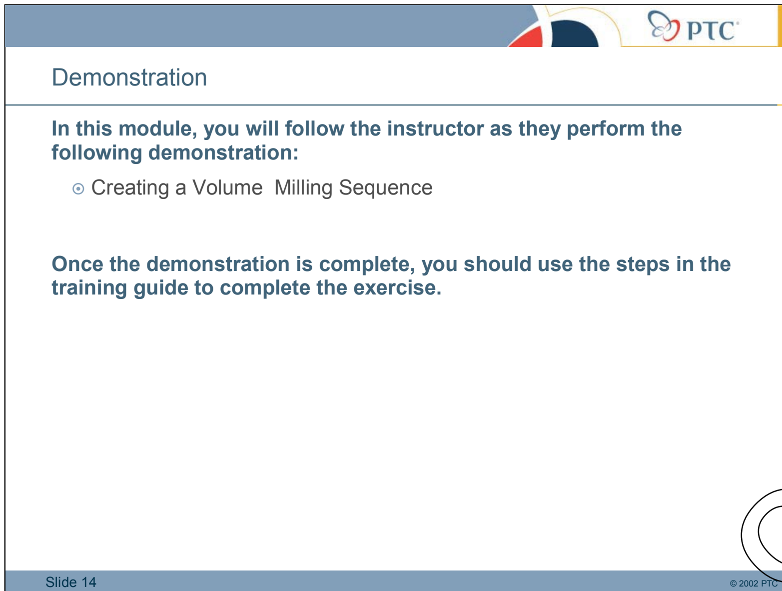


Stock Allowance

ROUGH_STOCK_ALLOW – controls stock on walls for roughing passes.

PROF_STOCK_ALLOW – controls the stock on the walls for profile passes. Is also used to calculate remaining stock when creating material removal features.

BOTTOM_STOCK_ALLOW – controls the stock on bottom faces for rough and profile passes. Defaults to PROF_STOCK_ALLOW if set to '-'.



Demonstration: Using Parameters and Setting up Site Files

Scenario

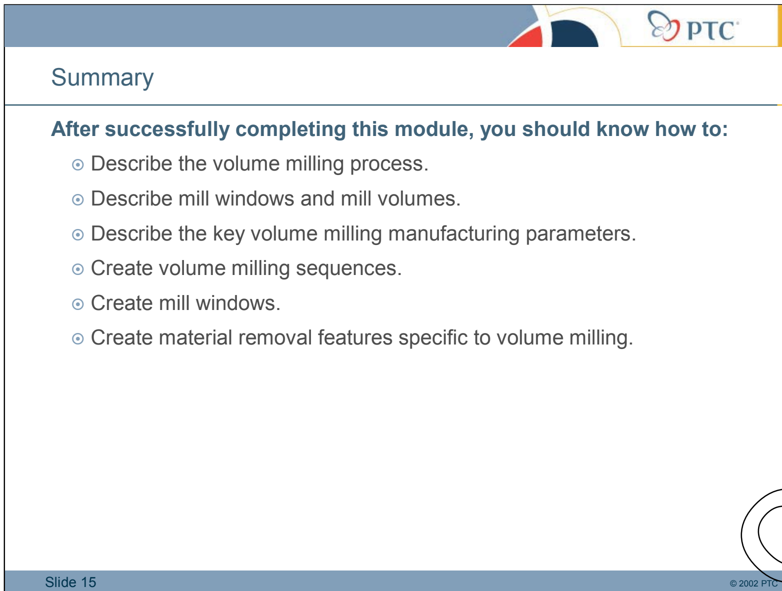
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the volume milling process.
- Describe mill windows and mill volumes.
- Describe the key volume milling manufacturing parameters.
- Create volume milling sequences.
- Create mill windows.
- Create material removal features specific to volume milling.

Module 9 Lab Exercises

Exercise 1: Creating Volume Milling Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create a volume milling sequence to rough out the material from inside the model, you modify key parameters to create an efficient tool path. Volume sequences can be used to rough out material as well as profile walls and face floors. Mill windows or mill volumes can be used to specify the volume of material to remove. You use a mill window to configure the volume to remove.

Objectives

After successfully completing this exercise, you will know how to:

- Create volume-milling sequences.
- Create mill windows.
- Create material removal features specific to volume milling.

Scenario

In this exercise, you continue manufacturing the aluminum cover by creating a volume milling sequence to rough out the material from inside the model. You also create a new mill window feature and reference it for the volume milling sequence.

Parameters

The following are key volume milling parameters.

Table 1: Parameter Descriptions

Parameter	Description
ROUGH_OPTION	Controls if a profiling and/or roughing pass is created.
SCAN_TYPE	The method of scanning the machined area.
CUT_ANGLE	The angle of cuts relative to NC sequence coordinate system.
STEP_OVER	Distance between passes.
NUMBER_PASSES	Number of passes per slice.
TOOL_OVERLAP	Tool step over calculated by overlap distance.
BOTTOM_SCALLOP_HGT	Step over calculated by scallop height.
STEP_OVER_ADJUST	Adjust STEP_OVER to provide complete coverage of machined area.
STEP_DEPTH	The incremental depth of each cut.
WALL_SCALLOP_HGT	Controls the step depth for angled walls.
MIN_STEP_DEPTH	Specifies the minimum allowable distance between slices.
ROUGH_STOCK_ALLOW	Controls stock remaining on walls for roughing passes.
PROF_STOCK_ALLOW	Controls stock remaining on walls for profiling passes.
BOTTOM_STOCK_ALLOW	Controls stock remaining on bottom faces for rough and profile passes.

Step 1. Locate and open the cover manufacturing model.

Before creating the volume milling sequence we need to locate the manufacturing model and open it using the folder navigator. When we have located the folder for this exercise we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module09**, right-click, and select **Make Working Directory.**
2. Open COVER.MFG. Turn off any displayed datum features.

Step 2. Create a volume milling NC sequence.

Using the Menu Manager, we can create a volume milling sequence. Notice how the site parameter file sets all the required manufacturing parameters; this includes reading in the speeds, feeds and depth-of-cut data for this tool. We can select the name and window options as these are the only sequence references we need to change. We can enter a name for the sequence to make it easy to identify. The FEM_25_0 tool is automatically selected because it was used in the previous sequence.

We can create a mill window using the select option. Using the tangent option, we can select any of the inner edges at the top of the pocket.

Notice the other options in the Mill Window dialog box: Plane, Tool Side, Depth, Offset and Inside Loops. The Plane option determines the height of the will window. The Mill Window is created on the Retract Plane by default.

We play the toolpath using the default parameters from the site parameter file. Notice the retracts in the middle of the slices when the toolpath hits a boundary. Notice also the warning messages at the bottom of the graphics window; this is because the site parameter file is setting cutter compensation to on, which is not required. We will correct this in the next step.

1. Using the Menu Manager, create a **Volume** milling sequence.
2. In the Seq Setup menu, select the **Name**, and **Window** check boxes, and complete the selection of options.
3. Name the sequence [**RGH_BOTTOM**].

Note:

The only option highlighted in the SEQ SETUP menu is Volume. This is because all of the other required options (Tool, Parameters, Coordinate System and Retract Plane) are already configured. The FEM_25_0 tool was used in the previous NC sequence, and will be used again by default. The site parameter file configures values for all the required parameters.

A volume sequence can use either a mill window or a mill volume.

4. Select **Create Wind** to create a mill window, name it [**WND_RGH_BOT**].

Note:

You can create manufacturing geometry such as mill windows before creating an NC sequence or during the creation of an NC sequence.

5. Change the view to the named view **OP010-TRI**.
6. Using the **Select** and **Tangent Chain** options, select one of the inner chain of edges at the top of the pocket, as shown in the following figure. Complete the selection of edges.

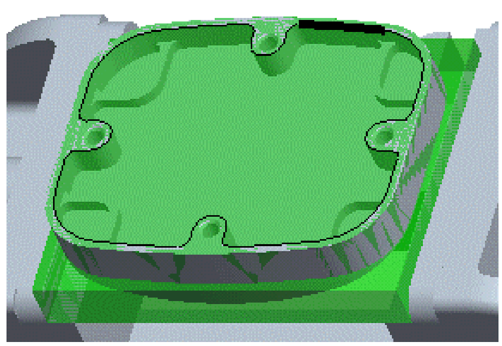


Figure 1: Showing selected edge and highlighted tangent chain of edges.

7. **Preview** the mill window, and complete the mill window configuration.

> **Note:**
> Select preview and notice that the mill window is created on the retract plane. The machining will begin at the top of workpiece. You can specify a different starting level for the machining by changing the plane in the Mill Window dialog box. You can also control the depth of machining and the offsets from the dialog box.

8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

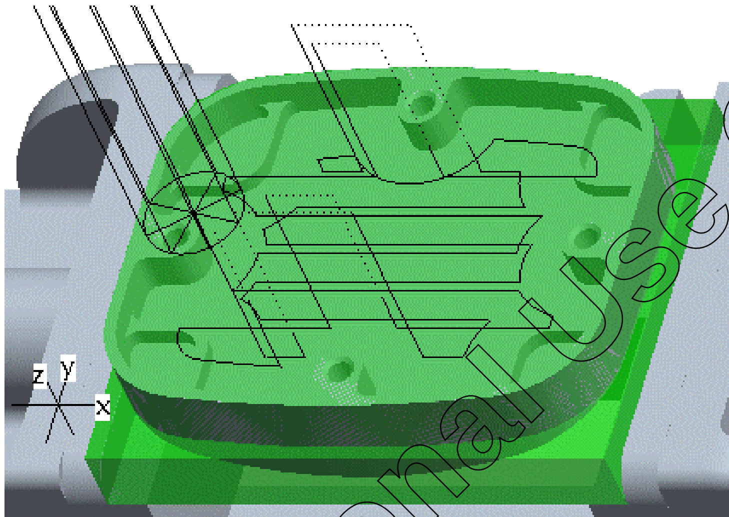


Figure 2: Showing unwanted retract moves.

> **Tool Path Review:**
> Notice the unwanted retract moves, this is controlled by the SCAN_TYPE parameter. To remove the retract moves, you can set the parameter to TYPE_2 (to go around islands) or TYPE_3 (to machine region by region). Notice also the warning messages at the bottom of the graphics window, this is because the site parameter file is turning on cutter compensation which is not required.

Step 3. Modify the sequence parameters to control the scan type, roughing option, and cutter compensation.

We can modify the SCAN_TYPE, CUTCOM and ROUGH_OPTION parameters to see the effect on the toolpath.

We can modify the SCAN_TYPE to TYPE_2 to eliminate retract moves, and set the CUTCOM to OFF to eliminate the warning messages. The TYPE_2 SCAN_TYPE will go around islands instead of over them. We can modify the ROUGH_OPTION to ROUGH_&_PROFILE. We can see that the tool now roughs each slice and then profiles the wall before moving to the next slice.

This can be seen more easily using NC Check. We can switch the NCCHECK_TYPE configuration option to NCCHECK. This will start the internal NCCHECK program instead of

Vericut. Using NCCHECK is generally faster than Vericut, but has fewer checking capabilities. Notice that the walls are now being machined. This is because the PROF_STOCK_ALLOW parameter is set to zero in the site parameter file. Also notice that the bottom surfaces are being machined. This is because the BOTTOM_STOCK_ALLOW parameter uses the PROF_STOCK_ALLOW setting if it is not configured.

1. Set the following parameters in the sequence:

Table 2: Changed Parameters

Manufacturing Parameter	Value	Section
SCAN_TYPE	TYPE_2	CUT OPTION
CUTCOM	OFF	MACHINE
ROUGH_OPTION	ROUGH_&_PROFILE	CUT OPTION

2. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

Figure 3: Showing resulting toolpath.

3. Using the **Tools** drop-down menu, click **Options**. In the Options dialog box, add the configuration option NCCHECK_TYPE, in the Value drop-down list click **NCCHECK**. **Apply** and **Close** the options.

> **Note:**
> Switching the NCCHECK_TYPE configuration option to NCCHECK starts the internal NCCHECK program instead of Vericut. Using NCCHECK is generally faster than Vericut, but has fewer checking capabilities.

4. In the Play Path menu, select **NC Check > Run** to play the toolpath. Close the NC Check menu when finished.

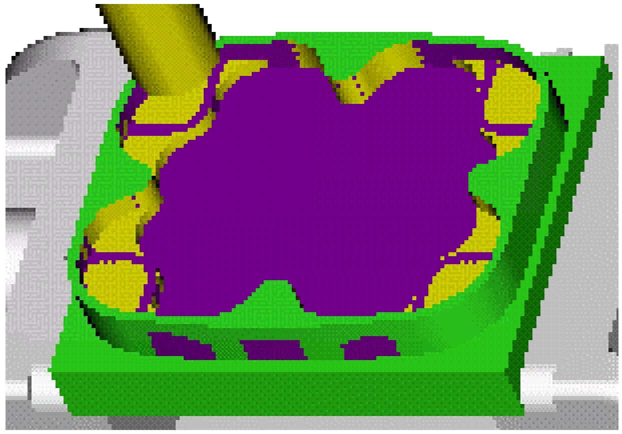


Figure 4: Showing resulting toolpath in NCCHECK.

Tool Path Review:

Setting the SCAN_TYPE to TYPE_2, makes the tool go around islands. Setting the ROUGH_OPTION to ROUGH_&_PROFILE, means the tool roughs each slice and then profiles the walls before going to the next depth. Notice that the walls and bottom faces are now machined with zero stock remaining. This is because PROF_STOCK_ALLOW is set to zero and BOTTOM_STOCK_ALLOW is set to '-'. In this case BOTTOM_STOCK_ALLOW uses the PROF_STOCK_ALLOW value, which is zero.

Step 4. Modify the sequence parameters to control the stock allowance and change the scanning method.

We can set the SCAN_TYPE, to TYPE_SPIRAL. We can modify the PROF_STOCK_ALLOW, ROUGH_STOCK_ALLOW and ROUGH_OPTION parameters to see the effect on the toolpath.

We set the ROUGH_OPTION back to ROUGH_ONLY. We modify the ROUGH_STOCK_ALLOW to 0.5. This leaves 0.5 mm stock on the sidewalls. We can modify the PROF_STOCK_ALLOW parameter to 0.5, which is also used by the BOTTOM_STOCK_ALLOW parameter and leaves 0.5 mm of stock on the bottom faces.

The PROF_STOCK_ALLOW parameter is also used in the calculation for the material removal feature.

We can play the toolpath using NCCHECK and see that stock is now left on the walls and the bottom surfaces.

1. Set the following parameters in the sequence:

Table 3: Changed Parameters

Manufacturing Parameter	Value	Section
SCAN_TYPE	TYPE_SPIRAL	CUT OPTION
ROUGH_OPTION	ROUGH_ONLY	CUT OPTION

Manufacturing Parameter	Value	Section
ROUGH_STOCK_ALLOW	0.5 *(0.02 in.)*	CUT PARAM
PROF_STOCK_ALLOW	0.5 *(0.02 in.)*	CUT PARAM
BOTTOM _STOCK_ALLOW	-	CUT PARAM

2. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

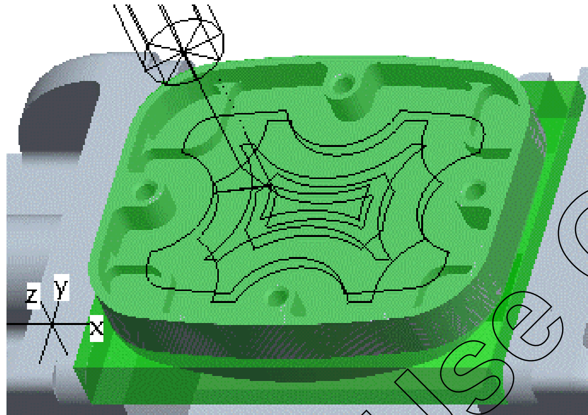


Figure 5: Showing resulting toolpath.

3. Play the toolpath using the **NC Check** option.

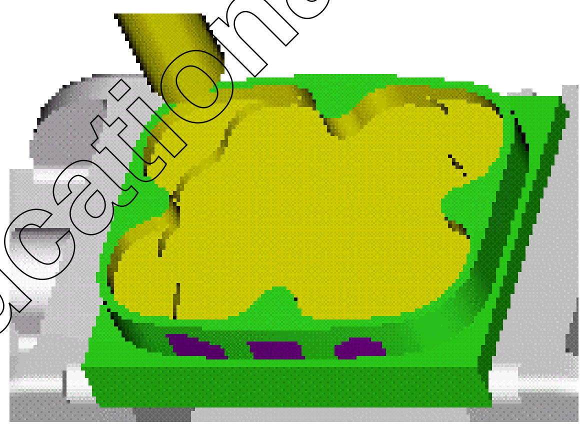


Figure 6: Showing resulting toolpath in NCCHECK.

Tool Path Review:

The ROUGH_STOCK_ALLOW parameter controls the amount of stock remaining on the walls. The BOTTOM_STOCK_ALLOW is set to (-), so it automatically uses the PROF_STOCK_ALLOW value of 0.5 *(0.02 in.)*. This leaves stock remaining on all of the bottom faces. When reviewing the toolpath using NC Check you can see the effect of these changes. Notice also how the scan type is now spiral.

The PROF_STOCK_ALLOW parameter is used in the next step to calculate the stock remaining after creating a material removal feature.

4. Complete the NC sequence.

Step 5. Create a material removal feature.

We can create a material removal feature to cut away the machined volume from the workpiece. We create a Material Removal feature using the ***Automatic*** *option. We select the* ***AutoAdd*** *option to pick the workpiece automatically. The material removal feature takes into account the amount of stock we leave on the surfaces based on the PROF_STOCK_ALLOW parameter. It does not currently take into account the size of the tool.*

1. Create a material removal feature for the RGH_BOTTOM sequence using the **Automatic** option.
2. Use the **AutoAdd** option to select the workpiece to intersect.
3. To see the remaining stock, select the **OP010-TOP** view.
4. Click the No Hidden icon to set the display to no hidden line.
5. Click **View > Display Settings > System Colors**. In the System Colors dialog box, clear the Blended Background check box to turn off the blended background.

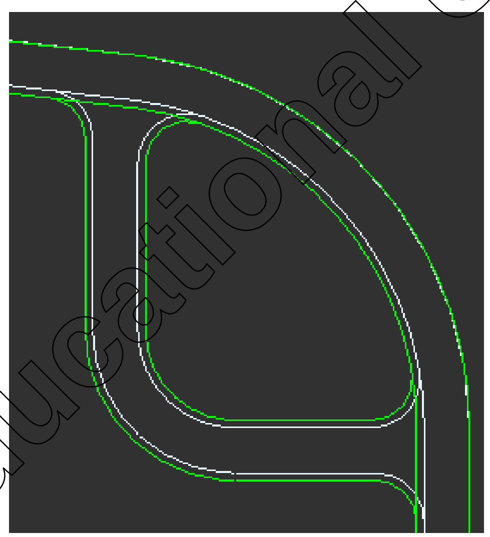


Figure 7: Showing stock remaining on vertical walls.

> **Note:**
> Notice the automatic cut made in the workpiece. The PROF_STOCK_ALLOW parameter was used to leave 0.5 mm *(0.02 in.)* of stock on the side and bottom surfaces.

6. Change the view back to the named view **OP010-TRI**.
7. Select the Blended Background check box to toggle the display of the blended background, then close the System Colors dialog box.
8. Click the Shading icon to set the display to shaded again.

9. Select and **hide** the mill window in the model tree.

10. Using the **Tools** drop-down menu, click **Options**, set the configuration option NCCHECK_TYPE back to **Vericut**, and **Apply** the options.

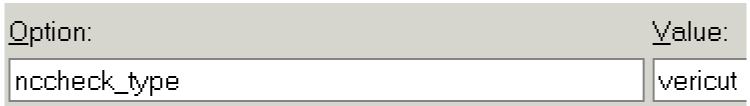


Figure 8: Setting NCCHECK_TYPE to Vericut.

11. Save the manufacturing model.

12. Close all windows and erase all components from memory.

PTC

Summary

After successfully completing this module, you should know how to:

- Describe the volume milling process.
- Describe mill windows and mill volumes.
- Describe the key volume milling manufacturing parameters.
- Create volume milling sequences.
- Create mill windows.
- Create material removal features specific to volume milling.

Module

10

Creating Local Milling Sequences

Introduction

Local milling sequences are typically used to remove material still remaining from a previous NC sequence. This is possible by using a smaller tool, and calculating the remaining material to be removed. There are four types of local milling sequences, and since each type of sequence can be used in different situations, it is important to know when and how to use each type of sequence.

Objectives

After completing this module, you will be able to:

- Describe the different types of local milling sequences.
- Describe the key local milling manufacturing parameters.
- Create local milling sequences.
- Create material removal features specific to local milling.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

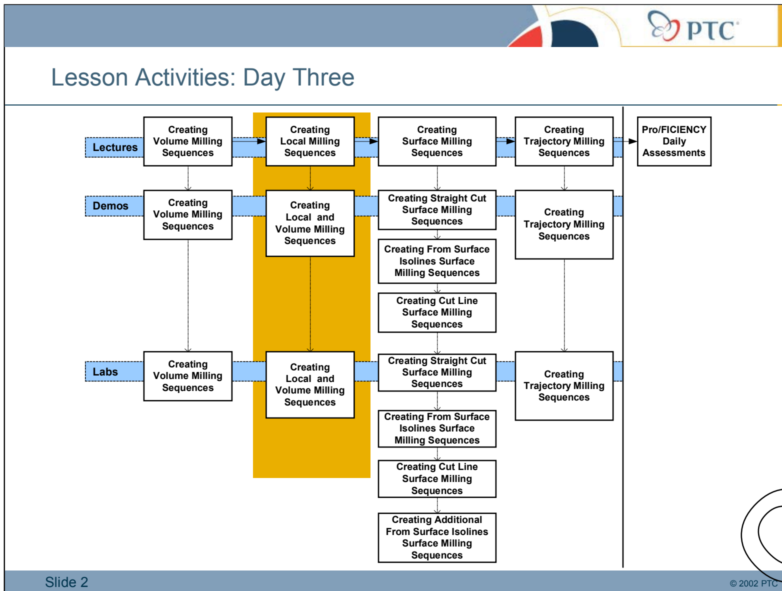
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

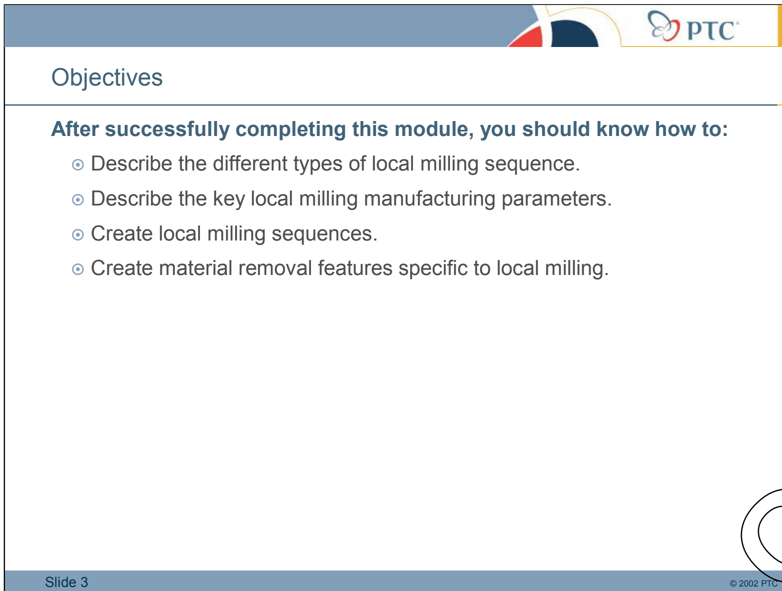
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 25 mins
- Labs : 50 mins
- Total: 1 hr 30 mins

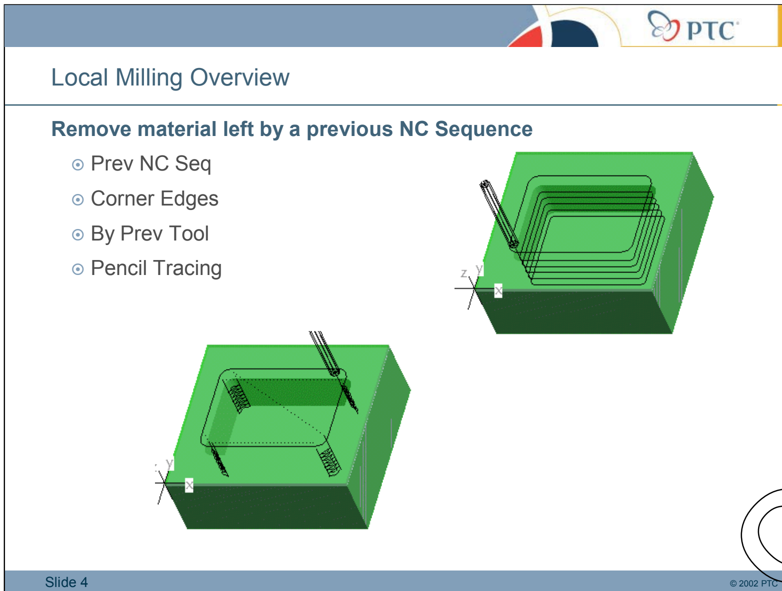


Overview

Local milling sequences are typically used to remove material still remaining from a previous NC sequence. This is possible by using smaller tools, and calculating the remaining material to be removed. There are four types of local milling sequences, and since each type of sequence can be used in different situations, it is important to know when and how to use each type of sequence.

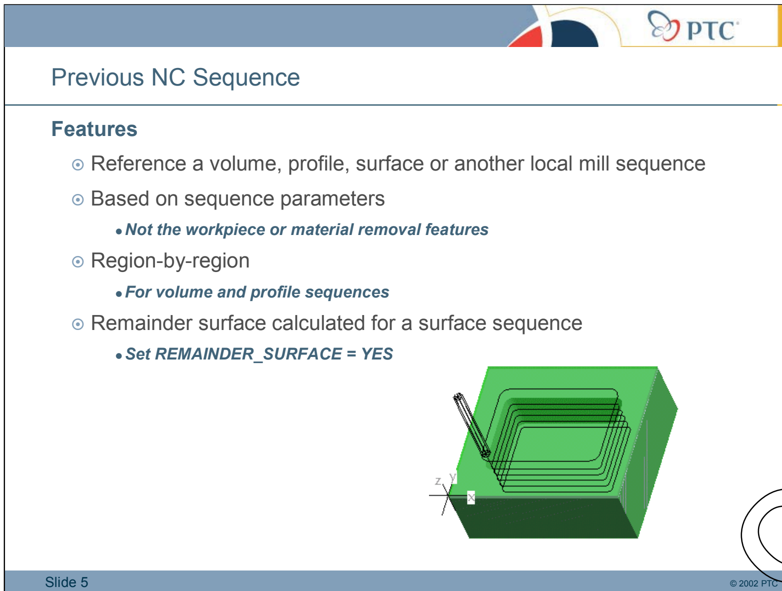
After completing this module, you will be able to:

- Describe the different types of local milling sequence.
- Describe the key local milling manufacturing parameters.
- Create local milling sequences.
- Create material removal features specific to local milling.



There are 4 types of Local mill sequences.

- Prev NC Seq: remove material left after a Volume, Profile, or another Local milling NC sequence, usually with a smaller tool. It does not use the workpiece or material removal features to determine the stock to remove, instead it uses the previous sequence parameters.
- Corner Edges: specify one or more corners to clean up by selecting edges.
- By Prev Tool: calculates the remainder material on specified surfaces after being machined by a larger tool; then removes this material by the current (smaller) tool.
- Pencil Tracing: cleans up edges of selected surfaces by creating a single-pass tool path along the corners.



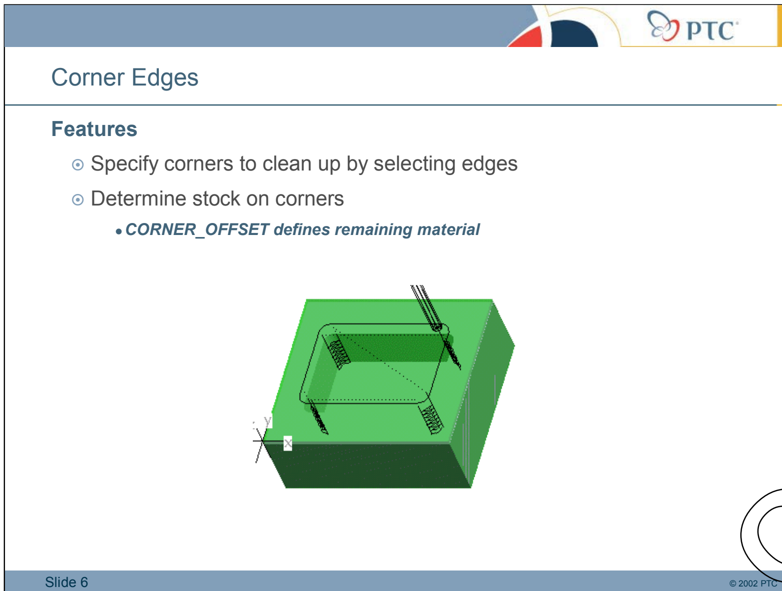
Previous NC Sequence

The previous sequence option of local milling allows you to reference the material left by a previously created volume, profile, surface or another local mill sequence.

The calculated material remaining from the previous sequence is based on the previous sequence parameters and not the mill volume or material removal features.

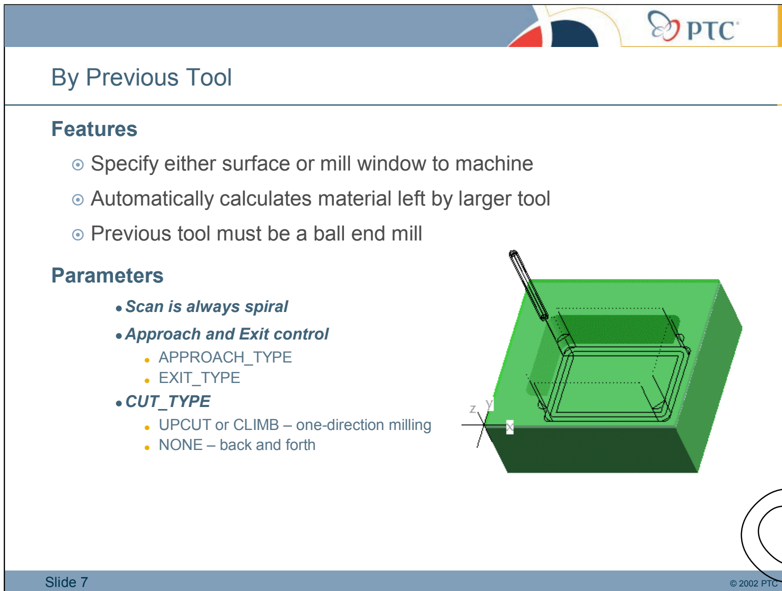
If you are using a volume or profile milling sequence as a reference, you can control the toolpath region by region.

If you are using a surface milling sequence as a reference, the system will calculate the remainder surface from the sequence. If you set the REMAINDER_SURFACE parameter to YES in the surface NC sequence, the remainder surface will be calculated as part of that sequence and it will save you calculation time in the local milling sequence.



Corner Edges

The corner edges option of local milling allows you to clean up corners by specifying which corners to machine. You specify the corners by selecting the vertical edge of the corner. The amount of material remaining from the previous tool is determined by the CORNER_OFFSET parameter.



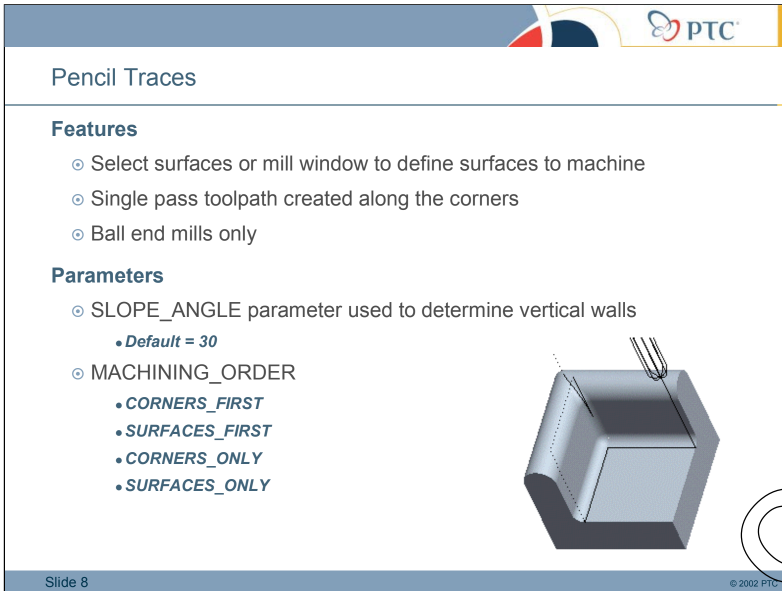
By Previous Tool

The by previous tool option of the local mill sequence allows you to finish machining surfaces based on the material left by a previous tool. You can specify the surfaces to machine by either selecting a group of surfaces or by using a mill window. If you use a mill window, all of the surfaces within the mill window are selected.

The system automatically calculates the material left by the previous tool. The previous tool must be a ball end mill.

Parameters to control the behavior of the toolpath:

- The toolpath will always be spiral.
- You can specify the approach and exit types.
- The CUT_TYPE can be specified as UPCUT or CLIMB, in which case the machine will only machine in one direction to maintain the cut type.
  - If you choose none for the CUT_TYPE, the system will scan back and forth to machine the surfaces without regard to the cutting type.

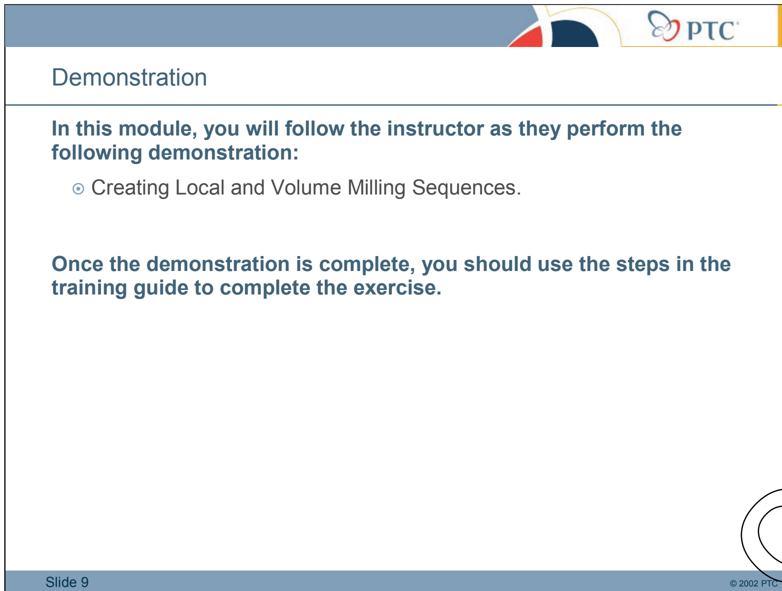


Pencil Traces

The pencil traces option of the local mill sequence lets you create a single pass toolpath along the corners of a model. You specify the surfaces to check for corners by either selecting the surfaces explicitly or by using a mill window to select the surfaces. The pencil trace only works with ball end mills.

There are two parameters to control the toolpath.

- The SLOPE_ANGLE parameter determines the angle at which a wall is considered vertical.
- The MACHINING_ORDER parameter determines which order to machine the corners and surfaces.
  - CORNERS_FIRST: vertical regions
  - SURFACES_FIRST: horizontal regions
  - CORNERS_ONLY: vertical regions
  - SURFACES_ONLY: horizontal regions



Demonstration: Creating a Local and Volume Milling Sequences

Scenario

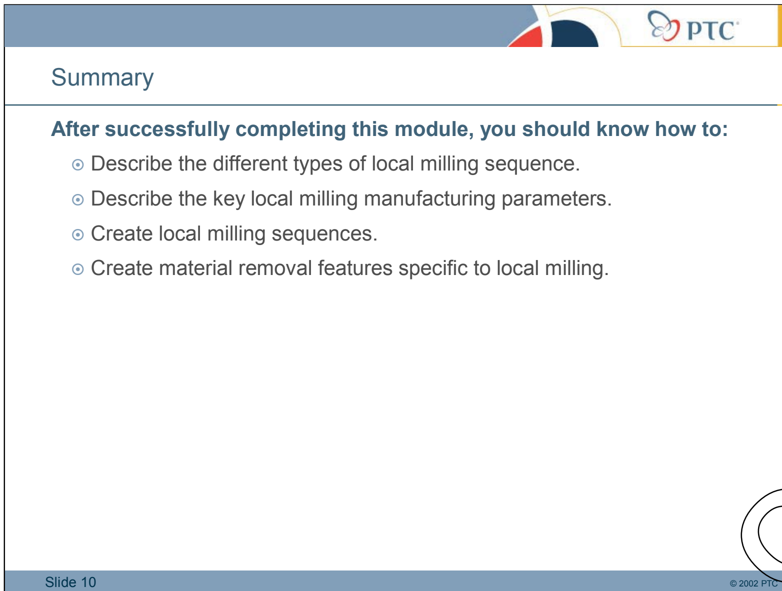
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the different types of local milling sequence.
- Describe the key local milling manufacturing parameters.
- Create local milling sequences.
- Create material removal features specific to local milling.

Module 10 Lab Exercises

Exercise 1: Creating Local and Volume Milling Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create a local milling sequence to remove the remaining material from the inside of the cover. You then modify parameters to make the tool path efficient. Local milling sequences are useful because they can automatically calculate the tool path required to remove remaining material based on previous sequence information and model geometry.

You also create a second fixture setup and operation to machine the opposite side of the cover. You continue the exercise by creating a volume milling sequence to remove most of the material from the top of the cover. You complete the exercise by creating a second volume milling sequence to machine the remaining material from the vertical walls on the outside of the cover.

Objectives

After successfully completing this exercise, you will know how to:

- Create local milling sequences.
- Create material removal features specific to local milling.
- Create volume milling sequences.

Scenario

You are now ready to continue machining the aluminum cover. You create a local milling sequence to remove the remaining material from the inside of the cover. You also create a second fixture setup and operation to machine the opposite side of the cover. You continue the exercise by creating a volume milling sequence to remove most of the material from the top of the cover. You complete the exercise by creating a second volume milling sequence to machine the remaining material from the vertical walls on the outside of the cover.

Step 1. Locate and open the cover manufacturing model.

Before creating the local milling sequence, we first need to locate the manufacturing model and open it using the folder navigator. Once we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module10**, right-click, and select **Make Working Directory.**
2. Open COVER.MFG. Turn off any displayed datum features.

Step 2. Create a local milling NC Sequence.

Using the Menu Manager, we can create a local milling sequence using the previous NC sequence option and reference the previous volume milling sequence. We can select the name, tool, and parameter options. We can enter a name for the sequence to make it easy to identify, and change the tool to a smaller diameter end mill. When setting the manufacturing parameters we can start by using the previous volume milling parameters, which need to be adjusted. So we can then read in the tool cutting data parameters to update the feed and speed, and depth-of-cut parameter values. We can also modify the STEP_DEPTH, STEP_OVER and the PROF_STOCK_ALLOW parameters and play the toolpath. Notice how the tool removes the remaining material on the walls and all the horizontal surfaces.

1. Using the Menu Manager, create a **Local** milling sequence using the **Prev NC Seq** option.
2. When prompted for a reference milling sequence, select **NC Sequence > RGH_BOTTOM > CUT MTN #1**.
3. Using the Menu Manager, select **Seq Setup**, then select the **Name, Tool** and **Parameters** check boxes, complete the selection of options.
4. Name the sequence [**FIN_BOTTOM**].
5. Change the tool to the **FEM_06_0** tool.
6. For the parameters, select the **Use Prev** option and use the **RGH_BOTTOM** parameters.
7. In the Mfg Params menu, select **Set** to modify the parameters.
8. In the Parameter Tree window, click **Edit > Copy From Tool > All > Roughing** to read in the roughing cutting data for the current tool.

Note:

Reading in the cutting data adjusts the CUT_FEED, STEP_DEPTH, STEP_OVER, and SPINDLE_SPEED parameters, these adjustments are based on workpiece material and tool parameters.

9. Modify the following parameters:

Table 1: Changed Parameters

Manufacturing Parameter	Value	Section
STEP_DEPTH	6 *(0.24 in.)*	CUT PARAM
STEP_OVER	3 *(0.12 in.)*	CUT PARAM
PROF_STOCK_ALLOW	0	CUT PARAM

10. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

11. Using **NC Check** start Vericut, then click on the Play to End icon to view the machining simulation.

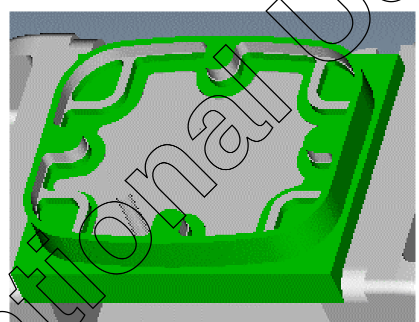


Figure 1: Showing the resulting toolpath in Vericut.

Tool Path Review:

The tool removes the remaining material on the walls and all the horizontal surfaces.

12. Close Vericut and complete the sequence.

Step 3. Create a material removal feature.

*We can create a material removal feature to cut away the machined volume from the workpiece. We create a material removal feature using the **Automatic** option. We select the **AutoAdd** option to pick the workpiece automatically.*

1. Create a Material Removal Feature for the FIN_BOTTOM sequence using the **Automatic** option.

2. Use the **AutoAdd** option to select the workpiece to intersect.

Note:
Notice the automatic cut made in the workpiece.

3. Save the manufacturing model.

Step 4. Create the second operation to machine the top of the cover.

We can create a second operation to machine the top of the cover. We create a new operation named OP020. We can create the machine zero position, ensuring that we create the co-ordinate system in the COVER.ASM assembly by selecting this assembly in the model tree. Selecting three surfaces on the workpiece, we can position the co-ordinate system in the front-left-top corner of the workpiece. We can set the direction of the axes to match our requirements, and rename the co-ordinate system to make it easier to identify. We can also create a retract surface and position it relative to our machine zero position. When the operation definition is complete we can rename the retract surface in the model tree to make it easier to identify

At this point, we can again specify the workpiece material for the operation, in this case aluminum.

We can open the Fixture Dialog box and name the fixture setup op_020. We can now add a component to the fixture setup; in this case we can select the VISE.ASM from the list. To ensure the vise assembly jaws are the correct distance apart, we select the OP020_INST from the list of instances. We can position the vise assembly using two mate constraints and an align constraint. Once assembled, we can close the Fixture Setup and Operation dialog boxes, and rename the retract plane in the model tree.

We finish the operation setup by creating three named views, as we did for operation 010. We can create a trimetric view, a top view and a front view.

1. Turn off the display of datum planes and datum axes. Turn the datum coordinate systems on if they are not already displayed.

Note:
Turn off the display of the blended background in the graphics window to make viewing of co-ordinate system axes easier.

2. Click **View > Display Settings > System Colors**, clear the Blended Background check box, and click **OK** to complete the configuration.

3. Using the Menu Manager, open the Operation Setup dialog box, click the Create New Operation icon and create a new operation named [**OP020**].

4. Switch the view to the **default** view and zoom into the model.

5. Create a machine zero co-ordinate system positioned at the front-top-left corner of the workpiece. In the Operation Setup dialog box, click the Machine Zero icon, then select **Create** in the MACH CSYS menu, when prompted to pick a model to create the coordinate system in, select the COVER.ASM assembly in the model tree.

6. To refresh the screen click **View > Repaint**.

> **Note:**
> To select multiple surfaces you must press and hold the CTRL key when selecting surfaces. To select hidden surfaces you must move the cursor over the hidden surface and right-click to highlight the hidden surface.

7. Select surfaces to configure the co-ordinate system origin. Press and hold CTRL, and select the front and top surfaces on the workpiece as shown in the following figure.

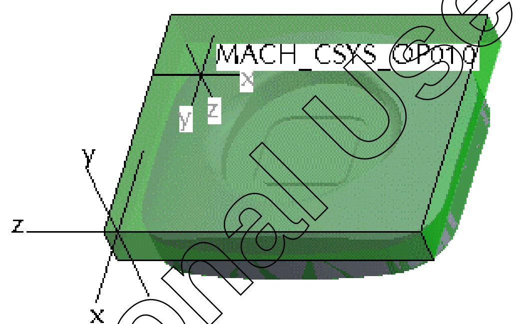


Figure 2: Showing selected front and top surfaces.

8. To select the hidden left surface move the cursor over the model as shown in the following figure, then press CTRL + right-click, when the hidden left surface highlights select it.

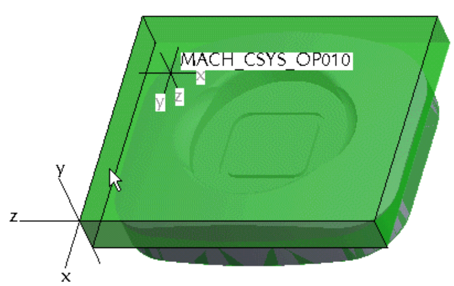


Figure 3: Showing selected left surface and cursor position.

> **Note:**
> The co-ordinate system origin will move to the front-top-left corner of the workpiece when all three surfaces have been selected correctly..

9. In the Coordinate System dialog box, select the Orientation tab. Use the drop-down axis options and the Flip button to modify the orientation of the axes as shown in the following figure.

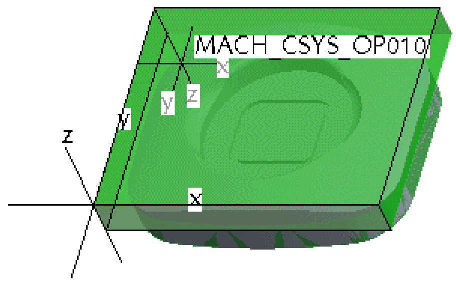


Figure 4: Showing re-oriented X, Y and Z axes.

10. In the Coordinate System dialog box, select the Properties tab, and rename the co-ordinate system to [**`mach_csys_op020`**]. Close the Coordinate System dialog box when finished.

Figure 5: Showing renamed co-ordinate system

11. In the Operation Setup dialog box, click the Retract Surface icon, select the **Along Z Axis** option and type [**`40`**] *(1.57 in.)* as the z distance. Close the Retract Selection dialog box.

12. Turn on the blended background. Click **View > Display Settings > System Colors**, select the Blended Background check box, and click **OK** to complete the configuration.

13. Select **aluminum** in the Stock Material drop-down list.

14. Turn off the display of datum planes, datum axes and datum co-ordinate systems.

15. Click the Fixture Setup icon to open the Fixture Set Up dialog box, type [**`op_020`**] in the Fixture Setup Name box.

16. Click the Add Component icon to add a component to the fixture setup, select **VISE.ASM** from the list of models, then select the **op020_inst** from the list of instances.

> **Tip:**
> The assembly process is easier if you do not display the vise assembly in the main graphics window when specifying assembly constraints; instead show the vise assembly in a separate window.

17. In the Component Placement dialog box, click the Show Component in a Separate Window icon, then remove the vise assembly from displaying in the graphics window by de-selecting the Show Component in Assembly window icon. Ensure both the manufacturing model and the vise assembly are clearly visible.

18. Assemble the vise assembly, as shown in the following figure. In the Component Placement dialog box, change the first constraint type from Automatic to **Mate**, then select the right surface on the workpiece and the small vertical surface on the clamp pad part.

19. Change the second constraint type from Automatic to **Mate**, then select the hidden bottom surface on the workpiece and the small horizontal surface on the clamp pad part.

20. Change the third constraint type from Automatic to **Align**, then select the front surface on the workpiece and the vertical front surface on the clamp pad part.

> **Tip:**
> You can use the spin, zoom, and pan commands on the vise assembly in the sub-window to make selection of surfaces easier.

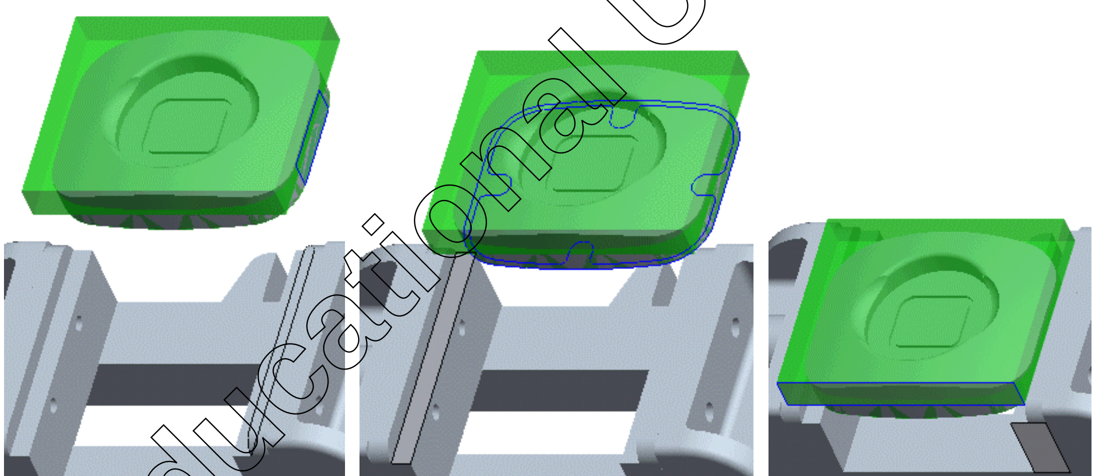


Figure 6: Showing the two mate and one align constraints; selected pairs of surfaces are highlighted.

21. When the three constraints have been created click the **Preview** option in the Component Placement dialog box to check the vise has been positioned correctly. Complete the component placement when satisfied.

22. Close the Fixture Setup dialog box.

23. Close the Operation dialog box.

24. Rename the retract surface from ADTM1 to [`retract_op020`] in the model tree.

25. Click **View > Orientation > Reorient**, and create a series of named views OP020-TRI, OP020-TOP, and OP020-FRONT as you did for operation 010.

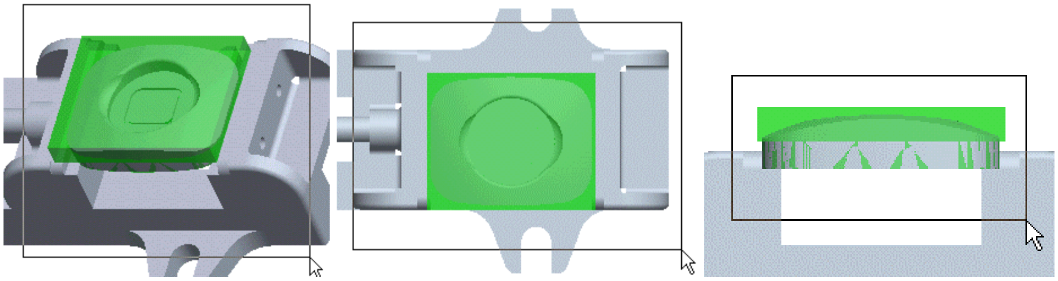


Figure 7: Showing named views.

Step 5. Create a mill window as a reference for the volume milling sequence.

We need to create a mill window to specify the volume we want to machine; during the configuration we select a number of optional elements to provide us with additional control over the machined volume.

We create a mill window and name it. Using the select and surf chain options we select the top of the workpiece. In the dialog box, we change the Tool Side option to Past, and change the Depth option and select the bottom surface of the pocket. We can also change the Offset option to 20. These changes provide more space for the tool to machine around the outer edges of the workpiece.

1. Using the Menu Manager, select **Mfg Setup > Mfg Geometry > Mill Window > Create Wind**, and name the mill window [**WND_RGH_TOP**].
2. When prompted to select a reference co-ordinate system select the MACH_CSYS_OP020 co-ordinate system from the model tree.
3. When prompted to select the window plane, select the RETRACT_OP020 datum plane from the model tree.
4. Change the view to the named view **OP020-TRI**.
5. Using the **Select** and **Surf Chain** options select the top surface of the workpiece, as shown in the following figure. Use the **Select All** option, and complete the selection.

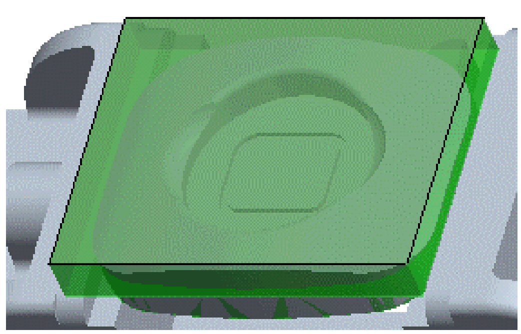


Figure 8: Showing selected top surface of the workpiece.

6. Click the Tool Side element in the dialog box, and using **Define** change the tool side to **Past**.

7. Click the Depth element in the dialog box, and using **Define** select the bottom surface of the pocket (not the bottom of the groove) as shown in the following figure.

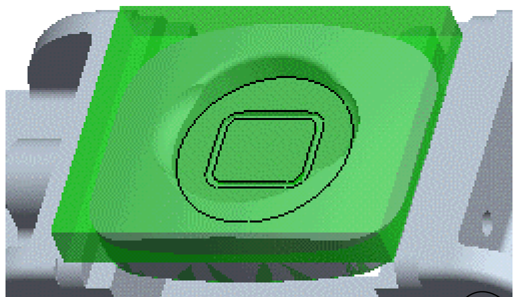


Figure 9: Depth surface for the Mill Window

8. **Preview** the mill window, and complete the mill window configuration.

9. Select the mill window in the model tree, right-click and select **Hide** from the drop-down list.

> **Note:**
> Setting the offset to 20.0 *(0.79 in.)* makes the mill window larger, and together with setting the tool side to past provides more space for the tool to machine around the outer edges of the workpiece.

Step 6. Create a mill volume as an alternative reference for the volume milling sequence.

We can create a mill volume as an alternative reference to specify the volume we want to machine; during the configuration we select a number of optional elements to provide us with additional control over the machined volume.

We create a mill volume and name it. Using extrude, solid, and one-side options, we select the top of the workpiece as the sketching plane; in sketcher we can select the edges of the workpiece to configure the volume boundary. When the sketch is complete we can specify the depth of the volume using the up to surface option and select the surface at the bottom of the pocket.

The mill volume has been extruded down to the bottom of the pocket in the reference model, to create the required volume we need to subtract the reference model geometry from the mill volume. We select the trim option and subtract the reference model geometry from the mill volume.

The side walls of the volume need to be off-set to provide enough room for the tool to machine around the sides of the reference model. We can select the four sidewalls and offset them by 20. These changes provide more space for the tool to machine around the outer edges of the workpiece.

We have now created a mill window and a mill volume which we can use as alternative references when volume milling the top of the workpiece.

1. In the Mfg Geometry menu select **Mill Volume > Create**, and name it [**VOL_RGH_TOP**].
2. Select Sketch, and accept the **Extrude** and **Solid**, and **One Side** options.
3. When prompted select the top surface of the workpiece as the sketching plane, and accept the arrow pointing down into the workpiece as the direction for creating the feature, see the figure below.

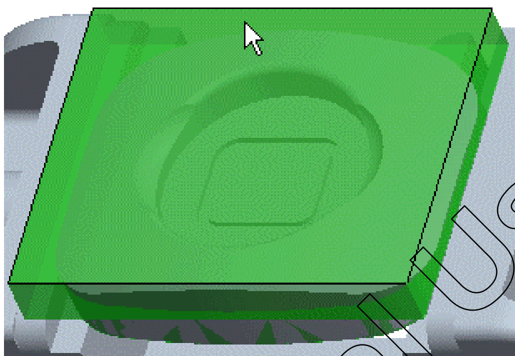


Figure 10: Showing selected top surface of the workpiece.

4. When prompted to select a sketching reference, select the front surface of the workpiece as the bottom sketch view reference, as shown in the following figure.

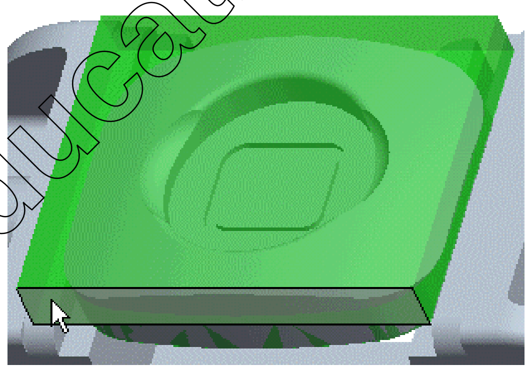


Figure 11: Showing selected front surface of the workpiece.

5. **Close** the references dialog box and accept that not enough references have been specified.
6. Change the view to the named view **OP020-TRI**.
7. Click the Create an Entity from an Edge icon and using the Loop option, select the top surface of the workpiece model as shown in the following figure.

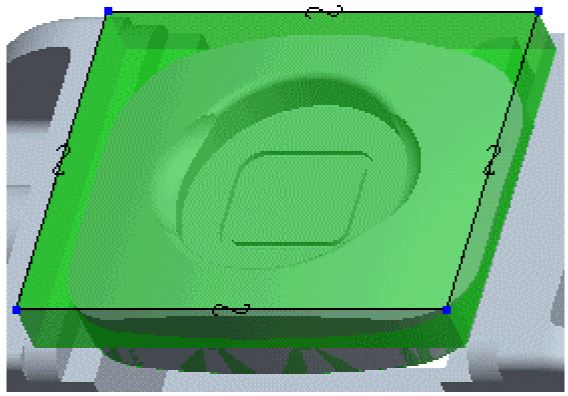


Figure 12: Showing selected loop surface.

8. Complete the sketch.

9. Use the **Up to Surface** depth option, and select the bottom surface of the pocket (not the bottom of the groove) as shown in the following figure.

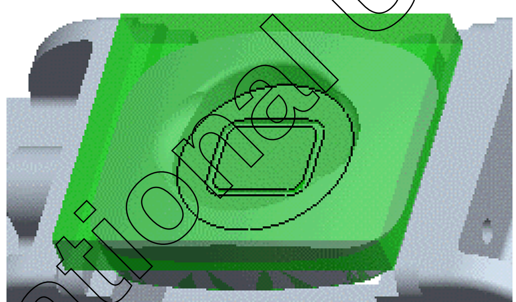


Figure 13: Depth surface for the Mill Volume.

10. Complete the volume by clicking **OK** in the Protrusion dialog box.

> **Note:**
> The mill volume has been extruded down to the bottom of the pocket in the reference model, to create the required volume we need to subtract the reference model geometry from the mill volume.

11. To subtract the reference model geometry from the mill volume, select **Trim** in the Create Vol menu.

12. When prompted to select a part for trimming, move the cursor over the reference model, right-click until the reference model highlights, then select it as the trimming part, as shown in the following figure.

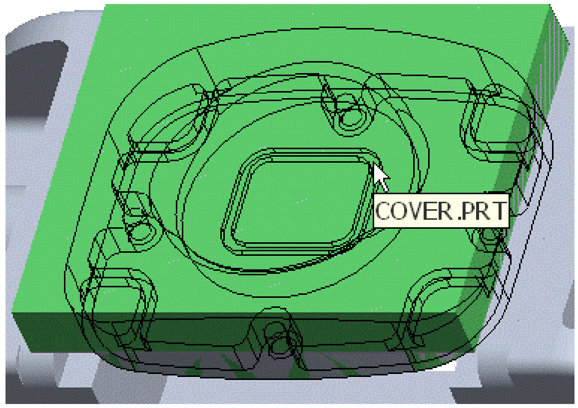


Figure 14: Showing reference part selected.

> **Note:**
> The side walls of the volume need to be off-set to provide enough room for the tool to machine around the sides of the reference model.

13. Select **Offset** in the Create Vol menu, then select the **Surfaces** and **Horizontal** options.

14. When prompted select the four vertical sidewalls of the volume shown in the following figure, and enter an offset distance of [**20**].

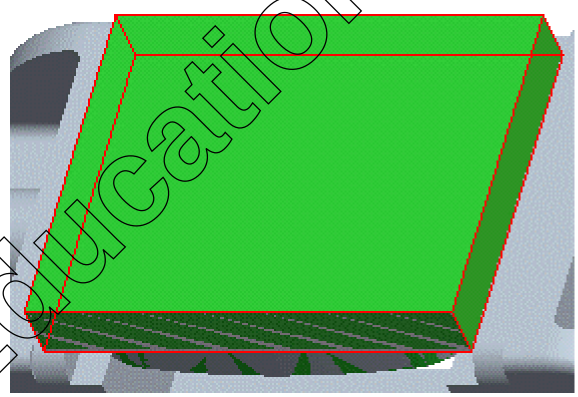


Figure 15: Showing selected four vertical sidewalls.

15. Complete the volume and return to the Manufacture menu.

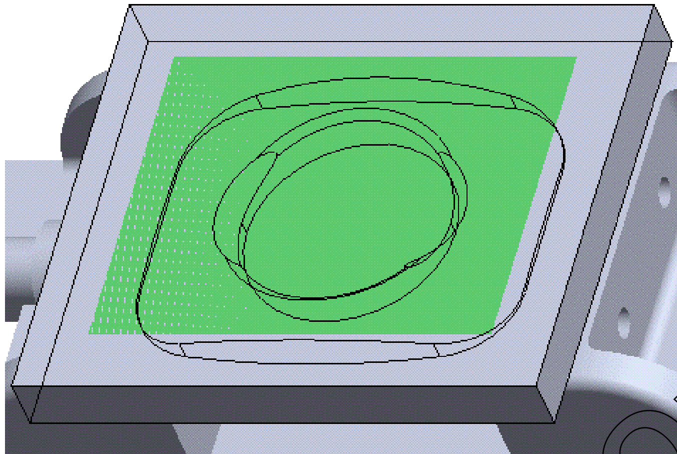


Figure 16: Showing resulting offset mill volume.

> **Note:**
> We have now created a mill window and a mill volume which we can use as alternative references when volume milling the top of the workpiece.

Step 7. Create a volume milling sequence to machine the top of the cover.

We can create a volume milling sequence for the top of the cover using a mill window. Using the menu manager, we can create a volume milling sequence. We can select the Name, and Approach Walls options. We can enter a name for the sequence to make it easy to identify, and change the tool to the 25 mm diameter end mill. When setting the manufacturing parameters we can start by using the previous volume milling parameters from operation 010, but these parameters need to be adjusted.

We edit the parameters and set TRIM_TO_WORKPIECE to YES and STEP_DEPTH to 5. This provides us with a reasonable step depth and ensures the tool does not machine outside the workpiece unless specified.

We can select the VOL_RGH_TOP mill volume to specify the volume to be machined.

We can now select the left and right edges of the mill window as Approach Walls. This enables the tool to plunge outside of the approach walls and approach safely through the side.

We play the toolpath to review the result.

As an alternative we can use the mill window to configure the volume of material to be machined, and review the results. We can select the mill window as an alternative reference, we need to use new approach walls by selecting the left and right edges of the mill window.

1. Using the Menu Manager, create a **Volume** milling sequence.
2. Select the **Name**, and **Appr Walls** check boxes, and complete the selection of options.

> **Note:**
> Notice when creating volume milling sequences, volume is selected by default.

3. Name the sequence [**RGH_TOP**].

4. Change the tool to the **FEM_25_0** tool.

5. For the parameters, select the **Use Prev** option and use the **RGH_BOTTOM** parameters.

6. Modify the following parameters:

Table 2: Changed Parameters

Manufacturing Parameter	Value	Section
STEP_DEPTH	*5 (0.20 in.)*	CUT PARAM
TRIM_TO_WORKPIECE	YES	CUT OPTION

7. Select **Select Vol,** in the Search Tool dialog box select the volume named VOL_RGH_TOP, and click **OK**, to complete the selection.

8. When prompted to define approach and exit walls belonging to VOL_RGH_TOP, select the left and right sides of the mill volume, as shown in the following figure.

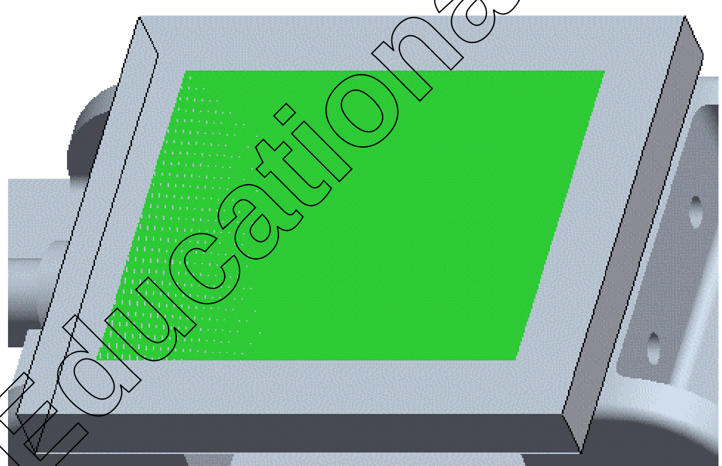


Figure 17: Mill Volume Approach Walls

> **Tips & Techniques:**
> When you specify approach walls with a mill volume, you select the surfaces to approach through. The toolpath will attempt to plunge outside of the material and enter through the approach walls.

9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

10. In the Play Path menu, select **NC Check** to start Vericut.

11. Click on the Play to End icon to view the machining simulation.

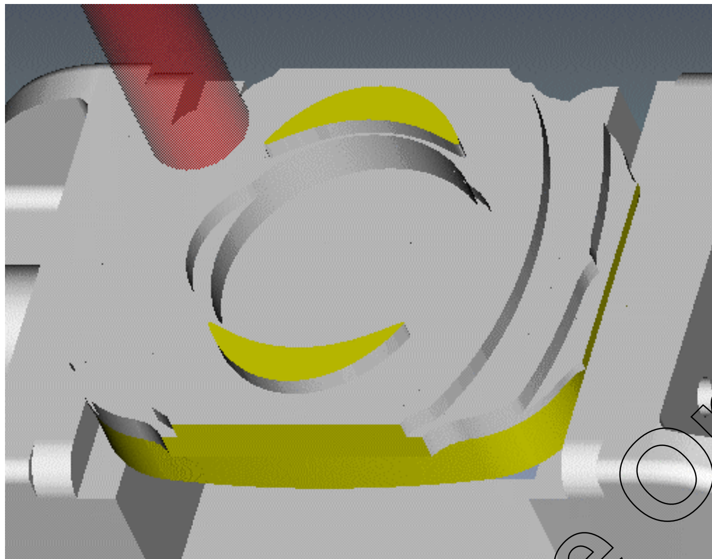


Figure 18: Showing resulting toolpath in Vericut.

Tool Path Review:

The tool creates a roughing toolpath, leaving 0.5 mm *(0.02 in.)* of stock on the vertical and flat surfaces. By using the approach walls, the tool plunges outside of the tool material.

12. Close Vericut.

Note:

As an alternative we can use the mill window to configure the volume of material to be machined, and review the results.

13. Select the mill volume in the model tree, right-click and select **Hide** from the drop-down list.
14. Select the mill window in the model tree, right-click and select **Unhide** from the drop-down list.
15. Using the Menu Manager, select **Seq Setup**, and select the **Window**, and **Appr Walls** check boxes.
16. Use the **Select Wind** option, then in the Search Tool dialog box click the **Find Now** button, and select the window named VOL_RGH_TOP, and click **OK**.
17. Accept the removal of the original approach walls, and then configure new approach walls by selecting the left and right edges of the mill window, as shown in the following figure.

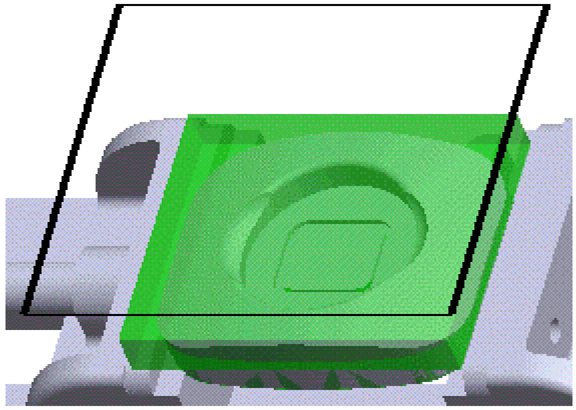


Figure 19: Mill Window Approach Walls

18. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

19. In the Play Path menu, select **NC Check** to start Vericut.

20. Click on the Play to End icon to view the machining simulation.

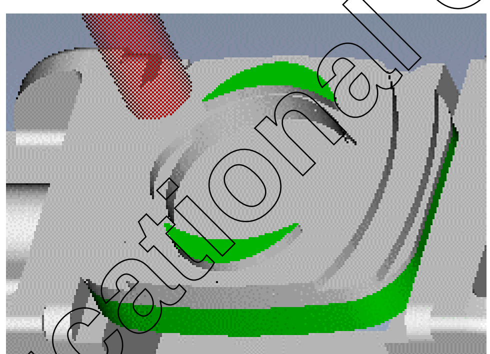


Figure 20: Showing resulting toolpath in Vericut

21. Close Vericut, complete the sequence, and return to the manufacture menu.

Step 8. Create a material removal feature.

*We can create a material removal feature to cut away the machined volume from the workpiece. We create a material removal feature using the **Automatic** option. We select the **AutoAdd** option to pick the workpiece automatically.*

1. Create a material removal feature for the sequence using the **Automatic** option.
2. Use the **AutoAdd** option to select the workpiece to intersect.
3. Save the manufacturing model.

Step 9. Create a volume milling sequence to machine the remaining material on the vertical walls on the top of the cover.

We can now create a volume milling sequence to machine the remaining material on the vertical walls of the top of the cover. Using the menu manager, we can create a volume milling sequence. We can select the Name, Parameters, and Window options. We can enter a name for the sequence to make it easy to identify. The parameters have been read from the site parameter file, we can adjust the parameters to machine only the vertical walls with zero stock allowance and provide a lead-in and lead-out motion. We can set the ROUGH_OPTION to PROF_ONLY, the ROUGH_STOCK_ALLOW to 0. We can also set the TANGENT_LEAD_STEP and NORMAL_LEAD_STEP to 10, and the OVERTRAVEL_DISTANCE to 5. We can reference the same mill window from the previous sequence.

We can review the toolpath and see that during the lead-in and lead-out moves the cutter compensation is applied. Also the over travel distance on each pass removes any witness lines from the machined surfaces.

1. Using the Menu Manager, create a **Volume** milling sequence.
2. Select the **Name**, **Parameters**, and **Window** check boxes.
3. Name the sequence [**PROF_TOP**].
4. Modify the following parameters:

Table 3: Changed Parameters

Manufacturing Parameter	Value	Section
ROUGH_OPTION	PROF_ONLY	CUT OPTION
ROUGH_STOCK_ALLOW	0	CUT PARAM
TANGENT_LEAD_STEP	10 *(0.39 in.)*	ENTRY/EXIT
NORMAL_LEAD_STEP	10 *(0.39 in.)*	ENTRY/EXIT
OVERTRAVEL_DISTANCE	5 *(0.20 in.)*	ENTRY/EXIT

5. Using the **Select Wind** option, click the **Find Now** button in the Search Tool dialog box and select **WND_RGH_TOP** from the list of mill windows.
6. Change the view to the named view **OP020-TRI**.
7. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

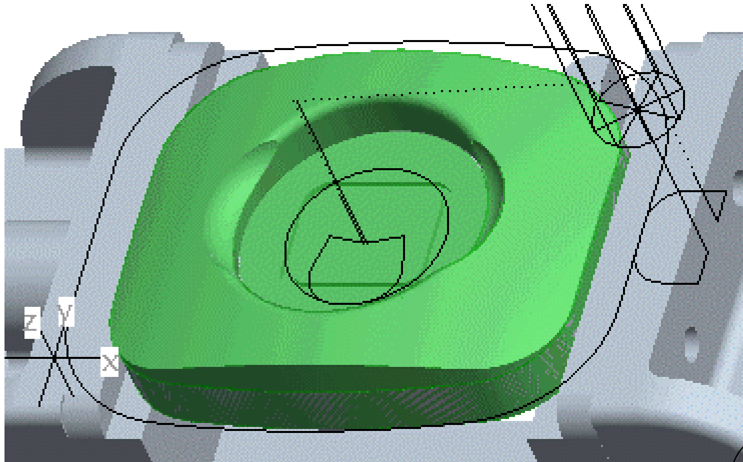


Figure 21: Showing the resulting toolpath.

Tool Path Review:

Notice during the lead-in and lead-out moves the cutter compensation is applied. The over travel distance on each pass removes any witness lines from the machined surfaces.

8. In the Play Path menu, select **NC Check** to start Vericut.
9. Click on the Play to End icon to view the machining simulation.
10. Close Vericut and complete the sequence.
11. Select and **Hide** the WND_RGH_TOP mill window in the model tree.
12. Save the manufacturing model.
13. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the different types of local milling sequences.
- Describe the key local milling manufacturing parameters.
- Create local milling sequences.
- Create material removal features specific to local milling.

Module

11

Creating Surface Milling Sequences

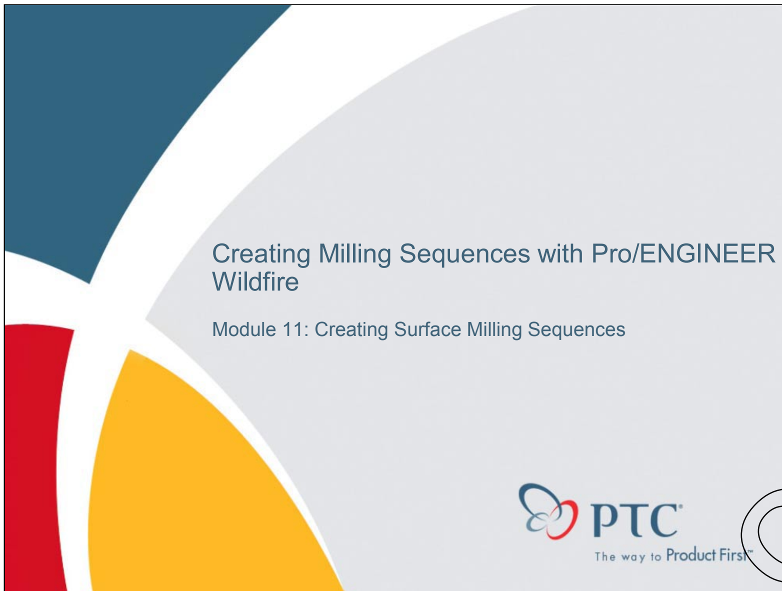
Introduction

Surface milling sequences enable you to mill contoured and horizontal surfaces. This is useful for finish machining contoured surfaces of components, or mold cavities. When creating surface milling sequences, mill surfaces can be used as alternate references when model surfaces are inappropriate, for example when the model being machined is made up of many surfaces, or if you need to extend the toolpath beyond the model. There are four types of surface milling sequence, providing you with a flexible approach to surface machining. It is an important skill to understand the differences between each type of surface milling sequence, and when to use each type of sequence. Once you have completed your NC sequence, you can represent the as-machined state of your workpiece by creating a material removal feature that removes the machined volume from the workpiece.

Objectives

After completing this module, you will be able to:

- Describe the different types of surface milling sequence.
- Describe the key surface milling manufacturing parameters.
- Describe methods for creating approach and exit moves.
- Create and refine surface milling sequences.
- Create mill surfaces relevant to surface milling.
- Create material removal features specific to surface milling.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

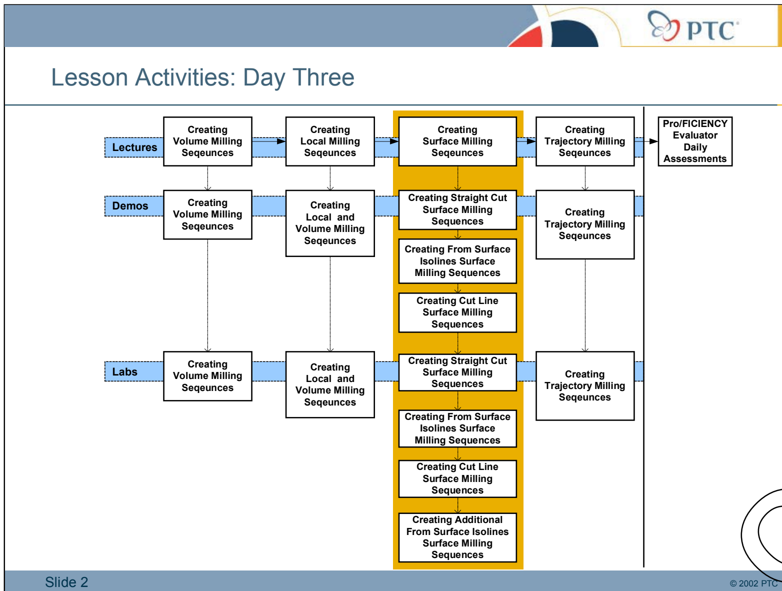
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

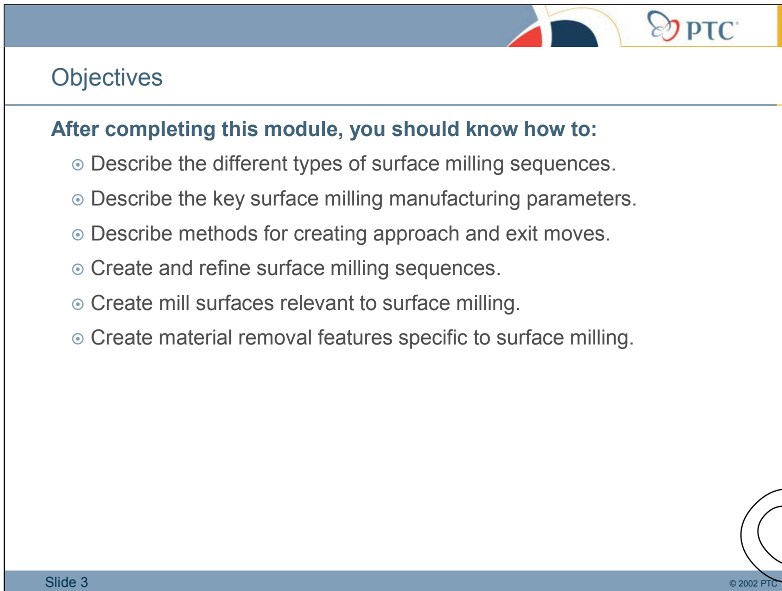
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 20 mins
- Demos (3) : 25 mins
- Labs (3 +1 challenge) : 45 mins
- Total: 1 hr 30mins

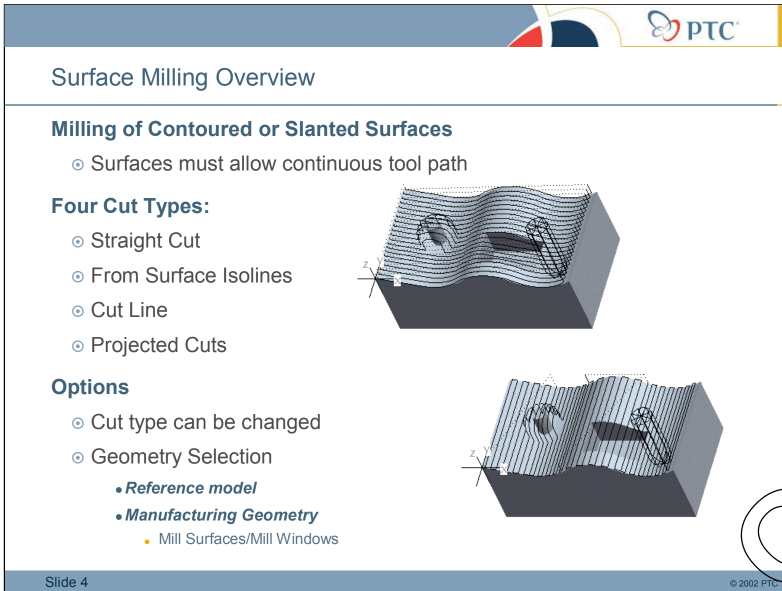


Overview

Surface milling sequences enable you to mill contoured and horizontal surfaces. This is useful for finish machining contoured surfaces of components, or mold cavities. When creating surface milling sequences, mill surfaces can be used as alternate references when model surfaces are inappropriate, for example when the model being machined is made up of many surfaces, or if you need to extend the toolpath beyond the model. There are four types of surface milling sequence, providing you with a flexible approach to surface machining. It is an important skill to understand the differences between each type of surface milling sequence, and when to use each type of sequence. Once you have completed your NC sequence, you can represent the as-machined state of your workpiece by creating a material removal feature that removes the machined volume from the workpiece.

After completing this module, you will be able to:

- Describe the different types of surface milling sequences.
- Describe the key surface milling manufacturing parameters.
- Describe methods for creating approach and exit moves.
- Create and refine surface milling sequences.
- Create mill surfaces relevant to surface milling.
- Create material removal features specific to surface milling.



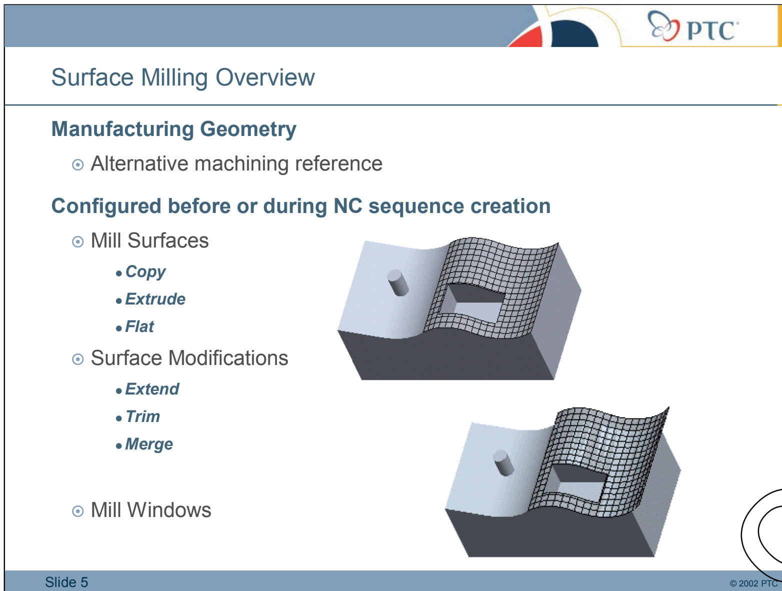
Surface Milling Process

Surface Milling is used to mill horizontal or contoured surfaces. The selected surfaces must allow for a continuous tool path. There are several methods of configuring the cut motion and generating the tool path:

- Straight Cut: mill the selected surfaces by a series of straight cuts.
- From Surface Isoline: mill the selected surfaces by following the surface u-v lines.
- Cut Line: mill the selected surfaces by defining the shape of the first, last, and some intermediate cuts. When the system generates the tool path, it gradually changes the shape of the cuts according to surface topology.
- Projected Cuts: mill the selected surfaces by projecting their contours on the retract plane, creating a "flat" tool path in this plane (using the appropriate scan type), and then projecting this tool path back on the original surface(s).

Options

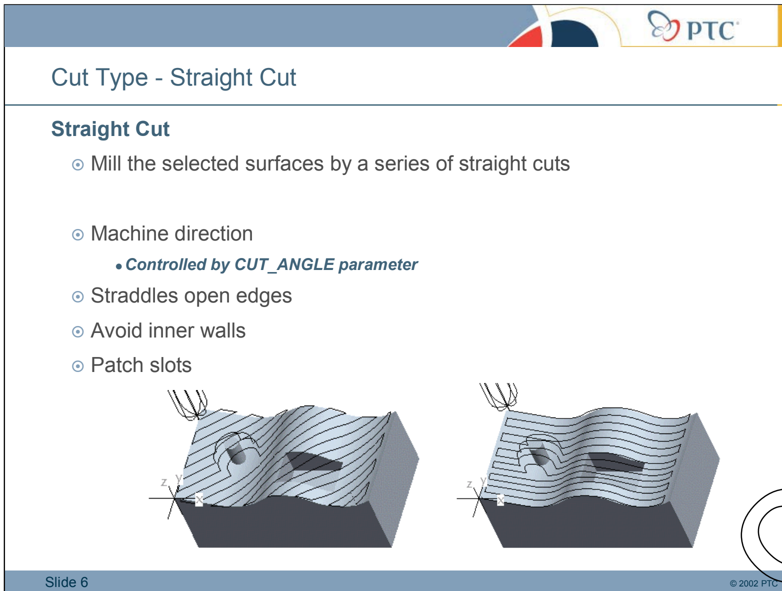
- The cut type can be changed during the configuration of the sequence.
- Selecting geometry:
  - You can select reference model surfaces for machining
  - Or create manufacturing geometry
    - Mill surfaces can be used.
    - All surfaces within a mill window will be selected.



Surface Milling Process

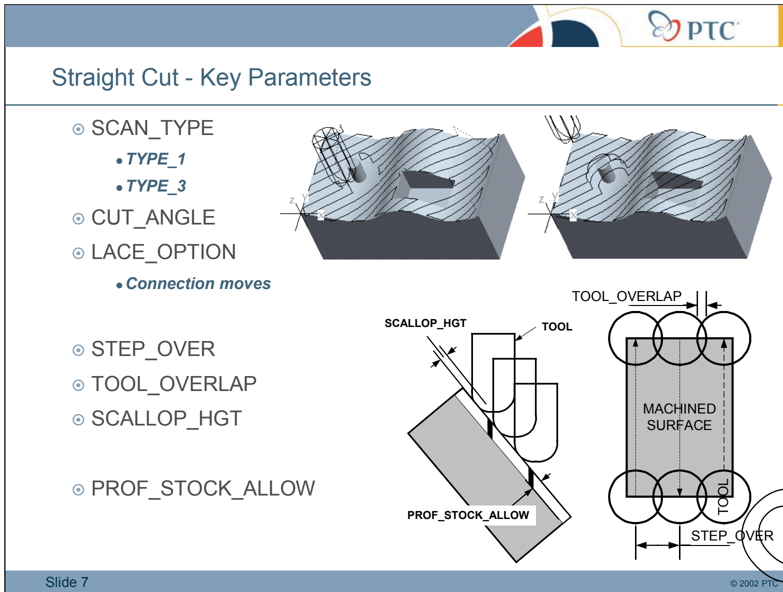
- Manufacturing geometry, can be used as an alternative machining reference.
- Manufacturing geometry can be configured before or during the creation of NC sequences.
  - Mill surfaces can be created, the most common options for surface milling are:
    - Copy: copy selected reference model surfaces.
    - Extrude: sketch a 2-D outline and extrude the outline to create a surface
    - Flat: sketch a flat 2-D outline to form the boundaries of a surface.
    - It is also possible to extend, trim and merge surfaces together.

    - Mill windows can also be used, in this case all surfaces within a mill window will be selected.



Straight Cut

- Machine direction controlled by cut angle.
- Completely machine the selected surface(s). If a surface is not bounded by walls on the outside, the tool will "straddle", that is, overrun the surface boundary by a half diameter.
- Any inner protrusions, as well as the outer walls extending up from the surface, will be avoided automatically. The stock allowance, if any, will apply to the side walls as well.
- If a surface is selected from model, any slots or holes on the surface will be "patched": by default the tool path will be generated as if they were not there.



Straight Cut Parameters

SCAN_TYPE: Refers to the way a milling tool scans the surfaces and avoids islands.

For straight cut surface milling the options are:

- TYPE_1—continuously machines the surface, retracts upon encountering islands.
- TYPE_3—the tool removes material from continuous zones defined by the island geometry, machining them in turn and moving around the islands.

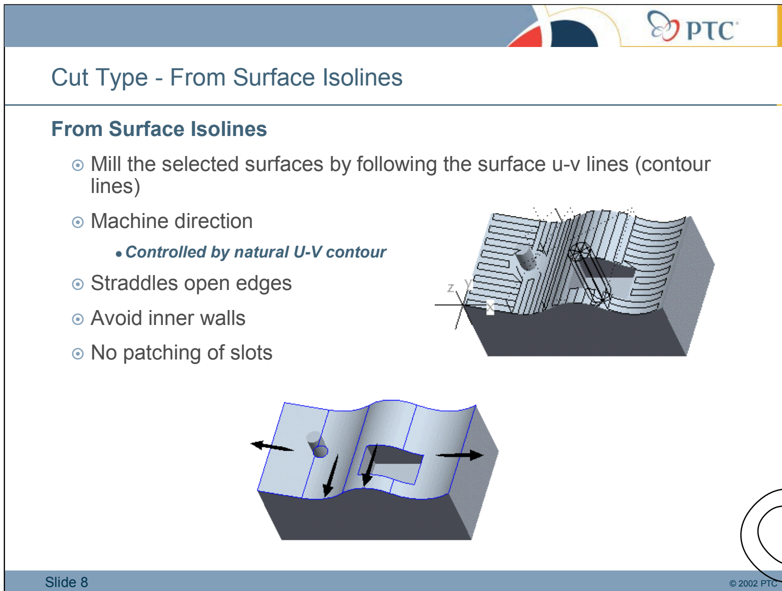
- CUT_ANGLE — the angle between the cut direction and the X-axis of the NC Sequence coordinate system.
- LACE_OPTION — if set causes the tool to cut back and forth, and specifies the shape of connection between the endpoints of neighboring cuts. There are various connection types.
  - LINE_CONNECT
  - CURVE_CONNECT
  - ARC_CONNECT
  - LOOP_CONNECT

Lateral control is through:

- STEP_OVER default parameter for controlling the step over within a pass.
- Or TOOL_OVERLAP can be used as an alternative method to control the step-over based on the tool overlap.
- Or SCALLOP_HEIGHT must be less than or equal to cutter radius, can also be used to calculate step-over.

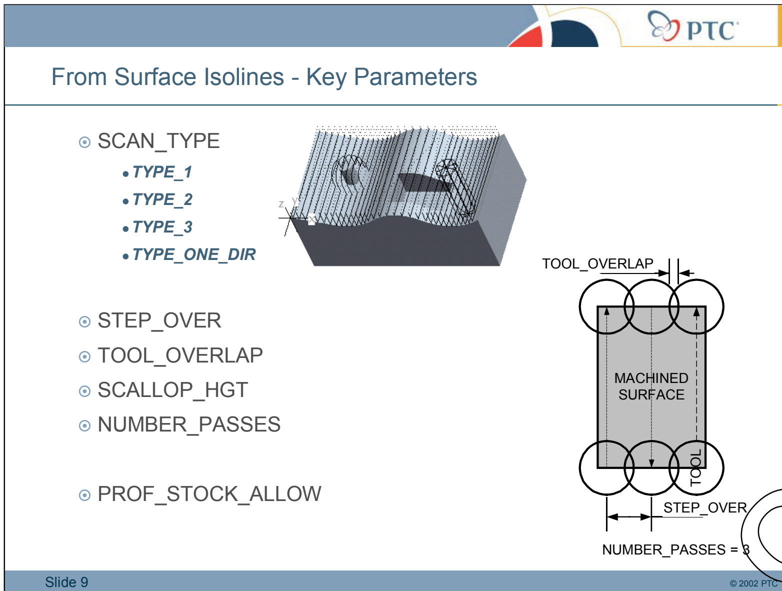
Whichever parameter produces the smallest step over is used.

- PROF_STOCK_ALLOW – controls the stock remaining on machined surfaces. Is also used to calculate remaining stock when creating material removal features. For surface milling this can be negative, allowing marching into the reference model, this can be useful when machining electrodes.



From Surface Isolines

- Machine direction controlled natural U-V contours of machined surfaces.
- Completely machine the selected surface(s). If a surface is not bounded by walls on the outside, the tool will "straddle", that is, overrun the surface boundary by a half diameter.
- Any inner protrusions, as well as the outer walls extending up from the surface, will be avoided automatically. The stock allowance, if any, will apply to the side walls as well.
- Internal holes and slots will *not* be "patched." For 3-Axis milling, the tool will move over internal slots or holes at a constant Z level, plunging or retracting at the lower edge as necessary. For 4- and 5-Axis milling, the tool will retract at all inner edges. If you don't want this to happen, use a mill surface, or select a surface from a mill volume.



From Surface Isolines Parameters

SCAN_TYPE: Refers to the way a milling tool scans the surfaces and avoids islands.

For from surface isolines surface milling the options are:

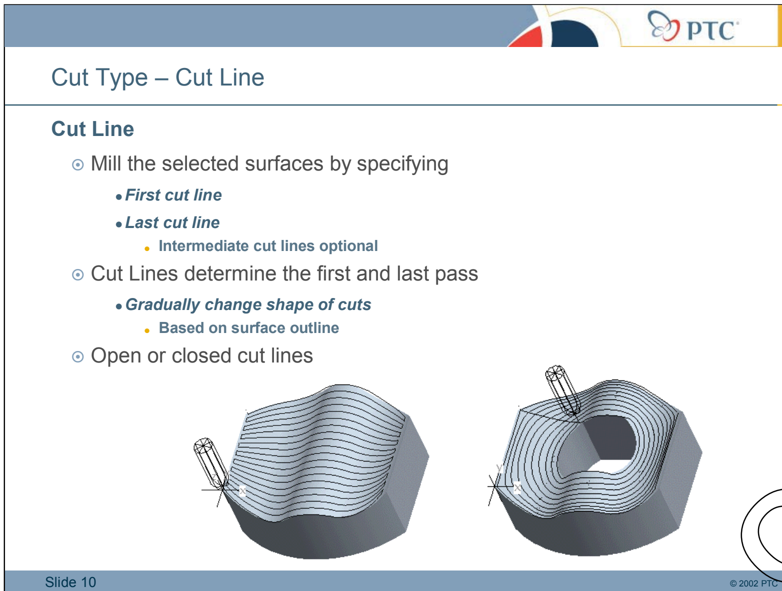
- TYPE_1—continuously machines the surface, retracts upon encountering islands.
- TYPE_2—continuously machines the surface without retract, moving around the islands
- TYPE_3—the tool removes material from continuous zones defined by the island geometry, machining them in turn and moving around the islands.
- TYPE_ONE_DIR—the tool cuts in one direction only.

Lateral control is through:

- STEP_OVER default parameter for controlling the step over within a pass.
- Or TOOL_OVERLAP can be used as an alternative method to control the step-over based on the tool overlap.
- Or SCALLOP_HEIGHT must be less than or equal to cutter radius, can also be used to calculate step-over.
- Or NUMBER_PASSES can be used to explicitly set the number of passes to take in each slice.

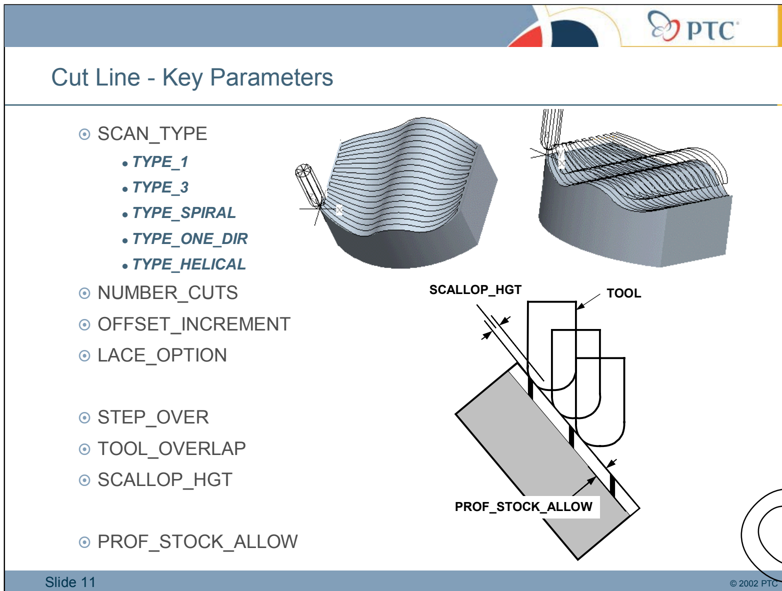
Whichever parameter produces the smallest step over is used.

- PROF_STOCK_ALLOW – controls the stock remaining on machined surfaces. Is also used to calculate remaining stock when creating material removal features. For surface milling this can be negative, allowing marching into the reference model, this can be useful when machining electrodes.



Cut Line

- Mill the selected surfaces by defining the shape of the first, last, and some intermediate cut lines.
- When the system generates the tool path, it gradually changes the shape of the cuts according to surface outline and geometry.
- Both open and closed cut lines are possible.



Cut Line Parameters

SCAN_TYPE: Refers to the way a milling tool scans the surfaces and avoids islands.

For cut LINE surface milling the options are:

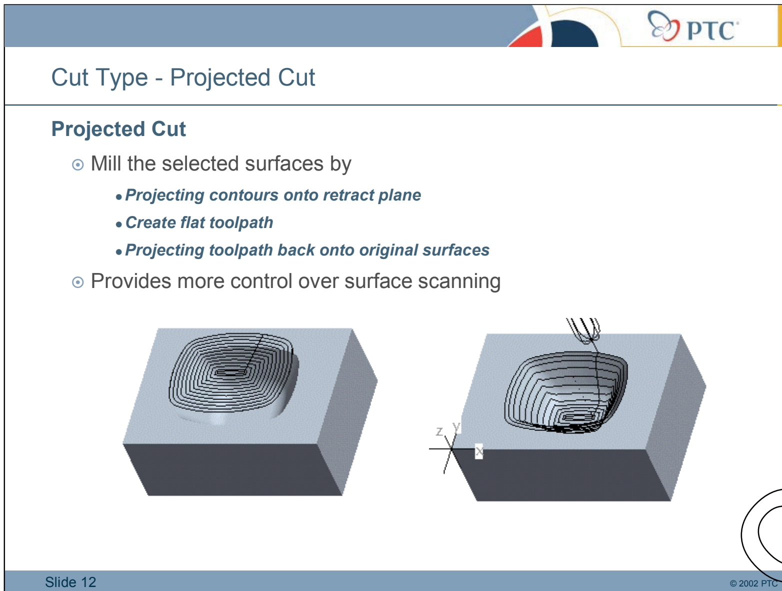
- TYPE_1—continuously machines the surface, retracts upon encountering islands.
- TYPE_3—the tool removes material from continuous zones defined by the island geometry, machining them in turn and moving around the islands.
- TYPE_SPIRAL—generates a spiral cutter path.
- TYPE_ONE_DIR—the tool cuts in one direction only.
- TYPE_HELICAL — the tool moves along a helix. Valid for Closed Cut Line machining only
- You can create multiple passes at different z-heights, using NUMBER_CUTS and OFFSET_INCREMENT
- LACE_OPTION — if set causes the tool to cut back and forth, and specifies the shape of connection between the endpoints of neighboring cuts.

Lateral control is through:

- STEP_OVER default parameter for controlling the step over within a pass.
- Or TOOL_OVERLAP can be used as an alternative method to control the step-over based on the tool overlap.
- Or SCALLOP_HEIGHT must be less than or equal to cutter radius, can also be used to calculate step-over.

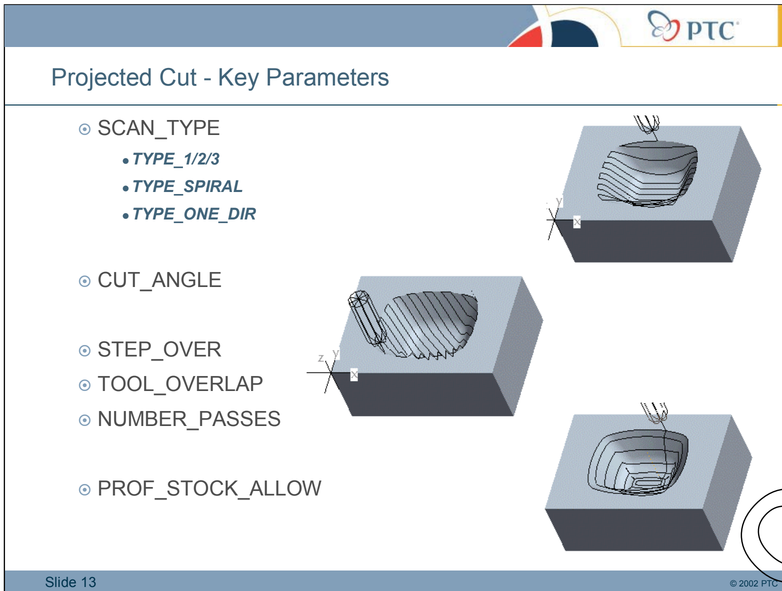
Whichever parameter produces the smallest step over is used.

- PROF_STOCK_ALLOW – controls the stock remaining on machined surfaces. Is also used to calculate remaining stock when creating material removal features. For surface milling this can be negative, allowing machining into the reference model, this can be useful when machining electrodes.



Projected Cut

- Mill the selected surfaces by projecting their contours onto the retract plane, creating a "flat" tool path in this plane (using the appropriate scan type), and then projecting this tool path back on the original surface(s). This method is available for 3 Axis surface milling only.
- Use projected cuts surface milling when you need more control over the way you scan the surfaces (for example, if you need to generate a spiral tool path).



Projected Cut Parameters

SCAN_TYPE: Refers to the way a milling tool scans the surfaces and avoids islands.

For straight cut surface milling the options are:

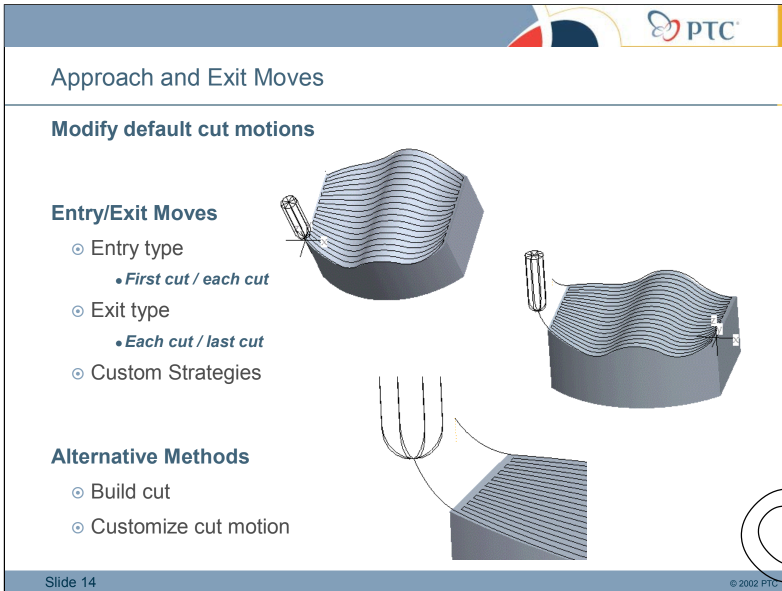
- TYPE_1—continuously machines the surface, retracts upon encountering islands.
- TYPE_2—continuously machines the surface without retract, moving around the islands
- TYPE_3—the tool removes material from continuous zones defined by the island geometry, machining them in turn and moving around the islands.
- TYPE_SPIRAL—generates a spiral cutter path.
- TYPE_ONE_DIR—the tool cuts in one direction only.
- CUT_ANGLE—the angle between the cut direction and the X-axis of the NC Sequence coordinate system.

Lateral control is through:

- STEP_OVER default parameter for controlling the step over within a slice.
- Or TOOL_OVERLAP can be used as an alternative method to control the step-over based on the tool overlap.
- Or NUMBER_PASSES can be used to explicitly set the number of passes to take in each slice.

Whichever parameter produces the smallest step over is used.

- PROF_STOCK_ALLOW – controls the stock remaining on machined surfaces. Is also used to calculate remaining stock when creating material removal features. For surface milling this can be negative, allowing marching into the reference model, this can be useful when machining electrodes.



Approach and Exit Moves

There are many times when additional approach and exit moves are required to improve a tool path.

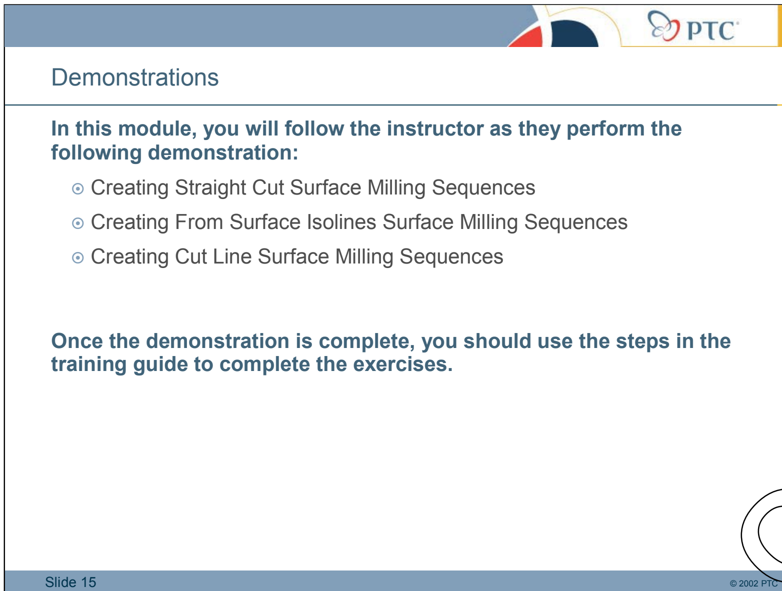
When creating NC sequences default cut motions are created based on selected geometry and manufacturing parameters. These default cut motions can be modified if required.

- The Approach/Exit option in the Sequence Setup menu provides access to the Entry/Exit dialog box
  - The Entry/Exit Move dialog box enables modification of the default cut motion and contains two group boxes.
  - You can select standard strategies for entry and exit.
  - You can specify different entry strategies for the first cut and the intermediate cuts, as well as different exit strategies for the intermediate cuts and the last cut.
  - Custom Strategies—Lets you define custom strategies for entry and exit.

Alternative methods for modifying the default cut motion are:

- Build cut in the Sequence Setup menu
- Customize in the NC sequence menu

Both options enable you to modify the existing cut motions and specify approach and exit moves.



Demonstration: Using Parameters and Setting up Site Files

Scenario

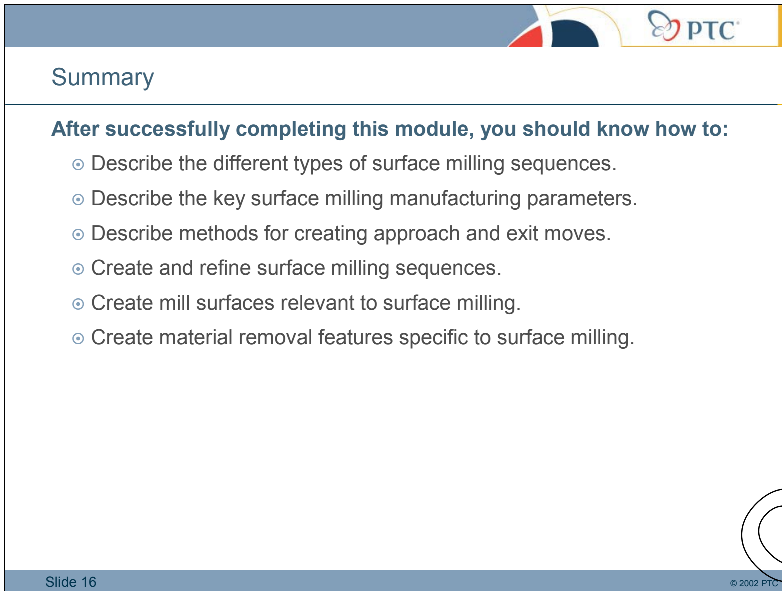
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the different types of surface milling sequences.
- Describe the key surface milling manufacturing parameters.
- Describe methods for creating approach and exit moves.
- Create and refine surface milling sequences.
- Create mill surfaces relevant to surface milling.
- Create material removal features specific to surface milling.

Module 11 Lab Exercises

Exercise 1: Creating Straight Cut Surface Milling Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create a mill surface as a reference for the surface milling sequences. You then create a semi-finish sequence on the top of the model by creating a straight cut surface milling sequence using 25-millimeter ball end mill. Before completing this sequence, you change the model reference to the mill surface enabling the tool to overlap the edges of the top surface. Mill surfaces are useful as they can be used as alternate references when model surfaces are inappropriate. You also use a solid model tool to enhance the display when reviewing tool paths.

Objectives

After successfully completing this exercise, you will know how to:

- Create surface-milling sequences using the straight cut option.
- Create mill surfaces.

Scenario

You continue manufacturing the aluminum cover by creating a number of surface milling sequences. This enables you to machine all the contoured surfaces on the top of the cover. You use different surface milling techniques based on the surface finish required on the product.

In this exercise, you create a mill surface as a reference for the surface milling sequences. You then create a semi-finish sequence on the top of the model by creating a straight cut surface milling sequence using a 25-millimeter ball end mill. You use a solid model tool to enhance the display during tool path review.

Parameters

The following are key surface milling parameters.

Table 1: Parameter Descriptions

Parameter	Description
SCAN_TYPE	The method of scanning the machined area.
CUT_ANGLE	The angle of cuts relative to NC sequence coordinate system.
LACE_OPTION	Causes the tool to cut back and forth, and specifies the shape of connection between the endpoints of neighboring cuts.
STEP_OVER	Distance between passes.
TOOL_OVERLAP	Tool step over calculated by overlap distance.
SCALLOP_HGT	Step over calculated by scallop height.
NUMBER_PASSES	Number of passes per slice. (From surface isolines and projected cut sequences only)
STEP_OVER_ADJUST	Adjust STEP_OVER to provide complete coverage of machined area.
PROF_STOCK_ALLOW	Controls stock remaining on machined surfaces.
NUMBER_CUTS	For cut line machining can specify multiple cuts and different z-depths.
OFFSET_INCREMENT	For cut line machining offset depth for multiple cuts.

Step 1. Locate and open the cover manufacturing model.

Before creating the volume milling sequence, we need to locate the manufacturing model and open it using the folder navigator. When we have located the folder for this exercise we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module11**, right-click, and select **Make Working Directory.**
2. Open COVER.MFG. Turn off any displayed datum features.

Step 2. Create a mill surface feature.

As an alternative to machining the reference model surfaces, we can create a mill surface to extend the machined surfaces beyond the outer edge of the model.

Mill surfaces are useful because they can be used as alternative references when model references are not appropriate.

We need to copy the surface from the top of the reference model and extend the outer edges.

To do this, we create a mill surface using the ***Copy*** *option. We select the top of the reference model, this surface is hidden under the workpiece surface so should be selected carefully. When we have copied the surface we can extend the outer edges by 2.5 mm, again ensure we pick the correct edge to extend.*

1. Using the Menu Manager, select **Mfg Setup > Mfg Geometry > Mill Surface > Create** to create a mill surface, name it [**finish_surface**].

2. Change the view to the named view **OP020-TRI**.

3. Select **Add > Copy > Done**, and select the hidden surface on the top cover of the reference model. Complete the selection and close the Surface Copy dialog box.

> **Tip:**
> Ensure you pick the hidden surface of the reference model, not the surface from the workpiece. Use right-click to highlight the hidden surface.

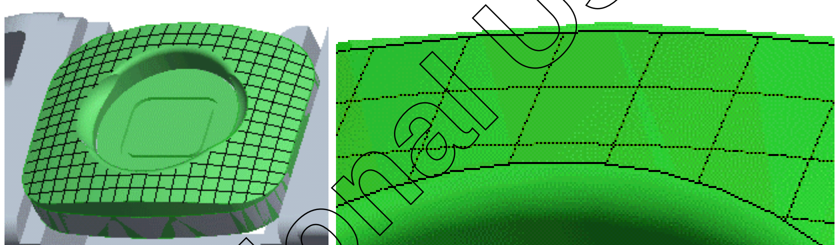


Figure 1: Selected hidden reference model surface

4. In the Surf Define menu, select **Extend** to extend the outer edges of the surface. Use the **Same Surf > Single Dst > Dist On Srf** options.

5. Using the **Tangent Chain** option, select one of the outer edges of the surface; again ensure you pick the hidden edge, not the workpiece edge. Complete the selection and select the same edge again to measure the extension distance. Type [**2.5**] *(0.10 in)* as the extend distance.

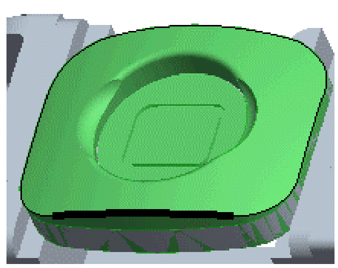


Figure 2: Tangent chain of edges and selected edge.

6. Complete the extension process, and return to the Manufacture menu.

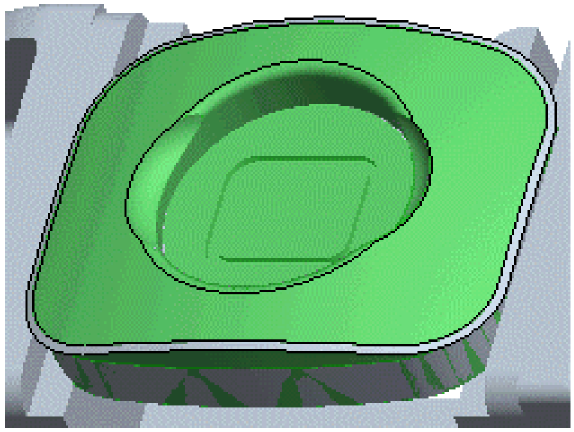


Figure 3: Showing extended surface.

> **Note:**
> Notice there are two new surface features in the model tree; one representing the original copied surface, and a second representing the extended surface.

Step 3. Create a semi-finish surface milling NC sequence using the straight cut option.

Using the Menu Manager, we can create a surface milling sequence. We can select the name, tool and parameter options as these are the additional sequence references we need to specify. We can enter a name for the sequence to make it easy to identify. We can retrieve the solid model tool to enhance the display when reviewing the tool path, and we can also specify the stock material for the tool and read in the speed, feed and depth-of-cut data for this tool. Notice how the site parameter file sets all the default manufacturing parameters. We can modify the PROF_STOCK_ALLOW to leave material on the machined surface, as this is only a semi-finishing tool path. We can also set the SCALLOP_HGT to control the step over between passes. We can select the mill surface we just created as the surface to be machined, ensuring that the upward facing side of the surface is machined. We can now specify the cut type, and accept the straight cut option.

We play the toolpath, and notice how the tool machines past the edges of the model surfaces to the edge of the larger mill surface. The tool also machines over the opening in the top surface, this can be changed using the SCAN_TYPE parameter. Notice the tool performs a lacing move between passes, this is controlled by the LACE_OPTION parameter, and the cut direction is parallel to the x-axis of the sequence co-ordinate system, controlled by CUT_ANGLE parameter.

1. Using the Menu Manager, create a **Surface Mill** milling sequence.
2. Select the **Name, Tool,** and **Parameters** check boxes.

> **Note:**
> Notice Surfaces and Define Cut are already selected, you are only selecting additional options that you need to change.

3. Name the sequence [**FIN_TOP_1**].

4. In the Tool Setup Dialog box, select **File**, and **Open Tool Library**, use the **By Copy** option and retrieve the BEM_25_0_SOLID, solid model tool. Set the tool number and tool offset number to [**19**].

5. Select the **Feeds-Speeds** tab, and click **Read DB** button to read in the cutting data for a roughing application and aluminum stock material.

6. **Apply** the changes and **Move** the tool to the existing pocket, finally select the tool for the sequence.

> **Note:**
> The solid model tool can be seen when playing the toolpath.

7. Modify the following parameters:

Table 2: Changed Parameters

Manufacturing Parameter	Value	Section
PROF_STOCK_ALLOW	0.15 *(0.006 in)*	CUT PARAM
SCALLOP_HGT	0.1 *(0.004 in)*	CUT PARAM

> **Note:**
> Notice how the site parameter file sets all the default values.

8. Select **Mill Surface > Done > Select Srf**, and use the **FINISH_SURFACE** from the list of surfaces.

9. Select **Okay,** to use the upward facing side of the surface. Use the **Select All** option to ensure the entire surface is used for machining, and complete the surface selection.

10. In the Cut Definition dialog box, accept the **Straight Cut** option, and close the dialog box.

11. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

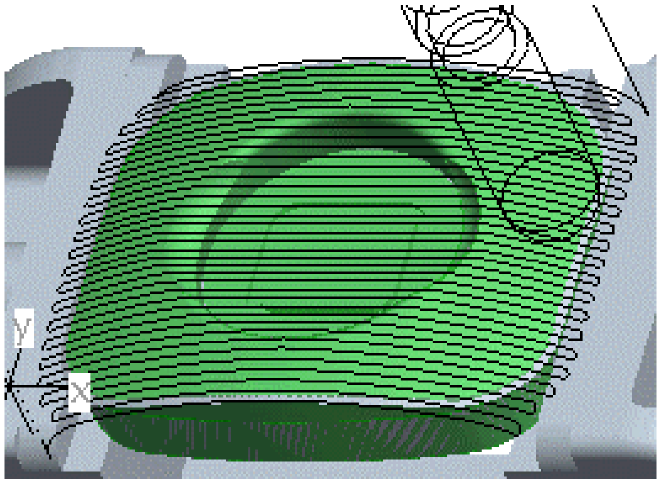


Figure 4: Showing the resulting toolpath.

Toolpath Review:

Notice how the tool machines past the edges of the model surfaces to the edge of the larger mill surface. The tool also machines over the opening in the top surface, this can be changed using the SCAN_TYPE parameter. Notice also the tool performs a lacing move between passes, this is controlled by the LACE_OPTION parameter, and the cut direction is parallel to the x-axis of the sequence co-ordinate system, which is controlled by CUT_ANGLE parameter.

Step 4. Modify the sequence parameters to control the scan type, the lacing move, and cutting angle.

We can modify the SCAN_TYPE to TYPE_3 to eliminate machining over the opening in the top surface. We can modify the LACE_OPTION to ARC_CONNECT to connect each pass with a smooth motion. We can also set the CUT_ANGLE to 90 degrees.

1. Set the following parameters in the sequence:

Table 3: Changed Parameters

Manufacturing Parameter	Value	Section
SCAN_TYPE	TYPE_3	CUT OPTION
LACE_OPTION	ARC_CONNECT	CUT OPTION
CUT_ANGLE	90	CUT PARAM

2. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

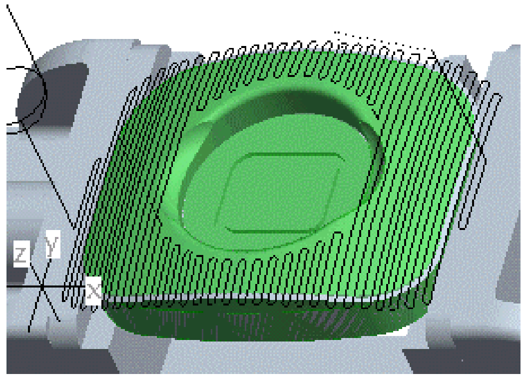


Figure 5: Showing the resulting toolpath.

3. In the Play Path menu, select **NC Check** to start Vericut.

4. Click on the Play to End icon to view the machining simulation.

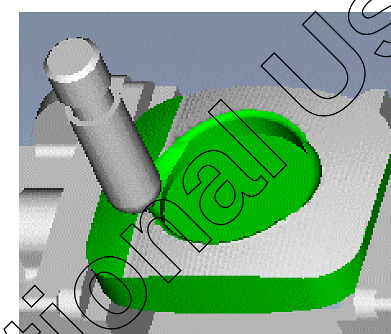


Figure 6: Showing the resulting toolpath in Vericut.

Tool Path Review:

Modifying the SCAN_TYPE to TYPE_3, eliminates machining over the opening in the top surface. Modifying the LACE_OPTION to ARC_CONNECT connects each pass with a smooth motion. Modifying the CUT_ANGLE to 90 degrees changes the direction of the cutting motions.

5. Close Vericut and complete the NC sequence

6. Save the manufacturing model.

Exercise 2: Creating From Surface Isolines Surface Milling Sequences

Introduction

In this demonstration, you add a finishing sequence to the top surface by creating a from isolines surface milling sequence. You also add a tangent approach and exit move to the tool path.

Objectives

After successfully completing this exercise, you will know how to:

- Create surface milling sequences using the from surface isolines option.
- Customize tool path approach and exit motions.

Scenario

You add a finishing sequence to the top surface by creating a from isolines surface milling sequence. You also add a tangent approach and exit move to the tool path.

Step 1. Create a finish surface milling NC sequence using the from surface isolines option.

Using the Menu Manager we can create a surface milling sequence. We can select the name and parameters options, as these are the additional sequence references we need to specify. We can enter a name for the sequence to make it easy to identify. There is no need to specify the tool as we are using the solid model tool from the previous sequence. For the parameters we can start by using the previous sequence parameters. As this is a finishing pass, we can modify the PROF_STOCK_ALLOW to leave zero material on the machined surface. We can also set the SCALLOP_HGT to provide a smaller step over between passes. We can select the same mill surface to be machined, ensuring that the upward facing side of the surface is machined. We can now specify the cut type, and select the from surface isolines option, the cut direction can be changed if required, but we can leave it unchanged.

We play the toolpath, and notice how the cut direction follows the contour of the surface instead of a true linear cut. Notice also how the tool makes a normal approach move when starting to machine the inside edge of the machined surface on the first pass, and the tool also makes a normal exit move on the last pass in that section of the tool path. This can be adjusted by specifying tangent approach and exit moves.

1. Using the Menu Manager, create a **Surface Mill** milling sequence.

> **Note:**
> Notice Surfaces and Define Cut are already selected, you are only selecting additional options that you need to change.

2. Select the **Name** and **Parameters** check boxes.

3. Name the sequence [**FIN_TOP_2**].

4. For parameters select the **Use Prev** option, and use the **FIN_TOP_1** parameters

5. Modify the following parameters:

Table 4: Changed Parameters

Manufacturing Parameter	Value	Section
PROF_STOCK_ALLOW	0.0	CUT PARAM
SCALLOP_HGT	0.025 *(0.001 in)*	CUT PARAM

6. Select **Mill Surface > Done > Select Srf**, and use the **FINISH_SURFACE** from the list of surfaces.

7. Select **Okay** to use the upward facing side of the surface. Use the **Select All** option to ensure the entire surface is used for machining, and complete the surface selection.

8. In the Cut Definition dialog box select the **From Surface Isolines** option, and **Preview** the machining scan. Close the dialog box when finished.

> **Note:**
> You can change the direction of cutting using the toggle icon in the dialog box.

9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

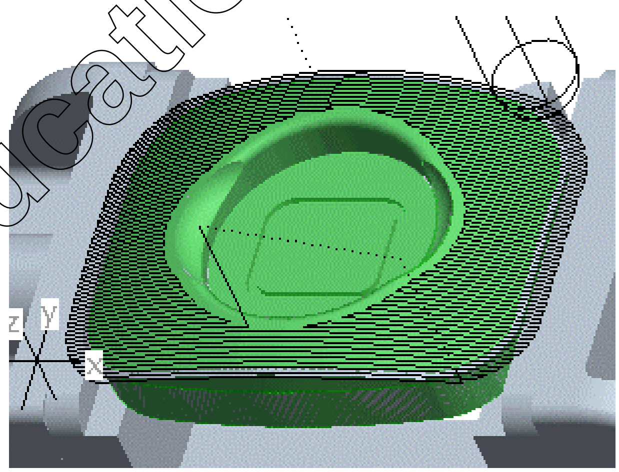


Figure 7: Showing the resulting toolpath.

> **Tool Path Review:**
> Notice how the cut direction follows the contour of the surface (u-v vectors) instead of a true linear cut. Notice also how the tool makes a normal approach move when starting to machine the inside edge of the machined surface on the first pass, and the tool also makes a normal exit move on the last pass in that section of the tool path. This can be adjusted by specifying tangent approach and exit moves.

Step 2. Create tangent approach and exit moves to remove witness lines from the curved surfaces on the inside edges of the machined surface.

Create tangent approach and exit moves to remove witness lines from the curved surfaces on the inside edges of the machined surface. We can modify the APPROACH_DISTANCE and EXIT_DISTANCE parameters to ***10****, and using the Approach/Exit option we can specify a* ***LINE_TANGENT*** *move for the first and last cut.*

We can play the toolpath and notice the additional tangent moves when approaching and moving away from the curved surfaces on the inside edge of the machined surface.

1. Modify the following parameters:

Table 5: Changed Parameters

Manufacturing Parameter	Value	Section
APPROACH_DISTANCE	10 *(0.40 in)*	ENTRY/EXIT
EXIT_DISTANCE	10 *(0.40 in)*	ENTRY/EXIT

2. Select **Seq Setup >Approach/Exit > Done.** In the Entry/Exit Move dialog box, specify a **LINE_TANGENT** move for the first and last cut, then close the dialog box.

3. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

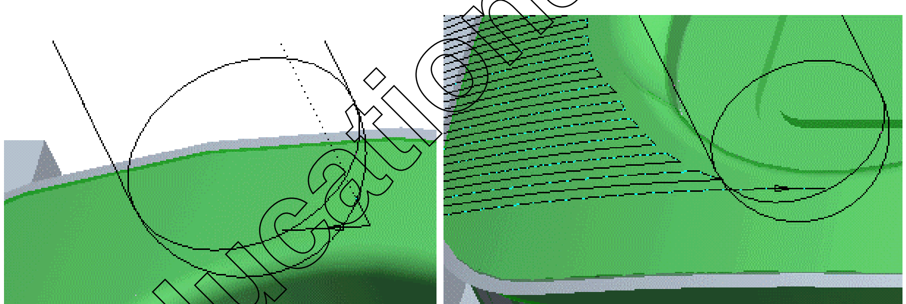


Figure 8: Showing additional tangent approach and exit moves.

Tool Path Review:

Notice the additional tangent moves when approaching and moving away from the curved surfaces on the inside edge of the machined surface.

4. Complete the sequence and return to the Manufacture menu.

5. Save the manufacturing model.

Exercise 3: Creating Cut Line Surface Milling Sequences

Introduction

In this demonstration, you add another finishing sequence to the top surface by creating a cut line surface milling sequence. This surface milling sequence provides a better surface finish than the previous finishing sequence, so you delete the second surface milling sequence you created.

Objectives

After successfully completing this exercise, you will know how to:

- Create surface milling sequences using the cut line option.

Scenario

You add another finishing sequence to the top surface by creating a cut line surface milling sequence. This surface milling sequence provides a better surface finish than the previous finishing sequence, so you delete the second surface milling sequence you created.

Step 1. Create a finish surface milling NC sequence using the cut line option.

Using the Menu Manager, we can create a surface milling sequence. We can select the name and parameters options, as these are the additional sequence references we need to specify. We can enter a name for the sequence to make it easy to identify. There is no need to specify the tool as we are using the solid model tool from the previous sequence. For the parameters we can start by using the previous sequence parameters. As this is a finishing pass, we can modify the PROF_STOCK_ALLOW to leave zero material on the machined surface. We can also set the SCALLOP_HGT to provide a smaller step over between passes. We can select the same mill surface to be machined, ensuring that the upward facing side of the surface is machined. We can now specify the cut type, and select the from surface isolines option, the cut direction can be changed if required, but we can leave it unchanged.

We play the toolpath, and notice how the cut direction follows the contour of the surface instead of a true linear cut. Notice also how the tool makes a normal approach move when starting to machine the inside edge of the machined surface on the first pass, and the tool also makes a normal exit move on the last pass in that section of the tool path. This can be adjusted by specifying tangent approach and exit moves.

1. Using the Menu Manager, create a **Surface Mill** milling sequence.
2. Select the **Name** and **Parameters** check boxes.
3. Name the sequence [**FIN_TOP_3**].
4. For parameters, select the **Use Prev** option, and use the **FIN_TOP_2** parameters.

5. Select **Mill Surface > Done > Select Srf**, and use the **FINISH_SURFACE** from the list of surfaces.

6. Select **Okay** to use the upward facing side of the surface. Use the **Select All** option to ensure the entire surface is used for machining, and complete the surface selection.

7. In the Cut Definition dialog box, select the **Cut Line** option, and for the cut line style select the **Closed Loops** option.

8. Click the Add a Cut Line icon to add a start cut line, select **Accept > Done** to use the outer boundary chain of edges shown in the following figure.

9. Using the **Select Start Point** option, select a start point approximately half way along the front edge of the mill surface as shown. Use the **Enter** option and type in [**0.5**], to position the point half way along the edge.

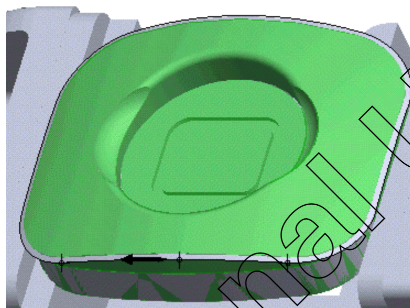


Figure 9: Outer cut line and start point highlighted.

10. Close the Add/Redefine Cut Line dialog box.

11. Click the Add a Cut Line icon to add an end cut line, select **Next > Accept > Done** to use the inner boundary chain of edges, shown in the following figure.

12. Using the **Select Start Point** option, select a start point approximately half way along the inner edge of the mill surface as shown. Use the **Enter** option and type in [**0.5**], to position the point half way along the edge.

13. Close the Add/Redefine Cut Line dialog box.

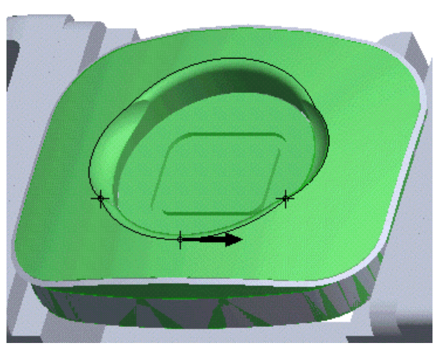


Figure 10: Inner cut line and start point highlighted.

14. Preview the machining scan, and close the Cut Definition dialog box.

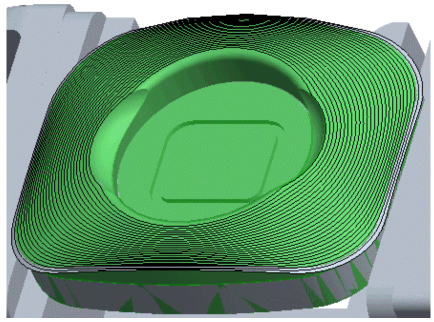


Figure 11: Previewed machining scan.

15. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

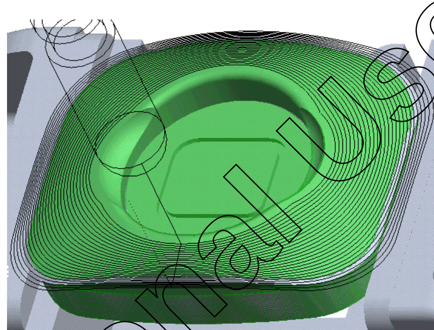


Figure 12: Showing the resulting toolpath.

Tool Path Review:

Notice how the tool path gradually changes shape between the start and end cut lines. It is also possible to modify the starting points of the cut lines as you did in this exercise.

16. Complete the sequence.

17. In the model tree, select **FINISH_SURFACE**, right-click and **Hide** the mill surface.

Step 2. Delete the second surface milling sequence.

We decide that this tool path provides a better surface finish than the previous tool path, so we can delete the from surface isolines sequence. We can select the sequence in the model tree and delete it. An alternative to deleting the sequence is to suppress the sequence; this enables us to resume the sequence later if required.

1. In the model tree, select the **FIN_TOP_2** sequence and delete it.

2. Save the manufacturing model.

Exercise 4: Creating Additional From Surface Isolines Surface Milling Sequences (Challenge)

Objectives

After successfully completing this exercise, you will know how to:

- Create surface milling sequences using the from surface isolines option.

Scenario

You create two more finish surface milling sequences to machine the left and right cut outs, and to machine the rounds on the top of the cover, you use the from surface isolines option.

Step 1. Create a surface milling sequence using the from surface isolines option to machine the left and right cut outs on the top of the model.

1. Using the Menu Manager, create a surface milling sequence.
2. Select the **Name, Tool** and **Parameters** check boxes.
3. Name the sequence [**FIN_CUT_OUTS**].
4. Change the tool to the **BEM_12_0** tool.
5. Modify the SCALLOP_HGT parameter to 0.025 *(0.001 in)*.
6. For the surfaces to be machined, select the two cut out surfaces on the reference model, as shown in the following figure.

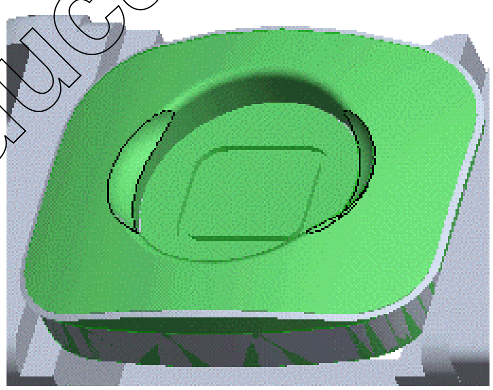


Figure 13: Showing left and right cut outs selected.

7. For the cut definition, select the **From_Surface_Isolines** option.
8. Modify the cut direction for both surfaces to be parallel to the y-axis of the machine co-ordinate system.

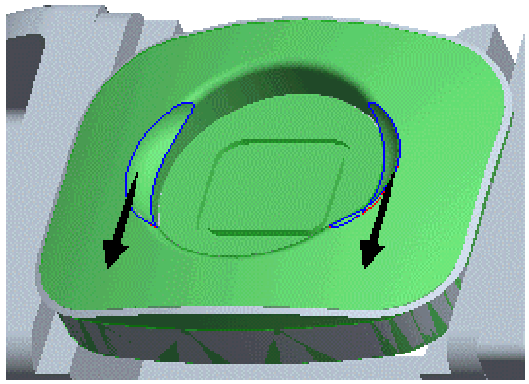


Figure 14: Showing cut direction.

> **Note:**
> The direction of cut and the order of machining can be modified in the Cut Definition dialog box.

9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

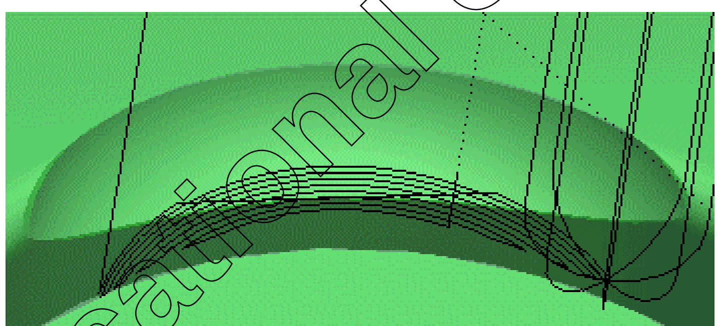


Figure 14: Showing one cut out surface being machined.

10. Complete the sequence.

Step 2. Create a surface milling sequence using the from surface isolines option to machine the rounds on the top of the model.

1. Using the Menu Manager, create a surface milling sequence.

2. Select the **Name,** and **Parameters** check boxes.

3. Name the sequence [**FIN_ROUNDS**].

4. For parameters select the **Use Prev** option, and use the **FIN_CUT_OUTS** parameters.

5. Modify the SCALLOP_HGT parameter to [**0.002**] *(0.0001 in).*

6. For the surfaces to be machined, select the two rounded surfaces on the reference model, as shown in the following figure.

7. For the cut definition select the **From_Surface_Isolines** option.

8. Modify the cut direction for both surfaces to be parallel to the x-axis of the machine co-ordinate system.

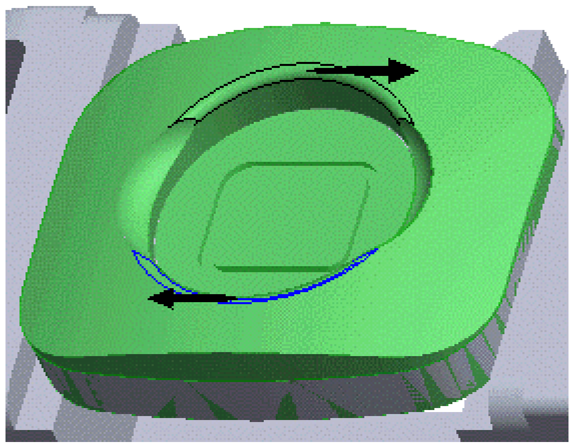


Figure 15: Showing selected surfaces and cut direction.

9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

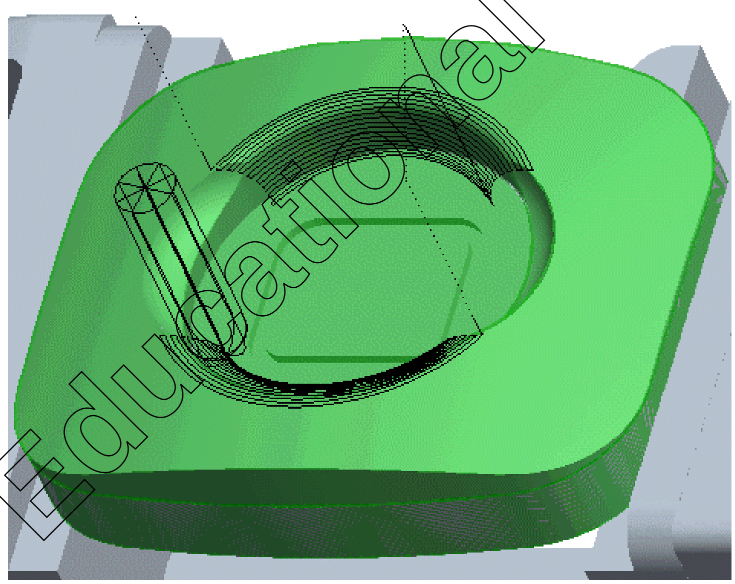


Figure 16: Showing the resulting toolpath.

10. Save the manufacturing model.

11. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the different types of surface milling sequence.
- Describe the key surface milling manufacturing parameters.
- Describe methods for creating approach and exit moves.
- Create and refine surface milling sequences.
- Create mill surfaces relevant to surface milling.
- Create material removal features specific to surface milling.

Module

12

Creating Trajectory Milling Sequences

Introduction

Trajectory milling allows you to sweep a tool along any user-defined trajectory. During the trajectory milling process, you must configure the trajectory for the tool to follow. Trajectory milling sequences can be used for milling slots where the shape of the tool corresponds to that of the slot, and they can also be used to chamfer edges. You can either use an edited tool, or you can sketch your own tool for the sequence; this enables you to specify the tool control point anywhere on the tool.

Objectives

After completing this module, you will be able to:

- Describe the trajectory milling process.
- Describe different methods of creating a trajectory.
- Describe how to create a sketched tool.
- Describe the features of multi-step and multi-pass trajectory milling.
- Create trajectory milling sequences by defining cut motions.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

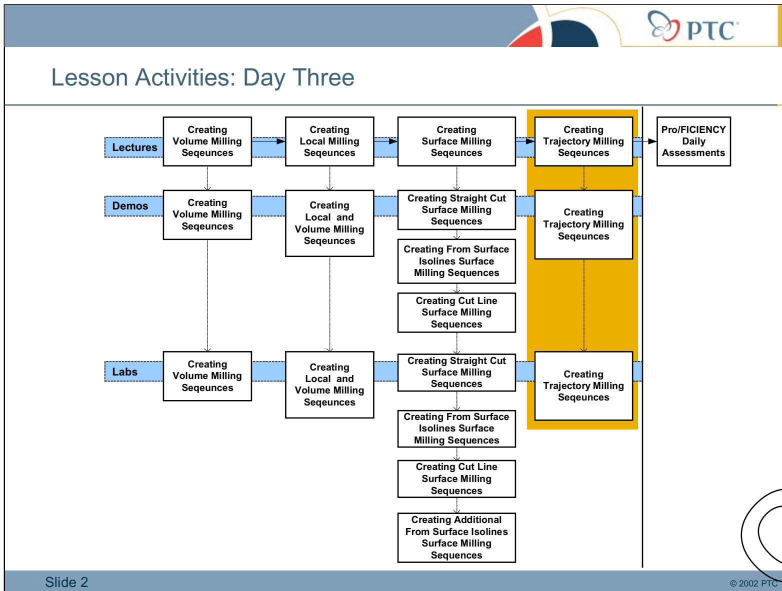
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

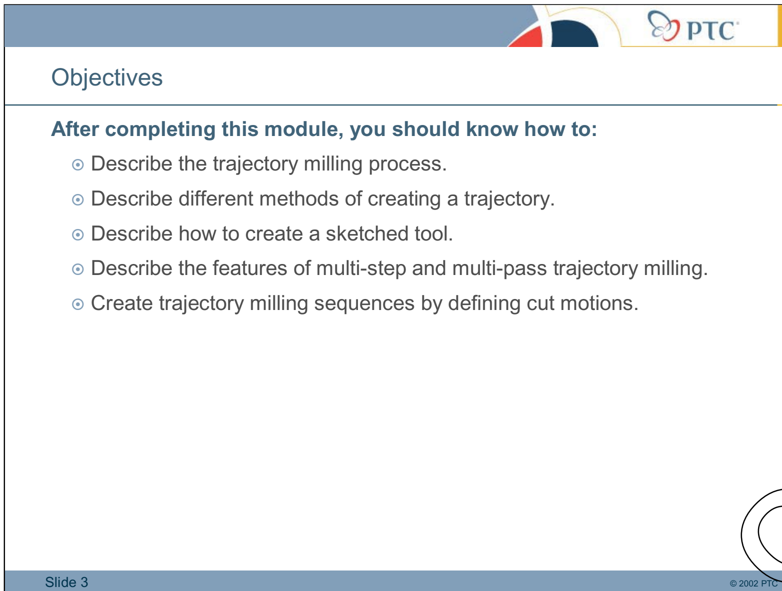
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 10 mins
- Demos : 20 mins
- Labs : 30 mins
- Total: 1 hr 00mins

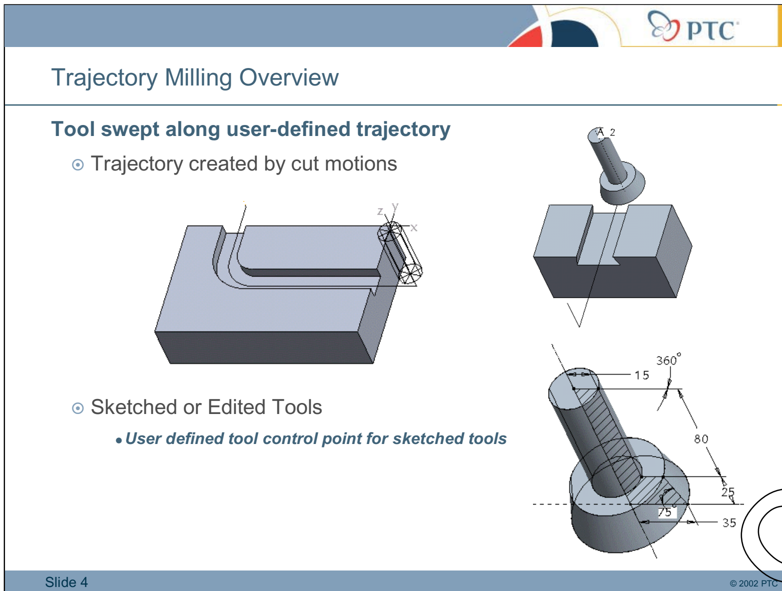


Overview

Trajectory milling allows you to sweep a tool along any user-defined trajectory. During the trajectory milling process you must configure the trajectory for the tool to follow. Trajectory milling sequences can be used for milling slots where the shape of the tool corresponds to that of the slot, they can also be used to chamfer edges. You can either use an edited tool, or you can sketch your own tool for the sequence; this enables you to specify the tool control point anywhere on the tool.

After completing this module, you will be able to:

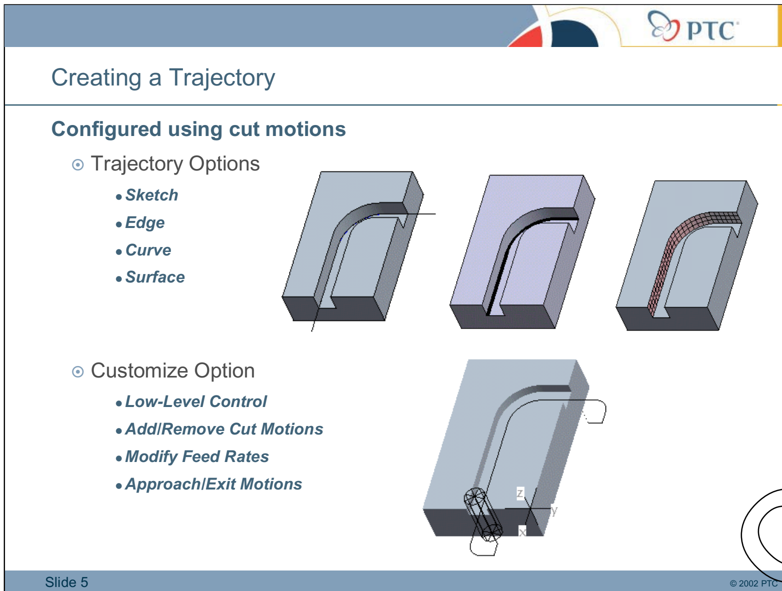
- Describe the trajectory milling process.
- Describe different methods of creating a trajectory.
- Describe how to create a sketched tool.
- Describe the features of multi-step and multi-pass trajectory milling.
- Create trajectory milling sequences by defining cut motions.



Trajectory Milling Process

Trajectory milling allows you to sweep a tool along any user-defined trajectory. It can be used for milling horizontal slots. To configure the tool path, you must specify the trajectory of the control point of the tool by creating cut motions using the customize functionality.

For 3-Axis Trajectory milling, you can either use edited tools, or sketch your own tool for the NC sequence. If you sketch a tool, you can create a user defined control point for the tool.



Creating a Trajectory

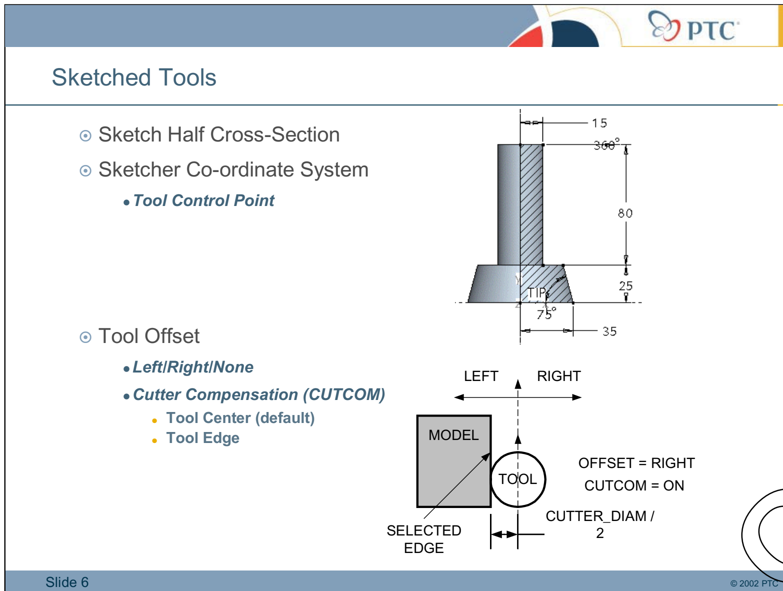
To configure the tool path, you must specify the trajectory of the control point of the tool by creating cut motions using the customize functionality.

Trajectory Options

- Sketch: sketch a trajectory
- Edge: use model edges
- Curve: use datum curves
- Surface: use surfaces (at a specified height)

Within the customize dialog box there are many options to provide more low-level control over the tool path. For example:

- You can add or remove cut motions.
- Modify feed rates.
- Add user defined approach and exit motions.



Sketched Tools

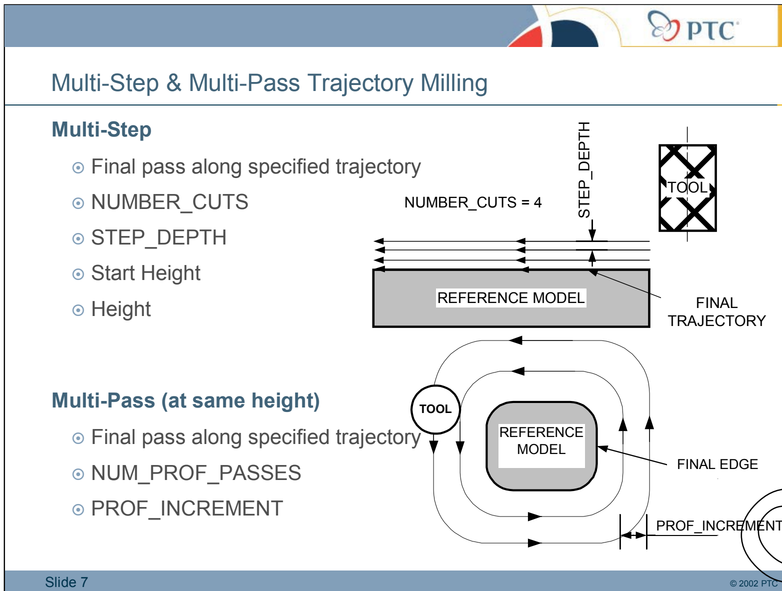
For sketched tools

- The tool is sketched as a revolved protrusion.
- The sketch represents half of the tool cross-section.
- The whole sketch must lie on one side of the axis of symmetry.
- The axis of symmetry must be vertical, with the sketch lying on the right.
- The section must be closed.
- You can specify a control point other than its tip by adding a sketcher coordinate system to the tool sketch. For edited tools, and for sketched tools with no control point specified, the tip of the tool will be used.

Tool Offset

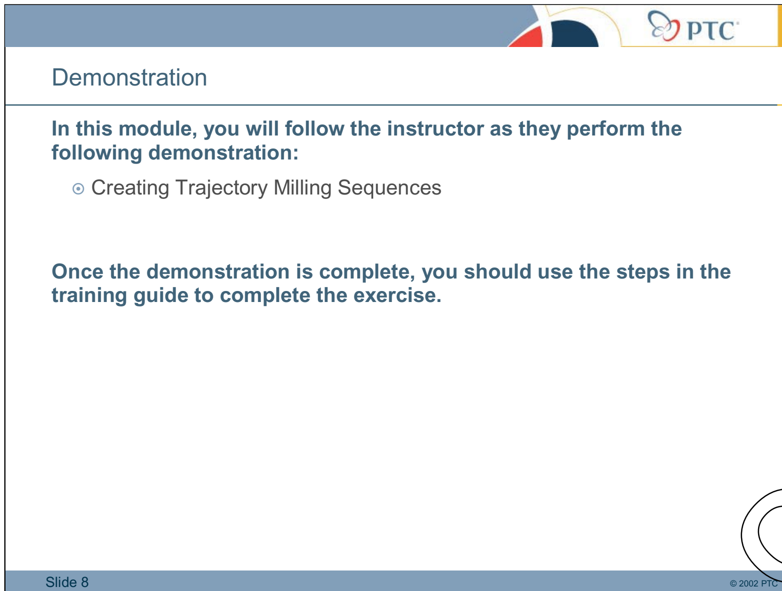
Specifies if the tool control point is offset from the trajectory:

- Can be specified during cut motion creation.
- Can be left/right or none.
- The offset is CUTTER_DIAM/2.
- You can also change cutter compensation from tool center to tool edge when configuring the workcell.



Multi-Step and Mutli-Pass

- Multi-step Trajectory milling can be created by specifying either the desired STEP_DEPTH or the desired NUMBER_CUTS. If both are specified, the higher of the two numbers of cuts will be used. The last pass will coincide with the specified trajectory.
- Start Height: can be used to specify an alternative start height to the top of workpiece.
- Height: can be used to specify alternative final height of tool path.
- Multi-pass Trajectory milling (a succession of trajectory passes with horizontal offset) can be performed by using the NUM_PROF_PASSES and PROF_INCREMENT parameters. The last pass will coincide with the specified trajectory.



Demonstration: Creating Trajectory Milling Sequences

Scenario

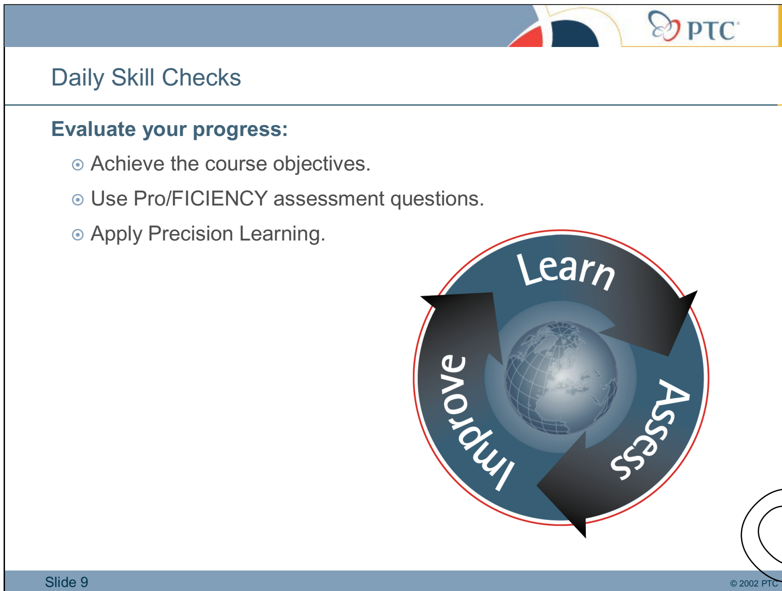
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY.
- **Improve:** The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Getting Started

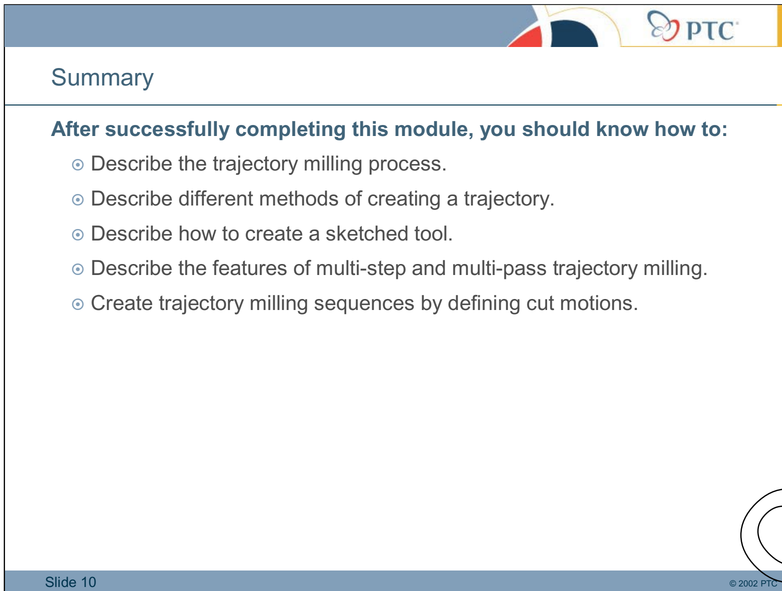
- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Tests

Description: For each course, 5 new 10 question exams based upon the topics covered each day.

How to use it:

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After successfully completing this module, you should know how to:

- Describe the trajectory milling process.
- Describe different methods of creating a trajectory.
- Describe how to create a sketched tool.
- Describe the features of multi-step and multi-pass trajectory milling.
- Create trajectory milling sequences by defining cut motions.

Module 12 Lab Exercises

Exercise 1: Creating Trajectory Milling Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create a trajectory milling sequence to machine the groove in the top portion of the model. You modify the tool path by adding and removing cut motions to provide a ramped approach into the slot. Trajectory milling sequences can be used for milling slots where the shape of the tool corresponds to that of the slot, and they can also be used to chamfer edges.

Objectives

After successfully completing this exercise, you will know how to:

- Create trajectory milling sequences by defining cut motions.

Scenario

You are now ready to continue machining the aluminum cover. You create a trajectory milling sequence to machine the groove in the top portion of the model. You modify the tool path by adding and removing cut motions to provide a ramped approach into the slot.

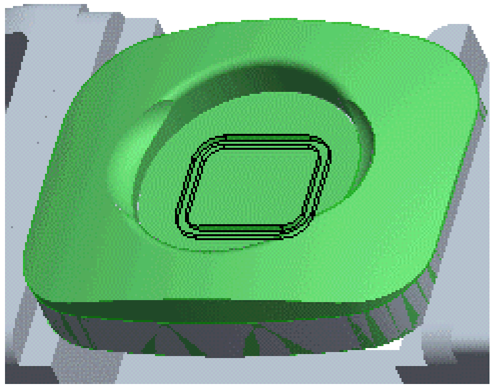


Figure 1: Showing groove to be machined.

Parameters

The following are key trajectory milling parameters.

Table 1: Parameter Descriptions

Parameter	Description
CUTCOM	Turns cutter compensation on or off in the CL data file.
NUMBER_CUTS	Calculate the step depth using number of cuts.
STEP_DEPTH	The incremental depth of each cut.
NUM_PROF_PASSES	Number of profile passes at each depth.
PROF_INCREMENT	Horizontal distance between passes.

Step 1. Locate and open the cover manufacturing model.

Before creating the trajectory milling sequence, we first need to locate the manufacturing model and open it using the folder navigator. Once we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module12**, right-click, and select **Make Working Directory**.
2. Open COVER.MFG. Turn off any displayed datum features.

Step 2. Create a trajectory milling NC Sequence to machine the groove in the top of the cover.

Using the Menu Manager, we can create a trajectory milling sequence. Notice how the Customize dialog box appears immediately. This is because a tool, site parameter file, machine co-ordinate system, and retract surface have already been specified for this operation. We can cancel out of the Customize dialog box, so we can specify a tool and name the sequence. We can select the *name, and tool options. We can enter a name for the sequence to make it easy to identify, and change the tool to a smaller diameter end mill wide enough to machine the slot.*

We now need to specify the trajectory we want the tool to follow. Using the Customize dialog box, we can insert an automatic cut motion, use the edge and tangent chain options and select the circular edge in the top-left corner of the groove, ensuring we pick the edge at the bottom of the groove. We can leave the cut direction pointing to the left; we can also specify the tool offset to the left, completing the cut motion. We can see this creates a toolpath that machines the groove in an anti-clockwise direction. Notice how the tool makes a plunge move at the start of the cut motion. This can be modified by removing and adding specific cut motions.

1. Using the Menu Manager, create a **Trajectory** milling sequence. **Cancel** out of the Customize dialog box to specify a different tool, and name the sequence first.

> **Note:**
> Notice how the Customize dialog box appears immediately. This is because a tool, site parameter file, machine co-ordinate system, and retract surface have already been specified for this operation.

2. Select **Seq Setup**, then select the **Name** and **Tool** check boxes.
3. Name the sequence [**FIN_GROOVE**].
4. Change the tool to the **FEM_03_0** tool.
5. Zoom into the area with the groove feature on the model.
6. To create a cut motion select **Customize**, and in the Customize dialog box, **Insert** an Automatic Cut motion.
7. Select **Edge**, accept the Edge Fit options, and using **Tangent Chain** select the circular edge at the bottom of the groove, as shown in the following figure.

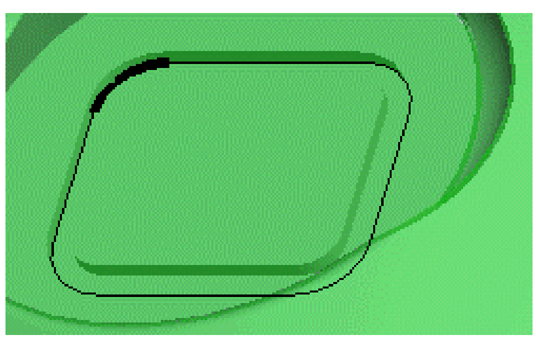


Figure 2: Showing selected edge and tangent chain of edges highlighted.

8. Leave the cut direction arrow pointing to the left, and select **Left** for the tool offset.

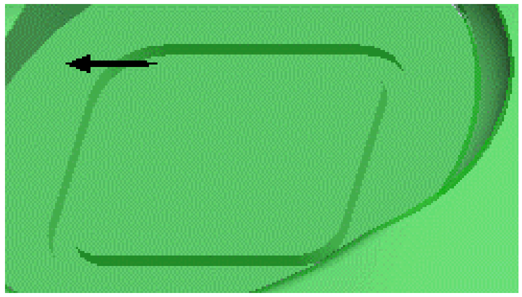


Figure 3: Showing the cut direction arrow pointing to the left.

9. Complete the cut motion, and close the Customize dialog box.

10. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

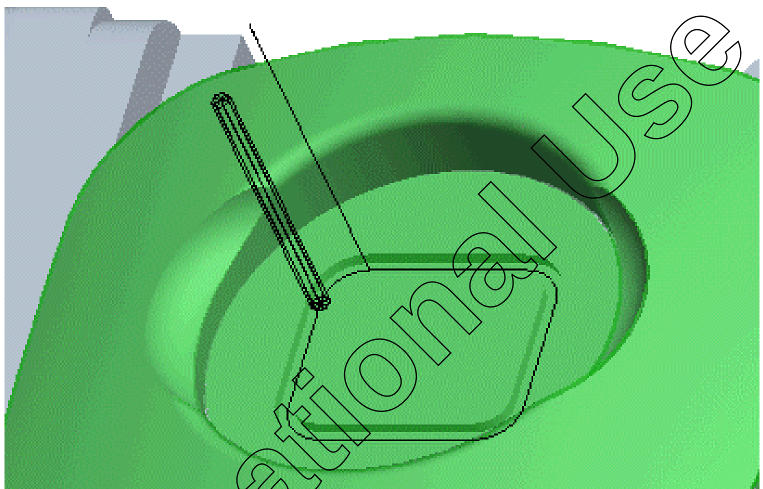


Figure 4: Showing the plunge move at the start of cut motion.

Tool Path Review:

The tool machines the groove but makes a plunge move at the start of the cut motion. This can be modified by removing and adding specific cut motions.

Step 3. Delete the auto plunge cut motion and create a ramp approach cut motion and a normal exit motion.

We can now modify the tool path to give the desired motion. Using the Customize dialog box, we can delete the auto plunge cut motion, and then insert a Go to point in front of the automatic cut motion. To create a ramp move, we need to offset this point by 10 in the X-increment and 5 in the Z-increment. We can also modify the retract move at the end of the tool path, select end of tool path in the dialog box, insert a Go delta motion, and specify a distance of 5 in the Z-increment direction. Completing the customization, we can review the tool path, and notice how the tool makes a ramp approach move and a normal exit move before moving to the retract plane.

1. Select **Customize** to open the Customize dialog box, select the **Auto Plunge** cut motion, and **Delete** it.

2. Select the **Automatic Cut** motion in the Customize dialog box, and insert a **Goto Point** motion from the drop-down list.

3. In the Goto Point dialog box, click the **Specify Point** button, using **Create** and **On Toolpath** options select a point at the start of the toolpath, as shown in the following figure.

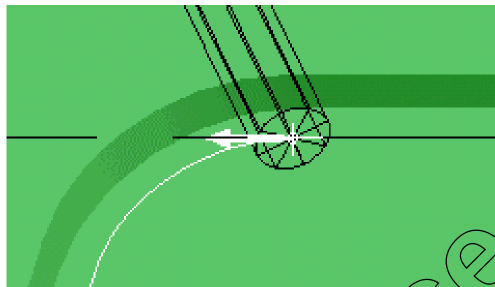


Figure 5: Showing selected goto point at the start of tool path.

4. Click the **Specify Offset** button, and type [**10**] *(0.39 in)* in the X-increment text box and [**5**] *(0.20 in)* in the Z-increment text box. Close the Offset and Goto Point dialog boxes.

> **Note:**
> We have created the first Goto point, we now need to ramp into the start of the automatic cut motion by creating a second Goto point at the start of the cut motion.

5. Select the **Automatic Cut** motion in the Customize dialog box, and insert another **Goto Point** motion from the drop-down list.

6. In the Goto Point dialog box, click the **Specify Point** button, using **Create** and **On Toolpath** options, select the same point at the start of the toolpath, as shown in the previous figure.

7. Click the **Feed** button, and change the feed rate from FREE to **CUT**, close the Feed Parameters dialog box and the Goto Point dialog box.

> **Note:**
> We now need to create an exit move to move clear of the groove at cutting feed rate.

8. Select the **end of toolpath** in the Customize dialog box and insert a **Go Delta** motion from the drop-down list, type [**5**] *(0.20 in)* in the Z-increment text box. Close the Goto Delta dialog box.

9. Close the Customize dialog box.

10. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

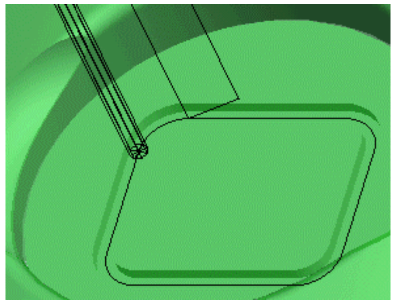


Figure 6: Showing ramp approach and normal exit motion.

Tool Path Review:

Notice how the tool makes a ramp approach move and a normal exit move before moving to the retract plane.

11. Save the manufacturing model.

12. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the trajectory milling process.
- Describe different methods of creating a trajectory.
- Describe how to create a sketched tool.
- Describe the features of multi-step and multi-pass trajectory milling.
- Create trajectory milling sequences by defining cut motions.

Module

13

Creating Holemaking Sequences

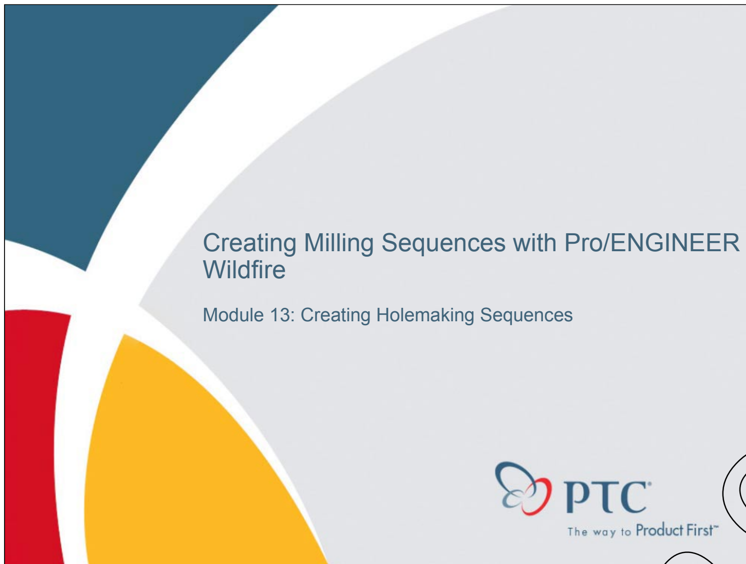
Introduction

Creating holemaking sequences enables you to create many different types of drilling cycles for machining. During the holemaking process, you need to select the appropriate cycle type; these include standard drilling, boring, tapping, reaming and countersink drill cycles. It is therefore important to understand when each type of drill cycle should be used. Another important stage in the holemaking process is creating hole sets that define the holes to be drilled. If you need to perform a series of holemaking sequences on the same set of holes, you can configure a drill group that simplifies the selection process. Also any holemaking sequences that reference a drill group will update if a drill group is modified.

Objectives

After completing this module, you will be able to:

- Describe the holemaking process.
- Describe the key holemaking manufacturing parameters.
- Describe different drill cycle types, and hole selection methods.
- Create and refine holemaking sequences.
- Create drill groups relevant to holemaking.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

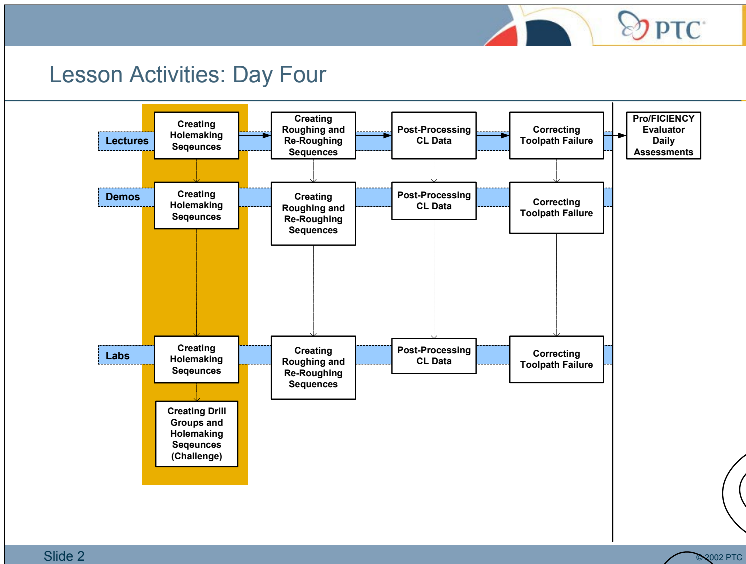
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

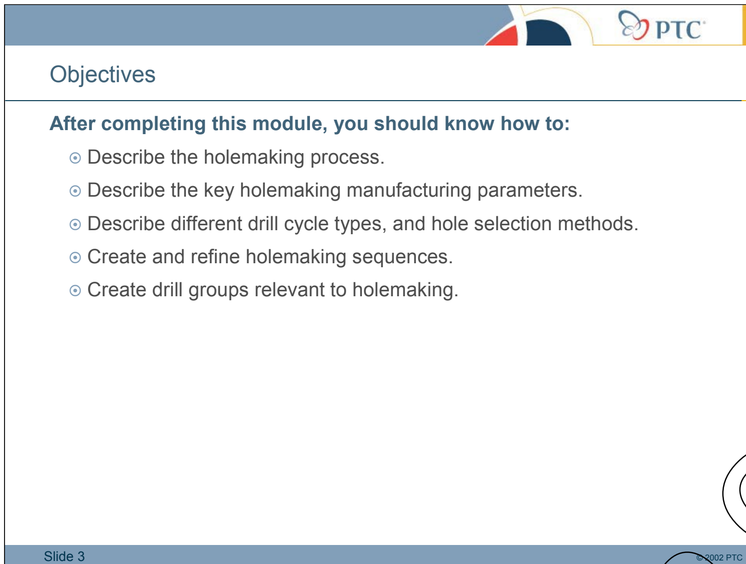
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos (1) : 20 mins
- Labs (2) : 55 mins
- Total: 1 hr 30 mins

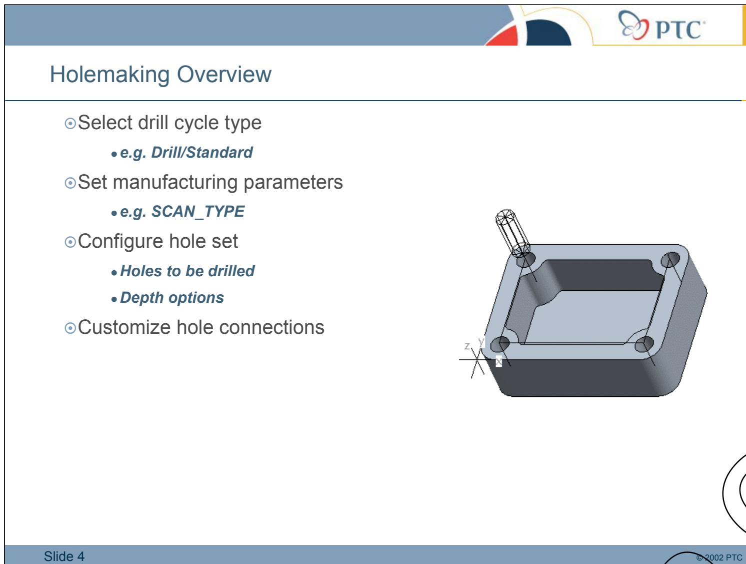


Overview

Creating holemaking sequences enables you to create many different types of drilling cycle for machining. During the holemaking process you need to select the appropriate cycle type; these include standard drilling, boring, tapping, reaming and countersink drill cycles. It is therefore important to understand when each type of drill cycle should be used. Another important stage in the holemaking process is creating hole sets that define the holes to be drilled. If you need to perform a series of holemaking sequences on the same set of holes, you can configure a drill group that simplifies the selection process. Also any holemaking sequences that reference a drill group will update if a drill group is modified.

After completing this module, you will be able to:

- Describe the holemaking process.
- Describe the key holemaking manufacturing parameters.
- Describe different drill cycle types, and hole selection methods.
- Create and refine holemaking sequences.
- Create drill groups relevant to holemaking.



Holemaking Process

You can create Holemaking sequences by:

- Selecting the appropriate holemaking cycle type, e.g. drilling or reaming
- Setting manufacturing parameters for feeds and speeds.
- Configuring the set of holes to be machined, by selecting the holes and specifying depth options.
- Specify the method for connecting the holes.

Cycle Types

Output different CYCLE/XXXX statements

- Drill
  - *Standard*
  - *Deep*
  - *Break Chip*
  - *Web*
  - *Back*
- Face
- Bore
- Countersink
- Tap
  - *Fixed/Floating*
- Ream
- Custom

TOOLS	CYCLE TYPES					
	DRILL	FACE	BORE	COUNTERSINK	TAP	REAM
DRILL	✓	✓	✓	✓		✓
COUNTERSINK	✓	✓	✓	✓		✓
TAP					✓	
REAM	✓	✓	✓			✓
BORE	✓	✓	✓			✓
CENTER-DRILL	✓	✓	✓	✓		✓
BACKSPOTTING	✓			✓		
END MILL	✓	✓	✓			✓

Drill Cycle Types

Depending on the cycle type selected a different CYCLE/XXXX statement is output in the CL data file

For Reference only, describe as required:

· **Drill** – Drill a hole. Depending on the additional option selected, the following statement will be output to the CL file:

· **Standard** (default) – CYCLE/DRILL

· **Deep** – CYCLE/DEEP

· **Break Chip** – CYCLE/BRKCHP

· **Web** – CYCLE/THRU (for multiple plates)

· **Back** – GOTO and SPINDLE statements for back spotting

· **Face** – Drill a hole with an optional dwell at final depth to help assure a clean surface at the bottom of the hole. The CYCLE/FACE statement will be output to the CL file.

· **Bore** – Bore a hole to create a finish hole diameter with high precision. The CYCLE/BORE statement will be output to the CL file.

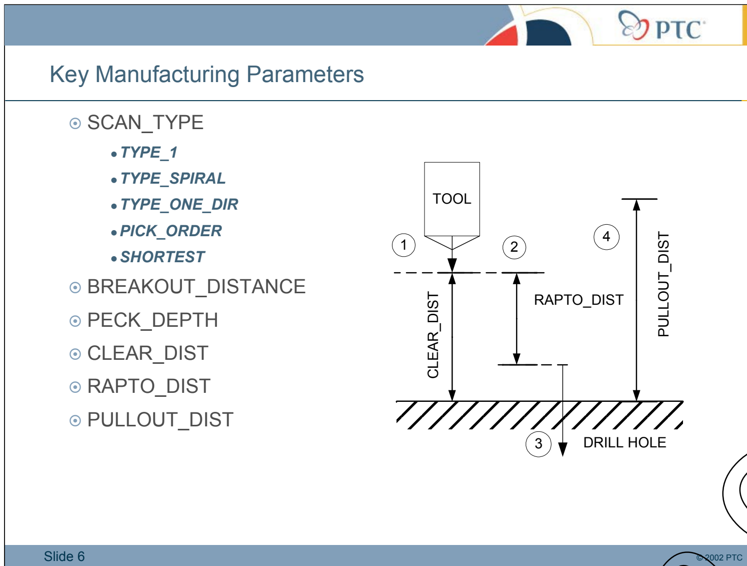
· **Countersink** – Drill a chamfer for a countersunk screw. The CYCLE/CSINK statement will be output to the CL file. If the Back option is selected together with Countersink, the system will perform back countersinking.

· **Tap** – Drill a threaded hole. Pro/NC supports ISO standard thread output. The CYCLE/TAP statement will be output to the CL file. Two additional options are available:

- Fixed – The feed rate is determined by the combination of thread pitch and spindle speed.
- Floating – Allows you to modify the feed rate using the parameter FLOAT_TAP_FACTOR.

· **Ream** – Create a precision finish hole. The CYCLE/REAM statement will be output to the CL file.

Custom – Create and use your own customized cycles for the current machine tool.



Manufacturing Parameters

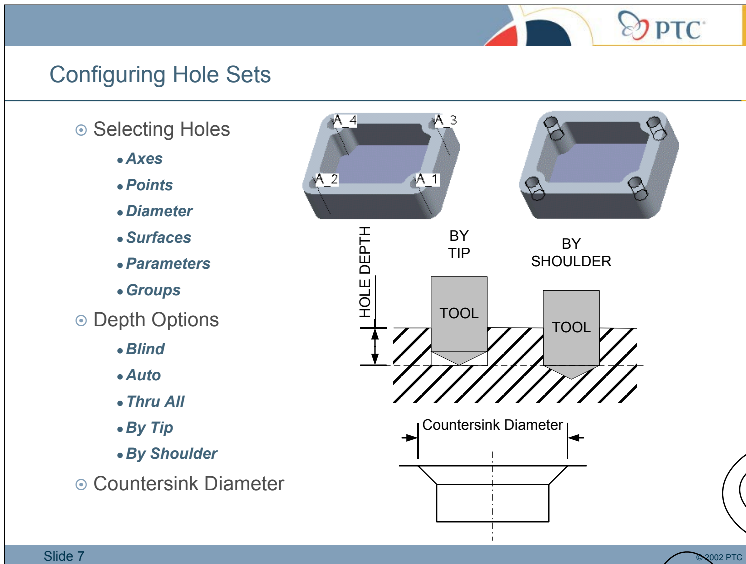
- SCAN_TYPE: Calculates the order for drilling selected holes

(descriptions for reference only)

  - TYPE_1—By incrementing the Y coordinate and going back and forth in the X direction.
  - TYPE_SPIRAL—Clockwise starting from the hole nearest to the coordinate system.
  - TYPE_ONE_DIR—By incrementing the X coordinate and decrementing the Y.
  - PICK_ORDER—The holes will be drilled in the same order as they are selected.
  - SHORTEST (default)—The system determines which order of holes results in the shortest machine motion time.

- BREAKOUT_DISTANCE: Adds the BREAKOUT_DISTANCE value to the Z depth in the CYCLE statements associated with holes drilled Thru All, and with through holes drilled using the Auto depth option.
- PECK_DEPTH: Depth increment for each drilling pass. Default value is 0. If you select DEEP drilling, you have to specify non-zero PECK_DEPTH.

- CLEAR_DIST: The clearance distance above the top of the hole at which the PLUNGE_FEED ends and the CUT_FEED begins.
- RAPTO_DIST: Allows for further rapid advance from CLEAR_DIST towards the top of the hole.
- PULLOUT_DIST: Allows for the tool to return to a point other than that defined by CLEAR_DIST.



Configuring Hole Sets

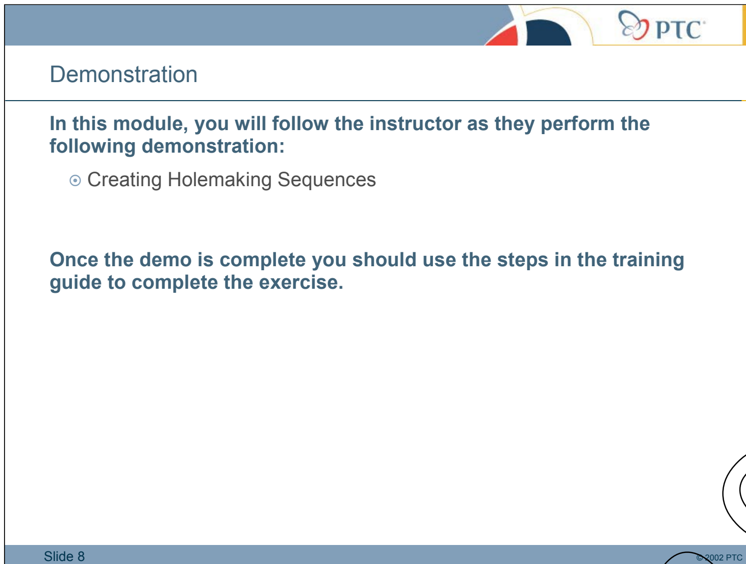
Configuring hole sets involves a number of steps

Selecting Holes: Specify the holes to be drilled using any combination of methods listed below:

- Axes—Specify holes by selecting individual hole axes.
- Points—Specify drill locations by selecting datum points or reading in a file with datum point coordinates.
- Diameters—Specify holes by entering diameter value(s).
- Surfaces—Specify holes by selecting surfaces of the reference part or workpiece.
- Parameters—Select holes with a certain parameter value.
- Groups—Select predefined drill groups. You can define groups of holes at setup time for later use in Holemaking NC sequences. Simplifying later hole selection.

Depth Options

- Blind—Drill from the start surface (or Z height) to specified depth. Specify the Start and End surfaces (by either selecting a surface or entering a Z depth).
- Auto—Depth of drilling is determined automatically, by referencing hole geometry.
- Thru All—Drill a through hole, from the retract surface all the way through the workpiece(s) or reference part(s) that the hole intersects.
- The Tool Depth option buttons, Shoulder and Tip, available for Blind and Auto drilling, determine if the drilling depth will be with reference to the shoulder or the tip of the tool.
- Depth for countersink drilling is defined by the start surface and the countersink diameter value, entered at the time of configuring the Hole Set. Instead of entering a countersink diameter, you can specify that the system automatically finds all the applicable chamfers and makes the necessary calculations. Countersink diameter is the final diameter of the hole after drilling, measured in the start surface.



Demonstration: Creating Holemaking Sequences

Scenario

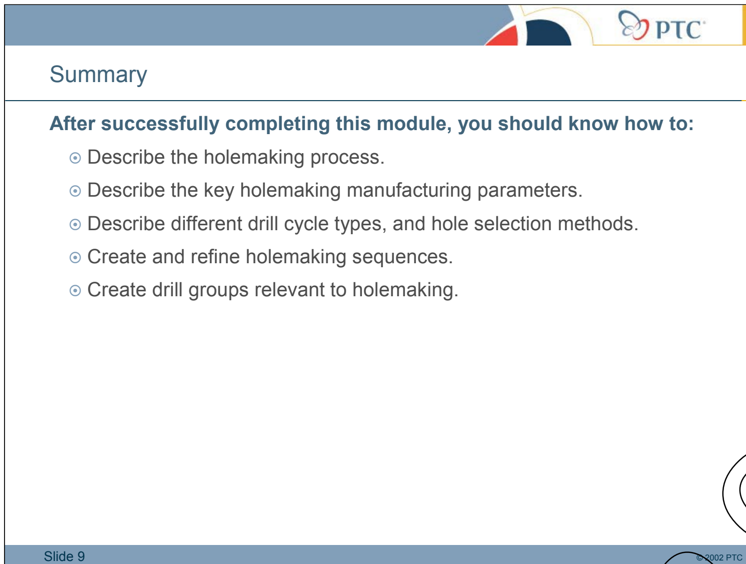
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercises (2)

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the holemaking process.
- Describe the key holemaking manufacturing parameters.
- Describe different drill cycle types, and hole selection methods.
- Create and refine holemaking sequences.
- Create drill groups relevant to holemaking.

Module 13 Lab Exercises

Exercise 1: Creating Holemaking Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you create holemaking sequences to machine the holes in the bottom of the cover model. You determine the hole size to enable correct selection of drilling tools. You also insert the holemaking sequences before the operation 020 sequences to ensure they are created in the correct order.

Objectives

After successfully completing this exercise, you will know how to:

- Identify and check model geometry and dimensions.
- Create holemaking sequences.

Scenario

The cover model has been completely machined except for the holes in the bottom of the model; these should have been drilled during operation 010. To create the holemaking sequences at the end of operation 010, you insert the sequences before the operation 020 sequences so ensuring they are created in the correct order. Before machining the holes, you need to identify and check the hole geometry and dimensions, enabling you to select the correct drilling tools.

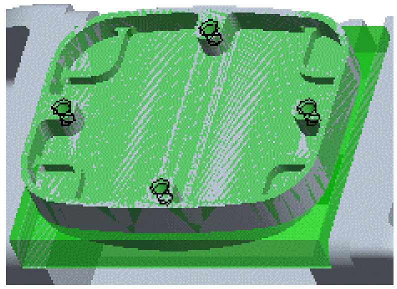


Figure 1: Showing holes to be machined.

Parameters

The following are key holemaking parameters.

Table 1: Parameter Descriptions

Parameter	Description
SCAN_TYPE	Calculates the order for drilling selected holes.
BREAKOUT_DISTANCE	Adds the value to the Z depth in the CYCLE statements for holes drilled Thru All, and through holes drilled using the Auto depth option.
PECK_DEPTH	Depth increment for each drilling pass. Default value is 0.
CLEAR_DIST	The clearance distance above the top of the hole at which the PLUNGE_FEED ends and the CUT_FEED begins.
RAPTO_DIST	Distance of further rapid advance from CLEAR_DIST towards the top of the hole.
PULLOUT_DIST	Distance the tool to returns to other than that defined by CLEAR_DIST.

Step 1. Locate and open the cover manufacturing model.

Before creating the holemaking sequences, we need to locate the manufacturing model and open it using the folder navigator. When we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module13**, right-click, and select **Make Working Directory**.
2. Open COVER.MFG.
3. Turn off any displayed datum features.

Step 2. Activate operation op010, and prepare to insert NC sequences before operation op020.

In the Operation dialog box, we can change the active operation to op010. To see the model more easily, we can change to the op010-tri view. Notice how the material removal features in op020 have removed the material from the other side of the cover. We can now activate insert mode after the last material removal feature for operation op010. Notice the Insert Mode text in *the bottom right corner of the screen, and the operation 020 material removal features have been suppressed along with all the operation 020 NC sequences. We are now ready to create the holemaking sequences.*

1. Using the Menu Manager, select **Mfg setup > Operation**, to open the Operation Setup dialog box. In the Operation Name drop-down list, click **op010** to change the active operation to op010. Close the Operation Setup dialog box, and return to the Manufacture menu.

2. Change the view to the named view **OP010-TRI**.

> **Note:**
> Notice how the material removal features in op020 have removed the material from the other side of the cover.

3. Check the model tree and note the position of the OP_020 setup, as shown in the following figure.

4. Using the Menu Manager, select **Machining > Utilities > Insert > Activate** to activate insert mode.

5. In the model tree, select the last material removal feature for operation op010, as shown.

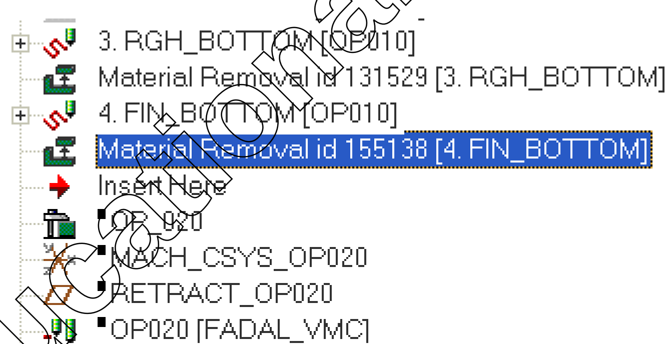


Figure 2: Showing insert mode activated after last material removal feature.

> **Note:**
> Notice the Insert Mode text in the bottom right corner of the screen. Notice also the operation 020 material removal features have been suppressed along with all the operation 020 NC sequences.

Step 3. Identify and check the hole dimensions.

Before creating the holemaking sequences, we need to check the hole dimensions. We can select and activate the cover part in the model tree; we can then select a hole and note its dimensions.

1. In the model tree, expand the listing for COVER_NC.ASM, then select COVER.PRT, right-click and **Activate** the part in the model tree.

2. Select one of the hole features to display the hole dimensions. Note down the hole dimensions, and return to the Manufacture menu.

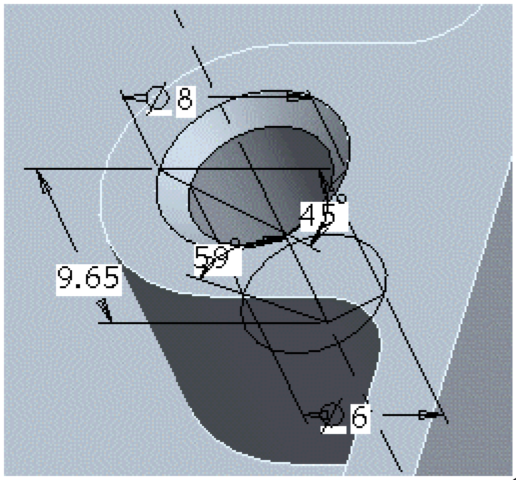


Figure 3: Showing selected hole and dimensions.

Note:

You should see that the hole diameter is 6.0 *(0.24 in)*, the depth is 9.65 *(0.38 in)*, and the countersink diameter is 8.0 *(0.31 in)* with a 45 degree chamfer angle. You need to center drill, drill and countersink the four holes.

Step 4. Create a holemaking NC sequence to center drill the four holes.

Using the Menu Manager we can create a holemaking sequence. We can specify the drill cycle type as standard. We can select the name, and tool options. Enter a name for the sequence to make it easy to identify, and change the tool to a center drill.

We can specify the CUT_FEED, SPINDLE_SPEED, and CLEAR_DIST parameters. Notice how the mill site parameter file does not set default values for a holemaking sequence, even though it is still active.

Using the Hole Set dialog box, we can use the Diameters tab and select the 6.0 diameter holes, this automatically selects all the 6.0 millimeter diameter holes on the model. As this is a center drilling sequence, we can specify the depth options as blind and tip and specify a depth of 5.0.

By playing the tool path and making the CL data visible, we can see the tool drills the four holes to a depth of 5.0 as specified. Notice the CYCLE/DRILL statement in the CL data file. You can use the SCAN_TYPE parameter to adjust the order and movement between holes; it is not necessary in this case.

1. Using the Menu Manager, create a **Holemaking** sequence; use the **Drill** and **Standard** drilling cycle options.

2. Select the **Name** and **Tool** check boxes.

3. Name the sequence [**CTR_DRILL**].

4. Change the tool to the **CTRDRILL_04** drill.

5. Modify the following parameters in the NC sequence:

Table 2: Changed Parameters

Manufacturing Parameter	Value
CUT_FEED	130 *(5.12 in)*
SPINDLE_SPEED	500
CLEAR_DIST	3 *(0.12 in)*

> **Note:**
> Notice how the mill site parameter file does not set default values for a holemaking sequence, even though it is still active.

6. In the Hole Set dialog box, use the **Diameters** tab, and click the **Add** button, select the **6.0** *(0.24 in)* diameter holes. Close the Hole Diameter dialog box, and **Preview** the selection.

7. Click the **Depth** button, then click the **Blind** and **Tip** options. In the Start Surface area, click the **Select** button and select the top surface of the workpiece as the start surface

8. In the End Surface area click the **Z Depth** option and type a depth of [**5.0**] *(0.20 in).* Close the Hole Set Depth dialog box.

9. Complete the hole set configuration.

10. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

11. Complete the sequence.

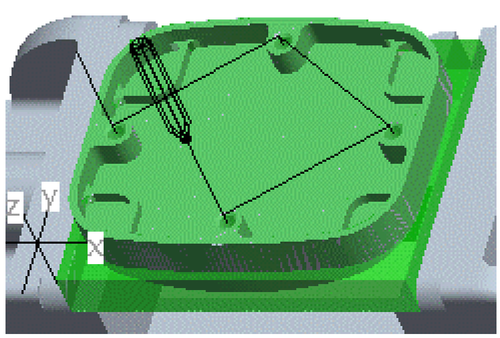


Figure 4: Showing the center drilling tool path

Tool Path Review:

The tool drills the four holes to a depth of 5.0 *(0.20 in)* as specified; notice the CYCLE /DRILL statement in the CL data file. You can use the SCAN_TYPE parameter to adjust the order and movement between holes, but it is not necessary in this case.

Step 5. Create a holemaking NC sequence to drill the four holes with a six-millimeter *(0.24 in)* diameter drill.

Using the Menu Manager, we can create a holemaking sequence. We can specify the drill cycle type as standard, select the name, and tool options. Enter a name for the sequence to make it easy to identify, and change the tool to a 6 millimeter drill. We can use the previous sequence manufacturing parameters.

Using the Hole Set dialog box, we can use the Use Prev option and select the first hole set. As this is a standard drilling sequence, we can specify the depth options as auto and tip, thus completing the hole set configuration.

By playing the tool path and making the CL data visible, we can see the tool drills the four holes, automatically calculating the correct depth. Notice the CYCLE /DRILL statement in the CL data file.

1. Using the Menu Manager, create a **Holemaking** sequence; use the **Drill** and **Standard**, drilling cycle options.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**DRILL_06**].
4. Change the tool to the **DRILL_06_0** drill.
5. Use the manufacturing parameters from the previous **CTRDRILL_04** sequence.
6. In the Hole Set dialog box, click the **Use Prev** button and select **Hole Set #1**.
7. Click on the **Depth** button and change the depth options to **Auto**, and **Shoulder**.
8. Complete the hole set configuration.
9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.
10. Complete the sequence.

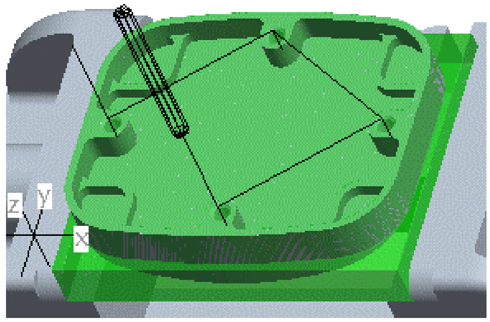


Figure 5: Showing the resultant tool path.

Tool Path Review:
In the CYCLE/DRILL statement the depth is set to drill to the bottom of the holes.

Step 6. Create a holemaking NC sequence to drill the four holes with a countersink drill.

Using the Menu Manager, we can create a holemaking sequence. We can specify the drill cycle type as countersink. We can select the name, and tool options. Enter a name for the sequence to make it easy to identify, and change the tool to a countersink drill. We can use the previous sequence manufacturing parameters.

Using the Hole Set dialog box, we can select the auto chamfer option, this automatically finds the holes on the model to chamfer, and calculates the countersink diameter. This completes the hole set configuration.

By playing the tool path and making the CL data visible, we can see the tool countersink drills the four holes. Notice in the CL data file the CYCLE/CSINK statement, where the diameter is automatically calculated correctly at 8.0.

1. Using the Menu Manager, create a **Holemaking** sequence; use the **Countersink** drilling cycle.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**CSINK**].
4. Change the tool to the **CSINK_12** drill.
5. Use the manufacturing parameters from the previous **DRILL_06** sequence.
6. In the Hole Set dialog box, select the **Auto Chamfer** check box.

Note:
Selecting Auto Chamfer automatically finds the holes on the model to chamfer, and calculates the countersink diameter.

7. Complete the hole set configuration.

8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

9. Complete the sequence, and return to the Manufacture menu.

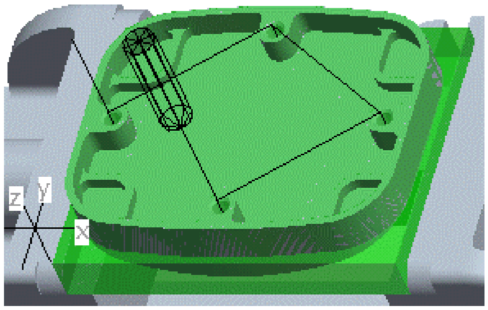


Figure 6: Showing the resultant tool path.

> **Tool Path Review:**
> The tool countersink drills the four holes as specified. Notice in the CYCLE/CSINK statement the diameter is automatically calculated correctly at 8.0 *(0.31 in)*.

10. Cancel insert mode; using the Menu Manager, select **Machining > Utilities > Insert > Cancel**, when prompted, click **Yes** to resume the operation 020 features. Return to the Manufacture menu.

> **Note:**
> Notice all the operation 020 features are resumed in the model tree.

11. Save the manufacturing model.

12. Close all windows and erase all components from memory.

Exercise 2: Creating Drill Groups and Holemaking Sequences (Challenge)

Objectives

After successfully completing this exercise, you will know how to:

- Create holemaking sequences.
- Create drill groups.

Scenario

In this exercise, you have a special request to machine holes in a new machining fixture that is required for another job. You create drill groups to enable easy selection of the holes for the holemaking sequences. You center-drill all the holes in the model. You create a holemaking sequence to drill the seven holes on the stepped face of the model. You then drill and tap the four holes in the bolt hole pattern on the top face of the model, and finish by drilling and boring the large hole on the top face.

Step 1. Locate and open the bracket manufacturing model.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module13\bracket**, right-click, and select **Make Working Directory.**
2. Open BRACKET.MFG.
3. Turn off any displayed datum features.

Step 2. Identify and check all the hole dimensions on the bracket part.

1. Select and activate the BRACKET.PRT part in the model tree.
2. Select one of the seven holes on the stepped face of the model. Note the hole dimensions.
3. Select one of the four holes in the bolt hole pattern on the top face of the model. Note the hole dimensions.

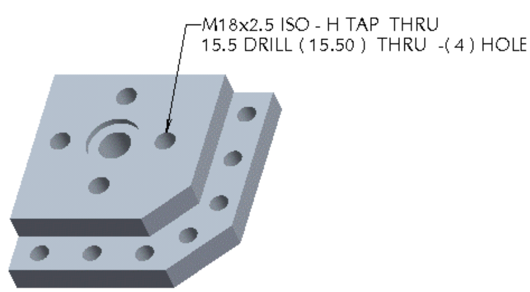


Figure 7: Showing the bracket part and 3D note.

> **Note:**
> Note the dimensions of the M18 tapped holes are also displayed in a 3D note.

4. Determine the hole and counter bore diameter of the large hole on the top face of the model.

5. Turn off the display of 3D notes.

Step 3. Create two drill groups, one for the seven holes on the stepped face and the other for the holes in the bolt hole pattern.

1. Using the Menu Manager, create a drill group of the seven holes on the stepped face of the model, select the holes by diameter size; name it [**group_14_0**].

2. Using the Menu Manager, create a drill group of the four holes that make up the bolt hole pattern on the top face of the model, select the holes by diameter size; name it [**group_m18_0**].

Step 4. Create a holemaking sequence to center drill all the holes.

> **Note:**
> An operation has already been setup, this includes a 3-axis milling workcell with tools, a machine zero co-ordinate system, and a retract surface.

1. Using the Menu Manager, create a holemaking sequence; use the **Standard** drilling cycle.

2. Select the **Name** and **Tool** check boxes.

3. Name the sequence [**CTR_DRILL**].

4. Change the tool to the **CTRDRILL_04** drill.

5. Modify the following parameters in the NC sequence:

Table 3: Changed Parameters

Manufacturing Parameter	Value
CUT_FEED	130 *(5.12 in)*
SPINDLE_SPEED	500
CLEAR_DIST	3 *(0.12 in)*

6. In the Hole Set dialog box, use the Groups tab to add the **group_M18_0** holes to the hole set.

7. Select the axis tab and select the large hole in the center of the top face of the model.

8. Set the depth options to **Blind** and **Tip**; select the top surface of the model as the start surface and specify an end surface Z-depth of [**4.0**] *(0.16 in)*. **Preview** the hole set.

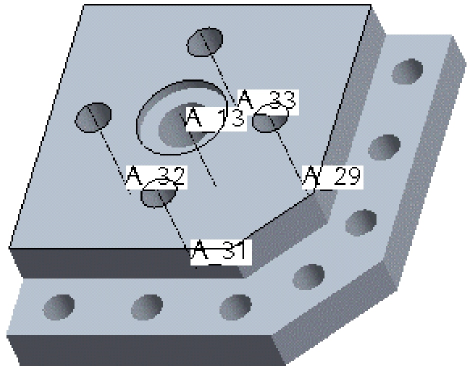


Figure 8: Showing the first hole set and starting surface.

9. Complete the hole set configuration.

10. Create a second hole set. In the Hole Set dialog box, use the Groups tab to add the **group_14_0** holes to the hole set.

11. Set the depth options to **Blind** and **Tip**; select the stepped surface of the model as the start surface and specify an end surface Z-depth of [**4.0**] *(0.16 in)*.

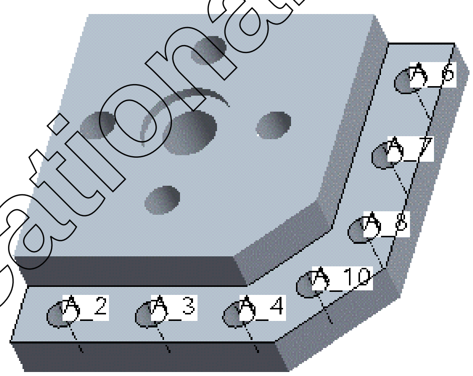


Figure 9: Showing the second hole set and starting surface.

12. Complete the hole set configuration.

13. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

14. Complete the NC sequence.

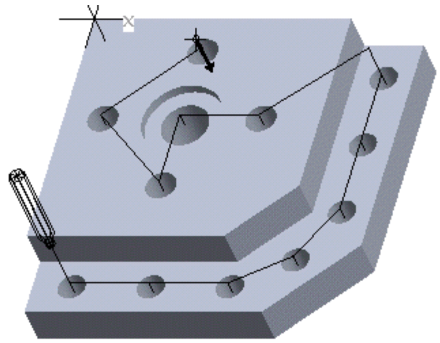


Figure 10: Showing the resulting tool path.

Step 5. Create a holemaking sequence to drill the seven holes on the stepped face of the model, use the fourteen millimeter diameter drill.

1. Using the Menu Manager, create a holemaking sequence; use the **Standard** drilling cycle.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**DRILL_14_0**].
4. Change the tool to the **DRILL_14_0** drill.
5. For the manufacturing parameters, select **Use Prev**, and select the previous sequence parameters, then modify the following parameter.

Table 4: Changed Parameters

Manufacturing Parameter	Value
BREAKOUT_DISTANCE	2.0 *(0.08 in)*.

6. In the Hole Set dialog box use the Groups tab to add the **group_14_0** holes to the hole set.
7. Set the depth option to **Thru All**, and select the bracket as the part to be used for depth calculations. **Preview** the hole set. Complete the hole set configuration.
8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.
9. Complete the NC sequence.

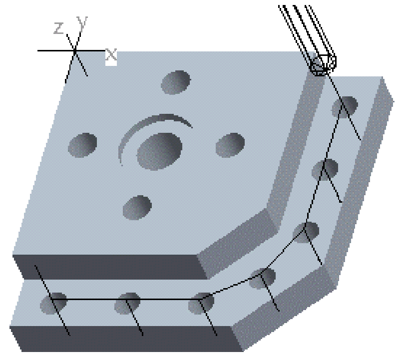


Figure 11: Showing the resulting tool path.

Step 6. Create a holemaking sequence using a tapping drill to drill the four holes in the bolt hole pattern.

1. Using the Menu Manager create a holemaking sequence; use the **Standard**, drilling cycle.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**DRILL_15_5**].
4. Change the tool to the **DRILL_15_5** drill.
5. For the manufacturing parameters, select **Use Prev**, and select the previous **drill_14_0** sequence parameters.
6. In the Hole Set dialog box, use the Groups tab to add the **group_m18_0** holes to the hole set.
7. Set the depth option to **Thru All**, and select the bracket as the part to be used for depth calculations. Complete the hole set configuration.
8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

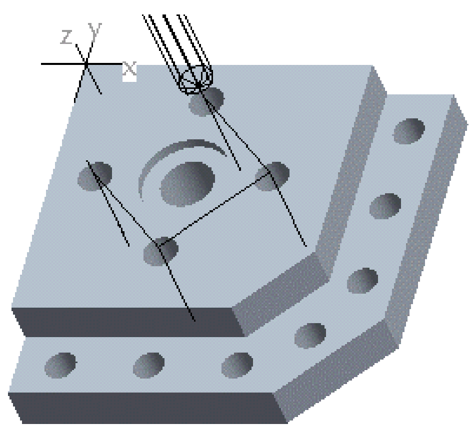


Figure 12: Showing the resulting tool path.

9. Complete the NC sequence.

Step 7. Create a holemaking sequence to tap the four holes in the bolt hole pattern.

1. Using the Menu Manager create a holemaking sequence; use the **Tap**, **Fixed** cycle.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**TAP_18_0**].
4. Change the tool to the **TAP_18_0** tool.
5. For the manufacturing parameters, select **Use Prev**, and select the previous **drill_14_0** sequence parameters.
6. Modify the following manufacturing parameters:

Table 5: Changed Parameters

Manufacturing Parameter	Value
THREAD_FEED	2.5 *(0.10 in)*
THREAD_FEED_UNITS	MMPR
SPINDLE_SPEED	300

7. In the Hole Set dialog box use the groups tab to add the **group_m18_0** holes to the hole set.
8. Set the depth option to **Thru All**, and select the bracket as the part to be used for depth calculations. **Preview** the hole set. Complete the hole set configuration.
9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.
10. Complete the NC sequence.

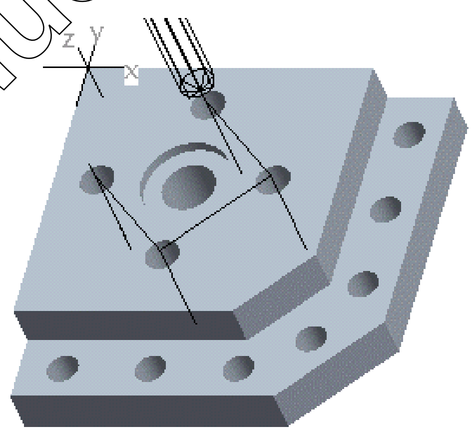


Figure 13: Showing the resulting tool path.

Note:
Notice the CYCLE/TAP statement in the CL data file.

Step 8. Create two holemaking sequences to drill and counter bore the large hole on the top face of the model.

1. Using the Menu Manager, create a holemaking sequence; use the **Standard** drilling cycle.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**DRILL_24_0**].
4. Change the tool to the **DRILL_24_0** drill.
5. For the manufacturing parameters, select **Use Prev**, and select the previous **drill_14_0** sequence parameters.
6. In the Hole Set dialog box, use the Axis tab and add the 24.0 *(0.94 in)* diameter hole to the hole set.
7. Set the depth option to **Thru All**, and select the bracket as the part to be used for depth calculations. Complete the hole set configuration.
8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.
9. Complete the NC sequence.
10. Using the Menu Manager, create a holemaking sequence; use the **Bore** cycle.
11. Select the **Name** and **Tool** check boxes.
12. Name the sequence [**BORE_40_0**].
13. Change the tool to the **BORE_40_0** drill.
14. For the manufacturing parameters, select **Use Prev**, and select the previous **drill_14_0** sequence parameters.
15. In the Hole Set dialog box, use the Axis tab and add the 40.0 *(1.57 in)* diameter hole to the hole set.
16. Set the depth options to **Blind** and **Tip**, select the surface on the top of the model as the start surface and the surface at the bottom of the counter bore as the end surface.
17. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

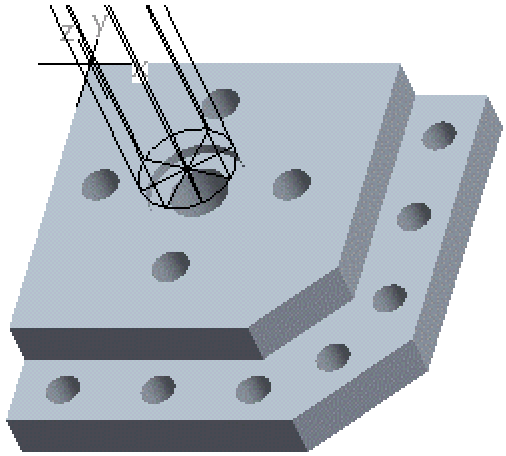


Figure 14: Showing the resulting tool path.

18. Complete the NC sequence.

19. Save the manufacturing model.

20. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the holemaking process.
- Describe the key holemaking manufacturing parameters.
- Describe different drill cycle types, and hole selection methods.
- Create and refine holemaking sequences.
- Create drill groups relevant to holemaking.

Module

14

Creating Roughing and Reroughing Sequences

Introduction

Roughing sequences are similar to volume milling sequences, but are intended specifically for high-speed mold machining, and especially for machining imported, non-solid geometry. Roughing sequences have additional capabilities aimed at producing more efficient tool paths when machining mold cavities or pockets.

Reroughing NC sequences creates tool paths to machine only the areas where a previous roughing or reroughing sequence could not reach. Typically they are performed with a smaller tool and machine the areas that the larger roughing cutter could not enter due to its size.

It is important to understand the differences between volume milling and roughing sequences and to use each sequence appropriately.

Objectives

After completing this module, you will be able to:

- Describe high speed machining strategies.
- Describe the roughing and reroughing process.
- Describe the key roughing and reroughing features.
- Describe the key roughing and reroughing manufacturing parameters.
- Create mill volumes and mill windows.
- Create volume and roughing sequences.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

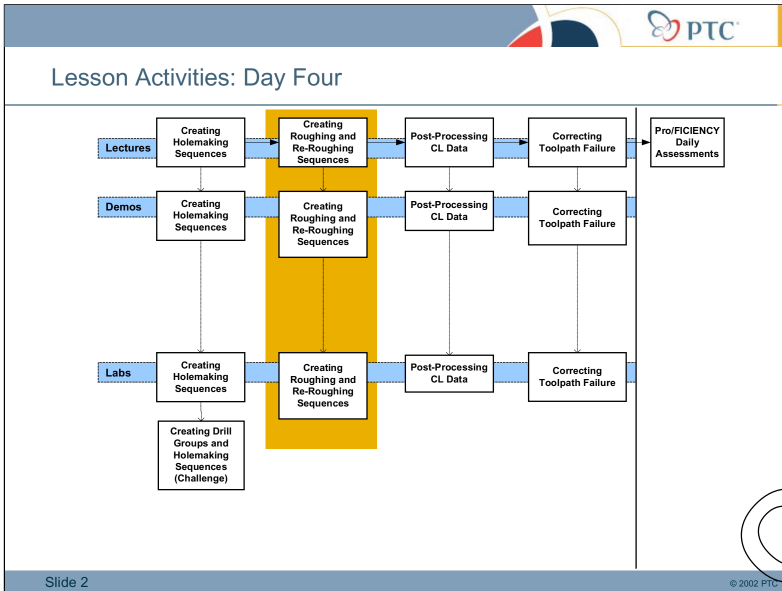
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

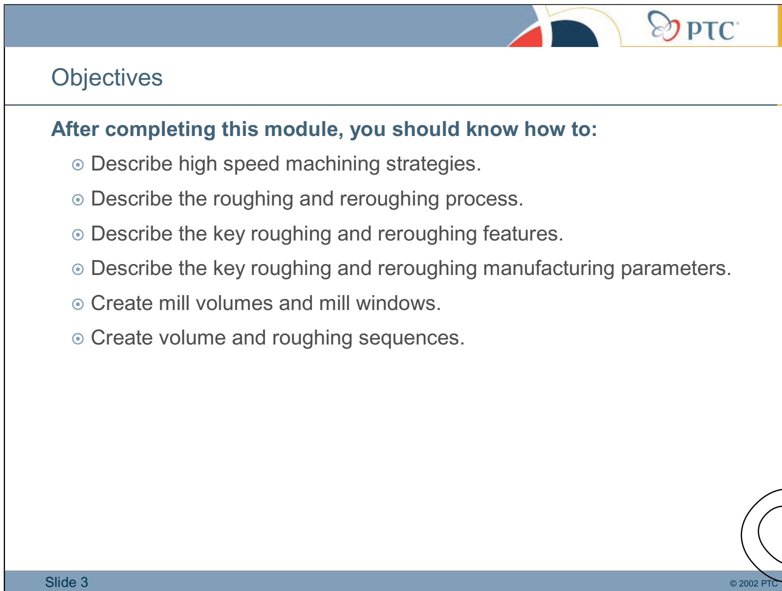
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos (1) : 25 mins
- Labs (1) : 50 mins
- Total: 1 hr 30 mins



Overview

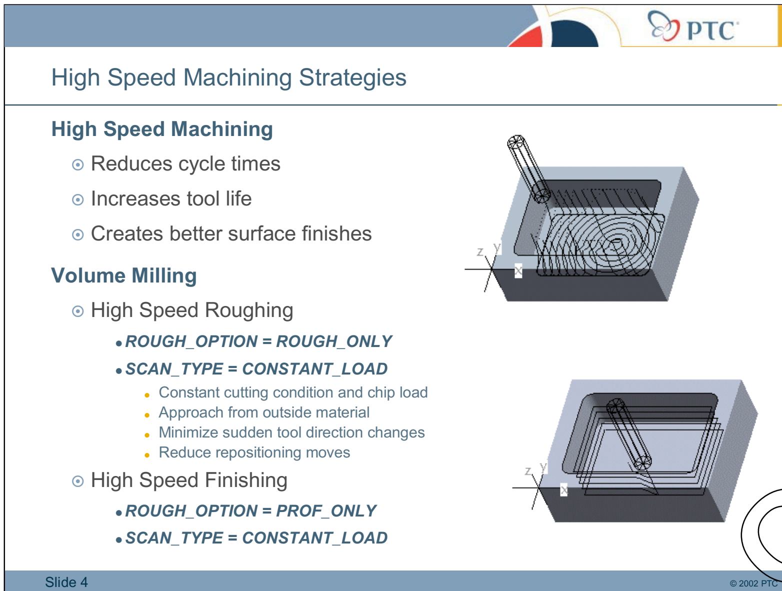
Roughing sequences are similar to volume milling sequences, but are intended specifically for high-speed mold machining, and especially for machining imported, non-solid geometry. Roughing sequences have additional capabilities aimed at producing more efficient tool paths when machining mold cavities or pockets.

Reroughing NC sequences create tool paths to machine only the areas where a previous roughing or reroughing sequence could not reach. Typically they are performed with a smaller tool and machine the areas that the larger roughing cutter could not enter due to its size.

It is important to understand the differences between volume milling and roughing sequences and to use each sequence appropriately.

After completing this module, you will be able to:

- Describe high speed machining strategies.
- Describe the roughing and reroughing process.
- Describe the key roughing and reroughing features.
- Describe the key roughing and reroughing manufacturing parameters.
- Create mill volumes and mill windows.
- Create volume and roughing sequences.



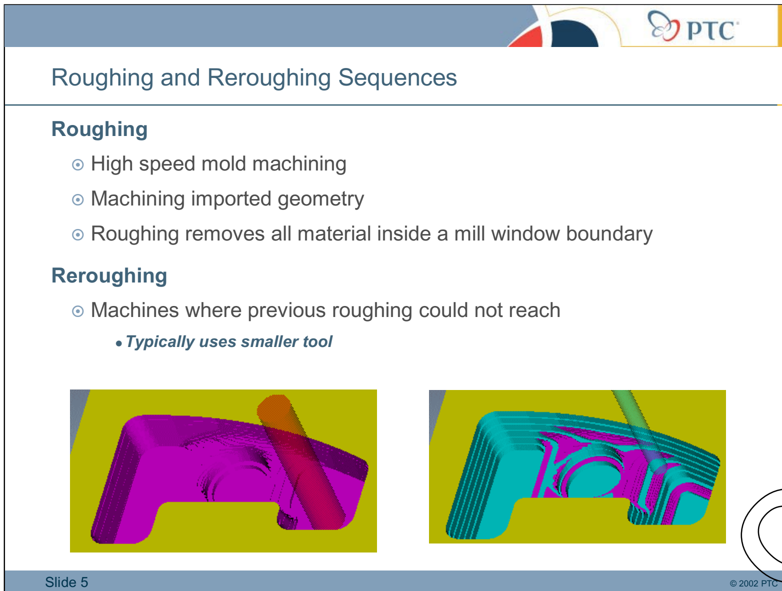
High Speed Machining Strategies

High Speed Machining provides the following benefits:

- Reduces cycle times
- Increases tool life
- Creates better surface finishes

In Volume Milling

- High Speed Roughing conditions can be created by setting
- ROUGH_OPTION to ROUGH_ONLY
- SCAN_TYPE to CONSTANT_LOAD
- This achieves the following:
  - Constant cutting condition and chip load
  - Approach from outside material
  - Minimize sudden tool direction changes
  - Reduce repositioning moves
- High Speed Finishing conditions can be created by setting
- ROUGH_OPTION to PROF_ONLY
- SCAN_TYPE to CONSTANT_LOAD
  - Achieves:
    - Constant cutting condition and chip load
    - Minimize sudden tool direction changes
    - Reduce repositioning moves

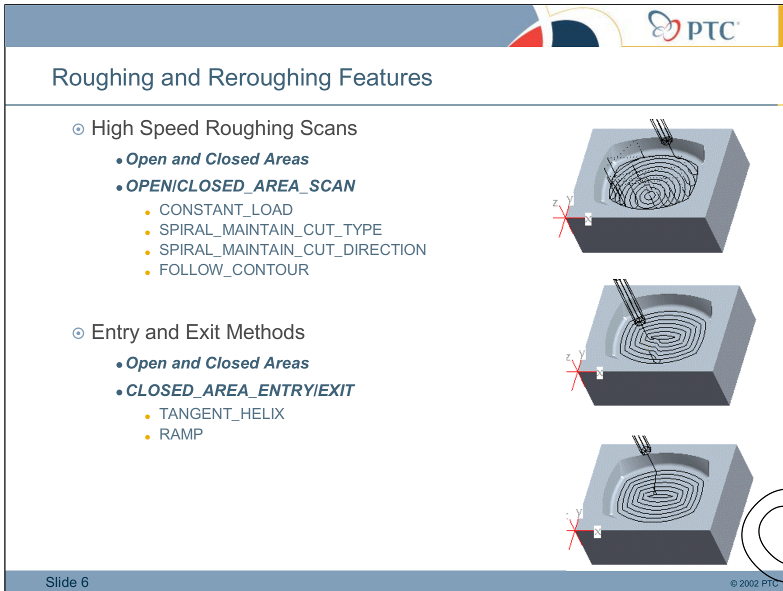


Roughing and Reroughing

These NC sequence types are designed specifically for high-speed mold machining, and especially for machining imported, non-solid geometry.

Roughing removes all material inside a mill window boundary, with additional depth control based on the depth of the mill window.

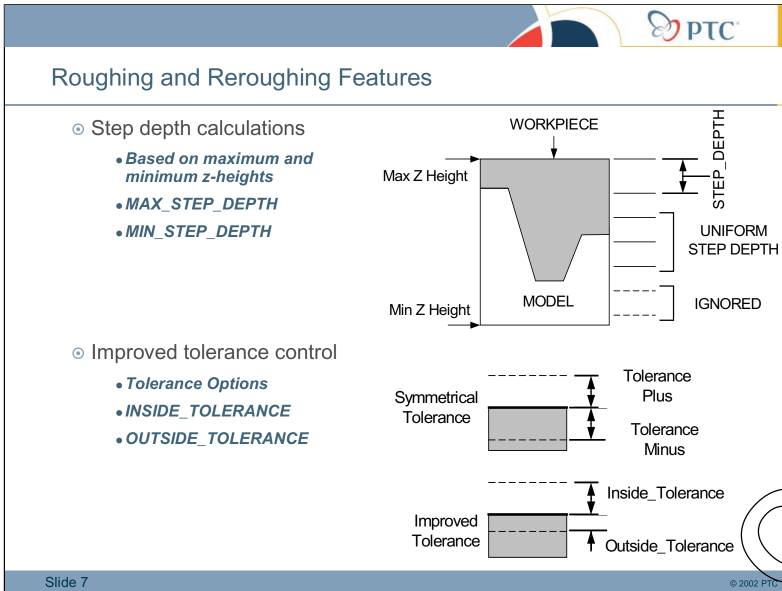
Reroughing NC sequences creates tool paths to machine only the areas where a previous Roughing sequence could not reach. Typically they are performed with a smaller tool and machine the areas that the larger roughing cutter could not enter due to its size. They do not attempt to remove scallops left between the slices of the referenced sequence.



Roughing and Reroughing Features

- **High Speed Roughing Scans**
  - Allows selection of a different high-speed scans for open and closed areas, assessed on a slice by slice basis.
  - Controlling parameters are:
  - OPEN/CLOSED_AREA_SCAN
    - CONSTANT_LOAD (default CLOSED)—creates a cutter path that produces an approximately constant tool load.
    - SPIRAL_MAINTAIN_CUT_TYPE—creates a spiral cutter path with reverse arc connections between cuts. This option minimizes retracts.
    - SPIRAL_MAINTAIN_CUT_DIRECTION—creates a spiral cutter path with S-shape connections between cuts. This option minimizes retracts.
    - FOLLOW_CONTOUR (default OPEN)—the shape of each cut follows the shape of the hard walls, maintaining fixed offset between the respective points of two successive cuts.

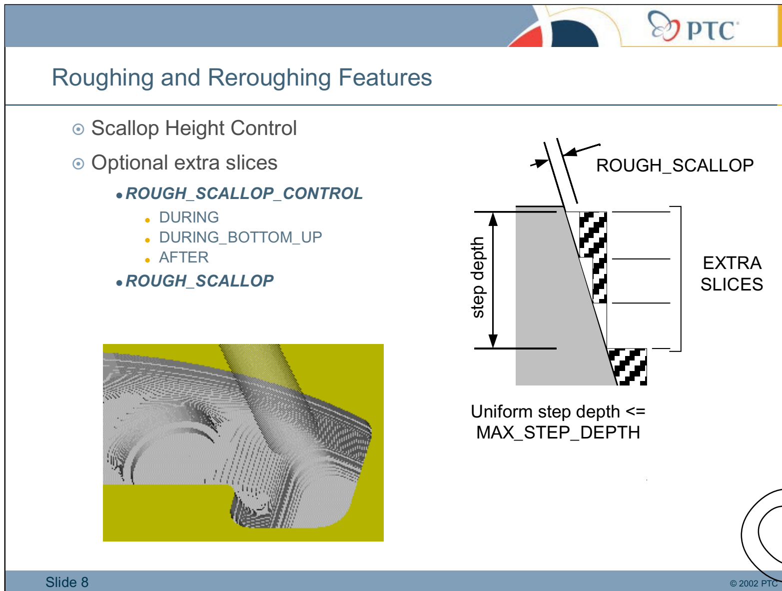
- **Entry and Exit Methods**
  - Provides different entry methods for open and closed areas. The tool enters open areas from the side. For closed areas, you can specify either helical or ramp top entry method.
  - Controlling parameters are:
  - CLOSED_AREA_ENTRY/EXIT
    - Approach or Exit move can be TANGENT_HELIX or RAMP



Roughing and Reroughing Features

- **Additional step depth calculations**
  - Calculates the step depth based on the maximum and minimum Z heights in the mill window and the MAX_STEP_DEPTH parameter value.
  - MAX_STEP_DEPTH
  - Specifies the maximum allowed step depth. After finding the positions of the highest and lowest slices, the system calculates the actual step depth, which is less than or equal to the specified MAX_STEP_DEPTH and results in a minimum number of uniformly spaced slices.
  - MIN_STEP_DEPTH
  - Lets you specify the minimum step depth allowed between the slices.

- **Improved tolerance control**
- The maximum distance that the straight line path of the tool deviates from the curved geometry is defined by tolerance. Tolerance band is normally symmetric, in Roughing and Reroughing you can specify different tolerance values for the two sides of the surface.



Roughing and Reroughing Features

Additional Scallop height control

After a first roughing sequence, the resulting scallop height does not always leave a constant thickness of material to remove for any subsequent finishing tool paths. Rough scallop control enables a semi-finishing toolpath to be generated with the same tool and during the same NC sequence, resulting in a smaller scallop height, and more uniform thickness of material remaining for any subsequent finishing tool paths.

ROUGH_SCALLOP_CONTROL

Controls if extra slices are created between regular roughing slices, and defines the machining order:

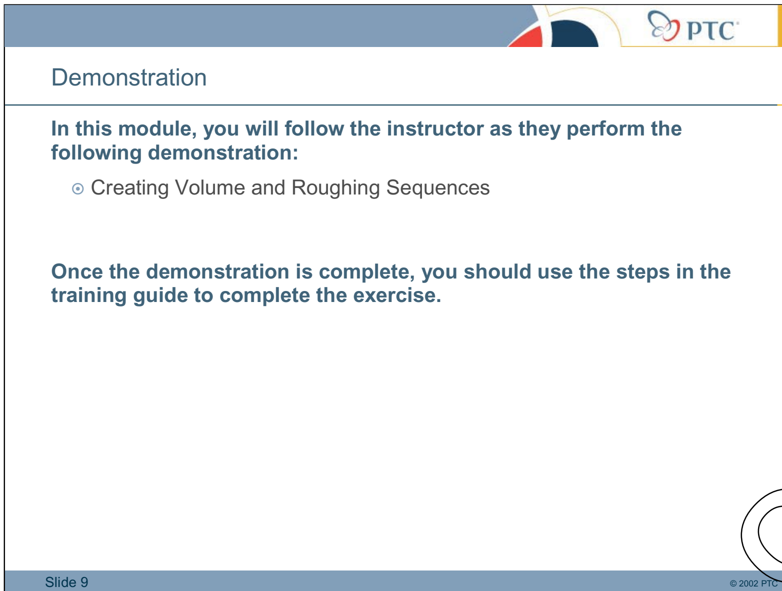
•NO—no extra slices are generated to control the scallop height.

•DURING—the system generates the extra slices and inserts them between the regular slices.

•DURING_BOTTOM_UP—the system generates the extra slices and inserts them between the regular slices, performing them from the bottom up to optimize the tool load.

•AFTER—the system generates the extra slices. The tool machines all the regular slices first, and then machines the extra slices to reduce the scallop height, where needed.

•ROUGH_SCALLOP: maximum scallop height for the steps between the regular slices.



Demonstration: Creating Volume and Roughing Sequences

Scenario

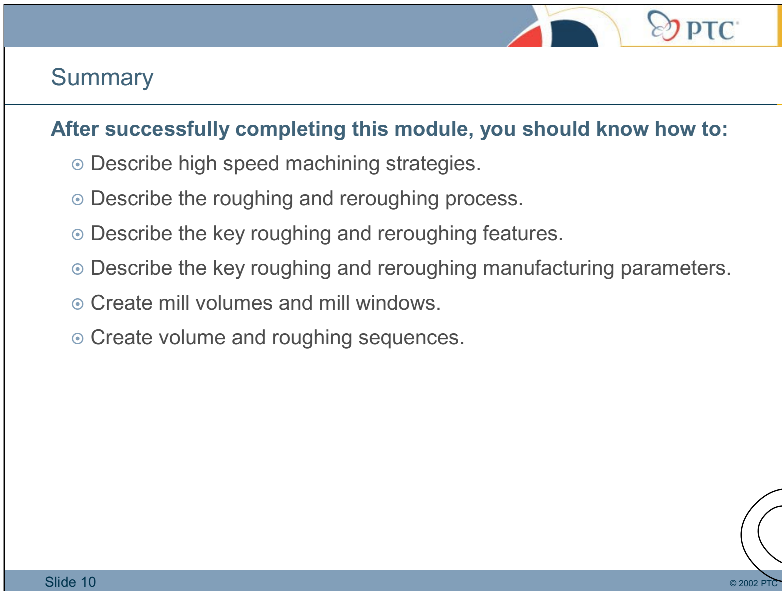
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercises (1)

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe high speed machining strategies.
- Describe the roughing and reroughing process.
- Describe the key roughing and reroughing features.
- Describe the key roughing and reroughing manufacturing parameters.
- Create mill volumes and mill windows.
- Create volume and roughing sequences.

Module 14 Lab Exercises

Exercise 1: Creating Roughing Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you machine a mold tool cavity using alternative volume milling and roughing sequences. You apply high speed machining methods to both sequences and compare the different resulting tool paths. You modify the roughing sequence and create extra slices to reduce the scallop height on the side-walls of the cavity. It is important to understand the differences between volume milling and roughing sequences and to use each sequence appropriately.

Objectives

After successfully completing this exercise, you will know how to:

- Create mill volumes and mill windows.
- Create volume and roughing sequences.

Scenario

You machine a mold tool cavity using alternative volume milling and roughing sequences. You start by opening an existing manufacturing model containing a mold cavity. You create a volume milling sequence to create a roughing toolpath in the cavity; you apply high speed machining methods to improve the efficiency of the tool path. You notice that the scallop height of the resulting tool path does not leave a uniform thickness of material to remove for any subsequent finishing tool paths. You create an alternative toolpath using a roughing sequence to machine the same cavity, again applying high speed machining methods. This toolpath creates roughing and semi-finish passes in the same NC sequence. You notice the scallop height of the resulting tool path is now reduced, leaving a more uniform thickness of material remaining for any subsequent finishing tool paths. (The next stage in the machining process would be to finish the mold cavity with a series of surface milling sequences; this is not done in this exercise).

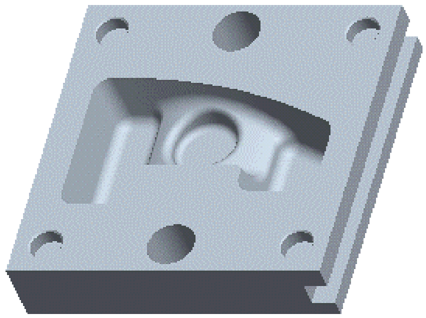


Figure 1: Showing the mold cavity to be machined.

Parameters

The following are key roughing parameters.

Table 1: Parameter Descriptions

Parameter	Description
OPEN_AREA_SCAN	Specifies the way the tool scans a horizontal machining slice that contains at least one open side. All the options are intended for high speed machining.
CLOSED_AREA_SCAN	Specifies the way the tool scans a horizontal machining slice that is completely surrounded by hard walls. All the options are intended for high speed machining.
CLOSED_AREA_ENTRY	Specifies the top entry method for closed areas.
CLOSED_AREA_EXIT	Specifies the exit method for closed areas.
MAX_STEP_DEPTH	Specifies the maximum allowed step depth.
MIN_STEP_DEPTH	Optionally specifies the minimum step depth allowed between the slices.
INSIDE_TOLERANCE	The portion of the tolerance band that falls inside the surface.
OUTSIDE_TOLERANCE	The portion of the tolerance band that falls outside the surface.
ROUGH_SCALLOP_CONTROL	Controls if extra slices are created between regular roughing slices. Also defines the machining order.
ROUGH_SCALLOP	Maximum scallop height for the steps between the regular slices.
RETRACT_OPTION	Controls the number and level of retracts.
CORNER_ROUND_RADIUS	Specifies the minimum radius allowed for concave corners in high speed machining.

Step 1. Locate and open the cover manufacturing model.

Before creating any milling sequences, we need to locate the manufacturing model and open it using the folder navigator. When we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module14**, right-click, and select **Make Working Directory.**
2. Open MOLD_CAVITY.MFG.
3. Turn off any displayed datum features.

Step 2. Review the face milling NC sequence.

Before creating any volume or roughing sequences, we need to check what NC sequences have already been created. Using the Menu Manager, we can select the face milling sequence, and play the toolpath. Satisfied with this toolpath, we can move on to creating more NC sequences.

1. Using the Menu Manager, select **Machining > NC Sequence > FACE_TOP**, to select the face milling sequence for redefining. **Suspend All** material removal features.
2. Play the tool path using the **Screen Play** option.
3. Complete the sequence.

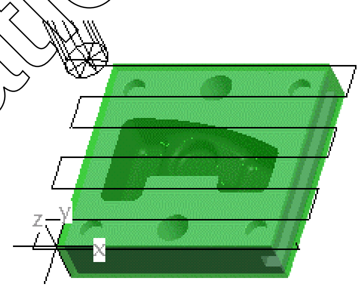


Figure 2: Showing the previously created face milling toolpath.

Step 3. Create a volume milling NC sequence to rough out the mold cavity.

Using the Menu Manager, we can create a volume milling sequence. We can select the name and tool options, and notice that parameters and volume are already checked off. We can enter a name for the sequence to make it easy to identify, and select the FEM_20_0 tool. We also need to specify parameters. To save time adjusting parameters, we can retrieve a stored set of roughing parameters that adhere to high speed machining methods. We can set TRIM_TO_WORKPIECE to YES, to avoid air machining, and set CUTCOM to OFF for this sequence.

We decide to create a mill volume to define the volume of material to be machined. We can name the mill volume to make it easy to identify. We decide to use the gather and Surf and Bound options to define the volume. Configuring a mill volume using the surface and boundaries method involves selecting a seed (or starting surface) and boundary surfaces. All surfaces within the configured boundary are "sewn" together to form a single quilt, which is then automatically extruded up to the retract plane to form the mill volume. We select one of the surfaces within the cavity as our seed surface, and select the top surface of the MOLD_CAVITY.PRT model as our boundary surface. When the mill volume is complete, we can select it in the model tree and hide it to make the display clearer.

We can play the tool path and notice that high-speed machining methods have been applied; this is because the SCAN_TYPE parameter is set to CONSTANT_LOAD. We can modify the SCAN_TYPE to SPIRAL_MAINTAIN_CUT_TYPE this reduces the number of retract moves in the tool path.

We can also play the tool path in Vericut. Notice the tool still leaves a visible scallop height on the sidewalls. We need to reduce the scallop height and leave a more uniform thickness of material remaining; this could be achieved by creating a semi-finishing tool path with a smaller tool. However, we want to try to use the same tool for the roughing and semi-finishing tool paths, enabling us to reduce machine cycle times. This can be achieved by creating a roughing sequence.

1. Using the Menu Manager, create a **Volume** milling sequence.
2. Select the **Name**, and **Tool** check boxes.
3. Name the sequence [**RGH_VOL**].
4. Change the tool to the **FEM_20_0** end mill.
5. To save time select **Retrieve**, and **Open** the ROUGHING_HSM_CAVITY.MIL parameters.
6. Modify the TRIM_TO_WORKPIECE parameter to **YES**.

> **Note:**
> The stored parameter file has optimum feed, speed, and depth-of-cut values to enable high speed machining.

7. Select **Create Vol** to create a mill volume, name it [**VOL_CAVITY**].
8. Select **Gather** and accept the Gather Steps options, then use **Surf & Bnd** to configure the gathering method.

> **Note:**
> Configuring a mill volume using the surface and boundaries method involves selecting a seed (or starting surface) and boundary surfaces. All surfaces within the configured boundary are "sewn" together to form a single quilt, which is then automatically extruded up to the retract plane to form the mill volume.

9. Move the cursor over a surface inside the mold cavity, then right-click, and select a seed surface inside the cavity as shown in the following figure. Move the cursor over the top surface of the mold cavity, right-click and select the surface of the mold cavity as the boundary surface as shown.

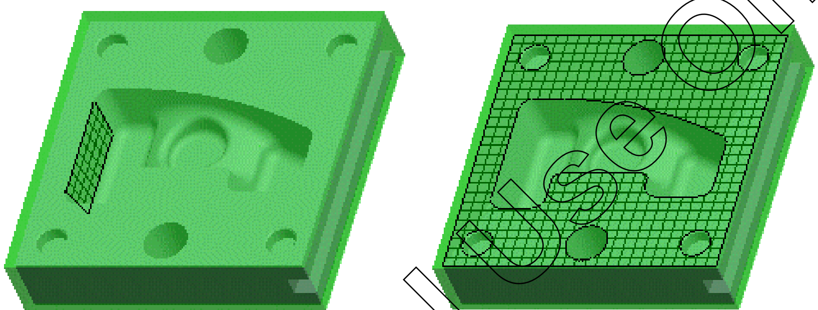


Figure 3: Showing seed and boundary surfaces selected.

10. Complete the mill volume.

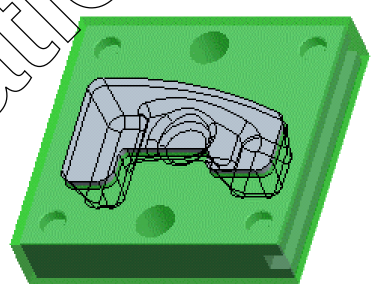


Figure 4: Showing the completed mill volume.

> **Note:**
> When the mill volume is configured we can hide it to make the display clearer.

11. Select the mill volume in the model tree, right-click and **Hide** it.

12. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

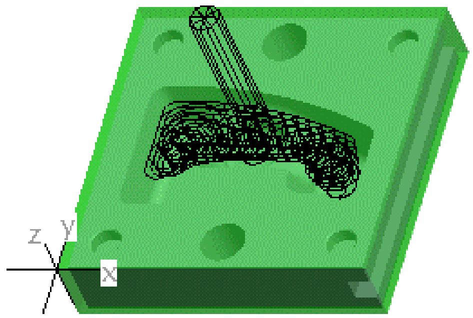


Figure 5: Showing the resulting tool path in default view.

13. Reorient the model and create a named front view as shown in the following figure.

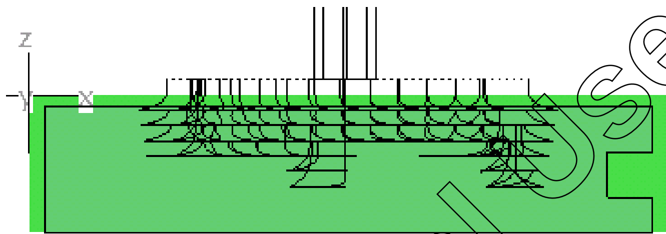


Figure 6: Showing the resulting tool path in front view.

> **Tool Path Review:**
> The SCAN_TYPE parameter is set to CONSTANT_LOAD, this means the tool path is following high speed machining guidelines. This involves reducing machine direction change, providing constant load during machining, reducing load and unload effects, and when necessary using a helical approach.

14. Change the SCAN_TYPE parameter to SPIRAL_MAINTAIN_CUT_TYPE.

15. Play the tool path using the **Screen Play** option. Ensure the CL data is visible. View the tool path in the default view and the named front view.

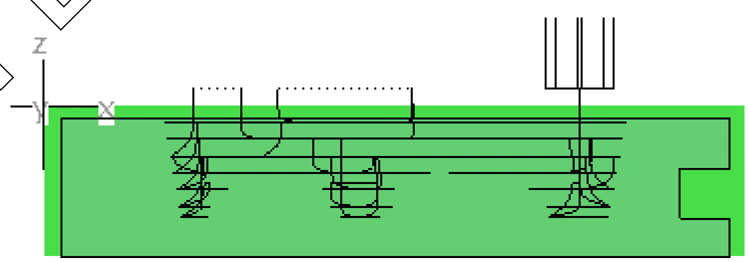


Figure 7: Showing the resulting tool path in front view.

Tool Path Review:

Notice the reduction in retract moves when setting the SCAN_TYPE to SPIRAL_MAINTAIN_CUT_TYPE.

16. Change the SCAN_TYPE parameter back to CONSTANT_LOAD.

17. Using **NC Check** start Vericut, then click on the Play to End icon to view the machining simulation.

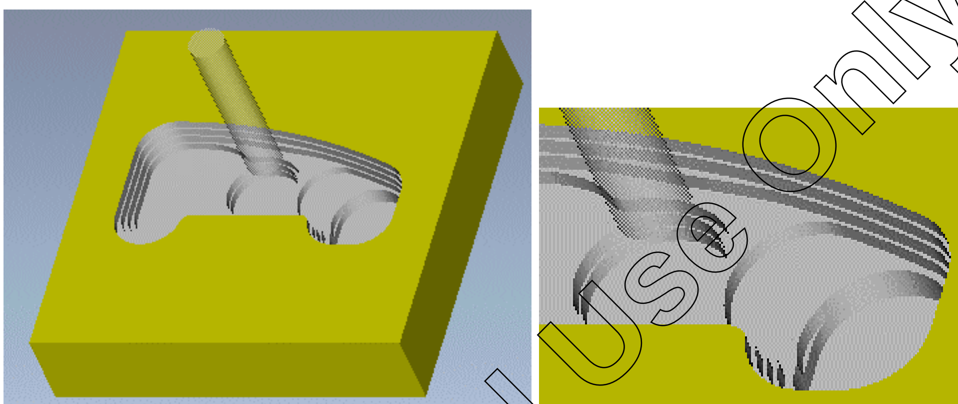


Figure 8: Showing the resulting tool path in Vericut.

Tool Path Review:

Notice the tool still leaves a visible scallop height on the side walls. We need to reduce the scallop height and leave a more uniform thickness of material remaining; this could be achieved by creating a semi-finishing tool path with a smaller tool. However, we want to use the same tool for the roughing and semi-finishing tool paths, enabling us to reduce machine cycle times. This can be achieved by creating a roughing sequence.

18. Close Vericut, and complete the sequence.

Step 4. Create a roughing NC sequence to rough out the mold cavity.

Using the Menu Manager, we can create a roughing sequence. We can select the name options and notice that parameters and window are already checked off. We can enter a name for the sequence to make it easy to identify, and use the same tool as before. We can also use the parameters from the volume milling sequence, then modify some of the parameters that are only available in roughing and reroughing sequences.

We need to create a mill window to define the volume of material to be machined. We can name the mill window to make it easy to identify. We can use the Select, and Tangent Chain options, and select one of the edges of the cavity geometry. With the mill window configuration complete, we can play the tool path and notice that high-speed machine methods have been applied.

However notice the tool is still making passes at approximately five-millimeter depth increments. We need to modify the ROUGH_SCALLOP_CONTROL parameter to create additional passes between the roughing passes. We can set the ROUGH_SCALLOP_CONTROL to DURING and set the ROUGH_SCALLOP to 0.4.

Replaying the tool path, notice the tool has created additional passes between the initial roughing passes. The ROUGH_SCALLOP_CONTROL parameter enables creation of the passes either during or after the initial roughing passes. The ROUGH_SCALLOP parameter specifies the maximum allowable scallop height on the walls of the cavity. The resulting scallop height is now reduced leaving a more uniform thickness of material remaining. Finally, we can also verify the toolpath in Vericut.

1. Using the Menu Manager, create a **Roughing** sequence.
2. Select the **Name** check box. Notice the parameters and window check boxes are already selected.
3. Name the sequence [**RGH_CAVITY**].
4. Select **Use Prev** and use the manufacturing parameters from the previous **RGH_VOL** sequence.
5. Modify the following parameters.

Table 2: Changed Parameters

Manufacturing Parameter	Value	Section
MAX_STEP_DEPTH	5 *(0.20 in)*	CUT PARAM
INSIDE_TOLRENACE	0.04 *(0.002 in)*	CUT PARAM
OUTSIDE_TOLERANCE	0.08 *(0.003 in)*	CUT PARAM

6. Select **Create Wind** to create a mill window, name it [**WND_CAVITY**].
7. Using the **Select** and **Tangent Chain** options select one of the edges of the cavity geometry, as shown in the following figure. Complete the mill window configuration.

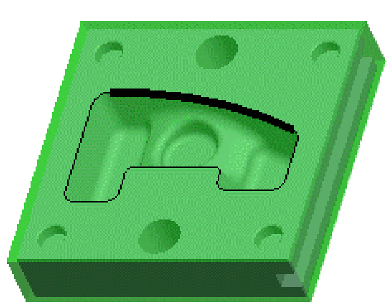


Figure 9: Showing the selected edge and highlighted tangent chain of edges.

8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible. View the tool path in the default view and the named front view.

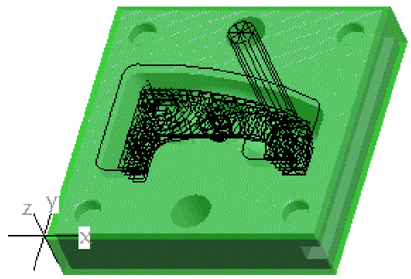


Figure 10: Showing the resultant tool path in default view.

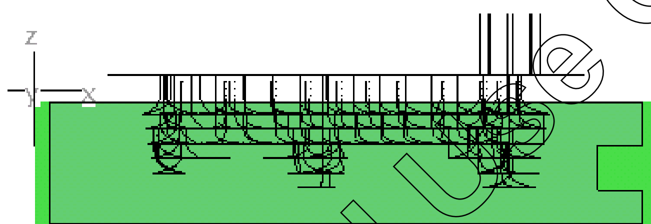


Figure 11: Showing the resultant tool path in front view.

Tool Path Review:

Notice the tool is still making passes at approximately 5 millimeter *(0.20 in)* depth increments. We need to modify the ROUGH_SCALLOP_CONTROL parameter to create additional passes between the roughing passes.

9. Modify the following parameters:

Table 3: Changed Parameters

Manufacturing Parameter	Value	Section
ROUGH_SCALLOP_CONTROL	DURING	CUT OPTION
ROUGH_SCALLOP	0.4 *(0.02 in)*	CUT PARAM

10. Play the tool path using the **Screen Play** option. Ensure the CL data is visible. View the tool path in the default view and the named front view.

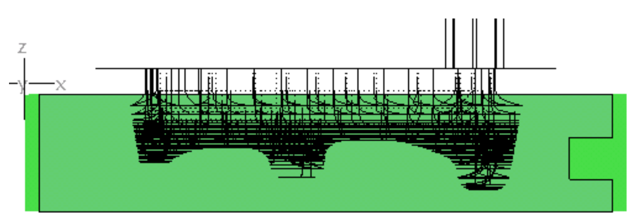


Figure 12: Showing the resultant tool path in front view.

11. Using **NC Check** start Vericut, then click on the Play to End icon to view the machining simulation.

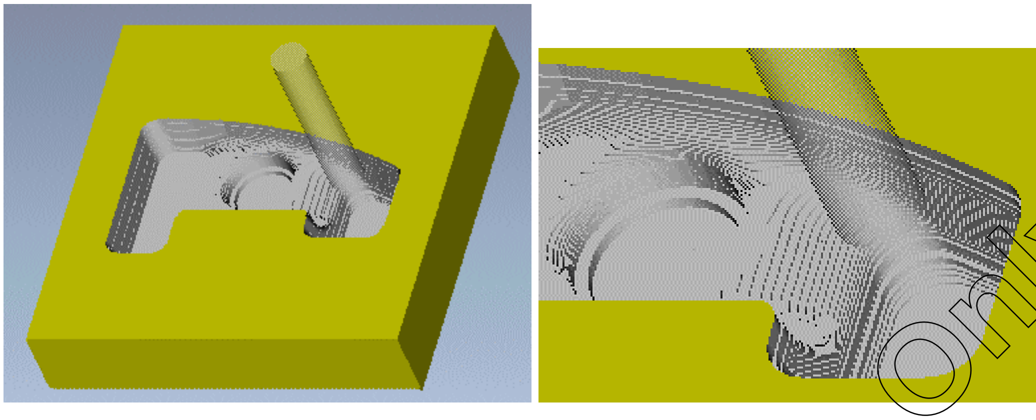


Figure 13: Showing the resulting tool path in Vericut.

> **Tool Path Review:**
>
> Notice the tool has created additional passes between the initial roughing passes. The ROUGH_SCALLOP_CONTROL parameter enables creation of the passes either during or after the initial roughing passes. The ROUGH_SCALLOP parameter specifies the maximum allowable scallop height on the walls of the cavity. The resulting scallop height is now reduced leaving a more uniform thickness of material remaining.

12. Close Vericut, and complete the sequence.

13. Save the manufacturing model.

14. Close all windows and erase all components from memory.

> **Note:**
>
> The next logical stage in the machining process would be to finish the mold cavity with a series of surface milling sequences; this is not done in this exercise as we have covered surface milling in an earlier exercise.

Summary

After successfully completing this module, you should know how to:

- Describe high speed machining strategies.
- Describe the roughing and reroughing process.
- Describe the key roughing and reroughing features.
- Describe the key roughing and reroughing manufacturing parameters.
- Create mill volumes and mill windows.
- Create volume and roughing sequences.

Module

15

Post-Processing CL Data

Introduction

Post-processing is the final stage in the manufacturing process. When operations have been completed you can create ASCII format CL data files for operations. CL data files can then be post-processed into specific machine control data (MCD) files using a post-processor. It is important to understand that changing NC sequences means you need to recreate the CL data file for the operation and post-process this file again to produce an updated MCD file.

Objectives

After completing this module, you will be able to:

- Describe the post-processing method.
- Create CL Data files.
- Create MCD files using a post-processor.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

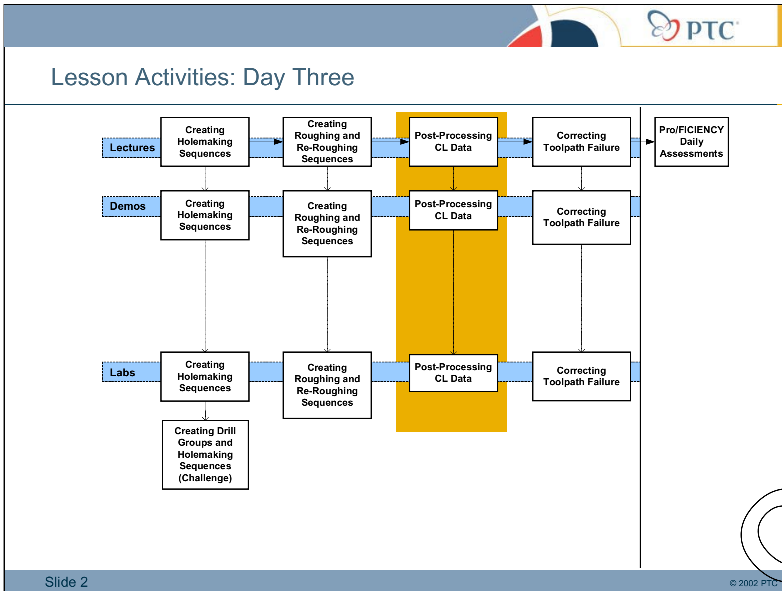
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

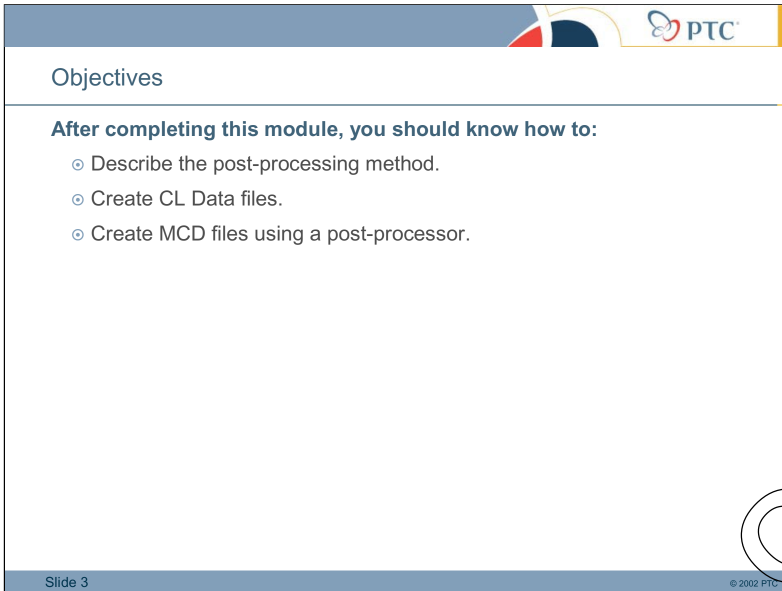
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 10 mins
- Demos (1) : 15 mins
- Labs (1) : 35 mins
- Total: 1 hr 00 mins

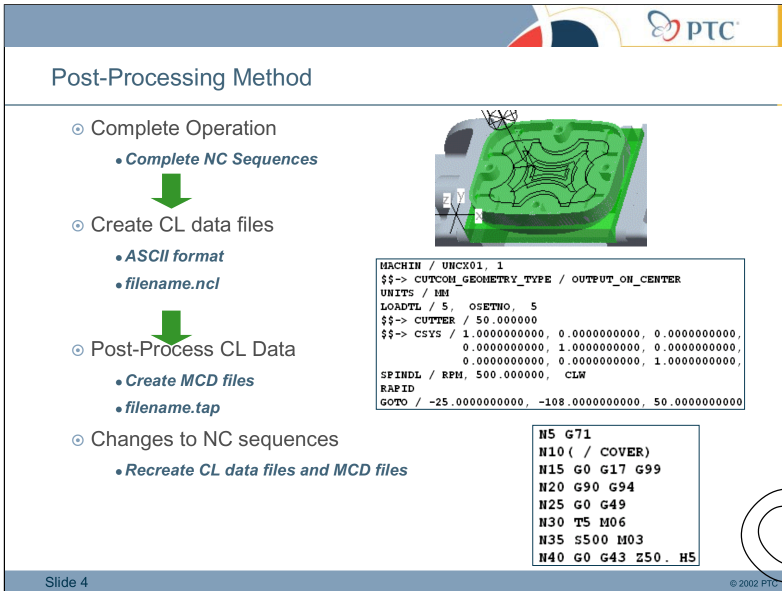


Overview

Post-processing is the final stage in the manufacturing process. When operations have been completed you can create ASCII format CL data files for operations. CL data files can then be post-processed into specific machine control data (MCD) files using a post-processor. It is important to understand that changing NC sequences means you need to recreate the CL data file for the operation and post-process this file again to produce an updated MCD file.

After completing this module, you will be able to:

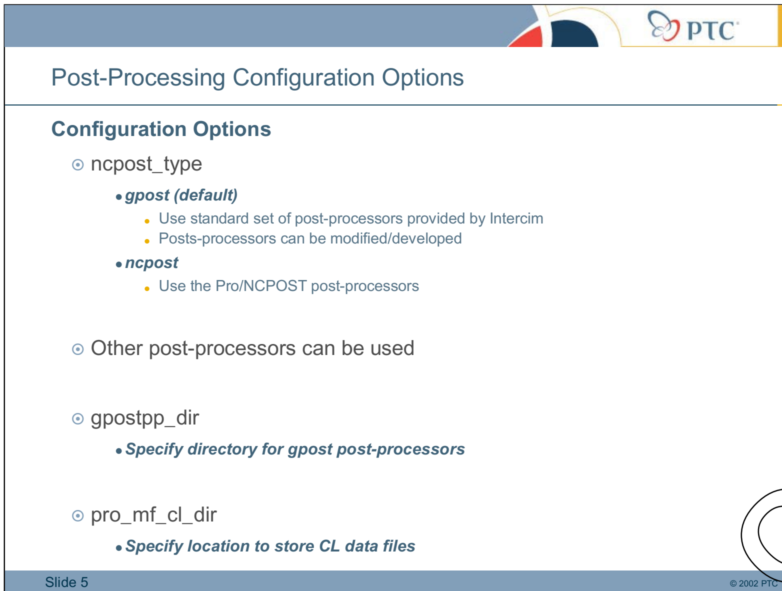
- Describe the post-processing method.
- Create CL Data files.
- Create MCD files using a post-processor.



Post-Processing Method

- You first need to complete the operation by creating all necessary NC sequences.
- When the operation is complete you can create CL data files, these are ASCII format files.
  - The default filename format is *filename*.ncl
- You can then post-process CL data files into specific machine control data (MCD) files.
  - The default filename format is *filename*.tap
  - You have an option to create the CL and MCD files simultaneously.

- Any changes to NC sequences means you must re-create the CL data files and MCD files.



Configuration Options

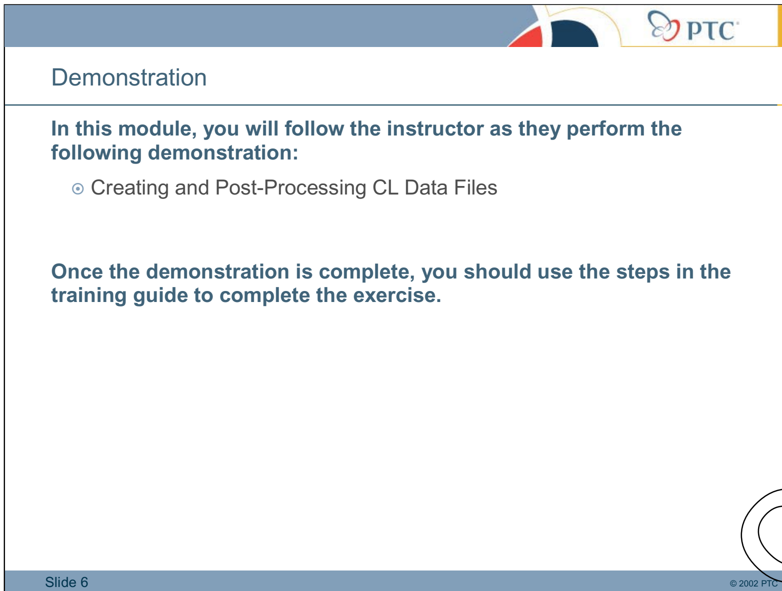
Each Pro/NC module includes a standard set of NC post-processors that can be used directly or modified using an optional module. You can control which post-processing module to use by setting the configuration option ncpost_type. The values are:

- gpost (default)—use the G-Post™ post-processors provided by Intercim Corporation.
- ncpost—use the Pro/NCPOST post-processors, provided by ICAM.

- Other post-processors capable of reading APT (automatically programmed tools) can also be used

- gpostpp-_dir
  - Specify the directory for gpost post-processors

- pro_mf_cl_dir
  - Specify location to store CL data files



Demonstration: Creating and Post-Processing CL Data Files

Duration: 15 minutes

Scenario

- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

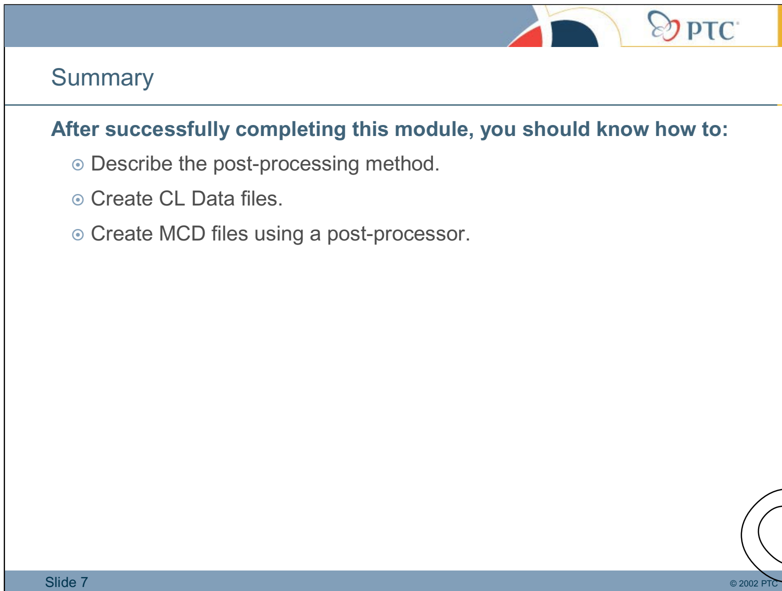
Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercises (1)

Duration: 35 minutes

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the post-processing method.
- Create CL Data files.
- Create MCD files using a post-processor.

Module 15 Lab Exercises

Exercise 1: Creating and Post-Processing CL Data Files

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you open the manufacturing model for the cover and create CL data files and MCD files for both operations. You then remove the trajectory milling NC sequence from operation 020, and re-create the CL data file and MCD file. It is important to understand that changing NC sequences means you need to recreate the CL data file for the operation and post-process this file again to produce an updated MCD file.

Objectives

After successfully completing this exercise, you will know how to:

- Create CL Data files.
- Create MCD files using a post-processor.

Scenario

You are ready to create MCD files for both operations in the aluminum cover. You open the manufacturing model for the cover and create CL data files and MCD files for both operations. It is decided that the last NC sequence in operation op020 is not required, as the grooves in the top of the model will be machined manually. You remove the trajectory milling NC sequence from operation op020, and re-create the CL data file and MCD file for operation op020.

Step 1. Locate and open the cover manufacturing model.

Before creating any CL data files, we need to locate the manufacturing model and open it using the folder navigator. When we have located the folder for this exercise, we can set this as our working directory, and open the manufacturing model of the cover. Before post-processing any files we can load a configuration file to ensure that we use the correct post-processor located in the posts directory.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module15**, right-click, and select **Make Working Directory.**
2. Open COVER.MFG.
3. Turn off any displayed datum features.
4. Before post-processing any files, load a configuration file. Click **Tools > Options**, in the Options dialog box, click the Open icon, select **config.pro** and **Open** it. **Apply** and **Close** the Options dialog box.

> **Note:**
> Loading the configuration file sets the gpostpp_dir option to .\posts, this specifies the directory for post-processors. There is a fadal-vmc post processor in this directory.

Step 2. Create a CL data file and MCD file for operation op010. Review the CL data file in Vericut.

We can create a CL data file for operation op010, at the same time we can create an MCD file, this enables us to create and post-process the CL data file at the same time. We can select the UNCX01.99 post-processor; as this is the fadal-vmc post. We have now created a CL data file named op010.ncl, and an MCD file named op010.tap.We can open and review these files in a text editor. Notice the PPRINT statements in the CL data file are transferred into the MCD file.

We can now start Vericut and review the CL data file.

1. Using the Menu Manager, select **CL Data > Output > Operation > OP010 > File**. Select the **MCD File** check box, and complete the file output. Click the **OK** button to create a CL data file for operation OP010.
2. Accept the **Verbose** and **Trace** options, and select the **UNCX01.P99** post-processor, when prompted type any number for the program number.
3. **Close** the information window, and return to the Manufacture menu.

> **Note:**
> You have now created a CL data file, named op010.ncl, and an MCD file named op010.tap.

4. In the navigator, select the Folder Browser tab, click the **module15** folder to view the contents in the browser.

5. In the Browser, select the **op010.tap** file, right-click and select **Open** to view the contents of the file in the browser, as shown in the following figure.

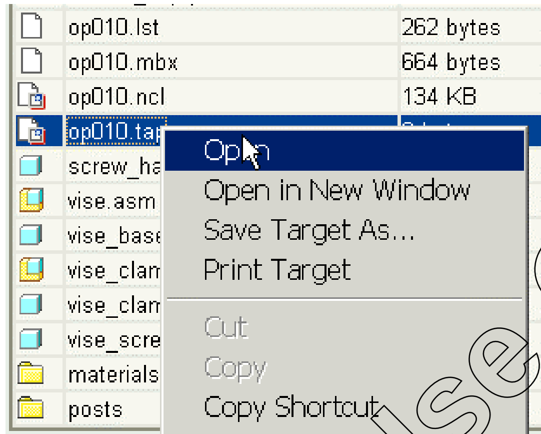


Figure 1: Showing contents of browser.

6. Click the Back icon at the top of the browser to return to viewing the files in the module15 folder, then minimize the browser window.

```
%
O1234
N1 (PRO/NC TOOL PATH : OP010)
N2 (PARTNO : COVER)
N3 G90 G80 G40 G17 G0
N4(DATE TIME : 02-DEC-02 14:46:30)
N5(      TOOL TABLE SUMMARY)
N6(TOOL NUMBER   TOOL ID       OFFSET NO   TOOL COMMENT)
N7(     1        CTRDRILL_04      1)
N8(     3        DRILL_06_0       3)
N9(     5        CSINK_12         5)
N10(     9        FEM_06_0         9)
N11(     15       FEM_25_0         15)
N12(     17       SH_MILL_50_0     17)
N13(NC SEQUENCE NAME : FACE_BOTTOM)
N14 G90 G80 G40 G17 G0
N15 T17 M6
N16 S1165 M3
N17 G0 G90 X-30. Y12.5
N18 G43 H17 Z40. M8
```

Figure 2: Showing start of op010.tap file.

Note:
Notice the PPRINT statements in the CL data file are transferred into the MCD file.

7. Using the Menu Manager, select **CL Data > NC Check > CL File**, and **Open** the op010.ncl file, then select **Done** to start Vericut. (You may need to wait a few moments for Vericut to start).

> **Note:**
> Selecting the NC Check option in the CL Data menu starts Vericut. This will open a separate Vericut window to perform the machining simulation.

8. In the Vericut window, click on the Setup Motion icon, and in the Motion dialog box drag the animation speed slider to the left, (this will reduce the speed of the animation). Then close the Motion dialog box.

9. Click on the View Toolpath File icon to see the CL data file during the simulation. Then click on the Play to End icon to view the machining simulation.

Figure 3: Showing CL data file op010.ncl in Vericut.

10. Close Vericut when finished, and return to the Manufacture menu.

Step 3. Activate operation op020.

> **Note:**
> Operation op020 is activated to ensure the CL data file is displayed correctly in Vericut.

1. Using the Menu Manager, select **Mfg Setup > Operation**, in the Operation dialog box activate operation **op020**. Close the dialog box, and return to the Manufacture menu.

2. Set the view to the named view **OP020-TRI**.

Step 4. Create a CL data file and MCD file for operation op020. Review the CL data file in Vericut.

We can create a CL data file for operation op020, at the same time we can create an MCD file, this enables us to create and post-process the CL data file at the same time. We can select the UNCX01.99 post-processor; as this is the fadal-vmc post. We have now created a CL data file *named op020.ncl, and an MCD file named op020.tap.We can open and review these files in a text editor.*

We can now start Vericut and review the CL data file.

1. Using the Menu Manager, select **CL Data > Output > Operation > OP020 > File**. Select the **MCD File** check box, and complete the file output. Click the **OK** button to create a CL data file for operation OP020.
2. Accept the **Verbose** and **Trace** options, and select the **UNCX01.P99** post-processor, when prompted type any number for the program number.
3. **Close** the information window, and return to the Manufacture menu.

> **Note:**
> You have now created a CL data file, named op020.ncl, and an MCD file named op020.tap.

4. Using the Menu Manager, select **CL Data > NC Check > CL File**, and **Open** the op020.ncl file, then select **Done** to start Vericut. (You may need to wait a few moments for Vericut to start).
5. Click on the View Toolpath File icon to see the CL data file during the simulation. Then click on the Play to End icon to view the machining simulation.

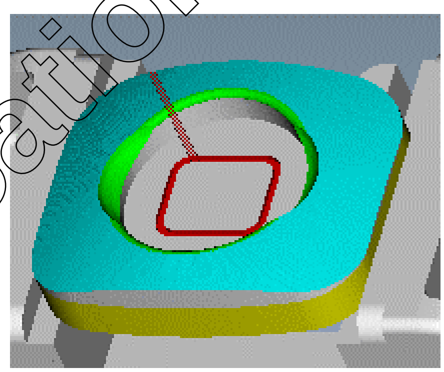


Figure 4: Showing CL data file op020.ncl in Vericut.

6. Close Vericut when finished, and return to the Manufacture menu.

Step 5. Suppress the last NC sequence named FIN_GROOVE, in operation op020. Recreate the CL data and MCD files, and review the CL data file in Vericut.

We can create a CL data file for operation op020, at the same time we can create an MCD file, this enables us to create and post-process the CL data file at the same time. We can select the UNCX01.99 post-processor; as this is the fadal-vmc post. We have now created a CL data file named op020.ncl, and an MCD file named op020.tap. We can open and review these files in a text editor.

We can now start Vericut and review the CL data file. Notice how the CL data file has updated and no longer machines the groove in the top of the model.

1. In the model tree, select the **FIN_GROOVE** sequence, right-click and **Suppress** it, click **OK** in the Suppress window.

2. Using the Menu Manager, select **CL Data > Output > Operation > OP020 > File**. Select the **MCD File** check box, and complete the file output. Click the **OK** button to create a CL data file for operation OP020.

3. Accept the **Verbose** and **Trace** options, and select the **UNCX01.P99** post-processor, when prompted type any number for the program number.

4. **Close** the information window, and return to the Manufacture menu.

5. Using the Menu Manager, select **CL Data > NC Check > CL File**, and **Open** the op020.ncl file, then select **Done** to start Vericut.

6. Click on the Play to End icon to view the machining simulation.

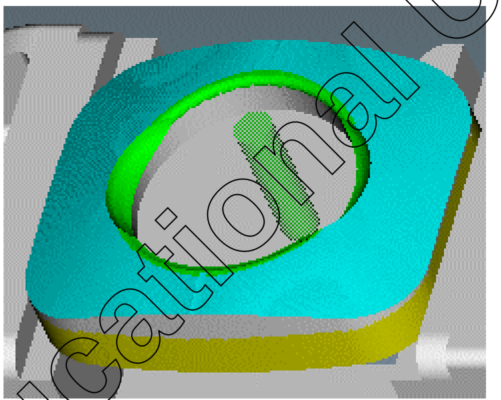


Figure 5: Showing the updated CL data file op020.ncl in Vericut.

> **Note:**
> Notice how the CL data file has updated and no longer machines the groove in the top of the model.

7. Close Vericut when finished, and return to the Manufacture menu.

8. Save the manufacturing model.

9. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the post-processing method.
- Create CL Data files.
- Create MCD files using a post-processor.

Module

16

Correcting Toolpath Failure

Introduction

NC sequences can sometimes fail after they have been created; this is normally due to sequence references changing or being removed. For example, a design engineer may change the diameter of a hole, or the shape of a pocket on a design model. In most cases, NC sequences will update automatically to reflect these changes, however if the original design references have been removed or drastically changed in size or shape, the associated NC sequence may fail. It is important to understand why NC sequences fail, and how to investigate and correct failed NC sequences.

Objectives

After completing this module, you will be able to:

- Describe the reasons for NC sequence failure.
- Describe how to investigate and correct failed NC sequences.
- Investigate and correct failed NC sequences.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

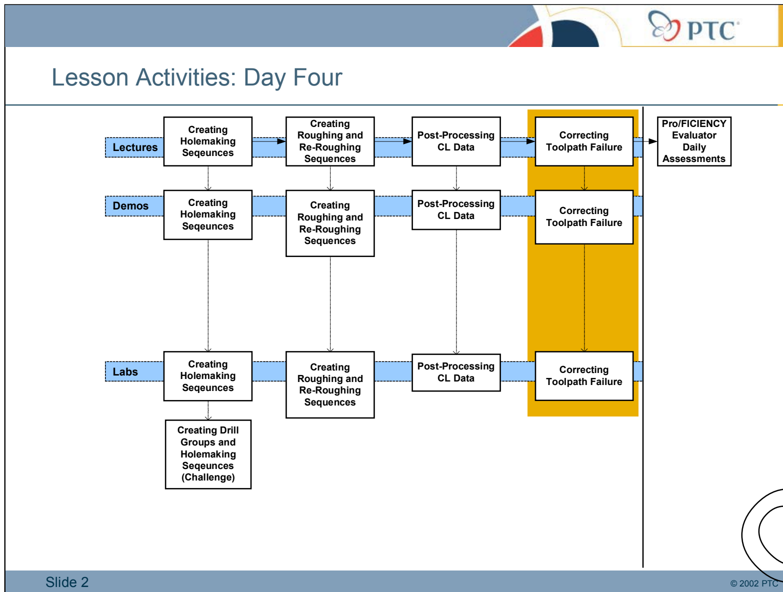
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

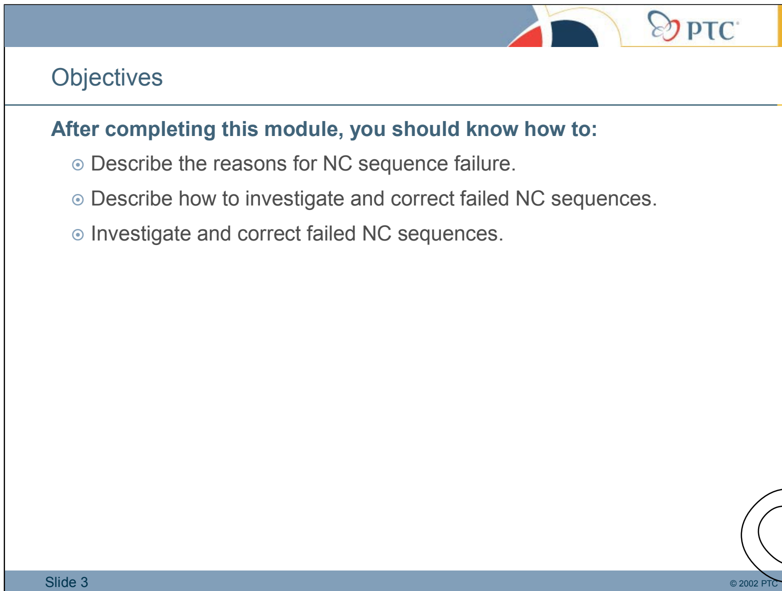
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 25 mins
- Labs : 50 mins
- Total: 1 hr 30mins

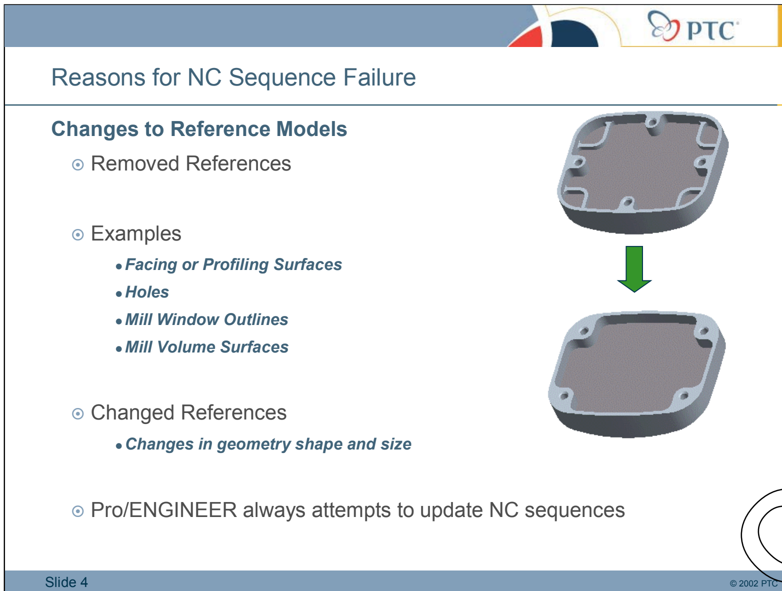


Overview

NC sequences can sometimes fail after they have been created; this is normally due to sequence references changing or being removed. For example a design engineer may change the diameter of a hole, or the shape of a pocket on a design model. In most cases NC sequences will update automatically to reflect these changes, however if the original design references have been removed or drastically changed in size or shape the associated NC sequence may fail. It is important to understand why NC sequences fail, and how to investigate and correct failed NC sequences.

After completing this module, you will be able to:

- Describe the reasons for NC sequence failure.
- Describe how to investigate and correct failed NC sequences.
- Investigate and correct failed NC sequences.



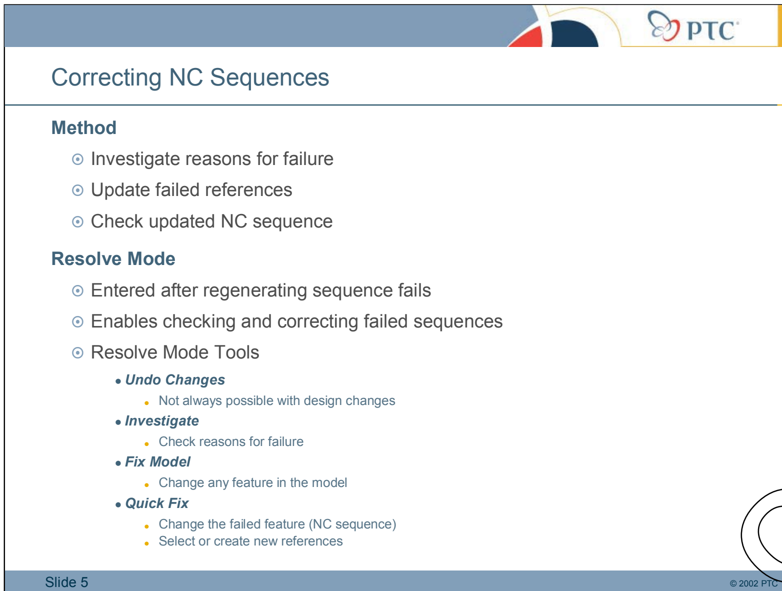
Reasons for NC Sequence Failure

Changes to reference models can cause NC sequences to fail, in most cases the NC sequence updates automatically.

If references are completely removed from the model, then new references will need to be specified, for example:

- Facing or Profiling surfaces
- Holes for drilling
- Mill window outlines
- Mill Volume surfaces

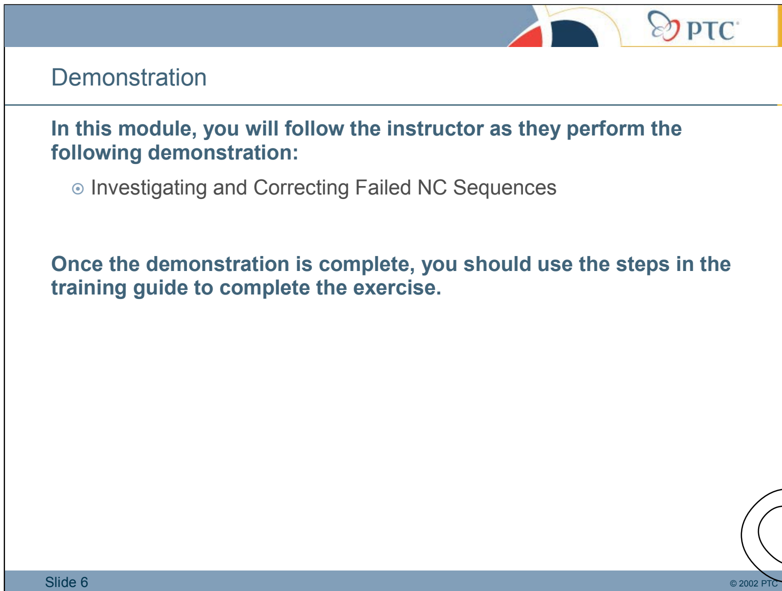
- Changed references can sometimes cause NC sequences to fail, but the changes to geometry shape and size normally need to be extreme.
- Pro/ENGINEER always attempts to update NC sequences in these situations. NC sequences will only fail if updating is not possible.



Correcting NC Sequences

- The method for correcting NC sequences, involves three steps:
  - Investigate the reasons for the NC sequence to fail, determine the missing or failed references, e.g. missing surfaces, holes, or edges.
  - Update the failed references, e.g. select new final surface for profile milling, or hole axis for holemaking.
  - Check the updated NC sequence performs correctly, usually by playing the toolpath.

- When an NC sequence fails you automatically enter resolve mode.
  - This provides a set of tools for checking and correcting failed sequences, the main tools are:
    - Undo Changes: undo all the changes since the last regeneration, not always possible if the reference model has already been saved in the changed state.
  - Investigate: check the reasons for the failure, usually missing references
  - Fix Model: change any feature in the model, for example you may decide to change reference model features in order to correct the problem.
  - Quick Fix: change the failed feature (NC sequence), change any of the references in the failed sequence, for example select new geometry references, update NC sequence parameters, or change tools.



Demonstration: Investigating and Correcting Failed NC Sequences

Duration: 25 minutes

Scenario

- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

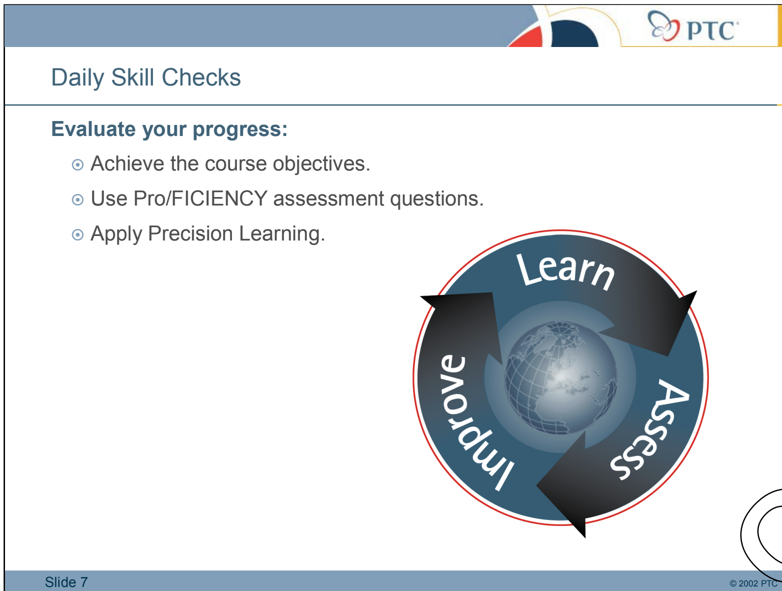
Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Duration: 50 minutes

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY.
- **Improve:** The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Getting Started

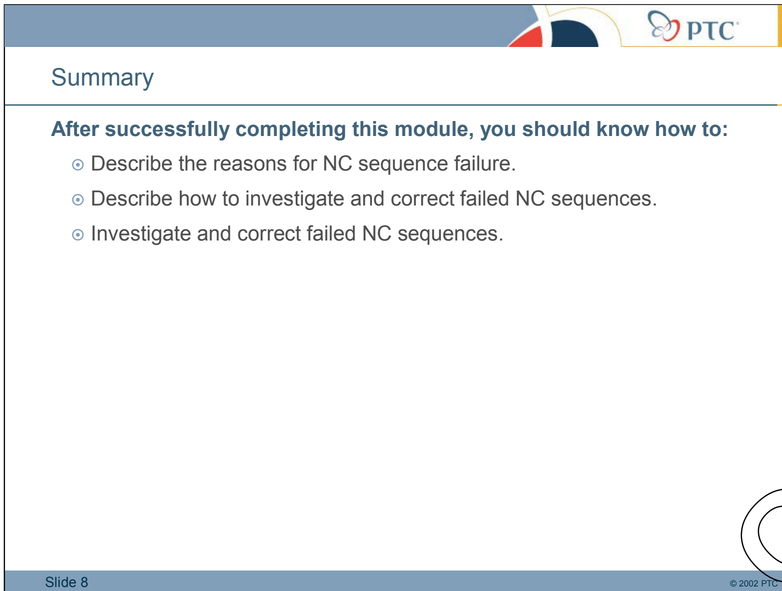
- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Tests

Description: For each course, 5 new 10 question exams based upon the topics covered each day.

How to use it:

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After successfully completing this module, you should know how to:

- Describe the reasons for NC sequence failure.
- Describe how to investigate and correct failed NC sequences.
- Investigate and correct failed NC sequences.

Module 16 Lab Exercises

Exercise 1: Investigating and Correcting Failed NC Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you open the updated reference model of the cover and review the design changes. You then open the manufacturing model of the cover, and for each feature that fails you investigate, identify and then update the missing references. It is important to understand why NC sequences fail, and how to investigate and correct failed NC sequences.

Objectives

After successfully completing this exercise, you will know how to:

- Investigate and correct failed NC sequences.

Scenario

A number of design changes occur on the aluminum cover, causing NC sequences to fail in the manufacturing model of the cover. You open the updated reference model of the cover and review the design changes. You then open the manufacturing model of the cover, and for each feature that fails you investigate, identify and then update the missing references.

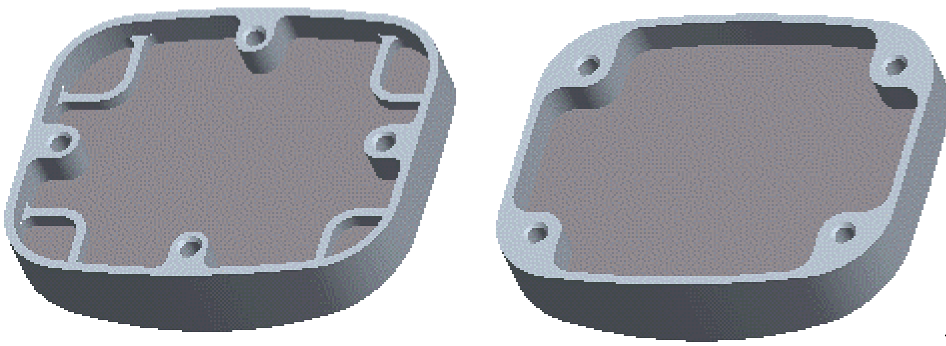


Figure 1: Showing the bottom view of original and modified cover model.

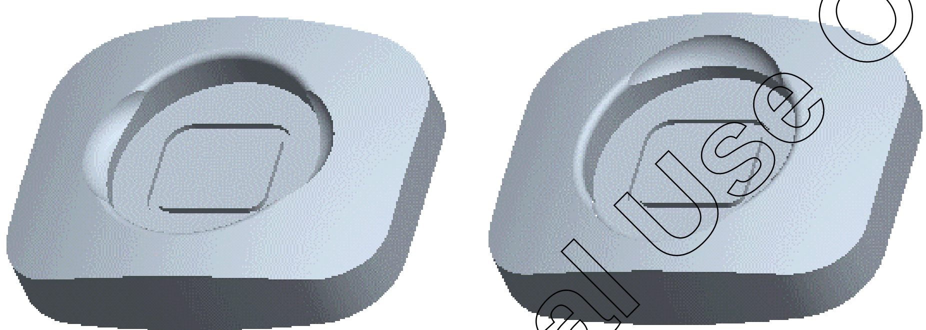


Figure 2: Showing the top view of original and modified cover model.

Step 1. Check the design changes in the cover reference model.

Before checking the design changes, we need to locate the reference model and open it using the folder navigator. When we have located the folder for this exercise, we can set this as our working directory, and open the reference model of the cover. Notice on the bottom of the cover the position of the drilled holes has been changed and the ribs have been removed. However, we can check that the size of the drilled holes has not changed. Notice on the top of the cover the original cut outs have been removed and replaced by new cut outs. In both cases, the design engineer has deleted the original features and replaced them with new features.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module16**, right-click, and select **Make Working Directory.**

2. Open COVER.PRT. Turn off any displayed datum features.

3. Change the view to the named view **BOTTOM-TRI**. Observe the design changes to the model. Check the size of the repositioned holes.

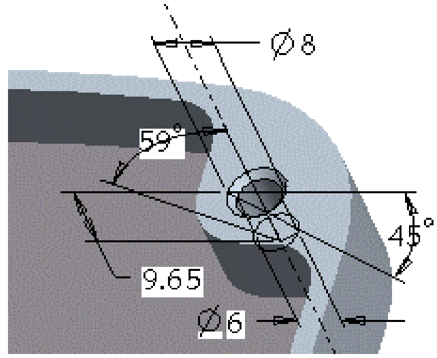


Figure 3: Showing hole size

4. Change the view to the **default** view. Observe the design changes to the model.

5. Close the window.

> **Note:**
> Notice on the bottom of the cover the position of the drilled holes has been changed and the ribs have been removed. Notice on the top side of the cover the original cut-outs have been removed and replaced by new cut-outs. In both cases, the design engineer has deleted the original features and replaced them with new features.

Step 2. Open the manufacturing model and correct the failed features in operation op010.

Before opening the cover manufacturing model, we should be aware that the model is likely to fail regeneration, and we will need to correct the failed features.

Opening the manufacturing model, we can first add the status column to the model tree. This provides information on which features have regenerated successfully in the manufacturing model. Notice in the model tree the mill window feature named WND_RGH_BOT has failed regeneration; also the failure diagnostics window states that feature references are missing. We can also hide the vise assembly to make the viewing of both sides of the model easier.

We can investigate the reasons for the feature failing and list the failed references in the reference information window. We can unhide the failed mill window feature in the model tree and redefine the references for the mill window. We can select the inner set of tangent chain edges at the top of the pocket, and preview the result. When we complete the mill window configuration, regeneration of the model continues automatically.

Notice in the model tree the holemaking NC sequence named CTR_DRILL has failed regeneration; this is the first of three holemaking features. We can use the Resolve Feature menu options to investigate and correct this problem by redefining the references of the failed feature. We can also check to see if the correct tool is being used, and then using the Hole Set dialog box we can use the Diameter tab and select the 6.0 diameter holes, this automatically selects all the 6.0-millimeter diameter holes on the model. As this is a center drilling sequence we can specify *the depth options as blind and tip and specify a depth of 5.0. By playing the tool path and making the CL data visible, we can see the tool drills the four holes to a depth of 5.0 as specified. When we complete the holemaking sequence, regeneration of the model continues automatically.*

Notice in the model tree, the holemaking NC sequence named DRILL_06 has now failed regeneration. We can correct this problem by redefining the references of the failed feature. We can also check to see if the correct tool is being used, and then using the Hole Set dialog box we can use the Use Prev option and select the first hole set. As this is a standard drilling sequence we can specify the depth options as auto and tip, this completes the hole set configuration. By playing the tool path and making the CL data visible we can see the tool drills the four holes, automatically calculating the correct depth.

Notice the next countersink holemaking sequence does not fail regeneration, this is because the sequence was configured to automatically drill all chamfered holes on the model.

1. Open COVER.MFG.

> **Note:**
> The manufacturing model has failed regeneration due to the design changes. You have been placed immediately into the Resolve Feature menu. A Failure Diagnostics window also provides information about the failed feature.

2. Move the position of the Failure Diagnostics window to make the model tree clearly visible.

3. In the model tree, click **Settings > Tree Columns**. In the Model Tree Columns dialog box, select the **Status** option and move it to the display column. Close the dialog box.

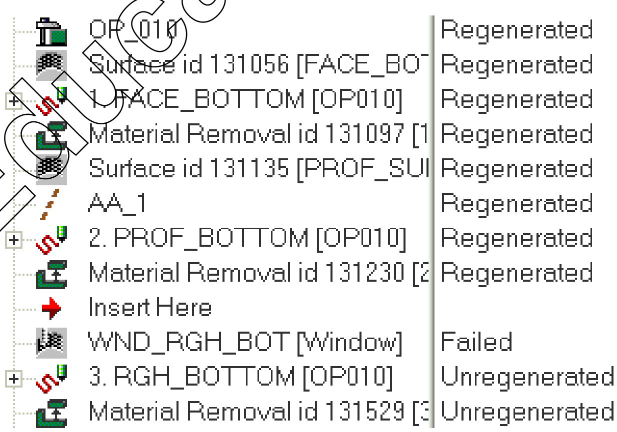


Figure 4: Showing the model tree with regeneration status column added.

4. Adjust the width of the model tree columns to make the feature names and regeneration status clearly visible. Note the failed feature is the mill window named WND_RGH_BOT.

> **Note:**
> Displaying the status column in the model tree provides information on which features have regenerated successfully in the COVER.MFG manufacturing model. Notice in the model tree the mill window feature named WND_RGH_BOT has failed regeneration, also the failure diagnostics window states that feature references are missing. The Resolve Feature menu options enable investigation and correction of failed features.

5. Read and minimize the Failure Diagnostics window. Change the view to **OP010-TRI**.

6. In the model tree, select the vise assembly, OP010_INST.ASM, right-click and **Hide** it from display.

> **Note:**
> Hiding the vise assembly makes it easier to view both sides of the model.

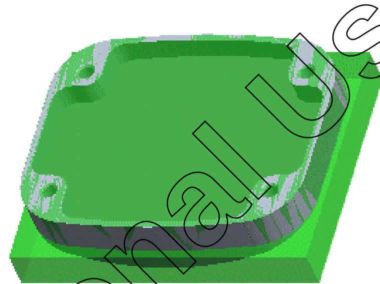


Figure 5: Showing the manufacturing model in resolve mode.

7. Using the Menu Manager, select **Investigate > Show Ref** to show the references of the failed feature.

> **Note:**
> Notice in the reference information window the missing references for the mill window are listed in the COVER.PRT model.

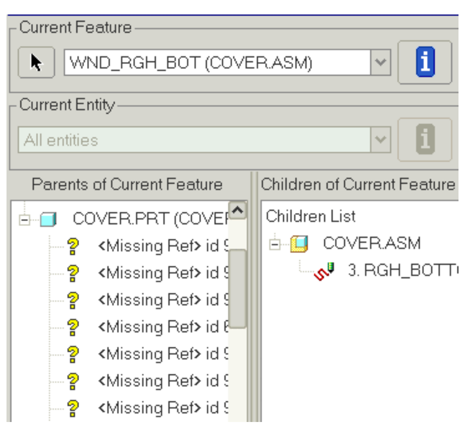


Figure 6: Showing a listing of missing references.

8. Close the Reference Information window.

9. Select the WND_RGH_BOT mill window feature in the model tree, right-click and **Unhide** it.

10. Using the Menu Manager, select **Quick Fix > Redefine**, and **Suspend All** child features.

11. Define the missing window references. Click the **Window** element and **Define** in the dialog box. Using the **Select** and **Tangent Chain** options, select the inner chain at the top of the pocket, then complete the selection.

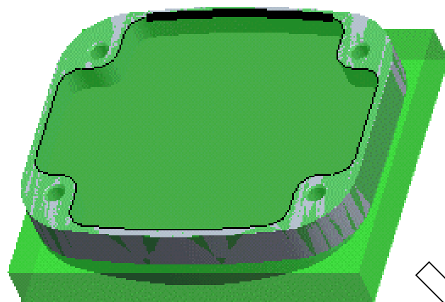


Figure 7: Showing the selected edge and highlighted tangent chain of edges.

12. Click the **Preview** button in the mill window.

> **Note:**
> Notice that the mill window outline has updated correctly and is created on the retract plane. Once you complete the mill window configuration regeneration of the model continues automatically.

13. Complete the mill window configuration.

Step 3. Correct the failed features in operation op020.

Notice in the model tree, the next sequence to fail is the surface milling sequence named FIN_CUT_OUTS in operation op020. This is because the reference surfaces for this NC sequence have been removed. First we need to change to the op020-tri view. We can correct this problem by redefining the references of the failed feature. Firstly, we can also check to see if the correct tool is being used. Then for the surfaces to be machined, select the two new cut out surfaces on the reference model, we also need to adjust the cut direction for each surface to ensure it is pointing in the desired direction. We can play the tool path and notice it has updated correctly. When we complete this NC sequence, notice that all the features now regenerate successfully.

You can continue by reviewing the other NC sequences to see if they need any adjustment.

Note:

Notice in the model tree, the next sequence to fail is the surface milling sequence named FIN_CUT_OUTS in operation op020. This is because the reference surfaces for this NC sequence have been removed.

1. Read the information in the Failure Diagnostics window, and then minimize the window.

```
- Reference surfaces are no longer available.
- Feature references are missing.
```

Figure 8: Showing information in the Failure Diagnostics window.

2. Change the view to the named view **OP020-TRI**.

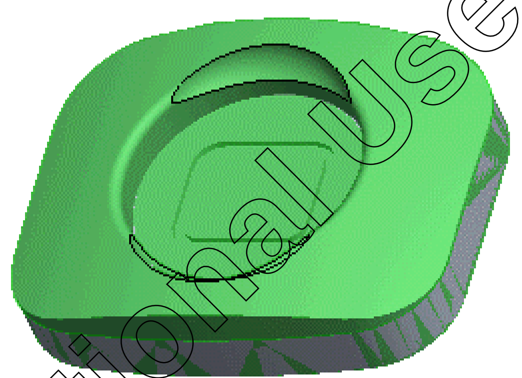


Figure 9: Showing the new cut outs in the top of the model.

3. Select **Quick Fix > Redefine**, and select the **References** check box.
4. Select the **Tool** check box; note the surfaces and define cut check box are already selected.
5. Check the BEM_12_0 tool is being used, and close the Tool Setup dialog box.
6. Accept the **Model** option, and for the surfaces to be machined, select the two new cut out surfaces on the reference model, shown in the following figure.
7. In the Cut Definition dialog box, select the first surface in the list and toggle the Direction of Cut icon as shown. Select the second surface in the list and again toggle the Direction of Cut icon as shown. Complete the cut definition.

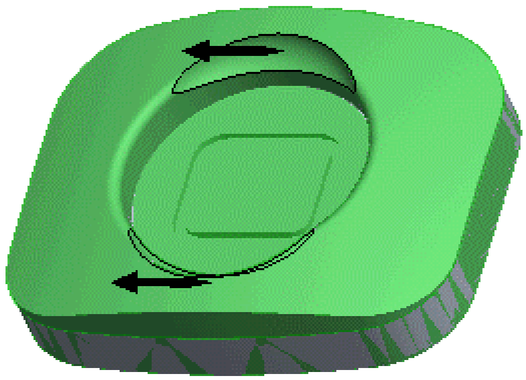


Figure 10: Showing selected surfaces and cut direction.

8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible

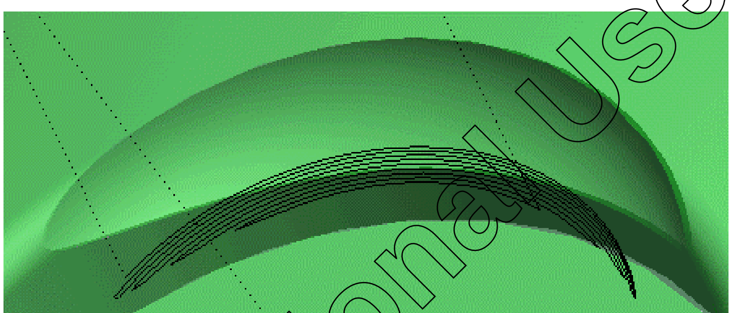


Figure 11: Showing one cut out surface being machined.

9. Complete the sequence, and select **Yes** to exit the Resolve Feature menu.

> **Note:**
> Notice all the features now regenerate successfully.

10. Continue by reviewing and checking the other NC sequences.

11. Save the manufacturing model.

12. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the reasons for NC sequence failure.
- Describe how to investigate and correct failed NC sequences.
- Investigate and correct failed NC sequences.

Module

17

Configuring the Manufacturing Environment

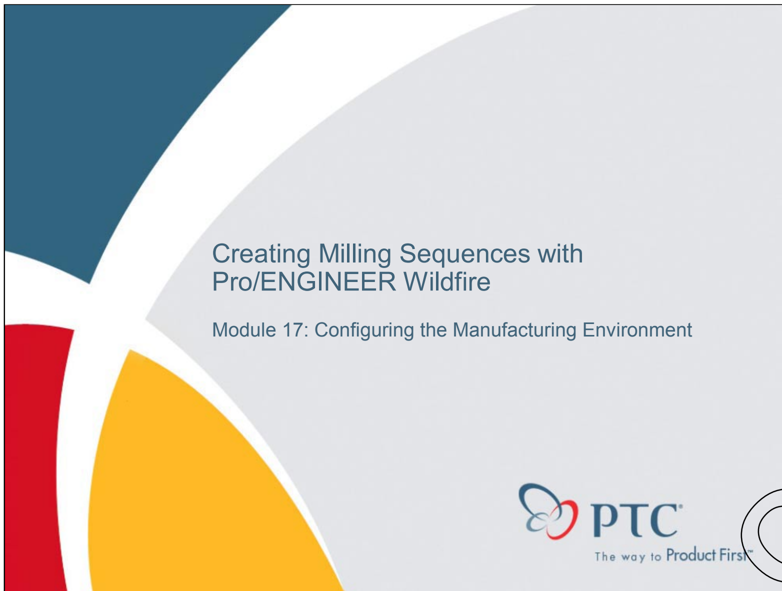
Introduction

There are many manufacturing configuration options that enable you to customize your working environment. For example, you can configure tool libraries and directories for workcells and manufacturing parameters, this enables you to store and then easily retrieve and use these items. Understanding manufacturing configuration options and methods is therefore an essential element of the manufacturing process, and enables you to create a more efficient working environment.

Objectives

After completing this module, you will be able to:

- Describe manufacturing configuration options.
- Create configuration options to store tools, material files, workcells, parameter files, and stock billets.
- Create configuration options to enhance your working environment.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

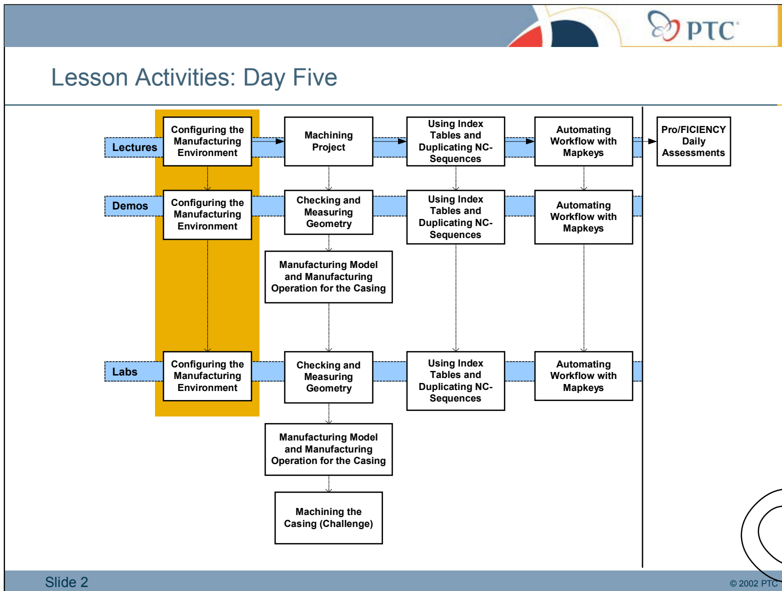
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

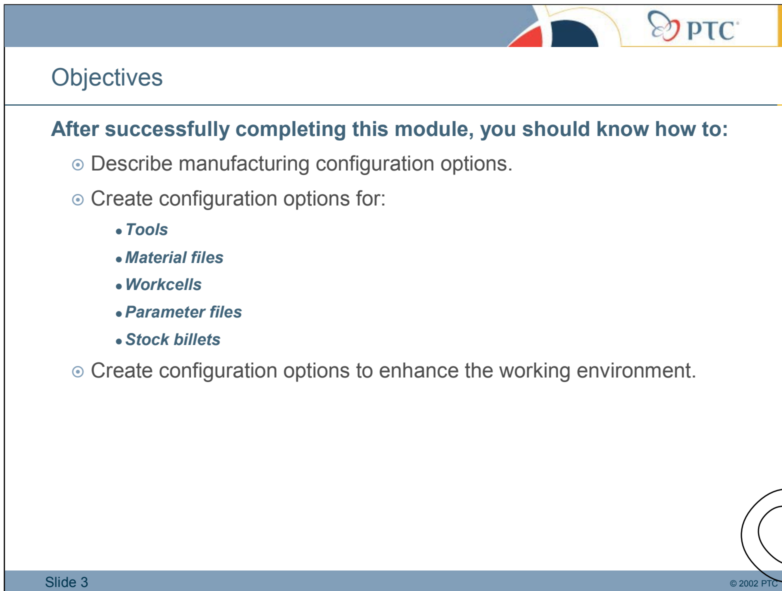
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire,* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 10 mins
- Demos : 20 mins
- Labs : 45 mins
- Total: 1 hr 15mins

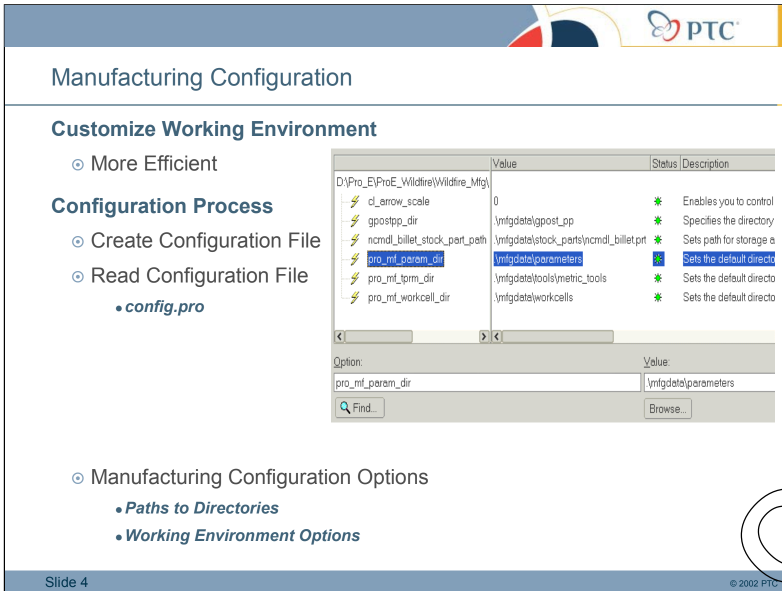


Overview

There are many manufacturing configuration options that enable you to customize your working environment. For example you can configure tools libraries, and directories for workcells and manufacturing parameters, this enables you to store and then easily retrieve and use these items. Understanding manufacturing configuration options and methods is therefore an essential element of the manufacturing process, and enables you to create a more efficient working environment.

After completing this module, you will be able to:

- Describe manufacturing configuration options.
- Create configuration options to store tools, material files, workcells, parameter files, and stock billets.
- Create configuration options to enhance the working environment.



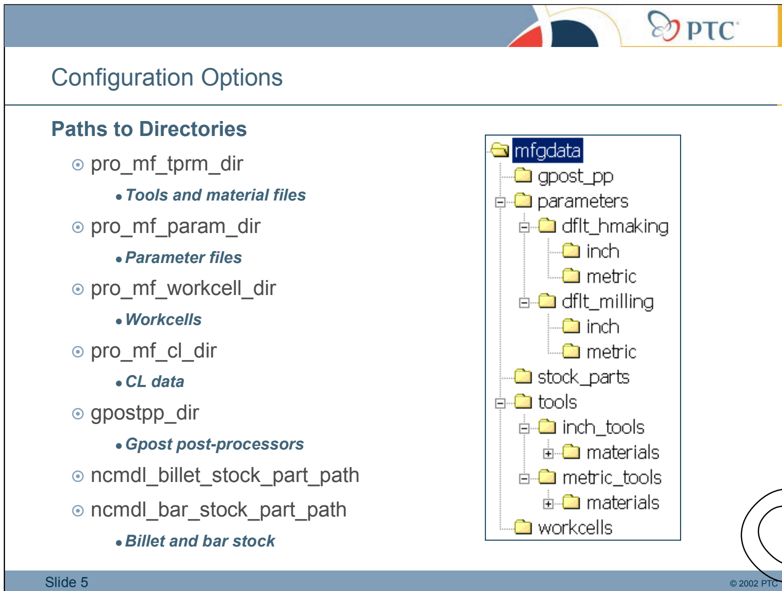
Manufacturing Configuration

- Manufacturing configuration options enable you to create a customized and more efficient working environment.

- The configuration process involves creating and reading a configuration file named config.pro.

The manufacturing configuration options can be broken into two broad categories:

- Options that specify paths to directories, for example a tools directory.
- Options that change the working environment, for example to specify the number of decimal places for X, Y, Z co-ordinates in CL data files.



Configuration Options

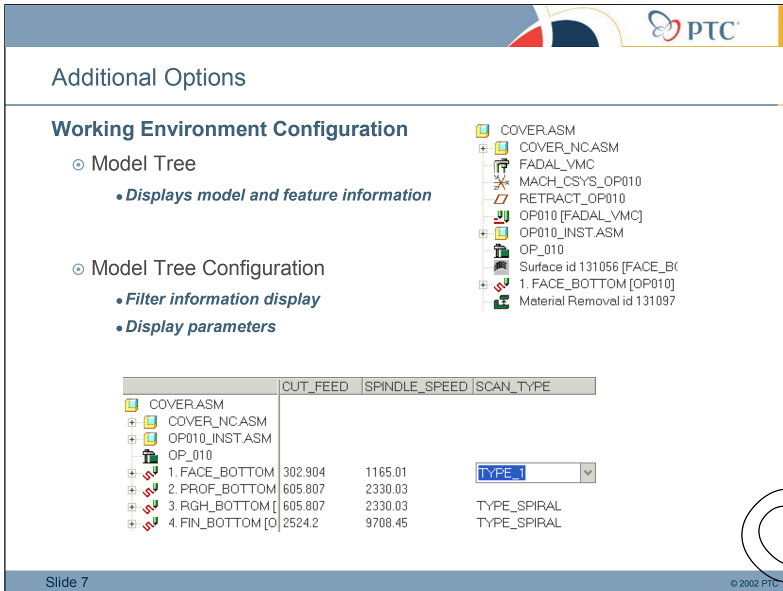
The main set of configuration options relating to paths to directories, this list is not comprehensive but provides the most commonly used options.

- Pro_mf_tprm_dir: specifies the default directory to store tool files
- Pro_mf_param_dir: specifies the default directory to store manufacturing parameter files
- Pro_mf_workcell_dir: specifies the default directory to store workcell files
- Pro_mf_cl_dir: specifies the default directory to store CL data files
- Gpostpp_dir: specifies the directory containing the post-processor files to be used by GPOST
- ncmdl_billet (and bar)_stock_part_path: specifies path to user defined default billet and bar stock

Configuration Options

The main set of configuration options relating to the working environment, this list is not comprehensive but provides the most commonly used options.

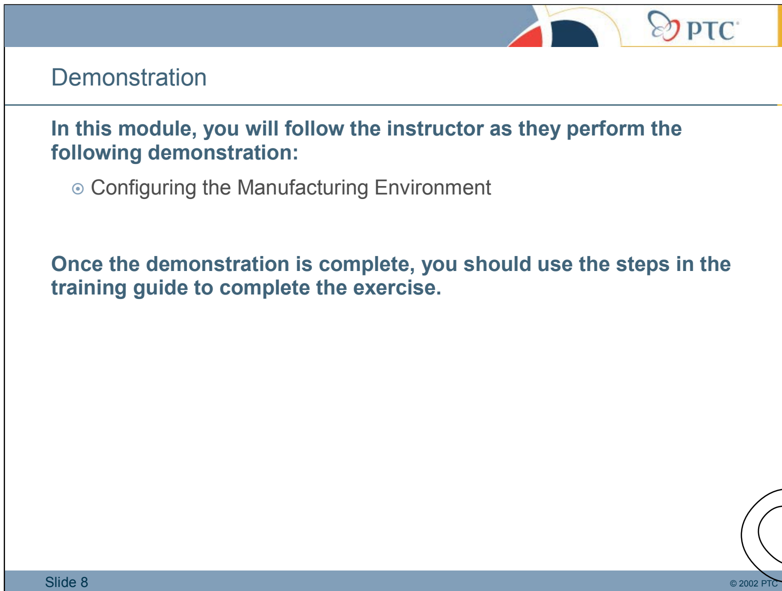
- mfg_workpiece_transparency: specifies workpiece transparency from 0.1 to 1(transparent), default is 0.5
- cl_arrow_scale: specifies size of toolpath arrow when displaying CL data.
- ncpost_type: specifies which post-processing module to use, default is gpost.
- nccheck_type : specifies which toolpath simulation module to use, default is Vericut.
- mfg_xyz_num_digits and mfg_ijk_num_digits: specifies the number of digits for xyz and ijk co-ordinates in cL data files, default is 10.
- store_tool_path_file: specifies if toolpath information is stored when creating an NC sequence, if used reduces time needed to regenerate the toolpath.



Additional Options

The model tree displays model and feature information.

Additional configuration of the working environment is possible by filtering information displayed in the model tree, and by displaying parameters for NC sequences in the model tree. Displayed parameters can be modified directly in the model tree.



Demonstration: Configuring the Manufacturing Environment

Duration: 20 minutes

Scenario

- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

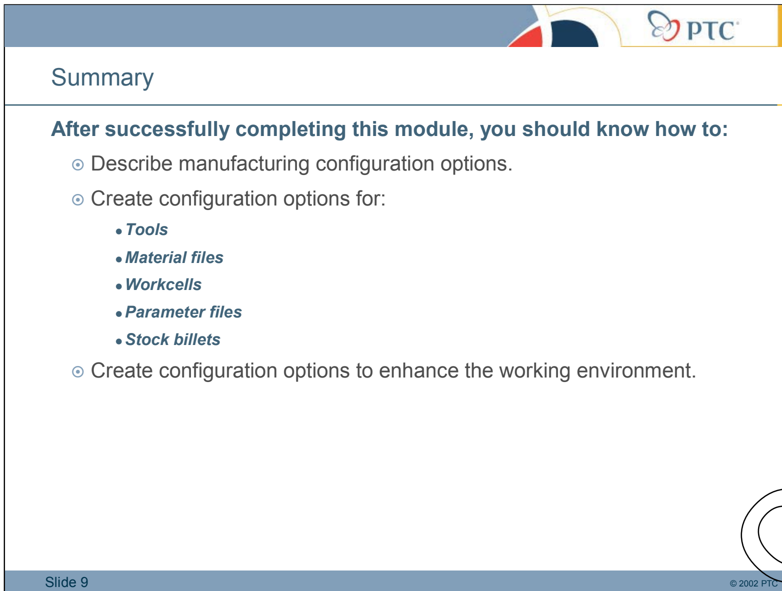
Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Duration: 45 minutes

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe manufacturing configuration options.
- Create configuration options to store tools, material files, workcells, parameter files, and stock billets.
- Create configuration options to enhance the working environment.

Module 17 Lab Exercises

Exercise 1: Configuring the Manufacturing Environment

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003051 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you need to setup a manufacturing environment for a new computer. The company already has a set of manufacturing standards, including tools, material files, workcells, and parameter files. You need to ensure that these configuration options work correctly on the new PC. Understanding manufacturing configuration options and methods is an essential element of the manufacturing process, and enables the creation of a more efficient working environment.

Objectives

After successfully completing this exercise, you will know how to:

- Create configuration options to store tools, material files, workcells, parameter files, and stock billets.
- Create configuration options to enhance your working environment.

Scenario

You need to setup a manufacturing environment for a new computer in your manufacturing group. Your company already has a set of manufacturing standards, including tools, material files, workcells, and parameter files. You need to ensure that these configuration options work correctly on the new computer. You check the contents of the folders containing the manufacturing standards, and then create a configuration file to utilize these standards. You test the configuration is working correctly by configuring a manufacturing operation, creating a face milling sequence, and post-processing a CL data file. You then enhance the working environment by configuring the model tree to display manufacturing parameters. You make the parameters more easily visible by moving the position of the model tree to the bottom of the graphics window; this is achieved by customizing the screen layout.

Step 1. Open the example manufacturing model.

Before configuring the manufacturing environment, we need to locate the example manufacturing model and open it using the folder navigator. Once we have located the folder for this exercise, we can set this as our working directory, and open the example manufacturing model.

1. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module17**, right-click, and select **Make Working Directory.**
2. Open EXAMPLE.MFG. Turn off any displayed datum features.

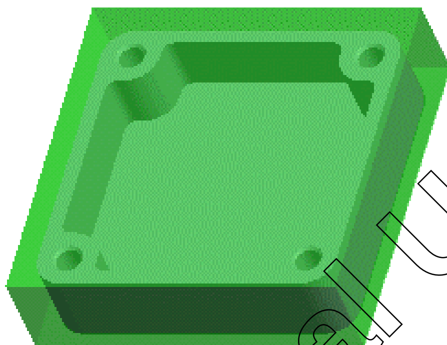


Figure 1: Showing the example manufacturing model.

> **Note:**
> Notice the reference model and workpiece have already been assembled into the manufacturing model.

Step 2. Check the contents of the mfgdata folder.

We need to check the contents of the mfg data folder that contains all the manufacturing configuration files. We can display the folder navigator, and select the mfgdata folder. We can expand all the sub-folders within the mfgdata folder, except the two materials folders as the listings in these folders is too long. We can also drag the navigator sash to make all the folder names visible.

We can see the company post processors are contained in the gpost_pp folder. We can specify the location of this folder later with a config.pro option. We can see that all the commonly used manufacturing parameters are stored within the parameters folder. Notice there are sub-folders for holemaking and milling parameters, and for imperial (inch) and metric units. There is also a folder for site parameter files. We can also select and open the face.mil parameter file to view the contents.

We can see that the stock_parts folder contains a user defined metric billet stock part, which is used with the Create Stock dialog box. User defined bar stock can also be configured if required.

We can see that all commonly used tools are stored within the tools folder. Notice there are sub-folders for imperial (inch_tools) and metric tools. We can also select and open the MMBUL10.tpm tool file to view the contents.

We can select the materials folder within the metric_tools folder and see within each materials folder there are sub-folders for different types of material. Each material type folder contains an equivalent tool file containing feed, speed and depth-of-cut data for each tool. This information can be used when configuring manufacturing parameters. We can open the STEEL-100BHN folder within the browser and select and open the MMBUL10.tpm material file to view the contents. Notice the feed, speed, and depth-of cut data, this is based on the tool diameter, number of teeth, and the workpiece material, in this case steel-100bhn.

We can see that the workcells folder contains the workcell configuration files.

Finally, we can see that the parameters folder also serves as the default folder for PPRINT files.

1. Click the Folder Browser tab to the left of the graphics window. Drag the navigator sash until the folder names are clearly visible.

2. Expand the module17 folder, and the mfgdata folder, and then expand the sub-folders within the mfgdata folder, as shown in the following figure. (Do not expand the two materials folders). Drag the navigator sash again until the folder names are clearly visible.

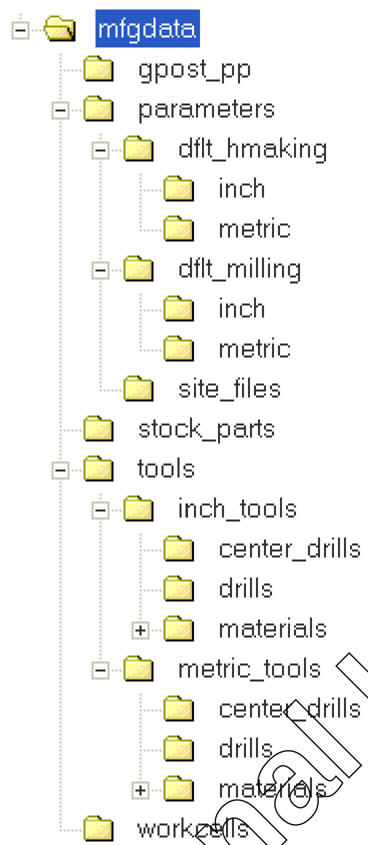


Figure 2: Showing the contents of mfgdata folder.

3. Select the **gpost_pp** folder and check its contents.

> **Note:**
> The gpost_pp folder contains the company post processors. We can specify the location of this folder with a config.pro option.

4. Within the parameters\dflt_milling folder select the **metric** folder. Check the contents of the folder in the browser.

> **Note:**
> Commonly used manufacturing parameters are stored within the parameters folder. Notice there are sub-folders for holemaking and milling parameters, and for imperial (inch) and metric units. There is also a folder for site parameter files.

File Name	Size	Modified
conv_surf.mil	1 KB	06-Jan-02 09:35:02 AM
face.mil	1 KB	06-Jan-02 09:26:00 AM
flatface.mil	1 KB	06-Jan-02 09:35:18 AM
pencil.mil	885 bytes	06-Jan-02 09:35:30 AM
profile.mil	865 bytes	06-Jan-02 09:35:42 AM
reroughing.mil	1 KB	06-Jan-02 09:30:30 AM
roughing_cavity.mil	1 KB	10-Jan-02 08:17:40 AM
roughing_core.mil	1 KB	06-Jan-02 09:32:08 AM
surf_cutline.mil	1 KB	06-Jan-02 09:32:42 AM
surf_iso.mil	981 bytes	06-Jan-02 09:33:28 AM
surf_projected.mil	1 KB	06-Jan-02 09:33:44 AM
surf_window.mil	486 bytes	06-Jan-02 09:33:50 AM

Figure 3: Showing parameter files in the metric folder.

5. Select the **face.mil** file and then right-click and select **Open** to view the contents of the file.

```
SCAN_TYPE              TYPE_3
CUT_TYPE               CLIMB
STEPOVER_ADJUST        YES
CUSTOMIZE_AUTO_RETRACT YES
STEP_DEPTH             CUTTER_DIAM/5
TOLERANCE              0.01
STEP_OVER              CUTTER_DIAM*.9
NUMBER_PASSES          0
ONE_PASS_OFFSET        0
BOTTOM_STOCK_ALLOW     0
```

Figure 4: Showing contents of the face.mil parameter file.

6. Click the Back arrow icon at the top of the browser to return to the contents of the metric folder.

7. Select the **stock_parts** folder, and check the contents of the folder in the browser.

8. Select the **ncmdl_billet.prt** file in the browser, wait for the part to appear in the preview window.

Note:

The stock_parts folder contains a user defined metric billet stock part, which is used with the Create Stock dialog box. User defined bar stock can also be configured if required.

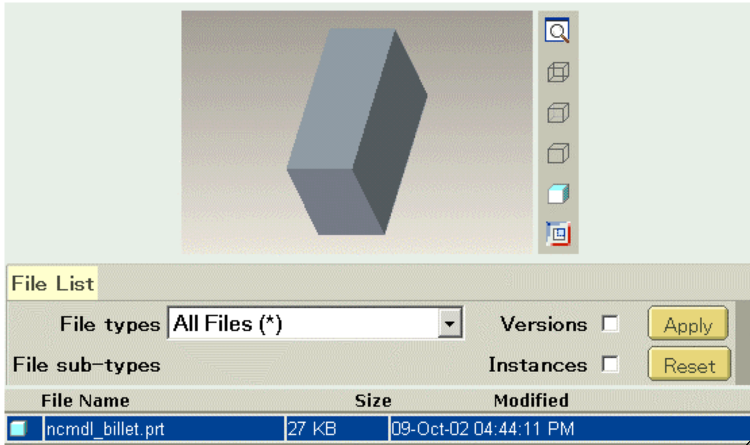


Figure 5: Showing the ncmdl_billet part.

9. Within the tools folder select the **metric_tools** folder. Check the contents of the folder in the browser.

10. Select the **MMBUL10.tpm** tool file and then right-click and select **Open** to view the contents of the tool file.

```
TOOL_ID              MMBUL10
TOOL_TYPE            MILLING
LENGTH_UNITS         MILLIMETER
CUTTER_DIAM          10.00000
CORNER_RADIUS        2.50000
SIDE_ANGLE           0.0
LENGTH               50.0
NUM_OF_TEETH         2
TOOL_MATERIAL        HSS
GAUGE_X_LENGTH       -
GAUGE_Z_LENGTH       -
COMP_OVERSIZE        -
TOOL_COMMENT         -
TOOL_LONG_FLAG       NO
```

Figure 6: Showing the contents of MMBUL10.tpm tool file.

11. Click the Back arrow icon at the top of the browser to return to the contents of the metric_tools folder.

12. Repeat steps 10 and 11 for the MMFLT10.tpm, and MMSPH10.tpm tool files.

> **Note:**
> All commonly used tools are stored within the tools folder. Notice there are sub-folders for imperial (inch_tools) and metric tools.

13. Within the metric_tools folder select the **materials** folder. Check the contents of the folder in the browser.

Note:

Notice also in each tools folder there is a materials sub-folder, and within each materials folder there are sub-folders for different types of material. Each material type folder contains an equivalent tool file containing feed, speed and depth-of-cut data for each tool based on workpiece material. This information can be used when configuring manufacturing parameters.

File Name	Size	Modified
ALUMINIUM		10-Dec-02 00:21:03 AM
AUSTEN		09-Dec-02 02:52:37 PM
BRASS-HARD		09-Dec-02 02:52:37 PM
BRASS-MEDIUM		09-Dec-02 02:52:36 PM
CASTIRON-HARD		09-Dec-02 02:52:36 PM
CASTIRON-MEDIUM		09-Dec-02 02:52:36 PM
COBALT		09-Dec-02 02:52:36 PM
COPPER		09-Dec-02 02:52:36 PM
FERR		09-Dec-02 02:52:36 PM
NICKEL		09-Dec-02 02:52:36 PM
STAINLESS-HARD		09-Dec-02 02:52:36 PM
STAINLESS-MEDIUM		09-Dec-02 02:52:36 PM
STEEL-100BHN		09-Dec-02 02:52:36 PM

Figure 7: Showing contents of metric_tools\materials folder

14. Double-click the STEEL-100BHN folder within the browser. Check the contents of the folder.

15. Select the **MMBUL10.tpm** material file and then right-click and select **Open** to view the contents of the tool material file.

```
CUT_DATA_UNITS       METRIC
APPLICATION_TYPE     ROUGHING
TOOL_SPINDLE_RPM     255
TOOL_RADIAL_DEPTH    4.8
TOOL_SURFACE_SPEED   8000
TOOL_FEED_RATE       51
TOOL_FEED_PER_UNIT   0.2
TOOL_AXIAL_DEPTH     8
APPLICATION_TYPE     FINISHING
TOOL_SPINDLE_RPM     255
TOOL_RADIAL_DEPTH    1
TOOL_SURFACE_SPEED   8000
TOOL_FEED_RATE       51
TOOL_FEED_PER_UNIT   0.2
TOOL_AXIAL_DEPTH     0.5
```

Figure 8: Showing contents of MMBUL.tpm tool material file.

Note:

Notice the feed, speed, and depth-of cut data, this is based on the tool diameter, number of teeth, and the workpiece material, in this case steel-100bhn.

16. Click the Back arrow icon at the top of the browser to return to the contents of the STEEL-100BHN folder.

17. Repeat steps 15 and 16 for the MMFLT10.tpm, and MMSPH10.tpm tool material files.

18. Compress the expanded materials folder in the navigator.

19. Within the navigator select the **workcells** folder, and check the contents of the folder in the browser.

Figure 9: Showing the contents of workcells folder.

> **Note:**
> The workcells folder contains the workcell configuration files.

20. Within the navigator select the **parameters** folder, and check the contents of the folder in the browser.

File Name	Size	Modified
pprint.ppr	2 KB	10-Dec-02 00:23:50 AM
dflt_hmaking		09-Dec-02 02:08:33 PM
dflt_milling		09-Dec-02 02:08:32 PM
site_files		10-Dec-02 10:27:45 AM

Figure 10: Showing the contents of parameters folder.

21. Select the **pprint.ppr** file and then right-click and select **Open** to view the contents of the PPRINT file.

22. Click the Back arrow icon at the top of the browser to return to the contents of the parameters folder.

> **Note:**
> The parameters folder also serves as the default folder for PPRINT files.

23. Collapse the browser window so that the model in the graphics window is visible.

Step 3. Create a configuration file with manufacturing options.

We need to create a config.pro file with a number of manufacturing options. Using the tool bar, we can open the Options dialog box. Notice as we type in each option we are prompted to complete the option name, always ensure the correct option names are used. For each option, *we can type in a value, set the cl_arrow_scale to zero, and specify default directories for post-processors, billet stock parts, manufacturing parameters, tools and workcells. When complete, we can apply the changes and save the config.pro file for future reference.*

1. In the menu bar at the top of the graphics window, click **Tools** > **Options** from the drop-down menu.

2. In the Options text box, enter the first option listed in table 1 below (cl_arrow_scale), in the Value text box enter [**0**]. Click the **Add/Change** button to add the option.

> **Note:**
> Notice as you type in each option the option name is completed for you.

3. Create the remaining options and values listed in table 1.

Table 1: Config.pro options

Option	Value
cl_arrow_scale	0
gpostpp_dir	.\mfgdata\gpost_pp
ncmdl_billet_stock_part_path	.\mfgdata\stock_parts\ncmdl_billet.prt
pro_mf_param_dir	.\mfgdata\parameters
pro_mf_tprm_dir	.\mfgdata\tools\metric_ tools
pro_mf_workcell_dir	.\mfgdata\workcells

> Note:
> For the purposes of simplifying this exercise all directory paths are given as relative to the current directory. It is recommended that absolute paths are used in a work situation.

4. **Apply** the changes, then click the Save File icon, and name it [**config.pro**]. **Close** the options window.

Step 4. Test the new configuration options; create a manufacturing operation.

We can now test the configuration options. We can open the Operation Setup dialog box, and the Machine Tool Setup dialog box; and then retrieve the hurco workcell, and PPRINT file. Notice how the system automatically uses the workcells and parameters folders. We can also retrieve the RGH_3_AXIS_MIL.sit site parameter file from the site_files folder. We can select the workpiece material from the Stock Material drop-down list; the list of materials corresponds to the material sub-folders within the tools folder.

We can then configure a program zero co-ordinate system and name it. We can also configure a retract surface, complete the operation and name the retract surface in the model tree.

1. In the navigator, select the Model Tree tab to make the model tree visible again.

2. Using the Menu Manager, select **Mfg Setup > Operation** to open the Operation Setup dialog box.

3. Open the Machine Tool Setup dialog box, click the Retrieve Machine icon and select the **hurco.gph** workcell.

4. In the Machine Tool Setup dialog box, select **PPRINT** and **Retrieve** the **pprint.ppr** file.

> **Note:**
> The config.pro options pro_mf_workcell_dir and pro_mf_param_dir are specifying the folders for workcells and PPRINT files.

5. In the Machine Tool Setup dialog box, select **Defaults**.

6. Using the **Current Directory** and **Mill** options, double-click the **site_files** folder, then select the **RGH_3_AXIS_MIL.sit** site parameter file.

7. Close the Machine Tool Setup dialog box.

8. In the Operation dialog box, select **STEEL-100BHN** in the Stock Material drop-down list.

> **Note:**
> The names of the folders in the metric_tools\materials folder specifies the list of materials to choose from.

9. Turn on the display of co-ordinate systems, if not already visible in the graphics window.

> **Note:**
> You can turn off the display of the blended background in the graphics window to make viewing of co-ordinate system axes easier.

10. Click **View > Display Settings > System Colors**, clear the Blended Background check box, and click **OK** to complete the configuration.

11. Create a machine zero co-ordinate system positioned at the front-left-top corner of the workpiece. Set the orientation of the axes as shown in the following figure. Rename the co-ordinate system to [**mach_csys_010**].

> **Tip:**
> When prompted create the co-ordinate system in the EXAMPLE.ASM assembly, select this assembly in the model tree. Refresh the graphics window before selecting the surfaces that configure the position of the co-ordinate system.

12. Create a retract surface along the Z-axis of the machine zero co-ordinate system, type [**30**] *(1.18 in)* as the z distance.

13. Close the Operation Setup dialog box.

14. Rename the retract surface to [**`retract_010`**] in the model tree.

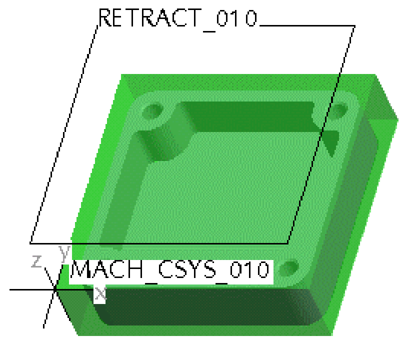


Figure 11: Showing program zero co-ordinate system and retract plane.

15. Turn on the blended background again. Click **View > Display Settings > System Colors**, select the Blended Background check box, and click **OK** to complete the configuration.

Step 5. Create a face milling sequence, retrieve a tool from the tool library, and retrieve a set of standard face milling parameters.

We start by creating a face milling sequence. We can retrieve a tool from the metric_tools folder; notice how by default we start searching for tools in the metric_tools folder.

We can review the manufacturing parameter values, notice how the site parameter file sets all the required manufacturing parameters; this includes reading in the speed, feed and depth-of-cut data for this tool.

We can now select the top surface of the part to machine, and play the toolpath. Notice how the toolpath follows the edge of the workpiece.

As an alternative to using site parameter files we can now retrieve the face.mil parameter file, located in the dflt_milling\metric folder. We can also change the TRIM_TO_WORKPIECE parameter back to YES to see the effect on the toolpath. Playing the toolpath we notice the dept-of-cut and step-over are different.

In summary, we can see that parameter values can be taken from site files, calculated from tool material files, or retrieved from stored parameter files. These options enable us to create NC sequences quicker and with less effort. They can also ensure that standard parameter values are used.

1. Turn off any displayed datum features.

2. Using the Menu Manager, create a **Face** milling sequence.

3. Select the **Parameters** check box.

4. In the Tools Setup dialog box, click the Retrieve Tools icon and select the **MMFLT20.tpm** tool from the metric_tools folder.

> **Note:**
> Notice how by default tools are retrieved from the metric_tools folder.

5. **Set** the parameters, view the parameter values and accept the default values.

> **Note:**
> These parameter values are taken from the site parameter file and the tool material file.

CUT_FEED	25
STEP_DEPTH	16
STEP_OVER	9.6
BOTTOM_STOCK_ALLOW	(-)
CUT_ANGLE	(0)
SCAN_TYPE	(TYPE_1)
SPINDLE_SPEED	127
COOLANT_OPTION	(ON)
CLEAR_DIST	(5)
APPROACH_DISTANCE	(-)
EXIT_DISTANCE	(-)

Figure 12: Showing default parameter values.

6. In the Surf Pick menu, accept the **Model** option to choose surfaces from the reference model.

7. Select the top surface of the reference model for the surface to machine, and complete the selection.

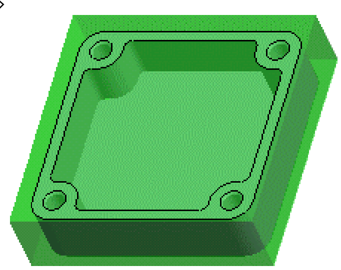


Figure 13: Showing the selected reference model surface.

8. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

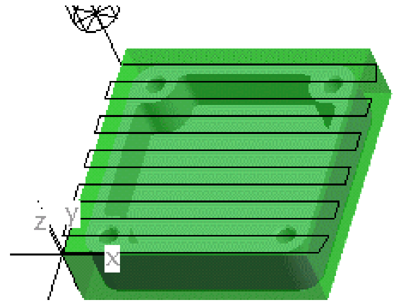


Figure 14: Showing the resulting tool path.

9. Using the Menu Manager, select **Seq Setup**, and select the **Parameters** check box. Using **Retrieve** double-click the **dflt_milling** and **metric** folders, then select the **face.mil** parameter file.

CUT_FEED	10
STEP_DEPTH	4
STEP_OVER	18
BOTTOM_STOCK_ALLOW	-
CUT_ANGLE	0
SCAN_TYPE	TYPE_3
SPINDLE_SPEED	1000
COOLANT_OPTION	ON
CLEAR_DIST	0.5
APPROACH_DISTANCE	-
EXIT_DISTANCE	-

Figure 15: Showing the face.mil file parameter values.

> **Note:**
> Notice all the parameter values are now taken from the face.mil parameter file.

10. Modify the TRIM_TO_WORKPIECE parameter to **YES**.

11. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

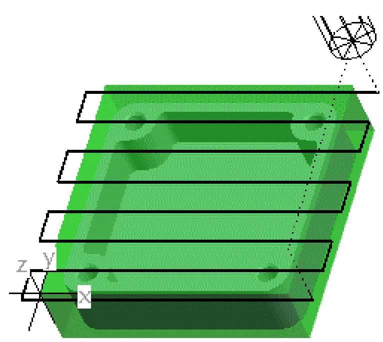


Figure 16: Showing resulting tool path.

Note:

Parameter values can be taken from site files, calculated from tool material files, or retrieved from stored parameter files.

12. Complete the NC sequence.

Step 6. Continue testing the new configuration options; create a CL data file and MCD file for operation op010.

The final test involves creating a CL data file and post-processing this file into an MCD file. We start by creating a CL data file, choose to create an MCD file at the same time, and select a post-processor from our gpost_pp folder. Notice only the post-processors located in the gpost_pp folder are listed for selection. Notice also the name of the machine type displays at the bottom of the graphics window when you highlight each post-processor in the menu.

1. Using the Menu Manager, select **CL Data > Output > Operation > OP010 > File**. Select the **MCD File** check box, and complete the file output. Click the **OK** button to create a CL data file for operation OP010.
2. Accept the **Verbose** and **Trace** options, and select the **UNCX01.P03** post-processor, (this is the hurco post-processor).

Note:

Notice only the post-processors located in the gpost_pp folder are listed for selection. Notice also the name of the machine type displays at the bottom of the graphics window when you highlight each post-processor in the menu.

Step 7. Configure the model tree to display manufacturing parameters.

We can enhance the working environment by configuring the model tree to display manufacturing parameters. Opening the Tree Columns dialog box, we can select the machining parameters option in the drop down list. We can then move relevant parameters into the displayed column. Once the configuration is complete, we can adjust the width of the parameter columns, and save the configuration. Notice it is not possible to see the parameter names without making the model tree wider. An alternative approach is to move the position of the model tree by customizing the screen layout. We can open the Customize Screen dialog box and in the navigators tab we can change the position of the model tree to below the graphics area. We can then complete the configuration; the customized window configuration is saved and can be retrieved when required. Moving the model tree to the bottom of the graphics window provides more space to add additional parameters to the model tree if desired.

1. Ensure the model tree is displayed in the navigator window.
2. In the model tree, click **Settings > Tree Columns**.

3. In the Model Tree Columns dialog box, click the Type drop-down list, and select the **Machining Params** option.

4. Move the **CUT_FEED**, **SCAN_TYPE**, and **STEP_DEPTH** parameters to the displayed column. Complete the configuration.

5. Adjust the width of the navigator and displayed parameter columns until they are clearly visible, as shown in the following figure.

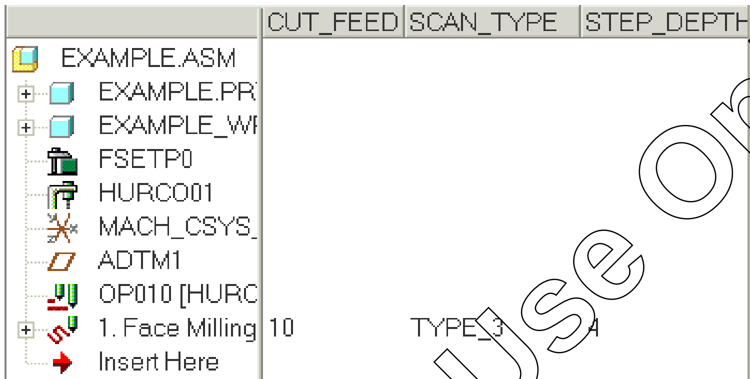


Figure 17: Showing displayed parameters in the model tree.

6. Click the **Settings** tab in the model tree, select the **Save Settings File** option from the drop-down list. Accept the tree.cfg default name and **Save** the tree configuration file.

> **Note:**
> Notice the selected parameters are displayed in the model tree. Notice also it is not possible to see the parameter names without making the model tree wider. An alternative approach is to move the position of the model tree by customizing the screen layout.

7. In the menu bar at the top of the graphics window, click **Tools > Customize Screen** from the drop-down menu.

8. In the Customize Screen dialog box, click the **Navigator Tabs** tab, in the Placement drop-down list, click **Below graphics area**, and in the Height text box change the height to [**5**]. Complete the configuration.

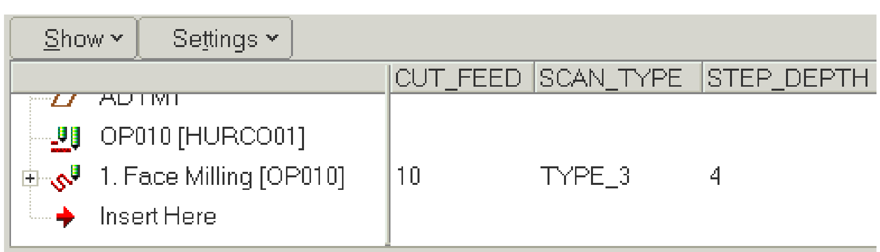


Figure 18: Showing model tree moved to the bottom of the graphics window.

> **Note:**
> This customized window configuration has been saved and can be retrieved when required. Moving the model tree to the bottom of the graphics window provides more space to add additional parameters to the model tree if desired.

9. Save the manufacturing model.

10. Close all windows and erase all components from memory.

11. To clear all configuration options, click **File > Exit > Yes** to exit from Pro/ENGINEER.

> **Note:**
> Exiting Pro/ENGINEER ensures all configuration options are cleared in preparation for the next module.

Summary

After successfully completing this module, you should know how to:

- Describe manufacturing configuration options.
- Create configuration options to store tools, material files, workcells, parameter files, and stock billets.
- Create configuration options to enhance your working environment.

Module

18

Machining Project

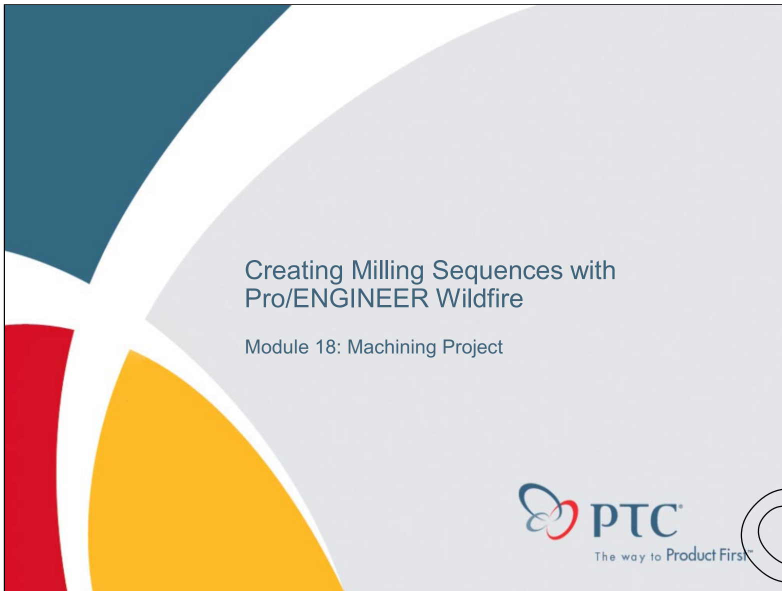
Introduction

When given a design model to machine, it is necessary to understand how to review and check model geometry and any associated model drawings. This provides a means to determine an appropriate machining strategy, and enables the correct selection of workcell, fixtures, tools, and manufacturing parameters during the machining process. When the machining process is complete, it is possible to create useful manufacturing information, such as route sheets, for manufacturing engineers and other downstream processes.

Objectives

After completing this module, you will be able to:

- Identify and check model geometry and dimensions.
- Open and review model drawings.
- Create manufacturing models and configure manufacturing operations.
- Complete machining operations and post-process CL data.
- Create route sheets and other useful manufacturing information.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

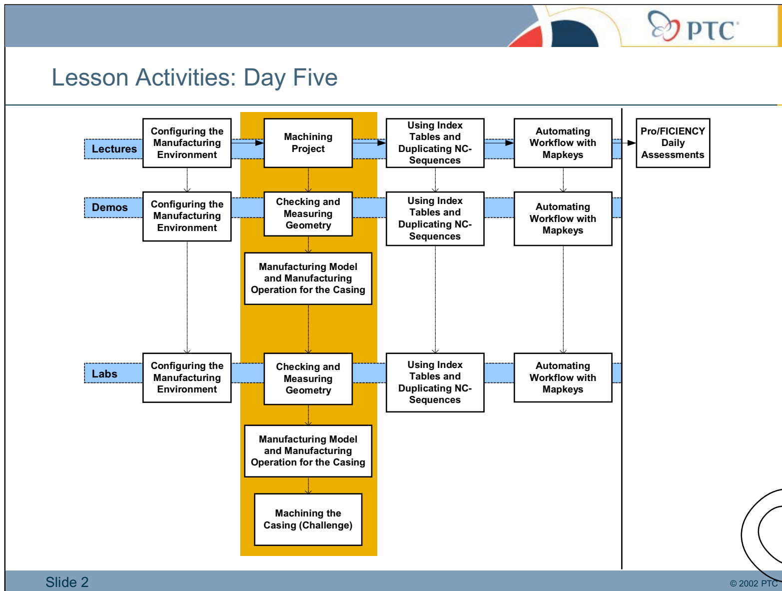
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

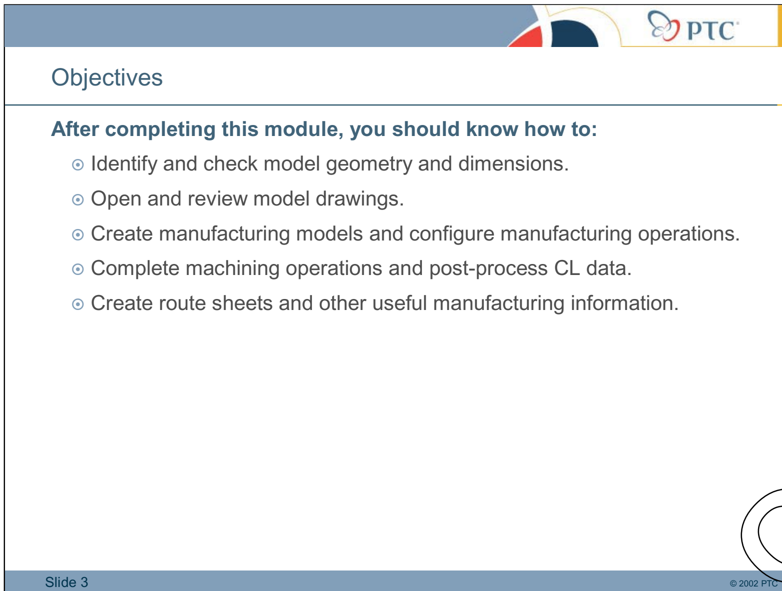
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 10 mins
- Demos : 25 mins
- Labs : 70 mins
- Total: 1 hr 45mins

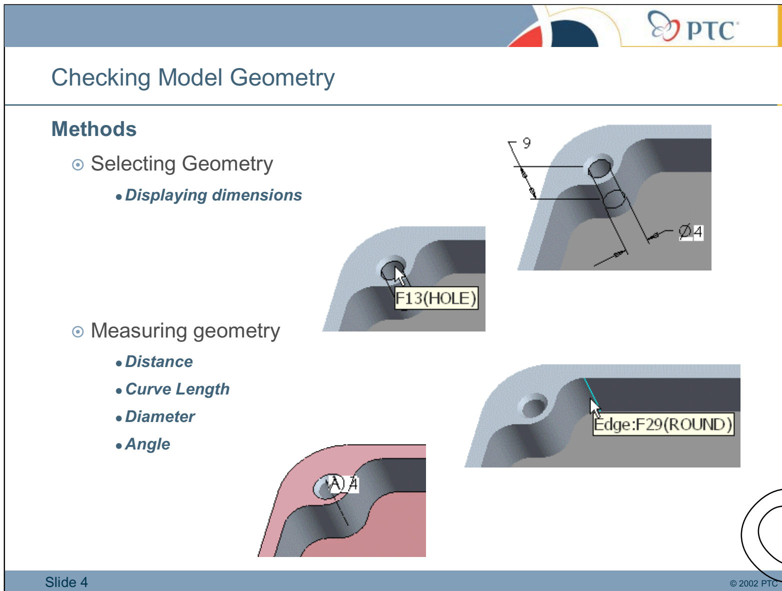


Overview

When given a design model to machine it is necessary to understand how to review and check model geometry and any associated model drawings. This provides a means to determine an appropriate machining strategy, and enables the correct selection of workcell, fixtures, tools, and manufacturing parameters during the machining process. When the machining process is complete, it is possible to create useful manufacturing information, such as a route sheets, for manufacturing engineers and other downstream processes.

After completing this module, you will be able to:

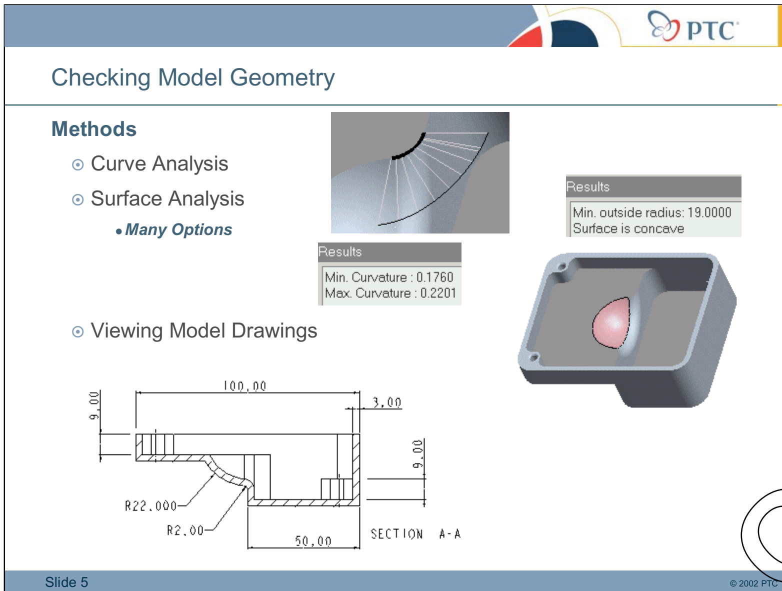
- Identify and check model geometry and dimensions.
- Open and review model drawings.
- Create manufacturing models and configure manufacturing operations.
- Complete machining operations and post-process CL data.
- Create route sheets and other useful manufacturing information.



Checking Model Geometry

There are several methods for checking model geometry.

- One method is to select model features and display dimensions.
- An alternative is the measure tool which provides a precise measurement capability. It is possible to select model geometry and make various measurement calculations, such as distance, curve length diameter and angle.



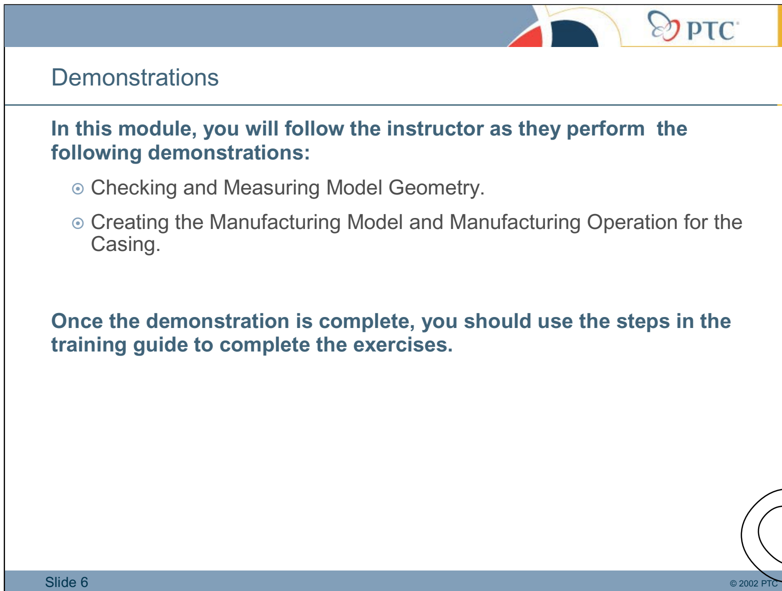
Checking Model Geometry

Other methods for checking model geometry are:

- Curve Analysis: There are several types of curve analysis available, the most commonly used are:
  - Curvature — display the curvature of the edge of curve at the selected point. Mathematically, the curvature is equal to 1/radius.
  - Radius — display the radius of the edge or curve at the selected point. The radius is equal to 1/curvature.

- Surface Analysis: Enables analysis of surface properties, there are fifteen types of surface analysis tools, such as
  - Radius:—calculate the smallest positive and negative radii for a surface of a part or assembly.
  - Slope—displays, in colors, the slope of a surface relative to a reference plane on a part.

- The most common means of checking model geometry are drawings.



Demonstrations (2):

- Checking and Measuring Model Geometry
- Creating the Manufacturing Model and Manufacturing Operation for the Casing

Scenario

- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

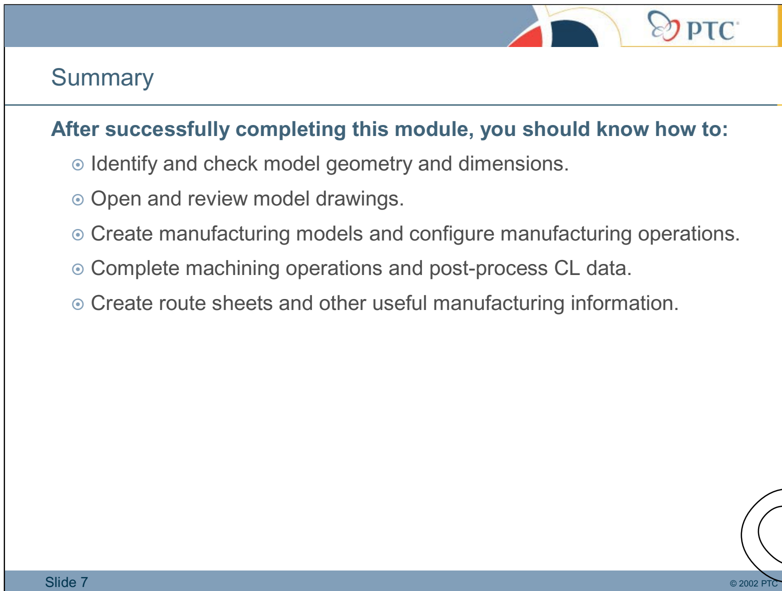
Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercises (3)

- Checking and Measuring Model Geometry
- Creating the Manufacturing Model and Manufacturing Operation for the Casing
- Machining the Casing (Challenge)

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Identify and check model geometry and dimensions.
- Open and review model drawings.
- Create manufacturing models and configure manufacturing operations.
- Complete machining operations and post-process CL data.
- Create route sheets and other useful manufacturing information.

Module 18 Lab Exercises

Exercise 1: Checking and Measuring Model Geometry

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you open the casing model, then check model dimensions, and measure model geometry using various tools. You also check dimensions in a model drawing,enabling you to determine a machining strategy, select appropriate tools and manufacturing parameters during the machining process.

Objectives

After successfully completing this exercise, you will know how to:

- Identify and check model geometry and dimensions.
- Open and review model drawings.

Scenario

You are given a casing model to machine. You use different techniques to determine key model dimensions and distances. This enables you to select correct tool sizes, and to determine a suitable machining strategy. You create a workpiece and manufacturing model and configure an operation. You then create the NC sequences required to machine the bottom of the casing, and post-process the CL data file for the operation. Finally, you identify the different types of manufacturing information that can be created, such as: operation, tooling, NC sequence information, and route sheets.

In this exercise, you check dimensions on the casing model, measure model geometry using various tools, and check dimensions in a model drawing. This enables you to determine a machining strategy, select appropriate tools and manufacturing parameters during the machining process.

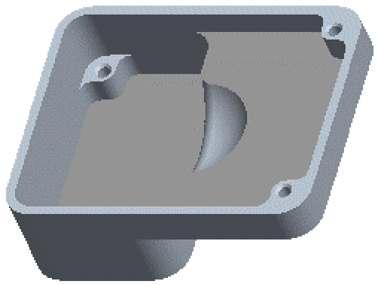


Figure 1: Showing the casing model.

Step 1. Open and check the casing model for key dimensions and sizes.

We can locate the folder for this exercise and set this as our working directory. Before opening the casing model, we can retrieve a window configuration file, returning the model tree to its default position.

We can now open the casing model. We need to select various features on the model to determine key dimensions. We start by selecting the round feature on the support pin. By selecting the edit option in the drop-down list, we can determine the radius value. We repeat this process for the protrusion feature and for the hole and chamfer. We can also determine the radius of the external round feature using the same method.

We could continue selecting features and highlighting model dimensions, however this may not always provide us with the information we require. For example, we need to check the depth of the steps in the pocket, and the internal radius in the deeper pocket. In this case, we can use the measure tool.

1. Start Pro/ENGINEER if it has not already been started for you.
2. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module18**, right-click, and select **Make Working Directory.**
3. Open CASING.PRT.
4. Turn off any displayed datum features.
5. Select the round feature on the pin support, as shown in the figure below. Then right-click and select **Edit** from the drop down list. Note the round radius value. Select in the background to de-select the highlighted geometry.

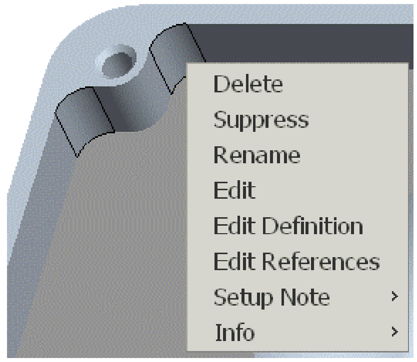


Figure 2: Showing the selected round feature.

6. Select the pin support protrusion feature. Then right-click and select **Edit** from the drop down list. Note the protrusion radius value. Select in the background to de-select the highlighted geometry.

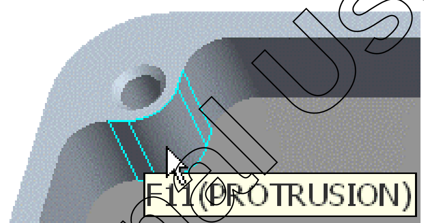


Figure 3: Showing the selected protrusion feature.

7. Select the hole feature, as shown in the following figure. Then right-click and select **Edit** from the drop down list. Note the hole diameter and depth. Select in the background to de-select the highlighted geometry.

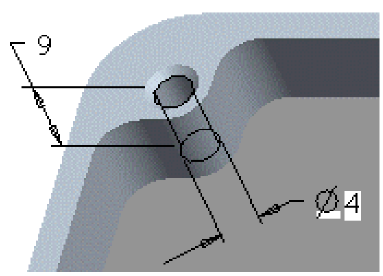


Figure 4: Showing the hole dimensions.

8. Select the hole chamfer feature as shown in the following figure. Then right-click and select **Edit** from the drop down list. Note the chamfer dimensions. Select in the background to de-select the highlighted geometry.

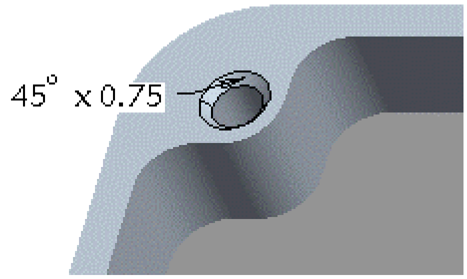


Figure 5: Showing the chamfer dimensions.

9. Select the external round feature. Then right-click and select **Edit** from the drop down list. Note the round radius value. Select in the background to de-select the highlighted geometry.

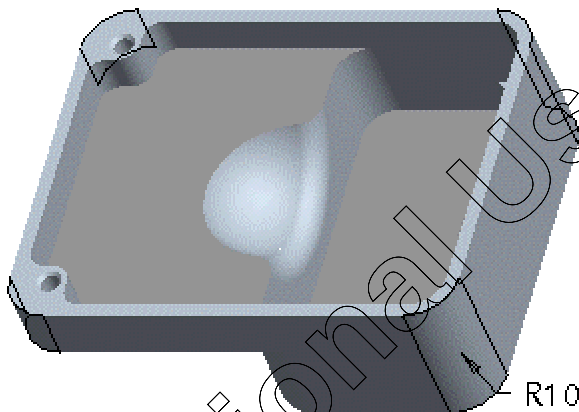


Figure 6: Showing the round radius.

Note:
We could continue selecting features and highlighting model dimensions, however this may not always provide us with the information we require. For example, we need to check the depth of the steps in the pocket, and the internal radius in the deeper pocket, but in this case we can use the measure tool.

Step 2. Use the measure tool to determine the depth of the steps in the pocket, and the internal radius of the round in the deeper pocket.

Opening the Measure dialog box, we can determine the depth of the shallow pocket by selecting surfaces at the top and bottom of the pocket. We can also determine the depth of the deeper pocket by selecting the bottom surface of this pocket. Notice in the Measure dialog box all distances are calculated from the first item selected until a new item is configured as the "From" position. We can also determine the internal round radius in the deeper pocket by selecting the internal rounded surface at the bottom of this pocket.

1. In the menu bar at the top of the graphics window, click **Analysis > Measure**. This makes the Measure dialog box appear.

2. In the Measure dialog box, click **Distance** from the type drop-down list.

3. Select the top surface on the rim of the casing, and then select the surface at the bottom of the shallower pocket, as shown in the following figure.

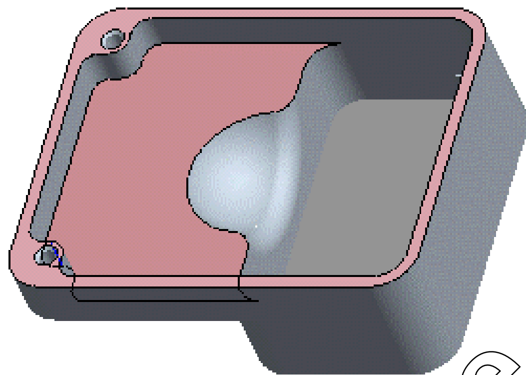


Figure 7: Showing the selected surfaces.

4. Note the distance between the two surfaces in the results section of the Measure dialog box.

5. Measure the distance from the top of the rim to the bottom of the deeper pocket by selecting the surface at the bottom of the deeper pocket.

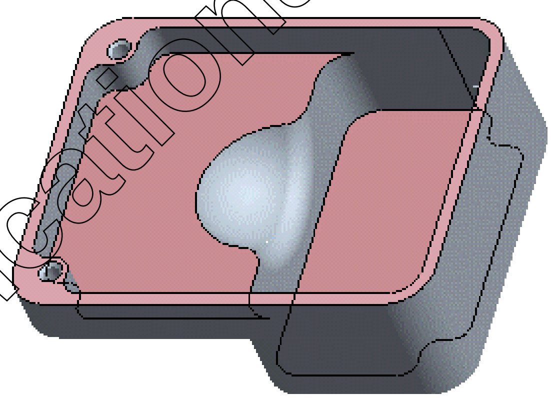


Figure 8: Showing the additional surface selected at the bottom of the deeper pocket.

6. Note the calculated distance in the results section of the Measure dialog box.

> **Note:**
> In the Measure dialog box, all distances are calculated from the first item selected until a new item is configured as the "From" position.

7. Measure the diameter of the internal round in the deeper pocket. In the Type drop-down list, click **Diameter** and select the internal round surface in the deeper pocket.

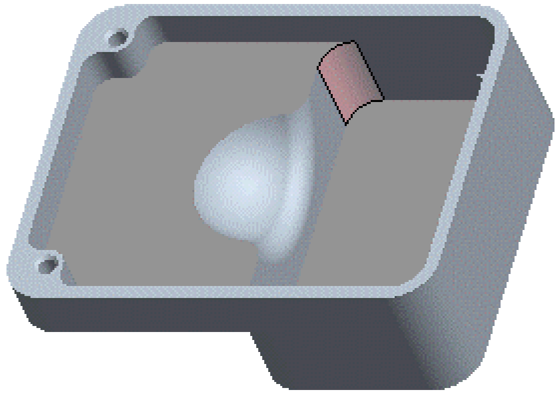


Figure 9: Showing the selected internal round surface.

8. Note the calculated diameter in the results section of the Measure dialog box.

> **Note:**
> There are six possible measurement types available within the Measure dialog box.

9. Close the Measure dialog box.

Step 3. Use the surface analysis tool to measure the radii of the two curved surfaces in the middle of the pocket.

We can use the Surface Analysis dialog box to determine the minimum radii on two curved surfaces. We open the Surface Analysis dialog box, and using the radius option we can select the larger curved surface in the middle of the pocket. We can then compute the minimum radius for the selected surface. We can repeat this process for the smaller curved surface in the middle of the pocket. The surface analysis tool is useful for determining minimum and maximum radius values on curved surfaces; this can help us select the correct tools for machining.

1. In the menu bar at the top of the graphics window, click **Analysis >Surface Analysis**. This makes the Surface Analysis dialog box appear.

2. In the Type drop-down list, click **Radius**, and select the larger curved surface in the middle of the pocket, as shown in the following figure.

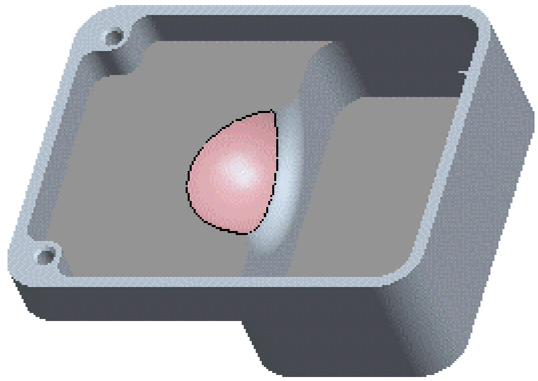


Figure 10: Showing the selected larger curved surface.

3. Click the **Compute** button to determine the minimum radius for the selected surface.
4. Select the smaller curved surface in the middle of the pocket, as shown in the following figure.

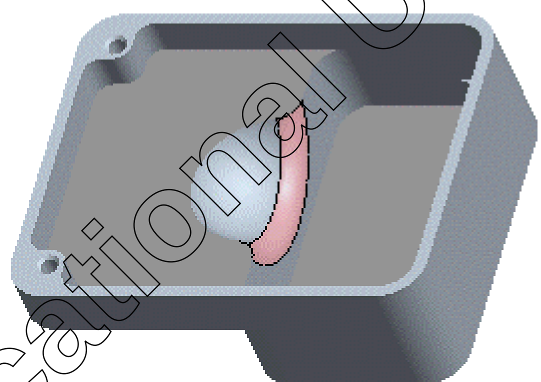


Figure 11: Showing the selected smaller curved surface.

5. Click the **Compute** button to determine the minimum radius for the selected surface.
6. Close the Surface Analysis dialog box.

Step 4. Check key dimensions in the casing model drawing.

Drawings can also be used to provide us with key information for manufacturing. In this case, we have a drawing with the minimum number of dimensions required to complete machining, and the drawing also enables us to check the component after machining. We can open the drawing and zoom-in to each view to determine any remaining key dimensions. Note the drawing can be rechecked during the machining process if required.

1. Open CASING.DRW.

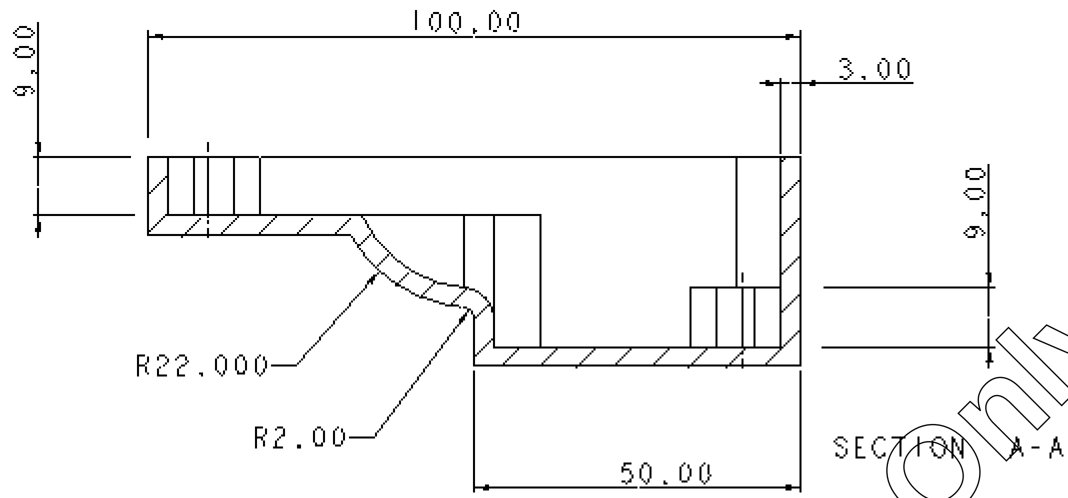


Figure 12: Showing the sectional view from casing model drawing.

2. Zoom-in to each view and determine any remaining key dimensions.

> **Note:**
> The drawing provides a minimum number of dimensions to enable machining of the component, and checking of the component after machining. The drawing can be rechecked during the machining process if required.

3. Close the drawing window.

Exercise 2: Creating the Manufacturing Model and Manufacturing Operation for the Casing

Introduction

In this demonstration, you create a workpiece for the casing model using the Create Stock dialog box. You create a manufacturing model, and you configure a manufacturing operation consisting of a workcell, milling site file, program zero co-ordinate system, and retract surface. You make use of the pre-configured manufacturing environment options.

Objectives

After successfully completing this exercise, you will know how to:

- Create workpieces using the Create Stock dialog box.
- Create manufacturing models.
- Configure manufacturing operations.

Scenario

In this exercise, you create a workpiece for the casing model using the Create Stock dialog box. You create a manufacturing model, and you configure a manufacturing operation consisting of a workcell, milling site file, program zero co-ordinate system, and retract surface. You make use of the pre-configured manufacturing environment options.

Step 1. Create an NC assembly for the casing; assemble the casing part as the reference model. Create the workpiece using the Create Stock dialog box.

Before creating any models, we need to load a configuration file, this enables us to access the company tools library, workcells, parameter files, stock parts and post-processors.

We can create a new NC model assembly, and assemble the casing part as the reference model. We can use the Create Stock dialog box to create our workpiece. We can type in stock allowances, and notice the stock outline update. This completes the configuration of the NC model assembly and workpiece.

1. Before creating any models load the configuration file config.pro. Click **Tools > Options**, use the Open a Configuration File icon and select the **config.pro** file from the module18 folder. Then **Apply** and, **Close** the Options dialog box.

> **Note:**
> To access the company standard manufacturing environment, we need to load the config.pro file which has all the required options and paths specified. This enables access to the company tools library, workcells, parameter files, stock parts and post-processors.

2. Create a new NC model assembly, click **File > New**, select the **Assembly** and **NC Model** options, and name the assembly [`casing_nc`]. When prompted select the **casing.prt** model from the list, (this assembles the reference model).

3. Using the Menu Manager, select **Create Stock**, in the Create Stock dialog box, click **Allowances** and type the following allowances.

Table 1: Stock Allowances

Allowance	Allowance values + and -
Length	5.0 *(0.20 in)*
Width	3.0 *(0.12 in)*
Thickness	5.0 *(0.20 in)*

> **Note:**
> Notice the stock outline update as you type in the allowances.

4. Close the dialog box.

5. Save the NC model assembly, and close the window.

Step 2. Create a new manufacturing NC assembly for the casing.

We start by creating a manufacturing NC assembly for the casing; a sensible naming convention should be used, so name the manufacturing model casing. When complete, we can save the manufacturing model. Notice the CASING.PRT file and CASING_NC_WRK.PRT file are automatically classified correctly as the reference model and workpiece.

1. Create a manufacturing NC assembly for the casing; click **File > New**, select the **Manufacturing** and **NC Assembly** options, and name it [`casing`].

2. Using the Menu Manager, select **Mfg Model > Assemble > Gen Assem**, and assemble CASING_NC.ASM into the manufacturing model.

3. Save the manufacturing model.

> **Note:**
> Notice the CASING.PRT file and CASING_NC_WRK.PRT file are automatically classified correctly as the reference model and workpiece.

Step 3. Configure a manufacturing operation for the casing.

We can open the Operation Setup dialog box, and the Machine Tool Setup dialog box; and then retrieve the mazak workcell, and PPRINT file. Notice how the system automatically uses the workcells and parameters folders. We can also retrieve the RGH_3_AXIS_MIL.sit site parameter *file from the site_files folder. We can select the workpiece material from the Stock Material drop-down list; the list of materials corresponds to the material sub-folders within the tools folder.*

We can then configure a program zero co-ordinate system and name it. We can also configure a retract surface, complete the operation and name the retract surface in the model tree.

> **Note:**
> The manufacturing configuration options should be accessible as you configure the manufacturing operation.

1. Using the Menu Manager, select **Mfg Setup** to open the Operation Setup dialog box.
2. Open the Machine Tool Setup dialog box; retrieve the **mazak.gph** workcell.
3. In the Machine Tool Setup dialog box, select **PPRINT** and retrieve the **pprint.ppr** file.
4. In the Machine Tool Setup dialog box, select **Defaults**.
5. Using the **Current Directory** and **Mill** options, retrieve from the **site_files** folder the **RGH_3_AXIS_MIL.sit** site parameter file.
6. Close the Machine Tool Setup dialog box.
7. In the Operation dialog box, select **STEEL-100BHN** in the Stock Material drop-down list.
8. Turn on the display of co-ordinate systems, if not already visible in the graphics window.
9. Click **View > Display Settings > System Colors**, clear the Blended Background check box, and click **OK** to complete the configuration.
10. Create a machine zero co-ordinate system positioned at the front-left-top corner of the workpiece. Set the orientation of the axes as shown in the following figure. Rename the co-ordinate system to [**mach_csys_010**].

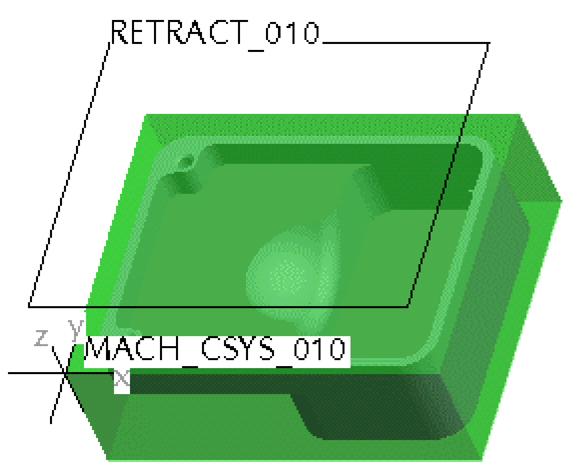


Figure 13: Showing the program zero co-ordinate system and retract plane.

Tip:

When prompted create the co-ordinate system in the CASING.ASM assembly, select this assembly in the model tree. Refresh the graphics window before selecting the surfaces that configure the position of the co-ordinate system.

11. Create a retract surface along the Z-axis of the machine zero co-ordinate system, type [**`30`**] as the z distance.

12. Close the Operation Setup dialog box.

13. Turn on the blended background again. Click **View > Display Settings > System Colors**, select the Blended Background check box, and click **OK** to complete the configuration.

14. Rename the retract surface to [**`retract_010`**] in the model tree.

15. Save the manufacturing model.

Exercise 3: Machining the Casing (Challenge)

Objectives

After successfully completing this exercise, you will know how to:

- Create NC sequences to machine the bottom of the casing.
- Create CL data files.
- Create MCD files using a post-processor.
- Create route sheets and other useful manufacturing information.

Scenario

In this exercise, you create a series of NC sequences to machine the bottom of the casing model. The steps in the exercise specify a suggested order for creating the NC sequences. You can machine the casing using different NC sequences and in a different order if desired. When the operation is complete you post-process a CL data file of the operation. You also identify the different types of manufacturing information that can be created such as operation, tooling, and NC sequence information, and route sheets.

Step 1. Create a face milling sequence to face down to the top of the casing model.

> **Note:**
> The objective is to face down to the top of the reference model leaving zero stock on the model.

1. Turn off any displayed datum features.
2. Using the Menu Manager, create a face milling sequence.
3. Retrieve a suitable tool from the metric_tools folder.
4. Modify parameters as required.

> **Note:**
> The parameter values are taken from the site parameter file and the tool material file.

5. Select the top surface of the reference model for the surface to machine.
6. Play the tool path. Ensure the CL data is visible.
7. Modify parameters and replay the tool path until satisfied.
8. Complete the NC sequence.

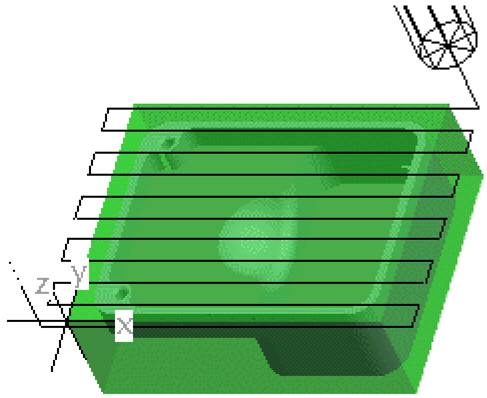


Figure 14: Showing the example facing tool path.

9. Create a material removal feature.

Step 2. Create a profile milling sequence to machine the outside profile of the casing.

> **Note:**
> The objective is to profile down the outside of the casing to a depth of 20 mm *(079 in).*

1. Using the Menu Manager, create a profile milling sequence.
2. Retrieve a suitable tool from the metric_tools folder.
3. Modify parameters as required.
4. Create an extruded mill surface as the profiling surface.
5. Play the tool path. Ensure the CL data is visible.
6. Modify parameters and replay the tool path until satisfied.
7. Complete the NC sequence.

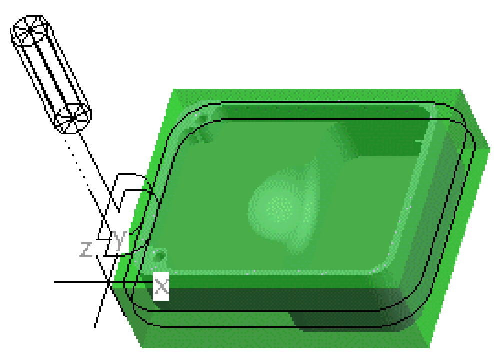


Figure 15: Showing the example profiling tool path.

8. Create a material removal feature, and hide the profile mill surface in the model tree.

Step 3. Create a volume milling sequence to rough out the pocket.

> **Note:**
> The objective is to rough out the pocket leaving zero stock on the flat horizontal surfaces and 0.75 mm *(0.03 in)* on the vertical walls and curved surfaces.

1. Using the Menu Manager, create a volume milling sequence.
2. Retrieve a suitable tool from the metric_tools folder.
3. Modify parameters as required.
4. Create a mill window to define the volume for machining.
5. Play the tool path. Ensure the CL data is visible.
6. Modify parameters and replay the tool path until satisfied.

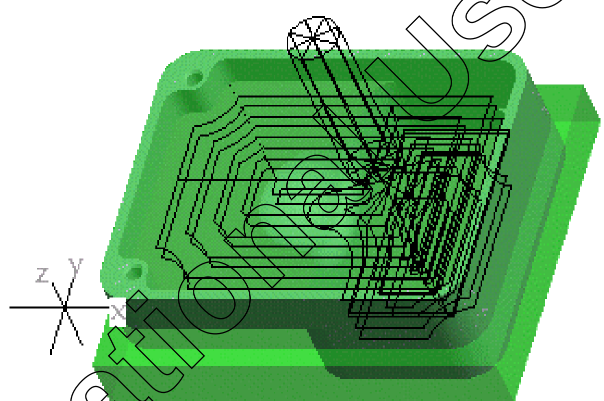


Figure 16: Showing the example rough volume tool path.

7. Complete the NC sequence, and hide the mill window in the model tree.

Step 4. Create a volume milling sequence to finish the vertical walls of the pocket.

> **Note:**
> The objective is to remove the remaining material from the vertical walls in the pocket.

1. Using the Menu Manager, create a volume milling sequence.
2. Retrieve a suitable tool from the metric_tools folder.
3. Modify parameters as required.
4. Create a mill volume to define the volume for machining.
5. Hide the mill volume in the model tree.

6. Play the tool path. Ensure the CL data is visible.

7. Modify parameters and replay the tool path until satisfied.

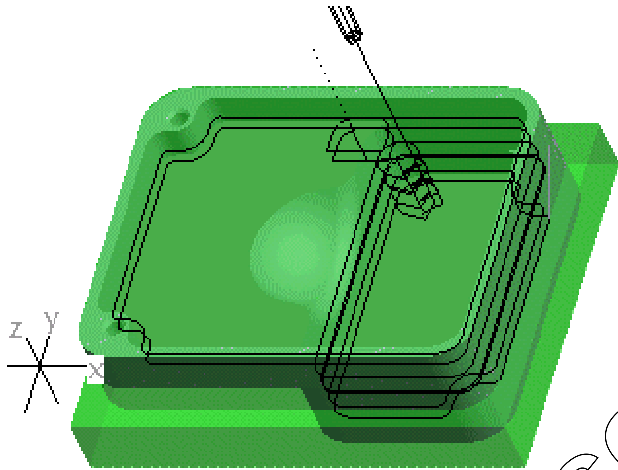


Figure 17: Showing the example finishing volume tool path.

8. Complete the NC sequence.

9. Create a material removal feature for the first (roughing) volume milling sequence.

Step 5. Create a semi-finish surface milling sequence to machine the curved surfaces in the middle of the pocket.

> **Note:**
> The objective is to machine the curved surfaces and leave 0.25 mm *(0.010 in)* stock remaining.

1. Using the Menu Manager, create a surface milling sequence.

2. Retrieve a suitable tool from the metric_tools folder.

3. Modify parameters as required.

4. Select the surfaces to be machined.

5. Configure the cut definition of the surface milling sequence.

6. Play the tool path. Ensure the CL data is visible.

7. Modify the parameters and cut definition; replay the tool path until satisfied.

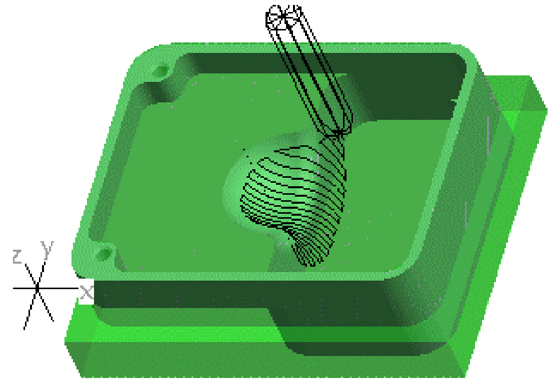


Figure 18: Showing the example semi-finishing surfacing tool path.

8. Complete the NC sequence.

Step 6. Create a finish surface milling sequence to machine the curved surfaces in the middle of the pocket.

> **Note:**
> The objective is to machine the curved surfaces and leave zero stock remaining.

1. Using the Menu Manager, create a surface milling sequence.
2. Retrieve a suitable tool from the metric_tools folder.
3. Modify parameters as required.
4. Select the surfaces to be machined.
5. Configure the cut definition of the surface milling sequence.
6. Play the tool path. Ensure the CL data is visible.
7. Modify the parameters and cut definition; replay the tool path until satisfied.

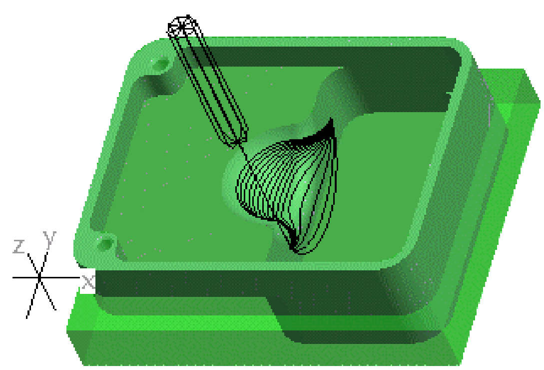


Figure 19: Showing the example finishing surfacing tool path.

8. Complete the NC sequence.

Step 7. Create two holemaking sequences to drill and countersink the four 4-millimeter holes.

1. Using the Menu Manager, create a holemaking sequence.
2. Specify the holemaking cycle type.
3. Retrieve a suitable drill tool from the metric_tools\drills folder.
4. Retrieve a standard set of drilling parameters from the dflt_hmaking\metric folder.
5. Configure the hole set.
6. Play the tool path. Ensure the CL data is visible.
7. Modify the parameters and hole set configuration; replay the tool path until satisfied.
8. Complete the NC sequence.
9. Using the Menu Manager, create a holemaking sequence.
10. Specify the holemaking cycle type.
11. Retrieve a suitable countersink drill tool from the metric_tools\drills folder.
12. Retrieve a countersink set of drilling parameters from the dflt_hmaking\metric folder.
13. Configure the hole set.
14. Play the tool path. Ensure the CL data is visible.
15. Modify the parameters and hole set configuration; replay the tool path until satisfied.
16. Complete the NC sequence.
17. Save the manufacturing model.

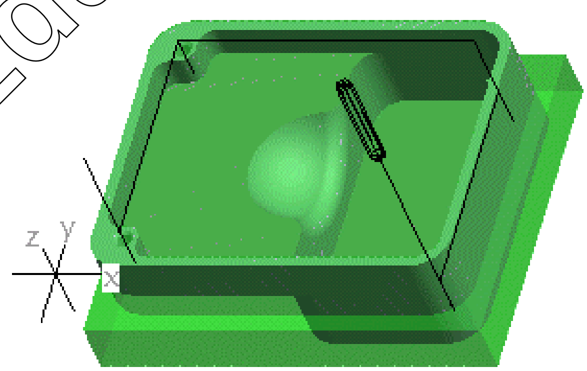


Figure 20: Showing the example holemaking tool path.

Step 8. Create a CL data file and MCD file for operation op010.

1. Using the Menu Manager create a CL data file for operation **OP010**, create an MCD file at the same time; accept the default filename of op010.ncl.
2. Accept the **Verbose** and **Trace** options, and select the **UNCX01.P01** post-processor, (the mazak post-processor).

> **Note:**
> You have now created a CL data file, named op010.ncl, and an MCD file named op010.tap.

Step 9. Check the different types of manufacturing information that can be created and create a route sheet.

1. Using the Menu Manager, select **Mfg Setup > Tooling > MAZAK10** to open the Tool dialog box.
2. Using the drop-down menu, click **View > All Tools**, note the tool information available and close the information window.
3. Click **View > Where Used**, note the tool information available and close the information window.

> **Note:**
> The tool information windows can be saved and printed if required.

TOOLS ARE USED IN THE FOLLOWING NC SEQUENCES:

TOOL ID	NC SEQUENCE NUMBER	FEATURE NUMBER	FEATURE ID	FEATURE TYPE
MMFLT20	1	7	11	MILLING
MMFLT14	2	12	14	MILLING
	3	17	362	MILLING
MMFLT04	4	23	3738	MILLING
MMSPH10	5	27	6010	MILLING
MMSPH08	6	30	6013	MILLING
CSINK_1_0	8	35	6021	HOLEMAKING
MMDRL_04	7	33	6020	HOLEMAKING

Figure 21: Showing tool where used information.

4. Close the Tool Setup dialog box.
5. In the menu bar at the top of the graphics window, click **Info > Manufacturing**. This makes the Manufacturing Info dialog box appear.
6. Click **Operation** and **Apply** to see the operation information window.

7. Note the operation information available and close the information window.

8. Click **NC sequence** and **Apply** to see the NC Sequence information window.

9. Note the NC sequence information available and close the information window.

10. Click **Route** and **Apply** to output information for the route sheet.

> **Note:**
> Note the run times for each NC sequence are shown. Fixture setup time can be specified during fixture configuration if required; alternatively the setup time can be edited manually in the route sheet.

```
|-----------------------------------------------------------------|
|                          Route Sheet                            |
|-----------------------------------------------------------------|
Assembly Name : CASING
Author        :


-----------------------------------------------------------------
Seq.  Machine and Sequence          Rmvd      Set Up     Run
 No.     Description                Vol.      Time       Time
                                  (cu. mm )   (min)      (min)

0010  MILL                          N/A        N/A       61.64
      Face Milling

0020  MILL                          N/A        N/A       26.428
      Profile Milling
```

Figure 22: Showing route sheet information.

11. Note the route sheet information available and close the information window.

12. Close the Manufacturing Information dialog box.

13. Close all windows and erase all components from memory.

14. To clear all configuration options, click **File > Exit > Yes** to exit from Pro/ENGINEER.

Summary

After successfully completing this module, you should know how to:

- Identify and check model geometry and dimensions.
- Open and review model drawings.
- Create manufacturing models and configure manufacturing operations.
- Complete machining operations and post-process CL data.
- Create route sheets and other useful manufacturing information.

Module

19

Using Index Tables and Duplicating NC Sequences

Introduction

Using indexing tables enables you to rotate components on the machine tool. This enables you to machine at different angles on a single component and to machine multiple components on different faces of an index table. In order to do this you need to specify one machine co-ordinate system, and multiple NC sequence co-ordinate systems. It is therefore important to understand the difference between machine and NC sequence co-ordinate systems. When using index tables, it is often possible to save time by patterning sequences or creating sequence subroutines, depending on your specific requirements. It is therefore important to understand the different types of CL data output created by patterning and creating subroutines.

Objectives

After completing this module, you will be able to:

- Describe the purpose of machine and NC sequence co-ordinate systems.
- Describe the differences between machine and NC sequence co-ordinate systems.
- Describe the differences between patterning NC sequences and creating NC sequence subroutines.
- Create NC sequences using different machine and NC sequence co-ordinate systems.
- Create NC sequence subroutines.
- Pattern NC sequences.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

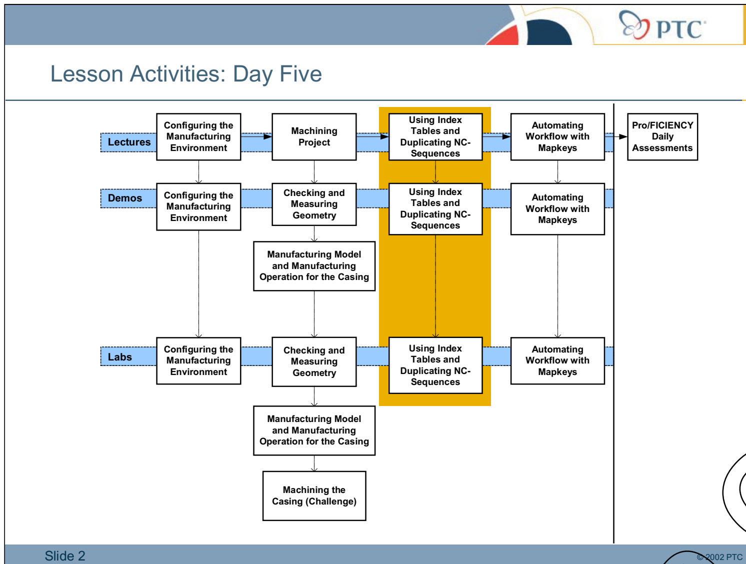
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

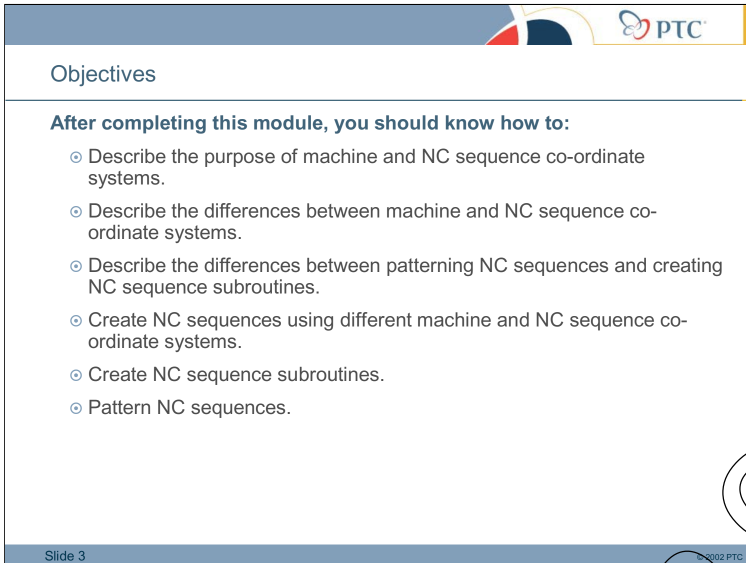
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 15 mins
- Demos : 25 mins
- Labs : 50 mins
- Total: 1 hr 30mins

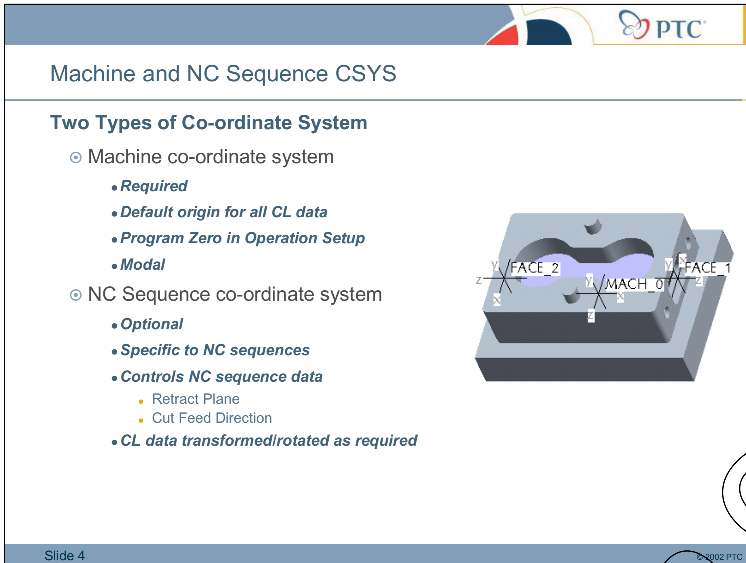


Overview

Using indexing tables enables you to rotate components on the machine tool. This enables you to machine at different angles on a single component and to machine multiple components on different faces of an index table. In order to do this, you need to specify one machine co-ordinate system, and multiple NC sequence co-ordinate systems. It is therefore important to understand the difference between machine and NC sequence co-ordinate systems. When using index tables it is often possible to save time by patterning sequences or creating sequence subroutines, depending on your specific requirements. It is therefore important to understand the different types of CL data output created by patterning and creating subroutines.

After completing this module, you will be able to:

- Describe the purpose of machine and NC sequence co-ordinate systems.
- Describe the differences between machine and NC-sequence co-ordinate systems.
- Describe the differences between patterning NC sequences and creating NC sequence subroutines.
- Create NC sequences using different machine and NC-sequence co-ordinate systems.
- Create NC sequence subroutines.
- Pattern NC sequences.



Machine and NC-sequence Co-ordinate systems

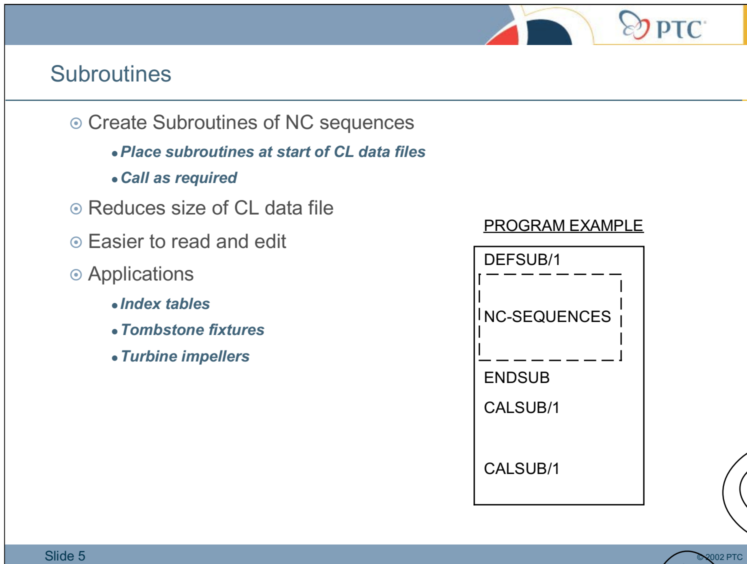
There are two types of co-ordinate system that can be specified in manufacturing.

- **Machine Co-ordinate system—**Acts as the default origin for all CL data. This coordinate system is specified at the time of operation setup. All NC sequences created within a certain operation will use the same Machine coordinate system. The co-ordinate system setting is modal; once set it does not change unless specified.

- NC Sequence—Is optional, and is specific to NC sequences. This coordinate system is specified at the time of NC sequence setup, and controls all the NC sequence data, such as retract surface and cut feed direction.

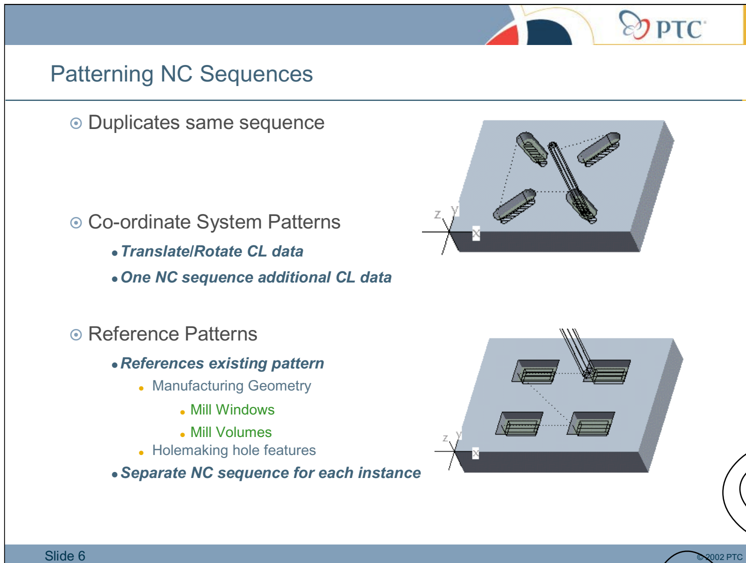
- If the Machine and NC Sequence coordinate systems are different, then, when creating an NC sequence, all CL data will be transformed and output in the coordinates of the Machine coordinate system.

- If the Z axes of the NC Sequence and Machine coordinate systems are not parallel, the tool orientation vector (i,j,k) or table rotation command will also be created in the CL data file. This depends on the configuration of the workcell and manufacturing parameters.



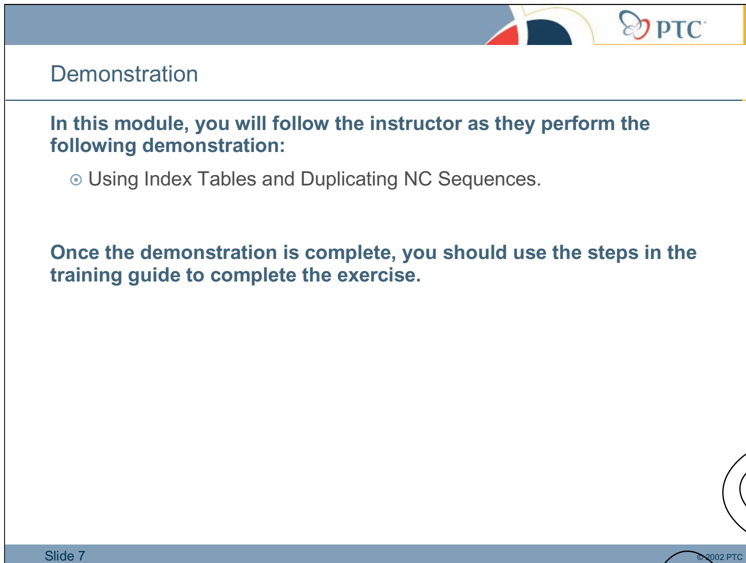
Subroutines

- Subroutine programming enables you to create NC sequences, place them as macros at the beginning of the CL file, and then call them from the main body of the CL file as many times as needed.
- This enhancement reduces the size of CL files, making them easier for the controller to handle and for the programmer to read and edit.
- Typical applications include tombstone work, multiple parts setup on a pallet, and turbine impellers where each blade is the same.



Patterning NC Sequences

- Patterning an NC sequence enables duplicating of the NC sequence multiple times.
- There are two types of NC Sequence pattern:
- **Coordinate patterns** are created by translating or rotating CL data with respect to either the NC Sequence or the Machine coordinate system.
- The result of this type of pattern is one NC sequence but additional CL data created in the operation.
- **Reference patterns** can be created only if the NC sequence to be patterned references a patterned feature (for example, patterned holes for Holemaking, or manufacturing geometry such as mill volumes or mill windows).
- The result of this type of pattern is separate NC sequences for each instance.



Demonstrations (1):

- Using Index Tables and Duplicating NC Sequences

Scenario

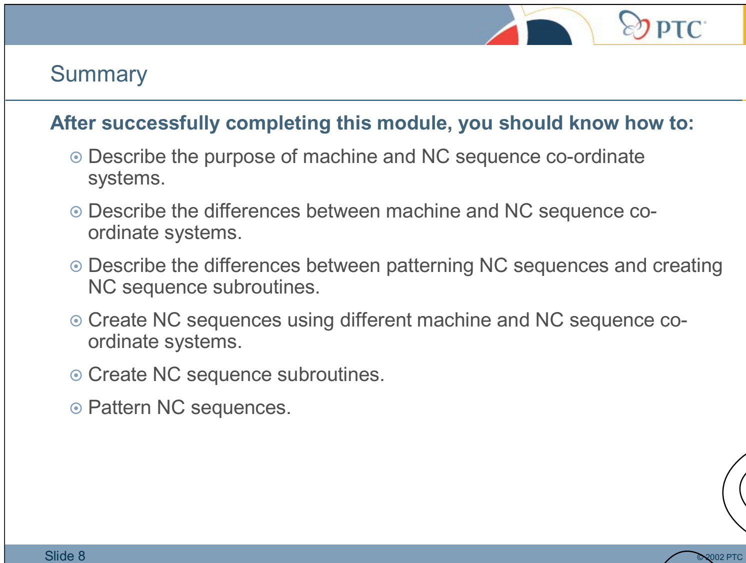
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable “Tell me—Show me—Let me do.”

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the purpose of machine and NC sequence co-ordinate systems.
- Describe the differences between machine and NC-sequence co-ordinate systems.
- Describe the differences between patterning NC sequences and creating NC sequence subroutines.
- Create NC sequences using different machine and NC-sequence co-ordinate systems.
- Create NC sequence subroutines.
- Pattern NC sequences.

Module 19 Lab Exercises

Exercise 1: Using Index Tables and Duplicating NC Sequences

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you are given a sub-plate component that requires machining on two faces. You place the component on a fixture that is located on a rotary index table. You create NC sequences to machine the two faces using one machine co-ordinate system and separate NC sequence co-ordinate systems for each face. It is therefore important to understand the difference between machine and NC sequence co-ordinate systems.

To save time, you create subroutines and patterns of the NC sequences and compare the different CL output. It is therefore important to understand the different types of CL data output created by patterning and creating sub-routines, and apply the most appropriate technique based on machining requirements.

Objectives

After successfully completing this exercise, you will know how to:

- Create NC sequences using different machine and NC sequence co-ordinate systems.
- Create NC sequence subroutines.
- Pattern NC sequences.

Scenario

You are given a sub-plate component that requires machining on two faces. You place the component on a fixture that is located on a rotary index table. You create NC sequences to machine the two faces using one machine co-ordinate system and separate NC sequence co-ordinate systems for each face. To save time, you create subroutines and patterns of the NC sequences and compare the different CL output.

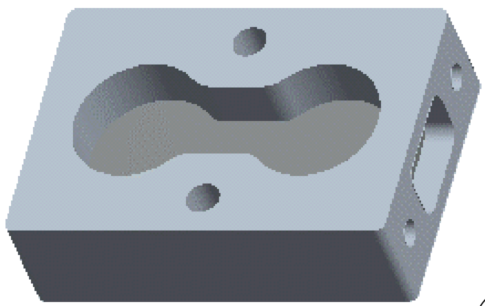


Figure 1: Showing the sub-plate model.

Step 1. Open and check the sub_plate model for key dimensions and sizes.

We can locate the folder for this exercise and set this as our working directory,

We can now open the sub_plate model. We need to select various features on the model to determine key dimensions. We can start by selecting one of the hole features on the right face, and by selecting the edit option in the drop-down list we can determine the hole diameter and depth. We can repeat this process to determine the radius of the round features in the pocket on the right face of the model, and the depth of the pocket feature.

1. Start Pro/ENGINEER if it has not already been started for you.
2. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module19**, right-click, and select **Make Working Directory.**
3. Open SUB_PLATE.PRT.
4. Turn off any displayed datum features.
5. Rotate the model, notice the pocket and two holes on the right face of the model, and the identical holes and a pocket on the left face of the model.
6. Select one of the hole features on the right face of the model, as shown in the following figure. Then right-click and select **Edit** from the drop down list. Note the hole diameter and depth. Select in the background to de-select the highlighted geometry.

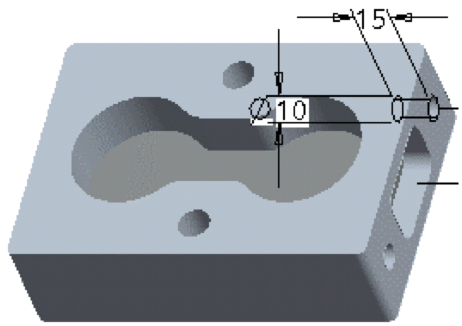


Figure 2: Showing the selected hole dimensions.

7. Select one of the round features in the pocket on the right face of the model. Then right-click and select **Edit** from the drop down list. Note radius size. Select in the background to de-select the highlighted geometry.

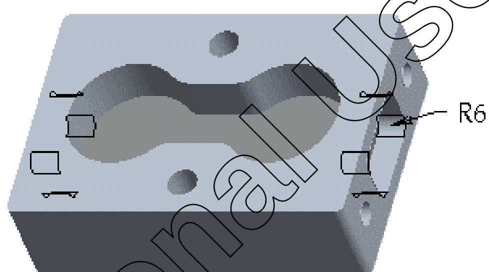


Figure 3: Showing the selected protrusion feature.

8. Select the pocket feature on the right face of the model. Then right-click and select **Edit** from the drop down list. Note the pocket dimensions. Select in the background to de-select the highlighted geometry.

9. Close the window.

Step 2. Create a manufacturing model and configure a manufacturing operation for the sub-plate.

We can create a new manufacturing NC assembly, and name it sub_plate; then load a configuration file to enable the manufacturing configuration options. We can assemble the sub_plate as a reference model. We decide that a workpiece is not required, because displaying material removal is not critical for this job, so there is no need to assemble a workpiece.

We can then open the Operation Setup dialog box, and the Machine Tool Setup dialog box; and create a workcell named makino. We can also retrieve the RGH_3_AXIS_MIL.sit site parameter file from the site_files folder.

We can then open the Fixture dialog box and name the fixture setup op_010. We can now add a component to the fixture setup; in this case we select the fixture part from the list. We can position the fixture using one mate constraint and an insert constraint. Notice only two constraints *are required because assumptions are made that enable the fixture to be placed. You can disable assumptions and add additional constraints If required. When assembled, we can close the fixture setup.*

In the Operation dialog box, we can select the workpiece material from the Stock Material drop-down list; the list of materials corresponds to the material sub-folders within the tools folder.

Ensuring that co-ordinate systems are displayed, we configure a program zero co-ordinate system by selecting the co-ordinate system on the fixture. There is no need to configure the retract surface because this will be configured individually for each NC sequence. This completes the operation configuration.

1. Load the configuration file **config.pro** from the module19 folder to enable the manufacturing configuration options.
2. Create a new manufacturing model for the mold cavity, click **File > New**, select the **Manufacturing** and **NC Assembly** options, and name it [**sub_plate**].
3. Using the Menu Manager, select **Mfg Model > Assemble > Ref Model** and assemble SUB_PLATE.PRT as the reference model.

> **Note:**
> Workpieces are optional components within the manufacturing model, in this case because material removal is not critical for the operation we decide not to use a workpiece.

4. Using the Menu Manager, open the Operation Setup dialog box.
5. Open the Machine Tool Setup dialog box, and retrieve the **makino.gph** workcell.
6. In the Machine Tool Setup dialog box, select **Defaults**.
7. Using the **Current Directory** and **Mill** options, retrieve from the **site_files** folder the **RGH_3_AXIS_MIL.sit** site parameter file.
8. Close the Machine Tool Setup dialog box.
9. Open the Fixture Set Up dialog box, type [**op_010**] as the fixture setup name.
10. Add a component to the fixture setup; select **fixture.prt** from the list.
11. Assemble the fixture, as shown in the following figures. Create a mate constraint and an insert constraint.

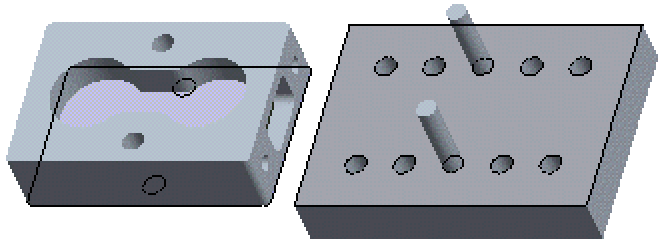


Figure 4: Showing selected surfaces for mate constraint.

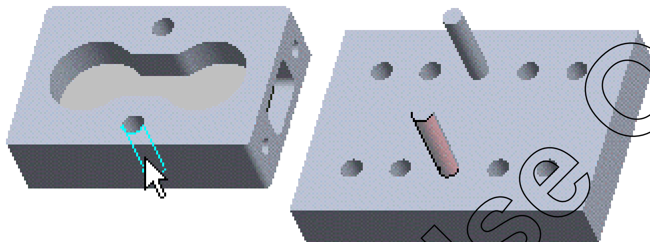


Figure 5: Showing selected surfaces for insert constraint.

> **Note:**
> Only two constraints are required because assumptions are made that enable the fixture to be placed. You can disable assumptions and add additional constraints If required.

12. Complete the fixture setup.

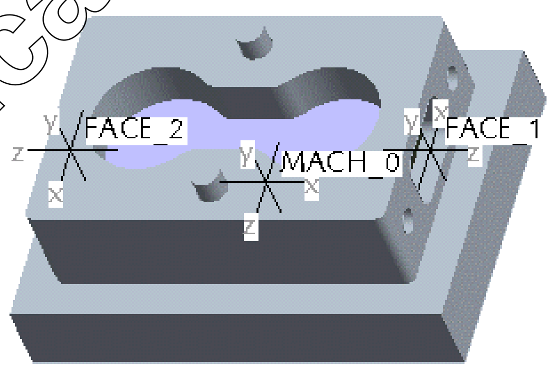


Figure 6: Showing the completed fixture set up.

13. In the Operation dialog box, select **STEEL-100BHN** in the Stock Material drop-down list.

14. Turn on the display of co-ordinate systems, if not already visible in the graphics window.

15. Click the Machine Zero icon, using the **Select** option select the co-ordinate system on the fixture named **MACH_0** as machine zero co-ordinate.

> **Note:**
> The retract surface will be configured individually for each NC sequence.

16. Close the Operation Setup dialog box. Save the manufacturing model.

Step 3. Create a holemaking sequence to drill the two 10-millimeter holes, on the right face.

Using the Menu Manager, we can create a holemaking sequence. We can specify the drill cycle type as standard. We can select Name, and Coord Sys as additional options to be configured. Enter a name for the sequence to make it easy to identify, and retrieve the 10-millimeter drill from the drills folder. We can also retrieve drilling parameters to set standard drilling parameter values.

We need to specify a NC sequence co-ordinate system and retract plane. We can select the face_1 coordinate system on the right face of the sub-plate, and specify the retract plane as 50-millimeters along the z-axis.

Using the Hole Set dialog box, we can use the axis tab and select the two holes on the right face. We can specify the depth options as auto and tip.

We can play the tool path to check the configuration, and then complete the sequence.

1. Using the Menu Manager, create a **Holemaking** sequence.
2. Specify the holemaking cycle type, as **Drill** and **Standard**.
3. Select the **Name** and **Coord Sys** check boxes in the Sequence Setup menu.
4. Name the sequence [**`drl_right_face`**].
5. In the Tools Setup dialog box, click the Retrieve Tools icon and select the **mmdrl_10.tpm** tool from the drills folder.
6. In the Parameters menu, select **Retrieve**, then double-click the **dflt_hmaking** and **metric** folders, and select the **drilling.drl** parameter file.
7. For the NC sequence co-ordinate system, select the FACE_1 coordinate system on the right face of the sub-plate.
8. In the Retract Selection dialog box, click the **Along Z Axis** button and type [**50**] *(1.97 in)* as the z distance. **Preview** the position of the retract plane, and close the dialog box.
9. In the Hole Set dialog box, click the **Add** button, and select the two holes on the right face for machining,
10. Complete the hole set configuration.
11. Play the tool path. Ensure the CL data is visible.
12. Complete the NC sequence.

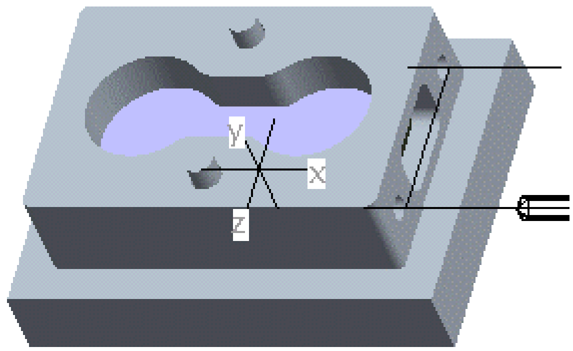


Figure 7: Showing resulting tool path.

Step 4. Create a holemaking sequence to drill the two 10-millimeter holes, on the left face.

We can now repeat the process to drill the two holes on the left face.

We can create a holemaking sequence, and specify the drill cycle type as standard. We can select Name, and Coord Sys as additional options to be configured. Enter a name for the sequence to make it easy to identify, and use the previous sequence drilling parameters.

We need to specify a NC sequence co-ordinate system and retract plane. We can select the face_2 coordinate system on the left face of the sub plate, and specify the retract plane as 50-millimeters along the z-axis.

Using the Hole Set dialog box, we can use the axis tab and select the two holes on the left face. We specify the depth options as auto and tip.

We can play the tool path to check the configuration, and then complete the sequence.

An alternative to creating individual NC sequences to machine each face is to create subroutines or pattern NC sequences. We can explore both options. First, we need to remove the second drilling sequence.

1. Using the Menu Manager, create a **Holemaking** sequence.
2. Specify the holemaking cycle type, as **Drill** and **Standard**.
3. Select the **Name** and **Coord Sys** check boxes in the Sequence sSetup menu.

> **Note:**
> Notice Retract is automatically selected in the Sequence Setup menu after Coord Sys is selected. There is no need to select tool as the same tool will be used.

4. Name the sequence [`drl_left_face`].
5. Use the previous sequence parameters.

6. For the NC sequence co-ordinate system select the FACE_2 coordinate system on the left face of the sub-plate.

7. In the Retract Selection dialog box, click the **Along Z Axis** button and type [**50**] *(1.97 in)* as the z distance. **Preview** the position of the retract plane, and close the dialog box.

8. In the Hole Set dialog box, click the **Add** button, and select the two holes on the left face for machining.

9. Complete the hole set configuration.

10. Play the tool path. Ensure the CL data is visible.

11. Complete the NC sequence.

> **Note:**
> An alternative to creating individual NC sequences to machine each face is to create subroutines or pattern NC sequences. We can explore both options. First we need to remove the second drilling sequence.

12. In the model tree, select the **drl_left_face** sequence, right-click and **Delete** it.

Step 5. Create a subroutine of the first NC sequence to drill holes in the left face of the sub plate. Check the subroutine output by editing the CL data file of the operation.

Using the Menu Manager, we can create a subroutine of an NC sequence and select the holemaking sequence. In the Subroutine dialog box, we can configure the subroutine to rotate the NC sequence about the y-axis of the machine co-ordinate system by 180 degrees. When the configuration is complete notice the subroutine is not listed in the model tree as an NC sequence, the only way to check the output of the subroutine is to create a CL data file.

Using the Menu Manager, we create a CL data file for the operation by using the edit option; this not only creates the CL data file but also enables us to view the resulting CL data immediately. In the CL edit menu, we can use the next option to single step through the tool path. Notice a subroutine definition at the beginning of the CL data file. The subroutine is called first to drill the holes on the right face, then a rotate command is output which (in reality) rotates the index table by 180 degrees. The subroutine is then called again to drill the holes on the left face.

An alternative is to remove the subroutine definition and call statements from the CL data file; this can be achieved by selecting the Copy CL option when configuring the subroutine. We can redefine the subroutine, and select the Copy CL option in the Subroutines dialog box. We can then re-edit the CL data file for the operation. Notice, there are now no subroutine definitions or call statements. The tool drills the holes on the right face first then outputs a rotate command that (in reality) rotates the index table by 180 degrees about the Y-axis. The tool then drills the holes on the left face.

The options within the Subroutine dialog box enable modification of CL data output to meet different requirements.

1. Using the Menu Manager, select **Machining > Subroutines** to create a subroutine; select the DRL_RIGHT_FACE sequence check box for the subroutine.

2. In the Subroutines dialog box, select the following options:

Table 1: Subroutine dialog box options

Option Name	Selection	Explanation
Pattern Style	Individual	Each selected sequence is patterned individually
CL Output Style	Absolute	CL data output in absolute mode
Pattern Type	Rotate	Rotating pattern
Base Coordinate System	Machine	Rotate about the machine co-ordinate system
Number	2	Number of instances, including the original NC sequence
Rotate Angle	180	Angle of rotation, output in CL data file
Rotate Axis	Y	Rotation axis, can be X, Y or Z.

3. **Close** the Subroutine dialog box, and complete the subroutine definition.

> **Note:**
>
> The selected dialog box options create a subroutine definition at the beginning of the CL data file. This subroutine will be called twice to machine the right and then the left faces of the sub-plate
>
> Notice the subroutine is not listed in the model tree as an NC sequence, the only way to check the output of the subroutine is to create a CL data file.

4. Using the Menu Manager, select **CL Data > Edit > Operation > OP010**, **Confirm** the creation of a new CL data file, and click **OK** to accept the name of op010.ncl.

5. In the CL Edit menu, use the **Next** option to single step through to the end of the CL data file.

```
DEFSUB / 1
CYCLE / DRILL, DEPTH, 15.000000, MMPM, 120.000000, CLEAR, 3.000000
GOTO / 75.0000000000, 20.0000000000, -35.0000000000,   $
1.0000000000, 0.0000000000, 0.0000000000
GOTO / 75.0000000000, 20.0000000000, 35.0000000000,   $
1.0000000000, 0.0000000000, 0.0000000000
CYCLE / OFF
RAPID
GOTO / 125.0000000000, 20.0000000000, 35.0000000000,   $
1.0000000000, 0.0000000000, 0.0000000000
ENDSUB
```

Figure 8: Showing the subroutine definition in CL data file.

> **Note:**
> Notice a subroutine definition at the beginning of the CL data file. The subroutine is called first to drill the holes on the right face, then a rotate command is output which (in reality) rotates the index table by 180 degrees. The subroutine is then called again to drill the holes on the left face.
>
> An alternative is to remove the subroutine definition and call statements from the CL data file. This is achieved by selecting the Copy CL option when configuring the subroutine.

6. Complete the editing, and return to the Manufacture menu.
7. Using the Menu Manager, select **Machining > Subroutines > Redefine** > **Ind Pat #1** to redefine the NC sequence subroutine, select the **Pattern Def** check box in the Redefine menu.
8. In the Subroutines dialog box, click the **Copy CL** check box, close the dialog box, and return to the Manufacture menu.
9. Using the Menu Manager, select **CL Data > Edit > Operation > OP010**, **Confirm** the creation of a new CL data file, and click **OK** to accept the name of op010.ncl.
10. In the CL Edit menu, use the **Next** option to single step through to the end of the CL data file.

> **Note:**
> Notice there are no subroutine definition or call statements. The tool drills the holes on the right face first then outputs a rotate command which (in reality) rotates the index table by 180 degrees about the Y-axis. The tool then drills the holes on the left face.
>
> The options within the Subroutine dialog box enable modification of CL data output to meet different requirements.

11. **Quit** the CL Edit menu.

Step 6. Create a volume milling sequence to machine the pocket on the right face of the sub-plate.

Using the Menu Manager, we create a volume milling sequence, and select Name and Coord Sys and Window as additional options in the Sequence Setup menu. We can name the sequence and retrieve the 10-millimeter flat end mill from the metric_tools folder. We can modify the ROUGH_STOCK_ALLOW to zero and set the SCAN_TYPE to TYPE_SPIRAL.

We need to specify a NC sequence co-ordinate system and retract plane. We can select the face_1 coordinate system on the right face of the sub-plate, and specify the retract plane as 50-millimeters along the z-axis.

We can create a mill window named wnd_right to define the volume for machining, and select the chain of edges at the top of the pocket on the right face to configure the mill window. Notice the *mill window is created at the retract plane height for the NC sequence. To avoid air machining, we need to redefine the mill window plane. In the Mill Window dialog box, we can select the plane option to define a new plane position, and select the right surface of the sub-plate, we can preview the mill window to confirm the change. This completes the mill window configuration. We can play the toolpath to check the sequence configuration and complete the sequence.*

1. Using the Menu Manager, create a **Volume** milling sequence.
2. Select **Name, Coord Sys** and **Window** check boxes.
3. Name the sequence [**vol_right**].
4. In the Tools Setup dialog box, click the Retrieve Tools icon and select the **mmflt10.tpm** tool from the metric_tools folder.
5. Modify the following parameters:

Table 2: Changed Parameters

Parameter	Value
ROUGH_STOCK_ALLOW	0
SCAN_TYPE	TYPE_SPIRAL
CUTCOM	OFF

6. For the NC sequence co-ordinate system select the FACE_1 coordinate system on the right face of the sub-plate.
7. In the Retract Selection dialog box, click the **Along Z Axis** button and type [**50**] *(1.97 in)* as the z distance. **Preview** the position of the retract plane, and close the dialog box.
8. Create a mill window named [**wnd_right**] to define the volume for machining.
9. Using the **Select** and **Tangent Chain** options, select the chain of edges at the top of the pocket on the right face. **Preview** the mill window.

Note:

Notice the mill window is created at the retract plane height for the NC sequence. To avoid air machining we need to redefine the mill window plane.

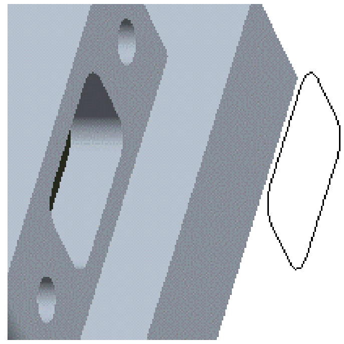


Figure 9: Showing the previewed mill window at height of retract plane.

10. In the Mill Window dialog box, select the **Plane** element and **Define** a new plane position.

11. Select the right surface of the sub-plate, as shown in the following figure. **Preview** the mill window, then complete the mill window configuration.

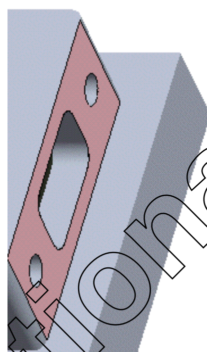


Figure 10: Showing the selected surface for mill window plane.

12. Play the tool path. Ensure the CL data is visible.

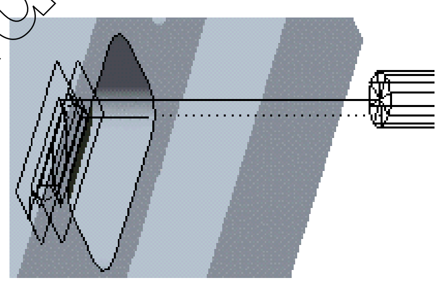


Figure 11: Showing the resulting tool path.

13. Complete the NC sequence, and hide the mill window in the model tree.

> **Note:**
> As an alternative to creating a subroutine, we can pattern the volume milling sequence to machine the pocket on the left face.

Step 7. Pattern the volume milling sequence to machine the pocket on the left face.

We can now pattern the volume milling sequence to machine the pocket on the left face. Using the Menu Manager, we can create a pattern of an NC sequence, and select VOL_RIGHT as the sequence to pattern. We can configure the pattern to rotate the NC sequence about the y-axis of the machine co-ordinate system by 180 degrees.

Notice the VOL_RIGHT volume milling sequence is now listed within a pattern in the model tree. The patterned sequence cannot be accessed directly for checking, the only way to check the output of the pattern is to create a CL data file. If required the pattern can be modified via the Menu Manager.

1. Using the Menu Manager, select **Machining > Utilities > Pattern > NC Sequence** to create a pattern of an NC sequence, select **VOL_RIGHT** as the sequence to pattern.

2. Progressively select the following options during the pattern configuration:

Table 3: Pattern Options

Menu	Option
Pat Type	Csys Pattern
Pat Type (first direction)	Rotate
Enter Value (rotation angle)	Enter, **180**
Rot Type	Mach Csys
Rot Type	Index Table
Rot Type	Y Axis
Rot Type	Done Rot
Enter Value (instances)	Enter, **2**
Pat Type (second direction)	None

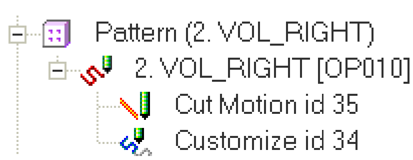


Figure 12: Showing pattern in model tree.

Note:
Notice the VOL_RIGHT volume milling sequence is now listed within a pattern in the model tree. The patterned sequence cannot be accessed directly for checking, because the only way to check the output of the pattern is to create a CL data file. If required the pattern can be modified via the Menu Manager.

Step 8. Edit a CL data file for operation op010.

Using the Menu Manager, we can create a CL data file for the operation using the edit option. This not only creates the CL data file but enables us to view the resulting CL data immediately. In the CL Edit menu, we can use the next option to single step through the tool path.

Notice the holes are drilled on the right and left faces using an NC sequence subroutine. Then the pocket on the right face is machined, and the pocket on the left face is machined using a patterned NC sequence.

1. Using the Menu Manager, select **CL Data > Edit > Operation > OP010**, **Confirm** the creation of a new CL data file, and click **OK** to accept the name of op010.ncl.
2. In the CL Edit menu, use the **Next** option to single step through to the end of the CL data file.

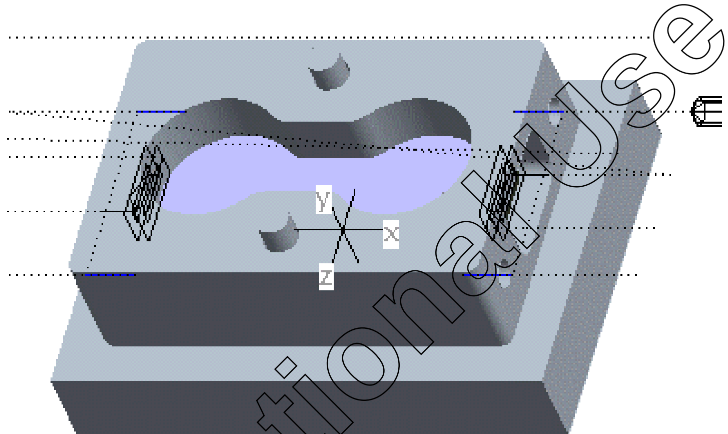


Figure 13: Showing the resulting operation tool path.

> **Note:**
> Notice the holes are drilled on the right and left faces using an NC sequence subroutine. Then the pocket on the right face is machined, and the pocket on the left face is machined using a patterned NC sequence.
>
> Patterns and subroutines provide a comprehensive set of options for duplicating NC sequences.

3. **Quit** the CL Edit menu.
4. Save the manufacturing model.
5. Close all windows and erase all components from memory.
6. To clear all configuration options, click **File > Exit > Yes** to exit from Pro/ENGINEER.

Summary

After successfully completing this module, you should know how to:

- Describe the purpose of machine and NC sequence co-ordinate systems.
- Describe the differences between machine and NC-sequence co-ordinate systems.
- Describe the differences between patterning NC sequences and creating NC sequence subroutines.
- Create NC sequences using different machine and NC-sequence co-ordinate systems.
- Create NC sequence subroutines.
- Pattern NC sequences.

Module

20

Automating Workflow with Mapkeys

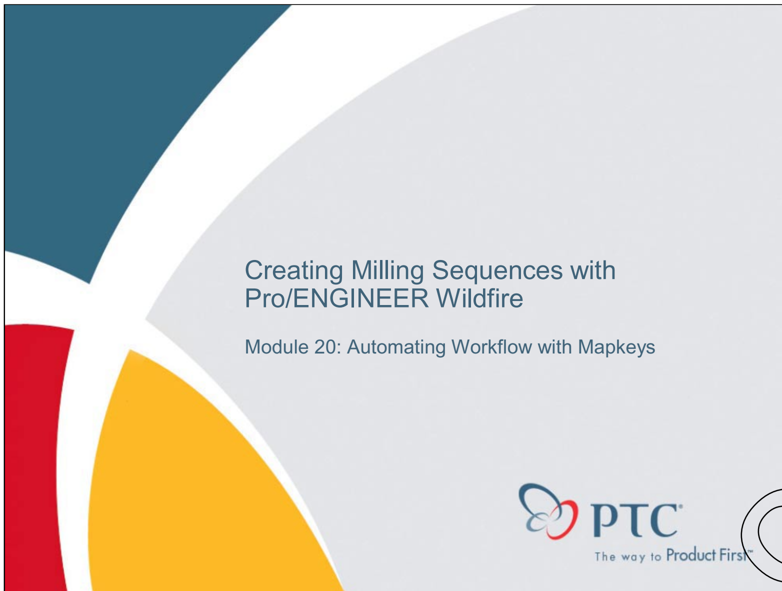
Introduction

Mapkeys enable you to save commonly used selection sequences into a macro called a mapkey. Mapkeys can be started via keyboard input, or they can be stored in your toolbar or menu bar, you can then use mapkeys with a single mouse click. Mapkeys are useful because they enable you to automate and speed up your workflow.

Objectives

After completing this module, you will be able to:

- Describe the uses of mapkeys.
- Describe the process of creating mapkeys.
- Create and use mapkeys.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

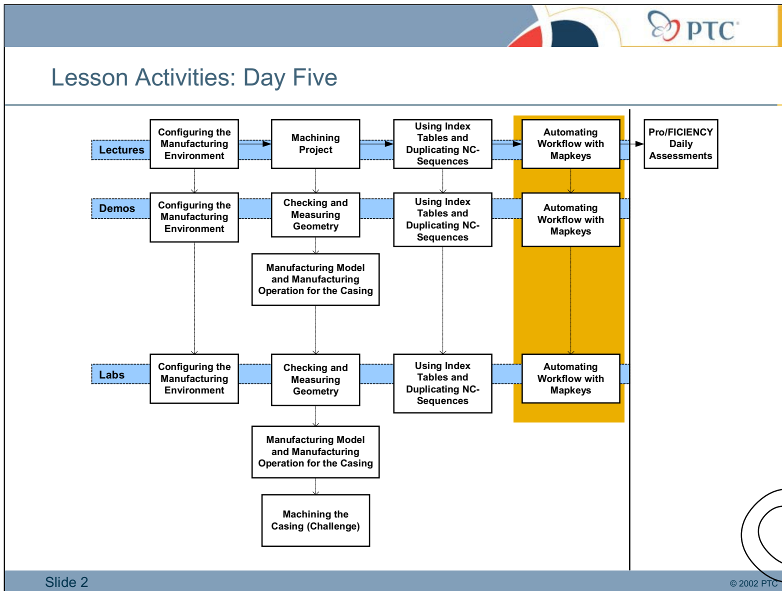
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

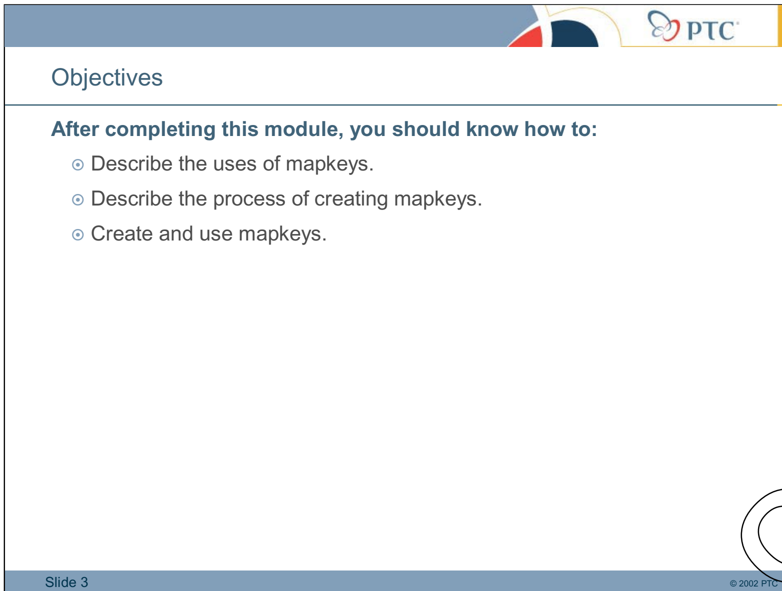
/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Duration

- Lecture: 10 mins
- Demos : 20 mins
- Labs : 30 mins
- Total: 1 hr 00mins

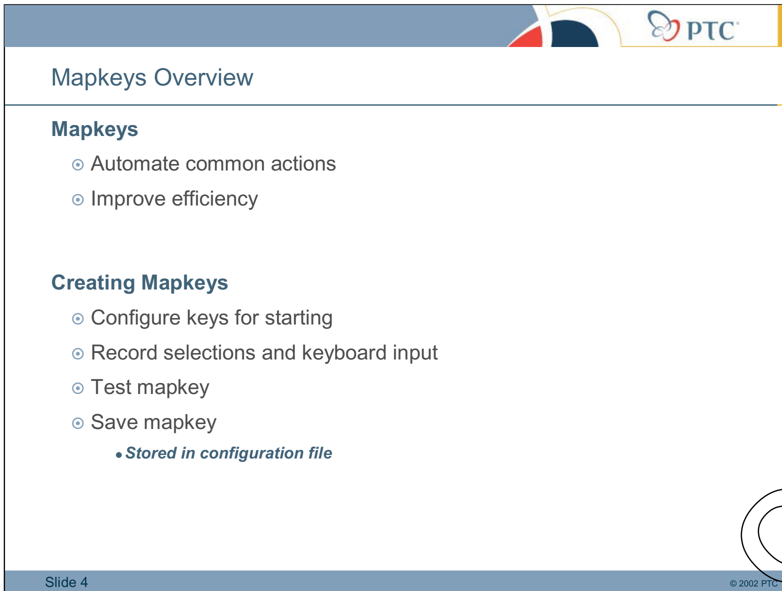


Overview

Mapkeys enable you to save commonly used selection sequences into a macro called a mapkey. Mapkeys can be started via keyboard input, or they can be stored in your toolbar or menu bar, where you can then use mapkeys with a single mouse click. Mapkeys are useful because they enable you to automate and speed up your workflow.

After completing this module, you will be able to:

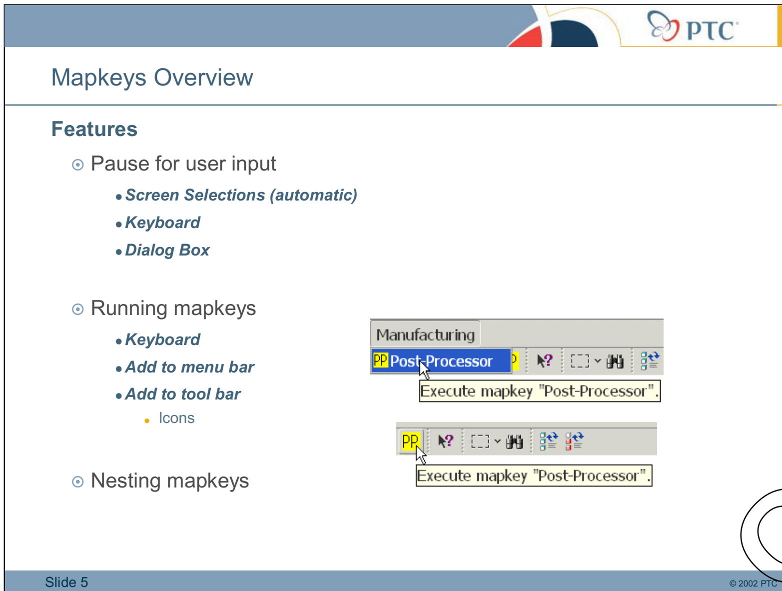
- Describe the uses of mapkeys.
- Describe the process of creating mapkeys.
- Create and use mapkeys.



Mapkeys Overview

- Mapkeys enable you to automate combinations of commonly used menu picks. You can create a mapkey for virtually any task you perform frequently within Pro/ENGINEER.
- Using mapkeys can save you time performing frequent tasks.

- Creating mapkeys involves:
  - Selecting a unique combination of keys to run the mapkey.
  - Recording the selections and keyboard input.
  - Testing the mapkey, ensuring the mapkey works as intended.
  - Saving the mapkey to a configuration file.
    - Mapkeys can be stored in configuration files for retrieval and future use.

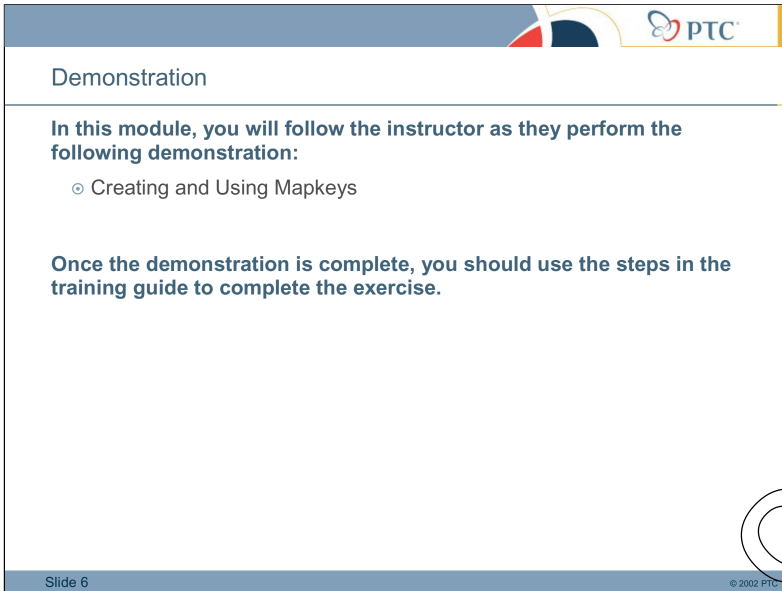


Mapkeys Overview

- When creating mapkeys, you can pause the mapkey for screen selections, this happens automatically.
- You can also optionally pause for user input from the keyboard or dialog box.

- Mapkeys can be run from the keyboard by typing in a unique combination of keys.
  - For example ‘md’ may run a mapkey to start the measure dialog box and prompt the user to measure the distance between two entities.
- Mapkeys can also be stored in toolbars as icons or in menu bars as additional menu commands.

- You can also nest one mapkey within another, so that one mapkey starts another. To do so, you include the mapkey name in the sequence of commands of the mapkey you are creating.



Demonstration: Creating and Using Mapkeys

Scenario

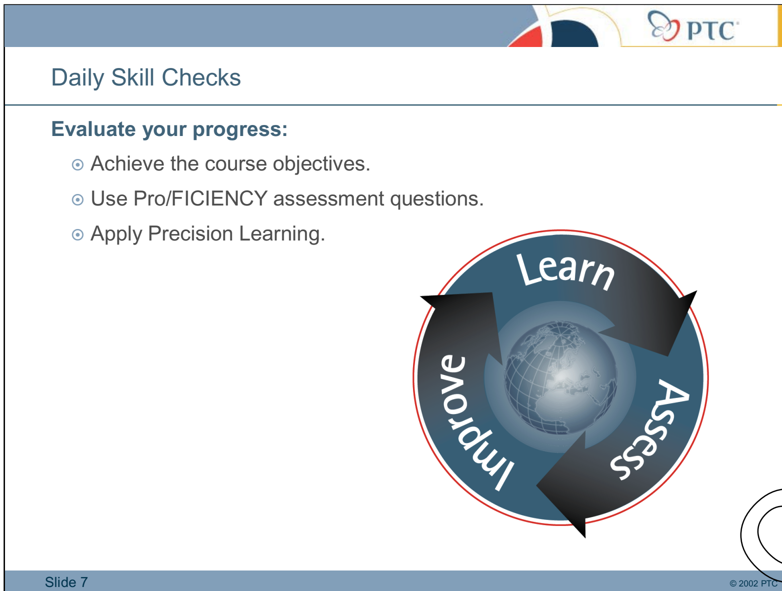
- Throughout this course, the exercise scenarios are the same as the demo scenarios.
- This has been done to build consistency between demos and lab exercises so as to enable "Tell me—Show me—Let me do."

Method

Use the script in the instructors guide, use the same models as in the exercises.

Exercise

Once the demo is complete the students should use the steps in the training guide to complete the exercises.



Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY.
- **Improve:** The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Getting Started

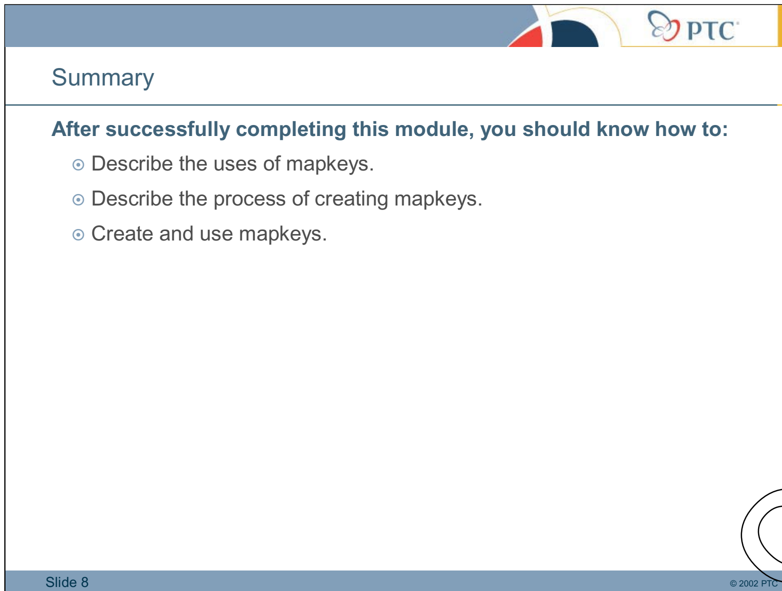
- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Tests

Description: For each course, 5 new 10 question exams based upon the topics covered each day.

How to use it:

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After successfully completing this module, you should know how to:

- Describe the uses of mapkeys.
- Describe the process of creating mapkeys.
- Create and use mapkeys.

Module 20 Lab Exercises

Exercise 1: Automating Workflow with Mapkeys

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2002380 or later.
- Download and install the class files **`wildfire_milling_330.tar.gz`** as described in the classroom setup notes.

Introduction

In this demonstration, you need to create a mapkey that records creating and post-processing a CL data file. You record the mapkey and ensure that the mapkey pauses for user input at the correct time. You verify the mapkey works correctly for all available post-processors. Mapkeys are useful because they enable you to automate and speed up your workflow.

Objectives

After successfully completing this exercise, you will know how to:

- Create and use mapkeys.

Scenario

To speed up the post-processing procedure, you need to create a mapkey that records creating and post-processing a CL data file. You record the mapkey and ensure that the mapkey pauses for user input at the correct time. You test the mapkey works correctly for all available post-processors.

Step 1. Open the cover manufacturing model.

We can locate the folder for this exercise and set this as our working directory. We then load a configuration file to enable the manufacturing configuration options. We can then open the cover manufacturing model.

1. Start Pro/ENGINEER if it has not already been started for you.

2. In the navigator, select the Folder Browser tab, and browse to **c:\users\student\wildfire_mfg_330\module20**, right-click, and select **Make Working Directory.**

3. Load the configuration file to enable the manufacturing configuration options. Click **Tools > Options**, use the Open a Configuration File icon and select the **config.pro** file from the module20 folder. Then **Apply** and, **Close** the Options dialog box.

4. Open COVER.MFG. Turn off any displayed datum features.

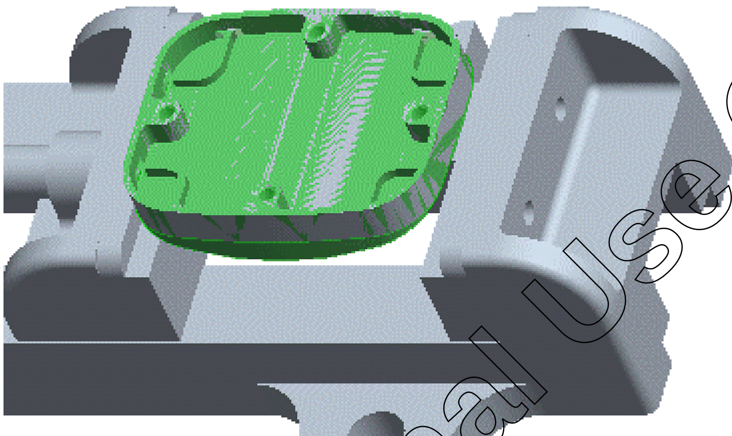


Figure 1: Showing the cover manufacturing model.

Step 2. Create and test a mapkey to create and post-process CL data files.

We need to record a mapkey to create and post-process a CL data file. Using the tool bar, we can open the Mapkeys dialog box. We configure a key sequence and name for the mapkey. We can then begin to record the mapkey. Using the Menu Manager, we can step through the commands to create a CL data file. When the list of operations displays, we pause the mapkey, type in a resume prompt, and then continue recording the mapkey. When the post-processor list displays, we pause again, type in a resume prompt and then complete the recording of the mapkey.

We can then run the mapkey, and select an operation and a post-processor when prompted. When satisfied the mapkey is running correctly, we can save the mapkey to the config.pro file, and close the Mapkeys dialog box.

1. In the menu bar at the top of the graphics window, click **Tools > Mapkeys**, in the Mapkeys dialog box, click the **New** button.

2. In the Record Mapkey dialog box, type [**pp**] in the Key Sequence text box, and type [**Post Process**] in the Name text box, then click **Record**.

> **Note:**
> Every menu pick is now recorded in the mapkey.

3. Using the Menu Manager, select **CL Data > Output > Operation**, when the operation list displays, click the **Pause** button in the Record Mapkey dialog box.

4. Type in the resume prompt [**Select Operation**], and click **OK** in the resume prompt window, then select **OP010** from the list of operations, and continue the mapkey by clicking the **Resume** button.

5. Continue recording the mapkey, in the Menu Manager select **File > MCD File > Done**, and click **OK** to accept the default name of op010. Accept the Verbose and Trace PP options, and select **Done**. When the PP List displays click the **Pause** option in the Record Mapkey dialog box.

6. Type in the resume prompt [**Select a Post-Processor**] and click **OK** in the resume prompt window, then select **UNCX01.P01** from the list of post-processors, wait for the post-processor to run, then complete the mapkey by clicking the **OK** button in the Record Mapkey dialog box.

7. Close the information window created when the post-processor was selected.

> **Note:**
> When you run the mapkey the resume prompts will suspend the mapkey until you have made a selection and then selected resume.

8. Test the mapkey; click the **Run** button in the Mapkeys dialog box.

9. When prompted select operation **op010**, and then click **Resume**.

10. When prompted select the **UNCX01.P01** post processor.

11. Check the mapkey ran successfully by reading the information window created when the post-processor was selected. Then close the information window.

12. Click **Resume** to complete the mapkey.

> **Note:**
> If the mapkey has failed for whatever reason delete the mapkey and create it again.

13. If the mapkey has failed, delete it and repeat steps 2 to 12 again.

14. When satisfied the mapkey is running correctly, click the **Save** button, to save the mapkey to the config.pro file.

15. **Close** the Mapkeys dialog box, and return to the Manufacture menu.

> **Note:**
> Mapkeys can be saved with other configuration options in the config.pro file, they can then be re-used when required.

Step 3. Run the mapkey from the keyboard; check it runs correctly for all post-processors.

We can now test the mapkey from the keyboard by typing in pp; this starts the mapkey. We can select operation op010 when prompted, and select the UNCX01.P01 post-processor when prompted. We repeat this process and check the mapkey is running correctly for all available post-processors.

In summary, the mapkey now runs correctly from the keyboard, and can be used to post-process any operation with any post-processor. This completes the mapkey configuration.

1. Run the mapkey from the keyboard by typing [**pp**].
2. When prompted select operation **op010**, and then click **Resume**.
3. When prompted select the **UNCX01.P01** post processor.
4. Check the mapkey ran successfully by reading the information window created when the post-processor was selected. Then close the information window.
5. Click **Resume** to complete the mapkey.
6. Repeat steps 1 to 4, each time select a different post-processor during step 3. Ensure the mapkey runs correctly for all post-processors.

> Note:
> The mapkey now runs correctly from the keyboard, and can be used to post-process any operation with any post-processor.

7. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the uses of mapkeys.
- Describe the process of creating mapkeys.
- Create and use mapkeys.

Appendix

A

Creating Auxiliary Milling Sequences

Introduction

Auxiliary NC sequences are created by manually building a CL data file. They can be used to specify the connecting tool motions and change the tool axis orientation, if needed, between two machining NC sequences. They also allow you to access the on-machine probe functionality by inserting CL commands. Auxiliary NC sequences are available for any workcell type, and can be performed with any type of tool. They can produce tool paths that cannot be created with other types of NC sequence.

Objectives

After completing this module, you will be able to:

- Describe the auxiliary milling process.
- Create auxiliary milling sequences by defining tool motions and inserting CL commands.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

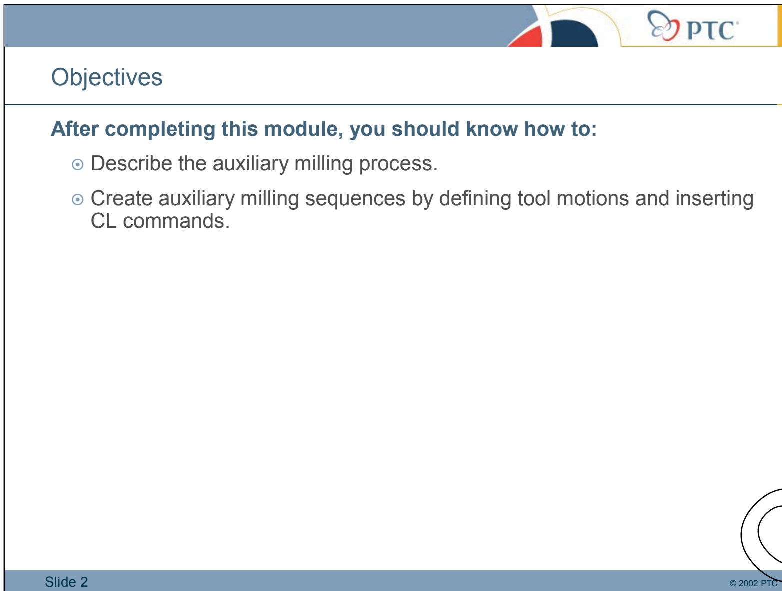
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in <u>Windmill</u>, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in <u>Windmill</u>, located in

/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology



Overview

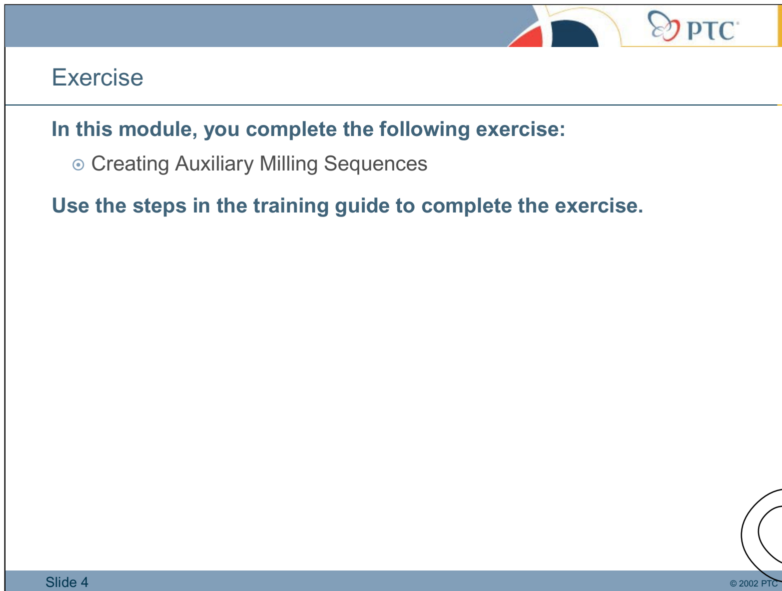
Auxiliary NC sequences are created by manually building a CL data file. They can be used to specify the connecting tool motions and change the tool axis orientation, if needed, between two machining NC sequences. They also allow you to access the on-machine probe functionality by inserting CL commands. Auxiliary NC sequences are available for any workcell type, and can be performed with any type of tool. They can produce tool paths that cannot be created with other types of NC sequence.

After completing this module, you will be able to:

- Describe the auxiliary milling process.
- Create auxiliary milling sequences by defining tool motions and inserting CL commands.

Auxiliary Milling

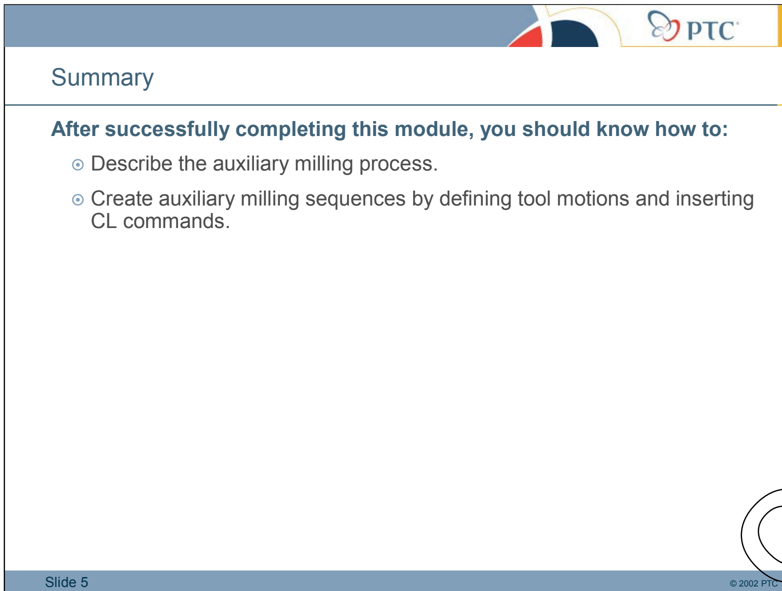
- Creating auxiliary NC sequences involves manually building the CL data file
  - You can configure tool motions by sketching or selecting existing geometry for the tool to follow.
  - You can also insert CL commands to control for example, feed rate, spindle speed, coolant, and cutter compensation. You can also insert CL commands that access on-machine probe functionality
  - You do not have to specify a tool for an Auxiliary NC sequence. You will be able to create Tool Motions even though no tool is specified.



Exercise: Creating Auxiliary Milling Sequences

Exercise

Use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the auxiliary milling process.
- Create auxiliary milling sequences by defining tool motions and inserting CL commands.

Appendix A Lab Exercises

Exercise 1: Creating Auxiliary Milling Sequences

Objectives

After successfully completing this exercise, you will know how to:

- Create auxiliary milling sequences by defining tool motions and inserting CL commands.

Scenario

You need to create an auxiliary milling sequence to profile the sidewalls of the pocket on the top face of the sub plate model. You create an auxiliary milling sequence by configuring tool motions and inserting CL commands. You check that the CL data in the tool path is correct.

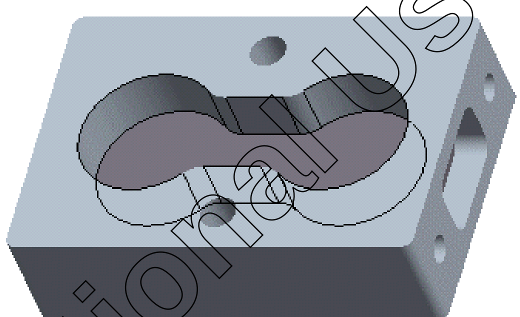


Figure 1: Highlighting pocket in sub plate model

Step 1. Locate and open the cover manufacturing model.

1. In the folder navigator browse through the folders and locate the directory: **c:\users\student\manufacturing_330\moduleA1**.

2. Open AUXILIARY.MFG.

3. Turn off any displayed datum features.

Step 2. Create an auxiliary milling NC Sequence to profile the pocket in the top of the sub plate.

1. Using the Menu Manager, create an auxiliary milling sequence.

2. Select the **FEM_012_0** tool.

3. Accept the default parameter settings.

> **Note:**
> Note the reduced list of parameters for auxiliary sequences. This is because the toolpath is created manually.

Step 3. Insert CL commands to turn on the spindle and coolant.

1. Using the Customize dialog box, insert a **CL Command**.
2. In the CL Command dialog box, select **Menu**.
3. To turn the spindle on, complete the following list of selections. Click **SELCTL-TMARK** > **SPINDL** > **ON** > **speed**, and type [**200**]. Click **RPM** > **CLW** > **QUIT OPTIONS** > **QUIT OPTIONS**. The command should appear in the Enter Command section.
4. To turn the coolant on, select **Menu** again and complete the following list of selections. Click **COOLNT-FROM** > **COOLNT** > **ON**. The command should appear in the Enter Command section.

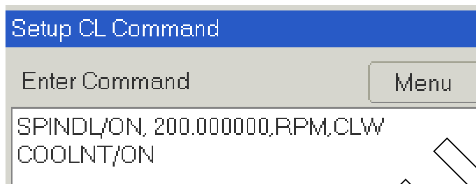


Figure 2: Showing the resulting CL commands.

5. Complete the CL command configuration.

> **Note:**
> Notice the CL commands appear in the CL Data window.

Step 4. Insert a Goto Point tool motion, and specify a feed rate.

1. Before proceeding, turn on the display of datum points.
2. Using the Customize dialog box, insert a **Goto Point**.
3. In the Goto Point dialog box, specify a Goto Target Point, **Select** the point on the left side of the pocket, as shown in the following figure.

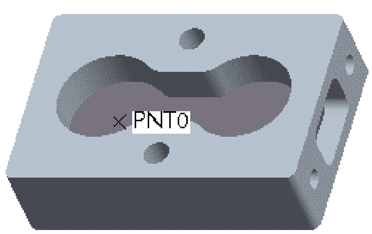


Figure 3: Showing point on left side of pocket

4. In the Goto Point dialog box, specify a feed rate of [**300**] *(11.81 in)*, specify the feed units as **MMPM**.
5. Complete the Goto Point configuration.

> **Note:**
> Notice the GOTO tool motion appears in the CL Data window.

Step 5. Insert a CL command to turn the cutter compensation on.

1. Using the Customize dialog box, insert a **CL Command**.
2. In the CL Command dialog box, select **Menu**.
3. To turn the CUTCOM on, complete the following list of selections. Click **COOLNT-FROM** > **CUTCOM** > **ON** > **QUIT OPTIONS** > **QUIT OPTIONS**. The command should appear in the Enter Command section.

```
17 SPINDL/ON, 200.000000,RPM,CLW
18 COOLNT/ON
19 RAPID
20 GOTO / 40.0000000000, 50.0000000000, 5.0000000000
21 FEDRAT / 300.000000, MMPM
22 GOTO / 40.0000000000, 50.0000000000, -20.0000000000
23 CUTCOM/ON
```

Figure 4: Showing the CL data file with inserted commands.

4. Complete the CL command configuration.

> **Note:**
> Notice the CUTCOM command appears in the CL Data window.

Step 6. Insert a follow sketch tool motion, to machine the profile of the pocket.

1. Using the Customize dialog box, insert a **Follow Sketch** tool motion.
2. In the Follow Sketch dialog box, select **Sketch**. Select the bottom surface of the pocket as the sketching plane, and select the right face of the sub plate as the right sketching reference.
3. In Sketcher, select the **Offset Edge** icon, and use the **Loop** option, then select the surface at the bottom of the pocket, type a value of [**-6**] *(0.24 in)* as the offset.

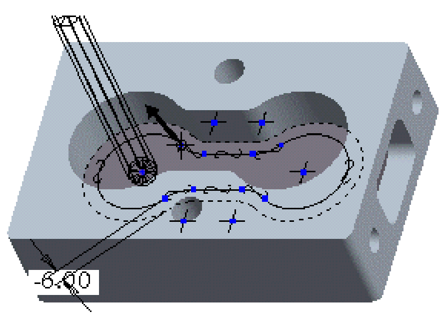


Figure 5: Showing offset loop of edges

4. Complete the sketch, and complete the follow sketch configuration.

> **Note:**
> Notice the follow sketch tool motion appears in the CL Data window.

Step 7. Insert a CL command to turn the cutter compensation off.

1. Using the Customize dialog box, insert a **CL Command**.
2. In the CL Command dialog box, select **Menu**.
3. To turn the CUTCOM on, complete the following list of selections. Click **COOLNT-FROM** > **CUTCOM** > **OFF** > **QUIT OPTIONS** > **QUIT OPTIONS**. The command should appear in the Enter Command section.
4. Complete the CL command configuration.

> **Note:**
> Notice the CUTCOM command appears in the CL Data window.

Step 8. Insert a Goto Point tool motion, to move the tool away from the profile.

1. Before proceeding turn on the display of datum points.
2. Using the Customize dialog box, insert a **Goto Point**.
3. In the Goto Point dialog box, specify a Goto Target Point, **Select** the point on the left side of the pocket, as before.
4. Complete the Goto Point configuration.

> **Note:**
> Notice the GOTO tool motion appears in the CL Data window.

5. Complete the customization.
6. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

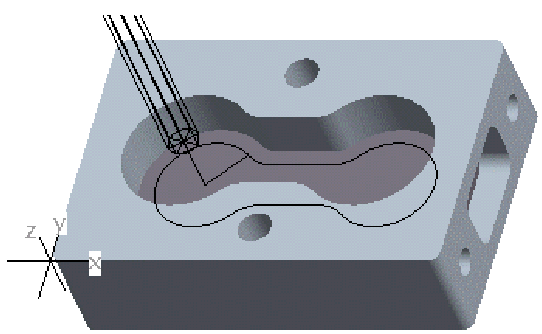


Figure 6: Showing the resulting tool path.

7. Check the format and position of the inserted CL data commands in the tool path file.

8. Save the manufacturing model.

9. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the auxiliary milling process.
- Create auxiliary milling sequences by defining tool motions and inserting CL commands.

Appendix B

Creating Pocket Milling Sequences

Introduction

Pocket milling sequences are typically used after volume milling to finish mill the surfaces of a pocket. The pocket can include horizontal, vertical, or slanted surfaces. Surfaces to be machined are selected from within the pocket geometry. The pocket walls are profile milled, and the bottom of the pocket is machined as in volume milling.

Objectives

After completing this module, you will be able to:

- Describe the pocket milling process.
- Create pocket milling sequences.

Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

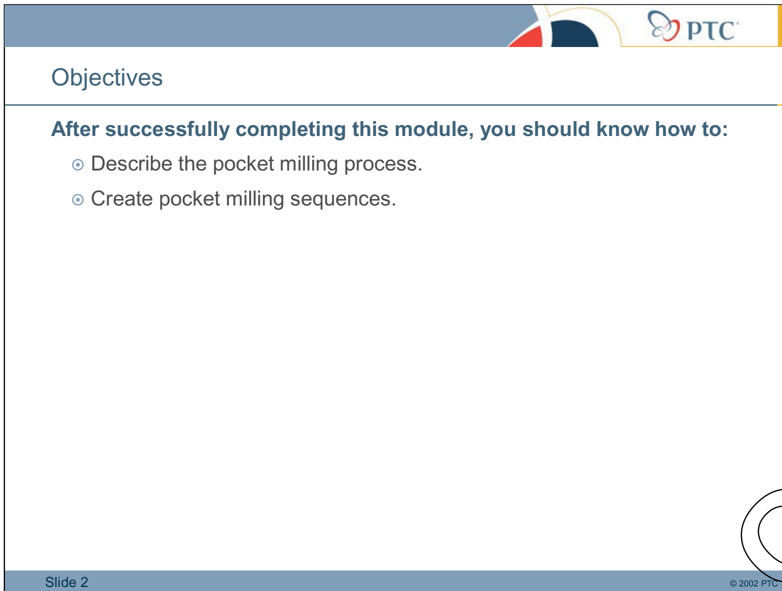
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN /T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology

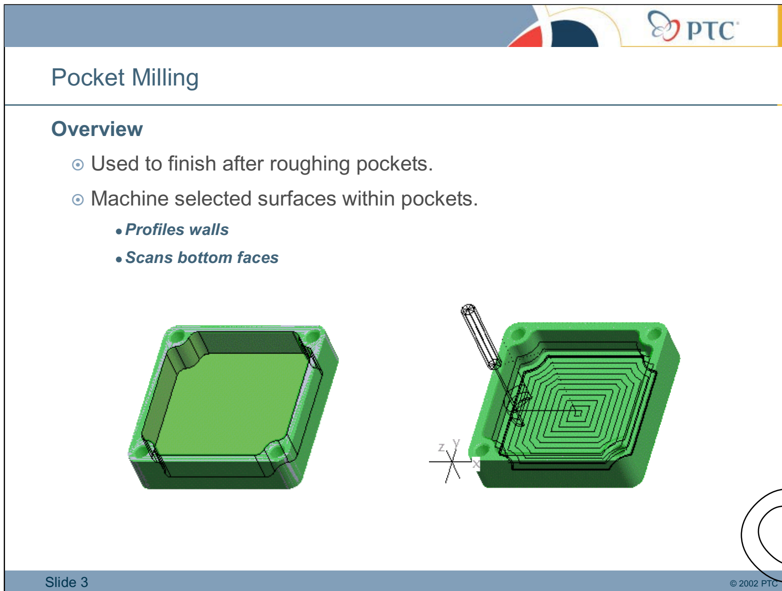


Overview

Pocket milling sequences are typically used after volume milling to finish mill the surfaces of a pocket. The pocket can include horizontal, vertical, or slanted surfaces. Surfaces to be machined are selected from within the pocket geometry. The pocket walls are profile milled, and the bottom of the pocket is machined as in volume milling.

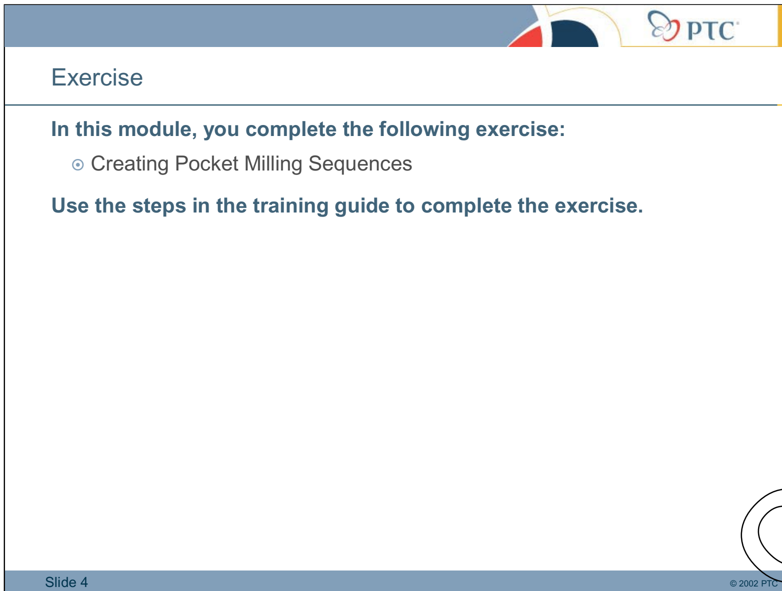
After completing this module, you will be able to:

- Describe the pocket milling process.
- Create pocket milling sequences.



Pocket Milling

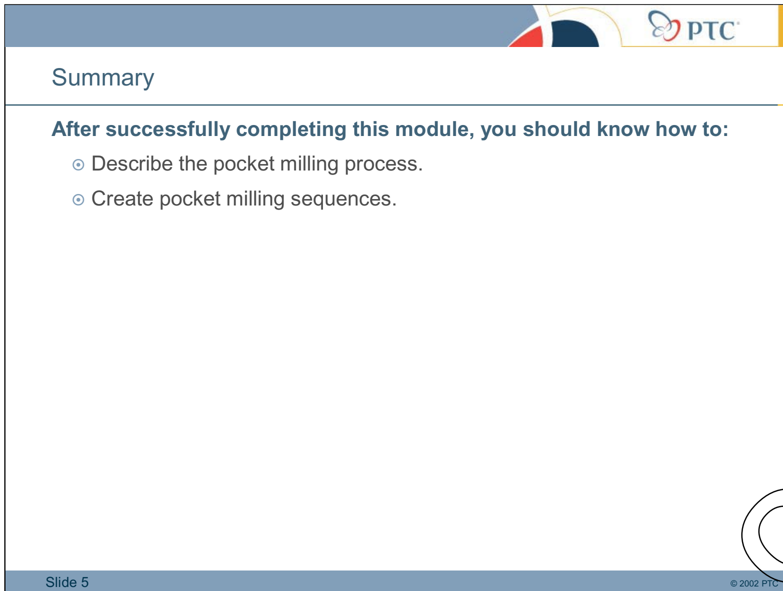
- Pocket milling sequences machine selected surfaces within pockets. The selected surfaces must allow for a continuous tool path, for instance, you cannot leave gaps in the selected surfaces.
  - The tool profile mills the walls, applying lead-in/lead-out and cutter compensation if required.
  - The tool machines the bottom faces of the pocket in the same way as a volume milling sequence.



Exercise: Creating Pocket Milling Sequences

Exercise

Use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Describe the pocket milling process.
- Create pocket milling sequences.

Appendix B Lab Exercises

Exercise 1: Creating Pocket Milling Sequences

Objectives

After successfully completing this exercise, you will know how to:

- Create pocket milling sequences.

Scenario

You need to create a pocket milling sequence to machine the sidewalls and bottom face of the pocket in the base cover model. A volume milling sequence has already roughed out the pocket leaving a small amount of material on all pocket faces. You create a pocket milling sequence to remove the remaining material in the pocket.

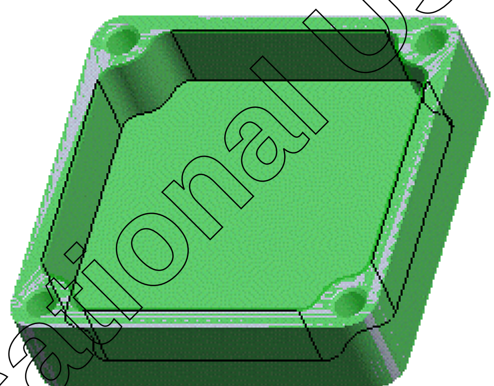


Figure 1: Highlighting pocket in base cover model

Step 1. Locate and open the cover manufacturing model

1. In the folder navigator browse through the folders and locate the directory: **c:\users\student\manufacturing_330\moduleA2**.
2. Open BASE_COVER.MFG.
3. Turn off any displayed datum features.

Step 2. Create a pocket milling NC Sequence to machine the pocket surfaces.

1. Using the Menu Manager, create a pocket milling sequence.
2. Select the **Name** and **Tool** check boxes.
3. Name the sequence [**FIN_POCKET**].
4. Select the **MMFLT08** tool.

5. Using the **Model** option, select all the surfaces within the pocket for machining.

> **Tip:**
> To select multiple surfaces press CTRL and select the pocket surfaces.

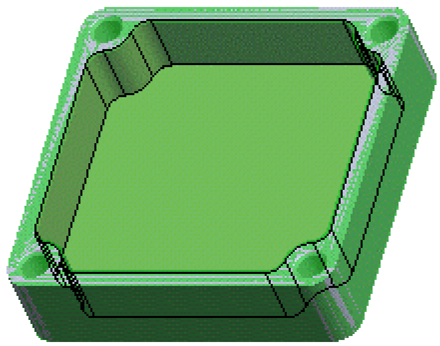


Figure 2: Showing selected surfaces in the pocket.

6. Complete the surface selection.

7. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

8. Modify the SCAN_TYPE parameter to TYPE_SPIRAL.

9. Play the tool path using the **Screen Play** option. Ensure the CL data is visible.

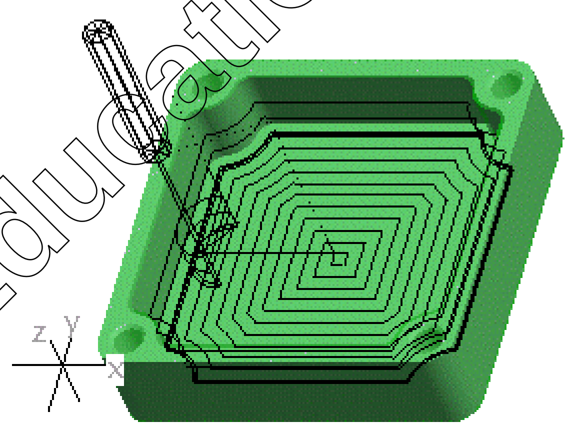


Figure 3: Showing the resulting tool path.

> **Toolpath Review:**
> Notice the tool profiles the walls first, applying cutter compensation and lead-in and lead-out moves. The tool then machines the bottom face of the pocket, leaving a small amount of material on the side walls. The tool then returns to profile the walls at the full depth of the pocket.

10. Complete the NC sequence.

11. Create a material removal feature for the pocketing sequence.

12. Save the manufacturing model.

13. Close all windows and erase all components from memory.

Summary

After successfully completing this module, you should know how to:

- Describe the pocket milling process.
- Create pocket milling sequences.

Appendix

C

Extra Models

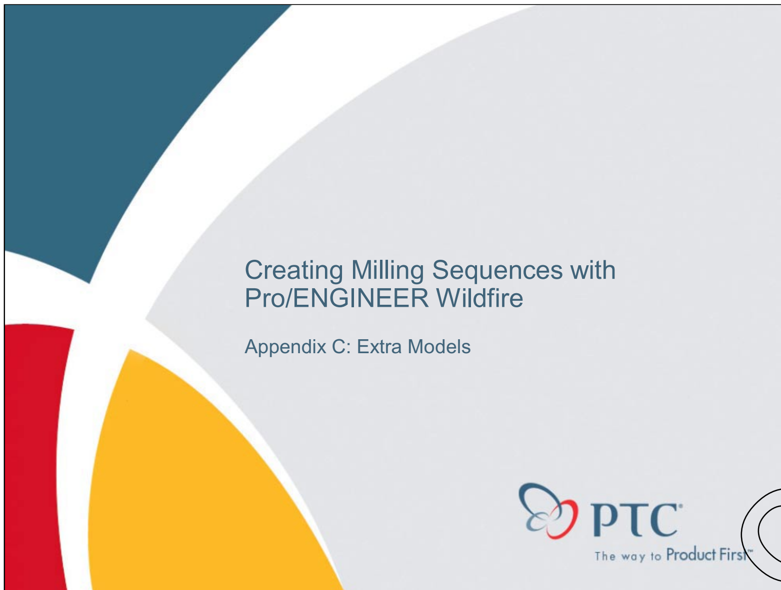
Introduction

This module provides a variety of extra models to practice manufacturing techniques.

Objectives

After completing this module, you will be able to:

- Create manufacturing models with appropriate operation set-up information.
- Create applicable NC Sequences based on model geometry.



Instructor Preparation

Before teaching *Creating Milling Sequences with Pro/ENGINEER Wildfire*, you **must** have read and **thoroughly understood** the following materials:

- General Information
  - Review Pro/ENGINEER Wildfire primer site at http://rdweb.ptc.com/primer/

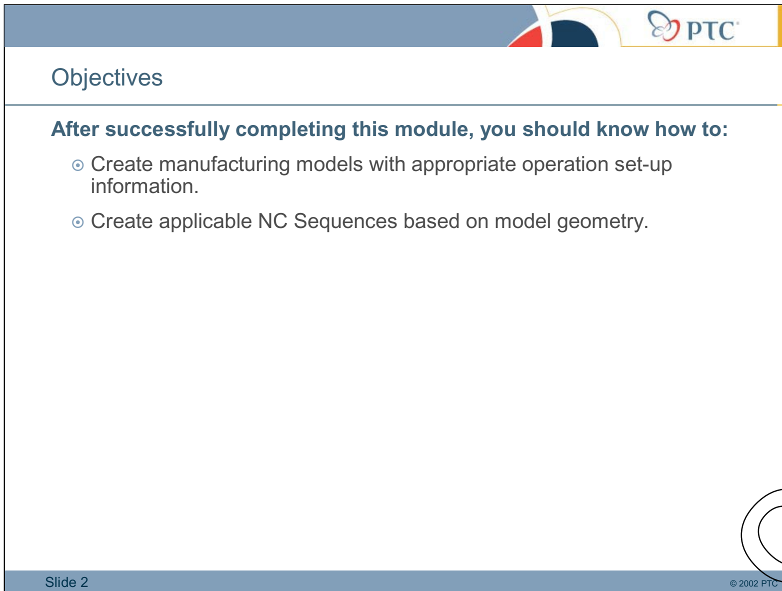
- Pro/E Wildfire Documentation
  - Review Pro/ENGINEER Wildfire documentation in Windmill, located in

/GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts

- Pro/E Wildfire Milling
  - Review Pro/ENGINEER Wildfire Milling training materials in Windmill, located in

/GS Ed Srvcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN
/T979-330-Instructor_Kit-EN

- Required Skills
  - Completion of *Pro/E Wildfire Foundation* or equivalent Pro/E skills
  - Completion of *Creating Milling Sequences with Pro/ENGINEER Wildfire* or equivalent Pro/E skills
  - Ability to install and configure Pro/E Wildfire
  - Ability to follow general classroom setup procedure
  - Good understanding of manufacturing processes and terminology

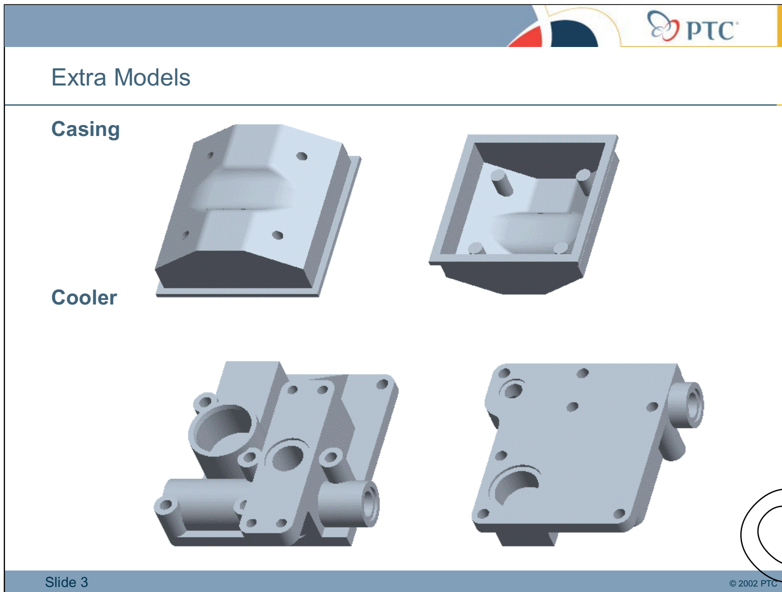


Overview

This module provides a variety of extra models to practice manufacturing techniques.

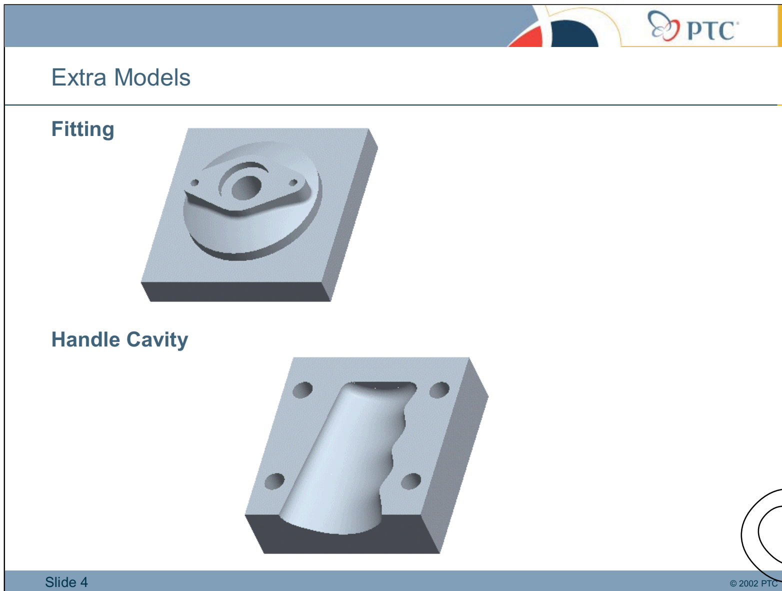
After completing this module, you will be able to:

- Create manufacturing models with appropriate operation set-up information.
- Create applicable NC Sequences based on model geometry.



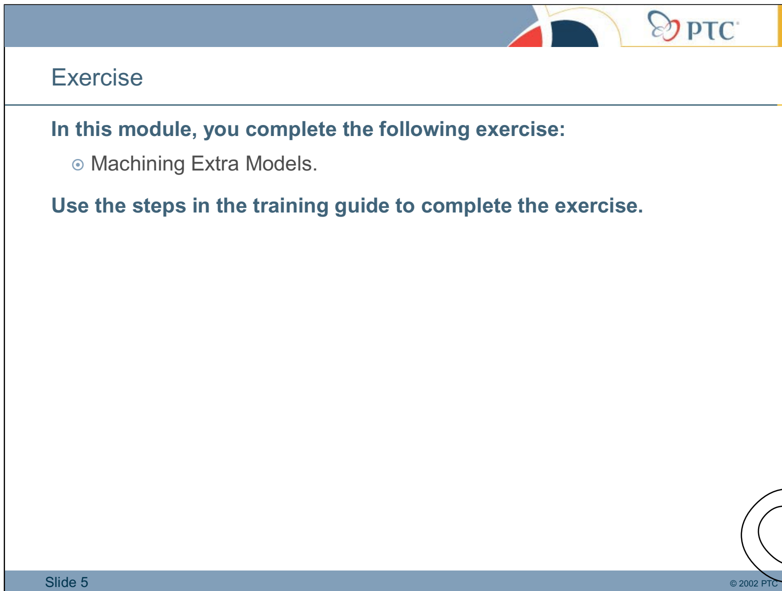
Extra Models

- Casing model
- Cooler model



Extra Models

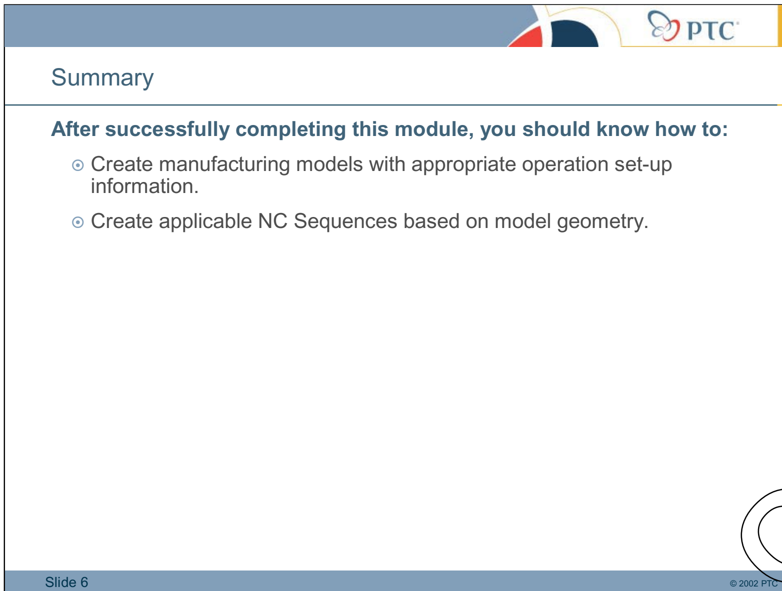
- Fitting model
- Handle Cavity model



Exercise: Machining Extra Models

Exercise

Use the steps in the training guide to complete the exercises.



Summary

After successfully completing this module, you should know how to:

- Create manufacturing models with appropriate operation set-up information.
- Create applicable NC Sequences based on model geometry.

Appendix C Lab Exercises

Exercise 1: Machining Extra Models

Objectives

After successfully completing this exercise, you will know how to:

- Create manufacturing models with appropriate operation set-up information.
- Create applicable NC Sequences based on model geometry.

Scenario

Select a model that best resembles the type of parts you machine.

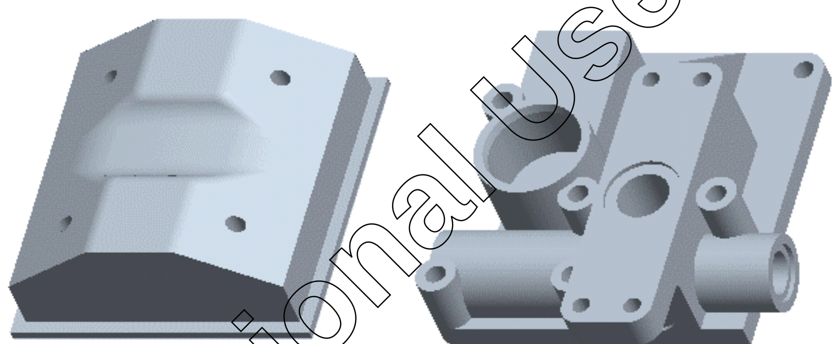


Figure 1: Showing casing and cooler models.

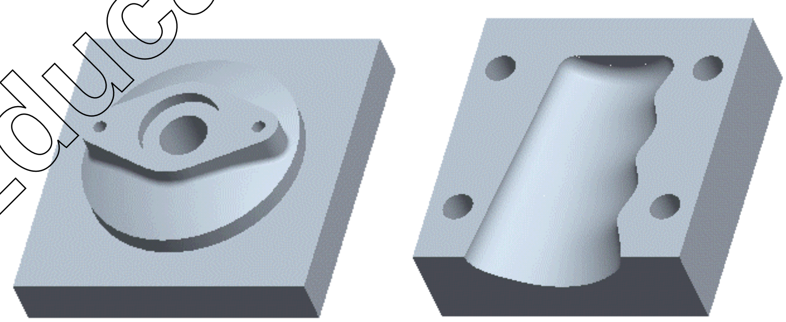


Figure 2: Showing fitting and handle cavity models.

Step 1. Locate the moduleA3 folder and machine a model of your choice.

1. In the folder navigator browse through the folders and locate the directory: **c:\users\student\manufacturing_330\moduleA3**.
2. Open a model of your choice and create a manufacturing model.
3. Create NC sequences to machine the model.

Summary

After successfully completing this module, you should know how to:

- Create manufacturing models with appropriate operation set-up information.
- Create applicable NC Sequences based on model geometry.