ADVM 0011 - THREE-DIMENSIONAL MODELING

Catalog Description

Formerly known as DES 11
Prerequisite: Completion of ADVM 2 or ADVM 3D or MECH 44 or ENGR 151 or ADVM 66 with grade of "C" or better; or equivalent as determined by instructor
Hours: 90 (36 lecture; 54 laboratory which may be scheduled TBA)
Description: Processes employed in developing design solutions using a feature based parametric solid modeler. Includes 3D part and assembly modeling, and the development of 2-dimensional part, assemblies, welding and sheet metal drawings per ASME standards. SolidWorks is the solid modeler used. (CSU, UC)

Course Student Learning Outcomes

- CSLO #1: Apply appropriate, current and relevant industry standards in preparing technical documentation for the appropriate discipline of study.
- CSLO #2: Apply welding symbols to engineering drawings showing the different types of welded joints.
- CSLO #3: Create custom drawing sheets, utilize drawing templates for drawing development, create drawing views that conform to AMSE/ANSI Standards and create bill of materials and design tables.
- CSLO #4: Create working drawings of parts using measuring devices to interpret feature size.

Effective Term

Fall 2022

Course Type

Credit - Degree-applicable

Contact Hours

90

Outside of Class Hours

72

Total Student Learning Hours

162

Course Objectives

Lecture Objectives:
I. Measurement and Gauging
1. Distinguish between the different types of measuring devices used in manufacturing.
II. Tolerance Dimensioning
1. Compare the different methods for specifying tolerance dimensions.
2. Distinguish between the practices for size designations.
3. Compare the methods for determining the fit between mating parts.
4. Compare the Basic Hole System to the Basic Shaft System.
5. Compare specifications of tolerance.
6. Discuss the American National Standard of limits and fits.
7. Calculate the prescribed fits and tolerances according to the ANSI standard fits tables.
8. Describe the disadvantages of accumulation of tolerances.
9. Discuss the Metric system for tolerances and fits.
10. Examine the foundations of geometric dimensioning and tolerancing at a rudimentary level.
III. Overview of three Dimensional Modeling
1. Differentiate between the different types of Solid Modelers.
2. Discuss the hardware requirements for three dimensional modeling.
IV. SolidWorks Software
1. Describe the user interface and explain the function of each major component.
V. Sketching
1. Create sketches that express the design intent.
2. Distinguish between the stages of the sketching process.
3. Demonstrate the requirements for sketch relations.
4. Analyze the sketch for complete dimensions.
5. Evaluate the sketch for design intent and edit the sketch accordingly.
VI. Sketch Contours
1. Analyze sketch contours and apply parameter changes.
2. Apply the rules that govern sketches relative to contours.
VII. Part Modeling
1. Demonstrate the stages of basic part modeling.
2. Utilize the terminology used in part modeling.
3. Identify the characteristics of boss features.
4. Identify the characteristics of cut features.
5. Demonstrate the rules that govern filleting and the propagation of fillets.
VIII. Revolved Features and Circular Patterns
1. Demonstrate the stages for revolved feature and circular pattern development.
2. Analyze the revolved features and circular patterns for meeting design intent.
3. Apply the rules that govern revolved features.
4. Utilize the characteristics of the feature manager.
5. Analyze revolved features for engineering changes and rebuild problems.
6. Demonstrate the use of equations for design intent.
IX. Configurations of parts
1. Demonstrate the use of part configurations.
2. Utilize the design table property manager.
3. Evaluate the design table for meeting the design intent.
X. Editing
1. Demonstrate the process of part editing.
2. Differentiate between the mechanisms for part editing.
XI. Assembly Modeling
1. Demonstrate the bottom-up philosophy of assembly modeling.
2. Analyze the stages in assembly modeling.
3. Utilize the feature manager design tree for assembly modeling.
4. Analyze assembly models for component properties.
5. Utilize the procedure for hiding components in an assembly.
6. Utilize the tools for analyzing assembly models.
XII. Drawings- ASME Compliance
1. Analyze the stages in creating drawings from part/assembly models.
2. Utilize the terminology used in drawing development.
XIII. Welding Representation
1. Differentiate between welding processes and describe the benefits of each.
2. Compare the different types of welded joints.
3. Compare the different types of welds and describe characteristics of each.
4. Apply welding symbols to engineering drawings.

Laboratory Objectives:
I. Measurement and Gauging
1. Create working drawings of parts using measuring devices to interpret feature size.
II. Tolerance Dimensioning
1. Create drawings to include the various methods for specifying tolerances to features on parts.
2. Demonstrate the methods for determining the fit between mating parts.
3. Utilize the different types of tolerance specifications.
4. Utilize the American National Standard of limits and fits.
5. Calculate the prescribed fits and tolerances according to the ANSI standard fits tables.
6. Utilize the Metric system for tolerances and fits.
7. Create drawings of parts and apply the symbols used in geometric dimensioning and tolerancing.
III. Overview of three Dimensional Modeling
1. Select the most appropriate modeling solution that meets the Design Intent.
IV. SolidWorks Software
1. Demonstrate the user interface and utilize the function of each major component.
V. Sketching
1. Create sketches that express the design intent.
2. Construct sketches utilizing the default planes.
3. Select sketch geometry to meet design intent.
4. Set up sketch geometry taking advantage of automatic relations.
5. Create sketches with fillets.
6. Create part models using boss and cut extrudes.
7. Evaluate the sketch for design intent and edit the sketch accordingly.
VI. Sketch Contours
1. Create sketch contours and develop part geometry from these contours.
2. Analyze sketch contours and apply parameter changes.
3. Create extruded part geometry using contour selection.
VII. Part Modeling
1. Create part geometry using the best profile of the part model.
2. Select the sketch plane based upon the best profile of the part model.
3. Construct the details of the part model following the design intent.
4. Compare the viewing options for part modeling.
5. Utilize the rules that govern filleting and the propagation of fillets.
VIII. Revolved Features and Circular Patterns
1. Analyze the revolved features and circular patterns for meeting design intent.
2. Create sketch geometry for revolved features.
3. Create revolved features from sketch geometry.
4. Create circular patterns of revolved features.
5. Construct revolved features that include chamfers.
6. Analyze revolved features for engineering changes and rebuild problems.
7. Utilize the use of equations for design intent.
IX. Configurations of parts
1. Create different configurations of part models.
2. Create design tables.
3. Utilize the design table property manager.
4. Utilize the design table for meeting the design intent.
X. Editing
1. Describe the stages in the process of part editing.
2. Demonstrate the mechanisms for part editing.
XI. Assembly Modeling
1. Utilize the feature manager design tree for assembly modeling.
2. Select the best method for the mating of components.
3. Create assembly models using part configurations.
4. Analyze assembly models for component properties.
5. Create assembly models using sub assemblies methods.
6. Utilize the tools for analyzing assembly models.
7. Create exploded assembly models.
8. Create physical simulations of assembly models.
9. Construct assembly drawings from assembly models.
XII. Drawings- ASME Compliance
1. Create custom drawing sheets.
2. Utilize drawing templates for drawing development.
3. Create drawing views that conform to AMSE Standards.
4. Create sketches in drawing views.
5. Select the appropriate settings for drawing views.
6. Create drawings utilizing appropriate drawing annotations.
7. Select the settings for drawings.
8. Create bill of materials and design tables.
XIII. Welding Representation
1. Apply welding symbols to engineering drawings.

General Education Information

- Approved College Associate Degree GE Applicability
- CSU GE Applicability (Recommended-requires CSU approval)
- Cal-GETC Applicability (Recommended - Requires External Approval)
- IGETC Applicability (Recommended-requires CSU/UC approval)

Articulation Information

- CSU Transferable
- UC Transferable

Methods of Evaluation

- Objective Examinations
  - Example: Describe the benefits of parametric modeling and how it influences product design. This question is evaluated relative to how well the student describes the benefits and how these benefits influence product design.
- Projects
  - Example: Students are to design a commercial quality metal structure following industry standards, and design criteria for optimum performance. Students will develop the 3D Models, Assemblies and complete set of detailed working drawings along with bills of materials for the project. Instructor evaluates the student performance of learned objectives such as part and assembly modeling, two dimensional part and assembly drawings that represent the design intent, accuracy to American Society of Mechanical Engineers (ASME) standards for compliance and the efficient use of a computer system. A point system is used and a letter grade assigned to the point totals.
- Skill Demonstrations
  - Example: Student will use proper techniques to develop CAD models and produce ASME compliant detail drawings from the models. Instructor evaluates the student performance of learned objectives such as part and assembly modeling, two dimensional part and assembly drawings that represent the design intent, accuracy to American Society of Mechanical Engineers (ASME) standards for compliance and the efficient use of a computer system. A point system is used and a letter grade assigned to the point totals.

Repeatable

No

Methods of Instruction

- Laboratory
- Lecture/Discussion
- Distance Learning

Lab:

1. The instructor provides physical parts and precision measuring equipment to students and guides them in the proper inspection and documentation techniques for reverse engineering a product design. Students will utilize the tools and parts to gather design criteria and data and synthesize the information onto design sketches.

Lecture:

1. The instructor will present to the students 3 dimensional parametric solid modeling methodologies. Students will discuss the various methods and strategize a plan to develop the models.

Typical Out of Class Assignments Reading Assignments

1. Students read chapter on assembly development and are expected to participate in the lecture/discussions based upon these readings. 2. Students are to construct a drawing, based upon their course readings, demonstrating the weekly-learning objectives. These weekly drawings are either freehand sketches and/or computer aided design (CAD) generated. The drawings are evaluated for compliance with American Society of Mechanical Engineers (ASME) standards. Critical thinking and problem solving are part of these assignments. 3. Students are required to search the Internet for articles that reference design for manufacture and assembly, then utilize their findings to design a commercial quality weldment, producing the 3D models and ASME documentation.

Writing, Problem Solving or Performance

1. Students will write a report comparing and contrasting methods employed in design for manufacture and assembly. 2. Students are required to prescribe appropriate fits and tolerances to mating parts in a working assembly. Calculations of Tolerance, Limits, Maximum and Least Material Condition, Minimum and Maximum Clearance are required.

Other (Term projects, research papers, portfolios, etc.)

1. Students are required to develop a portfolio that contains samples of their semester assignments to show potential employers the engineering design concepts studied.

Required Materials

- Parametric Modeling with SOLIDWORKS
  - Author: Shih
  - Publisher: Schroff Development Corporation Publications
  - Publication Date: 2017
  - Text Edition:
  - Classic Textbook?:
  - OER Link:
  - OER:

Other materials and-or supplies required of students that contribute to the cost of the course.