**CHAMP Course Map**

|  |  |
| --- | --- |
| **Course Name:** CAD 255 | |
| **Instructor Name:** Community College of Denver | **Date:** July 2016 |
| Course Competencies:   1. Locate given points in a coordinate system. 2. Discuss the right hand rule, viewpoint, standard views, plan view, and isometric views. 3. Explain view orientation in relationship to 3D coordinate systems. 4. Define, create and use drawing, work, and sketching planes to develop a model profile. 5. Discuss proper profile creation and limitations. 6. Use pre-constructed solid shapes as building blocks. 7. Identify hidden lines. 8. Discuss advantages and disadvantages of one technique over another. 9. Discuss the three-dimensional modeling process for parametric models and non- parametric models. 10. Create extrusions of profiles to three-dimensional parametric models. 11. Discuss the theory of constraints and how they drive the model construction. 12. List default constraints and explain their control over parametric geometry. 13. Solve basic geometry constructions using constraints. 14. Develop a new set of constraints by removing, modifying or adding constraints. 15. Use standard parametric construction techniques to create boxes, cylinders, and spheres. 16. Create sweeps using linear and circular operations. 17. Discuss the theory of adding holes, fillets and chamfers to parametric models. 18. Recognize the limitations of and restrictions of placing features on models. 19. Demonstrate how to correct over defined sketches. 20. Assign dimensions to the parametric model. | |

**Course Materials (Text, Edition and any other publisher items)**

**Textbooks and/or Resources:** **Engineering Drawing and Design 6th Edition, Madson and Madson, 2016**

**Resources:**

| **Module # and Title** | **CCNS Competencies** | **Content, Activities or Challenges**  **(Learner Interaction**  **& Engagement)** | **Assessments, Rubrics (Feedback)** | **Publish to OER** |
| --- | --- | --- | --- | --- |
| Module 1:  Introduction to SolidWorks | 1, 2 | 1. Discuss Syllabus and Outline 2. Lecture on setting up a Project 3. Lecture on launching and viewing a part in Inventor 4. Assignment: Using the SolidWorks Interface    1. Launch the software    2. Open a new file    3. View Project Dialog box    4. Select the correct environment configuration in SolidWorks |  | * Syllabus and Outline |
| Module 2:  Modeling in SolidWorks: 3D Solid Modeling | 3, 4, 5, 6, 7, 8 | 1. View the 21 parts of the Oscillating Steam Engine in the Syllabus 2. Lecture on each type of the following modeling techniques    1. Extrude (Add/Subtract)    2. Revolve (Add/Subtract)    3. Hole (Depth/Type/Thread Control)    4. Shell    5. Sweep    6. Coil 3. Modeling Assignment: Sketch and constrain the individual parts and extrude them into 3D space 4. Save all parts in the correct file structure, as explained in the naming conventions document | Modeling Assignment Assessment - Individual part files turned in and graded based on 3D Grading Rubric | * Oscillating Steam Engine Drawing * Oscillating Steam Engine Parts Drawings * 3D Grading Rubric |
| Module 3:  Assembling in SolidWorks: 3D Solid Modeling Assembly | 10, 11 | 1. Lecture on Assembling in SolidWorks    1. Mate    2. Angle    3. Tangent    4. Insert    5. How to control the location of parts relative to each other 2. Assembly Assignment: Assemble all the parts of the Oscillating Steam Engine in SolidWorks | Assembly Assignment Assessment – Assembly file turned in and graded based on 3D Grading Rubric |  |
| Module 4:  Controlling Exploded Assemblies: SolidWorks Presentation | 12 | 1. Lecture on “Explode Component” 2. Assignment: In SolidWorks, space apart every part of Oscillating Steam Engine into an exploded assembly | Exploded Assembly Assignment Assessment – Exploded Assembly file turned in and graded based on 3D Grading Rubric |  |
| Module 5:  Creating a Document Set: SolidWorks Drawing | 9 | 1. Lecture on how to create a document set 2. Document Set Assignment:    1. Use the drawing section in SolidWorks to control the location of parts, assemblies, in a ANSI or ISO drawing format.    2. Add the correct dimensions, annotations with proper document control | 1. Final Project:    1. Correct Part Models, feedback from 3D Grading Rubric    2. Correct Assemblies, feedback from 3D Grading Rubric    3. Correct Drawings, feedback from 3D Grading Rubric |  |