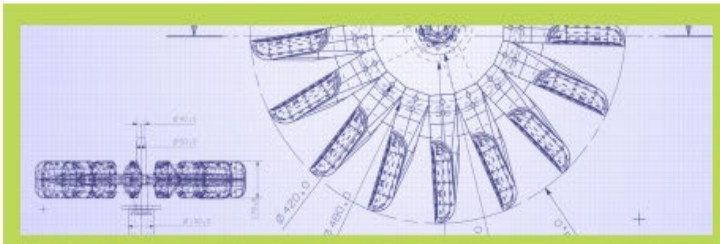
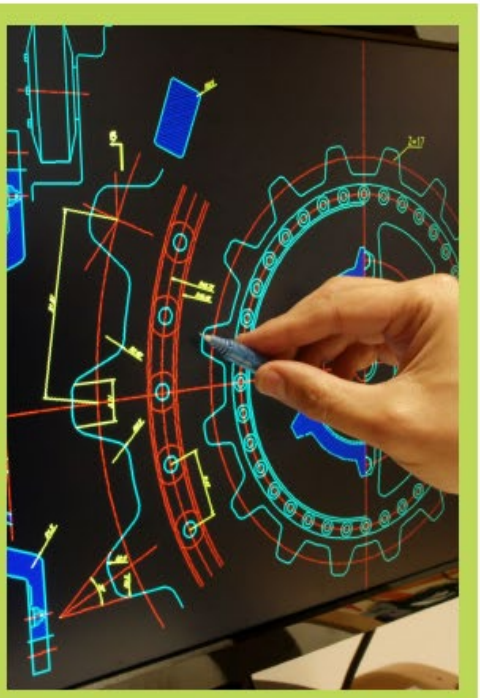
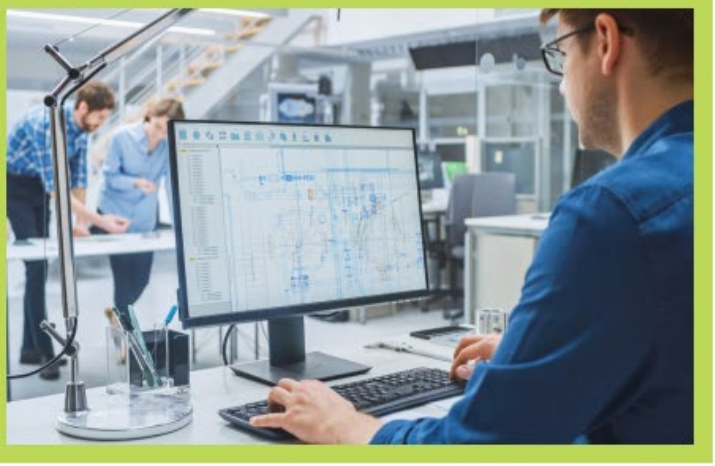
Drafting II - Engineering

TRADE, TECHNOLOGY, ENGINEERING, AND INDUSTRIAL EDUCATION | Career and Technical Education

IV22







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Introduction

This curriculum guide for IV22 Drafting II - Engineering, was developed to assist teachers in preparing students to meet the North Carolina State Board of Education's guiding vision, "Every public school student in North Carolina will be empowered to accept academic challenges, prepared to pursue their chosen path after graduating high school, and encouraged to become lifelong learners with the capacity to engage in a globally-collaborative society." This course is based on state and national content standards and it is rigorous and relevant. Business and industry representatives reviewed the standards and provided input on the content for this course as one that helps to prepare students for high-skill, high-wage, or in demand occupational opportunities. It also infuses technology and active learning tools throughout the curriculum to teach today's generation of students. The CTE Course Management System includes the course standards and information, the career pathway, and equipment list. As presented in the course essential standards 1.00-6.00, an understanding of Engineering Drafting Concepts is covered including the following topics.

1.00 Understand Basic Engineering Design and Industry Terms

2.00 Understand Engineering Design Concepts and Principles to Solve Problems

3.00 Understand Manufacturing Processes

4.00 Apply Parametric-Solid Modeling Techniques to create a 3D Model

5.00 Understand Conventional Dimensioning & Tolerancing

6.00 Understand Procedures to Create Working Drawings of a 3D Model

Aligned to the course standards and each indicator, this guide contains a culminating question, essential questions, unpacked content, resources, instructional activities and additional textbook and online resources as needed. It incorporates and enhances appropriate content outlined in the North Carolina Standard Course of Study. The proof-of-learning will be either a 100-item multiple choice post-assessment at the standard level and administered through the NC

Instructional Management System or an obtained Industry Credential (Autodesk Certified User-Inventor or Certified SolidWorks Associate).

Culminating Question

This question is central to the purpose of the standard. It requires students to think about the knowledge that will be learned.

Essential Questions

Essential questions are used to guide students' learning and are geared toward uncovering a topic. All essential questions for this course are derived directly from the unpacked content.

Unpacked Content & Resources

The unpacked content comes from the objectives listed on the course blueprint. Autodesk and SolidWorks provide online resources used to develop the unpacked content specific to the software provided by each PSU.

Instructional Activities

Individual and group activities will be listed in the Instructional Activities section. Instructional activities reflect "best practice" as determined by highly qualified and successful teachers. The activities follow the unpacked content that is designed to build understanding of the indicator.

Vocabulary/Content Literacy Terminology

There are a variety of research-based activities that effectively introduce and reinforce vocabulary for any subject. This course provides instructional flexibility to utilize any tool that achieves the intended result which is to understand and recall key terms necessary for further development of this objective. When learning a new vocabulary, it is critical that students know how to recognize/read a word, spell the word, define it and obtain a visual clue for context. Sample activities are provided in the file below. Also provided in this guide are the Content Literacy Terminology for each indicator.


Vocabulary Activity Options.docx

Guest Speakers, Virtual Field Trips & Field Experiences

Industry involvement is critical for a deeper student understanding of content/concepts. Educators can help participants receive the most from these visits by preparing for the visit, having participants take notes during the visit, and then reflecting on the visit. These types of activities are not limited to just one standard/objective.

Additional Resources

Textbook & Online

Autodesk and SolidWorks provide content; previous Curriculum Guides for course also provided content. Additional textbooks can be selected by individual PSUs for content; however, no specific textbook was referenced for this Guide. Referenced websites are functional as of the publication date of this curriculum guide. No guarantee can be made as to the continued functionality, but a generic internet search may yield additional resources and websites.

Curriculum Projects

Incorporate hands-on projects that become the instructional method through which students acquire understanding of the content. Students may address these learning outcomes simultaneously, rather than in the sequential manner occurring in traditional courses. The learning outcomes; therefore, are not specified for coverage during a specific week of the semester but are tied to projects and can be acquired at any point. For more information on successful projects visit AutoCAD Resource, SolidWorks Resource and/or the shared Moodle for the course.

CTSO

This course emphasizes Career and Technical Student Organization (CTSO) competitive events for SkillsUSA; however these were not directly written into Curriculum Guide. For more information on SkillsUSA visit: https://www.skillsusa.org/.

Acknowledgements

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Course	**IV22 Drafting II - Engineering**			
Essential Standard	1.00	B2	5%	Understand Basic Engineering Design and Industry Terms.
Indicator	1.01	N/A	N/A	Understand the concept of Engineering Design.
Culminating Question	**What is engineering design?** ● What part does engineering design play in Industry?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the concept of engineering design.

INSTRUCTIONAL ACTIVITIES- 1.01	
A. Content Literacy Terminology	
Resource(s)	**(See 1.01.1)**
B. Understand the concept of engineering Design.	
Activity	Watch, Think, Write-Engineering Design Introduction

Teacher Instructions	• Facilitate accessing prior knowledge by asking students to call out or write on board/paper words they associate with “Engineering” & “Design.”
	• Facilitate “Watch, Think, Write” Activity.
	  • *Watch:* Select video ahead of time for students or allow them to select individually. Multiple videos can be used for the whole class or small groups. Pairs should have the same video. Determine groups/pairs ahead of time if needed.
	  • *Think:* Facilitate discussion (as a whole class, in pairs or small groups) of videos/segment(s). No writing allowed. The time given for this can be adjusted depending on class and content.
	  • *Write:* Students should write independently (prompts can be given as needed).
	  • *Share:* Facilitate students sharing their Reviews. This does not have to be between students who were in the same group/pair as before.
	• Review indicator definitions with students. Facilitate full group discussion of follow-up questions.

Student Directions	• Call out or write on board/paper words you associate with “Engineering” & “Design.”
	• Participate in “Watch, Think, Write” Activity.
	  • *Watch:* Watch selected video or select a video.
	  • *Think:* Discuss (as a whole class, in pairs or small groups) the videos/segment(s). No writing allowed.
	  • *Write:* Write a short video review (4-5 sentences) highlighting the important facts.
	  • *Share:* Share your review.
	• Participate in review of definitions and discussion of follow-up questions.

Resource	 Watch, Think, Write, Share- Engine
Website Resources- Example Videos	
YouTube. (2021). Engineering Design Process.	https://www.youtube.com/watch?v=b0ISWaNoz-c&t=12s
YouTube. (2021). The Engineering Design Process: A Taco Party.	https://www.youtube.com/watch?time_continue=3&v=MAhpfFt_mWM&feature=emb_logo

YouTube. (2021). The Engineering Design Process.	https://www.youtube.com/watch?v=oBqGoXCBHtk

Content Literacy Terminology-1.01.1	
Autodesk	Umbrella company which creates 2D/3D design, engineering, and entertainment software such as AutoCAD, Revit, and Inventor.
Autodesk Inventor	CAD software for professional-grade 3D mechanical design, documentation, and product simulation tools.
Engineering	The branch of science and technology concerned with the design, building, and use of engines, machines, and structures.
Engineering Design	The method that engineers use to identify and solve problems.
Manufacturing	The production of products/articles on a large scale, using labor and machinery, tools, and chemical or biological processing.

Course	IV22 Drafting II - Engineering			
Essential Standard	1.00	B2	5%	Understand Basic Engineering Design and Industry Terms.
Indicator	1.02	N/A	N/A	Understand Career Opportunities related to Engineering Design: ● Engineering Services: Civil, Electrical, Mechanical ● Machine Tool Designers ● Marine Engineering Services ● Petroleum Engineering services ● Aerospace Engineering ● Industrial Design/Product Design ● New and Emerging Technologies
Culminating Question **Essential Questions**	**What career opportunities are related to engineering design?** ● What are your interests and how do they relate to the Drafting career cluster- engineering? ● What pathways are available to obtain a career in Drafting? ● What career opportunities are available in engineering design?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Identify personal interests and their relationship to the Drafting career cluster- engineering.
c. Identify pathways available to obtain a career.
d. Understand various career opportunities related to engineering design.

INSTRUCTIONAL ACTIVITIES- 1.02	
A. Content Literacy Terminology	
Resource(s)	**(See 1.02.1)**
B. Identify personal interests and their relationship to the Drafting career cluster. **C. Identify pathways available to obtain a career.** **D. Understand various career opportunities related to engineering design.** *Note: Activity includes all Unpacked Content.*	
Activity	Interest Identification-Career Pathway Opportunities
Teacher Instructions	● Select an online Career Assessment link to share with students ahead of time. ● Provide digital or hard copy access to *Engineering & Manufacturing Employment-Career Exploration*. ● Facilitate student progress on assessment. ● Facilitate students' selection of career from list in Part Two or assign. Variety amongst all students is recommended. ● Facilitate creation of pamphlet for assigned/selected career in software such as Microsoft Publisher. Review and/or provide rubric for understanding of expectations. ● Facilitate presentations of information in class Pamphlets can be printed and shared with other students outside of class for program promotion as well. ● Facilitate students developing short slide presentations to accompany and/or enhance pamphlets. ● Facilitate completion of peer and/or personal review of work. Provide teacher feedback via rubric.
Student Directions	● Complete an online career assessment. ● Select a career to expand on and identify possible career pathways related to Engineering and/or Manufacturing. ● Create a pamphlet for career in software such as Microsoft Publisher following guidelines in rubric. ● Develop a short slide presentation with information. ● Share your pamphlet with the class. ● Complete peer and/or personal review of work.
Resource(s)	 Engineering & Manufacturing Emp

Content Literacy Terminology- 1.02.1	
Aerospace Engineering	Design primarily aircraft, spacecraft, satellites, and missiles. In addition, they test prototypes to make sure that they function according to design. concerned with the development of aircraft and spacecraft. It has two major and overlapping branches: aeronautical engineering and astronautical engineering.
Civil Engineer	The oldest engineering profession according to some sources they design structures, environmental systems, and various construction projects and may do analysis and design for materials and structural systems for buildings, aircraft, etc.
Electrical Engineer	Design, develop, test, and supervise the manufacturing of electrical equipment, such as electric motors, radar and navigation systems, communications systems, and power generation equipment. Electronics engineers design and develop electronic equipment, such as broadcast and communications systems—from portable music players to global positioning systems (GPSs).
Industrial Design	Develop the concepts for manufactured products, such as cars, home appliances, and toys. They combine art, business, and engineering to make products that people use every day. Industrial designers consider the function, aesthetics, production costs, and the usability of products when developing new product concepts.
Machine Tool Designers	Also known as “Machine Tool Builder” a corporation or person that builds machine tools, usually for sale to manufacturers, who use them to manufacture products. A machine tool builder runs a machine factory, which is part of the machine industry.
Marine Engineering Services	The discipline of applying engineering sciences, including mechanical engineering, electrical engineering, electronic engineering, and computer science, to the development, design, operation and maintenance of watercraft propulsion and on-board systems and oceanographic technology. It includes but is not limited to power and propulsion plants, machinery, piping, automation and control systems for marine vehicles of any kind, such as surface ships and submarines. This includes the engineering of boats, ships, oil rigs and any other marine vessel or structure, as well as oceanographic engineering.
Mechanical Engineer	Comparable to general engineering (with more emphasis on creative abilities of the drafter: creativity, ingenuity and technical knowledge) they work from sketches or just a memo describing a new product idea and determine how or if ideas might work and provide accurate drawings and specifications for proposed products

Petroleum Engineering Services	Design and develop methods for extracting oil and gas from deposits below the Earth's surface. Petroleum engineers also find new ways to extract oil and gas from older wells. Petroleum engineers divide themselves into several types such as the following: • Reservoir engineers work to optimize production of oil and gas via proper placement, production rates, and enhanced oil recovery techniques. • Drilling engineers manage the technical aspects of drilling exploratory, production and injection wells. • Production engineers, including subsurface engineers, manage the interface between the reservoir and the well, including perforations, sand control, downhole flow control, and downhole monitoring equipment; evaluate artificial lift methods; and select surface equipment that separates the produced fluids (oil, natural gas, and water).
Product Design	Create a range of items, from everyday products, such as mobile phones, household appliances and cars; to larger items, such as industrial tools, equipment, and machinery. A Product Designer might work on new products or improve existing ones, and use the understanding of technology, materials and manufacturing methods to improve the design and usability of an item.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	1.00	B2	5%	Understand Basic Engineering Design and Industry Terms.
Indicator	1.03	N/A	N/A	Understand terms related to Engineering Design and Manufacturing: ● Design Iterations/Design Intent ● Conceptual Development ● 3D Laser Scanning ● Reverse Engineering ● Parametric Modeling ● Finite Element Analysis ● Rapid Prototyping/Additive Manufacturing ● Ergonomics
Culminating Question **Essential Questions**	**What are some of the terms associated with engineering design and manufacturing?** ● How are the major terms and concepts associated with engineering design and manufacturing connected? ● How are the concepts and trends related to engineering design and manufacturing used in industry?			

UNPACKED CONTENT

a. Content Literacy Terminology
b. Understand terms related to engineering design and manufacturing.
c. Understand concepts related to engineering design and manufacturing and how they apply to industry.

INSTRUCTIONAL ACTIVITIES- 1.03	
A. Content Literacy Terminology	
Resource(s)	**(See 1.03.1)**
B. Understand terms related to engineering design and manufacturing. **C. Understand concepts related to Drafting career cluster and how they apply to industry.** *Note: Activity includes all Unpacked Content.*	
Activity	Independent Exploration of Topics
Teacher Instructions	● Provide a digital or hard copy of *What Did You Learn?* ● Facilitate student time to explore and read about assigned concept(s) individually. Facilitate students watching videos, reading articles and visiting websites provided by Instructor (selected links to also be provided ahead of time.) Facilitate student completion of *What Did You Learn?* ● Facilitate whole-class or small group discussion. *Note:* This activity can also be done as a JigSaw activity with homogenous small groups "specializing" in certain topics and then returning to teach the whole class or heterogeneous groups about what they learned.
Student Directions	● Explore assigned concept(s) by watching videos, reading articles and visiting websites provided ● Fill out *What Did You Learn?* ● Participate in whole-class or small group discussion.
Resource	What Did You Learn.docx
Website Resources	
Design Iterations/Design Intent Example: INTERACTIONDESIGN FOUNDATION. (2021). Design Interaction. What is Design Interaction?	https://www.interaction-design.org/literature/topics/design-iteration
Design Iterations/Design Intent Example: YouTube. (2021). Interactive Design in Action.	https://www.youtube.com/watch?v=Rnsk5lA52ps
Conceptual Development Example: PDF. (2021).	https://appinventor.mit.edu/explore/sites/all/files/teachingappcreation/unit1/DesignUnit1.pdf

Introduction to Design and the Concept Development Process.	
Conceptual Development Example: YouTube. (2021). Design – Concept Development.	https://www.youtube.com/watch?v=Lq-Uly0iPTw
3D Laser Scanning Example: LASERDESIGN. (2021). 3D Scanning Technology — Hard Work That Looks Like "Magic."	https://www.laserdesign.com/what-is-3d-scanning
3D Laser Scanning: Example: YouTube. (2021). 3D scanner. How it works?	https://www.youtube.com/watch?v=p33qxgdEwVc
Reverse Engineering Example: PHYSICALDIGITAL 3D Optical Measurement. (2021). What is Reverse Engineering?	https://physicaldigital.com/what-is-reverse-engineering/
Reverse Engineering Example: YouTube. (2021). What is Reverse Engineering?	https://www.youtube.com/watch?v=CjMAMzke7nw
Parametric Modeling Example: Design World (2020) What is Parametric Modeling?	https://www.designworldonline.com/what-is-parametric-modeling/
Parametric Modeling	https://www.youtube.com/watch?v=uSEzWfGgP2Y

Example: YouTube. (2021). What is Parametric Modeling? - Introduction to Parametric Modeling.	
Finite Element Analysis Example: MANOR Tool & Manufacturing Company. (2021). The Benefits of Finite Element Analysis in Manufacturing.	https://www.manortool.com/finite-element-analysis
Finite Element Analysis Example: YouTube. (2021). What is Finite Element Analysis? FEA explained for beginners.	https://www.youtube.com/watch?v=boSLQYhDXoE
Rapid Prototyping/ Additive Manufacturing Example: SYNECTIC Product Development. (2019). What is Rapid Prototyping? A Rapid Prototyping FAQ.	https://synectic.net/what-is-rapid-prototyping/
Rapid Prototyping/ Additive Manufacturing Example: EMPIRE GROUP. (2021). 3D Printing vs. Rapid Prototyping vs. Additive Manufacturing - What's The Difference?	https://www.empiregroupusa.com/empire-group-blog/whats-the-difference-between-3d-printing-vs-rapid-prototyping-vs-additive-manufacturing
Rapid Prototyping/ Additive Manufacturing	https://www.ge.com/additive/additive-manufacturing

Example: GE Additive. (2021). What is Additive Manufacturing?	
Rapid Prototyping/ Additive Manufacturing Example: Fractory. (2021). Engineering Blog. What Is Rapid Prototyping?	https://fractory.com/what-is-rapid-prototyping/
Ergonomics Example: Humanscale. (2021). What is Ergonomics?	https://www.humanscale.com/ergonomics/what-is-ergonomics/index.cfm
Ergonomics Example: YouTube. (2021). ERGONOMICS IN MANUFACTURING.	https://www.youtube.com/watch?v=3_wBB_zNz0c

Content Literacy Terminology- 1.03.1	
3D laser scanning	A non-contact, non-destructive technology that digitally captures the shape of physical objects using a line of laser light. 3D laser scanning is a way to capture a physical object's exact size and shape into the computer world as a digital 3-dimensional representation.
Additive Manufacturing	The technologies that build 3D objects by *adding* layer-upon-layer of material, whether the material is plastic, metal, concrete.
Conceptual Development	A set of activities that are carried out early in the systems engineering life cycle to collect and prioritize operational needs and challenges, develop alternative concepts to meet the needs, and select a preferred one as the basis for subsequent system or capability development and implementation.
Design intent	Used to describe how the model should be created and how it should behave when it is changed. Not just about the size and shape of features, but includes tolerances, manufacturing processes, relationship between features, dimensions, and the use of equations.
Design Iterations	Methodology based on a cyclic process of prototyping, testing, analyzing, and refining a product or process.
Engineering	The branch of science and technology concerned with the design, building, and use of engines, machines, and structures.
Engineering Analysis	To break down an object, system, problem, or issue into its basic elements to get at its essential features and their relationships to each other and to external elements.
Engineering Design	The method that engineers use to identify and solve problems.
Ergonomics	The process of designing or arranging workplaces, products and systems so that they fit the people who use them. Applies to the design of anything that involves people–workspaces, sports and leisure, health and safety.
Finite Element Analysis	A numerical method for solving problems of engineering and mathematical physics. FEA is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements.
Industrial Design	Develop the concepts for manufactured products, such as cars, home appliances, and toys. They combine art, business, and engineering to make products that people use every day. Industrial designers consider the function, aesthetics, production costs, and the usability of products when developing new product concepts.
Manufacturing	The production of products/articles on a large scale, using labor and machinery, tools, and chemical or biological processing.
Parametric Modeling	Uses the computer to design objects or systems that model component attributes with real world behavior. Parametric modeling allows designers to define entire shapes, not just specific parts.
Product Design	Create a range of items, from everyday products, such as mobile phones, household appliances and cars; to larger items, such as industrial tools, equipment, and machinery. A Product Designer might work on new products or improve existing ones, and use the understanding of technology, materials, and manufacturing methods to improve the design and usability of an item.

Rapid Prototyping	A group of techniques used to quickly fabricate a scale model of a physical part or assembly using three-dimensional computer aided design. It is used to test the efficiency of a part or product design before it is manufactured in larger quantities.
Reverse Engineering	Taking apart an object to see how it works in order to duplicate or enhance the object. To disassemble and examine or analyze in detail (a product or device) to discover the concepts involved in manufacture usually in order to produce something similar.

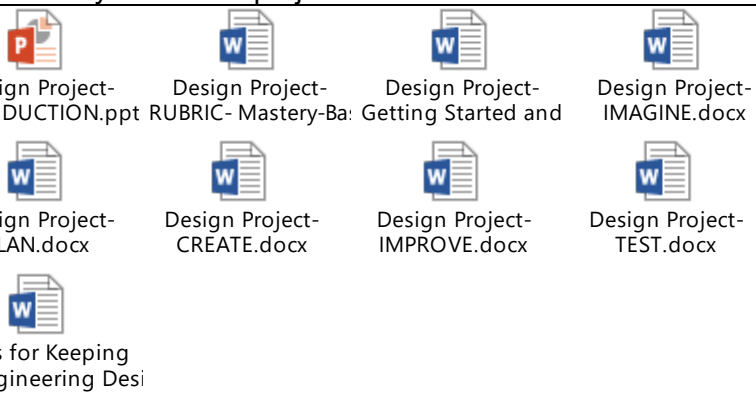
Course	IV22 Drafting II - Engineering			
Essential Standard	2.00	B2	5%	Understand Engineering Design Concepts and Principles to Solve Problems.
Indicator	2.01	N/A	N/A	Understand the concepts and principles of the engineering design process.
Culminating Question **Essential Question**	**What are the concepts and principles of the engineering design process?** ● How is the engineering design process used to solve a design problem?			

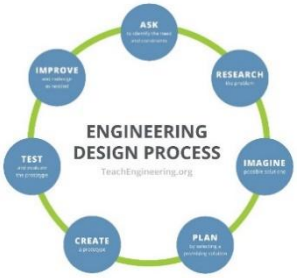
UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply the concepts and principles of the engineering design process.

INSTRUCTIONAL ACTIVITIES- 2.01	
A. Content Literacy Terminology	
Resource(s)	**(See 2.01.1)**
B. Apply the concept and principles of the engineering design process.	
Activity	Individual or Small Group Design Project-Engineering Design Process Introduction. *Note: Possible combination with creating and maintaining an Engineering Notebook (Indicator 2.02.)*
Teacher Instructions	● Share Design Project Files with students as they progress through activity. Activity best suited after Indicator 1.01 Activity-*Watch, Think, Write- Engineering Design Introduction.* File *Design Project-INTRODUCTION* PowerPoint Presentation to be used as daily guide and for Introduction. ● *Introduction:* Assign a design problem to students (i.e., 10-piece Lego set, grill tool, kitchen gadget.) Review and/or provide rubric (*Design Project-Rubric*) for understanding of expectations as well as *Design Project-INTRODUCTION* PowerPoint Presentation as an overview of project. Students can begin creating their website (*Design Project-Getting Started and ASK.RESEARCH.)* ● *Ask & Research:* Facilitate student research using *Design Project-Getting Started and ASK.RESEARCH.* Students will compile research in a document and transfer to websites. ● *Imagine:* Facilitate student creation of multiple design ideas and sketches. Students will take pictures of design sketches and transfer to websites with short written descriptions. Resource: *Design Project-IMAGINE* ● *Plan:* Facilitate students analyzing each idea. Students will document analysis and transfer to websites with short written descriptions. Resource: *Design Project-IMAGINE.* Review with students and guide students to selection of final design idea. Student to complete a detailed design drawing of the final idea (this can be completed in isometric, oblique, or as a Multiview). Students will take pictures of final detail design drawing and transfer to websites with short written description(s). ● *Create:* Facilitate students creating a 3D model of all parts, needed assemblies, a set of basic working drawings, and a presentation while keeping a log of work time/progress (see *Design Project-CREATE*.) ● *Test*: Facilitate students' creation of a physical model. Emphasize how this would be different in Industry (materials, methods, etc.) Students will take a picture of the model and transfer to websites with a short written description along with answered questions from *Design Project-TEST*. ● *Improve:* Facilitate students writing a flection on their design and the process using *Design Process-IMPROVE* as a guide. ● Facilitate students presenting their overall projects. *Notes:* This Activity is designed to span multiple class periods and other standards. Create, Test, & Improve are designed to be completed once students have a good understanding of software/content. Students can

	be required to keep an Engineering Notebook common to Industry Standards to document the process.
Student Directions	● *Introduction:* Review assigned Design problem and provided rubric (*Design Project-Rubric*) for understanding of expectations, as well as *Design Project-INTRODUCTION* PowerPoint Presentation. Begin creating your website (*Design Project-Getting Started and ASK.RESEARCH.)* ● *Ask & Research:* Research using *Design Project-Getting Started and ASK.RESEARCH.* Compile research in a document and transfer to your website. ● *Imagine:* Create multiple design ideas and sketches *(*see *Design Project-IMAGINE*). Take pictures of design sketches and transfer to your website with short written descriptions. ● *Plan:* Analyze each idea. Complete document analysis and transfer to your website with short written descriptions (see *Design Project-IMAGINE).* Complete a detailed design drawing of your final idea (this can be completed in isometric, oblique, or as a Multiview). Take pictures of final detail design drawing and transfer to your website with short written description(s).

	● *Create:* Create a 3D model of all parts, needed assemblies, a set of basic working drawings and a presentation while keeping a log of work time/progress (see *Design Project-CREATE*). ● *Test*: Create a physical model. Take pictures of the model and transfer to your website with a short written description along with answered questions from *Design Project-TEST*. ● *Improve:* Write a reflection on your design and the process using *Design Process-IMPROVE* as a guide. Add this to your website. ● Present your overall project
Resource(s)	

Content Literacy Terminology- 2.01.1	
Brainstorming	Technique that allows teams to tap into all the expertise of a group in order to develop the most successful solution to a design challenge.
Engineering Design Process	A series of steps that guide engineering teams as they solve problems. Step 1: Ask: Identify the need and constraints. Step 2: Research the problem. Step 3: Imagine: Develop possible solutions. Step 4: Plan: Select a promising solution. Step 5: Create: Build a prototype. Step 6: Test and evaluate prototype. Step 7: Improve: Redesign as needed.  Resource: https://www.teachengineering.org/design/designprocess
Engineering Notebook	A bound design notebook that can be used to reconstruct your work, even years after the original project. Other engineers should be able to use the notebook to reconstruct your work. The notebook will be used to determine the rightful owner of patents and other proprietary ideas.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	2.00	B2	5%	Understand Engineering Design Concepts and Principles to Solve Problems.
Indicator	2.02	N/A	N/A	Understand the concepts to create an Engineering Notebook. Maintain for all class projects.
Culminating Question **Essential Question**	**What is the purpose of an Engineering Design Notebook?** ● What are the procedures for keeping and maintaining an Engineering Notebook?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Create and Maintain an Engineering Notebook.

INSTRUCTIONAL ACTIVITIES- 2.02	
A. Content Literacy Terminology	
Resource(s)	**(See 2.02.1)**
B. Create and maintain an Engineering Notebook.	
Activity	Engineering Design Notebook Application *Note:* Possible combination with applying the concepts and principles of the Engineering Design Process. *(Indicator 2.01)*
Teacher Instructions	● Provide hard copy (and or digital copy) of *Rules for Keeping an Engineering Design Notebook* to students. Facilitate review, discussion, and reading of rules as a whole-class. ● Facilitate students keeping notebooks for project(s).
Student Directions	● Participate in review, discussion, and reading of rules as a whole-class. ● Keep a notebook for project(s).
Resource(s)	 Rules for Keeping an Engineering Desi

Content Literacy Terminology- 2.02.1	
Engineering Notebook	A bound design notebook that can be used to reconstruct your work, years after you have completed the original project. Other engineers should be able to use the notebook to reconstruct your work. The notebook will be used to determine the rightful owner of patents and other proprietary ideas.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	3.00	B2	15%	Understand Manufacturing Processes.
Indicator	3.01	N/A	N/A	Understand the various manufacturing processes that can be used to produce a part.
Culminating Question **Essential Questions**	**What are some manufacturing processes which can be used to produce a part?** ● Why is it important to understand various manufacturing processes and materials? ● What are the major phases of manufacturing? ● What major classifications of materials are used to manufacture a part?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Recognize the importance of understanding various manufacturing processes and materials.
c. Identify the phases of the manufacturing process.
d. Identify various types of materials used in manufacturing.
e. Understand various manufacturing processes used to create a part.

INSTRUCTIONAL ACTIVITIES- 3.01	
A. Content Literacy Terminology	
Resource(s)	**(See 3.01.1)**
B. Recognize the importance of understanding various manufacturing processes and materials. **C. Identify the phases of the manufacturing process.** **D. Identify various types of materials used in manufacturing.** **E. Understand various manufacturing processes used to create a part.** *Note: Activity includes all Unpacked Content for Indicator and combines with Indicator 3.05.*	
Activity	Manufacturing Processes Introduction
Teacher Instructions	● Provide students a hard copy of *Manufacturing Processes- Introduction-Presentation- Guided Notes.* Facilitate whole-class guided instruction using *Manufacturing Processes- Introduction-Presentation* while students fill in notes. ● Provide digital copy of *Manufacturing Processes- Introduction-Presentation* for reference while students answer leading questions *Manufacturing Processes-Introduction-Leading Questions*. ● Break students into small groups or pairs and facilitate student led discussion on individual answers to leading questions.
Student Directions	● Participate in whole-class guided instruction while filling in guided notes. ● Answer leading questions using the provided PowerPoint Presentation. ● Demonstrate your understanding of major terms/concepts associated with manufacturing processes by discussing answers to leading questions in small groups or pairs.
Resource(s)	Manufacturing Processes- Introduct Manufacturing Processes- Introduct Manufacturing Processes- Introduct Manufacturing Processes- Introduct

Content Literacy Terminology-3.01.1	
Flat Pattern	Also known as a stretch-out, the shape of the sheet metal part before it is formed that show bend lines, bend zones, punch locations, and the shape of the entire part flattened.
Annealing	The process generally used to soften metal by heating followed by slow cooling.
Assembly	The process where various parts are put together to complete the product.
Blind Hole	A hole that cuts into but does not pass completely through the object
Blow Molding	Plastic manufacturing process used in the production of hollow products such as bottles and containers. The molten plastic enters around a tube that also forces air inside the material, which forces it against the interior surface of the mold to create the shape desired.
Boring	Finishing process in which a cutting tool enlarges a hole to a more accurate size.
Casting	Rough forming metal process of producing a part by pouring molten metal into a mold consisting of damp sand (sand casting) or cavities between metal dies (die casting).
CNC	(Computer Numerical Control) Controlling and operating machines by means of preprogrammed, coded, tool-cutting instructions.
Coating	Finishing process in which a covering is applied to a surface.
Counterbore	To enlarge the end of a drill hole to a specific diameter and depth in order to recess a mating part.
Countersink	To recess a hole with a conically (cone) shaped tool to provide a seat for flat head screws.
Deburring	Finishing process that removes built-up burrs and other imperfections from a finished surface.
Die Casting	Manufacturing process where malleable material is forced by high pressure into a mold cavity. It results in increased surface quality of the product.
Drilling	The process of cutting a cylindrical hole into an object.
Extrusion	Plastic manufacturing process used to make continuous shapes such as moldings, tubing, bars, water hose, weather stripping, and any part that has a constant shape. This process creates the desired continuous shape by forcing molten plastic through a metal die.
Finishing	Machining processes used to increase accuracy of a part or surface treating such as grinding, milling, polishing, and deburring.
Folding	Manufacturing process in which sheet metal is bent into specific shapes.
Forging	Rough forming metal process of heating it in a fire or furnace and striking or hammering it.
Hardening	The process which requires heating and then rapid cooling in oil or water.
Heat Treating	The process of changing the properties of metals by heating and cooling.
Injection Molding	Most used plastic manufacturing process for creating thermoplastic products. The process involves injecting molten plastic material into a mold that is in the form of the desired shape. Injection molding is

	used to create products such as housings for electronic implements, automotive interior components, food storage containers, and components for medical applications.
Inorganic materials	Materials which do not contain carbon (non-living sources). Examples used in manufacturing include, but are not limited to rocks/minerals, glass, ceramics, and metals.
Machining	Any of various processes in which a piece of raw material is cut into a desired final shape and size by a machine.
Manufacturing Materials	Materials for production are divided into three general categories: metal, plastic, and inorganic materials.
Metal	Ferrous (iron), nonferrous, and alloy (mixture) material characterized by high electrical and thermal conductivity as well as by malleability, ductility, and high reflectivity of light.
Milling	Finishing process using rotary cutters to remove material.
Plastic	Synthetic or semi-synthetic organic compounds (polymers) characterized by being generally low cost, durable, strong for their weight, electrically and thermally insulative, and resistant to shock, corrosion, chemicals, and water.
Plating	Finishing process in which a thin layer of metal coats a substrate
Polishing	Finishing process of smoothing a surface by rubbing an abrasive or using a chemical action.
Reaming	Finishing process in which a cutting tool produces a hole with very smooth sides or slightly removing material.
Rough Forming	Process of shaping, fusing or joining two pieces of material by means of heat.
Spotface	The cutting of a shallow counterbore, usually about .0625 deep (depth symbol is omitted). The spotface depth does not need to be specified. The spotface provides an accurate bearing surface for the underside of a bolt head.
Stamping	Manufacturing process used to convert flat metal sheets into specific shapes.
Surface Treating	A broad range of processes that alter the surface to achieve a certain property such as resistance to corrosion or wear.
Surfacing	Any of various processes in which material surface is improved.
Thermoforming	Plastic manufacturing process used to make all types of thin-walled products, such as containers, guards, fenders, food packages, and cosmetic packaging. The process works by taking a sheet of plastic material and heating it until it softens. Vacuum pressure is then applied to suck the hot material down against the mold to conform to the desired shape.
Thermoplastic	Plastic process in which material may be heated and formed by pressure. Upon reheating, the shape can be changed. Most plastic products are made with this process because they are easier to mold into various shapes. These products cannot be used where heat might exist.
Thermosetting	Plastic process in which products are formed into a permanent shape by heat and pressure and may not be altered after curing. This process is more expensive and can be more difficult than others because thermoset materials cannot be re-melted once they have

	been melted and formed for the first time. Thermoset products are the choice when the product is used in an application where heat exists such as plastic parts found on or near the engine of a car. The most common production process is die casting.
Through Hole	A hole that passes all the way through the object.
Welding	Assembly process by the fusion or joining of two pieces of metal by means of heat, with or without the application of pressure.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	3.00	B2	15%	Understand Manufacturing Processes.
Indicator	3.02	N/A	N/A	Understand Rough Forming: Casting, Forging, and Welding.
Culminating Question **Essential Questions**	**What manufacturing processes are used to produce a part specific to rough forming?** ● How is casting used in Industry to rough form a part? ● How is forging used in Industry to rough form a part? ● How is welding used in Industry to rough form a part?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand casting as it is used in the manufacturing Industry.
c. Understand forging as it is used in the manufacturing Industry.
d. Understand welding as it is used in the manufacturing Industry.
e. Understand various manufacturing processes used to create a part specific to rough forming.

INSTRUCTIONAL ACTIVITIES- 3.02	
A. Content Literacy Terminology	
Resource(s)	**(See 3.02.1)**
B. Understand casting as it is used in the manufacturing Industry. **C. Understand forging as it is used in the manufacturing Industry.** **D. Understand welding as it is used in the manufacturing Industry.** **E. Understand various manufacturing processes used to create a part specific to rough forming.** *Note: Activity includes all Unpacked Content in steps and combines with Indicator 3.03.*	
Activity	Student Online Resource Creation-Manufacturing Processes
Teacher Instructions	● Provide copy of *Student Online Resource Creation- Manufacturing Processes- Instructions* and *Student Online Resource Creation- Manufacturing Processes- Rubric*. Facilitate explanation of project using rubric to set clear expectations. ● Facilitate students working to create their resource/examples for each type of Manufacturing Process following guidelines. ● Facilitate students sharing the link to their digital resource. ● Provide feedback to students using rubric. *Note:* This project is better suited for most students to be broken into multiple parts/days. Teacher, individual, or peer review is encouraged to take place before final submission.
Student Directions	● Explain multiple types of manufacturing processes used in Industry by creating resources/examples for each type of Manufacturing Process following provided guidelines. ● Share the link to your digital resource. ● Provide feedback using rubric.
Resource(s)	Student Online Resource Creation- Student Online Resource Creation-

Content Literacy Terminology- 3.02.1	
Annealing	The process used to soften metal by heating followed by slow cooling.
Casting	Rough forming metal process of producing a part by pouring molten metal into a mold consisting of damp sand (sand casting) or cavities between metal dies (die casting).
Die Casting	Manufacturing process where malleable material is forced by high pressure into a mold cavity. It results in increased surface quality of the product.
Forging	Rough forming metal process of heating it in a fire or furnace and striking or hammering it.
Hardening	The process which requires heating and then rapid cooling in oil or water.
Heat Treating	The process of changing the properties of metals by heating and cooling.
Machining	Any of various processes in which a piece of raw material is cut into a desired final shape and size by a machine.
Rough Forming	Process of shaping, fusing, or joining two pieces of material by means of heat.
Welding	Assembly process by the fusion or joining of two pieces of metal by means of heat, with or without the application of pressure.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	3.00	B2	15%	Understand Manufacturing Processes.
Indicator	3.03	N/A	N/A	Understand Finishing: Drilling, Machining, and Surfacing.
Culminating Question **Essential Questions**	**What manufacturing processes are used to produce a part specific to finishing?** ● What concepts and terms are associated with drilling and holes? ● How is drilling used in industry? ● How is machining used in industry? ● Hw is surfacing used in industry?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Identify concepts and terms associated with drilling and holes.
c. Understand drilling as it is used in the manufacturing Industry.
d. Understand machining as it is used in the manufacturing Industry.
e. Understand surfacing as it is used in the manufacturing Industry.
f. Understand various manufacturing processes used to create a part specific to finishing.

INSTRUCTIONAL ACTIVITIES- 3.03	
A. Content Literacy Terminology	
Resource(s)	**(See 3.03.1)**
B. Identify concepts and terms associated with drilling and holes.	
Activity	Teacher Led Concept Simplification-Drilling Boring Reaming... "Making Sense of it All"
Teacher Instructions	● Facilitate whole-class discussion/clarification of concepts associated with manufacturing holes using *Teacher Led Concept Simplification-Drilling Boring Reaming... Making Sense of it All-PowerPoint Presentation.* Select videos to accompany presentation topics.
Student Directions	● Differentiate between terms associated with drilling in manufacturing by participating in class discussion/clarification of concepts associated with manufacturing holes.
Resource(s)	 Teacher Led Concept Simplificati
C. Understand drilling as it is used in the manufacturing Industry. **D. Understand machining as it is used in the manufacturing Industry.** **E. Understand surfacing as it is used in the manufacturing Industry.** **F. Understand various manufacturing processes used to create a part specific to finishing.** *Note: Activity includes all Unpacked Content in steps and combines with Indicator 3.02.*	
Activity	Student Online Resource Creation-Manufacturing Processes
Teacher Instructions	● Provide copy of *Student Online Resource Creation- Manufacturing Processes- Instructions* and *Student Online Resource Creation-Manufacturing Processes- Rubric*. Facilitate explanation of project using rubric to set clear expectations. ● Facilitate students working to create resources/examples for each type of Manufacturing Process following guidelines. ● Facilitate students sharing the link to their digital resource. ● Provide feedback to students using rubric. *Note:* This project is better suited for most students to be broken into multiple parts/days. Teacher, individual, or peer review is encouraged to take place before final submission.
Student Directions	● Explain multiple types of manufacturing processes used in Industry by creating resources/examples for each type of Manufacturing Process following provided guidelines. ● Share the link to your digital resource. ● Provide feedback using rubric.
Resource(s)	 Student Online Resource Creation-  Student Online Resource Creation-

Content Literacy Terminology-3.02.1	
Blind Hole	A hole that cuts into but does not pass completely through the object.
Boring	Finishing process in which a cutting tool enlarges a hole to a more accurate size.
Coating	Finishing process in which a covering is applied to a surface.
Counterbore	To enlarge the end of a drill hole to a specific diameter and depth in order to recess a mating part.
Countersink	To recess a hole with a conically (cone) shaped tool to provide a seat for flat head screws.
Deburring	Finishing process that removes built-up burrs and other imperfections from a finished surface.
Drilling	The process of cutting a cylindrical hole into an object.
Finishing	Machining processes used to increase accuracy of a part or surface treating such as grinding, milling, polishing, and deburring.
Heat Treating	The process of changing the properties of metals by heating and cooling.
Machining	Any of various processes in which a piece of raw material is cut into a desired final shape and size by a machine.
Milling	Finishing process using rotary cutters to remove material.
Plating	Finishing process in which a thin layer of metal coats a substrate.
Polishing	Finishing process of smoothing a surface by rubbing an abrasive or using a chemical action.
Reaming	Finishing process in which a cutting tool produces a hole with very smooth sides or slightly removing material.
Spotface	The cutting of a shallow counterbore, usually about .0625 deep (depth symbol is omitted). The spotface depth does not need to be specified. The spotface provides an accurate bearing surface for the underside of a bolt head.
Surface Treating	A broad range of processes that alter the surface to achieve a certain property such as resistance to corrosion or wear.
Surfacing	Any of various processes in which material surface is improved.
Through Hole	A hole that passes all the way through the object.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	3.00	B2	15%	Understand Manufacturing Processes.
Indicator	3.04	N/A	N/A	Understand Assembly techniques.
Culminating Question **Essential Questions**	**What manufacturing processes are used to assemble parts?** ● What are the two main types of holes used in assemblies? ● What types of fasteners are used in assemblies? ● How are holes designed with threads to fit various types of fasteners?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Identify the main types of holes used in assemblies.
c. Identify the major categories of fasteners.
d. Understand holes designed with threads to fit various types of fasteners.
e. Understand the manufacturing processes used to assemble parts.

INSTRUCTIONAL ACTIVITIES- 3.04	
A. Content Literacy Terminology	
Resource(s)	**(See 3.04.1)**
B. Identify the main types of holes used in assemblies. **C. Identify the major categories of fasteners.** **D. Understand holes designed with threads to fit various types of fasteners.** **E. Understand the manufacturing processes used to assemble parts.** *Note: Activity includes all Unpacked Content in steps and combines with part of Indicator 4.04.*	
Activity	Teacher-Led Overview-Holes, Threads, and Fasteners
Teacher Instructions	● Facilitate whole-class instruction using *Teacher-Led Overview-Holes Threads and Fasteners-PowerPoint Presentation*. Show real life examples of terms/concepts in presentation (i.e., show students what an actual rivet looks like and how it functions).
Student Directions	● Participate in whole-class instruction and identify common types or holes and fasteners with provided examples.
Resource(s)	 Teacher-Led Overview- Holes Thr

Content Literacy Terminology-3.04.1	
Assembly	The process where various parts are put together to complete the product.
Blind Hole	A hole that cuts into but does not pass completely through the object.
Counterbore	To enlarge the end of a drill hole to a specific diameter and depth in order to recess a mating part.
Countersink	To recess a hole with a conically (cone) shaped tool to provide a seat for flat head screws.
Fastener	A hardware device that mechanically joins or affixes two or more objects together. Common types include, but are not limited to, screws, bolts (with nuts), rivets, pins, and keys.
Spotface	The cutting of a shallow counterbore, usually about .0625 deep (depth symbol is omitted). The spotface depth does not need to be specified. The spotface provides an accurate bearing surface for the underside of a bolt head.
Thread	A uniformed ridge/cut in the form of a helix on a fastener used to hold parts together, adjust parts and/or transmit power.
Through Hole	A hole that passes all the way through the object.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	3.00	B2	15%	Understand Manufacturing Processes.
Indicator	3.05	N/A	N/A	Understand various manufacturing materials: Metal, Plastic, and Inorganic Material.
Culminating Question **Essential Questions**	**What types of materials are used in manufacturing?** ● Why is it important to understand various materials used in manufacturing? ● What major classifications of materials are used to manufacture a part?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Recognize the importance of understanding various materials used in manufacturing.
c. Identify various types of materials used in manufacturing.

INSTRUCTIONAL ACTIVITIES- 3.05	
A. Content Literacy Terminology	
Resource(s)	**(See 3.05.1)**
B. Recognize the importance of understanding various materials used in manufacturing. **C. Identify various types of materials used in manufacturing.** *(Note: Activity includes all Unpacked Content in steps and combines with Indicator 3.01.*	
Activity	Manufacturing Processes-Introduction
Teacher Instructions	● Provide students a hard copy of *Manufacturing Processes-Introduction-Presentation- Guided Notes.* Facilitate whole-class guided instruction using *Manufacturing Processes- Introduction-Presentation* while students fill in notes. ● Provide digital copy of *Manufacturing Processes- Introduction-Presentation* for reference while students answer leading questions *Manufacturing Processes-Introduction-Leading Questions*. ● Break students into small groups or pairs and facilitate student led discussion on individual answers to leading questions.
Student Directions	● Participate in whole-class guided instruction while filling in guided notes. ● Answer leading questions using the provided PowerPoint Presentation. ● Demonstrate your understanding of major terms/concepts associated with manufacturing processes by discussing answers to leading questions in small groups or pairs.
Resource(s)	Manufacturing Processes- Introduct Manufacturing Processes- Introduct Manufacturing Processes- Introduct Manufacturing Processes- Introduct

Content Literacy Terminology-3.05.1	
Inorganic materials	Materials which do not contain carbon (non-living sources.) Examples used in manufacturing include, but are not limited to rocks/minerals, glass, ceramics, and metals.
Manufacturing Materials	Materials for production are divided into three general categories: metal, plastic, and inorganic materials.
Metal	Ferrous (iron), nonferrous, and alloy (mixture) material characterized by high electrical and thermal conductivity as well as by malleability, ductility, and high reflectivity of light.
Plastic	Synthetic or semi-synthetic organic compounds (polymers) characterized by being generally low cost, durable, strong for their weight, electrically and thermally insulative, and resistant to shock, corrosion, chemicals, and water.

Course	IV22 Drafting II - Engineering			
Essential Standard	4.00	C3	30%	Apply Parametric-Solid Modeling Techniques to create a 3D Model.
Indicator	4.01	N/A	N/A	Apply the techniques to complete 2D sketches using the appropriate draw tools.
Culminating Question **Essential Questions**	**What techniques are used to create 2D sketches with appropriate draw tools?** ● How is the User Interface navigated? ● What are the major environments provided by a 3D modeling software program? ● What are the basic draw and modify tools used to create a sketch?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Navigate the user interface of a 3D modeling software.
c. Understand the major environments provided by a 3D modeling software program.
d. Understand the basic draw and modify tools used to create a sketch.
e. Apply techniques used to create 2D sketches with appropriate draw & modify tools.

INSTRUCTIONAL ACTIVITIES-4.01	
A. Content Literacy Terminology	
Resource(s)	**(See 4.01.1)**
B. Navigate the user interface of a 3D Modeling software.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Complete video or written tutorial session(s.)
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
C. Understand the major environments provided by a 3D Modeling software program.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Complete video or written tutorial session(s.)
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
D. Understand the basic draw and modify tools used to create a sketch.	
Activity	Read Aloud & Guided Practice-Sketches
Teacher Instructions	● Provide a hard copy of *Guided Practice- Sketches*. Facilitate reading aloud of Introduction. ● Direct students through steps to create a sketch and further into guided practice for each type of draw and modify tool given. These do not have to be specific drawings as much as just practice with tools.
Student Directions	● Listen to reading aloud of Introduction. ● Locate and use basic draw and modify tools by creating a sketch following steps and completing the guided practice for each type of draw and modify tool given. These do not have to be specific drawings but a practice with tools.
Resource(s)	Guided Practice-Sketches.docx
E. Apply techniques used to create 2D sketches with appropriate draw & modify tools.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Complete video or written tutorial session(s.)
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-4.01.1	
Active Sketch	The current sketch to edit.
Adaptive Geometry	Geometry that can have two-way dependencies. One part is designated as fixed geometry, and its adaptive counterparts update when the fixed geometry is changed. Whenever a part is reassigned from adaptive to fixed geometry, then changes made to that part update the remaining adaptive counterparts.
Arc Sketch Tool	Sketch drawing command/tool used to draw curved lines by locating the center point of arc, points of tangency or 3-points.
Assembly File (.iam)	A type of file used within a constraint-based CAD system to organize individual parts and/or assemblies to create a more complex representation of a product that contains information about how parts are constrained relative to one another. Also, a main primary environment in the 3d Modeling Software Autodesk Inventor.
Associative Geometry	Geometry that has a one-way parametric relationship between parent and child geometry. Child geometry projected from a parent part is dependent on parent geometry. Child geometry updates when the parent is modified.
Chamfer Sketch Tool	Sketch modify command/tool used to bevel corners at desired angle or distance.
Circle Sketch Tool	Sketch drawing command/tool used to sketch circles. Can be drawn by locating the center point and entering radius or diameter or locating points along the perimeter of the circle.
Circular Pattern Sketch Tool	Sketch modify command/tool used to repeat a sketch multiple times in a circular direction around a center point at a given distance in one action.
Constraint-based modeling software	A type of CAD software that uses feature definitions (extrude, revolve, fillet, etc.), dimensional and geometric constraints (equal, parallel, concentric, etc.), and a feature tree (how the features are arranged) to define 3D solid models. Examples: SolidWorks®, Inventor®, and Pro/Engineer®
Construction Geometry	Geometry used to assist in the creation of sketches or features, but not used to define profiles or paths. The line style designates a curve as construction geometry.
Construction plane or Workplane	The most common type of construction geometry within constraint-based CAD systems. They are planes in 3D space used to define global (world) and local (user defined) coordinate systems. They can be imaginary planes or surfaces on the existing solid model.
Coordinate system	The set of magnitudes (visually represented by the X, Y, and Z axes) that determine the position of points, lines, curves, and planes in part and assembly files. By default, a grid displays on the active sketch plane of the coordinate system.
Copy Sketch Tool	Sketch modify command/tool used to duplicate existing geometry.
Drawing File (.dwg)	A file used to create traditional 2D documentation of objects within a constraint-based CAD system. They typically include traditional views (top, front, right-side, bottom, left-side, rear, auxiliary, sectional, and pictorials), dimensions (standard, tolerance, and geometric), and

	annotations or notes (including title blocks and borders). Also, a main primary environment in the 3d Modeling Software Autodesk Inventor.
Ellipse Sketch Tool	Sketch drawing command/tool used to draw ellipses by locating the center point and then major and minor axis.
Extend Sketch Tool	Sketch modify command/tool that extends geometry to a fence.
Feature	A physical portion of a solid model that appears in the feature tree. They can be sketches, extrudes, revolves, sweeps, lofts, fillets, chamfers, etc.
Feature definition	The method a constraint-based CAD system uses to keep track of the parameters for each individual feature that makes up a solid model. Swept features are defined by a construction plane or work plane, a sketch or profile with dimensional and geometric constraints, a path or direction, and a distance or angle. Other features such as fillets, chamfers, and shells are not defined by a sketched profile but by other parameters usually selected from a dialog box within the software.
Feature tree	A list of the geometric features that exist within a model file in the order in which they are interpreted by the modeler (sometimes called a browser, modeling tree, history, or feature manager design tree). Features in the tree can be construction geometry (origins, planes, axes, etc.), part features (extrudes, revolves, sweeps, lofts, fillets, etc.), or components in an assembly file.
Fillet Sketch Tool	Sketch modify command/tool used to round off corners to desired radius.
Graphics Window	The active modeling area in which sketches, constraints, features, parts, and assemblies are created and edited. In the graphics window, models can be rotated, zoomed in and out, and view characteristics such as appearance, material, and light defined.
Heads-Up Display (HUD)	Also referred to as Dynamic Input, the user interface near the cursor in the Sketch Environment to help you keep your focus in the sketching area. Value input fields near the cursor display information that is dynamically updated as the cursor moves.
Line Sketch Tool	The most basic sketch drawing command used to draw straight or angled lines. The cursor can be used to snap to the origin or existing points of other sketch entities.
Mini-toolbar	Located near a selected object in the graphics window, the mini-toolbar displays in-canvas buttons that allow quick access to frequently used commands and command options.
Mirror Sketch Tool	Sketch modify command/tool used to create mirrored sketch geometry based on a selected reference axis or entity.
Move Sketch Tool	Sketch modify command/tool used to relocate existing geometry.
Offset Sketch Tool	Sketch modify command/tool used to create a parallel entity at a desired distance from the object.
Parametric Dimension	Dimensions are constraints that control sketch size. The sketch geometry resizes when you change the dimension value. Together, geometric constraints and dimensional constraints control the size and shape of sketches used to create features.
Part File (.ipt)	An individual solid model file within a constraint-based CAD system that contains information about the part's 2D and 3D geometry, appearance, material properties, and annotations or notes. Also, a

	main primary environment in the 3d Modeling Software Autodesk Inventor.
Point Sketch Tool	Sketch drawing command/tool that places a point onto a sketch for reference or dimension.
Polygon Sketch Tool	Sketch drawing command/tool used to draw regular polygons. Number of sides and center of the polygon are entered to create the polygon.
Presentation File (.ipn)	A file used to create scenes, exploded views, animations, and other stylized views of an assembly to aid you in documenting your design. Also, a main primary environment in the 3D Modeling Software Autodesk Inventor.
Primary Environments	Drawing and modeling purpose/task driven environments provided by different Inventor products. Base environments include: Part, Drawing, Sheet Metal Part, Assembly, Weldment Assembly, Presentation.
Profile	A closed loop defined by sketched or reference geometry that represents a cross section of a feature. An open profile defined by sketched segments, arcs, or splines can define a surface shape or extend to boundaries to close a region. A profile can enclose islands.
Project	A means to organize Autodesk Inventor files and maintain valid links logically between files. A project consists of a home folder, a project file that specifies the paths to the locations of the files in the project, and the local and network folders containing Autodesk Inventor files. You can have various projects as needed to manage your work. The project file for each project must be maintained in the project home folder. A project file is a text file with an. ipj extension.
Rectangle Sketch Tool	Sketch drawing command/tool used to sketch rectangles and squares. Can be drawn corner to corner or from the center of a rectangle.
Rectangular Pattern Sketch Tool	Sketch modify command/tool used to repeat a sketch multiple times in a linear direction in one action.
Rotate Sketch Tool	Sketch modify command/tool used to rotate selected objects about a specified point.
Sketch (or Profile)	Within the context of constraint-based modeling, the 2D geometry created on a construction plane or workplane which is used with some type of sweeping operation (extrude, cut-extrude, revolve, cut-revolve, loft, sweep, etc.) to create a solid model.
Sketch Plane	A planar face or work plane on which the current sketch is created.
Slot	An elongated hole.
Slot Sketch Tool	Sketch drawing command/tool used to draw slots by locating the centers of arcs or center point of slot. Can also draw curved slots that follow a 3-point or center point arc.
Spline Sketch Tool	Sketch drawing command/tool used to draw irregular curved lines that can be adjusted with handles.
Template	An assembly, part, or drawing file that contains predefined file properties. To create a file based on a template, open a template file, create the content, and then save it with a unique file name. Pre-defined properties can include visible default reference planes, customized grid settings, color scheme, drafting standards, and such.
Text Sketch Tool	Sketch drawing command/tool that adds text to a sketch which can then be extruded as a solid/void model.

Trim Sketch Tool	Sketch modify command/tool used to remove unwanted sketch geometry that intersects with other geometry.
Unconsumed Sketch	A sketch in a part or assembly model that was not used in a feature. An unconsumed sketch can be used to show assembly layout and develop design concepts. You can display unconsumed sketches in drawing views.
Visibility	A characteristic of an assembly component that determines whether it appears in the graphics window. In large assemblies, it is useful to turn off the visibility of components not needed in the current design.

Course	IV22 Drafting II - Engineering			
Essential Standard	4.00	C3	30%	Apply Parametric-Solid Modeling Techniques to create a 3D Model.
Indicator	4.02	N/A	N/A	Apply the techniques of Adding Constraints: Geometric and Dimensional.
Culminating Question **Essential Questions**	**What are the techniques used to add constraints: geometric and dimensional?** ● What is the difference between a geometric and dimensional constraint? ● What are the various types of geometric sketch constraints used in 3D modeling software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the difference between geometrical and dimensional constraints.
c. Understand the various types of sketch geometric constraints.
d. Apply techniques used to add sketch constraints: geometric and dimensional.

INSTRUCTIONAL ACTIVITIES-4.02	
A. Content Literacy Terminology	
Resource(s)	**(See 4.02.1)**
B. Understand the difference between Geometrical and Dimensional Constraints.	
Activity	Small Group Guided Content Reading-Constraints
Teacher Instructions	● Break students into heterogeneous groups ahead of time and provide groups a hard copy of *Peer Teaching- Constraints.* ● Facilitate individuals defining pre-reading terms in Part One. Students may use the internet, professional texts or class textbooks to define terms. ● Facilitate group discussion of definitions for Part Two. ● Facilitate students reading the passage in small groups and following instructions for Part Three: Selecting, Verifying, Organizing and then Analyzing chosen excerpts. ● Facilitate each group sharing review/comparisons of lists and/or the final statement.
Student Directions	● Define pre-reading terms in Part One. You may use the internet, professional texts or class textbook to define terms ● Discuss definitions for Part Two. ● Read the passage in your small group and follow instructions for Part Three: Select, Verify, Organize and then Analyze chosen excerpts. ● Share review/comparisons of lists and/or the final statement. .
Resource(s)	 Small Group Guided Content Rea
C. Understand the various types of sketch Geometric Constraints.	
Activity	Peer Teaching-Constraints
Teacher Instructions	● Assign students a geometric constraint from *Peer Teaching-Constraints* ahead of time. Constraints are listed by difficulty from top to bottom. Students can also be paired as needed. Variety in assignment of topics amongst class as a whole will guarantee peer teaching at the end of activity covers all concepts. ● Read instructions to whole-class. Facilitate students creating presentations. Students will be able to use the internet, *Peer Teaching- Constraints* document and software as resources for creating their presentations. ● Facilitate students sharing presentations.
Student Directions	● Listen to instructions. Create a presentation on your assigned geometric constraint. You may use the internet, *Peer Teaching-Constraints* document and software as resources for creating your presentation. ● Share your presentation and listen to others to determine the difference and usage of geometric constraints.
Resource(s)	 Peer Teaching-Constraints.docx

D. Apply techniques used to add sketch Constraints: Geometric and Dimensional.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Complete video or written tutorial session(s.)
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-4.02.1	
Active Sketch	The current sketch to edit.
Adaptive Geometry	Geometry that can have two-way dependencies. One part is designated as fixed geometry, and its adaptive counterparts update when the fixed geometry is changed. Whenever a part is reassigned from adaptive to fixed geometry, then changes made to that part update the remaining adaptive counterparts.
Associative Geometry	Geometry that has a one-way parametric relationship between parent and child geometry. Child geometry projected from a parent part is dependent on parent geometry. Child geometry updates when the parent is modified.
Angle Constraint	An assembly constraint that controls the angle between planes on two components in an assembly.
Coincident Sketch Constraint	A geometric constraint that constrains two points together or constrains one point to a curve. When this constraint is applied to the center points of two circles, arcs, or ellipses, the result is the same as the concentric constraint.
Collinear Sketch Constraint	A geometric constraint that causes two or more-line segments or ellipse axes to lie along the same line. In an assembly, a collinear constraint is achieved with a mate constraint between two lines, edges, or axes.
Concentric Sketch Constraint	A geometric constraint that causes two circles, arcs, or ellipses to have the same center point. The result is the same as that of a coincident constraint applied to the centers of the curves.
Constraint	Rules that govern the position, slope, tangency, dimensions, and relationships among sketch geometry or the relative position between parts in an assembly. Geometric constraints control the shapes and relationships among sketch elements or assembly components. Dimensional constraints control size. Applying constraints removes degrees of freedom.
Constraint-based modeling software	A type of CAD software that uses feature definitions (extrude, revolve, fillet, etc.), dimensional and geometric constraints (equal, parallel, concentric, etc.), and a feature tree (how the features are arranged) to define 3D solid models. Examples: SolidWorks®, Inventor®, and Pro/Engineer®.
Construction plane or Workplane	The most common type of construction geometry within constraint-based CAD systems. They are planes in 3D space used to define global (world) and local (user defined) coordinate systems. They can be imaginary planes or surfaces on the existing solid model.
Construction Geometry	Geometry used to assist in the creation of sketches or features, but not used to define profiles or paths. The line style designates a curve as construction geometry.
Dimensional Constraint	Parametric dimensions that control sketch size. When dimensions are changed, the sketch resizes. Dimensional constraints can be expressed as numeric constants, as variables in equations, or in parameter files.
Equal Sketch Constraint	A geometric constraint that causes selected arcs and circles to have the same radius or selected lines to have the same length.

Explicit constraints	Constraints which the user must apply by completing some type of command action.
Fixed Sketch Constraint	Sketch constraint that fixes points and curves in position relative to the sketch coordinate system.
Geometrical Constraint	Rules that define the geometric relationships of sketch elements and control how a sketch can change shape or size. Geometric constraints are coincident, collinear, concentric, equal, fix, horizontal, parallel, perpendicular, tangent, and vertical. For example, two lines may be defined as parallel, equal, or collinear (in the same line).
Horizontal Sketch Constraint	A geometric constraint that positions selected lines, ellipse axes, or pairs of points parallel to the X axis of the sketch coordinate system or positions 2 selected points (curve endpoint, center, midpoint, or sketch point) the same distance from the primary axis (same Y coordinate).
Implicit constraints	Constraints which get applied automatically by the software when the user sketches lines. Examples: the horizontal and vertical constraints that are applied to lines when they are sketched.
Insert Constraint	Places a planar and axial mate as a single constraint between selected cylindrical faces or edges.
Parallel Sketch Constraint	A geometric constraint that causes two or more lines or ellipse axes to have the same slope and orientation.
Parametric Dimension	Dimensions are constraints that control sketch size. The sketch geometry resizes when you change the dimension value. Together, geometric constraints and dimensional constraints control the size and shape of sketches used to create features.
Perpendicular Sketch Constraint	A geometric constraint that causes two lines or ellipse axes to lie at right angles to one another.
Tangent Sketch Constraint	A geometric constraint that causes two curves to have the same slope at the point where they intersect. For example, a line can be tangent to an arc, circle, or ellipse, but two lines cannot be tangent to one another.
Vertical Constraint	A geometric constraint that positions selected lines, ellipse axes, or pairs of points parallel to the Y-axis of the sketch coordinate system (same X coordinate).

Course	**IV22 Drafting II - Engineering**			
Essential Standard	4.00	C3	30%	Apply Parametric-Solid Modeling Techniques to create a 3D Model.
Indicator	4.03	N/A	N/A	Apply the techniques of Creating 3D Parts from constrained sketches using: Extrude, Revolve, Sweep and Loft.
Culminating Question	**How are the techniques applied in the software to create 3D Parts from constrained sketches using: Extrude, Revolve, Sweep and Loft?**			
Essential Questions	● What do the terms Extrude, Revolve, Sweep and Loft mean regarding 3D modeling? ● How does the Boolean operation apply to creating 3D parts with Extrude, Revolve, Sweep and Loft? ● What are the techniques used in the software to create 3D Parts from constrained sketches using: Extrude, Revolve, Sweep and Loft?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand concepts/terms related to Extrude, Revolve, Sweep and Loft.
c. Understand Boolean.
d. Understand the techniques of creating 3D parts from constrained sketches using: Extrude, Revolve, Sweep and Loft
e. Apply the techniques of creating 3D parts from constrained sketches using: Extrude, Revolve, Sweep and Loft

INSTRUCTIONAL ACTIVITIES-4.03	
A. Content Literacy Terminology	
Resource(s)	**(See 4.03.1)**
B. Understand concepts/terms related to Extrude, Revolve, Sweep and Loft. *Note: Activity combines with part of Indicator 4.04.*	
Activity	Individual Research Activity-3D Modeling Definitions
Teacher Instructions	• Provide digital copy of *Individual Research Activity- 3D Modeling Definitions*. Read instructions. Facilitate students completing research and compiling image examples for terms. • Collect examples or have students share examples for review/formative assessment.
Student Directions	• Define common terms related to 3D Modeling commands by completing research and compiling image examples for terms. • Participate in review.
Resource(s)	 Individual Research Activity- 3D Modelin
C. Understand Boolean. *Note: Activity combines with part of Indicator 4.04*	
Activity	Guided Concept Understanding-Boolean
Teacher Instructions	• Facilitate guided instruction and concept review using *Guided Concept Understanding-Boolean PowerPoint Presentation*.
Student Directions	• Participate in class/small group discussion. • Select the appropriate Boolean operation when applying 3D modeling techniques.
Resource(s)	 Guided Concept Understanding- Boc
D. Understand the techniques of creating 3D parts from constrained sketches using: Extrude, Revolve, Sweep and Loft. *Note: Activity combines with part of Indicator 4.03.*	
Activity	Teacher Facilitated Content Review-3D Modeling Terms
Teacher Instructions	• Facilitate guided instruction/practice of 3D Modeling commands following *Teacher Facilitated Content Review-3D Modeling Terms-PowerPoint Presentation* and students completing basic part operations along with teacher-led examples.
Student Directions	• Create basic parts in software using 3D modeling concepts techniques by participating in guided instruction.
Resource(s)	 Teacher Facilitated Content Review- 3D
E. Apply the techniques of creating 3D parts from constrained sketches using: Extrude, Revolve, Sweep and Loft.	
Activity	Software Tutorials
Teacher Instructions	• Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.

	Additional Resource: Student Application-Isometric Part Examples is available for examples of some concepts, but not all in this indicator (originally sourced from: Spencer, H. C., Dygdon, J. T., & Novak, J. E. (1995). Basic Technical Drawing.) (Activity can combine with Indicators 4.03, 4.04, 7.01, 7.02, 7.03, & 7.05)
Student Directions	● Complete video or written tutorial session(s.)
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Student Application- Isometr

Content Literacy Terminology-4.03.1		
Active Part	The part that is being edited. In an open part file, the part is active and available for edit. In an assembly file, select the part in either the browser or the graphics window before it can be edited. If a part or subassembly was previously hidden or designated as background, it must be enabled before activation. The active part may be edited.	
Active Sketch	The current sketch to edit.	
Axis of Revolution	The centerline of a revolved feature.	
Asymmetric	An option in both the Extrude and Revolve commands that allows a 2D profile to be extruded or revolved in both positive and negative directions simultaneously with different linear or angular values.	
Base Feature	The first feature created in a part. May be an imported base solid (.sat or .step file format), in which case the base feature is a fixed size. Sketched or placed features add details to the base feature and are positioned relative to one another using dimensional or geometric constraints. The base feature should represent the most basic shape in the part.	
Boolean	Combining two or more objects by creating intersections, subtractions, or unions of mass.	
	Cut (Subtract) Operation	One of three Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. A cut operation removes the volume of a sketched feature from an existing feature. Not available for base features.
	Intersect Operation	One of three Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. An intersect operation creates a feature from the shared volume of a sketched feature and an existing feature. Material not included in the shared volume is deleted. Not available for base features.
	Join (Union) Operation	One of three Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. A join operation adds the volume of a sketched feature to the existing feature. Not available for base features.
Child	In a hierarchical design relationship, a child element is dependent on another (parent) element. A typical example is a feature, such as a cut, that depends on a base feature. In the browser, the child, or dependent feature, is indented under its parent. A child feature can be a parent to other features. In most cases, deleting the child feature has no effect on the parent feature.	
Degree of Freedom	The variables by which an object can move. Each object has six degrees of freedom; three translational (linear movement along the X, Y, or Z axes) and three rotational (rotation about the X, Y, or Z axes).	
Dependent Features	Features that are geometrically dependent on another feature, such as chamfers and fillets.	

Extrude	A feature created by adding depth to a sketched profile. Feature shape is controlled by profile shape, extrusion extent, and taper angle. Unless the extruded feature is the first feature, its relationship to an existing feature or body is defined by selecting a Boolean operation (join, cut, or intersect) and the participating bodies for the operation if multiple bodies exist. Optionally, can create a new body.
Feature	A physical portion of a solid model that appears in the feature tree. They can be extrudes, revolves, sweeps, lofts, fillets, chamfers, etc.
Feature definition	The method a constraint-based CAD system uses to keep track of the parameters for each individual feature that makes up a solid model. Swept features are defined by a construction plane or workplane, a sketch or profile with dimensional and geometric constraints, a path or direction, and a distance or angle. Other features such as fillets, chamfers, and shells are not defined by a sketched profile but by other parameters usually selected from a dialog box within the software.
Feature tree	A list of the geometric features (sometimes called a browser, modeling tree, history, or feature manager design tree) that exist within a model file in the order in which they are interpreted by the modeler. Features in the tree can be construction geometry (origins, planes, axes, etc.), part features (extrudes, revolves, sweeps, lofts, fillets, etc.), or components in an assembly file.
Loft	3D modeling technique where, at least two, 2D profiles on separate construction planes are created and the software interpolates the solid geometry between the two profiles.
Parameter	Used to define the size and shape of features and to control the relative positioning of components within assemblies. Can be expressed as equations to define the relationships between geometric elements relative to one another. Changes to one element update the other.
Parent	In a hierarchical system, a parent object owns dependent child objects. Deleting a parent deletes dependent children objects. For example, deleting a plate also deletes the pattern of holes on the plate. Deleting a child has no effect on its parent object.
Part File (.ipt)	An individual solid model file within a constraint-based CAD system that contains information about the part's 2D and 3D geometry, appearance, material properties, and annotations or notes.
Path	The trajectory of a sweep feature. A path can be an open or closed loop consisting of lines, arcs, ellipses, or circles, with a specified start point.
Profile	A closed loop defined by sketched or reference geometry that represents a cross section of a feature. An open profile defined by sketched segments, arcs, or splines can define a surface shape or extend to boundaries to close a region. A profile can enclose islands.
Revolve	A solid feature created by revolving a profile around an axis. Unless the revolved feature is the first feature, its relationship to an existing feature or body is defined by selecting a Boolean operation (join, cut, or intersect), and the participating bodies for the operation if multiple bodies exist. Optionally, can create a new body.

Shared Sketch	A sketch used by more than one feature: for example, a sketch containing hole centers for different hole features.
Sweep	A feature created by moving a profile along a path. A sweep feature usually requires two sketches, a profile, and a path on intersecting planes.
Void	A group of faces that define an internal hollow space. For example, when a cube is shelled without removing a face, the result is a void inside the cube.

Course	IV22 Drafting II - Engineering			
Essential Standard	4.00	C3	30%	Apply Parametric-Solid Modeling Techniques to create a 3D Model.
Indicator	4.04	N/A	N/A	Apply the techniques of adding Placed Features: Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features.
Culminating Question **Essential Questions**	**How are the techniques applied in the software to create 3D Parts from constrained sketches using; Extrude, Revolve, Sweep and Loft?** ● What do the terms Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features mean in regards to 3D modeling? ● How does the Boolean operations apply to creating 3D parts with Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features? ● What are the techniques used in the software to create 3D Parts with Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand concepts/terms related to Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features.
c. Understand Boolean.
d. Understand terms related to holes in 3D Modeling.
e. Understand the techniques of Creating 3D Parts from constrained sketches using: Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features.
f. Apply the techniques of Creating 3D Parts from constrained sketches using: Hole, Fillet, Chamfer, Shell, threads, and pattern features.

INSTRUCTIONAL ACTIVITIES-4.04	
A. Content Literacy Terminology	
Resource(s)	**(See 4.04.1)**
B. Understand concepts/terms related to Hole, Fillet, Chamfer, Shell, threads, and pattern features. *Note: Activity combines with part of Indicator 4.03.*	
Activity	Individual Research Activity-3D Modeling Definitions
Teacher Instructions	● Provide digital copy of *Individual Research Activity- 3D Modeling Definitions*. Read instructions. Facilitate students completing research and compiling image examples for terms. ● Collect examples or have students share examples for review/formative assessment.
Student Directions	● Define common terms related to 3D Modeling commands by completing research and compiling image examples for terms. ● Participate in review.
Resource(s)	 Individual Research Activity- 3D Modelin
C. Understand Boolean. *Note: Activity combines with part of Indicator 4.03.*	
Activity	Guided Concept Understanding-Boolean
Teacher Instructions	● Facilitate guided instruction and concept review using *Guided Concept Understanding-Boolean PowerPoint Presentation*.
Student Directions	● Participate in class/small group discussion. ● Select the appropriate Boolean operation when applying 3D modeling techniques.
Resource(s)	 Guided Concept Understanding- Boc
D. Understand terms related to holes in 3D Modeling. *Note: Activity combines with Indicator 3.04*	
Activity	Teacher-Led Overview-Holes, Threads, and Fasteners
Teacher Instructions	● Facilitate whole-class instruction using *Teacher-Led Overview-Holes Threads and Fasteners-PowerPoint Presentation*. Show real life examples of terms/concepts in presentation (i.e., show students what an actual rivet looks like and how it functions).
Student Directions	● Participate in whole-class instruction and identify common types or holes and fasteners with provided examples.
Resource(s)	 Teacher-Led Overview- Holes Thr
E. Understand the techniques of Creating 3D Parts from constrained sketches using: Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features. *Note: Activity combines with part of Indicator 4.03.*	
Activity	Teacher Facilitated Content Review-3D Modeling Terms

Teacher Instructions	● Facilitate guided instruction/practice of 3D Modeling commands following *Teacher Facilitated Content Review-3D Modeling Terms-PowerPoint Presentation* and students completing basic part operations along with teacher led examples.
Student Directions	● Create basic parts in software using 3D modeling concepts techniques by participating in guided instruction.
Resource(s)	 Teacher Facilitated Content Review- 3D
F. Apply the techniques of Creating 3D Parts from constrained sketches using: Hole, Fillet, Chamfer, Shell, threads, and pattern features.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU. *Additional Resource: Student Application-Isometric Part Examples* is available for examples of some concepts, but not all in this indicator (originally sourced from: Spencer, H. C., Dygdon, J. T., & Novak, J. E. (1995). Basic Technical Drawing.) (Activity can combine with Indicators 4.03, 4.04, 7.01, 7.02, 7.03, & 7.05)
Student Directions	● Complete video or written tutorial session(s.)
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Student Application- Isometr

Content Literacy Terminology-4.04.1		
Active Part	The part that is being edited. In an open part file, the part is active and available for edit. In an assembly file, select the part in either the browser or the graphics window before it can be edited. If a part or subassembly was previously hidden or designated as background, it must be enabled before activation. The active part may be edited.	
Active Sketch	The current sketch to edit.	
Base Feature	The first feature created in a part. May be an imported base solid (.sat or .step file format), in which case the base feature is a fixed size. Sketched or placed features add details to the base feature and are positioned relative to one another using dimensional or geometric constraints. The base feature should represent the most basic shape in the part.	
Boolean	Combining 2 or more objects by creating intersections, subtractions, or unions of mass.	
	Cut Operation	One of 3 Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. A cut operation removes the volume of a sketched feature from an existing feature. Not available for base features.
	Intersect Operation	One of 3 Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. An intersect operation creates a feature from the shared volume of a sketched feature and an existing feature. Material not included in the shared volume is deleted. Not available for base features.
	Join Operation	One of three Boolean operations (cut, join, and intersect) that define the relationship between a sketched feature and an existing feature. A join operation adds the volume of a sketched feature to the existing feature. Not available for base features.
Chamfer	3D modeling technique is used to cut a corner at an angle.	
Child	In a hierarchical design relationship, a child element is dependent on another (parent) element. A typical example is a feature, such as a cut, that depends on a base feature. In the browser, the child, or dependent feature, is indented under its parent. A child feature can be a parent to other features. In most cases, deleting the child feature has no effect on the parent feature.	
Constraint	Rules that govern the position, slope, tangency, dimensions, and relationships among sketch geometry or the relative position between parts in an assembly. Geometric constraints control the shapes and relationships among sketch elements or assembly components. Dimensional constraints control size. Applying constraints removes degrees of freedom.	

Construction plane or Workplane	The most common type of construction geometry within constraint-based CAD systems. They are planes in 3D space used to define global (world) and local (user defined) coordinate systems. They can be imaginary planes or surfaces on the existing solid model.
Consumed Sketch	A sketch incorporated into a feature, such as a sketch used in an extrusion. By definition, the sketch is consumed by the feature.
Counterbore Hole	Type of hole includes a cylindrical hole with a given angle, and a smaller diameter hole that may have a given depth or pass through the part.
Countersink Hole	Type of hole includes a conical hole with a given depth, and a smaller diameter hole that may have a given depth or pass through the part.
Degree of Freedom	The variables by which an object can move. Each object has six degrees of freedom; three translational (linear movement along the X, Y, or Z axes) and three rotational (rotation about the X, Y, or Z axes).
Dependent Features	Features that are geometrically dependent on another feature, such as chamfers and fillets.
Drilled Hole	Hole with a specified diameter and are flush with the planar face.
Duplicated Feature	A feature that has been copied and arrayed in a rectangular or circular pattern or mirrored.
Feature	A physical portion of a solid model that appears in the feature tree. They can be extrudes, revolves, sweeps, lofts, fillets, chamfers, etc.
Feature definition	The method a constraint-based CAD system uses to keep track of the parameters for each individual feature that makes up a solid model. Swept features are defined by a construction plane or workplane, a sketch or profile with dimensional and geometric constraints, a path or direction, and a distance or angle. Other features such as fillets, chamfers, and shells are not defined by a sketched profile but by other parameters usually selected from a dialog box within the software.
Feature tree	A list of the geometric features (sometimes called a browser, modeling tree, history, or feature manager design tree) that exist within a model file in the order in which they are interpreted by the modeler. Features in the tree can be construction geometry (origins, planes, axes, etc.), part features (extrudes, revolves, sweeps, lofts, fillets, etc.), or components in an assembly file.
Fillet	3D modeling technique is used to cut round an edge to a specified arc radius.
Grounded Component	A part or subassembly for which all six degrees of freedom were removed relative to the assembly origin. You can position the part or subassembly without reference to other parts. It is fixed in space. The first part or subassembly placed in an assembly file is grounded automatically, although the ground can later be deleted and relocated, if needed.
Hole	A geometric feature defined by hole type, placement, size, and dimensions. A hole requires a center point sketch for its origin and placement.
Loft	3D modeling technique where, at least two, 2D profiles on separate construction planes are created and the software interpolates the solid geometry between the two profiles.
Material Properties	Properties defined by the material definition stored in individual part files. Properties include Young's modulus, Poisson's ratio, specific

	heat, density, yield strength, ultimate strength, linear expansion, and thermal conductivity.
Parameter	Used to define the size and shape of features and to control the relative positioning of components within assemblies. Can be expressed as equations to define the relationships between geometric elements relative to one another. Changes to one element update the other.
Parametric	System that allows designers to define entire shapes, not just specific parts.
Parent	In a hierarchical system, a parent object owns dependent child objects. Deleting a parent deletes dependent children objects. For example, deleting a plate also deletes the pattern of holes on the plate. Deleting a child has no effect on its parent object.
Part File (.ipt)	An individual solid model file within a constraint-based CAD system that contains information about the part's 2D and 3D geometry, appearance, material properties, and annotations or notes.
Path	The trajectory of a sweep feature. A path can be an open or closed loop consisting of lines, arcs, ellipses, or circles, with a specified start point.
Pattern	Multiple instances of a placed or sketched feature arrayed in a specified pattern. Patterns are defined by type (rectangular or circular), orientation, number of features, and spacing between features.
Physical Properties	Physical properties of a part, including mass, radii of gyration, volume, principal moments, products, center of gravity, and principal axes. Mass properties are calculated with respect to the sketch coordinate system and the density assigned to an object through a material property attribute.
Shell	3D modeling technique used to "hollow out" solids.
Drilled Hole	Holes have a specified diameter and are flush with the planar face.
Solid Body	An enclosed 3D body that has volume.
Spotface Hole	A hole that includes an enlarged, recessed space for a flush/recessed head.
Sweep	A feature created by moving a profile along a path. A sweep feature usually requires two sketches, a profile, and a path on intersecting planes.
Taper	Sets positive or negative taper angle for sweeps, extrusions, and coils normal to the sketch plane.
Tapped Hole	A hole with a defined thread.
Thread	A uniformed ridge/cut in the form of a helix on a fastener used to hold parts together, adjust parts and/or transmit power.
Void	A group of faces that define an internal hollow space. For example, when a cube is shelled without removing a face, the result is a void inside the cube.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	4.00	C3	30%	Apply Parametric-Solid Modeling Techniques to create a 3D Model.
Indicator	4.05	N/A	N/A	Apply adaptive feature parts and subassemblies.
Culminating Question	**How are the techniques applied in the software to create Adaptive Feature parts and subassemblies?**			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply adaptive feature parts and subassemblies.

INSTRUCTIONAL ACTIVITIES-4.05	
A. Content Literacy Terminology	
Resource(s)	**(See 4.05.1)**
B. Apply adaptive feature parts and subassemblies**.**	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU. *Additional Resource: Student Application-Isometric Part Examples* is available for examples of some concepts, but not all in this indicator (originally sourced from: Spencer, H. C., Dygdon, J. T., & Novak, J. E. (1995). Basic Technical Drawing.) (Activity can combine with Indicators 4.03, 4.04, 7.01, 7.02, 7.03, & 7.05)
Student Directions	● Create multiple parts/projects applying 3D modeling concepts.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-4.05.1	
Active Part	The part that is being edited. In an open part file, the part is active and available for edit. In an assembly file, select the part in either the browser or the graphics window before it can be edited. If a part or subassembly was previously hidden or designated as background, it must be enabled before activation. The active part may be edited.
Active Sketch	The current sketch to edit.
Adaptive Geometry	Geometry that can have two-way dependencies. One part is designated as fixed geometry, and its adaptive counterparts update when the fixed geometry is changed. Whenever a part is reassigned from adaptive to fixed geometry, then changes made to that part update the remaining adaptive counterparts.
Associative Geometry	Geometry that has a one-way parametric relationship between parent and child geometry. Child geometry projected from a parent part is dependent on parent geometry. Child geometry updates when the parent is modified.
Adaptive Feature	A feature that can resize when constrained to other features. Individual part features may be designated as adaptive using the context menu. Features are constrained to a specified size, unless they are designated as adaptive.
Adaptive Part	Underconstrained part geometry can resize when designated as an adaptive part in an assembly. Assembly constraints position adaptive parts relative to other parts and adapt the part topology to fully constrained part features. Features that were under-constrained in the part file can resize according to assembly constraints and positions of other parts.
Adaptive Subassembly	A component that contains underconstrained parts or subassemblies. When an adaptive subassembly is constrained within its parent assembly, or to a component in another assembly, underconstrained geometry in an adaptive part resizes. For example, adaptive piston and rod subassemblies are sized and positioned when inserted in an air cylinder assembly.
Angle Constraint	An assembly constraint that controls the angle between planes on two components in an assembly.
Assembly (.iam)	Two or more components (parts or subassemblies) considered as a single model. An assembly typically includes multiple components positioned absolutely and relatively (as required) with constraints that define both size and position. Assembly components may include features defined in place in the assembly. Mass and material properties may be inherited from individual part files.
Assembly Features	Features (chamfers, extrude cuts, holes, weld beads) that are created and saved in the assembly file. The topology is not passed down to the component files. Used to define specific manufacturing processes such as match drilling and post-weld machining.
Assembly Constraints	Rules that determine how parts in an assembly are placed relative to other parts in the assembly. Constraints remove degrees of freedom. Assembly constraints include angle, flush (insert), mate, and tangent. Constraints may be placed between faces of features, part edges,

	points, inferred axes and part work features such as planes, axes, and points.
Assembly Pattern	In an assembly, components that are arranged in a circular or rectangular pattern. All elements in the pattern are identical. Assembly patterns are used to place multiple bolts in holes, or to position any component or components in a symmetrical arrangement. An assembly pattern can be associative to a feature pattern, updating when the feature pattern is edited.
Assembly File (.iam)	A type of file used within a constraint-based CAD system to organize individual parts and/or assemblies to create a more complex representation of a product that contains information about how parts are constrained relative to one another.
Attribute	A data management description of information associated with a part of subassembly. In an assembly or a bill of material, an attribute typically refers to a file-level property to incorporate into a structured bill of material.
Base Feature	The first feature created in a part. May be an imported base solid (.sat or .step file format), in which case the base feature is a fixed size. Sketched or placed features add details to the base feature and are positioned relative to one another using dimensional or geometric constraints. The base feature should represent the most basic shape in the part.
Bidirectional associativity	A term used to describe the relationship between part, assembly, and drawing files within a constraint-based solid modeler. Within constraint-based modelers, changes to any of these files (parts, assemblies, or drawings) are automatically updated in all linked files (e.g., a change to the part file automatically generates changes to the assembly and drawing files or a change to the drawing file automatically generates changes to the assembly and part files).
Bill of Materials	For an assembly, a document that describes components details such as component name, material, and quantity. Bills of material can be single or multiple levels (exploded).
Child	In a hierarchical design relationship, a child element is dependent on another (parent) element. A typical example is a feature, such as a cut, that depends on a base feature. In the browser, the child, or dependent feature, is indented under its parent. A child feature can be a parent to other features. In most cases, deleting the child feature has no effect on the parent feature.
Convert Entities or Project Geometry	Creates sketch geometry on construction plane from existing selected solid model geometry such as a face or edge.
Consumed Sketch	A sketch incorporated into a feature, such as a sketch used in an extrusion. By definition, the sketch is consumed by the feature.
Constraint	Rules that govern the position, slope, tangency, dimensions, and relationships among sketch geometry or the relative position between parts in an assembly. Geometric constraints control the shapes and relationships among sketch elements or assembly components. Dimensional constraints control size. Applying constraints removes degrees of freedom.

Degree of Freedom	The variables by which an object can move. Each object has six degrees of freedom; 3 translational (linear movement along the X, Y, or Z axes) and 3 rotational (rotation about the X, Y, or Z axes).
Dependent Features	Features that are geometrically dependent on another feature, such as chamfers and fillets.
Derived Part	A new part that uses an existing Autodesk Inventor part as its base feature. A part can be scaled or mirrored when inserted into a file, and additional features can further modify the body. A derived part is linked to the original part and can be updated to reflect changes to the original part.
Driven Dimension	A parametric dimension that determines the size of sketch geometry and resizes the sketch when its value changes.
Duplicated Feature	A feature that has been copied and arrayed in a rectangular or circular pattern or mirrored.
Flush Constraint	An assembly constraint that points the surface normals of selected faces in the same direction.
Grounded Component	A part or subassembly for which all six degrees of freedom were removed relative to the assembly origin. You can position the part or subassembly without reference to other parts. It is fixed in space. The first part or subassembly placed in an assembly file is grounded automatically, although the ground can later be deleted and relocated, if needed.
Insert Constraint	Places a planar and axial mate as a single constraint between selected cylindrical faces or edges.
Insert Point	A user-defined point at which the cursor is attached when a sketched symbol is inserted into a drawing. If you do not specify an insert point for a sketched symbol, the cursor is attached to the center of the symbol geometry.
Included Geometry	Model edges, vertices, 2D lines and arcs inserted from existing parts into a 3D sketch using the Include command. Included geometry is converted to 3D geometry and can be used in a path sketch for a 3D sweep feature.
Interchangeable Parts	Parts that are made to easily fit mating parts without additional machining at the time of assembly.
iProperties	File properties that can be used to find and manage Autodesk Inventor files. iProperties are also used to maintain and update information automatically in title blocks, parts lists, bills of material, and sketched symbols. See also design properties.
iFeature	Features, sketches, or subassemblies that can be used in more than one design are designated as iFeatures and saved in a file with an .ide extension. To add an iFeature to a part, use Windows Explorer to drag the file name and drop it in the active part file. To change the size of an iFeature, edit its sketch, feature definition, or link it to parameters that define its size. You can precisely position an iFeature using geometric constraints and dimensions.
Library	The locations of files not edited. Libraries can include purchased or standard parts, Mechanical Desktop parts used in Autodesk Inventor assemblies, iPart factories and members, or other internally developed standard parts. A library is often referenced by multiple projects. Each project specifies the locations of its libraries.

Mate Constraint	An assembly constraint that joins elements together with a surface normal orientation and an optional offset. A planar mate constraint usually moves two external part faces so that their surface normals point in opposite directions. Mate constraints can be used to join points, lines, edges, or axes together and to adapt diameters of unconstrained cylinders.
Parameter	Used to define the size and shape of features and to control the relative positioning of components within assemblies. Can be expressed as equations to define the relationships between geometric elements relative to one another. Changes to one element update the other.
Parametric Dimension	Dimensions are constraints that control sketch size. The sketch geometry resizes when you change the dimension value. Together, geometric constraints and dimensional constraints control the size and shape of sketches used to create features.
Parent	In a hierarchical system, a parent object owns dependent child objects. Deleting a parent deletes dependent children objects. For example, deleting a plate also deletes the pattern of holes on the plate. Deleting a child has no effect on its parent object.
Project	A means to organize Autodesk Inventor files and maintain valid links logically between files. A project consists of a home folder, a project file that specifies the paths to the locations of the files in the project, and the local and network folders containing Autodesk Inventor files.You can have various projects as needed to manage your work. The project file for each project must be maintained in the project home folder. A project file is a text file with an .ipj extension.
Projected Geometry	Geometry (model edges, vertices, work axes, work points, or other sketch geometry) projected onto the active sketch plane as reference geometry. Can include edges of a selected assembly component that intersects the sketch plane when it was cut in an assembly cross section.
Rotation Constraint	A motion constraint that specifies rotation of one part relative to another part using a specified ratio. Used to specify motion of gears and pulleys, for example.
Shared Sketch	A sketch used by more than one feature. For example, a sketch containing hole centers for different hole features.
Subassembly	An assembly file used in another assembly. The subassembly behaves as a single unit, such as a motor with a gear reducer. Parts can originate in part files, as OLE objects, or as iFeatures, or can be imported from the Mechanical Desktop or other CAD system.
Tangent Assembly Constraint	Constraint that can be applied between cylindrical, conical, and toroidal faces or circular arc edges. On selected components, one component moves toward another, and contacts at the point of tangency. In assemblies, tangency can be inside or outside a curve, depending on the direction of the selected surface normal.
Unidirectional associativity	Within constraint-based modelers, changes to the part file automatically generate changes to assembly and drawing files, but not vice versa.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	5.00	B2	15%	Understand Conventional Dimensioning & Tolerancing.
Indicator	5.01	N/A	N/A	Understand intermediate dimensioning techniques.
Culminating Question **Essential Questions**	**How are standard intermediate dimensioning techniques used in technical drawing?** ● What are the foundational techniques for dimensioning technical drawings? ● What are the intermediate hidden line techniques for dimensioning technical drawings? ● What are the intermediate centerline techniques for dimensioning technical drawings? ● What are the intermediate special circumstance techniques for dimensioning technical drawings? ● How are holes and threads notated on Technical drawings?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Review foundational dimensioning technique.
c. Understand intermediate hidden line techniques.
d. Understand intermediate centerline techniques.
e. Understand special circumstance dimensioning techniques.
f. Understand techniques for dimensioning holes and thread on technical drawings.

INSTRUCTIONAL ACTIVITIES-5.01	
A. Content Literacy Terminology	
Resource(s)	**(See 5.01.1)**
B. Review basic dimensioning technique.	
Activity	Guided Review and Tutorial Creation
Teacher Instructions	● Provide students with a copy of *Guided Review and Creation-Dimensions*. Facilitate students answering Pre-Questions as means of linking prior knowledge. Students should return to these questions after the PowerPoint Presentation to check answers. ● Facilitate whole-class or small-group instruction of foundational techniques using *Guided Review and Creation-Dimensions-Presentation*. Facilitate students recording guidelines as they appear. ● Facilitate whole-class, small group, pair or individual answering of Understand Questions. Formatively assess answers. ● Provide instruction for Demonstrate portion of activity. Assign students rules with emphasis on variety throughout class. Facilitate students creating a short video on their assigned guidelines using provided example objects as needed. Students will upload videos to FlipGrid. ● Facilitate students watching at least one video on all guidelines.
Student Directions	● Answer Pre-Questions ● Participate in review and record guidelines as they appear. Return to pre-questions after the PowerPoint Presentation to check answers. ● Answer Understand questions. ● Create a short video on your assigned guidelines using provided example objects as needed. Upload videos to FlipGrid. ● Watch at least one video on all guidelines.
Resource(s)	Guided Review and Creation- Dimensior Guided Review and Creation- Dimensior
Website Resource	
Flipgrid. (2021). Empower your voice.	https://info.flipgrid.com/
C. Understand intermediate hidden line techniques.	
Activity	Student Reasoning-Intermediate Hidden Line Technique
Teacher Instructions	● Provide copy of *Student Reasoning-Intermediate Hidden Line Technique*. Facilitate students reading through the table individually or in pairs and determining their reasoning for “why” each of the intermediate hidden line techniques. ● Facilitate small group or whole-class discussion on each technique when completed.
Student Directions	● Read through the table and determine reasoning for “why” each of the intermediate hidden line techniques. ● Discuss your reasoning.

Resource(s)	 Student Reasoning- Interme
D. Understand intermediate centerline techniques.	
Activity	Student Reasoning-Intermediate Centerline Technique
Teacher Instructions	● Provide copy of *Student Reasoning-Intermediate Centerline Technique*. Facilitate students reading through the table individually or in pairs and determining their reasoning for "why" each of the intermediate centerline techniques. ● Facilitate small group or whole-class discussion on each technique when completed.
Student Directions	● Read through the table and determine reasoning for "why" each of the intermediate centerline techniques. ● Discuss your reasoning.
Resource(s)	 Student Reasoning-Intermec
E. Understand special circumstance dimensioning technique. **F. Understand techniques for dimensioning holes and thread on technical drawings.** *(Note: Activity includes Unpacked Content for both E & F.)*	
Activity	Guided Questions-Special Circumstances and Intermediate Dimension Techniques-Leading Questions
Teacher Instructions	● Provide copies of *Guided Questions-Special Circumstances and Intermediate Dimension Techniques-Leading Questions* and associated *Guided Questions-Special Circumstances and Intermediate Dimension Techniques-PowerPoint Presentation*. ● Facilitate students answering corresponding leading questions with PowerPoint presentation. ● Facilitate small group or whole-class discussion on answers and each technique when completed.
Student Directions	● Progress through presentation while answering corresponding leading questions. ● Discuss answers and techniques.
Resource(s)	 Guided Questions- Special Circumstanc  Guided Questions- Special Circumstanc

Content Literacy Terminology-5.01.1	
Aligned Dimension	A linear dimension parallel to a line spanning the minimum distance between two measurement points.
Angular Dimension	A dimension that denotes the angle formed by two lines.
Arrowhead	Small triangular shape that serves as a termination point of a dimension line (Mechanical or ISO drawings) or leader.
Center Mark	Equal length t-shape that serves as a dimensioning reference to centers of holes and axes of symmetry.
Chamfer	Applies an angled corner to two intersecting lines or edges.
Diameter	The distance from the outer edge to outer edge of a circle through its center (Ø).
Dimension Line	Thin & dark, continuous lines that run between extension lines.
Dual Dimensioning-Bracket Method	Dimensioning which shows both metric and decimal inch dimensioning on the same drawing. Millimeter value is enclosed in square brackets.
Dual Dimensioning-Position Method	Dimensioning which shows both metric and decimal inch dimensioning on the same drawing. Millimeter value is placed above (or below) the inch value or separated by a dash.
Extension Line	Thin & dark, continuous lines that extend out past the feature being measured.
Fillets	Applies a rounded corner of a certain radius to two intersecting lines or edges.
Linear Dimension	A dimension/annotation with a thin, solid line extending from a note and terminates with an arrowhead.
Location Dimension	Label the location of each geometric feature within an object or view.
Radius	The distance from the outer edge of a circle to its center (R).
Rectangular Coordinate Dimensioning	Type of dimensioning system is specifically used when computer-controlled production machines are used to manufacture parts.
Round	Fillets created on convex edges.
Size Dimension	Label the length of the overall width, height, and depth of an object.
Tabular Dimensioning	Dimensioning is used when a series of parts consists of the same features or geometry but vary in dimension. Letters are used in place of dimension values, and the values are then placed in a table. Most standard parts are dimensioned this way in catalogs, the machinery handbook, and in the back of most textbooks.

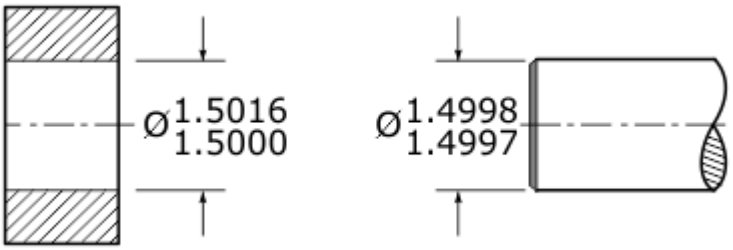
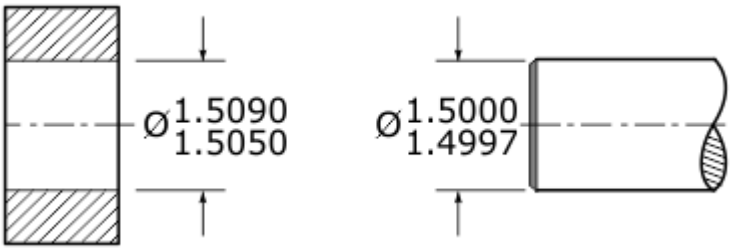
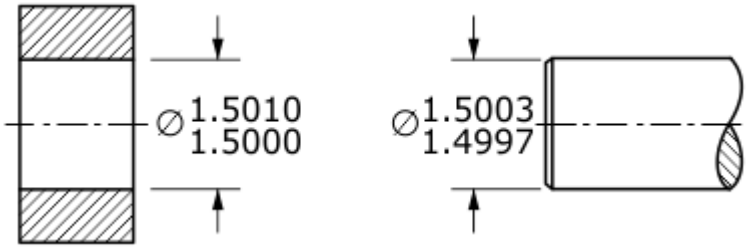
Course	**IV22 Drafting II - Engineering**			
Essential Standard	5.00	B2	15%	Understand Conventional Dimensioning & Tolerancing.
Indicator	5.02	N/A	N/A	Understand basic concepts of determining and creating tolerances.
Culminating Question **Essential Questions**	**How are tolerances determined and created in manufacturing?** ● What is the purpose of tolerance? ● What are the basic concepts associated with tolerances? ● How are tolerances determined?			

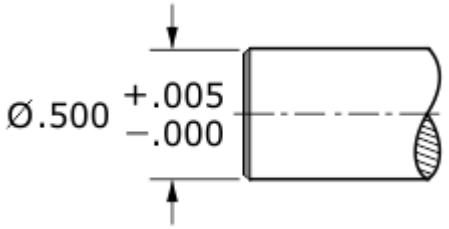
UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the purpose of tolerances.
c. Understand the basic concepts and terms associated with tolerances.
d. Understand how tolerances are determined.

INSTRUCTIONAL ACTIVITIES-5.02	
A. Content Literacy Terminology	
Resource(s)	**(See 5.02.1)**
B. Understand the purpose of tolerances.	
Activity	Small Group Guided Content Reading-Tolerances
Teacher Instructions	● Assign students into homogeneous groups (by reading level) ahead of time. Preselect articles related to topic at various reading levels. Provide students a copy of *Small Group Guided Content Reading-Tolerances*, links to articles, and 5 index cards for each student. ● Facilitate individuals defining pre-reading terms in Part One. Students may use the internet, professional texts or class textbooks to define terms. ● Students will select 5 excerpts from articles in which they find importance. ● Break class into small groups. Facilitate group discussion of selected experts for Part Three. Students will select the 5 most important excerpts. Facilitate students recording this information on a large format and identifying article information (board/construction paper). ● Facilitate groups sharing with the class recorded information. *Note:* Teacher can provide Content Literacy Terms before and/or after activity
Student Directions	● Define pre-reading terms in Part One using the internet, professional texts or class textbook. ● Read your provided article. ● Select five excerpts from articles with great importance. ● As a group, discuss selected experts for Part Three. Select the overall five most important excerpts Record this information on a large format and identify article information (board/construction paper). ● Share with the class your group's recorded information. .
Resource(s)	 Small Group Guided Content Rea
Website Resource(s): Example Articles	
Design World. (2012). Why It's Important to Always Use Tolerances.	https://www.designworldonline.com/why-its-important-to-always-use-tolerances/
Pacific Research Laboratories. (2021). Why are Tolerances Important in Manufacturing?	https://www.pacific-research.com/why-are-tolerances-important-in-manufacturing-prl/

Ritebearing Corporation. (2014). Important Benefits of Using Tolerances.	https://www.ritbearing.com/blog/archive/important-benefits-of-using-tolerances/
C. Understand the basic concepts and terms associated with tolerances. **D. Understand how tolerances are determined.** *(Note: Activity includes Unpacked Content for both C & D.)*	
Activity	Guided Questions-Tolerances & Fits
Teacher Instructions	● Provide copies of *Guided Questions- Tolerances & Fits- Leading Questions* and *Guided Questions- Tolerances & Fits- Presentation. Students*. Facilitate students answering corresponding leading questions while progressing through PowerPoint Presentation. ● Facilitate small group or whole-class discussion on each tolerance concept & fit when completed.
Student Directions	● Progress through PowerPoint Presentation, individually, answering corresponding leading questions. ● Discuss each tolerance concept & fit.
Resource(s)	 Guided Questions- Tolerances & Fits- L  Guided Questions- Tolerances & Fits- P

Content Literacy Terminology-5.02	
Basic Hole System	The design size of the hole is the basic size, and the allowance is applied to the shaft. 
Basic Shaft System	The design size of the shaft is the basic size, and the allowance is applied to the hole. 
Basic Size	The basic size is the size to which allowances and tolerances are added to get the limits of size.
Bilateral Tolerance	Tolerance allows variation in both directions.
Bilateral Unequal Tolerance	A tolerance that allows variation in both directions but is not equal in both directions.
Clearance Fit	An internal member fits in an external member and always leaves a space between the parts.
Design Size	The design size is the size from which the limits of size are derived by the application of tolerances.
Interference Fit	An internal member is larger than the external member and surface collision is the result.
Limit dimension	A tolerancing method showing the maximum and minimum size values. The maximum dimension is placed above the minimum dimension. When expressed in a single line, the lower limit precedes the upper limit.
Nominal Size	Designation used for the purpose of general identification (usually expressed as a fraction).
Tolerance	Amount a specific dimension can vary (the difference between the limits).
Transition Fit	An internal member is sized such that clearance or interference may result with the external member after manufacturing. 

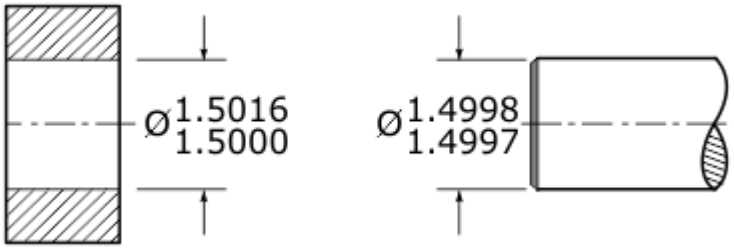
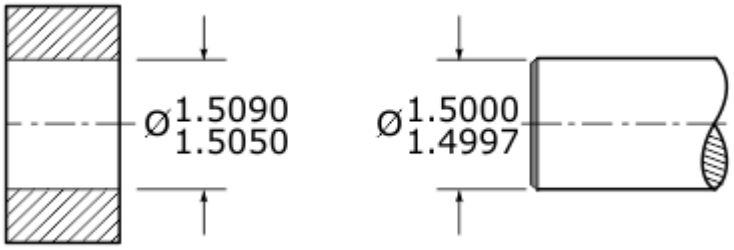
Unilateral Tolerance	Tolerance that allows variation in one direction. 

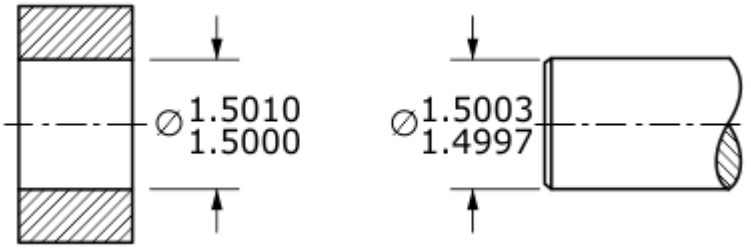
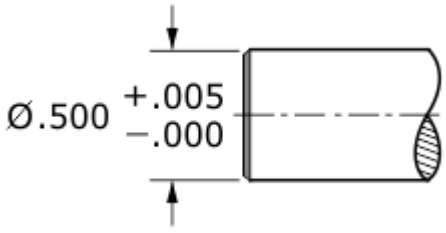
Course	**IV22 Drafting II - Engineering**			
Essential Standard	5.00	B2	15%	Understand Conventional Dimensioning & Tolerancing.
Indicator	5.03	N/A	N/A	Understand drawings with intermediate &/or tolerance dimensions.
Culminating Question **Essential Questions**	**How are tolerances and intermediate dimension techniques represented on technical drawings?** ● How are intermediate dimensions interpreted on technical drawings? ● How are tolerance dimensions interpreted on technical drawings?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand drawings with intermediate dimensions.
c. Understand drawings with tolerance dimensions.

INSTRUCTIONAL ACTIVITIES-5.03	
A. Content Literacy Terminology	
Resource(s)	**(See 5.03.1)**
B. Understand drawings with intermediate dimensions.	
Activity	Linking Industry Examples-Intermediate Dimensions
Teacher Instructions	● Provide students with *Linking Industry Examples- Intermediate Dimensions* and review instructions for the project. Facilitate students completing a presentation on concepts and locating real-world industry examples. Emphasize students should select *technical drawings* that display concepts and not just images. ● Facilitate students sharing final presentations in small-groups/pairs or combine presentations from student examples and share with the whole-class as review.
Student Directions	● Create a presentation on concepts and locate real-world industry examples. Select *technical drawings* that display concepts and not just images. ● Share presentations.
Resource(s)	 Linking Industry Examples- Intermedi
C. Understand drawings with tolerance dimensions.	
Activity	Linking Industry Examples-Tolerance
Teacher Instructions	● Provide students with *Linking Industry Examples- Tolerance Dimensions* and review instructions for the project. Facilitate students completing a presentation on concepts and locating real-world industry examples. Emphasize students should select *technical drawings* that display concepts and not just images. ● Facilitate students sharing final presentations in small-groups/pairs or combine presentations from student examples and share with the whole-class as review.
Student Directions	● Create a presentation on concepts and locate real-world industry examples. Select *technical drawings* that display concepts and not just images. ● Share presentations.
Resource(s)	 Linking Industry Examples- Tolerance

Content Literacy Terminology-5.03	
Aligned Dimension	A linear dimension parallel to a line spanning the minimum distance between two measurement points.
Angular Dimension	A dimension that denotes the angle formed by two lines.
Arrowhead	Small triangular shape that serves as a termination point of a dimension line (Mechanical or ISO drawings) or leader.
Basic Dimensioning	A numerical value used to describe the theoretically exact size, profile, orientation, or location of a feature.
Basic Hole System	The design size of the hole is the basic size, and the allowance is applied to the shaft. 
Basic Shaft System	The design size of the shaft is the basic size, and the allowance is applied to the hole. 
Basic Size	The basic size is the size to which allowances and tolerances are added to get the limits of size.
Bilateral Tolerance	Tolerance allows variation in both directions.
Bilateral Unequal Tolerance	A tolerance that allows variation in both directions but is not equal in both directions.
Center Mark	Equal length t-shape that serves as a dimensioning reference to centers of holes and axes of symmetry.
Chamfer	Applies an angled corner to two intersecting lines or edges.
Clearance Fit	An internal member fits in an external member and always leaves a space between the parts.
Design Size	The design size is the size from which the limits of size are derived by the application of tolerances.
Diameter	The distance from the outer edge to outer edge of a circle through its center (Ø).
Dimension Line	Thin & dark, continuous lines that runs between extension lines.
Dual Dimensioning-Bracket Method	Dimensioning which shows both metric and decimal inch dimensioning on the same drawing. Millimeter value is enclosed in square brackets.
Dual Dimensioning-Position Method	Dimensioning which shows both metric and decimal inch dimensioning on the same drawing. Millimeter value is placed above (or below) the inch value or separated by a dash.
Extension Line	Thin & dark, continuous lines that extend out past the feature being measured.

Fillets	Applies a rounded corner of a certain radius to two intersecting lines or edges.
Interference Fit	An internal member is larger than the external member and surface collision is the result.
Limit dimension	A tolerancing method showing the maximum and minimum size values. The maximum dimension is placed above the minimum dimension. When expressed in a single line, the lower limit precedes the upper limit.
Linear Dimension	A dimension/annotation with a thin, solid line extending from a note and terminates with an arrowhead.
Location Dimension	Label the location of each geometric feature within an object or view.
Nominal Size	Designation used for the purpose of general identification (usually expressed as a fraction).
Radius	The distance from the outer edge of a circle to its center (R).
Rectangular Coordinate Dimensioning	Type of dimensioning system is specifically used when computer-controlled production machines are used to manufacture parts.
Round	Fillets created on convex edges.
Size Dimension	Label the length of the overall width, height, and depth of an object.
Tabular Dimensioning	Dimensioning is used when a series of parts consists of the same features or geometry but vary in dimension. Letters are used in place of dimension values, and the values are then placed in a table. Most standard parts are dimensioned this way in catalogs, the machinery handbook, and in the back of most textbooks.
Tolerance	Amount a specific dimension can vary (the difference between the limits).
Transition Fit	An internal member is sized such that clearance or interference may result with the external member after manufacturing. 
Unilateral Tolerance	Tolerance that allows variation in one direction. 

Course	IV22 Drafting II - Engineering			
Essential Standard	6.00	C3	30%	Apply Procedures to Create Working Drawings of a 3D Model.
Indicator	6.01	N/A	N/A	Apply the concepts and techniques of creating working drawings of single parts.
Culminating Question **Essential Questions**	**How are the concepts and techniques of creating working drawings of single parts used in the software?** ● What are detail drawings and how are they used in Manufacturing? ● What type of information is displayed on detail drawings?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand what detail drawings are used for in manufacturing.
c. Understand the type of information included on detail drawings.

INSTRUCTIONAL ACTIVITIES-6.01	
A. Content Literacy Terminology	
Resource(s)	**(See 6.01.1)**
B. Understand what detail drawings are used for in manufacturing.	
Activity	Teacher Led-Introduction
Teacher Instructions	● Facilitate introduction to detail drawings using *Guided Presentation-Detail Working Drawings of a 3D Model*. Students should follow along with a copy of the presentation.
Student Directions	● Participate in whole-class instruction on Detail Drawings to name the main parts of a detail drawing and their purpose.
Resource(s)	 Guided Prestentation- Detai
C. Understand the type of information included on detail drawings.	
Activity	Print Reading for Comprehension
Teacher Instructions	● Provide students with a copy of *Print Reading for Comprehension-Cap Block-Detail Drawings*. Facilitate students completing questions and checking answers in small groups, partners, or whole-class. ● Provide students with a copy of *Print Reading for Comprehension-Rear Packing Gland-Detail Drawings*. Facilitate students completing questions and checking answers in small groups, partners, or whole-class. *Note:* Second assignment can be used as an assessment of understanding.
Student Directions	● Complete *Print Reading for Comprehension-Cap Block-Detail Drawings* questions and check answers. ● Complete *Print Reading for Print Reading for Comprehension-Rear Packing Gland-Detail Drawings* questions and check answers.
Resource(s)	 Print Reading for Comprehension- Ca  Print Reading for Comprehension- Re

Content Literacy Terminology-6.01.1	
Assembly Drawing	A technical drawing which includes view(s) to describe assembled positions, identify parts by number, and have a parts list.
Detail Drawing	A technical drawing which provides a detailed description of the geometric form of a part including dimensions.
Detail View	An enlarged view of a specified portion of another drawing view.
Drawing File .idw or .dwg	Primary file type in which sheets/documents are set-up and working drawings sets are created.
Exploded Assembly Drawing	A technical drawing which includes view(s) to describe assemblies by moving components out from their assembled position.
JPEG	A type of image file format.
PDF	(Portable Document Format) A file format used to save files that cannot be modified but still easily shared and printed.
Presentation File .ipn	Primary file type in which are used to create an exploded view or animation.
Title Block	The area on a drawing sheet that identifies the owner, includes a description of the drawing, and provides other relevant information.
Working Drawings	The set of technical drawings used during the manufacturing phase of a product.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	6.00	C3	30%	Apply Procedures to Create Working Drawings of a 3D Model.
Indicator	6.02	N/A	N/A	Apply the concepts and techniques of creating multi-view projection and the various views that are needed to document a 3D model including: Base Views and Projected Views.
Culminating Question	**How are the concepts and techniques of creating multi-view projection and the various views that are needed to document a 3D model including: Base Views and Projected Views used in the software?**			
Essential Questions	● What are the types of views and drawings used in manufacturing? ● How are views of an object created in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the types of views and drawings used in manufacturing.
c. Understand how to create multiple types of views in the software.

INSTRUCTIONAL ACTIVITIES-6.02	
A. Content Literacy Terminology	
Resource(s)	**(See 6.02.1)**
B. Understand what detail drawings are used for in manufacturing.	
Activity	Teacher Led Review of Material-Types of Drawings
Teacher Instructions	● Facilitate review of material using *Teacher Led Review of Material-Types of Drawings-PowerPoint Presentation*.
Student Directions	● Participate in whole-class instruction on Types of Drawings to name the types of drawings and views used in manufacturing.
Resource(s)	 Teacher Led Review of Material- Types o
C. Understand how to create multiple types of views in the software.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying various views.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-6.02.1	
Base View	The primary view first created in a drawing and source of subsequent views.
Cutting Plane	Plane which creates a slice into an object or entity.
Cutting Plane Line	Lines used to indicate where the section or cut is made. Arrowheads on a cutting plane line indicate the direction of sight.
Detail Drawing	A technical drawing which provides a detailed description of the geometric form of a part including dimensions.
Detail View	An enlarged view of a specified portion of another drawing view.
Drawing File .idw or .dwg	Primary file type in which sheets/documents are set-up and working drawings sets are created.
Foreshorten Surface	A surface that is not true size or not true shape.
Inclined Surface	A surface that's true size and shape cannot be viewed in frontal, horizontal, or profile planes.
Isometric View	A type of view where a 3D model is situated so that each of the three fundamental model planes (X, Y, and Z) has an equal degree of exposure relative to the view angle of the observer.
JPEG	A type of image file format .
Line of sight "LOS"	Represents the direction you are looking at an object.
PDF	(Portable Document Format) a file format used to save files that cannot be modified, but still easily shared and printed.
Presentation File .ipn	Primary file type in which are used to create an exploded view or animation.
Projected View	An orthographic or isometric view that is generated from a base view or other existing view.
Reference Plane or Reference Line	A plane or line that is parallel to the inclined surfaces.
Section	Drawing which shows interior detail or space of an object or entity.
Title Block	The area on a drawing sheet that identifies the owner, includes a description of the drawing, and provides other relevant information.

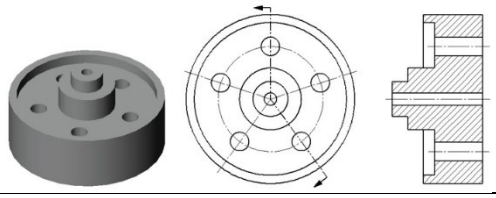
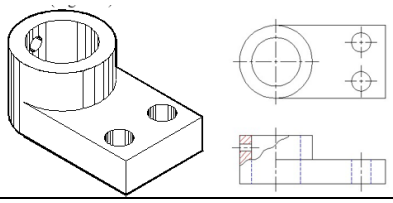
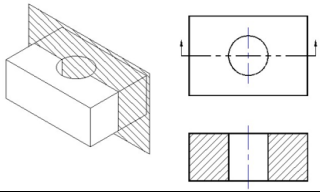
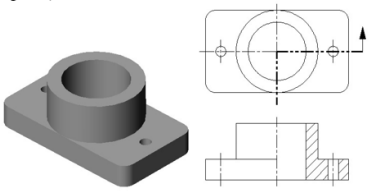
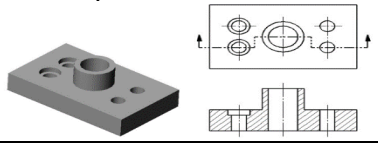
Course	**IV22 Drafting II - Engineering**			
Essential Standard	6.00	C3	30%	Apply Procedures to Create Working Drawings of a 3D Model.
Indicator	6.03	N/A	N/A	Apply concepts of Section Views.
Culminating Question **Essential Questions**	**How are the concepts and techniques of creating section views used in the software?** ● Why are section views included in Working Drawings? ● What are the major types of section views used in Technical Drawing? ● What are special circumstances that apply to section views for specific parts/features? ● How are section views of an object created in the software?			

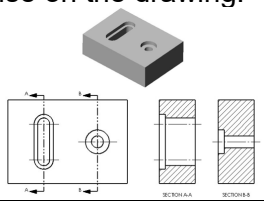
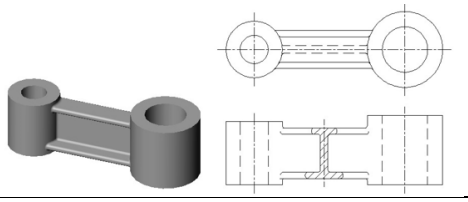
UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the purpose of section views and major concepts associated with them.
c. Understand the major types of section views used in Technical Drawings.
d. Understand special circumstances that apply to section views.
e. Understand how section views are created in the software.

INSTRUCTIONAL ACTIVITIES-6.03	
A. Content Literacy Terminology	
Resource(s)	**(See 6.03.1)**
B. Understand the purpose of section views and major concepts associated with them. **C. Understand the major types of section views used in Technical Drawings.** *Note: Activity includes Unpacked Content for both B & C.*	
Activity	Independent Practice-Linking Prior and New Knowledge-Sections
Teacher Instructions	● Provide a copy of *Independent Practice- Linking Prior and New Knowledge- Sections*. Facilitate students timing themselves reading Part 1 and recording progress. Have students write the main idea of the paragraph in their words after reading. ● Facilitate student-led assessment of concepts in Part Two where. students will identify terms they feel as though they "can define", "have seen or heard" or "no clue." ● Facilitate students defining each term and find a synonym for words identified as "no clue." ● Facilitate student research on terms in Part Three. Students can share information in pairs or small-groups. ● Review all concepts using student examples and correct definitions when completed.
Student Directions	● Time yourself reading Part 1 and record progress. ● Write the main idea of the paragraph after reading. ● Identify terms as "can define", "have seen or heard" or "no clue." ● Define each term and find a synonym for words identified as "no clue." ● Research terms in part 3. Share information in pairs or small-groups. ● Participate in review of all concepts, examples and correct definitions.
Resource(s)	 Independent Practice- Linking Pric
D. Understand special circumstances that apply to section views.	
Activity	Student-Led Discovery-Conventions in Sections
Teacher Instructions	● Provide copy of *Student-Led Discovery-Conventions in Sections*. Facilitate students completing presentation/research. ● Review all concepts using student examples and correct definitions when completed.
Student Directions	● Complete presentation/research outlined in *Student-Led Discovery-Conventions in Sections.* ● Identify major concepts associated with special circumstances in sections with review.

Resource(s)	 Student-Led Discovery- Conventi
E. Understand special circumstances that apply to section views.	
Activity	Software Tutorials
Teacher Instructions	• Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	• Create multiple parts/projects applying Section Views.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-6.03.1	
Aligned Section	Sectional view used to include details of a part by “bending” the cutting plane. Commonly used to section a round object with asymmetrical spokes. 
Broken-out Section	Sectional view used when only a portion of the object’s interior shapes needs to be sectioned. A broken-out section may be used when a full section or half section is not necessary. A freehand break line is used to separate the sectioned details from the non-sectioned parts. 
Cutting Plane	Plane which creates a slice into an object or entity.
Cutting Plane Line	Lines used to indicate where the section or cut is made. Arrowheads on a cutting plane line indicate the direction of sight.
Full Section	Sectional view in which half of the object is imagined to be cut away. 
Half Section	Sectional view in which a quarter of the object is imagined to be cut away. Commonly used in symmetrical objects or entities. 
Hatch	Pattern fill of an established area with boundaries.
Lug	Projection built onto part used for the purpose of attachment.
Offset Section	Sectional view in which the cutting plane is bent or “offset”. The change of plane that occurs when the cutting plane is bent 90 degrees is not represented with lines in the sectional view. 

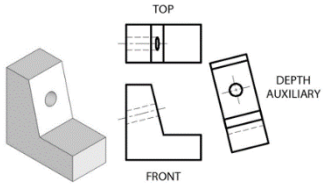
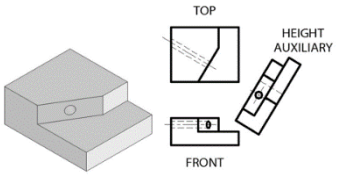
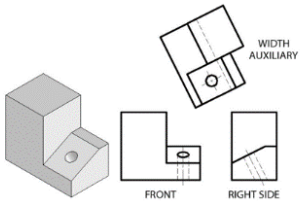
Removed Section	Sectional view is taken from its normal position and shown somewhere else on the drawing. 
Revolved/Rotated Section	Sectional view used to represent the cross-sectional shape of a handle, spoke, or other elongated features. Create the section by cutting through the feature and revolving the section 90 degrees. Superimpose the sectional view on the orthographic view. 
Rib	Long raised piece of stronger or thicker material across a surface or through a structure, and typically serving to support or strengthen between two planes.
Section	Drawing which shows interior detail or space of an object or entity.
Section Line	Thin, visibly lined areas which represent materials that have been in contact with the cutting plane line. The general universal section line symbol is uniformly spaced at 45 degrees. Instead of using section lines, parts may be shaded with a solid fill or hatch. Do not draw section lines parallel or perpendicular to visible edges.
Spoke	Rod radiating from middle (hub) to edge (rim) of a wheel serving to support.
Web	Long raised piece of stronger or thicker material across a surface or through a structure, and typically serving to support or strengthen between three or more plans.

Course	IV22 Drafting II - Engineering			
Essential Standard	6.00	C3	30%	Apply Procedures to Create Working Drawings of a 3D Model.
Indicator	6.04	N/A	N/A	Apply concepts of Auxiliary Views.
Culminating Question **Essential Questions**	**How are the concepts and techniques of creating auxiliary views used in the software?** ● What is the purpose of an auxiliary view? ● What are types of auxiliary views and drawings used in manufacturing? ● How are auxiliary views determined for an object? ● How are auxiliary views of an object created in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the purpose of auxiliary views.
c. Understand the main types of auxiliary views.
d. Determine correct auxiliary views for an object.
e. Understand how to create auxiliary views in the software.

INSTRUCTIONAL ACTIVITIES-6.04	
A. Content Literacy Terminology	
Resource(s)	**(See 6.04.1)**
B. Understand the purpose of auxiliary views. **C. Understand the main types of auxiliary views.** *Note: Activity includes Unpacked Content for both B & C.*	
Activity	Guided Presentation-Auxiliary Drawings
Teacher Instructions	● Facilitate whole-class guided instruction using *Guided Presentation-Auxiliary Drawings-PowerPoint Presentation*.
Student Directions	● Name the types of auxiliary views used in manufacturing by participating in whole-class instruction on types of drawings.
Resource(s)	Guided Presentation- Auxili
D. Determine correct auxiliary views for an object.	
Activity	Student Practice-Auxiliary View Matching
Teacher Instructions	● Provide copy of *Student Practice- Auxiliary View Matching*. Facilitate students numbering scrap paper 1-10. Facilitate students matching examples and recording selections. Students can work in small-groups, pairs, or individually. ● Provide formative assessment as needed for students.
Student Directions	● Number scrap paper 1-10. Determine auxiliary views for various objects by recording selections for each auxiliary drawing.
Resource(s)	Student Practice- Auxiliary View Matcl
E. Understand how to create auxiliary views in the software.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying auxiliary views..
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-6.04.1	
Auxiliary View	An orthographic view that is drawn on any plane other than from the frontal, horizontal, or profile plane.
Base View	The primary view first created in a drawing and source of subsequent views.
Depth Auxiliary	The inclined surface will appear foreshortened in a top view and a right-side view. The inclined surface will appear as a line or edge in the front view. 
Elevation/Height Auxiliary	The inclined surface will appear foreshortened in a front view and a right-side view. The inclined surface will appear as a line or edge in the top view. 
Foreshorten Surface	A surface that is not true size or not true shape.
Inclined Surface	A surface that's true size and shape cannot be viewed in frontal, horizontal, or profile planes.
Isometric View	A type of view where a 3D model is situated so that each of the three fundamental model planes (X, Y, and Z) has an equal degree of exposure relative to the view angle of the observer.
Line of sight "LOS"	Represents the direction you are looking at an object.
Primary Auxiliary View	A single view projected from one of the six principal views that provides a true size and true shape image of an inclined surface.
Projected View	An orthographic or isometric view that is generated from a base view or other existing view.
Reference Plane or Reference Line	A plane or line that is parallel to the inclined surfaces.
True Size and Shape	A surface that is perpendicular to the projection plane.
Width Auxiliary	The inclined surface will appear foreshortened in a top view and a front view. The inclined surface will appear as a line or edge in the right or left-side view. 

Course	**Drafting II- Engineering**			
Essential Standard	7.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Model.
Indicator	7.01	N/A	N/A	Apply the concepts and techniques of creating Working Drawings of single parts.
Culminating Question	**How are the concepts and techniques of creating Working Drawings of single parts used in the software?**			
Essential Questions	● How are concepts of creating Working Drawings of a single part applied in the software? ● How are techniques for creating Working Drawings of a single part applied in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply concepts of creating Working Drawings of single parts.
c. Apply techniques for creating Working Drawings of single parts.

INSTRUCTIONAL ACTIVITIES-7.01	
A. Content Literacy Terminology	
Resource(s)	**(See 7.01.1)**
B. Apply concepts of creating Working Drawings of single parts. **C. Apply techniques for creating Working Drawings of single parts.** *Note: Activity includes Unpacked Content for both B & C.*	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU. *Additional Resource: Student Application-Isometric Part Examples* is available for examples of some concepts, but not all in this indicator (originally sourced from: Spencer, H. C., Dygdon, J. T., & Novak, J. E. (1995). Basic Technical Drawing.) (Activity can combine with Indicators 4.03, 4.04, 7.01, 7.02, 7.03, & 7.05)
Student Directions	● Create multiple parts/projects applying the creation of Working Drawings.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-7.01.1	
Assembly Drawing	A technical drawing which includes view(s) to describe assembled positions, identify parts by number, and have a parts list.
Detail Drawing	A technical drawing which provides a detailed description of the geometric form of a part including dimensions.
Detail View	An enlarged view of a specified portion of another drawing view.
Drawing File .idw or .dwg	Primary file type in which sheets/documents are set-up and working drawings sets are created.
Exploded Assembly Drawing	A technical drawing which includes view(s) to describe assemblies by moving components out from their assembled position.
JPEG	A type of image file format.
PDF	(Portable Document Format) a file format used to save files that cannot be modified, but still easily shared and printed.
Presentation File .ipn	Primary file type in which are used to create an exploded view or animation.
Title Block	The area on a drawing sheet that identifies the owner, includes a description of the drawing, and provides other relevant information.
Working Drawings	The set of technical drawings used during the manufacturing phase of a product.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	7.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Mode.
Indicator	7.02	N/A	N/A	Apply the concepts and techniques of creating Multi-view projection and the various views that are needed to document a 3D model including: Base Views and Projected Views.
Culminating Question	**How are the concepts and techniques of creating Multi-view Projection and the various views that are needed to document a 3D model including: Base Views and Projected Views used in the software?**			
Essential Questions	● What types of views and drawings are used in manufacturing? ● How are views of an object created in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply the types of views and drawings used in manufacturing.
c. Apply multiple types of views in the software.

INSTRUCTIONAL ACTIVITIES-7.02	
A. Content Literacy Terminology	
Resource(s)	**(See 7.02.1)**
B. Apply the types of views and drawings used in manufacturing. **C. Apply multiple types of views in the software.** *Note: Activity includes Unpacked Content for both B & C.*	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU. *Additional Resource: Student Application-Isometric Part Examples* is available for examples of some concepts, but not all in this indicator (originally sourced from: Spencer, H. C., Dygdon, J. T., & Novak, J. E. (1995). Basic Technical Drawing.) (Activity can combine with Indicators 4.03, 4.04, 7.01, 7.02, 7.03, & 7.05)
Student Directions	● Create multiple parts/projects applying multiple types of views used in manufacturing.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

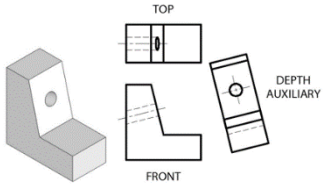
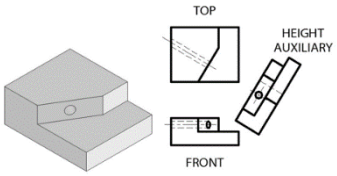
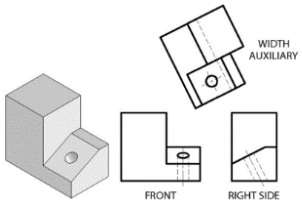
Content Literacy Terminology- 7.02.1	
Base View	The primary view first created in a drawing and source of subsequent views.
Cutting Plane	Plane which creates a slice into an object or entity.
Cutting Plane Line	Lines used to indicate where the section or cut is made. Arrowheads on a cutting plane line indicate the direction of sight.
Detail Drawing	A technical drawing which provides a detailed description of the geometric form of a part including dimensions.
Detail View	An enlarged view of a specified portion of another drawing view.
Drawing File .idw or .dwg	Primary file type in which sheets/documents are set-up and working drawings sets are created.
Foreshorten Surface	A surface that is not true size or not true shape.
Inclined Surface	A surface that's true size and shape cannot be viewed in frontal, horizontal, or profile planes.
Isometric View	A type of view where a 3D model is situated so that each of the three fundamental model planes (X, Y, and Z) has an equal degree of exposure relative to the view angle of the observer.
JPEG	A type of image file format.
Line of sight "LOS"	Represents the direction you are looking at an object.
PDF	(Portable Document Format) a file format used to save files that cannot be modified, but still easily shared and printed.
Presentation File .ipn	Primary file type in which are used to create an exploded view or animation.
Projected View	An orthographic or isometric view that is generated from a base view or other existing view.
Reference Plane or Reference Line	A plane or line that is parallel to the inclined surfaces.
Section	Drawing which shows interior detail or space of an object or entity.
Title Block	The area on a drawing sheet that identifies the owner, includes a description of the drawing, and provides other relevant information.

Course	**IV22 Drafting II - Engineering**			
Essential Standard	7.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Mode.
Indicator	7.04	N/A	N/A	Apply concepts of Auxiliary Views.
Culminating Question **Essential Questions**	**How are the concepts and techniques of creating auxiliary views in the software?** ● How are auxiliary views created for an object by hand? ● How are auxiliary views of an object created in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply procedures for creating auxiliary views by hand.
c. Apply auxiliary views in the software.

INSTRUCTIONAL ACTIVITIES-7.04	
A. Content Literacy Terminology	
Resource(s)	**(See 7.04.1)**
B. Apply procedures for creating auxiliary views by hand.	
Activity	Guided Presentation-Auxiliary Drawings
Teacher Instructions	● Provide copies of *Guided Directions- Creating an Auxiliary View* and *Guided Directions- Practice- Auxiliary Views Activity*. Facilitate student creation of the first auxiliary example as a whole-class or small-group instruction following the provided steps. ● Facilitate students creating second example independently and check for correctness using *Guided Directions-Answers*. Students should also answer conclusion questions. ● Facilitate review of conclusion questions and provide formative feedback or assistance as needed.
Student Directions	● Create the first auxiliary example as a whole-class or small-group following the provided steps. ● Create the second example independently and check for correctness using *Guided Directions-Answers*. ● Answer conclusion questions. ● Participate in review of conclusion questions.
Resource(s)	Guided Directions of Creating an Auxiliary ' Guided Directions-Answers.docx Guided Directions-Practice- Auxiliary Vie
C. Apply auxiliary views in the software.	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying Auxiliary Views..
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource Independent Practice- Auxiliary Vie

Content Literacy Terminology-7.04.1	
Auxiliary View	An orthographic view that is drawn on any plane other than from the frontal, horizontal, or profile plane.
Base View	The primary view first created in a drawing and source of subsequent views.
Depth Auxiliary	The inclined surface will appear foreshortened in a top view and a right-side view. The inclined surface will appear as a line or edge in the front view. 
Elevation/Height Auxiliary	The inclined surface will appear foreshortened in a front view and a right-side view. The inclined surface will appear as a line or edge in the top view. 
Foreshorten Surface	A surface that is not true size or not true shape.
Inclined Surface	A surface that's true size and shape cannot be viewed in frontal, horizontal, or profile planes.
Isometric View	A type of view where a 3D model is situated so that each of the three fundamental model planes (X, Y, and Z) has an equal degree of exposure relative to the view angle of the observer.
Line of sight "LOS"	Represents the direction you are looking at an object.
Primary Auxiliary View	A single view projected from one of the six principal views that provides a true size and true shape image of an inclined surface.
Projected View	An orthographic or isometric view that is generated from a base view or other existing view.
Reference Plane or Reference Line	A plane or line that is parallel to the inclined surfaces.
True Size and Shape	A surface that is perpendicular to the projection plane.
Width Auxiliary	The inclined surface will appear foreshortened in a top view and a front view. The inclined surface will appear as a line or edge in the right or left-side view. 

Course	**IV22 Drafting II - Engineering**			
Essential Standard	7.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Mode.
Indicator	7.05	N/A	N/A	Apply the techniques of adding annotations, dimensions and tolerances, using ANSI standards that best describe manufacturing processes.
Culminating Question **Essential Questions**	**How are the techniques of adding annotations, dimensions and tolerances, using ANSI standards that best describes manufacturing processes applied in the software?** ● How are the techniques for adding Annotations and dimensions using ANSI standards to basic parts applied? ● How are the techniques of adding tolerances, using ANSI standards applied to manufacturing drawings?			

UNPACKED CONTENT

a. Apply the techniques of adding annotations and dimensions using ANSI standards to basic parts.
b. Apply the techniques of adding tolerances, using ANSI standards to manufacturing drawings.
c. Apply the techniques of adding annotations, dimensions and tolerances, using ANSI standards that best describe manufacturing processes.

INSTRUCTIONAL ACTIVITIES-7.05	
D. Apply the techniques of adding annotations and dimensions using ANSI standards to basic parts. **E. Apply the techniques of adding tolerances, using ANSI standards to manufacturing drawings.** **F. Apply the techniques of adding annotations, dimensions and tolerances, using ANSI standards that best describes manufacturing processes.** *Note: Activity includes Unpacked Content for both A, B & C.*	
Activity	Software Tutorials
Teacher Instructions	● Facilitate students completing video or written tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying multiple types of annotations, dimensions, and tolerances.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource