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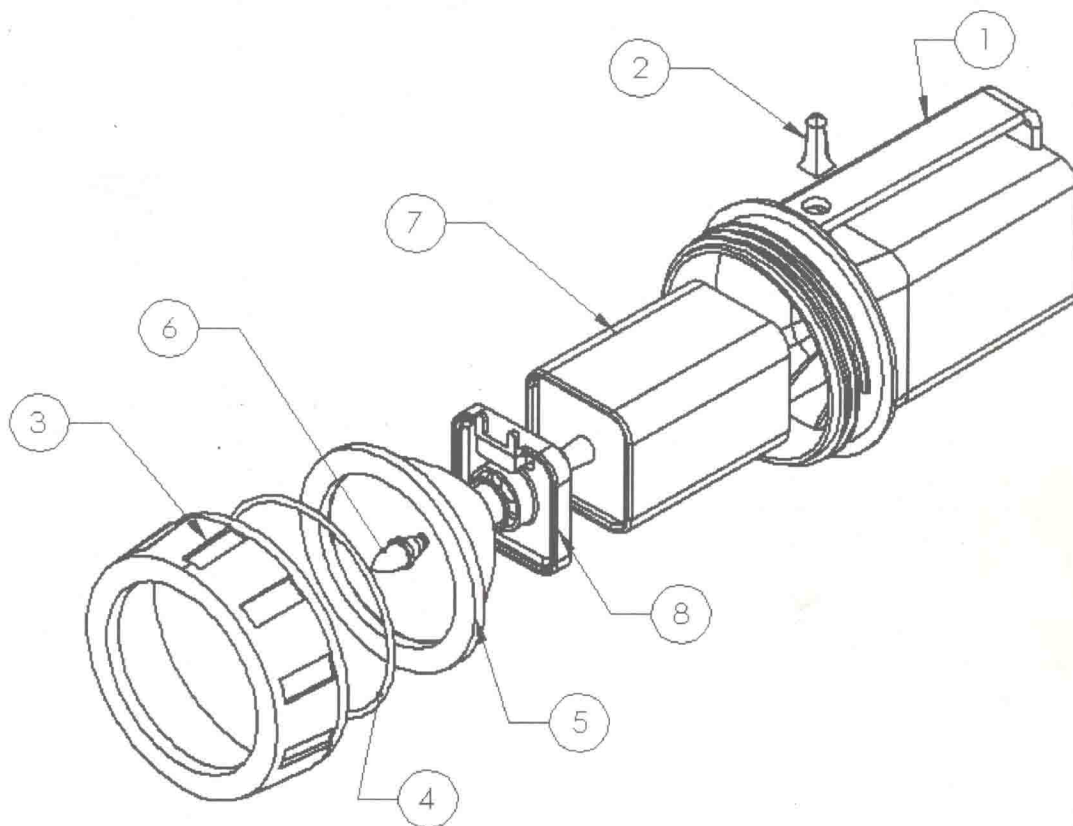
# SolidWorks Tutorial

## 2001Plus

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A Competency Project Based Approach  
Utilizing 3D Solid Modeling

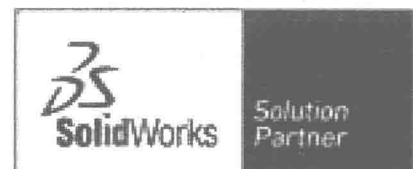
David C. Planchard & Marie P. Planchard



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# **SolidWorks Tutorial**

## **2001 Plus**

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### **A Competency Project Based Approach Utilizing 3D Solid Modeling**

David C. Planchard & Marie P. Planchard

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## About the Authors

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David and Marie Planchard are co-founders of D&M Engineering and are active industry and education consultants. They are co-authors of the following SDC Publication books:

- **Engineering Design with SolidWorks 1999, 2000, 2001, 2001Plus**
- **Drawing and Detailing with SolidWorks 2001/2001Plus**
- **SolidWorks Tutorial 2001**
- **Applications in Sheet Metal Using Pro/SHEETMETAL & Pro/ENGINEER**
- **An Introduction to Pro/SHEETMETAL**

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## INTRODUCTION

### Preface

SolidWorks 2001Plus Tutorial was written to assist students, designers, engineers and professionals. SolidWorks 2001Plus Tutorial is a subset of Engineering Design with SolidWorks. The book provides a quick introduction to SolidWorks. The book is focused on providing a solid foundation in SolidWorks using competency-based projects. Desired outcomes and usage competencies are listed for each project.

Commands are presented in a step-by-step progressive approach.

A competency project based curriculum is provided. The learning process is explored through a series of design situations, industry scenarios, projects and objectives.

A progressive learning approach is addressed in each chapter. Each chapter identifies a project with reflective information from the previous project situation. The book is designed to compliment the on-line tutorials contained within SolidWorks.

The authors developed the industry scenarios by combining their own industry experience with the knowledge of engineers, department managers, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of a 3D model.

Please contact us with any comments, questions or suggestions on this book.

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Information in this text is provided from the ASME Engineering Drawing and Related Documentation Publications:

ASME Y14.1 1995  
ASME Y14.2M-1992 (R1998)  
ASME Y14.3M-1994 (R1999)  
ASME Y14.5M-1994

The illustrations and part documents were recreated in SolidWorks. Note: By permission of The American Society of Mechanical Engineers, Codes and Standards, New York, NY, USA. All rights reserved.

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Notes:

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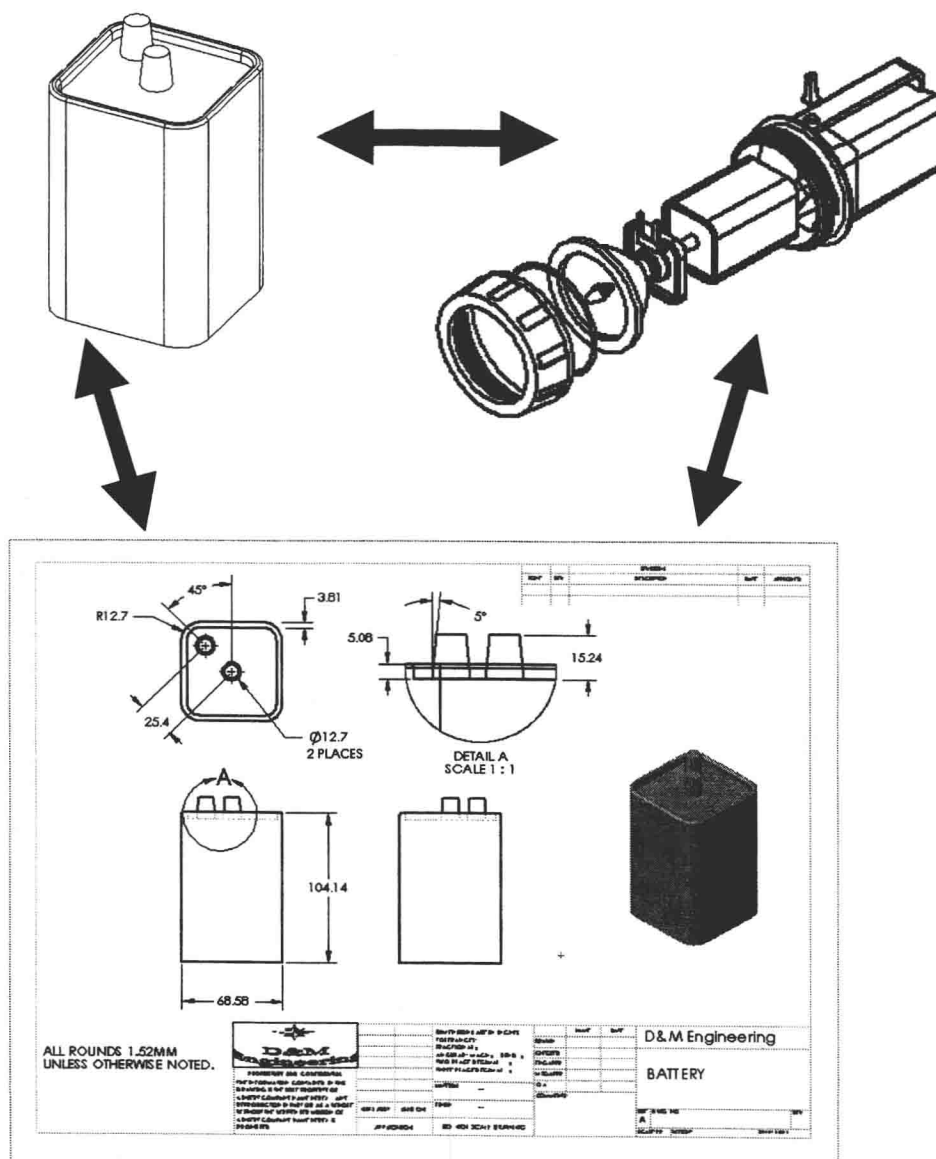
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## What is SolidWorks?

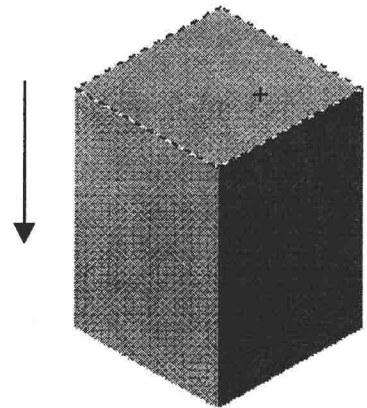
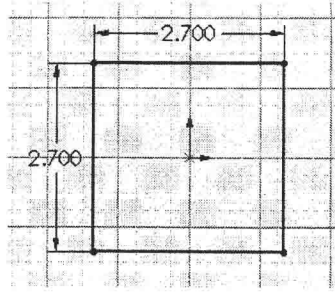
SolidWorks is a design automation software package used to produce parts, assemblies and drawings. SolidWorks is a Windows native 3D solid modeling CAD program. SolidWorks provides easy to use, highest quality design software for engineers and designers who create 3D models and 2D drawings ranging from individual parts to assemblies with thousands of parts.

SolidWorks Corporation, headquartered in Concord, Massachusetts USA, develops and markets innovative design solutions for the Microsoft Windows platform. More information on SolidWorks and its family of products can be found at their URL, [www.SolidWorks.com](http://www.SolidWorks.com).

In SolidWorks, you create 3D parts, assemblies and 2D drawings. The part, assembly and drawing documents are all related.



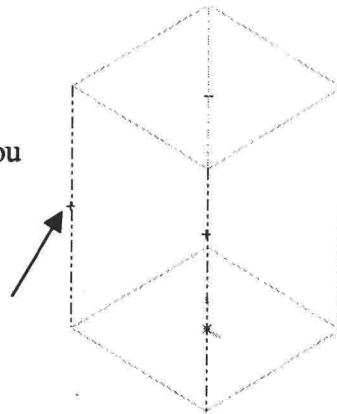
The building blocks of parts are called features. Features such as the Extruded-Boss, Cut, Hole, Fillet, Chamfer and others are used to create parts. Some features are sketched, such as an Extruded-Boss.



Other features are created by selecting edges or faces of existing features,

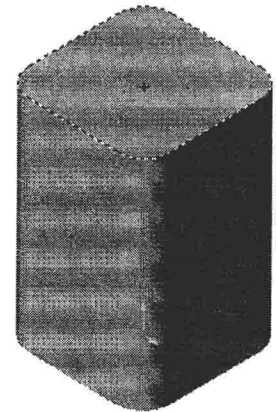
such as a Fillet.

Dimensions drive features. Change a dimension and you change the size of the part.

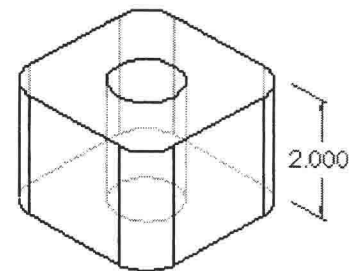
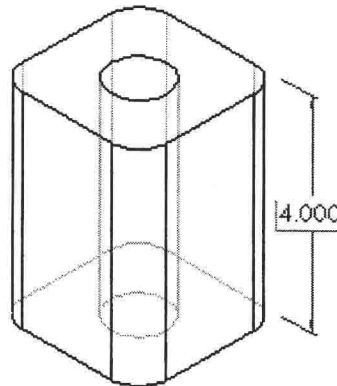


Geometric relationships are used to maintain the intent of the design.

Create a hole that penetrates through a part. SolidWorks maintains relationships through the change.



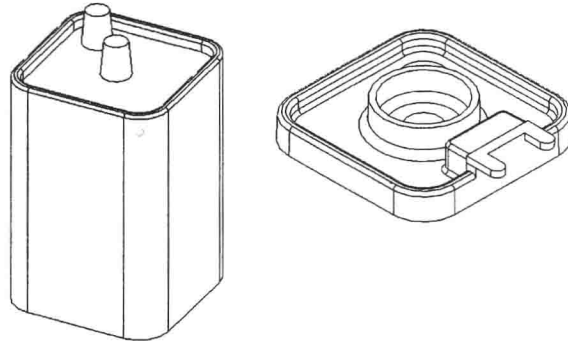
The step-by-step approach used in this text allows you to create parts, assemblies and drawings. The text also allows you to modify and change all components of the model. Change is an integral part of design. Let's begin.



## Overview of Projects

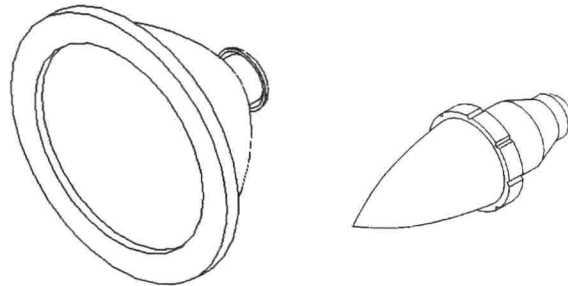
### Project 1: Extrude Features

How do you start a design in SolidWorks? What is the design intent? How do you take a customer's requirements and convert them into a model? Project 1 introduces the basic concepts behind SolidWorks. In Project 1 you create two parts: BATTERY and BATTERY PLATE. You are exposed to the following features: Extrude Base, Extrude Boss, Extrude Cut and Fillet.



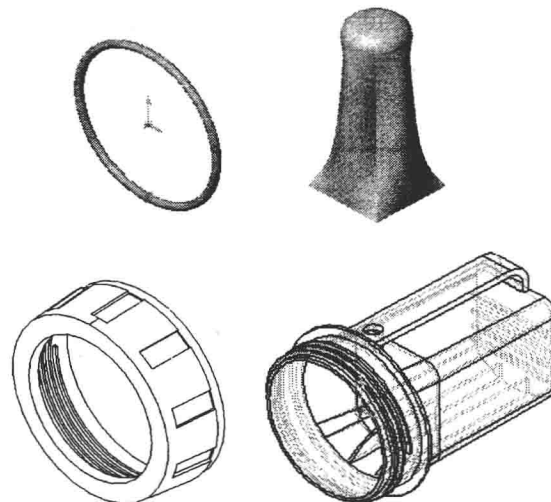
### Project 2: Revolve Features

Project 2 introduces the Revolve features. The LENS and BULB are created with the Revolve Base. Explore the Revolve Cut and Revolve Boss features. Circular Pattern, Dome feature and Transparent Feature Properties are introduced.



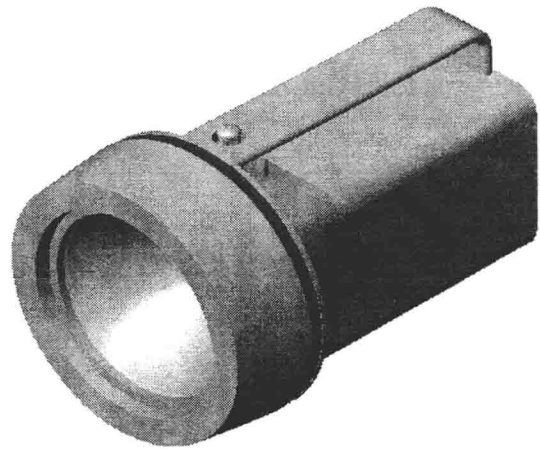
### Project 3: Sweep and Loft Features

Project 3 covers the development of the Sweep and Loft features and strengthens the use of previously defined features. The O-RING utilizes a Sweep Base feature. The SWITCH utilizes the Loft Base feature. The LENS CAP and HOUSING are developed with numerous features. A helical curve creates the thread for the LENS CAP and HOUSING.



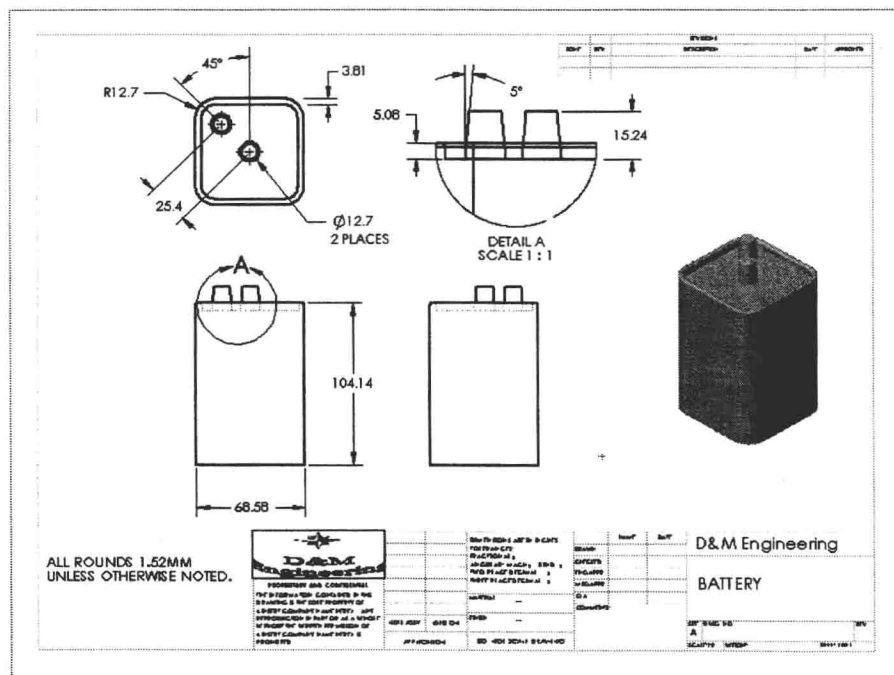
### Project 4: Assembly Fundamentals

Project 4 develops the LENSAND BULB, BATTERYANDPLATE, and LENSANDCAP assemblies. These assemblies combined with the SWITCH and HOUSING components to complete the FLASHLIGHT assembly.




### Project 5: Drawing Fundamentals

Project 5 covers the development of a customized drawing template. Develop a company logo. Create a BATTERY drawing with five views. Develop and incorporate a Bill of Materials into the FLASHLIGHT assembly drawing.



## Command Syntax

The following command syntax is used throughout the text. Commands that require you to perform an action are displayed in **Bold** text.

Format	Convention	Example
<b>Bold</b>	<p>All commands actions.</p> <p>Selected icon button.</p> <p>Selected geometry: line, circle, arc, point and text</p> <p>Value entries.</p>	<p>Click <b>Save</b>. Click <b>Tools</b>, <b>Options</b> from the Main menu.</p> <p>Click <b>Rectangle</b>  from the Sketch Tools toolbar.</p> <p>Select the <b>centerpoint</b>. Drag the <b>circle</b> downward. Click the <b>arc</b>.</p> <p>Enter <b>3.0</b> for Radius. Click <b>60mm</b> from the Depth spin box.</p>
Capitalized	<p>Filenames, part names, assembly, component and drawing names.</p> <p>First letter in a feature name.</p>	<p>The <b>BATTERY</b> is contained inside the <b>FLASHLIGHT</b> assembly.</p> <p>Click the <b>Fillet</b> feature. Click the <b>Extrude Base</b> feature.</p>

Project dimensions are provided in both inches and millimeters. Inches are the primary design units and millimeters are the secondary design units. Millimeters are displayed in brackets [x].

Part dimensions are displayed with three decimal places for inches and two decimal places for millimeters. Control individual decimal display in your drawing to adhere to the following drawing standards: ANSI or ISO. Use PartEnglishTemplate and AssemblyEnglishTemplate for inch units. Use the [PartMetricTemplate] and [AssemblyMetricTemplate] for millimeter units.

This text follows the ASME Y14 Engineering Drawing and Related Documentation Practices for drawings. Display of dimensions and tolerances are as follows:

<b>TYPES of DECIMAL DIMENSIONS (ASME Y14.5M)</b>			
<b>Description</b>	<b>Example MM</b>	<b>Description</b>	<b>Example INCH</b>
Dimension is less than 1mm. Zero precedes the decimal point.	0.9 0.95	Dimension is less than 1 inch. Zero is not used before the decimal point.	.5 .56
Dimension is a whole number. Display no decimal point. Display no zero after decimal point.	19	Express dimension to the same number of decimal places as its tolerance. Add zeros to the right of the decimal point.	1.750
Dimension exceeds a whole number by a decimal fraction of a millimeter. Display no zero to the right of the decimal.	11.5 11.51	If the tolerance is expressed to 3 places, then the dimension contains 3 places to the right of the decimal point.	

<b>TABLE 1</b>		
<b>TOLERANCE DISPLAY FOR INCH AND METRIC DIMENSIONS (ASME Y14.5M)</b>		
<b>DISPLAY</b>	<b>INCH</b>	<b>METRIC</b>
Dimensions less than 1	.5	0.5
Unilateral Tolerance	1.417 <sup>+0.005</sup> <sub>-0.000</sub>	36 <sup>0</sup> <sub>-0.5</sub>
Bilateral Tolerance	1.417 <sup>+0.010</sup> <sub>-0.020</sub>	36 <sup>+0.25</sup> <sub>-0.50</sub>
Limit Tolerance	.571 .463	14.50 11.50



**Windows 95/98/2000/NT4.0 Terminology**

The mouse pointer provides an integral role in executing SolidWorks commands. The mouse pointer executes commands, selects geometry, displays Pop-Up menus and provides information feedback. A summary of mouse pointer terminology is displayed below:

Item	Description
Click	Press and release the left mouse button.
Double-click	Double press and release the left mouse button.
Click inside	Press the left mouse button. Wait a second and then press the left mouse button inside the text box. This technique is used to modify Feature names in the FeatureManager design tree.
Drag	Point to an object, press and hold down the left mouse button. Move the mouse pointer to a new location, release the left mouse button.
Right-click	Press and release the right mouse button. A Pop-up menu is displayed. Use the left mouse button to select a menu command.
Tool Tip	Position the mouse pointer over an Icon (button). The command is displayed below the mouse pointer.
Mouse pointer feedback	Position the mouse pointer over various areas of the sketch: part, assembly or drawing. The cursor provides feedback depending upon the geometry.