

PTC Global Services



Fast Track to Pro/ENGINEER Wildfire

T994-330-01



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Fast Track to Pro/ENGINEER Wildfire

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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.

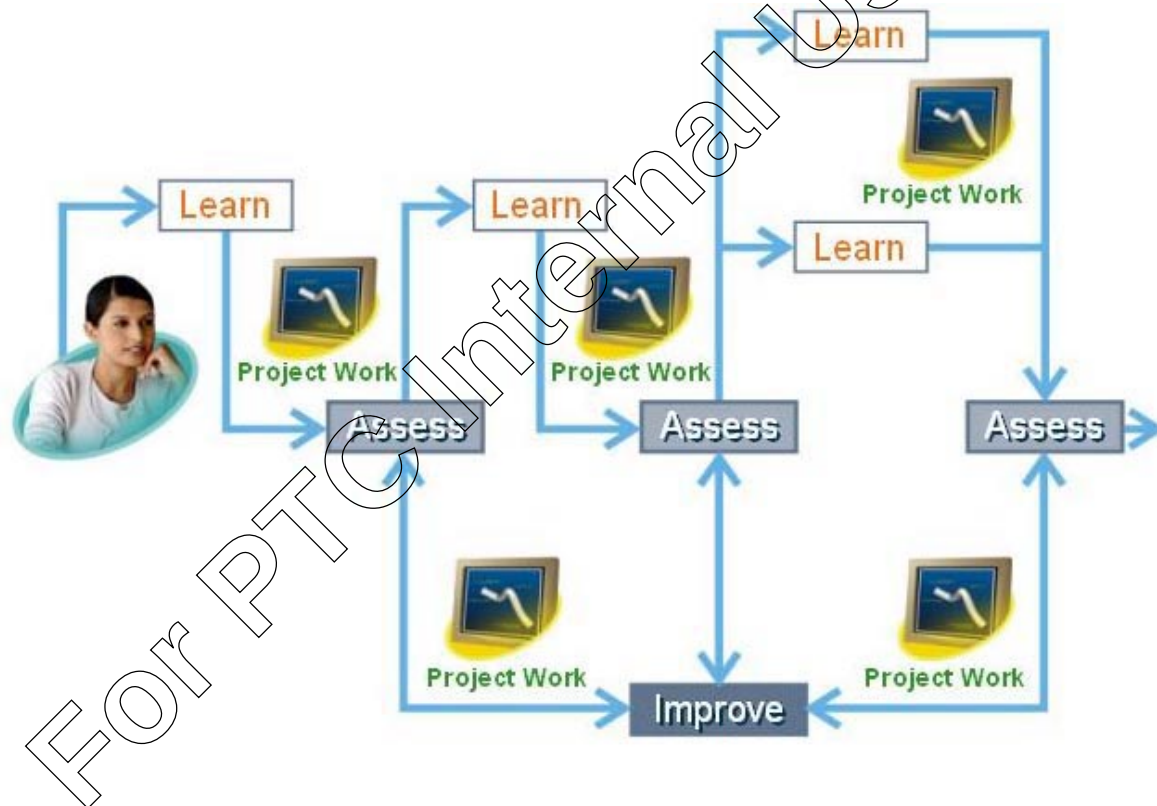
Precision Learning

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.

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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 <i>Dimensioning Scheme</i>
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 <i>Swept Forms</i>
Resolve Mode	Create some simple models and make them fail.
Resolve Mode	Web lesson <i>Resolve Mode</i>

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	
Advanced 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

Fast Track to 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	2003100

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Training Agenda

Fast Track to Pro/ENGINEER Wildfire

Day 1

Introduction to Pro/ENGINEER Wildfire

Editing Design Models

Creating Reference Geometry

Creating Direct Features

Day 2

Capturing Design Intent with Sketcher

Creating Sketch-based Features I

Creating Sketch-based Features II

Project I

Day 3

Duplicating Design Model Features

Enhancing Design Models

Assembling Components

Project II

Day 4

Creating Mechanism Connections

Creating Drawings

Resolving Regeneration Failures

Project III

Day 5

Managing Assemblies

Analyzing and Optimizing Design Models

Project IV

Table of Contents

Fast Track to Pro/ENGINEER Wildfire

THE PRECISION LEARNING METHODOLOGY.....	i
CONTINUOUS IMPROVEMENT	ii
PRECISION LEARNING IN THE CLASSROOM	iii
EDUCATION CIRCUIT EXAMPLE	iv
INTRODUCTION TO PRO/ENGINEER WILDFIRE	1-1
Module 1 Lab Exercises.....	1-3
Exercise 1: Previewing and Opening Design Models.....	1-3
Exercise 2: Orienting Design Models	1-8
Summary	1-12
EDITING DESIGN MODELS	2-1
Module 2 Lab Exercises.....	2-3
Exercise 1: Selecting Components and Features	2-3
Exercise 2: Editing Design Models	2-9
Exercise 3: Creating and Managing Layers	2-16
Summary	2-18
CREATING REFERENCE GEOMETRY	3-1
Module 3 Lab Exercises.....	3-3
Exercise 1: Accessing Pro/ENGINEER Wildfire Help Center.....	3-3
Exercise 2: Creating Reference Geometry	3-5
Exercise 3: Creating Advanced Reference Geometry.....	3-12
Exercise 4: Creating and Managing Layers	3-20
Summary	3-22
CREATING DIRECT FEATURES	4-1
Referencing PTC Help for Module 4 Topics	4-3
Accessing Pro/ENGINEER Wildfire Help Center	4-3
Module 4 Lab Exercises.....	4-4
Exercise 1: Creating Direct Features – Holes	4-4
Exercise 2: Creating Direct Features – Rounds	4-10
Exercise 3: Creating Direct Features – Chamfers.....	4-22

Exercise 4: Creating Direct Features – Drafts	4-28
Exercise 5: Creating Direct Features – Shells	4-32
Exercise 6: Creating and Managing Layers	4-34
Exercise 7: Creating Direct Features on the Chuck (Challenge)	4-36
Exercise 8: Creating Direct Features on the Flywheel (Challenge)	4-38
Exercise 9: Shelling the Engine Cover (Challenge)	4-42
Exercise 10: Creating Drafts on the Cylinder (Challenge)	4-43
Summary	4-45
 CAPTURING DESIGN INTENT WITH SKETCHER	 5-1
Referencing PTC Help for Module 5 Topics	5-3
Accessing Pro/ENGINEER Wildfire Help Center	5-3
Module 5 Lab Exercises	5-4
Exercise 1: Creating New Design Models Using Default Templates	5-4
Exercise 2: Capturing Design Intent with Sketcher	5-7
Exercise 3: Creating Datum Features (Challenge)	5-22
Summary	5-25
 CREATING SKETCH-BASED FEATURES I	 6-1
Referencing PTC Help for Module 6 Topics	6-3
Accessing Pro/ENGINEER Wildfire Help Center	6-3
Module 6 Lab Exercises	6-4
Exercise 1: Creating Extruded and Revolved Features	6-4
Exercise 2: Creating Ribs and Group Datum Features	6-16
Exercise 3: Completing the Piston (Challenge)	6-21
Summary	6-24
 CREATING SKETCH-BASED FEATURES II	 7-1
Referencing PTC Help for Module 7 Topics	7-3
Accessing Pro/ENGINEER Wildfire Help Center	7-3
Module 7 Lab Exercises	7-4
Exercise 1: Creating Sweeps	7-4
Exercise 2: Creating Blends	7-7
Exercise 3: Sweeping Along a 3-D Trajectory	7-10
Exercise 4: Creating a Blend Using Grouped Datum Features (Challenge)	7-12
Summary	7-15
 PROJECT I	 8-1
Module 8 Lab Exercise	8-3

Objective 1: Creating the Piston_Pin	8-3
Objective 2: Creating the Piston.....	8-5
Objective 3: Creating the Connecting_Rod	8-10
Objective 4: Creating the Crankshaft	8-15
Objective 5: Creating the Engine_Block.....	8-22
Objective 6: Creating the Impeller_Housing.....	8-31
Objective 7: Creating the Frame	8-40
Summary	8-46

DUPLICATING DESIGN MODEL FEATURES

9-1

Referencing PTC Help for Module 9 Topics	9-3
Accessing Pro/ENGINEER Wildfire Help Center	9-3
Module 9 Lab Exercises.....	9-4
Exercise 1: Patterning Features.....	9-4
Exercise 2: Copying Features	9-16
Exercise 3: Mirroring Models.....	9-21
Exercise 4: Creating and Managing Layers	9-23
Summary	9-25

ENHANCING DESIGN MODELS USING FAMILY TABLES, RELATIONS, AND PARAMETERS

10-1

Referencing PTC Help for Module 10 Topics	10-3
Accessing Pro/ENGINEER Wildfire Help Center	10-3
Module 10 Lab Exercises.....	10-4
Exercise 1: Creating Relations.....	10-4
Exercise 2: Creating Family Tables	10-9
Exercise 3: Creating Family Tables (Challenge).....	10-13
Summary	10-14

ASSEMBLING COMPONENTS

11-1

Referencing PTC Help for Module 11 Topics	11-3
Accessing Pro/ENGINEER Wildfire Help Center	11-3
Module 11 Lab Exercises.....	11-4
Exercise 1: Assembling Using Constraints	11-4
Exercise 2: Assembling Using Component Interfaces	11-11
Exercise 3: Assembling Flexible Components	11-17
Exercise 4: Assembling Additional Engine Components (Challenge)	11-22
Summary	11-28



PROJECT II	12-1
Module 12 Lab Exercises	12-3
Objective 1: Creating the Impeller part	12-3
Objective 2: Completing the Frame	12-13
Objective 3: Completing the Impeller Housing	12-19
Objective 4: Continuing the Engine_Block	12-25
Objective 5: Creating Assemblies	12-30
Summary	12-37
 CREATING MECHANISM CONNECTIONS	 13-1
Referencing PTC Help for Module 13 Topics	13-3
Accessing Pro/ENGINEER Wildfire Help Center	13-3
Module 13 Lab Exercises	13-4
Exercise 1: Assembling Using Joint Connections	13-4
Exercise 2: Assembling Using Gear Connections	13-11
Exercise 3: Completing the Drill Assembly (Challenge)	13-17
Summary	13-22
 CREATING DRAWINGS	 14-1
Referencing PTC Help for Module 14 Topics	14-3
Accessing Pro/ENGINEER Wildfire Help Center	14-3
Module 14 Lab Exercises	14-4
Exercise 1: Creating Drawing Views	14-4
Exercise 2: Detailing Drawings	14-14
Summary	14-34
 RESOLVING REGENERATION FAILURES	 15-1
Referencing PTC Help for Module 15 Topics	15-3
Accessing Pro/ENGINEER Wildfire Help Center	15-3
Module 15 Lab Exercises	15-4
Exercise 1: Obtaining Design Information	15-4
Exercise 2: Resolving Part Failures	15-8
Exercise 3: Resolving a Part Failure from a Drawing (Challenge)	15-15
Exercise 4: Resolving Assembly Failures (Challenge)	15-18
Summary	15-21
 PROJECT III	 16-1
Module 16 Lab Exercises	16-3
Objective 1: Creating Mechanism Connections	16-3



Objective 2: Creating a Drawing	16-10
Objective 3: Completing the Engine Block part.....	16-25
Summary	16-44

MANAGING ASSEMBLIES 17-1

Referencing PTC Help for Module 17 Topics	17-3
Accessing Pro/ENGINEER Wildfire Help Center	17-3
Module 17 Lab Exercises.....	17-4
Exercise 1: Creating Style and Simplified Representations	17-4
Exercise 2: Using Skeletons to Design Models.....	17-12
Exercise 3: Creating View Orientations and Explode States (Challenge)	17-21
Summary	17-37

ANALYZING AND OPTIMIZING DESIGN MODELS 18-1

Referencing PTC Help for Module 18 Topics	18-3
Accessing Pro/ENGINEER Wildfire Help Center	18-3
Module 18 Lab Exercises.....	18-4
Exercise 1: Analyzing Design Models	18-4
Exercise 2: Analyzing the Fuel Tank Volume.....	18-9
Summary	18-12

PROJECT IV 19-1

Module 19 Lab Exercises.....	19-3
Objective 1: Analyzing the AC-40 for Interferences	19-3
Objective 2: Creating and Assembling the Bolt.....	19-10
Objective 3: Creating and Assembling the Engine Head	19-14
Objective 4: Creating and Assembling the Glow Plug.....	19-25
Objective 5: Creating and Assembling the Muffler (Challenge).....	19-30
Objective 6: Creating and Assembling the Carburetor (Challenge).....	19-38
Objective 7: Creating and Assembling the Engine Cover (Challenge).....	19-44
Objective 8: Creating and Assembling the Flange (Challenge).....	19-48
Objective 9: Creating and Assembling the Nut (Challenge).....	19-51
Summary	19-55

Introduction to Pro/ENGINEER Wildfire

Introduction

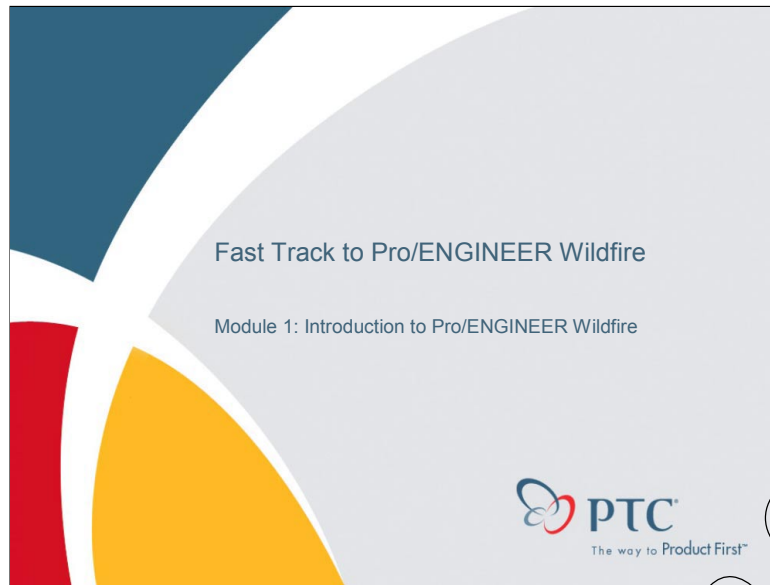
Pro/ENGINEER Wildfire enables you to quickly create, manipulate and orient design models within an intuitive, Windows™-style interface.

Using the Pro/ENGINEER Wildfire navigator, you can easily locate, preview, and open design models. You can also access the Web and collaborate with other designers.

Objectives

After completing this module, you will be able to:

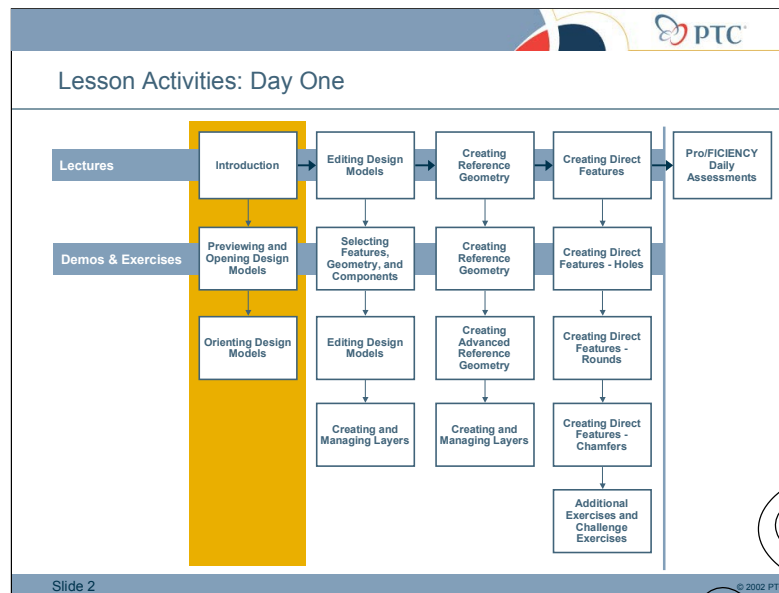
- Navigate the main Pro/ENGINEER Wildfire interface.
- Preview, open, and orient design models.
- Manage Pro/ENGINEER Wildfire memory and directories.



Instructor Preparation

Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** 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 Foundation
 - Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (2): 20 mins
- Labs (2): 30 mins
- Total: 1hr



Objectives

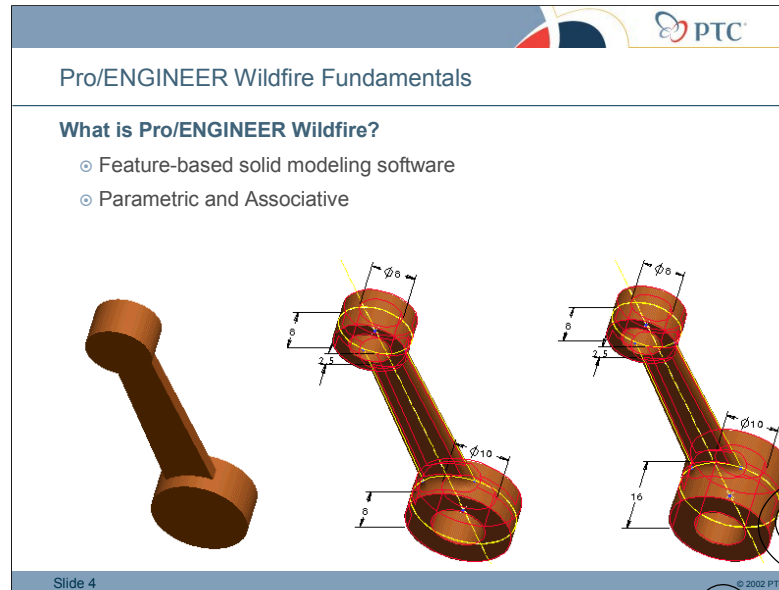
After completing this module, you will be able to:

- Navigate the main Pro/ENGINEER Wildfire interface.
- Preview, open, and orient design models.
- Manage Pro/ENGINEER Wildfire memory and directories.

Slide 3

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- With Pro/ENGINEER Wildfire, you can quickly create design models in an intuitive, Windows™-style interface.
- Manipulating and orienting design models is an important basic skill that you will use whenever working in Pro/ENGINEER Wildfire.
- The navigator is another important feature of Pro/ENGINEER Wildfire. The navigator enables you to quickly locate, preview, and open design models.



Feature-based

- Models are constructed using a series of features. Each feature builds upon the previous feature. Individually each feature can be simple, but taken together all features create complex parts and assemblies.

Solid modeling

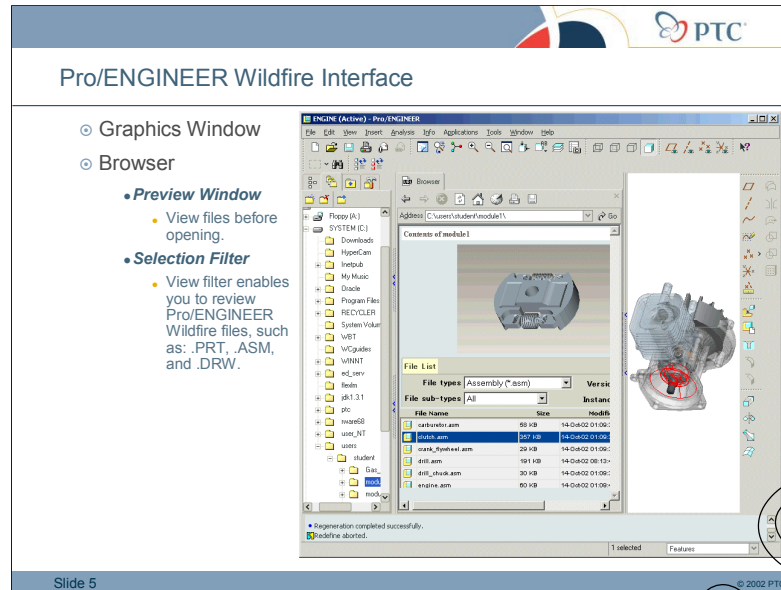
- All models contain properties such as mass, volume, center of gravity, surface area, etc... As features are added or removed from the model, then the properties update. For example if you add a hole to a block, then the mass of the block will change. In addition, solid models enable tolerance analysis and clearance/interference checking.

Parametric

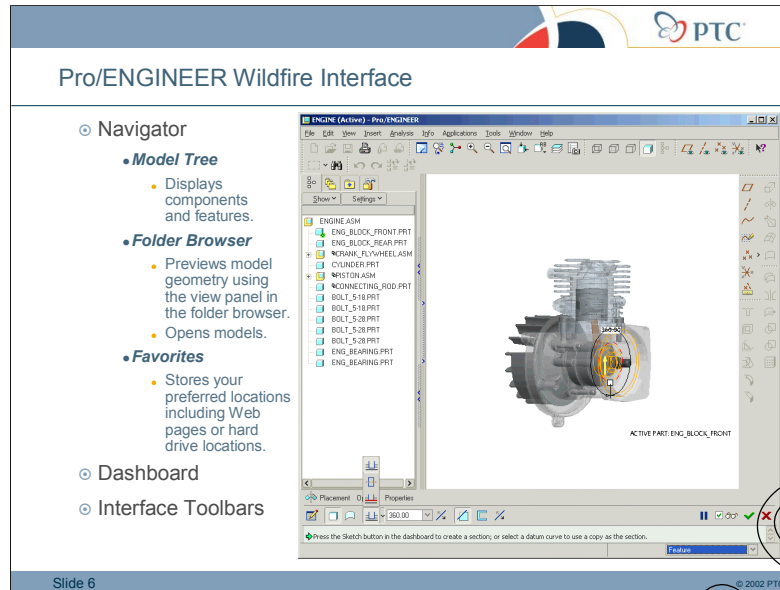
- The models are value driven using dimensions and parameters. If you change a dimension of the model, then the 3-D representation of the model will change. This is not the case in a 2-D modeling software because the length of a line will not change if you change the dimension.

Associative

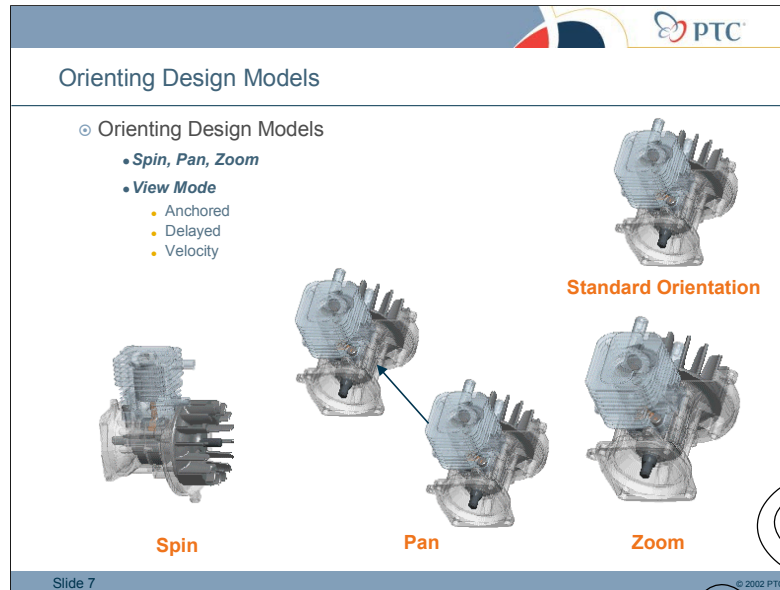
- All deliverables are based off of the part model. If you change a dimension on the part model, then all deliverables such as assemblies, drawings, and manufacturing files will reflect the change. For example if we have a connecting rod that is a cast part, and we change the length of the connecting rod. Then the deliverables such as the drawing of the connecting rod and the tools path associated to the manufacturing file will also update to reflect the change in length.
- Note that associativity is bi-directional, which means that if you change the dimension of a part within an assembly, then the part viewed individually will also change.



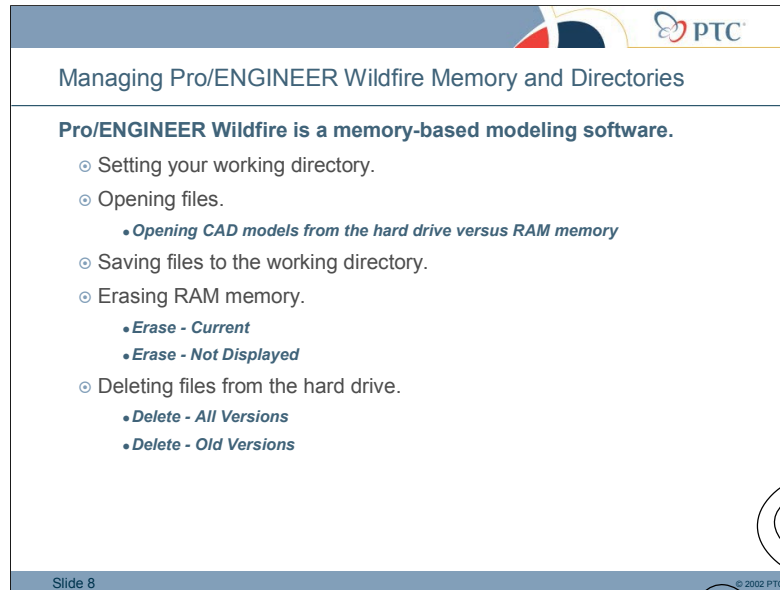
- In Pro/ENGINEER Wildfire, you design solid and sheet metal models in the graphics window.
- Design models consist of parts and assemblies.
- The Selection Filter is a drop-down list located on the bottom right of the graphics window.
- You can control the type of geometry to be selected on the model using the options in the filter.
- To design the model, you use simple features as the building blocks.
- Each feature is created by referencing other features.
- You can also create drawings using design information from the existing models.
- Parts, assemblies and drawings are completely associative, modifying one updates the others.



- The model tree provides the order in which features are created and regenerated.
- Features can be selected to perform operations such as editing, deleting, and hiding.
- You use the folder browser to navigate the local folders, and preview and open design models.
- You preview model geometry using the view panel in the folder browser.
- Model geometry displays in the default 3-D saved orientation.
- You can easily identify the model you want to open, instead of only being able to view the name of the model.
- You can open design models directly from the folder browser after previewing the geometry.
- You use the Web browser to connect to the Web for help information, downloading supplier parts, and performing design collaboration.
- When you design solid models, you use feature tools to create direct and sketch-based features.
- You use datum tools to create non-solid reference geometry.
- You use editing tools to create curves and surfaces, duplicate and edit solid features.
- The dashboard displays on the bottom left of the graphics window. It guides you through the sequence of steps required when creating and redefining features.



- 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 the orientations for later use.
- 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 referencing geometry such as an edge or surface with View Mode.



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.

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.

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 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

- ◉ Previewing and Opening Design Models
- ◉ Orienting Design Models

Slide 9

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You will follow along while I demonstrate:

1. Previewing and Opening Design Models
2. Orienting Design Models

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- ◉ Navigate the main Pro/ENGINEER Wildfire interface.
- ◉ Preview, open, and orient design models.
- ◉ Manage Pro/ENGINEER Wildfire memory and directories.

For PTC Internal Use Only

Module 1 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demonstration for customers:

- Practice running the demonstration so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Conventions

- Demonstration script is written in italics.

Exercise 1: Previewing and Opening Design Models

Introduction

In this demonstration, we preview models and documents using the embedded Web browser in the Pro/ENGINEER Wildfire interface. Using the Wildfire navigator, we can preview all the assembly components of any model before opening it. These simple features save a lot of time in the design process.

Objectives

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

- Locate design data using the Pro/ENGINEER Wildfire navigator.
- Preview and open design models.

Scenario

Cordless Power Tools, Inc. (CPT) designs and manufactures hand-held power tools, including cordless drills, circular saws, and hammer drills. CPT is currently developing a new cordless drill that uses a gasoline-powered two-cycle engine instead of a battery-powered electric motor. The gasoline-powered engine produces more power than a battery-operated motor. It is marketed to commercial and industrial customers requiring drills that have a high torque capacity for drilling through steel and concrete, and a higher MTBF (Mean Time Before Failure) ratings. The gas-powered drill lasts four to five times longer than battery-powered drills, and can also be used in locations without a reliable power supply.

The CPT engineering design team completed its first gas-powered drill prototype assembly. The pre-production design review identified several problems with the virtual assembly. The engineering department manager has assigned you to correct these problems. Before you can implement any design modifications, you must review the existing drill components. You navigate to the latest design model, and then open and review the assembly components.



Figure 1: The Gas-powered Drill Assembly

Step 1. Locate the design model specification documents by browsing your department's Web site using the Pro/ENGINEER Wildfire navigator.

In this demonstration, we use the main Pro/ENGINEER Wildfire components to review the ECOs (Engineering Change Orders) filed by the pre-production design team. These ECOs have identified problems with the virtual assembly in the drill assembly prototype.

Starting with the navigator, notice how the tabs enable us to quickly change between the model tree, the folder browser, the personal favorites folder, and to different Web site connections.

The navigator enables us to work outside of Pro/ENGINEER Wildfire. We can access suppliers and contractors via their Web sites. We can also access PDM (Product Data Management) systems.

Using the navigator, we access our department's Web site, and then locate and review the ECOs.

1. Start Pro/ENGINEER Wildfire.
2. Review the links in the browser. Interactive tours, tutorials, and quick reference materials are available. You can also use the Menu Mapper to map menu options from previous releases to Pro/ENGINEER Wildfire.
3. Minimize the navigator and the browser.

4. Restore the navigator.
5. Using the folder navigator, browse to the following location:
c:\users\student\fast_track_330\module_01-02\ECO.
6. Select the **ECO** folder to open the browser. Double-click on the *.htm files to review the ECO documents.

Step 2. Add a Favorite within Pro/ENGINEER Wildfire.

***Instructor:** The current link to Pro/FICIENCY is <http://www-ed.ptc.com/ptc/tbls>. However, it may soon be changed to <http://www-ed.ptc.com/ptc/proficiency>. Test the link before writing it on the board for the students to refer to in this step.*

1. Select the Favorites tab.
2. In the browser, enter the link to Pro/FICIENCY provided by your instructor.
3. Add a favorite named **Pro/FICIENCY**.

Step 3. Browse the folders containing the design models, and then use the navigator to set your working directories for the project.

Next, we switch back to the navigator folder view. This enables us to browse through the different folders and find the design models referenced in the ECOs. We also set a working directory. Any files that we create or modify are automatically saved to this folder.

1. Select the folder navigator, and browse to the following folder:
c:\users\student\fast_track_330\module_01-02\.
2. Right-click and select **Make Working Directory**.

Step 4. Preview the design model geometry and locate the top-level assembly.

The navigator enables us to filter out everything but the files we need. First, we review all the files in top-level assembly folder, and then we filter out everything except for the part and assembly files. Next, we preview the files without opening them in Wildfire. The Preview feature enables us to find the smallest assembly to work with. It can also help us find the top-level assembly in the folder.

1. In the browser, select **Part (*.prt)** from the File Types drop-down list. Click **Apply**.

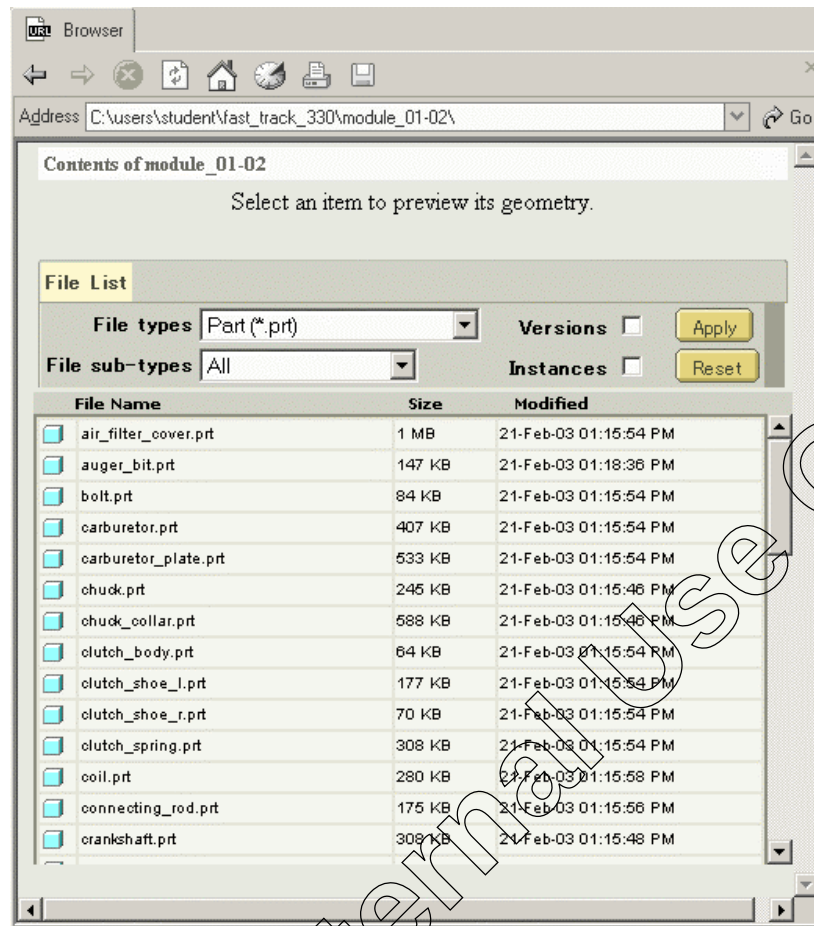


Figure 2. Configure the Navigator Display

2. In the browser, select and preview the following parts: AIR_FILTER_COVER.PRT, CONNECTING_ROD.PRT, and FLYWHEEL.PRT.

Note:

You can use the mouse to Spin/Pan/Zoom in the preview window.

3. In the browser, select **Assembly (*.asm)** from the File Types drop-down list. Click **Apply**.
4. Select DRILL.ASM to preview it.

Step 5. Open the top-level assembly.

We have located the top-level assembly. Next, we open the assembly.

1. Drag DRILL.ASM into the graphics window to open the model.
2. Turn off the display of all datum features using the toolbar.



Figure 3: Viewing DRILL.ASM in Pro/ENGINEER Wildfire

Note:

To open the assembly, you can also double-click the name of the model, or use the Open icon near the Preview window.

This completes the exercise.

Exercise 2: Orienting Design Models

Introduction

In this demonstration, we spin, pan, and zoom the models. These options enable us to quickly focus on specific areas of interest within models.

Objectives

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

- Orient the assembly components into any necessary view.

Scenario

After locating and opening the design model, you review its features. You orient the drill assembly and view it from several different angles. Orienting design models is important because it enables you to focus on the components and features that you need to revise.

Step 1. Review the ENGINE.ASM components using the spin, pan, zoom, and advanced view options.

We opened the engine assembly in Pro/ENGINEER Wildfire. Next, we manipulate the view angles in order to examine the modifications recommended in the ECOs.

We use the middle-drag function for all the View orientations.

To spin, cursor over the model and middle-drag. The model updates as we drag the mouse.

To pan the model, press SHIFT + middle-drag the model.

To zoom, press CTRL + middle-drag the model.

These middle-drag functions help us to quickly orient and evaluate a design.

In addition, with the advanced View options, we can delay the update of spin, pan and zoom.

With a 3-D model, we can set a constant velocity of change to the View options.

Each of these capabilities minimizes the amount of time it takes to evaluate a model.

1. Select ENGINE.ASM from the model tree, right-click, and select **Open**.

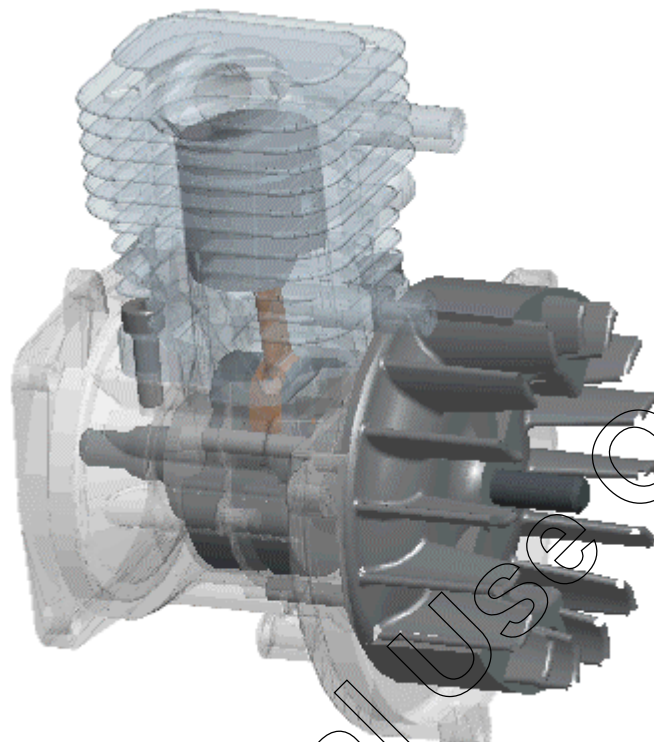


Figure 4: ENGINE.ASM

2. To spin, middle-drag in the graphics window.
3. Click **View > Orientation > Standard Orientation** to return the ENGINE.ASM to the standard orientation.

Note:

You can increase graphics performance by adjusting or disabling Transparency . Click **View > Display Settings > Model Display**. Select the Shade tab, and either set the Stippled option or disable the Transparency option.

4. Use the toolbar to disable the Spin Center.
5. Place the mouse cursor near the top of the CYLINDER.PRT, and middle-drag to spin ENGINE.ASM.
6. Place the mouse cursor near the bottom of ENGINE.ASM, and middle-drag to spin. Notice the center of rotation is the mouse cursor position.

Note:

With the Spin Center off, the mouse cursor position becomes the spin center.

7. Click **View > Orientation > Standard Orientation** to return the ENGINE.ASM to the standard orientation.
8. Use the toolbar to enable the Spin Center.

9. To zoom, move the mouse cursor over the CONNECTING_ROD.PRT, and press CTRL + middle-drag vertically, as shown in the following figure.

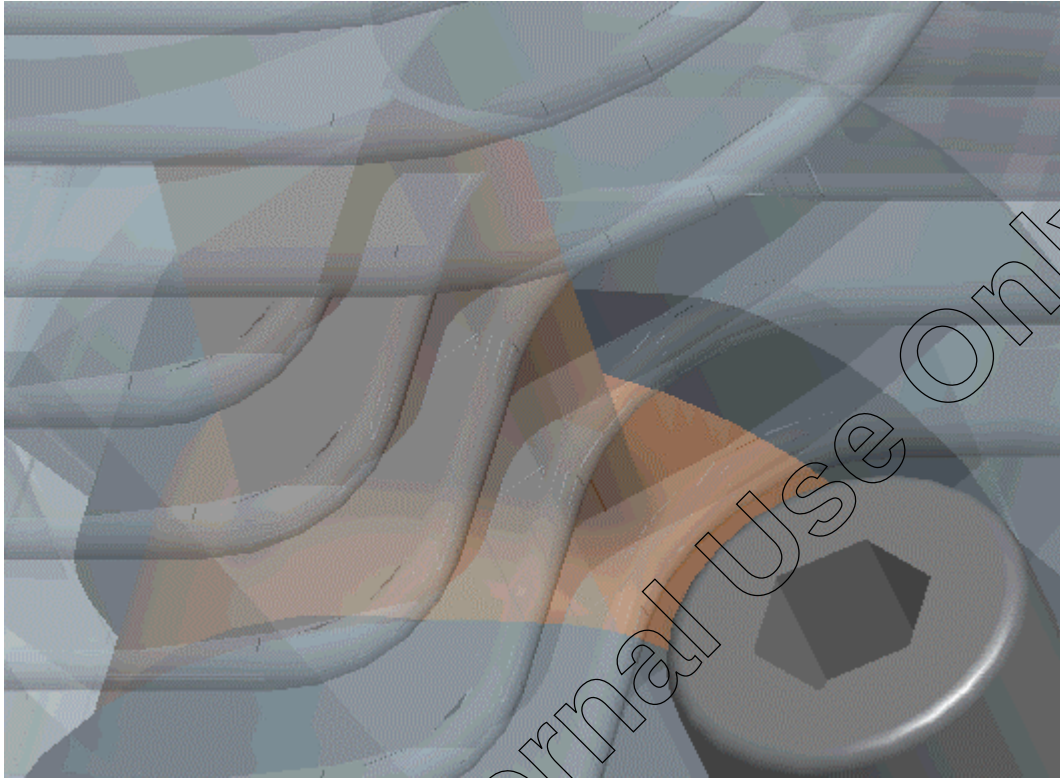


Figure 5: Panning and Zooming the ENGINE.ASM

10. Click **View > Orientation > Previous** to return to the previous view.
11. To spin the model to view FLYWHEEL.PRT, middle-drag in the graphics window.
12. To pan the model to center FLYWHEEL.PRT in the graphics window, press SHIFT + middle-drag.
13. To zoom, place the mouse cursor over a cooling fin, and press CTRL + middle-drag vertically.

Note:

You can also use Quick Zoom if you have a wheel mouse. To use Quick Zoom, position the mouse cursor, and then roll the wheel button.

14. Return to the standard orientation using the toolbar.
15. To turn ENGINE.ASM, move the mouse cursor over the model, press CTRL + middle-drag horizontally.
16. Press CTRL-D to return to the standard orientation. Activate View Mode using the toolbar.
17. Right-click and select **Anchored**. Spin, pan and zoom ENGINE.ASM.

18. Right-click and select **Delayed**. Spin, pan and zoom ENGINE.ASM.

19. Right-click and select **Velocity**. Spin, pan and zoom ENGINE.ASM.

Note:

Using View Mode, you can also toggle the Spin Center on and off. Additionally, you can middle click on geometry for edge or surface specific rotation.

20. Click **File > Close Window** to close the ENGINE.ASM window, and return to the DRILL.ASM.

21. Erase the models from memory. Click **File > Erase >Current > OK** to remove the DRILL.ASM from RAM memory.

This completes the exercise.

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Summary

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

- Navigate the main Pro/ENGINEER Wildfire interface.
- Preview, open, and orient design models.
- Manage Pro/ENGINEER Wildfire memory and directories.

For PTC Internal Use Only

Editing Design Models

Introduction

By selecting individual features and geometry, you can identify features, geometry, and components to edit. For example, before you add a hole feature to a part, you must select the placement and location references. This is a routine skill that is used throughout the modeling process.

Selection is important because after “selecting” on a feature(s), model(s), or geometry, you will be able to make modifications to the selected item(s). To make modifications to models, you must edit the model. Editing is important because a majority of new product designs are based upon designs that have already been completed.

Objectives

After completing this module, you will be able to:

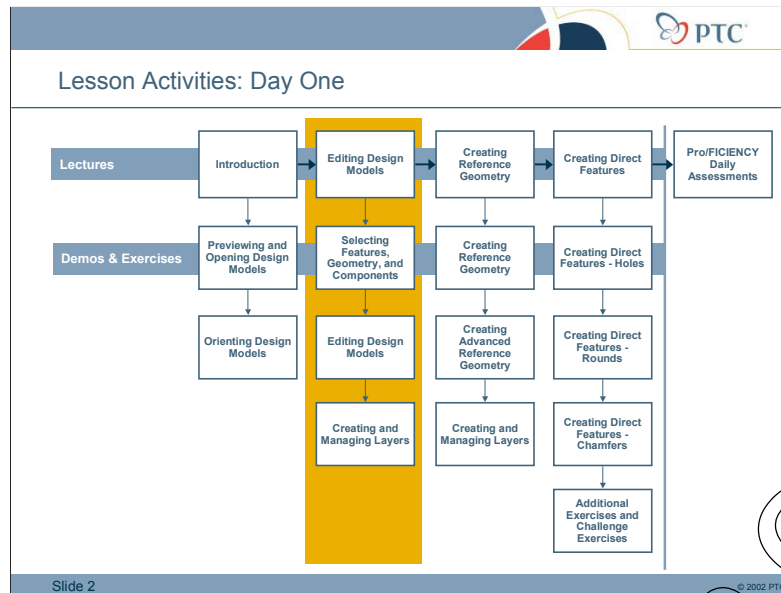
- Select features, geometry, and components of models.
- Edit design models.
- Describe how editing design models affects parent/child relationships.
- Manage layers when editing design models.

Fast Track to Pro/ENGINEER Wildfire

Module 2: Editing Design Models



For PTC Internal Use Only



Duration

- Lecture: 15 mins
- Demos (2): 30 mins
- Labs (2): 30 mins
- Total: 1hr 15 mins



Objectives

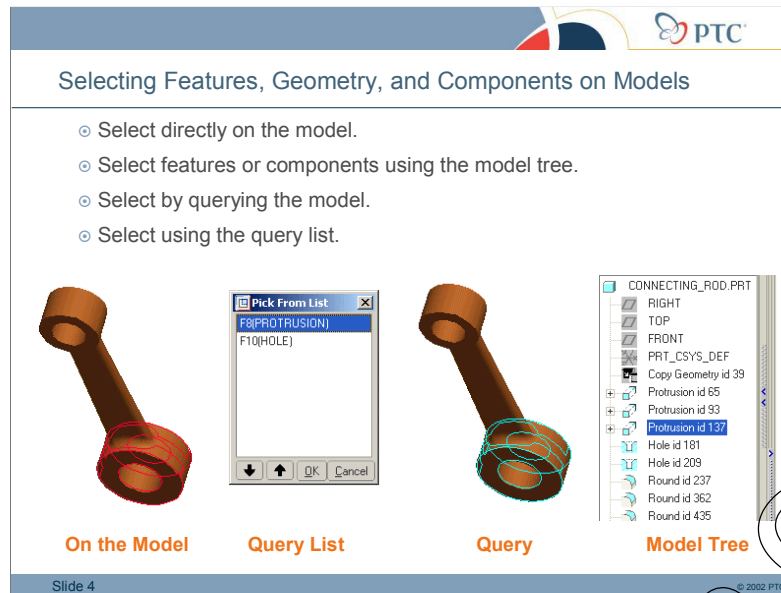
After completing this module, you will be able to:

- Select features, geometry, and components of models.
- Edit design models.
- Describe how editing design models affects parent/child relationships.
- Manage layers when editing design models.

Slide 3

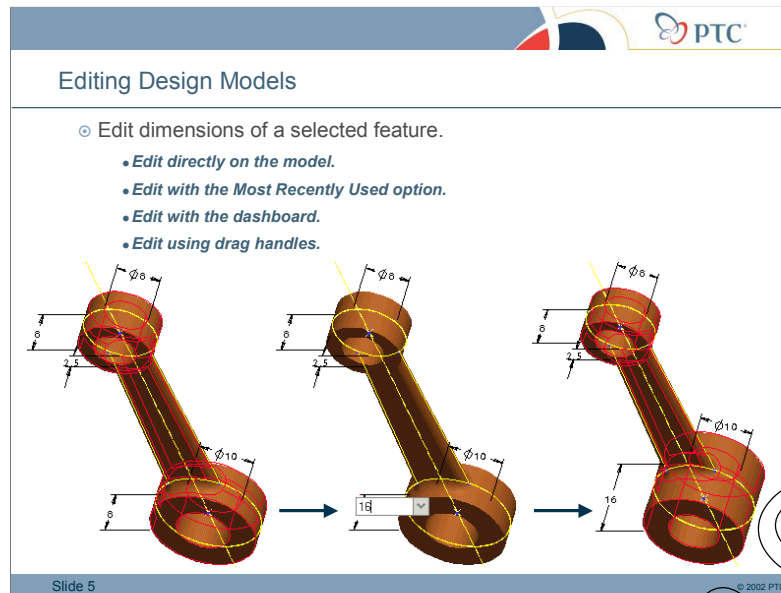
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- You select individual features and geometry throughout the modeling process.
- You select features to identify geometry and components that you want to edit.
- For example, before you insert a hole feature to a part, you select the placement and location references.



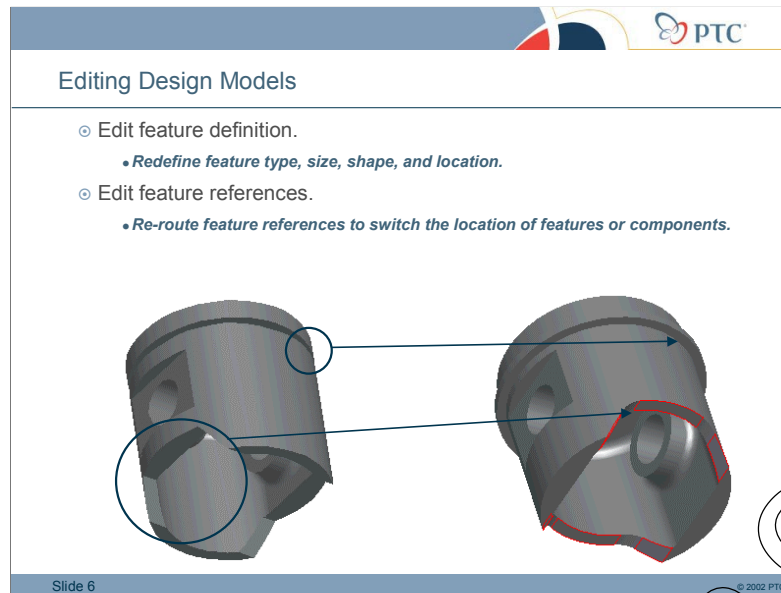
Selection is important because after “selecting” on a feature(s), model(s), or geometry, you will be able to make modifications to the selected item(s).

- When you select a model directly in the Pro/ENGINEER Wildfire workspace, the blue edges designate a pre-selected item.
- When you left-click on an item in a model, it highlights in red indicating that it is selected. Notice that you can query through items using right-click.
- When you select components and features, they highlight with a red wireframe.
- Selected edges highlight in bold red.
- Selected surfaces highlight as a red-shaded item.
- You control what to select using the selection filter. This is useful for selecting features that are visible upfront on the screen.
- To select a model using the query list, you activate the query list, and then select an item in the Pro/ENGINEER Wildfire workspace. Pro/ENGINEER Wildfire lists all the visible and hidden features. Then, you select the desired feature from the list.
- To select features or components using the model tree, you navigate through the list of features or components in the model tree, and select the desired feature. Pro/ENGINEER Wildfire highlights the selected feature or component in the graphic window.



You can edit dimensions once geometry features have been selected. This is important because a majority of new products are based upon designs that have already been completed.

- You change the size and location of a feature by editing the corresponding dimensions in the design model.
- The model must be regenerated every time you edit the dimensions of a feature.
- The dimensions of the selected geometry display in the graphic window.
- When you select a dimension, a drop-down list displays the most recent values of the model. You select a suitable value and regenerate the model.
- When you create or redefine a feature, you can edit the dimensions of the feature in the dashboard, or drag the dimension handles of the model to the desired value



Additionally, you can edit the feature definition or feature references once you have selected geometry features. This is important because by editing the references, you can change the feature's parents. This is also important because by editing the feature's definition, you can drastically alter the feature. For example, you can change a cut that removes material into a protrusion that adds material to a part.

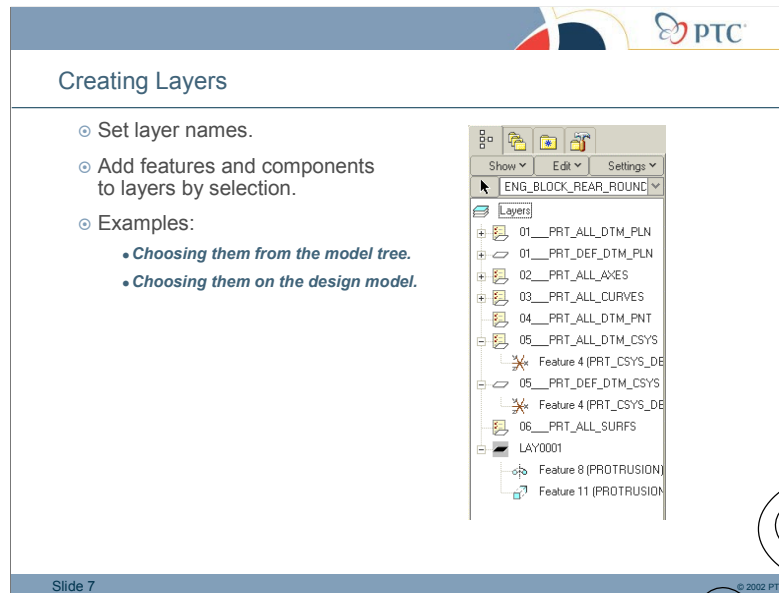
- The figures in the slide illustrate the **Edit Definition** and **Edit References** options.
- The small circle illustrates a cut that has been redefined to create a protrusion on the outside of the piston.
- The large circle illustrates a cut and a round that have been rerouted to different planes.
- You will practice both of these changes in the lab exercises.

Note to instructor:

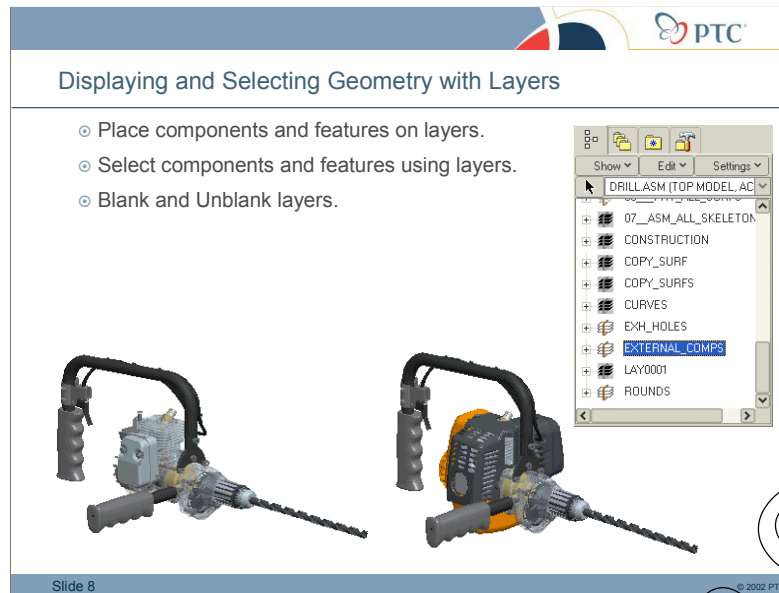
(In the context of this slide, references determine the placement of features or components.

Depending on students understanding, you may have to elaborate on this definition.

However, try to keep it simple as future exercises will expand the students understanding).



- Organizing geometry items using layers enables you to selectively perform operations.
- For example, you can control layer display on the screen, as well as delete and suppress layer items from the model.
- When you use default templates for creating new design models, a set of default layers is automatically created.
- Pro/ENGINEER Wildfire adds geometry items corresponding to each default layer as the geometry items are created.
- For example, all default datum planes are automatically added to the default layer 01_PRT_DEF_DTM_PLN.
- You can select geometry items, include all types of features and components by:
 - Using the model tree.
 - Using the graphics window.



You use layers to select features and components.

- When you select the `EXTERNAL_COMPS` layer, the items on the layer highlight in a green wireframe.
- You use layers to control the display of features and components in the graphics window.
- You can select the features associated to the layers (such as default datum planes and axes) to edit, delete, or blank the features.
- Blanking layers in part mode only controls the display of non-solid geometry. For example if you blank a layer containing the holes of a part, then the axes associated to those holes will be removed from the graphics window.
- When blanking layers in assembly mode, you can also controls the display of parts. For example if you blank a layer containing the parts of an assembly (see figures in slide), then the components are removed from the graphics window.

PTC

Parent/Child Relationships – Editing Design Models

When editing design models, you must consider the parent/child relationships that already exist. Changes made to a parent feature affects its children.

- Edit
 - *Children of feature or component update as edits are regenerated.*
- Edit Definition/Edit References
 - *Enables you to change the parent of the feature or component.*
- Suppress/Resume
 - *Enables you to remove a feature or component and their children from graphics window and regeneration cycle.*
- Delete
 - *Deletes all children of the selected feature or component.*
- Hide/Unhide
 - *Does not affect parent/child relationships.*
- Blank/Unblank (Layers)
 - *Does not affect parent/child relationships.*

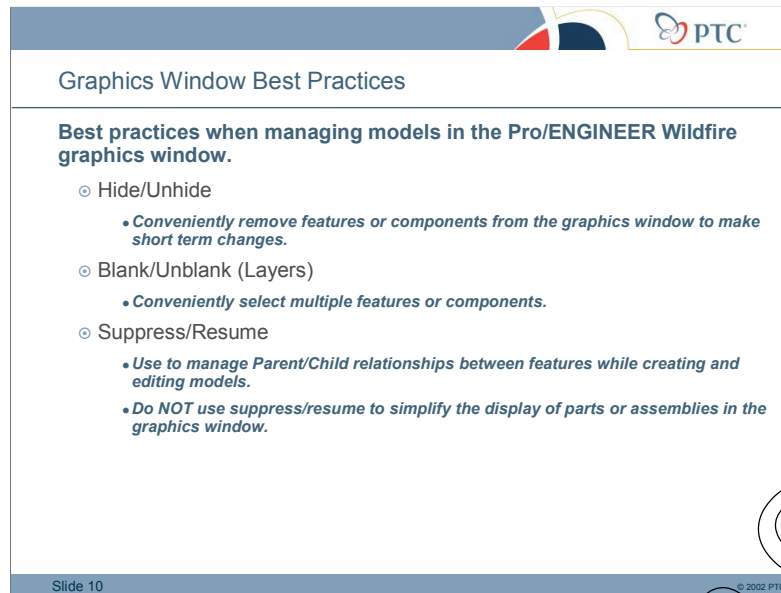
Slide 9 © 2002 PTC

Note to Instructors:

Use this slide to teach students how the functionality they will use in the exercises is affected and how it affects parent/child relationships. It may be easier to talk through the slide after the demo.

Hence as an instructor, you should inform students that they will see the functionality in the demonstration. So:

- 1) *Inform students that we have these best practices for them to follow.*
- 2) *Present demonstration.*
- 3) *Review parent/child relationships after demonstration.*



Note to Instructors:

These best practices are a reference for students. As an instructor, you should inform students that they will see the functionality in the demonstration. So:

- 1) Inform students that we have these best practices for them to follow.
- 2) Present demonstration.
- 3) Review best practices after demonstration.

Note that in some cases, these best practices were not followed when creating this course simply because of the time that would be required to implement these practices. Students should understand this point as it is important!

Best practices when managing models in the Pro/ENGINEER Wildfire graphics window.

- Hide/Unhide
 - Recommend that you use only to temporarily remove features or components in the graphics window in order to make a change. Unhide the features or components once you have completed the change.
- Blank/Unblank (Layers)
 - Recommend that you use layers to remove datum features of parts and assemblies in the graphics window on a long term basis.
 - Recommend that you use layers to select multiple features or components.
- Suppress/Resume
 - Use to manage Parent/Child relationships between features while creating and editing models. For example, you may want to temporarily suppress a series of features to be able to more easily create new features, and then resume the suppressed features. You may also want to temporarily suppress a feature to try out a different feature in the same location. You then could resume or delete accordingly to achieve the desired result.
 - Recommend that you do not use suppress/resume to simplify the display of parts or assemblies in the graphics window because of Parent/Child relationships.

Please note that to remove multiple components from the graphics window, it is recommended to use simplified representations. This will be covered in a later module.



Demonstrations

Modifying Design Models

- ◉ Selecting Components and Features
- ◉ Editing Design Models
- ◉ Creating and Managing Layers

Slide 11

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You will follow along while I demonstrate:

1. Selecting Components and Features
2. Editing Design Models
3. Creating and Managing Layers

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- ◉ Select features, geometry, and components of models.
- ◉ Edit design models.
- ◉ Describe how editing design models affects parent/child relationships.
- ◉ Manage layers when editing design models.

For PTC Internal Use Only

Module 2 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demonstration for customers:

- Practice running the demonstrations, so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003100 or later.
- Download and install the class files fast_track_330.tar.gz as described in the classroom setup notes.

Exercise 1: Selecting Components and Features

Introduction

In this demonstration, we select and review features on a Pro/ENGINEER Wildfire assembly.

This skill enables us to identify items on models that we want to edit. For example, before we can add a hole feature to a part, first we must select the placement surface and identify necessary references.

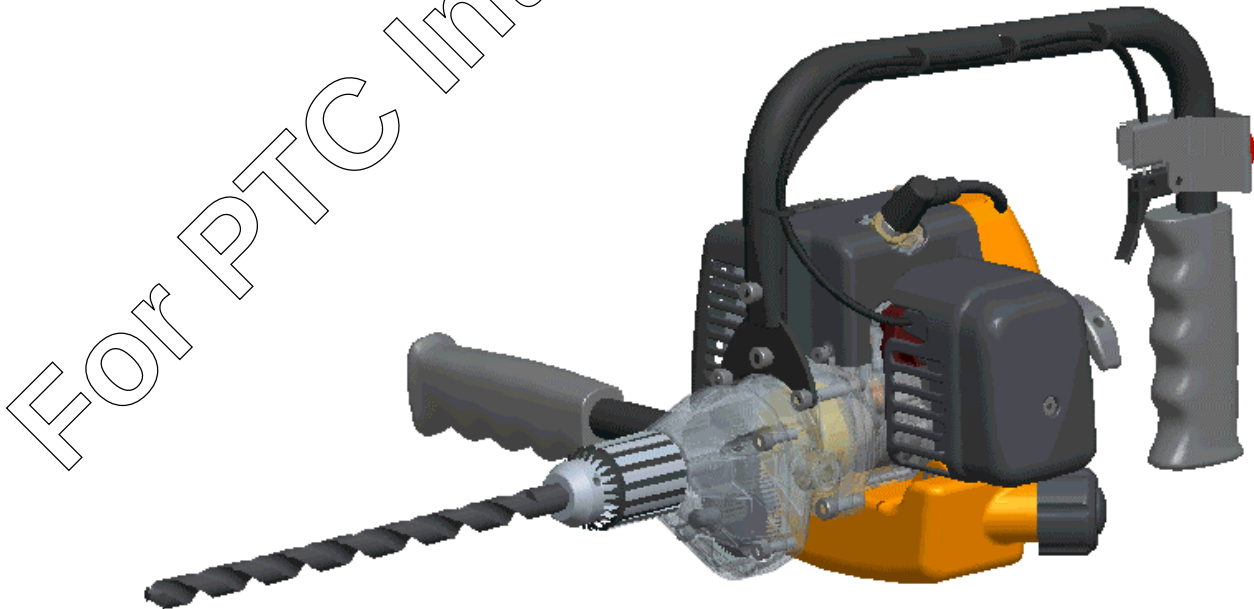


Figure 1: DRILL.ASM

Objectives

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

- Select part features and assembly component geometry.

Scenario

To continue your assessment of the drill assembly, you select the designated components and review their features. At this point, you must determine how the parts were designed, and then compare the existing features against the ECO. This enables you to plan the necessary design changes.

Step 1. Select the drill assembly components.

To continue assessing the drill assembly, we must select the designated components and review their features.

At this point, we want to determine how the parts were designed and compare the existing features against the ECO. This will help us plan the necessary design changes.

First, we locate the drill assembly with the navigator, and then we open the assembly and view its components in the model tree.

Note that the components from the ECOs are the connecting rod and flywheel parts.

1. If necessary, start Pro/ENGINEER Wildfire.
2. If necessary, navigate to **c:\users\student\fast_track_330\module_01-02**, right-click and select **Make Working Directory**.
3. Open DRILL.ASM.
4. Using the model tree, expand ENGINE.ASM and CRANK.ASM.
5. Press and hold CTRL, and select CONNECTING_ROD.PRT and FLYWHEEL.PRT.

Note:

If you have properly selected a component, you should be able to right-click in the graphics window to display list of actions that you can perform on the component.

6. Click in the graphics window to de-select all components.
7. Cursor over CONNECTING_ROD.PRT, right-click to query, and then select it.
8. Cursor over FLYWHEEL.PRT, press CTRL + right-click to query, and then select it.

Note:

If you are having trouble locating components, use the model tree to locate the proximity of a component. Then, select the model directly in the graphics window.

9. Click in the graphics window to de-select all components.
10. Cursor over CONNECTING_ROD.PRT, right-click and select **Pick From List**. Highlight CONNECTING_ROD.PRT and select it.

Note:

Press and hold the right mouse button briefly to activate Pick From List.

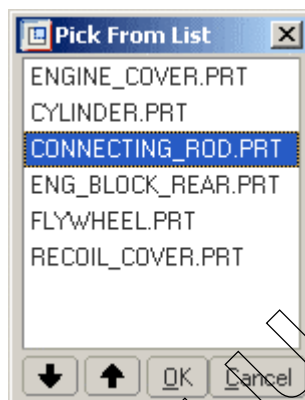


Figure 2: Select CONNECTING_ROD.PRT Using the Query List

11. Cursor over FLYWHEEL.PRT, press CTRL + right-click and select **Pick From List**. Highlight FLYWHEEL.PRT and select it.
12. To de-select all components, click **Edit > Select > Deselect All**.

Note:

Repainting the screen does not de-select items.

Step 2. Hide and unhide ENGINE_COVER.PRT.

Hiding components allows us to remove components from the display window. The benefit is that we can easily view other components contained within the hidden component. By unhiding the component, we once again display it in the Pro/ENGINEER Wildfire modeling workspace.

There are multiple methods to hide and unhide components. We can select on the components in the graphics window and right-click. We can also select on the components in the model tree and right-click. Once we select the components, we can use the View menu to hide and unhide them.

1. Select ENGINE_COVER.PRT, right-click and select **Hide**, as shown in the following figure.



Figure 3: Hidden ENGINE_COVER.PRT

2. Select ENGINE_COVER.PRT again, right-click and select **Unhide**, as shown in the following figure.



Figure 4: Unhidden ENGINE_COVER.PRT

Note:

You can Hide/Unhide using the model tree, menus, and the right mouse button.

3. De-select all components.

Step 3. Open GEARBOX_REAR.PRT and review its features.

We can select the assembly components or specific component features using the model tree, graphics window, or the query list. These options provide great flexibility.

1. Using the graphics window, select GEARBOX_REAR.PRT, right-click and select **Open**.
2. Click **Tools > Model Player**. Go to the beginning of the model. Select the **Display Each Feature** check box and go to the last feature in the model. Notice how the features build upon each other to create the finished model. Close the model player.
3. Using the model tree, select and review one protrusion feature, one round feature, one hole feature, and one group of GEARBOX_REAR.PRT.
4. Using the graphics window, de-select the features.
5. Press and hold CTRL, and select all four of the previous features at once in the model tree.
6. De-select the features.

Note:

When reviewing features, look at any data about the feature by expanding the feature in the model tree or by reviewing the feature's information.

7. Using the graphics window, cursor over the hole feature, and then select the hole feature, as shown in the following figure

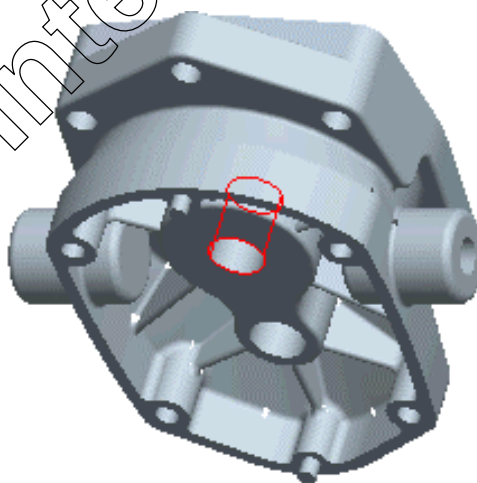


Figure 5: Directly Select the Hole in the Graphics Window

Note:

Notice that the feature pre-highlights in cyan wireframe before you select it. The selected feature highlights in a red wireframe.

8. De-select the feature.

9. Using the graphics window, select the round feature, as shown in the following figure.

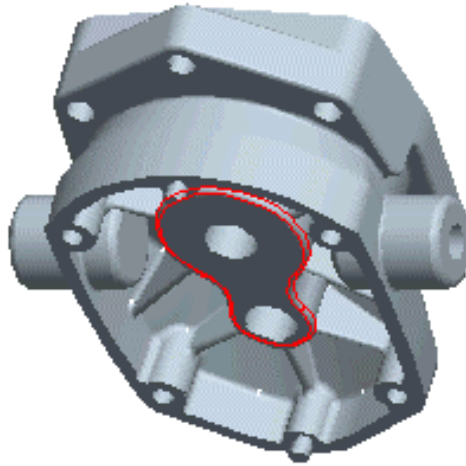


Figure 6: Directly Select a Round Feature

10. De-select the feature.
11. Cursor over the model, right-click to query, and then select the protrusion, as shown in the following figure.

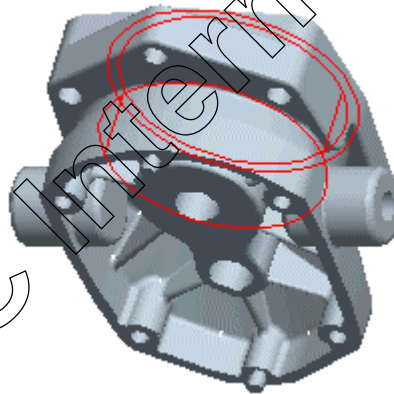


Figure 7: Select a Protrusion in the Graphics Window

12. De-select the feature.
13. Using the query list, select and review one protrusion feature, one round feature, and one hole feature of GEARBOX_REAR.PRT. Notice that the number of features found in the list will depend on where you selected on the model.
14. De-select all features.
15. Close the GEARBOX_REAR.PRT window to return to the DRILL.ASM window, and then proceed to the next exercise.

This completes the exercise.

Exercise 2: Editing Design Models

Introduction

In this demonstration, we select specific components and features to edit. Most engineering work involves changing a design that has already been built. The benefit is that we save time, since we do not have to start a new 3-D design model.

Objectives

- Edit the dimensions of design models.

Scenario

You have reviewed the ECO and confirmed the problems in the actual design model. Now you update the affected components specified in the ECO.

Step 1. Edit the length of the STANDARD_BIT.PRT (in DRILL.ASM).

Now that we know how to select individual parts and features, we can quickly edit the parts specified in the ECO. Initially we change the drill bit part. Once we complete the changes, we regenerate and save the part before closing the Pro/ENGINEER Wildfire window and moving on to the next part.

1. Using the graphics window, select STANDARD_BIT.PRT, right-click and select **Open**.
2. Using the graphics window, double-click the MAIN protrusion to display the dimensions, as shown in the following figure.

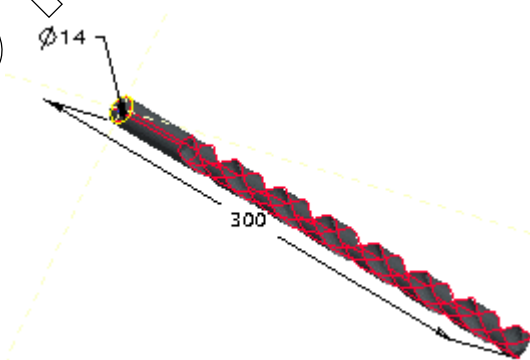


Figure 8: STANDARD_BIT.PRT Dimensions

3. Double-click the 300 dimension and enter **275**.
4. Click **Edit > Regenerate** to update the model.
5. Select the MAIN protrusion, right-click, and select **Edit Definition**.

6. In accordance with the ECO specifications, drag the depth handle to **250**.
7. Use the dashboard to complete the feature, and regenerate the model using the toolbar.
8. Click **File > Save** to save the model.
9. Close the STANDARD_BIT.PRT window to return to the DRILL.ASM window.

Step 2. Open the ENGINE.ASM into a new graphics window.

To simplify the model display in the graphics window, we can also open a new graphics window with a smaller set of component data.

To open the engine sub-assembly into a new graphics window, we right-click in the model tree.

This right-click option saves time by eliminating searching for models in a separate Open dialog box.

1. Using the model tree, select ENGINE.ASM, right-click and select **Open**.
2. Using the model tree, review the different subassemblies and parts contained within ENGINE.ASM.

Step 3. Edit the number of cooling fins on FLYWHEEL.PRT (in ENGINE.ASM).

We use the model tree to display the dimensions of the flywheel part so that we can edit the number of cooling fins specified in the ECO.

This is another example of how Pro/ENGINEER Wildfire's functionality can enhance our productivity. Notice that we do NOT have to open the flywheel into a separate window to edit the number of fins.

1. Select FLYWHEEL.PRT, right-click and select **Activate**.
2. Query and select one of the cooling fin protrusions, and then right-click and select **Edit**.
3. Double-click the value for the quantity of cooling fins, and edit from 16 to **12**.
4. Use the toolbar, regenerate the model. The model appears as shown in the following figure.

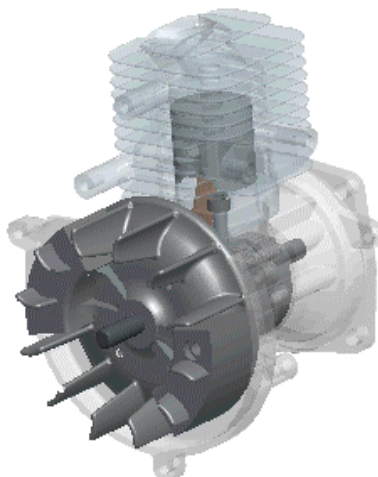


Figure 9: FLYWHEEL.PRT with 12 Cooling Fins

Step 4. Edit the CONNECTING_ROD.PRT dimensions.

The ECO recommends editing the connecting rod. We edit the model by selecting it directly in the graphics window. We must hide a few parts so that we can see the connecting rod. Therefore, to hide the cylinder and block parts, we use the right-click method, and then we also experiment using the toolbar menu. Again, we regenerate and save the parts once they are completed.

1. Using the model tree, select ENGINE.ASM, right-click and select **Activate**.
2. Using the graphics window, press and hold CTRL, and select CYLINDER.PRT, ENG_BLOCK_REAR.PRT, and ENG_BLOCK_FRONT.PRT
3. Right-click and select **Hide**. Zoom to view the CONNECTING_ROD.PRT.
4. Select CONNECTING_ROD.PRT, right-click and select **Activate**.
5. Cursor over the center of CONNECTING_ROD.PRT, and right-click to query. Select the underlying protrusion (**F6**), right-click and select **Edit**.
6. Edit the lower width of CONNECTING_ROD.PRT from 10 to **12**, and edit the upper width from 8 to **10**, as shown in the following figure.

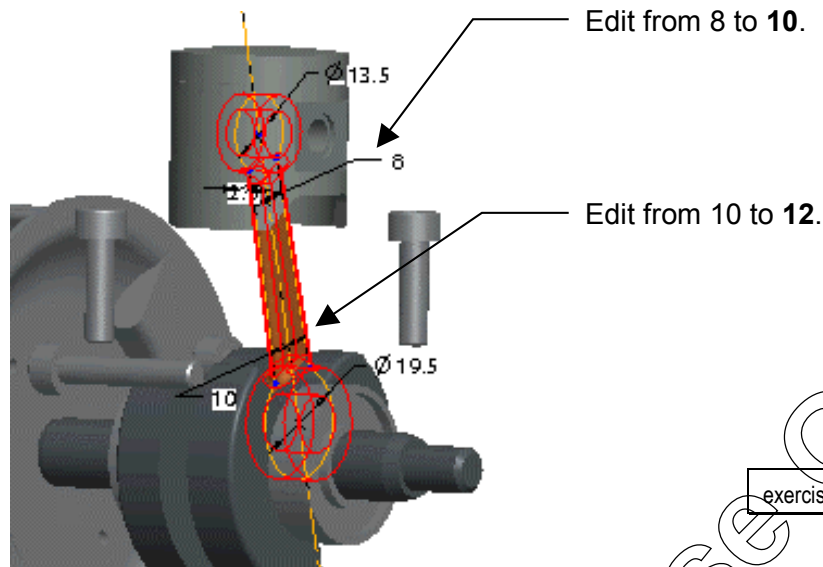


Figure 10: Edit CONNECTING_ROD.PRT

7. Regenerate the model.
8. Using the model tree, press and hold CTRL, and select CYLINDER.PRT, ENG_BLOCK_REAR.PRT, and ENG_BLOCK_FRONT.PRT
9. Right-click and select **Unhide**.
10. Save the model.

Note:

Saving the assembly saves the components in the assembly.

Step 5. Redefine and suppress/resume the PISTON.PRT features.

In this step, we familiarize ourselves with the functionality commonly used when editing 3-D designs. This functionality is important because most engineering work begins with a model that is already designed to save time and development costs.

1. Select and open PISTON.PRT into a separate window.
2. Click **Edit > Resume > Last** to resume the last suppressed feature.

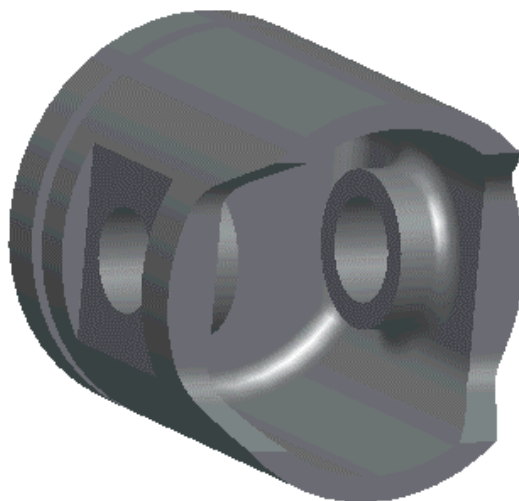


Figure 11: PISTON.PRT

Note:

There is only one suppressed round.

3. Using the model tree, select **Cut id 1204**, right-click and select **Suppress**. Allow **Round id 1688** to also be suppressed due to the parent-child relationship.

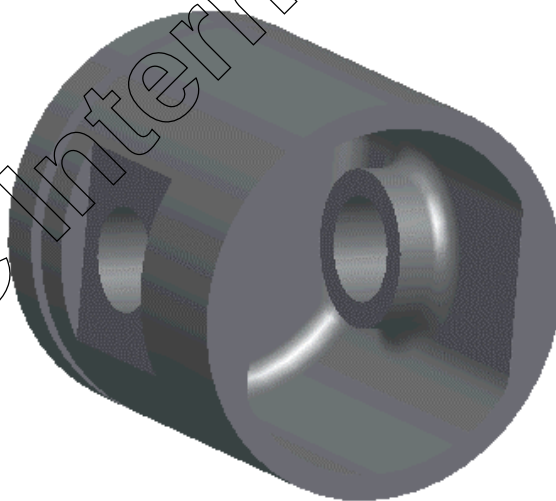


Figure 12: PISTON.PRT with Cut and Round Suppressed

4. To view the suppressed features in the model tree, click **Settings > Tree Filters**. Select the **Suppressed Objects** check box and continue.
5. Using the model tree, select the two suppressed features, right-click and select **Resume**.
6. Using the model tree, select **Cut id 1963**, right-click and select **Edit Definition**.
7. Using the dashboard, disable the **Remove Material** option.

8. Using the dashboard, redefine the existing section, and enter the sketch.
9. Double-click to edit the 1.5 dimension to **-1.5**, as shown in the following figure.

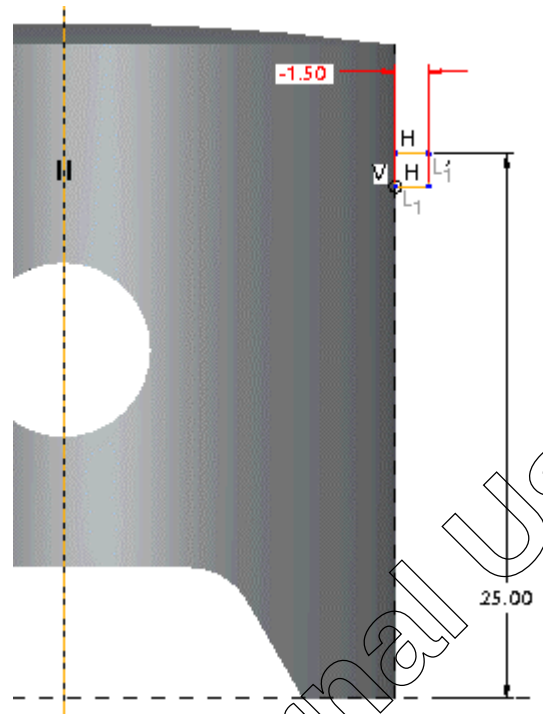


Figure 13: Alter the Cut to Create a Protrusion

10. Complete the sketch and complete the feature, as shown in the following figure.

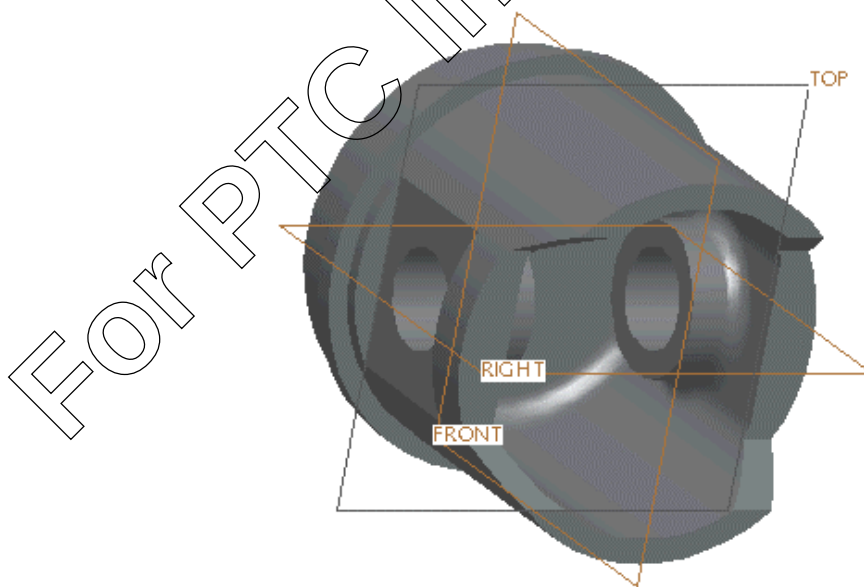


Figure 14: The Completed Protrusion

11. Using the model tree, select **Cut id 1204**, right-click and select **Edit References**. When prompted to roll back the model, click **No**.
12. To specify an alternate reference for the sketching plane, select datum **RIGHT** in the model tree.
13. Click **Same Ref** for the remaining three references, and accept the default direction. Refer to the following figure.

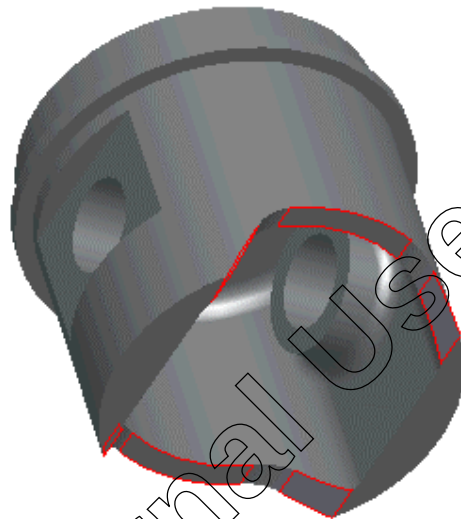


Figure 15: Re-routed the Cut Reference

14. Save the model.
15. Activate the DRILL.ASM window by clicking **Window > DRILL.ASM**.
16. Click **File > Close Window** to close the DRILL.ASM window.
17. Click **File > Erase > Not Displayed** to erase models from session. Note that only files not contained in the graphics window are removed from RAM memory.
18. Close all windows.
19. Erase all models from session.

This completes the exercise.

Exercise 3: Creating and Managing Layers

Introduction

In this demonstration, we create layers in order to simplify the displayed information.

Layers allow us to quickly remove information from the graphics window, such as features, components, and references.

For example, we can blank all datum points used to create the trajectory of a curve, while displaying all datum points that define connections between movable parts. Layers are especially useful when working with complicated designs because they simplify the model's appearance.

Objectives

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

- Create layers containing components.

Scenario

You create layers at the assembly levels in order to organize components and control their display in the graphics window. This is important because you can easily select all components on the layer instead of selecting them individually in the model tree or graphics window.

Step 1. Create a layer containing exterior parts of DRILL.ASM.

In this exercise, we initially create a layer containing some of the exterior components of the drill assembly. A coworker could easily use this layer to blank components or to select multiple components. Note: we will later learn a different method to remove components from the graphics window that also minimizes information stored in RAM memory.

1. Open DRILL.ASM.
2. From the model tree, click **Show > Layer Tree**. Right-click and select **New Layer**. Enter **EXTERNAL_COMPS** as the name.
3. Click **Show > Model Tree**. Select COIL.PRT, ENGINE_COVER.PRT, and their corresponding bolts. Then select RECOIL.ASM, FUEL_TANK.ASM, and CARBURETOR.ASM. Complete the layer.
4. Using the model tree, click **Show > Layer Tree**. Select the EXTERNAL_COMPS layer, right-click and select **Blank Layer**. Repaint the screen.

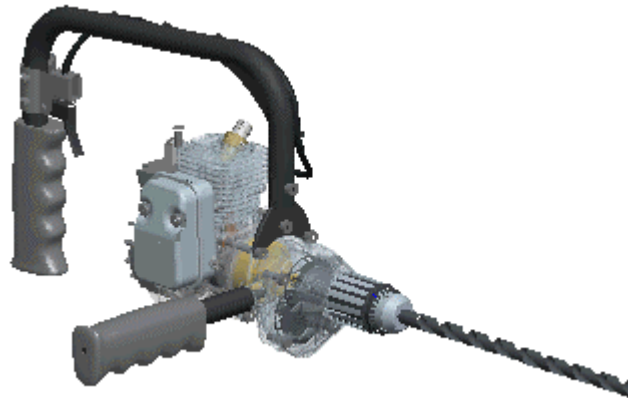


Figure 16: Components Blanked

5. Select the EXTERNAL_COMPS layer, right-click and select **Unblank Layer**. Repaint the screen again.



Figure 17: Components Unblanked

Note:

Unlike hide, layer status is not saved by default when the model is saved.

6. Select the EXTERNAL_COMPS layer, right-click, and select **Select Items**. Cursor over the graphics window, right-click and select **Suppress > OK**.
7. Resume all features.
8. Save the model
9. Close all windows.
10. Erase all models from memory.

Summary

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

- Select features, geometry, and components of models.
- Edit design models.
- Describe how editing design models affects parent/child relationships.
- Manage layers when editing design models.

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Creating Reference Geometry

Introduction

Reference geometry enables you to easily locate base model features in a three-dimensional space. To begin the design of a new model, you create the reference geometry known as datum planes.

Datum planes are used throughout the modeling process, when assembling components or creating features. Additional reference geometry, such as datum axes, datum points, datum coordinate systems, and datum curves offer alternate methods of placing or creating features and components.

Layers enable you to simplify geometry selection by temporarily hiding or displaying specific model features or assembly components in the graphics window.

Objectives

After completing this module, you will be able to:

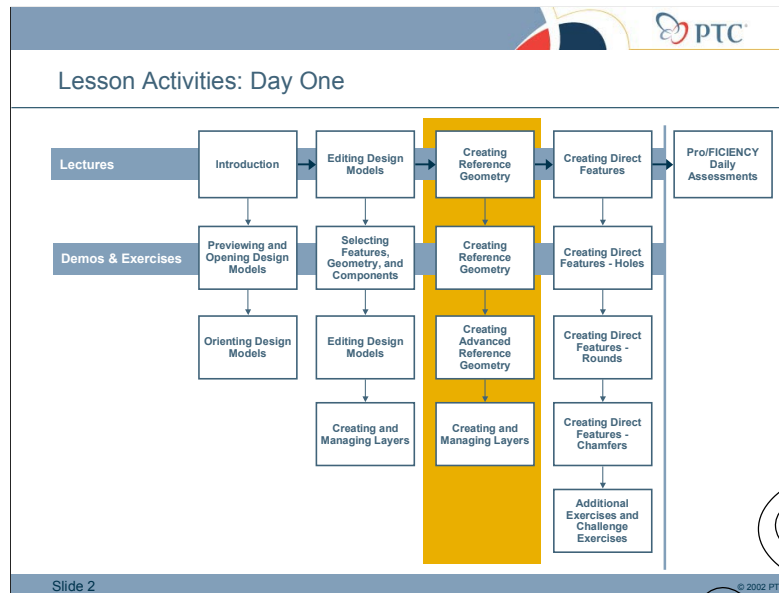
- Create reference geometry.
- Create features using reference geometry.
- Describe how creating reference geometry affects parent/child relationships.
- Manage layers when creating reference geometry.

Fast Track to Pro/ENGINEER Wildfire

Module 3: Creating Reference Geometry



For PTC Internal Use Only



Duration

- Lecture: 10 mins
- Demos (3): 30 mins
- Labs (3): 50 mins
- Total: 1hr 30min



Objectives

After completing this module, you will be able to:

- Create reference geometry.
- Create features using reference geometry.
- Describe how creating reference geometry affects parent/child relationships.
- Manage layers when creating reference geometry.

Slide 3

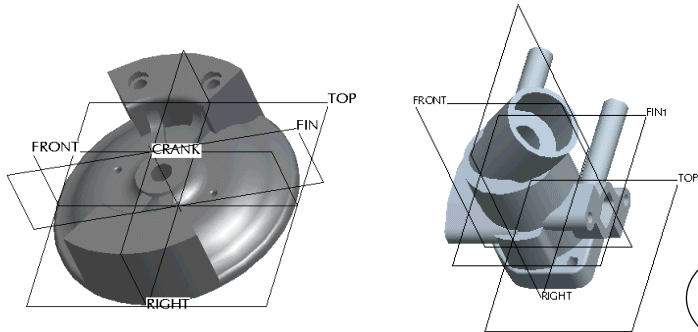
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- Before creating models, you locate the base model features in three-dimensional space (3-D).
- Reference geometry allows you to easily locate the features in the base models.
- The first step in designing a new model is to create reference geometry known as datum planes.
- Datum planes are used throughout the modeling process.
- Base model features are placed relative to these datum planes.
- Datum planes are also used at other times in the modeling process, such as assembling components.
- Other references, such as datum axes, datum points, datum coordinate systems, and datum curves, provide alternate methods of placing or creating features and components.

PTC

Creating Reference Geometry

- Creating Datum Planes
 - Use templates to automatically create default datum planes.
 - Set placement references and geometric constraints.
 - Use to sketch, orient, and place references for features and components.



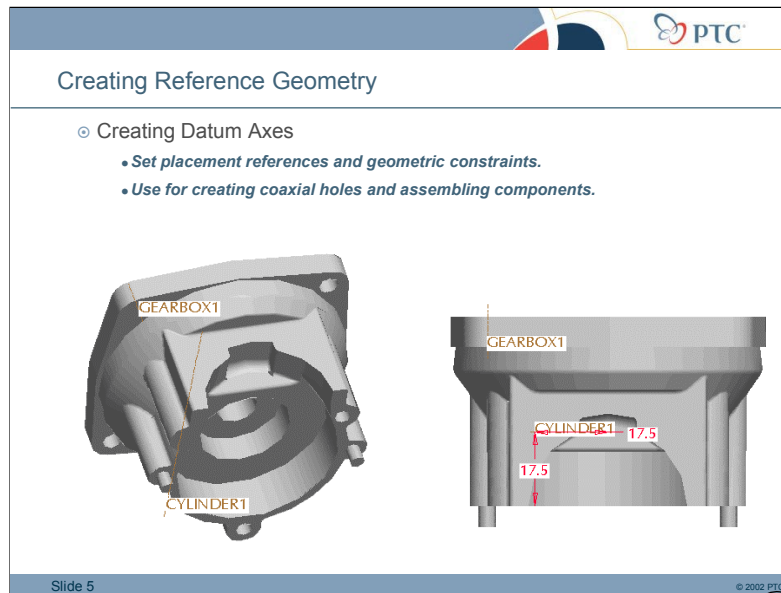
Slide 4

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Note to Instructors:

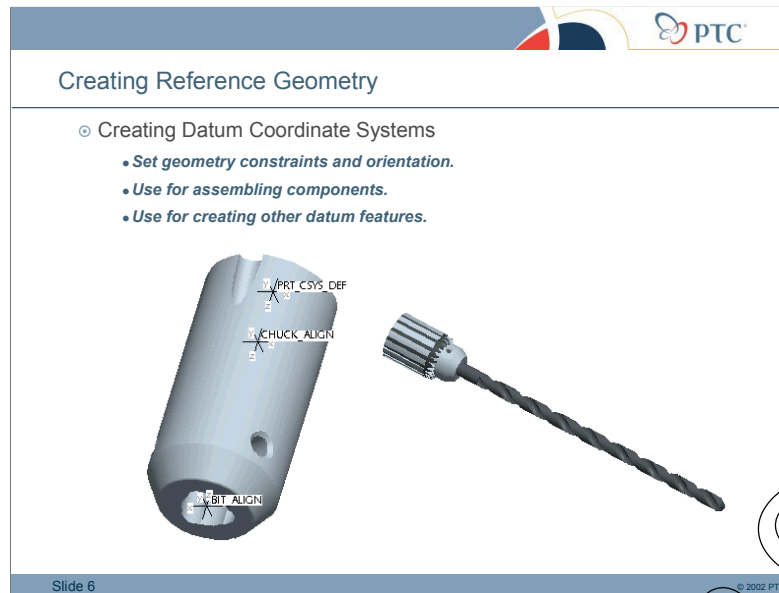
Use the figures to describe how the new feature will be placed in reference to the FIN and FIN1 datum planes. It is also important to note that these datum planes will be used to capture design intent. However, tell students that the design intent portions will be seen in a later module. Yet it is important to think about design intent while they are building models.

- This slide illustrates how default datum planes are used as references to create models.
- This slide also illustrates setting geometric constraints, for example, offsetting a plane from a surface; or constraining thru an axis and at an angle to a plane.
- At least one geometric constraint has to be set when you create datum planes. For example, the offset plane uses just a distance.

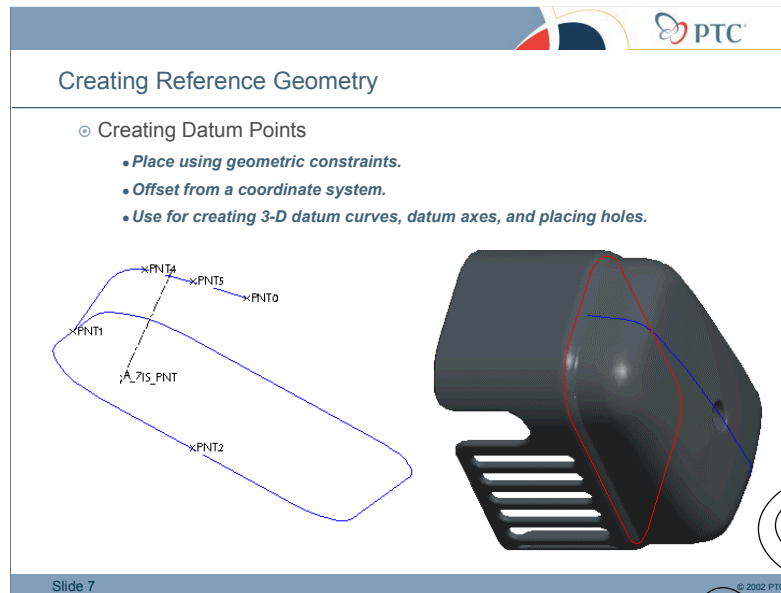


Datum axes also capture design intent in the form of hole locations and mounting information when parts are assembled.

- The figures in this slide illustrate creating datum axes using cylindrical and linear references.
- Datum axes serve as good reference geometry when creating coaxial holes and assembling components using constraints.

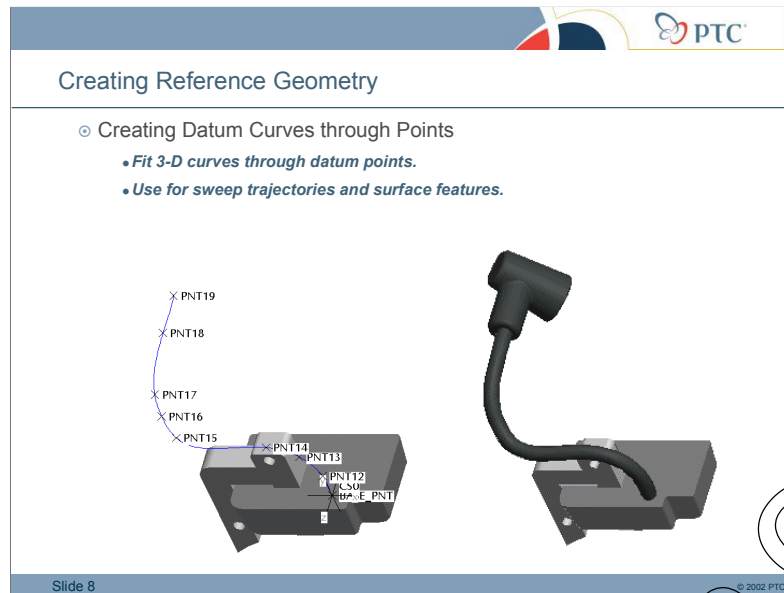


- To create a coordinate system, you set geometric constraints and also orient the X, Y and Z axes of the coordinate system. *Cartesian is most commonly used.*
- The figures in this slide illustrate orienting the coordinate system using datum planes, datum axes and solid surfaces.
- Datum coordinate systems provide helpful references when you are:
 - Assembling components using constraints.
 - Creating datum features, such as datum points and datum curves.



- The figures in this slide illustrate a model with a set of datum points created using different methods dependent on the references selected, for example:
 - Length-ratio on a curve.
 - Using actual length of a curve.
 - Offsetting from a datum plane.
 - On a Vertex.
 - At the center of an arc.
 - At the intersection of a Plane and a curve.
- Datum points can also be created by defining X, Y and Z offset values from a coordinate system. This creates a set of points that can be referenced to create 3-D curves for sweep trajectories.

Note to the Instructor: You will discuss creating sketcher points in Module 5- Sketcher.



- The slide illustrates a 3-D curve through a series of datum points on the block.
- You can optionally set the curve to be tangent or curvature continuous at the start and end points.
- 3-D curves serve as trajectories when creating variable section sweeps and surface features.

Note to Instructor:

- You will explain how to create sketched datum curves in in Module 5- Sketcher.



Parent/Child Relationships – Creating Reference Geometry

When creating reference geometry, parent/child relationships are created. Changes made to a parent feature affects its children.

- Datum Planes
- Datum Axes
- Datum Coordinate Systems
- Datum Points
- Datum Curves through Points

Slide 10

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Use this slide to teach students how the functionality they will use in the exercises is affected and how it affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

For example:

Datum Planes are children to their references, such as an axis and another datum plane.

Datum axes can be children to cylindrical surfaces, datum planes, flat surfaces, and datum points.



Graphics Window Best Practices

Best practices when managing models in the Pro/ENGINEER Wildfire graphics window.

- ◉ Hide/Unhide
 - Conveniently remove features or components from the graphics window to make short term changes.
- ◉ Blank/Unblank (Layers)
 - **Conveniently remove datum features of parts and assemblies from the graphics window on a long term basis.**
 - Conveniently select multiple features or components.
- ◉ Suppress/Resume
 - Use to manage Parent/Child relationships between features while creating and editing models.
 - Do NOT use suppress/resume to simplify the display of parts or assemblies in the graphics window.

Slide 11 © 2002 PTC

Note to Instructors:

Use this slide to review best practices with students. Only the red line is different than the slide in module 2, and this is based off of what they will see in the demonstration and exercise in this module. While this slide is repeated from the previous module, it is important to note the first bullet under Blank/Unblank (Layers) is new.

These best practices are a reference for students. Note that in some cases, these best practices were not followed when creating this course simply because of the time that would be required to implement these practices. Students should understand this point as it is important!

Best practices when managing models in the Pro/ENGINEER Wildfire graphics window.

- Hide/Unhide
 - Recommend that you use only to temporarily remove features or components in the graphics window in order to make a change. Unhide the features or components once you have completed the change.
- Blank/Unblank (Layers)
 - Recommend that you use layers to remove datum features of parts and assemblies in the graphics window on a long term basis.
 - Recommend that you use layers to select multiple features or components.
- Suppress/Resume
 - Use to manage Parent/Child relationships between features while creating and editing models. For example, you may want to temporarily suppress a series of features to be able to more easily create new features, and then resume the suppressed features. You may also want to temporarily suppress a feature to try out a different feature in the same location. You then could resume or delete accordingly to achieve the desired result.
 - Recommend that you do not use suppress/resume to simplify the display of parts or assemblies in the graphics window because of Parent/Child relationships.

Please note that to remove multiple components from the graphics window, it is recommended to use simplified representations. This will be covered in a later module.

Using PTC Help

Referencing Pro/ENGINEER Wildfire Help Center to access:

- Functional Areas
 - Contains Pro/ENGINEER Wildfire reference guides that are tailored to specific module purposes.
 - These reference guides include:
 - Functionality theory
 - Functionality uses
- Quick Links
 - Contains links to additional Pro/ENGINEER Wildfire reference information and tutorials.

Slide 12

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So if you wanted to get more information on creating datum planes or layers, you can PTC Help to search for and read reference information.

Note to Instructors:

It is critical that students understand that the PTC Training guides will no longer contain detailed theory about Pro/ENGINEER Wildfire functionality. The training guides serve as a high-level outline that facilitates the progress of the course by introducing functionality topics. The theory behind these topics is then explained during the instructor demonstration.

If students want to read more about Pro/ENGINEER Wildfire functionality, they can use the Pro/ENGINEER Wildfire Help Center to search and find the desired functionality. Note that all Pro/ENGINEER Wildfire Reference guides are contained within the Help Center. These guides contain more information than all previous training guides, including:

- *Functionality theory – Learn what the desired functionality accomplishes and how you can benefit by using it.*
- *Functionality uses – Learn how to use desired functionality within Pro/ENGINEER Wildfire.*

It is important that students learn how to use these reference guides because this information is available to them at work by loading Pro/HELP from the standard Pro/ENGINEER Wildfire installation disks.

The picture in this slide displays the Help Center home page.

The Functional Areas contain the Pro/ENGINEER Wildfire Reference Guides. By choosing the Site Map, you will see that these guides have been divided up further according to functionality. Again, these guides include both functionality theory and uses.

The Quick Links provide additional reference information and tutorials. Recommended reading includes the What's New section, which if you open the Web version documents several functionality areas such as Snapping Drag Handles, Working with Rounds, etc...

You should also be familiar with Global Search capabilities and the Quick Reference Card.



Demonstrations

- ◉ Creating Reference Geometry
- ◉ Creating Advanced Reference Geometry
- ◉ Creating and Managing Layers

Slide 13

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You will follow along while I demonstrate:

1. Creating Reference Geometry
2. Creating Advanced Reference Geometry
3. Creating and Managing Layers

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- ◉ Create reference geometry.
- ◉ Create features using reference geometry.
- ◉ Describe how creating reference geometry affects parent/child relationships.
- ◉ Manage layers when creating reference geometry.

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Module 3 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demonstration for customers:

- Practice running the demonstration so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Accessing Pro/ENGINEER Wildfire Help Center

Objectives

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

- Locate reference information about specific Pro/ENGINEER Wildfire functionality.

Scenario

Use Pro/ENGINEER Wildfire Help Center to reinforce your understanding of key concepts covered in this module by performing a Global Search for information.

Step 1. Locate reference information about specific Pro/ENGINEER Wildfire functionality.

In this demonstration, we use the main Pro/ENGINEER Wildfire Help Center to ensure our understanding of specific Pro/ENGINEER Wildfire functionality. By understanding the purpose of specific functionality, we can better determine when to use the functionality when designing our products.

1. Start Pro/ENGINEER Wildfire.
2. Access the Help Center by clicking **Help > Help Center**.
3. Begin a search on Datum Planes by clicking **Global Search**.
4. Enter **Datum Planes** in the Search For text box. Move **Part Modeling** to Selected Functional Areas by selecting it and clicking **>**. Click **Search**.

Note:

By clicking Site Map, you can determine the Pro/ENGINEER Wildfire modules contained within each Functional Area.

5. In the Search Results area, click on **About Datum Planes**. Reference information about datum planes is now displayed in the Help Center.

Note:

You may have to scroll the Search Results area to find this selection.

Step 2. Finding related reference information by using the Help Center interface.

1. Click **Show Navigation** in the Help Center Interface.
2. Click >> to open the Contents to display the next topic.

Note:

You now have opened a display that shows you exactly where your topic is located within the Pro/ENGINEER Wildfire Functional Area. Additionally, you see the related topics such as About Datum Plane UI, To Create a Datum Plane, etc...

3. Scroll to the end of the Help Center page. Note that there are additional links to related information.

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Axes	Part Modeling	About Datum Axes
Coordinate System	Part Modeling	About Coordinate Systems
Points	Part Modeling	About the Datum Point Feature
Curves Through Points	Part Modeling	To Create a Datum Curve Through Points
Layers	Fundamentals	About Layer Display

Exercise 2: Creating Reference Geometry

Introduction

In this demonstration, we create basic reference geometry in the form of datum planes and datum axes. In Pro/ENGINEER Wildfire, datum planes and axes are used as building blocks when constructing 3-D geometry. For example, before building a part, we start with three datum planes that reference where the part is located in “space.” This routine skill is used throughout the modeling process.

Objectives

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

- Create angle and offset datum planes.
- Create datum axes through cylinders and on surfaces.

Scenario

You have completed making the changes ordered in the ECO. Now, you have been assigned to create reference features on existing components.

Step 1. To design the blades on FLYWHEEL.PRT, create a datum plane at an angle to the FRONT datum plane.

To quickly create features, we must be able to locate the features in 3-D space. Reference geometry allows us to locate the features.

Before we begin building features, first we create the datum planes. We use the datum planes to sketch the profile of a cooling fin. We will not sketch the cooling fin in this step.

However, it is important that we create the cooling fin so that we can pattern the cooling fin later. To accomplish this, we create a datum plane at an angle to the FRONT datum plane.

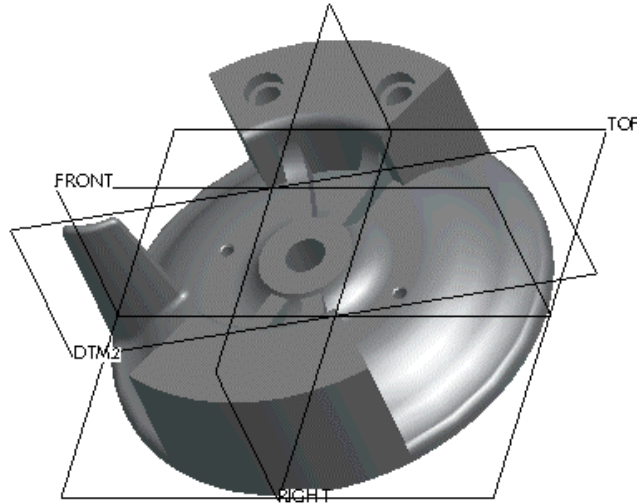


Figure 1: FLYWHEEL.PRT with Completed Cooling Fin and Datum Plane

1. If Pro/ENGINEER Wildfire is open, close all the windows and erase all the models from session. Otherwise, start Pro/ENGINEER Wildfire.
2. Select the folder navigator, and browse to the following folder:
C:\users\student\fast_track_330\module_03.
3. Right-click and select **Make Working Directory**.
4. Using the browser, select and open FLYWHEEL.PRT. Display all datum features.
5. Select the FRONT plane. Press and hold CTRL and select the RIGHT plane.
6. Start the Datum Axis tool. Notice an axis has been created.
7. With the axis still selected, right-click and select **Rename**. Enter **CRANK** as the name.
8. With the axis still selected, start the Datum Plane tool. Press and hold CTRL, and select the FRONT plane. Drag the angle handle and enter **11.25** as the angle value.

Note:

Negative values are used to flip the direction of dimensions.

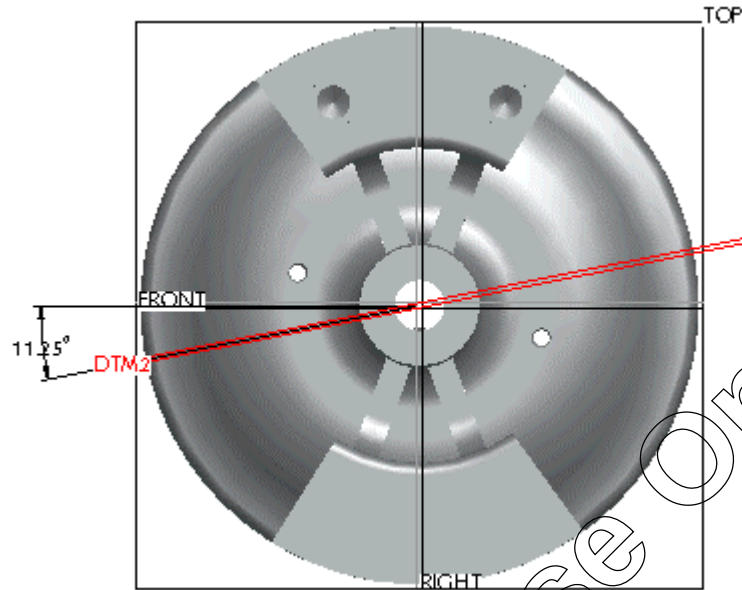


Figure 2: Creating the Datum Plane for the Cooling Fin

9. Complete the feature. This plane will be used in the construction of a cooling fin.
10. With the plane still selected, rename it to **FIN**.

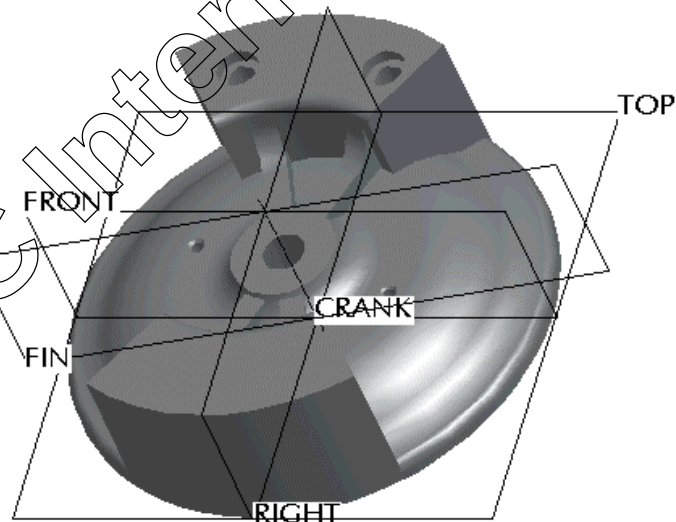


Figure 3: The Completed the Datum Plane

11. Save the model.
12. Close the window.

Step 2. Create a datum plane at an offset distance to design the fins on CYLINDER.PRT.

In the second step, we create an offset datum plane. Offset datum planes are created at a set distance parallel to another datum plane or a two-dimensional surface. Offset planes provide another option for creating datum planes.

Again, we use the existing datum plane to sketch the profile of a cooling fin. We will not sketch the cooling fin in this step. However, it is important that we create the cooling fin referencing this plane, so that we can pattern the cooling fin later. .

1. Open CYLINDER.PRT.
2. Start the Datum Plane tool. Select the CYL_INTF plane. Drag the offset handle and enter **15**, as shown in the following figure.

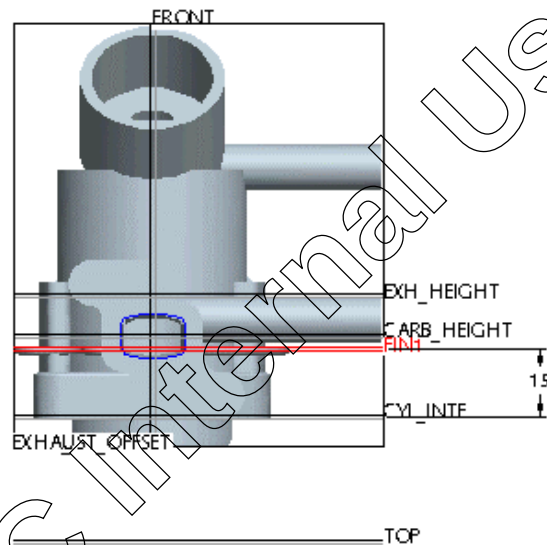


Figure 4: Offset the FIN1 Datum Plane

3. Select the Properties tab and enter **FIN1** as the name. Complete the feature, as shown in the following figure. This plane will be used in the construction of a cooling fin.

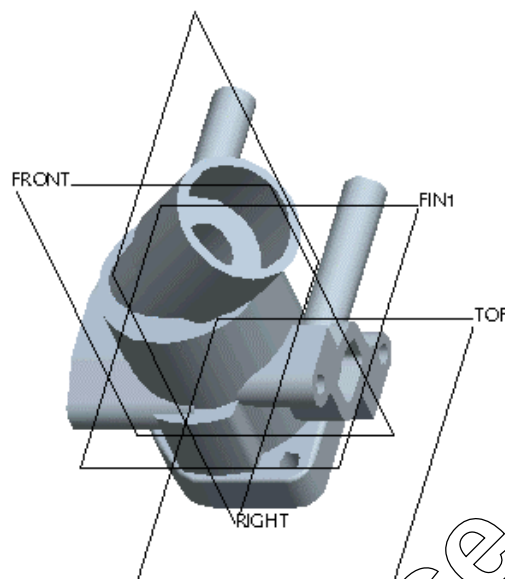


Figure 5: The Completed Datum Plane

4. Save the model.
5. Close the window.

Step 3. Create datum axes on ENG_BLOCK_FRONT.PRT.

In the third step, we create references called datum axes.

Datum axes are used to place features such as coaxial holes, or used as a hinge for rotating other features.

By using axes for hole location references, we reduce the number of parent child relations that can interfere with re-using the parts in the future.

In this demonstration, we create datum axes that reference curved and planar surfaces. The references will enable us to easily locate coaxial holes. The holes will be used as interfaces so that other parts can be bolted to the front engine block.

1. Open ENG_BLOCK_FRONT.PRT.
2. Using the toolbar, orient to the 3D_BACK view.
3. Start the Datum Axis tool. Select the cylindrical surface, as shown in the following figure.

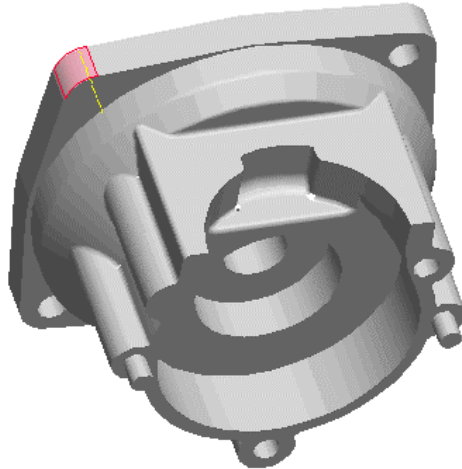


Figure 6: Selecting a Cylindrical Surface

4. Using the Properties tab, enter **GEARBOX1** as the name. Complete the feature.
5. Select the planar surface, as shown in the following figure.

Note:

Using the Smart filter, select the parent feature first, and then select its surface.

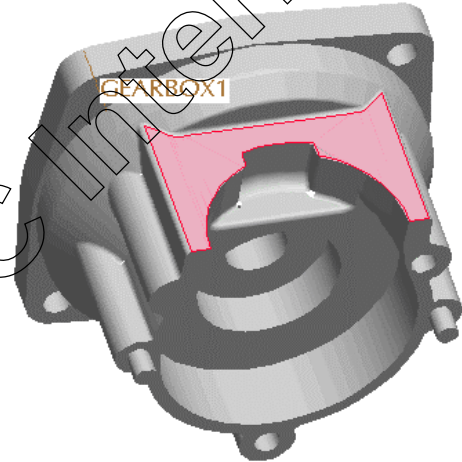


Figure 7: Selecting a Planar Surface

6. Start the Datum Axis tool and orient to the TOP view. Display Datum Planes if necessary.
7. Drag the reference handles to snap to the FRONT and RIGHT planes. Then drag the location handle approximately as shown in the following figure.

Note:

If you are having trouble dragging the reference handle to the datum plane, try dragging the handle to the datum plane's name.

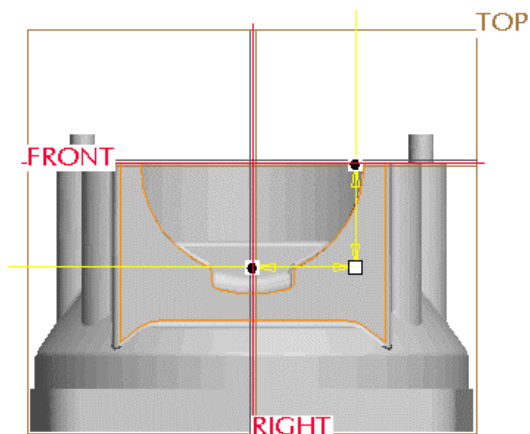


Figure 8: Positioning Drag Handles

Note:

Instead of dragging the reference handles, you can also right-click and select **Offset References**, and then select the offset planes directly.

8. Enter **17.5** for each offset value. Enter **CYLINDER1** as the name. Complete the feature.

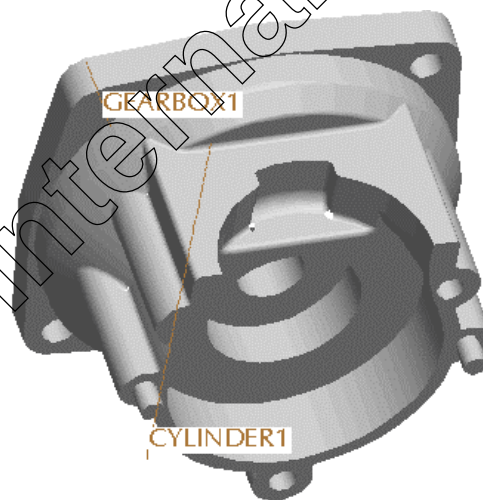


Figure 9: Two Axes Created

Note:

Datum features can be renamed during or after creation. Renaming datum features allows for quick identification when creating subsequent features.

9. Save the model.
10. Close the window.
11. Erase all models from memory.

Exercise 3: Creating Advanced Reference Geometry

Introduction

In this demonstration, we create datum coordinate systems, datum points, and datum curves to use as references. We use these skills to quickly build references within the models.

For example, we use the datum coordinate systems within parts to quickly assemble the parts to each other.

We also create datum points to use as references for axes, holes and curves.

Objectives

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

- Create datum coordinate systems in order to assemble multiple parts.
- Create datum points to use as references for axes, holes and curves.

Scenario

You have been assigned to create datum coordinate systems, datum points, and datum curves to use as references for assembling components. You also use datum coordinate systems for designing features in the engine, the drill chuck, and the handle sub-assemblies.

Step 1. Create datum coordinate systems in CHUCK.PRT.

Again, reference geometry allows us to locate the features in 3-D space.

In this exercise, we use the advanced datum references: datum coordinate systems, datum points, and datum curves.

These features allow us to locate the features while minimizing the amount of parent/child relationships.

In this first step, we create a datum coordinate system. Datum coordinate systems are used as location references to assemble parts and subassemblies.

We create two datum coordinate systems using different methods.

In the first method, we align the coordinate system.

In the second method, we offset the coordinate system from the first.

We also rename the coordinate systems so that we can easily locate them when we assemble the components.

The coordinate systems are used to assemble the drill bit and the drill chuck collar.



Figure 10: The Drill Bit Assembly

1. Open CHUCK.PRT.
2. Start the Datum Coordinate System tool. Select axis **A_2**, press and hold CTRL, and select the bottom surface of the hole (id 1885), as shown in the following figure.
3. Select the ORIENTATION tab. Use the Z-axis from PRT_CSYS_DEF to determine **Z**. Use the X-axis from PRT_CSYS_DEF to determine **X**. Enter **CHUCK_ALIGN** as the name, and then complete the feature. This coordinate system will be used to assemble the drill chuck collar.

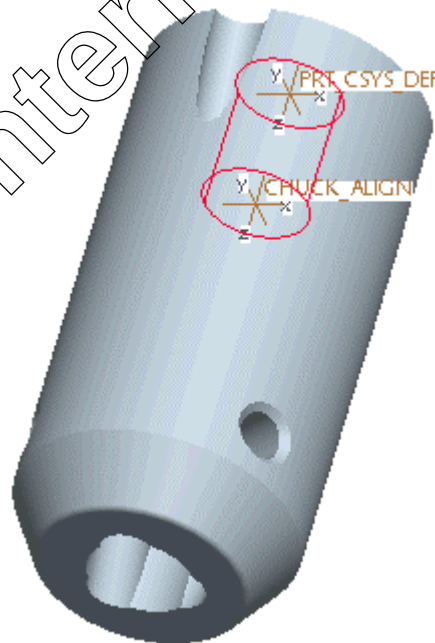


Figure 11: CHUCK_ALIGN Coordinate System

4. With the CHUCK_ALIGN coordinate system selected, start the Datum Coordinate System tool. Drag the location handle in the Z direction, and enter **50** as the offset.

5. Select the ORIENTATION tab. Rotate **180** degrees about X and Z. Enter **BIT_ALIGN** as the name. Complete the feature. This coordinate system will be used to assemble the drill bit.

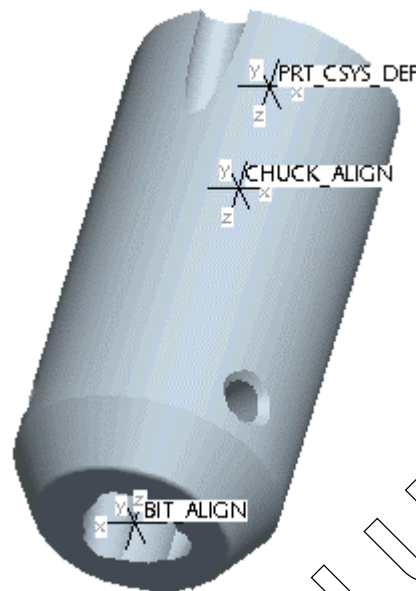


Figure 12: The Completed CHUCK.PRT

6. Save the model.

Step 2. Create a curve through a series of datum points. Use it to design the throttle cable part sweep trajectory.

Next, we create the datum curve through a series of datum points. Datum curves are frequently used as a profile for a cut or protrusion geometry feature.

First, we create a series of datum points by offsetting them to a coordinate system.

Then, we create the curve through the points.

Finally, we set an option on the curve to make it normal to the starting surface.

The benefit in creating a datum curve is shown in the following figure. We can easily use it to model the protrusion that will represent the wire that will lead to the spark plug.



Figure 13: The Completed COIL.PRT

1. Open COIL.PRT.
2. Start the Offset Coordinate Datum Point tool. Select the CSO Csys.
3. Create an array of eight datum points with the offset values shown in the following figure.

Note:

The point names are not important in this case.

× PNT19

× PNT18

× PNT17

× PNT16

× PNT15

× PNT14

× PNT13

× PNT12

× CSO

× BASE_PNT

	Name	X Axis	Y Axis	Z Axis
1	PNT12	0.00	10.00	0.00
2	PNT13	-5.00	20.00	0.00
3	PNT14	-15.00	25.00	0.00
4	PNT15	-45.00	30.00	0.00
5	PNT16	-50.00	35.00	-5.00
6	PNT17	-52.00	40.00	-10.00
7	PNT18	-52.00	47.00	-30.00
8	PNT19	-52.00	47.75	-45.00

Figure 14: The Datum Points Offset of Coordinate System CSO

4. Complete the feature.
5. Insert a datum curve, and click **Thru Points > Done**. Select BASE_PNT, and then select PNT12 to create the curve.

6. Click **Done**, and then double-click **Tangency**. Click **Surface > Normal**, and select the large horizontal surface. Accept the default direction and complete the feature.

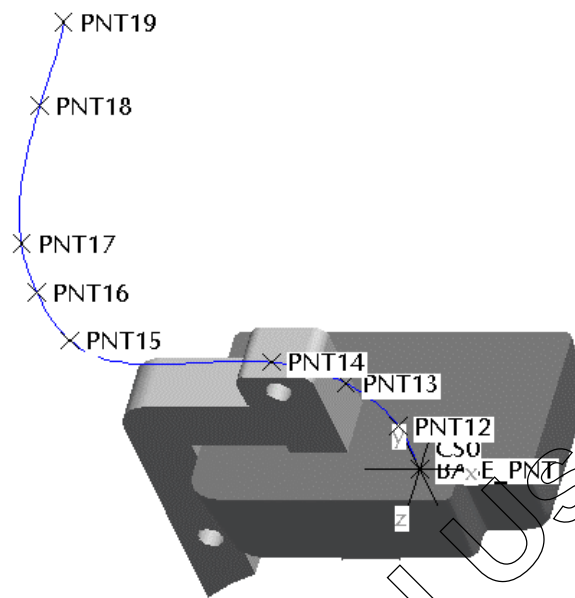


Figure 15: Create the Curve through Points

7. Save the model.
8. Close the window.
9. Erase all models from memory.

Step 3. (OPTIONAL) Create datum points on AIR_FILTER_COVER.PRT.

Instructor: Skip this step for the demo.

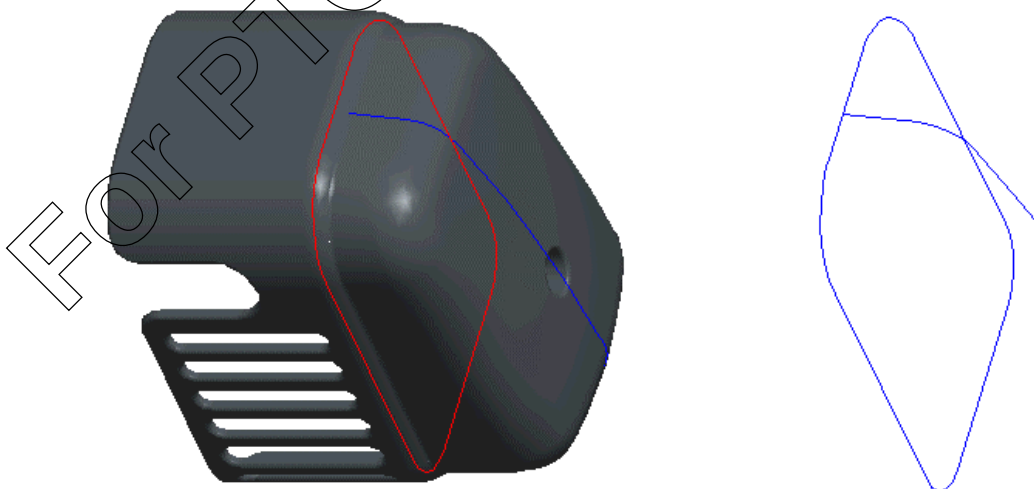


Figure 16: Completed AIR_FILTER_COVER.PRT and Profile Datum Curve

1. Open AIR_FILTER_COVER.PRT.
2. Start the Datum Point tool, and create an array of points, as shown in the following figure:

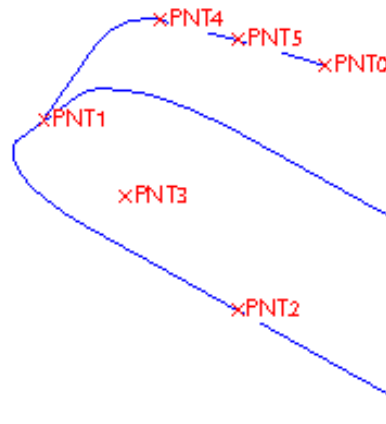


Figure 17: Completed Point for AIR_FILTER_COVER.PRT

- Select the vertex to create **PNT0**.
- Query and select the oval curve, press and hold CTRL, then query and select the L-shaped curve to create **PNT1**.
- Select the front edge of the oval curve, press and hold CTRL, and select DTM1 to create **PNT2**.
- Select the upper left radius of the oval curve, and then change the Reference option from **On** to **Center** to create **PNT3**.
- Query and select the entire L-shaped curve. Enter **0.5** as the offset ratio value to create **PNT4**.
- Query and select the entire L-shaped curve again. Select the **Reference** option, and select DTM1. Enter **25** as the offset.

3. Complete the datum point feature.

Note:

The previous series of datum points should constitute a single datum point feature.

4. Click **Edit > Setup > Name > Other**, and select PNT3. Enter **AXIS_PNT** as the name. Click **Done**.
5. Select the datum point feature, and then select **AXIS_PNT**.
6. Start the Datum Axis tool, press and hold CTRL, and select the **RIGHT** plane. Complete the feature.

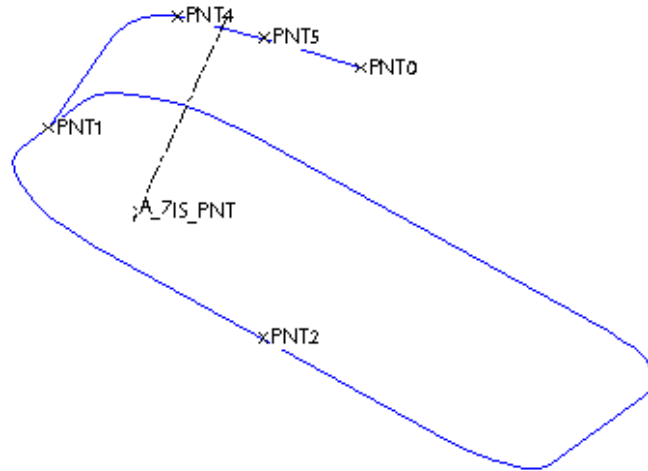


Figure 18: Normal to Right Axis through AXIS_PNT

7. Save the model.
8. Close the window.
9. Erase all models from memory.

Step 4. (OPTIONAL) Create a datum point on a surface

Instructor: Skip this step for the demo.

1. Open RECOIL_COVER.PRT.
2. Start the Datum Point tool.
3. Query and select the surface shown in the following figure.

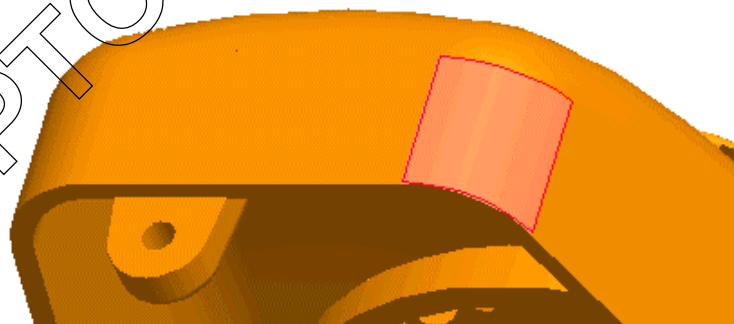


Figure 19: Selecting a Surface

4. Right-click and select **Offset References**. Press and hold CTRL and select the FRONT and RIGHT datum planes. Enter **-19** and **25** respectively.
5. Complete the feature. This point can be used to create an on-point hole.

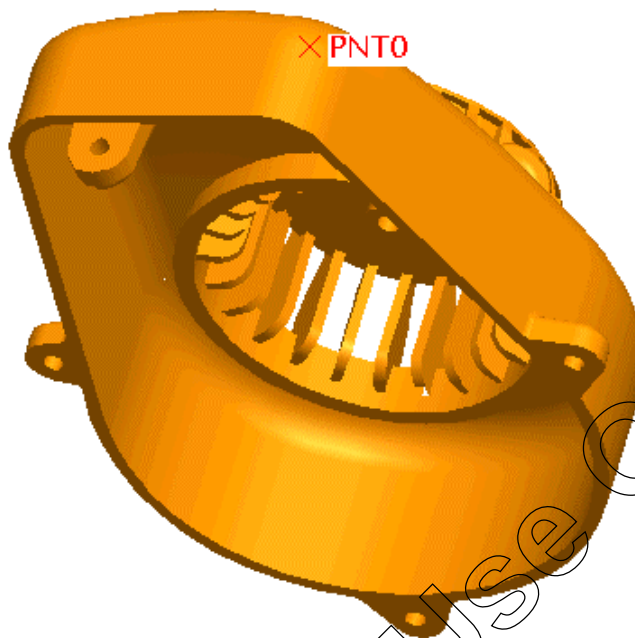


Figure 20: Datum Point On-Surface.

6. Save the model.
7. Close the window.
8. Erase all models from memory.

Exercise 4: Creating and Managing Layers

Introduction

In this demonstration, we create layers in order to simplify the displayed information.

Layers allow us to quickly remove information from the graphics window, such as features, components, and references.

For example, we can blank all datum points used to create the trajectory of a curve, while displaying all datum points that define connections between movable parts.

We can also blank all surfaces that were used as references.

Layers are especially useful when working with complicated designs because they simplify the model's appearance.

Objectives

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

- Create layers containing reference geometry features.

Scenario

You create layers at assembly and part levels in order to organize datum features. It is important to note that layers of individual parts can be seen at the assembly level, which enables you to easily manage their display in the graphics window.

Step 1. Create a layer that contains datum planes from DRILL.ASM and CYLINDER.PRT.

In this exercise, we initially create a layer that will contain the assembly level datum planes.

Afterwards, we open a part on the assembly and create a layer with the same name to contain the part level datum planes. Note that when you now look at the layer in the assembly, the part level layer is also present.

1. Close all windows and erase all models from session.
2. Navigate to: **C:\users\student\fast_track_330\module_03\assy_layer**.
3. Right-click and select **Make Working Directory**.
4. Open DRILL.ASM.
5. Display datum planes if necessary.
6. In the layer tree, right-click and select **New Layer**. Enter **DTMS** as the name.
7. Using the graphics window, select the ASM_FRONT, ASM_RIGHT, and ASM_TOP planes to add them to the layer. Complete the layer.
8. Show the model tree and open CYLINDER.PRT.

9. In the layer tree, right-click and select **New Layer**. Enter **DTMS** as the name.
10. Select the FRONT, RIGHT, and TOP planes to add them to the layer. Complete the layer.
11. Close the window to return to DRILL.ASM. The layer tree should appear as shown in the following figure.

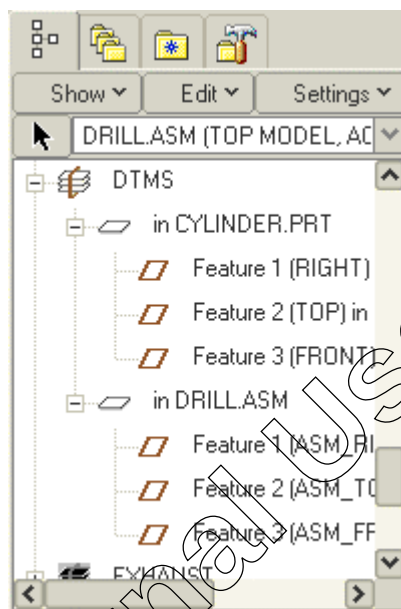


Figure 21: Create the DTMS layer.

12. Blank the DTMS layer and repaint the screen. Notice that some planes are still visible.
13. Click **Edit > Save Status** to save the current layer status.

Note:

Layer status is not saved by default when the model is saved.

14. Save the model.
15. Close all windows.
16. Erase all models from memory.

This completes the exercise.

Summary

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

- Create reference geometry.
- Create features using reference geometry.
- Describe how creating reference geometry affects parent/child relationships.
- Manage layers when creating reference geometry.

For PTC Internal Use Only

Creating Direct Features

Introduction

There are many ways to create geometric features in Pro/ENGINEER Wildfire. You can use traditional approaches that begin with 2-D sketches, and then proceed to solid extrusions, dimensioning, and placement.

Alternatively, you can use direct features to quickly add features such as holes, chamfers, and rounds, by placing these pre-defined shapes onto the design model. This process simplifies the feature creation process. You omit the sketch phase and move directly to placement and dimensioning on the design model.

Objectives

After completing this module, you will be able to:

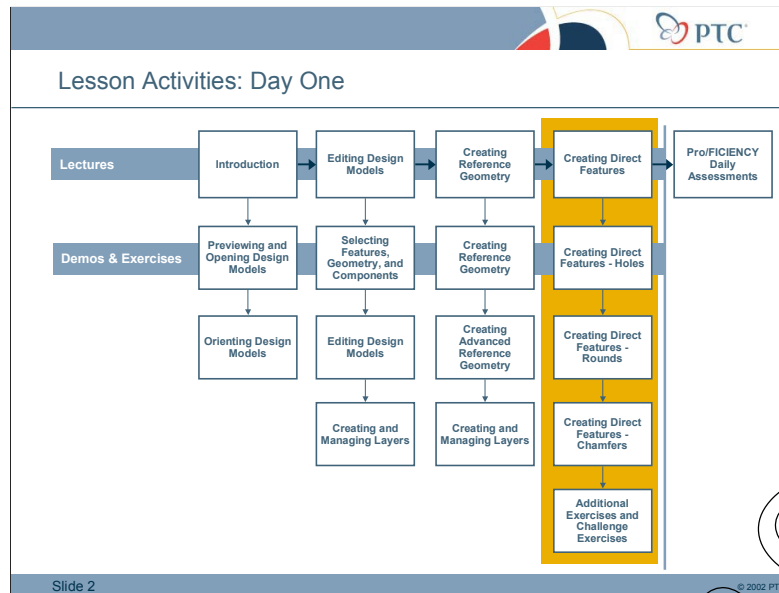
- Create direct features such as holes, rounds, chamfers, drafts, and shells.
- Describe how creating direct features affects parent/child relationships.
- Create and manage layers when creating direct features.

Fast Track to Pro/ENGINEER Wildfire

Module 4: Creating Direct Features



For PTC Internal Use Only



Duration

- Lecture: 10 mins
- Demos (5): 30 mins
- Labs (5): 80 mins
- Total: 2 hrs

PTC

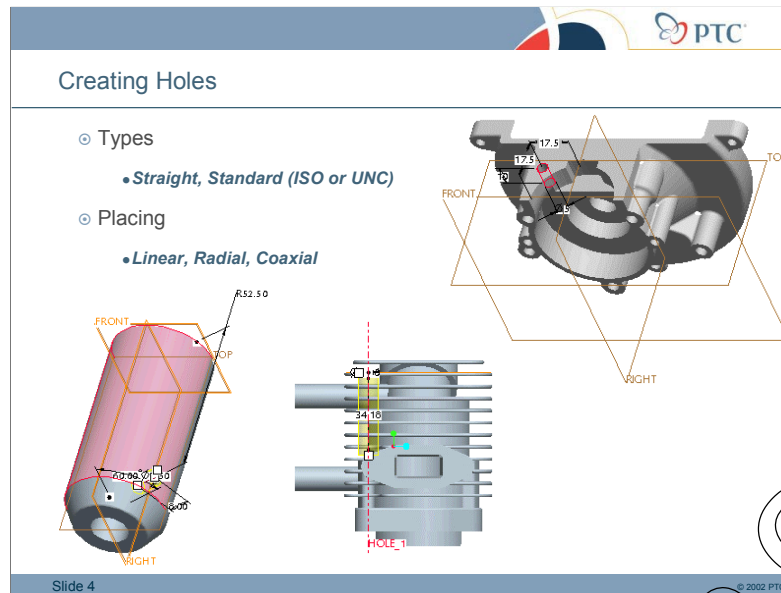
Objectives

After completing this module, you will be able to:

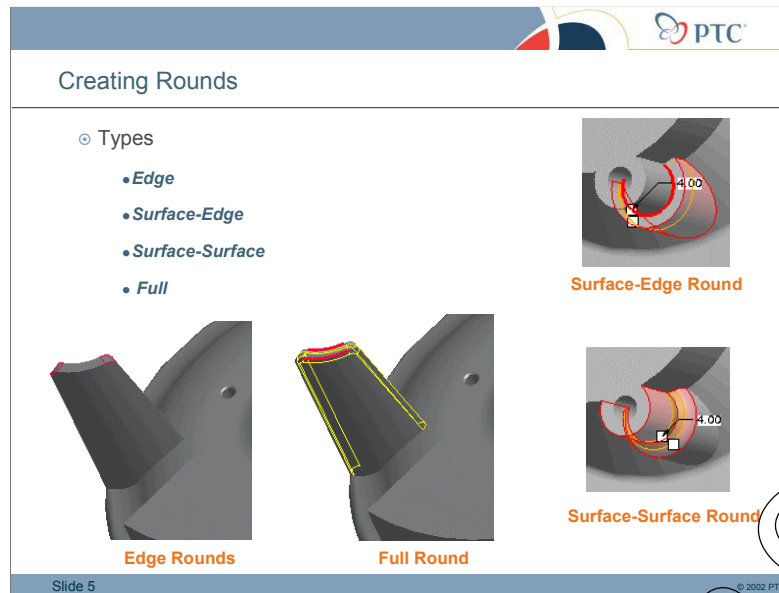
- Create direct features such as holes, rounds, chamfers, drafts, and shells.
- Describe how creating direct features affect parent/child relationships.
- Create and manage layers when creating direct features.

Slide 3

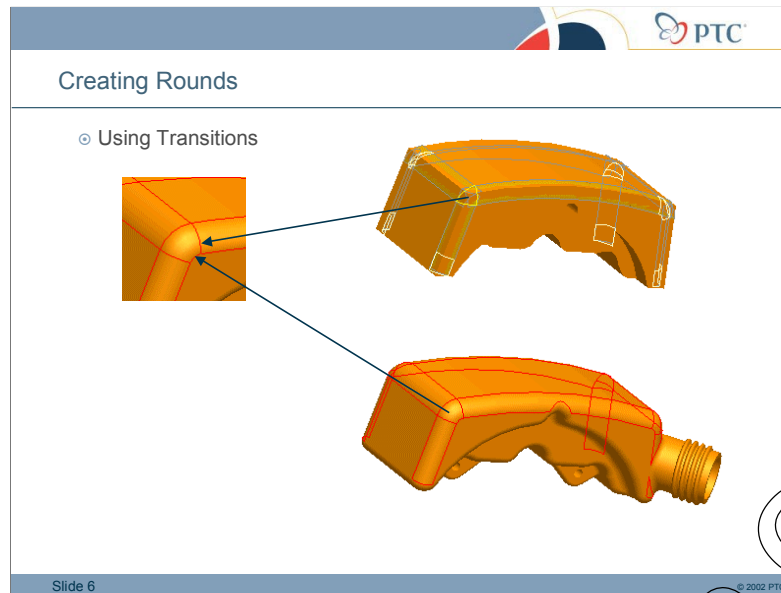
- There are many ways to create geometric features in Pro/ENGINEER Wildfire.
- You can use traditional approaches that begin with 2-D sketches and then proceed to solid extrusions, dimensioning, and placement.
- Alternatively, you can use “direct” features.
- Direct features enable you to quickly add features (such as holes, chamfers, rounds, and so on) by placing these pre-defined shapes on design models.
- This process simplifies and speeds the feature creation process because you skip the sketch phase and move directly to placement and dimensioning on the design model.



- Holes are direct features with pre-defined shapes that remove material from design models.
- You have to define the size and location in the design model when creating them.
- The figures in this slide display examples of straight and standard holes.
 - Types
 - Straight, Standard
 - Placing
 - Linear, Radial, Coaxial
- For straight holes, you define the diameter and locating references on the model.
- For standard holes, you can choose from standard ISO or UNC sizes and optionally display the hole information as notes on the screen.
- The linear hole uses linear dimensions to be located on the block.
- The coaxial hole references the cylinder's datum axis for placement.
- The radial hole placed on a cylinder references the end surface of the cylinder for location and the radial hole on the plate references a datum plane for the angle and the top planar surface for placement.
- The block containing the standard holes include a counter-sink, counter-bore and tapped hole. The block is made transparent to visualize the shape of these standard holes.

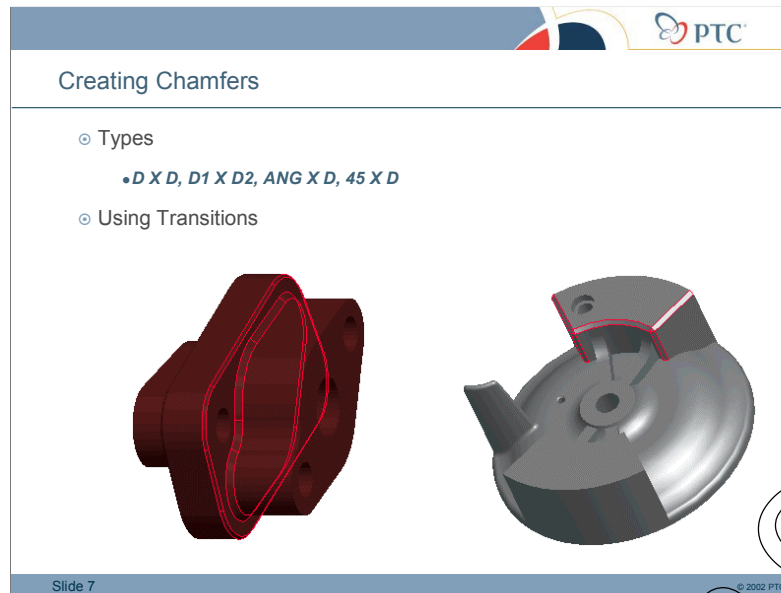


- Rounds add or remove material by creating a smooth transition between adjacent surfaces and edges selected as references.
- You have to define the size and location in the design model.
 - Types
 - Edge, Edge-Surface, Surface-Surface, Full
- The figures in this slide display examples of the different types of rounds that add and remove material.



Transitions enable you to define the appearance of rounds where multiple round sets intersect in a design model. This is important if you have a specific preference to how your products 'looks'.

- The block has a corner sphere type transition defined where the three round sets meet.



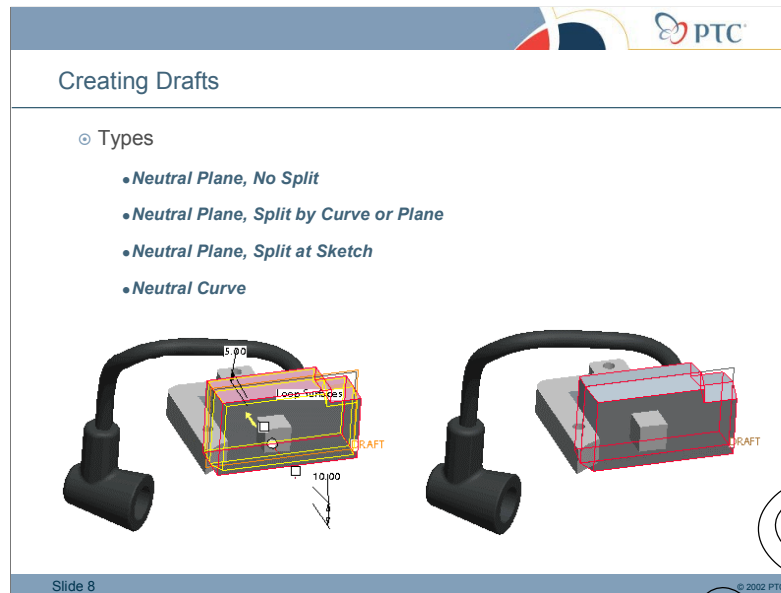
- Chamfers add or remove material by creating a bevelled surface between adjacent surfaces and edges selected as references.
- You define the size and location in the design model.

Types

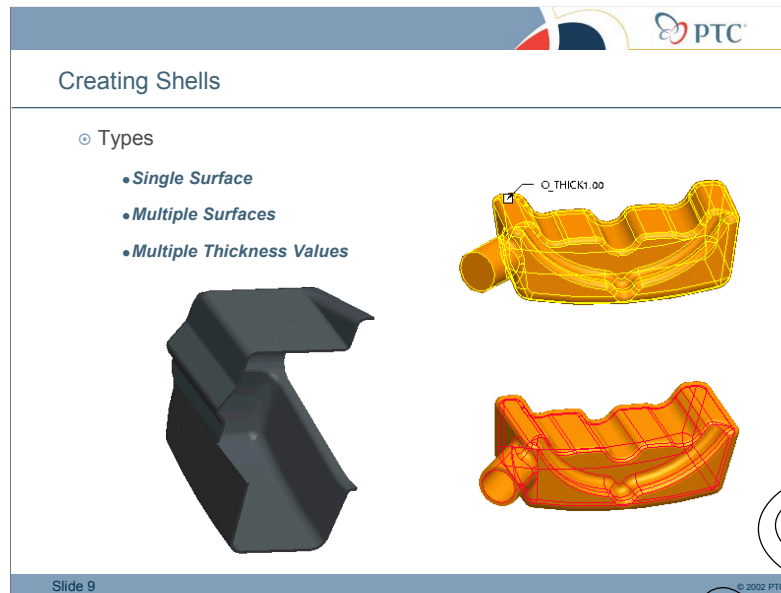
- D X D, D1 X D2, ANG X D, 45 X D
- The figures display chamfers that use different dimensioning schemes.

Using Transitions

- Like round transitions, you can optionally define transitions where multiple chamfer sets intersect in a design model to assign a preferred appearance.
- The block has a patch type transition defined where the three chamfer sets meet.



- Drafts are used as finishing features in molded and cast parts.
 - Types
 - Neutral Plane, No Split
 - Neutral Plane, Split by curve or plane
 - Neutral Plane, Split at sketch
 - Neutral Curve
- To create a draft, you select the surfaces to be drafted, a neutral plane or curve that defines the hinge, the pull direction and the draft angle.
- You create drafts with or without splits and select curves, planes or sketched sections to define the split.
- This slide displays a draft with no split on the top cylinder surfaces, and a draft with a split at the central datum plane on the surfaces of the base block.
- The draft angle ranges from – 30 deg to + 30 deg.

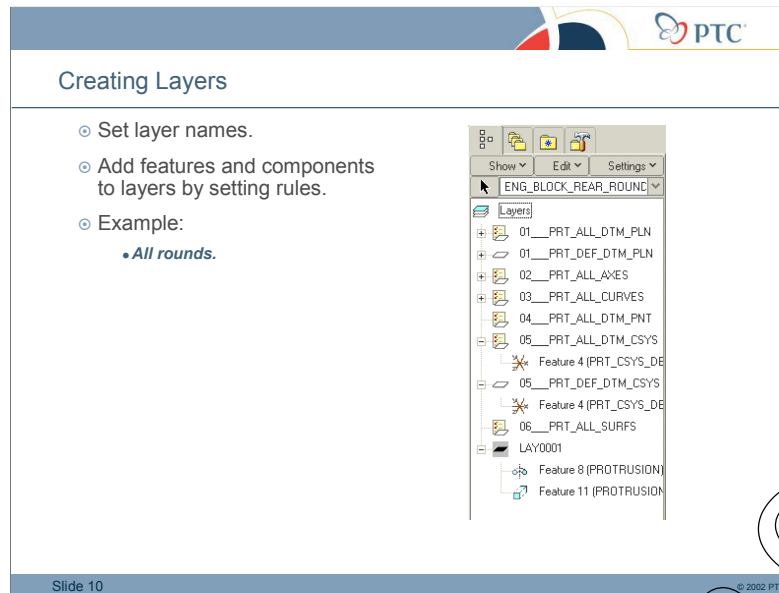


Shells remove surfaces to hollow out a design model leaving walls with specified thickness values.

- Types

- Single Surface
- Multiple Surfaces
- Multiple Thickness Values

- The figures on this slide display shells removing single and multiple surfaces with constant and varying thickness walls.
- It is a good design practice to create shells as finishing features and not early in the design process.



- Organizing geometry items using layers enables you to selectively perform operations.
- For example, you can control their display on the screen, as well as delete and suppress them from the model.
- When you use default templates for creating new design models, a set of default layers is automatically created.
- Pro/ENGINEER Wildfire adds geometry items corresponding to each default layer as the geometry items are created.
- For example, all default datum planes are automatically added to the default layer 01_PRT_DEF_DTM_PLN.
- You can select geometry items, include all types of features and components by:
 - Using the model tree.
 - Using the graphics window.
 - Setting rules: for example, selecting all the holes and datum axes from the model.



Parent/Child Relationships – Creating Direct Features

When creating direct features, parent/child relationships are created. Changes made to a parent feature affects its children.

- Holes
- Rounds
- Chamfers
- Drafts
- Shells

Slide 11

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Note to Instructors:

Use this slide to teach students how the functionality they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

All direct features are children to their references. For example, coaxial holes are children to the axis and surface that they are located on.

Edge rounds are children to the selected edge curves. Surface to surface rounds are children to two surfaces.



Demonstrations

- ◉ Creating Direct Features – Holes
- ◉ Creating Direct Features – Rounds
- ◉ Creating Direct Features – Chamfers
- ◉ Creating Direct Features – Drafts
- ◉ Creating Direct Features – Shells
- ◉ Creating and Managing Layers

Slide 12

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You will follow along while I demonstrate:

1. Creating Direct Features – Holes
2. Creating Direct Features – Rounds
3. Creating Direct Features – Chamfers
4. Creating Direct Features – Drafts
5. Creating Direct Features – Shells
6. Creating and Managing Layers

Then, you will practice the same steps in the laboratory exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- ⦿ Achieve the course objectives.
- ⦿ Use Pro/FICIENCY assessment questions.
- ⦿ Apply Precision Learning.



Slide 13

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Precision Learning

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

Note to Instructors:

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 Exams

Description: For each course, Pro/FICIENCY exams are 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:

- Create direct features such as holes, rounds, chamfers, drafts, and shells.
- Describe how creating direct features affect parent/child relationships.
- Create and manage layers when creating direct features.

For PTC Internal Use Only

Referencing PTC Help for Module 4 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Rounds	Part Modeling	About the Round Feature
Transitions	Part Modeling	About Round Transitions About Chamfer Transitions
Chamfers	Part Modeling	About the Chamfer Feature
Drafts	Part Modeling	About the Draft Feature
Shell	Part Modeling	About the Shell Feature
Layers	Fundamentals	About Layer Display

Module 4 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Conventions

- **Demo script** is formatted in italic.

Exercise 1: Creating Direct Features – Holes

Introduction

In this demonstration, we create basic hole features. Basic holes ease the workload, because we do not have to sketch them to create them. We can simply select the hole that we want to create, place them onto the model, and dimension them as needed.

Objectives

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

- Create basic holes

Scenario

In order to assemble the drill, you must create holes on a number of parts that will act as bolt interfaces. You create the holes with pre-defined shapes that remove material on the front engine block, rear engine block, and cylinder, so that bolts may be inserted during assembly.

Step 1. Create coaxial holes on ENG_BLOCK_FRONT.PRT.

Now that we know how to select, edit, and locate features, we are ready to create direct features. Direct features are geometry features that are commonly used when developing parts.

In this first demonstration, we create straight holes. Straight holes contain positional references, a diameter, and a depth. They are used in many places including part-to-part interfaces where the parts are bolted together.

In the initial step, we create two coaxial holes.

Notice that the mouse never leaves the graphics window.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to:
C:\users\student\fast_track_330\module_04.
3. Open ENG_BLOCK_FRONT.PRT. Turn on Datum Axis display.
4. Select the GEARBOX1 axis from the model tree.
5. Start the Hole tool. Right-click and select **Secondary References Collector**. Select the surface shown in the following figure.

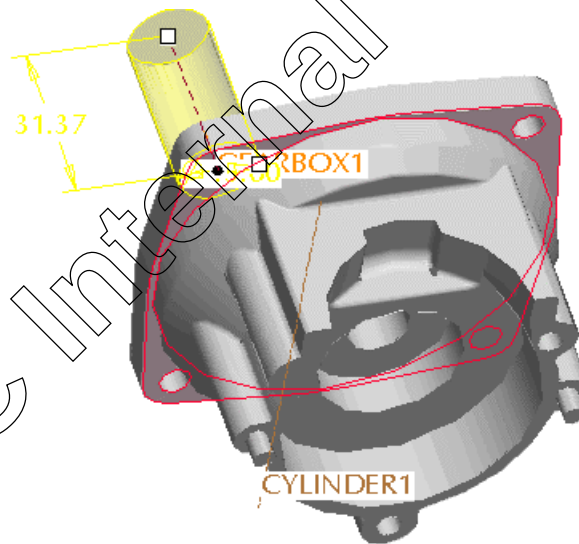


Figure 1: Selecting a Surface for the First Hole

6. Cursor over the Depth handle. Right-click and select **To Next**. Double-click on the diameter dimension and enter **6**. Complete the feature.
7. Start the Hole tool and select the CYLINDER1 axis. Right-click and select **Secondary References Collector**. Select the surface shown in the following figure.

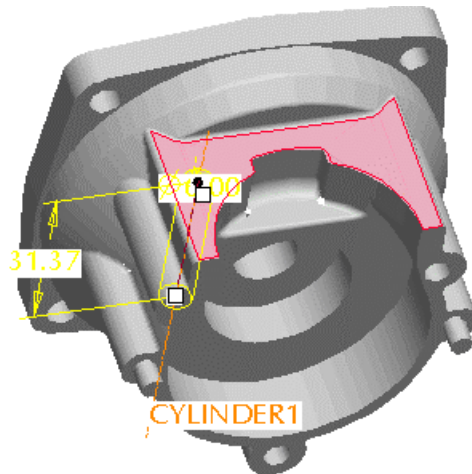


Figure 2: Selecting a Surface for the Second Hole

8. Using the dashboard, enter **10** as the depth and enter **5** as the diameter. Complete the feature.
9. Save the model.

Step 2. Create a linear hole on ENG_BLOCK_REAR_HOLE.PRT

Next, we create another hole. In this instance, we right-click to select the reference planes.

1. Open ENG_BLOCK_REAR_HOLE.PRT.
2. Start the Hole tool, and select the large horizontal surface.
3. Right-click and select **Secondary References Collector**. Select the RIGHT plane, press and hold CTRL, and select the FRONT plane.

Note:

You can also drag the reference handles to snap to references.

4. Drag the location, radius, and depth handles to approximate positions, and then size the hole, as shown in the following figure.
5. Using the graphics window, double-click to edit the dimensions as shown in the following figure:

- Depth = **10**
- Diameter = **5**
- Distance from RIGHT plane = **17.5**
- Distance from FRONT plane = **17.5**

Note:

When locating features with reference planes, you may have to use negative values.

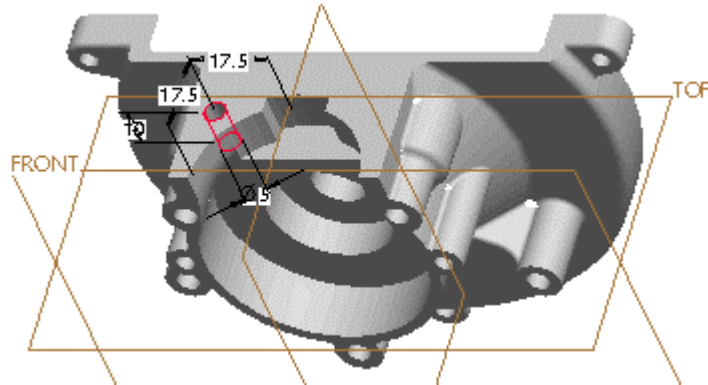


Figure 3: Creating a Linear Hole Using the Lead Workflow

6. Complete the feature.
7. Save the model.

Step 3. Edit the definition of a hole on CYLINDER.PRT.

In this step, we experiment with depth options on an existing hole.

1. Open CYLINDER.PRT.
2. Select HOLE_2 using the model tree, and edit its definition.

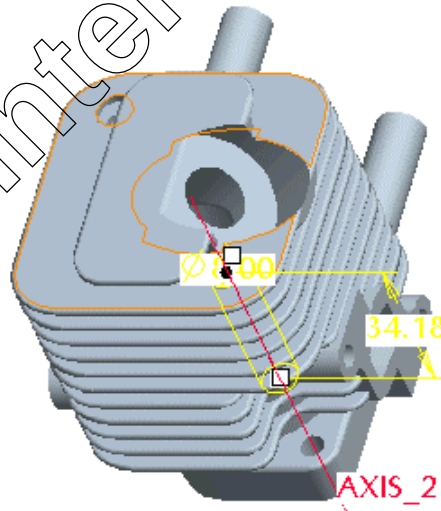


Figure 4: Editing the Definition of HOLE_2

3. Drag the depth handle to **20** and preview the feature. Resume the dashboard.
4. Cursor over the Depth handle. Right-click and select **To Next**.
5. Preview the feature and then Resume the dashboard.
6. Cursor over the Depth handle. Right-click and select **Through All**.

7. Preview the feature and then Resume the dashboard.
8. Cursor over the Depth handle. Right-click and select **To Selected**. Select the bottom surface of the fin as shown in the following figure.

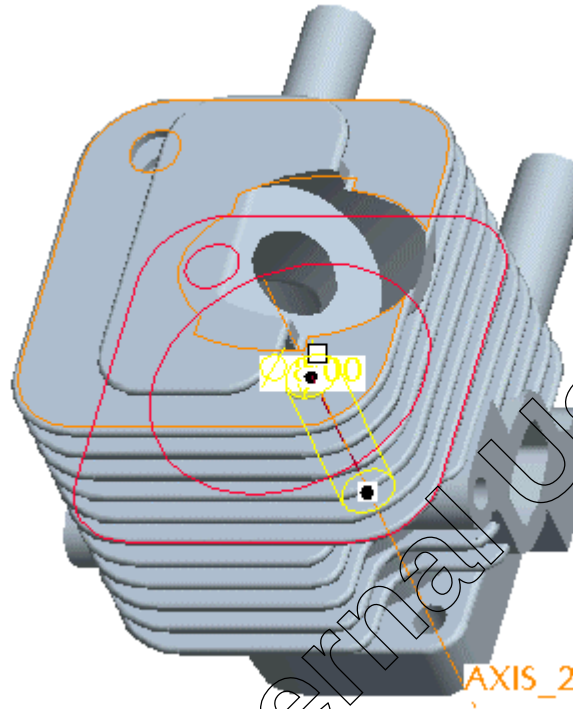


Figure 6: Selecting a Surface for Depth

9. Complete the feature.
10. Save the model.
11. Close all windows.
12. Erase all models from memory.

Step 4. (OPTIONAL) Create an on-point hole.

Instructor: Skip this step for the demo.

1. Open RECOIL_COVER.PRT.
2. Start the Hole tool, and select PNT0.
3. Set the depth to **To Next** and the diameter to **12**. Complete the feature.

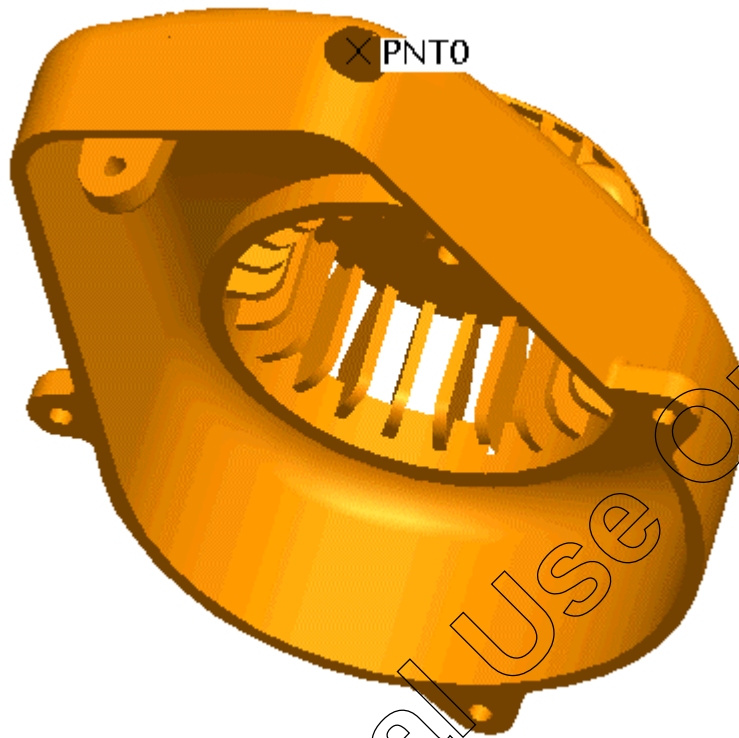


Figure 6: On-Point Hole

4. Save the model.
5. Close all windows.
6. Erase all models from memory.

This completes the exercise.

Exercise 2: Creating Direct Features – Rounds

Introduction

In this demonstration, we create basic round features.

Basic rounds are important because they ease the workload. We do not have to sketch the features to create them. We simply select the edge and/or surfaces that we want to round and dimension.

Objectives

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

- Create basic rounds.

Scenario

In order to reduce stresses and remove sharp edges within the drill components, you add rounds to the front gearbox and connecting rod.

Additionally, rounds add aesthetic value to parts. Preview and edit the fuel tank's round transitions to create an improved appearance.

Step 1. Create the first of three edge round features on GEARBOX_FRONT.PRT.

Basic rounds are rounds that contain a positional references and a radius. They are used to smooth potentially sharp edges and for aesthetic value.

In this initial step, we create three rounds to reduce the amount of stress within the part by smoothing sharp edges.

We create the initial round by following the workflow to select the edges. Note that by selecting both edges, we set a dependency that both edges will use the same radius dimension.

Since we do not want both edges to have the same value for their radius, we remove one edge reference.

It is important to learn the different round selections and workflows to find the best process to increase productivity.

(Note to Instructor: Discuss config option millimeter_grid_interval.)

1. Open GEARBOX_FRONT.PRT.
2. Start the Round tool, and select the large circular edge. Press and hold CTRL and select the small circular edge. Refer to the following figure.
3. Drag the radius handle to approximately 1.50. Using the dashboard, enter **1.9** as the radius.

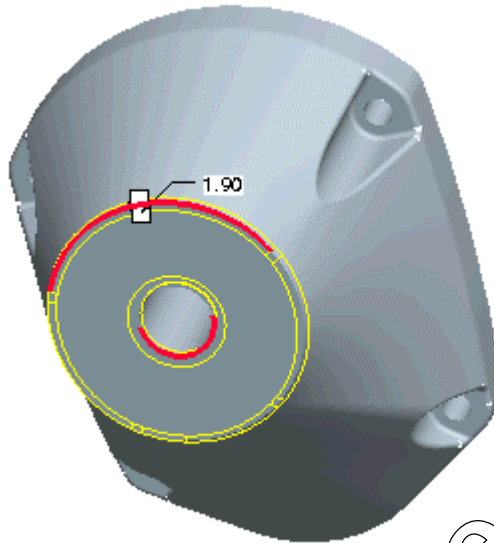


Figure 7: Edge References for the Round

4. Using the dashboard, select the SETS tab, and select the first edge reference (F7), as shown in the following figure. Right-click and select **Remove**.

Note:

You can also de-select references by selecting the reference again while holding CTRL.

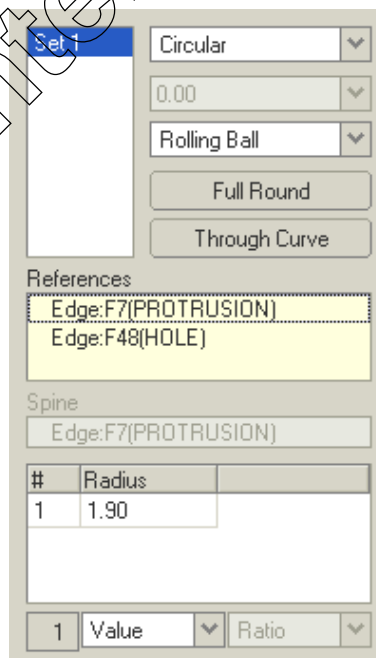


Figure 8: Deleting a Reference in Round Set

5. Select the large circular edge again from the model. A second set should appear in the dashboard, as shown in the following figure.

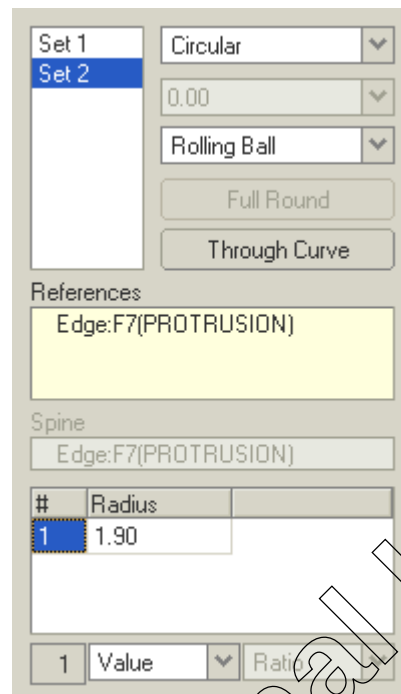


Figure 9: Adding a Second Round Set

6. Edit the second round set radius to 2.
7. Complete the feature.

Note:

Selecting references *without* the CTRL key creates independent sets.
Selecting *with* the CTRL key adds/removes references to the same set.

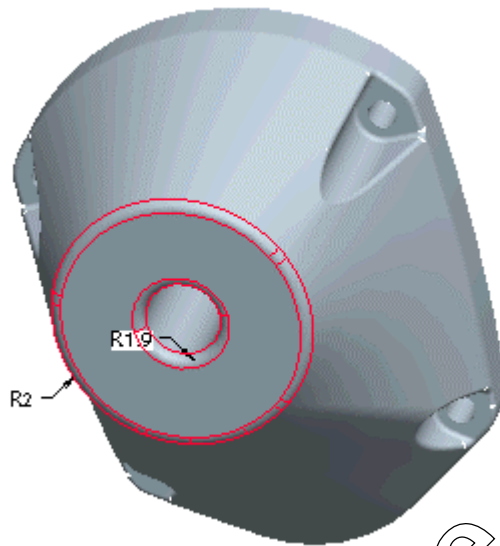


Figure 10: Completed Rounds with Dimensions Displayed

8. Save the model.

Step 2. Create the second of three edge round features on GEARBOX_FRONT.PRT.

Now we will reorient the model and create a tangent chain round.

1. Orient to the **3D_BACK** view.

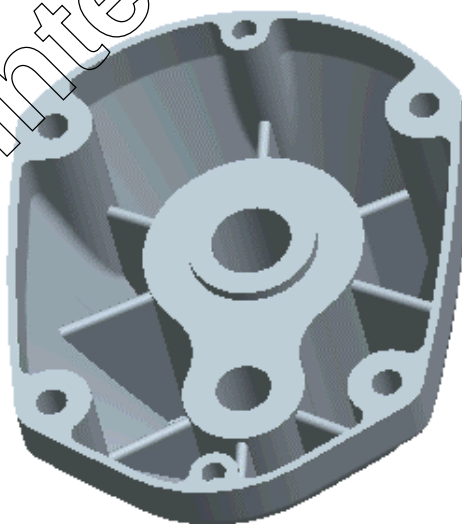


Figure 11: Reorienting the Model

2. Select the edge, as shown in the following figure.

Note:

To select an edge, select the parent feature first, then the edge.

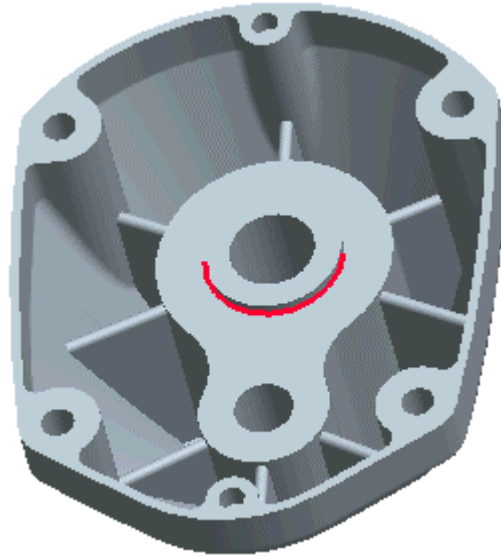


Figure 12: Selecting the Edge Chain

3. Start the Round tool and edit the radius to 1.
4. Select the tangent chain, as shown in the following figure.

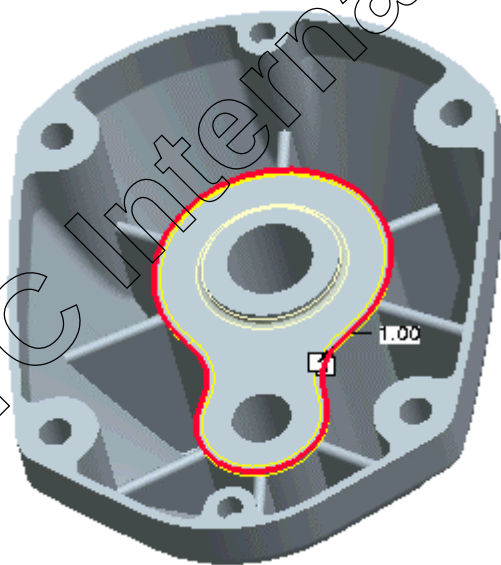


Figure 13: Editing the Radius

5. Drag the reference handle and edit the radius to 4, as shown in the following figure.

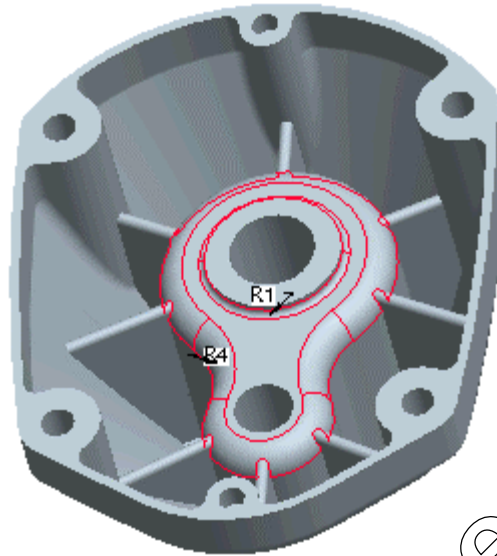


Figure 14: Rounds with Dimensions Displayed

6. Complete the feature.
7. Save the model.

Step 3. (OPTIONAL) Create the third of three edge round features on GEARBOX_FRONT.PRT.

Instructor: Skip this step for the demo.

1. Start the Round tool. Select the first edge, as shown in the following figure. Press CTRL and select the seven additional references to the same round set, as shown in the following figure.

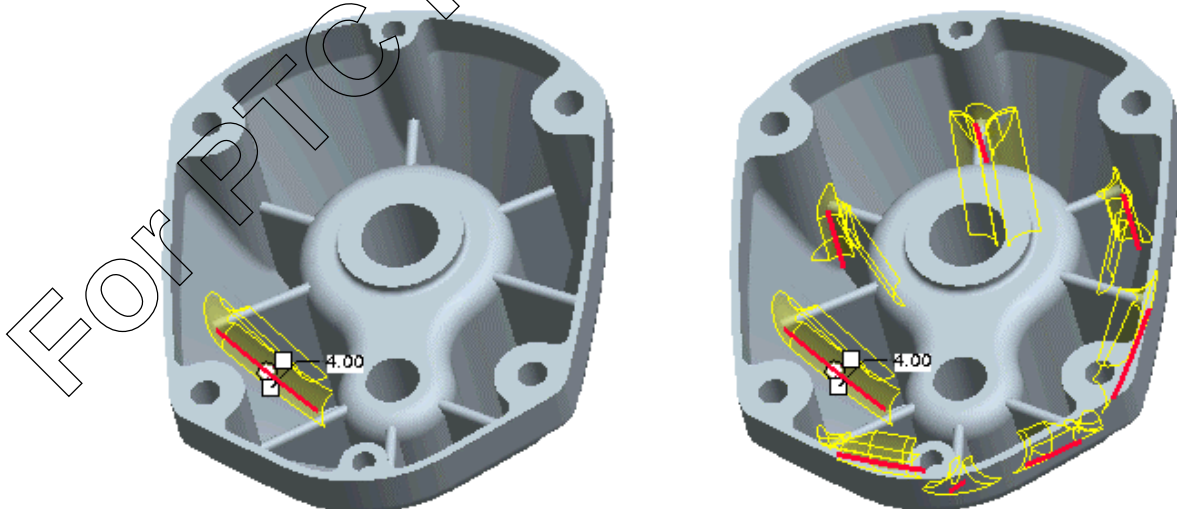


Figure 15: Selecting the Multiple Round References

2. Edit the radius to 1.

3. Complete the feature.

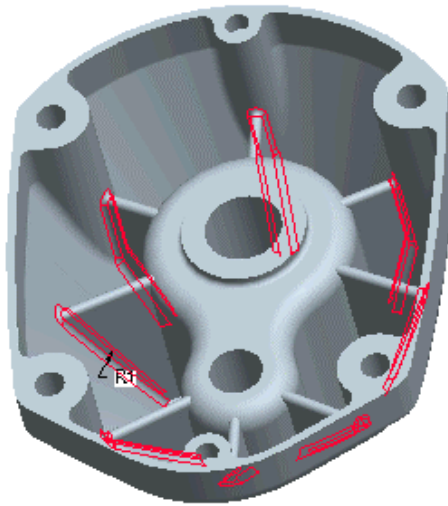


Figure 16: Completed Rounds with Dimension Displayed

4. Save the model.

Step 4. Create full rounds on CONNECTING_ROD.PRT.

Next we add rounds to the connecting rod.

It is a standard practice to reduce the amount of stress within the part.

We create a surface-to-surface round along the length of the connecting rod.

Then we round the additional edges where the length meets the rod's interfacing features.

1. Open CONNECTING_ROD.PRT.
2. Select the FULL_ROUND from the model tree, and edit its definition.
3. Right-click and select **Add Set**.
4. Select the large rectangular surface. Press CTRL + right-click to query to the back surface, and select it. Then, select the thin surface on the right to complete the round set, as shown in the following figure.

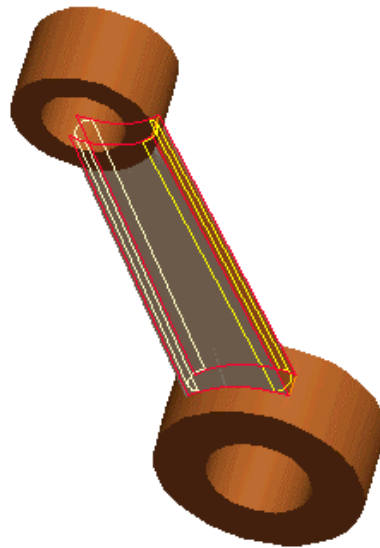


Figure 17: Creating the Initial Surface-To-Surface Round Set

5. Select the **Sets** tab from the dashboard. View the contents of Set 1 and Set 2.
6. Complete the feature.

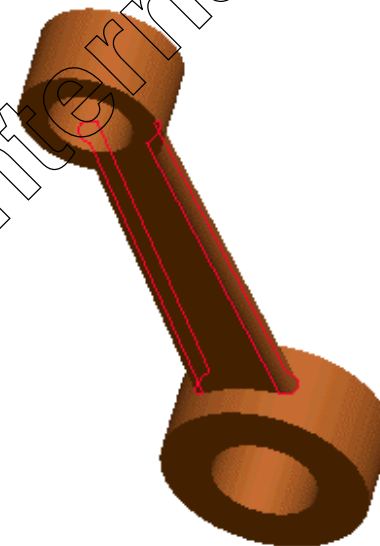


Figure 18: The Completed Round Feature

Step 5. Create edge rounds on CONNECTING_ROD.PRT.

Now that we have tangent chains in the geometry, we can create the final rounds.

1. Start the Round tool, and select the two edges shown in the following figure, adding them to the same set.
2. Edit the radius to 2. Complete the feature.

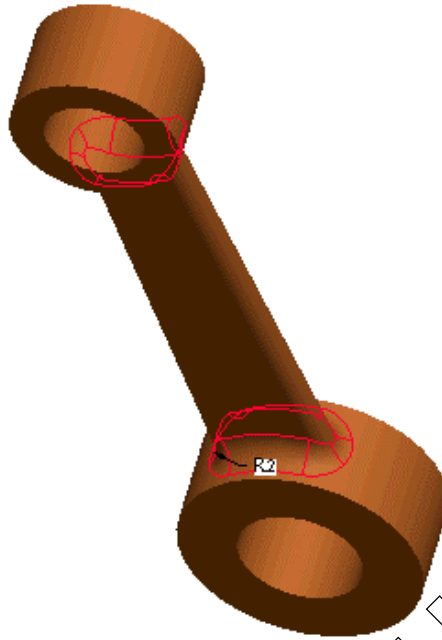


Figure 19: The Complete Rounds

3. Save the model.
4. Close the window.

Step 6. Attempt an edge chain round on ENG_BLOCK_REAR_ROUNDS.PRT.

Again in the rear engine block, we want to reduce the stresses that will be present at various sharp edges.

1. Open ENG_BLOCK_REAR_ROUNDS.PRT. Orient the model, as shown in the following figure.
2. Start the Round tool. Press and hold CTRL and select the two edges, as shown in the following figure. Edit the radius to 2.

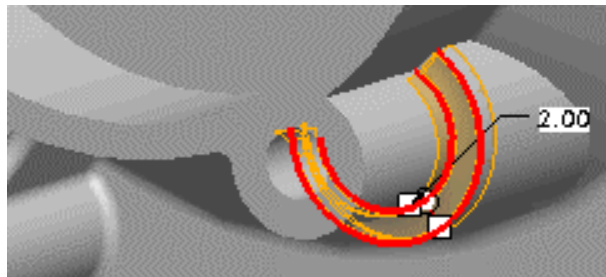


Figure 20: The Edge Round

3. Preview the feature. Note that the feature fails because of the rounds overlap each other.
4. Resume the Round tool and cancel the feature.

Step 7. Create a surface-to-edge round on ENG_BLOCK_REAR_ROUNDS.PRT.

After initially trying an edge-chain round, we use an edge-to-surface round to smooth a sharp edge.

1. Start the Round tool, and select the larger cylindrical surface. Press and hold CTRL and select the edge (bold red highlight), as shown in the following figure. Edit the radius to 4.

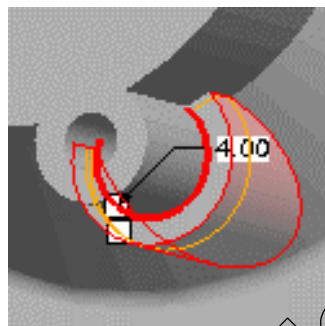


Figure 21: Creating a Surface-to-Edge Round

Note

A larger radius is possible on this surface-to-edge round.

2. Complete the feature.

Step 8. Create a surface-to-surface round on ENG_BLOCK_REAR_ROUNDS.PRT.

Now we will create a surface-to-surface round.

1. Start the Round tool, and select the smaller cylindrical surface. Press and hold CTRL and select the surface of the previous round, as shown in the following figure. Edit the radius to 4.

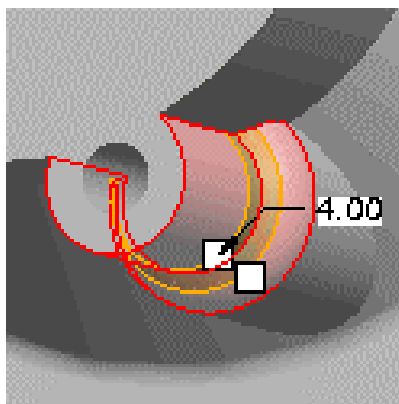


Figure 22: A Surface-to-Surface Round

2. Complete the feature.

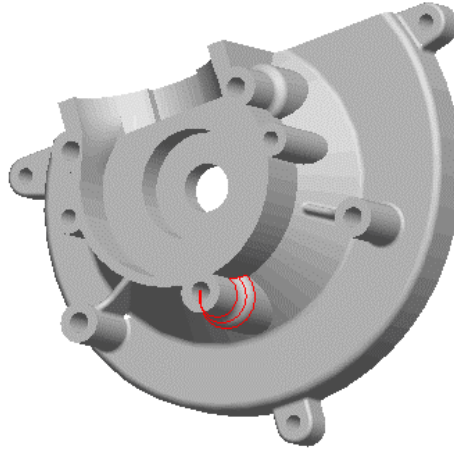


Figure 23: Completed Model

3. Save the model
4. Close all windows.

Step 9. (OPTIONAL) Edit the round transitions on FUEL_TANK_ROUNDS.PRT.

Instructor: Skip this step for the demo.

1. Open FUEL_TANK_ROUNDS.PRT.
2. Orient to the TRANSITIONS view.
3. Using the model tree, select the ROUND_TRANSITION feature, right-click and select **Edit Definition**.
4. Switch to Transitions Mode, and select the nearest transition, as shown in the following figure.

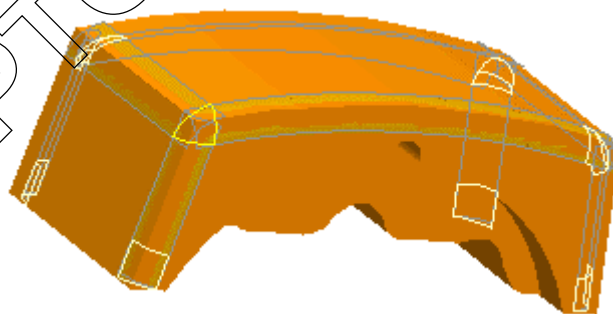


Figure 24: Round Transitions

5. Use the dashboard to change the round transition type to **Intersect**.
6. Right-click on the graphics window, and select **Corner Sphere** as the transition type.
7. Select **Corner Sweep** as the transition type.

8. Complete the feature.

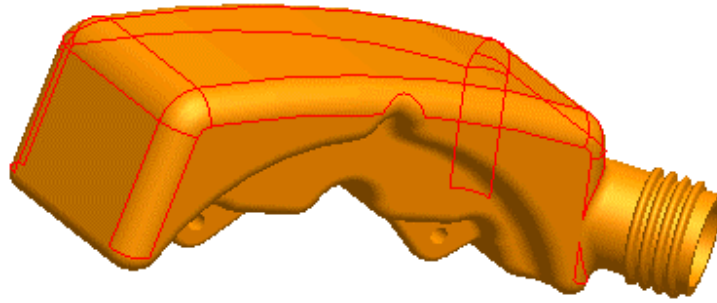


Figure 25: Completed Round Transitions

9. Save the model.
10. Close the window.
11. Erase all models from memory.

This completes the exercise.

Exercise 3: Creating Direct Features – Chamfers

Introduction

In this demonstration, we create chamfer features.

We do not have to sketch chamfers to create them.

We simply select the edge that we want to chamfer and dimension as needed.

Objectives

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

- Create chamfers.

Scenario

In order to create interfaces between moving parts, it is often necessary to create chamfers between the parts. In other instances, a chamfer produces a different aesthetic appearance than a round by smoothing sharp edges.

In this exercise, you create multiple chamfers on the drill's crankshaft where it interfaces with the engine block. You also create aesthetic chamfers on the manifold.

Step 1. Create first of three chamfer features on CRANKSHAFT.PRT.

The third type of direct feature is the chamfer. They are used to smooth potentially sharp edges and for aesthetic value.

The first three of the four chamfers that we create are on the crankshaft.

The initial chamfer we create is on the ends of the crankshaft, where the part interfaces with the engine. This chamfer removes material from the sharp edges.

Even though the chamfer is created as a single set, the two edges are independent of each other.

1. Open CRANKSHAFT.PRT.
2. Start the Chamfer tool. Select the far left circular edge, and then select the far right circular edge of the crankshaft, as shown in the following figure.

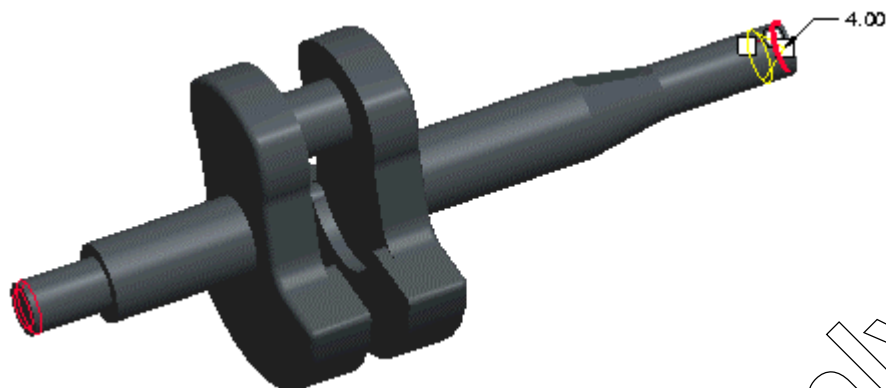


Figure 26: Creating Two Edge Chamfers

3. Drag the distance handles for each chamfer. Note that they are independent of each other.
4. In the dashboard, select the SETS tab. Select **SET2**, right click, and select **Delete**.
5. Press and hold CTRL and select the right circular edge. Drag the distance handles for the chamfer. Note that the chamfers belong to the same set.
6. Using the dashboard, edit the chamfer distance to 1.
7. Complete the feature.

Note:

Selecting references *without* the CTRL key creates independent sets.
Selecting *with* the CTRL key adds/removes references to the same set.

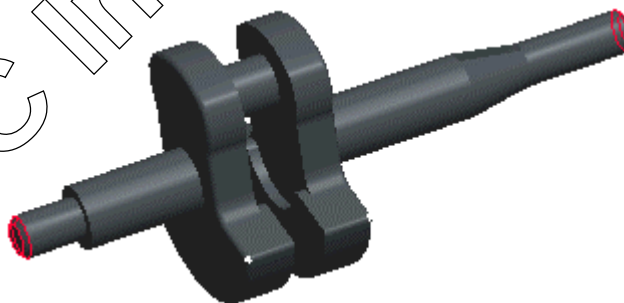


Figure 27: Completed D1xD1 Edge Chamfer

Step 2. Create second of three chamfer features on CRANKSHAFT.PRT.

The second chamfer is on the sharp edge on the short side of the crankshaft. It also removes material. We create the chamfer and test a variety of dimensioning methods before selecting an angular dimension with a single distance scheme (Angle x D).

1. Select the edge shown in the following figure.

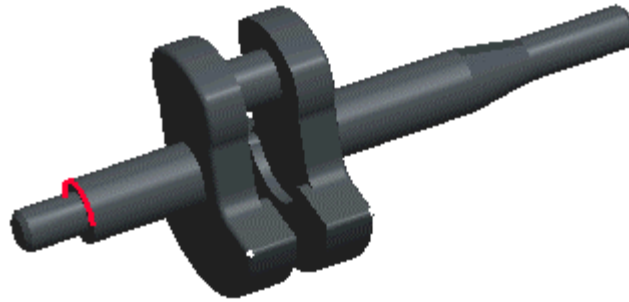


Figure 28: Creating a Second Edge Chamfer

2. Start the Chamfer tool. Using the dashboard, select the **D1 x D2** dimension scheme. Drag the distance handles, as shown in the following figure.

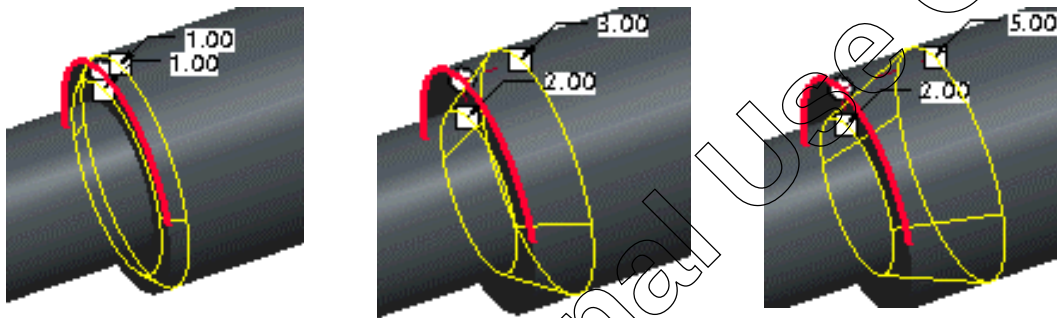


Figure 29: Edit the D1 x D2 chamfer.

3. Select the **Angle x D** dimension scheme. Using the graphics window, double-click on the angle dimension, and enter **15**. Edit the distance to **3**.
4. Complete the feature.

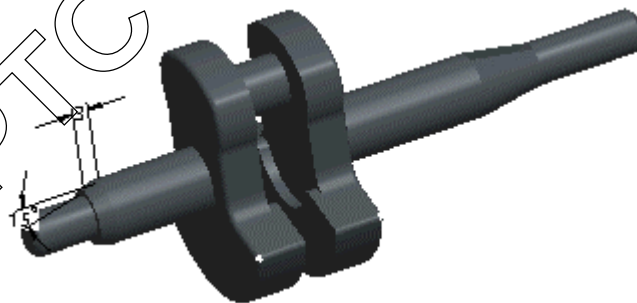


Figure 30: Completed Angle x D Edge Chamfer

5. Save the model.

Step 3. (OPTIONAL) Create third of three chamfer features on CRANKSHAFT.PRT.

Instructor: Skip this step for the demo.

1. Create a chamfer on the edges, as shown in the following figure, making both edges part of the same set. Edit the chamfer distance to 2.

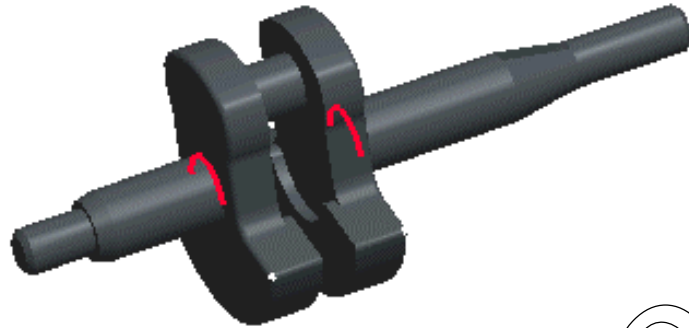


Figure 31: Adding Material by Creating a D x D Edge Chamfer

2. Complete the feature.

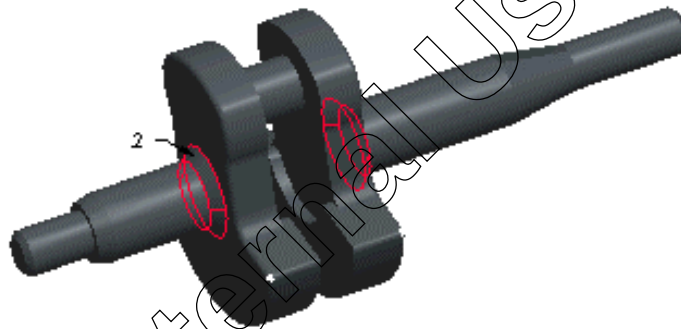


Figure 32: The Completed CRANKSHAFT.PRT

3. Save the model.

Step 4. (OPTIONAL) Create a chamfer on MANIFOLD.PRT.

Instructor: Skip this step for the demo.

1. Open MANIFOLD.PRT.
2. Select the tangent chain of edges, as shown in the following figure.

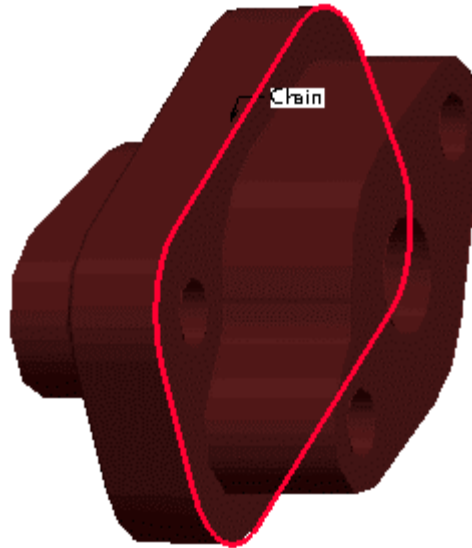


Figure 33: The Tangent Edge Chain

3. Start the Chamfer tool and edit the chamfer distance to **0.5**.
4. Select the edge shown in the following figure to create a second set. Edit the chamfer distance to **1**.

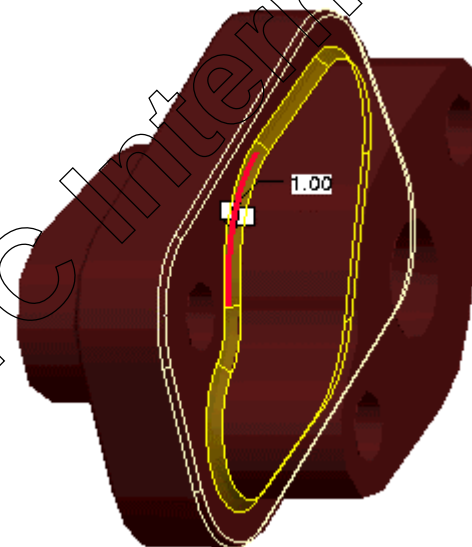


Figure 34: The Second Tangent Edge

5. Complete the feature.

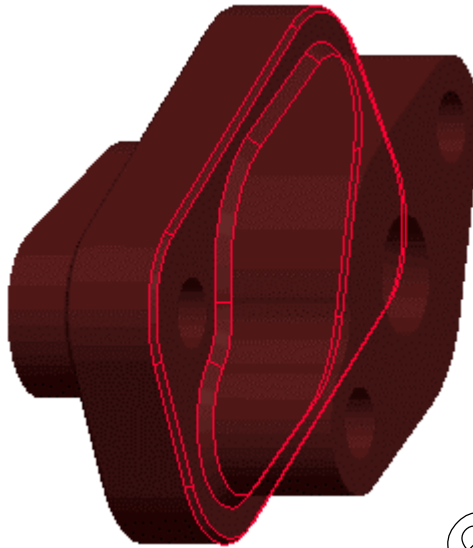


Figure 35: The Completed MANIFOLD.PRT

6. Save the model.
7. Close all windows.
8. Erase all models from memory.

This completes the exercise.

Exercise 4: Creating Direct Features – Drafts

Introduction

In this demonstration, we create draft features.

Objectives

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

- Create drafts.

Scenario

In order to extract parts from casts or molds, it is often necessary to create drafted edges.

In this exercise, you create drafts on the drill's coil.

Step 1. Edit a draft on COIL.PRT.

The fourth type of direct feature that we create is called a draft feature.

Drafts are useful features to use on parts that are either molded or cast. In this case, we place the drafts on the drill's coil.

1. Open COIL.PRT. Orient to the DRAFT view.
2. Using the model tree, select DRAFT_1. Edit the feature and enter 7 for the angle dimension. Regenerate the model.

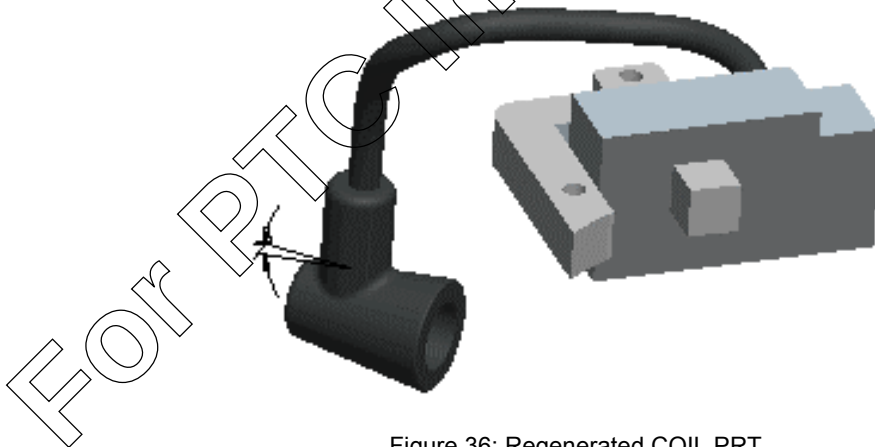


Figure 36: Regenerated COIL.PRT

Step 2. Create a draft on COIL.PRT.

1. Start the Draft tool. Press and hold CTRL and select the two horizontal surfaces, as shown in the following figure. Right-click and select **Draft Hinges**. Select the DRAFT plane as the draft hinge.

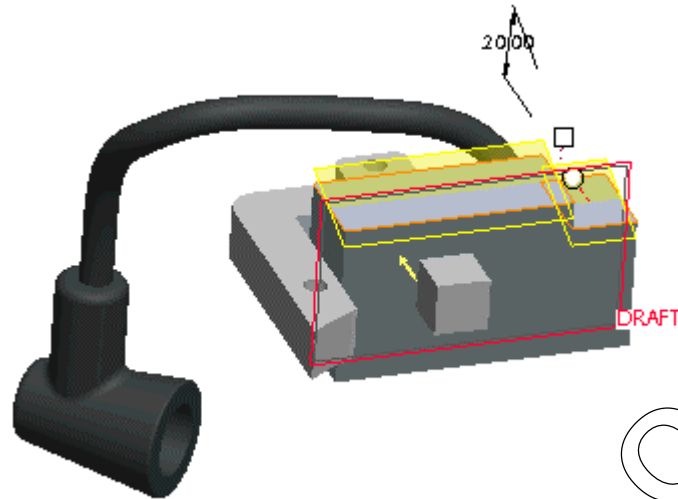


Figure 37: A 20° Draft on Two Surfaces

2. Drag the angle handle upwards, and edit the angle dimension to **20**.
3. Complete the feature.

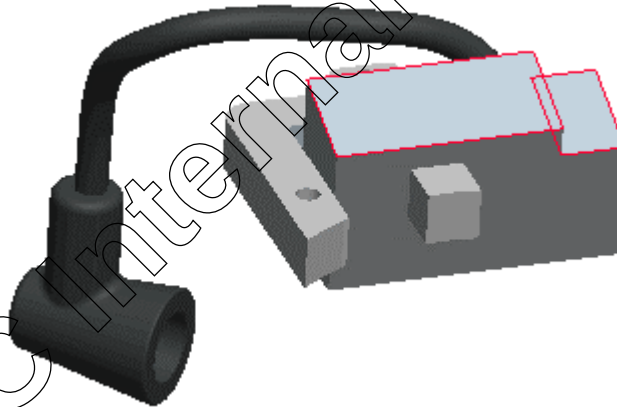


Figure 38: The Completed Draft Feature

4. Save the model.

Step 3. (OPTIONAL) Create a split draft on COIL.PRT.

Instructor: Skip this step for the demo.

1. Suppress the previous draft feature.
2. Start the Draft tool. Select the front surface, as shown in the following figure. To select the Loop Surfaces, press and hold SHIFT, and select an outer edge of the surface.

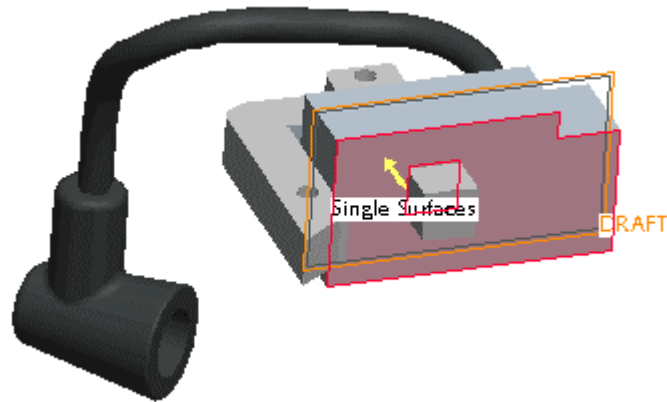


Figure 39: Selecting the Draft Surface

3. Right-click and select **Draft Hinges**. Select the DRAFT plane.
4. Drag the angle handle downward to approximately 5.
5. Using the dashboard, select the REFERENCES tab. Select the **Split by Draft Hinge** option.
6. Drag the angle handles, and edit the angle dimensions to 5 and 10 respectively, as shown in the following figure.

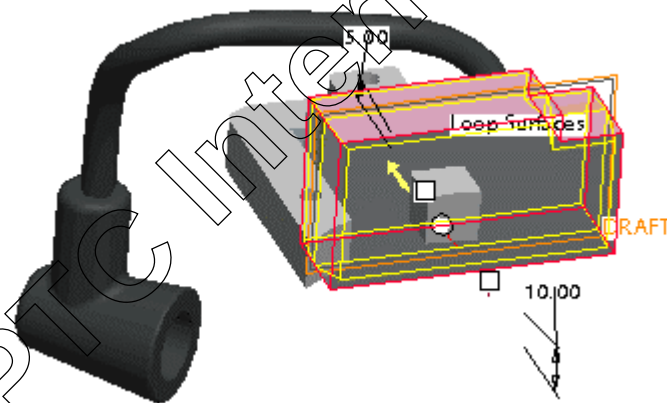


Figure 40: Splitting the Draft at the Hinge

7. Complete the feature.

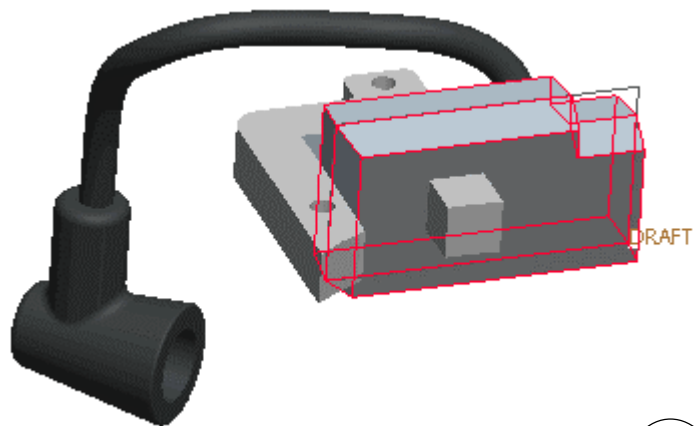


Figure 41: The Completed COIL.PRT

8. Save the model.
9. Close the window.
10. Erase the model from memory.

This completes the exercise.

Exercise 5: Creating Direct Features – Shells

Introduction

In this demonstration, we create a shell feature.

Objectives

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

- Create shells.

Scenario

In order to complete the drill's fuel tank, you hollow out the part and create an opening. A shell feature enables you to easily complete this task. You shell the fuel tank so that the drill will have a location to store gasoline.

Step 1. Create a shell on FUEL_TANK_SHELL.PRT.

Shells allow us to remove the inner material from parts – hollowing the part out. This is a convenient feature for molded parts meant to store liquids or powders.

In this exercise, we shell the fuel tank. First, we select a surface reference to shell. Then, we select all references to create the openings and complete the feature.

1. Open FUEL_TANK_SHELL.PRT.
2. Start the Shell tool. Select the flat circular surface on the filler neck.
3. Edit the thickness to 1, as shown in the following figure. Preview the feature.

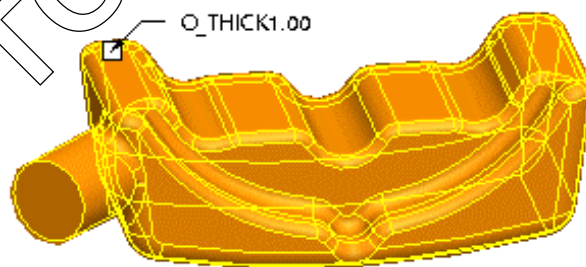


Figure 42: Create the Shell Thickness

4. Resume the Shell tool and edit the thickness to 3.
5. Complete the feature.

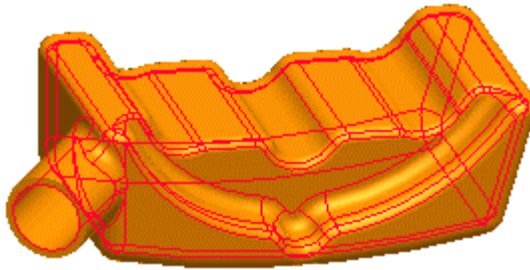


Figure 43: The Completed FUEL_TANK_SHELL.PRT

6. Save the model.
7. Close the window.
8. Erase the model from memory.

This completes the exercise.

For PTC Internal Use Only

Exercise 6: Creating and Managing Layers

Introduction

In this demonstration, we create layers in order to simplify the displayed information.

Layers allow us to quickly remove information from the graphics window, such as features, components, and references.

For example, we can blank all datum points used to create the trajectory of a curve, while displaying all datum points that define connections between movable parts.

We can also blank all surfaces that were used as references.

Layers are especially useful when working with complicated designs because they simplify the model's appearance.

Objectives

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

- Create layers containing direct features.

Scenario

You create layers at the part level in order to organize direct feature geometry. Note that by creating this layer, you can easily select all features within the layer.

Step 1. Create a layer that contains rounds in GEARBOX_FRONT.PRT.

In this exercise, we create a layer containing all of the rounds on the front gearbox. We will create the layer using a rule that will add all rounds to the layer, and add any additional rounds to the layer once they are created. A coworker could easily use this layer to select all rounds on the gearbox.

1. Open GEARBOX_FRONT_LAYER.PRT.
2. Using the model tree, select **Round id 8947**, right-click and select **Suppress**.
3. Click **Settings > Tree Filters** to view the suppressed features in the model tree.
4. Select the **Suppressed Objects** check box.
5. Select and resume the round feature.
6. Use the layer tree to create a layer named **ROUNDS**.
7. Select the RULES tab, and click **Edit Rules**. Using the Search tool, find all the round features. Complete the search and the layer.
8. Select the ROUNDS layer, right-click, and **Select Items**. Cursor over the graphics window, right-click and select **Suppress > OK**.

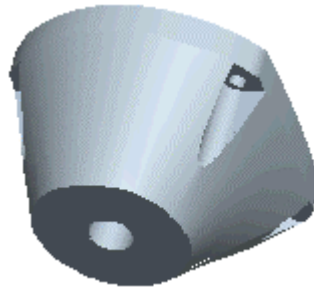


Figure 44: All rounds suppressed

9. Resume all round features.
10. Save the model.
11. Close the window.
12. Erase the model from memory.

This completes the exercise.

Please continue with the Pro/FICIENCY Daily Assessment Checks, and if time allows, please continue on with the challenge exercises.

Exercise 7: Creating Direct Features on the Chuck (Challenge)

Objective

This exercise demonstrates radial holes placed on a cylindrical surface. It is a challenge exercise. The workflow used to create radial holes is nearly identical to that of linear holes demonstrated in the previous exercises.

In addition, you create two chamfers with uncommon dimension schemes.

Scenario

Add additional hole and chamfer features to the chuck part. Create the chuck-key hole, and then chamfer the hole to enable an easier key insertion. Then, create a chamfer on the bottom of the chuck to aid in clearance.

Step 1. Create the radial hole on CHUCK.PRT.

1. Open CHUCK.PRT.
2. Using the **Radial** option, create a hole on the surface, as shown in the following figure.
 - Distance from FRONT plane = **52.50mm**
 - Angle from RIGHT plane = **60.00°**
 - Diameter = **5.50mm**
 - Depth = **8.0mm**

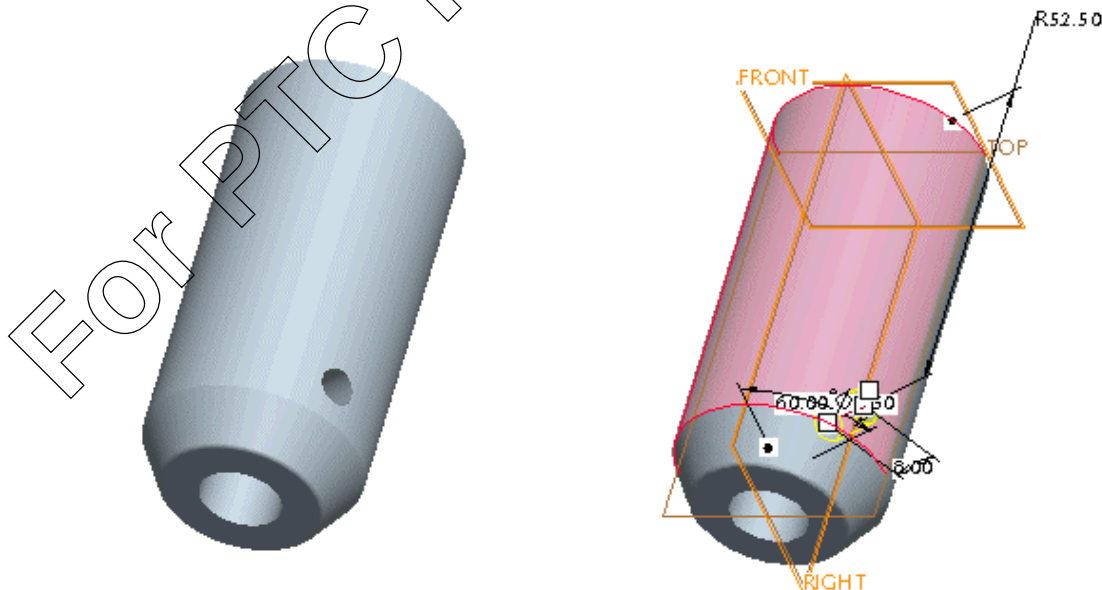


Figure 45: CHUCK.PRT with Radial Hole and Dimensions

Step 2. Create chamfers on CHUCK.PRT.

1. Using the **Angle x D** option, create the chamfer, as shown in the following figure. Edit the angle to **10** and the distance to **5**.

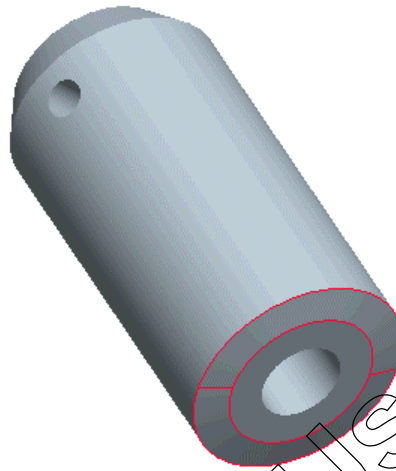


Figure 46: The End Chamfer

2. Using the **D1 x D2** option, create the chamfer, as shown in the following figure. Edit the distances to **1.4** and **0.8** respectively.

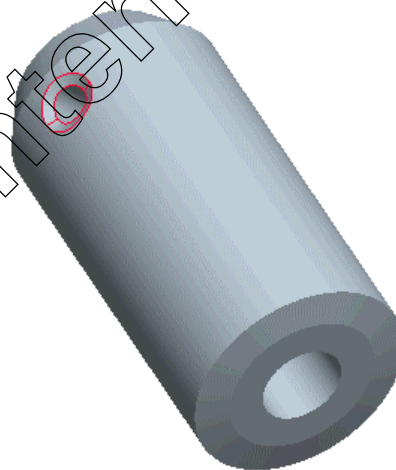


Figure 47: The Hole Chamfer

3. Save the model.

This completes the challenge exercise.

Exercise 8: Creating Direct Features on the Flywheel (Challenge)

Objective

This challenge exercise demonstrates radial holes placed on the surface of the flywheel, and dimensioned relative to a center axis. You also experiment with options for STD holes.

A series of rounds are created in the proper order to provide tangency for the next round in the series. Additionally, chamfer transitions are explored.

Scenario

Create additional hole, round, and chamfer features on the flywheel. Create a small hole for mounting a different type of ratchet in the leaf-blower engine assembly. Create a countersink hole to balance the flywheel as it is rotated. Remove remaining sharp edges with rounds and chamfers.

Step 1. Create two radial holes on FLYWHEEL.PRT.

1. Open FLYWHEEL.PRT.
2. Using the options for **Radial** and **Standard**, create a hole, as shown in the following figure
 - Thread size = **M3x.5**
 - Depth = **Thru All**
 - Radius from CRANK axis = **20.0mm**
 - Offset from FRONT plane = **15.0°**

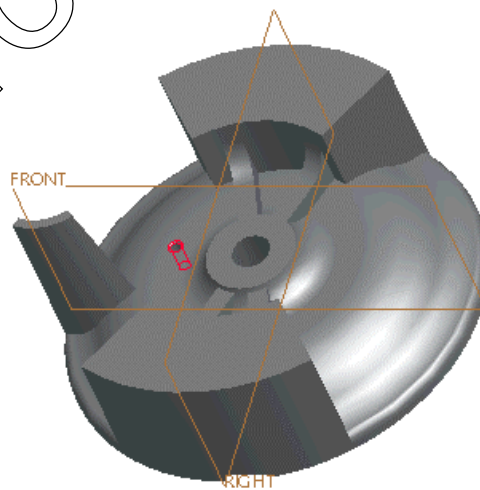


Figure 48: Initial Radial Hole

3. Click **Tools > Environment**, clear the 3D NOTES check box, and continue.

4. Using the options for **Radial** and **Standard**, create the hole, as shown in the following figure.

- Thread size = **M5x.8**
- Depth = **12.6mm**
- Radius from CRANK axis = **35.0mm**
- Offset from RIGHT plane = **23.0°**

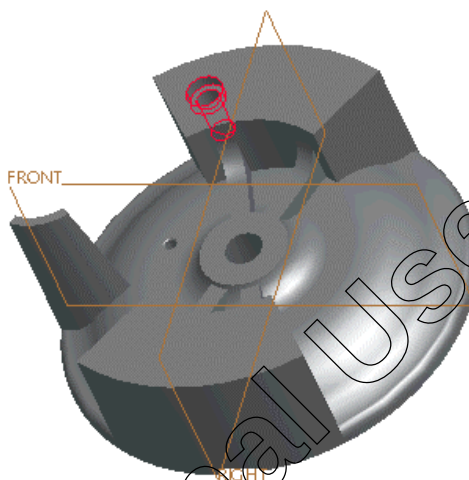


Figure 49: Second Radial Hole

Step 2. Create three rounds on FLYWHEEL.PRT.

1. Create a round with a radius of 2, as shown in the following figure.

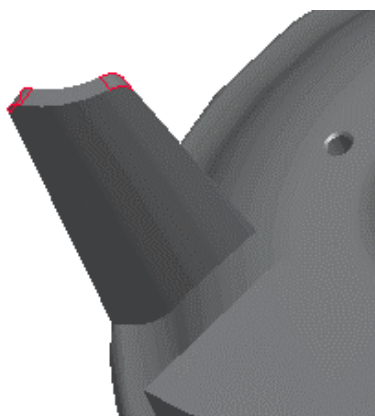


Figure 50: Two Edge Rounds

2. Using the **Full Round** option, create a round referencing two edges, as shown in the following figure.

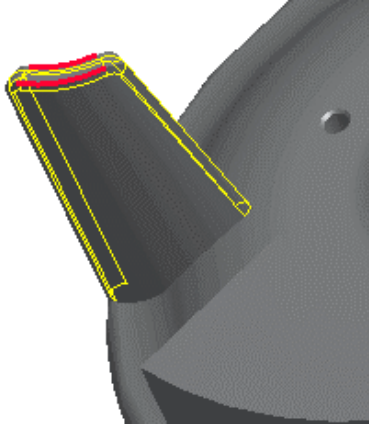


Figure 51: Full Round on Tangent Edge Chains

3. Create a round referencing two surfaces with a radius of **1.5**, as shown in the following figure.

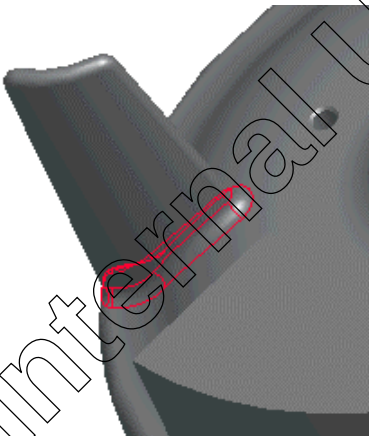


Figure 52: Surface-to-Surface Round

Step 3. Create a transitional chamfer on the on FLYWHEEL.PRT.

1. Create a chamfer with a single set, as shown in the following figure.
2. Use the **Corner Plane** option, edit the distance to **1.25**, and specify both transitions.

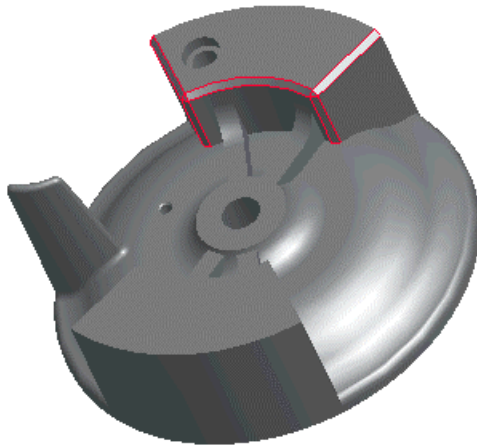


Figure 53: Single Dimension, Two-set Chamfer

3. Save the model.

This completes the challenge exercise.

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Exercise 9: Shelling the Engine Cover (Challenge)

Objective

This challenge exercise enables you to experiment with removing multiple surfaces with a shell feature.

Scenario

Shell the engine cover. Notice the part was created as a shelled solid. This is the easiest and most efficient method to create the part's geometry.

Step 1. Create a shell on ENGINE_COVER_SHELL.PRT.

1. Open ENGINE_COVER_SHELL.PRT.
2. Create a shell, selecting four surfaces to be removed. Edit the thickness to **2**, as shown in the following figure.

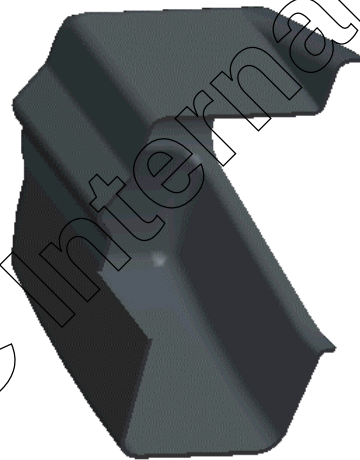


Figure 54: Shelling ENGINE_COVER_SHELL.PRT

3. Save the model.

This completes the challenge exercise.

Exercise 10: Creating Drafts on the Cylinder (Challenge)

Objective

This challenge exercise enables you to experiment with creating additional drafts.

Drafts can be created quickly, since the surfaces to be drafted are tangent and can be easily selected.

Scenario

Create drafts on the fuel intake and exhaust features. The drafts improve the flow of the fuel and exhaust fumes.

Step 1. Create more drafts on CYLINDER_DRAFT.PRT.

1. Open CYLINDER_DRAFT.PRT.
2. Create a draft on the surfaces shown in the following figure. The outer mating surface should remain the same size, and the surfaces should taper inward. Edit the angle to 3.

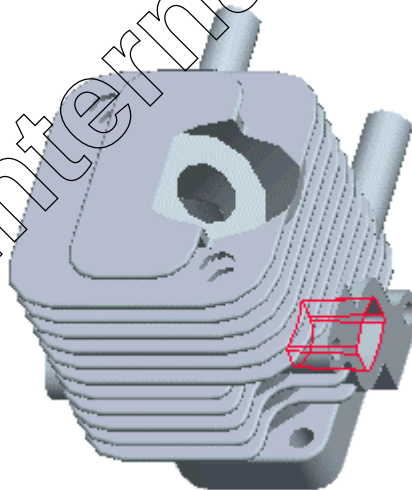


Figure 55: Initial Draft Feature

3. Create a second draft using the same procedure, as shown in the following figure.

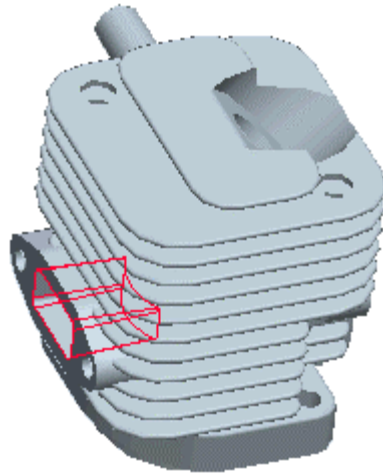


Figure 56: Second Draft Feature

4. Using the model tree, reorder the features below the EXH_CUT feature.
5. Save the model.
6. Close all windows.
7. Erase all models from memory.

This completes the challenge exercise.

Please continue with the Pro/FICIENCY Daily Assessment Checks if you have not already completed it.

Summary

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

- Create direct features such as holes, rounds, chamfers, drafts, and shells.
- Describe how creating direct features affect parent/child relationships.
- Create and manage layers when creating direct features.

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Capturing Design Intent with Sketcher

Introduction

There are several ways to create geometric features in a new design model. Sketching datum curves involves selecting the feature's sketching plane and orientation, and then sketching, dimensioning, and constraining the curve.

Along with datum planes and axes, datum curves also serve as geometric references. For example, datum planes, axes, and curves can capture geometric design intent in the form of a skeleton. Skeletons enable you to communicate your design intent.

Objectives

After completing this module, you will be able to:

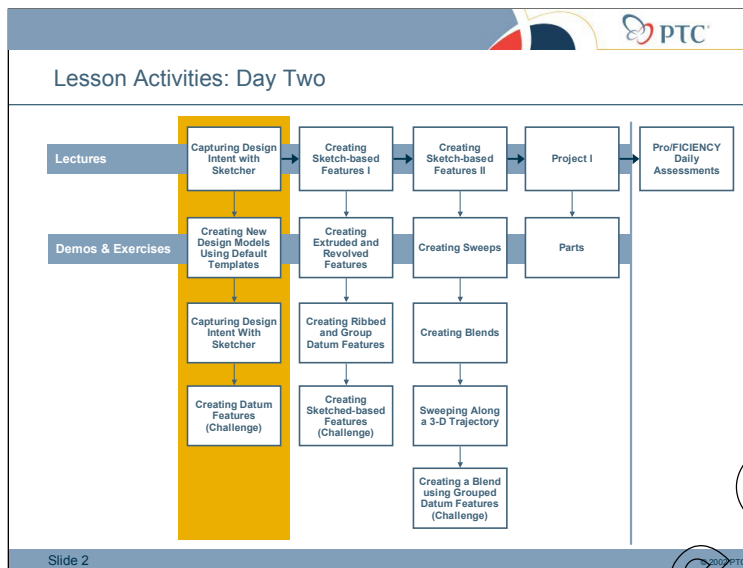
- Start new design models using default templates.
- Capture design intent with Sketcher.
- Create design frameworks with datum features.
- Describe how sketched reference geometry affects parent/child relationships.



Instructor Preparation

Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** 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 Foundation
 - Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 5 mins
- Demos (2): 30 mins
- Labs (2): 55 mins
- Total: 1 hr 30 mins



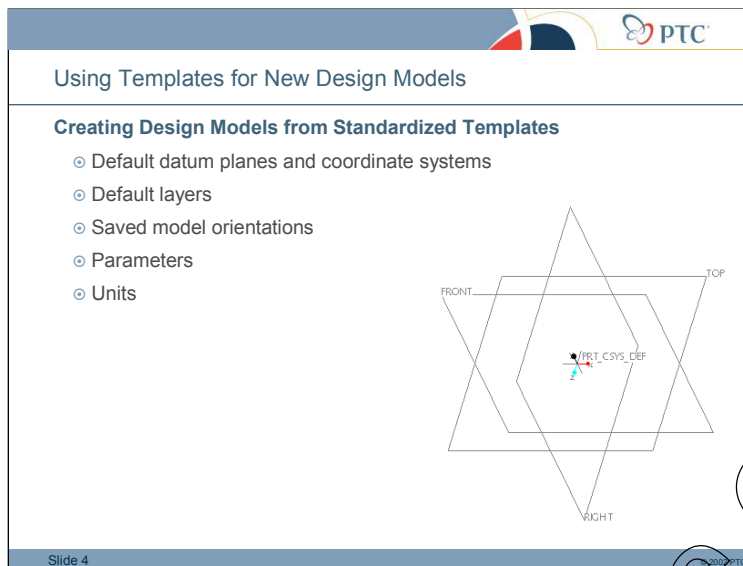
Objectives

After completing this module, you will be able to:

- Start new design models using default templates.
- Capture design intent with Sketcher.
- Create design frameworks with datum features.
- Describe how sketched reference geometry affects parent/child relationships.

Slide 3

- You can create geometric features for new design models by sketching datum curves.
- You select the feature's sketching plane and orientation, sketch the curve, and then dimension and constrain the curve.
- Like datum planes and axes, datum curves provide geometric references.
- For example, datum planes, axes, and curves can capture geometric design intent in the form of a skeleton.
- Skeletons enable you to easily communicate your design intent to other engineers.



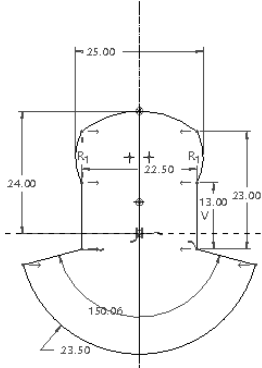
- Default templates are an initial design model or assembly that contain common features, layers, view orientations, parameters, and units.
- Default templates are used when you first create any new part or assembly. You simply select a default template when you start to create a new part.
- By using a standard template, you can eliminate future design problems by using a set corporate standard. This standard template will enable all engineers to have a consistent starting point.
- A standard template ensures that all the necessary elements required by your company will be included in your design.
- For example, if your company requires that each model has a primary engineer assigned, then the default template will have a MODEL_BY parameter to designate the primary design engineer.
- **Benefit:** Start all of your designs for new parts and assembly designs using a standard template customized to your company's needs.

PTC

Sketching Datum Curves

Capturing the Design Intent

- Select a sketching plane.
 - Plane or surface will be made parallel the computer screen.
- Select orientation reference.
 - An additional plane or surface that determines 3-D orientation.
- Select orientation direction.
 - A default orientation is selected based off of the current model orientation.
 - You can manually override the default orientation.



Slide 5

- Design intent ensures predictable results when a model is modified.
- Datum curves serve as references. They can exist separately as their own feature, or as the starting point when you create sketch-based features.
- Datum curves are used when you want to create reference geometry or sketch-based features. Datum curves are created in the Pro/ENGINEER Wildfire sketcher after selecting the sketching plane and the references.
- The sketching plane can be either a datum plane or a flat surface of an existing feature.
- The orientation reference determines the 3-D orientation of the sketch. This reference is also either a datum plane or a flat surface and must be normal to the sketch plane.
- The orientation reference can be manually set to face Top, Bottom, Right, or Left. These directions are named to reflect how the reference will orient with regard to the computer screen. For example, a surface will orient facing left if manually set to Left. Note that when you have completed a sketch, depending on how you chose your reference plane will make the sketch look differently in the Standard Orientation.

PTC

Sketching Datum Curves

Capturing the Design Intent

- Sketch the curve.
 - References
 - Lines
 - Construction lines
- Constrain the curve.
 - Horizontal, vertical, parallel, perpendicular
 - Equal length, equal radii, tangent
 - Midpoint, same point
 - Symmetric about centerline
- Dimension the curve.
 - Weak and strong dimensions
 - Reference dimensions

Slide 6

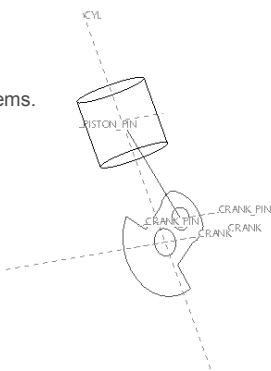
- Design intent ensures predictable results when a model is modified.
- Datum curves allow you to capture design intent.
- Design logic is set by sketching, dimensioning and constraining the curve.
- Constraining the curve is an extremely important means to capture design intent. Note that as you add constraints, you minimize the number of dimensions required to document your design intent.
- When dimensioning a sketch, it is important to create dimensions that capture your design intent because these dimensions are displayed when you edit the model and when you create drawings of the model. Note to instructor: save the discussion of weak versus strong dimensions for the demonstration.
- **Benefit:** Establishes a design intent so that a model can be modified with predictable results.

PTC

Creating Design Frameworks with Datum Features

Communicating Design Intent using Reference Geometry

- Creating datum curves.
- Creating datum planes.
- Creating datum axes.
- Creating datum coordinate systems.
- Creating datum points.
- Creating datum surfaces.



Slide 7

- Skeletons serve as references to new parts and features.
- They exist as their own model, and serve to communicate design intent to others.
- Skeletons are used when beginning a top-down design project.
- You create skeletons like other models and assemble them to one another.
- Later, you reference the skeletons to ensure that all parts are designed and dimensioned so that they assemble correctly the first time with no interferences.
- Skeletons help communicate your design intent to other engineers, so that they will be able to design their parts to fit correctly the first time all parts are assembled.
- **Benefit:** Datum features help to communicate a precise design intent to other engineers in your design projects.

PTC

Parent/Child Relationships – Capturing Design Intent

When capturing design intent with sketcher, parent/child relationships are created. Changes made to a parent feature affects its children.

- Default Datum Features within Part Templates
 - Many model features dependant on default datum planes
- Sketched datum curves
 - Sketch plane and reference plane
 - Sketcher references
 - Dimensions
 - Constraints

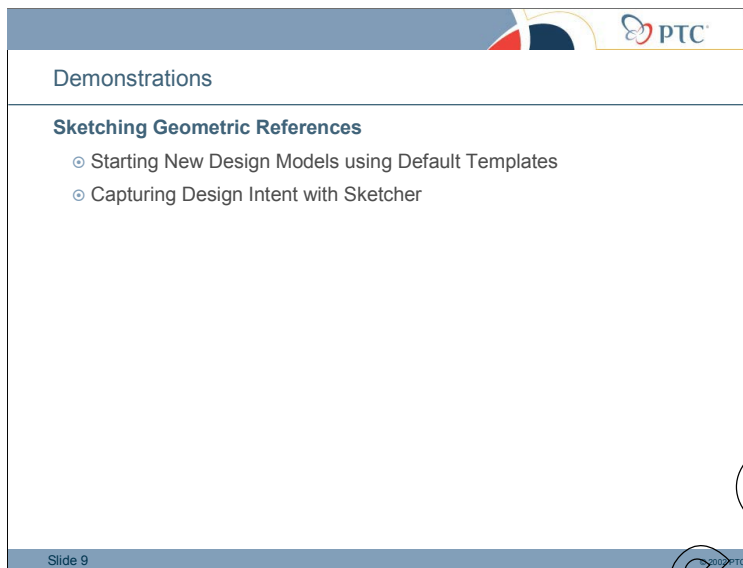
Slide 8

Note to Instructors:

Use this slide to teach students how the functionality that they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

There is no dependency created to the actual part template, however most model features will be dependant on the default datum planes.

All sketched datum curve features are children to their references. For example, sketched datum curves will be children to the planes and surface that they are drawn on and referenced to. Additionally, any constraints to other sketched entities will be a parent feature. For example, constraining the edge of the sketch to make it parallel to a previous sketch based feature will be a parent of the new feature.



You will follow along while I demonstrate:

1. Starting New Design Models using Default Templates
2. Sketching Datum Curves

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Start new design models using default templates.
- Capture design intent with Sketcher.
- Create design frameworks with datum features.
- Describe how sketched reference geometry affects parent/child relationships.

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Referencing PTC Help for Module 5 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Parameters	Fundamentals	About User Parameters
Units	Fundamentals	About Measurement Units To Select a Measurement Units System
Sketched Curves	Part Modeling	About Sketched Datum Curves About Sketched Datum Curves UI
Sketcher References	Part Modeling	About References About Sketcher Mode With Improved Intent Manager
Sketcher Constraints	Part Modeling	About Using Sketcher Constraints Supported Constraints
Dimensions	Part Modeling	About Dimensioning Basics To Create Reference Dimensions

Module 5 Lab Exercises

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 2003100 or later.

Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Creating New Design Models Using Default Templates

Introduction

In this demonstration, we create two parts using the templates provided. Templates, also known as start parts, enable us to assign a part a name and use pre-defined settings and features. For example, it is standard for templates to contain three default datum planes, a datum coordinate system, assigned units, and default layers. Using templates to design new models enables us to set a corporate standard – saving upfront setup time and editing time later in the design process.

Objectives

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

- Create parts from templates.

Scenario

You have been assigned to review the default file templates to ensure that the correct units are being used. In the past, incorrect units have caused costly design changes later in the development process. You create two parts using the templates provided. These part models will be used later in the drill assembly.

Step 1. Start a new design model with the default template.

Before beginning the design of any new part, it is a best practice to create new part models based on the default part template files.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to:
C:\users\student\fast_track_330\module_05.
3. Click **File > New > Part**, and enter **Sketched_Curve_1** as the name. Note that the **Use Default Template** option is enabled.
4. Display all datum features. Select each feature in the model tree to highlight it in the graphics window.

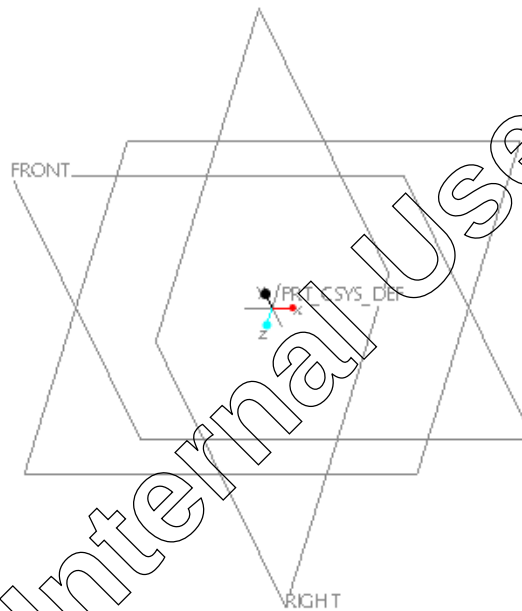


Figure 1: Default Part Features

5. Using the layer tree, review the default layers. Note that the icons for the two DTM_PLN layers are different, and continue.
6. Select the 01_PRT_DEF_DTM_PLN layer, right-click and select **Layer Properties**. Select the RULES tab. Note that no rules exist.
7. Repeat the procedure to view the properties of the 01_PRT_ALL_DTM_PLN layer. Note that rules exist for this layer.
8. Click **Edit > Setup > Units**. Note the active unit system and continue.

Note:

A customized default unit system is used in this course. You can use the Units Manager to select an existing units system, or create your own. You can also set any unit system to be your default using a CONFIG.PRO options file.

9. Click **Tools > Parameters**. Enter **TBD** as the Description value, and enter your name as the Modeled_By value. Complete the parameter editing.
10. Save the model.
11. Close the window.
12. Erase the model from memory.

This completes the exercise.

For PTC Internal Use Only

Exercise 2: Capturing Design Intent with Sketcher

Introduction

In this demonstration, we constrain, dimension, and create datum curves. Datum curves are used as the starting point for features, as well as references. Therefore, it is important to know how to properly constrain and dimension datum curve sketches to ensure predictable results with later modifications. Applying this logic into sketches is also known as documenting your design intent.

Objectives

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

- Create, constrain, and dimension sketched datum curves.

Scenario

Before you begin designing the model features, your manager has asked you to capture the design intent by constraining and dimensioning specific reference curves. It is critical that you learn how to properly constrain, dimension, and create sketched curves. Future product designs will typically re-use previous design information. If your design information cannot be easily re-used, it will result in lost design time and cost overruns.

After you impart the design intent, you sketch additional datum curves for use in skeletons.

Step 1. Capture the design intent by constraining CURVE1.

In this exercise, we focus on capturing our product design intent by using sketcher constraints, dimensions, and curves.

A properly designed model will modify in a predictable way so that new variations of the product can be made more quickly. For example, more cooling fins can be added to the cylinder or flywheel simply by changing a single dimension.

In this initial step, we focus on using constraints to capture design intent on sketched datum curves.

Constraints enable us to create perpendicular, parallel, horizontal, and vertical sketch entities, as well as set entities to have the same length, radius, or alignments.

Besides capturing design intent, constraints also enable us to eliminate unneeded dimensions.

1. Open SKETCHER_CONSTRAINTS.PRT.
2. Using the model tree, select the CURVE1 feature, right-click and select **Edit Definition > Sketch**.
3. Using the toolbar, toggle off dimensions.
4. Impose sketcher constraints, and select the Equal constraint. Select the two circles.

5. Select the Horizontal constraint, and select the two circle centers. Complete the constraints.
6. Select and drag the circle centers and radii to test the constraints.
7. Impose sketcher constraints, and select the Same Point constraint. Select a circle center and the horizontal sketcher reference to align the circles, as shown in the following figure.

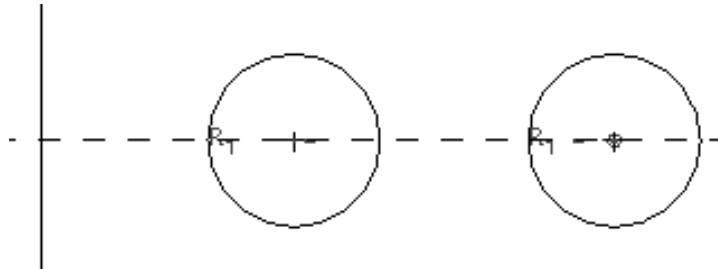


Figure 2: Aligning and Constraining the Circles

Note:

The **Undo** and **Redo** commands are located on the toolbar.

8. Complete the sketch and the feature.

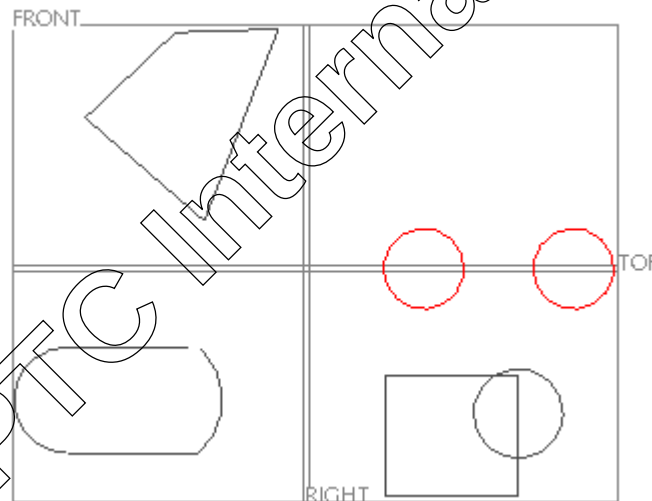


Figure 3: The Completed CURVE1 Feature

Step 2. Capture the design intent by constraining CURVE2.

Now we will constrain a second sketch.

1. Using the model tree, select the CURVE2 feature, right-click and select **Edit Definition > Sketch**.
2. Impose sketcher constraints, and select the Midpoint constraint. Select the circle center and the adjacent vertical line.

3. Use dynamic trim to remove the intersecting entities.
4. Drag the rectangle, as shown in the following figure.

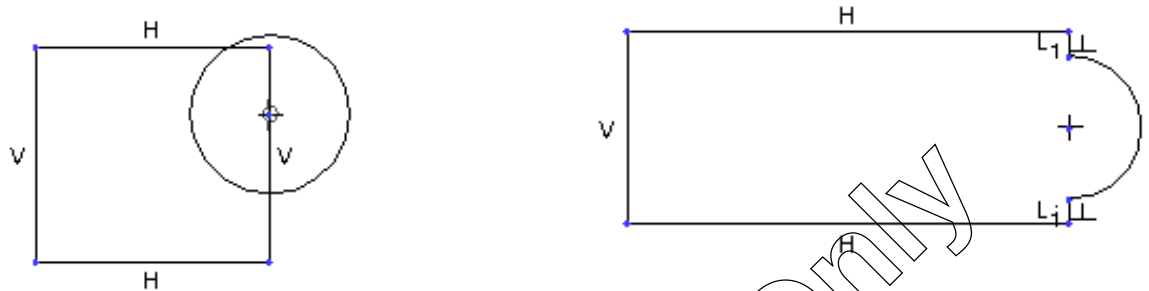


Figure 4: Constraining, Trimming and Dragging CURVE2

5. Complete the sketch and the feature.

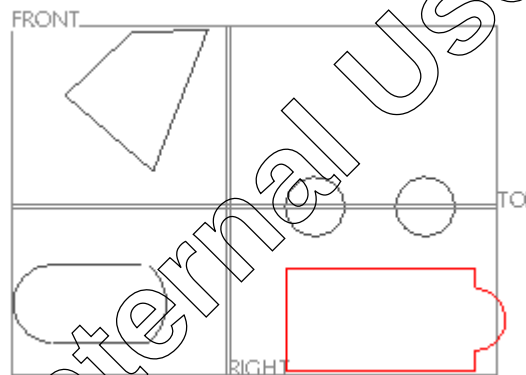


Figure 5: The Completed CURVE2 Feature

Step 3. Capture the design intent by constraining CURVE3.

Instructor: Skip this step for the demo.

1. Select CURVE3, and edit the definition of its sketch.
2. Impose sketcher constraints, and constrain the two lines near the upper right vertex to be perpendicular.
3. Constrain the top and bottom lines to be both parallel and equal. Constrain the leftmost line to be vertical.
4. Constrain lines to lie on the same point as the horizontal and vertical sketcher references, as shown in the following figure.

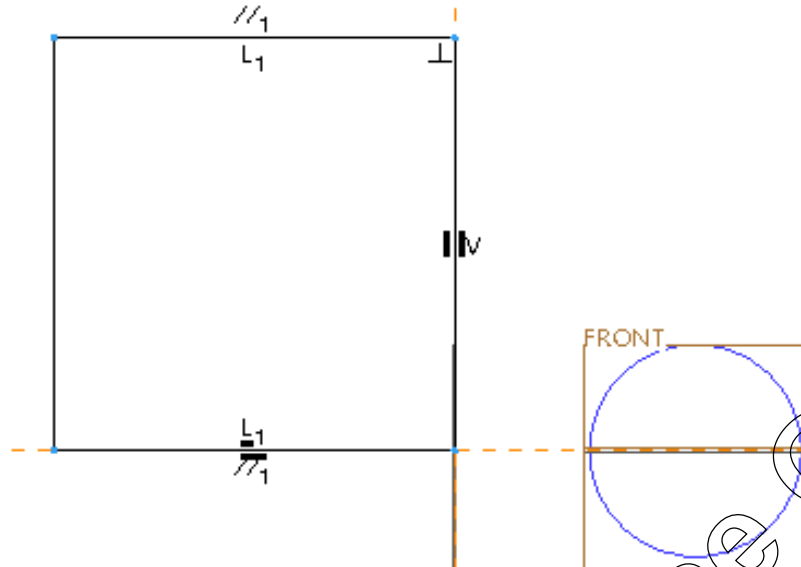


Figure 6: Editing and Constraining CURVE3

5. Complete the sketch and the feature.

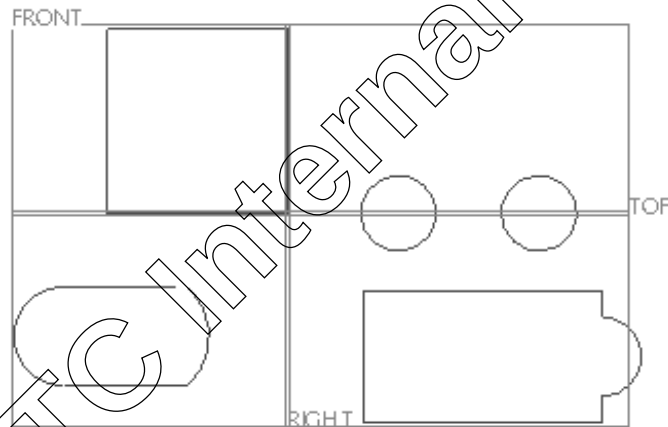


Figure 7: The Completed CURVE3 Feature

Step 4. Capture the design intent by constraining CURVE4.

Now we will constrain another sketch.

1. Select CURVE4, and edit the definition of its sketch.
2. Impose sketcher constraints, and constrain the points near the 'gap' to lie on the same point.
3. Constrain the right side arc to be tangent to the upper line.
4. Constrain the arc centers to be symmetric about the centerline. Drag the centerline to view its effect, as shown in the following figure.

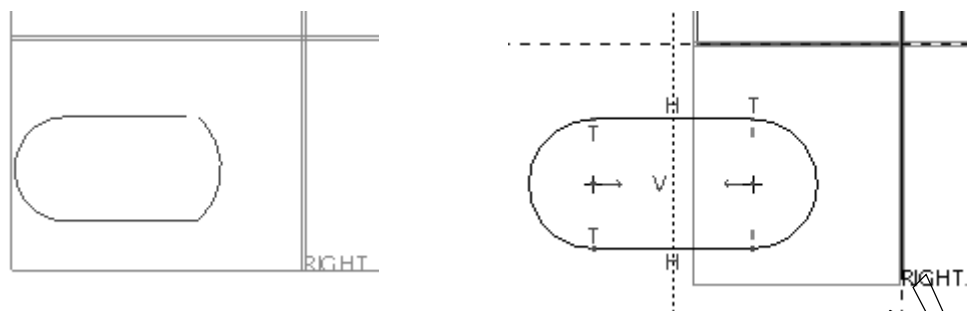


Figure 8: Editing and Constraining CURVE4

5. Complete the sketch and the feature.

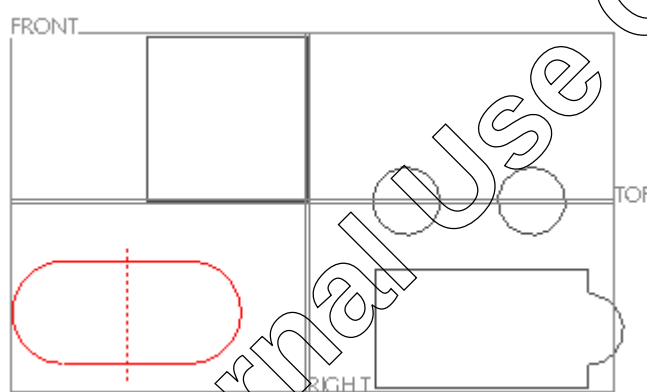


Figure 9: The Completed CURVE4 Feature

6. Save the model.

Step 5. Capture the design intent by dimensioning sketched entities on SKETCHER_DIM_MOD1.PRT.

Next, we document design intent by dimensioning sketched entities. Dimensioning enables us to set numerical values to our sketches so that the sketch is parametric.

Parametric sketches enable us to change the dimension, which will then predictably update the feature.

For example, we have a block with a hole in the upper right corner. If we change the block to double width, does the hole stay in the far right corner or is it now in the center of the block? To properly document design intent, we dimension the hole so that it remains in the upper right corner.

In the initial part of this step, we create a new dimensioning scheme for our curved feature. Then we examine how we can change dimensions by using the mouse wheel.

1. Open SKETCHER_DIM_MOD1.PRT.

2. Select the curve and edit the definition of its sketch.
3. Using the toolbar, toggle on dimensions, and toggle off constraints.
4. Create the dimensions shown in the following figure, and then double-click to edit their values. Do not leave any weak dimensions.

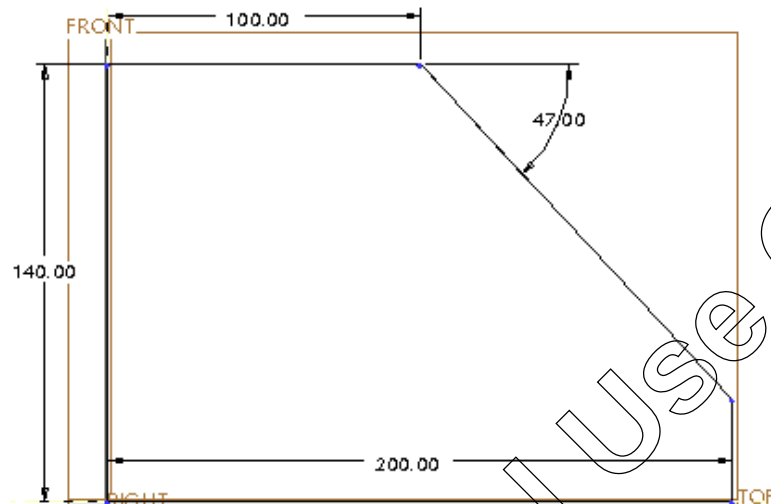


Figure 10: The Modified SKETCHER_DIM_MOD1.PRT

5. Select the 47 dimension, and then use the toolbar to modify its value. Scroll its value, and then enter **30**.
6. Select the 140 dimension, scroll its value, and then enter **120**, as shown in the following figure. Complete the modification.

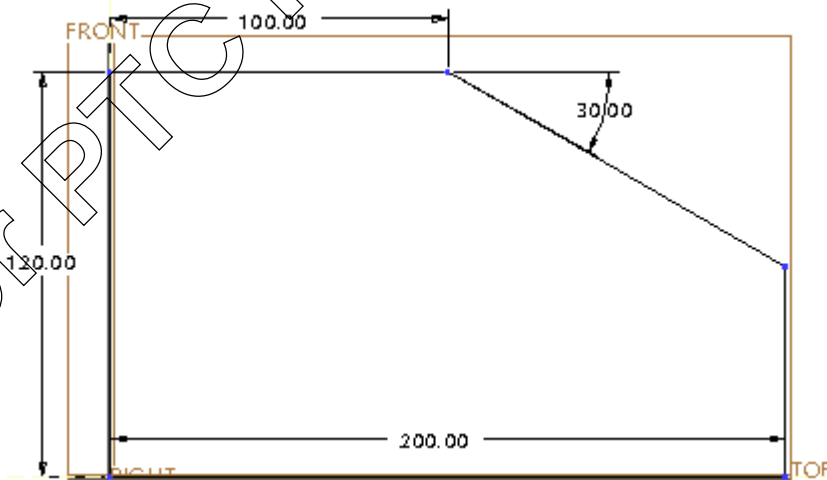


Figure 11: Editing SKETCHER_DIM_MOD1.PRT

7. Complete the sketch and the feature.
8. Save the model.

Step 6. Capture the design intent by dimensioning sketched entities on SKETCHER_DIM_MOD2.PRT.

In the second sketch, we examine how scaling the sketched curve rapidly modifies dimensions. We mirror the entities in order to complete the sketched curve.

1. Open SKETCHER_DIM_MOD2.PRT.

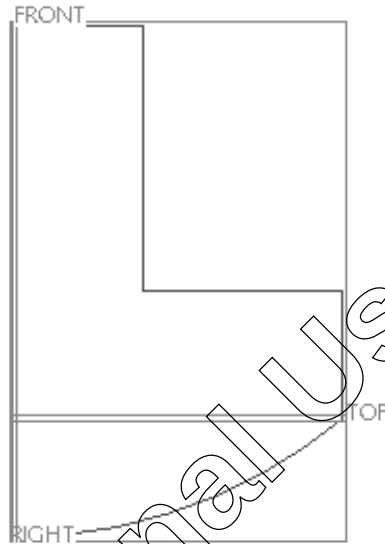


Figure 12: SKETCHER_DIM_MOD2.PRT

2. Select the curve and edit the definition of its sketch.
3. Using the toolbar, toggle on constraints if necessary.
4. Create dimensions, as shown in the following figure. Ignore the dimensional values for now.

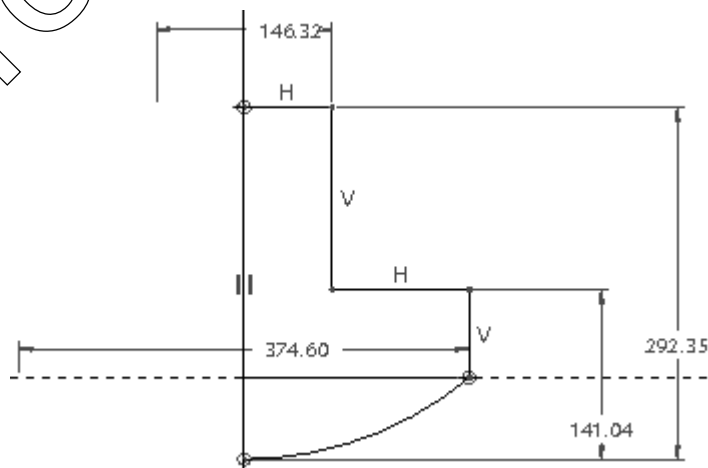


Figure 13: Editing SKETCHER_DIM_MOD2.PRT

5. Drag a window to select all dimensions. Use the toolbar to modify their value, and select the **Lock Scale** option. Select the major vertical dimension (292.35), and enter **16**. Complete the modification, as shown in the following figure.

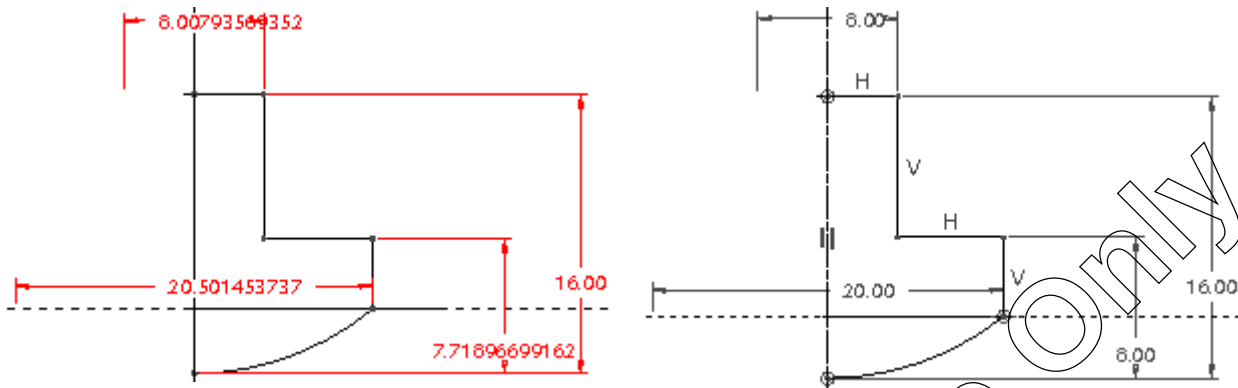


Figure 14: Modifying the Dimensional Values

6. Drag a window to select all the sketched geometry. Mirror the sketched entities, and select the vertical centerline, as shown in the following figure. Recreate dimensions as necessary.

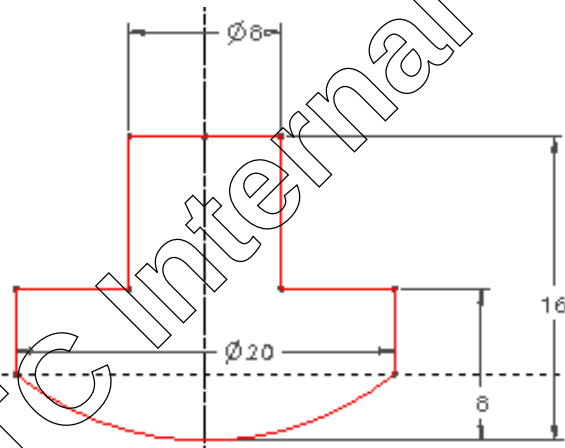


Figure 15: Mirroring the Sketched Entities

7. Complete the sketch and the feature.
8. Save the model.

Step 7. Capture the design intent by dimensioning sketched entities on SKETCHER_DIM_MOD3.PRT.

Instructor: Skip this step for the demo.

1. Open SKETCHER_DIM_MOD3.PRT.
2. Select the curve and edit the definition of its sketch.
3. Create dimensions, as shown in the following figure. Ignore the dimensional values for now.

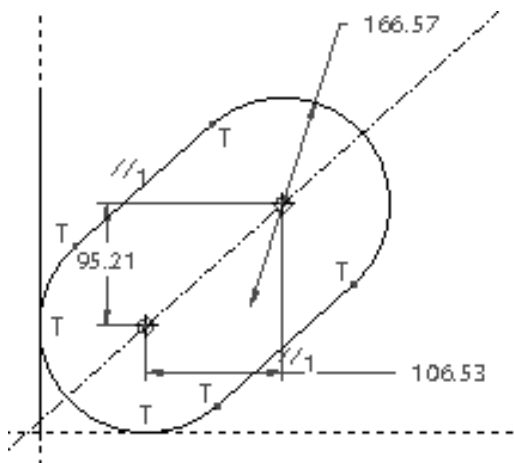


Figure 16: Dimensioning the Curve

4. Create dimensions, as shown in the following figure. Delete any conflicting horizontal and vertical dimensions, and ignore the dimensional values for now.

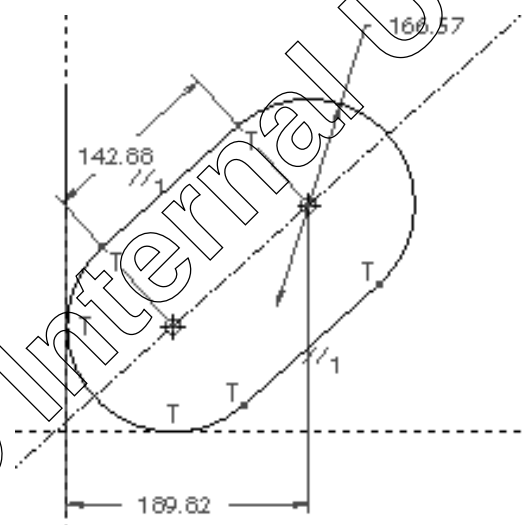


Figure 17: Changing the Curve Dimensioning Scheme

5. Drag a window to select all dimensions. Use the toolbar to modify their value, and clear the **Regenerate** check box. Modify the dimension values, and complete the modification, as shown in the following figure.

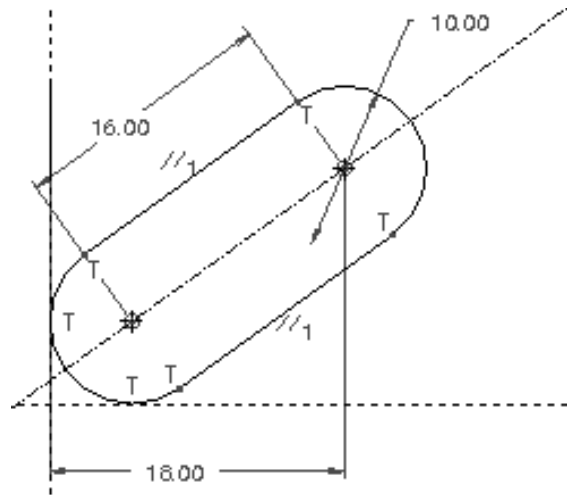


Figure 18: Editing and Regenerating the Dimensional Values

6. Complete the sketch and the feature.
7. Save the model.

Step 8. Create the initial sketched datum curves on SKETCHED_CURVE_1.PRT.

Now that we have constrained and dimensioned the sketched curves, we can create the sketches. The initial two curves that we create provide the base feature. The final curve that we create on the surface of the protrusion will be used as a reference for a future crank connection.

1. Open SKETCHED_CURVE_1.PRT that was created in the previous exercise.
2. Start the Sketched Datum Curve tool. Select datum FRONT and click **Sketch**.
3. Accept the default references and continue.
4. Sketch horizontal and vertical centerlines snapping to the references.
5. Sketch a rectangle, allowing it to snap symmetrically about the centerlines. Ignore the dimensional values for now.

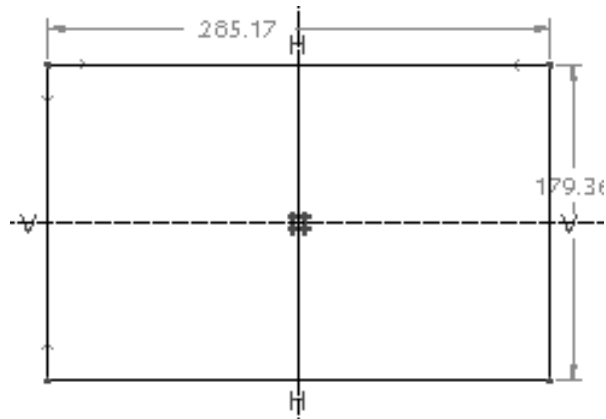


Figure 19: Sketching a Datum Curve

6. Create two circular fillets, and constrain them to be equal. Ignore the dimensional values for now.

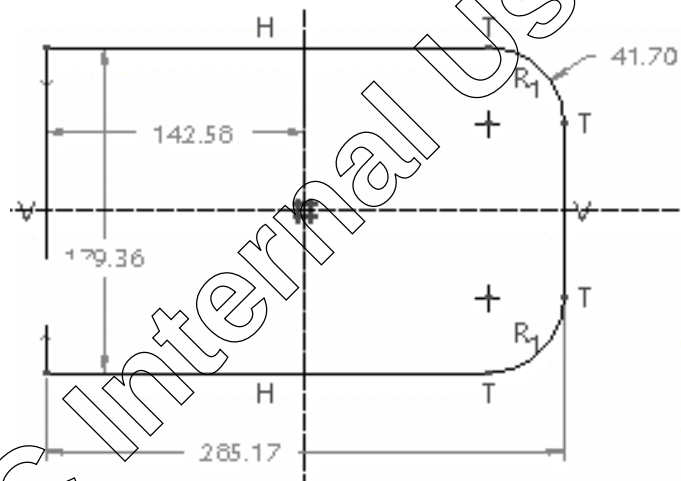


Figure 20: Editing the Datum Curve

7. Create sketcher points where the vertical lines intersect the horizontal reference. Create a symmetry constraint between the sketcher points and the vertical centerline.
8. Use **Lock Scale** to modify the dimension values. Then edit the values individually as shown in the following figure.

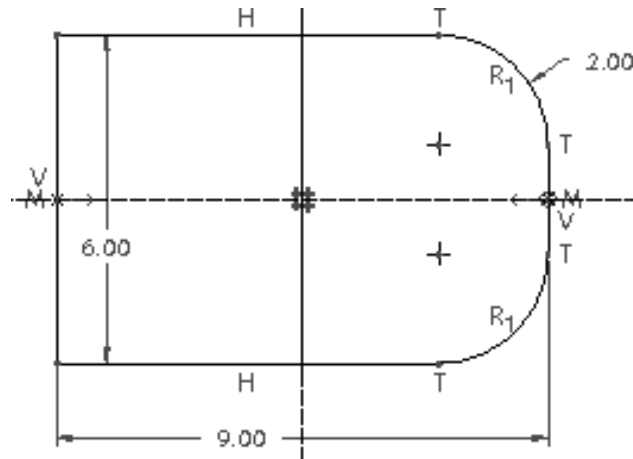


Figure 21: Adding and Constraining the Sketcher Points

9. Complete the sketch and the feature.

10. Save the model.

Step 9. Create a second sketched datum curve.

Instructor: Skip this step for the demo.

1. Start the Sketched Datum Curve tool, and select datum FRONT. Select **Bottom** as the sketch orientation for the RIGHT plane. Click **Sketch** and accept the default references.
2. Note the difference in orientation from the previous sketch.
3. Sketch vertical and horizontal centerlines.
4. Sketch a line, an arc, a line, and an arc in sequence, as shown in the following figure.

Note:

To quickly create this sketch, use the right-click menu.

5. Constrain the arc centers to be symmetric about the horizontal centerline, and then dimension, as shown in the following figure.

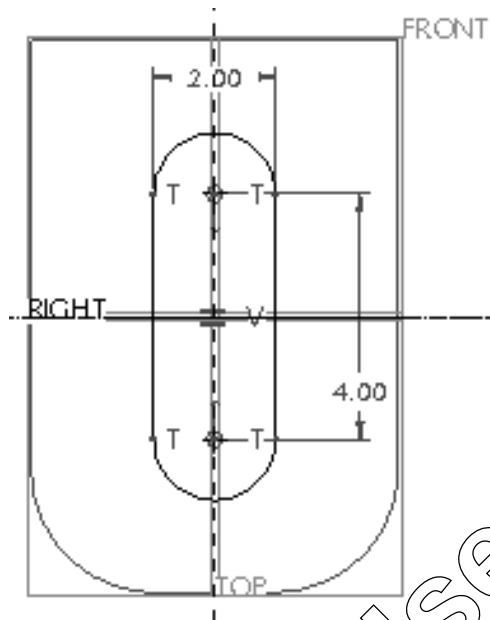


Figure 22: Creating a Second Datum Curve

6. Complete the sketch and the feature.

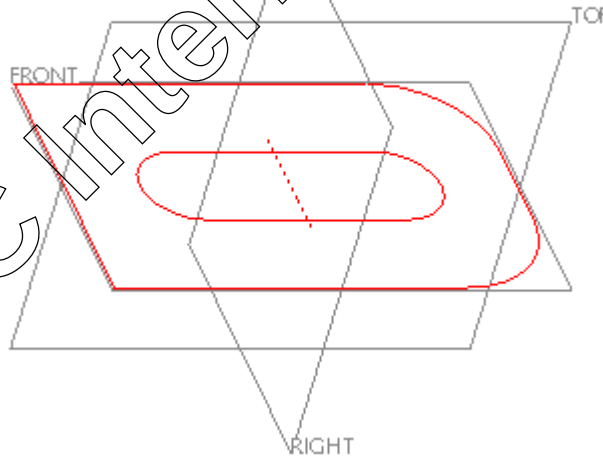


Figure 23: The Completed Datum Curves

7. Save the model.

Step 10. Create the sketched datum curves on an existing model.

In this example, we will create a sketched curve on an existing model.

1. Open SKETCHED_CURVE_2.PRT.

2. Select the front surface of the protrusion, start the sketched datum curve tool, and click **Sketch**.
3. Select the sloped surface and axis A_2 as references.

Note:

References that have been specified in sketcher but that are not used are automatically deleted by Pro/ENGINEER Wildfire once you have completed the sketch.

4. Sketch a line starting from the axis, allowing it to be parallel to the sloped reference. Right-click and select **Construction**. Sketch a circle and dimension, as shown in the following figure.

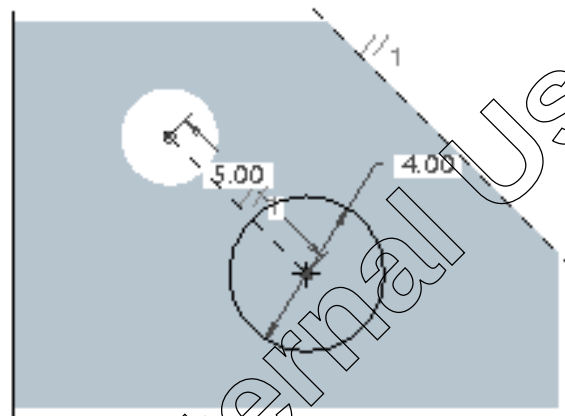


Figure 24: Creating a Datum Curve on the Protrusion

5. Using the toolbar to create entities from edges, select the edges of the hole.
6. Sketch lines using the 2-tangent option. Use dynamic trim to remove unwanted entities, as shown in the following figure.

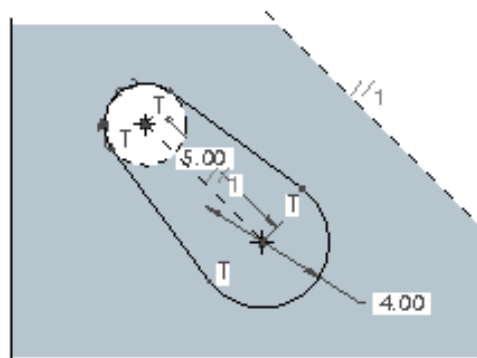


Figure 25: Edit the Datum Curve Using Edges and Trimming

7. Sketch a sketcher point and dimension, as shown in the following figure. Confirm that the sketcher point is constrained to the arc.

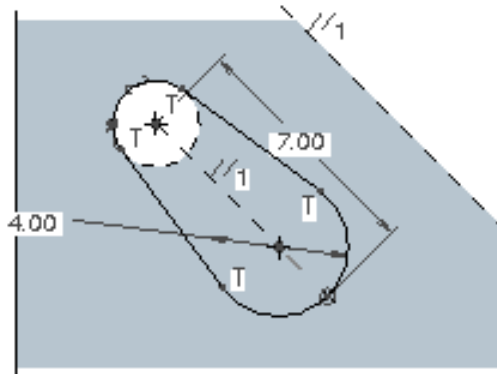


Figure 26: Dimensioning the Sketch

8. Complete the feature.

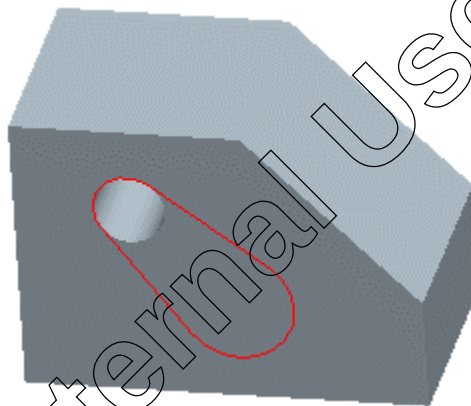


Figure 27: The Completed SKETCHED_CURVE_2.PRT

9. Save the model.
10. Close all windows.
11. Erase all models from memory.

This completes the exercise.

Exercise 3: Creating Datum Features (Challenge)

Objective

In this challenge exercise, you create sketched curves and other reference datum features to form a skeleton of a part. Skeletons are used to document conceptual designs using simple geometry to represent features.

Scenario

Your manager has asked you to create the skeleton of the crankshaft. You must create the skeleton so that others may reference your design later to create the solid part.

Step 1. Create a skeleton of the CRANKSHAFT.PRT.

1. Create a new part named CRANKSHAFT.PRT using the default template.
2. Select the RIGHT and TOP planes, and create an axis named **CRANK** to represent the crank, as shown in the following figure.

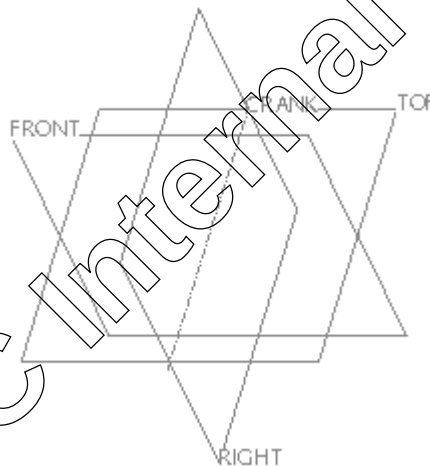


Figure 28: Creating CRANKSHAFT.PRT

3. Create a datum plane offset **14** from the TOP plane named **OFFSET**.
4. Select the OFFSET and RIGHT planes, and create an axis named **CRANK_PIN** to represent the crankshaft pin.
5. Create a datum plane offset **12.5** from the FRONT plane named LOBE_FRONT.
6. Create a datum plane offset **-12.5** from the FRONT plane named LOBE_REAR.
7. Create a sketched datum curve on the FRONT plane, as shown in the following figure.

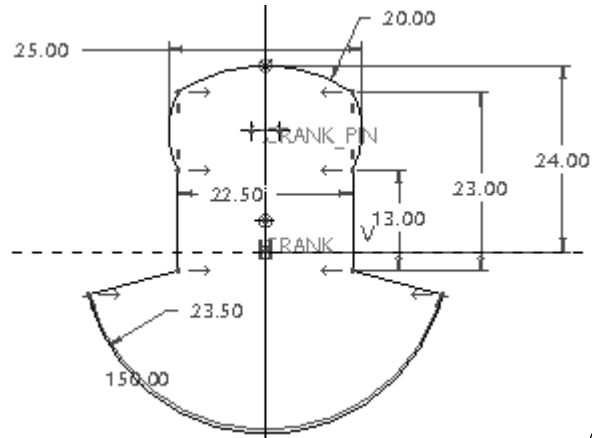


Figure 29: Create the Lobe Skeleton

Note:

If you are having difficulty sketching this curve, you can import the sketch. Click **Sketch > Data From File**, and select the appropriate *.sec file.

8. Create two additional sketched datum curves to represent the pin and the crank, as shown in the following figures.
9. Rename the 10mm diameter curve to **PIN_DIA**.
10. Rename the 12mm curve to **CRANK_DIA**.

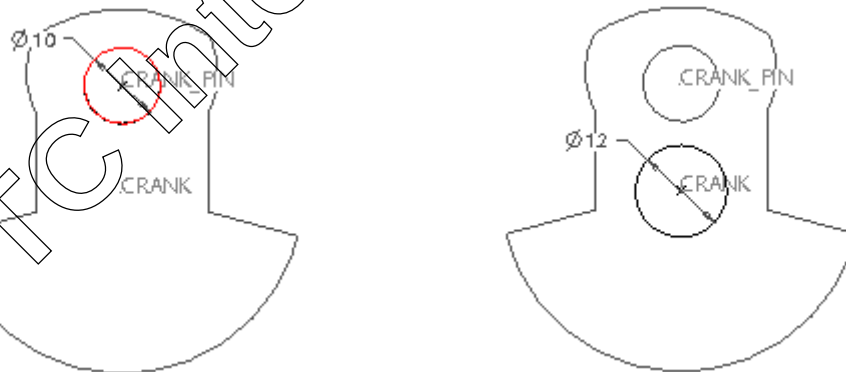


Figure 30: Creating the Skeleton Features to Represent the Pin and Crank

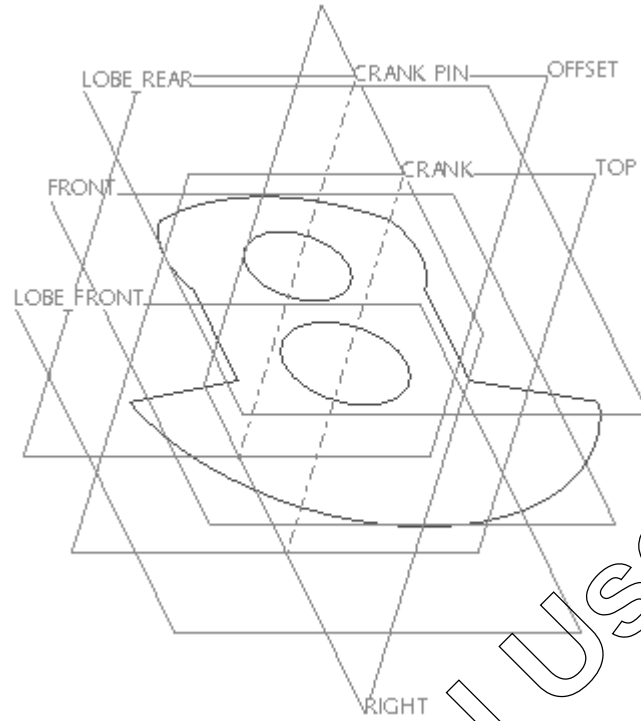


Figure 31: The Complete CRANKSHAFT.PRT

11. Save the model.
12. Close the window.
13. Erase the model from memory.

This completes the exercise.

Summary

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

- Start new design models using default templates.
- Capture design intent with Sketcher.
- Create design frameworks with datum features.
- Describe how sketched reference geometry affects parent/child relationships.

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Creating Sketch-based Features I

Introduction

The size and shape of sketch-based features are defined by creating 2-D sketches or by selecting existing sketched curves. Depending on the complexity of a sketch-based feature or its intended use, additional internally grouped datum features may be required.

Objectives

After completing this module, you will be able to:

- Create extruded, revolved, and ribbed from 2-D sketches.
- Create grouped datum features when creating features.
- Describe how creating sketch-based features affects parent/child relationships.



Instructor Preparation

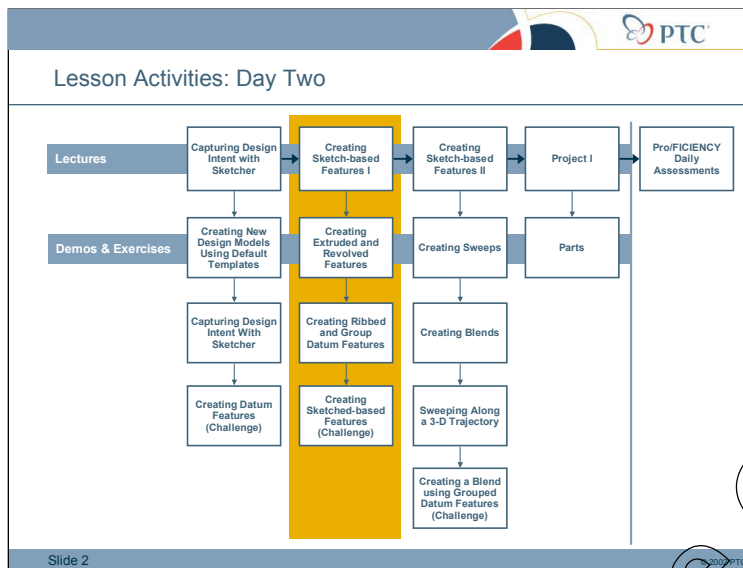
Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read and thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (2): 20 mins
- Labs (2): 30 mins
- Total: 1 hr



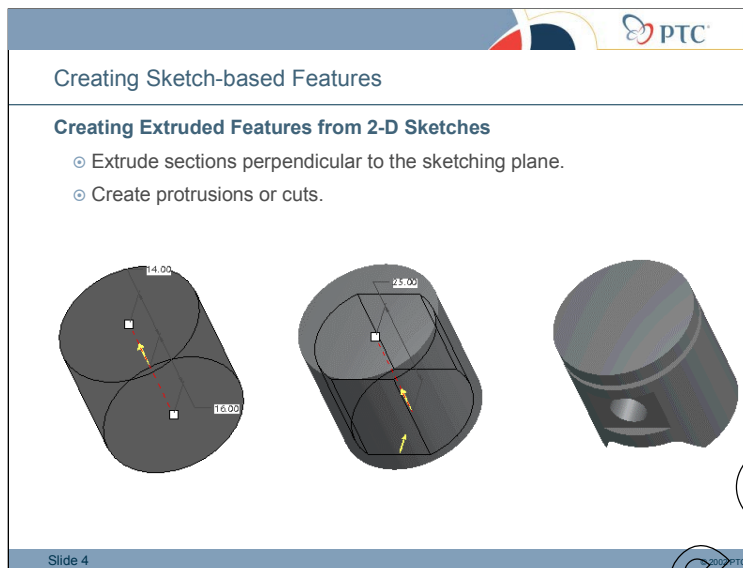
Objectives

After completing this module, you will be able to:

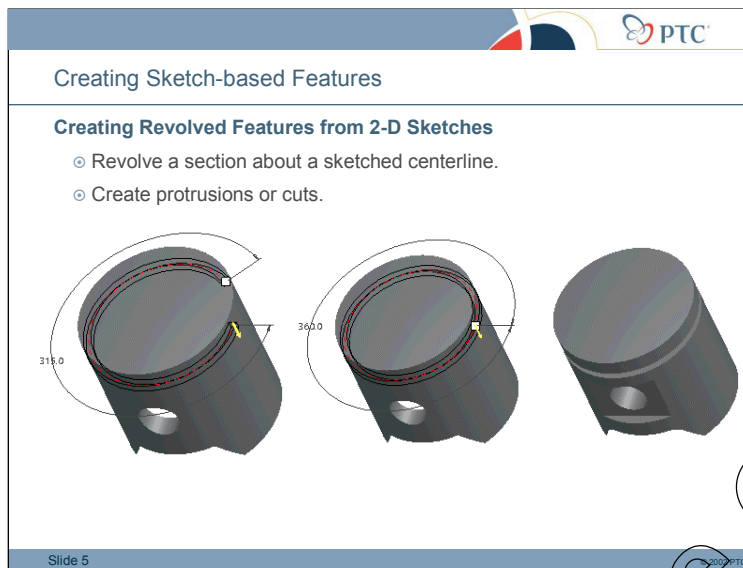
- Create extruded, revolved, and ribbed from 2-D sketches.
- Create grouped datum features when creating features.
- Describe how creating sketch-based features affects parent/child relationships.

Slide 3

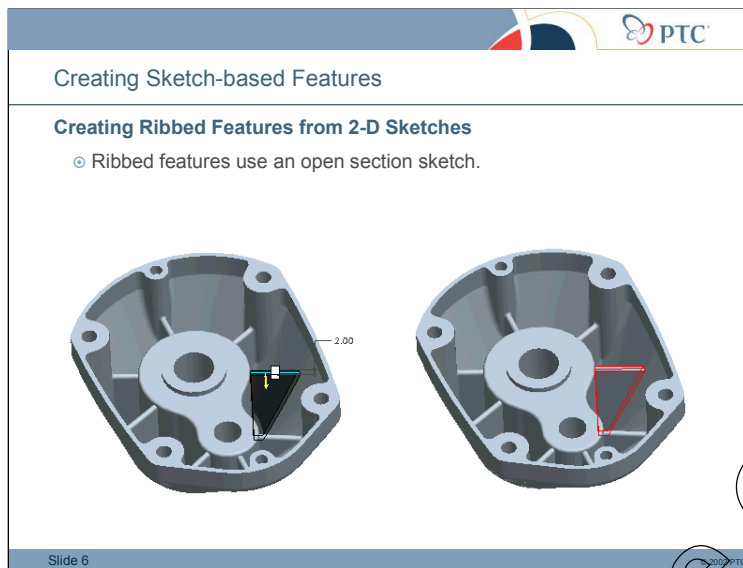
- The size and shape of sketch-based features are defined by creating 2-D sketches, or by selecting existing sketched curves.
- Depending on the complexity of a sketch-based feature or its intended use, additional internally grouped datum features may be required.



- An extruded feature is a sketch that extrudes a section perpendicular from a sketching plane to create a cut, protrusion, or surface.
- You use extruded features when you want to create or cut solid models.
- You do this by:
 - Selecting a sketching plane and orientation.
 - Selecting references.
 - Sketching, constraining, and dimensioning internal sections via curves. (You can also reference previous curves.)
 - Designating the feature as a protrusion, cut, or surface.
 - Selecting an extrude depth.



- A revolved feature is a sketch that is rotated about a centerline to create a cut, protrusion, or surface.
- You use revolved features when you want to create or cut solid models.
- You do this by:
 - Selecting a sketching plane and orientation.
 - Selecting references.
 - Sketching a centerline.
 - Sketching, constraining, and dimensioning internal sections via curves. (You can also reference previous curves.)
 - Designating the feature as a protrusion, cut, or surface.
 - Selecting a revolve angle.



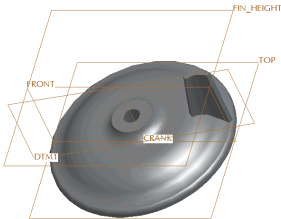
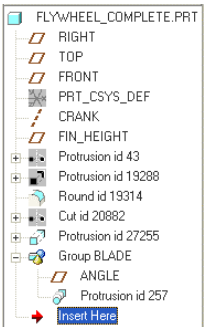
- A rib feature is similar to a direct feature, such as a hole or round, except that it requires a sketched or reference datum curve.
- Ribs are used to strengthen parts.
- The Rib tool enables you to create rib features quicker than creating and sketching a protrusion. The rib also conforms to existing planar or cylindrical geometry.

PTC

Creating Sketch-based Features

Creating Grouped Datum Features when Defining 2-D Sketches.

- Create datum references on-the-fly when creating features.
- Datum references are grouped with the feature being created.
- Datum references can be used when duplicating features.

Slide 7

- Grouped features are used to organize features within the model tree by cascading them underneath a group.
- If you create a datum feature within the context of another tool, group features are automatically created.
- For example, if you create a datum plane while creating another feature such as an extrude, the extrude and the plane are grouped automatically.
- Datum features in the group are automatically hidden. And, all the dimensions are available, if you modify the group.



Parent/Child Relationships – Creating Sketch-based Features I

When creating sketch-based features, parent/child relationships are created. Changes made to a parent feature affects its children.

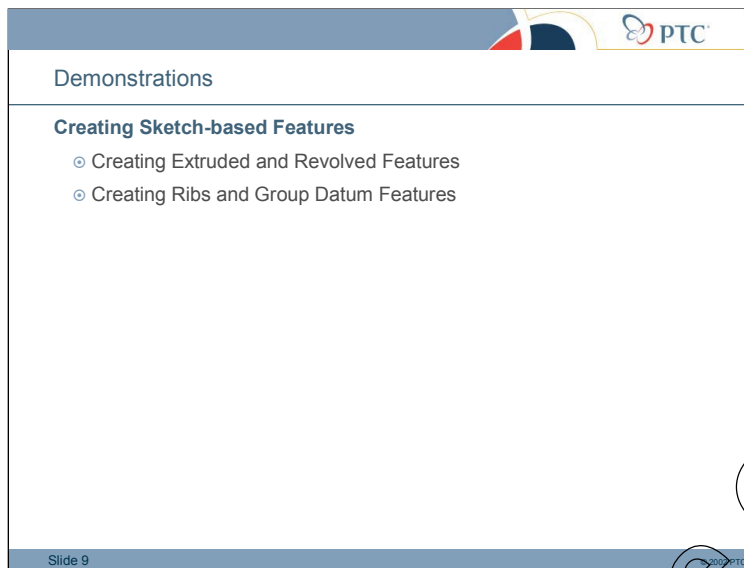
- Extruded features
- Revolved features
- Rib features
- Grouped features

Slide 8

Note to Instructors:

Use this slide to teach students how the functionality that they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

All sketched datum curve features are children to their references. For example, sketched datum curves will be children to the planes and surface that they are drawn on and referenced to. Additionally, any constraints to other sketched entities will be a parent feature. For example, constraining the edge of the sketch to make it parallel to a previous sketch based feature will be a parent of the new feature.



You will follow along while I demonstrate:

1. Creating Extruded and Revolved Features
2. Creating Ribs and Group Datum Features

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Create extruded, revolved, and ribbed from 2-D sketches.
- Create grouped datum features when creating features.
- Describe how creating sketch-based features affects parent/child relationships.

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Referencing PTC Help for Module 6 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Extrude Feature	Part Modeling	Example: Different Types of Extrude Feature About the Extruded User Interface
Revolve Feature	Part Modeling	About the Revolve Feature Example: Different Types of Revolved Feature About the Revolve User Interface About Sections Used for Revolved Features
Rib Features	Part Modeling	About the Ribbed Feature About the Ribbed User Interface Sketching Ribbed Features
Groups	Part Modeling	About Local Groups About Working with Groups

Module 6 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Creating Extruded and Revolved Features

Introduction

In this demonstration, we create three parts using extruded and revolved protrusions and cuts. Extruded cuts and protrusions are one of the primary methods for creating model geometry by extruding a section perpendicular from a sketching plane. Revolved features are used create cuts and protrusions by rotating a section about a centerline. The majority of the model geometry we create will use these two methods of creating cuts and protrusions.

Objectives

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

- Create extruded parts
- Create revolved parts

Scenario

As part of the new design for the gas powered drill, you have been assigned to build three parts: the crankshaft, piston, and connecting rod. You must adhere to any reference geometry that has already been designed for these parts. Finally, change the parts to their assigned colors before saving the parts.

Step 1. Create an extruded shaft on CRANKSHAFT.PRT using an external sketch.

Our manager has assigned us to create the drill's crankshaft referencing the geometry features that a team member created. This will ensure that all parts will fit together correctly when assembled in the future. Note the crankshaft will be used in the drill's engine assembly.

In this exercise, we create the crankshaft referencing the three datum curves that have already been created. We start by building the main rod of the crankshaft. Initially, we will not be concerned by the cut that will occur in the rod. We will make this cut at a later time.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to:
C:\users\student\fast_track_330\module_06
3. Open CRANKSHAFT.PRT.
4. Start the Extrude tool, and select the large circular curve, as shown in the following figure.
5. Read the prompt, and click **Continue**.
6. Using the dashboard, select the OPTIONS tab, and select **Blind** for SIDE2. Drag the depth handles and edit the dimensions to **46.5** and **73.5**, as shown in the following figure.

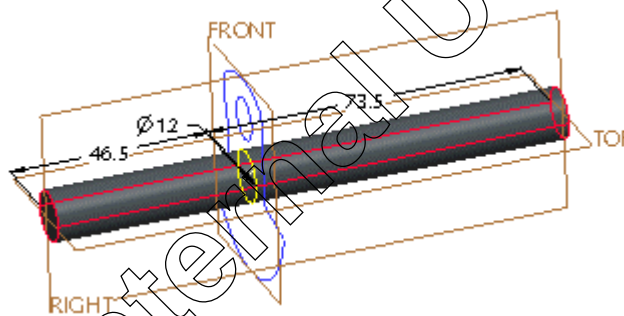


Figure 1: Extruding the Rod of CRANKSHAFT.PRT

7. Complete the feature.

Step 2. Create an extruded lobe on CRANKSHAFT.PRT using an external sketch.

After creating the main rod, we create a protrusion that will serve as the base of the two lobes of the crankshaft. This protrusion will also reference a pre-defined curve.

1. Using the model tree, select the LOBE curve, and start the Extrude tool.
2. Read the prompt, and click **Continue**.
3. Drag the depth handle, right-click and select **To Selected**. Select the LOBE_FRONT plane.
4. Using the dashboard, select the OPTIONS tab, and select **To Selected** for SIDE2. Select the LOBE_REAR plane.
5. Complete the feature.

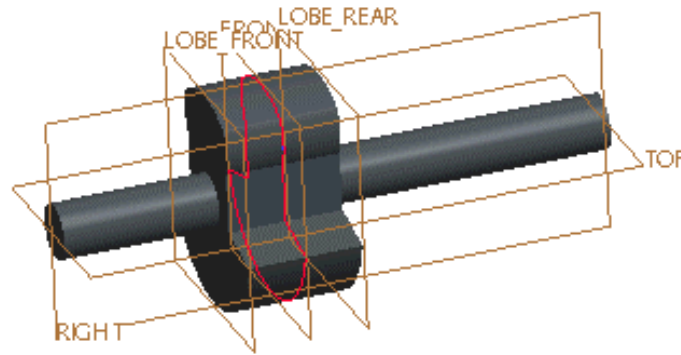


Figure 2: Extruding the Lobe of CRANKSHAFT.PRT

Note:

Notice that much of the design intent for this model originated in the skeleton.

Step 3. Create an extruded cut on the lobe feature of CRANKSHAFT.PRT using an external sketch.

Then, we extrude a cut to form the two lobes again referencing a datum curve. We create a datum curve and reference that curve in a revolved cut at the end of the main rod.

1. Start the Extrude tool. Using the model tree, select the PIN_DIA curve.
2. Read the prompt and click **Continue**.
3. Using the dashboard, extrude symmetrically on both sides, and edit the dimension to **10**.
4. Using the dashboard, remove material and flip the material direction.
5. Complete the feature.

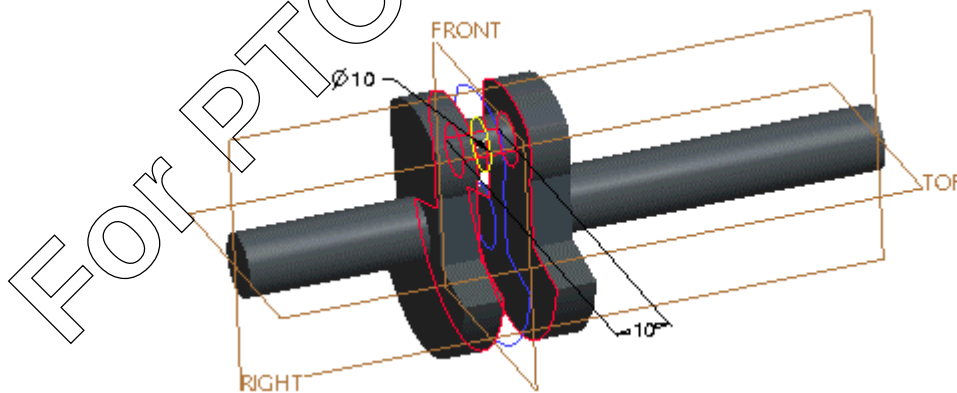


Figure 3: Extruding a Cut on the Lobe of CRANKSHAFT.PRT

Step 4. Create a revolved cut on the shaft feature of CRANKSHAFT.PRT using an external sketch.

Finally, we delete various reference geometry to confirm that no parent-child relationships were created when sketching certain features. We also change the color of the crankshaft to allow it to be distinguished during assembly.

1. Orient the model approximately horizontal.
2. Start the Sketched Datum Curve tool, select the RIGHT plane, and click **Sketch**.
3. Specify references as the upper circular surface and right flat surface of the crankshaft.
4. Sketch a horizontal centerline. Sketch two lines snapping to the references. Dimension the sketch, as shown in the following figure.

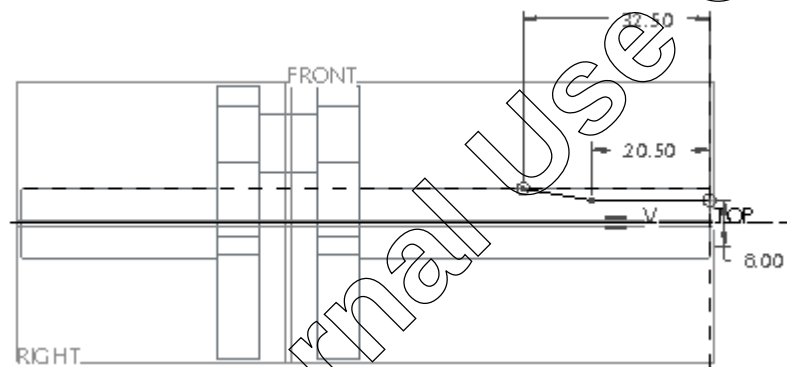


Figure 4: Sketching a Reference Datum Curve

5. Complete sketch and the feature, as shown in the following figure.

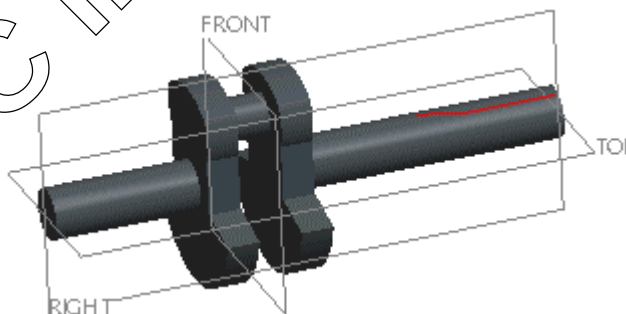


Figure 5: Completed Datum Curve Feature

6. With the previous curve still selected, start the Revolve tool.
7. Read the prompt and click **Continue**.
8. Using the dashboard, revolve the feature as a solid and remove material.
9. Accept the default of **360** degrees and complete the feature, as shown in the following figure.

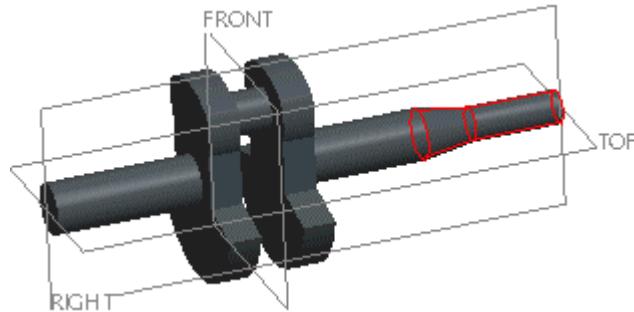


Figure 6: Complete the CRANKSHAFT.PRT Cut

10. Using the model tree, select and delete the four datum curves.

Note:

If the external curves are not intended to be re-used, it is a best practice to delete them.

Step 5. Complete CRANKSHAFT.PRT.

Instructor: Skip this step for the demo.

1. Click **View > Color and Appearance**. From the Appearance Editor, click **File > Open**, and select DRILL_COLOR_APPEARANCES.DMT from the folder located up one level.
2. Click **Override All**, select the **Gray_dark** appearance, and click **Apply**. Complete the appearance edit.
3. Save the model.
4. Close the window.

Step 6. Extrude the base feature on CONNECTING_ROD.PRT using an internal sketch.

Next, we create the drill's connecting rod. To ensure that all parts fit together correctly during assembly, we reference the geometry features previously created by another team member. Note that the connecting rod will be used in the drill's engine assembly.

In this exercise, we start by extruding the base feature of the connecting rod. We extrude the end protrusions of the rod. We create holes where the rod assembles to the piston pin and crankshaft. Finally, we change the color of the connecting rod to distinguish it for assembly.

1. Open CONNECTING_ROD.PRT.
2. Examine the Copy Geometry feature, as shown in the following figure. Note that these references are copied from a skeleton model and convey critical mounting information.

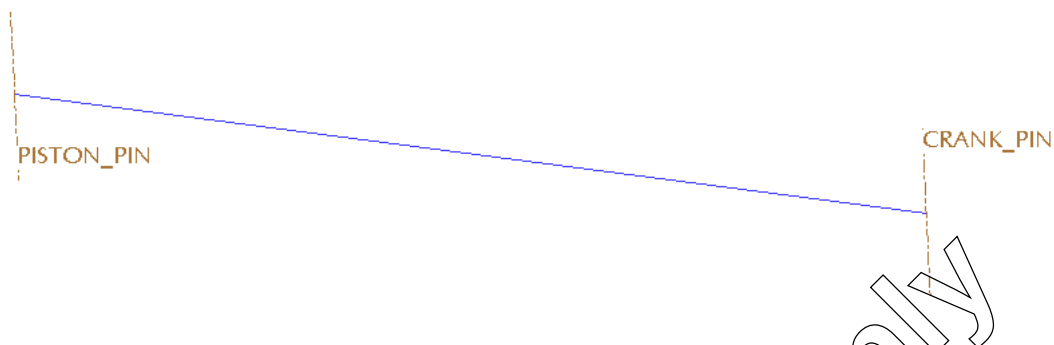


Figure 7: Examining Copy Geometry Features

Note:

By using skeletons and copy geometry features, you can distribute design tasks to multiple engineers.

3. Start the Extrude tool. Using the dashboard, create a sketch. Select the FRONT datum plane and orient the sketch view so that the RIGHT datum faces **Top**.
4. Click **Sketch** and select the CRANK_PIN and PISTON_PIN axes as references.
5. Sketch a horizontal centerline, and then sketch, constrain, and dimension the remaining geometry, as shown in the following figure.

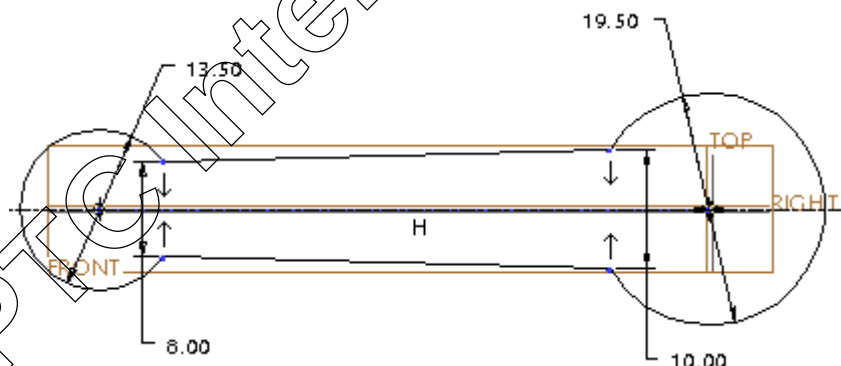


Figure 8: Sketching the Initial CONNECTING_ROD.PRT Protrusion

6. Complete the sketch. Drag the depth handle, right-click and select **Symmetric**, and then enter **2.5** as the depth.
7. Complete the feature.

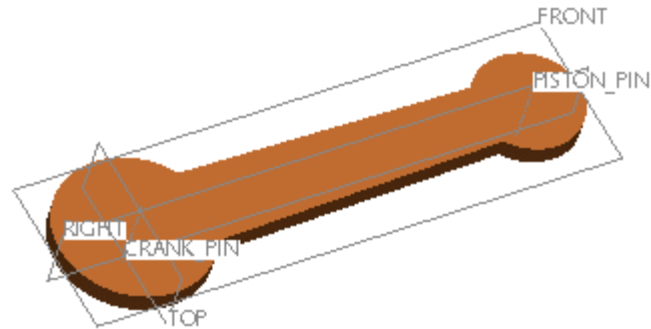


Figure 9: Completed Extruded Protrusion

Step 7. Create protrusions on the ends of CONNECTING_ROD.PRT using internal sketches.

We will now create protrusions referencing existing geometry.

1. Start the Extrude tool. Using the dashboard, create a sketch. Select the **FRONT** datum plane. Orient the sketch view so that the **RIGHT** datum faces **Top**.
2. Click **Sketch** and accept the default references.
3. Using the Concentric Circle option, sketch a circle, as shown in the following figure.

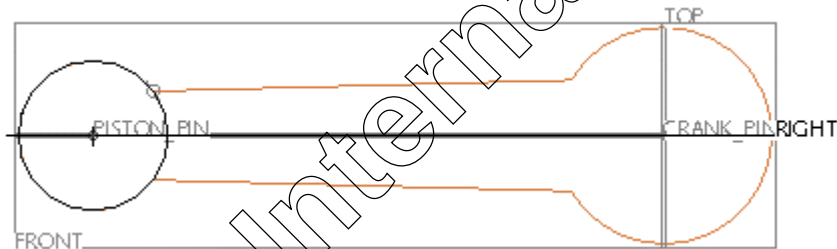


Figure 10: Sketching the Protrusion

4. Complete the sketch. Using the dashboard, enter **8** as the symmetrical depth.
5. Complete the feature.

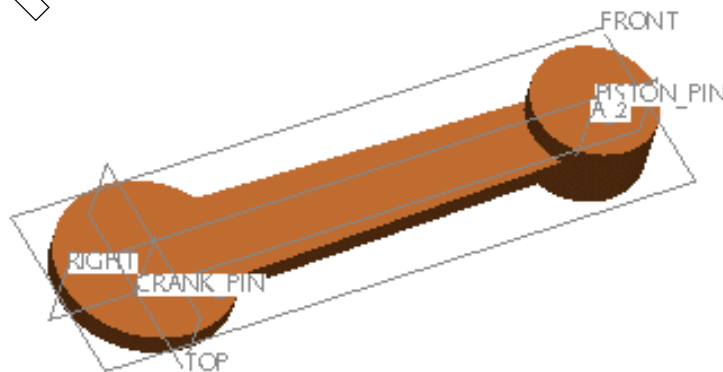


Figure 11: Completed Protrusion

6. Repeat the previous procedure to create another protrusion. For the depth, reference the surfaces of the previous protrusion, as shown in the following figure.

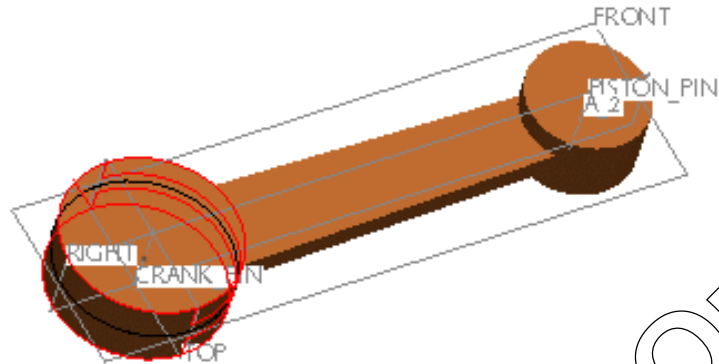


Figure 12: Completed Protrusion

7. Save the model.
8. Close the window.

Step 8. Create PISTON.PRT by extruding and revolving features with internal sketches.

In this exercise, we create the piston by extruding and revolving cuts and protrusions. Note that we sketch all of the features in this part.

1. Create a new part named PISTON.PRT, using the default template.
2. Create two datum axes through the default datum planes. Name the axes PISTON_PIN and CYLINDER, as shown in the following figure.

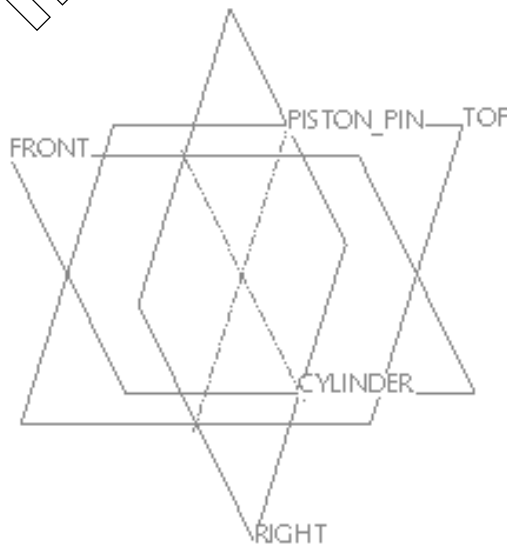


Figure 13: Creating the PISTON_PIN and CYLINDER Axes

3. Select the TOP datum plane. Start the Extrude tool. Using the dashboard, create a sketch. Note that the TOP plane is selected as the sketching plane.
4. Click **Sketch** and accept the default references. Sketch a circle with a **30.5** diameter, as shown in the following figure. Complete the sketch.
5. Set the depth for SIDE 1 to **14** and the depth for SIDE 2 to **16**. Complete the feature.

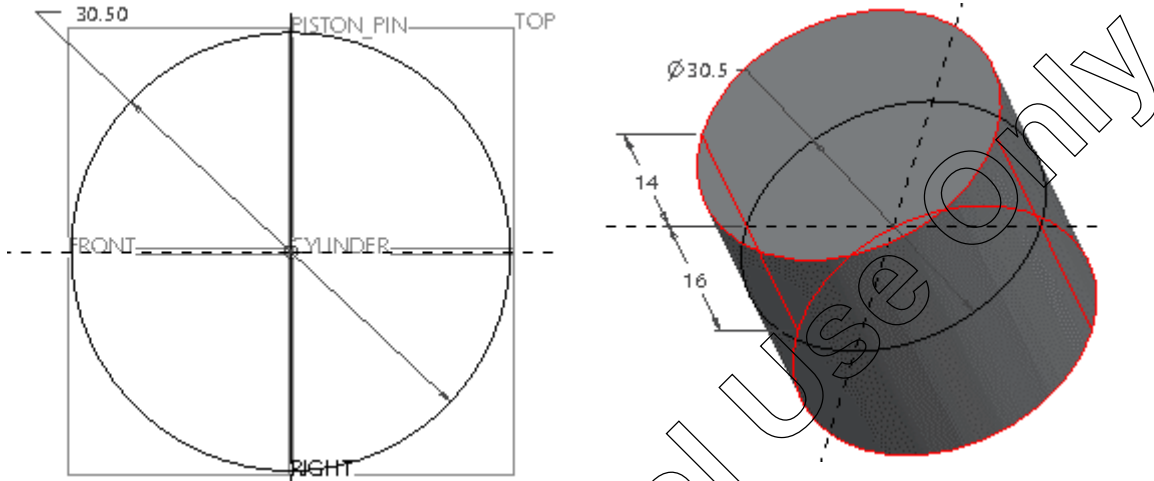


Figure 14: Completing the Initial Protrusion

Step 9. Create a revolved protrusion with an internal sketch.

Instructor: Skip this step for the demo.

1. Click **View > Color and Appearance**. Select the **Gray_med** appearance, and click **Apply**. Complete the appearance edit.
2. Start the Revolve tool. Using the dashboard, create a sketch, and then select the FRONT datum plane.
3. Click **Sketch**, and select the top and right surfaces of PISTON.PRT as references.
4. Sketch a vertical centerline on the center reference.
5. Sketch an arc and a short vertical line, as shown in the following figure. Constrain the arc perpendicular the centerline.
6. Complete the sketch, revolve **360** degrees, and complete the feature.

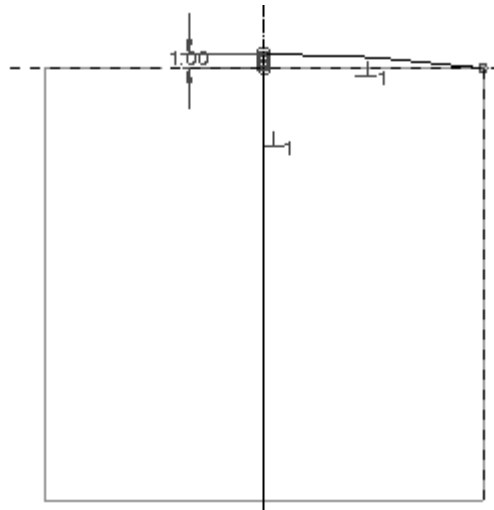


Figure 15: Revolving a Protrusion on PISTON.PRT

Step 10. Create the cavity and clearance cut on PISTON.PRT by extruding cut features with internal sketches.

Instructor: Skip this step for the demo.

1. Start the Extrude tool. Using the dashboard, create a sketch, and select the BOTTOM surface of the model.
2. Click **Sketch**, and then select the circular surface of PISTON.PRT as a reference. Sketch and dimension using a horizontal centerline for symmetry, as shown in the following figure.
3. Complete the sketch. Using the dashboard, flip the direction of the depth, enter **25** as the depth, and remove material. Complete the feature.

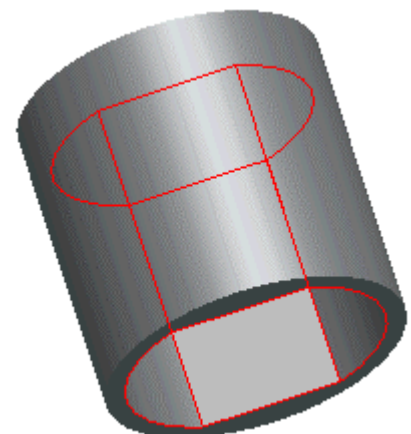
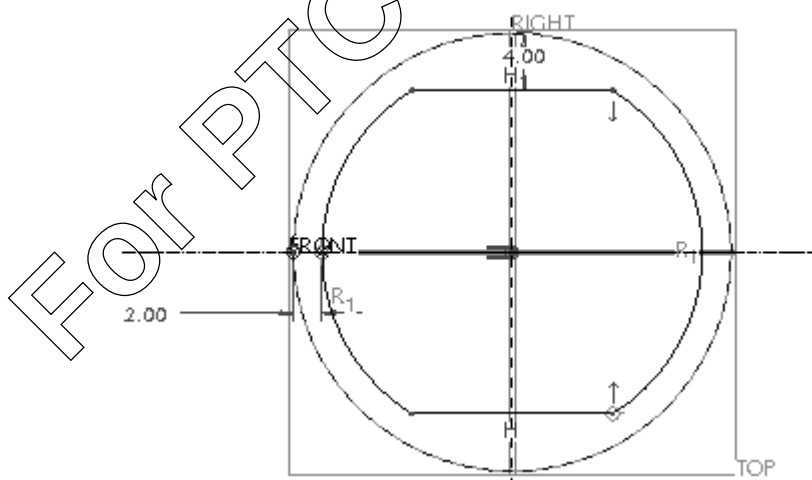


Figure 16: Sketching the Extruded Cut and the Completed Cut

4. Start the Extrude tool. Using the dashboard, create a sketch, and select the FRONT datum plane.
5. Click **Sketch** and select the lower surface of PISTON.PRT as a reference. Sketch and dimension, as shown in the following figure, using a vertical centerline for symmetry.
6. Complete the sketch. Using the dashboard, specify to remove material, and set the depth for Side 1 and Side 2 to **Thru All**, as shown in the following figure. Complete the feature.

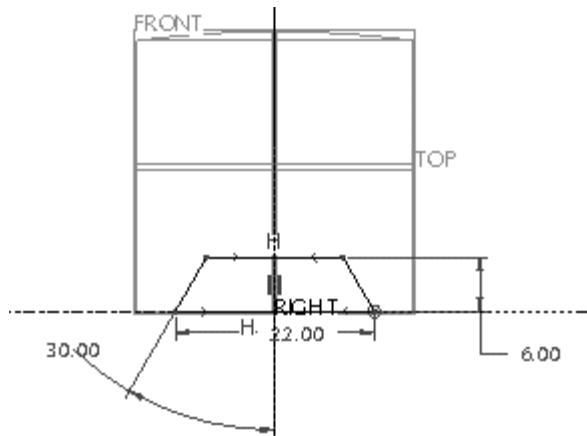


Figure 17: Sketching the Extruded Cut and the Completed Cut

Step 11. Create additional PISTON.PRT features.

Instructor: Skip this step for the demo.

1. Create a coaxial hole referencing the PISTON_PIN axis and the FRONT plane, as shown in the following figure. Set both depths to **Thru All** and the diameter to **8**.

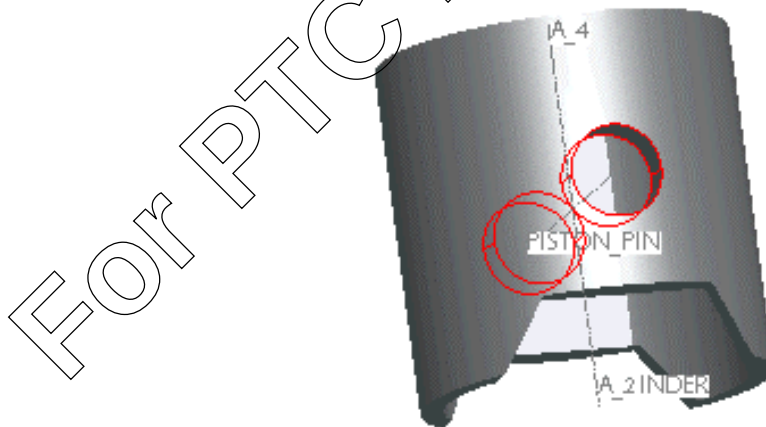


Figure 18: Creating a Coaxial Hole on PISTON_PIN

2. Create a **360** degree revolved a cut referencing the FRONT plane, as shown in the following figure.

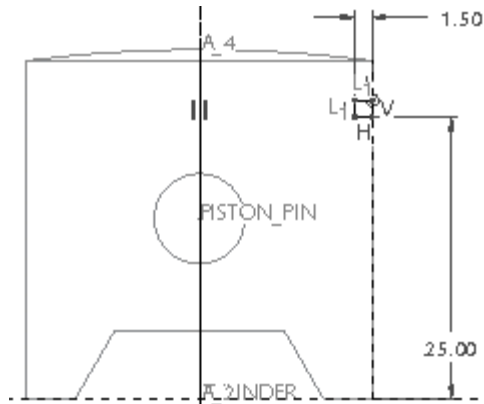


Figure 19: Sketching the Cut and the Completed Cut Feature

3. Create an extruded cut from the RIGHT plane **Thru All**, as shown in the following figure.

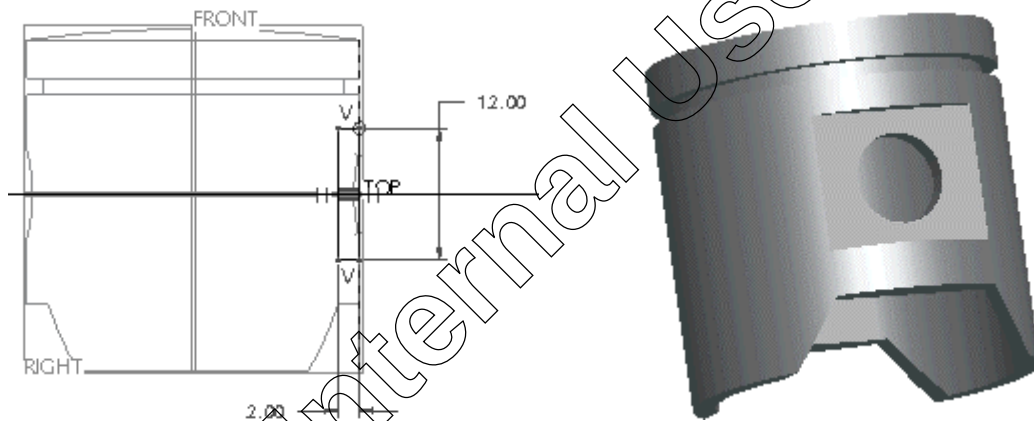


Figure 20: Sketching the Cut and the Completed Cut Feature

4. Save the model
5. Close the window.
6. Erase all models from memory.

This completes the exercise.

Exercise 2: Creating Ribs and Group Datum Features

Introduction

In this demonstration, we create ribs on three parts by sketching the rib and setting a thickness. Ribs are a primary engineering method used to strengthen parts. The Rib tool enables you to quickly create rib features.

Objective

After successfully completing this exercise, you know how to:

- Create rib features.

Scenario

Product testing has indicated that the gearbox front part, engine block rear part, and recoil cover part need additional ribs to strengthen the parts against premature failure. Your assignment is to add the additional rib features.

Step 1. Create a rib on GEARBOX_RIB.PRT, using an external section.

Instructor: Skip this step for the demo.

1. Open GEARBOX_RIB.PRT.
2. Orient to the RIB view.
3. Using the model tree, select the RIB_CRV and start the Rib tool, as shown in the following figure.

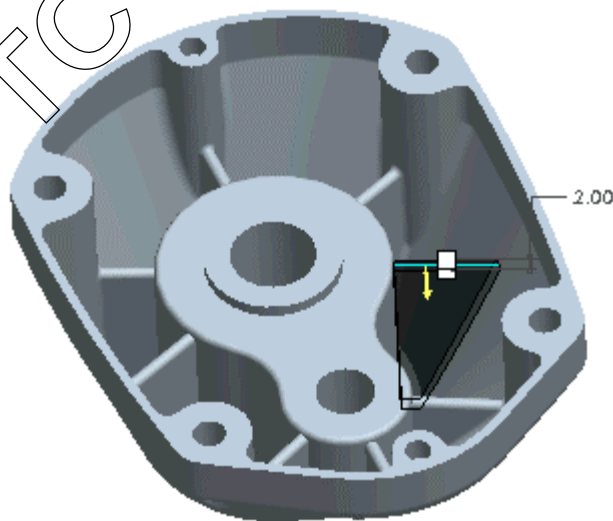


Figure 21: Creating the Rib Feature

4. Drag the thickness handle and enter **2**. Complete the feature.

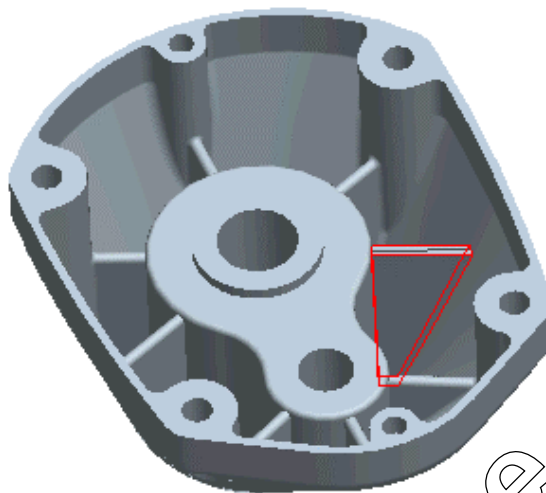


Figure 22: Completed Gearbox with Rib

5. Using the model tree, delete the RIB_CRV.
6. Save the model.
7. Close the window.

Step 2. Create a rib on RECOIL_COVER.PRT using a grouped datum.

This rib is on the recoil cover. It is created as a GROUP feature in addition to a datum plane. To accomplish this group feature, we start the rib tool to start building the rib. Then, we start the datum plane tool to create the angled RIB plane. After completing the plane, Pro/ENGINEER Wildfire continues with the rib feature, which we sketch on the rib plane. After completing the feature, the rib plane and the rib is displayed in the model tree as a grouped feature.

Grouped features allow us to organize the model tree and manipulate large numbers of features simultaneously. For example, we can edit both the RIB plane's angle and re-dimension the rib sketch.

1. Open RECOIL_COVER.PRT.
2. Start the Rib tool. Pause the dashboard and start the Datum Plane tool. Select the MAIN axis, press CTRL and select the TOP plane. Accept the default angle of **45** degrees, and then name the plane **RIB**, and complete the feature.

Note:

The dashboard can be paused automatically by starting any datum tool while the dashboard is open.

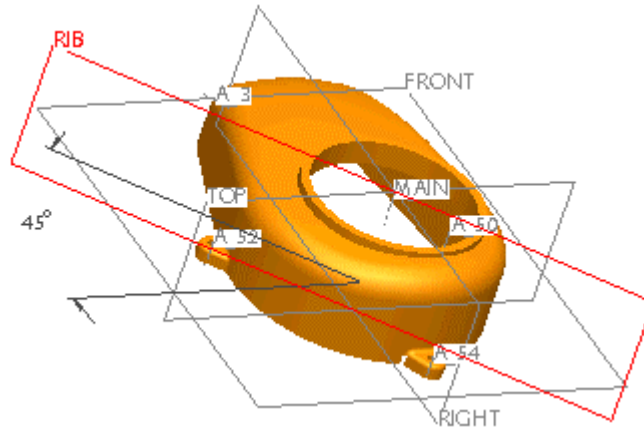


Figure 23: Creating RIB Plane

3. Using the dashboard, resume the Rib tool and create a sketch. Select **RIB** as the sketching plane. Select the **FRONT** plane and select **Bottom** to orient the sketch.
4. Click **Sketch** and select the **MAIN** axis and the upper horizontal surface as references.
5. Click **Sketch > Data from File**, and open **RIB_SKETCH.SEC**.
6. Drag the location handle to position the sketch, as shown in following figure. Specify a Scale of **1** and a Rotation of **0**. Complete the section placement.

Note:

You can move the location of the drag handle within the section by clicking on it with the right mouse button.

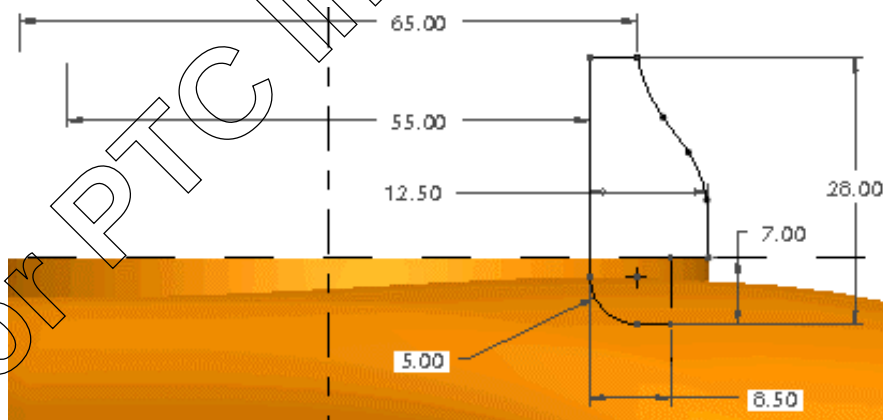


Figure 24: Creating the Rib Sketch

7. Complete the sketch and enter **2** as the thickness.
8. Complete the feature and rename the group to **RIB_1**.

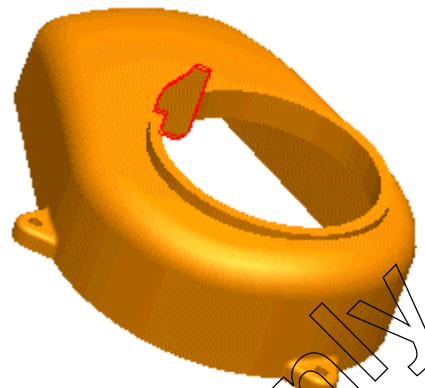


Figure 25: Model Tree and Completed Rib Feature on RECOIL_COVER.PRT

Note:

The RIB plane is created internally in a Group with the rib feature.

9. Save the model.
10. Close the window.

Step 3. Create a rib on ENG_BLOCK_REAR.PRT.

Instructor: Skip this step for the demo.

1. Open ENG_BLOCK_REAR.PRT.
2. Start the Rib tool. Using the dashboard, create a sketch. Select the RIB plane as the sketching plane. Using the model tree, select the FRONT plane and select **Top** to orient the sketch.
3. Click **Sketch** and select the 3 surfaces as references highlighted in red, as shown in the following figure. Sketch and dimension a line and an arc. Complete the sketch.

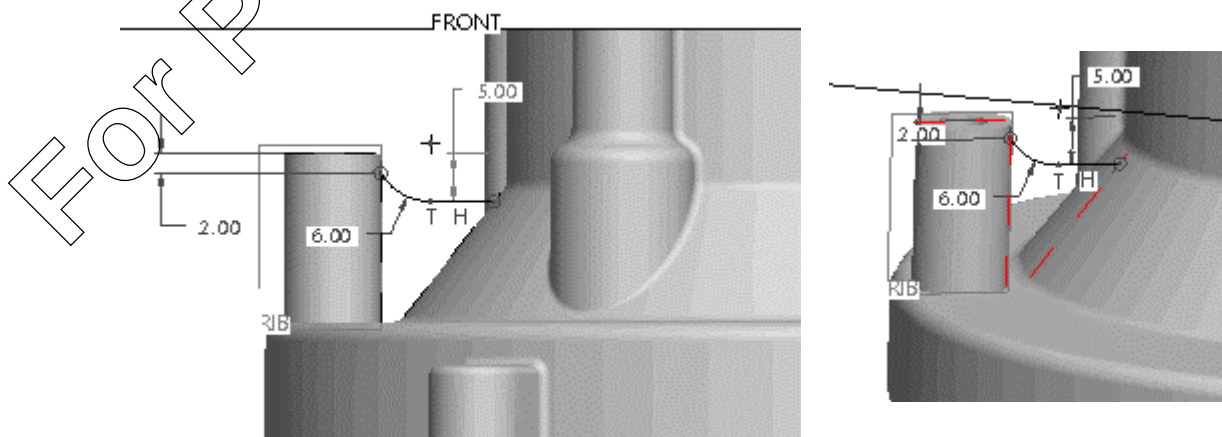


Figure 26: Sketching the Rib on ENGINE_BLOCK_REAR.PRT

4. If necessary, flip the material direction. Drag the thickness handle and enter **2**. Complete the feature.

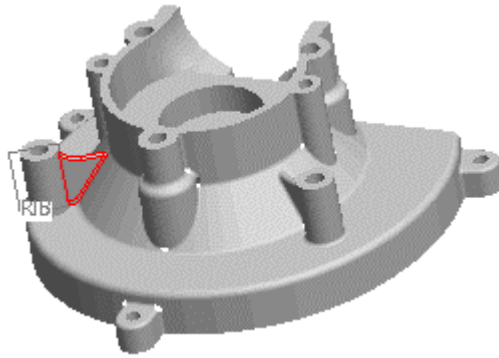


Figure 27: Complete Rib Feature on ENGINE_BLOCK_REAR.PRT

5. Save the model.
 6. Close the window.
 7. Erase all models from memory.
- This completes the exercise.

Exercise 3: Completing the Piston (Challenge)

Objective

In this exercise, you complete the piston by using various techniques learned previously.

Scenario

You are assigned to complete the piston. Balance the piston, add retaining features that will hold the piston pin, and round off edges to reduce stresses.

Step 1. Complete PISTON.PRT.

1. Open PISTON.PRT.
2. Extrude a cut from the RIGHT plane **Thru All**, as shown in the following figure.

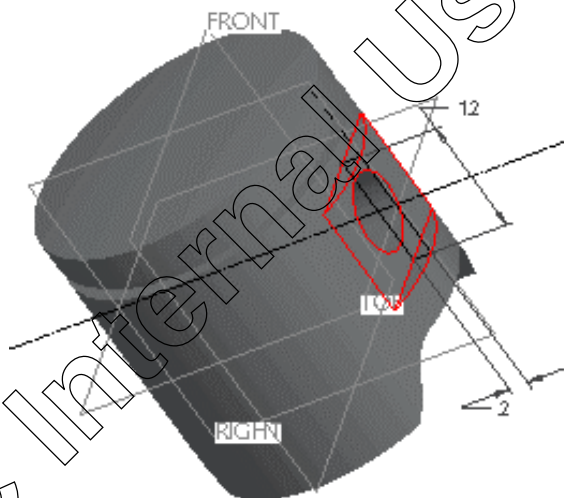


Figure 28: Creating an Extruded Cut

3. Extrude a thin protrusion symmetrically from the FRONT plane, as shown in the following figure. Specify a thickness of **2**.

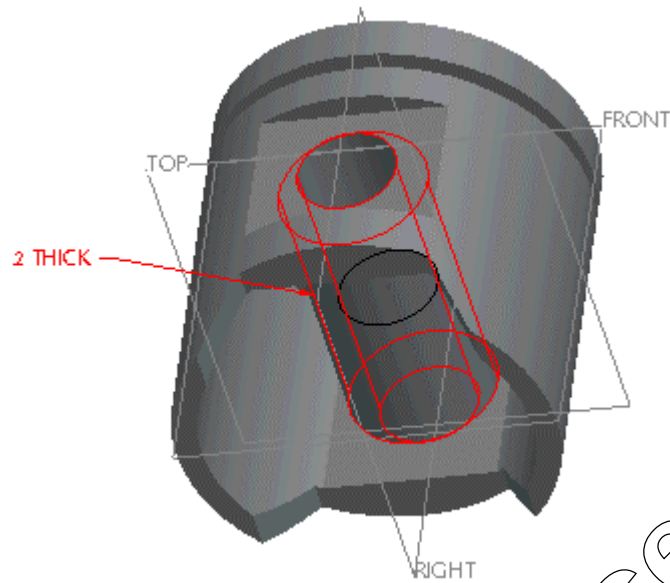


Figure 29: Creating a Thin Protrusion

4. Extrude a cut symmetrically from the FRONT plane with a depth of **14**, as shown in the following figure.

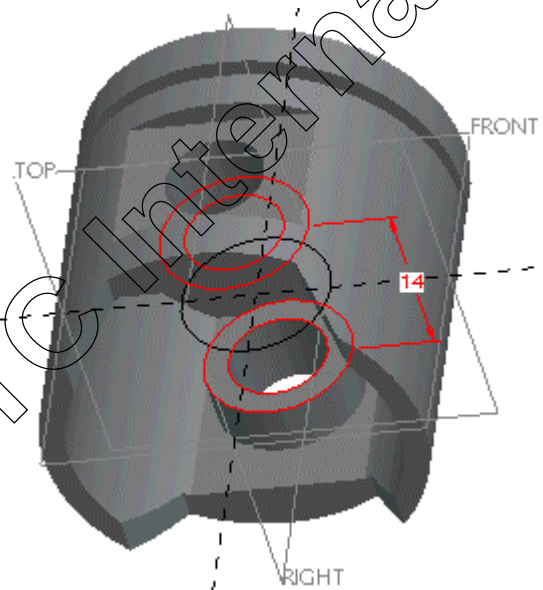


Figure 30: Cutting the Thin Protrusion

5. Create four round features, as shown in the following figures.

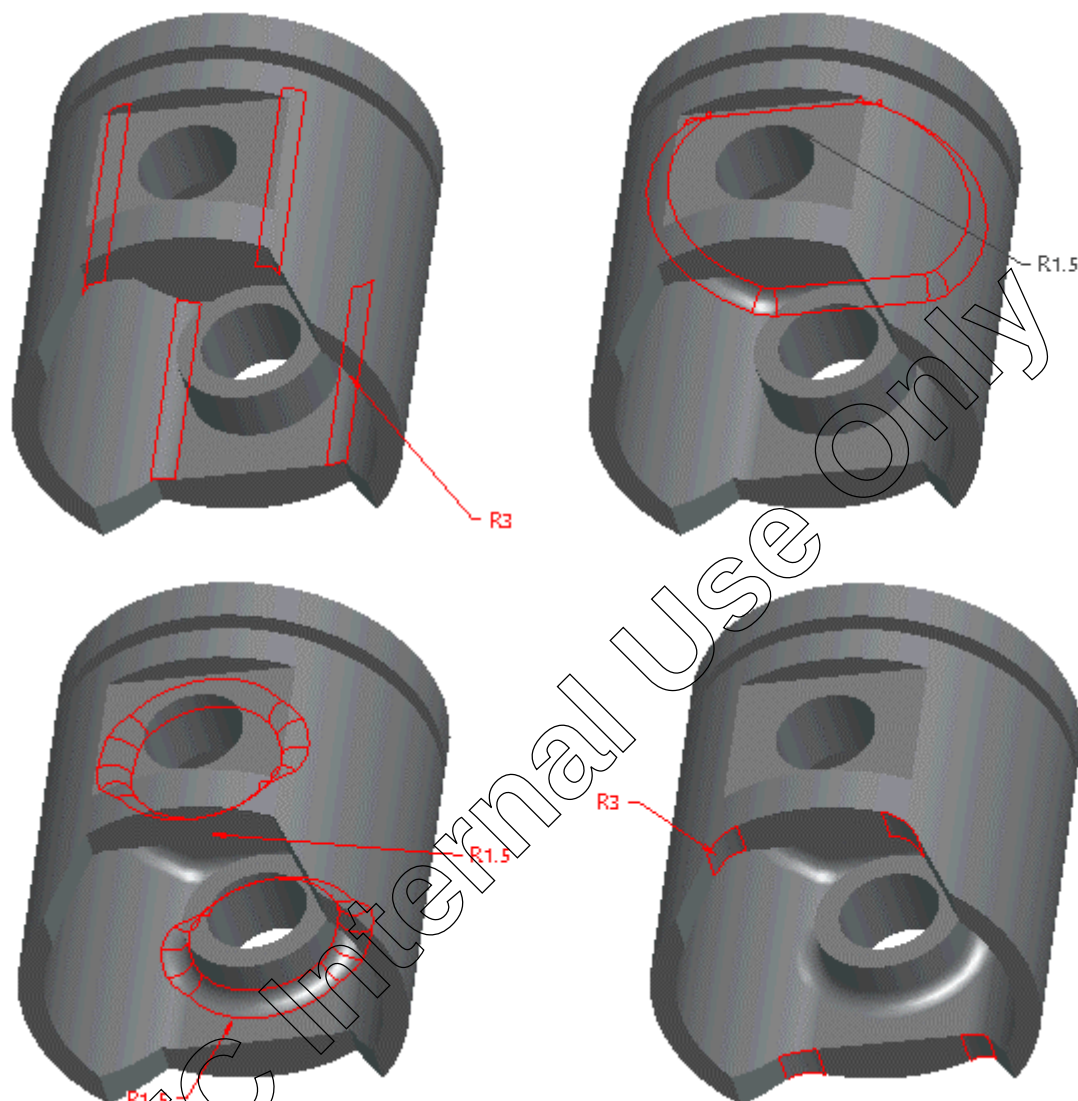


Figure 31: Creating Rounds on PISTON.PRT

6. Save the model
7. Close the window.
8. Erase the model from memory.

This completes the exercise.

Summary

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

- Create extruded, revolved, ribbed, swept, and blended features from 2-D sketches.
- Create grouped datum features when creating features.
- Describe how creating sketch-based features affect parent/child relationships.

For PTC Internal Use Only

Creating Sketch-based Features II

Introduction

The size and shape of sketch-based features are defined by creating 2-D sketches or by selecting existing sketched curves. Depending on the complexity of a sketch-based feature or its intended use, additional internally grouped datum features may be required.

Objectives

After completing this module, you will be able to:

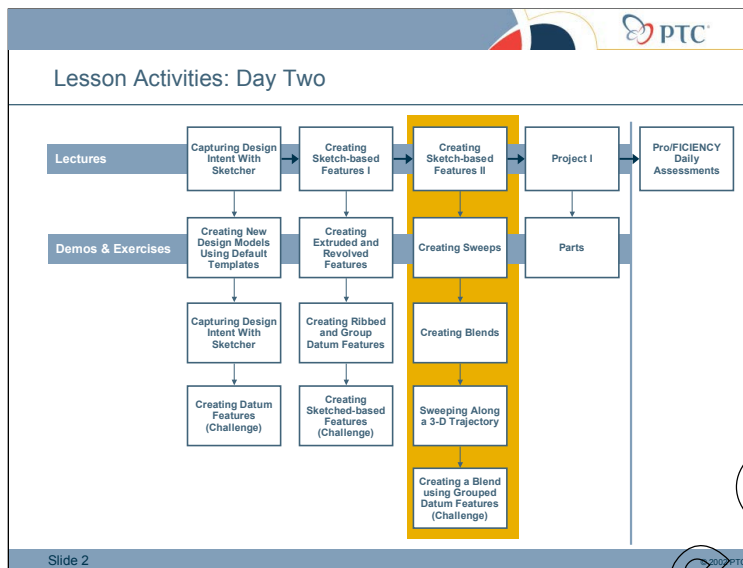
- Create swept and blended features from 2-D sketches.
- Describe how creating sketch-based features affects parent/child relationships.



Instructor Preparation

Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read and thoroughly understand** 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 Foundation
 - Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 5 mins
- Demos (2): 20 mins
- Labs (2): 35 mins
- Total: 1 hr



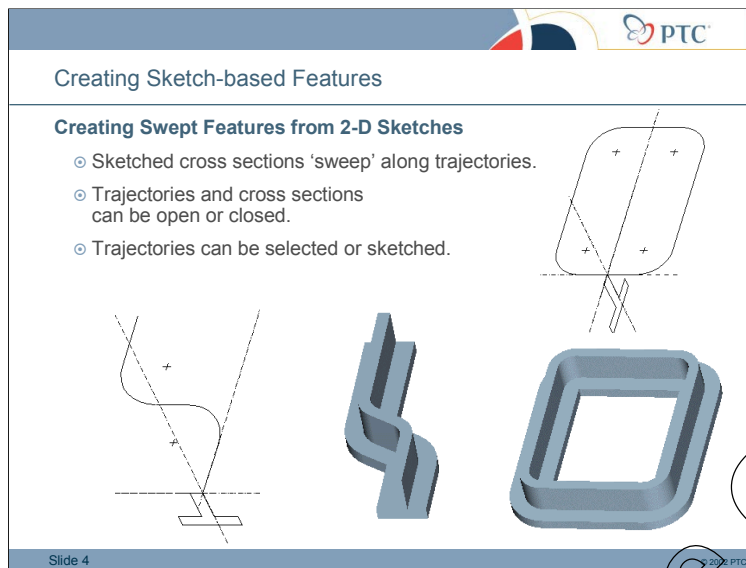
Objectives

After completing this module, you will be able to:

- Create swept and blended features from 2-D sketches.
- Describe how creating sketch-based features affects parent/child relationships.

Slide 3

- The size and shape of sketch-based features are defined by creating 2-D sketches, or by selecting existing sketched curves.
- Depending on the complexity of a sketch-based feature or its intended use, additional internally grouped datum features may be required.



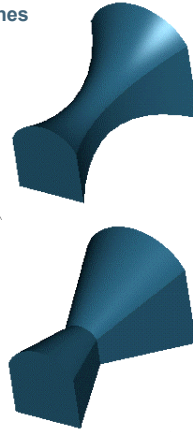
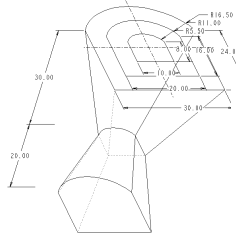
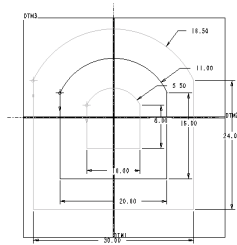
A sweep feature “sweeps” a sketched curve along a trajectory.

- You create a sweep feature when you want to create a constant cross section model that can follow a curve.
- Protrusions following a trajectory are quicker to model, rather than using extrude and revolve features.

Creating Sketch-based Features

Creating Parallel Blended Features from 2-D Sketches

- Blend sections with straight or smooth transitions.
- Sketch at least two sections on the same plane.



Slide 5

- Blend features create cuts and protrusions that use different cross sectional sketches.
- You create blend features when you need to create models that contain different transitional cross sections.
- Protrusions created using blends are quicker to model, rather than using extrude and revolve features.



Parent/Child Relationships – Creating Sketch-based Features II

When creating sketch-based features, parent/child relationships are created. Changes made to a parent feature affects its children.

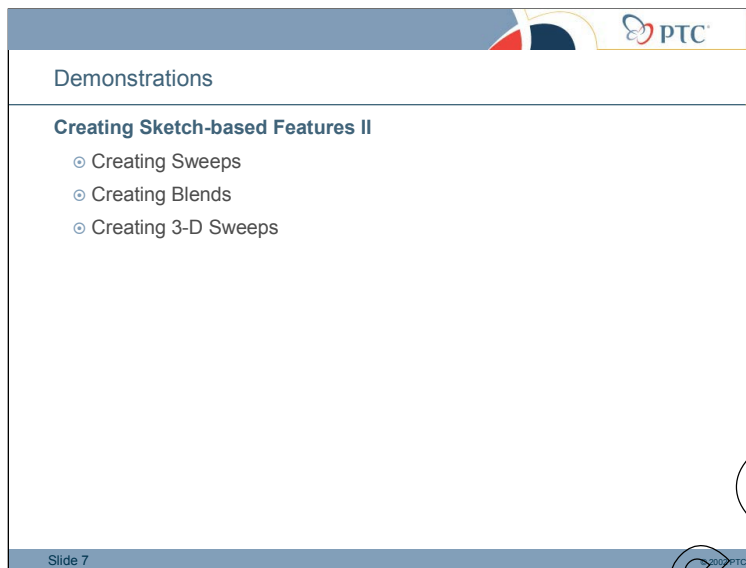
- Swept features
- Blend features

Slide 6

Note to Instructors:

Use this slide to teach students how the functionality that they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

All sketched datum curve features are children to their references. For example, sketched datum curves will be children to the planes and surface that they are drawn on and referenced to. Additionally, any constraints to other sketched entities will be a parent feature. For example, constraining the edge of the sketch to make it parallel to a previous sketch based feature will be a parent of the new feature.



You will follow along while I demonstrate:

1. Creating Sweeps
2. Creating Blends
3. Creating 3-D Sweeps

Then, you will practice the same steps in the laboratory exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 8

Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY assessment questions.
- **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 Exams

Description: For each course, 5 new 10 question Pro/FICIENCY 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:

- Create swept and blended features from 2-D sketches.
- Describe how creating sketch-based features affects parent/child relationships.

Referencing PTC Help for Module 7 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Sweep	Part Modeling	Sweep Geometry Example: Sweeps To Create a Swept Feature To Create a Three-Dimensional Sweep
Parallel Blend	Part Modeling	About Parallel Blends Example: Different Blend Geometries To Create a Parallel Blend with a Regular Section

Module 7 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Creating Sweeps

Introduction

In this demonstration, we create sweeps. Sweeps are protrusions or cuts that take a sketched cross-section and “sweep” it along a trajectory. The trajectory can be either a datum curve or a sketched curve.

Sweeps are important because they allow us to create geometry using one feature whereas it might take us several extrude and revolve features to perform the same task.

Objective

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

- Create sweep features.

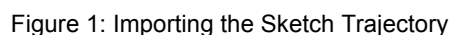
Scenario

You are assigned to create the main handle of the drill assembly. Use the initial trajectory sketch provided by Product Marketing to sweep a protrusion for the handle.

Step 1. Create the swept protrusion that forms the HANDLE_MAIN.PRT.

We are assigned to create the main handle feature for the drill assembly. Product Marketing performed an ergonomic study and determined the appropriate trajectory profile for the handle. We need to sketch the trajectory profile and create a swept protrusion to model the handle. The advantage of using a swept protrusion is that it would take several extrude and revolve features to create the same model versus a single sweep feature.

If you are having difficulty with the sketch, you can import the sketch by clicking **Sketch > Data from File**. Open **HANDLE_MAIN.SEC**. Right-click on the location handle and snap it to the centerlines in the section. Drag the location handle to snap the section to the sketcher references.



6. Specify a Scale of **1** and a Rotation of **0**. Complete the section placement.
7. Complete the sketch.
8. Sketch a circle at the intersection of the centerlines, as shown in the following figure.



9. Complete the sketch and complete the feature.

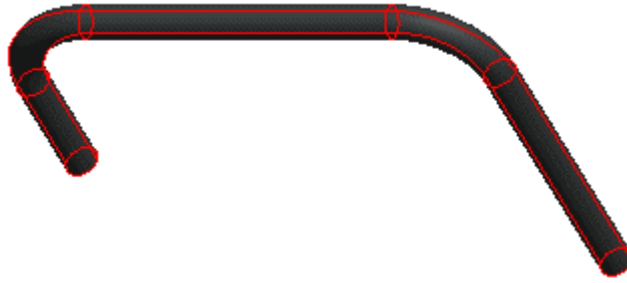


Figure 3: Completed Swept Feature on HANDLE_MAIN.PRT

10. Save the model.

11. Close the window.

This completes the exercise.

Exercise 2: Creating Blends

Introduction

In this demonstration, we create blends. Blends are protrusions or cuts that merge several cross-sections along a straight line by setting a depth.

For example, we could create a blend that starts as a circle and transforms into a thin flat rectangle, such as a flat-head screwdriver. Blends are important because they allow us to create geometry using one feature whereas it might take us several extrude and revolve features to perform the same task.

Objective

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

- Create blend features.

Scenario

You are assigned to create the blend that will form part of the gearbox housing. Use the sections provided to complete the blend feature.

Step 1. Create GEARBOX_BLEND.PRT using a parallel blend feature.

We are assigned to begin creating the front gearbox part. After creating the part using the default template, we begin to create the initial feature, which is a blend feature. We start by importing the sections. Note that the second blended section is the same as the initial section because we want a constant profile.

We import a different sketch for the final section. Note that this section has multiple points on it.

There is the same number of points (10) on this section as there is for the previous section.

The number of connecting points should match to ensure a smooth transition in the blended feature. We line up the initial points of the sections to also ensure a smooth transition. Otherwise our feature will have a “twisted” look.

1. Create a new part named GEARBOX_BLEND.PRT.
2. Click **Insert > Blend > Protrusion > Done > Done**.
3. Select datum FRONT, and click **Ok > Default**.
4. Accept the default references and click **Sketch > Data from File**. Open GEARBOX_SECTION1-2.SEC.
5. Specify a Scale of **1** and a Rotation of **0**.

6. Drag the location handle to snap the centerlines in the section to the sketcher references, as shown in the following figure.
7. Complete the section placement.

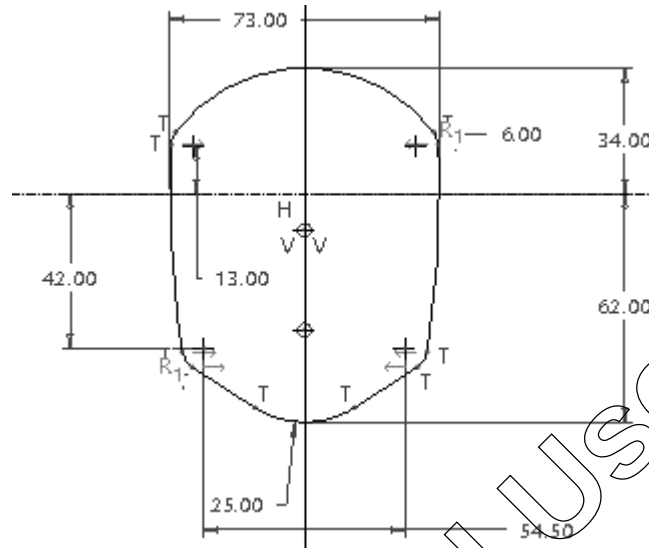


Figure 4: Importing the Initial Blend Sketch

8. Right-click and select **Toggle Section**.
9. Sketch using the GEARBOX_SECTION1-2.SEC file.
10. Specify a Scale of **1** and a Rotation of **0**.
11. Drag the location handle to snap the section directly over the previous section.
12. Complete the section placement.
13. Right-click and select **Toggle Section**.
14. Sketch using the GEARBOX_SECTION3.SEC file.
15. Drag the location handle to snap the section to the sketcher references, as shown in the following figure.
16. Specify a Scale of **1** and a Rotation of **0**.
17. Complete the section placement.

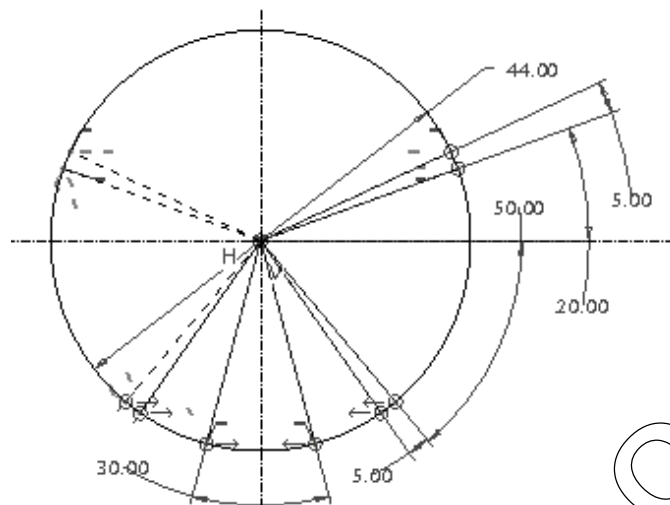


Figure 5: Importing the Final Section's Sketch

18. In the current section, select the vertex corresponding to the start point in the previous sections. Right-click and select **Start Point**.
19. Complete the sketch.
20. Enter **9** as the depth for Section 2, and **43** as the depth for Section 3.
21. Complete the feature.

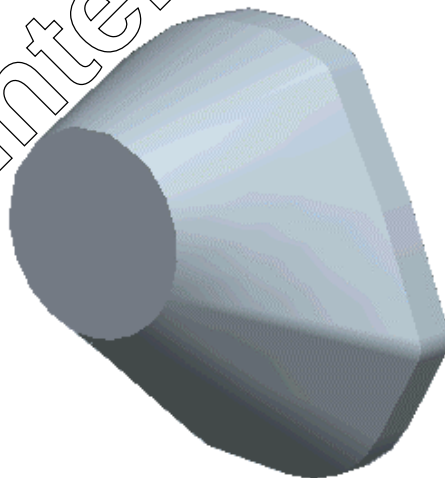


Figure 6: Completed Blend Feature on GEARBOX_BLEND.PRT

22. Save the model.
23. Close the window.
24. Erase all models from memory.

This completes the exercise.

Exercise 3: Sweeping Along a 3-D Trajectory

Objective

In this exercise, you learn how to create a swept protrusion using a 3-D trajectory.

Scenario

You are assigned to complete the coil by creating a sweep that will represent the coil's wire along a 3-D curve.

Step 1. Create the COIL.PRT wire using a sweep.

1. Open COIL.PRT.
2. Create a variable section sweep using the trajectory and section shown the following figure.

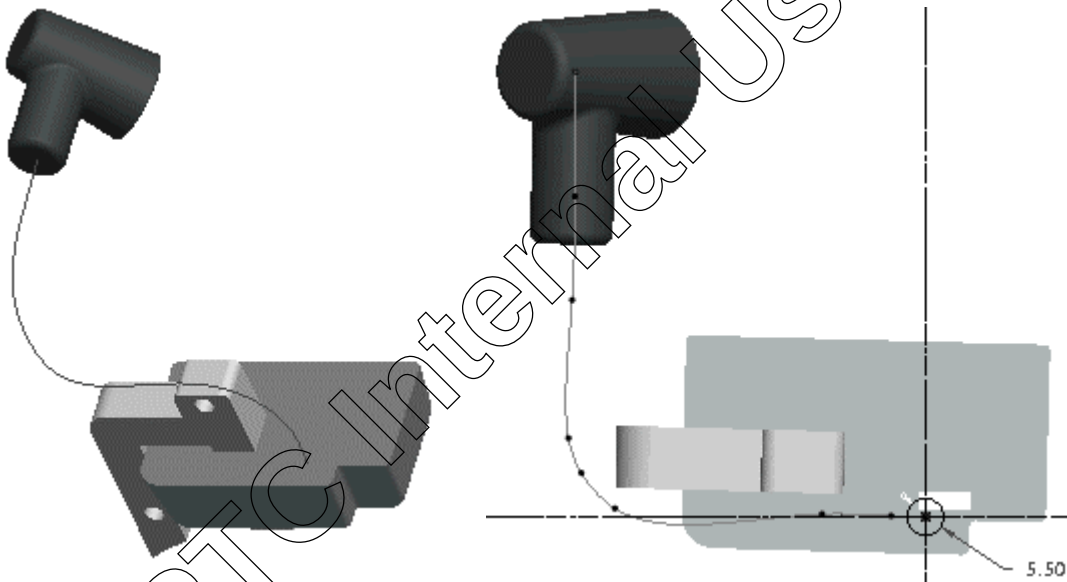


Figure 7: Sketching the Sweep

3. Using the dashboard, specify Solid, and complete the feature.
4. Apply the black appearance from the drill palette to the appropriate surfaces, as shown the following figure.



Figure 8: Completed COIL.PRT

5. Save the model
6. Close the window.
7. Erase the model from memory.

This completes the exercise.

Exercise 4: Creating a Blend Using Grouped Datum Features (Challenge)

Objective

In this exercise, you learn how to create local groups and populate them with features.

Scenario

You are assigned to create the initial blade of the flywheel. The design specification document recommends using a thin protrusion to create this feature.

Step 1. Insert a blend on FLYWHEEL.PRT.

1. Open FLYWHEEL.PRT.
2. Create a datum plane named ANGLE through the CRANK axis and **11.25** degrees from the FRONT plane.
3. Click **Insert > Blend > Thin Protrusion > Done > Done**. Select datum TOP as the sketching plane.
4. Accept the default direction, click **Bottom**, and select the ANGLE plane.
5. Select the CRANK axis and the ANGLE plane as references.
6. Sketch 2 construction circles and an arc to form the base of the blade, as shown in the following figure.

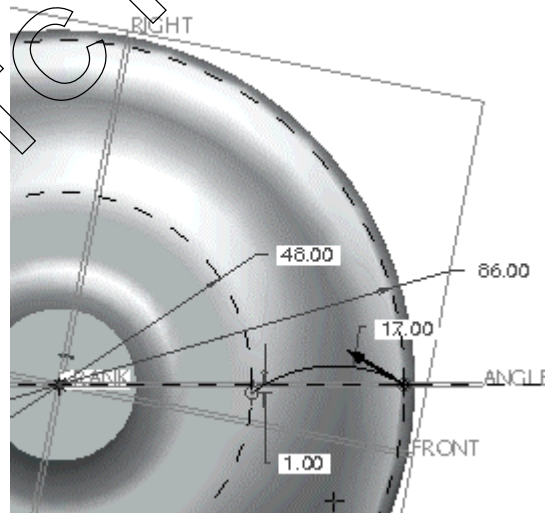


Figure 9: Creating the Base Blade Sketch

7. Toggle the section and sketch an arc to form the tip of the blade, as shown in the following figure.

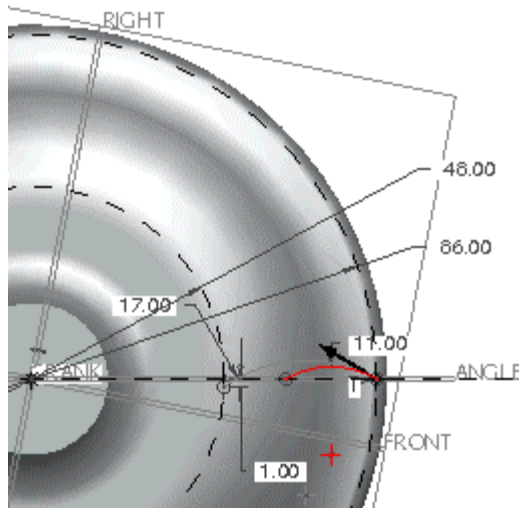


Figure 10: Creating the Upper Blade Profile

8. Complete the sketch, click **Both**, and enter **2.1** as the thickness.
9. Click **Thru Until > Done**, select the FIN_HEIGHT plane, and complete the feature.

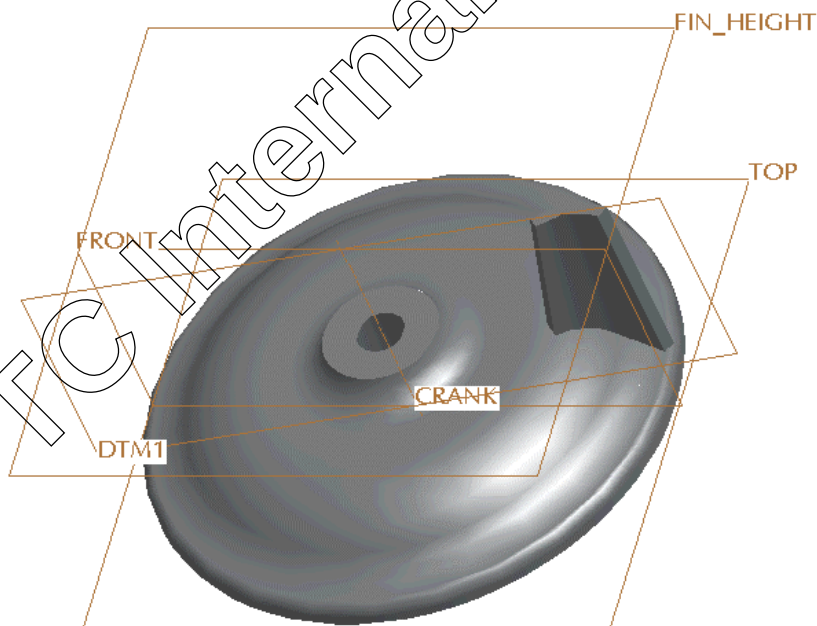


Figure 11: Completed Blend

10. Click **Edit > Feature Operations > Group > Local Group**. Enter **Blade** as the name. Select the ANGLE plane and the protrusion and click **Done**. Refer to the following figure.

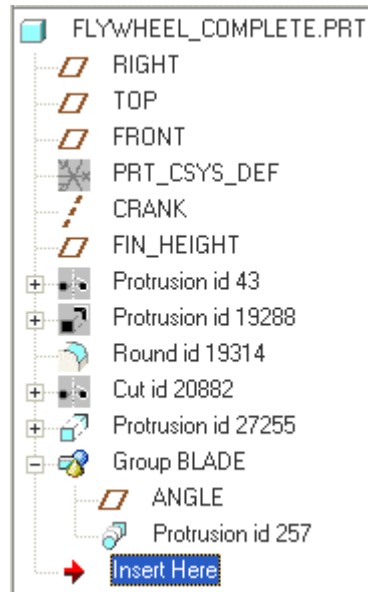


Figure 12: BLADE Group in Model Tree

11. Save the model
12. Close the window.
13. Erase the model from memory.

This completes the exercise.

Summary

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

- Create swept and blended features from 2-D sketches.
- Describe how creating sketch-based features affects parent/child relationships.

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Project I

Introduction

Using the Pro/ENGINEER Wildfire, complete the following parts based upon the materials taught up to this point in class.

Objectives

After completing this module, you will be able to:

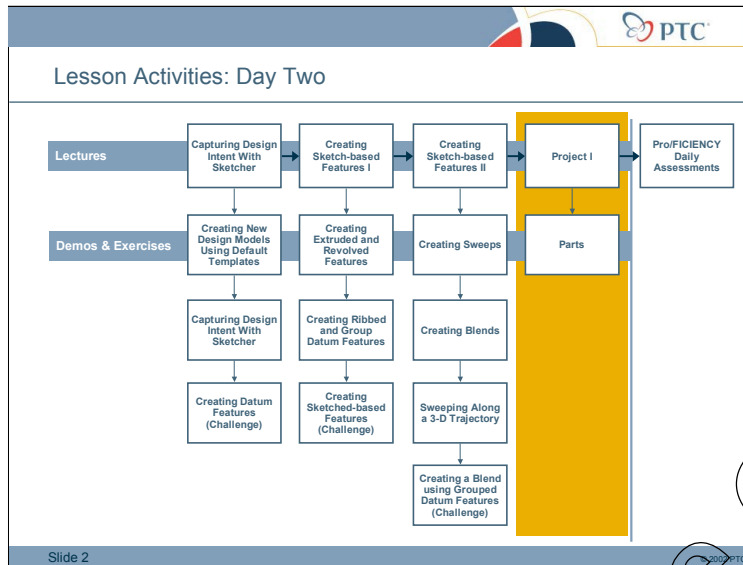
- Create the Piston Pin.
- Create the Piston.
- Create the Connecting Rod.
- Create the Crankshaft.
- Create the Engine Block.
- Create the Impeller Housing.
- Create the Frame.

Fast Track to Pro/ENGINEER Wildfire

Module 8: Project I



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Duration

Lecture: 10 mins

Labs: 1 hr 50 mins

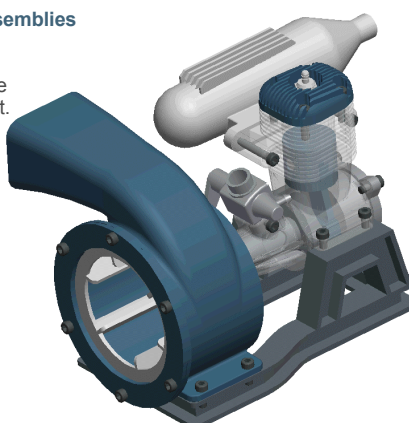
Total: 2 hr

PTC

Project Lab: The Air Circulator

AC-40: 16 Parts in 4 Assemblies

- Start from 'scratch'.
- Completed models are saved for each Project.
- Completed models are downloadable for students.



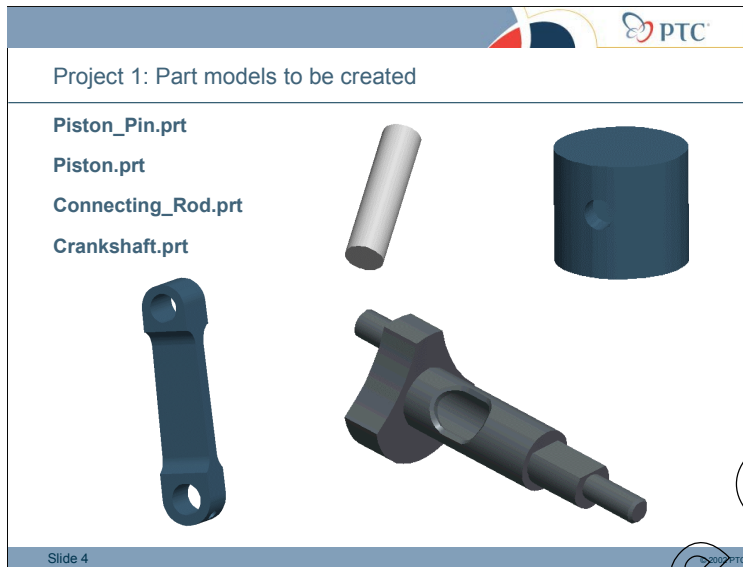
Slide 3

In the project, you will be creating the AC-40 air circulator. You will be starting this project at the beginning, which means that you can continue to work on this project once you leave the classroom. Hence you do not have to worry about completing the project.

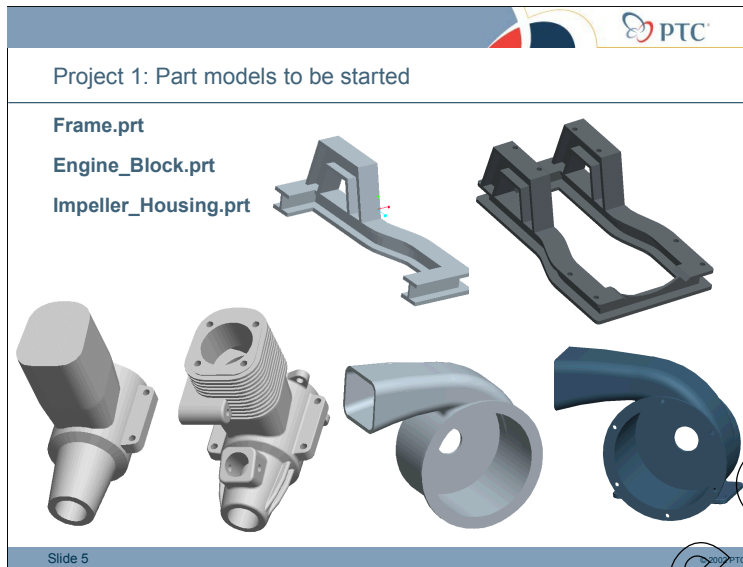
You should save the project within the Project folder in the classroom folder structure.

Additionally, the completed classroom files will eventually be uploaded to ProjectLink.

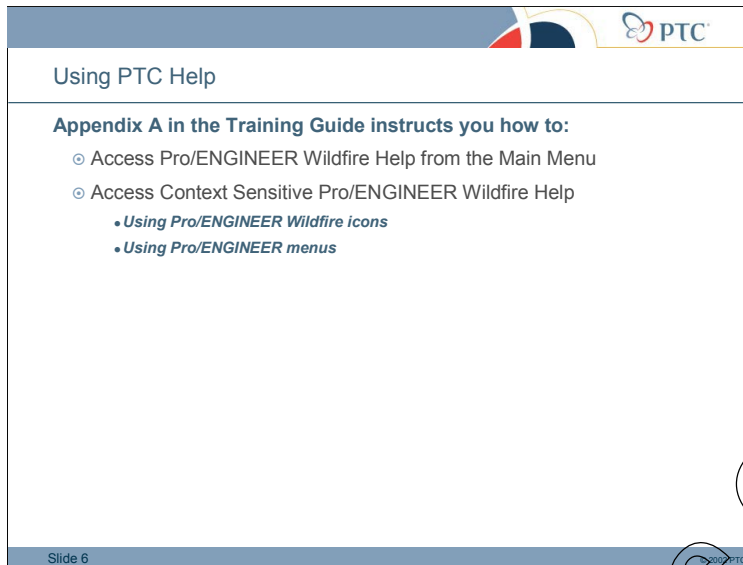
Finally, the project is not meant to be finished during the class. It is expected that most students will be finishing project 3 at the end of Friday. Project 4 can be completed at home.



There are no notes with this slide. This slide is only to display the parts that students will complete during this project module.



There are no notes with this slide. This slide is only to display the parts that students will start building during this project module.



Note to Instructors:

The main purpose of this slide is to direct you to Appendix A. Once at Appendix A, discuss with students the other methods to get help within Pro/ENGINEER Wildfire. In particular, talk about the Context Sensitive help.

You may wish to do a quick 2 minute demonstration on using context sensitive help. Perform this demonstration first by loading a simple part (for example the Cylinder.prt) and then by using the following picks:

- **Help > What's This?** and choose the **Regenerate Icon** at the top of the screen. Explain how Help automatically retrieves the information on Regenerate. Proceed to quick demo 2 (next bullet).
- **Help > What's This?** and choose the **Round Tool icon** at the right of the screen. Again, explain how help retrieves all information about the icon you chose. Proceed to quick demo 3 (next bullet).
- **Help > What's This? > Insert > Draft.** Explain that Context Sensitive Help also allows you to find help within the Pro/ENGINEER Wildfire menus. Proceed to quick demo 4 (next bullet).
- **Edit > Setup > Help > What's This?** and choose **Shrinkage** from the Part Setup Menu Manager. Explain that Context Sensitive Help also allows you to get help on the Menu Manager picks too.
- End of demos – Let students begin Project I.

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Module 8 Lab Exercise

Scenario

ACME Inc. is a multinational conglomerate that develops and markets several consumer, industrial, and defense goods. The Light Industrial Division of ACME Inc. creates a number of products including industrial fans, heating, air conditioning, pumps, and lighting. You work for the Light Industrial Division of ACME Inc., which has just started to use Pro/ENGINEER Wildfire for its product designs.

Upon returning from Pro/ENGINEER Wildfire training, you are assigned to create the AC-40 Air Circulator. As the name implies, the AC-40 is an air circulation device. Its primary purpose is to enable fast effective clean up of flooded homes and businesses that have no power due to the flood. Another variation of the AC-40 is currently being designed to use electric power. A further variation that includes an air compressor and condenser that will provide back-up air conditioning to high-rise buildings is also being considered.

In this initial phase of the project, you are to begin building the parts that will be assembled into the AC-40 based upon the tasks that you have learned thus far in the training course.

Objectives

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

- Create the Piston Pin.
- Create the Piston.
- Create the Connecting Rod.
- Create the Crankshaft.
- Create the Engine Block.
- Create the Impeller Housing.
- Create the Frame.

Objective 1: Creating the Piston_Pin

Step 1. Create PISTON_PIN.PRT.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from session. Otherwise, start Pro/ENGINEER Wildfire.
2. Using the folder navigator, browse to the following location:
c:\users\student\fast_track_330\Project. Make this folder the working directory.
3. Create a new part named PISTON_PIN.PRT.

Note:

Use the default template for all models in the project.

4. Using the Extrude tool, create a protrusion as shown in the following figure.

Design Intent:

The model is to be centered on the FRONT datum plane.

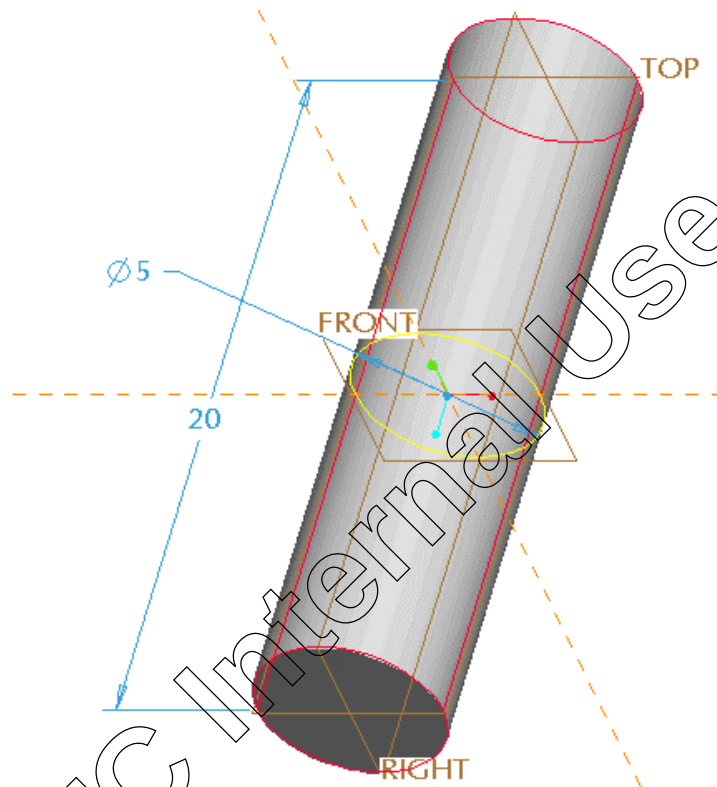


Figure 1: Creating a Protrusion.

Note:

Dimensions on features and sketches have been set to **blue** for printed clarity. If you wish to reproduce this effect, set the 'Letter' system color to RGB 52, 160, 220.

5. This model is now complete. Save the model and close the window.

This completes the exercise.

Objective 2: Creating the Piston

Step 1. Create PISTON.PRT. Create the main body of the model.

1. Create a new part named PISTON.PRT.
2. Using the Extrude tool, create a protrusion as shown in the following figure.

Design Intent:

This feature is to be created such that the TOP datum plane is the top of the model.

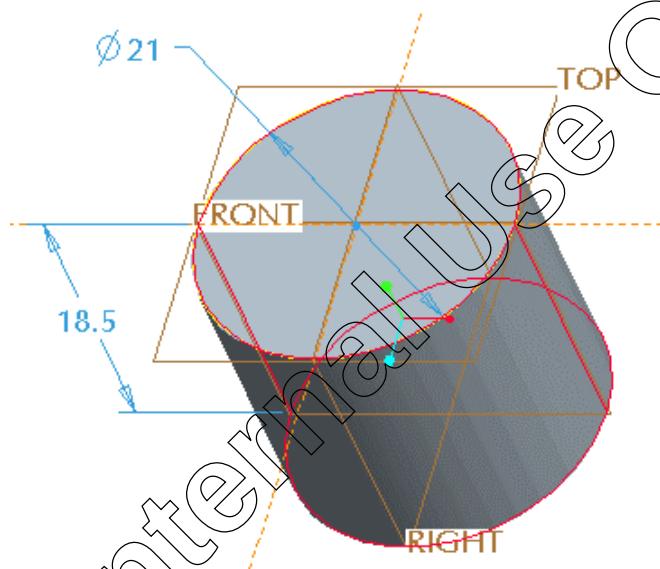


Figure 2: Creating a Protrusion

Step 2. Create a cut to hollow out the PISTON.PRT for the CONNECTING_ROD.PRT.

1. Start the Extrude tool and then create a datum plane offset downward from TOP as shown in the following figure.

Design Intent:

This feature is to be created such that it is dimensioned the TOP datum plane is the top of the model.

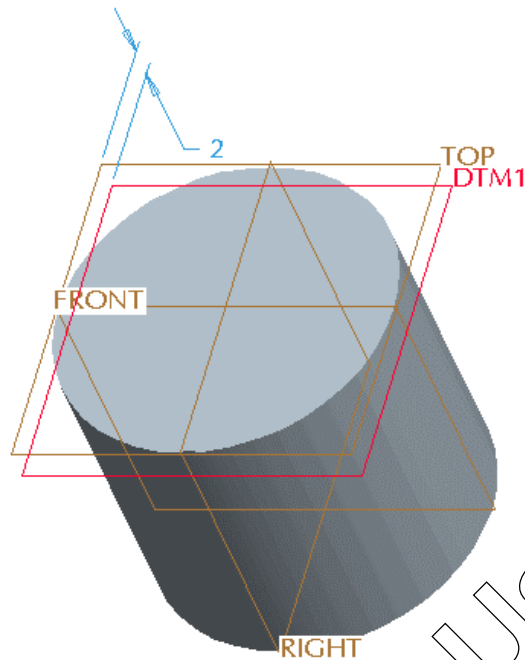


Figure 3: Creating Datum Plane

2. Select the previous plane (DTM1) as a sketching plane and sketch as shown below.

Design Intent:

The sketch should be oriented such that the RIGHT plane is vertical.

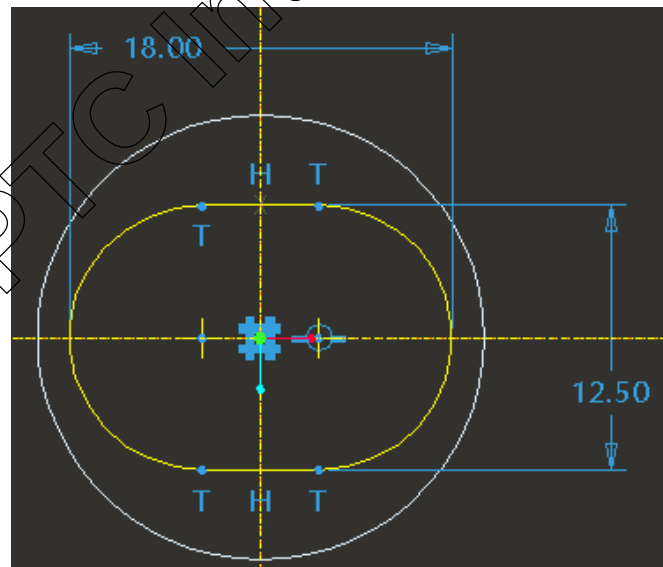


Figure 4: Creating a Sketch

3. Toggle the feature to Remove Material, and set the depth to extrude through the model as shown in the following figure.

Design Intent:

Regardless of the length of the original protrusion, the cut should extrude through the entire model.

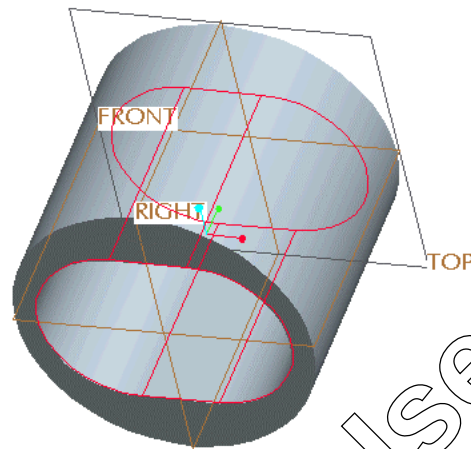


Figure 5: Creating a Cut

4. Rename the group to OVAL_CUT.

Step 3. Create a second cut.

1. Create a circular cut with the dimensioning scheme shown in the following figure.

Design Intent:

The depth should be dimensioned from the bottom surface of the model.

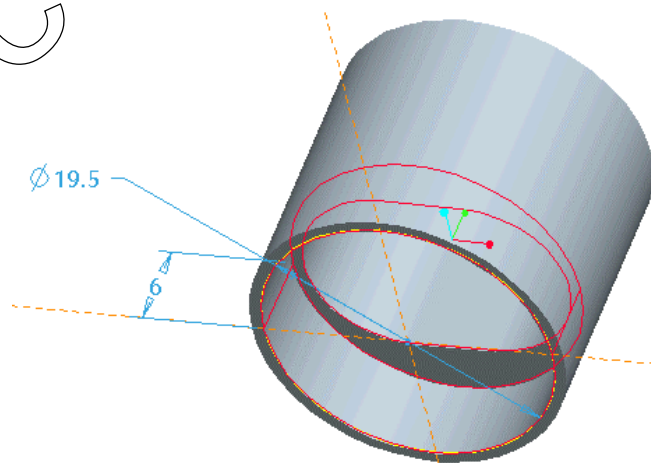


Figure 6: Creating a Second Cut

Step 4. Create a Hole to attach the PISTON_PIN.PRT.

1. Start the Hole tool and then create a datum plane offset from the TOP datum as shown in the following figure.

Design Intent:

The hole should be dimensioned from the top surface of the model.

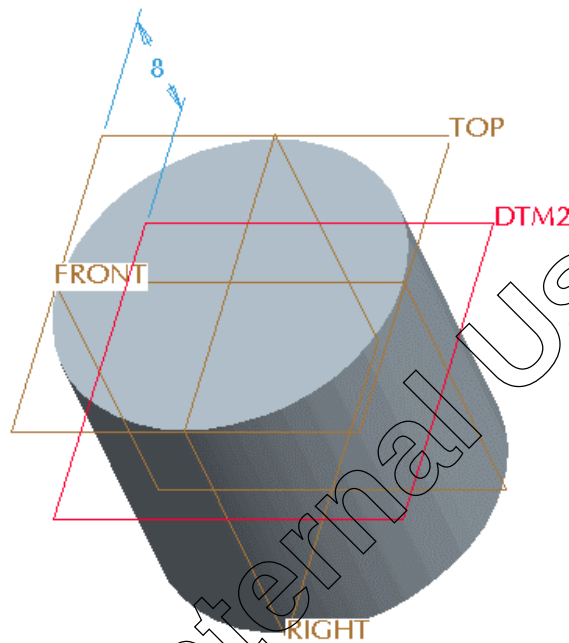


Figure 7: Creating a Datum Plane

2. Create a Datum Axis at the intersection of the previous datum plane (DTM2) and RIGHT as shown in the following figure.

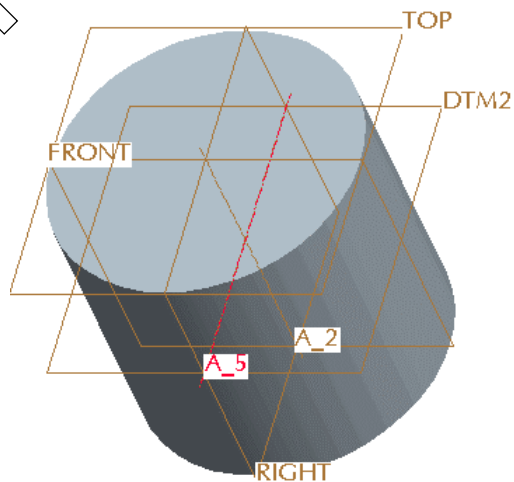


Figure 8: Creating a Datum Axis

3. Create a Hole on the previous axis (A_5), as shown in the following figure.

Design Intent:

The hole should be placed on the FRONT plane, and its depth should extend through the entire model in both directions.

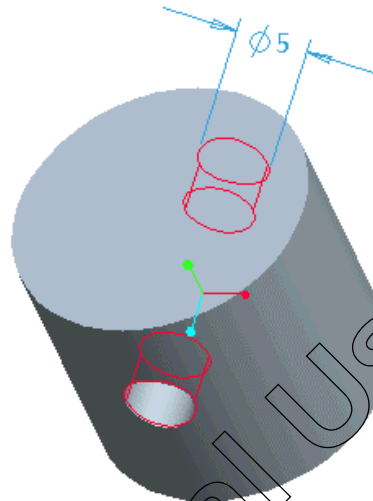


Figure 9: Creating a Hole

4. Rename the group to PIN_HOLE.
 5. This model is now complete. Save the model and close the window.
- This completes the exercise.

Objective 3: Creating the Connecting_Rod

Step 1. Create CONNECTING_ROD.PRT. Create the main body of the model.

1. Create a new part named CONNECTING_ROD.PRT.
2. Start the Extrude tool, and sketch the following on the FRONT datum plane as shown in the following figure. Sketch Axis Points on the arc centers. (HINT: Sketch two circles and then two 2-tangent lines.)

Design Intent:

The sketch should be oriented such that the RIGHT plane is vertical. Also, holes will later be placed on the arc centers.

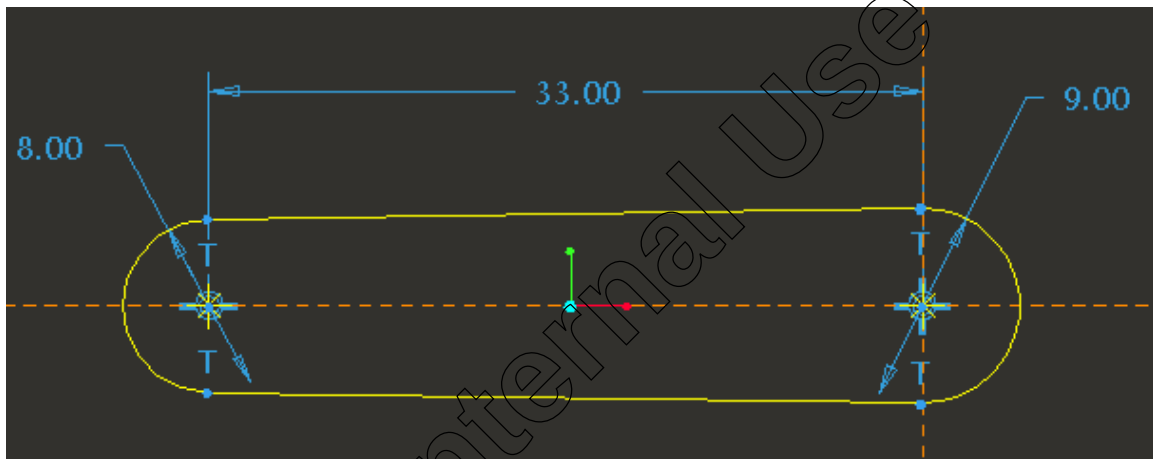


Figure 10: Creating a sketch

3. Set the depth as shown in the following figure.

Design Intent:

The feature should be symmetric about the FRONT plane.

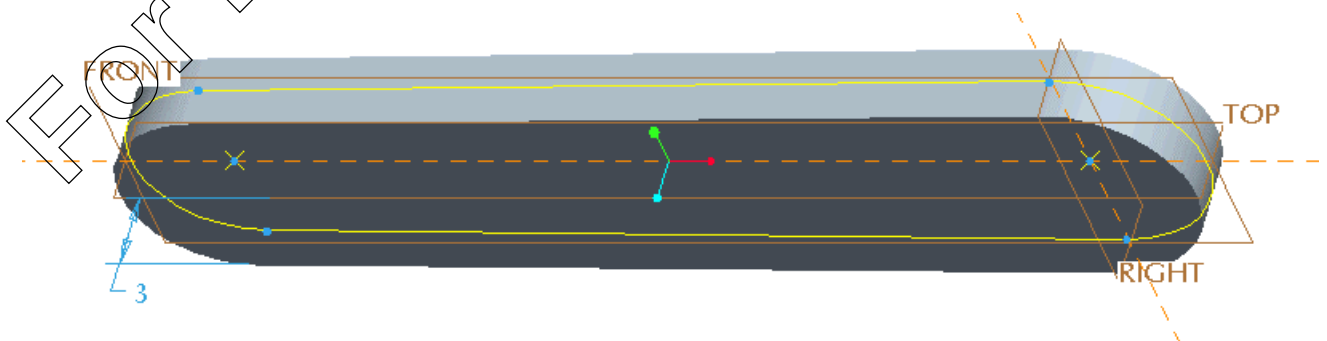


Figure 11: Creating a Protrusion

Step 2. Create protrusions on the ends of the model to support the CRANKSHAFT.PRT and PISTON_PIN.PRT.

1. Create a protrusion on the right side of the model as shown in the following figure. Use the Use Edge tool in sketcher to generate the arc and horizontal lines by referencing existing geometry.

Design Intent:

The feature should be symmetric about the FRONT plane, and should only use the dimensions shown.

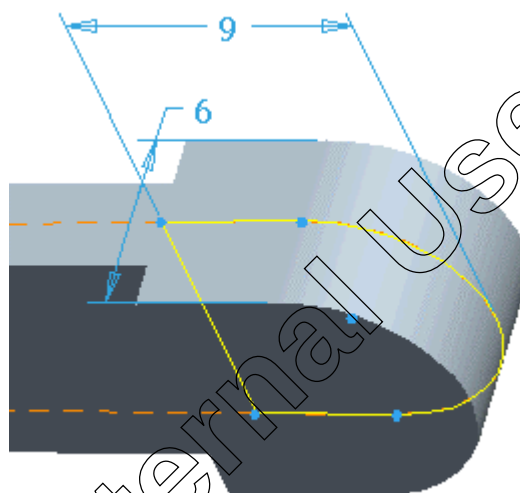


Figure 12: Creating a Protrusion

2. Create a similar protrusion on the left side of the model, as shown in the following figure.

Design Intent:

The feature should be symmetric about the FRONT plane, and should only use the dimensions shown.

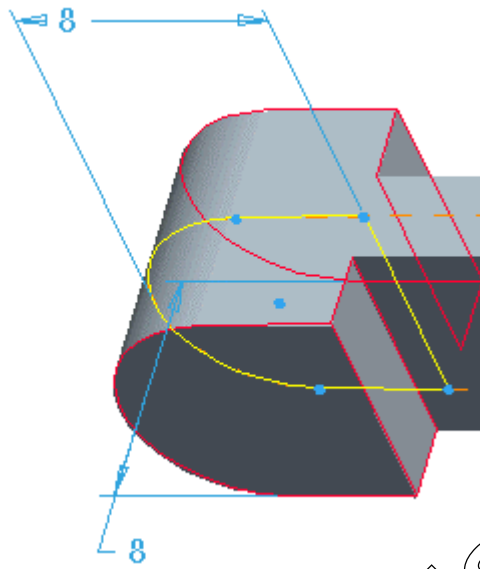


Figure 13: Creating a Protrusion

Step 3. Create holes on the ends of the model for the CRANKSHAFT.PRT and PISTON_PIN.PRT.

1. Create a hole placed on the axis on the right side of the model, as shown in the following figure.

Design Intent:

The hole depths should extend through the entire model.

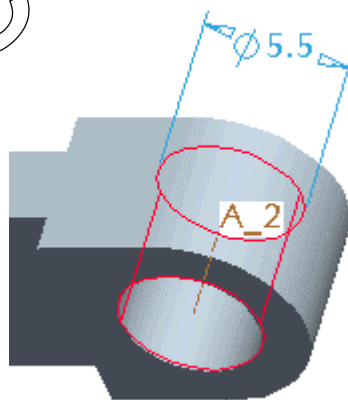


Figure 14: Creating a Hole

2. Create a similar hole placed on the axis on the left side of the model, as shown in the following figure.

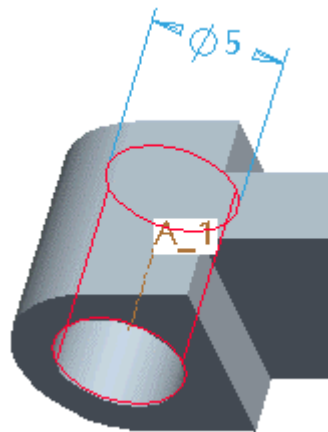


Figure 15: Creating a Hole

Step 4. Create a round to add strength.

1. Create a round with 4 surface-to-edge round sets as shown in the following figure.

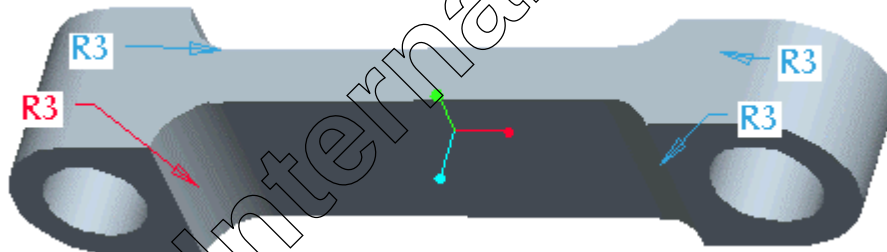


Figure 16: Creating a Round

Design Intent:

The rounds should be tangent to the surface and pass through the edge.

Step 5. Create a lubrication hole on the CRANKSHAFT.PRT end of the model.

1. Create a hole on the right side of the model, as shown in the following figure.

Design Intent:

The holes should be dimensioned radially from the TOP plane. Also, the hole depths should extend only through the next surface.

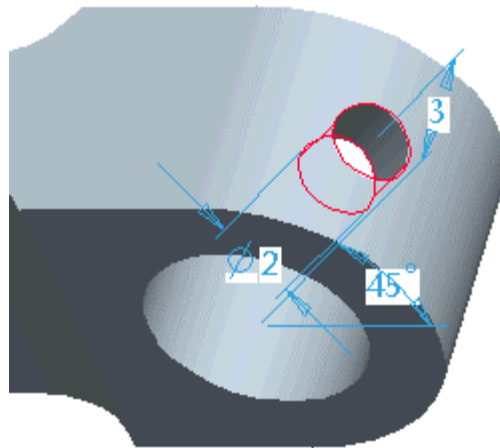


Figure 17: Creating a Hole

2. This model is now complete. Save the model and close the window.
This completes the exercise.

Objective 4: Creating the Crankshaft

Step 1. Create CRANKSHAFT.PRT. Create the main body of the model.

1. Create a new part named CRANKSHAFT.PRT.
2. Using the Extrude tool, create a protrusion as shown in the following figure.

Design Intent:

This feature is to be extruded from the FRONT datum plane, towards you.

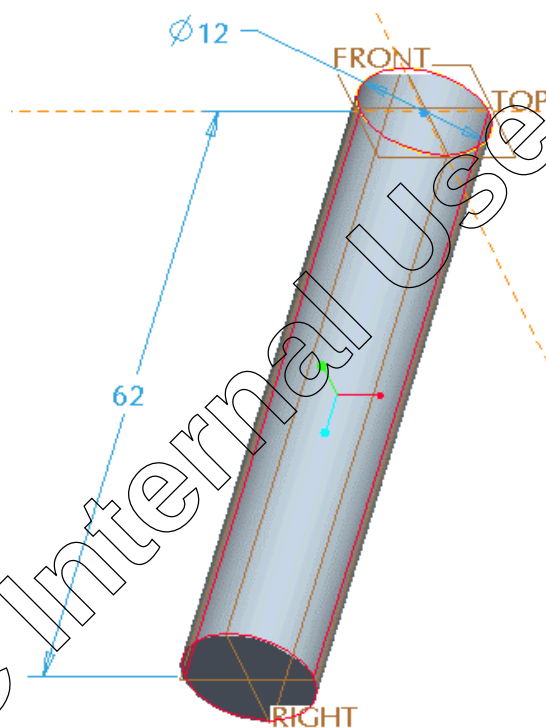


Figure 18: Creating a Protrusion

Step 2. Create the CRANKSHAFT.PRT lobe

1. Start the Extrude tool, and sketch on the FRONT plane as shown in the following figure. Use a construction circle to aid in creating the sketch. Sketch an axis point as shown, offset 8.50 from the horizontal reference.

Design Intent:

The sketch should be oriented such that the RIGHT plane is vertical. Also, a protrusion will later be centered on the axis point.

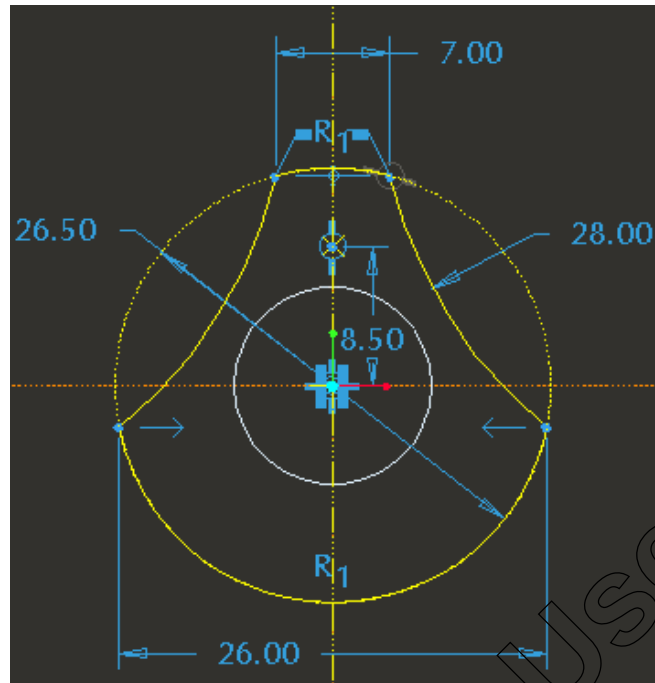


Figure 19: Creating a Sketch

2. Set the depth as shown in the following figure.

Design Intent:

The feature should be created from the FRONT plane forward.

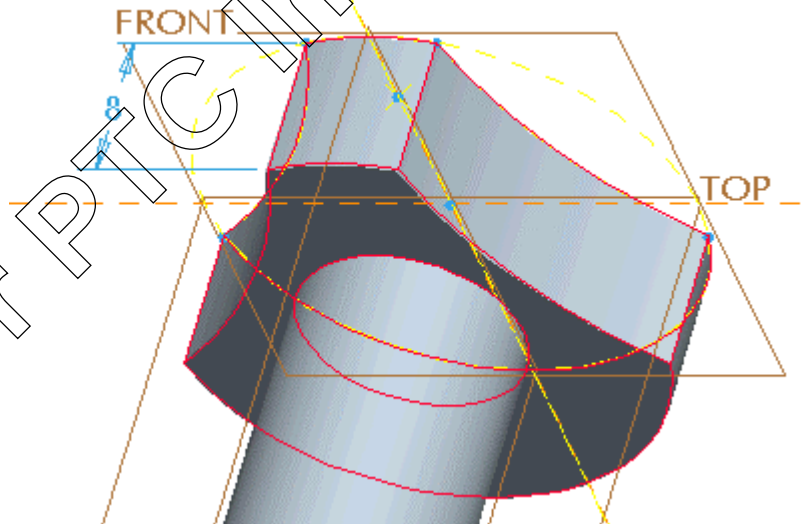


Figure 20: Creating a Protrusion

Step 3. Create the CRANKSHAFT.PRT pin.

1. Using the Extrude tool, create the protrusion as shown in the following figure.

Design Intent:

This feature should be centered on the axis from the previous protrusion.

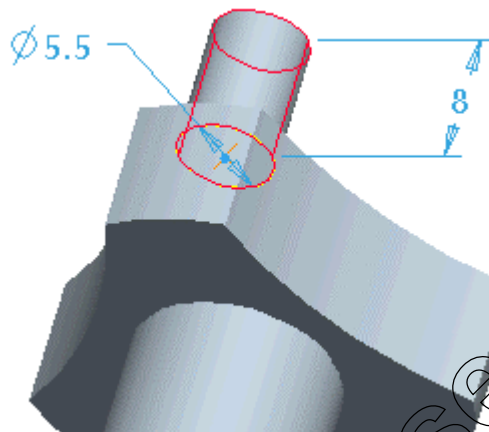


Figure 21: Creating a Protrusion

Step 4. Create a hole to hollow out a portion of the CRANKSHAFT.PRT.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial to the original cylindrical feature, and should be placed on the rear model surface.

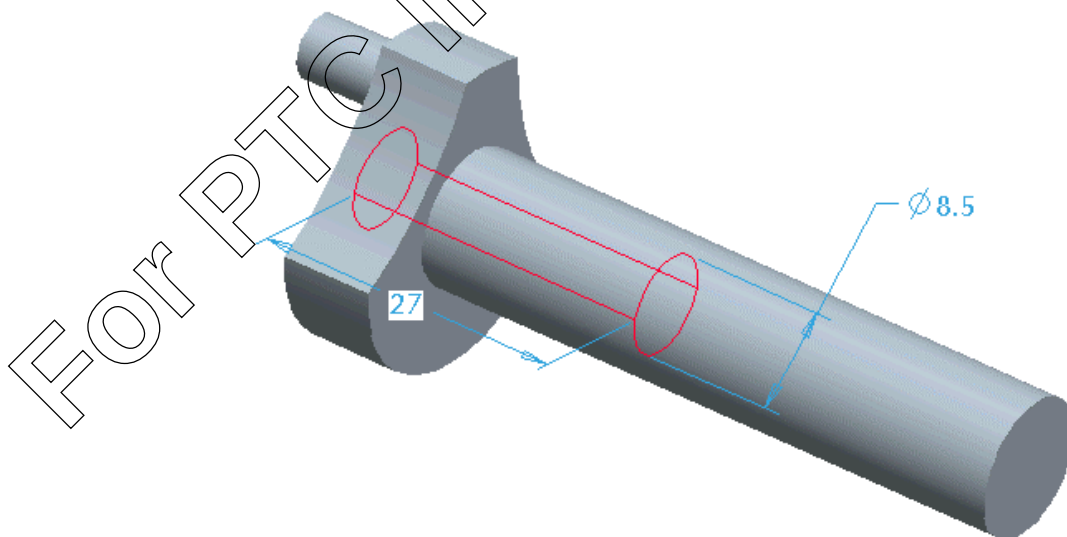


Figure 22: Creating a Hole

Step 5. Create a cut and chamfer to allow intake gases to enter the crankcase.

1. Start the Extrude tool, then create a datum plane as shown in the following figure.

Design Intent:

The datum plane should pass through the central axis and at an angle to TOP. This plane will be used as a sketching plane.

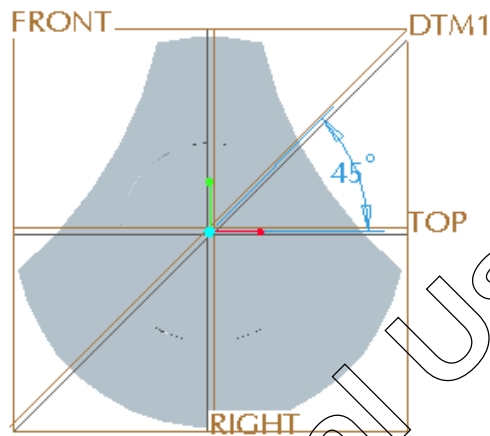


Figure 23: Creating a Datum Plane

2. Select the previous datum plane (DTM1) as a sketching plane, and sketch as shown in the following figure.

Design Intent:

Orient the sketch so that the FRONT plane is vertical. Reference the FRONT plane and the axis.

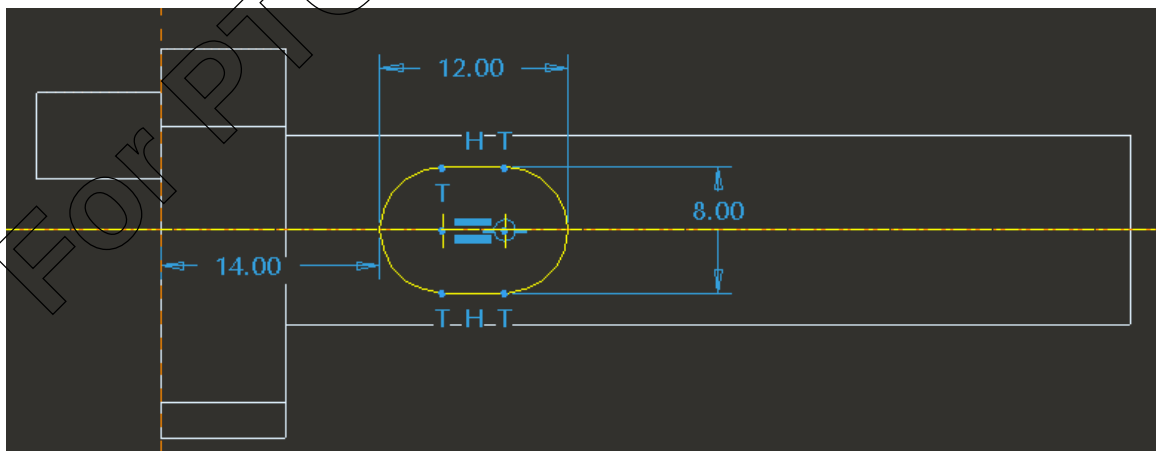


Figure 24: Creating a Sketch

3. Toggle the Extrude to remove material. Complete the cut as shown in the following figure.

Design Intent:

The depth for this feature should extend though the entire part.

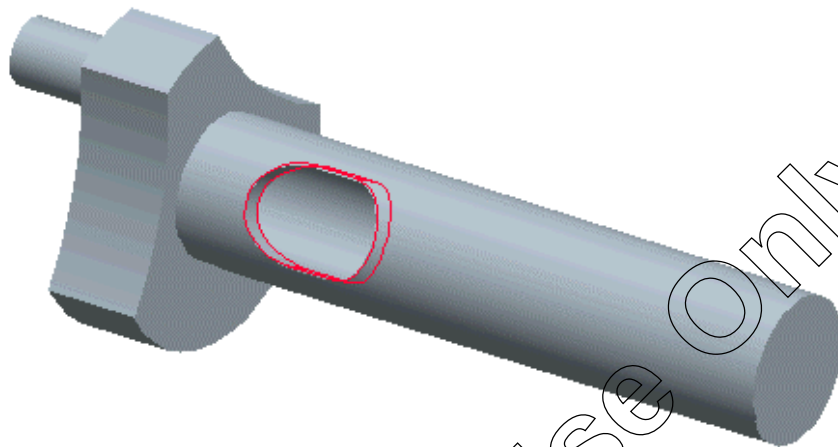


Figure 25: Creating a Cut

4. Rename the group to CRANK_CUT.

Step 6. Create a cut that will be used to interlock with the IMPELLER.PRT.

1. Start the Extrude tool. Select the surface shown in the following figure as the sketching plane.

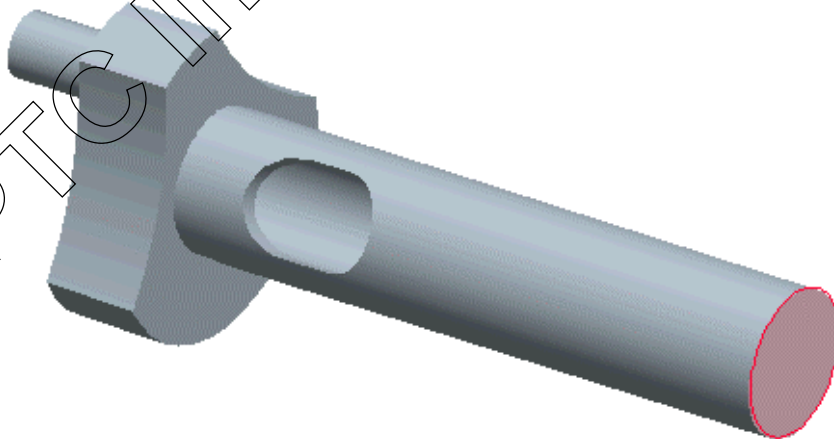


Figure 26: Sketching Plane

2. Sketch as shown in the following figure.

Design Intent:

Orient the sketch as shown in the following figure.

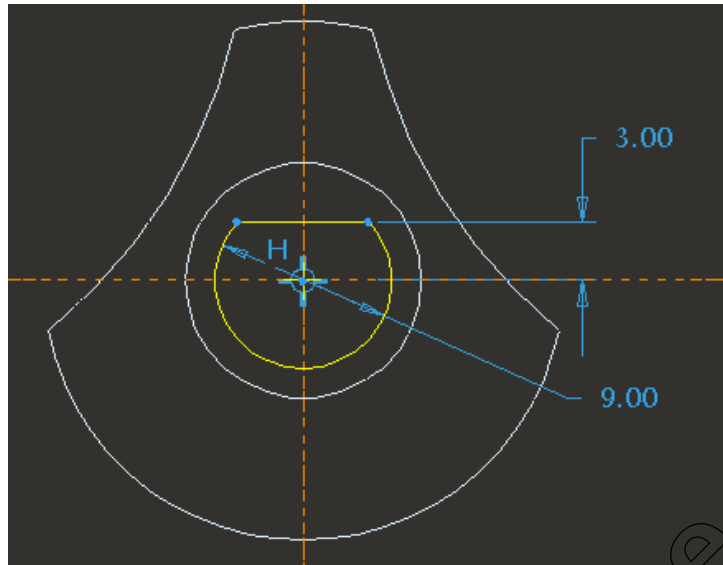


Figure 27: Creating a Sketch

3. Toggle the Extrude to remove material, and set the depth as shown in the following figure.

Design Intent:

The cut feature should remove material on the outside of the sketch.

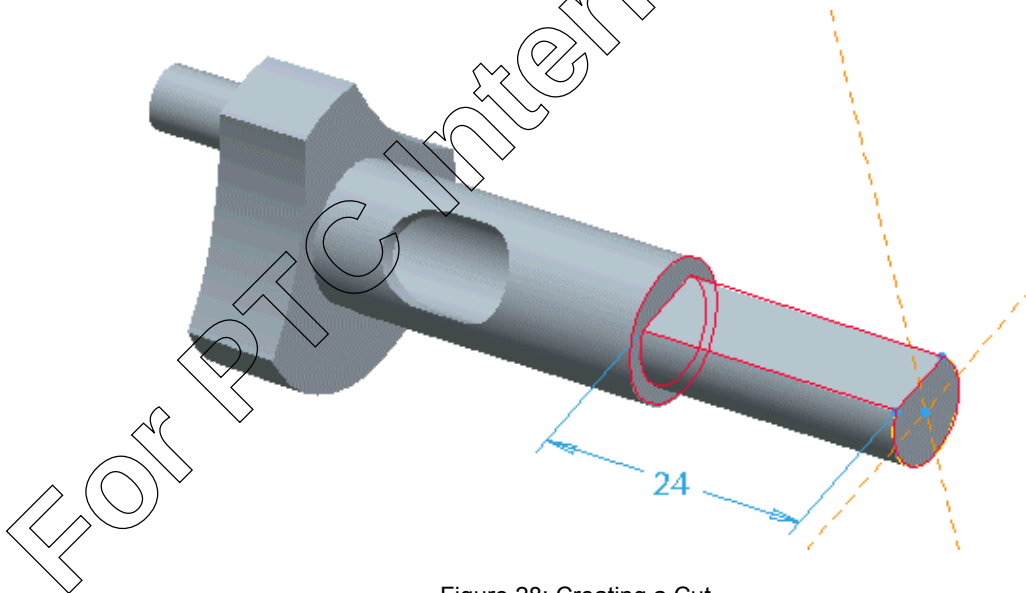


Figure 28: Creating a Cut

Step 7. Create a cut to form a circular shaft that will be threaded later.

1. Create a circular cut using the previous procedure, as shown in the following figure.

Design Intent:

The sketch for this feature should be concentric to the original cylindrical feature.

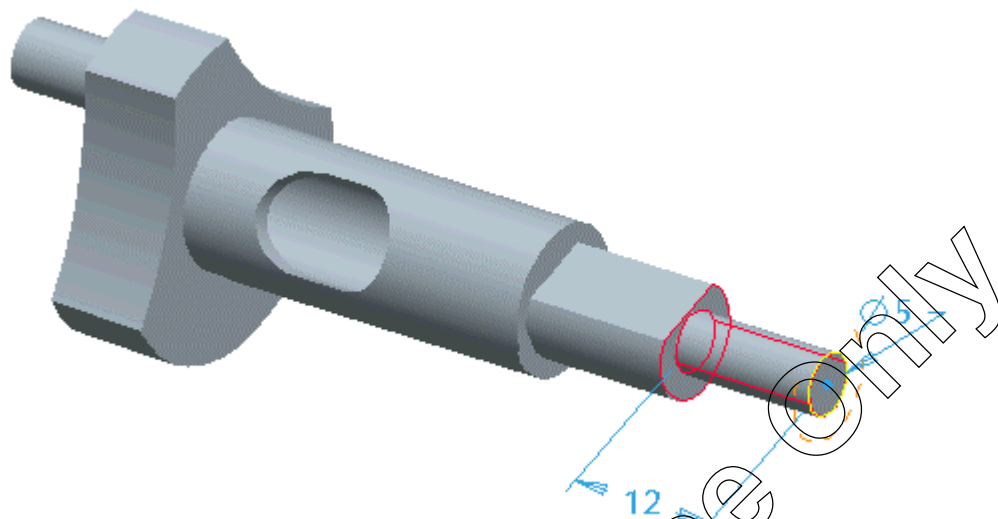


Figure 29: Creating a Cut

Step 8. Create a chamfer to bevel the edges of the cut and the end of the CRANKSHAFT.PRT.

1. Create a chamfer as shown in the following figure.

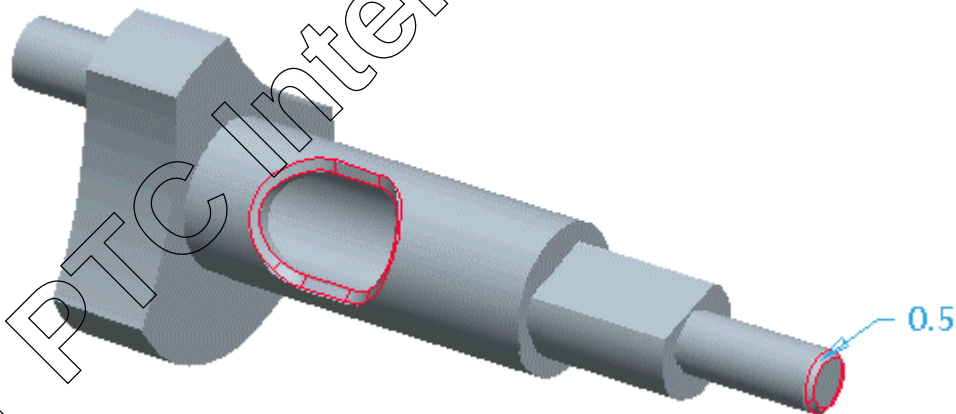


Figure 30: Creating a Chamfer

Design Intent:

Both legs of the Chamfer should have the same dimension (DxD).

2. This model is now complete. Save the model and close the window.

This completes the exercise.

Objective 5: Creating the Engine_Block

Step 1. Create ENGINE_BLOCK.PRT. Create the main body of the model.

1. Create a new part named ENGINE_BLOCK.PRT.
2. Using the Revolve tool, create a protrusion as shown in the following figure. Sketch on the RIGHT plane.

Design Intent:

Orient the sketch so that the TOP plane is horizontal.

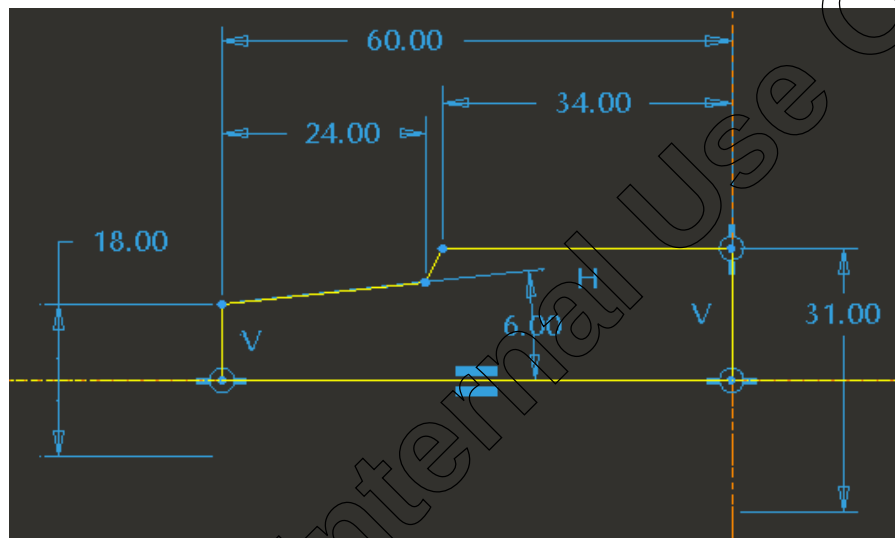


Figure 31: Creating a Sketch

3. Complete the feature.

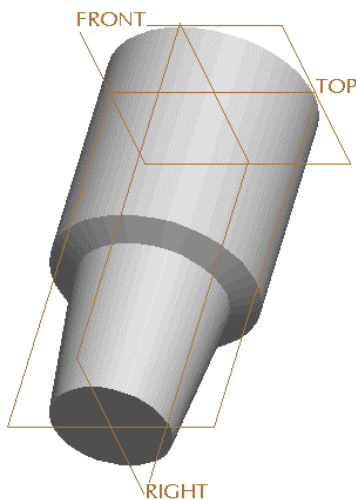


Figure 32: Creating a Protrusion

Step 2. Create datum features that will be used to construct other features.

1. Create a datum plane offset from FRONT, as shown in the following figure. Rename the plane to CTR.

Design Intent:

This plane will be used as a reference for the next feature.

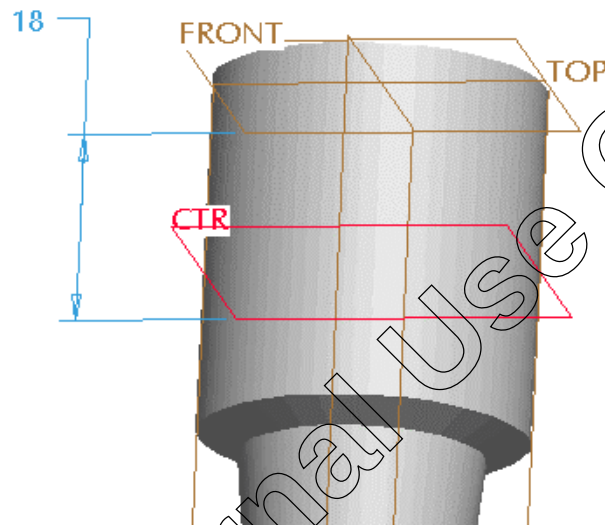


Figure 33: Creating a Datum Plane

2. Create a datum axis at the intersection of the RIGHT and the CTR planes, as shown in the following figure. Rename the axis to CYL.

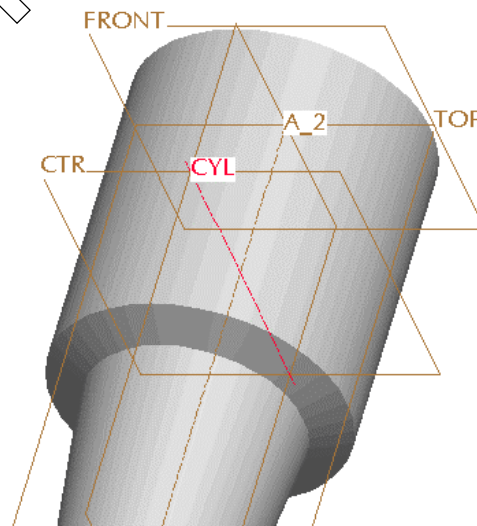


Figure 34: Creating a Datum Axis

Step 3. Create a protrusion forming the cylinder.

1. Start the Extrude tool and then create a datum plane offset from the TOP plane, as shown in the following figure.

Design Intent:

This plane will be used as a sketching plane for the next feature.

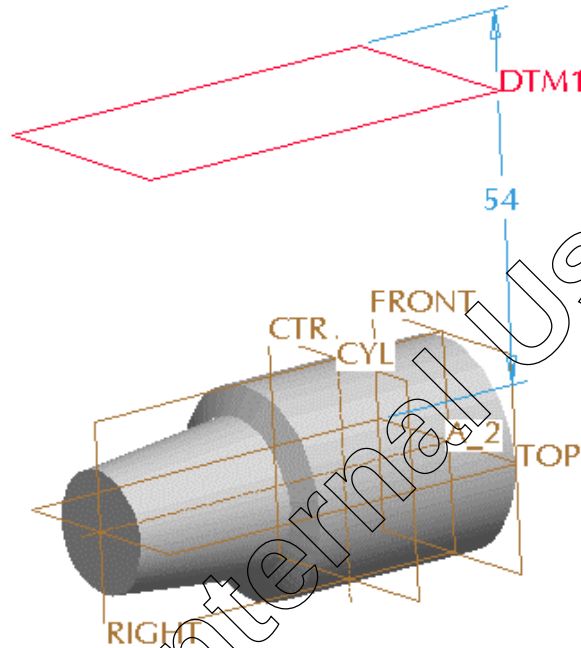


Figure 35: Creating a Datum Plane

2. Select the previous datum plane (DTM1) as a sketching plane, and sketch as shown in the following figure.

Design Intent:

Orient the sketch so that the RIGHT plane is vertical. The sketch should be symmetrical about vertical reference. The vertical dimensions should reference the CTR plane.

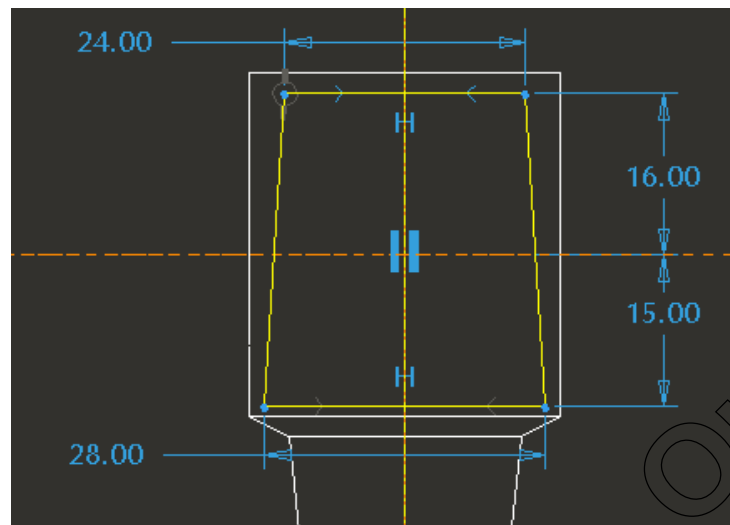


Figure 36: Creating a Sketch

3. Set the feature depth to extend up to the next feature, as shown in the following figure.

Design Intent:

The protrusion must conform to the cylindrical shape.

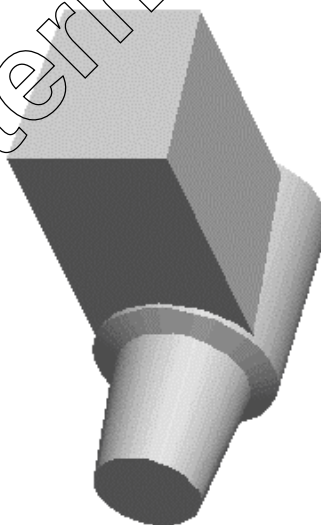


Figure 37: Creating a Protrusion

4. Rename the group to CYLINDER.

Step 4. Create rounds and draft to shape the cylinder.

1. Create a round as shown in the following figure.

Design Intent:

Since the rear edges need to be a larger radius than the front edges, create a single round feature with 2 round sets.

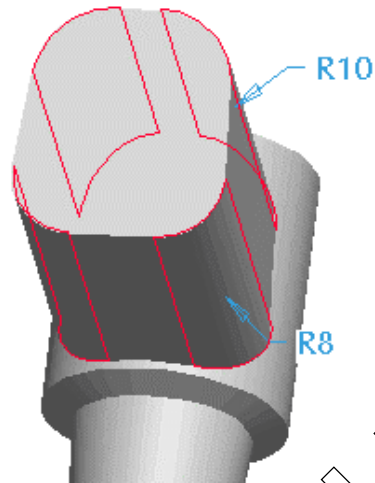


Figure 38: Creating a Round

2. Start the Draft tool, and then create a datum plane offset from the upper surface, as shown in the following figure.

Design Intent:

This plane will be used to split the draft.

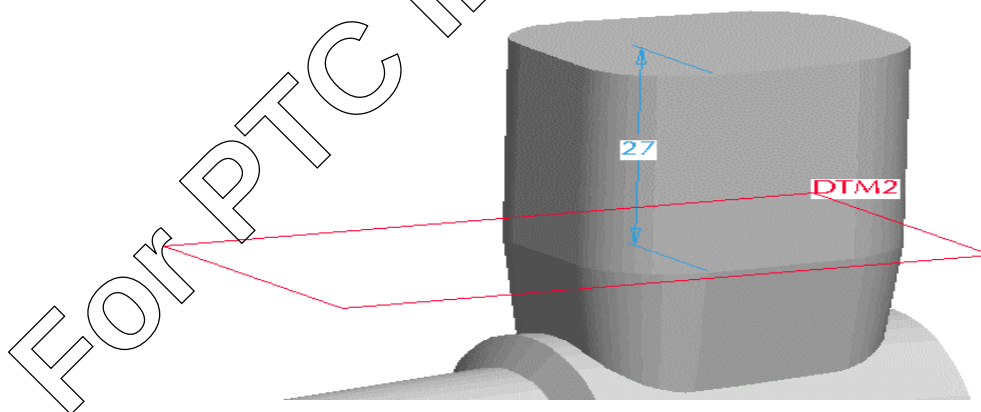


Figure 39: Creating a Datum Plane

3. Select a surface of one of the previous rounds to draft, and select the previous offset plane (DTM2) as the draft hinge. Create a draft of 6 degrees as shown in the following figure.

Design Intent:

The draft should be split by the draft hinge, and drafted only on the lower side.

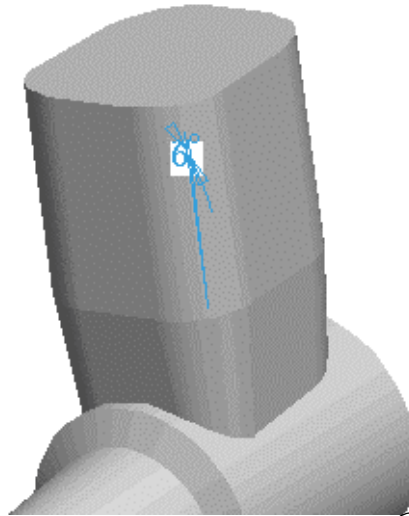


Figure 40: Creating a Draft Feature

4. Rename the group to DRAFT.
5. Create a round as shown in the following figure.

Design Intent:

This round makes the all the edges of the cylinder tangent.

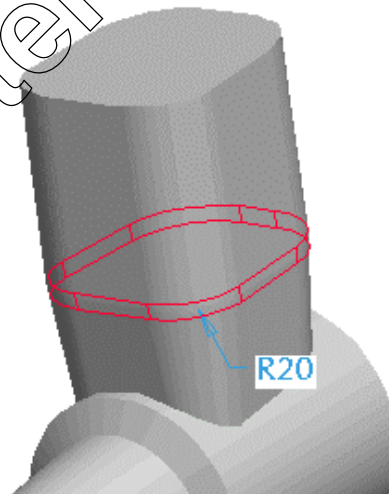


Figure 41: Creating a Round

Step 5. Create a protrusion to be used as a mounting flange.

1. Start the Extrude tool and create a sketch on the TOP plane as shown in the following figure. Sketch axis points on the arc centers.

Design Intent:

Sketch a closed loop using the references shown. The flange should be symmetric about the horizontal reference to the CTR plane. Coaxial Holes will be placed using the axis points.

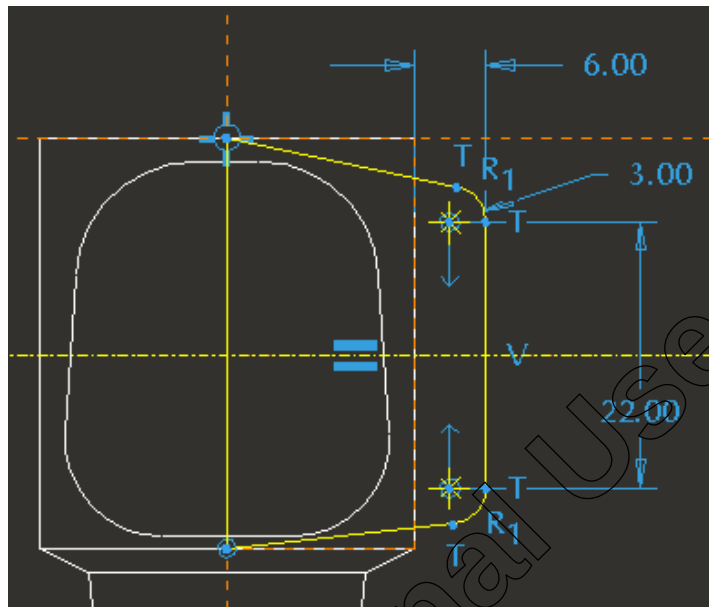


Figure 42: Creating a Sketch

2. Set the depth as shown in the following figure.

Design Intent:

The depth of the feature should extend from the TOP plane upward.

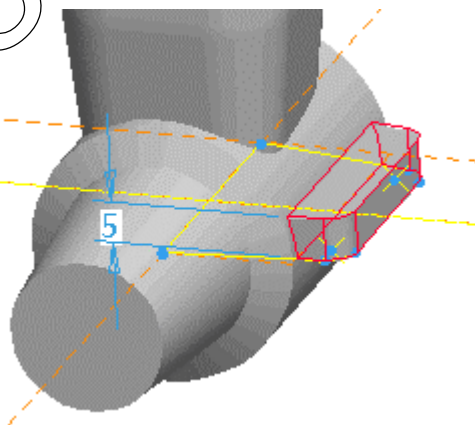


Figure 43: Creating a Protrusion

Step 6. Create holes for mounting bolts.

1. Create holes as shown in the following figures.

Design Intent:

The holes should be coaxial. The hole depths should extend through the next surface.

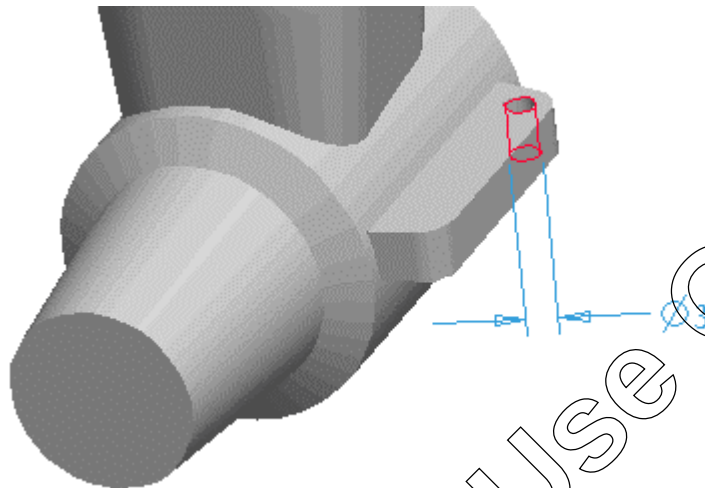


Figure 44: Creating a Hole

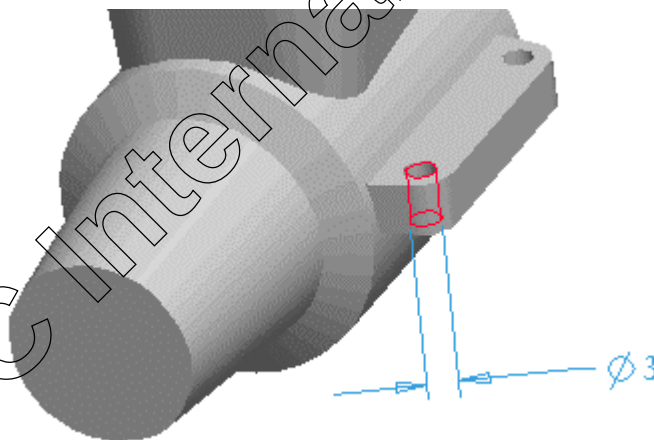


Figure 45: Creating a Hole

2. Create a group called MOUNT1 containing the flange and the holes.

Step 7. Create Holes for the crankshaft and the crankcase.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and should extend through the entire part.

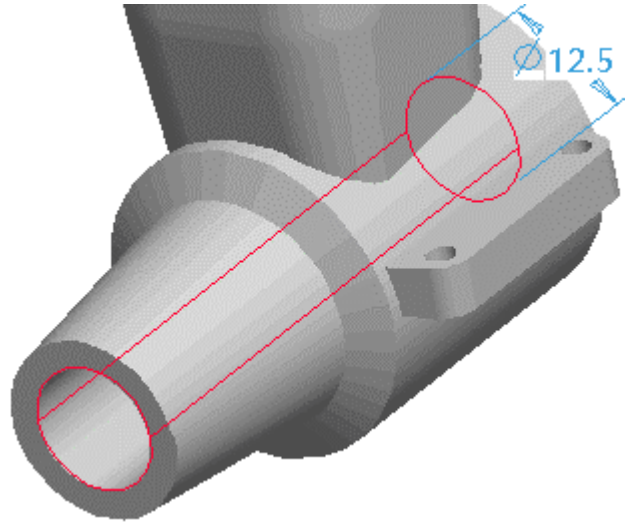


Figure 46: Creating a Hole

2. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and should have the depth shown.

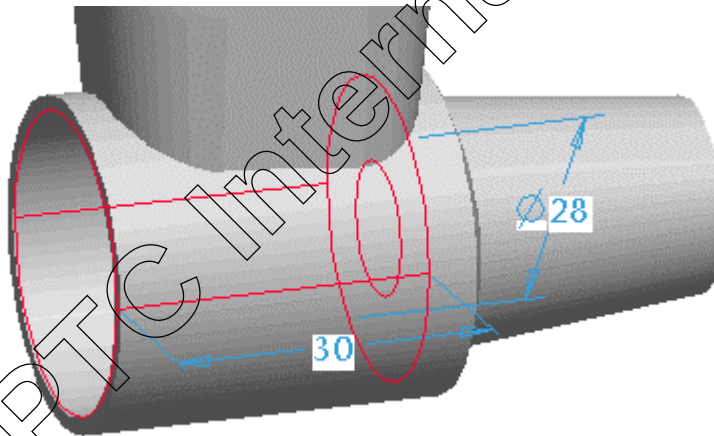


Figure 47: Creating a Hole

3. This model will be completed later in the project. Save the model and close the window.
This completes the exercise.

Objective 6: Creating the Impeller_Housing

Step 1. Create IMPELLER_HOUSING.PRT. Create the main body of the model.

1. Create a new part named IMPELLER_HOUSING.PRT.
2. Using the Extrude tool, create a protrusion as shown in the following figure.

Design Intent:

This feature should have asymmetrical depths from the FRONT plane as shown in the following figure.

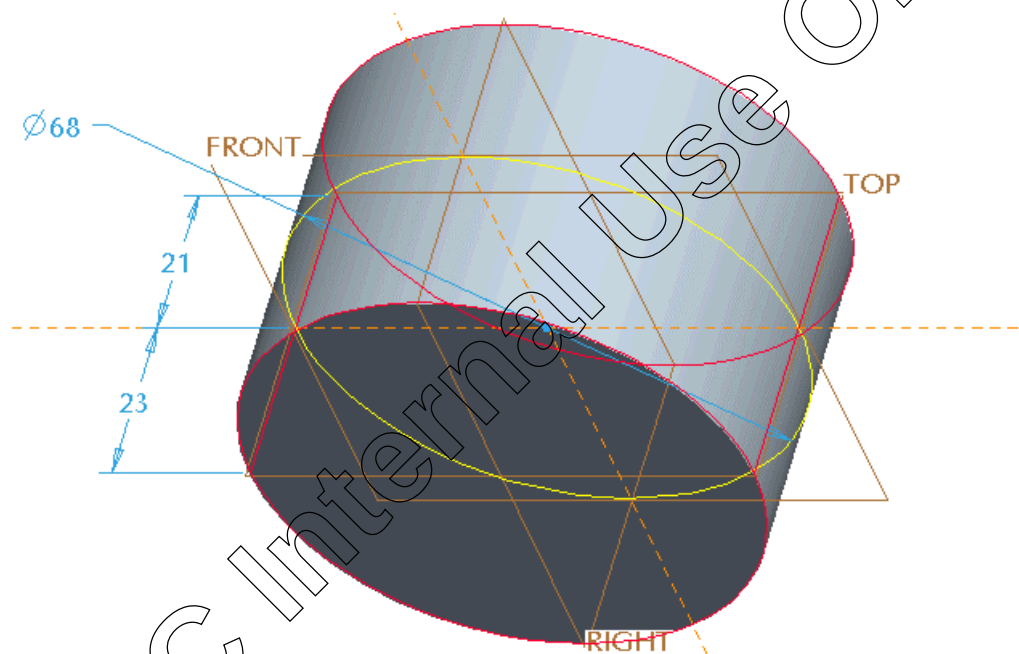


Figure 48: Creating a Protrusion

Step 2. Create a revolved protrusion to begin forming the curved portion of the housing.

1. Start the Revolve tool, and sketch on the TOP plane, as shown in the following figure.

Design Intent:

Orient the sketch so that the FRONT plane is horizontal. The Sketch should be symmetric about a centerline on the horizontal reference. The axis of revolution should be offset 20 from the vertical reference.

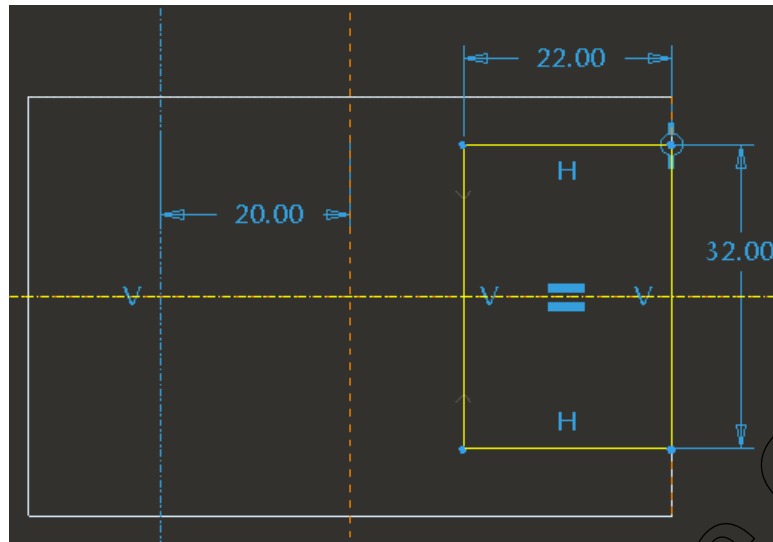


Figure 49: Sketching the Trajectory

2. Revolve the protrusion as shown in the following figure.

Design Intent:

The protrusion should be revolved 90 degrees.

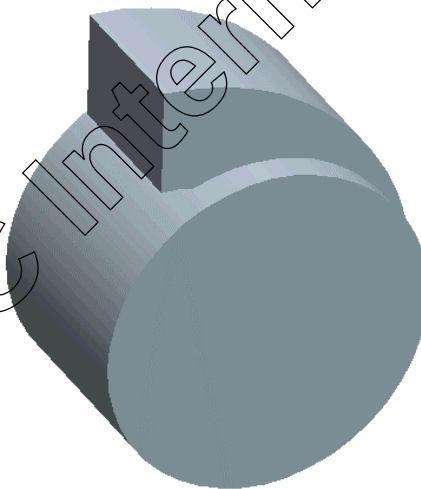


Figure 50: Completed Revolve

Step 3. Create a blend to complete the curved portion of the housing.

1. Insert a Blend as a Protrusion, using all defaults. Select the surface shown below for the sketching plane.

Design Intent:

This blend will use 3 parallel sections. The first section of the blend will be placed on the surface of the previous revolve.

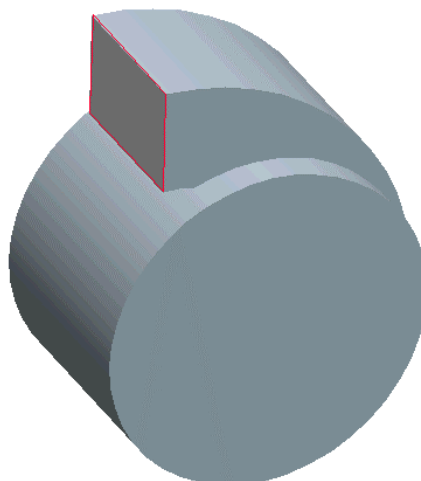


Figure 51: Sketching Plane

2. Start the Use Edge tool and select the Loop option. Select the rectangular surface shown in the following figure to create the first section. Sketch a centerline on the vertical reference.

Design Intent:

The first section of the blend needs to be the same size as the sweep section. Subsequent sections will need to be symmetric.



Figure 52: Selecting edges for Section 1

3. Toggle to section 2. Start the Offset Edge tool and select the Loop option. Select the rectangular surface again to create the second section, as shown in the following figure.

Design Intent:

The second section should be evenly offset from the revolve feature.

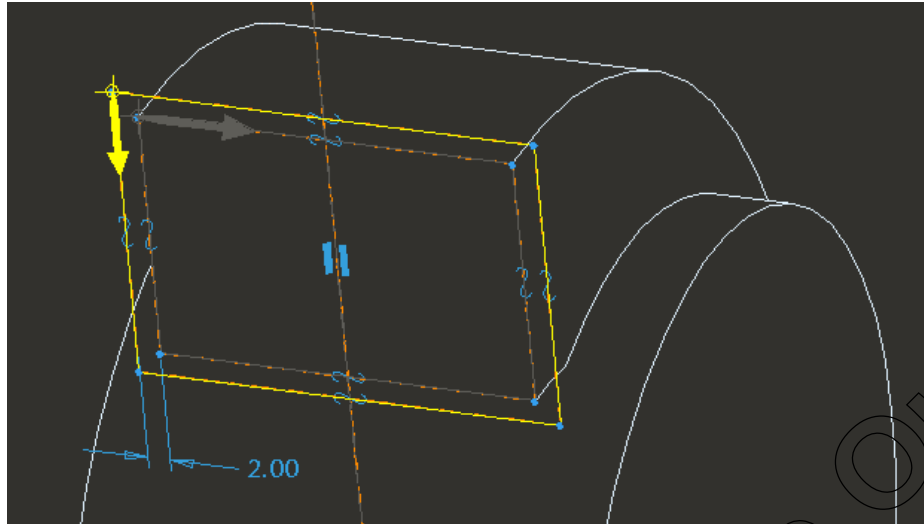


Figure 53: Sketching Section 2

4. Toggle to section 3. Sketch a rectangle, as shown in the following figure.

Design Intent:

The upper horizontal edge of the third section should lie over the upper horizontal edge of the second section. Also, the sketch is symmetric about the vertical reference.

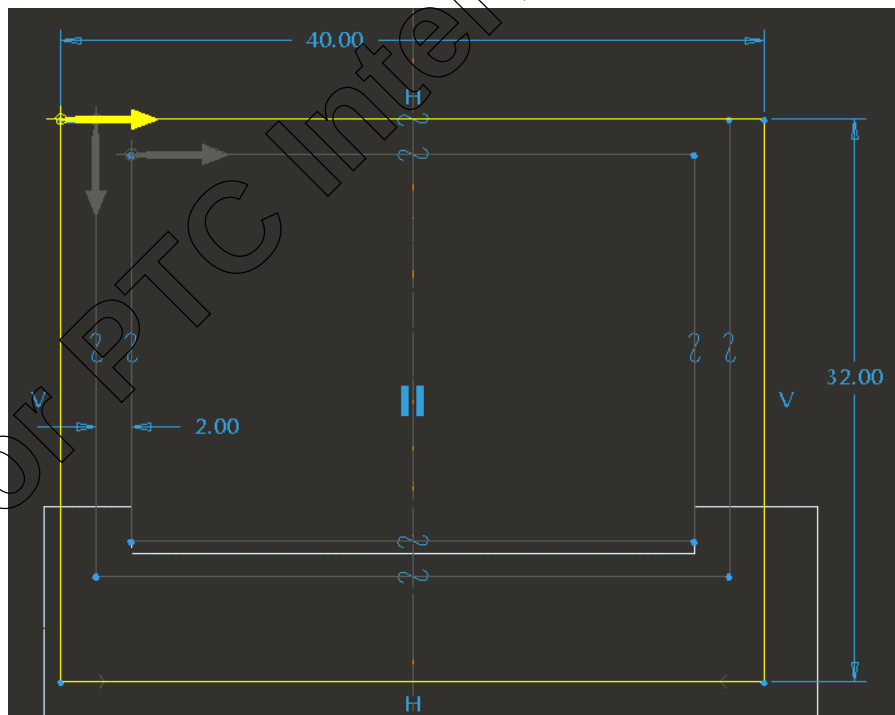


Figure 54: Sketching Section 3

Note:

The start points must be located in the same corner, but do not have to point in the same direction.

5. Complete the feature, entering the depths shown in the following figure.

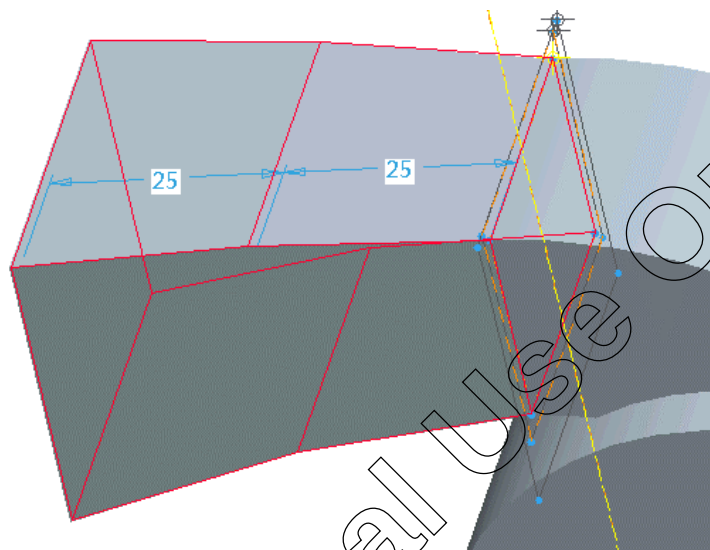


Figure 55: Blend Created

6. Notice that there is a 'sharp edge' at the location of section 2. Edit the definition of the blend, and toggle from Straight to Smooth, as shown in the following figure.

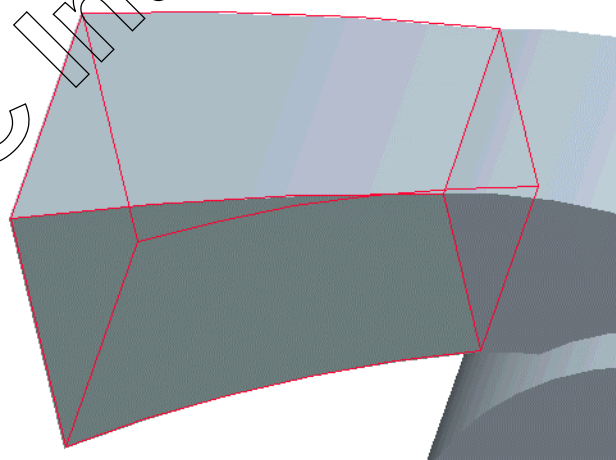


Figure 56: Blend with Smooth Transitions

Step 4. Create a series of rounds to shape the housing for maximum airflow.

1. Create a round on the 3 edges shown in the following figure.

Design Intent:

This round sets up tangent chains for the next round.

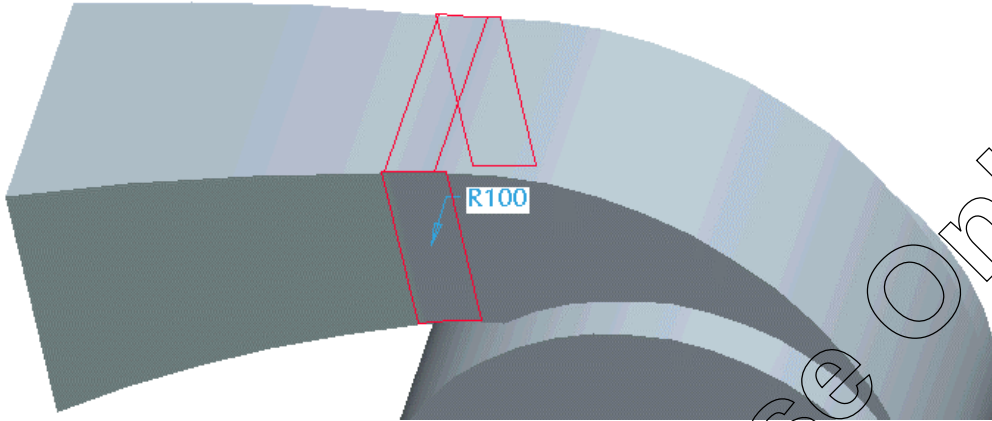


Figure 57: Creating a Round

2. Create a round on the 4 tangent edge chains as shown in the following figure. Use an intent chain for easy selection.

Design Intent:

The rounds on the 4 edges should be controlled by a single value.

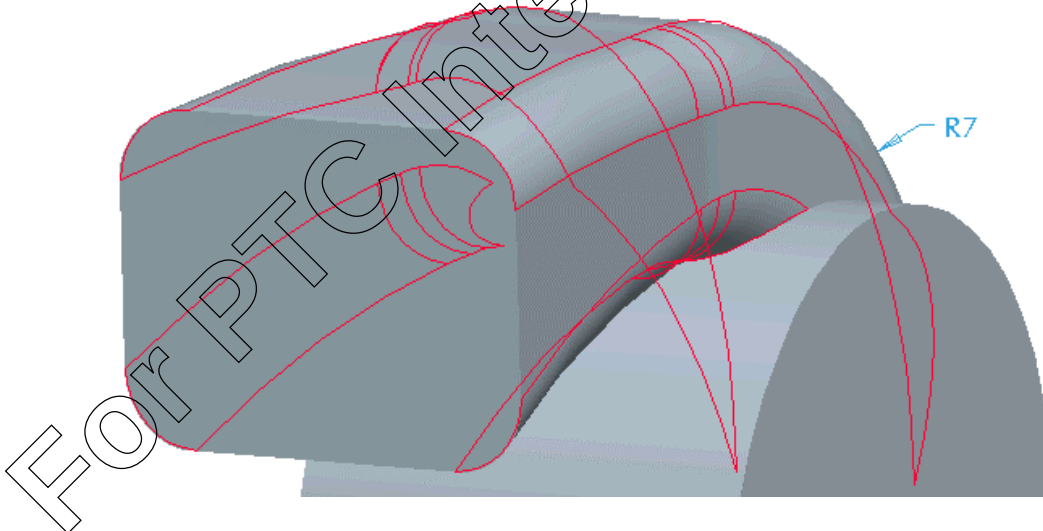


Figure 58: Creating a Round

3. Create a round on the edge shown in the following figure.

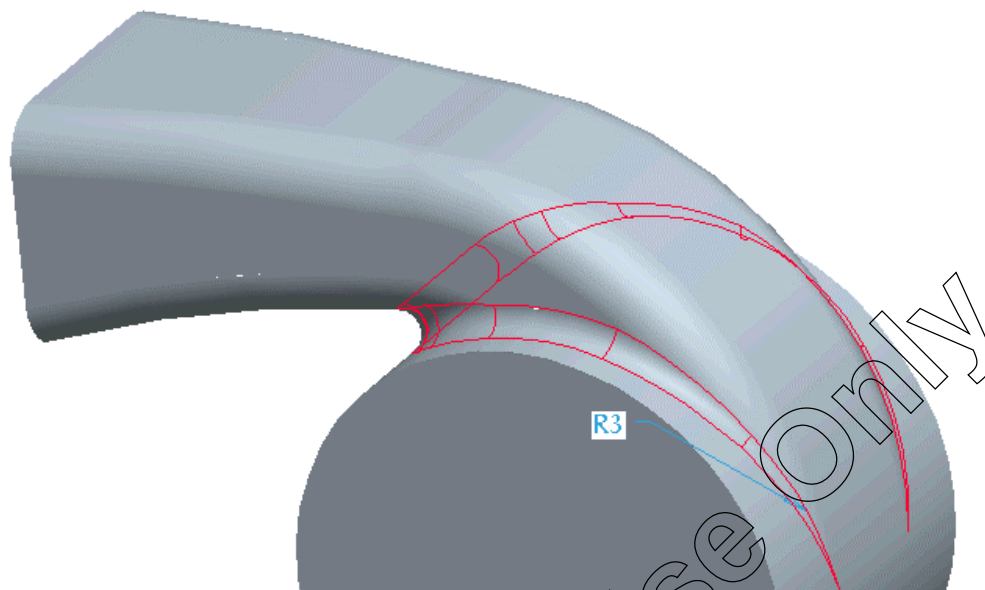


Figure 59: Creating a Round

Step 5. Create a shell to 'hollow out' the housing.

1. Start the Shell tool, and select appropriate surfaces to remove, as shown in the following figure. Specify a shell thickness of 2.

Design Intent:

The shell should remove the surfaces for the entry and exit of the airflow.

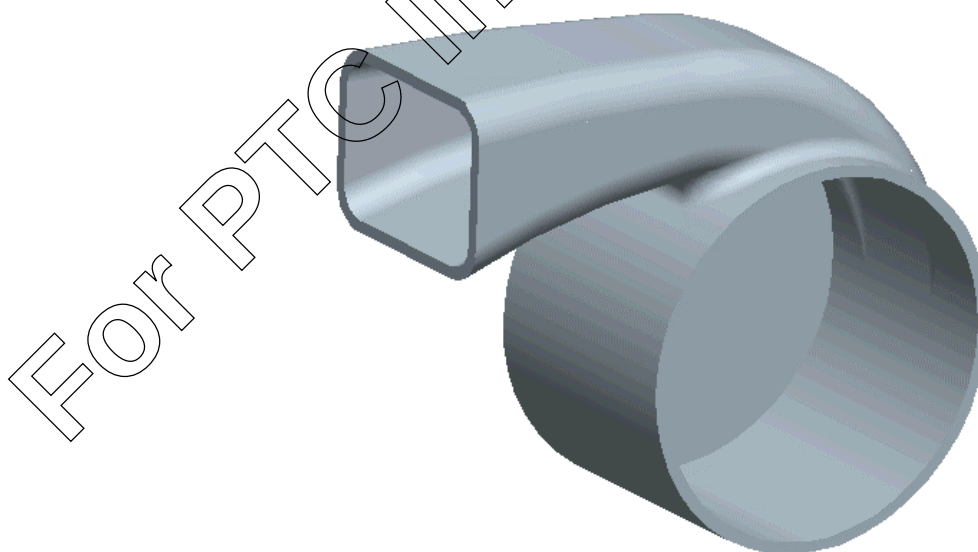


Figure 60: Creating a shell

Step 6. Create a hole for the impeller collar.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and have no depth dimension.

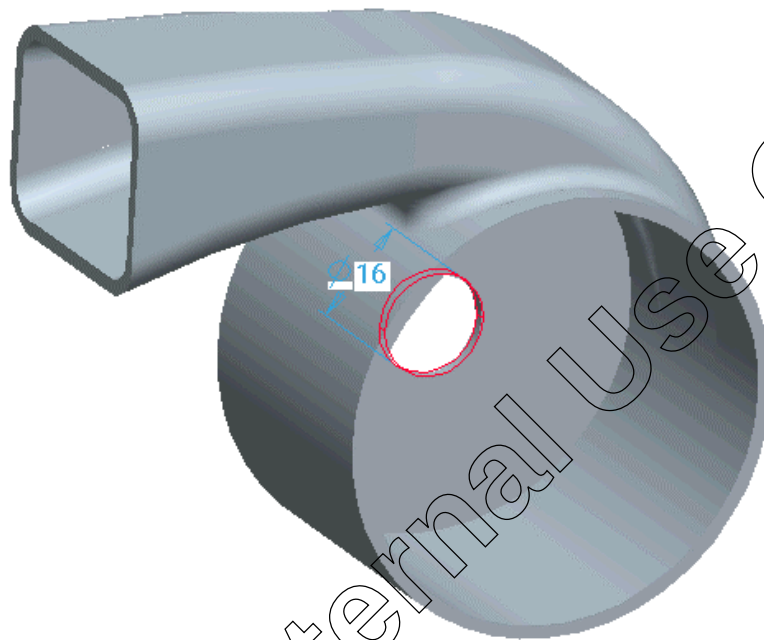


Figure 61: Creating a Hole

Step 7. Create a protrusion to serve as a bolting flange.

1. Create a protrusion as shown in the following figure. Sketch 2 concentric circles.

Design Intent:

The protrusion should be sketched on the front 'thin' surface of the model. The inner circle should reference the inner diameter, and the outer should be dimensioned. Extrude its depth inward over the model.

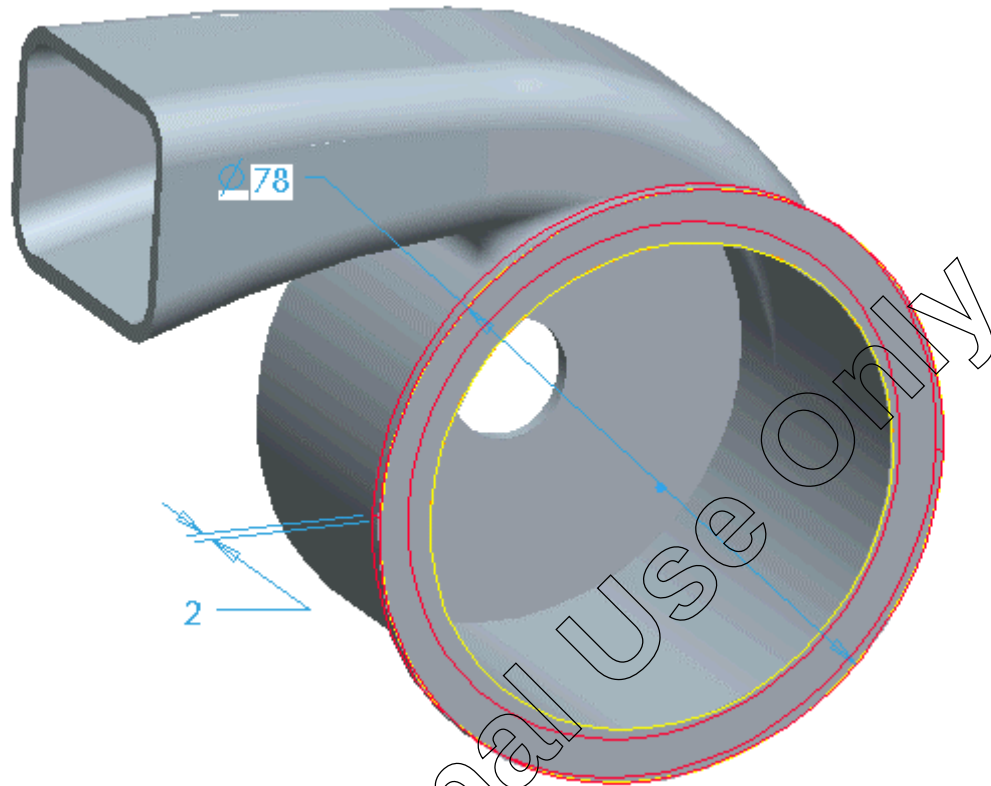


Figure 62: Creating a bolting flange

2. This model will be completed later in the project. Save the model and close the window. This completes the exercise.

Objective 7: Creating the Frame

Step 1. Create FRAME.PRT. Create the main body of the model.

1. Create a new part named FRAME.PRT.
2. Insert a Sweep as a Protrusion. Sketch the trajectory on the TOP plane, as shown in the following figure.

Design Intent:

Orient the sketch so that the RIGHT plane is vertical. The start point is to be placed in the upper left corner.

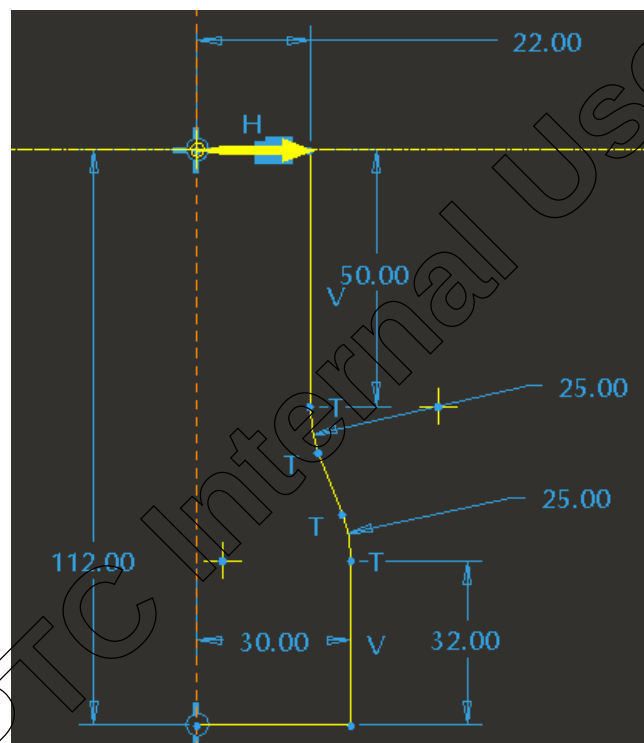


Figure 63: Sketching the Trajectory

3. Sketch the cross-section as shown in the following figure.

Design Intent:

The section is to be created such that the trajectory cross-hairs are located in the lower center of the sketch.

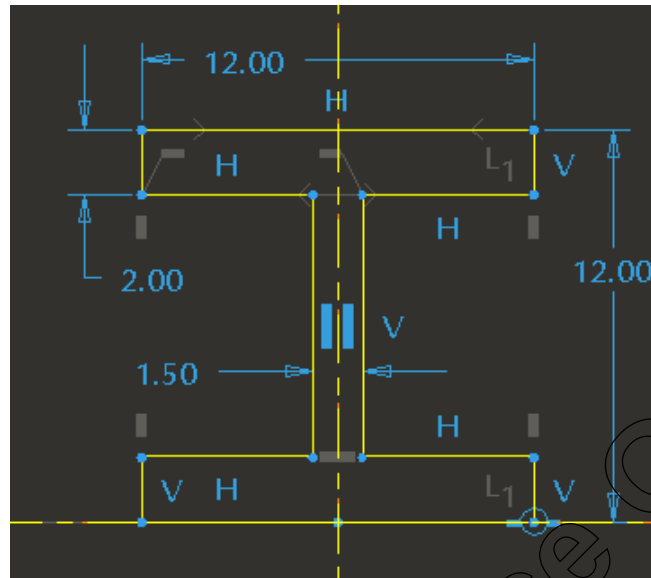


Figure 64: Sketching the Cross-Section

4. Save a copy of the sketch, and name it I-BEAM.SEC. Complete the feature.

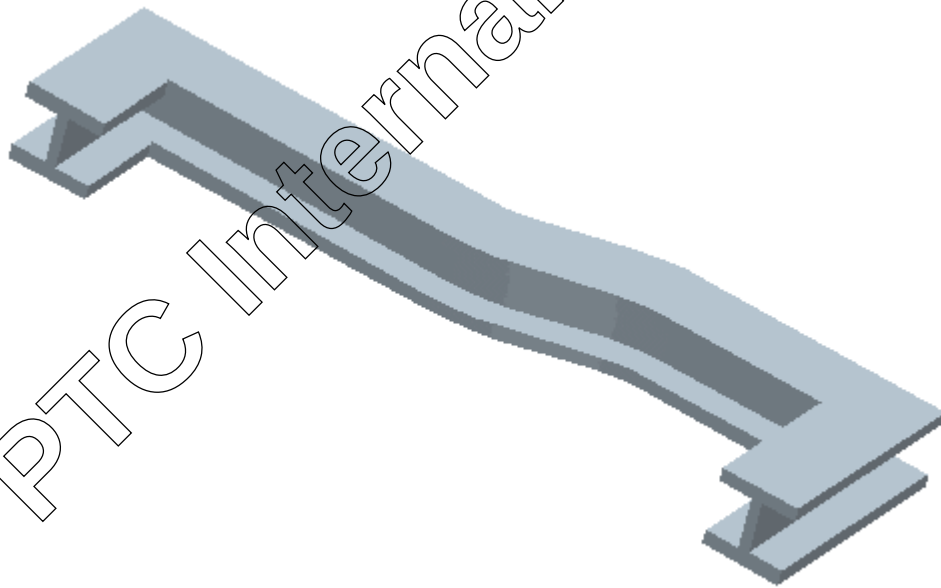


Figure 65: Completed Sweep

Step 2. Create a second sweep to support the engine.

1. Create a datum plane offset from RIGHT, as shown in the following figure. This plane will be used as a sketching plane for the next feature.

Design Intent:

The sketching plane for the next feature needs to be centered on the existing sweep.

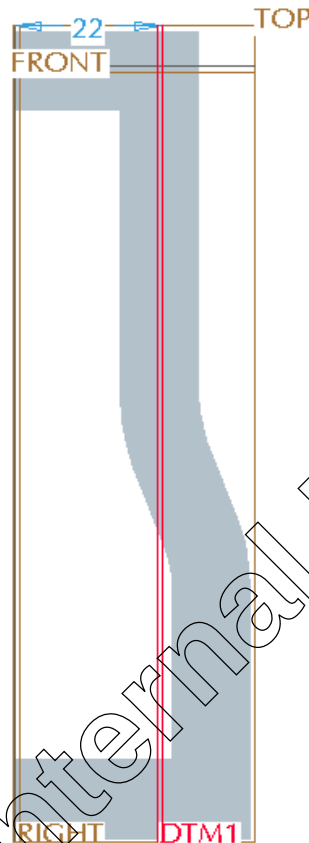


Figure 66: Creating a Datum Plane

2. Insert another Sweep as a Protrusion. Sketch the trajectory on the previous datum plane (DTM1), as shown in the following figure.

Design Intent:

Orient the sketch so that the TOP plane is horizontal. The sketch should reference FRONT and the upper model surface, and be symmetric around a center line.

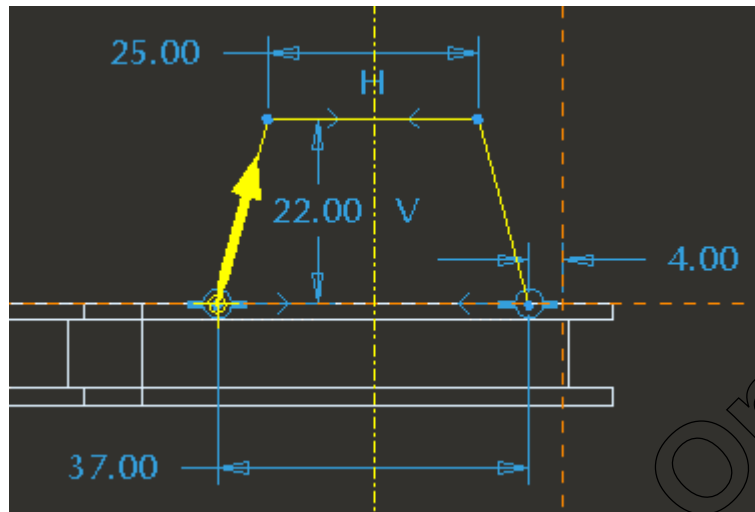


Figure 67: Sketching the Trajectory

3. Specify the Free Ends option.
4. To create the cross section, sketch using the I-BEAM.SEG file created previously.

Design Intent:

The section is to be scaled at 1.0, and rotated 90 degrees. Center the trajectory on the sweep trajectory cross hairs.

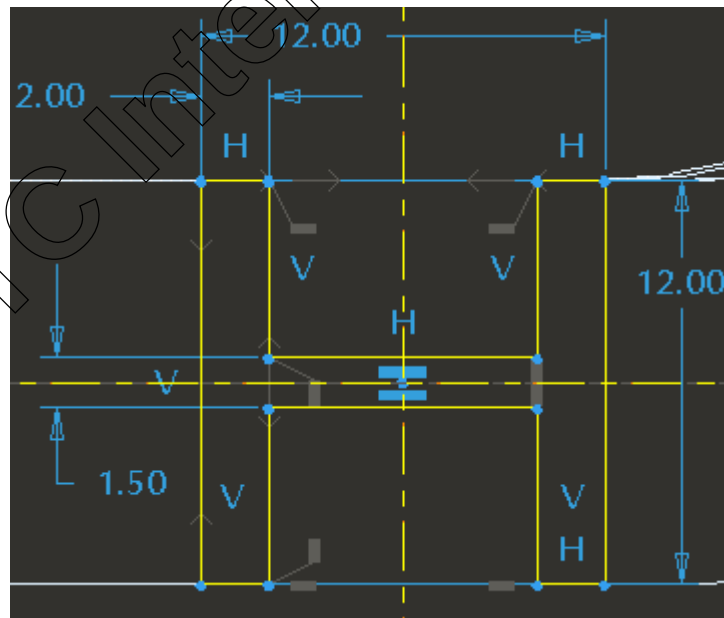


Figure 68: Creating the Sketch

5. When prompted, do NOT align any of the entities.
6. Complete the feature. Notice that the ends of the sweep are not flush.

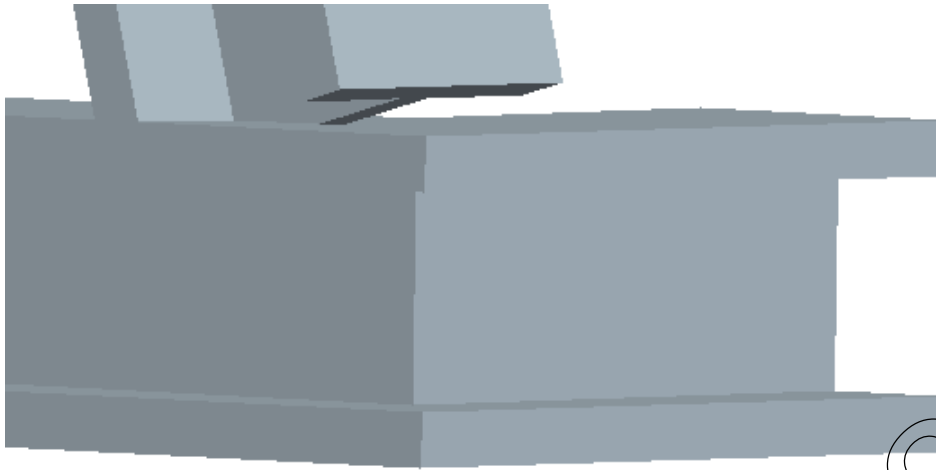


Figure 69: Sweep set to Free Ends

7. Edit the definition of the last sweep, and set the attributes to Merge Ends.



Figure 70: Sweep set to Merge Ends

8. Create a group named ENG_SWEEP containing the offset datum plane and the second sweep.
9. Hide the datum plane in the group.

Note:

The sweep feature does not automatically group datum features.

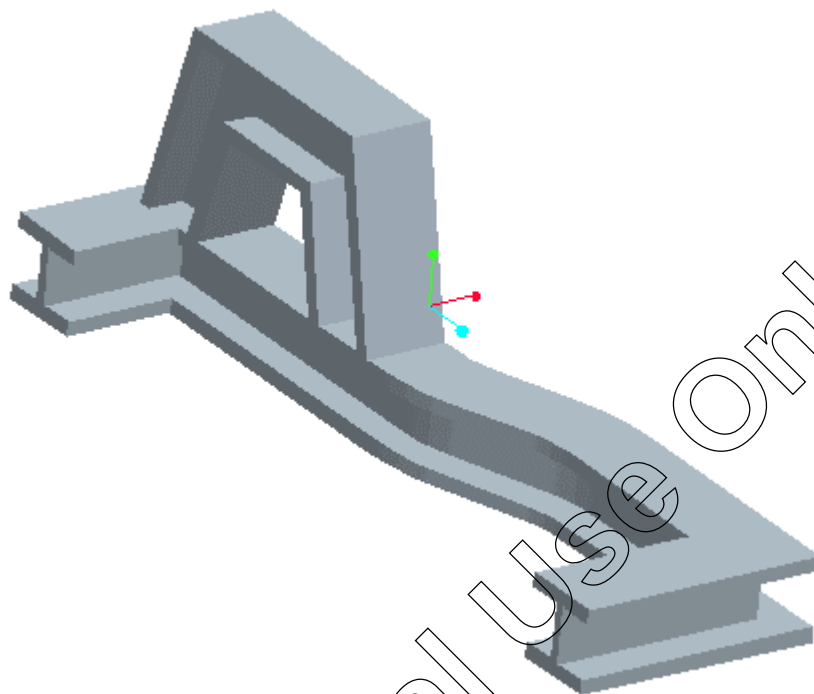


Figure 71: Partially completed FRAME.PRT

10. This model will be completed later in the project. Save the model and close the window.

This completes the exercise

Summary

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

- Create the Piston Pin.
- Create the Piston.
- Create the Connecting Rod.
- Create the Crankshaft.
- Create the Engine Block.
- Create the Impeller Housing.
- Create the Frame.

For PTC Internal Use Only

Duplicating Design Model Features

Introduction

When creating design models, you can quickly duplicate features or components. You can also incorporate your design intent into models by using mathematical relations. In this module, you learn how to use the pattern tool to create multiple instances of a single feature. You also learn how to use the copy tool to create a single instance of multiple features. Finally, you learn how to create left- and right-handed models by mirroring existing design models.

Objectives

After completing this module, you will be able to:

- Duplicate features using Pattern.
- Duplicate features using Copy.
- Mirror existing part models.
- Create symmetrical part models.
- Create and manage layers of duplicated features.
- Describe how copy, pattern, mirrored parts, and symmetrical parts affect parent/child relationships.



Instructor Preparation

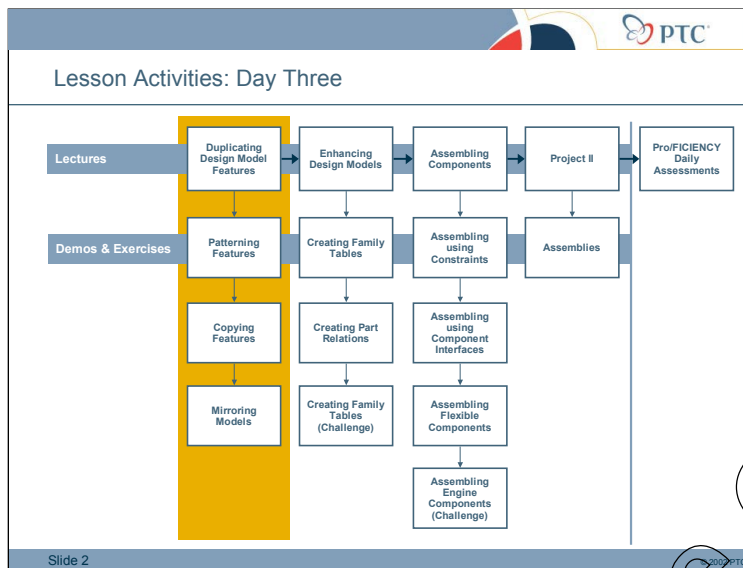
Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must** read and **thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 15 mins
- Demos (3): 35 mins
- Labs (3): 50 mins
- Total: 1 hr 30 mins



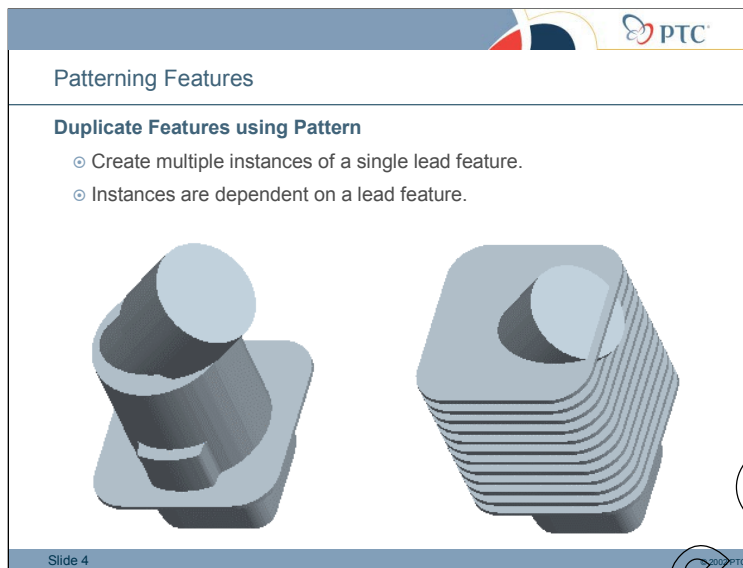
Objectives

After completing this module, you will be able to:

- Duplicate features using Pattern.
- Duplicate features using Copy.
- Mirror existing part models.
- Create symmetrical part models.
- Create and manage layers of duplicated features.
- Describe how copy, pattern, mirrored parts, and symmetrical parts affect parent/child relationships.

Slide 3

- When creating design models, you can quickly duplicate features or components.
- You can also incorporate your design intent into models by using mathematical relations.
- In this module, you learn how to use the pattern tool to create multiple instances of a single feature.
- You also learn how to use the copy tool to create a single instance of multiple features.
- In addition, you learn how to create left- and right-handed models by mirroring existing design models.
- Finally, you learn how to create relations between features in design models.



- Patterning enables you to copy a feature using a repeatable scheme.
- This scheme can be either a dimension scheme or a fill scheme.
- Using a dimension scheme, you simply select the dimensions that you want to pattern, set a distance, and determine the total number of instances.
- To pattern according to a defined region, you can use a fill scheme.
- For example, a fill scheme can be used to pattern the holes on a telephone's receiver.
- Using Patterning, you can easily propagate similar features across a part or assembly.
- Additionally, if you change the lead feature, then the entire pattern will update to reflect that change.
- You can also edit a pattern using a pattern table, which is essentially a matrix of the different patterned features.

Benefit:

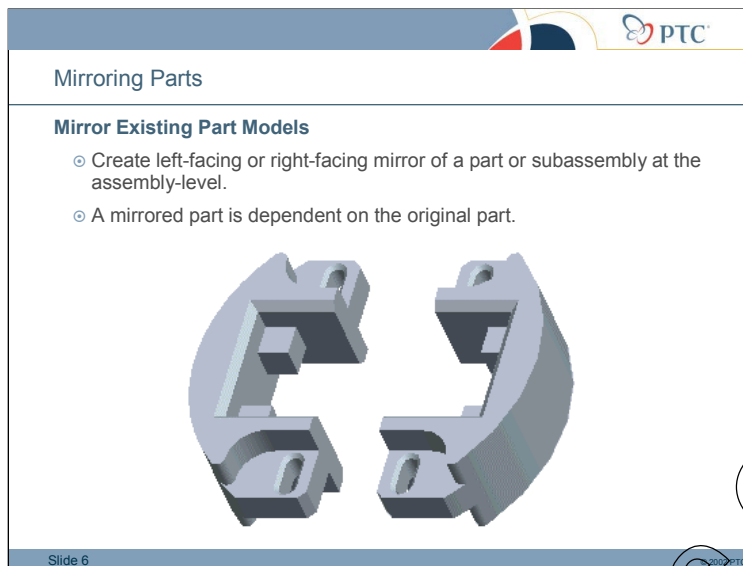
- Patterning enables you to quickly propagate similar features across a part or assembly.



- Using the copying feature, you can duplicate one or more features on a part.
- Copied features can be located using:
 - New references.
 - Same references.
 - Move by translation or rotation.
 - Mirror
- You copy features when you want to create an exact instance that re-uses multiple features that have already been created on the part.
- Copied features that are moved can be patterned.
- You save time by not recreating the two or more features that you copy.

Benefit:

- Duplicate features save you time by eliminating recreation of multiple features.



- By mirroring existing part models, you can recreate an entire part on an assembly that is mirrored about datum plane.
- You create mirrored parts in assembly mode.
- The mirrored part is created after you select a part to mirror and a datum plane that the part is to be mirrored about.
- Mirrored parts significantly increase your productivity by eliminating the need to recreate a symmetrically opposite part.

Benefit:

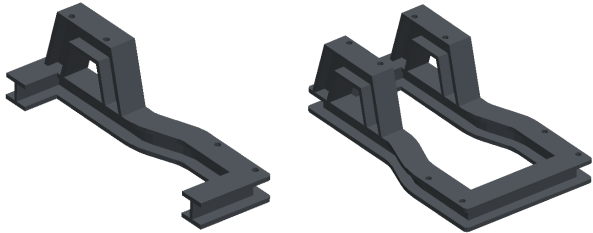
- Increases productivity by eliminating the need to recreate a symmetrically opposite part.

PTC

Creating Symmetrical Part Models

Design half of model and mirror to create complete part

- Uses the Mirror Tool to mirror the entire model.
- Creates a 'mirror' feature in the model tree.
- Modified features that are mirrored are updated upon regeneration.



Slide 7

Building symmetrical design models can be significantly sped up by creating half of the model and then mirroring the model to complete the part as shown in the figures above.

This is different than mirroring features as it uses the mirror tool to mirror the model instead of using feature operations > copy > mirror.

Additionally, this functionality creates a mirror feature in the model tree instead of 'copied' features.



Parent/Child Relationships – Duplicating Design Model Features

When duplicating design model features, parent/child relationships are created. Changes made to a parent feature affects its children.

- Pattern Features
- Copy Features
- Mirror Parts
 - *Creates mirrored part*
- Symmetrical Parts
 - *Creates mirror feature*

Slide 8

Note to Instructors:

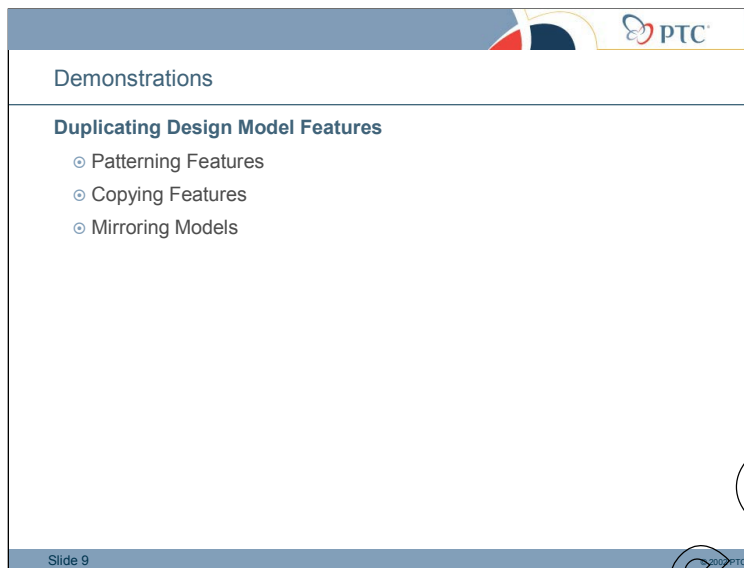
Use this slide to teach students how the functionality they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

Pattern features are children to the initial feature in the pattern. They are also children to that feature's parent.

Copy features are children to their location references. They are also children to the feature that was copied, if it was selected to be dependent.

Mirror parts are directly dependent on the parent part.

Symmetrical parts are created by mirroring all the features in the model tree. Hence, the mirror feature that is created is directly dependent on all features that come before it in the model tree.



You will follow along while I demonstrate:

1. Patterning Features
2. Copying Features
3. Mirroring Models

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Duplicate features using Pattern.
- Duplicate features using Copy.
- Mirror existing part models.
- Create symmetrical part models.
- Create and manage layers of duplicated features.
- Describe how copy, pattern, mirrored parts, and symmetrical parts affect parent/child relationships.

For PTC Internal Use Only

Referencing PTC Help for Module 9 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Pattern	Part Modeling	About the Pattern User Interface About Pattern Regeneration Options About Fill Patterns
Copy	Part Modeling	Using the COPY FEATURE Menu About the Copy Command
Mirror	Part Modeling	About Mirrored Features To Mirror All the Geometry in a Part Using the COPY FEATURE Menu To Copy Features by Mirror

Module 9 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Patterning Features

Introduction

In this demonstration, we create a variety of common patterns using pattern features. Using patterns, we can create an array of features based on a single feature or a group of features. Patterns create multiple features or groups, which eliminates the need to recreate the feature or group. For example, we can pattern a single cut that is used to vent hot gases to create a four by four cut array.

Objectives

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

- Create patterns of features and groups.
- Create fill patterns.

Scenario

You are assigned to work on the cylinder part of the engine assembly. You must create the cylinder's cooling fins by patterning the initial cooling fin by selecting a dimension and choosing the number of patterned features. You will also pattern a rib feature on the recoil cover part and multiple cut features on the engine cover part using the same method of choosing a dimension.

Afterwards, you will pattern a hole feature on the muffler part using a fill pattern. A fill pattern creates patterned features dependent on the configurable settings. Patterning enables you to quickly create multiple features that are the same as the parent feature, but are at a different location.

Step 1. Pattern the CYLINDER.PRT cooling fins.

In this initial step, we pattern the cooling fin on the engine's cylinder. First, we select the cooling fin group and pattern it by selecting the linear dimension. The linear dimension represents the direction that we wish the pattern to follow. Afterward, we set the pattern's dimensional increment and the total number of instances. Note that after completing the pattern we can add features to the initial group and have them reference the pattern.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to
C:\users\student\fast_track_330\module_09.
3. Open CYLINDER.PRT.
4. Examine the FIN group in the model tree.

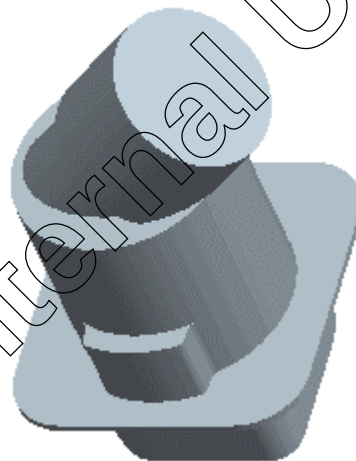


Figure 1: Examining the CYLINDER.PRT

5. Select the FIN group and start the Pattern tool.
6. Select the **15** dimension and enter **5** as the increment.
7. Use the dashboard to enter **12** as the quantity, and complete the feature, as shown in the following figure.

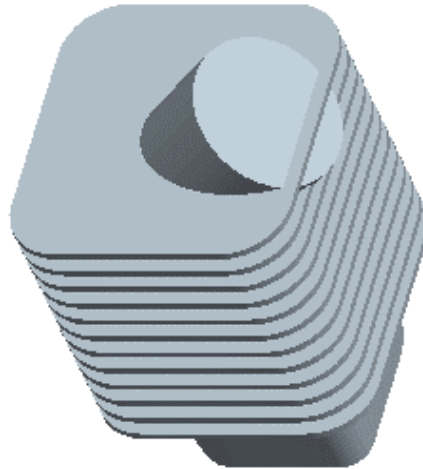


Figure 2: Patterning the Fin

Step 2. Pattern the FIN_CUT.

1. Display suppressed features in the model tree, and resume the FIN_CUT group.
2. Select the FIN_CUT group and start the Pattern tool. Select **Dimension** as the pattern type.
3. Select the **15** dimension and enter **5** as the increment.
4. Use the dashboard to enter **5** as the quantity and complete the feature, as shown in the following figure.

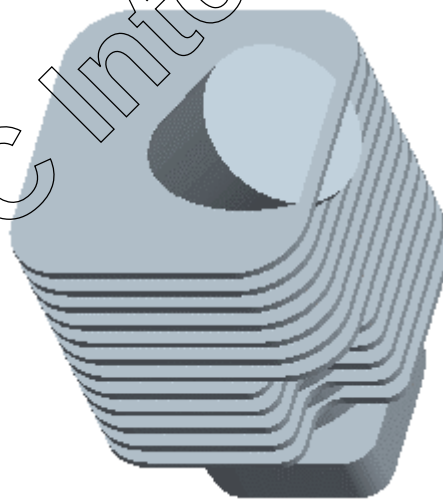


Figure 3: Patterning the Fin Cut

Step 3. Create and reference pattern a round.

1. Zoom in to the original fin.

2. Start the Round tool. Press and hold CTRL and select the top and bottom surfaces of the original fin, and then select the outer thin surface of the original fin to create a full round, as shown in the following figure. Complete the feature.

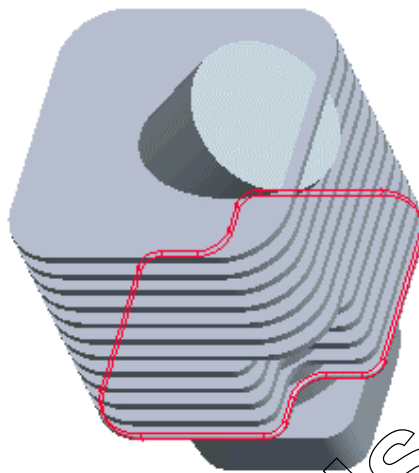


Figure 4: Creating a Round

3. With the previous round still selected, start the Pattern tool to create a reference pattern as shown in the following figure.

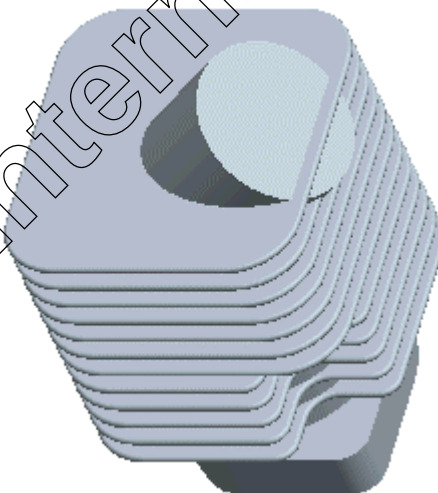


Figure 5: Patterning the Round by Reference Pattern

Note:

Reference patterns can be created with many feature types. However, the feature must reference the leading feature in the pattern. For direct features such as rounds and chamfers, the reference pattern is created automatically. For other features, you must select the reference pattern option.

4. Save the model.
5. Close the window.

Step 4. Pattern the ribs on the RECOIL_COVER.PRT.

In the next step, we create another pattern that follows a rotational scheme. Note that the overall steps are the same as the linear pattern except that we choose an angular dimension in order to pattern the rotation about an axis.

1. Open RECOIL_COVER.PRT.
2. Examine the RIB_1 group in the model tree as shown in the following figure.

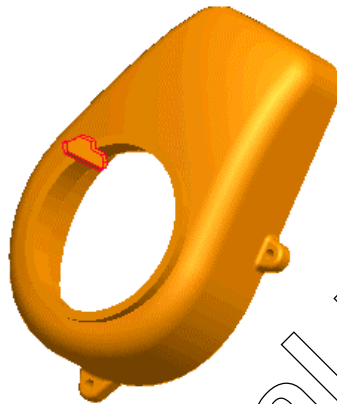


Figure 6: Examining the RECOIL_COVER.PRT

3. Select the RIB_1 group and start the Pattern tool.
4. Select the **45** degree dimension and enter **15** as the increment.
5. Use the dashboard to enter **24** as the quantity and complete the feature, as shown in the following figure.

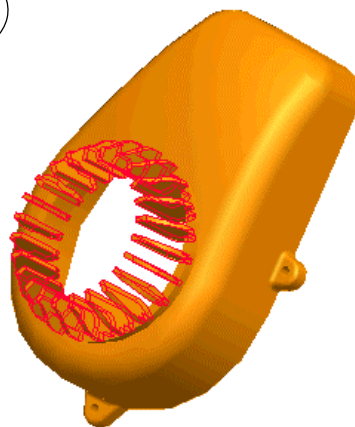


Figure 7: Patterning the Rib on the RECOIL_COVER.PRT

6. Save the model.
7. Close the window.

Step 5. Pattern the top vents.

1. Open ENGINE_COVER.PRT.
2. Orient to the TOP view.
3. Select the TOP_VENT group from the model tree and start the Pattern tool. Select the **44** dimension and enter **8** as the increment.
4. Right-click and select **Direction 2 dimensions**. Select the **22** dimension and enter **-23** as the increment.
5. Select the **Dimensions** tab in the dashboard to view the dimension values. Also notice that each direction defaults to a quantity of 2. Complete the feature.

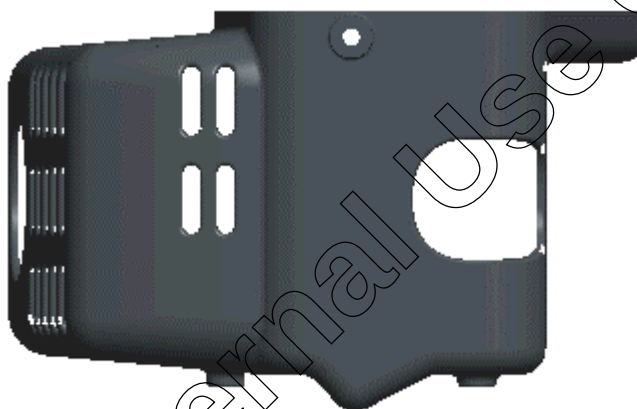


Figure 8: Creating a Pattern in Two Directions

6. Edit the pattern. Enter **4** for the horizontal quantity, and **3** for the vertical quantity. Regenerate the model, as shown in the following figure.

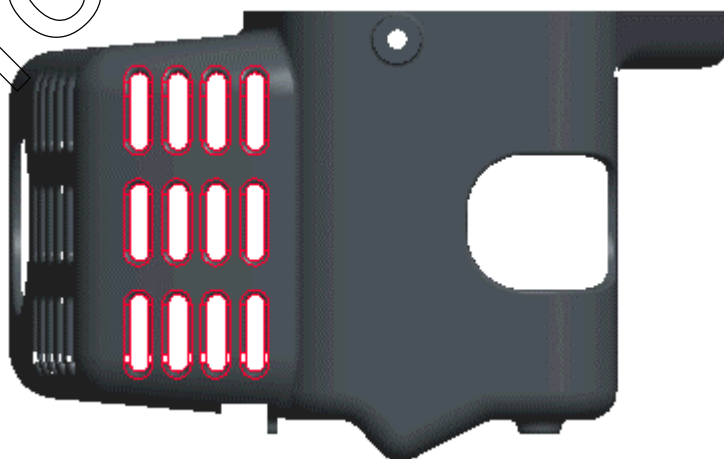


Figure 9: Editing the Top Vent Pattern

7. Save the model.

Step 6. Pattern the vents on ENGINE_COVER.PRT.

Instructor: Skip this step for the demo.

1. Orient to the FRONT view.
2. Select the FRONT_CUT from the model tree and start the Pattern tool.
3. Select the **30** dimension and enter **8** as the increment. Enter **11** as the quantity and complete the feature, as shown in the following figure.



Figure 10: Creating the Front Cut Pattern

4. Examine the original FRONT_CUT feature to determine why the cut narrowed at the end of the pattern.

Note:

The pattern narrowed due to the construction geometry in the FRONT_CUT feature that captures the design intent.

Step 7. Create and pattern a round feature.

Instructor: Skip this step for the demo.

1. Insert a round with a radius of **1** on the edge of the first cut in the FRONT_CUT pattern.
2. With the round still selected, start the Pattern tool to create a reference pattern, as shown in the following figure.



Figure 11: Creating a Reference Pattern

Step 8. Pattern the back vents.

***Instructor:** Skip this step for the demo.*

1. Orient to the BACK view.
2. Click **Edit > Feature Operations > Group > Local Group**. Enter **BACK_VENT** as the name. Select BACK_CUT and BACK_ROUND and click **Done**.
3. With the group still selected, start the Pattern tool. Select the **30** dimension and enter **8** as the increment. Enter **6** as the quantity and complete the feature, as shown in the following figure.

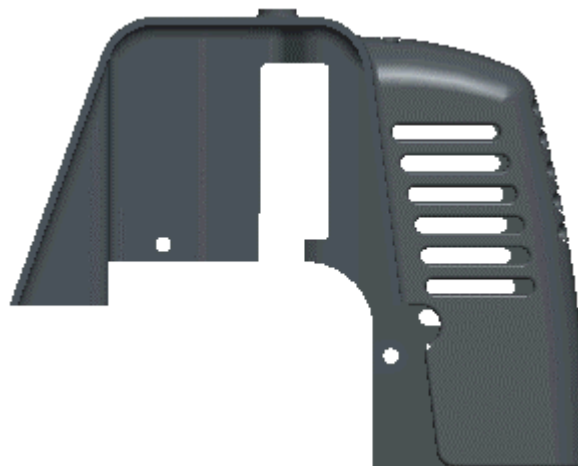


Figure 12: Creating a One-Direction Pattern

4. Notice how the sixth pattern instance is very close to the 'U' shaped cut. With the pattern still selected, right-click, and select **Edit Definition**.
5. Press and hold CTRL and select the **24** dimension, and enter **-1** as the increment.
6. Select the **Dimensions** tab, and notice the second dimension to the first direction. Enter **11** for the quantity and complete the feature, as shown in the following figure.

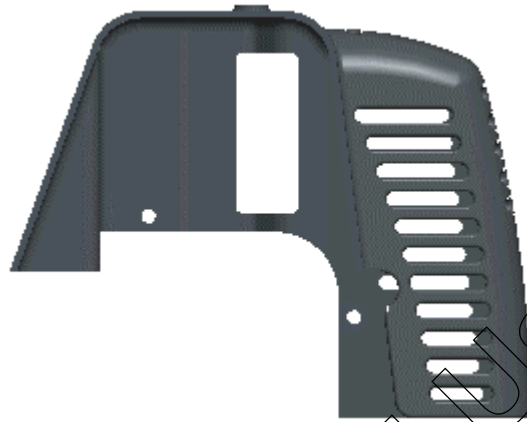


Figure 13: Redefining the Pattern to use Two Dimensions

7. Save the model.
8. Close the window.

Step 9. Create a fill pattern on MUFFLER.PRT.

In the final step, we create a fill pattern. Fill patterns differ from the patterns we have created thus far by enabling us to create a pattern to fill an area. The area can be manually selected, and the type of fill can be altered between multiple choices such as square, diamond, circle, curve, and spiral. We also can alter the spacing between the features and offset the fill type at an angle to provide the precise pattern that we need. The advantage to using fill patterns is that we can create aesthetic patterns within a single feature instead of having to create multiple patterns to achieve a desired appearance.

1. Open MUFFLER.PRT.
2. Turn off the display of datum axes.
3. Select the small hole and start the Pattern tool. Select the circular curve to copy its section into the pattern.
4. Edit the spacing between pattern member centers to **5**, as shown in the following figure.



Figure 14: Creating a Circular Fill Pattern

5. Use the dashboard to space the pattern members in a **Diamond** pattern. Then space the members in a **Triangle** pattern, as shown in the following figure.

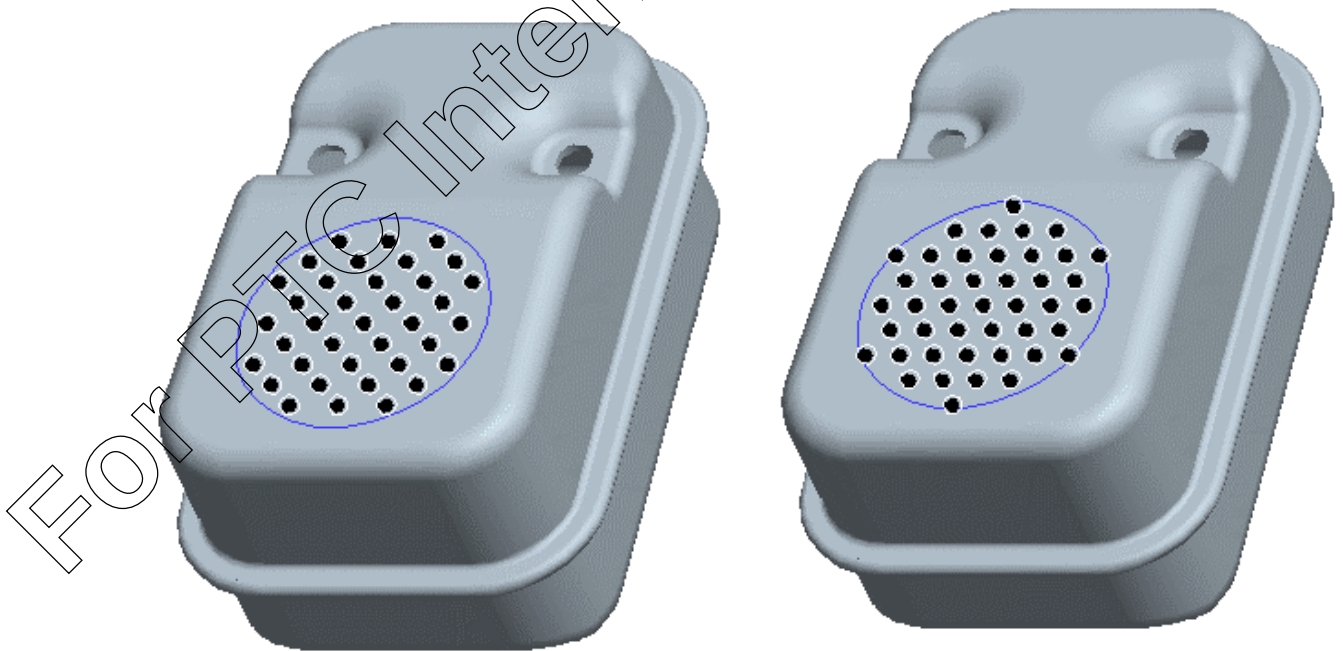


Figure 15: Examining Diamond (left) and Triangle (right) Fill Patterns

6. Use the dashboard to space the pattern members in a **Circle** pattern. Edit the radial spacing to **7**, as shown in the following figure.



Figure 16: Examining the Circular Fill Pattern

7. Space the pattern members along a **Curve**.
8. Space the pattern members in a **Spiral** pattern. Edit the rotation of the grid to **180** degrees.
9. Space the pattern members in a **Square** pattern. Edit the boundary value to **-1**. Edit the grid rotation to **45** degrees.
10. Select the two pattern members nearest to the front edge to disable them as shown in the following figure. Complete the feature.



Figure 17: Disabling Patterned Holes (left) and the Completed Pattern (right)

Step 10. Edit the fill pattern.

Instructor: Skip this step for the demo.

1. Select and hide **Curve id 5103**, which was originally referenced by the fill pattern.
2. Edit the pattern. Enter **40** for the curve diameter and regenerate the model.
3. Edit the definition of the pattern to disable a total of four holes, as shown in the following figure. Complete the feature.

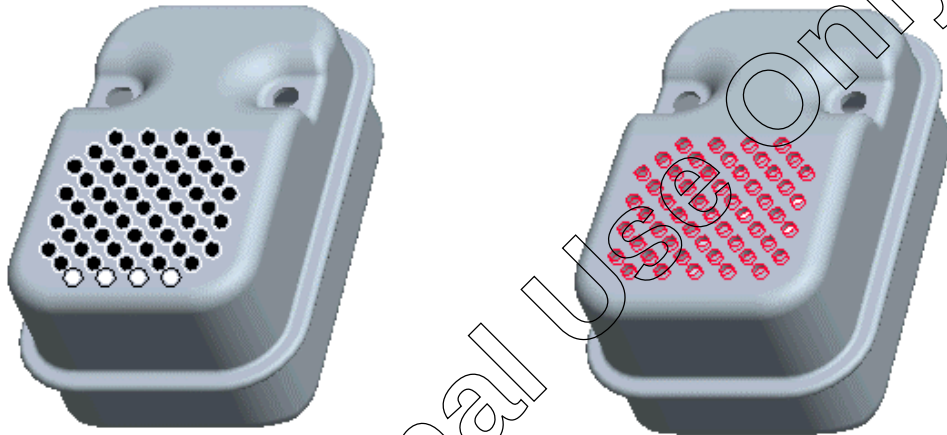


Figure 18: Disabling Holes (left) and Completing the Pattern (right)

4. Save the model.
5. Close the window.

This completes the exercise.

Exercise 2: Copying Features

Introduction

In this demonstration, we reuse features or groups by copying them and placing them in other locations on a part. Copying features or groups saves time because we do not have to recreate the feature. We select the feature or group, select the references, select whether to translate or rotate the group, and select any dimensions to vary within the group. We can also select whether the new feature or group will be dependent or independent on the original.

Objectives

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

- Copy features or groups by translation and rotation.
- Copy features to new references.

Scenario

You are assigned to complete the rear engine block design. The remaining task is to create a boss feature on the engine block. You must reuse a previously created boss feature by copying and rotating the feature. These boss features will be used as interfaces for assembly.

You will also create a copy of a mounting ear feature on the carburetor part and translate the copy to a new location. Afterwards, you will create two newly copied features by mirroring both the original and copied ear features to the opposite side of the part.

Step 1. Copy a boss on ENG_BLOCK_REAR.PRT.

In this initial step, we copy a group that creates a boss feature on the rear engine block. In order to keep the model organized, we insert this group directly after the group we intend to copy. This feature will be rotated.

1. Open ENG_BLOCK_REAR.PRT.
2. Display datum axes, and orient to the RIB view.
3. Drag the **Insert** indicator directly after the BOSS_1 group.
4. Click **Edit > Feature Operations > Copy > Move > Dependent > Done**.
5. Select the BOSS_1 group and click **Done > Rotate > Crv/Edg/Axis**.
6. Select the MAIN axis and click **Okay**. Enter **100** degrees, and click **Done Move > Done > Ok**.

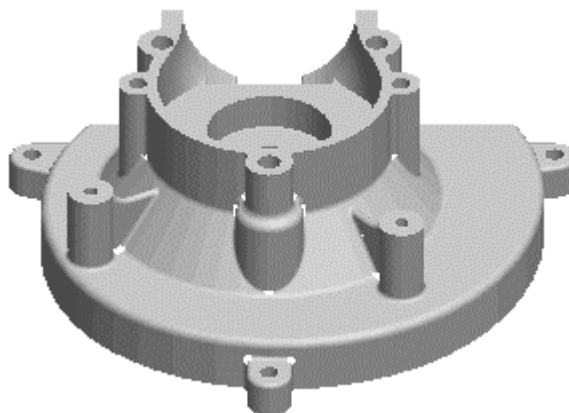


Figure 19: Copied Group

7. Rename the copied group to BOSS_2.
8. Select the **Insert** indicator, right-click and select **Cancel > Yes**.

Step 2. Edit the copied features.

1. Edit the BOSS_2 group. Enter **120** degrees for the angle and regenerate the model.

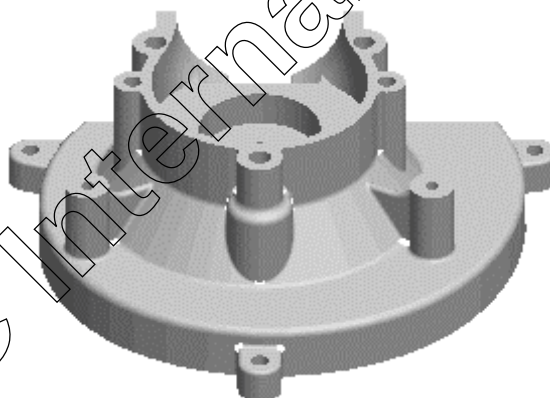


Figure 20: Completing the Copy-Rotate Feature

2. Edit the hole diameter in BOSS_1 to **5**, and regenerate the model. Notice the hole in BOSS_2 updates.
3. Save the model.
4. Close the window.

Step 3. Copy the features on CARBURETOR.PRT.

In this next step, we copy and translate two features on the carburetor, the newly copied feature and its parent. We mirror the features about a plane.

1. Open CARBURETOR.PRT.

2. Click **Edit > Feature Operations > Copy > Move > Dependent > Done**.
3. Select the EAR and HOLE features from the model tree and click **Done > Translate**.
4. Select the FRONT plane from the model tree and click **Okay**.
5. Enter **23.5** and click **Done Move > Done > Ok**.

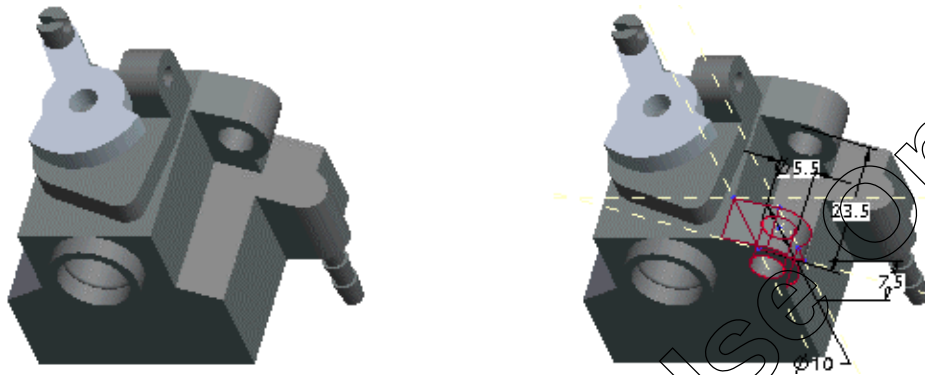


Figure 21: Examining the CARBURETOR.PRT (left) and Copying the Ear and Hole Features (right)

6. Click **Copy > Mirror > Dependent > Done**.
7. Select the EAR, HOLE, and COPIED_GROUP features from the model tree and click **Done**.
8. Select the RIGHT plane to complete the feature, as shown in the following figure.

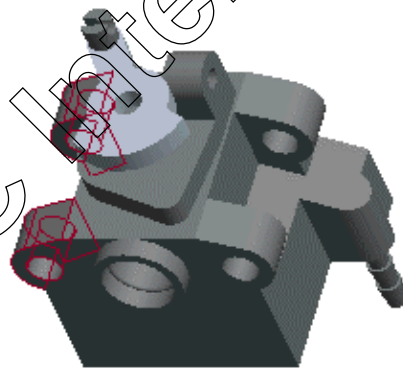


Figure 22: Mirroring Features about a Plane.

9. Save the model.
10. Close the window.

Step 4. (OPTIONAL) Copy the features in the HANDLE_MAIN.PRT.

Instructor: Skip this step for the demo.

1. Open HANDLE_MAIN.PRT.
2. Click **Edit > Feature Operations > Copy > Same Refs > Dependent > Done**.

3. Select the TAB1 group from the model tree and click **Done**.
4. Select the **80** and **20** dimensions as shown in the following figure and click **Done**.

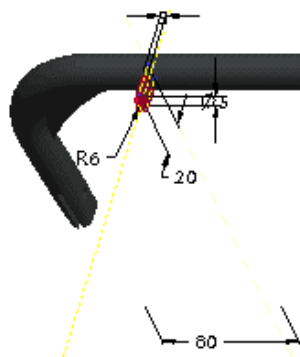


Figure 23: Examining TAB1

5. Enter **140** and **17**, and click **Ok** to complete the feature as shown in the following figure.

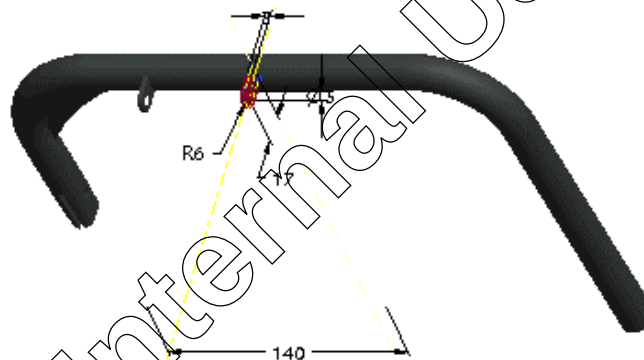


Figure 24: Creating a Dependent Copy of TAB1

6. Rename the COPIED_GROUP to TAB2.

Step 5. (OPTIONAL) Create a second copy of TAB1.

Instructor: Skip this step for the demo.

1. Click **Copy > Move > Dependent > Done**.
2. Select the TAB1 group from the model tree and click **Done > Translate**.
3. Select the RIGHT plane from the model tree and click **Okay**. Enter **120** and click **Rotate > Crv/Edg/Axis**. Select the TAB_PIVOT axis and click **Flip > Okay**.
4. Enter **30** degrees, and click **Done Move**. Select the **20** dimension and click **Done**.
5. Enter **14** and complete the feature as shown in the following figure.

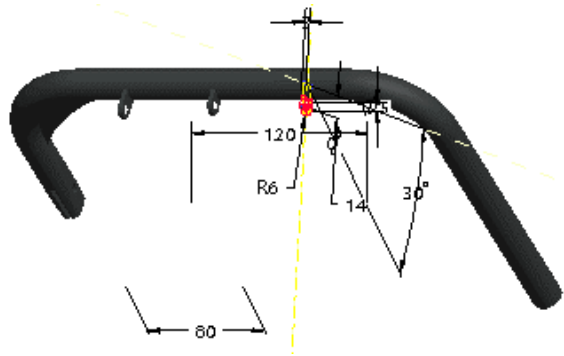


Figure 25: Translating and Rotating a Feature

6. Rename the group to TAB3.

Step 6. (OPTIONAL) Create a third copy of TAB1.

Instructor: Skip this step for the demo.

1. Click **Copy > New Refs > Dependent > Done**.
2. Select the TAB1 group from the model tree and click **Done**. Select the **80** and **20** dimensions and click **Done**.
3. Enter **50** and **23**. Select the MOUNT2 plane and the MOUNT plane as alternate references.
4. Click **Same > Okay > Okay > Done**. Refer to the following figure.

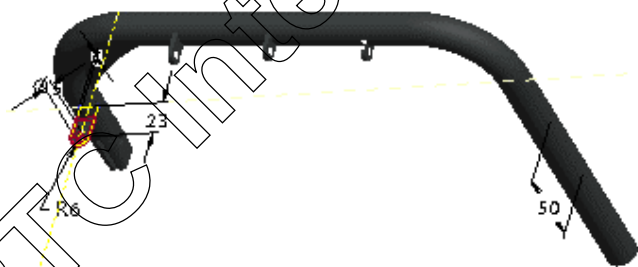


Figure 26: Copying a Group to New References

5. Rename the COPIED_GROUP to TAB4.

Step 7. (OPTIONAL) Edit the TAB features.

Instructor: Skip this step for the demo.

1. Edit the TAB1 group, and edit thickness from 5 to **3**. Regenerate the model. Notice that all the tabs update their width, but retain varied lengths.
2. Save the model.
3. Close the window.

This completes the exercise.

Exercise 3: Mirroring Models

Introduction

In this demonstration, we reuse previously created part geometry to create new models that are mirror images. Mirroring a part is efficient, because you simply select a plane to mirror the part.

Objectives

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

- Create new parts by mirroring.

Scenario

You are assigned to complete the modeling of both Clutch Shoes. Note that one of the Clutch Shoes is complete. Create the opposite Clutch Shoe.

Step 1. Mirror the CLUTCH_SHOE_L.PRT.

In this exercise, we reuse previously created part geometry to create new models that are mirror images.

1. Open CLUTCH_SHOE_L.PRT.
2. Orient to the TOP view and examine the part. Note that the slots are offset by different values.
3. Create an assembly named **CLUTCH_SHOE_TEMP** using the default template.
4. Assemble the CLUTCH_SHOE_L.PRT, and use a default constraint.
5. Click **Insert > Component > Create > Part > Mirror**. Enter **CLUTCH_SHOE_R** as the name and click **OK**.
6. Select the CLUTCH_SHOE_L.PRT as the part reference.
7. Select the 'T' shaped surface on the CLUTCH_SHOE_L.PRT as the planar reference.
8. Complete the mirror and save the model.

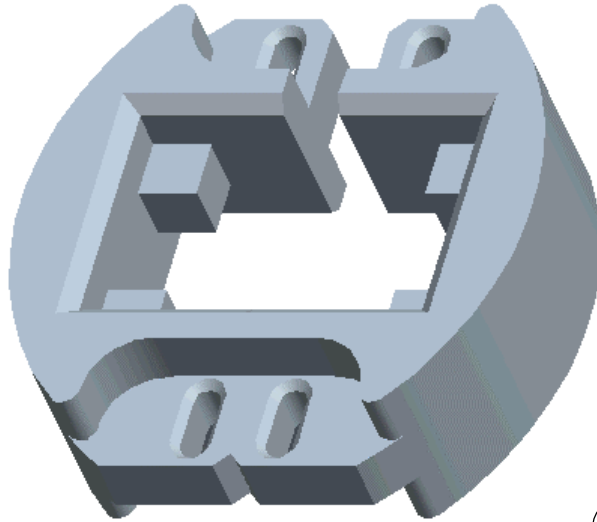


Figure 27: Mirroring the CLUTCH_SHOE_L.PRT

Note:

When creating mirror parts, never choose an assembly-level datum plane as the mirror reference. If you select a datum plane instead of a surface, it is necessary to select a part-level datum plane as the mirror plane to avoid referencing a plane in a temporary assembly. It will not affect the end result if the parts interfere with one another when mirrored.

9. Click **File > Delete > All Versions > Yes > Ok**. (Do NOT delete either CLUTCH_SHOE)
 10. Open the CLUTCH_SHOE_R.PRT to examine the part. Note that the part contains a single feature. This feature is only dependent on the CLUTCH_SHOE_L.PRT.
 11. Close all windows.
 12. Erase all models from memory.
- This completes the exercise.

Exercise 4: Creating and Managing Layers

Introduction

In this demonstration, we create layers in order to simplify the displayed information.

Layers allow us to quickly remove information from the graphics window, such as features, components, and references.

Layers are especially useful when working with complicated designs because they simplify the model's appearance.

Objectives

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

- Create layers containing duplicated features.

Scenario

You create layers at the part level in order to organize direct feature geometry. By creating this layer, you can easily select all features within the layer.

Step 1. Create a layer that contains rounds in MUFFLER_LAYER.PRT.

In order to simplify the display of patterned features within a model, we create a layer that blanks some of the hole axes on the muffler part. Notice that by blanking this layer, we reduce the amount of clutter on the screen. This will enable us to easily assemble the muffler by referencing the mounting hole axes.

1. Open MUFFLER_LAYER.PRT.
2. Using the toolbar, display the datum axes.

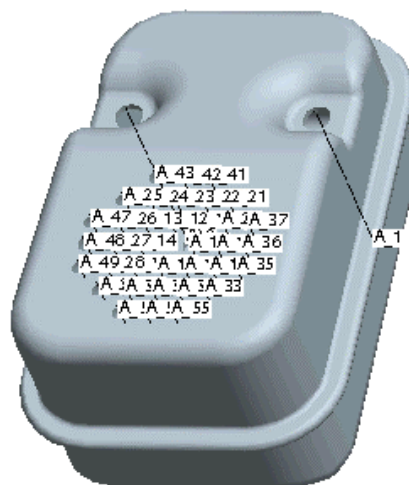


Figure 28: Axes Displayed

3. Using the layer tree, create a layer named **EXH_HOLES**.
4. Using the model tree, expand the pattern of holes. Select the first hole in the pattern, press and hold SHIFT, and select the last hole in the pattern. Complete the layer.
5. Blank the EXH_HOLES layer and repaint the screen. Axes A_1 and A_2 should remain visible.

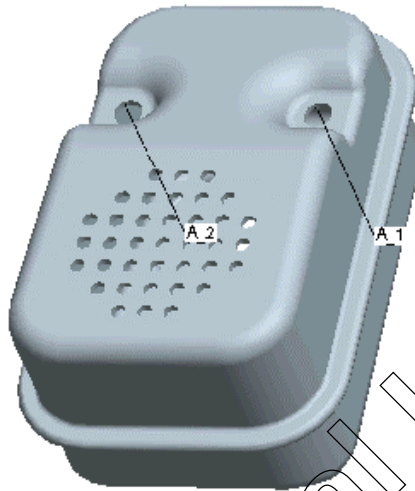


Figure 29: Axes Blanked

6. Save the layer status.
7. Save the model.
8. Close the window.
9. Erase the model from memory.

Summary

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

- Duplicate features using Pattern.
- Duplicate features using Copy.
- Mirror existing part models.
- Create symmetrical part models.
- Create and manage layers of duplicated features.
- Describe how copy, pattern, mirrored parts, and symmetrical parts affect parent/child relationships.

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Enhancing Design Models using Family Tables, Relations, and Parameters

Introduction

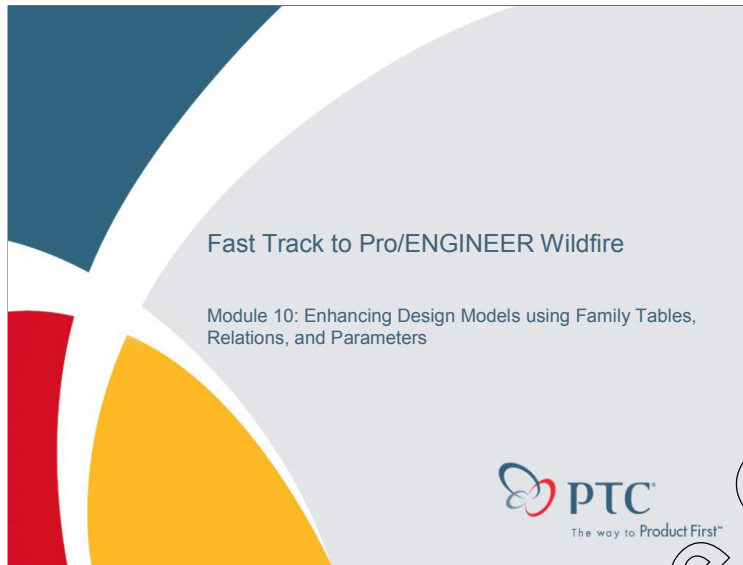
When creating design models, you can re-use geometry to quickly create multiple instances by varying the design of existing models using family tables. You can also incorporate your design intent into models by using mathematical relations.

In this module, you learn how to create multiple design variations of models using family tables. You also learn how to create relations between features in design models.

Objectives

After completing this module, you will be able to:

- Create design model variations using family tables.
- Configure relations that capture a part's design intent.
- Create parameters to capture additional information.
- Describe how family tables, relations, and parameters affect parent/child relationships.



Instructor Preparation

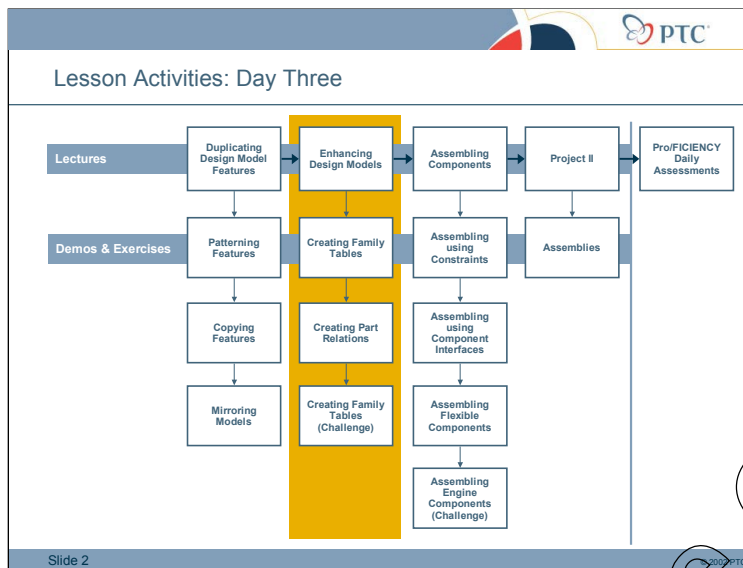
Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 5 mins
- Demos (3): 20 mins
- Labs (3): 35 mins
- Total: 1 hr



Objectives

After completing this module, you should be able to:

- Create design model variations using family tables.
- Configure relations that capture a part's design intent.
- Create parameters to capture additional information.
- Describe how family tables, relations, and parameters affect parent/child relationships.

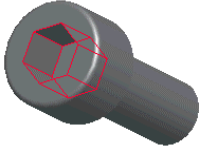
Slide 3

- When you create design models, you can re-use geometry from other design models.
- When you start a new design model, you can create inheritance features to transfer geometry from other models.
- You can twist, bend, stretch, and scale design models into various shapes using warp features.
- You can also quickly create multiple instances by varying the design of existing models using family tables.

Creating Family Tables

Creating Variations of Design Models using Family Tables

- Create family table instances by varying features, dimensions, and parameters.



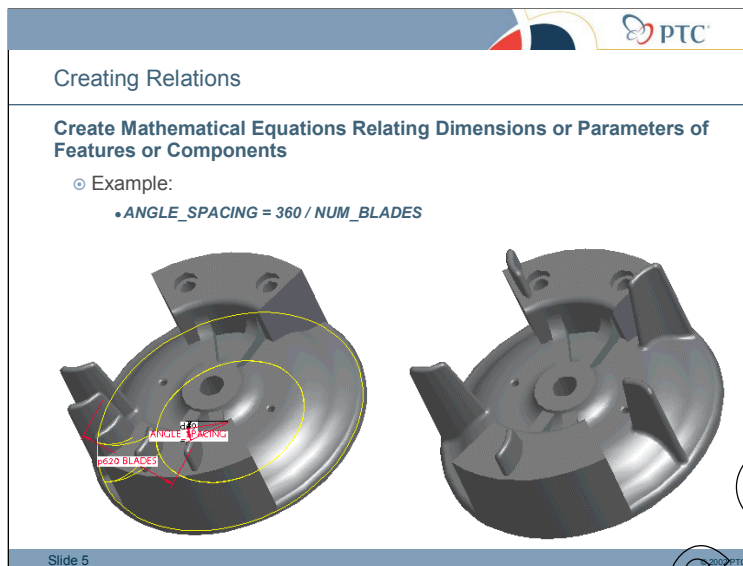
Type	Instance Name	d1 BOLT_DIA	d0 BOLT_LENGTH	F95 HEX	F178 TORX	F155 [CHAMFR]	DESCRIPTION
	BOLT	6.00	15.00	Y	Y	Y	Hex Head
	BOLT_6-15	6.00	15.00	Y	N	Y	Hex Head
	BOLT_6-20	6.00	20.00	Y	N	Y	Hex Head
	BOLT_6-36	6.00	36.00	Y	N	Y	Hex Head
	BOLT_5-15	5.00	15.00	Y	N	N	Hex Head
	BOLT_5-18	5.00	18.00	Y	N	N	Hex Head
	BOLT_5-24	5.00	24.00	Y	N	N	Hex Head
	BOLT_5-28	5.00	28.00	Y	N	N	Hex Head
	BOLT_5-50	5.00	50.00	Y	N	N	Hex Head
	BOLT_5-60	5.00	60.00	Y	N	N	Hex Head
	BOLT_4-12	4.00	12.00	N	Y	N	Torx Head
	BOLT_4-15	4.00	15.00	N	Y	N	Torx Head
	BOLT_4-18	4.00	18.00	N	Y	N	Torx Head

Slide 4

- Using a family table is a quick way to create a variety of parts or assemblies by designating dimensions, features, and parts that can change.
- You select items and place them into a matrix ("table") that represents the family.
- To create a family table, you:
 - Create generic model.
 - Create a table.
 - Add columns containing dimensions, parameters, or features that you want to vary.
 - Add rows with the names of instances.
 - Verify that the instances created can be regenerated.
- Family tables enable you to quickly create a family of virtual parts and assemblies by setting variable features and dimensions.

Benefit:

- Quickly create a family table of parts and assemblies by setting variable features and dimensions.



- Part relations provide you with a means to impart further design intent on a part.
- You create part relations when you want to automate the regeneration of feature dimensions or parameters that fulfill your design intent.
- Part relations enable you to add design intent to a model. By changing a single dimension or parameter, multiple other dimensions will change.
- For example, on a flywheel, if you change the angular dimensioning scheme of the cooling fin, the number of fins on the flywheel automatically changes.

Benefit:

- Part relations add design intent to models enabling them to operate logically.



Creating Parameters

Add additional information to your parts and assemblies that capture relevant design information.

- Real Number, Integer, String, and Yes/No
- Can be driving parameters or driven by parameters
- Examples:
 - *Designer name*
 - *Company name*
 - *Cost*
 - *Material*
 - *Make/Buy*

Slide 6

Parameters are important because they enable you to add additional information about parts and assemblies to the part or assembly file. This information can be extracted at any time in a drawing. It can also be extracted in Intralink or Windchill once the system is configured.



Note to Instructors:

Use this slide to teach students how the functionality that they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

Family tables are children to the generic part.

Relations drive how dimensions and patterns change. In all cases, one dimension will be the driving dimension. This dimension is the parent to the driven dimension, which cannot be changed independently by using the Edit command.

Parameters that contain data about the part such as cost and material generally do not create any parent/child relationship. Parameters can also be set as driven by or driving. Driven by parameters change their values based off of some other dimension or parameter. Driving parameters have children which change their value based upon the driving parameter.



You will follow along while I demonstrate:

1. Creating Family Tables
2. Creating Part Relations

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Create design model variations using family tables.
- Configure relations that capture a part's design intent.
- Create parameters to capture additional information.
- Describe how family tables, relations, and parameters affect parent/child relationships.

Referencing PTC Help for Module 10 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Family Tables	Fundamentals	About Saving Family Tables About Modifying Family Tables About Instance Accelerator Files
Relations	Fundamentals	About Relations Functions Used in Relations Operators Used in Relations Tip: Using Comments in Relations Rules for Creating Relations in Parts
Parameters	Fundamentals	About User Parameters Parameter Symbols Used in Relations

Module 10 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Creating Relations

Introduction

In this demonstration, we create relations. Relations enable you to enhance part and assembly designs by configuring simple mathematical formulas that can automate design intent. For example, you can create a relation that sets the angle of the spokes on an alloy car wheel to be symmetrical. This means that if you have three spokes, the spokes will be **120°** apart from each other. Four spokes would be **90°**.

Capturing design intent by configuring relations is important because it enables you to easily update a part of assembly simply by changing one dimension.

Objectives

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

- Configure relations that capture design intent.

Scenario

For an upcoming design review with a manufacturing contractor, your manager has asked you to add intelligence to the flywheel model. Your manager wants to showcase that the number of cooling fins may change in future iterations of the engine. However, the cooling fins must remain evenly patterned during the design review.

Step 1. Edit the FLYWHEEL.PRT.

In this initial step, we add a relation that enables the number of cooling fins to remain evenly spaced when the pattern is edited.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to
C:\users\student\fast_track_330\module_10.
3. Open FLYWHEEL.PRT.
4. Select the BLADE group from the model tree, and start the Pattern tool.
5. Select the **11.25** degree dimension and enter **22.5** as the increment.
6. Use the dashboard to enter **2** as the quantity and complete the feature, as shown in the following figure.

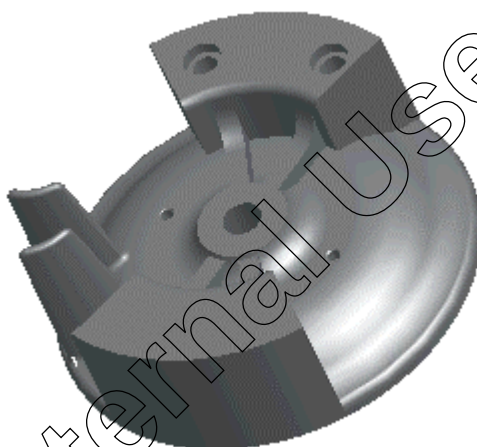


Figure 1: Patterning the Blade

7. Edit the pattern to change the number of instance to **5**. Regenerate the model, and notice that all patterned blades have incremental spacing of **22.5°**.
8. Edit the second group within the pattern. Select the **22.5°** dimension, and edit its properties to change its name to **ANGLE_SPACING**.
9. Edit the name of the quantity dimension to **BLADE_QTY**.

Best Practice:

It is a best practice to rename dimensions that are used in relations to make identification easier.

Step 2. Create relations on the FLYWHEEL.PRT.

1. Click **Tools > Relations**. Enter the following to create a comment:
/* Space blades equally
2. On the next line, enter **ANGLE_SPACING = 360 / BLADE_QTY**.

3. Complete the relation and regenerate the model. Notice the blades are now equally spaced as shown in the following figure.

Notes:

The */** designates a comment within a relation.

You can enter in dimension symbols manually or use the relations toolbar to select them from the graphics window.

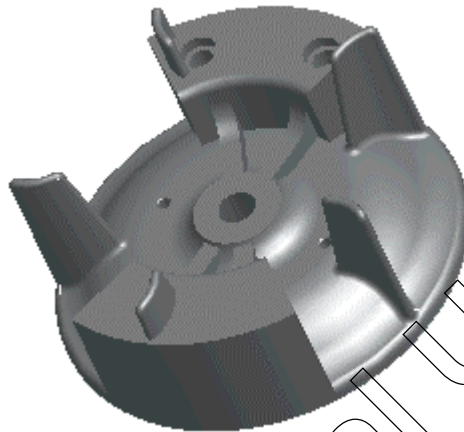


Figure 2: Pattern with Relation

4. Edit the pattern, and enter **16** for the quantity. Regenerate the model.

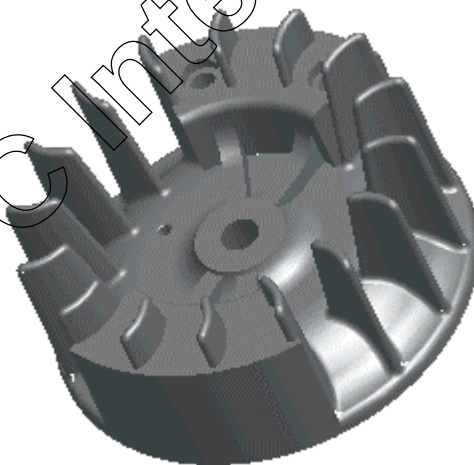


Figure 3: Completed FLYWHEEL.PRT

5. Save the model.
6. Close the window.

Step 3. Edit the BOLT.PRT.

Instructor: Skip this step for the demo.

1. Open BOLT.PRT.
2. Edit the protrusion that forms the bolt head and click **Info > Switch Dimensions**.
3. Note that the dimension names have already been edited. Edit the HEX cut to view the names of the dimensions, as shown in the following figure.

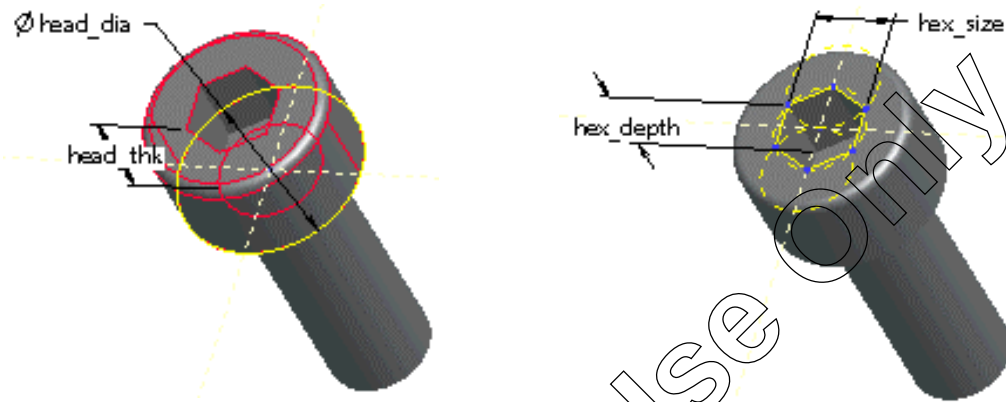


Figure 4: Examining the BOLT.PRT Head (left) and Examining the Hex Cut (right)

Step 4. Create relations on the BOLT.PRT.

Instructor: Skip this step for the demo.

1. Click **Tools > Relations**. Enter the following on a new line to create a comment:
/* Adjust bolt head to changing diameters
2. Enter the following on new lines:
HEX_DEPTH = BOLT_DIA - 1
HEX_SIZE = BOLT_DIA - 1
HEAD_THK = BOLT_DIA
HEAD_DIA = BOLT_DIA * 1.8
3. Complete the relation and regenerate the model. Notice that the bolt head now updates according to the relations as shown in the following figure.

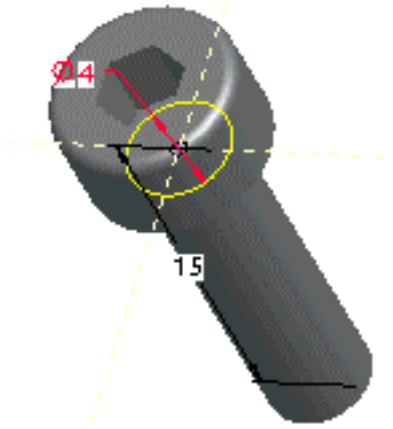


Figure 5: Regenerating after Adding the Relations

4. Click **Info > Switch Dimensions** to view the dimension values.
5. Edit the diameter of the bolt shaft to **4** and regenerate. Edit the diameter to **6** and regenerate.
6. Note that the bolt head updates according to the relations, as shown in the following figure.

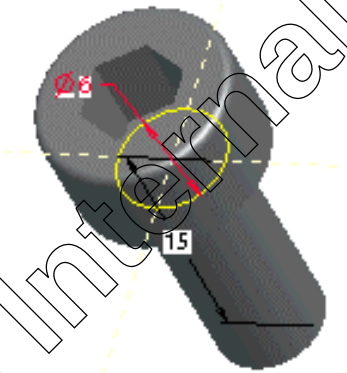


Figure 6: Completed BOLT.PRT

7. Save the model.

This completes the exercise.

Exercise 2: Creating Family Tables

Introduction

In this demonstration, we create a family of parts using family tables. Family tables enable us to create a large number of parts based on a single design. This is done by designating the dimensions, features, and parameters to be modified. The modifications are stored in a table. For example, we can create a family of screw parts where we change the screw length, diameter, and threads per millimeter.

Objectives

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

- Create new parts using a family table.

Scenario

Due to the lack of availability of project personnel from an overseas supplier, it is necessary for your company to quickly model the bolts that will be used within the drill assembly. You are assigned to take the current generic bolt and create a family of bolts using family tables.

Step 1. Prepare BOLT.PRT for the family table.

In this step, we examine the bolt that will become our generic bolt. Note that relations enable this bolt's head size and hex interface size to automatically change when the diameter changes. There is also a special TORX feature (a special type of screw) available for use in specific applications. Finally, we assign a name to the dimension that represents the bolt's length.

1. If necessary, open BOLT.PRT.
2. Use the model tree to suppress the HEX feature. A second cut that was previously hidden by the HEX feature is now visible.
3. Rename the CUT feature shown in the following figure to **TORX**.

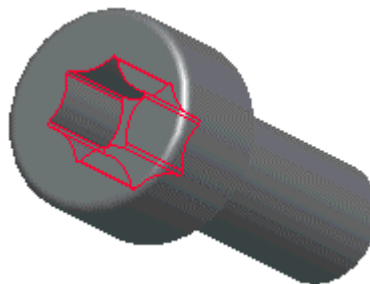


Figure 7: Highlighted TORX Feature

4. Show suppressed features in the model tree. Resume the HEX feature.
5. Select and edit the protrusion forming the shaft on BOLT.PRT. Select the length dimension, right-click, and select **Properties**.
6. Select the **Dimension Text** tab and edit the Name from d123 to **BOLT_LENGTH**. Complete the edit.
7. With the dimensions still visible, click **Info > Switch Dimensions** to view the dimension symbols as shown in the following figure.

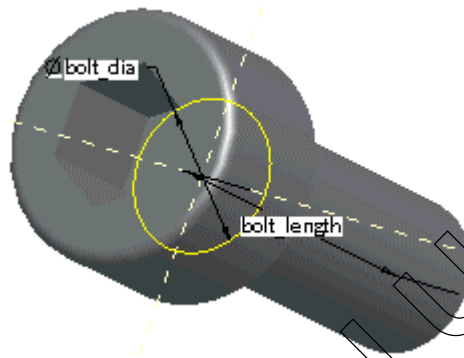


Figure 8: Editing the Dimension Name

8. Click **Info > Switch Dimensions** again to view the dimensions as values, and repaint.
9. Click **Tools > Parameters** to view the parameters associated to BOLT.PRT.
10. Click **Parameters > Add Parameter** to create a parameter to track cost.
11. Enter **Cost** as the name of the parameter.
12. Enter **Real Number** as the parameter type.
13. Enter **0.05** for the parameter value.
14. Enter **Hex Head** in the value column as the Description parameter. Enter **ACME Bolts** in the value column as the Modeled_By parameter. Complete the parameter editing.

Step 2. Create a family table.

In this next step, we create the family table that enables us to specify changes in the bolt's length and diameter. We are also able to interchange the HEX head bolt and the TORX head bolt in the model. Also note that we add a chamfer feature to all bolts that have a diameter of 6.0mm.

1. Click **Tools > Family Table**. Use the toolbar to Add columns to the table. Select the shaft of the BOLT.PRT and then select the diameter dimension and the length dimension.
2. Click **Feature**, and select the **HEX**, **TORX**, and **Chamfer** features from the model tree.

3. Click **Parameter**, and select both the **DESCRIPTION** and **COST** parameters then click **Insert Selected**.
4. Click **Close** from the Select Parameter dialog box. Click **OK** from the Family Items dialog box.
5. Use the toolbar to Insert a new row (instance) into the table. Enter **BOLT_6-15** as the Instance Name.
6. Enter **6** for the BOLT_DIA and **15** for the BOLT_LENGTH.
7. Select **Y** for HEX, **N** for TORX, and **Y** for CHAMFR.
8. Enter **Hex Head** for DESCRIPTION and **0.05** for COST.

Note:

In a family table, rows are referred to as instances, and become nearly equivalent of another model. In addition, dimension columns allow values to be entered, while feature columns can be resumed or suppressed by selecting Y or N.

Step 3. Create the remaining instances.

Instructor: Demonstrate the creation of BOLT_5-18 and BOLT_4-15 only.

1. Create several more instances for the BOLT.PRT, as shown in the following figure. Note that the HEX cut is used for all 6 and 5 diameter instances, while the TORX cut is used only on the 4 diameter instances. Also, the chamfer is only used on the 6 diameter instances.

Note:

If this model did not have relations to control the head size with respect to the diameter, several more columns would need to be created in the table.

Type	Instance Name	d1 BOLT DIA	d0 BOLT_LENGTH	F85 HEX	F178 TORX	F155 [CHAMFR]	DESCRIPTION	COST
	BOLT	6.00	15.00	Y	Y	Y	Hex Head	0.050
	BOLT_6-15	6.00	15.00	Y	N	Y	Hex Head	0.050
	BOLT_6-20	6.00	20.00	Y	N	Y	Hex Head	0.060
	BOLT_6-36	6.00	36.00	Y	N	Y	Hex Head	0.070
	BOLT_5-15	5.00	15.00	Y	N	N	Hex Head	0.040
	BOLT_5-18	5.00	18.00	Y	N	N	Hex Head	0.050
	BOLT_5-24	5.00	24.00	Y	N	N	Hex Head	0.055
	BOLT_5-28	5.00	28.00	Y	N	N	Hex Head	0.060
	BOLT_5-50	5.00	50.00	Y	N	N	Hex Head	0.070
	BOLT_5-60	5.00	60.00	Y	N	N	Hex Head	0.080
	BOLT_4-12	4.00	12.00	N	Y	N	Torx Head	0.030
	BOLT_4-15	4.00	15.00	N	Y	N	Torx Head	0.035
	BOLT_4-18	4.00	18.00	N	Y	N	Torx Head	0.040

Figure 9: Creating Family Table Instances

Note:

Family Table instances can be entered manually, using cut/paste, importing from a text file, or even by using Excel. Also, there is a **copy instance with increments** option that enables you to generate arrays of incrementing sizes for instances.

2. Use the toolbar to Verify all instances.
3. Use the toolbar to Preview the **BOLT_6-15**, **BOLT_5-18**, and **BOLT_4-15** instances.

Step 4. Examine additional instances and open the Generic.

Instructor: Skip this step for the demo.

1. Select the **BOLT_4-18** instance and click **Open**. Examine its features.
2. Close both windows.
3. Open the BOLT.PRT. Note that the generic model or any of its instances can be opened.
4. Open the **BOLT_6-20** instance. Examine its features.
5. Close the window.
6. Open BOLT.PRT. Then, open **The Generic**.
7. Save the model.
8. Close all windows.
9. Erase all models from memory.

This completes the exercise.

Exercise 3: Creating Family Tables (Challenge)

Objective

In this exercise, you practice creating family tables based on the experience you gained from previous exercises.

Scenario

You are assigned to create a family table of drill bits that may be sold as extras to the drill.

Step 1. Create a family table of STD_BIT.PRT.

1. Open STD_BIT.PRT.

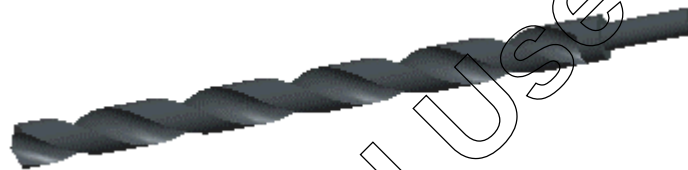


Figure 10: Examining the STD_BIT.PRT

2. Edit the dimension names of the MAIN_PROTRUSION to be **BIT_LENGTH** and **BIT_DIA**.
3. Create a family table with **18** instances, where the bit's diameter increases from **8** to **25**. With each diameter increase of **1**, add **8** to the instance length.
4. Since the drill chuck only accepts up to **12** diameter shafts, add a column for **REDUCTION_CUT** for the instances with a shaft diameter greater than **12**.
5. Edit the instance names to reflect their size (ex: STD_BIT_14-266).
6. Verify all instances.
7. Preview and open various instances.
8. Save the generic model.

This completes the exercise.

Summary

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

- Create design model variations using family tables.
- Configure relations that capture a part's design intent.
- Create parameters to capture additional information.
- Describe how family tables, relations, and parameters affect parent/child relationships.

For PTC Internal Use Only

Assembling Components

Introduction

Most commercial product designs contain many components. Pro/ENGINEER Wildfire enables you to manually and automatically locate components in an assembly using the constraints and component interfaces.

Additionally, the assembly components are flexible. You can assemble a component multiple times, while varying the dimensions as each component is added.

Objectives

After completing this module, you will be able to:

- Create assembly models.
- Assemble components using constraints.
- Assemble components using component interfaces.
- Assemble flexible components.
- Describe how assembling components affects parent/child relationships.



Instructor Preparation

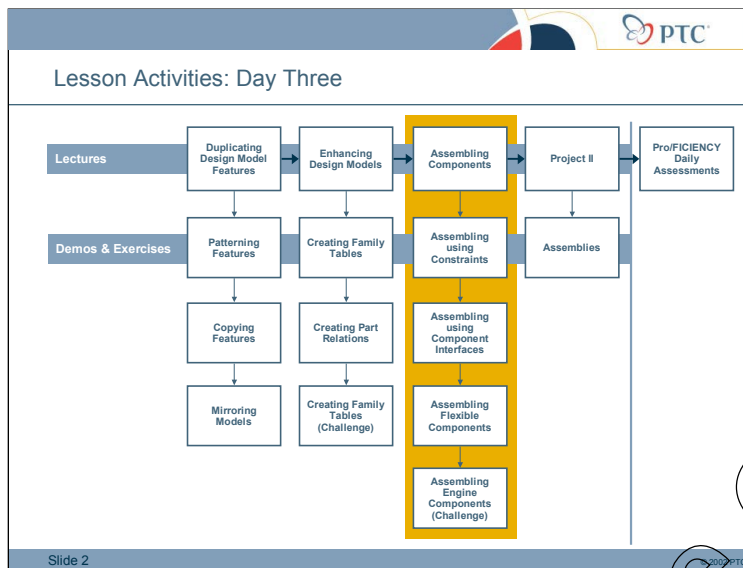
Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (3): 40 mins
- Labs (3): 70 mins
- Total: 2 hr



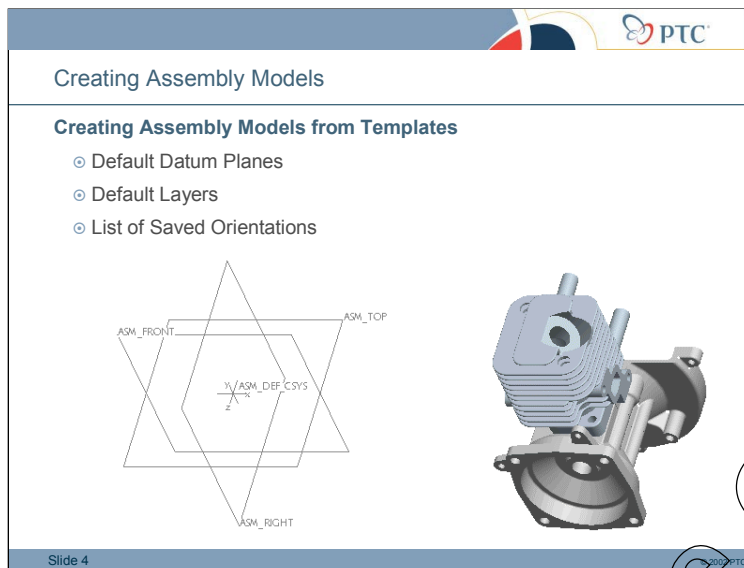
Objectives

After completing this module, you will be able to:

- Create assembly models.
- Assemble components using constraints.
- Assemble components using component interfaces.
- Assemble flexible components.
- Describe how assembling components affects parent/child relationships.

Slide 3

- Most commercial product designs contain many components.
- Pro/ENGINEER Wildfire enables you to manually and automatically locate components in an assembly using the constraints and component interfaces.
- Additionally, the assembly components are flexible. You can assemble a component multiple times, while varying the dimensions as each component is added.



- Assembly templates are similar to part templates.
- Assembly templates save you time by providing common default datum planes, layers, and saved view orientations.
- When you first create a new assembly, you apply an assembly template.



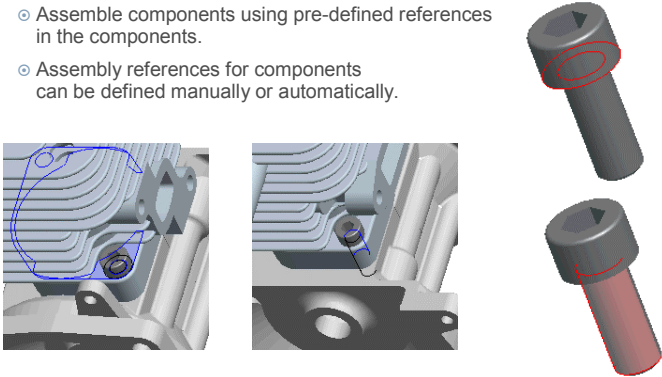
- Constraints are applied between parts within an assembly.
- You use constraints to assembly parts.
- Constraints enable you to give your assemblies design intent, so that when you change a dimension on a part, the assembly reacts as predicted.

PTC

Assembling Components using Component Interfaces

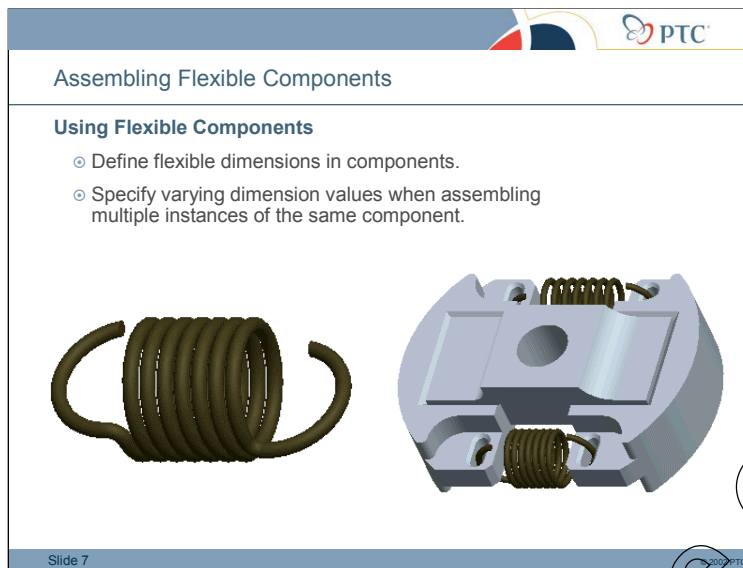
Creating Component Interfaces

- Assemble components using pre-defined references in the components.
- Assembly references for components can be defined manually or automatically.

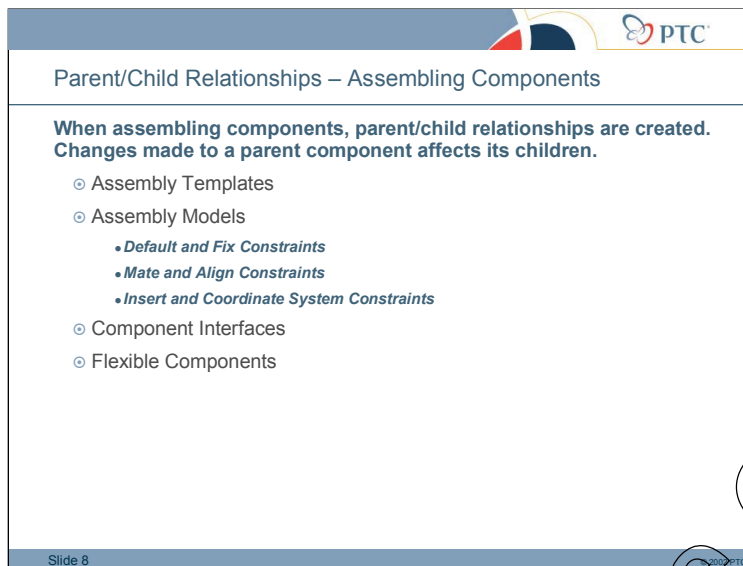


Slide 6

- You use component interfaces to specify references on a component.
- Then, when you assemble a component, you can simply select the references on the assembly that correlate to the pre-defined component references.
- Component interfaces are ideal for common parts that are generally assembled using the same references. A bolt is a good example.
- Using component interfaces saves time. When you assemble a common component, you do not have to select references on the component. While this might seem insignificant when assembling a single part, imagine you are assembling 30 or more instances of a common bolt.



- Flexible components can have varied dimensions within a single assembly.
- For example, the camshaft springs in the engine of a car can be either **flexed** or **relaxed**, depending on the current state of the piston's rotation.
- With Pro/ENGINEER Wildfire, all camshaft springs can be assembled in their natural state. Some springs can be in a flexed state, while others springs can be in relaxed state.
- This is an important feature because your 3-D models can appear exactly as they will when they are manufactured.
- In addition, the Bill of Material can list two of the same spring part, but the assembly can display visually different springs parts.



Note to instructors:

Use this slide to teach students how the functionality that they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

NOTE: It might be best to review this information after completing the demonstrations.

Like part templates, assembly templates do not have parent/child relationships.

Assembly models are the parent to all part models within the assembly. Assembly models can also be children if they are assembled to other assembly models.

Component interfaces create parent/child relationships between the predefined component interfaces and the manually selected assembly references.

Flexibly components contain parent/child relationships like other components except that they are saved in varying states of regeneration.



You will follow along while I demonstrate:

1. Assembling using Constraints
2. Assembling using Component Interfaces
3. Assembling using Flexible Components

Then, you will practice the same steps in the laboratory exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 10

Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY assessment questions.
- **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 Exams

Description: For each course, 5 new 10 question Pro/FICIENCY 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:

- Create assembly models.
- Assemble components using constraints.
- Assemble components using component interfaces.
- Assemble flexible components.
- Describe how assembling components affects parent/child relationships.

Referencing PTC Help for Module 11 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Constraints	Assembly and Welding	About Placing Components Placement Constraint Types Using the Component Placement Dialog Box About Freeform Mouse-Driven Component Manipulation Constraint Orientation Assumptions
Component Interfaces	Assembly and Welding	About Component Interfaces
Flexible Components	Assembly and Welding	About Flexible Components

Module 11 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Assembling Using Constraints

Introduction

In this demonstration, we create assemblies using automatic constraints.

Automatic constraints enable you to easily select the features of the parts being assembled.

Pro/ENGINEER Wildfire automatically mates, aligns, or inserts the parts together.

Objectives

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

- Assemble part components using constraints.

Scenario

Now that a majority of parts for the drill assembly have been modeled, you are assigned to begin creating several subassemblies using automatic constraints to assemble the individual parts.

Step 1. Create ENGINE.ASM.

Initially, we create the assembly using a default template, and then assemble the rear engine block using the default constraint.

We assemble the front engine block and the cylinder using Insert and Mate constraints.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise, start Pro/ENGINEER Wildfire.
2. Set the working directory to:
C:\users\student\fast_track_330\module_11.

3. Click **File > New > Assembly**. Enter **Engine** as the name, and use the default template.
4. Using the toolbar, add a component to the assembly. Open ENG_BLOCK_REAR.PRT and create a Default constraint. Complete the component placement.
5. Turn off the display of all datum features. Using the toolbar, add another component to the assembly. Open ENG_BLOCK_FRONT.PRT.
6. Press CTRL + ALT + right-drag to position the component approximately in place. Refer to the following figure for the next few steps.
7. Zoom in and select the cylindrical surface of an alignment pin and a cylindrical surface on the corresponding hole. Note that this has created an Insert constraint.

Note:

To easily select surfaces, zoom in before selecting.

8. Repeat the process to select the other alignment pin and hole. Press CTRL + ALT + right-drag. Note the component now moves according to its constraints.
9. Select the two mating surfaces. Note this has created a Mate constraint. Specify the Offset to be **Coincident**. Complete the component placement.

Note:

Depending on the proximity of the component, the mate constraint may prompt for an offset value. In this case, accept the default offset, and then change the mate offset to **Coincident**.

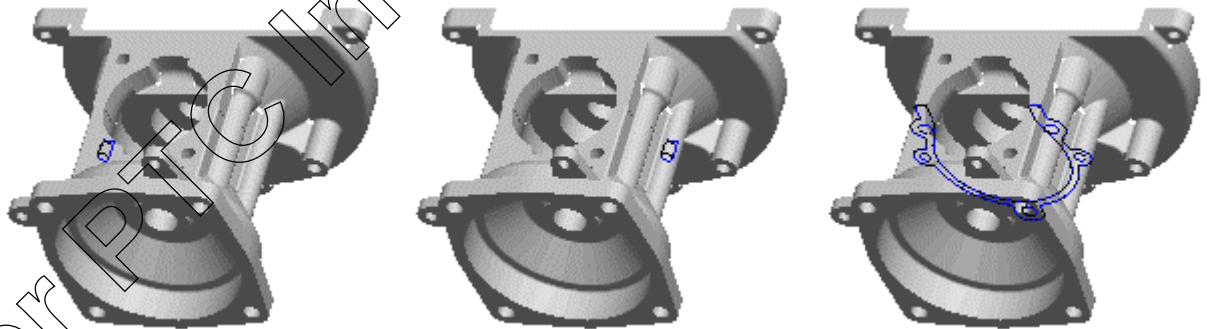


Figure 1: Assembling the ENG_BLOCK_FRONT.PRT

10. Orient to the standard orientation. Using the toolbar, add a component to the assembly. Open CYLINDER.PRT.
11. Press CTRL + ALT + right-drag to position the component approximately in place. Refer to the following figure for the next few steps.
12. Zoom in and select the cylindrical surfaces of a pair of corresponding holes. Note this has created an Insert constraint.

13. Repeat the process to select the other pair of corresponding holes.
14. Press CTRL + ALT + right-drag. Note the component now moves according to its constraints.
15. Select the two Mating surfaces. Note this has created a Mate constraint. Specify the Offset to be **Coincident**. Complete the component placement.

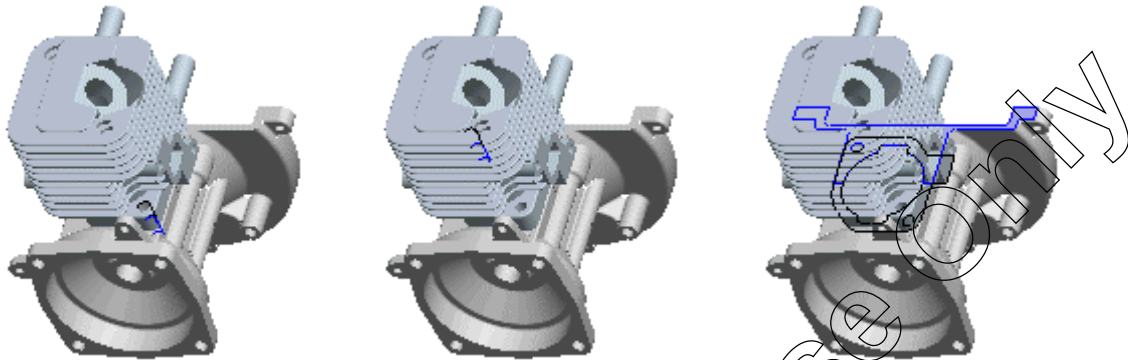


Figure 2: Assembling the CYLINDER.PRT

16. Save the model.
17. Close the window.

Step 2. Create DRILL_CHUCK.ASM.

Next, we create the drill chuck assembly.

We create the assembly using the default template, and then we assemble the chuck using the default constraint.

We use the insert and align offset constraints to assemble the chuck collar, and assemble the standard bit using coordinate systems.

1. Create a new assembly named **Drill_Chuck**.
2. Assemble CHUCK.PRT using a default constraint.
3. Assemble CHUCK_COLLAR.PRT. Drag the component into an approximate position. Select the major cylindrical surface to create an Insert constraint.
4. Drag the component towards the front of the assembly. Select the two planar surfaces on the back side to create an Align constraint. Enter **-1** as the offset.
5. Complete the component placement, as shown in the following figure.

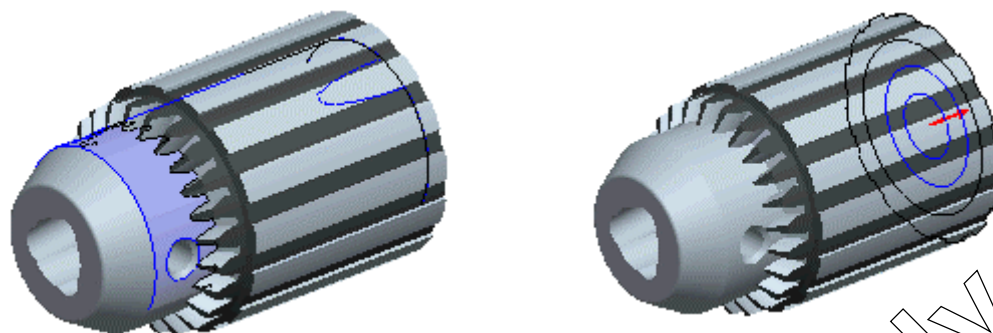


Figure 3: Assembling the CHUCK_COLLAR.PRT

Note:

The CHUCK_COLLAR is assembled as fully constrained by using the **Allow Assumptions** option. In this case, the rotation of the component is not important.

6. Assemble the STANDARD_BIT.PRT. Drag the component into an approximate position.
7. Display the Coordinate Systems. Select the BIT_CSYS and BIT_ALIGN coordinate systems to create a Coord Csys constraint.
8. Complete the component placement, as shown in the following figure.
9. Turn off display of Coordinate Systems.



Figure 4: Completed DRILL_CHUCK.ASM

10. Save the model.
11. Close the window.

Step 3. Create CRANK.ASM.

Instructor: Skip this step for the demo.

1. Click **File > New > Assembly**. Enter **Crank** as the name, and use the default template.
2. Using the toolbar, add a component to the assembly. Open the CRANKSHAFT.PRT and create a Default constraint. Complete the component placement.

3. Using the toolbar, add another component to the assembly. Open the FLYWHEEL.PRT.
4. Press CTRL + ALT + right-drag and CTRL + ALT + middle-drag to position the component approximately in place. Refer to the following figure for the next few steps.
5. Manually specify a Mate constraint and select the 2 conical surfaces, as shown in the following figure.
6. Display datum planes. Add a constraint, and select the KEY planes to create an Align constraint. Set the Offset option to be **Coincident**.
7. Complete the component placement, as shown in the following figure.

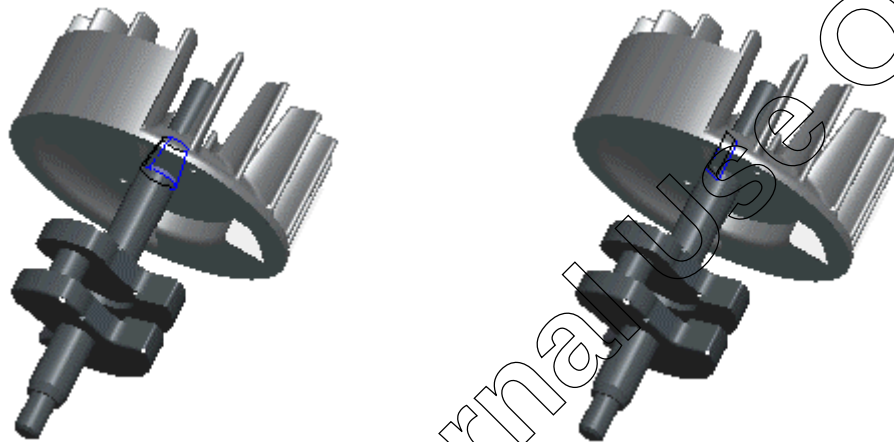


Figure 5: Assembling the FLYWHEEL.PRT

8. Turn off display of datum planes.
9. Save the model.
10. Close the window.

Step 4. (OPTIONAL) Create PISTON.ASM.

Instructor: Skip this step for the demo.

1. Create a new assembly named PISTON.
2. Assemble PISTON.PRT using a default constraint.
3. Assemble PISTON_PIN.PRT. Drag the component into an approximate position. Select cylindrical surfaces to create an Insert constraint.
4. Start the Search tool. Find and select the FRONT plane from PISTON.PRT.
5. Start the Search tool again. Find and select the FRONT plane from PISTON_PIN.PRT. Specify an Align Coincident constraint.

Note:

Depending on the proximity of the component, the align constraint may prompt for an offset value. In this case, accept the default offset, and then change the align offset to **Coincident**.

6. Complete the component placement, as shown in the following figure.

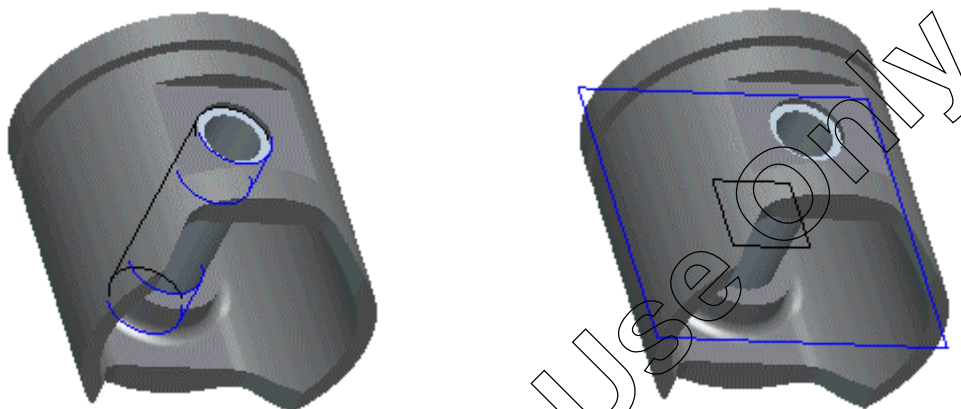


Figure 6: Assembling the PISTON_PIN.PRT

Note:

The PISTON_PIN was considered Fully Constrained with the use of the Allow Assumptions option. In this case, we do not have a preference with the orientation of the PISTON_PIN in the PISTON. If orientation is important, disable the Allow Assumptions option and create a third constraint.

7. Assemble PISTON_RING.PRT. Drag the component into an approximate position.
8. Select a planar surface from the bottom of the ring and the bottom of the ring groove to create a Mate Coincident constraint.
9. Select the cylindrical surfaces from the outside of the PISTON_RING and PISTON to create an Insert constraint.
10. Press CTRL + ALT + middle-drag to rotate PISTON_RING.PRT, and then locate the gap as desired. Create a Fix constraint to fully constraint the component.
11. Complete the component placement, as shown in the following figure.



Figure 7: Completed PISTON.ASM

12. Save the model.

13. Close the window.

This completes the exercise.

Exercise 2: Assembling Using Component Interfaces

Introduction

In this demonstration, we assemble components using pre-defined component interfaces.

Objectives

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

- Assemble part components using component interfaces.

Scenario

You are assigned to continue assembling the engine of the drill by locating the bolts on the assembly. A co-worker recommends that you use component interfaces to take advantage of their productivity enhancement when assembling the same component multiple times.

Step 1. Define the component interfaces.

Before we can assemble parts using component interfaces, we must define the component interfaces. Using component interfaces saves time.

When you assemble a common component, you do not have to select references on the component. While this might seem insignificant when assembling a single part, imagine that you are assembling 30 or more instances of a common bolt.

First, we create the component interface on a generic bolt. Then, we assemble five instances of the bolt to the engine using various techniques.

1. Open BOLT.PRT and click **The Generic > Open**.
2. Click **Edit > Setup > Comp Interface**. Edit the Interface Name to BOLT.
3. Specify an Insert constraint and select the cylindrical surface, as shown in the following figure.
4. Click **Add**, specify a Mate constraint, and then select the planar surface, as shown in the following figure.



Figure 8: Creating the Insert and Mate Component Interfaces

5. Complete the interface.
6. Save the model.
7. Close the window.

Step 2. Assemble BOLT.PRT utilizing component interfaces.

Now that we have defined the component interface on a generic bolt, we can assemble the bolt using component interfaces to the engine assembly.

In this step, we assemble five instances of the bolt on the engine assembly.

If we assemble the initial component automatically, the mated surfaces default to an offset setting.

Instead of setting an offset distance, we make the part coincident to the assembly. We can use these settings for all of the bolts, but we can also autoplacement the bolt.

We can locate multiple bolts using the same ASSEMBLY dialog box.

We can assemble bolts directly from the navigator browser.

All of these assembly methods using component interfaces enhance our productivity. We can select the method best suited to the situation.

1. Open ENGINE.ASM.
2. Select and hide CYLINDER.PRT. Review the bolt placement holes, as shown in the following figure.

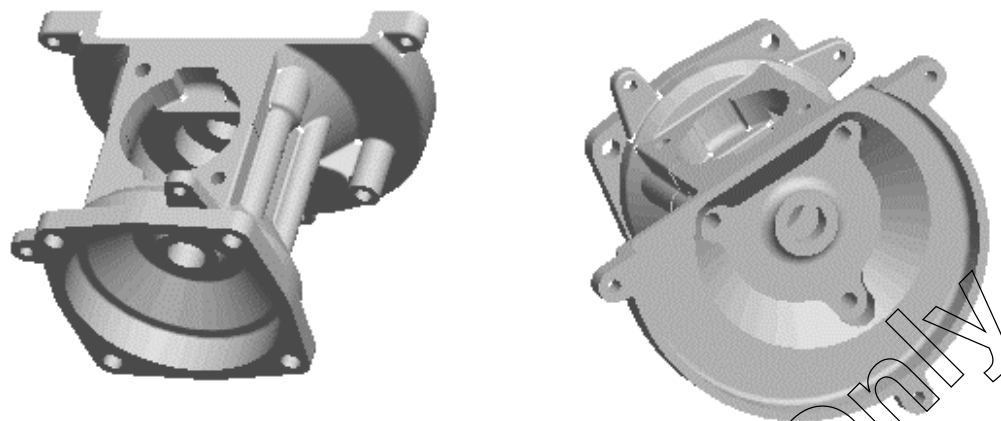


Figure 9: Reviewing the Bolt Placement Holes

3. Unhide CYLINDER.PRT and assemble BOLT.PRT. Open the BOLT_5-18 instance.
4. Select the BOLT interface, and then click **Ok**.
5. Select insert and mating surfaces to assemble the component, as shown in the following figure.

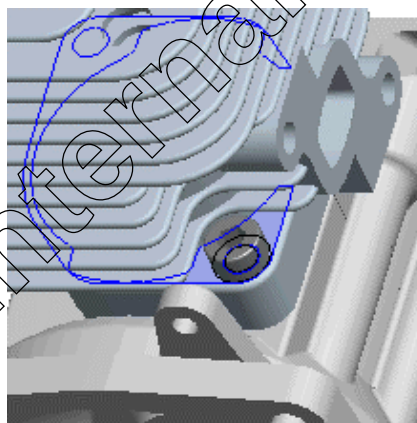


Figure 10: Assembling the 5-18 BOLT.PRT Instance with the Component Interface

6. Complete the component placement.

Step 3. Assemble the second bolt.

Instructor: Skip this step for the demo.

1. Assemble BOLT.PRT again, and then open the BOLT_5-18 instance.
2. Select the BOLT interface, and then click **Ok**.
3. Select insert and mating surfaces to assemble the component, as shown in the following figure.

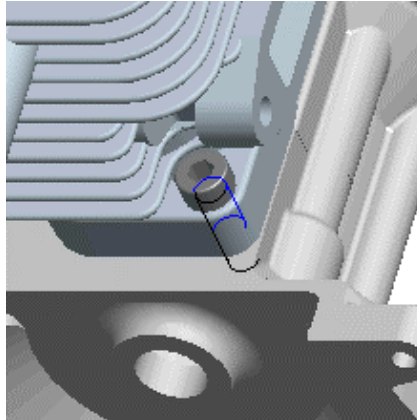


Figure 11: Assembling a Second 5-18 BOLT.PRT Instance with the Component Interface

4. Complete the component placement.

Step 4. Assemble the third bolt.

1. Assemble the BOLT.PRT again, and then open the BOLT_5-28 instance.
2. Select the BOLT interface, and then click **Auto Place**.
3. Select an approximate location to automatically assemble the bolt, as shown in the following figure. Select among the placement options (rows) to find the appropriate constraint combination. Click **Apply** and leave the dialog box open.

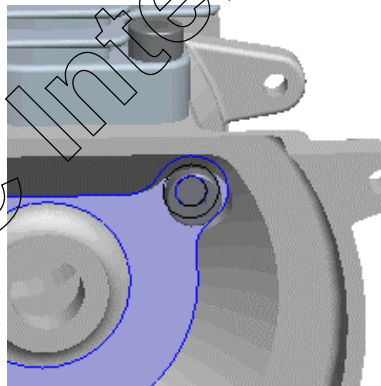


Figure 12: Assembling the 5-28 BOLT.PRT Instance Automatically

Step 5. Assemble the fourth bolt.

Instructor: Skip this step for the demo.

1. Select an approximate location to automatically assemble the bolt, as shown in the following figure. Select among the placement options (rows) to find the appropriate constraint combination. Click **Ok** to complete the placement.

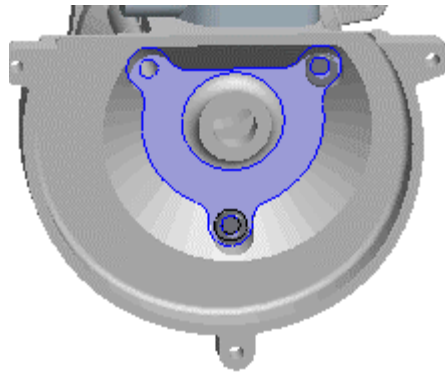


Figure 13: Assembling the 5-28 BOLT.PRT Instance from the Same Dialog Box

Step 6. Assemble the fifth bolt.

1. Minimize the model tree. Using the navigator, browse to BOLT.PRT.
2. Zoom in to the remaining hole, and then drag the bolt directly into the hole. Open the BOLT_5-28 instance. Refer to the following figure.

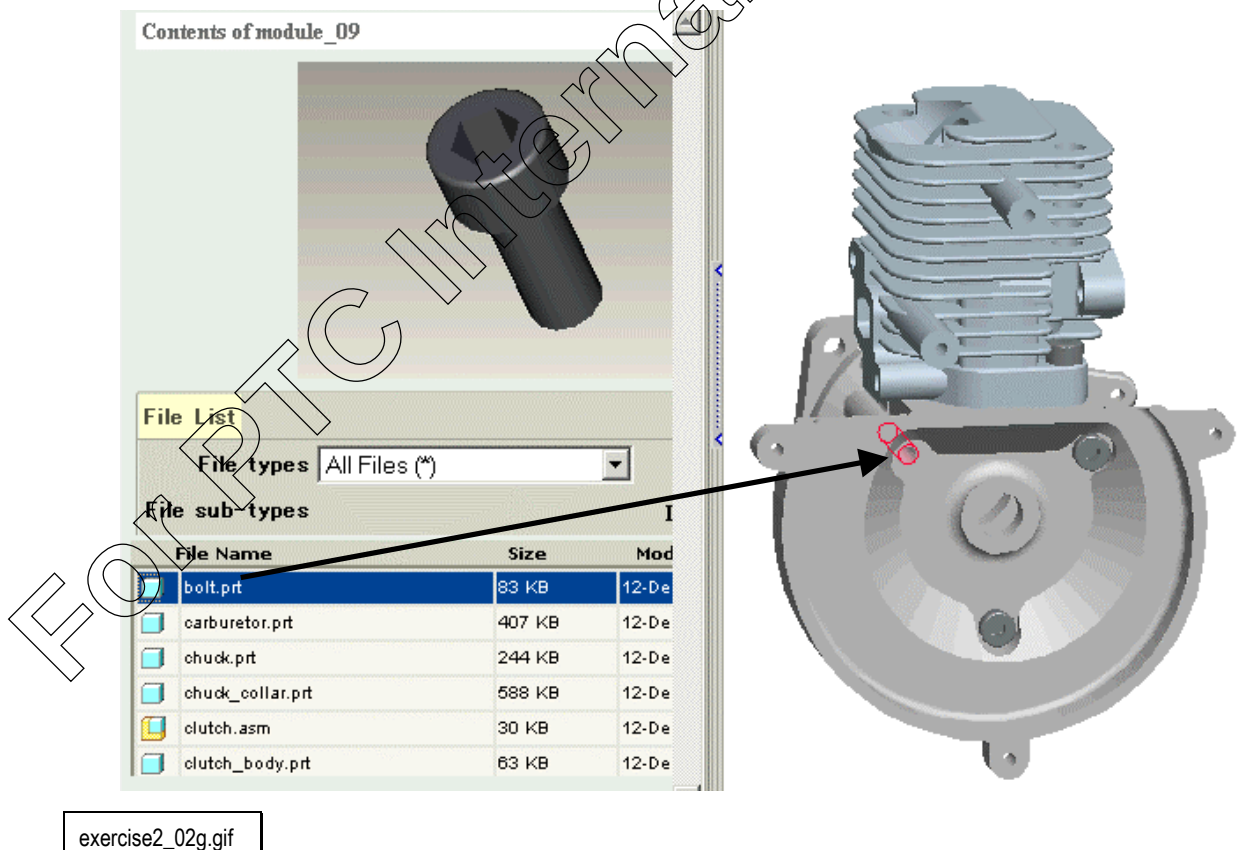


Figure 14: Assembling a Component Directly from the Navigator

Note:

If BOLT.PRT does not align properly, edit the definition of the appropriate BOLT.PRT and change its constraints accordingly.

3. The assembled bolts should appear as shown in the following figure.

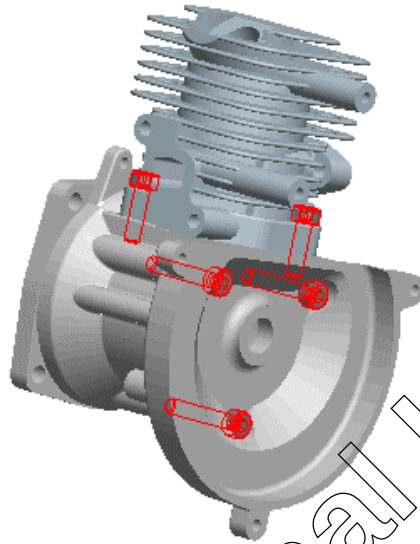


Figure 15: Completed ENGINE.ASM

4. Save the model.
5. Close the window.

This completes the exercise.

Exercise 3: Assembling Flexible Components

Introduction

In this demonstration, we assemble flexible components.

We define the clutch spring as a flexible component, and then we assemble two clutch springs to the clutch assembly.

Objective

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

- Assemble part components using flexible components.

Scenario

In order to create real-life figures for the Marketing Department's brochures, you are assigned to assemble the drill's clutch using flexible components. First, define the clutch springs as flexible components. Then, assemble the clutch springs into the clutch assembly.

Step 1. Define the component flexibility.

In this step, we define the helical length of the spring as a flexible dimension. This allows the dimension to be changed once it is assembled.

1. Open CLUTCH_SPRING.PRT.



Figure 16: CLUTCH_SPRING.PRT

2. Using the model tree, select the SPRING_LENGTH point, right-click and select **Edit**.
3. Edit the value from 18 to **22**, and then regenerate the model, as shown in the following figures.

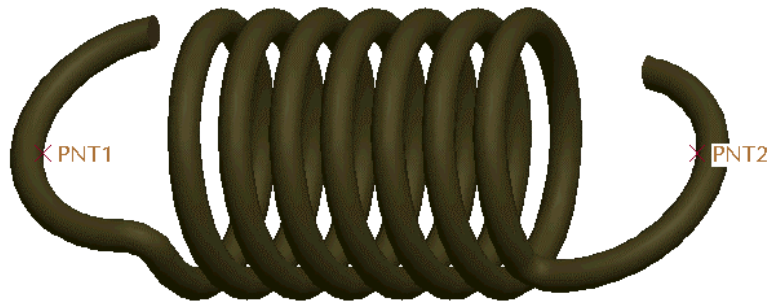


Figure 17: Editing the Spring Pitch by Changing the Length

4. Repeat the previous procedure to edit the dimension back to **18**, and then regenerate.
5. Click **Edit > Setup > Flexibility**. Select the SPRING_LENGTH point.
6. Select the **18** dimension (length), and then click **Ok**. Complete the Flexibility definition.
7. Save the model.
8. Close the window.

Step 2. Assemble the first CLUTCH_SPRING.PRT.

Now that we have defined the clutch spring as a flexible component, we can assemble the clutch spring into the clutch.

The clutch spring is an ideal component to define as flexible, since the distance between the left and right clutch shoes are different.

To assemble the first clutch spring, we align the PNT1 and SPRING1 points as coincident. We align the PNT2 and SPRING1 points as coincident.

To assemble the second clutch spring, we align the PNT1 and SPRING2 points as coincident. We align the PNT2 and SPRING2 points as coincident.

Finally, we note that the Bill of Materials (BOM) lists two clutch springs.

1. Open CLUTCH.ASM.
2. Display Datum Points. Note that the distance between the upper and lower sets of points is different.

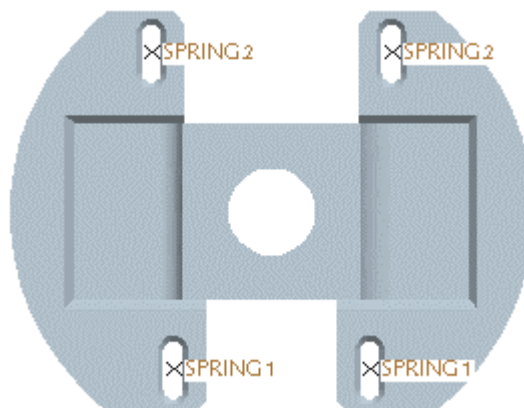


Figure 18: Examining the Distance between the Clutch Shoes

3. Click **Insert > Component > Flexible**. Open CLUTCH_SPRING.PRT.
4. Change the Method to **Distance**. Select both SPRING1 points.
5. Click **Close > Placement**, and then drag the component into an approximate position near the lower set of points.
6. Select the PNT1 and SPRING1 points to create an Align Coincident constraint. Press CTRL + ALT + middle-drag to rotate the spring slightly away from the assembly.
7. Select the PNT2 and SPRING1 points to create a second Align Coincident constraint, as shown in the following figure.
8. Press CTRL + ALT + middle-drag to rotate the spring hooks vertical, avoiding interference. Add a fix constraint to fully constrain CLUTCH_SPRING.PRT.

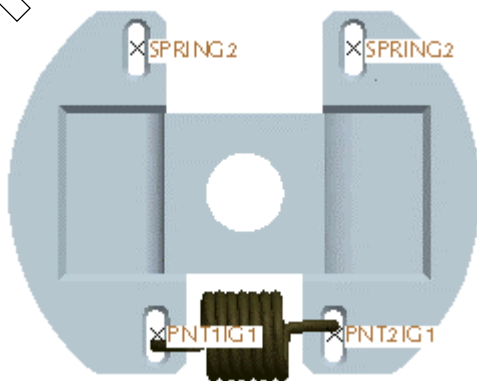


Figure 19: Inserting the CLUTCH_SPRING.PRT as a Flexible Component

9. Complete the component placement.

Step 3. Assemble the second CLUTCH_SPRING.PRT.

1. Click **Insert > Component > Flexible**. Open CLUTCH_SPRING.PRT.

2. Change the Method to **Distance**. Select both SPRING2 points.

Note:

If changes are made such that the components of the assembly are moved or changed, then the CLUTCH_SPRING.PRT will update.

3. Click **Close > Placement**. Drag the component into an approximate position near the upper set of points.
4. Select the PNT1 and SPRING2 points to create an Align Coincident constraint. Press CTRL + ALT + middle-drag to rotate the spring slightly away from the assembly.
5. Select the PNT2 and SPRING2 points to create a second Align Coincident constraint, as shown in the following figure.
6. Press CTRL + ALT + middle-drag to rotate the spring hooks vertical, avoiding interference. Add a fix constraint to fully constrain CLUTCH_SPRING.PRT.
7. Complete the component placement, as shown in the following figure.

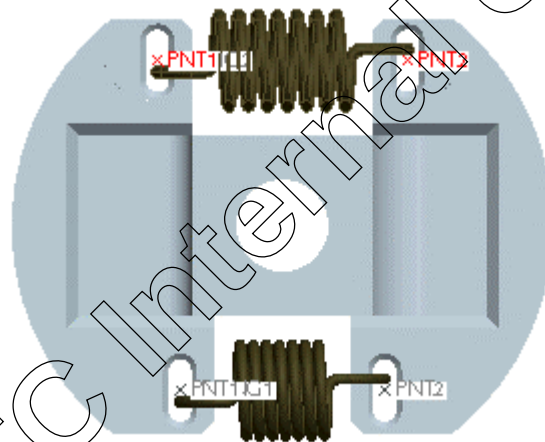


Figure 20: Inserting another CLUTCH_SPRING.PRT as a Flexible Component

Note:

Note that the CLUTCH_SPRING.PRT models in the model tree have symbols indicating flexible components.

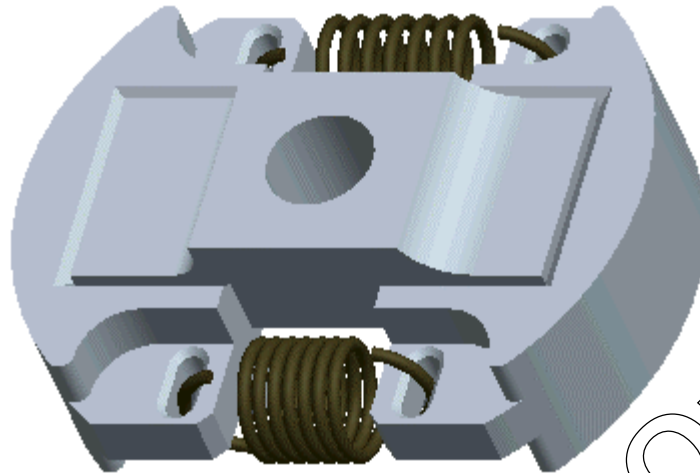


Figure 21: Completed CLUTCH.ASM

8. Click **Info > Bill Of Materials > Ok**. Note that the BOM lists two CLUTCH_SPRING models.
 9. Save the model.
 10. Close the window.
 11. Erase all models from memory.
- This completes the exercise.

Exercise 4: Assembling Additional Engine Components (Challenge)

Objective

In this exercise, you continue assembling the drill's engine.

Scenario

Complete the assembling of the drill's engine by placing and constraining the spark plug, muffler, bolts, manifold, carburetor, engine bearings, and the coil.

Step 1. Assemble SPARK_PLUG.PRT.

1. Open ENGINE.ASM.
2. Using the model tree, click **Settings > Tree Filters > Features > Ok**.
3. Drag the Insert indicator directly after CYLINDER.PRT.
4. Assemble SPARK_PLUG.PRT. Select the existing component interface, and then click **Auto Place**. Select the cylindrical surface just inside the spark plug hole. Complete the component placement, as shown in the following figure.

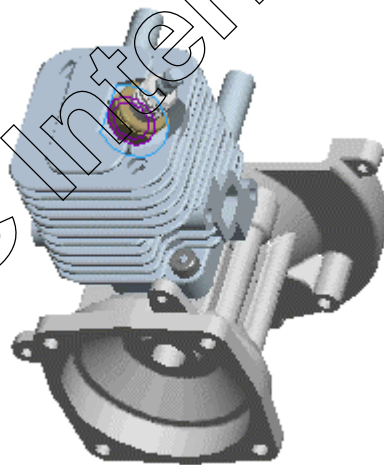


Figure 22: Assembling the SPARK_PLUG.PRT

5. Select the Insert indicator, right-click, and select **Cancel > Yes**.

Step 2. Assemble the MUFFLER.PRT.

1. Assemble MUFFLER.PRT to the cylinder, as shown in the following figure.
2. Assemble two BOLT_5-50 instances to fasten MUFFLER.PRT to CYLINDER.PRT, as shown in the following figure.

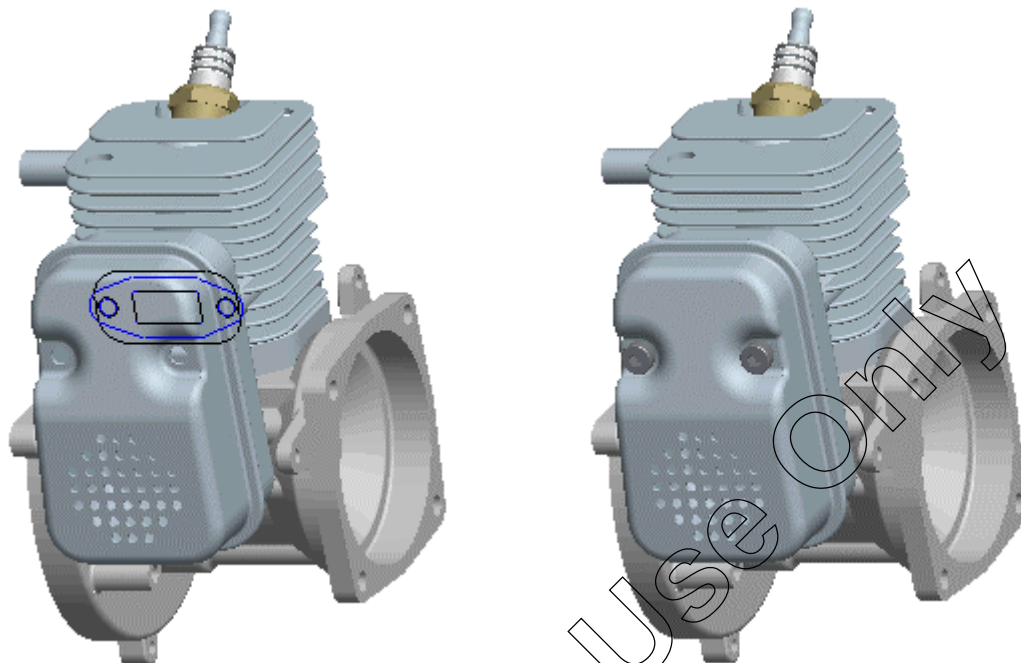


Figure 23: Assembling the MUFFLER.PRT (left) and Assembling Two 5-50 BOLT.PRT instances (right)

3. Save the model.

Step 3. Assemble CARBURETOR.ASM.

1. Using the default template, create a new assembly named **Carburetor**.
2. Using a default constraint, assemble MANIFOLD.PRT.
3. Save the assembly and close the window to return to the ENGINE.ASM window.
4. Assemble CARBURETOR.ASM, as shown in the following figure. Note that this produces a sub-assembly in the model tree.

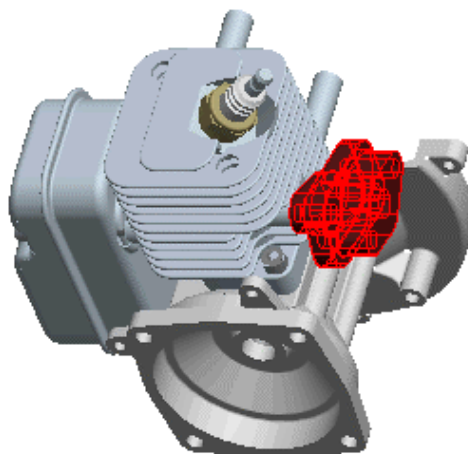


Figure 24: Assembling CARBURETOR.ASM to ENGINE.ASM

5. Using the model tree, select CARBURETOR.ASM, right-click, and select **Activate**.
6. Assemble CARBURETOR.PRT, as shown in the following figure.

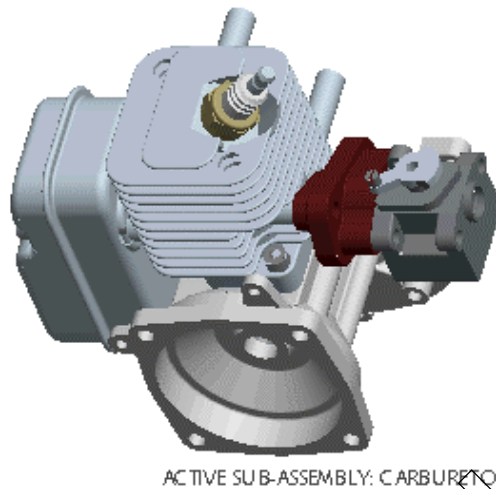


Figure 25: Assembling CARBURETOR.PRT to CARBURETOR.ASM

7. Assemble two BOLT_5-24 instances to fasten MANIFOLD.PRT to CYLINDER.PRT, as shown in the following figure.

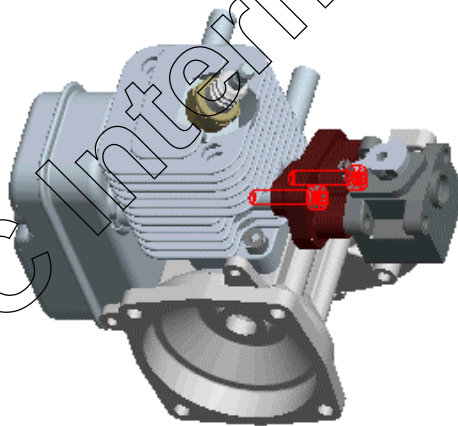


Figure 26: Assembling Two 5-24 BOLT.PRT Instances

8. Save the model.

Step 4: Assemble ENG_BEARING.PRT.

1. Using the model tree, select ENGINE.ASM, right-click, and select **Activate**.
2. Drag the Insert indicator directly after ENG_BLOCK_REAR.PRT.

Note:

If the **Insert** indicator is not visible, display features in the model tree.

3. Assemble ENGINE_BEARING.PRT, as shown in the following figure.

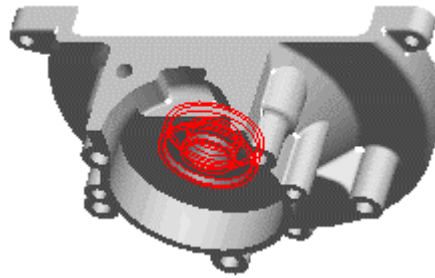


Figure 27: Assembling ENGINE_BEARING.PRT to ENG_BLOCK_REAR.PRT

4. Drag the Insert indicator directly after ENG_BLOCK_FRONT.PRT.
5. Hide ENG_BLOCK_REAR.PRT and ENGINE_BEARING.PRT.
6. Assemble another ENGINE_BEARING.PRT to ENG_BLOCK_FRONT.PRT, as shown in the following figure.

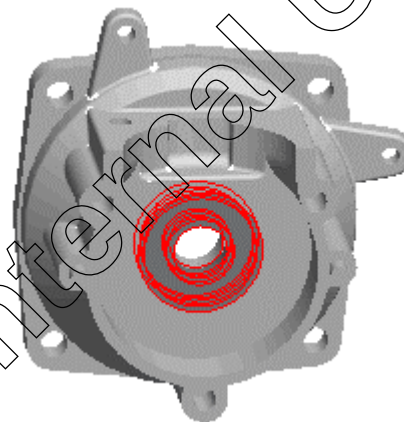


Figure 28: Assembling ENGINE_BEARING.PRT to ENG_BLOCK_FRONT.PRT

7. Unhide the hidden components.

Note:

Selecting **Unhide All** will display many unwanted features within all components.

8. Select the Insert indicator, right-click, and select **Cancel**.
9. Save the model.

Step 5. Assemble the COIL.PRT.

1. Assemble COIL.PRT, as shown in the following figure.

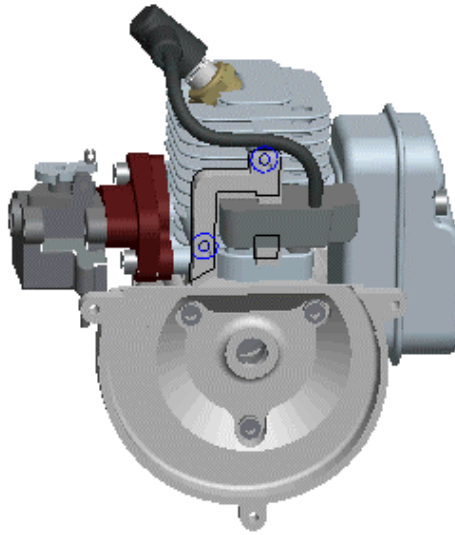


Figure 29: Assembling the COIL.PRT to the ENGINE.ASM

2. Assemble two BOLT_4-18 instances to fasten COIL.PRT to CYLINDER.PRT, as shown in the following figure.

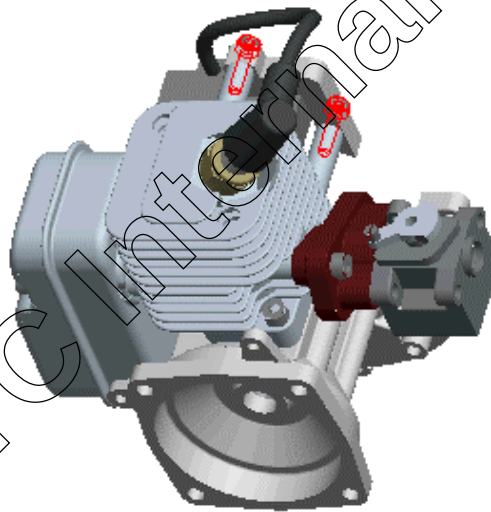


Figure 30: Assembling Two 4-18 BOLT.PRT Instances

3. Save the model.

Step 6. Restructure the components in DRILL.ASM.

In this initial step, we restructure components within the drill, so that critical moving and stationary components of the engine are placed together.

1. Create a new assembly named DRILL, using the default template.
2. Assemble ENGINE.ASM using a default constraint.

3. Click **Edit > Restructure**. Select SPARK_PLUG.PRT, select DRILL.ASM, and then click **Done**.
4. Repeat the procedure to restructure the following components into the drill: MUFFLER.PRT, BOLT_5-50.PRT (x2), CARBURETOR.ASM, COIL.PRT, and BOLT_4-18.PRT (x2), as shown in the following figure.

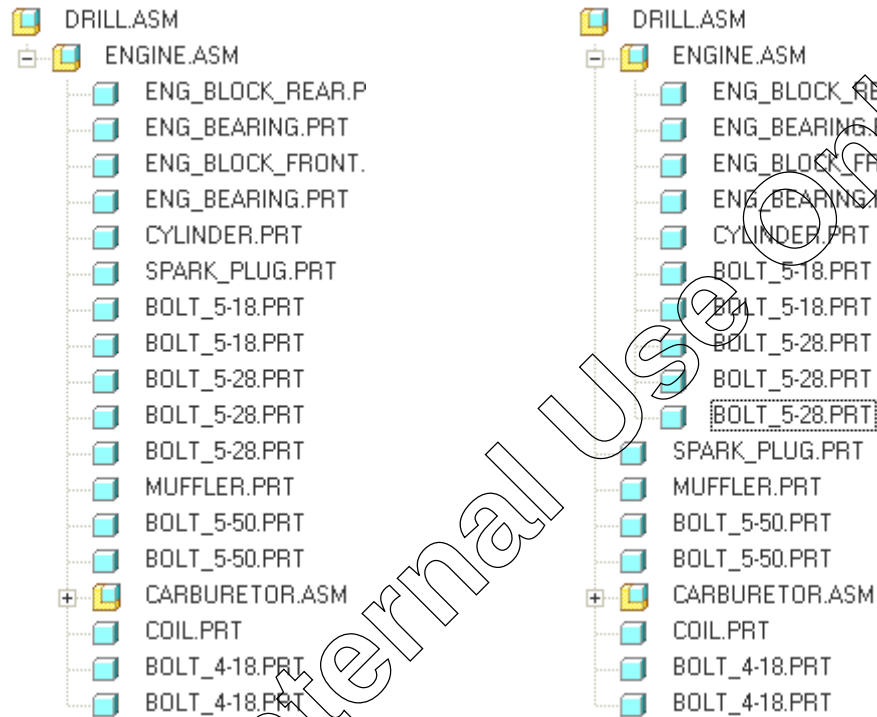


Figure 31: Restructuring DRILL.ASM

5. Save the model.
6. Close all windows.
7. Erase all models from memory.

This completes the exercise.

Summary

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

- Create assembly models.
- Assemble components using constraints.
- Assemble components using component interfaces.
- Assemble flexible components.
- Describe how assembling components affects parent/child relationships.

For PTC Internal Use Only

Project II

Introduction

Using the Pro/ENGINEER Wildfire, complete the following parts based upon the materials taught up to this point in class.

Objectives

After completing this module, you will be able to:

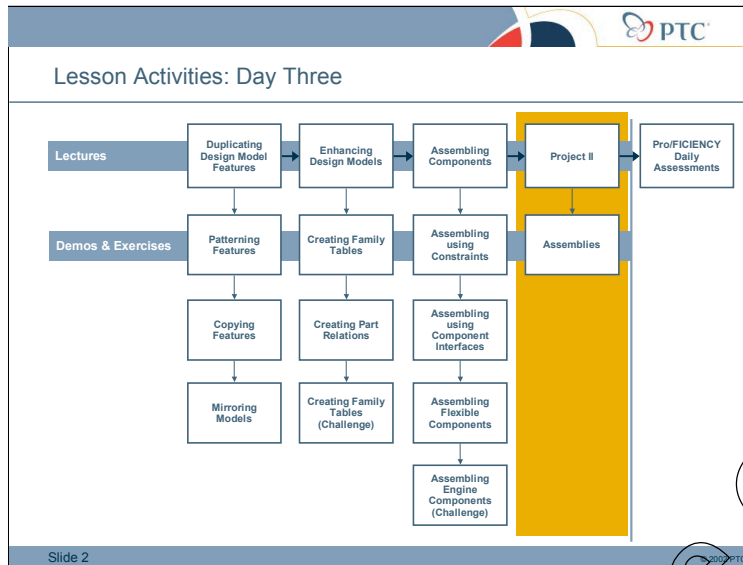
- Create the Impeller.
- Complete the Frame.
- Complete the Impeller Housing.
- Continue Creating the Engine Block.
- Create the Piston assembly.
- Create the Engine assembly.
- Create the Blower assembly.
- Create the AC-40 assembly.

Fast Track to Pro/ENGINEER Wildfire

Module 12: Project II

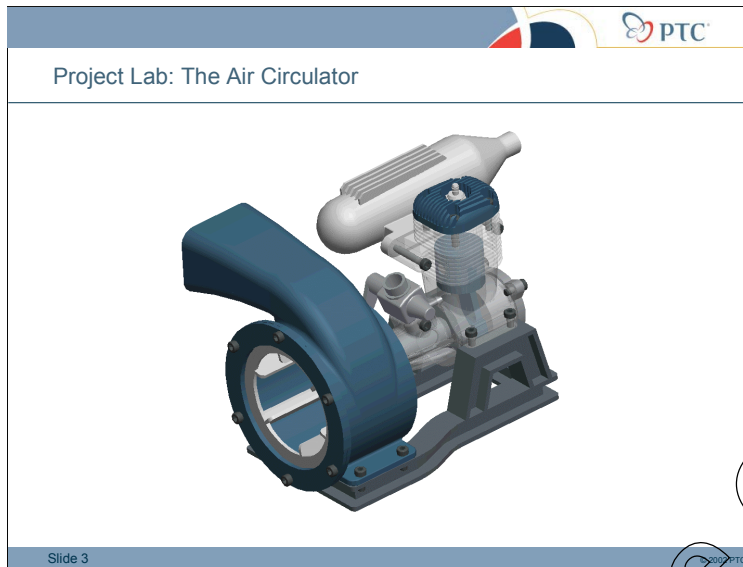


For PTC Internal Use Only



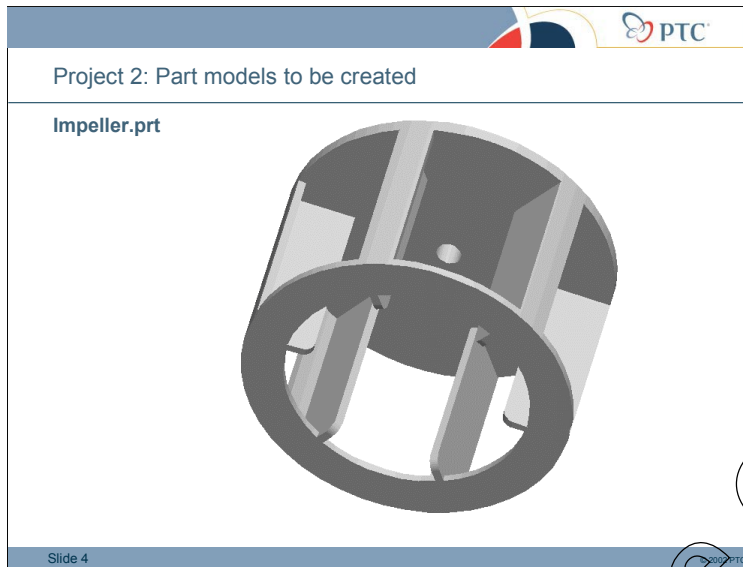
Duration

Total: 1 hr 30 mins

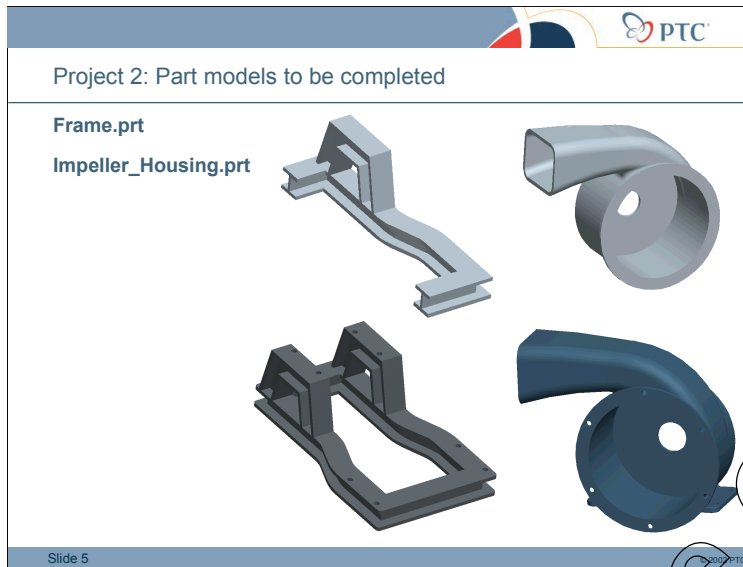


Instructor:

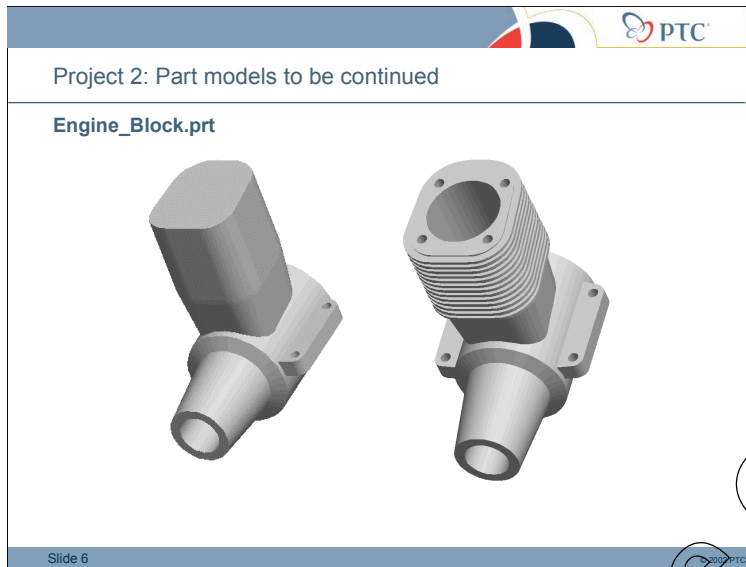
Use the following slides to brief the students on what will be accomplished during this phase of the project.



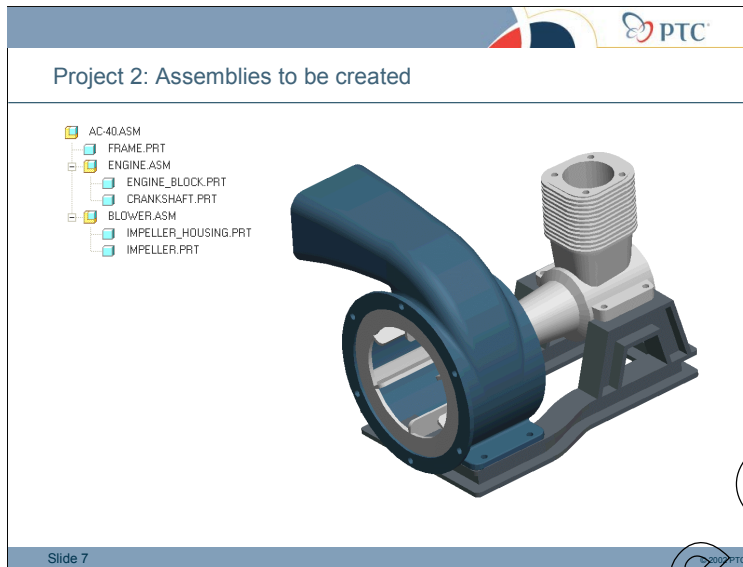
There are no notes for this slide.



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There are no notes for this slide.

Module 12 Lab Exercises

Scenario

ACME Inc. is a multinational conglomerate that develops and markets several consumer, industrial, and defense goods. The Light Industrial Division of ACME Inc creates a number of products including industrial fans, heating, air conditioning, pumps, and lighting. You work for the Light Industrial Division of ACME Inc., which has just started to use Pro/ENGINEER Wildfire for its product designs.

Upon returning from Pro/ENGINEER Wildfire training, you are assigned to create the AC-40 Air Circulator. As the name implies, the AC-40 is an air circulation device. Its primary purpose is to enable fast effective clean up of flooded homes and businesses that have no power due to the flood. Another variation of the AC-40 is currently being designed to use electric power. A further variation that includes an air compressor and condenser that will provide back-up air conditioning to high-rise buildings is also being considered.

In the second phase of the project, you continue building the parts that will be assembled into the AC-40. You will also complete parts and begin building assemblies based upon the tasks that you have learned thus far in the training course.

Objectives

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

- Create the Impeller.
- Complete the Frame.
- Complete the Impeller Housing.
- Continue Creating the Engine Block.
- Create the Piston assembly.
- Create the Engine assembly.
- Create the Blower assembly.
- Create the AC-40 assembly.

Objective 1: Creating the Impeller part

Step 1. Create IMPELLER.PRT.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from session. Otherwise, start Pro/ENGINEER Wildfire.
2. Using the folder navigator, browse to the following location:
c:\users\student\fast_track_330\Project. Make this folder the working directory.
3. Create a new part named IMPELLER.PRT.

4. Create a protrusion as shown in the following figure. Sketch on FRONT.

Design Intent:

The feature is to be created from the FRONT plane forward.

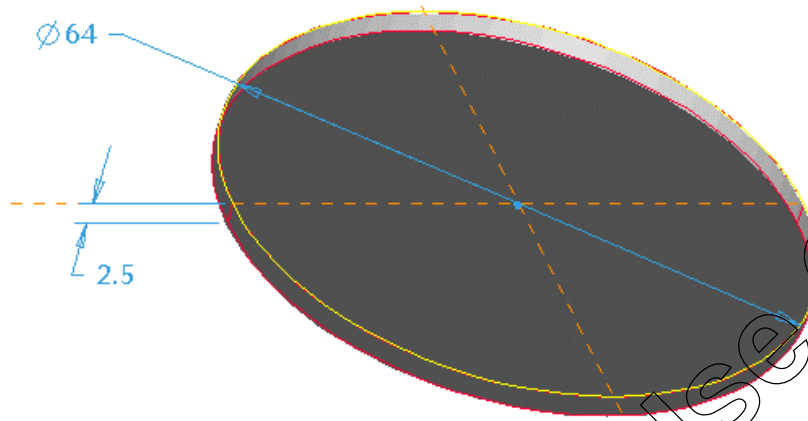


Figure 1: Creating a Protrusion.

Step 2. Create a datum plane.

1. Create a datum plane offset from FRONT, as shown in the following figure. Name the plane WIDTH.

Design Intent:

This plane will be used with several features to control the overall width of the part.

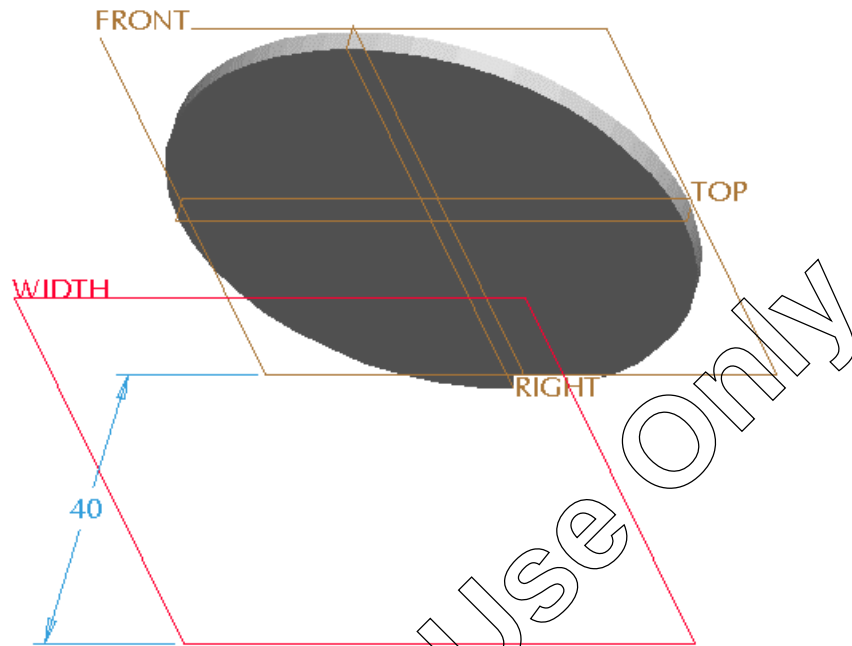


Figure 2: Creating a Datum Plane

Step 3. Create a protrusion for an impeller blade, suitable for a rotational pattern.

1. Start the Extrude tool. Create a datum plane through the center axis and at an angle to TOP, as shown in the following figure.

Design Intent:

This plane will be used as the reference plane for a protrusion which will be rotationally patterned.

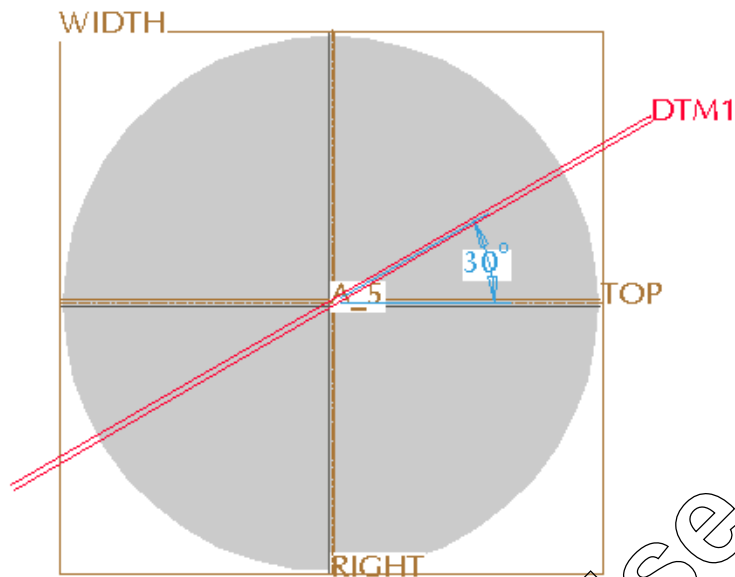


Figure 3: Creating a Datum Plane

2. Select the front surface of the previous protrusion as a sketching plane, and select the previous angled datum plane as the horizontal reference. Sketch a line and a center-end arc as shown in the following figure.

Design Intent:

The sketch should only reference the center axis, the angled datum, and the cylindrical surface.

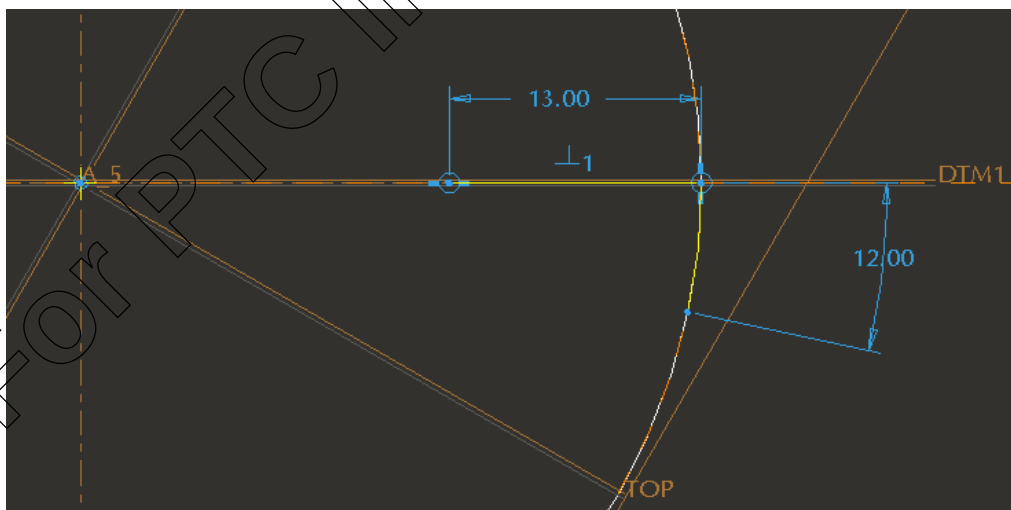


Figure 4: Creating a Sketch

3. Specify the Thin option. Set the depth up to the WIDTH plane.

Design Intent:

Thicken the sketch 1.5 towards the inside of the 'L' shape . The depth should update if the WIDTH plane is modified.

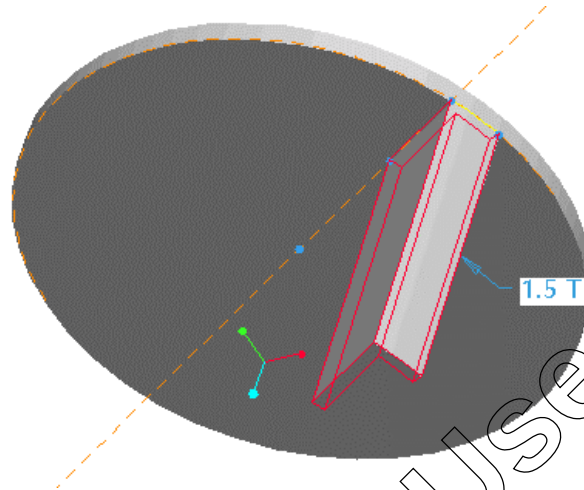


Figure 5: Creating a Protrusion

4. Rename the group to BLADE.

Step 4. Duplicate the feature using a pattern.

1. Pattern the BLADE group as shown in the following figure.

Design Intent:

The blades need to be identical, but the quantity may change.

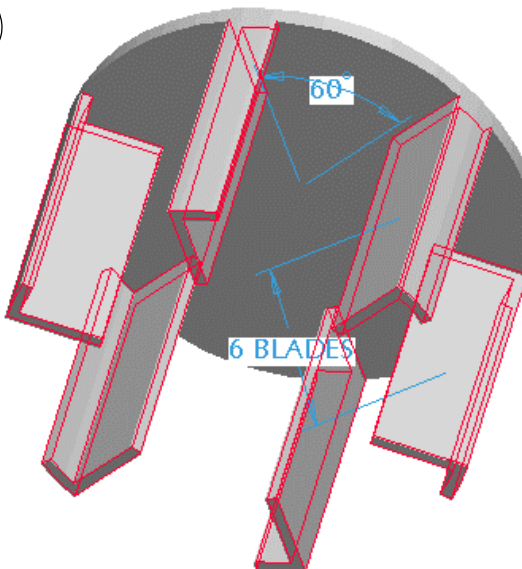


Figure 6: Creating a Pattern

2. Edit the pattern quantity to 4.

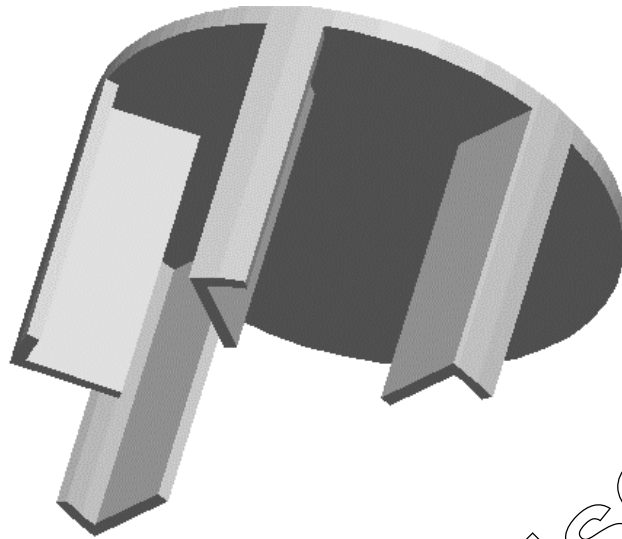


Figure 7: Modified Pattern

3. Edit the pattern and select the 60 degree dimension. Edit its properties to change its name to ANG_SPACING.
4. Select the quantity 4 dimension. Edit its properties to change its name to QTY_BLADES.
5. Enter the following as a relation:
/* This relation maintains equal blade spacing
ANG_SPACING = 360 / QTY_BLADES
6. Regenerate the model. The blades should be equally spaced.
7. Test the relation by modifying the quantity to 5. Modify the quantity back to 6.

Step 5. Create and pattern a round to smooth out the sharp corner of the blades.

1. Create a round as shown in the following figure.

Design Intent:

The round must be placed on the original blade to reference pattern.

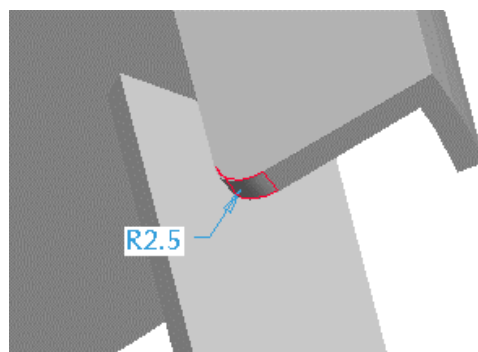


Figure 8: Creating a Round

2. Pattern the round using the Reference option.

Design Intent:

The number of rounds must update with the changing qty of blades.

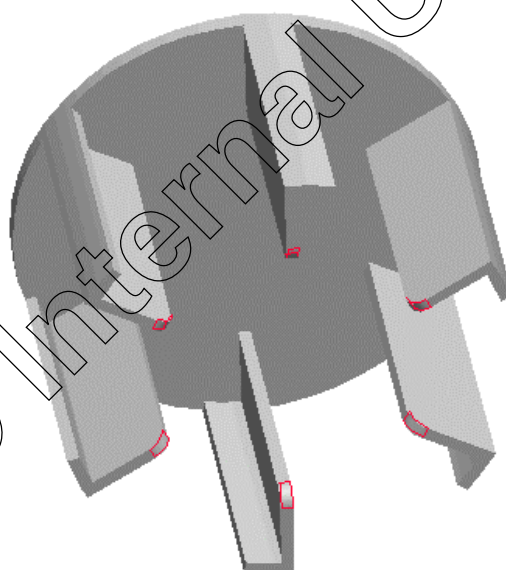


Figure 9: Round Reference Pattern

Step 6. Create a protrusion to support the tips of the blades.

1. Create a protrusion as shown in the following figure. Sketch 2 concentric circles on the WIDTH plane.

Design Intent:

The depth of this feature should extend from the WIDTH plane inward over the model.

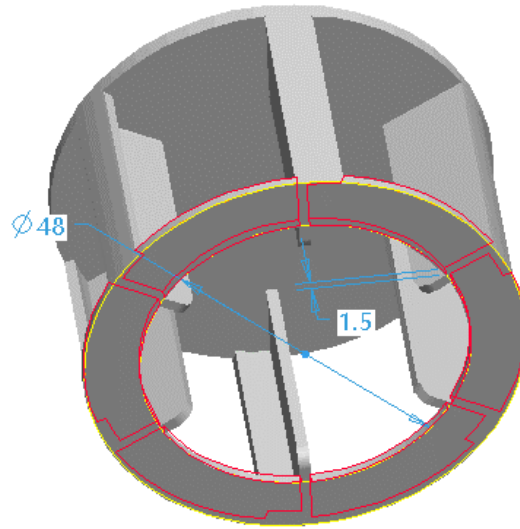


Figure 10: Creating a Protrusion

Step 7. Create a protrusion to form the attachment collar for the CRANKSHAFT.PRT.

1. Start the Extrude tool, and select the surface shown below as a sketching plane.



Figure 11: Sketching Plane

2. Create a circular protrusion with the dimensioning scheme shown in the following figure.

Design Intent:

The depth should be dimensioned from the inside surface of the model.

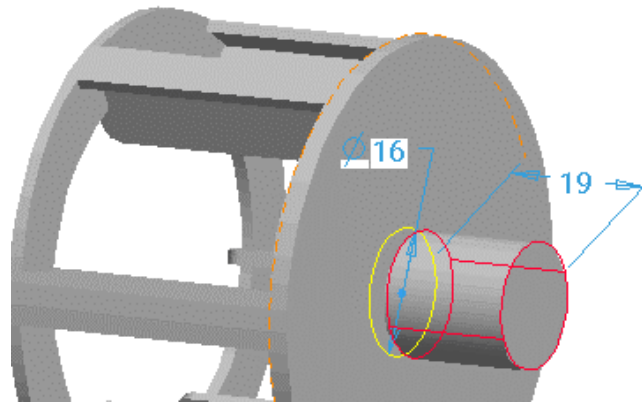


Figure 12: Creating a Protrusion

Step 8. Create a cut used to interlock the IMPELLER.PRT with the CRANKSHAFT.PRT.

1. Create a cut with the dimensioning scheme shown in the following 2 figures.

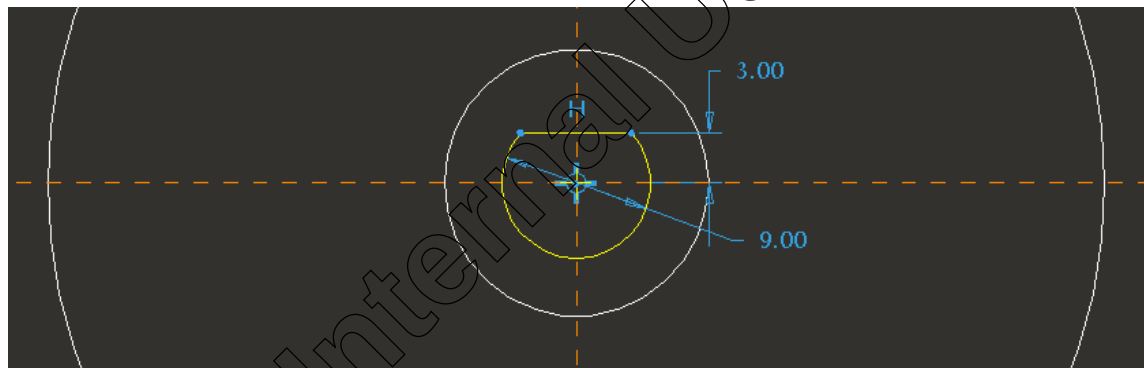


Figure 13: Creating a Sketch

Design Intent:

The depth should be dimensioned as shown in the following figure.

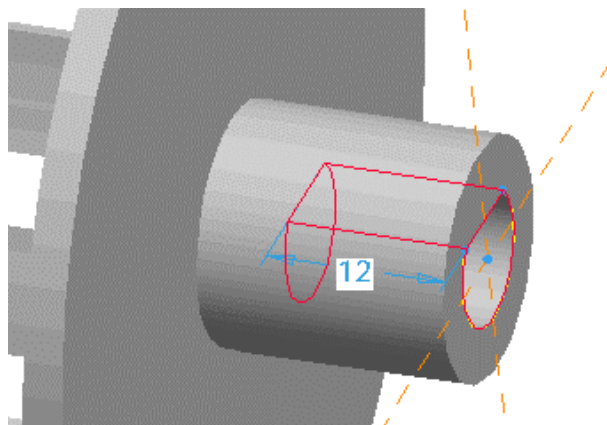


Figure 14: Creating a Cut

Step 9. Create a hole for the CRANKSHAFT.PRT to slide through.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial and should extend through the next surface.

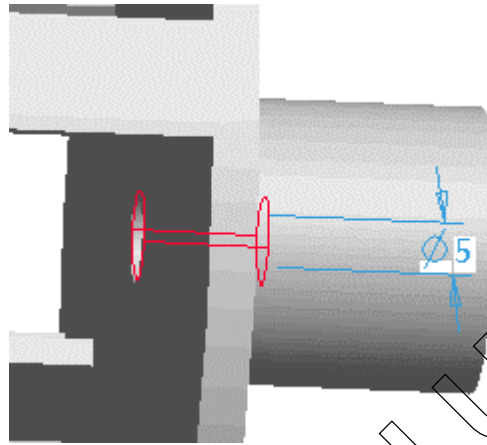


Figure 15: Creating a Hole

2. This model is now complete. Save the model.
3. Close the window.

This completes the exercise.

Objective 2: Completing the Frame

Step 1. Mirror the FRAME.PRT model.

1. Open FRAME.PRT.

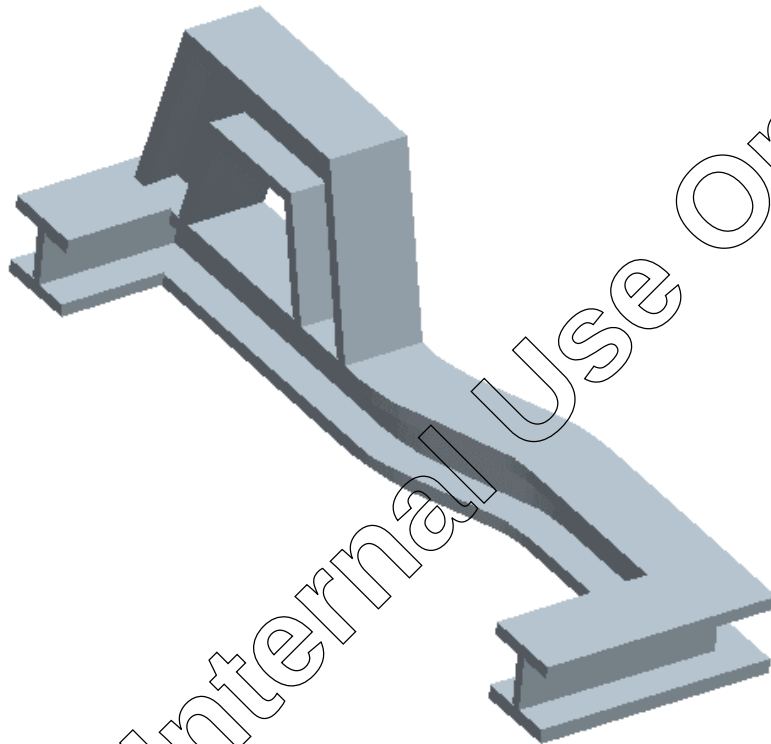


Figure 16: FRAME.PRT Model

2. Select the part node from the model tree and mirror the part around the RIGHT plane.

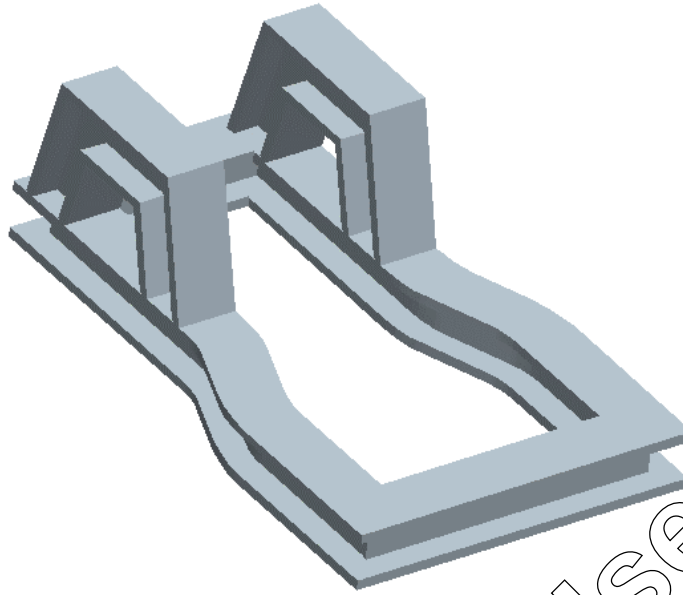


Figure 17: Mirror Created

Step 2. Create a series of mounting holes in the FRAME.PRT.

1. Create two holes with the dimensioning schemes shown in the following figures. Create both holes on the 'original' side of the model.

Design Intent:

The first hole feature should be dimensioned from the edge for the 6 dimension. The hole depths should extend only through the next surface.

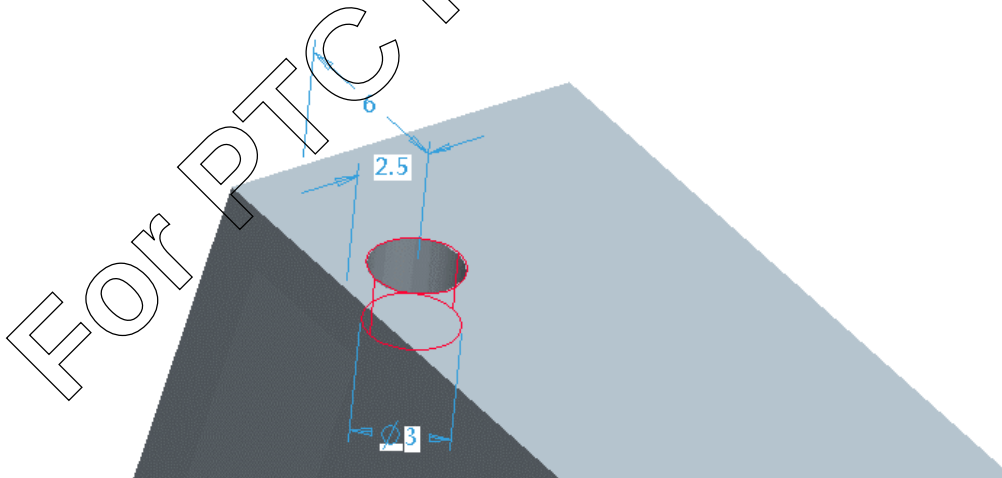


Figure 18: Creating a Hole in the Upper Right Corner of the FRAME.PRT

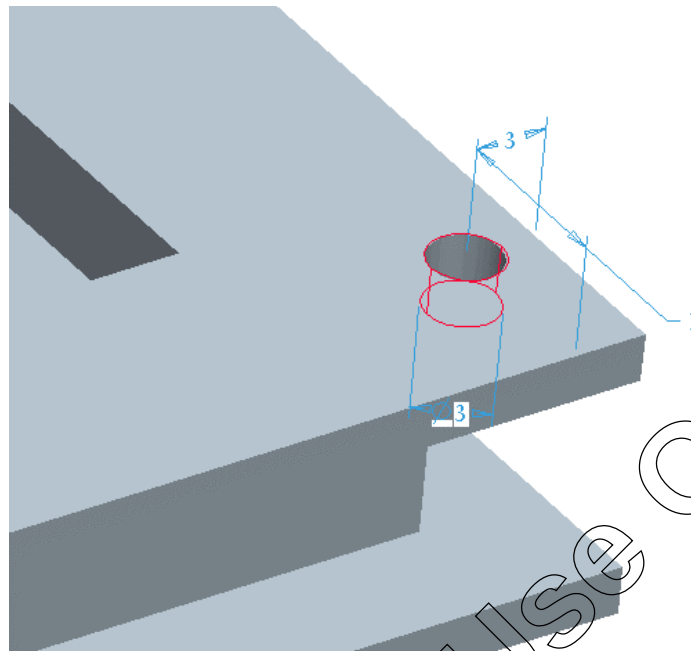


Figure 19: Creating a Hole in the Lower Right Corner of the FRAME.PRT

2. Copy each hole using the dimensioning schemes shown in the following figures.

Design Intent:

The copied holes should be translated normal to the FRONT plane. Also the diameter of the hole should be dependent on the original hole's diameter.

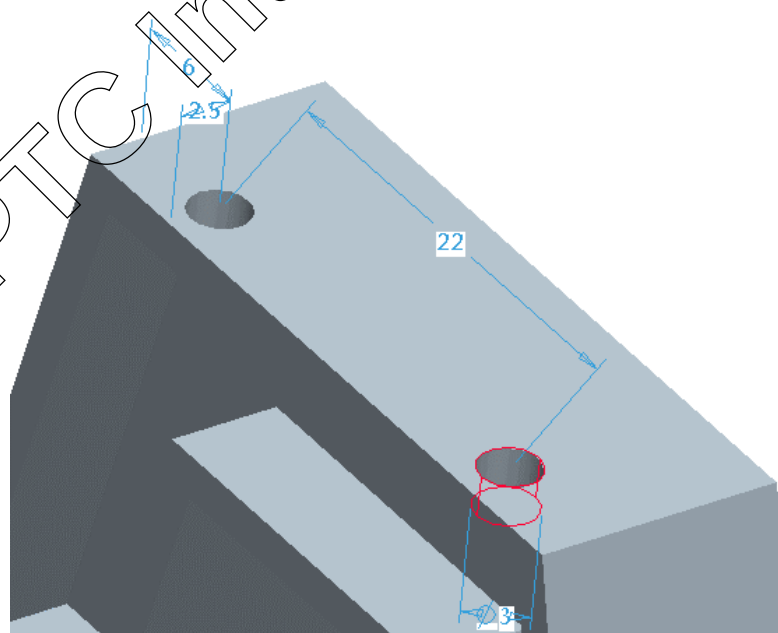


Figure 20: Copying the First Hole

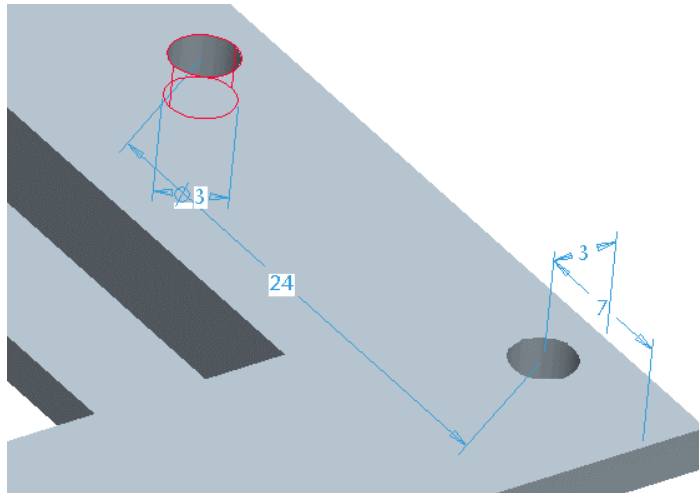


Figure 21: Copying the Second Hole

3. Since there is only one hole in each of the copied groups, ungroup them.

Step 3. Create a Round to smooth out sharp corners on the FRAME.PRT.

1. Create a single round feature rounding the FIVE edges shown in the following figure.

Design Intent:

The round on the front and rear edges should have the same radius.

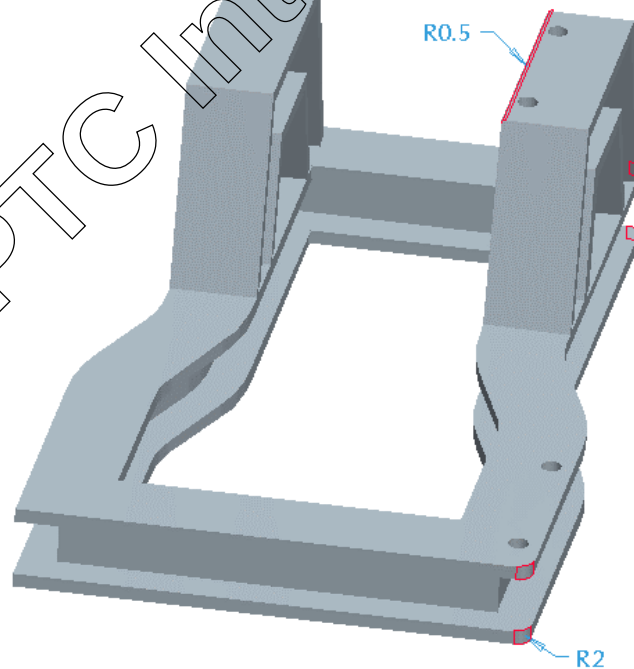


Figure 22: Creating a Round

Step 4. Mirror the holes and round onto the left side of the model.

1. Reorder the Mirror in the model tree to be the last feature.

Design Intent:

This part should always remain symmetric. Any features created before the Mirror feature will be mirrored.

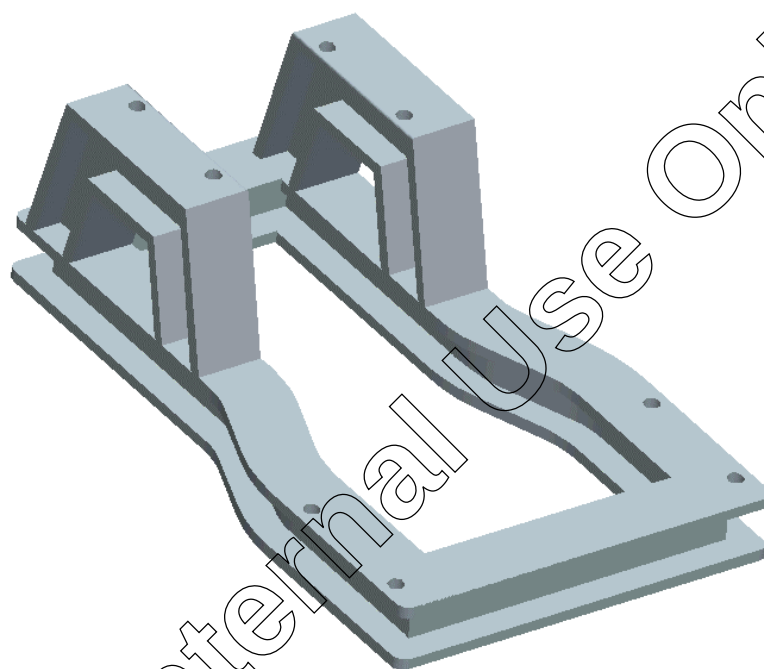


Figure 23: Mirror Feature Reordered

Step 5. Write a relation to keep the second sweep centered on the first.

1. Display the dimensions of the trajectory of the first sweep. Select the 22 dimension. Edit its properties to change its name to FRAME_WIDTH.

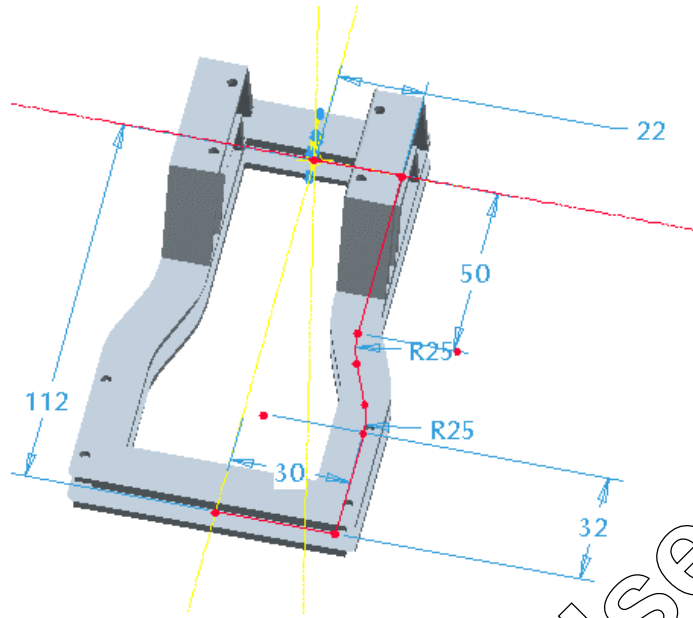


Figure 24: First Sweep Dimensions

2. Display the dimensions of the offset datum plane in the ENG_SWEEP group. Edit the value of the 22 dimension and enter **FRAME_WIDTH**. Select **Yes** to create the relation.

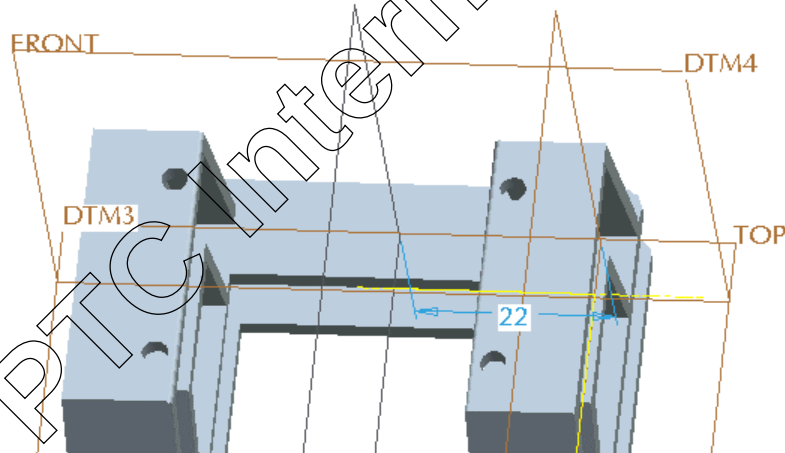


Figure 25: Offset plane dimensions

Design Intent:

The second sweep must automatically update its width if the width of the first sweep is changed.

3. This model is now complete. Save the model.
4. Close the window.

This completes the exercise.

Objective 3: Completing the Impeller Housing

Step 1. Create a pattern of mounting holes for the FLANGE.PRT.

1. Open IMPELLER_HOUSING.PRT.

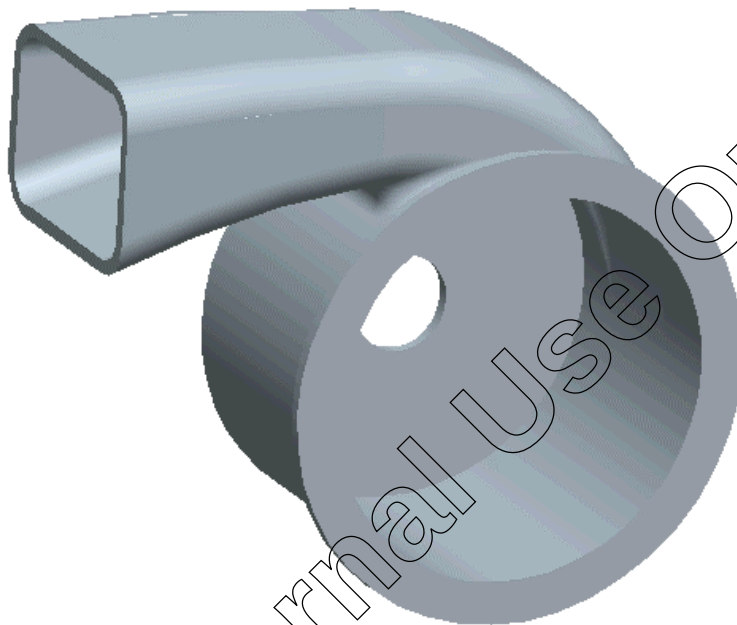


Figure 26: IMPELLER_HOUSING.PRT Model

2. Create a hole as shown in the following figure.

Design Intent:

The hole should be dimensioned radially from the center axis and the TOP plane. Its depth should be through the next surface.

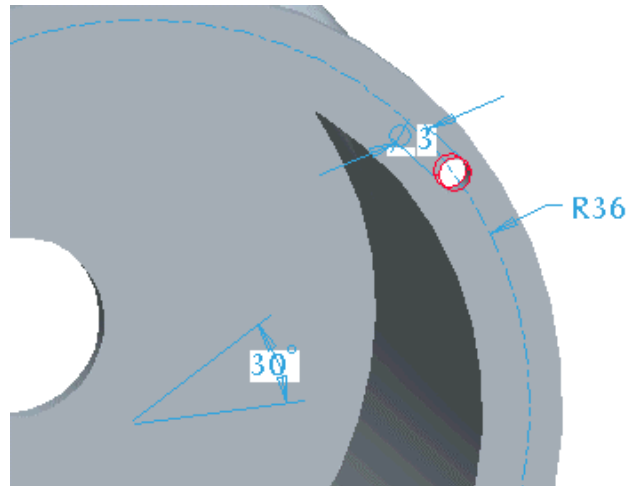


Figure 27: Creating a Hole

3. Pattern the hole as shown in the following figure.

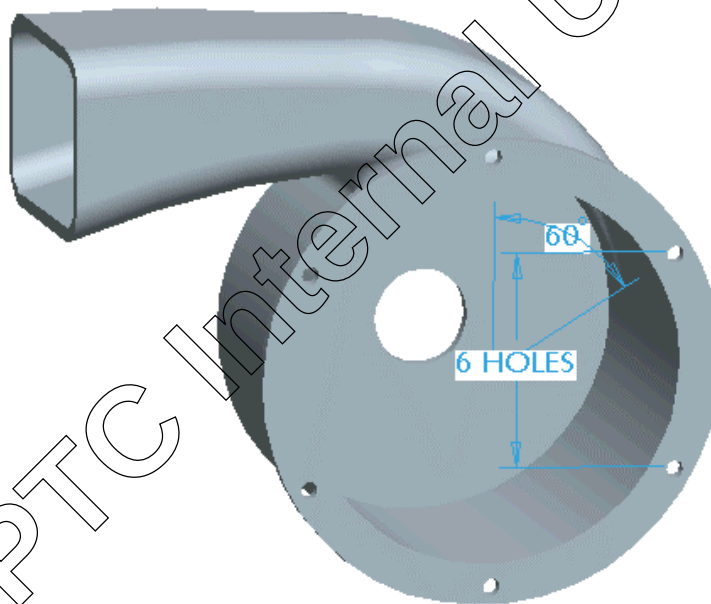


Figure 28: Hole Pattern

Step 2. Create a protrusion to be used as a mounting flange.

1. Create a protrusion on the FRONT plane. Flip the viewing direction, and sketch as shown in the following figure.

Design Intent:

The feature should be on the right side when oriented to the default 3D view (the TOP plane should be oriented to face Top). The cylindrical surface of the first feature should be referenced in the sketch.

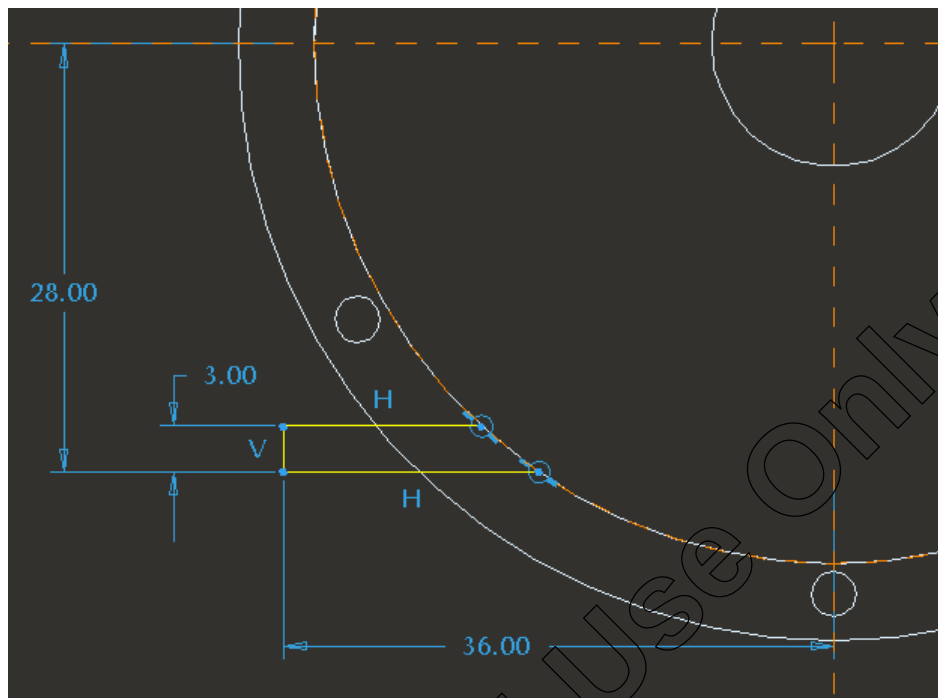


Figure 29: Creating a Sketch

2. Specify a depth as shown in the following figure.

Design Intent:

The feature should be symmetric about the FRONT plane.

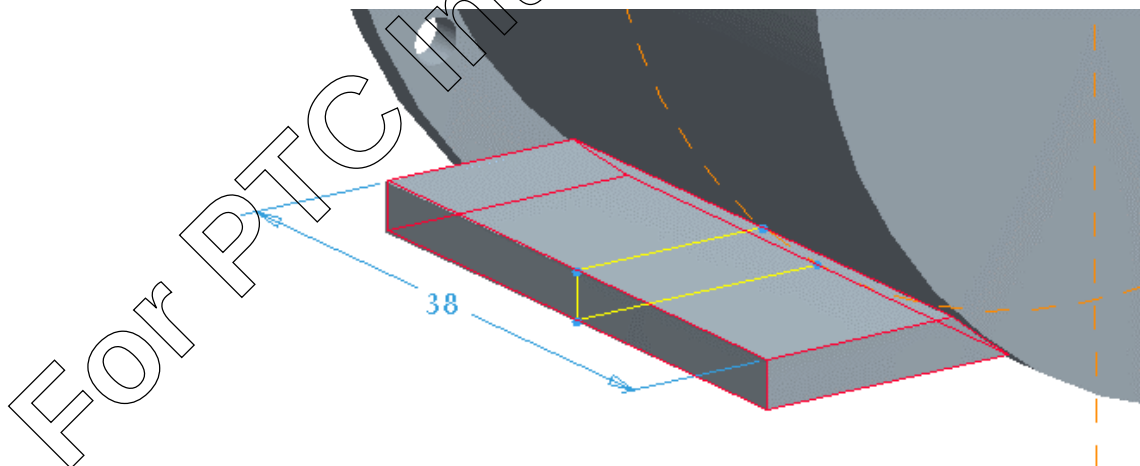


Figure 30: Creating a Protrusion

Step 3. Create mounting holes.

1. Create a hole as shown in the following figure.

Design Intent:

The hole depth should extend through the next surface.

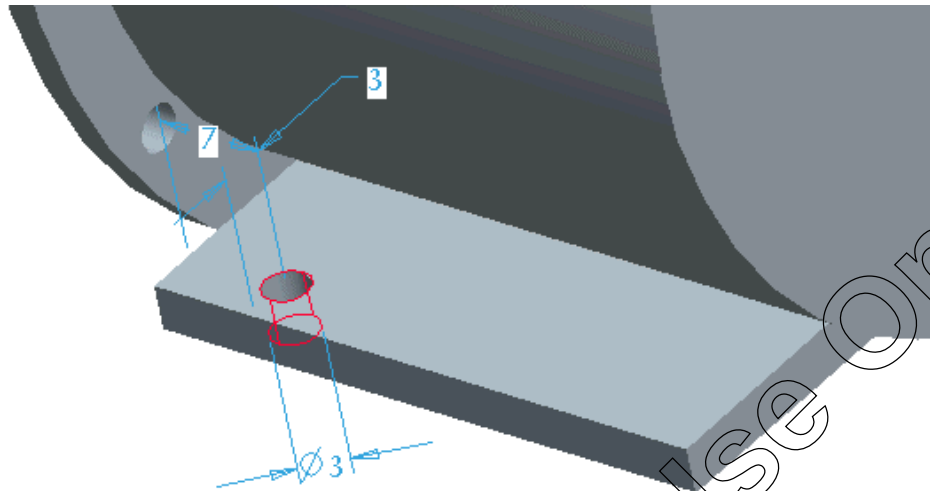


Figure 31: Creating a Hole

2. Mirror the previous hole about the FRONT plane, as shown in the following figure.

Design Intent:

The holes should be symmetric about the FRONT plane.

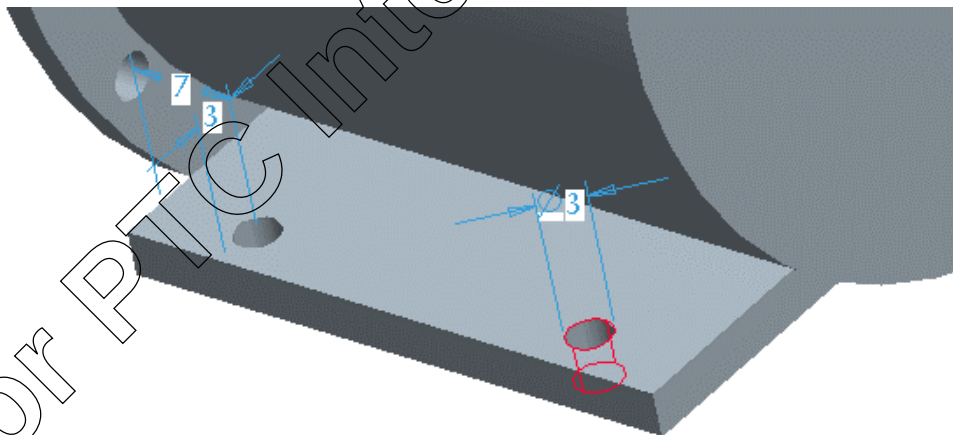


Figure 32: Creating a Hole

3. Since there is only one hole in the copied group, ungroup it.

Step 4. Create rounds to add strength and smooth out sharp corners.

1. Create a round as shown in the following figure.

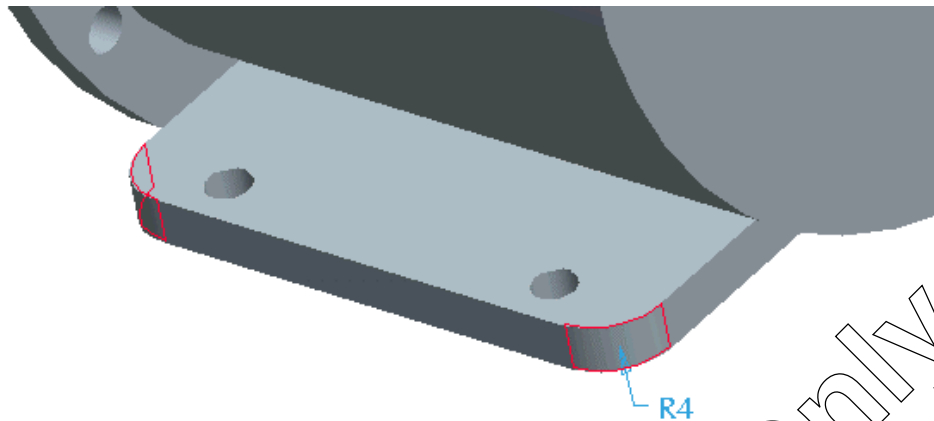


Figure 33: Creating a Round

Design Intent:

The round feature should have one radius dimension for both edges.

2. Create a round as shown in the following figure.

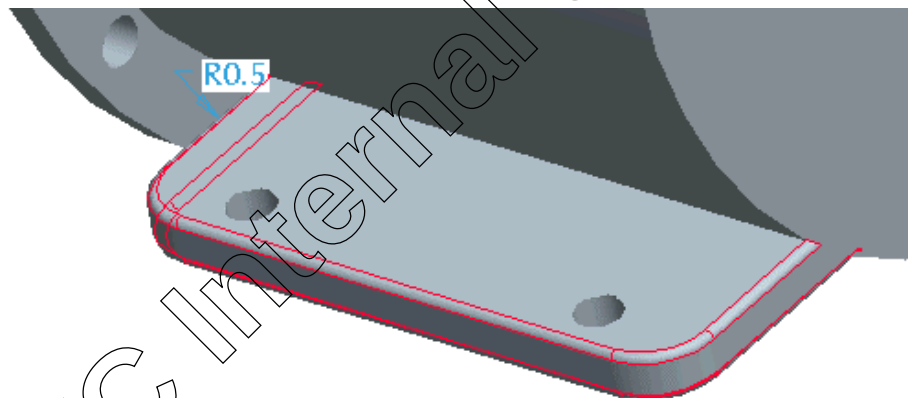


Figure 34: Creating a Round

Design Intent:

This round sets up a tangent chain for the next round.

3. Create a round as shown in the following figure.

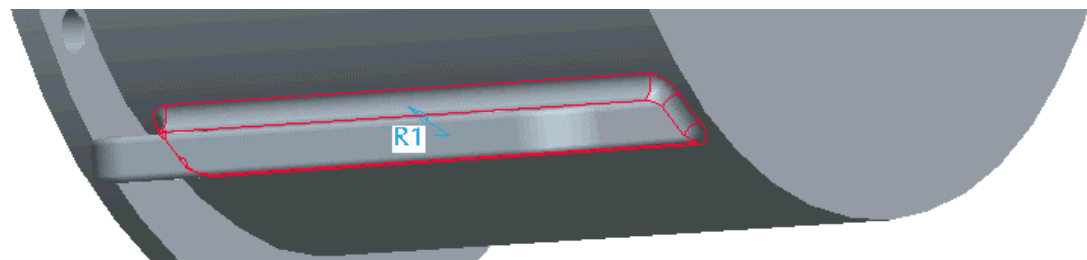


Figure 35: Creating a Round

Design Intent:

This round adds strength to the mounting flange.

4. Create a group containing the protrusion, the two holes, and the 3 rounds called FLANGE1.

Step 5. Create Flange2.

1. Mirror Flange1 to the other side of the model, as shown in the following figure.

Design Intent:

The flange should be mirrored about the RIGHT plane.

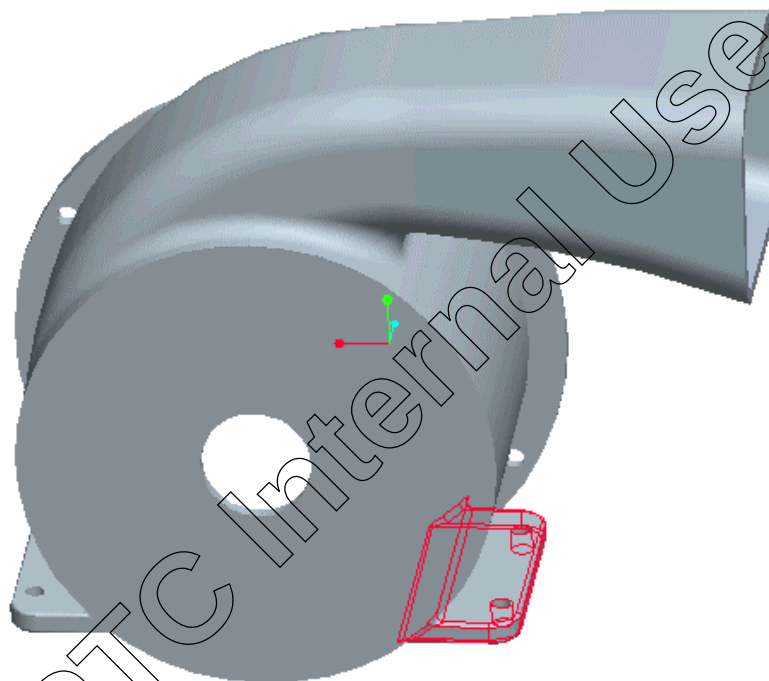


Figure 36: Flange1 Mirrored

2. Rename the group to FLANGE2.
3. This model is now complete. Save the model.
4. Close the window.

This completes the exercise.

Objective 4: Continuing the Engine_Block

Step 1. Mirror the engine mount.

1. Open ENGINE_BLOCK.PRT.

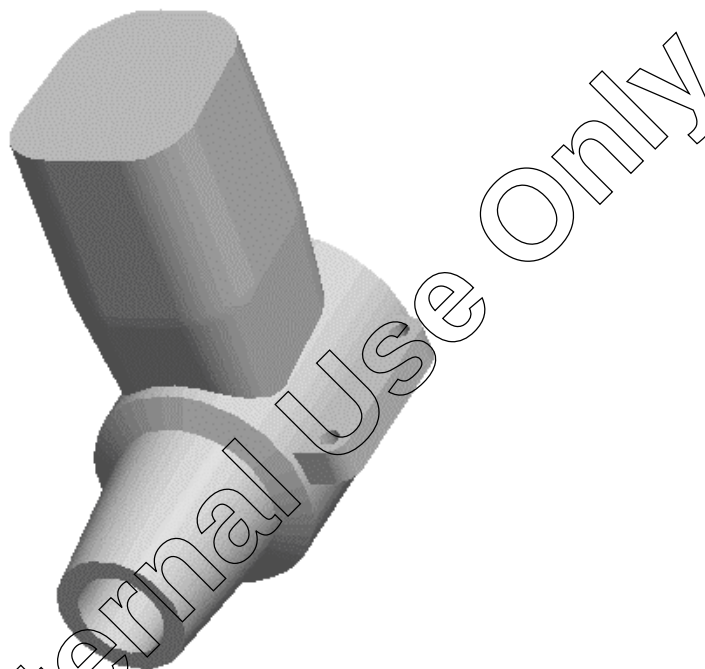


Figure 37: Engine Block Model

2. In the model tree, drag the Insert indicator to be just after Mount1.
3. Mirror Mount1 to the other side of the model. Cancel Insert Mode. Rename the copied group to Mount2. Refer to the following figure.

Design Intent:

Both flanges need to be ordered before the holes to maintain a 'hollowed out' crankcase.

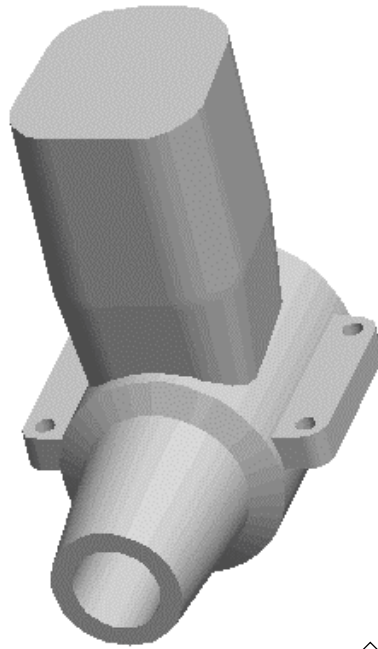


Figure 38: Flange Mirrored

Step 2. Create a cooling fin.

1. Start the Extrude tool. Create a datum plane offset downward from the top model surface, as shown in the following figure.

Design Intent:

This plane will be used as a sketching plane for the next feature.

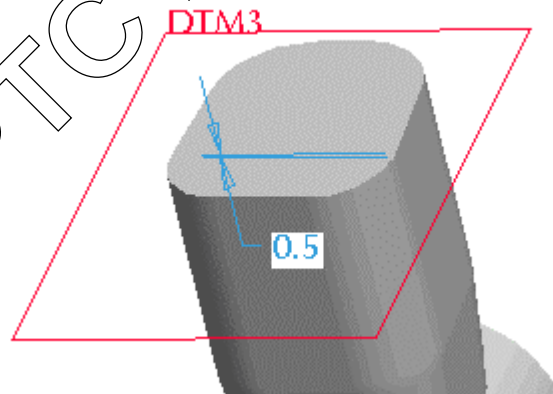


Figure 39: Creating a Datum Plane

2. Select the previous offset plane as a sketching plane, and sketch as shown in the following figure. (Hint: Use the Offset Edge tool with the Loop option.)

Design Intent:

The sketch for the cooling fin should be evenly offset from the cylinder.

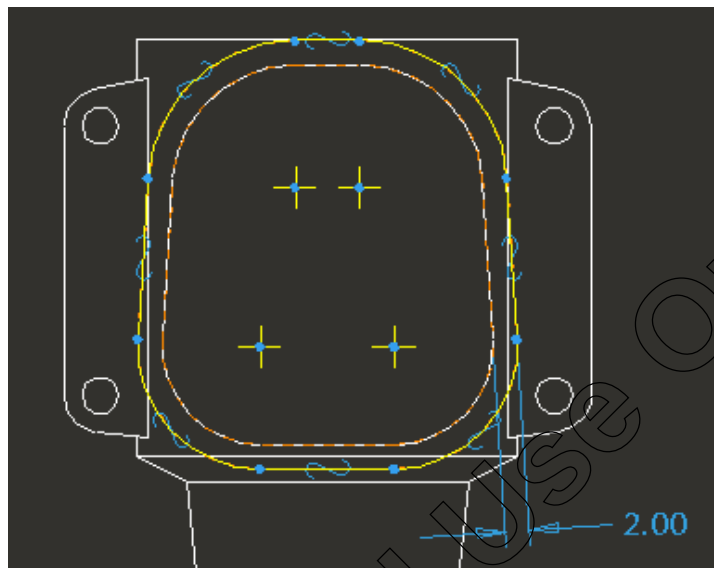


Figure 40: Creating a Sketch

3. Set the depth for the protrusion as shown in the following figure.

Design Intent:

The depth direction should be downward.

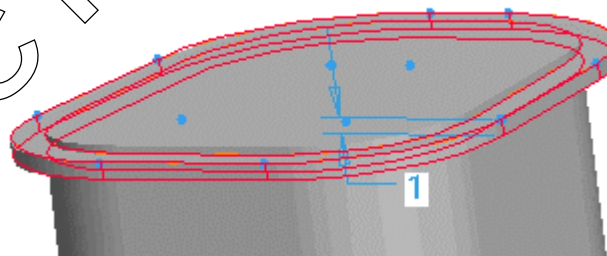


Figure 41: Creating a Protrusion

4. Rename the group to FIN.

Step 3. Create a pattern of cooling fins.

1. Pattern the cooling fin as shown in the following figure.

Design Intent:

If the thickness of the first cooling fin changes, all must update.

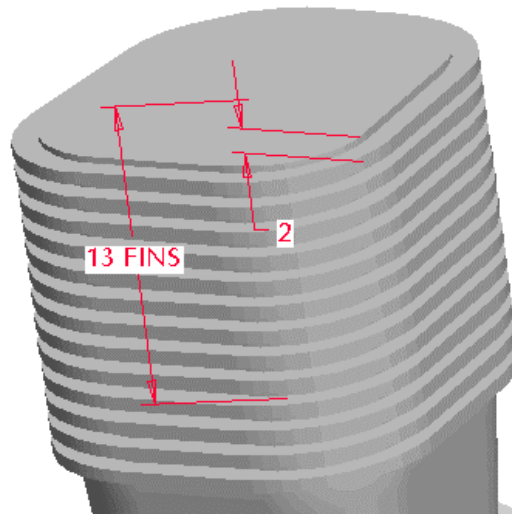


Figure 42: Patterning a Protrusion

Step 4. Create a pattern of bolting holes for the ENGINE_HEAD.PRT.

1. Create a hole as shown in the following figure

Design Intent:

The hole should be placed radially (using the diameter option) to the CYL axis and the RIGHT plane.

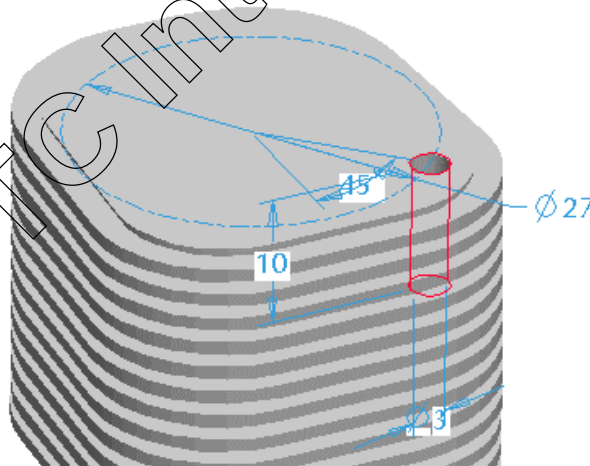


Figure 43: Creating a Hole

2. Create a pattern as shown in the following figure.

Design Intent:

The holes should be 90 degrees apart.

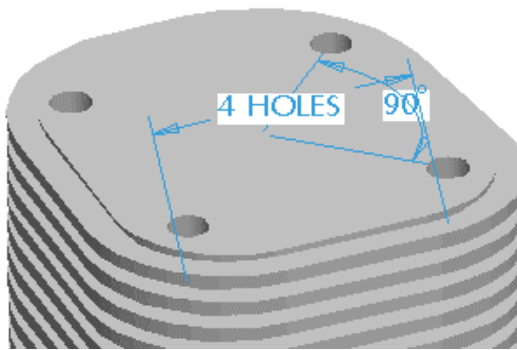


Figure 44: Patterning a Hole

Step 5. Create the cylinder bore.

1. Create a hole as shown in the following figure. Rename the hole to BORE.

Design Intent:

The hole depth should extend to the next surface.

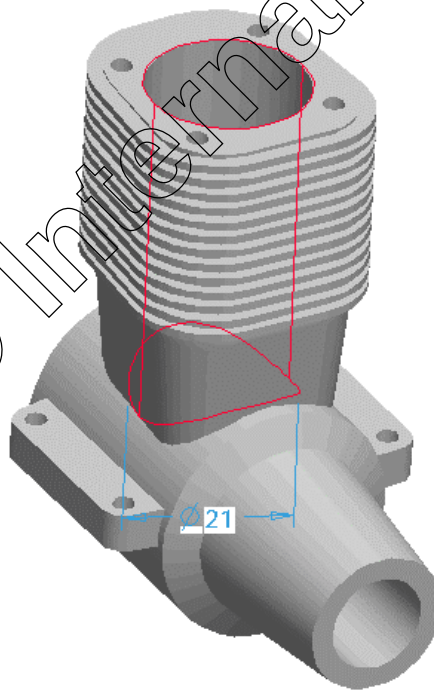


Figure 45: Creating a Hole

2. This model will be completed in a later exercise. Save the model.
3. Close the window.

This completes the exercise.

Objective 5: Creating Assemblies

Step 1. Create PISTON.ASM.

Note:

When components are assembled together, they will be assigned color appearances from the **project_palette.dmt** file in the PROJECT directory. You can load the *.dmt file or create colors in the appearance editor using these RGB color codes.

Black = 96, 102, 102

Blue_Dark = 80, 124, 153

Grey_Med = 160, 163, 166

Grey_Dark = 128, 140, 145

Silver = 255, 255, 255

Silver_Transparent = 255, 255, 255 (Transp 70%)

1. Create a new assembly named PISTON.ASM.
2. Assemble PISTON.PRT using a default constraint. Apply the Blue_Dark appearance.

Design Intent:

The first component in an assembly should be assembled by default.



Figure 46: Assembling the PISTON.PRT

3. Assemble the PISTON_PIN.PRT. Insert the cylindrical surfaces of the PISTON_PIN and the hole in the PISTON.PRT. Align the FRONT planes to center the component.

Design Intent:

The PISTON_PIN.PRT must remain centered in the PISTON.PRT.

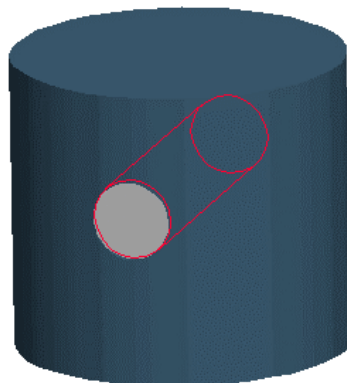


Figure 47: Assembling the PISTON_PIN.PRT

4. Save the model.

Step 2. Create ENGINE.ASM.

1. Create a new assembly named ENGINE.ASM.
2. Assemble ENGINE_BLOCK.PRT using a default constraint. Apply the Silver appearance.

Design Intent:

The first component in an assembly should be assembled by default.

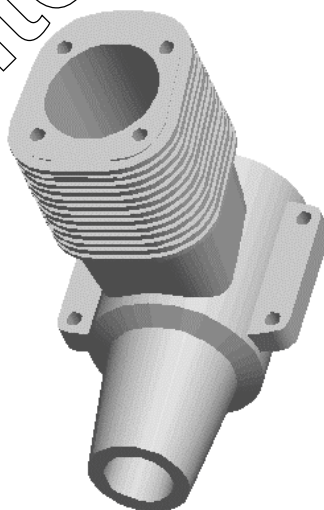


Figure 48: Assembling the ENGINE_BLOCK.PRT

3. Assemble CRANKSHAFT.PRT. Mate the two surfaces shown below.

Design Intent:

The rotation position of the CRANKSHAFT.PRT is not important at this stage.

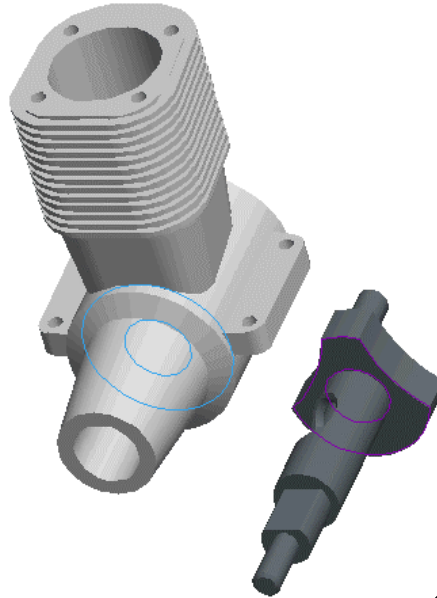


Figure 49: Selecting Mating Surfaces

4. Align the two surfaces shown below. Apply the Grey_Dark appearance to the CRANKSHAFT.PRT.

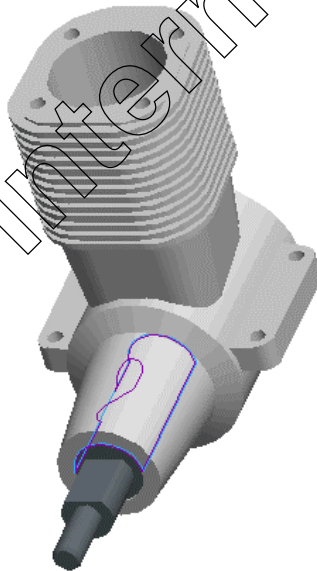


Figure 50: Selecting Insert Surfaces

5. Save the model.

Step 3. Create BLOWER.ASM.

1. Create a new assembly named BLOWER.ASM.

2. Assemble IMPELLER_HOUSING.PRT using a default constraint. Apply the Blue_Dark appearance.

Design Intent:

The first component in an assembly should be assembled by default.



Figure 61: Assembling the IMPELLER_HOUSING.PRT

3. Assemble the IMPELLER.PRT, using an Insert and a Mate Offset constraint. Apply the Silver appearance.

Design Intent:

For clearance purposes, the IMPELLER.PRT must be offset from the rear of the IMPELLER_HOUSING.PRT by 1.0.

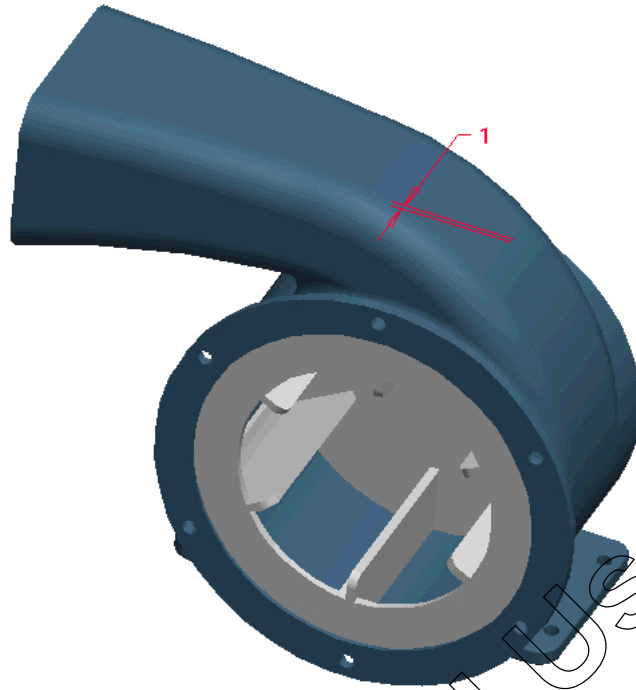


Figure 52: Assembling the IMPELLER.PRT

4. Save the model.

Step 4. Create the AC-40 assembly.

1. Create a new assembly named AC-40.ASM.
2. Assemble FRAME.PRT using a default constraint. Apply the Grey_Dark appearance.

Design Intent:

The first component in an assembly should be assembled by default.

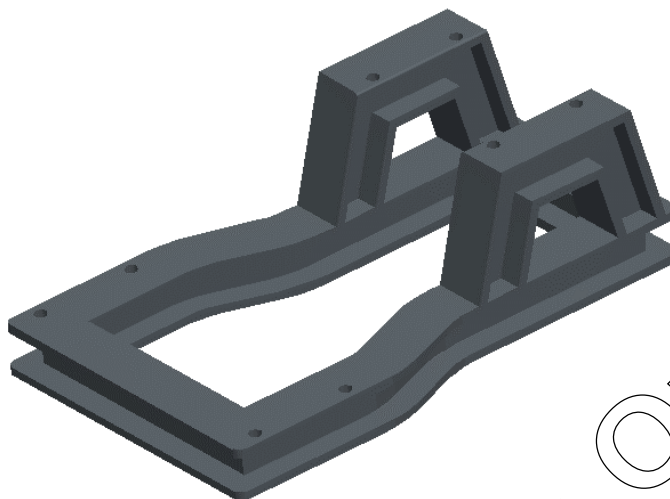


Figure 53: Assembling the FRAME.PRT

3. Assemble ENGINE.ASM. Use a mate constraint and two insert constraints between bolt holes.

Design Intent:

The ENGINE.ASM position must update to changes in the FRAME.PRT bolt holes.

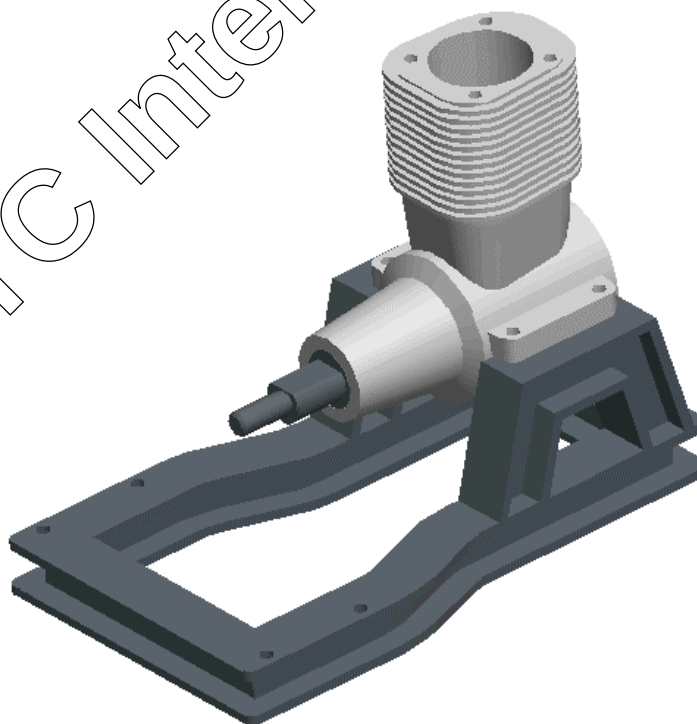


Figure 54: Assembling the Engine

4. Assemble BLOWER.ASM. Use a mate constraint and two insert constraints between bolt holes.

Design Intent:

The BLOWER.ASM position must update to changes in the FRAME.PRT bolt holes. Also, the IMPELLER.PRT will be interlocked with the CRANKSHAFT.PRT in the next project, its rotation is not important at this stage.

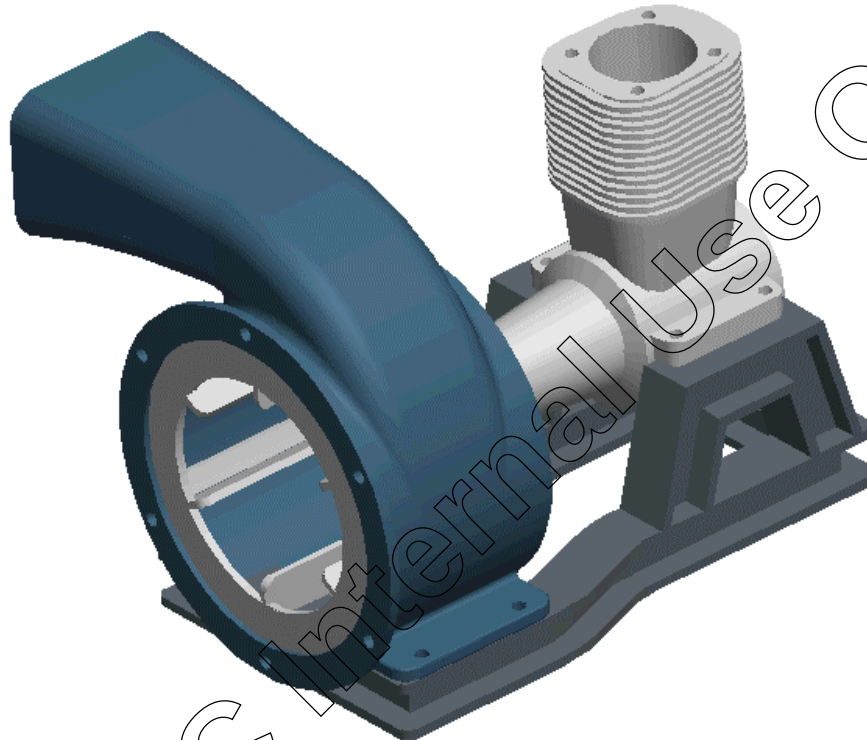


Figure 55: Assembling the BLOWER.ASM

5. This model will be completed later in the project. Save the model.
6. Close the window.

This completes the exercise.

Summary

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

- Create the Impeller.
- Complete the Frame.
- Complete the Impeller Housing.
- Continue Creating the Engine Block.
- Create the Piston assembly.
- Create the Engine assembly.
- Create the Blower assembly.
- Create the AC-40 assembly.

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For PTC Internal Use Only

Creating Mechanism Connections

Introduction

Many product designs include both static and dynamic components. Pro/ENGINEER Wildfire enables you to assemble dynamic components using several connection types. You can also add dynamic entities such as motors, which can be used to analyze motion in your product designs.

Objectives

After completing this module, you will be able to:

- Assemble the components using connections such as joints and gears.
- Create mechanism entities, such as motors, to simulate motion in assemblies.
- Describe how creating mechanism connections affect parent/child relationships.



Instructor Preparation

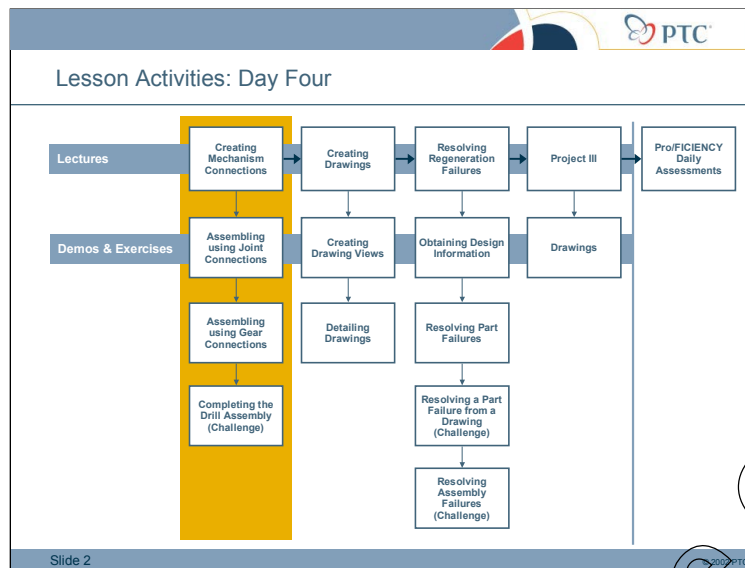
Before teaching *Fast Track to Pro/ENGINEER Wildfire Foundation*, you **must read and thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (2): 40 mins
- Labs (2): 40 mins
- Total: 1 hr 30 min



Objectives

After completing this module, you will be able to:

- Assemble components using connections such as joints and gears.
- Create mechanism entities, such as motors, to simulate motion in assemblies.
- Describe how creating mechanism connections affect parent/child relationships.

Slide 3

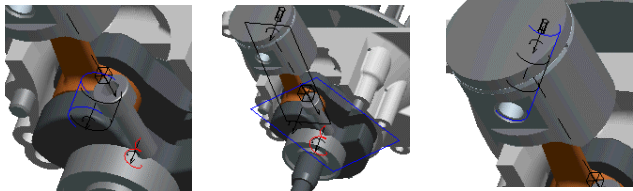
- Many product designs include both static and dynamic components.
- Pro/ENGINEER Wildfire enables you to assemble dynamic components using several connection types.
- You can also add dynamic entities, such as springs and motors, to analyze motion in your product designs.
- In this module, you learn how to assemble components into mechanism assemblies, and how to simulate motion.

PTC

Assembling Components Using Connections

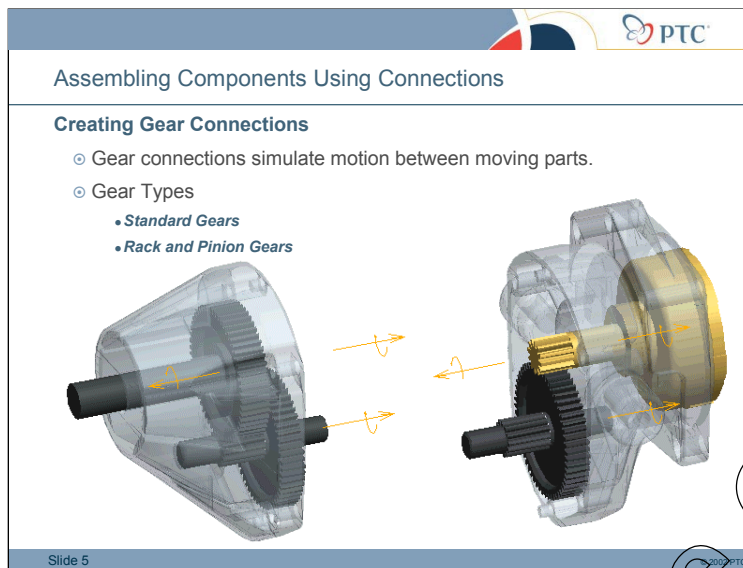
Creating Joint Connections

- Simulate motion between moving parts.
- Dynamically drag components through their range of motion and check for interferences.
- Joint connection types:
 - *Pin, Cylinder, Slider*



Slide 4

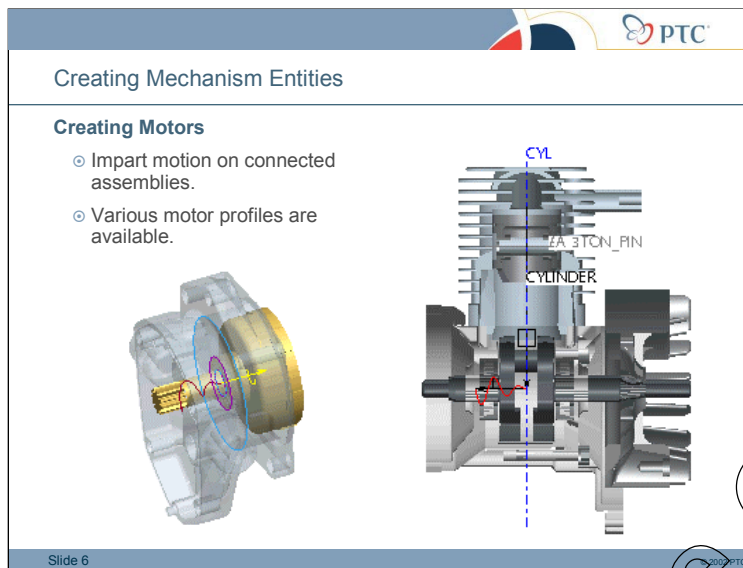
- A mechanical connection is a method of constraining parts so that they form a joint.
- Creating a joint constraint is similar to creating constraints between assembled parts.
- Joint constraints enable you to create true-to-life connections between parts so that you can simulate motion between moving parts.
- Using joints, you can dynamically drag components through their range of motion and check for interferences.



- Gear connections enable you to create true-to-life connections between parts so that you can simulate motion between moving parts.
- You do not have to have gear teeth to model gears using Mechanism. You can have two shafts. The actual part geometry is not used in the gear calculation.
- Unlike gears, cams do use the modeled geometry.

Benefit:

- Create assemblies and analyze assembly motion using “real” connections.



- A motor is a mechanism entity used to impart motion on moving modeled parts.
- You use a motor to impart motion onto your Pro/ENGINEER Wildfire assemblies by selecting a motor profile (for example: constant, ramp, cosine), and then entering the motor conditions.



Parent/Child Relationships – Assembling Components

When assembling components using connections, parent/child relationships are created. Changes made to a parent component affects its children.

- Assembly Models
 - Pin, Cylinder, and Slider Connections
 - Gears
- Servo Motors

Slide 7

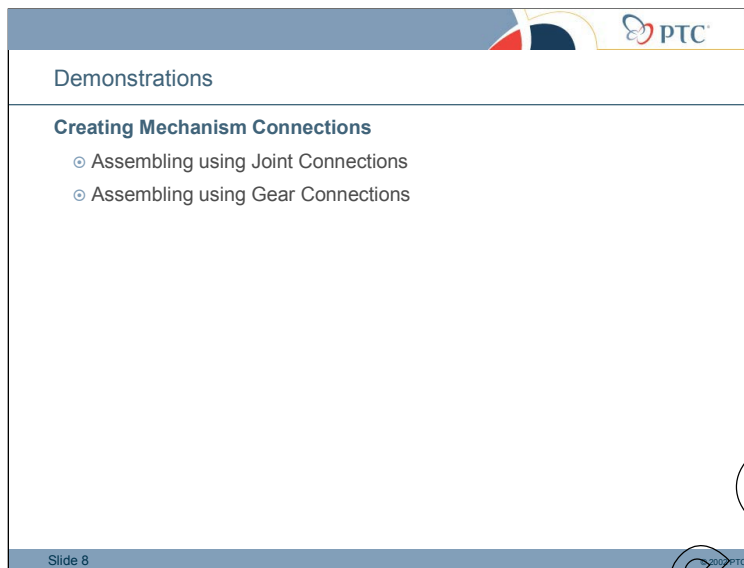
Note to Instructors:

Use this slide to teach students how the functionality they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

NOTE: *It might be best to review this information after completing the demonstrations.*

Assembly models are the parent to all part models within the assembly. Assembly models can also be children if they are assembled to other assembly models.

Servo motors are children of the joint connections that they reference.



You will follow along while I demonstrate:

1. Assembling using Joint Connections
2. Assembling using Gear Connections

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Assemble components using connections such as joints and gears.
- Create mechanism entities, such as motors, to simulate motion in assemblies.
- Describe how creating mechanism connections affect parent/child relationships.

Referencing PTC Help for Module 13 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
About Joint Connections	Simulation	Joint Types To Add a Component with Joint Connections About Joint Connections Degrees of Freedom
About Gear Connections	Simulation	About Gear Pairs Defining Standard Gear Pairs
Servo Motors	Simulation	About Servo Motors (review Mechanism Design, not Design Animation) About Magnitude Settings

Module 13 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Assembling Using Joint Connections

Introduction

In this demonstration, we create joint connections between the moving parts of the drill assembly. Joint connections enable us to assemble moving components within an assembly. For example, we create a connection that enables the crankshaft to rotate, which in turn imparts motion on the connecting rod causing the piston assembly to move up and down.

Joint connections impart a true range of motion to an assembly versus a static assembly. This enables us to check the range of motion and examine the assembly for interferences.

Objectives

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

- Create joint connections.

Scenario

You are assigned to examine the range of motion within the engine and the drill. You must assemble the moving components using joint connections, and then run analyses so that Quality Assurance can run interference checks in the future.

Step 1. Assemble CRANK.ASM with joint connections.

Now that we have finished restructuring the engine, we can begin to assemble the moving components. The first component that we assemble using joint connections is the crank assembly.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all components from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to: **C:\users\student\fast_track_330\module_13**.
3. Open DRILL.ASM. Select ENGINE.ASM, right-click and select **Activate**.
4. Assemble CRANK.ASM, and then click **Connections**. Set the connection type to **Pin**.
5. For the Axis Alignment constraint, select the cylindrical surfaces of the shaft on CRANKSHAFT.PRT, and select the central hole on ENG_BLOCK_FRONT.PRT.
6. For the Translation constraint, use the Search tool to select the FRONT planes from CYLINDER.PRT and CRANKSHAFT.PRT.
7. Complete the component placement, as shown in the following figure.

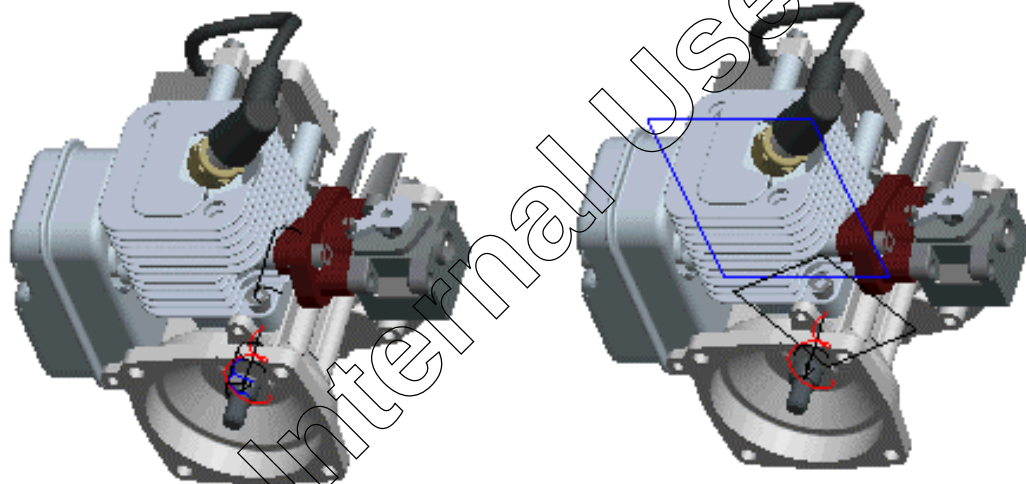


Figure 1: Creating Pin (left) and Translation (right) Constraints

8. Using the model tree, confirm that CRANK.ASM is within ENGINE.ASM.

Step 2. Assemble PISTON.ASM with joint connections.

After the crank assembly is connected, we can assemble the piston assembly with joint connections.

1. Using the model tree, open ENGINE.ASM.
2. Assemble PISTON.ASM, and click **Connections**. Set the connection type to **Slider**.
3. Drag the component into an approximate position.
4. For the Axis Alignment constraint, use the Search tool to select the CYLINDER axis from PISTON.PRT and the CYL axis from CYLINDER.PRT.
5. If necessary, drag PISTON.ASM above CYLINDER.PRT.

6. For the Rotation constraint, use the Search tool to select the FRONT planes from PISTON.PRT and CYLINDER.PRT.
7. Drag PISTON.ASM into CYLINDER.PRT.
8. Complete the component placement, as shown in the following figure.

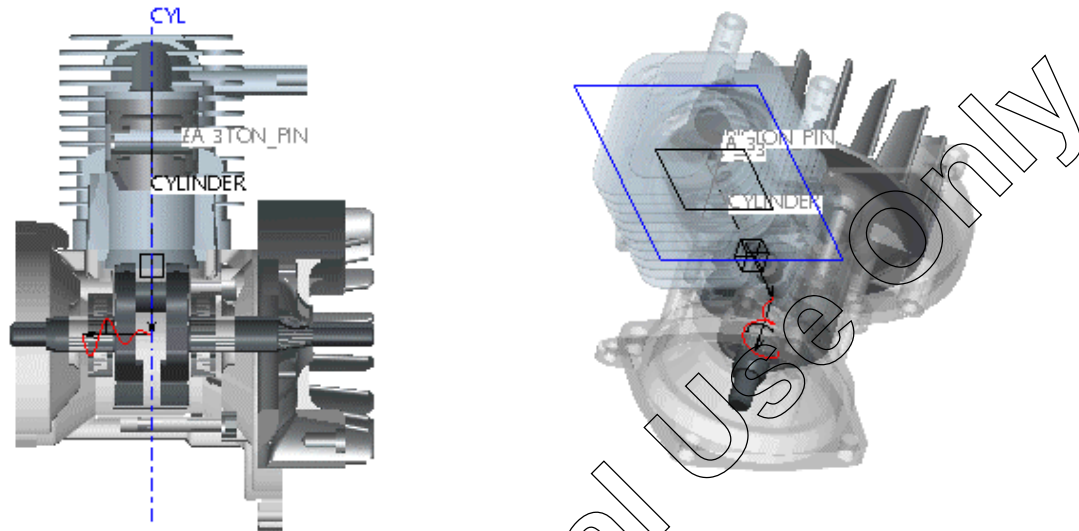


Figure 2: Creating Slider (left) and Translation (right) Constraints

9. Save the model.

Step 3. Assemble the CONNECTING_ROD.PRT with joint connections.

Next, we assemble the connecting rod to the engine and the piston using joint connections.

1. Hide CYLINDER.PRT and ENG_BLOCK_FRONT.PRT.
2. Assemble CONNECTING_ROD.PRT, and click **Connections**. Set the connection type to **Pin**.
3. Drag the component into an approximate position.
4. For the Axis Alignment constraint, select the appropriate cylindrical surfaces from CRANKSHAFT.PRT and CONNECTING_ROD.PRT.
5. For the Translation constraint, use the Search tool to select the FRONT planes from CONNECTING_ROD.PRT and CRANKSHAFT.
6. Add a second connection. Set the connection type to **Cylinder**.
7. For the Axis Alignment select the appropriate cylindrical surfaces of PISTON_PIN.PRT and CONNECTING_ROD.PRT.
8. Complete the component placement, as shown in the following figure.

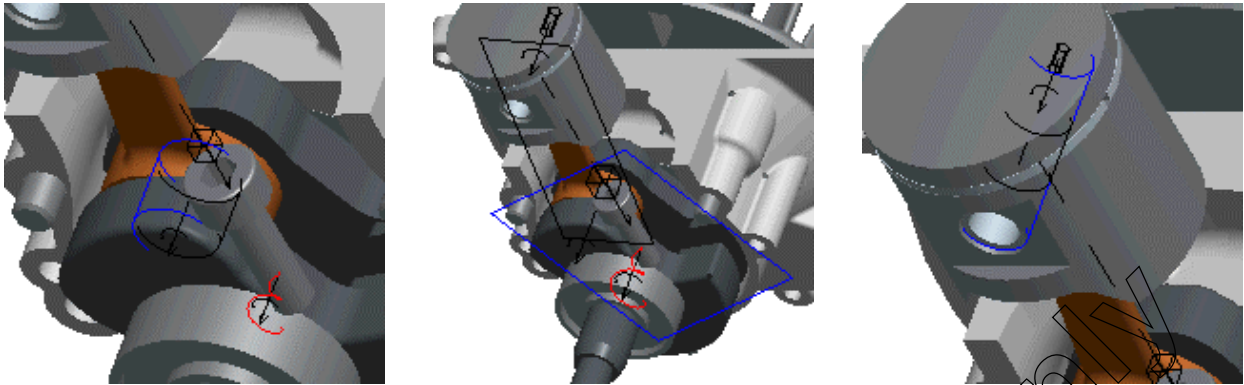


Figure 3: Creating Pin (left and middle) and Cylindrical (right) Constraints

9. Unhide CYLINDER.PRT and ENG_BLOCK_FRONT.PRT.
10. Apply the **Default_transparent** appearance to CYLINDER.PRT and the **Silver_transparent** appearance to ENG_BLOCK_FRONT.PRT and ENG_BLOCK_REAR.PRT.

Note:

On-screen performance with transparency enabled can be improved using the **Stippled** option. Click **View > Display Settings > Model Display**, and then select the SHADE tab. You can also toggle Transparency on and off.

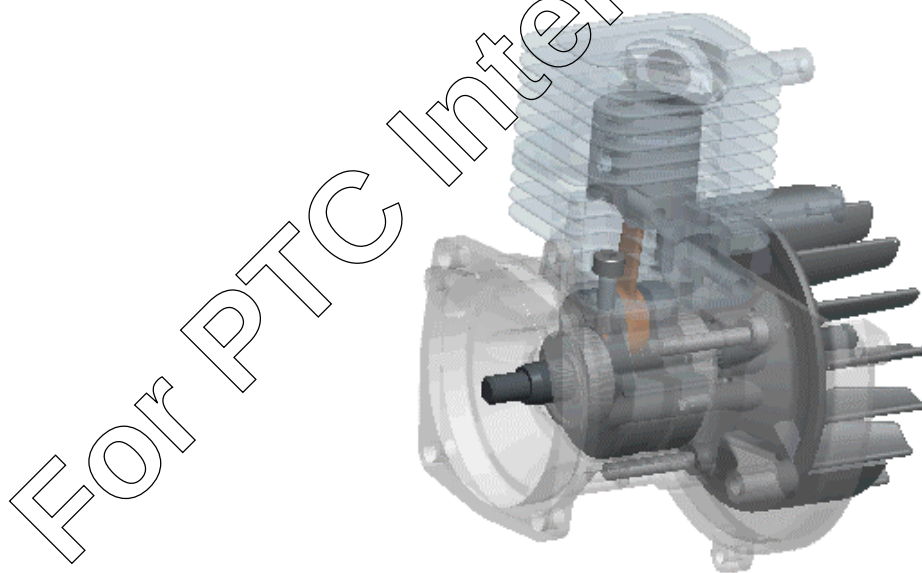


Figure 4: Applying Transparent Appearances to the ENGINE.ASM

11. Save the model.

Step 4. Run the mechanism.

Now that we have assembled all of the critical moving components, we can add a motor and create a mechanism analysis. We use the motor to impart motion on the moving parts. And, the analysis is used to set operating conditions. Finally, we run the analysis to check the operation.

1. Orient ENGINE.ASM, as shown in the following figure. Click **Applications > Mechanism** to enable Mechanism mode.
2. Using the toolbar, drag CRANK.ASM by selecting a blade on FLYWHEEL.PRT to ensure that the mechanism is working properly. Middle-click when finished dragging.

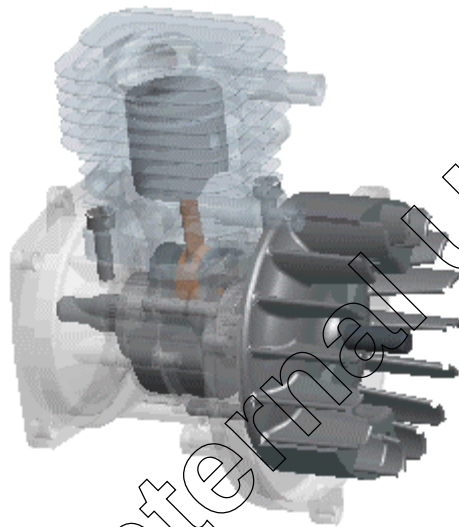


Figure 5. Orienting the ENGINE.ASM

3. Using the toolbar, define a Servo Motor and click **New**. Enter **Engine** as the name.
4. For the driven Joint Axis, select the Pin Joint between CRANKSHAFT.PRT and ENG_BLOCK_FRONT.PRT.

Note:

Notice the CRANK.PRT and FLYWHEEL.PRT, which are the moving bodies, highlight in cyan. The ENG_BLOCK_FRONT.PRT, ENG_BLOCK_REAR.PRT and CYLINDER.PRT, which are the stationary bodies, highlight in green.

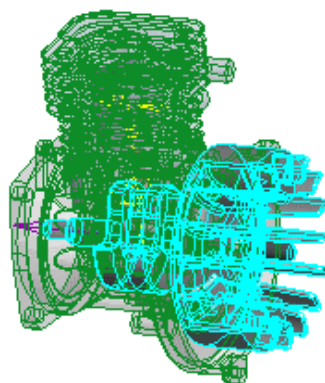


Figure 6: Defining the Moving and Stationary Bodies

5. Select the PROFILE tab. Set the Specification to **Velocity**. Enter **90** as the constant magnitude value (deg/sec). Complete the motor definition.
6. Using the toolbar, run an Analysis and click **New**. Enter **Run_Engine** as the name.
7. Enter **4** as the end time, and then click **Run**.
8. After the analysis run is finished, click **Ok > Close** to complete the analysis.
9. Using the model tree, expand the PLAYBACKS node. Select **Run_Engine**, right-click and select **Play**. Increase the speed to maximum, and play the animation.

Note:

You can run a mechanism with some components hidden, or Hide/Unhide components while a mechanism is running.

10. When you are finished, stop the animation, and close the ANIMATE dialog box.
11. Using the model tree, select the **Run_Engine** playback, right-click and select **Save > Save**.
12. Click **Applications > Standard** to enable Standard mode.
13. Save the model.

Step 5. (OPTIONAL) Assemble CLUTCH.ASM to DRILL.ASM and run the mechanism.

Instructor: Skip this step for the demo.

1. Close the ENGINE.ASM window to return to the DRILL.ASM.
2. Assemble the CLUTCH.ASM. Drag the component to an approximate position. Select appropriate surfaces to create an Insert constraint and a Mate Coincident constraint.
3. Add a third constraint. Using the Search tool, select the RIGHT planes in CRANKSHAFT.PRT and CLUTCH_BODY.PRT. Specify an Align Coincident constraint.
4. Complete the component placement, as shown in the following figure.

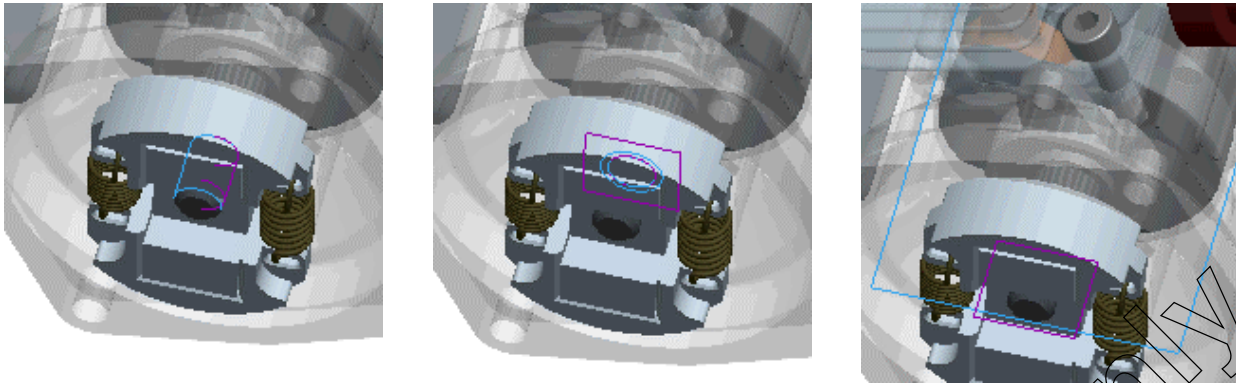


Figure 7: Creating the Insert (left), Mate Coincident (middle), and Align Coincident (right) Constraints

Note:

The orientation of CLUTCH.ASM will be dependent on when the mechanism was stopped playing.

5. Click **Applications > Mechanism** to enable Mechanism mode.
6. Using the toolbar, define an Analysis. Enter **Run_Drill** as the name. Enter **4** for the end time, and then click **Run**.

Note:

Saved analyses are only accessible from the assembly in which they were created.

7. After the analysis run is finished, click **Ok > Close** to complete the analysis.
8. Using the model tree, expand the **Playbacks** node. Select **Run_Drill**, right-click and select **Play**. Increase the speed to maximum, and play the animation. When you are finished, stop the animation, and close the ANIMATE dialog box.
9. Using the model tree, select the **Run_Drill** playback, right-click and select **Save > Save**.
10. Save the model.
11. Close the window

This completes the exercise.

Exercise 2: Assembling Using Gear Connections

Introduction

In this demonstration, we create gear connections between the moving parts of the drill assembly. Gear connections enable us to accurately assemble moving components within an assembly.

For example, using gear connections, we can simulate the internal mechanism motion of a wristwatch, and also impart motion from the internal watch gears to the time hands on the external face of the watch.

Using gear connections, we can model reductions between gear trains showing the increase or decrease in rotational movement.

Objectives

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

- Create joint connections.

Scenario

You are assigned to examine the range of motion within the drill's gearbox. You must assemble the moving components using gear connections and run an analysis, so that Quality Assurance can run interference checks in the future.

Step 1. Create Gear connections.

In this first step, we create gear connections between two gear pairs.

1. Open GEARBOX_CHUCK.ASM.
2. Turn off display of all datum features.
3. Click **Applications > Mechanism** to enable Mechanism mode.
4. Notice the existing Pin connections, as shown in the following figure.

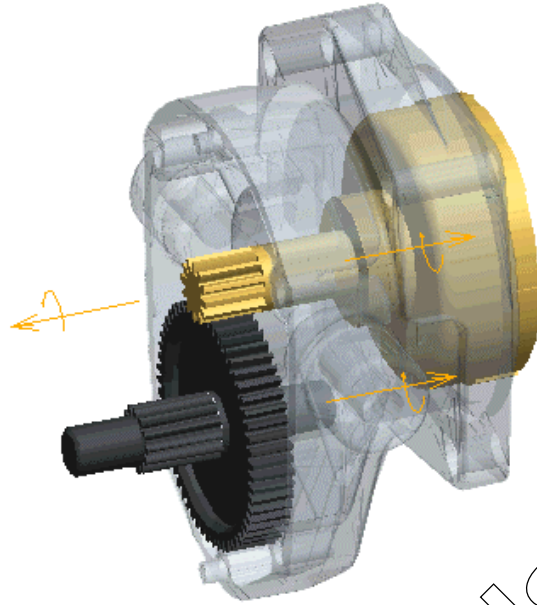


Figure 8: GEARBOX_CHUCK.ASM

5. Using the toolbar, drag the PRIMARY_GEAR_SHAFT.PRT by selecting a point on the drum. Note that the REDUCTION_GEAR_SHAFT.PRT remains stationary. Middle-click when finished.
6. Using the toolbar, define a Gear-Pair Connection and click **New**. To define Gear1, select the joint axis on PRIMARY_GEAR_SHAFT.PRT. Enter **11** for the pitch diameter.
7. Select the Gear2 tab, and select the joint axis on REDUCTION_GEAR_SHAFT.PRT. Enter **46** for the pitch diameter.
8. Complete the gear definition.
9. Using the toolbar, drag the PRIMARY_GEAR_SHAFT.PRT by selecting a point on the drum. Notice that the REDUCTION_GEAR_SHAFT.PRT now rotates proportionately. Middle-click when finished.
10. Unhide GEARBOX_FRONT.PRT and FINAL_GEAR_SHAFT.PRT.
11. Hide GEARBOX_REAR.PRT and PRIMARY_GEAR_SHAFT.PRT.

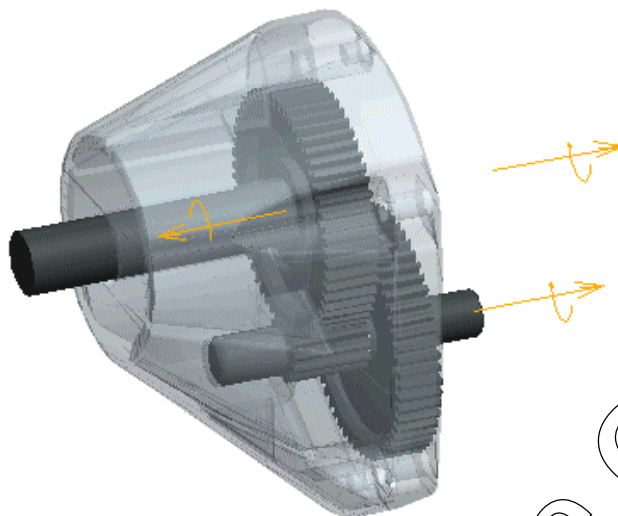


Figure 9: Components Hidden

12. Use the toolbar to define a Gear-Pair Connection and click **New**. To define Gear1, select the joint axis on REDUCTION_GEAR_SHAFT. Enter **12** for the pitch diameter.
13. Select the Gear2 tab and select the joint axis on FINAL_GEAR_SHAFT.PRT. Enter **46** for the pitch diameter.
14. Complete the gear definition.
15. Unhide GEARBOX_REAR.PRT and PRIMARY_GEAR_SHAFT.PRT. Display Coordinate Systems. Click **Applications > Standard**.
16. Assemble the DRILL_CHUCK.ASM. Select the CHUCK and CHUCK_ALIGN coordinate systems. Complete the component placement, as shown in the following figure.

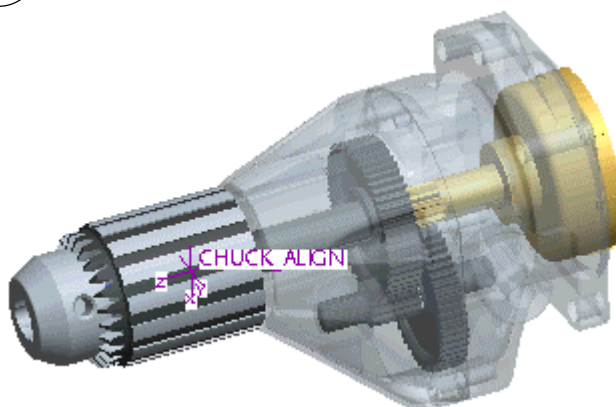


Figure 10: Assembling the DRILL_CHUCK.ASM

17. Turn off display of Coordinate Systems

18. Save the model.

Step 2. Run GEARBOX_CHUCK.ASM mechanism.

Now that the gears have been constrained, we create a motor and run an analysis.

1. Enable Mechanism mode.
2. Using the toolbar, drag the PRIMARY_GEAR_SHAFT.PRT by selecting a point on the drum. Note that the gears 'mesh' and the DRILL_CHUCK.ASM rotates proportionately. Middle-click when finished.

Note:

The Gear connection does not use the geometry of the gear teeth. Instead, it simply applies the ratio of the pitch diameters when the mechanism is run.

3. Use the toolbar to create a Servo Motor and click **New**. Enter **Gearbox** as the name.
4. For the driven Joint Axis, select the Pin Joint on PRIMARY_GEAR_SHAFT.PRT.
5. Click **Flip** so that the magenta arrow, which indicates the direction of rotation, points into GEARBOX_CHUCK.ASM.

Note:

Notice that the *moving* bodies highlight in cyan and the *stationary* bodies highlight in green.

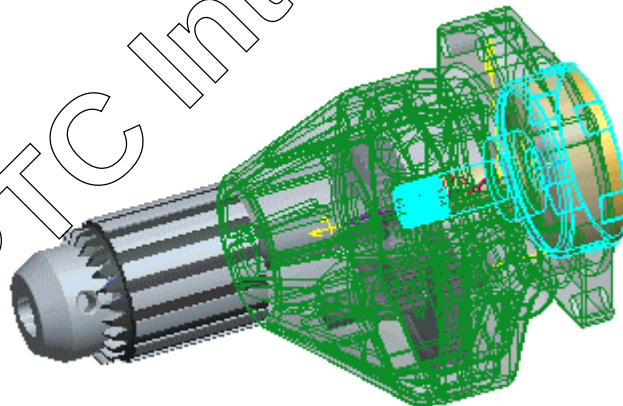


Figure 11: Defining the Moving and Stationary Bodies

6. Select the PROFILE tab and set the Specification to **Velocity**. Enter **90** as the constant magnitude value (deg/sec). Complete the motor definition.
7. Using the toolbar, run an Analysis and click **New**. Enter **Run_Gearbox** as the name.
8. Enter **8** for the end time and click **Run**. After the analysis run is finished click **Ok > Close** to complete the analysis.

9. Using the model tree, expand the PLAYBACKS node. Select **Run_Gearbox**, right click and select **Play**. Increase the speed to maximum, and play the animation.
10. When you are finished, stop the animation, and close the ANIMATION dialog box.
11. Using the model tree, select the **Run_Gearbox** playback, right-click and select **Save**.
12. Click **Applications > Standard** to return to the GEARBOX_CHUCK.ASM window.
13. Save the model.
14. Close the window.

Step 3. (OPTIONAL) Assemble GEARBOX_CHUCK.ASM to DRILL.ASM.

Instructor: Skip this step for the demo.

Note:

This exercise assumes that the previous optional exercise has been completed.

1. Open DRILL.ASM.
2. Assemble GEARBOX_CHUCK.ASM to DRILL.ASM, as shown in the following figure.



Figure 12: Assembling the GEARBOX_CHUCK.ASM to DRILL.ASM

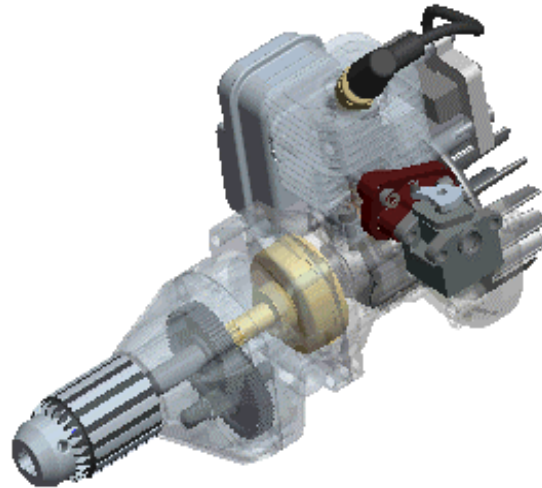


Figure 13: DRILL.ASM with GEARBOX_CHUCK.ASM

3. Enable Mechanism mode.
4. Edit the existing **Run_Drill** analysis. Select the MOTORS tab and notice that this analysis has already a servo motor. Add the GEARBOX motor to the motor list. Leave the other default settings and run the analysis.
5. After the analysis run is finished, click **Ok** to complete the analysis. Using the model tree, select the RUN_DRILL playback, and play it. Increase the speed to maximum and play. Stop and close the PLAYBACK dialog box when finished.
6. Enable Standard mode. Save the motion result.
7. Using the model tree, show Features and Suppressed Objects. Examine DRILL_CHUCK.ASM. Select the AUGER_BIT and resume it.
8. Enable Mechanism mode.
9. Run the mechanism.
10. Enable Standard mode.
11. Suppress the AUGER_BIT.
12. Save the model.
13. Close all windows.
14. Erase all models from memory.

This completes the exercise.

Exercise 3: Completing the Drill Assembly (Challenge)

Objective

In this exercise, you complete the assembly of the drill using techniques that you have learned.

Scenario

You have completed all of the parts for the new drill. You are now assigned to complete the drill assembly so that drawings and documentation work can be completed.

Step 1. Complete the DRILL.ASM.

1. Open DRILL.ASM, if necessary.
2. Disable the model transparency.
3. Activate GEARBOX_CHUCK.ASM. Assemble four BOLT_5-18 instances, as shown in the following figure.

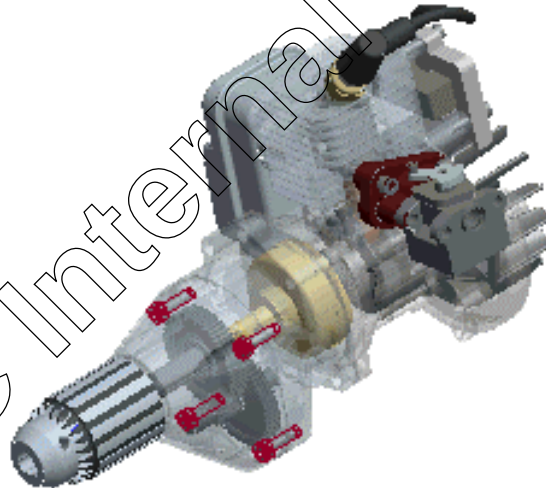


Figure 14: Securing GEARBOX_FRONT.PRT to GEARBOX_REAR.PRT

4. Activate DRILL.ASM and assemble the FUEL_TANK.ASM. Activate FUEL_TANK.ASM and assemble two BOLT_5-15 instances, as shown in the following figure.

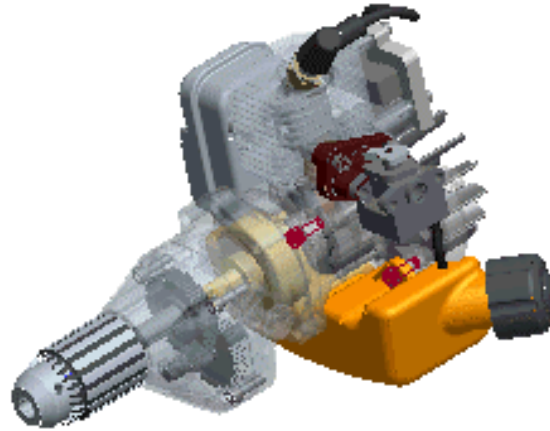


Figure 15: Assembling the FUEL_TANK.ASM

5. Activate CARBURETOR.ASM and assemble CARBURETOR_PLATE.PRT. Assemble two BOLT_5-60 instances, as shown in the following figure.

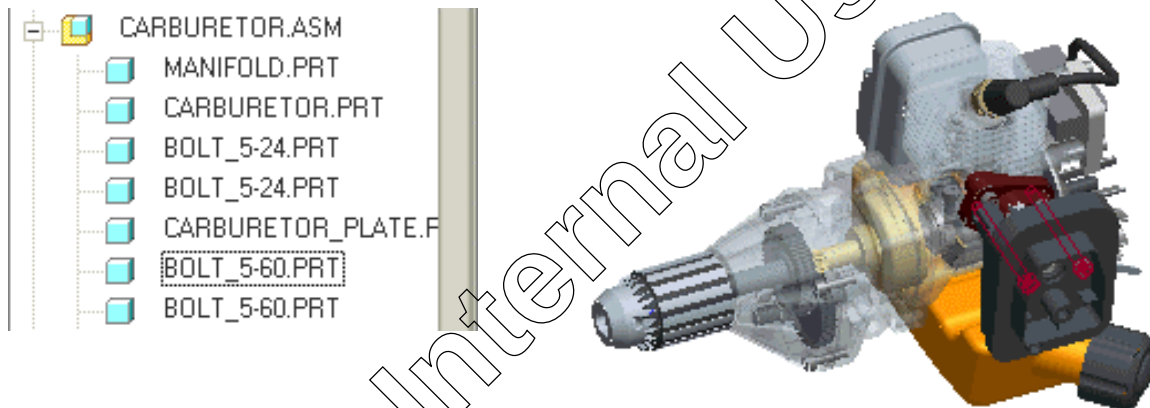


Figure 16: Assembling the CARBURETOR_PLATE.PRT

6. With CARBURETOR.ASM still active, assemble the AIR_FILTER_COVER.PRT. Then assemble one BOLT_5-24 BOLT instance, as shown in the following figure.

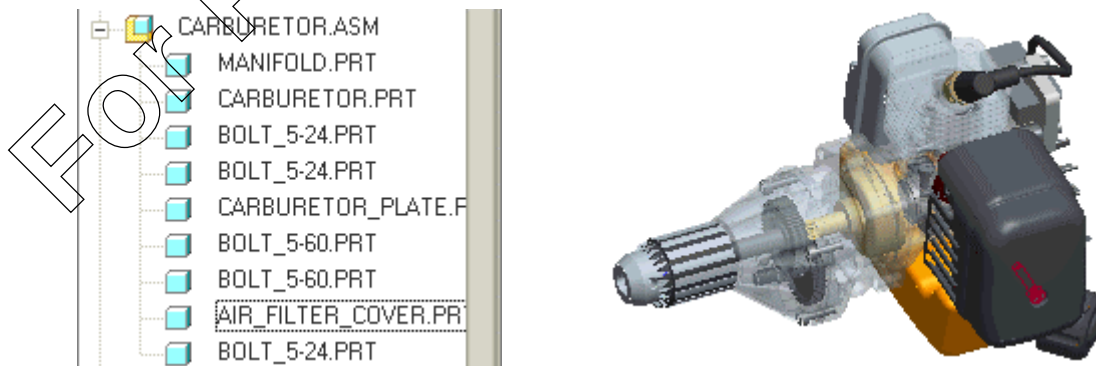


Figure 17: Assembling the AIR_FILTER_COVER.PRT

7. Activate DRILL.ASM and assemble the RECOIL.ASM.
Activate RECOIL.ASM and assemble three BOLT_4-15 instances.
8. With RECOIL.ASM still active, assemble the RECOIL_GRIP.PRT using the available Csys. Reorder RECOIL_GRIP.PRT before the bolts, as shown in the following figure.

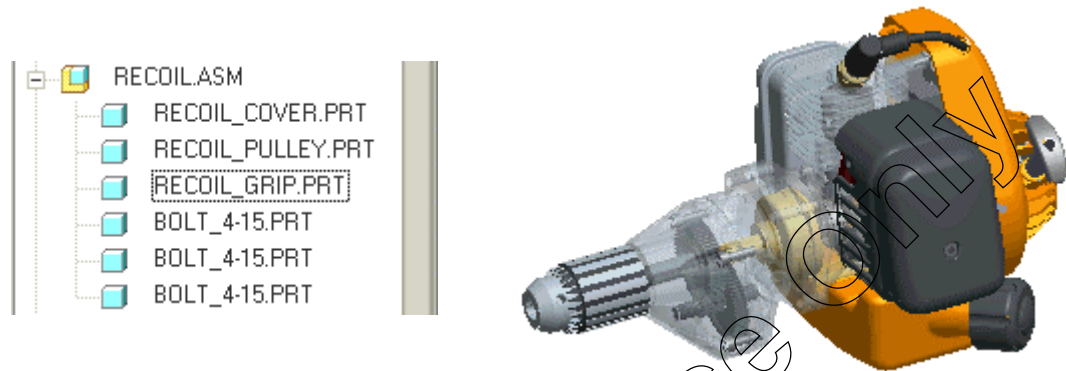


Figure 18: Assembling the RECOIL_GRIP.PRT to RECOIL.ASM

9. Activate DRILL.ASM and assemble ENGINE_COVER.PRT. Assemble three BOLT_4-12 instances to fasten ENGINE_COVER.PRT, as shown in the following figure.



Figure 19: Assembling the ENGINE_COVER.PRT

10. Assemble the HANDLE_MAIN.ASM, as shown in the following figure.

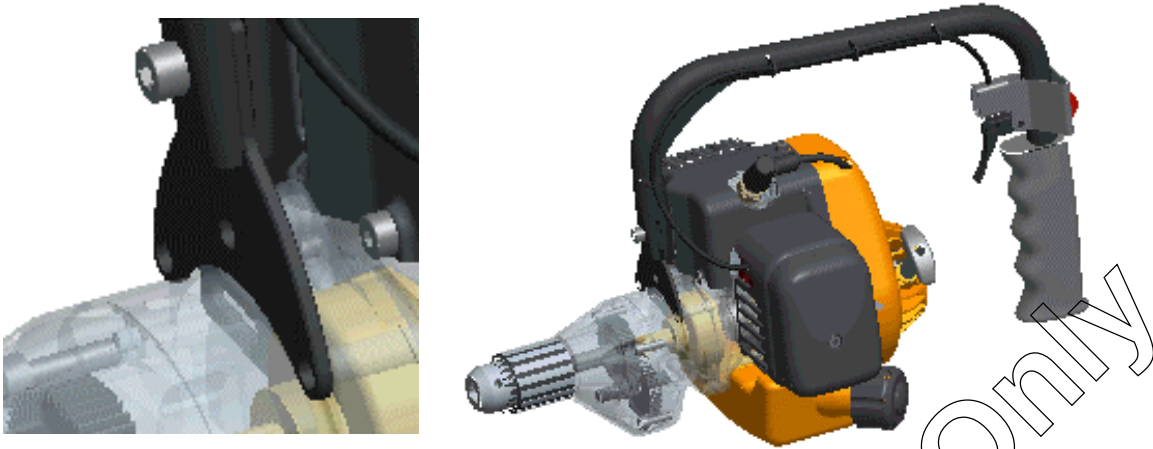


Figure 20: Assembling the HANDLE_MAIN.ASM

11. Assemble two BOLT_6-36 instances to fasten HANDLE_MAIN.ASM and GEARBOX_REAR.PRT to ENG_BLOCK_FRONT.PRT.
12. Assemble one 6-15 BOLT.PRT instance to fasten HANDLE_MAIN.ASM to GEARBOX_REAR.PRT.
13. Assemble two BOLT_6-20 instances to fasten GEARBOX_REAR.PRT to ENG_BLOCK_FRONT.PRT.
14. Assemble the HANDLE_SIDE.ASM, as shown in the following figure. Note that the handle can be assembled to either side.

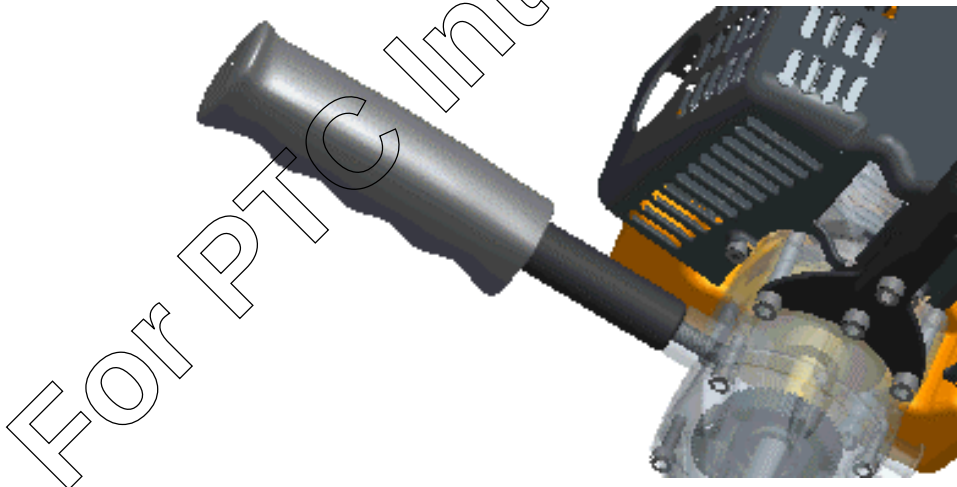


Figure 21: Assembling the HANDLE_SIDE.ASM

15. Using the model tree, show features and suppressed objects
16. Resume the drill bit of your choice.



Figure 22: Completed DRILL.ASM

17. Save the model.
 18. Close all windows.
 19. Erase all models from memory.
- This completes the exercise.

Summary

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

- Assemble the components using connections such as joints and gears.
- Create mechanism entities, such as motors, to simulate motion in assemblies.
- Describe how creating mechanism connections affect parent/child relationships.

For PTC Internal Use Only

Creating Drawings

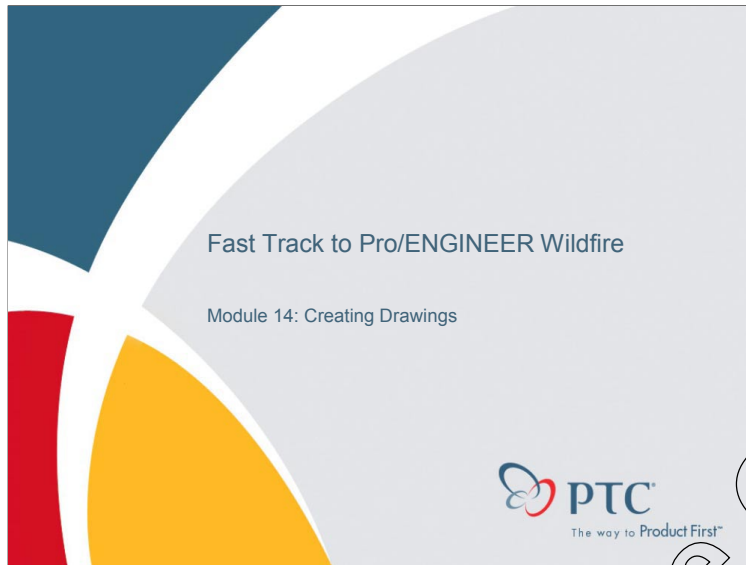
Introduction

Drawings are used for documenting the final production design of parts and assembly models. They typically contain two- and three-dimensional design models views, as well as dimensions, notes, and Bills of Material. Drawings are frequently used in the manufacture of product designs. Due to their importance, they are usually stored in secure locations and are tightly controlled by configuration- and release-management procedures.

Objectives

After completing this module, you will be able to:

- Create drawings of design models.
- Detail drawings with dimensions and notes.
- Describe how creating drawings affects parent/child relationships.



Instructor Preparation

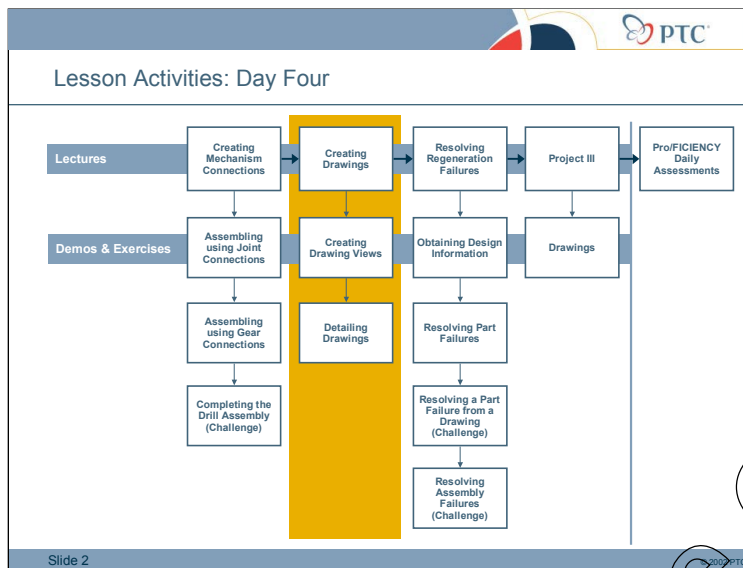
Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read and thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (2): 40 mins
- Labs (2): 40 mins
- Total: 1 hr 30 min



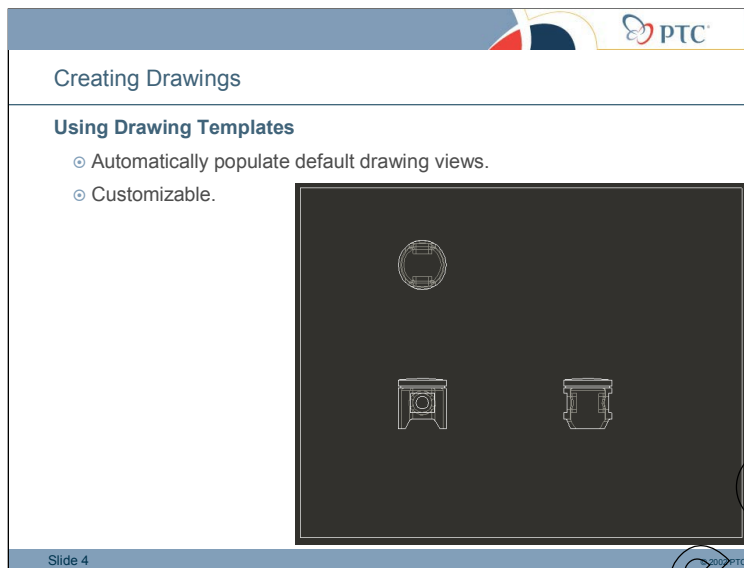
Objectives

After completing this module, you will be able to:

- ◉ Create drawings of design models.
- ◉ Detail drawings with dimensions and notes.
- ◉ Describe how creating drawings affect parent/child relationships.

Slide 3

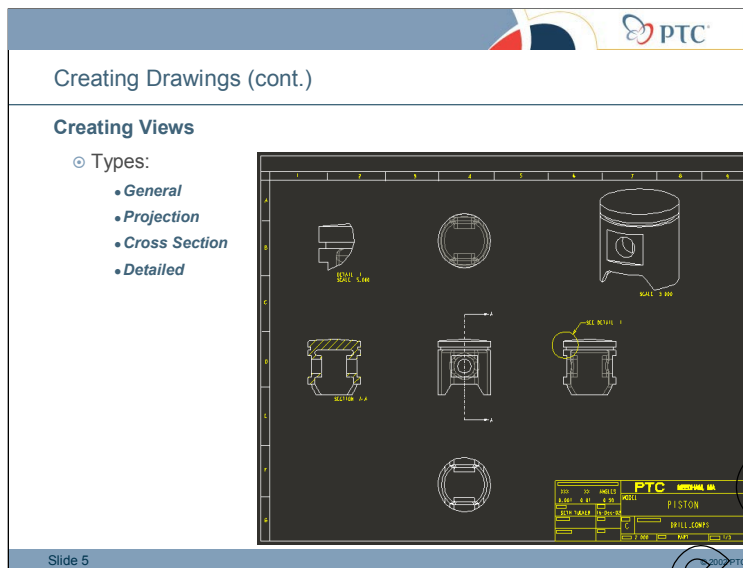
- Drawings are used for documenting the design of part and assembly models.
- They are an important part of manufacturing processes.
- They typically contain multiple 2-D (two-dimensional) and 3-D (three-dimensional) views of design models, as well as dimensions, notes, and bill of materials lists.



- Like part and assembly templates, a drawing template provides you with a starting point to create your drawings.
- A drawing template can be customized so that your company formats and standards can be saved.
- Templates enable you to create drawings in a fraction of the time that it would take to sketch in 2-D.
- You use drawing templates when you want to create a standardized drawing.

Benefit:

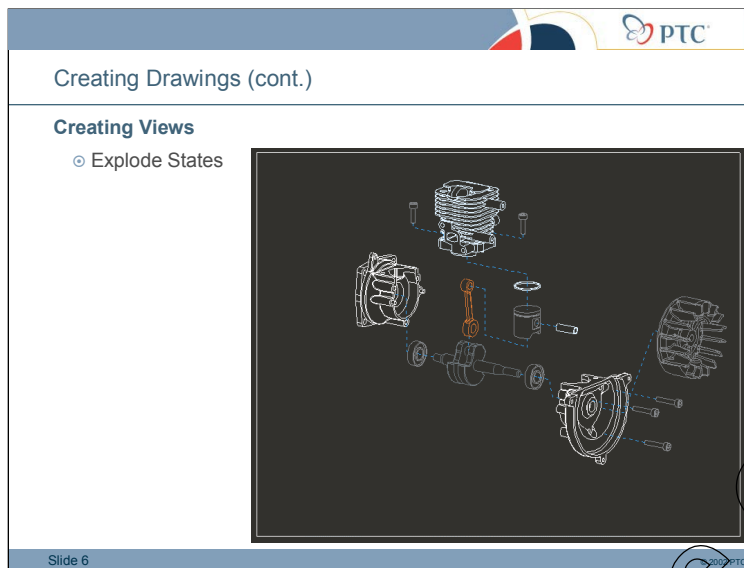
- Drawing templates can store company standard formats. This enables you to automatically create drawings in a fraction of the time it would take to sketch them.



- You use drawing views to place 2-D or 3-D views of your models onto a drawing format or empty template.
- You use drawing views in place of a drawing template, or to delete or replace views on the template.
- You can use drawing views to customize the view perspective of your models.
- You use customized drawings based on 3-D models. Manufacturing personnel use this information to build your product.

Benefit:

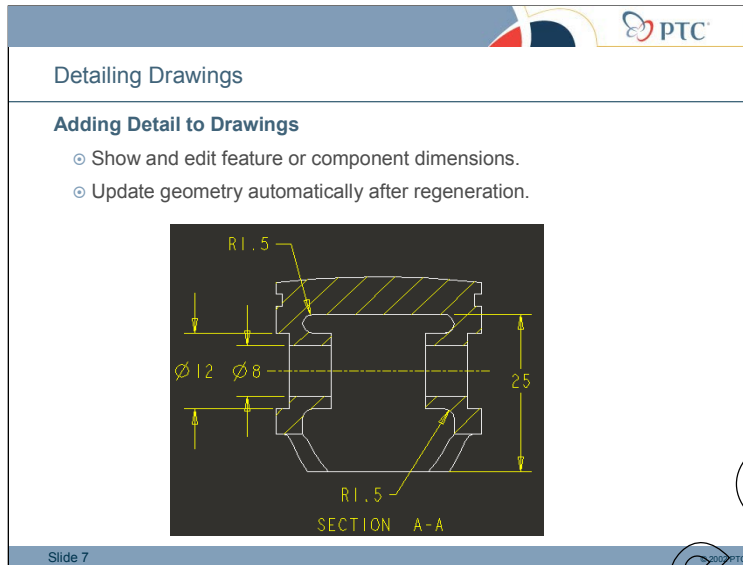
- Create customized drawings based on 3-D models that display information needed by manufacturing to produce your product.



- Exploded views are used to illustrate assembly and dis-assembly (taking a product apart).
- In addition, BOM balloons and a table indexing the parts can also be added to the drawing; this allow readers to easy reference the component information.
- With exploded views, you can create customized drawings based on 3-D models; these can display information needed by manufacturing personnel.

Benefit:

- Create customized drawings based on 3-D models that display information needed by manufacturing to produce your product.



- Dimensioning detail enables you to add dimensions directly from the model's features or the assembly's constraints.
- You add dimensions by simply selecting the feature and enabling/disabling the dimensions to produce the drawing needed by your manufacturing operations.
- Dimensions for your drawings are taken directly from the model features or from the assembly constraints; there is no need to recreate sketches.
- Another benefit to dimensioning your drawings is that you can take advantage of Pro/ENGINEER Wildfire's bi-directional associativity. This means that no matter where you change a feature dimension – all model deliverables will update to the new dimension. This can save significant time, such as when manufacturing reports that a certain dimension needs to be changed because they cannot manufacture the part.

Benefit:

- Bi-directional associativity enables you to change dimensions anywhere. All the model deliverables update to the new dimension.

PTC

Detailing Drawings (cont.)

Adding Notes to Drawings

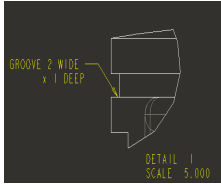
- Simple Notes
- Parametric Notes

Adding Sheets to Drawings

- Move detail items between sheets.

Adding Bill of Materials (BOM)

- BOM Balloons



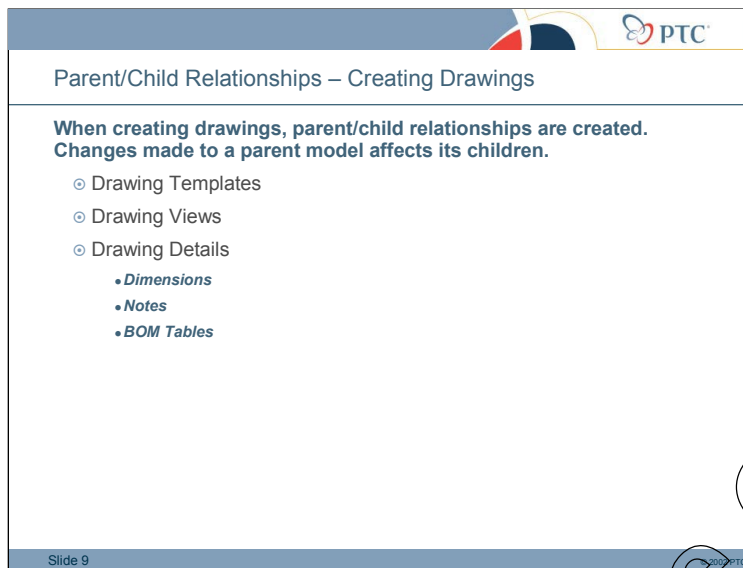
ITEM	NAME	QTY
1	BOLT_5-16	2
2	BOLT_5-20	3
3	CONNECTING_ROD	1
4	CRANKSHAFT	1
5	CYLINDER	1
6	ENG_BEARING	2
7	ENG_BLOCK_FRONT	1
8	ENG_BLOCK_REAR	1
9	FLYWHEEL	1
10	PISTON	1
11	PISTON_PIN	1
12	PISTON_RING	1

Slide 8

- You use notes to communicate critical assembly design information.
- Notes can be either simple or parametric.
- Simple notes are just “dumb” text – only the notes change when you edit them.
- Parametric notes are associative – deliverables affected by the note (related to the note) will update if you change the note.
- BOM (Bill of Materials) tables can be used to detail the location and number of parts included in the assembly for manufacturers.
- BOM tables can be also be parametric, which means that the table updates whenever you add or delete a part to the assembly.
- Notes, views, and other detail items can be moved from one sheet to another.

Benefit:

- Communicate critical design information to others.
- Parametric notes associatively update deliverables when they are modified.



Note to Instructors:

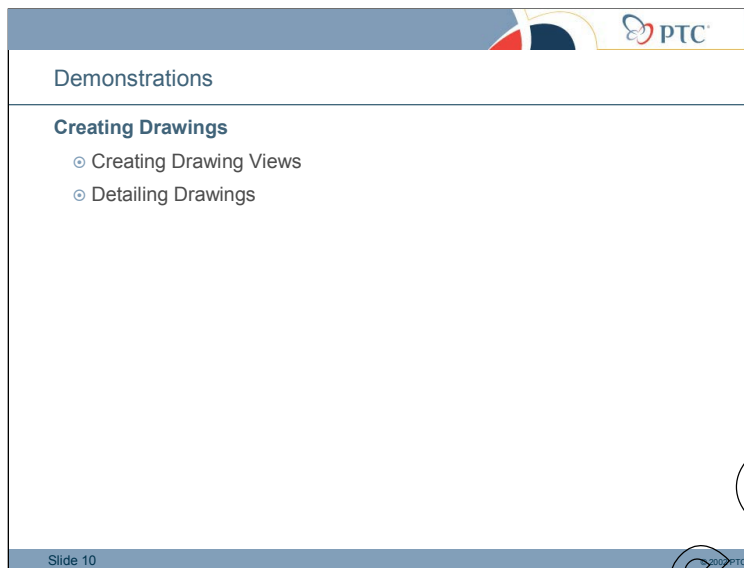
Use this slide to teach students how the functionality that they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

NOTE: It might be best to review this information after completing the demonstrations.

Like part and assembly templates, drawing templates do not have parent/child relationships.

Drawing views are children to either the Saved Views in the part or of the reference orientations selected.

Drawing details such as dimensions, notes and BOM tables are generally children to the model. There are exceptions. For example, notes have to be parametric notes and not 'dumb' text notes.



You will follow along while I demonstrate:

1. Creating Drawing Views
2. Detailing Drawings

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Create drawings of design models.
- Detail drawings with dimensions and notes.
- Describe how creating drawings affect parent/child relationships.

Referencing PTC Help for Module 14 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Templates	Detailing	To Create a Drawing Using a Drawing Template
Views	Detailing	About the Basic View Types About Inserting Drawing Views To Add Models to the Drawing
Layers	Detailing	About Layers in Drawing Mode
Explode States	Detailing	About Exploded Assembly Views
Dimensions	Detailing	About Applying Dimensions Dimensioning In Pro/E Wildfire
Assembly Drawings	Detailing	About Assembly Drawings
Notes	Detailing	About Note Parameters
Multisheet Drawings	Detailing	About Multisheet Drawings
Formats	Detailing	About Using Formats in a Drawing

Module 14 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Creating Drawing Views

Introduction

In this demonstration, we create a multi-sheet drawing. This drawing will include several drill components by placing views of the components onto a drawing template. Views and templates are important because they enable you to reuse existing models. They save time that normally would be used to sketch the models in 2-D.

Objectives

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

- Create drawings.
- Add views to drawings.

Scenario

The Manufacturing Operations Department (“Manufacturing”) needs to begin planning the tooling necessary to create the Piston and the Handle Bracket parts. Additionally, Manufacturing wants an initial drawing of the completed engine assembly. This is needed to begin retooling of the assembly shop floor and to determine if the engine can be assembled in the available space.

You are assigned to create a drawing for Manufacturing. You must create a drawing that shows projected, cross sectional, and detailed views of the piston. You must also add a second sheet to the drawing that shows projected and flat state views of the handle bracket. Finally, you must add a third sheet that shows the engine assembly in an exploded state.

Step 1. Create a saved view and cross section.

In this step, we create a drawing of the piston by selecting a template and associating the model to the template. The template then automatically orients the piston into three 2-D views. No additional sketching is necessary. When we create drawings in Pro/ENGINEER Wildfire, we simply place models and then orient the views.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to **C:\users\student\fast_track_330\module_14**.
3. Open PISTON.PRT.
4. Orient the model as shown in the following figure. Use the View Manager to save the orientation as **3D**.
5. Click **Tools > Model Sectioning > New**.
6. Enter **A** as the name, and click **Done**. Select datum **RIGHT** as the sectioning plane. Then click **Display > Show X-section**. Close the Cross Section dialog box when finished.

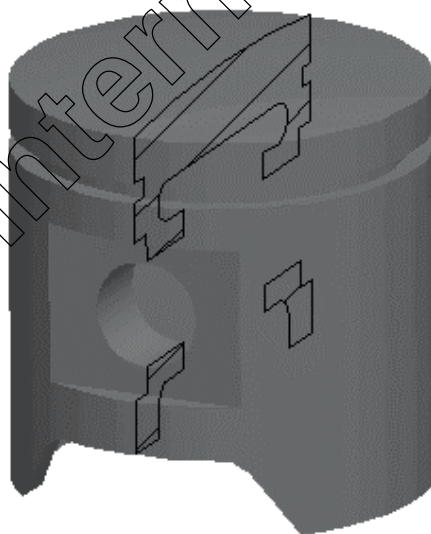


Figure 1: Creating the 'A' Cross Section

7. Save the model and close the window.

Step 2. Create a new drawing with default views.

1. Click **File > New > Drawing** and enter **Drill_Comps**. Click **Ok** to continue, using the default template.

2. Click **Browse** and open PISTON.PRT. Select **C_drawing** as the Template and click **OK**.
3. Turn off display of all datum features and Repaint the screen. Note that the template automatically creates three standard views (front, top, right).

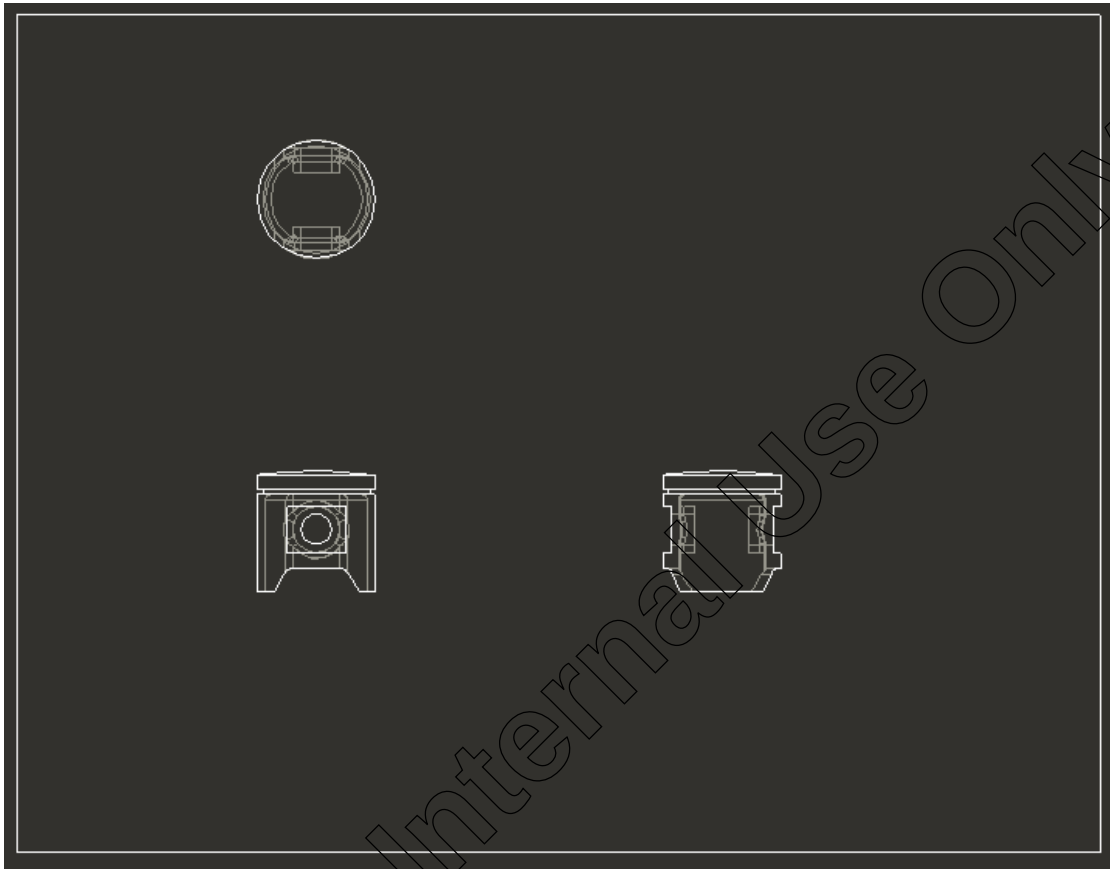


Figure 2: Creating DRILL_COMPS.DRW

Step 3. Create projected views of the PISTON.PRT.

1. Right-click and select **Insert Drawing View**. Accept all defaults and click **Done**. Select a point under the front view, as shown in the following figure.

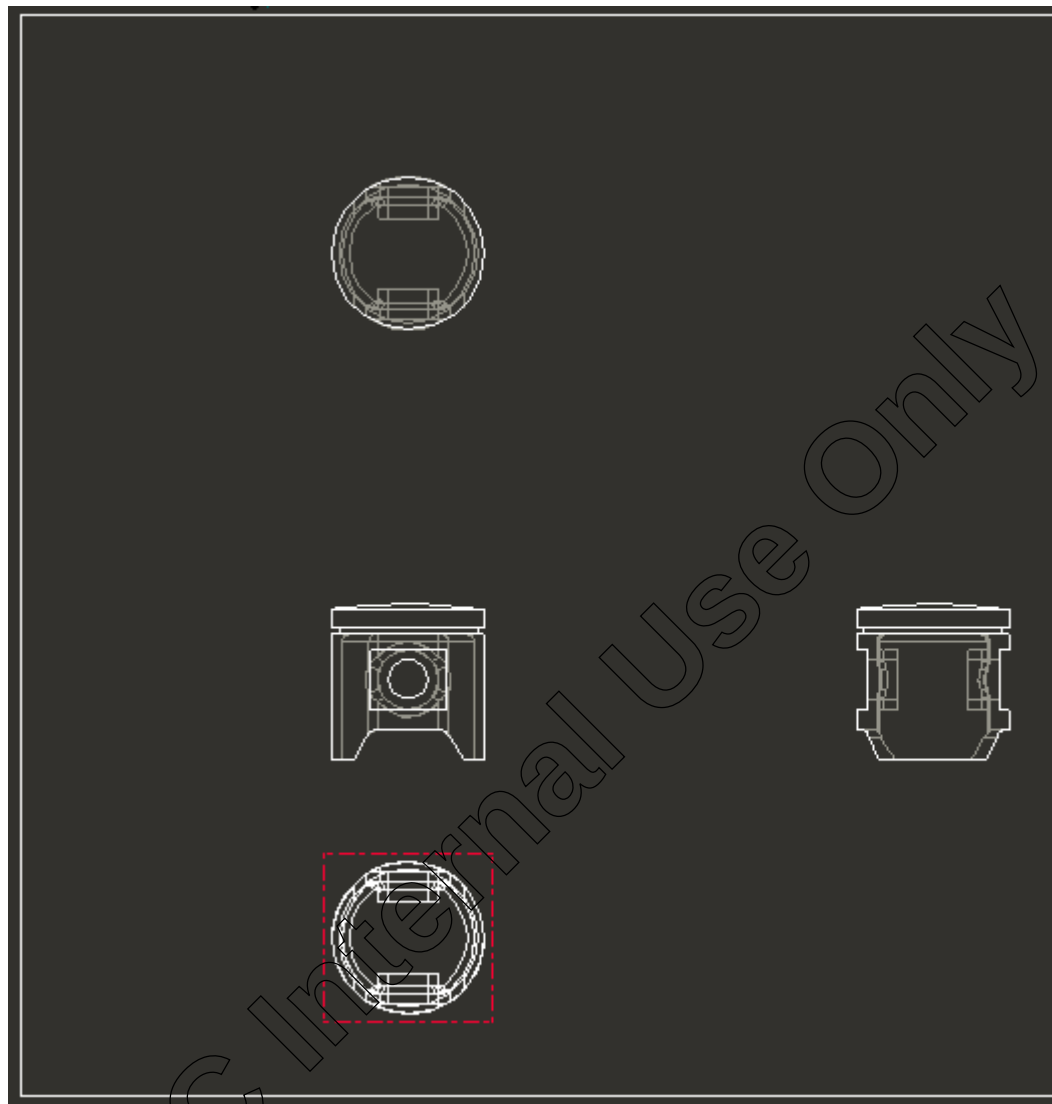


Figure 3: Inserting a Projection View

2. Select on the background to de-select the previous view.
3. Right-click and select **Insert Drawing View**. Click **Section > Done > Done**. Select a point to the left of the front view, as shown in the following figure.
4. Click cross-section **A**, and select the front view to place the section arrows.

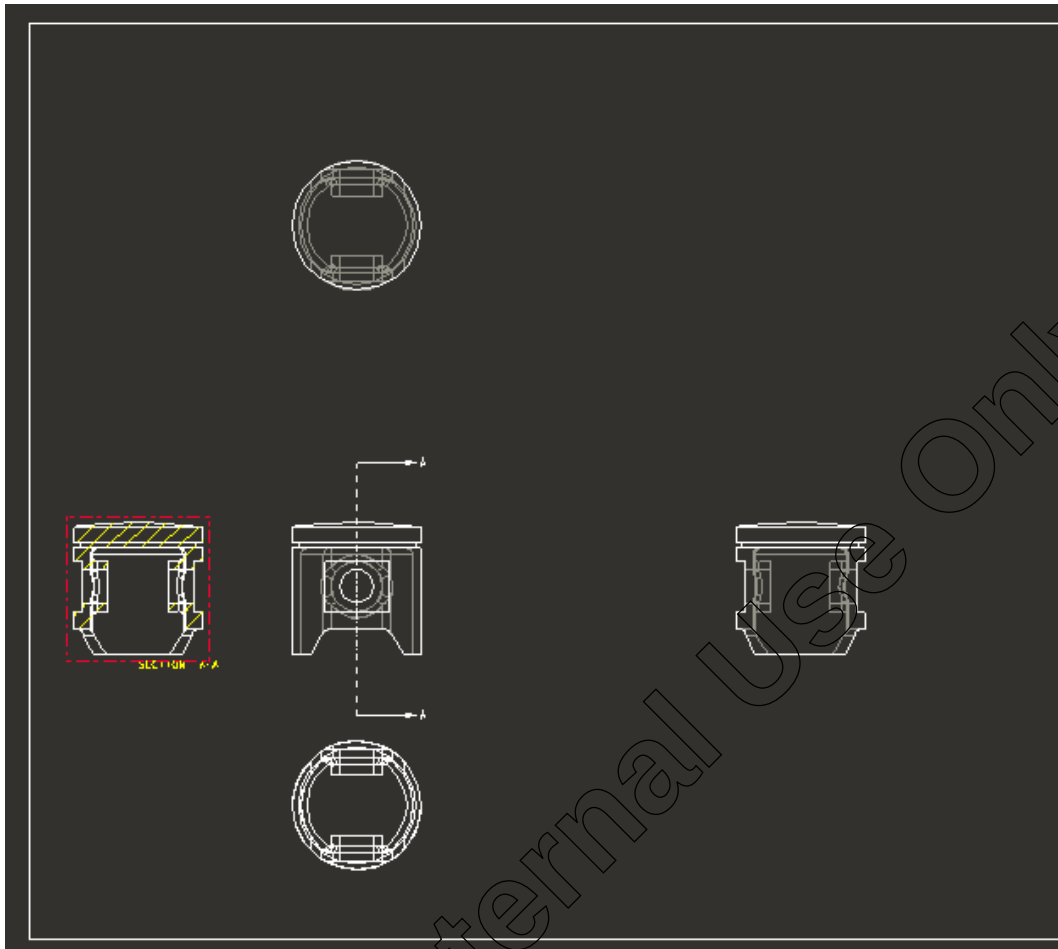


Figure 4: Creating Cross Section A

Step 4. Insert a Detail view.

1. Select on the background to de-select the previous view.
2. Right-click and select **Insert Drawing View**. Click **Detailed > Done**. Select a point to the left of the top view, as shown in the following figure.
3. Enter **5** for the scale. Zoom in to the ring groove cut on the right view. Select an edge of the cut (F12).
4. Select points to form a spline around the ring groove cut. Middle-click when finished.
5. Enter **1** for the name and click **Circle**. Select a location for the note above the right view.
6. Use the toolbar to refit the screen.

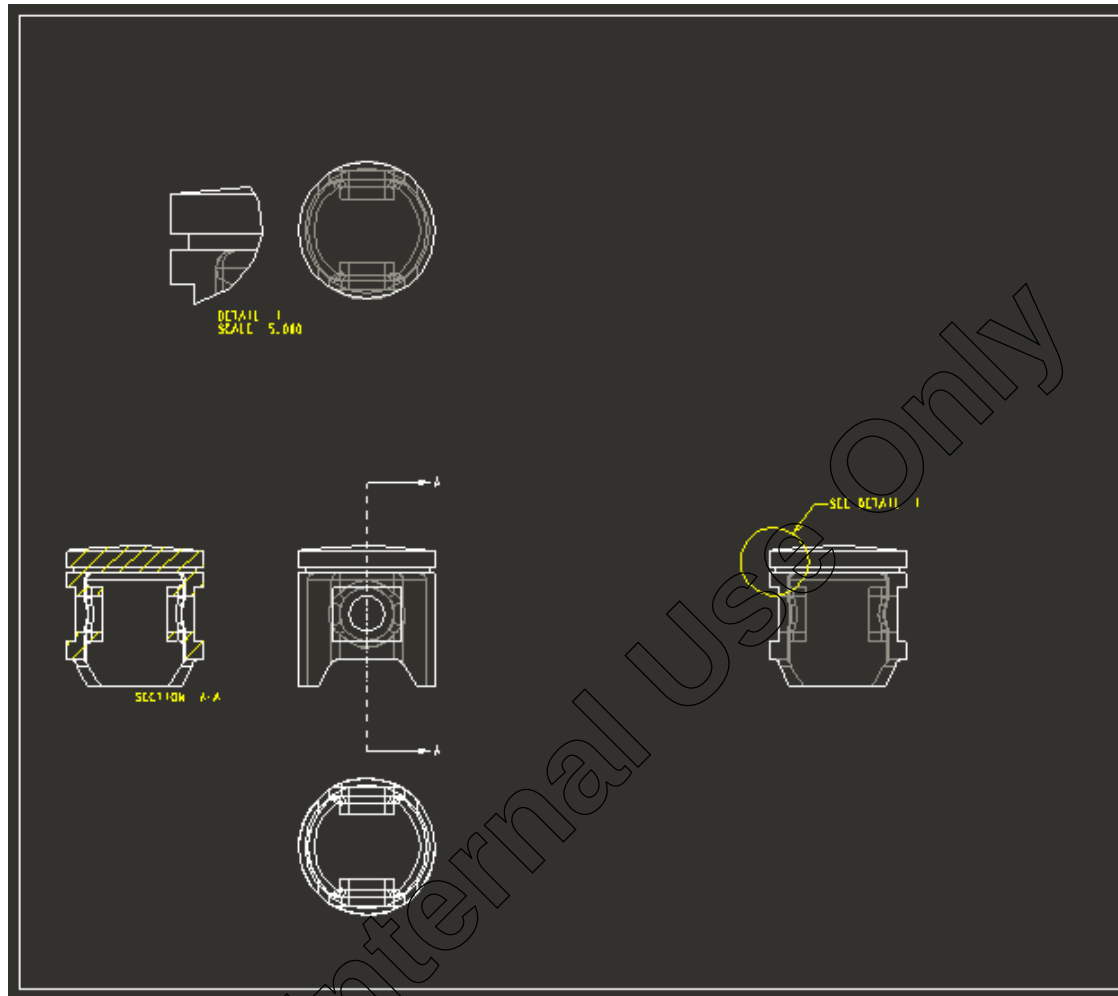


Figure 5. Inserting a Detailed View of the PISTON.PRT Ring

Step 5. Insert a General View.

1. Select anywhere on the background to de-select the previous view.
2. Right-click and select **Insert Drawing View**. Click **General > Scale > Done**. Select a point in the upper-right corner.
3. Enter **3** for the scale. Click **Saved Views** and select **3D**. Click **Set > Ok**. Refer to the following figure.

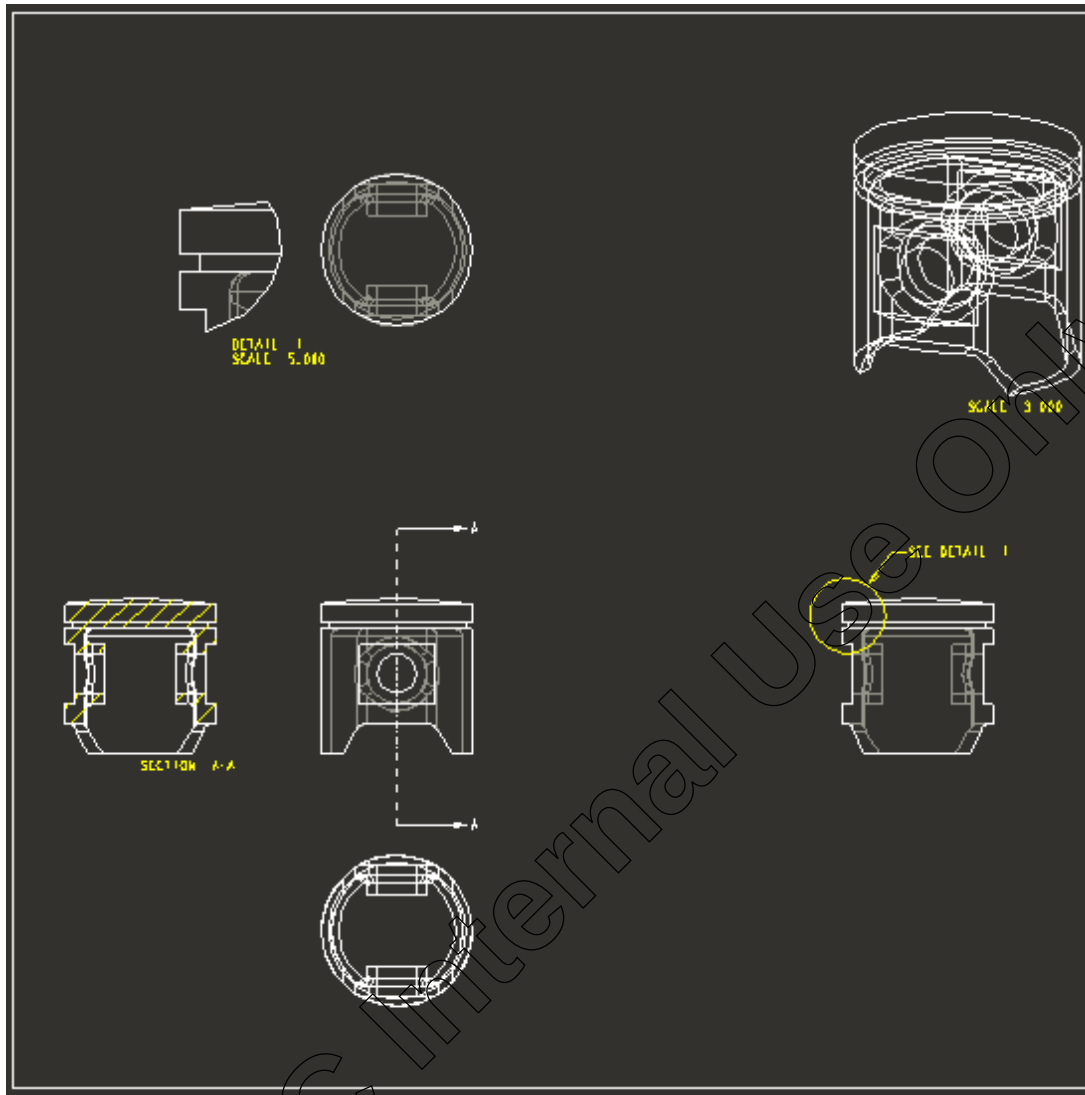


Figure 6: Inserting a Scaled General View

Step 6. Move views and change view display.

1. Right-click and uncheck **Lock View Movement**. Select and drag views to arrange as shown in the following figure.

Note:

Moving views and dimensions utilize a “click and release” then “click and drag” workflow.

2. Press and hold CTRL and select the cross-section, bottom, and general views. Right-click and select **Properties**. Click **View Disp > No Hidden > Done > Done**.
3. Select the cross-section view, right-click and select **Properties**. Click **View Disp > No Disp Tan > Done > Done**. Refer to the following figure.

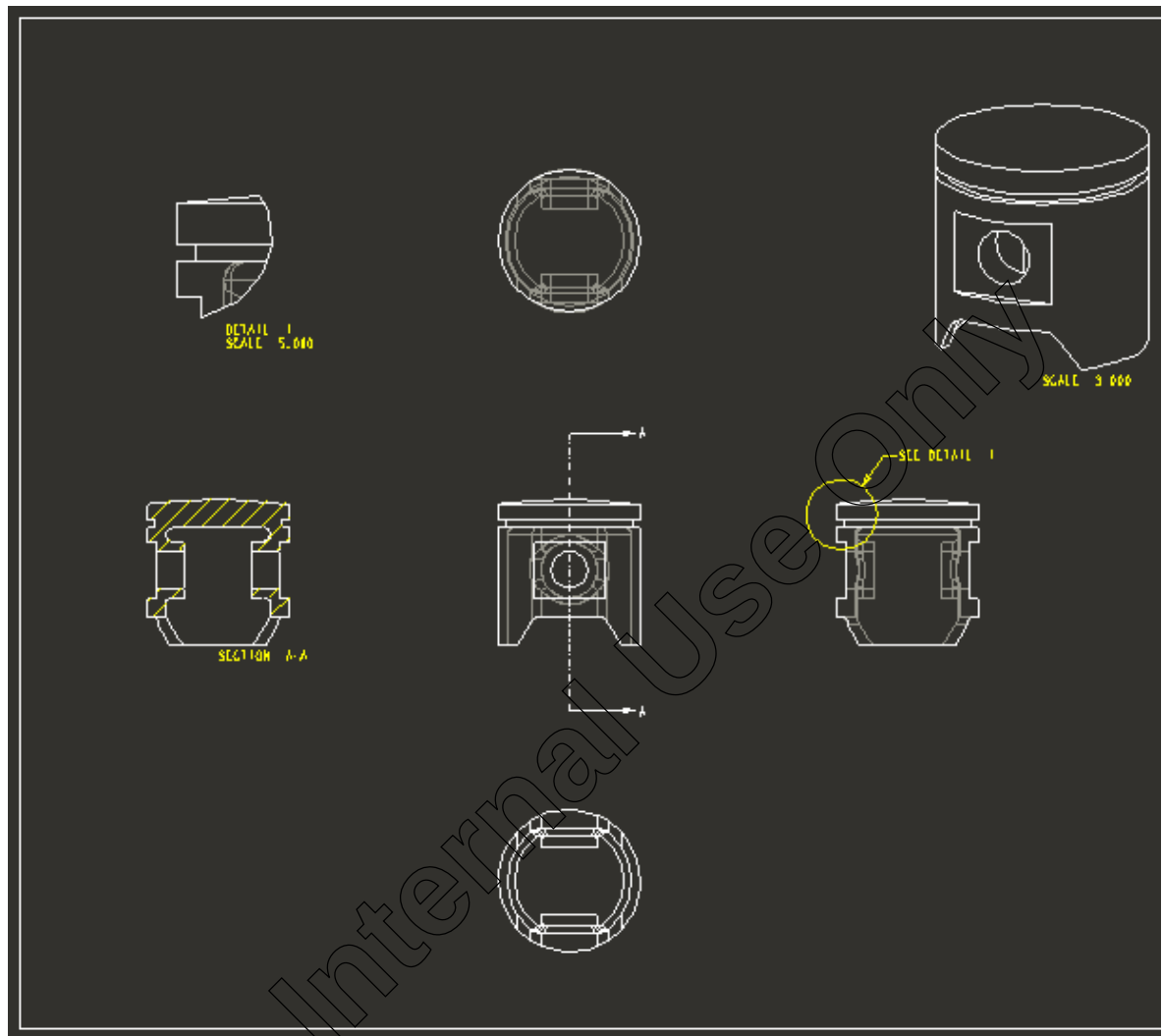


Figure 7. Changing the View Display of Bottom, Cross-section, and General Views

4. Save the drawing.

Step 7. Create a view of ENGINE.ASM.

Finally, we add a second sheet to the drawing; this one places a view of the exploded engine assembly.

1. Click **Insert > Sheet**.
2. Click **File > Properties > Drawing Models > Add Model**. Open ENGINE.ASM.
3. Right-click and select **Insert Drawing View**. Click **Done**.
4. Select a point near the upper right corner of the sheet as shown in the following figure.
5. Select the saved view EXPLODE_ORIENTATION and click **Set > Ok**.

6. Double-click the sheet scale and enter **1**.
7. Select the view and edit its properties. Click **View Disp > No Hidden > Model Color > Done > Done**.
8. Click **File > Properties > Drawing Options**.
9. Scroll to find the section entitled “These options control layers.”
10. Select the option for **ignore_model_layer_status** and set it to **No**.
11. Click **Add/Change > OK > Done/Return**.
12. Click **View > Update > Current Sheet** to update the view display, as shown in the following figure.

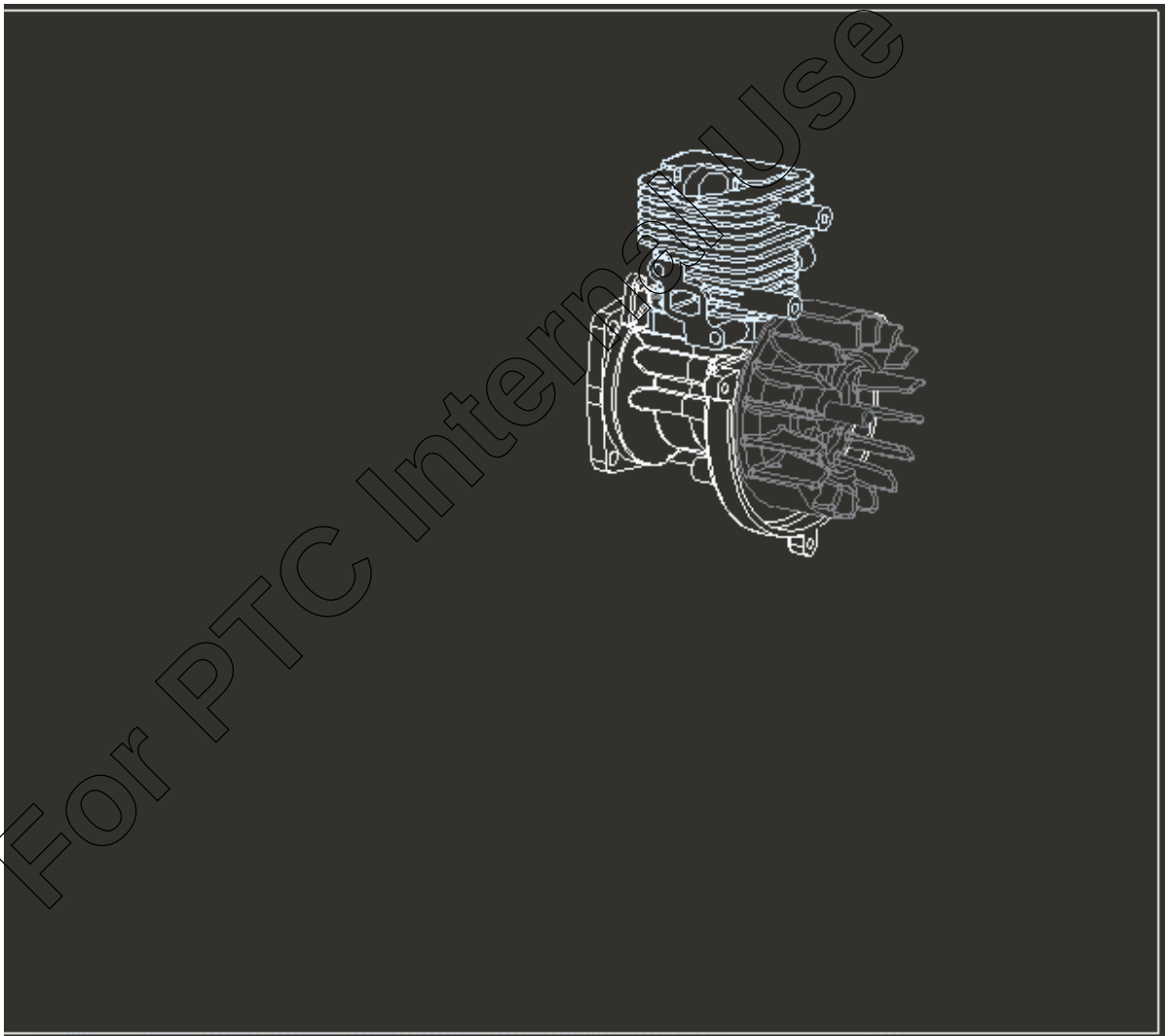


Figure 8: Adding the ENGINE.ASM View

13. Select the view and edit its properties. Click **View Type > Exploded > Done**. Select ENGINE_EXPLODE and click **Done > Done**.
14. Move the view as shown in the following figure.

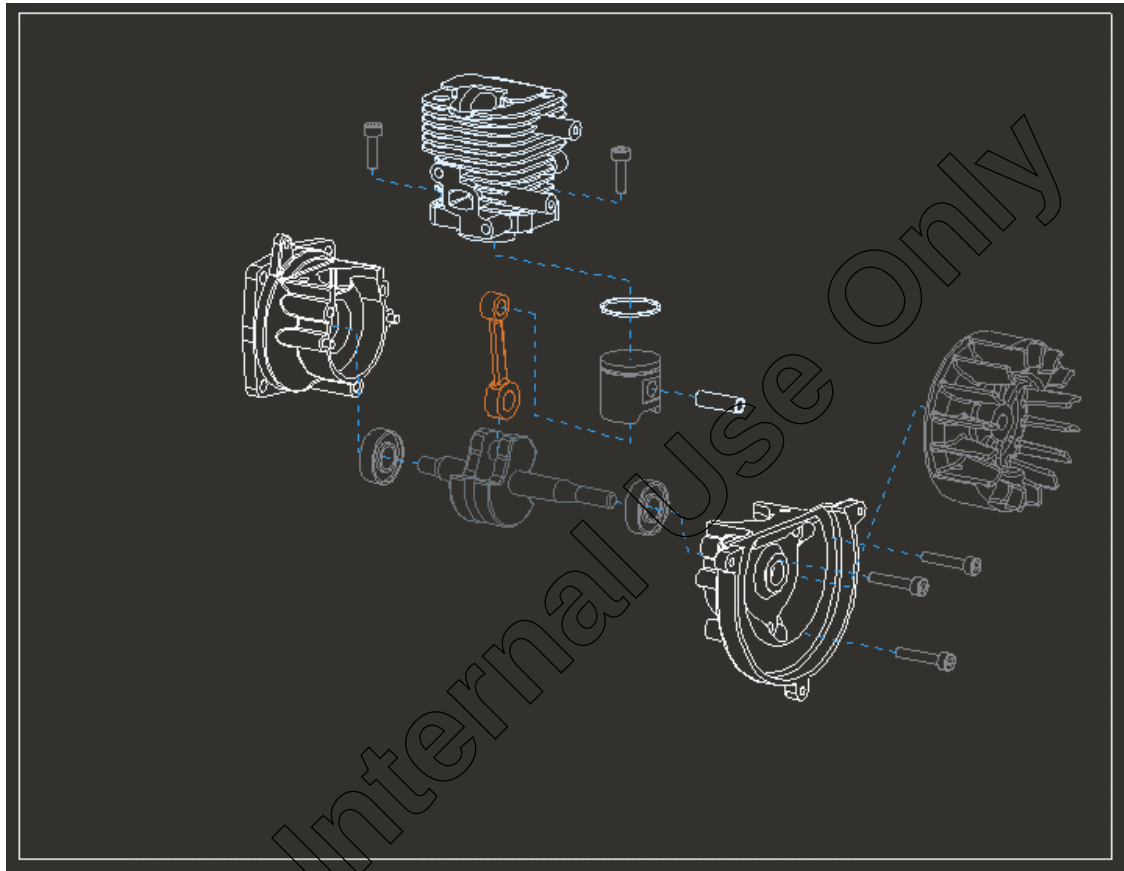


Figure 9: Exploding the ENGINE.ASM View

15. Save the drawing.
This completes the exercise.

Exercise 2: Detailing Drawings

Introduction

In this demonstration, we continue to work on the drawing created for the piston, handle bracket, and engine assembly. We do this by adding details to the drawing in the form of dimensions, notes, and a BOM.

Objectives

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

- Apply a format to a drawing.
- Add details to a drawing.

Scenario

Continue working on the piston and engine drawings and add the dimensional detail needed by Manufacturing. You need to add the piston dimensions that are necessary for machining. Additionally, you need to add a BOM and balloons that label each part the engine assembly.

Step 1. Apply a drawing format to Sheet 1.

In this first step, we apply a format to the drawing. The format will help identify us, our company, the part/assembly in the drawing, and other relevant information.

1. If necessary, open DRILL_COMPS.DRW
2. Use the toolbar to navigate to Sheet 1.
3. Use the toolbar to set PISTON to be the current drawing model.
4. Double-click on the sheet size to access the Page Setup dialog box.
5. Browse for a new format for Sheet 1. Look in the **module_14** folder and open **c_format_generic.frm**.
6. Click **OK** and enter your name at the prompt. Refer to the following figure.

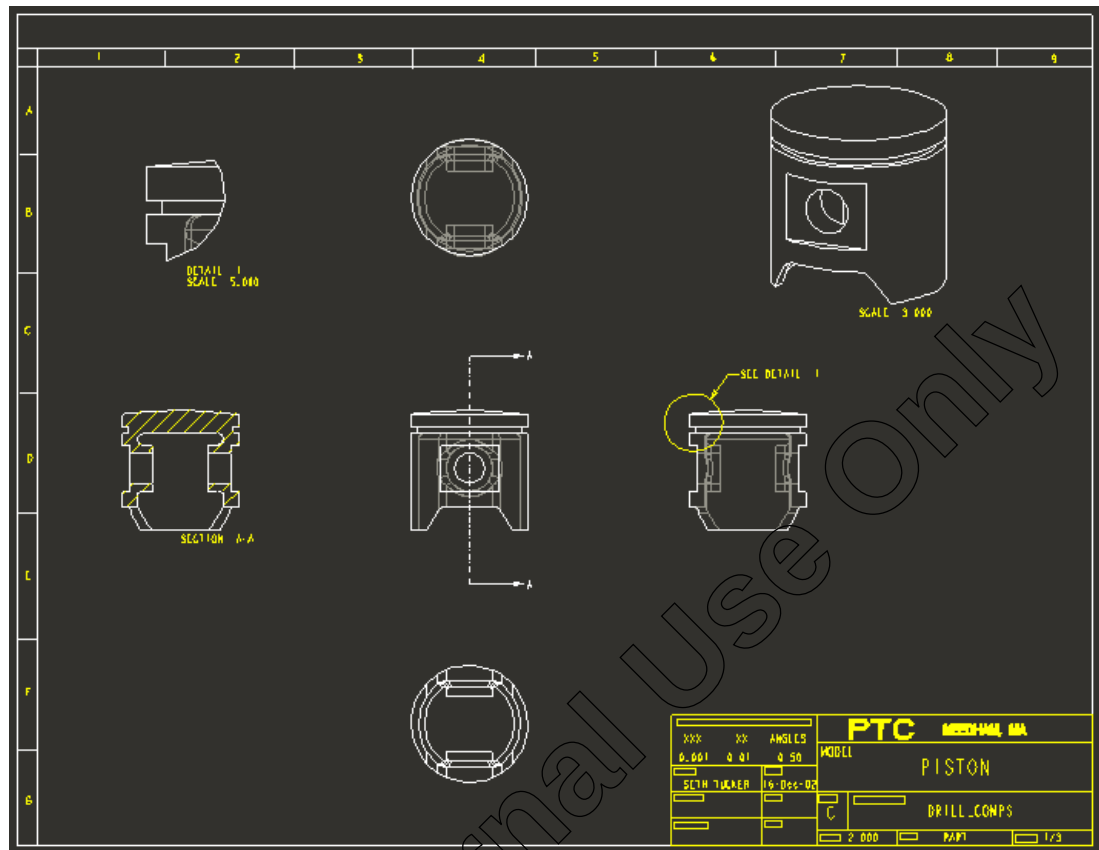


Figure 10: Applying a Drawing Format to Sheet 1

Note:

You are placing a format file (*.frm) on the drawing. The drawing Title Block is set up to automatically fill in several values, such as the part name. Other information, such as your name, must be entered manually.

Step 2. Apply a drawing format to Sheet 2.

Instructor: Skip this step for the demo.

1. Navigate to Sheet 2. Use the toolbar to set ENGINE to be the current drawing model.
2. Double-click on the sheet size to access the Page Setup dialog box.
3. Browse for a new format for Sheet 2.
4. Look in the **module_14** folder and open **c_format_generic.frm**.
5. Click **OK** and enter your name at the prompt. Edit the sheet scale to **0.75**. Refer to the following figure.

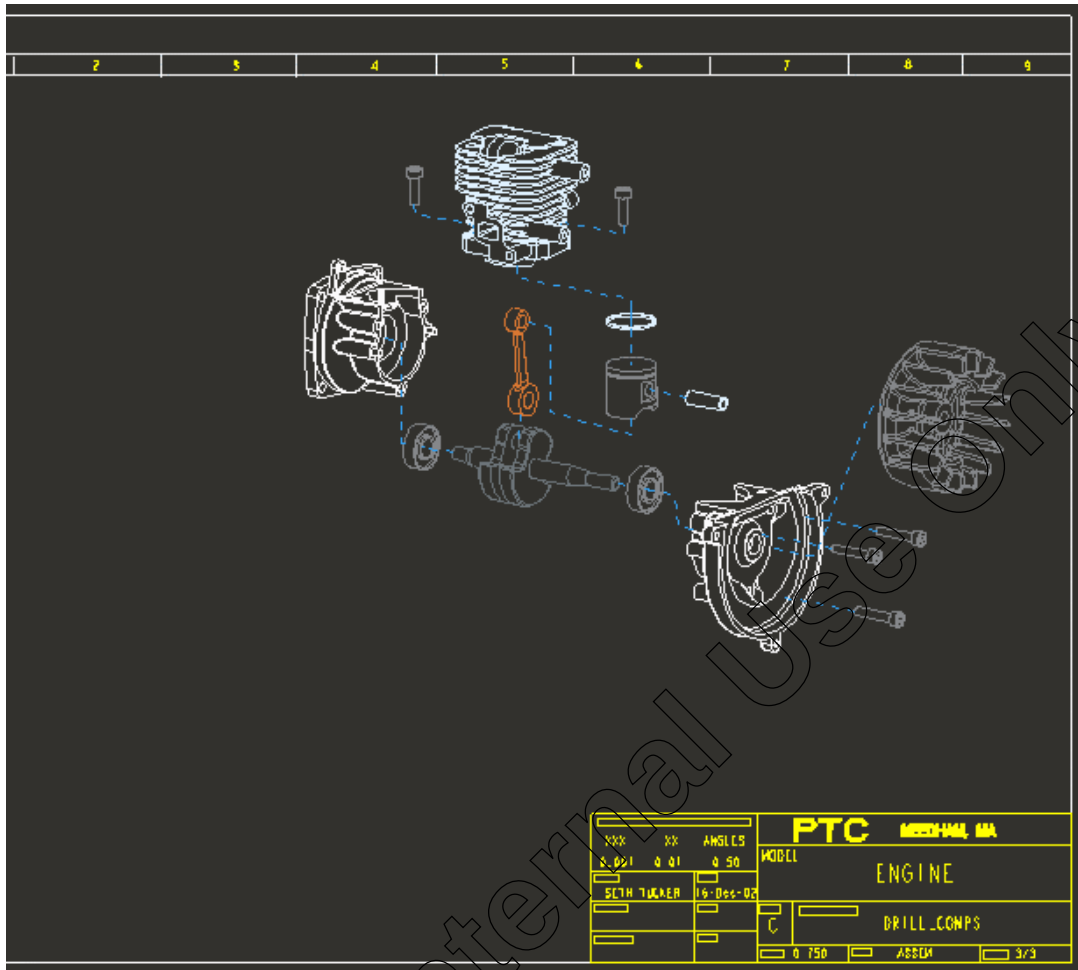


Figure 11: Applying a Drawing Format to Sheet 3

6. Save the drawing.

Step 3. Show dimensions on the PISTON.PRT.

In this step, we add dimensions to the piston so that manufacturers can machine it from either cast or solid stock materials. Note that we can display dimensions without having to redraw the part – the information is taken directly from the 3-D model. We can display all dimensions at once or we can select specific features or views to add dimensional detail. We can also format the drawing's dimensions in order to give it an organized appearance.

1. Navigate to Sheet 1 and set PISTON.PRT to be the current drawing model.
2. Click **View > Show and Erase**. Select the Dimension icon and click **Show All > Yes**, as shown in the following figure.

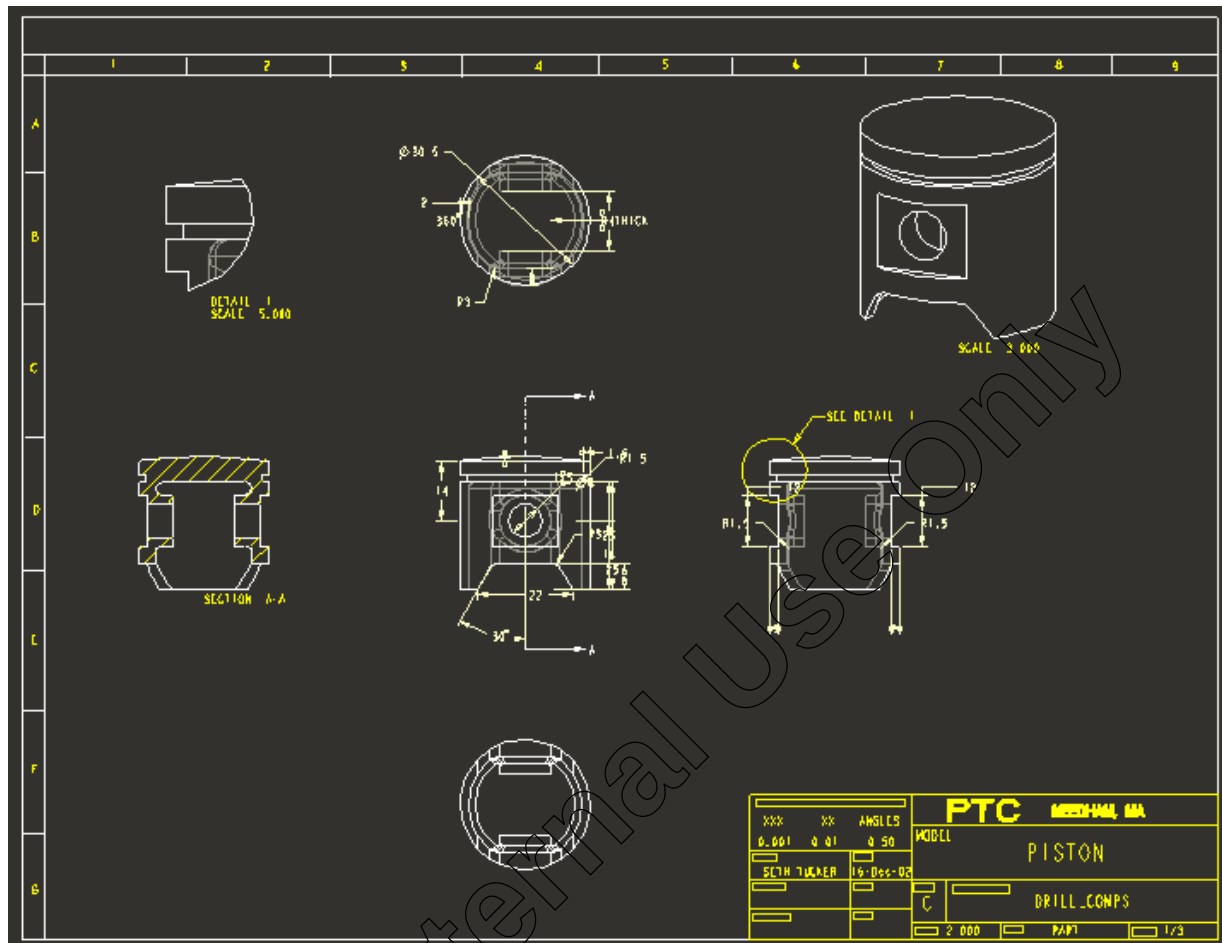


Figure 12: Showing All Dimensions

3. Click **Erase All**.
4. Show by **Feature** and select the ring groove cut (F12) in the detail view. Note that the dimensions are displayed in various views, as shown in the following figure.

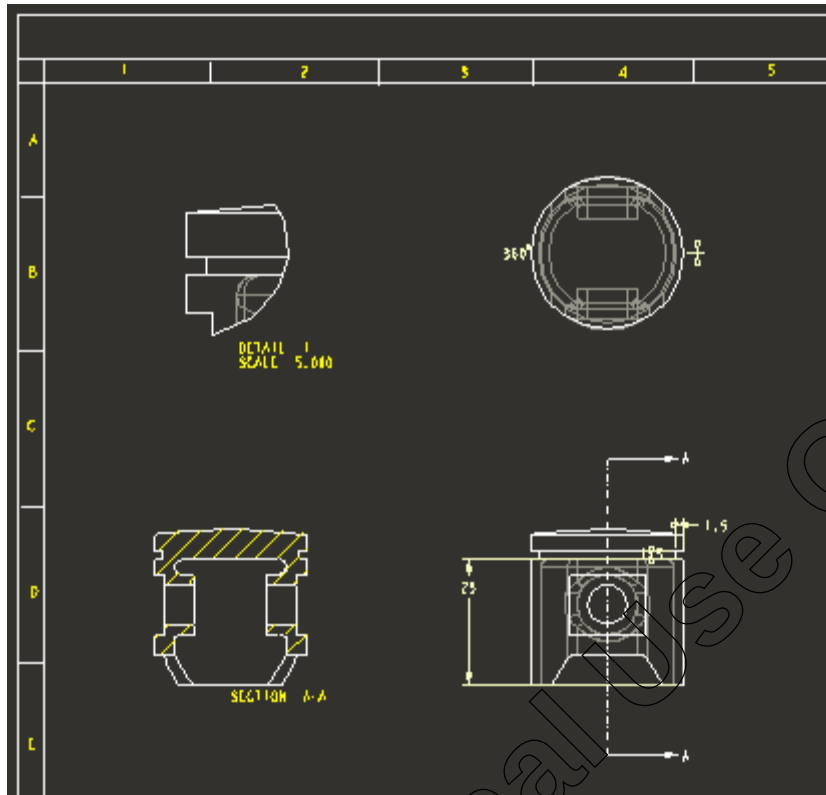


Figure 13: Showing All Ring Groove Dimensions

5. Click **OK > Erase All**.
6. Show by **Feature and View** and select the ring groove cut (F12) in the detail view. Note that the dimensions are displayed in this view only, as shown in the following figure.
7. Click **OK > Sel to Keep**. Press and hold CTRL, select the two **1.5** dimensions, and then click **OK**. Refer to the following figure.

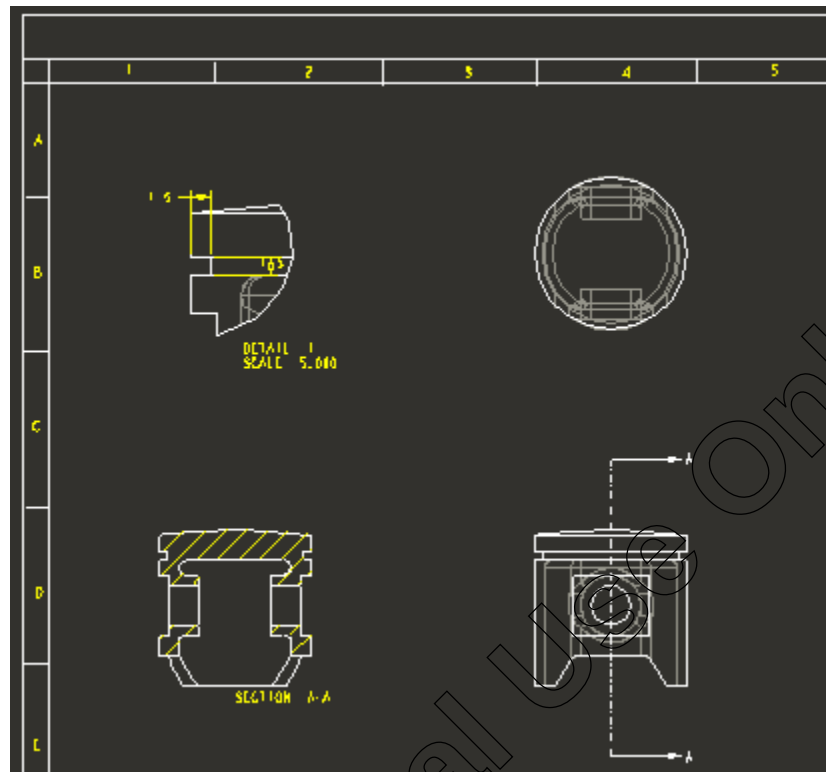


Figure 14: Showing the Ring Groove Dimensions on the Detailed View

8. Select the cut feature (F16) in the bottom view, as shown in the following figure.
9. Click **OK > Accept All**.

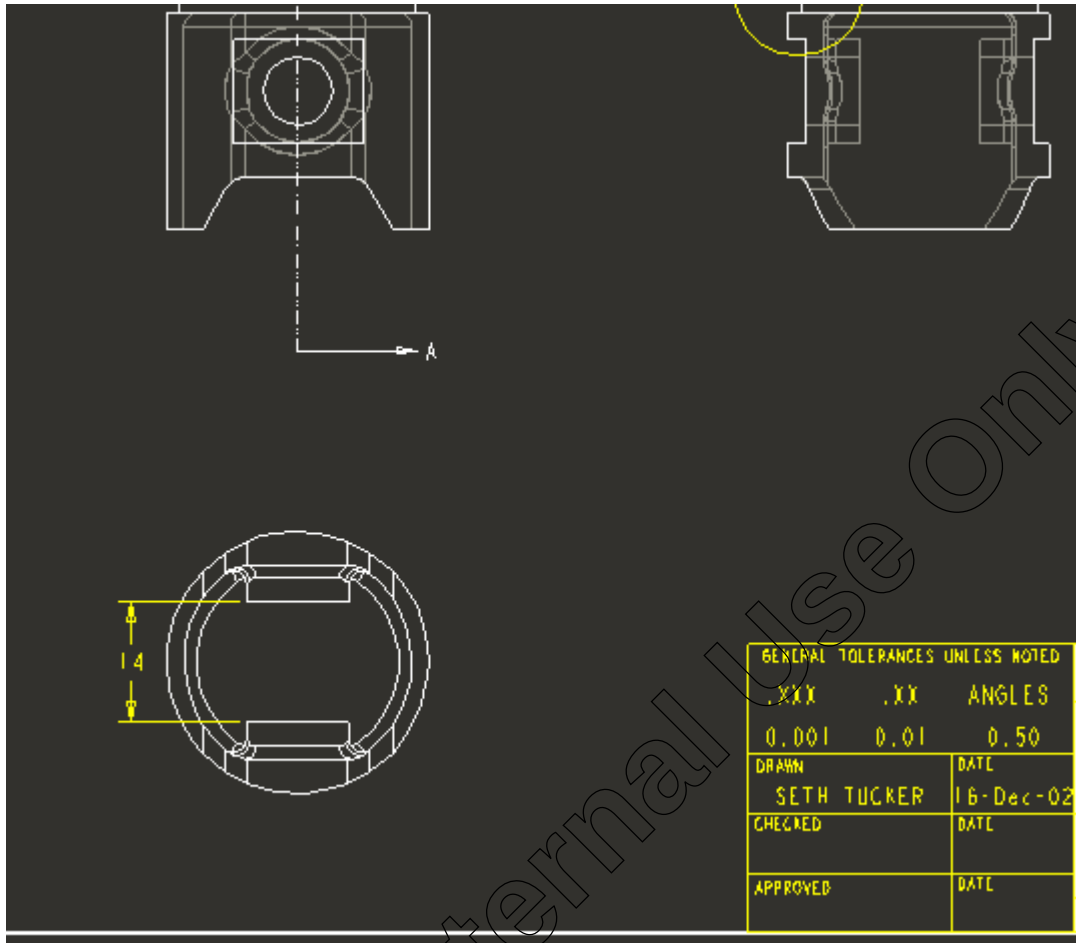


Figure 15: Showing Cut F16 on the Bottom View

10. Show by **View**, and select the front view as shown in the following figure.

11. Click **Ok > Accept All**.

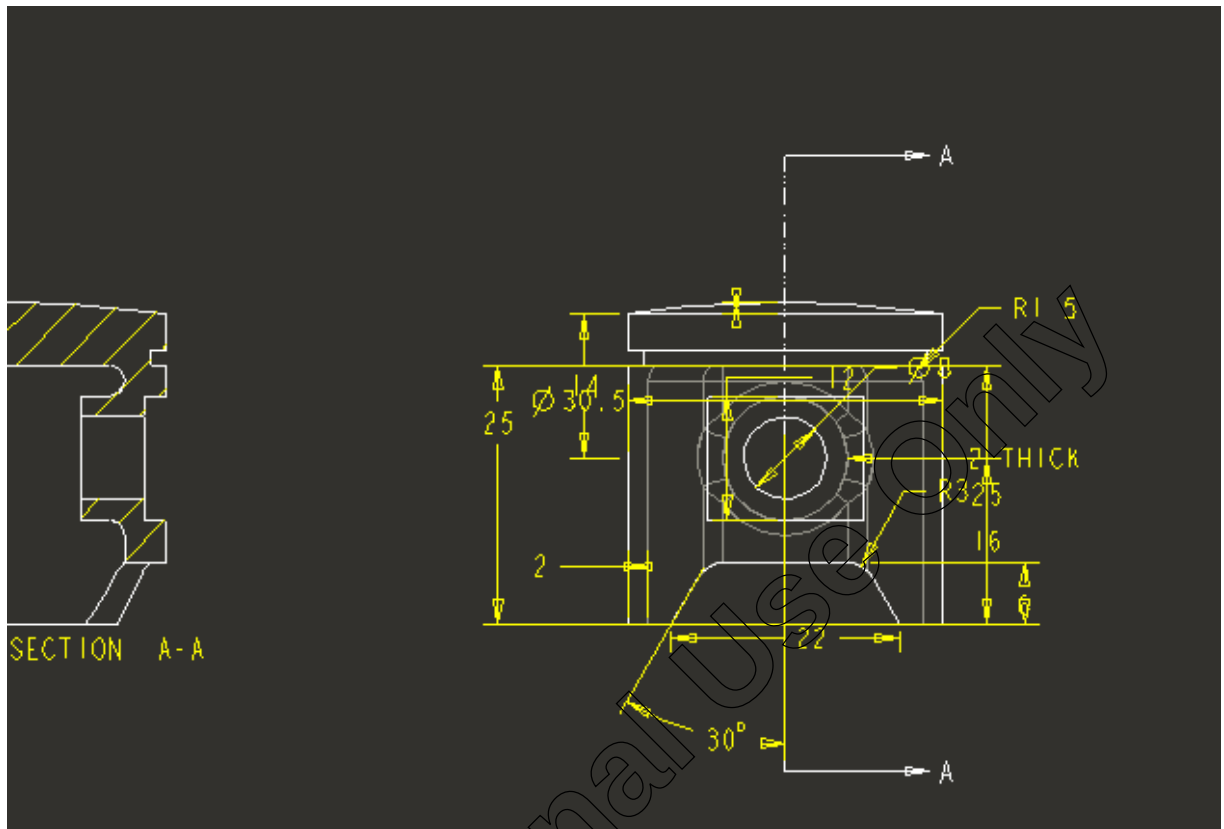


Figure 16: Showing All Dimensions on the Front View

12. Click **Show All > Yes > Accept All**. Refer to the following figure.

13. Close the Show/Erase dialog box.

14. Select on the background to de-select all dimensions.

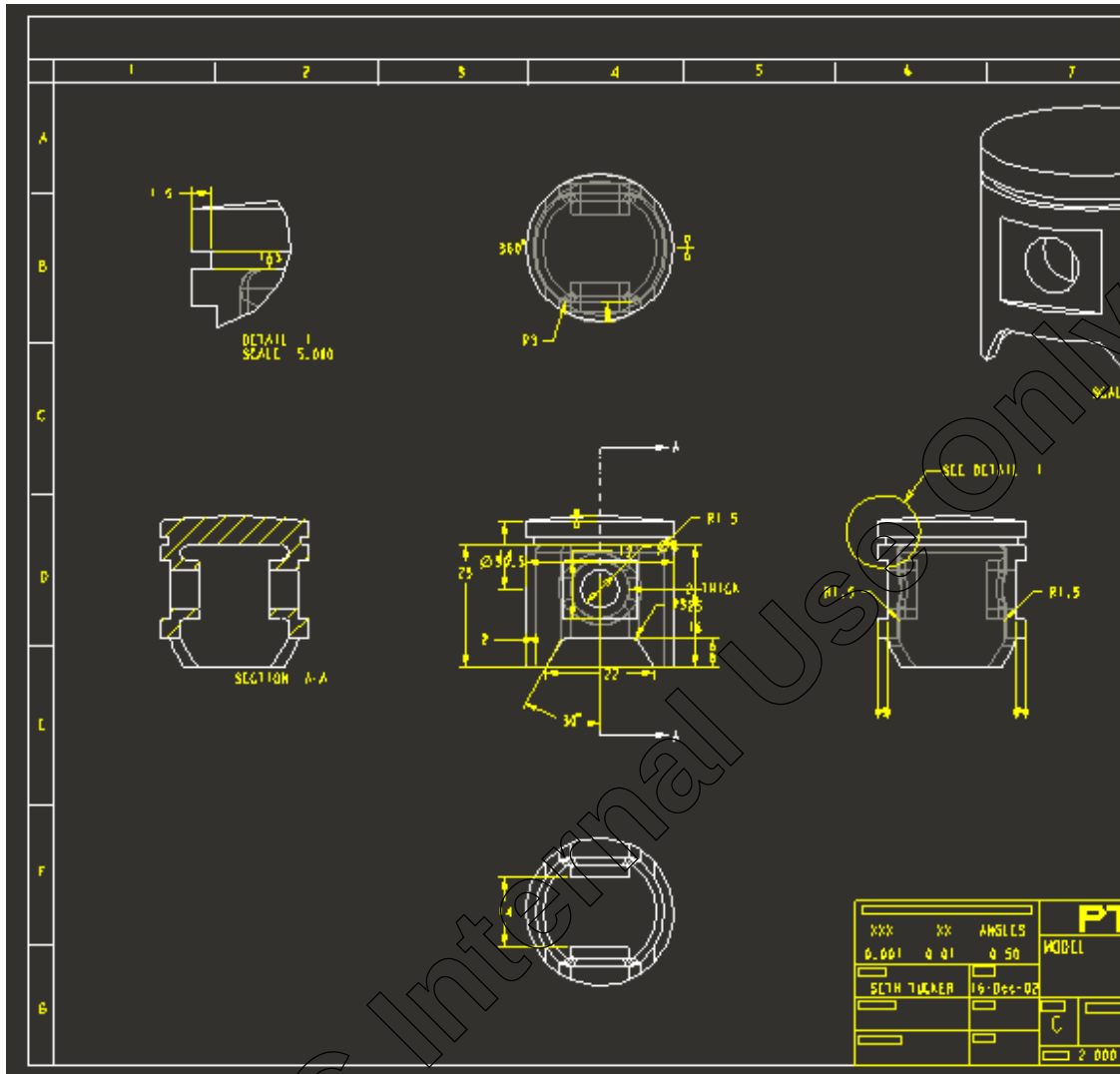


Figure 17: Showing All Drawing Dimensions

Step 4. Cleanup dimensions, and create a 'driven' dimension.

1. Drag a window to select all dimensions in the front, top, and detail views.
2. Click **Edit > Cleanup > Dimensions**. Clear the **Create Snap Lines** checkbox, and click **Apply > Close**.
3. Select on the background to de-select all dimensions. Refer to the following figure.

4. Click **Insert** > **Dimension** > **New Reference** and select the edges shown in the following figure to create a vertical **12** diameter dimension. Click **Return** when finished.

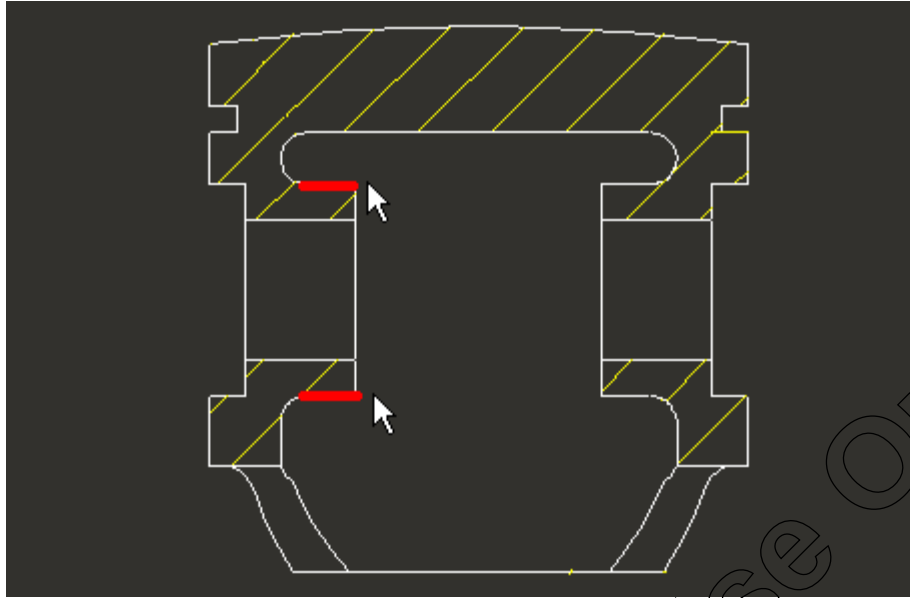


Figure 19: Selecting References for Diameter Dimension

5. Select the **12** dimension, and access its properties. Select the **Dimension Text** tab and select the $\varnothing$ text symbol for the Prefix.

Step 5. Use various detail options to manually arrange the dimensions.

Instructor: Quickly demonstrate this step, showing one example of each.

1. Manually cleanup the drawing, as shown in the following figure:
 - Select and drag views and dimensions.
 - Select a dimension, and drag its handles to move dimension lines or text.
 - Select a dimension, right-click and select **Flip Arrows**.
 - Select a dimension, right-click, select **Move Item to View**, and select a view.
 - Select a dimension, right-click, select **Erase**, and select on the background.
 - Use CTRL to select two dimensions, right-click and select **Align Dimensions**.

Notes:

Drawing dimensions that are underneath other dimensions can be selected by using a “cursor over, right-click to query, and then select” workflow.

There are multiple **12** and **25** dimensions that belong to completely different features.

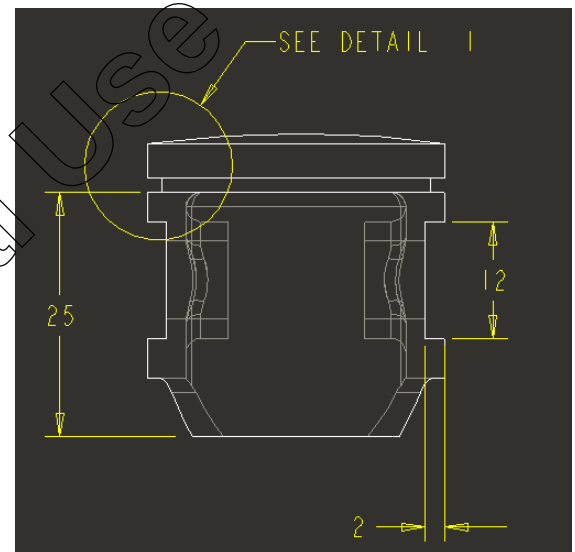
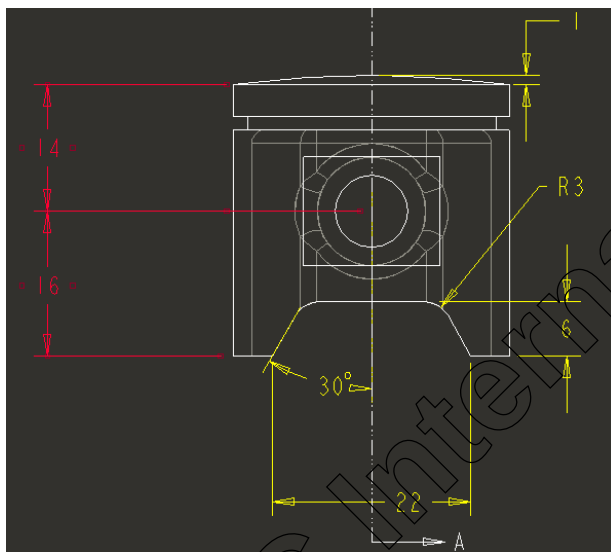
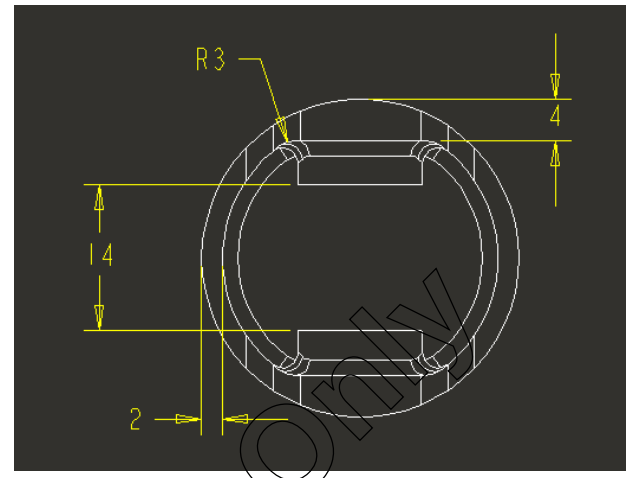
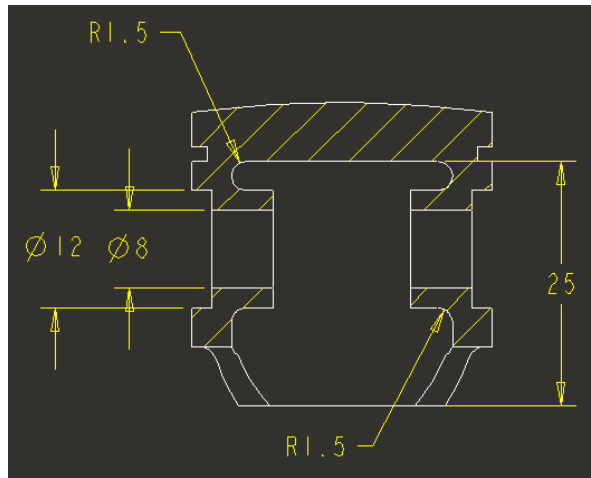


Figure 20: Manually Detailed Drawing

Step 6. Show the axes.

1. Click **View > Show and Erase**. De-select the Dimensions icon, and select the Axes icon.
2. Click **Show All > Yes > Sel to Keep**.
3. Press and hold CTRL and select the axes shown in the following figure. Click **OK**.
4. Close the Show / Erase dialog box.

Step 7. Create a note and edit dimensions.

- The **&d41:0** syntax stands for **&dimension_symbol:component_symbol**.

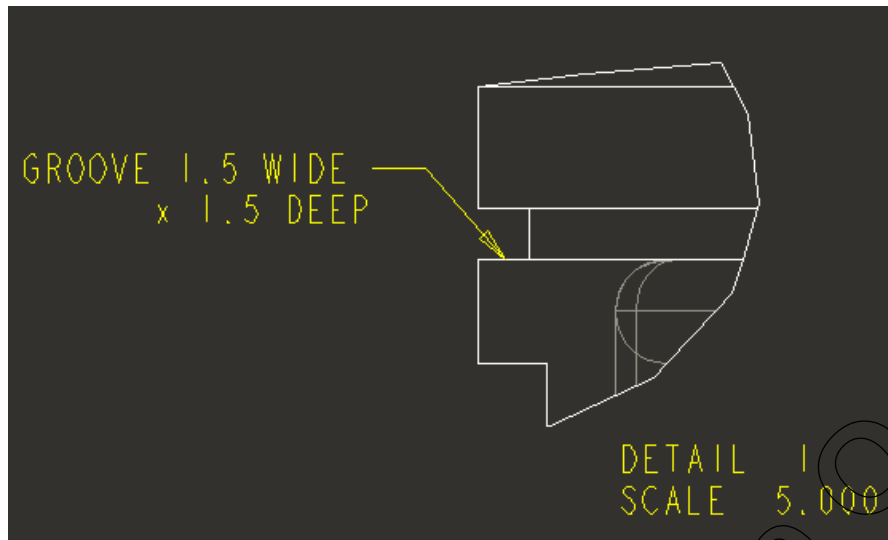


Figure 23: Adding a Detailed Note to the Drawing

6. Edit the dimensions in the note to **2 wide** and **1 deep**.
7. Edit the groove height dimension from **25** to **26.25**.
8. Regenerate the model. Notice the views update, as shown in the following figure.

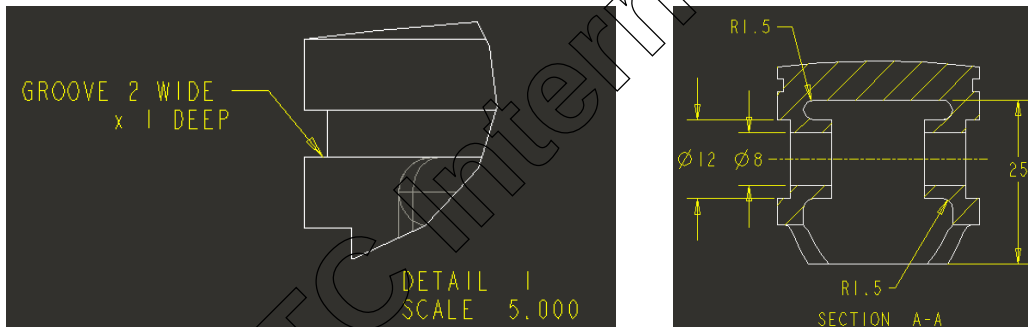


Figure 24: Editing the Piston Dimensions

Note:

If you open PISTON.PRT you will also see that the dimensions have updated due to Pro/ENGINEER Wildfire's bi-directional associativity.

9. Save the drawing.

Step 8. Create a BOM and balloons for the ENGINE.ASM.

In this step, we add detail to the engine assembly drawing by importing a BOM table. BOM tables are important because they list the number of parts and their locations in an assembly. Note that the BOM table we import is parametric, which means that it self updates whenever we add or delete a part to the assembly.

1. Navigate to Sheet 2.
2. Set ENGINE.ASM as the current drawing model.
3. If necessary, modify the sheet scale to **0.75**.
4. Click **Table > Insert > Table from File**. Open BOM_TABLE.TBL and place it in the upper-left corner of the drawing, as shown in the following figure.

ITEM	NAME	QTY
1	BOLT_5-18	2
2	BOLT_5-28	3
3	CONNECTING_ROD	1
4	CRANKSHAFT	1
5	CYLINDER	1
6	ENG_BEARING	2
7	ENG_BLOCK_FRONT	1
8	ENG_BLOCK_REAR	1
9	FLYWHEEL	1
10	PISTON	1
11	PISTON_PIN	1
12	PISTON_RINGS	1

Figure 25: Inserting a Bill of Materials Table

Note:

This parametric table was created with repeat regions. Repeat regions are table cells that contain parametric notes; these enable the notes to automatically update if components are assembled, deleted, renamed, suppressed, and so on.

5. Click **Table > BOM Balloons > Set Region**. Select in the center of the table.
6. Click **Create Balloon > Show All**. Drag and arrange the balloons, as shown in the following figure.

Note:

You can also use **Edit > Cleanup > BOM Balloons** to organize the balloons.

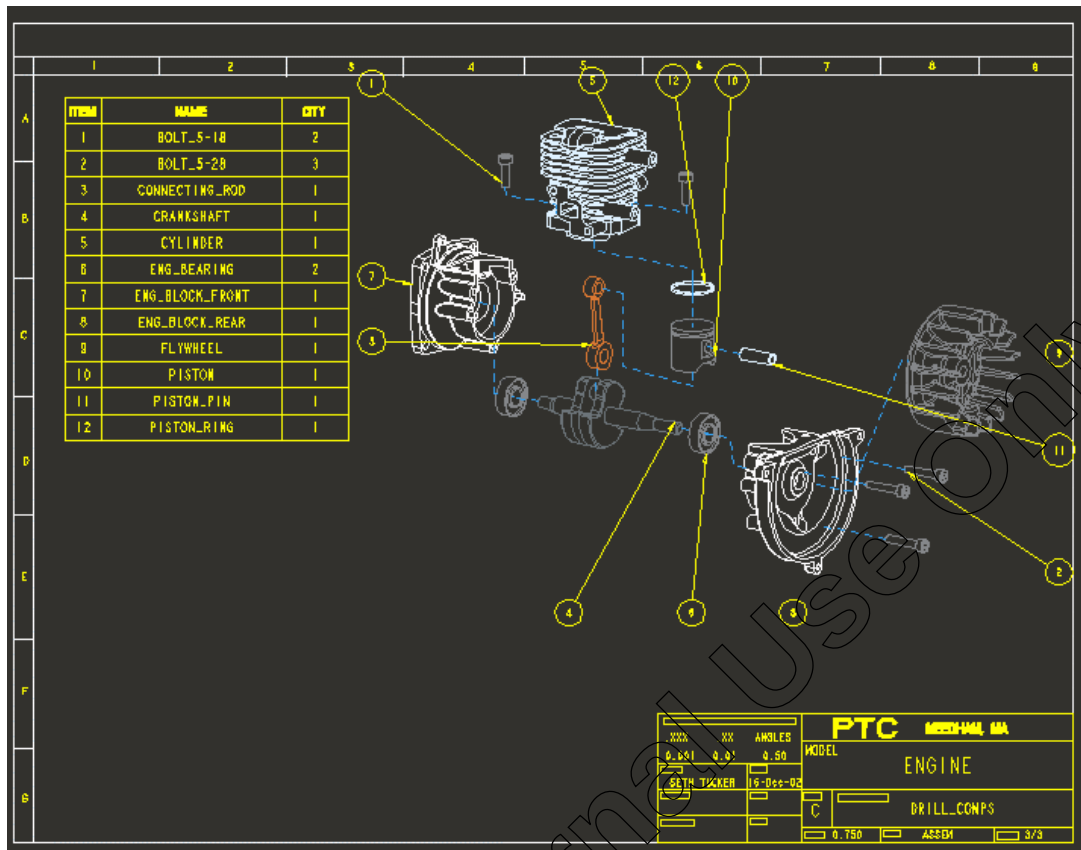


Figure 26: Inserting BOM Balloons

Step 9. (OPTIONAL) Resume a component and update the BOM.

Instructor: Skip this step for the demo.

1. Open ENGINE.ASM.
2. Resume the SPARK_PLUG.PRT.
3. Disable transparency.

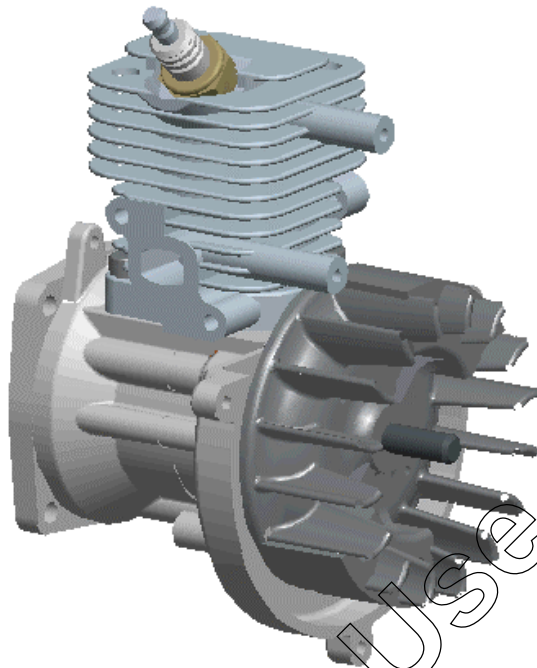


Figure 27: Assembling the SPARK_PLUG.PRT

4. Use the View Manager to activate the explode state named ENGINE_EXPLODE.
5. Click **View > Explode > Edit Position**. Click **Copy Pos**, and select the CYLINDER.PRT and then the SPARK_PLUG.PRT. Repaint the screen.
6. Zoom in to SPARK_PLUG.PRT. Click **Translate** and select **Plane Normal**. Select the sloped surface under SPARK_PLUG.PRT, and explode it as shown in the following figure. Click **Ok**. Repaint the screen.
7. Click **View > Explode > Offset Lines > Create**. Create an offset line for the SPARK_PLUG.PRT, as shown in the following figure.

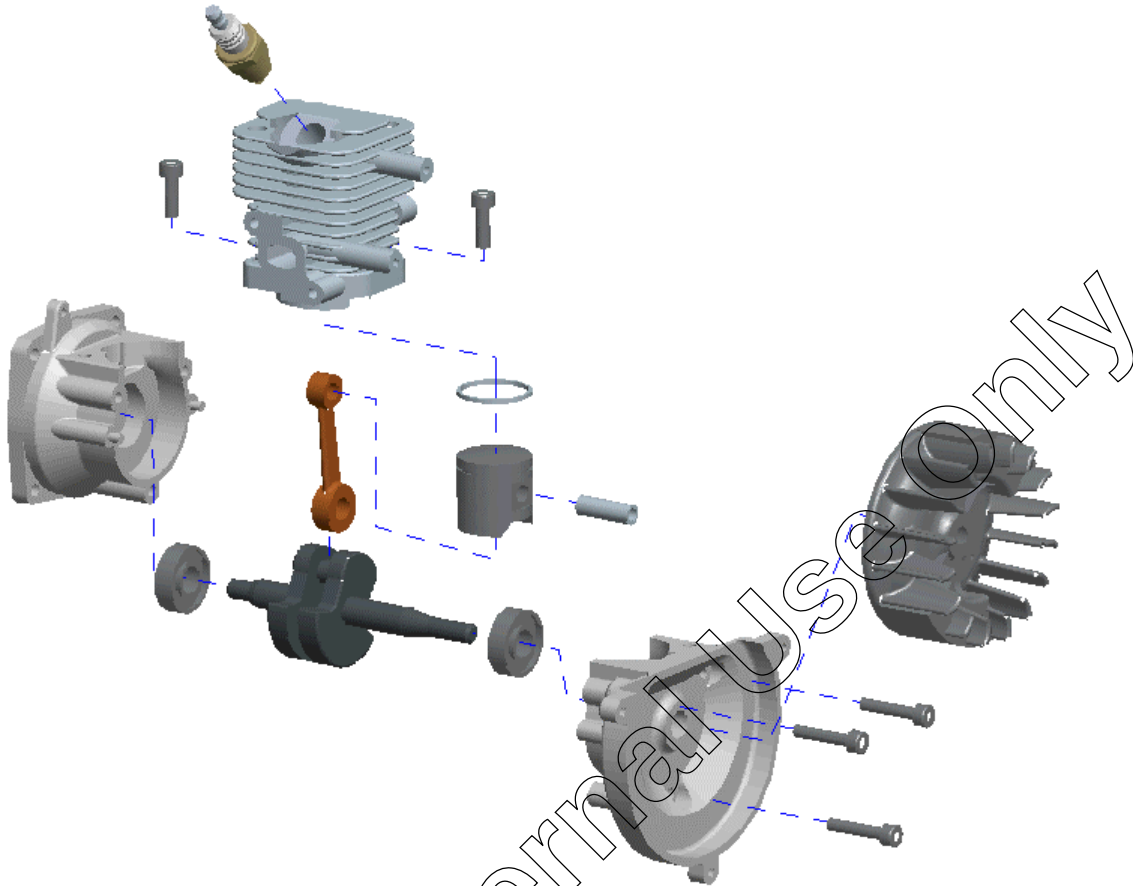


Figure 28: Creating a New Explode State

8. Use the View Manager to save a new explode state called **Exp_2**. Close the View Manager.
9. Open DRILL_COMPS.DRW.
10. Note that the BOM table has automatically updated.
11. Select the view and edit its properties.
12. Click **Mod Explode** and double-click **Exp_2**. Click **Close**.
13. Click **Table > BOM Balloons > Create Balloon > By Comp** and select SPARK_PLUG.PRT. Click **Ok** and move the balloon as desired. Refer to the following figure.

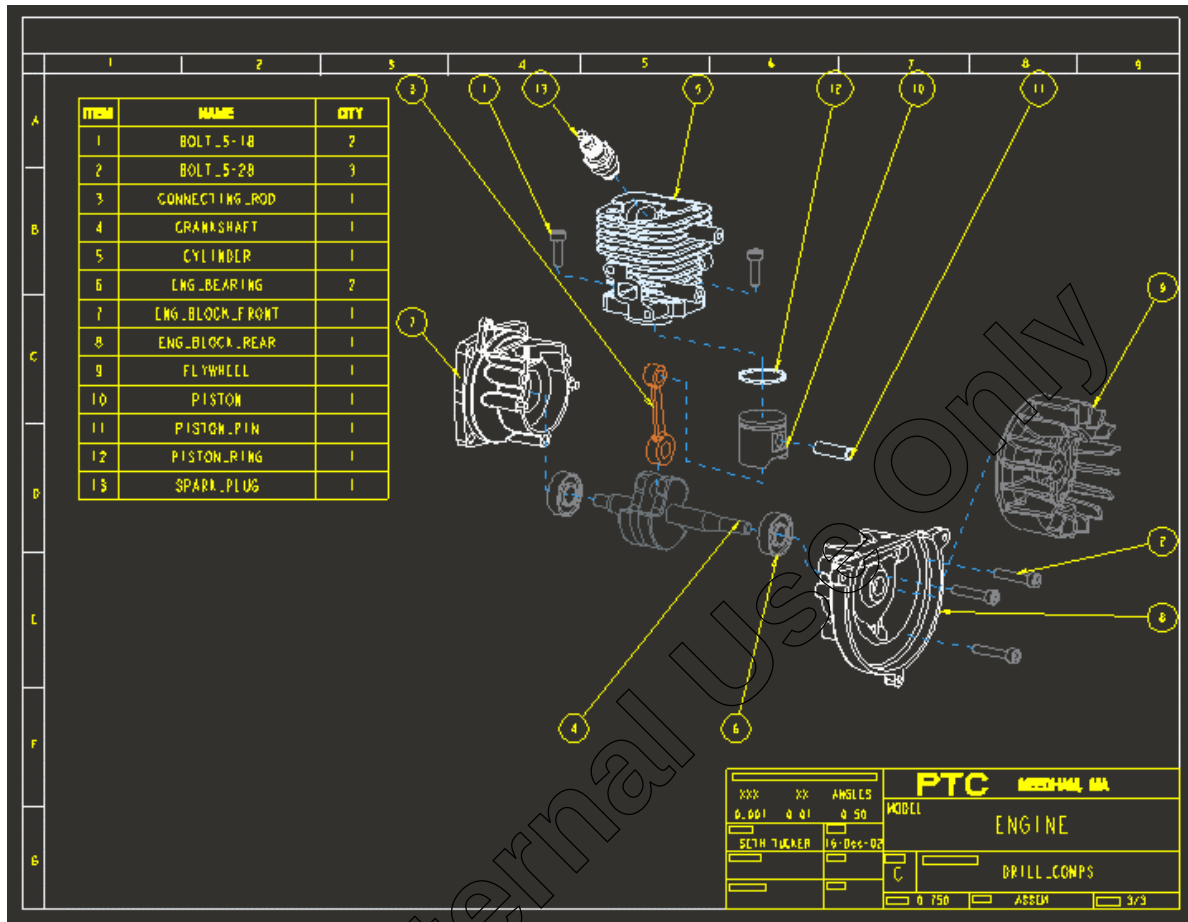


Figure 29: Updating the Drawing

14. Save the drawing.
15. Close all windows.
16. Erase all models from memory.

This completes the exercise.

Summary

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

- Create drawings of design models.
- Detail drawings with dimensions and notes.
- Describe how creating drawings affects parent/child relationships.

For PTC Internal Use Only

Resolving Regeneration Failures

Introduction

When you use features as design model “building blocks,” you create several references and parent-child relationships between the features. By using references and relationships, you can capture design intent and create robust, parametric models.

In this module, you learn how to obtain information about the references created between components, as well as between features. You also learn how to resolve regeneration failures that are caused by inadvertently created references and relationships. Finally, you learn how to resolve regeneration failures that can result from editing parts, assemblies, and drawings.

Objectives

After completing this module, you will be able to:

- Obtain design information in part and assembly models.
- Resolve regeneration failures in parts, assemblies, and drawings.
- Describe how parent/child relationships affect regeneration failures.



Instructor Preparation

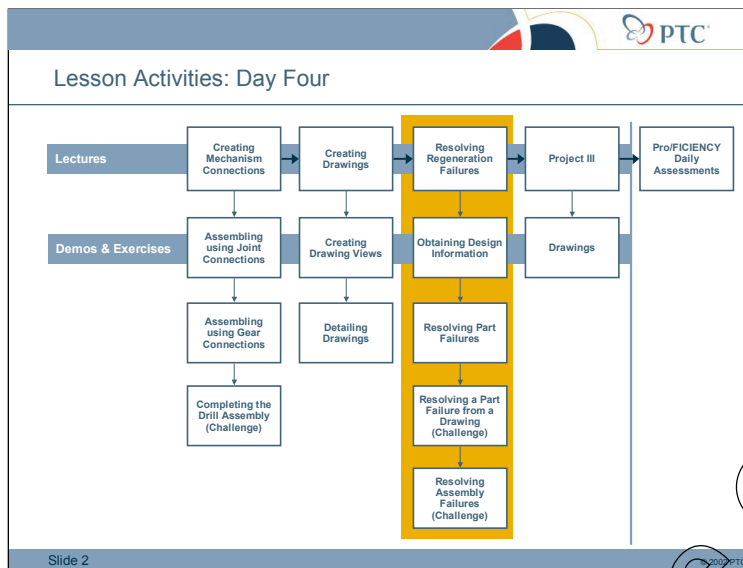
Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** the following materials:

- 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 Foundation

- Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (2): 40 mins
- Labs (2): 40 mins
- Total: 1 hr 30 min



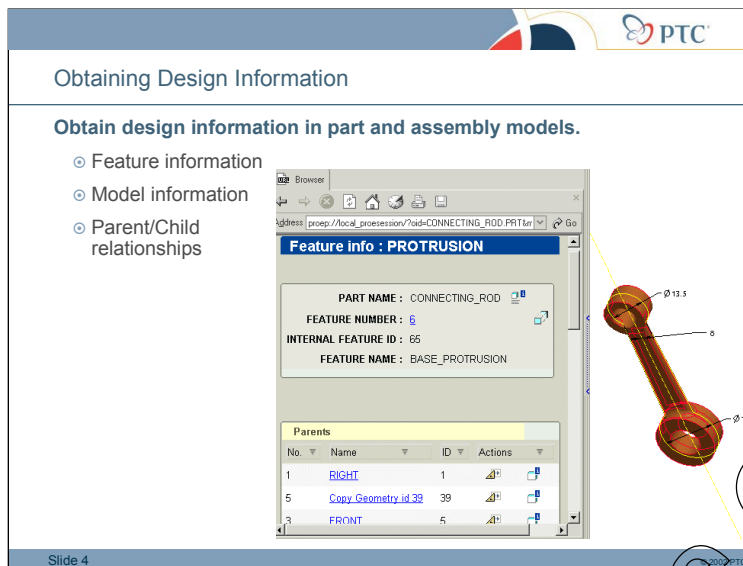
Objectives

After completing this module, you will be able to:

- Obtain design information in part and assembly models.
- Resolve regeneration failures in parts, assemblies and drawings.
- Describe how parent/child relationships affect regeneration failures.

Slide 3

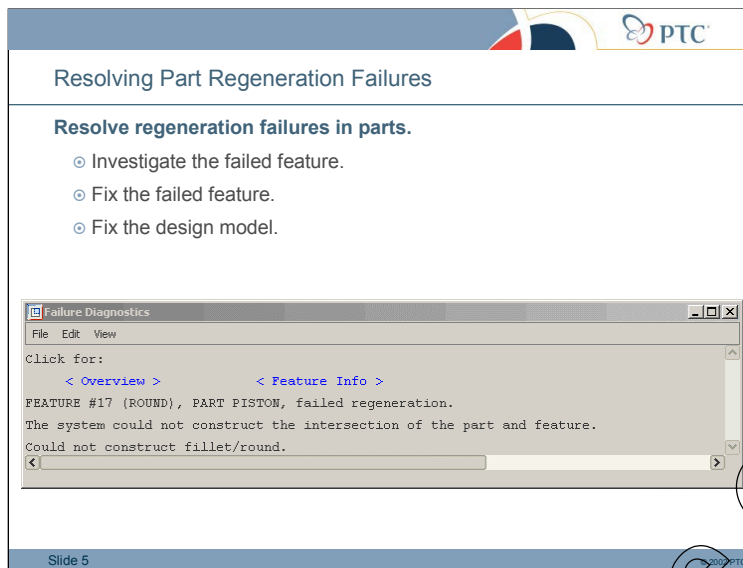
- When you use features as the 'building blocks' of a design model, you create several references and parent-child relationships between the features.
- By using references and relationships, you can capture your design intent, and create robust, parametric models.
- In this module, you learn how to obtain information about the references created between components, as well as features.
- You also learn how to resolve regeneration failures that are caused by inadvertently created references and relationships.
- Finally, you learn how to resolve regeneration failures that can result from editing parts, assemblies and drawings.



- Features contain information that describes how they are created.
- This information includes dimensions, sketches, parents, and children, as well as other information.
- Models also contain information that describes how they are created.
- This information includes all features within the model tree, model name, units, cross sections, and reference assemblies.
- Feature and model information can be examined at any time.
- This information is particularly useful when features or models fail during regeneration.
- By examining this information, you can determine what elements within a feature or model need to be fixed to resolve the failure.

Benefit:

- By examining feature and model information, you can determine what elements within a feature or model need to be fixed to resolve the failure.



- You use Failure Diagnostics when parts fails upon regeneration.
- By investigating the failed feature, you can determine what cause the failure and fix it.
- The Failure Diagnostics mode helps you to resolve failures so that errors are not repeated, and time is not wasted.
- By fixing errors, you also make your models more robust.

Benefit:

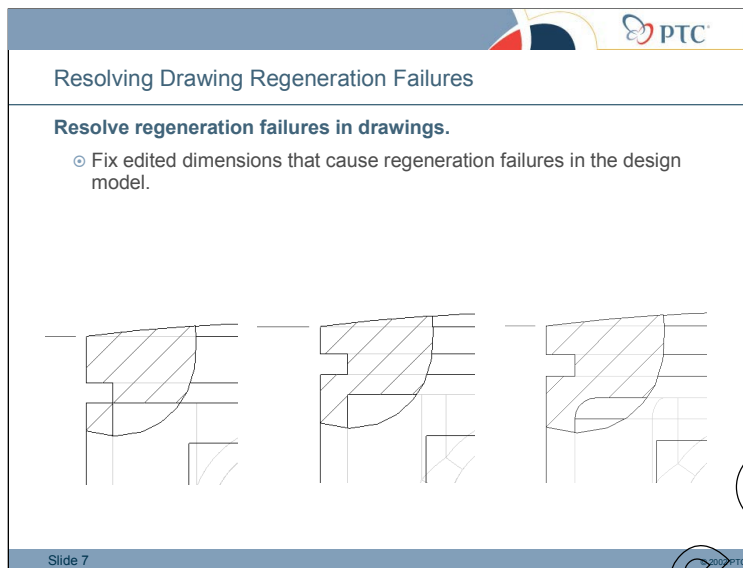
- By resolving failures, you save other individuals within company time because they will not repeat the error.



- You use Failure Diagnostics when assemblies fail upon regeneration.
- The Failure Diagnostics mode helps you to resolve failures so that errors are not repeated, and time is not wasted.
- By fixing errors, you also make your models more robust.

Benefit:

- By resolving failures, you save other individuals within company time because they will not repeat the error.



- You use failure diagnostics when parts within a drawing fail upon regeneration. By investigating the failed feature, you can determine what caused the failure.
- The Failure Diagnostics mode helps you to resolve failures so that errors are not repeated, and time is not wasted.
- By fixing errors, you also make your models more robust.

Benefit:

- By resolving failures, you save other individuals within company time because they will not repeat the error.



Parent/Child Relationships – Resolving Regeneration Failures

If model changes cause parent/child relationships to fail, or cause model geometry to become invalid, models enter a Resolve Mode. By understanding parent/child relationships, you can fix the model or the failing feature and resolve regeneration failures.

- Part Failures
- Assembly Failures
- Drawing Failures

Slide 8

Note to Instructors:

Use this slide to teach students how the functionality they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

NOTE: It might be best to review this information after completing the demonstrations.

Regeneration failures are generally due to an failure within a parent/child relationship. For example when an assembly model (parent) cannot find a part model (child of assembly file) within the working directory, there will be a regeneration failure. In many cases, fixing the parent/child relationship will resolve the regeneration failure.



You will follow along while I demonstrate:

1. Obtaining Design Information
2. Resolving Part Failures

Then, you will practice the same steps in the laboratory exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 10

Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY assessment questions.
- **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 Exams

Description: For each course, 5 new 10 question Pro/FICIENCY 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:

- Obtain design information in part and assembly models.
- Resolve regeneration failures in parts, assemblies and drawings.
- Describe how parent/child relationships affect regeneration failures.

Referencing PTC Help for Module 15 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Resolve	Part Modeling	The Resolve Environment About Resolving Feature Failures Example: Misalignment Example To Use the FEAT FAILED Menu To Use the RESOLVE FEAT Menu To Use the INVESTIGATE Menu To Use the FIX MODEL Menu
Resolve	Assembly and Welds	About Resolving Failures

Module 15 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Obtaining Design Information

Introduction

In this demonstration, we examine parts to obtain feature and model design information. Features contain information that describes how they are created. This information includes dimensions, sketches, parents, and children as well as other information.

Like feature information, models also contain information that describes how they are created. This information includes all features within the model tree, model name, units, cross-sections, and reference assemblies.

Feature and model information identifies what elements within a feature or model need to be fixed to resolve a failure that occurs after regeneration.

Objective

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

- Examine parts to obtain design information.

Scenario

In order to be prepared to fix failed parts and features, you must examine the connecting rod, recoil pulley, and muffler. You must familiarize yourself with the dimensions, sketches, parents, and children.

Step 1. Obtain the design information from the CONNECTING_ROD.PRT.

In this initial step, we examine the connecting rod. Note that we can find vast amounts of information on features, such as dimensions, sketches, parents, and children.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to **C:\users\student\fast_track_330\module_15\part**.
3. Open CONNECTING_ROD.PRT.
4. Click **Info > Model**. Minimize the model tree to and resize the browser to view both the browser and the model, as shown in the following figure.

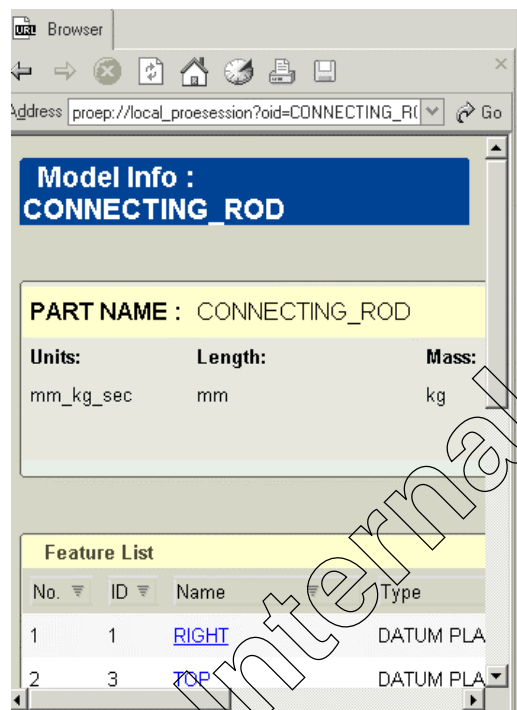


Figure 1: Viewing the CONNECTING_ROD.PRT Model Information

5. Examine the model information. Note that unit information is available, as well as information on each CONNECTING_ROD.PRT feature.
6. Highlight several features by selecting in the Name column. Repaint after selecting each feature.

Note:

Features with the **---** symbol for Name are features that have not been renamed.

7. Highlight the BASE_PROTRUSION. Obtain Feature Info by selecting the appropriate icon in the Actions column.
8. Notice the browser gives general feature information, parent/child information, feature element information, section information, and dimension information.
9. Scroll to the Dimensions section. Select various dimensions to highlight them in the graphics window, as shown in the following figure.

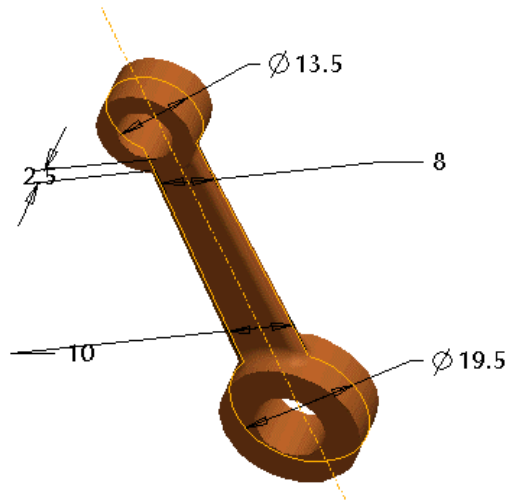


Figure 2: Displaying the Feature's Dimensions

10. Repaint the screen. Scroll to the Children section. Highlight various child features.
11. Click Back in the browser. Highlight the SIDE_ROUND. Obtain Feature Info by selecting the appropriate icon in the Actions column. Repaint and then highlight this feature's children.
12. Minimize the browser.
13. Select BASE_PROTRUSION in the graphics window. Right-click and select **Info > Parent/Child**.
14. Expand the Copy Geom feature in the Parents list. Notice that the feature references Axes within the Copy Geometry feature as Parents, and that SIDE_ROUND appears as a Child.
15. Select SIDE_ROUND in the Child list, right-click and select **Set Current**. Expand BASE_PROTRUSION in the parent list and highlight its reference surfaces.

Note:

The PARENT/CHILD dialog box provides more detailed information than can be obtained from the Feature Info browser.

16. Repaint and highlight features in the Child list. Select **Round id 362**, right-click and select **Set Current**. Expand SIDE_ROUND in the Parent list and highlight the reference edges.
17. Close the PARENT/CHILD dialog box.
18. Suppress SIDE_ROUND using the model tree. Notice that the child features highlight, and click **Options**. Select **Suspend** for both child features. Click **Ok**.
19. Notice that the child rounds regenerate, but create different geometry without the parent round creating a tangent edge. Resume all features.
20. Close the window.

Note:

The **Suppress** option will suppress the child features along with the parent features. The **Suspend** option suppresses the parent features, and regenerates the children features without the parent features. This may result in a resolve, but it can be a viable method of deleting parents

Step 2. Obtain design information from the RECOIL_PULLEY.PRT.

In the next step, we examine the recoil pulley. In particular, we obtain the design information for the LEFT_TOOTH feature, and note that it contains an open ended section. We also note that the RIGHT_TOOTH is dependent on the LEFT_TOOTH.

1. Open RECOIL_PULLEY.PRT.
2. Select the LEFT_TOOTH feature from the model tree. Right-click and select **Info > Feature**.
3. Notice that the feature has an open section, and that the RIGHT_TOOTH feature is a child.
4. Obtain Feature Info for the RIGHT_TOOTH feature by selecting the appropriate icon in the Actions column. Notice that the RIGHT_TOOTH feature also has an open section.
5. Close the window.

Step 3. Obtain the design information from the MUFFLER.PRT.

Instructor: Skip this step for the demo

1. Open MUFFLER.PRT.
2. Select the initial hole in the pattern using the model tree. Right-click and select **Info > Feature**.
3. Notice the general information lists the hole as 1 of 37. Also notice that the Side One depth element is defined as THRU NEXT.
4. Minimize the browser. Right-click and select **Info > Parent/Child**.
5. Expand the shell feature in the Parent list and highlight the reference surface. Close the dialog box.

Note:

This hole was placed on the inside of the shell with its depth defined upward through the next surface.

6. Close the window.
7. Erase all models from memory.

This completes the exercise.

Exercise 2: Resolving Part Failures

Introduction

In this demonstration, we learn how to resolve regeneration failures. We do this by creating a failure by modifying parts and regenerating them. Then we investigate and resolve the failure.

Objectives

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

- Investigate part failures.
- Resolve part failures.

Scenario

Due to some manufacturing difficulties, you are assigned to edit the following models so that they can be manufactured: connecting rod, recoil pulley, and muffler. You must complete the edits as instructed.

Step 1. Redefine a feature to “inadvertently” create a missing reference.

In this initial step, we edit the sketch of the MAIN_PROTRUSION of the connecting rod. First, we delete a line and sketch an arc in its place. Next, we get a regeneration failure. We determine that we deleted a reference required by a round. By editing the round's references, we resolve the failure.

1. If necessary, set the working directory to **C:\users\student\fast_track_330\module_15\part**.
2. Open CONNECTING_ROD.PRT.
3. Expand the BASE_PROTRUSION in the model tree. Select its section, right-click and select **Edit Definition > Sketch**.
4. Delete the “vertical” line on the right. Read the prompt and click **Yes**. Sketch an arc in its place, as shown in the following figure.

Note:

The dimensioning scheme is not important for this exercise.

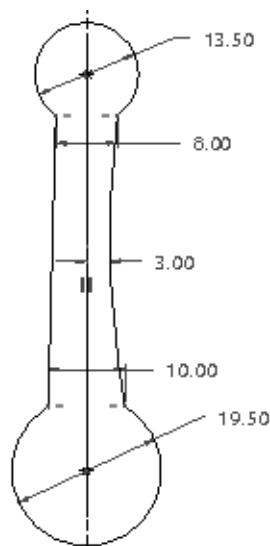


Figure 3: Sketching an Arc

5. Complete the sketch and the feature.

Step 2. Resolve the missing reference failure.

1. Read the FAILURE DIAGNOSTICS window. Note that the failing round is Feature 11, and that it is missing references.
2. Click **Investigate > Backup Model > Confirm**. Open CONNECTING_ROD.PRT
3. In the model tree, click **Settings > Tree Columns**. Add **Feat#** to the Displayed list, and click **OK**.
4. Select feature **11** from the model tree to highlight the feature on the model. Notice the failing feature is the SIDE_ROUND.
5. Click **Current Model** to return to the current (failing) model. Click **Show Ref**. Expand the BASE_PROTRUSION in the Parent list. Highlight the surfaces and notice there is a missing surface reference. Close the References dialog box.
6. Click **Quick Fix > Redefine > Confirm**. Select the **Sets** tab and select Set2. Right-click and select **Driving Surface**. Select the thin arc-shaped surface to create the round. Complete the feature.
7. Click **Yes** to exit resolve mode.



Figure 4: Resolved Round Feature

Step 3. Redefine a feature to avoid creating a missing reference.

1. Expand the BASE_PROTRUSION in the model tree. Select its section, right-click and select **Edit Definition > Sketch**.
2. Select the arc on the right side. Mirror the arc, selecting the vertical centerline as a reference.
3. With the mirrored arc still selected, click **Edit > Replace**. Select the remaining line and click **Yes**. Refer to the following figure.

Note:

The dimensioning scheme is not important for this exercise.

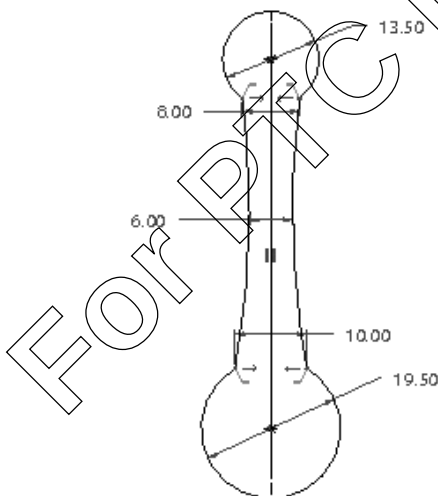


Figure 5: Replacing Sketched Entities (left) and the Completed CONNECTING_ROD.PRT (right)

4. Complete the sketch and the feature. Notice there are no feature failures.

Note:

By using **Replace** when deleting sketched entities, you can transfer references to the new entity and prevent the model from entering Resolve mode.

5. Save the model.
6. Close the window.

Step 4. Resolve an open section failure.

In the next step, we modify the tooth height of the recoil pulley. Due to a data entry error, we accidentally enter the wrong number for the height. After editing the number again, we get a regeneration error. Upon investigation we determine that the tooth sketch is incomplete. After editing the sketch, we resolve the failure.

1. Open RECOIL_PULLEY.PRT.
2. Edit the LEFT_TOOTH height to **7** and regenerate, as shown in the following figure.

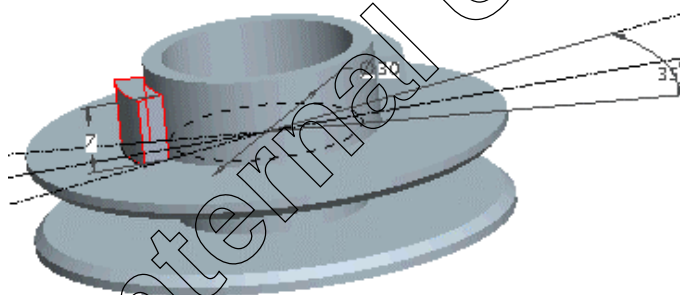


Figure 6: Editing the Left Tooth Height

3. Edit the LEFT_TOOTH height to **11** and regenerate. The protrusion fails.
4. Read the FAILURE DIAGNOSTICS window. Note that the system could not intersect the part with the feature.
5. Click **Quick Fix > Redefine > Confirm**.
6. Use the dashboard to edit the **Sketch**.
7. Sketch an arc to close the sketch, as shown in the following figure.

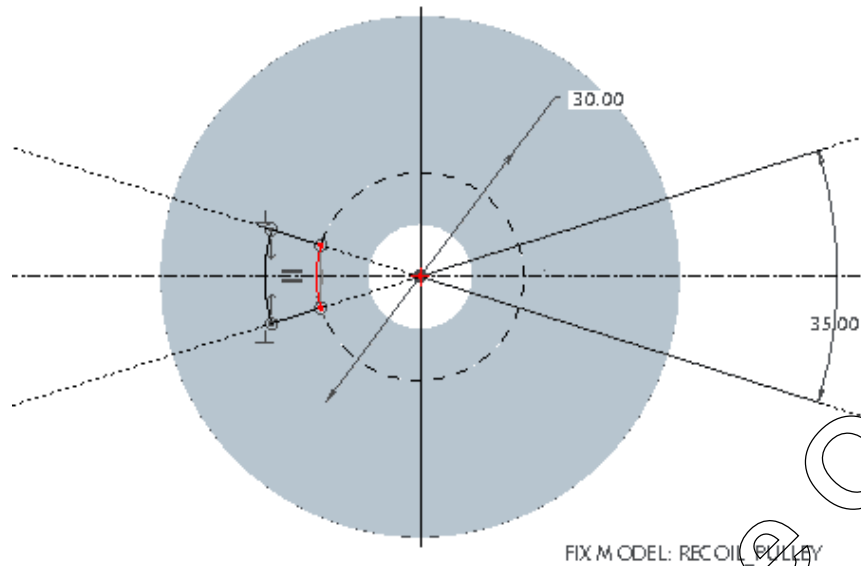


Figure 7: Resolving the Left Tooth Failure

8. Complete the sketch and the feature.
9. The RIGHT_TOOTH fails with a similar message. Resolve the RIGHT_TOOTH failure using previous techniques.

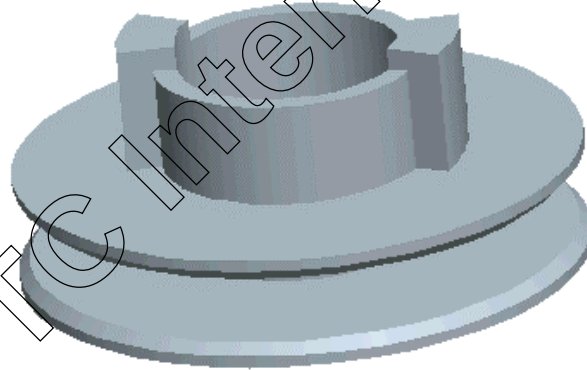


Figure 8: Resolved RECOIL_PULLEY.PRT

10. Click **Yes** to exit Resolve mode.
11. Save the model.
12. Close the window.
13. Erase all models from memory.

Step 5. (OPTIONAL) Resolve a pattern failure.

Instructor: Skip this step for the demo.

1. Open MUFFLER.PRT.

Note:

All patterns are created as General by default. General Patterns are the most robust, but take longer to regenerate. Identical Patterns are the least robust, but are faster to regenerate.

2. Drag the **Insert** indicator before the Shell feature.
3. Start the Extrude tool and select the existing curve. Edit the depth to **2**, and complete the feature, as shown in the following figure.

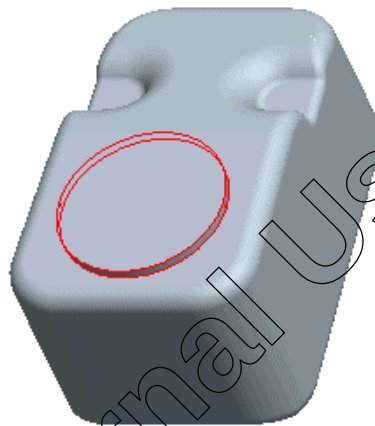


Figure 9: Completed Protrusion

4. Create two rounds with a radius of **2**, as shown in the following figure.

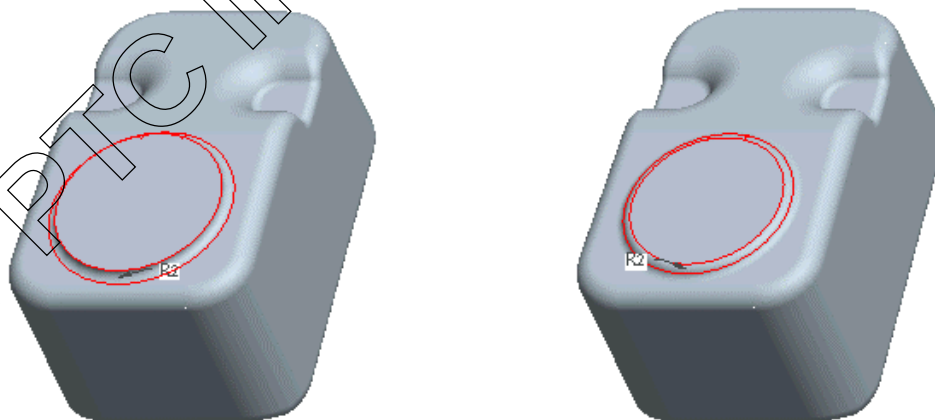


Figure 10: Creating Two rounds

5. Select the **Insert** indicator, right-click and select **Cancel > Yes**. The pattern fails.
6. Read the FAILURE DIAGNOSTICS window.
7. Click **Fix Model > Feature > Redefine**. Select the Pattern from the model tree.

8. Select the **Options** tab from the dashboard, and click **Variable**. Complete the feature.
9. Click **Yes** to exit Resolve mode.
10. View the model in No Hidden display to notice that some instances intersect the round.

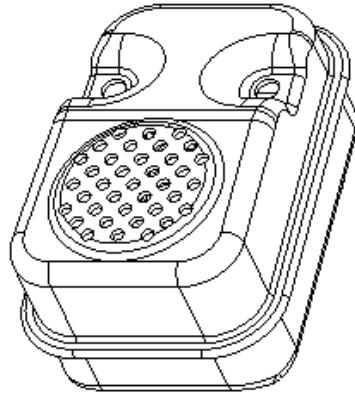


Figure 11: Viewing the MUFFLER.PRT with No Hidden Lines

11. Save the model.
12. Close the window.
13. Erase all models from memory.

This completes the exercise.

Exercise 3: Resolving a Part Failure from a Drawing (Challenge)

Objective

In this exercise, you learn how to resolve failures within drawings.

Scenario

The drill drawing has been shipped to Manufacturing so that tooling can begin. However, the drawing failed when Manufacturing made a change. You are assigned to resolve the failures within the drill drawing so that manufacturing can continue with their planning.

Step 1. Edit the piston.

1. Set the working directory to **C:\users\student\fast_track_330\module_15\drw**.
2. Open **DRILL_COMPS.DRW**.
3. Zoom in to the detail view of the ring groove. Edit the depth to **2**. Use the toolbar to Regenerate the model. A round fails, and the model opens.
4. Read the **FAILURE DIAGNOSTICS** window.
5. Click **Investigate > Backup Model > Confirm**. Open **PISTON.PRT**.
6. In the model tree, click **Settings > Tree Columns**. Add **Feat #** to the Displayed list, and click **Ok**. Select feature **17** from the model tree to highlight the feature on the model.
7. Click **Current Model** to return to the current (failing) model. Click **Quick Fix > Suppress > Confirm > Suppress All**. Click **Yes** to exit Resolve mode.
8. Examine the updated groove feature in the detail and section views.
9. Select the front view and access its properties. Click **View Type > Section > Done > Local > Done**. Select section **B** and middle-click to not place section arrows.
10. In the front view, select an edge of the ring groove cut and sketch a spline around it. Middle-click to finish the spline and click **Done > Done**.
11. Select the hatch lines, right-click, and select **Properties**. Click **Spacing > Half > Done**, as shown in the following figure.
12. Examine the front view. Note that in the new cross section that the geometry is not connected. The groove needs to be moved upwards, to reconnect the geometry.

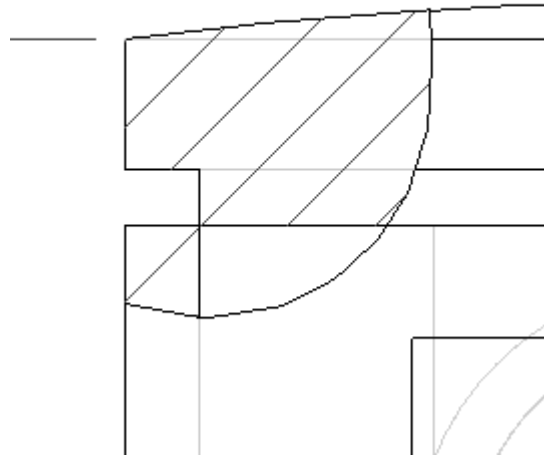


Figure 12: Creating the Cross-sectional Spline

Step 2. Resolve the failure.

1. Edit the groove height dimension from **25** to **26.5** and regenerate.
2. Note that the geometry around the cut is now connected in the local cross section.

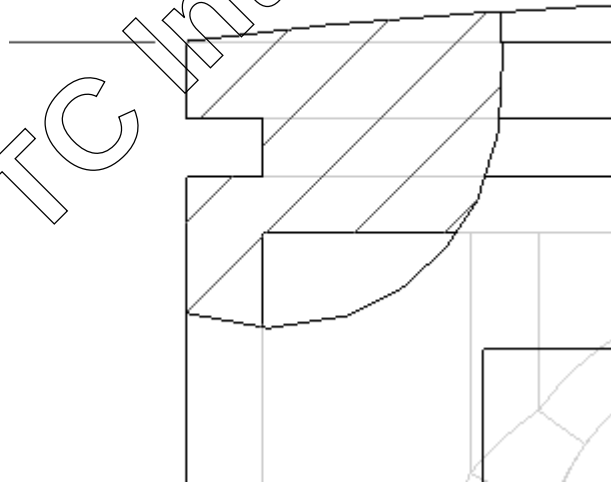


Figure 13: Examining the Edited Cross-section

3. Use the model tree to open PISTON.PRT. Resume all suppressed features. Close the window to return to DRILL_COMPS.DRW.
4. Examine the groove cut in various drawing views.

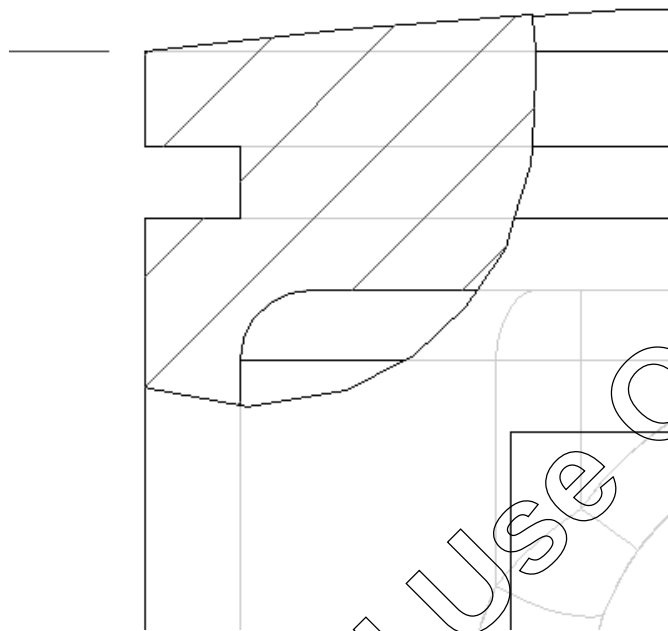


Figure 14: Resolved PISTON.PRT

5. Save the drawing.
 6. Erase all models from memory.
- This completes the exercise.

Exercise 4: Resolving Assembly Failures (Challenge)

Objective

In this exercise, you learn how to resolve failures within assemblies.

Scenario

The engine assembly has been shipped to Manufacturing so that tooling can begin. However, the assembly failed. You are assigned to resolve the failures within the engine assembly so that manufacturing can continue with their planning.

Step 1. Examine the ENGINE.ASM.

1. Set the working directory to **C:\users\student\fast_track_330\module_15\assy.**
2. Open ENGINE.ASM. Notice there are no failures.
3. Close the window.
4. Erase all models from memory.

Step 2. Edit the ENGINE.ASM components to “inadvertently” create failures.

1. Within Windows Explorer, move the PISTON_RING.PRT to **C:\users\student\fast_track_330\module_15\assy\Ring_Folder.**
2. Open PISTON.PRT.
3. Select the RING_GROOVE cut, and edit the definition of the sketch. Delete the line and sketch an arc, as shown in the following figure.

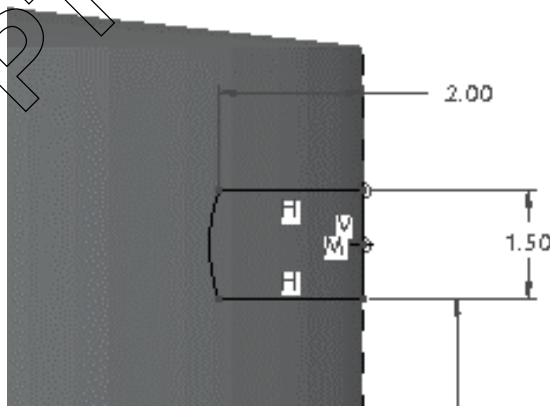


Figure 15: Sketching the Arc on the Groove Cut

4. Complete the sketch and the feature.

5. Save the model.
6. Close the window.
7. Open ENG_BLOCK_FRONT.PRT.
8. Edit the CYL_HOLE dimensions to **18.5** and **22.5**, as shown in the following figure.

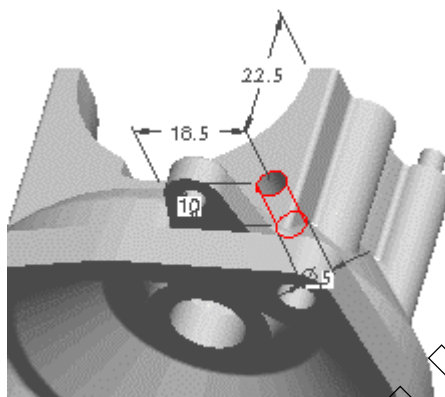


Figure 16: Editing the CYL_HOLE Placement Dimensions

9. Regenerate the model.
10. Save the model.
11. Close the window.

Step 3. Resolve the PISTON_ASSY.ASM failures.

1. Open PISTON_ASSY.ASM.
2. Read the FAILURE DIAGNOSTICS window. Note that the piston ring failed with two reasons listed.
3. To locate the ring model, click **Quick Fix > Find Component**. Navigate to the RING_FOLDER and open the PISTON_RING.PRT.
4. The PISTON_RING.PRT fails again. Click **Quick Fix > Redefine > Done**. Select the cylindrical surface of PISTON.PRT. Complete the component placement.
5. Click **Yes** to exit Resolve mode.
6. Open PISTON_RING.PRT.
7. Click **File > Backup > Ok** to save PISTON_RING.PRT to the current directory.

Note:

A copy of PISTON_RING remains in the PISTON_RING subfolder.

8. Close all windows.

Step 4. Resolve the ENGINE.ASM failure.

1. Open ENGINE.ASM.
2. The CYLINDER.PRT fails. Read the FAILURE DIAGNOSTICS window.
3. Click **Fix Model > Modify > Mod Part**. Select the ENGINE_BLOCK_FRONT.PRT.
4. Use the Search tool to find the feature named CYL_HOLE. Click **Ok**.
5. Edit the CYL_HOLE dimensions back from 18.5 and 22.5 to **17.5** and **20.5** respectively.
6. Click **Regenerate > Done > Yes** to exit Resolve mode.
7. Save the model.
8. Close all windows.
9. Erase all models from memory.

This completes the exercise.

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Summary

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

- Obtain design information in part and assembly models.
- Resolve regeneration failures in parts, assemblies and drawings.
- Describe how parent/child relationships affect regeneration failures.

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Project III

Introduction

Using the Pro/ENGINEER Wildfire, complete the following parts based upon the materials taught up to this point in class.

Objectives

After completing this module, you will be able to:

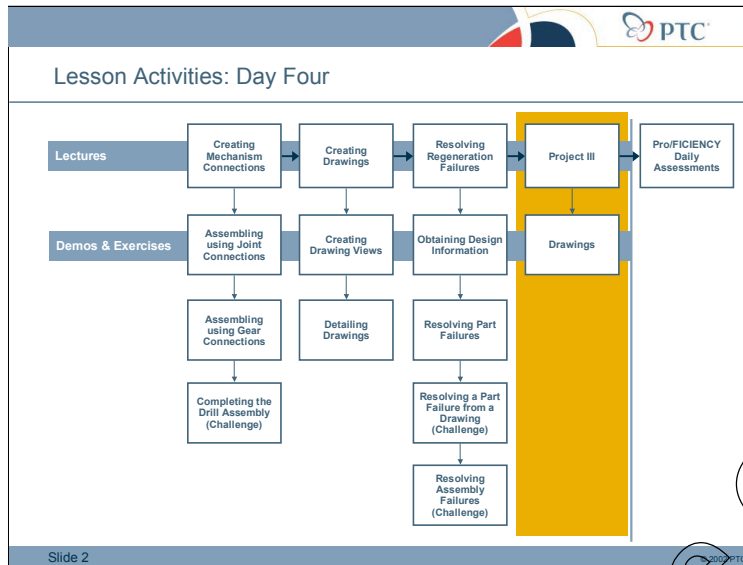
- Create Mechanism Connections.
- Create a Drawing.
- Complete the Engine Block.

Fast Track to Pro/ENGINEER Wildfire

Module 16: Project III

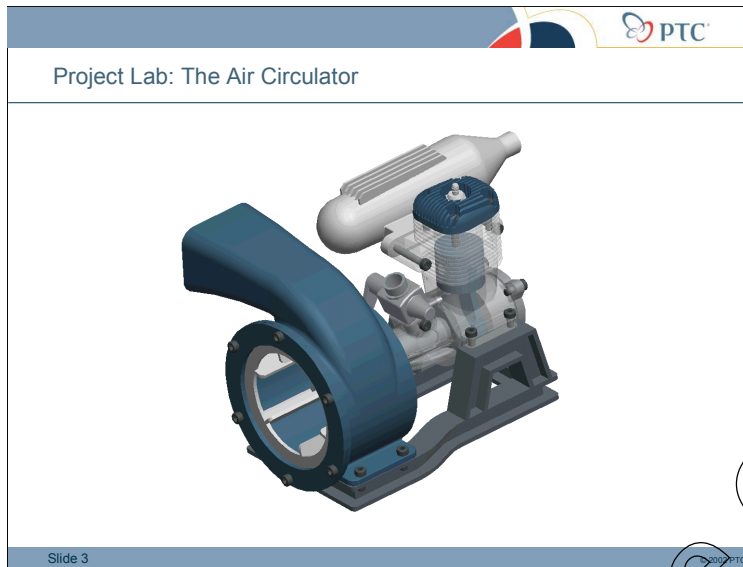


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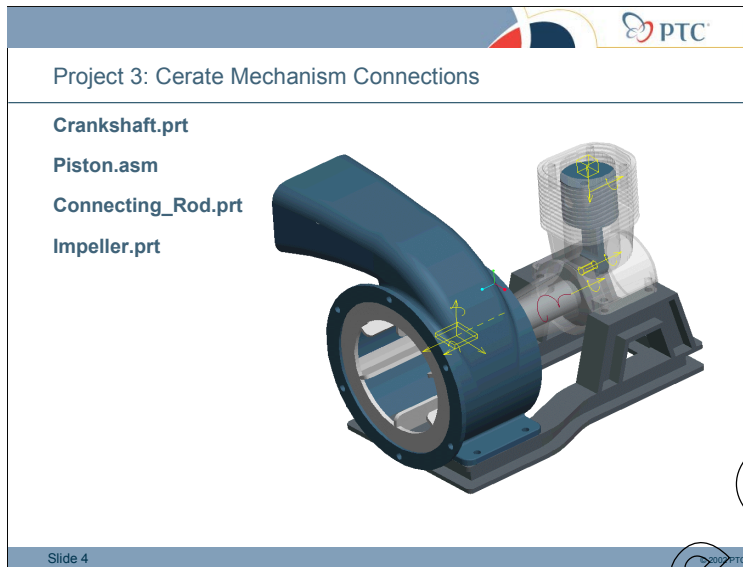
Duration

Total: 1 hr 30 min



Instructor:

Use the following slides to brief the students on what will be accomplished during this phase of the project.

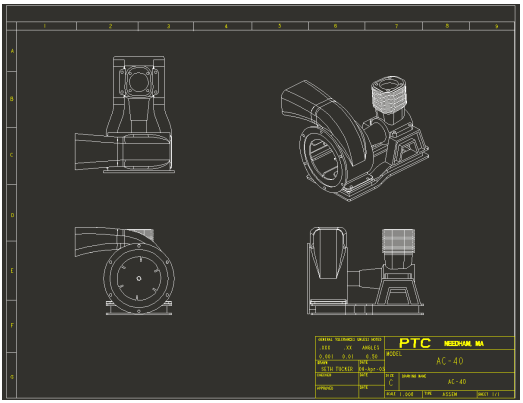


There are no notes for this slide.



Project 3: Create a Drawing: Sheet 1

AC-40.asm



PTC		WILMINGTON, MA	
DATE	01-01-2011	MODEL	AC-40
BY	J. J. J.	REV	0
APP'D		DATE	01-01-2011
CHK'D		DATE	01-01-2011
DRW'G		DATE	01-01-2011
ISSUED		DATE	01-01-2011

Slide 5

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There are no notes for this slide.

PTC

Project 3: Create a Drawing: Sheet 2

Engine_Block.prt

The drawing shows four views of an engine block: a front view (top left), a top view (bottom left), a side view (bottom center), and a section view (bottom right). Dimensions are provided for various features, including a 1.5 inch diameter hole, a 1.5 inch diameter hole, a 1.5 inch diameter hole, and a 1.5 inch diameter hole. A section line is shown in the side view, labeled 'SECTION A-A'.

PTC		REVISION	
DATE	01/01/2011	BY	01/01/2011
TIME	10:00	TIME	10:00
USER	01/01/2011	USER	01/01/2011
NAME	01/01/2011	NAME	01/01/2011
DESCRIPTION	01/01/2011	DESCRIPTION	01/01/2011

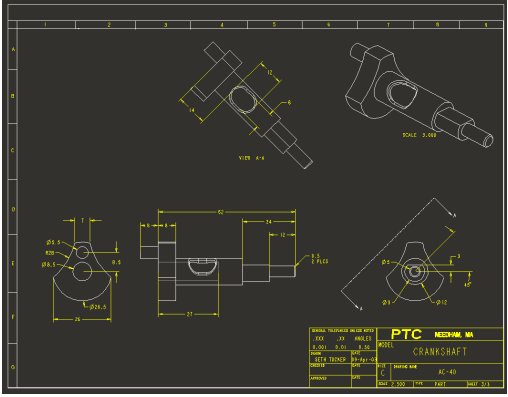
Slide 6

There are no notes for this slide.



Project 3: Create a Drawing: Sheet 3

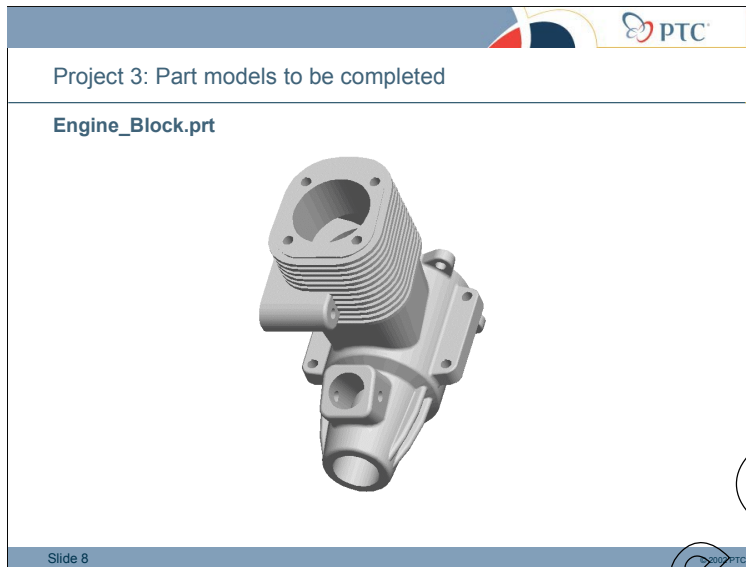
Crankshaft.prt



PTC		NICHOLAS, IN	
DATE	08/01/11	DESIGNER	CRANKSHAFT
BY	08/01/11	CHECKED	
APP'D		DATE	08/01/11
REV		BY	
REV		DATE	

Slide 7

There are no notes for this slide.



There are no notes for this slide.

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Module 16 Lab Exercises

Scenario

ACME Inc. is a multinational conglomerate that develops and markets several consumer, industrial, and defense goods. The Light Industrial Division of ACME Inc. creates a number of products including industrial fans, heating, air conditioning, pumps, and lighting. You work for the Light Industrial Division of ACME Inc., which has just started to use Pro/ENGINEER Wildfire for its product designs.

Upon returning from Pro/ENGINEER Wildfire training, you are assigned to create the AC-40 Air Circulator. As the name implies, the AC-40 is an air circulation device. Its primary purpose is to enable fast effective clean up of flooded homes and businesses that have no power due to the flood. Another variation of the AC-40 is currently being designed to use electric power. A further variation that includes an air compressor and condenser that will provide back-up air conditioning to high-rise buildings is also being considered.

In the third phase of the project, you continue building the parts that will be assembled into the AC-40. You will also continue to build the assembly using mechanism connection. Finally, you will create a drawing based upon the tasks that you have learned thus far in the training course.

Objectives

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

- Create Mechanism Connections.
- Create a Drawing.
- Complete the Engine Block.

Objective 1: Creating Mechanism Connections

Step 1. Create a connection on CRANKSHAFT.PRT.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from session. Otherwise, start Pro/ENGINEER Wildfire.
2. Using the folder navigator, browse to the following location:
c:\users\student\fast_track_330\Project. Make this folder the working directory.
3. Open ENGINE.ASM. Apply the Silver_Transparent appearance to the ENGINE_BLOCK.PRT.

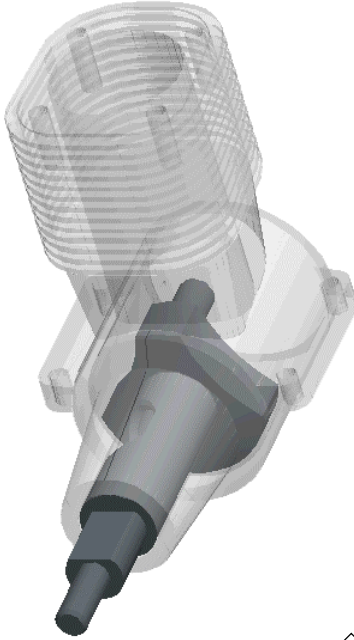


Figure 1: Transparent Appearance

4. Edit the definition of CRANKSHAFT.PRT. Enable Connections. Notice that the Insert and Mate constraints are converted into a Pin joint. Complete the redefinition.

Design Intent:

The crankshaft requires mechanism movement.

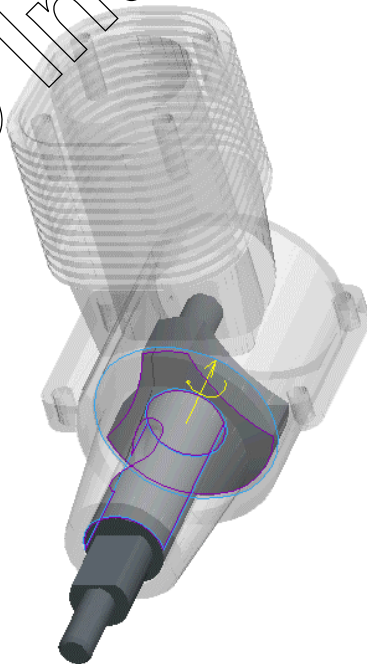


Figure 2: Pin Joint Connection

Step 2. Assemble the PISTON.ASM.

1. Assemble PISTON.ASM. Enable connections and select a Slider connection.
2. Select the cylindrical surfaces from the PISTON.PRT and the ENGINE_BLOCK.PRT for the Axis Alignment constraint.
3. Select the FRONT datum plane from the PISTON.PRT and the CTR datum plane from the ENGINE_BLOCK.PRT for the Rotation constraint. Complete the connection. Refer to the following figure.

Design Intent:

The PISTON.ASM requires mechanism movement. Leave it in mid-air for now.

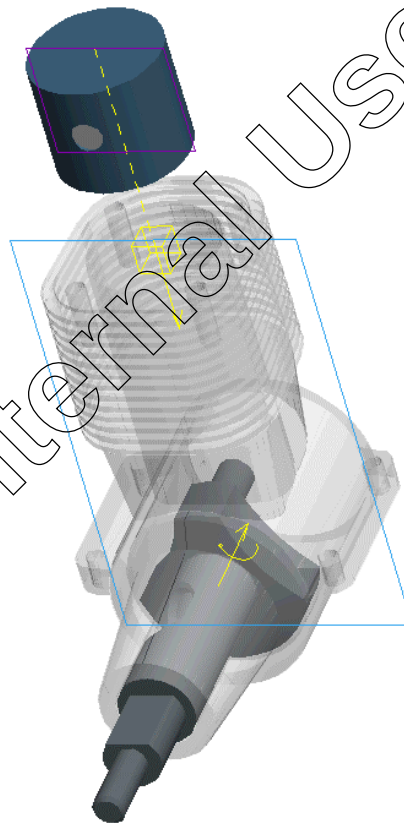


Figure 3: Slider Joint Connection

Step 3. Assemble the CONNECTING_ROD.PRT.

1. Assemble CONNECTING_ROD.PRT. Enable connections and select a Pin joint connection.
2. Select the cylindrical surfaces from the CONNECTING_ROD.PRT and the PISTON_PIN.PRT for the Axis Alignment constraint. (Select the end of the CONNECTING_ROD.PRT *without* the small oil hole)

3. Select the FRONT datum planes from the PISTON_PIN.PRT and the CONNECTING_ROD.PRT for the Translation constraint. Complete the connection and apply the Blue_Dark appearance to the CONNECTING_ROD.PRT. Refer to the following figure.

Design Intent:

The CONNECTING_ROD.PRT requires mechanism movement. Leave it in mid-air for now.

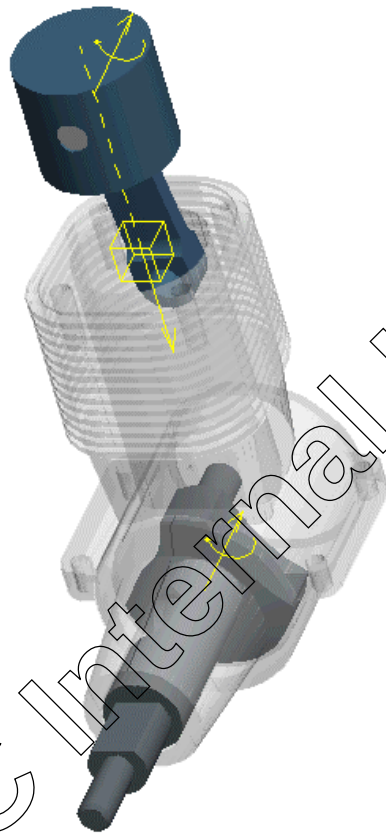


Figure 4: Pin Joint Connection

4. Add a second connection and select a Cylinder joint connection.
5. Select the cylindrical surfaces from the CONNECTING_ROD.PRT and the CRANKSHAFT.PRT.
6. Complete the connection. Refer to the following figure

Design Intent:

The CONNECTING_ROD.PRT requires mechanism movement, and attaches to both the PISTON.ASM and the CRANKSHAFT.PRT.

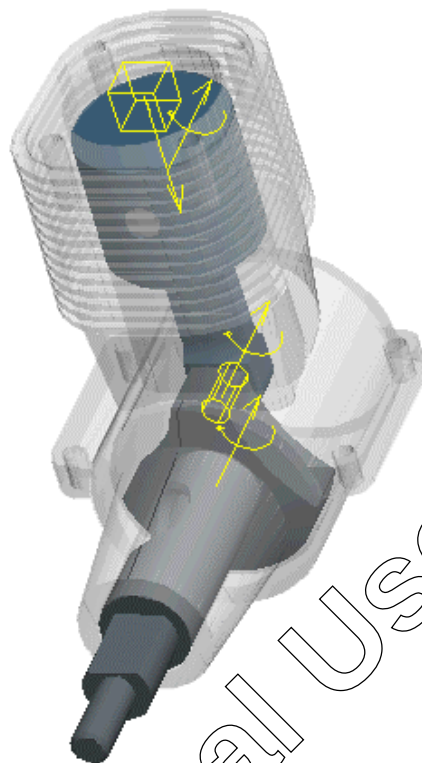


Figure 5: Engine Connections Complete

7. Save the assembly.

Step 4. Establish a pin connection on the IMPELLER.PRT.

1. Open the AC-40.ASM.
2. Select the BLOWER.ASM and open it. Edit the definition of the IMPELLER.PRT and enable connections. Notice that the Insert and Mate Offset constraints are converted into a Pin joint.
3. Complete the redefinition.

Design Intent:

The IMPELLER.PRT requires mechanism movement.

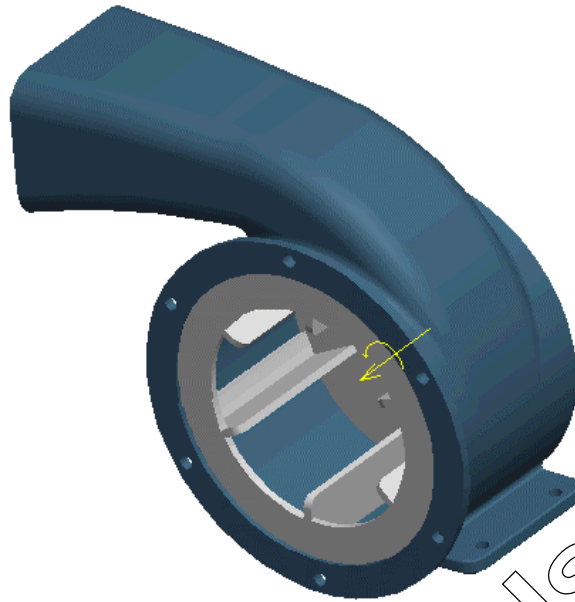


Figure 6: Pin Joint Connection

4. Close the window to return to the AC-40.ASM.

Step 5. Connect the IMPELLER.PRT to the CRANKSHAFT.PRT.

1. From the AC-40.ASM window, edit the definition of the BLOWER.ASM.
2. Enable Connections to convert the existing mate and insert constraints to a Rigid connection.
3. Add a Planar Connection. Select the planar surfaces on the CRANKSHAFT.PRT and the IMPELLER.PRT as shown in the following figure. Complete the redefinition.

Design Intent:

The assembly motion requires a connection between the CRANKSHAFT.PRT and the IMPELLER.PRT.

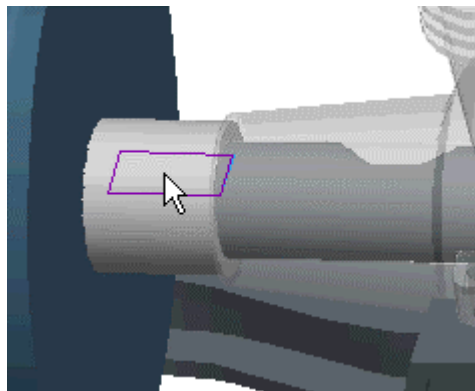


Figure 7: Selecting Hidden Planar Surfaces

Step 6. Create a servo motor and run the mechanism.

1. Enable Mechanism mode. Test the mechanism movement by dragging an impeller blade.
2. Create a servo motor. Select the pin connection on the CRANKSHAFT.PRT. Flip the direction arrow to face the BLOWER.ASM.
3. Specify the profile as Velocity with a constant magnitude of 90. Complete the definition of the servo motor.

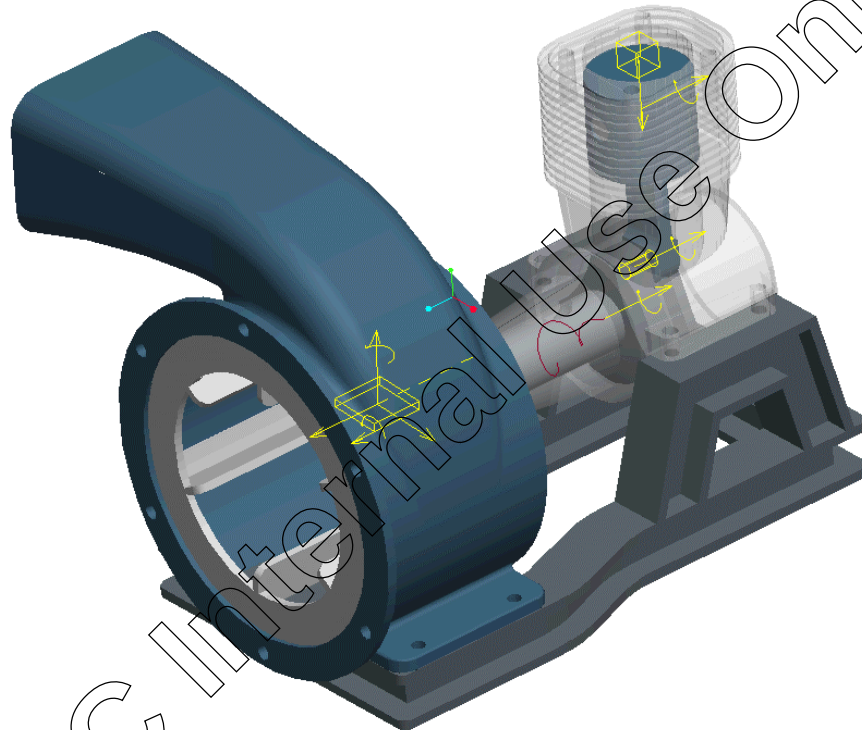


Figure 8: Mechanism Mode

4. Create an Analysis. Edit the end time to 4.0.

Design Intent:

The Velocity is 90 deg/sec. With a time duration of 4 seconds, the crankshaft will rotate a full 360 degrees.

5. Run the Analysis. Then play the Playback in a continuous loop.
6. Stop the playback when finished. Do not save the playback.
7. This model will be completed later in the project. Save the model and close the window.

This completes the exercise.

Objective 2: Creating a Drawing

Step 1. Create a saved view.

1. Open the AC-40.ASM if necessary.
2. Orient the model as shown in the following figure.

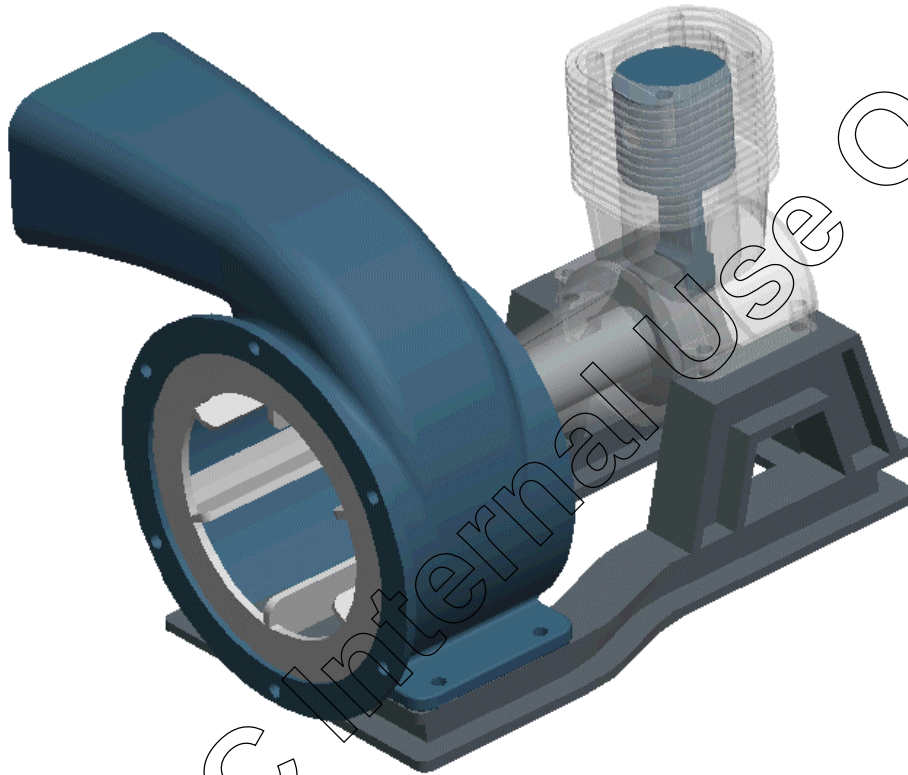


Figure 9: Orienting the Model

3. Save the view as 3D_1.

Step 2. Create a Drawing.

1. Create a drawing named AC-40.DRW. Accept all defaults.
2. Change the view display for all views to No Hidden.

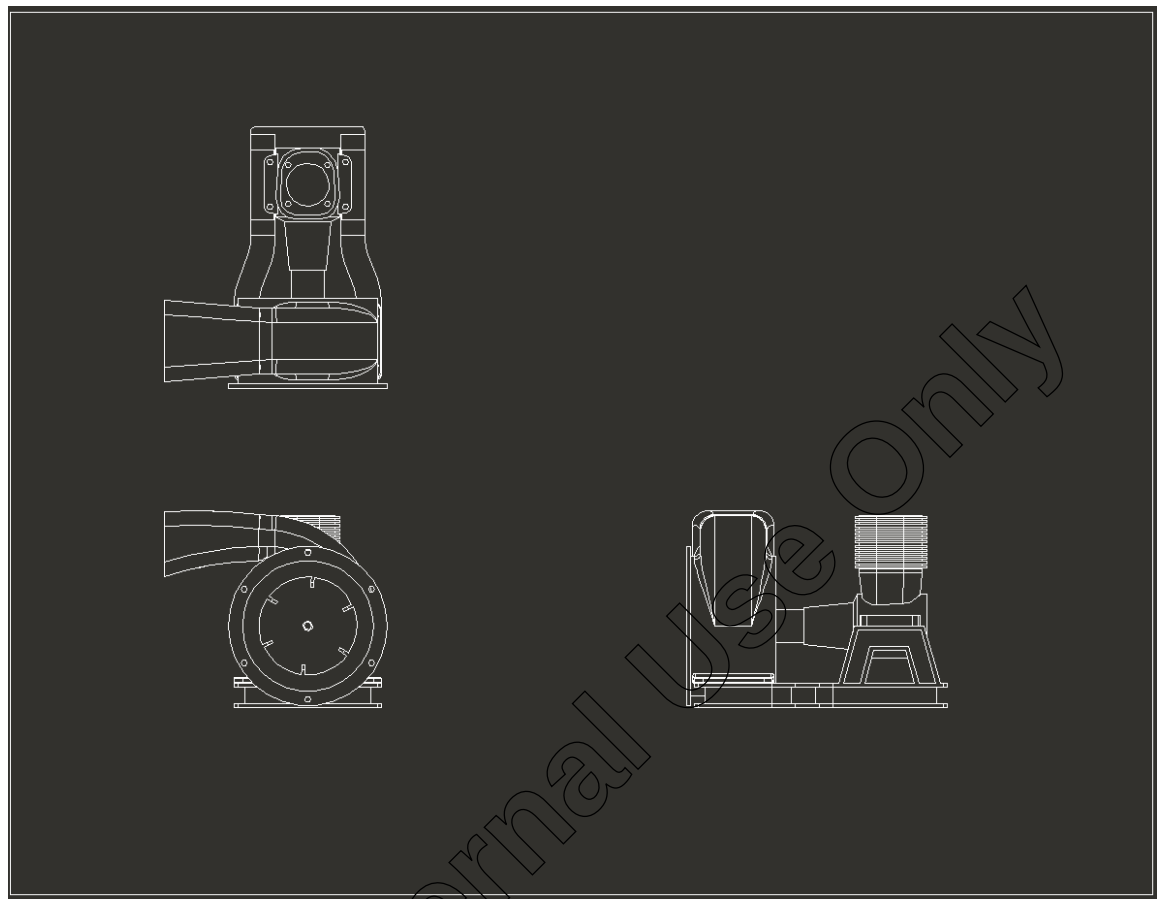


Figure 10: New Drawing

Note:

Pro/ENGINEER Wildfire automatically creates 3 standard views based on the default drawing template.

3. Insert a general view using the saved view 3D_1 in the upper right corner of the sheet. Edit its properties to not display hidden lines. Refer to the following figure.

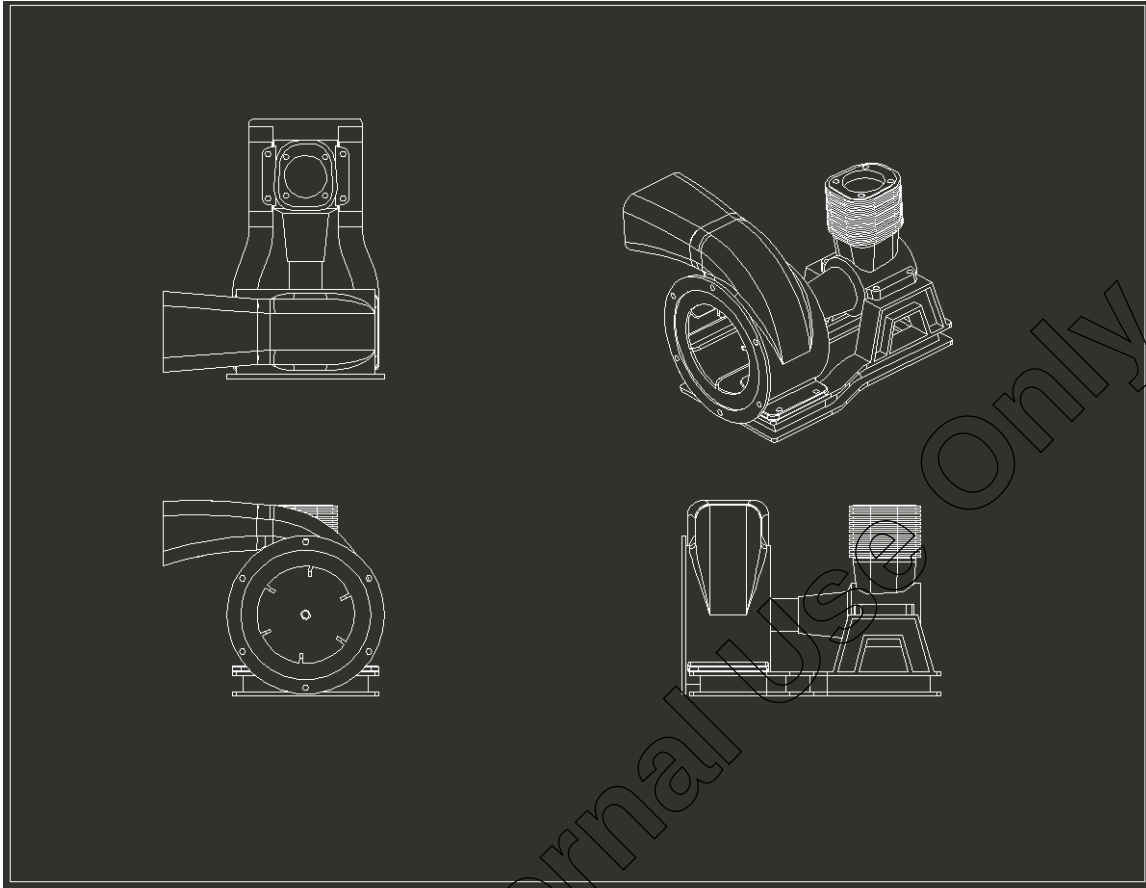


Figure 11: General View Added

Step 3. Insert views on the second sheet.

1. Insert a second drawing sheet.
2. Add the ENGINE_BLOCK.PRT model to the drawing.
3. Insert a general view in the lower left corner of the sheet. Orient the view to FRONT.
4. Modify the sheet scale to 1.75, and then insert two projected views. Change the view display for all views to No Hidden. Refer to the following figure.

Note:

For any sheet beyond sheet 1, you must create views manually.

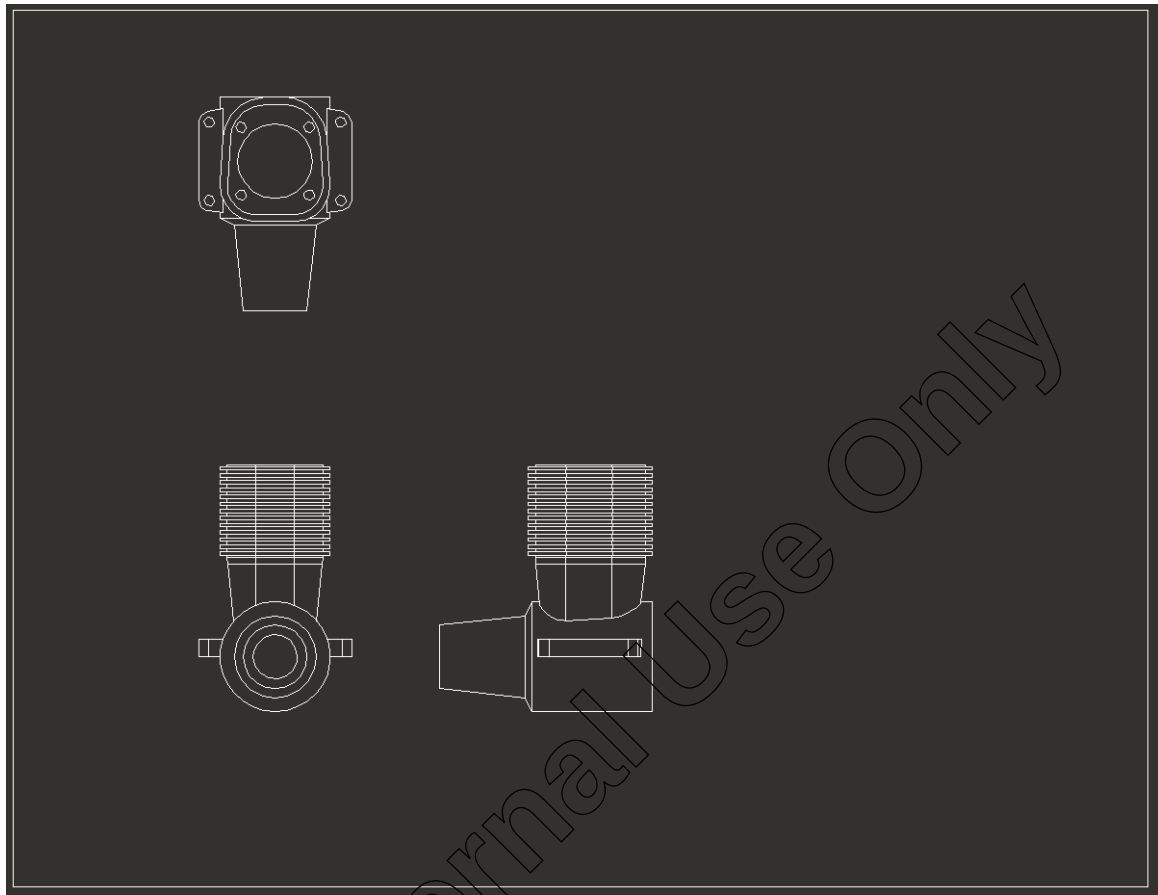


Figure 12: Inserting Views

5. Edit the properties of the RIGHT view to create a cross-section named A. Display datum planes and select datum RIGHT to create the cross-section. Show the cross-section arrows in the FRONT view. Refer to the following figure.

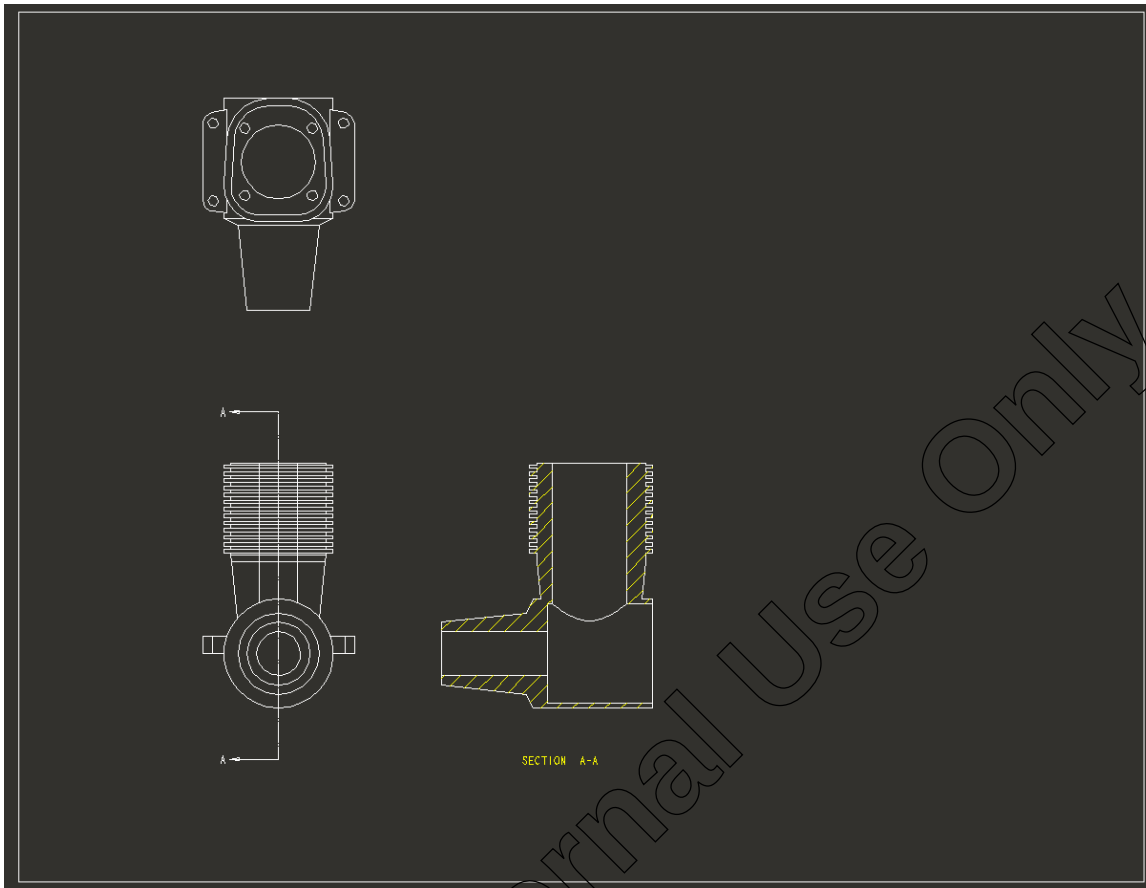


Figure 13: Creating a Cross-Section

6. Insert a detail view of the cooling fins. Name the detail **1** and use a scale of **10**. Then edit the properties of the cross-hatching in the detail to be a tighter spacing. Refer to the following figure.

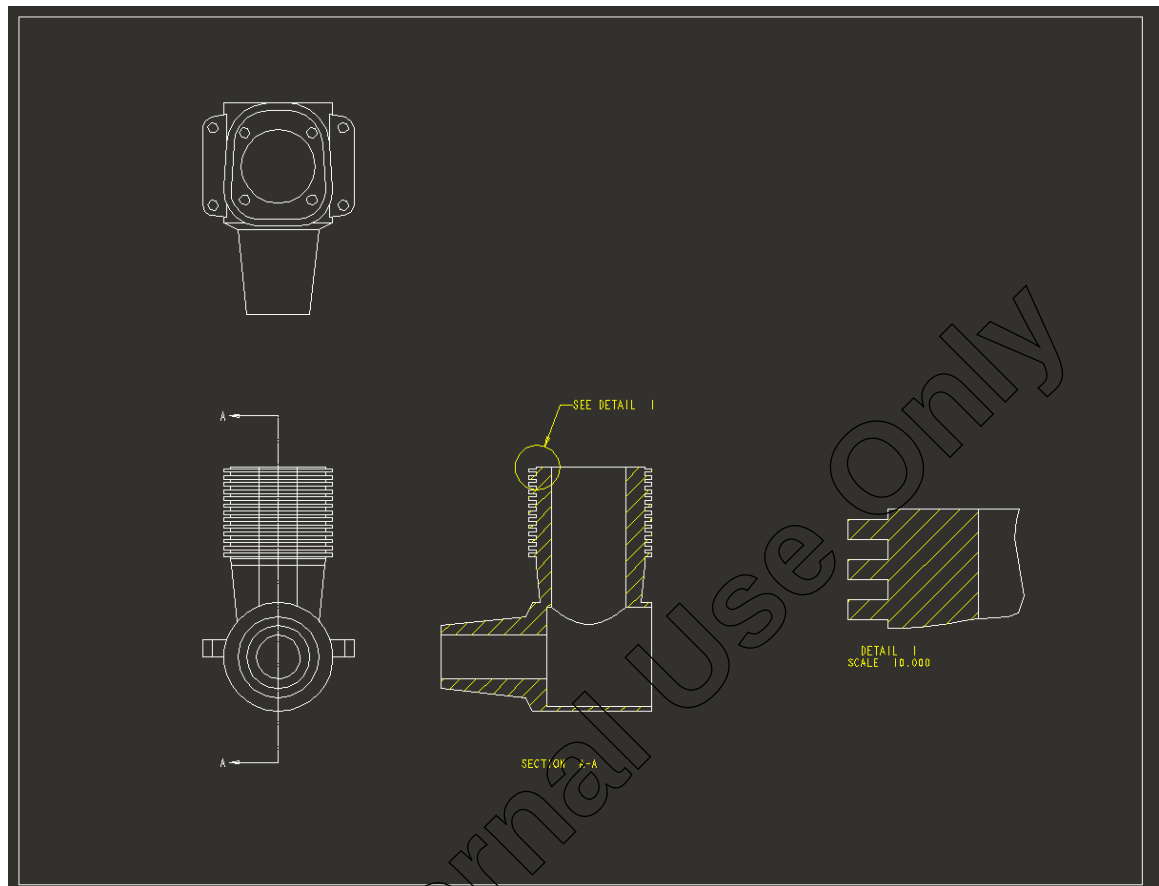


Figure 14: Inserting a Detail View

7. Insert a general view in the upper right corner of the sheet. Use the Default orientation and a scale of 2. Refer to the following figure.
8. If necessary, edit its properties to not display hidden lines.

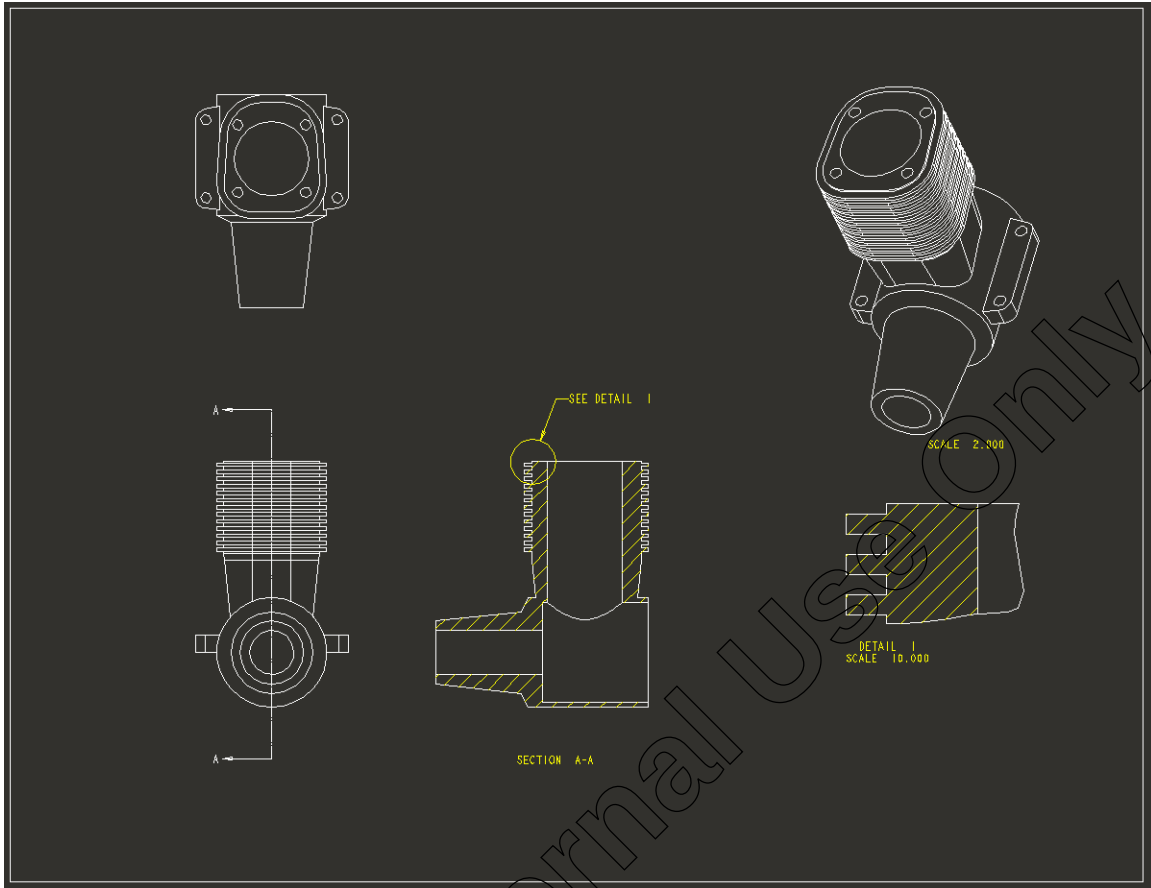


Figure 15: Inserting a General View

Step 4. Add details to sheet 2.

1. Show a few primary dimensions as shown in the following figure. Then display the quantity of FINS in the model by showing Notes.
2. Incorporate the quantity value into the Detail 1 note by entering its symbolic representation followed by **FINS**

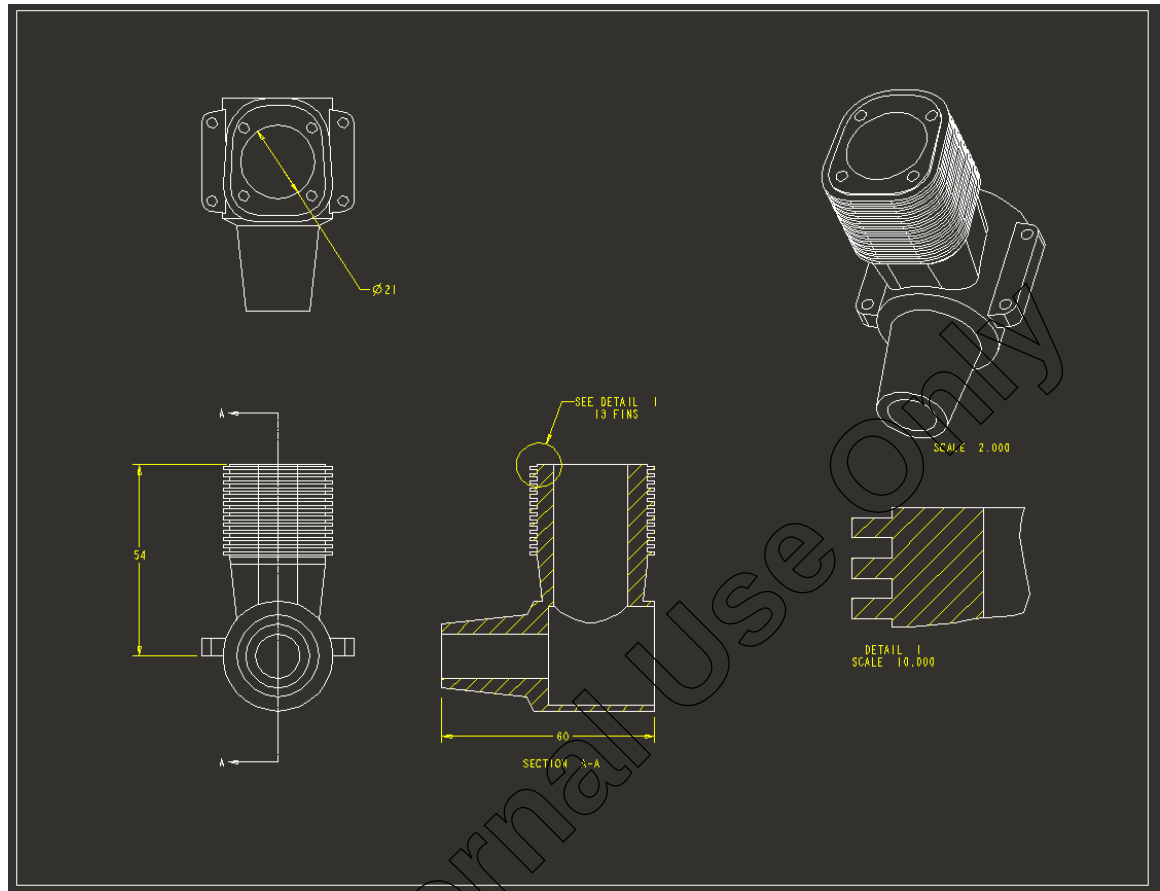


Figure 16: Drawing Details

Step 5. Insert views on sheet 3.

1. Insert a third drawing sheet.
2. Add the CRANKSHAFT.PRT model to the drawing.
3. Insert a general view in the lower center of the sheet. Orient the view to LEFT.
4. Modify the sheet scale to 2.5, and then insert two projected views. Refer to the following figure.
5. Edit the properties for the general view to display hidden lines, and the two projected views not to display hidden lines.

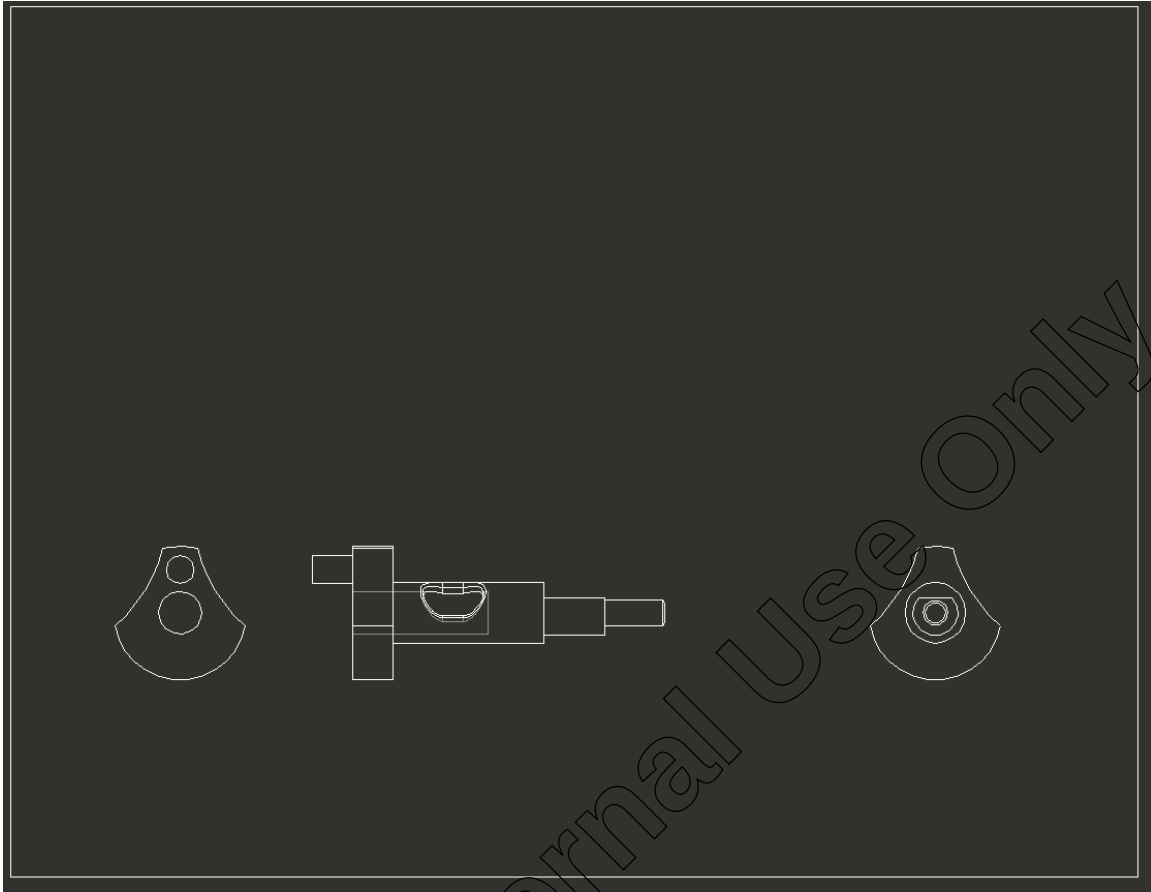


Figure 17: Inserting Views

6. Set the Config option **make_proj_view_notes** to **Yes**.
7. Unhide DTM1 in the CRANK_CUT group using the model tree. Display datum planes and repaint the screen.
8. Insert an Auxiliary view. Select a location above the general view, select DTM1 to create the view. Change the view display to No Hidden.
9. Rename the view to **A**. Add cross-section arrows for view A as shown in the following figure.

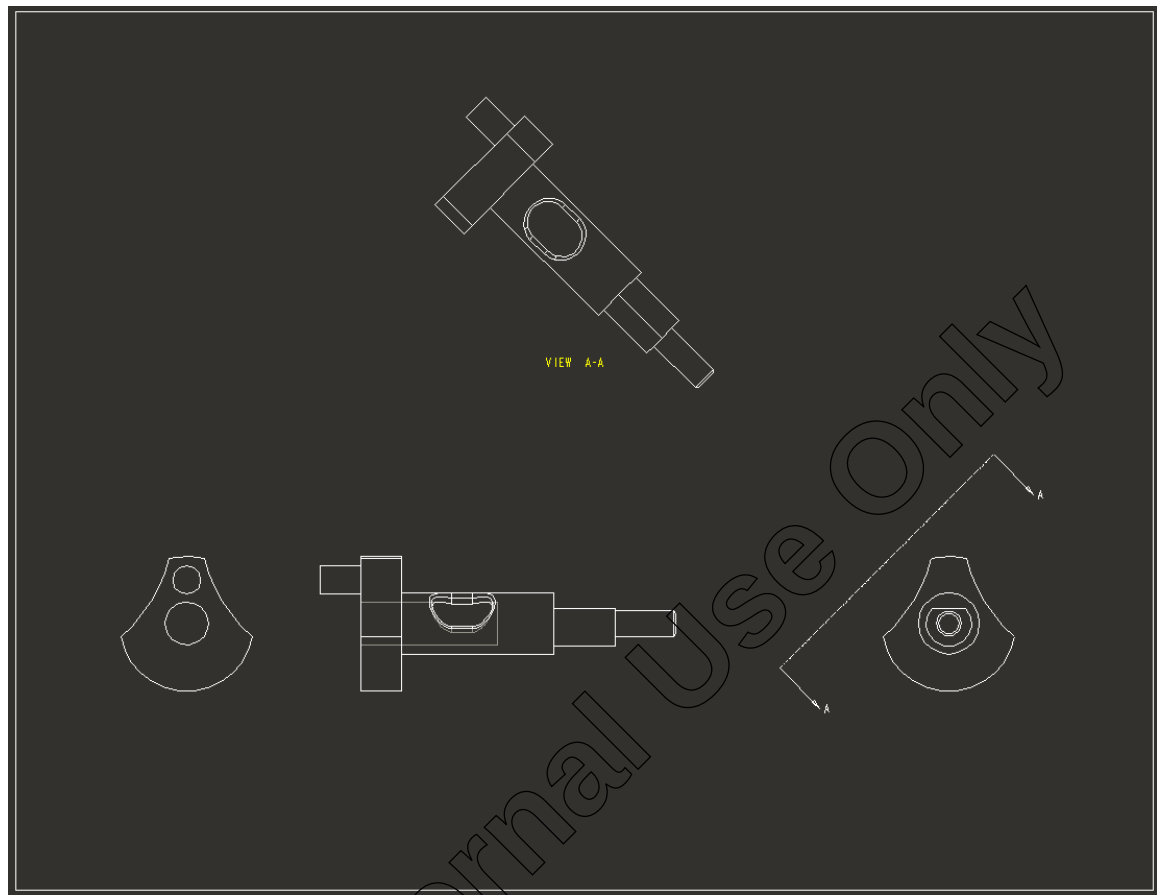


Figure 18: Inserting an Auxiliary View

10. Open the CRANKSHAFT.PRT. Save a view orientation named CRANK_3D as shown in the following figure.

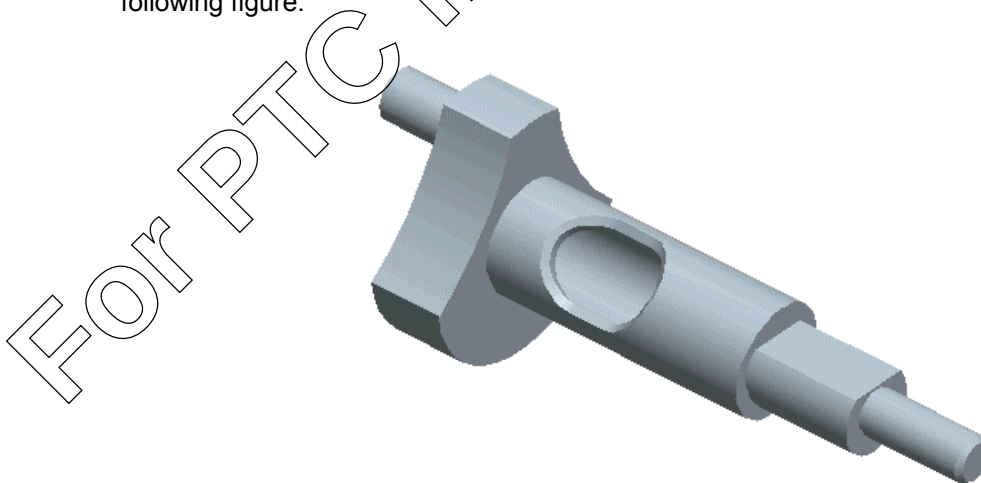


Figure 19: CRANK_3D View Orientation

11. Insert a general view that references the CRANK_3D orientation in the upper right corner of the sheet. Use a scale of **3**. Change the view display to No Hidden. Refer to the following figure.

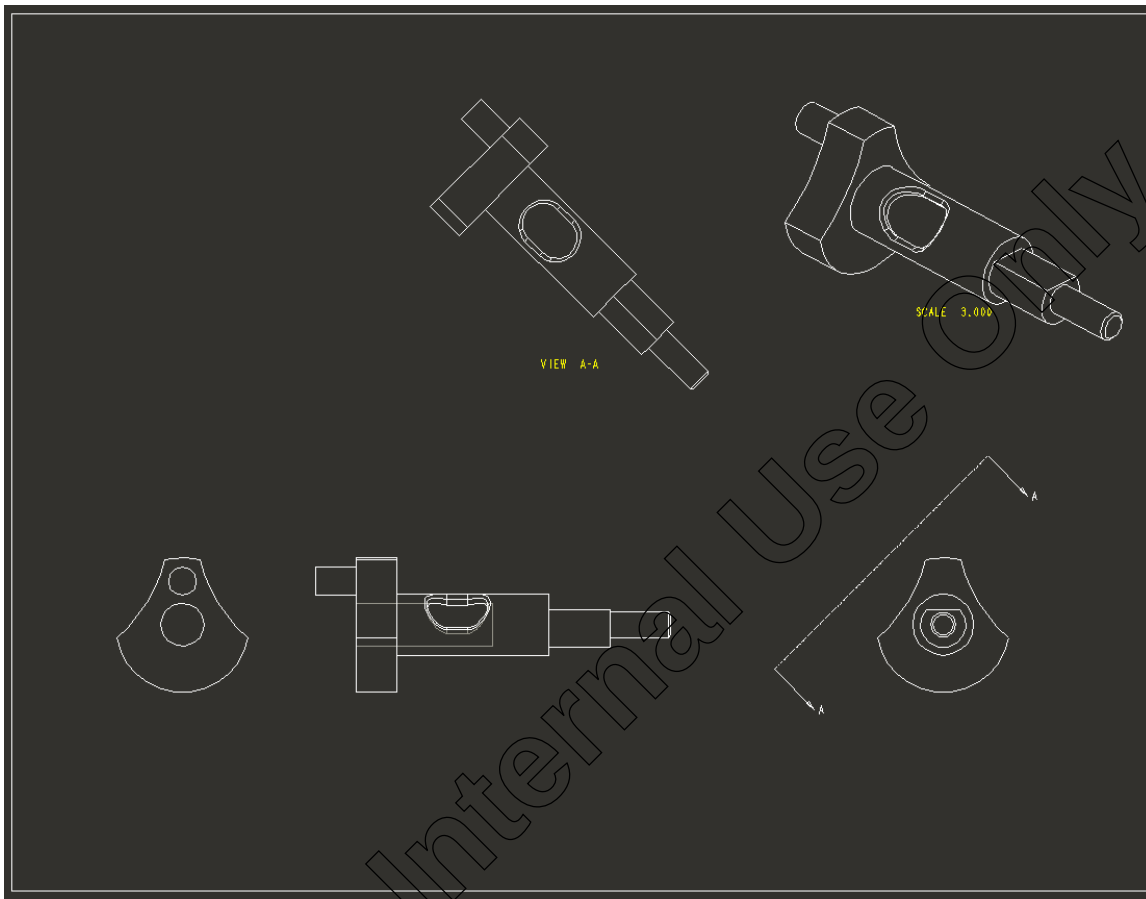


Figure 20: Inserting a General View

Step 6. Dimension sheet 3.

1. Show all dimensions.
2. Manipulate the dimensions so they appear approximately as in the next four figures.

Note:

Edit the properties of the chamfer dimension to add the “CHMFR 2 PLCS” text.

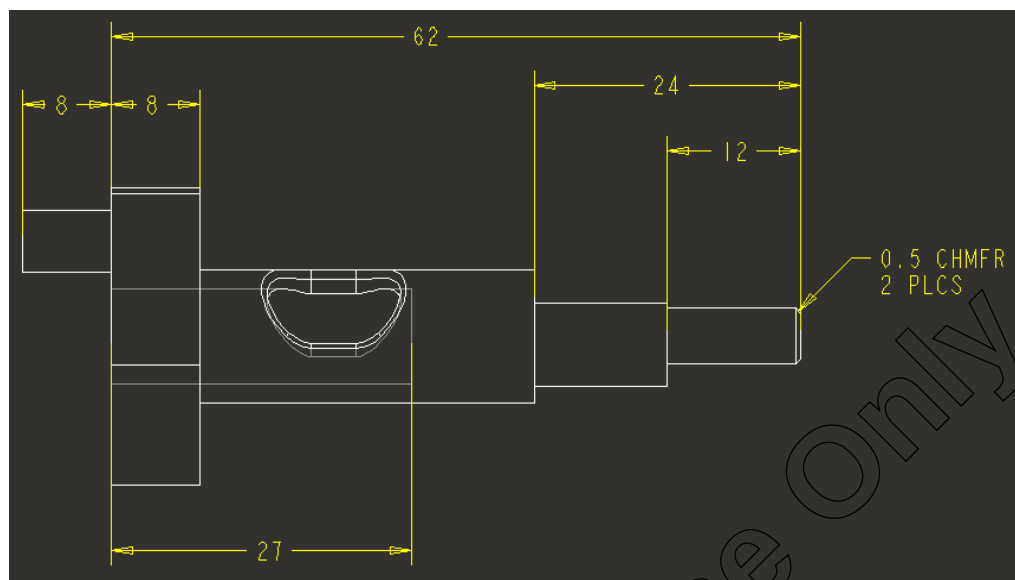


Figure 21: Dimensions on the General View

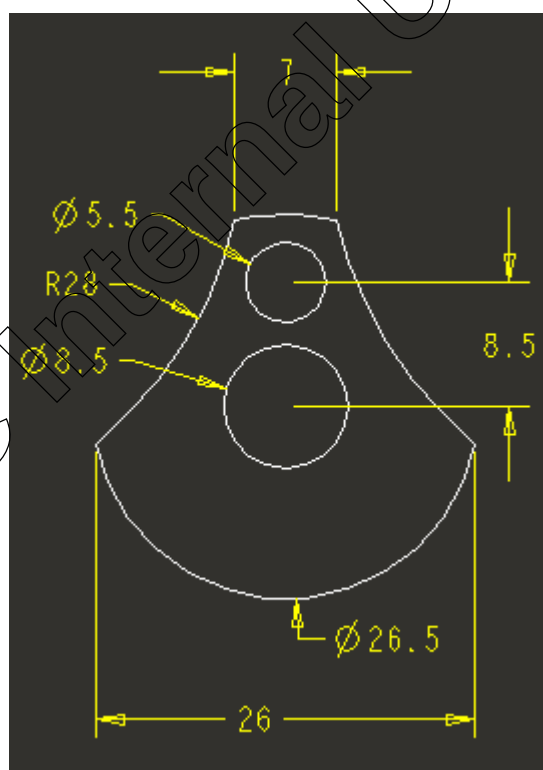


Figure 22: Dimensions on the Left Projection View

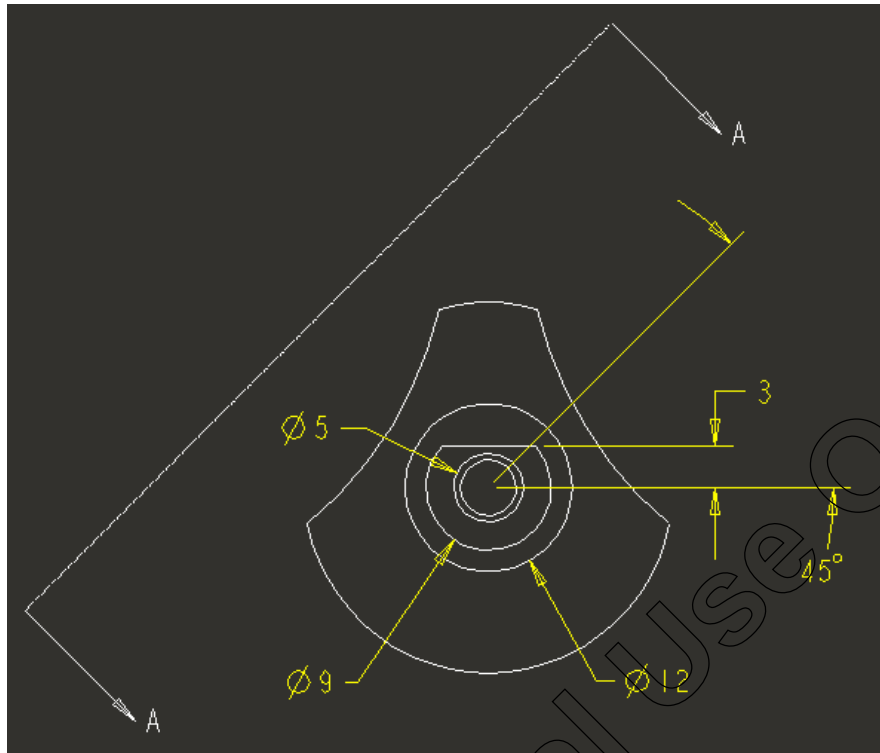


Figure 23: Dimensions on the Right Projection View

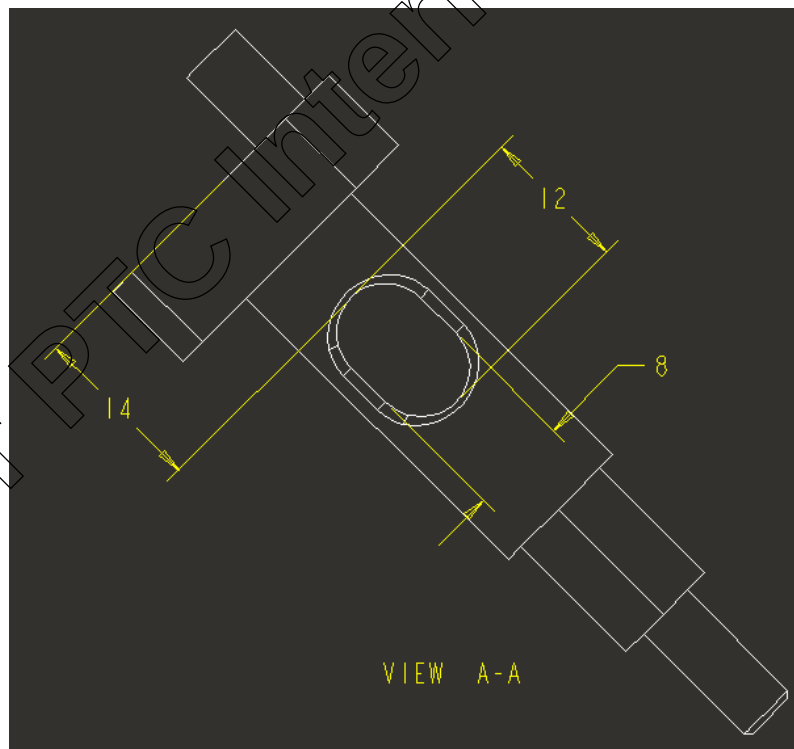


Figure 24: Dimensions on the Auxiliary View

3. The drawing should appear as shown in the following figure.

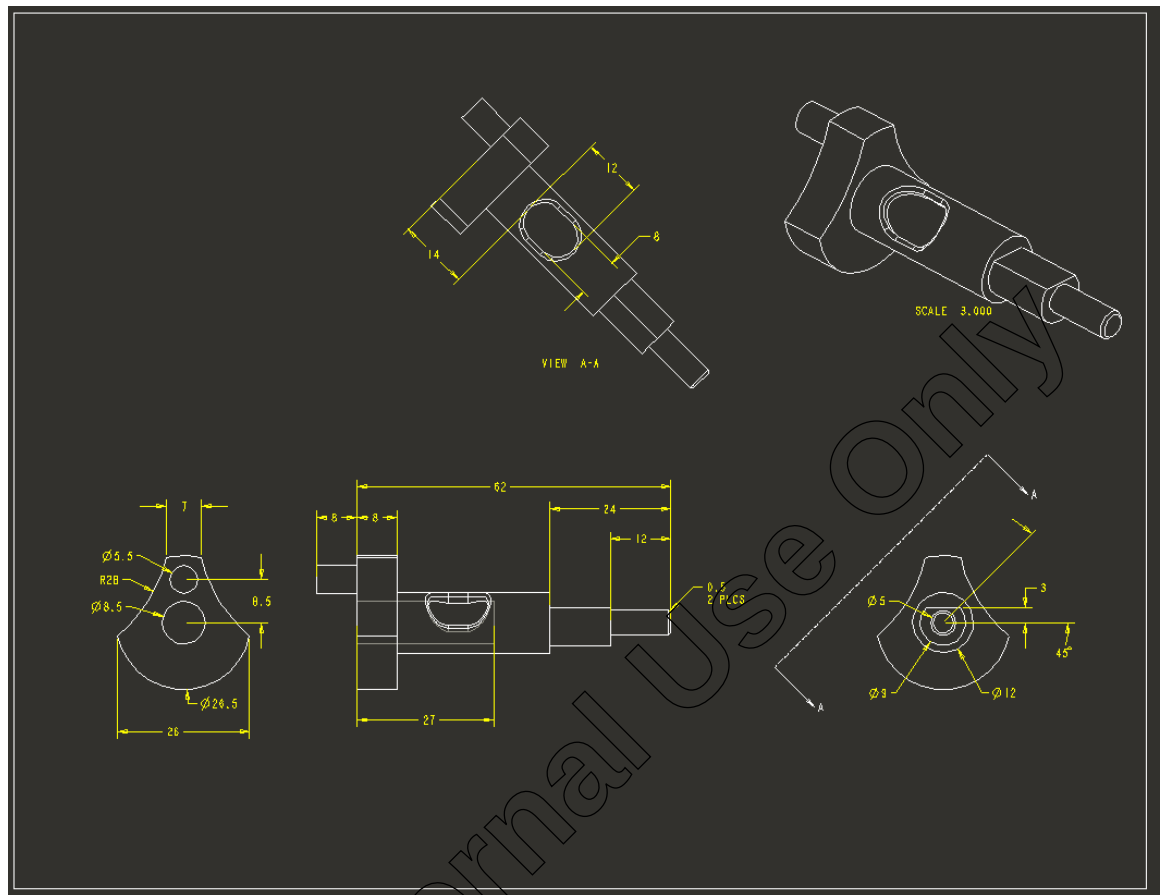


Figure 25: Dimensioning Complete

Step 7. (OPTIONAL) Apply a drawing format to all sheets.

1. Navigate to Sheet 1.
2. Ensure that AC-40 is the active drawing model.
3. Apply the **C_format_generic.frm** to the drawing sheet. The format is located in your PROJECT folder. Enter your name at the prompt.

Note:

If the project folder is not available, you can apply your company's C size format.

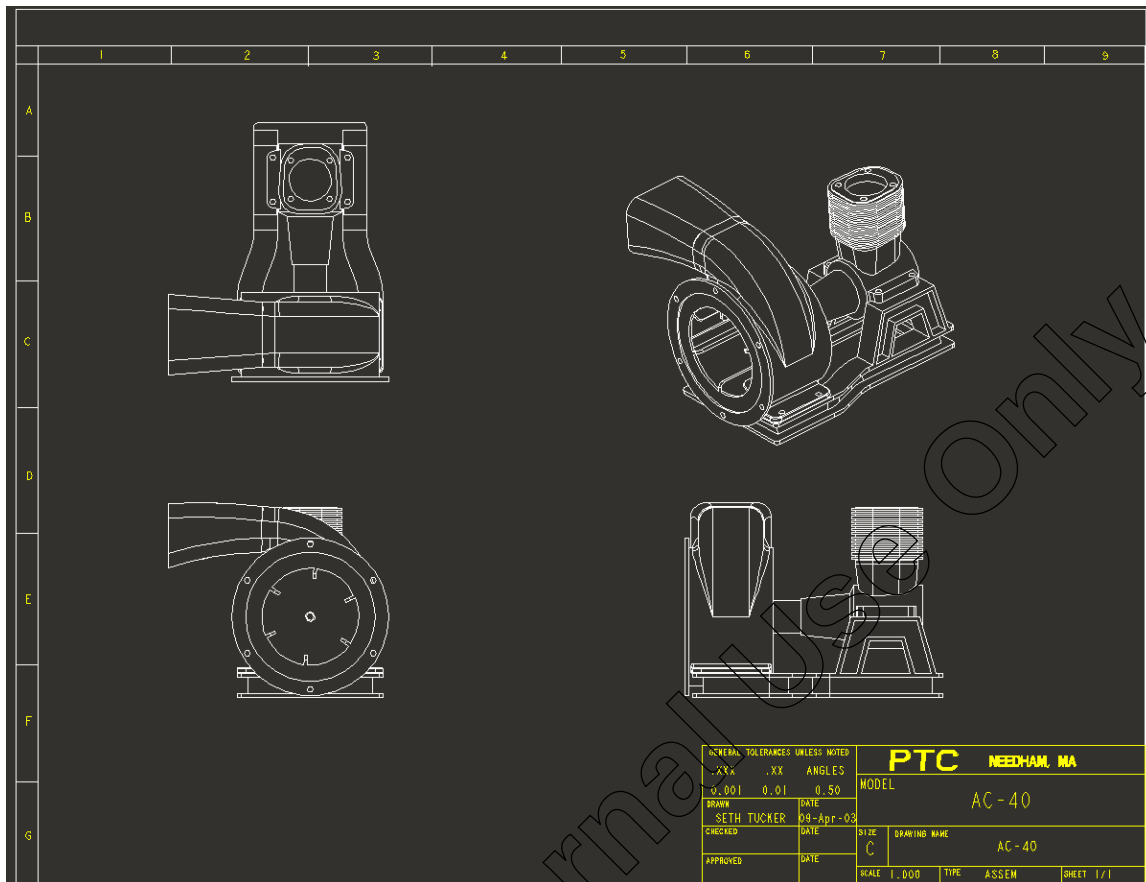


Figure 26: Applying a Drawing Format

4. Navigate to Sheet 2.
5. Ensure that ENGINE_BLOCK is the active drawing model.
6. Apply the **C_format_generic.frm** to the drawing sheet. Enter your name at the prompt.
7. Navigate to Sheet 3.
8. Ensure that the CRANKSHAFT is the active drawing model.
9. Apply the **C_format_generic.frm** to the drawing sheet. Enter your name at the prompt.
10. The drawing is now complete. Save the drawing and close the window.

This completes the exercise.

Objective 3: Completing the Engine Block part

Step 1. Open the model and create the exhaust features.

1. Open ENGINE_BLOCK.PRT.

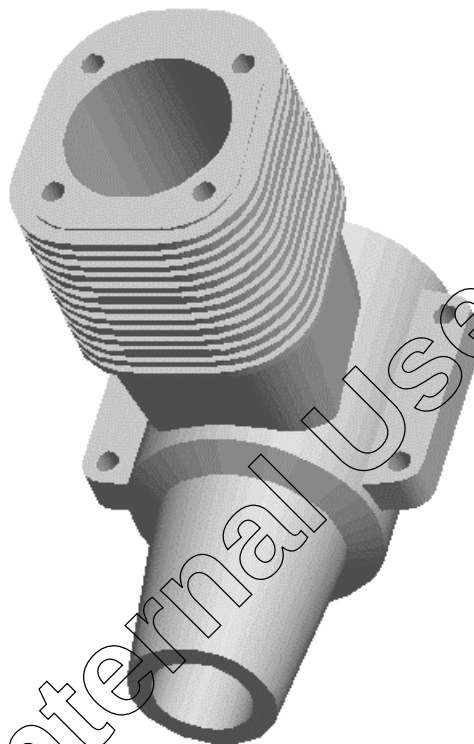


Figure 27: Engine Block

2. Create a datum plane offset from TOP as shown in the following figure. Rename the plane to EXH_HEIGHT.

Design Intent:

This plane will be used in the construction of multiple features.

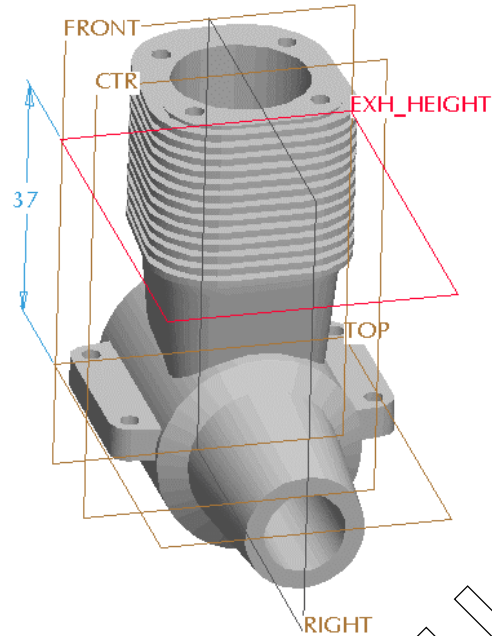


Figure 28: Creating a Datum Plane

3. Suppress the pattern of cooling fins.

Design Intent:

Suppressing the pattern reduces clutter on the screen when creating the next feature.

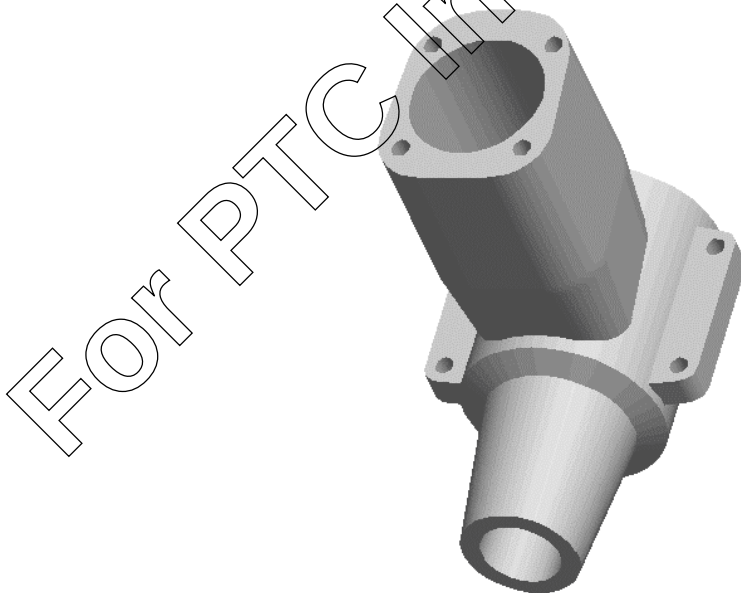


Figure 29: Pattern Suppressed

4. Create a protrusion, and sketch on the RIGHT plane. Sketch axis points on the center of each arc. Refer to the following figure.

Design Intent:

The sketch should be oriented such that the TOP plane is horizontal and flip the view direction such that it is viewed from the left. Also, holes will later be placed on the axes.

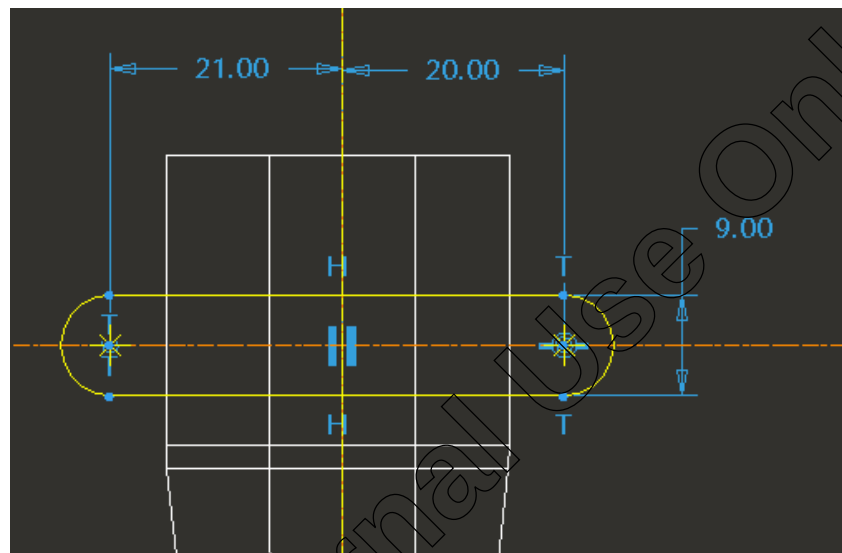


Figure 30: Creating a Sketch

5. Specify a depth as shown in the following figure.

Design Intent:

The depth should be dimensioned from the RIGHT plane.

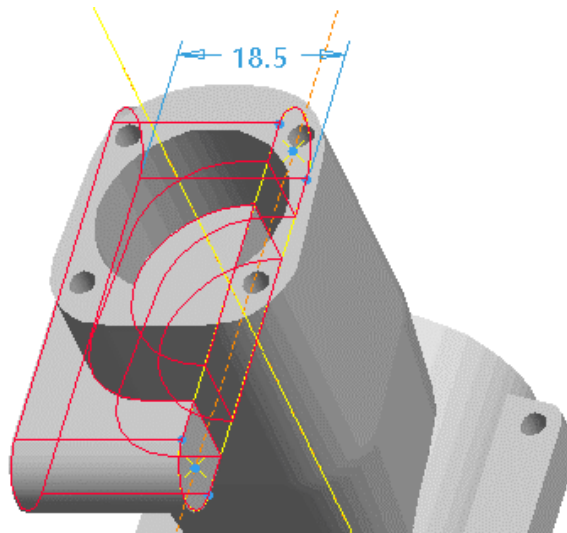


Figure 31: Creating a Protrusion

6. Rename the protrusion to EXH_MOUNT.

Step 2. Resume and Reorder Features. Then create bolt holes for the MUFFLER.PRT.

1. Show suppressed features in the model tree. Resume the pattern of cooling fins.
2. Reorder the BORE hole to be directly after the EXH_MOUNT feature.

Design Intent:

The BORE feature needs to come after the protrusion in order to hollow out the cylinder.

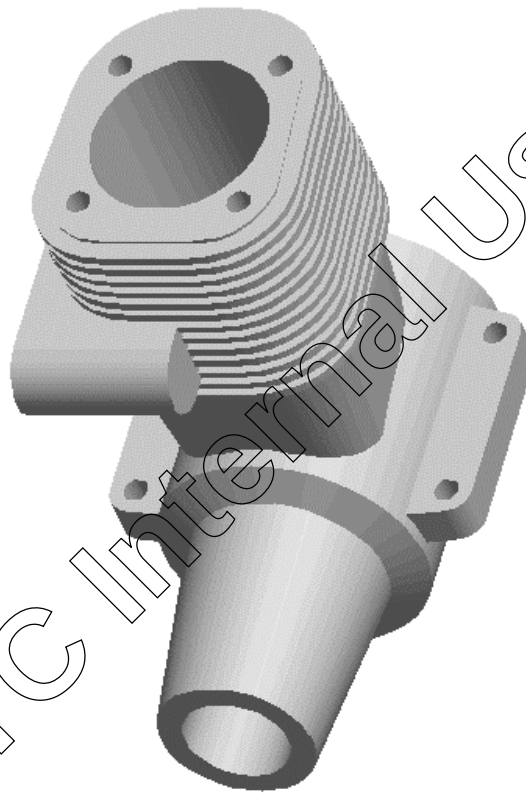


Figure 32: Features Resumed and Reordered

3. Create two holes, as shown in the following figures.

Design Intent:

The holes should be coaxial, and their depths should extend through the next surface.

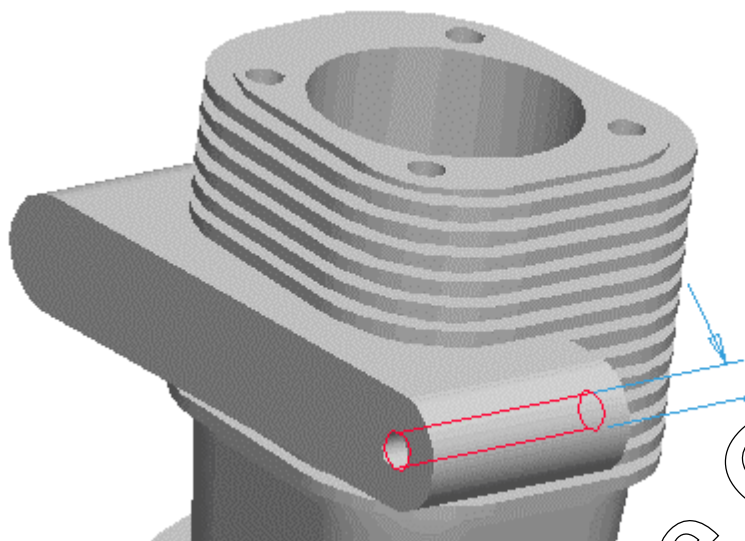


Figure 33: Creating a Hole on the Front

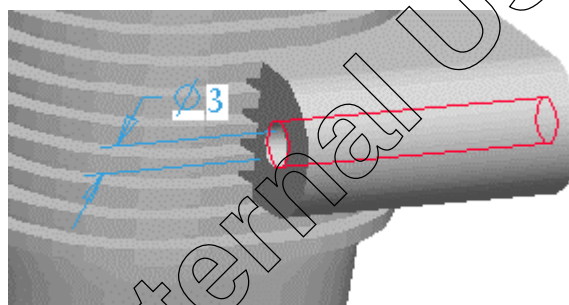


Figure 34: Creating a Hole on the Back

Step 3. Create a cut to allow the exhaust gases to flow from the cylinder.

1. Create an extruded cut on the EXH_HEIGHT plane. Sketch as shown in the following figure.

Design Intent:

The sketch should reference the cylinder bore and the EXH_MOUNT feature. Orient the sketch so that the FRONT plane is horizontal.

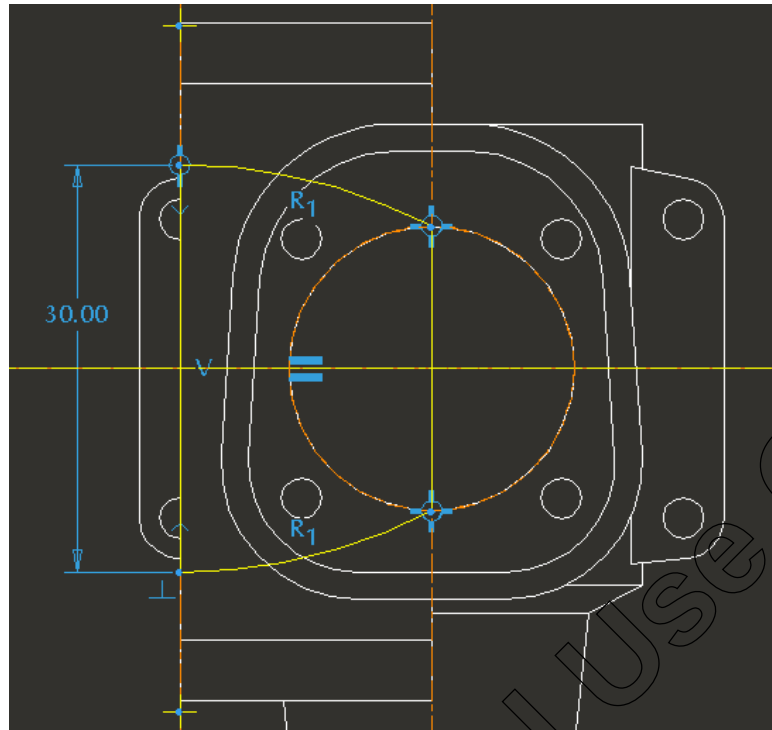


Figure 35: Creating a Sketch

2. Specify a depth as shown in the following figure.

Design Intent:

The flange should be symmetric about the EXH_HEIGHT plane.

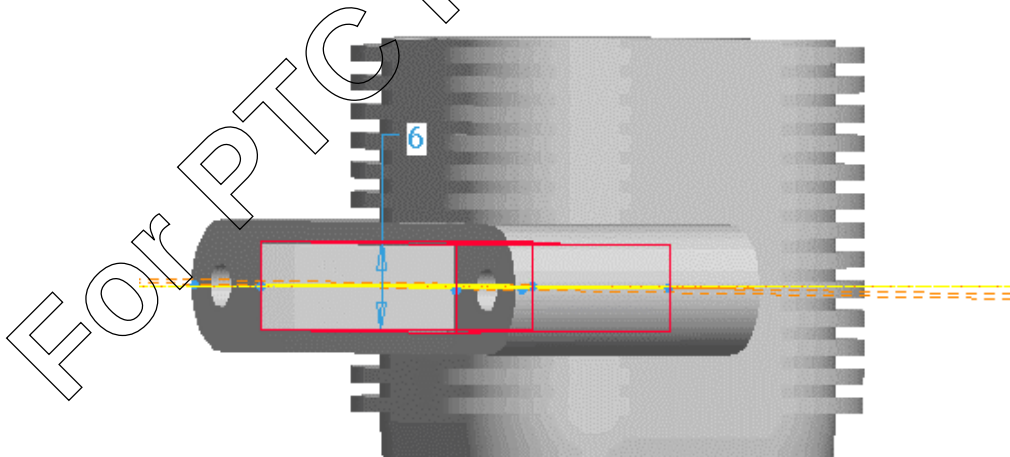


Figure 36: Creating a Cut

Step 4. Create features for the CARBURETOR.PRT mount.

1. Start the Extrude tool, and then create an axis as shown in the following figure.

Design Intent:

The axis should be dimensioned from the CTR and TOP planes.

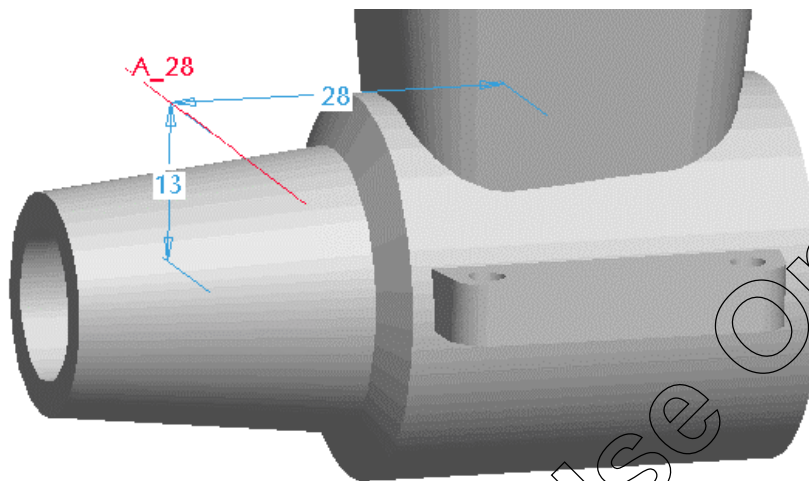


Figure 37: Creating an Axis

2. Create a datum plane through the previous axis and **22** degrees from TOP, as shown in the following figure.

Design Intent:

This plane will be used as a sketching plane for the following feature.

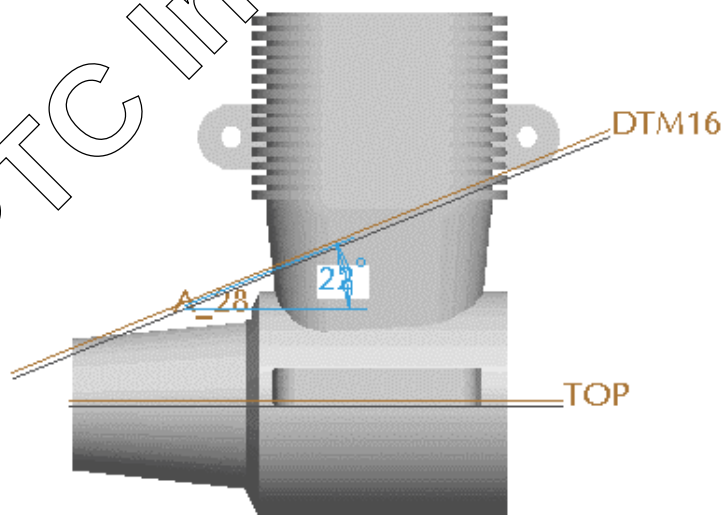


Figure 38: Creating a Datum Plane

3. Select the previous plane as the sketching plane. Sketch an axis point at the center of the sketch. Reference the previous axis and the RIGHT plane. Refer to the following figure.

Design Intent:

The sketch should be symmetric. A hole will be created using the axis.

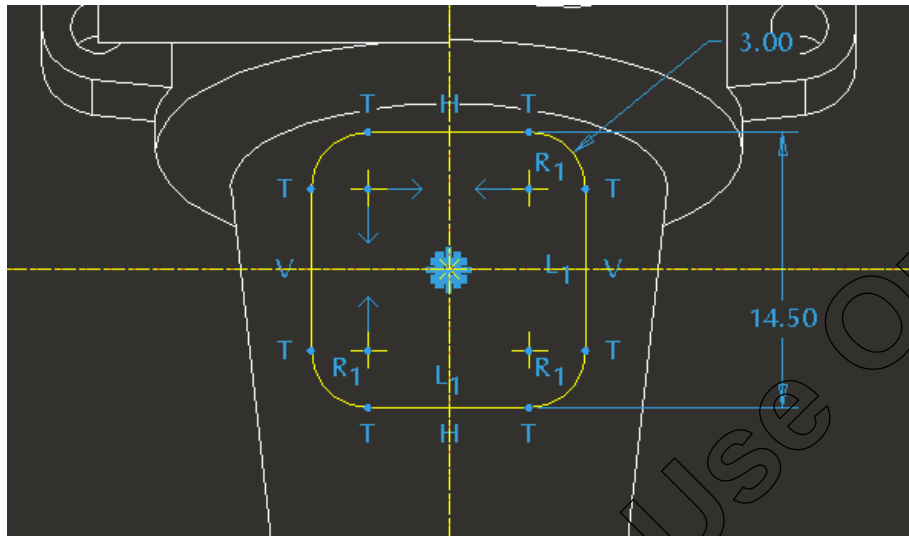


Figure 39: Creating a Sketch

4. Specify a 2-sided depth, extruding **5.0** upward, and downward onto the part surface. Refer to the following figure.

Design Intent:

The feature should contour to the conical surface.

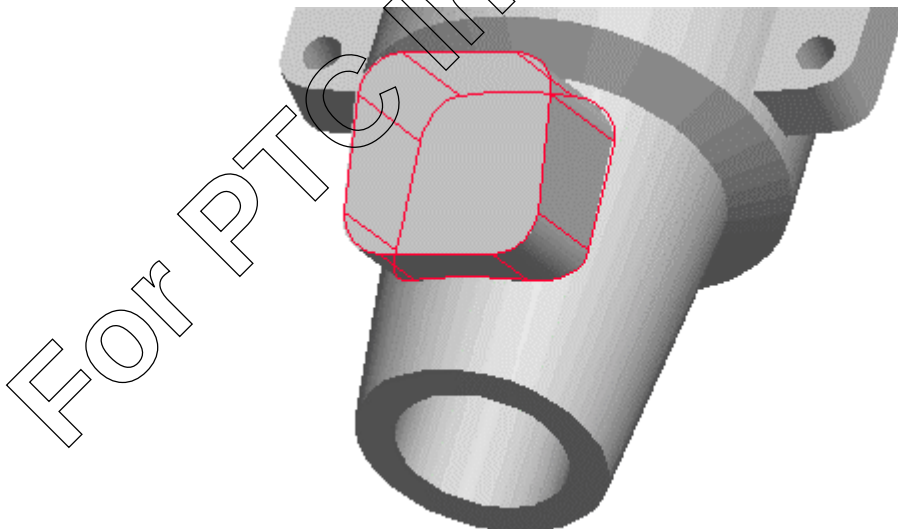


Figure 40: Creating a Protrusion

5. Rename the group to CARB_MOUNT.

Step 5. Create a hole for the intake gases to flow into the crankcase.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and its depth should extend through the next surface.

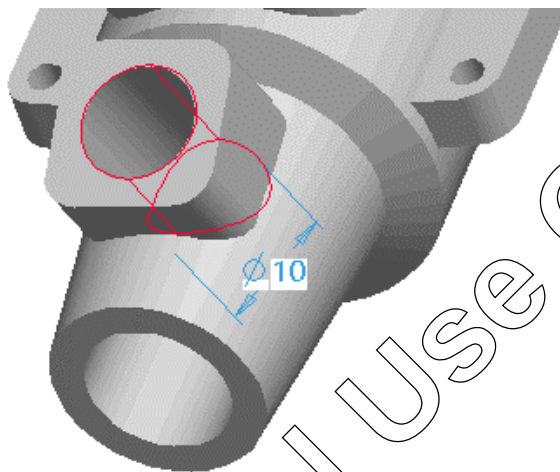


Figure 41: Creating a Hole

Step 6. Create a hole for bolting on the CARBURATOR.PRT.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and should extend through the entire model in both directions.

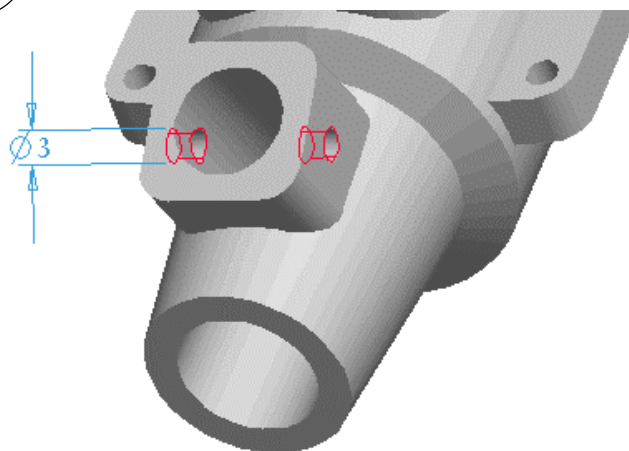


Figure 42: Creating a Hole

2. Expand the CARB_MOUNT group. Select the two previous holes and drag them into the bottom of the group.

Step 7. Create mounting tabs for the ENGINE_COVER.PRT.

1. Start the Extrude tool. Then create a datum plane through the main axis and at an angle to TOP.

Design Intent:

This plane will be used as a reference plane for the following feature.

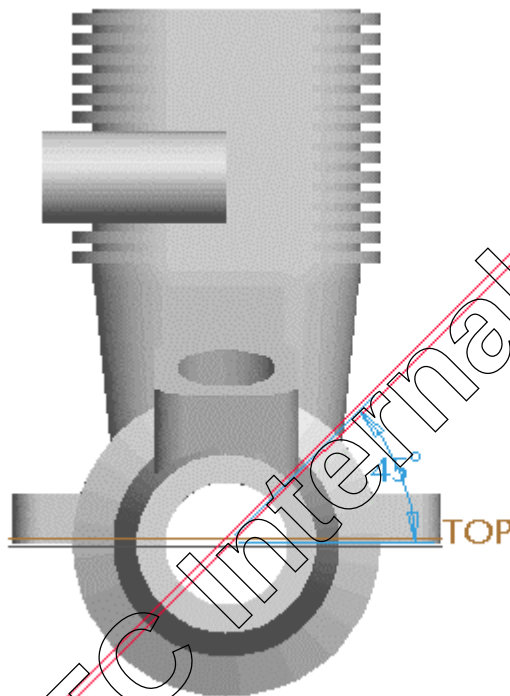


Figure 43: Creating a Datum Plane

2. Sketch on the thin rear surface of the model. Sketch as shown below, referencing the cylindrical surface. Place an axis point at the center of the arc.

Design Intent:

The sketch should be oriented such that the previous angled datum plane is horizontal. The sketch is also symmetric about the horizontal reference.

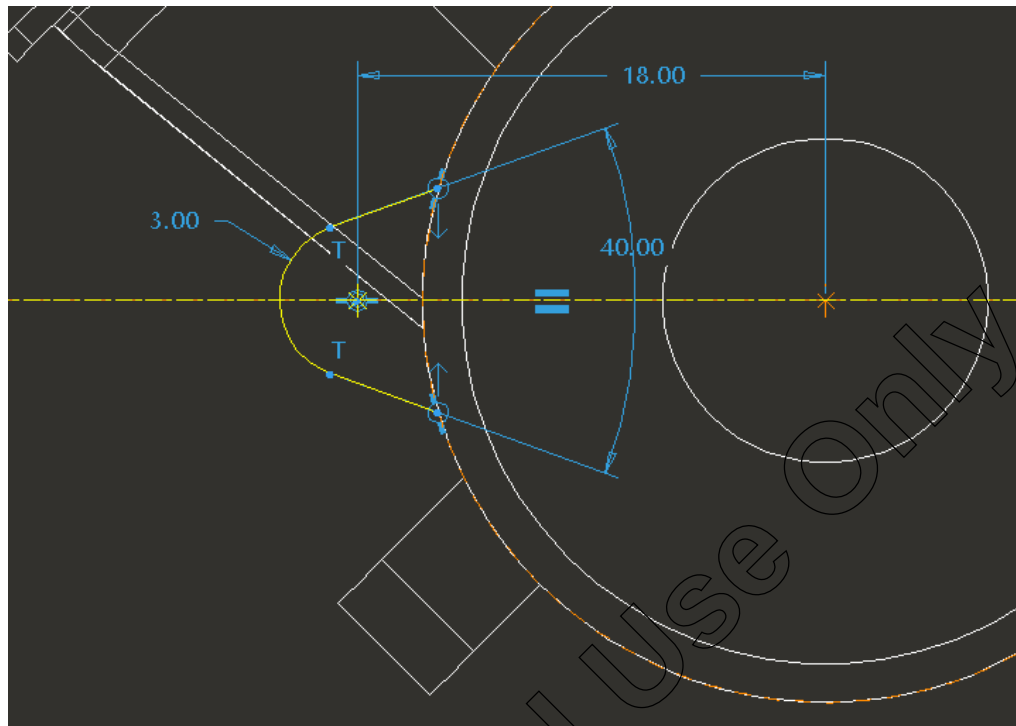


Figure 44: Creating a Sketch

3. Specify a depth as shown in the following figure.

Design Intent:

The depth should extend from the rear of the model forward.

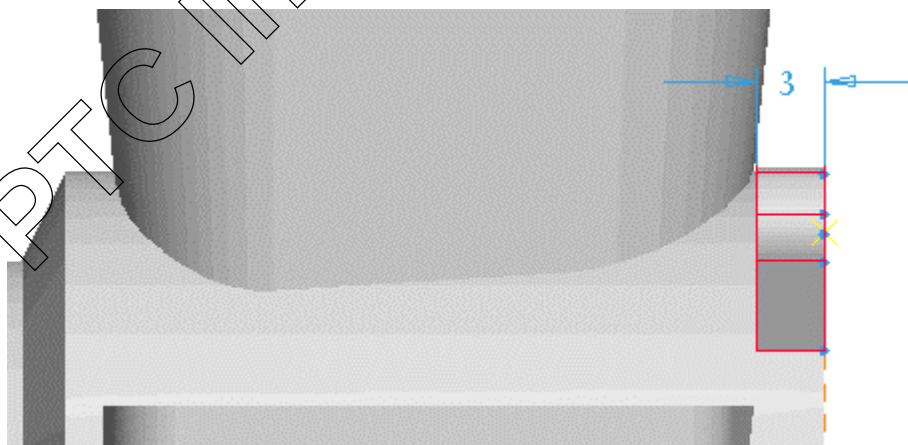


Figure 45: Creating a Protrusion.

4. Rename the group to **TAB**.
5. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and should extend through the next surface.

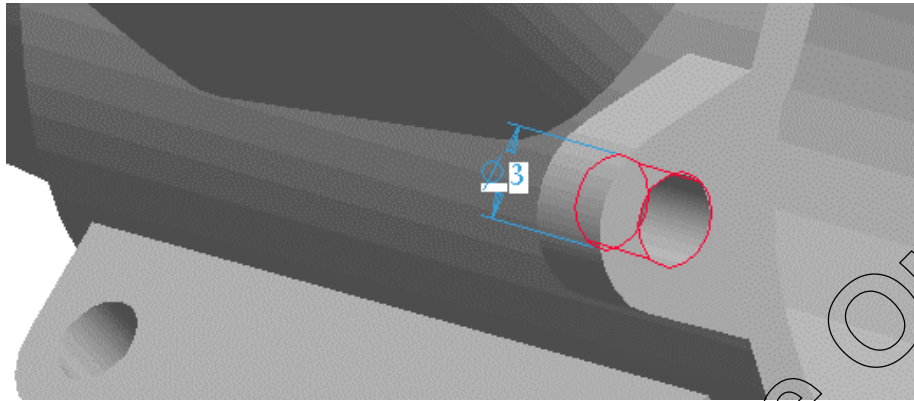


Figure 46: Creating a Hole

6. Create a round as shown in the following figure.

Design Intent:

This round sets up a tangent chain for the next round.

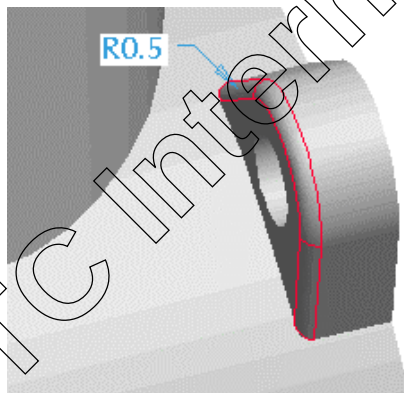


Figure 47: Creating a Round

7. Create a round as shown in the following figure.

Design Intent:

This round makes all three sides of the tab tangent to the model, leaving the rear surface planar.

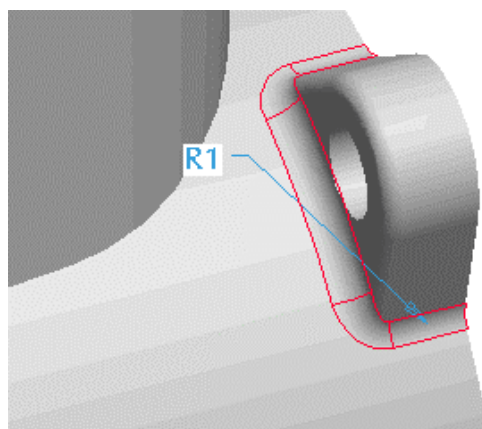


Figure 48: Creating a Round

8. Expand the TAB group. Select the two previous rounds and the hole and drag them into the bottom of the group.

Design Intent:

The group will allow patterning of multiple features at once.

9. Pattern the TAB group as shown in the following figure.

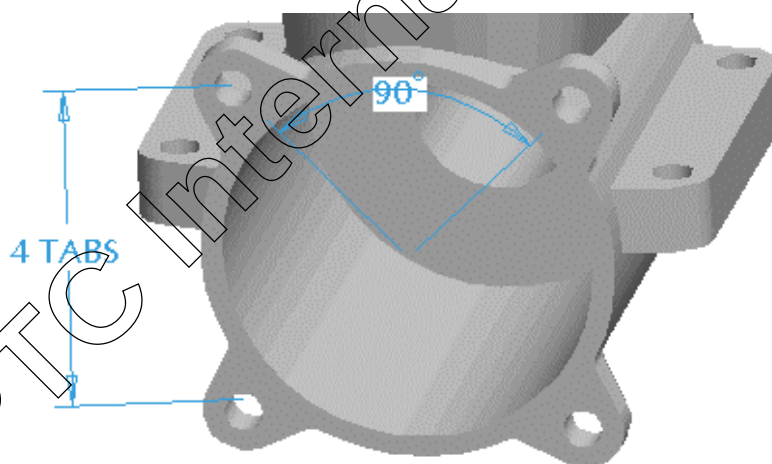


Figure 49: Pattern Created

Step 8. Create rounds to smooth out several sharp corners and to add strength.

1. Create a round on the four edges as shown in the following figure.

Design Intent:

This round sets up a tangent chain for the next round. All four edges should have the same radius.

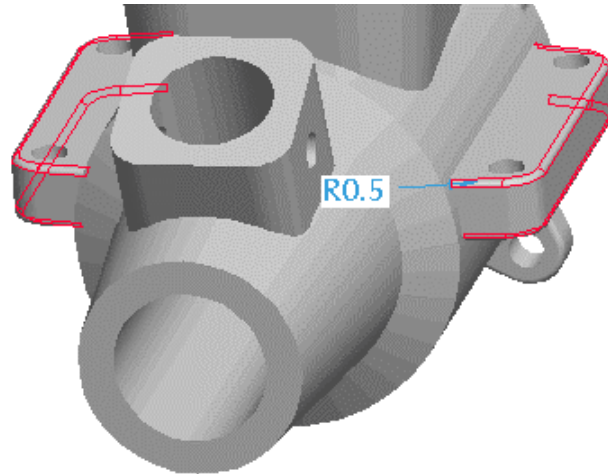


Figure 50: Creating a Round

2. Create a round on the four edges as shown in the following figure.

Design Intent:

All four edges should have the same radius.

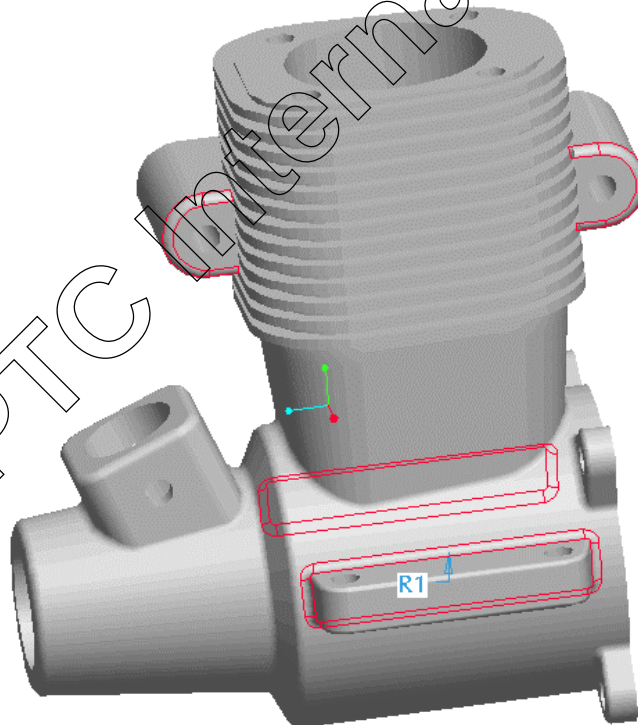


Figure 51: Creating a Round

3. Create a round on the six edges as shown in the following figure.

Design Intent:

All six edges should have the same radius.

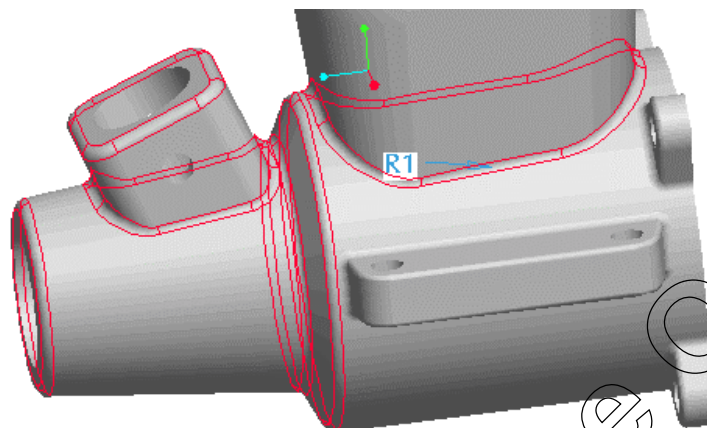


Figure 52: Creating a Round

Step 9. Create a Rib to strengthen the front portion of the part.

1. Start the Rib tool, and then create a datum plane as shown in the following figure.

Design Intent:

The datum plane should pass through the central axis and be 10 degrees from TOP. It will be used for a sketching plane for the next feature.

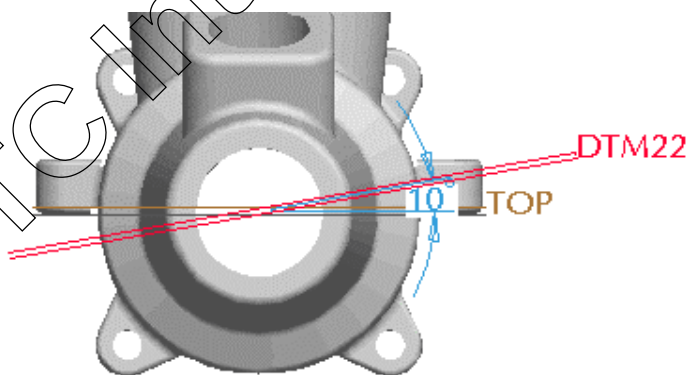


Figure 53: Creating a Datum Plane

2. Sketch on the previous plane, as shown in the following figure.

Design Intent:

Orient the sketch so that the CTR plane is vertical. The sketch should reference the original revolved protrusion.

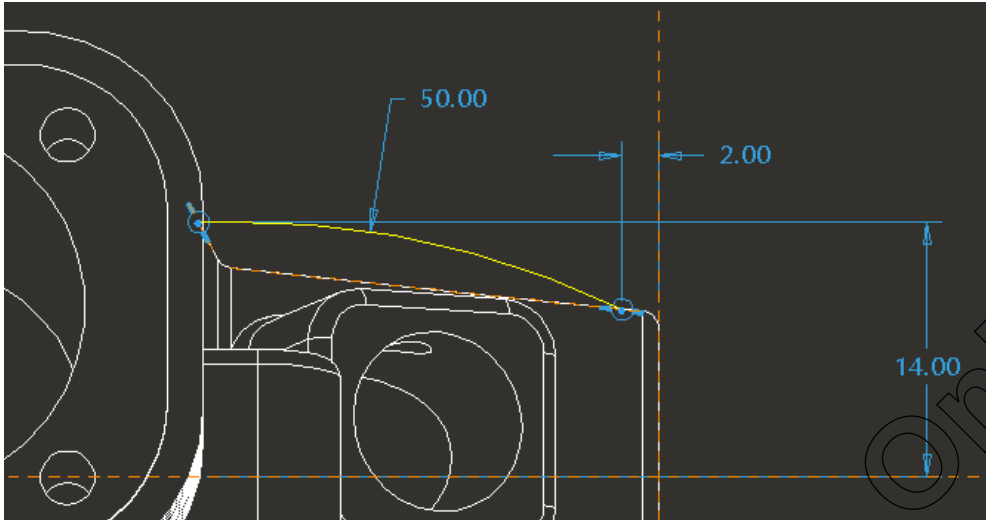


Figure 54: Creating a Sketch

3. Specify a thickness as shown in the following figure.

Design Intent:

The depth should extend symmetric from the sketching plane.

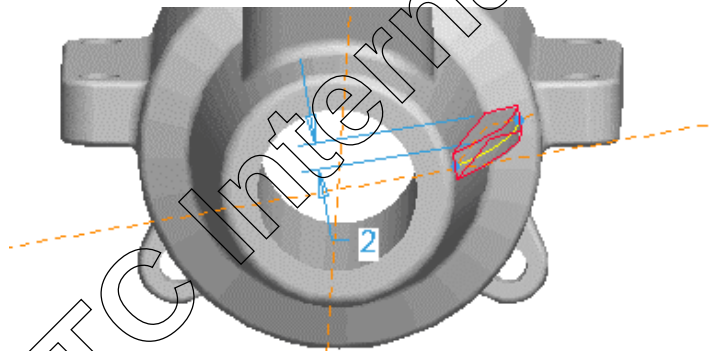


Figure 55: Creating a Rib

4. Rename the group to **RIB**.

Step 10. Create rounds on the rib.

1. Create a round as shown in the following figure.

Design Intent:

The round should be created so it doesn't require a radius dimension.

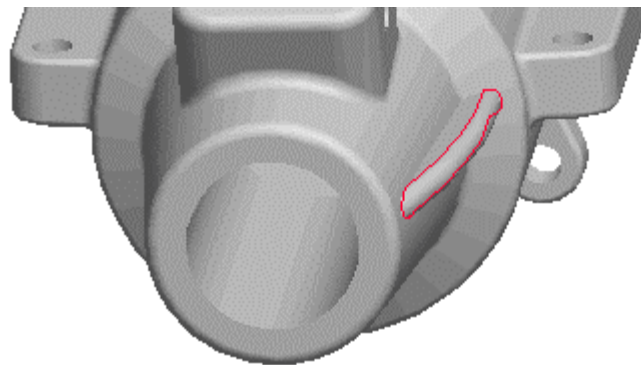


Figure 56: Creating a Full Round

2. Create a round as shown in the following figure.

Design Intent:

The previous rounds allow this round to form a complete loop.

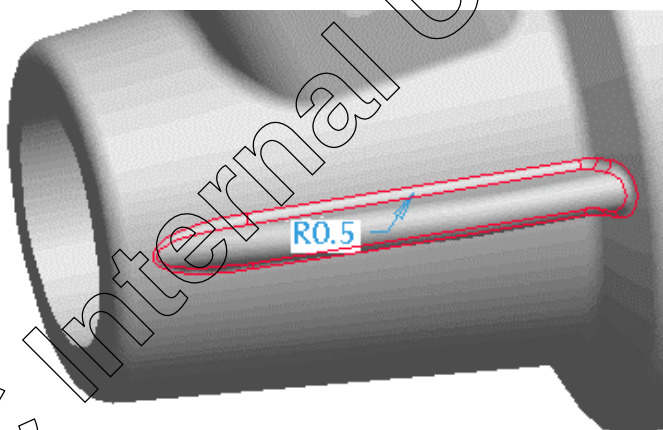


Figure 57: Creating a Round

3. Expand the RIB group. Select the two previous rounds and drag them into the bottom of the group.

Design Intent:

The group will allow copying multiple features at once.

Step 11. Duplicate the rib several times.

1. Copy the RIB group as shown in the following figure. Rename the group to **RIB2**.

Design Intent:

The copied rib should be rotated 20 degrees clockwise about the central axis, maintaining the same thickness as the original.

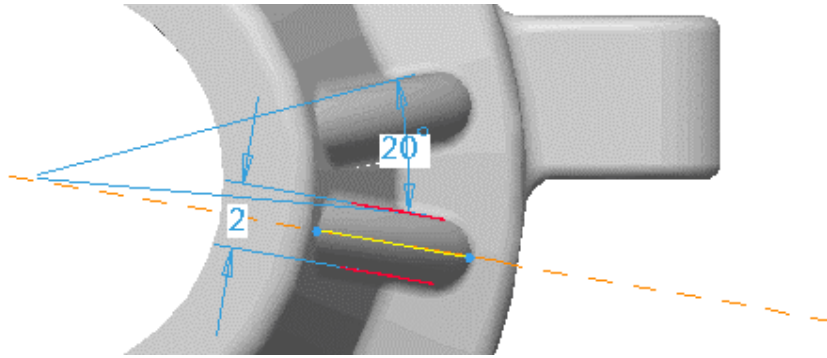


Figure 58: Rib Copy Rotated

2. Copy the original RIB group as shown in the following figure. Rename the group to **RIB3**.

Design Intent:

The copied rib should be rotated 100 degrees clockwise about the central axis, entering a greater thickness than the original.

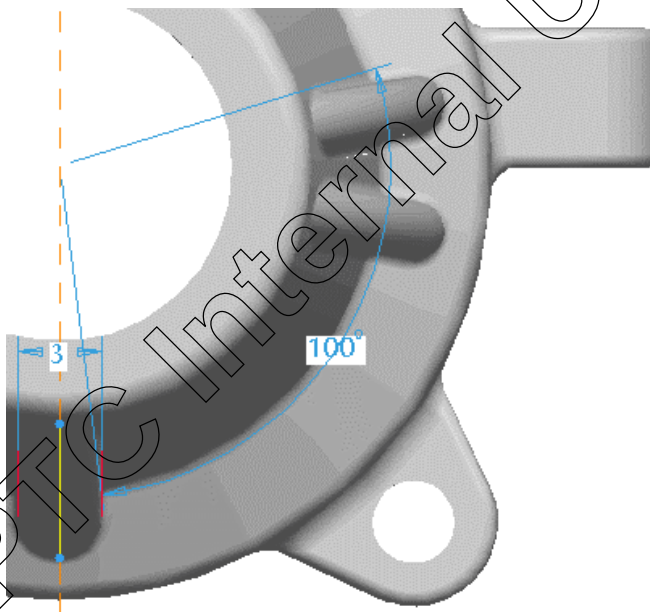


Figure 59: Rib Copy Rotated

3. Mirror the original 2 ribs about the RIGHT datum plane as shown in the following figure. Rename the group to **RIB4-5**.

Design Intent:

These ribs should maintain the same thickness as the first two ribs.

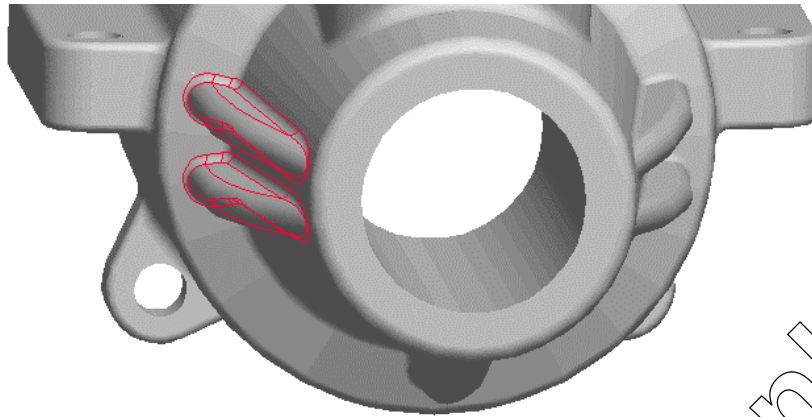


Figure 60: Ribs Copy Mirrored

4. The model should appear as shown in the following figure.

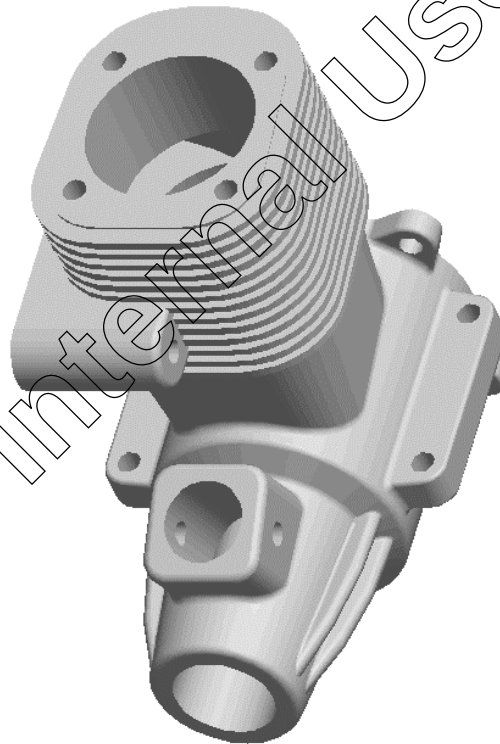


Figure 61: Completed ENGINE_BLOCK.PRT

5. Open the AC-40 drawing. Observe the changes in the ENGINE_BLOCK.PRT views on sheets 1 and 2.
 6. Save the drawing and close the window.
 7. This model is now complete. Save the model and close the window.
- This completes the exercise.

Summary

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

- Create Mechanism Connections.
- Create a Drawing.
- Complete the Engine Block.

For PTC Internal Use Only

Managing Assemblies

Introduction

Assemblies can be created, simplified and organized using various assembly management techniques.

Simplified representations enable you to manage large assemblies by removing unneeded details from your computer's graphics and RAM memory.

Skeletons enable you to capture design intent using reference geometry such as axes, curves, planes, and surfaces. This information can be shared with part models and referenced when creating features. One benefit of using skeletons is that as they are updated, the part models automatically update, too.

View orientations provide you with zoom and rotational settings that make it easy to achieve a desired view. Explode states enable you to capture assembly parts in various states of assembly/disassembly. These states can be easily referenced when creating drawings and assembly/disassembly procedures.

Objectives

After completing this module, you will be able to:

- Create style and simplified representations of assembly models.
- Design models using skeletons.
- Create view orientations of assembly models.
- Create exploded states of assembly models.
- Describe how managing assemblies enables you to manage parent/child relationships.



Instructor Preparation

Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** the following materials:

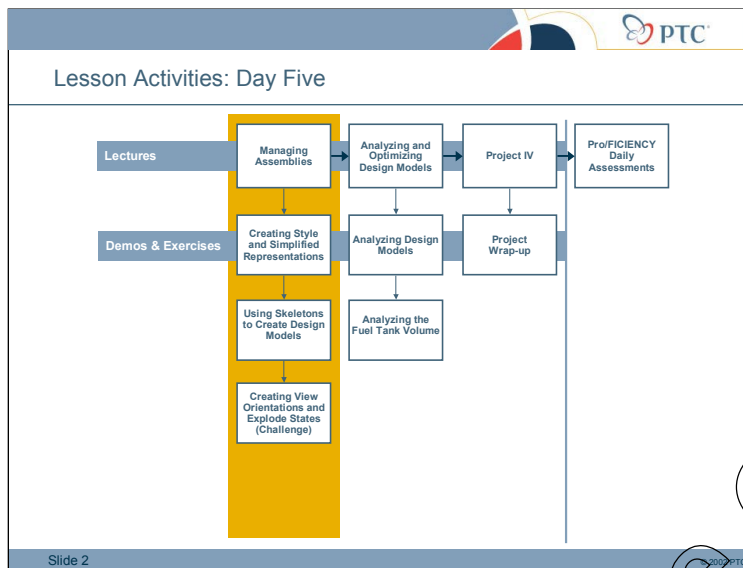
- General Information

- 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 Foundation

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

/GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN.



Duration

- Lecture: 10 mins
- Demos (2): 40 mins
- Labs (2): 40 mins
- Total: 1 hr 30 mins

PTC

Objectives

After completing this module, you will be able to:

- Create style and simplified representations of assembly models.
- Design models using skeletons.
- Create view orientations of assembly models.
- Create exploded states of assembly models.
- Describe how managing assemblies enables you to manage parent/child relationships.

Slide 3

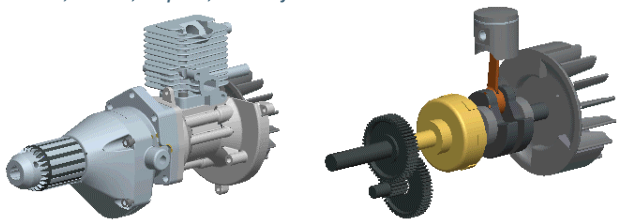
- You reduce large assembly file size by creating shrinkwraps.
- Shrinkwraps are a representation of a part or an assembly that captures the shape of the model.
- Shrinkwraps increase your computer's performance by reducing part and assembly file size up to 90%.
- You use top-down design techniques to create part models by referencing information contained in skeletons.
- Skeletons provide you with a collection of reference geometry such as axes, curves, planes, and surfaces.
- Skeletons enable you to create solid models that assembly correctly.

PTC

Creating Simplified Representations of Assembly Models

Creating Simplified Representations

- Simplify design models
 - *Increase on-screen manipulation speed*
 - *Customize an assembly for a specific task*
 - *Remove components without affecting parent/child relationships*
- Specify a representation.
 - *Master, Exclude, Graphics, Geometry*



Slide 4

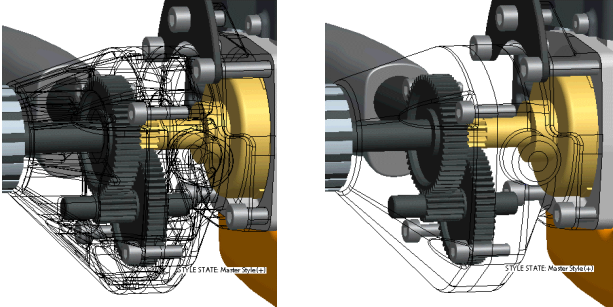
- A Simplified Representation is used to manage large assemblies so that the assemblies do not impair your computer's performance.
- Simplified representations accomplish this by removing unneeded information from the display or your computer's RAM memory.
- You use simplified representations when you have an assembly that is causing your computer to work slower than acceptable when processing rotations, pans, and zooms, or when working with other Pro/ENGINEER Wildfire functionality.
- Simplified representations enable you to increase your productivity by increasing your computer's performance.
- Simplified representations also can be tailored to specific tasks such as assembling a component.

PTC

Creating Style Representations of Assembly Models

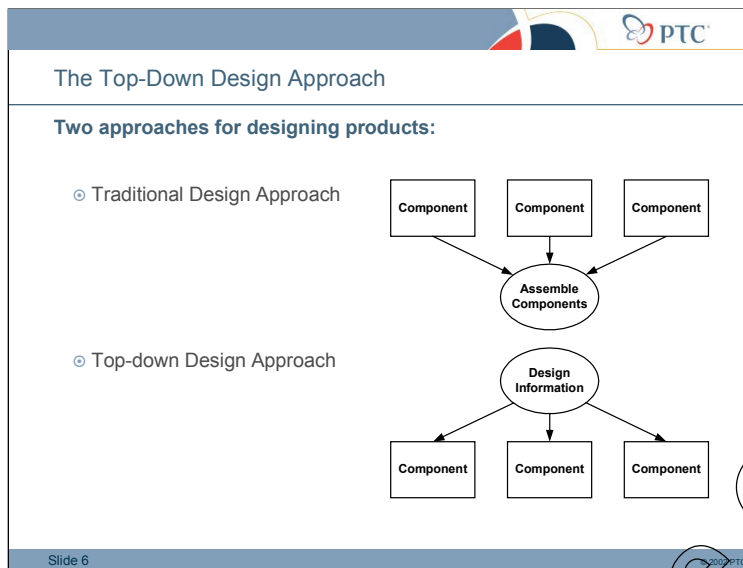
Creating Style Representations

- Style representations define different display states for components.
 - *Wireframe, Shaded, Hidden, No Hidden*

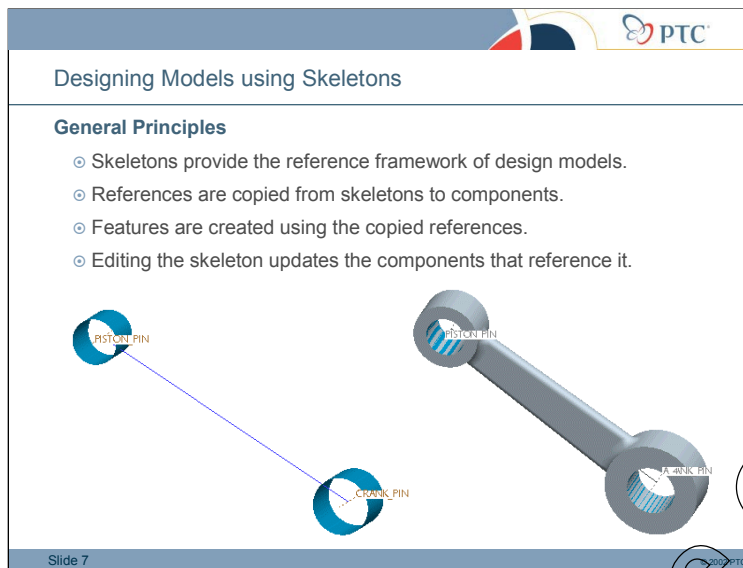


Slide 5

- A Style Representation is a view state that contains saved information about specific view settings of models. These view settings can be either Wireframe, Shaded, Hidden, or No Hidden Lines.
- You use Style Representations to remove graphics that are unnecessary or undesired to your current work tasks.
- Style representations enable you to configure the view state of specific models.



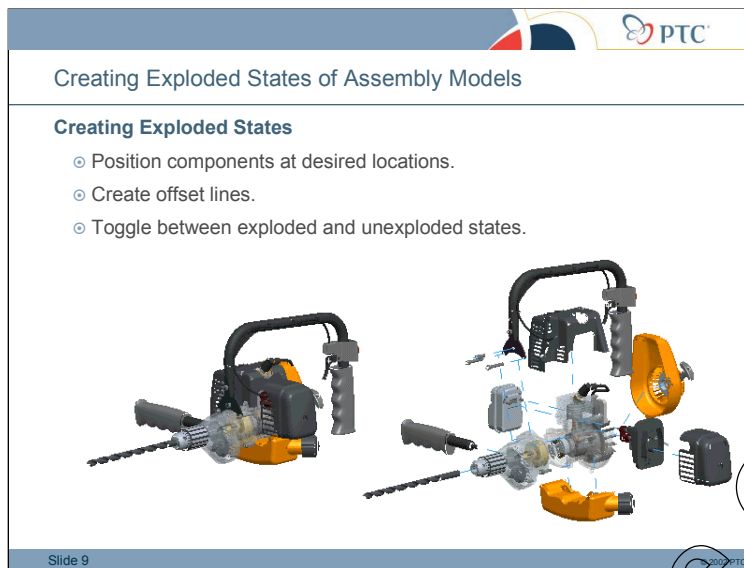
- Using the traditional assembly design approach, an engineer designs individual components independent of the assembly. The designer places components in subassemblies, and then brings those subassemblies together to develop the top-level assembly.
- After creating the assemblies, a designer often discovers that the models do not meet the design criteria (for example, a critical interface on two models does not match). This must be corrected by manually adjusting each model.
- As an assembly grows, detecting these inconsistencies and correcting them can take a lot of time. If significant changes occur that affect many components, such as changing the overall width of the design, the designer must manually identify and modify each component to make the adjustment.
- A design originates as a concept, and then gradually evolves into a complete product with individual parts and subassemblies modeled in Pro/ENGINEER Wildfire.
- The designer treats the components as part of a system from the beginning, considering the interactions between components.
- The term “top-down design” refers to the method of placing critical information in a high-level location, and then communicating that information to the lower levels of the product structure.
- As the design develops, more specific information becomes available and is incorporated into the design.
- By capturing the overall design information in one centralized location, it becomes easier to make significant design changes.
- Because the information is contained in one place—with all subcomponents looking to that location—if you change this information, the system updates all other components.



- In Pro/ENGINEER Wildfire, a skeleton acts as a 3-D layout of the assembly. Skeletons are the initial components in the model tree and the assembly.
- Skeletons act as a central location for storing design information relating to the assembly. The system conveys this information to all of the components in the assembly. When the skeleton changes, the components that use this information also change.
- Skeletons provide a location for design data that is critical to the entire assembly. You avoid having to “search” through all of the components for the critical dimensions and parameters. They are contained in one place.
- Because skeletons are fairly simple parts, the system quickly regenerates and displays them. You can visualize the entire assembly with only that part, without having to work on the entire assembly.
- Skeletons assist in assembling complex mechanisms. For example, if two components depend on each other for their position, the order in which you place the components in the assembly prevents you from capturing this dependence when placing them. The first component cannot obtain any placement information from the second one, as the second one does not yet exist in the assembly. To capture this information, you can create a skeleton, and then assemble both models to the skeleton.
- Because the other components in the assembly are located using references from the skeleton, you can minimize interdependencies between the major components of the assembly.
- Because the components are assembled to the skeleton and not to each other, you can place finished components in the assembly even if other components are not yet complete. By placing design information common to several components in the skeleton, and then distributing this information to the individual parts, you can minimize the external references created between the parts in the assembly.
- The **Reference Control** tool in Pro/ENGINEER Wildfire has a special setting which allows you to reference only the assembly skeleton.



- Views enable you to save a specific orientation and zoom setting of a part or an assembly.
- By changing to a saved view, Pro/ENGINEER reverts to the saved views orientation and zoom setting.
- If you find that you consistently need to view a certain feature or part, you can use either the mouse or the ORIENTATION dialog box to create the view in the Pro/ENGINEER graphics window, and then save the view using the View Manager.
- Saved views will save you unnecessary pan, zoom, and rotating time. Saved views can also be placed directly into drawings.



- Explode states enable you to view your assembly in various views that illustrate assembly/disassembly procedures.
- You use explode states to capture the assembly and disassembly processes for documentation purposes.
- Explode states enable you to easily save assembly/disassembly views that can later be directly imported into drawings for documentation purposes.

PTC

Parent/Child Relationships – Managing Assemblies

Allows you to manage and control the flow of parent/child relationships. Changes made to a skeleton model can update several components accordingly.

- Top-Down Design / Skeletons

Does not affect parent/child relationships

- Simplified Representations
- Style Representations
- View Orientations
- Explode States
 - Offset lines will be removed if parents are deleted

Slide 10

Note to Instructors:

Use this slide to teach students how the functionality they will use in the exercises is affected by and affects parent/child relationships. Feel free to use the previous slides in the module to describe the parent/child relationships if students have trouble understanding it during the demo.

Simplified representations are the only means to remove models from working memory without affecting the parent/child relationships. This provides you with a powerful means to work on large assemblies by excluding parts and sub-assemblies or by removing graphic and geometric data.

Top-down design and skeletons help you manage parent/child relationships by keeping all (or as many as you desire) of the parent child relationships within the skeletal curves, planes, axes, and surfaces.

View orientations and explode states belong to the parent part or assembly.



Graphics Window Best Practices

Best practices when managing models in the Pro/ENGINEER Wildfire graphics window.

- ◉ Hide/Unhide
 - Conveniently remove features or components from the graphics window to make short term changes.
- ◉ Blank/Unblank (Layers)
 - Conveniently remove datum features of parts and assemblies from the graphics window on a long term basis.
 - Conveniently select multiple features or components.
- ◉ Suppress/Resume
 - Use to manage Parent/Child relationships between features while creating and editing models.
 - Do NOT use suppress/resume to simplify the display of parts or assemblies in the graphics window.
- ◉ Simplified Representations
 - Conveniently remove parts and assemblies from the graphics window and RAM memory on a long term basis.
 - Conveniently work on parts and assemblies on an as-needed basis.
- ◉ Style Representations
 - Conveniently display selected parts and assemblies as wireframe, hidden line, no hidden line, and solid regardless of Pro/ENGINEER Wildfire shading status.

Slide 11

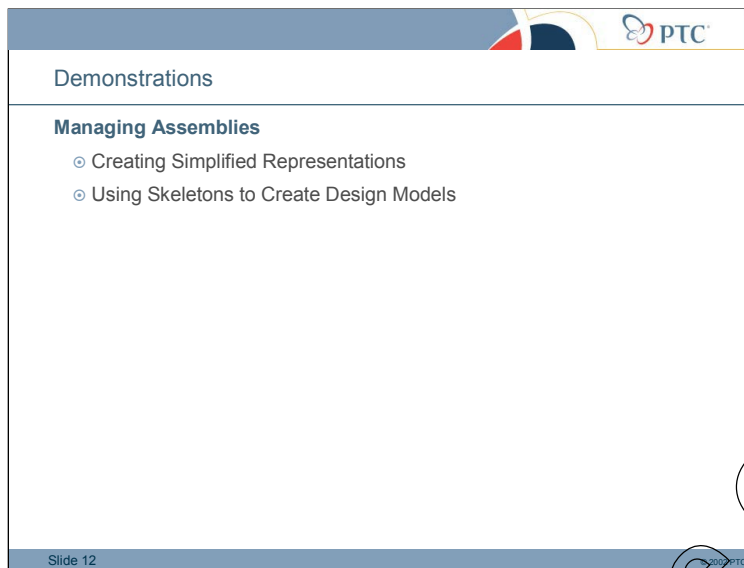
Note to Instructors:

Use this slide to review best practices with students. While this slide is repeated from the previous module, it is important to note that the Simplified Representations bullet is new.

These best practices are a reference for students. Note that in some cases, these best practices were not followed when creating this course simply because of the time that would be required to implement these practices. Students should understand this point as it is important!

Best practices when managing models in the Pro/ENGINEER Wildfire graphics window.

- **Hide/Unhide**
 - *Recommend that you use only to temporarily remove features or components in the graphics window in order to make a change. Unhide the features or components once you have completed the change.*
- **Blank/Unblank (Layers)**
 - *Recommend that you use layers to remove datum features of parts and assemblies in the graphics window on a long term basis.*
 - *Recommend that you use layers to select multiple features or components.*
- **Suppress/Resume**
 - *Use to manage Parent/Child relationships between features while creating and editing models. For example, you may want to temporarily suppress a series of features to be able to more easily create new features, and then resume the suppressed features. You may also want to temporarily suppress a feature to try out a different feature in the same location. You then could resume or delete accordingly to achieve the desired result.*
 - *Recommend that you do not use suppress/resume to simplify the display of parts or assemblies in the graphics window because of Parent/Child relationships.*
- **Simplified Representations**
 - *Conveniently remove parts and assemblies from the graphics window and RAM memory on a long term basis.*
 - *Conveniently work on parts and assemblies on an as-needed basis.*
- **Style Representations**
 - *Conveniently display selected parts and assemblies as wireframe, hidden line, no hidden line, and solid regardless of Pro/ENGINEER Wildfire shading status.*



You will follow along while I demonstrate:

1. Creating Simplified Representations
2. Using Skeletons to Create Design Models

Then, you will practice the same steps in the laboratory exercises.

Summary

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

- Create style and simplified representations of assembly models.
- Design models using skeletons.
- Create view orientations of assembly models.
- Create exploded states of assembly models.
- Describe how managing assemblies enables you to manage parent/child relationships.

Referencing PTC Help for Module 17 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Simplified Representations	Assembly and Welds	About Creating Simplified Representations Example: To Create a Simplified Representation
Skeletons	Assembly and Welds	About Skeleton Models About Assembly Mode Functionality Tip: Allow Multiple Skeleton Replacement Copies
Top-Down Design	Assembly and Welds	About Geometry Features
View Orientations	Fundamentals	Orientation Dialog Box About Orienting the Model View About Saved Views
Explode States	Assembly and Welds	About Exploded Views About Using Offset Lines in Exploded Views

Module 17 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Creating Style and Simplified Representations

Introduction

In this demonstration, we create simplified representations of parts and assemblies. Simplified representations reduce the graphical and geometry information on parts that are not part of the work routine. By reducing the amount of displayed information, we are able to increase the computer's graphical performance.

Also, simplified representations increase graphical performance because fewer models are stored in RAM memory, which increases computational time when rotating/panning/zooming.

Objectives

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

- Create style representations.
- Create simplified representations.
- Assemble components using simplified representations.

Scenario

Marketing needs additional graphics of the drill with the common pieces removed. You are assigned to remove certain parts using style representations so that Marketing can create their necessary graphics.

Additionally, several interns have asked whether there is a method to increase their computer performance when working on the Drill assembly. In particular, they would appreciate several efficient simplified representations to work with when completing their job tasks. However, they

cannot create the simplified representations because their computer stops responding when running the full model. Hence, you are assigned to create the following simplified representations.

Step 1. Create a style representation for the gearbox.

Marketing needs more graphics of the drill with the common pieces removed. In this exercise, we remove certain parts using style representations.

1. If necessary, open DRILL.ASM.
2. If necessary, click **View > Explode > Unexplode View**.
3. Orient to the 3D_1 view.
4. If necessary, disable the model transparency.
5. Select GEARBOX_FRONT.PRT and GEARBOX_REAR.PRT. Click **View > Display Style > Wireframe**.
6. Orient to the GEARBOX view.

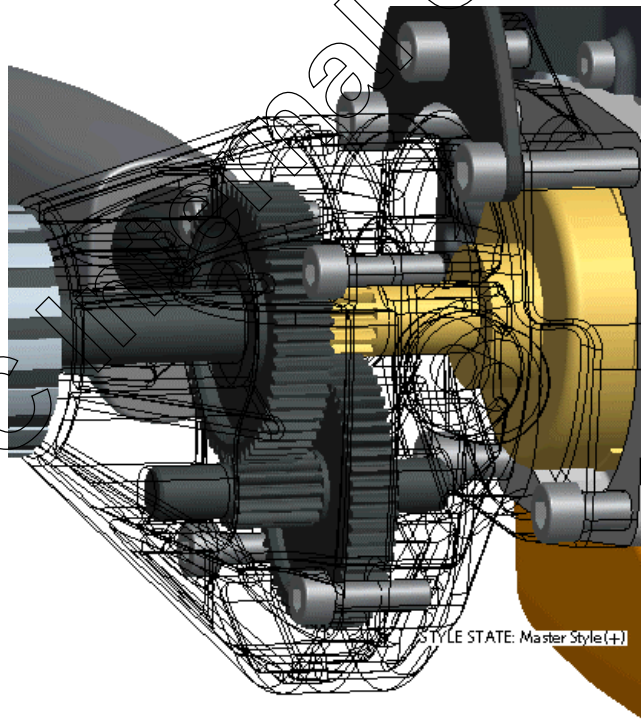


Figure 1: Setting Components to Wireframe

7. Select GEARBOX_FRONT.PRT and GEARBOX_REAR.PRT. Click **View > Display Style > Hidden Line**.
8. Select GEARBOX_FRONT.PRT and GEARBOX_REAR.PRT. Click **View > Display Style > No Hidden**.

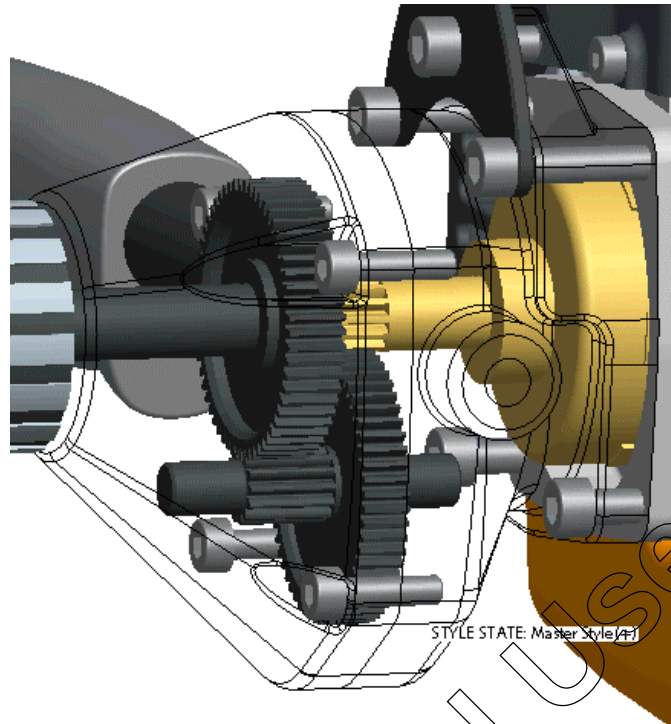


Figure 2: Setting Components to No Hidden Lines

9. Start the View Manager, and select the **STYLE** tab. Click **New**. Enter **Gearbox_No_Hidden**.
10. Double-click on **Master Style**. Leave the View Manager open for the next step.

Step 2. Create a Style Representation for the covers.

Instructor: Skip this step for the demo.

1. Using the model tree, select **ENGINE_COVER.PRT**, **RECOIL.ASM**, **AIR_FILTER_COVER.PRT**, and **CARBURETOR_PLATE.PRT**
2. Click **View > Display Style > No Hidden**. Repaint if necessary.
3. Using the View Manager, click **New**. Enter **Covers_No_Hidden**.

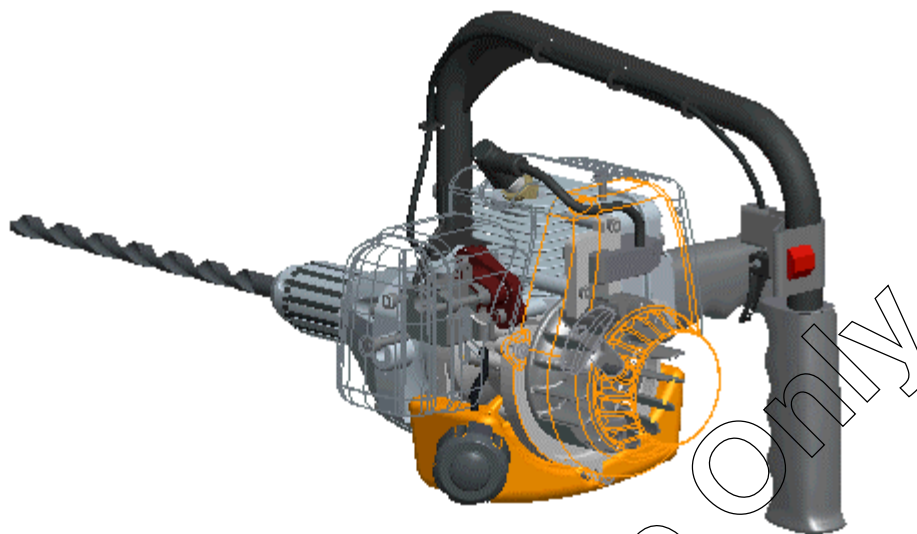


Figure 3: Setting Components to No Hidden Lines

4. Double-click on **Master Style**. Close the View Manager.
5. Save the model.

Step 3. Create the No Covers and No Hardware representations.

Simplified representations enable us to increase our graphical performance.

In this next step, we create several simplified reps and examine their effect on our computer by observing which specific components can be erased from RAM memory.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise, start Pro/ENGINEER Wildfire.
2. Set the working directory to: **C:\users\student\fast_track_330\module_17.**
3. Open DRILL.ASM.
4. Select ENGINE_COVER.PRT, RECOIL.ASM, AIR_FILTER_COVER.PRT, and CARBURETOR_PLATE.PRT.
5. Click **View > Representation > Exclude**.
6. Start the View Manager, and select the SIMP REP tab. Click **New**. Enter **No_Covers**.
7. Orient the model as shown in the following figure.



Figure 4: Creating the No_Covers Simplified Representation

8. Select the SIMP REP tab and double-click on **Master Rep**. Close the View Manager.
9. Using the Search tool, locate all components named BOLT. Click **OK** to select the models.
10. Click **View > Representation > Exclude**.
11. Using the View Manager, save a Simplified Rep named **No_Hardware**.
12. Orient the model as shown in the following figure.

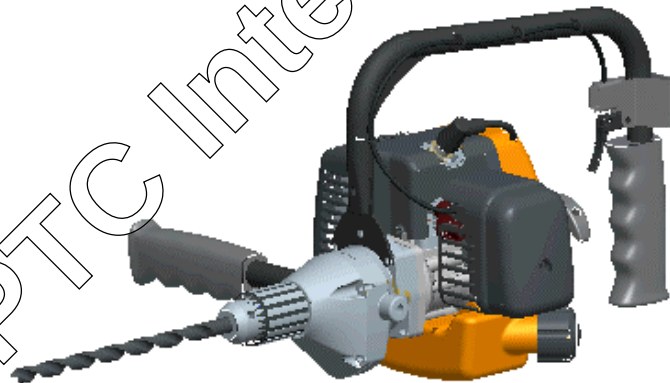


Figure 5: Creating the No_Hardware Simplified Representation

13. Double-click on **Master Rep**. Close the View Manager.

Step 4. Create the Powertrain representation.

1. Using the Search tool, locate all components with the **Include Submodels** check box cleared. Click **Ok** to select the models.
2. Press and hold CTRL and de-select ENGINE.ASM, GEARBOX_CHUCK.ASM. Then, select STANDARD_BIT.PRT.

3. Click **View > Representation > Exclude**. Refer to the following figure.
4. Use the View Manager to save a Simplified Rep named **Powertrain**. Close the View Manager.

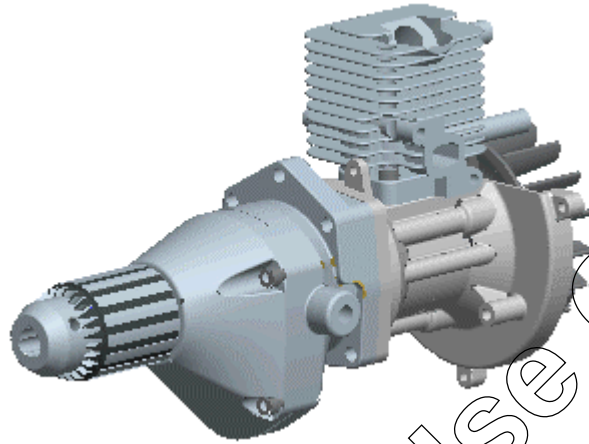


Figure 6: Creating the Powertrain Simplified Representation

Step 5. Create the Internal Rep.

Instructor: Skip this step for the demo.

1. Using the Search tool, locate and select all components named BOLT. Press and hold CTRL, and then select DRILL_CHUCK.ASM, GEARBOX_FRONT.PRT, GEARBOX_REAR.PRT, CYLINDER.PRT, ENG_BLOCK_FRONT.PRT, and ENG_BLOCK_REAR.PRT.
2. Click **View > Representation > Exclude**.
3. Use the View Manager to save a Simplified Rep named **Internal**.

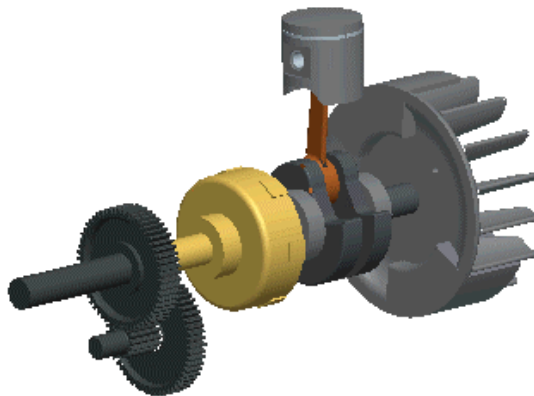


Figure 7: Creating the Internal Simplified Representation

4. Double-click on **Master Rep**. Close the View Manager.
5. Save the model.

Step 6. (OPTIONAL) Experiment with simplified representations.

Instructor: Skip this step for the demo.

1. Start the View Manager and activate the internal simplified rep. Click **Properties**, select CLUTCH.ASM, and set it to **Master Rep**.
2. Click **List**. Update the internal simplified rep. Close the View Manager.
3. Click **File > Erase > Not Displayed**. Note that several components could be removed from memory. Click **Ok**.
4. Start the View Manager and activate the **Master Rep**. Notice that components are being retrieved from disk.
5. Select the GEARBOX_FRONT.PRT and AIR_FILTER_COVER.PRT. Click **View > Representation > Graphics Only**. Using the View Manager, save the rep as **Graphics_Test**.

Note:

The display of components set to Graphics Rep is dependent on the setting of the save_model_display CONFIG.PRO option. By default, models are displayed in wireframe, but can be set to different levels of quality for shading.

6. Click **File > Erase > Not Displayed**. Note that these two components could be removed from memory. Click **Ok**.
7. Using the View Manager, activate the **Master Rep**. Notice that components are being retrieved from disk.
8. Using the model tree, activate HANDLE_MAIN.PRT. Notice that the feature tools in the toolbar are available, indicating that you could create features to the model while in this rep.
9. Activate DRILL.ASM. Select the HANDLE_MAIN.ASM. Click **View > Representation > Geometry Only**.
10. Select HANDLE_MAIN.PRT. Right-click and attempt to activate this model. Note that activate is not available since feature data has been removed.
11. Click **File > Erase > Not Displayed**. Note that this component could be removed from memory. Click **Ok**.
12. Using the View Manager, activate the **Master Rep**. Notice that components are being retrieved from disk.
13. Save the model.
14. Close the window.
15. Erase all models from memory.

Step 7. Assemble using simplified representations.

Instructor: Skip this step for the demo.

1. Erase all models from session. Click **File > Open**. Select DRILL.ASM, and then click **Open Rep**. Select the POWERTRAIN rep, and then click **Ok**.
2. Click **File > Erase > Not Displayed**. Notice that there are no components available to be erased.
3. Select GEARBOX_CHUCK.ASM and exclude it. Select CYLINDER.PRT, ENG_BLOCK_FRONT.PRT, and ENG_BLOCK_REAR.PRT, and set them to **Graphics Only**.
4. Select FLYWHEEL.PRT and CRANKSHAFT.PRT, and set them **Geometry Only**. Using the View Manager, save the rep as **Assemble_Ratchet**.
5. Click **File > Erase > Not Displayed**. Note that several components are available to be erased from memory. Click **Ok**.
6. Assemble RATCHET.PRT using an Insert constraint and Mate Coincident constraint, as shown in the following figure.

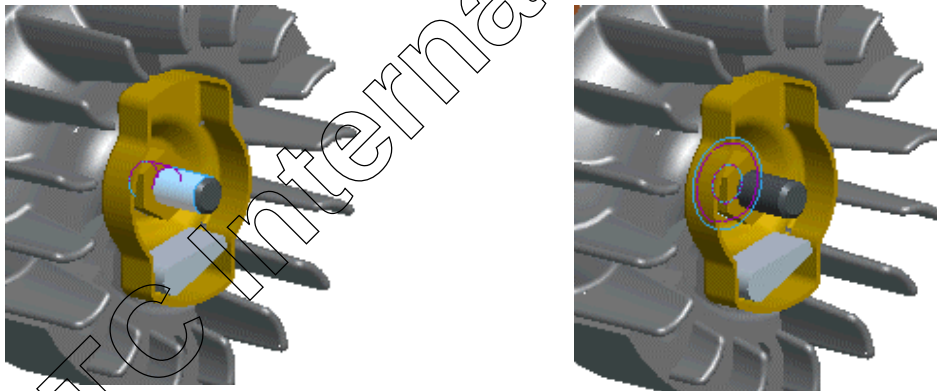


Figure 8: Assembling the RATCHET.PRT

7. Complete the component placement. Activate the **Master Rep**. Notice that components are being retrieved from disk.
8. Using the model tree, reorder RATCHET.PRT so that it comes immediately before RECOIL.ASM.
9. Save the model.
10. Close the window.
11. Erase all models from memory.

This completes the exercise.

Exercise 2: Using Skeletons to Design Models

Introduction

In this demonstration, we create a solid model based on information stored in a skeleton part. Skeletons enable us to create solid models that assembly correctly. They also provide a collection of reference geometry, such as axes, curves, planes, and surfaces.

Objective

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

- Create a solid model by referencing information stored in a skeleton model.

Scenario

You are assigned to create the connecting rod by using skeleton geometry designed by another engineer. You must use the reference information contained in the skeleton to design the connecting rod within the drill's engine assembly.

Step 1. Investigate properties of an existing skeleton.

In this first step, we examine the skeleton geometry that enables us to visually determine how the desired part will fit into the assembly. We open the skeleton part and note the references for our design. Finally, we create a copy geometry feature that enables us to copy references from the skeleton model to other models.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from memory. Otherwise, start Pro/ENGINEER Wildfire.
2. Set the working directory to: **C:\users\student\fast_track_330\module_17\Skeleton.**
3. Open DRILL.ASM.
4. Using the model tree, expand SKELETON.ASM and ENGINE.ASM.

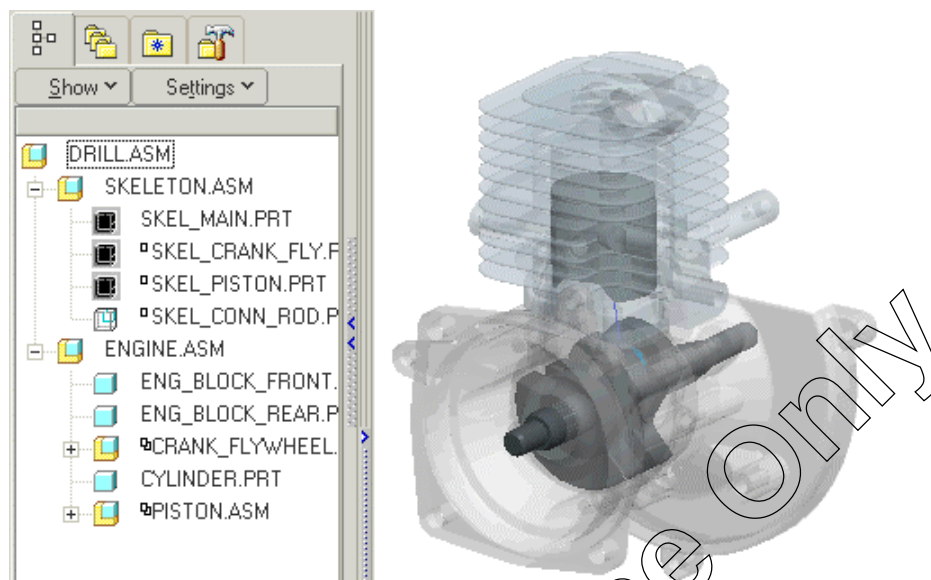


Figure 9: Viewing the DRILL.ASM

Note:

This assembly was created differently than the previous DRILL.ASM you created. Top-down design techniques were used to create skeleton models, which are assembled using mechanism connections in a sub-assembly. The skeleton models consist of datum and surface geometry for critical sizing and mounting point information. The skeleton geometry was then transferred to each solid part and used as references when creating the features. The “missing” connecting rod will be created using top-down design techniques.

5. Using the View Manager, orient to the **3D_1** view, and activate the **Outer_Invisible** style representation.

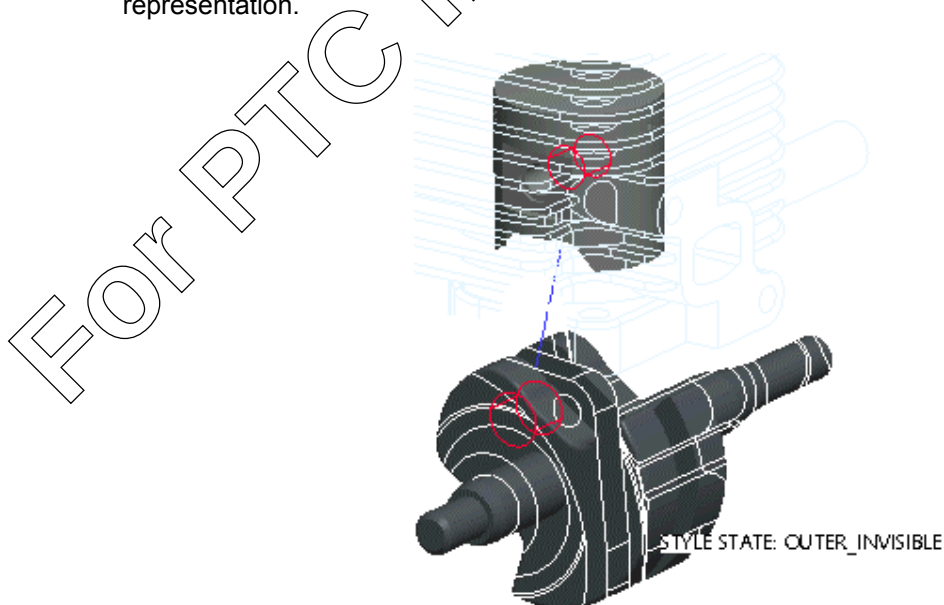


Figure 10: Outer Invisible Style Rep

6. Close the View Manager and enable Mechanism mode.
7. Using the model tree, expand the **Analyses** node. Select the **ENGINE** analysis, right-click and select **Run**. Note that since the mechanism was defined between the skeleton components, the solid part models do not have to exist in order to view the motion.
8. Enable Standard mode. Click **Yes** to discard the playback file.
9. Using the model tree, select SKEL_CONN_ROD.PRT and open it. Notice that it contains surfaces, curves, and axes.

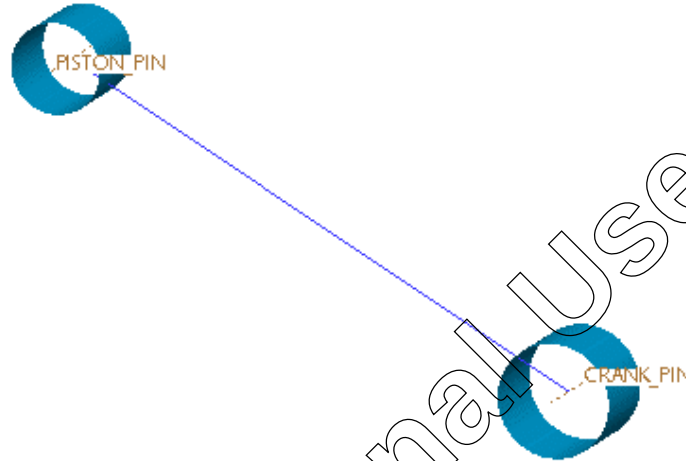


Figure 11: Examining SKEL_CONN_ROD.PRT

10. Close the window to return to DRILL.ASM.

Step 2. Transfer the skeleton geometry to the connecting rod part.

Now we have captured the references in a publish geometry feature. The next step is to create a new part model that imports the published information within the engine assembly. We reference this information to build the connecting rod features.

1. Activate ENGINE.ASM.
2. Click **Insert > Component > Create**. Select **Part**. Enter **Connecting_Rod** as the name. Click **Browse**. Open START.PRT. Click **OK**.
3. Display only the datum coordinate systems. Select the two visible coordinate systems to assemble CONNECTING_ROD.PRT to SKEL_CONN_ROD.PRT, using a Coord Sys constraint.

Note:

The underlying skeleton is created with mechanism connections. This means CONNECTING_ROD.PRT will inherit the mechanism motion from the skeleton component to which it is assembled.

4. Complete the component placement. Display features using the model tree.
5. Using the View Manager, activate the **Master Style** rep and the **Outer_Excluded** simplified rep.
6. Using the model tree, select PISTON.ASM and CRANK_FLYWHEEL.ASM. Click **View > Representation > Exclude**. Using the View Manager, save the rep as **Conn_Rod_Only**. Close the View Manager.
7. Zoom SKEL_CONN_ROD.PRT to the datum axes.
8. Activate CONNECTING_ROD.PRT. Click **Insert > Shared Data > Copy Geometry**.
9. Double-click **Surface Refs**. Set the selection filter to **Quilt**. Press and hold CTRL and select both cylindrical surface quilts. Click **OK**.
10. Double-click **Curve Refs**. Select the datum curve, and then click **Done**.
11. Double-click **Misc Refs**. Click **Axis**. Press and hold CTRL and select the CRANK_PIN and PISTON_PIN axes. Click **Ok > Ok**, and then complete the feature.
12. Using View Manager, activate the **Master Style** rep and the **Outer_Excluded** simplified rep. Close the View Manager.
13. Save the assembly.

Step 3. Create solid geometry for CONNECTING_ROD.PRT.

Next, we build the individual connecting rod features.

1. Using the model tree, open CONNECTING_ROD.PRT. Notice that it contains a single copy geometry feature.

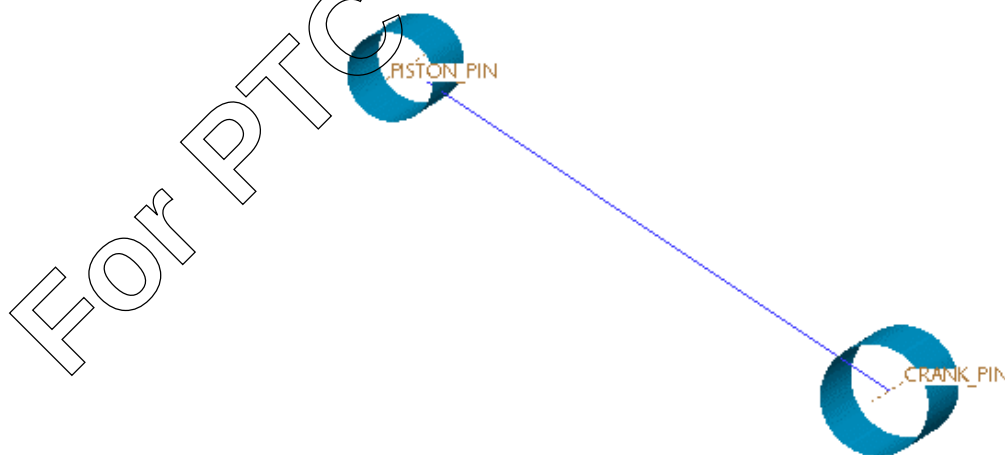


Figure 12: CONNECTING_ROD.PRT with Copy Geometry

2. Start the **Extrude** tool. Create a sketch using the dashboard. Select datum FRONT as the sketching plane and click **Sketch**. Select the two axes as references.

3. Sketch using the CONN_ROD.SEC file. Scale to **1.0** and locate so that the circle centers snap to the axis references. Refer to the following figure.

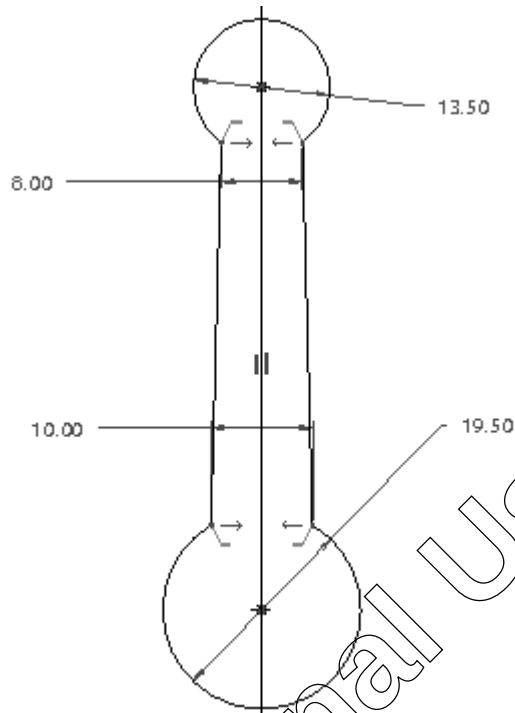


Figure 13: Sketching the CONNECTING ROD.PRT Based off of Skeleton References

4. Complete the sketch. Using the dashboard, enter **2.5** as the symmetrical depth.
5. Complete the feature.

Step 4. Analyze the mechanism motion.

Finally, we examine how the new connecting rod fits into the assembly. To accomplish this task, we run the engine analysis within Mechanism mode and check for interference. After the interference areas are located, we can edit the necessary parts and create a virtual model that is free of interference. This enables us to limit the number of engine prototypes that must be built to formally test the drill.

1. Close the window and return to DRILL.ASM. Notice that CONNECTING_ROD.PRT now has solid geometry features.

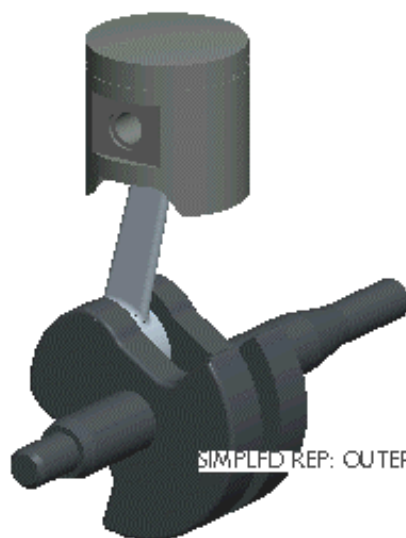


Figure 14: Activating the Outer Excluded Simplified Rep

2. Enable Mechanism mode.
3. Using the model tree, expand the **Analyses** node. Select the **Engine** analysis, right-click and select **Run**.
4. Using the model tree, select the **Playbacks** node. Right-click and select **Play**. Select **Global Interference**. Play the current results set. Increase the speed to maximum. Play the animation. Note that interference exists because of the length of CONNECTING_ROD.PRT.

Note:

Continuous interference also occurs at the ends of the connecting rod (where holes have not yet been created).

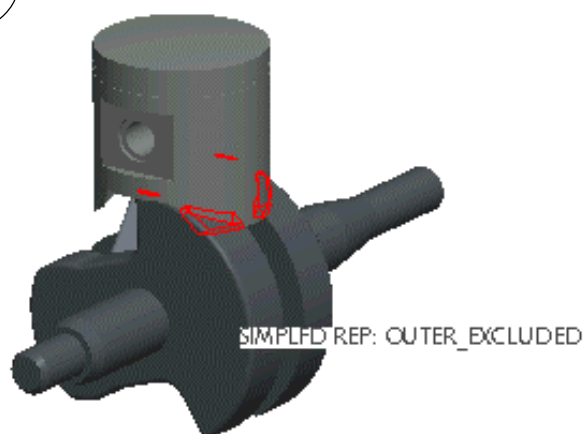


Figure 15: Examining the Mechanism for Global Interferences

5. Finish viewing. Stop the animation. Close the ANIMATION and PLAYBACKS dialog boxes.

6. Enable Standard mode. Click **Yes** to discard the playback.

Step 5. Resolve the major interference.

1. Open SKEL_CONN_ROD.PRT. Edit the datum curve. Enter **51.5** for the length. Regenerate the model, then close the window.
2. Regenerate the assembly. Enable Mechanism mode.
3. Using the model tree, expand the **Analyses** node. Select the **ENGINE** analysis, right-click and select **Run**.
4. Using the model tree, select the **Playbacks** node. Right-click and select **Play**. Select **Global Interference** and play the current results set. Increase the speed to maximum. Play the animation.
5. Observe that PISTON.PRT and CRANKSHAFT.PRT no longer interfere with each other. Finish viewing. Stop the animation. Close the ANIMATION and PLAYBACKS dialog boxes.
6. Enable Standard mode. Click **Yes** to discard the playback file.
7. Save the model.

Step 6. (OPTIONAL) Complete designing CONNECTING_ROD.PRT

Instructor: Skip this step for the demo.

1. Open CONNECTING_ROD.PRT.
2. Create the extruded protrusion as shown in the following figure. Reference the existing solid for the diameter and the underlying surface geometry for the depth. This is shown in the following figure.

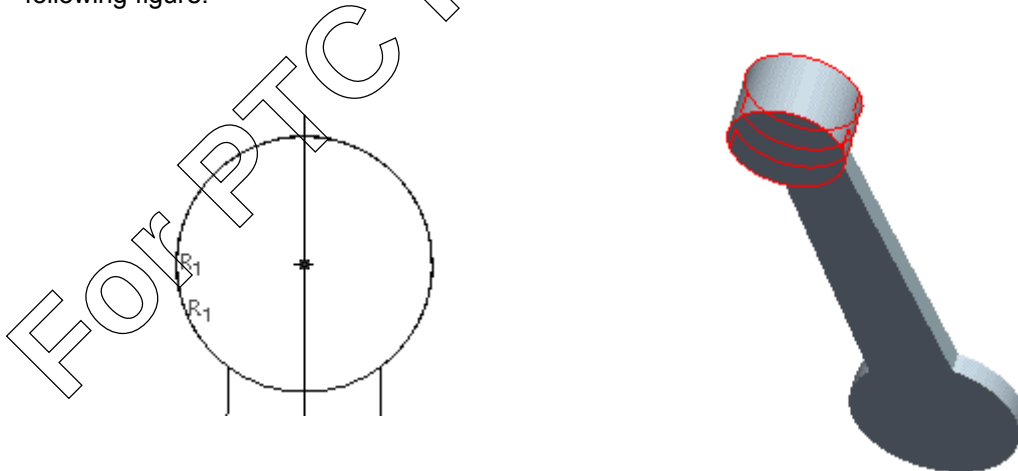


Figure 16: Sketching the Feature (left) and the Completed Extrude Feature (right)

3. Repeat the procedure for the other end of CONNECTING_ROD.PRT.



Figure 17: Creating a Second Protrusion Referencing the Geometry Feature

4. Create an extruded cut referencing the underlying surface geometry for the diameter.

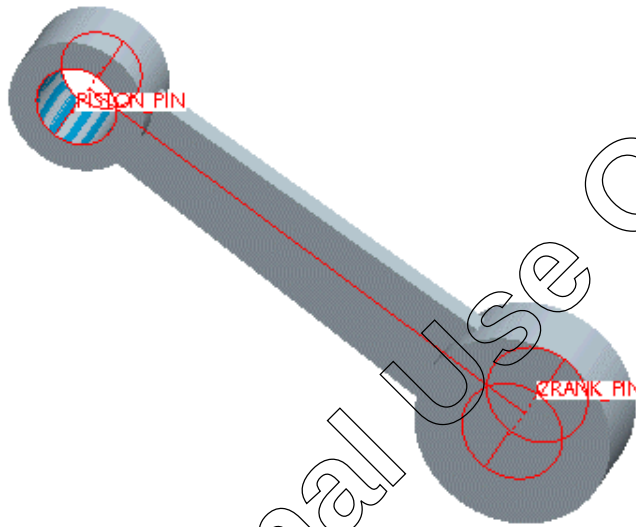


Figure 18: Creating a Cut Referencing the Copy Geometry Feature

5. Repeat the procedure for the other end of CONNECTING_ROD.PRT.

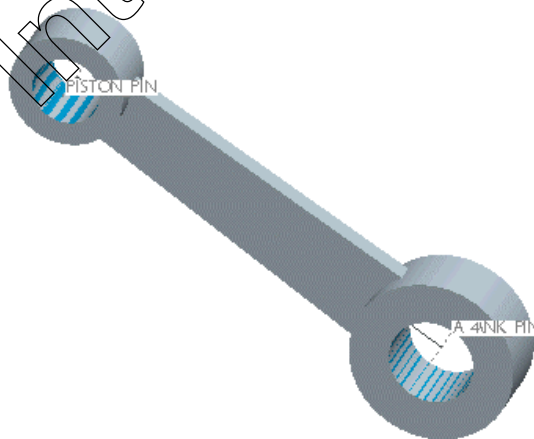


Figure 19: Creating a Second Cut Referencing the Copy Geometry Feature

6. Create a round as shown in the following figure.

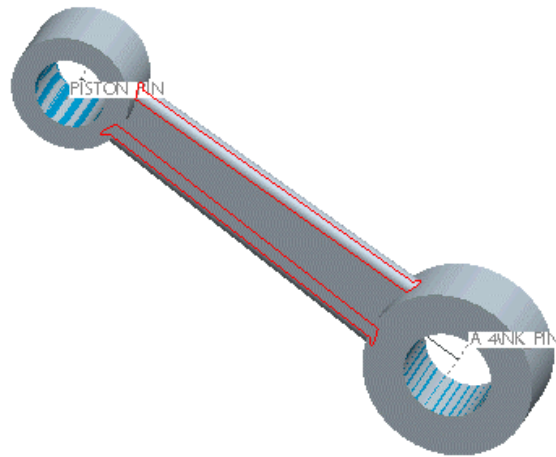


Figure 20: Creating a Two Set Surface-to-Surface Round Feature

7. Create a round with a radius of **2**, as shown in the following figure.

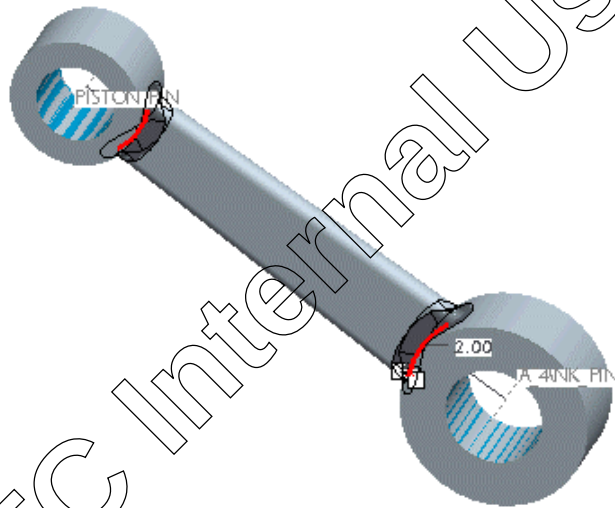


Figure 21: Creating a Single Set Edge Round Feature

8. Apply the **Copper** appearance to the part.
9. Re-run the mechanism analysis.
10. Playback the animation and check for interference.
11. Save the model.
12. Close all windows.
13. Erase all models from memory.

This completes the exercise.

Exercise 3: Creating View Orientations and Explode States (Challenge)

Introduction

In this demonstration, we create saved views of parts and assemblies. Views enable us to save a specific orientation and zoom setting of a part or an assembly. View orientations provide us with zoom and rotational settings that make it easy to achieve a desired view. These view orientations save time because we do not have to rotate/pan/zoom.

In this demonstration, we also create explode states of assemblies. Explode states display assemblies in various configurations of assembly/disassembly. They are useful in drawings and in documenting assembly/disassembly procedures.

Objectives

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

- Create saved view orientations.
- Create explode states of assemblies.

Scenario

You are assigned to create various saved orientation views of parts and assemblies that will be used during design reviews. Other departments and suppliers of your company using ProductView can then access these views.

You are also assigned to create basic explode states of the drill, which will be use later by the Documentation department to create field maintenance manuals for your customers.

Step 1. Orient the GEARBOX_FRONT.PRT.

Instructor: Skip this step for the demo.

1. Set the working directory to: **C:\users\student\fast_track_330\module_17.**
2. Open GEARBOX_FRONT.PRT.
3. Orient the model, as shown in the following figure.

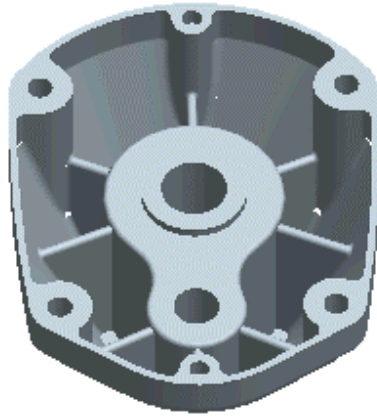


Figure 22: Orienting GEARBOX_FRONT.PRT.

Note:

The term 'orient' describes a combination of rotate, zoom, and pan operations.

4. Start the View Manager, and select the ORIENT tab. Click **New**. Enter **3D_Back** to save the orientation. Close the View Manager.
5. Using the toolbar, orient to the standard orientation, and then orient to the 3D_BACK view.
6. Save the model, and close the window.

Step 2. Orient RECOIL_COVER.PRT.

In this step, we create a saved view of the rope stud feature on the recoil cover. Manufacturing requested this view as part of a feasibility study.

1. Open RECOIL_COVER.PRT.
2. Start the View Manager, and select the ORIENT tab. Double-click to examine the existing saved orientations.
3. Orient to VIEW0001, as shown in the following figure. Close the View Manager.

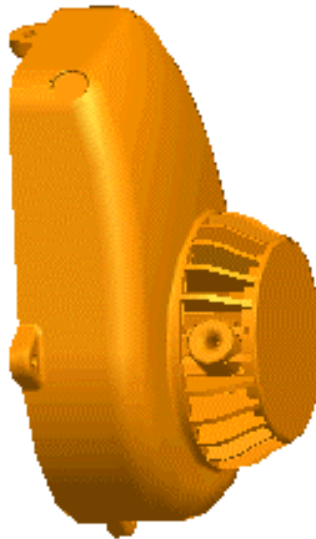


Figure 23: Orienting RECOIL_COVER.PRT

4. Click **View > Orientation > Reorient**. Select **Orient By Reference** as the type. Select appropriate surfaces to face the front and top, and orient the model, as shown in the following figure. Click **Ok**.



Figure 24: Reorienting the RECOIL_COVER.PRT

5. Zoom in on the rope stud feature, as shown in the following figure.

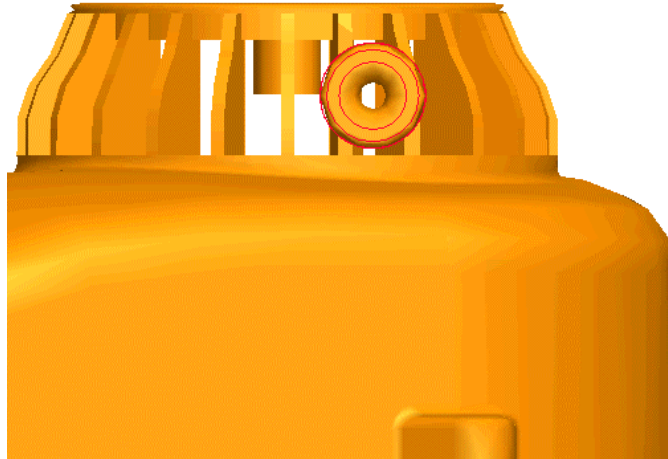


Figure 25: Zooming In on the Rope Stud

6. Start the View Manager, and select the ORIENT tab. Click **New**. Enter **Rope_Stud** to save the orientation.
7. Close the View Manager. Using the toolbar, orient to the standard orientation, then orient to the ROPE_STUD view.
8. Save the model, and close the window.

Step 3. Create the GEARBOX saved orientation.

In addition to Manufacturing's request, Product Marketing has requested several saved views for use in their product brochures. In this step, we must zoom in close enough to the desired parts or Marketing will call back and request additional pictures.

1. Open DRILL.ASM.
2. Start the View Manager, and select the ORIENT tab. Double-click to examine the existing saved orientations.
3. Enable model transparency, if necessary.
4. Orient the model to view GEARBOX_CHUCK.ASM in more detail, as shown in the following figure.

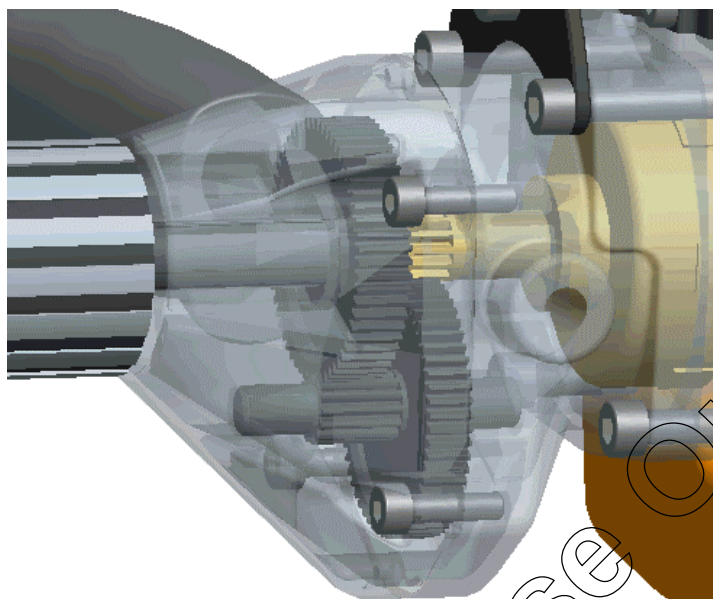


Figure 26: Saving the Gearbox Orientation

Note:

You can leave the View Manager open during next few steps.

5. Start the View Manager, and select the **ORIENT** tab. Click **New**. Enter **Gearbox**.
6. Double-click on **Standard Orientation**.

Step 4. Create saved orientations for the Carburetor and Trigger.

Instructor: Skip this step for the demo.

1. Hide AIR_FILTER_COVER.PRT.
2. Orient the model to view the CARBURETOR.ASM in more detail, as shown in the following figure.
3. Using the View Manager, and save the orientation as **Carburetor**.

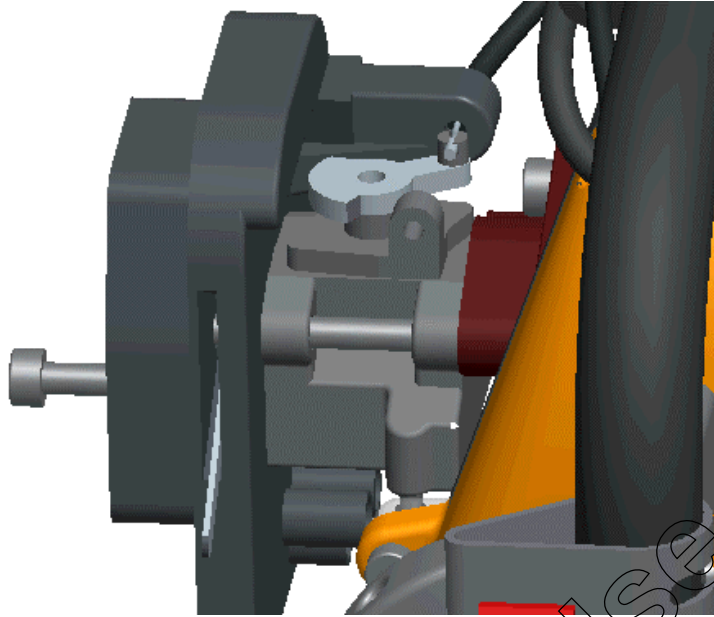


Figure 27: Saving the Carburetor Orientation

4. Double-click on **Standard Orientation**, and unhide AIR_FILTER_COVER.PRT.
5. Orient the model to view TRIGGER.PRT in more detail, as shown in the following figure.
6. Using the View Manager, and save the orientation as **Trigger**.
7. Double-click on **Standard Orientation**.

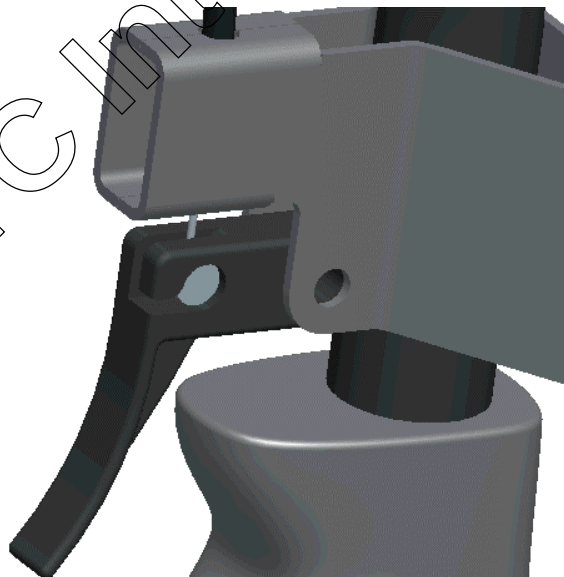


Figure 28: Saving the Trigger Orientation

Step 5. Save the 3D_1 and 3D_2 views.

1. Using the toolbar, reorient the view, and then select **Dynamic Orient** as the type.
2. Toggle **Spin Center Axis** to spin. Enter **-40** for X, **-25** for Y, and **-5** for Z.
3. Click **Saved Views**. Enter **3D_1** as the name. Click **Save**.



Figure 29: Saving 3D_1 Orientation

4. Enter **90** for Y. Enter **3D_2** as the name. Click **Save**. Leave the dialog box open for the following step.



Figure 30: Saving 3D_2 Orientation

Step 6. Save the 3D_3 and 3D_4 views.

Instructor: Skip this step for the demo.

1. Enter **180** for Y. Enter **3D_3** as the name. Click **Save**.



Figure 31: Saving 3D_3 Orientation

2. Enter **-90** for Y. Enter **3D_4** as the name. Click **Save > Ok**.



Figure 32: Saving 3D_4 Orientation

3. Using the toolbar, orient to the various saved views.
4. Save the model.

Step 7. Explode DRILL.ASM.

In this step, we create explode states of the drill assembly for use by Product Documentation in their field service manuals. Explode states display assemblies in various configurations of assembly/disassembly. They are useful in drawings and in documenting assembly/disassembly procedures.

1. Orient the 3D_1 view.

Note:

If necessary, disable model transparency for faster performance.

2. Click **View > Explode > Edit Position**. Select **Plane Normal** as the Motion Reference type. Select the planar surface on the tip of CHUCK.PRT as the reference.
3. Select the STANDARD_BIT.PRT, and then select a new location in the graphics window, as shown in the following figure. Repaint the screen.

Note:

The mouse operations for exploding components use a “click-release, move the mouse, click-release” workflow sequence.

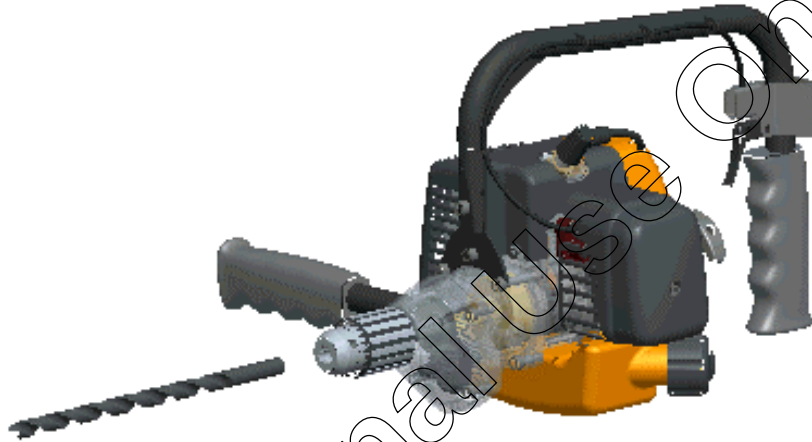


Figure 33: Exploding the STANDARD_BIT.PRT

4. Click **Preferences > Move With Children > Close**.
5. Cursor over CHUCK.PRT. Right-click to query GEARBOX_CHUCK.ASM, and then select it. Drag to the left to explode, as shown in the following figure. Repaint the screen.

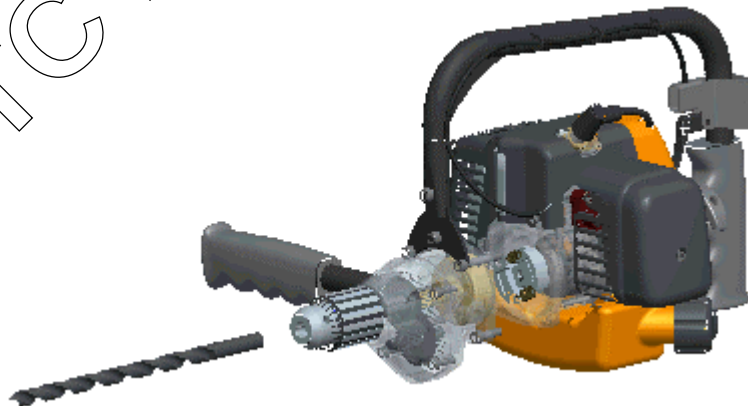


Figure 34: Exploding the GEARBOX_CHUCK.ASM

Note:

Many of the following steps will require the **Move with Children** option.

6. Query and select HANDLE_MAIN.ASM. Drag slightly to the left to explode, as shown in the following figure. Repaint the screen.

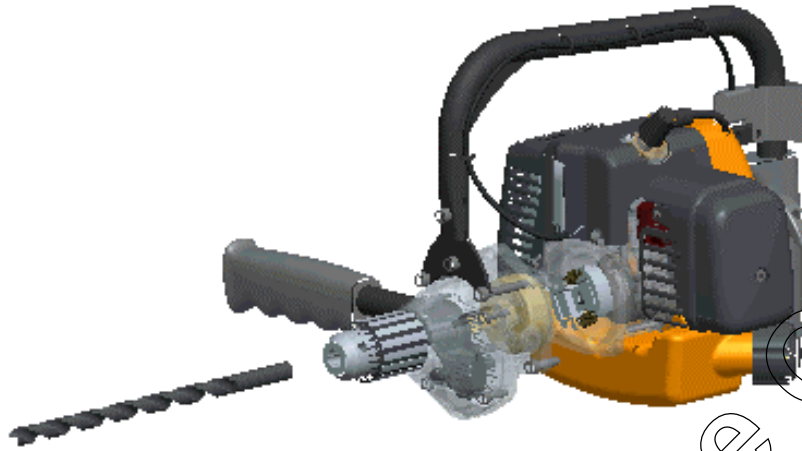


Figure 35: Exploding the HANDLE_MAIN.ASM

7. Select the planar surface on the top of the ENGINE_COVER.PRT as the new motion reference.
8. Query and select HANDLE_MAIN.ASM. Drag up to explode, as shown in the following figure. Repaint the screen.



Figure 36: Exploding the HANDLE_MAIN.ASM

9. Query and select ENGINE_COVER.PRT. Drag up to explode, as shown in the following figure.



Figure 37: Exploding the ENGINE_COVER.PRT

10. Select the planar surface on the tip of CHUCK.PRT as the new motion reference.
11. Query and select RECOIL.ASM. Drag to the right to explode, as shown in the following figure.



Figure 38: Exploding the RECOIL.ASM

12. Click **Ok** to complete the exploded state.

Step 8. Explode the remaining DRILL.ASM components.

Instructor: Skip this step for the demo.

1. Click **View > Explode > Edit Position**.
2. Select **Plane Normal** as the Motion Reference type.

3. Using the Search tool, select the ASM_RIGHT datum plane as the new motion reference.
4. Query and select CARBURETOR.ASM. Drag to the right to explode, as shown in the following figure.

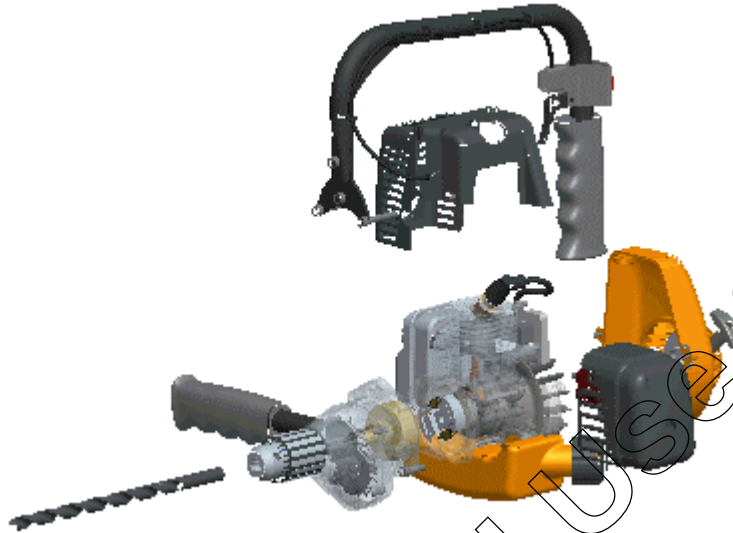


Figure 39: Exploding the CARBURETOR.ASM

5. Query and select AIR_FILTER_COVER.PRT. Drag to the right to explode, as shown in the following figure.



Figure 40: Exploding the AIR_FILTER_COVER.PRT

6. Query and select HANDLE_SIDE.ASM. Drag to the left to explode, as shown in the following figure.

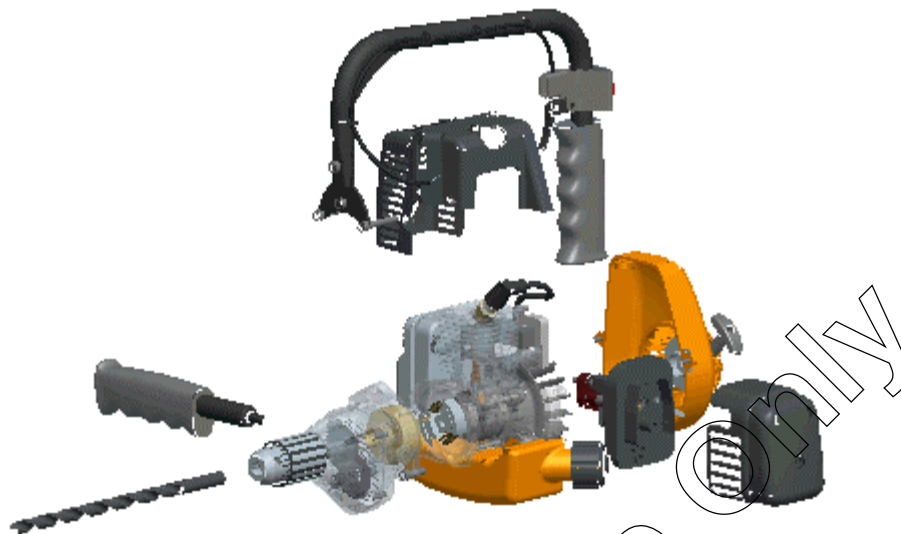


Figure 41: Exploding the HANDLE_SIDE.ASM

7. Select any horizontal surface or plane as the new motion reference.
8. Query and select RECOIL.ASM. Drag up to explode, as shown in the following figure.

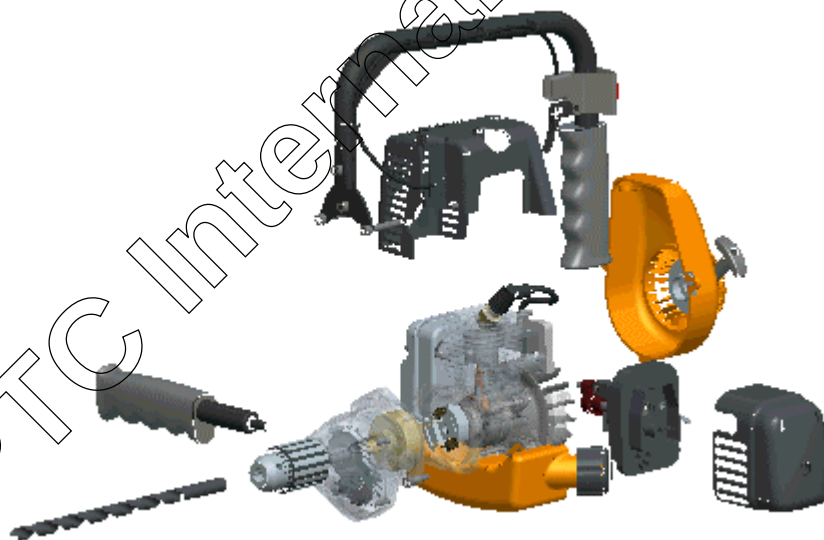


Figure 42: Exploding the RECOIL.ASM

9. Query and select FUEL_TANK.ASM. Drag down to explode, as shown in the following figure.

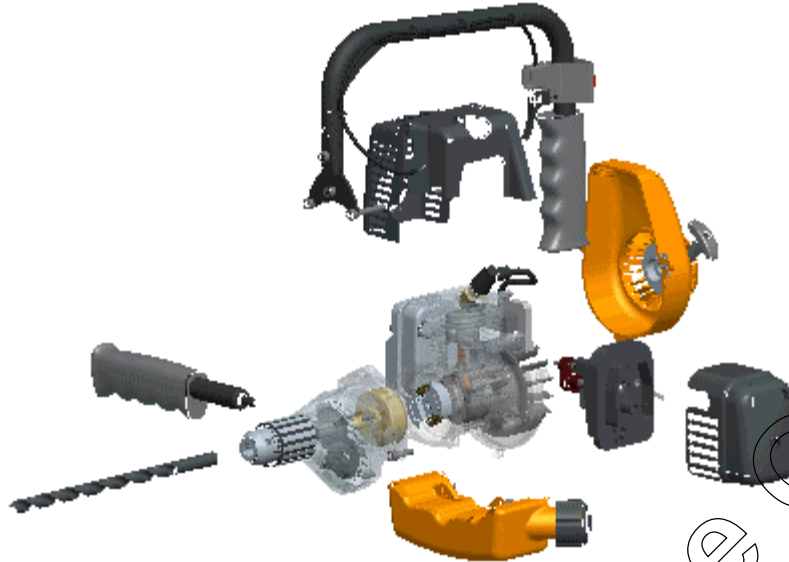


Figure 43: Exploding the FUEL_TANK.ASM

10. Query and select MUFFLER.PRT. Drag left to explode, as shown in the following figure.

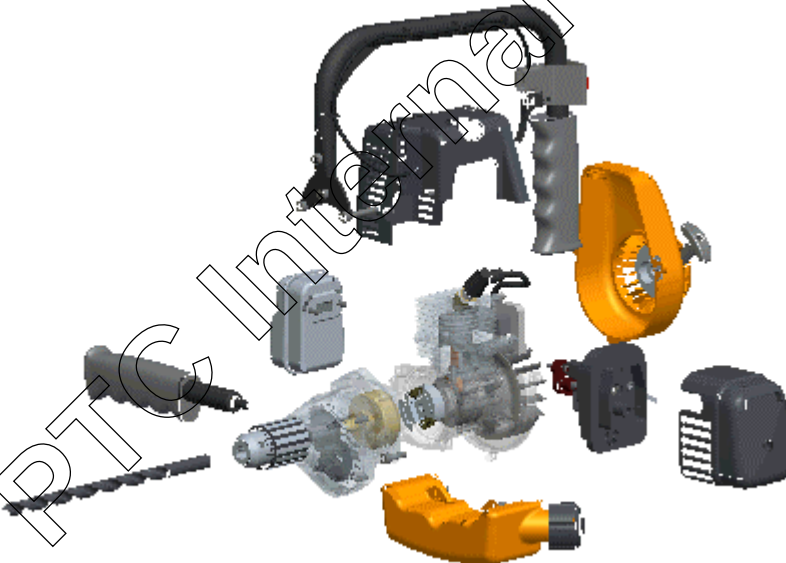


Figure 44: Exploding the MUFFLER.PRT

11. Select any vertical surface or plane as the new motion reference.
12. Explode the three main handle/gearbox bolts.
13. Click **Ok** to complete the exploded state.



Figure 45: Creating the Main Handle/Gearbox Bolts Explode State

Step 9. Save the exploded state and create offset lines.

1. Start the View Manager, and select the EXPLODE tab. Click **New**. Enter **Drill_Exp**.
2. In the View Manager, right-click and select **Unexplode**.
3. Double-click on DRILL_EXP. Close the View Manager.
4. Save the assembly.
5. Using the toolbar, turn off display of datum axes.

Note:

Even though datum axes are not visible due to layers, turning them off in the toolbar prevents them from highlighting in the graphics window.

6. Click **View > Explode > Offset Lines > Create > Axis**.
7. Zoom in to view the gap between STANDARD_BIT and CHUCK.PRT.
8. Select the appropriate cylindrical surfaces from the STANDARD_BIT and CHUCK.PRT to create an offset line.

Note:

When selecting surfaces for axial offset lines, select as if you were assembling with an Insert constraint.

9. Click **Surface Norm**. Select a location on the surface underneath the ENGINE_COVER.PRT and a location on the surface on the top of the CYLINDER.PRT to create an offset line.

Note:

When selecting locations on surfaces for Surface Normal offset lines, select corresponding locations that line up as directly as possible.

10. Click **Ok > Quit**. Start the View Manager, and select the EXPLODE tab. Select DRILL_EXP, and then right-click and select **Update**. Close the View Manager.
11. Using previous techniques, create additional offset lines, as shown in the following figures.

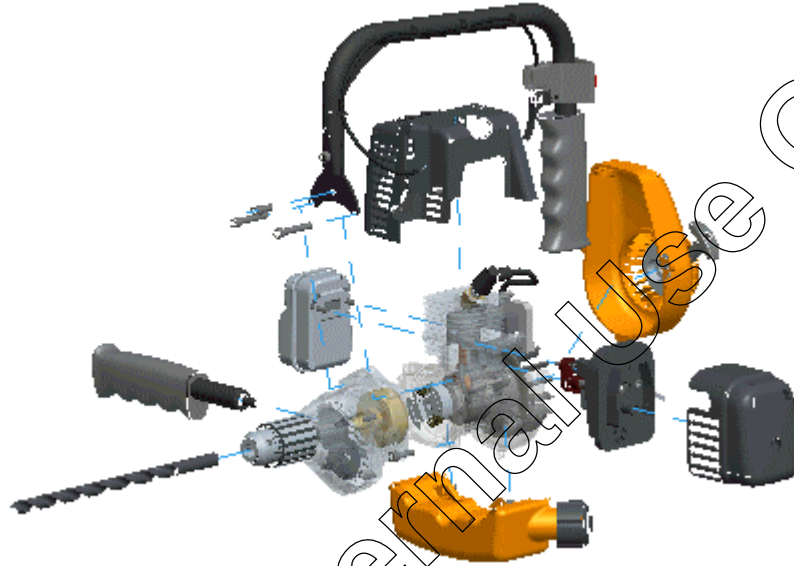


Figure 46: Creating Explode Lines

12. Using the View Manager, update the DRILL_EXP.
13. Save the model.
14. Close all windows.
15. Erase all models from memory.

This completes the exercise.

Summary

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

- Create style and simplified representations of assembly models.
- Design models using skeletons.
- Create view orientations of assembly models.
- Create exploded states of assembly models.
- Describe how managing assemblies enables you to manage parent/child relationships.

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Analyzing and Optimizing Design Models

Introduction

You can create various types of analyses, such as measuring distances, calculating mass properties and interferences, and examining surface curvature and continuity in design models. These analyses can be saved as features that update along with changes to the design models. Analysis features can then be used in conjunction with other parameters in iterative design studies.

Objectives

After completing this module, you will be able to:

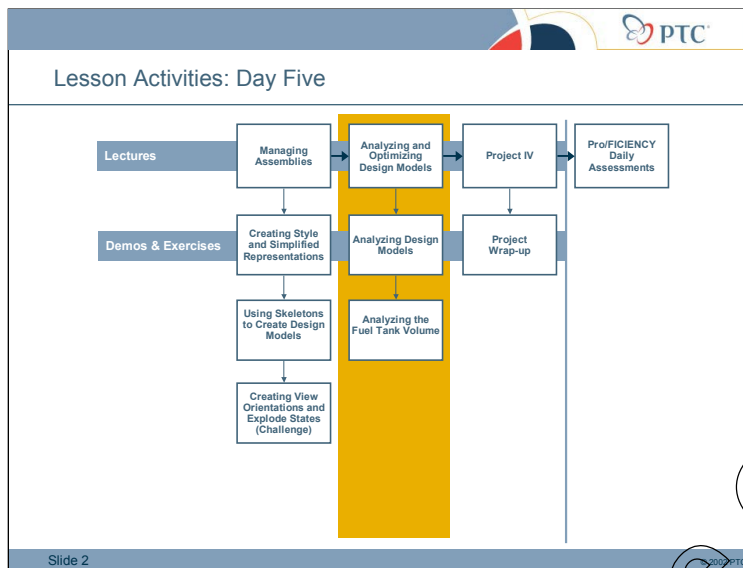
- Create various types of design model analyses.
- Create analysis features.



Instructor Preparation

Before teaching *Fast Track to Pro/ENGINEER Wildfire*, you **must read** and **thoroughly understand** 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 Foundation
 - Review Pro/ENGINEER Wildfire Foundation training materials in Windmill, located in /GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T978-330-Instructor_Kit-EN



Duration

- Lecture: 10 mins
- Demos (2): 40 mins
- Labs (2): 40 mins
- Total: 1 hr 30 mins



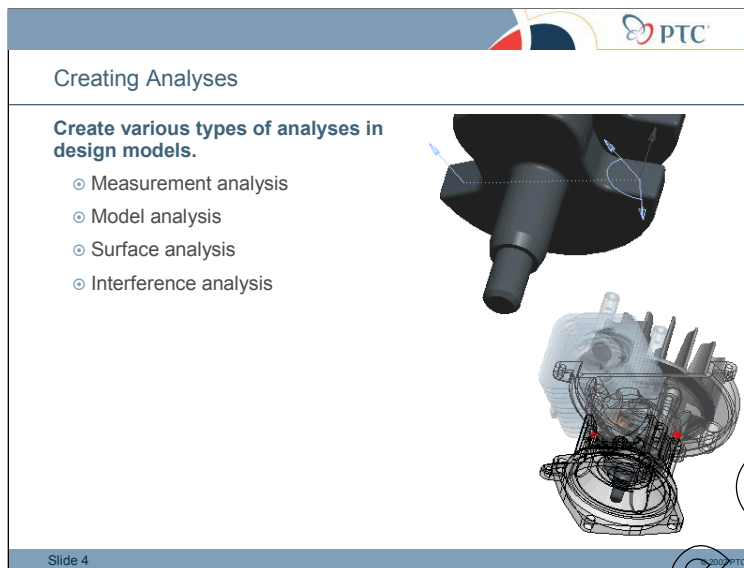
Objectives

After completing this module, you will be able to:

- Create various types of design model analyses.
- Create analysis features.

Slide 3

- You can create various types of analyses, such as measuring distances, calculating mass properties and interferences, and examining surface curvature and continuity in design models.
- These analyses can be saved as features that update along with changes to the design models.
- Analysis features can then be used in conjunction with other parameters in iterative design studies.
- In this module, you learn how to analyze design models, create analyses features, and conduct feasibility and optimization studies.



Analyses enable you to inspect and query your models for specific information.

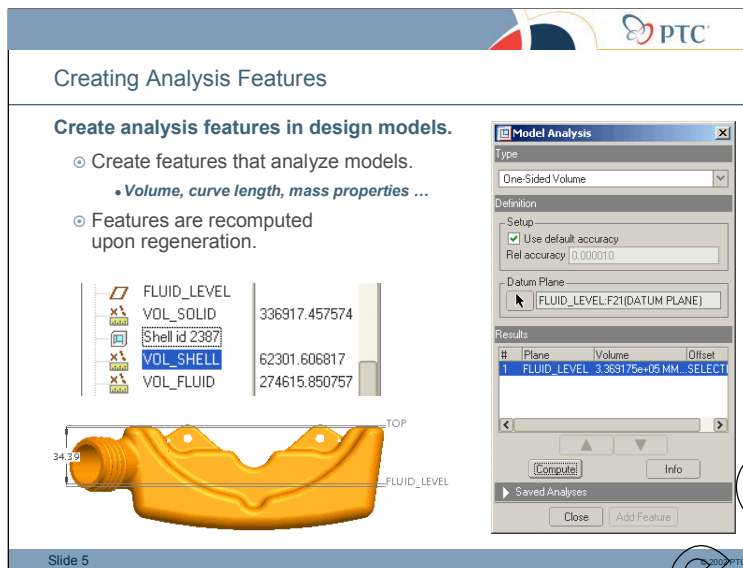
You can use analyses to:

- Measure distances, angles, curve lengths, areas, and diameters.
- Calculate interferences, clearances, and mass properties.

Analyses are important to you because they can aid you in determining whether all design specification are being met. For example, you can check for interference between parts. You can also ensure that a part is balanced by checking to ensure its center of gravity is along its axis of rotation.

Benefit:

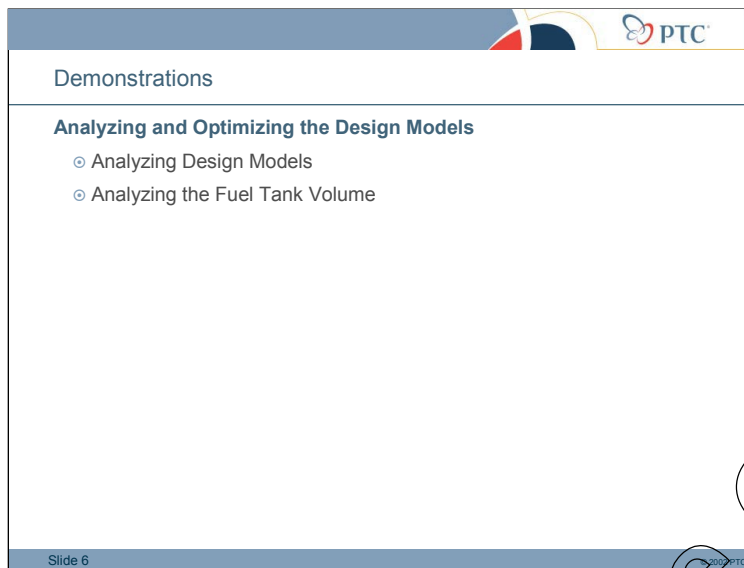
- Analyses are important as they can help you ensure that all design specifications are being incorporated into your models.



- Analysis features enable you to save an analysis as a feature within the model tree.
- Therefore, you do not have to keep recreating analyses, you only have to check the analysis features.
- By saving an analysis as a feature, you eliminate the need to recreate the analysis to ensure design specifications.
- Analysis features update when the design model is edited and regenerated.
- Analysis features can be referenced in feasibility and optimization studies.

Benefit:

- By saving an analysis as a feature, you eliminate the need to recreate the analysis to ensure design specifications.
- Analysis features update when the design model is edited and regenerated.
- Analysis features can be referenced in feasibility and optimization studies.



You will follow along while I demonstrate:

1. Analyzing Design Models
2. Analyzing the Fuel Tank Volume

Then, you will practice the same steps in the laboratory exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 7

Precision Learning

- **Learn:** by listening to lectures, watching demos, and completing lab exercises.
- **Assess:** your progress with Pro/FICIENCY assessment questions.
- **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 Exams

Description: For each course, 5 new 10 question Pro/FICIENCY 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:

- Create various types of analyses in design models.
- Create analysis features.

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Referencing PTC Help for Module 18 Topics

Accessing Pro/ENGINEER Wildfire Help Center

Here are some other suggested keywords and phrases that you can perform a Global Search on to find additional information.

Keyword or Phrase	Functional Area	Link
Analysis	Fundamentals	About Model Analysis About Analyzing Curves About Part Comparison About the Analysis Menu

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Module 18 Lab Exercises

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 2003100 or later.
- Download and install the class files **fast_track_330.tar.gz** as described in the classroom setup notes.

Exercise 1: Analyzing Design Models

Introduction

In this demonstration, we analyze various parts and a subassembly of the drill. Using Pro/ENGINEER Wildfire, we can run many different types of analyses, including mass property, surface, measurements, and interferences. These types of analyses are important, if you are designing components with a specific goal in mind, such as maintaining a part's center of gravity on its rotational axis.

Objectives

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

- Configure the mass properties of a part material.
- Analyze components for distance, angles, surface area, and interferences.

Scenario

While designing the drill, your manager asks you to confirm that the component models adhere to the design specifications. For example, you need to verify the mass properties of the fuel tank to confirm that the raw material estimates are correct. You need to calculate the surface area of the cylinder to confirm the proper thermal cooling. Also, the crankshaft's center of gravity needs to be along the axis of rotation. Finally, you need to check the engine assembly for interferences between its components.

Step 1. Analyze the FUEL_TANK.PRT.

In this initial step, we analyze the fuel tank in order to ensure that Procurement gets a proper quote on raw materials. First, we check the system of units that the part was designed in, so that

we can avoid any calculation mistakes. We then assign a material mass property for the plastic used to build the fuel tank. Finally, we compute the volume and mass of materials needed to create the fuel tank.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from session. Otherwise start Pro/ENGINEER Wildfire.
2. Set the working directory to
C:\users\student\fast_track_330\module_18.
3. Open FUEL_TANK.PRT.
4. Click **Edit > Setup > Units**. Examine the current system of units and click **Close**.
5. Click **Mass Props** and enter the density for the nylon fuel tank: **1.2e-6** (kg/mm³). Click **Ok**.
6. Click **Analysis > Model Analysis**. Select **Model Mass Properties** and click **Compute**.
7. Notice the volume (0.10 liters) and mass (0.12kg) of material. Close the Model Analysis dialog box.
8. Save the model.
9. Close the window.

Step 2. Analyze the mass properties of the CYLINDER.PRT.

In the next step, we analyze the cylinder to check the amount of surface area available for cooling. This is important, as the engine assembly is air-cooled. Again, we check the system of units and assign a material mass property for aluminum. Finally, we compute the surface area and make other needed measurements on the cylinder.

1. Open CYLINDER.PRT.
2. Click **Edit > Setup > Units**. Examine the current system of units and click **Close**.
3. Click **Mass Props** and enter the density for the aluminum cylinder: **2.7e-6** (kg/mm³). Click **Ok**.
4. Click **Analysis > Model Analysis**. Select **Model Mass Properties** and click **Compute**.
5. Notice the surface area of the model (64,258 mm²). Close the Analysis dialog box.

Step 3. Measure the CYLINDER.PRT.

1. Click **Analysis > Measure**. Select **Diameter** as the type and select the cylindrical surface of the spark plug hole. Note that the diameter is 13.75.
2. Select **Distance** as the type and select the carburetor interface and exhaust interface surfaces. Note that the distance is 62.

3. Select **Area** as the type. Select the engine block interface surface. Note that the area is 1021. Close the Measure dialog box.

Note:

Measurements can be saved by name or can be made into an Analysis Feature that displays in the model tree.

4. Save the model and close the window.

Step 4. Analyze the mass properties of the CRANKSHAFT.PRT.

Instructor: Skip this step for the demo.

1. Open CRANKSHAFT.PRT.
2. Click **Edit > Setup > Units**. Examine the current system of units and click **Close**.
3. Click **Mass Props** and enter the density for the steel crankshaft: $7.8e-6$ (kg/mm³). Click **Ok**.
4. Click **Analysis > Model Analysis**. Select **Model Mass Properties** and click **Compute**.
5. Note that the center of gravity is below the axis of revolution.

Step 5. Measure the CRANKSHAFT.PRT.

Instructor: Skip this step for the demo.

1. Click **Analysis > Measure**. Select **Curve Length** as the type. Select the straight edge of the keyway. Note that the distance is 11.77.
2. Select **Angle** as the type. Select the two planar surfaces as, shown in the following figure. Note that when you flip the appropriate arrow, the angle is approximately 150 degrees.

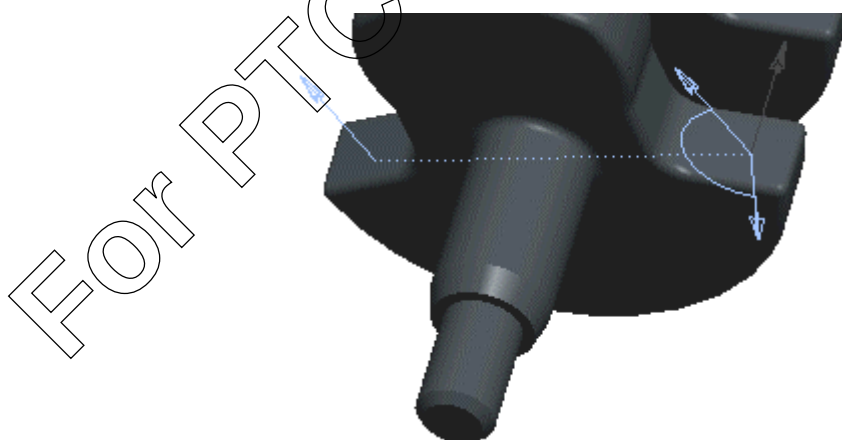


Figure 1: Measuring the Angle Between the Planar Surfaces

3. Save the model.
4. Close the window.

Step 6. Analyze the mass properties of the ENGINE.ASM.

In the last step, we analyzed the engine assembly for interferences. This is an extremely useful capability because we can eliminate all undesired interferences before any part of the engine is manufactured.

1. Open ENGINE.ASM.
2. Click **Analysis > Model Analysis**. Select **Assembly Mass Properties** and click **Compute**. Note the displayed values for volume, surface area, density, mass, and center of gravity.

Note:

The components of the engine have been assigned appropriate density values for either steel or aluminum. If the density values had not been defined, then Pro/ENGINEER Wildfire would prompt you when computing the mass properties.

Step 7. Analyze and resolve interferences within ENGINE.ASM components.

1. From the Model Analysis dialog, select **Global Interference** as the type and click **Compute**. Notice that there are three pairs of interfering components.

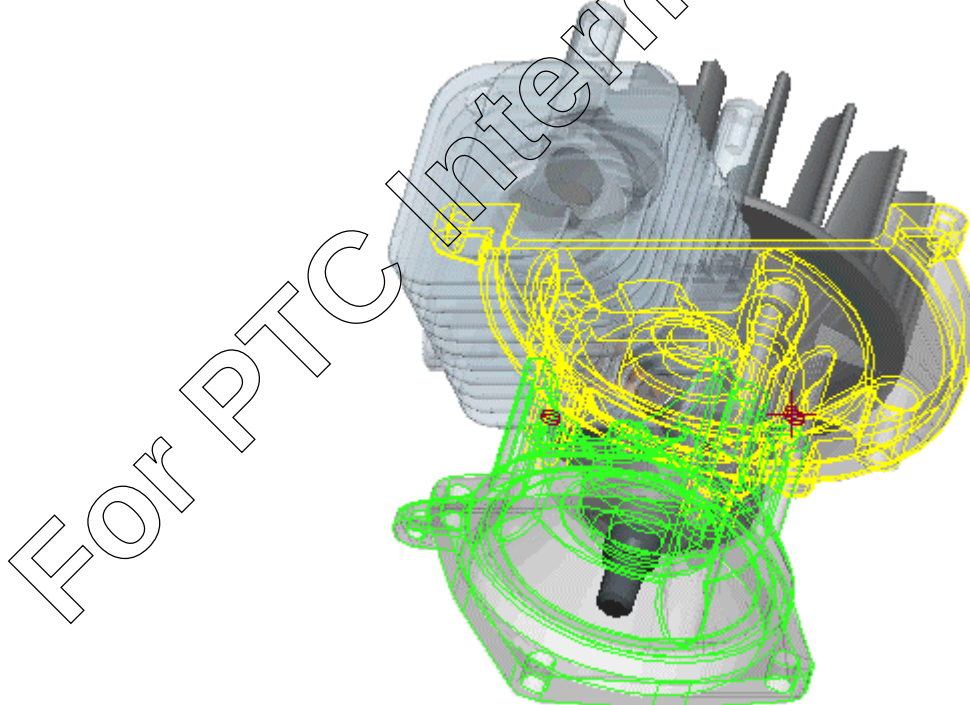


Figure 2: Global Interference: Pair1

2. Highlight each to display the interference. Note the ENG_BLOCK_FRONT.PRT interferes with the alignment pins, and the CYLINDER.PRT interferes with the bolts.

Step 8. Resolve the interference.

Instructor: Skip this step for the demo.

1. Resolve the interferences in CYLINDER.PRT by editing the CYL_BLOCK hole diameters to **5.5**. Resolve the interferences in ENG_BLOCK_FRONT.PRT by editing the depth of the ALIGN_PINS to **5**.
2. Regenerate the assembly. Re-Compute the Global Interference check. There should be no interfering parts.
3. Save the model.
4. Close all windows.
5. Erase all models from memory.

This completes the exercise.

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Exercise 2: Analyzing the Fuel Tank Volume

Introduction

In this demonstration, we create analysis features. Analysis features are important because they run a pre-configured analysis and can export the results as a parameter. The outputted parameters can be used to modify the model to ensure that it conforms to its design intent.

Objectives

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

- Create analysis features.
- Create feasibility studies.

Scenario

The design specification of the fuel tank requires that the fuel tank hold at least **0.35** liters of fuel. The design specification also requires that the fuel tank have a means to identify when it is half full. You are assigned to ensure that the fuel tank has been properly designed to this specification.

Step 1. Calculate the fluid volume of the FUEL_TANK.PRT.

In this initial step, we set up analysis features to compute the volume of the fuel tank. Once we determine the current volume of the fuel tank, we can then make any necessary modifications to achieve 0.350 liters of volume. We also create a datum plane that will represent the current level of fuel in the fuel tank. By moving the datum plane, we can simulate empty to full conditions.

1. Open FUEL_TANK.PRT.

Note:

Since Pro/ENGINEER Wildfire only measures the volume of solid material present, we will measure the volume of the model before and after the shell feature. By subtracting these values, we obtain the fluid volume.

2. Orient to the FRONT view.
3. Drag the **Insert** indicator before the shell feature.
4. Create a datum plane offset from the TOP plane by **-20**. Name the plane FLUID_LEVEL.
5. Use the toolbar to create an Analysis Feature.

Note:

You must press ENTER when entering values for Analysis Features.

6. Enter **VOL_SOLID** as the Name and select **Model Analysis** as the Type.
7. Click **Next** and select **One-Sided Volume** as the type. Select the FLUID_LEVEL plane, Flip the direction downward, and click **Okay > Close**. Rename the parameter to **VOLUME**. Complete the feature.

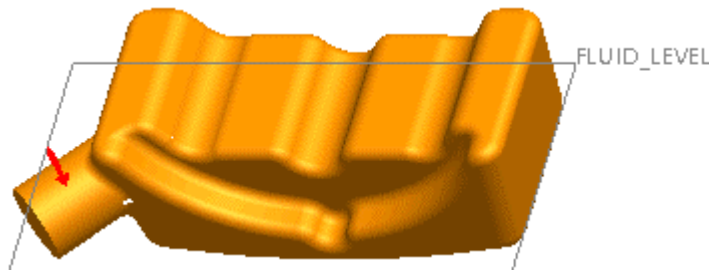


Figure 3: Configuring the Model Analysis for the Analysis Feature

8. In the model tree, click **Settings > Tree Columns**. Select **Feat Params** as the Type, enter VOLUME, and click **OK**. Notice that the solid volume below the plane is approximately **0.336** liters.

Note:

A mass properties analysis can also be used to calculate the volume. However, it will not permit a plane to be selected.

9. Select the **Insert** indicator, and drag after the shell feature.
10. Use the toolbar to create a second Analysis Feature. Enter **VOL_SHELL** as the Name and select **Model Analysis** as the Type.
11. Click **Next** and select **One-Sided Volume** as the type. Select the FLUID_LEVEL plane, Flip the direction downward, and click **Okay > Close**. Rename the parameter to **VOLUME**. Complete the feature.
12. Use the toolbar to create a third Analysis Feature. Enter **VOL_FLUID** as the Name and select **Relation** as the Type.
13. Click **Next** and enter the following:

$$\text{volume} = (\text{volume:FID\_VOL\_SOLID} - \text{volume:FID\_VOL\_SHELL})/1,000,000$$

Note:

By dividing the right side of the above relation by 1,000,000, you effectively convert mm^3 to Liters.

14. Click **Ok** and complete the feature.

Note:

FID is an acronym for Feature ID.

15. Notice that in the model tree the calculated fluid volume under the plane is **0.274** liters.

Note:

The remaining features do not affect the volume of the fuel tank, so it is acceptable to have them after the volume calculation.

16. Select the **Insert** indicator, right-click and select **Cancel > Yes**.

17. Save the model.

Step 2. Experiment with fluid levels.

1. Edit the FLUID_LEVEL plane to **35** and regenerate. Note the volumes have updated.
2. Simulate a full tank of fuel by editing the FLUID_LEVEL plane to **0** and regenerate. Note that the total capacity is **0.333** liters.
3. This does not fulfill the design specification, which calls for the FUEL_TANK.PRT to hold at least **0.35** liters.
4. Edit the BASE_PROTRUSION height and change the 59 dimension to **65** and regenerate. Note the volumes have updated, but that the volume is larger than **0.35** liters.
5. Edit the BASE_PROTRUSION height and change the 65 dimension to **61** and regenerate.
6. Note that when $d5 = 61$ the volume is approximately $350,000\text{mm}^3$ or 0.35 liters.

Note:

By using a sensitivity analysis, Pro/ENGINEER Wildfire enables you to find the dimensions that have the largest impact on changing the volume. After the analysis, you can plot out a value that will satisfy the requirements.

For example, you can create a sensitivity analysis on the BASE PROTRUSION'S d5 dimension that determines the volume when $d5 = 59$ through 65. After running the analysis, the plot that you create would illustrate that when $d = 61$ the volume is approximately 0.35 liters.

7. Save the model.
8. Close the window.
9. Erase the model from memory.

This completes the exercise.

Summary

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

- Create various types of analyses in design models.
- Create analysis features.

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Project IV

Introduction

Using the Pro/ENGINEER Wildfire, complete the following parts based upon the materials taught up to this point in class.

Objectives

After completing this module, you will be able to:

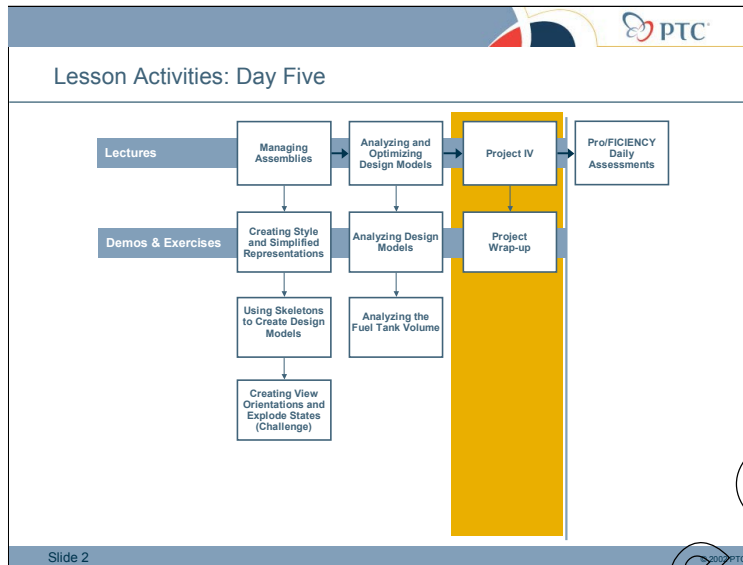
- Analyze the AC-40 for Interferences.
- Create and Assemble the Bolt.
- Create and Assemble the Engine Head.
- Create and Assemble the Glow Plug.
- Create and Assemble the Muffler.
- Create and Assemble the Carburetor.
- Create and Assemble the Engine Cover.
- Create and Assemble the Flange.
- Create and Assemble the Nut.

Fast Track to Pro/ENGINEER Wildfire

Module 19: Project IV

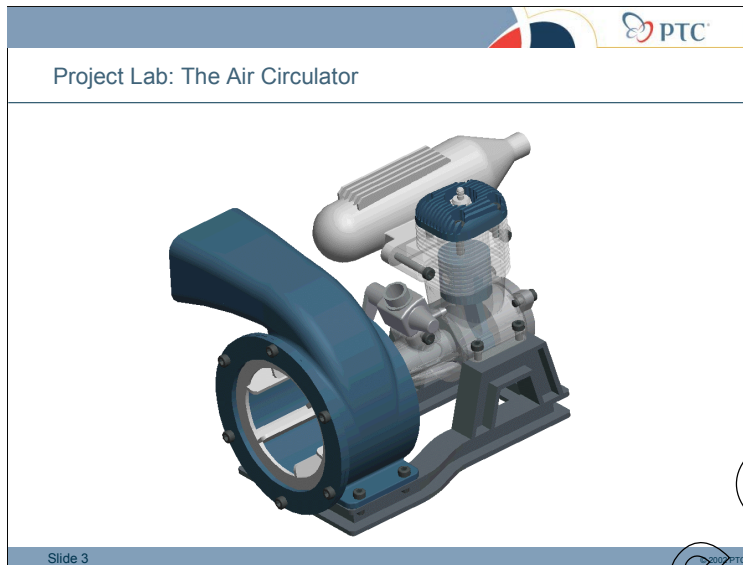


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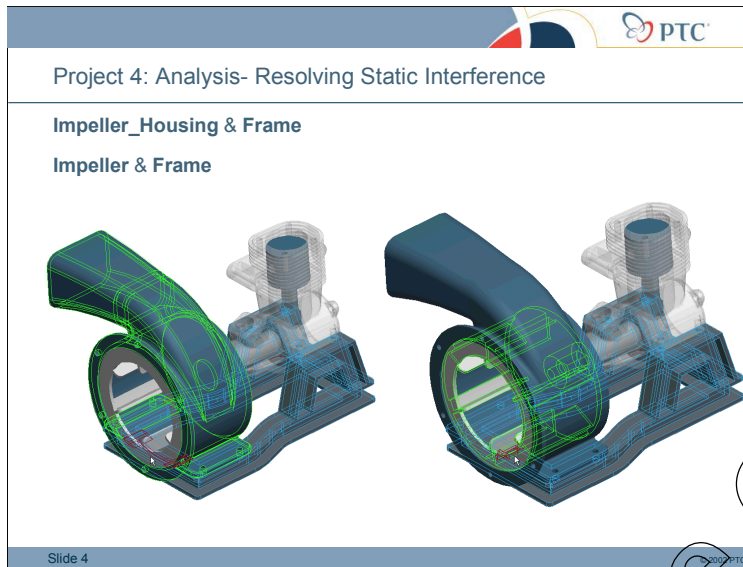


Duration

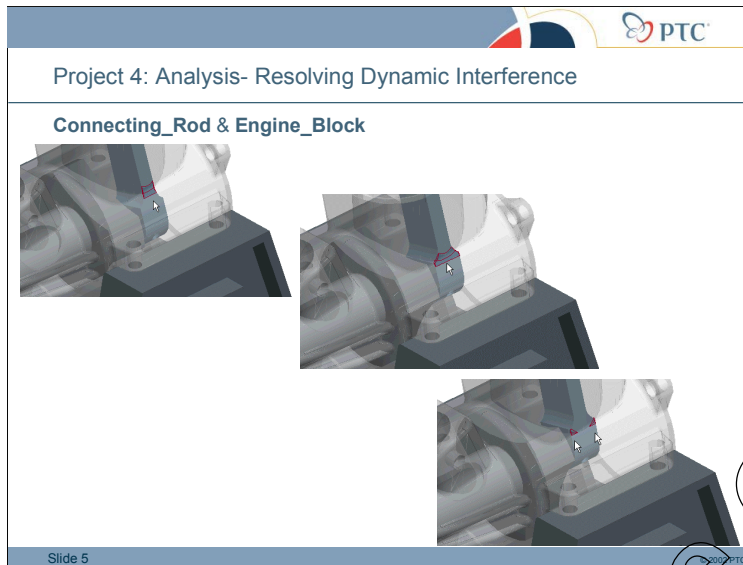
Total: 2 hr



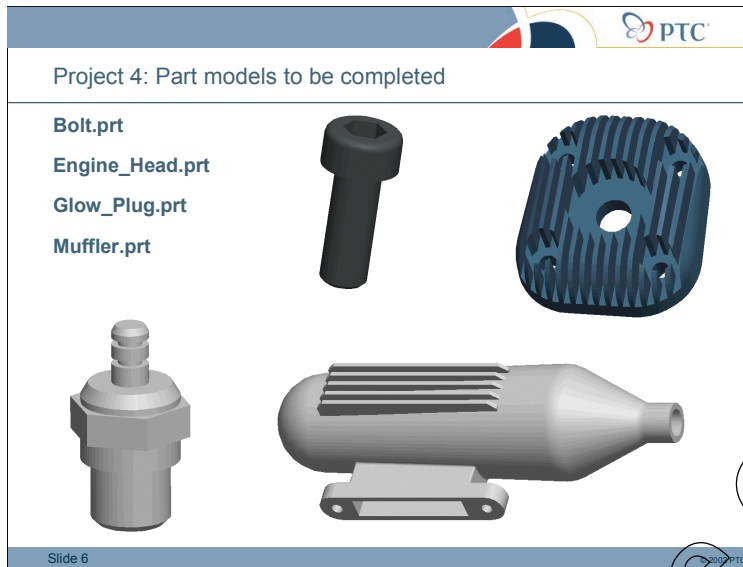
Use the remaining class time to complete the project. Note that most students (95%) will not complete the project. This is intentional. The files will be made available to them via PROJECTLINK.



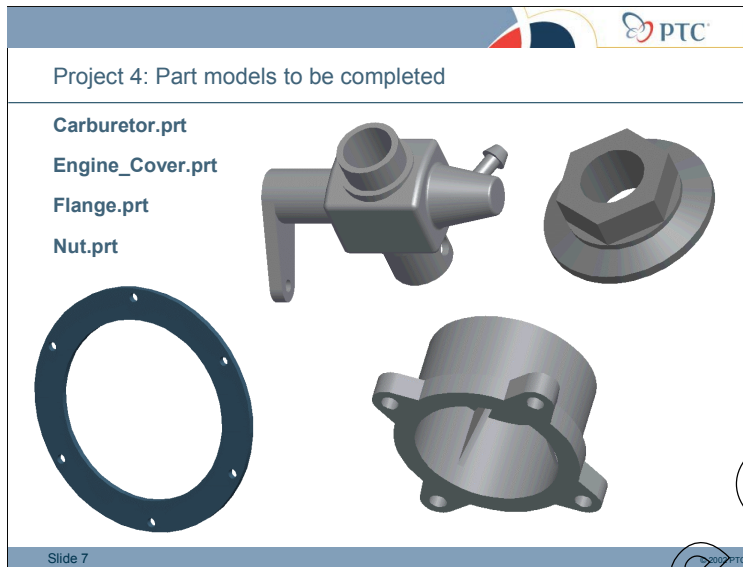
There are no instructor notes.



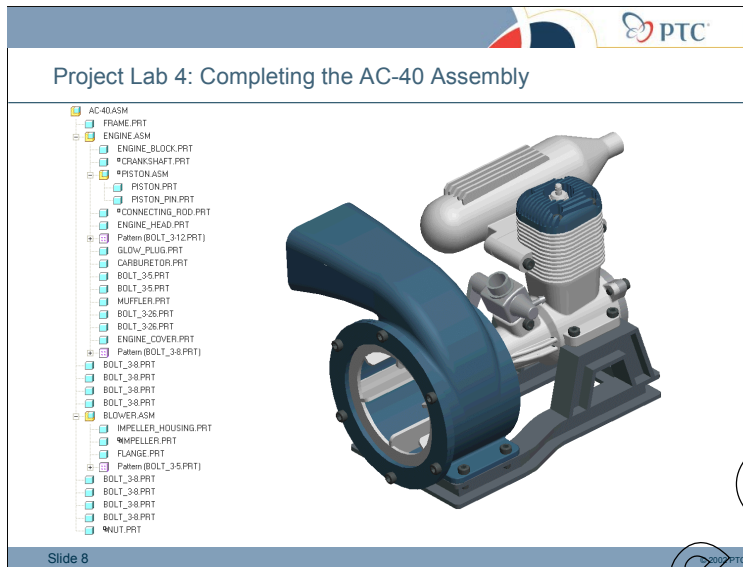
There are no instructor notes.



There are no instructor notes.



There are no instructor notes.



There are no instructor notes.

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Module 19 Lab Exercises

Scenario

ACME Inc. is a multinational conglomerate that develops and markets several consumer, industrial, and defense goods. The Light Industrial Division of ACME Inc creates a number of products including industrial fans, heating, air conditioning, pumps, and lighting. You work for the Light Industrial Division of ACME Inc., which has just started to use Pro/ENGINEER Wildfire for its product designs.

Upon returning from Pro/ENGINEER Wildfire training, you are assigned to create the AC-40 Air Circulator. As the name implies, the AC-40 is an air circulation device. Its primary purpose is to enable fast effective clean up of flooded homes and businesses that have no power due to the flood. Another variation of the AC-40 is currently being designed to use electric power. A further variation that includes an air compressor and condenser that will provide back-up air conditioning to high-rise buildings is also being considered.

In the final phase of the project, you create and complete the parts that will be assembled into the AC-40. You will also check the assembly to ensure that there are no interfering parts.

Objectives

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

- Analyze the AC-40 for Interferences
- Create and Assemble the Bolt
- Create and Assemble the Engine Head.
- Create and Assemble the Glow Plug.
- Create and Assemble the Muffler.
- Create and Assemble the Carburetor.
- Create and Assemble the Engine Cover.
- Create and Assemble the Flange.
- Create and Assemble the Nut.

Objective 1: Analyzing the AC-40 for Interferences

Step 1. Analyze static interferences in the AC-40 assembly.

1. If Pro/ENGINEER Wildfire is open, close all windows and erase all models from session. Otherwise, start Pro/ENGINEER Wildfire.
2. Using the folder navigator, browse to the following location:
c:\users\student\fast_track_330\Project. Make this folder the working directory.

3. Open the AC-40.ASM. Activate Mechanism mode. Drag the IMPELLER.PRT to position the PISTON.ASM at approximately Top Dead Center (TDC), as shown in the following figure.
4. Enable Standard mode.

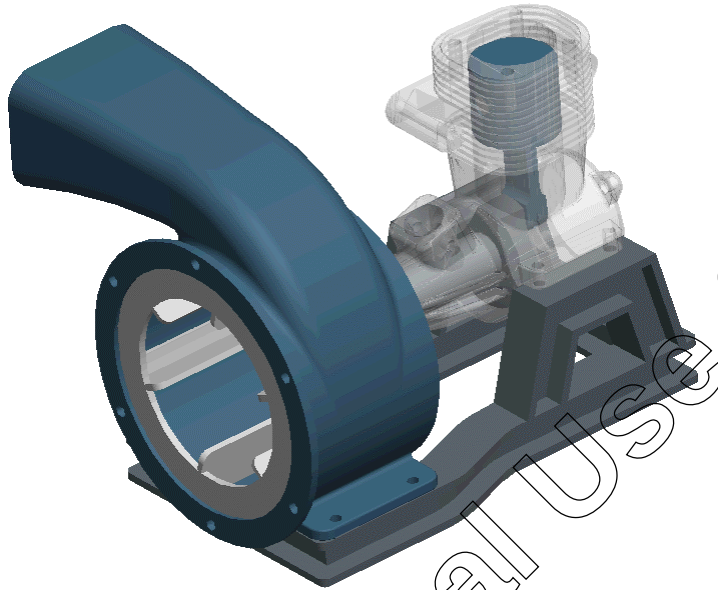


Figure 1: AC-40 Assembly

5. Run a Model Analysis to compute Global Interference. Select pair 1. Notice the IMPELLER_HOUSING.PRT interferes with the FRAME.PRT.

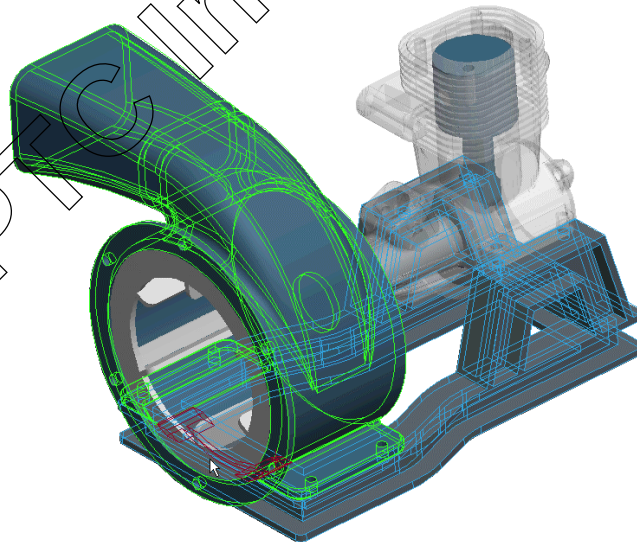


Figure 2: Interference Pair 1

6. Select pair 2. Notice the IMPELLER.PRT also interferes with the FRAME.PRT.

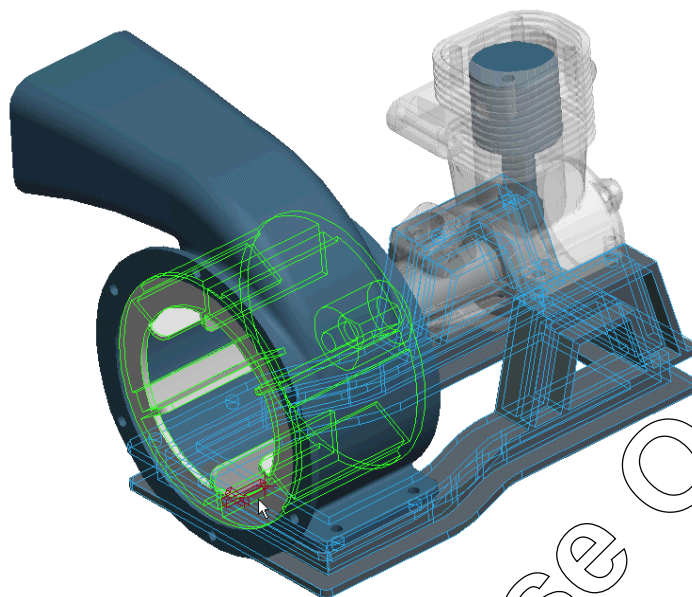


Figure 3: Interference Pair 2

Step 2. Resolve the interferences with the FRAME.PRT.

1. Open FRAME.PRT.

Design Intent:

There should be no interference among models when the assembly is not in motion.

2. Create a cut as shown in the following figures.

Design Intent:

Orient the sketch so that the TOP plane is horizontal. Use the RIGHT plane and model surfaces for references.

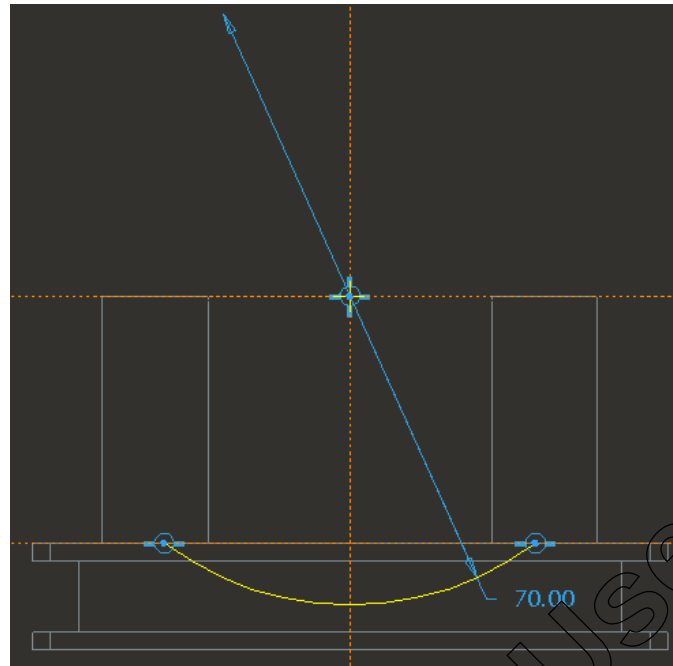


Figure 4: Creating a Protrusion

Design Intent:

The depth should only extend only through the surfaces shown below.



Figure 5: Creating a Cut

3. Save the model and close the window to return to the AC-40.ASM
4. Re-run the Global Interference check. No models should interfere.

Step 3. Analyze dynamic interferences.

1. Enable Mechanism mode, and run the analysis.
2. Playback the analysis results, enabling the Global Interference option.

3. Notice that interference occurs twice per revolution between the CONNECTING_ROD.PRT and the ENGINE_BLOCK.PRT.

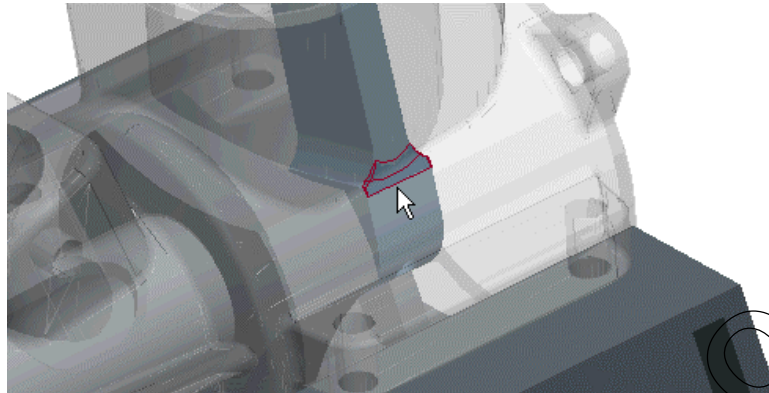


Figure 6: Interference Detected

Design Intent:

The connecting rod should not interfere as it will cause engine failure.

Step 4. Resolve the interference.

1. Enable Standard mode.
2. Orient the AC-40 assembly to the BACK view.
3. Activate the ENGINE_BLOCK.PRT. Sketch a cut on the CTR plane as shown in the following figure.

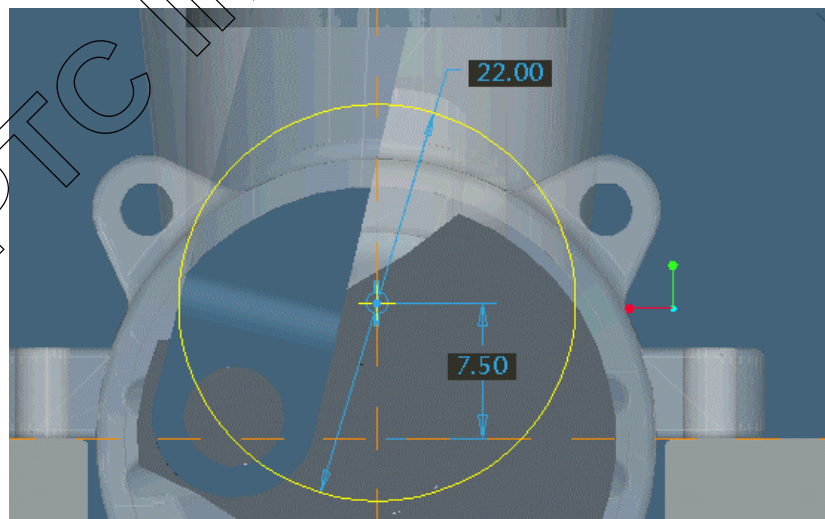


Figure 7: Creating a Cut

4. Specify a symmetric width of 7.

Design Intent:

Material should be removed on the inside of the sketch.

5. Create a round with a radius of 5 on the lower horizontal edges created by the previous cut, as shown in the following figure.

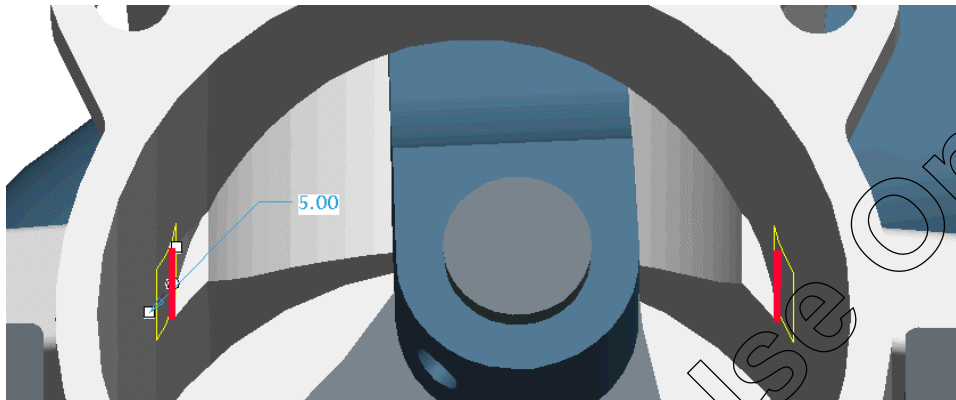


Figure 8: Creating a Round

6. Activate the AC-40.ASM. Open the CRANKSHAFT.PRT.
7. Edit the dimension controlling the height of the crank pin to 8, as shown in the following figure.

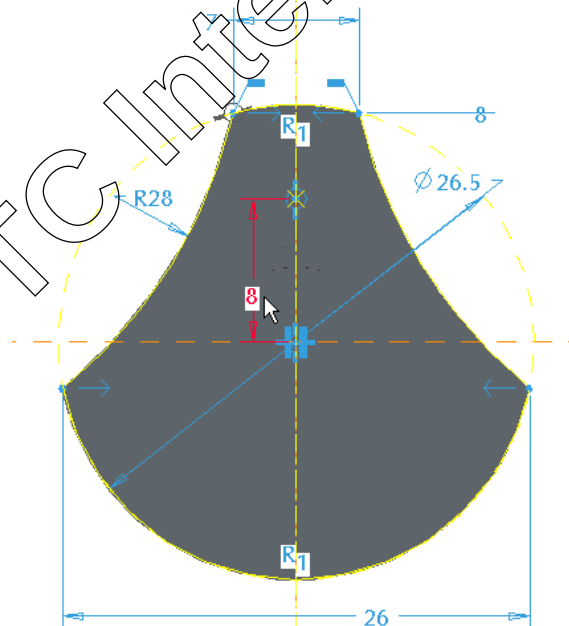


Figure 9: Modified Dimension

8. Regenerate the CRANKSHAFT.PRT. Close the window to return to the AC-40.ASM.

9. Enable Mechanism mode, and run the analysis. Playback the analysis results, enabling the Global Interference option.
 10. Notice that no interference occurs between the CONNECTING_ROD.PRT and the ENGINE_BLOCK.PRT.
 11. The assembly will be continued in later exercises. Save the model and close the window.
- This completes the exercise.

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Objective 2: Creating and Assembling the Bolt

Step 1. Create the BOLT.PRT.

1. Create a new part named BOLT.PRT.
2. Using the following figure, create a series of features to create the model.

Design Intent:

You may want to create the features in the following order:

- Create a protrusion representing the bolt shaft.
- Create a protrusion for the bolt head
- Create a hexagonal cut. Use a construction circle to easily create the sketch.
- Create a round and a chamfer

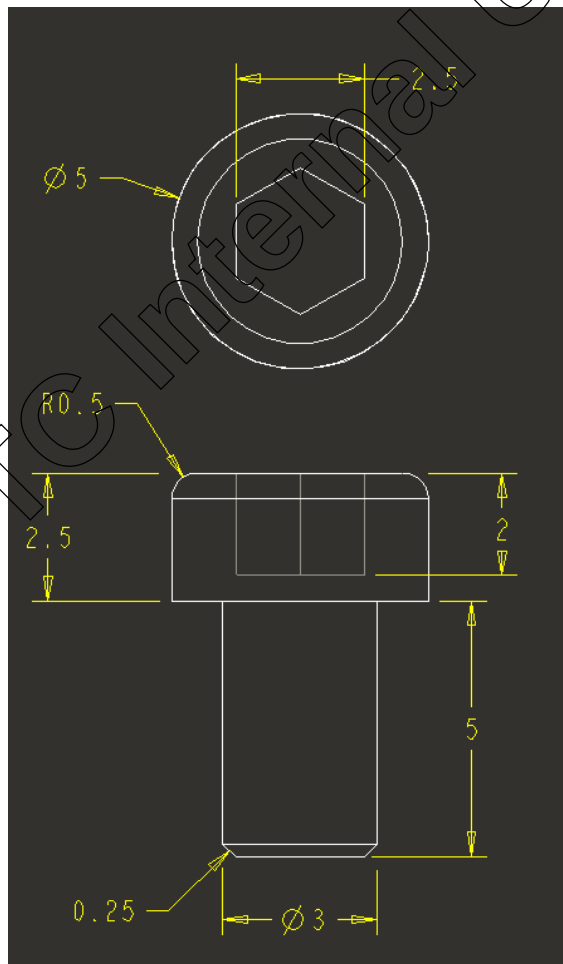


Figure 10: Bolt Specifications

3. Apply the Black appearance to the part.

Step 2. Create a family table to vary the bolt length.

1. Edit the properties of the dimension shown in the following figure to change its name to **bolt_length**.

Design Intent:

Descriptive names make dimensions easy to recognize.

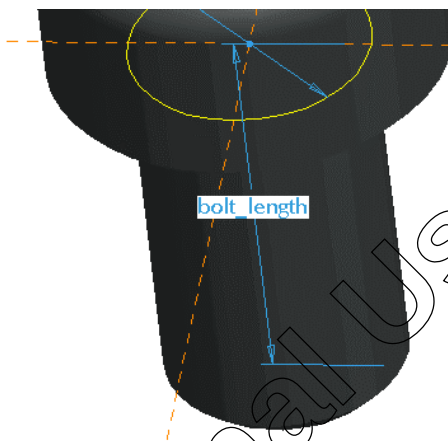


Figure 11: Editing Dimension Name

2. Create a family table varying the bolt length as shown in the following figure.

Instance Name	d0 BOLT_LENGTH
BOLT	5.00
BOLT_3-5	5.00
BOLT_3-8	8.00
BOLT_3-12	12.00
BOLT_3-26	26.00

Figure 12: Creating a Family Table

Design Intent:

The assembly requires bolts with various lengths.

Step 3. Assemble the bolt to fasten the ENGINE.ASM and BLOWER.ASM subassemblies.

1. Setup a component interface named BOLT. Select an insert surface from the bolt shaft and a mating surface under the bolt head.
2. Save the model and close the window.
3. Open the AC-40.ASM.

4. Display features in the model tree in order to display the Insert indicator.
5. Collapse all parts and assemblies to view only the top-level models.
6. Drag the Insert indicator to after the ENGINE.ASM and before BLOWER.ASM.
7. Assemble four of the BOLT_3-8 instances as shown in the following figure. Use the BOLT interface to speed assembly.

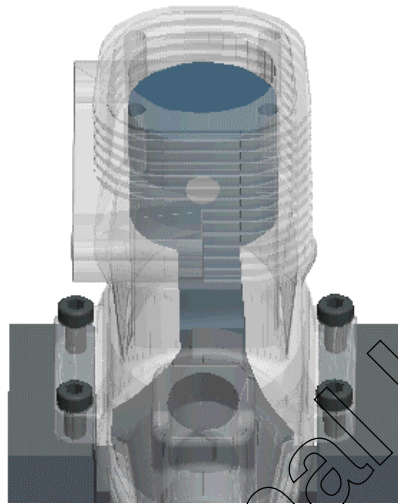


Figure 13: Bolts Assembled

Design Intent:

The bolts should come directly after the ENGINE.ASM in the model tree.

8. Drag the insert indicator to just after the BLOWER.ASM.
9. Assemble four of the BOLT_3-8 instances as shown in the following figure. Use the pre-defined BOLT interface to speed assembly.

Design Intent:

The bolts should come directly after the BLOWER.ASM in the model tree.

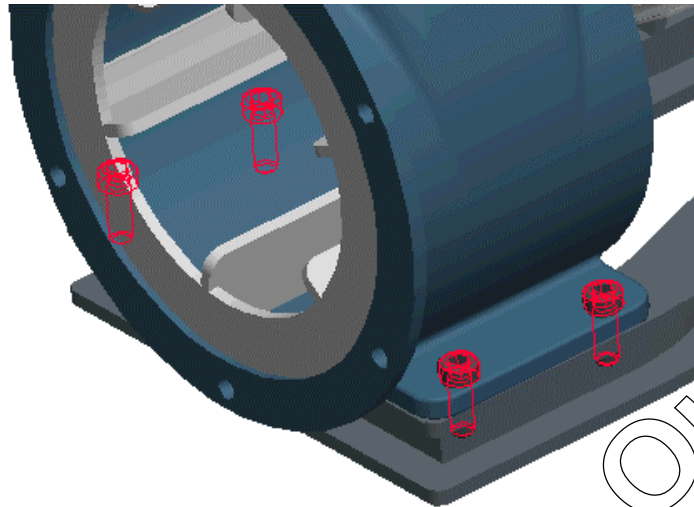


Figure 14: Bolts Assembled

10. The model tree should appear as in the following figure.

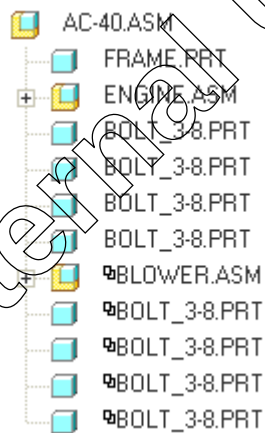


Figure 15: Model Tree

11. The assembly will be continued in later exercises. Save the model and close the window.

This completes the exercise.

Objective 3: Creating and Assembling the Engine Head

Step 1. Create the ENGINE_HEAD.PRT. Create the main body of the model.

1. Create a new part named ENGINE_HEAD.PRT.
2. Create a protrusion on the TOP plane. Sketch as shown in the following figure.

Design Intent:

The sketch should be symmetric about the vertical reference.

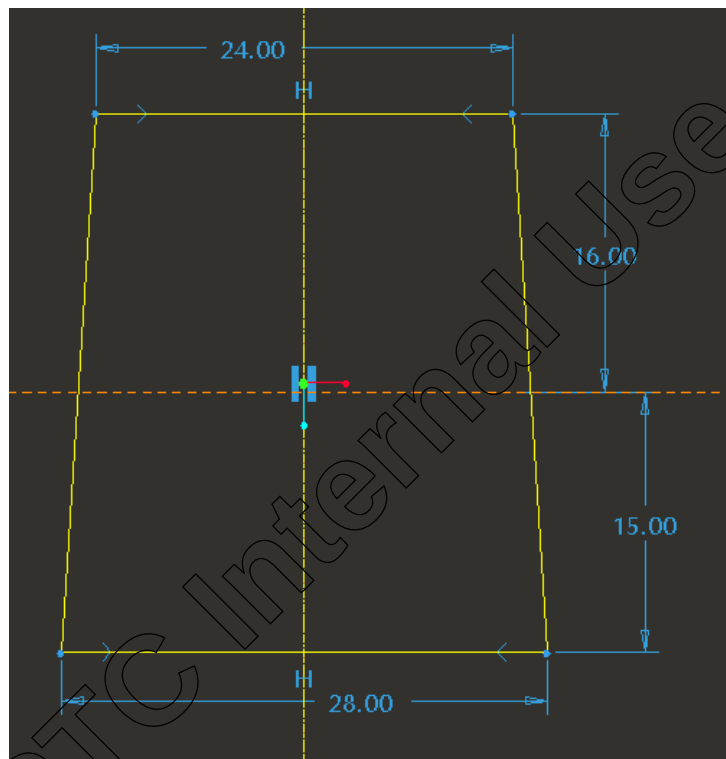


Figure 16: Creating a Sketch

3. Specify a depth of **0.50**, upward from the TOP plane.
4. Create a round as shown in the following figure.

Design Intent:

The edges in the back require a larger radius than the front edges.

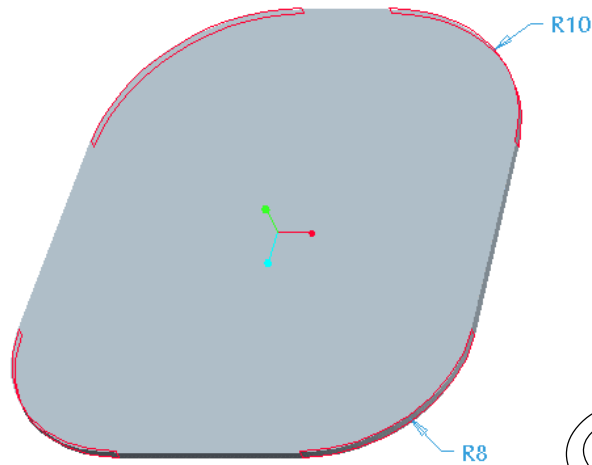


Figure 17: Creating a Round

5. Create a protrusion on the top surface of the previous protrusion, as shown in the following 2 figures.

Design Intent:

The sketch for this feature should be evenly offset 2 from the previous sketch.

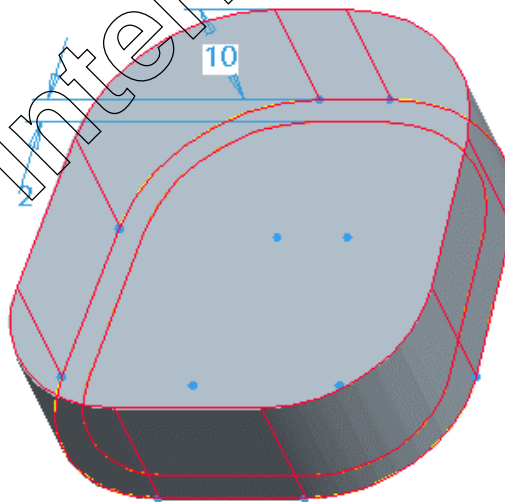


Figure 18: Creating a Protrusion

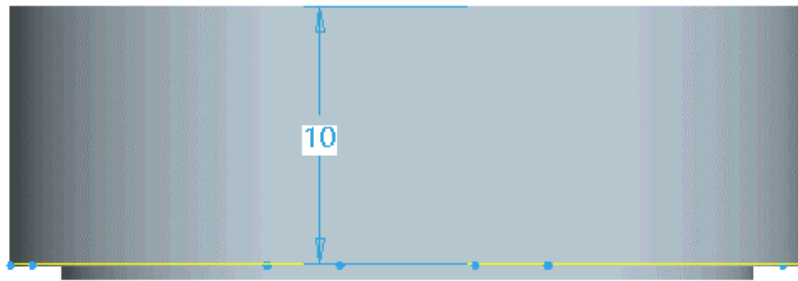


Figure 19: Creating a Protrusion

6. Create a draft of 5 degrees, as shown in the following figure.

Design Intent:

The top surface of the model should get smaller with the draft, while the bottom surface should remain the same size.

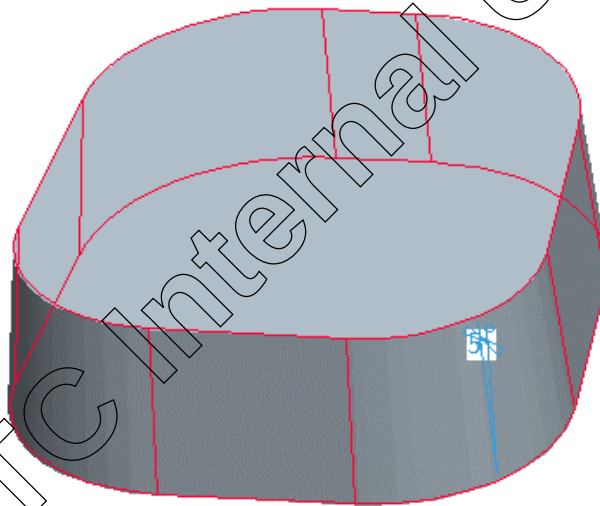


Figure 20: Creating a Draft

7. Create a round as shown in the following figure.

Design Intent:

This round should have a constant radius.

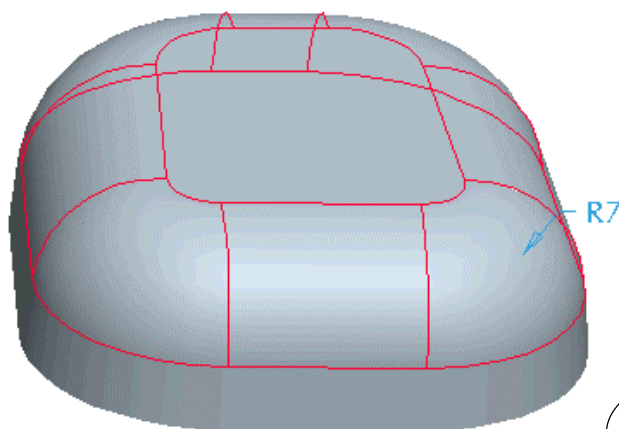


Figure 21: Creating a Round

Step 2. Create holes for the GLOW_PLUG.PRT.

1. Start the Hole tool. Create an axis at the intersection of the FRONT and RIGHT planes. Select the axis to locate the hole, as shown in the following figure.

Design Intent:

The hole should extend through the entire part.

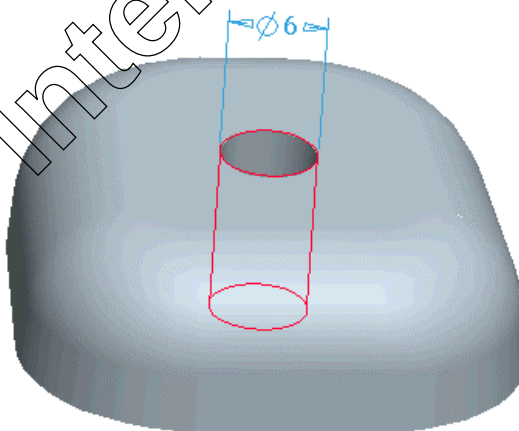


Figure 22: Creating a Hole

2. Name the group CENTER_HOLE.
3. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial to the previous hole.

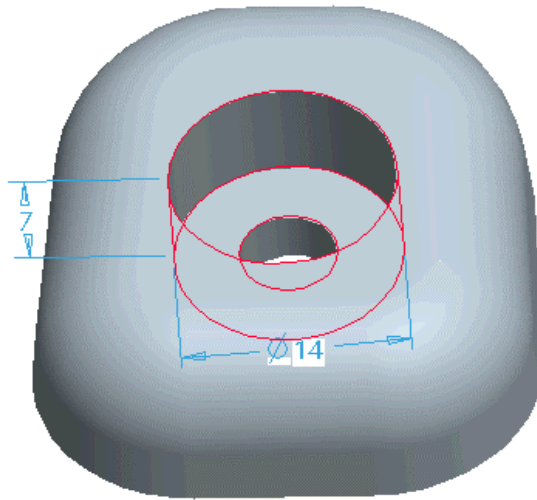


Figure 23: Creating a Hole

4. Drag the hole into the CENTER_HOLE group.

Step 3. Create a protrusion to form a 'hemi' combustion chamber.

1. Create a revolve on the RIGHT plane. Sketch as shown in the following figure.

Design Intent:

The sketch should reference the surface of the CENTER_HOLE, and the lower model surface. Specify the axis of revolution as the vertical reference.

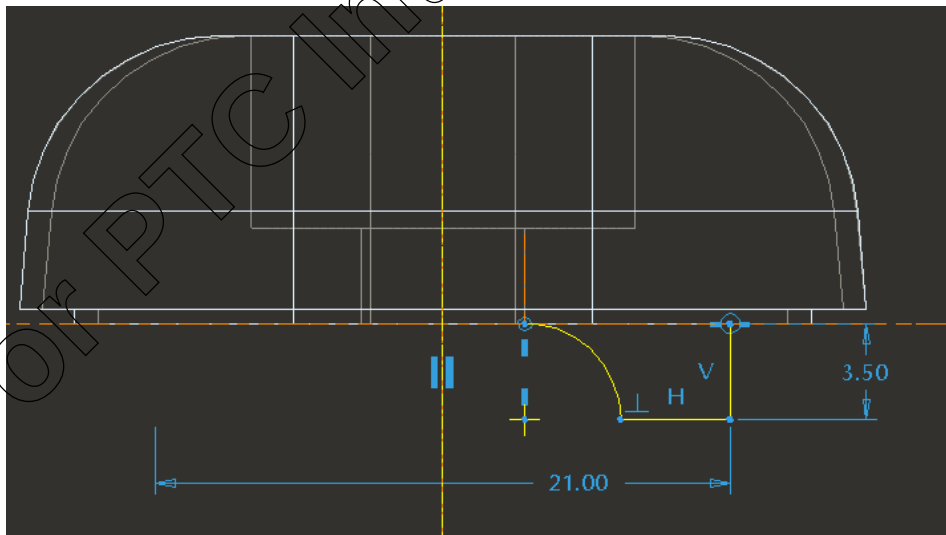


Figure 24: Creating a Sketch

2. Revolve the protrusion 360 degrees, as shown in the following figure.

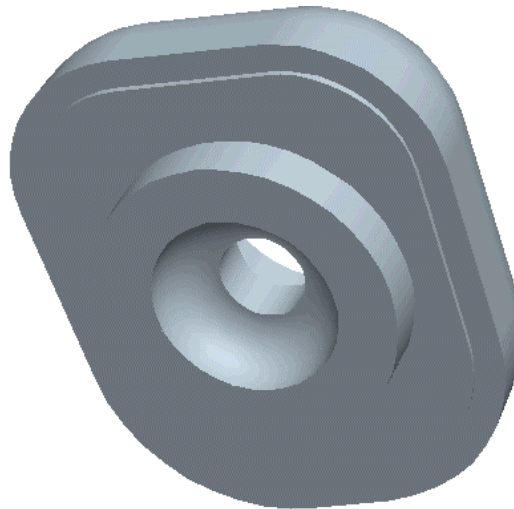


Figure 25: Creating a Protrusion

Step 4. Create cooling fins.

1. Create a thin cut on the top surface of the model. Sketch as shown in the following figure.

Design Intent:

The sketch should reference the outermost model edges.

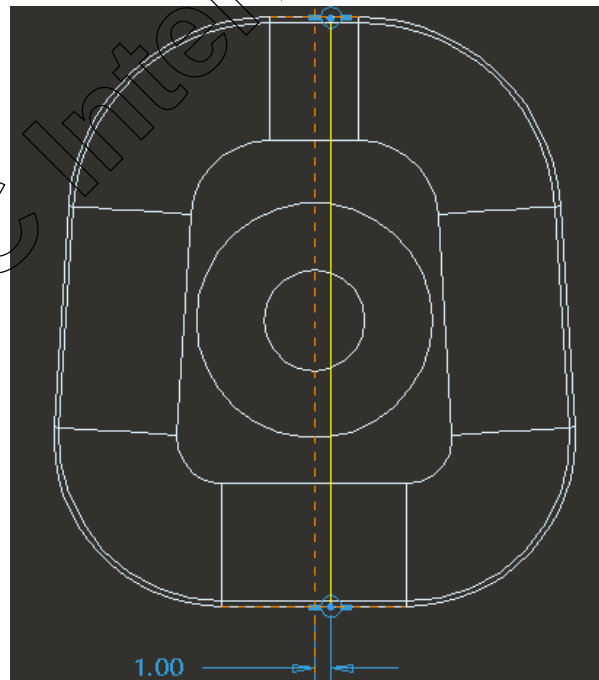


Figure 26: Creating a Sketch

2. Specify a symmetric width of 1.25 and the depth to be up to the surface of the hole as shown in the following figure.

3. Pattern the cut as shown in the following figure.

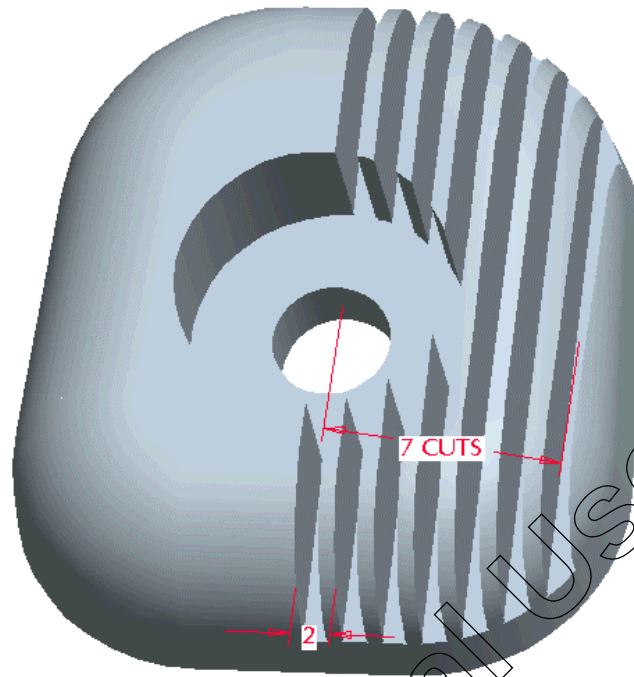


Figure 27: Pattern of Cuts

4. Mirror the pattern about the RIGHT plane. Rename the group to MIRROR_CUTS.

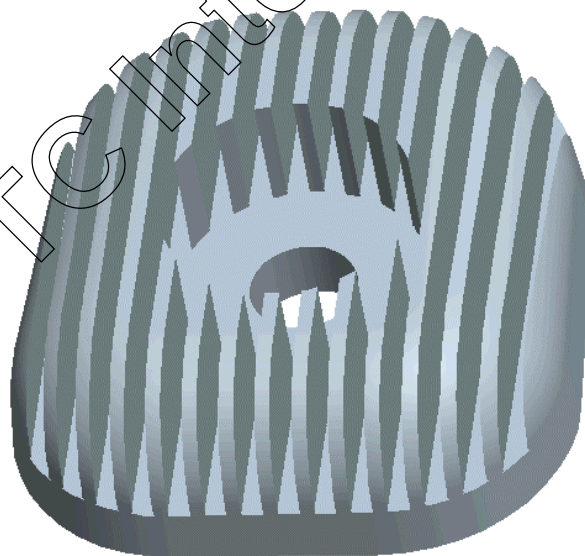


Figure 28: Pattern Mirrored

Step 5. Create mounting holes.

1. Orient to the BOTTOM view and create a hole as shown in the following figure.

Design Intent:

The hole should be radial (using the diameter option) from the center axis.

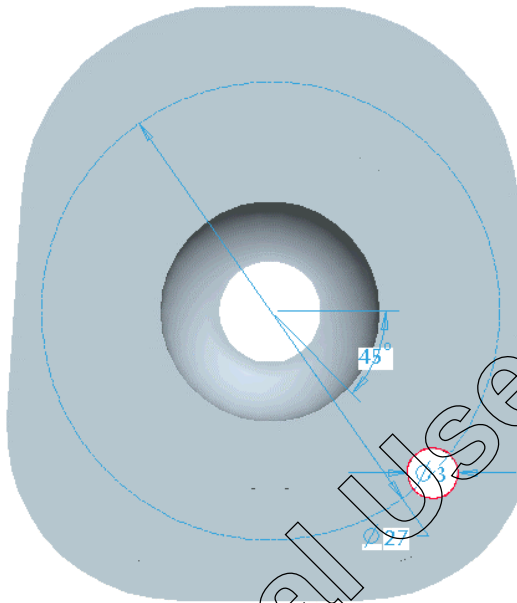


Figure 29: Creating a Hole

2. Pattern the hole as shown in the following figure.

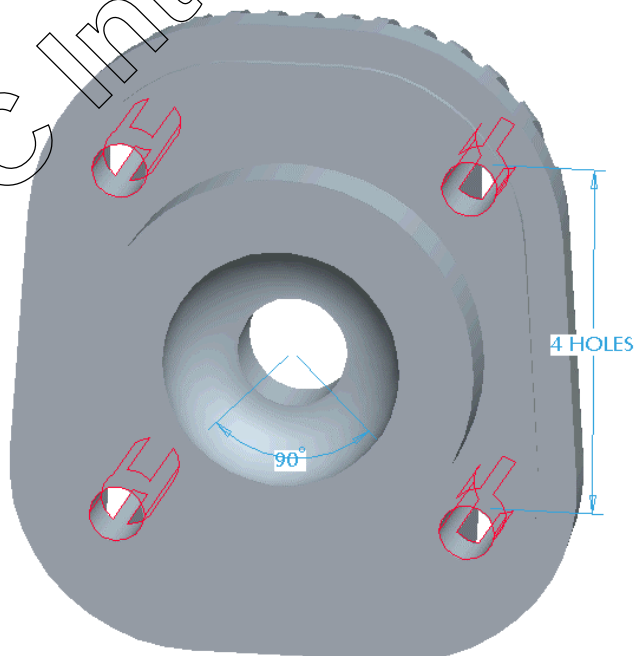


Figure 30: Pattern Created

3. Create and reference pattern a hole as shown in the following figure.

Design Intent:

The hole must be placed on the leader of the previous pattern.

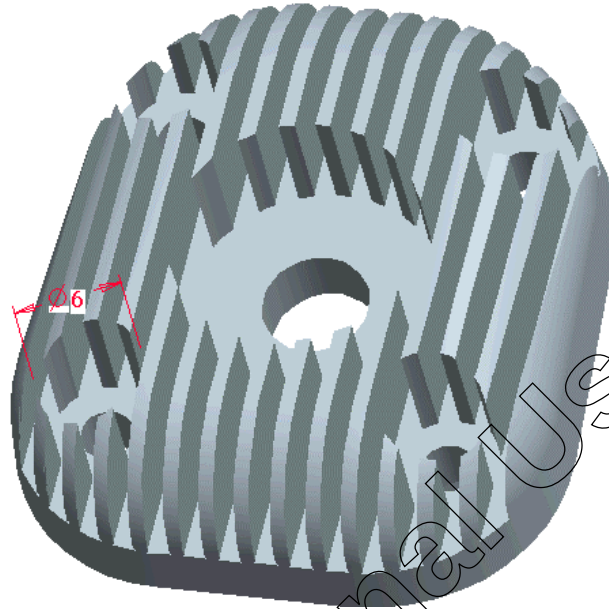


Figure 31: Creating a Hole and Reference Pattern

4. Apply the Blue_Dark appearance. Save the model and close the window.

Step 6. Assemble the ENGINE_HEAD.PRT and its bolts.

1. Open the AC-40.ASM. Activate the ENGINE.ASM.
2. Assemble the ENGINE_HEAD.PRT as shown in the following figure.

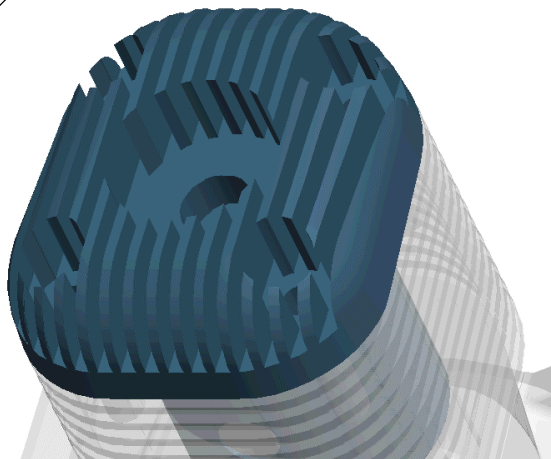


Figure 32: ENGINE_HEAD.PRT Assembled

3. Assemble a BOLT_3-12 instance as shown in the following figure.

Design Intent:

The bolt should be assembled to the leader of the hole pattern in the ENGINE_HEAD.PRT.

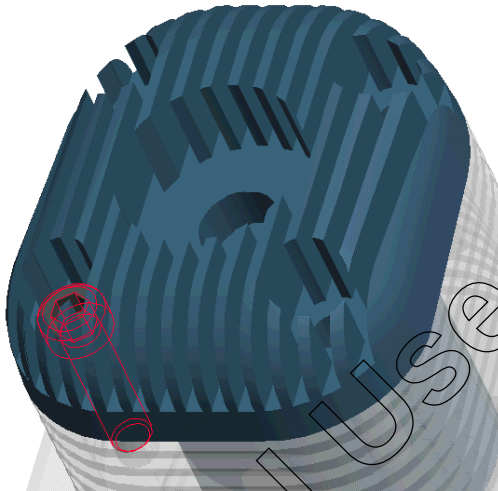


Figure 33: Bolt Assembled

4. Reference pattern the bolt as shown in the following figure.

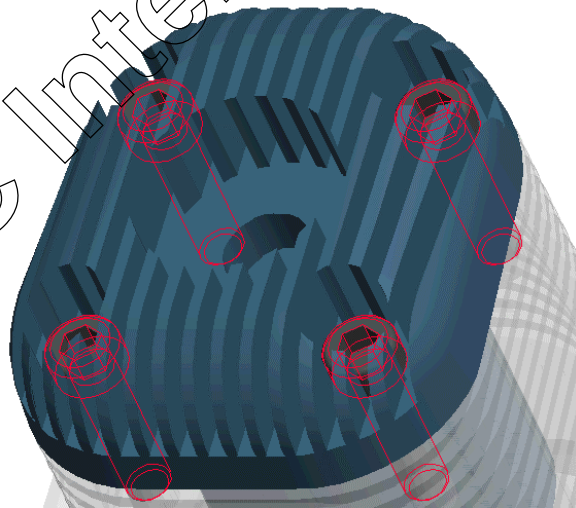


Figure 34: Bolt Patterned

5. The model tree should appear as shown in the following figure.

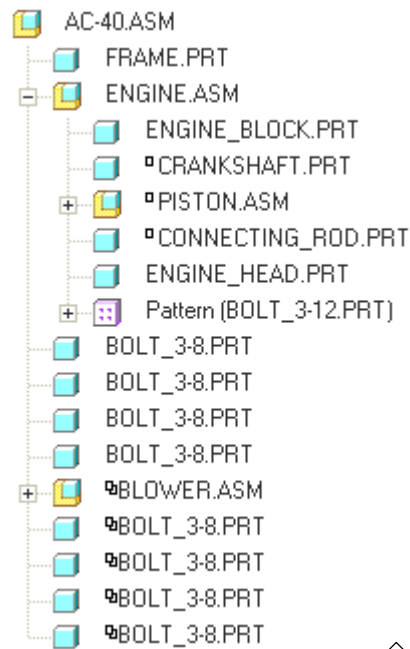


Figure 35: Model Tree

6. The assembly will be continued in later exercises. Save the model and close the window.

This completes the exercise.

Objective 4: Creating and Assembling the Glow Plug

Step 1. Create the GLOW_PLUG.PRT. Create the main body of the model.

1. Create a new part named GLOW_PLUG.PRT.
2. Create a revolved protrusion on the FRONT plane. Sketch as shown in the following figure.

Design Intent:

The sketch should be symmetric about the vertical reference.

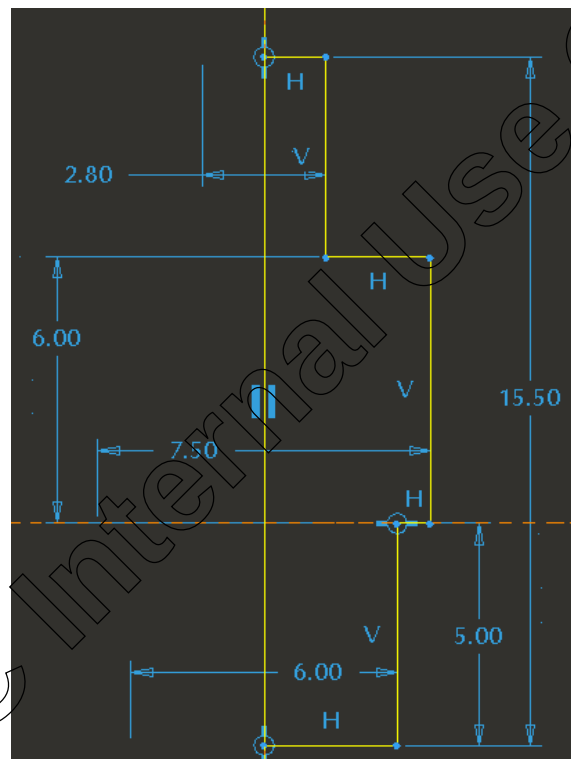


Figure 36: Creating a Sketch

3. Revolve the feature 360 degrees as shown in the following figure.

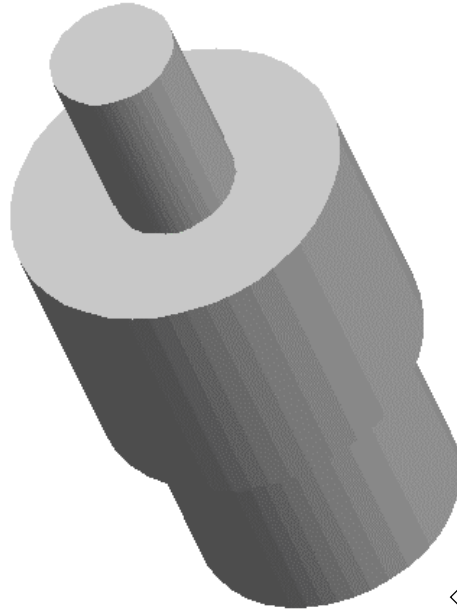


Figure 37: Creating a Protrusion

4. Apply the Silver appearance to the model.

Step 2. Create of additional features on the model.

1. Create a series of features as shown in the following figure.

Design Intent:

You may want to create the features in the following order:

- Create a protrusion representing the hex, offset 2 from the mating surface. Use a construction circle to easily create the sketch.
- Create a hole in the base of the model. The hole should be coaxial.
- Create a revolved cut to form the first groove.
- Copy the revolved cut to create a second groove.

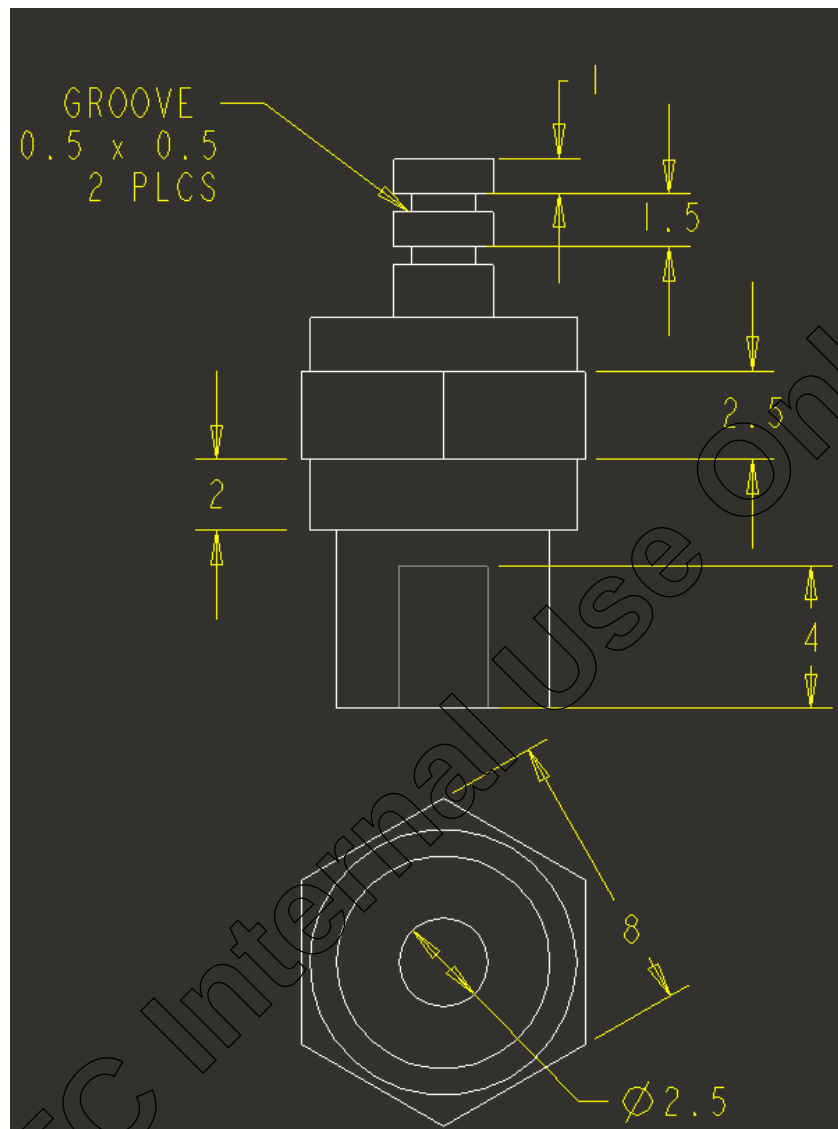


Figure 38: Glow Plug Specifications

Step 3. Complete the model.

1. Create a chamfer as shown in the following figure.

Design Intent:

The chamfer should reference multiple edges, and use multiple dimensioning schemes.

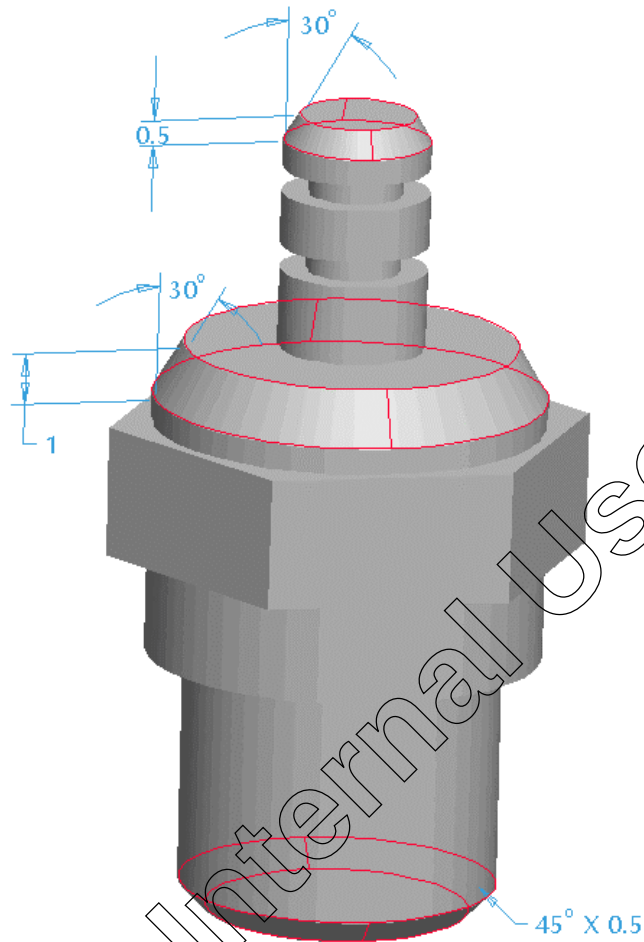


Figure 39: Creating a Chamfer

2. Save the model and close the window.

Step 4. Assemble the GLOW_PLUG.PRT.

1. Open the AC-40.ASM. Activate the ENGINE.ASM.
2. Assemble the GLOW_PLUG.PRT as shown in the following figure.

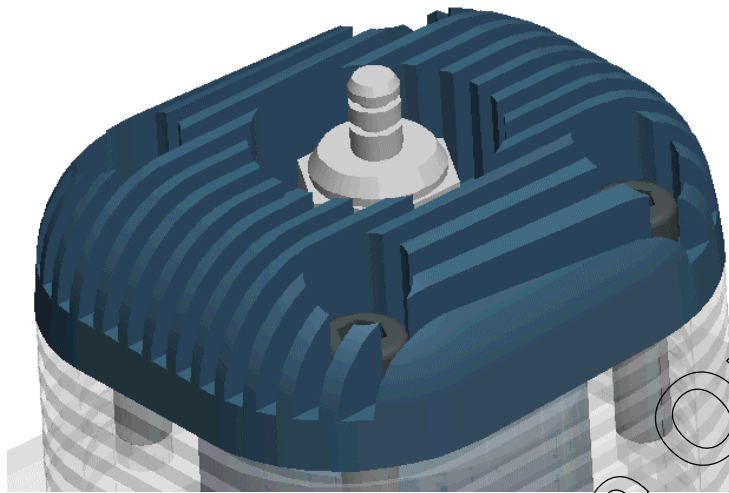


Figure 40: GLOW_PLUG.PRT Assembled

3. The model tree should appear as in the following figure.

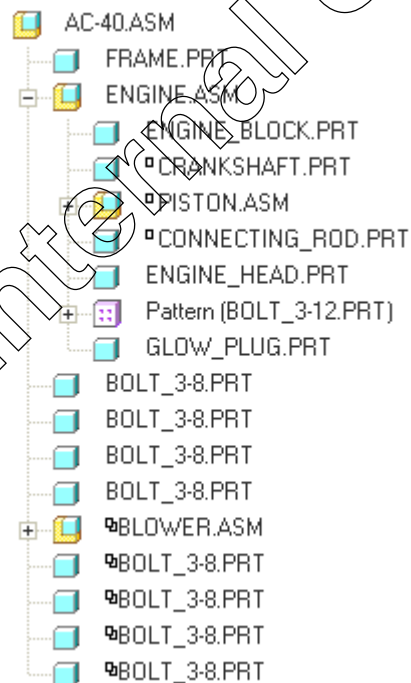


Figure 41: Model Tree

4. The assembly will be continued in later exercises. Save the model and close the window.
This completes the exercise.

5. Start the Extrude Tool, and then create a datum plane named OFFSET as shown in the following figure.

Design Intent:

The datum plane will be used as a reference plane in the next feature.

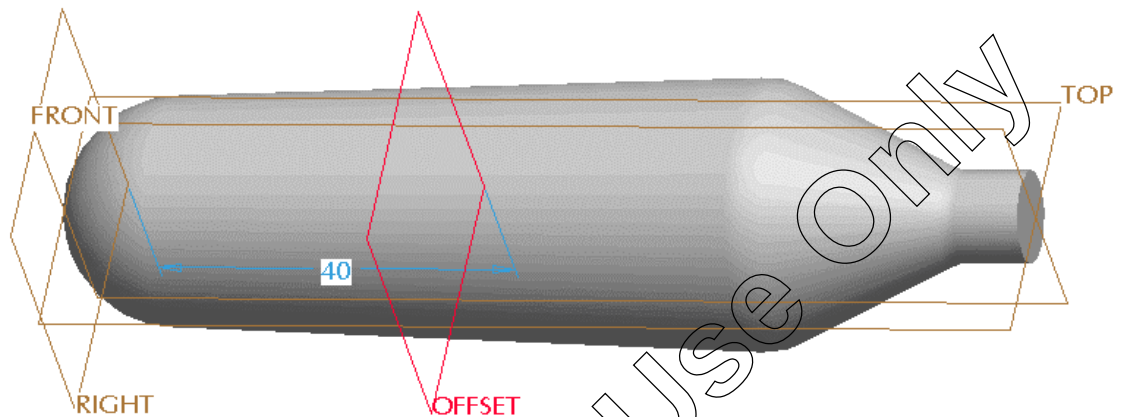


Figure 44: Creating a Datum Plane

6. Select the datum FRONT as a sketching plane, and the previous protrusion as a vertical reference. Sketch as shown in the following figure.

Design Intent:

The sketch should be symmetric about the OFFSET plane.

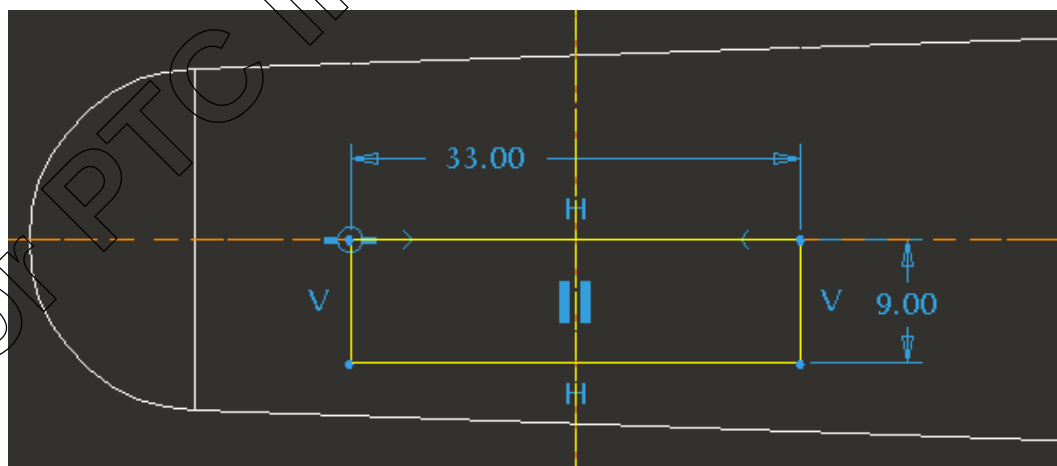


Figure 45: Creating a Sketch

7. Specify a depth of 24 from the FRONT plane. Name the group ENG_MOUNT.
8. Create a shell with a thickness of 1.5 as shown in the following figure.

Design Intent:

The shell should remove 2 surfaces.

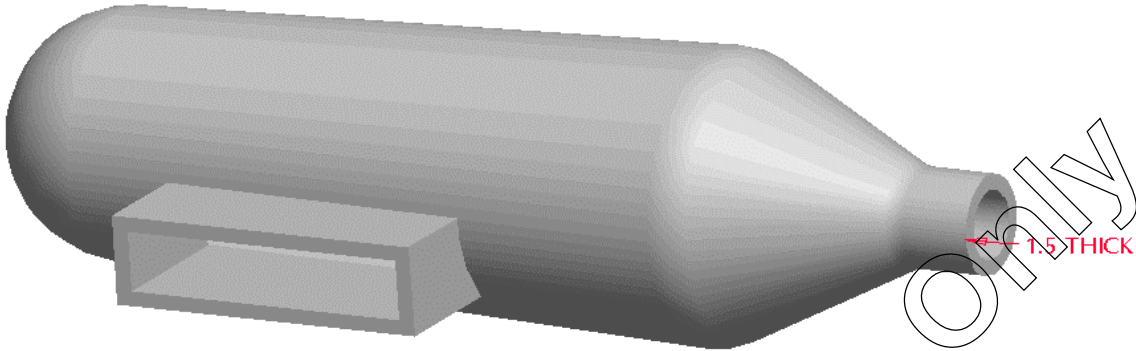


Figure 46: Creating a Shell

Step 2. Create mounting tabs.

1. Create a protrusion as shown in the following figure. Sketch an axis point at the center of the arc.

Design Intent:

Reference existing geometry. Dimension the arc from the OFFSET plane. A hole will later be placed using the axis point.

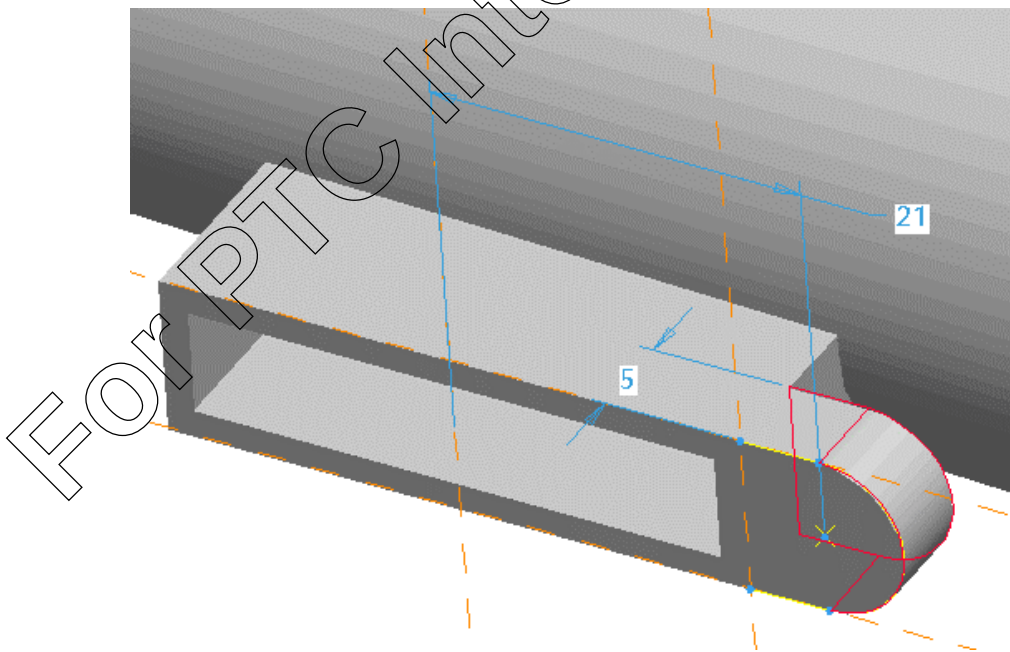


Figure 47: Creating a Protrusion

2. Create a similar protrusion on the opposite side.

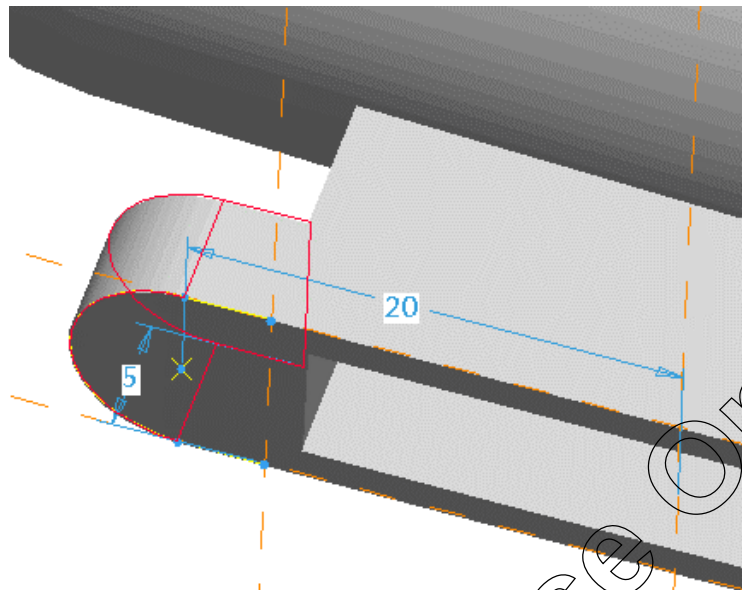


Figure 48: Creating a Protrusion

Step 3. Create cooling fins.

1. Start the Extrude tool. Then create a datum plane as shown in the following figure.

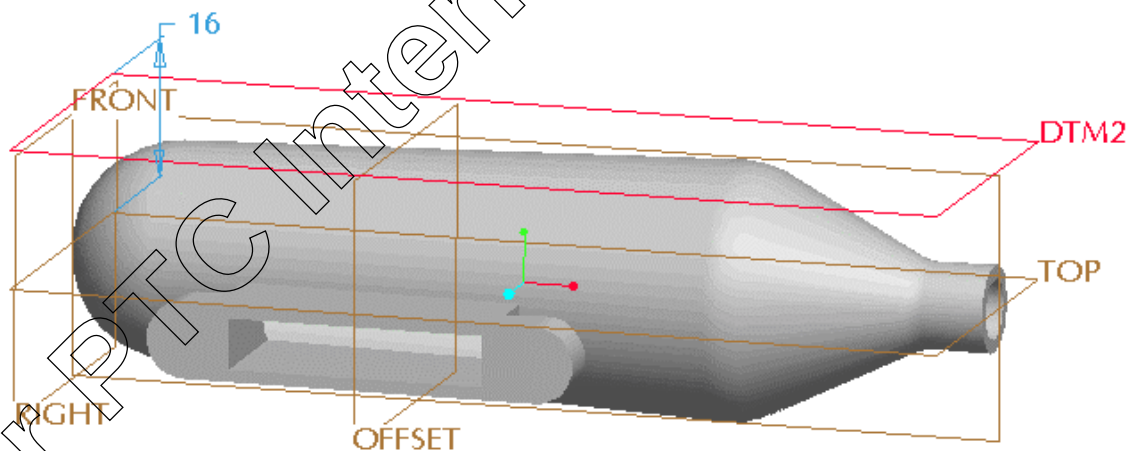


Figure 49: Creating a Datum Plane

Design Intent:

This plane will be used as a sketching plane for the next feature.

2. Select the previous plane as a sketching plane. Sketch as shown in the following figure.

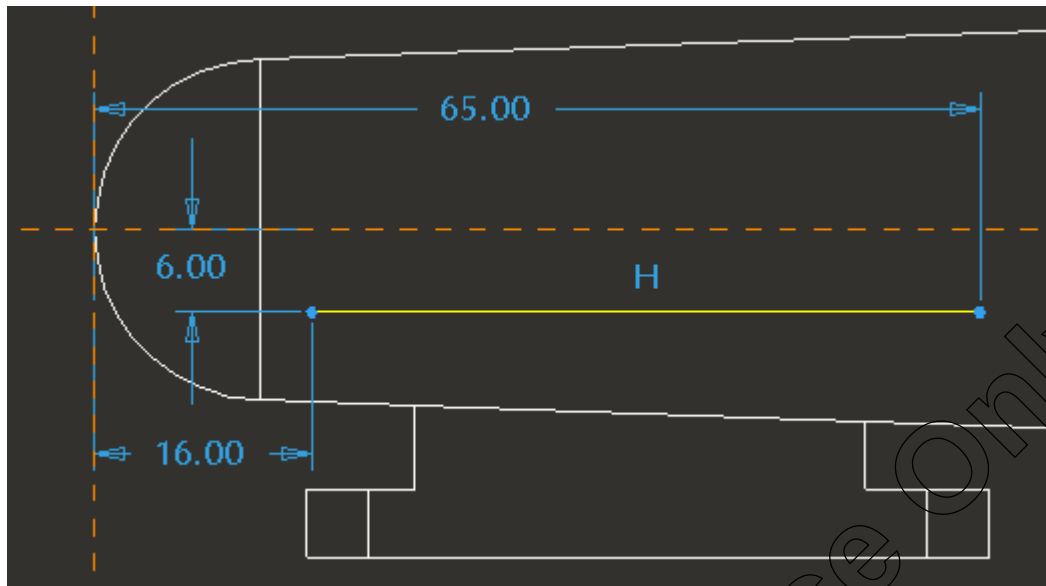


Figure 50: Creating a Sketch

Design Intent:

This protrusion will use the Thin option.

3. Specify a symmetric thickness of **1.0**, and a depth up to the revolved surface. Name the group FIN.
4. Pattern the FIN group as shown in the following figure.

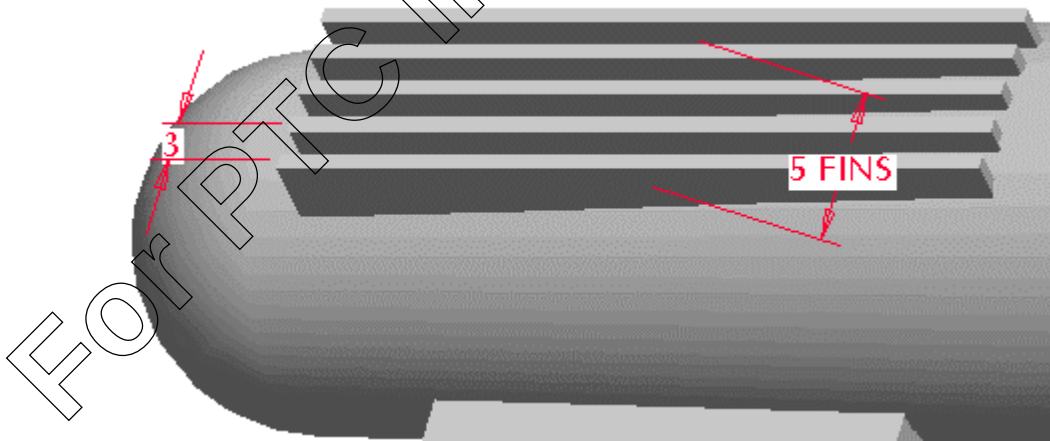


Figure 51: Creating a Pattern

5. Create a chamfer and reference pattern it as shown in the following figure.

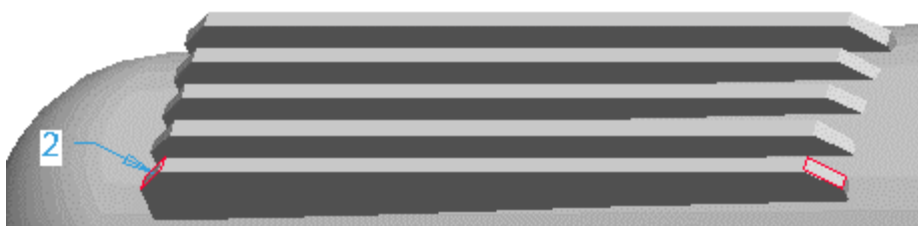


Figure 52: Chamfer Reference Pattern

Step 4. Create bolt holes and rounds.

1. Create two holes as shown in the following figure.

Design Intent:

The holes should be coaxial and equal in size.

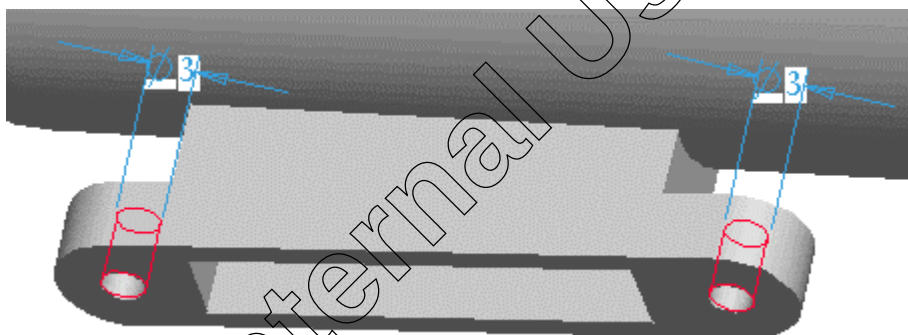


Figure 53: Creating Holes

2. Create a series of three separate round features, as shown in the following figure.

Design Intent:

The rounds should be created in an order that sets up tangent chains for the next round.

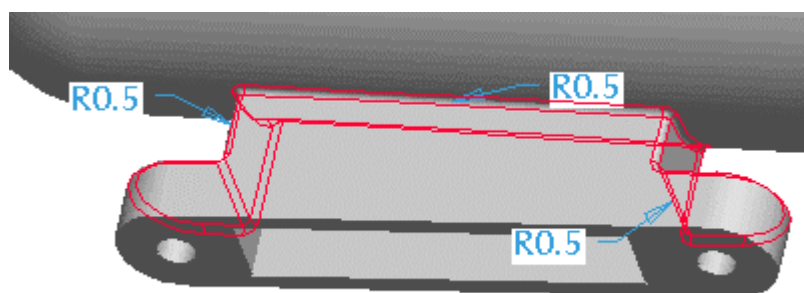


Figure 54: Creating Rounds

3. Apply the Silver appearance. Save the model and close the window.
4. Open the AC-40.ASM. Activate the ENGINE.ASM.
5. Assemble the MUFFLER.PRT and its corresponding bolts shown in the following figure.

Design Intent:

The BOLT_3-26 instances should be used.

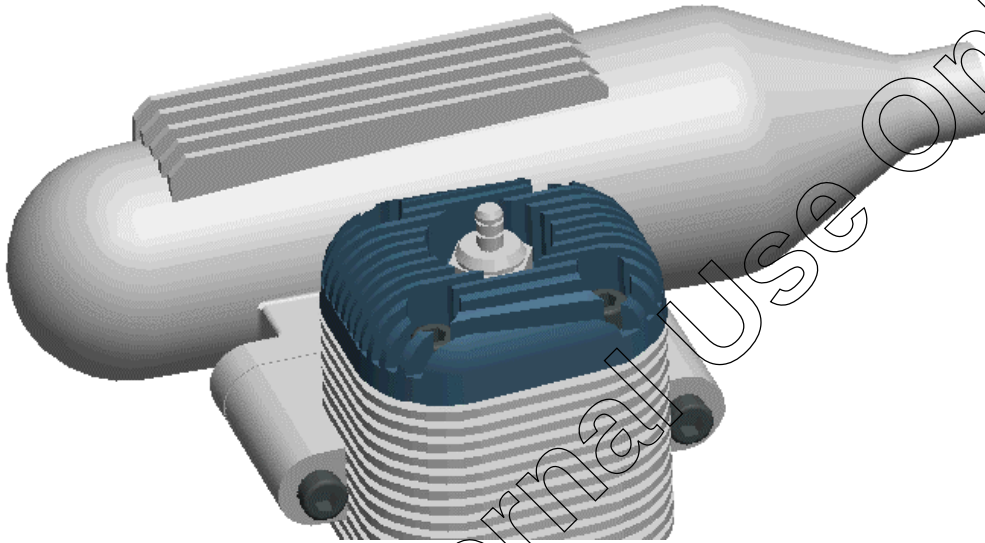


Figure 55. MUFFLER.PRT Assembled

6. The model tree should appear as in the following figure.

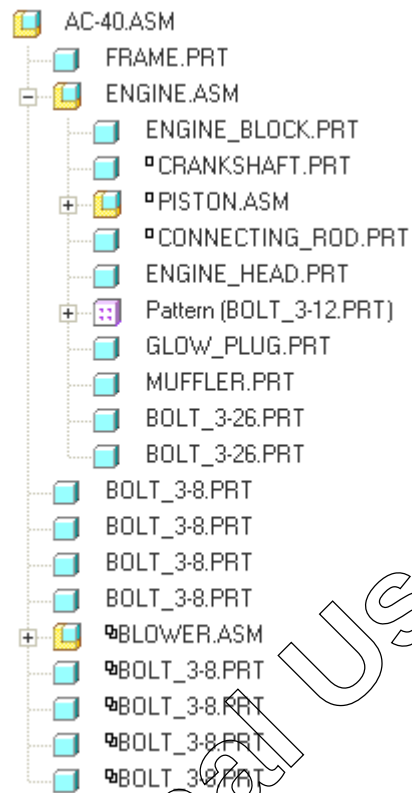


Figure 56: Flange1 Mirrored

7. The assembly will be continued in later exercises. Save the model and close the window.
This completes the exercise.

Objective 6: Creating and Assembling the Carburetor (Challenge)

Step 1. Create the CARBURETOR.PRT. Create the main body of the model.

1. Create a new part named CARBURETOR.PRT.
2. Create a series of features as shown in the following figure.

Design Intent:

You may want to create the features in the following order:

- Create a protrusion main 10 diameter cylinder, with asymmetrical blind depths.
- Create a 12 diameter cylinder, with a symmetrical depth.
- Create the 'rectangular' protrusion as shown in the right view.
- Create a revolved protrusion to form the cone shape.

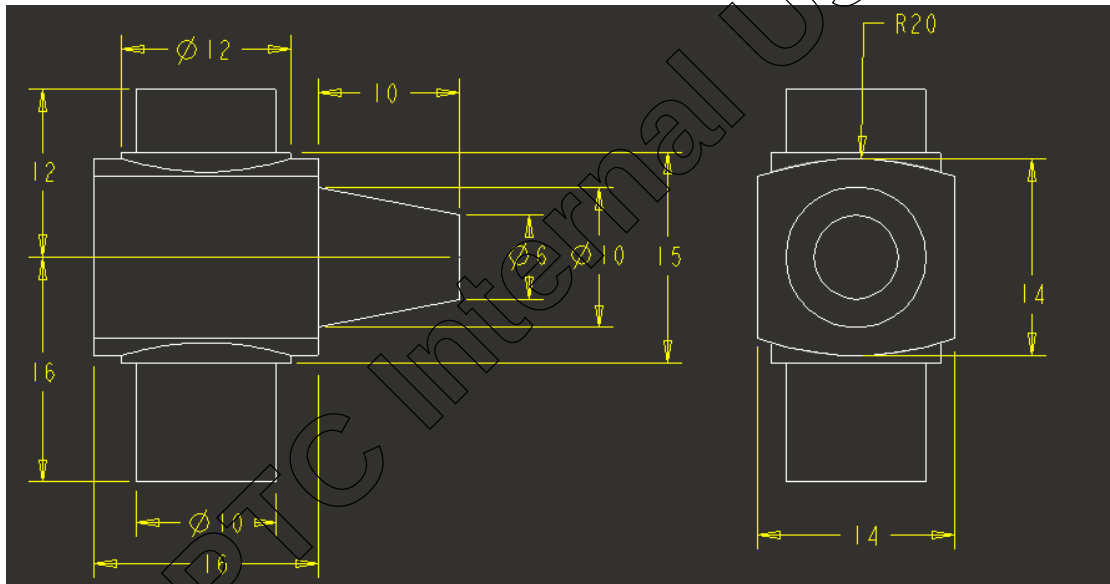


Figure 57: CARBURETOR.PRT Specifications

3. The model should appear as shown in the following figure.

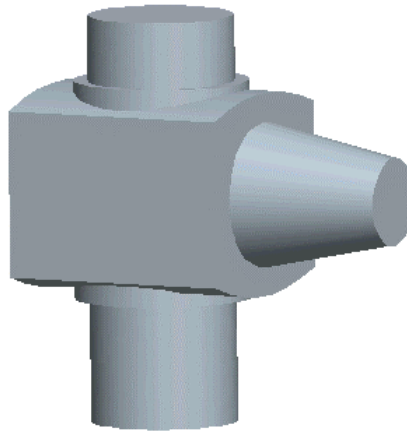


Figure 58: CARBURETOR.PRT Model

Step 2. Create a hole to allow the intake of air, and the fuel inlet nozzle.

1. Create a hole as shown in the following figure.

Design Intent:

The hole should be coaxial, and extend through the entire model.

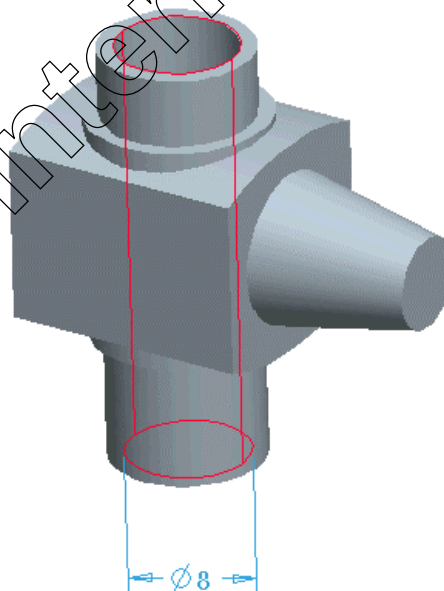


Figure 59: Creating a Hole

2. Create a revolved protrusion on TOP. Sketch as shown in the following figure.

Design Intent:

Use a sketcher point to locate the 'pivot' in the sketch.

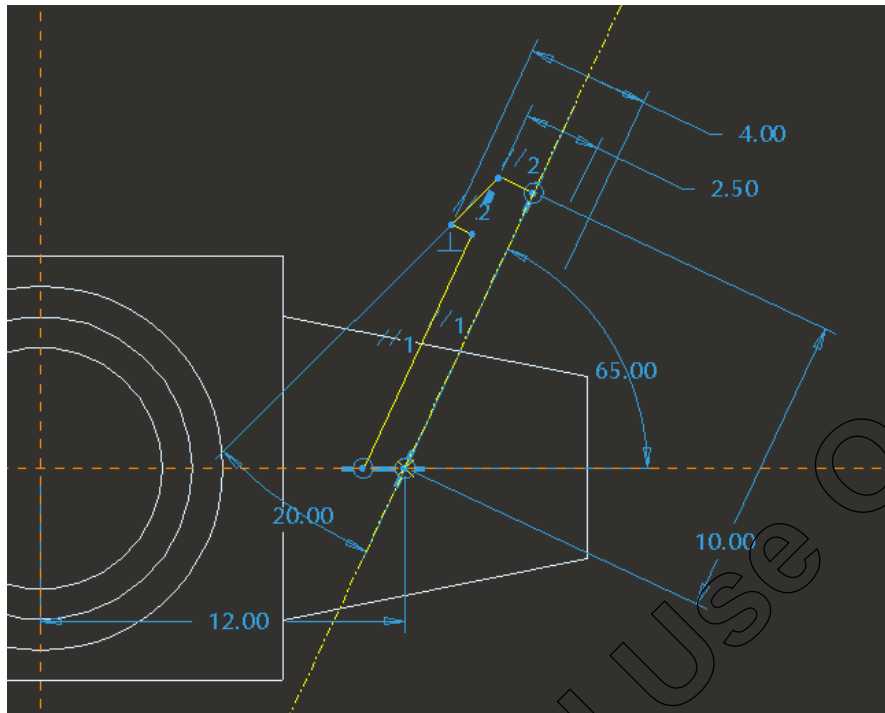


Figure 60: Creating a Sketch

Step 3. Create features to represent the throttle arm.

1. Create a protrusion as shown in the following figure.

Design Intent:

The depth should be dimensioned from the outer model surface.

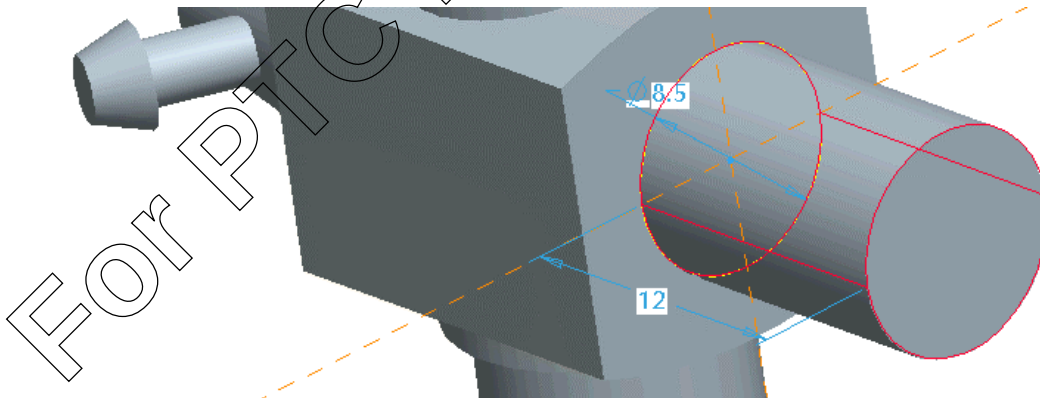


Figure 61: Creating a Protrusion

2. Create a protrusion on the end surface of the previous protrusion. Sketch as shown in the following figure.

Design Intent:

Dimension the angle from the TOP plane. Sketch an axis point at the center of the 5 diameter arc.

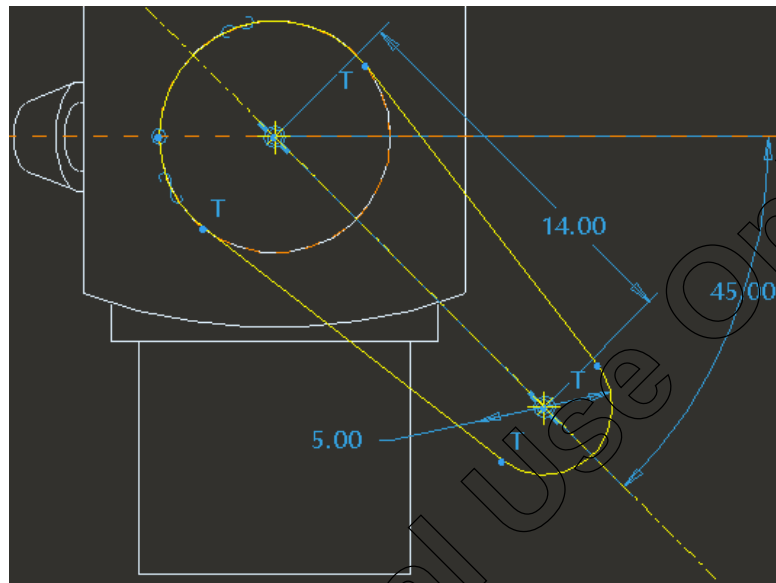


Figure 62: Creating a Protrusion

3. Specify a depth of 2, dimensioned inward toward the model.

Step 4. Create holes and rounds.

1. Create three holes as shown in the following figure.

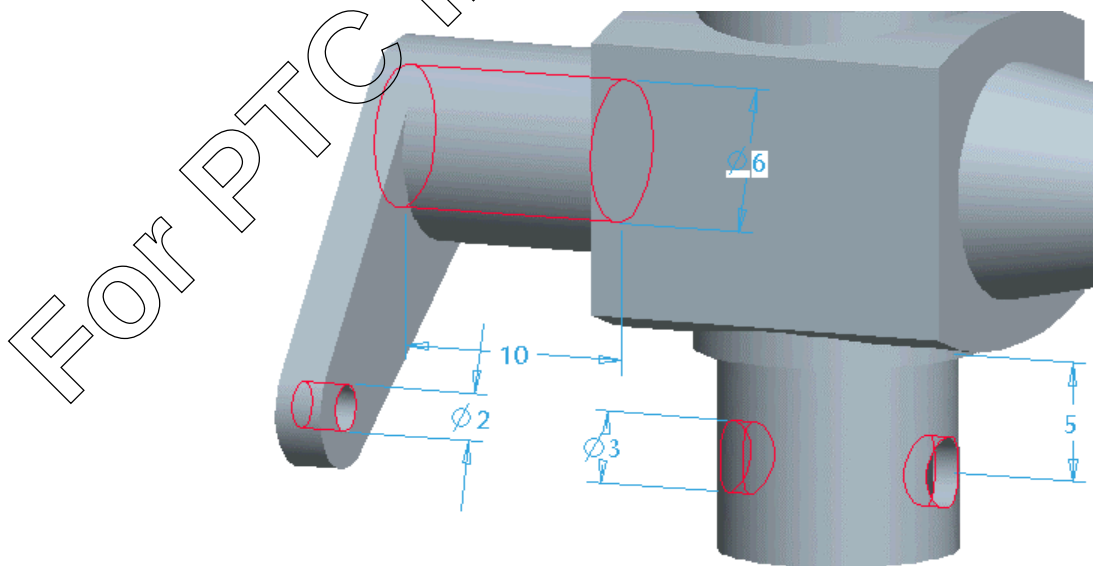


Figure 63: Creating Holes

2. Create two round features as shown in the following figure.

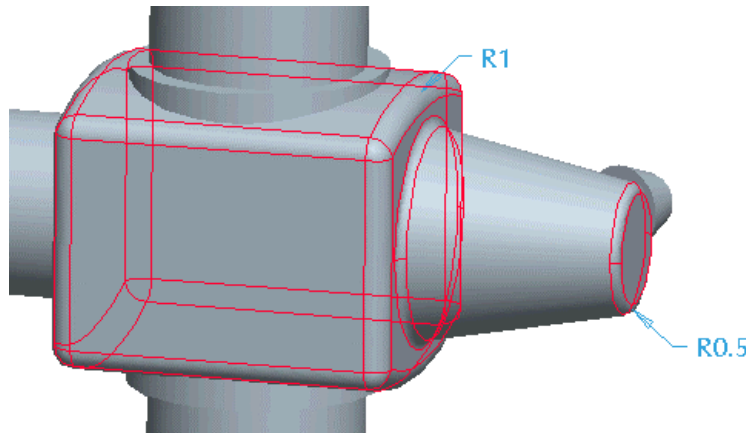


Figure 64: Creating a Round

Step 5. Assemble the CARBURETOR.PRT and its corresponding bolts.

1. Apply the Grey_Med appearance. Save the model and close the window.
2. Open the AC-40.ASM. Activate the ENGINE.ASM.
3. Assemble the CARBURETOR.PRT and its corresponding bolts shown in the following figure.

Design Intent:

The BOLT_3-5 instances should be used.

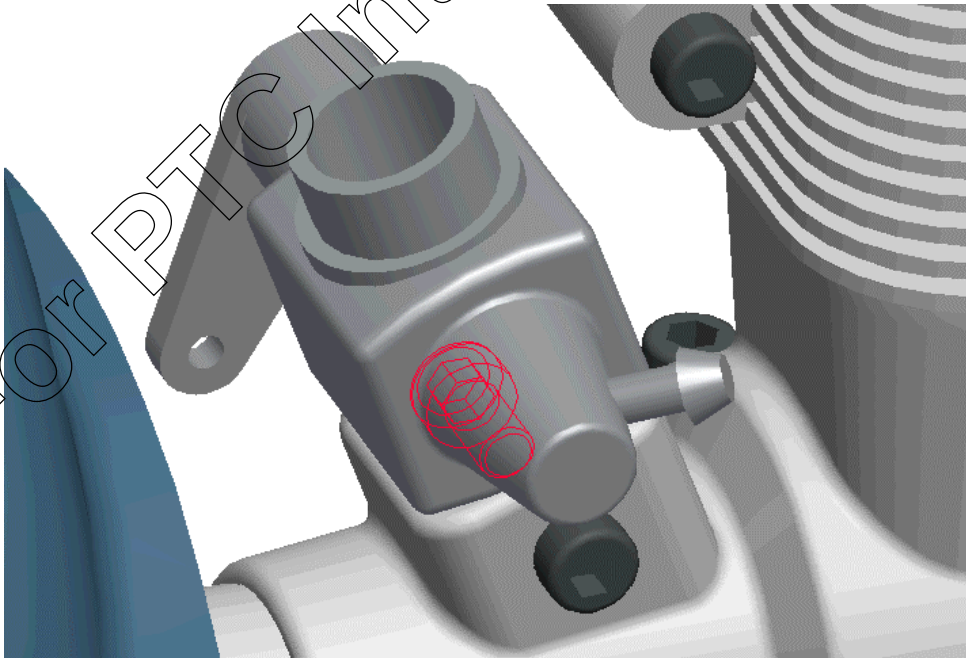


Figure 65: CARBURETOR.PRT Assembled

4. The model tree should appear as in the following figure.

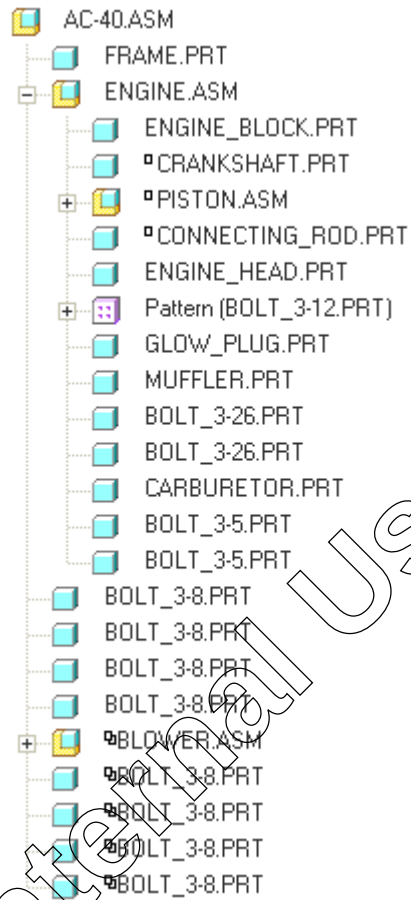


Figure 66: Flange1 Mirrored

5. The assembly will be continued in later exercises. Save the model and close the window.
This completes the exercise.

Objective 7: Creating and Assembling the Engine Cover (Challenge)

Step 1. Create the ENGINE_COVER.PRT.

1. Create a new part named ENGINE_COVER.PRT.
2. Create a series of features as shown in the following figure.

Design Intent:

You may want to create the features in the following order:

- Create a 31 diameter disc-shaped protrusion.
- Create a 28 diameter protrusion.
- Create a 24 diameter hole placed on a plane 2.0 from the end surface.
- Create a protrusion for the tab on an angular datum.
- Pattern the tab.
- Create and reference pattern the hole..

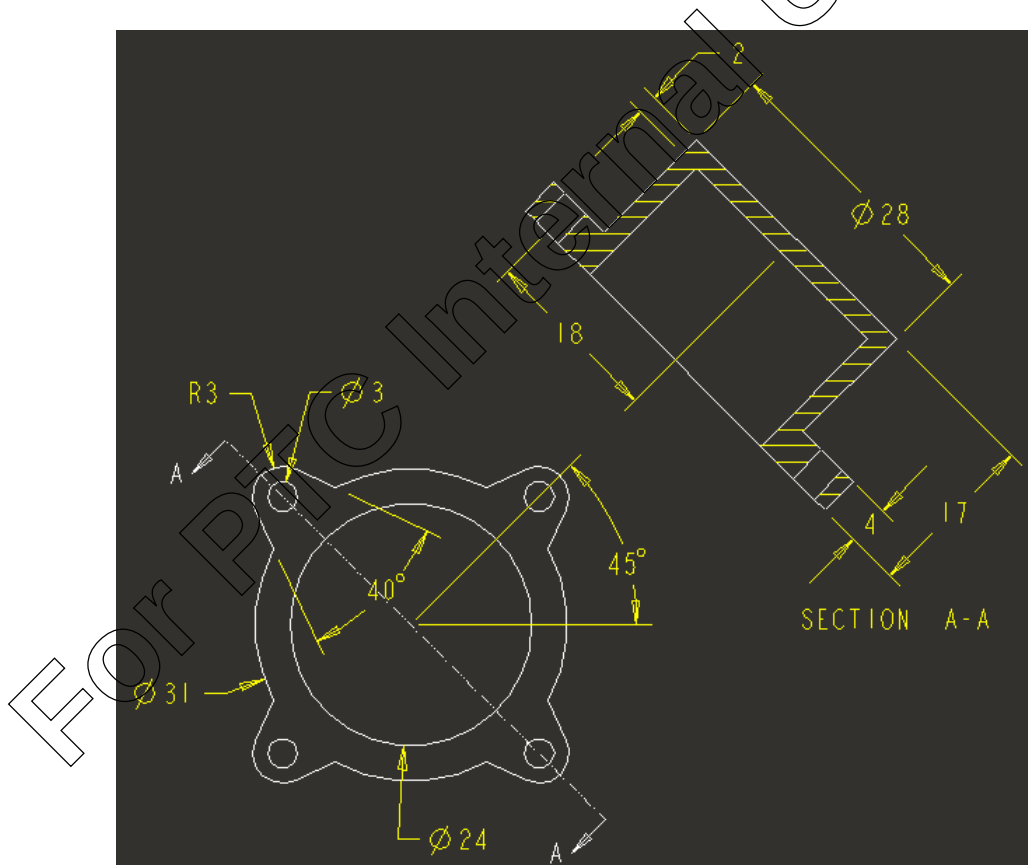


Figure 67: ENGINE_COVER.PRT Specifications

3. Apply the Grey_Med appearance. The model should appear as shown in the following figure.

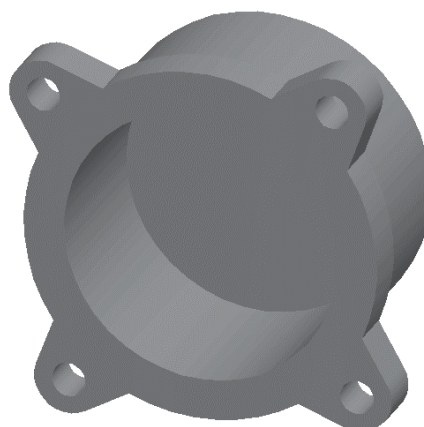


Figure 68: ENGINE_COVER.PRT Model

Step 2. Create ribs to strengthen the part.

1. Create a Rib sketched on the internal datum from the pattern leader of the tabs. Sketch as shown in the following figure.

Design Intent:

Sketching on a previously patterned datum will allow a reference pattern.

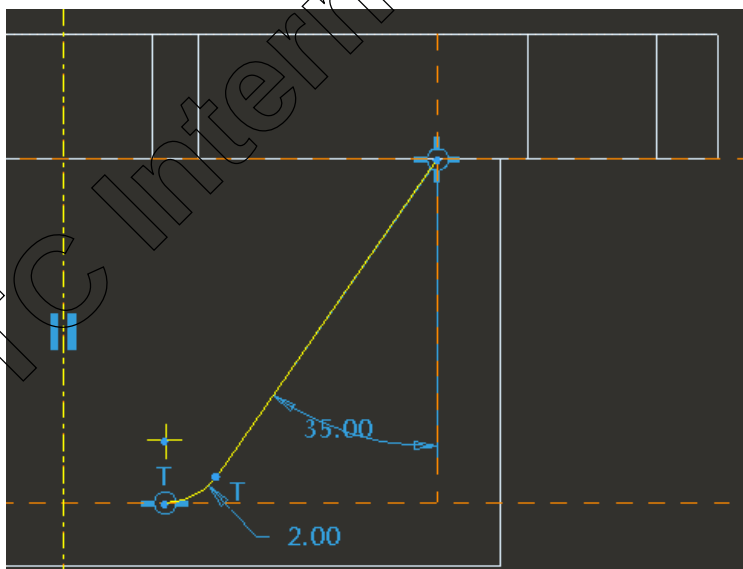


Figure 69: Creating a Sketch

2. Specify a thickness of 1.0. Create a reference pattern of the rib.

Design Intent:

Ribs should be placed adjacent to each tab.



Figure 70: Reference Pattern of Ribs

3. Save the model and close the window.

Step 3. Assemble the ENGINE_COVER.PRT.

1. Open the AC-40.ASM. Activate the ENGINE.ASM.
2. Assemble the ENGINE_COVER.PRT and its corresponding bolts shown in the following figure.

Design Intent:

The BOLT_3-8 instances should be reference patterned in the tab holes.

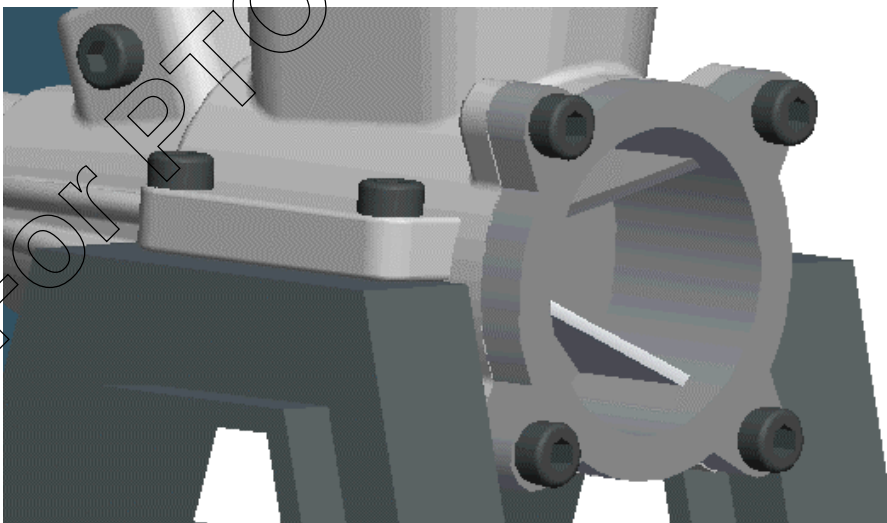


Figure 71: ENGINE_COVER.PRT Assembled

3. The model tree should appear as in the following figure.

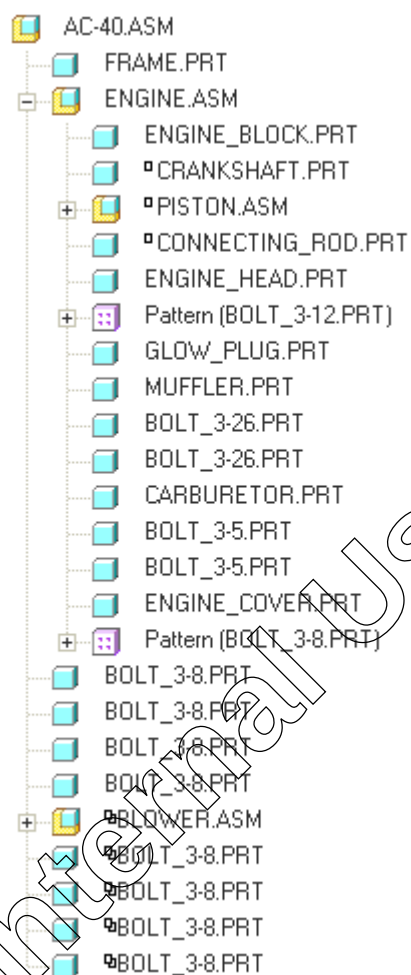


Figure 72: Creating a Hole

4. The assembly will be continued in later exercises. Save the model and close the window.

This completes the exercise.

Objective 8: Creating and Assembling the Flange (Challenge)

Step 1. Create the FLANGE.PRT.

1. Create a new part named FLANGE.PRT.
2. Create a series of features as shown in the following figure.

Design Intent:

You may want to create the features in the following order:

- Create a 78 OD and 58 ID disc-shaped protrusion.
- Create and pattern a radial hole.

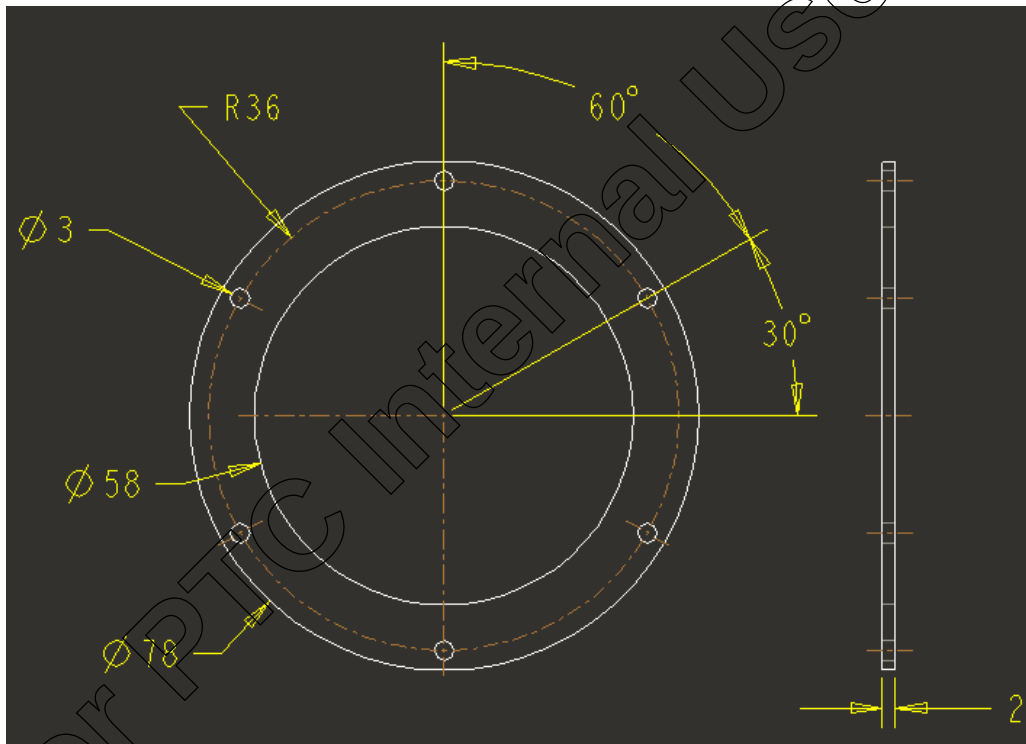


Figure 73: FLANGE.PRT Specifications

3. Apply the Blue_Dark appearance. The model should appear as shown in the following figure.

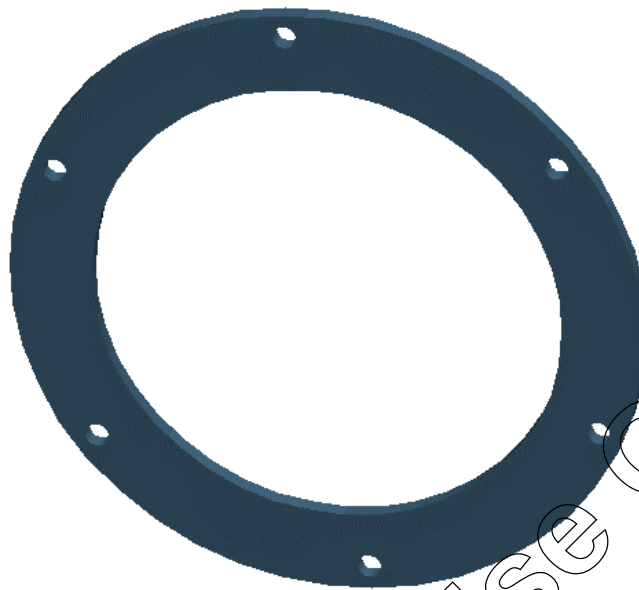


Figure 74: FLANGE.PRT Model

4. Save the model and close the window.

Step 2. Assemble the FLANGE.PRT.

1. Open the AC-40.ASM. Activate the BLOWER.ASM.
2. Assemble the FLANGE.PRT as shown in the following figure.

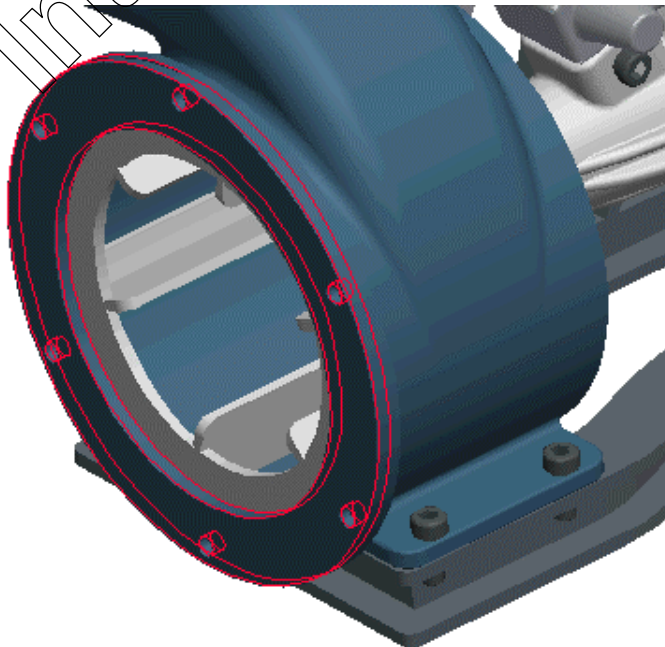


Figure 75: FLANGE.PRT Assembled.

3. Assemble and reference pattern BOLT_3-5 instances as shown in the following figure.

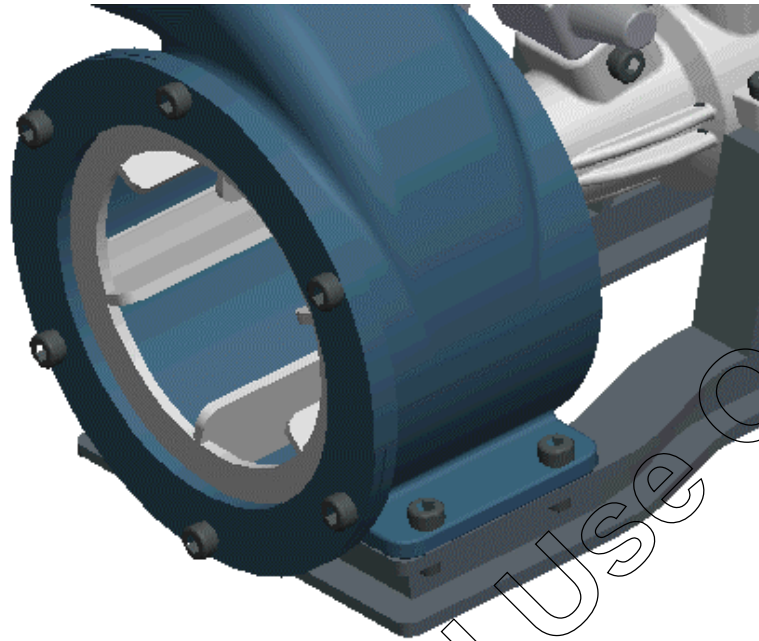


Figure 76: Bolts Patterned

4. The model tree should appear as shown in the following figure.

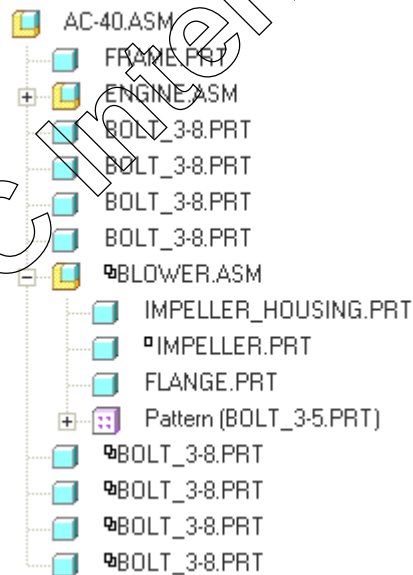


Figure 77: Creating a Protrusion

5. The assembly will be completed in the next exercise. Save the model and close the window.
This completes the exercise.

Objective 9: Creating and Assembling the Nut (Challenge)

Step 1. Create the NUT.PRT.

1. Create a new part named NUT.PRT.
2. Create a revolved protrusion. Sketch on the FRONT plane as shown in the following figure.

Design Intent:

The sketch should be revolved about the vertical reference.

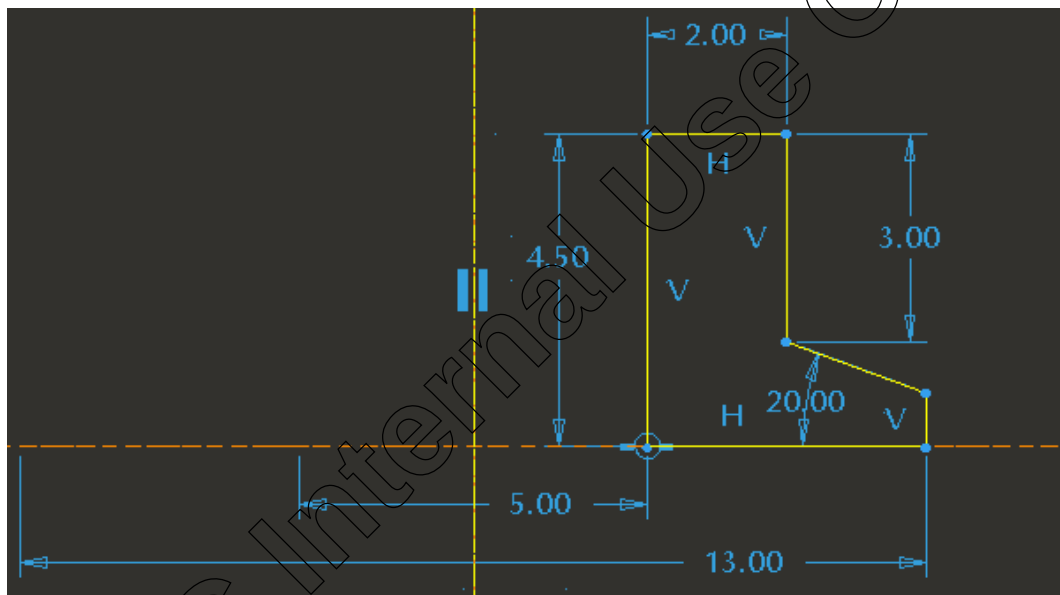


Figure 78: Creating a Sketch

3. The model should appear as shown in the following figure.

Design Intent:

Revolve the sketch 360 degrees.

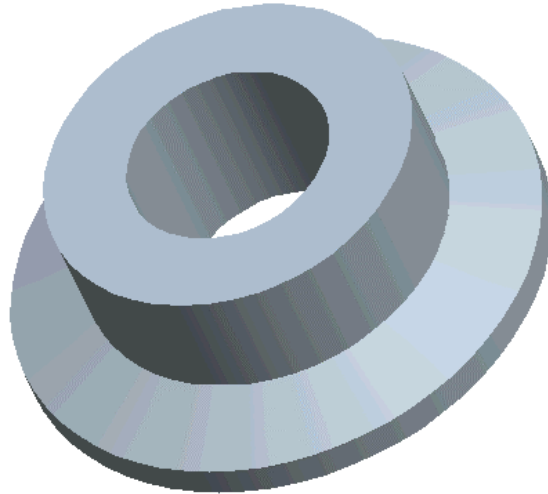


Figure 79: Creating a Protrusion

4. Create a cut with the depth as shown in the following figure.

Design Intent:

Sketch a hex shape by referencing the existing geometry. No sketch dimensions are required. Remove material on the outside of the sketch.

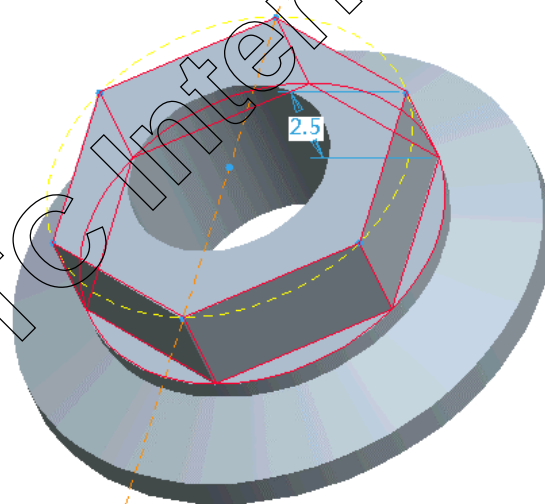


Figure 80: Creating a Cut

5. Apply the Grey_Med appearance. Save the model and close the window.

Step 2. Assemble the NUT.PRT.

1. Open the AC-40.ASM. Activate the top level assembly.
2. Assemble the NUT.PRT as shown in the following figure.

Design Intent:

Align Orient on of the faces on the nut with the flat on the crankshaft, so that the nut will rotate with the mechanism properly.

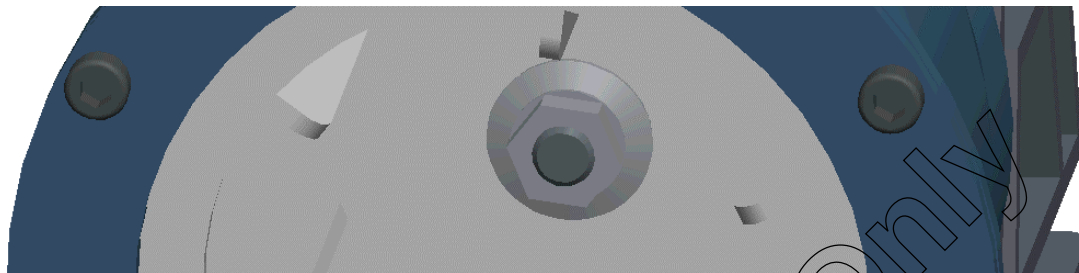


Figure 81: NUT.PRT Assembled.

3. The model tree should appear as shown in the following figure.

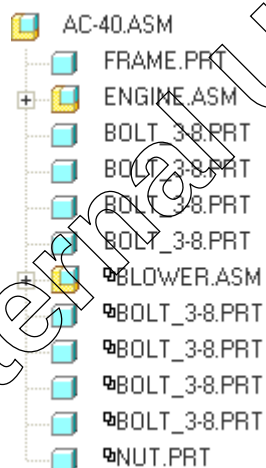


Figure 82: Model Tree

4. The assembly should appear as shown in the following figure.

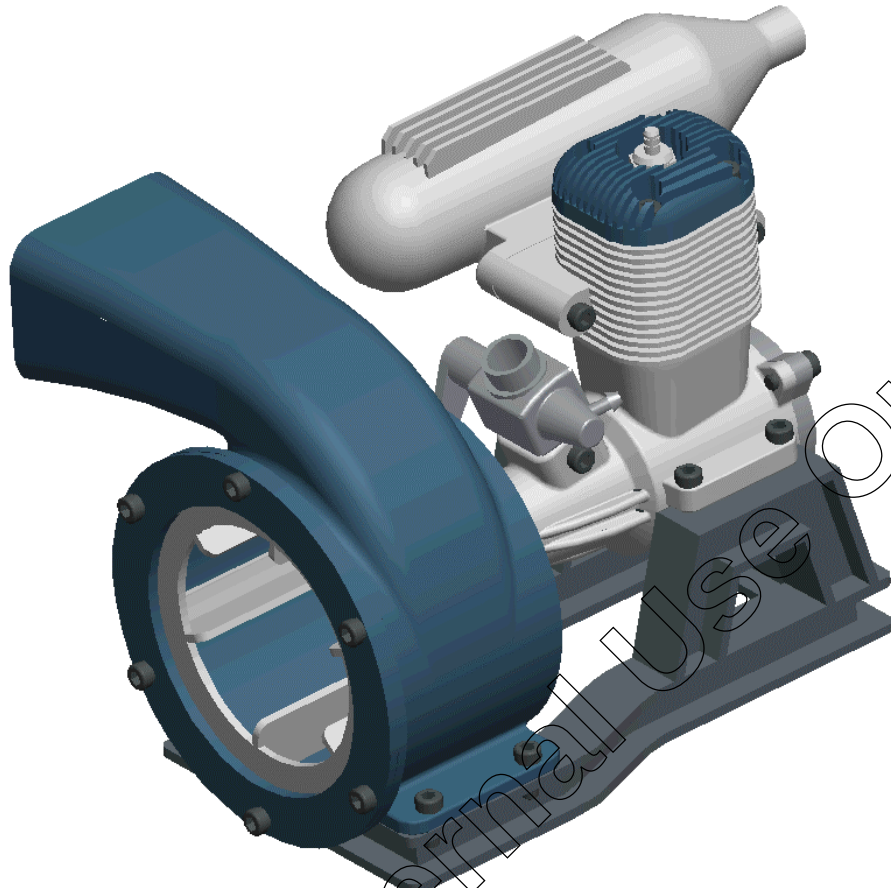


Figure 83: AC-40 Completed

5. The AC_40 assembly is complete. Save the model and close the window.

This completes the exercise.

Summary

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

- Analyze the AC-40 for Interferences.
- Create and Assemble the Bolt.
- Create and Assemble the Engine Head.
- Create and Assemble the Glow Plug.
- Create and Assemble the Muffler.
- Create and Assemble the Carburetor.
- Create and Assemble the Engine Cover.
- Create and Assemble the Flange.
- Create and Assemble the Nut.

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