

STEM ACADEMY

SOLIDWORKS® 2013



Teacher Answer Key

Teacher Name

Class

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Chapter 1

Introduction to SolidWorks

Lesson 1.1 What is SolidWorks:

- Explain what SolidWorks is used for
- Modify a part and understand how the changes are updated throughout the program
- Name the three primary environments and identify what they are used for

Lesson 1.2 User Interface:

- Identify the key elements of the Command Manager
- Name the key features of the User Interface

Lesson 1.3 Navigation Control:

- Utilize the View Orientation button to rotate parts
- Describe how to use the mouse to zoom and rotate views.

Chapter 1

Lesson 1: What is SolidWorks?

Lecture

Answer the following questions during the lecture portion of the video. Please just watch and record your answers without using the software.

What is SolidWorks?

A 3D parametric modeling software

Define mechanical design?




Anything made or operated by machinery

Define parametric and explain why SolidWorks is considered a parametric modeling program?

Parametric means that the shape of the geometry can be changed at any time.

Complete the following table with the correct extension and description

Three Primary File Types

File Type	Extension	Icon	Description
Part	.sldprt		A single part
Assembly	.sldasm		A group of parts that form an assembly
Drawing	.slddrw		A layout that contains views of parts and assemblies

Demonstration

Follow along with the demonstration and increase the length of the Arm part file. Take a screen shot of the assembly to prove that you were able to complete the demonstration correctly. Remember to crop the image so that just the assembly is showing and not the other parts of the screen.



Activity

Change the length of the Arm part file back to 3". Take a screen shot of the assembly to prove that you were able to complete the Activity correctly. Remember to crop the image so that just the assembly is showing and not the other parts of the screen.



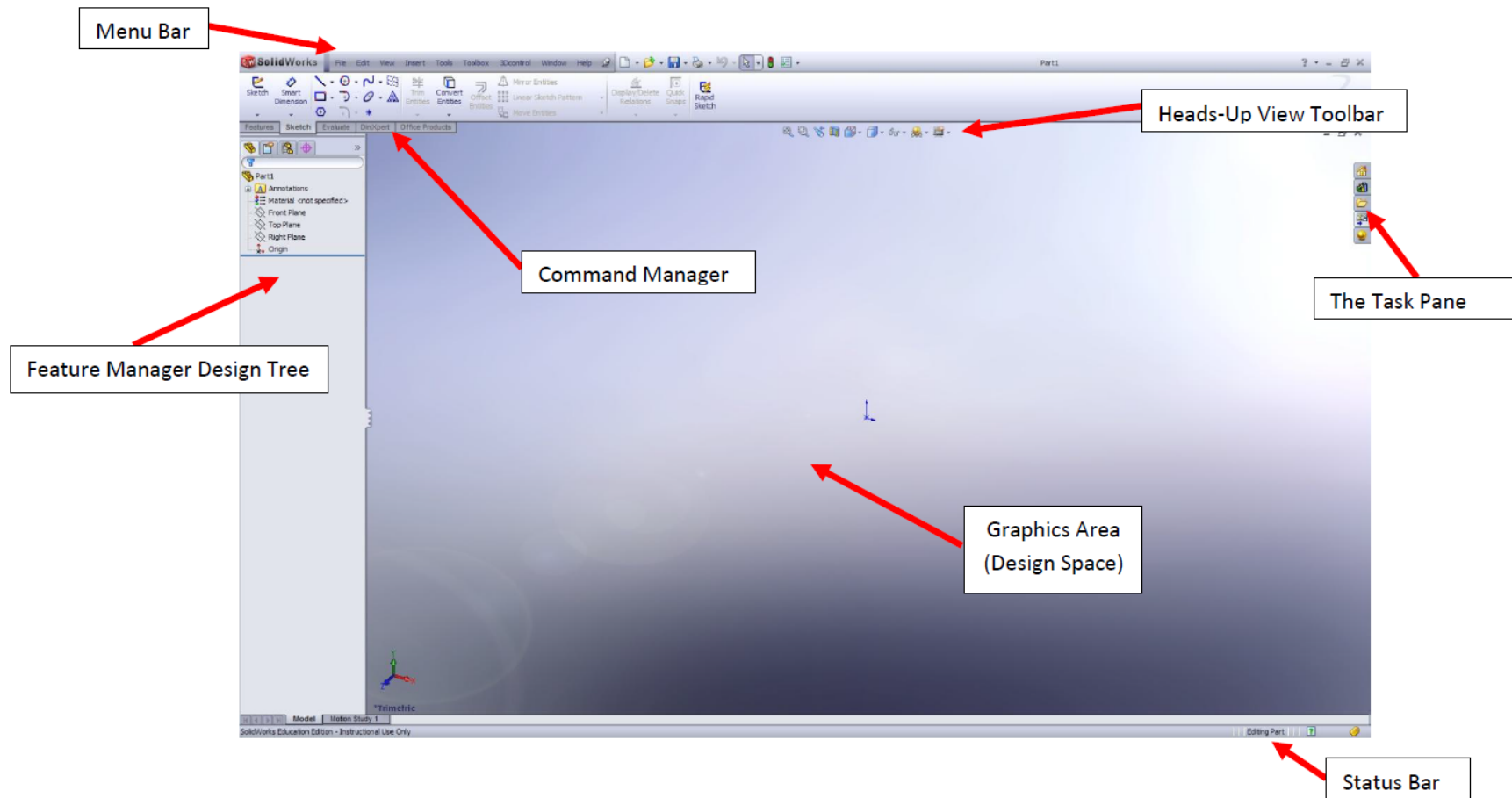
Chapter 1

Lesson 2: User Interface

LECTURE

Record more detailed information about each area of the User Interface on the next page

Basic SolidWorks Layout



User Interface

Describe the main functions of the following items

What types of headings are found in the menu bar?

File, Edit, View, Insert, Tools, Toolbox

Command Manager

This is where the main commands of SolidWorks can be accessed in button form

Feature Manager/Design Tree

This is a listing of all of the changes that have been made to a part.

Heads-Up View Toolbar

This is where view commands can be accessed from without having to go in to menus

Graphics Area

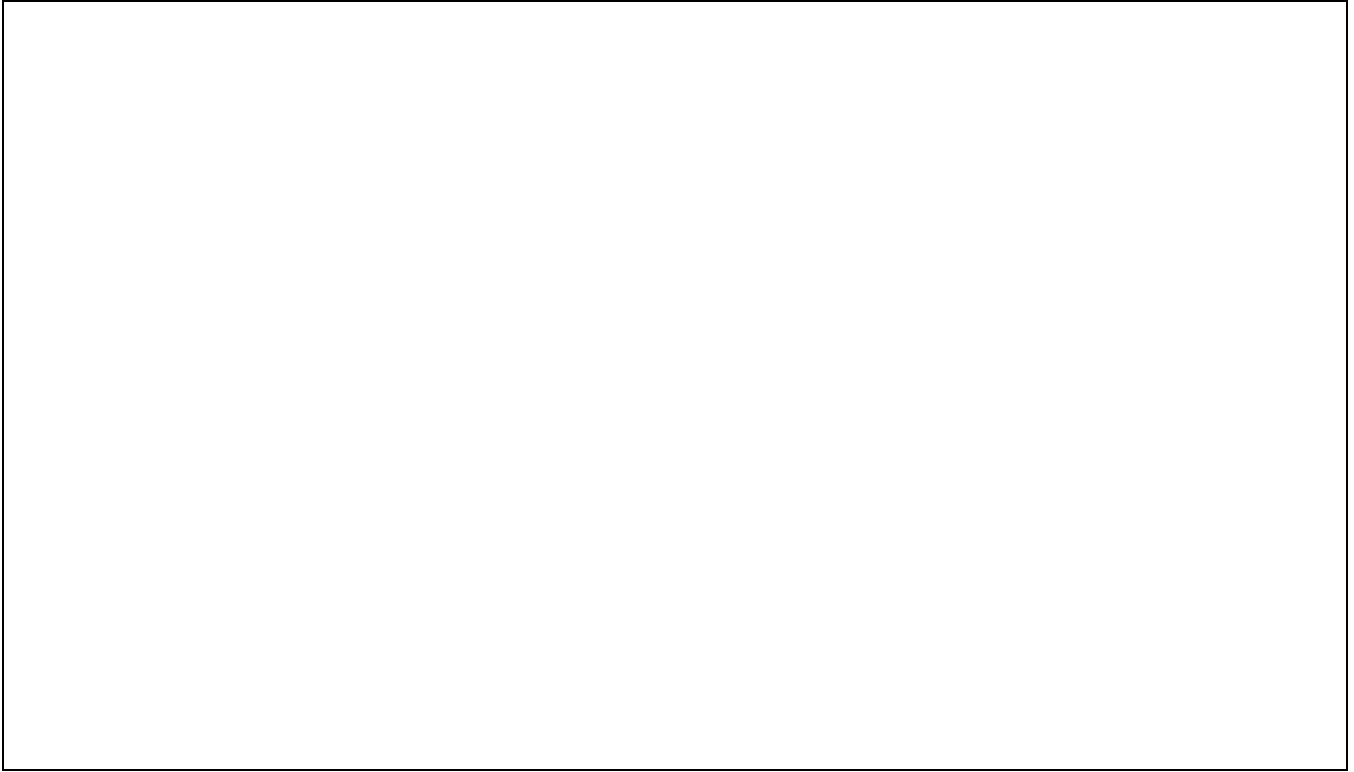
This is the main workspace of SolidWorks where your geometry is displayed.

Status Bar

This shows system and document information such as the units system.

Demonstration

Take a screen shot of the help menu to show that you know how to access it



Activity

No Activity for this lesson

Chapter 1

Lesson 3: Navigation Control

Lecture

What does scrolling the mouse wheel do?

Zoom in and out

What does the “Zoom to Fit” command do?

Zooms so that all available geometry is displayed within the display

What can I press and hold to quickly rotate an object?

The scroll wheel on the mouse

How can you switch between different major views (Front, Right, Top, etc.) in SolidWorks?

By clicking on the view orientation button in the heads up display toolbar

What is a “normal to” view and what is the keyboard shortcut to get there?

Normal to means looking directly at a particular plane. Control-8

Demonstration

Follow along with the demonstration and view the right hand side of the die, make sure you are "normal to" and take a screenshot.



Activity

Use the view orientation tool and show what side of the die you are looking at when you see 3 dots. (Is it the front, right, back, etc.?)

A horizontal rectangular box with a thin black border, intended for a text response to the activity question.

Chapter 2

Navigating through the Environments

Lesson 2.1 Creating a Simple Part:

- Describe the importance of a Closed Profile
- Identify the two categories of constraints used to define a sketch
- List the guidelines for creating a sketch
- Choose the best plane to begin the first sketch
- Identify the different options for creating a Rectangle
- Construct an object using both Join and Cut Extrude options
- Identify a Placed Feature
- Compare a Fillet and a Chamfer

Lesson 2.2 Creating a Simple Assembly:

- Identify the four types of Assembly Constraints
- Create an Assembly using Mate and Flush Constraints
- Define Degrees of Freedom

Lesson 2.3 Creating an Exploded View:

- Identify why an Exploded View is useful
- Demonstrate how to animate an Exploded View

Lesson 2.4 Creating a Simple Drawing:

- Explain the uses of a Drawing File
- Modify drawing orientation, scale and style
- Create a multi-view drawing using part files
- Add a Parts List and Balloons to an Exploded View

Lesson 2.5 Common Mistakes:

- Utilize Repair Sketch to identify a problem
- Describe sketching techniques that should be avoided

Chapter 2

Lesson 1: Creating a Simple Part

Lecture

What is a profile?

A series of lines which form a closed shape.

What are constraints?

Things that lock in the size, position, or location of a particular piece of geometry.

What are the two types of constraints used when creating a sketch?

Smart Dimensions and Relations

What are the guidelines for creating sketches?

1. Keep it simple
2. Start on the origin
3. Add relations as necessary
4. Add dimensions
5. Fully define your sketches

What button do we press once we have completed our profile?

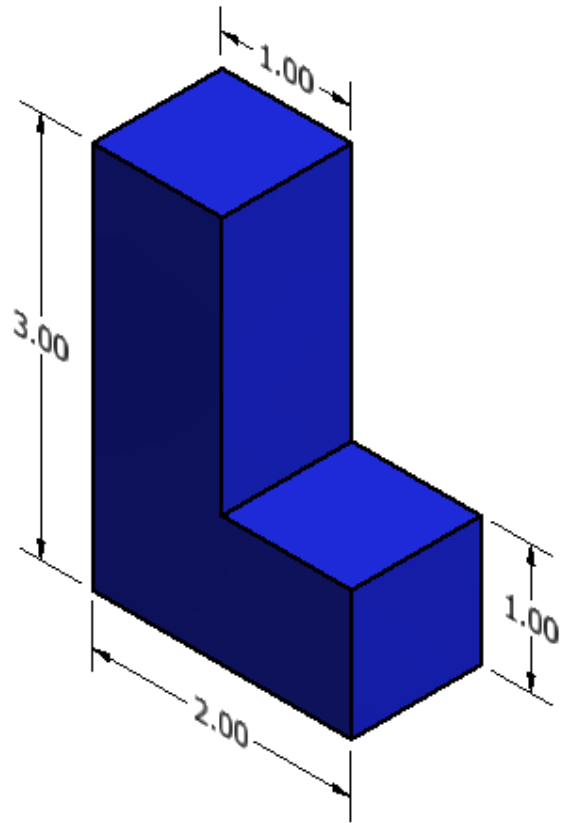
Escape

What does the extrude command do?

Creates a 3 dimensional solid from your 2 dimensional profile

Demonstration

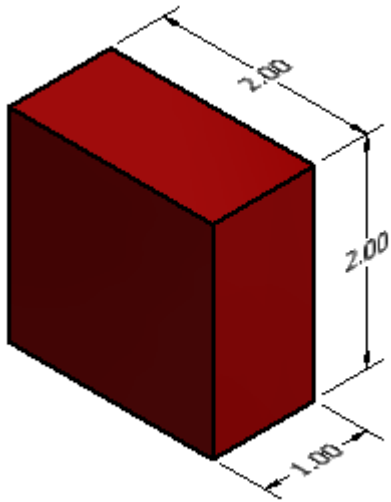
Follow along with the video demonstration and complete the following shape. Save as "L Block" in the Mosaic Puzzle Folder



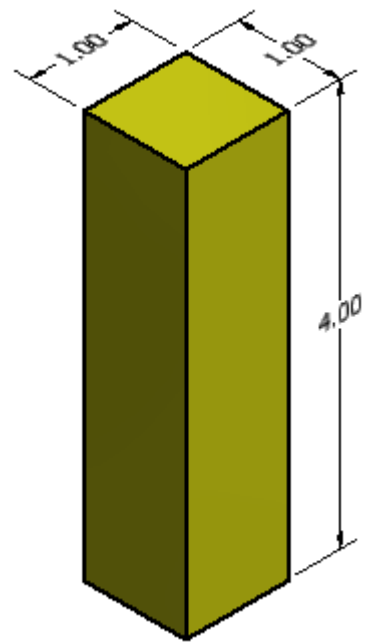
Activity

Create the remaining shapes on your own. Begin by making a new Part file and make sure to fully constrain each sketch. Save each part in the proper folder (Mosaic Puzzle) using the names listed.

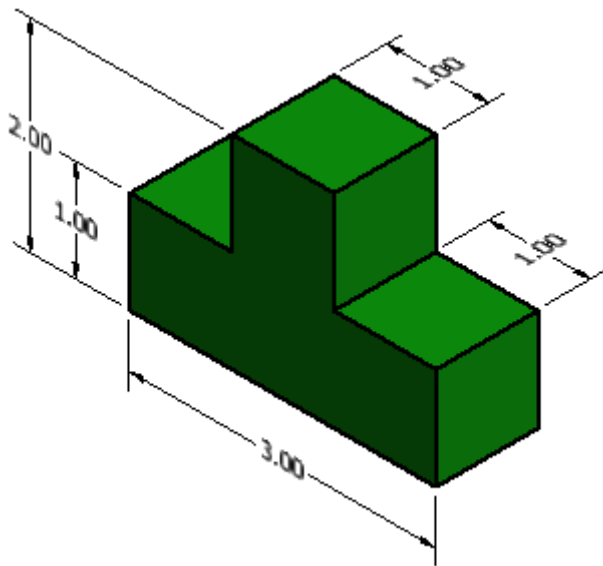
Square Block



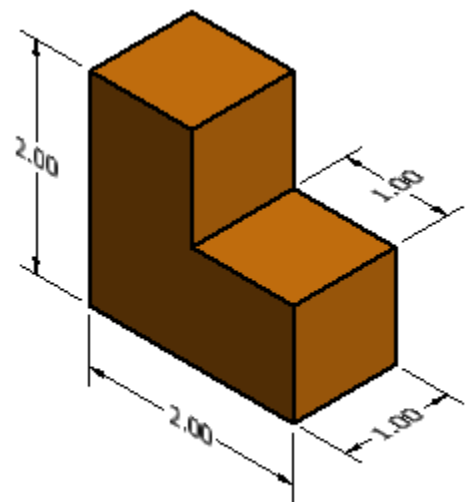
Rectangle Block



T Block



L Block Small



Chapter 2

Lesson 1a: Creating a Simple Part

Lecture

Which plane should be selected if you are going to build the first sketch from the top view?

The top plane

What are the five options for creating a rectangle?

Corner, Center, 3 Point Corner, 3 Point Center, and Parallelogram

How do you change the direction of the extrusion?

By clicking on the black and blue arrows in the dialog box or by clicking and dragging on the arrow in the graphics area.

How should a model be viewed before an extrusion is made?

The sketch should be viewed normal to, but while you are setting up the extrusion it is often helpful to look at it at an angle to visualize the depth of the extrusion.

What is the difference between a fillet and a chamfer?

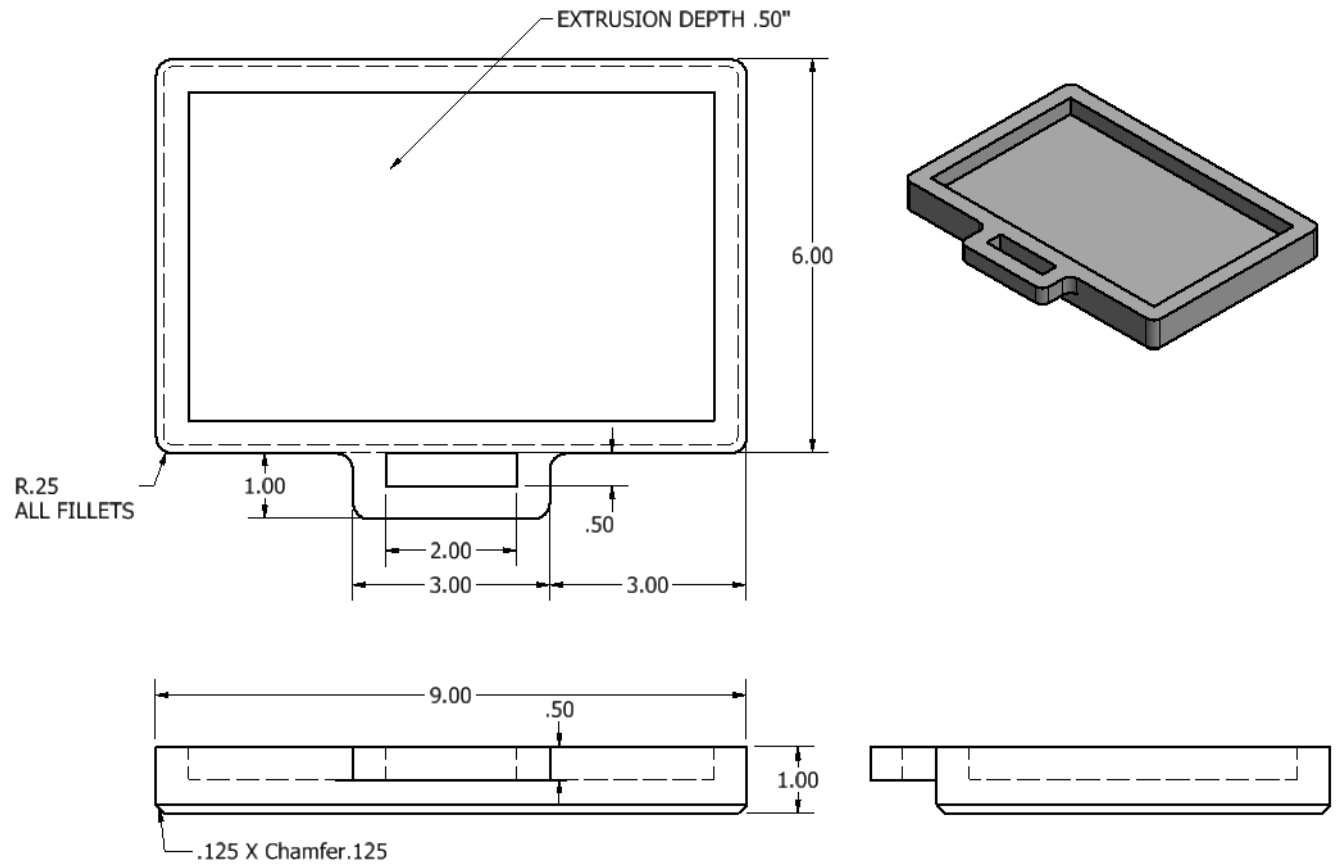
A fillet is a rounded edge and chamfer is a straight cut off of the edge

What are three ways that we can define a chamfer?

Angle, Distance-Distance, and Vertex

Demonstration

Follow along with the video demonstration and complete the following part. Click Save when you are done and name the part "Mosaic Puzzle Case". Be sure to save the part into the Mosaic Puzzle folder.



Activity

No Activity for this lesson. Please continue on to 2.2 Creating Simple Assemblies after completing the Demonstration.

Chapter 2

Lesson 2: Creating a Simple Assembly

Lecture

The first part placed into an assembly is considered _____

Fixed

What are the five main types of mates?

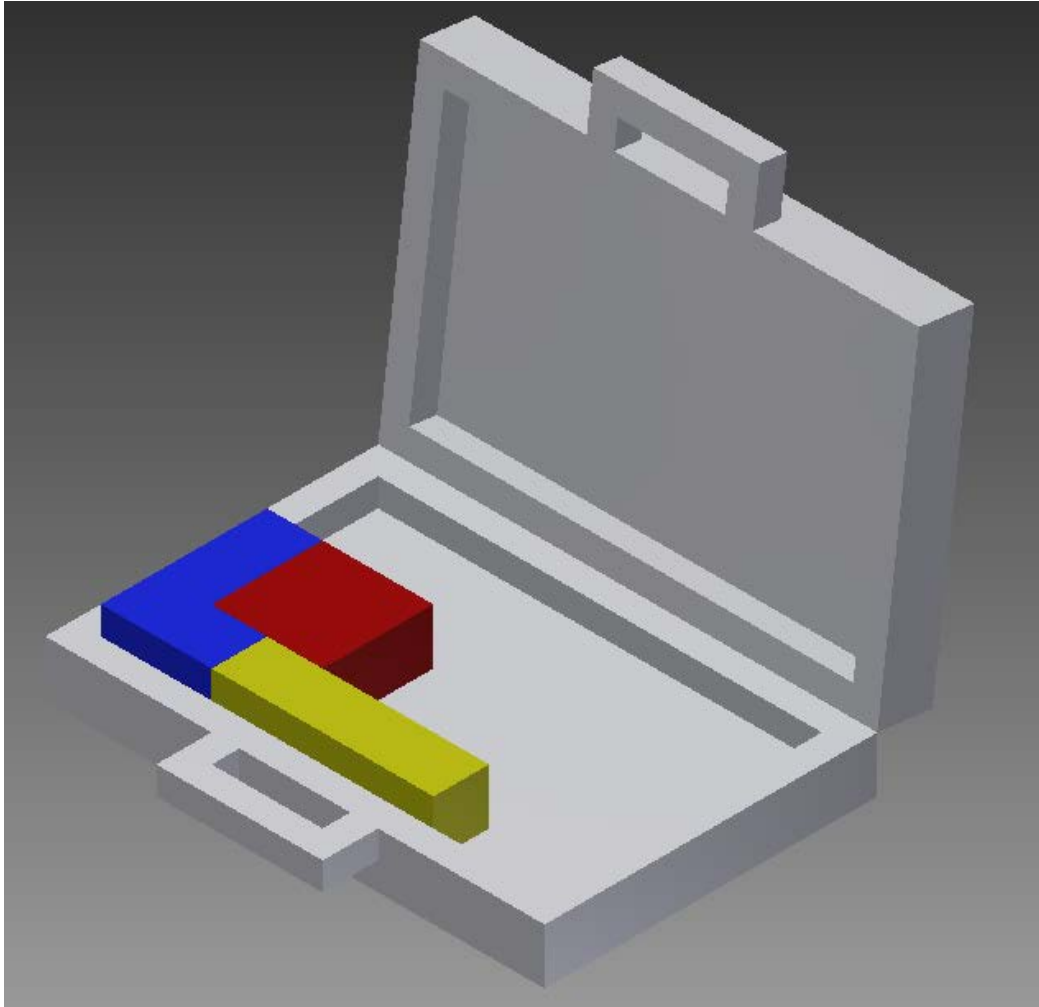
Coincident, Parallel, Perpendicular, Tangent, and Concentric

How many degrees of freedom are in an unconstrained part in an assembly?

6 (3 axial and 3 linear)

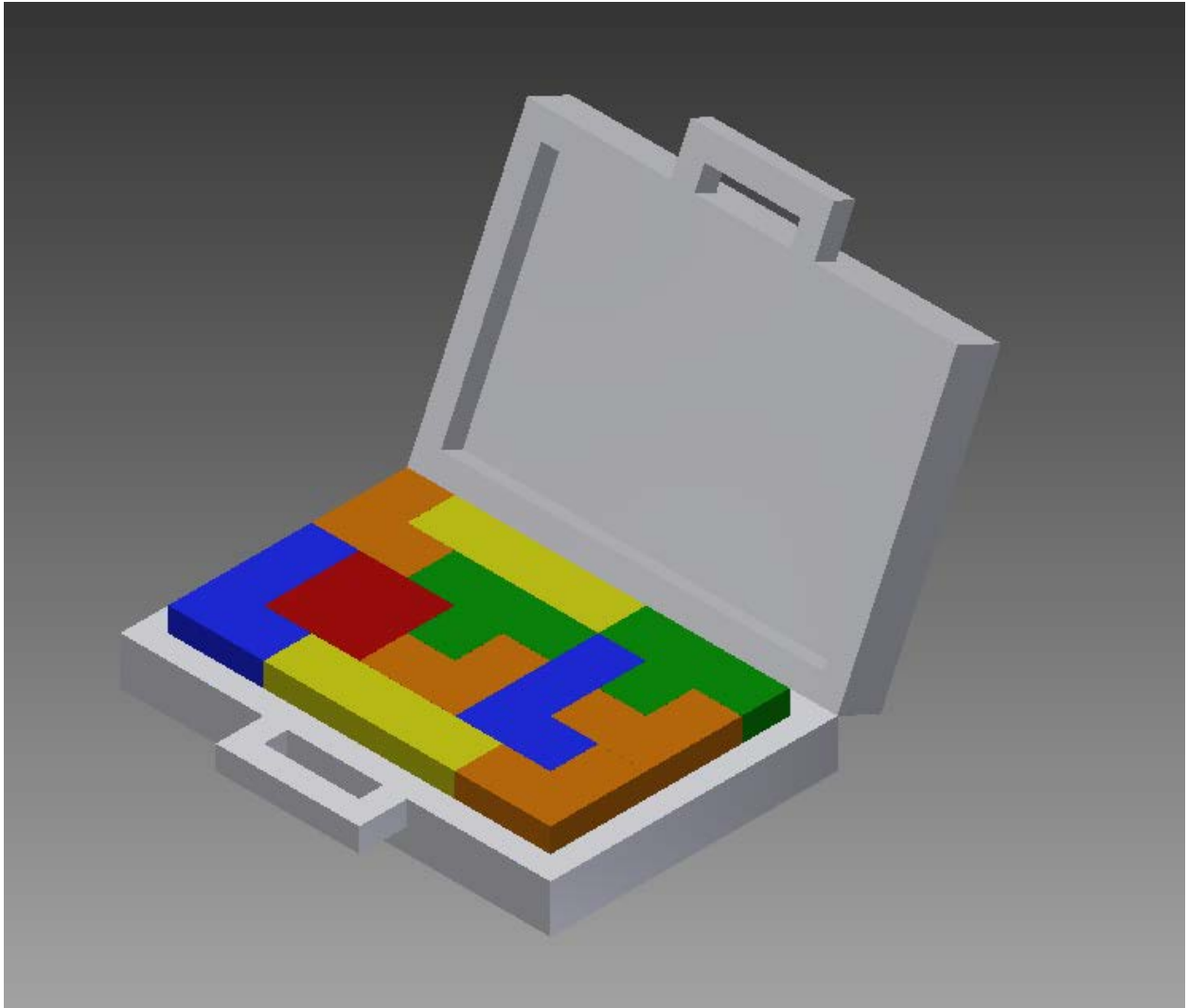
Demonstration

Follow along with the video demonstration and complete the following assembly.



Activity

Finish creating the assembly and Save As Mosaic Puzzle Assembly in the Mosaic Folder.



Challenge:

Create an assembly but try and find a different combination that still fits in the base without any space between parts. If you can't figure out a new combination, try creating new puzzle piece that would allow you to solve the puzzle.

Chapter 2

Lesson 3: Creating an Exploded View

Lecture

What is an exploded view?

An exploded view is a way of examining an arrangement of parts so that you can see hidden components or features that would not be outwardly apparent otherwise.

Why would we want to show an exploded view?

To show how components fit together

What is the only file type that can be used to create exploded views?

An assembly

What are the rules for creating an exploded view?

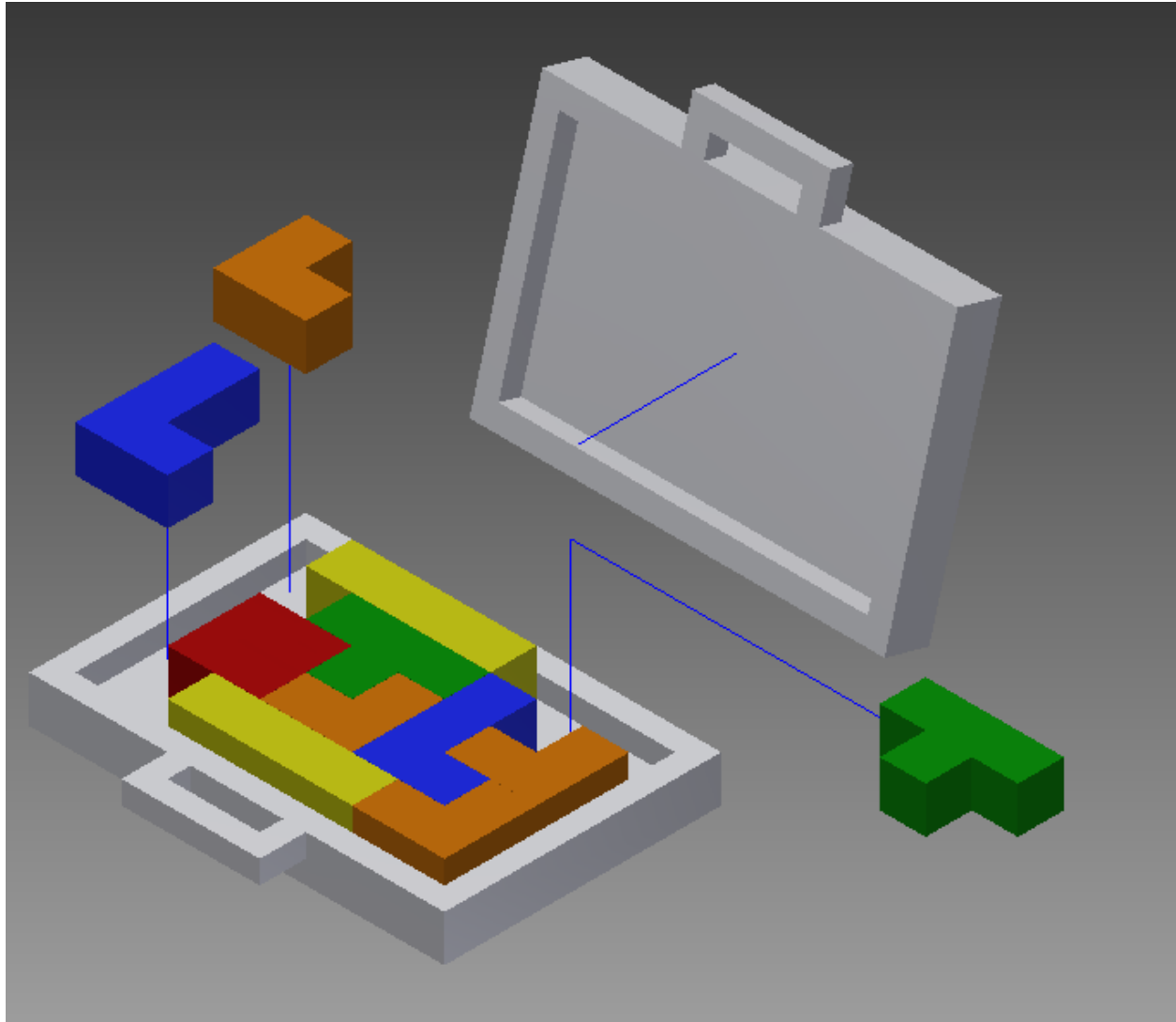
- 1) Have all of the pieces equally spaced, 2) You want to explode and collapse the parts in the order they would be assembled or disassembled as much as possible

Can exploded views be animated?

Yes, and saved as video files

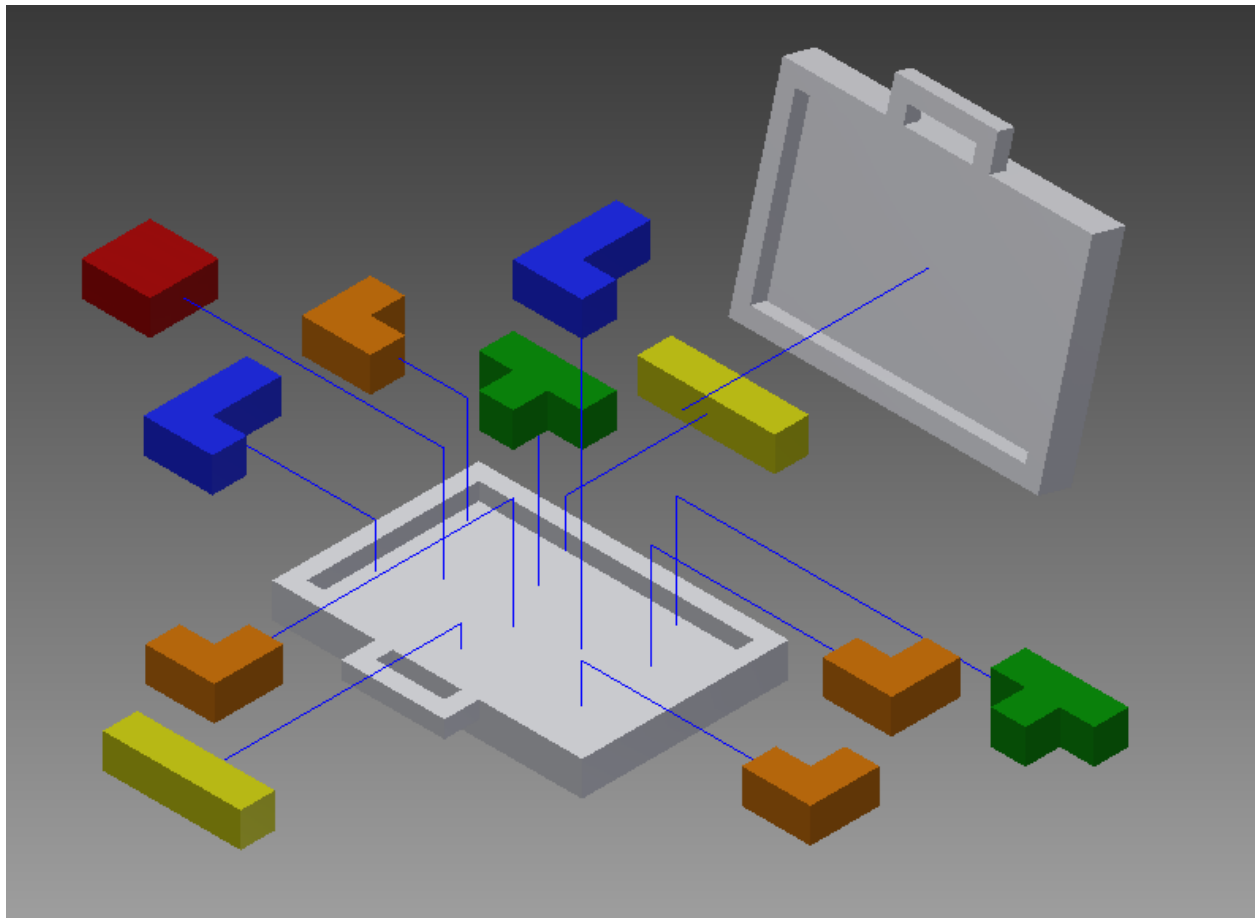
Demonstration

Follow along with the video demonstration and complete the following exploded view. Save the file as “Mosaic Exploded” in the Mosaic Puzzle Folder when it is completed



Activity

Finish creating the exploded view by tweaking the remaining parts. The final exploded view should resemble the picture below, but there may be slight differences depending on your approach. Save the file as “Mosaic Exploded” in the Mosaic Puzzle Folder when it is completed.



Challenge

Create an animated video of the parts moving back into the case. Try and make sure none of the blocks go through each other as they move.

Chapter 2

Lesson 4: Creating a Simple Drawing

Lecture

What is the easiest way to think of a Drawing file?

As a sheet of paper

Why do we need drawing sheets?

We often need to present our 3D ideas on 2D media or to people without access to 3D tools.

What are the dimensions of an “A” size sheet?

8.5”x11”

What is the first view called that is placed on a drawing?

The base view

What views does a multiview drawing typically show?

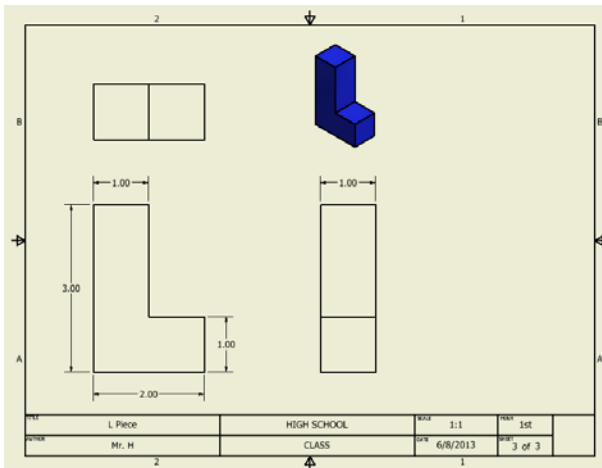
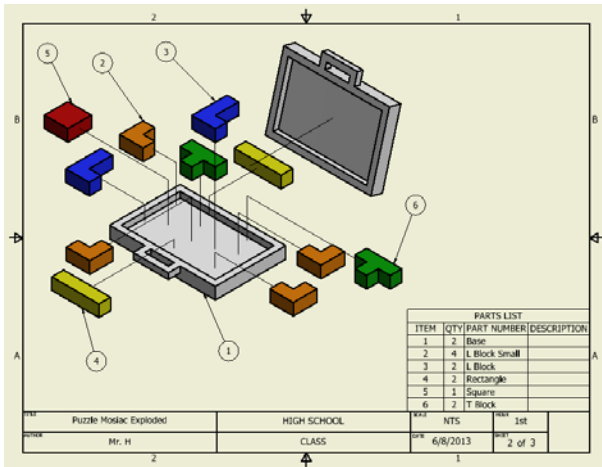
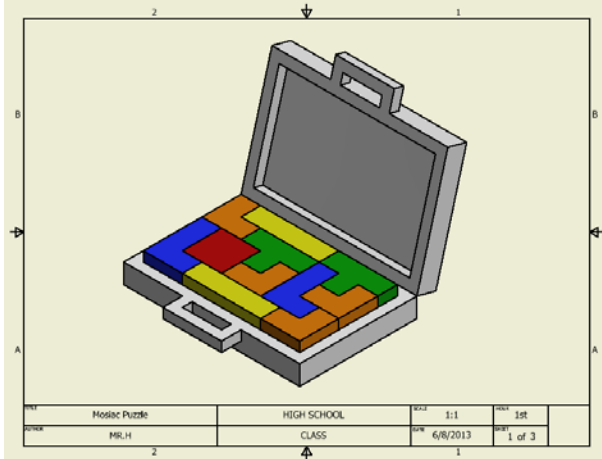
The top, front, and right

What type of files can be placed on a sheet?

Parts and assemblies

Demonstration

Follow along with the video demonstration and complete the following drawings.



Activity

Finish the remaining drawing sheets. Create a new sheet for each piece. Make a front, top, right and Shaded Isometric similar to the "L Piece" sheet that we created during the demonstration. Basically all of the remaining sheets should have the same format, but laying out the different parts. Make sure to talk with your teacher about proper dimensioning standards and fully dimension the remaining parts. Ask your teacher for specific instructions on how to turn in the completed drawings.

Challenge

Create a multiview similar to the pieces you just completed, but using the Case part. Use correct layout and dimensioning techniques to properly annotate the part.

Chapter 2

Lesson 5: Common Mistakes

Lecture

How do you bring back a tool bar if you have lost it?

Click and drag on it and place it back on one of the docking icons

How can you tell if a profile will not extrude as a solid?

It will try to extrude out as a “thin feature” or rim.

What is one easy way to diagnose a problem with a sketch?

Use the repair sketch command

What should you do if the Repair Sketch command will not resolve the problem?

Manually examine the corner to make sure they are coincident

What are two techniques you should try to avoid doing while creating a sketch?

- 1) Drawing two separate line segments where only one should be.
- 2) Drawing lines over other lines.

What is an unconsumed sketch?

A sketch which has not been used by a feature

How do you get back in to a sketch?

Right click on it in the feature manager and select “edit sketch.”

Demonstration

No Demonstration

Activity

No Activity

Try to avoid making these common mistakes in the future and use this knowledge fix them if you do.

Chapter 3

Creating Complex Sketches and Constraints

Lesson 3.1 Geometric Constraints:

- Define each of the available geometric constraints
- Utilize geometric constraints and dimensions to fully constrain a sketch
- Demonstrate how to show and delete geometric constraints

Lesson 3.2 Creating Complex Sketches:

- Define a feature in a part file
- Utilize Arc, Circle, Point and Polygon tools in a sketch
- Demonstrate the use of Trim, Extend, and Offset
- Utilize a construction line in a sketch
- Create text and use it for an emboss

Lesson 3.3 Mass Properties

- Describe the steps necessary to change the materials of a part
- Calculate properties such as Density, Volume, Weight, and Surface Area
- Changing Visual Style and Appearance

Chapter 3

Lesson 1: Geometric Relations

Lecture

What are relations and why do we need them?











Relations geometrically confine sketches, we need them because numerical constraints do not tell us the physical position of the geometric entity.


How do you show or hide relations in SolidWorks?

Go in to the view tool menu and select "Relations"

What is an implied relation?

A relation that the computer automatically assumes you want based upon how the geometry is drawn.

Symbol	Name	Function
	Horizontal	Locks the entities in a horizontal position
	Vertical	Locks the entities in a vertical position
	Collinear	Places the entities on the same line
	Perpendicular	Places the entities at a right angle to one another
	Parallel	Places the lines so that they never touch, even if extended
	Equal	Equalizes the dimension of the entities
	Lock	Locks the entities in place (only to be used as a last resort)
	Tangent	Places a line and circle so that they touch at only one point
	Concentric	Places the entities so that they share the same center
	Midpoint	Places the end point of one line coincident with the midpoint of another.

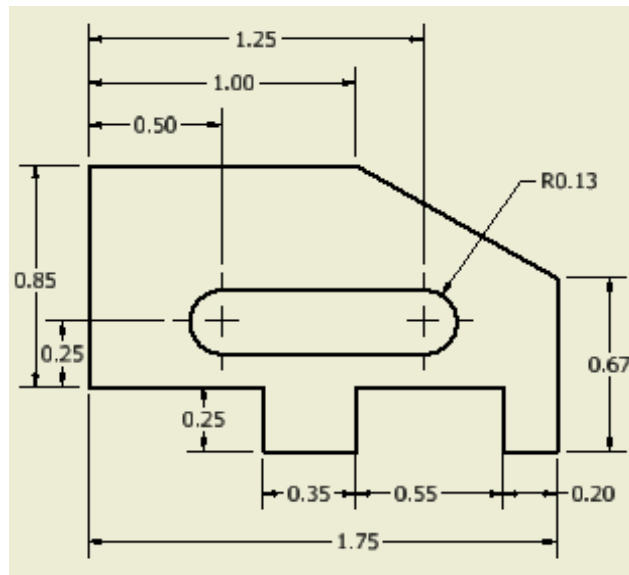
	Coincident	Places two entities so that they are in contact with one another.
---	------------	---

How do we delete a geometric constraint?

Click on the green relation icon in the graphics area and press delete (or right click and select delete).

Demonstration

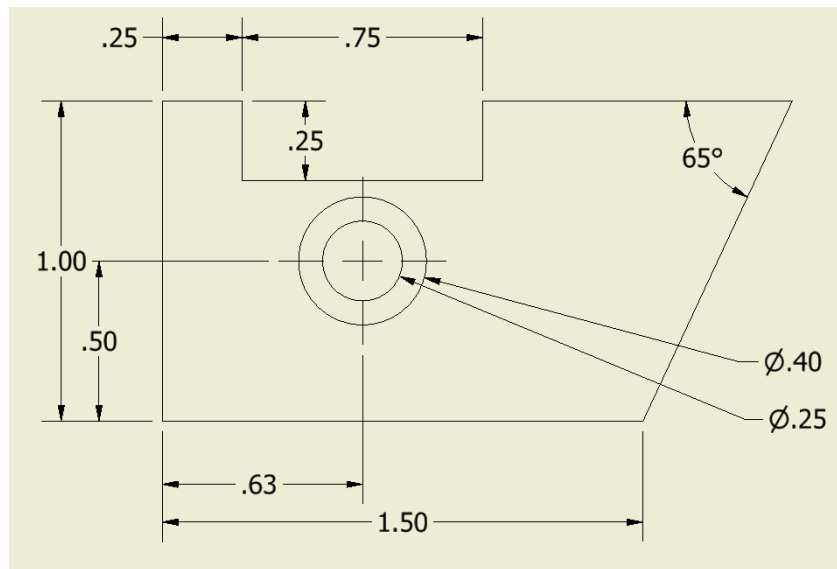
Open the part named "Geometric Relations Bad Sketch Demo" Use Geometric Relations and Dimension Constraints to fix this poor sketch until it is fully constrained. Place a screen shot of the finished sketch into the box below.



Place screen shot here

Activity

Open the part named "Geometric Relations Bad Sketch Activity." Use the drawing below to add Geometric and Dimension Constraints to fix this poor sketch until it is fully constrained. Place a screen shot of the finished sketch in the box below.



Place screen shot here

Chapter 3

Lesson 2: Creating Complex Sketches

Lecture

An extrusion and a cut are both examples of a _____.

Feature

A circle dimension defaults to _____ and an arc dimension defaults to _____.

Diameter, Radius

Typically a point is used to locate the center of a _____.

Circle (or arc)

What is a fillet?

A rounded edge

What is the difference between an inscribed and circumscribed polygon?

An inscribed polygon will be inside a circle and a circumscribed polygon will be outside of a circle

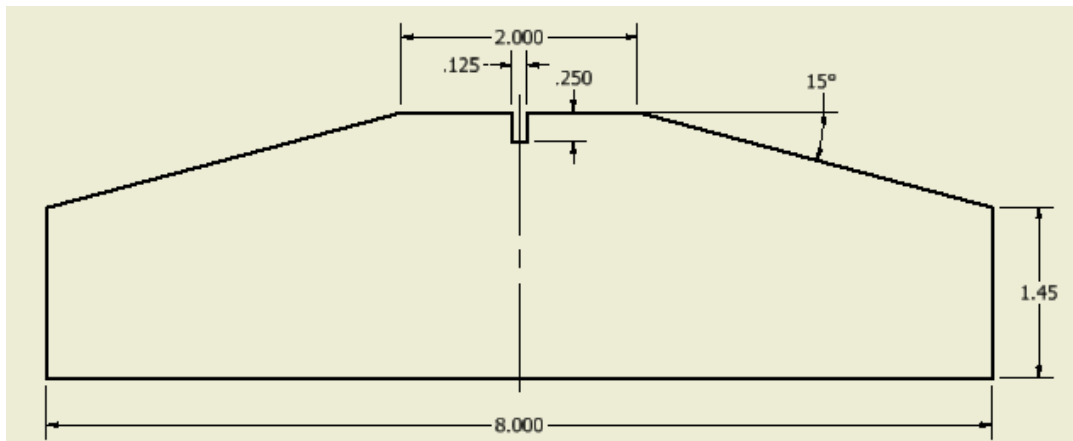
Why are centerlines useful?

The allow the creation of reference geometry that SolidWorks will not try and use for features.

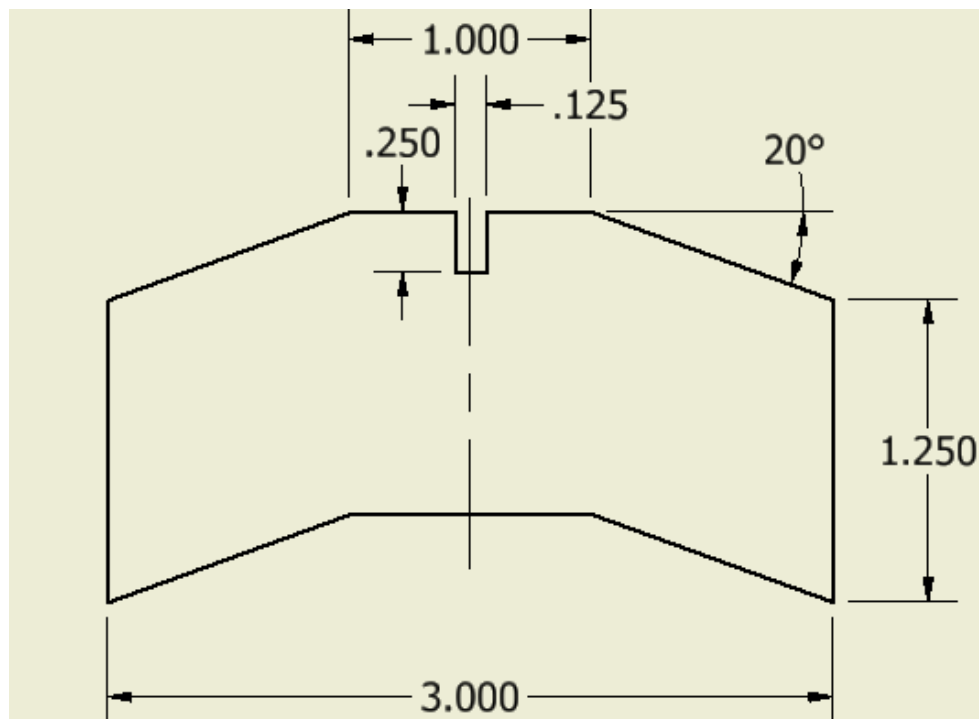
Activity

Create the Front Wing and Rear Wing using the dimensions below and save the parts with the names given in the Balsa Wood Plane folder. Make sure the sketches are fully constrained and the thickness to extrude is .125"

Balsa Wood Front Wing



Balsa Wood Rear Wing



Balsa Wood Sheet Set

Create a front view of each part on your custom title block as shown below. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Balsa Wood Sheet Set into the 3.2 Balsa Wood folder. Be sure to label each view and put the correct scale.

The drawing shows three parts of a balsa wood plane. The Rear Wing (SCALE 1 : 1) is a trapezoidal shape with a notch at the top center. The Front Wing (SCALE 1 / 2) is a similar trapezoidal shape. The Balsa Wood Plane Body (SCALE 1 : 1) is a long, narrow fuselage with a curved upper surface, a circular hole, a rectangular slot, and a tail section with a vertical fin and a hexagonal hole. The drawing is enclosed in a frame with dimensions 2 and 1 at the top and bottom, and A and B on the sides. A title block is located at the bottom of the drawing.

TITLE	BALSA WOOD PLANE PARTS	HIGH SCHOOL	SCALE	VARIES	HOUR
AUTHOR	MR. H	CLASS	DATE	8/26/2013	SHEET
					2 of 4

Challenge

Assemble all of the components of the Balsa Wood Plane.

Chapter 3

Lesson 3: Mass Properties

Lecture

What is the default material called in SolidWorks?

Default grey plastic

How do you change the materials of a part?

Right click on the materials heading in the feature manager and select a new material

What is the difference between changing the material and changing the color?

The material has physical properties attached to it while an appearance is just how it looks

How do you access the Mass Properties?

Click on the evaluate tab then click on the mass properties button.

Give two examples of properties that are determined by the material and two things that are not.

Density and Mass

Volume and Surface Area

Which tab can be used to change the visual style, turn on shadows or view on object in perspective view?

The view settings button in the heads up display toolbar

Demonstration

Follow the instructions given and close the part without saving changes when completed.

Activity

Create a part with the dimensions is 2" x 2" x 27". Change the material type to Gold and fill in the blanks to the Physical Properties below.

Mass _____ Surface Area _____ Volume _____

Challenge

How much would a 2" x 2" x 27" bar of gold be worth? (Research current gold prices)

What would the mass and value of a part be if it was made of copper?

Chapter 4

Creating Parts with Multiple Features

Lesson 4.1 Creating Complex Parts:

- List the steps used to create a strategy for modeling a part

Lesson 4.2 Creating Complex Parts with Curves:

- Determine the correct front view
- Describe a placed feature
- Demonstrate various ways to create a hole

Lesson 4.3 Reference Planes:

- Create a variety of Reference planes
- Utilize project geometry

Lesson 4.4 Pattern Tools:

- Define Work Features
- Utilize Linear Pattern, Circular Pattern, and Mirror

Chapter 4

Lesson 1: Creating Complex Parts

Lecture

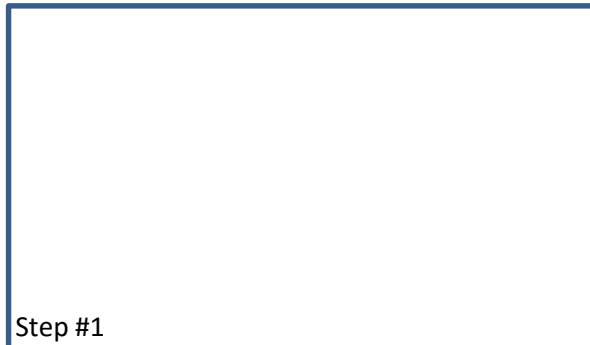
What is the first extrusion in a part called?

The base feature.

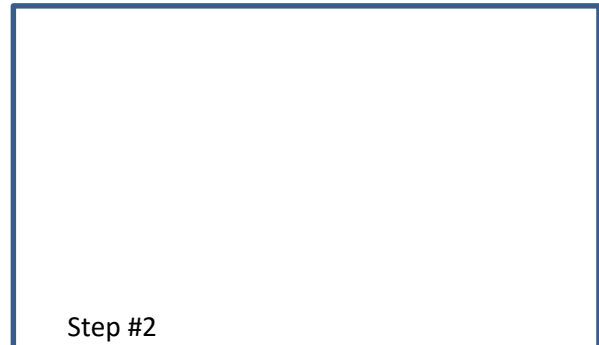
List three questions you should ask yourself before beginning a part?

- 1) What plane should I start my first sketch on?
- 2) What should the base feature be?
- 3) How should the part be extruded?

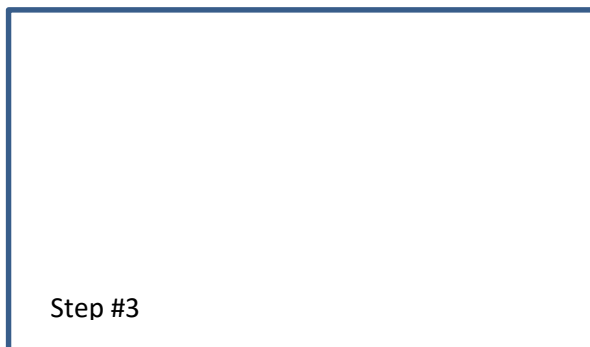
In the boxes below sketch the strategy that will be used to create the model



Step #1



Step #2

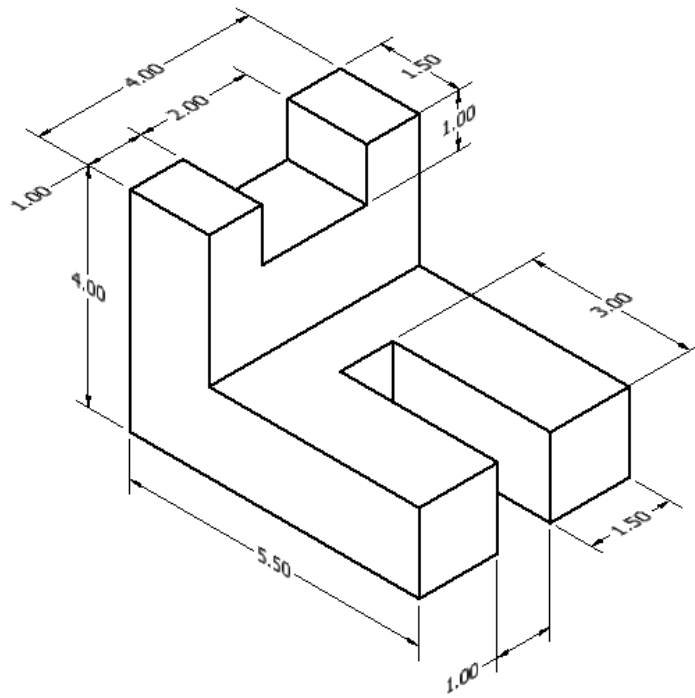


Step #3

Demonstration

Save this part as Isometric Demo into the 4.1 Creating Complex Parts folder.

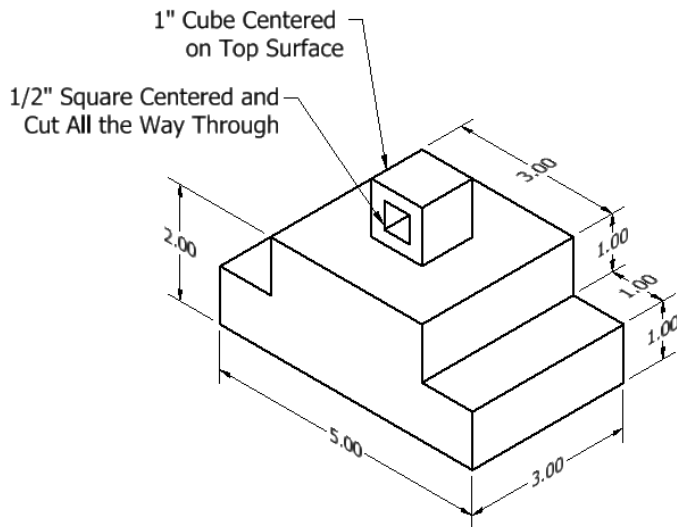
Isometric Demo



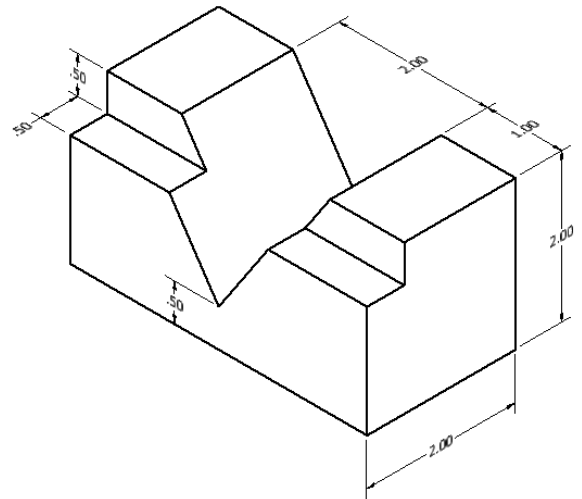
Activity

Create the following parts and save each one with the name listed in the 4.1 Creating Complex Parts folder.

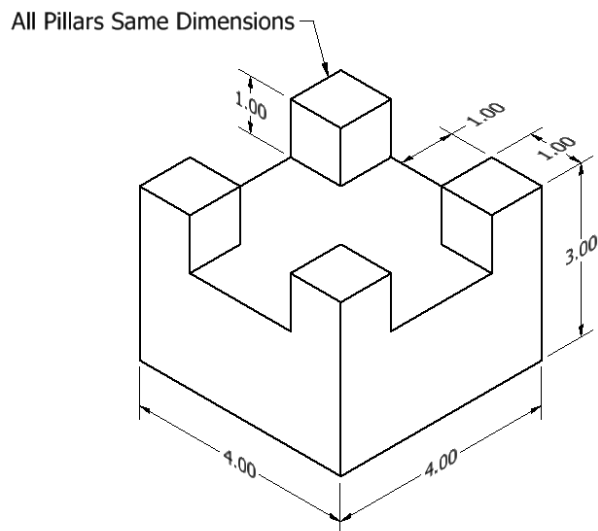
Isometric #1



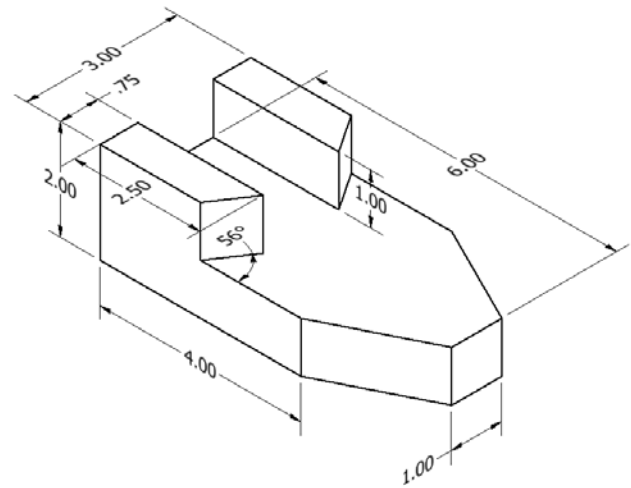
Isometric #2



Isometric #3

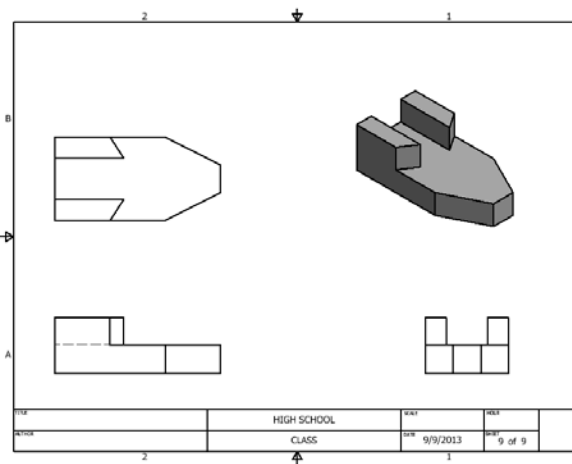
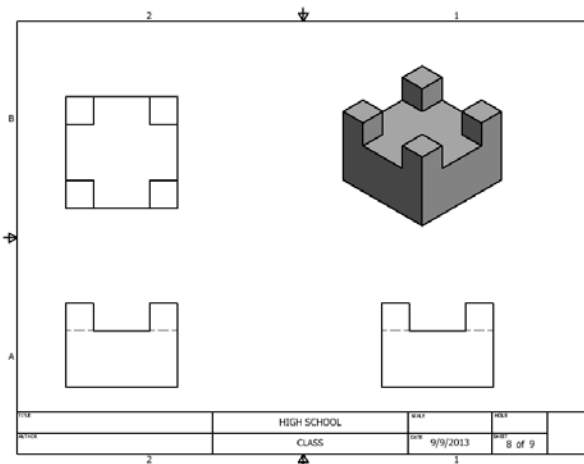
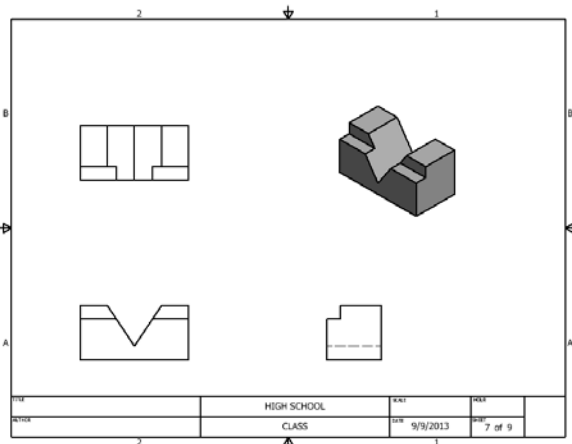
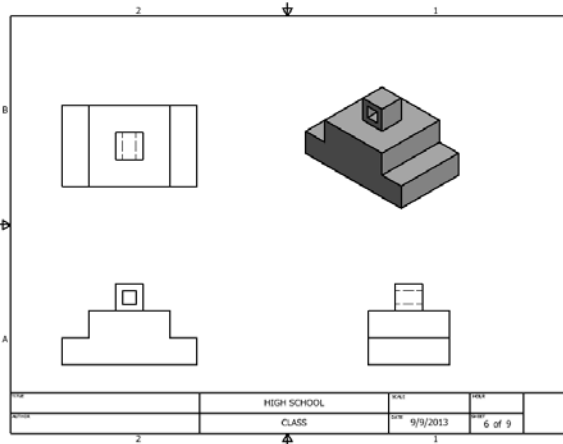
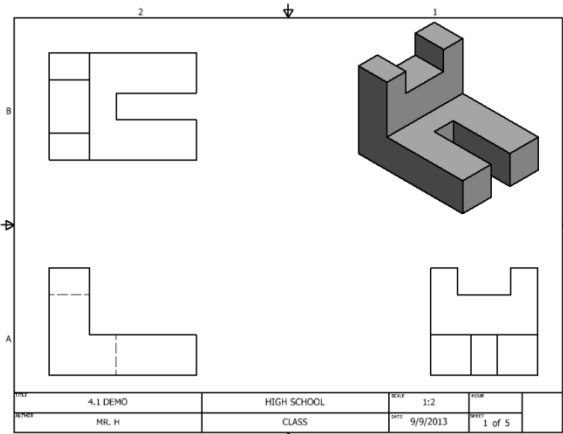


Isometric #4



Complex Part Sheet Set

Create a multiview of each part on your custom title block as shown below. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as *Creating Complex Parts Sheet Set* into the 4.1 folder.



Chapter 4

Lesson 2: Creating Complex Parts with Curves

Lecture

What is a good rule of thumb for determining the best front view?

The front view typically has the most detail of the part.

Why would you select a different origin plane when starting a new drawing?

If the part has more detail in another view starting it in another plane may be advantageous.

What is a placed feature?

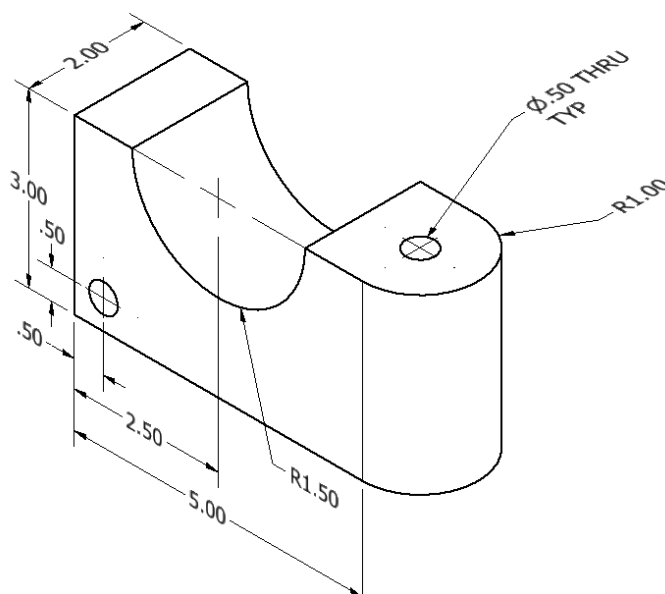
A placed feature is a feature which is applied to another feature, such as a fillet or chamfer. This is opposed to a feature which is made from a sketch (such as an extrusion).

When would you want to use the hole wizard?

When you need to place holes which are going to be used for fasteners like bolts.

Demonstration

Save this part as *Curved Part Demonstration* into the 4.2 *Creating Complex Parts with Curves* folder.



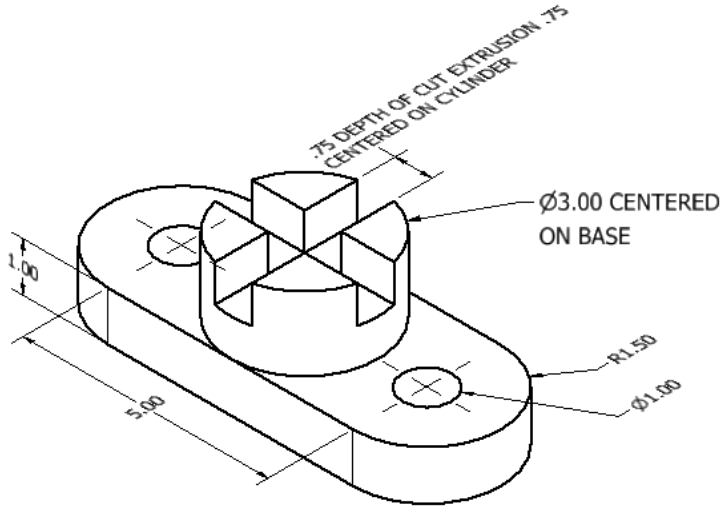
Activity

Create the

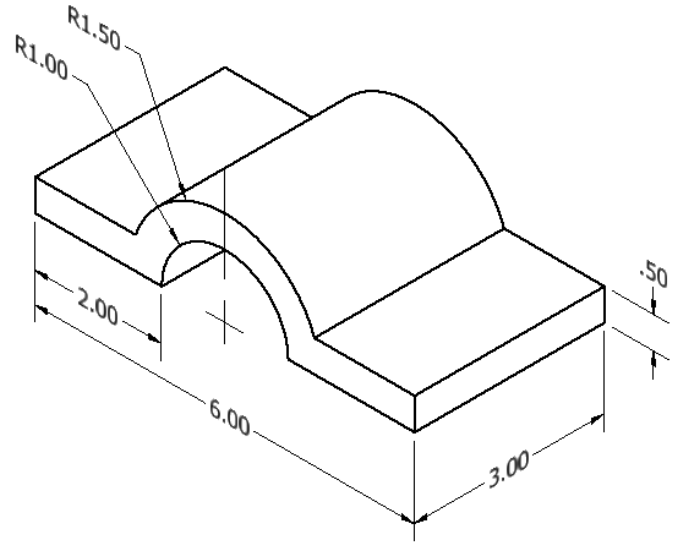
following parts and save

each one with the name listed into the 4.2 Creating Complex Parts with Curves folder.

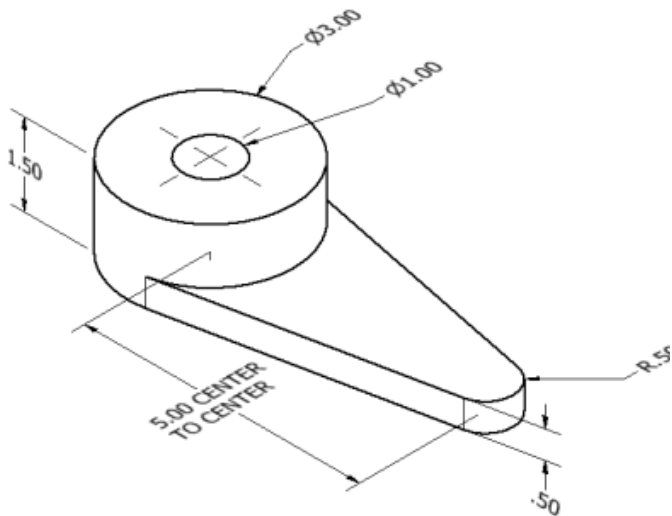
Curved Part 1



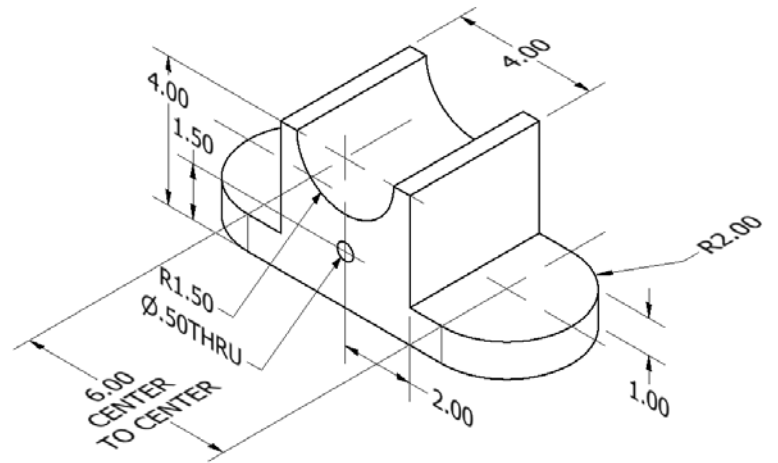
Curved Part 2



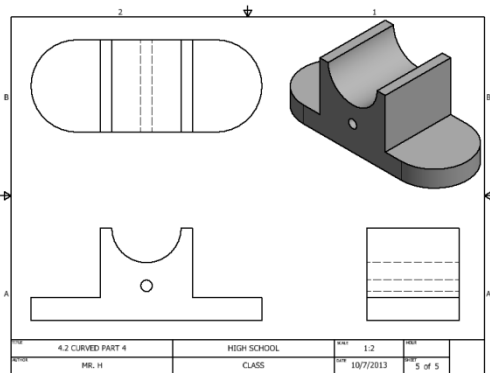
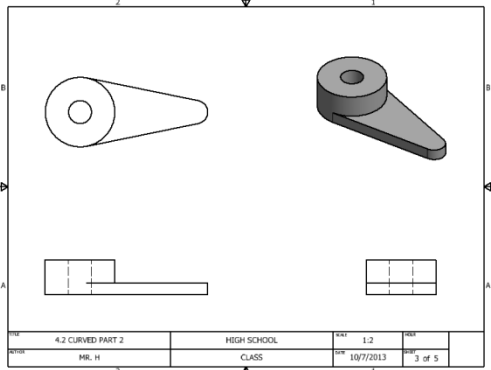
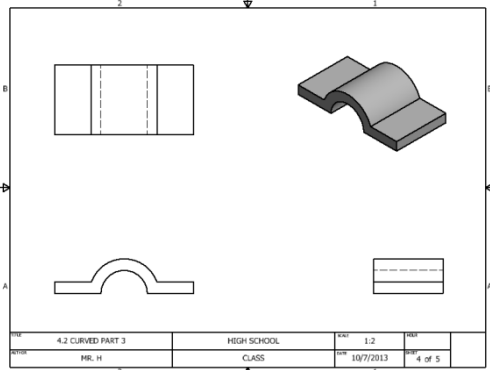
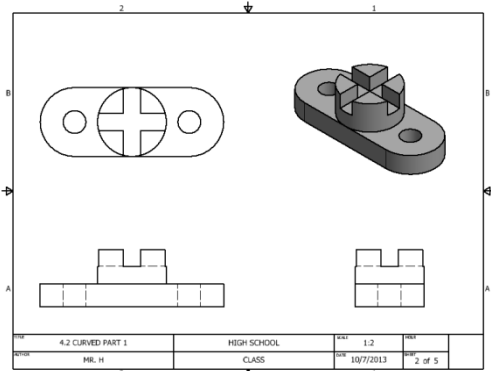
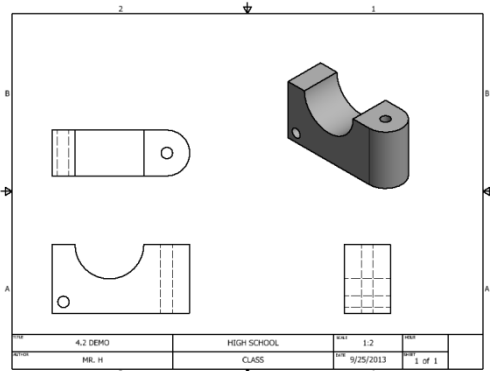
Curved Part 3



Curved Part 4

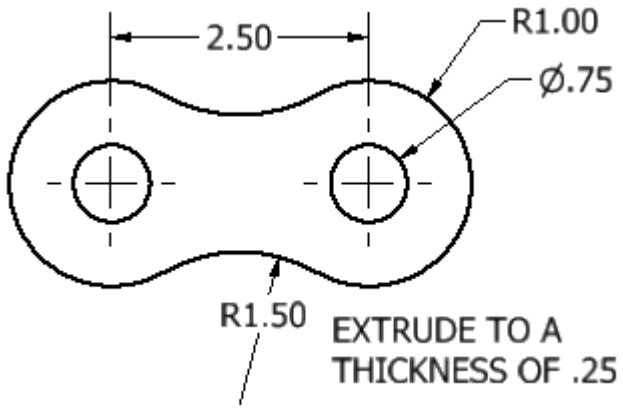


Create a multiview of each on your custom title block as shown on the next page. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as *Creating Complex Parts with Curves Sheet Set* into the 4.2 *Creating Complex Parts with Curves* folder.

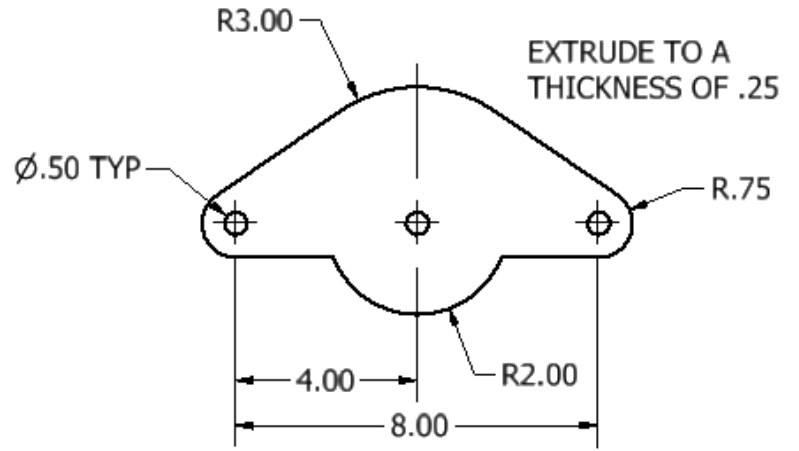


Challenge

Curved Challenge #1



Curved Challenge #2



Chapter 4

Lesson 3: Reference Planes

Lecture

What are Reference Planes used for?

Reference planes are used to draw sketches on planes which are not either on the original 3 planes or are not on a planer face available on the geometry.

How do you access the original Planes?

Right click on them in the feature manger

Describe a good strategy to use if an object has symmetry.

Create half the part on one side of a plane, then mirror all of the symmetric feature over that plane.

What does Convert Entities do?

Convert entities projects sketch lines from an already completed face on to another plane.

How can you tell which way a Reference Plane is facing?

- 1) The direction the triad is facing OR
- 2) (If your computer has high enough graphics capability) the color of the plane in the graphics area.

Is it necessary to specify a particular type of Reference Plane in order to create it? Explain.

No, but you do have to tell the computer on which features or geometry you would like the new plane based off of.

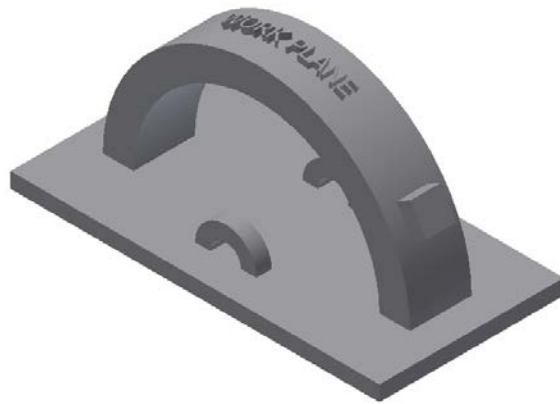
How do you turn off the visibility of a Reference Plane?

Right click on it and click on the eyeglasses.

Demonstration

Save this part as Work Plane Demo into the 4.3 Work Plane folder.

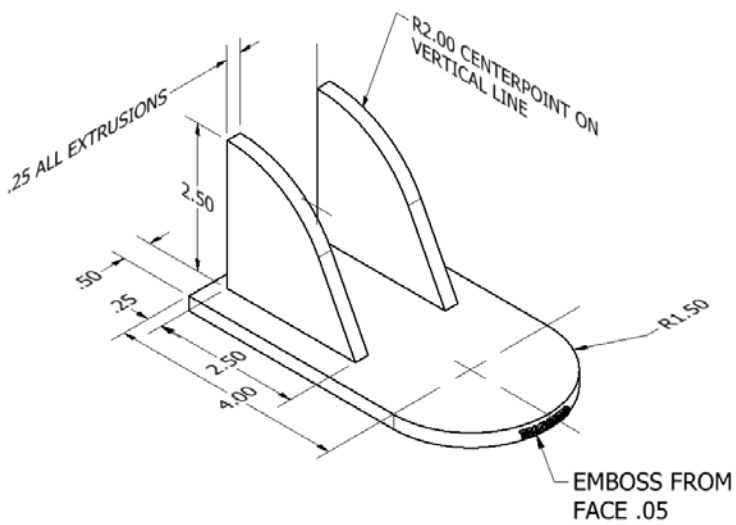
Work Plane Demo



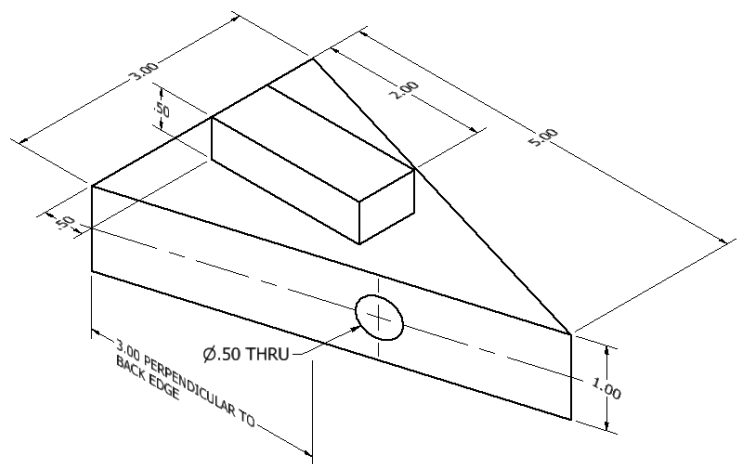
Activity

Create the following parts and save each one with the name listed into the 4.3 Work Plane folder.

Work Plane 1

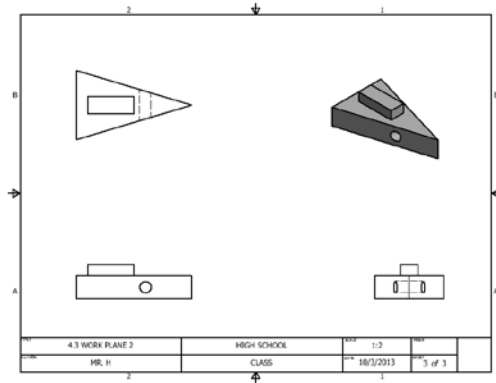
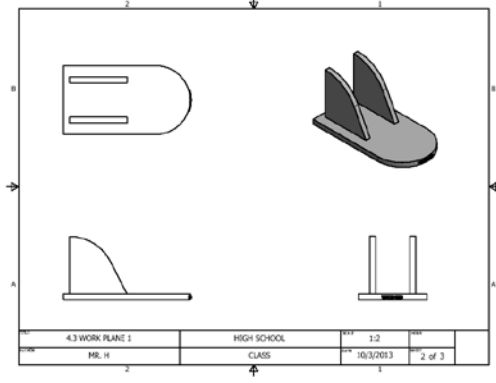
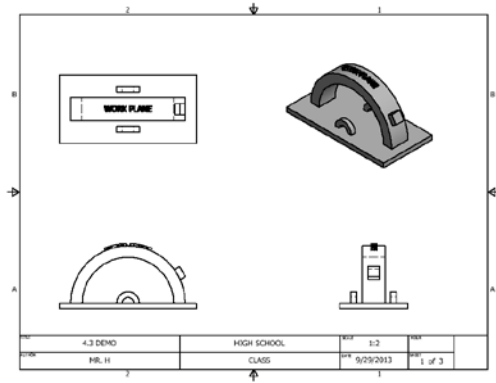


Work Plane 2



Work Plane Sheet Set

Create a multiview of each on your custom title block as shown on the next page. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Work Plane Sheet Set into the 4.3 Work Plane folder.



Chapter 4

Lesson 4: Pattern Tools

Lecture

What are the three types of pattern tools that can be applied to sketches and features?

Linear Pattern, Circular Patter, and Mirror

Define Linear Pattern.

A regularly repeating entity along a line

While creating a rectangular pattern what can be selected for defining pattern directions?

Linear edges

How can a linear pattern be used to generate patterns that are not rectangular?

By only selecting one direction to create a pattern which only goes in one direction

Define Circular Pattern.

Regularly repeating entities which are all located the same distance from a centerpoint

While creating a circular pattern what can be selected for defining the pattern?

Any circular edge or cylindrical face

Define Mirror.

A copy of an entity which is repeated across a plane or planer face.

While creating a mirror what can be selected for defining the operation?

A plane or planer face.

Is it possible to pattern entire parts? Is it possible to pattern parts in an assembly?

Yes, and Yes

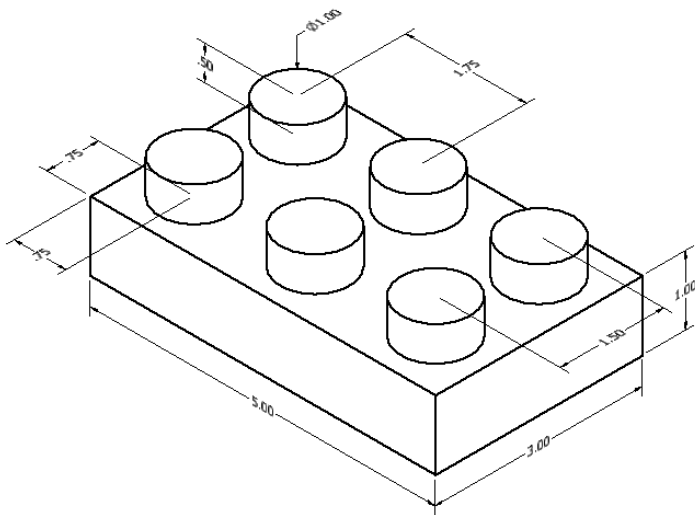
My preference is to pattern the _____ instead of the _____.

Feature instead of the sketch. Features are easier to edit later on.

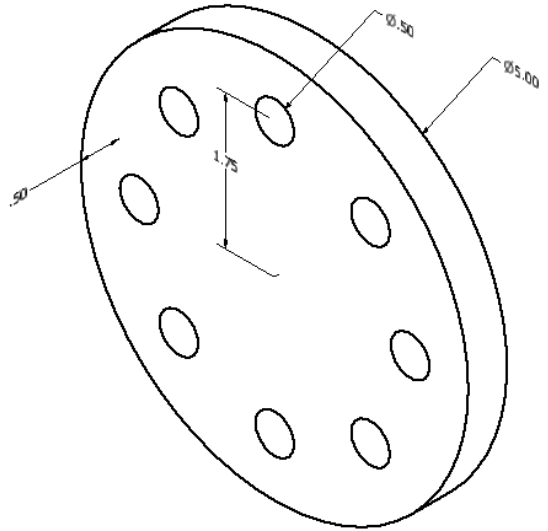
Demonstration

Save each demonstration part with the name provided into the 4.4 Pattern Tools folder.

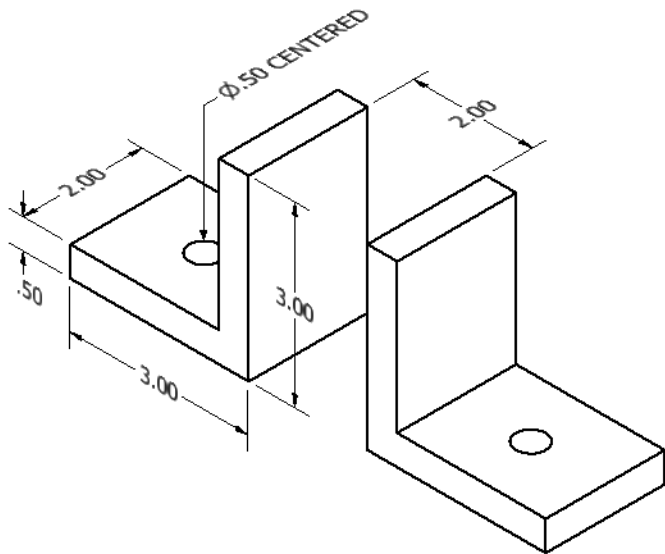
Rectangular Pattern Demo



Circular Pattern Demo



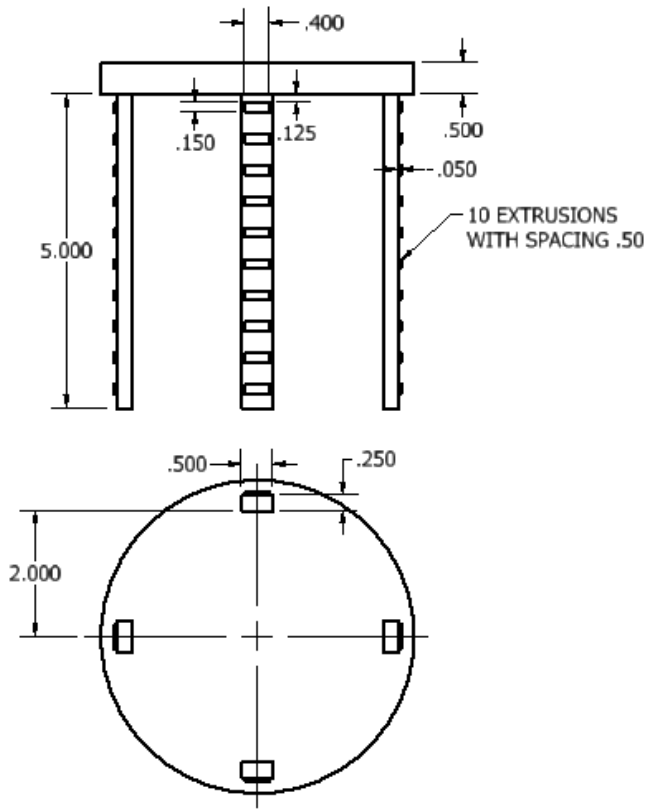
Mirror Demo



Activity

Create the following part and save with the name listed into the 4.4 Pattern Tools folder.

End Table



Pattern Tools Sheet Set

Create a multiview of each on your custom title block as shown on the next page. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Pattern Tools Sheet Set into the 4.4 Pattern Tools folder.

NAME	MIRROR	SCHOOL	HIGH SCHOOL	SCALE	1:2	DATE	10/3/2013	PAGE	6 of 6
DESIGNER	MR. H	CLASS	CLASS						

NAME	CIRCULAR PATTERN	SCHOOL	HIGH SCHOOL	SCALE	1:1	DATE	10/3/2013	PAGE	5 of 6
DESIGNER	MR. H	CLASS	CLASS						

NAME	RECTANGULAR PATTERN	SCHOOL	HIGH SCHOOL	SCALE	1:1	DATE	10/3/2013	PAGE	4 of 6
DESIGNER	MR. H	CLASS	CLASS						

Technical drawing with dimensions: 400, 1.500, .150, .125, .500, .050, 5.000, 10 EXTRUSIONS WITH SPACING .50, .500, .250, 2.000.

NAME		SCHOOL	HIGH SCHOOL	SCALE		DATE	10/10/2013	PAGE	7 of 7
DESIGNER		CLASS	CLASS						

Chapter 5

Creating Advanced Features

Lesson 5.1 Revolve:

- Demonstrate how to create a revolve feature
- Utilize the Center line tool to create a revolve

Lesson 5.2 Loft:

- Demonstrate how to create a loft feature

Lesson 5.3 Sweep:

- Explain the difference between a path and a profile
- Demonstrate how to create a sweep feature

Lesson 5.4 Shell:

- Demonstrate how to create a shell feature

Lesson 5.5 Helix and Spiral:

- Demonstrate how to create a coil feature

Lesson 5.6 Advanced Hole Wizard:

- Compare the differences between hole types
- Explain how to read a thread note
- Demonstrate how to create Counterbore, Countersink, and Tapped Holes

Lesson 5.7 SolidWorks Toolbox:

- Demonstrate how to load in the SolidWorks Toolbox
- Demonstrate how use the Toolbox with the Hole Wizard to populate fasteners

Chapter 5

Lesson 1: Revolve

Lecture

What is a Revolve?

A 2d profile “extruded” around an axis instead of in a line.

Where should the line that defines the axis of rotation be drawn?

On the outside of the 2d profile.

What happens if a profile is not connected to the centerline?

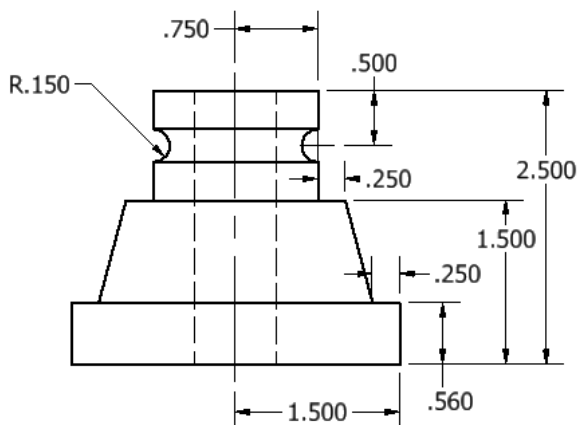
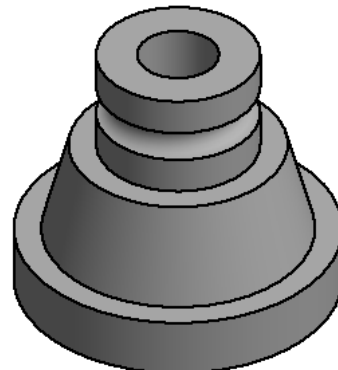
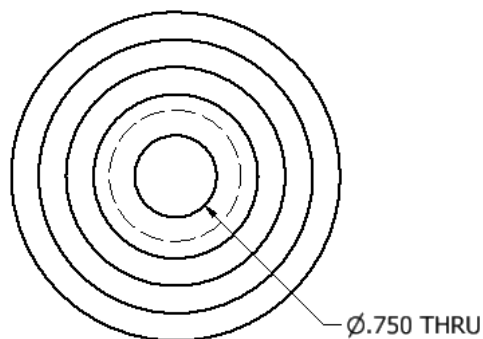
The revolve will take place offset from the centerline, creating a rim.

What is the advantage of using a centerline as the axis?

SolidWorks will not attempt to do anything else with that line.

Demonstration

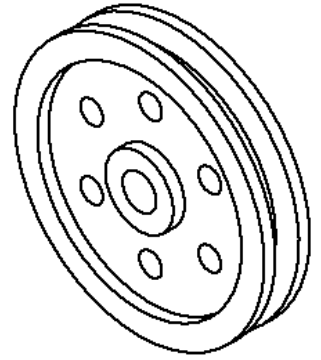
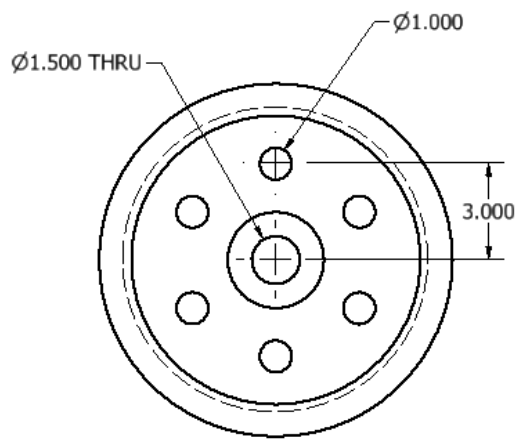
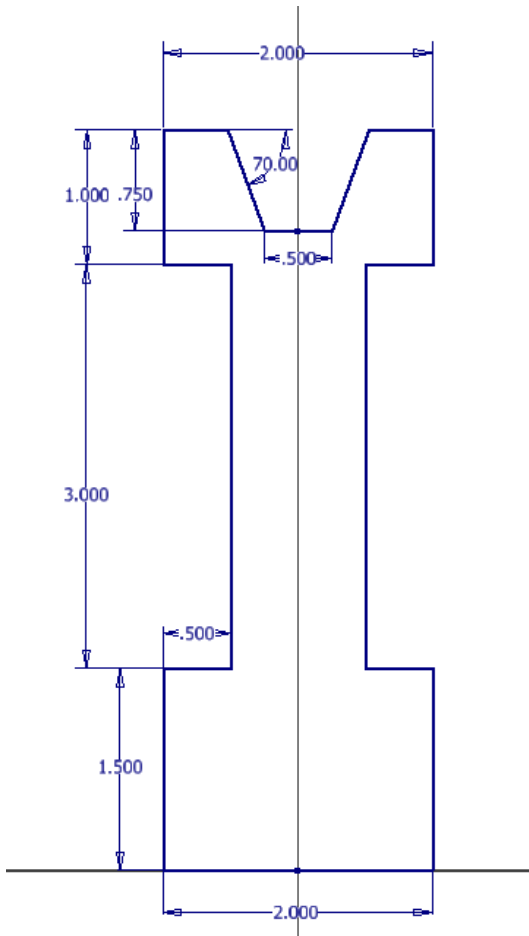
Save the demonstration file as Revolve Demo into the 5.1 Revolve folder.



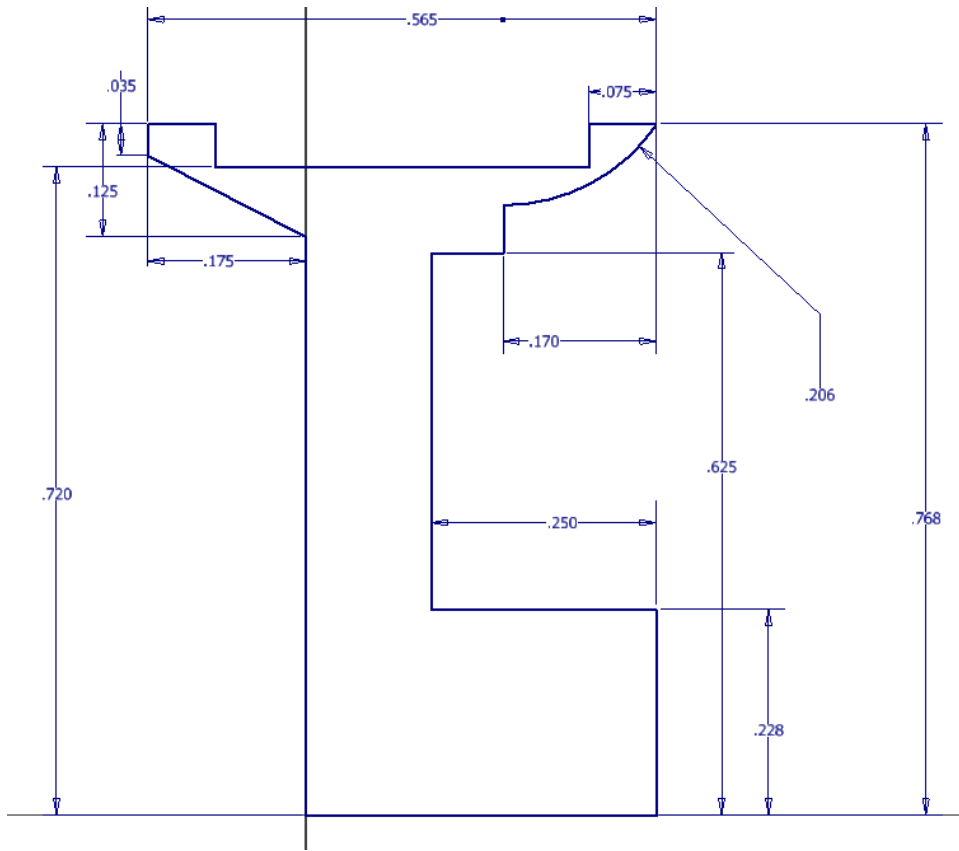
Activity

Create the following parts and save with the name listed into the 5.1 Revolve folder.

Pulley

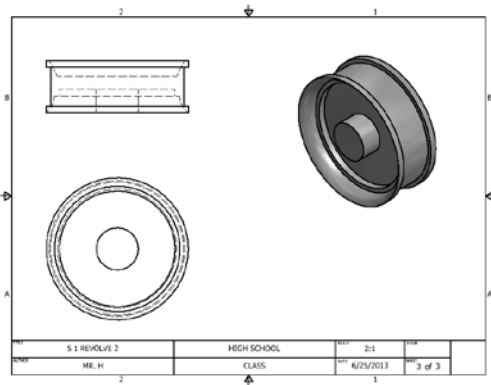
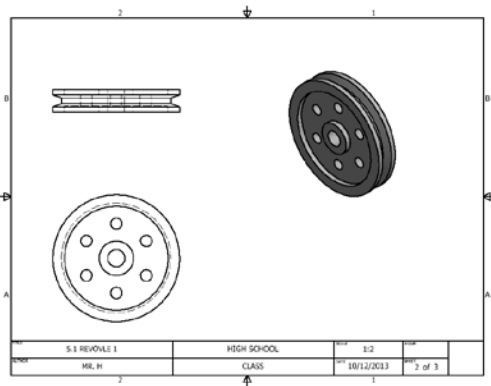
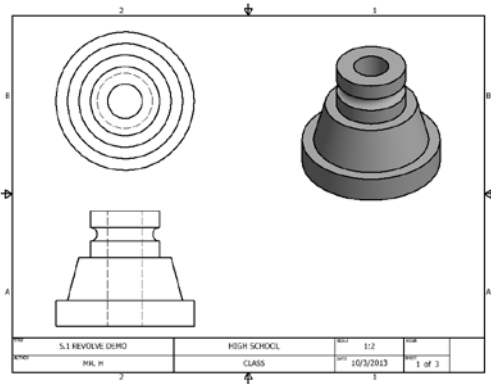


Rim Blank



Revolve Sheet Set

Create a multiview of each on your custom title block as shown on the next page. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Revolve Sheet Set into the 5.1 Revolve folder.



Challenge

Customize the rim blank by cutting or adding additional features and using circular pattern to make the best design.

Chapter 5

Lesson 2: Loft

Lecture

What is a Loft?

A 3d solid formed by linking the profiles of multiple, parallel sketches together.

What is the minimum number of sketches necessary to create a loft? Maximum?

Minimum, 2. No maximum.

Can the loft command cut as well?

Yes

When should we use the loft command?

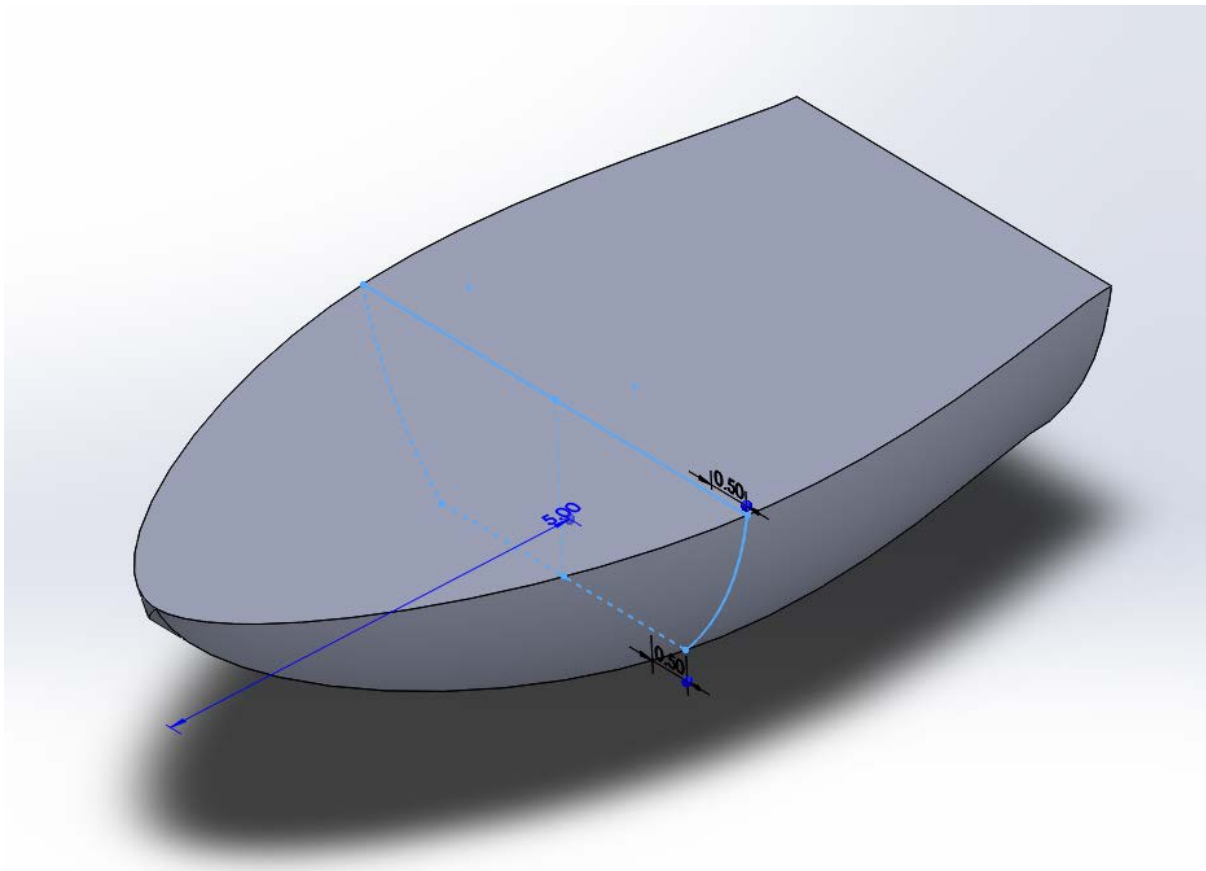
When you need to create geometry with smooth curves, it is particularly good at making “organic” forms.

Give three real world examples of when a loft might be used.

Many possible answers. A mouse, a handle, video game controller, a spoon, boat hull, airplane wing, propeller, etc.

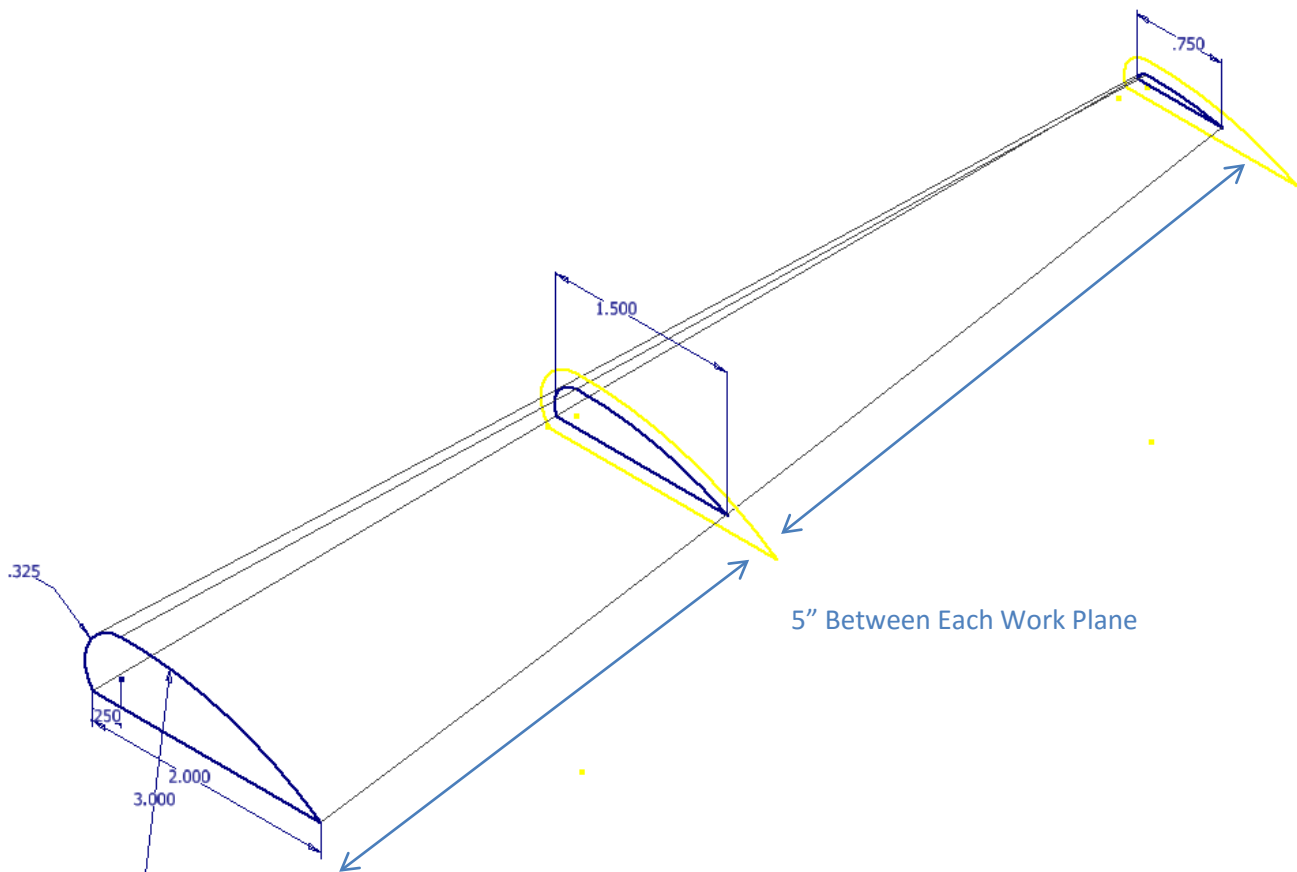
Demonstration

Follow along with the demonstration and create the boat hull part. Save this part as Boat Hull into the 5.2 Loft folder.



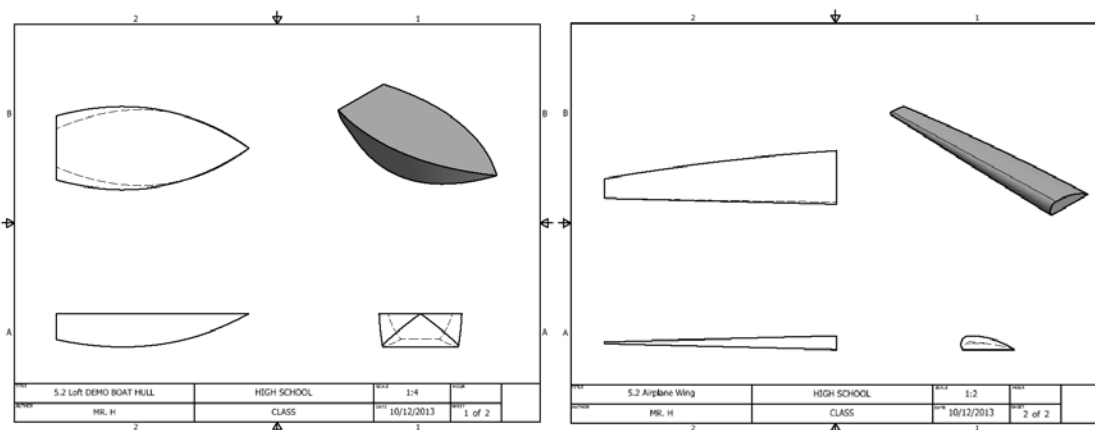
Activity

Create the Airplane Wing using the loft command. Hint: Use Project Geometry and offset once you have created the initial sketch. Save as Airplane Wing in the 5.2 Loft folder.



Loft Sheet Set

Create a multiview of each on your custom title block as shown below. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Loft Sheet Set into the Lesson 5.2 Loft folder.



Challenge

Research how to create a loft using the centerline or rails option. See if you can create a more complex loft using either of those options.

Chapter 5

Lesson 3: Sweep

Lecture

What is the Sweep command?

A profile that has been extruded along a user defined path, rather than in a straight line as in an extruded boss/base (although you could sweep along a straight line).

What is necessary to create a sweep?

A profile sketch and a path sketch which intersect with one another.

Can the sweep command cut?

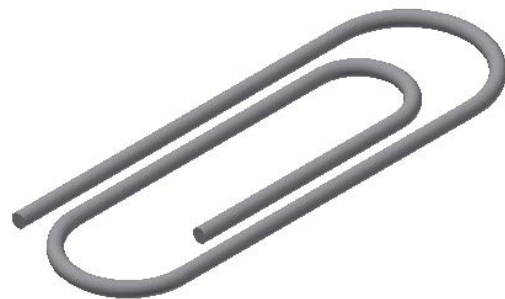
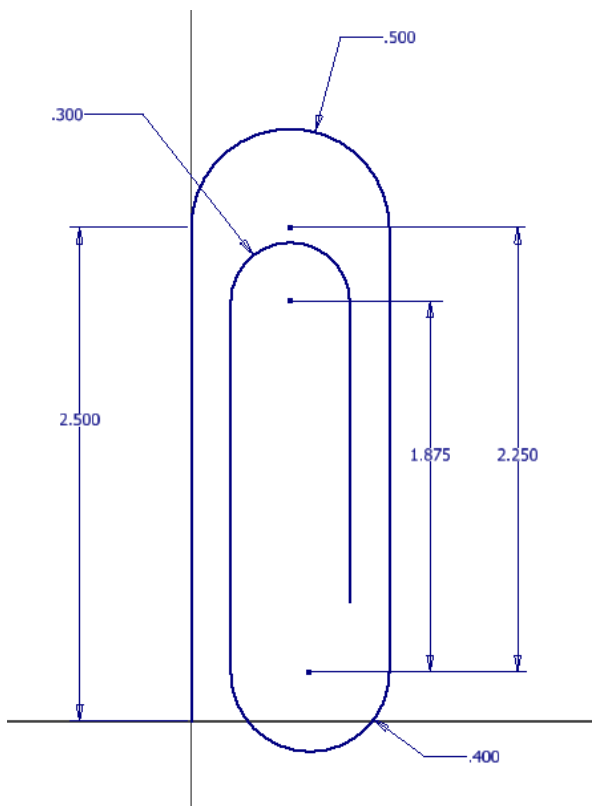
Yes

Is it only possible to sweep circles?

No, any sketch which has a point can be swept.

Demonstration

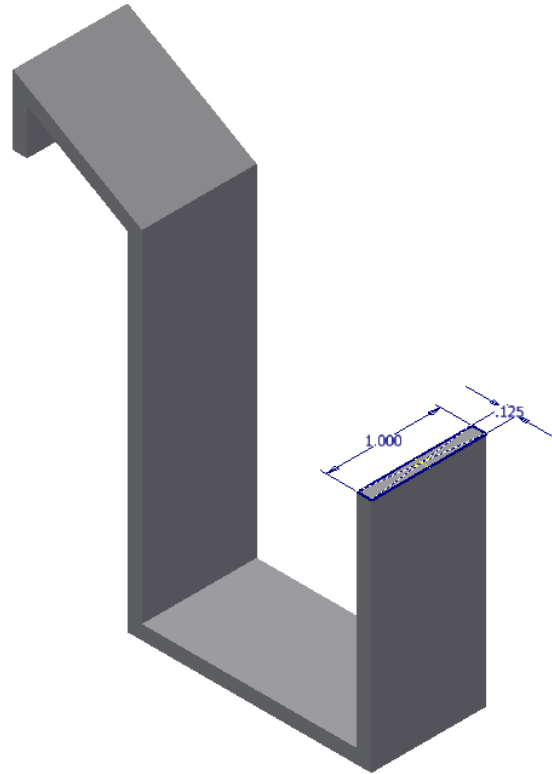
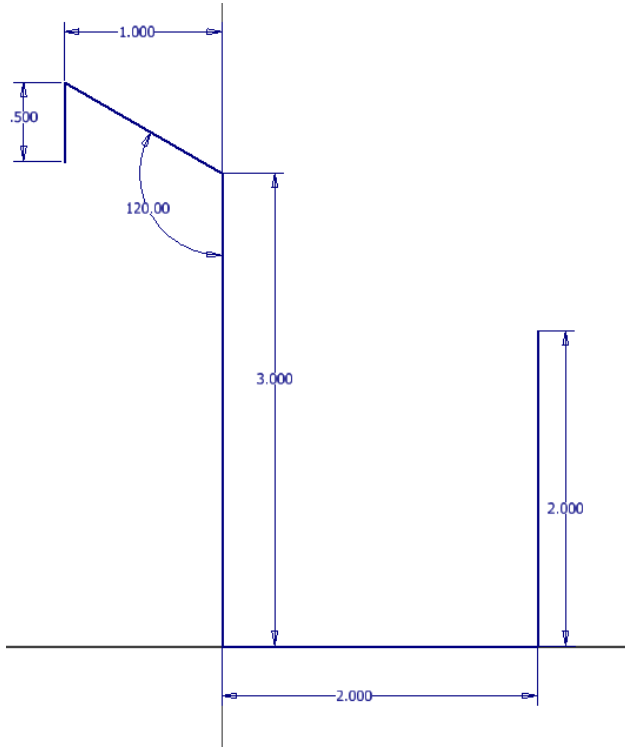
Save this part as Paper Clip into the Lesson 5.3 Sweep folder.



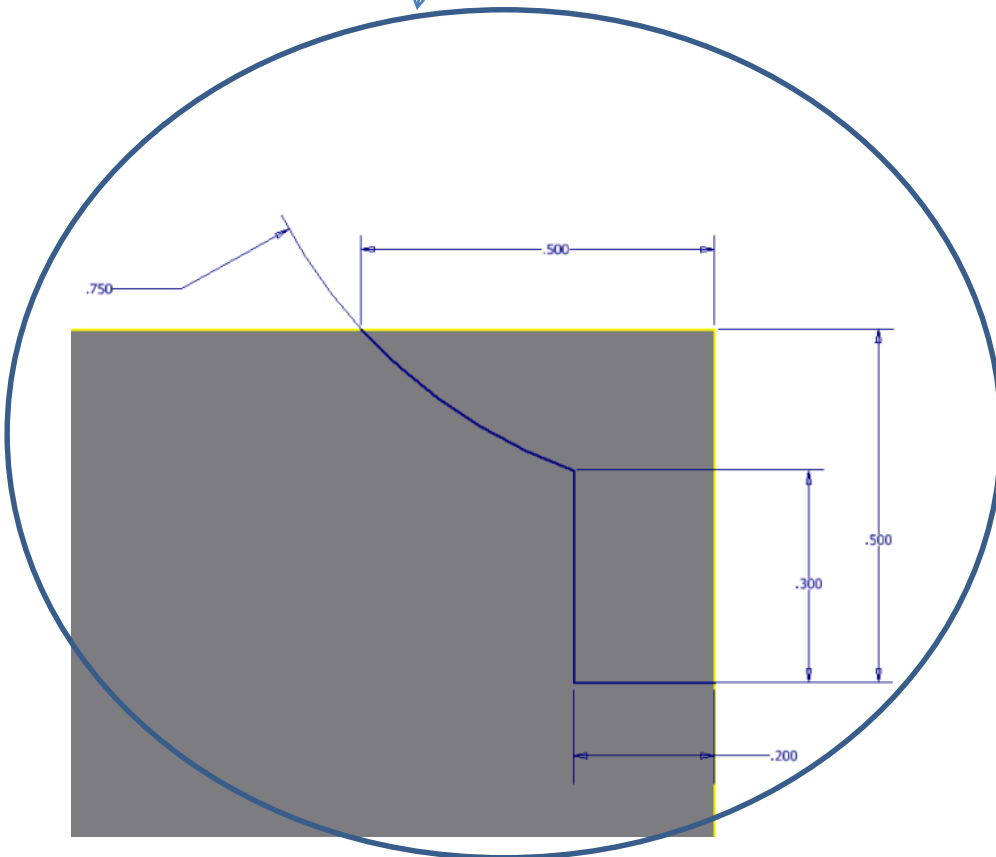
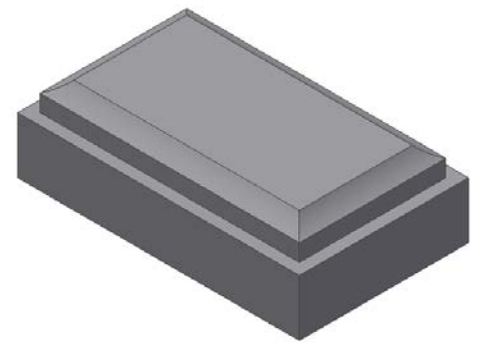
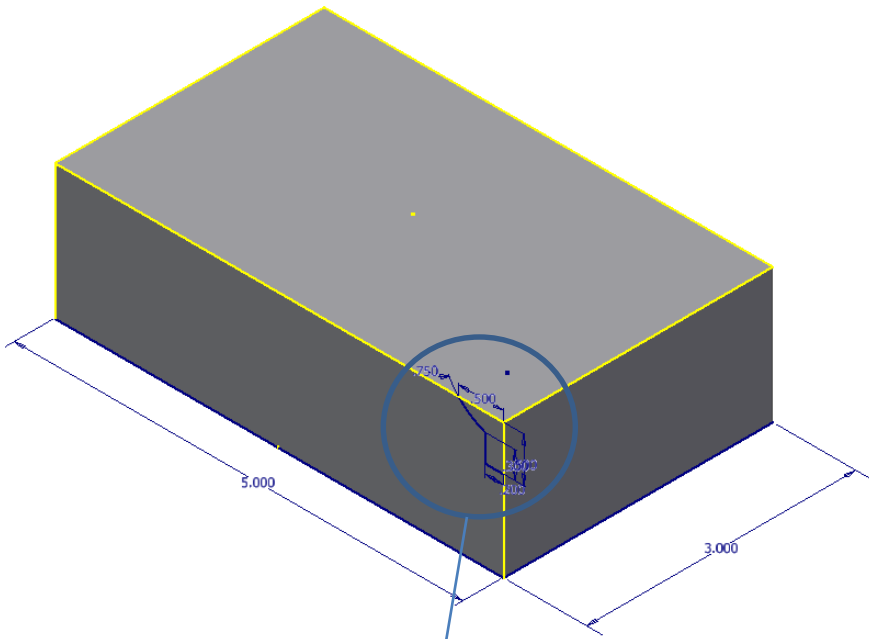
Activity

Create the following parts and save each one with the name listed in the 5.3 Sweep folder.

Bracket

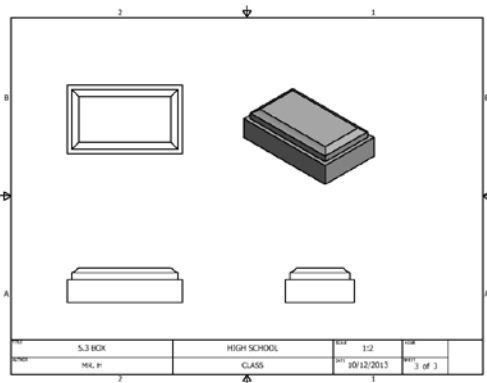
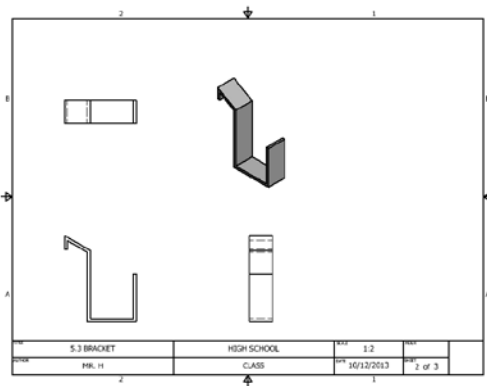
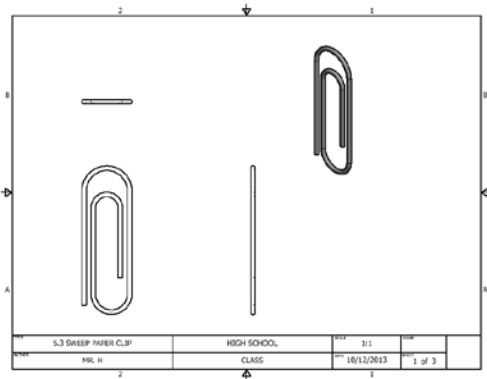


Box



Sweep Sheet Set

Create a multiview of each on your custom title block as shown on the next page. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Sweep Sheet Set into the Lesson 5.3 Sweep folder.



Challenge

Create a sketch using work planes that goes in the X, Y and Z direction. Create a sweep that follows that path.

Chapter 5

Lesson 4: Shell

Lecture

What does the shell command do?

Removes the interior volume of a solid and removes specified faces.

Does a face need to be removed in order to create a shell?

No, you can hollow the object and keep all of the faces.

Is a shell applied to an individual extrusion or an entire part?

The entire part.

What is the difference between the outside and inside option?

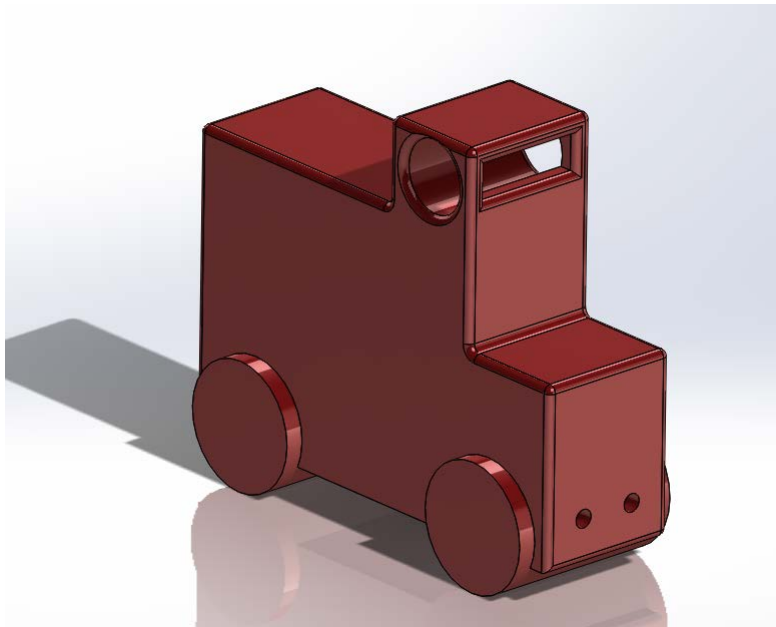
The inside removes material from the interior while the outside add material to the exterior.

How do you specify a unique thickness for a face?

Click in the multi-thickness box, and select the faces, and specify the thickness for that face.

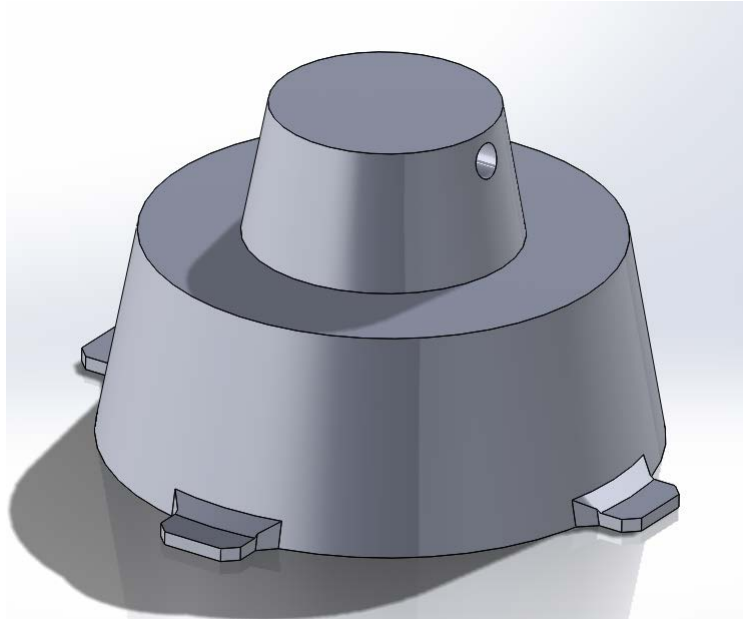
Demonstration

Open the part named *Shell Demo* in the *Lesson 5.4 Shell* folder. Follow the demonstration and save when complete.



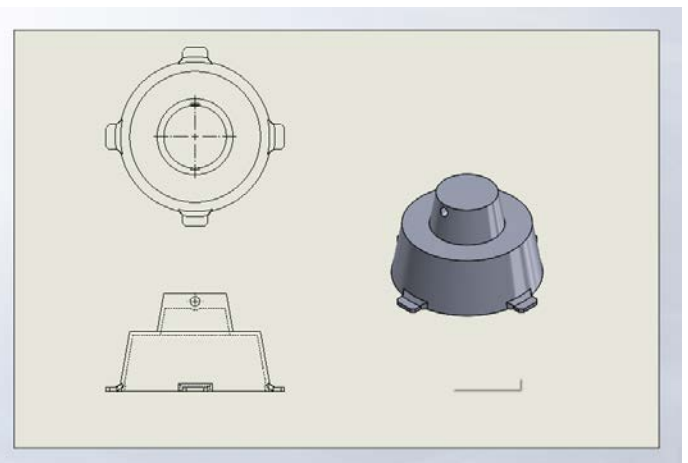
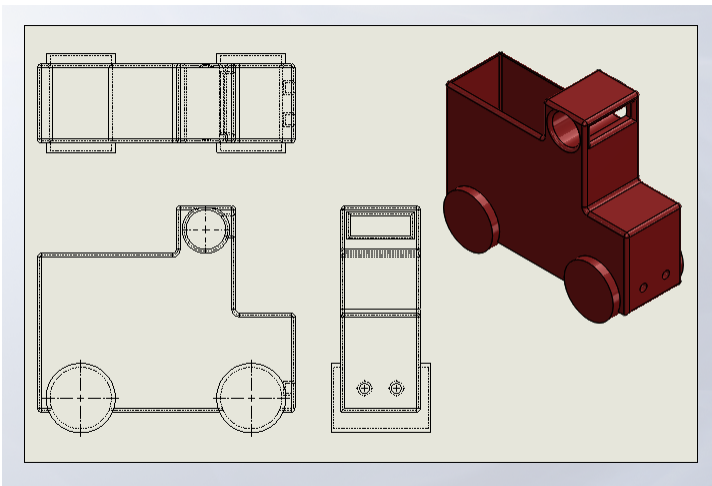
Activity

Open the file named Shell Activity. Shell the part to a thickness of .1 inches except for the top face, make that face a unique thickness at .4 inches, remove the bottom face. Click save when complete.



Shell Sheet Set

Create a multiview of each on your custom title block as shown on the next page. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Shell Sheet Set into the Lesson 5.4 Shell folder.



Chapter 5

Lesson 5: Helix and Spirals

Lecture

What does the helix/spiral command do?

Creates a “line” that is in the shape of a helix or spiral, this new piece of geometry is not a sketch.

What needs to be included in a sketch to create a helix or spiral?

A circle.

What is the difference between a helix and a spiral?

A helix is 3 dimensional and a spiral is 2 dimensional.

What is “pitch”?

The distance between the rotations of the helix or spiral.

What are three ways we can define a helix?

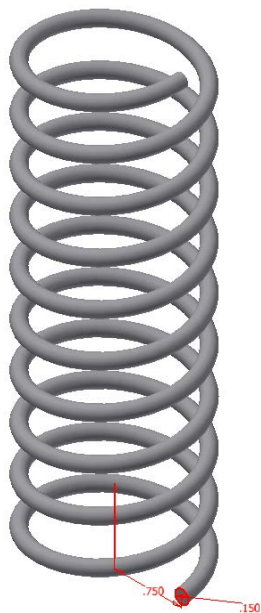
- 1) Height and pitch
- 2) Pitch and number of revolutions
- 3) Number of revolutions and pitch

Demonstration

Save this part as Coil Demo into the 5.5 Helix and Spirals folder.

Pitch = .5”

Revolution = 10

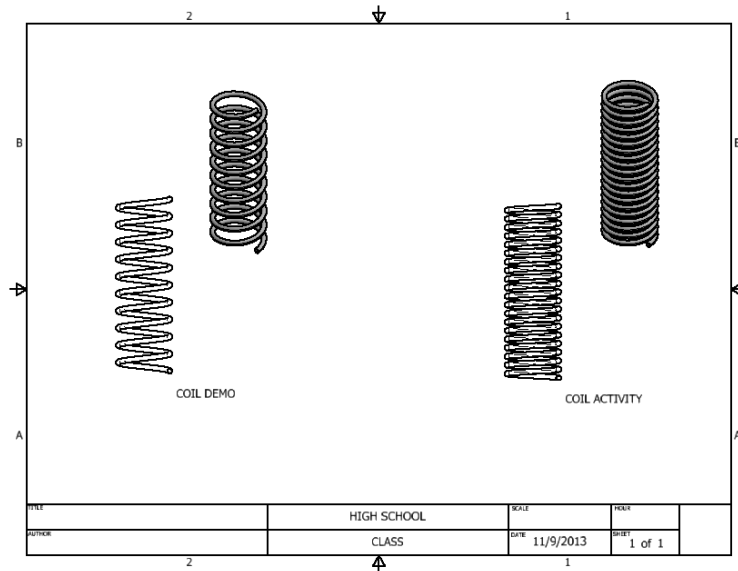


Activity

Open the Coil Demo part that you just created. Edit the feature so the pitch is .25 and the revolution is 20. Do a Save As and name the new part Coil Activity.

Coil Sheet Set

Create a front view with an isometric on your custom title block as shown below. Add text to label each part and place both parts on the same sheet. Ask your teacher if the parts should be dimensioned or if there are any additional requirements. Save the drawing file as Coil Sheet Set into the Lesson 5.5 Coil folder.



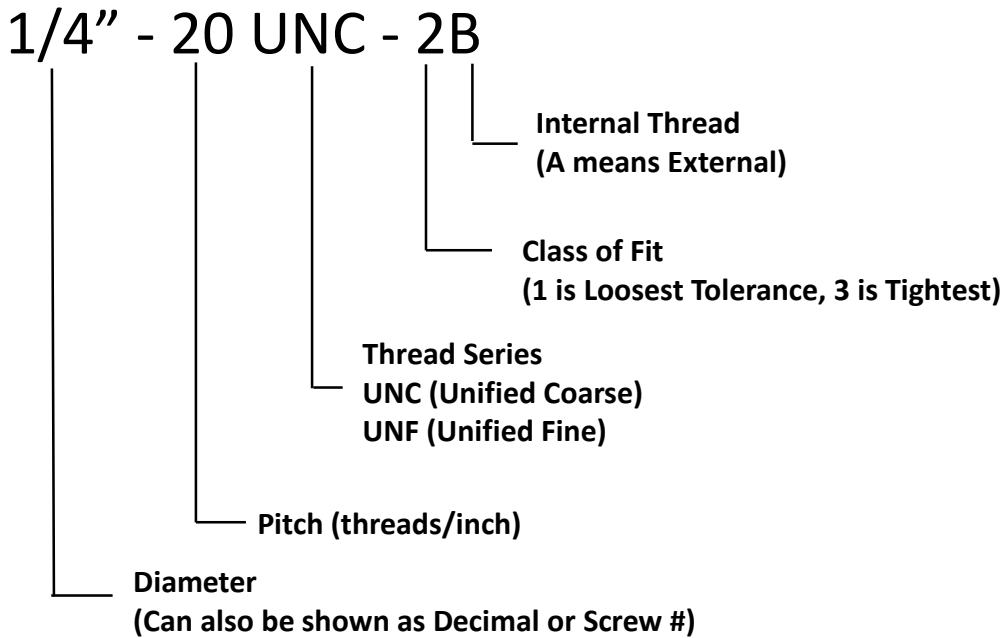
Challenge

Use planes and cuts to create flat tops and bottoms on a coil.

Chapter 5

Lesson 6: Advanced Hole Wizard

Lecture



What term is used in the hole wizard dialog box to create a threaded hole?

Strait tap.

What thread type is most typically used in the United States?

Unified Thread Standard

What does a thread label with "Class B" signify?

Internal Thread

How can you tell if a hole is threaded in the graphics area, without looking at the note?

If it has a dashed line surrounding the hole.

What is a Counterbore Hole?

A hole with a depressed cavity where the fastener head can sit flush with the surface of the workpiece.

Why does SolidWorks oversize a counterbored hole?

To accommodate for the size of the fastener itself.

What symbol is used to denote a Counterbore?

A wide, square shaped U.

What is a Countersink hole?

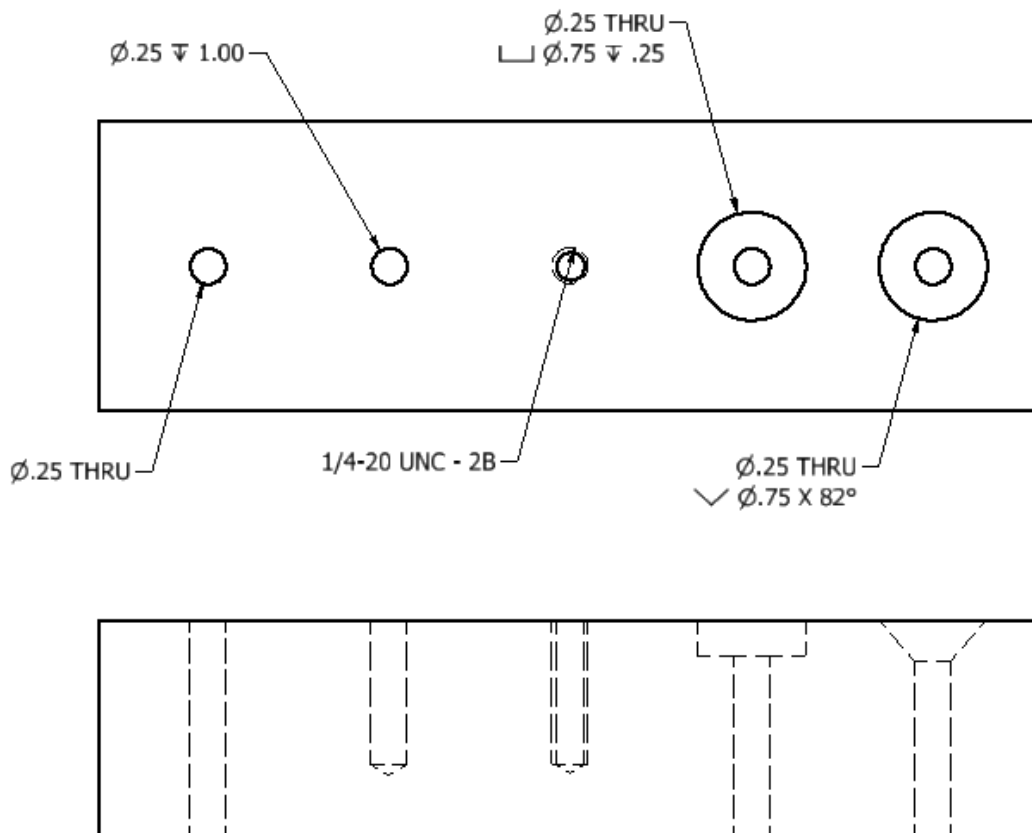
A hole where the top is tapered so that the faster can sit flush with the top of the workpiece with no gaps.

What symbol is used to denote a Countersink?

A "V"

Demonstration

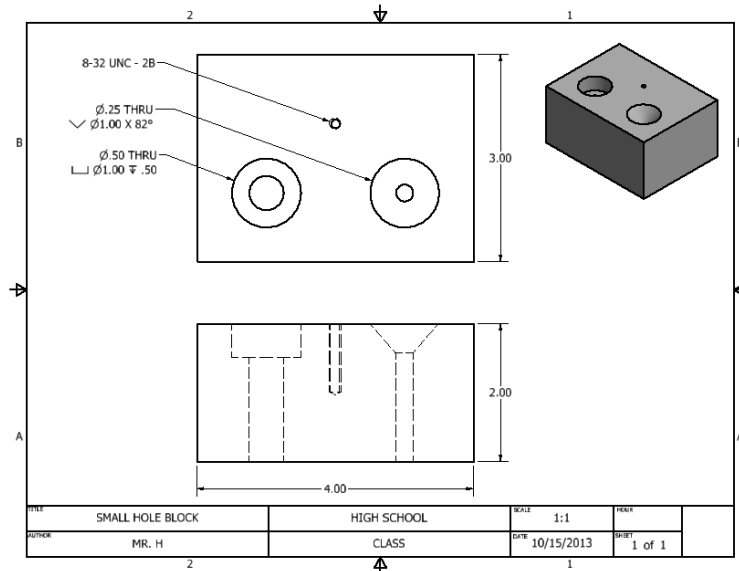
Save this part as Hole Demo into the Lesson 5.6 Advanced Holes and Threads folder.



Activity

Create the following block and then create the holes shown. Save the part as *Small Hole Block* in the 5.6 Advanced Hole and Thread folder. Place the views on your custom title block and dimension as shown below. Save the drawing as *Hole and Thread Sheet Set*.

Hole and Thread Sheet Set



Challenge

Research bolt sizes and create a realistic bolt with a thread.

Chapter 5

Lesson 7: SolidWorks Toolbox

Lecture

Is the Toolbox normally activated in SolidWorks?

No, you have to turn it on.

Where can you find the Toolbox components?

In the resource library on the right hand side of the screen.

What happens if you place a toolbox component in to empty space?

It will enter the design space as a configurable part.

What is meant by the Toolbox components being configurable?

You can edit their properties.

How are Toolbox components indicated in the Assembly Manager?

By a small bolt icon.

Can you mate Toolbox components?

Yes, like any other components in an assembly.

What happens if you place a toolbox component in to a hole created by the Hole Wizard?

It will automatically size itself.

What does the smart fasteners command do?

Automatically populates hole wizard holes with the correct fasteners.

Demonstration

Open the file called "Hole Block". Follow along with the demonstration and save when complete.

Activity

No Activity or prints required for this lesson

Challenge

Create 2 parts and attach them with Toolbox Fasteners.

Chapter 6

Creating Advanced Drawings

Lesson 6.1 Special Drawing Views:

- Demonstrate how to create an Auxiliary View
- Demonstrate how to create a Section View
- Demonstrate how to create a Detail View
- Demonstrate how to create a Break.

Lesson 6.2 Advanced Dimensioning:

- Explain the use of centerlines and center marks
- Demonstrate how change the size of a center mark
- Demonstrate the use of baseline dimensions
- Demonstrate the use of the text tool in a drawing file
- Modify a dimension's text

Chapter 6

Lesson 1: Special Drawing Views

Lecture

What does the projected view do?

Creates a view which is projected either horizontally or vertically from the base view.

List three special types of views that can be used to clarify a drawing.

Auxiliary, Section, and Detail

When is an auxiliary view used?

To show a projected view which cannot be shown from either a horizontal or vertical direction.

What does an auxiliary view do?

Shows a "normal to" view from a user defined direction.

When is a section view used?

To see the interior of an object.

What type of line is used to define where a part is sectioned?

A section line.

What type of lines is used to represent materials in a section view?

Hatched lines.

What does a break line represent?

It shows where a lengthy object is cut down to size so that it can fit on a printed page.

When is a detail view used?

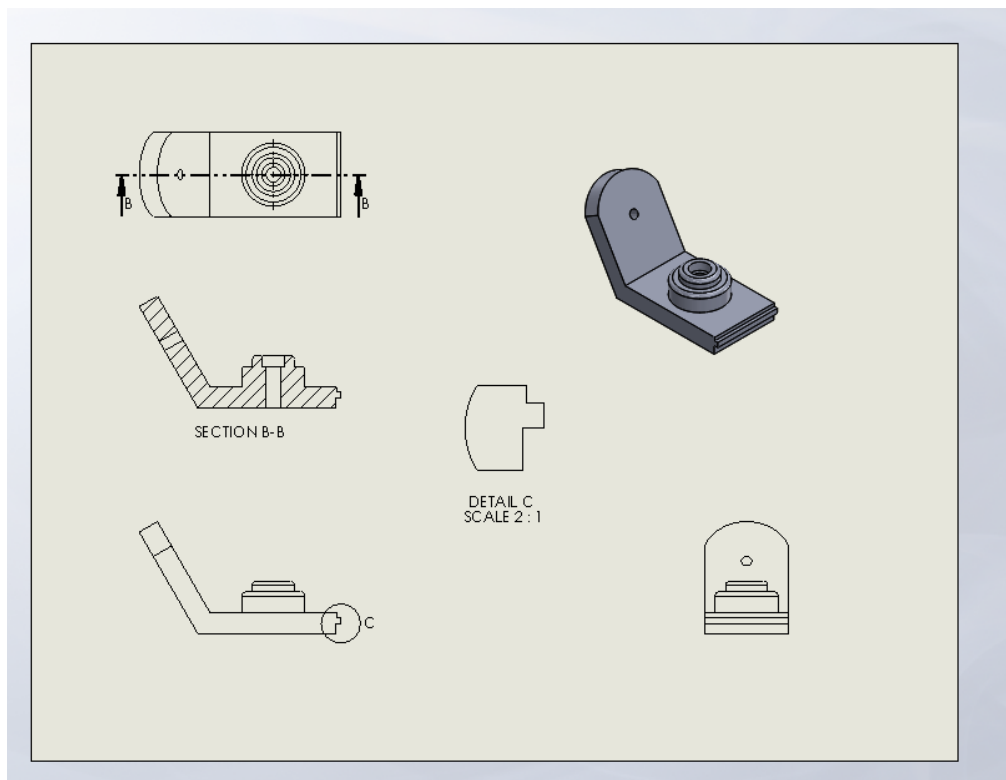
To show small details that would be difficult to see in the normally scaled drawings.

Demonstration

Open the .idw file named *Special Drawing Views Demo*. Make changes as demonstrated and save when finished.

Activity

Use the file named *Special Drawing Views Activity* to create a sheet using a Section, Auxiliary, and Detail view as shown below.



Challenge

Use these skills to create special views of projects you create in the future.

Chapter 6

Lesson 2: Advanced Dimensioning

Lecture

Holes are typically dimensioned to what?

Their centerpoint

How do you change the size of a centermark?

Select the centermark then uncheck the “use document setting” box, put a custom value in the “mark size” box.

What does a baseline dimension do?

Dimensions a number of attributes to a common line (the baseline).

How do you insert a baseline dimension?

Click on the smart dimension flyout and select baseline dimension, click on the line you wish to use as your baseline, then click on the entities you wish to dimension.

What are three changes that can be made while editing a dimension?

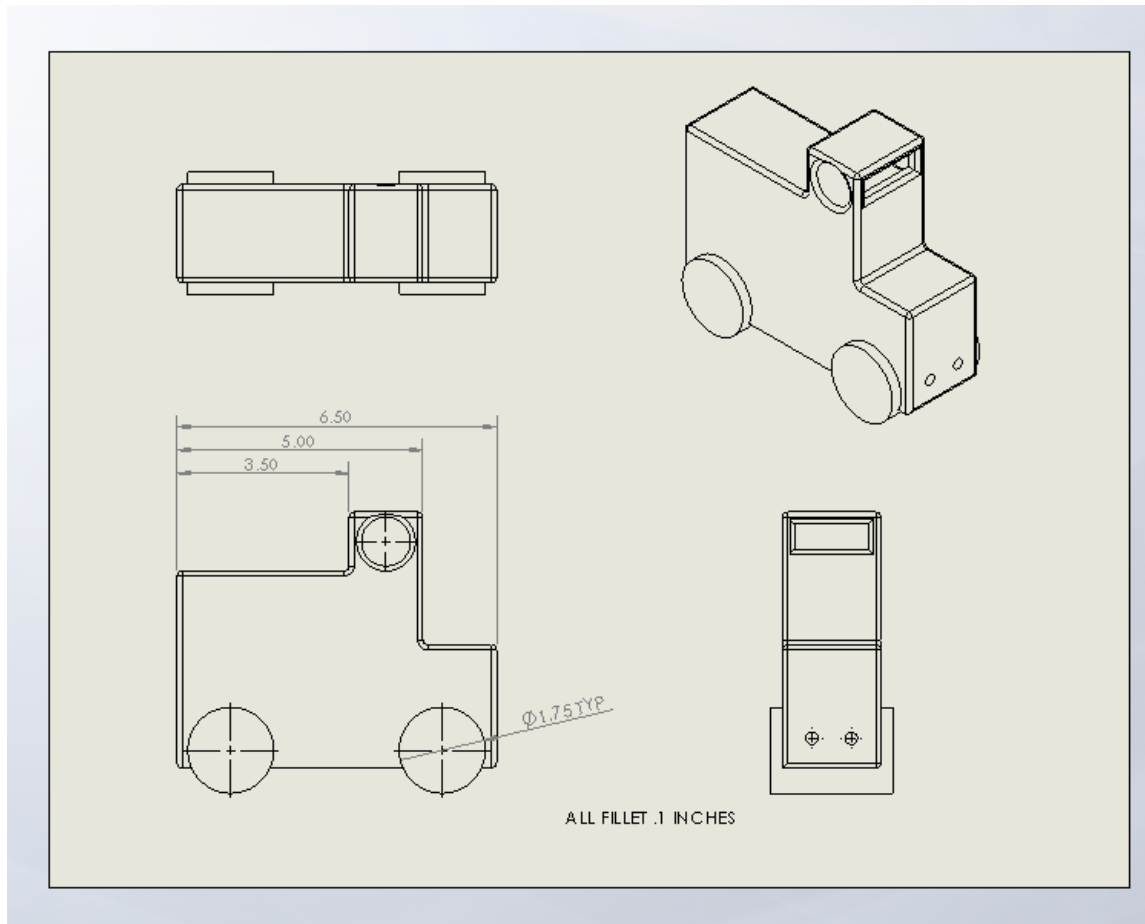
- 1) The text
- 2) The position
- 3) The size

How can you add text to a drawing?

Use the note command located in the annotation tab.

Demonstration

Open the drawing file named *Shell Demo – Advanced Dimensioning*. Follow the demonstration and save the file when completed.



Activity

No Activity

Challenge

Open the drawing sheet created in 4.2 named *Curved Part Sheet Set*. Properly dimension the four parts using the techniques learned.

Chapter 7

Creating Advanced Assemblies

Lesson 7.1 More Assembly Constraints:

- Demonstrate the use of Angle, Tangent and Insert Assembly Constraints

Lesson 7.2 Create Component:

- Demonstrate the process of creating a Component based off of another part in an assembly
- Define Adaptive

Lesson 7.3 Sub-Assemblies and Pattern Tools:

- Demonstrate the process of creating a Sub-Assembly
- Demonstrate the process of creating a Pattern or Mirror of a Component in an assembly

Chapter 7

Lesson 1: More Assembly Mates

Lecture

Other than coincident, what are the other basic mate commands?

Parallel, Perpendicular, Concentric, Tangent, Lock

Can a component still rotate if it has only a concentric mate?

Yes

How can you flip the alignment of a mate?

While in the mate command click on the aligned/anti-aligned buttons.

How can you edit a mate?

Right click on the mate and select "edit mate."

Where can you find a list of all of the mates in an assembly?

In the assembly manager under the paperclip icon.

How can you tell what mates are on a particular component?

Click on the component, then click on the properties tab on the top of the assembly manger.

How can you create folders to organize your mates?

Right click on the mate and select "add to folder"

Demonstration

Open the assembly file named More Assembly Mates Demo. Take a screen shot after completing the demonstration to show that it was completed correctly. Save the completed assembly.

Place Screen Shot Here

Activity

Create a new assembly file. Use the parts located in the Post and Beam folder to create the assembly. Use the Post and Beam .pdf (also in the Post and Beam folder) to help identify parts in more detail and get an overview of how the parts connect to create the assembly. Save as My Post and Beam Assembly when complete and place a screen shot below.

Place Screen Shot Here

Chapter 7

Lesson 2: Editing Components

Lecture

What is a shortcut to opening parts already present in an assembly?

Hold down the control key and click and drag on the component to copy it.

What happens when one instance of a part is edited?

All instances of that part will be edited.

What does the Edit Component command do?

Allows you to edit a part in the context of the assembly.

How do you tell between the component being edited and the rest of the assembly?

The component will be solid and the rest of the assembly will be translucent.

Demonstration

Open the assembly called Interlocking Components. Follow along with the demonstration and create a screen shot of the completed shape when you have finished.

Place Screenshot Here

Activity

Create a new part, called Component C, which will fit in between component A and B, as well as change shape when you edit either component A or B.

Challenge

Use create component to make a new part in one of the open ended design projects assigned by your teacher.

Chapter 7

Lesson 3: Sub-Assemblies & Pattern Tool

Lecture

What is it called when an assembly is placed into another assembly?

A subassembly.

How can you get a sub assembly to move and function?

Right click on the subassembly and select “component properties” in the dialogue box that comes up find the “solve as” box and set it to “Flexible.”

Can an entire part be patterned or mirrored in an assembly?

Yes.

How can you tell if a part is patterned or mirrored?

It will have a mirror or pattern icon next to it in the assembly manager.

Demonstration

Open the Linkage Machine assembly. Follow along with the demonstration and create a screen shot of the completed assembly when you have finished. Save the Assembly as Extended Linkage Machine.

Place Screenshot Here

Activity

Create a new subassembly and integrate it in to the Linkage Machine, see how you can change the motion by adding in other components. The holes and pins have a diameter of .5" and the pins have a depth of 1" with the holes having a depth of .5".

Place Screenshot Here