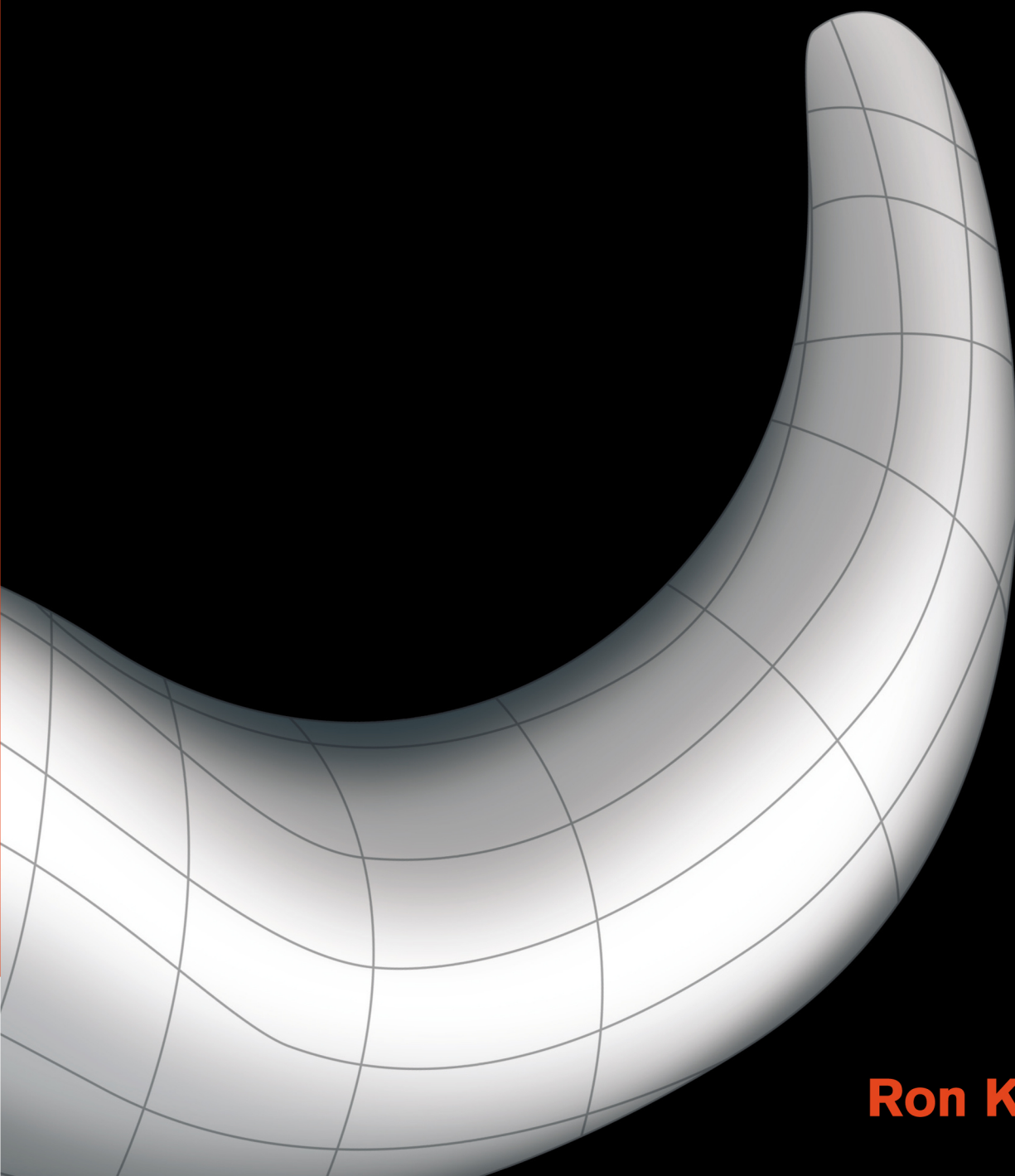


Inside RhinoCeros[®] 5



Ron K. C. Cheng

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Inside Rhinoceros[®] 5

RON K. C. CHENG



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INTRODUCTION

INTRODUCING RHINOCEROS

Rhinoceros, also known as Rhino, is a 3D surface modeling application. It has already gained popularity in the industry, including industrial design, marine design, jewelry design, CAD/CAM, rapid prototyping, reverse engineering, graphic design, and multimedia. It is based on the popular NURBS (nonuniform rational B-spline) mathematics, which enables the construction of free-form organic surfaces that are compatible with most other computer models used in the industry.

To cope with rapid prototyping and some animation and visualization applications that use faceted approximation instead of NURBS surfaces to represent 3D free-form objects, Rhino also provides a set of polygon mesh tools.

Making NURBS surfaces usually requires a set of curve frameworks. Therefore, Rhino has a set of curve tools. To enable modification and improvisation of surface models, Rhino incorporates a comprehensive set of editing, transformation, and analysis tools. Apart from that, it has tools for outputting 2D engineering drawings and highly photorealistic rendered images and animations.

AUDIENCE AND PREREQUISITES

This book is intended for students and practitioners in the following industry: industrial design, marine design, jewelry design, CAD/CAM, rapid prototyping, reverse engineering, graphic design, and multimedia. There is no prerequisite needed for several reasons, because the book is written in a nonlinear manner, enabling novices to learn the application in a short period of time, and allowing experienced users to discover more advanced use of Rhino in design.

PHILOSOPHY AND APPROACH

With learning-by-doing in mind, this book stresses application, using Rhinoceros as a tool to produce the digital model of a design in mind for downstream processes. In addition, this book has a balanced emphasis on modeling projects and computer modeling concepts, bridging the theoretical and software-oriented approaches to modeling in the computer, because theory is relatively useless without hands-on experience and vice versa.

To enable both novices and more advanced users to learn about Rhino in a nonlinear way, this book has two major components: a set of hands-on modeling projects and the main chapters. The set of modeling projects is placed at the beginning of the book, allowing all readers to have hands-on experience on using Rhino to construct some real-world objects. In order to provide a clear picture about what can be produced by using Rhino, the main chapters are written logically, with concepts explained together with simple tutorials focusing on individual elements.

INTRODUCTION

This book is written to Rhinoceros Release 5, addressing the new tools provided as well as tools that are already provided in previous releases.

At the beginning of this book, there is a set of four easy-to-follow, hands-on modeling projects. In the first project, guides are provided to construct the chassis and mechanical components of the famous bubble car. The second project completes the bubble car by making the free-form bodywork. The third project relates to a set of small objects to highlight the use of Rhino in various ways. Finally, in the fourth project, a way to assemble a set of components is suggested.

There are 14 main chapters. Chapter 1 provides an overview about digital modeling, explains the concepts of surface modeling, and introduces Rhinoceros as a digital modeling tool, as well as an examination of the Rhino user interface,

including its key functional components. Chapter 2 familiarizes you with the concepts of construction plane, basic geometry construction and modification methods, layer management, and surface display methods. Chapter 3 provides you with a clear understanding on the kind of surfaces that you can construct by using Rhino and the framework of points and curves required for building such surfaces. Chapters 4 through 6 delineate the ways to construct and manipulate points and curves for making surfaces. After discussing curves, in Chapter 7 you learn more detailed surface manipulation methods. Chapter 8 covers the manner in which Rhino represents solids in the computer and various ways of solid construction. In Chapter 9, you will learn the ways to construct and manipulate polygon meshes, which are significantly used for rapid prototyping.

Chapter 10 explains various methods to transform curves, surfaces, and polygon meshes. Chapter 11 deals with analyzing curves, surfaces, and polygon meshes. In Chapter 12, you learn about grouping drawing objects, construction of data blocks, and use of work sessions. Chapter 13 addresses producing 2D engineering drawings from 3D models, as well as import and export of files. Chapter 14 explains how rendered images and animations can be produced from 3D models.

SUPPLEMENTS TO THIS BOOK

Student Companion Site

A Student Companion Website is available containing Rhinoceros files for the project exercises.

Accessing a Student Companion Website from CengageBrain:

1. Go to <http://www.cengagebrain.com>
2. Enter author, title, or ISBN in the search window.
3. When you arrive at the Product page, click on the Access Now tab.
4. Click on the resources listed under Book Resources in the left navigation pane to access the project files.

Instructor Site

An Instruction Companion Website containing supplementary material is available. This site contains an Instructor Guide containing answers to the end-of-chapter review questions. Contact Cengage Learning or your local sales representative to obtain an instructor account.

Accessing an Instructor Companion Website from SSO Front Door

1. Go to <http://login.cengage.com> and login using the Instructor email address and password.
2. Enter author, title, or ISBN in the Add a title to your bookshelf search box and click on Search button.
3. Click Add to My Bookshelf to add Instructor Resources.
4. At the Product page click on the Instructor Companion site link.

New Users

If you are new to Cengage.com and do not have a password, contact your sales representative.

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ABOUT THE AUTHOR

Highly influenced and inspired by Jean Piaget's constructivism theory, Seymour Papert's constructionism theory, and Jerome Seymour Bruner's cognitive learning theory, Cheng, with master degree in education as well as master degree in engineering, designed and pioneered the integrative project learning approach in the Hong Kong Polytechnic University over a decade ago, training its undergraduate as well as postgraduate students. After serving the university for over thirty-five years at the age of sixty, leading engineering design and integrative project learning in the Industrial Center, Cheng now began his second phase of life, continuing to assert and advocate the learning approach that he pioneered in various education institutions, and working on a number of industrial automation projects as well as engineering design projects, leading to commercialization of scientific research.

Modeling Projects

We will begin with a set of modeling projects, which are intended to let you appreciate how to use Rhinoceros as a tool in computer modeling. These modeling projects are followed by the main chapters, delineating various computer modeling concepts and Rhinoceros commands in a logical way. It is hoped that you will find these modeling projects interesting and not too difficult to follow. However, if you are a novice in surface modeling or using Rhinoceros and find these modeling projects too difficult, it is suggested that you should first go to the main chapters and then come back to the modeling projects later. Figures P-1 through P-8 show the rendered views of these projects.



FIGURE P-1 *Bubble car chassis modeling project*
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FIGURE P-2 *Bubble car body modeling project*
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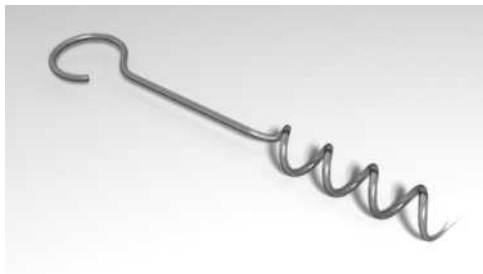


FIGURE P-3 *Small objects modeling project 1*
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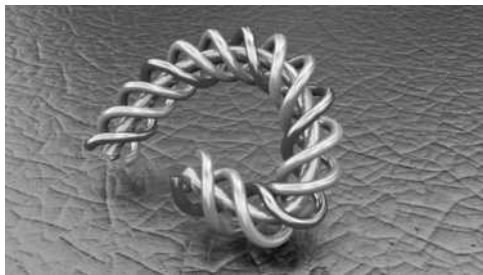


FIGURE P-4 *Small objects modeling project 2*
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FIGURE P-5 *Small objects modeling project 3*
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FIGURE P-6 *Small objects modeling project 4*
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FIGURE P-7 *Small objects modeling project 5*
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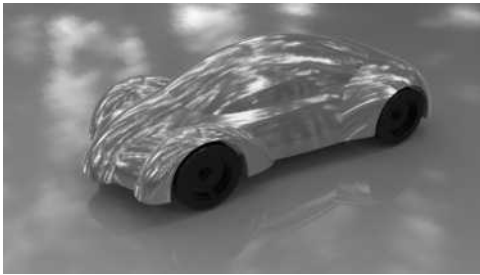


FIGURE P-8 *Toy car modeling project*
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NURBS CURVES, NURBS SURFACES, POLYGON MESHES, AND SOLIDS

As will be explained in detail in the main chapters of this book, Rhinoceros is a 3D modeling tool and we can use it to construct three major kinds of geometric objects: NURBS curves, NURBS surfaces, and polygon meshes. Among them, we use curves as framework on which surfaces are constructed. As for NURBS surfaces and polygon meshes, they are two distinct ways of composing a 3D surface model, with NURBS surfaces used for making accurate models and polygon meshes used for approximated representation of 3D objects.

In Rhinoceros terms, by joining two or more contiguous NURBS surfaces together, a polysurface is formed. If a single NURBS surface, such as a sphere or an ellipsoid, or a polysurface forms a closed loop enclosing a volume without any gap or opening, a solid is implied.

DESIGN DEVELOPMENT

Prior to working on the modeling projects, let us have a quick review on the two-stage iterative design development process.

Top-Down Thinking Process

With an idea or concept in mind, you start deconstructing the 3D object into discrete surface elements by identifying and matching various portions of the object with various types of surfaces. Unless the individual surfaces are primitive surfaces, like box, sphere, or ellipsoid, you need to think about the shapes and locations of curves and points required to build the surface, as well as the surface construction commands to be applied on the curves and points. This is the top-down thinking process.

Bottom-Up Construction Process

After this thinking process, you create the curves and points and apply appropriate surface construction commands. By putting the surfaces together properly in 3D space, you obtain the 3D object. This is the bottom-up construction process. If the surfaces you construct do not conform to your concept, you think about the curves and points again.

Detailing

With practice you gain the experience that makes this process more efficient. Logically, you construct points and curves, and from these you construct surfaces and solids. In detailing your design, you add surface features. To construct surface features that conform in shape to existing surfaces, you create points and curves from existing objects.

SURFACE MODELING PROJECTS

The prime goal of surface modeling is to construct a set of surfaces and put them together to represent an object depicting a design. Constructing surfaces inevitably requires curves and points. In comparison to surface construction, curve construction is a tedious job.

Because the location and shape of the curves have a direct impact on the shape of surfaces constructed from them, the particulars of most curves in these modeling projects are given to you. While working on these modeling projects, you should try to relate the 3D curves to the 3D surfaces. It is hoped that you reverse the process, seeing the curves when a surface is given.

MODELING PROJECT

1

Bubble Car Chassis

INTRODUCTION

This modeling project, together with the next one, concerns making a 1- to 10-scale model of the famous bubble car, which was first designed and produced by an Italian refrigerator manufacturer about six decades ago. The design was later on franchised to a number of car manufacturers, who built the car in huge quantities, with a variety of configurations. The computer model that you are going to construct, as shown in Figures P1-1 and P1-2, is based on an earlier model.



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FIGURE P1-1 Right-front view of the Rhinoceros bubble car model rendered by using Flamingo nXt



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FIGURE P1-2 Rear-right view of the Rhinoceros bubble car model rendered by using Flamingo nXt

OBJECTIVES

After working on this modeling project, you should be able to

- Demonstrate a general understanding on using Rhinoceros's simple surface and solid modeling tools

OVERVIEW

This bubble car is chosen as a Rhinoceros computer modeling project because the car has simple engineering structure and a stylish body. Besides, we may find a lot of information about it by searching over the Internet, using the keyword "Isetta." For the sake of simplicity, construction of this model will be carried out in two unique modeling project sessions. In the first session, we will work on the engineering components. Making the engineering components can help you understand the process of constructing geometric shapes of regular pattern. In the second session, we will concentrate on body panel parts. Building the body panels lets you experience how free-form surfaces can be constructed. The components that we will construct here are shown in Figure P1-3. The main body panels, consisting of an egg-like shell (Figure P1-4) and a refrigerator-like front door (Figure P1-5), will be dealt with in the next session.



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FIGURE P1-3 Bubble car mechanical components to be constructed in this project



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FIGURE P1-4 Rendered bubble car body panels to be constructed in the next project



FIGURE P1-5 Bubble car's refrigerator-like front door panel to be constructed in the next project

CONSTRUCTING THE MECHANICAL COMPONENTS

Figure P1-6 shows the bottom views of the mechanical components we will make in this project. They are wheels, tires, rear axle, chassis, rear suspension, engine, exhaust system, intake system, gas tank, front suspension, drive controls, floor panels, and seat.

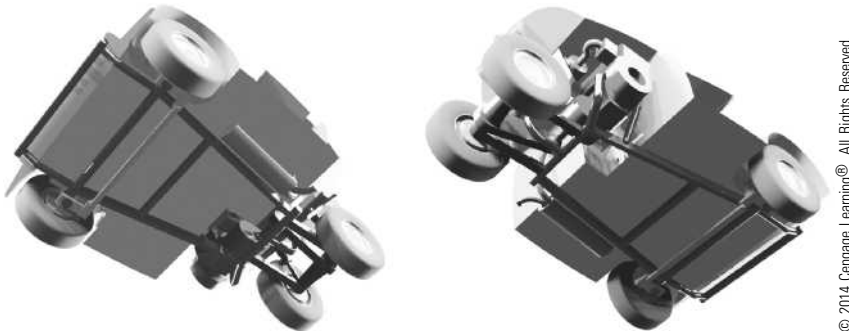


FIGURE P1-6 Bottom views of the mechanical components of the bubble car model

Template Files

To build the mechanical components, we need NURBS curves as wireframe skeletons, upon which we will build the NURBS surfaces, polysurfaces, and solids. Because curve building is a tedious task, we have constructed all the required curves and placed them in various layers of a file located on the Student Companion Website. Consult the preface of this text for information on how to access the Student Companion Website and download all of the Rhinoceros files. As the computer model is a 1- to 10-scale model of the real car, we have simplified most of the component parts appropriately. Now open the template file.

1. Select File > Open and select the file Bubble Car 1.3dm from the Modeling Project 1 folder that you downloaded from the Student Companion Website.

In the folder, you will find other template files depicting various stages of model construction. Therefore, at any stage if you want to have a break, you do not need to save your file. Next time when you continue with your tutorial, you need only to select one of the templates in the folder.

Bubble Car's Tires and Wheels

A default Rhinoceros file has four viewports; each has its own construction plane. As shown in Figure P1-6, the Top viewport is maximized and there are several curves, residing on three separate layers. Curves X, Y, and Z serve to indicate the

rear track, wheelbase, and front track, respectively. They are used for reference purpose, positioning the tires and wheels. Curves A and D depict the cross-section of the tire and wheel. We will revolve them to construct a set of revolved surfaces.

SURFACE CREASE

Because the curves you will use to construct revolved surfaces have kink points and you have to fillet the surface at the kinks, prior to surface construction, you need to perform the following steps to split the resulting surface at creases in order to produce a polysurface instead of a single surface.

2. Type CreaseSplitting at the command area.

A way to execute a command is to type its command name at the command area.

3. Select Yes at the command area.

Subsequently constructed surfaces from curves with kink points will be constructed as polysurfaces. Otherwise, they will be constructed as a single surface with creases.

Now you can use the curves to construct revolved surfaces.

4. Select Surface > Revolve.

5. Select curve A, indicated in Figure P1-7, and press the ENTER key.

6. Set object snap mode to End and then select endpoints B and C.

7. Select the FullCircle option on the command area.

A polysurface depicting the wheel of the bubble car is constructed on the current layer.

8. Set current layer to Tire.

You will construct the next object on this layer.

9. Select Surface > Revolve.

10. Select curve D, indicated in Figure P1-7, and press the ENTER key.

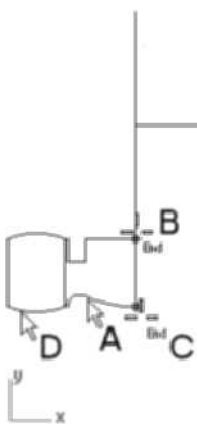
11. Select endpoints B and C and select the FullCircle option on the command area.

Another polysurface representing the tire is constructed.

12. Turn off layers Curves Wheel and Curves Tire.

The curves are not required anymore.

Top



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FIGURE P1-7 Curves for making the tire and wheel and curves indicating the wheelbase and track

Because curves A and D are sets of curve elements joined together, the outcome is sets of surfaces joined together or a single surface, depending on the CreaseSplitting setting. As each of these outcome surfaces encloses a volume without any opening, it is regarded as a solid in Rhinoceros terms.

The edges of the polysurface representing the tire are to be rounded by filleting. Basically, there are two different ways to construct a filleted edge: by using the fillet command from the Surface menu or the fillet command in the Solid menu. Because the surfaces are already joined as a polysurface during their construction, we will use the solid filleting command to round off two edges, as follows.

13. Select Solid > Fillet Edge > Fillet Edge.
14. Select the Current Radius option on the command area and set it to 2 mm.
15. Select edges A and B, indicated in Figure P1-8, and press the ENTER key twice.

Fillet surfaces are constructed at the edges.

SURFACE CREASES

If the revolved surface is a single surface because prior setting of CreaseSplitting is not Yes, the resulting surfaces are single surfaces, not a polysurface. Naturally, you cannot fillet along a single surface; therefore, you have to select Surface > Surface Edit Tools > Divide Surface; select A, indicated in Figure P1-8, and repeat steps 13 through 15 again.

We will now copy the tire and wheel from the rear axis to the front axis of the car, using the front and rear track lines as references.

16. Select Transform > Copy.
17. Select the tire B and wheel C (Figure P1-8) and press the ENTER key.
18. Select endpoints D and E and press the ENTER key.

The tire and wheel are copied.

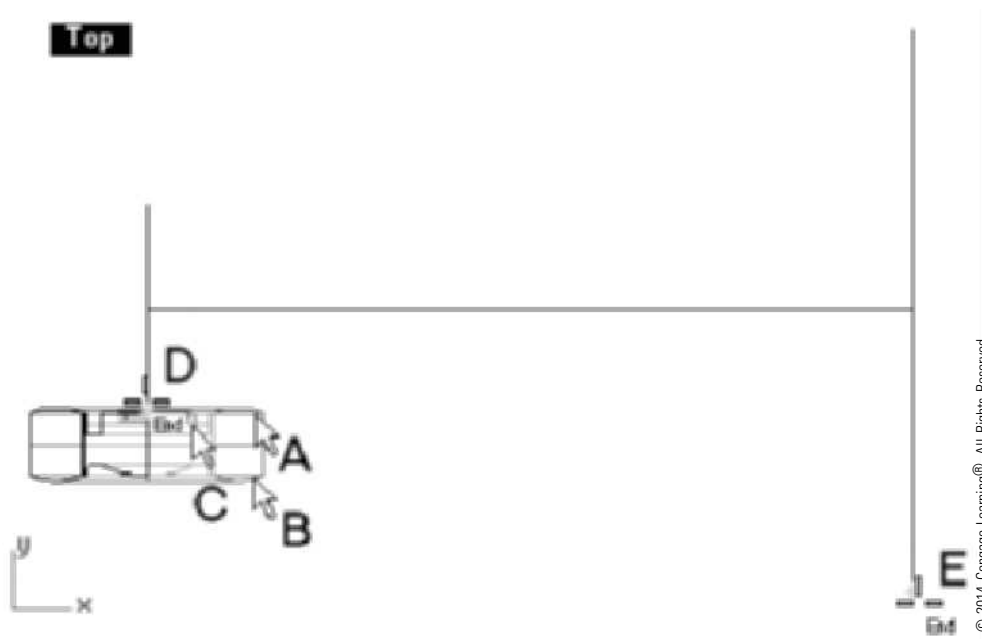


FIGURE P1-8 Tire being filleted and tire and wheel being copied

As the car is symmetric left and right, we will make a mirror copy of the tires and wheels, as follows.

19. Select Transform > Mirror.
20. Select A, B, C, and D, indicated in Figure P1-9, and press the ENTER key.