Drafting III - Engineering

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

IV23

PUBLIC SCHOOLS OF NORTH CAROLINA
State Board of Education | Department of Public Instruction
www.dpi.nc.gov

Introduction

This curriculum guide for IV23 Drafting III - 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 Requirements for Engineering and Manufacturing Employment

2.00 Apply Engineering Design Concepts and principles to solve problems

3.00 Understand Advanced Manufacturing Processes

4.00 Apply Advanced 3D Parametric-Solid Model and Assembly Creation Techniques

5.00 Apply Procedures to Create Working Drawings of a 3D Model and Assembly

6.00 Apply Procedures to Create Sheet Metal Parts

7.00 Apply Procedures to create Geometric Dimensions and Tolerances

8.00 Apply Procedures to create a professional Portfolio

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 Professional- Inventor or Certified SolidWorks Professional).

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 Indicators 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 and 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/Indicator.

Additional Resources

Textbook & Online

Autodesk and SolidWorks provide content. Previous Curriculum Guides for the 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 the Curriculum Guide. For more information on SkillsUSA visit: https://www.skillsusa.org/.

Acknowledgements

North Carolina TTEI would like to thank the following educators who assisted with the development of this course:

- Daron Atkins, Surry County Schools
- Jeremiah Blango, Cabarrus County Schools
- Darrin Bridges, Mooresville Graded School District
- Blair Deen, New Hanover County Schools
- Cary Gluf, Cabarrus County Schools
- Matt Hall, Rockingham County Schools
- Stephen Herrington, Onslow County Schools
- David Lambert, Guilford County Schools
- Kim Osborne, Guilford County Schools
- Andy Owens, Catawba County Schools
- Christy Pieper, Cabarrus County Schools
- Paul Satenstein, Cabarrus County Schools
- Stephen Thacker, Cabarrus County Schools
- Nancy Wills, Cabarrus County Schools

A special thank you goes to the following business and industry representatives who assisted in the development of the course content:

- Kris Dell, Territory Account Consultant, Applied Software
- Mike Leary, Territory Sales Senior Manager, SolidWorks Education

State Staff for Career and Technical Education
Craig Pendergraft, Trade, Technology, Engineering, and Industrial Education Consultant
Paul Heidepriem, Executive Director, North Carolina SkillsUSA
Delores P. Ali, Program Director for CTE Curriculum
Felicia Gray-Watson, Program Director for CTE Special Projects
Angela LeMay, Program Director for CTE Assessment and Human Capital
Kimberly MacDonald, Program Director for CTE Budget, Reporting and Analysis
Marty Tobey, Program Director for CTE Regional Services
Trey Michael, CTE State Director

Course	**IV23 Drafting III - Engineering**			
Essential Standard	1.00	B2	5%	Understand Requirements for Engineering and Manufacturing Employment.
Indicator	1.01	N/A	N/A	Understand the requirements of becoming a professional Engineer.
Culminating Question **Essential Question**	**What are the requirements to become a professional Engineer?** ● How can you complete a path to become an engineer?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand career pathways for Engineers.

INSTRUCTIONAL ACTIVITIES-1.01	
A. Content Literacy Terminology	
Resource(s)	**(See 1.01.1)**
B. Understand career pathways for Engineers. *Note: Activity combines with Indicator 1.02.*	
Activity	Self-Reflection and Research-Careers
Teacher Instructions	● Provide access to S*elf-Reflection and Research- Careers*. Facilitate students acquiring personal statistics and information for Parts One and Two. ● Facilitate students completing research using the internet on provided careers in Part Three. ● Facilitate students answering questions in Part Four using research and personal statistics. Provide websites/resources for students as needed.
Student Directions	● Acquire personal statistics and information for Parts One and Two. ● Complete research using the internet on provided careers in Part Three. ● Outline your progress towards a degree in Engineering by answering questions in Part Four using research and personal statistics.
Resource(s)	 Self Reflection and Research- Careers.d

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.
Manufacturing	The production of products/articles on a large scale, using labor and machinery, tools, and chemical or biological processing.
Types of Degrees/Certifications	Certification, License, Associates, Bachelors, Masters, and Doctorate

Course	IV23 Drafting III - Engineering			
Essential Standard	1.00	B2	5%	Understand Requirements for Engineering & Manufacturing Employment.
Indicator	1.02	N/A	N/A	Understand Education and Professional qualifications for becoming employed in Engineering, Manufacturing, or related fields. ● Engineering Services: Civil, Electrical, Mechanical ● Machine Tool Designers ● Machinist / CNC technician ● Certified Welder ● Marine Engineering Services ● Petroleum Engineering Services ● Aerospace Engineering ● Industrial / Product Design ● New and Emerging Technologies
Culminating Question **Essential Questions**	**What are the educational and professional qualifications for becoming employed in engineering, manufacturing, or related fields?** ● What qualifications are required to achieve careers in various engineering and manufacturing fields? ● What are some new and emerging technologies in engineering career fields?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand qualifications to achieve careers in various engineering and manufacturing fields.
c. Understand new and emerging technologies in engineering and manufacturing fields.

INSTRUCTIONAL ACTIVITIES-1.02	
A. Content Literacy Terminology	
Resource(s)	**(See 1.02.1)**
B. Understand qualifications to achieve careers in various engineering fields. **C. Understand new and emerging technologies in engineering and manufacturing fields.** *Note: Activity includes all Unpacked Content for Indicator and combines with Indicator 1.01.*	
Activity	Self-Reflection and Research-Careers
Teacher Instructions	● Provide access to S*elf-Reflection and Research- Careers*. Facilitate students acquiring personal statistics and information for Parts One and Two. ● Facilitate students completing research using the internet on provided careers in Part Three. ● Facilitate students answering questions in Part Four using research and personal statistics. Websites/resources can be provided as needed.
Student Directions	● Acquire personal statistics and information for Parts One & Two. ● Complete research using the internet on provided careers in Part Three. ● Outline your progress towards a degree in Engineering by answering questions in Part Four using research and personal statistics.
Resource(s)	Self Reflection and Research- Careers.d IV23 Drafting III Engineering - 1.02.p

Content Literacy Terminology-1.02.1	
Aerospace Engineering	Designs aircraft for NASA, public transportation and military Applications and may work with sub-systems, such as electrical, mechanical, structural, etc.
Civil Engineer	They design structures, environmental systems, and various construction projects and may do analysis and design for materials and structural systems for buildings, aircraft, etc.
CNC Technician	Also called a computer numerically controlled operator, is a machinist and programmer who uses computer-driven machines and tools to make precision components for aerospace, computer, medical, and recreational equipment.
Electrical Engineer	Designs electric power devices, controls, mechanisms, and electrical systems. Works with power transmission, analog and digital circuits, and communications.
Machine Tool Designers	Work to engineer the designs of new equipment used in industrial manufacturing. Some tool designers work for large companies that provide specialty machines for manufacturers in a variety of areas, such as forging, parts manufacturing, machine shops and other manufacturing industries.
Machinist	A person who operates and/or repairs machinery in an industrial/manufacturing setting.
Marine engineering services	Field that deals with the design, development, production, and maintenance of the equipment used at sea and onboard sea vessels like boats, ships etc.
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	Field of engineering which deals with the activities related to the production of Hydrocarbons, which can be either crude oil or natural gas.
Types of Degrees/Certifications	Certification, License, Associates, Bachelors, Masters, and Doctorate

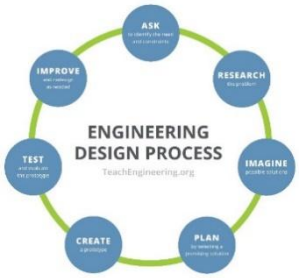
Course	IV23 Drafting III - Engineering			
Essential Standard	2.00	C3	5%	Apply Engineering Design Concepts and Principles to Solve Problems.
Indicator	2.01	N/A	N/A	Apply the concepts and principles of the Engineering Design Process.
Culminating Question **Essential Question**	**What are the concepts and 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 concept 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. *Note: Activity should take place in conjunction with Indicator 2.02.*	
Activity	Individual or Small Group Design Project-Engineering Design Process Introduction
Teacher Instructions	• Share Design Project Files with students as they progress through activity. • *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 and 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 reflection on their design and the process using *Design Process-IMPROVE* as a guide. • Facilitate students presenting their overall projects. *Notes:* This activity should expand on project(s) from Engineering II. It 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 should be required to keep an engineering notebook

	common to Industry Standards to document the process (see Indicator 2.02).
Student Directions	● Keep an engineering design notebook to document your project. ● *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 and 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)	 Design Project-INTRODUCTION.ppt  Design Project-RUBRIC- Mastery-Ba  Design Project-Getting Started and  Design Project-IMAGINE.docx  Design Project-PLAN.docx  Design Project-CREATE.docx  Design Project-IMPROVE.docx  Design Project-TEST.docx  Rules for Keeping an Engineering Desi

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 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	IV23 Drafting III - Engineering			
Essential Standard	2.00	C3	5%	Apply Engineering Design Concepts and Principles to Solve Problems.
Indicator	2.02	N/A	N/A	Apply the concepts to create an Engineering Notebook. Maintain for all class projects.
Culminating Question **Essential Question**	**What are the rules for keeping and maintaining an engineering notebook?** ● What is the purpose of an engineering design 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. *Note: Activity should take place in conjunction with Indicator 2.01.*	
Activity	Engineering Design Notebook Application
Teacher Instructions	● Provide a copy of *Rules for Keeping an Engineering Design Notebook* to students. Facilitate review/discussion/reading of rules as a whole-class. ● Facilitate students keeping a design notebook for each project. *Notes:* This Activity is designed to span multiple class periods and other standards. Should combine with Indicator 2.01 to apply the concept and principles of the engineering design process.
Student Directions	● Participate in review/discussion/reading of rules as a whole-class. ● Keep a design notebook for each project.
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, 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	IV23 Drafting III - Engineering			
Essential Standard	3.00	B2	10%	Understand Advanced Manufacturing Processes.
Indicator	3.01	N/A	N/A	Understand the concepts of Advanced Manufacturing Processes: ● Mechatronics ● CNC ● Robotics ● New and Emerging Technologies (Not to be tested on Post assessment)
Culminating Question **Essential Questions**	**What are the concepts involved with advanced manufacturing?** ● How does mechatronics apply to advanced manufacturing? ● How does CNC apply to advanced manufacturing? ● How does robotics apply to advanced manufacturing? ● How do new and emerging technologies apply to advanced manufacturing?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand mechatronics.
c. Understand CNC.
d. Understand robotics.
e. Understand new and emerging technologies in advanced manufacturing.

INSTRUCTIONAL ACTIVITIES-3.01	
A. Content Literacy Terminology	
Resource(s)	**(See 3.01.1)**
B. Understand mechatronics. **C. Understand CNC.** **D. Understand robotics.** **E. Understand new and emerging technologies in advanced manufacturing.** *Note: Activity includes all Unpacked Content for Indicator and combines with Indicator 3.02.*	
Activity	Qualitative Research-Advanced Manufacturing Processes
Teacher Instructions	● Provide a copy of *Qualitative Research- Advanced Manufacturing Processes* instructions for activity. Facilitate whole-class or small group discussion of concepts/definitions pulled from NCSCOS on finding credible resources. Facilitate students reading terms aloud. ● Facilitate students researching in Part One of activity for note cards. Determine needed time limits for each concept according to individual learning needs. Students should research each concept. ● Facilitate students choosing one concept (or multiple) to write a short (minimum three minute) informative speech discussing the concept and related new emerging technologies. ● Facilitate draft revisions and recordings as needed. ● Facilitate students listening to at least one recording on each concept other than their own and giving constructive, peer feedback.
Student Directions	● Participate in whole-class or small group discussion of concepts/definitions pulled from NCSCOS on finding credible resources. Read aloud terms if prompted. ● Complete research in Part One of activity for note cards for each concept. ● Define advanced manufacturing concepts by choosing a concept(s) to write a short (minimum three minute) informative speech discussing the concept and related new emerging technologies. ● Participate in draft revisions and record. ● Listen to at least one recording on each concept other than your own and give constructive, peer feedback.
Resource(s)	 Qualitative Research- Advanced

Content Literacy Terminology-3.01.1	
Assembly	Last stage of the Manufacturing Process where various parts are put together to complete the product. **OR** An environment within Autodesk Inventor where parts are put together with the file type being *.iam.*
CNC	Process in which the functions and motions of a machine tool are controlled by means of a prepared program containing coded alphanumeric data.
Finishing	After Forming this is the last step in the Manufacturing Process for an individual part before it goes to Assembly. Might include polishing, burnishing, deburring, surface treating, coating, and plating.
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	A manufacturing term encompassing a broad range of technologies and techniques. It can be roughly defined as the process of removing material from a workpiece using power-driven machine tools to shape it into an intended design.
Manufacturing Materials	Materials for production are divided into three general categories: metal, plastic, and inorganic materials.
Manufacturing Process	The steps through which raw materials are transformed into a final product.
Mechatronics	A multidisciplinary field that combines several types of engineering (electrical, computer, and mechanical) which refers to the skill sets needed in the contemporary, advanced automated manufacturing industry.
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.
Robotics	The intersection of science, engineering and technology that produces machines that substitute for (or replicate) human actions.
Rough Forming	First stage of the manufacturing process which consists of shaping the part by casting, forging, and/or welding.
Surfacing	Any of various processes in which material surface is improved.

Course	IV23 Drafting III - Engineering			
Essential Standard	3.00	B2	10%	Understand Advanced Manufacturing Processes
Indicator	3.02	N/A	N/A	Understand the concepts of Additive Manufacturing processes; file formats (.STL, etc.) Design Considerations, Design Intent.
Culminating Question **Essential Questions**	**What are the concepts involved with advanced manufacturing?** ● What is additive manufacturing? ● What file formats are used in additive manufacturing? ● What is design intent?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand additive manufacturing.
c. Understand file formats used in additive Manufacturing.
d. Understand design intent.

INSTRUCTIONAL ACTIVITIES-3.02	
A. Content Literacy Terminology	
Resource(s)	**(See 3.02.1)**
B. Understand additive manufacturing. **C. Understand file formats used in additive manufacturing.** *Note: Activity includes Unpacked Content B & C and combines with Indicator 3.01.*	
Activity	Qualitative Research-Advanced Manufacturing Processes
Teacher Instructions	● Provide a copy of *Qualitative Research- Advanced Manufacturing Processes* instructions for activity. Facilitate whole-class or small group discussion of concepts/definitions pulled from NCSCOS on finding credible resources. Facilitate students reading terms aloud.. ● Facilitate students researching in Part One of activity for note cards. Students should research each concept. ● Facilitate students choosing one concept (or multiple) to write a short (minimum three minute) informative speech discussing the concept and related new emerging technologies. ● Facilitate draft revisions and recordings as needed. ● Facilitate students listening to at least one recording on each concept other than their own and giving constructive, peer feedback.
Student Directions	● Participate in whole-class or small group discussion of concepts/definitions pulled from NCSCOS on finding credible resources. Read aloud terms if prompted. ● Complete research in Part One of activity for note cards for each concept. ● Define advanced manufacturing concepts by choosing a concept(s) to write a short (minimum three minute) informative speech discussing the concept and related new emerging technologies. ● Participate in draft revisions and record. ● Listen to at least one recording on each concept other than your own and give constructive, peer feedback.
Resource(s)	 Qualitative Research- Advanced
D. Understand design intent.	
Overarching concept which should be discussed for all projects and software activities. Therefore, no specific activity(s) provided.	

Content Literacy Terminology-3.02.1	
.STL file Type	File format native to the stereolithography CAD software widely used for rapid prototyping, 3D printing and computer-aided manufacturing. STL files describe only the surface geometry of a three-dimensional object without any representation of color, texture, or other common CAD model attributes.
Additive Manufacturing	The technologies that build 3D objects by *adding* layer-upon-layer of material, whether the material is plastic, metal, concrete.
Additive Manufacturing	The technologies that build 3D objects by adding layer-upon-layer of material, whether the material is plastic, metal, concrete.
Design Intent	Process used to describe how the model should be created and how it should behave when it is changed. Design intent is not just about the size and shape of features, but includes tolerances, manufacturing processes, relationship between features, dimensions, and the use of equations.
Manufacturing Materials	Materials for production are divided into three general categories: metal, plastic, and inorganic materials.
Manufacturing Process	The steps through which raw materials are transformed into a final product.

Course	IV23 Drafting III - Engineering			
Essential Standard	3.00	B2	10%	Understand Advanced Manufacturing Processes.
Indicator	3.03	N/A	N/A	Understand the concepts of Threads and Fasteners used in Assemblies.
Culminating Question **Essential Questions**	**What are the concepts of Threads and Fasteners used in Assemblies?** ● What is the purpose of threads and fasteners in assemblies? ● What are some common types of fasteners used in assemblies? ● What are the common types of threads? ● How are threads represented on a technical drawing? ● How are threads notated on a technical drawing?			

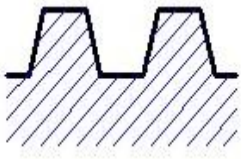
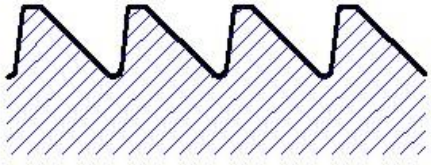
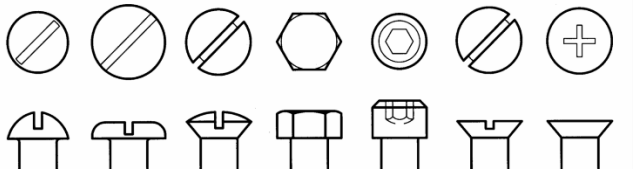
UNPACKED CONTENT

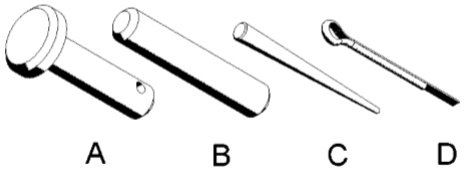
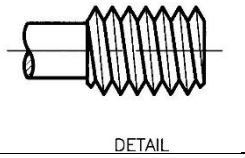
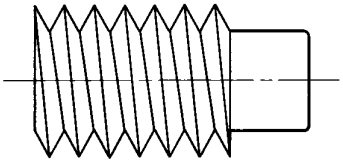
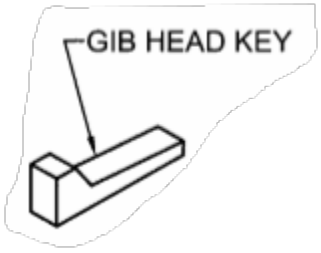
a. Content Literacy Terminology.
b. Understand the purpose of threads and fasteners in assemblies.
c. Identify common types of fasteners used in assemblies.
d. Understand common thread types.
e. Understand how to represent threads on a technical drawing.
f. Understand how to notate threads on a technical drawing.

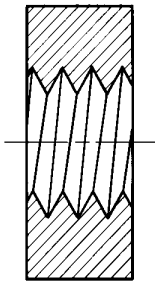
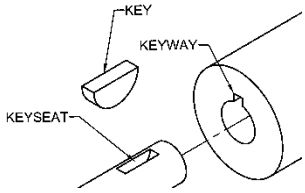
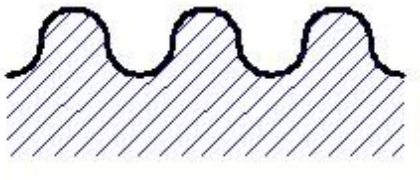
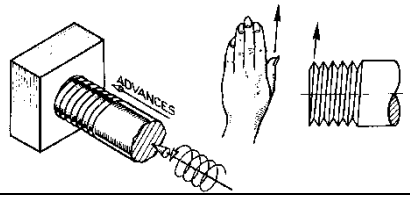
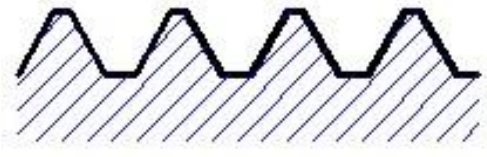
INSTRUCTIONAL ACTIVITIES-3.03	
A. Content Literacy Terminology	
Resource(s)	**(See 3.03.1)**
B. Understand the purpose of threads and fasteners in assemblies.	
Activity	Scavenger Hunt-Threads and Fasteners Introduction
Teacher Instructions	● Facilitate introduction to concepts using *Scavenger Hunt-Threads and Fasteners Introduction*. Provide real life examples for students to see for slides 4 & 5 if available. Prompt students to come up with real world examples for slides 6-9. Explain final scavenger hunt activity on slide 10. ● Facilitate students exploring a room or building to find examples of each purpose (up to five for each). Students should take pictures of examples individually, in pairs, or small groups. ● Facilitate students compiling their examples in a presentation. ● Facilitate students sharing their presentation with whole-class or in small groups.
Student Directions	● Participate in introduction to concepts. Provide real world examples for slides 6-9. ● Explore a room or building to find examples of each purpose (up to five for each). Take pictures of examples. ● Compile your examples in a presentation. ● Share your presentation with the whole-class or in small groups.
Resource(s)	 Scaveneger Hunt-Threads and Fasten
C. Identify common types of fasteners used in assemblies.	
Activity	Visual Peer Teaching-Fastener Types
Teacher Instructions	● Provide digital copy of *Visual Peer Teaching- Fastener Types*. Explain expectations on slide 3. ● Facilitate students finding two image examples for each type of fastener notated by a yellow star using the internet. One image should be a graphic representation (i.e. technical drawing which includes the type of fastener) and the other a real life object. ● Facilitate students sharing presentations in pairs, small groups or for the whole-class.
Student Directions	● Identify common types of fasteners used in assemblies by finding two image examples for each type of fastener notated by a yellow star using the internet. One image should be a graphic representation (i.e. technical drawing which includes the type of fastener) and the other a real life object. ● Share your presentation in pairs, small groups or for the whole-class.
Resource(s)	 Visual Peer Teaching- Fastener
D. Understand common thread types.	
Activity	Classification and Selection-Thread Forms-Presentation

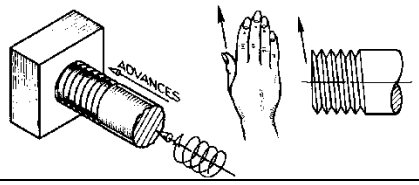
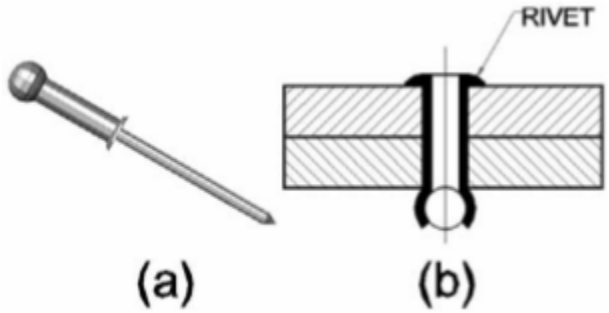
Teacher Instructions	● Provide digital copies of *Classification and Selection-Thread Forms- Presentation* and *Classification and Selection-Thread Forms-Leading Questions*. Facilitate students answering leading questions using the information provided in the PowerPoint Presentation. ● Formatively check for understanding and/or review with the whole-class to check for understanding.
Student Directions	● Identify common types of threads by answering leading questions independently using the information provided in the PowerPoint Presentation. ● Participate in review of information.
Resource(s)	 Classification and Selection- Thread Fc  Classification and Selection- Thread Fc
E. Understand how to represent threads on a technical drawing.	
Activity	Image Identification-Thread Representation
Teacher Instructions	● Provide digital copy of *Image Identification-Thread Representation- Presentation* and a hard copy of *Image Identification-Thread Representation- Worksheet*. Facilitate students labeling images in worksheet using the information provided in the PowerPoint Presentation ● Formatively check for understanding and/or review with the whole-class using *Image Identification-Thread Representation-Worksheet-Answer Key.*
Student Directions	● Identify common parts and representations of threads by labeling images on the worksheet using the information provided in the PowerPoint Presentation. ● Participate in review of images.
Resource(s)	 Image Identification- Threa  Image Identification- Threa  Image Identification- Threa
F. Understand how to notate threads on a technical drawing.	
Activity	Notation Deciphering-Threads and Fasteners
Teacher Instructions	● Provide hard copy of *Notation Deciphering- Threads and Fasteners- Guided Notes*. Facilitate introduction to concepts using *Notation Deciphering-Threads and Fasteners-Presentation*. Students will copy definitions for Unified Threads (gray slides in presentation) and label Metric Thread parts (slide 38) on Guided Notes. ● Provide copies of *Notation Deciphering-Threads and Fasteners-Independent Practice* and *Notation Deciphering-Threads and Fasteners-Resource*. Facilitate students working individually or in pairs to decipher each of the provided examples. They may use the internet or other resource(s) for help. ● Formatively check for understanding and/or review with the whole-class.

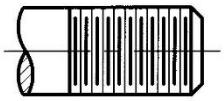
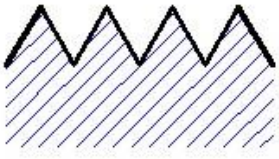
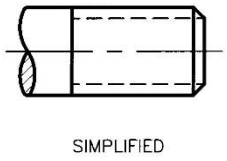
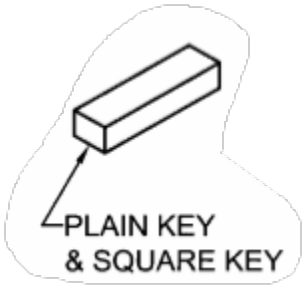
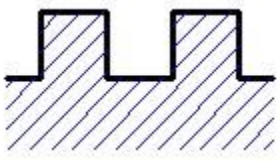
Student Directions	● Copy definitions for Unified Threads (gray slides in presentation) and label Metric Thread parts (slide 38) on Guided Notes. ● Decipher the provided thread notations working individually or in pairs. You may use the internet or other resource(s) for help. ● Participate in review of notations.
Resource(s)	 Notation Deciphering- Threa  Notation Deciphering- Threa  Notation Deciphering- Threa  Notation Deciphering- Threa

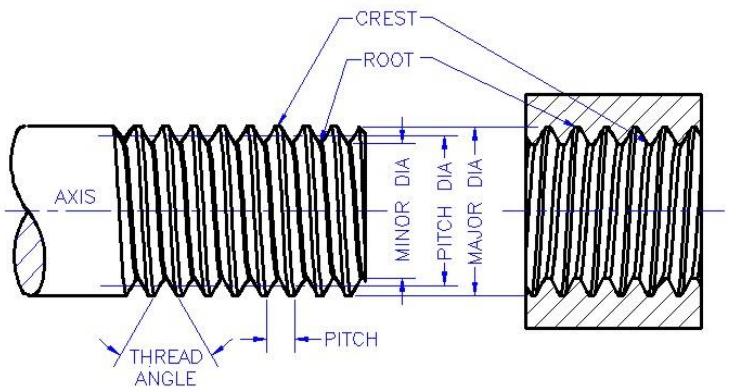
Content Literacy Terminology-3.03.1	
ACME Thread	Standard thread form. Modification of the square thread. Stronger and easier to cut than the square thread. Used to transmit power. Also known as a worm thread. 
Assembly	Last stage of the Manufacturing Process where various parts are put together to complete the product. **OR** An environment within Autodesk Inventor where parts are put together with the file type being *.iam.*
Blind Hole	A hole that cuts into but does not pass completely through the object.
Buttress thread	Standard thread form. Designed to transmit power in one direction. Used on large guns, screw jacks, and Bumper Jacks. 
Class 1	A class of threaded fit. Loose fit, used for rough work.
Class 2	A class of threaded fit. Free/Standard fit, general-purpose use of most bolts and nuts.
Class 3	Medium fit, used for the better grades of work, such in automobiles.
Class 4	Close fit used where a very snug fit is required, as in aircraft engines.
Class Fit	There are four standardized classes of fit. The term fit refers to how closely the screw fits in the threaded hole. That is the amount of play between the two parts.
Common Fastener Head Types	 Left to Right: (a) Round, (b) Pan, (c) Oval, (d) Hex, (e) Socket, (f) Flat, and (g) Phillips.
Common Types of Bolts	Hex, Lag, Carriage, Eye, J, and U *(Note: this is not all types).*
Common Types of Keys	Square, Gib, Pratt and Whitney, and Woodruff *(Note: this is not all types).*
Common Types of Pins	Clevis (a), Straight (b), Taper (c), and Cotter (d)

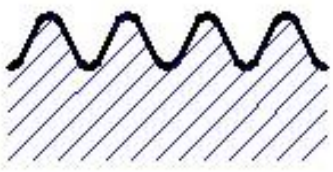
	 *(Note: this is not all types.)*
Common Types of Screws	Machine, Cap, Sheet Metal, Wood and Set *(Note: this is not all types).*
Counterbore	To enlarge the end of a drill hole to a specific diameter and depth in order to recess a mating part.
Countersink	To recess a hole with a conically (cone) shaped tool to provide a seat for flat head screws.
Crest	The shallowest thread cut. It can be rounded or flat
Detailed Representation	One of the three ways to represent threads on a technical drawing which shows all details and is true to form. Complex to draw and used mostly on only large threads. 
External Thread	Helix shape cut off part material or applied to the shaft. 
Fasteners	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.
Gib Head Key	Same as the square key except that it has a gib head, which provides for easy removal. 
Helix	The spiral grooves cut into the surface of cylinders. This is the same form made by wrapping a copper wire around a cylinder to form a spring/coil.
Internal Thread	Helix shape cut into absence of material or applied to hole.

	
Keys	Fasteners used to prevent relative movement between wheels, pulleys, gears, cranks, and similar parts to a shaft. Made of three pieces: Keyseat, Key, and Keyway. 
Knuckle Thread	Standard thread form. Typically rolled from sheet metal. Sometimes formed by casting. Used in light bulbs, sockets, and jar and bottle tops. 
Lathe	A machine used to produce large threads.
Lead	The distance a thread moves in one revolution.
Left-Handed Thread	(LH) – Advances into a nut when turned counterclockwise. 
Major Diameter	The largest diameter of a thread. Measured from crest to crest.
Manufacturing Process	The steps through which raw materials are transformed into a final product.
Metric Thread	Standard thread form. Adopted in 1946 by the International Organization for Standardization (now ISO). 
Minor Diameter	The smallest diameter of a thread.

Pin	Type of fastener classified under two separate groups. One, which allows the assembly of parts that might require the need for quick release. The other uses semi-permanent fasteners, designed with an interference fit that would require the aid of tools for installation or removal.
Pitch	The distance from one point on the thread to the corresponding point on the next form. The pitch of a thread is usually expressed in tables in terms of the number of threads per inch.
Pratt & Whitney Key	Rectangle in shape with semi-cylindrical ends. 
Right-Handed Thread	(RH) – Advances into a nut when turned clockwise. All threads are understood to be RH unless designated specifically LH in the thread note. 
Rivet	Permanent fasteners. Generally used to hold sheet metal or rolled steel shapes together. The smooth cylindrical shaft is inserted into the aligned holes of matting parts. The tail is then formed to create a head on the opposite end of the shaft.  *Before installation (a), After installation (b)*
Root	The deepest thread cut. It can be rounded or flat.
Schematic Representation	One of the three ways to represent threads on a technical drawing which shows threads per inch, but not true to overall form.

	 SCHEMATIC
Sharp V or Sellars Thread	Standard thread form. First U.S. Standard Thread. Now used on brass pipe work. 
Simplified Representation	One of the three ways to represent threads on a technical drawing which removes most detail and is not true to overall form. Most common practice because of time. Hidden lines are drawn parallel to the axis at the approximate depth of the thread. The crest line is represented by a visible line, also parallel to the axis, in viewing the major diameter of the thread.  SIMPLIFIED
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.
Square Key	Design is used for heavy-duty functions. Sometimes referred to as a flat key. The widths of keys generally used are about one-fourth the shaft size. One half of the key is sunk into the shaft. The depth of the keyway or the keyseat is measured on the side – not the center.  PLAIN KEY & SQUARE KEY
Square Thread	Standard thread form. Theoretically the ideal thread for power transmission. Tough to cut the threads because of the 90° angles. 

Tap Tool	A small fluted cutting tool with cutting teeth shaped to form small internal threads.
Thread Angle	The angle formed by the walls of the thread. The angle created by the slope of the thread is a standard 60 degrees.
Thread Depth	Refers to the distance from the top of the thread (crest), to the bottom of the thread (root).
Thread Form	Also referred to as the thread type or shape of thread.
Thread Parts: ● Axis ● Thread Angle ● Pitch ● Crest ● Root ● Minor Diameter ● Pitch Diameter ● Major Diameter	
Thread Series	A standard based upon the number of threads/inch for a specific nominal diameter. Standards for standard inch units are: Coarse (C), Fine (F), Extra-Fine (EF).
Threaded Fastener Types	Screw and Bolt
Threads	A uniformed ridge/cut in the form of a helix on a fastener used to hold parts together, adjust parts and/or transmit power.
Threads per Inch	A measure of the number of crests per unit of length measured along the axis of the thread. The number of threads/inch for a thread series is given by standard and may be found in thread tables.
Through Hole	A hole that passes all the way through the object.
Tight Fit	A class of threaded fit. Can be started by hand but requires assistance (tools) to advance threads. Common for set screws. Used in permanent assemblies.
Unified Thread	Standard thread form. Adopted by the U.S., British, and Canada after World War II as an ANSI Standard. Mainly used for fastening and adjusting. 
Unthreaded Fastener Types	Rivet, Pin and Key

Whitworth Thread	Standard thread form. First standard in England. Has been mostly replaced by the Unified thread. 
Woodruff Key	Semicircular in shape and is often used in machine-tool work. The bottom of the key fits into a semicircular key slot cut with a Woodruff cutter and the top into a rectangular slot. 

Course	IV23 Drafting III - Engineering			
Essential Standard	4.00	C3	30%	Apply Advanced 3D Parametric-Solid Model and Assembly Creation Techniques
Indicator	4.01	N/A	N/A	Apply advanced techniques to complete a 2D sketch using the appropriate draw tools
Culminating Question **Essential Question**	**What are the advanced techniques to complete a 2D sketch using the appropriate draw tools?** ● What are the advanced constraint-based modeling concepts including 2D sketching/draw tools?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand advanced constraint-based modeling concepts including 2D sketch concepts.
c. Apply advanced techniques to complete a 2D sketch using the appropriate draw tools.

INSTRUCTIONAL ACTIVITIES-4.01	
A. Content Literacy Terminology	
Resource(s)	**(See 4.01.1)**
B. Understand Advanced Constraint-Based Modeling concepts including 2D sketch concepts. *Note: Activity combines with Indicators 4.02 & 4.05.*	
Activity	Classification Organizer-Advanced Constraint-Based Parametric Solid Modeling
Teacher Instructions	● Provide copy of C*lassification Organizer-Advanced Constraint-Based.Parametric Solid Modeling*. Facilitate read-aloud of Introduction. ● Explain CREATE portion of activity. Facilitate students creating a graphic organizer individually on scrap paper with all concepts included in the document.Provide copy of *Classification Organizer- Advanced Constraint-Based.Parametric Solid Modeling- Expanding Resource* for students to use as a reference. Students may also use the internet for help understanding terms and making connections. ● Break students into small groups or pairs to review graphic organizers. Facilitate groups/pairs analyzing, recreating or making changes as needed to come to a consensus on one design of the organizer. ● Facilitate student movement to outside space with large concrete surfaces (i.e. parking lot or black top). Provide each group with multiple pieces of sidewalk chalk. Facilitate groups recreating the decided upon graphic organizer on a larger scale. Students must add at least 6 pictures/drawings/images/sketches to the larger organizer for concepts. ● Facilitate students sharing and reviewing each group/pair's graphic organizer. ● Students will have a connected understanding of advanced concepts in parametric modeling.
Student Directions	● Participate in the read-aloud of Introduction. ● Demonstrate an understanding of the connection of advanced concepts in parametric modeling by creating a graphic organizer individually on scrap paper with all concepts included in the document. You may use *Classification Organizer-Advanced Constraint-Based.Parametric Solid Modeling-Expanding Resource* or the internet as a reference. ● In groups/pairs analyze, recreate or make changes as needed to come to a consensus on one design of the organizer. ● Recreate the decided upon graphic organizer on a larger scale. You must add at least 6 pictures/drawings/images/sketches to the larger organizer for concepts. ● Share and review each group/pair's graphic organizer.

Resource(s)	Classification Organizer- Advance Classification Organizer- Advance
C. Apply Advanced techniques to complete a 2D sketch using the appropriate draw 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	● Create multiple parts/projects applying concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-4.02.1	
Active Sketch	The current sketch to edit.
Arc Sketch Tool	Sketch drawing command/tool used to draw curved lines by locating centerpoint of arc, points of tangency or 3-points.
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.
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.
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.
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.
Driven Dimension	A parametric dimension that determines the size of sketch geometry and resizes the sketch when its value changes.
Ellipse Sketch Tool	Sketch drawing command/tool used to draw ellipses by locating centerpoint and then major and minor axes.
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 extrudes, revolves, sweeps, lofts, fillets, chamfers, etc.
Feature tree	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 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.

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.
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.
Loop	A closed sketch shape that can include sketched curves, edges, and planar faces as boundaries.
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 an 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.
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 the center are entered to create a polygon.
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.
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.
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.
Shared Sketch	A sketch used by more than one feature. For example, a sketch containing hole centers for different hole features.

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 centers of arcs or centerpoint of slot. Can also draw curved slots that follow a 3-point or centerpoint arc.
Spline Sketch Tool	Sketch drawing command/tool used to draw irregular curved lines that can be adjusted with handles.
Text Sketch Tool	Sketch drawing command/tool that adds text to a sketch that can be extruded or cut into a solid 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	IV23 Drafting III - Engineering			
Essential Standard	4.00	C3	30%	Apply Advanced 3D Parametric-Solid Model and Assembly Creation Techniques.
Indicator	4.02	N/A	N/A	Apply advanced techniques of adding constraints: Geometric and Dimensional.
Culminating Question **Essential Question**	**What are advanced techniques of adding constraints: geometric and dimensional in the software?** ● What are the advanced constraint-based modeling concepts including adding constraints?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand advanced constraint-based modeling concepts including adding constraints.
c. Apply advanced techniques of adding constraints: geometric and dimensional.

INSTRUCTIONAL ACTIVITIES-4.02	
A. Content Literacy Terminology	
Resource(s)	**(See 4.02.1)**
B. Understand advanced constraint-based modeling concepts including adding constraints. *Note: Activity combines with Indicators 4.02 & 4.05.*	
Activity	Classification Organizer-Advanced Constraint-Based Parametric Solid Modeling
Teacher Instructions	● Provide copy of C*lassification Organizer-Advanced Constraint-Based.Parametric Solid Modeling*. Facilitate read-aloud of Introduction. ● Explain CREATE portion of activity. Facilitate students creating a graphic organizer individually on scrap paper with all concepts included in the document. Provide copy of *Classification Organizer- Advanced Constraint-Based.Parametric Solid Modeling- Expanding Resource* for students to use as a reference. Students may also use the internet for help understanding terms and making connections. ● Break students into small groups or pairs to review graphic organizers. Facilitate groups/pairs analyzing, recreating or making changes as needed to come to a consensus on one design of the organizer. ● Facilitate student movement to outside space with large concrete surfaces (i.e. parking lot or black top). Provide each group with multiple pieces of sidewalk chalk. Facilitate groups recreating the decided upon graphic organizer on a larger scale. Students must add at least 6 pictures/drawings/images/sketches to the larger organizer for concepts. ● Facilitate students sharing and reviewing each group/pair's graphic organizer.
Student Directions	● Participate in the read-aloud of Introduction. ● Demonstrate an understanding of the connection of advanced concepts in parametric modeling by creating a graphic organizer individually on scrap paper with all concepts included in the document. You may use *Classification Organizer-Advanced Constraint-Based.Parametric Solid Modeling-Expanding Resource* or the internet as a reference. ● In groups/pairs analyze, recreate or make changes as needed to come to a consensus on one design of the organizer. ● Recreate the decided upon graphic organizer on a larger scale. You must add at least 6 pictures/drawings/images/sketches to the larger organizer for concepts. ● Share and review each group/pair's graphic organizer.
Resource(s)	Classification Organizer- Advance Classification Organizer- Advance

C. Apply Advanced techniques to complete a 2D sketch using the appropriate draw 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	● Create multiple parts/projects applying concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-4.02.1	
Active Sketch	The current sketch to edit.
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.
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. The sketch is consumed by the feature.
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.
Driven Dimension	A parametric dimension that determines the size of sketch geometry and resizes the sketch when its value changes.
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.
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 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.
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.
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.
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.
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.
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.
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	IV23 Drafting III - Engineering			
Essential Standard	4.00	C3	30%	Apply Advanced 3D Parametric-Solid Model and Assembly Creation Techniques.
Indicator	4.03	N/A	N/A	Apply advanced techniques of creating 3D Parts from constrained sketches using: Extrude, Revolve, Sweep and Loft.
Culminating Question **Essential Questions**	**What are the advanced techniques for creating 3D parts from constrained sketches using; Extrude, Revolve, Sweep and Loft in the software?** ● How is Extrude used to create a 3D part in an advanced setting in the software? ● How is Revolve used to create a 3D part in an advanced setting in the software? ● How is Sweep used to create a 3D part in an advanced setting in the software? ● How is Loft used to create a 3D part in an advanced setting in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply advanced techniques of creating 3D parts from constrained sketches using Extrude.
c. Apply advanced techniques of creating 3D parts from constrained sketches using Revolve.
d. Apply advanced techniques of creating 3D parts from constrained sketches using Sweep.
e. Apply advanced techniques of creating 3D parts from constrained sketches using Loft.

INSTRUCTIONAL ACTIVITIES-4.03	
A. Content Literacy Terminology	
Resource(s)	**(See 4.03.1)**
B. Apply advanced techniques of creating 3D parts from constrained sketches using Extrude.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
C. Apply advanced techniques of creating 3D Parts from constrained sketches using Revolve.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
D. Apply advanced techniques of creating 3D Parts from constrained sketches using Sweep.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
E. Apply advanced techniques of creating 3D Parts from constrained sketches using 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.
Student Directions	● Create multiple parts/projects applying concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

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	IV23 Drafting III - Engineering			
Essential Standard	4.00	C3	30%	Apply Advanced 3D Parametric-Solid Model and Assembly Creation Techniques
Indicator	4.04	N/A	N/A	Apply Advanced techniques of adding placed features: Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features
Culminating Question **Essential Questions**	**What are the advanced techniques of adding placed features: Hole, Fillet, Chamfer, Shell, Threads, and Pattern Features in the software?** ● How is Hole used to add a placed feature in the software? ● How is Fillet used to add a placed feature in the software? ● How is Chamfer used to add a placed feature in the software? ● How is Shell used to add a placed feature in the software? ● How are Threads used to add a placed feature in the software? ● How is Pattern used to add a placed feature in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply advanced techniques of adding the placed feature: Hole.
c. Apply advanced techniques of adding the placed feature: Fillet.
d. Apply advanced techniques of adding the placed feature: Chamfer.
e. Apply advanced techniques of adding the placed feature: Shell.
f. Apply advanced techniques of adding the placed feature: Thread.
g. Apply advanced techniques of adding the placed feature: Pattern.

INSTRUCTIONAL ACTIVITIES-4.04	
A. Content Literacy Terminology	
Resource(s)	**(See 4.04.1)**
B. Apply advanced techniques of adding the placed feature: Hole.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
C. Apply advanced techniques of adding the placed feature: Fillet.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
D. Apply advanced techniques of adding the placed feature: Chamfer.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
E. Apply advanced techniques of adding the placed feature: Shell.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
F. Apply advanced techniques of adding the placed feature: Pattern.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
G. Apply advanced techniques of adding the placed feature: Thread.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

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 two or more objects by creating intersections, subtractions, or unions of mass.	
	Cut 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 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. 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	IV23 Drafting III - Engineering			
Essential Standard	4.00	C3	30%	Apply Advanced 3D Parametric-Solid Model and Assembly Creation Techniques.
Indicator	4.05	N/A	N/A	Apply advanced techniques of adaptive features, parts, and subassemblies.
Culminating Question **Essential Questions**	**What are the advanced techniques of adaptive features, parts, and subassemblies in the software?** ● What are the advanced constraint-based modeling concepts including adaptive features, adaptive parts, and subassemblies? ● How are the advanced techniques of adaptive features applied in the software? ● How are the advanced techniques of adaptive parts applied in the software? ● How are the advanced techniques of subassemblies applied in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand advanced constraint-based modeling concepts including adaptive features, adaptive parts, and subassemblies.
c. Apply advanced techniques of adaptive features.
d. Apply advanced techniques of adaptive parts.
e. Apply advanced techniques of subassemblies.

INSTRUCTIONAL ACTIVITIES-4.05	
A. Content Literacy Terminology	
Resource(s)	**(See 4.05.1)**
B. Understand advanced constraint- based modeling concepts including adaptive features, adaptive parts, and subassemblies. *Note: Activity combines with Indicators 4.01 & 4.02.*	
Activity	Classification Organizer-Advanced Constraint-Based Parametric Solid Modeling
Teacher Instructions	● Provide copy of C*lassification Organizer-Advanced Constraint-Based.Parametric Solid Modeling*. Facilitate read-aloud of Introduction. ● Explain CREATE portion of activity. Facilitate students creating a graphic organizer individually on scrap paper with all concepts included in the document.Provide copy of *Classification Organizer- Advanced Constraint-Based.Parametric Solid Modeling- Expanding Resource* for students to use as a reference. Students may also use the internet for help understanding terms and making connections. ● Break students into small groups or pairs to review graphic organizers. Facilitate groups/pairs analyzing, recreating or making changes as needed to come to a consensus on one design of the organizer. ● Facilitate student movement to outside space with large concrete surfaces (i.e. parking lot or black top). Provide each group with multiple pieces of sidewalk chalk. Facilitate groups recreating the decided upon graphic organizer on a larger scale. Students must add at least 6 pictures/drawings/images/sketches to the larger organizer for concepts. ● Facilitate students sharing and reviewing each group/pair's graphic organizer. ● Students will have a connected understanding of advanced concepts in parametric modeling.
Student Directions	● Participate in the read-aloud of Introduction. ● Demonstrate an understanding of the connection of advanced concepts in parametric modeling by creating a graphic organizer individually on scrap paper with all concepts included in the document. You may use *Classification Organizer-Advanced Constraint-Based.Parametric Solid Modeling-Expanding Resource* or the internet as a reference. ● In groups/pairs analyze, recreate or make changes as needed to come to a consensus on one design of the organizer. ● Recreate the decided upon graphic organizer on a larger scale. You must add at least 6 pictures/drawings/images/sketches to the larger organizer for concepts. ● Share and review each group/pair's graphic organizer.

Resource(s)	Classification Organizer- Advance Classification Organizer- Advance
C. Apply advanced techniques of adaptive 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.
Student Directions	● Create multiple parts/projects applying concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
D. Apply advanced techniques of adaptive parts.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
E. Apply advanced techniques of 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.
Student Directions	● Create multiple parts/projects applying concept(s).
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 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. 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; 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.
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.

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 or 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 as many 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	IV23 Drafting III - Engineering			
Essential Standard	5.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Model and Assembly
Indicator	5.01	N/A	N/A	Apply the concepts and techniques of creating working drawings ● Assembly Drawings ● Detail Drawings ● Parts List ● Balloons
Culminating Question **Essential Questions**	**How are concepts and techniques applied to create Working Drawings (including Assembly Drawings, Detail Drawings, parts lists and balloons) in the software?** ● What are the purposes of Working Drawings? ● What is included in a set of Working Drawings? ● How are concepts and techniques of creating Assembly Drawings applied in the software? ● How are concepts and techniques of creating Detail Drawings applied in the software? ● How are concepts and techniques of creating Parts Lists applied in the software? ● How are concepts and techniques of creating balloons applied in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the purpose of Working Drawings.
c. Understand what is included in a set of Working Drawings.
d. Apply the concepts and techniques of creating Assembly Drawings.
e. Apply the concepts and techniques of creating Detail Drawings.
f. Apply the concepts and techniques of creating a parts List.
g. Apply the concepts and techniques of creating balloons.

INSTRUCTIONAL ACTIVITIES-5.01	
A. Content Literacy Terminology	
Resource(s)	**(See 5.01.1)**
B. Understand the purpose of Working Drawings. **C. Understand what is included in a set of Working Drawings.** *Note: Activity includes Unpacked Content for both B & C.*	
Activity	Guided Instruction-Working Drawings
Teacher Instructions	• Provide students with hardcopy of *Guided Instruction-Working Drawings-Student* (recommended to print 3-6 slides per sheet). Facilitate whole-class guided instruction using *Guided Instruction- Working Drawings- Teacher* and students filling in blanks. • Students will be able to identify the reason for Working Drawings and standards.
Student Directions	• Participate in whole-class guided instruction. Fill in blanks on guided notes to identify the reason for Working Drawings and standards.
Resource(s)	Guided Instruction-Working Drawings- Guided Instruction-Working Drawings-
D. Apply the concepts and techniques of creating Assembly Drawings.	
Activity	Software Tutorials and Projects
Teacher Instructions	• Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	• Create multiple parts/projects applying concept(s). *Application-Working Drawings-Assembly* available as sample project.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource Application-Working Drawings-
E. Apply the concepts and techniques of creating Detail Drawings.	
Activity	Software Tutorials and Projects
Teacher Instructions	• Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	• Create multiple parts/projects applying concept(s). *Application-Working Drawings-Assembly* available as sample project.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource Application-Working Drawings-
F. Apply the concepts and techniques of creating a parts list.	
Activity	Software Tutorials and Projects
Teacher Instructions	• Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	• Create multiple parts/projects applying concept(s). *Application-Working Drawings-Assembly* available as sample project.

Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Application- Working Drawings-
G. Apply the concepts and techniques of creating balloons.	
Activity	Software Tutorials & Projects
Teacher Instructions	● Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying concept(s). *Application-Working Drawings-Assembly* available as sample project.
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Application- Working Drawings-

Content Literacy Terminology-5.01.1	
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.
Parts Bubble	Circled number label attached to a leader which refers to a parts list.
Parts List	List included on drawings which shows all or specified parts and subassemblies.
PDF	(Portable Document Format) a file format used to save files that cannot be modified but still easily shared and printed.
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.

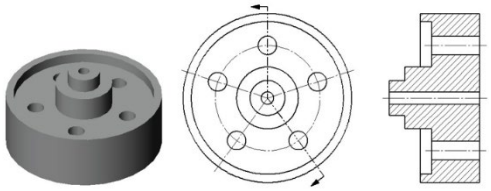
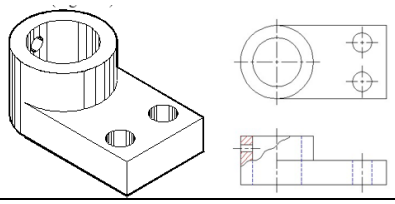
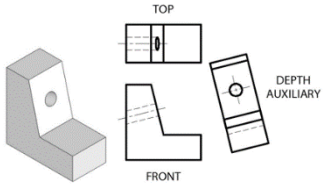
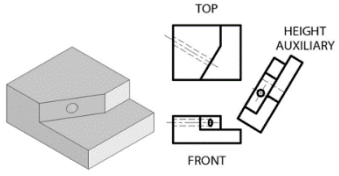
Course	IV23 Drafting III - Engineering			
Essential Standard	5.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Model and Assembly.
Indicator	5.02	N/A	N/A	Apply multi-view projection and the various views that are needed to document a 3D model including: Base, Projected, Section, and Auxiliary Views.
Culminating Question	**How are concepts of multiview projection and techniques for the various views needed to document a 3D model including: base, projected, section, and auxiliary views applied in the software?**			
Essential Questions	● How is multiview projection (including base views and projected views) applied in the software? ● What are the types of section drawings and special circumstances that apply in sections? ● How are section drawings created in the software? ● What are the major types of auxiliary views? ● How are auxiliary views created in the software?			

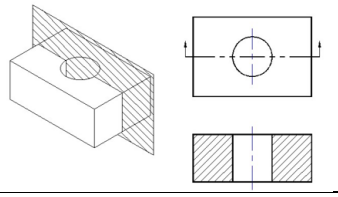
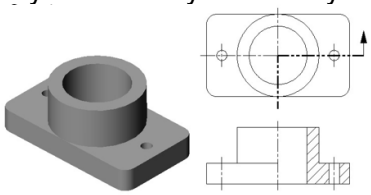
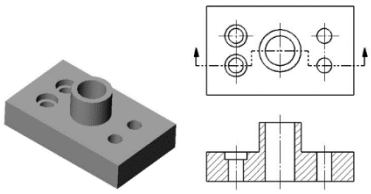
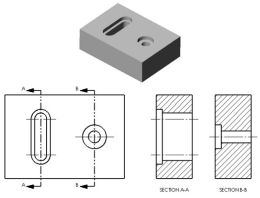
UNPACKED CONTENT

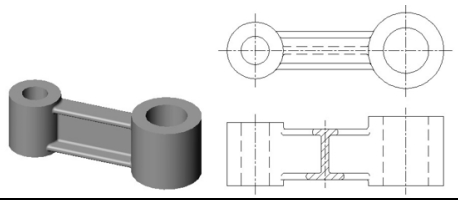
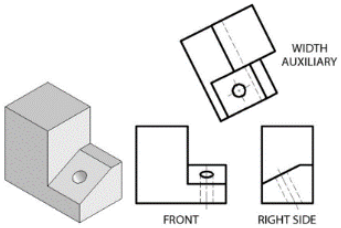
a. Content Literacy Terminology.
b. Apply multiview projection (base and projected views) in the software.
c. Review the major types of section views and special circumstances.
d. Apply section views in the software.
e. Review the major types of auxiliary views.
f. Apply auxiliary views in the software.

INSTRUCTIONAL ACTIVITIES-5.02	
A. Content Literacy Terminology	
Resource(s)	**(See 5.02.1)**
B. Apply Multiview projection (base & projected 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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
C. Review the major types of section views and special circumstances.	
Activity	Autonomous Application-Sections Review
Teacher Instructions	• Provide a copy of *Autonomous Application- Sections Review*. Facilitate students creating examples of each type of section in the software and pasting a screenshot into the document. Tutorial(s) can be provided for students.
Student Directions	• Create examples of each type of section in the software and paste a screenshot into the document.
Resource(s)	 Autonomous Application- Section
D. Apply section 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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
E. Review the major types of auxiliary views.	
Activity	Student Practice-Auxiliary View Matching
Teacher Instructions	• Provide copy of *Student Practice-Auxiliary View Matching-PowerPoint Presentation*. • Facilitate students matching examples and recording on scrap paper individually, in pairs or small groups. • Provide formative assessment as needed for students.
Student Directions	• Number scrap paper 1-10. • Determine auxiliary views for various objects by matching examples. Record choices on scrap paper.
Resource(s)	 Student Practice-Auxiliary View Matcl
F. 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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-5.02.1	
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. 
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.
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.
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. 
Drawing File .idw or .dwg	Primary file type in which sheets/documents are set-up and working drawings sets are created.
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.

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. 
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.
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. 
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.
Removed Section	Sectional view is taken from its normal position and shown somewhere else on the drawing. 

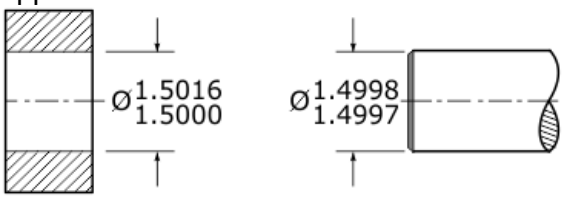
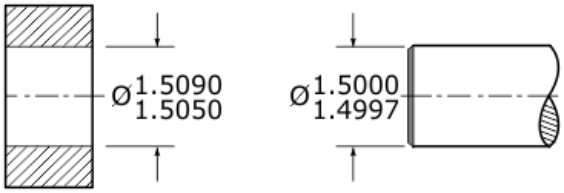
Revolved/Rotated Section	Sectional view used to represent the cross-sectional shape of a handle, spoke, or other elongated features. Create the section by cutting through the feature and revolving the section 90 degrees. Superimpose the sectional view on the orthographic view. 
Rib	Long raised piece 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.
True Size and Shape	A surface that is perpendicular to the projection plane.
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.
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. 

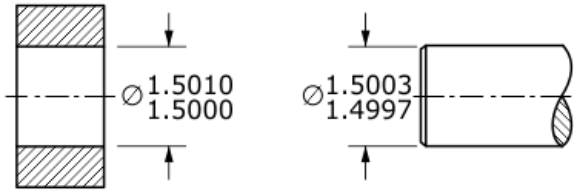
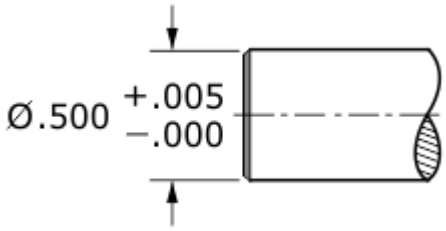
Course	IV23 Drafting III - Engineering			
Essential Standard	5.00	C3	20%	Apply Procedures to Create Working Drawings of a 3D Model and Assembly.
Indicator	5.03	N/A	N/A	Apply the techniques of adding Annotations, Dimensions, and Tolerances.
Culminating Question **Essential Questions**	**How are techniques applied for adding annotations, dimensions, and tolerances in the software?** ● How are annotations applied in the software to Working Drawings? ● How are dimensions applied in the software to Working Drawings? ● What are the concepts of GD&T? ● How are tolerances applied in the software to Working Drawings?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Apply annotations to Working Drawings in the software.
c. Apply dimension to Working Drawings in the software.
d. Understand geometric and tolerance dimensions.
e. Apply tolerances to Working Drawings in the software.

INSTRUCTIONAL ACTIVITIES-5.03	
A. Content Literacy Terminology	
Resource(s)	**(See 5.03.1)**
B. Apply annotations to Working Drawings in the software.	
Activity	Software Tutorials
Teacher Instructions	● Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying concepts. *Application-Working Drawings-Assembly* is available as a project for indicator (*Note: combines with additional Indicators as well*).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Application-Working Drawings-
C. Apply dimension to Working Drawings in the software.	
Activity	Software Tutorials
Teacher Instructions	● Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying concepts. *Application-Working Drawings-Assembly* is available as a project for indicator (*Note: combines with additional Indicators as well*).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Application-Working Drawings-
D. Understand geometric and tolerance dimensions.	
Note: See Essential Standard 7.00	
E. Apply tolerances to Working Drawings in the software.	
Activity	Software Tutorials *Note: Also see Essential Standard 7.00*
Teacher Instructions	● Use or create video tutorial session(s) which align with the current version being used in PSU.
Student Directions	● Create multiple parts/projects applying concepts. *Application-Working Drawings-Assembly* is available as a project for indicator (*Note: combines with additional Indicators as well*).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource  Application-Working Drawings-

Content Literacy Terminology-5.03.1	
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.
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 and dark, continuous lines that run between extension lines.
Drawing File .idw or .dwg	Primary file type in which sheets/documents are set-up and working drawings sets are created.
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 and 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.
Hatch	Pattern fill of an established area with boundaries.
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. 
True Size and Shape	A surface that is perpendicular to the projection plane.
Unilateral Tolerance	Tolerance that allows variation in one direction. 

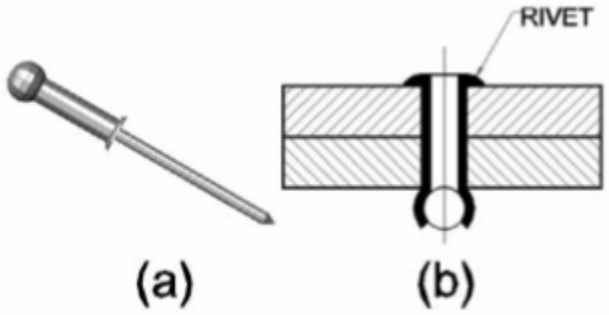
Course	IV23 Drafting III - Engineering			
Essential Standard	6.00	C3	10%	Apply Procedures to Create Sheet Metal Parts.
Indicator	6.01	N/A	N/A	Apply the concepts used to Create sheet metal parts.
Culminating Question **Essential Question**	**How are techniques applied to create sheet metal parts in the software?** ● What are the concepts associated with sheet metal processing and patterns?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand concepts associated with sheet metal and patterns.
c. Apply concepts used to create sheet metal parts.

INSTRUCTIONAL ACTIVITIES-6.01	
A. Content Literacy Terminology	
Resource(s)	**(See 6.01.1)**
B. Understand concepts associated with sheet metal and patterns. *Note: Activity combines with Indicators 6.02 & 6.03.*	
Activity	Research Compilation-Sheet Metal and Pattern Development
Teacher Instructions	● Provide copy of *Research Compilation-Sheet Metal and Pattern Development*. Facilitate students researching answers to leading questions. ● Facilitate students sketching for Part Two. ● Break students into small groups or pairs to review answers to questions and share sketches. ● Students will be able to identify common concepts related to sheet metal and patterns.
Student Directions	● Identify common concepts related to sheet metal and patterns by researching answers to leading questions and completing sketches. ● Review answers to questions and share sketches.
Resource(s)	 Research Compilation- Sheet
C. Apply concepts used to create sheet metal parts 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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-6.01.1	
Bend	In sheet metal design, a corner seam can be converted to a bend. A bend is the small radius between faces that represents the fabrication process in which bends are applied to a flat pattern to form a 3D part.
Bend allowance	In sheet metal design, a factor applied when converting a bend to an unfolded condition. Material thickness, material type, and bend radius are used to calculate the amount of distortion.
Bend lines	In sheet metal design, bend lines can be displayed in the flat pattern drawing view to show the start, center, and end of the bend zone.
Bend relief	In sheet metal design, a small notch cut at the end of a bend to prevent material from tearing.
Bend zone	In sheet metal design, the area where material deformation takes place, often referred to in the flat pattern as the area between bend lines.
Corner break	In sheet metal design, a corner break applies either a radius or chamfer edge to the end of a flange.
Corner relief	In sheet metal design, a small notch that prevents material from tearing is placed where two bends join.
Corner seam	In sheet metal design, a corner seam mates edges where two bends join, including the alignment or overlap of edges, miter of angled flanges, and inserts corner relief.
Cut	In sheet metal design, cuts are sketched shapes that define material to remove from a single face, a specified depth, or an entire 3D part.
Cut length	For frames, the overall length of the raw stock required to create a structural member (including enough material to accommodate end treatments on either end).
Cut list	A cut list is a table listing components with dimensions and cut treatment information for fabrication and accounting purposes. Each treatment on a frame member produces a custom property called CUTDETAIL(n), which you can add as manufacturing notes in the cut list.
Cut 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.
Flat Pattern	The shape of the sheet metal part before it is formed, to create drawings for manufacturing. Flat patterns show bend lines, bend zones, punch locations, and the shape of the entire part with all bends flattened and bend factors considered. Also known as stretch-out.
Folding	Sheet metal pattern forming method.
Hemming	A sheet metal feature that folds back at the edge of a part so that the two layers are flush. A hem can be open, closed, double, or teardrop.
Joggle	In sheet metal design, a means to offset material so that material can be overlapped.
Lateral Surface	All sides of an object, excluding its base and top.

Rivet	Permanent fasteners. Generally used to hold sheet metal or rolled steel shapes together. The smooth cylindrical shaft is inserted into the aligned holes of matting parts. The tail is then formed to create a head on the opposite end of the shaft.  *Before installation (a), After installation (b)*
Rolling	Sheet metal pattern forming method in which sheet metal is formed into a circular shape.
Seaming	A sheet metal feature that folds back at the edge of a part and uses a seam to join the two layers.
Sheet metal bend plate	In sheet metal design, that portion of the (flat or folded) model that contains the material that deforms during a bend or fold.
Sheet metal edge	In sheet metal design, the thin face (material thickness) at the edge of a part. In part modeling, edges are the boundary between faces on the solid model.
Sheet metal face	In sheet metal design, faces (the sketched outline of material added to a part) are planar and have a uniform thickness defined by the reserved sheet metal parameter Thickness. In part modeling, faces have no thickness and are bounded by edges.
Sheet metal plate	In sheet metal design, a plate that can be used synonymously with a face.
Soldering	A fabrication process in which two or more metal items are joined together by melting and then flowing a filler metal into the joint.
Stamping	Process of placing flat sheet metal in either blank or coil form into a stamping press where a tool and die surface forms the metal into a net shape.
Truncated Cylinder	The geometric solid produced when a cylinder is cut by a plane that is not parallel to the base.
Welding	A fabrication process that joins materials, usually metals or thermoplastics, by using high heat to melt the parts together and allowing them to cool, causing fusion.

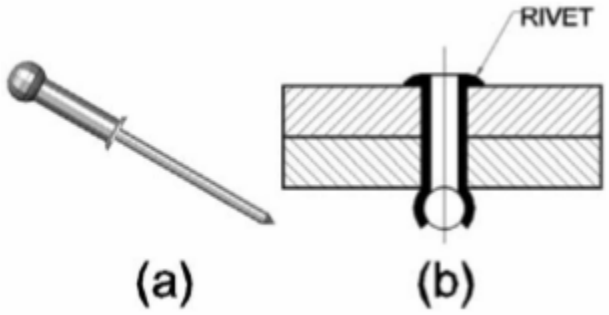
Course	IV23 Drafting III - Engineering			
Essential Standard	6.00	C3	10%	Apply Procedures to Create Sheet Metal Parts.
Indicator	6.02	N/A	N/A	Apply the concepts used to Modify sheet metal parts.
Culminating Question **Essential Question**	**How are techniques applied to modify sheet metal parts in the software?** ● What are the concepts associated with sheet metal processing and patterns?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand concepts associated with sheet metal and patterns.
c. Apply concepts used to modify sheet metal parts.

INSTRUCTIONAL ACTIVITIES-6.02	
A. Content Literacy Terminology	
Resource(s)	**(See 6.02.1)**
B. Understand concepts associated with sheet metal and patterns. *Note: Activity combines with Indicators 6.01 & 6.03.*	
Activity	Research Compilation-Sheet Metal and Pattern Development
Teacher Instructions	● Provide copy of *Research Compilation-Sheet Metal and Pattern Development*. Facilitate students researching answers to leading questions. ● Facilitate students sketching for Part Two. ● Break students into small groups or pairs to review answers to questions and share sketches. ● Students will be able to identify common concepts related to sheet metal and patterns.
Student Directions	● Identify common concepts related to sheet metal and patterns by researching answers to leading questions and completing sketches. ● Review answers to questions and share sketches.
Resource(s)	 Research Compilation- Sheet
C. Apply concepts used to modify sheet metal parts 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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-6.02.1	
Bend	In sheet metal design, a corner seam can be converted to a bend. A bend is the small radius between faces that represents the fabrication process in which bends are applied to a flat pattern to form a 3D part.
Bend allowance	In sheet metal design, a factor applied when converting a bend to an unfolded condition. Material thickness, material type, and bend radius are used to calculate the amount of distortion.
Bend lines	In sheet metal design, bend lines can be displayed in the flat pattern drawing view to show the start, center, and end of the bend zone.
Bend relief	In sheet metal design, a small notch cut at the end of a bend to prevent material from tearing.
Bend zone	In sheet metal design, the area where material deformation takes place, often referred to in the flat pattern as the area between bend lines.
Corner break	In sheet metal design, a corner break applies either a radius or chamfer edge to the end of a flange.
Corner relief	In sheet metal design, a small notch that prevents material from tearing is placed where two bends join.
Corner seam	In sheet metal design, a corner seam mates edges where two bends join, including the alignment or overlap of edges, miter of angled flanges, and inserts corner relief.
Cut	In sheet metal design, cuts are sketched shapes that define material to remove from a single face, a specified depth, or an entire 3D part.
Cut length	For frames, the overall length of the raw stock required to create a structural member (including enough material to accommodate end treatments on either end).
Cut list	A cut list is a table listing components with dimensions and cut treatment information for fabrication and accounting purposes. Each treatment on a frame member produces a custom property called CUTDETAIL(n), which you can add as manufacturing notes in the cut list.
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.
Flat Pattern	The shape of the sheet metal part before it is formed, to create drawings for manufacturing. Flat patterns show bend lines, bend zones, punch locations, and the shape of the entire part with all bends flattened and bend factors considered. Also known as stretch-out.
Folding	Sheet metal pattern forming method.
Hemming	A sheet metal feature that folds back at the edge of a part so that the two layers are flush. A hem can be open, closed, double, or teardrop.
Joggle	In sheet metal design, a means to offset material so that material can be overlapped.
Lateral Surface	All sides of an object, excluding its base and top.

Rivet	Permanent fasteners. Generally used to hold sheet metal or rolled steel shapes together. The smooth cylindrical shaft is inserted into the aligned holes of matting parts. The tail is then formed to create a head on the opposite end of the shaft.  *Before installation (a), After installation (b)*
Rolling	Sheet metal pattern forming method in which sheet metal is formed into a circular shape.
Seaming	A sheet metal feature that folds back at the edge of a part and uses a seam to join the two layers.
Sheet metal bend plate	In sheet metal design, that portion of the (flat or folded) model that contains the material that deforms during a bend or fold.
Sheet metal edge	In sheet metal design, the thin face (material thickness) at the edge of a part. In part modeling, edges are the boundary between faces on the solid model.
Sheet metal face	In sheet metal design, faces (the sketched outline of material added to a part) are planar and have a uniform thickness defined by the reserved sheet metal parameter Thickness. In part modeling, faces have no thickness and are bounded by edges.
Sheet metal plate	In sheet metal design, a plate that can be used synonymously with a ace.
Soldering	A fabrication process in which two or more metal items are joined together by melting and then flowing a filler metal into the joint.
Stamping	Process of placing flat sheet metal in either blank or coil form into a stamping press where a tool and die surface forms the metal into a net shape.
Truncated Cylinder	The geometric solid produced when a cylinder is cut by a plane that is not parallel to the base.
Welding	A fabrication process that joins materials, usually metals or thermoplastics, by using high heat to melt the parts together and allowing them to cool, causing fusion.

Course	IV23 Drafting III - Engineering			
Essential Standard	6.00	C3	10%	Apply Procedures to Create Sheet Metal Parts.
Indicator	6.03	N/A	N/A	Apply how to create a Flat Pattern.
Culminating Question **Essential Questions**	**How are techniques applied to create flat patterns in the software?** ● What are the concepts associated with sheet metal processing and patterns? ● What is the purpose of a flat pattern? ● What do common flat patterns look like?			

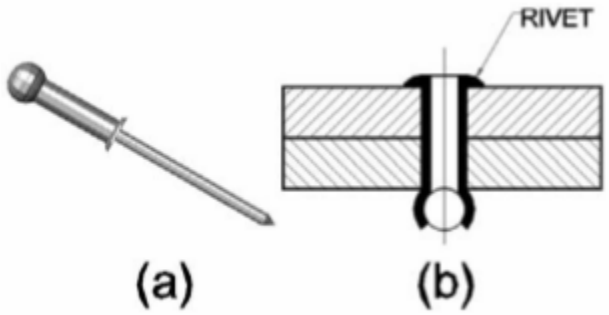
UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand concepts associated with sheet metal and patterns.
c. Understand the purposes of flat patterns.
d. Understand common flat patterns.
e. Apply concepts used to create flat patterns in the software.

INSTRUCTIONAL ACTIVITIES-6.03	
A. Content Literacy Terminology	
Resource(s)	**(See 6.03.1)**
B. Understand concepts associated with sheet metal and patterns. *Note: Activity combines with Indicators 6.01 & 6.02.*	
Activity	Research Compilation-Sheet Metal and Pattern Development
Teacher Instructions	● Provide copy of *Research Compilation-Sheet Metal and Pattern Development*. Facilitate students researching answers to leading questions. ● Facilitate students sketching for Part Two. ● Break students into small groups or pairs to review answers to questions and share sketches. ● Students will be able to identify common concepts related to sheet metal and patterns.
Student Directions	● Identify common concepts related to sheet metal and patterns by researching answers to leading questions and completing sketches. ● Review answers to questions and share sketches.
Resource(s)	 Research Compilation- Sheet
C. Understand the purposes of flat patterns. **D. Understand common flat patterns.** *Note: Activity applies to both C & D.*	
Activity	Project-Based Learning Activity-Pattern Development
Teacher Instructions	● Facilitate whole-class direct instruction using *Project-Based Learning Activity-Pattern Development-Teacher Led Review and Student Creation Instructions*. ● Facilitate instructions and expectations for the project (end of presentation). ● Facilitate students creating their own stretch-out or flat pattern on paper. ● Facilitate students creating a physical model using available resources. ● Facilitate student creation of their 3D Digital model in the software. ● Provide feedback using *Project-Based Learning-Pattern Development-Mastery Based Rubric*. *Note*: Students will need basic knowledge of sheet metal in software prior to creating in the software (indicators 6.01 & 6.02 and subgoal E of 6.03)
Student Directions	● Participate in direct instruction on Pattern Development. ● Create a stretch-out or flat pattern on paper. ● Create a physical model using available resources. ● Create your 3D Digital model in the software.

Resource(s)	Project-Based Learning Activity- Pa Project-Based Learning- Pattern De
E. Apply concepts used to create sheet metal parts 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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-6.03.1	
Bend	In sheet metal design, a corner seam can be converted to a bend. A bend is the small radius between faces that represents the fabrication process in which bends are applied to a flat pattern to form a 3D part.
Bend allowance	In sheet metal design, a factor applied when converting a bend to an unfolded condition. Material thickness, material type, and bend radius are used to calculate the amount of distortion.
Bend lines	In sheet metal design, bend lines can be displayed in the flat pattern drawing view to show the start, center, and end of the bend zone.
Bend relief	In sheet metal design, a small notch cut at the end of a bend to prevent material from tearing.
Bend zone	In sheet metal design, the area where material deformation takes place, often referred to in the flat pattern as the area between bend lines.
Corner break	In sheet metal design, a corner break applies either a radius or chamfer edge to the end of a flange.
Corner relief	In sheet metal design, a small notch that prevents material from tearing is placed where two bends join.
Corner seam	In sheet metal design, a corner seam mates edges where two bends join, including the alignment or overlap of edges, miter of angled flanges, and inserts corner relief.
Cut	In sheet metal design, cuts are sketched shapes that define material to remove from a single face, a specified depth, or an entire 3D part.
Cut length	For frames, the overall length of the raw stock required to create a structural member (including enough material to accommodate end treatments on either end).
Cut list	A cut list is a table listing components with dimensions and cut treatment information for fabrication and accounting purposes. Each treatment on a frame member produces a custom property called CUTDETAIL(n), which you can add as manufacturing notes in the cut list.
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.
Flat Pattern	The shape of the sheet metal part before it is formed, to create drawings for manufacturing. Flat patterns show bend lines, bend zones, punch locations, and the shape of the entire part with all bends flattened and bend factors considered. Also known as stretch-out.
Folding	Sheet metal pattern forming method.
Hemming	A sheet metal feature that folds back at the edge of a part so that the two layers are flush. A hem can be open, closed, double, or teardrop.
Joggle	In sheet metal design, a means to offset material so that material can be overlapped.
Lateral Surface	All sides of an object, excluding its base and top.

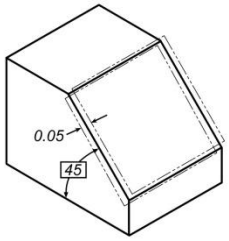
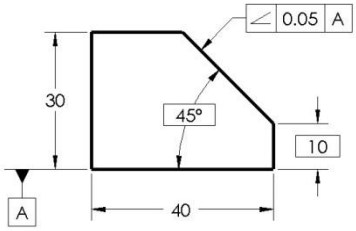
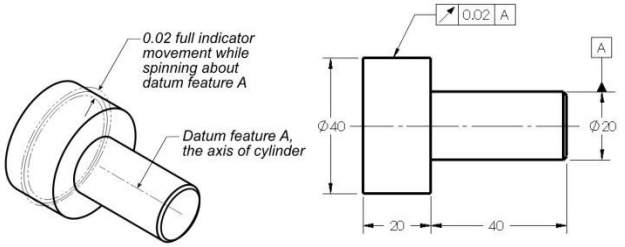
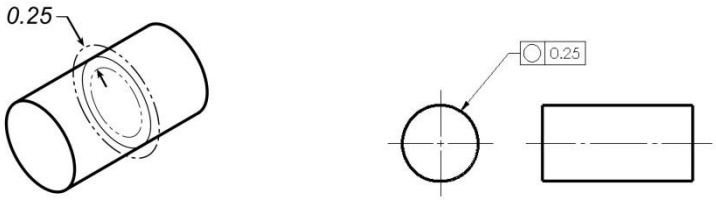
Parallel Line Development	Method of pattern development used to develop patterns of square, rectangular and cylindrical shapes (prisms). The method divides the surface into a series of parallel lines to determine the shape of a pattern.
Radial Line Development	Method of pattern development used to develop patterns for objects that have a tapering form with lines converging to a common point, called the apex point.
Rivet	Permanent fasteners. Generally used to hold sheet metal or rolled steel shapes together. The smooth cylindrical shaft is inserted into the aligned holes of matting parts. The tail is then formed to create a head on the opposite end of the shaft.  *Before installation (a), After installation (b)*
Rolling	Sheet metal pattern forming method in which sheet metal is formed into a circular shape.
Seaming	A sheet metal feature that folds back at the edge of a part and uses a seam to join the two layers.
Sheet Metal Bend Plate	In sheet metal design, that portion of the (flat or folded) model that contains the material that deforms during a bend or fold.
Sheet metal edge	In sheet metal design, the thin face (material thickness) at the edge of a part. In part modeling, edges are the boundary between faces on the solid model.
Sheet metal face	In sheet metal design, faces (the sketched outline of material added to a part) are planar and have a uniform thickness defined by the reserved sheet metal parameter Thickness. In part modeling, faces have no thickness and are bounded by edges.
Sheet metal plate	In sheet metal design, a plate that can be used synonymously with a face.
Soldering	A fabrication process in which two or more metal items are joined together by melting and then flowing a filler metal into the joint.
Stamping	Process of placing flat sheet metal in either blank or coil form into a stamping press where a tool and die surface forms the metal into a net shape.
Truncated Cylinder	The geometric solid produced when a cylinder is cut by a plane that is not parallel to the base.
Welding	A fabrication process that joins materials, usually metals or thermoplastics, by using high heat to melt the parts together and allowing them to cool, causing fusion.

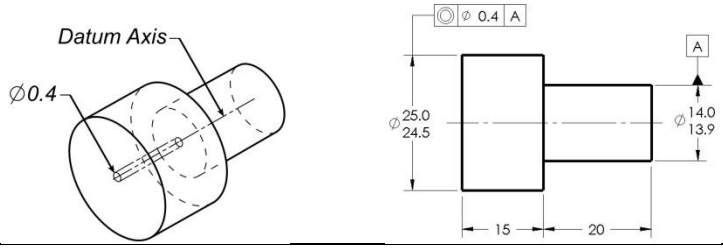
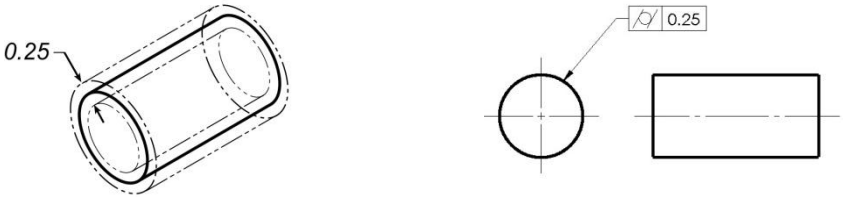
Course	IV23 Drafting III - Engineering			
Essential Standard	7.00	B2	8%	Understand Procedures to create Geometric Dimensions and Tolerances.
Indicator	7.01	N/A	N/A	Understand geometric dimensioning and tolerancing terms, symbols, and concepts.
Culminating Question **Essential Questions**	**What are the concepts for geometric dimensioning and tolerancing?** ● What is the purpose of GD&T? ● How is a datum applied to certain types of features? ● What are the types of tolerances and their zones?			

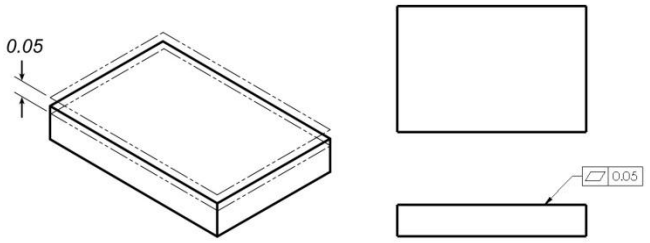
UNPACKED CONTENT

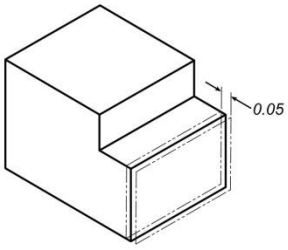
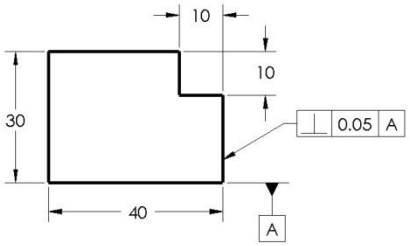
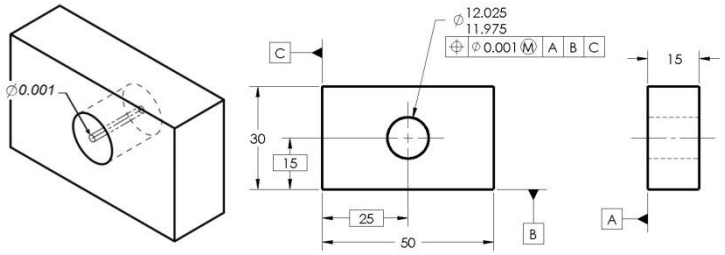
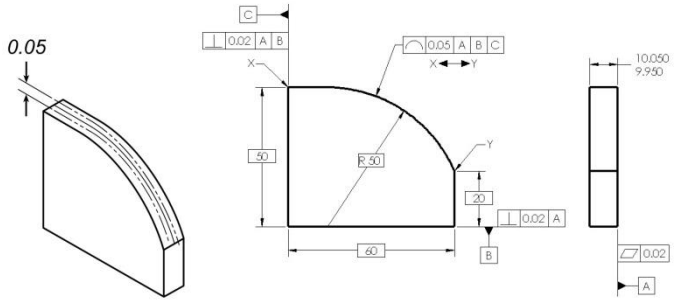
a. Content Literacy Terminology.
b. Understand the purpose of GD&T.
c. Understand how a datum is applied to a feature.
d. Understand the types of tolerances and their zones.

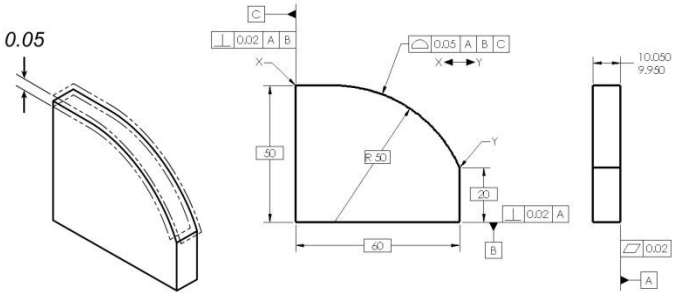
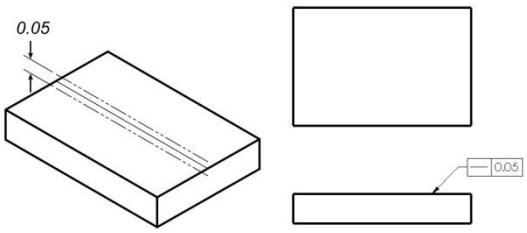
INSTRUCTIONAL ACTIVITIES-7.01	
A. Content Literacy Terminology	
Resource(s)	**(See 7.01.1)**
B. Understand the purpose of GD& T. **C. Understand how a datum is applied to a feature.** *Note: Activity applies to both B&C and combines with part of Indicator 7.02.*	
Activity	Guided Questioning- Geometric Dimensioning and Tolerances- Introduction and Datums
Teacher Instructions	• Provide copy of *Guided Questioning- Geometric Dimensioning and Tolerances- Introduction and Datums- Leading Questions* and *Guided Questioning- Geometric Dimensioning and Tolerances- Introduction and Datums- Presentation*. Facilitate students working independently to answer leading questions using the PowerPoint Presentation as a resource. • Facilitate whole-class, small-groups, or individuals review of questions/concepts. • Students will be able to identify datums and how they are applied in GD&T.
Student Directions	• Identify datums and how they are applied in GD&T by working independently to answer leading questions using the PowerPoint Presentation as a resource. • Participate in review of questions/concepts.
Resource(s)	Guided Questioning- Geom Guided Questioning- Geom
D. Understand the symbols associated with tolerances and their zones. *Note: Activity combines with part of Indicator 7.02.*	
Activity	Structured Notes-Making Sense of Tolerances
Teacher Instructions	• Provide a hardcopy of *Structured Notes-Making Sense of Tolerances-Handout* and digital copy of *Structured Notes-Making Sense of Tolerances-Reading* or *Structured Notes-Making Sense of Tolerances-Presentation* according to learning needs. Facilitate students completing structured notes as explained on the front page for each concept and draw the zone in given squares. • Facilitate whole-class, small group or individual review using *Structured Notes-Making Sense of Tolerances-Presentation*. • Students will be able to identify the types of tolerances included in GD&T and describe their zones.
Student Directions	• Identify the types of tolerances included in GD&T and describe their zones by completing structured notes using the provided resource. • Participate in review of concepts.
Resource(s)	Structured Notes- Making Sense of To Structured Notes- Making Sense of To Structured Notes- Making Sense of To

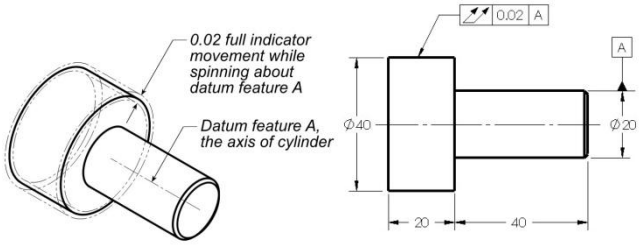
Content Literacy Terminology-7.01.1	
Angularity	Orientation tolerance of a surface, center plane, or axis at a specified angle (other than 90º) from a datum plane or axis. All points on the surface must lie between two parallel planes. Perfect geometry is located using basic dimensions. The shape of the tolerance zone is a *3D area* between two parallel planes.  
Circular Runout	Type of Runout (composite) Tolerance in which all points on the surface must lie between two concentric circles relative to the datum feature. The shape of the tolerance zone is the *2D area* between the two concentric circles. 
Circularity (Roundness)	Form tolerance specifying a tolerance zone bounded by two concentric circles within which each circular element of a surface must lie. All points on the surface must lie between two concentric circles. The shape of the tolerance zone is a *2D area* between two concentric circles. 
Clearance Fit	An internal member fits in an external member and always leaves a space between the parts.

Concentricity	Location tolerance where the median points of all diametrically opposed elements of a figure of revolution are congruent with the axis of a datum feature. All points on the axis must lie within a cylinder relative to the datum axis. The shape of the tolerance zone is a *3D area* within the cylinder. 
Cylindricity	Form tolerance of a surface of revolution in which all points of the surface are equidistant from a common axis. All points on the surface must lie between two concentric cylinders. The shape of the tolerance zone is a *3D area* between two concentric cylinders. 
Datum	A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.
Datum Feature	An actual feature of a part that is used to establish a datum.
Datum Feature Simulator	A surface of adequately precise manufacturing or inspection equipment contacting the datum feature(s). Used to establish the simulated datum(s). Typically, this surface must be at least 10 times better in quality (flatness) than the tolerances specified on the drawing.
Datum Feature Symbol	The symbolic means of indicating a datum feature. It consists of a capital letter enclosed in a square frame and a leader line extending from the frame to the concerned feature, terminating with a triangle.
Datum Reference Frame	Three perpendicular theoretically established planes (similar to the Cartesian Coordinate System) used in GD&T to orient or locate features on a part. Planes are established relative to features on the actual object.
Design Size	The design size is the size from which the limits of size are derived by the application of tolerances.
Feature	A general term applied to a physical portion of a part, such as a surface, pin, tab, hole, or slot.
Feature of Size	A cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.
Feature without Size	Typically, this is singular a planar surface.

Flatness	Form tolerance in which all surface elements are in one plane. All points on the surface must lie between two parallel planes. The shape of the tolerance zone is a *3D area* between two parallel planes. 
Geometric Dimensioning & Tolerancing (GD&T)	An international engineering language that is used on engineering drawings to describe products in three dimensions. It is a precise mathematical language that describes the form, orientation and location of part features in zones of tolerance. Advantages: • Clearer intent of the designer. • Better communication throughout the design process. • Better choices for manufacturing/machining. • Better/more accurate choices for inspection. • Leaves almost nothing that can be interpreted more than one way.
Least Material Condition (LMC)	The geometric tolerance applies only at the feature's least material within the stated limits of size.
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.
Location Dimension	Label the location of each geometric feature within an object or view.
Maximum Material Condition (MMC)	The geometric tolerance applies only at the feature's maximum material within the stated limits of size.
Parallelism	Orientation tolerance of a surface or center plane, equidistant at all points from a datum plane; or an axis, equidistant along its length from one or more datum planes or a datum axis. All points on the surface must lie between two parallel planes. The shape of the tolerance zone is a *3D area* between two parallel planes. 

Perpendicularity	Orientation tolerance of a surface, center plane, or axis at a right angle to a datum plane or axis. All points on the surface must lie between two parallel planes. The shape of the tolerance zone is a *3D area* between two parallel planes.  
Position	Location tolerance which defines a zone within which the center, axis, or center plane of a feature of size is permitted to vary from a true position. All points on the axis must lie within a cylinder. The cylinder is located with basic dimensions from the datums. The shape of the tolerance zone is a *3D area* within the cylinder. 
Profile of a Line	Type of profile tolerance in which each point on the specified path must lie between two parallel contours. The shape of the tolerance zone is a *2D area* between the two contours. Perfect geometry is located with basic dimensions. 

Profile of a Surface	Type of profile tolerance in which Each point on the surface must lay between two parallel/ concentric contours. The shape of the tolerance zone is a *3D area* between the two contours. Perfect geometry is located with basic dimensions. 
Profile Tolerance	The outline of an object in a given plane. Profiles are formed by projecting a three-dimensional figure onto a plane or by taking cross sections through the figure.
Rectangular Coordinate Dimensioning	Type of dimensioning system is specifically used when computer-controlled production machines are used to manufacture parts.
Regardless of Feature Size	The geometric tolerance applies at any increment of size of the feature within its size tolerance.
Runout	Composite tolerance used to control the functional relationship of one or more features of a part to a datum axis.
Size Dimension	Label the length of the overall width, height, and depth of an object.
Straightness	Form tolerance where an element of a surface or a centerline is a straight line. Each longitudinal element on the surface must lie between two parallel lines. The shape of the tolerance zone is a *2D area* between two parallel lines. 
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).

Total Runout	Type of Runout (composite) Tolerance in which all points on the surface must lie between two concentric cylinders relative to the datum feature. The shape of the tolerance zone is the *3D area* between the two concentric cylinders. 

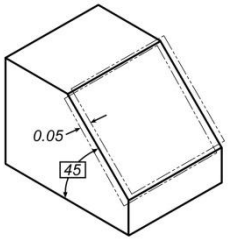
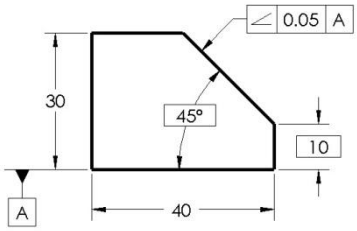
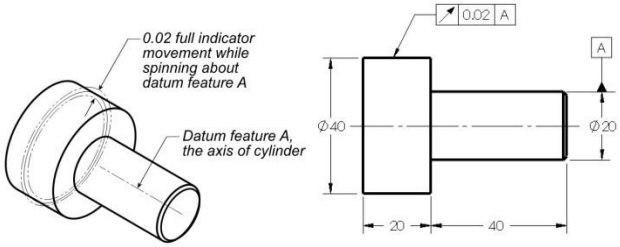
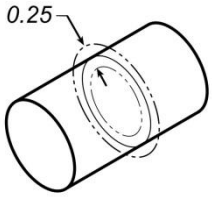
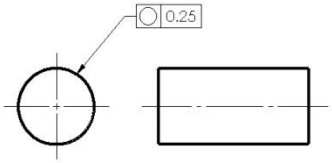
Course	IV23 Drafting III - Engineering			
Essential Standard	7.00	B2	8%	Understand Procedures to create Geometric Dimensions and Tolerances.
Indicator	7.02	N/A	N/A	Apply geometric dimensioning and tolerancing terms, symbols, and concepts.
Culminating Question **Essential Questions**	**What are the concepts for geometric dimensioning and tolerancing?** ● What is the purpose of GD&T? ● How is a datum applied to certain types of features? ● What are the symbols used for tolerances and GD&T? ● What is the purpose of the symbols used in GD&T? ● How is a Feature Control Frame read and created?			

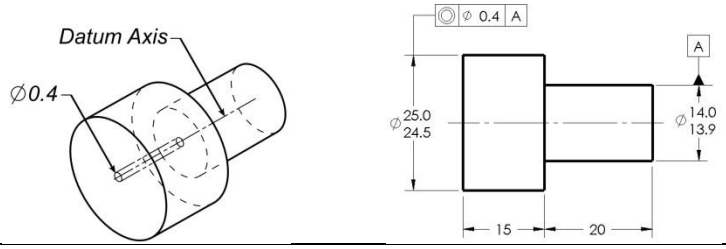
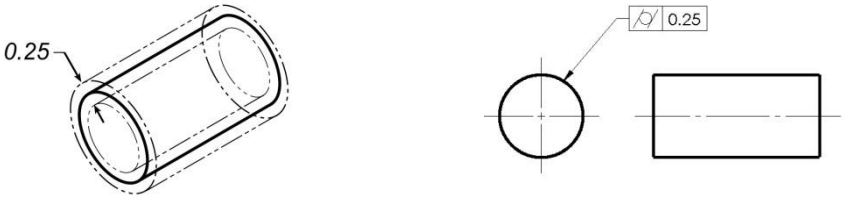
UNPACKED CONTENT

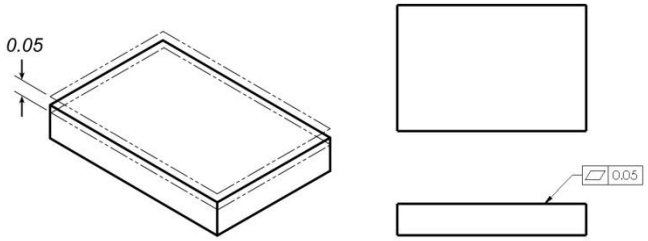
a. Content Literacy Terminology.
b. Understand the purpose of GD& T.
c. Understand how a datum symbol is applied to a feature.
d. Identify the symbols associated with tolerances and their zones.
e. Understand the symbols associated with GD&T.
f. Understand Feature Control Frames.

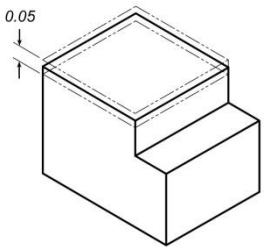
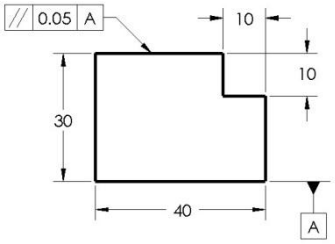
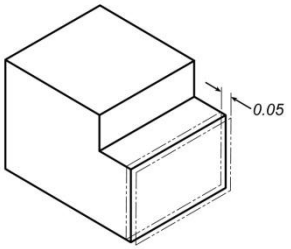
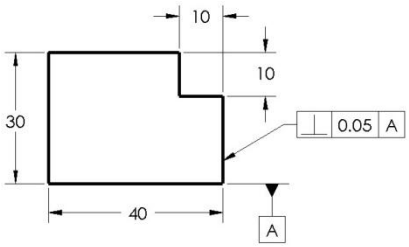
INSTRUCTIONAL ACTIVITIES-7.02	
A. Content Literacy Terminology	
Resource(s)	**(See 7.02.1)**
B. Understand the purpose of GD& T. **C. Understand how a datum symbol is applied to a feature.** *Note: Activity applies to both B &C and combines with part of Indicator 7.01.*	
Activity	Guided Questioning-Geometric Dimensioning and Tolerances-Introduction and Datums
Teacher Instructions	• Provide copy of *Guided Questioning- Geometric Dimensioning and Tolerances- Introduction and Datums- Leading Questions* and *Guided Questioning- Geometric Dimensioning and Tolerances- Introduction and Datums- Presentation*. Facilitate students working independently to answer leading questions using the PowerPoint Presentation as a resource. • Facilitate whole-class, small-groups, or individuals review of questions/concepts. • Students will be able to identify datums and how they are applied in GD&T.
Student Directions	• Identify datums and how they are applied in GD&T by working independently to answer leading questions using the PowerPoint Presentation as a resource. • Participate in review of questions/concepts.
Resource(s)	Guided Questioning- Geom Guided Questioning- Geom
D. Identify the symbols associated with tolerances and their zones. *Note: Activity combines with part of Indicator 7.01.*	
Activity	Structured Notes-Making Sense of Tolerances
Teacher Instructions	• Provide a hardcopy of *Structured Notes-Making Sense of Tolerances-Handout* and digital copy of *Structured Notes-Making Sense of Tolerances-Reading* or *Structured Notes-Making Sense of Tolerances-Presentation* according to learning needs. Facilitate students completing structured notes as explained on the front page for each concept and draw the zone in given squares. • Facilitate whole-class, small group or individual review using *Structured Notes-Making Sense of Tolerances-Presentation*. • Students will be able to identify the types of tolerances included in GD&T and describe their zones.
Student Directions	• Identify the types of tolerances included in GD&T and describe their zones by completing structured notes using the provided resource. • Participate in review of concepts.
Resource(s)	Structured Notes- Making Sense of To Structured Notes- Making Sense of To Structured Notes- Making Sense of To

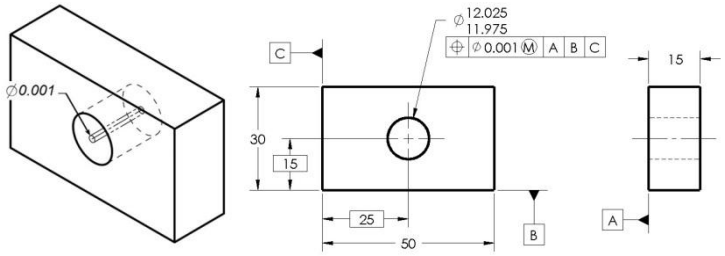
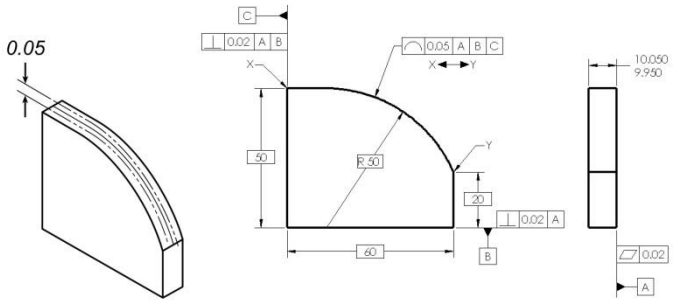
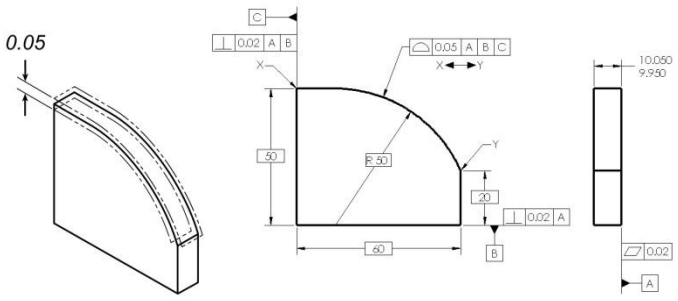
E. Understand the symbols associated with GD&T.	
Activity	Listing and Grouping with Leading Questions- Geometric Characteristics Symbols
Teacher Instructions	● Place students in pairs. Provide hardcopy of *Listing and Grouping with Leading Questions-Geometric Characteristics Symbols* to all students. ● Facilitate students working in pairs to fill in the table and answer questions. ● Facilitate whole-class review using student answers to drive discussion. ● Students will be able to explain concepts associated with GD&T. *Notes:* Activity designed to follow *Activity Structured Notes-Making Sense of Tolerances.*
Student Directions	● Explain concepts associated with GD&T by working in pairs to fill in the table and answer questions. ● Participate in a whole-class review of concepts.
Resource(s)	 Listing and Grouping with Lead
F. Understand Feature Control Frames.	
Activity	Interactive Notes-Geometric Dimensioning and Tolerances-Feature Control Frames
Teacher Instructions	● Provide hardcopy of *Interactive Notes-Geometric Dimensioning and Tolerances-Feature Control Frames-Handout*. Facilitate whole-class direct instruction using *Interactive Notes-Geometric Dimensioning and Tolerances-Feature Control Frames-Presentation* while students complete interactive notes.
Student Directions	● Read a Feature Control Frame by completing the interactive notes along with the guided instruction.
Resource(s)	 Interactive Notes-Geometric Dimensio  Interactive Notes-Geometric Dimensio

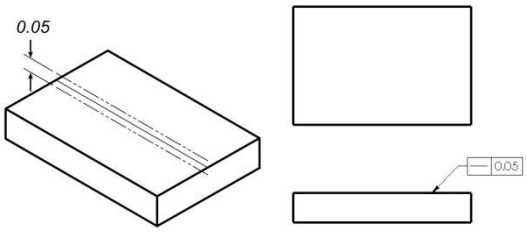
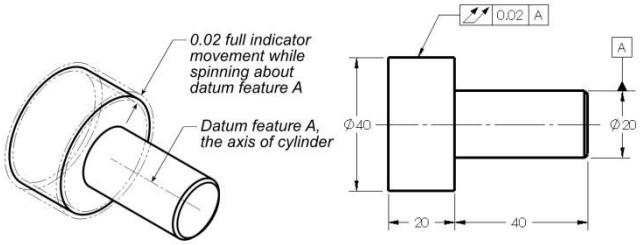
Content Literacy Terminology-7.02.1	
Angularity	Orientation tolerance of a surface, center plane, or axis at a specified angle (other than 90º) from a datum plane or axis. All points on the surface must lie between two parallel planes. Perfect geometry is located using basic dimensions. The shape of the tolerance zone is a *3D area* between two parallel planes.  
Circular Runout	Type of Runout (composite) Tolerance in which all points on the surface must lie between two concentric circles relative to the datum feature. The shape of the tolerance zone is the *2D area* between the two concentric circles. 
Circularity (Roundness)	Form tolerance specifying a tolerance zone bounded by two concentric circles within which each circular element of a surface must lie. All points on the surface must lie between two concentric circles. The shape of the tolerance zone is a *2D area* between two concentric circles.  

Concentricity	Location tolerance where the median points of all diametrically opposed elements of a figure of revolution are congruent with the axis of a datum feature. All points on the axis must lie within a cylinder relative to the datum axis. The shape of the tolerance zone is a *3D area* within the cylinder. 
Cylindricity	Form tolerance of a surface of revolution in which all points of the surface are equidistant from a common axis. All points on the surface must lie between two concentric cylinders. The shape of the tolerance zone is a *3D area* between two concentric cylinders. 
Datum	A theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.
Datum Feature	An actual feature of a part that is used to establish a datum.
Datum Feature Simulator	A surface of adequately precise manufacturing or inspection equipment contacting the datum feature(s). Used to establish the simulated datum(s). Typically, this surface must be at least 10 times better in quality (flatness) than the tolerances specified on the drawing.
Datum Feature Symbol	The symbolic means of indicating a datum feature. It consists of a capital letter enclosed in a square frame and a leader line extending from the frame to the concerned feature, terminating with a triangle.
Datum Reference Frame	Three perpendicular theoretically established planes (similar to the Cartesian Coordinate System) used in GD&T to orient or locate features on a part. Planes are established relative to features on the actual object.
Feature	A general term applied to a physical portion of a part, such as a surface, pin, tab, hole, or slot.
Feature Control Frame	Frame containing GD&T measurements, datums and symbols used to specify acceptable tolerance zones for the features relative to the Datum Reference Frame.
Feature of Size	A cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.

<table>
<tr><td>Feature without Size</td><td>Typically, this is singular a planar surface.</td></tr>
<tr><td>Flatness</td><td>Form tolerance in which all surface elements are in one plane. All points on the surface must lie between two parallel planes. The shape of the tolerance zone is a 3D area between two parallel planes.
 
</td></tr>
<tr><td>GD&T Symbols</td><td>
<table>
<tr><th></th><th>TYPE OF TOLERANCE</th><th>CHARACTERISTIC</th><th>SYM</th></tr>
<tr><td rowspan="4">FOR INDIVIDUAL FEATURES</td><td rowspan="4">FORM</td><td>STRAIGHTNESS</td><td></td></tr>
<tr><td>FLATNESS</td><td></td></tr>
<tr><td>CIRCULARITY</td><td></td></tr>
<tr><td>CYLINDRICITY</td><td></td></tr>
<tr><td rowspan="2">FOR INDIVIDUAL OR RELATED FEATURES</td><td rowspan="2">PROFILE</td><td>PROFILE OF A LINE</td><td></td></tr>
<tr><td>PROFILE OF A SURFACE</td><td></td></tr>
<tr><td rowspan="8">FOR RELATED FEATURES</td><td rowspan="3">ORIENTATION</td><td>ANGULARITY</td><td></td></tr>
<tr><td>PERPENDICULARITY</td><td></td></tr>
<tr><td>PARALLELISM</td><td></td></tr>
<tr><td rowspan="3">LOCATION</td><td>POSITION</td><td></td></tr>
<tr><td>CONCENTRICITY</td><td></td></tr>
<tr><td>SYMMETRY</td><td></td></tr>
<tr><td rowspan="2">RUNOUT</td><td>CIRCULAR RUNOUT</td><td></td></tr>
<tr><td>TOTAL RUNOUT</td><td></td></tr>
</table>
</td></tr>
<tr><td>Geometric Dimensioning & Tolerancing (GD&T)</td><td>An international engineering language that is used on engineering drawings to describe products in three dimensions. It is a precise mathematical language that describes the form, orientation and location of part features in zones of tolerance. Advantages:
Clearer intent of the designer.
Better communication throughout the design process.
Better choices for manufacturing/machining.
Better/more accurate choices for inspection.
Leaves almost nothing that can be interpreted more than one way.</td></tr>
<tr><td>Least Material Condition (LMC)</td><td>The geometric tolerance applies only at the feature’s least material within the stated limits of size.</td></tr>
<tr><td>Maximum Material Condition (MMC)</td><td>The geometric tolerance applies only at the feature’s maximum material within the stated limits of size.</td></tr>
</table>

Parallelism	Orientation tolerance of a surface or center plane, equidistant at all points from a datum plane; or an axis, equidistant along its length from one or more datum planes or a datum axis. All points on the surface must lie between two parallel planes. The shape of the tolerance zone is a *3D area* between two parallel planes.  
Perpendicularity	Orientation tolerance of a surface, center plane, or axis at a right angle to a datum plane or axis. All points on the surface must lie between two parallel planes. The shape of the tolerance zone is a *3D area* between two parallel planes.  

Position	Location tolerance which defines a zone within which the center, axis, or center plane of a feature of size is permitted to vary from a true position. All points on the axis must lie within a cylinder. The cylinder is located with basic dimensions from the datums. The shape of the tolerance zone is a *3D area* within the cylinder. 
Profile of a Line	Type of profile tolerance in which each point on the specified path must lie between two parallel contours. The shape of the tolerance zone is a *2D area* between the two contours. Perfect geometry is located with basic dimensions. 
Profile of a Surface	Type of profile tolerance in which Each point on the surface must lay between two parallel/ concentric contours. The shape of the tolerance zone is a *3D area* between the two contours. Perfect geometry is located with basic dimensions. 
Profile Tolerance	The outline of an object in a given plane. Profiles are formed by projecting a three-dimensional figure onto a plane or by taking cross sections through the figure.
Regardless of Feature Size	The geometric tolerance applies at any increment of size of the feature within its size tolerance.

Runout	Composite tolerance used to control the functional relationship of one or more features of a part to a datum axis.
Straightness	Form tolerance where an element of a surface or a centerline is a straight line. Each longitudinal element on the surface must lie between two parallel lines. The shape of the tolerance zone is a *2D area* between two parallel lines. 
Tolerance	Amount a specific dimension can vary (the difference between the limits).
Total Runout	Type of Runout (composite) Tolerance in which all points on the surface must lie between two concentric cylinders relative to the datum feature. The shape of the tolerance zone is the *3D area* between the two concentric cylinders. 

Course	IV23 Drafting III - Engineering			
Essential Standard	8.00	C3	5%	Apply Procedures to create a Professional Portfolio.
Indicator	8.01	N/A	N/A	Apply the methods used for creating a digital portfolio.
Culminating Question **Essential Questions**	**What are the methods for creating a digital portfolio?** ● What are the common types of portfolios and what is their purpose? ● What is commonly included in a portfolio? ● How are artifacts best selected and documented for a portfolio? ● How are digital portfolios created?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand the common types of portfolios (digital and hard copy) and their purpose.
c. Understand what is included in a portfolio.
d. Identify appropriate artifacts.
e. Create a digital portfolio.

INSTRUCTIONAL ACTIVITIES-8.01	
A. Content Literacy Terminology	
Resource(s)	**(See 8.01.1)**
B. Understand the common types of portfolios (digital & hard copy) and their purpose. **C. Understand what is included in a portfolio.** **D. Identify appropriate artifacts.** *Note: Activity includes Unpacked Content for B, C &D.*	
Activity	Concept Introduction-Portfolios
Teacher Instructions	● Facilitate whole-class direct instruction using *Concept Introduction-Portfolios-Presentation*. Select industry examples to share with students (or previous student work when available) after presentation/direct-instruction is complete. ● Facilitate student list creation of possible artifacts/topics they might like to include in personal portfolios. Students will look through personal work from Cluster Courses (i.e. Drafting I, ENG II, other CTE courses), as well as outside of courses. Students can include work which is not directly related to the course, however the goal is to highlight work related to Engineering and this course's Essential Standards/Indicators.
Student Directions	● Participate in whole-class direct instruction on portfolios. Review provided examples. ● Select appropriate artifacts for a digital portfolio by creating a list of possible artifacts/topics you might like to include in your personal portfolio. Look through personal work from Cluster Courses (i.e. Drafting I, ENG II, other CTE courses), as well as outside of courses. An artifact can be included which is not directly related to the course, however the goal is to highlight work related to Engineering and this course's Essential Standards/Indicators.
Resource(s)	 Concept Introduction- Portfo
E. Create a digital portfolio.	
Activity	Digital Portfolios
Teacher Instructions	● Select main source to house/create portfolio according to availability within PSU or student ability. Common sources are slide presentations (Google or PowerPoint) and/or websites (examples of free resources listed below). ● Select main components/sections of the portfolio (website), but additional can be allowed. Main components/sections should reflect Essential Standard/Indicators for the course listed in Blueprint. ● Provide guidance/requirements for artifacts according to completed coursework/project(s). Provide examples of artifact descriptions and assistance to students as needed. Provide assistance to students as needed for inclusion of multimedia.

	● Facilitate students composing a digital portfolio showcasing their work/talents. ● Facilitate students conducting self and/or peer reviews of portfolios. ● Facilitate students sharing portfolios with other students, parents and/or stakeholders when the final draft is completed.
Student Directions	● Compose a digital portfolio showcasing your work/talents by selecting appropriate artifacts. ● Conduct self and/or peer reviews of portfolios. ● Share your portfolio with other students, parents and/or stakeholders when the final draft is completed.
Resource(s)	 Website Portfolio Rubric.docx
Website Resource(s): Free Online Website Creator Examples:	
WIX. (2021). Create a Website You're Proud Of.	https://www.wix.com
Weebly. (2021). Build a professional website that grows with your business.	https://www.weebly.com

Content Literacy Terminology-8.01.1	
Artifact	An included work example or sample in a portfolio.
Portfolio	A digital or hard-copy purposeful collection of significant artifacts that provides tangible evidence of work and abilities.

Course	IV23 Drafting III - Engineering			
Essential Standard	8.00	C3	5%	Apply Procedures to create a Professional Portfolio.
Indicator	8.02	N/A	N/A	Apply the procedures to create Presentations, Rendered Images and Animated Assemblies.
Culminating Question **Essential Questions**	**What are the methods for creating presentations, rendered images and animated assemblies in the software?** ● What are the concepts and terminology associated with presentation files in the software? ● How are presentations created in the software? ● How are rendered images created in the software? ● How are animated assemblies created in the software?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand terminology and concepts associated with presentation files.
c. Apply procedures for creating presentations.
d. Apply procedures for creating rendered images.
e. Apply procedures for creating animated assemblies.

INSTRUCTIONAL ACTIVITIES-8.02	
A. Content Literacy Terminology	
Resource(s)	**(See 8.02.1)**
B. Understand terminology and concepts associated with presentation files.	
Activity	Content Reading Comprehension-Renderings and Presentations
Teacher Instructions	● Provide copies of *Content Reading Comprehension- Renderings and Presentations- Passage* and *Content Reading Comprehension- Renderings and Presentations- Questions*. Facilitate read-aloud of questions before passage as an example of a test-taking technique. ● Facilitate students reading the passage and answering questions. ● Facilitate whole-class review of answers and class discussion on topic. ● Students will be able to define common terms/concepts associated with presentation files.
Student Directions	● Define common terms/concepts associated with presentation files by reading the passage and answering associated questions. ● Participate in whole-class review of answers and class discussion on topic.
Resource(s)	 Content Reading Comprehension- Re  Content Reading Comprehension- Re
C. Apply procedures for creating presentations.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
D. Apply procedures for creating rendered images.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource
E. Apply procedures for creating animated assemblies.	
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 concept(s).
Resource(s)	Autodesk Inventor Resource and SolidWorks Resource

Content Literacy Terminology-8.02.1	
Rendering	Images generated by the software to show the part of assembly as realistically as possible according to assigned materials.
Materials	A combination of different property sets referred to as assets (physical and appearance).
Physical Assets	Properties which provide information about material composition used for simulation, analysis, and to determine properties such as mass, area, and volume.
Appearance Assets	Determine the visual aspects of a material and are assigned to any face, feature, part, or assembly.
Appearance Properties	Properties which provide visual feedback for a material, such as color, texture, bump mapping, and grain.
Animation	A type of presentation file used to show the step-by-step assembly, disassembly, or maintenance operations of your design.
Keyframe	Starting and ending positions at specific times used in the creation of an animation.
Interpolation	Process in which the software calculates intermediate positions between keyframes.
Presentation File (.ipn)	A type of file used within a constraint-based CAD system used to develop scenes, exploded views, animations, and other stylized views of an assembly to aid you in documenting your design.
Presentation View	A specialized assembly view. You develop the specialized views, such as exploded views, in a presentation file and then use them to create drawing views or other presentations.
Scene	A set of 3D objects, including models, lights, and cameras used to create a rendering or animation.
Tweak	Adjustments made to assembly components in an exploded view. You specify the distance and direction of movement to provide better visibility of the components and their relationships.
Exploded View	A view in which parts of an assembly are separated for unobstructed viewing. The direction and distance that parts can be moved are dependent on the settings that you specify when setting up the view. Exploded views are defined in an assembly presentation file and then used to add exploded views to a drawing.

Course	IV23 Drafting III - Engineering			
Essential Standard	8.00	C3	5%	Apply Procedures to create a Professional Portfolio.
Indicator	8.03	N/A	N/A	Apply the process of creating a Professional Resume.
Culminating Question **Essential Questions**	**What are the procedures for creating a professional resume?** ● How is a word processor used to create a document like a resume? ● What are the proper sections and materials/information to include on a professional resume? ● How is a professional resume adapted to fit an individual?			

UNPACKED CONTENT

a. Content Literacy Terminology.
b. Understand common word processing concepts.
c. Identify proper sections and appropriate material to include on a professional resume.
d. Create a professional resume.

INSTRUCTIONAL ACTIVITIES-8.03	
A. Content Literacy Terminology	
Resource(s)	**(See 8.03.1)** *Note:* Some definitions pulled from Microsoft Doc Glossary (see reference below).
B. Understand common Word Processing concepts. **C. Identify proper sections and appropriate material to include on a professional resume.** **D. Create a professional resume.** *Note: Activity includes all Unpacked Content for Indicator.*	
Activity	Guided Worksheet and Student Creation-Resume Workshop
Teacher Instructions	● Provide digital copy of *Guided Worksheet-Resume*. Preselect career readiness video from Industry to show to students in presentation. Use *Guided Worksheet-Resume-Presentation* for whole-class or small group direct instruction. Pause and facilitate students filling in their copy of the worksheet as each section is reviewed/covered (gray slides). ● Facilitate student creation of personal resumes with information from the worksheet. Online resources can be used for word processing software as needed. Students should share drafts with someone at home to help with review/ideas.
Student Directions	● Fill in your copy of the worksheet as each section is reviewed/covered (gray slides) on Resumes. ● Create your personal resumes with information from the worksheet. Use online resources for help with word processing software as needed. ● Share a draft with someone at home to help with review/ideas.
Resource(s)	Guided Worksheet-Resume.docx Guided Worksheet-Resume- Presentatic
Website Recourse(s): Word Processing Tutorials & Terminology	
Microsoft Support. (2021). Word for Windows Training	https://support.microsoft.com/en-us/office/word-for-windows-training-7bcd85e6-2c3d-4c3c-a2a5-5ed8847eae73
Microsoft MS-DOC. (2021). Glossary: Microsoft Docs.	https://docs.microsoft.com/en-us/openspecs/office_file_formats/ms-doc/951dd5ff-6eb5-4265-b8c2-f4b7f3d745ca

Content Literacy Terminology-8.03.1	
Resume	A formal document which itemizes qualifications for a position.
Human Resources	Personnel who plan, coordinate, and direct the administrative functions of an organization such as recruiting, interviewing, and hiring/firing.
Extracurricular	Outside of regular school/education.
Summary Statement	Short paragraph or bulleted section of a resume in which an applicant briefly communicates career/education highlights and strengths.
Objective Statement	One to two sentence section of resume in which applicant quickly communicates career goals/aspirations.
Transferable Skill	Qualities of an employee or applicant that can be transferred from one job to another.
Serif	Certain fonts that have a slight projection finishing off a stroke of a letter.
Sans Serif	Certain fonts that do not have any projections finishing off a stroke of a letter.
Career Readiness Skills	Foundational skills needed to be minimally qualified for a specific occupation as determined through a job analysis or occupational profile. (Resource www.ACT.org)