Course Title:	Content Area:	Grade Level:	Credit (if applicable)
Three Dimensional Modeling	Career and Technical Education (CTE): Engineering and Technical Sciences	9-12	0.5 (45 - 75 minute classes)

Course Description:

This course provides an introduction to creating, editing, and analyzing 3D models. Develops foundational skills to work with, and navigate the digital 3D modeling workspace to create 3D objects. Examines basic elements of the 3D development of modeling, texturing, lighting, animating, and rendering. Along with building foundational skills in 3D modeling, students work toward the advanced skills of creating 3D models with moveable parts.

Aligned Core Resources:	Connection to the *BPS Vision of the Graduate*
Autodesk Fusion 360 **Autodesk Inventor** **Ultimaker Cura** **Parametric Modeling with Autodesk Fusion** Spring 2024 Edition. Shih, Randy H. Oregon Institute of Technology. SDC Publications. ©2024 ISBN-13: 978-1-63057-686-8 ISBN-10: 1-63057-686-7 SDC Publications, P.O. Box 1334, Mission, KS 66222 913-262-2664, www.SDCpublications.com	**Critical Thinking and Problem Solving** • Collect, assess and analyze relevant information • Reason effectively. Use systems thinking • Make sound judgments and decisions. Identify, define and solve authentic problems and essential questions. • Reflect critically on learning experience, processes and solutions • Transfer knowledge to other situations
Additional Course Information: ***Knowledge/Skill Dependent courses/prerequisites***	**Link to *Completed Equity Audit***
As an introductory course, no prior knowledge of 3D modeling or CAD software is required. Students do need to be experienced with the Windows operating system and have an introductory background in drafting 3D parts which can be developed with TinkerCAD and/or SketchUp.	3D Modeling Equity Curriculum Review Fall 2024

Standard Matrix

Advance CTE Standard	Unit 1 CAD Fundamentals	Unit 2 Part & Assembly Modeling	Unit 3 3D Solid Modeling
ESS01.02: Demonstrate language arts knowledge and skills required to pursue the full range of post-secondary education and career opportunities. • Comprehend key elements of written information such as cause/effect, comparisons/contrasts, conclusions, context, purpose, charts /tables/graphs, evaluation/critiques, sequence,	✔	✔	✔

summaries, and technical subject matter. ● Evaluate oral and written information for accuracy, adequacy/sufficiency, appropriateness, clarity, conclusions/solutions, fact/opinion, propaganda, relevancy, validity, and relationship of ideas.			
ESS01.03: Demonstrate mathematics knowledge and skills required to pursue the full range of post-secondary education and career Opportunities. ● Demonstrate knowledge of basic arithmetic operations such as addition, subtraction, multiplication, and division. ● Apply data and measurements to solve a problem. ● Analyze data when interpreting operational documents.	✔	✔	✔
ESS02.01: Select and employ appropriate reading and communication strategies to learn and use technical concepts and vocabulary in practice. ● Demonstrate use of content, technical concepts and vocabulary when analyzing information and following directions. ● Interpret information, data, and observations to apply information learned from reading to actual practice. ● Communicate information, data, and observations to apply information learned from reading to actual practice		✔	✔
ESS02.05 Use correct grammar, punctuation and terminology to write and edit documents.	✔	✔	✔
ESS03.01 Employ critical thinking skills independently and in teams to solve problems and make decisions (e.g., analyze, synthesize and evaluate). ● Analyze elements of a problem to develop creative solutions. ● Use structured problem-solving methods when developing proposals and solutions. ● Identify alternatives using a variety of problem-solving and critical thinking skills.	✔	✔	✔
ESS 04.10 Employ computer operations applications to manage work tasks. ● Manage computer operations. ● Manage file storage. ● Compress or alter files.	✔	✔	✔
ESS06.01 Implement personal and jobsite safety rules and regulations to maintain safe and healthful working conditions and environments. ● Assess workplace conditions for safety and health. ● Align safety issues with appropriate safety standards to ensure a safe workplace/jobsite. ● Identify safety hazards common to workplaces. ● Identify safety precautions to maintain a safe worksite. ● Select appropriate personal protective equipment as needed for a safe workplace/jobsite.		✔	✔

ESS07.03 Employ teamwork skills to achieve collective goals and use team members' talents effectively. ● Work with others to achieve objectives in a timely manner. ● Demonstrate teamwork processes that provide team building, consensus, continuous improvement, respect for the opinions of others, cooperation, adaptability, and conflict resolution. ● Take responsibility for shared group and individual work tasks. ● Assist team members in completing their work. ● Adapt effectively to changes in projects and work activities.	✔	✔	✔
MNPA01.01.04 Monitor fabrication of the product using process control data. ● Use process control data to ensure that the manufacturing process complies with standards. ● Identify possible consequences resulting from failure to perform operations safely. ● Verify that the product meets customer specifications. ● Complete, maintain, and forward to proper parties product and process documentation. ● Verify that production operations comply with all health, safety, and environmental policies and procedures.		✔	✔
MNPA01.01.05 Inspect the product to verify that it meets specifications. ● Verify the calibration of the testing equipment. ● Follow the established sampling plan and inspection policies/procedures. ● Predict consequences of failure to identify promptly any product and production process that do not meet specifications. ● Complete inspection documents accurately and forward them to proper parties. ● Following appropriate testing/production tools and procedures. ● Make adjustments needed to keep the production process within specifications.		✔	✔
ACC01.01 Perform math operations such as estimating and distributing materials and supplies to complete jobsite/workplace tasks. ● Use basic math functions to complete jobsite/workplace tasks. ● Use geometric formulas to determine areas and volumes of various structures. ● Use appropriate formulas to determine ratios, fractions, and proportion measures. ● Use appropriate formulas to determine measurements of dimensions, spaces and structures. ● Conceptualize a three-dimensional form from a two-dimensional drawing to visualize proposed work.	✔	✔	✔
ACPA06.01 Develop technical drawings drafted by hand and computer generated plans to design structures.	✔	✔	✔

• Draw and sketch by hand to communicate ideas effectively • Learn to read and produce technical drawings, understanding the significance of each line in a drawing.			
ACPA06.02 Employ appropriate representational media to communicate concepts and design. • Convey graphic information using multi-dimensional drawings. • Build models using referenced drawings and sketches. • Utilize computer technology when communicating concepts and designs.	✔	✔	✔

Unit Links

Unit Title:

Unit 1 CAD Fundamentals

In this unit, students will learn the foundational principles of creating 3D parts using CAD software. Topics covered include sketching, extrusion, revolve, fillets, chamfers, patterns, and feature-based modeling. Through guided exercises and projects, students will develop proficiency in generating precise and complex part models, applying geometric constraints, and understanding parametric modeling techniques.

Relevant Standards: Bold indicates priority

ESS01.02: Comprehend key elements of written information such as cause/effect, comparisons/contrasts, charts /tables/graphs, evaluation/critiques, sequence, summaries, and technical subject matter.

ESS01.03: Demonstrate knowledge of basic arithmetic operations such as addition, subtraction, multiplication, and division. Apply data and measurements to solve a problem.

ESS 02.05 Use correct grammar, punctuation and terminology to write and edit documents.

ESS03.01 Employ critical thinking skills independently and in teams to solve problems and make decisions (e.g., analyze, synthesize and evaluate): Use structured problem-solving methods when developing proposals and solutions.

ESS 04.10 Manage file storage and compress or alter files.

ESS07.03 Demonstrate teamwork processes that provide team building, consensus, continuous improvement, respect for the opinions of others, cooperation, adaptability, and conflict resolution. Assist team members in completing their work. Adapt effectively to changes in projects and work activities.

ACC01.01 Use appropriate formulas to determine ratios, fractions, and proportion measures.
Use appropriate formulas to determine measurements of dimensions, spaces and structures.
Conceptualize a three-dimensional form from a two-dimensional drawing to visualize proposed work.

ACPA06.01 Learn to read and produce technical drawings, understanding the significance of each line in a drawing.

ACPA06.02 Convey graphic information using multi-dimensional drawings.

Essential Question(s):	Enduring Understanding(s):
<u>Lesson 1: Intro to Fusion 360</u> ● How do I create an Autodesk education account and get a Fusion 360 license or are assigned a Fusion 360 license? ● What is the purpose and benefits of a Folder-Level Project? ● How can I invite and manage members to a Folder-Level Project Folder? ● How can I remove and/or deactivate members from a Folder-Level Project? ● What is the purpose of archiving a Project on Fusion Team? <u>Lesson 2: Sketching</u> ● What is applied to help define a profile in a sketch? ● What type of dimensions can be created with the sketch Dimension tool? ● While in a sketch, what do blue, and black lines represent?	A Fusion Team Is where you access your Fusion 360 and project data. Data is stored in the cloud. Control roles/permissions, files, versions, markups, add comments, and recover deleted files are controlled in the cloud. You can invite members to your Fusion Team and control their access and you can be invited into others' Fusion Team. You can administer more than one Fusion Team. Projects are used to manage data and control access. A project can be setup for a class or for a specific design You can invite members to the project and set how the project can be found by members called project security types. Folders are used to manage data. In a Folder-Level Project, you can set a role for each member per folder. When you set a role, it will be the default for subfolders. Each folder can have members with different roles.

● While in a sketch, how can you apply a horizontal/vertical constraints to a midpoint? ● How do you know when a profile is fully constrained? ● How do you edit a sketch? ● How can we copy geometry onto an active sketch? <u>Lesson 3: Part Modeling</u> ● How do you know when a profile is closed? ● When creating features, what are the three main operation types you use to define a component? ● Can you use the extrude command on the solid tab to extrude an open profile? ● When creating a profile that will be revolved, why would you change a line to a centerline? ● When placing threaded hole, what option do you use to create true modeled threads? ● When editing a fillet or chamfer feature, how do you add or remove edges to the selection set? ● When creating a circular pattern, what can be used as the axis of rotation? ● What construction plane method would you use to create a construction plane an equal distance from two existing faces or planes. ● What construction plane method is used to create a construction plane that touches a circular face on one location and at and specified angle? ● What is the difference between changing the Physical Material and the Appearance of a component? ● How do you find the area and mass information about a component? ● When using the measure command, what information can you obtain? ● After creating a Section Analysis, how do you control its visibility? <u>Lesson 4: Assembly Modeling</u> ● How do you know if you are working on a component or an assembly? ● How do you turn a component file into an assembly file which only has local components? ● What are the differences and benefits of bottom-up and top-down assemblies? ● How many assemblies can a component be inserted (referenced) into? ● After converting a body into a component, what happens to the features that were used to create the body? ● When would you apply an As-Built Joint? ● When applying an assembly joint, how do you filter for points on a specific face? ● When would you create a Rigid Group? ● Are Contact Sets or Joint Limits the preferred	A component can be created in its own file or within the context of an assembly design file. Components can be placed together to create an assembly or subassembly. When modeling a component, the 2D outline will be used to create the base feature (first feature). It is the feature other features will add material to or remove material from. Extruded or revolved the profile which will be known as the base feature. Continue modeling by creating additional features. Features can add or remove material. A sketch is used to locate where profile geometry will be placed. A sketch can consist of 2D or 3D geometry, constraints and dimensions, and is usually turned into 3D geometry. Dimensions are added to sketch geometry to define the size and location of the geometry. The geometry changes size to match the value of the dimension. This is referred to as parametric dimensions. Sketch constraints help define the shape of the profile and create relationships. To add sketch constraints, a sketch must be active. Features are used to define the shape of a design. The first feature created for a component is referred to as a base feature. If the feature is created after another feature was created, the features can add and remove material. When using tools like; extrude, revolve, sweep, loft you have five operation options: join, cut, intersect, body, and new component. The Modeling Hierarchy includes (from basic to complex): A body which is a continuous 3D object; it is a solid or surface that is used to model a component. Bodies are stationary. Then there is a component which is an envelope that contains bodies, sketches, a coordinate system, & reference geometry. A component can contain multiple features and must have at least one body. Components can move. Finally there is an assembly that is a grouping of 1 or more components and has its own coordinate system. Assemblies allow the assignment of position and motion of components using joints. Assembly creation methods can be either top-down where single design file for assembly and its components create components inside a single design file or bottom-up where separate design files for components and assemblies Insert component files into assembly files. Components are required to apply assembly joints. When a component is created from a body, and when the component is active, the features that were used to create the body are not shown in the Timeline. The features remain at the top-level assembly.

method for modeling? • Why would you set a joint's limits? <u>Lesson 5: Drawings</u> • What command do you use to create orthographic views that are projected relative to a parent view? • When creating a drawing of a component, when do you first define the sheet size? • When placing a drawing view, what are the four Appearance and Edge Styles? • When creating a projected drawing view, how many orientations can you create? • When would you create a Detail View? • When a parent view moves, what happens to the child orthographic views? • When you change the scale of a Base View, what happens to the child views? • When annotating a drawing view, how do you add centerlines and center marks? • When annotating a drawing, how do you add text to a dimension? • When annotating a drawing with the general dimension tool and you select the outside circle of a counterbore hole, what information will be displayed? • When adding a Parts List to a drawing, where do you change a components Part Number, Part Name and Description? • When exporting a drawing, what file formats can be created?	When designing, you may have multiple components that should be locked together in their current location. Instead of applying multiple Rigid - As-Built Joints, you can create one Rigid Group. Components in a Rigid Group are treated as a single object when you move them in the canvas or apply joints to the Rigid group. To prevent the Rigid Group from moving in the parent assembly, include the parent assembly in the Rigid Group. When designing an assembly that has motion, you can simulate real-world conditions by using Contact Sets and Joint Limits. Contact Sets apply to components so they only move or stop when they touch each other. Contact sets use physical contact between components to limit movement. This method requires more computer computation than Joint Limits. Joint Limits are values that define the range of motion allowed for components to move or rotate. Use joint limits to evaluate a design by limiting motion to a minimum and maximum values. Joint Limits are preferred over contact set because they require less computer computation. To document a design (component, assembly or animation) you can create a new drawing file that will document the design for manufacturing or customer use. Fusion 360 generates a 2D projection of the component(s) you select. The first drawing view is referred to as a base view. Once you place the base view in the drawing, you can generate orthogonal and isometric projected views from it. Fusion 360 calculates the most appropriate scale for the base view at the time you create the drawing.
Demonstration of Learning:	**Pacing for Unit**
Completion of lesson activities and exercises in Fusion 360 (models showing work in timeline).	18 - 75 minute classes
Family Overview (link below)	**Integration of Technology:**
Three Dimensional Modeling Family Overview (2024)	Use of the Autodesk Fusion 360 cloud-based program on desktop or laptop computers. Models may be created through additive manufacturing techniques (3D Printing) or subtractive manufacturing techniques (laser engraving, CNC machining).
Opportunities for Interdisciplinary Connections:	**Aligned Unit Materials, Resources, and Technology (beyond core resources):**
This unit teaches the foundational knowledge and skills required to create 2D and 3D models in Fusion 360. *Mechanical Engineering*: Fusion 360 is primarily used in mechanical engineering for designing parts, assemblies, and creating technical drawings. Students can explore concepts of material properties, forces, and stress analysis through	• https://www.autodesk.com/education/support • Students create an Autodesk education account and acquire a Fusion 360 license • Educator assigns students' a Fusion 360 license • Getting Started with Fusion Team for Education • Fusion 360 Educators Forum • https://www.autodesk.com/learn/ • Autodesk Learning Catalog

simulations and design testing. *Architecture*: Fusion 360 can help architecture students model building structures and interior designs. *Art and Design*: Fusion 360 can be used in the creation of sculptures, installations, and interactive art.	
Unit-specific Vocabulary:	**Anticipated misconceptions:**
Lesson 1 Fusion Team (file cabinet), Project (drawer), Folders (organize data); Canvas, Fusion Team Roles: Team Administrator, Team Member, Project Contributor; Folder Member Roles: Viewer, Reader, Editor, Manager, Administrator; Sketch; Dimensions, Constrained geometry, X, y, z axis orientation Lesson 2 Component, Assembly, Extrude, Revolve, Plane, Planar face, Base feature, Horizontal/Vertical, Coincident, Tangent, Equal, Parallel, Perpendicular, Fix/UnFix, Midpoint, Concentric, Collinear, Symmetry, Curvature, Timeline, Browser, Visibility, Driving dimension, Driven dimension, Project geometry Lesson 3 Extrude, Thin-extrude, Profile, Symmetric, Half-length, Whole-length, Taper angle, Join, Cut, Intersect, New Body, New Component, Revolve, Centerline, Hole, Counterbore, Countersink, Threads, Fillet, Chamfer, Pattern features, Mirror features, Shell feature, Construction geometry, Tangent, Perpendicular, Torus, Inspection, Section analysis Lesson 4 Assemblies, Subassemblies, Components, Bodies, Joints, As-built joints, Assembly Joints, Rigid, Revolute, Slider, Cylindrical, Pin-Slot, Planar, Ball, Faces, Parent Assembly, Rigid groups, Contact sets, Joint limits Lesson 5 Drawing Views, Base, Projected, Section, Detail, Annotations, ASME, ISO, Sheet, Title Block, Center Marks, Centerlines, Slice, Parent View, Text Notes, Hole Notes, Thread Notes, Parts List, Balloons, Export, PDF, DWG, DXF, CSV	Misconception 1: Failure is a sign of incompetence Failure is a natural and necessary part of learning and innovation. Every CAD project involves trial and error, experimentation and feedback, and iteration and improvement. Failure is not something to be ashamed of, but to be embraced as an opportunity to grow and refine your ideas. Misconception 2: Failure is a waste of time and resources Failure is not a waste, but an investment in your future success. Failure helps you to discover new possibilities, identify problems and solutions, and gain valuable insights and knowledge. Failure also helps you to develop resilience, adaptability, and creativity, which are essential skills for any CAD professional. Misconception 3: Failure is final and irreversible. Failure is not final, but temporary and reversible. CAD software allows you to save, edit, undo, and redo your work at any stage of the design process. You can always recover from a failure, learn from your mistakes, and make changes or adjustments to your model. Failure is not the end, but the beginning of a new challenge. Misconception 4: Failure is a solo experience Failure is not a solo experience, but a shared one. You are not the only one who fails in CAD, and you can benefit from the support and feedback of others who have faced similar challenges. You can also share your failures with others, and learn from their experiences and perspectives. Failure is a source of collaboration and inspiration.
Connections to Prior Units:	**Connections to Future Units:**
This is the first unit in this course.	This unit provides the foundational knowledge and skills for the applications of solid modeling in the remaining units in this course.
Differentiation through *Universal Design for Learning*	
UDL Indicator and Teacher Actions	

Expression and Communication: Build fluencies with graduated levels of support for practice and performance
- Provide Computer-Aided-Design (CAD), notation (writing) software, or mathematical notation software support documentation
- Use web applications (e.g., animations, presentations)
- Provide differentiated models to emulate (i.e. models use differing approaches, strategies, skills, etc.)
- Provide scaffolds that can be gradually released with increasing independence and skills
- Provide differentiated feedback (e.g., feedback customized to individual learners)
- Provide multiple examples of novel solutions to authentic problems

Supporting Multilingual/English Learners	
Related *CELP standards:*	**Learning Targets:**

An EL can participate in grade appropriate oral and written exchanges of information, ideas, and analyses, responding to peer, audience, or reader comments and questions.
I can create a manufacturing file using Fusion360.

Level 1: With prompting and support, use a very limited set of strategies to:
- Follow basic instructions to create a manufacturing file using Fusion360
- Use simple vocabulary to describe the steps involved in creating a manufacturing file
- Respond to yes/no questions and some basic questions about creating a manufacturing file

Level 2: With prompting and support, use an emerging set of strategies to:
- Follow instructions to create a manufacturing file using Fusion360
- Participate in short exchanges about creating manufacturing files using academic and domain-specific vocabulary
- Respond to simple questions about the process of creating a manufacturing file

Level 3: With guidance and support, use a developing set of strategies to:
- Discuss and ask questions about creating manufacturing files using Fusion360
- Use academic and domain-specific vocabulary to describe the steps involved in creating a manufacturing file
- Add relevant information and evidence to discussions about creating manufacturing files
- Restate key ideas about creating manufacturing files expressed by others

Level 4: Use an increasing range of strategies to:
- Engage in discussions and written exchanges about creating manufacturing files on various topics and issues using academic and domain-specific vocabulary
- Build on the ideas of others regarding creating manufacturing files
- Clearly express own ideas about creating manufacturing files with specific evidence
- Ask and answer questions to clarify steps and concepts related to creating manufacturing files
- Summarize key points discussed about creating manufacturing files

Level 5: Use a wide range of strategies to:
- Participate in extended discussions and written exchanges about creating manufacturing files on substantive topics and issues using academic and domain-specific vocabulary
- Build on the ideas of others to deepen discussions about creating manufacturing files
- Clearly and persuasively express own ideas about creating manufacturing files with specific and relevant evidence
- Refer to specific evidence from texts or research to support ideas about creating manufacturing files
- Ask and answer questions that probe reasoning and claims related to creating manufacturing files
- Summarize key points and evidence discussed about creating manufacturing files

General Teacher Resources for Fusion 360

Learn about the Fusion Interface Video
Setting Preferences in Fusion
Display Settings

Use the Marking Menu (most common commands - different for each workspace)
Use the Toolbox
Using Fusion Team
Difference Between TinkerCAD and Fusion 360
Product Documentation
CAD Projects Fusion Library and Learning Catalog
New to Fusion Videos (Autodesk)

Lesson Sequence	Learning Target/Success Criteria	Task/ Assessment	Resources
1 Intro to Fusion 360	I can describe Fusion 360 as a cloud-enabled platform for design, understand the structure of a Fusion Team, access and manage project data, set up folder-level projects, assign roles and permissions, deactivate users, archive projects, and identify resources for support. Students will be able to: ● Describe Fusion 360 as a Cloud Enabled Platform for Design ● Understand the structure of a Fusion Team ● Access their data ● Set up a Folder-Level Project ● Add folders with roles ● Remove and deactivate students ● Archive a project ● Know where to get help	Getting Started Step 1: Create an Autodesk Account & Acquire a Fusion 360 License Step 2: In Fusion 360, Create a Folder-Level Project Step 3: In Fusion 360, Create Folders Step 4: Invite New Members to a Folder Step 5: Set Members' Role for Folders Step 6: Change Members' Role for the Fusion Team Archive a Project: Step 1: Remove Students from a Folder-Level Project Step 2: Deactivate Members from Your Fusion Team Step 3: Archive a Project on Fusion Team	Fusion 360 Program 01 Introduction to Fusion 360, Fusion Team and Folder-Level Projects Presentation Fusion 360 Fundamentals Course - Import Script Project Data Importer Files
2 Sketching	I can understand the modeling process in Fusion 360, create and edit sketches, navigate the sketching plane using the mouse, add dimensions and constraints to sketches, sketch on the face of a part, and project geometry from a 2D sketch. Students will be able to: ● Understand the modeling process ● Create sketches ● Utilize the Fusion 360 Interface ● Utilize the mouse to navigate on the sketching plane ● Add dimensions to a sketch ● Add constraints to a sketch ● Edit sketches ● Sketch on the face of a part ● Project geometry from a 2D	Setup a project, axis, and units. Exercise 1: Sketching (no dimensions) Exercise 2: Sketching (with dimensions) Exercise 3: Sketching on a Part's Face Exercise 4: Dimensioning Exercise 5: Sketch Constraints Exercise 6: Edit Sketch Exercise 7: Project Geometry	Fusion 360 Program 02 Fusion 360 Fundamentals Course - Sketching Presentation

	sketch		
3 Part Modeling	I can create and modify features in Fusion 360, including extrude, revolve, hole, fillet, chamfer, pattern, mirror, and shell features, use construction geometry, change materials, and inspect my design. Students will be able to: ● Create extrude features ● Create revolve features ● Create hole features ● Create fillet and chamfer features ● Pattern and mirror features ● Create a shell feature ● Create construction geometry ● Change material and inspect the design	Exercise 1: Extrude Features Exercise 2: Thin Extrude Feature Exercise 3: Revolve Feature Exercise 4: Hole Features Exercise 5: Fillets and Chamfer Features Exercise 6: Rectangular and Circular Patterns Exercise 7: Mirror Exercise 8: Shell Feature Exercise 9: Construction Geometry Exercise 10: Inspect – Change Material, Properties, Measure, Section Analysis	Fusion 360 Program 03 Fusion 360 Fundamentals Course - Part Modeling
4 Assembly Modeling	I can manage assemblies in Fusion 360, create components from bodies, use the move/copy, align, and assembly joint tools, create rigid groups, and apply contact sets and joint limits. Students will be able to: ● Manage assemblies ● Create components from bodies ● Utilize the move/copy, align, and assembly joint tools ● Create rigid groups ● Utilize contact sets and joint limits	Exercise 1: Managing Assemblies Exercise 2: Create Components From Bodies Exercise 3: Move Align Joints Exercise 4: Rigid Groups Exercise 5: Contact Sets and Joint Limits	Fusion 360 Program 04 Fusion 360 Fundamentals Course - Assembly Modeling Presentation
5 Drawings	I can create and edit drawing views (base, projected, section, detail), adjust their scale, appearance, and edge visibility, control section and detail view areas, add annotations, centerlines, center marks, dimensions, notes (text, hole, and thread), parts lists, and balloons, and manage sheets and title blocks. I can also export a drawing and edit document settings, including text in the title block. Students will be able to: ● Create drawing views; base, projected, section, detail ● Edit drawing views ○ Move drawing views. ○ Delete drawing views. ○ Change the scale, appearance, and edge visibility of drawing views. ○ Control what is displayed in a section view. ○ Change the area displayed in	Exercise 1: Create Drawing Views Exercise 2: Edit Drawing Views Exercise 3: Annotating a Drawing Exercise 4: Sheets, Borders and Title Blocks Exercise 5: Export a Drawing	Fusion 360 Program 05 Fusion 360 Fundamentals Course - Drawings Presentation

	a detail view ● Add annotations ○ Centerlines and Center Marks ○ Dimensions ○ Text, Hole, and Thread Notes ○ Parts List and Balloons ● Edit a sheet and title block ○ Change the sheet size. ○ Add, rename, and delete a sheet. ○ Edit a document's settings. ○ Edit text in a title block. ○ ● Export a drawing		

Unit Title:

Unit 2: Part & Assembly Modeling

In this unit, students will explore the assembly modeling process, focusing on the integration and interaction of multiple parts to create complex assemblies. Topics covered include component hierarchy, constraints, mates, fasteners, interference detection, exploded views, and bill of materials (BOM). Through hands-on projects, students will develop skills in assembling, simulating motion, and documenting assemblies for manufacturing and visualization purposes. Students will further develop their skills with technical drawing and annotation within the CAD environment.

Relevant Standards: Bold indicates priority

ESS01.02: Comprehend key elements of written information such as cause/effect, comparisons/contrasts, charts /tables/graphs, evaluation/critiques, sequence, summaries, and technical subject matter. Evaluate written information for accuracy, adequacy/sufficiency, clarity, solutions, and relationship of ideas.

ESS01.03: Apply data and measurements to solve a problem. Analyze data when interpreting operational documents.

ESS02.01: Demonstrate use of content, technical concepts and vocabulary when analyzing information and following directions. Interpret and communicate information, data, and observations to apply information learned from reading to actual practice.

ESS 02.05 Use correct grammar, punctuation and terminology to write and edit documents.

ESS03.01 Analyze elements of a problem to develop creative solutions. Use structured problem-solving methods when developing proposals and solutions. Identify a variety of problem-solving and critical thinking skills.

ESS 04.10 Manage computer operations, manage file storage, and compress or alter files.

ESS06.01 Identify safety hazards common to workplaces. Identify safety precautions to maintain a safe worksite. Select appropriate personal protective equipment as needed for a safe workplace/jobsite.

ESS 07.03 Work with others to achieve objectives in a timely manner. Demonstrate teamwork processes that provide consensus, continuous improvement, respect for the opinions of others, cooperation, adaptability, and conflict resolution. Take responsibility for shared group and individual work tasks. Assist team members in completing their work. Adapt effectively to changes in projects and work activities.

MNPA01.01.04 Use process control data to ensure that the manufacturing process complies with standards. Identify possible consequences resulting from failure to perform operations safely. Verify that the product meets customer specifications.

MNPA01.01.05 Inspect the product to verify that it meets specifications. Verify the calibration of the testing equipment. Follow the established sampling plan and inspection policies/procedures. Predict consequences of failure to identify promptly any product and production process that do not meet specifications. Make adjustments needed to keep the production process within specifications.

ACC01.01 Use geometric formulas to determine areas and volumes of various structures. Use appropriate formulas to determine ratios, fractions, and proportion measures. Use appropriate formulas to determine measurements of dimensions, spaces and structures. Conceptualize a three-dimensional form from a two-dimensional drawing to visualize proposed work.

ACPA06.01 Learn to read and produce technical drawings, understanding the significance of each line in a drawing.

ACPA06.02 Convey graphic information using multi-dimensional drawings. Build models using referenced drawings and sketches. Utilize computer technology when communicating concepts and designs.

Essential Question(s):

- How do I use CAD assembly tools to combine multiple part files into a common assembly file?
- How do you make part updates within an assembly?
- How do I apply constraints to parts within an assembly?
- How are assembly files used in industry?
- How do presentation files help show a working assembly (optional)?
- What are the components of a technical drawing?
- What are the dimensioning/annotation standards for technical drawings?
- How do you create a technical drawing using CAD software?
- How are drawing sheets used in industry?

Enduring Understanding(s):

- **Part modeling:** Before assembling components, students should be proficient in creating individual parts using CAD modeling tools. This includes creating sketches, extruding, revolving, sweeping, and applying features to generate complex shapes.
- **Component hierarchy:** Students need to understand the concept of component hierarchy within an assembly. This includes defining relationships between parts, such as mates, constraints, and alignments, to accurately position and connect components.
- **Constraints and mates:** Learning how to apply constraints and mates is essential for assembling components in CAD. Constraints define how parts relate to each other geometrically, while mates specify how parts are positioned and oriented relative to one another.
- **Assembly structure:** Students should grasp the structure of an assembly, including the organization of components within a hierarchical tree or list. This involves understanding how to create subassemblies, insert parts into assemblies, and manage assembly components efficiently.
- **Interference detection:** Understanding how to detect and resolve interference between components is crucial for ensuring that the assembled product functions correctly. CAD software often provides tools for detecting and visualizing interferences, allowing students to identify and address potential issues.
- **Assembly motion and animation:** Students may need to learn how to simulate assembly motion and create animations to visualize how components interact and move within the assembly. This involves defining motion constraints and creating motion paths to animate the assembly.
- **Fasteners and joints:** Students should understand how to incorporate fasteners, such as screws, bolts, nuts, and joints, such as hinges and bearings, into assemblies. This includes selecting appropriate standard components from libraries or modeling custom fasteners as needed.
- **Exploded views:** Learning how to create exploded views helps students communicate the assembly process visually by showing how components fit together and how they are assembled or disassembled step by step.
- **Bill of Materials (BOM):** Understanding how to generate a bill of materials from an assembly is essential for documenting the components required to build the product. CAD software typically provides tools for automatically generating BOMs based on the components in the assembly.
- **Collaboration and sharing:** Students should learn how to share and collaborate on assemblies with others, including methods for exchanging CAD files, managing revisions, and incorporating feedback from team members or stakeholders.
- **Drawing setup**: Understanding how to set up drawing templates, including title blocks, units, scales, and layers, is essential for creating standardized technical drawings.
- **Geometry creation**: Proficiency in creating and modifying basic geometric shapes such as lines, circles, arcs, polygons, and ellipses is fundamental to creating technical drawings.
- **Dimensioning**: Students need to learn how to add dimensions accurately to their drawings, including linear dimensions, angular dimensions, radial dimensions, and ordinate dimensions.
- **Text and annotations:** Students should understand how to add text, labels, symbols, and other annotations to convey important information on their technical drawings.
- **Orthographic projection:** Understanding orthographic projection principles, including creating and aligning multiple views (e.g., front, top, side views), is crucial for accurately representing three-dimensional objects in two dimensions.
- **Section views and detail views**: Students should learn how to create section views to show internal features of objects and detail views to magnify specific areas of interest within a drawing.
- **Symbols and standards:** Familiarity with industry-standard symbols, abbreviations, and drawing conventions (such as ANSI, ISO, or ASME standards) is essential for creating professional-quality technical drawings.
- **Plotting and printing:** Knowing how to set up plot configurations, scale drawings for printing, and create PDF or physical prints is necessary for sharing technical drawings with others.
- **File management:** Understanding how to organize and manage CAD files, including naming conventions, file

formats, and version control, helps students maintain an efficient workflow and collaborate effectively.
- **Practice and application:** Regular practice and application of CAD skills through exercises, projects, and real-world applications are essential for students to develop proficiency in creating technical drawings using CAD software.

Demonstration of Learning:	**Pacing for Unit**
Completion of lesson activities and exercises in Fusion 360 Design.	14 class periods (75 minute classes)
Family Overview (link below)	**Integration of Technology:**
Three Dimensional Modeling Family Overview (2024)	Use of the Autodesk Fusion 360 cloud-based program on desktop or laptop computers. Models may be created through additive manufacturing techniques (3D Printing) or subtractive manufacturing techniques (laser engraving, CNC machining).
Opportunities for Interdisciplinary Connections:	**Aligned Unit Materials, Resources, and Technology (beyond core resources):**
Mechanical Engineering: Fusion 360 is primarily used in mechanical engineering for designing parts, assemblies, and creating technical drawings. Students can explore concepts of material properties, forces, and stress analysis through simulations and design testing. *Architecture*: Fusion 360 can help architecture students model building structures and interior designs. *Art and Design*: Fusion 360 can be used in the creation of sculptures, installations, and interactive art.	Autodesk Fusion in 90 Minutes Course Files 3D Content Central – Large variety of free CAD models
Unit-specific Vocabulary:	**Anticipated misconceptions:**
Mates, constraints, assembly tools, presentation, exploded view, balloon notes, technical drawing, drawing sheets, dimensions, annotation, multiview, isometric, orthographic, sectional, auxiliary, extension lines, leader lines, hidden lines, center lines, object lines, construction lines, cutting plane, hidden features	• Under constraining or over constraining assemblies • Use design intent and parametric modeling to prevent or reduce CAD assembly errors • BOM Mistakes Every CAD Designer Makes • Starting a model with the primary planes on a corner instead of center. • Not setting up a good file naming system. • Not setting up Toolbox, templates, libraries, favorites. • Becoming too reliant on configurations. • Becoming too reliant on multi-body techniques. • Making sketch fillets/chamfers instead of feature fillets. • Using sketch patterns and not Mirroring sketch entities. • Deleting things and recreating instead of editing. • Nott assigning a CAD Administrator. • Failing to keep sketches simple. • Not having a real plan for model dimensions. • Dimensioning everything in a sketch instead of minimizing dimensions by using construction lines and relations.
Connections to Prior Units:	**Connections to Future Units:**
This unit utilizes and expands upon on	Students will apply the information for modeling parts and

knowledge and skills learned in Unit 1: Intro to Fusion 360	assemblies in the next unit for learning how CAD-CAM integration shapes modern manufacturing practices and can impact future design and engineering practices.

Differentiation through *Universal Design for Learning*

UDL Indicator & Teacher Actions:

Expression and Communication: Build fluencies with graduated levels of support for practice and performance
- Provide Computer-Aided-Design (CAD), notation (writing) software, or mathematical notation software support documentation
- Use web applications (e.g., animations, presentations)
- Provide differentiated models to emulate (i.e. models use differing approaches, strategies, skills, etc.)
- Provide scaffolds that can be gradually released with increasing independence and skills
- Provide differentiated feedback (e.g., feedback customized to individual learners)
- Provide multiple examples of novel solutions to authentic problems

Supporting Multilingual/English Learners

Related *CELP standards* and Learning Targets:

An EL can participate in grade appropriate oral and written exchanges of information, ideas, and analyses, responding to peer, audience, or reader comments and questions. **I can create a manufacturing file using Fusion360.**

Level 1: With prompting and support, use a very limited set of strategies to:
- Follow basic instructions to create a manufacturing file using Fusion360
- Use simple vocabulary to describe the steps involved in creating a manufacturing file
- Respond to yes/no questions and some basic questions about creating a manufacturing file

Level 2: With prompting and support, use an emerging set of strategies to:
- Follow instructions to create a manufacturing file using Fusion360
- Participate in short exchanges about creating manufacturing files using academic and domain-specific vocabulary
- Respond to simple questions about the process of creating a manufacturing file

Level 3: With guidance and support, use a developing set of strategies to:
- Discuss and ask questions about creating manufacturing files using Fusion360
- Use academic and domain-specific vocabulary to describe the steps involved in creating a manufacturing file
- Add relevant information and evidence to discussions about creating manufacturing files
- Restate key ideas about creating manufacturing files expressed by others

Level 4: Use an increasing range of strategies to:
- Engage in discussions and written exchanges about creating manufacturing files on various topics and issues using academic and domain-specific vocabulary
- Build on the ideas of others regarding creating manufacturing files
- Clearly express own ideas about creating manufacturing files with specific evidence
- Ask and answer questions to clarify steps and concepts related to creating manufacturing files
- Summarize key points discussed about creating manufacturing files

Level 5: Use a wide range of strategies to:
- Participate in extended discussions and written exchanges about creating manufacturing files on substantive topics and issues using academic and domain-specific vocabulary
- Build on the ideas of others to deepen discussions about creating manufacturing files
- Clearly and persuasively express own ideas about creating manufacturing files with specific and relevant evidence
- Refer to specific evidence from texts or research to support ideas about creating manufacturing files
- Ask and answer questions that probe reasoning and claims related to creating manufacturing files

Lesson	Learning Target/Success	Task/Assessment	Resources

Sequence	Criteria		
1	I can download dataset files and open them in Fusion 360, describe the different sections of the user interface, access projects and designs through the Data Panel, create new projects, upload and open files, change views using zoom, pan, and orbit functions, and make edits to features or components using the Browser and Timeline. Students will be able to: • Download dataset files and open in Fusion 360. • Describe the different sections of the user interface. • Access projects and designs using the Data Panel. • Create a new project. • Upload a file and open it. • Change the view for an object and use the zoom, pan, and orbit functions. • Make edits to a feature or component and use the Browser and Timeline.	• Demonstrate navigation of the Fusion 360 user interface. • Distinguish between the design, render, animation, simulation, and manufacture workspaces. • Upload the Tumbler file into the student's Fusion account (2 separate assembly files in the design). • Demonstrate use of different views with Tumbler files. • Access edit features of the Tumbler by clicking on the timeline.	• Lecture Slides • Discover the UI Step by Step Slides • Discover the User Interface Step by Step Guide • Use Navigation & Viewing Tools Step by Step Slides • Use Navigation & Viewing Tools Step by Step Guide • Dataset: Autodesk_Tumbler.f3z
2	I can create a 2-dimensional sketch, apply dimensions and constraints, add geometric constraints, and ensure sketches are fully constrained in Fusion 360. Students will be able to: • Create a 2-dimensional sketch. • Apply dimensions and constraints. • Add geometric constraints to a sketch. • Create fully constrained sketches.	• Create the design file and save as MyTumbler. • Utilize imperial units for the project. • Create a sketch in the xy plane. • Draw lines with exact dimensions. • Confirm if the Tumbler sketch is constrained using line colors AND the lock symbol. • Create a new design file named Tumbler_Coaster.	• Lecture Slides • Create a Fully Constrained Sketch Step by Step Slides • Create a Fully Constrained Sketch Step by Step Guide • Practice Solution: Create a Sketch Step by Step Slides • Practice Exercise: Create a Sketch Step by Step Guide • Dataset::Tumbler Coaster
3	I can create 3D bodies from 2D sketches, extrude and revolve profiles, hollow solid bodies using the Shell command, and create round and smooth edges with the	• Make an extrusion on the Tumbler and remove it. • Hollow out the inside of the Tumbler. • Smooth the edges of the Tumbler.	• Lecture Slides • Create Features from 2D Sketches Step by Step Slides • Create Features from 2D Sketches Step by Step

	Fillet and Chamfer tools in Fusion 360. Students will be able to: ● Create 3D bodies from 2D sketches. ● Extrude and revolve a profile. ● Hollow a solid body by using the Shell command. ● Create round and smooth edges using the Fillet and Chamfer tools.		Guide ● Practice Solution: Extrude a Sketch Profile Step by Step Slides ● Practice Exercise: Extrude a Sketch Profile Step by Step Guide ● Students use their saved Tumbler file and Tumbler Coaster file from prior lesson.
4	I can start a new component in a top-down assembly, use existing geometry to create new components, consider tolerance for different manufacturing processes, add threads to components, toggle between components, apply the correct joints to model motion, and evaluate the differences between various types of joints in Fusion 360. Students will be able to: ● Start a new component in a top-down assembly. ● Use existing geometry to create new components. ● Consider tolerance when designing for different manufacturing processes. ● Add threads to a component. ● Toggle between components. ● Apply the correct joints to model motion. ● Evaluate the difference between many types of joints.	● Distinguish between bodies and components. ● Define top-down and bottom-up assemblies. ● Make the top level of the Tumbler assembly active. ● Create a new component named Gasket in an assembly file. ● Create an extrusion on the top of the Tumbler so the Gasket has a feature in its timeline. ● Create a sketch for the Tumbler Lid using the shell and hole tools. ● Distinguish between different joint types. ● Apply the correct joints to control the motion. ● Apply rigid joint types, rigid groups, and as-built joints. ● Create a simple/basic Table for the Tumbler to sit on using as-built joints.	● Lecture Slides ● Create new components in an assembly Step by Step Slides ● Create new components in an assembly Step by Step Guide ● Use existing geometry to create a design Step by Step Slides ● Use existing geometry to create a design Step by Step Guide ● Position components using jointsStep by Step Slides ● Position components using joints Step by Step Guide ● Practice Solution: Create New Components Step by Step Slides Video ● Practice Exercise: Create Components and As Built Joints Step by Step Guide ● Table Drawings and Datasets ● Students use their saved Tumbler file from prior lesson.
5	I can create a sweep, use the Measure tool to determine the sizes of previously modeled parts, insert a separate design file into an assembly using the bottom-up approach, create the base view in a new detailed drawing, apply appropriate dimensioning techniques, and insert notes into a detailed drawing in Fusion 360. Students will be able to: ● Create a sweep. ● Use the Measure tool to determine sizes of previously	● Create a customized straw for your tumbler cup that should fit within the opening of the lid and reach the bottom of the tumbler. ● Add the tumbler cup to a table and apply a planar joint to allow a sliding motion on the table top only. ● Create dimensioned drawings to be used in manufacturing processes. ● Use basic fundamentals of dimensioning to create professional-looking	● Lecture Slides ● Create a Sweep Feature Step by Step Slides ● Create a Sweep Feature Step by Step Guide ● Practice Exercise: Create a Bottom Up Assembly Step by Step Guide ● Practice Solution Create a Bottom Up Assembly Video ● Create Dimensional Drawings Step by Step Slides ● Create Dimensional Drawings Step by Step

	modeled parts. ● Insert a separate design file into an assembly using the bottom-up approach. ● Create the base view in a new detailed drawing. ● Apply appropriate dimensioning techniques. ● Insert notes to a detailed drawing.	drawings for the Tumbler, Coaster, Straw, and Table.	Guide ● Practice Exercise: Create a Drawing with Multiple Views and Dimensions Step by Step Guide ● Practice Exercise: Create a Drawing with Multiple Views and Dimensions Step by Step Slides Video ● Use saved Tumbler, Tumbler Coaster, Straw, and Table files from prior lessons.
6	I can apply visual settings, modify Render workspace settings, submit a render to the cloud, share a link to the design, and prepare and export files for 3D printing in Fusion 360. Students will be able to: ● Apply visual settings. ● Modify Render workspace settings. ● Submit a render to the cloud. ● Share a link to the design. ● Prepare and export files for 3D printing.	● Change the model appearance to create a realistic rendering of the objects and their environment that include the Tumbler, coaster, Lid, Straw, and Table. ● Share a link to the design. ● Prepare and export files for 3D printing.	● Lecture Slides ● Render a Project Environment Step by Step Slides ● Render a Project Environment Step by Step Guide ● Practice Exercise: Create a Render Step by Step Guide ● Practice Exercise: Create a Render Step by Step Slides ● Export Share Print Step by Step Slides ● Export Share Print Step by Step Guide ● Practice Exercise: Prepare a Design for 3D Printing Step by Step Guide ● Practice Exercise: Prepare a Design for 3D Printing Step by Step Slides ● Use saved Tumbler, Tumbler Coaster, Straw, and Table files from prior lessons.

Unit Title:

Unit 3: 3D Solid Modeling

In this unit, students will utilize the different design setups to virtually examine the manufacturing of their parts, to help make corrections and informed decisions on the manufacturing process being utilized. Students will learn how to implement the steps of the design process for transforming ideas into a production part. For existing parts which may need to be created or modified, the use of reverse engineering will give students the opportunity to recreate or improve on existing designs. Students will use additive or subtractive manufacturing processes to create a part or parts needed for an assembly.

Relevant Standards: Bold indicates priority

ESS01.02: Comprehend key elements of written information such as cause/effect, comparisons/contrasts, charts /tables/graphs, evaluation/critiques, sequence, summaries, and technical subject matter. Evaluate written information for accuracy, adequacy/sufficiency, clarity, solutions, and relationship of ideas.

ESS01.03: Apply data and measurements to solve a problem. Analyze data when interpreting operational documents.

ESS02.01: Demonstrate use of content, technical concepts and vocabulary when analyzing information and following directions. Interpret and communicate information, data, and observations to apply information learned from reading to actual practice.

ESS 02.05 Use correct grammar, punctuation and terminology to write and edit documents.

ESS03.01 Analyze elements of a problem to develop creative solutions. Use structured problem-solving methods when developing proposals and solutions. Identify alternatives using a variety of problem-solving and critical thinking skills.

ESS 04.10 Manage computer operations, manage file storage, and compress or alter files.

ESS06.01 Identify safety hazards common to workplaces. Identify safety precautions to maintain a safe worksite. Select appropriate personal protective equipment as needed for a safe workplace/jobsite.

ESS 07.03 Work with others to achieve objectives in a timely manner. Demonstrate teamwork processes that provide consensus, continuous improvement, respect for the opinions of others, cooperation, adaptability, and conflict resolution. Take responsibility for shared group and individual work tasks. Assist team members in completing their work. Adapt effectively to changes in projects and work activities.

MNPA01.01.04 Use process control data to ensure that the manufacturing process complies with standards. Identify possible consequences resulting from failure to perform operations safely. Verify that the product meets customer specifications.

MNPA01.01.05 Inspect the product to verify that it meets specifications. Verify the calibration of the testing equipment. Follow the established sampling plan and inspection policies/procedures. Predict consequences of failure to identify promptly any product and production process that do not meet specifications. Make adjustments needed to keep the production process within specifications.

ACC01.01 Use geometric formulas to determine areas and volumes of various structures. Use appropriate formulas to determine ratios, fractions, and proportion measures. Use appropriate formulas to determine measurements of dimensions, spaces and structures. Conceptualize a three-dimensional form from a two-dimensional drawing to visualize proposed work.

ACPA06.01 Learn to read and produce technical drawings, understanding the significance of each line in a drawing.

ACPA06.02 Convey graphic information using multi-dimensional drawings. Build models using referenced drawings and sketches. Utilize computer technology when communicating concepts and designs.

Essential Question(s):

- What are the core differences between CAD and CAM, and how do they complement each other in the design and manufacturing process?
- How do you create and manipulate a design using Fusion360, from initial sketch to finalized part?
- What steps are involved in creating a manufacturing file using Fusion360, and how do you ensure the proper setup and settings for the chosen manufacturing process
- How does the design process facilitate the transformation of an idea into a solution, and what are the key steps involved?
- What are the differences between additive and subtractive manufacturing, and how do they influence the design and production of parts?
- How can Fusion 360 be utilized to create files for different manufacturing processes, such as 3D printing, laser cutting, and CNC machining?

Enduring Understanding(s):

Overview

- Computer-aided manufacturing (CAM) uses computer systems to automate manufacturing processes to make products with very high accuracy and precision. These manufacturing processes are performed by computer numeric control (CNC) machines. The manufacturing processes that these machines perform are milling, turning, cutting (laser, waterjet and plasma), CNC routing, electrical discharge machining (EDM), welding, 3D printing, etc..

Understanding the Design-to-Manufacturing Process:

- Students will understand the sequential process from conceptualizing a design to manufacturing a physical part, involving both CAD and CAM.
- They will comprehend how CAD is used for designing digital models, while CAM is utilized for generating instructions for manufacturing machines.
- Students will grasp the relationship between CAD and CAM, recognizing how they work together to streamline the production process.
- The CAM process starts with CAD where the geometry for the part to be manufactured is generated. The CAD geometry file is then converted to a suitable format that a CNC machine can convert into machine language. The machine language contains all the instructions needed for the CNC machine to make the part. A CAM file is a set of G codes and M codes giving instructions to machines for the manufacturing process.

Applying Manufacturing Principles:

- Students will understand the principles behind manufacturing setups and processes.
- They will know how to create manufacturing setups in Fusion360 and select appropriate processes for a given part.
- They will be able to adjust manufacturing settings based on the chosen process to optimize production.

Integration of Design and Manufacturing:

- Students will understand the importance of integrating design and manufacturing considerations throughout the process.
- They will recognize how design decisions impact manufacturability and vice versa.
- They will be able to make informed design choices that take into account manufacturing constraints and requirements.

Problem-Solving and Adaptability:

- Students will develop problem-solving skills and adaptability in using CAD/CAM software.
- They will be able to troubleshoot issues that arise during the design or manufacturing process.
- They will understand the importance of flexibility in adjusting designs or manufacturing setups based on feedback or changing requirements.

Manufacturing Methods and Their Implications:

- Students will understand the differences between additive and subtractive manufacturing.
- They will recognize that additive manufacturing builds parts layer by layer, while subtractive manufacturing removes material from a solid block.
- They will be able to evaluate the benefits and drawbacks of each manufacturing method in terms of cost, speed, complexity, and material usage.

Application of CAD/CAM Tools:

- Students will understand how to use Fusion 360 to create files for different manufacturing processes.
- They will be able to create .stl files for 3D printing, PDF files for laser cutting, and G-code for CNC machining.
- They will recognize the importance of selecting the appropriate file format and settings based on the requirements of the manufacturing process.

- Design Constraints and Evaluation:
- Students will understand the importance of evaluating design solutions to ensure they meet design constraints.
- They will be able to identify and prioritize design constraints such as size, material, functionality, and cost.
- They will critically evaluate design solutions and make revisions as needed to meet desired criteria.

Interdisciplinary Skills and Collaboration:

- Students will recognize the interdisciplinary nature of the design and manufacturing process.
- They will understand the importance of collaboration between designers, engineers, and manufacturers to develop and produce successful products.
- They will develop communication skills necessary for effective collaboration and the ability to integrate feedback into the design process.

Demonstration of Learning:	**Pacing for Unit**
Completion of lesson activities and exercises in Fusion 360 (Manufacturing). Fabrication of designs.	13 - 75 minute classes
Family Overview (link below)	**Integration of Technology:**
Three Dimensional Modeling Family Overview (2024)	Use of the Autodesk Fusion 360 cloud-based program on desktop or laptop computers. Models may be created through additive manufacturing techniques (3D Printing) or subtractive manufacturing techniques (laser engraving, CNC machining).
Opportunities for Interdisciplinary Connections:	**Aligned Unit Materials, Resources, and Technology (beyond core resources):**
Mechanical Engineering: Fusion 360 is primarily used in mechanical engineering for designing parts, assemblies, and creating technical drawings. Students can explore concepts of material properties, forces, and stress analysis through simulations and design testing. *Architecture*: Fusion 360 can help architecture students model building structures and interior designs. *Art and Design*: Fusion 360 can be used in the creation of sculptures, installations, and interactive art.	• The Embedded Design Process: CAD/CAM and Prototyping • 3D printer(s) and Laser printer(s) • Machines - milling/ router/ lathe • 3D Content Central – Host an enormous variety of free CAD models
Unit-specific Vocabulary:	**Anticipated misconceptions:**
2d Contour, 3d Modeling Tool, Adaptive Clearing, Additive Manufacturing, Assembly, Baseline, CAD (Computer-Aided Design), CAM (Computer-Aided Manufacturing), Constraints, CNC (Computer Numerical Control), CNC Machining, Design Process, Evaluation, Collet, Caliper, Chamfer, Cutting, Design File, Clearance Height, Extrude, Feeds And Speeds, Fillet, Filament, Fusion 360, G-Code, Interdisciplinary, Iterative, Laser Cutting, Lead In/Out, Manufacturing File, Manufacturing Process, Manufacturing Setup, Navigation Bar, Orientation, PDF (Portable Document Format), Projection,Prototyping, Ramping, Rapid Movement, Revolve, Shell, Sketch, Sketch Plane, STL File, Sepover, Stock, Subtractive Manufacturing, Technical Drawing, Toolpath, 3d Printing, View Cube, X Axis, Y Axis, Z Axis, Z Slice	• There are tools and commands for parts, others for assemblies, and others for drawings. • Know the difference between the Design and Manufacturing locations and capabilities in Fusion. • The "toggle" contexts: while editing a sketch, when editing a component in assembly context, when making an exploded view, or when editing the sheet format, you are unable to do unrelated things until you close the active toggle context. • Randomly placing the first part in an assembly because you don't know to hit the green checkmark and fix it at the origin. • Using assembly mates as a way to validate modeling precision rather than a tool for restricting degrees of freedom of components. • Using the wrong tool for the desired manufacturing process. • Ensuring all values are reviewed and simulated prior

	to fabrication.
Connections to Prior Units:	**Connections to Future Units:**
In the prior two units in this course students developed their skills for computer aided design in Autodesk Fusion along with creating industry standard drawings of their designs. These skills provide the foundation for computer aided manufacturing where students learn to fabricate their designs in additive and subtractive manufacturing.	This is the final unit in this course. After successfully completing this course, students will have solid foundational knowledge, understanding, and skills of three dimensional design and manufacturing which can be used in advanced coursework for CAD and CAM.

Differentiation through *Universal Design for Learning* UDL Indicator and Teacher Actions:

Expression and Communication: Build fluencies with graduated levels of support for practice and performance
- Provide CAD, notation (writing) software, or mathematical notation software support documentation
- Use web applications (e.g., animations, presentations)
- Provide differentiated models to emulate (i.e. models use differing approaches, strategies, skills, etc.)
- Provide scaffolds that can be gradually released with increasing independence and skills
- Provide differentiated feedback (e.g., feedback customized to individual learners)
- Provide multiple examples of novel solutions to authentic problems

Supporting Multilingual/English Learners

Related *CELP standards* and Learning Targets:

An EL can participate in grade appropriate oral and written exchanges of information, ideas, and analyses, responding to peer, audience, or reader comments and questions.
I can create a manufacturing file using Fusion360.

Level 1: With prompting and support, use a very limited set of strategies to:
- Follow basic instructions to create a manufacturing file using Fusion360
- Use simple vocabulary to describe the steps involved in creating a manufacturing file
- Respond to yes/no questions and some basic questions about creating a manufacturing file

Level 2: With prompting and support, use an emerging set of strategies to:
- Follow instructions to create a manufacturing file using Fusion360
- Participate in short exchanges about creating manufacturing files using academic and domain-specific vocabulary
- Respond to simple questions about the process of creating a manufacturing file

Level 3: With guidance and support, use a developing set of strategies to:
- Discuss and ask questions about creating manufacturing files using Fusion360
- Use academic and domain-specific vocabulary to describe the steps involved in creating a manufacturing file
- Add relevant information and evidence to discussions about creating manufacturing files
- Restate key ideas about creating manufacturing files expressed by others

Level 4: Use an increasing range of strategies to:
- Engage in discussions and written exchanges about creating manufacturing files on various topics and issues using academic and domain-specific vocabulary
- Build on the ideas of others regarding creating manufacturing files
- Clearly express own ideas about creating manufacturing files with specific evidence
- Ask and answer questions to clarify steps and concepts related to creating manufacturing files
- Summarize key points discussed about creating manufacturing files

Level 5: Use a wide range of strategies to:
- Participate in extended discussions and written exchanges about creating manufacturing files on substantive topics and issues using academic and domain-specific vocabulary
- Build on the ideas of others to deepen discussions about creating manufacturing files
- Clearly and persuasively express own ideas about creating manufacturing files with specific and relevant evidence
- Refer to specific evidence from texts or research to support ideas about creating manufacturing files
- Ask and answer questions that probe reasoning and claims related to creating manufacturing files

● Summarize key points and evidence discussed about creating manufacturing files			
Lesson Sequence	**Learning Target**	**Success Criteria/ Assessment**	**Resources**
1	I can explain the difference between and characteristics of Computer-Aided Design (CAD) and Computer-Aided Manufacturing (CAM). Students will be able to: ● Explain the difference between Computer Aided Design (CAD) and Computer Manufacturing Design (CAM). ● Describe what CAD is and how it is used. ● Describe what CAM is and how it is used.	Computer generated diagram comparing and contrasting CAD and CAM.	● CAD/CAM Presentation ● Video demonstrations illustrating differences between CAD/CAM and final products. ● Manufacture overview
2	I can create a manufacturing file in Fusion 360, set up a new manufacturing process, select the appropriate process, and adjust manufacturing settings accordingly. Students will be able to: ● Create a manufacturing file in Fusion360. ● Create a new manufacturing setup. ● Select the proper manufacturing process ● Change and adjust manufacturing settings based on the chosen manufacturing process.	1-Caliper_Front_FirstOp_With_NoSetUp 2-Caliper_Front_FirstOp_With_SetUp 3-Caliper_Front_FirstOp_Complete	01 Fusion 360 CAM Fundamentals, 2.5 Axis Toolpath operations Presentation Milling Basics
3	I can describe and explain each step of the design process, apply the steps to transform an idea into a solution, and evaluate the solution to ensure it meets design constraints. Students will be able to: ● Describe the use of the design process. ● Explain each step in the design process. ● Apply each of the steps of the design process to take an idea and turn it into a solution. ● Evaluate the design solution to ensure it effectively meets design constraints.	4-Caliper_Front_Setup With Toolpaths 5 - 3D Milling - Overview of Toolpaths 6-3D Milling - Overview of toolpaths_StarterFile	02 Fusion 360 CAM Fundamentals, 3 Axis Toolpath operations Presentation 2D adaptive milling tutorial
4	I can explain the difference between additive and subtractive manufacturing, identify three additive and three subtractive manufacturing processes along with their benefits and drawbacks, and demonstrate the use of Fusion 360 to create files for exporting to a device to produce a part. Students will be able to:	Computer generated diagram comparing and contrasting additive and subtractive manufacturing.. Additive FFF CAM Files Additive SLA CAM Files	Additive & Subtractive Manufacturing Presentation Manufacturing Product Samples Additive FFF tutorial Additive SLA tutorial

	● Explain the difference between additive and subtractive manufacturing? ● Name 3 additive manufacturing processes with the benefits and drawbacks of each. ● Name 3 subtractive manufacturing processes with the benefits and drawbacks of each ● Demonstrate the use of Fusion 360 to create the necessary file(s) needed to export to a device to create a part.		
5	I can create a physical part by using Fusion 360 to generate an .stl file for 3D printing in MakerPrint, export a design as a PDF for laser printing in CorelDRAW, and create G-code for CNC machining to process a material. Students will be able to: ● Create a physical part utilizing Fusion 360 to create a .stl file and export it into MakerPrint for setup for a 3D printer. ● Create a physical part utilizing Fusion 360 for export as a pdf file into CorelDRAW for setup of a laser printer. ● Create G-code utilizing Fusion 360 to export into a CNC machine for subtractive processing of a material.	CAM Files for Selected Projects Fabricated Projects	3D Printing from Autodesk Fusion Presentation Laser Engraving from Autodesk Fusion Presentation