

COBALT



XENON

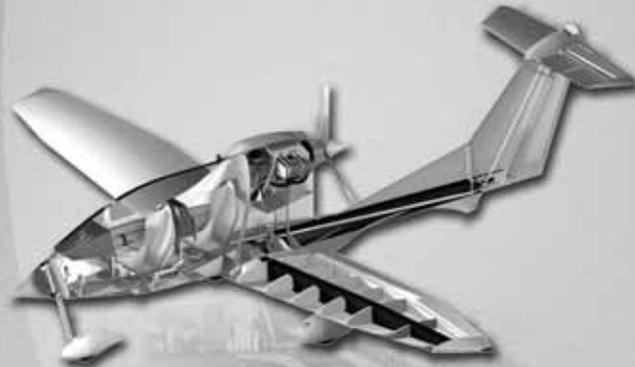


ARGON

Designer **Elements**

v8 Getting Started

 **ASHLAR·VELLUM™**
Software that works the way you think.



Designer **Elements**
v8 Getting Started

Cobalt™, Argon™, Xenon™

August 2009

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Table of Contents

Introduction to Ashlar-Vellum 3-D Modeling	1-1
Ashlar-Vellum 3D Modeling Products	1-1
Some Things to Know	1-4
Tutorial Introduction	1-5
The 3D Modeling Environment	2-1
Exercise 1: Ashlar-Vellum's User Interface	2-1
Exercise 2: Exploring the Drafting Assistant™	2-9
Exercise 3: Saving Preferences	2-17
Exercise 4: Selecting Objects	2-20
Exercise 5: Drawing Technique Options	2-24
Wireframe Modeling	3-1
Exercise 1: Setting Up the File	3-2
Exercise 2: Creating the Wheel Axles	3-8
Exercise 3: Creating the Chassis Profile	3-13
Exercise 4: Editing the Profile	3-19
Solid Modeling	4-1
Exercise 1: Extruding the Chassis Profile	4-1
Exercise 2: Setting Up the Mounting Platform	4-5
Exercise 3: Creating a Platform Cutout	4-11
Exercise 4: Creating the Chassis Cutout	4-18
Exercise 5: Basic Rendering	4-27

Exercise 6: Advanced Rendering	4-29
Exercise 7: Calculating the Chassis' Physical Properties	4-33
The Detail Drawing	5-1
Exercise 1: Creating a 2D Detail Drawing	5-2
Exercise 2: Creating a Section View	5-4
Exercise 3: Adding Dimensions	5-8
Index	1-1

Introduction to Ashlar-Vellum 3-D Modeling

Ashlar-Vellum 3D 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 contains all of the fundamental features of the modelers. Xenon and Cobalt add functionality to that foundation that greatly enhances productivity for certain design processes. The differences in Argon, Xenon, and Cobalt were developed to provide value and choice for 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 offer a wide variety of modeling tools, but Argon provides the means to precisely share data with other CAD/CAM/CAE packages through DXF/DWG, STEP, IGES, Parasolids (UG, SolidEdge, SolidWorks), and ACIS, SAT (AutoCAD, Inventor, Mechanical DeskTop) translators.

Xenon

Building on everything found in Argon, Xenon provides additional functionality 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 so that any modifications to the original shape are propagated throughout the design. View the history tree for curves, surfaces and solids using the 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. A 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 remitted as needed.
4. Surfaces are stitched into a watertight body represented as a solid.
5. The 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 changes require modifications on the original curves.
9. Curves are modified and steps 2 - 7 automatically update through associativity.

Cobalt

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.

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

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

Some Things to Know

Documentation

The graphics in this manual apply to both Macintosh and Windows operating systems. For those instances when a platform-specific reference occurs, a Windows graphic is used.

Serial Number/Registration Code

Your serial number and registration code are required to operate the software. The serial number and registration code are emailed to you upon purchase. If the registration code is not there please call 800-966-2349 to request it.

Enter your name and company name to personalize your copy of the software.

Recommended System Requirements

At the time of publication of the document, the following system configuration is recommended.

Windows

- Windows XP/Vista
- 32 MB Accelerated Video Card w/ OpenGL
- 512 MB RAM
- 250 MB Free hard drive space
- DVD Drive or High Speed Internet

Macintosh

- OS X Leopard
- 32 MB Accelerated Video Card w/ Open GL
- 512 MB RAM
- 1 GB Allocated virtual memory
- 250 MB Free hard drive space
- DVD Drive or High Speed Internet

Tutorial Introduction

This tutorial is a learning, productivity and reference tool that will introduce 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

Proceeding through the exercises, choose commands contained in submenus of other menus, like the pull down menu. For example, to set a color, go 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 their significance is instantly recognized and they can be easily located for future reference.

Tip

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

Tech Note

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

Referral

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

Step Conventions

The numbered tasks in the exercises describe the activity to perform, and the bulleted steps beneath the numbers direct how to accomplish the task. If you already know how to accomplish the task, 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-Vellum's 3D modeling products. If you are adventuresome, open a new document for your explorations and then switch back to the tutorial document 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 3D 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 starting 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.

Pointer

Depending on it's location, the point is an arrow or may look like the current tool.



The 3D Modeling Environment

Ashlar-Vellum's 3D design environment was developed to enable the designer to focus on the part being designed not on the software used to design it. This chapter presents the following:

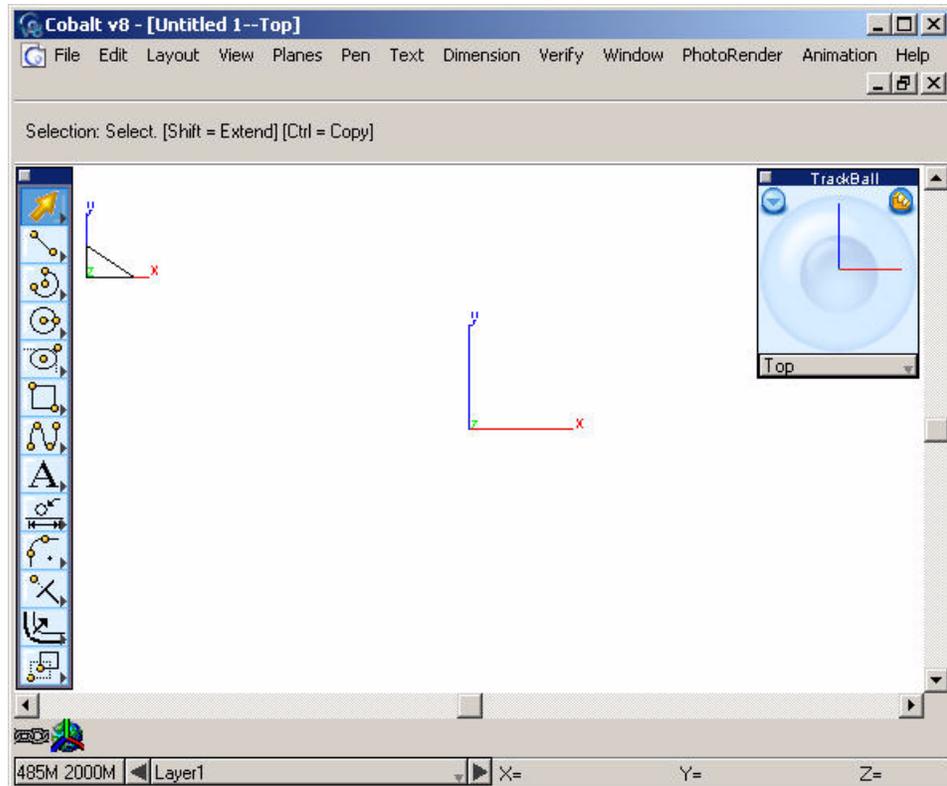
- Familiarization with the user interface
- Exploring the Drafting Assistant
- Saving preferences
- Selecting objects
- Introduction to drawing techniques

Exercise 1: Ashlar-Vellum's User Interface

This exercise introduces the user interface.

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

The program opens, displaying an untitled document with an empty drawing area.

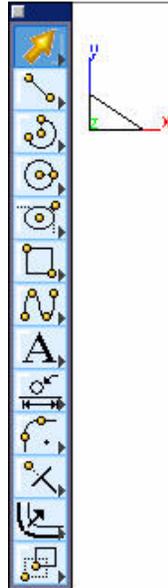


2. Take a look at the menu bar across the top. There are thirteen menus.

File	Contains commands that affect entire documents (files), including opening and closing files, setting preferences and defining the <i>Print Setup</i> (Windows) or <i>Page Setup</i> (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 <i>Grid</i> , <i>Layer Manager</i> , <i>Group</i> and <i>Align</i> .

View	Contains commands for displaying the document including setting and selecting 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 the text (lower, upper and title capitals).
Dimension	Contains commands that specify dimensions, their format and tolerance.
Verify	Contains commands to obtain information about a file and specific objects in a 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 photo-realistically rendering the geometry and setting rendering options.
Animation	Contains commands for generating Quicktime movies.
Help	Contains the commands to access the on-line help file, updating the software from the website and connecting to technical support.

3. Take a closer look at the main tool palette.



4. This tool palette contains drawing and editing tools used for constructing, editing and annotating geometry as developing parts and assemblies. All of the 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 tools

Select object(s) and points in the drawing area and copy properties from one object to another.



Line tools

Create line segments (by two endpoints as well as by mid-point and endpoint), connected lines, a line parallel to existing lines and points. When creating 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, circles specified by 2 opposite points, circles specified by 3 points and circles tangent to two objects.



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, star polygons, arbitrary polygons and polygons from curves.



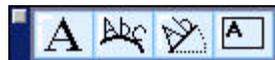
Spline tools

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 it is possible to 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. 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

Add dimensions as desired to drawings.



Fillet/Chamfer tools

Construct fillets and chamfers from corners formed by nonparallel lines or curves. Fillets and chamfers are automatically trimmed by default but this can be overridden to prevent trimming.



Trim tools

Lengthen or shorten lines and curves. Generally, the procedure is to select the limiting object(s) before selecting 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 and location as well as duplicating objects in a number of specific patterns.



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



When selecting a tool and moving 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 that should be specified. 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.

6. Take a look at the Message Line at the top of the 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 guides each step for using the tool and any additional options. These options are available for a number of tools in the software.

7. 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*

8. Move the smart pointer and a rubberband image appears. (The smaller circle in the graphic below is the smart pointer for the **Opposite-Point Circle** tool.)



9. 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.

10. Look at the Status Line at the bottom of the drawing area.



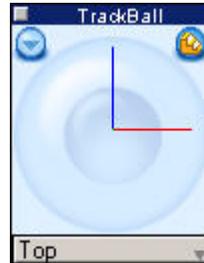
When selecting a tool, the Status Line contains data fields that provide information about the geometry being created, such as measurements, angles, coordinates and the like.

The Status Line can be used 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 an object is constructed, the data field that can be changed is active.

12. Choose **Window>Trackball**.

The Trackball displays.



The trackball rotates the view orientation of the geometry in the drawing area. This Sphere Trackball drags the 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:

Set the angle increment for the Step Trackball using the View Rotation Options dialog box. See Chapter 6 of the *User Guide* for information.

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

14. Click the sphere button on the right side of the Trackball title bar to return to the Sphere Trackball.

Exercise 2: Exploring the Drafting Assistant™

The Drafting Assistant™ is the feature that separates Ashlar-Vellum 3D modeling products from other design and drafting software. The Drafting Assistant™ thinks like a designer. It automatically knows where construction lines are typically wanted and displays them temporarily when necessary.

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.

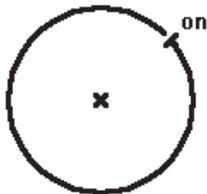
When the pointer is in the drawing area, the Drafting Assistant™ locks onto specific points on existing objects as the pointer is moved near them.

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.

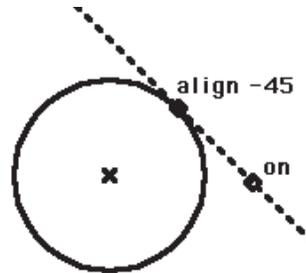


The pointer becomes a crosshair.

3. Move the pointer across 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 shows when the snap point has locked onto the following points of an object.

center



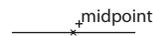
Indicates the center of an arc or circle. Move the pointer across the arc or circle to display *on* for the arc or circle, then move the pointer near the center to display the *center* point notation.

endpoint



Indicates the endpoints of lines, arcs, circles, ellipses and splines.

midpoint



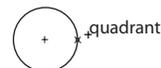
Indicates the midpoint of lines, arcs, circles, ellipses and splines.

intersection



Shows the intersection of two curves including permanent and construction lines.

quadrant



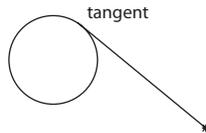
Displays the quadrant points on an arc or circle displayed at 3 o'clock, 6 o'clock, 9 o'clock and 12 o'clock.

vertex



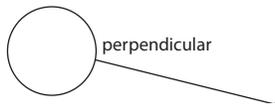
Indicates the vertices of an ellipse, spline, or dimension point.

tangent



Shows the tangent points of an arc or circle. If a point is clicked on an arc or circle and the pointer is dragged away at about a 45° angle, the Drafting Assistant locks onto a tangent.

perpendicular



Shows the perpendicular points off any curve. If a point is clicked on a curve and dragged away at a 90° angle the Drafting Assistant locks onto a perpendicular.

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

5. Click to place the final point.

Once a point is set, the Drafting Assistant™ construction line disappears. The 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.

- Click somewhere in the drawing to set the first point.

As the pointer is moved, a dynamic construction line extends automatically from the last point created.



- 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.

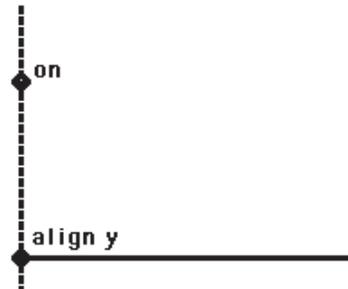


The first line segment is completed.

- Move the pointer over the left endpoint of the line segment.

When this kind of action is performed over one or more points, they are activated. The Drafting Assistant™ automatically displays construction lines relative to them.

- Click on the endpoint to place the first point of another segment.
- Move the pointer vertically up from that point.



- 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.



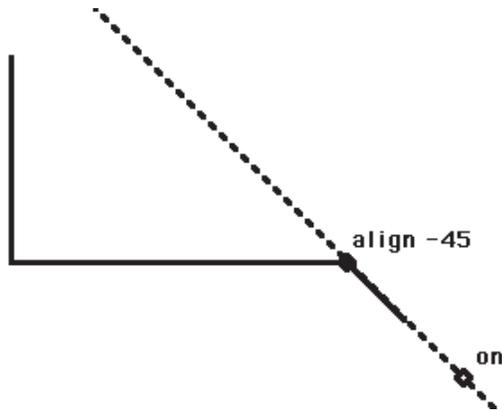
- Drag to the **Connected Lines** tool and release the mouse button. The **Connected Lines** tool now displays selected in the palette.



This tool creates connected line 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.

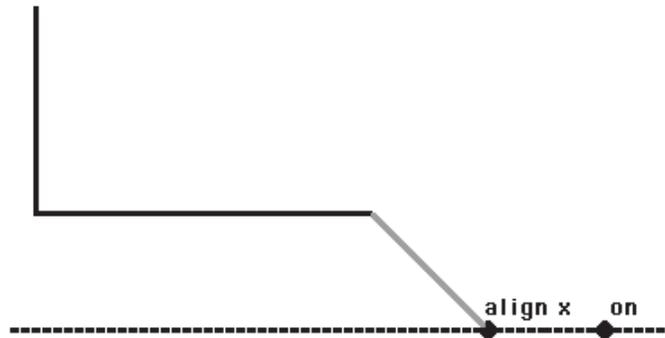
A 45° dynamic construction line appears.



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.

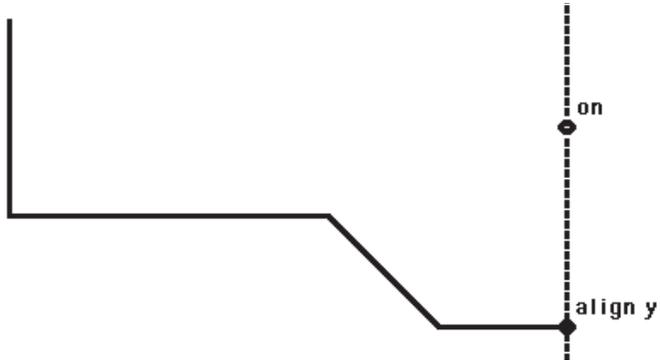


19. Click the next point somewhere to the right of the last point.

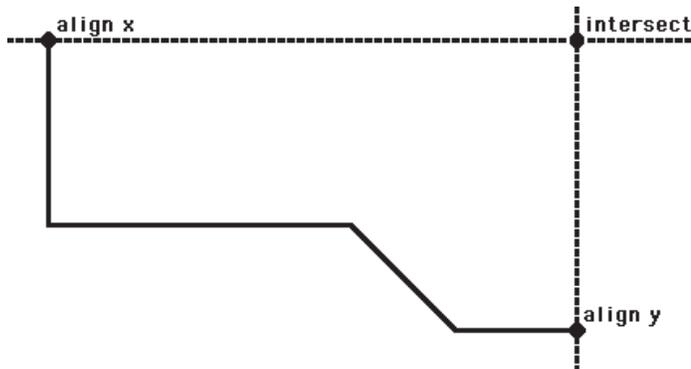


Notice that the point is placed on the construction line even though the pointer wasn't exactly on that line when the mouse was clicked.

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

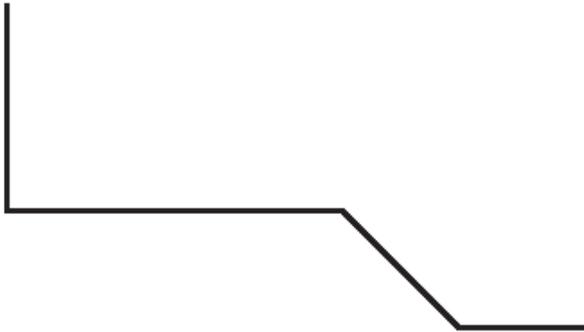


21. Continue moving the pointer so that it is aligned with the top endpoint of the vertical line segment and the last point placed. Two construction lines will display, as shown here.



These construction lines are the Drafting Assistant's attempt to anticipate the 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 as desired.

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

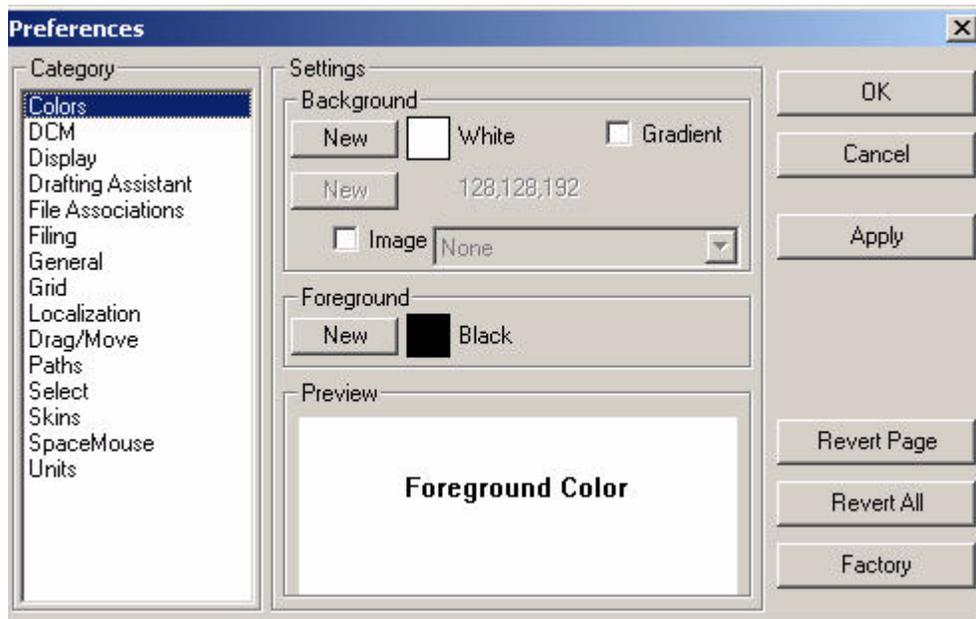
This exercise was a very simple example of the power of the Drafting Assistant™. The Drafting Assistant™ helps with more complex drawings, including 3D, as discovered in the next three chapters.

Exercise 3: Saving Preferences

Cobalt, Xenon and Argon save preferences for a particular session and for the style in which you like to work.

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

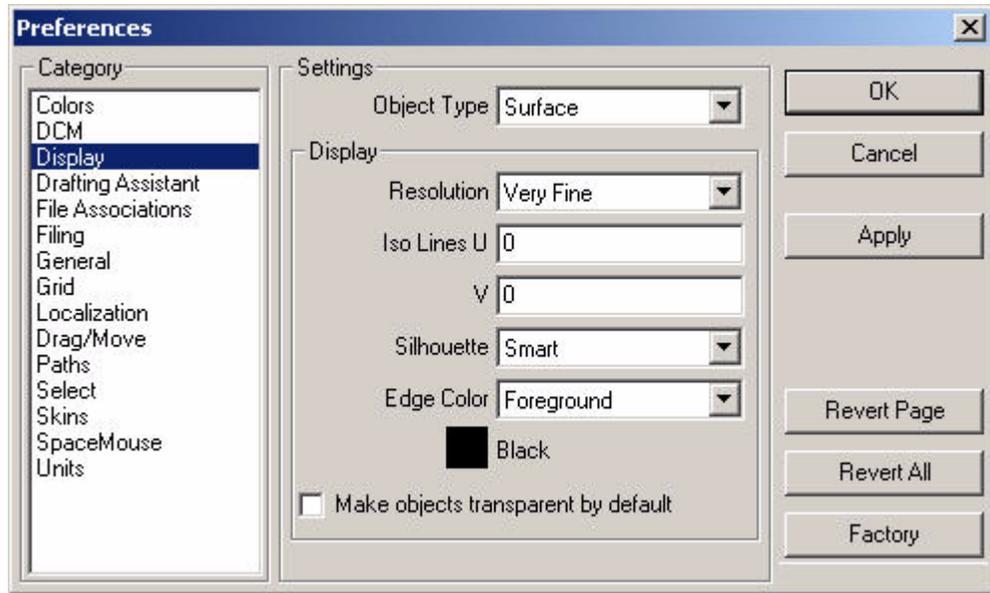
The Preferences dialog box contains a category list of preference groups, group settings and dialog box buttons.



The preference categories include Colors, DCM, Display, Drafting Assistant, File Associations, Filing, General, Grid, Localization, Drag/Move, Paths, Select, Skins, SpaceMouse and Units.

3. Choose the Display category.

4. From the Object Type section choose Surface.



Set Resolution, Iso Lines, Silhouette and Edge Color for surface and solid object types in this section.

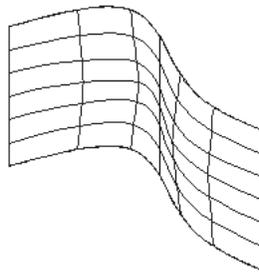
Resolution

Controls how accurately an object's curves appear. 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 visually appealing 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* for information on each of these settings.

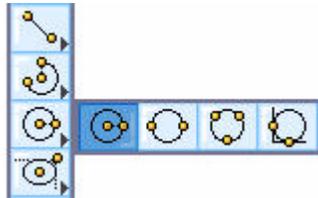
When any of the preference settings are changed, the changes apply only to new objects.

Exercise 4: Selecting Objects

Once an object is created, it is possible to make changes. In order to do so, first select the object. In Ashlar-Vellum 3D modeling products, the step at which the object is selected depends on which operation is performed. When an object is selected it highlights until another object is selected or it is deselected by clicking in the drawing area. Select an object with the **Selection** tool or by the hollow selection tool that appears temporarily when using one of the **Editing** 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 the drawing area.
3. Select the **Single Line** tool.

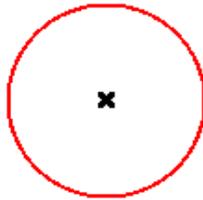


4. Create a line of any length.
5. Choose the **Selection** tool, the first tool in the main tool palette.



6. Place the pointer over the circle and click.

The circle highlights red (the default selection color set in Preferences).

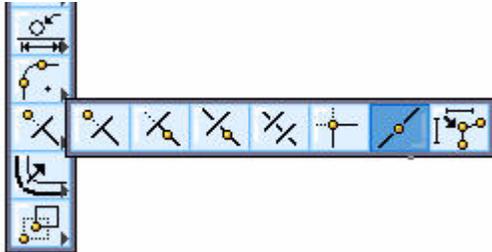


7. Click in the drawing area to deselect the circle.

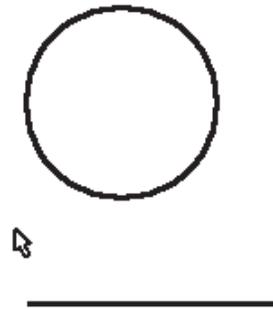
Selecting and Modifying Object with an Editing Tool

When editing an object, it may be necessary to select an object after selecting the tool.

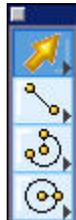
1. Select the **Divide at Location** tool.



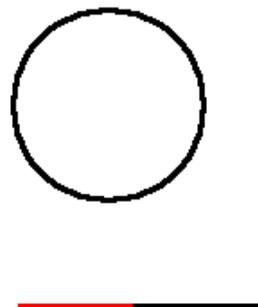
2. Move the pointer into the drawing area. The pointer has become a hollow selection tool.



3. Select the line.
The line highlights and the next step with the tool can be performed.
4. Click on the line at some location.
The line divides at that location even though it is not seen.
5. To verify the division, choose the **Selection** tool.



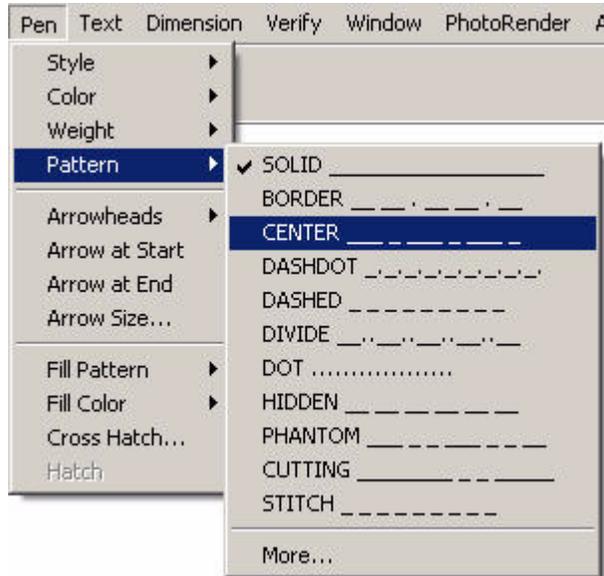
6. Click on the line at some location and only a portion of the line highlights showing that the line has been divided.



Selecting an Object and Modifying it with a Command

Edit an object simply by changing one of its properties. For example, if a center pattern is applied to a circle to indicate a bolt-hole, it might be necessary to change the pen pattern.

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



The circle pen pattern changes to a center pattern display.

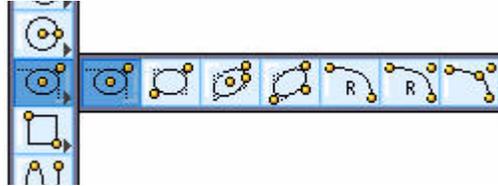


Exercise 5: Drawing Technique Options

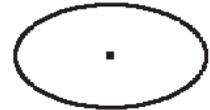
In Cobalt, Argon and Xenon all objects are created by clicking. As geometry is being created a rubberband image of the geometry displays showing how the object will appear when the last point is clicked.

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

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

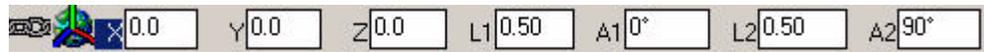


2. Move the pointer to the drawing area and click to set the center point of the ellipse.
3. Move the pointer to the another location. (It doesn't matter where.) While doing this a rubberband image of the line appears.



When moving the pointer, the rubberband image adjusts accordingly.

4. Click to set the corner point of the control rectangle of the ellipse, as indicated in the Message Line.
5. Notice that the X field in the Status Line at the bottom of the drawing area is highlighted.

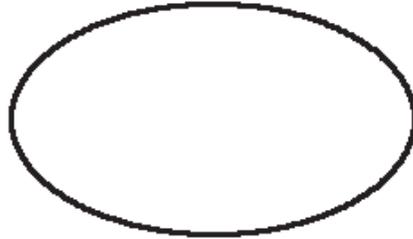


This field represents the X location of the center point of the ellipse.

Immediately after construction the X location or any other value in the Status Line can be changed. If the geometry is not satisfactory, press BACKSPACE (Windows) or DELETE (Macintosh).

6. Use the TAB key to move to the L1 data field.
7. Type **1.75**.
8. Press TAB twice to highlight the L2 data field.
9. Type **1.0**.

10. Press ENTER (Windows) or RETURN (Macintosh) and the ellipse updates.



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

Congratulations! You have completed the introductory section of the Cobalt, Xenon and Argon tutorial. The next section will introduce wireframe modeling.

Wireframe Modeling

Just as there are several ways of approaching a design problem, Cobalt, Xenon and Argon have different ways of modeling objects based on the software features you choose to use in the design environment.

Our 3D modeling products create wireframe, surface and solid models. This section of the tutorial introduces 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.

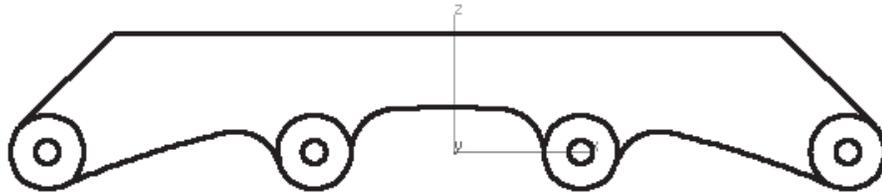
Wireframes in Cobalt, Argon and Xenon consist of points, lines, arcs, circles, ellipses, conics, splines or a combination of these. These same wireframes are also collectively referred to as curves. Although wireframes are limited in the amount of model content they represent, they are powerful building 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.

This chapter demonstrates how to:

- Set up a 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.



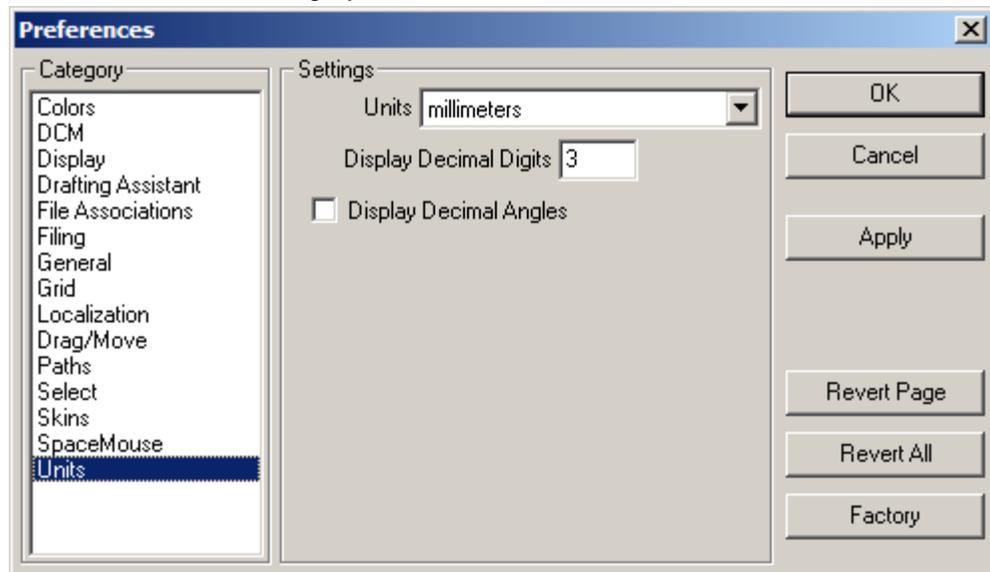
Exercise 1: Setting Up the File

Set some preferences for the file as well as the view scale to fit the geometry.

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

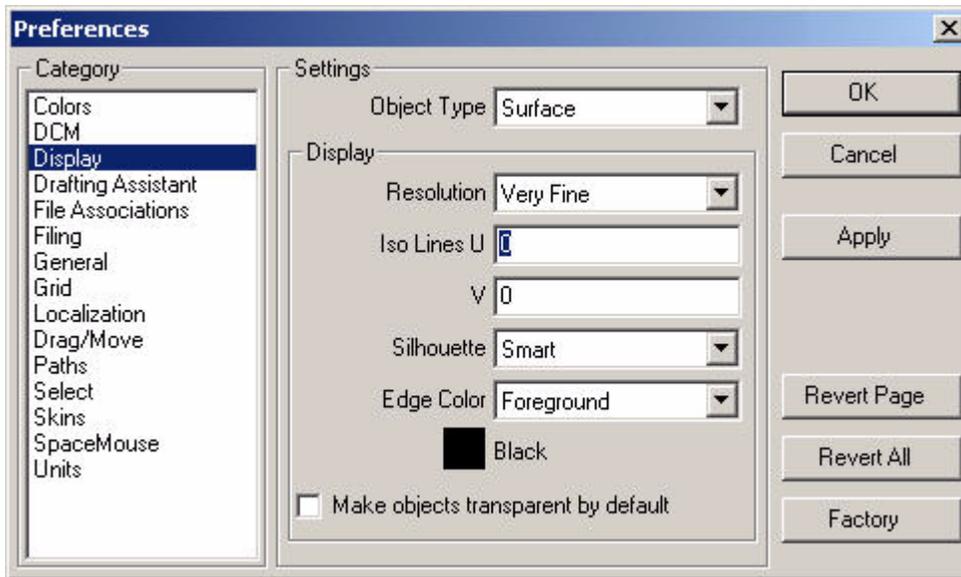
If the software is not already running, launch the program. An untitled file automatically displays. Do not open a new file.

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.

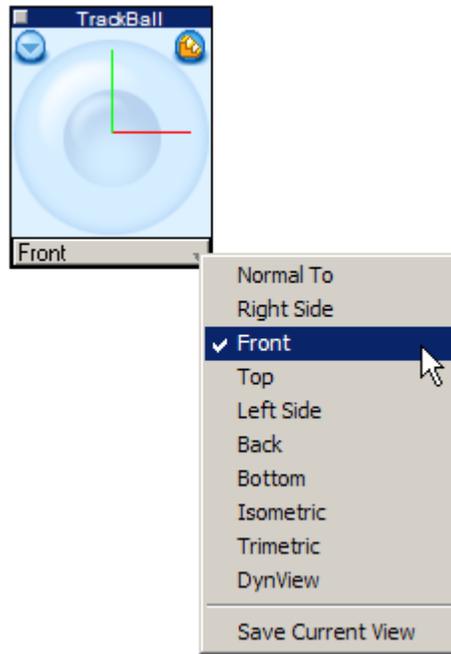
5. Select the *Display* category.



There are three object types: Curve, Surface and Solid; and five resolution settings: Coarse, Medium, Fine, Very Fine and Super Fine. The resolution for each object type can be set 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 settings.
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 the drawing area shows the Front plane graphically.

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

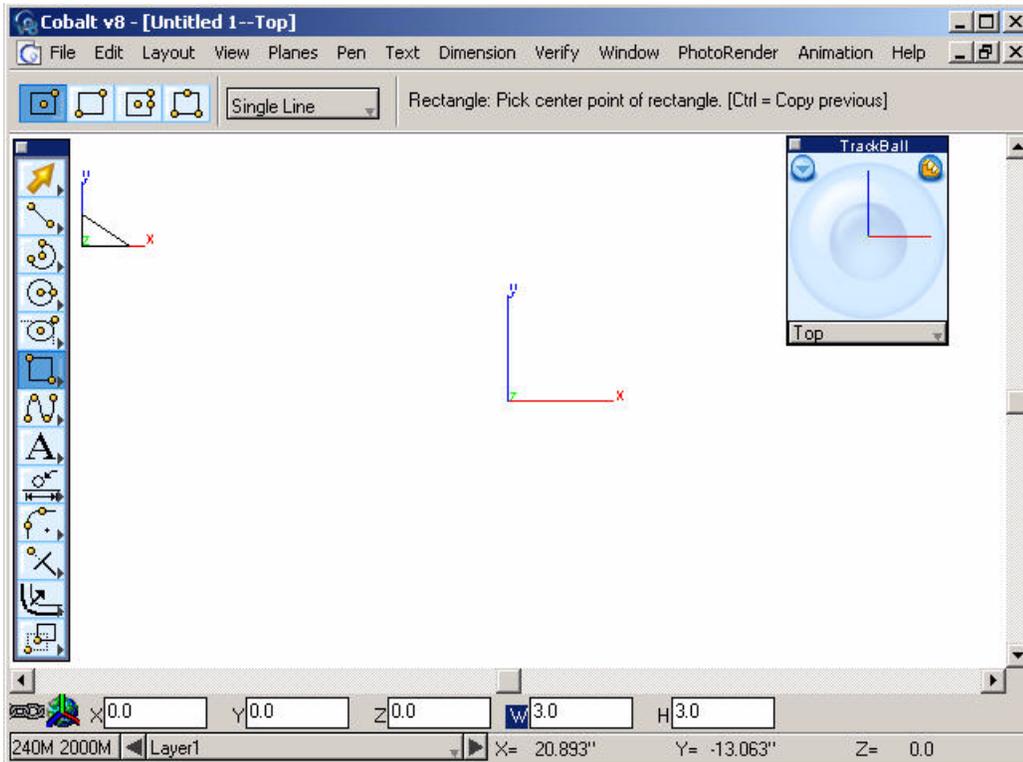


The Message Line includes an additional palette with four types of rectangle tools: polygon by center and point (**Center/Corner Rectangle** tool), polygon by diagonal (**Diagonal Rectangle** tool), polygon by center/axis (**Center/Axis Rectangle** tool) and polygon by major and minor axis (**Major/Minor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

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



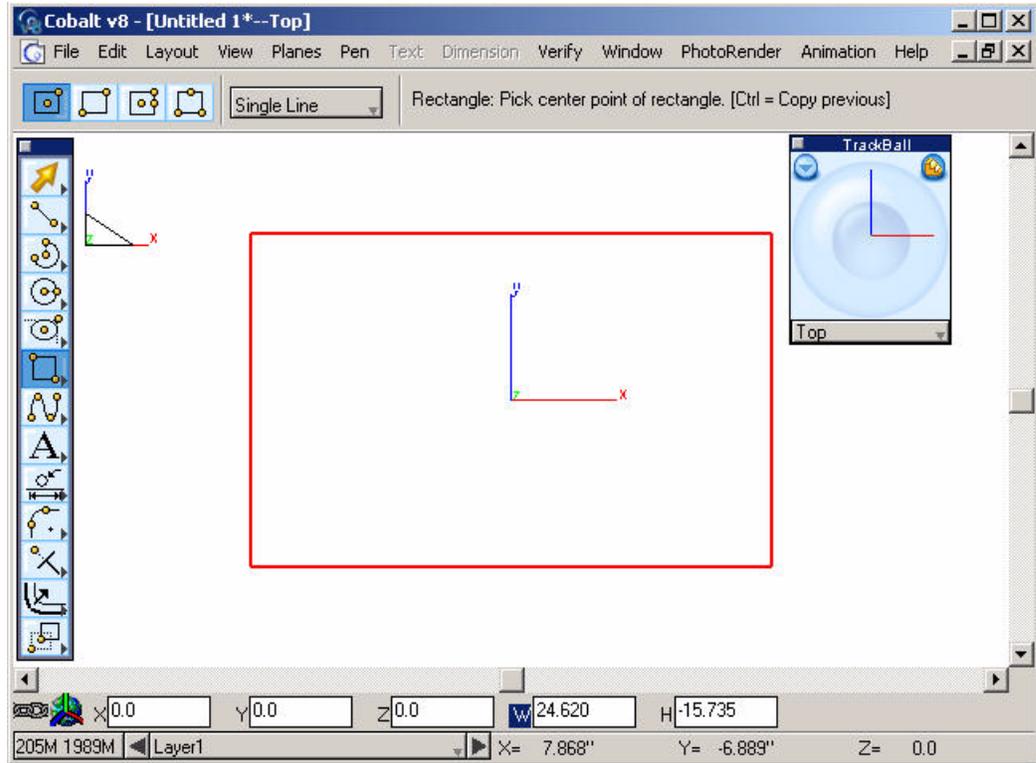
16. Move the rectangle pointer to the origin of the drawing located at the intersection of the X, Y and Z axes.



Notice that the Message Line provides 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.

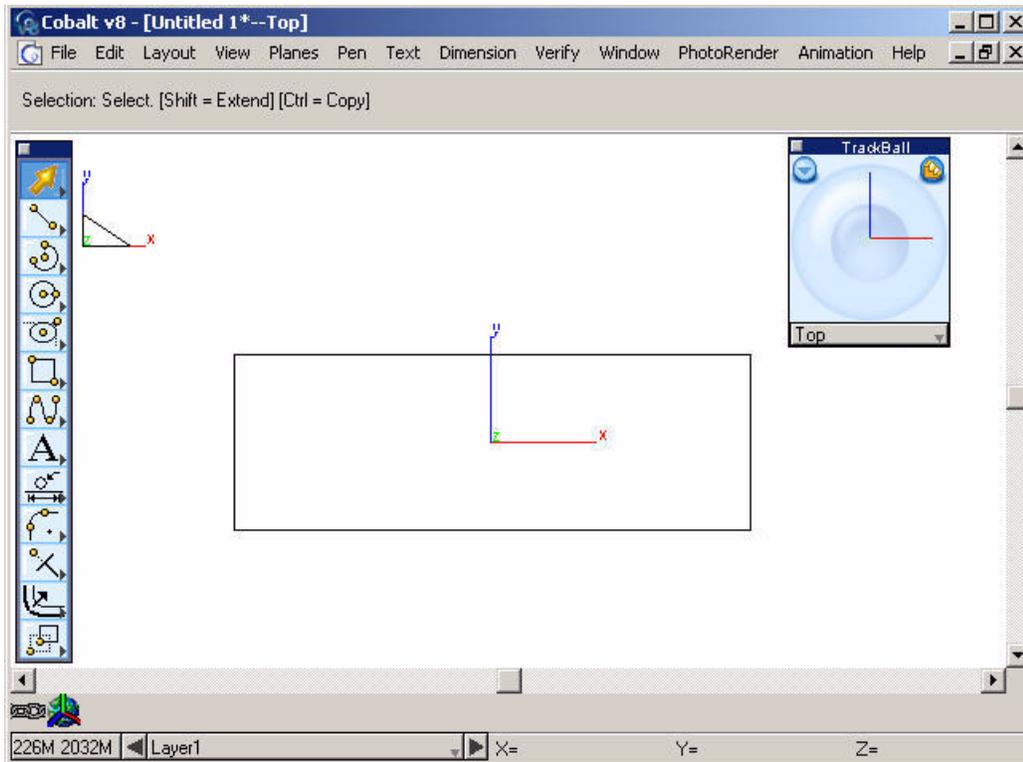


The W or width data field is highlighted in the Status Line below the drawing.

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**.

The view rescales to fit the rectangle.



25. Delete the rectangle.

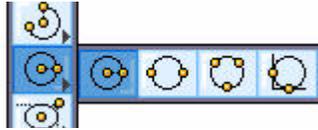
- Using the **Selection** tool, drag a selection fence around the rectangle (hold down mouse and drag a selection fence around the rectangle).
- Then press DELETE.
- Close the Trackball if it is taking up too much room in the drawing area.

The file is set to begin creating the skate. The rectangle was temporarily created to set the viewing scale to the monitor's screen size.

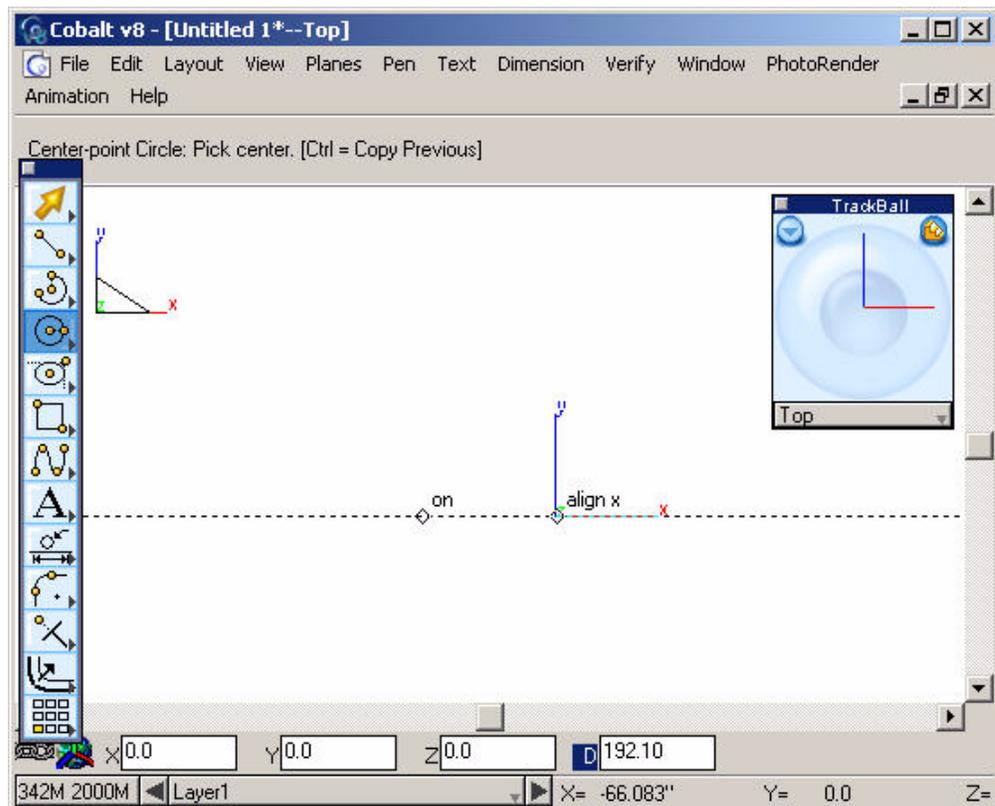
Exercise 2: Creating the Wheel Axles

This exercise creates the wireframe wheel axles using the **Center-Point Circle** tool, construction lines and the **Linear Duplicate** tool. Start the design in 2D and move on to 3D in the next chapter. 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.



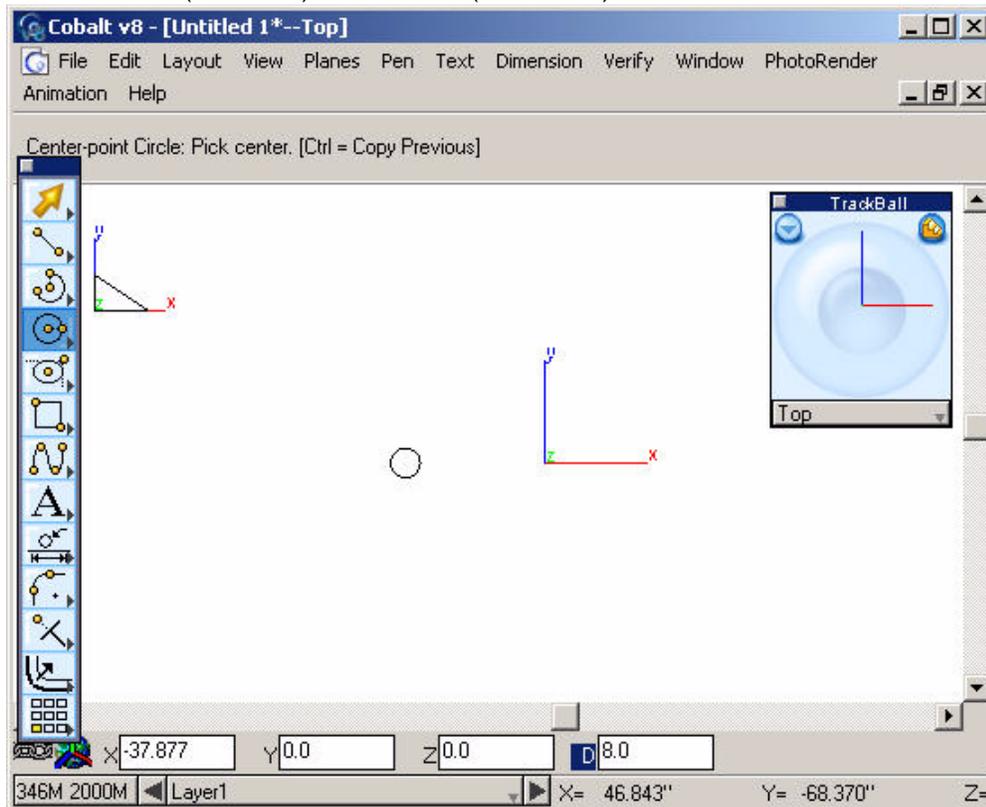
- Move the pointer to the origin to wake up a temporary horizontal construction line.



- Now move the pointer to the left, along the construction line, as shown above.
- Near the left of the 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 the pointer is moved 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).

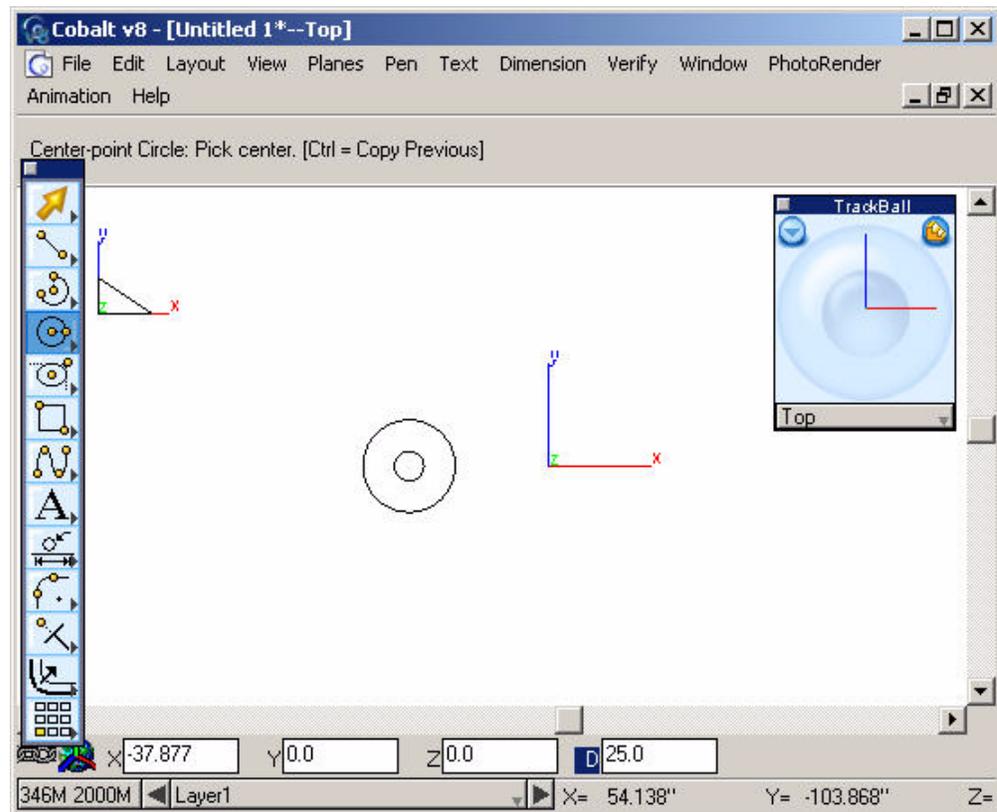


This circle represents the first axle hole.

- 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.

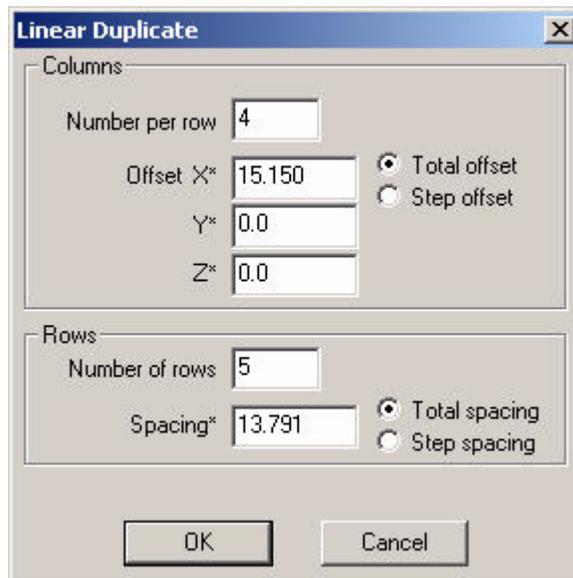


2. Select the two circles.
 - Choose the **Selection** tool.
 - Drag a selection fence around the two circles.
The two circles are selected.

3. Create three copies of these two circles across the 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.

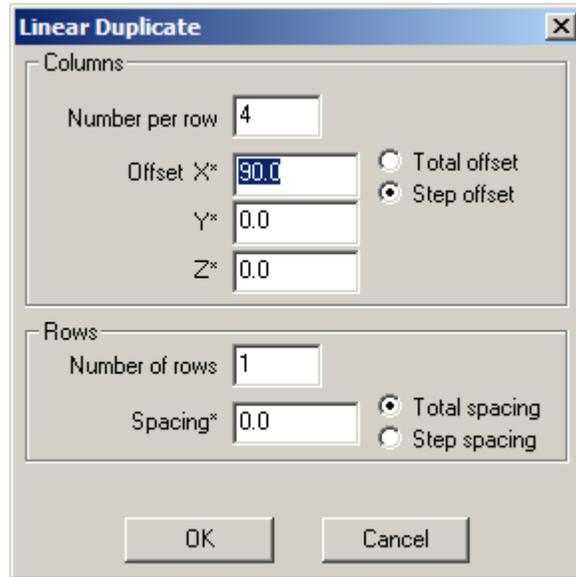


The Linear Duplicate dialog box appears.



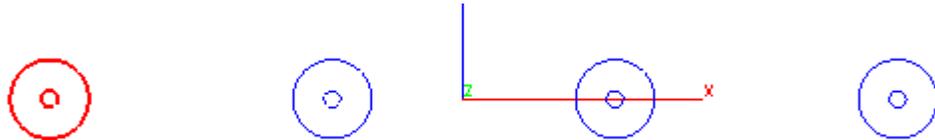
- In the *Number per row* field of the Columns section, type **4**.
- In the *Offset X* field, type **90**.
- Select the *Step offset* option, if it is not already selected.

- In the *Number of rows* field of the Rows section, type 1.



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

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



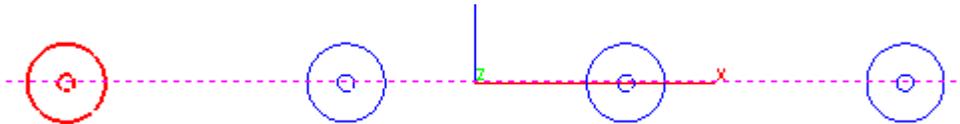
4. Save the file as *Inline Skate Profile*.
 - Choose **File>Save** or use CTRL+S (Windows) or ⌘+S, (Macintosh).
 - In the dialog box, type **Inline Skate Profile**.
 - Click Save.

Exercise 3: Creating the Chassis Profile

This exercise creates the chassis profile using construction lines, the **Parallel Line** tool, **3-point 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.

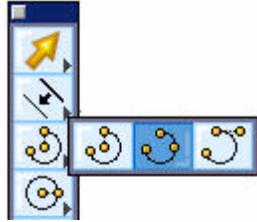


- Place the pointer, shown at the right, over the construction line and drag a new line up a short distance from the original line. ⇄
- In the Offset data field of the Status Line, type **15** and press ENTER (Windows) or RETURN (Macintosh).

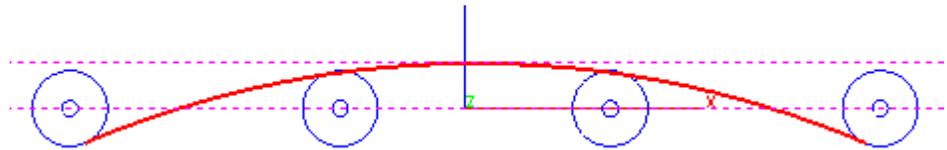


3. Create an arc, tangent to the left-most 25 mm circle, the upper construction line and the right-most 25 mm circle. This creates the shape for the bottom of the chassis.

- 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) 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. Use this line as a reference for setting the height of the chassis.

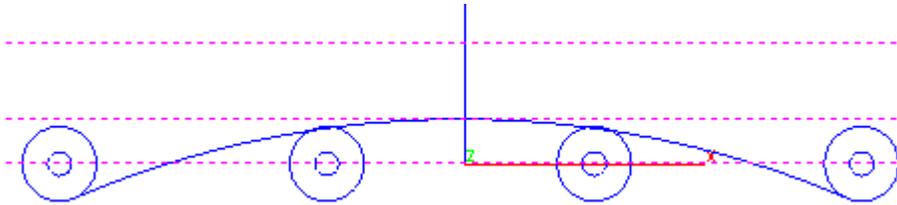
- In the **Line** tool palette, select the **Parallel Line** tool.



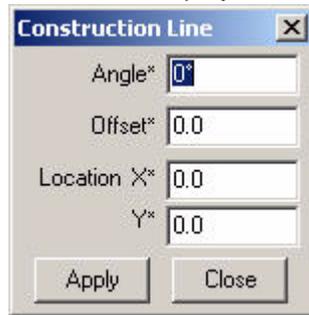
Notice that it is not necessary to open the subpalette to select the tool. When a tool is selected, it becomes the tool displayed in the palette.

- 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).



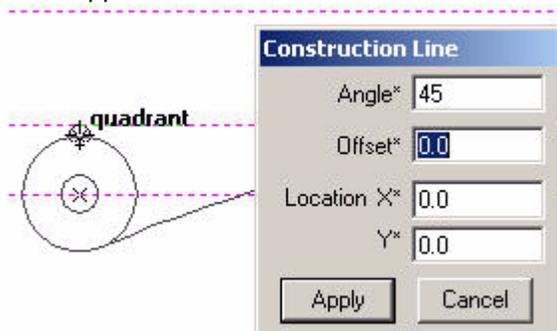
5. Draw a tangent line from the outer edge of the left-most 25 mm circle to the upper construction line at a 45° angle.
 - Choose **Layout>Construction** to display the Construction Line dialog box.



Tip:

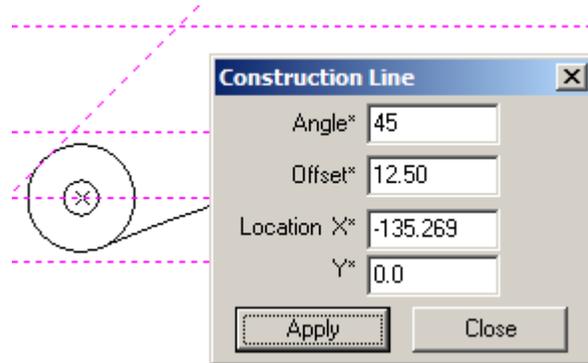
Click on the data field name to highlight the value in the field so it is ready to be overwritten. Click within the data field to leave the existing data unhighlighted, ready to be edited.

- 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 it is possible to 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 that intersects the upper construction line.



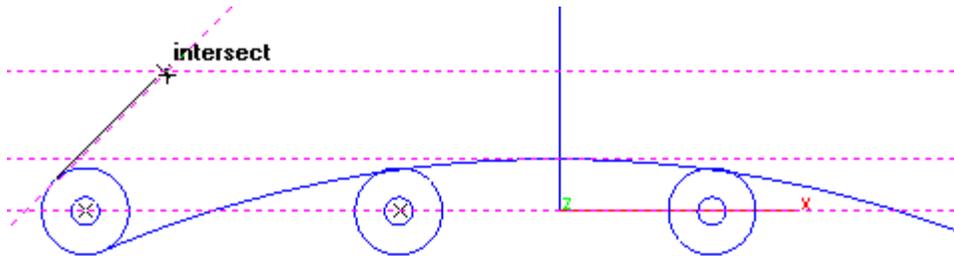
- Click Cancel to close the Construction Line dialog box.
- In the **Line** tool palette, select the **Single Line** tool.



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

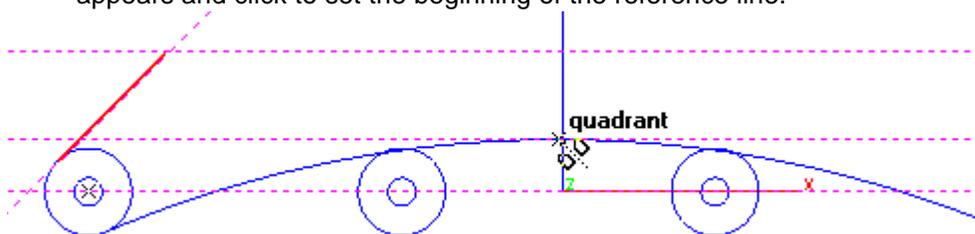


- Click to place the endpoint of the line at the noted intersection.
6. Using the arc's middle quadrant point as a reference, mirror a copy of the tangent line to the right side. This will create a new tangent line touching the right-most circle.
- In the **Transformation** tool palette, select the **Mirror**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



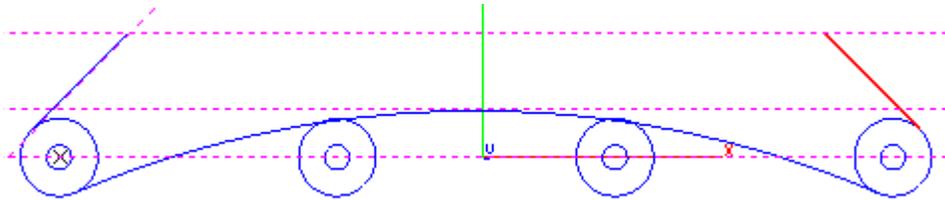
Make sure that the single line created in the previous step is still selected.

- 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).

A mirrored copy of the tangent line appears.

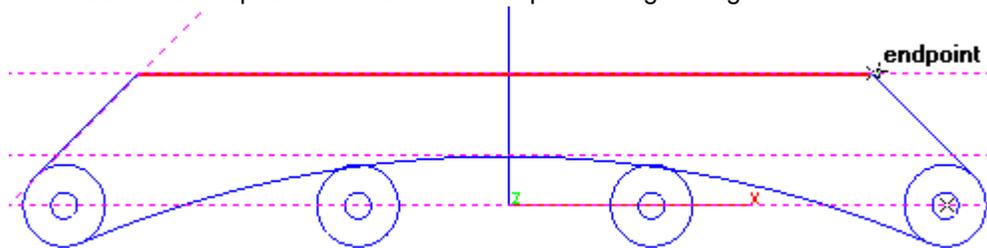


7. Draw a line between the top endpoints of the two tangent lines.

- Make sure that the Work Plane snap is on.
- In the **Line** tool palette, select the **Single Line** tool.



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

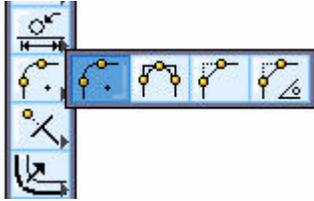
- Choose **File>Save** or use CTRL+S (Windows) or ⌘+S (Macintosh).

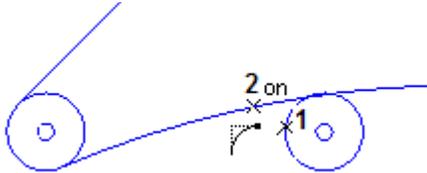
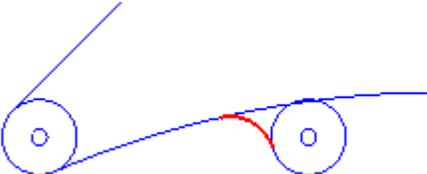
The basic chassis profile is created.

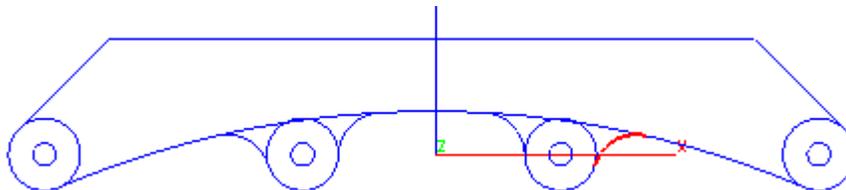
Exercise 4: Editing the Profile

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

1. Fillet the two innermost 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.

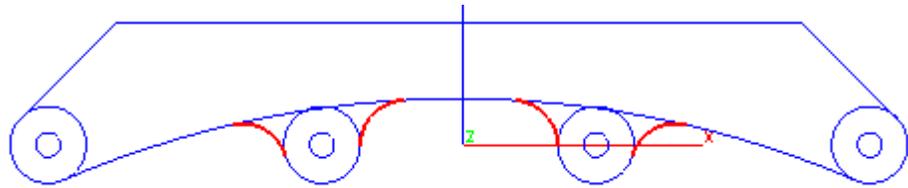


- In the Radius data field of the Status Line, type **15**.
 - Hold down the CTRL key (Windows) or the OPTION key (Macintosh) to prevent trimming of the line, as mentioned in the Message Line.
 - Click on the left side of the first of the two innermost circles (point 1 in the graphic) to select the first fillet entity.
- 
- A diagram showing a profile with two innermost circles. A red 'X' labeled '1' is placed on the left side of the first circle, indicating the selection point for 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.
- 
- A diagram showing the same profile as above, but now a red fillet is visible on the left side of the first circle. A red 'X' labeled '2 on' is placed on the arc to the left of the circle, indicating the selection point for the second fillet entity.

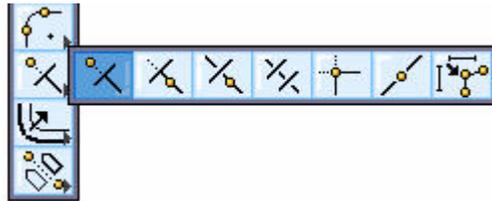


2. Trim the arc between the fillets on the left innermost circle and between the fillets on the right innermost circle.

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

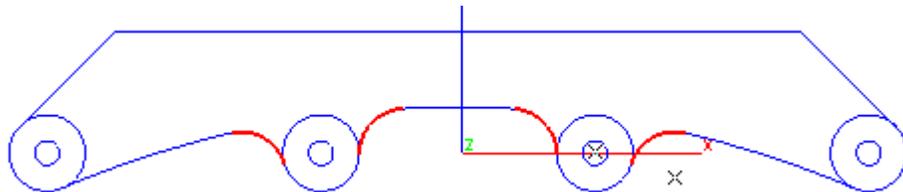


- Choose the **Simple Trim** tool from the **Curve Extras** tool palette.

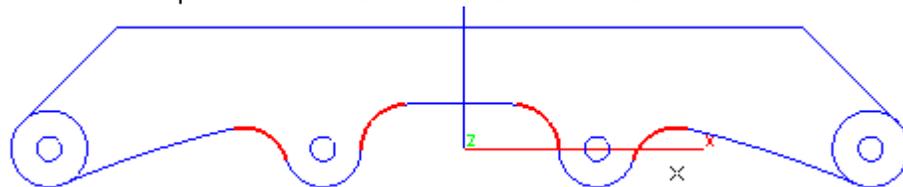


- Click on the arc section between the fillets for the left innermost circle and the arc section between the fillets for the right innermost circle.

The arcs are trimmed.

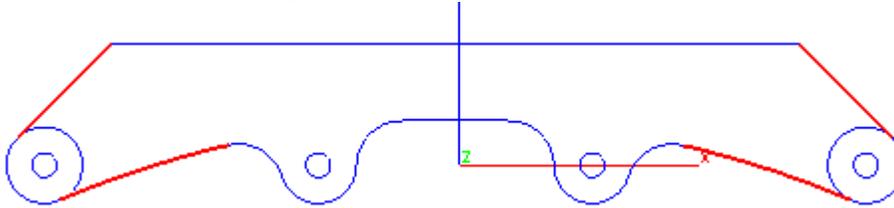


3. Now trim the portion of the inner circles between these same fillets.

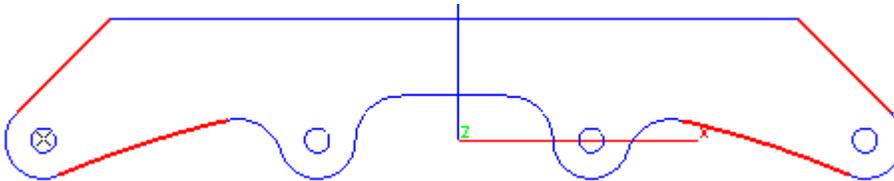


4. Trim the remaining portion of the left and right outermost circles to complete the wireframe.
 - Choose the **Selection** tool.

- 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! The wireframe profile for the inline skate chassis is completed now. In the next chapter, 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, continue designing the inline skate chassis using the 2D profile created in the last chapter to produce the following solid model:

This chapter demonstrates 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

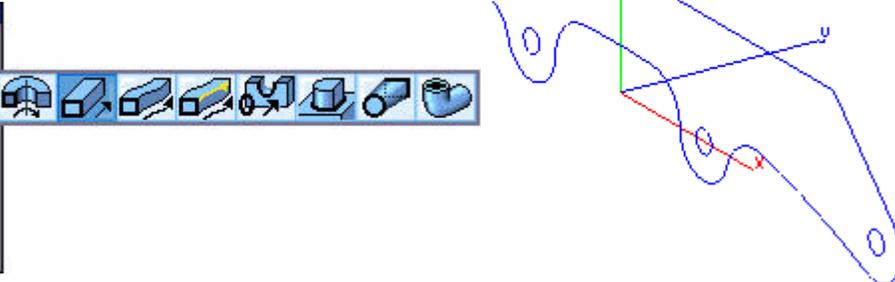
In this exercise, extrude the profile using the **Extrude Solid** tool.

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 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.



5. From the Trackball menu, choose Trimetric.
6. From the **Profiles** tool palette, choose the **Extrude Solid** tool.



The pointer becomes a hollow selection arrow.

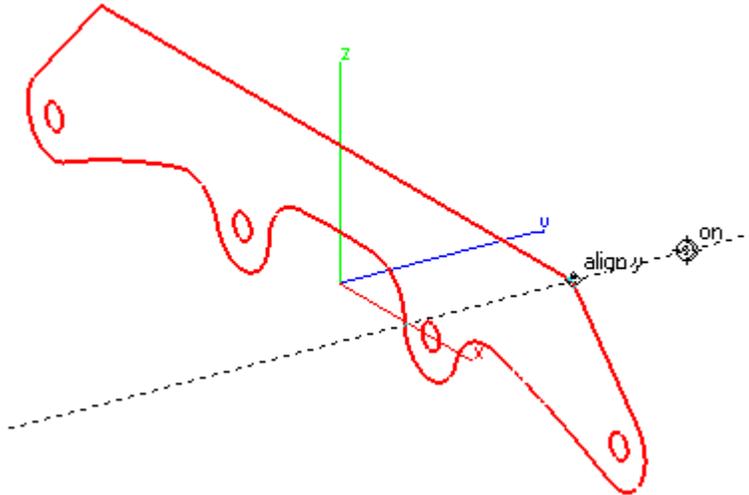


7. Drag a selection fence around the entire chassis profile to select it. The hollow selection arrow becomes a bulls-eye pointer.

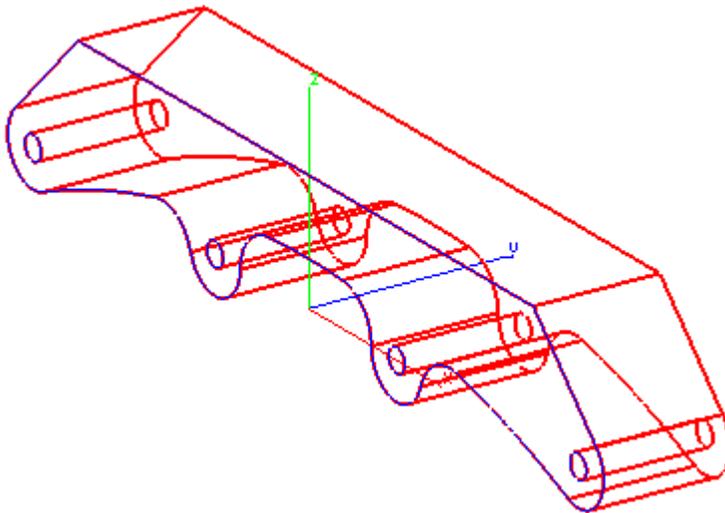


8. Click on a point on the profile. This specifies the beginning point of the extrusion.

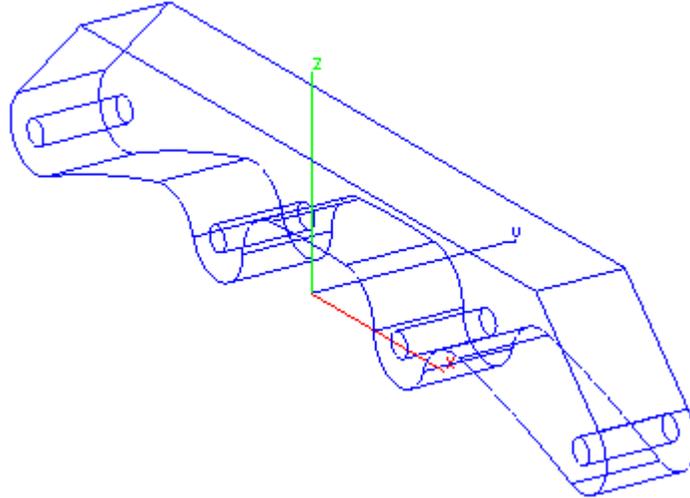
9. Move the pointer along the Y axis, using the Drafting Assistant as a guide.



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



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



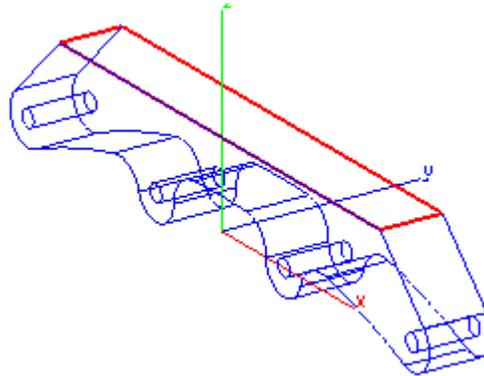
12. Save the file.

Exercise 2: Setting Up the Mounting Platform

In this exercise, 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.

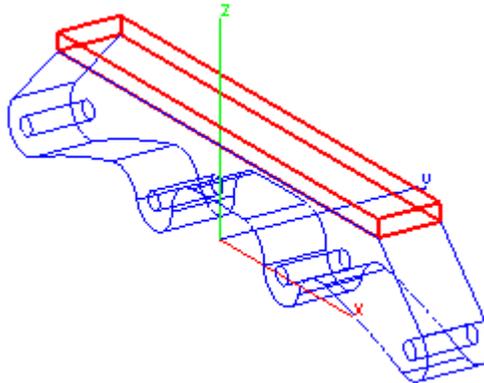
1. Extrude the top face of the chassis up 8 mm with a 10° draft.

- 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.



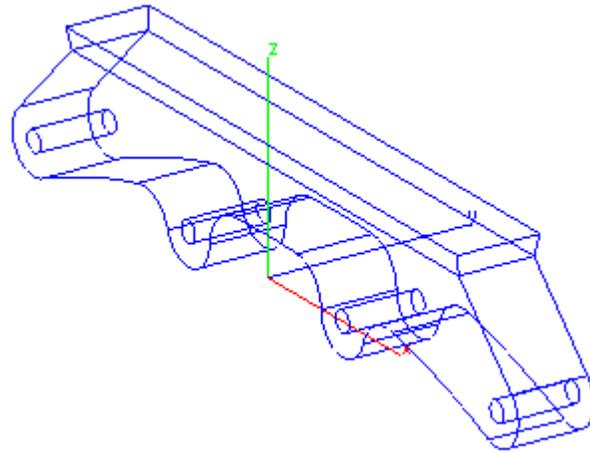
The top face extrudes.

- 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.



An 8 mm platform is created.

Use the scroll bar to adjust the drawing area if the entire chassis is not seen.

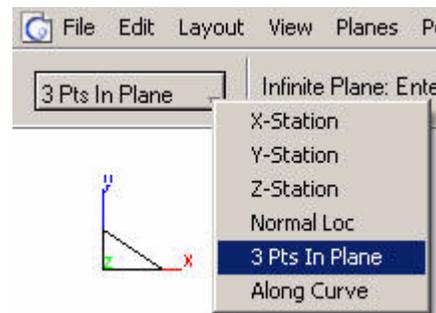


2. Trim the 8 mm extrusion to share the same plane as the sloped face on the front and back of the chassis.

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



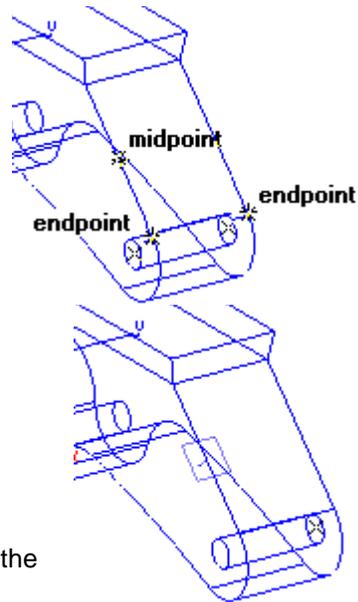
- In the Message Line, choose the *3 Pts In Plane* option from the pull-down menu.



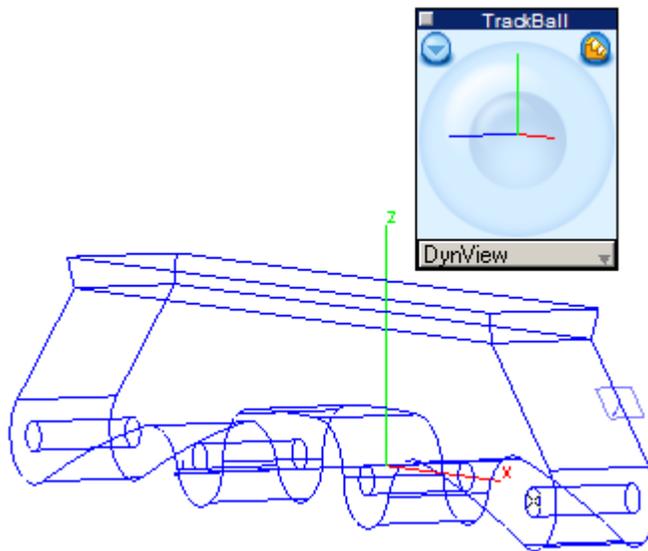
- Click three points on the sloped face to create an infinite plane at the same angle as the face.

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

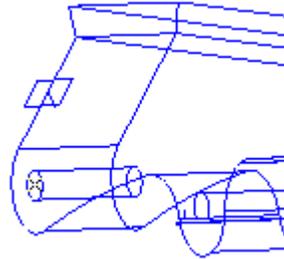
The infinite plane is created at the angle of the sloped face.



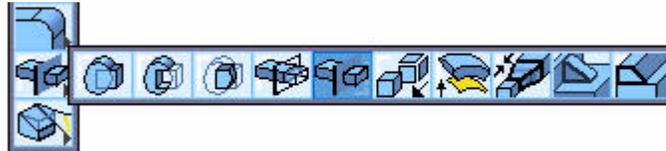
- Using the Trackball, rotate the chassis so that the back sloped face is visible.



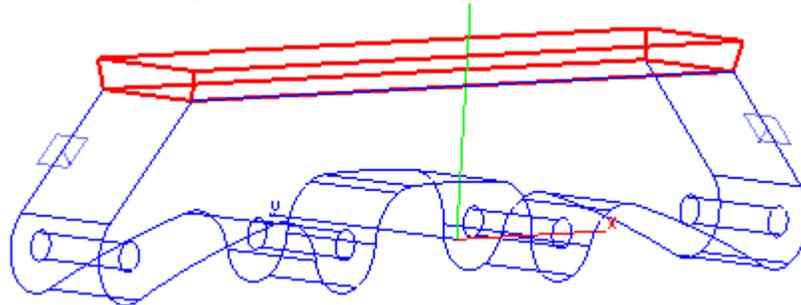
- Click three points on this face to create an infinite plane.



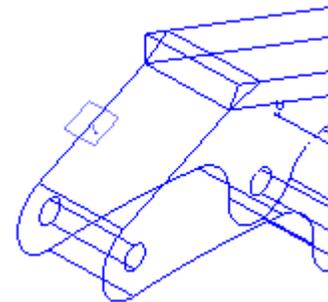
3. Split the 8 mm platform using the infinite plane on both sides of the chassis.
 - In the **Solids Utilities** tool palette, choose the **Split Solid** tool.



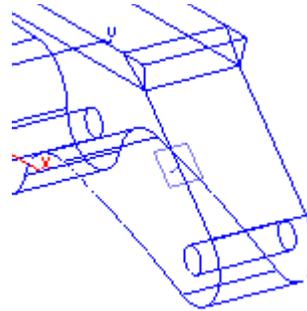
- Select the 8 mm platform created in step 1.



- 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 Trimetric.
- 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**.

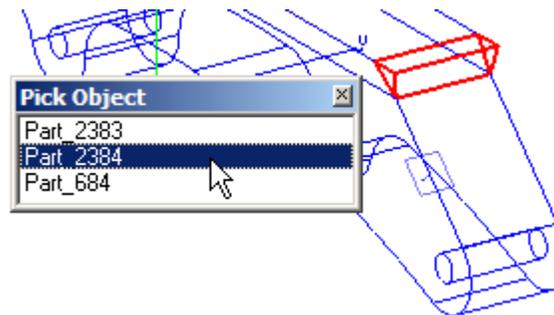


- Select *Hide* in the Show-Hide window.

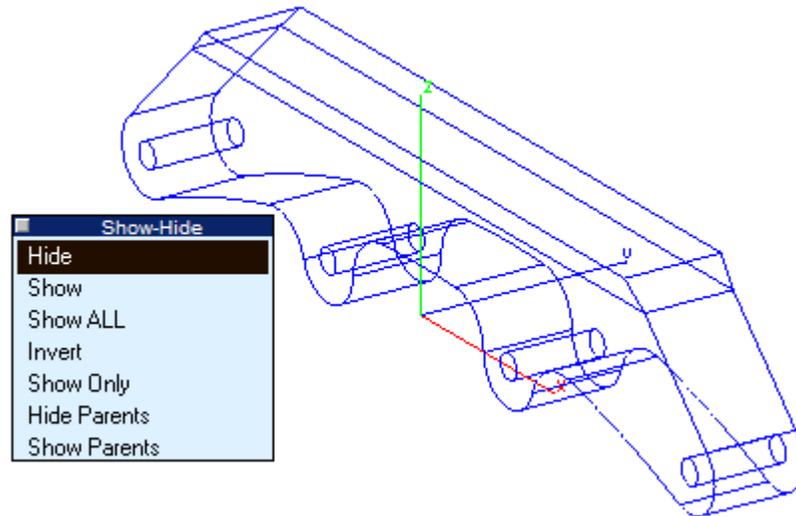
- Select the small split portion of the platform.

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.



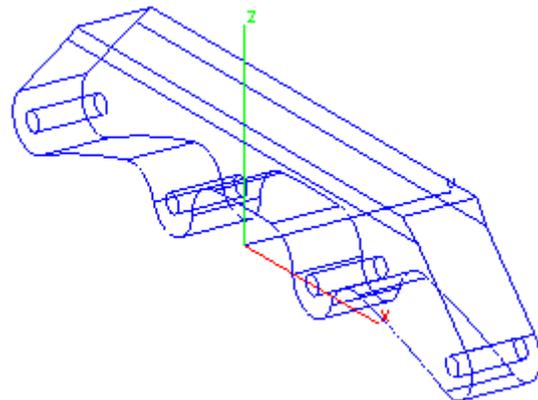
- In the Trackball menu, choose Trimetric.



- 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, create a cutout through the chassis using the *Snap Options* command, the **Parallel Line** tool, the **3-point 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 the screen, choose **View>Zoom All**.

2. Choose **Window>Snaps**.

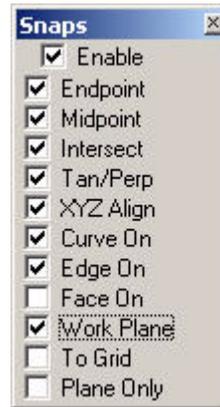
The Snap Options dialog box displays.

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

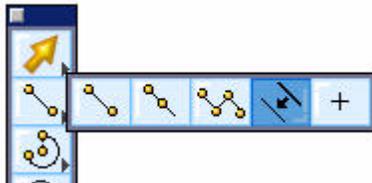
By choosing this option you can create curves in one 2D plane while using the chassis as a reference.

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.

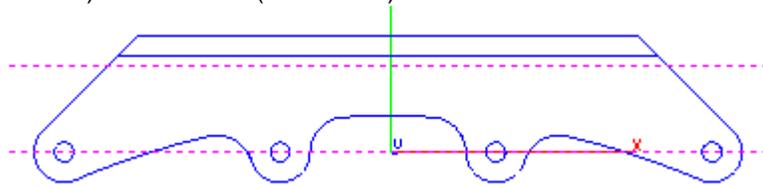


6. Create a new horizontal construction line offset from the axle center, 35 mm up.
 - In the **Line** tool palette, select the **Parallel Line** tool.

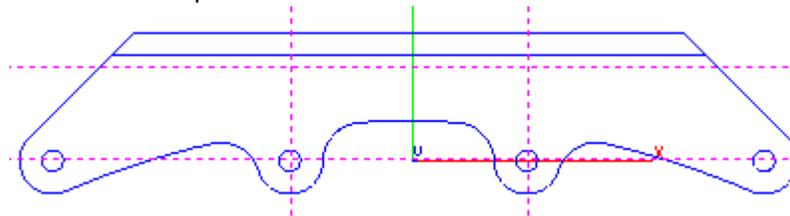


- Place the pointer over the construction line and drag a new line up a short distance from the original line.

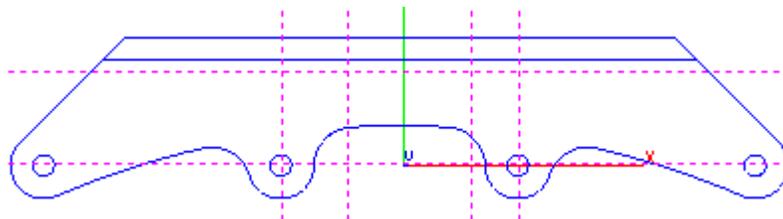
- Type **35** in the Offset data field of the Status Line and press ENTER (Windows) or RETURN (Macintosh).



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.

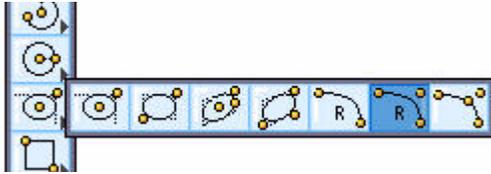


8. Using the **Parallel Line** tool, create two additional vertical construction lines. Place one 25 mm to the right of the left vertical construction line. Place another 18 mm to the left of the right vertical construction line.
 - Place the pointer over the left vertical construction line.
 - Drag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

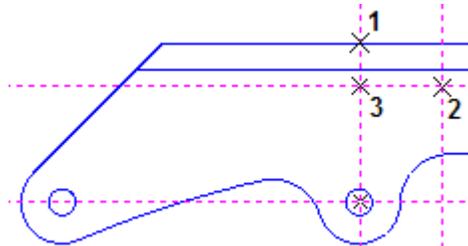


9. Create two conics using the construction lines as a guide.

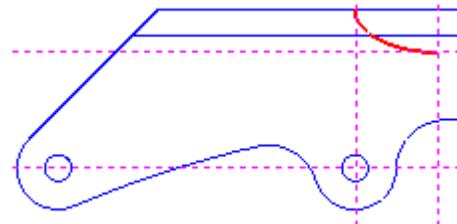
- In the **Ellipse/Conic** tool palette, choose the **3-point Conic** tool.



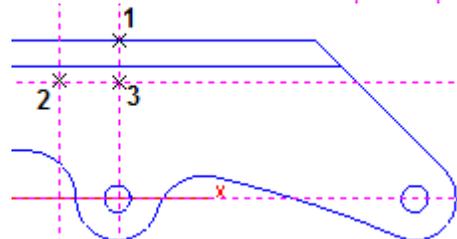
- Click the points for the left conic in the order (1, 2, 3) indicated in the graphic.



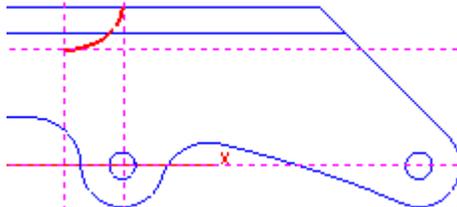
The conic appears.



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



A second conic appears.

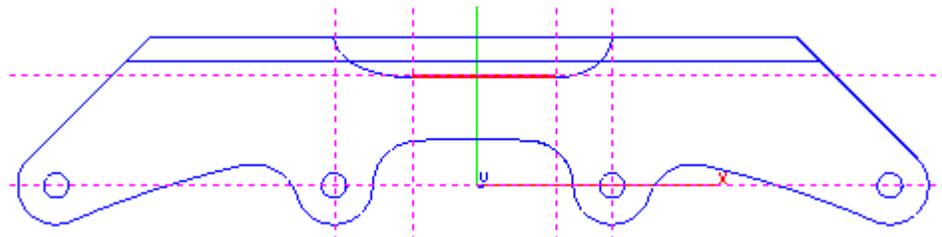


10. Connect the bottom of the conics with a line.

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



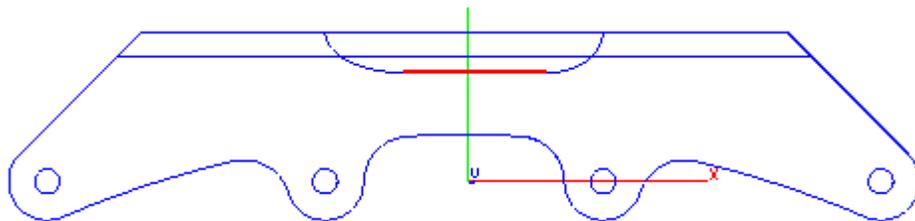
- Place the first point of the line at the lower point of the first conic.
- Place the end point of the line at the bottom of the lower point of the second conic.



11. Delete the construction lines.

- Choose **Layout>Delete Constructions**.

The constructions 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.

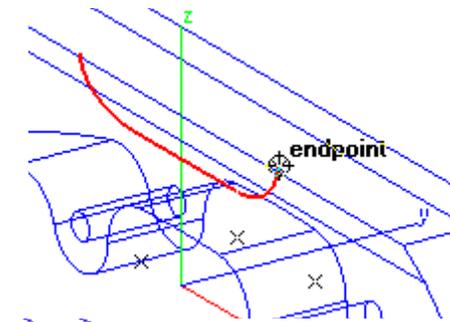
13. Now create a surface from the profile along the width of the platform.
- Change the view to Trimetric using the Trackball menu.
 - In the **Surface** tool palette, choose the **Sweep 2-Pt Surface** tool.



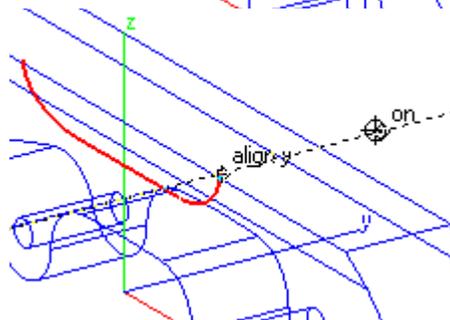
- Hold down the SHIFT key and select the two conics and the line just created.

The pointer becomes a cross hair.

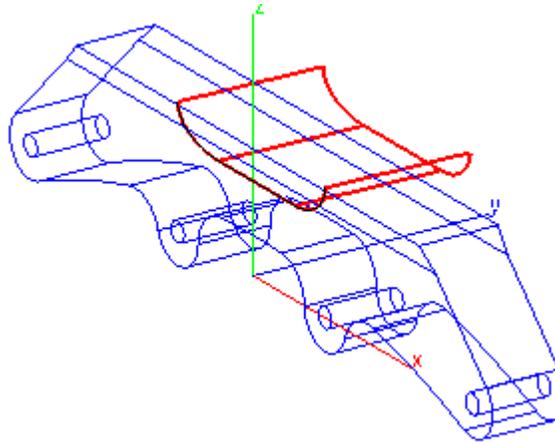
- Click on the endpoint of the conic.



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

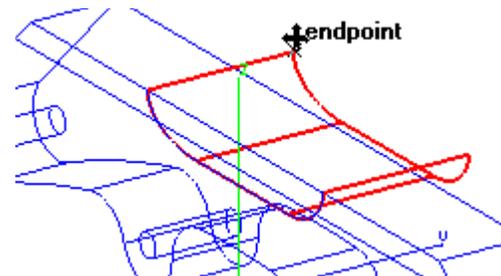


The surface is generated.

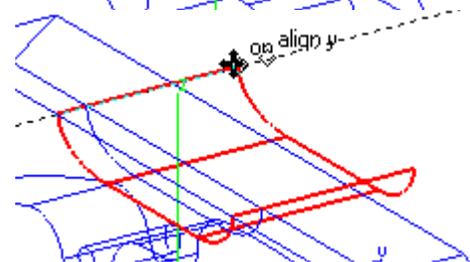


14. Drag the surface along the Y axis so that surface completely covers the top of the platform.

- 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.

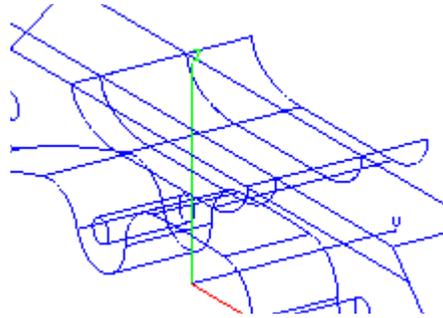
- In the **Solids Utilities** tool palette, choose the **Split Solid** tool.



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

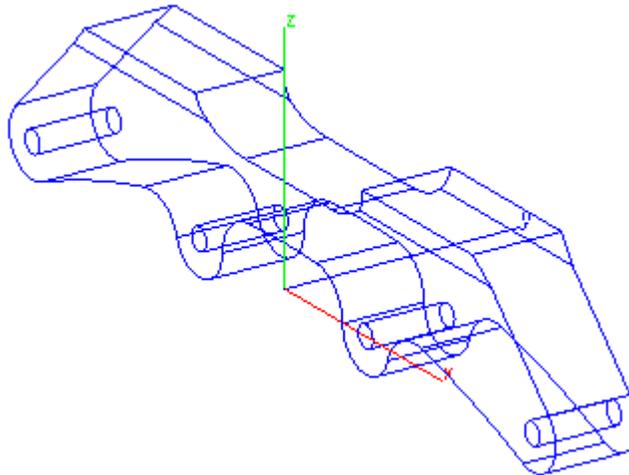
16. Show only the chassis.

- 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.



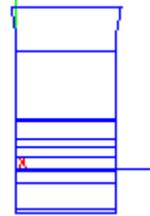
Close the Show-Hide window by clicking on the close button in its title bar.

17. Save the file.

Exercise 4: Creating the Chassis Cutout

Now 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.

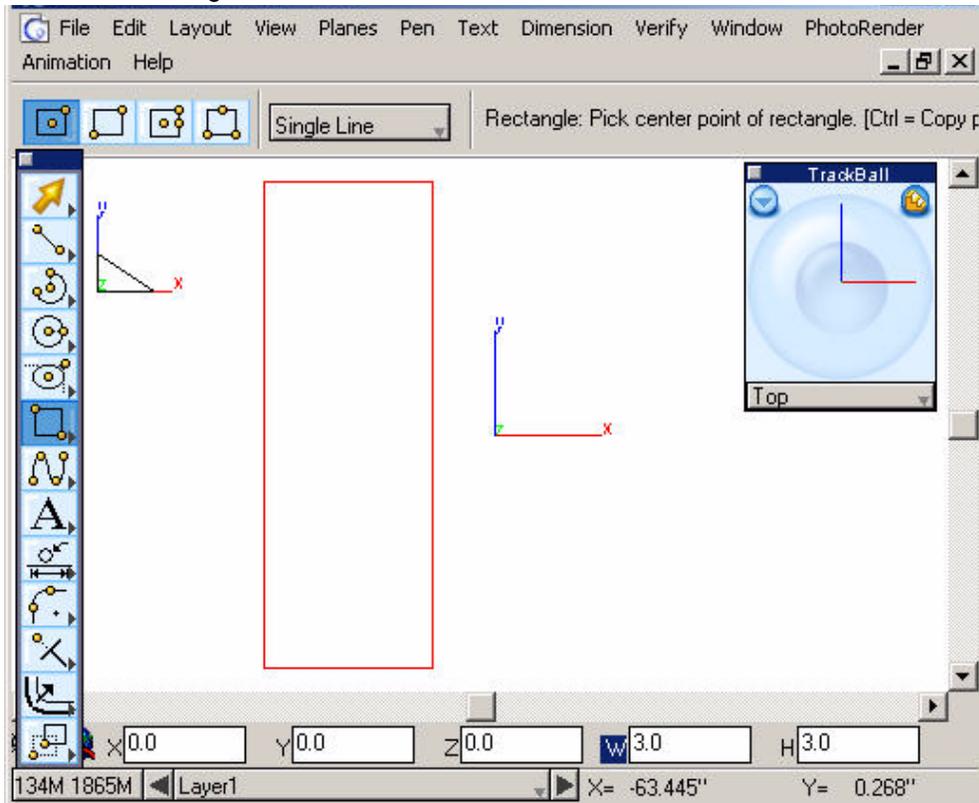


- Choose the **Center/Corner Rectangle** tool, the first tool in the subpalette.



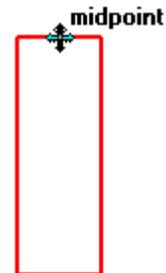
- Move the rectangle pointer to the left of the chassis and click to place the center point of the rectangle.
- Move the pointer away from the center and click anywhere to place the corner of the rectangle.
- Type **24.5** in the highlighted *W* (width) data field of the Status Line.
- TAB once to activate the *H* (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.

It doesn't matter which midpoint of the chassis the rectangle snapped to, as we'll soon see.

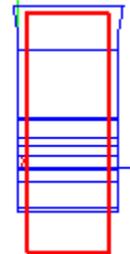
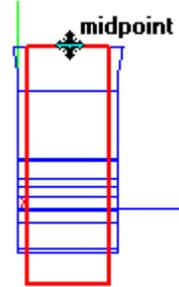
4. With the rectangle still selected, use the **Move** tool to move it -2.5 mm in the Z direction.
- In the **Transformation** tool palette, choose the **Move** tool.



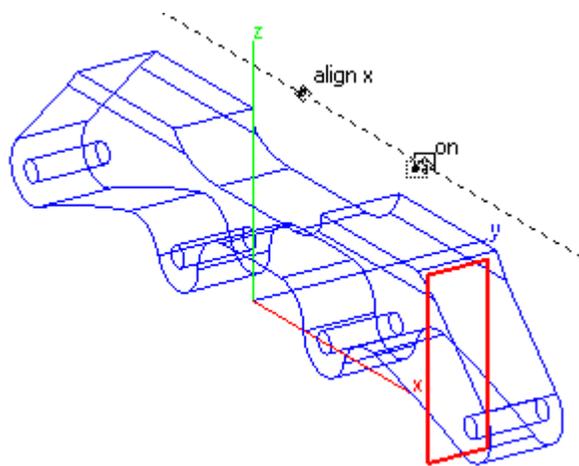
- Type **-2.5** in the dZ data field of the Status Line and press ENTER (Windows) or RETURN (Macintosh).

The rectangle moves down 2.5 mm.

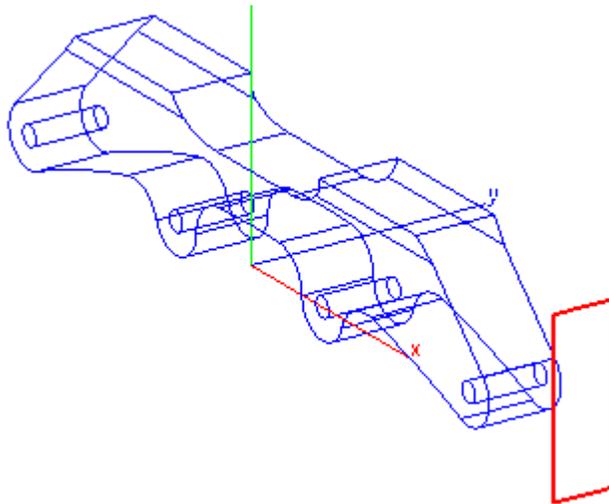
5. Change the view to Trimetric using the Trackball menu.
 6. Use the **Move** tool to move the rectangle along the X axis outside of the chassis.
- Click a point outside the chassis. It doesn't matter where because this point and the next are used as reference points for setting the move distance.



- 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.



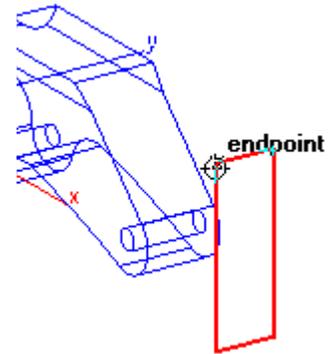
Check the Status Line to see that only the dX field contains a non-zero value. If the dY and dZ data fields contain values other than 0, type **0** in them and press ENTER (Windows) or RETURN (Macintosh).

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

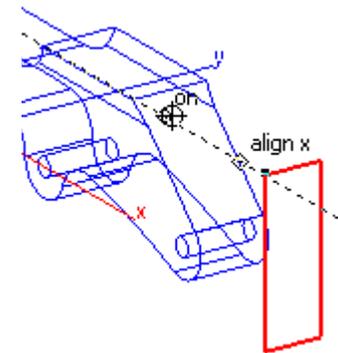
7. In the **Solids Features** tool palette, choose the **Cutout Feature** tool.



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.

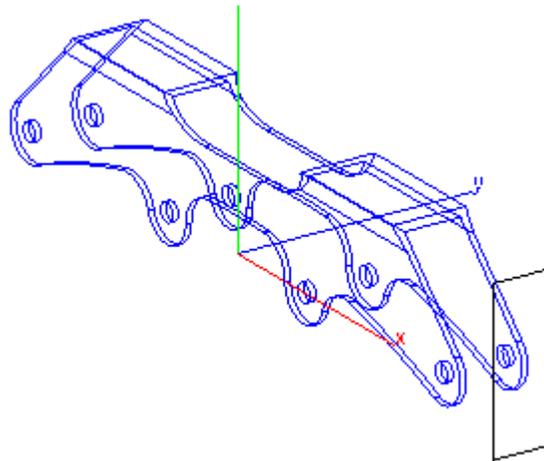
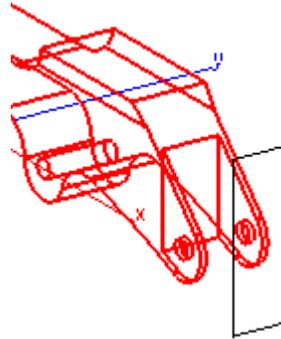
12. In the highlighted Distance field of the Status Line, type **500**.

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 the rectangle is moved in step 6. 500 should be long enough.

Make sure the Draft Angle is set to 0.

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

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



14. 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.

15. Place 3 mm blends on each side of the inside of the tunnel at each mounting plate.

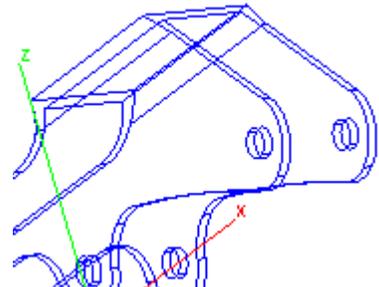
- In the **Solid Features** tool palette, choose the **Blend** tool.



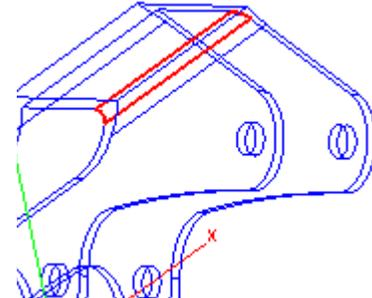
- In the Message Line, make sure the first tool, **Constant Blend**, is selected.



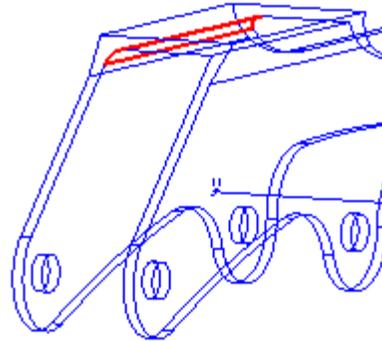
- Type **3** in the R (radius) data field of the Status Line.
- Rotate the chassis using the Trackball to better see the top right inside edge.



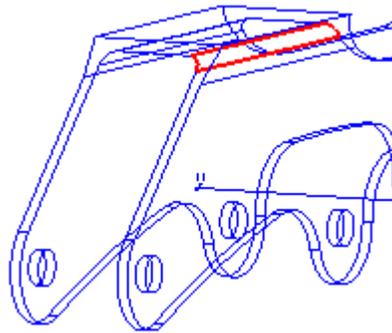
- Click on the right inside top edge of the tunnel.



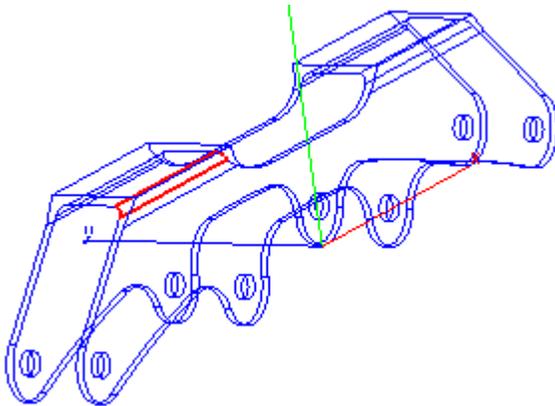
- Rotate the chassis to clearly see the left inside edge.



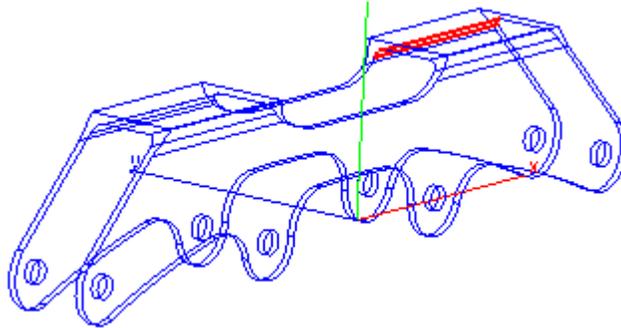
- Click on the left inside top edge of the tunnel.



- 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.
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.

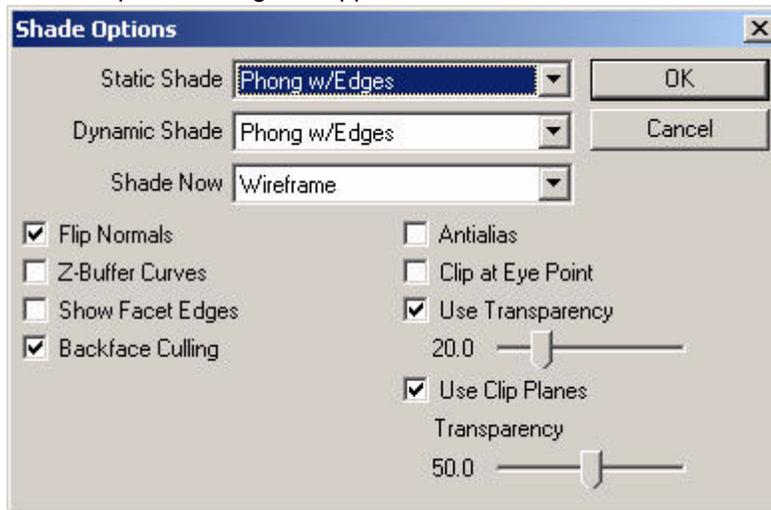
16. Save the file.

Exercise 5: Basic Rendering

Render the chassis using the basic rendering capabilities in Ashlar-Vellum Cobalt, Xenon or Argon.

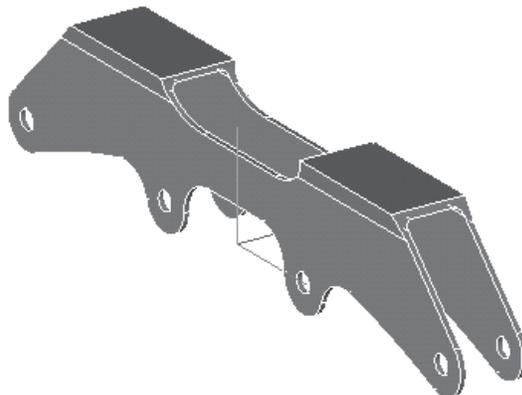
1. Choose **View>Shade Options**.

The Shade Options dialog box appears.

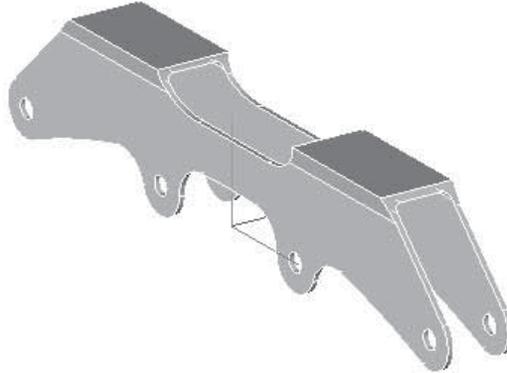


2. Use the pull-down menu to set the *Static Shade*, *Dynamic Shade* and *Shade Now* options to Phong w/Edges.
3. Click OK to save the settings and close the dialog box.
4. Choose **PhotoRender>Raytrace Preview Render [Shadows Off]**.

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.



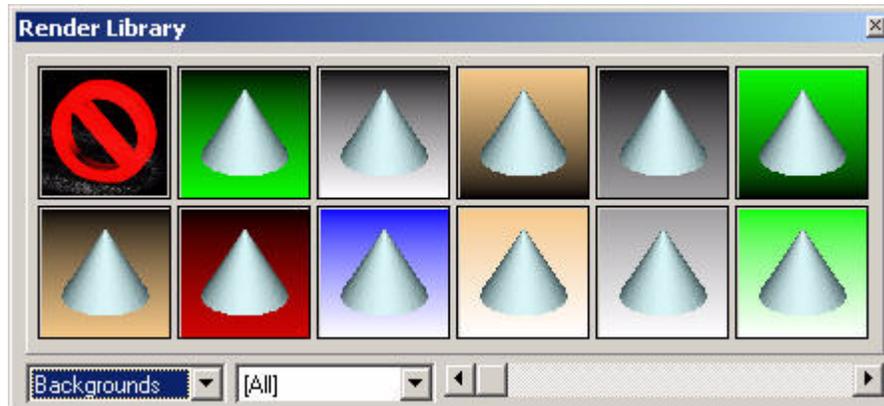
Rotate the chassis to see it from different angles.

Exercise 6: Advanced Rendering

Create more realistic visual affects by applying a material and displaying the model using photo-realistic rendering.

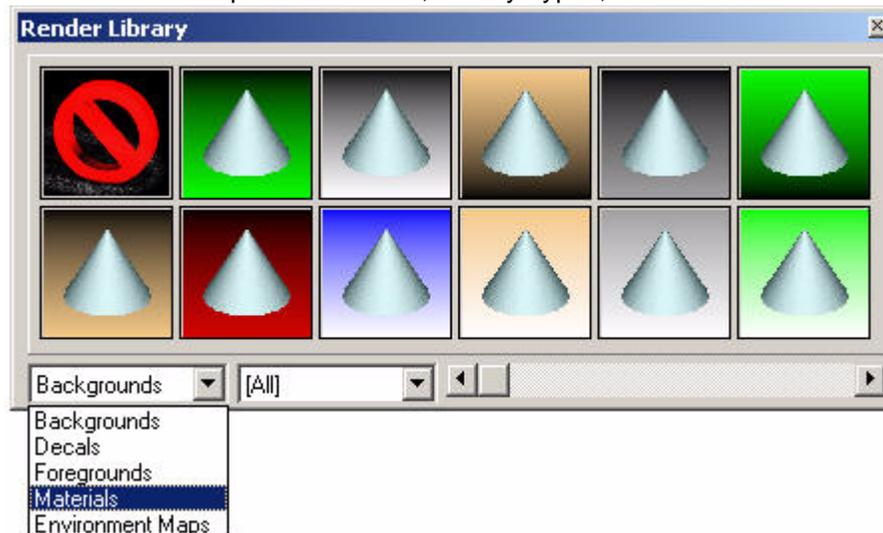
1. Choose **Window>Render Library**.

The Render Library dialog box displays on the screen.

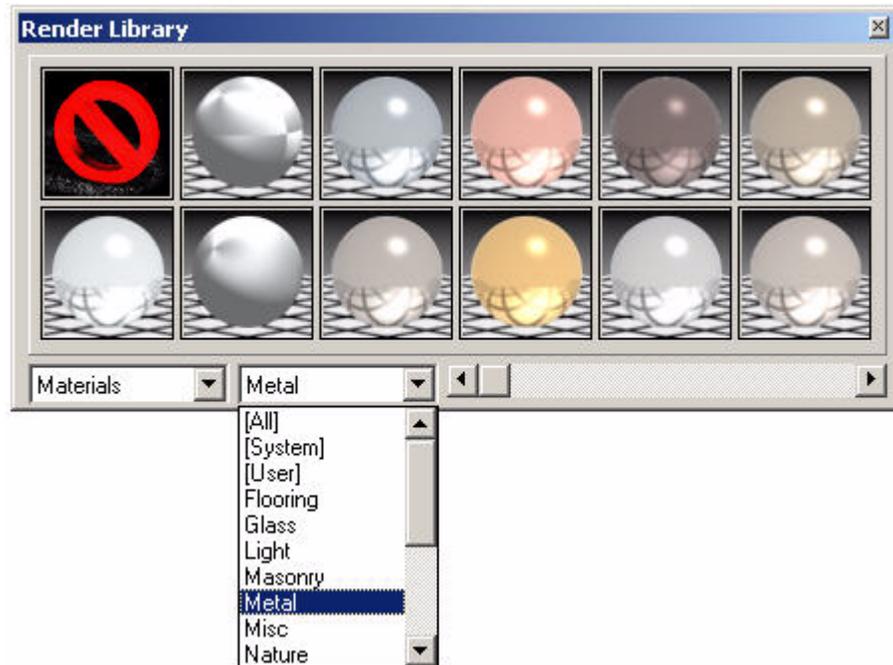


This library is divided into library types, including flooring, glass, masonry, metal and the like.

2. From the 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 first window in the second row. The title bar should display *Render Library -- Metal.Aluminum*.



5. Drag the pointer from the Render Library dialog box to the chassis. While dragging the pointer becomes an application symbol shown here.



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



Now render the chassis using photo-realistic rendering.

7. Choose **PhotoRender>Auto Full Render [Shadows On, Antialias]**.
The software redraws the chassis with the new rendering.



8. Apply different materials and textures to the chassis simply by selecting the material and dragging it to the chassis. Then view the effect by using one of the advanced rendering commands, *Raytrace Preview Render [Shadows Off]*, *Raytrace Preview Render [Shadows On]*, *Auto Full Render [Shadows Off]* and *Auto Full 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.

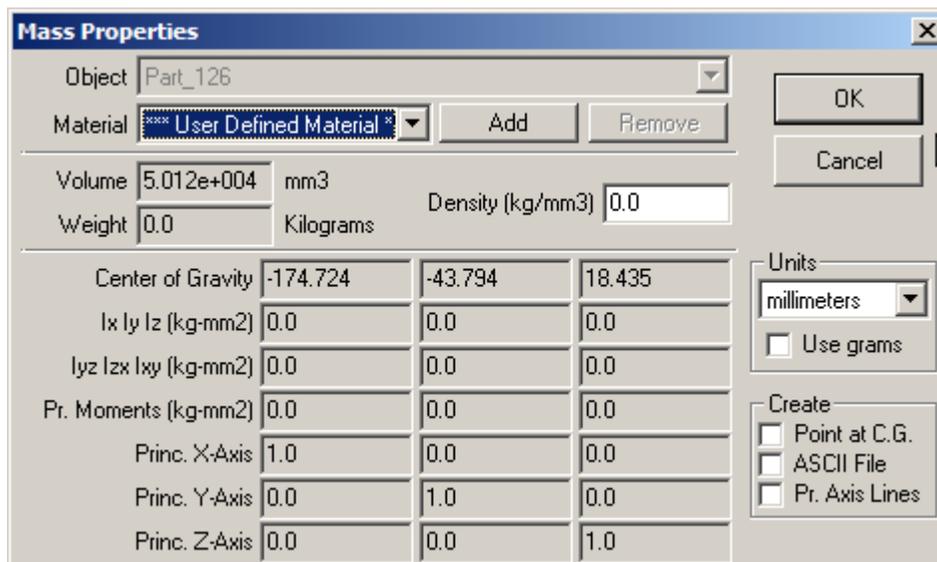
The basic inline skate chassis is modeled in the integrated design environment. But it is not finished yet.

Exercise 7: Calculating the Chassis' Physical Properties

Calculate a model's physical properties 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.

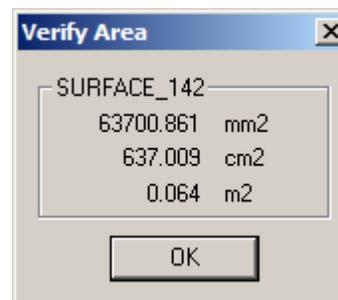
3. Choose **Verify>Area**.

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

Click OK to close the dialog box. Explore other physical properties if necessary.

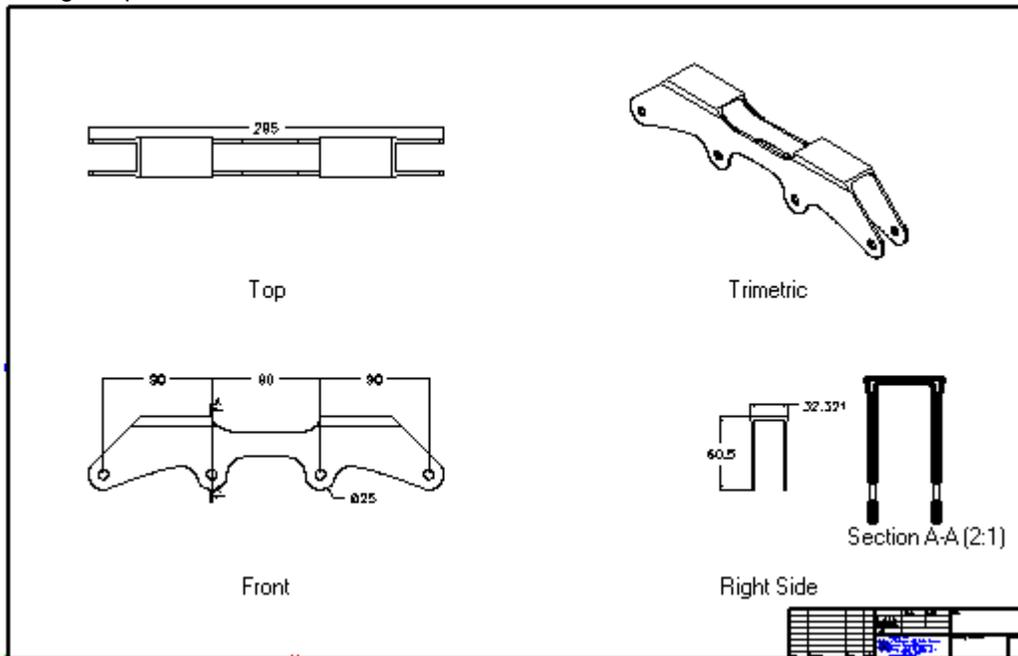
4. Save the file.

The inline skate chassis is completed. In the next chapter, produce a 2D drawing, add dimensions and create a section view.



The Detail Drawing

After creating the model, create production drawings. Display the model at various angles, place dimensions and even create a section cut of the model.



This chapter demonstrates how to:

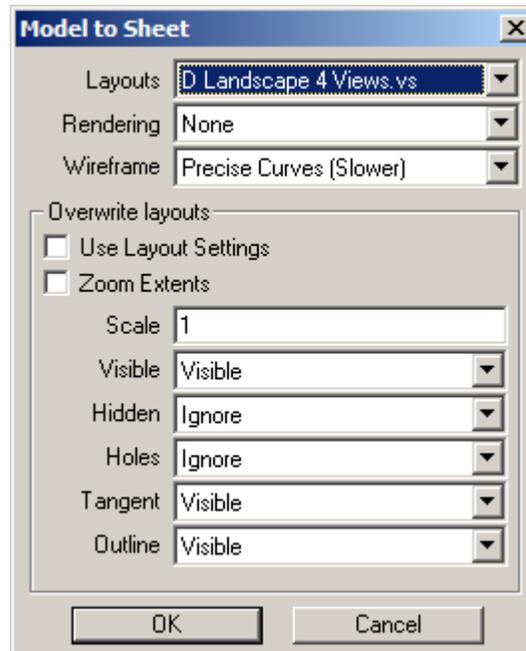
- Create a detail drawing
- Create a section view (Cobalt & Xenon only)
- Place dimensions

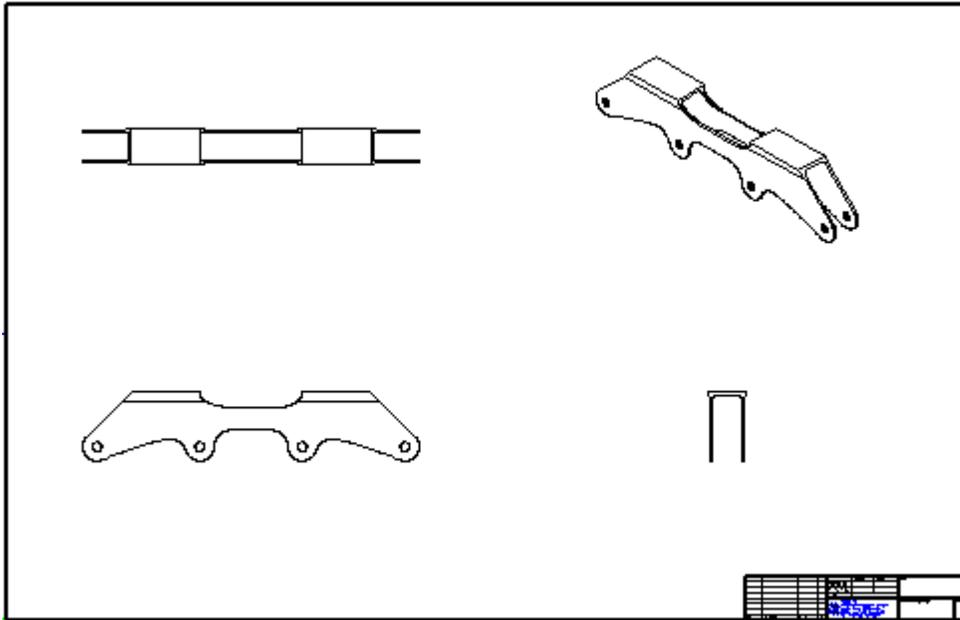
Exercise 1: Creating a 2D Detail Drawing

In this exercise, 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. Choose *None* option for *Rendering*.
6. The *Wireframe* option should be set to *Precise Curves (Slower)*.
7. In the *Scale* data field, accept the default of **1**.
8. 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 the wrong layout is inadvertently selected, choose **Edit>Undo**.

9. Save the file as *Inline Skate Production*.
 - Choose **File>Save As**.
 - In the File name data field, type **Inline Skate Production**.
 - Click **Save**.

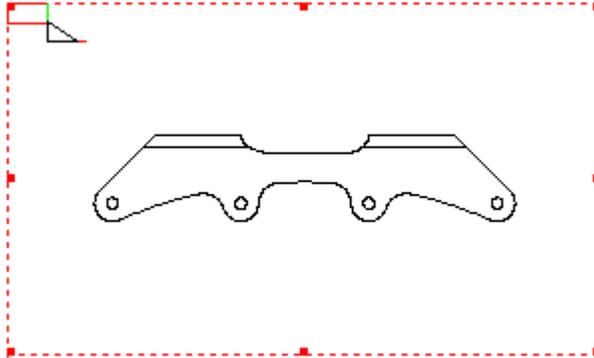
Exercise 2: Creating a Section View



The following exercise can be done only in the Cobalt and Xenon programs. Argon users should skip ahead to exercise 3.

Now add a section view to the drawing using the **Section** tool.

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



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

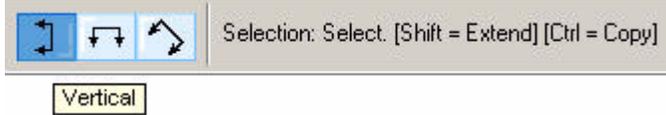


The **Sheet** tool palette displays in the drawing area.

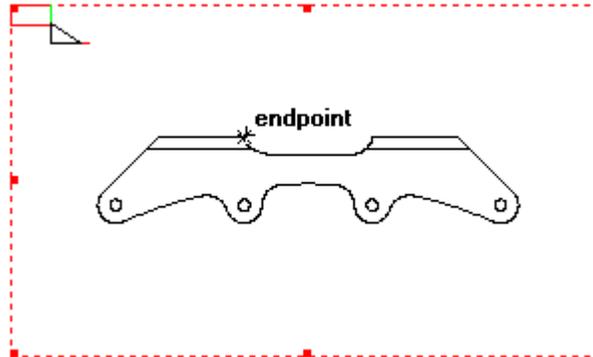
3. Choose the **Section** tool from the **Sheet** tool palette.



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

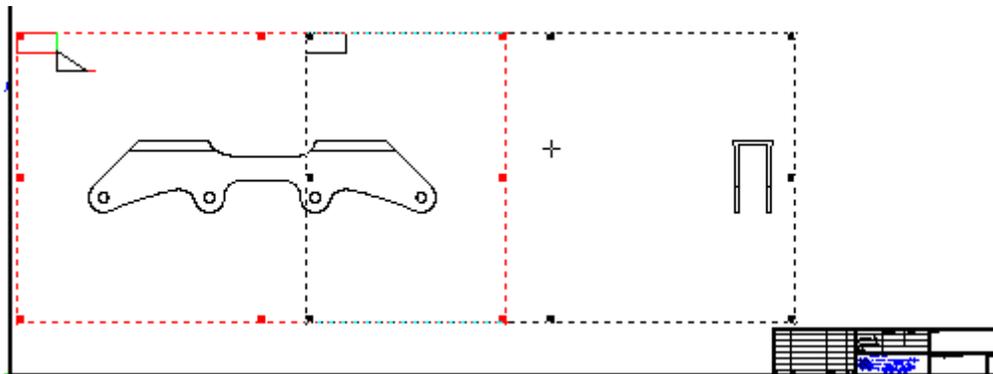


- Pick the view again.
- Click on the endpoint indicated in the graphic here.



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

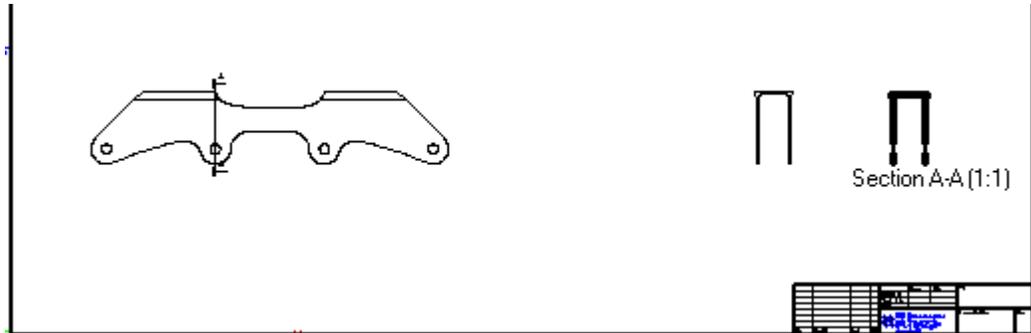
In this case drag it to the right over the title box at the bottom of the drawing. Notice that it seems like only the view boundary is dragged.



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

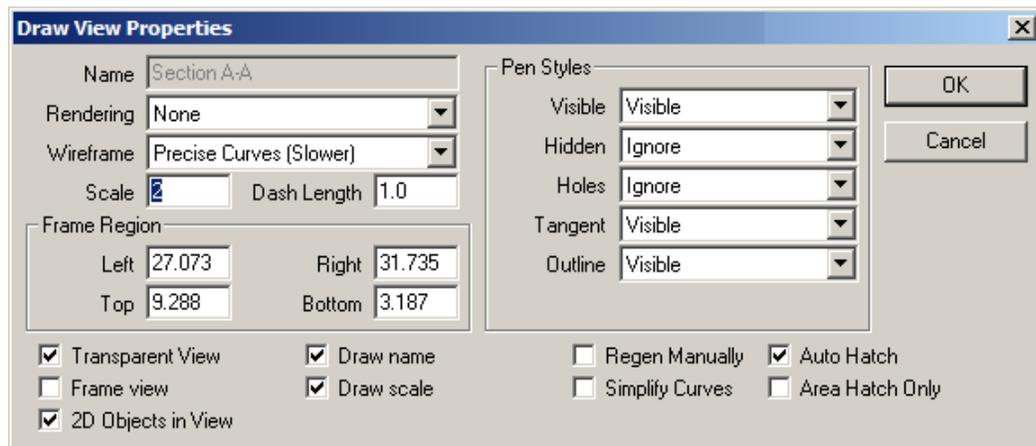
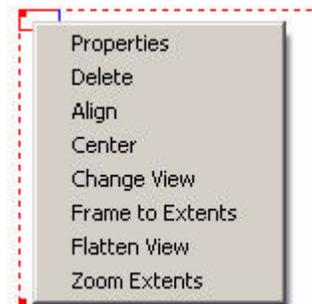


The chassis section appears, labeled Section A-A. Notice that the Front view is also labeled at the section location.



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.

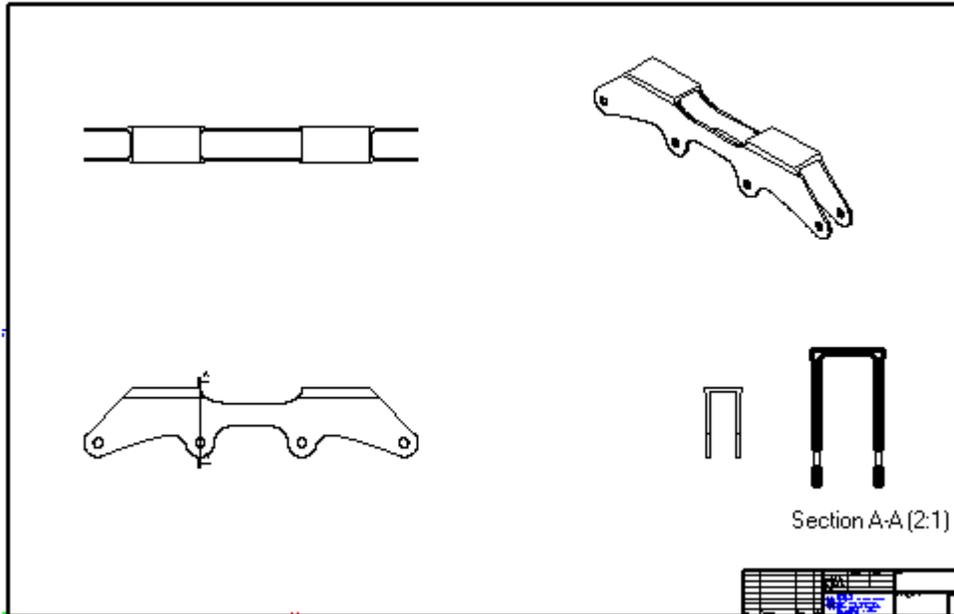
The Draw View Properties dialog box displays.



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

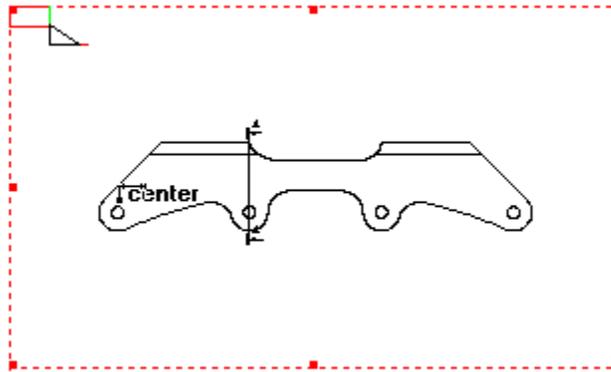
Since the section cut is not seen because of the scale change, it is necessary to resize the drawing view.

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

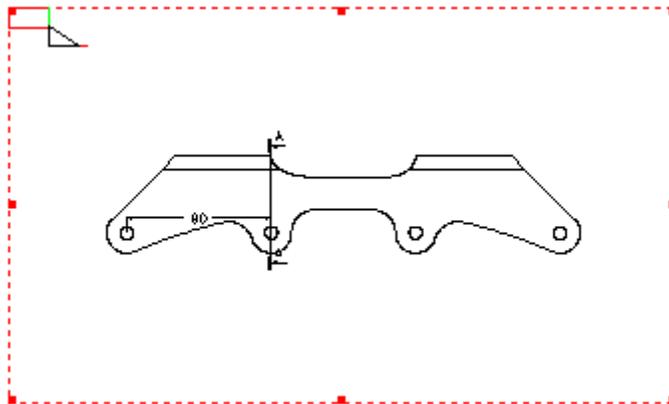


13. Close the **Sheet** tool palette.
14. Save the drawing.

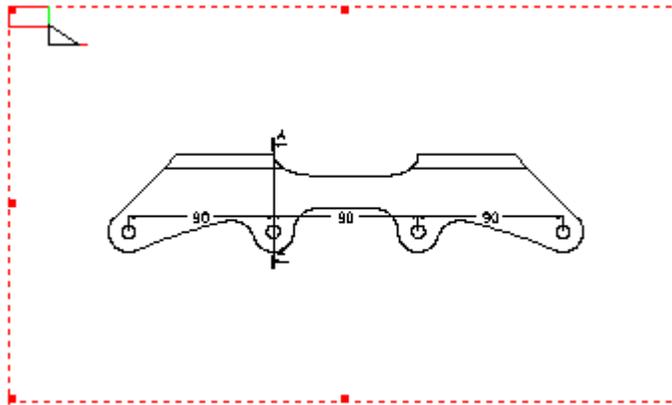
17. Move the hot spot of the pointer over to the left most axle of the chassis such that the center point notation appears.



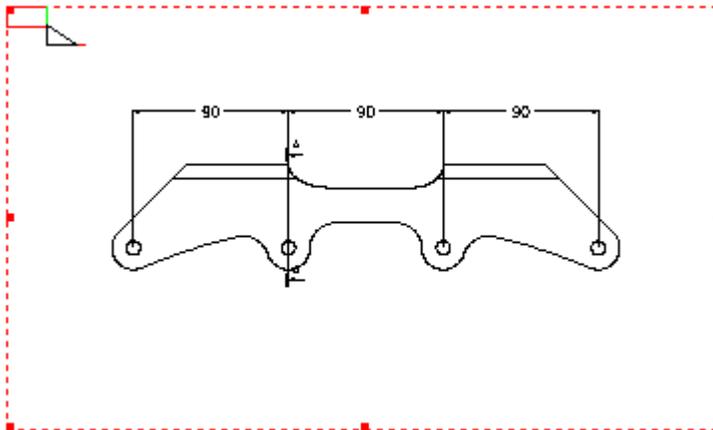
18. Click to place the first point.
 19. Move the pointer to the center of the axle hole to the right and click to place the next point.
- A dimension appears.



20. Click at the centers of the next two axles to finish placing the horizontal chain dimension.



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

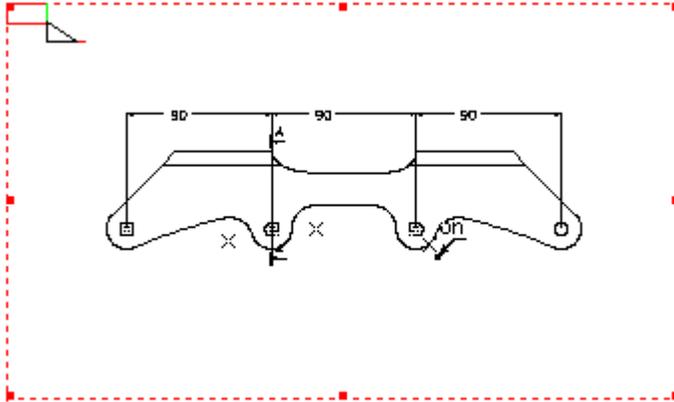


24. Select the **Diameter Arrow Out Dimension** tool from the **Dimension** tool palette.

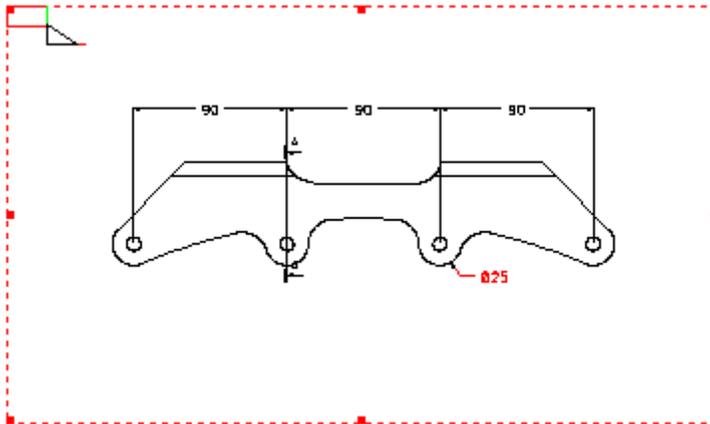
The pointer becomes the diameter arrow pointer.



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

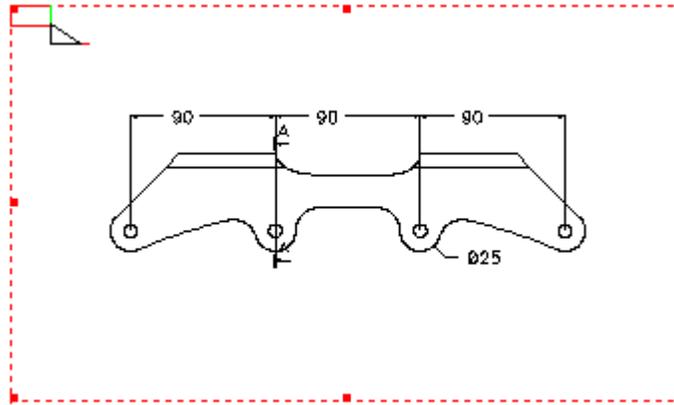


26. Click to place the dimension.

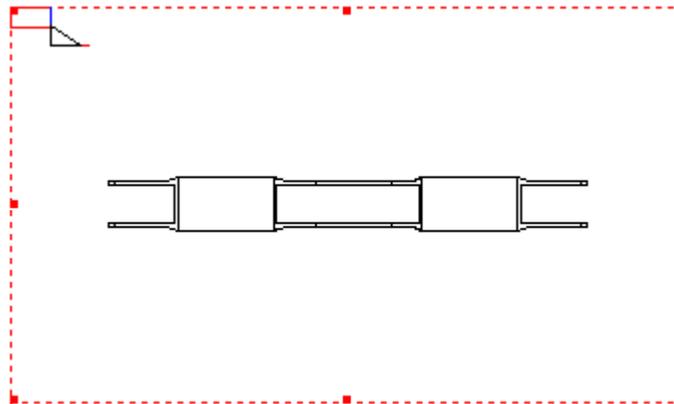


27. Choose the **Selection** tool and select the chain dimensions and the section dimension, in addition to the diameter dimension already selected.
28. Choose **Dimension>Size** and select 24.

The dimensions text resizes.

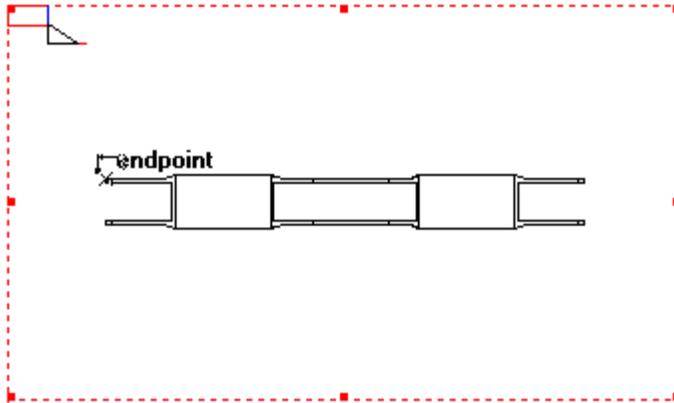


29. Choose the **Selection** tool and select the Top view.

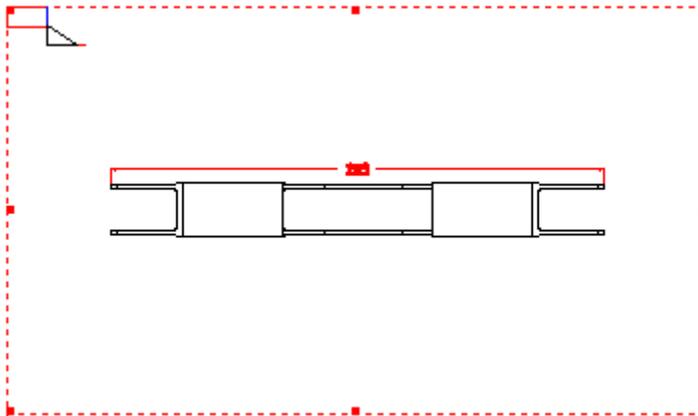


30. From the **Horizontal Dimension** tool subpalette, choose the **Horizontal Dimension** tool. (See page 5-8 for graphic illustration.) Choose **Horizontal** option from the Message Line.

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

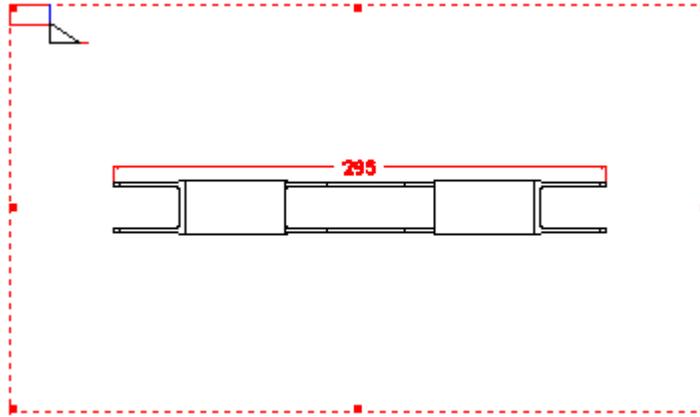


32. Click to place the first point.
33. Move the pointer to the endpoint at the opposite side of the chassis and click to place the other point.

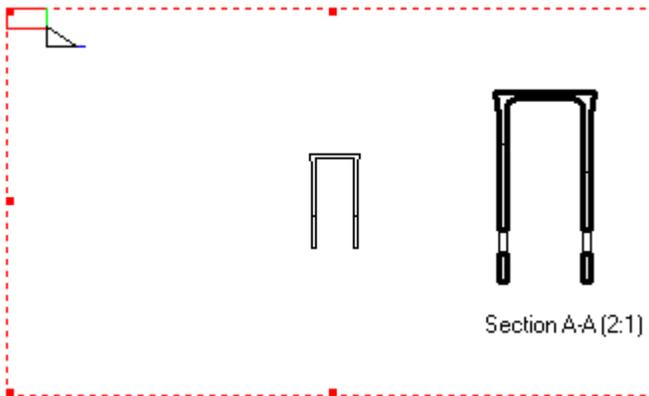


34. Choose **Dimension>Size** and select 24.

The dimensions text resizes.



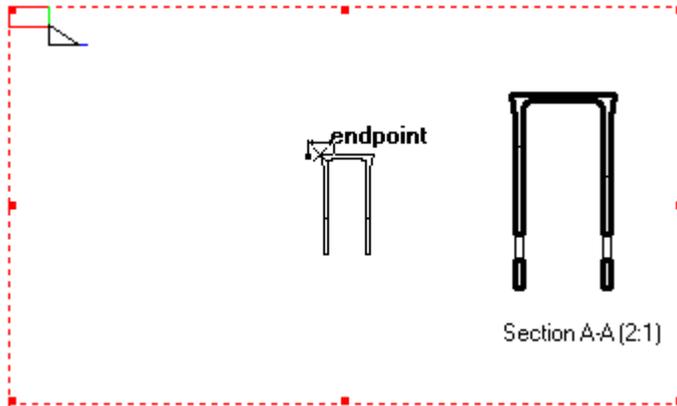
35. Choose the **Selection** tool and select the Right Side view.



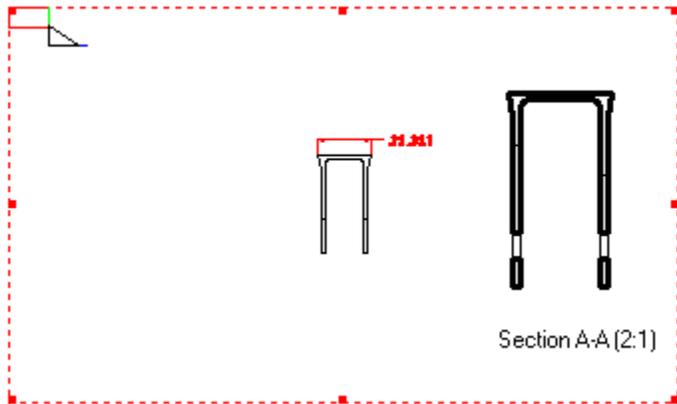
36. Select the **Horizontal Dimension** tool again.



37. Click the top left endpoint of the chassis.



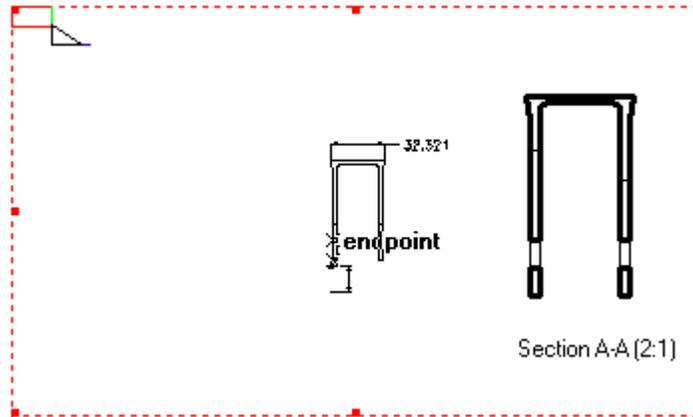
38. Click the top right endpoint and the dimension appears.



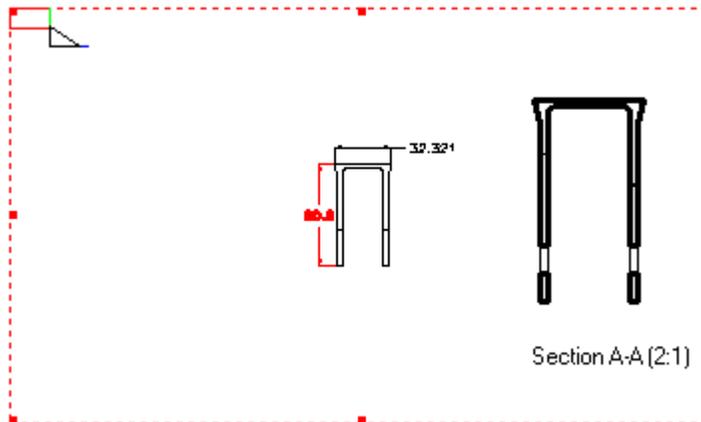
39. Choose the **Vertical Dimension** tool, as shown here.



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



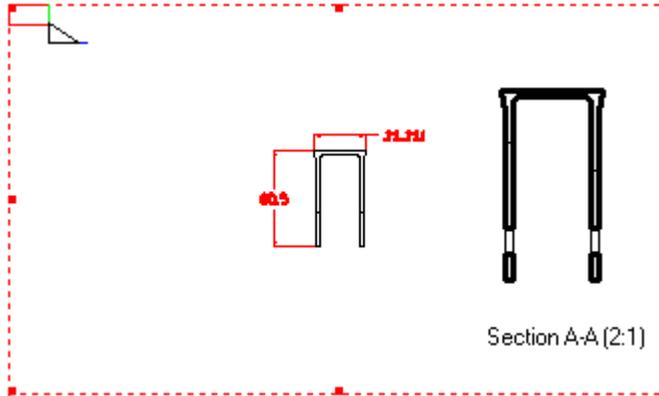
41. Click the second point at the top left endpoint.



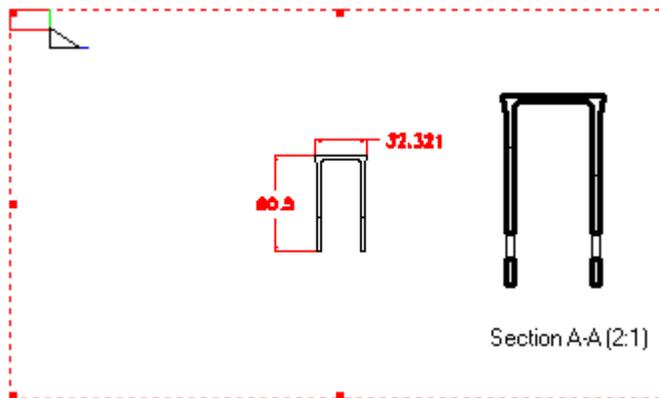
42. Place the pointer over the center of the text so that a four way arrow appears.



43. Drag the dimension to the left so that it does not cover the part.

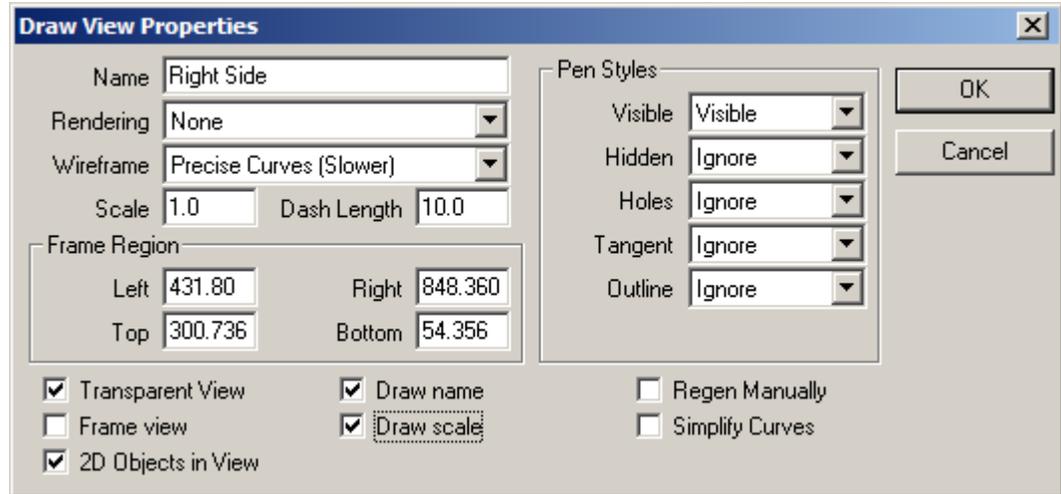


44. Choose the **Selection** tool.
45. Hold down the SHIFT key and select the horizontal dimension in addition to the vertical dimension still selected.
46. Choose **Dimension>Size** and select 24.
- The dimensions text resizes.



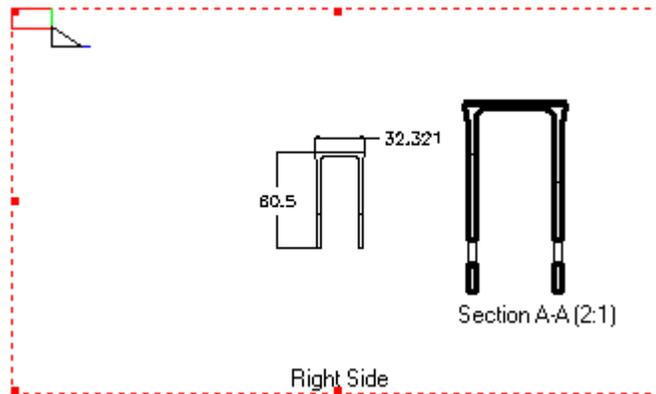
47. To complete the tutorial, add view names to the drawing views.
- Click on the top left corner of the drawing view to display the Drawing View menu.

- Choose the *Properties* command and the dialog box displays.



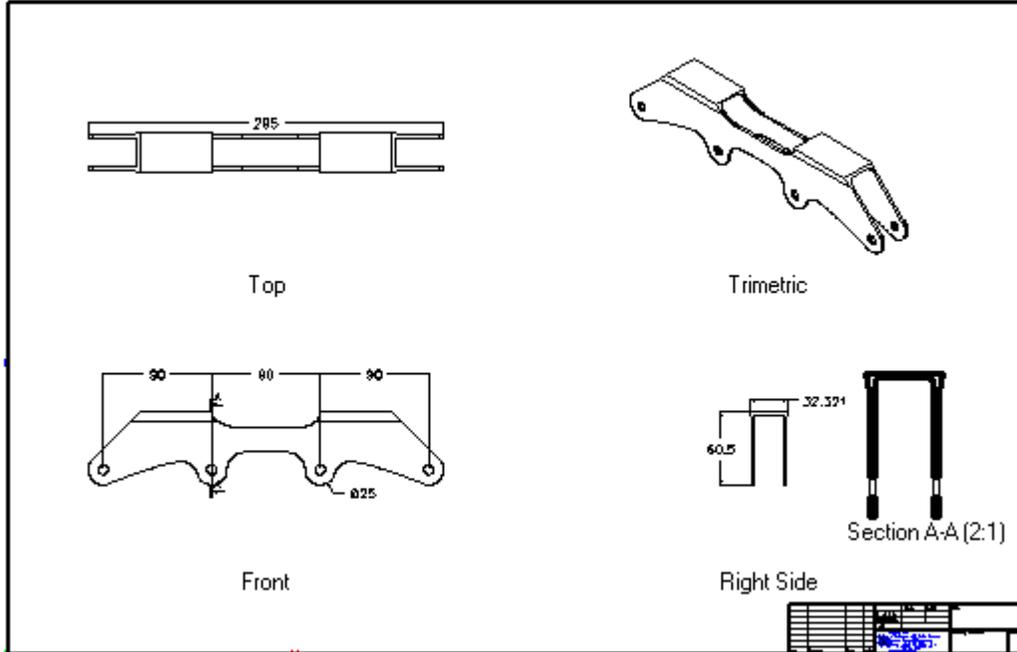
- Select the *Draw name* option. Draw scale option can be chosen also.
- Click OK.

The dialog box closes and the name, Right Side, and scale 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! A 2D detail drawing with a section view and dimensions is generated. You've also completed the Ashlar's 3D modeling products tutorial.

Index

Numerics

- 2-Entity Fillet tool 3-19
- 2-Point Center Ellipse tool 2-24
- 3-Point Arc tool 3-14
- 3-Pt Conic tool 4-13

A

- Animation 2-3
- Anti-alias 4-31
- Arc tools 2-4
 - 3-Point 3-14
- Area 4-33
- Ashlar-Vellum Modeling Products 1-1
 - Argon 1-1
 - Cobalt v5 1-3
 - Xenon 1-2
- Authorization code 1-4

B

- Blend tool 4-24

C

- Center/Corner Rectangle tool 3-5, 4-18
- Center-Point Circle tool 2-20, 3-8
- Chamfer tools 2-6
- Circle tools 2-5
 - Center-Point 2-20, 3-8
 - Opposite-Point 2-6, 2-9
- Color
 - Edge 2-19
 - Pen 4-28
 - Preferences 2-17
- Conic tools 2-5
 - 3-Pt 4-13
- Connected Lines tool 2-13
- Constant Blend tool 4-24
- Construction line
 - Command 3-15
 - Delete 3-18, 4-14

- Dynamic 2-12

- Stroke 3-13

- Curve Extras tools 2-6

- Divide at Location 2-21

- Simple Trim 3-20

- Cutout Feature tool 4-22

D

- Delete construction lines 3-18, 4-14

- Detail drawing 5-1

- Layouts 5-3

- Scale 5-3

- Section tool 5-4

- Section view 5-4

- Diameter Arrow Out Dimension tool 5-10

- Dimension

- Diameter Arrow Out 5-10

- Horizontal 5-12

- Horizontal Chain 5-8

- Menu 2-3

- Size 5-11

- Tool palette 5-8

- Vertical 5-15

- Display (preferences) 2-17, 3-3

- Divide at Location tool 2-21

- Documentation 1-4

- Style conventions 1-7

- Drafting Assistant 2-9

- Construction line 2-12

- Preferences 2-17

- Rubberband image 2-24

- Snap points 2-10

- Snaps command 4-11

- Drawing techniques 2-24

- Drawing view

- Frame to Extents 5-7

- Name 5-18

- Properties 5-6

- Scale 5-6

- Dynamic construction line 2-12

E

Edit
 Menu 2-2
 Objects 2-21, 2-23
Ellipse tools 2-5
 2-Point Center 2-24
Endpoint 2-10
Environment (Vellum Solids) 2-1
Extrude Solid tool 4-2

F

File
 New 3-2
 Save 3-12
Fillet tools 2-6
 2-Entity 3-19
Frame to Extents command 5-7

G

Grid 2-17

H

Helix Curve tool 2-5
Help 2-3
Hide (Show) command 4-9
Horizontal Chain Dimension tool 5-8
Horizontal Dimension tool 5-12
Hot spot 2-6

I

Infinite Plane tool 4-6
Intersect (Drafting Assistant) 2-10
Iso lines 2-18

L

Layout
 Menu 2-2
 Model to Sheet 5-3
Line tools 2-4
 Connected 2-13
 Parallel 3-13, 4-11
 Single 2-9, 2-11, 3-16, 4-14
Linear Duplicate tool 3-11

M

Margin notes 1-7
Menu 1-7
 Dimension 2-3
 Edit 2-2
 File 2-2
 Layout 2-2
 Pen 2-3
 Planes 2-3
 Render 2-3
 Text 2-3
 Verify 2-3
 View 2-3
 Window 2-3
Menu bar 2-2
Message Line 2-7
Midpoint 2-10
Mirror tool 3-17
Model to Sheet command 5-2
Modeling
 Solid 4-1
 Wireframe 3-1
Move tool 4-20

O

Object
 Area 4-33
 Editing 2-21, 2-23
 Selecting 2-20
 Show-Hide 4-9
Offset
 Construction line 3-15
 Linear duplicate 3-11
 Parallel line 3-13
Opposite-Point Circle tool 2-6, 2-9

P

Parallel Line tool 3-13, 4-11
Pen
 Color 4-28
 Menu 2-3
 Pattern 2-23

- Phong w/Edges 4-27
- Photorealistic rendering 4-31
- PhotoRender 4-27
 - Advanced 4-29
 - Basic 4-27
 - Library 4-29
 - Materials 4-29
 - Menu 2-3
 - Phong w/Edges 4-27
 - Raytrace 4-31
- Plane
 - Infinite 4-6
 - Project to Work Plane 4-11
- Planes menu 2-3
- Pointer
 - Smart 2-6
 - Stroke 3-13
- Polygon tools 2-5
 - Rectangle 3-4, 4-18
- Preferences 3-2
 - Color 2-17
 - Display 2-17, 3-3
 - Drafting Assistant 2-17
 - Filing 2-17
 - Grid 2-17
 - Save 2-17
 - Select 2-17
 - Units 2-17, 3-2
- Properties
 - Area 4-33
 - Drawing view 5-6
 - Model 4-33
 - Section view 5-6
- Q**
- Quadrant 2-10
- R**
- Raytrace rendering 4-31
- Rectangle tool 3-4, 4-18
- Render
 - Anti-alias 4-31
 - Photorealistic 4-31
- Resolution 2-18
 - Settings 3-3
- Rubberband image (Drafting Assistant) 2-24
- S**
- Save
 - File 3-12
 - Preferences 2-17
- Scale 5-3, 5-6
- Section tool 5-4
- Section view 5-4
 - Properties 5-6
- Selection
 - Objects 2-20
 - Preferences 2-17
 - Tool 2-4, 2-11
- Serial number 1-4
- Sheet tools 5-5
- Show-Hide command 4-9, 4-17
- Silhouette 2-19
- Simple Trim tool 3-20
- Single Line tool 2-9, 2-11, 3-16, 4-14
- Smart pointer 2-6
- Snap
 - Command 4-11
 - Drafting Assistant 2-10
 - Points 2-10
 - Project to Work Plane 4-11
- Solid modeling 4-1
- Solid tools
 - Constant Blend 4-24
 - Cutout Feature 4-22
 - Extrude 4-2
 - Split 4-16
 - Split Solid 4-8
 - Union 4-10
- Solids tool palette 4-2
- Sphere Trackball 2-8
- Spline tools 2-5
- Split Solid tool 4-8, 4-16

- Spot (hot) 2-6
- Status Line 2-7
- Step Trackball 2-8
- Stroke
 - Construction line 3-13
 - Pointer 3-13
- Submenu 1-7
- Surface tools
 - Infinite Plane 4-6
 - Sweep 4-15
- Surfaces tool palette 4-2
- Sweep Surface tool 4-15
- System Requirements 1-5

T

- Text
 - Menu 2-3
 - Tools 2-5
- Tool palette
 - Dimension 5-8
 - Solids 4-2
 - Surfaces 4-2
 - Wireframe (main) 2-4
- Trackball
 - Sphere 2-8
 - Step 2-8
 - View rotation options 2-8
- Transformation tools 2-6
 - Linear Duplicate 3-11
 - Mirror 3-17
 - Move 4-20
- Trim tools 2-6
- Tutorial Introduction 1-6

U

- Union Solid tool 4-10
- Units 2-17, 3-2
- User interface 2-1

V

- Verify menu 2-3
- Vertex 2-11

- Vertical Dimension tool 5-15
- Vertical Section tool 5-5
- View menu 2-3

W

- Window menu 2-3
- Wireframe modeling 3-1
- Wireframe tool palette 2-4
- Work plane 4-11

Z

- Zoom All 3-6, 4-11