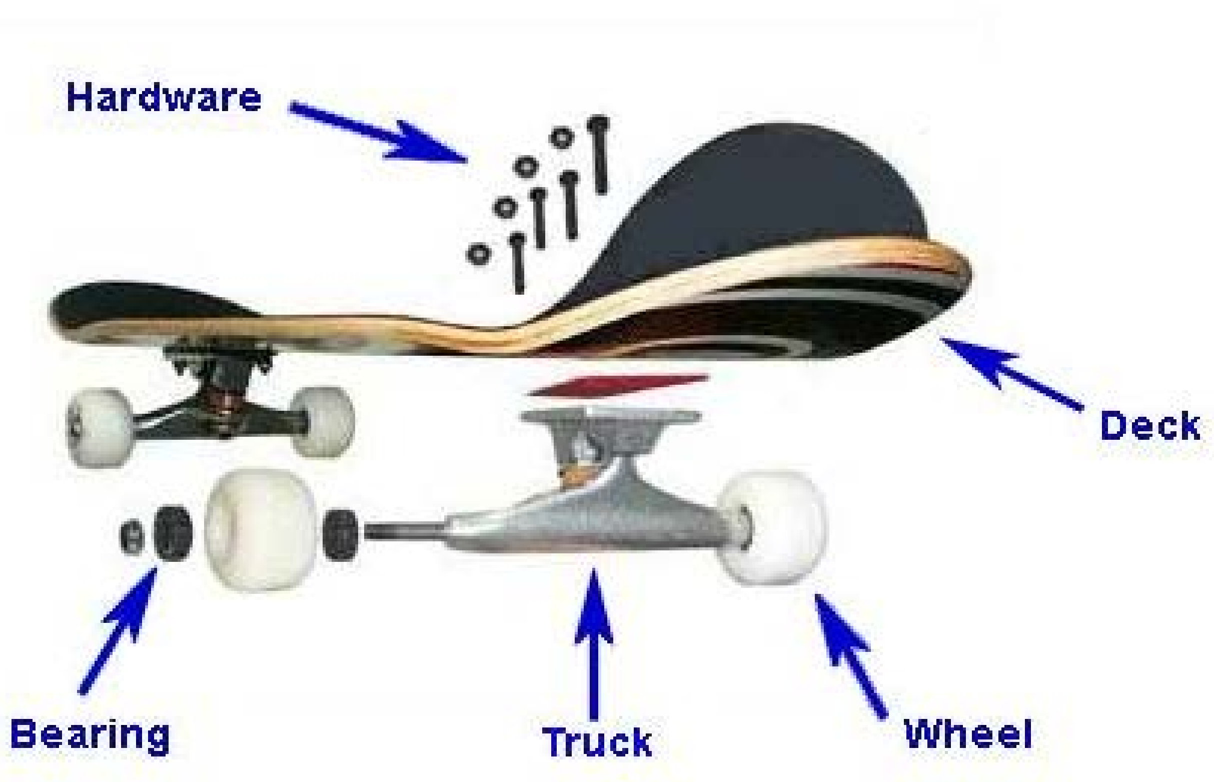
Introduction to SolidWorks second edition

SOLIDWORKS



Mario H. Castro-Cedeño, PE
Certified SolidWorks Associate

Motivation for Writing this Book

My reason for writing another “Introduction to SolidWorks” book is to emphasize the role and importance of computer aided design (CAD) and engineering drawings in communicating design ideas. The old adage “a picture is worth a thousand words” is true in mechanical engineering design. Often it is impossible to describe a complex design in words and we must use engineering drawings.

Many introductory CAD tutorials focus only on the software and train students to be skilled CAD operators. This book also places equal emphasis on creating engineering drawings that comply with accepted design standards. Although much effort and resources have been expended in technologies that reduce or eliminate the need for paper documents, most organizations still depend on engineering drawings in paper or electronic form.

Engineering drawings are still the most important and efficient method of communication between the design and manufacturing functions. They are also necessary for the maintenance and repair of consumer and industrial products as varied as automobiles, airplanes, earth moving equipment and cruise ships. Lastly, engineering drawings are legal documents and commonly used to determine and apportion fault in product liability lawsuits. For all these reasons, every engineering and manufacturing professional must be proficient at creating, understanding and using engineering drawings.

Target Audience

This book is written primarily for classroom instruction at the high school, vocational and college levels. With that audience in mind, SolidWorks commands are introduced while designing a skateboard.

Experienced CAD professionals that want to learn SolidWorks on their own will also find the book useful. The book is organized in a way that allows experienced users to easily find and learn the commands needed to customize SolidWorks, and to create solid models, assemblies and drawings.

All readers, students, as well as practicing professionals, will find that this is also an excellent reference book. The alphabetical index at the end of the book can be used to find information about the most common SolidWorks commands and examples of how they are used. For those interested in more thorough command of SolidWorks beyond the introductory level, the website provides additional instruction and references.

Prerequisites

To get the maximum benefit from this book, the reader must be computer literate and familiar with the complementary roles of hardware and software. In addition, the reader must be familiar with the Windows operating system. SolidWorks was one of the earliest CAD programs written specifically to run within Windows and takes full advantage of the Windows interface, and file naming and handling conventions.

How to Use this Book

This book has two goals: first, it is a tutorial that teaches how to use SolidWorks at an introductory level, and second, the book also teaches how engineering drafting conventions and standards are used to communicate design ideas. The main text teaches the commands needed to create a solid model of a skateboard and then to use the model to produce working drawings. If the reader follows only the main text, the book is a step-by-step tutorial to gain an introductory knowledge of SolidWorks and of its most useful commands. To get the best results, read the step-by-step tutorial and practice with your SolidWorks software and computer as you read. The CD available in our website contains an audiovisual tutorial that follows the main text and is intended as an alternative for readers who prefer to see a demonstration instead of following written instructions. The CD can be ordered from the website www.cadeducators.com.

We also recommend the use of the SolidProfessor video course as an optional way of studying SolidWorks in depth and for updating skills annually when a new version is released. You can find additional information at www.solidprofessor.com. Each Lesson has a list of the relevant topics in SolidProfessor. It is recommended that you watch the SolidProfessor topics listed before or after the lesson to maximize your learning.

The information that is not directly related to the skateboard model is included in the book's sidebars and in the supporting website. Each sidebar is self-contained and should be read independently from the main text. The sidebars will:

1) explain the use of SolidWorks commands and command options in detail, or
2) introduce and explain drafting conventions and standards, or
3) explain the role of CAD in the design process.

All the sidebars in a lesson should be read before moving to the next lesson.

The website is an integral part of the book. It has information about annual enhancements to the software and advanced topics beyond the introductory level. If you find that a figure in the book does not match your version of SolidWorks, visit the website to see what has changed.

Finally, we recommend that you subscribe to SolidProfessor while you use this book to learn SolidWorks. This is optional, but the information provided by SolidProfessor is an excellent reinforcement to the material learned in the book. SolidProfessor also covers advanced topics not included in the book or in the website. You can register to use SolidProfessor in our website www.cadeducators.com or in the SoildProfessor website http://www.solidprofessor.com/. Each Lesson in this book has a list of the useful topics in SolidProfessor that you can use to maximize your learning.

Together, the written and CD tutorials, the sidebars, the website and the SolidProfessor lessons recommended, will accommodate different learning styles. If a subject must be studied in greater depth than it is covered in the book or the website, consult the references at the end of each lesson or the SolidProfessor curriculum.

This book is divided into sections and each Section is further divided into Lessons. Every Lesson has practice exercises and questions. The practice exercises give students the opportunity to use the SolidWorks commands learned in the Lesson in new situations. For example, although English units are used in the text, some practice exercises use metric units instead. Questions are used to reinforce and expand the student's knowledge of design and drafting practices. It is important to complete the questions and practice exercises to achieve the most learning possible.

The Appendix includes two complete sets of working drawings. One set uses the ANSI standard and English units (inch and pound) and the other set uses the ISO standard and metric units (millimeter and gram). The drawings are to be used by the student to determine the dimensions needed to create the solid models, assemblies and drawings. Because the student is expected to find in the drawings the dimensions needed to create the models, the text rarely reveals them after the first few Lessons. The benefit of this approach is that the student becomes familiar with drawings and the information they contain. Classroom experience has shown that students will be comfortable with this approach.

The Appendix also has a list of all the commands on the Main Drop-down Menu and in the CommandManager. To find additional information about each command and how it is used, the reader can use SolidWorks Help or the alphabetical index at the end of the book.

Conventions

- Click is used when pressing the left-button on the mouse.
- Right-click is used when pressing the right-hand-button.
- Double-click is used when pressing the left-button of the mouse two times quickly.
- Right-drag is used to initiate mouse gestures. In the graphics area, press the right button and drag. The tool highlighted is the one in the drag direction.
- Click, hold and drag are the sequences used for moving toolbars or icons from one location to another.
- Help➔SolidWorks Help is the sequence of clicks to achieve an action.
- **Bold** letters are used to indicate Windows commands.
- ***Italic Bold*** letters are used to indicate SolidWorks commands or input request.
- Steps are numbered sequentially from the beginning to the end of the book. This is to facilitate communication between the student and the teacher. The numbered steps also permit the combination of two books into one document. The SolidWorks tutorial is the sequence of steps and can be followed without interference from the drafting practices lessons in the sidebars. The numbered steps teach SolidWorks and the sidebars teach drafting.

Acknowledgements

Publishing a book requires the effort of many contributors. Although the author is given the majority of the credit, all authors know that the support team can make or break the project, irrespective of his talent and effort. In my case, I have many to thank for their help in bringing this book to life. At the risk of leaving many unnamed, I feel compelled to name Ms. Victoria MacKinnon for painstakingly editing the original manuscript and crafting the format. I also must name Mr. William Toft for his editing of the technical content and the creation of the video that explains how to model the skateboard.

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Section I – Preliminaries

Lesson 1 – Introducing SolidWorks

1.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Explain how to start SolidWorks.
- Explain the SolidWorks Interface.
- Explain how SolidWorks files are named.
- Explain the difference between a part and an assembly of parts.
- Explain the difference between a part drawing and an assembly drawing.
- Explain concurrent engineering.

1.2 Introduction

SolidWorks is a modern computer aided design (CAD) program. It enables designers to create a mathematically correct solid model of an object that can be stored in a database. When the mathematical model of a part or assembly is associated with the properties of the materials used, we get a solid model that can be used to simulate and predict the behavior of the part or model with finite element and other simulation software. The same solid model can be used to manufacture the object and also contains the information necessary to inspect and assemble the product. The marketing organization can produce sales brochures and videos that introduce the product to potential customers. SolidWorks and similar CAD programs have made possible concurrent engineering, where all the groups that contribute to the product development process can share information on real-time.

Concurrent Engineering

The practice of sharing the solid model throughout the organization is called concurrent engineering. It can reduce the time it takes to develop a new product. Previous to concurrent engineering, the design group had to complete the design before the manufacturing organization decided how to make each part. For example, if a part is injection molded, the manufacturing organization must design the mold. With concurrent engineering, manufacturing personnel does not have to wait for the drawings of the part to be complete and can use the solid model to design the mold in parallel. If the design of the part changes, the new dimensions are available to the manufacturing organization in real-time and the mold design always reflect the latest information. In addition to reducing the product development time, this early involvement of the manufacturing organization promotes better communication that reduces errors and improves product quality.

The ultimate expression of concurrent is the integrated product team (IPT). The team includes representation from all the company stakeholders including the design and manufacturing organizations, the field maintenance organization, and the marketing, accounting and legal departments.

1.3 Starting SolidWorks

Figure 1.1 – SolidWorks Icon

Step 1: To start SolidWorks, click on the SolidWorks icon shown in Figure 1.1. It should be on your desktop. If the SolidWorks icon is not on your desktop, you can start the program from the start menu by clicking ***Start→All Programs→SolidWorks→SolidWorks***. You can also use Windows Explorer or “Windows Search” to find the file **sldworks.exe** in the directory ***C:\Program Files\SolidWorks\SolidWorks***. Double-click to start the program.

After you start SolidWorks you will see a familiar Windows desktop. Notice, in Figure 1.2 that:

1. The SolidWorks banner is at the top-left corner.
2. The Windows **Main Drop-down Menu** is adjacent to the banner. It includes the Windows drop-down menus **File**, **View**, **Tools** and **Help**.
3. Other menus will appear when needed, depending on what you are doing.

SolidWorks and Windows

SolidWorks is fully integrated into Windows. The look and feel of the SolidWorks desktop is similar to the Windows desktop. SolidWorks Part files are stored as *.sldprt, where * is the filename. Assembly files are stored as *.sldasm and Drawing files as *.slddrw. It is also possible to save templates for parts, assemblies and drawings as *.prtdot, *.asmdot and *.drwdot. Templates customize how SolidWorks looks and works.

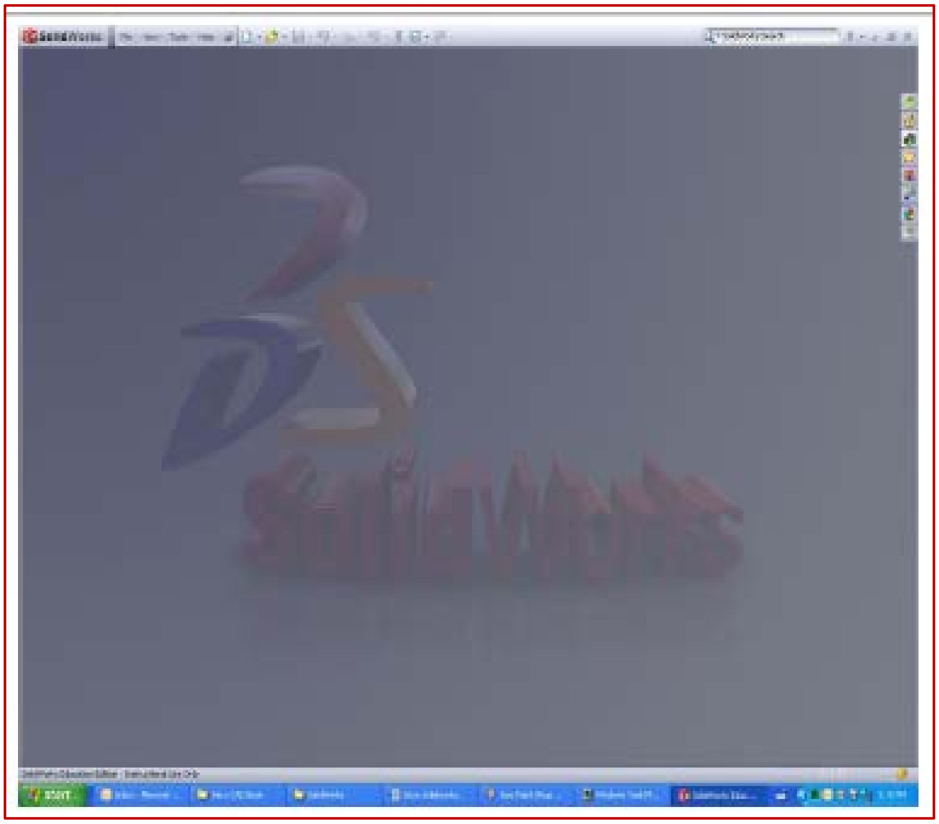


Figure 1.2 – SolidWorks Desktop

If you do not see the **Main Drop-down Menu**, move your mouse cursor over the SolidWorks banner to show it. To keep it visible, push on the pin.

When the pin is horizontal, you can toggle between the **Main Drop-down Menu** and the ***Standard*** toolbar (with icons) immediately to the right of the **Main Drop-down Menu** by moving the mouse cursor over the banner.

Finally, if you wish to see the content of a drop-down menu, click on its label.

4. The ***Standard*** toolbar, includes the icons for commonly used Windows and SolidWorks commands; for example **New**, **Open**, **Save** and **Print** a document, ***Rebuild*** your model and ***Undo*** and ***Redo***. The content of the toolbar can vary because it is possible to customize it. We will learn to customize toolbars in Lesson 2.

   Notice that if you place the mouse cursor over one of the icons, you can see what command it represents and also the command's keyboard-shortcut equivalent.

5. On the bottom of the screen you can see the Windows **Start Button** and **Taskbar**.

6. On the right side of the screen you can find the ***Task Pane***. It has icons for ***SolidWorks Resources***, ***Design Library*** and for the ***SolidWorks File Explorer***, which works like Windows Explorer.

   The ***Task Pane*** can be shown by clicking on one of the icons or hidden by clicking on the Graphics Area. The pin can be pushed to keep the ***Task Pane*** open and it must be horizontal to allow to toggle between visible and hidden.

   Next, explore the ***TaskPane*** and its contents. The ***SolidWorks Resources*** include the ***Online Tutorial*** and ***What's New***, as well as access to the ***SolidWorks Forum***. In ***Design Library*** you can find ***Toolbox*** and ***3DContentCentral***. Both, ***Toolbox*** and ***3DContentCentral*** have models of parts that can be re-used in new assemblies. This is very convenient because screws, bearings, linear actuators and other catalog parts don't have to be re-drawn. ***Toolbox*** resides on your desktop but it is an extra cost item and could be missing in your installation. ***3DContentCentral*** requires internet access.

Step 2: To open a new document, click on the icon **New** in the **Standard toolbar** (or click **File→New** in the **Main Drop-down Menu**).

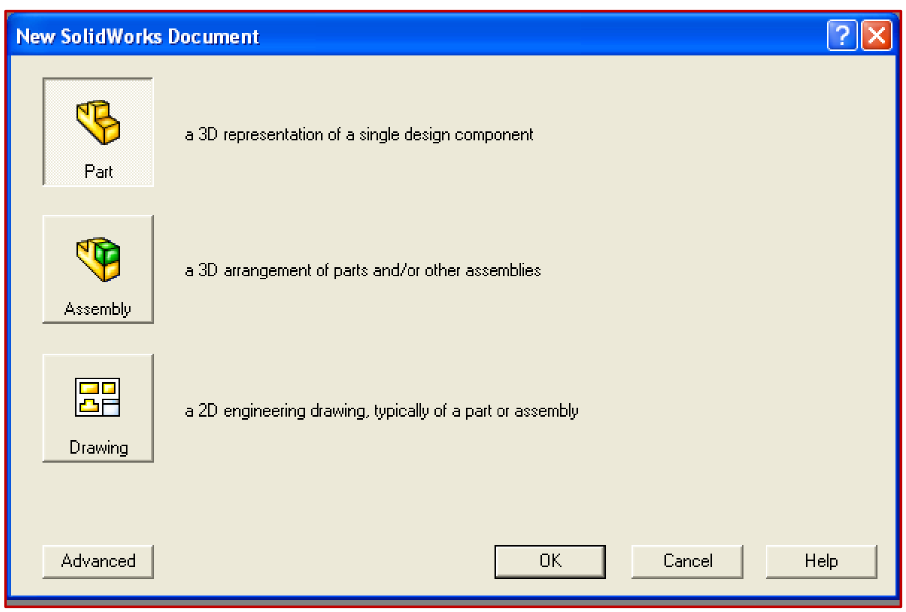


Figure 1.3a – Novice Screen for a New Document

To open an existing document, you must click the **Open** icon in the **Standard toolbar** (or **File→New** in the **Main Drop-down Menu**).

When we create a new document, we get either the novice or the expert screen in Figures 1.3a and 1.3b. It is possible to toggle between these options by clicking on the ***Advanced*** or ***Novice*** buttons at the lower left corner of the menu box. The advantage of the ***Advanced*** menu is that you can select from the customized templates that we will create in Lesson 2.

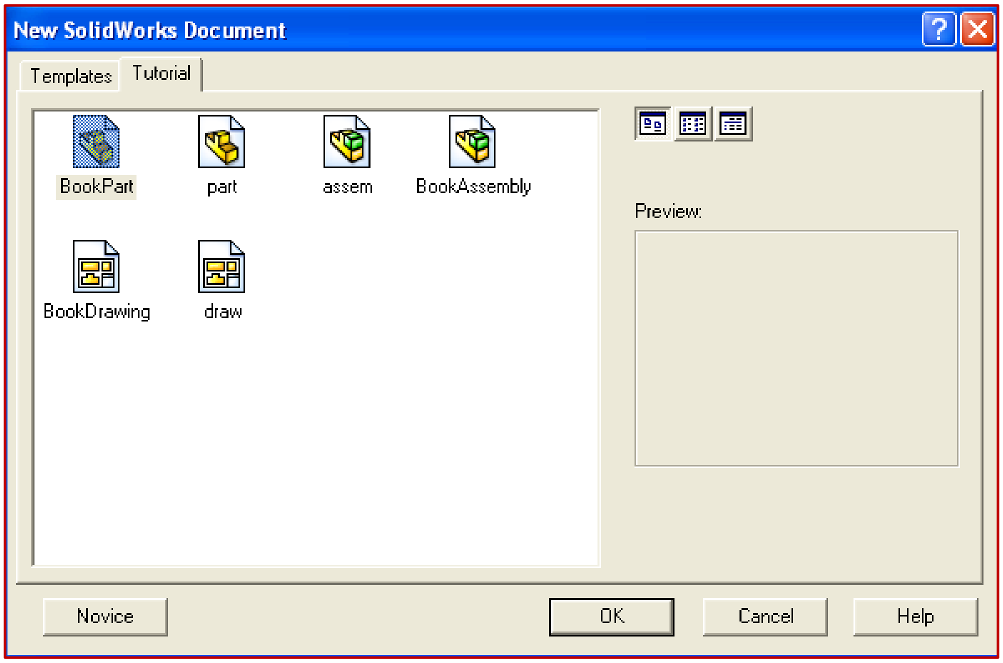


Figure 1.3b – Advanced Screen for a New Document

Notice that there are three kinds of documents in SolidWorks:

(1) parts,

(2) assemblies of parts, and

(3) drawings of parts or assemblies.

We will study them in that order in future lessons.

1.4 The SolidWorks Interface

Step 3: When you open a new ***Part*** document you get additional toolbars and pull-down menus as shown in Figure 1.4.

✓ Toolbars and commands are context sensitive. The commands that are available and related to what the user is doing are in color and the commands that are not available are shown gray.

A new toolbar is the ***View Heads→Up*** toolbar, at the top-center of the ***Graphics Area***. It can be customized, as we shall see later. The question mark at the lower-right corner can be clicked to get help.

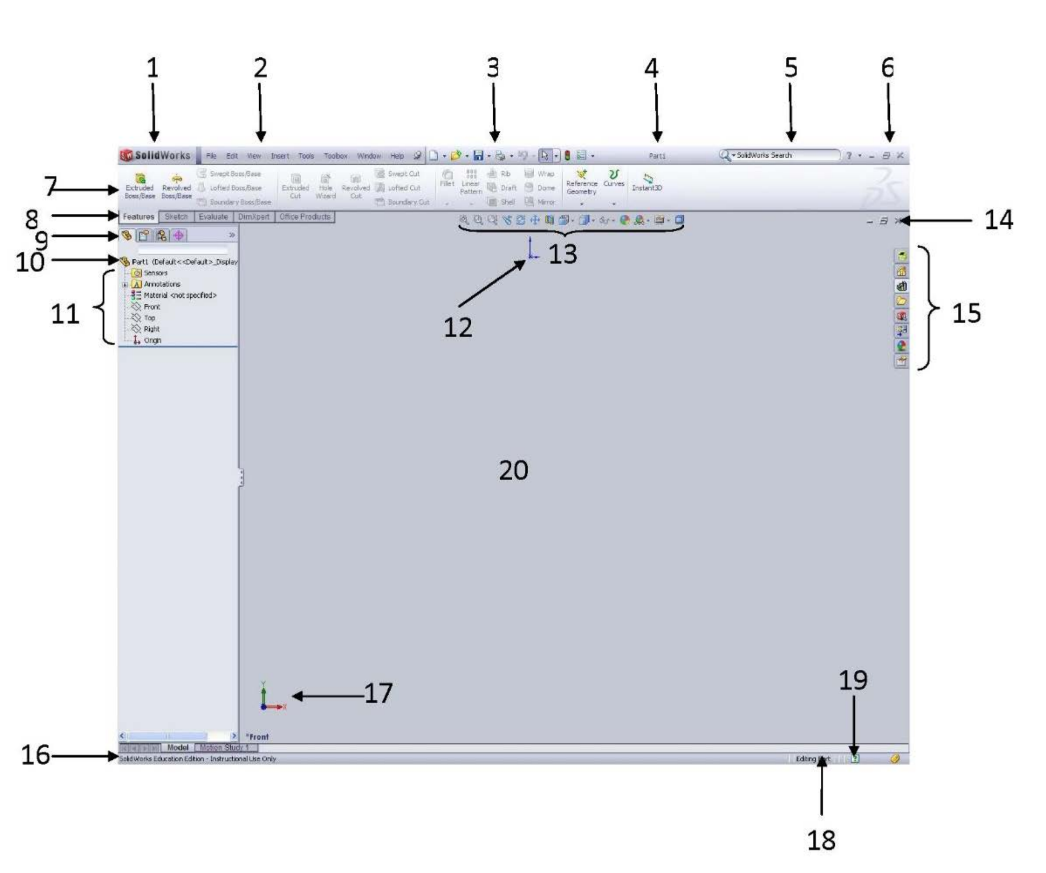


Figure 1.4 – SolidWorks Desktop for a New Part Document

Table 1.1 – SolidWorks Desktop for a New Part Document

1-	SolidWorks logo	11-	FeatureManager Design Tree (also called the Feature Tree)
2-	Main Drop-down Menu	12-	Origin (red or blue)
3-	Quick Access Toolbar (also called the Standard Toolbar)	13-	View (Heads-Up) toolbar
4-	File name	14-	Minimize/Maximize/Close window (see also 6)
5-	SolidWorks search	15-	Task pane
6-	Help (?) & Minimize/Maximize/Close window (see also 14)	16-	SolidWorks command description
7-	CommandManager	17-	Reference triad
8-	CommandManager toolbars tabs	18-	SolidWorks status bar & units selection
9-	Feature/Property/Configuration managers tabs	19-	Quick tips
10-	File name	20-	Graphics area

Parts, Assemblies and Drawings

An assembly is a group of parts that together make the complete product or a portion of the product. For example, an automobile is an assembly of parts, but for convenience, it can be subdivided into a frame sub-assembly, a powertrain sub-assembly, the air conditioning sub-assembly, etc. The powertrain sub-assembly can be further subdivided into the engine sub-assembly, the transmission sub-assembly, the wheel-tire sub-assembly, etc.

Parts and assemblies are a convenient way to subdivide a product because they allow division of labor and specialization. If production resources, including employees, equipment and facilities, are dedicated exclusively to producing only one or a few parts or assemblies they can achieve efficiencies that are not possible otherwise. Specialization is the basis of mass production. The engine block, for example, is usually a casting. The engine camshaft is machined from a steel bar and the pistons can be cast or forged. Manufacturing each part requires different machinery and expertise. A typical automobile today has thousands of parts and dozens of sub-assemblies produced in manufacturing facilities located all over the world.

Designers and design engineers create detail or part drawings primarily to enable the manufacturing of parts by different people in different locations. This requires that each detail or part drawing have all the information needed to make the part, and nothing extraneous that can cause confusion. For example, the part drawing for a camshaft must have one or more pictorial views of the part, derived from the solid model, and additional information such as dimensions and tolerances, material, finish, etc. On the other hand, there is no need for information about other engine parts such as the valves. Thus, each of the camshaft detail drawing and the valve detail drawing will have all the information needed to make each part but no information about how they fit together. The information needed to assemble the camshaft and the valve together belongs on the engine sub-assembly drawing.

The first step when modeling a product is to break down the assembly into sub-assemblies and parts. This is a very important step that will impact the quality and cost of the final product and the complexity of the model. Although there are exceptions to the rules, the following are important considerations when deciding what should be a part:

1. Components made of different materials should be different parts. For example, a part should not be made of sheet metal with rubber bumper(s). Instead, create a sub-assembly that contains sheet metal part(s) and rubber part(s).
2. Components made with different manufacturing processes should be different parts. For example, a typical steel file drawer has many sheet metal and machined parts assembled into drawers, cabinets, etc. This is also true for the finishing process. It is not practical to paint a portion of a part and chrome-plate another portion. It is more appropriate to have an assembly that contain painted parts and plated parts.
3. Weldments are different. A Weldment is one part, even though it is made from many separate pieces. It should be drawn with all the final dimensions and tolerances after all the pieces are welded together. On the other hand, a book case is typically drawn as individual detail drawings for each shelf, plus an assembly drawing. The choice can be based on the degree of control available. A weldment is typically handmade and a book case is made with woodworking machinery.
4. Use sub-assemblies when the parts complement to enable a function. For example, the wheel and tire together will enable the automobile to roll over the pavement.

In addition to the detail or part drawings used to make the parts, the design organization will create assembly drawings to provide information about how the parts assemble together. The information in an assembly drawing is different than the information in a detail drawing because they have different purposes. The detail drawing is used to make the parts and the assembly drawing is used to assemble the parts into a product.

Practice Exercises

1. Click ***Help→SolidWorks→Help*** on the **Main Drop-down Menu** and study the section ***User Interface***.
2. Click ***Help→SolidWorks→Help*** on the **Main Drop-down Menu** and use ***Search*** to find information about the ***CommandManager***.
3. Click ***Help→SolidWorks→Tutorials*** on the **Main Drop-down Menu** and find the Tutorial Lesson 1. Complete the exercise. (Caution: The units in the Tutorial are mm. A common error by beginners is to draw with the wrong units. The result will look identical in the monitor screen, but a part drawn in mm will be smaller than the same part drawn in inches.)
4. On the ***Task Pane***, click ***SolidWorks Resources*** click to visit the ***SolidWorks Discussion Forum***. (You have to register to gain access.) Follow one thread and write a paragraph describing the discussion.
5. On the ***Task Pane***, find ***SolidWorks Resources*** and locate the most convenient SolidWorks user group that you can visit.
6. On the Task Pane, open the ***Design Library*** and click to visit ***3D Content Central***. Download one file and open it from within SolidWorks. (Note: You have to register or log-in before you can download files.)

Questions

1. You will need a partner to do this exercise. First, find the drawings of the deck in the Appendix. Describe the deck to your partner in words. Your partner will draw the part while you describe it. You can answer the questions that your partner asks for clarification and provide unrequested hints, but do not reveal the drawing or look at your partner's representation of the part. When the drawing is complete, answer the following questions:
    a. How accurate is your partner's final representation of the part? Is the result more accurate if both of you know about skateboarding and have the same mental image of the deck?
    b. How easy (or difficult) was it to communicate the information needed to draw the part? How long did it take?
2. Research the origins and history of skateboarding and skateboards.
3. Research the history of computers and CAD.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

High Tech Hot Shots: Careers in Sports Engineering – by C. Baine, The National Society of Professional Engineers, Alexandria, VA, US

Engineering a Totally Rad Skateboard with Max Axiom, Super Scientist – by T. Enz, Capstone Press, North Mankato, MN, US

Videos from SolidProfessor SolidWorks for Beginners

- About This Course
- What is SolidWorks?
- Interface Tour

Videos from SolidProfessor 3D Skills

- The Sketcher

Internet Resources

CADeducators.com website: http://www.cadeducators.com

CAD History: http://www.cadhistory.net/

CAD news and information: http://www.caddigest.com/subjects/solidworks/index.htm

Skateboarding history: http://en.wikipedia.org/wiki/Skateboarding

Skateboard design: http://en.wikipedia.org/wiki/Skateboard

Skateboarding news: http://skateboard.about.com/

Skateboarding science: http://exploratorium.edu/skateboarding/trick02.html

Lesson 2 – Customizing SolidWorks

2.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Explain how to customize SolidWorks using ***View➔Toolbars➔Customize*** and ***Tools➔Customize*** in the **Main Drop-down Menu** or ***Options➔Customize*** in the **Quick Access Toolbar**.
- Explain how to customize SolidWorks using ***Tools➔Options*** in the **Main Drop-down Menu** or ***Options*** in the **Quick Access toolbar**.
- Explain how to customize SolidWorks by creating a part template.
- Explain how to customize SolidWorks using ***Tools➔Add-Ins*** in the **Main Drop-down Menu** or ***Options➔Add-Ins*** in the **Quick Access toolbar**.
- Explain where to find SolidWorks files.

2.2 Introduction

SolidWorks can be customized to look and work according to your preferences, or to follow company standards. In this lesson, we will customize SolidWorks to reflect customary US practice.

2.3 Customizing the Toolbars

Step 4: Click ***View➔Toolbars*** to see all the toolbars that are available. Toolbars that are visible have a check mark. See Figure 2.1.

- ✓ Verify that the following toolbars are visible: ***CommandManager, View (Heads-Up),*** and ***Task Pane.***
- ✓ Check and uncheck one or more toolbars and see what difference it makes, but remember to undo your changes before proceeding to the next step.
- ✓ All toolbars can be moved to a new location by clicking and dragging. The ***CommandManager*** can be moved by clicking and dragging to the

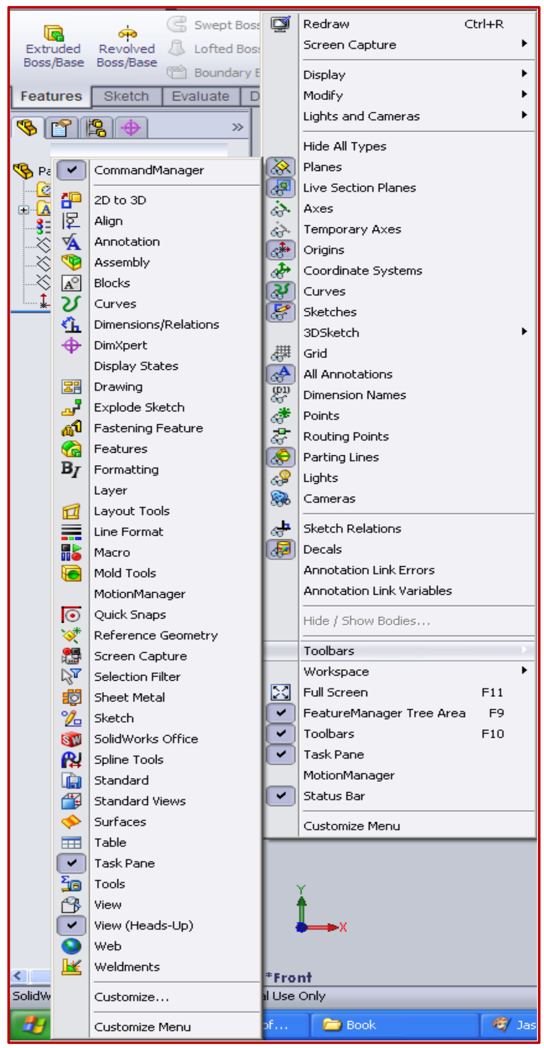


Figure 2.1 – *View➔Toolbars*

arrows on the center-top/right/left of the graphics area.

- ✓ If you click ***Options→Customize*** in the **Quick Access toolbar**, the menu in Figure 2.2 appears. It is another way of selecting which toolbars can be seen on the desktop. You can select which toolbars are shown by clicking to add the checkmark.
- ✓ Two other ways of getting the menu in Figure 2.2 are by clicking ***Tools→Customize*** in the **Main Drop-down Menu** or by clicking ***View→Toolbars→Customize.***

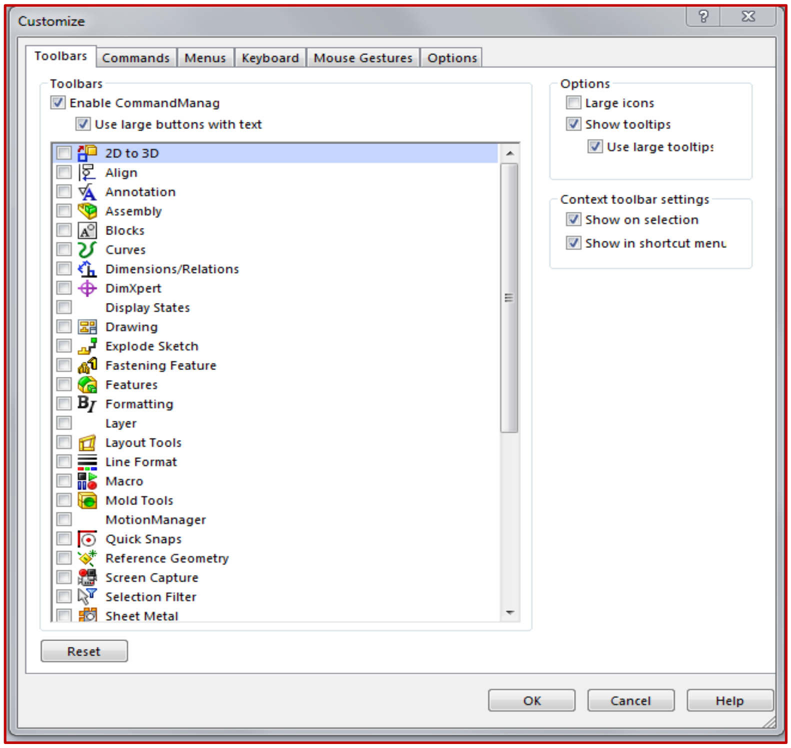


Figure 2.2 – *View→Toolbars→Customize*

Step 5: The commands in a toolbar can be changed. Click ***Options→Customize*** and then select the ***Commands*** tab to get the menu in Figure 2.3. This is a list of all the toolbars in SolidWorks. They are called ***Categories*** on this menu.

If you click a ***Category***, the icons shown on the right side of the menu box are the commands that belong in the toolbar. If you place the mouse cursor over each icon, you can see what command it represents. Some commands are not included in the default version of a toolbar, but can be added.

To add a command, click and drag the icon to a toolbar. It will remain in the toolbar when you click OK to accept. You can place any command in any toolbar, even if it is unrelated, but it is best to keep toolbars small and concise.

To remove a command from a toolbar, drag the icon from the toolbar back to the ***Customize*** menu.

Step 6: In the ***CommandManager*,** click the tab ***Sketch*** to show the sketch commands. Next, on ***Options→ Customize→Commands→Sketch*** in Figure 2.3, locate the ***Ellipse*** tool. Click and drag it to the ***Sketch*** toolbar in the ***CommandManager*** and click OK. This will add the new command.

- ✓ To remove the ***Ellipse*** icon from the ***CommandManager*** toolbar, click and drag the icon back and release the mouse button.

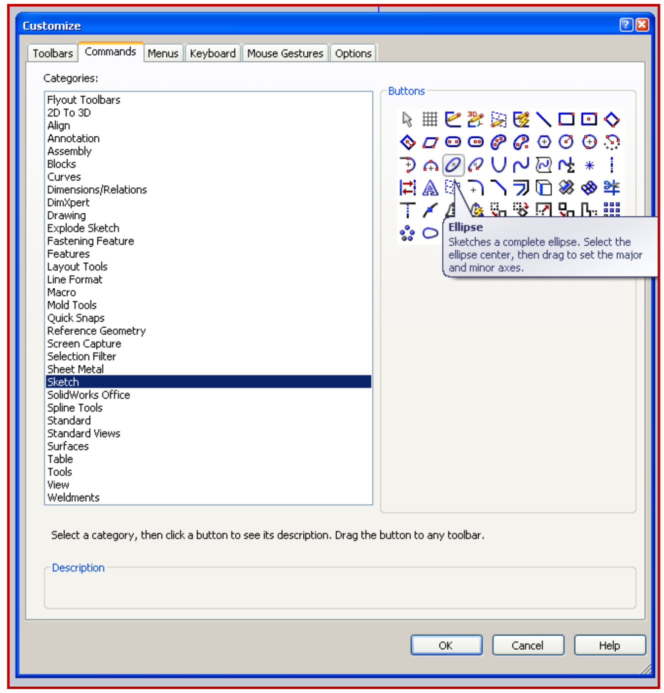


Figure 2.3 – *View→Toolbars→Customize→ Commands→Sketch→Ellipse*

2.4 Customizing with *Tools→Options*

Another way of customizing SolidWorks is by selecting ***Tools→Options*** in the **Main Drop-down Menu** (or ***Options*** in the **Quick Access toolbar**). The menu in Figure 2.4 will appear. It has two tabs, ***System Options*** and ***Document Properties***.

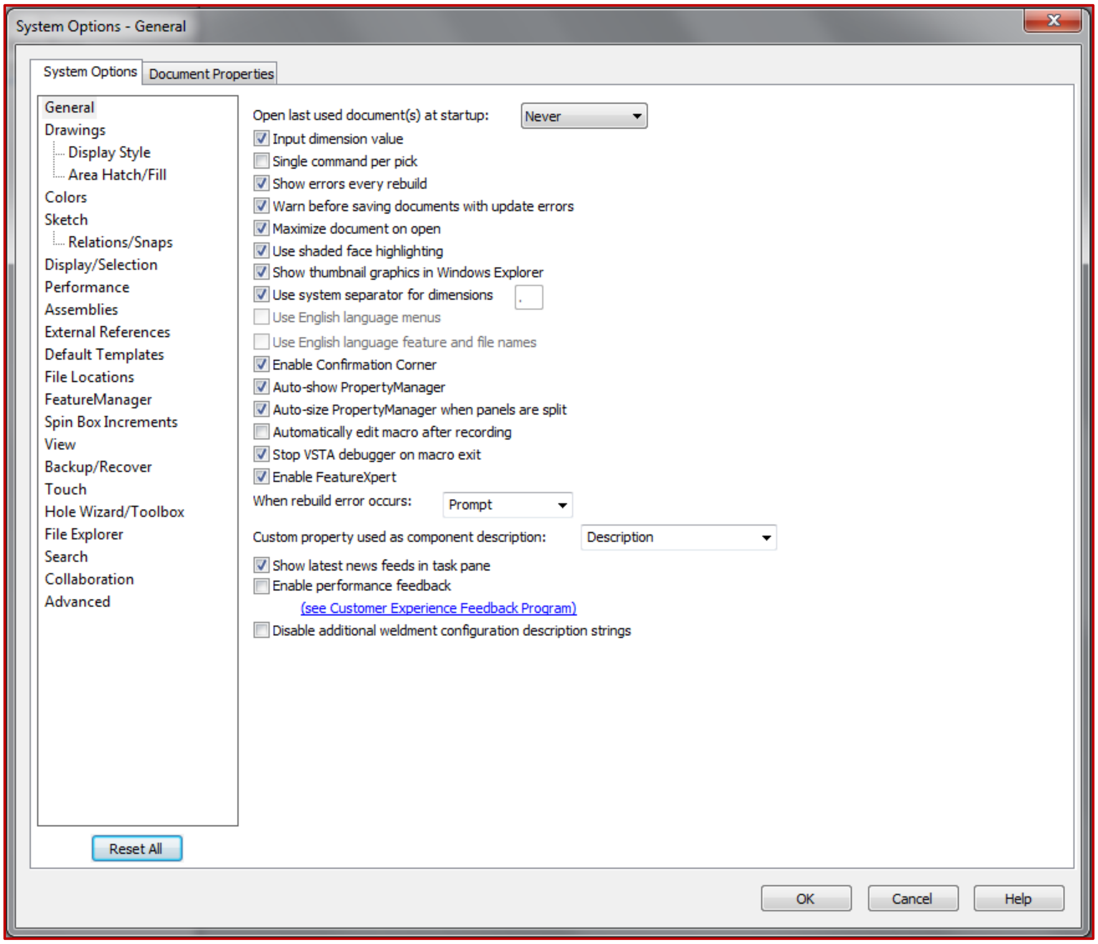


Figure 2.4 – *Tools→Options*

Step 7: System Options.

- ✓ Select ***Display Style*** on the ***System Options*** tab and verify that ***Tangent edges in new views***: **Removed**.
- ✓ Select ***Display/Selection*** on the ***System Options*** tab and verify that ***Projection Type for four viewport:*** is **Third Angle**. Also, ***Part/Assembly tangent edge display:*** **Removed**.
- ✓ Select ***FeatureManager*** and change ***Design Binder→Show*** and ***Origin→Show.***

Step 8: Document Properties.

- ✓ Click the ***Document Properties*** tab and select ***ANSI*** in ***Overall drafting standard***. See Figure 2.5.
- ✓ Click ***Units*** and select ***IPS*** (inch-pound-seconds).
- ✓ Click OK.

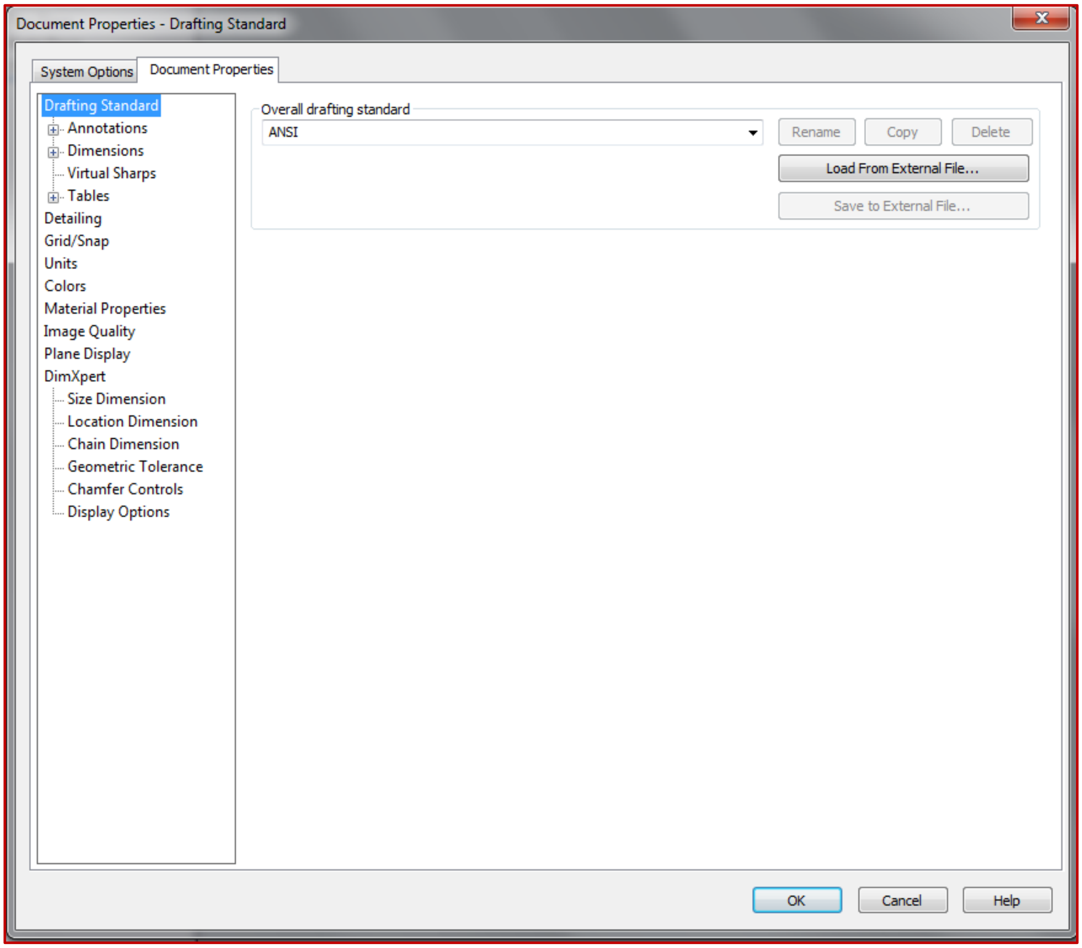


Figure 2.5 – *Tools→Options→Document Properties*

2.5 Custom Properties

Custom Properties allow storing information that will be used in drawings and in SolidWorks calculations. Click **File→Properties** in the **Main Pull-down menu** to get the screen in Figure 2.6. On the ***Summary*** tab, Figure 2.6a, we will fill as a minimum the ***Author*** and the ***Title***. On the ***Custom*** tab, Figure 2.6b, fill the first column (Property Name) of the first four lines by typing 1) Description, 2) Weight, 3) Material and 4) Finish. On the second column (Type) select Text from the pull-down menu for all the lines. Finally, on the third column (Value/Text Expression) leave 1) blank (to be filled later for each part/configuration). For line 2) select MASS from the pull-down menu on the right of the cell. For lines 3) and 4) leave blank.

You can add other properties depending on the needs of your team.

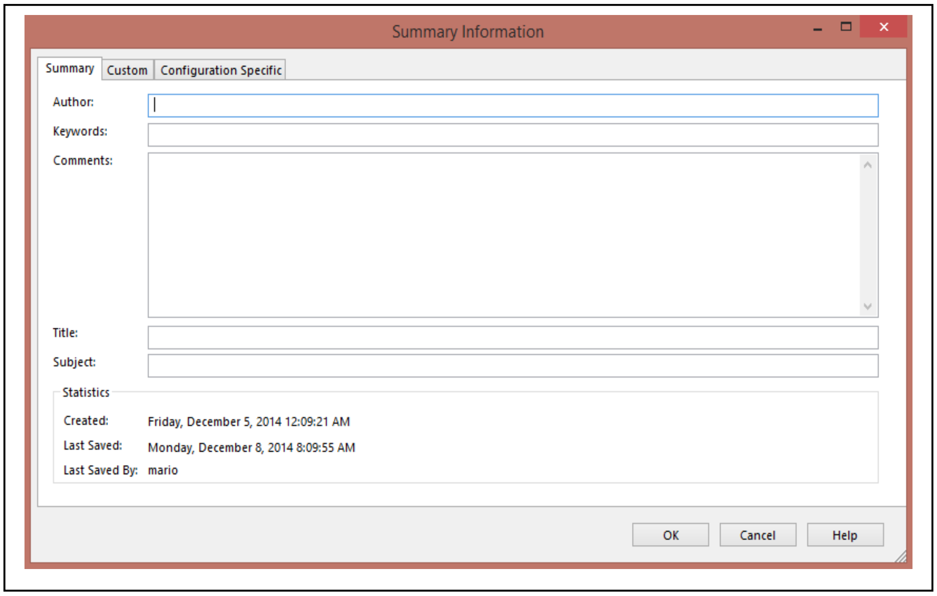


Figure 2.6a – File→Properties, Summary tab

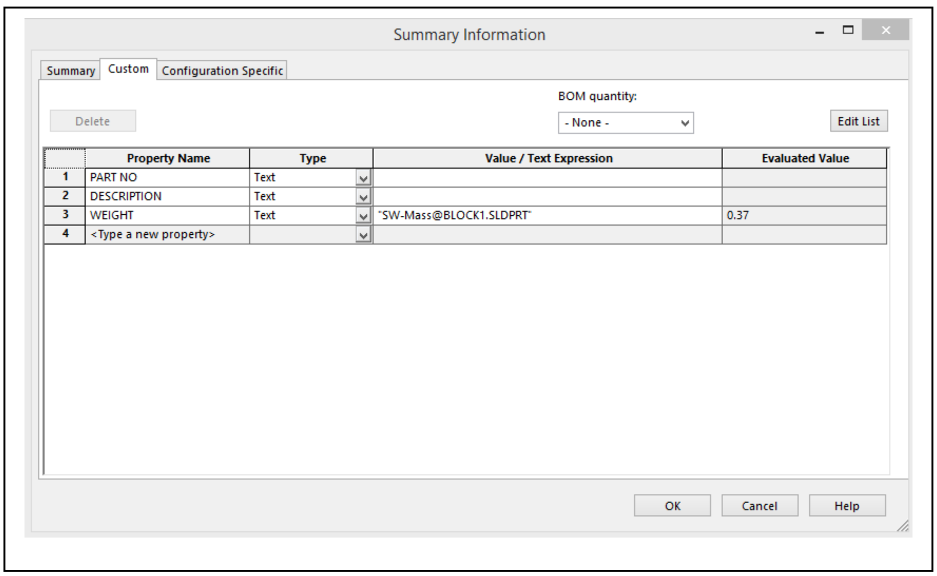


Figure 2.6b – File→Properties, Custom tab

2.6 Saving a Template

After you have customized SolidWorks you can save a template. The Template will bring your choice of toolbars and icons to every new document. Creating and using a template is the best way to insure that all the models, assemblies and drawings you create follow the drafting standard (e.g. ANSI or ISO) and units (e.g. inches or millimeters) that you want.

Step 9: Click ***File→Save As*** and select ***Part Template*** on the ***Save as type*** pull-down menu. See Figure 2.6

- ✓ Name the template **InchPart.prtdot** and save it for future use.
- ✓ After you save your template, close it before you draw and save a part. If you Fail to do this, SolidWorks will save your part as the template
- ✓ The template is stored in the computer. If you want to store the template in your removable or network storage, follow the instructions in **Step 10**.

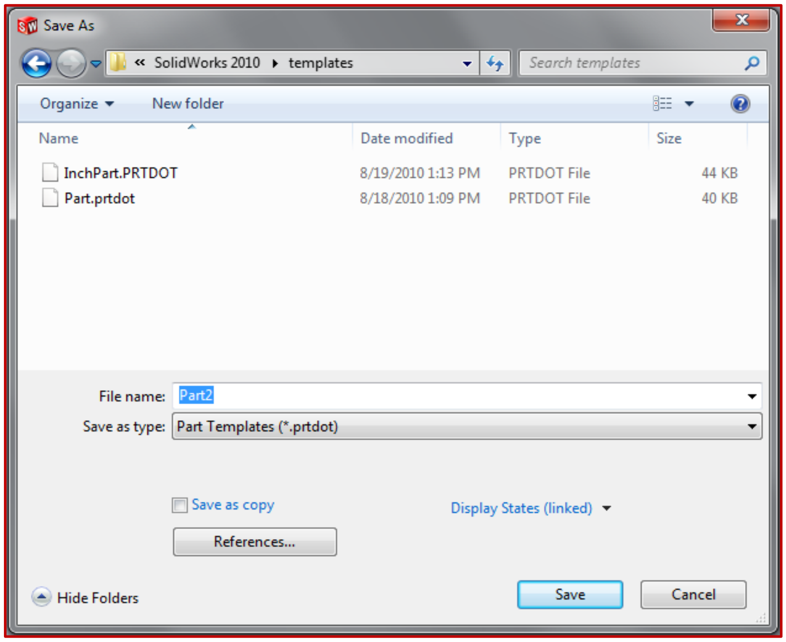


Figure 2.7 – Saving a Part Template

Step 10: If you wish to save your templates on removable or network storage or in a sub-directory in your computer, you must first tell the SolidWorks the name of your storage device and its location.

- ✓ First, save one or more templates to the location you wish to use.
- ✓ In the **Quick Access toolbar** click ***Options→ System Options→File Locations***. See Figure 2.7.
- ✓ In the Drop-down menu for ***Show folders for:*** select Document Templates.
- ✓ In ***Folders*** you will see the present location of the templates you see in Figure 1.3a and Figure 1.3b.
- ✓ Click ***Add*** and browse to your removable storage device or desired storage location (where you stored the templates earlier).
- ✓ Click OK and you will see the new location added.
- ✓ Click OK again and answer Yes to add the new location.
- ✓ Repeat **Step 9** to save one or more templates in the new location.
- ✓ Close your template document(s).
- ✓ Open a new part and select the ***Advanced*** option. You will notice a new tab for your new location. Note: the tab will not be visible if you did not save a template first.

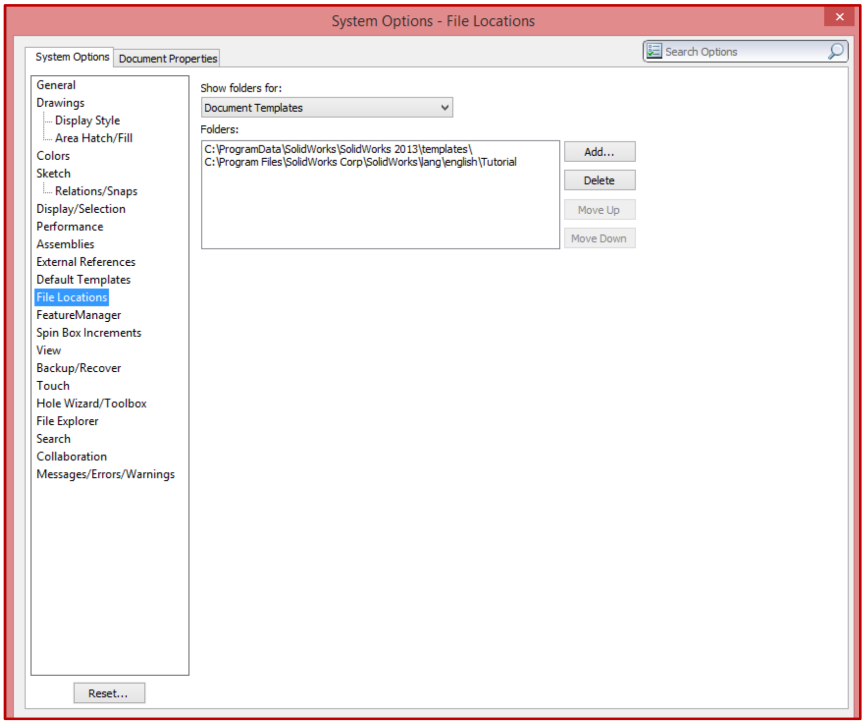


Figure 2.8 – SolidWorks File Location

2.7 Customizing with Add-Ins

Add-Ins provide additional features and capabilities to SolidWorks. Some Add-Ins cost extra and may not be available in your version of the software. They are not part of your template, but once they are enabled during the session they will remain active until the end of the session. Although Add-Ins are very useful, they require additional memory. They should be kept disabled until needed.

Step 11: Click on ***Tools→Add-Ins*** to see which ones are available in your version of SolidWorks. See Figure 2.8.

- ✓ If available, verify that the following ***Add-Ins*** have a check mark: ***SolidWorksToolbox*** *and* ***SolidWorksToolbox Browser***.
- ✓ You can add other ***Add-Ins***, but remember that they use memory and could slow your computer.

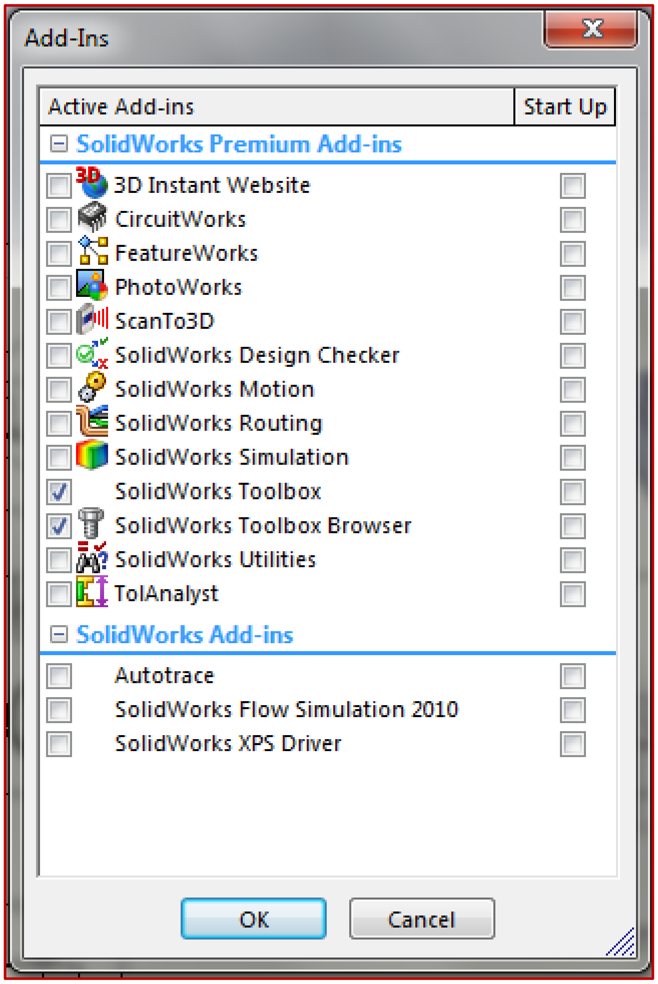


Figure 2.9 – *Add-Ins* Menu

2.8 Customizing with the View Menu

The ***View*** menu on the ***Main Pull-down Menu*** is shown in Figure 2.9 and in Appendix D3. The entities highlighted will show on your SolidWorks graphics area. If the Origins are not highlighted, they will not be visible. Of special interest are the ***Temporary Axes***. You will need them to use as axes of rotation in some commands. If you cannot find an axis to rotate about, click ***View*** and highlight ***Temporary Axes***.

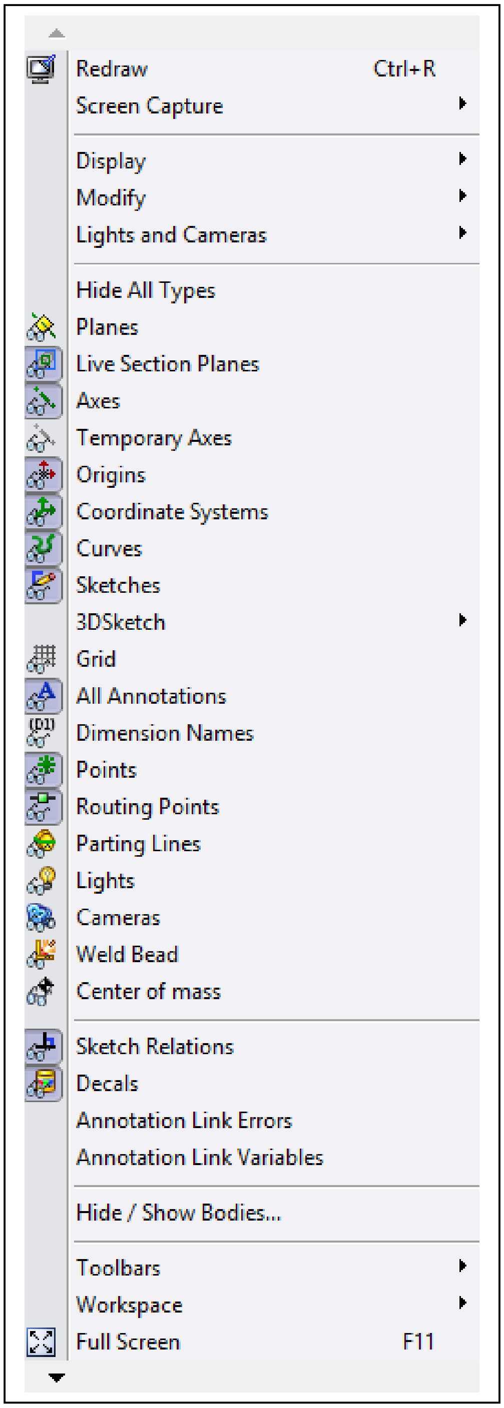


Figure 2.10 – View Pull-down Menu

Practice Exercises

1. Click ***Help→SolidWorks Tutorials*** on the **Main Drop-down Menu** and study what tutorials are available.
2. Add and then remove the command for a partial ellipse to the ***Sketch*** toolbar in the ***CommandManager***.
3. Create a part template for millimeter units. Set decimal places for length and for angles equal to zero. Select the ANSI standard. Call the template **mmPart.prtdot**.
4. Use **File Explorer** to find the template that you created in **Step 11**.
5. Save your Deck file in PDF format.

Questions

1. Search the Internet to find information about the following document formats supported by SolidWorks. Write a brief description of each one, including advantages, disadvantages and when you would use them. (Hint: You can search the internet using Google)

ACIS	TIFF	VRML
STEP	JPEG	PDF
STL	IGES	Parasolid

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor SolidWorks for Beginners

- View Manipulation
- Starting a Part
- Starting a Sketch
- Sketch Tools

Videos from SolidProfessor Core Concepts for Parts and Assemblies

- Picking a plane
- Design Intent
- Custom Properties

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks website: http://www.solidworks.com

SolidWorks models for download: http://www.3dcontentcentral.com/3dcontentcentral/

Popular CAD models for download: http://grabcad.com/library/software/solidworks

SolidWorks training files: http://www.solidworks.com/sw/support/807_ENU_HTML.htm

Section II – Modeling Simple Parts

Lesson 3 – Modeling the Skateboard Deck Using the Command *Extruded Boss/Base*

3.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Model simple parts in SolidWorks.
- Explain and use the ***CommandManager*** toolbar.
- Explain and use the ***FeatureManager design tree***.
- Explain and use the ***PropertyManager*** menu.
- Explain and use the ***Fillet*** and ***Chamfer*** commands in the ***Features*** tab of the ***CommandManager***.
- Explain and use the command ***Smart dimensions*** in the ***Sketch*** tab of the ***CommandManager*** or ***Tools→Dimensions→Smart*** in the **Main Drop-down Menu**.
- Explain and use the command ***Add Relations*** in the ***Display/Delete Relations*** pull-down menu of the ***Sketch*** tab of the ***CommandManager*** or in ***Tools→Relations→Add*** in the **Main Drop-down Menu**.
- Learn to use the ***Hole Wizard***.
- Learn to constrain a model by using ***Dimensions*** and ***Sketch Relations***.
- Explain the difference between ***Sketch*** and ***Feature***.
- Explain and use ***Inferencing***.

3.2 Introduction

When you open the template for a part, all the commands needed to create a solid model become available. The ***CommandManager*** toolbar will show tabs for the ***Sketch*** toolbar and the ***Features*** toolbar. As the name implies, the first can be used like a pencil and paper to create a 2D cross-section of the part. The second has the commands needed to extrude the 2D cross-section into a 3D solid.

Many parts can be modeled by following five modeling steps:

Modeling 1 – Select a sketch plane.

✓ A common error made by beginners learning SolidWorks is forgetting to select the sketch plane. Forgetting to select the sketch plane can:

1) create a sketch in the wrong plane, or

2) merge the new sketch with the previous sketch, or
3) result in an error message.

If you find yourself in one of these three these situations, use the Esc key to exit, and try again.

Note: Sketch based features (e.g. ***Extrude Boss/Base*** and ***Revolved Boss/Base***) can be created without selecting a plane first and while changing the order of the steps. After you become more proficient using SolidWorks you may choose to use these commands as sketch based features, but early in your learning it is advantageous to use the five steps as a standard procedure that guide you and prevent you from getting lost in the software.

Modeling 2 – Sketch a cross-section of the part.

- ✓ At this stage of the sketch, exact dimensions are not important. They will be added later.
- ✓ It is helpful to use ***Inferencing***. It makes the SolidWorks intelligent. ***Inferencing*** can guess the user's intent and will suggest (or assume) appropriate relations between sketch entities. Some examples of relations are parallel, perpendicular, midpoint, etc.
- ✓ One example of ***Inferencing*** is the dashed line that appears when the cursor is aligned with the origin in Figure 3.1.
- ✓ For practice, use ***Inferencing*** to draw a horizontal line across the origin. 1)Click the ***Line*** tool in the ***CommandManager*** in the ***Sketch tab***. 2)Hover the pencil cursor at a location that gives you the ***Inferencing*** line. 3)Click and drag across the origin and release to get the line. 4)Explore what happens when you slowly move the cursor up and down before releasing the left mouse button.
- ✓ Even the most complex cross-section can be sketched. Maintain, as best as possible, the original proportions and relations (i.e. parallel, perpendicular, etc.). Be careful of unintended relations due to ***Inferencing.*** If necessary, break a complicated sketch into multiple simpler sketches. If the sketch becomes something you do not want, use ***Undo*** to step back and try something else.

Sketch Planes and View Manipulation

To sketch geometry in SolidWorks you must select a sketch plane first. SolidWorks gives you three choices: the Front, Top and Right planes at the beginning of the FeatureManager design tree (See Figure 1.4). I a latter Lesson we will learn that we can also sketch on any existing part surface that we created earlier.

It is important to select a plane that will simplify creating the assembly. For example, if you look at drawing SKBD100 in Appendix A, you can see multiple views of the skateboard assembly. There are four views in total. The view on the right is called the isometric view because the orientation of the part to each rectangular axis (x, y, z) is the same, 30 degrees. The isometric view is sometimes the preferred view for making hand sketches because it shows parts and assemblies the way we see them from a distance.

The three views on the left are called orthogonal views because they are rotated 90 degrees from each other. The view in the middle is called the Front view. The upper view is called the Top view and the lower view is called the Bottom view. They represent how the skateboard looks if we look from the top, edgewise and from the bottom.

Notice that when sketching the line that will become the deck, we chose to sketch in the Front plane. If we had sketched in the right plane, the top of the skateboard would be in the Top view. In the Top view of the orthogonal projection, the skateboard would look standing up on its edge. This would look unstable when compared to the horizontal length that we have. It takes a little planning before we pick a sketch plane and start sketching. The best approach is to sketch an isometric view of the part or assembly first, then plan how you go from sketch to solid, and then decide which one is the Front plane, the Top plane and the Right plane. Alternatively, another way of selecting a sketch plane is to draw a part the way it looks in the assembly. If you can visualize or see the part in the assembly, you should draw accordingly. For example, the front view of the wheel in drawing SKBD121 looks identical to the wheel in the assembly SKBD100.

The angle at which we look at the part or assembly is called the view orientation. During the process of creating a part or constructing an assembly we need to change the view orientation to see the part from different angles. There are many tools in SolidWorks to do that. The easiest tool to use is the space bar in your keyboard. If you tap on the space bar, you will get the orientation menu and the ***View Selector cube***. The ***View Selector cube*** is explained in Appendix I. The orientation menu can also be seen if we click on the icon ***View Orientation*** on the ***View (Heads Up) toolbar*** (see Figure 1.4).

When sketching, it is good practice to sketch when we can see the sketching plane in full size. This can be accomplished by clicking to make the sketch plane active, and then clicking ***View Orientation→Normal To*** in the ***View (Heads Up) toolbar***.

Another way to change the view orientation is to use your mouse. If you press on the mouse middle button, and move the mouse, you will make your part rotate. Sometimes it is difficult to get the part to look the way you want, but it is easier if you move in one direction at a time, first horizontally and then vertically. You can also rotate and translate the part without rotation if you click on the appropriate icons in the ***View (Heads Up) toolbar***.

Zooming tools include ***Zoom to Fit***, which will make the part or assembly occupy all the screen. It can also be used to bring back the model if you lost it. It can be found in the ***View (Heads Up) toolbar*** or by taping the key F in your keyboard. ***Zoom to Area*** is also in the ***View (Heads Up) toolbar*** and requires that we first click the icon, then press and hold the left button of the mouse, move to highlight the area we want, and release the button.

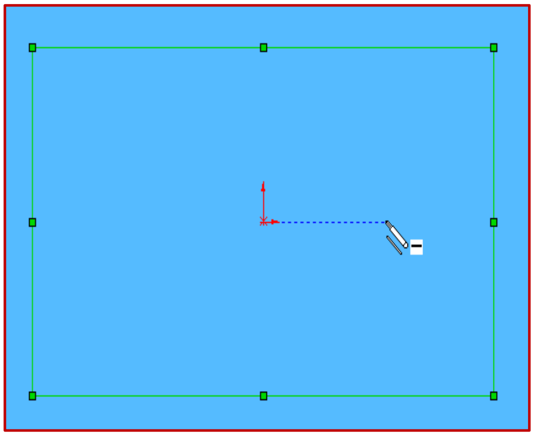


Figure 3.1 – Example of *Inferencing*

Smart Dimension

Smart Dimension is the best command for dimensioning geometry in SolidWorks. It can be used as follows:

1) click a line to dimension,
2) click two points to dimension the distance,
3) click a line and a point to dimension the distance,
4) click two parallel lines to dimension the distance and
5) click two lines that are not parallel to dimension the angle.

Just by making the length of a line the desired value does not create a dimension. Always use Smart Dimension.

Inferencing

Inferencing will indicate that your line is horizontal or vertical, or that your mouse pointer is coincident or parallel to another line. This is a very useful feature when sketching.

Unfortunately, sometimes SolidWorks will make the wrong assumption. To delete a sketch relation, click ***Tools→Relations→Display/Delete*** on the **Main Drop-down Menu** (or the delete key on your keyboard**)**. You can click ***Tools→Relations→Add*** to add relations that were not captured with ***Inferencing***.

You can see which relations you have in your sketch by clicking ***Tools→Relations→Display***.

It is possible to override ***Inferencing***. On the **Main Menu** click ***Tools→Sketch Settings→Sketch Relations*** to toggle ***Inferencing*** on/off. If the symbols displaying sketch relations clutter your sketch, you can turn them on/off with ***View→Sketch Relations*** on the **Main Menu**. The checkmark indicates that they are visible.

Modeling 3 – Constrain your sketch by adding dimensions and sketch relations.

✓ The sketch is fully defined when no blue lines are visible and all the lines are black.

✓ Blue lines mean that you have to use ***Smart Dimensions*** and/or ***Add Relation*** commands to fully define and constrain the sketch. All the dimensions are in the drawings provided in the Appendix.

✓ The sequence or order of your commands is important. Sometimes, as you define dimensions and sketch relations, the sketch will change into something that does not resemble what you want. Step back by using ***Edit→Undo*** on the **Main Drop-down Menu** (or click the ***Undo*** icon in the ***Quick Access Menu***) and try a different sequence of commands. In most situations, going back one step is enough to fix sketches.

Constraining Sketches

SolidWorks will allow the creation of features, parts and assemblies with sketches that are not fully defined (they still have blue lines), but it is very likely that the model will fail in the future. It is a best practice to fully define dimensions and relations to get a fully constrained sketch before proceeding to create a solid.

Occasionally, after we add all the information we think is necessary, the sketch is still blue. This means that the sketch is not fully defined.

To find what information is missing, we must ask the following two questions:

1) Did I specify the sizes of all the lines, circles, etc.?
2) Did I specify how far from the origin they are located?

Another way of finding what is missing is by clicking and dragging the geometry. If it moves up-down, for example, it means that it is not fixed in that direction and it needs a vertical distance to the origin. If a line can stretch or a circle can change its diameter, it is because that dimension has not been specified.

Overdefined Sketches

Sometimes we provide more information than is necessary to define a sketch and the color of the lines changes to yellow or red. Yellow means that the geometry is over defined and has too many dimensions and/or relations. Red means that the geometry is impossible. To correct, undo the last step or delete one or more **Dimensions** or **Relations and try something different**. Always remember that the order in which **Dimensions** and **Relations** are specified is important. Try a different sequence.

If you get the warning "***Make Dimension driven?"*** you must decide based on design intent. If you choose to make the new dimension driving, you can help to resolve the problem when you use the SketchXpert.

- ✓ On the lower right corner of your desktop (the status bar), click the red ***Over Defined*** warning.
- ✓ On the left side of the screen you will get two options
  1) ***Diagnose*** and
  2) ***Manual Repair***.
- ✓ ***Diagnose*** will present you with a list of possible solutions. You must choose the one that agrees with your design intent.
- ✓ Manual Repair will list the conflicting Dimensions and Relations. You must delete the unnecessary ones.

Modeling 4 – Use one or more command(s) from the ***Features*** toolbar to create the solid that you want.

- ✓ The ***Features*** toolbar has commands that can be used to transform 2D sketches into 3D solids. The commands can be additive (***Extruded Boss/Base*** and ***Revolved Boss/Base***) or subtractive (***Extruded Cut*** and ***Revolved Cut***).
- ✓ SolidWorks calls the 3D solids created with this toolbar ***Features***, and they are listed in the ***FeaturesManager Design Tree*** (see Figure 1.4) in the same the order in which they are created.
- ✓ Complex 3D solids and parts are created by repeating modeling steps 1 to 4 as needed.

- ✓ The default option is to combine ***Features*** automatically as they are created, but it is possible to switch-off this option in the ***PropertyManager Design Tree*** to create multiple bodies.

Modeling 5 – Document your design intent in the ***Design Binder*** for future reference.

- ✓ Models, assemblies and drawings are created to communicate ideas. Documenting important information such as materials, requirements, references and sources of information, and renaming the features to make them easy to recognize is important. Never skip this step. As a minimum, record your name and the date you created the model in the ***Design Binder***.

The first four steps can be remembered as ***PSDF: Plane, Sketch, Dimension, Feature***.

Configuration Management

Designs are usually re-used and/or modified over the production life of a product. Even if the product is no longer manufactured or sold, the company must maintain an inventory of replacement parts. For example, the B-52 bomber was designed during the 1950s and production ended during the 1960s but it is still in the U.S. Air Force inventory and the plan is to keep it flying until the middle of the 21st century.

When product modifications and upgrades are needed, the changes are easier and safer if the design is well documented. Unfortunately, well documented designs are the exception. In well-run organizations, the responsibility of enforcing proper record keeping and of maintaining this information belongs to the Configuration Control or Configuration Management office. They use software variously known as product data managers (PDM), product life-cycle management (PLM), revision control software or change control software.

The most important reason for documentation and formal record keeping is legal liability. Drawings, notebooks, e-mails and other communication can and will be used as legal evidence to determine liability in court cases. Poor quality documentation can result in a guilty verdict in a court case.

3.3 The Skateboard Deck

First, we will model the deck of the skateboard. Drawing SKTBD001 in Appendix A has all the information we need. We will use the five modeling steps in Section 3.2 to create the solid model.

Step 12: Open a new document using the **InchPart** template and click to select the ***Front Plane*** in the ***FeatureManager design tree*** as shown in Figure 3.2. This will be the plane that we will use to sketch the cross-section of the part. In a cube, this is the plane in front of you. The other planes we could use to sketch are the top plane and the right side plane. A good idea is to hand sketch the assembly that you need and plan how you will create each part, including what plane is the best to use for sketching.

- ✓ When you place the mouse cursor over the ***Front Plane*** icon in the ***FeatureManager***, a large red rectangle becomes visible. This is the front plane that we will use to sketch the cross-section of the skateboard. The square is only a portion of a plane that extends to infinity in the X and Y directions. A sketch outside the square is still in the front plane.
- ✓ On the ***FeatureManager design tree***, click the ***Front*** plane to sketch the cross-section of the part.

Step 13: In the ***CommandManager*****,** click the ***Sketch*** toolbar. This will make it active and the sketch commands will become visible. Next, click the ***Line*** command and sketch the skateboard cross-section as shown in Figure 3.3.

- ✓ To sketch the line, click to start the line, drag the line to the location of the end point and release. To end the line, double-click after you release the mouse button.

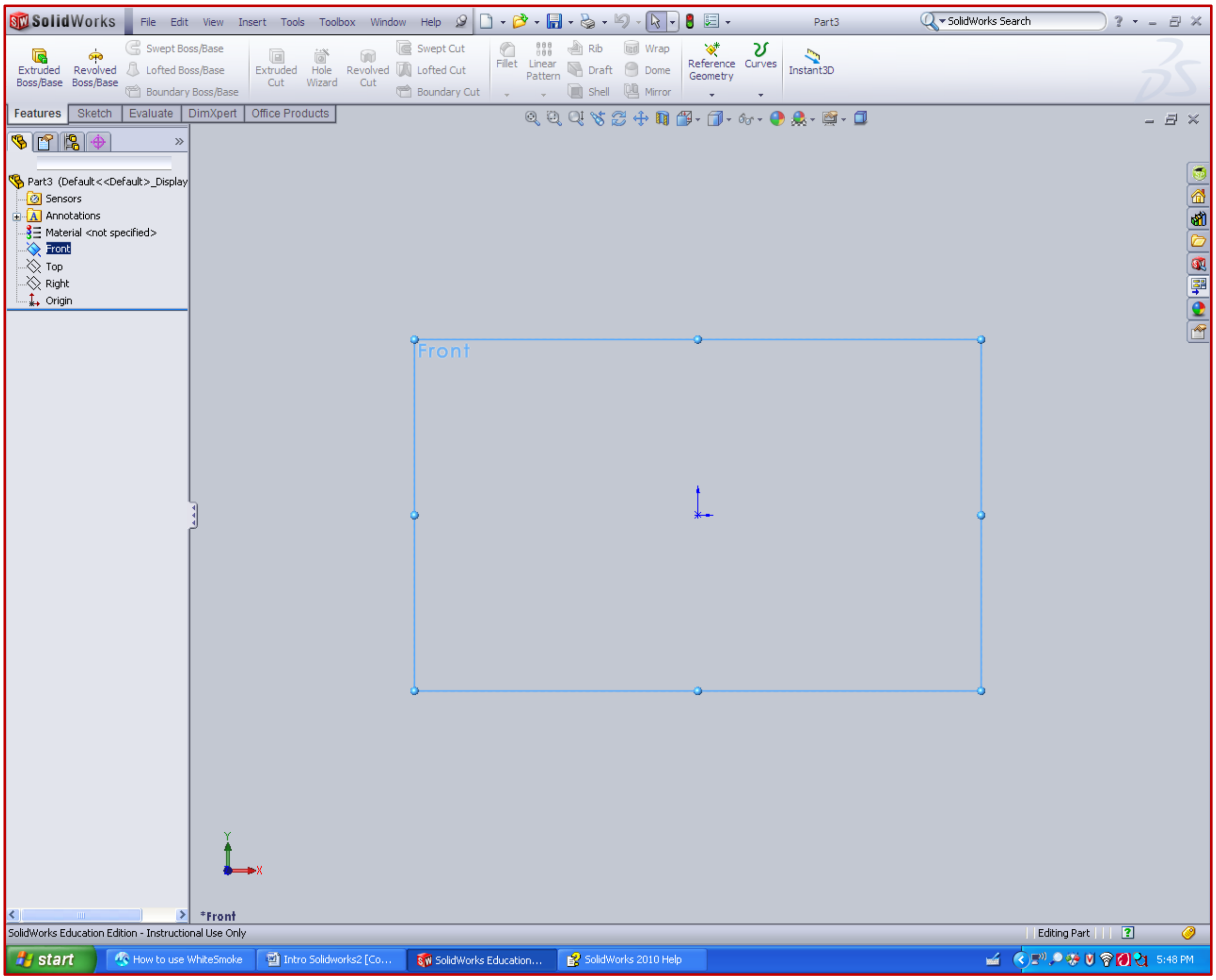


Figure 3.2 – Selecting the Front Plane

- ✓ To create a chain of lines, click-drag-release and repeat click-drag-release for the second line. Repeat drag and release until you have all the lines you need. To exit, you can double click or use Esc. Do not break the horizontal line into two lines. It must be one single line that extends past the origin.

- ✓ With SolidWorks it is not necessary for the sketch to have exact dimension or angle. In this step, we only need approximate geometry. The next step will be to fully define, dimension and constrain the geometry.
- ✓ Notice that when the ***Line*** tool is active, the mouse cursor will change into a symbol that looks like a pencil. Also, two icons on the confirmation corner at the top-right corner of the graphics area become visible. The two icons and the status bar at the bottom indicate that we are in sketch editing mode. To quit the sketch and retain your work, click the pencil icon. To discard your work, click the X icon.
- ✓ To delete a line, click the line and then press the Delete key in your keyboard. Alternatively, click in the graphics area and drag your mouse over the line. Note:The complete line must be inside the highlighted area. Next, release the mouse button and the line will change color. Finally, press the Delete key in your keyboard to erase the line.

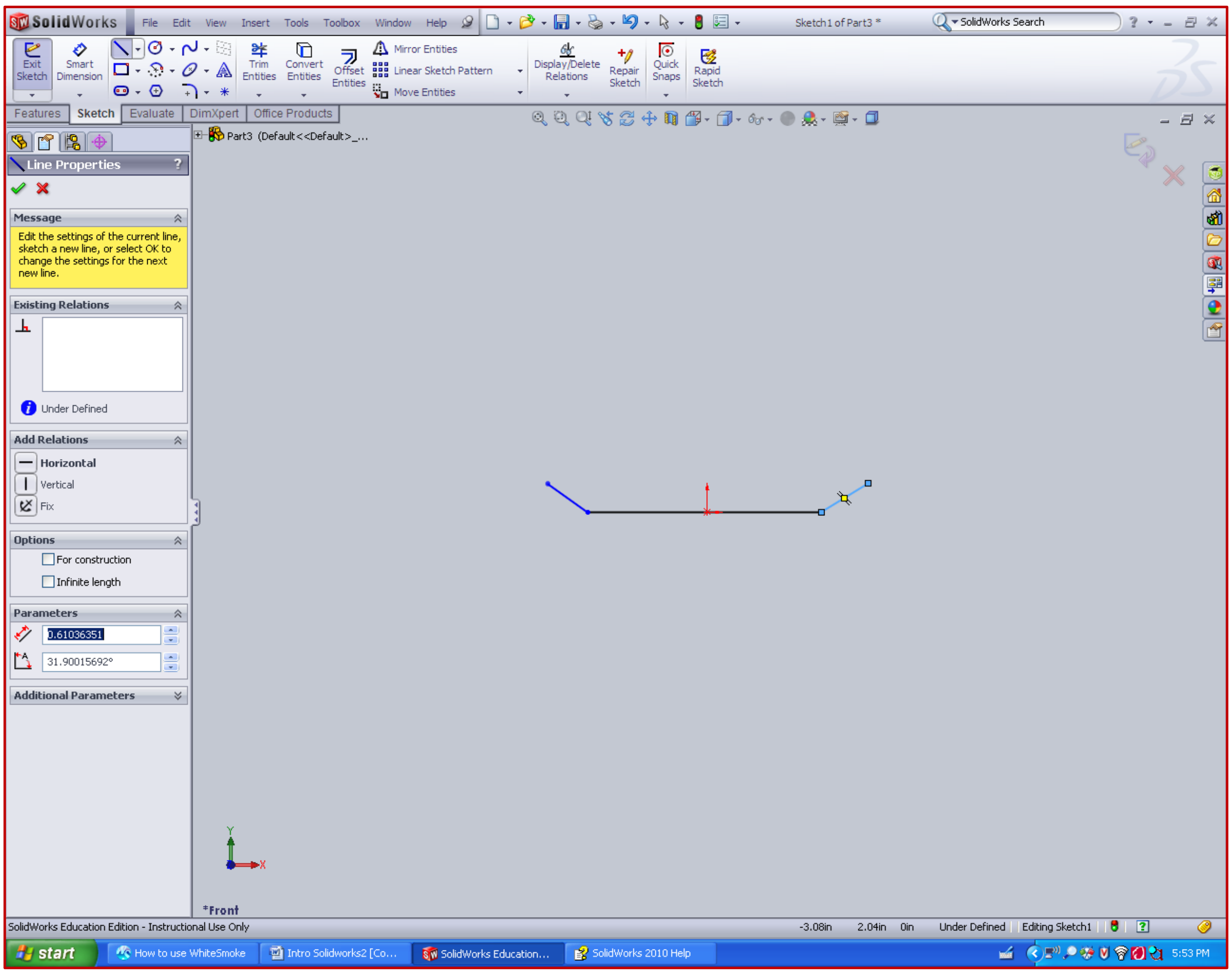


Figure 3.3 – Sketching the Skateboard Cross-section

Step 14: Use ***Smart Dimension*** and ***Add Relation*** in the ***Display/Delete Relations*** pull down menu to fully define the geometry. When this step is complete there should not be any blue lines.

- ✓ To input the skateboard dimensions, click the ***Smart Dimension*** tool in the ***Sketch*** toolbar of the ***CommandManager*** or ***Tools➔Dimensions➔Smart***. The mouse cursor changes to a dimension symbol.
- ✓ Click the deck's horizontal line, drag the mouse, and release. Click again to place the dimension. An input box will appear. Double click on the number in the input box and type the correct dimension, which in our case is 24 inches. If part of the sketch moves out of view, click the ***Zoom to fit*** icon in the ***View (Heads-Up)*** toolbar (see Figure 1.4).
- ✓ Repeat for the right and left lines and make them 4 inches in length.
- ✓ To dimension an angle, click on two lines and SolidWorks will show the angles between the lines. You must move around the vertex to see the various complementary and supplementary angles. The angles in our skateboard are 20 degrees as shown in Figure 3.4.

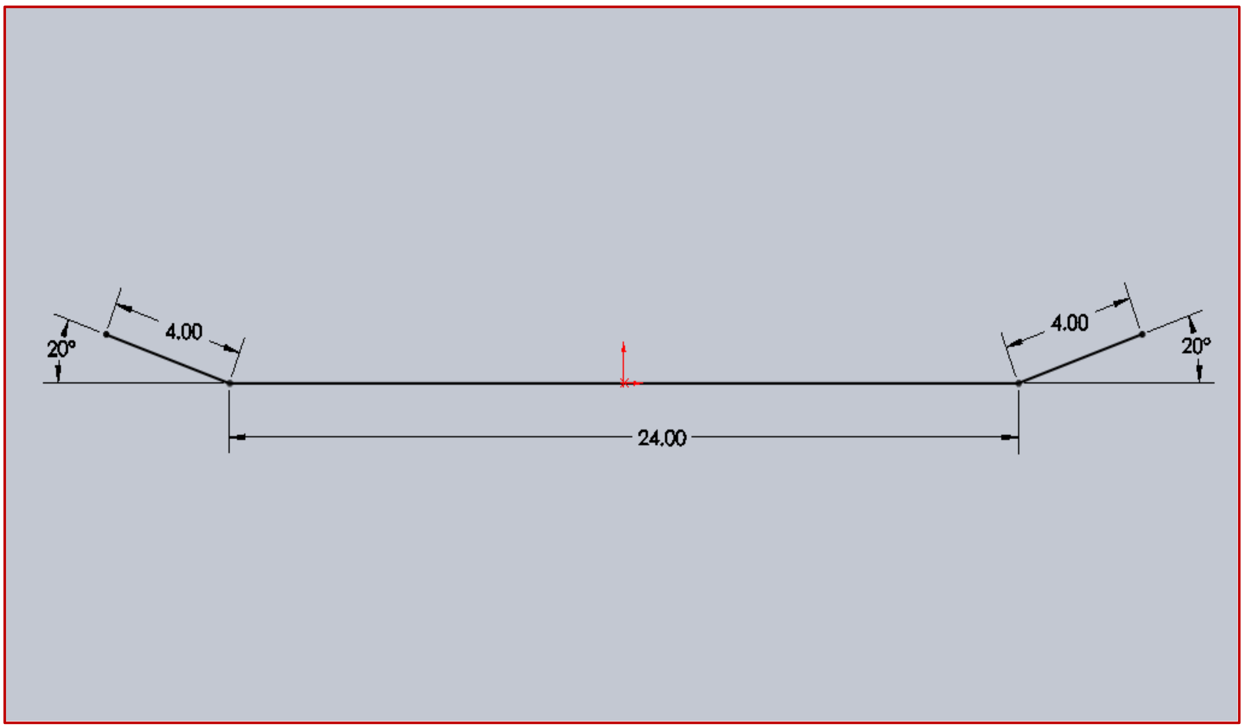


Figure 3.4 – Sketch with Dimensions

- ✓ Even when all lines have been dimensioned, they are still blue. This means that the sketch is not fully defined. SolidWorks needs more information. In addition to the dimensions, we must provide the location of the lines. We will do that by making the sketch symmetric about the origin. This will locate directly the horizontal line and indirectly the left and right lines, which are connected to the horizontal line.

- ✓ Because the location of the lines is not fixed in space, we can click and drag them to a new location. Verify that the lines move.
- ✓ To fix the lines, click ***Tools→Relations→Add*** in the **Main Drop-down Menu** or ***Display/Delete Relations→Add Relation*** in the **Quick Access toolbar**.
- ✓ In the dialog box shown in Figure 3.5, verify that you have the blue box labeled ***Selected Entities.***
- ✓ Click the horizontal line and then the origin to select those entities. They will appear in the dialog box.
- ✓ Click ***Midpoint*** in the ***Add Relations*** dialog box that appears and accept by clicking the check mark at the top of the box.
- ✓ All lines will turn black to indicate that the sketch is now fully defined.

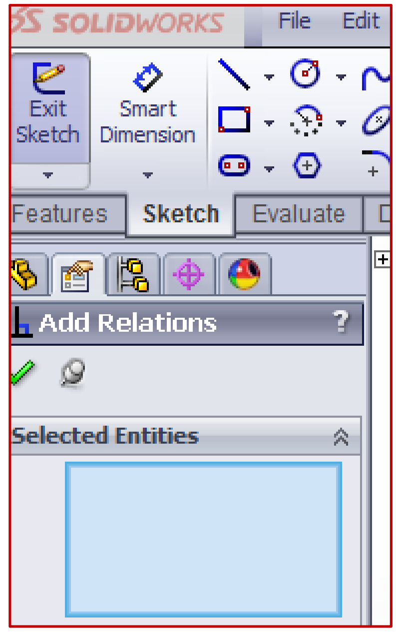


Figure 3.5 – *Tools→Relations→ Add Relations*

- ✓ You can see what relations you have in your sketch by clicking ***View→Sketch Relations*** in the **Main Drop-down Menu.** To see the relations the icon must be highlighted. If your sketch is cluttered, you can toggle them off. If you need to see them, you can toggle them on.

Step 15: Next, use the ***Extruded Boss/Base*** command to create the width and the thickness of the skateboard. It can be found in the ***CommandManager*** and in the **Main Drop-down Menu**.

- ✓ Click the ***Features*** tab in the ***CommandManager*** to make the toolbar active and show the commands available.
- ✓ Click the ***Extruded Boss/Base*** command to get the dialog box in Figure 3.6. Select the ***Mid Plane*** option for ***Direction 1*** and the ***Thin Feature****.* Type the dimension 8 inches for D1 and 1/2 inches down for T1. The Up and Down directions can be selected by choosing ***One-Direction*** in the first dialog box and then pressing the button with the arrows. When you toggle the button with the arrows you alternate between extruding up and extruding down. Verify that your extrusion is down.
- ✓ Click the check mark to accept. The preview should look like Figure 3.6.
- ✓ ***Direction 1*** is the width (in and out of the screen). ***Thin Feature*** is for the thickness. Extruding down puts the origin in the top surface.

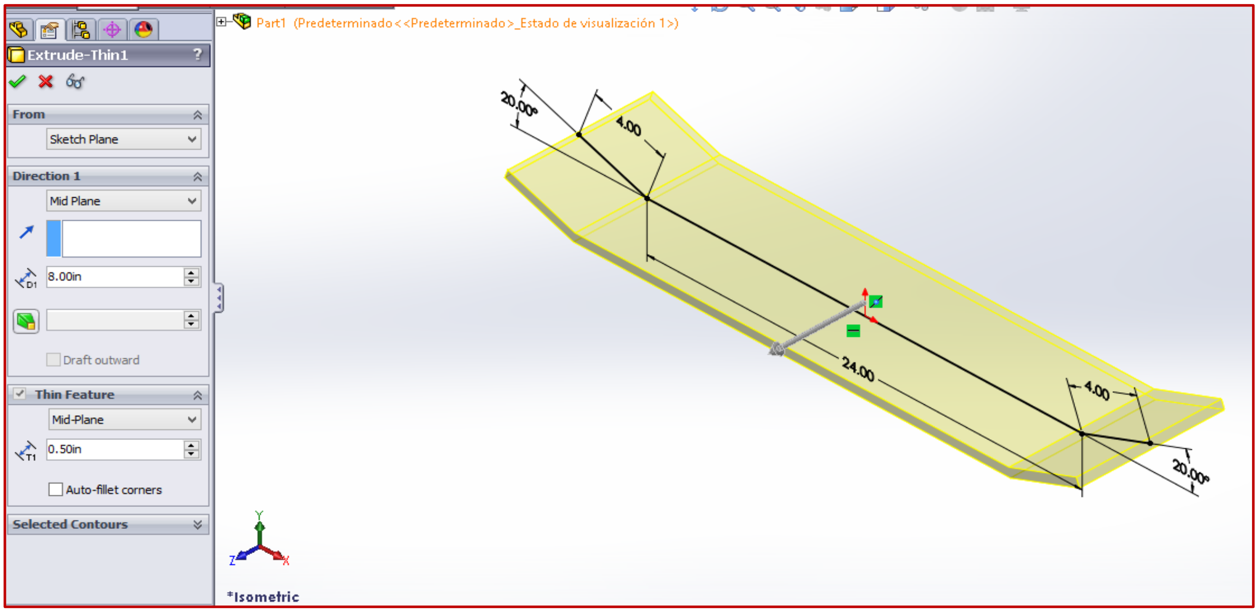


Figure 3.6 – Extrusion of the Sketch

At this point our model begins to looks like a skateboard, but it still needs a few details.

Step 16: Create rounded corners by using the ***Fillet*** command and then you get the chamfer the edges of the deck.

- ✓ Click on the ***Fillet*** tool in the ***Features*** toolbar to get the dialog in Figure 3.7.
- ✓ Verify that you get the dialog box ***Items to Fillet***. Click on all four corners of the skateboard and notice that the edges are now listed in the dialog box.
- ✓ When clicking the corners, perhaps one or more corners are hidden from view. If you are in ***Display Style→Shaded With Edges*** (in the ***View Heads-Up*** toolbar), click ***Display Style→Hidden Lines Visible*** to see through the Deck.
- ✓ Type the fillet radius of 4 inches.
- ✓ Verify that the ***Full preview*** option is selected on the ***Items to Fillet*** menu to match Figure 3.7.
- ✓ Use the commands in the ***View Heads-Up*** toolbar to move, rotate and zoom.

After the preview shows all corners with the round fillets, click the check mark to accept. If for some reason the preview does not show the rounded corners, or looks different from the one on the figure, there is something wrong. Verify that all the options in your screen match the one shown in the figure. As a last resort, cancel the ***Fillet*** command by clicking X on the ***Confirmation Corner*** and repeat these steps.

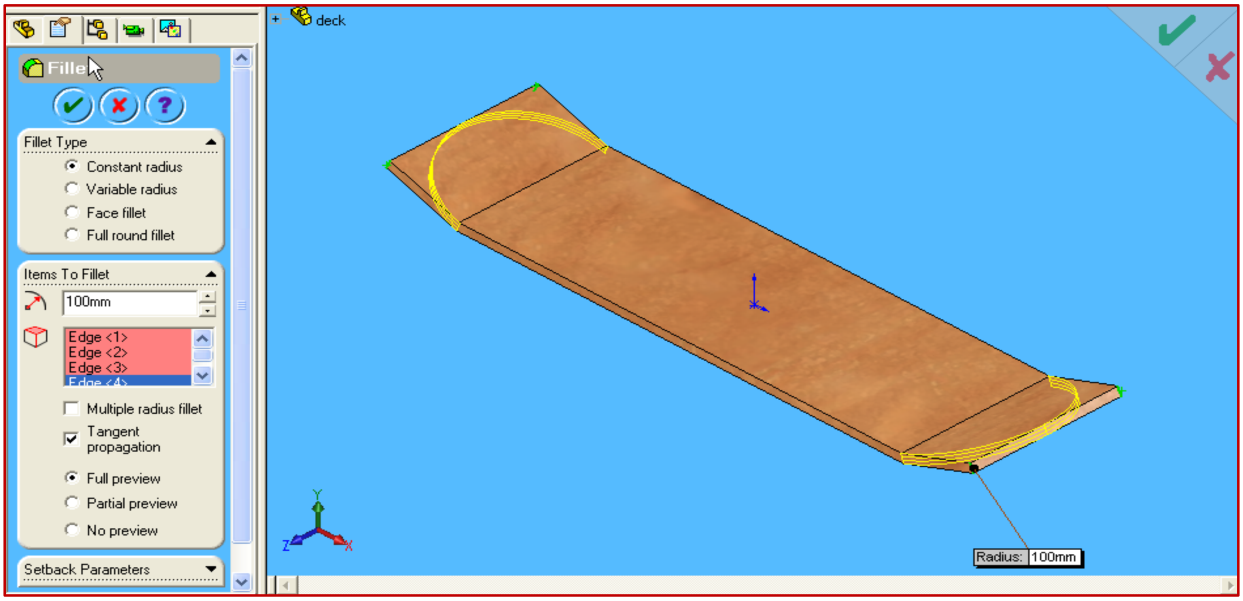


Figure 3.7a – Fillet Menu and Pre-view

- ✓ To chamfer the edges of the deck, click on the edges around the top and bottom surfaces. (Use ***Display Style→Hidden Lines Visible*** when convenient.)
- ✓ In the ***Features*** tab of the ***CommandManager***, click the ***Fillet*** icon pull down menu and select the ***Chamfer*** command.
- ✓ Select the ***Angle distance*** option and 0.125 inch and 45 degrees. See Figure 3.7b.
- ✓ Click to accept.

Display Style

SolidWorks can display your part in different styles. On the ***View (Heads Up) toolbar*** click on the ***Display Style*** icon to see your options. The most cpu intensive option is ***Shaded with Edges***. ***Wireframe*** is the easiest to display. In the early days of CAD, wireframes were the norm. Today, equipment is powerful enough that it is not important which display style is used. Still, some display styles are more descriptive than others and you might want to experiment to see which one you prefer. As an exercise, change the display style of the skateboard deck to see the difference. Many SolidWorks users believe that shaded with edges is the most realistic display style. Other options like Hidden Lines Visible or Wireframe are useful because it is possible to see details that are hidden in other views.

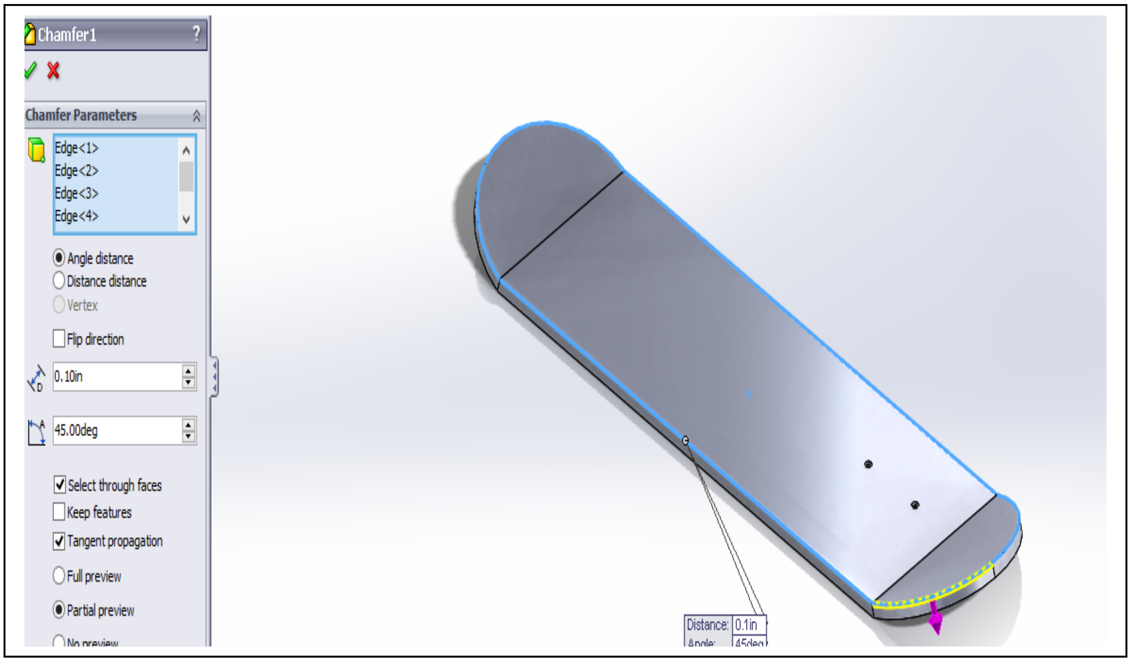


Figure 3.8b – Chamfer Menu

Step 17: In Figure 3.1, the sketch of the skateboard cross-section show a sharp transition between the horizontal line and the two ends that slope at 20 degrees. In the real world, this is impossible. We need a 1/4 inch radius on each of the transitions to allow for the bending radii at the two locations. To make the change we must edit the skateboard cross-section sketch.

- ✓ On the ***CommandManager***, select the ***Sketch*** *tab*.
- ✓ On the ***Sketch*** toolbar, click ***Sketch Fillet***. This is a fillet created while sketching and can be contrasted to the feature fillet created in **Step 16**.
- ✓ On the dialog box, type 0.25.
- ✓ Click on each of the two intersection points.
- ✓ Click the check mark to accept.

Step 18: Next, use the ***Hole Wizard*** to create the holes needed to attach the Truck and the wheels.

- ✓ Click the skateboard top surface to make it the sketching plane before you click the ***Hole Wizard*** command. The ***Hole Wizard*** icon is located in the ***Feature*** tab of the ***CommandManager*** or you can find it in the **Main Drop-down Menu** at ***Insert→Feature→Hole Wizard***.
- ✓ Click ***Hole Wizard*** to get the dialog box in Figure 3.8. Select #8 countersunk flathead screws. Because the holes must be drilled through the thickness select ***Thru All*** as the end condition. Verify that your menu choices match the figure.
- ✓ Next, click the ***Position*** tab to locate the holes.

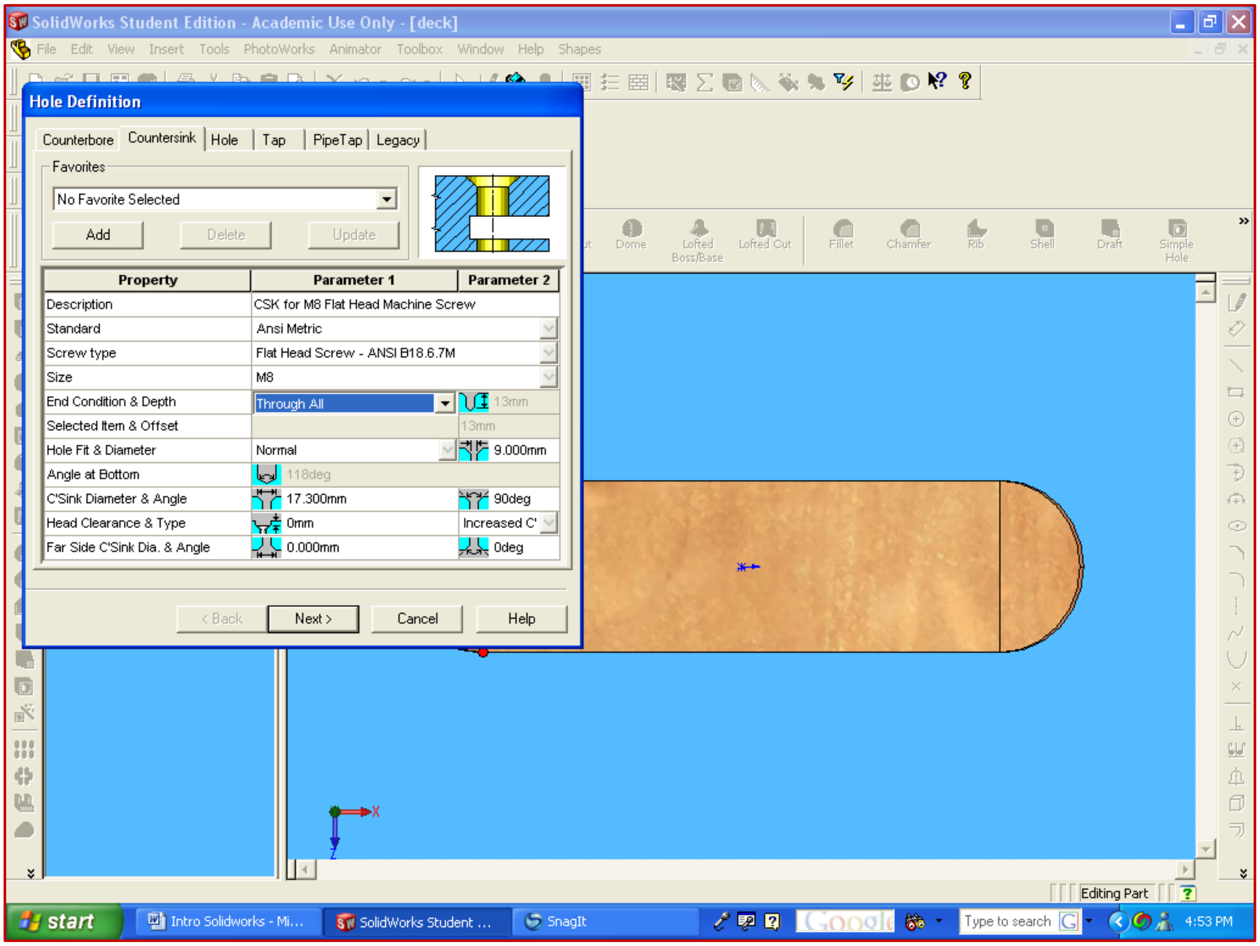


Figure 3.9 – Hole Wizard Dialog Box

- ✓ SolidWorks asks for the location of the hole(s) and the cursor becomes a pencil. Click to mark the approximate location of two parallel holes as shown in Figure 3.9. Only the approximate location is required. Like any SolidWorks sketch, we will define the exact locations and sketch relations after we create the geometry.
- ✓ The locations of the holes will be defined next. Click the ***Smart Dimension*** tool and use it to dimension the distance between the leading hole and the line where the deck is bent as shown in Figure 3.10. Type the distance 2 1/4 inches.
- ✓ Dimension the distance between the two holes by clicking each hole and typing the dimension 3 inches.
- ✓ Locate the front hole from the edge of the skateboard deck. The distance is 3 inches.

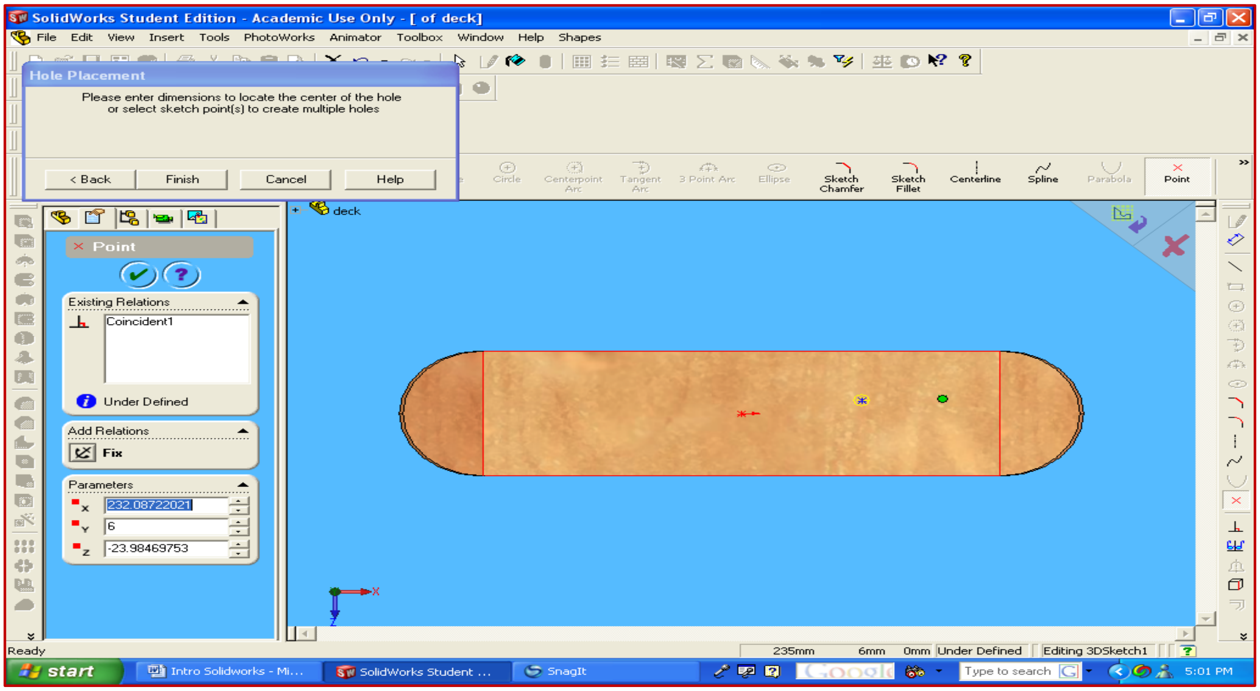


Figure 3.10 – *Hole Wizard→Positions* Tab

- ✓ If the two holes are not parallel, we must create a horizontal sketch relation between them first. Click ***Tools→Relations→Add*** in the **Main Drop-down Menu** or in the ***CommandManager*** click the pull-down menu ***Display/Delete Relations*** and select Add Relation.
- ✓ Click on the two points at the center of the holes and select ***Horizontal*** in the ***Add Relation*** dialog box. Click on the check mark to accept. (Note: If you used ***Inferencing*** to create two parallel holes, this step may not be necessary.)
- ✓ The holes will change from blue to black to indicate that the sketch is now fully defined.
- ✓ We are finally ready to create the countersunk holes. Click the check mark again to accept.

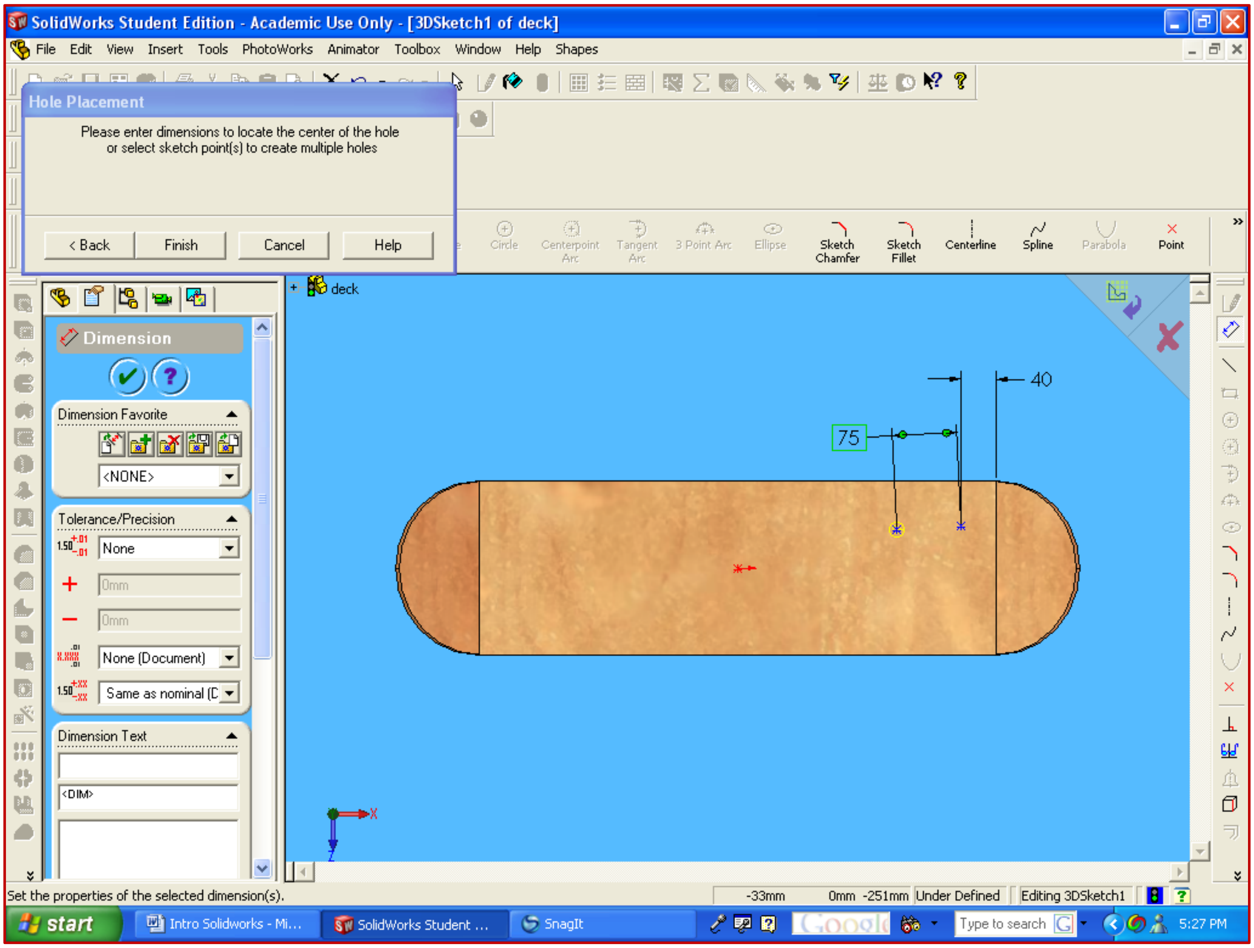


Figure 3.11 – Hole Locations

Step 19: Use the ***Mirror*** command to duplicate the two holes.

- ✓ Click the ***Mirror*** command in the ***Features*** toolbar. The dialog box is shown in Figure 3.12. The features we want to duplicate are the ***CSK*** (countersunk) holes for flat head machine screws. It should be visible in the ***Features to Mirror*** dialog box. If ***CSK*** is in the ***Features to Mirror*** dialog box, skip the next paragraph.
- ✓ If CSK is not in the ***Features to Mirror*** dialog box, click the box to make it active and notice the new location of the ***FeatureManager design tree***. It has moved to the Graphics Area to create space for the ***Mirror*** command dialog box. Click on the (+) sign of the ***FeatureManager design tree*** to expand the feature tree. As expected, this is a list of all the features created so far. Find and click on the ***CSK*** feature. It should be the last line in the feature tree. This will add ***CSK*** to the ***Features to Mirror*** dialog box.
- ✓ The other information that SolidWorks needs to mirror the holes is the mirror plane. Click on the ***Mirror Face/Plane*** dialog box to make it active and then click ***Front Plane*** in the ***FeatureManager design tree***. SolidWorks will show a preview of the new holes.
- ✓ Click the check mark to accept.

✓ Repeat the ***Mirror*** command across the ***Right*** plane to copy the **Mirror1** feature to the other side, as seen in Figure 3.13.

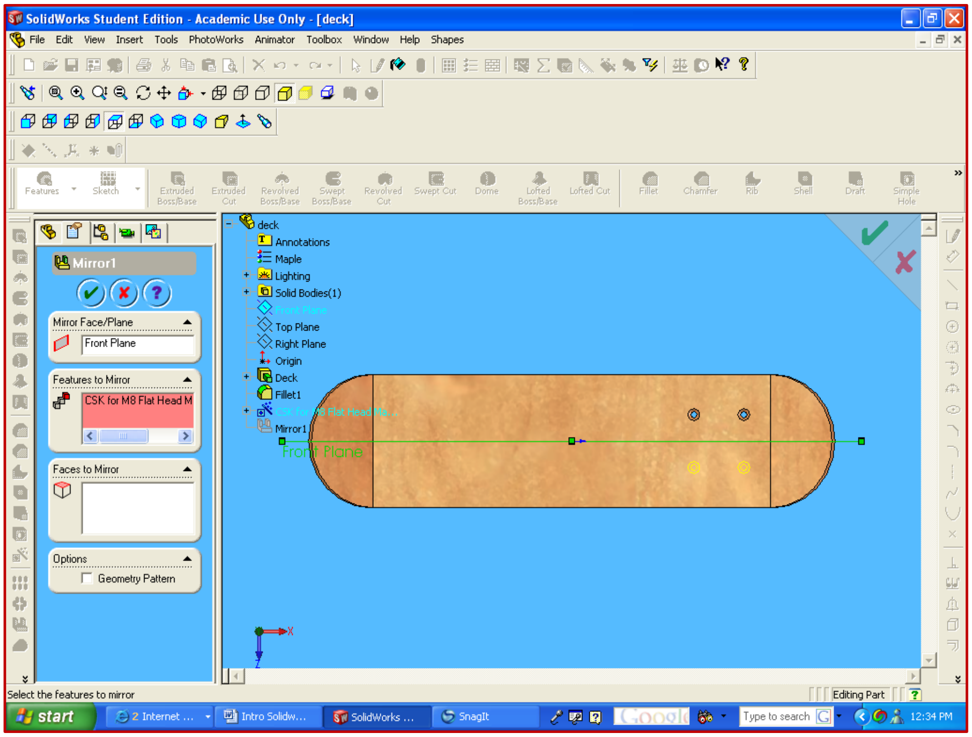


Figure 3.12 – Duplicating Two Holes Across the Front plane

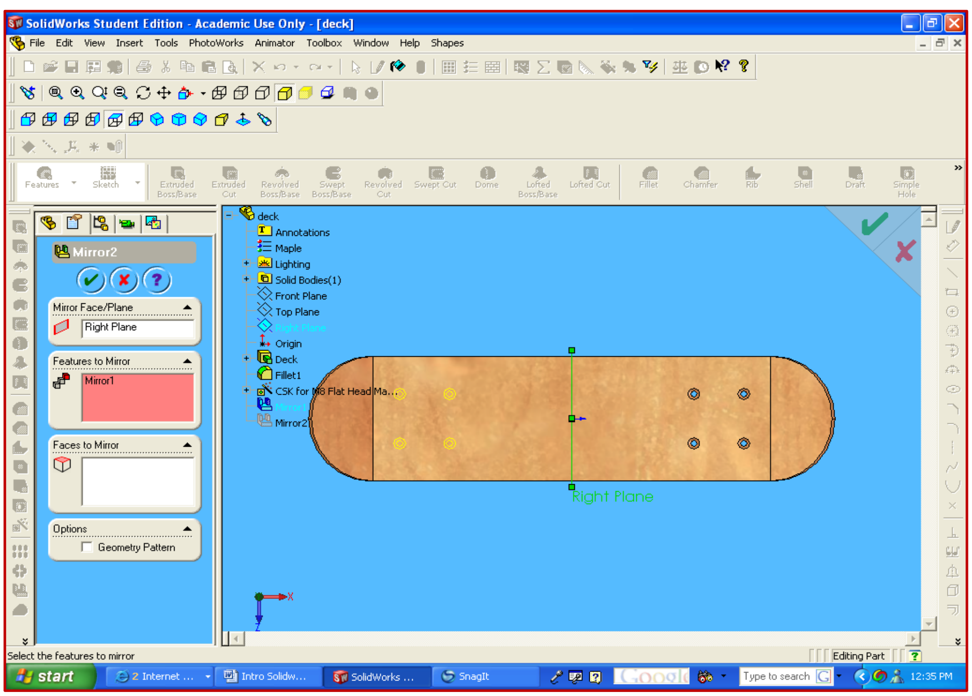


Figure 3.13 – Duplicating the 4-hole Pattern Across the Right plane

Step 20: Our skateboard model is now complete, but we need to save design information that others will need to understand our design.

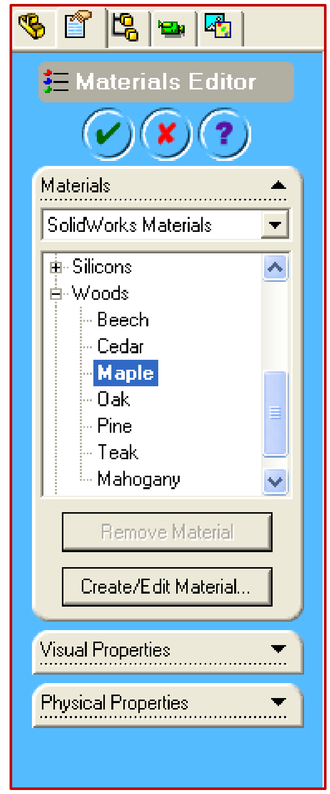


Figure 3.14 – *Materials Editor*

- ✓ Expand the Design Binder in the Feature Tree by clicking the + sign.
- ✓ Double-click Design Journal.doc to open.
- ✓ Type a description of the part, its use and the material.
- ✓ Type your name and the date for future reference. Type **Save** and then type **Close**.
- ✓ The skateboard deck is made of pine wood. Right-click the ***Material*** icon in the ***FeatureManager design tree*** and then click ***Edit Material*** to get the ***Material Editor***. Find the folder for ***Woods*** and click to show the types of wood available. Select **Pine** and click the check mark to accept. See Figure 3.14.
- ✓ Click ***File→Properties*** and fill your name (***Author***) and SKATEBOARD DECK (***Title*** and also ***Description***) on the ***Summary tab*** and 1) SKBD101 (***Part No***). SolidWorks will assign the weight automatically if you selected the material.

Step 21: Rename the features with more descriptive names.

- ✓ This is shown in Figure 3.15, where the names assigned by SolidWorks to the deck features have been replaced by more descriptive names.
- ✓ To change a Feature name, first make it active by clicking, and then click and hold for a second and release. You can now type the new name.
- ✓ Another way to change the name is to right-click over the name and select ***Feature Properties***. You can now type the new name.
- Notice that we did not save the part as DECK because in most engineering organizations drawings are saved and retrieved by the drawing number and not by the name. The reason is that the number avoids duplication when we have, for example, two or more gears or pins. Another reason for using numbers is that the practice is a remnant from a long time ago when it was easier to automate the saving and retrieving of numbers instead of letters or names.
- Another peculiarity in the file name is that all the letter are upper case. In Engineering drawings and sketches, lower case letters are never used. Hence, SKBD is the correct form and not skbd or Skbd.
- ✓ The skateboard deck is finally complete.
- ✓ Create a folder to store all your skateboard files and save your model as file SKBD101.

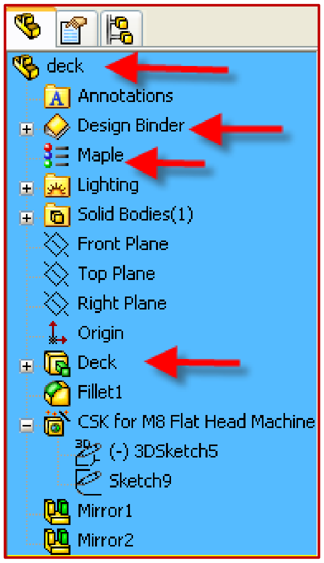


Figure 3.15 – Renaming Features

Part Numbers and Part Names

Parts are tracked by their part number, not their part name. The reason is that it is easier to organize, store and retrieve a sequence of numbers than descriptive names.

This is the reason why SolidWorks stores the name you use to save your file as the part number.

A part number will typically include information about the project and the assembly where it belongs. For the skateboard deck, we will use SKBD100 for the top level assembly and SKBD101, SKBD102, etc. for the drawings. Drawing number SKBD110 is the Truck sub-assembly and the drawings that belong are saved as SKBD111, SKBD112, etc.

Practice Exercises

1. Click ***Help→SolidWorks Help*** on the **Main Drop-down Menu**, expand the ***Troubleshooting*** topic at the end of the Topics list (the left column) and study the sections ***Troubleshooting Resources*** and ***Errors***.
2. Use Help→**Search** to learn about ***Sketch Relations.*** Write, in your own words a definition of the term and describe how you can take advantage of this feature of SolidWorks.
3. Sketch a square and then use ***Extruded Boss/Base*** to create a bar with a square cross-section. Click on ***Draft On/Off*** and see how your part changes.
4. Explore the ***Extruded Boss/Base*** command. What other options are available besides ***Mid Plane***?
5. Click ***Tools→Mass Properties*** and find the volume and weight of the skateboard deck. (Hint: to get the correct weight, you must first define the material in the ***FeatureManager design tree*** – see Step 20 and Step 73).
6. Re-draw the parts modeled in this lesson using millimeters instead of inches. Use the mmPart template and the dimensions in the detail drawing in Appendix B.

Questions

1. Compare maple and pine as the material for the Deck. What are the advantages and disadvantages of each material? Which one you recommend? List the requirements and assumptions on which you base your recommendation.

2. Search the Internet for information about Product Data Management. Describe what it is in your own words.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor 3D Skills

- Lines
- Geometric Relations
- Dimensions
- Extruded Boss

Videos from SolidProfessor SolidWorks for Beginners

- Editing Geometry
- Hole Wizard
- Mirror Feature

Videos from SolidProfessor Core Concepts for Parts and Assemblies

- Chamfers
- Fillets
- Sketch Fillet
- 3D Mirroring
- Sketch Mirroring
- Fully Defined Sketches
- Over Defined Sketches

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks website: http://www.solidworks.com

Lesson 4 – Modeling the Wheel Using the Revolve Command

4.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Explain and use the ***Corner Rectangle*** command.
- ✓ Explain and use the ***Revolve*** command.
- Explain and use the ***Centerline*** command.
- Explain and use the command ***Convert Entities*** and ***Offset Entities***.
- Explain why the Wheel Sub-assembly was not created as a single part.
- Create part models using more than one ***PSDF*** sequence.
- Learn to use the drawings in Appendix A to find the dimensions you need.

4.2 Introduction

The ***Revolve Boss/Base*** command, like the ***Extruded Boss/Base*** command in Lesson 3 can be used to create 3D solids from 2D sketches. Both commands belong in the ***Features*** toolbar. Extrude creates solids by elongating the 2D sketch in one or more directions and Revolve creates solids by rotating the sketch around an axis of revolution.

4.3 Modeling the Wheel

The wheels and tires should be two different parts because they are different materials and are made using completely different manufacturing processes. Skateboard wheels can be injection molded using a hard grade of polyurethane, a type of synthetic rubber. Skateboard wheels can also be made from steel. The tires are made with a softer polyurethane. The assembly of the wheel and tire is made by placing the wheel in a mold and pouring soft polyurethane around it.

To create a model of the wheel, we will follow the five steps we learned at the beginning of Lesson 3:

1) select a plane,
2) sketch,
3) add dimensions,
4) use the ***Revolve*** command in the ***Features*** toolbar and, finally,
5) document the design information and references.

Remember ***PSDF***: ***Plane***, ***Sketch***, ***Dimension*** and ***Feature***.

Step 22: Open a new part with the **InchPart** template and select the ***Top*** plane for sketching.

- ✓ If you hand sketch the skateboard to see how the various parts fits into the assembly, you will notice that the Top plane is the best to sketch this part.
- ✓ Refer to the skateboard assembly drawing, drawing SKBD100 in the Appendix and Figure 4.2 below, the cross-section of the wheel. The cross-section will revolve around the line.

Step 23: Create the wheel using the ***Revolve*** command. The wheel and the tire are both made of polyurethane. The wheel is a harder grade of polyurethane, compared to the tire.

- ✓ In the ***Top*** plane, sketch the rectangle in Figure 4.1 using the ***Corner Rectangle*** command on the ***Sketch*** tab of the ***CommandManager***.
- ✓ Notice that ***Corner Rectangle*** is one of many ways of creating a rectangle. If you click on the pull-down menu, you will see that you can make a ***Center Rectangle***, a ***3 Point Rectangle*** and a ***Parallelogram.*** Type each of these commands in the Search box, select SolidWorks Help in the pull-down menu and click the magnifying glass to see how they are used.

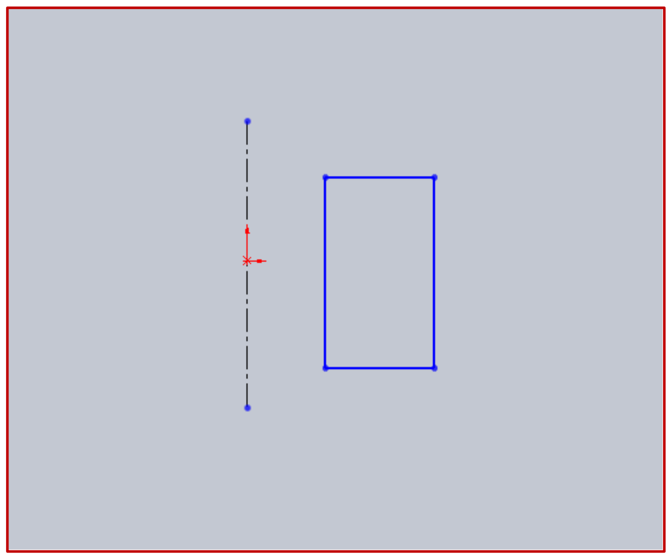


Figure 4.1 – Wheel Sketch

- ✓ In the ***Sketch*** tab, click the ***Centerline*** command on the ***Line*** pull-down menu.
- ✓ Sketch the construction line shown in the figure. The ***Centerline*** is parallel to the sides of the rectangle. Note: Centerlines or construction lines are for the benefit of humans and invisible to some SolidWorks calculations. In this example, the Centerline is the axis of rotation but is not part of the solid created.
- ✓ Use the ***Smart Dimension*** tool to fully define the sketch. The dimensions are shown in the detail drawing of the wheel in the Appendix (SKBD121).
- ✓ Notice that the height of the rectangle is 1.25 inch. It is the height of the cylinder that will be created when the rectangle rotates around the axis.
- ✓ The distance from the left vertical line of the rectangle to the centerline is one half of 0.87 inch (you can type 0.435 inch in the input box or 0.87/2). The inside diameter of the wheel is indicated by the Greek letter Φ and equals 0.87. The units are given in the Title Block, the rectangle in the lower right corner of the drawing. The Title Block has the name of the part (WHEEL), the part number (SKBD121), the material (POLYURETHANE) and above the material it says “DIMENSIONS ARE IN INCHES).

- ✓ Notice that the drawing says 2X (i.e. two times) before the diameter. The reason is that there are two pockets for two bearings. The 2X indicates that both pockets have the same diameter.
- ✓ The distance from the right vertical line of the rectangle to the center line is one-half of 1.37 inch (0.685 inch). The outside diameter of the wheel in 1.37 inches.
- ✓ The width of the rectangle is one-half of 1.37-0.87 inch (0.25 inch). We must divide by two because when we revolve the rectangle, the cross-section will show two rectangles. One rectangle on each side of the centerline. This dimension is not in the drawing because it is redundant. It can be calculated from the other dimensions given.
- ✓ Redundant dimensions are called “double dimensioning” and when shown in a drawing they must be enclosed in parentheses. They are called reference dimensions and are shown for convenience. Remember the rule: All the dimensions must be in the drawing but never use double dimensions.
- ✓ Notice that the sketch is still free to move up and down if we pull on the top or bottom blue lines because it is not fully constrained. To fully constrain we must dimension the distance from the origin to the top or bottom line. Click the height or 1.25 inch dimension and it will show in the input box. Next divide by two (/2). This defines the location of the line as ½ of the height and fixes the rectangle. It is important to click on the height and not type 1.25” because the number could change in the future. All the lines will turn black and the rectangle cannot move anymore.
- ✓ Next, we will revolve the rectangle around the centerline. See Figure 4.2.
- ✓ On the ***Features*** tab, Click ***Features→Revolved Boss/Base*** to get the menu in Figure 4.2. The only input request is the axis of revolution.

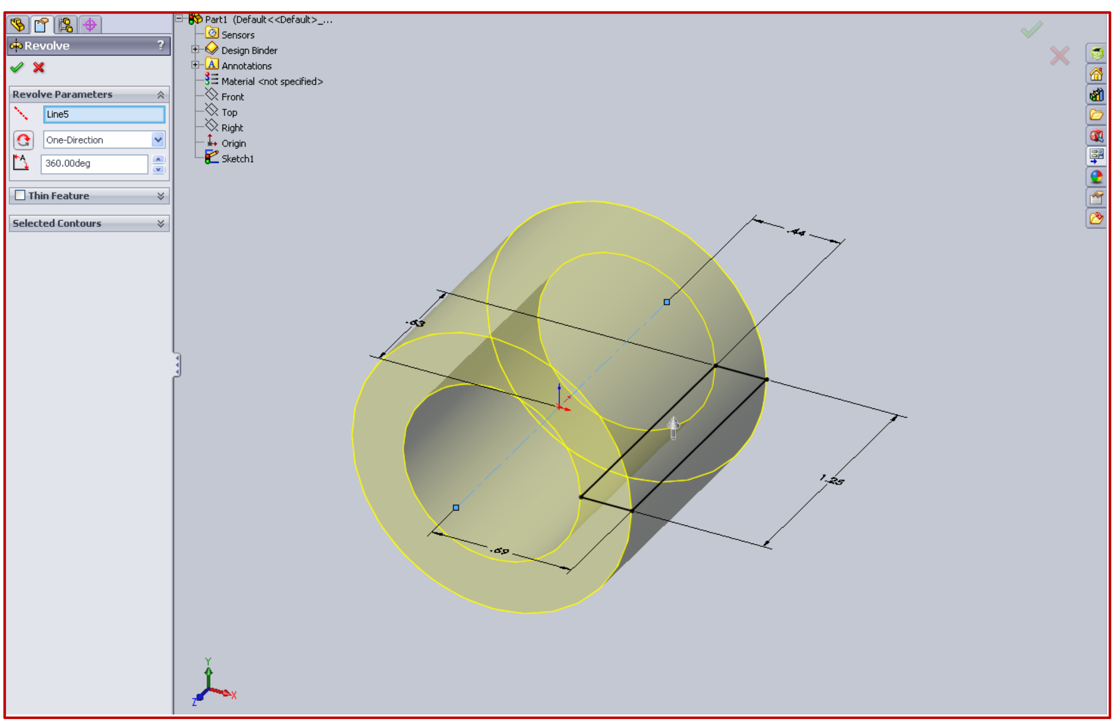


Figure 4.2 – *Features→Revolved Boss/Base*

- ✓ Move the cursor over the centerline and SolidWorks will show a preview of the revolved solid in Figure 4.2.
- ✓ Click the check mark to accept.

Step 24: Next, we will create a spacer to separate the two bearings. We always use two bearings instead of only one to balance the loads acting on the part.

- ✓ Select the ***Front*** plane. Click ***View Orientation→Normal To*** in the ***View (Heads Up) toolbar*** to see the sketching plane full size.
- ✓ Click the ***Sketch*** icon on the ***Sketch*** tab in the ***CommandManager***.
- ✓ Click ***Convert Entities*** and then click the inside diameter of the cylinder to create a circle on the ***Front*** plane with the same diameter as the inside diameter of the cylinder. The new circle will be the larger diameter of the spacer we are going to create.

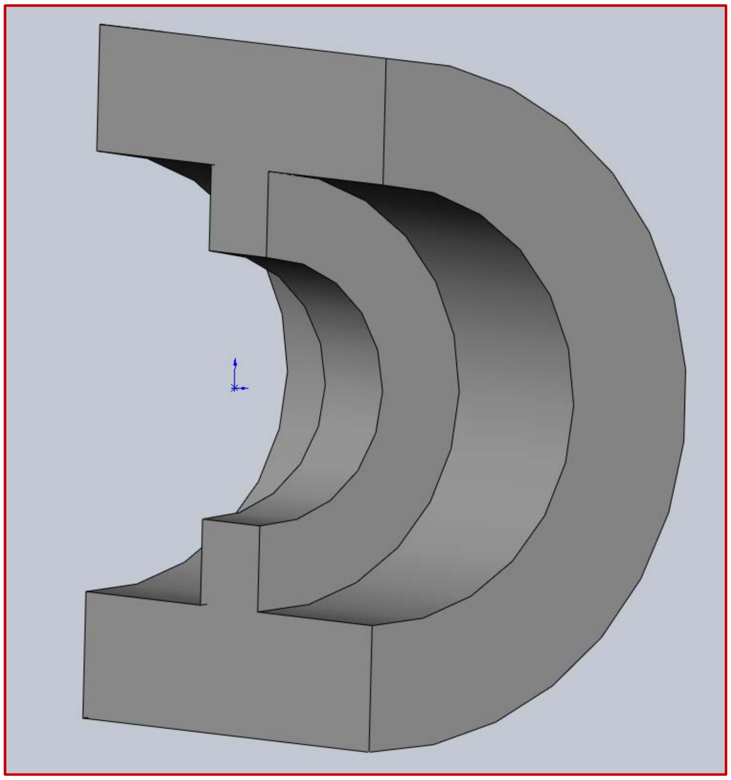


Figure 4.3 – Skateboard Wheel

- ✓ Click the line created and then click ***Offset Entities*** in ***the CommandManager→Sketch tab***. This will be the smaller diameter of the spacer. From the drawing of the wheel, notice that we must type (0.87-0.25)/2. This will be the offset needed in the input box. Remember that both circles were sketched on the ***Front*** plane. Notice that you can type the equation to calculate the offset.
- ✓ Extrude the spacer by clicking ***Features→Extruded Boss/Base*** in the ***Main Pull-down menu*** or ***Extrude Boss/Base*** in the ***Features tab*** in the ***CommandManager***.
- ✓ In ***Direction 1***, use the pull-down menu to select ***Midplane.***
- ✓ Type the thickness of the spacer in ***D1***. In the drawing SKBD121 we can see that the thickness of the spacer is 1.25-2(0.5) = 0.25 inch. (Remember that the total height of the cylinder is 1.25 inches. Also, remember that there are two pockets for bearings and each pocket is 0.5 in deep).
- ✓ Click the check mark to accept.
- ✓ To see the interior of the part, click ***View→Display→Section View*** (You can find ***Section View*** on the ***Main Drop-down Menu*** at the top of your screen or in the ***View***

Heads-up toolbar. See Figure 1.4). Select the Top plane or right plane to cut the part and create a cross-section like Figure 4.3.

✓ To go back to the complete part, click on the ***Section View*** command again.

Step 25: Save the important design information:

✓ Make the material polyurethane**.**

✓ Fill the Design Journal with your name, date, part description and material.

✓ In ***File→Properties*** type your name (***Author***), Skateboard Wheel (***Title*** and ***Description***) and SKBD121 (***PART NO***)

✓ Save your part file as SKBD121.

Practice Exercises

1. Model the skateboard rubber tire.
2. Model the wheel using the ***Extruded Boss/Base*** command instead of ***Revolve***. (Hint: Sketch two circles and extrude, or use ***Extruded Boss/Base*** and then ***Extruded Cut***.)
3. Practice creating a) a center rectangle, b) a 3 Point rectangle c) a parallelogram.
4. Use the command ***Offset Entities*** to create the bearings spacer inside diameter. (Hint: Offset the spacer's outside diameter by 0.17 inch.)
5. Explain why the best plane for the wheel sketch is the Top plane. Propose an alternative and its advantages and disadvantages.
6. Re-draw the parts modeled in this Lesson using millimeters instead of inches. Use the mmPart template and the dimensions in the detail drawing in the Appendix.
7. It is possible to ***Extrude Boss/Base*** one circle or two concentric circles but not three circles. Explain why?

Questions

1. Explain why the wheel and the tire are each individual parts with separate detail drawings. Discuss the advantages and disadvantages of the having only one detail drawing for both.

4. What is the hardness range of polyurethane?

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor 3D Skills

- Rectangles
- Circles & Arcs
- Revolved Boss

Videos from SolidProfessor SolidWorks for Beginners

- Revolve Feature

Videos from SolidProfessor Core Concepts for Parts and Assemblies

- Convert Entities
- Offset Entities

Internet Resources

CADeducators.com website: http://www.cadeducators.com

Polyurethane properties: http://www.polyurethanes.org/en/what-is-it

Design tables and information: http://www.engineeringtoolbox.com/

Lesson 5 – Modeling Miscellaneous Rubber Parts

5.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Make changes to an existing solid model.
- Sketching on faces.
- Explain and use the ***Dome*** command.
- Explain how to toggle between a dome and a dome cavity.
- Explain and use the command ***Extruded Cut***.
- Explain and use the ***Draft On/Off*** option in ***Extruded Cut***.

5.2 Introduction

The truck assembly has two spacers. We will model the first by sketching two concentric circles with the command ***Extruded Boss/Base***.

To create the second spacer, we will edit the ***FeatureManager design tree*** to change the thickness of the first spacer. It is not necessary to start a new solid model because SolidWorks allows the modification of existing models. Remember to save the two models as separate files called TopSpacer and BottomSpacer.

Finally, we will model the Truck Bumper using the command ***Dome***. We will need two sequences of ***PSDF*** (***Plane***, ***Sketch***, ***Dimension*** and ***Feature***).

5.3 Modeling the Top Spacer

The top and bottom spacers have the same cross-sections, but different thickness.

Step 26: Open a new InchPart document.

- ✓ In drawing SKBD113 in Appendix A, find the best plane to draw the two concentric circles that will be extruded to get the doughnut shaped spacers. I should be the ***Top*** plane.

- ✓ Sketch two concentric circles as shown in drawing SKBD113 in Appendix A and in Figure 5.1.

- ✓ Use ***Smart Dimension*** to add the inside diameter and the outside diameter in Figure 5.1. The dimensions can be found in drawing SKBD113 in Appendix A.
- ✓ Use ***Extruded Boss/Base*** to create the thickness for the Top spacer and the ***Fillet*** command to round the edges.
- ✓ The thickness is found in the spacer thickness table in SKBD113.
- ✓ Use a midplane extrusion. Click OK to accept.

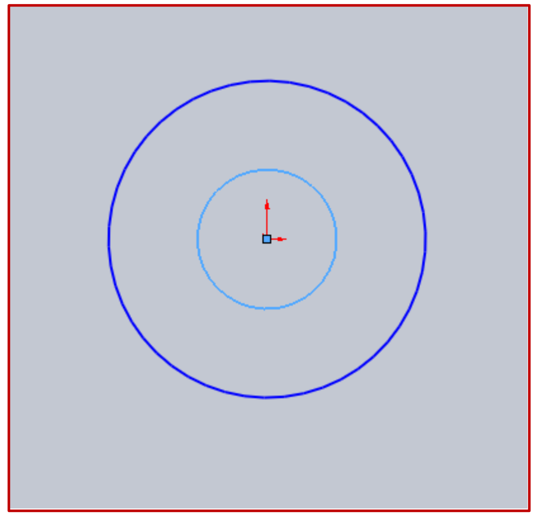


Figure 5.1 – Sketch for TopSpacer

Step 27: Use the ***Fillet*** command to cut the 90 degree edges around the tire.

- ✓ Hold the control key and click each of the edges of the tire, and then click the ***Fillet*** command on the ***Features*** tab in the ***CommandManager.*** The input box will show Edge1 and Edge2 as the items to fillet. Type the radius of the fillet to add a fillet around the edges. The completed spacer is shown in Figure 5.2a.
- ✓ Right-click ***Material*** and then click ***Edit Material.*** Select Polyurethane rubber as the material.
- ✓ Click the arrows to the right side of the ***Feature/Property/Configuration*** managers' tabs. This will show the Display Pane. See Figure 5.2b.
- ✓ Select the column under the multicolor ball and click the first line. Click Appearance to open the ***PropertyManager*** for the part. Click a shade of red you like and then the checkmark to accept.
- ✓ Record your name, date, and other information in the Design Journal.
- ✓ Fill the information in the ***File→Properties*** form.
- ✓ Save your model as SKD113.

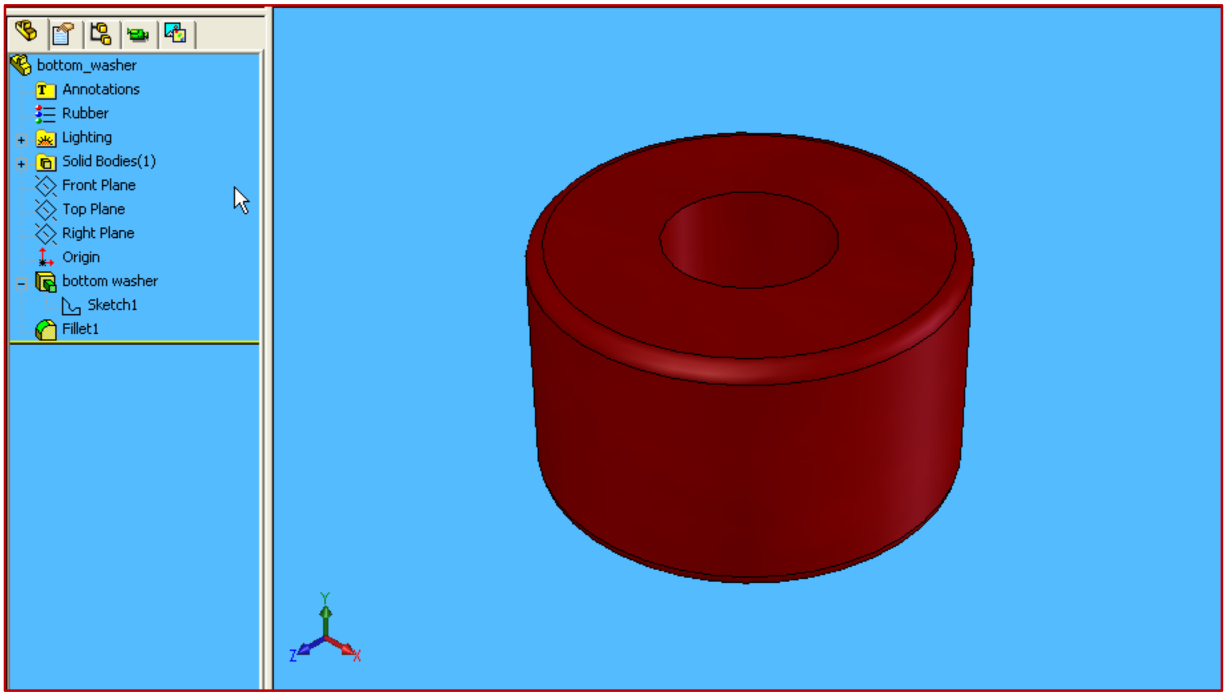


Figure 5.2a – TopSpacer

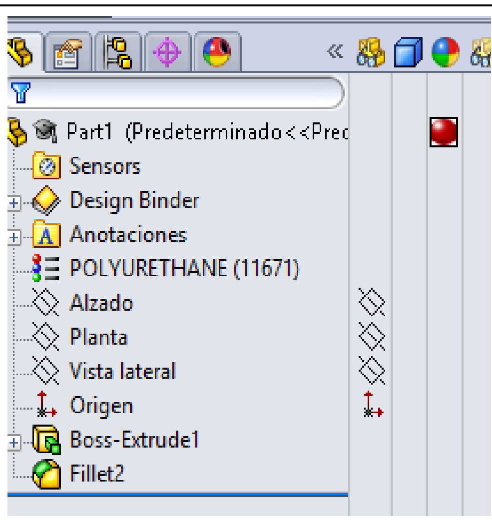


Figure 5.3b – Display Pane

Changes

Changes are inevitable in the design process because design is a learning experience. As you explore alternatives, you learn that some are better than others. Thus, the better ideas replace the not-so-good choices and you have to constantly update your model to reflect the latest ideas. Fortunately, SolidWorks minimizes the work when we make changes. You can make changes in the sketch and in the extrude distance. It is also possible to change the sketch plane.

The problem with changes is that there will always be a different, or maybe even better, way of accomplishing a design goal. This can result in a never ending process of "improvements". At some point in time the design must be considered complete and ready for manufacture. Always keep in mind that "better is the enemy of good enough". If a design solution satisfies the requirements agreed to at the beginning of the design, do not change it for the sake of a "better design". Beware also of "creeping requirements". New requirements cost more and take more time.

Step 28: To create the Bottom Spacer:

- ✓ Begin with the TopSpacer.
- ✓ Right-click the ***Extruded Boss/Base*** feature in the ***FeatureManager design tree*** and select the ***Edit Feature*** icon.
- ✓ Change D1, the thickness of the spacer, to 0.50 inch.
- ✓ Click the check mark to accept.
- ✓ Use ***Save As*** in the ***Main Drop-down Menu*** to save the new spacer as SKBD115.

Note: We edited the extrusion ***Feature*** to change the length of the extrusion. If we wanted to change the diameter of the extrusion we would edit the ***Sketch***.

5.4 Modeling the Truck Bumper

Step 29: To create the Truck Bumper, refer to drawing SKBD114 in Appendix A for dimensions:

- ✓ Select the ***Top*** Plane and sketch a circle.
- ✓ Dimension the diameter.
- ✓ Use the ***Extruded Boss/Base*** command to create a cylinder 0.20 inch long. Use the ***Blind*** option and the Up direction. Click the check mark to accept.
- ✓ Use the command ***Rotate View*** on the ***View (Has-Up) toolbar*** to see the bottom surface of the cylinder.
- ✓ Click the bottom surface of the cylinder to make it active.
- ✓ Click the ***Insert→Features→Dome*** command on the **Main Drop-down Menu** or click the ***Dome*** icon on the ***Features*** tab in the ***CommandManager***.
- ✓ Type the radius of the dome on the input box.
- ✓ If ***Show preview*** has a check mark, you will see a preview of the dome in Figure 5.3.
- ✓ Click the check mark to accept the dome.

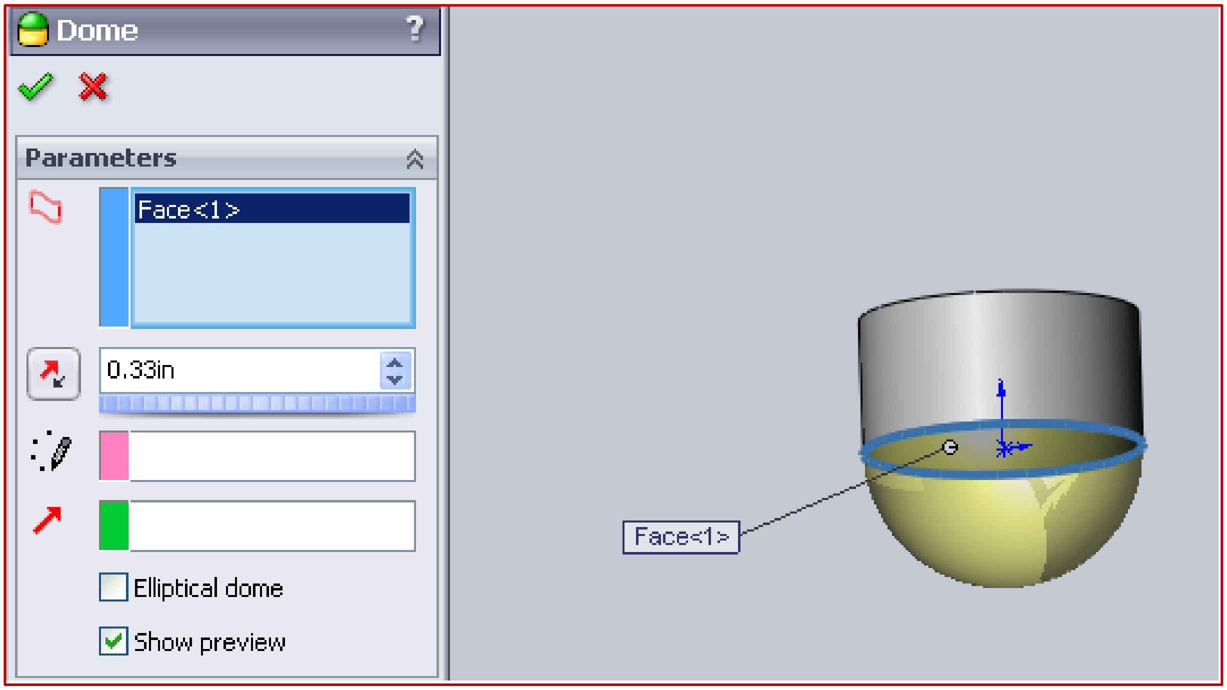


Figure 5.4 – Dome Command

Step 30: To create the interior of the part, click the flat surface opposite the dome.

- ✓ Click the ***Sketch*** icon on the ***Sketch*** tab of the ***CommandManager***.
- ✓ After clicking the ***Sketch*** icon you will notice that ***Offset Entities*** will change from gray to color. This means that it is now available for use.
- ✓ Click ***Offset Entities***.
- ✓ Type 0.10 inch on the input box.
- ✓ The new offset circle is smaller than the original circle. You can use the ***Reverse Direction*** arrows, if necessary.
- ✓ Click the check mark to accept.
- ✓ Click ***Insert➔Cut➔Extrude*** on the **Main Drop-down Menu**, or ***Extruded Cut*** on the ***Features*** tab.
- ✓ Use the blind option and type the depth of the ***Extruded Cut*** in the drawing. See Figure 5.4 below and notice that the ***Extruded Cut*** stops where the dome starts.
- ✓ Click the ***Draft On/Off*** icon and type 5 (degrees) draft.
- ✓ Click the check mark to accept.
- ✓ Use ***Section View*** in the ***View Heads-Up*** toolbar to verify that the diameter is smaller at the end of the extrusion.

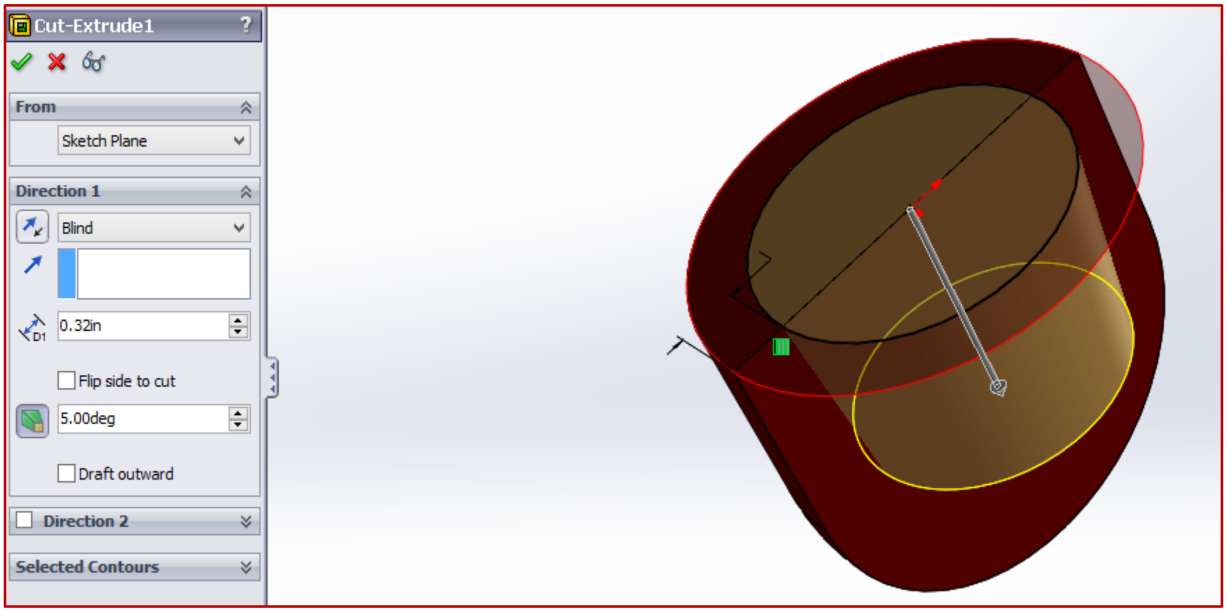


Figure 5.5 – *Insert→Cut-Extrude*

Step 31: To create a dome cavity, click on the surface created by the ***Extruded Cut*** to make it active.

- ✓ Click the ***Dome*** command.
- ✓ Type the value of the inside radius (.18 inch) on the input box.
- ✓ Click the ***Reverse Direction*** arrows to create a cavity. See Figure 5.5.
- ✓ Click the check mark to accept.

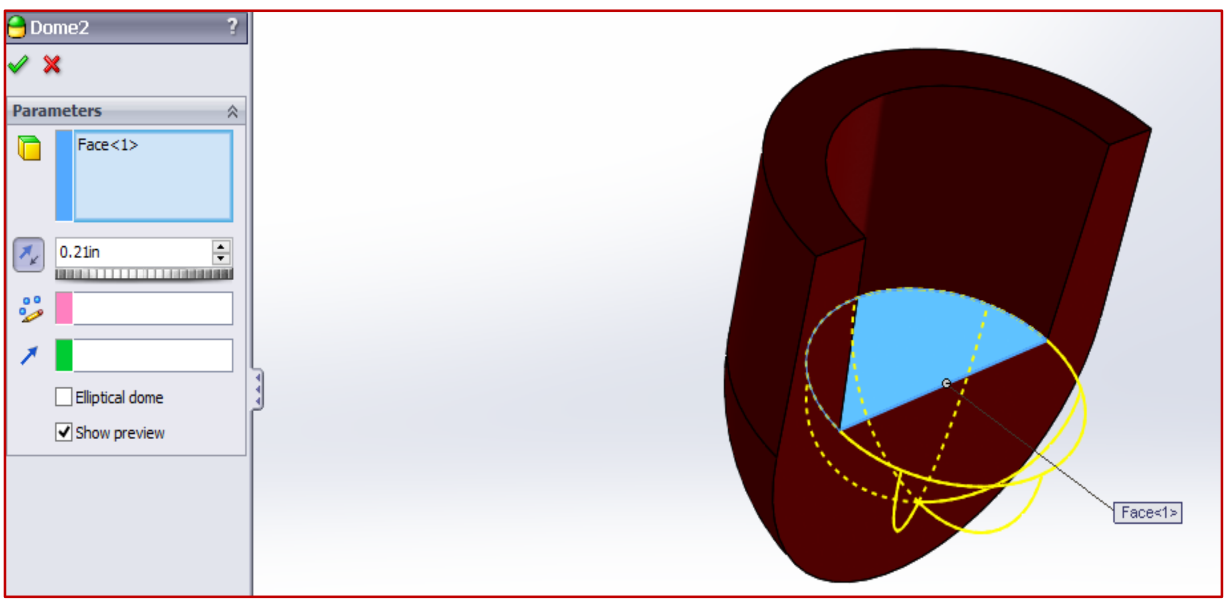


Figure 5.6 – Dome Cavity

Step 32: Make the material rubber and the color to red. Make an appropriate entry in the Design Journal and in the ***Properties*** form. Save your model as SKBD114.sldprt. The ***Title*** and ***Description*** are Front Bumper.

Practice Exercises

1. Model the spacer using ***Revolve Boss/Base*** instead of ***Extruded Boss/Base***.
2. Create models of the parts in this Lesson using millimeters instead of inches. Use the mmPart template and the dimensions in the detail drawing in Appendix B.

Questions

1. Create engineering working drawings for your bed, naming parts and assemblies? Make sketches of the bed assembly and of each of the parts and sub-assemblies. Explain why you chose to define each of the parts and assemblies the way you did. Which SolidWorks planes you would choose to sketch each of the parts? How would you transform the sketches into solid models?
2. Choose a typical consumer product and describe how you would divide it into parts and assemblies. Some examples of products include a bicycle, a tennis racket, an inline skate, a ball-point pen or a shoe. On which planes would you create each of the SolidWorks sketches? How would you transform those sketches into solid models?

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor SolidWorks for Beginners

- Rollback Bar and Order of Features

Videos from SolidProfessor Core Concepts for Parts and Assemblies

- Extruded Cut
- Revolved Cut
- Sketches on Faces

Videos from SolidProfessor Case Studies

- Editing Geometry

Internet Resources

CADeducators.com website: http://www.cadeducators.com

Polyurethane properties: http://www.polyurethanes.org/en/what-is-it

Design tables and information: http://www.engineeringtoolbox.com/

Lesson 6 – Editing Parts

6.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Edit SolidWorks part models.
- Explain and use the ***Rollback Bar.***
- Explain why the Wheel Sub-assembly was not created as a single part.
- Explain and use the feature command ***Circular Pattern***.

6.2 Introduction

One of the reasons for the widespread use of CAD software is the increase in productivity that can be achieved with their use. Before CAD, making a change required erasing the old geometry very carefully to avoid untidiness, and then adding the changes. If the new geometry did not fit in the space available, the change could require a new drawing. With SolidWorks, old models can be changed and re-used easily.

6.3 Adding Bumps to the Wheel

To get good mechanical grip between the wheel and the tire, we will add bumps to the outside surface of the wheel (see the ***Front View*** in drawing SKBD121). These bumps will prevent the tire from slipping around the wheel and will match pockets in the rubber tire (see SKBD122).

Step 33: Navigate to file **SKBD121.sldprt** and Click ***File→Open*** on the **Main Pull-down Menu** to open the document.

Step 34: We will use the ***Rollback Bar*** as a time machine. If we move the bar up before the ***Chamfer*** feature (and after ***Revolve***), we can create the bumps before creating the ***Chamfer***.

- ✓ First, move the mouse cursor over the line and it will change into a hand.
- ✓ Click and drag the line to the new location between the ***Revolve*** feature and the ***Chamfer*** feature.
- ✓ The ***Chamfer*** feature changes to gray, meaning that it is not active.

Step 35: Select the ***Front Plane*** as the sketch plane and click the ***Sketch*** command on the ***Sketch*** tab of the ***CommandManager***.

Step 36: Use the ***Rectangle*** tool to sketch Figure 6.1.

- ✓ Notice that ***Inferencing*** changes the color of the circumference to red to indicate that the lower left corner of the rectangle is coincident with the edge of the wheel.
- ✓ If you cannot sketch a rectangle with the corner in the desired location, try ***Zoom***. Sometimes lines, circles and other geometric entities cannot be made active because they are too small.
- ✓ If ***Zoom*** does not work, use ***Display/Delete Relations→Add Relation*** on the ***Sketch tab*** in the ***CommandManager*** to make the corner of the square coincident with the edge.

Step 37: Add the dimensions in Figure 6.1 to fully define the sketch.

- ✓ You can make the right corner and the circle coincident instead of using the dimension 0.250.

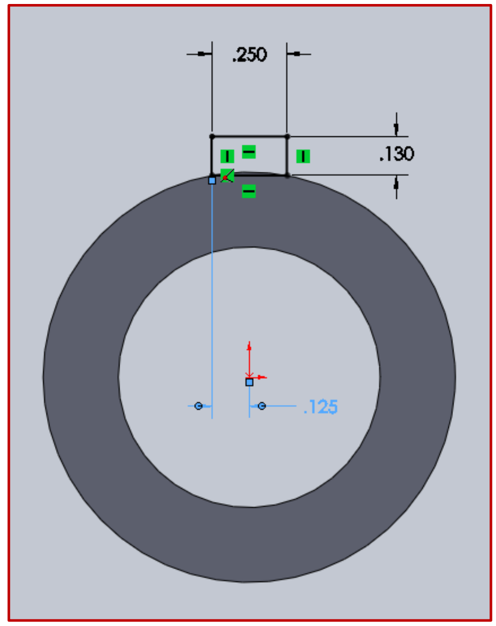


Figure 6.1 – Sketch for the Wheel Bump

Step 38: Use ***Extruded Boss/Base*** using the ***Midplane*** option and the length of the bumps in SKBD121 to get Figure 6.2. Notice that the bumps do not extend to the edges of the wheel.

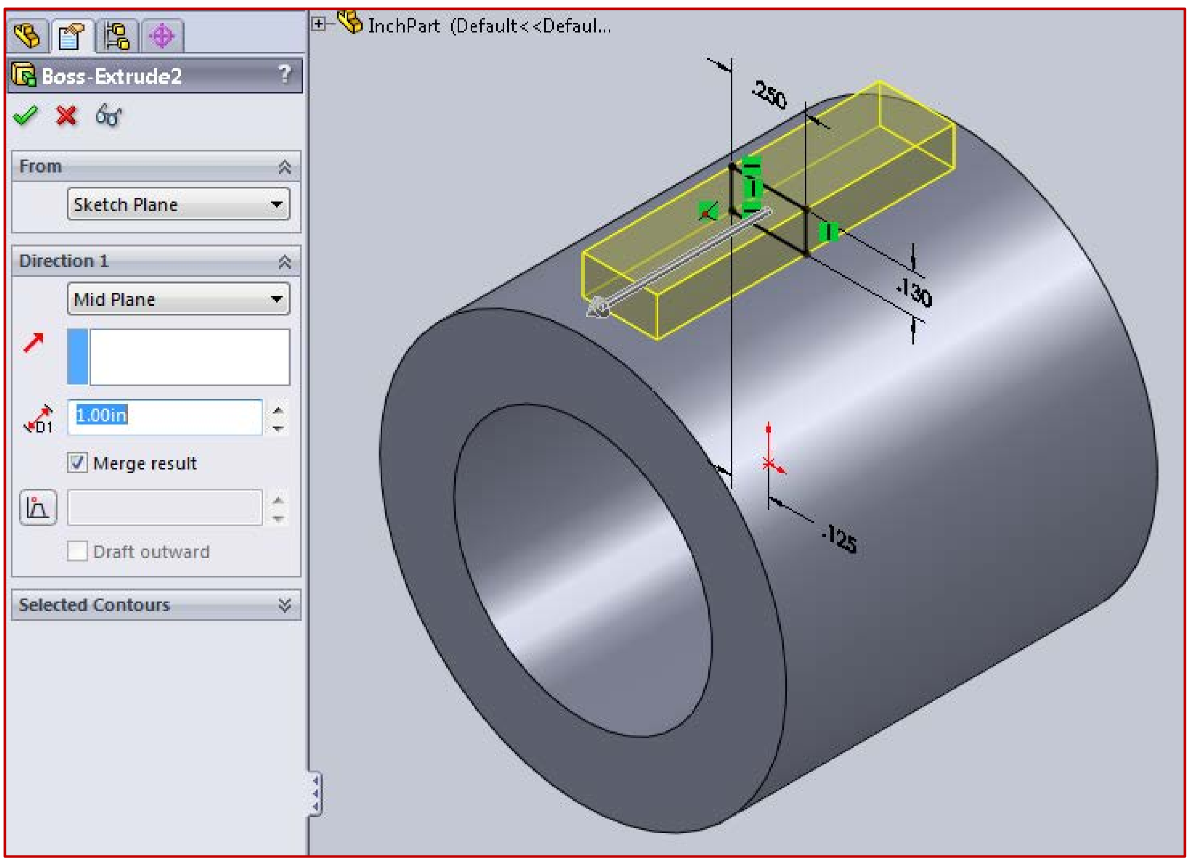


Figure 6.2 – Wheel Bump Extrusion

Step 39: Next, we will copy the bump into a circular pattern.

- ✓ Click on the ***Linear Pattern→Circular Pattern*** icon in the ***Features toolbar*** in the CommandManager. The first dialog box is the axis around which the pattern rotates.
- ✓ Click on the dialog box to make it active.
- ✓ Next, click the origin of the part in the center of the wheel. If an axis does not show in the options dialog box, it is because ***Temporary Axes*** are not visible.
- ✓ To make the axis of rotation visible, click on ***View→Temporary Axes*** on the **Main Drop-down Menu**. The origin of the part and axis of rotation can be seen now and can be used to create the pattern.
- ✓ Click the input box again to make it active and then the temporary axis at the center of the part. This will fill the input box.
- ✓ To create 12 equally spaced repetitions including the original bump, fill all the option boxes in the dialog box as shown in Figure 6.3. Notice that there will be 12 bumps equally distributed around the outside surface of the cylinder.

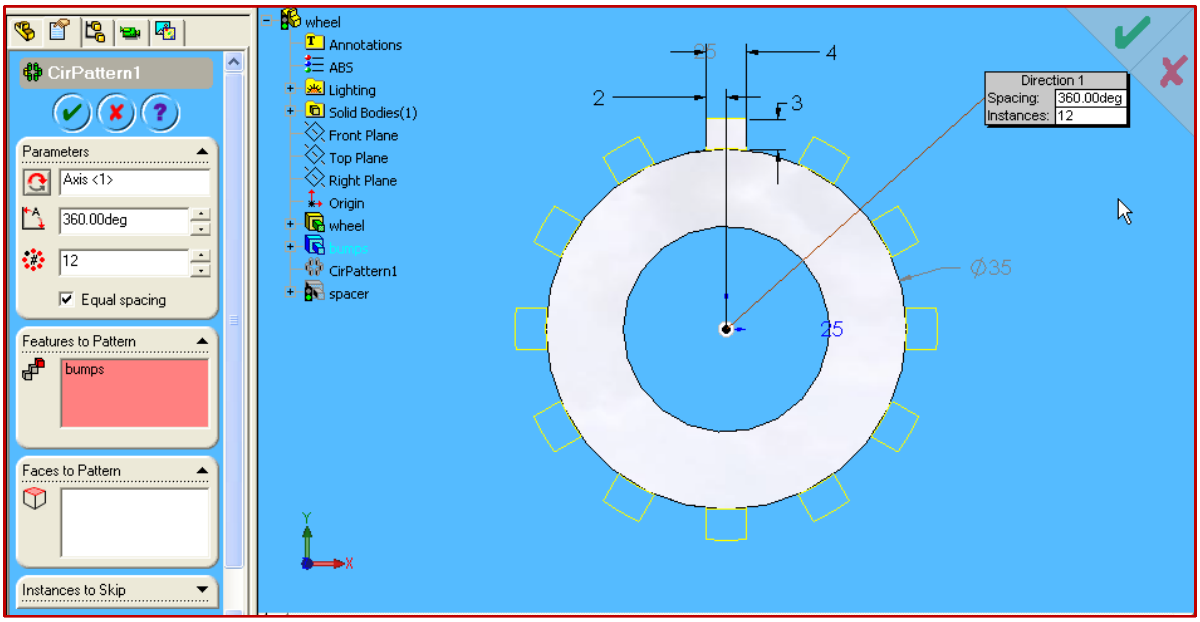


Figure 6.3 – Preview of the Circular Pattern of Bumps

- ✓ Click the ***Feature to Pattern*** input box and the ***Extruded Boss/Base*** feature in the ***FeatureManager design tree***.
- ✓ Notice that the ***FeatureManager design tree*** moved to the right of the dialog box.
- ✓ Click the + sign to expand the ***FeatureManager design tree***.
- ✓ Click the feature we extruded in **Step 37** at the bottom of the design tree.
- ✓ Click the check mark to accept the change and before saving your work, record on the Design Journal the date when you added the bumps.
- ✓ Verify that your work was saved as SKBD121.

6.4 Configurations

In Lesson 5 we created two spacers. The Top spacer is 0.38 inch in thickness and the Bottom spacer is 0.3125 inch in thickness. Because the two spacers are so similar, SolidWorks allows the storing of the model information as variations or configurations of the basic donut design. In the following two steps we will create a solid model with two configurations to replace solid models SKBD113 and SKBD115. We will call the new solid model SKBD116.

Step 40: Open SKBD113 (the Top spacer) and create a new configuration.

- ✓ Click the ConfigurationManager Tab (see Item 9 in Figure 1.4).
- ✓ Right-click the first line in the ConfigurationManager tree.
- ✓ This will present the input menu.
- ✓ Type BottomSpacer on the first input box for the ***Configuration name***.
- ✓ Type any appropriate information in the next two input boxes for ***Description*** and ***Comments***.
- ✓ On the pull-down menu, select ***Configuration name***.
- ✓ Expand ***Advanced Options*** and check ***Suppress operations*** so that different configurations can have different geometry.
- ✓ Click the check-mark to accept.

Step 41: Rename and change dimensions of configurations.

- ✓ You should have two configurations now. One is called ***Default*** and the other ***BottomSpacer***.
- ✓ Click slowly two times on the ***Default*** configuration. This will allow changing the name.
- ✓ Type TopSpacer to replace ***Defaul***t.
- ✓ Experiment by double-clicking alternately on TopSpacer and BottomSpacer. This will alternate between the two.
- ✓ Make active the BottomSpacer by double clicking the new configuration.
- ✓ Click the ***FeatureManager Tab*** (see Item 9 in Figure 1.4).
- ✓ Right-click the ***Extruded Boss/Base*** feature on the Design tree.
- ✓ Select ***Edit Feature*** to show the input for ***Extruded Boss/Base***.
- ✓ Type the new thickness (D1) of 0.50 inch.

✓ Check “This configuration” to make this thickness unique to the BottomSpacer.

✓ Click the check-mark to accept.

✓ Return to the ConfigurationManager and double-click on the two configurations to see how they change thickness.

✓ Save the model as SKBD116 and “Spacers” for ***Title***.

Practice Exercises

1. Create the part **Tire.sldprt** with the pockets for the bumps in the wheel**.** Use the dimensions in drawing SKBD122. Hints: Use the following commands, ***Temporary Axes***, ***Linear Pattern→Circular Pattern*** on the ***CommandManager***
2. Modify the skateboard deck to have two configurations. One in which the material is pine and the other made of maple.

Questions

1. Explain the need for the “bumps” in the wheel and the pockets in the tire.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor for Beginners

- Rollback Bar and Order of Features

Videos from SolidProfessor Core Concepts for Parts and Assemblies

- Automatic Relations
- Numeric Sketch Input
- Task Pane Resources
- Rapid Sketch
- Configurations - Intro
- Configurations - Dimension Changes
- Configurations – Feature Suppression
- Configurations - Table

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks tutorials: http://www.solidworks.com/sw/resources/solidworks-tutorials.htm

SolidProfessor tutorials: http://www.solidprofessor.com/

Section III – Modeling Complex Parts

Lesson 7 – Modeling the Truck Axle

7.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Model complex parts by breaking them into multiple simple parts.
- ✓ Learn to create a tapered extrusion.
- Explain how to create sketch planes.
- ✓ Learn to create and explain the command ***Trim Entities***.
- Explain and use the command ***Insert→Annotations→Cosmetic Thread***.
- Explain and use the ***Mirror*** command on the ***Features*** tab on the ***CommandManager***.
- Explain and use ***Multibodies***.

7.2 Introduction

Complex parts must be planned. They are created by combining simple solids. In many complex parts, it is necessary to create new sketch planes.

7.3 Modeling the Truck Axle

The truck is a combination of four solid bodies:

1) the axle,
2) a seat for the two rubber spacers,
3) a reinforcing rib to join the axle and the seat, and
4) a tapered pivot.

See drawing SKBD112 in Appendix A for reference.

Step 42: Create the middle shaft of the axle.

✓ Select the ***Right*** plane, click the ***Sketch*** command on the ***Sketch tab*** of the ***CommandManager*** and then click the ***Circle*** command.

✓ Use ***Smart Dimension*** to make the diameter 0.63 inch.

✓ Use ***Extruded Boss/Base*** to create a cylinder 2.5 inches long. Use the ***Blind*** extrusion option.

✓ Click the check mark to accept.

Step 43: Create the smaller diameter section of the axle.

- ✓ Click one of the end faces of the cylinder.
- ✓ In the ***View Heads-Up*** toolbar click ***View Orientation→Normal To***.
- ✓ Click the surface to make it active.
- ✓ Click the ***Sketch*** command in the ***Sketch tab*** of the ***CommandManager.***
- ✓ Click the ***Circle*** command on the ***Sketch tab*** of the ***CommandManager***.
- ✓ Make the diameter 0.25 inch.
- ✓ Use ***Extruded Boss/Base*** to extrude another cylinder to a length of 1.00 inch. Use the ***Blind*** option.
- ✓ Click the check mark to accept.
- ✓ Click the ***Mirror*** command on the ***Features tab*** of the ***CommandManager***. This will mirror the ½ axle we have into the full length.
- ✓ Click the axle to fill the ***Bodies to Mirror*** input box. The ½ axle consists of the two merged cylinders created thus far.
- ✓ The mirror plane is the ***Right*** plane
- ✓ Click the check mark to accept.

Step 44: Create the threaded tip of the axle.

- ✓ Select the end face of the second cylinder you created as the new sketch plane.
- ✓ Click ***View Orientation→Normal To*** in the ***View (Heads-Up) toolbar***.
- ✓ Click the edge of the circle to make it active.
- ✓ Click the command ***Sketch*** on the ***Sketch tab*** of the ***CommandManager.***
- ✓ Click ***Convert Entities*** to add the edge of the surface to the new sketch.
- ✓ Click the check mark to accept.
- ✓ Click ***Extruded Boss/Base*** and select ***Blind.*** Type 0.75 inch for the length D1.
- ✓ Uncheck ***Merge result*** on the properties menu.
- ✓ Click the check mark to accept.
- ✓ Notice that a new folder named ***Solid Bodies*** is now visible in the ***FeatureManager design tree***. Open the folder and verify that you have two bodies. This SolidWorks ***Multibody*** model was created because we unchecked ***Merge result*** when we created the new feature.
- ✓ Select the circular surface of the new cylinder and click ***Insert→Annotations→Cosmetic Thread.*** Select ANSI Inch in the ***Standard*** input box. Select ¼-28 UNF thread in the ***Size*** input box and ***Up to next*** for end condition.

- ✓ UNC means that this is a coarse thread, the most commonly used. Fine thread (UNF) is used for high stress applications. UNF requires more “turns” to travel a distance compared to UNC and requires a little more labor to use.
- ✓ Click the check mark to accept.

Step 45: Change the appearance of the surface to simulate threads.

- ✓ Right-click the surface of the new cylinder body.
- ✓ Click the ***Appearances*** icon (this is the 4-color ball).
- ✓ Click ***Face***. The ***PropertyMenu*** appears at the left side of the screen and the ***Appearances/Scenes*** appears on the right side.
- ✓ Expand ***Appearances (color)*** on the right-hand menu.
- ✓ Expand ***Miscellaneous***.
- ✓ Click ***Pattern***.
- ✓ Click the ***screw thread*** pattern.
- ✓ Click the ***Mapping*** tab on the left hand menu.
- ✓ Select ***Axis direction: XZ*** on the pull-down menu and ***Big mapping size***. You may need to Zoom on the thread to see the difference.
- ✓ Experiment with ***Rotation***.
- ✓ Click the Mirror command again to duplicate the new body and the threads on the opposite side.

Step 46: Next, ***Mirror*** the new body across the ***Right*** plane.

- ✓ Click ***Insert→Mirror/Pattern→Mirror*** on the ***Main Pull-down Menu***, or click ***Linear Pattern→Mirror*** on the ***Feature tab*** of the ***CommandManager***.
- ✓ Select the ***Right Plane*** as the ***Mirror Face/Plane*** and the latest extrusions as the ***Features to Mirror***. See Figure 7.1.
- ✓ Click the check mark to accept.
- ✓ Notice that there are now three bodies in the ***Solids*** folder.
- ✓ It is important to know that the ***Mirror*** command works with ***Features*** or ***Bodies***, but they have to be duplicated in separate steps. ***Mirror*** will not work with ***Features*** and ***Bodies*** at the same time.

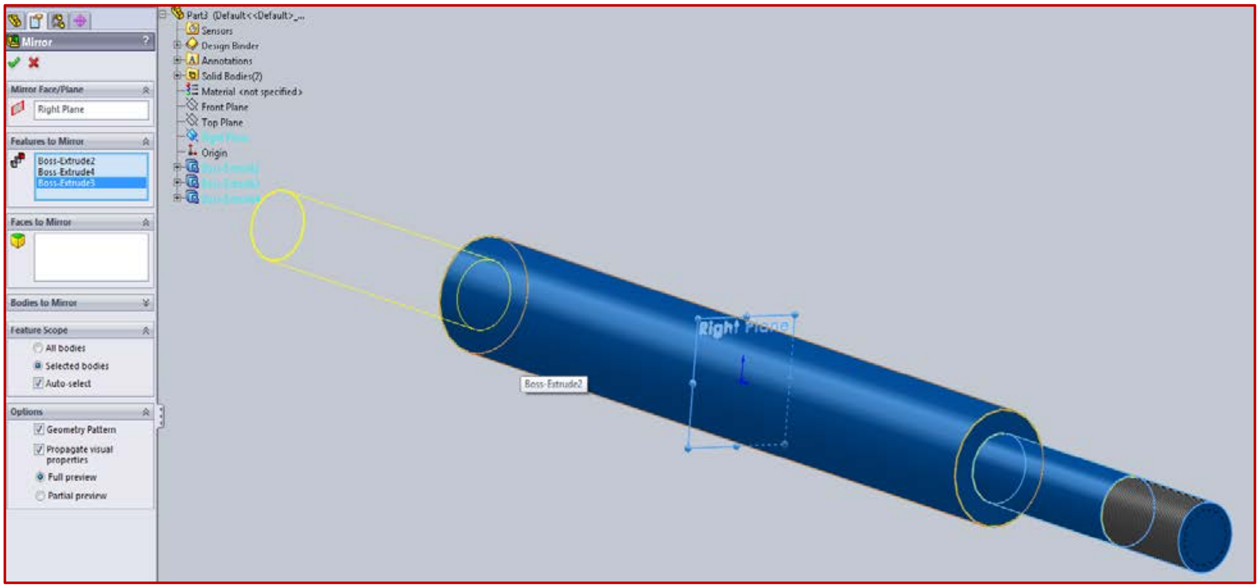


Figure 7.1 – Truck Axle

Step 47: Next, we will create a seat for the rubber spacers.

- ✓ Select the top plane and sketch a circle with the dimensions shown in Figure 7.2. (Pick the ***Top Plane***, click ***Sketch*** in the ***Sketch tab*** and then click ***Circle***).
- ✓ Warning: The circle must overlap with the axle cylinder. If the circle is tangent or detached, the two solids will not merge and you will get another body in your multibody model.

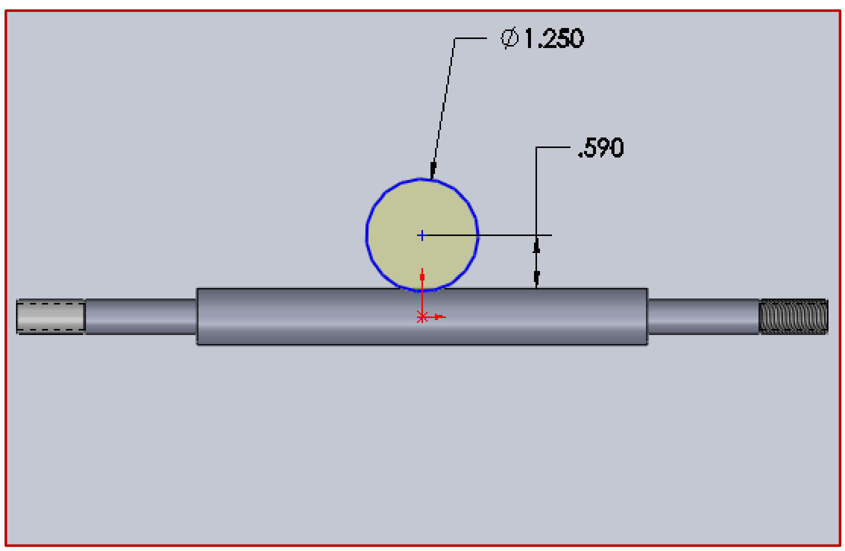


Figure 7.2 – Seat for the Rubber Spacers

- ✓ Add a vertical relation between the center of the circle and the origin to keep them aligned. Use the command ***Display/Delete Relation→Add Relation*** in the ***Sketch tab***.
- ✓ Extrude the circle using the ***Mid Plane*** option as shown in Figure 7.3. The extrusion distance is 0.38 inch.

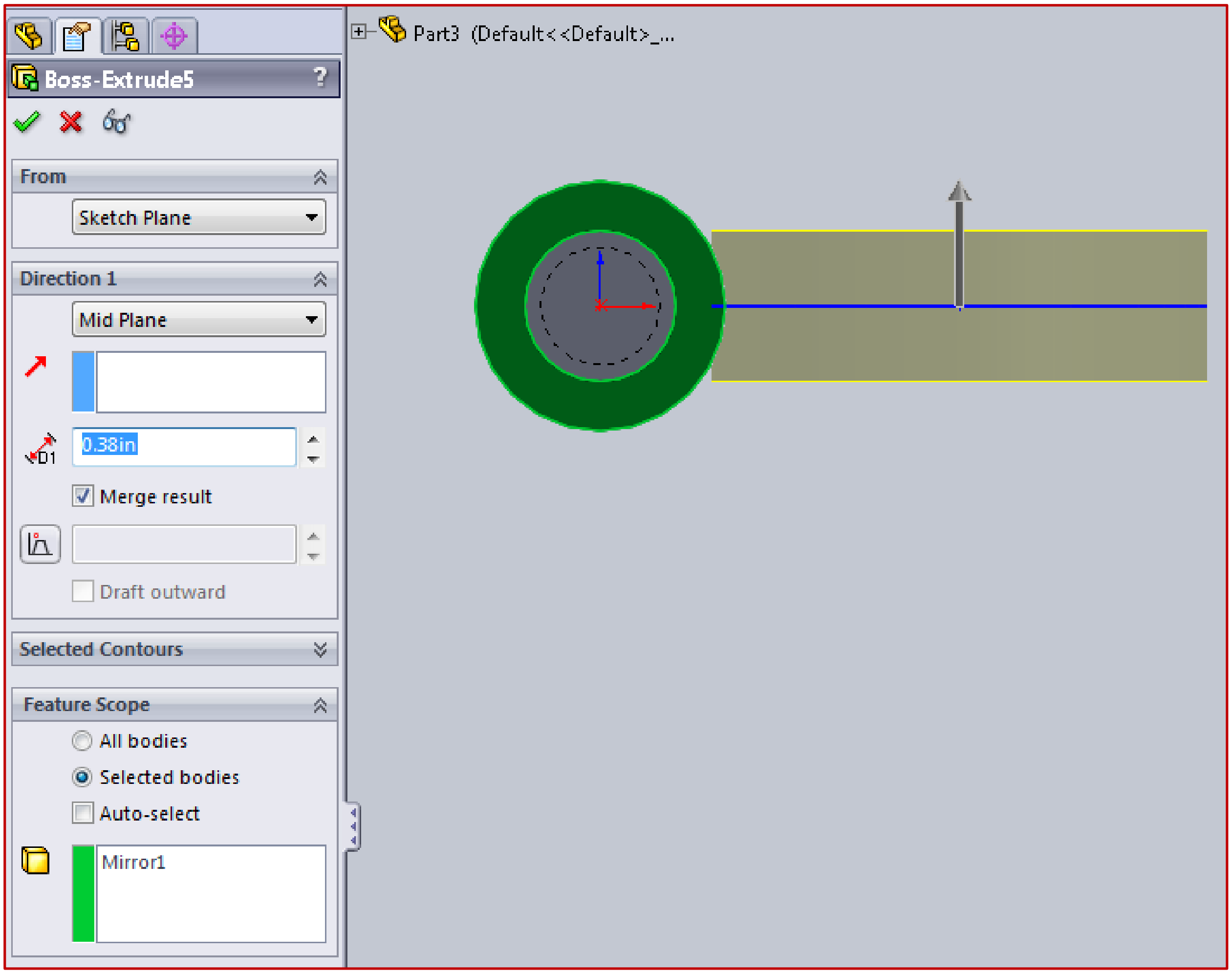


Figure 7.3 – Extruded Seat

Step 48: Create the truck pivot.

- ✓ First, we will create a sketch plane rotated 32 degrees from the ***Front Plane.***
- ✓ Click ***Insert→Reference Geometry→Plane***.
- ✓ For the ***First Reference*** select the ***Front Plane*** from the ***FeatureManager design tree***, which moved to the right of the input dialog.
- ✓ Click the angle icon and type 32degrees.
- ✓ For the ***Second Reference*** click the axis of the shaft.
- ✓ If the axis of the shaft is not visible, click ***View→Temporary Axes*** on the **Main Drop-down Menu**. The axis is now visible as a dashed line. Click the line.
- ✓ Click the check mark to accept. Remember that planes in SolidWorks are infinite. The location of the plane that appears is not important.

- ✓ The new plane created is rotated from the ***Front Plane*** 32 degrees using the shaft axis for rotation. See Figure 7.4.

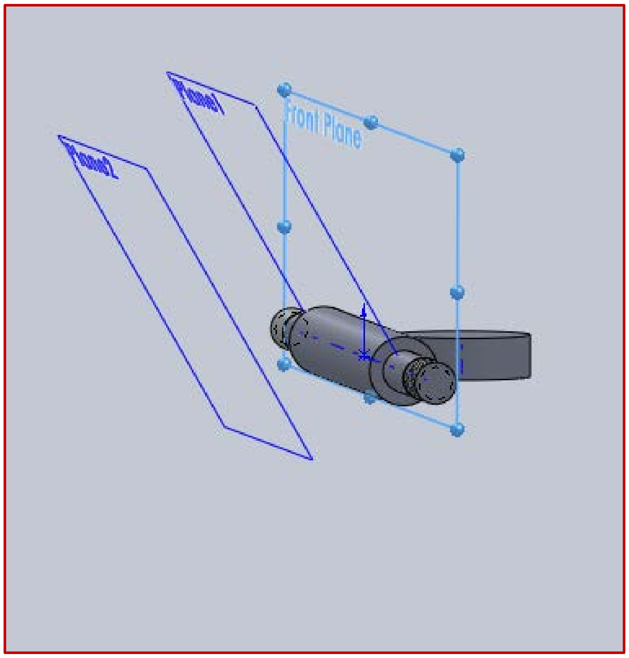


Figure 7.4 – Front, Rotated and Parallel Planes

Step 49: Create another plane parallel to the rotated plane.

- ✓ Click ***Insert→Reference Geometry→ Plane***.
- ✓ For the ***First Reference*** click the plane that we created in the previous step.
- ✓ Click the ***Offset distance*** icon to create a parallel plane 1.06 inches away.
- ✓ Click the check mark.
- ✓ Notice that rotation requires two references (the plane to be rotated and the axis of rotation) while offset needs only one reference (the offset).

Step 50: Create the base of the dome at the tip of the truck pivot.

- ✓ Verify that the new offset plane is active. Its color is different from other visible planes.
- ✓ In the ***Heads-up View*** toolbar, click on ***View Orientation*** and select ***Normal To***. The view can be rotated 180 degrees to the opposite side by clicking ***View Orientation→Normal To*** a second time.
- ✓ Click the Sketch icon on the Sketch tab of the CommandManager. Click the Circle command and draw a circle on the offset plane as shown in Figure 7.5. The diameter is 0.4 inches.
- ✓ The center of the circle is aligned with the origin of the axle.
- ✓ Notice that the origin is not on the offset plane, but if you move the cursor over the origin, ***Inferencing*** will show the location of the point in the offset plane that is aligned with the origin. If the blue coordinates are not visible, click ***View*** on the ***Main Pull-down Menu*** and verify that ***Origins*** is active.

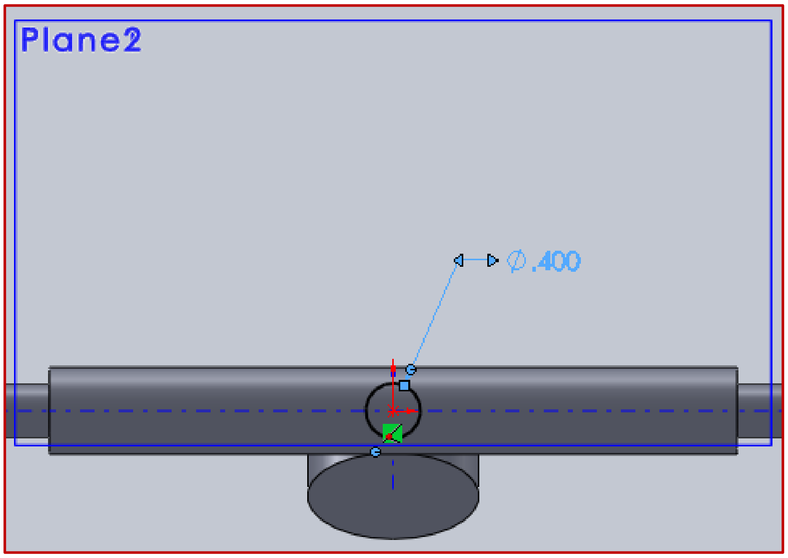


Figure 7.5 – Sketch for Pivot Tip

- ✓ Also notice that the diameter of the circle is 0.4 inch because it must match the dome of radius 0.2 inch.
- ✓ With the sketch active on the design tree, use ***Extruded Boss/Base*** to create the pivot. Use the ***Draft On/Off*** option and type 5 degrees. Select ***Up to Surface*** for the end condition. See Figure 7.6. If necessary, you can toggle the draft to match the Figure.
- ✓ Click the check mark to accept.

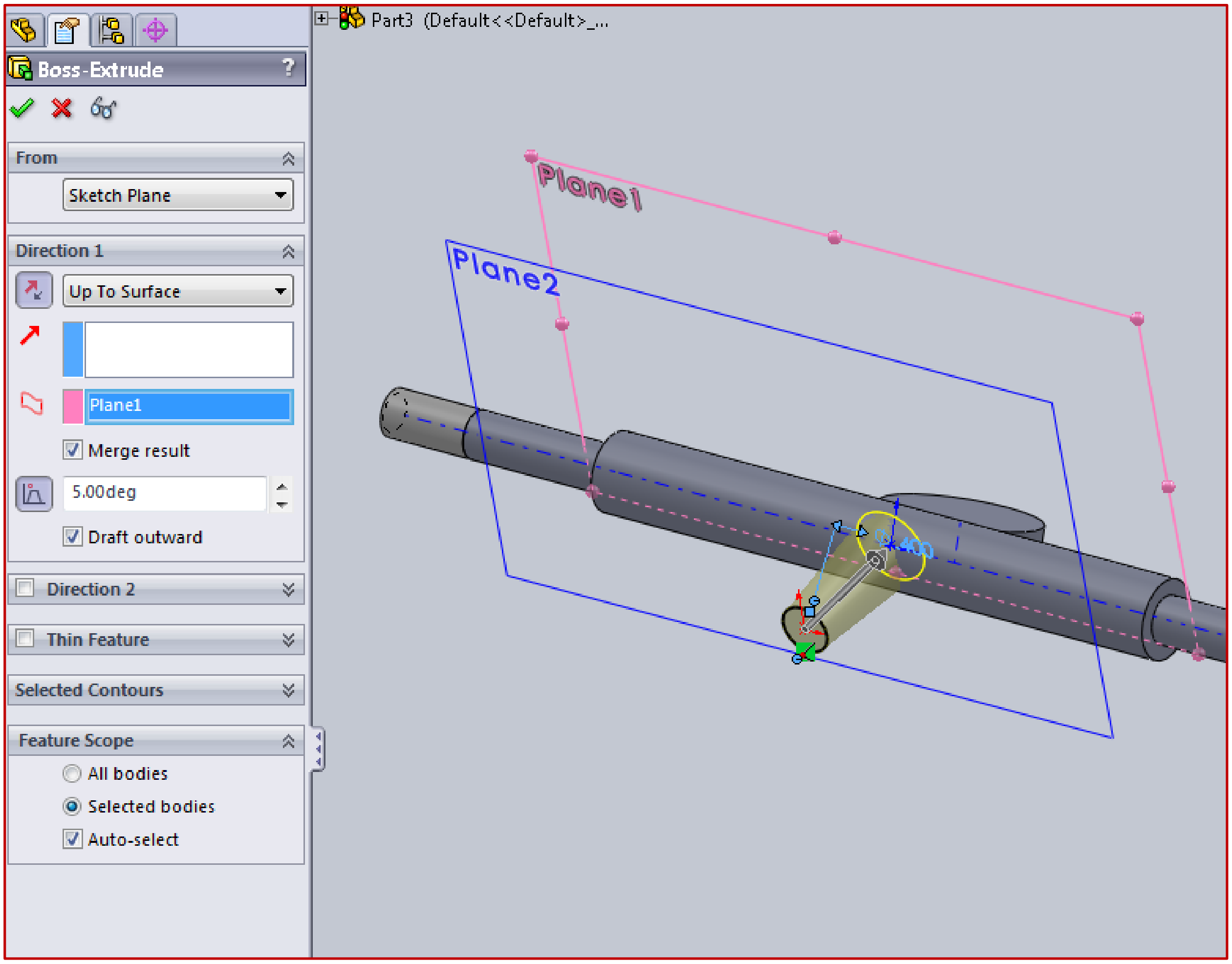


Figure 7.6 – Tapered Extrusion for Pivot

Step 51: Create a dome at the end of the pivot.

- ✓ Select the surface at the end of the pivot as the sketch surface.
- ✓ Click ***Insert→ Features→Dome***. The radius is 0.20 inch. See Figure 7.7.

For the stiffener, we will sketch in the ***Top*** plane.

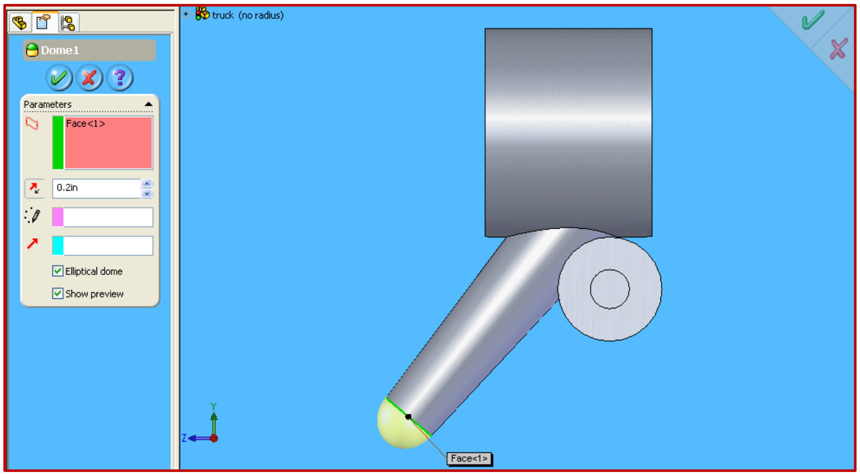


Figure 7.7 – Pivot Dome

Step 52: Create the stiffener.

- ✓ Click ***Top Plane*** and sketch the triangle in Figure 7.8.
- ✓ Pay attention to the sketch relations and angles to achieve a fully defined sketch.
- ✓ Use ***Extruded Boss/Base***. Select ***Mid Plane*** and type a thickness distance ***D1*** of 0.125 inch.
- ✓ Please note that if we use 2 decimal places in the drawing the number will show as 0.13.

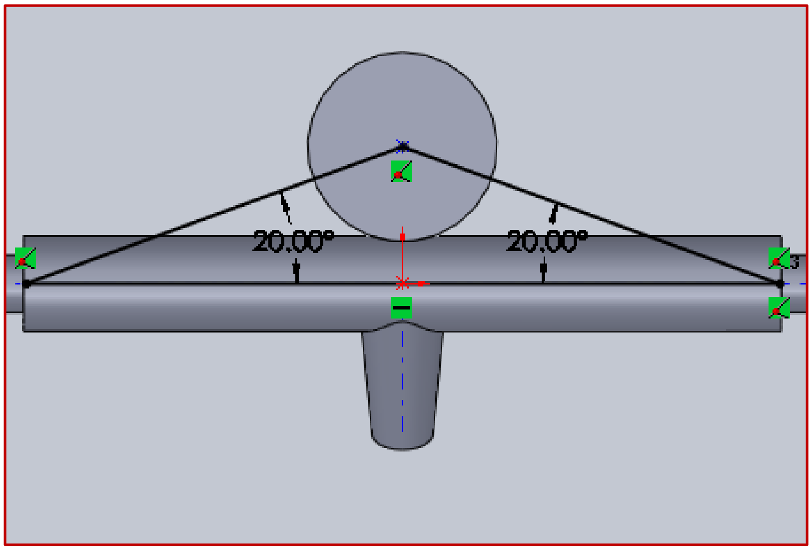


Figure 7.8 – Sketch for Stiffener

Step 53: Create the top pocket for the rubber spacers.

- ✓ Sketch a circle1 inch in diameter as shown in Figure 7.9.
- ✓ Use ***Extruded Cut*** to create a pocket for the rubber spacer. The depth is 0.125 inch.

Step 54: Use the ***Mirror*** command in the ***Features*** toolbar to create the second pocket in the opposite surface. See Figure 7.10. The mirror plane is the ***Top Plane***.

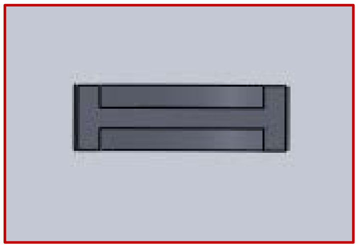


Figure 7.10 – Spacer Pockets

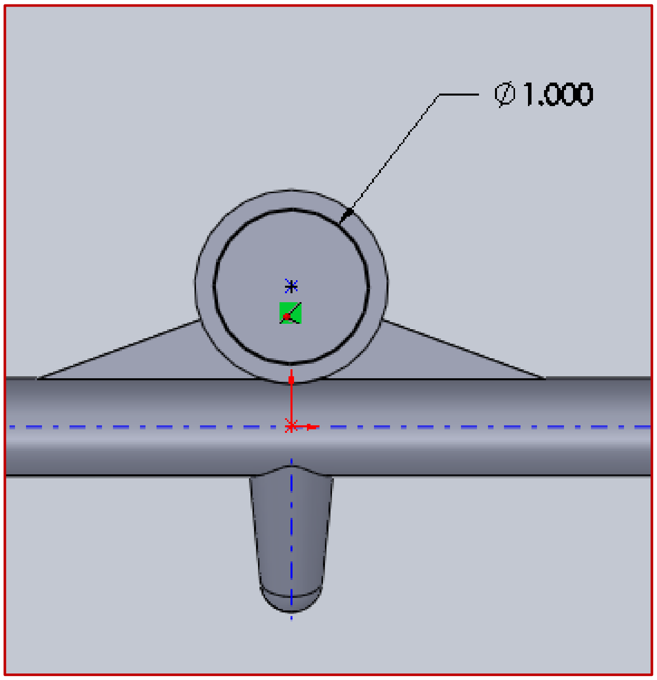


Figure 7.9 – Sketch for Pocket

Step 55: Sketch a slot for the bolt that attaches the truck to the skateboard.

- ✓ The sketch is shown in Figure 7.11. Use the ***Centerpoint Straight Slot*** on the ***Sketch*** tab to create the sketch.
- ✓ Use ***Extruded Cut*** to create the slot.

Step 56: The Truck Axle is complete. Save your work as SKBD112. Complete the appropriate entries in the ***Design journal*** and in the ***Properties*** form.

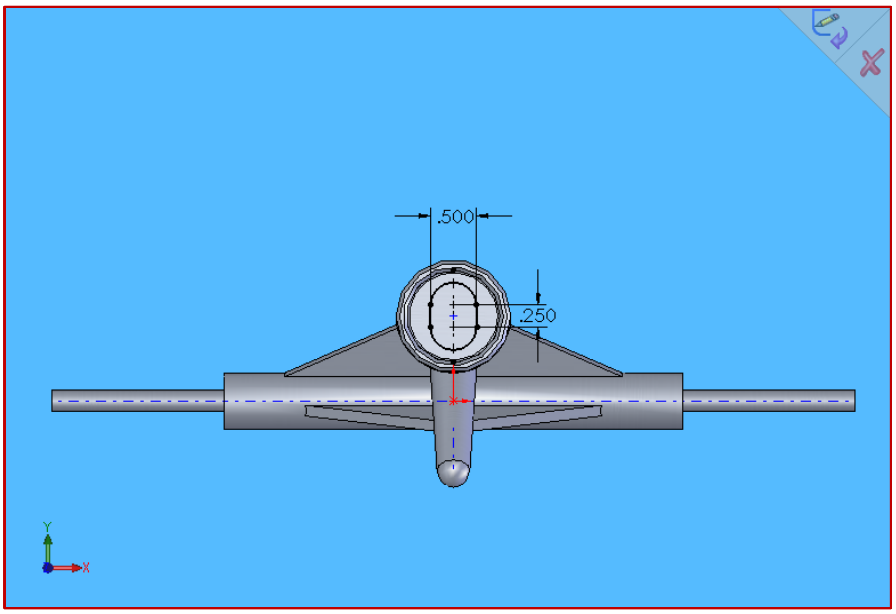
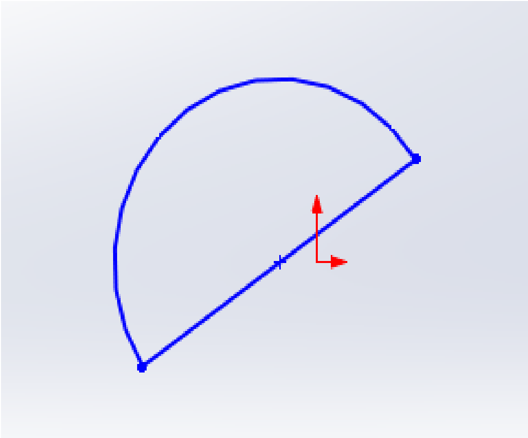
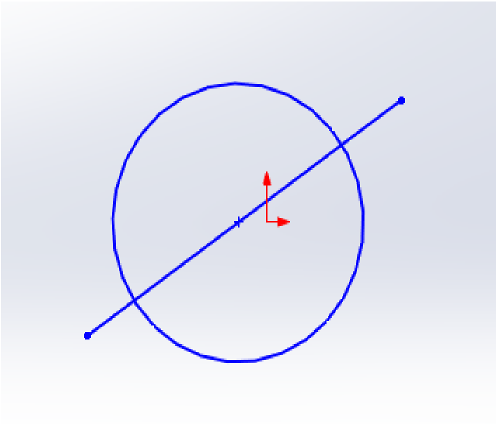


Figure 7.11 – Sketch for Slot

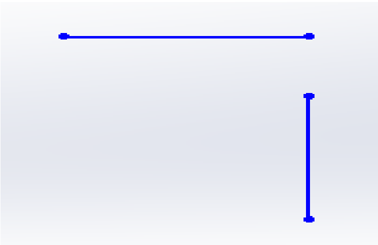
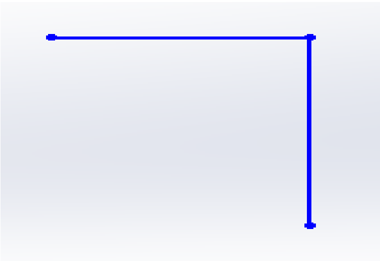
Trim Entities

The command ***Trim Entities*** can be used to cut a portion of a line or curve. You can find the icon in the ***Sketch*** tab of the CommandManager. For example, the sketch on the left can be generated from the sketch on the right by using the command ***Trim Entities→ Trim to closest*** to cut the segments that are not needed.

Extend Entities

You can find the command ***Extend Entities*** in the pull-down menu under ***Trim Entities*** in the ***Sketch*** tab of the ***CommandManager***. It can be used to extend a line or curve to meet another element (point, line or curve). It cannot be used to just make the line or curve longer into empty space. For example, the figure on the left can become the figure on the right with the use of this command to extend the vertical line.

Practice Exercises

1. Re-draw the Truck Axle modeled in this Lesson using millimeters instead of inches. Use the mmPart template and the dimensions in the detail drawing in Appendix B.
2. Create the pivot using ***Lofted Boss/Base*** instead of the tapered ***Extrude Boss/Base***.

Questions

1. Give one example of a complex part and the strategy you would use to create a model of the part.
2. What manufacturing process and material do you recommend for the truck axle?

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor Core Concepts for Parts and Assemblies

- Planes
- Trim Tools
- Mirroring Bodies
- Troubleshooting Parts – Introduction
- Troubleshooting Sketches
- Troubleshooting Features

Videos from SolidProfessor Advanced Parts

- Dome
- Multibody Parts
- Creating Multiple Bodies

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks tutorials: http://www.solidworks.com/sw/resources/solidworks-tutorials.htm

SolidProfessor tutorials: http://www.solidprofessor.com/

Lesson 8 – Modeling the Truck Base

8.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Explain how to link dimensions.
- Explain how to use equations for dimensions.
- Explain how to use the command ***Extruded Boss/Base*** when the extrusion is offset from the sketching plane.
- Explain how to extrude in two directions simultaneously.

8.2 Introduction

It is possible to link dimensions and also to use equations to define distances. In this chapter we will learn both as we create the base of the skateboard truck. We will also learn that the Extrude Boss/Base command does not have to start from the sketching plane.

8.3 Modeling the Truck Base

The truck base is a combination of three features: the rectangular base, a cylinder extrusion for the spacers and a raised Pivot pocket with a socket hole for the truck pivot. We will first create the base, then the cylinder extrusion and finally the Pivot pocket.

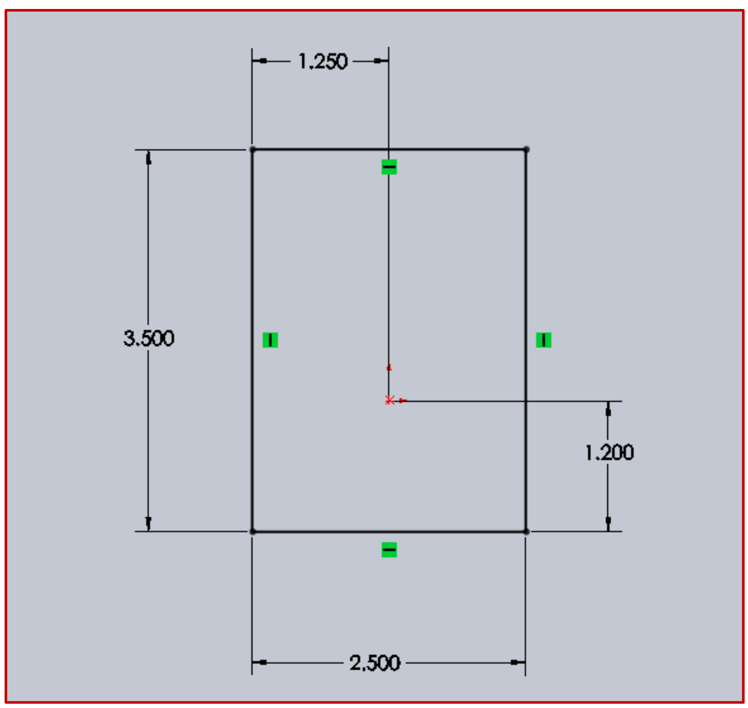


Figure 8.1 – Sketch for Truck Base

Step 57: Create the rectangular base.

✓ Open a part document and select the **Inch** template.

✓ Sketch on the ***Top Plane,*** the rectangle in Figure 8.1.

✓ Click ***Extruded Boss/Base,*** select ***Blind*** a thickness of 1/8 inch in the down direction so the origin is on the top surface.

Step 58: Create the top-right corner hole.

- ✓ Select the top surface of the base to make it active.
- ✓ Click ***Circle*** on the ***Sketch*** tab and sketch a circle on the top-right corner of the plate.
- ✓ Click ***Smart Dimension*** and then click the diameter of the circle.
- ✓ Type “0.25” on the input box.
- ✓ In your keyboard, press Enter two times to accept the value (or the check mark to accept the equation). Notice that when you press “Enter” the first time SolidWorks creates an equation.
- ✓ Enter the other locating dimensions by typing “=” and then clicking the diameter of the circle.
- ✓ Note: We are following the rule of thumb that the center of the hole should be a minimum of one diameter from the edges of the plate. This is a design rule of thumb that prevents stress concentration and possible failure if the hole is too close to the edge. You can use a larger separation. In addition, the bolt heads should not extend past the edge of the plate and we can make the distance larger.

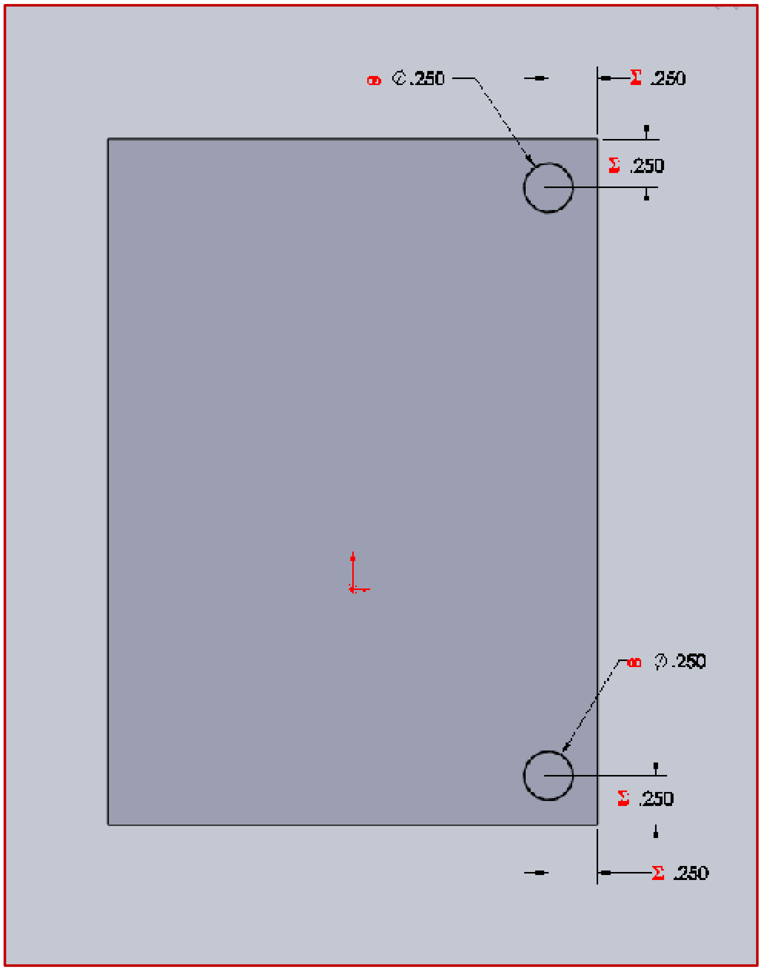


Figure 8.2 – Corner Holes

Step 59: Add a second hole at the bottom-right corner of the plate.

- ✓ Click the ***Circle*** command again and draw another circle in the lower corner of the Base.
- ✓ Click ***Smart Dimension*** and then click the circumference of the new circle.
- ✓ In the input box type “=”.
- ✓ Click the diameter of the previous circle. This will enter the name of the previous circle into the input box. Now the two diameters are the same.
- ✓ Enter the other locating dimensions in the same fashion, typing “=” and then clicking the equivalent dimension in the first circle.
- ✓ Your sketch should look like Figure 8.2. The ∑ indicates that we have equations for the dimensions.

- ✓ Click the check mark to accept the sketch.

Step 60: Use the ***Mirror Entities*** command on the ***Sketch*** tab to create the other two holes.

- ✓ Click the pull-down menu for the line command and select the centerline.
- ✓ Sketch a vertical centerline from the origin. The length of the line is not important. You will notice that the end point is blue, or undefined. You can constrain it by using ***Smart Dimension***, or by clicking on the point, which opens the ***PropertyManager,*** and selecting ***Fix*** in ***Add Relations***.
- ✓ Click ***Mirror Entities*** on the ***Sketch*** tab.
- ✓ For ***Entities to mirror***, select the two holes.
- ✓ For ***Mirror about***, select the centerline.
- ✓ Click the check mark to accept the ***Mirror*** command.
- ✓ Click ***Extruded Cut*** on the ***Features*** menu. Select ***Through All*** or ***Up to Next***.
- ✓ Click the check mark to accept.

Fix Relation

The "Fix" relation can be used to fix the location of the points in a spline. They are also useful to fix the end points in a construction line that will be used for the mirror or sweep commands. They should not be used instead of proper modeling using dimensions and ***Sketch Relations***.

Sketch Clutter

It is possible to have too much information in your sketch. Figure 8.2, for example shows dimensions and sketch relations. To turn-off the sketch relations, use ***View→Sketch relations*** on the **Main Drop-down Menu** to toggle on and off. Other useful toggles are ***View→Temporary Axis*** and ***View→Hide All Types***.

Changing Linked Values and Equations

Links can be broken and equations can be changed. Double click on the number and use the smart dimension pull-down menu to make changes.

Step 61: The cylinder extrusion with the pocket for the bottom spacer is tilted at an angle from the horizontal surface of the base. First, we must create a sketch plane. Then, we will sketch a circle and extrude the cylinder.

- ✓ Select the top surface of the base for sketching and sketch a centerline as shown in Figure 8.3. The line passes though the origin (a coincident relation). If the origin is not visible, click ***View*** on the ***Main Pull-down Menu*** and click the ***Origins*** icon.
- ✓ Click the pencil to accept the sketch.
- ✓ To create a new plane by rotating the top surface about the centerline, first click the Top surface of the base.

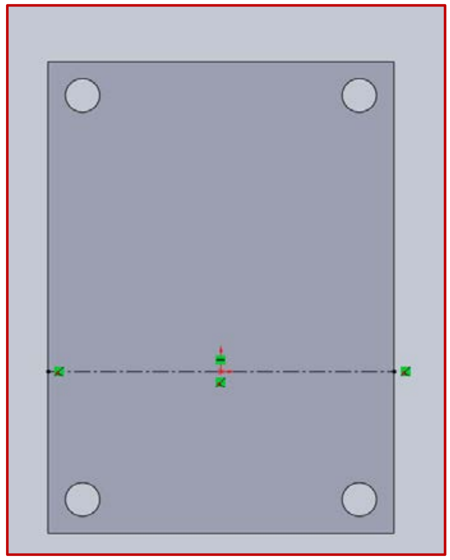


Figure 8.3 – Axis of Rotation for Top Plane

- ✓ Click ***Insert→Reference Geometry→Plane*** and enter the top surface and the centerline we created in the previous step in the input boxes. This will rotate the top surface with the centerline as the axis of rotation. Rotate the top surface 5 degrees clockwise. See Figure 8.4.
- ✓ Experiment with the ***Flip*** option to see how it changes the new plane. Your new plane should match Figure 8.4.
- ✓ Click the check mark to accept.

Step 62: Select the plane we created in the previous step and sketch a circle with the center in the origin and a diameter of 1.25 inch.

Step 63: Extrude the cylinder in both directions as shown in Figure 8.4. Click the check mark to accept.

- ✓ In one direction the extrusion is ***Blind*** and in the other it is ***Up To Surface***. The ***Blind*** extrusion is 1.00 inch. For ***Face/Plane*** click the ***Top*** surface of the Base.
- ✓ Notice the 1 degree draft on the extrusion.

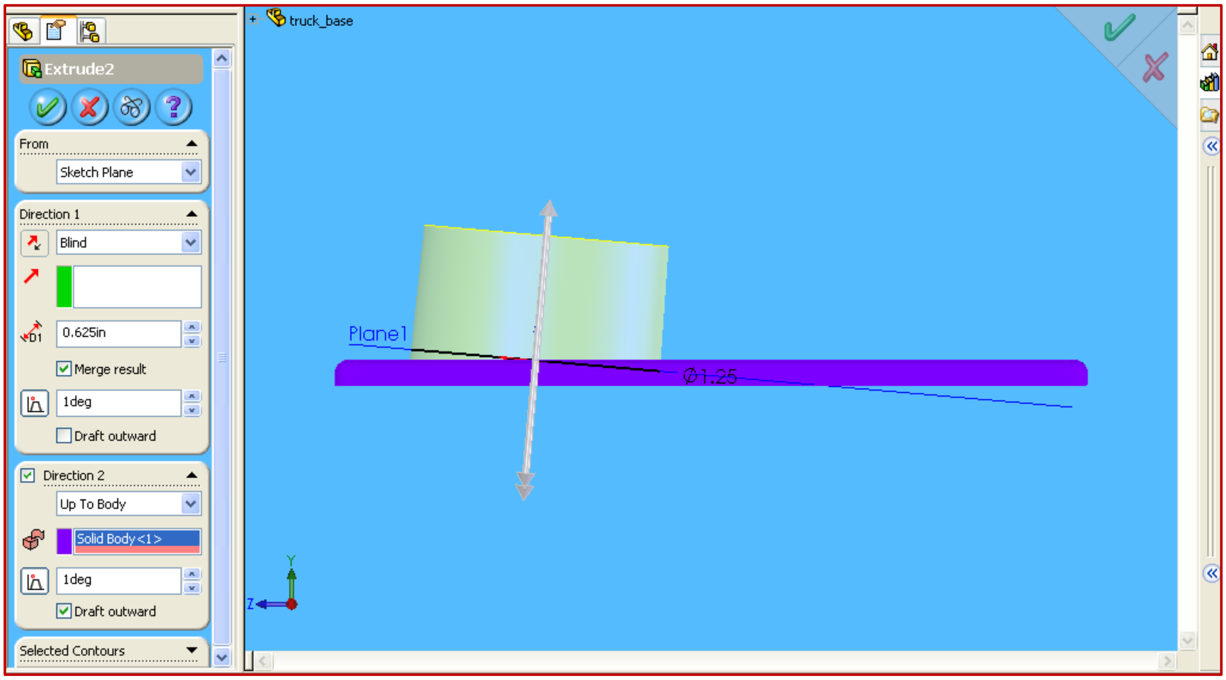


Figure 8.4 – Truck Base

Step 64: Create the pocket for the rubber spacer (Part SKBD115).

- ✓ Select the top surface of the newly created cylinder for sketching.
- ✓ In the ***CommandManager→Sketch*** toolbar click on ***Offset Entities***. See Figure 8.5.
- ✓ In the ***Offset Entities*** dialog box, type 0.120 inch offset.
- ✓ Click the check mark to accept.
- ✓ Click ***Extruded Cut*** in the ***Features*** toolbar of the ***CommandManager.*** See Figure 8.6. The depth of the pocket is 0.125 inch.

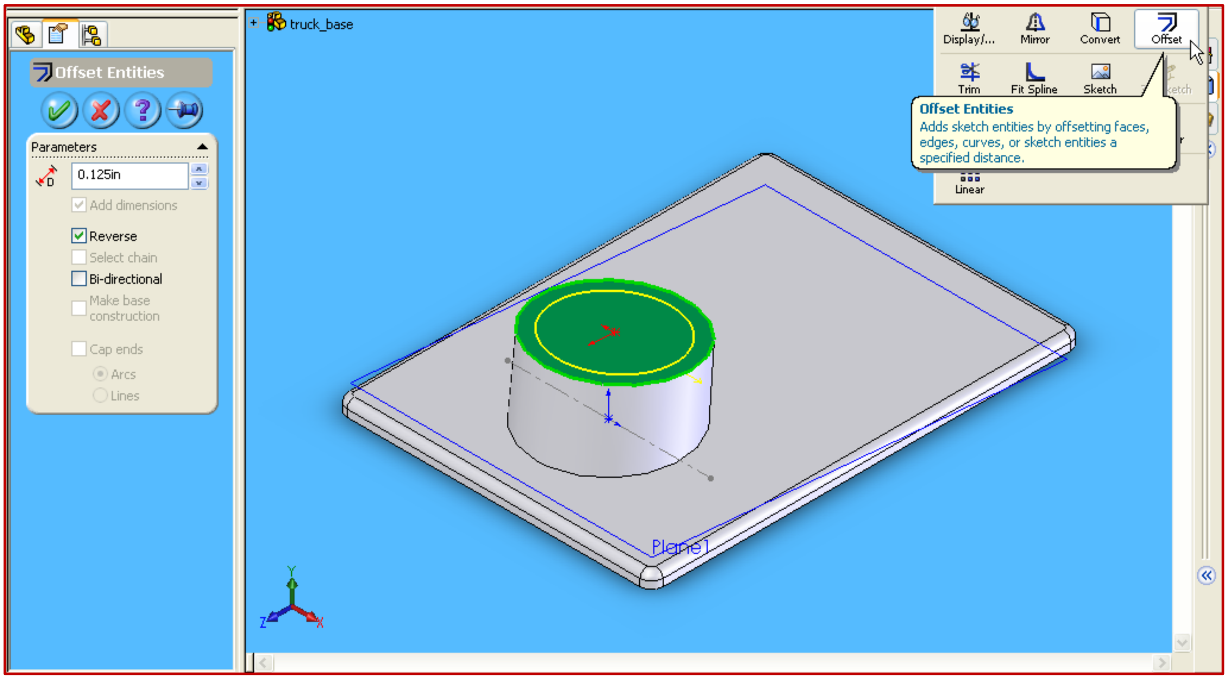


Figure 8.5 – *Offset* Command

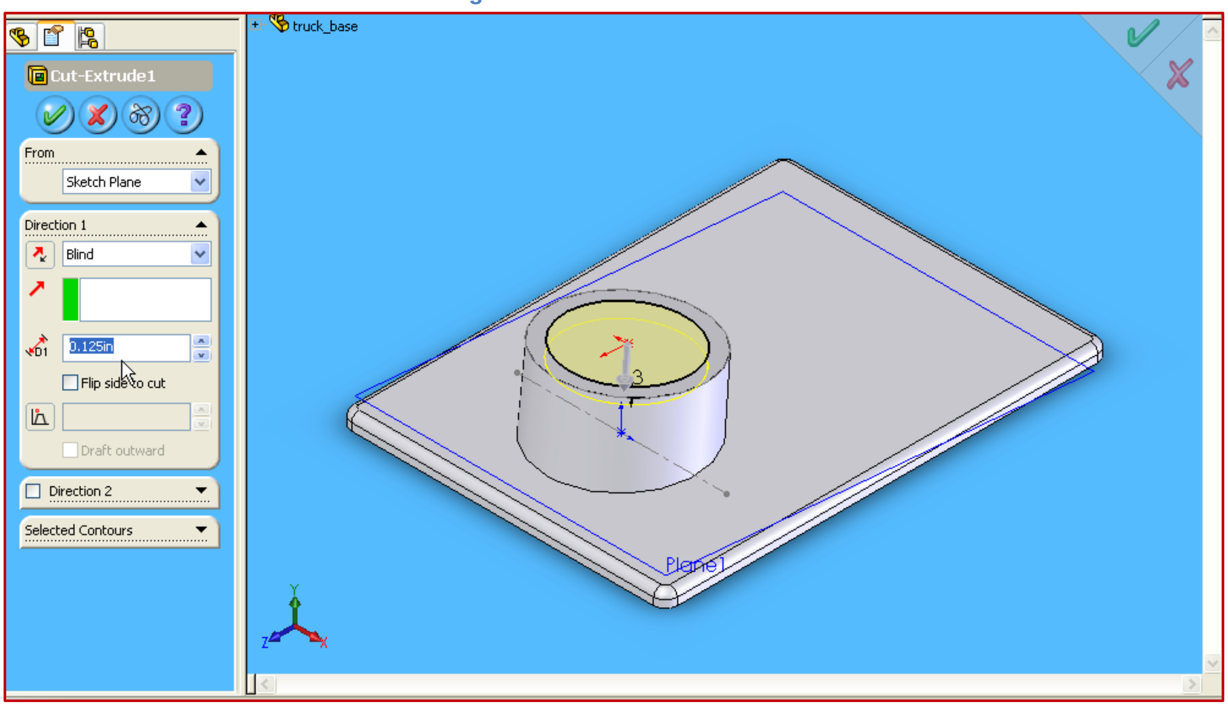


Figure 8.6 – *Extruded Cut* Pocket

Step 65: Select the surface of the pocket and sketch a circle 3/8 inch in diameter with the center coincident with the origin, and click ***Extruded Cut*** using ***Through All***. Remember to use ***Smart Dimensions*** to enter the diameter and add a collinear relation if needed. See Figure 8.7. This is for the kingpin that will attach the Truck Base to the Deck.

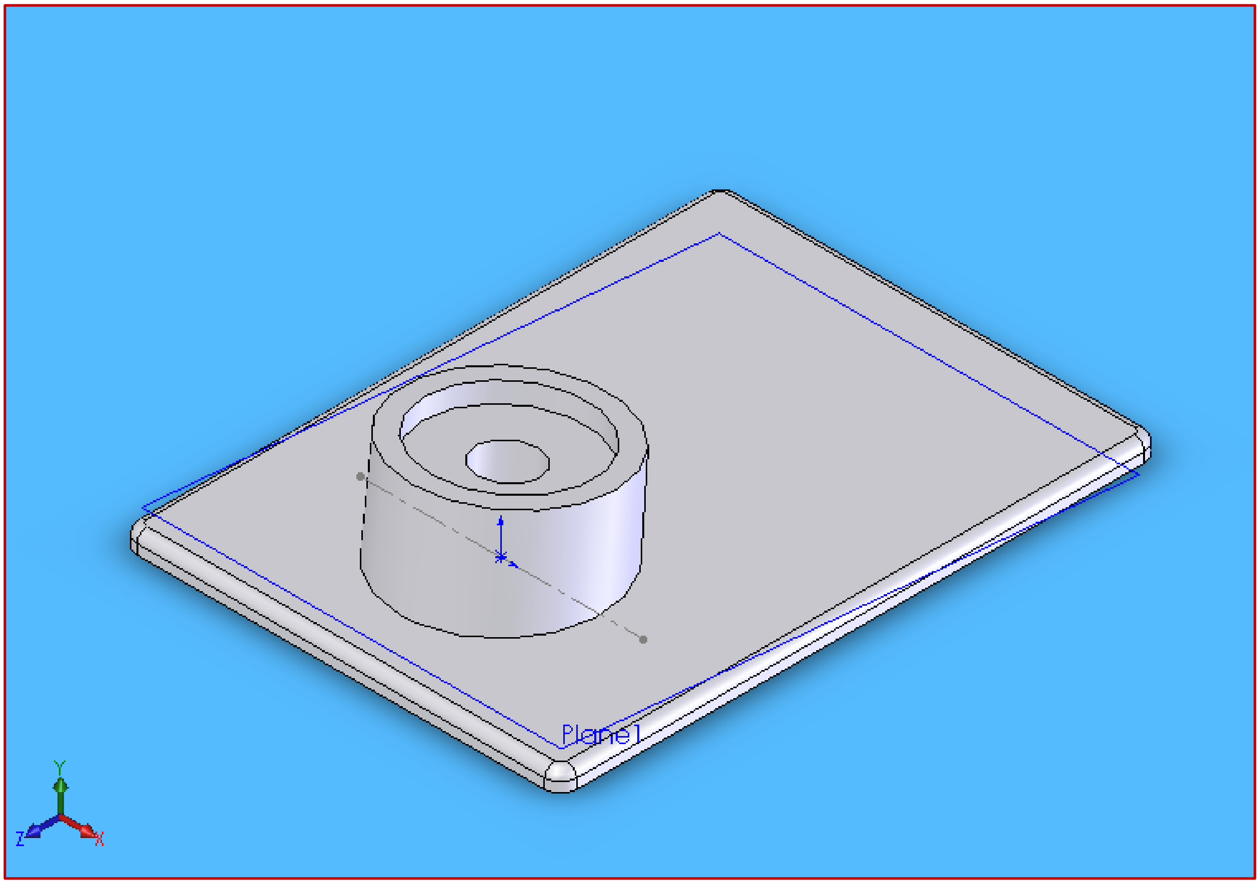


Figure 8.7 – Hole for Bolt

Step 66: Select the surface of the pocket again and sketch the square shown in Figure 8.8. The sides of the square are 0.75 inch and it is centered on the origin.

- ✓ Use the ***Center Rectangle*** option.

Step 67: Click on the ***Extruded Cut*** icon in the ***Features*** toolbar and select the options in the dialog box on Figure 8.9.

- ✓ In Figure 8.9, notice that the pocket starts at an offset of 0.20 inch from the sketch surface and ends with the ***Up To Next*** end condition in Direction 1.
- ✓ Click the check mark to accept.

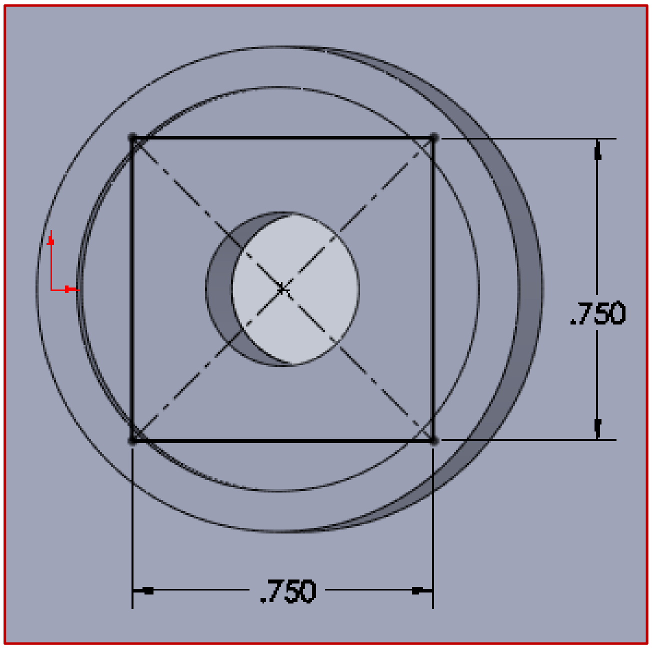


Figure 8.8 – Rectangle Sketch for Bottom Pocket

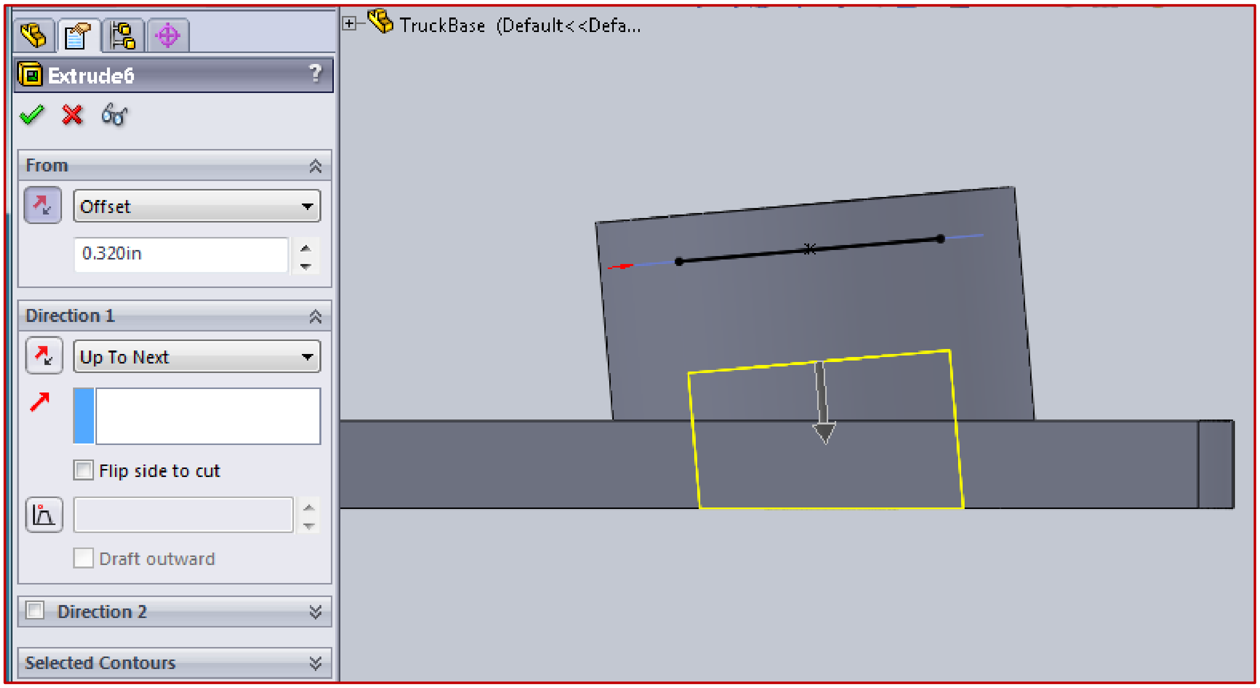


Figure 8.9 – Bottom Pocket

Step 68: To create a pocket for the Truck Pivot, we will start by creating a plane at a 128 degrees tilt from the top face of the base. Then we will sketch a cross-section at the end face of the base and then extrude the cross-section up to the plane created. See Figure 8.10.

✓ Select the top face of the base as the sketch plane.

✓ Create a centerline across the face of the base at a distance of one inch from the edge of the base. See Figure 8.10.

✓ Click the Pencil to accept the sketched centerline.

✓ Next, create a new plane by rotating the top surface of the base 128 degrees clockwise. See Figure 8.10. Use the command Insert→Reference Geometry→Plane with the top face of the Base as the ***First Reference*** and the centerline created earlier as the ***Second Reference***.

✓ Click the check mark to accept.

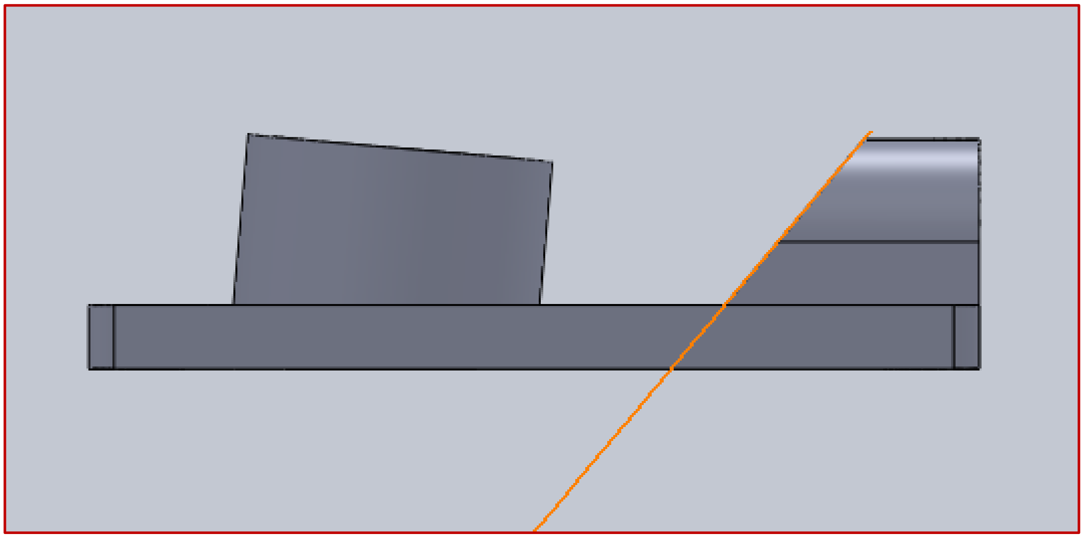


Figure 8.10 – Extrusion for the Truck Pivot Pocket

Step 69: Sketch the cross section at the end of the Truck Base. See Figure 8.11.

- ✓ Select the vertical surface on the back of the rectangular base to make it active.
- ✓ Sketch the cross section shown in Figure 8.11. Notice that the center of the arc and the origin have a vertical relation. Also, the end of the arc and the straight lines are tangent.
- ✓ Use the commands ***Smart Dimensions*** and ***Display/Delete Relations→Add Relation*** to add the relations shown. The order you use is important. If the sketch is distorted by a command or relation, click **Undo** and try another sequence.

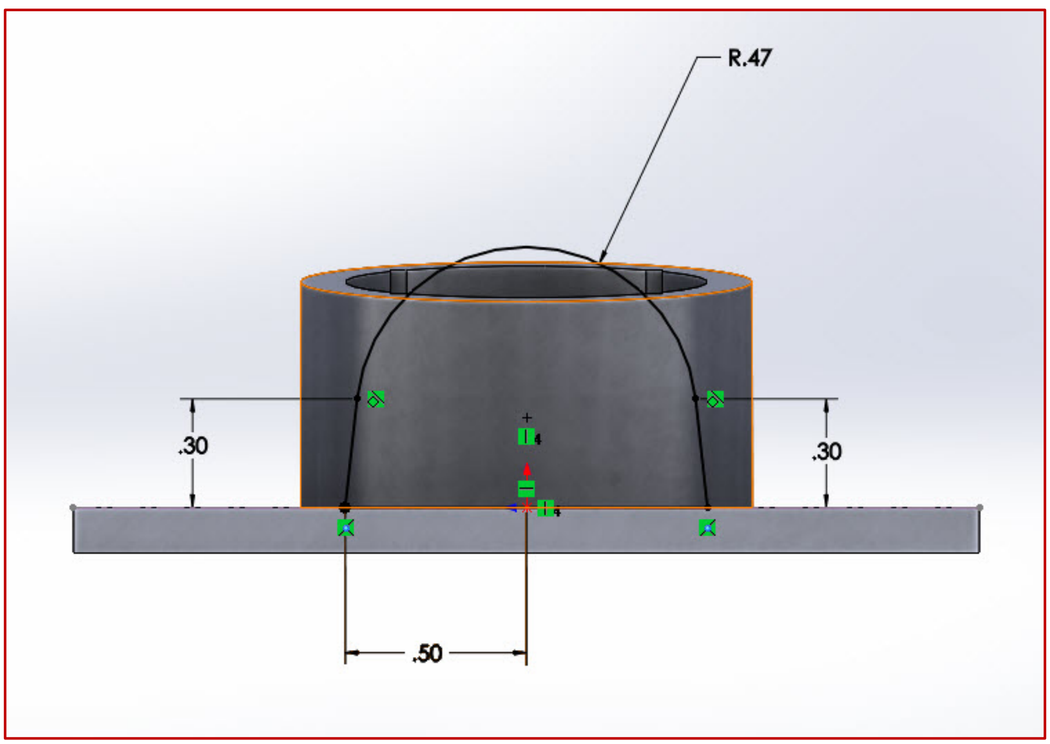


Figure 8.11 – Sketch for the Back of the Pivot Pocket

Step 70: Extrude the cross-section from the sketch plane to the plane that was rotated earlier to 128 degrees. The Extruded Boss/Base input panel is shown in Figure 8.12. Notice that the extrusion is ***Up To Surface*** and the surface is the plane created earlier.

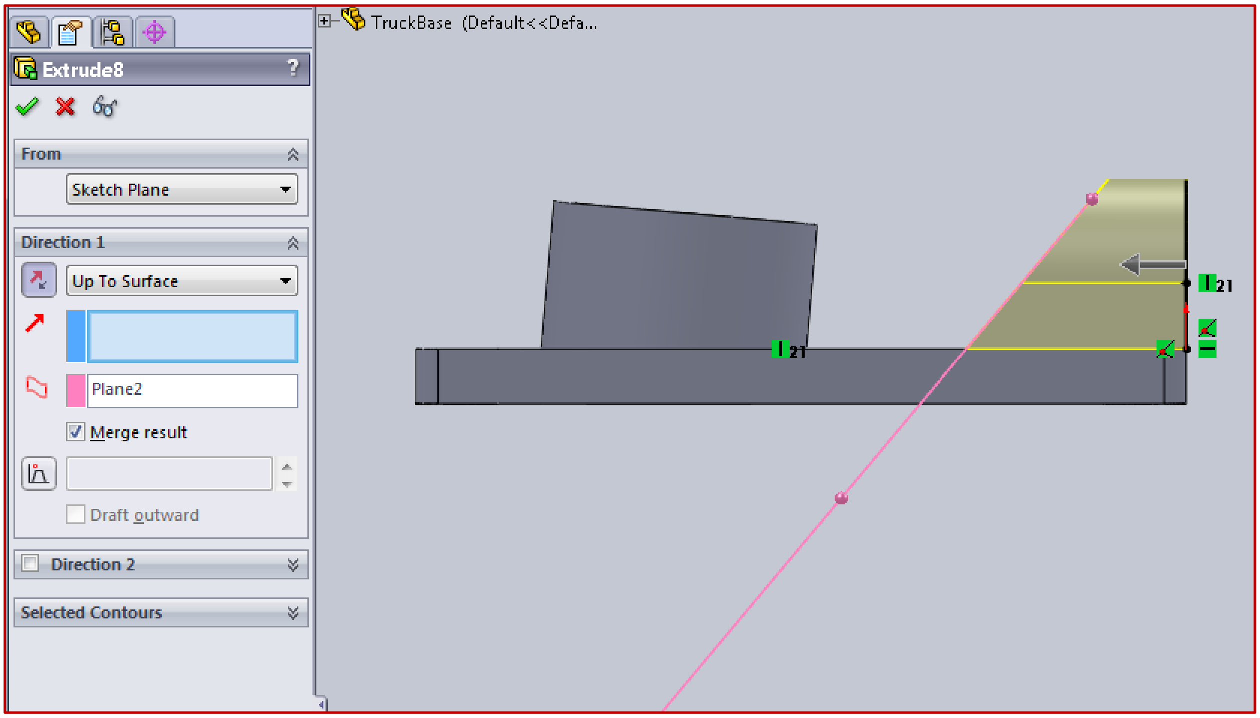


Figure 8.12 – Extrusion of the Pivot Pocket

Step 71: Create a dome cavity for the Truck Pivot.

- ✓ Select the Front face of the extrusion created on **Step 70** and click Circle to sketch the circle shown in Figure 8.13.
- ✓ Use the ***Smart Dimensions*** command to get the sketch.
- ✓ If your sketch is not fully defined (black), Use ***Add Relation*** and ***Vertical*** to align the center of the circle and the center of the base.
- ✓ Use ***Extruded Cut*** to a depth of 0.06 inch.
- ✓ Click the check mark to accept.

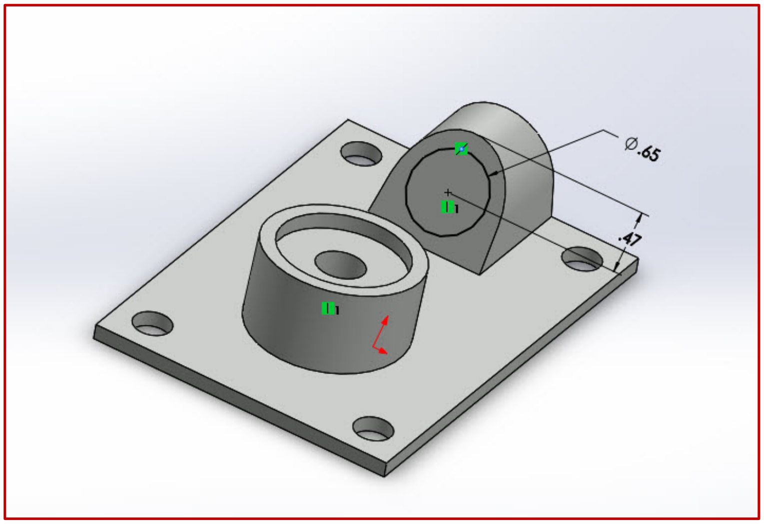


Figure 8.13 – Sketch for the Dome

- ✓ Select the surface at the bottom of the ***Extruded Cut*** as the sketch plane.
- ✓ Use the ***Insert→Features→ Dome*** command. The distance is 0.35 inch.
- ✓ If necessary, toggle the ***Reverse Direction*** icon to create a cavity instead of a dome. See Figure 8.14.
- ✓ Click ***Section View*** to check your work. Verify that the dome does not cut the bottom surface of the base.

Step 72: Use the ***Fillet*** command to create the 1/8 inch rounded corners on the base and a 0.03 inch chamfer around the top surface of the base. The edge of the seat for the bottom spacer are also chamfered 0.03 inch. See Figure 8.14.

Step 73: Calculate the volume of material needed to make this part and its mass.

- ✓ Right-click ***Material*** in the ***FeatureManager design tree***.
- ✓ Click on ***Edit Material***.

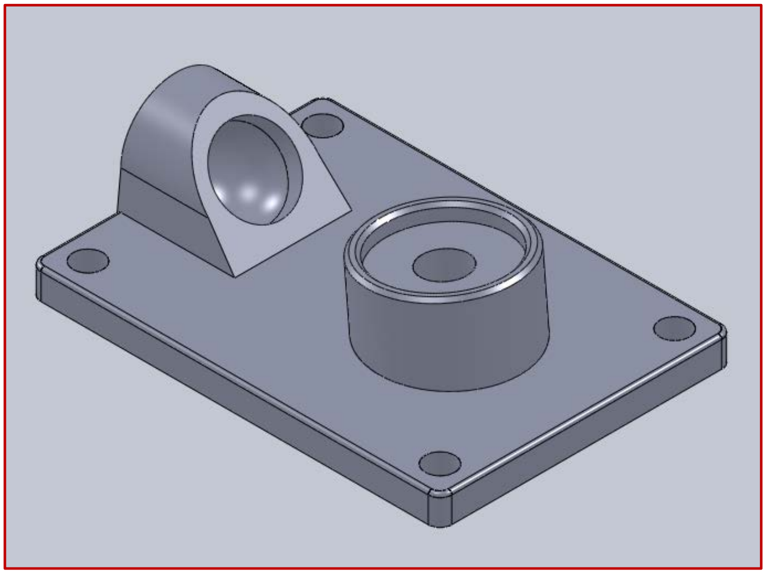


Figure 8.14 – Truck Base Model

- ✓ Find and highlight die-cast Zinc alloy AC43A.
- ✓ Click to ***Apply*** and then click ***Close***
- ✓ Click on the ***Evaluate*** tab in the ***CommandManager*** and then click ***Mass Properties***.

Step 74: Document your design by filling the ***Design journal*** and the ***Properties*** form. Record the volume and weight of the part. The material is die-cast Zinc.

- ✓ Finally, save the part as SKBD111.sldprt.
- ✓ If necessary click ***View*** in the ***Main Pull-down Menu*** and then click ***Hide All Types*** so the planes, axis and other confusing entities are not visible.

Evaluate, Measure and Mass Properties

The ***Evaluate*** tab in the ***CommandManager*** allows us to measure and calculate mass and section properties of parts and assemblies. In the case of mass properties, SolidWorks needs to know which material(s) are used. To choose a material, right-click the Material icon on the ***FeatureManager design tree*** to get 10 common choices. You can get additional choices by clicking on ***Edit Material***. Custom Materials are used for proprietary alloys or heat treatments that have properties different from published values.

By clicking the ***Mass Properties*** icon in the ***Evaluate*** tab of the ***CommandManager***, we get the volume, mass, density and surface area of a part or assembly. We can also get the center of mass and the moment of inertia. An example is shown below.

```
Mass properties of Part1
   Configuration: Predeterminado
   Coordinate system: -- default --

Density = 0.04 pounds per cubic inch

Mass = 0.04 pounds

Volume = 1.20 cubic inches

Surface area = 11.05 square inches

Center of mass: ( inches )
    X = 0.00
    Y = -0.07
    Z = 0.00

Principal axes of inertia and principal moments of inertia: ( pounds * square inches )
Taken at the center of mass.
     Ix = (1.00, 0.00, 0.00)      Px = 0.01
     Iy = (0.00, 0.00, -1.00)     Py = 0.01
     Iz = (0.00, 1.00, 0.00)      Pz = 0.01

Moments of inertia: ( pounds * square inches )
Taken at the center of mass and aligned with the output coordinate system.
    Lxx = 0.01          Lxy = 0.00          Lxz = 0.00
    Lyx = 0.00          Lyy = 0.01          Lyz = 0.00
    Lzx = 0.00          Lzy = 0.00          Lzz = 0.01

Moments of inertia: ( pounds * square inches )
Taken at the output coordinate system.
    Ixx = 0.01          Ixy = 0.00          Ixz = 0.00
    Iyx = 0.00          Iyy = 0.01          Iyz = 0.00
    Izx = 0.00          Izy = 0.00          Izz = 0.01
```

With the ***Measure*** tool it is possible to obtain diameter, length and distance between two points.

Section Properties will calculate the properties of a cross-section instead of the properties of the part calculated earlier.

Practice Exercises

1. Re-draw the Truck Base modeled in this Lesson using millimeters instead of inches. Use the mmPart template and the dimensions in the detail drawing in Appendix A.
2. Explain the options you have when using ***Extrude Boss/Base*** for: 1) start of the extrusion (***From***)**, 2)** ***Direction 1*** and 3) ***Thin Feature***.

Questions

1. Compare aluminum and zinc as materials for manufacturing the Axle and the Base. What are the relative advantages of each material? Which one would you recommend?
2. Verify that the head of the screw used on Figure 8.2 will not exceed the edges of the base.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor 3D Skills Core Concepts for Parts and Assemblies

- Planes 2
- Measure
- Materials
- Mass Properties

Videos from SolidProfessor Advanced Parts

- SketchXpert

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks tutorials: http://www.solidworks.com/sw/resources/solidworks-tutorials.htm

SolidProfessor tutorials: http://www.solidprofessor.com/

Lesson 9 – Importing Models from the Internet

9.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Find models of mechanical parts in ***Design Library***, ***Toolbox*** and in the Internet, and import them into SolidWorks.
- Import models from other CAD software using IGES, STEP and other neutral file translators.
- Use JPEG, TIF, VRML and other common file structures in SolidWorks.

9.2 Introduction

The ability to import and re-use models is one of the most useful capabilities of CAD software. Many manufacturers and suppliers of bearings, fasteners, actuators and other hardware have their catalogs on the Internet and you can download 3D models of their products. You can also re-use your old models and those of your co-workers. Many companies have libraries of commonly used parts and assemblies for re-use.

9.3 Importing from Toolbox

SolidWorks includes various libraries of components for re-use. One of them is the Toolbox. Although the primary use of Toolbox is to introduce hardware components into Assemblies, it can also be used to create parts when purchased hardware is modified. The modified purchased hardware must have a drawing and, hence, a part number. In this section, we will import a bearing and in a latter lesson we will make a drawing.

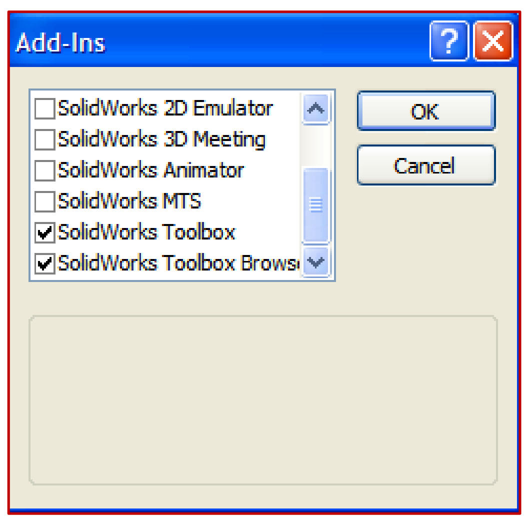


Figure 9.1 – Tools Add-Ins

Step 75: To make the toolbox available, click ***Tools→Add-ins*** on the **Main Drop-down Menu** or click ***Options→Add In*** in the **Quick Access Toolbar** to get the menu in Figure 9.1. Verify that ***Toolbox*** and ***Toolbox Browser*** have a check mark.

Step 76: Next, in the Task Pane expand ***Design Library→Toolbox→ANSI Inch→Bearings→Roller Bearings.*** See Figure 9.2.

✓ In the Roller Bearings folder, right-click on the needle roller bearings and select ***Create Part***.

✓ Select bearing 14NIU15, which is ½ inch thick with 0.875 inch outside diameter and 0.25 inch bore. Select the Detailed Display in the pull-down menu. In the Parts Number input box type SKBD123.

✓ Save the part for future use. Call it SKBD123.

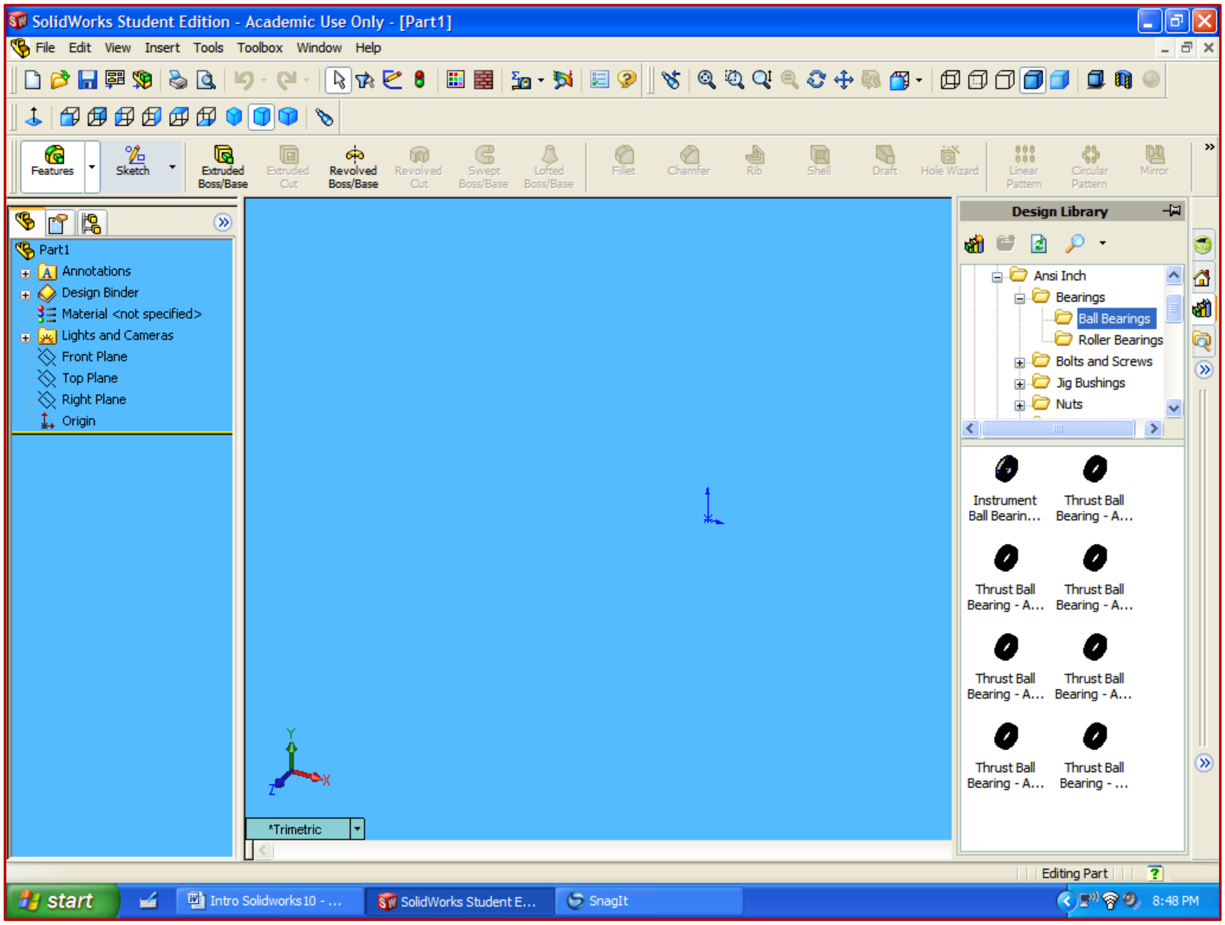


Figure 9.2 – Toolbox Menu

9.4 Importing from the Internet

Another source of models is the Internet. SolidWorks maintains a library of free models at http://www.3dcontentcentral.com/3DContentCentral/ . The models are contributed by manufacturers that hope to sell you the components that you download. You need to join to download from this website, but there is no charge for joining or downloading models.

Step 77: Login to 3DContentCentral and select a Nylock self-locking nut. Follow the instructions to download and save your model for future use.

9.5 Importing Standard Fasteners

Fasteners are available from either Toolbox or from the Internet on 3DContentCentral.

9.6 Importing and Exporting IGES and STEP Files

IGES and STEP are popular formats for CAD. They are open standards or neutral files that many CAD programs support. They can be used to transfer models between CAD systems. The process is not perfect, unfortunately, and some information is always lost in the translation. Some proprietary translators claim to transfer more complete information and to have a lower error rate than the open standards.

Step 78: To import or export models using one of these neutral files, select the appropriate File Type when opening or saving your model as shown in Figure 9.3.

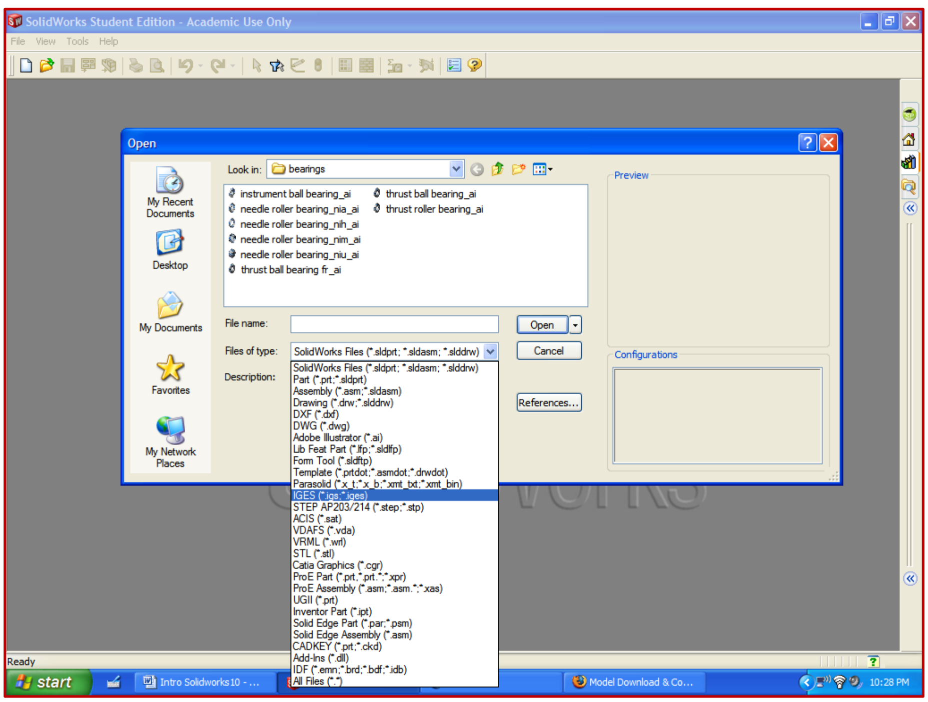


Figure 9.3 – File Format Options

9.7 Importing an Image for the Background

We can add a background to a SolidWorks part or assembly. The combination can be used in a marketing brochure or maintenance manual. In this section we will add a background to the skateboard deck. We will import an image in the TIFF format.

Step 79: To insert a picture in a SolidWorks model, right click in the ***Graphics Area*** and select ***Edit Scene.***

- ✓ On the input Background, select one of the options available: color, gradient, image or use environment.
- ✓ For Image, Browse to the image you wish to use. One source of images is Google.com, or use one of the background images provided by SolidWorks.
- ✓ Check Stretch image to fit SolidWorks window.
- ✓ Click the check mark to accept and save your work.

Step 80: The background can be removed by right-clicki***ng the Graphics Area, then clicking Edit Scene*** and selecting ***None*** on the ***Background*** pull-down menu.

Figure 9.4 – Adding a Background to the Solid Model

Practice Exercises

1. Import the screws needed in the skateboard assembly. You will need eight flathead screws and two hex head screws. They can be imported from the Toolbox, from SolidWorks 3D Content Central, or from a manufacturer's catalog on the Internet. Consult the assembly drawing in Appendix A for the size.

Questions

1. Find three or more websites where you can find SolidWorks models of commercial components.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor Toolbox and Design Library

- Toolbox Overview
- Setting Up/Adding In the Toolbox
- Inserting Toolbox Components
- Design Library Introduction

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks models for download: http://www.3dcontentcentral.com/3dcontentcentral/

Popular CAD models for download: http://grabcad.com/library/software/solidworks

Supliers CAD models for download: http://cad.thomasnet.com/cadmodels.html

CAD Models for download: http://b2b.partcommunity.com/community/

Section IV – Modeling Assemblies

Lesson 10 – Creating Assemblies and Sub-Assemblies

10.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Customize SolidWorks by creating an assembly template.
- Explain and create assemblies and sub-assemblies.
- Apply assembly mates.
- Explain and use the ***Hide Component*** and ***Change Transparency*** commands.

10.2 Introduction

When we open a new assembly document, the software activates all the commands needed to assemble two or more parts that were modeled previously. Most of these commands are included in the ***Assembly*** toolbar. They include commands to move and rotate the individual parts or the whole assembly. The ***Mate*** command can establish a relationship between different parts. Some examples of ***Mate*** between parts are concentric, parallel and coincident. Most are equivalent to the ***Sketch Relations*** studied before, with the exception that they are used with parts instead of lines.

In this Lesson, first we will customize SolidWorks by creating a template that we can use with assemblies. Next, we will create the tire and wheel sub-assembly and then the truck sub-assembly. Before creating this assembly, you must create the tire with the pockets that will match the bumps on the wheel (see Practice Exercise 2 in Lesson 6 and drawing SKBD122 on Appendix A). Finally, we will learn to explode an assembly and to check for interference.

10.3 Creating the Assembly Template

To create the assembly template, open a new document and select an ***Assembly*** document. The only critical option to be selected is the units that will be used, inches or millimeters. We will verify that we work with inches and that the ***Design Journal*** is visible. We will accept all other default options.

Step 81: Click Options on the Quick Access Toolbar or Tools→Options on the Main Pull-down Menu.

✓ Select the ***System Options*** tab.

- ✓ Click ***FeatureManager*** and verify that the pull-down menu for ***Design Binder*** says ***Show***.
- ✓ Select the ***Document Properties*** tab and click ***Units*** to get Figure 10.1.
- ✓ Verify that the selection is ***IPS*** (inch, pound, second).
- ✓ Click OK.
- ✓ In ***File→Properties***, create the following ***Custom Properties***: 1) Description (leave column 3 blank), 2) Weight (select SW-Mass in column 3), Material (type "- -" or "NOT APPLICABLE" in column 3) and Finish (type "- -" or "NOT APPLICABLE" in column 3).
- ✓ Save the inch assembly template as InchAssembly.asmdot.

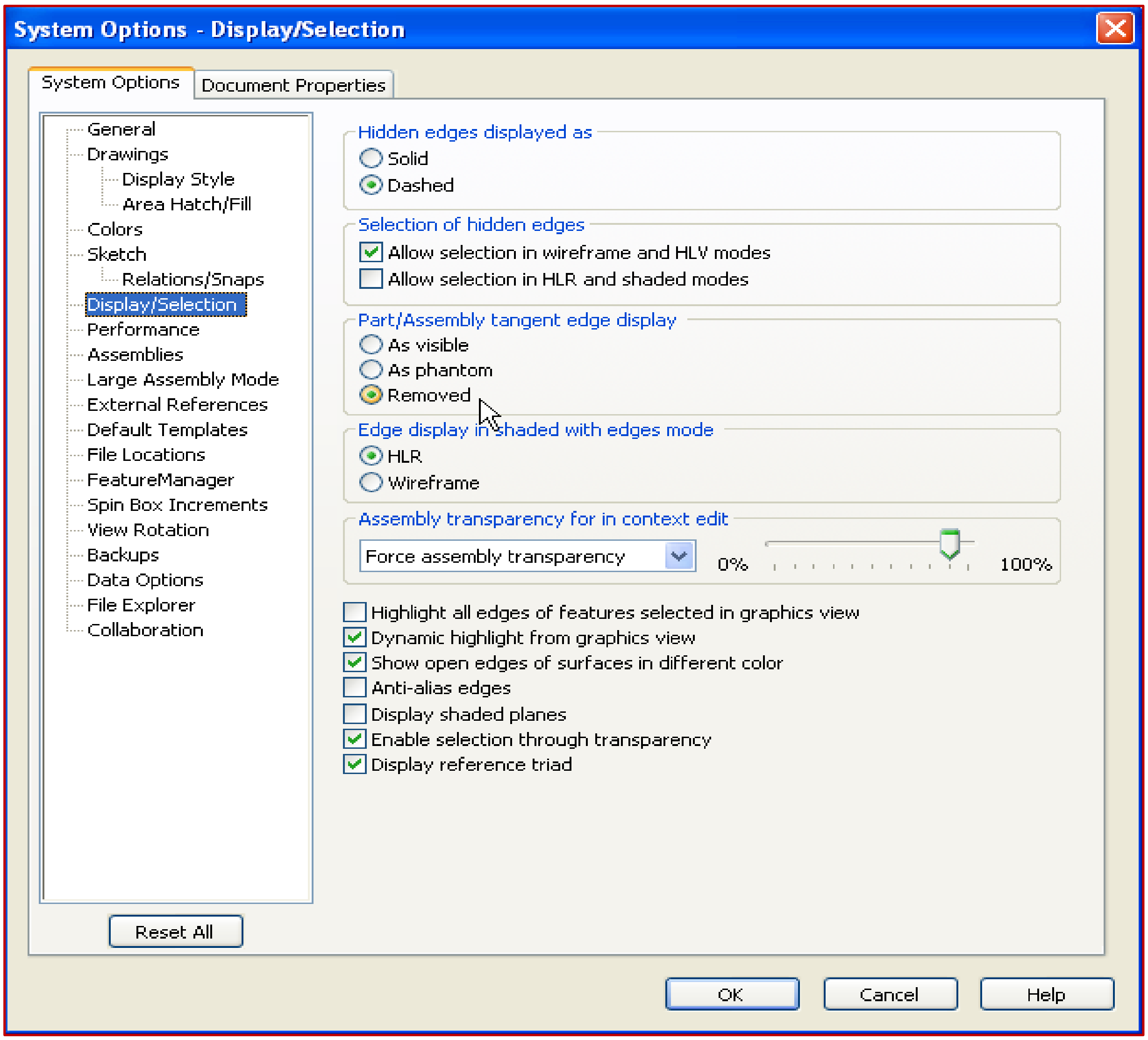


Figure 10.1 – *Tools→Options*

10.4 The Tire and Wheel Sub-Assembly

The first part introduced to an assembly will remain fixed. Subsequent parts added to the assembly can move relative to the fixed part. It is important to plan which part should be introduced first. For the wheel sub-assembly we will start with the wheel.

Step 82: Open a new document and select the ***InchAssembly*** template.

- ✓ The input menu will ask you what part you want to add. Browse through your saved parts until you find the skateboard wheel.
- ✓ Click to select the wheel (SKBD121) and then click ***Open***.
- ✓ Move the cursor to the location where you want to locate the wheel. Remember that once the wheel is positioned it will not move.
- ✓ To make the wheel coincident with the origin, click the check mark. If the origin is not visible, click ***View→Origins*** on the **Main Drop-down Menu**. See Figure 10.2.

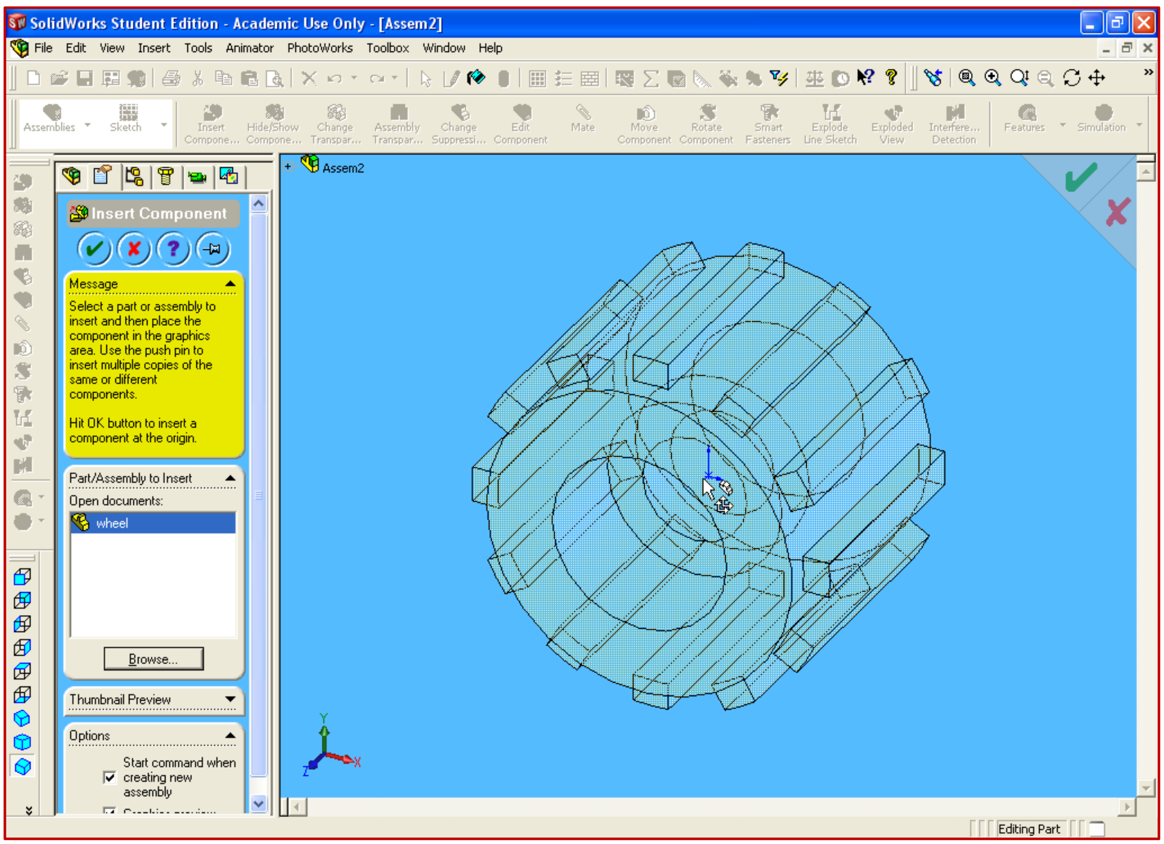


Figure 10.2 – Adding the Wheel to the Assembly

Assemblies

Products, devices and structures are usually assemblies of parts. The reason for breaking your design into parts is to facilitate manufacturing. Each part is made of a different material or manufacturing process. Parts are typically manufactured and handled separately and come together at assembly. The SolidWorks assembly model can be used to create an assembly drawing that will help the assembler. In addition, the SolidWorks assembly can be used to verify that parts do not have interferences and that mechanisms will work as expected. If used, this capability can reduce or entirely eliminate errors.

Sub-assemblies are used to facilitate assembly. For example, in our skateboard the wheels and tires are assembled together during manufacture by casting the rubber compound around the wheel. The result is a tire and wheel sub-assembly that for all practical purposes can be treated as a single part. Another example of a sub-assembly in the skateboard is the truck sub-assembly consisting of the wheels, tires and truck. In the automobile industry, engines are assembled and tested independently before they are added to the automobile frame at final assembly.

Step 83: Next, add the tire to the assembly. Click ***Insert Components*** in the ***CommandManager*** or select ***Insert→Component→Existing Part*** on the **Main Drop-down Menu**.

- ✓ Browse to the tire model (SKBD122), click to select and click OK to bring the tire to the graphics area. See Figure 10.3.
- ✓ Drag and release the tire on the graphics area.
- ✓ You have two options to mate the two parts. The quick and easy option is to click the check mark to make the axis of the tire and axis of the wheel coincident. This will work if the bumps and pockets match exactly. The more complex approach is explained in the next step.
- ✓ Drag and drop the tire to one side of the wheel.

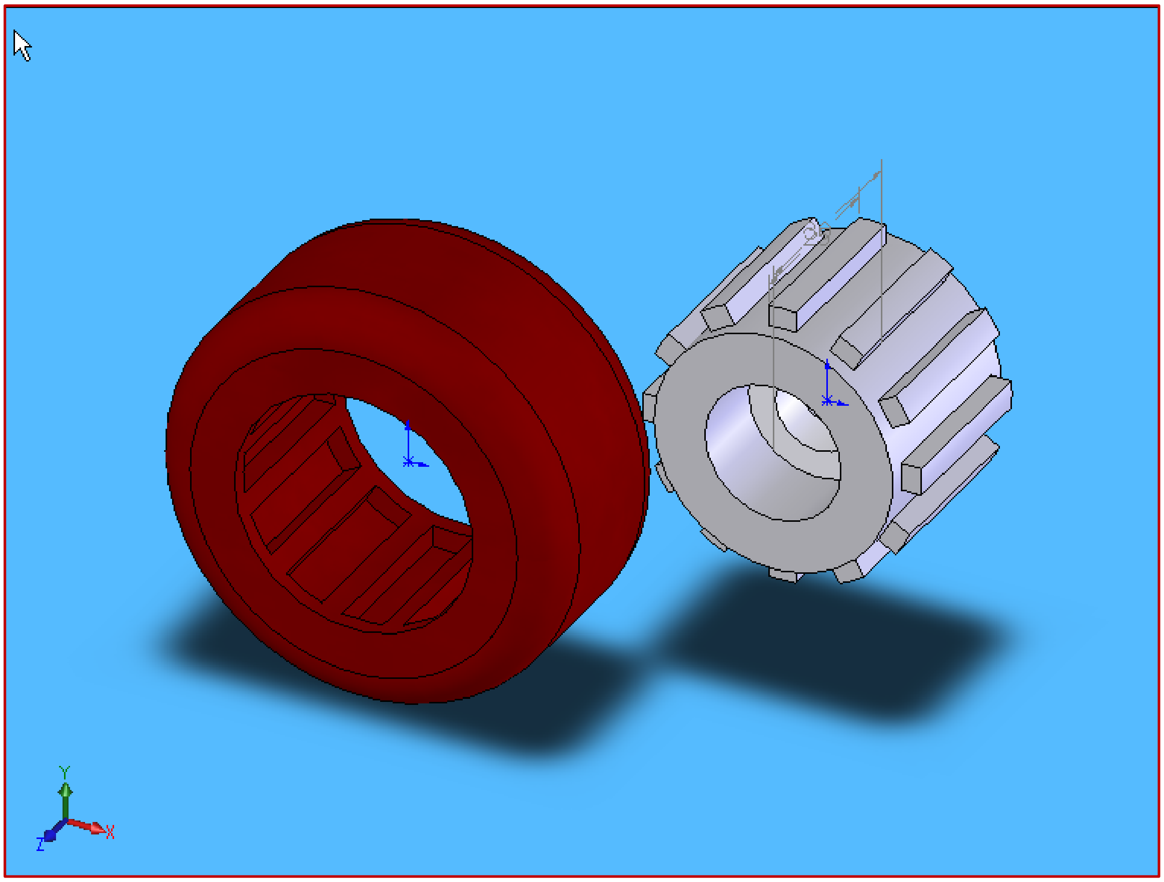


Figure 10.3 – Adding the Tire to the Assembly

Step 84: To achieve the correct orientation between the two parts, we will use ***Assembly Mates***.

- ✓ First, we will make the wheel and the tire ***Concentric***. Click the command ***Mate*** on the ***CommandManager*** toolbar and click the inner rim of the tire and the outside edge of the wheel. Select ***Concentric*** on the ***Standard Mates*** panel. See Figure 10.4.
- ✓ Notice that the wheel will not move if you click and drag. It is fixed. If you click and drag the tire, it will move only along the axis.
- ✓ Open the ***FeatureManager design tree*** (to the right of the input panel) and notice, that the assembly design tree is composed of the parts included in the assembly. If you click the + sign you can see each part's ***FeatureManager design tree*** and the steps that created each part.
- ✓ At the end of the design tree you will find the ***Mates Folder***. If you open the ***Mates Folder*** you will see the mates used to create your assembly. If you right click a mate and select ***Edit Feature***, you can change to another type of mate.

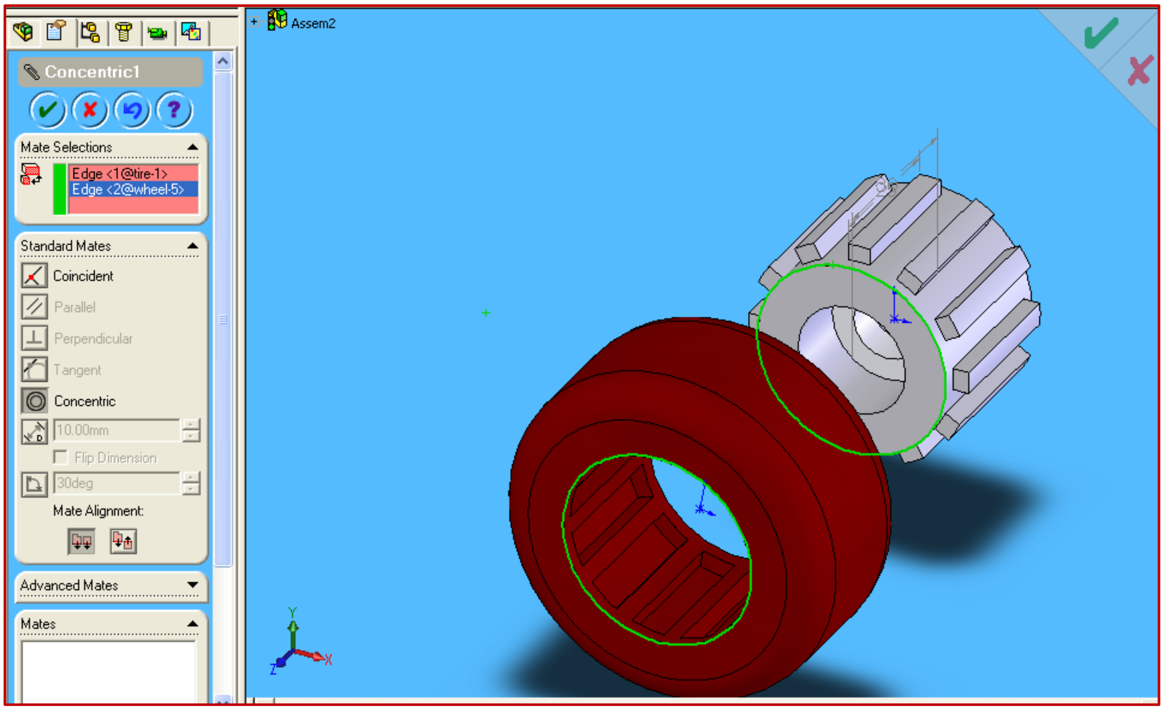


Figure 10.4 – Selecting Concentric *Mate*

- ✓ Next, select the flat surface of one of the wheel bumps and the equivalent surface in the tire and make them ***Parallel***.
- ✓ Finally, make the face of the wheel and the side face of the tire ***Coincident***.
- ✓ Check your work using the ***Section View*** icon on the ***View (Heads-Up) toolbar*** or select ***Display Style→Wireframe or Hidden Lines Visible*** on the ***View (Heads-Up) toolbar.***

Step 85: Add the bearings. The bearings can be added clicking ***Insert Component*** or clicking ***Design Library→Toolbox***.

- ✓ To add the bearings using ***Insert Component***, first click the icon on the ***CommandManager***.
- ✓ Browse to the Bearing model (optional SKBD123) created in ***Step 76***.
- ✓ Import the model into the assembly.
- ✓ Make the diameters concentric and the two faces that touch coincident.

Adding the bearings importing directly from Toolbox.

- ✓ To add the second bearing importing directly from the ***Toolbox***, click ***Design Library→Toolbox→ANSI Inch→Bearings→Roller Bearings*** on the ***Task pane*** and click and drag the needle roller bearing to its position on the wheel.
- ✓ Release the mouse button.

- ✓ Press the Escape key on your keyboard to stop adding bearings.

Adding the second bearing using the command *Mirror Component*.

- ✓ Another way of adding a second bearing is to ***click Linear Components→Mirror components***. This will copy the bearing to the other side.
- ✓ Check your work using the ***Section View,*** the ***Wireframe*** or the ***Hidden Lines Visible*** commands.
- ✓ Add the appropriate information to the ***Design journal*** and to the ***Properties*** form.
- ✓ Save the model as SKBD120.sldasm.

Practice Exercises

1. Create a millimeter assembly template. Save it as mmAssembly.asmdot.
2. Create the Truck Sub-Assembly drawing SKBD110 in Appendix A. (Hint: Use the following assembly order: Base→Bottom spacer→Front bumper→Truck axle→Top spacer)
3. Create the top-level skateboard assembly drawing SKBD100 in Appendix A. (Hint: Use Toolbox for the hardware, i.e. the screws and nuts.)
4. Describe the differences in the ***FeatureManager design tree*** for parts and assemblies.

Questions

1. Describe the advantages and disadvantages of using sub-assemblies in your design.
2. List one sub-assembly in a typical automobile.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoilldWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidProfessor 3D Skills

- Introducing Assemblies
- Starting an Assembly
- Inserting Components
- Adding Mates

Videos from SolidWorks for Beginners

- Working With Sub-Assemblies

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks tutorials: http://www.solidworks.com/sw/resources/solidworks-tutorials.htm

SolidProfessor tutorials: http://www.solidprofessor.com/

Lesson 11 – Detecting Interference, Editing and Exploding the Assembly and Creating Multiple Assembly Configurations

11.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Check an assembly for interferences and detect collisions
- Explain and use ***Top Down Design*** and ***In-Context*** editing.
- Explode and collapse SolidWorks assemblies.
- Animate the explosion or collapse of an assembly.
- Create multiple versions of an assembly.

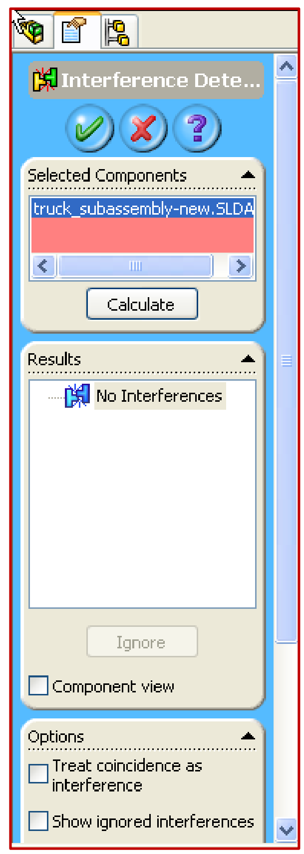


Figure 11.1 – Checking for Interference in the Assembly

11.2 Introduction

One benefit of solid modeling is that we can check for interference between parts. It is cheaper and faster to fix interference problems in the model than in real parts. Now, we will check the tire sub-assembly for interferences. If you have completed the Truck sub-assembly (Practice Exercise 2 in Lesson 10 and drawing SKBD110) and the Top Skateboard assembly (Practice Exercise 3 in Lesson 10 and drawing SKBD100) you can check those also.

11.3 Detecting Interferences

Step 86: Click **Open** and select the tire sub-assembly.

✓ On the **Main Drop-down Menu**, select ***Tools→Interference*** to get the menu in Figure 11.1. There should be no interferences detected as shown in the figure.

✓ To correct interferences, you can edit the ***Part*** model or edit ***In-Context.*** Any changes will be reflected in the assembly and in the assembly drawing.

Editing ***In-Context***.

- ✓ In the ***FeatureManager design tree***, right-click on the part that you want to edit and select ***Open Part*** or ***Edit Part***.
- ✓ ***Open Part*** will open the ***Part*** file and you will work on the part without the assembly.
- ✓ Clicking ***Edit Part*** will do two things:
  1) The design tree of the part being edited will become blue, and
  2) In the solid model of the assembly, only the part being edited will remain solid and all the others will become transparent. You will be making changes in the context of the assembly.
- ✓ Click the + sign to open the part design tree.
- ✓ Make changes to the features or sketches.
- ✓ To return to the assembly from the edit mode, right-click on the ***Top Assembly*** (first) icon in the design tree and select the ***Edit Assembly*** icon. All the parts will cease to be transparent.

11.4 Detecting Collision

Detecting collision when a mechanism moves is another useful feature in SolidWorks. Collisions are expensive and time consuming to fix. If you have completed the Truck sub-assembly you can collisions when the truck axle rotates about its pivot.

Step 87: Click Move Component or Rotate Component.

- ✓ On the ***PropertyMenu***, click ***Collision Detection***.
- ✓ Drag or rotate the moving part.
- ✓ Depending of the options selected, you can stop the motion of colliding parts, make them change color, or make a sound.

11.5 The Exploded Assembly

Exploded assemblies are very useful. It is easier to understand complex assemblies and the assembly sequence when the components are shown exploded.

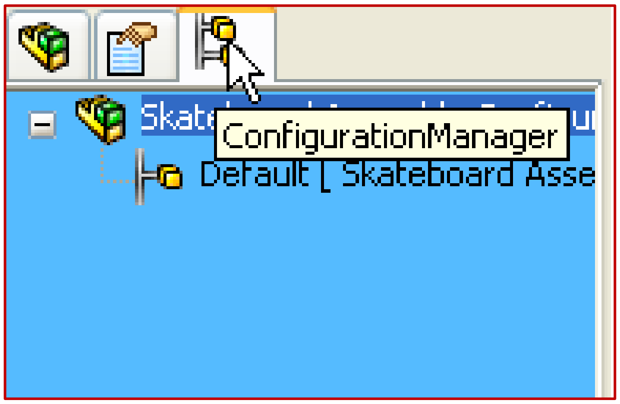


Figure 11.2 – ConfigurationManager showing the Default Configuration

Step 88: Exploding the tire assembly.

- ✓ Click on the ***ConfigurationManager*** icon to get Figure 11.2.
- ✓ The figure shows only one configuration called ***Default***. It is the name that SolidWorks gave the assembly we created earlier.
- ✓ Right-click ***Default*** and select ***New Exploded View*** to get the menu in Figure 11.3***.***

Step 89: Click on one assembly component to show the manipulator handle.

- ✓ Click and drag the handle in the direction you wish to explode.
- ✓ Release the mouse button to release the part.
- ✓ Repeat for every part until you have the fully exploded configuration that you want. See Figure 11.4.
- ✓ Notice that you can drag any number of components simultaneously by first clicking to make them active, and then dragging the manipulator handle.

Step 90: After saving the exploded configuration, it is possible to toggle between the exploded and normal configuration by right-clicking on ***AssemblyConfiguration*** and changing between ***Collapse*** and ***Explode***.

Step 91: Right-click again on AssemblyConfiguration and select Animate collapse or Animate explode.

- ✓ The menu in Figure 11.5 will appear to allow the user to control the animation. Table 11.1 shows what options are available.

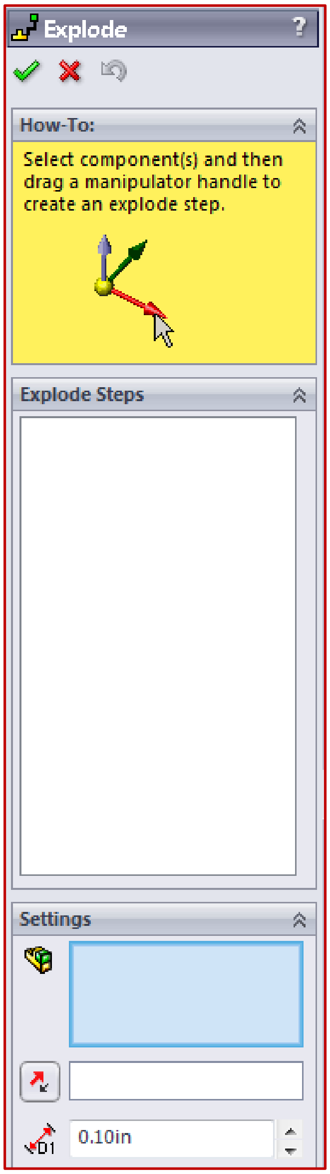


Figure 11.4 – Creating the Exploded View

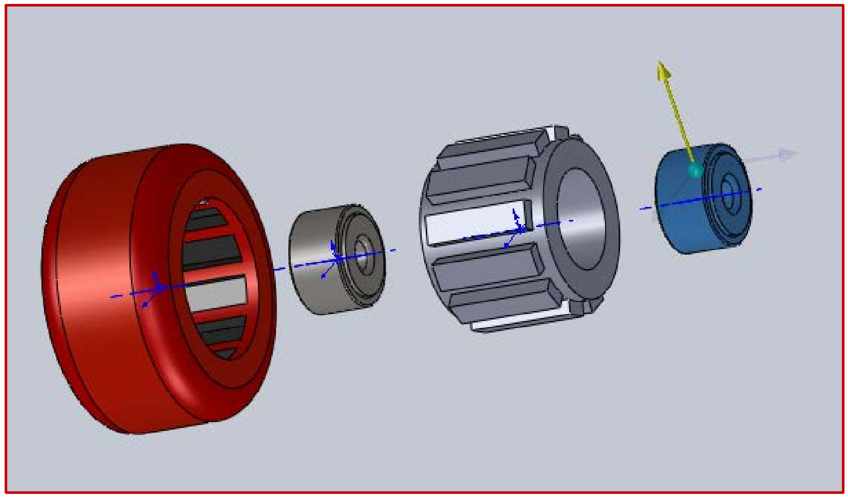


Figure 11.3 – Exploded Assembly

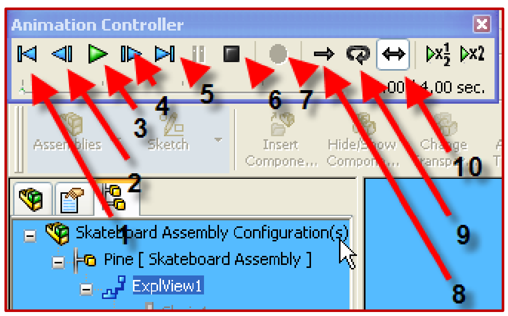


Figure 11.5 – Explode Animation Menu

Table 11.1 – Explode Animation Control

Icon Number	Action
1	Return the animation to frame one. Click **Start** after selecting this choice.
2	Rewind (play back) the animation. Click **Start** after selecting this choice.
3	**Start**
4	Fast forward. Click **Start** after selecting this choice.
5	Pause the animation. Click **Start** after selecting this choice.
6	**Stop**
7	Save the animation.
8	When 8 is selected, the animation plays start-to-end once.
9	When 9 is selected, the animation loops or plays beginning-to-end continuously.
10	When 10 is selected the animation reciprocates between explode-collapse-explode.

11.6 Multiple Assembly Configurations

In Section 11.5 we created an exploded wheel assembly. In this section we will create a new configuration with a green tire.

Step 92: Right-**c**lick the Tire and select ***Edit Feature***.

- ✓ Verify that you have the ***FeatureManager design tree***.
- ✓ Notice that the Tire feature changes to blue to show that the tire is being edited.
- ✓ On the Feature/Property/Configuration manager tabs (see Figure 1.4), click the arrow to show the Display Pane (see Figures 11.6 and 11.7).
- ✓ Under the colored ball for "Appearances", click the line for the tire and select Appearance.
- ✓ Select and click on a green color that you like and then click on the checkmark to accept.

Step 93: Right-click the Tire again and select ***Edit Assembly*** to save the changes and return to the assembly.

Figure 11.16a- Feature/Property/Configuration manager tabs.

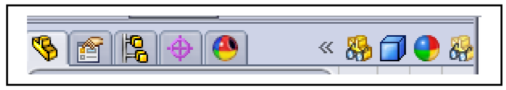


Figure 11.16b- Feature/Property/Configuration manager tabs with Display Pane.

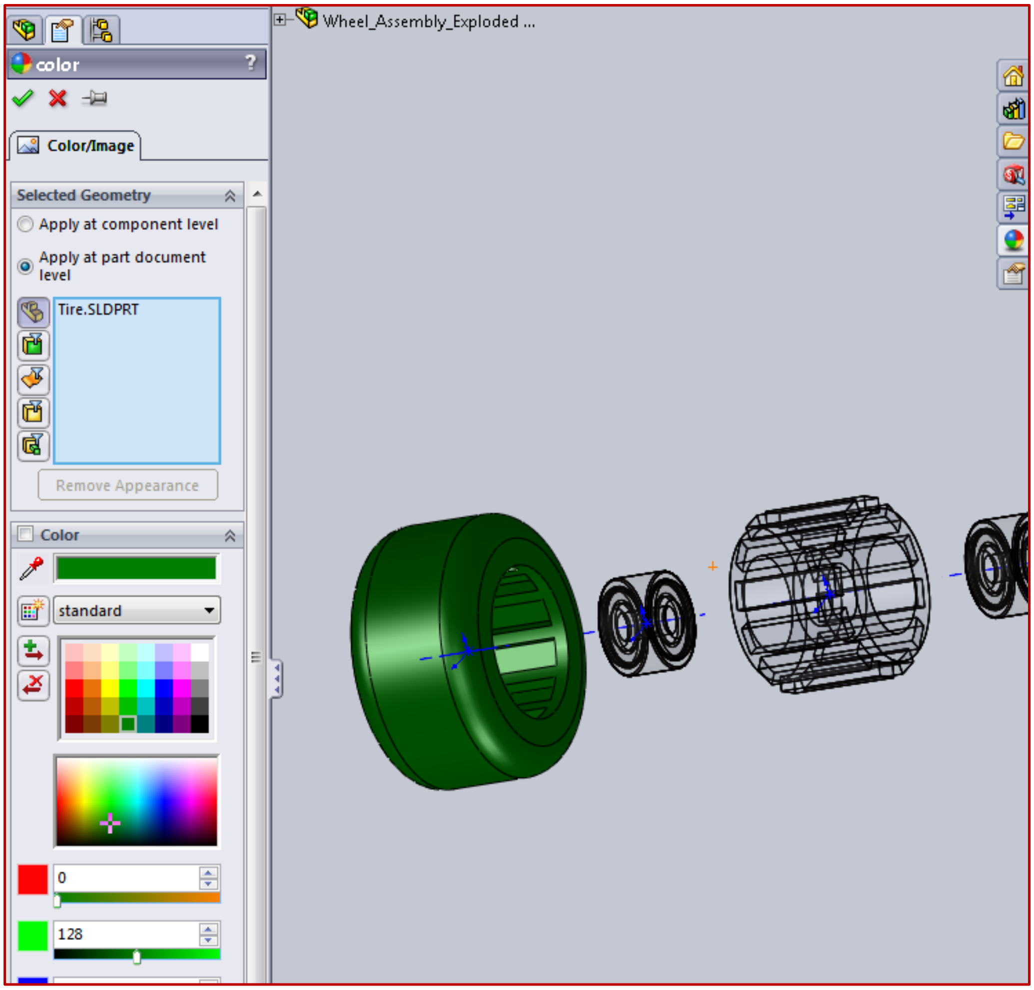


Figure 11.7 – Green Tire

Step 94: Create a new configuration.

- ✓ Click the ***ConfigurationManager*** tab.
- ✓ Click **Wheel_ Assembly** and right-click to show the menu.

- ✓ Click ***Add Configuration***. Name the new configuration "Green" and click the check mark to accept.
- ✓ A new configuration named "Green" is created.

Step 95: Save the model. It will have two configurations.

Top Down Design

Top down design can be used to make changes to a component that does not fit correctly into an assembly. It can be also used to create a new component or a new feature in an existing component, if they are part of an assembly. Although top down design is an advanced topic and is beyond the scope of this book, it is useful to have an introduction to the subject because it is a powerful tool.

One variation of top down design is in-context design. For example, if the diameter of the tire must be made larger and we want to verify that it is not too large and it still fits under the deck, we can use in-context design. On the skateboard final assembly (SKBD100):

- ✓ Notice that we have in the ***FeatureManager design tree***, the design trees for each part. They can be expanded by clicking on the (+) sign. Similarly, the steps can be hidden by clicking the (-) sign.
- ✓ Right-click on the tire (or the tire feature in design tree) to get a pull-down menu and click the icon ***Edit Part*** to edit the part in the context of the assembly.

  Note: Alternatively, we can click the icon Open Part to edit the part by itself without the assembly.
- ✓ On the design tree, click the feature or the sketch that must be changed.
- ✓ When the changes have been completed, you can return to the assembly by first right-clicking on the part or the design tree to get the pull-down menu and then clicking ***Edit Assembly***.

Practice Exercises

1. Move the wheel bearings from the Wheel Assembly to the Truck Sub-Assembly.

Practice Problems

1. Discuss the advantages and disadvantages of including the wheel bearings in the Wheel Assembly vs. the Truck Sub-Assembly.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

Videos from SolidWorks Core Concepts for Parts and Assemblies

- Interference Detection
- Dynamic Clearance
- Collision Detection
- Exploding Assemblies
- Hide/Show Components
- Introduction to Top Down Design
- Intro to Top Down Design II

Videos from SolidProfessor Advanced Assemblies

- In-Context Features – Part A

Internet Resources

CADeducators.com website: http://www.cadeducators.com

SolidWorks tutorials: http://www.solidworks.com/sw/resources/solidworks-tutorials.htm

SolidProfessor tutorials: http://www.solidprofessor.com/

Section V – Creating Engineering Drawings

Drafting Standards

Drafting standards are needed because drawings must be understood by every interested party irrespective of language, cultural or geography barriers. The easiest way to accomplish this is with a set of codified rules available to all. The Drafting standards list the rules to be used when creating or interpreting technical drawings. The two most widespread drafting standards are ANSI and the ISO. The first because it is the most common in the United States and the second because it is the most popular in the rest of the world. The ISO standard allows for customization and most countries have their local version. The ANSI standard has converged towards ISO and to some extent it can be said that ANSI is the U.S. version of the ANSI standard.

The ANSI Drafting Standard

ANSI is the abbreviation for American National Standards Institute. ANSI is a private, non-profit organization whose charter is to coordinate the development of voluntary consensus standards. Also, ANSI is the United States representative to ISO, an international organization also involved in the creation of standards. ANSI´s mission is:

> To enhance both the global competitiveness of U.S. business and the U.S. quality of life by promoting and facilitating voluntary consensus standards and conformity assessment systems, and safeguarding their integrity. (From http://www.ansi.org)

The members of ANSI are companies, government agencies, universities and individuals interested in the creation, use, and maintenance of standards and on the certification of products and personnel to the standards. ANSI coordinates but not produce the standards. The standards are produced by the member companies, organizations and individuals that have an interest in the subject. Typically, a draft of the standard produced by a member is circulated to others for changes and eventual approval by consensus.

Technical drawings that follow the ANSI standard are drawn in inches or millimeters and are projected in the third quadrant or angle.

The ISO Drafting Standard

The organization known as ISO is chartered to create standards that are accepted and can be used worldwide. The name ISO means equal and is derived from the Greek word isos *(ἴσος)*. ISO was chosen to avoid having different abbreviations in the various languages. ISO's headquarter is in Geneva, Switzerland and its members represent 164 countries as of 2014. This compares with 193 countries in the United Nations. Thus, almost all the countries of the world with manufacturing or international trade economic sectors are represented in ISO. Each country sends a delegation to the annual general assembly and provides experts to staff the various committees. Most delegations are the host country´s standards organization. For example, ANSI (American National Standards Institute) represents the United States in the ISO general assembly and in committees and is one of the important standards organization in the United States.

ISO drawings use strictly millimeters only and are projected in the first quadrant or angle.

Lesson 12 – Creating Detail Drawings

12.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Explain the need for working drawings.
- Explain the difference between detail drawings and assembly drawings.
- Customize SolidWorks by creating a drawing template.
- Explain and create detail drawings in SolidWorks.
- Explain and create assembly drawings in SolidWorks.
- Explain the detail drawing title block.
- Create the drawing for the Skateboard Deck. Instructions for creating other drawings can be found at the website www.cadeducators.com or in the step by step videos also available in the website.

12.2 Introduction

Working drawings are the primary means of communication between designers and fabricators. They describe each part needed to make a product and how they are assembled. There are three kinds of drawings in a complete set of working drawings:

1) detail drawings,
2) assembly drawings, and
3) the bill of materials.

Sometimes, the assembly drawing and the bill of materials are combined into one drawing.

Each detail drawing has the instructions for making one part. The instructions include all dimensions, the material to be used, and the finish, such as painting plating or protecting with grease. In some situations, even the storage instructions are recorded in the drawing.

Assembly drawings describe how the parts are assembled. They are rarely used by the person making a part. Instead, they are the instructions for the person or persons responsible for the assembly of the product. Every assembly drawing needs a bill of materials or BOM. The BOM is a list of all the parts needed to assemble the product or sub-assembly. In practice, the person assembling the product will first verify that all the parts are available before assembly begins.

Drawings are also used by the maintenance organization when a repair or upgrade is necessary. For this reason, maintenance information should be included as notes in the detail or assembly drawings. Finally, the design and the maintenance organizations need information about the changes made to the product over the years. In this role, drawings are historical documents that can be used to reconstruct the original design and all the changes. This information is usually found in the change block.

12.3 Creating a Drawing Template

We will create a drawing template to customize SolidWorks. Drawing templates must follow standards, for example ANSI or ISO.

Step 96: Open a new document and specify ***Drawing*** as the type of document. A new drawing Format and Size selection panel is shown in Figure 12.1.

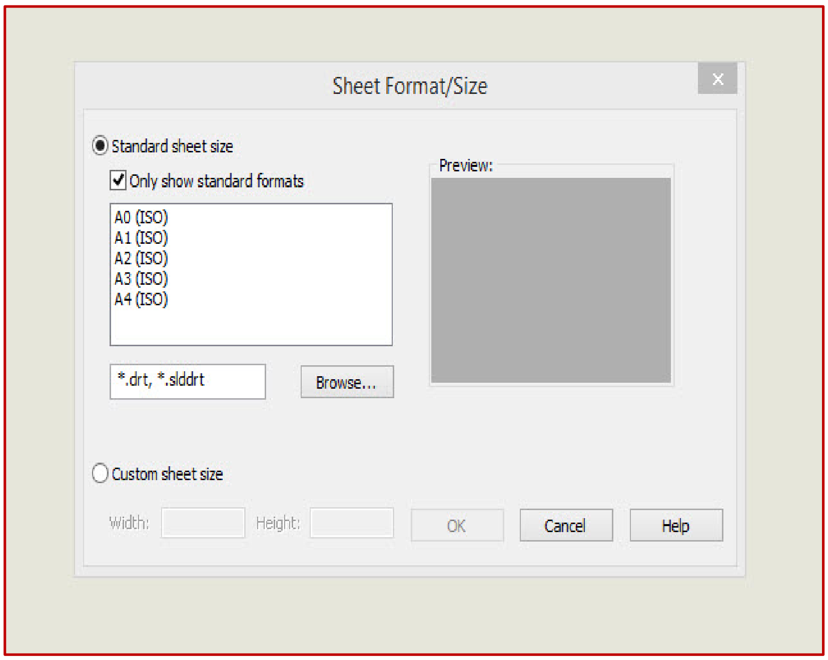


Figure 12.1 – New Drawing Format and Size Properties

- ✓ Click ***Browse*** and navigate to the location where SolidWorks stores the drawings templates in your computer or network. See Figure 12.2a.
- ✓ Select ***a – landscape.slddrt***.
- ✓ Click ***Open*** and then click ***OK***.
- ✓ You will get the Drawing document on Figure 12.2b.

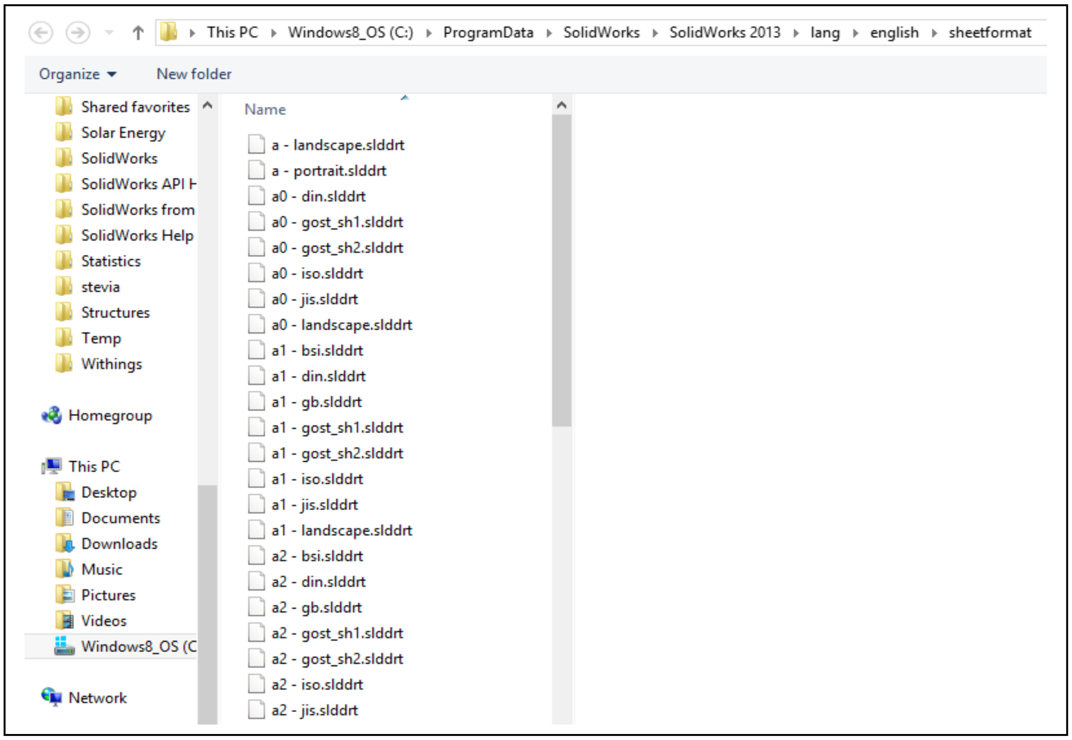


Figure 12.1a – SolidWorks Drawing Format Templates

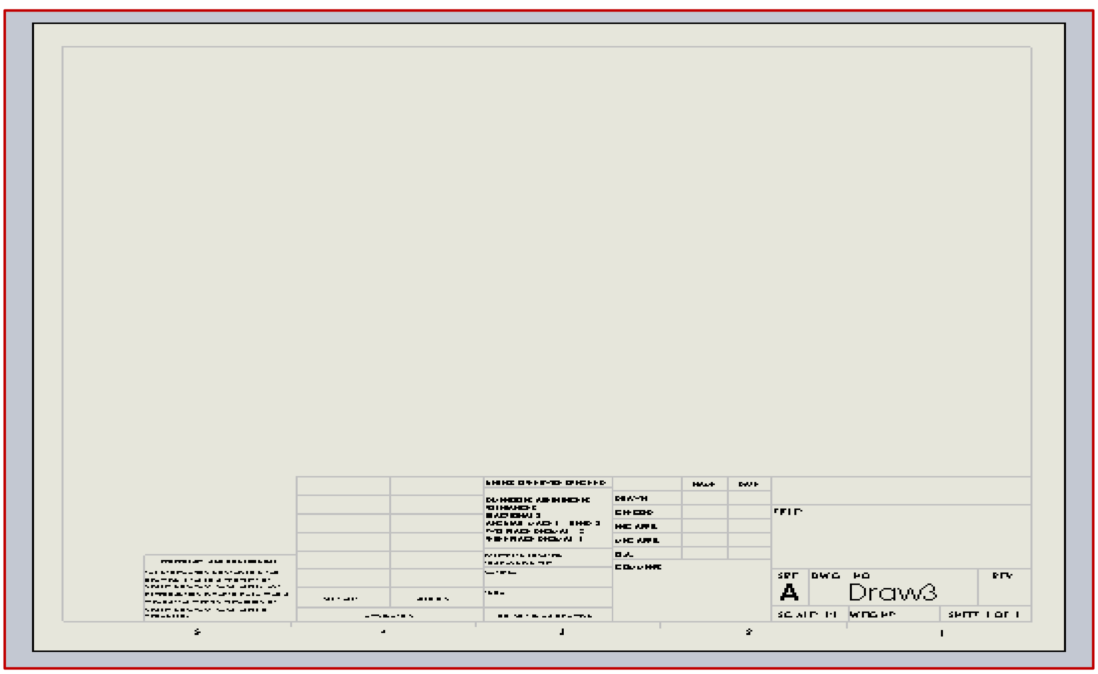


Figure 12.2b – New Drawing Document

Step 97: We will now create a template for the A-size Landscape drawings.

- ✓ Click on ***Tools→Customize*** on the ***Main Pull-down Menu*** or on ***Options→Customize*** on the ***Quick Access Toolbar*** and select the ***Toolbars*** tab to get the dialog box in Figure 12.3.
- ✓ Verify that the ***CommandManager*** is enabled.
- ✓ Verify that the ***Task Pane*** and ***View (Heads-Up)*** toolbars are enabled.
- ✓ Verify that in ***Context toolbar settings***, ***Show on selection*** and ***Show on shortcut menu*** have a check mark. See Figure 12.3.

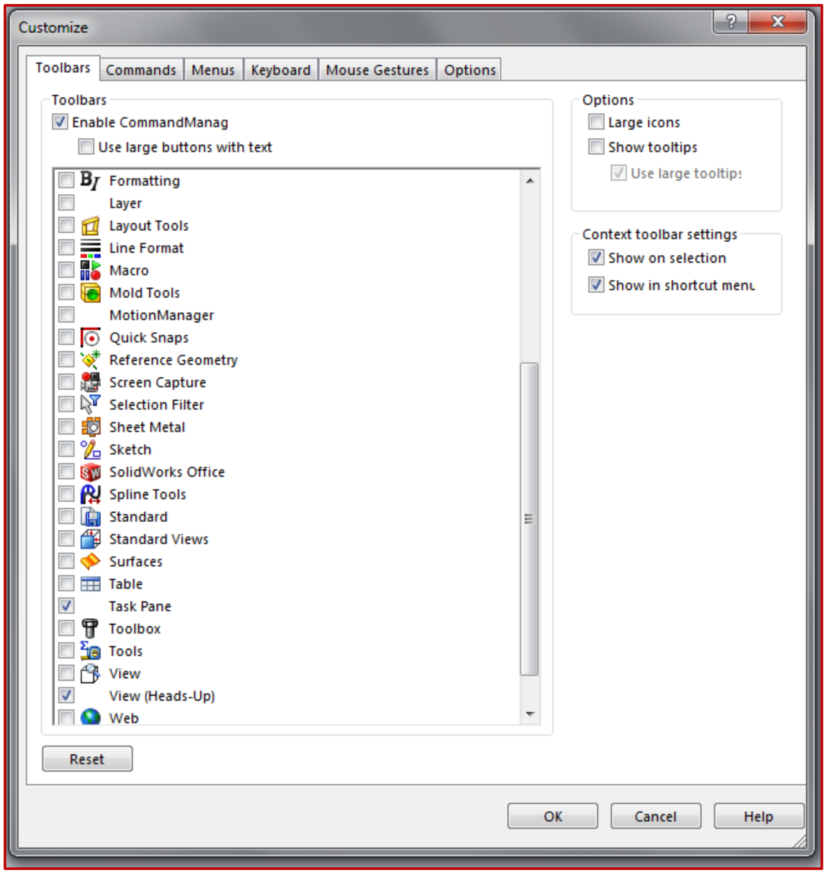


Figure 12.2 – *Tools→Customize→Toolbars*

Step 98: Select System Options

- ✓ Click ***Tools→Options*** on the ***Main Pull-down Menu*** or ***Options*** on the ***Quick Access Toolbar*** and select the ***System Options*** tab.
- ✓ Click ***Display Style*** and verify that hidden lines and tangent edges are removed.
- ✓ In ***Tools→Options*** select the ***System Options*** tab, click ***FeatureManager*** and verify that the ***Design Binder*** pull-down menu says ***Show***.
- ✓ Your screen should look like Figure 12.4.

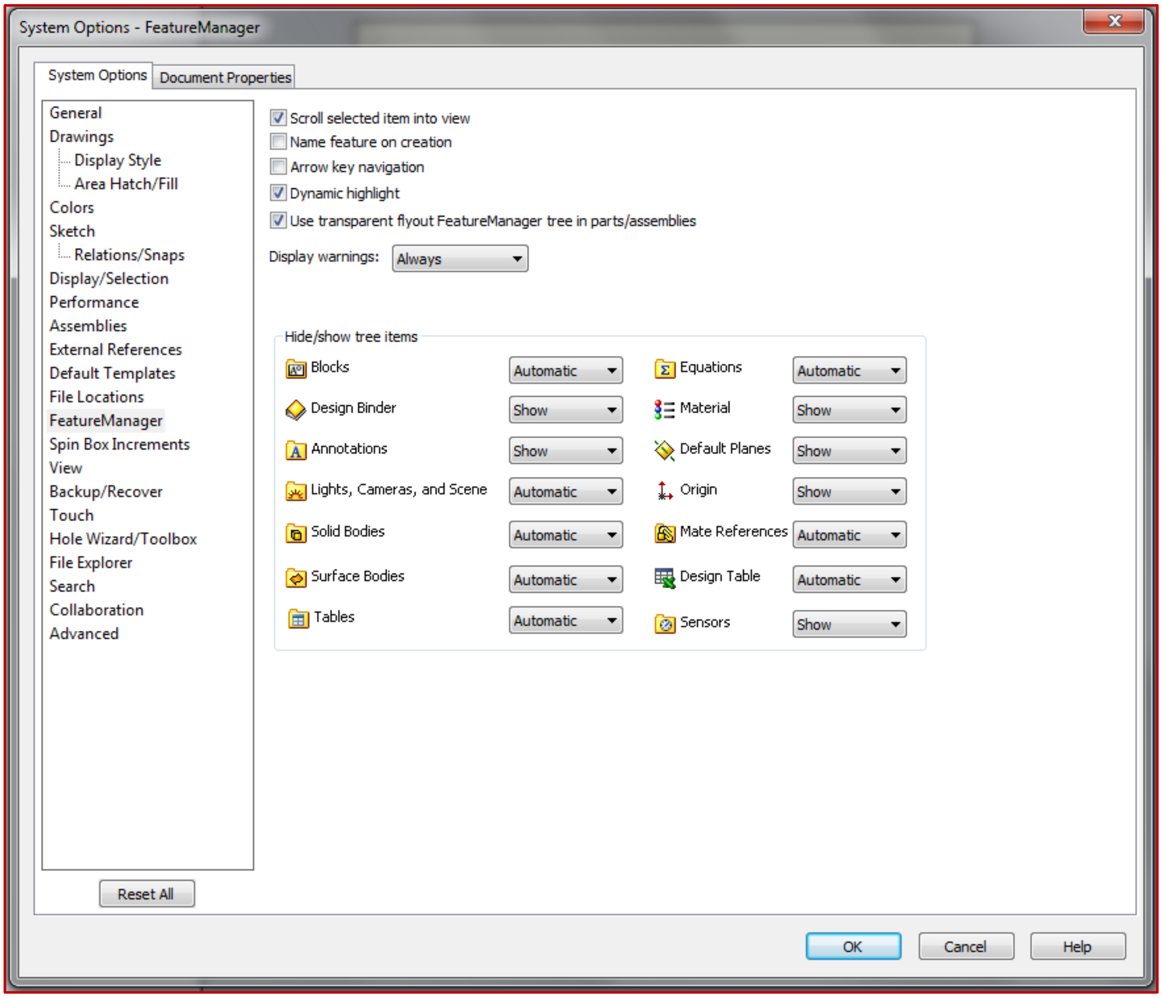


Figure 12.3 – *Tools→Options→System Options*

Step 99: Select Document Properties

- ✓ Select the ***Document Properties*** tab and verify the following choices:
- ✓ ***Drafting Standard –*** Verify that ***ANSI*** is selected as the drafting standard. See Figure 12.5.
- ✓ ***Units –*** Select ***IPS*** (inches, pounds, and seconds).
- ✓ Click ***OK***.

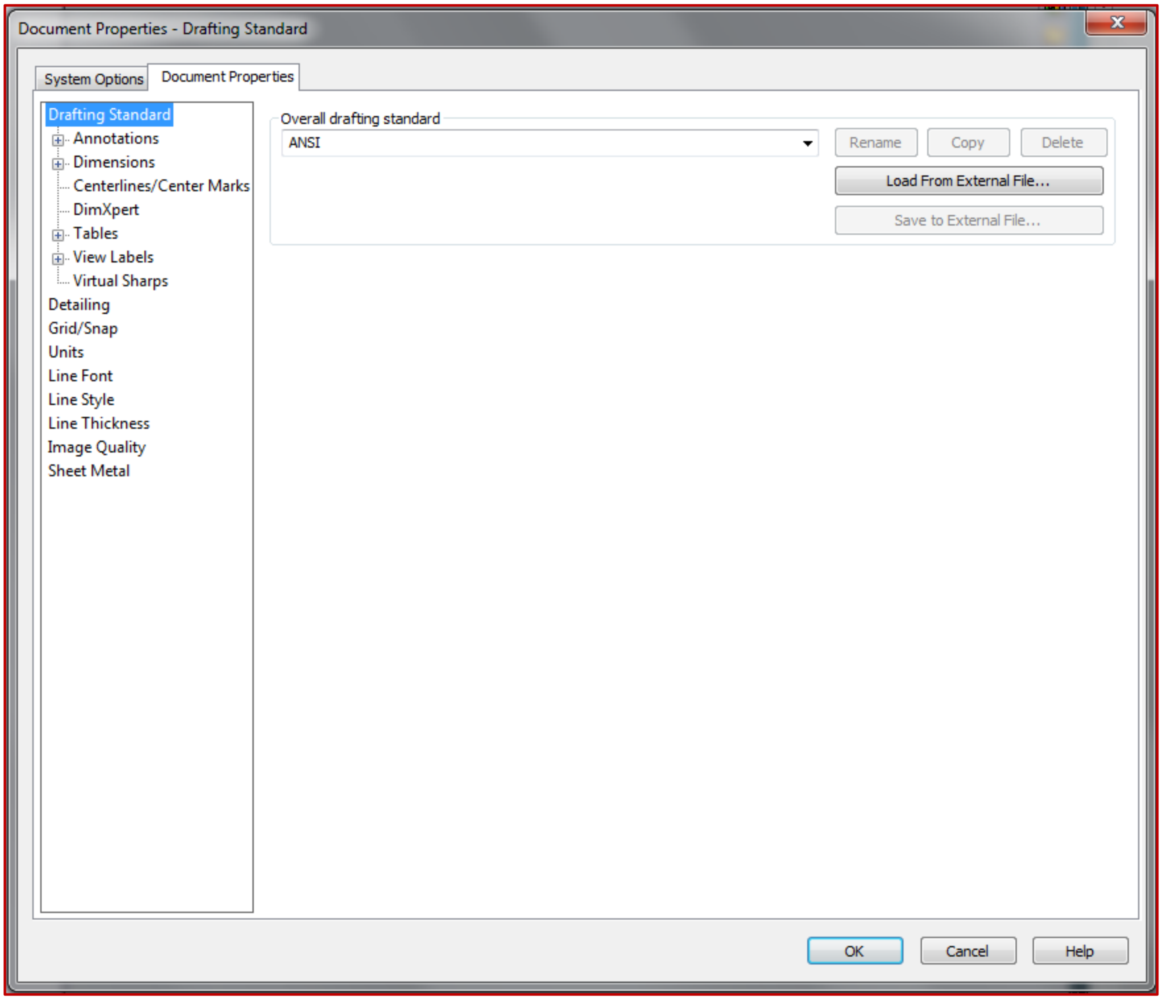


Figure 12.4 – *Tools→Options→Document Properties*

Step 100: Enter the Title Block information.

✓ Your screen should look like Figure 12.2b.

✓ Right-click in the graphics area and select ***Edit Sheet Format*** to change the Title Block.

✓ On the space above the Title, type your name or the name of your school or company. To type your name, right click and select ***Annotations→Note***.

✓ After you finish typing, click to locate the name and then press **Escape** on your keyboard. This will prevent duplicating the name everywhere you click.

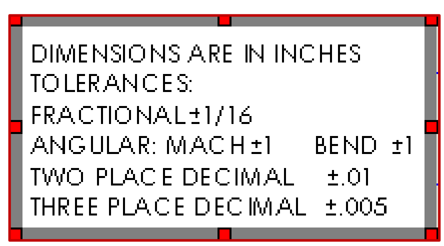


Figure 12.5 – Tolerances in the Title Block

✓ Double-click the box with the ***TOLERANCES*** shown on figure 12.6. Now you can make changes. We will add the tolerances for machining and bending in degrees.

✓ Click after **FRACTIONAL±** and add 1/16.

✓ Click after **ANGULAR: MACH±** and add 1.

- ✓ Add 1 after **BEND±**. to indicate 1 degree tolerance.
- ✓ After **TWO PLACE Decimal ±** add 0 .01 and for **THREE PLACE DECIMAL ±** add 0.005.
- ✓ See Figure 12.6.
- ✓ Click the **Edit Sheet** icon on the upper right corner of the screen or Right-click the graphics area and select **Edit Sheet**.
- ✓ Click ***File→Save as*** and select ***Drawing Template*** in the pull down menu. Name the template ANSIinchLandscapeA.drwdot.

Tolerances

The title block tolerances are the default values and are included in the organization's drafting practices manual. They are based on industry experience and manufacturing capabilities. The best tolerance is one that insures that the product can be assembled easily and works as designed. Geometric Dimensioning and Tolerancing as defined in ASME Y14.5 and its ISO equivalent, defines the rules and practices that will achieve optimum tolerances consistently.

12.4 Creating a Detail Drawing of the Skateboard Deck

Each part in a product must have a detail drawing. A detail drawing has all the information needed to manufacture the part, including dimensions, the material, heat treatment, finish and all other relevant information. In addition, a detail drawing's title block will have the part name and part number, the name of the designer and other information that can help locate old drawings and the people involved in their creation.

Cleaning-Up your Model or Drawing

As you create your model or drawing you can find that you need information such as axes, planes, origins, relationships and other references. However, such information do not belong in a drawing and can be distracting when you show your models to others. To clean-up your model or drawing for presentation to others, click ***View→Hide All Types***. This will hide all unnecessary references, which can be brought back by clicking again. You can also choose to hide specific information only.

Step 101: Open a new document and select the A-size drawing template created in the previous section. Notice that the ***CommandManager*** now has tabs for ***View Layout*** and ***Annotation***.

- ✓ Click ***Insert→Drawing View→ Model*** on the ***Main Pull-down Menu*** or click ***Model View*** on the ***CommandManager*** to get the dialog box in Figure 12.7.
- ✓ Click ***Browse*** and locate your skateboard deck model.

✓ Click ***Open*** or double click the model.

✓ Click the ***Next*** arrow to get the dialog box in Figure 12.8.

✓ In ***Orientation***, select the ***Front*** view, drag the cursor to the location you want place the Deck and click and release the left mouse button.

✓ The first view that you drag to the graphics area becomes your **Front View**.

✓ If you drag and click you can also place the **Top View**, the **Right View** and the **Isometric View** at 45 degrees. These are the typical views used in technical drawings. See drawing SKBD101 in Appendix A.

✓ Click the check mark or press Escape on the keyboard to accept your choice of views.

✓ If you selected the best sketching plane when you created the model, you have appropriate views. If you did not choose a good sketching plane, you can try again by undoing what you have and selecting another starting view to get a better combination of views. This can be confusing.

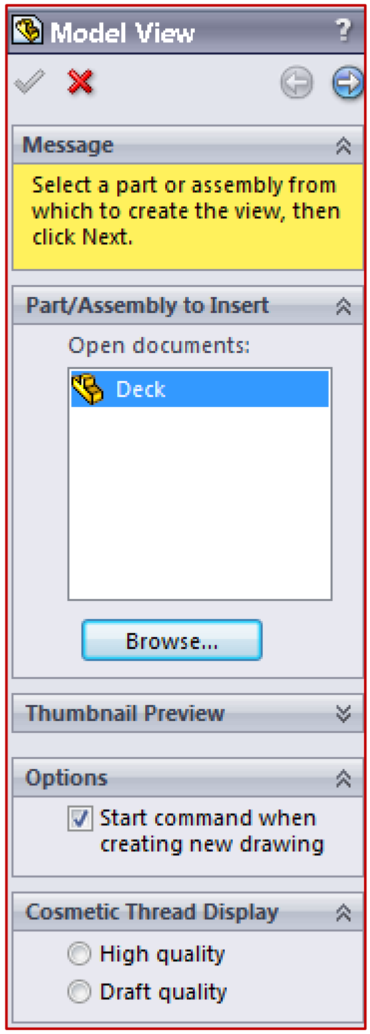


Figure 12.6 – *Insert→Drawing View→Model Property Manager*

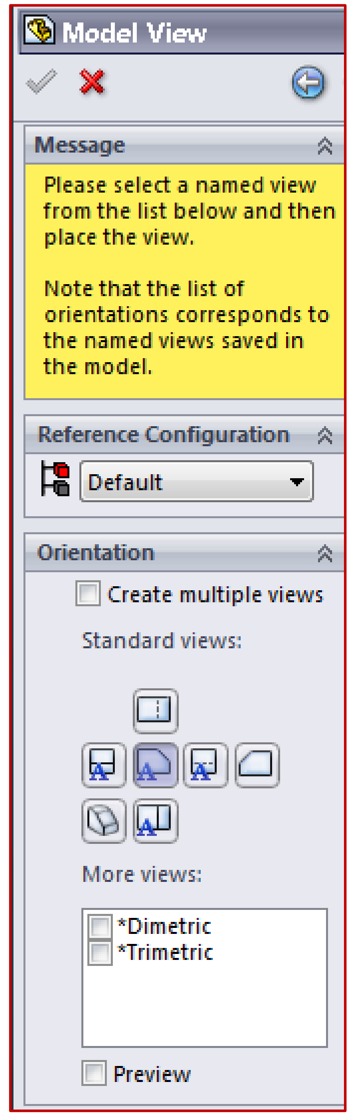


Figure 12.7 – Insert Front View

✓ To move the ***Front*** view to a new location, move your cursor over the ***Front*** view dashed lines or model and click and drag. Release at the new location. If you have additional views, you will notice that all the views move together.

✓ To move the other views, you also click on the dashed line or the part, but you can move in only one direction. Only the Front View, the first one placed on the graphics area, can move in two directions.

✓ If you need additional views after you accept your earlier choice, click the ***Projected View*** icon on the ***View Layout*** tab, click a view and drag and release to create additional views.

✓ In our deck example we need the **Front**, **Top**, **Right** and **Isometric Views**. See drawing SKBD101 in Appendix A.

- ✓ If you click one view to make it active and then click ***Display Style*** on the ***View (Heads-Up) toolbar*** you will notice that the view has ***Hidden Lines Removed***. Most engineering organizations use this option instead of showing hidden lines because hidden lines can result in confusion when drawing complex parts. Instead of hidden lines, most organizations use sections and additional views to show details not seen on the primary views.

Part Names and Part Numbers

Parts are tracked by their part number, not their part name. The reason is that it is easier to organize, store and retrieve a sequence of numbers than descriptive names. A part number will typically include information about the project and the assembly where it belongs.

This is the reason why SolidWorks stores the name you use to save your drawing as the part number.

- ✓ Change the **Isometric View** to include ***Shaded with Edges*** to get a more realistic representation.

- ✓ Click and drag each of the views to notice that:
    1. When moving the ***Front*** view, the ***Top*** and ***Right*** views also move.
    2. The ***Top*** view moves only up and down and will always stay aligned with the ***Front*** view.
    3. The ***Right*** view also stays aligned with the ***Front*** view.
    4. The ***Isometric*** view is free to move in any direction.

- ✓ If you can see the blue origin in your drawing, you can toggle on/off by clicking ***View*** on the Main Pull-down Menu and clicking ***Origins***. Other ***View*** menu options include: ***Temporary Axes***, ***Points***, ***Sketch Relations*** and others. They do not belong in a drawing. For drawings, it is best to select ***Hide All Types***.
- ✓ Click ***View→Hide All Types*** and save your drawing as SKBD101.

Drafting Standards

Drawings are legal documents and are in court to adjudicate fault in negligence cases. For that reason, they must be accurate and correct. Many reviews by multiple reviewers are needed to insure accuracy and correctness and this makes the process of producing drawings labor and time intensive.

Although computers and CAD software like SolidWorks have reduced the time and effort necessary to create drawings, they have not replaced humans. The creation of drawings is more art than science and only humans can decide what is appropriate and looks best. There are guidelines developed from experience that can help produce attractive and complete drawings with all the information needed to build a product or structure. These guidelines are usually collected in standards and drafting practice manuals. Two popular standards are ANSI and ISO. ANSI (American National Standards Institute) is common in the United States, Canada and the U.K. while ISO (International Organization for Standardization) is used worldwide. In addition, every engineering organization will have a proprietary drafting practices manual.

Step 102: Next, we will insert vertical break and detail views.

- ✓ Vertical and horizontal breaks are used when objects have one dimension that is significantly larger than the others, for example flagpoles and pipelines. The break allows them to be drawn in a scale that shows the smaller dimensions clearly.
- ✓ Click to select the ***Front*** view and click ***Insert→Drawing View→Vertical Break*** **on the Main Pull-down** Menu or click ***Break*** on the ***CommandManager→View Layout tab*** to get the ***PropertyManager***.
- ✓ Move the cursor to the location where you want to place the first break line and click.
- ✓ Double-click to place the second line.
- ✓ Click the check mark or press **Escape** on the keyboard to accept.
- ✓ Repeat the previous step to create a break in the ***Top*** view. The results are shown in the detail drawing SKBD101 in Appendix A.
- ✓ To see the edge chamfer clearly, we will create a detail view.
- ✓ Click ***Insert→Drawing View→Detail***.
- ✓ Click in the middle of the area that you want to include in your detail view and drag the cursor to create a circle.
- ✓ Release the mouse button.
- ✓ Drag the detail view to the location you want and click again to release.
- ✓ Save your drawing again. Verify that the file name is SKBD101.drw.

Step 103: Dimensioning is both, a science with rules, and also art, where the designer must be aware of aesthetics.

- ✓ It is possible to conceive multiple arrangements of dimensions that are valid and acceptable but different.
- ✓ The most important rules are:

1. The drawing must have all the dimensions needed to fabricate the part.
2. Dimensions shall never be repeated. Never double dimension.
3. Dimension the most logical view.
4. Group related dimensions in the same view.
5. Use clear dimensions that can be interpreted only one way.
6. Minimize the accumulation of tolerances.
7. Avoid dimensions to hidden lines, if possible.
8. Never dimension inside the part.

All the rules can be overruled to make the drawing clear and aesthetically pleasant.

Step 104: To create dimensions in SolidWorks drawings, use ***Smart Dimension*** and:

1. click a line to display its length dimension, or
2. click each of the two points at the end of a line to show the distance between the points, which is also the length of the line, or
3. click two parallel lines to display the distance between the lines, or
4. click two lines that are not parallel to display the angle, or
5. click a diameter of a circle or an arc to display their diameter or radius.

✓ When you click a dimension it becomes active and you can see the dimension ***PropertyManager.*** See Figure 12.9.

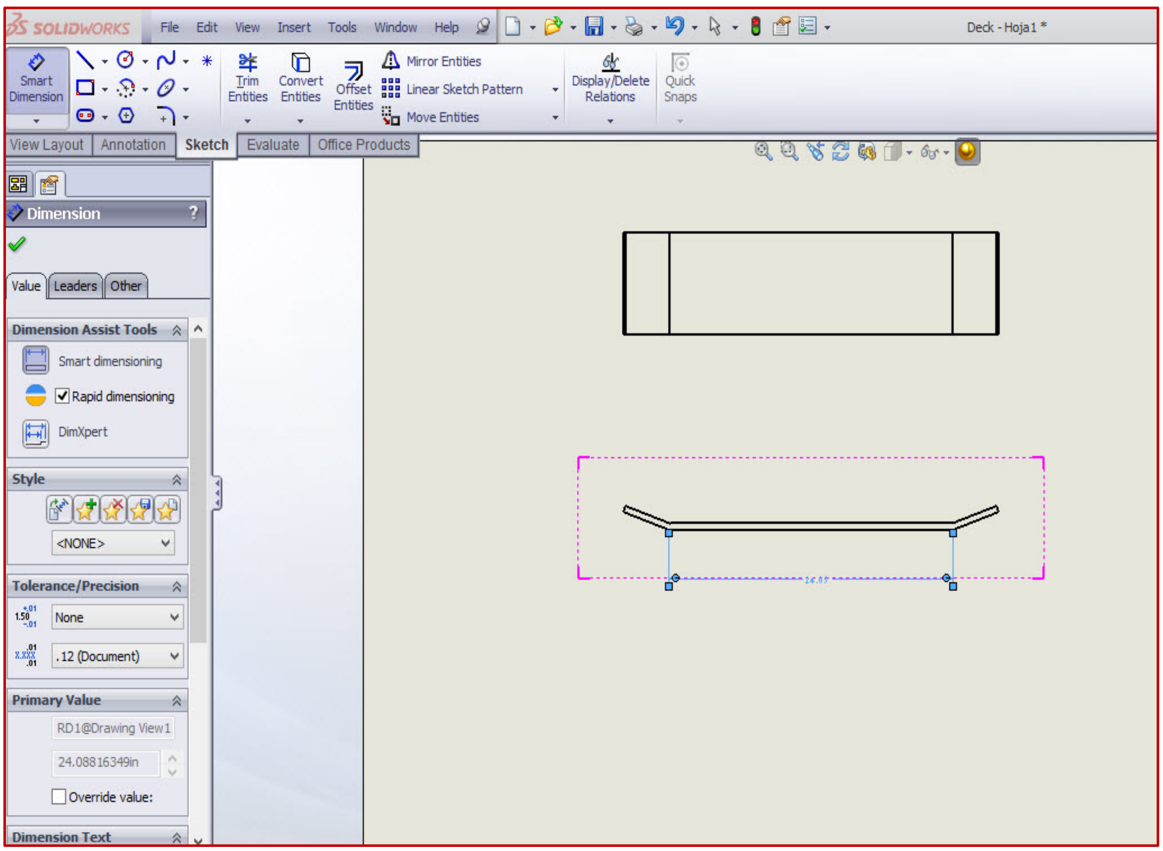


Figure 12.9 – *PropertyManager*

- ✓ The options in the ***PropertyManager*** can be used to tailor how dimensions are displayed. For example:
    1. Change the number of decimal points in the dimension in the ***Tolerance/Precision*** box. You can also add tolerances to your dimensions by using the appropriate menu choice. See Figure 12.10.
    2. You can add symbols in the ***Dimension Text*** box. In the example, 2X was added by clicking to the left of **<DIM>** and adding the text.
    3. The arrows can be placed inside or outside of the leader lines by selecting the proper choice on the ***Leaders*** tab.
- ✓ Click Save As and use SKBD101 as the filename.

Step 105: The last step is to fill the missing information in the title block. To toggle between making changes to the drawing and making changes to the title block:

- ✓ Right-click in a blank space in the graphics area and select ***Edit Sheet Format***.
- ✓ Be aware that right-clicking close to one of the views will make it active and ***Edit Sheet Format*** will not be available.

Step 106: In the ***Edit Sheet Format*** mode, the information in the title block can be added or changed.

- ✓ Double click the location to open the input box and type the information.
- ✓ Click to accept the text and press escape to finish.
- ✓ If double clicking does not open the input box, right-click to show the menu and select ***Annotation→Note***, drag the annotation box to the desired location, and click to release.
- ✓ Add the name of the part, the material and the finish as shown in SKBD101.
- ✓ To exit the ***Edit Sheet Format*** mode, right-click in a blank space in the graphics area and select ***Edit Sheet***, or click on the Edit Sheet icon on the upper-right corner of your screen.
- ✓ Open the ***Design Binder*** and add your name, date and other useful information to the Journal and save your SkateboardDeck drawing again as SKBD101.

Using Drawing Tables

Storing drawings used to be a very expensive requirement. Even today, with CAD and electronic drawings, Storage is still a concern. As a result, most organizations minimize the number of drawings created. In situations where the dimensions of similar parts are different, it is possible to use a Table to specify multiple parts in the same drawing. Drawing SKBD116 representing the two spacers is an example. To create a Table, click ***Insert→Table→General Table***.

12.5 Creating Detail Drawings for Commercial Items

Some engineering organizations produce a separate detail drawing for each commercial part, such as a bearing or a specific type of nut or bolt. Other organizations do not produce drawings for commercial items and simply keep the information in a database and print what is needed.

Commercial items are not manufactured by the company that uses them. They are purchased by what is typically called the Purchase or Procurement group. The information they need is the part number or identifying number (sometimes called the SKU or Stock Keeping Unit), the quantity of parts needed and the name, address and telephone number of one or more suppliers. In addition, a picture of the part can be helpful. Figure 12.10 is an example of a detail drawing for a commercial bearing.

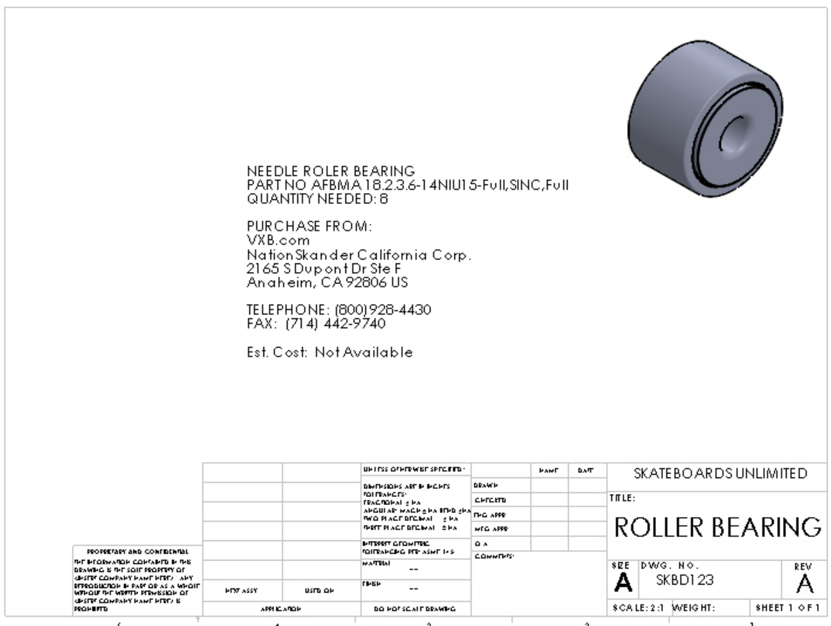


Figure 12.10- Example of Detail Drawing for a Commercial Item

12.6 Creating e-Drawings

SolidWorks eDrawings can be used to share solid models and drawings when the recipient does not have SolidWorks. A free viewer can be downloaded from http://www.edrawingsviewer.com/ or in http://www.solidworks.com. Also, it was probably installed on your computer as part of the SolidWorks installation.

Step 107: There are two ways to create an eDrawing:

1) use the command **Save As** on the **File** pull-down menu of the **Main Drop-down Menu** and select ***eDrawing*** on the pull-down menu for ***Save as type***, or

2) click **File→Publish eDrawing File** on the **File** pull-down menu of the **Main Drop-down Menu**.

Sharing Drawings

Often, it is necessary to share drawings and other design information with co-workers, customers and other stakeholders. Unfortunately, to open SolidWorks files we need the software. If you want to share drawings and the recipient does not have SolidWorks, you can use the command ***Save As*** and select PDF or JPEG format. Most personal computers will be able to open these formats. If the recipient has AutoCAD software, you can save your files as DXF (Drawing Exchange File). Other file formats that you can use are IGES, STEP and STL. All are available in some versions of SolidWorks. Translators to most CAD formats, including ProE, CATIA, Rhino, etc., are also available.

If you save your drawings as detached drawings (.slwddrw), the recipient does not need the solid models to open the drawings, but must have SolidWorks to open them. Finally, another useful alternative is to save your work as e-Drawings. This is a free application available at SolidWorks.com that you can use to view and mark-up drawings.

12.7 Using the Spell Checker

To use the spell checker we must first verify that SolidWorks has access to a dictionary. If your computer has Microsoft Office installed, you have satisfied this requirement.

Step 108: To spell-check you document:

- ✓ Click ***Tools→Spelling*** on the **Main Drop-down Menu**.
- ✓ The spelling checker will report any typographical error in the drawing.

Practice Exercises

1. Describe the ***FeatureManager design tree*** for a drawing.
2. Check the spelling in your Skateboard Deck drawing (SKBD101).
3. Click ***Help→SolidWorks Help*** in the **Main Drop-down Menu** and select ***Contents*** and study the section ***Derived Drawing Views***. List and describe what views are available in SolidWorks.

Questions

1. Suggest what manufacturing processes can be used to make one part in the skateboard.
2. Develop an assembly sequence to put together the skateboard.
3. What is ANSI?
4. What is ISO?

5. ***Click Tools→Options→Document Properties*** and find what other drafting standards are available besides ANSI and ISO. Where are they popular?
6. Explain what is/are:
   a. Nominal size.
   b. Basic size.
   c. Actual size.
   d. True position.
   e. True radius.
   f. Limit dimensions.
   g. Reference dimensions.
   h. Datum.
   i. Dimension lines.
   j. Extension lines.
   k. Leader lines.
   l. Section view.
   m. Detail view.
   n. Broken-out section.

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

ANSI Standard
Engineering Drawing and Design – by D.A. Madsen and D.P. Madsen, Cengage Learning, Clifton Park, NY, US

Interpreting Engineering Drawings – C. Jensen and J.D. Helsel, Cengage Learning, Clifton Park, NY, US

British (BSI) Standard
Manual of Engineering Drawing – by C.H. Simmons, N. Phelps and D.E. Maguire, Elsevier, Oxford, UK

ISO Standard
Manual of Engineering Drawing – by C.H. Simmons, N. Phelps and D.E. Maguire, Elsevier, Oxford, UK

The Mechanical Engineering Drawing Desk Reference – by Paul Green, CreateSpace, Seattle, WA, US

Videos from SolidProfessor 3D Skills

- Introduction to Drawings
- Model View
- Section View
- Detail View

Videos from SolidProfessor SolidWorks for Beginners

- Associativity
- Annotations

Videos from SolidProfessor Core Concepts for Drawings

- Auxiliary View
- Broken View
- Scale
- Custom Properties and Parametric Notes

Videos from SolidProfessor Intro to eDrawings

- eDrawings Overview

Internet Resources

CADeducators website: http://www.cadeducators.com/ansi-english.php

http://www.cadeducators.com/iso-english.php

American National Standards Institute (ANSI) website: http://www.ansi.org

International Standards Organization (ISO) website: http://www.iso.org

Wikipedia.com: http://en.wikipedia.org/wiki/Engineering_drawing

NASA Technical Drawings: http://history.nasa.gov/diagrams/diagrams.htm

Lesson 13 – Creating the Assembly Drawing with the BOM

13.1 Lesson Objectives

After completing this Lesson, you will be able to:

- Explain and create assembly drawings in SolidWorks.
- Explain and create the Bill of Materials.
- Explain the assembly drawing title block.
- Explain and use balloons in the assembly drawing.
- Explain and use the ***Balloon*** and ***AutoBalloon*** commands.

13.2 Introduction

The purpose of the assembly drawing is to provide the information needed to assemble the product. The person or team assembling the product from parts does not need the exact dimensions of the parts, or the material used to make them. What they need is an exploded assembly drawing showing the sequence in which the components are added to the assembly.

A Bill of Materials (BOM) is a list of all the parts and commercial components in the assembly. Often it is included in the assembly drawing, but for large assemblies it is more convenient if the BOM is on a separate sheet. In practice, the person assembling the product will verify that all the parts are available by comparing the bill of materials with the parts available. Typically they do this before the process of assembly begins.

In this Lesson we will create an assembly drawing and BOM for the wheel and tire assembly.

13.3 Creating the Assembly Drawing

The assembly drawing will use the ANSIinchLandscapeA template we created earlier.

Step 109: Open a new drawing document and select the template ANSIinchLandscapeA.

- ✓ Click ***Insert→Drawing View→Model*** on the ***Main Pull-down Menu*** or click ***Model View*** on the ***CommandManager***
- ✓ Click Browse and find the model of the tire assembly.
- ✓ Click the tire assembly and then click OK. This will get the tire assembly model into the drawing document.

- ✓ In ***Orientation***, select the ***Isometric*** view. We could also choose to present the assembly as multiple orthogonal views, similar to a detail drawing.
- ✓ At the bottom of the properties menu, click ***More Options***.
- ✓ Select ***Show in exploded state***.
- ✓ Click the check mark to accept.
- ✓ Repeat ***Insert→Drawing View→Model***, but do not explode the assembly.
- ✓ Click each of the views and select ***Display State→Shaded With Edges***.
- ✓ Click **Save As** and use the file name SKBD120.

Step 110: Right-click and choose ***Edit Sheet Format*** and fill the title block.

- ✓ Tolerances, materials and finish are never filled on assembly drawings. This is because each of the parts in the assembly can be made of a different material and can be manufactured with different tolerances.
- ✓ The wheel assembly drawing can be found in Appendix A and can be used as a reference when completing the title block.
- ✓ Save your drawing again.

Adding Sheets to your Drawing

As your drawing becomes more complex, you may decide to use multiple pages or sheets. To add additional sheets to your drawing, right click in the Graphics area, away from your drawing, and select “Add sheet”.

To delete a sheet click “Delete”.

13.4 Adding the BOM

Step 111: Click ***Insert→Tables→Bill of Materials*** on **the Main Pull-down Menu**.

- ✓ Click the skateboard assembly to open the BOM dialog box in Figure 13.1. Accept the default options.
- ✓ Click the check mark to accept and drag the mouse to position the BOM table on the top-left corner. See Figure SKBD120 in Appendix A.
- ✓ To add a column or row to the BOM table, right-click over the table and select ***Insert→Column*** or ***Insert→Row.***
- ✓ To type or make changes, double-click on a table cell.
- ✓ To exit the edit mode, click outside the BOM.

Step 112: Add balloons one at a time with the ***Balloon*** command in the ***CommandManager's Annotations*** tab, or use the ***AutoBalloon*** command. The balloons can be moved with click-drag.

- ✓ Notice that SolidWorks already knows which part you want to label with a balloon. You provided this information when you created the assembly.

Step 113: Step 98: Save your drawing a third time. Remember that the filename is SKBD120.

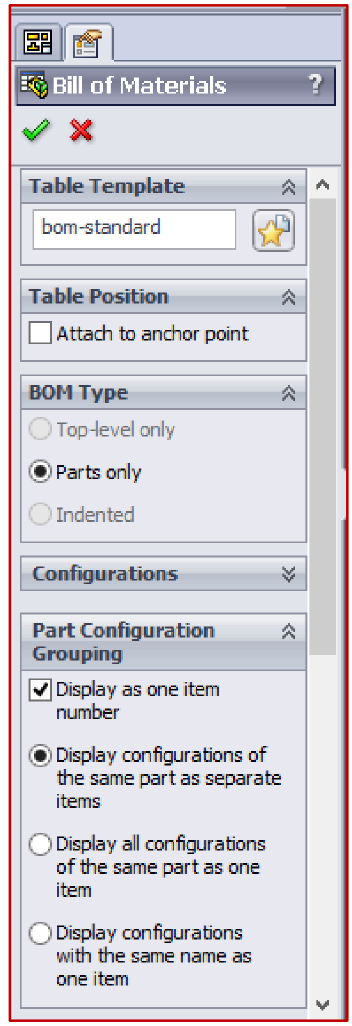


Figure 13.1 – Bill of Materials

Practice Exercises

1. Move the BOM to Sheet 2 of the assembly drawing. (Hint: Right-click and then click ***Add Sheet***. Notice that ***Sheet1*** and ***Sheet2*** tabs are shown at the bottom of the ***FeatureManager design tree***.)
2. Add a Part Description column to the BOM table.
3. Create the assembly drawing for the Truck Sub-Assembly. (Hint: You can use the drawing in the Appendix for reference.)
4. Create the drawing for the skateboard top assembly. (Hint: You can use the drawing in the Appendix for reference.)
5. Find the keyboard shortcut to spell check a document. (Hint: You will find the ***Spell Check*** command in ***Tools→Customize→Keyboard.*** The command is in the ***Others*** category. If you lose your geometry after you spell-check, use the ***Zoom to Fit*** and ***Rotate View*** commands to bring it back into view.

Problems

1. List the parts in a pencil. Do you recommend the use of sub-assemblies?

References

SolidWorks Bible – by Matt Lombard, John Wiley and Sons, Indianapolis, IN, US

SolidWorks Office Training Manual – by SolidWorks Corporation, Waltham, Massachusetts, US

SolidWorks 2014 Short and Simple – by OnlineInstructor.org

Introduction to Using SoildWorks – by W.E. Howard and J.C. Musto, McGraw Hill, New York, NY, US

ANSI Standard

Engineering Drawing and Design – by D.A. Madsen and D.P. Madsen, Cengage Learning, Clifton Park, NY, US

Interpreting Engineering Drawings – C. Jensen and J.D. Helsel, Cengage Learning, Clifton Park, NY, US

British (BSI) Standard

Manual of Engineering Drawing – by C.H. Simmons, N. Phelps and D.E. Maguire, Elsevier, Oxford, UK

ISO Standard
The Mechanical Engineering Drawing Desk Reference – by Paul Green, CreateSpace, Seattle, WA, US

Videos from SolidProfessor 3D Skills

- Bill of Materials
- Balloons

Videos from SolidProfessor SolidWorks for Beginners

- Assembly Drawing

Videos from SolidProfessor Core Concepts for Drawings

- Formatting Dimensions
- Custom Templates
- Custom Properties and Parametric Notes

Internet Resources

CADeducators website: http://www.cadeducators.com/ansi-english.php

http://www.cadeducators.com/iso-english.php

American National Standards Institute (ANSI) website: http://www.ansi.org

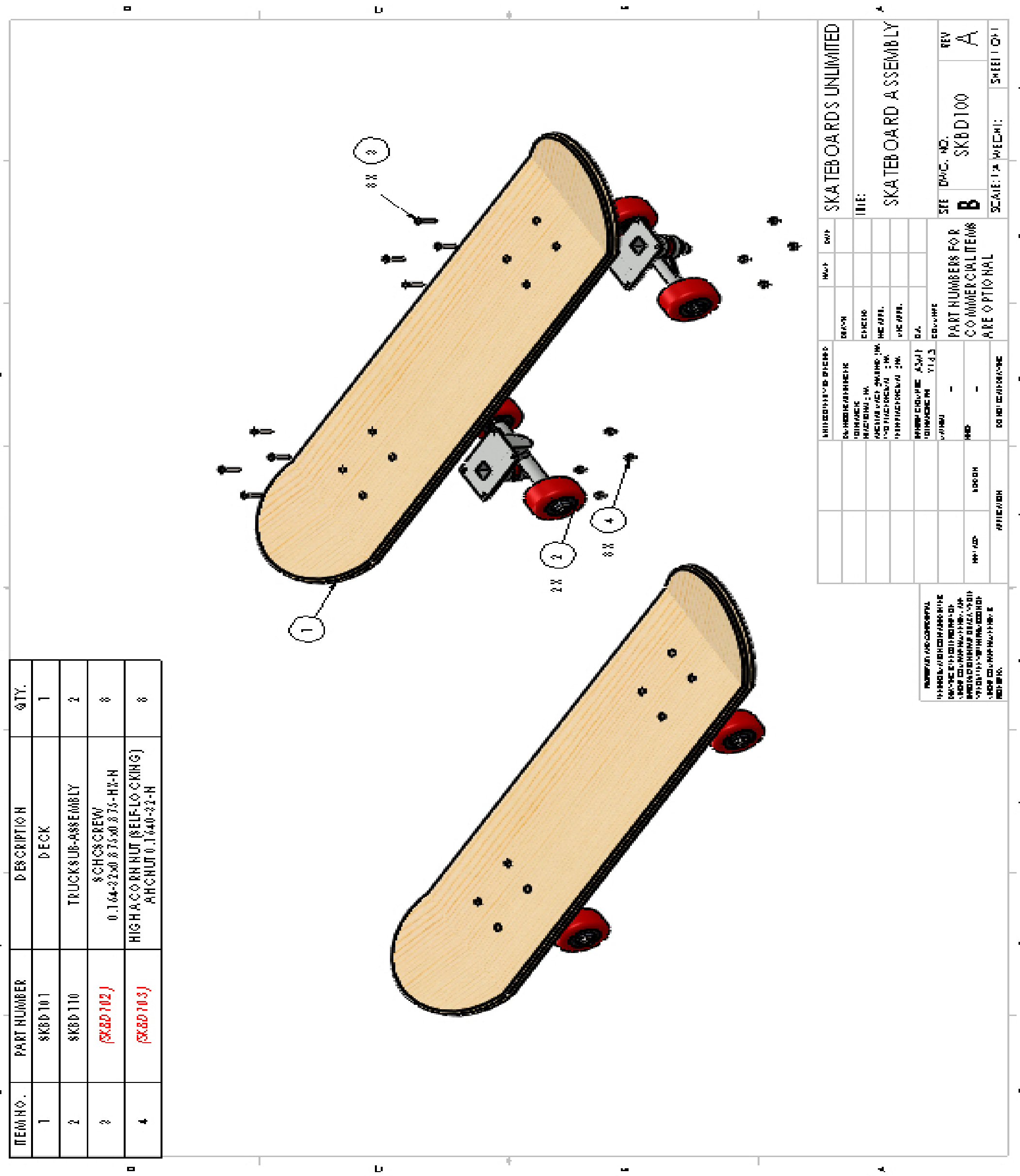
International Standards Organization (ISO) website: http://www.iso.org

Wikipedia.com: http://en.wikipedia.org/wiki/Engineering_drawing

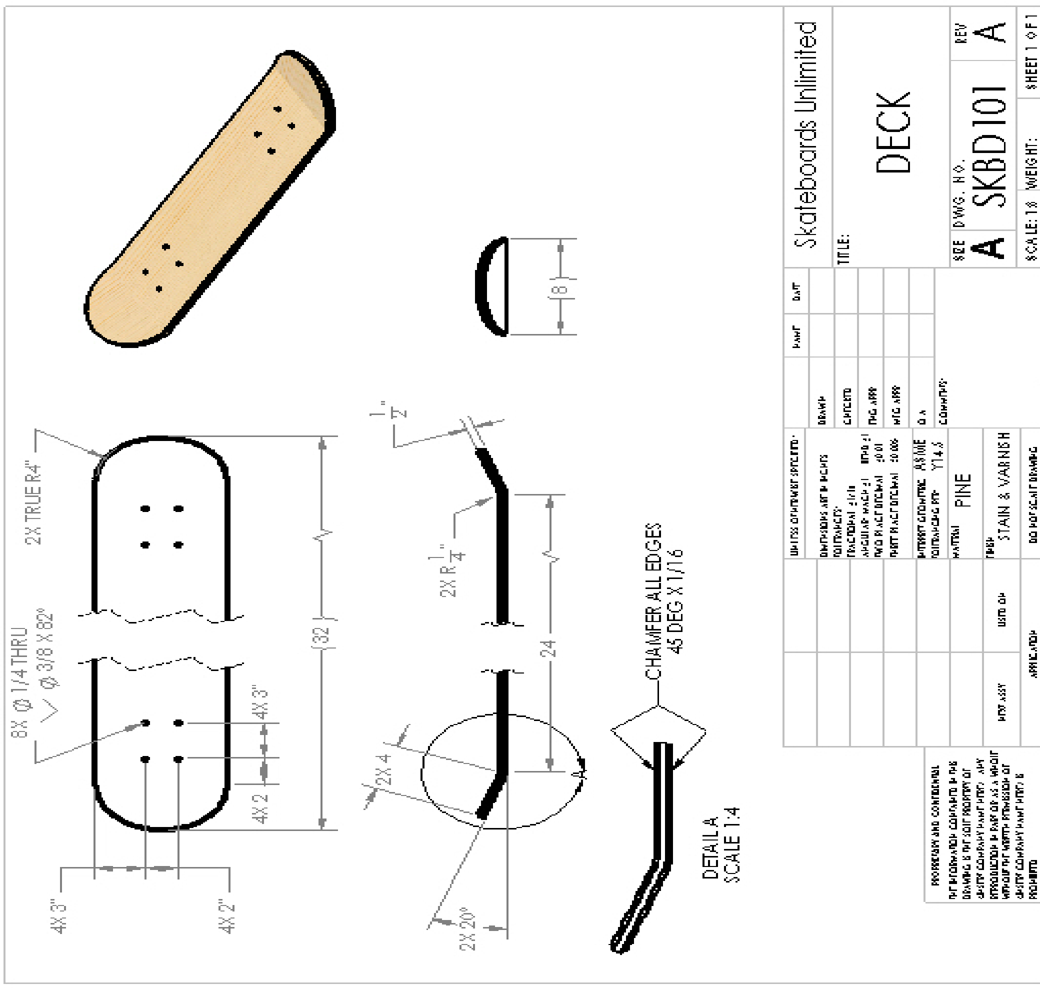
NASA Technical Drawings: http://history.nasa.gov/diagrams/diagrams.htm

Appendix A – Skateboard Working Drawings – ANSI/Inches

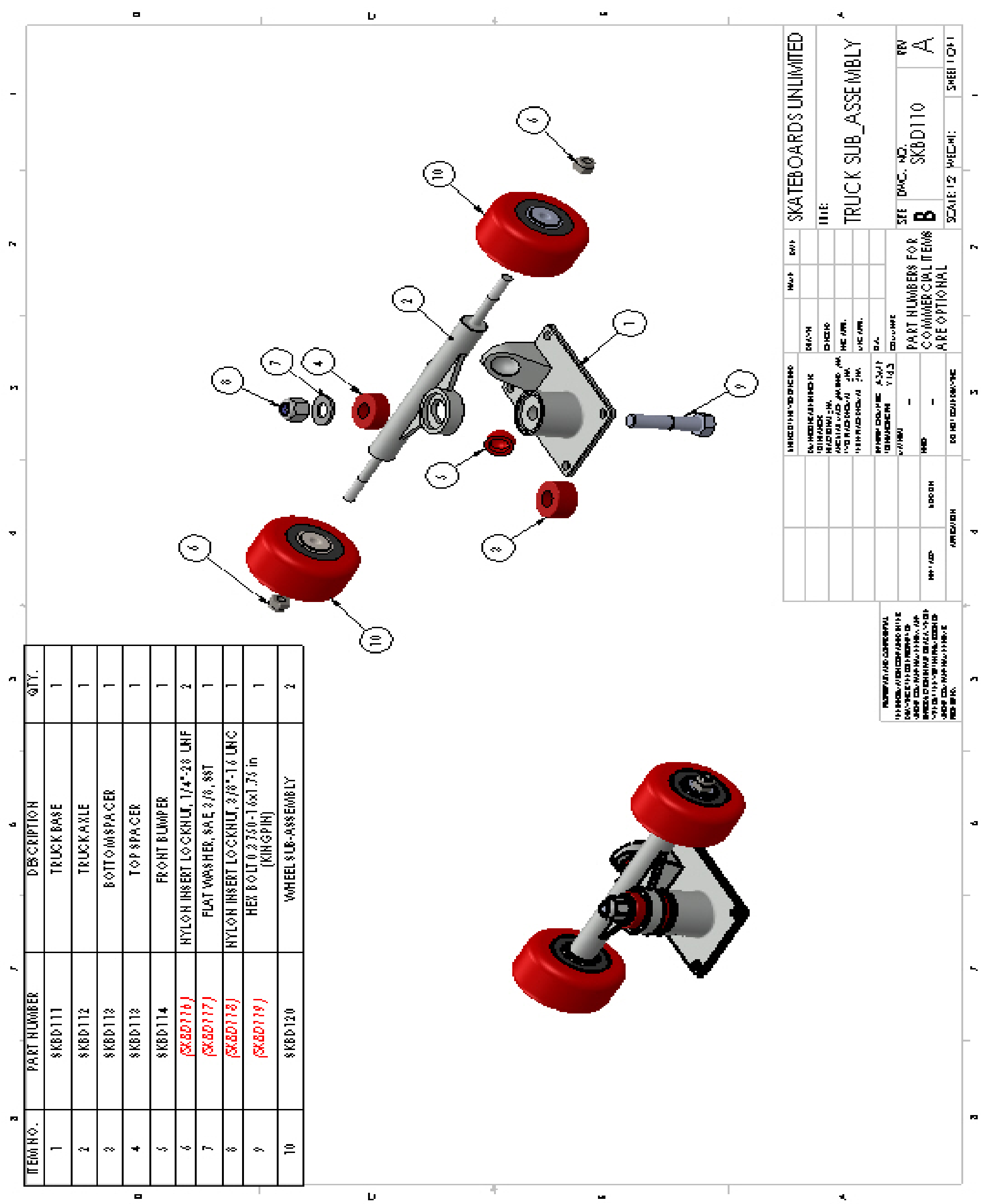
A1 – SKATEBOARD ASSEMBLY



A2 – DECK

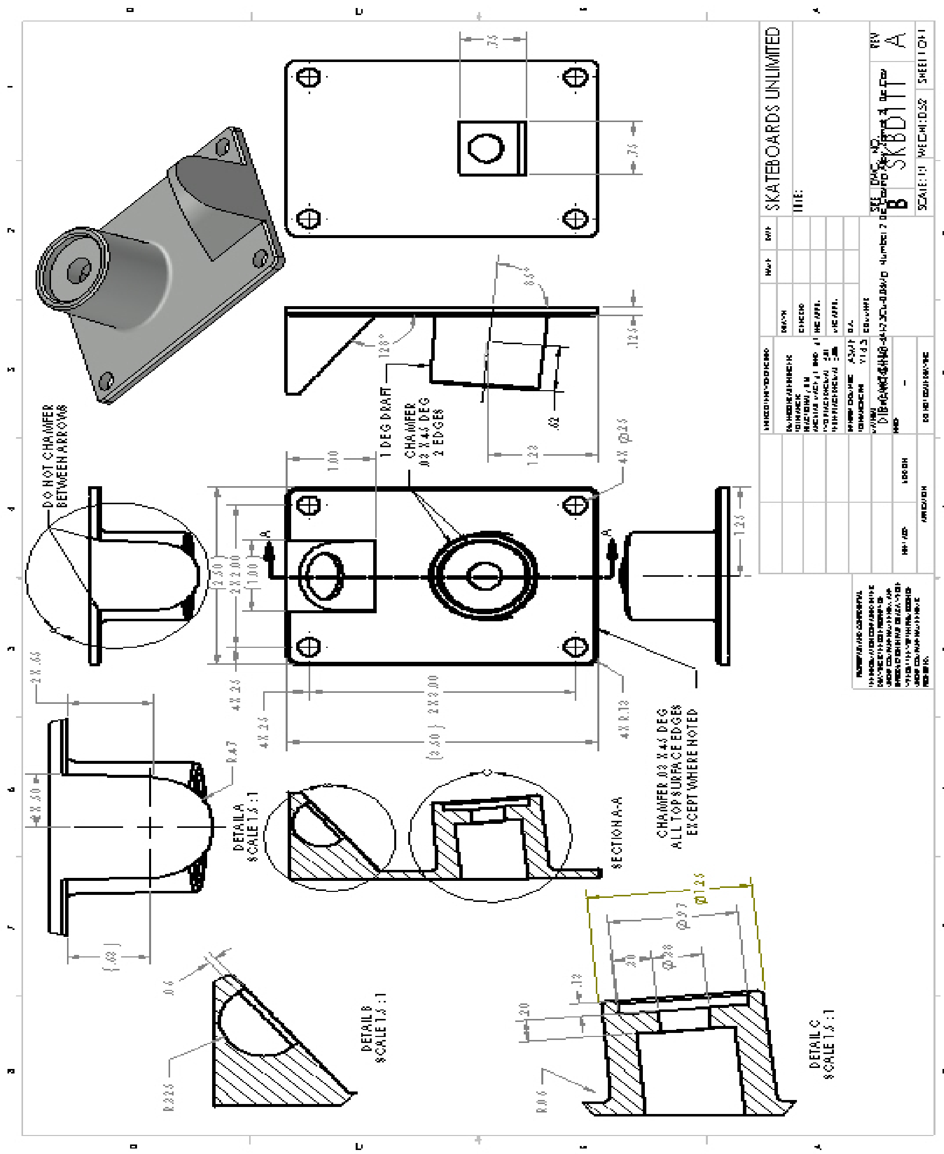

2X TRUE R4"
8X ∅ 1/4 THRU
∅ 3/8 X 82°
4X 3"
4X 2"
32
24
8
2X R 1/4"
2X 4
2X 20°
A
CHAMFER ALL EDGES
45 DEG X 1/16
DETAIL A
SCALE 1:4
Skateboards Unlimited
TITLE:
DECK
SIZE
A
DWG. NO.
SKBD101
REV
A
PINE
STAIN & VARNISH
SCALE: 1:8
WEIGHT:
SHEET 1 OF 1

A3 – TRUCK SUB-ASSEMBLY

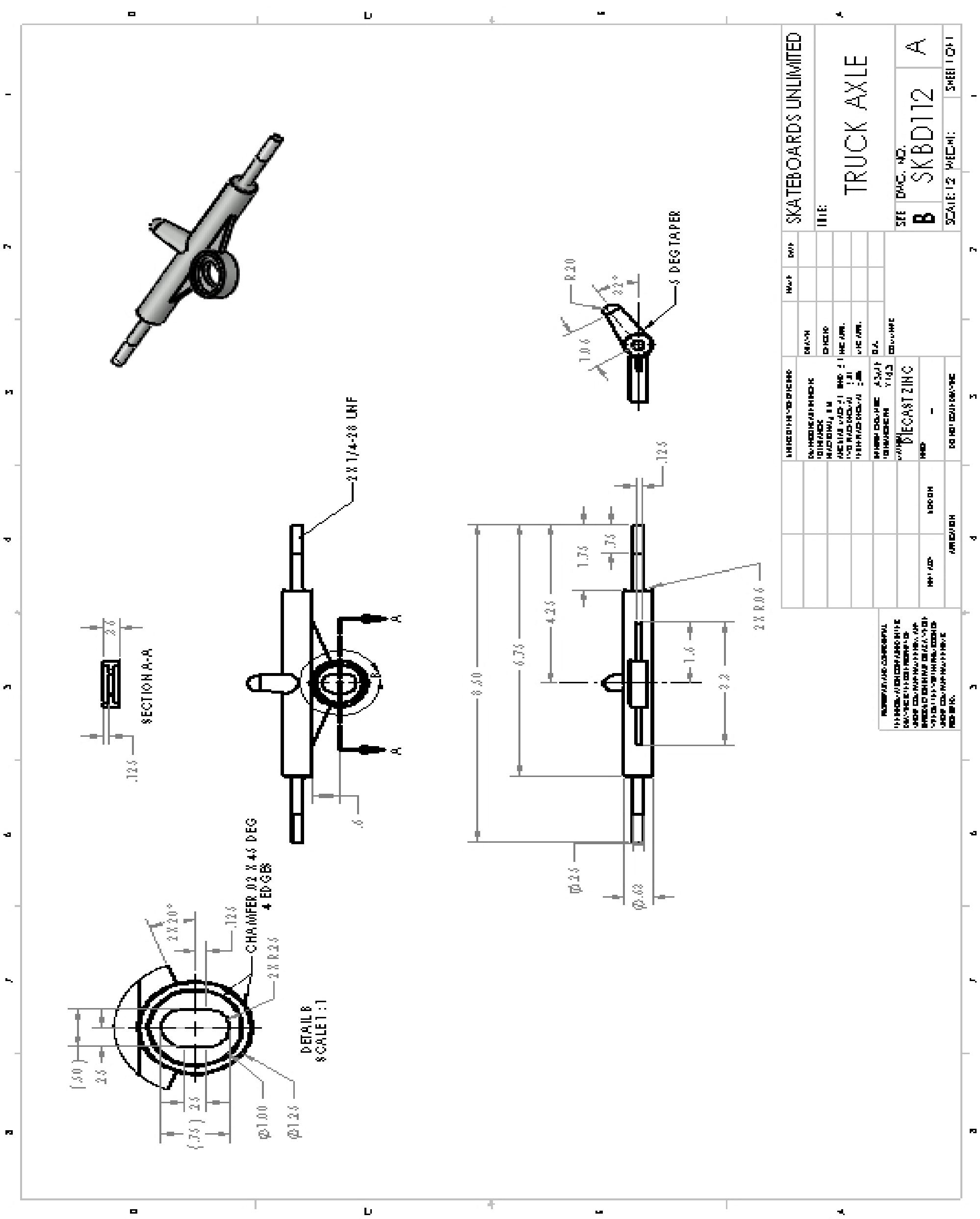


ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	SKBD111	TRUCK BASE	1
2	SKBD112	TRUCK AXLE	1
3	SKBD113	BOTTOM SPACER	1
4	SKBD113	TOP SPACER	1
5	SKBD114	FRONT BUMPER	1
6	(SKBD116)	NYLON INSERT LOCKNUT, 1/4"-28 UNF	2
7	(SKBD117)	FLAT WASHER, SAE 3/8, SST	1
8	(SKBD118)	NYLON INSERT LOCKNUT, 3/8"-16 UNC	1
9	(SKBD119)	HEX BOLT 0.3750-16x1.75 in (KINGPIN)	1
10	SKBD120	WHEEL SUB-ASSEMBLY	2

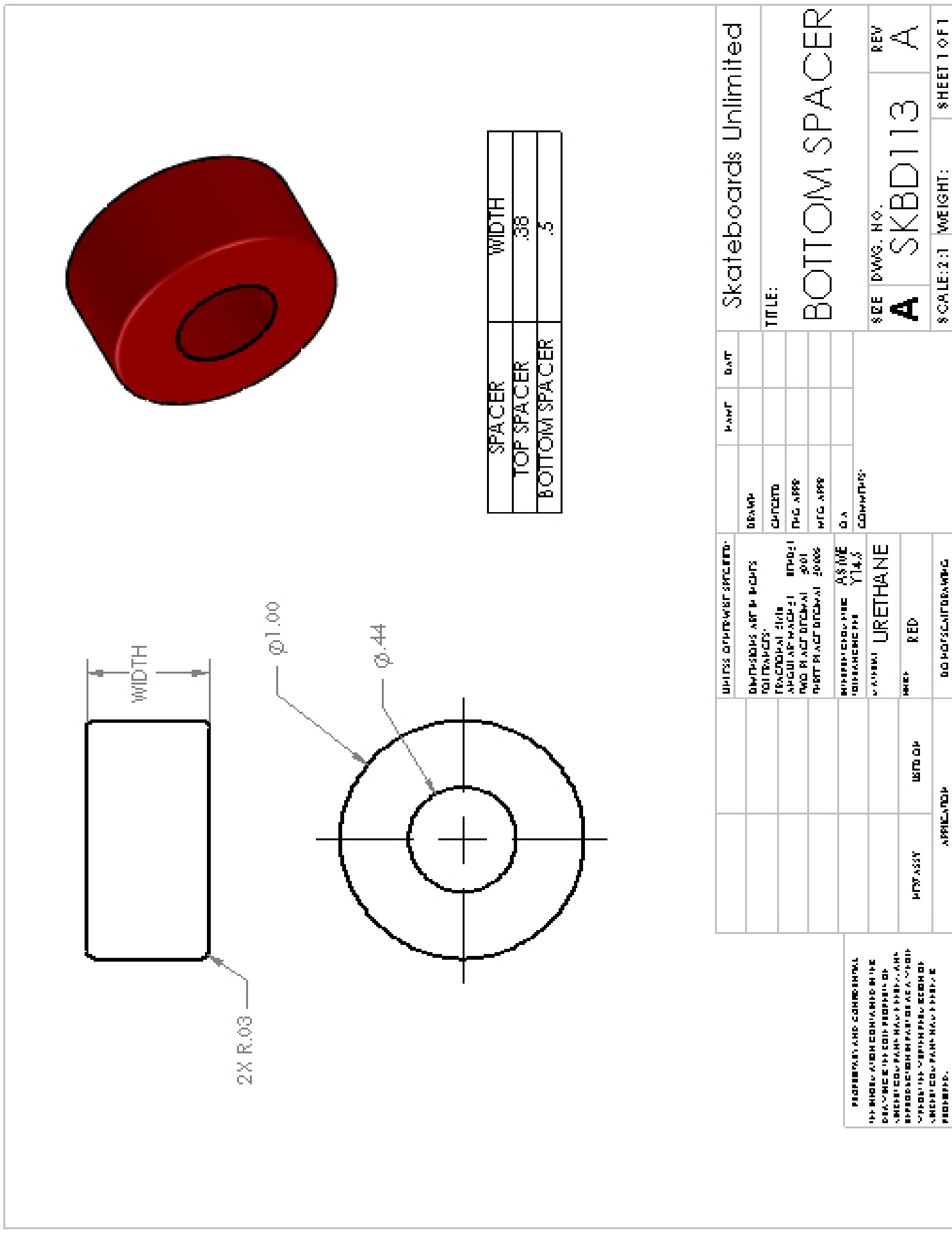
A4 – BASE


SKATEBOARDS UNLIMITED
SECTION A-A
DETAIL A
SCALE 1.5 : 1
DETAIL B
SCALE 1.5 : 1
DETAIL C
SCALE 1.5 : 1
DO NOT CHAMFER BETWEEN ARROWS
1 DEG DRAFT
CHAMFER .02 X 45 DEG ALL TOP SURFACE EDGES EXCEPT WHERE NOTED

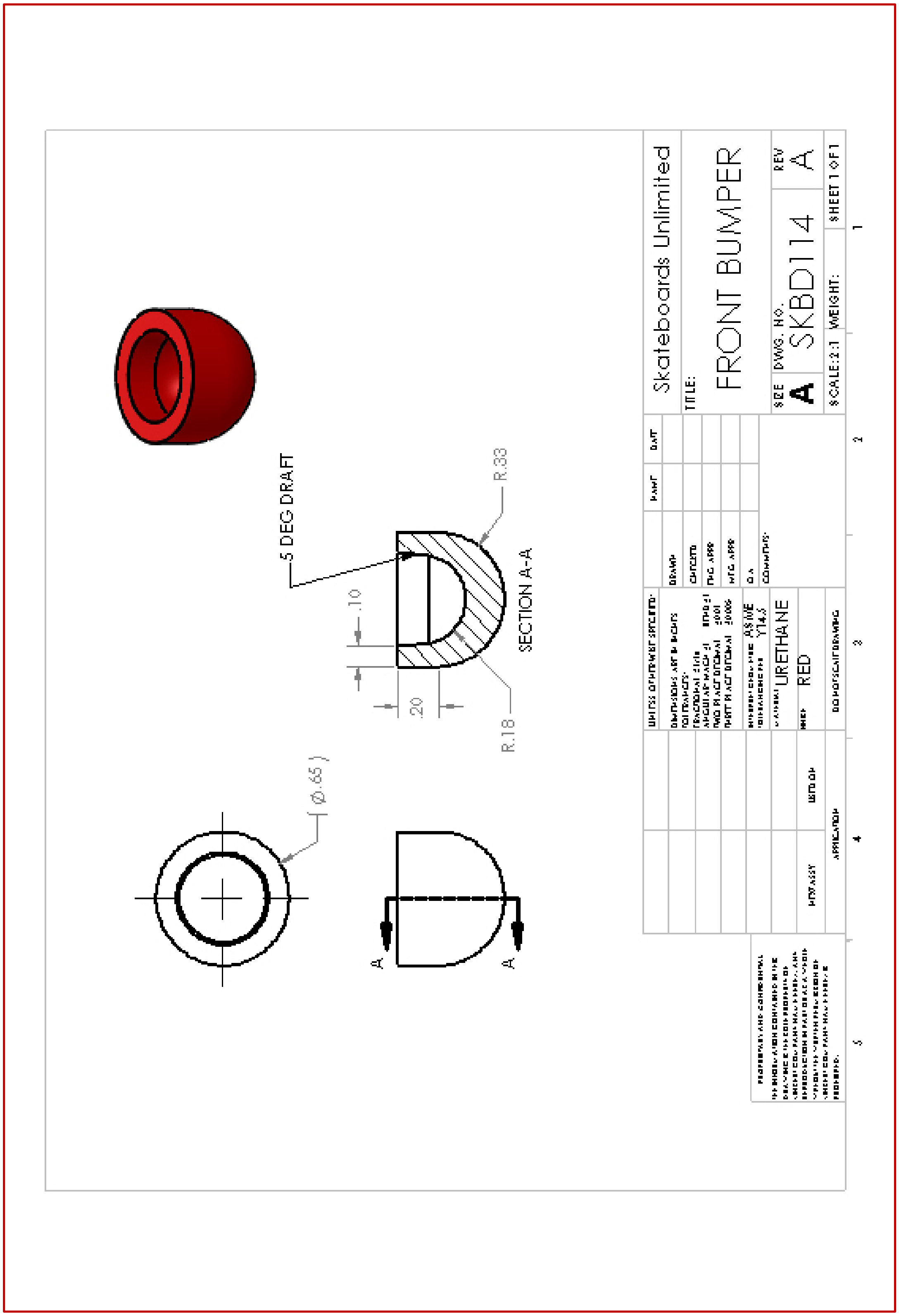
A5 – TRUCK AXLE



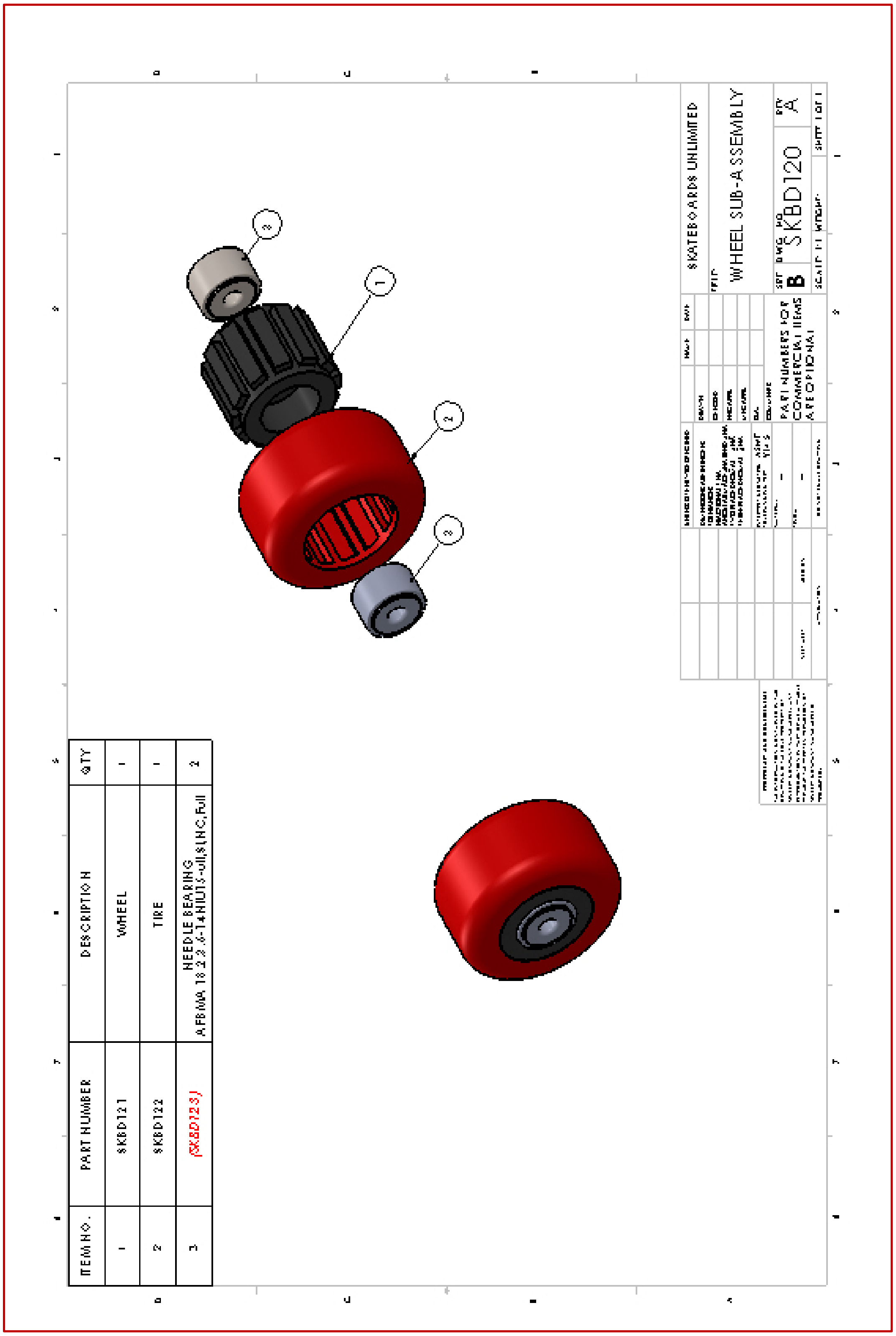
A6 – SPACER


Skateboards Unlimited
TITLE:
BOTTOM SPACER
SIZE
A
DWG. NO.
SKBD113
REV
A
SCALE:2:1
WEIGHT:
SHEET 1 OF 1
SPACER
WIDTH
TOP SPACER
.38
BOTTOM SPACER
.5
WIDTH
Ø1.00
Ø.44
2X R.03
ASME Y14.5
URETHANE
RED

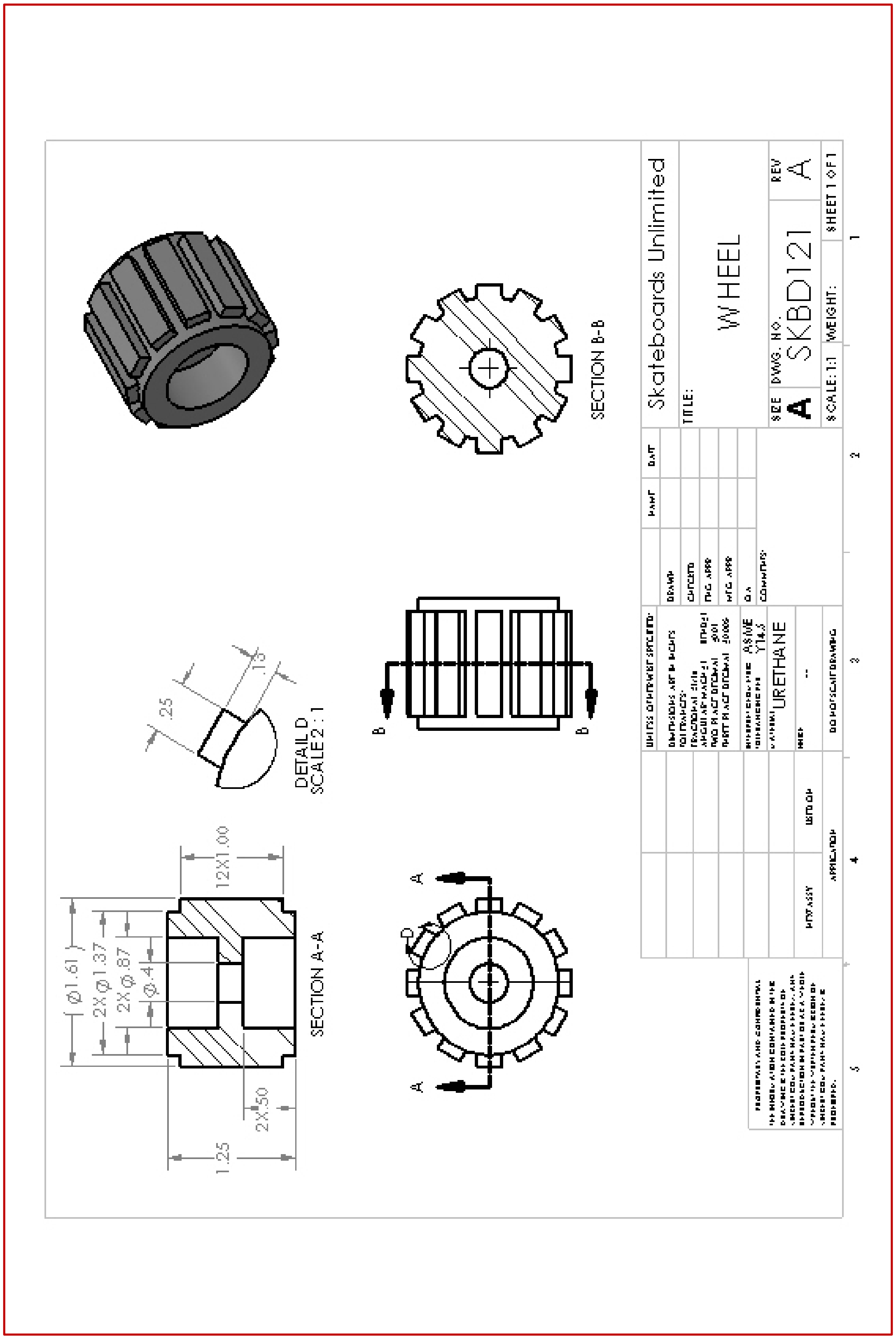
A7 – FRONT SPACER



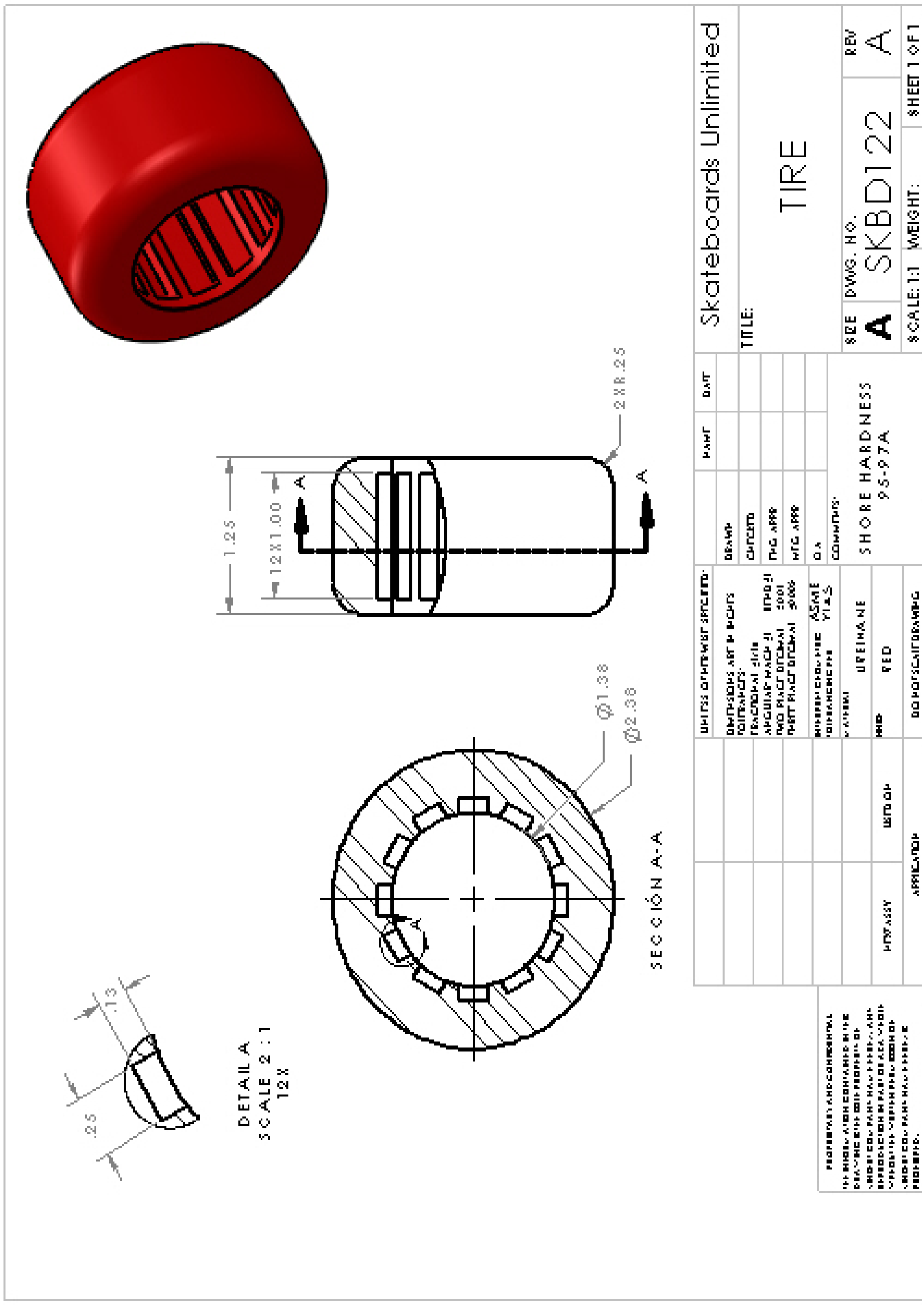
A8 – WHEEL ASSEMBLY



A9 – WHEEL

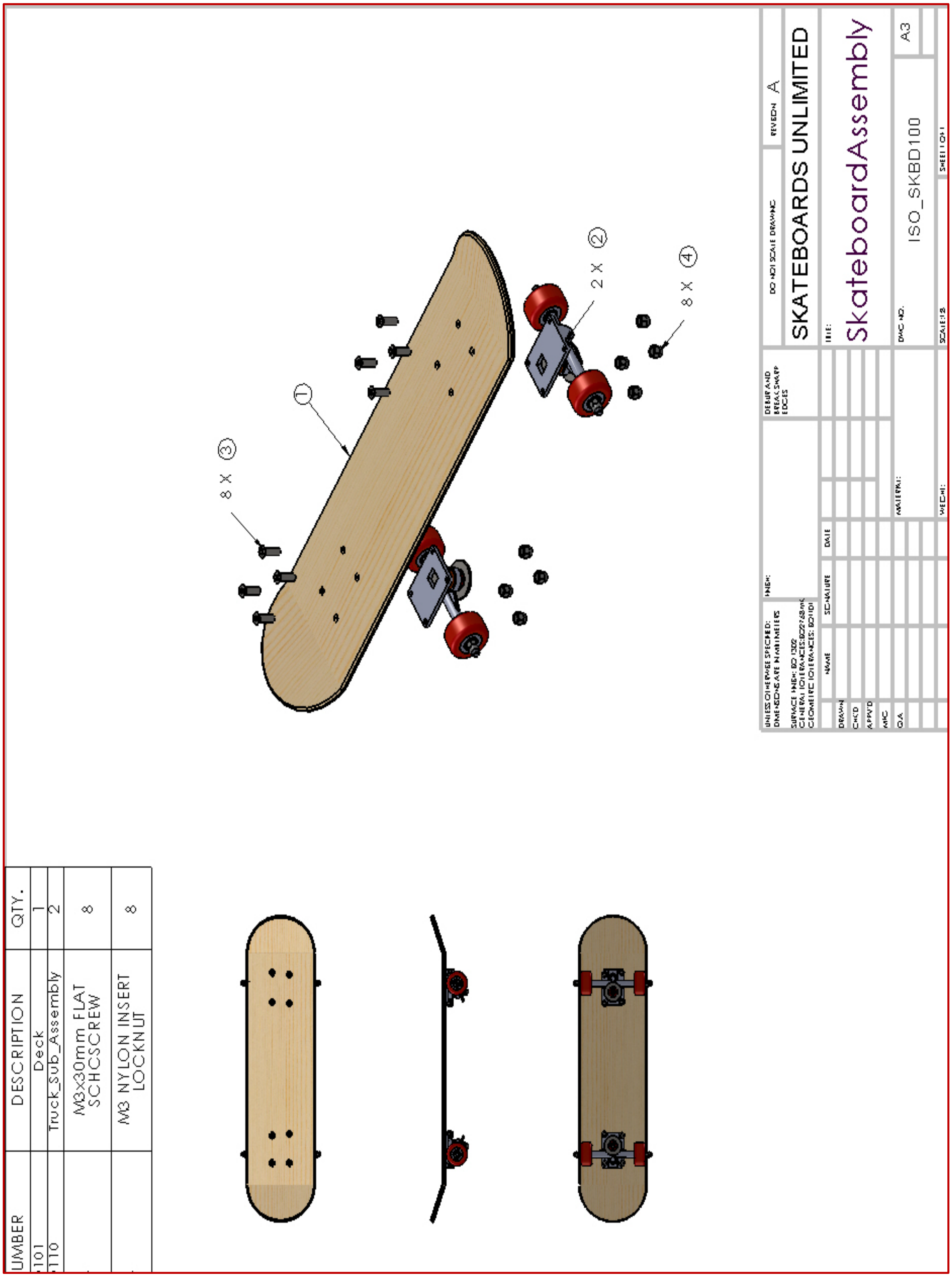

Skateboards Unlimited
TITLE:
WHEEL
SIZE
A
DWG. NO.
SKBD121
REV
A
SCALE: 1:1
WEIGHT:
SHEET 1 OF 1
URETHANE
SECTION A-A
SECTION B-B
DETAIL D
SCALE 2 : 1
12X1.00
1.25
2X.50
.25
.15

A10 – TIRE

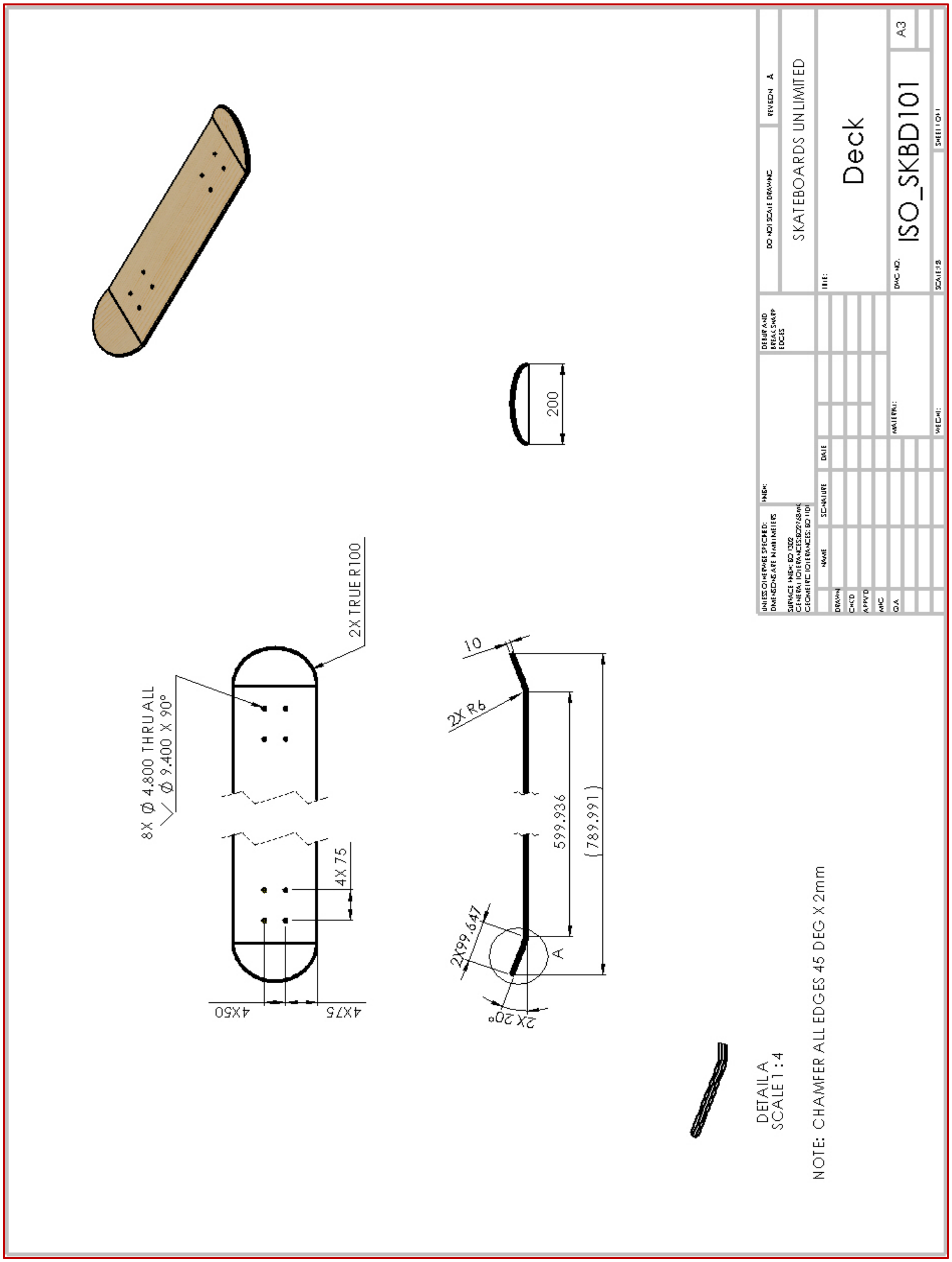

Skateboards Unlimited
TITLE:
TIRE
SIZE
A
DWG. NO.
SKBD122
REV
A
SCALE: 1:1
WEIGHT:
SHEET 1 OF 1
SHORE HARDNESS
95-97A
URETHANE
RED
DO NOT SCALE DRAWING
1.25
12X1.00
2X R.25
Ø1.38
Ø2.38
SECCIÓN A-A
DETAIL A
SCALE 2 : 1
12X
.25
.13

Appendix B – Skateboard Working Drawings – ISO/millimeters

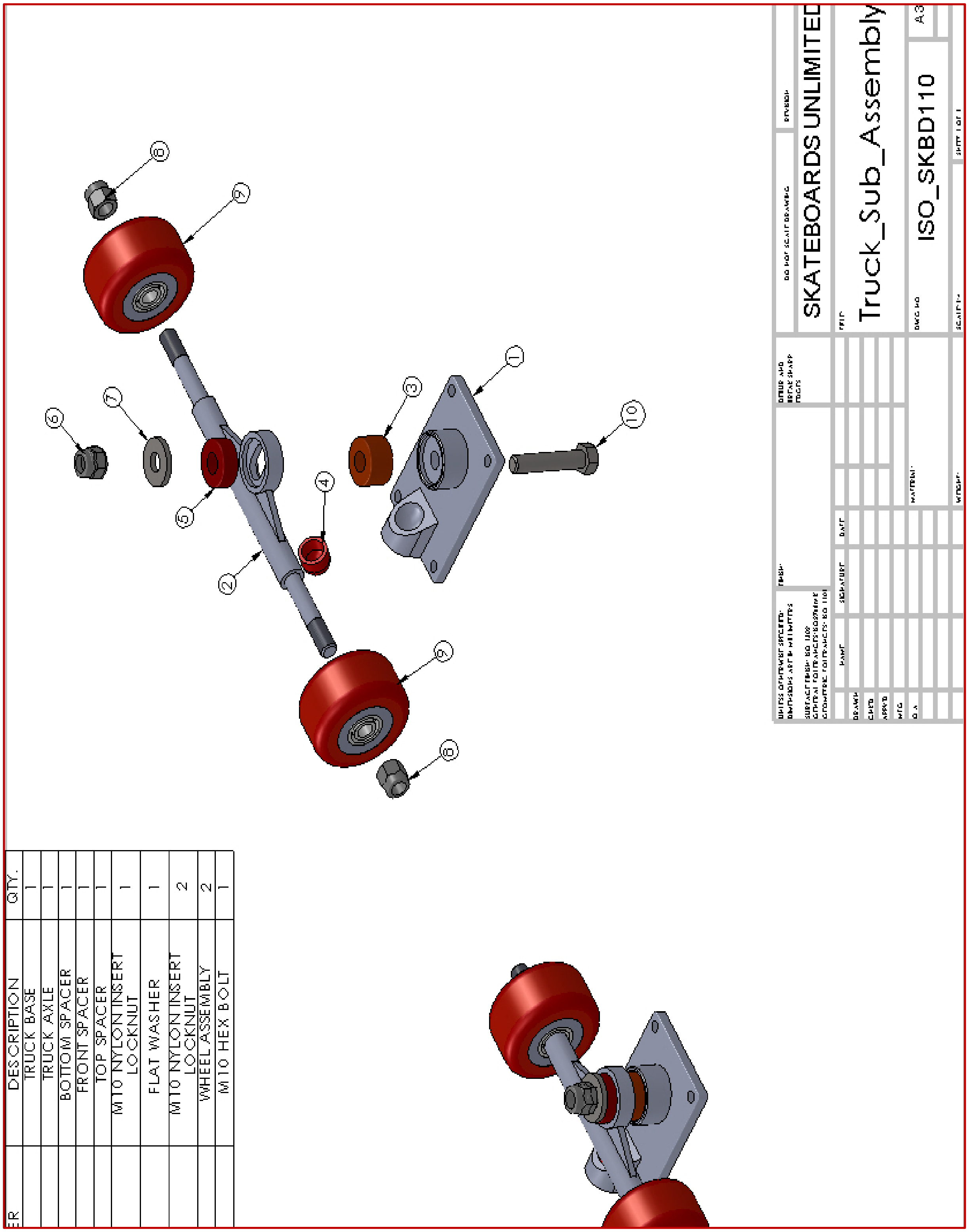
B1 – SKATEBOARD ASSEMBLY



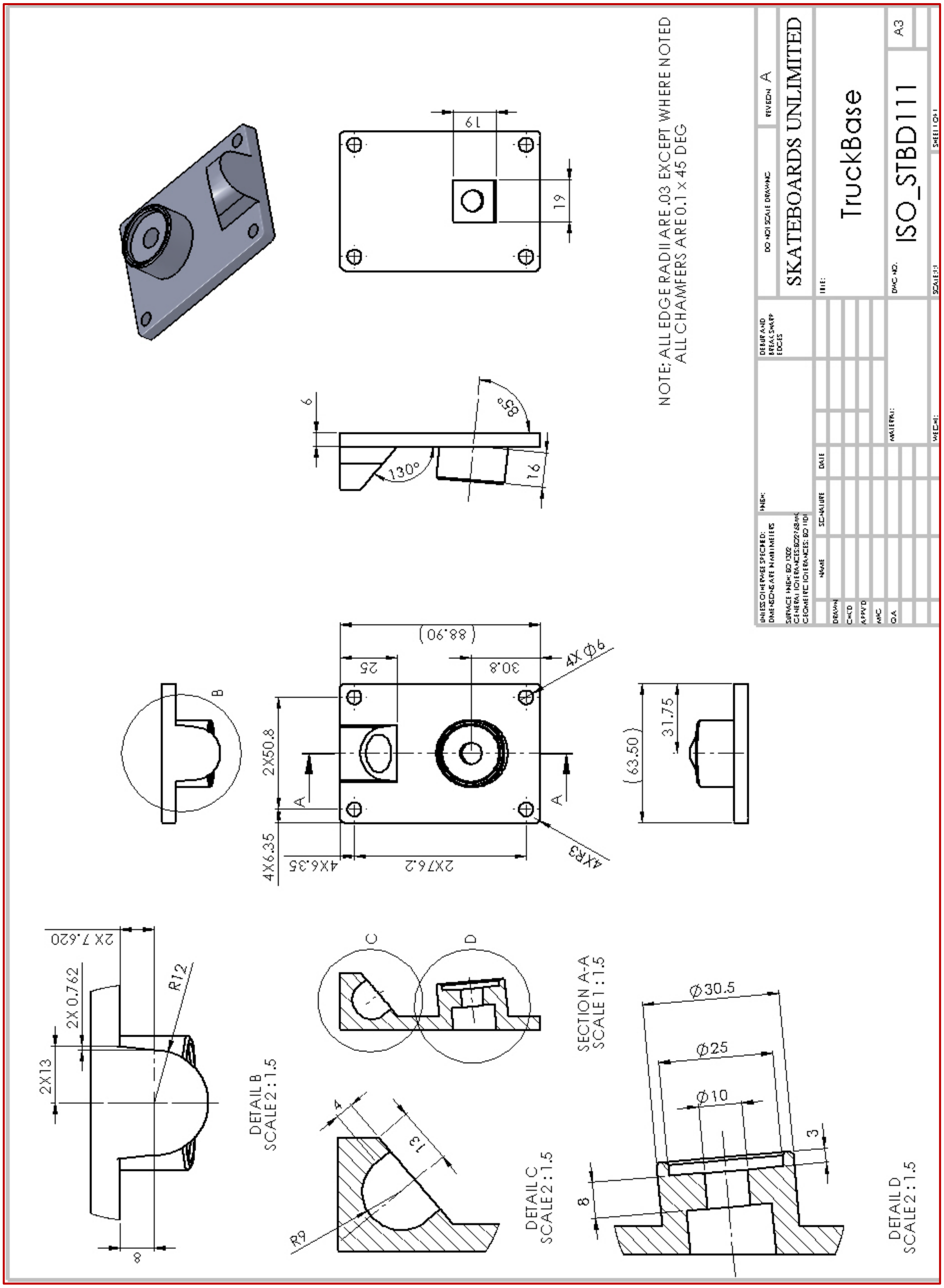
B2 – DECK


SKATEBOARDS UNLIMITED
Deck
ISO_SKBD101
A3
8X Ø 4.800 THRU ALL
Ø 9.400 X 90°
2X TRUE R100
4X75
4X50
4X75
200
10
2X R6
599.936
(789.991)
2X99.647
2X 20°
A
DETAIL A
SCALE 1 : 4
NOTE: CHAMFER ALL EDGES 45 DEG X 2mm

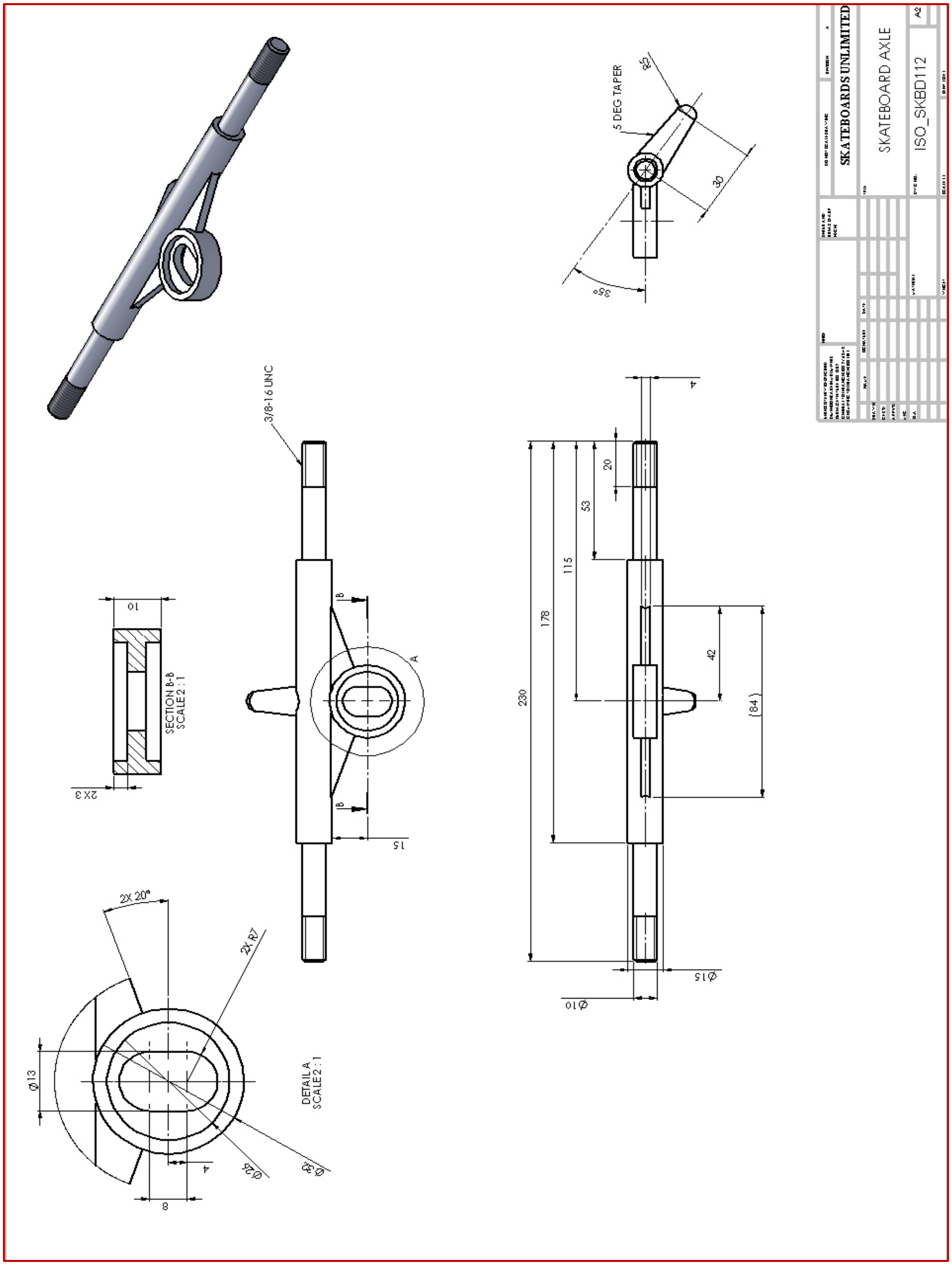
B3 – TRUCK SUB-ASSEMBLY



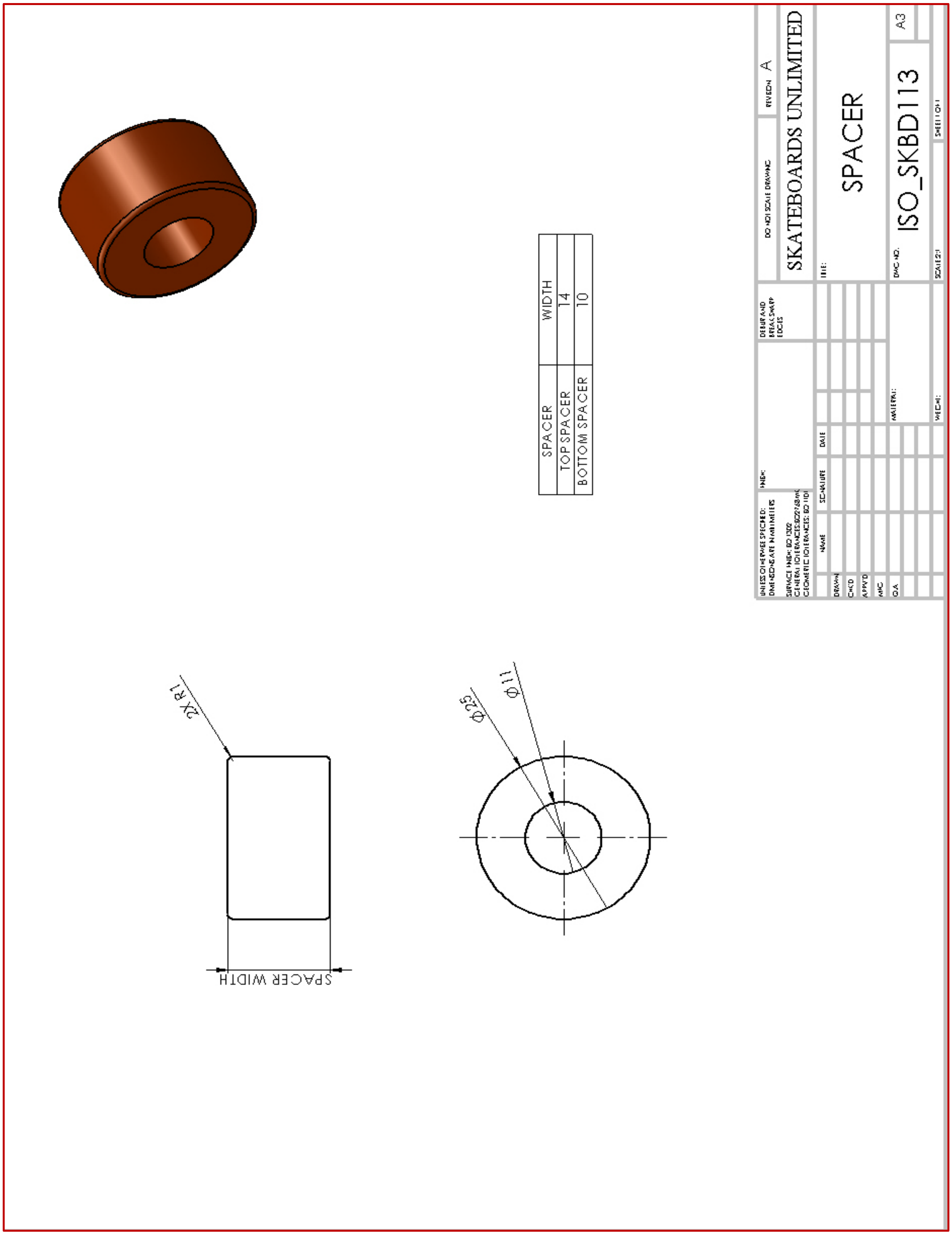
B4 – TRUCK BASE



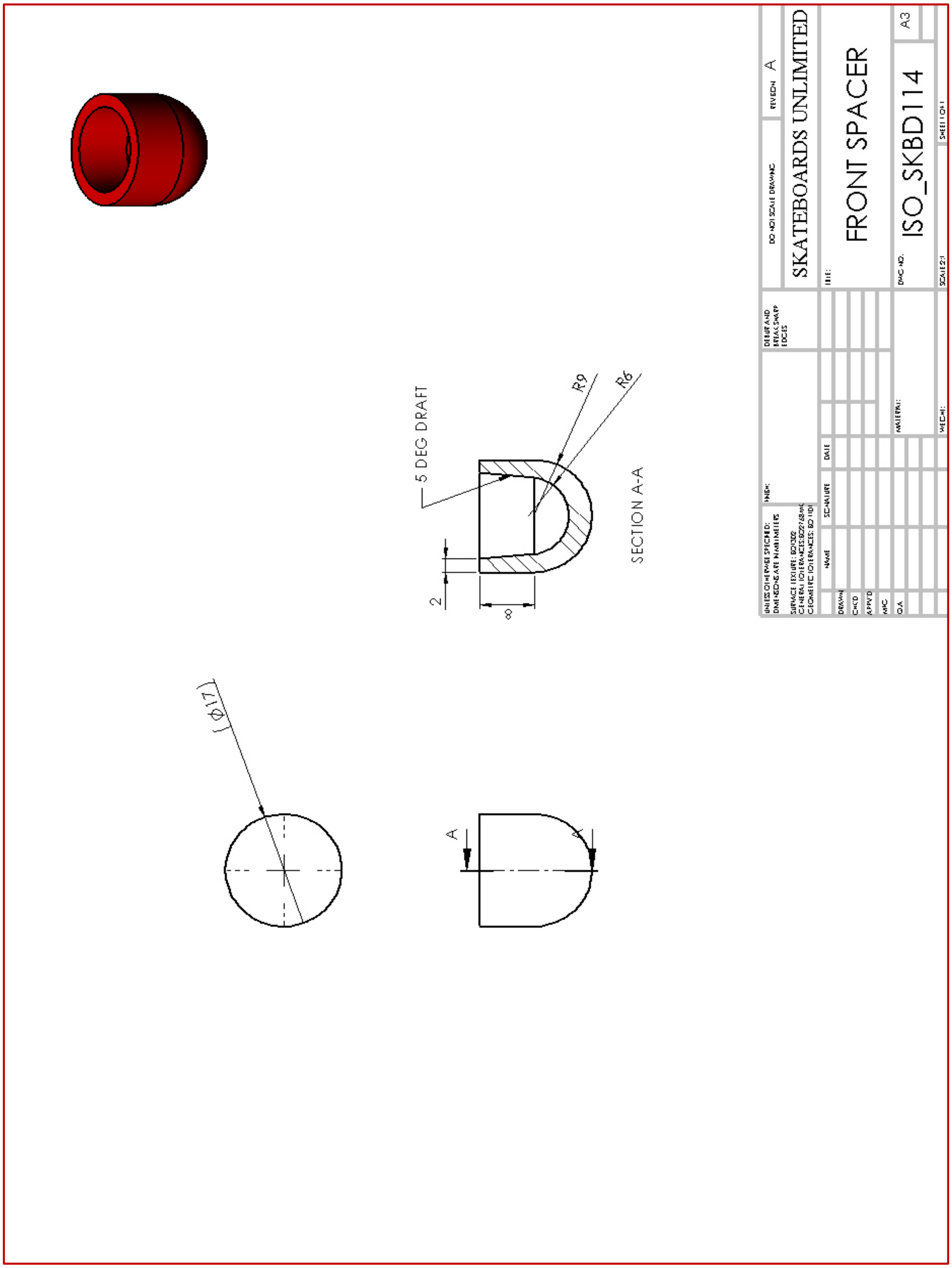
B5 – SKATEBOARD AXLE



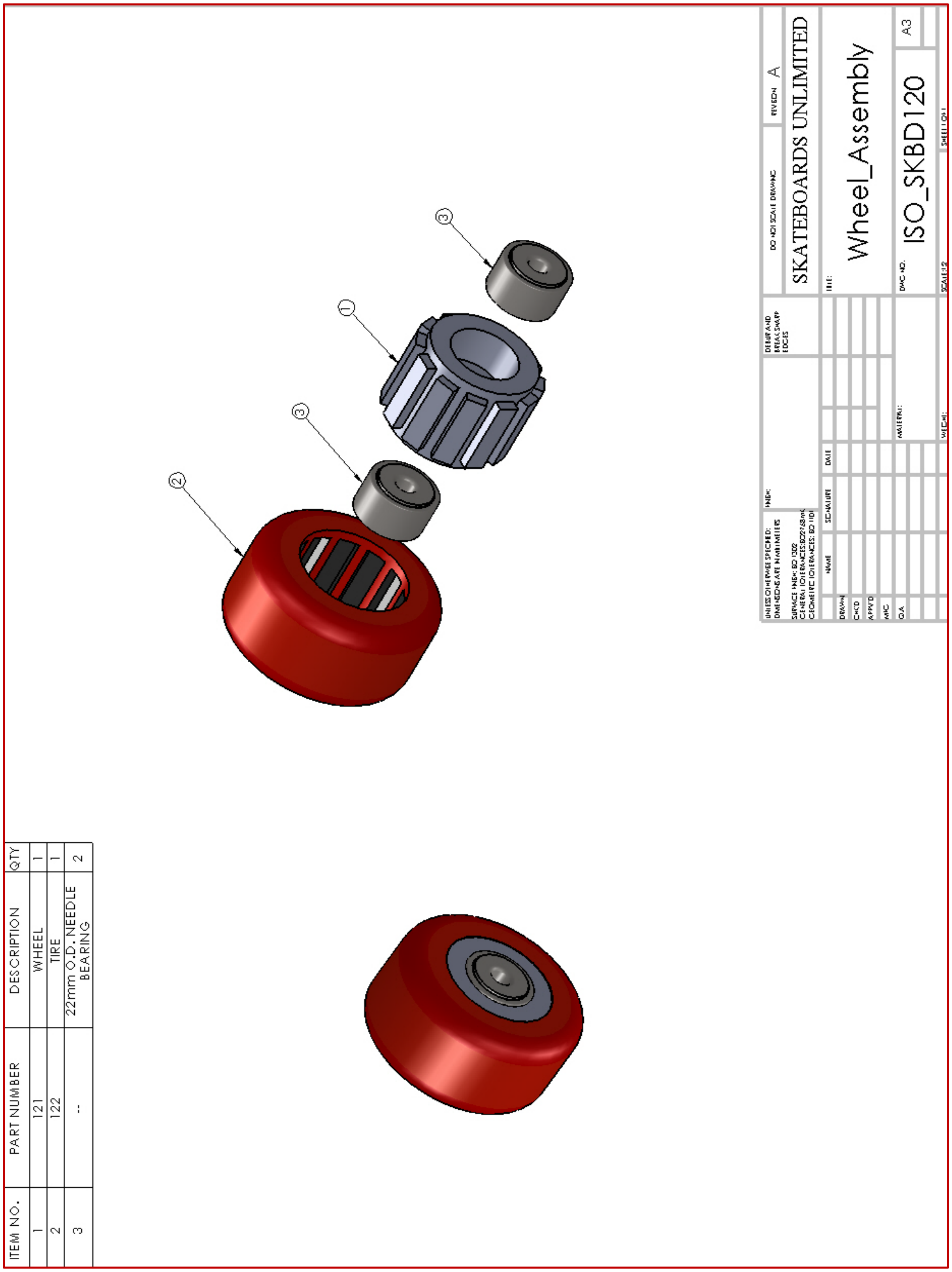
B6 – SPACER


SPACER
WIDTH
TOP SPACER
14
BOTTOM SPACER
10
DO NOT SCALE DRAWING
REVISION
A
SKATEBOARDS UNLIMITED
SPACER
ISO_SKBD113
A3
2X R1
Ø25
Ø11
SPACER WIDTH

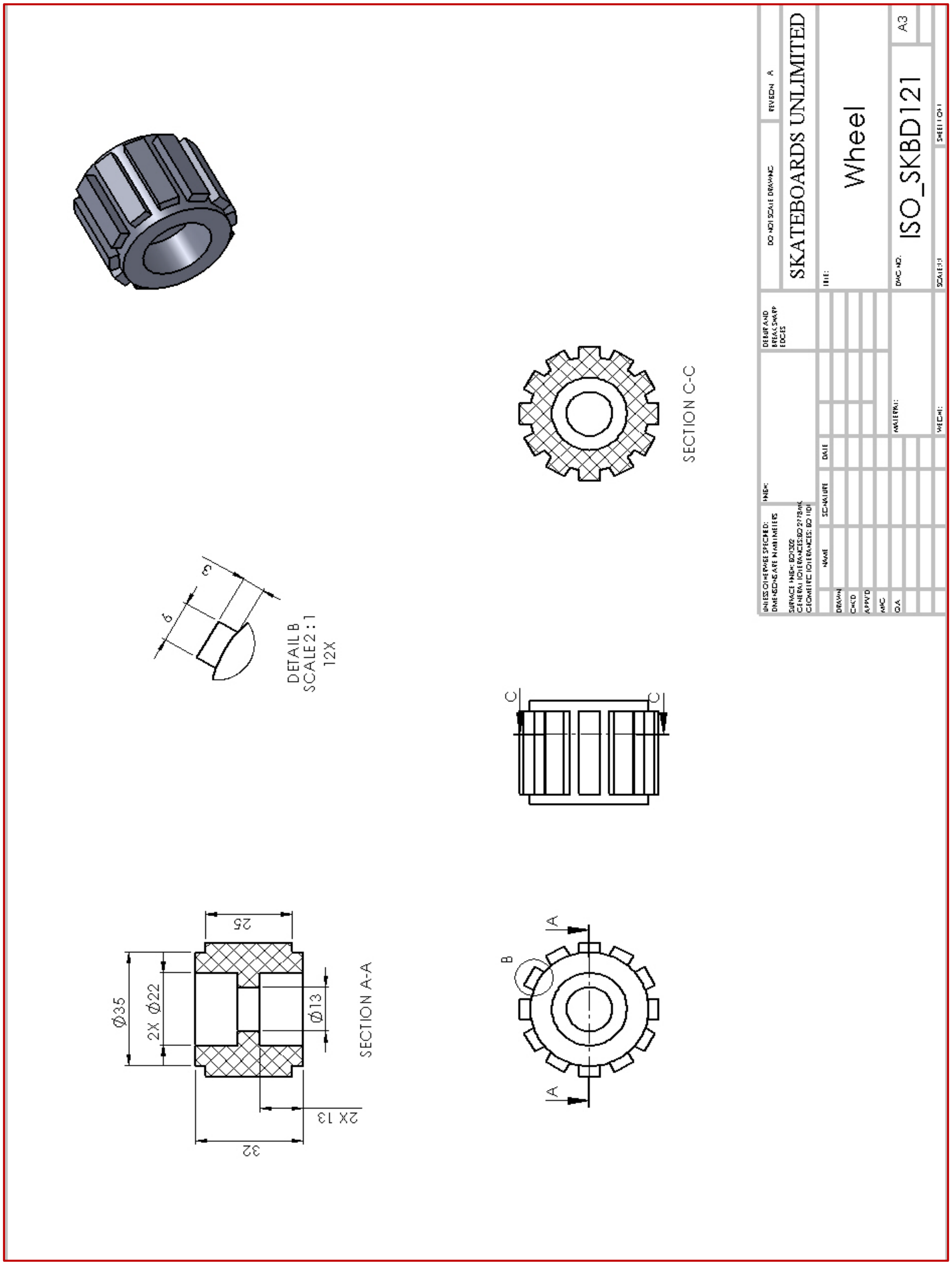
B7 – FRONT SPACER



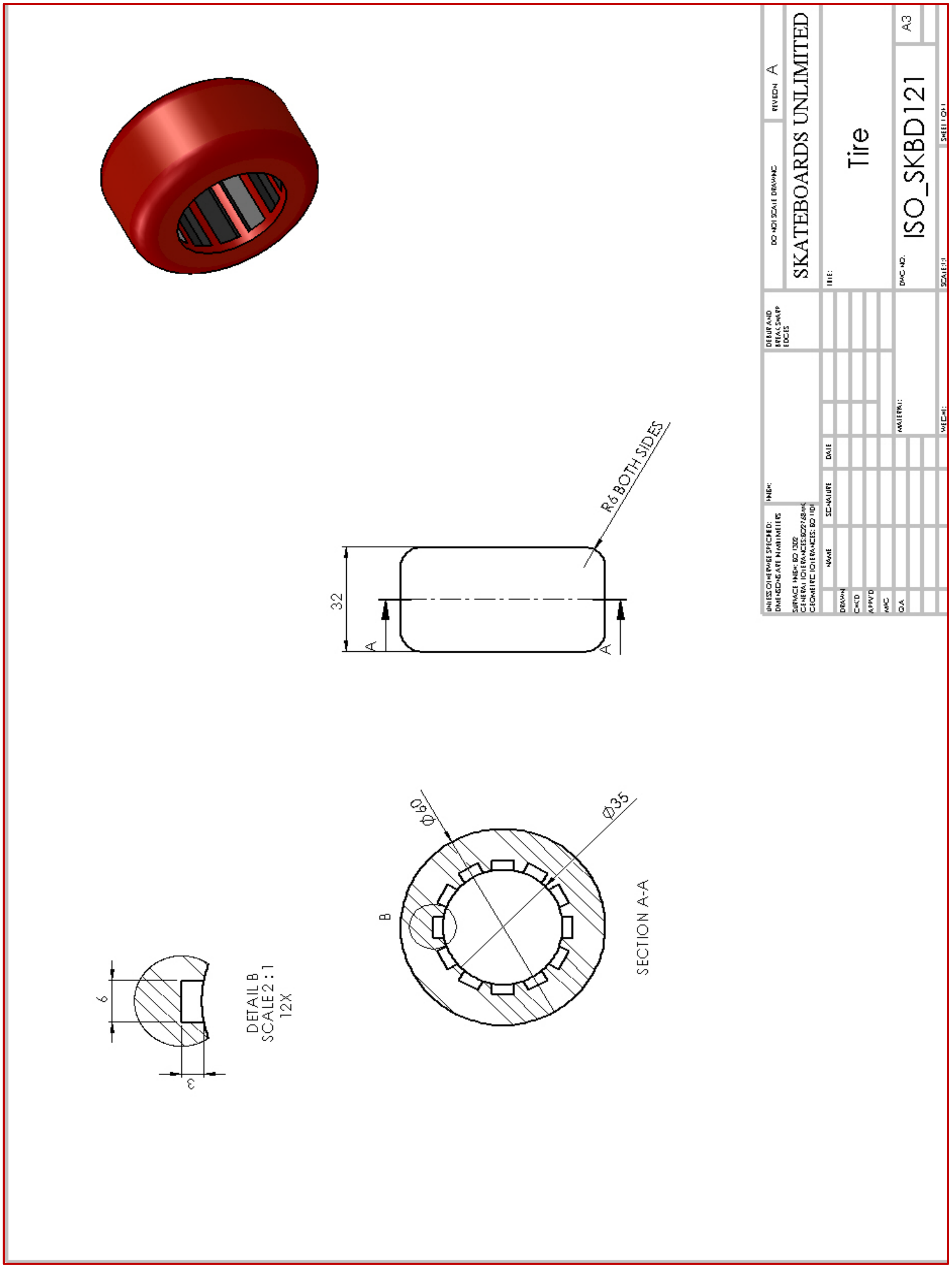
B8 – WHEEL ASSEMBLY



B9 – WHEEL


SKATEBOARDS UNLIMITED
Wheel
ISO_SKBD121
A3
SECTION C-C
DETAIL B
SCALE 2 : 1
12X
SECTION A-A
Ø35
2X Ø22
Ø13
25
32
2X 13

B10 – TIRE


SKATEBOARDS UNLIMITED
Tire
ISO_SKBD121
A3
R6 BOTH SIDES
32
Ø60
Ø35
B
SECTION A-A
DETAIL B
SCALE 2 : 1
12X
6
3

Appendix C – SolidWorks Task Pane

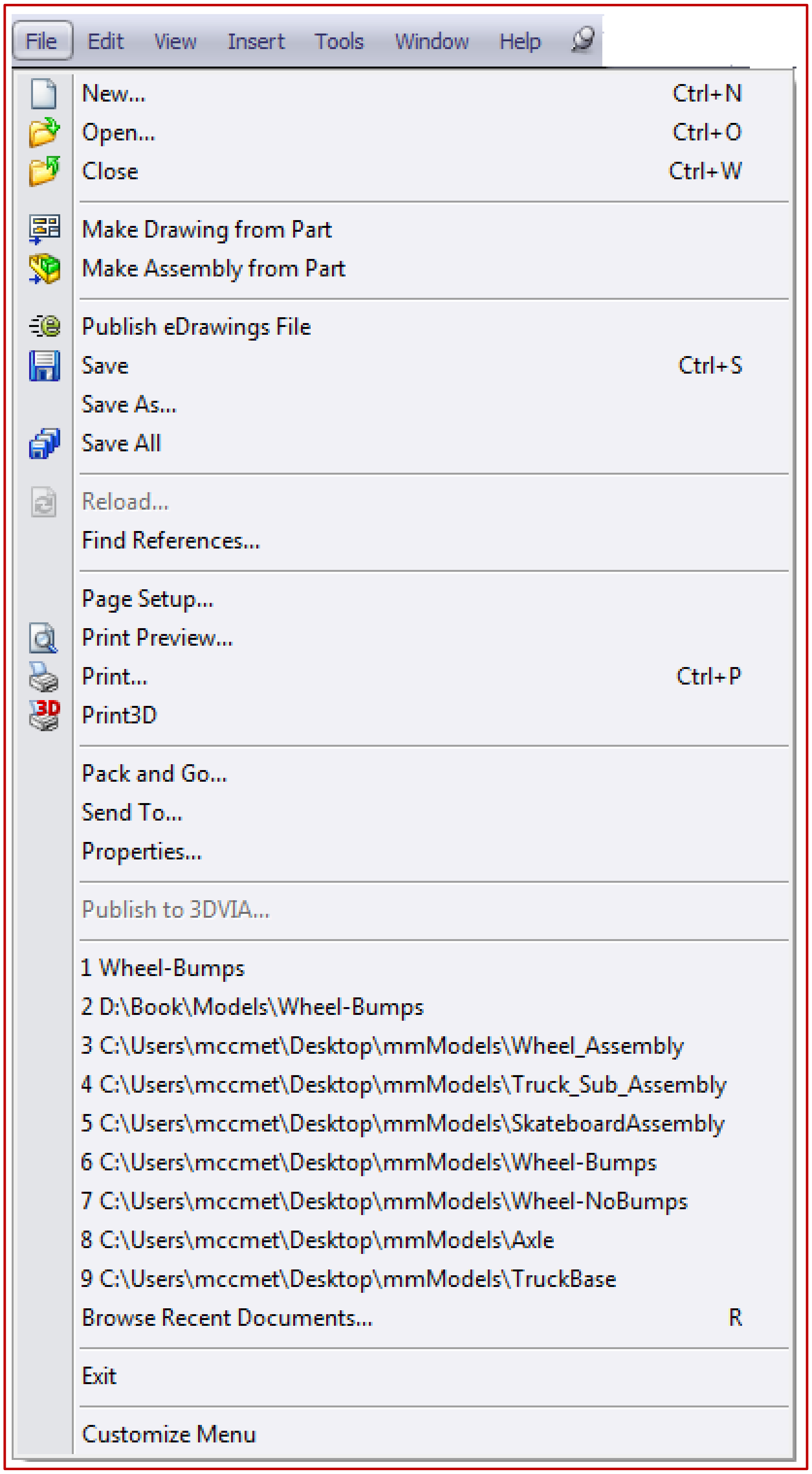
C1 – Tabs in the ***Task Pane*** (From: SolidWorks Help)

Figure C1 shows the various tabs in the ***Task Pane*** (See Figure 1.4) and the information and resources it contains. Clicking any of the tabs will expand the ***Task Pane*** and clicking the graphics area will close it. Clicking the push-pin will keep the ***Task Pane*** open.

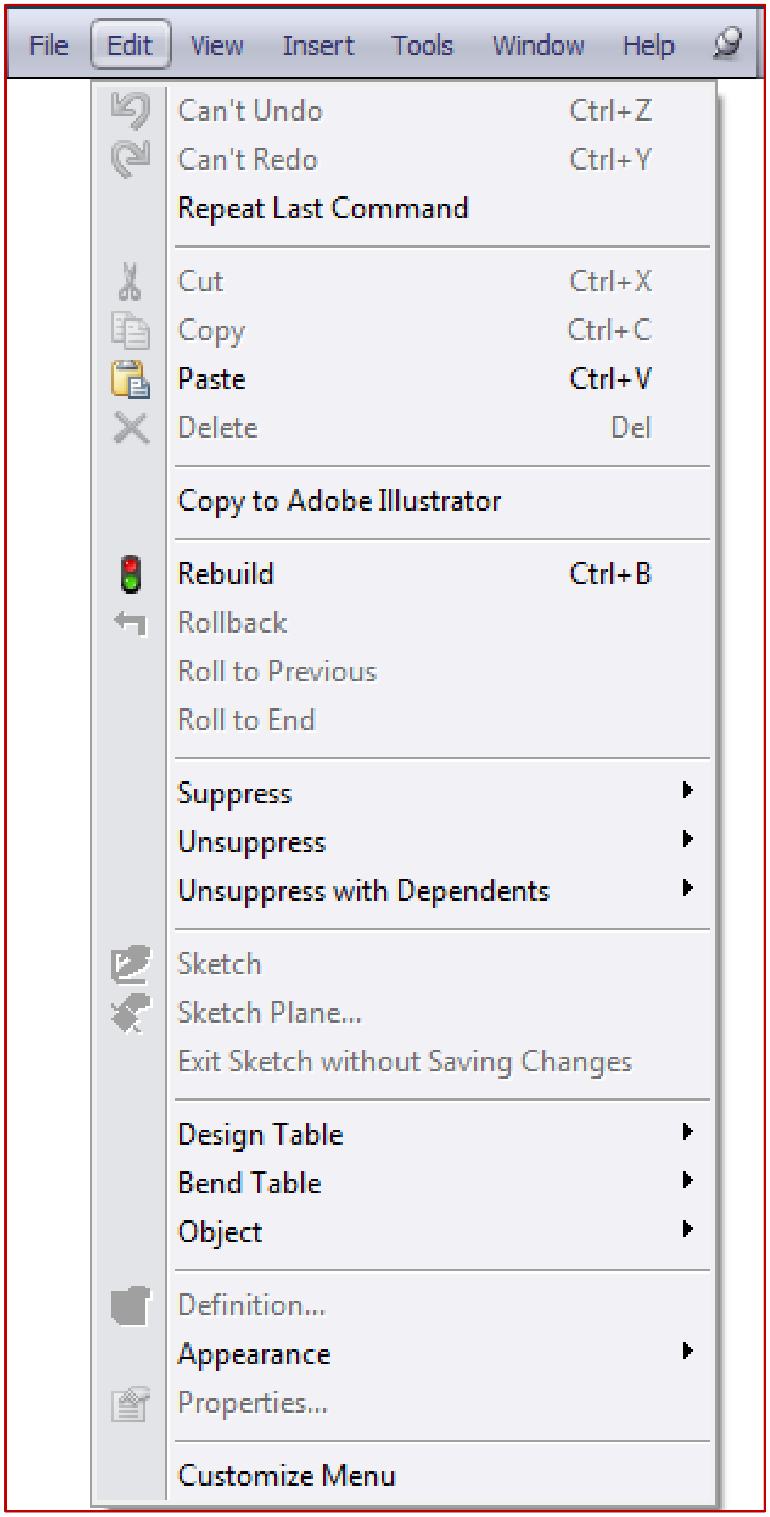
SolidWorks Resources	Groups of commands for **Getting Started**, **Community**, and **Online Resources**, plus **Tip of the Day**.
Design Library	Reusable parts, **assemblies**, and **other elements**, including Library Features.
File Explorer	Duplicate of Windows Explorer on your computer, plus **Recent Documents** and **Open in SolidWorks**. If SolidWorks Workgroup PDM is added in, the tab changes to .
Search	Results of search operations.
View Palette	Images of standard views, annotation views, section views, and flat patterns (sheet metal parts) to drag onto a drawing sheet.
Document Recovery	If auto-recovery is enabled in **Tools** > **Options** > **System Options** > **Backup/Recover** and the system terminates unexpectedly, recovered files appear on this tab the next time you start the application.
Appearances, Scenes, and Decals	Library of appearances, scenes, and decals.
Custom Properties	Enter custom properties in SolidWorks files.
SolidWorks Forum	Browse the SolidWorks Discussion Forum from directly within the Task Pane.

Appendix D – SolidWorks Main Pull-down Menu Commands

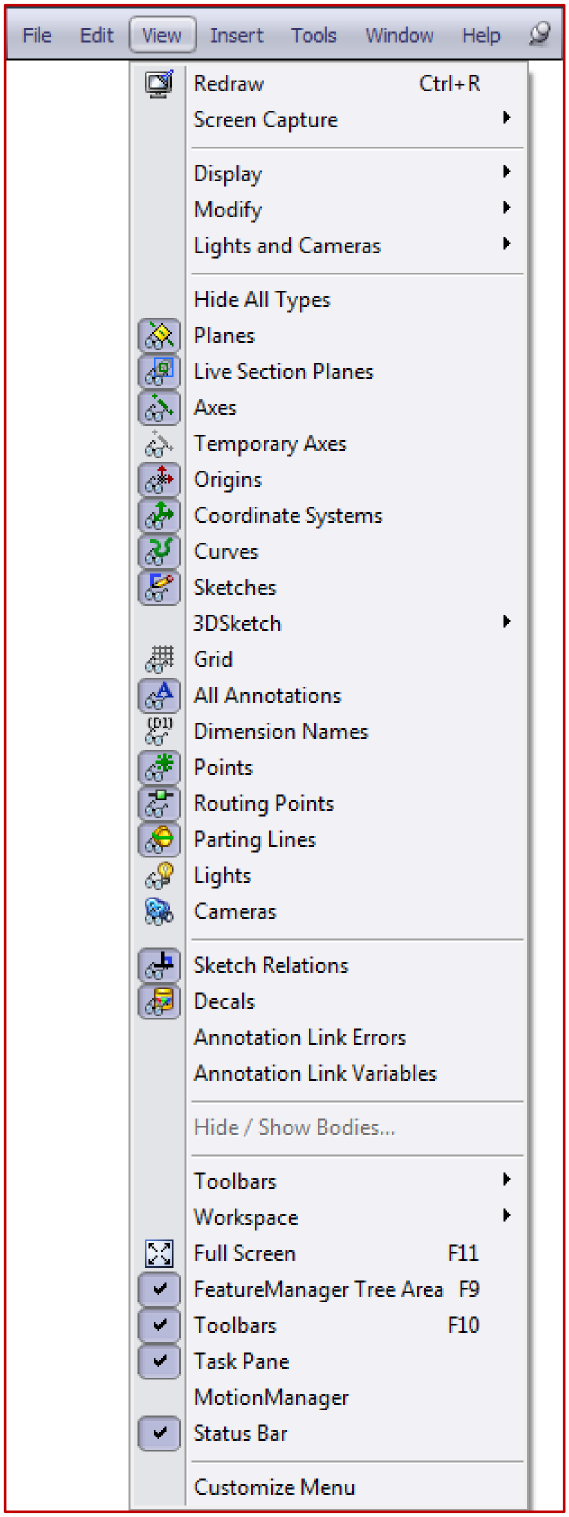
D1 – File Sub-Menu



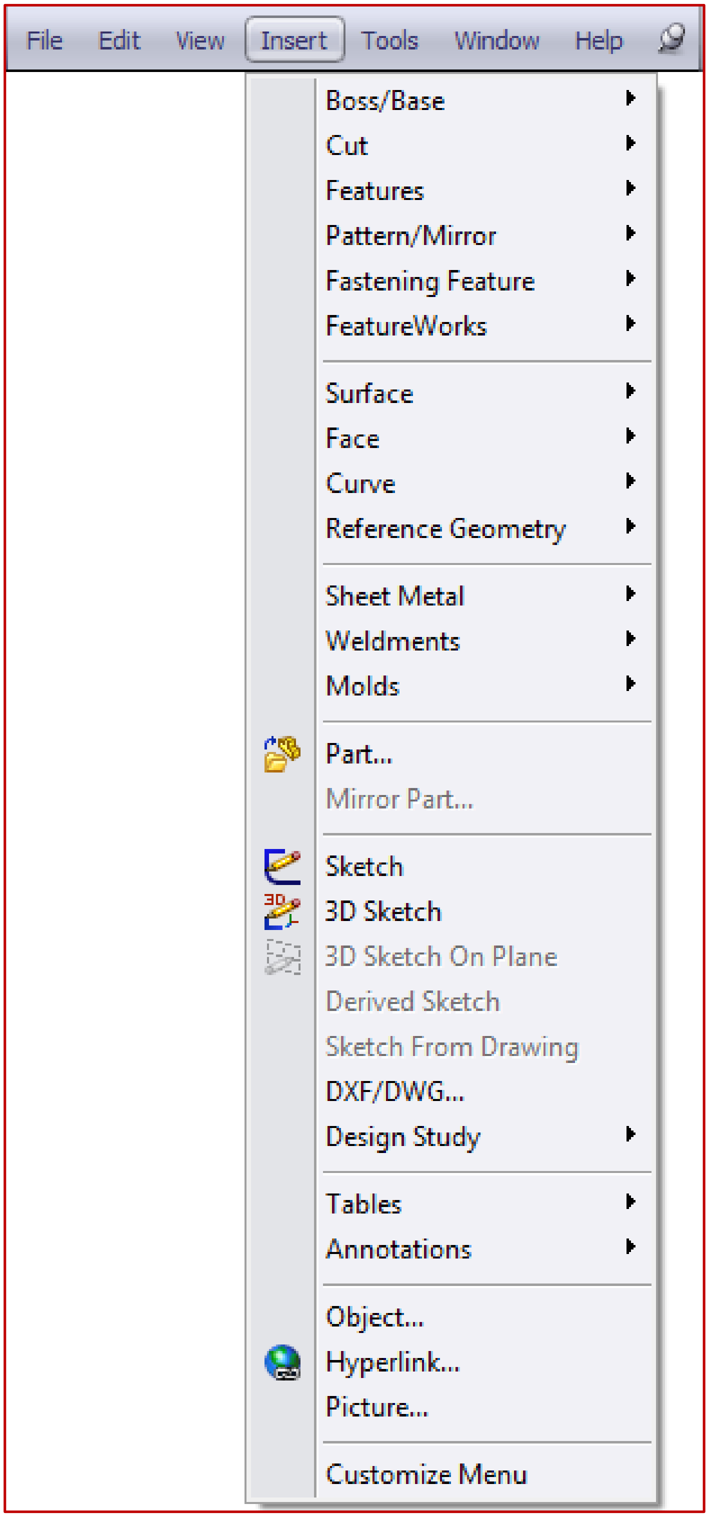
D2 – Edit Sub-Menu



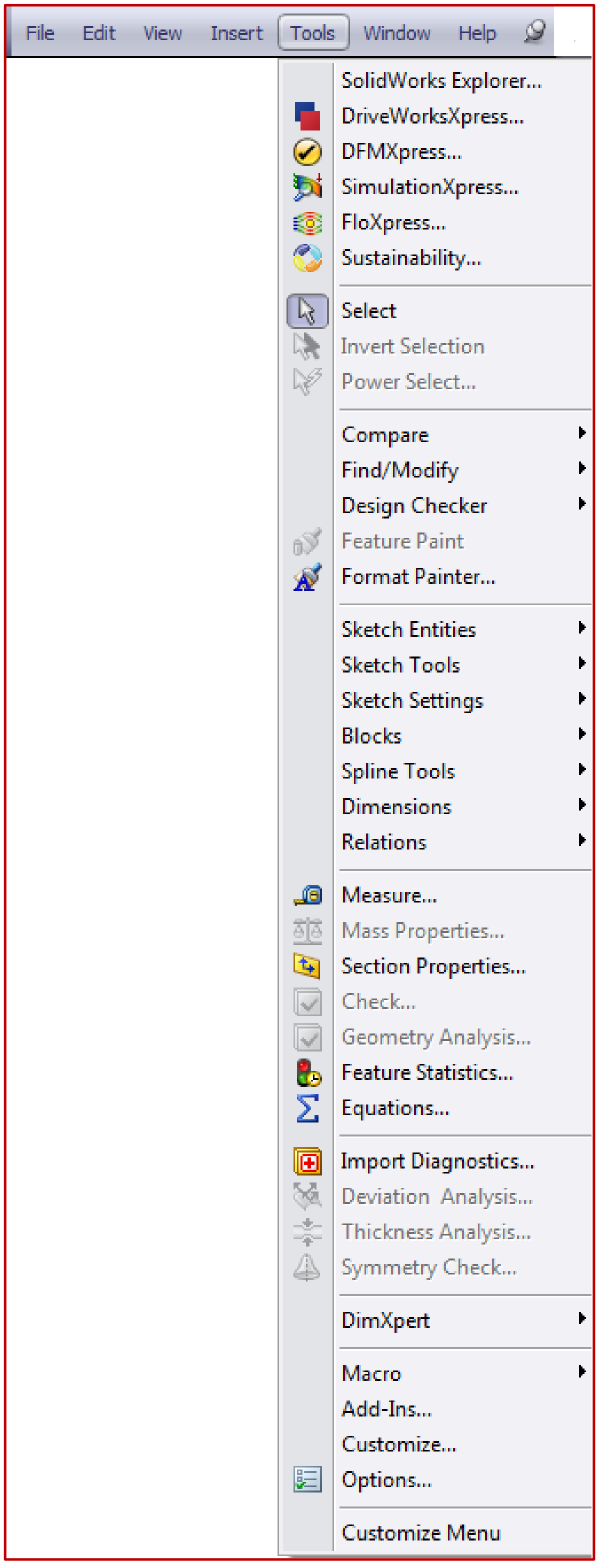
D3 – View Sub-Menu



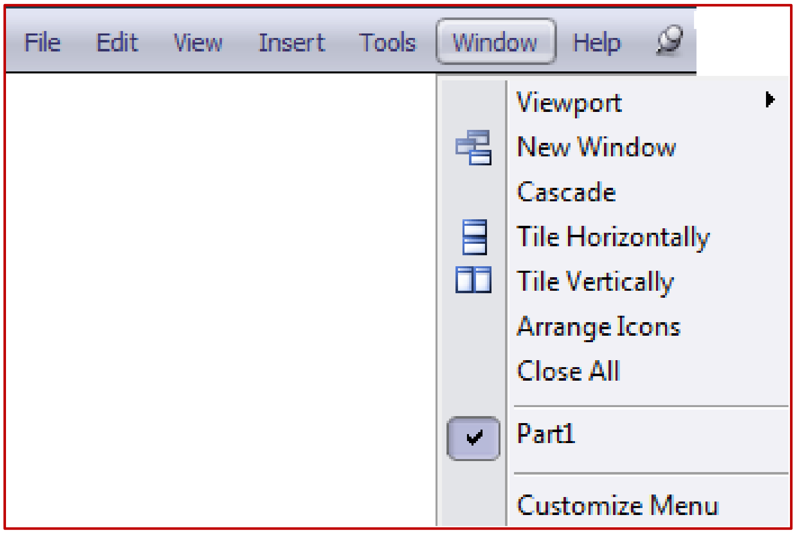
D4 – Insert Sub-Menu



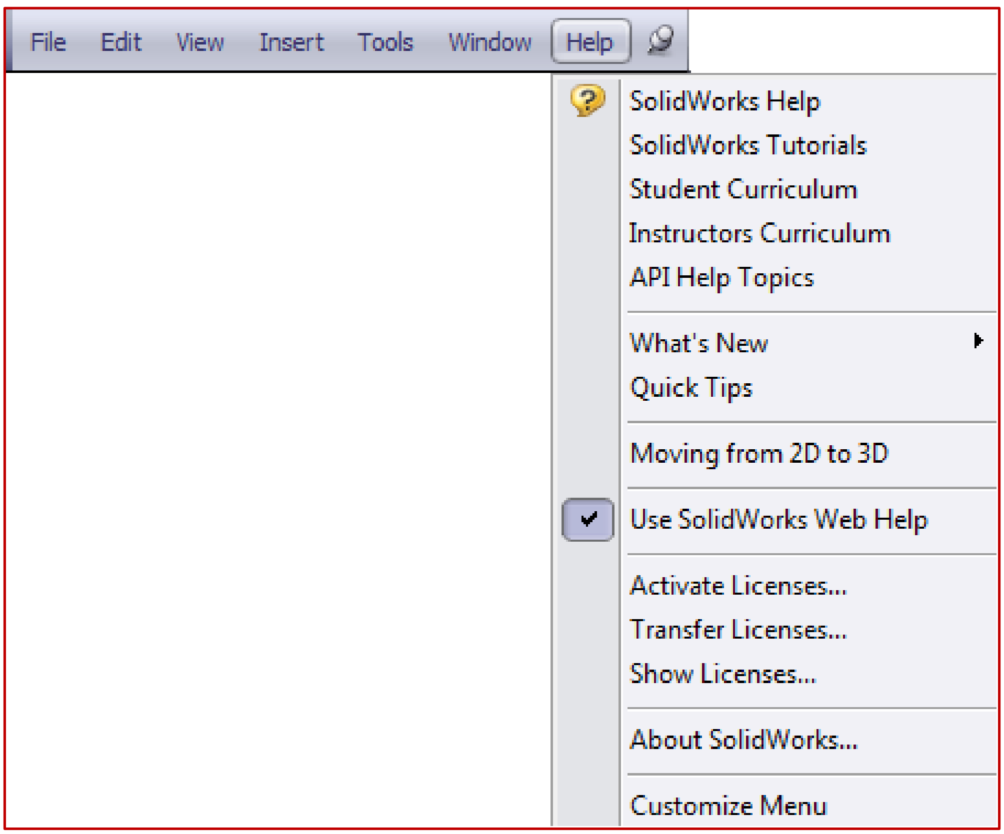
D5 – Tools Sub-Menu



D6 – Window Sub-Menu

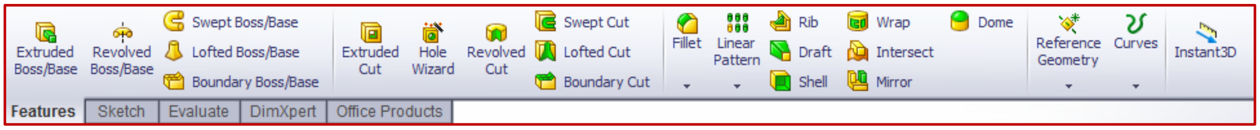


D7 – Help Sub-Menu

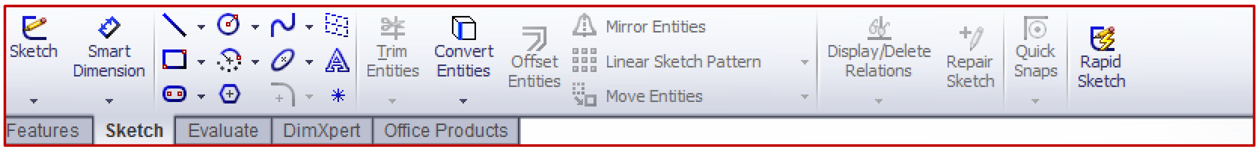


Appendix E – SolidWorks CommandManager

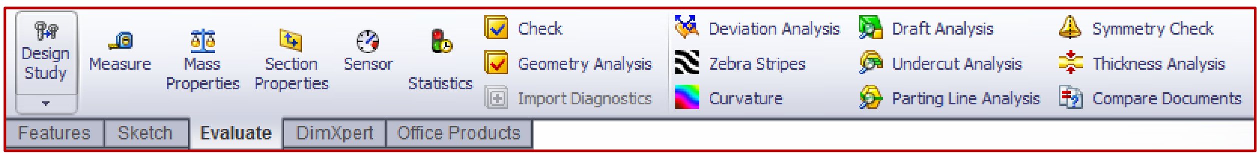
E1 – Features



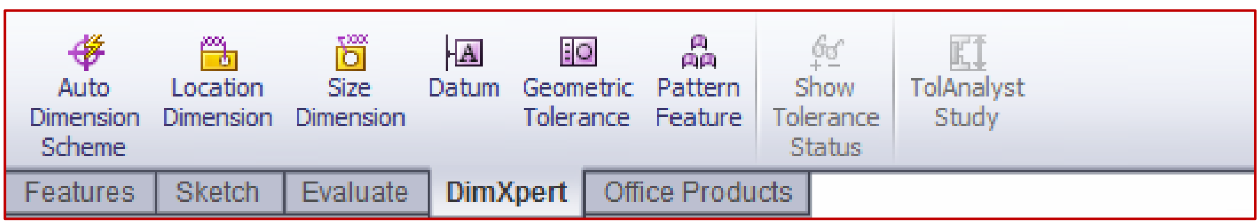
E2 – Sketch



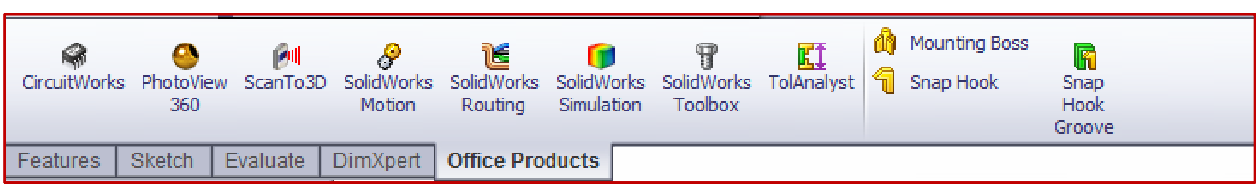
E3 – Evaluate



E4 – DimXpert

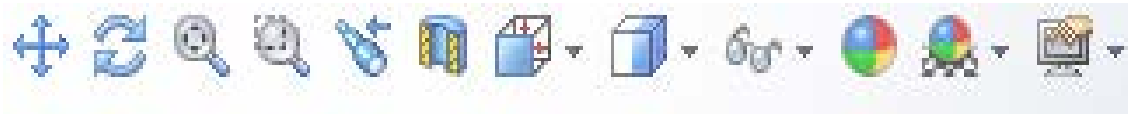


E5 – Office Products



Appendix F – View (Heads-Up) Toolbar

F1 – View (Heads-Up) Toolbar

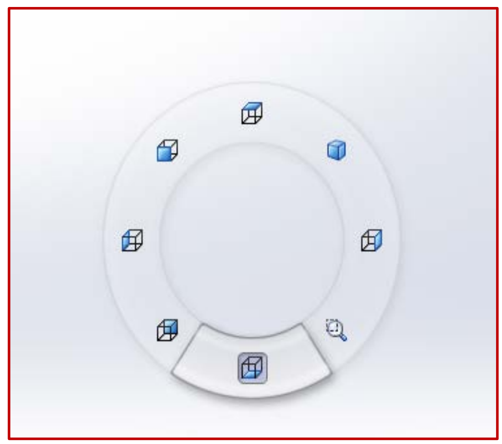


Appendix G – Mouse Gestures

G1 – Mouse Gestures

Mouse gestures increase your productivity by presenting quick access common SolidWorks commands. To enable Mouse Gestures. To enable/disable mouse gestures, click ***Tools→Customize*** in the **Quick Access Toolbar*.*** On the ***Mouse Gestures*** tab, select ***Enable/Disable*** and the number of gestures you want. ***Mouse Gestures*** are enabled by default during installation.

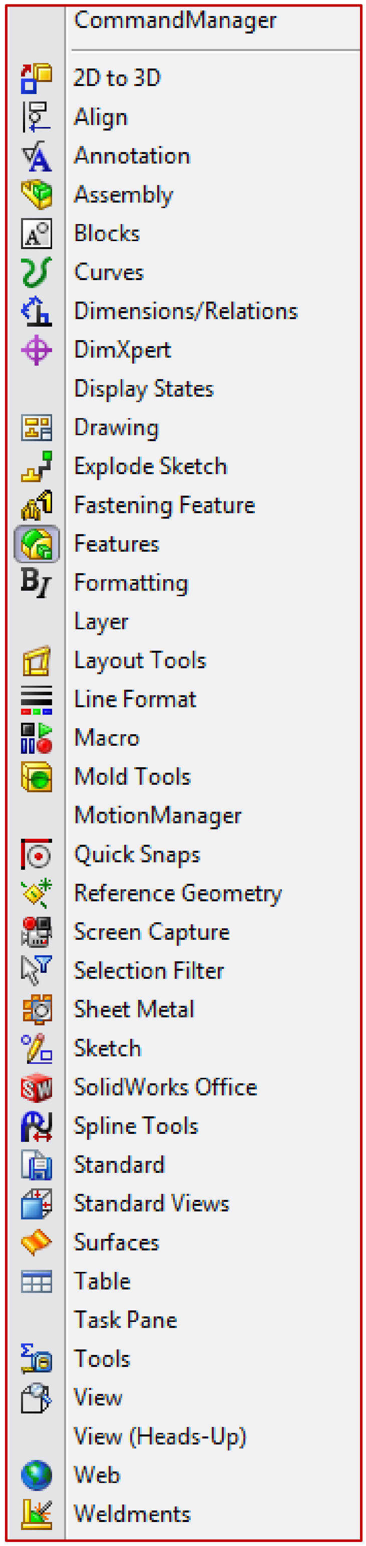
To use Mouse Gestures, right-drag your mouse by pressing the right button and dragging. The drag direction will highlight one of the tools. To select another tool continue pressing and drag in the direction you want.



Appendix H – List of SolidWorks Toolbars

H1 – List of SolidWorks Toolbars

You can display the list of available Toolbars by right-clicking anywhere in the ***CommandManager***. You can also display any of the Toolbars by clicking and highlighting them (see the icon ***Features*** below).



Appendix I – SolidWorks Keyboard Commands

I1 – SolidWorks Keyboard Commands

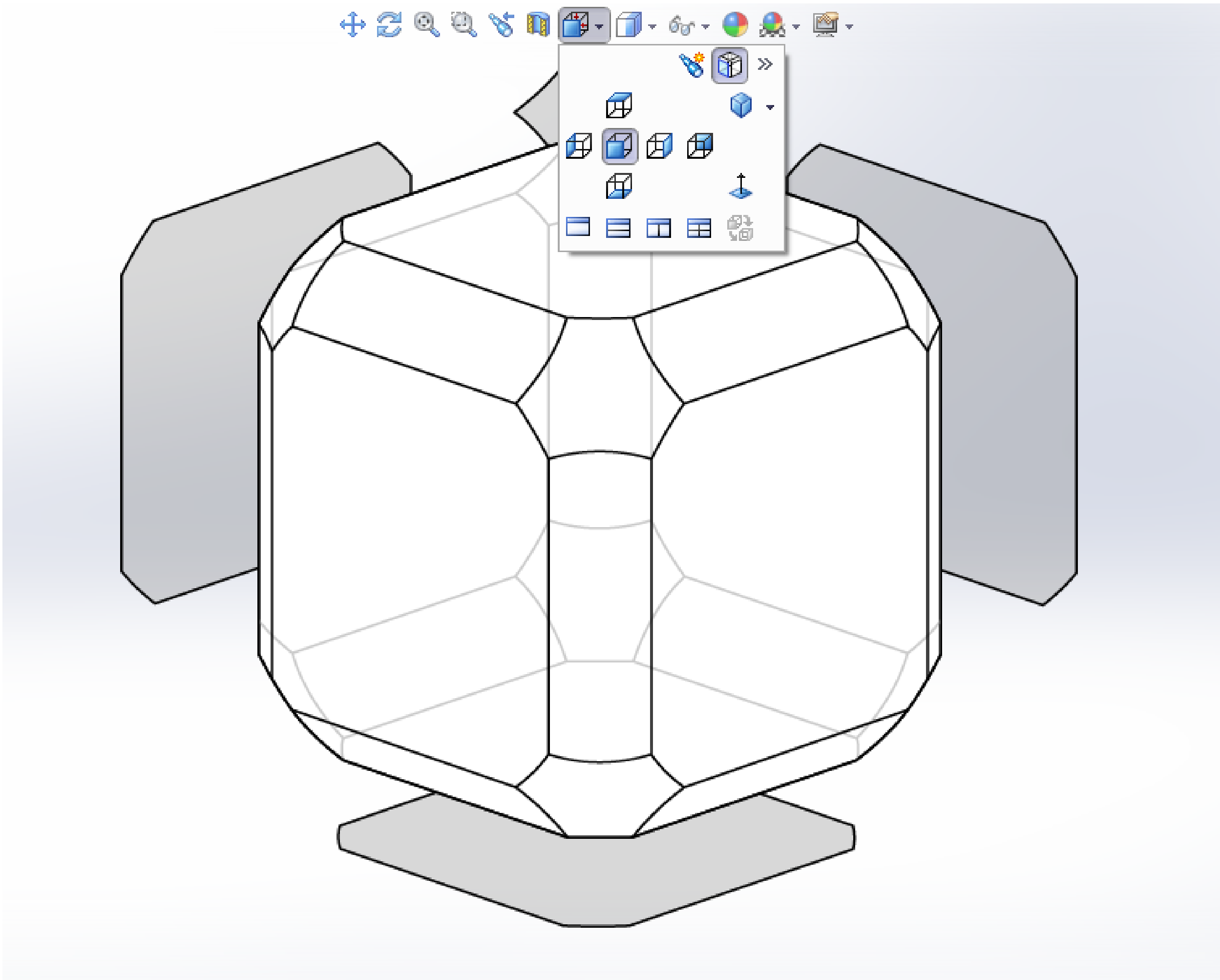
Keyboard Commands are shortcuts you can use to instead of the icons in the Toolbars.

Category	Command	Shortcut(s)
File	New..	Ctrl+N
File	Open..	Ctrl+O
File	Close..	Ctrl+W
File	Save..	Ctrl+S
File	Print..	Ctrl+P
File	Browse Recent Documents..	R
Edit	Undo..	Ctrl+Z
Edit	Redo..	Ctrl+Y
Edit	Repeat Last Command..	Enter
Edit	Cut..	Ctrl+X
Edit	Copy..	Ctrl+C
Edit	Paste..	Ctrl+V
Edit	Delete..	Delete
Edit	Rebuild..	Ctrl+B
View	Redraw..	Ctrl+R
View	Orientation..	SpaceBar
View	Zoom to Fit..	F
View	Quick Snaps..	F3
View	Full Screen..	F11
View	FeatureManager Tree Area..	F9
View	Toolbars..	F10
View	Task Pane..	Ctrl+F1
Tools	Line..	L
Others	Front	Ctrl+1
Others	Back	Ctrl+2
Others	Left	Ctrl+3
Others	Right	Ctrl+4
Others	Top	Ctrl+5
Others	Bottom	Ctrl+6
Others	Isometric	Ctrl+7
Others	Normal To	Ctrl+8
Others	Command option toggle	A
Others	Expand/Collapse Tree	C
Others	Collapse all Items.	Shift+C
Others	Filter Edges	E
Others	Find/Replace	Ctrl+F
Others	Next Edge	N
Others	Force Regen	Ctrl+Q
Others	Magnifying Glass	G
Others	Shortcut Bar	S
Others	Filter Vertices	V
Others	Toggle Selection Filter Toolbar	F5
Others	Toggle Selection Filters	F6
Others	Spell Checker	F7
Others	Filter Faces	X
Others	Accept Edge	Y
Others	Zoom Out	Z
Others	Zoom In	Shift+Z
Others	Previous View	Ctrl+Shift+Z

Appendix J – SolidWorks View Selector cube

J1 – SolidWorks View Selector cube

You can use the View Selector cube to choose the model view (i.e. the angle from which you see your model or part). On the View (Heads-up) Toolbar, click ***Orientation*** and select the View Selector cube (see the Figure below). Your model will be shown with ***Parts*** in-context, meaning that you will be able to see and choose the view that best shows what you want to see.



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BIOGRAPHY

Mario H. Castro-Cedeño has over 30 years of experience as an engineer working in industry, government and academia. His duties have included mechanical and materials engineering, corrosion research, project engineering and project management. He also has occupied positions in engineering management.

In this book he teaches how to use SolidWorks as a tool for developing and presenting engineering ideas and concepts. In addition to teaching the basics of SolidWorks, he shares his experience related to how these tools are used in an engineering organization.

The book is divided into the following five sections:

1. Introducing and Customizing SolidWorks
2. Modeling Simple Parts
3. Modeling Complex Parts
4. Modeling Assemblies
5. Creating Engineering Drawings

Unique amongst introductory SolidWorks tutorials, this book teaches how to read engineering drawings in parallel with learning to use SolidWorks. This is done by re-creating the model from the drawings included.