



COBALT



XENON



ARGON

Designer**Elements**

3D Modeling Getting Started

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Ashlar-Vellum Modeling Products

Ashlar extends the Vellum family of products with the introduction of Argon, Xenon and Cobalt. The three products were created specifically for designers who need to generate or manipulate precise 3D models. Argon provides the base foundation of the modelers whereas Xenon and Cobalt add functionality that greatly enhances productivity for certain design processes. The differences in Argon, Xenon, and Cobalt were developed to provide value and choice for consumer and product designers who frequently have diverse design tool requirements due to their unique environments.

Argon

Argon provides the core foundation for all modeling products. Based upon the ACIS geometry engine, Argon integrates precise curve, surface, and solid modeling design methods through the easy to use Vellum interface. Designers can use curves to create surfaces/solids, surfaces to create solids, and then choose from one of the many feature operations such as shelling and blending to further refine their design.

Not only does Argon provide a wide variety of modeling tools, but Argon provides means to precisely share data with other CAD/CAM/CAE packages through DXF/DWG STEP, IGES, PRO/E, Parasolids (UG, SolidEdge, SolidWorks), and ACIS (AutoCAD, Inventor, Mechanical DeskTop) translators.

Xenon

Xenon adds additional functionality to the Argon core package by enabling associativity between the design data. Associativity is a transparent relationship created between the original geometry and the resulting shape. These relationships are

stored in a history tree such that any modifications to the original shape can be replayed to update the design. You can view the history tree for curves, surfaces and solids using Design Explorer. Associativity is a critical component for rapidly exploring design concepts and iterations through modifications.

The following scenario is an example of how associativity can enhance a designer's productivity.

1. Designer creates a collection of curves defining an organic shape in space.
2. Surfaces are added to the curves.
3. Surfaces are trimmed to other surfaces or relimited as needed.
4. Surfaces are stitched into a watertight body represented as a solid.
5. Body is shelled and faces drafted for manufacturing demands.
6. Body edges are smoothed with any of the seventeen blend or chamfer tools.
7. A 2D drawing is generated consisting of side, front, top and isometric views.
8. Design requires modifications on the original curves.
9. Curves are modified and steps 2 -7 automatically update through associativity.

Cobalt v5

Cobalt is the next generation of Ashlar's Vellum Solids modeling program. Cobalt adds more drawing composition, assembly modeling, bill of materials, and GD&T tools relative to Xenon. Cobalt is the package of choice for creating industry standard drawings.

Another major feature of Cobalt is that it provides an equation-driven parametric constraint management system. Using constraints, the designer can over, under or fully constrain profiles to move in relations relative to other curves and/or dimension values. One of the more exciting features using the constraint system is to animate dimension values by providing minimum/maximum ranges and the number of steps to evaluate. Below is a collection of dimension and geometry constraints supported by Cobalt v5.

- Horizontal, vertical to reference system
- Coincident, colinear, parallel, concentric
- Tangent, symmetric, perpendicular, offset, equal lengths
- Horizontal, vertical, parallel, radial, and diameter distance constraints

Documentation

Most of the graphics in this manual apply to both Macintosh and Windows operating systems. For those instances requiring a platform specific reference, a Windows graphic is used.

Installing Ashlar-Vellum 3D Modeling Products

You can find instructions for quick and easy installation on the CD-ROM that is included with your product or, if you downloaded an Ashlar-Vellum Modeling product from our web site, running the installer will place an informational Read Me on your hard drive.

Serial Number/Registration Code

Your serial number and registration code are required to operate the software; the serial number and registration code are located inside the CD package. If the registration code is not there please call 800-966-2349 to request it. If you purchased through our website they were sent to you via email.

Enter your name and company name to personalize your copy of the software. If you choose not to register your copy, your Ashlar-Vellum Modeling product will run in demonstration mode for fourteen (14) days. For information regarding the purchase of Ashlar-Vellum products in the US call 1.800.877.2745.

It is important to register your Ashlar-Vellum Modeling product so we can provide you with the best possible service. Registered users of our software are eligible for free technical support (for thirty [30] days after receipt of Registration code), information regarding new versions and products, discounts and special offers on new products.

System Requirements

Windows

Minimum Requirements

- Windows 98
- Pentium II
- Video Card

- 256 MB RAM
- 150 MB Free hard drive space
- CD-ROM

Macintosh

- OS 8.6
- G3 Native
- Video Card w/ OpenGL
- 256 MB RAM
- 512 MB Allocated virtual memory
- 150 MB Free hard drive space
- CD-ROM

Recommended Requirements

Windows

- Windows NT/2000
- Pentium 4 or AMD Athlon
- 32 MB Accelerated Video Card w/ OpenGL
- 512 MB RAM
- 250 MB Free hard drive space
- CD-ROM

Macintosh

- OS 9.1
- G4
- 32 MB Accelerated Video Card w/ Open GL
- 512 MB RAM
- 1 GB Allocated virtual memory
- 250 MB Free hard drive space

- CD-ROM

Tutorial Introduction

This tutorial is a learning, productivity and reference tool that will introduce you to some of the most commonly used features and functions in the Ashlar-Vellum modeling products: Cobalt, Xenon and Argon. You'll also be introduced to our world renowned Drafting Assistant that anticipates your next design action, making Ashlar-Vellum software work the way you think. You'll discover how easily you can go from 2D design to 3D with the Drafting Assistant and create your own designs.

The exercises are organized into chapters beginning with an introduction of Ashlar-Vellum's integrated design environment and continuing with the creation and modification of an inline skate chassis.

The majority of the exercises in this tutorial apply to all three products. However, both Xenon and Cobalt feature intelligent history-based associativity that speeds the design revision process and aids in design management. The design modification exercise featuring the Design Explorer applies only to Cobalt and Xenon. Although Cobalt includes additional features not available in Xenon or Argon, the exercises in this tutorial do not address these features.

By the end of this tutorial, we hope you will have obtained enough familiarity with the model creation tools to create your own parts and explore features and tools not covered in these exercises.

Menus and Submenus

Choosing Commands

As you proceed through the exercises, you will be directed to choose commands contained in submenus of other menus, like the pull down menu. For example, you might be asked to set a color by going to the Color submenu of the Pen menu. That will be shown as ***Pen>Color***.

Margin Notes

Margin notes provide alternative ways of doing a procedure or may refer to another section or chapter for related information. There are three types of margin notes: Tip, Tech Note and Referral. These notes are given special treatment so that you can instantly recognize their significance and locate them for future reference.

Tip

A tip provides instructions for getting the most out of Ashlar's 3D modeling products. Tips may show you how to speed up an operation or perform some timesaving drawing technique.

Tech Note

A technical note provides additional technical information that may help you when using a tool or an alternative way of doing a procedure in an exercise.

Referral

A referral directs you to related information contained elsewhere in the manual for the particular topic being addressed.

Style Conventions

This manual uses various style conventions which highlight certain terms or phrases. The list below includes an explanation and an example in parentheses. The conventions are as follows:

Bold	Tool palette names (Light palette); tool names (Single Line tool); Keyboard-entered text; Definition terms (as shown in these style conventions)
<i>Italic</i>	Terms used for the first time in a chapter; Drafting Assistant notations (<i>midpoint</i>); tool and dialog box options (<i>Angle</i> data field); section references (<i>User Guide</i>); Message Line directions (<i>Single Line: Pick the beginning point.</i>); margin note headings (<i>Tip</i>); menu commands (<i>Zoom Previous</i>); filenames (<i>prefs.ini</i>); stand alone extensions (<i>.dwg</i>); directory names; drawing names
<i>Bold and Italic</i>	Command series (<i>Layout>Preferences>Snap</i>)
ALL CAPITALS	Key names on the keyboard (ENTER, RETURN)
Title Capitalization	Dialog box names (Edit Objects); menu names (Pen menu); special Vellum phrases (the Drafting Assistant)
all lower case	File names (<i>prefs.ini</i>); stand-alone file extensions (<i>.dwg</i>)

Step Conventions

The numbered tasks in the exercises describe the activity you are to perform, and the bulleted steps beneath the numbers tell you how to accomplish the task. If you already know how to accomplish the task, you should do it without following the bulleted directions, then proceed to the next numbered task.

Exploring

Some users like to go off on their own to explore while going through the exercises of a tutorial. This is an excellent way to learn more about Ashlar's 3D modeling products. If you are adventuresome, open a new document for your explorations and then switch back to the tutorial document when you want to continue with the exercises.

Occasionally, the tutorial may verify a position or entry that seems obvious to you. If the condition is vital to the next step and you might have inadvertently deviated from the tutorial path, verification (for example, "the x,y location is 0,0") has been added to ensure that you get the correct result from the exercise.

There's More than One Way

Ashlar-Vellum Modeling products often provide more than one way to perform a task. This tutorial describes only one method at a time and may show you a different way to do the same task later. When you start to develop a preference, feel free to substitute your own method for whatever is suggested here, provided you're certain that your method produces the same outcome as the tutorial.

Basic Terminology

This manual uses the following terms for mouse activities:

Pointer

An arrow or any other graphic symbol that allows selection or creation of an object. Move the pointer to point to a command or an object on the screen. Depending on its location, the pointer is an arrow or may look like the current tool.



Arrow Pointer



Selection Arrow



Center-Point Circle

To move the pointer, move the mouse on the mouse pad.

Point	Move the mouse until the pointer is over the item you want.
Press	Press and hold down the mouse button.
Click	Quickly press and release the mouse button once.
Double-click	Click the mouse button twice, quickly in succession.
Drag	Press and hold down the mouse button, move the mouse, then release the mouse button.

3D Modeling Environment

Ashlar-Vellum's 3D design environment was developed to allow you to focus on the part you are designing rather than on the software. In this chapter you'll do the following:

- Familiarize yourself the user interface
- Explore the Drafting Assistant
- Save preferences
- Select objects
- Be introduced to drawing techniques

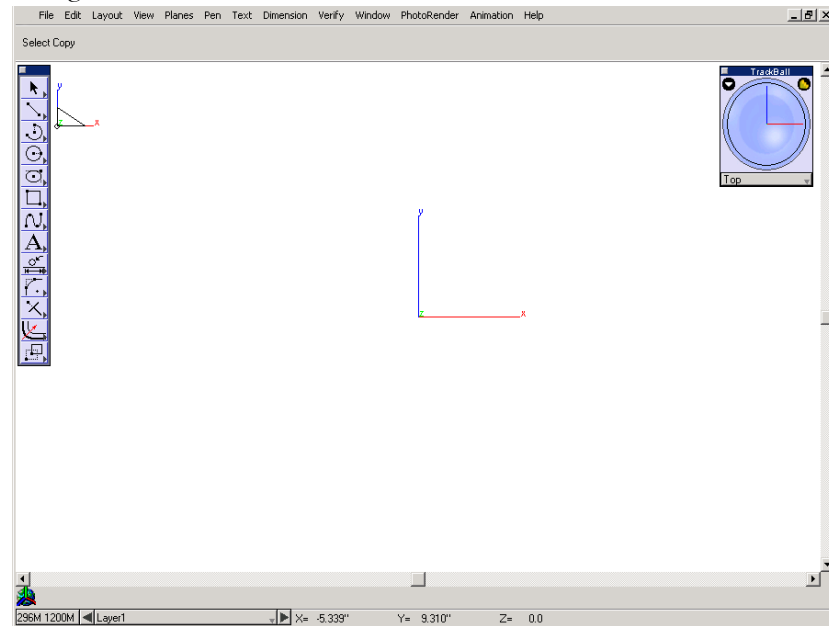
Exercise 1: Ashlar-Vellum's User Interface

This exercise introduces you to the user interface.

1. Launch one of our 3D modeling products (Cobalt, Xenon or Argon).

Our Modeling product opens, displaying an untitled document with an empty

drawing area.



2. Take a look at the menu bar. There are twelve menus.

File

Contains commands that affect entire documents (files), including opening and closing files, setting preferences and defining your *Print Setup* (Windows) or *Page Setup* (Macintosh).

Edit

Contains commands to select and manipulate objects. These include such things as copying and pasting as well as changing an object's direction, resolution, layer and type.

Layout

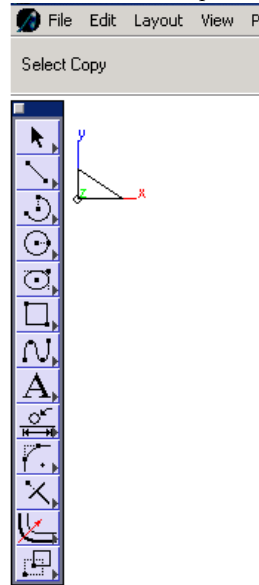
Contains commands and settings that specify the drawing area and provide program features and functionality such as the *Grid*, *Layer Manager*, *Group* and *Align*.

View

Contains commands for displaying your document and includes choosing and setting views and zoom options.

Planes	Contains commands for choosing and defining the work plane.
Pen	Contains commands to specify pen characteristics (color, weight, and pattern), polygon patterns, polygon fill, crosshatching and arrows.
Text	Contains commands to set the font, size, style and case of your text (lower, upper and title capitals).
Dimension	Contains commands that specify dimensions, their format and tolerance.
Verify	Contains commands to obtain information about your file and specific objects in your file. These include properties of an object, direction, curvature and object counts.
Window	Contains commands for displaying tool palettes, the Design Explorer, Trackball and Showing or Hiding objects.
PhotoRender	Contains commands for rendering your geometry and setting options for rendering.
Animation	Contains commands for generating Quicktime movies.
Help	Contains the commands to access the on-line help file.

3. Take a look more closely at the main tool palette.



4. This tool palette contains drawing and editing tools used for constructing, editing and annotating geometry as you develop parts and assemblies. Most of these tools in the palette have subpalettes containing tools with related functions. The arrow in the lower right corner of the tool icon represents the presence of a subpalette.

Selection tool

Select object(s) and points in the drawing area.



Line tools

Create line segments, connected lines, line parallel to existing lines and points. As you create a line, the coordinate locations, line length and angle from horizontal appear in the Status Line.



Arc tools

Create arcs with center, beginning and end points.



Circle tools

Create circles with a center point and diameter.



Ellipse/Conic tools

Create ellipses with a center, major axis point and minor axis point or conic curves defined by a start point, end point, shoulder and slope control points.



Polygon tools

Create rectangles, inscribed polygons, circumscribed polygons, arbitrary polygons and polygons from curves.



Spline tools

Create smooth, free-form curves passing through specified points - NURBS (Non-Uniform Rational B-Splines). NURB splines don't develop kinks as a spline is altered and you can make changes to a specific area without affecting the remainder of the spline. This palette also includes the **Helix Curve** tool.



Text tools

Create text for annotating designs in the drawing area. You can set the text characteristics, such as font, text size and case. The **Text** tools in our 3D modeling products do not use text boxes but text objects that can be moved independently.



Dimension tools

The Dimension tool palette has been added to the main tool palette for easy access.



Fillet/Chamfer tools

Construct fillets and chamfers from corners formed by nonparallel lines or curves. Our 3D modeling products automatically trim fillets and chamfers by default but you can override this to prevent trimming.



Trim tools

Lengthen or shorten lines and curves. Generally, you select the limiting object(s) before you select a specific trim tool.



Curve Extras tools

Contains tools for offsetting, extruding, revolving, exploding (creating) curves from edges and projecting curves.



Transformation tools

Contains tools and commands for transforming geometry with respect to scale, location and the like. These **Transformation** tools include: **Move**, **Rotate**, **Expand/Shrink** and **Mirror**.



5. Select the **Opposite-Point Circle** tool, the first tool in the **Circle** tools subpalette.



When you select a tool and move the pointer into the drawing area, the pointer becomes a *smart pointer* representing the tool and displaying indicators for multi-step procedures.



On one side of the smart pointer is a dot, the *hot spot*, showing the point you should specify. This dot indicates that the first click of the mouse places a point on one edge of a circle being created. The dot changes position on the pointer during each step of construction.

- Take a look at the Message Line at the top of your drawing area.

The Message Line provides instructions for using the current tool. Since the **Opposite-Point Circle** tool is selected, the Message Line reads, *Opposite-Point Circle: Pick first point on circle [Ctrl = Copy Previous (Windows) or Option = Copy Previous (Macintosh)].*

Notice that the Message Line tells you not only the step for using the tool but notifies you of an additional option. These options are available for a number of tools in our 3D modeling products.

- Click the first point of the circle. The hot spot on the smart pointer moves to the other side of the circle pointer icon.



The Message Line now reads, *Opposite-Point Circle: Pick opposite point on circle*

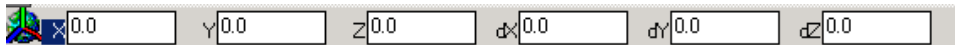
- Move the smart pointer. As you do so, a rubberband image appears. (The smaller circle, in the graphic here, is the smart pointer for the **Opposite-Point Circle** tool.)



- Click to indicate the second point of the circle.

The circle appears and the hot spot on the smart pointer returns to its original position for creating another circle.

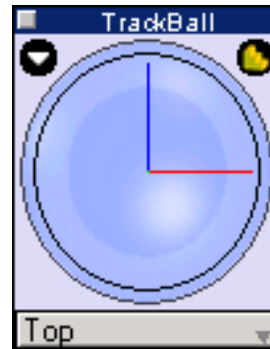
- Take a look at the Status Line at the bottom of the drawing area.



When you select a tool, the Status Line contains data fields that provide information about the geometry being created, such as measurements, angles, coordinates and the like.

You can use the Status Line in three ways: to create an object with keyboard entries, to edit an object that was just created and is still selected, and to create additional wireframe objects using the current tool.

11. Notice that the X data field is highlighted. Whenever you construct an object, the data field that you are most likely to change is active.
12. Choose **Window>Trackball**.
The Trackball displays.



The trackball lets you rotate the view orientation of your geometry in the drawing area. This Sphere Trackball allows you to drag your view to the desired rotation.

13. Click the arrow button on the right side of the Trackball title bar.
The Trackball changes to the Step Trackball.



Tech Note:

You can set the angle increment for the Step Trackball using the View Rotation Options dialog box. See Chapter 7 of the *User Guide* section for information.

This trackball gives you the ability to rotate your views in regular angle increments or in a continuous movement. By clicking an arrow, your drawing rotates 5° in the direction selected.

14. Click the arrow button on the Trackball title bar again to return to the Sphere Trackball.

Exercise 2: Exploring the Drafting Assistant™

The Drafting Assistant™ is the feature that separates our 3D modeling products from other design and drafting software. The Drafting Assistant™ thinks like a designer. It automatically knows where you typically want construction lines and displays them temporarily when you need them.

The Drafting Assistant™ also makes it easy to select existing points for construction by displaying information about the pointer's location in the drawing area. If a Drafting Assistant™ notation displays when you click, the construction snaps onto the geometry being created.

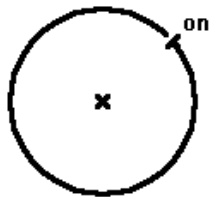
When the pointer is in the drawing area, the Drafting Assistant™ locks onto specific points on existing objects as you move the pointer near them.

1. Create a circle using the **Opposite-Point Circle** tool. The circle size does not matter.
2. Select the **Single Line** tool, the second tool from the top in the main tool palette.

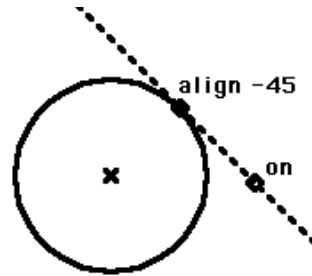


The pointer becomes a crosshair.

3. Move the pointer across around the edge of the circle. Click the first point when an *on* notation appears and the first point of the line is on the circle.



4. Move along the circle in a 45° angle and notice that a Drafting Assistant™ construction line appears.



The Drafting Assistant™ also tells you when the snap point has locked onto the following points of an object.

center



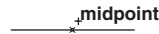
The center of an arc or circle. Move the pointer across the arc or circle to display *on* for the arc or circle, then move the pointer near the center to display the *center* point notation.

endpoint



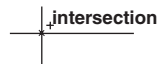
The endpoint of lines, arcs, circles, ellipses and splines.

midpoint



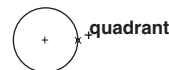
The midpoint of lines, arcs, circles, ellipses and splines.

intersection



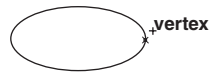
The intersection of two curves (permanent and the Drafting Assistant's dynamic construction lines).

quadrant



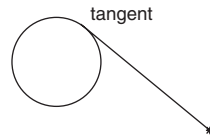
Quadrant points on an arc or circle displayed at 3 o'clock, 6 o'clock, 9 o'clock and 12 o'clock.

vertex



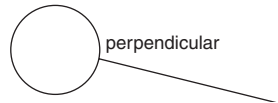
The vertices of an ellipse, spline, or dimension point.

Tangent



Tangent points of an arc or circle. If you click a point on an arc or circle and drag the pointer away at about a 45° angle, the Drafting Assistant locks onto a tangent.

Perpendicular



Perpendicular points off any curve. If you click a point on a curve and drag away at a 90° angle the Drafting Assistant locks onto a perpendicular.

The Drafting Assistant™ also displays the three types of dynamic construction lines you'll most likely use during geometry construction. These appear temporarily to help you align geometry along the X-axis, Y-axis and 45° angle (if in the Top plane).

5. Click to place the final point.

Once you've set a point, the Drafting Assistant™ construction line disappears. Your drawing is not cluttered with unnecessary, extraneous lines.

6. Double-click on the **Selection** tool, the first tool in the main tool palette, to select all geometry.



7. Press BACKSPACE (Windows) or DELETE (Macintosh) to delete all geometry from the drawing.

8. Select the **Single Line** tool again.
9. Click somewhere in your drawing to set the first point.
As you move the pointer, a dynamic construction line extends automatically from the last point you created.

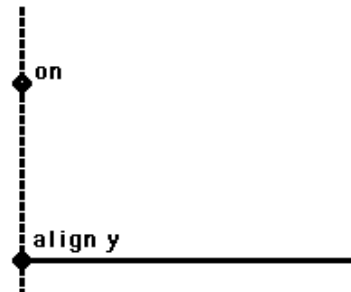


10. Move the pointer along the temporary horizontal construction line and click to place the second point of the line. The exact distance does not matter for this segment length or any of the segments created in the rest of this exercise. Just approximate the line length using the graphics as a reference.



You've completed the first line segment.

11. Move the pointer over the left endpoint of the line segment.
When you perform this kind of action over one or more points, you activate them. The Drafting Assistant™ automatically displays construction lines relative to them.
12. Click on the endpoint to place the first point of another segment.
13. Move the pointer vertically up from that point.



14. Click to place the endpoint of this new line segment.



15. Select the **Connected Lines** tool.

- Place the pointer over the **Single Line** tool and press the mouse button to display the **Line** tools subpalette.



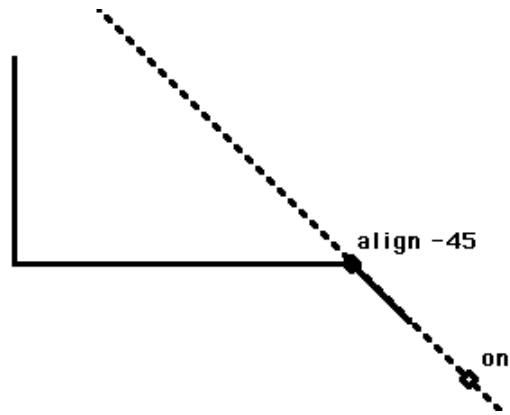
- Drag to the **Connected Lines** tool and release the mouse button. The **Connected Lines** tool now displays selected in the palette.



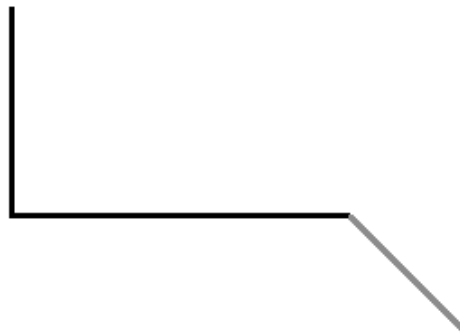
This tool creates connected lines segments rather than single line segments created with the **Single Line** tool.

16. Move the pointer to the right endpoint of the horizontal line to activate that control point and then move the pointer in a 45° direction.

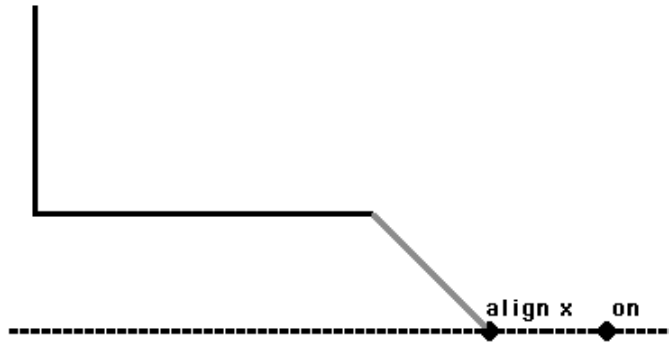
A 45° dynamic construction line appears.



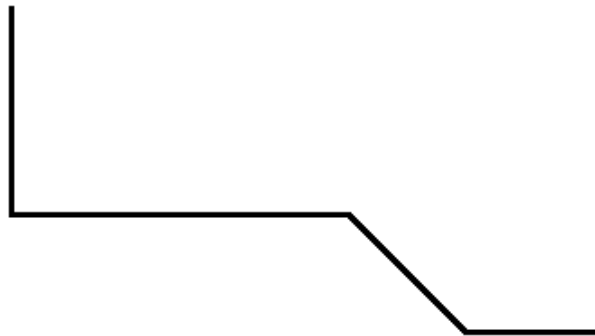
17. Click on a point along the 45° construction line to place the next point.



18. Now move the pointer away from the point horizontally so that a construction line appears aligned along the X-axis. Don't worry if the pointer isn't exactly on the displayed construction line as long as the *align x* and *on* notations display.

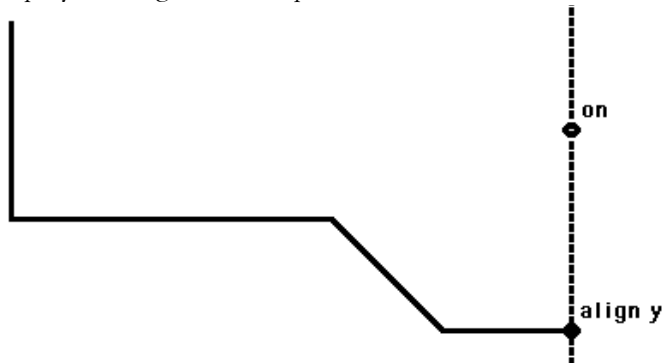


19. Click the next point somewhere to the right of the last point.

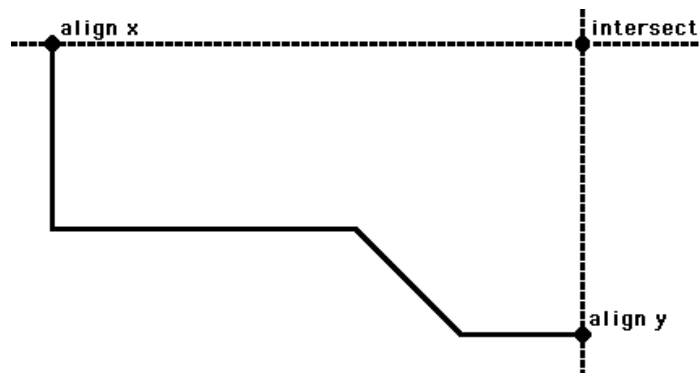


Notice that the point is placed on the construction line even though the pointer wasn't exactly on that line when you clicked the mouse.

20. Move the pointer vertically up from the point and a vertical dynamic construction line displays through that last point.

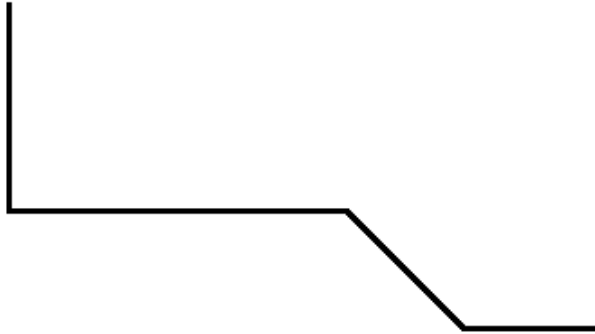


21. Continue moving the pointer so that it is aligned with the top endpoint of the vertical line segment and the last point placed. Two construction lines will display, as shown here.



These construction lines are Drafting Assistant's attempt to anticipate your next action by displaying a possible location for the endpoint. The Drafting Assistant™ displays *intersect*, *align x* and *align y* notations.

22. Double-click to place the last point along that construction line, as indicated in the Message Line.



23. Save (or discard) this file if you wish.

- Choose **File>Save**.
- Type the desired file name.
- Click OK.

This exercise was a very simple example of the power of the Drafting Assistant™. The Drafting Assistant™ will help you with more complex drawings, including 3D, as you'll discover in the next three chapters.

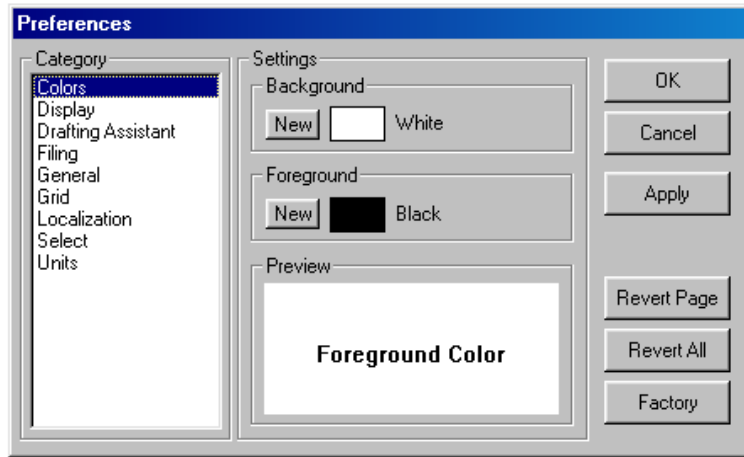
Exercise 3: Saving Preferences

Our 3D modeling products allow you to save preferences for a particular session and in the style in which you work.

1. Choose **File>New** to display an untitled drawing.
2. Choose **File>Preferences**.

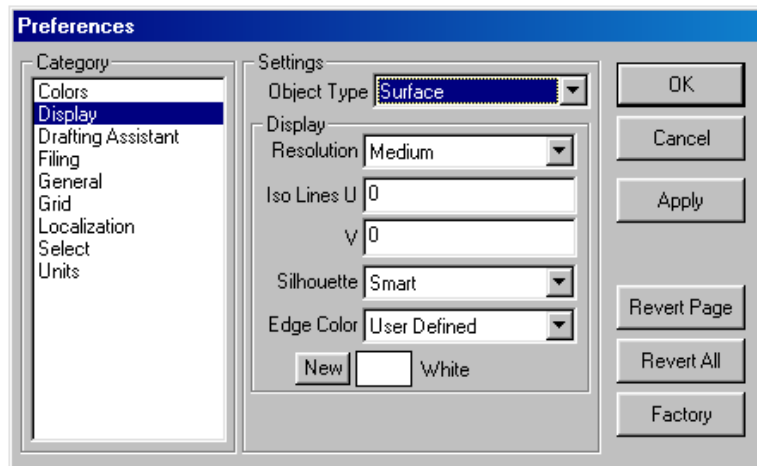
The Preferences dialog box contains a category list of preference groups, group

settings and dialog box buttons.



The preference categories include Colors, Display, Drafting Assistant, Filing, General, Grid, Select and Units.

3. Choose the Display category.
4. From the Object Type section choose Surface.



For surface and solid object types you have the ability to set, Resolution, Iso Lines, Silhouette and Edge Color.

Resolution

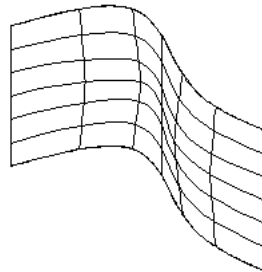
Controls how accurately an object's curves appear. You can set the curve resolution to Coarse,

Medium, Fine, Very Fine and Super Fine. An object with a Coarse resolution draws more quickly but may be less appealing visually than an object with a Super Fine resolution.

Iso Lines

Iso (isopram) lines control the number of *U* and *V* lines displayed for a surface or solid object. Iso lines are constant parameter curves that lie on an object. *U* and *V* are the letters used to define these lines (and their coordinates) in parameter space and are standard for the industry. A zero in both fields turns off iso lines. *U/V* values may enhance the visual appearance of a surface or solid at the expense of drawing speed.

The graphic below shows a surface with both the *U* and *V* iso lines set to five.



Silhouette

Controls the silhouette edge draw mode. A silhouette is a set of edges that separates visible faces from invisible faces of an object with respect to a given viewing direction. For example, a sphere has no real edges. The silhouette feature reads the sphere and displays a circle.

There are three options, Off, On and Smart. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. If the Smart mode is selected, silhouettes will be dynamically drawn based on performance considerations.

Edge Color

Controls the edge color of rendered objects separate from the entity itself. The pull-down menu

provides four options, Foreground, Background, Entity and User-Defined. To specify a user-defined color, click on New, choose a color in the palette and click OK. The new color displays in the Edge Color window with its color values.

5. Explore the various categories and settings available. See the *User Guide* section for information on each of these settings.

When you change any of the preference settings, the changes apply only to new objects.

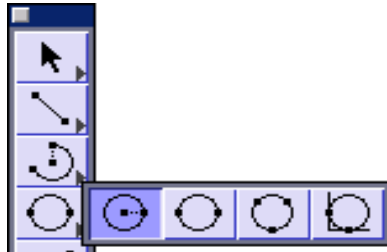
Exercise 4: Selecting Objects

Once you create an object, you will undoubtedly want to make changes. In order to do so, you first have to select the object. In our 3D modeling products, the step at which you select the object depends on which operation you want to perform.

When you select an object it highlights until you select another object or deselect it by clicking in the drawing area. You can select an object with the **Selection** tool or by the hollow selection tool that appears temporarily when using one of the **Edit-*ing*** tools. Selecting an object does not affect the properties of the object. You can select both objects and points.

Selecting an Object

1. Select the **Center-Point Circle** tool.



2. Create a circle of any size in your drawing area.

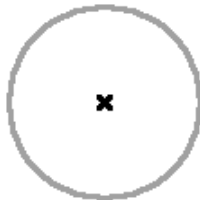
3. Select the **Single Line** tool.



4. Create a line of any length.
5. Choose the **Selection** tool, the first tool in the main tool palette.



6. Place the pointer over the circle and click.
The circle highlights red (the default selection color set in Preferences).

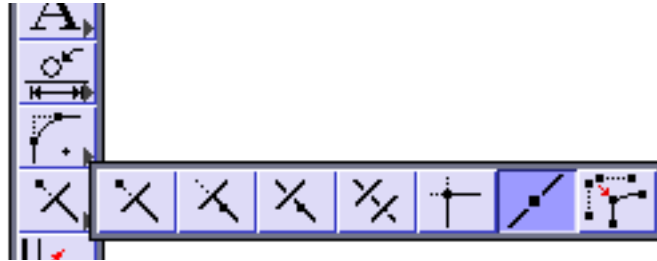


7. Click in the drawing area to deselect the circle.

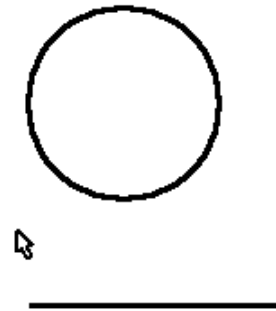
Selecting and Modifying Object with an Editing Tool

When editing an object, you may be asked to select an object after selecting the tool.

1. Select the **Divide at Location** tool.



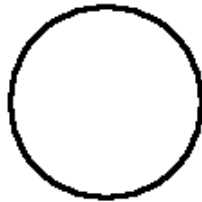
2. Move the pointer into the drawing area. The pointer has become a hollow selection tool.



3. Select the line.
The line highlights and you can now perform the next step with the tool.
4. Click on the line at some location.
The line divides at that location even though you can't see it.
5. To verify the division, choose the **Selection** tool.



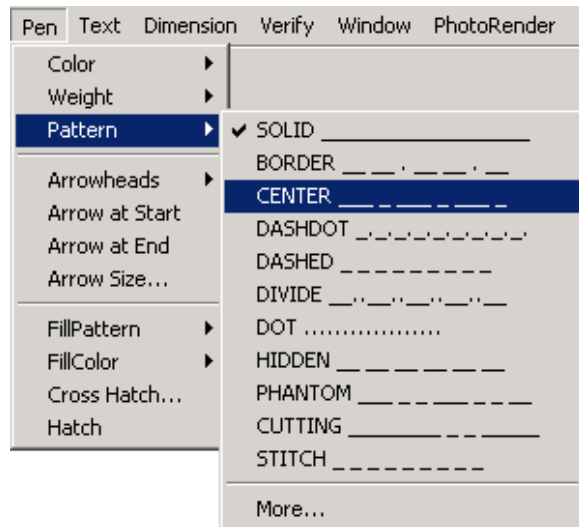
- Click on the line at some location and you will see that only a portion of the line highlights showing that the line has been divided.



Selecting an Object and Modifying it with a Command

You can also edit an object simply by changing one of its properties. Maybe you want to apply a center pattern to a circle to indicate a bolt-hole. You'll need change the pen pattern.

- Using the **Selection** tool, select the circle.
- Choose **Pen>Pattern>Center**.



The circle pen pattern changes to a center pattern display.



Exercise 5: Drawing Technique Options

In our 3D modeling products, all objects are created by clicking. As you create geometry a rubberband image of the geometry displays showing you how the object will appear when you click the last point.

As mentioned previously, each dot on the icon represents a point you must place. The smart pointer indicates the order for designating points.

1. Select the **2-Point Center Ellipse** tool.



2. Move the pointer to the drawing area and click to set the center point of the ellipse.

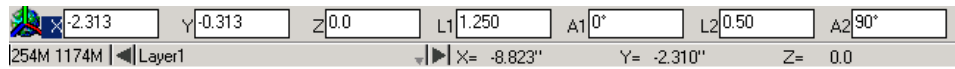
3. Move the pointer to the another location. (It doesn't matter where.) As you do a rubberband image of the line appears.

If you keep moving the pointer, the rubberband images adjusts accordingly.



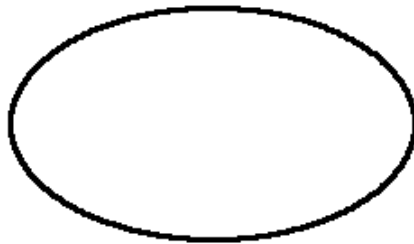
4. Click to set the corner point of the control rectangle of the ellipse, as indicated in the Message Line.

- Notice that the X field in the Status Line at the bottom of the drawing area is highlighted.



This field represents the X location of the center point of the ellipse. Immediately after construction you can change the X location or any other value in the Status Line. If the geometry is not satisfactory, press BACKSPACE (Windows) or DELETE (Macintosh).

- Use the TAB key to tab to the L1 data field.
- Type **1.75**.
- Press TAB twice to highlight the L2 data field.
- Type **1.0**.
- Press ENTER (Windows) or RETURN (Macintosh) and the ellipse updates.



Using the Status Line you can sketch out your drawing without having the exact values for geometry.

Congratulations! You have completed the introductory section of our 3D modeling products tutorial. The next section will introduce you to wireframe modeling.

Wireframe Modeling

Just as there are several ways of approaching a design problem, our 3D modeling products have different ways of modeling objects based on the software features you choose to use in the design environment.

In our 3D modeling products you can create wireframe, surface and solid models. This section of the tutorial introduces you to wireframe modeling, the most basic modeling type. A wireframe consists of geometry that makes up the edges of an object. Wireframe relates to the concept of a wire that is bent to follow an object's edges. A wireframe model is the simplest mathematical representation of an object.

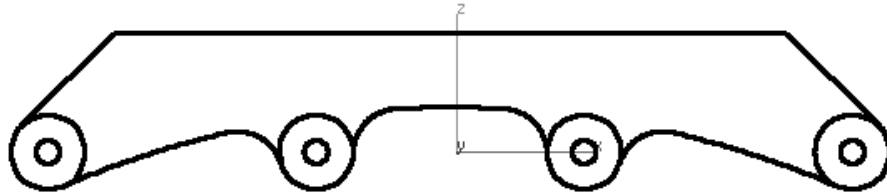
Wireframes in our 3D modeling products consists of points, lines, arcs, circles, ellipses, conics, splines or a combination of these. These same wireframes are also collectively referred to as curves. Although wireframes are limited in the amount of model content they represent, they are powerful building blocks for creating more complex surface and solid models.

A wireframe model can often be used in place of a physical prototype for running simulations. These digital models also can be used for visual inspection, measuring distances between points within the model and observing the visual and real intersection of lines.

In this chapter you'll learn how to:

- Set up your file
- Create wheel axles
- Create the chassis profile

Now let's move on to creating a wireframe profile of the in-line skate chassis, as shown below.



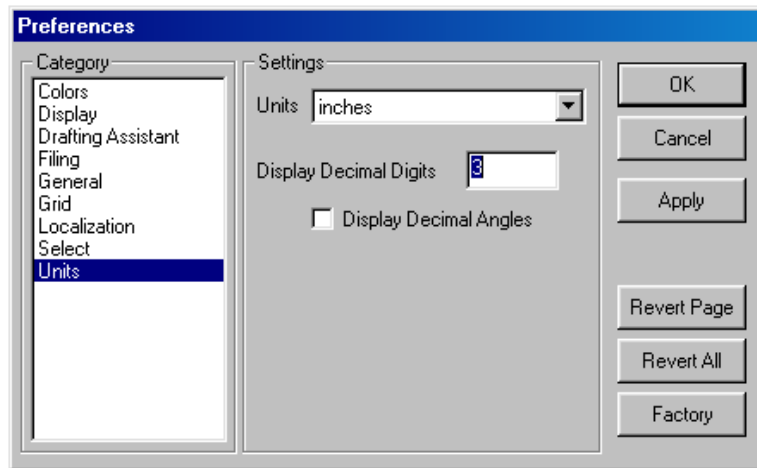
Exercise 1: Setting Up the File

In this exercise you're going to set some preferences for the file as well as the view scale to fit your geometry.

1. Choose **File>New**. An untitled file displays on your screen.

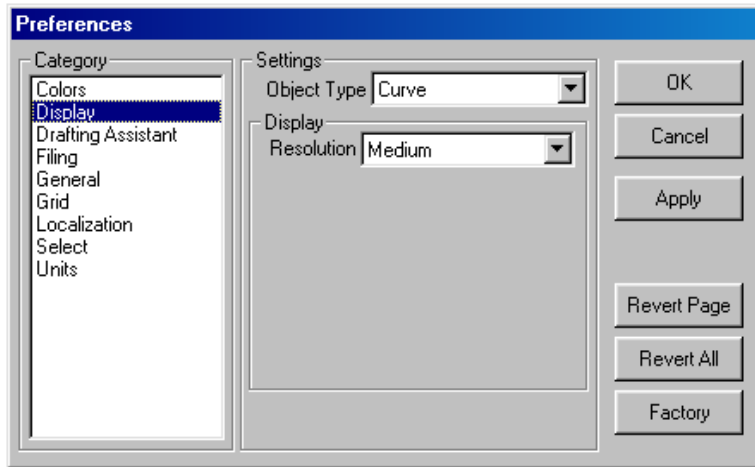
If Ashlar's 3D modeling product is not already running, launch the program. An untitled file automatically displays. You do not have to open a new file.

2. Choose **File>Preferences** to open the Preferences dialog box.
3. Select the *Units* category and set the units to millimeters.



4. Click **Apply** to save the setting.

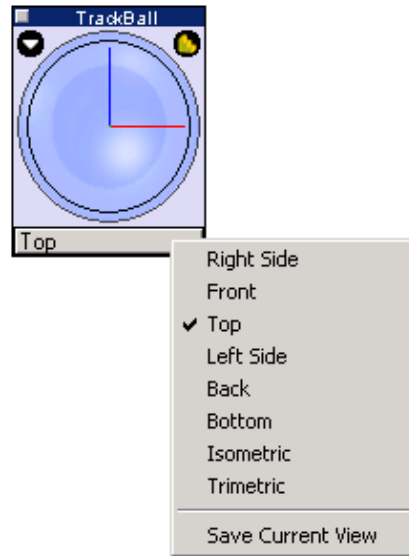
5. Select the *Display* category.



You have three object types, Curve, Surface and Solid and five resolution settings, Coarse, Medium, Fine, Very Fine and Super Fine. You can set the resolution for each object type separately.

6. From the *Object Type* menu, choose Curve, if it is not already selected.
7. From the *Resolution* menu, choose Medium.
8. Set the resolution for the Surface and Solid object types to medium also.
9. Click Apply to save the setting
10. Click OK to close the dialog box.

11. Choose **Window>Trackball** to display the Trackball, if it is not already displayed.



12. Click on the Trackball view menu.
13. Choose Front from the list.

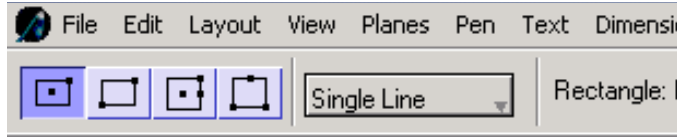
The view changes to Front. Notice that the Triad in the upper left corner of your drawing area shows the Front plane graphically.

14. From the main tool palette, select the **Rectangle** tool.

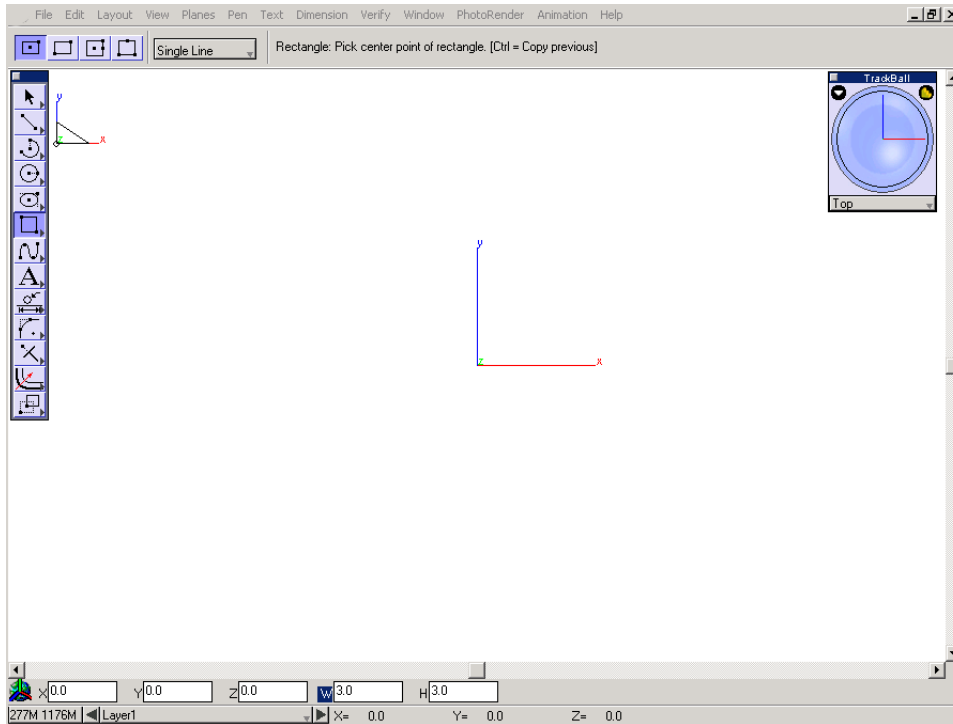


The Message Line includes an additional palette with four types of rectangle tools. polygon by center and point on (**Center/Corner Rectangle** tool), polygon by diagonals (**Diagonal Rectangle** tool), polygon by center/axis (**Center/Axis Rectangle** tool) and polygon by major and minor axis (**Major/Minor Axis Rectangle** tool).

15. Choose the **Center/Corner Rectangle** tool, the first tool in the subpalette.



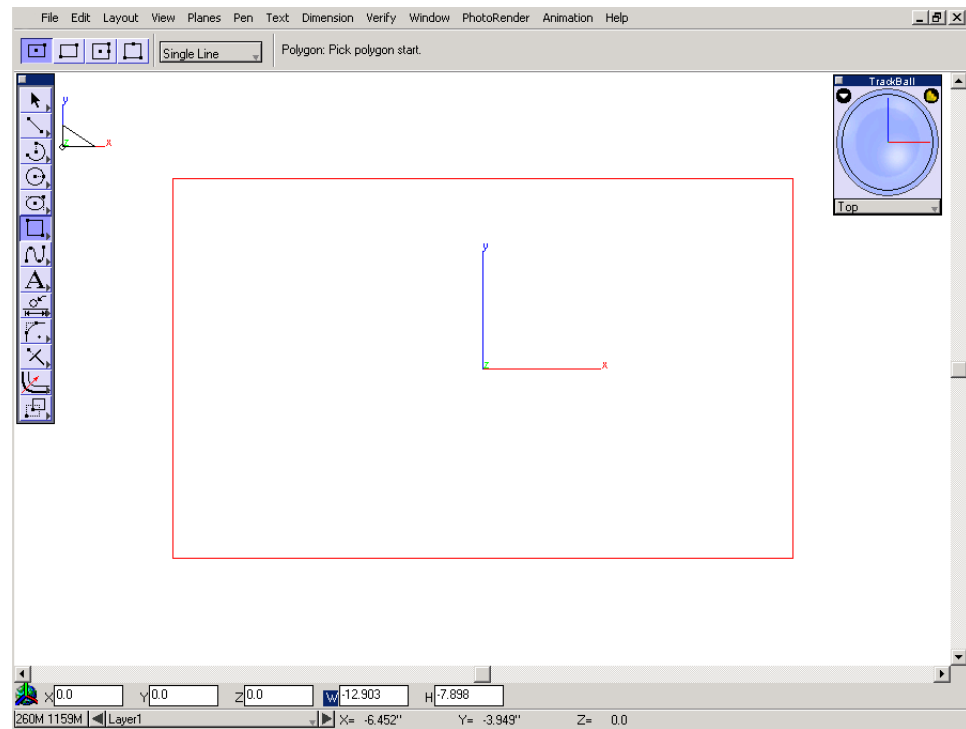
16. Move the rectangle pointer to the origin of your drawing located at the intersection of the X, Y and Z axes.



Notice that the Message Line provides you instructions for using the tool.

17. Click at the origin to place the center point of the rectangle.
18. Move the pointer away from the origin and a rubberband image of the rectangle appears.

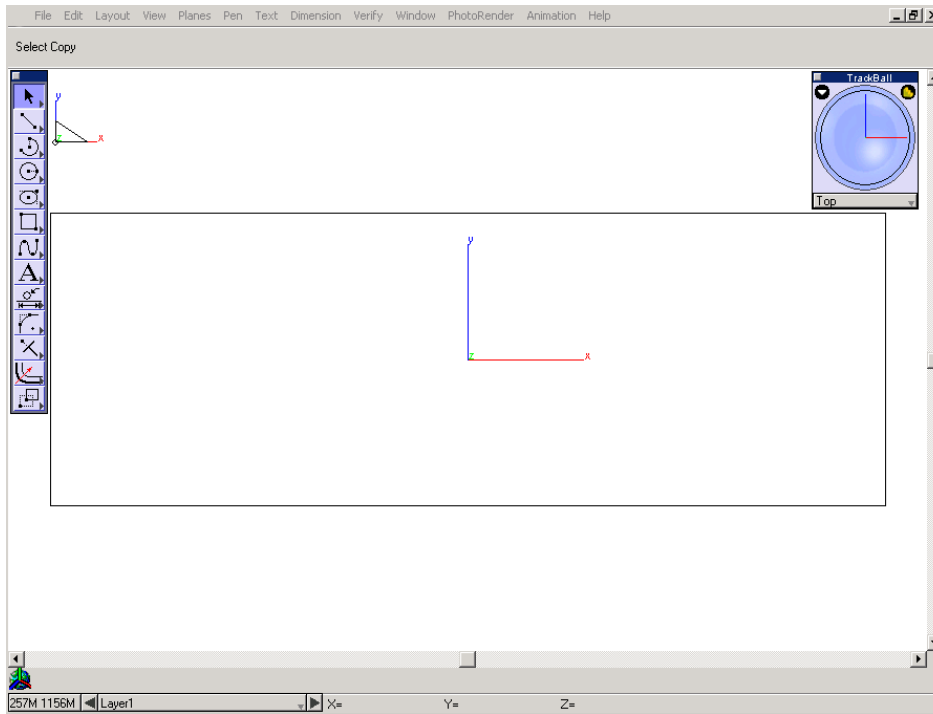
19. Click anywhere to place the corner of the rectangle.



The W or width data field is highlighted in the Status Line below your drawing.

20. Type **400** in the data field.
21. Press TAB to activate the H or height data field.
22. Type **140** into that data field.
23. Press ENTER (Windows) or RETURN (Macintosh) and the rectangle resizes to these new values.
24. Choose **View>Zoom All**.

Your view rescales to fit the rectangle.



25. Delete the rectangle.

- Using the **Selection** tool, drag a selection fence around the rectangle (hold down mouse and drag a selection fence rectangle around the two circles).
- Then press DELETE.
- Close the Trackball if it is taking up too much room in your drawing area.

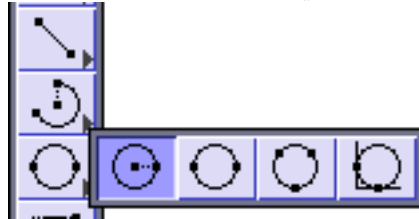
You've now set up your file to begin creating the skate. The rectangle was temporarily created to set the viewing scale to your monitor's screen size.

Exercise 2: Creating the Wheel Axles

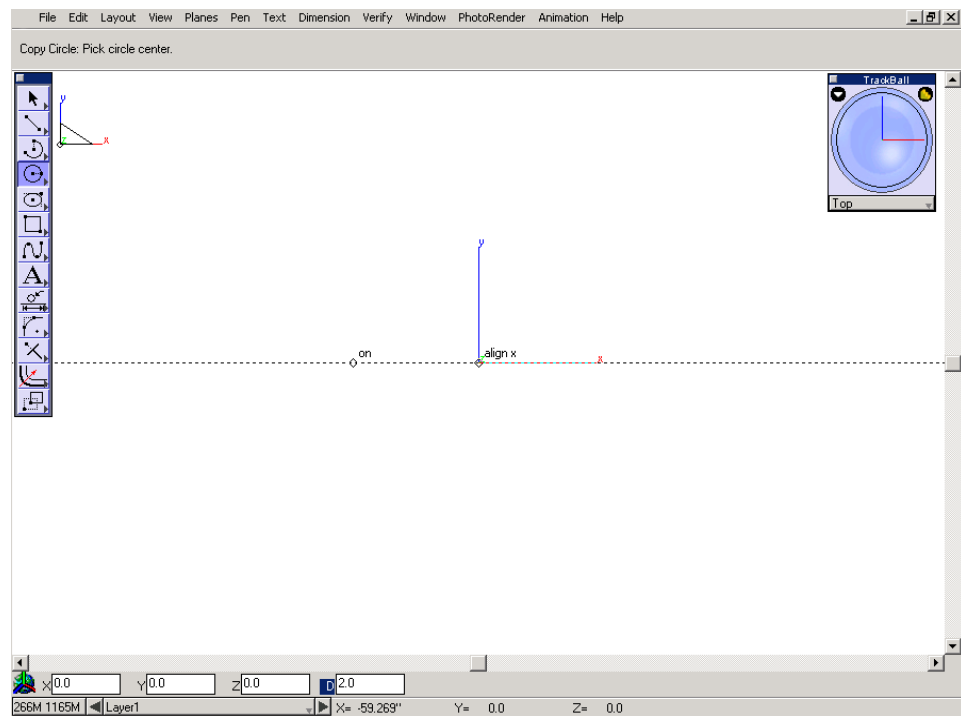
In this exercise you'll create the wireframe wheel axles using the **Center-Point Circle** tool, construction lines and the **Linear Duplicate** tool. You'll start the design in 2D and move on to 3D in the next chapter. You will also assume that the wheels are 82 millimeters in diameter, standard for in-line skates.

1. Create two concentric circles with diameters of 8 mm and 25 mm to the left of the origin.

- From the main tool palette, select the **Center-Point Circle** tool.



- Move the pointer to the origin to wake up a temporary horizontal construction line.

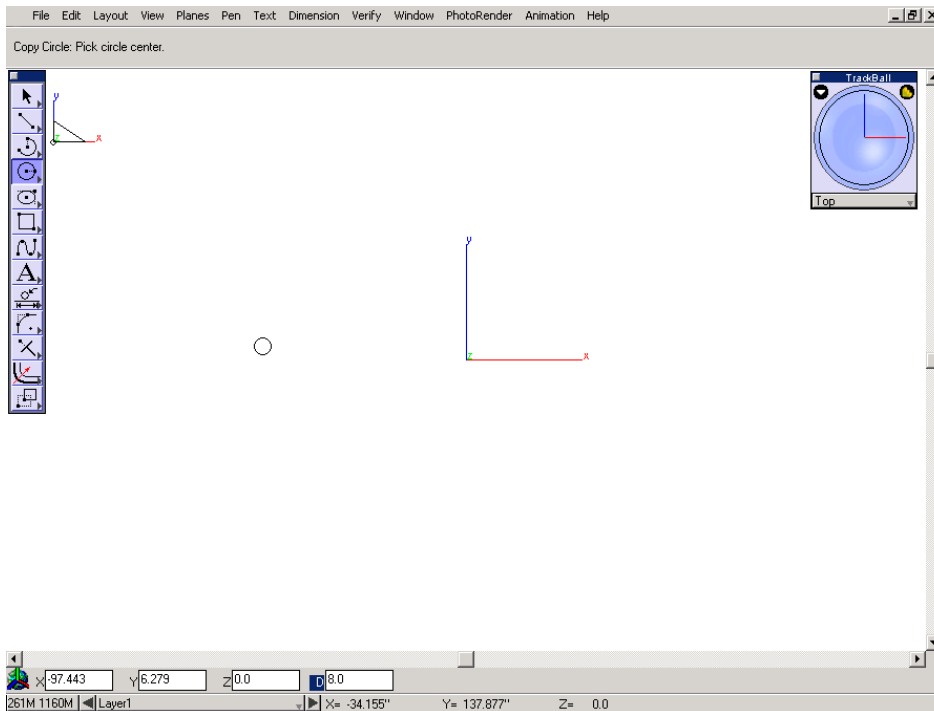


- Now move the pointer to the left, along the construction line, as shown above.
- Near the left of your drawing area, click the center point of the circle.

- Move the pointer a small distance away from the first point and click the second point. The exact distance does not matter.

As you move the pointer to place the second point, a rubberband image of the circle appears.

- In the highlighted D (diameter) field of the Status Line, type **8** and press ENTER (Windows) or RETURN (Macintosh).

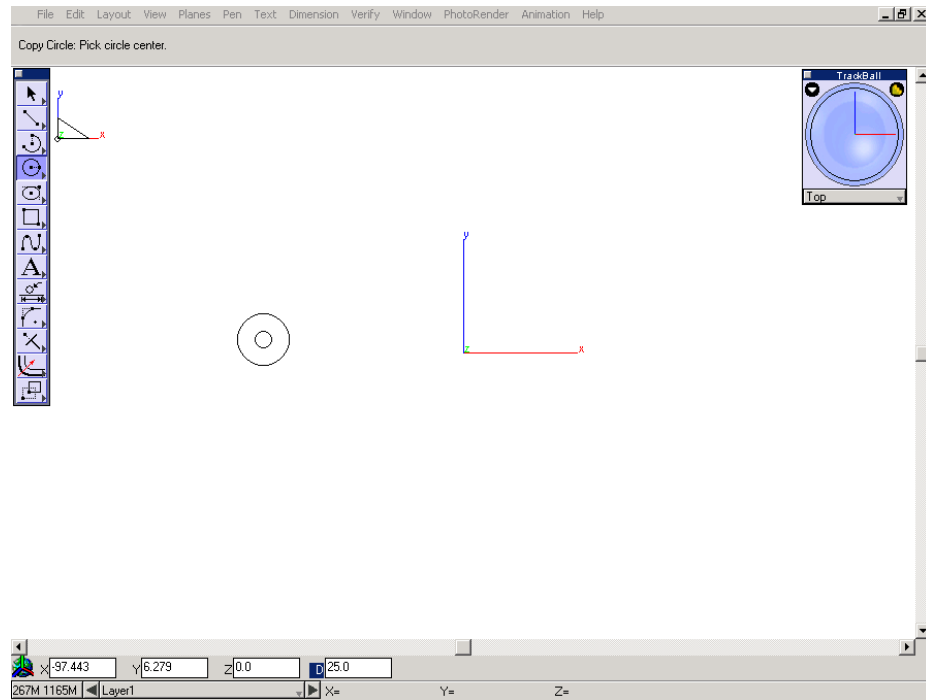


This circle represents the first axle hole.

- Move the pointer to the center of the hole (the *center* notation should appear) and click to place the center point for the second circle.
- Move the pointer out some distance and click to place the second point. Again, the exact distance does not matter.
- Type **25** in the highlighted D (diameter) field and press ENTER (Windows) or RETURN (Macintosh).

Another circle is created with the same center point and represents the wheel

material around the axle hole.



2. Select the two circles.
 - Choose the **Selection** tool
 - Drag a selection fence around the two circles.The two circles are selected.
3. Create three copies of these two circles across your drawing area in the X direction with a step offset of 90 mm for the four circles.
 - Select the **Linear Duplicate** tool from the **Move** tool palette.



The Linear Duplicate dialog box appears.

Linear Duplicate

Columns

Number per row: 5

Offset X*: 2.0 Total offset
 Step offset

Y*: 0.0

Z*: 0.0

Rows

Number of rows: 3

Spacing*: 2.0 Total spacing
 Step spacing

OK Cancel

- In the Number per row field of the Columns section, type **4**.
- In the Offset X field, type **90**.
- Select the Step offset option, if it is not already selected.
- In the Number of rows field of the Rows section, type **1**.



Linear Duplicate

Columns

Number per row: 4

Offset X*: 90 Total offset
 Step offset

Y*: 0.0

Z*: 0.0

Rows

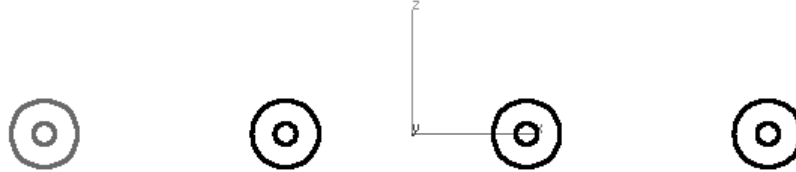
Number of rows: 1

Spacing*: 50.80 Total spacing
 Step spacing

OK Cancel

- Click OK to close the dialog box and create the duplicates.

Three additional axles appear, lying along the same X axis as the original axle.



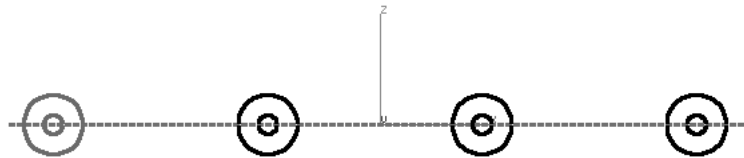
4. Save the file as *Inline Skate Profile*.
 - Choose **File>Save** (CTRL+S, Windows or ⌘+S, Macintosh).
 - In the dialog box, type **Inline Skate Profile**.
 - Click Save.

Exercise 3: Creating the Chassis Profile

In this exercise, you'll create the chassis profile using construction lines, the **Parallel Line** tool, **3-Pt Arc** tool, the **Mirror** tool and the **Single Line** tool.

1. Create a horizontal stroke construction line through the origin.
 - Hold down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh). The pointer becomes the stroke pointer (⌘).
 - Place the stroke pointer over the origin and drag to the left.

A horizontal construction is created which will serve as the base construction for the inline skate chassis layout.

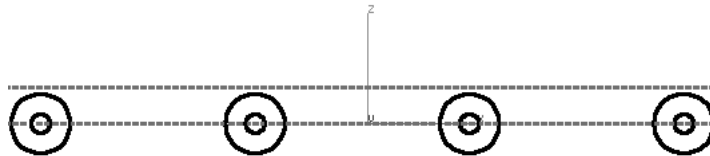


2. Create a new horizontal construction line offset from the axle center, 15 mm above the first construction line. This line will be the actual layout line used for the remainder of these exercises.

- In the **Line** tool palette, select the **Parallel Line** tool.



- Place the pointer, shown at the right, over the construction line and drag a new line up a short distance from the original line. ⇔
- In the Offset data field of the Status Line, type **15** and press ENTER (Windows) or RETURN (Macintosh).

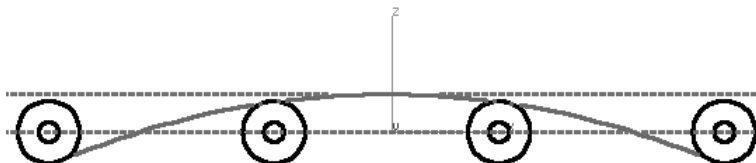


3. Create an arc, tangent to the left-most 25 mm circle, the upper construction line and the right-most 25 mm circle. This creates the shape for the bottom of the chassis.

- In the **Arc** tool palette, select the **3-Point Arc** tool.



- Hold down the CTRL key (Windows) or the OPTION key (Macintosh).
- Click the left-most 25 mm circle, the upper construction line and the right-most 25 mm circle.



By holding down the CTRL key (Windows) or OPTION key (Macintosh) you create an arc tangent to the circles and construction line.

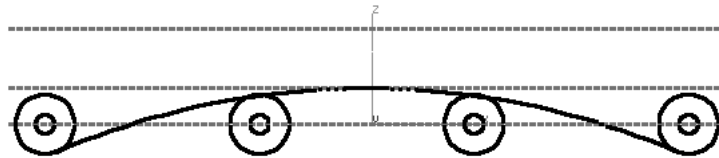
4. Create a new horizontal construction line offset from the axle centers, 40 mm above the first construction line. You'll use this line as a reference for setting the height of the chassis.

- In the **Line** tool palette, select the **Parallel Line** tool.



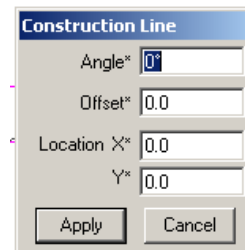
Notice that you did not have to open the subpalette to select the tool. When you select a tool, it becomes the displayed tool in the palette.

- Place the pointer over the first construction line (through the center of the axle holes) and drag a new line up a short distance from the original line.
- In the Offset data field of the Status Line, type **40** and press ENTER (Windows) or RETURN (Macintosh).



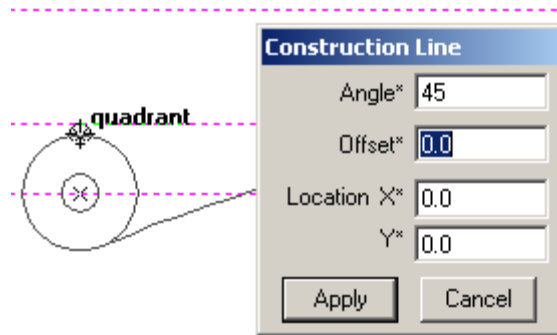
5. Draw a tangent line from the outer edge of the left-most 25 mm circle to the upper construction line at a 45° angle.

- Choose **Layout>Construction** to display the Construction Line dialog box.



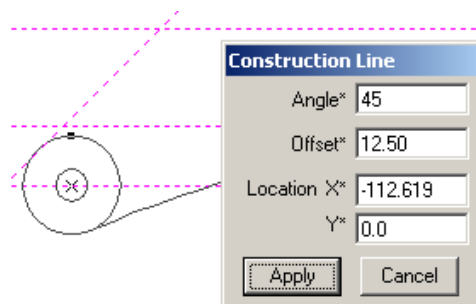
- Click on the Angle name to activate its data field.
- Type **45**.

- Click on the Offset name to activate its field.
- Move the pointer to the top quadrant of the left outer circle such that a *quadrant* notation appears.



Notice that the pointer has become a target cursor indicating that you can click in the drawing area and have the values entered automatically.

- Drag the pointer from this *quadrant* point to the *center* point of the circle. (12.50 should appear in the offset field).
- Click on the Location X name to activate its field.
- Click on the center of the circle.
- Click Apply in the dialog box and a construction line appears tangent to the outer circle and that intersects the upper construction line.

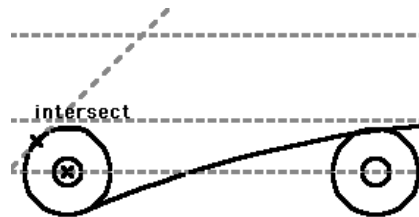


- Click Cancel to close the Construction Line dialog box.

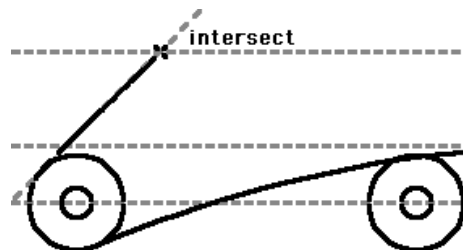
- In the **Line** tool palette, select the **Single Line** tool.



- Move the pointer to the intersection of the outer circle and the 45° construction line such that an *intersect* notation appears.



- Click to place the beginning point on the circle.
- Move the pointer to the intersection of the 45° construction line and the upper construction line such that an *intersect* notation appears.

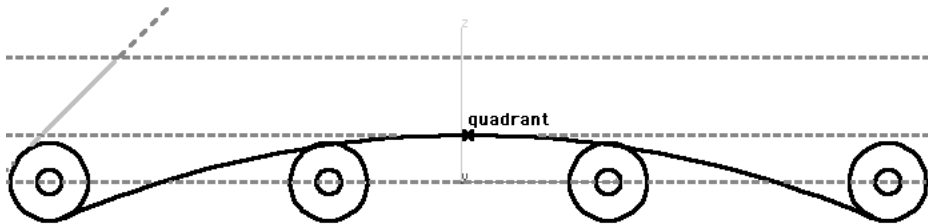


- Click to place the endpoint of the line at the noted intersection.
6. Using the arc's middle quadrant point as a reference, mirror a copy of the tangent line to the right side. This will create a new tangent line touching the rightmost circle.
 - In the **Transformation** tool palette, select the **Mirror** tool.



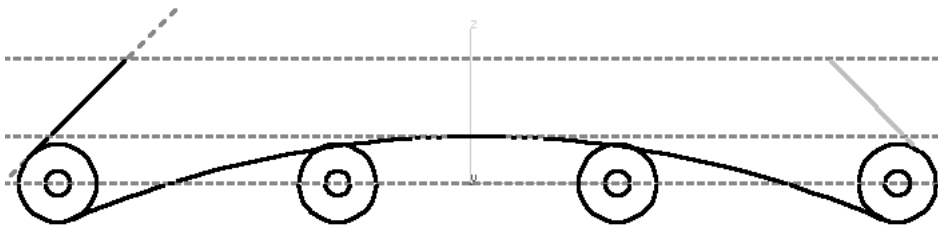
Make sure that the single line created in the previous step is still selected.

- Move the pointer over the center of the arc such that the *quadrant* notation appears and click to set the beginning of the reference line.



- Hold down the CTRL key (Windows) or OPTION key (Macintosh).
- Move the pointer vertically down from the previous point, such that a *perpendicular* notation appears.
- Click to set the end of the reference line. (If you have difficulty with this due to axis interference, choose **Layout>Hide Axis**.)

A mirrored copy of the tangent line appears.



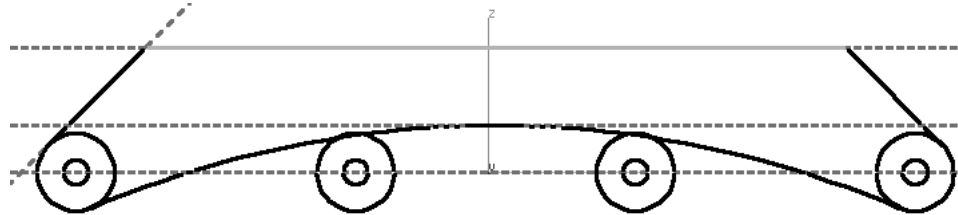
7. Draw a line between the top endpoints of the two tangent lines.

- In the **Line** tool palette, select the **Single Line** tool.



- Click the first point of the line at the top endpoint point of the left tangent line.

- Click the endpoint of the line at the top endpoint of the right tangent line.



8. Delete the construction lines. They are no longer needed.
 - Choose **Layout>Delete Constructions** to delete the construction lines.
9. Save the file.
 - Choose **File>Save** or CTRL+S (Windows) or ⌘+S (Macintosh).

You've now created the basic chassis profile.

Exercise 4: Editing the Profile

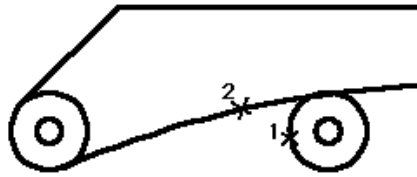
In this exercise, you'll edit the profile using the **2-Entity Fillet** tool and the **Simple Trim** tool.

1. Fillet the two inner-most 25 mm circles between the circle and the arc with 15 mm fillets without trimming the circles.
 - In the **Fillet/Chamfer** tool palette, select the **2-Entity Fillet** tool.



- In the Radius data field of the Status Line, type **15**.
- Hold down the CTRL key (Windows) or the OPTION key (Macintosh) to prevent trimming of the line, as mentioned in the Message Line.

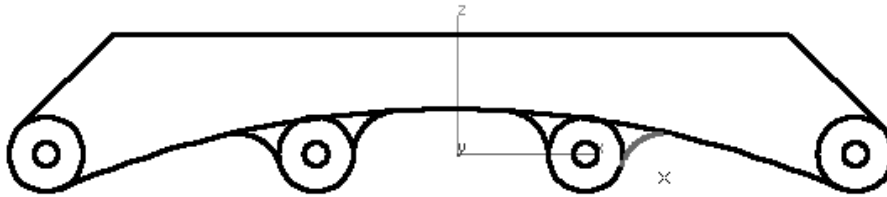
- Click on the left side of the first of the two inner-most circles (point 1 in the graphic) to select the first fillet entity.



- Click on the arc to the left of that 25 mm circle (point 2 in the graphic) and the first fillet is created.

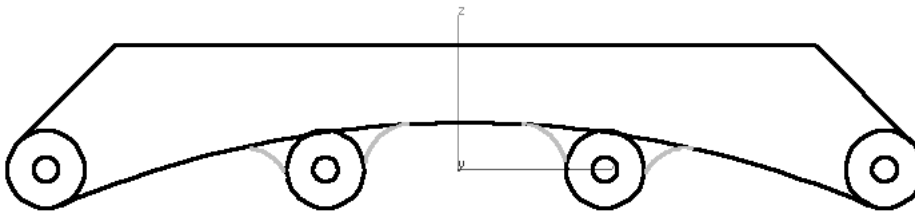


- Fillet the other side of the circle.
- Now fillet both sides of the other 25 mm circle.

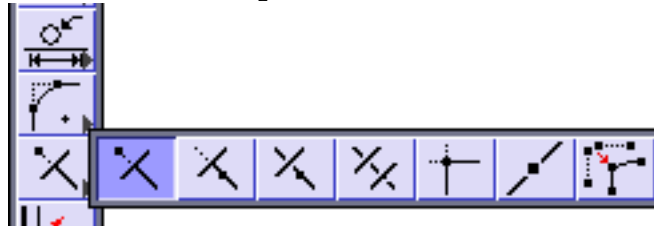


2. Trim the arc between the fillets on the left innermost circle and between the fillets on the right innermost circle.

- Choose the **Selection** tool.
- Select the four fillets you just created.

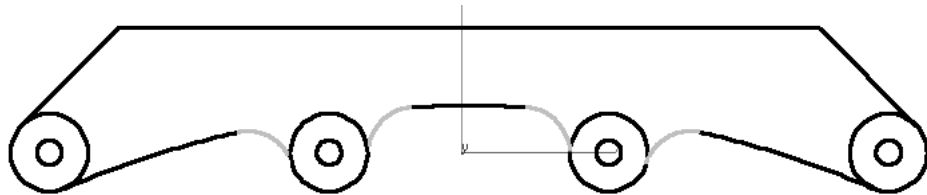


- Choose the **Simple Trim** tool from the **Curve Extras** tool palette.

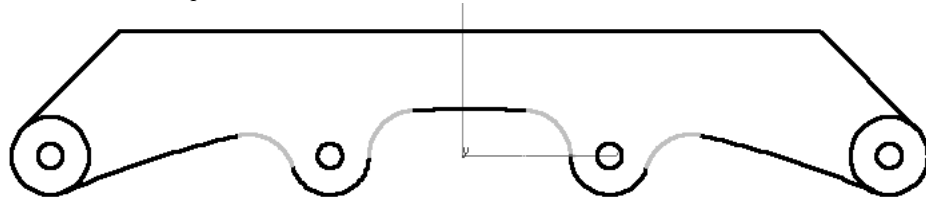


- Click on the arc section between the fillets for the left innermost circle and the arc section between the fillets for the right innermost circle.

The arcs are trimmed.

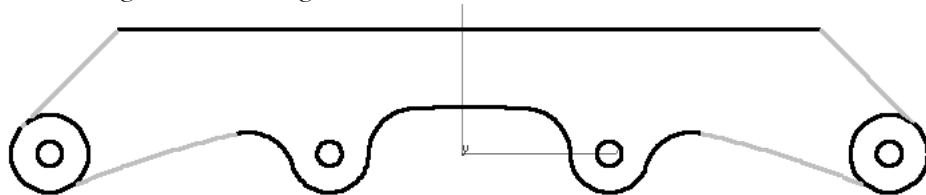


3. Now trim the portion of the inner circles between these same fillets.



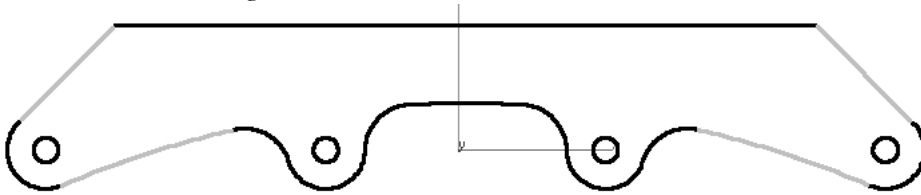
4. Trim the remaining portion of the left and right outermost circles to complete the wireframe.

- Choose the **Selection** tool.
- Hold down the SHIFT key and select the two tangent lines and the left-most and right-most arc segments.



- Choose the **Simple Trim** tool again.

- Trim the circle segments.



5. Click away from the geometry to deselect everything.
6. Save the file as, *Inline Skate Profile*.
 - Choose **File>Save**.

Congratulations! You have now completed the wireframe profile for the inline skate chassis. In the next chapter, you'll turn this 2D geometry into a 3D part.

Solid Modeling

A solid model is a 3D representation of an object with its inside and outside material defined mathematically. Unlike wireframes and surfaces, which define lengths and areas, solid models accurately provide physical information about objects. They can also generate stereolithography models, useful rapid prototyping concepts, or create forms for molds or castings.

In these exercises, you'll continue designing the inline skate chassis using the 2D profile created in the last chapter to produce the following solid model:



In this chapter you'll learn how to:

- Extrude a profile

- Set up a mounting platform on the chassis
- Create a platform and a chassis cutout
- Perform basic and advanced photorealistic rendering
- Calculate the chassis' physical properties

Exercise 1: Extruding the Chassis Profile

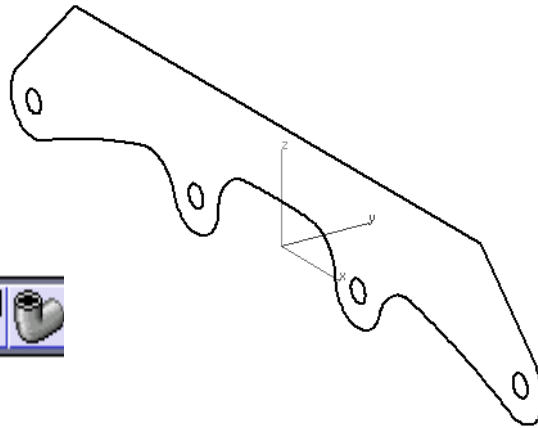
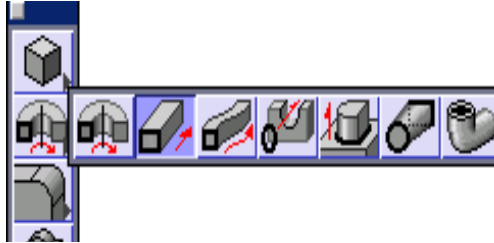
In this exercise you will extrude the profile using the **Extrude Solid** tool.


1. Open the file, *InLine Skate Profile*, if it is not already open.
2. Save the file as *InLine Skate Solid*.
 - Choose **File>Save As**.
 - In the Save document as dialog box, type **Inline Skate Solid**.
 - Click Save.
3. Choose **Window>Trackball** to display the Trackball.
4. Choose **Window>Surfaces** and **Window>Solids** if the **Surfaces** and **Solids** tool palettes are not displayed under the main tool palette.



- From the Trackball menu, choose Trimetric.

- From the **Profiles** tool palette, choose the **Extrude Solid** tool.

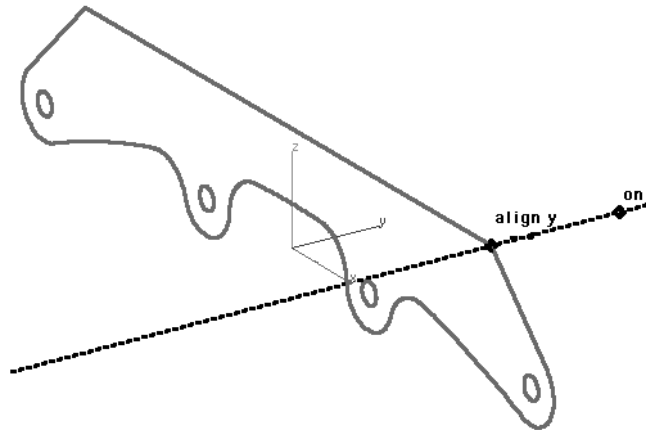


 The pointer becomes a hollow selection arrow.

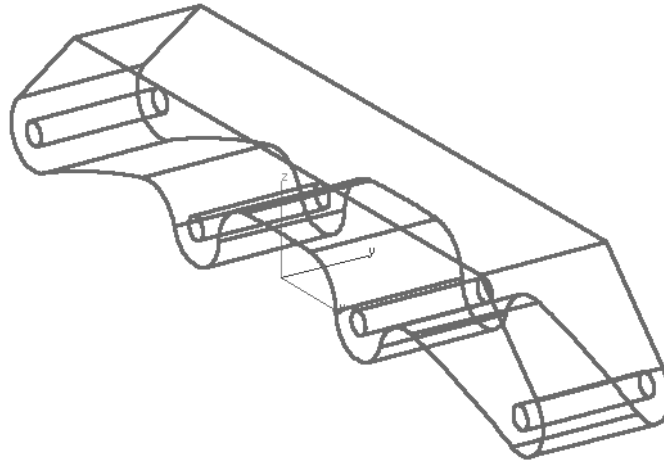
- Drag a selection fence around the entire chassis profile to select it. The hollow selection arrow becomes an bulls-eye pointer.



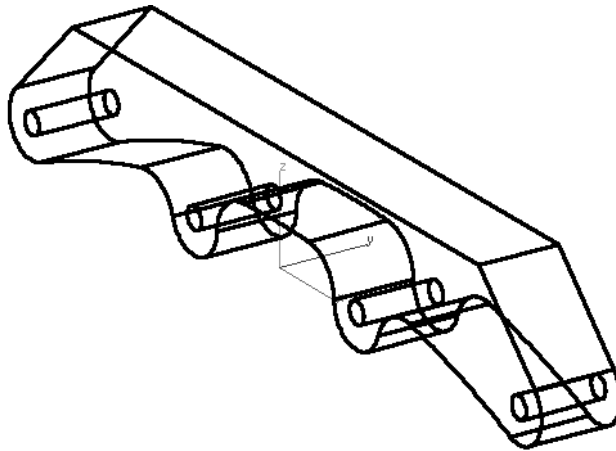
- Click on a point on the profile. You are specifying the beginning point of the extrusion.
- Move the pointer along the Y axis, using the Drafting Assistant as a guide.



- Click along the Y axis (it doesn't matter where) to finish defining the extrusion direction and length.



- In the highlighted Distance data field of the Status Line, type **29.5** and press ENTER (Windows) or RETURN (Macintosh).



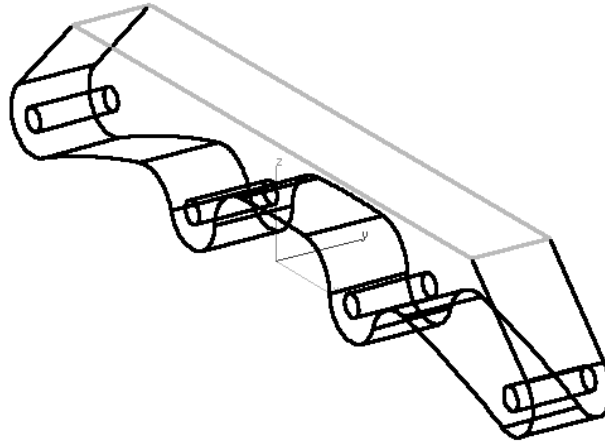
- Save the file.

Exercise 2: Setting Up the Mounting Platform

In this exercise you'll set up the mounting platform for the skate using the **Extrude Solid** tool, the **Infinite Plane** tool, the **Split Solid** tool, the **Union Solid** tool and the *Show/Hide* command.

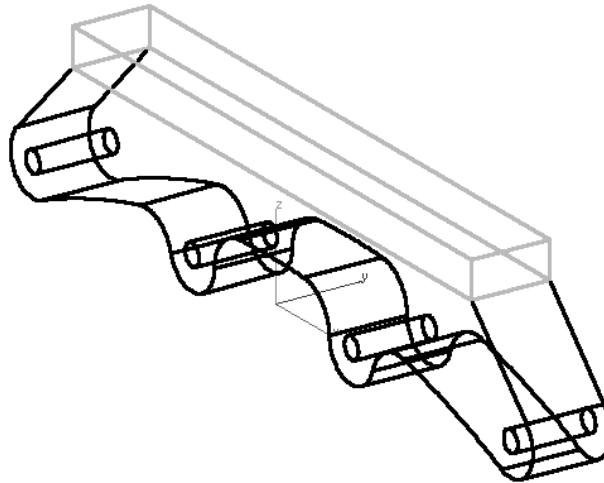
1. Extrude the top face of the chassis up 8 mm with a 10° draft.

- With the **Extrude Solid** tool still active, select the top face of the chassis.
- Click a point on the edge of a face.
- Move the pointer along the Z axis and click some distance above the first point.



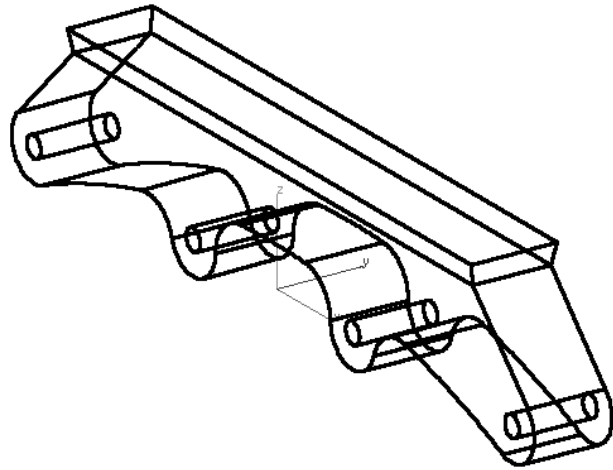
The top face extrudes.

- Type **8** in the highlighted Distance data field of the Status Line and **10** in the Draft Angle field. (Use the TAB key to move between fields.)
- Press ENTER (Windows) or RETURN (Macintosh) to update the extrusion.



You've created an 8 mm platform.

If you can't see the entire chassis, use the scroll bar to adjust the drawing area.

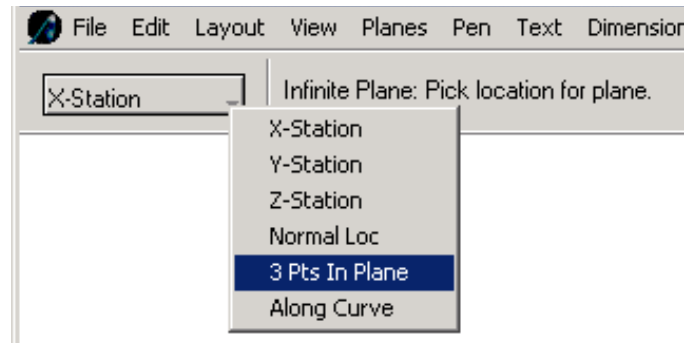


2. Trim the 8 mm extrusion to share the same plane as the sloped face on the front and back of the chassis.

- In the **Surfaces** tool palette, choose the **Infinite Plane** tool.

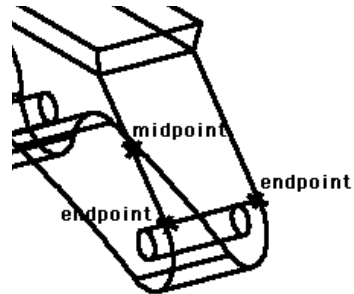


- In the Message Line, choose the *3 Pts In Plane* option from the pull-down menu.

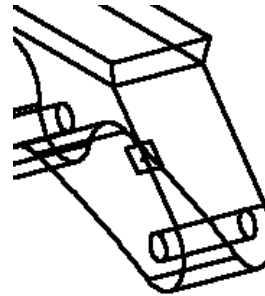


- Click three points on the sloped face to create an infinite plane at the same angle as the face.

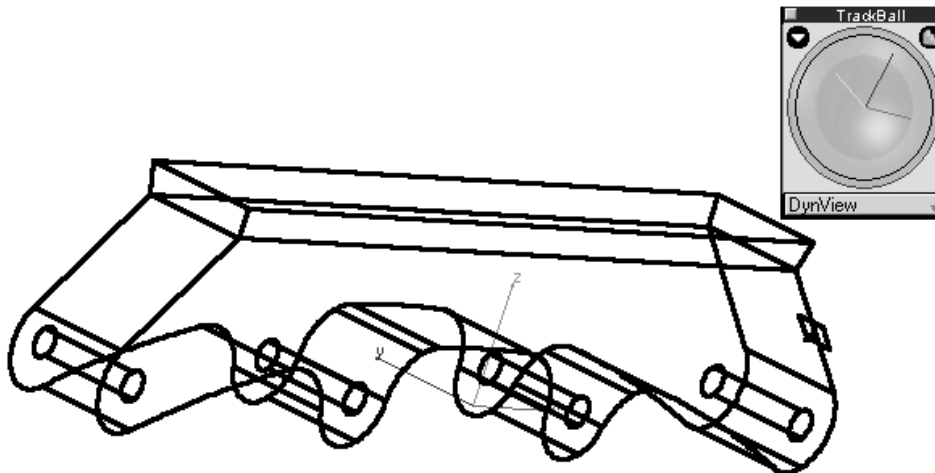
In the graphic here, the midpoint of the left edge, bottom left and bottom right endpoints were selected.



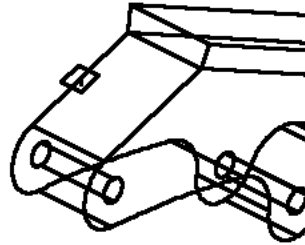
The infinite plane is created at the angle of the sloped face.



- Using the Trackball, rotate the chassis so that the back sloped face is visible.



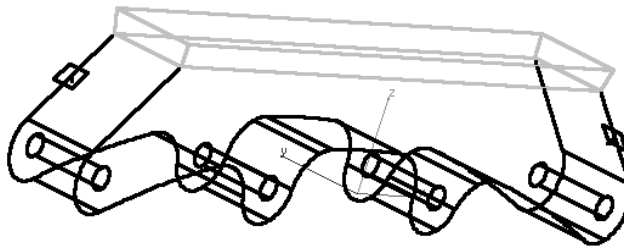
- Click three points on this face to create an infinite plane.



3. Split the 8 mm platform using the infinite plane on both sides of the chassis.
 - In the **Solids Utilities** tool palette, choose the **Split Solid** tool.



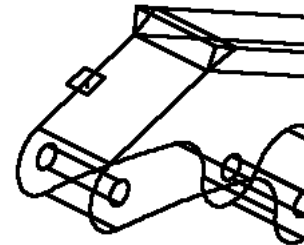
- Select the 8 mm platform you created in step 1.



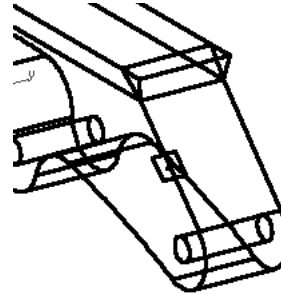
- Select the infinite plane on this back face of the chassis.

The 8 mm platform is split by the plane.

- In the Trackball menu, change the view to Tri-metric.
- Select the 8 mm platform again.



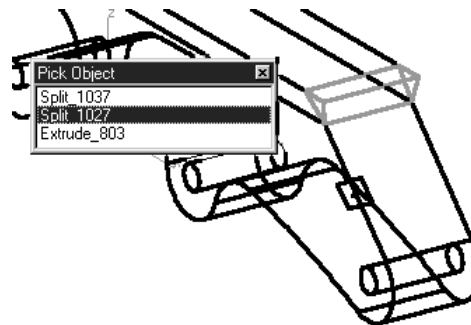
- Select the infinite plane on this side of the chassis and the 8 mm platform is split by the plane.
4. Hide the two split portions of the platform and the two infinite planes.
- Choose **Window>Show-Hide**.



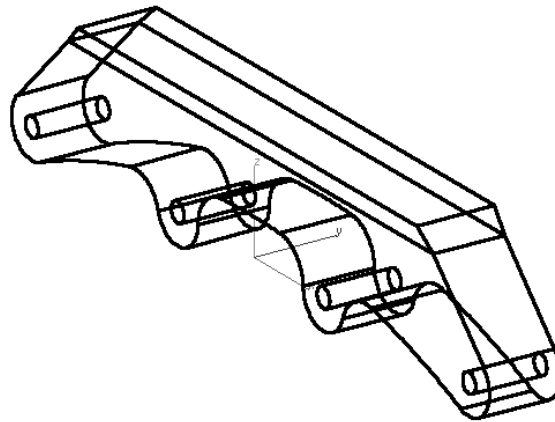
- Select *Hide* in the Show-Hide window.
- Select the small split portion of the platform.

The split portion hides.

- Select the infinite plane to hide it.
- Rotate the chassis to display the back side and hide the split portion of the platform and the infinite plane.



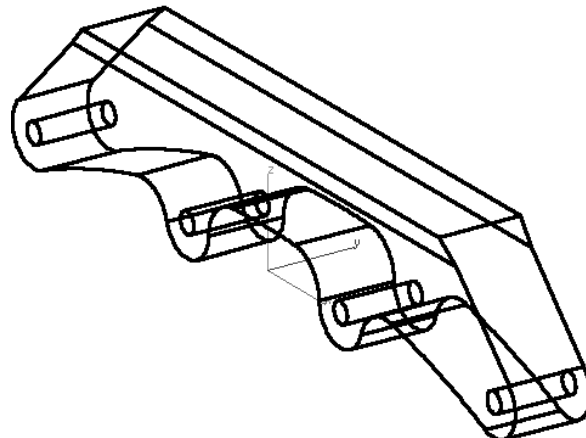
- In the Trackball menu, choose Trimetric.



- Close the Show-Hide dialog box by clicking on the close button in its title bar.
5. Union the 8 mm platform with the rest of the chassis.
 - In the **Solids Utilities** palette, choose the **Union Solid** tool.



- Select the chassis.
 - Select the 8 mm platform.
- The two objects union into one solid.
6. Save the file.



Exercise 3: Creating a Platform Cutout

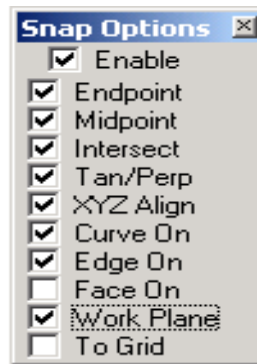
In this exercise, you'll create a cutout through the chassis using the *Snap Options* command, the **Parallel Line** tool, the **3 Pt Conic** tool, the **Single Line** tool, the **Sweep Surface** tool, the **Split Solid** tool and the *Show/Hide* command.

1. Use the Trackball to display the Front view.

This automatically sets the plane to Front. If the chassis does not completely display on your screen, choose **View>Zoom All**.

2. Choose **Window>Snaps**.

The Snap Options dialog box displays.



3. Select the *Project to Work Plane* option and a check mark appears.

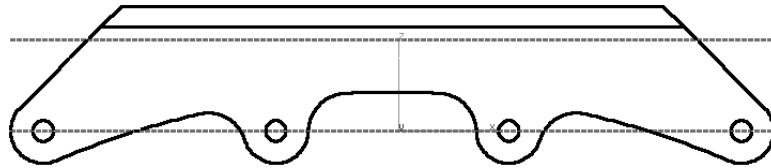
By choosing this option you can create curves in one 2D plane while using the chassis as a reference.

4. Close the dialog box.
5. Create a horizontal stroke construction line through the axle centers.
 - Hold down the SHIFT+CTRL keys (Windows) or the ⌘ key (Macintosh). The pointer becomes the Stroke pointer (⌘).
 - Place the Stroke pointer over an axle center and drag to the left.
 - A horizontal construction is created.
6. Create a new horizontal construction line offset from the axle center, 35 mm up.
 - In the **Line** tool palette, select the **Parallel Line** tool.

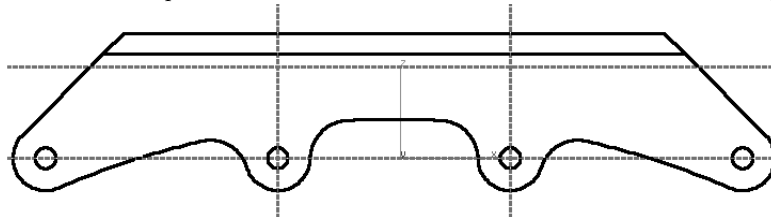


- Place the pointer over the construction line and drag a new line up a short distance from the original line.

- Type **35** in the Offset data field of the Status Line and press ENTER (Windows) or RETURN (Macintosh).

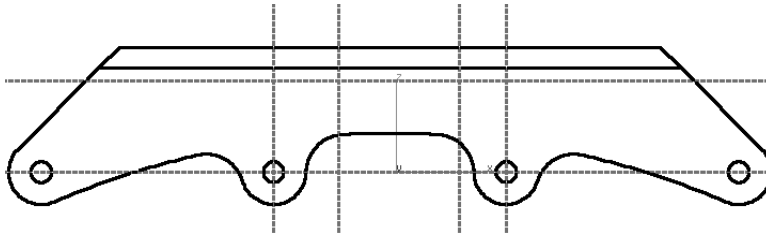


7. Create two vertical construction lines through the centers of the two inner wheel axles.
 - Hold down the SHIFT+CTRL keys (Windows) or the **⌘** key (Macintosh).
 - Place the Stroke pointer over each axle center for each axle and drag up.



8. Using the **Parallel Line** tool, create two additional vertical construction lines. Place one 25 mm to the right of the left vertical construction line. Place another 18 mm to the left of the right vertical construction line.
 - Place the pointer over the left vertical construction line.
 - Drag to the pointer to the right.
 - In the Offset data field of the Status Line, type **25** mm and press ENTER (Windows) or RETURN (Macintosh).
 - Place the pointer over the right vertical construction line.
 - Drag to the pointer to the left.

- In the Offset data field, type **18 mm** and press ENTER (Windows) or RETURN (Macintosh).

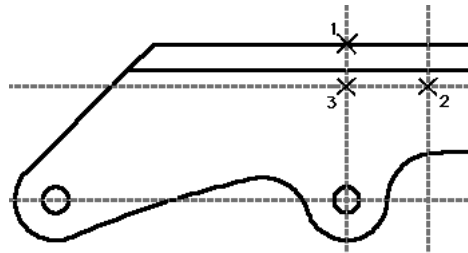


9. Create two conics using the constructions lines as a guide.

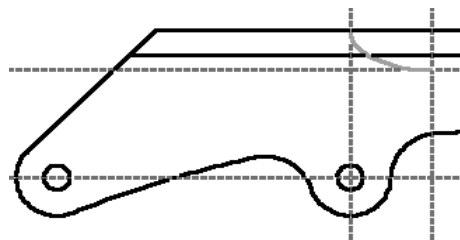
- In the **Ellipse/Conic** tool palette, choose the **3-Pt Conic** tool.



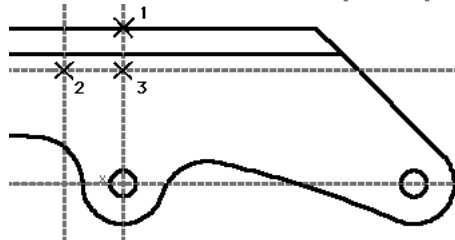
- Click the points for the left conic in the order (1, 2, 3) indicated in the graphic.



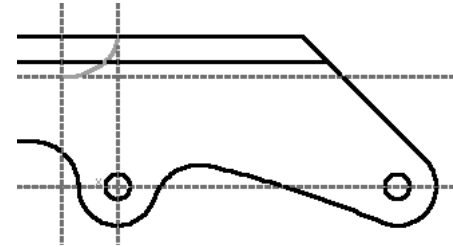
The conic appears.



- Click the points for the right conic in the order (1, 2, 3) indicated in the graphic.



A second conic appears.

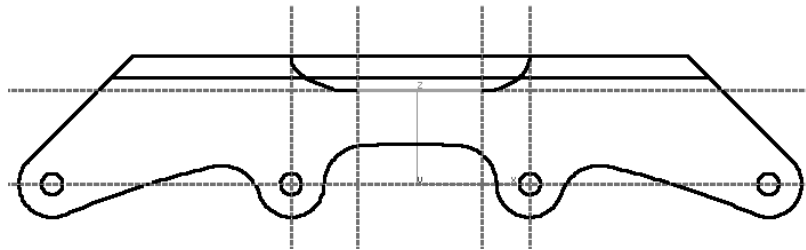


10. Connect the bottom of the conics with a line.

- In the **Line** tool palette, select the **Single Line** tool.



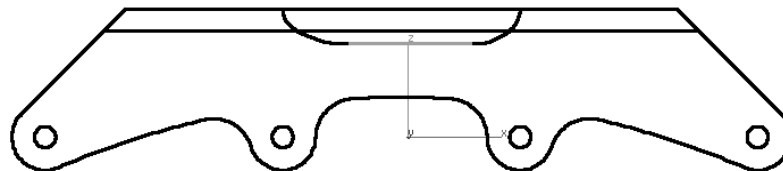
- Place the first point of the line at the lower point of the first conic.
- Place the end point of the line at the bottom of the lower point of the second conic.



11. Delete the construction lines.

- Choose **Layout>Delete Constructions**.

The construction lines are deleted.



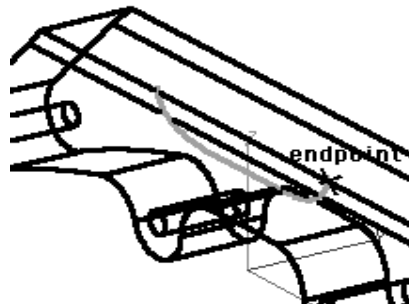
12. Turn off the Snap option, *Project to Work Plane*.

- Choose **Window>Snaps**.
 - In the Snap Options dialog box, turn off the *Project to Work Plane* option by clicking in the check box.
 - Close the Snap Options dialog box.
13. Now create a surface from the profile along the width of the platform.
- Change your view to Trimetric using the Trackball menu.
 - In the **Surface** tool palette, choose the **Sweep Surface** tool.

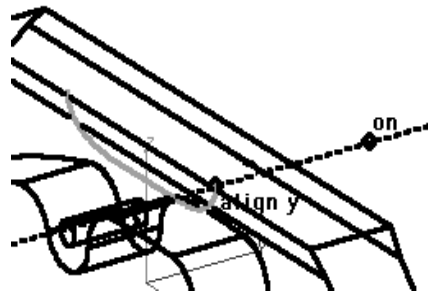


- Hold down the SHIFT key and select the two conics and the line you just created.
- The pointer becomes a bulls-eye.

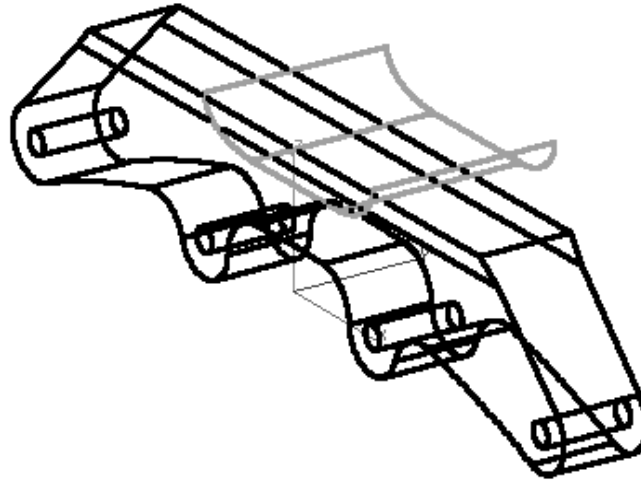
- Click on the endpoint of the conic.



- Move the pointer along the Y axis beyond the end of the chassis.
- Click to place the second point.

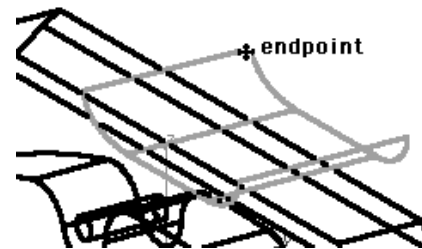


The surface is generated.

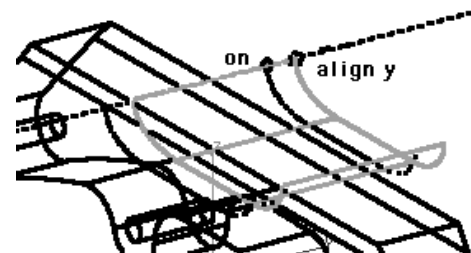


14. Drag your surface along the Y axis so that surface completely covers the top of the platform.

- Choose the **Selection** tool and select the surface.
- Place the pointer at an endpoint.



- Drag along the Y axis some distance with the help of the Drafting Assistant's dynamic construction line.



15. Split the chassis with the surface.

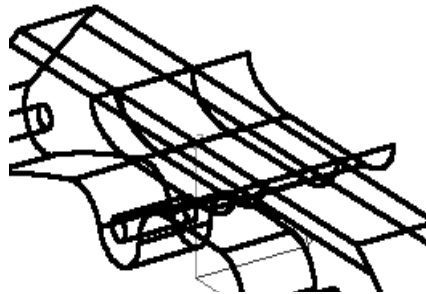
- In the **Solids Utilities** tool palette, choose the **Split Solid** tool.



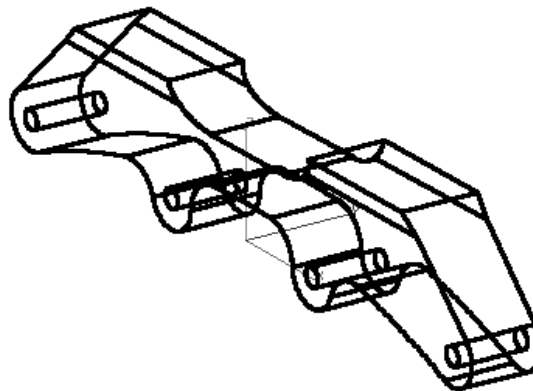
- Select the chassis.
- Select the surface and the solid is split.

16. Show only the chassis.

- Choose **Window>Show-Hide**.
- Select the *Show Only* option.



- Select the chassis.
- The curves, surface and split solid portion are hidden. Only the chassis displays.
- Close the Show-Hide window by clicking on the close button in its title bar.

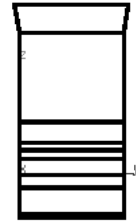


17. Save the file.

Exercise 4: Creating the Chassis Cutout

Now you're going to create a cutout through the length of the chassis where the wheels would be placed. You'll use the **Rectangle** tool, the **Move** tool, the **Cutout Feature** tool and the **Constant Blend** tool.

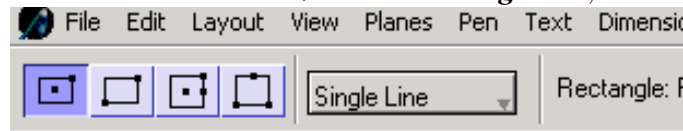
1. Switch the view to Side using the Trackball menu.
2. In the empty space to the left of the chassis, draw a rectangle 24.5 mm wide by 70 mm high.



- From the main tool palette, select the **Rectangle** tool.

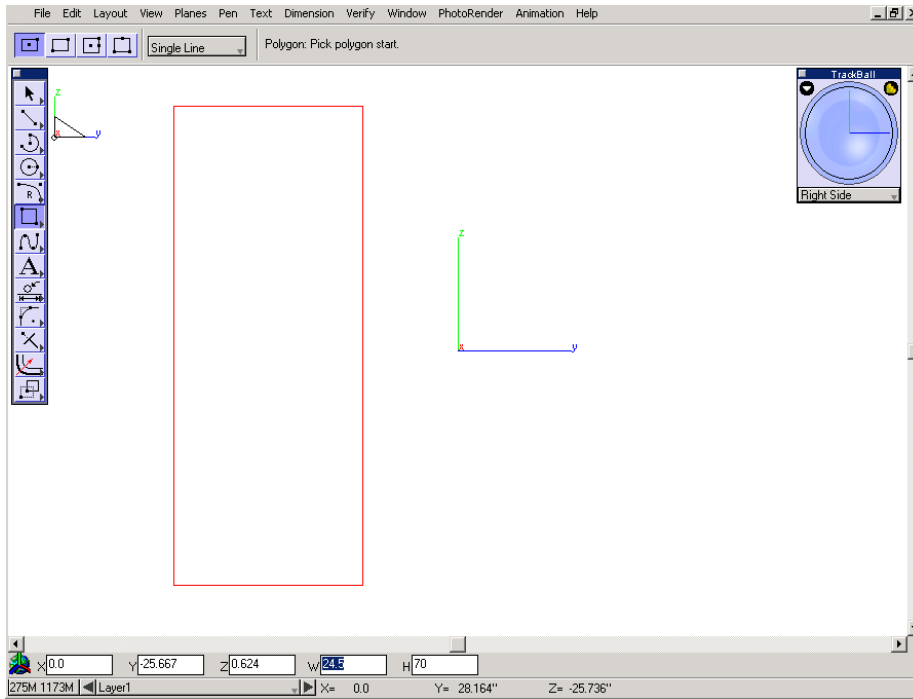


- Choose the **Center/Corner Rectangle** tool, the first tool in the subpalette.



- Move the rectangle pointer to the left of the chassis and click to place the center point of the rectangle.
- Move the pointer away from the center and click anywhere to place the corner of the rectangle.
- Type **24.5** in the highlighted W (width) data field of the Status Line.
- TAB once to activate the H or height data field.
- Type **70** into that data field
- Press ENTER (Windows) or RETURN (Macintosh).

- The rectangle resizes to these new values.



3. Move the rectangle using the midpoint of the top line and align it with the top midpoint of the chassis.
 - Select the rectangle with the **Selection** tool.
 - Place the pointer at the midpoint of the top horizontal line such that the midpoint notation appears.



- Drag the rectangle from this midpoint and align it with the midpoint of the top of the chassis.

You won't know which midpoint of chassis the rectangle snapped to but that does not matter as you'll soon see.

4. With the rectangle still selected, use the **Move** tool to move it -2.5 mm in the Z direction.

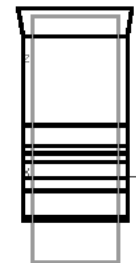
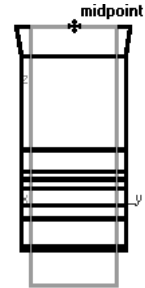
- In the **Transformation** tool palette, choose the **Move** tool.



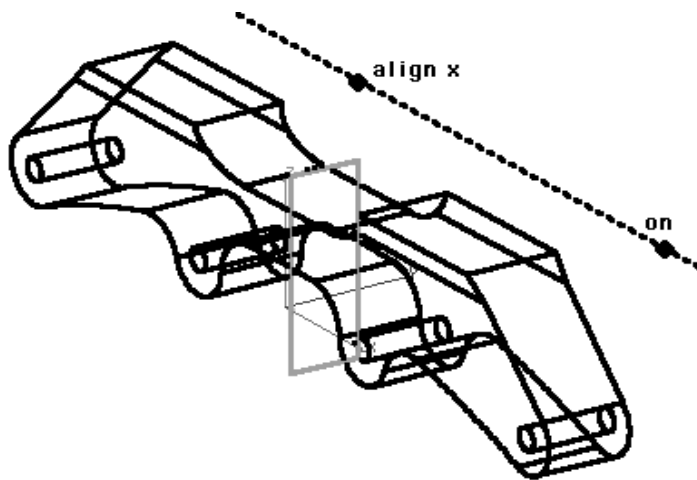
- Type **-2.5** in the dZ data field of the Status Line and press ENTER (Windows) or RETURN (Macintosh).

The rectangle moves down 2.5 mm.

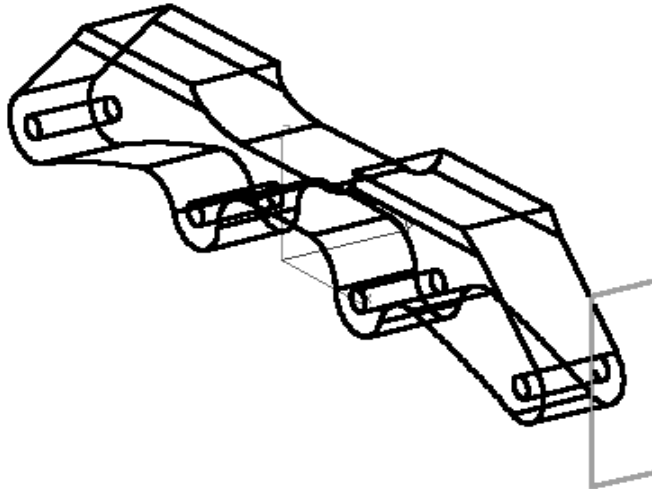
5. Change the view to Trimetric using the Trackball menu.
6. Use the **Move** tool to move the rectangle along the X axis outside of the chassis.
 - Click a point outside the chassis. It doesn't matter where because you are using this point and the next as a reference point for setting the move distance.



- Using the Drafting Assistant, click the second point far enough along the X axis such that the rectangle moves in front of the chassis.



The rectangle moves outside the chassis.



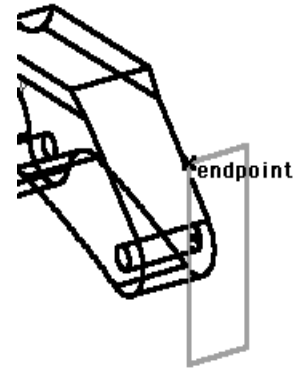
Check the Status Line to see that only the dX field contains a non-zero value. If the dY and dZ data fields contain values other than 0, type **0** in them and press ENTER (Windows) or RETURN (Macintosh).

If the rectangle is not completely within the drawing area, use the scroll bars to correct it.

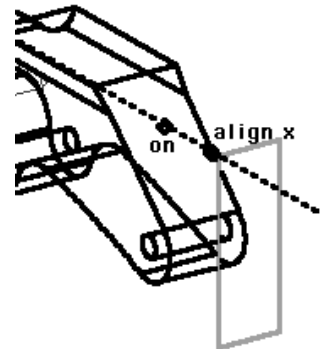
7. In the **Solids Features** tool palette, choose the **Cutout Feature** tool.



8. Select the solid chassis.
9. Drag a selection fence around the rectangle to select it.
10. Click on a top endpoint of the rectangle.



11. Click along the X axis towards the chassis to specify the direction of the cutout.



A cutout appears through the chassis to the depth shown in the Status Line.

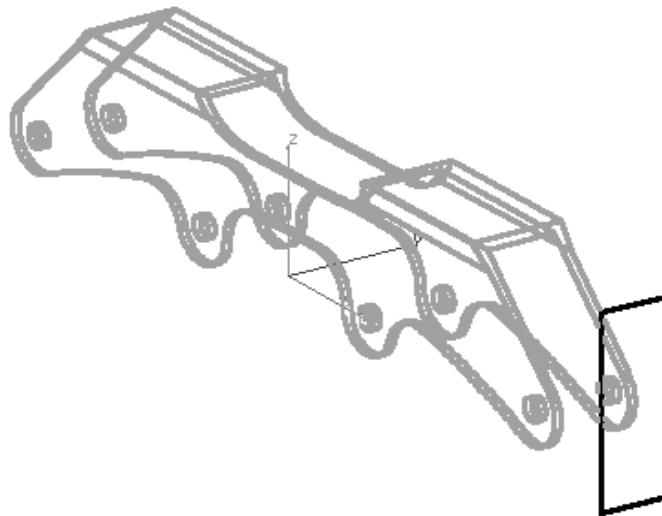
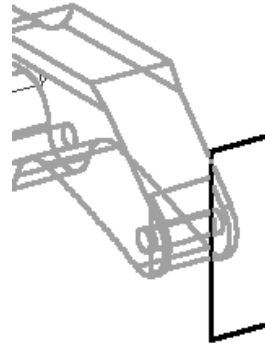
- In the highlighted Distance field of the Status Line, type **500**.

This is a value greater than the length of the chassis (300 mm) so that the cutout goes completely through the chassis. This value depends on how far away from the chassis you moved the rectangle in step 6. 500 should be long enough.

Make sure the Draft value is set to 0.

- Press ENTER (Windows) or RETURN (Macintosh).

The cutout tunnel extends through the chassis and subtracts the intersecting solid material.



- Hide the rectangular profile.
 - Choose **Window>Show-Hide**.
 - Select the *Hide* option.
 - Drag a selection fence around the rectangle to hide it.
 - Close the Show-Hide window.
- Place 3 mm blends on each side of the inside of the tunnel at each mounting plate.

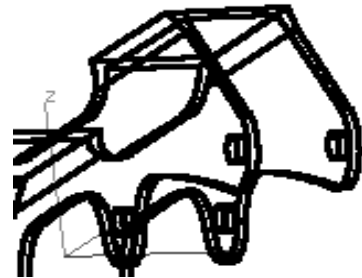
- In the **Solid Features** tool palette, choose the **Blend** tool.



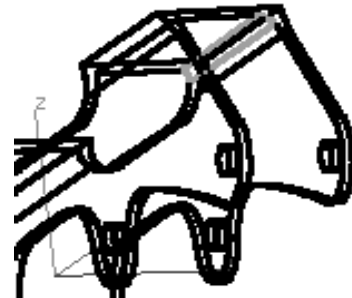
- In the Message Line, make sure the first tool, **Constant Blend**, is selected.



- Type **3** in the R (radius) data field of the Status Line.
- Rotate the chassis using the Trackball to better see the top right inside edge.



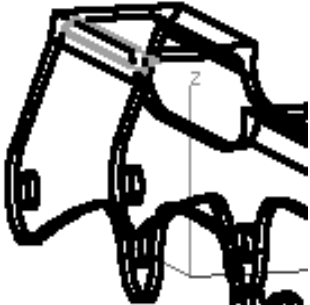
- Click on the right inside top edge of the tunnel.



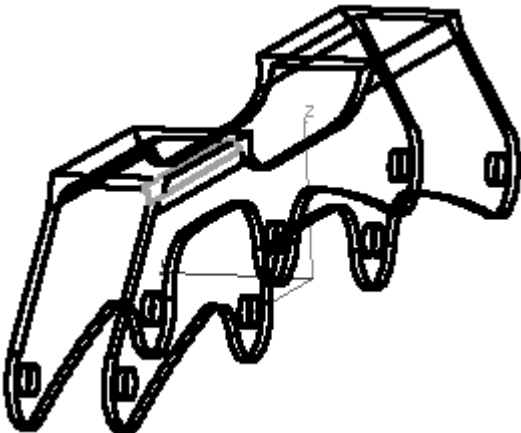
- Rotate the chassis so you can clearly see the left inside edge.



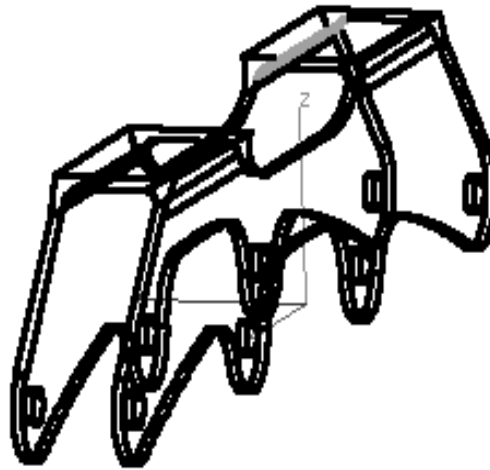
- Click on the left inside top edge of the tunnel.



- Rotate the chassis so that the back mounting plate and tunnel display.



- Click on the inside edges of the tunnel to blend these edges rotating as needed.



- Choose the **Selection** tool.
You are doing this so you do not inadvertently try to blend something else if you click in the drawing area.
- Change the view to Trimetric using the Trackball.

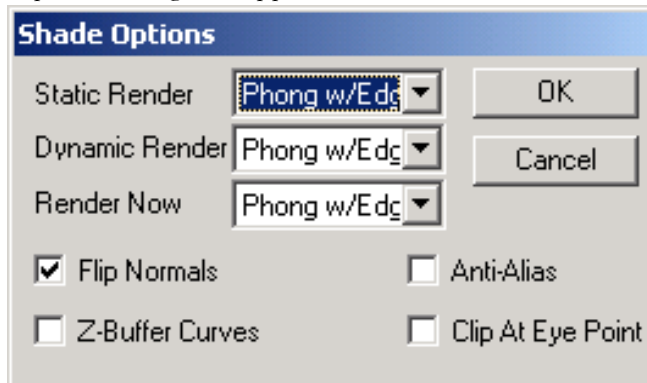
16. Save the file.

Exercise 5: Basic Rendering

You're now going to render your chassis using Ashlar's 3D modeling products (Cobalt, Xenon or Argon) basic rendering capability.

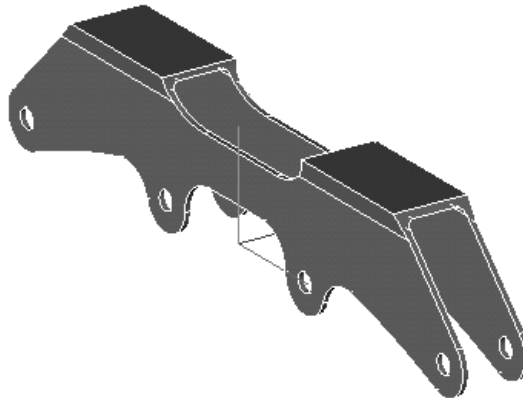
1. Choose ***View>Shade Options***.

The Shade Options dialog box appears.



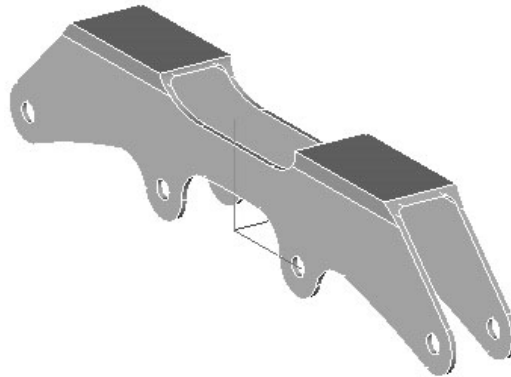
2. Use the pull-down menu to set the *Static Render*, *Dynamic Render* and *Render Now* options to Phong w/Edges.
3. Click OK to save the settings and close the dialog box.
4. Choose **PhotoRender>Preview Render(shadows off)**.

The chassis renders.



5. Change the color of the chassis using the color palette.
 - Select the chassis.

- Choose **Pen>Color** and select the desired color.



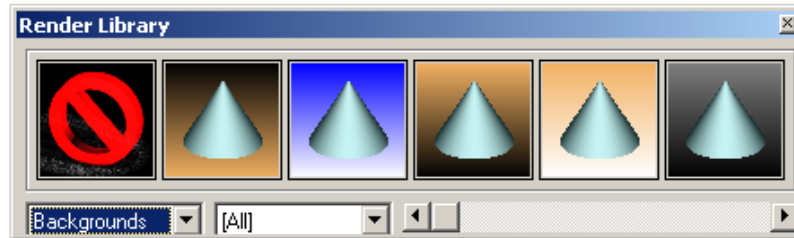
You can also rotate the chassis to see it from different angles.

Exercise 6: Advanced Rendering

You can create more realistic visual affects by applying a material and displaying your model using photorealistic rendering.

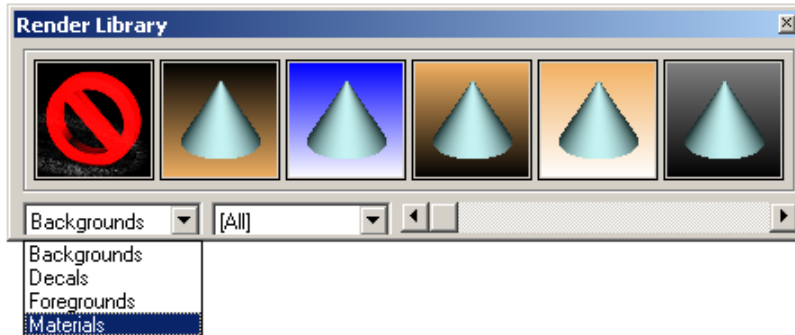
1. Choose **Window>Render Library**.

The Render Library dialog box displays on your screen.

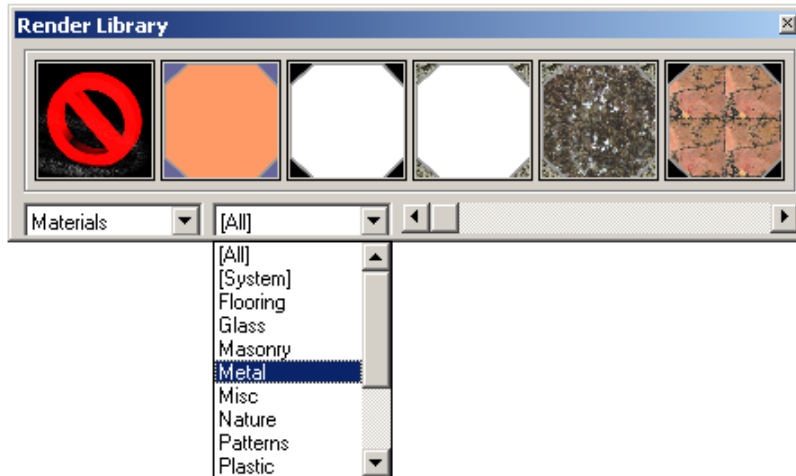


This library is divided into library types, including flooring, glass, masonry, metal and the like.

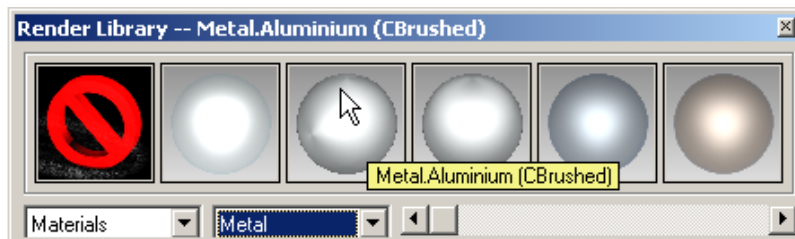
- From left most pull-down menu, Library Types, select Materials.



- From the next pull-down menu, Categories, select Metal for the material.



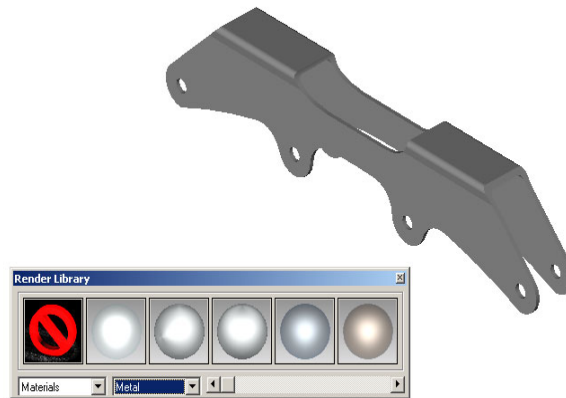
- Place the pointer over the third window. The title bar should display *Render Library -- Metal Aluminium (CBrushed)*.



5. Drag the pointer from the Render Library dialog box to the chassis. As you do so the pointer becomes an application symbol shown here.



6. Release the mouse button when the chassis becomes selected and the material is applied.



Now you want to render the chassis using photorealistic rendering.

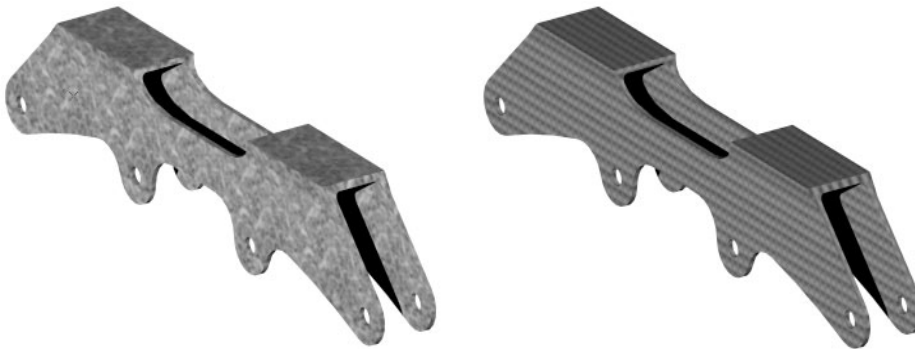
7. Choose ***PhotoRender>Raytrace Render [shadows on, Anti-Alias]***.

Ashlar's 3D modeling product redraws your chassis with the new rendering.



8. Apply different materials and textures to your chassis simply by selecting the material and dragging it to your chassis. Then view the effect by using one of the advanced rendering commands, *Preview Render [shadows off]*, *Preview Render [shadows on]*, *Raytrace Render [shadows off]* and *Raytrace Render [shadows on, Anti-Alias]*.

The left graphic here shows the chassis rendered with a bubble wrap miscellaneous material and the right graphic shows it rendered with a Knurl 2 tiled texture material.



9. Close the Render Library dialog box.
10. Save the file.

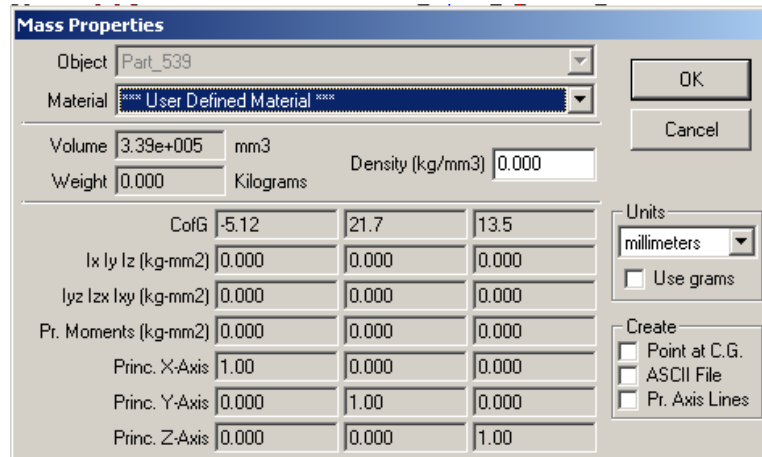
You're done. You've modeled the basic inline skate chassis in Ashlar's 3D modeling products' integrated design environment. But you're not finished yet.

Exercise 7: Calculating the Chassis' Physical Properties

You can calculate a model's physical properties using Ashlar's 3D modeling products, such as Mass Properties, area, interference between parts and other properties.

1. Select the chassis.
2. Choose **Verify>Properties**.

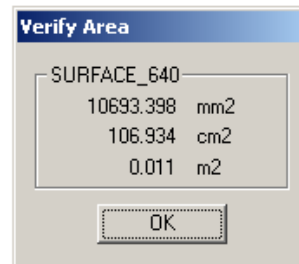
The Mass Properties dialog box appears displaying the Properties of the chassis.



Click OK to close the dialog box.

3. Choose **Verify>Area**.

The Verify Area dialog box appears displaying the area of the chassis.



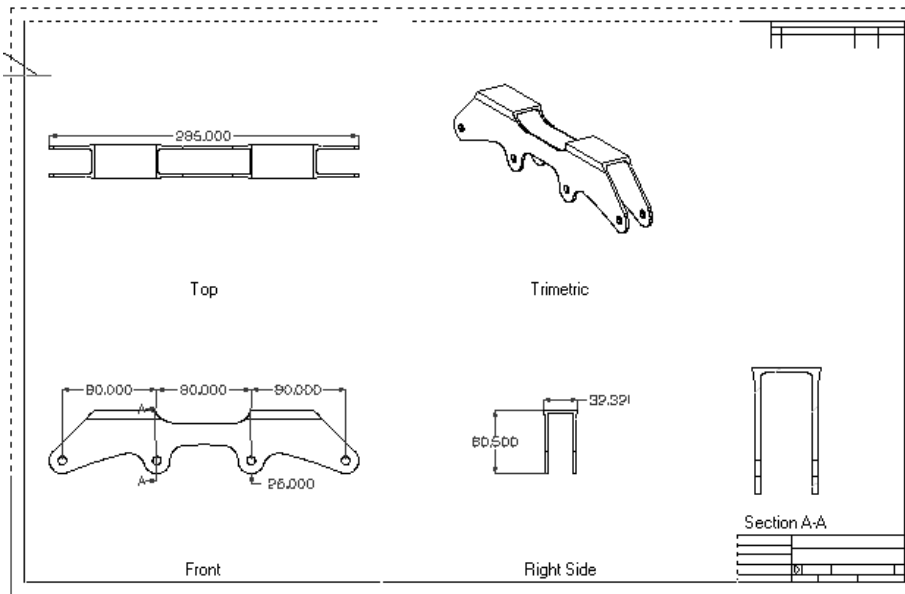
Click OK to close the dialog box. You can explore other physical properties as you wish.

4. Save the file.

You've completed the inline skate chassis. In the next chapter you produce a 2D drawing, add dimensions and create a section view.

Detail Drawing

After you've created the model, you may want to create production drawings. You probably will want to display your model at various angles, place dimensions and even create a section cut of the model. In these exercises you'll do just that for the chassis to create the following detail drawing.



In this chapter you'll learn how to:

- Create a detail drawing

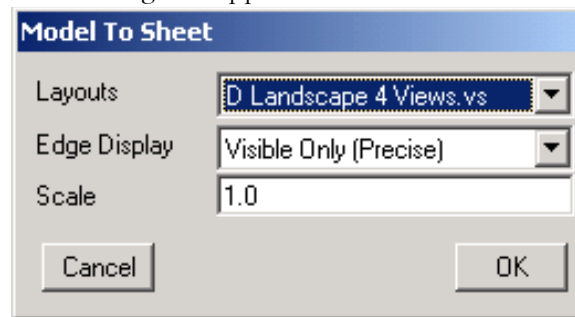
- Create a section view
- Place dimensions

Exercise 1: Creating a 2D Detail Drawing

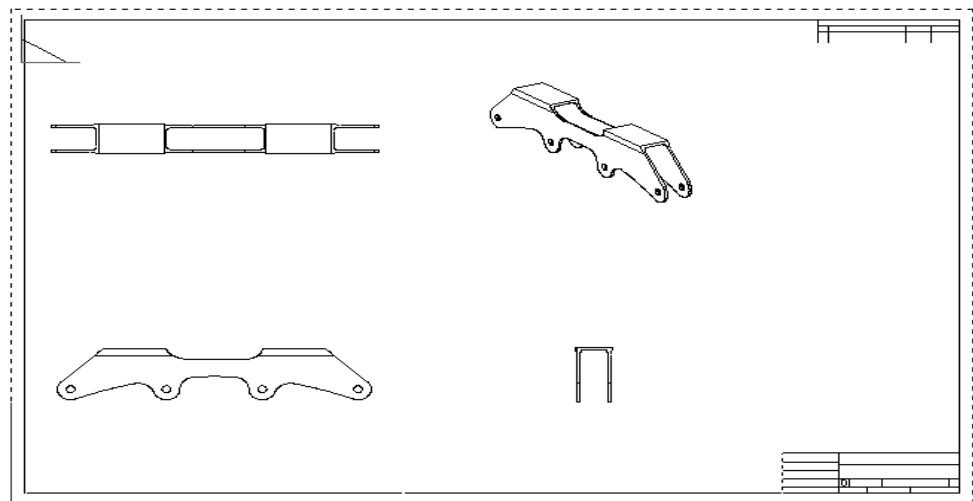
In this exercise you'll generate a 2D detail drawing using the *Model to Sheet* command.

1. Open the file, *Inline Skate Solid*, if it is not already open.
2. Select the inline chassis.
3. Choose **Layout>Model to Sheet**.

The Model to Sheet dialog box appears.



4. From the *Layouts* pull-down menu, select D Landscape 4Views.vs.



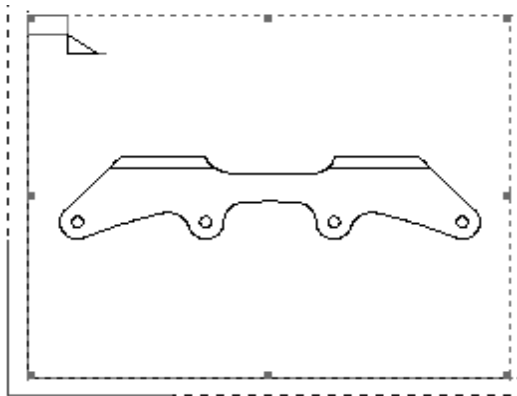
5. The *Edge Display* option should be set to Visible Only (Precise).
6. In the *Scale* data field, accept the default of **1**.
7. Click OK to close the dialog box and generate the drawing.
The detail drawing displays the chassis in Top, Front, Side and Trimetric views.
If you inadvertently selected the wrong layout, choose **Edit>Undo**.
8. Save the file as *Inline Skate Production*.
 - Choose **File>Save As**.
 - In the File name data field, type **Inline Skate Production**.
 - Click Save.

Exercise 2: Creating a Section View

The following exercise can be done only in the Cobalt™ Designer Elements program. Argon™ and Xenon™ skip ahead to exercise 3.

Now you'll add a section view to your drawing using the **Section** tool.

1. Select the Front view of the chassis.
 - Move your pointer over the Front view of the chassis.
 - Click on the chassis. The Front drawing view of the chassis activates.



Cobalt Only

2. Choose **Window>Sheet Tools**.



The **Sheet** tool palette displays in the drawing area.

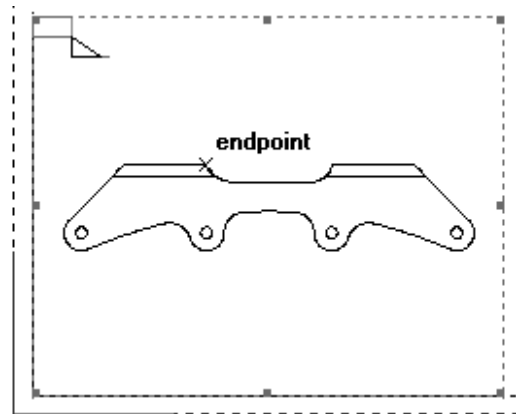
3. Choose the **Section** tool from the **Sheet** tool palette.



4. In the Message Line, make sure the **Vertical Section** tool is selected.



5. Pick the view again.
6. Click on the endpoint indicated in the graphic here.



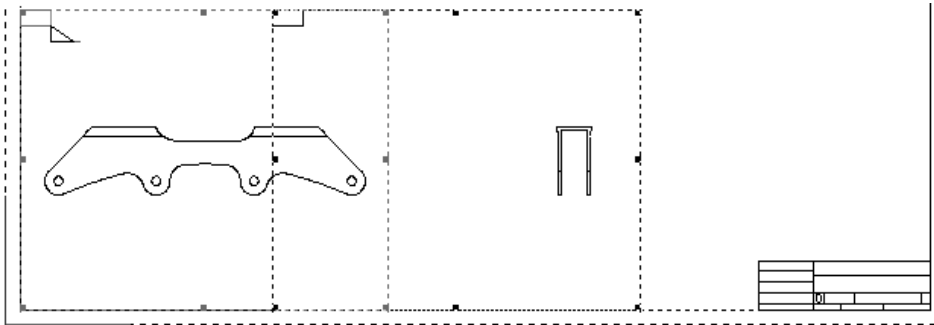
7. Drag the section view to the final position, as instructed in the Message Line.

In this case drag it to the right over the title box at the bottom of the drawing. You



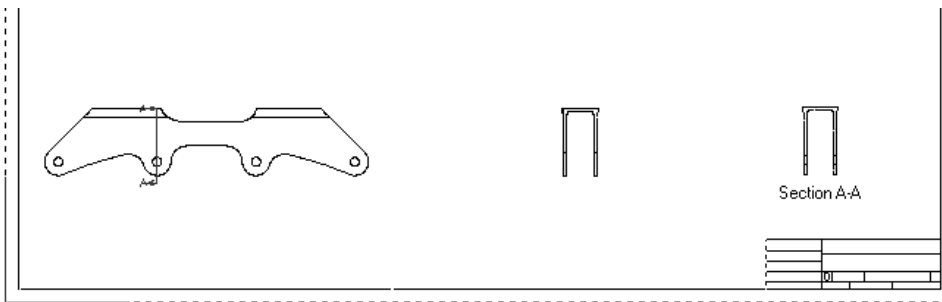
Cobalt Only

will notice that you only seem to be dragging the view boundary.



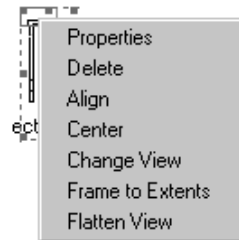
8. Click to place the final location of the section view.

The chassis section appears, labeled Section A-A. Notice that the Front view is also labeled at the section location.

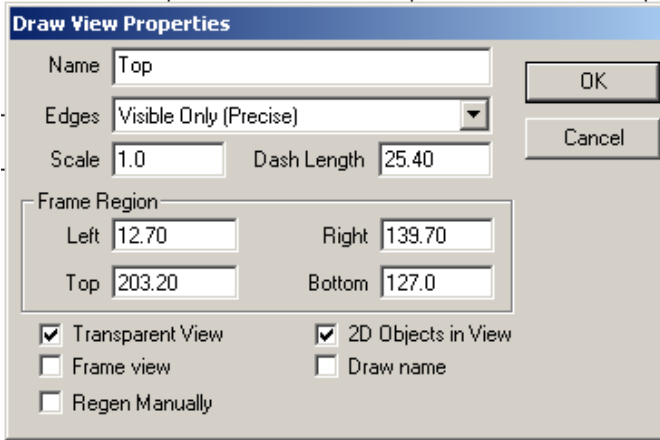


Cobalt Only

9. Increase the scale of the section view to 2.
 - Choose the **Selection** tool.
 - Select the section view by clicking on it.
 - In the upper left corner of the section view, click in the box area to display the Drawing View menu.
 - Select the *Properties* command.



The Draw View Properties dialog box displays.



Cobalt Only

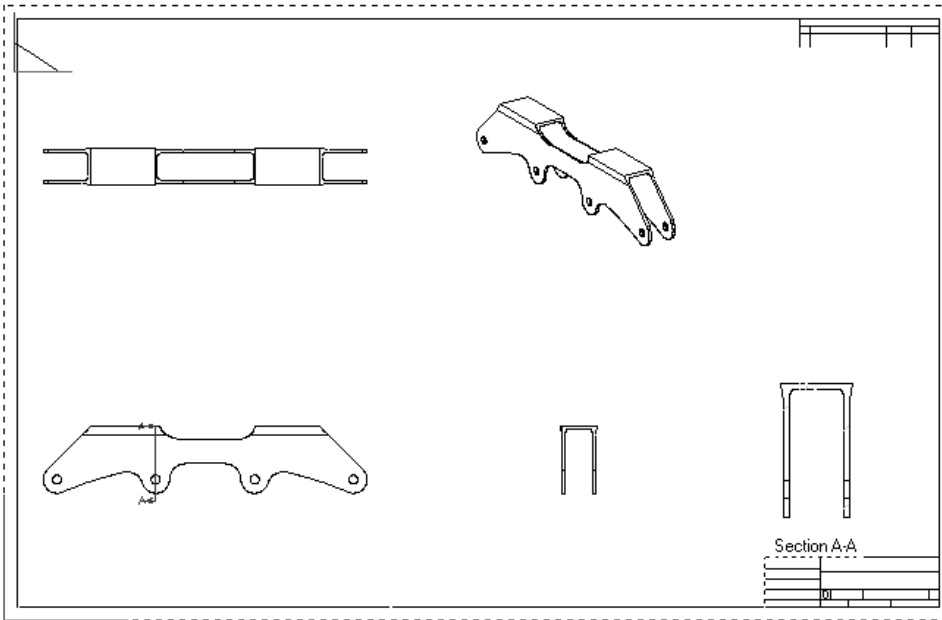
- Type **2.0** in the Scale data field and click OK to close the dialog box.

Since you now cannot see the section cut because of the scale change, you will need to resize the drawing view.

10. Click in the upper left corner of the view to display the Drawing view menu again.
11. Choose the *Frame to Extents* command.

The view resizes to frame the section cut.

- Click outside of the dotted lines of the drawing to deselect the section view.



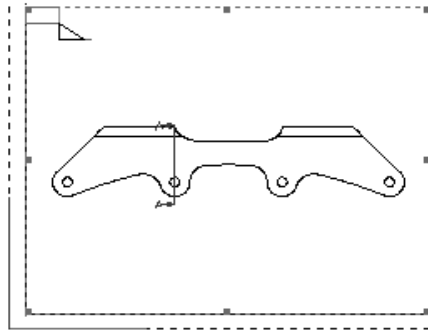
- Close the **Sheet** tool palette.
- Save your drawing.

Exercise 3: Adding Dimensions

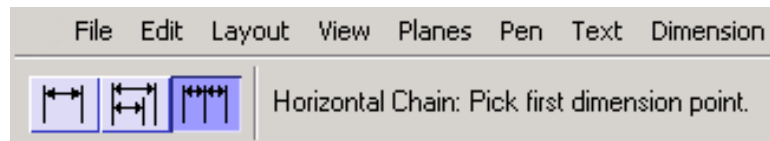
In this exercise you're going to add a few dimensions to your drawing using the **Horizontal Chain** Dimension tool, the **Dimension Arrow Out** Dimension tool, the **Horizontal** Dimension tool and the **Vertical** Dimension tool.

Tear off the **Dimension** palette from the main tool palette (to rotate the palette to the horizontal position, right click on the palette and chose "Flip Horizontal").

- Using the **Selection** tool, select the Front view.



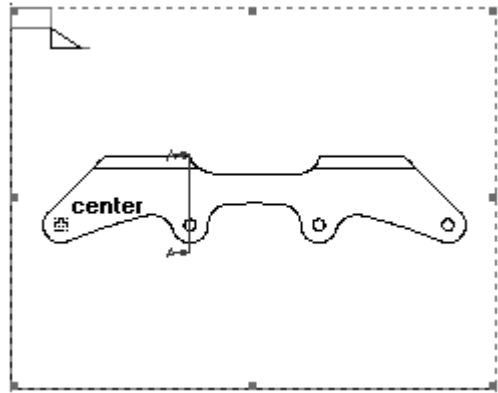
- In the Dimension palette, choose the **Horizontal Chain** Dimension tool.
 - Move your pointer to the top tool in the **Dimension** tool palette.
 - Drag the pointer to the third tool, the **Horizontal Chain** Dimension tool, to select it.



- The pointer becomes a horizontal dimension tool icon.

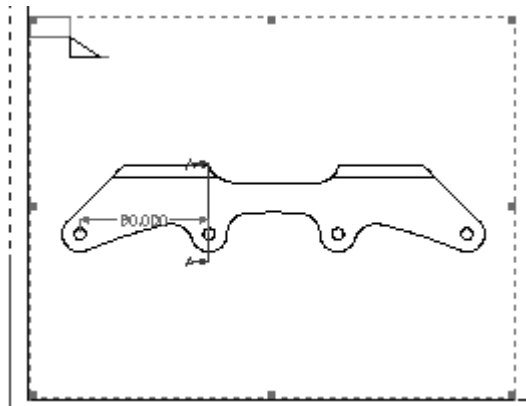


17. Move the hot spot of the pointer over to the left most axle of the chassis such that the center point notation appears.

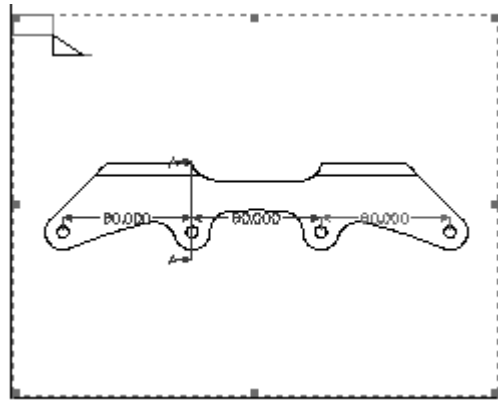


18. Click to place the first point.
19. Move the pointer to the center of the axle hole to the right and click to place the next point.

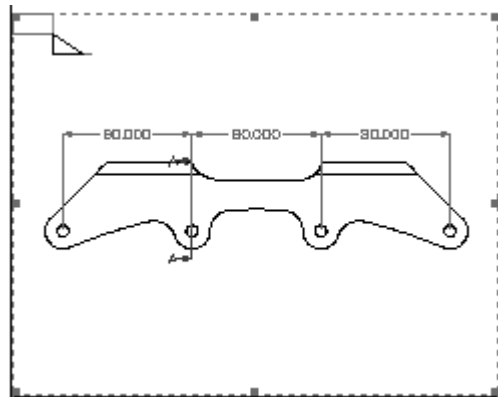
A dimension appears.



- Click at the centers of the next two axes to finish placing the horizontal chain dimension.



- Choose the **Selection** tool.
- Hold down the SHIFT key and select all three chain dimensions.
- Drag the dimensions above the chassis.

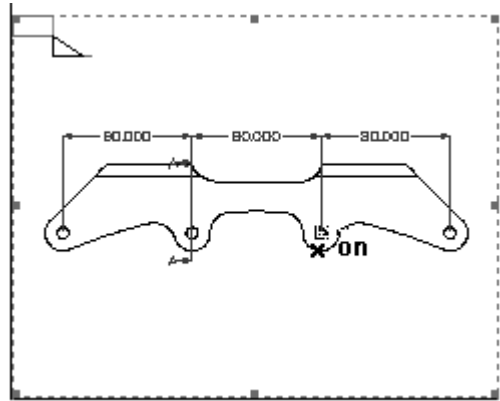


- Select the **Diameter Arrow Out Dimension** tool from the **Dimension** tool palette.

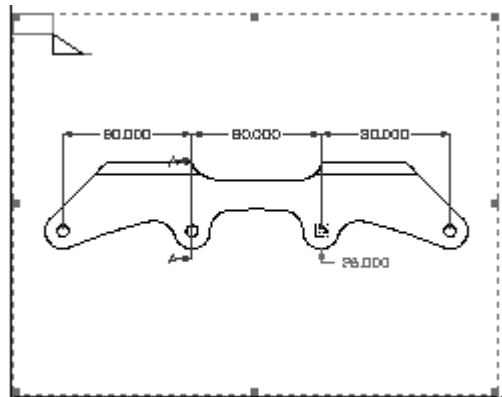
The pointer becomes the diameter arrow pointer.



25. Move the pointer to the edge of the third axle from the left such that an *on* notation appears.

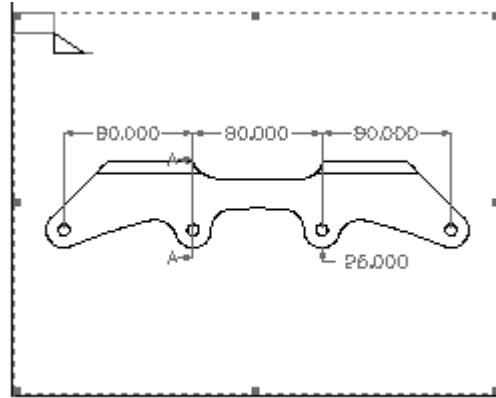


26. Click to place the dimension.

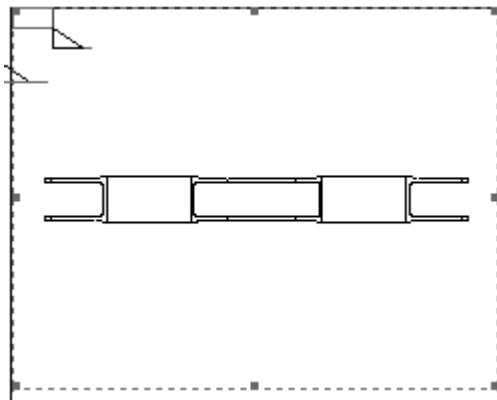


27. Choose the **Selection** tool and select the chain dimensions and the section dimension, in addition to the diameter dimension already selected.
28. Choose **Dimension>Size** and select 24.

The dimensions text resizes.

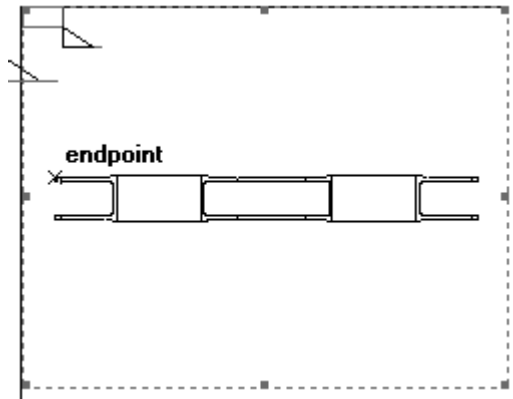


29. Choose the **Selection** tool and select the Top view.

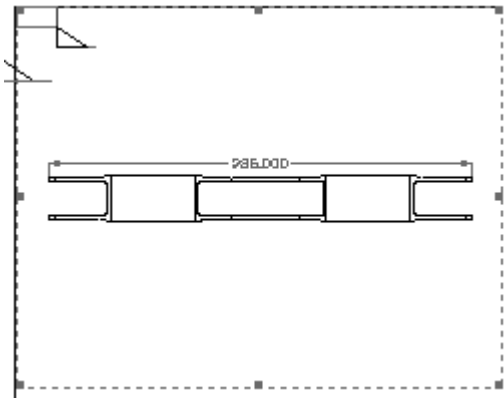


30. From the **Horizontal Dimension** tool subpalette, choose the **Horizontal Dimension** tool. (See page 6-8 for graphic illustration.)

31. Move the pointer to the top left endpoint of the chassis such that an *endpoint* notation appears.

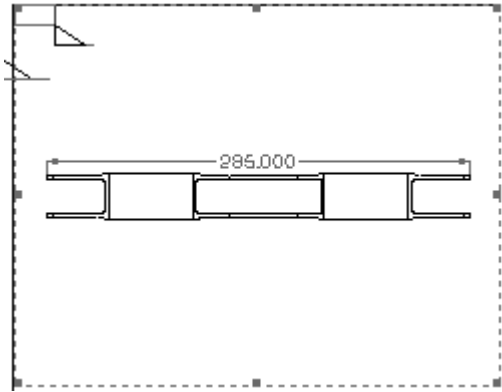


32. Click to place the first point.
33. Move the pointer to the endpoint at the opposite side of the chassis and click to place the other point.

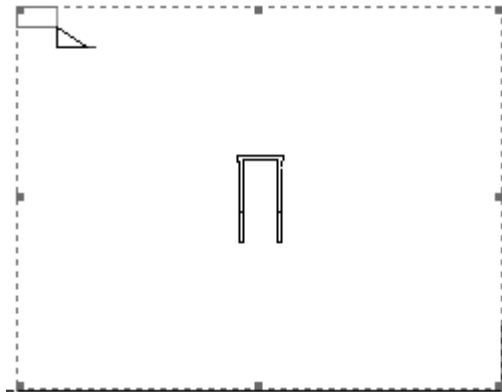


34. Choose **Dimension>Size** and select 24.

The dimensions text resizes.



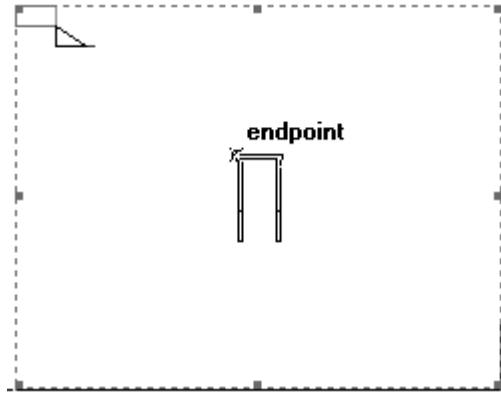
35. Choose the **Selection** tool and select the Right Side view.



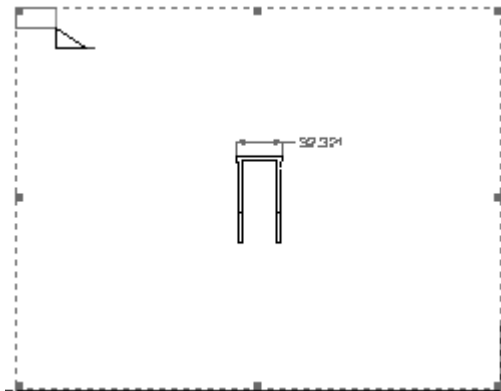
36. Select the **Horizontal Dimension** tool again.



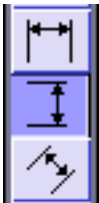
37. Click the top left endpoint of the chassis.



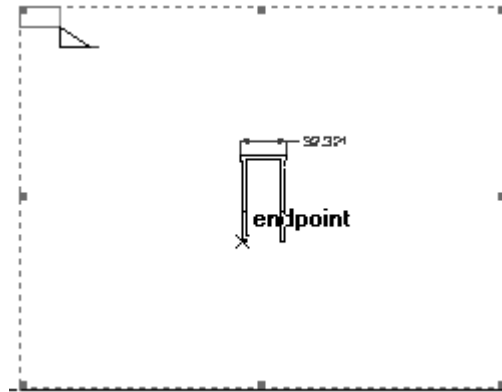
38. Click the top right endpoint and the dimension appears.



39. Choose the **Vertical Dimension** tool, as shown here.



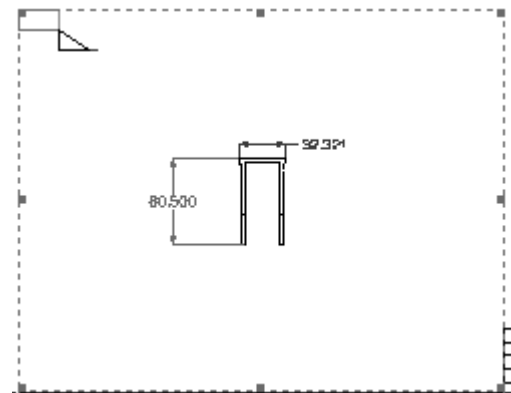
40. Move the pointer to the bottom left endpoint of the chassis and click at the endpoint.



41. Click the second point at the top left endpoint.
42. Place the pointer over the center of the text so that a four way arrow appears.

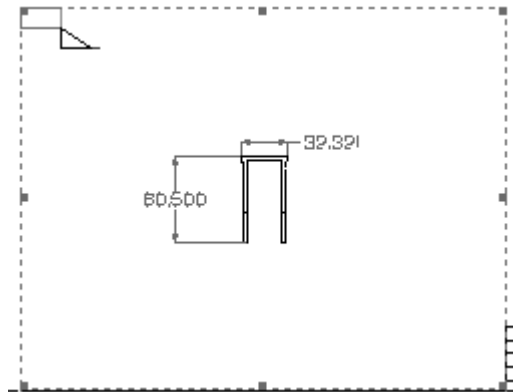


43. Drag the dimension to the left so that it does not cover the part.
44. Choose the **Selection** tool.
45. Hold down the SHIFT key and select the horizontal dimension in addition to the vertical dimension still selected.



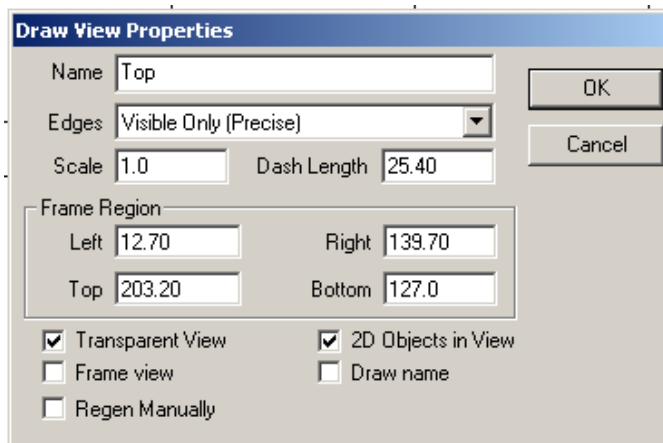
46. Choose **Dimension>Size** and select 24.

The dimensions text resizes.



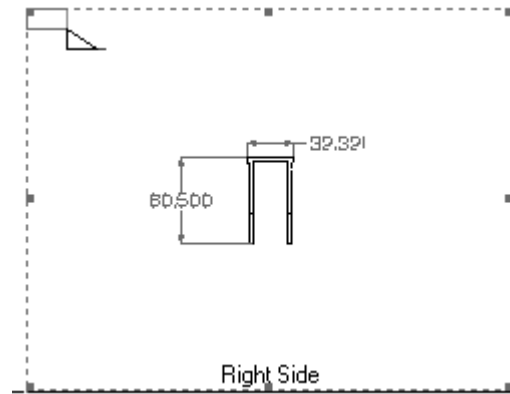
47. To complete the tutorial, add view names to the drawing views.

- Click on the top left corner of the drawing view to display the Drawing View menu.
- Choose the *Properties* command and the dialog box displays.

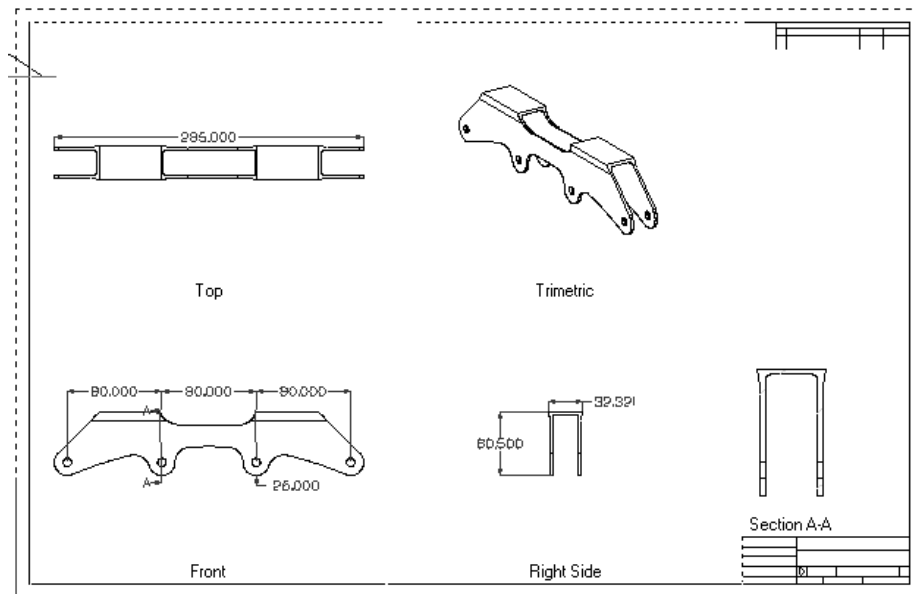


- Select the *Draw name* option.
- Click OK.

The dialog box closes and the name, Right Side appears.



- Select the views separately and repeat this procedure to display their respective view names.
- Click outside of the drawing area to deselect everything.



48. Save the file.

Congratulations! You've generated a 2D detail drawing with a section view and dimensions. You've also completed the Ashlar's 3D modeling products tutorial.

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