

Fundamentals of Drafting/SolidWorks Design

Cochranton Junior Senior High School

Applied Engineering and Technology

Course Description: This course deals with various aspects of mechanical and computer aided drafting. It offers the opportunity to learn drafting techniques used in today's industry. The first half of the class will focus on mechanical drawing aspects such as 2-D drawings, multi-view drawings, and 3-D drawings. The second half of the year will focus on computer aided drafting where students will use 3-D parametric software to design, engineer, test, and analyze different mechanical parts. This class is an excellent prerequisite for students perusing a career in mechanical, civil, or other aspects of engineering or design.

Unit Title: Introduction and Orientation

Suggested time frame: 1 weeks

Sept 1-Sept 8

Standards: 3.4.10.B4

Big Idea: Technology development has been evolutionary.

Essential Questions:

- What are some of the most primitive ways communications systems allowed us to share our ideas?
- What are your expectations for this course?
- Do you feel being able to be creative or innovative is a skill all designers must have?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none"> Describe local program and expectations, policies, and procedures. Describe local program and vocational center policies and procedures including dress code, attendance, academic requirements, discipline, shop/lab rules and regulations. Give a brief overview of the course. Explain to students what Drafting and CADD is, why it is important, and how it will be delivered. Compare and contrast local program and school policies to expectations of employers. Preview course objectives, program policy, and the industry standards 	<ul style="list-style-type: none"> -Policies -Employment -Assessment -Objectives 	<ul style="list-style-type: none"> -Assess student orientation, policy, and procedure knowledge through instructor observations and a written unit test. File the completed test to document student mastery of the school and program policies and procedures. -Have students evaluate themselves weekly using the provided rubrics for each drawing. -Have students peer review work and make corrections. 	<ul style="list-style-type: none"> -Student and Safety In Technology Education Contract - Orientation Test -Various Bellringers/Anticipatory Sets

Unit Title: Basic Fractions and Math for Drafting

Suggested time frame: 3 Class Periods

Standards: CC.2.1.HS.F.5, 3.4.10E4

Big Idea: Basic mathematic concepts are used in everyday life and no more importantly than in Fundamentals of Drafting/SolidWorks Design

Essential Questions:

- How does individual construction and architectural prints tell you information properly?
- Why do you think basic measuring skills are important to this trade and skill-set?
- How can using a simple calculation such as area relate directly to construction technologies?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• a. Define terms related to construction math.• Add, subtract, multiply, and divide whole numbers, decimals, and fractions.• Convert whole numbers to fractions and convert fractions to whole numbers.• Convert decimals to percent and convert percent to decimals.• Convert fractions to decimals.• Convert fractions to percent.• Demonstrate reading a tape measure to the nearest 16th.	<ul style="list-style-type: none">-Denominator-Numerator-Fractions-Improper Fractions-Proper Fractions-Reciprocate-Conversion-Match-Blueprint-Equivalent	<ul style="list-style-type: none">- Have students define, illustrate, and give examples of terms related to construction math. Examples should include how to use these terms in basic math principles as well as real-world situations. The terms can include but are not limited to acute angle, adjacent angles, angle, area, bisect, borrow, carry, circle, circumference, convert, cubic, decimal, degree, denominator, diagonal, diameter, difference, digit, English ruler, equilateral triangle, equivalent fractions, formula, fraction, improper fraction, invert, isosceles triangle, long division, machinist's ruler, meter, metric ruler, mixed number, negative numbers, numerator, obtuse angle, opposite angles,	<ul style="list-style-type: none">-Card Stock-Measuring pre-test-Measuring Test-Various Bellringers-Sample Blueprints-Access to various rooms within the school setting to compare blueprint difference.

		<p>percent, perimeter, pi, place value, positive numbers, radius, rectangle, remainder, right angle, right triangle, square.</p> <p>-Have students estimate their own height and the height of a partner in centimeters. Then have them pair up and measure their heights with a tape measure to see how closely they estimated.</p> <p>- Have students use sample blueprints to calculate the square footage of a house plan. As an extension, have students work as a team to use measurement tools to determine the square 52 footage of the classroom, building, cafeteria, and so forth</p>	
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<u>Unit Title:</u>	Freehand Sketching in Technical Drawing
<u>Suggested time frame:</u>	1 week (4-5 Class Periods)
<u>Standards:</u>	3.4.10.C1
<u>Big Idea:</u>	Freehand Sketching yet simple can effectively get your design or idea across to another individual
<u>Essential Questions:</u>	<ul style="list-style-type: none"> -What type of lines allow us to see the outline or edges of a drawing? -What lines are used to represent the centers or arcs, circles and used to show symmetry of a drawing? -How important is proportion in Freehand Sketching?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none"> • Explain why Freehand Sketching is Important • Identify the thickness and kinds of lines used in freehand sketching • Demonstrate the sketching of a straight line • Demonstrate the estimating of proportions • Demonstrate the sketching of an arc, a circle and an ellipse • List the steps in sketching a view 	<ul style="list-style-type: none"> -Lead Types and Identification -Freehand vs Technical -Snapping -Parallel -Perpendicular -Tangent -Concentric -Coincident -Proportion -Ellipse -Major Axis Minor Axis 	<ul style="list-style-type: none"> Have each student complete a test to identify specific tools. - Utilize Content written and performance assessments - Evaluate the scenario through teacher assessment of appropriateness. -Evaluate the selection of the proper tool for the assigned project and demonstration of its use 	<ul style="list-style-type: none"> -Heavy bonded Drafting Paper -.03, .05, .07 Pencils -Drafting Dots -Drafting Boards -PDF of Freehand Sketching Lesson -Digital Version of Freehand Sketching Lesson -

<u>Unit Title:</u>	Geometry in Technical Drawing
<u>Suggested time frame:</u>	2 weeks
<u>Standards:</u>	3.4.10.E7 & 3.4.10.E4
<u>Big Idea:</u>	Geometry in Technical Drawing in a required concept to implement in all fields of engineering

<u>Essential Questions:</u>	How would one create a perpendicular bisector to solve a technical drawing problem?
	How can creating internal and external fillets (tangent arcs) add structural strength to a part?
	-Which fields of engineering and design require knowledge of geometric construction?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none"> • Enlarge or reduce a drawing sheet by use of a diagonal • Bisect a horizontal and vertical line • Divide a line into equal parts • Bisect an angle • Draw Regular Polygons • Draw Arcs in various relationships to straight lines • Draw Ellipses • Create geometric based technical drawings implementing skills taught from lesson. 	<ul style="list-style-type: none"> -Diameter -Chord -Complementary Angles -Supplementary Angles -Trapezoid -Pentagon -Octagon -Inscribed -Circumscribed -Sphere -Hexagon -Bisect -Ellipse 	<ul style="list-style-type: none"> - Students will create geometric problems from the handouts provided on Bisecting Angles, Bisecting Lines, and Arcs, Constructing Equilateral Triangles, Creating Tangent Arcs to straight lines and other Arcs and creating Ellipses. Each day students will learn 2-3 new skills and given an anticipatory set the following day on that information until all skills are taught. At the end of the unit will produce two technical drawings that implement the skills taught in the unit from the handouts provided 	<ul style="list-style-type: none"> -Basic Technical Drawing Book (Spencer,Dygdon,Novak) -Various Bellringers/Anticipatory Sets -Guided Review Sheets -Technical Drawing Paper (Green) -Drafting Boards -Drafting Pencils -Compass -45/90 and 30/60/90 Triangles -Eraser Shields -Circle Templates (Large/Small) -Masking Tape -T-Square -PDF of End of Unit Drawings -PDF of Geometric Skills

Unit Title: Views of Objects

Suggested time frame: 2 Weeks

Standards: 3.4.10.E7 & 3.4.10.E4

Big Idea: Apply geometric concepts to model and solve real world problems.

Evaluate structure design as related to function, considering such factors as style, convenience, safety, and efficiency.

Essential Questions:

- How can a Multiview drawing assist in the manufacturing of that part?
- In what type of careers might you need to know how to read a Multiview drawing and why?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Explain the difference view of an object.• Explain how an object can be revolved to illustrate different views• Use the idea of a “glass box” to explain views• Identify the “necessary views” of a given object that is to be drawn• Identify objects that usually require only two views• Sketch three or more views of an object• Place views correctly on the page	<ul style="list-style-type: none">-Front View-Front Elevation-Right-Side View-Right-Side Elevation-Hidden Line-Visible Line-Center Line	<ul style="list-style-type: none">- Using a prebuilt and fabricated “Glass Box” out of Lexan with hinges, students will place 3D printed objects with in and sketch what the object looks like from the front, top and right view. We will unlock the box to view the views of the object in their proper orientation.-Students will construct Multiview drawings on graph paper to scale from the sketching problems using the PDF created by the teacher. Homework will be assigned daily.	<ul style="list-style-type: none">-Pitsco “Ortho Box”-Basis Technical Drawing Book (Spencer,Dygdaon,Novak)-3D Printed Objects-Expo Markers-Graph Paper-PDF Files

Unit Title: SolidWorks Essentials: Lesson 1 Basic User Interface

Suggested time frame: 5 days

Standards: 3.4.10.C1

Big Idea: Understanding how a computer-based program functions is just as important as the program itself. Navigating the program and recognizing its potential makes the tool itself more useful.

Essential Questions:

- What type of design software is SolidWorks?
- How can understanding a program like SolidWorks help you in a career after high school, secondary school, or a trade school?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Describe the key characteristics of a parametric based modeler program such as SolidWorks.• Distinguish between sketched and applied features• Identify the principal components of the SolidWorks user interface• Identify how different dimensioning methodologies convey different design intents.	<ul style="list-style-type: none">-Sketch Based-Feature Based-Feature Manager Design Tree-Command Manager-Home Screen-Keyboard Shortcuts-Mouse Gestures-Status Bar-Heads-Up Toolbar-Methodology-Potter's Wheel Approach-Manufacturing Approach-Layer Cake Approach	<ul style="list-style-type: none">- Students will navigate through SolidWorks software program recognizing and understanding each toolbar, tab and icon.- Students will download parts from SolidWorks Essentials Training Files and determine sketched based features vs. applied features.-	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

<u>Unit Title:</u>	SolidWorks Essentials: Lesson 2 Introduction to Sketching
<u>Suggested time frame:</u>	5 days
<u>Standards:</u>	3.4.10.C1
<u>Big Idea:</u>	Understanding how 3D modeling software has advanced the areas of design in respect to CAD/CAM.
<u>Essential Questions:</u>	<ul style="list-style-type: none"> -What type of design software is SolidWorks? -How can understanding a program like SolidWorks help you in a career after high school, secondary school, or a trade school?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none"> • Create a new part using SolidWorks Parametric Modeling Software • Insert a new sketch onto a specific plane within SolidWorks • Add sketch geometry to lines angles and arcs. • Establish sketch relations between pieces of geometry • Understand the state of a sketch • Extrude the sketch into a solid • Capture Design Intent 	<ul style="list-style-type: none"> -Plane -Parallel -Perpendicular -Colinear -Equal -Vertical -Horizontal -Co-Radial -Concentricity -Extruded Boss Base 	<ul style="list-style-type: none"> - Students will create basic sketched geometry using automatic relations, dimensions, added relations and equations. -Students will design parts using SolidWorks software, apply geometric relations, and extrude the sketch into a parametric solid. - Students will complete exercises to display competency in the skills of this lesson. 	<ul style="list-style-type: none"> -SolidWorks Essentials 2022 Book -SolidWorks Essentials Training Files -SolidWorks Software -3D Printing Technologies (MakerBot, Prusa, Formlabs)

<u>Unit Title:</u>	SolidWorks Essentials: Lesson 3 Basic Parts Modeling
<u>Suggested time frame:</u>	8 days
<u>Standards:</u>	3.4.10.C1
<u>Big Idea:</u>	3D parametric modeling is a skill that can not only assist you in the fields and careers of design but also fulfill necessary skills needed in the trade of manufacturing and machining.
<u>Essential Questions:</u>	<p>-What profile best represents this part and on which plane would you create this part from?</p> <p>-Why is Hole Wizard such an essential tool within SolidWorks Software?</p>

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none"> Choose the best profile for sketching Choose the proper sketch plane Extrude a sketch as a cut Create Hole Wizard holes Insert fillets on a solid Use the editing tools edit sketch, edit feature and rollback Make a basic drawing of a part Make a change to a dimension Demonstrate the associativity between the model and its drawing. 	<ul style="list-style-type: none"> -Counter Bore -Countersink -Hole -Tapped Hole -Clearance Hole -Profile -Extruded Cut -Tangent Arc -Tangency 	<ul style="list-style-type: none"> - Students will discuss the considerations that need to be made before creating a part and show the process of creating a simple one. -Students will download parts from Canvas that will test their knowledge on which profile is the best for starting a sketch. - Students will complete exercises to display competency in the skills of this lesson. -Students will be tested at this end of this lesson and must use all skills learned through lessons 1-3 in order to complete the 3D modeled part. 	<ul style="list-style-type: none"> -SolidWorks Essentials 2022 Book -SolidWorks Essentials Training Files -SolidWorks Software -3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 4 Symmetry and Draft

Suggested time frame:

10 days

Standards:

3.4.10.C1-3

Big Idea:

Manufacturing processes such as injection molding require parts to be designed using a specific process. This process should be exemplified within the design process.

Essential Questions:

- Which tools within SolidWorks software are essential in order to design a part that would need to be injection molded?
- How can end conditions used within Extruded Boss base play such a huge role in capturing design intent?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Use the view display and modifications commands• Edit the definition and parameters of a feature and regenerate the model• Use UP TO NEXT and MIDPLANE end conditions to capture design intent• Use symmetry in the sketch	<ul style="list-style-type: none">-Injection Molding-Draft-Draft Analysis-Midplane-Mirror Entities-Center Line-Normal To-Diameter-Radius-Up to Next-Up to Surface-Arc-	<ul style="list-style-type: none">- Students will discuss and create a symmetrical part that will require draft in order to be injection molded.-Students will download parts from Canvas that will test their knowledge on which profile is the best for starting a sketch.- Students will complete exercises to display competency in the skills of this lesson.-Students will be tested at this end of this lesson and must use all skills learned through lessons 1-4 in order to complete the 3D modeled part.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 5 Patterning

Suggested time frame:

5 days

Standards:

3.4.10.C1-3

Big Idea:

Patterns are the best method for creating multiple instances of one or more features when the design intent is for the features to always remain the same. Use of patterns is preferable to other methods for several different reasons.

Essential Questions:

- Which tools within SolidWorks software are essential in order to design a part that would need to be injection molded?
- How can end conditions used within Extruded Boss base play such a huge role in capturing design intent?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Create a linear pattern• Add a circular pattern• Use geometry patterns properly• Create and use the reference geometry types axes and planes• Create a mirror pattern• Use the pattern seed only option with a linear pattern• Add a sketch driven pattern• Automate the process of fully defining a sketch	<ul style="list-style-type: none">-Linear Pattern-Circular Pattern-Reference Geometry-Pattern Seed-Axes-Planes-Mirror-Sketch Driven Pattern	<ul style="list-style-type: none">- Students will create patterns of features when reuse of geometry is needed, changes can be passed onto other instances within the pattern, patterns created at the part level are reusable at the assembly level as feature driven patterns and finally when the use of smart fasteners can save time and capture the design intent of the part/assembly.- Students will complete exercises to display competency in the skills of this lesson.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 6 Revolved Features

Suggested time frame:

7 days

Standards:

3.4.10.C1-3

Big Idea:

Revolved features are a basic 3D skill that all designers and CAD operators must know how to use. Most if not all manufacturing-based parts that are made on a lathe should be designed using this type of tool to assist in the manufacturing process.

Essential Questions:

- How can first pass stress analysis using finite element analysis assist in the design of a part that must meet certain factors of safety?
- When determining the mass of an object, what other two factors must be known in order to do so?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Create revolved features• Apply special dimensioning techniques to sketches for revolved features• Use the straight slot sketch tool• Use the multibody solid technique• Create a sweep feature• Calculate the physical properties of a part• Perform rudimentary, first pass stress analysis using finite element analysis.	<ul style="list-style-type: none">-Revolved Boss Base-Multi-body Solid-Hub-Axis of Revolution-Mass Properties-Density-Volume-Finite Element Analysis-Simulation-Sweep Boss Base-3 Point Arc-Straight Slot	<ul style="list-style-type: none">- Students will create a case study part using revolved features, 3 points arcs, multibody solids, sweep boss base, and various other sketch tools.- Students will complete exercises to display competency in the skills of this lesson.-Students will be tested at the end of this lesson using the part Movie Film Reel. Students Must be all tools necessary from lessons 1-6 to do so. Students will also need to apply materials and determine mass of the object to pass their test.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 7 Shelling and Ribs

Suggested time frame:

5-8 days

Standards:

3.4.10.C1-3

Big Idea:

Shelling, ribs and draft are all tools necessary to design parts that are commonly used in the plastics industry. These tools are necessary skills needed in the manufacturing industry within our local area and region.

Essential Questions:

- How can adding a rib to a part that is to be molded or injection molded add strength to the part?
- What is draft and why is it necessary for parts to have it that are to be manufacturing using the injection molding process?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Apply draft to model faces• Perform shelling operations• Use the rib tool and determine proper surfaces in order to sketch• Create planes using faces, edges and vertices.• Create offset planes• Create thin features• Use the Draft Analysis tool in order to determine faces to apply draft to• Create full round face fillets	<ul style="list-style-type: none">-Draft Analysis-Draft-Rib-Full Round Face Fillet-Offset Plane-Thin Feature	<ul style="list-style-type: none">- Students will create a case study part using draft, draft analysis, rib, shelling, plane creation and thin features in order to display competency during the lesson.- Students will complete exercises 30-35 to display competency in the skills of this lesson.-Students will be tested at the end of this lesson. Students must be all tools necessary from lessons 1-7 to do so.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 8 Editing Tools

Suggested time frame:

7 days

Standards:

3.4.10.C1-3

Big Idea:

Nondestructive testing of a model can yield many important insights as how the model was created, the relationships that were established, and changes that can be incorporated. This section will focus on using editing tools in conjunction with rollback to “interrogate” the model.

Essential Questions:

- How can finding and repairing problems in a part be sure a key skill in solid modeling?
- What is the main difference between a Warning Marker and an Error Marker?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Diagnose various problems in a part• Repair sketch geometry problems• Use the rollback bar• Repair dangling relations and dimensions• Use the FeatureXpert to repair filleting problems• Use the FilletXpert to add fillets	<ul style="list-style-type: none">-Top Level Error-Warning Marker-Error Marker-Magnifying Glass Tool-FilletXpert-FeatureXpert	<ul style="list-style-type: none">- Students will open pre-create parts designed by SolidWorks in order to learn to tool necessary in order to fix and repair parts and sketches.- Students will complete exercises 36-41 to display competency in the skills of this lesson.-Students will be tested at the end of this lesson. Students Must be all tools necessary from lessons 1-7 to do so.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

<u>Unit Title:</u>	SolidWorks Essentials: Lesson 9 Editing Design Changes
<u>Suggested time frame:</u>	2-4 days
<u>Standards:</u>	3.4.10.C1-3
<u>Big Idea:</u>	Repairing 3D parametric models using editing tool in software such as SolidWorks is just as important as learning the design process itself.
<u>Essential Questions:</u>	<p>-How can dependencies such as parent and child within SolidWorks play such an important role in the editing of a part?</p> <p>-How can tools such as FilletXpert and SketchXpert save valuable time when designing?</p>

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none"> Understand how modeling techniques influence the ability to modify a part Utilize all the available tools to edit and makes changes to a part. Use Sketch Contours to define the shape of a feature 	<ul style="list-style-type: none"> -Rollback Bar -Edit Feature -Edit sketch plane -Reorder -Change Dimension -Edit Sketch 	<ul style="list-style-type: none"> - Students will utilize editing tools to modify the geometry and recapture design intent. Many of the most used editing commands: Edit Sketch, Edit Feature, edit sketch plane, Reorder, Rollback and Change dimensions are used in this lessons case study. - Students will complete exercises 42-45 to display competency in the skills of this lesson. 	<ul style="list-style-type: none"> -SolidWorks Essentials 2022 Book -SolidWorks Essentials Training Files -SolidWorks Software -3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 10 Configurations

Suggested time frame:

4-7 days

Standards:

3.4.10.C1-3

Big Idea:

Configurations allow the designer to represent more than one version of a part within a single SolidWorks file. This tool allows the user/designer to save time and file space therefore making the process more streamlined.

Essential Questions:

- How can equations within a part allow the part to capture the design intent of the part itself?
- Utilizing the design library can be an extremely useful tool within a company in order to maximize and streamline their parts, fasteners and assemblies. How so?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Use global variables to tie values together• Create equations• Use configurations to represent different versions of a part within a single SOLIDWORKS file• Suppress and unsuppress features• Change dimension values by configuration• Suppress features by configuration• Understand the ramifications of making changes to parts that have configurations• Use the design library to insert features into a part	<ul style="list-style-type: none">-Suppress-Unsuppress-Configuration-Design Library-Task Pane-Equations-Global Variables-Equalities-Design Tables	<ul style="list-style-type: none">- Students will utilize configurations to represent more than one version of the part in the same file. They will do so by creating a case study-based part and editing downloaded SolidWorks parts to learn the skills for this lesson.- Students will complete exercises 46-50 to display competency in the skills of this lesson.-Student will create their own version of a part with multiple configurations for their test at the end of this unit.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 11 Creating Drawings

Suggested time frame:

4-5 days

Standards:

3.4.10.C1-3

Big Idea:

Drawings of parts and assemblies are the most important part of the design and manufacturing process. Without a detailed drawing detailing all components of the part, sizes, tolerances and finishes, the part would be impossible to manufacture and produce.

Essential Questions:

- Which type of view allows the user to display the inner workings of a part to see features that one would not be able to see from any standard view?
- Why are tolerances so important on a drawing when the part is to be machined?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Create several type of drawing views• Modify drawing views by alignment and tangent edges• Add annotations to a drawing	<ul style="list-style-type: none">-Model View-Section View-Detail View-Annotations-Projected View-Break View-Tangent Edges-Sheet Formats-Datum Feature Symbols-Surface Finish Symbols-Centerlines-Geometric Tolerances	<ul style="list-style-type: none">- Students will create varying views of the downloaded part Ratchet Body from SolidWorks Essentials. Views that will be created in the Ratchet Body drawing will be tested on the exercises at the end of the lesson- Students will complete exercises 51-53 to display competency in the skills of this lesson.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)

Unit Title:

SolidWorks Essentials: Lesson 12 Bottom-Up Assembly Modeling

Suggested time frame:

5-7 days

Standards:

3.4.10.C1-3

Big Idea:

Creating assemblies allows the user or designer to check for things that you cannot generally check for in 2D drawings or 2D creation. Collisions, interferences, and gaps with parts can all be detected using 3D Modeling Software.

Essential Questions:

- How can the state of the component determine the movement along its axis of movement?
- Why are tolerances so important on a drawing when the part is to be machined?

Competency	Vocabulary	Strategy	Resource
<ul style="list-style-type: none">• Create a new assembly• Insert components into an assembly using all available techniques• Add mating relationships between components• Insert sub-assemblies• Utilize the assembly-specific aspects of the FeatureManager design tree to manipulate and manage the assembly	<ul style="list-style-type: none">-Assembly-Sub-Assembly-Mate-Inset Component-Smart Mate-Design Library-Rotate-Move-Fixed Component-Degrees of Freedom-Width-Distance-Coincident-Concentric-Tangent-Pack and Go	<ul style="list-style-type: none">- Students in this lesson will create two subassemblies of a Universal Joint. Once both sub-assemblies are complete a main assembly will be created allowing the Universal Joint to move freely.- Students will complete exercises 54-58 to display competency in the skills of this lesson.	<ul style="list-style-type: none">-SolidWorks Essentials 2022 Book-SolidWorks Essentials Training Files-SolidWorks Software-3D Printing Technologies (MakerBot, Prusa, Formlabs)