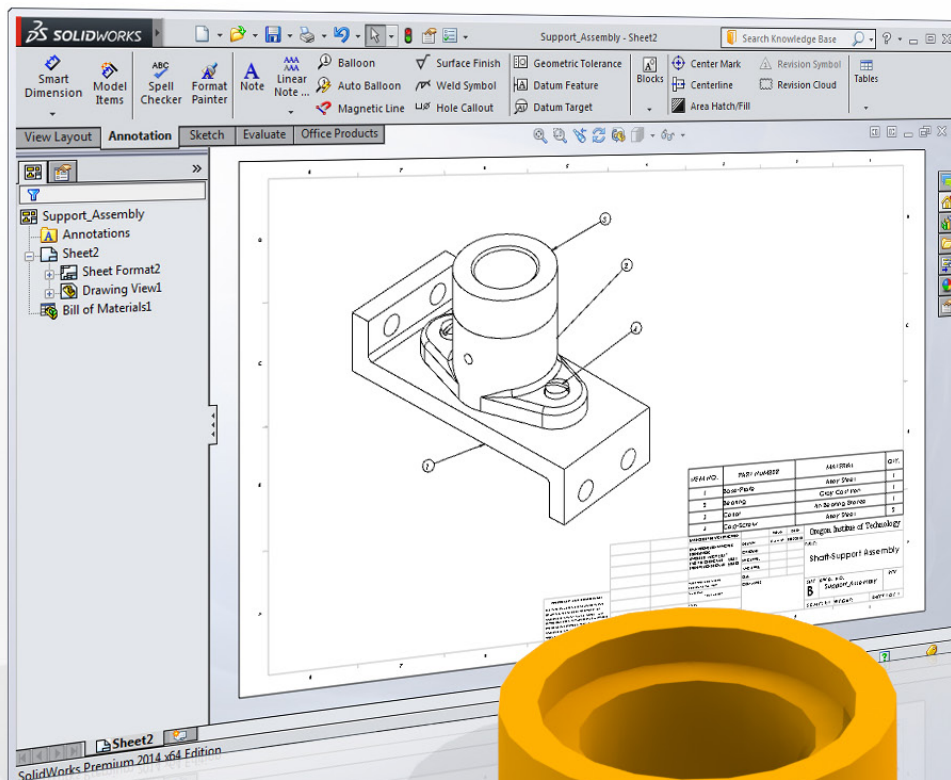


SolidWorks® 2014 and Engineering Graphics

An Integrated Approach



Randy H. Shih

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Certified SolidWorks Associate (CSWA) Exam Overview

The Certified SolidWorks Associate (CSWA) Exam is a performance-based exam. The examination is comprised of 10 – 20 questions to be completed in three hours. The test items will require you to use the *SolidWorks* software to perform specific tasks and then answer questions about the tasks.

Performance-based testing is defined as *Testing by Doing*. This means you actually perform the given task then answer the questions regarding the task. Performance-based testing is widely accepted as a better way of insuring the user has the skills needed, rather than just recalling information.

The CSWA examination is designed to test specific performance tasks in the following areas:

Sketch Entities – Lines, Rectangles, Circles, Arcs, Ellipses, Centerlines

Objectives: Creating Sketch Entities.

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Objectives: Creating Fillets and Chamfers.

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Standard Mates – Coincident, Parallel, Perpendicular, Tangent, Concentric, Distance, Angle

Objectives: Applying Standard Mates to Constrain Assemblies.

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Objectives: Creating Reference Planes, Axes, and Mate References

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Drawing Sheets and Views

Objectives: Creating and Setting Properties for Drawing Sheets; Inserting and Editing Standard Views.

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Annotations

Objectives: Creating Annotations.

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