Roland®

FUELING THE ROLAND UNIVERSITY PASSION

Transforming Ideas Into Reality

Product Design & Manufacturing I

Design 3D prototype models using Rhinoceros software. Produce actual prototypes using Roland Modela Player software and the Roland MDX-15 milling machine.

Teaching Module

doMORE™

Product Design & Manufacturing I

Product Design & Manufacturing I

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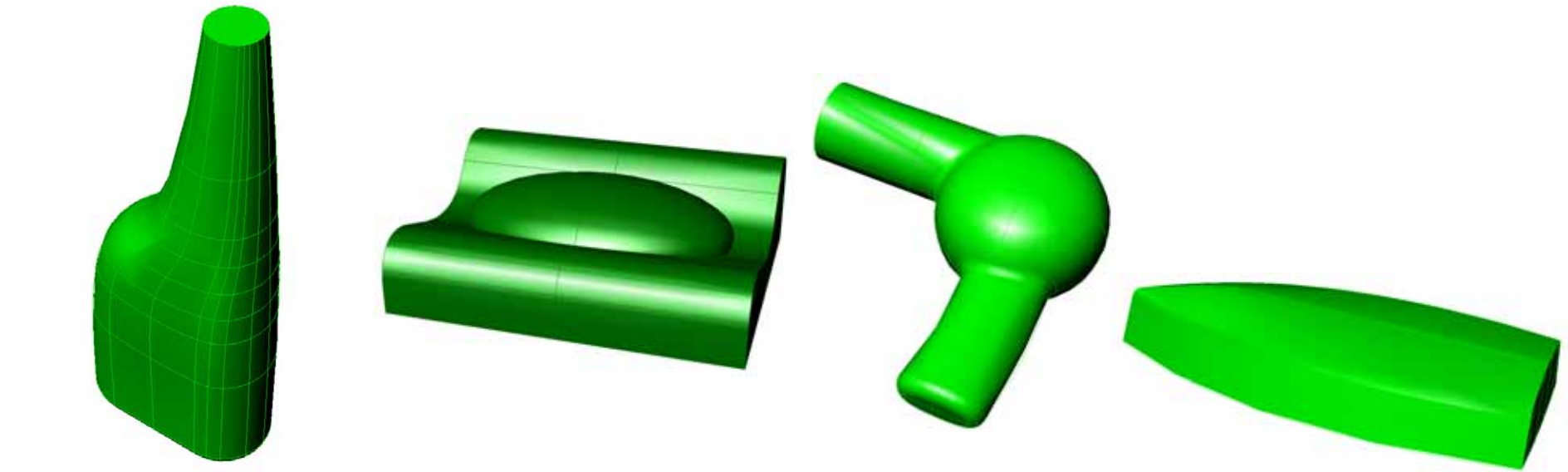


Overview

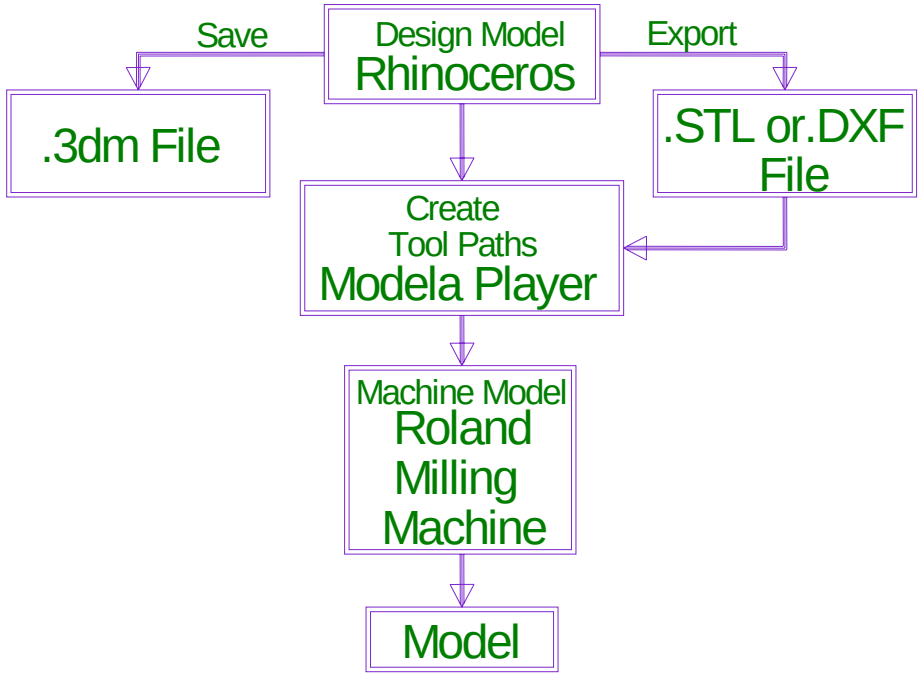
Welcome to the Roland DGA Product Design & Manufacturing I Curriculum Series. The purpose of Series 1 is to teach students the basics of Rhinoceros software to design 3D prototype models and machine these models using Roland Modela Player software and Roland milling machines, in particular the MDX-15. Further series will be oriented towards more specific industrial applications.

Included is a set of 9 lessons. The first lesson teaches the basics of using Rhinoceros software. Lesson 2 models a bottle and Lesson 3 creates the toolpaths for the bottle. Lesson 4 models a desk weight using solids and its companion, Lesson 5, creates the toolpaths. A hair dryer is created and machined in Lesson 6 and 7. Lessons 8 and 9 model and machine a boat. The lessons are designed to be completed in 10 one-hour class periods. If your students are already proficient with Rhino, they can skip Lesson 1 that is designed to be completed in 2 class periods.

As a student proceeds through the lessons, steps and procedures that they have already done and should have mastered will be explained in less detail. It is very important that you emphasize with your students that they don't go through the lesson like a robot but instead learn how to do each procedure so they can use these skills to create their own models.

All of the parts are machined using machining wax. This material is easily cut and has a relatively high melting temperature. Candle wax cannot be used, as it would melt on the machine tool bit. A 1/8" ball nose bit will be used to machine all of the parts.

The following is a flow diagram of the process the lessons will follow to create and machine the models.



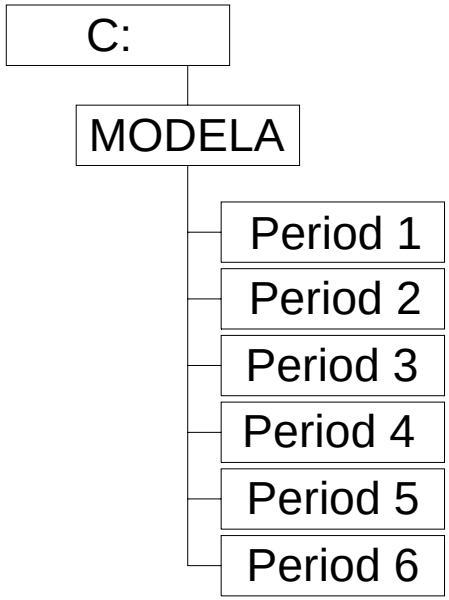
Instructor Responsibilities

To successfully use the Product Design & Manufacturing Series I in your classroom you will need to be responsible for the following areas.

1) Set up the directory system for the students to save their files.

In the Series the students are taught to save their files on the C: drive on the computer, in their period directory under a directory called MODELA.

You will need to set up a directory structure as diagrammed.

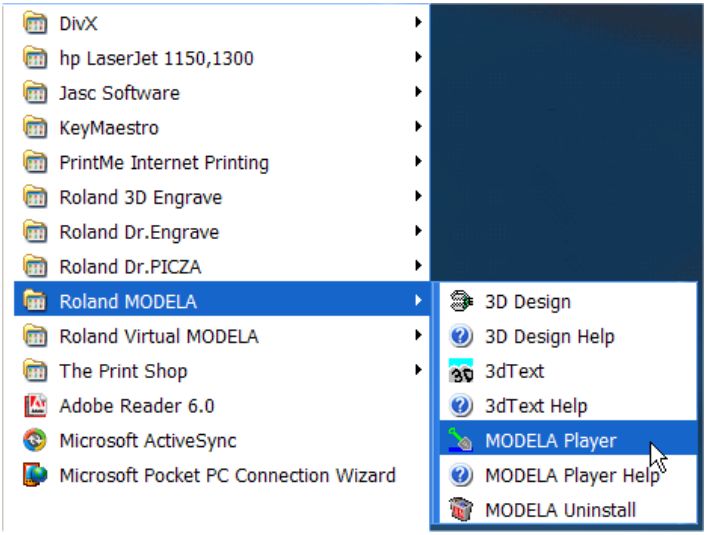


2) Install the Rhino and Roland Modela Player software on the desired computers.

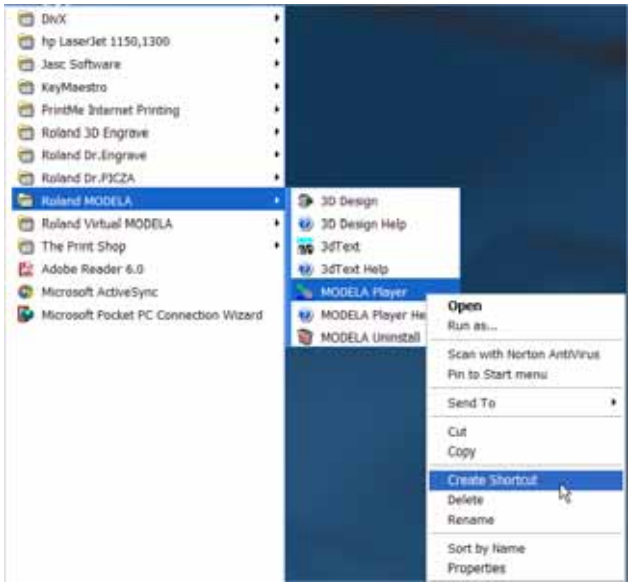
You will need to copy the Modela Player Icon on to the desktop.

The following explains the steps to copy the Modela icon to the desktop:

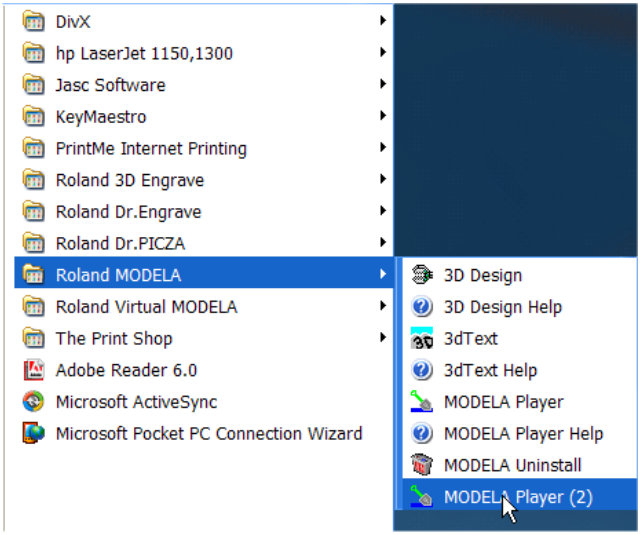
From the Start menu, **locate the Modela Player program**.



Right click on Modela Player and select Create Shortcut from the menu.

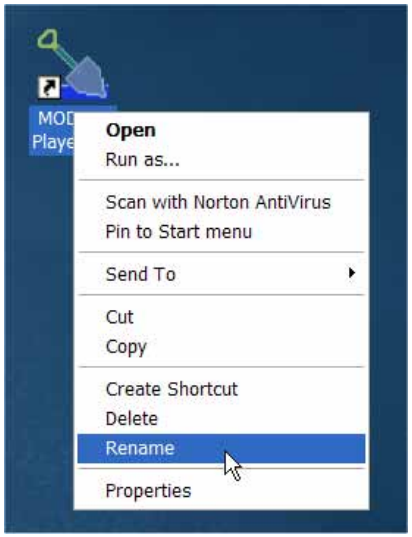


Left click and hold on the new shortcut and **drag it on to the desktop**. **Release** the mouse button.

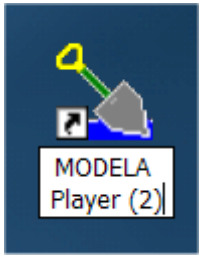


Result

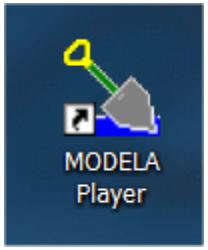
Right click on the desktop icon and **pick (left click) Rename**.



Move the cursor to the end of the (2) as displayed and **backspace the (2) away**. **Press Enter**.



Result



3) Read the MDX-15 manual and practice the setup so you can instruct your students.

4) Go over safety instructions with the students using the MDX-15.

5) Set up a schedule for use of the software and machines (see classroom suggestions).

Trouble Shooting

The following is a list of typical trouble shooting items if the students are unable to send toolpaths to the milling machine.

1) The milling machine is not connected to the computer and powered up.

2) The student has not followed the instructions accurately for the Modela Player chapters.

    a) The part is not sized properly.

    b) The cutting depth is not set properly.

    c) They have not set the machine type correctly.

3) The printer driver has not been set to the MDX-15. This can be done from the File menu in the Modela Player.

Classroom Organization Suggestions

The following are some suggestions to help in the flow of students through the Production Design & Manufacturing Series I.

1) Have students work in pairs.

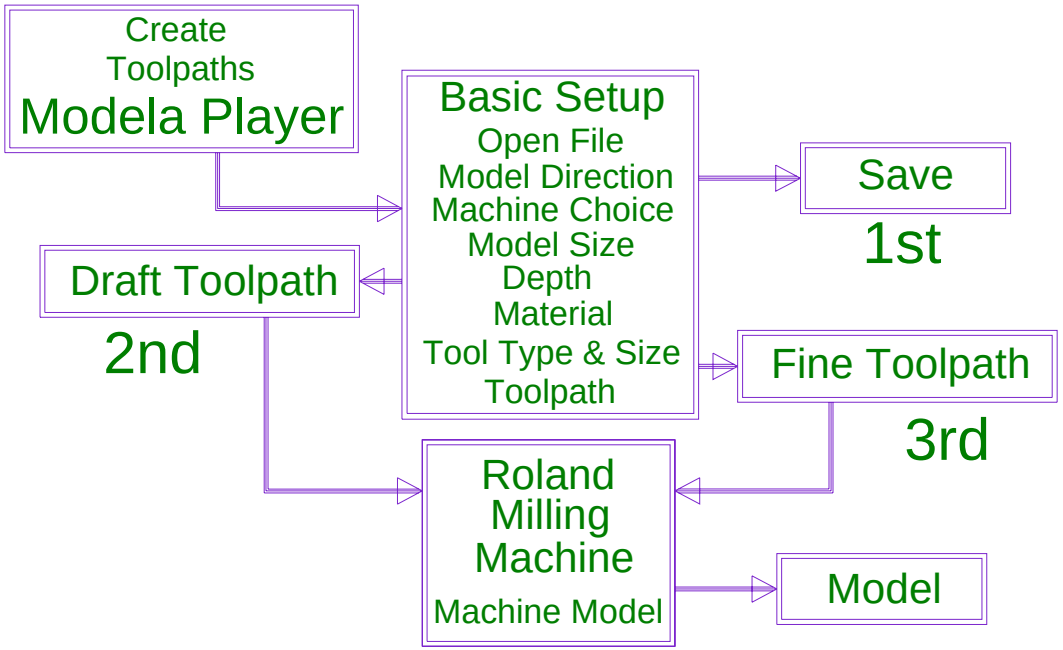
2) Create a master schedule for the semester, with 10 days for each pair of students. Two teams can be working on the Series at one time.

3) Stagger the start dates by one day for the two sets of teams so that the MDX-15 is assigned to each team on alternating days. This will maximize MDX-15 usage.

4) Go over the MDX-15 usage with each set of teams on their third scheduled day if they are covering Lesson 1 on Rhino Basics.

Modela Player Information

The Modela Player is a simple fast and easy way to create toolpaths. Displayed is the process for creating the toolpaths and sending them to the mill.



After one becomes proficient with the software, it should take no more than five minutes to open a file and create the Draft toolpaths for the model. However, every item in the basic setup needs to be considered, set or checked.

It is important to note that after everything has been completed to the point of creating the Draft toolpaths, the file can be saved and it will retain all of the settings. The file can be opened; the Toolpath icon selected to create the toolpaths and sent to the machine. If a team finishes the Rhino lesson before the end of a class period they can set up the model in Modela to the point of creating the Draft toolpaths.

Machining Times

The following is a list of approximate machining times for each of the four parts in the series using a Roland MDX-15 milling machine.

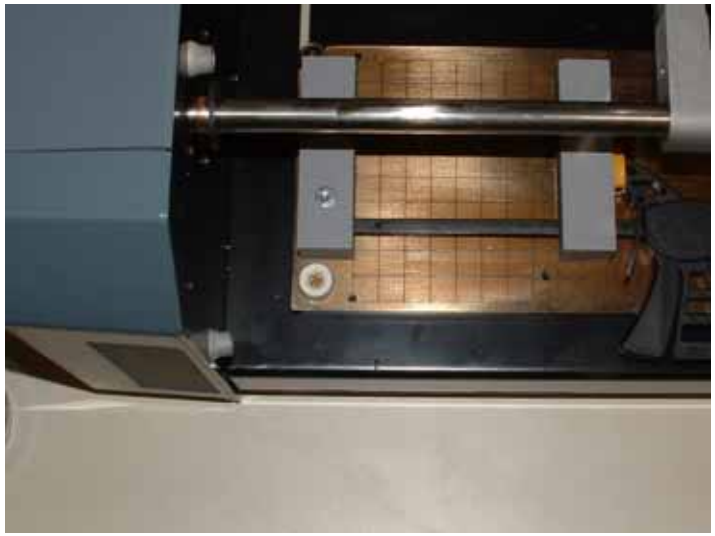
Bottle	Draft - 17 minutes Fine - 12 minutes	
		Total – 29 minutes
Desk Weight	Draft - 22 minutes Fine - 9 minutes	
		Total – 31 minutes
Hair Dryer	Draft - 25 minutes Fine - 12 minutes	
		Total – 37 minutes
Boat	Draft - 28 minutes Fine - 11 minutes	
		Total – 39 minutes

Material Orientation (MDX -15)

The following is a series of pictures displaying the orientation of the machining wax in the MDX-15.

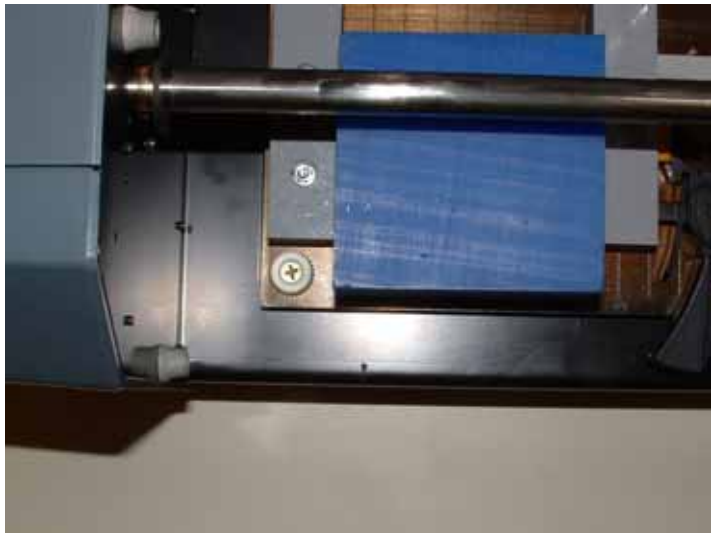
Using the supplied vise:

Pictured is the vise plate installed in the MDX-15.



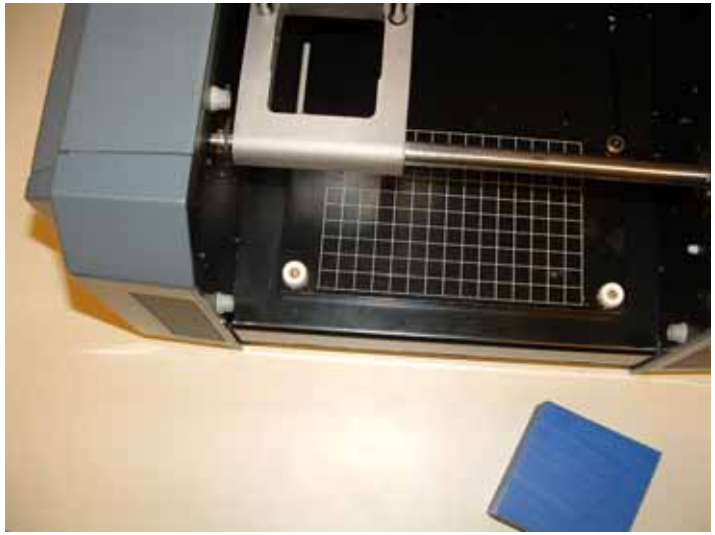
The wax is to be lined up with the lower left corner of the grid.

Pictured is the wax clamped in the vise.



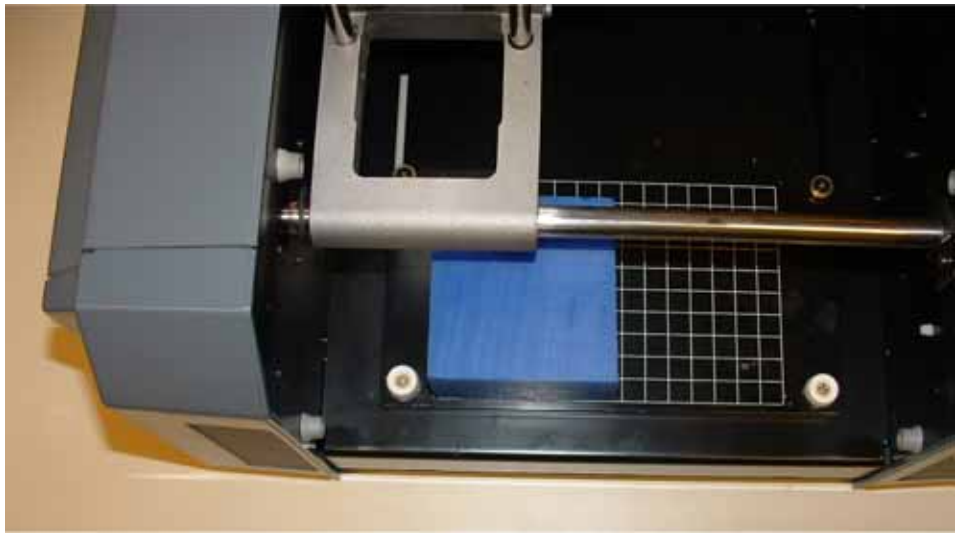
Using the standard grid plate:

Pictured is the standard plate installed in the MDX-15.



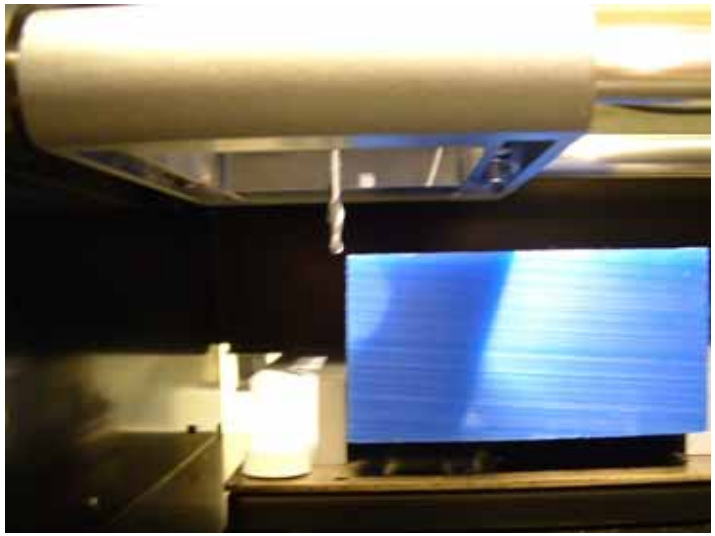
The wax is to be attached with two sided carpet tape and lined up with the lower left corner of the grid.

Pictured is the wax taped to the standard plate.



Pictured is the bit positioned properly to begin machining. It is located, even with the top of the wax and at the lower left corner.

This is done by taking the MDX-15 off of View and using the Down button on the Front control panel to lower the bit.



Academic Competencies

The following are a list of the academic competencies that can be taught, reinforced and enriched by the Product Design & Manufacturing I Series. They are based on the Washington State Essential Academic Learning Requirements.

Reading

Use word and recognition skills and strategies to read and comprehend text.

Demonstrate evidence of reading comprehension.

Expand comprehension by analyzing, interpreting, and synthesizing information and ideas in informational text.

Read to learn new information.

Read to perform a task.

Read for career application.

Mathematics

Understand and apply concepts and procedures from measurement --- attributes and dimensions , approximation and precision and systems and tools.

Understand and apply concepts and procedures from geometric sense --- properties and relationships and locations and transformations.

Analyze information from a variety of sources; use models, known facts, patterns and relationships to validate thinking.

Predict results and make conjectures based on problem analysis situations.

Draw conclusions and verify results --- support mathematical arguments, justify results, and check for reasonableness of solutions.

Science

Apply science knowledge and skills to solve or meet challenges.

Know that science and technology are human endeavors, interrelated to each other, to society, and to the workplace.

1 Rhinoceros Basics

The Rhinoceros Basics lesson is designed to give you the skills and knowledge to create models that can be machined with Roland software and milling machines. It is very important that you master this introduction. This is the key to your success in completing the following lessons.

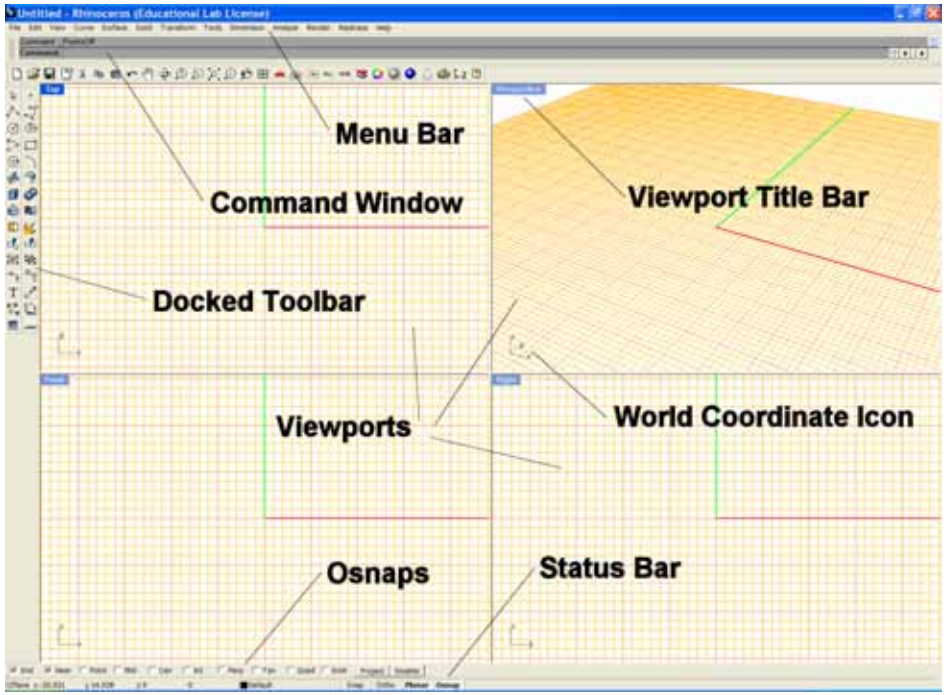
Starting Rhino

To start Rhino, boot up the computer.

Double click on the Rhinoceros icon on the Windows desktop.

The Screen Area

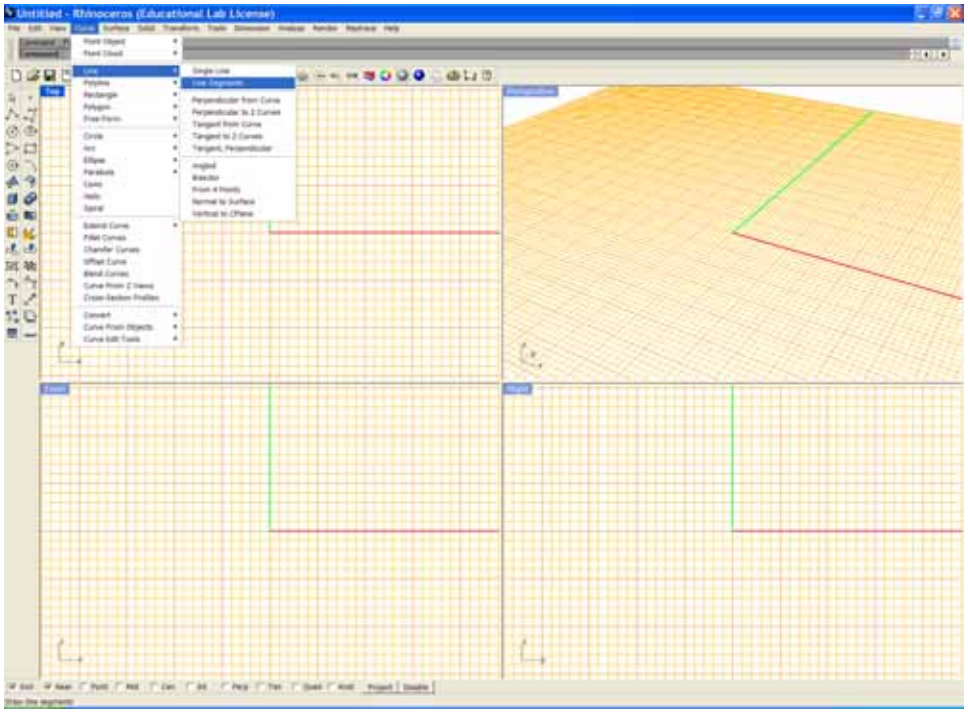
Displayed is the Rhino screen area. You should familiarize yourself with the different areas of the screen.

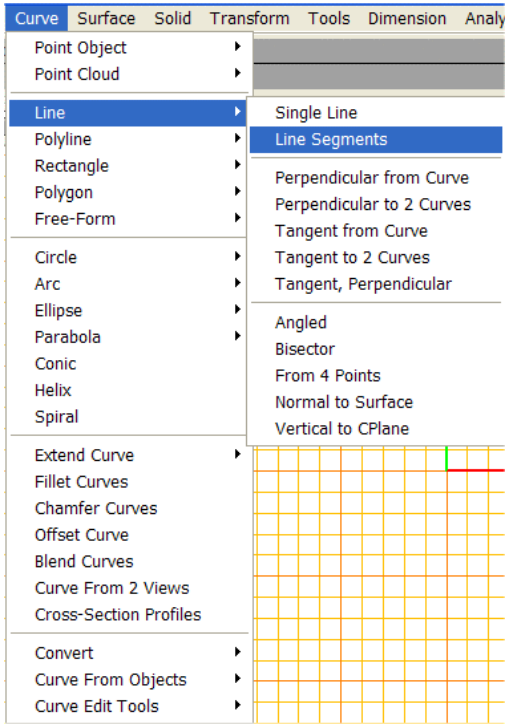


Accessing Commands

Pull Down Menus

Most of the Rhino commands can be run from pull down menus. The pull down menus are listed across the top of the graphics screen (menu bar).

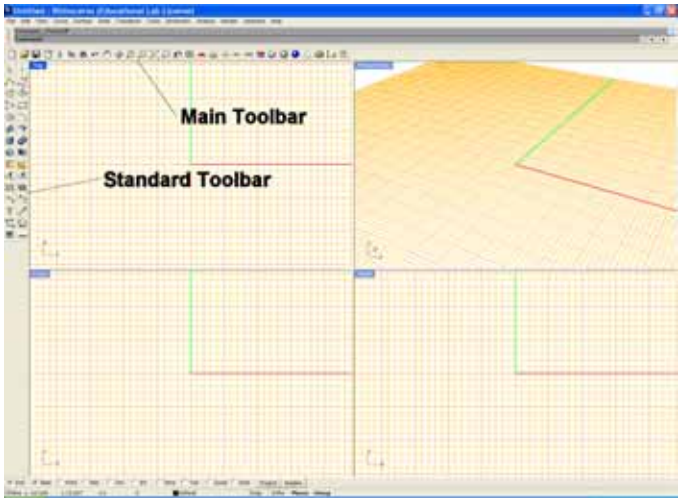




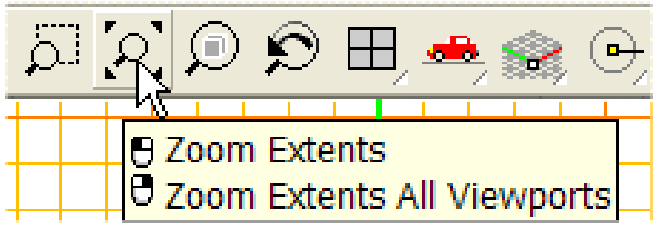
Toolbars

Rhino has a series of toolbars that can be used to access commands. When Rhino is loaded the Main toolbar sits on top of the graphics screen below the command window. The Standard toolbar is on the left of the graphics screen.

These toolbars can be moved and others added. It is STRONGLY suggested that the toolbar layout not be modified for these lessons.



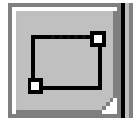
Many of the toolbar icons have two commands assigned to them. One is started with a left mouse button, the other with a right mouse button.



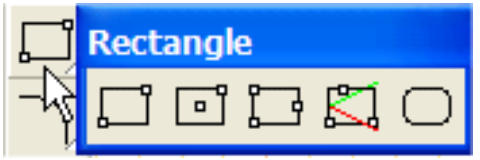
In this example, a left mouse pick will run a Zoom Extents command and a right pick will run Zoom Extents All. Zooms will be explained later in the introduction. The text box that appears when you hold the mouse pointer over a command icon is called a **tooltip**.

Flyouts

On many of the toolbar icons there is a small triangle in the bottom right corner of the icon.



By picking and holding with the left mouse button or right clicking a flyout menu will appear. This menu has other commands associated with the icon you selected. To use one of the flyout icons move your mouse to it and right click.



Controlling Commands

Canceling a Command

To cancel a Rhino command, **press the Esc key** on the keyboard.

Repeating a Command

There are various ways to repeat the last command.

1) Press the right mouse button while over a viewport.

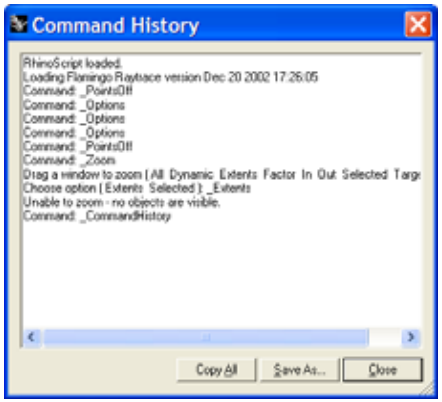
2) Press the space bar on the keyboard.

3) Press the Enter key on the keyboard.

Note: These actions will also end a command in most cases.

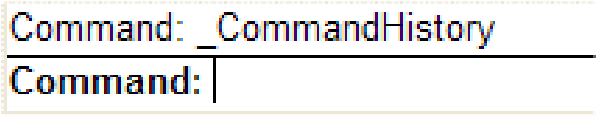
Command History

To view the previous commands that you have run, press the F2 key on the keyboard. The scroll bar can be used to view commands that are off the screen.



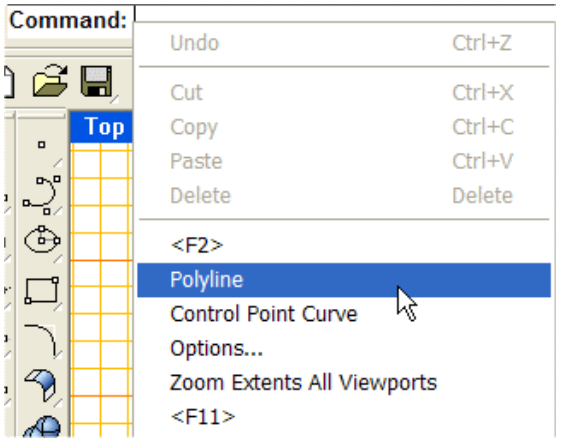
Pick the X in the upper right of the box to close it.

The command history can also be used to start commands that you already have used.



Place your pointer in the command line without a command running.

Right click and the last nine commands will appear. **Left click** on the one that you want to run.



Viewports and Work Planes

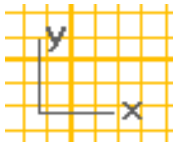
Rhino works in 3D space. For the purposes of this introduction, space can be looked at as a box. The size of the box can be any size needed to create the desired model.

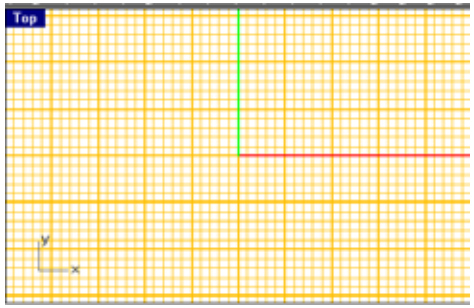
Rhino, when started comes up with four viewports, Top, Front, Right Side and Perspective.

The Top viewport is looking down on the box that represents space. The Front viewport is looking at the front of the box and so forth.

X, Y and Z-axes are used to draw in each viewport.

In the Top viewport the X-axis runs left and right and the Y axis, up and down. This is called the world coordinates. Where the two thick lines meet is X, Y zero.





The Perspective viewport is the same as the top viewport but looking at it from a different point of view.

The two other viewports also have icon displaying the X, Y, and Z-axes but in relation to the world coordinates. However, when working in the Front or Right viewports the X-axis runs left and right and the Y-axis, up and down. How to work in the viewports will be discussed later.

You can create more viewports and resize them as desired but for these lessons that will not be necessary.

NOTE: It is very important to look at all viewports while drawing to make sure that your geometry is correct. Often, the drawing looks correct in one viewport but you can see the error in another.

Each of the standard viewports has a work plane assigned to it. For our purposes, these are the sides of the box. 2D geometry is created on the work plane.

In Rhino, as you change viewports from Top, Front and Side you automatically change work planes. The Top and Perspective viewports have the same work plane.

Rhino also allows you to create work planes as needed. These can be looked at as a flat piece of paper oriented in space wherever desired.

This exercise will help you visualize these concepts.

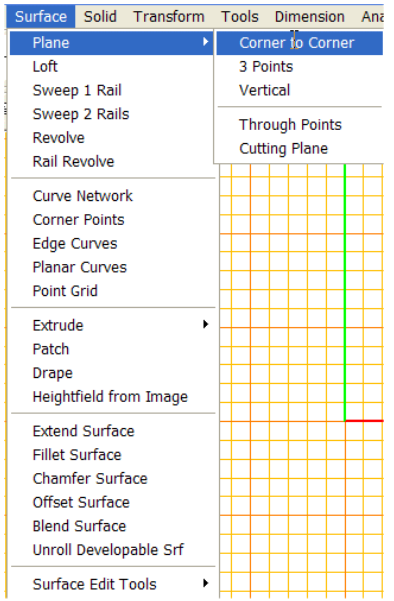
1) Start up Rhino:

To start Rhino, boot up the computer.

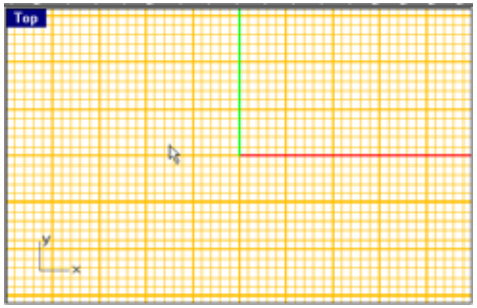
Double click on the Rhinoceros icon on the Windows desktop.

Use the Surface pull down menu as displayed and pick the following:

Surface - Plane – Corner to Corner



3) **Put your pointer in the Top viewport**.

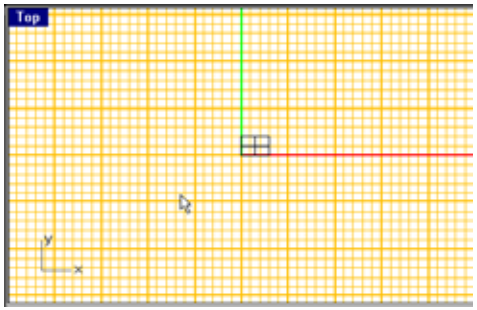


4) Input the following (bold items) to the command prompts in the command line.

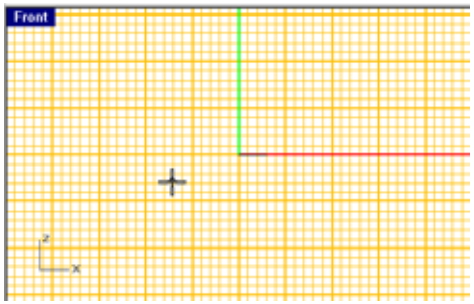
Note: Enter means to press the Enter Key on the keyboard.

First corner of plane (3Point Vertical Center Deformable): **0 Enter**

Other corner or length: **3,2 Enter**



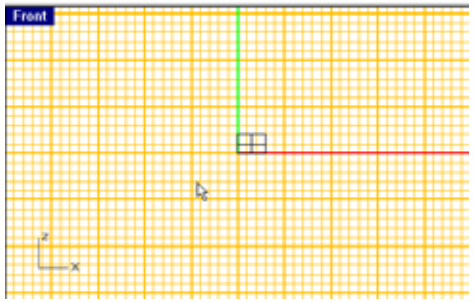
5) **Restart the Surface – Plane command** (press the space bar) **and put the pointer in the Front viewport.**



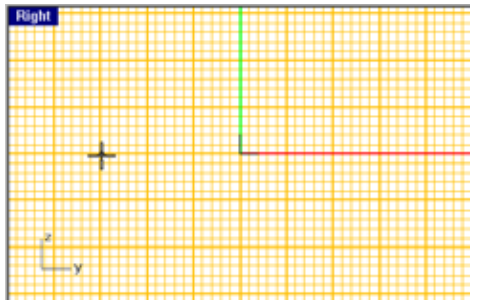
6) Input the following (bold items) to the command prompts in the command line.

First corner of plane (3Point Vertical Center Deformable): **0 Enter**

Other corner or length: **3,2 Enter**



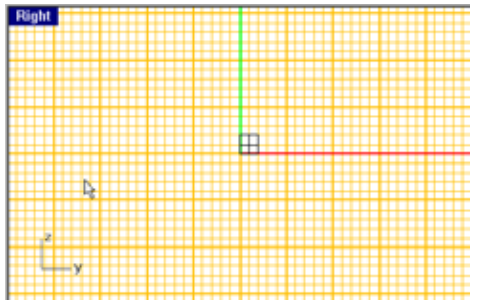
7) **Restart the Plane command** (press the space bar) **and put the pointer in the Side viewport.**



8) Input the following (bold items) to the command prompts in the command line.

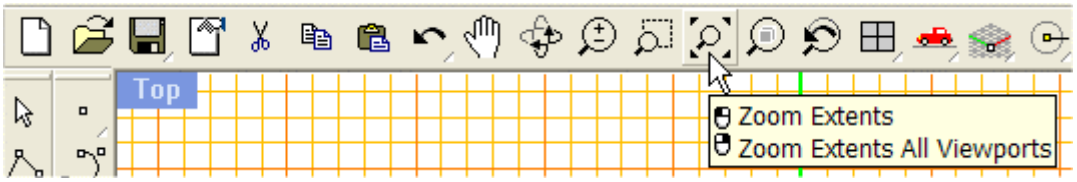
First corner of plane (3Point Vertical Center Deformable): **0 Enter**

Other corner or length: **2,2 Enter**

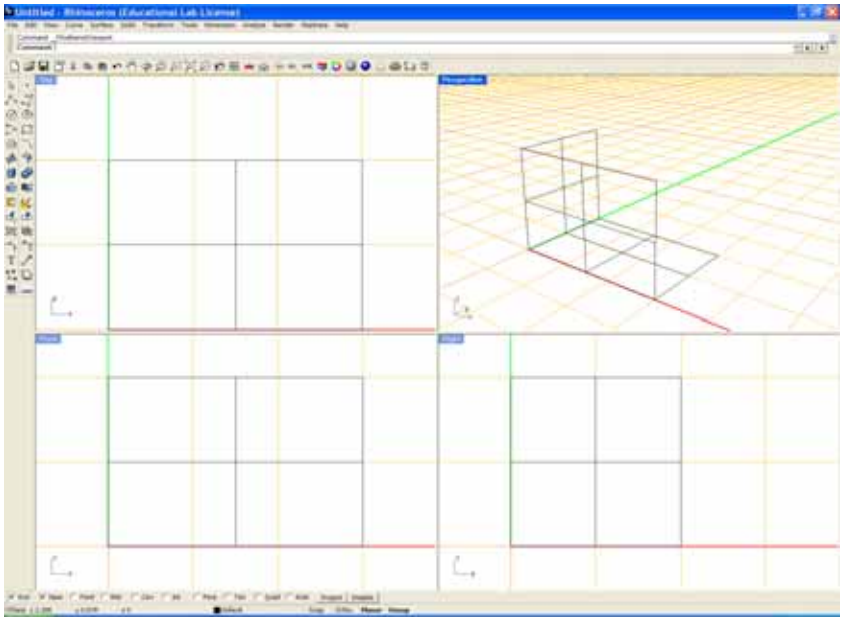


9) To see the results more clearly, **right click on the Zoom Extents ALL** icon as displayed.

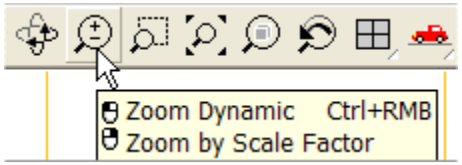
This command zooms the view to the overall size of the model.



Result

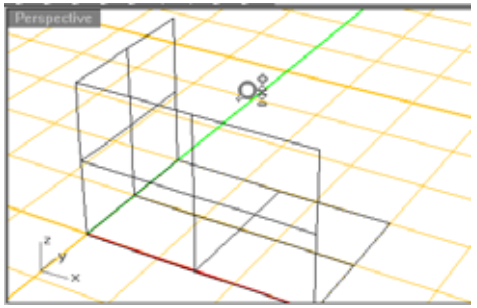


10) Because it uses the extents of the Top viewport the model is not completely displayed in the Perspective viewport. To correct this:

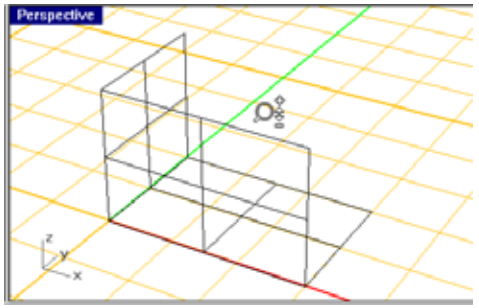


Pick the Zoom Dynamic command from the top toolbar.

Move the cursor to the Perspective viewport as displayed.

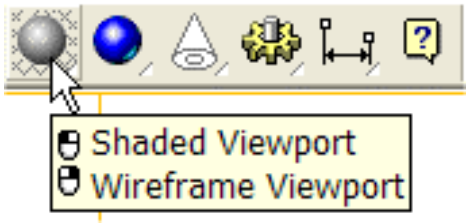


Hold the left mouse button down and move the mouse downward until the model fits into the viewport.



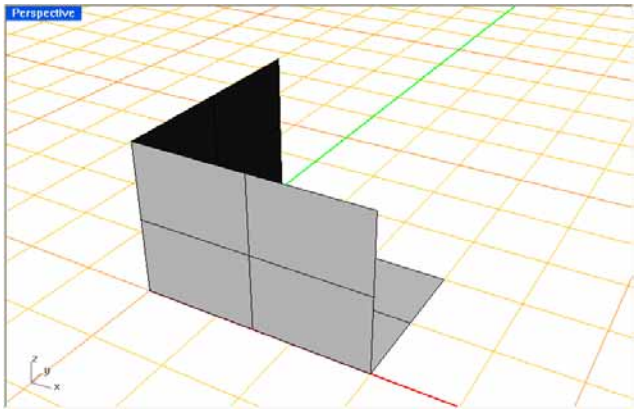
Release the mouse button.

11) To more easily view the planes you have created, they can be shaded.

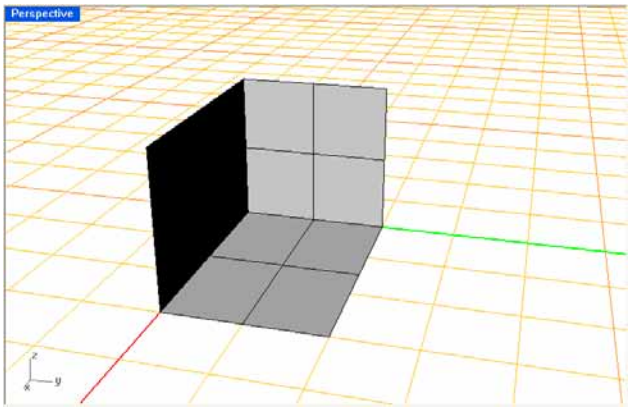


Left click on the Shade icon on the top toolbar.

Result



12) To look at the model from another direction, it can be rotated.



Right click and hold the Perspective viewport.

Move the mouse to rotate the view.

Release the mouse button when you are finished.

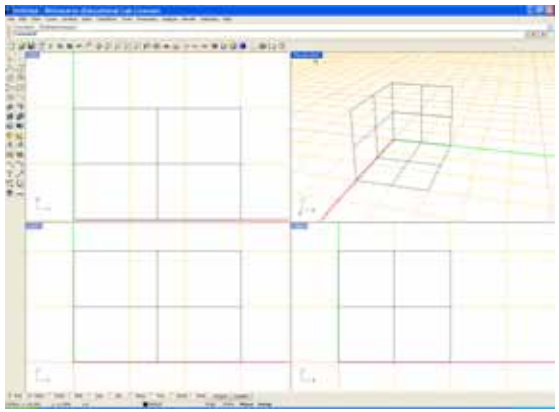
Right click on the Shade icon to remove the shading.

As you can see each of the rectangles were created on the work plane in each viewport. Most CAD modeling programs only have one work plane available at a time. This is a great strength of Rhino but it does require the user to keep track of the viewport they are working in.

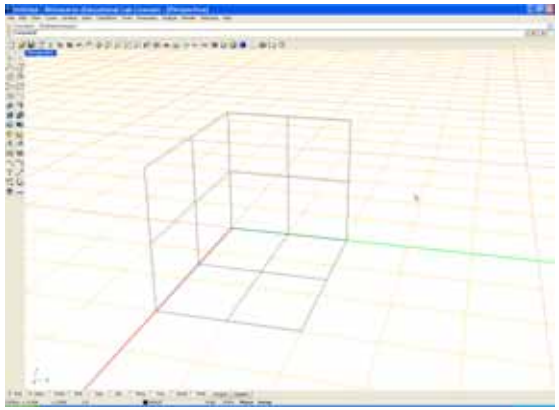
Single Viewport

To fill the graphics screen with a single viewport, **double click on the viewport title**.

In this case, Perspective.

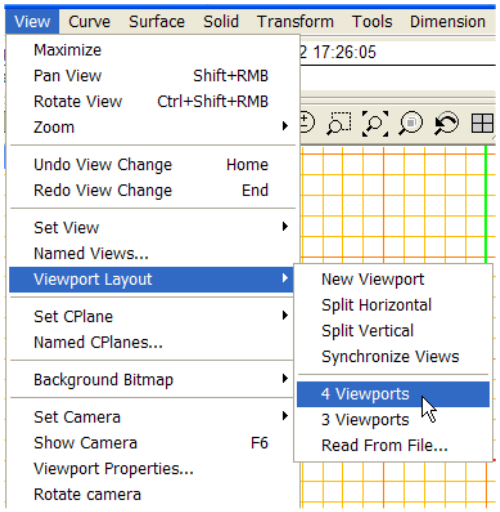


Result



To return to multiple viewports, **double click on the viewport title**.

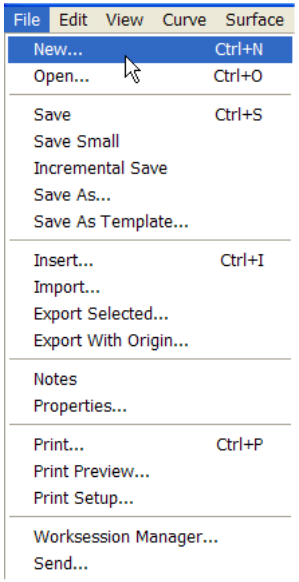
Resetting the Viewports



If your viewport settings get completely out of alignment do the following from the pull down menu.

View – Viewport Layout – 4 Viewports

Starting a new model while in Rhino

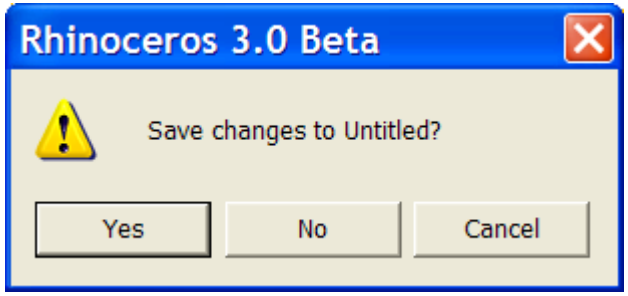


To start a new drawing session while in Rhino:

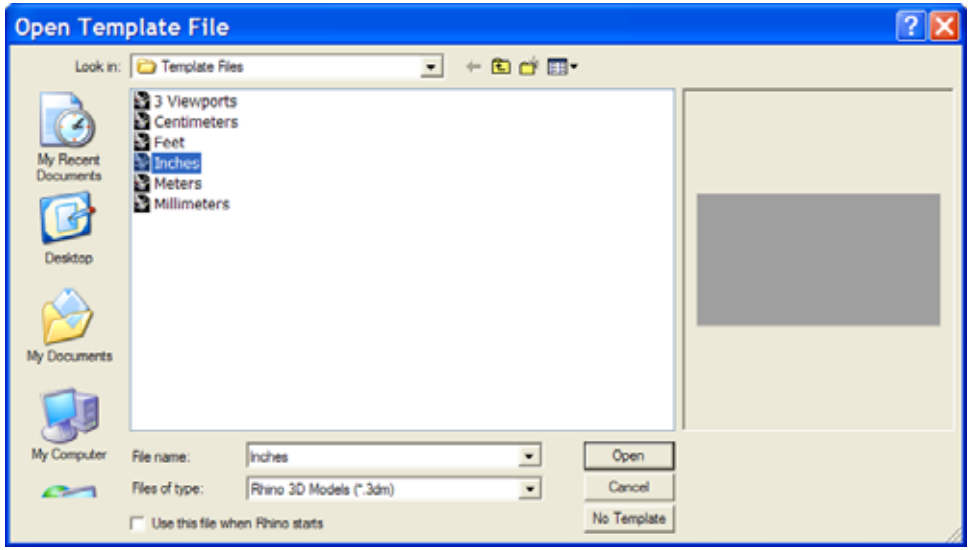
From the pull down menu:

File – New…

Select No, to not save the model.



Select Inches and Open to complete the process.



Modeling Accurately

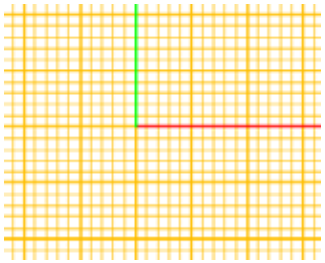
Being able to draw and model accruately is a very important skill in any CAD or 3D modeling program. There are various ways to do this. Some you will use more than others. However it is very important that you master all of these skills.

Grid and Snap

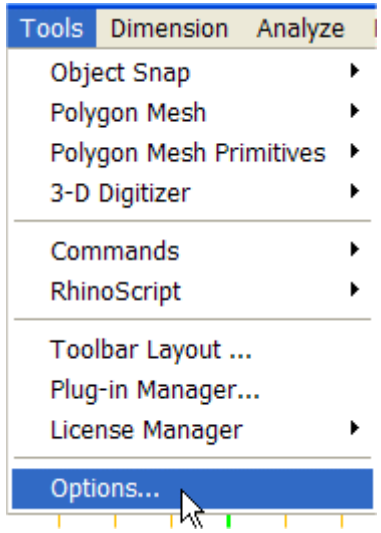
The simplest method of drawing accurately is using Grid and Snap. This method only works if what you are modeling has dimensioning that is evenly divisable, such as ¼ ., ½ and 1. Otherwise using Grid and Snap will not be effective.

In Rhino, Grid lines are on the screen in a vertical and horizontal pattern. By default, every fifth one is thicker. These are reference lines that allow the user to track (count) distances.

Snap causes the pointer to jump at a preset increment, for example, 1 unit usually based from the 0,0 point of the X, Y-axes.



1) **Start** a new Rhino drawing.

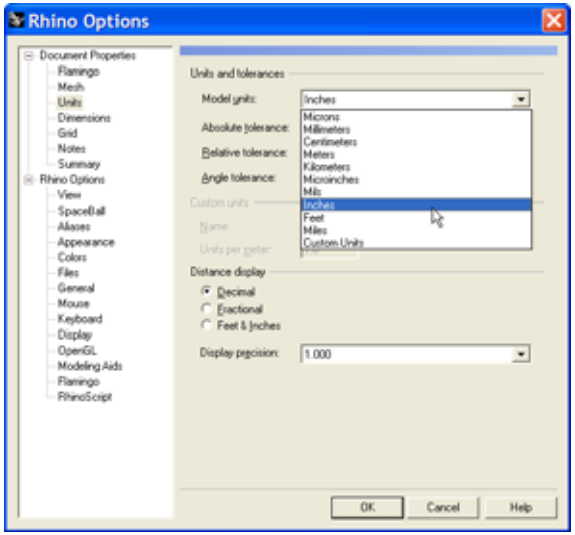


2) To check and/or setup Grid and Snap:

From the pull down menus:

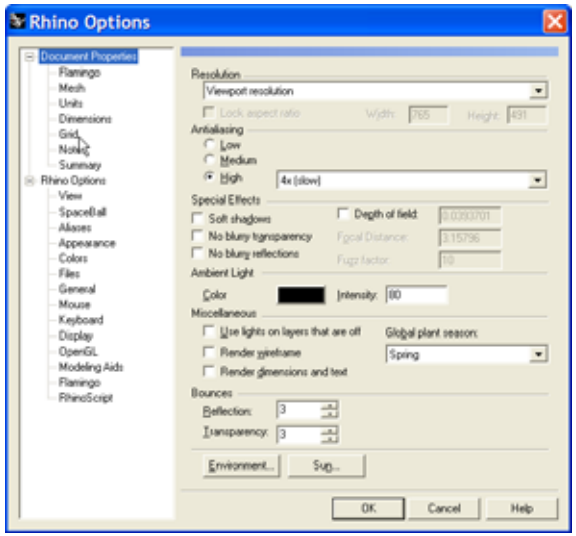
Tools – Options…

3) From the Options dialogue box:



On the left side of the dialogue box, **select Units** and **Inches**, if they are not already selected.

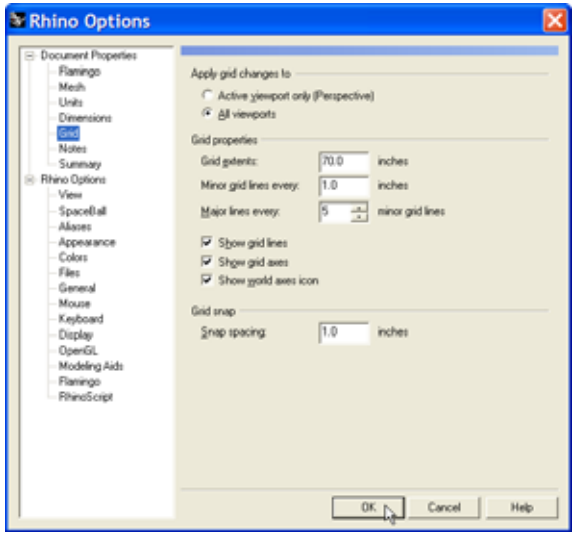
Pick the Grid tab from the left side of the dialogue box.



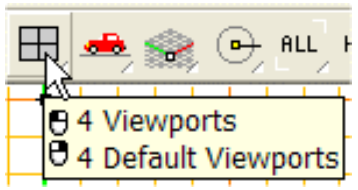
The Grid tab values should be set as shown.

If it is not, set to these values.

Pick OK.



To zoom into the complete viewport area, **pick the 4 Viewports icon** from the Main toolbar.



There are two ways to turn on Snap.

One is to press the **F9 key on the keyboard**.

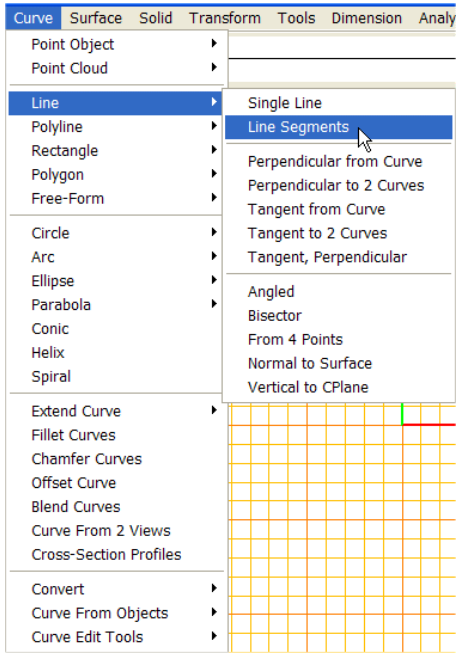
or

Pick Snap at the bottom of the graphics screen so that it is bold (ON) and not grayed out (OFF).

4) To see how grid and snap work:

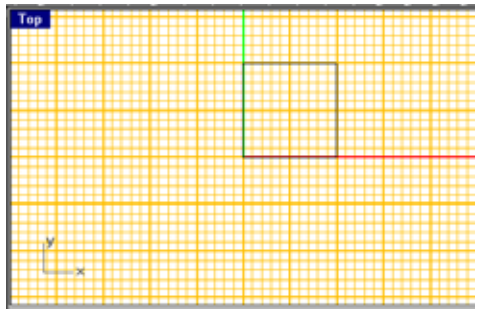
From the pull down menu:

Curve – Line – Line Segments



With the pointer in the Top viewport:

Pick four points as displayed, creating a 10 by 10 unit box.



5) You my have found it hard to keep the lines on the axis as they would grab the snap point to the side of where you wanted to pick. To solve this, you can turn on Ortho.

Ortho causes the lines to only extend either straight up/down or across.

This is a handy tool even when using commands other than snap.

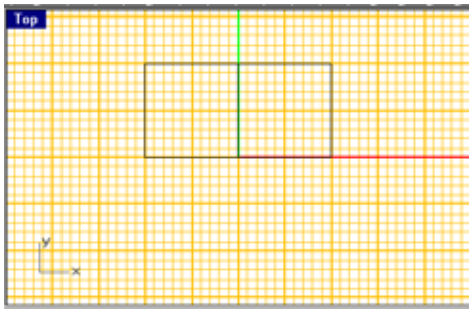
There are two ways to turn on Ortho.

One is to press the **F8 key on the keyboard**.

or

Pick Ortho at the bottom of the graphics screen so that it is bold (ON) and not grayed out (OFF).

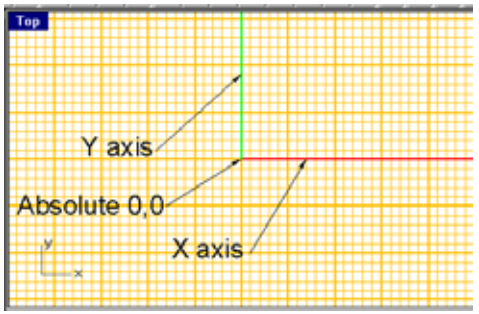
6) **Restart** the line segments command and **draw** a box to the left of the one you previously created.



Coordinate Systems

Coordinate systems are the main tools used to create accurate models. Rhino supports three types: Absolute Cartesian; Relative Cartesian and Polar.

Absolute Cartesian Coordinates

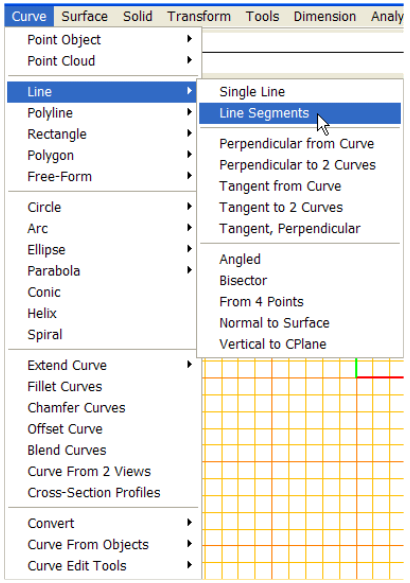


Absolute Cartesian coordinates are based on X, Y, and Z-axes. And at this point we will ignore the Z component.

When inputting these coordinates you type:

X-value, Y-value, Z –value i.e 3,2

The Z value is optional.



1) **Start** a new Rhino drawing with the **Millimeters template**.

2) Start the line segments command.

From the pull down menu:

Curve – Line – Line Segments

With the pointer in the Top viewport:

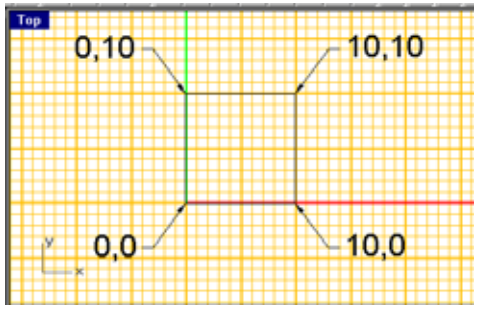
Start of first line: **0 Enter**

End of line (Undo): **10,0 Enter**

End of line. Press Enter when done (Undo): **10,10 Enter**

End of line. Press Enter when done (Undo): **0, 10 Enter**

End of line. Press Enter when done (Undo): **C Enter**



The “C” in the 5th entry stands for “close” and closes the box.

Relative Cartesian Coordinates

Relative Cartesian coordinates are based on X, Y, and Z-axes relative to the last point you have drawn.

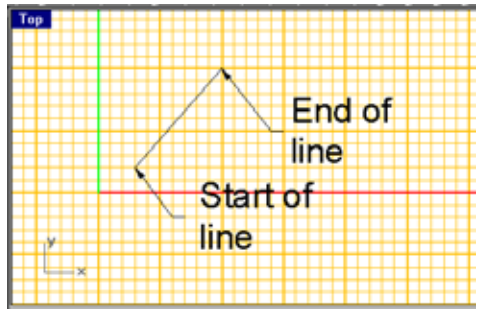
When inputting these coordinates you type:

R X-value, Y-value, Z –value i.e R3, 2 The R can be in either upper or lower case.

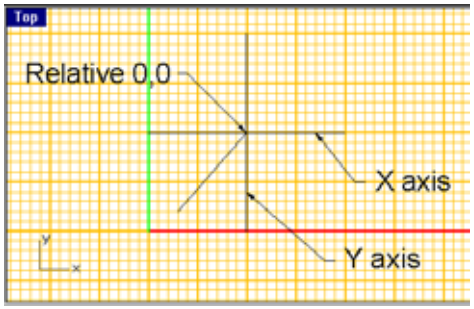
The Z-value is optional.

Let’s take a look about how Relative Cartesian coordinates work.

A line has been drawn.



The end of the line becomes the Relative 0,0 point.



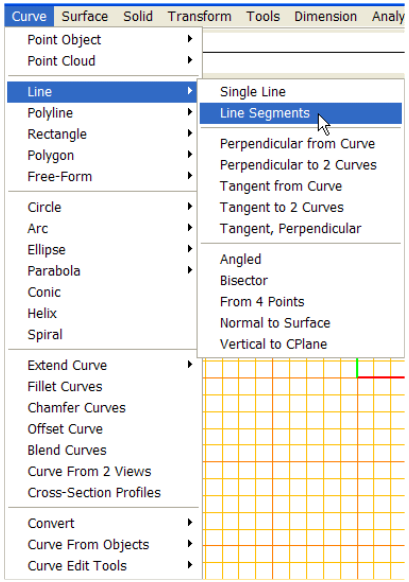
This is much easier than always calculating the position from absolute 0,0.

1) Start a new Rhino drawing

2) Start the line segments command.

From the pull down menu:

Curve – Line – Line Segments



With the pointer in the Top viewport:

Start of first line: **3,3 Enter** (this is an absolute entry)

End of line (Undo): **R10,0 Enter**

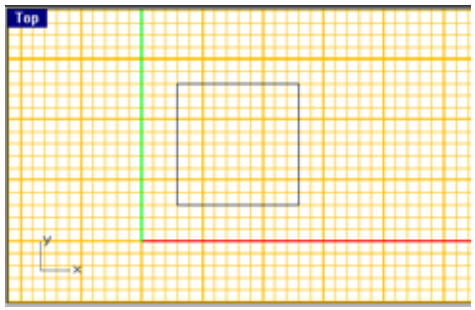
End of line. Press Enter when done (Undo): **R0,10 Enter**

(notice the –10 in the next input)

End of line. Press Enter when done (Undo): **R-10, 0 Enter**

End of line. Press Enter when done (Undo): **C Enter**

End of line. Press Enter when done (Undo): **Enter**



Polar Coordinates

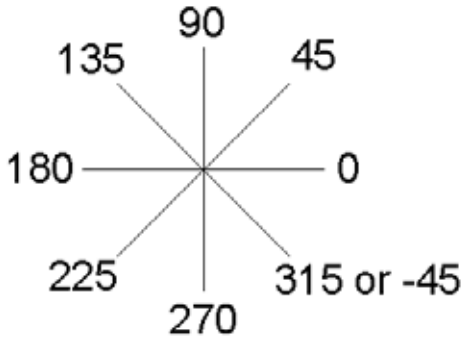
Polar coordinates work with angles and distances and are relative to the last point entered as with Relative Cartesian coordinates.

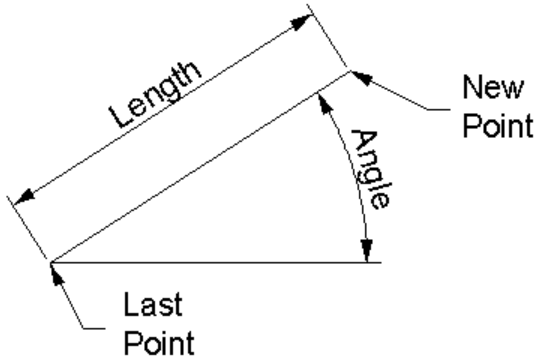
The angles work as displayed. Any increment in-between is valid.

The distance is the length of line, etc. that you are drawing.

When inputting these coordinates you type:

R distance < angle



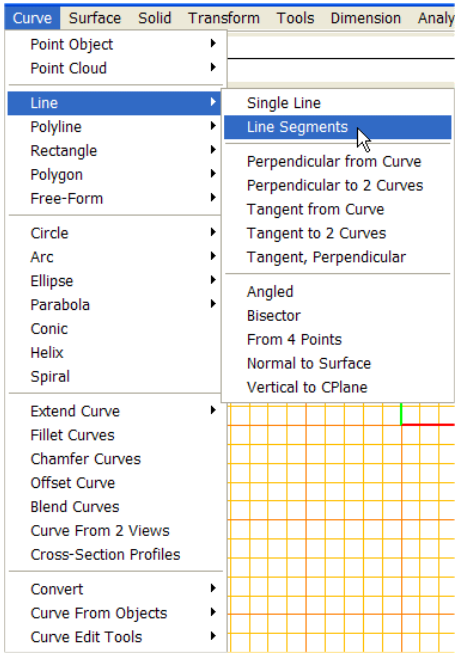


1) **Start** a new Rhino drawing with the **Millimeters template**.

2) Start the line segments command.

From the pull down menu:

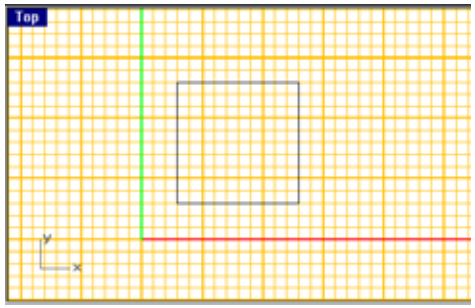
Curve – Line – Line Segments



With the pointer in the Top viewport:

Start of first line: **3,3 Enter** (this is an absolute entry)

End of line (Undo): **R10<0 Enter**



End of line. Press Enter when done (Undo): **R10<90 Enter**

End of line. Press Enter when done (Undo): **R10<180 Enter**

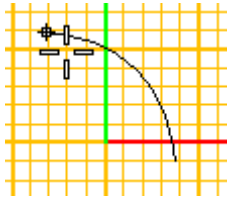
End of line. Press Enter when done (Undo): **C Enter**

Object Snaps

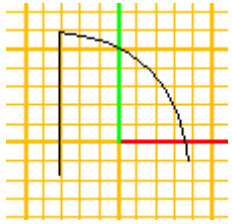
Object snaps are command options that allow the user to attach an accuracy of 14 decimal places to different parts of existing geometry.

For example, if you want to draw a line from the end of an arc, you would use an endpoint Osnap at the start of the line command to exactly select the end of the arc:

Start of the line using an endpoint Osnap.

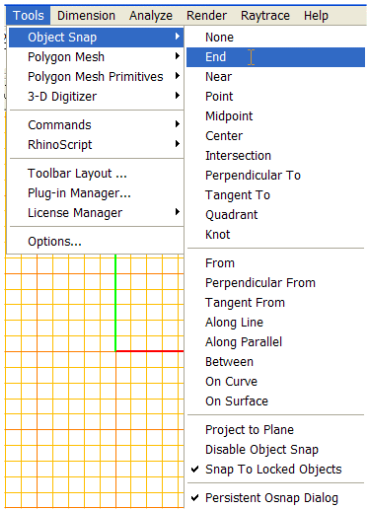


Result



Osnaps can be accessed by:

Tools – Object Snap – pick the desired Osnap



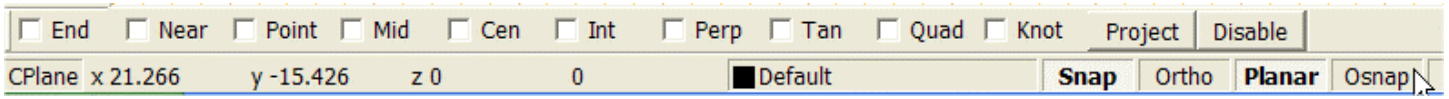
Selecting an Osnap this way turns it on only for one pick.

Note: There are Osnaps listed here that are not accessible from the toolbar discussed below.

or

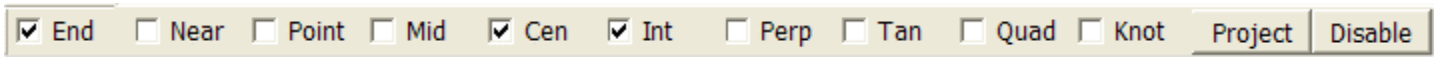
Use the Osnap toolbar at the bottom of the graphics screen.

If it is not displayed select the Osnap button at the bottom of the screen and it will appear.



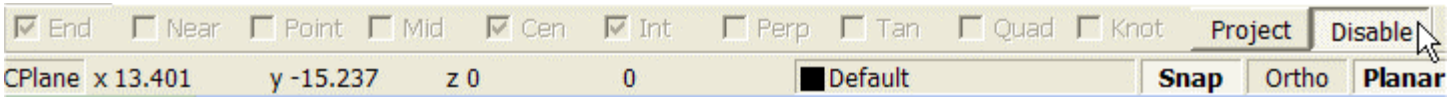
Select any of the Osnaps that you want to use.

These are running Osnaps. They are on for every pick.



To turn these running Osnaps off, **pick Disable**.

To turn them back on, **pick Disable again**.

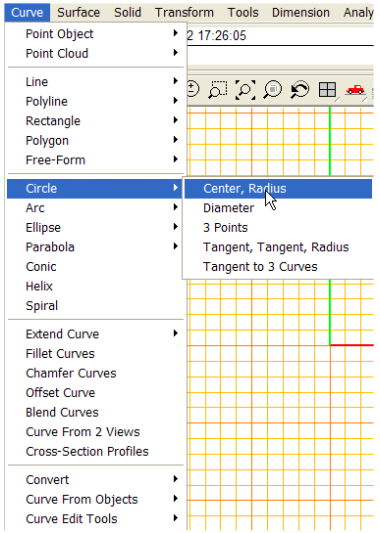


1) **Start** a new drawing with the **Millimeters template**.

2) Start the Circle command

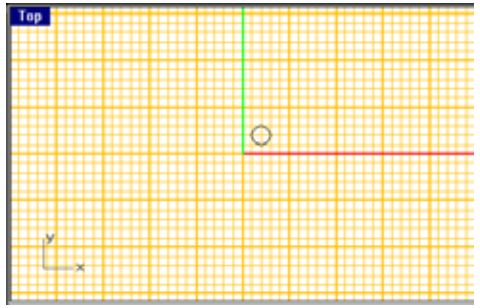
With the pointer in the Top Viewport:

Curve – Circle – Center, Radius



Center of circle (Deformable Vertical Diameter 3Point Tangent AroundCurve): **2,2 Enter**

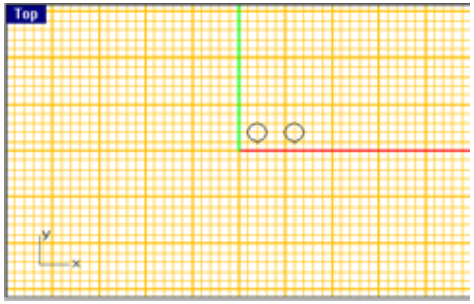
Radius <1> (Diameter): **1 Enter**



Press Enter to re-start the command.

Center of circle (Deformable Vertical Diameter 3Point Tangent AroundCurve): **6,2 Enter**

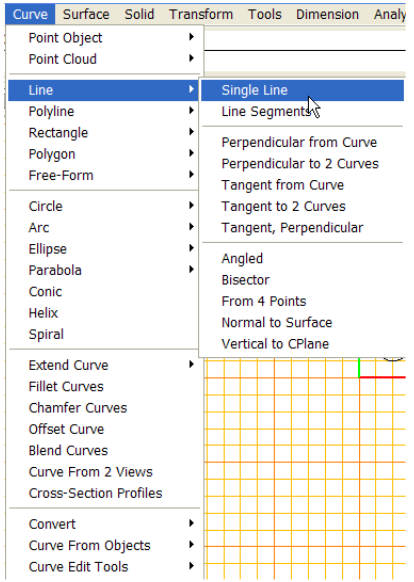
Radius <1> (Diameter): **1 Enter**



3) Start the Single Line command.

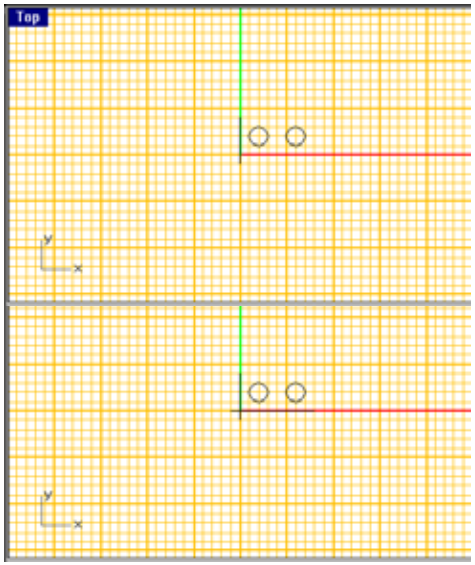
With the pointer in the Top Viewport:

Curve – Line – Single Line



Start of first line (Normal Angle Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **0,-1 Enter**

End of line (Undo): **0,4 Enter**



Press Enter to re-start the command.

Start of first line (Normal Angle Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides):: **-1,0 Enter**

End of line (Undo): **8,0 Enter**

Press Enter to re-start the command.

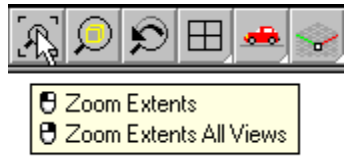
Start of first line (Normal Angle Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **4,0 Enter**

End of line (Undo): **R0,4 Enter**

4) Zoom up on the drawing.

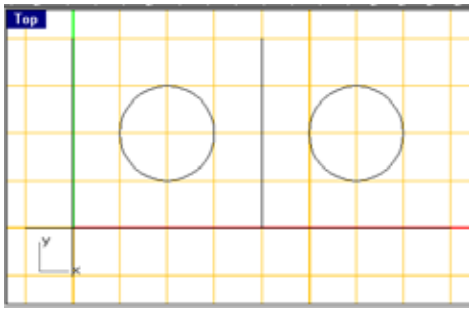
View – Zoom – Extents All

Or

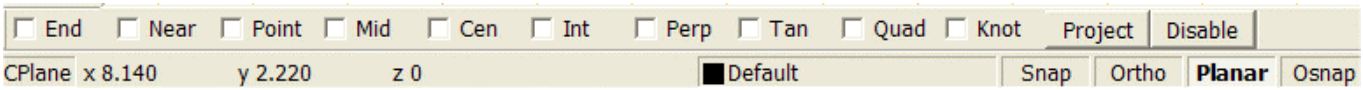


Right click on the Zoom Extents All icon from the top tool bar.

Result



5) Have the Osnap toolbar displayed with Snap and Ortho off.

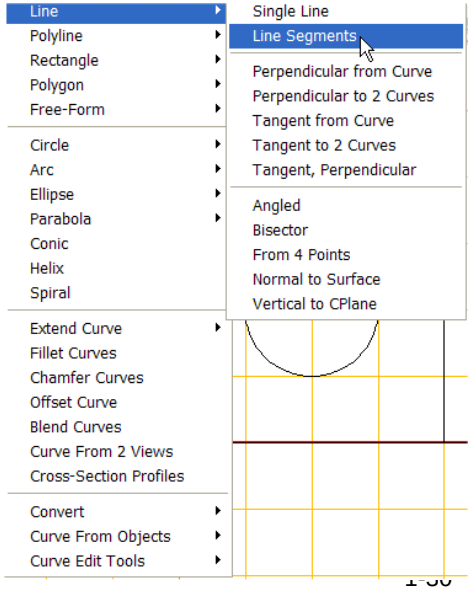


The following drawing is an example of using Osnaps.

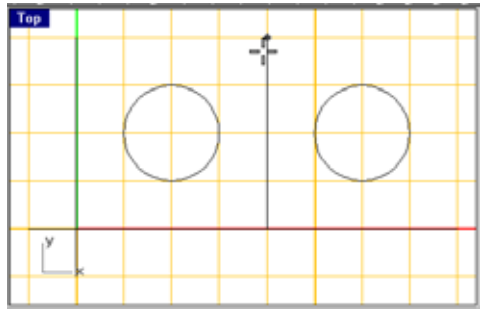
NOTE: At each step check the needed Osnap in the toolbox and un-check any others.

Start the Line Segments, Line command.

Curve - Line - Line Segments

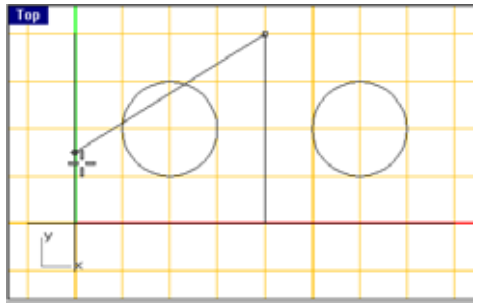


All of the following picks are in the Top Viewport



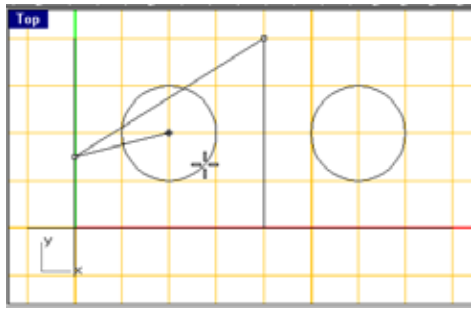
Start of first line: **Pick as displayed with End Osnap**

(as you move close to the endpoint the pointer will snap to the end of the line)



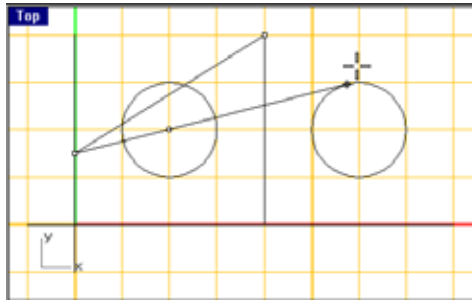
End of line (Undo): **Pick as displayed with Mid Osnap**

(Midpoint)



End of line. Press Enter when done (Undo): **Pick as displayed with Cen Osnap**

(Center)

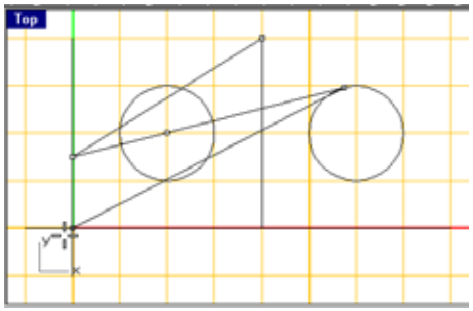


End of line. Press Enter when done (Close Undo): **Pick as displayed with Tan Osnap**

(Tangent)

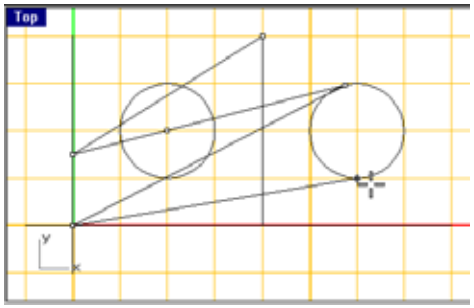
End of line. Press Enter when done (Close Undo): **Pick as displayed with Int Osnap**

(Intersection)



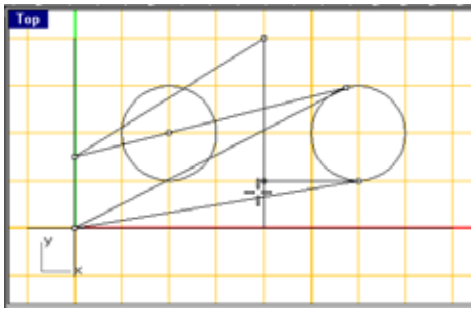
End of line. Press Enter when done (Close Undo): **Pick as displayed with Quad Osnap**

(Quadrant - there are four quadrant points on a circle, every 90 degrees)

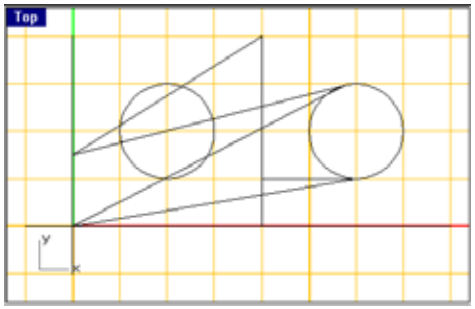


End of line. Press Enter when done (Close Undo): **Pick as displayed with Perp Osnap**

(Perpendicular)



End of line. Press Enter when done (Close Undo): **Enter** – to end the command



Saving and Opening a Drawing/Model

Save a drawing

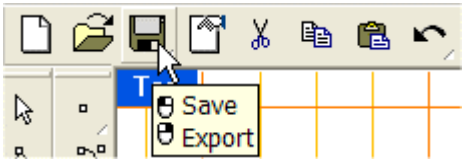
We will save your Osnap drawing

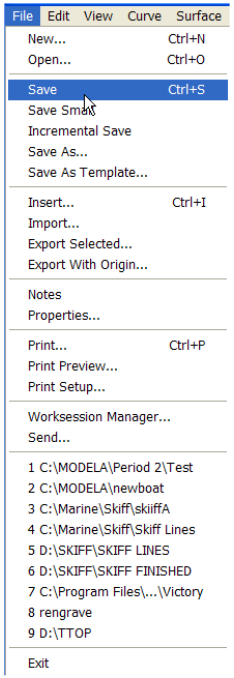
1) To save the drawing:

File – Save

or

Pick the Save icon from the top tool bar.

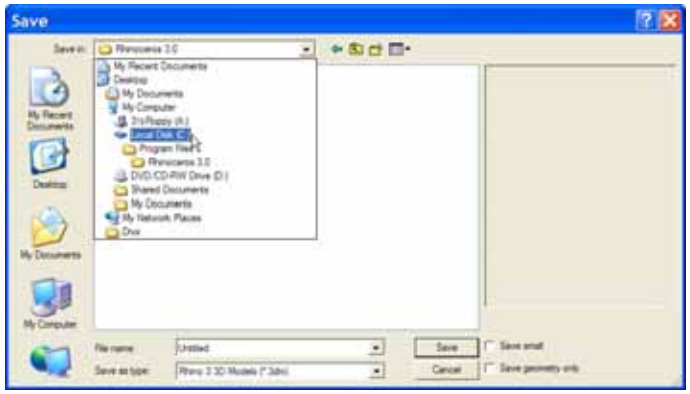




In the dialogue box, **pick the Arrow** across from the Save in box.

Click on C:

This should show in the Save in box.



Double click on MODELA

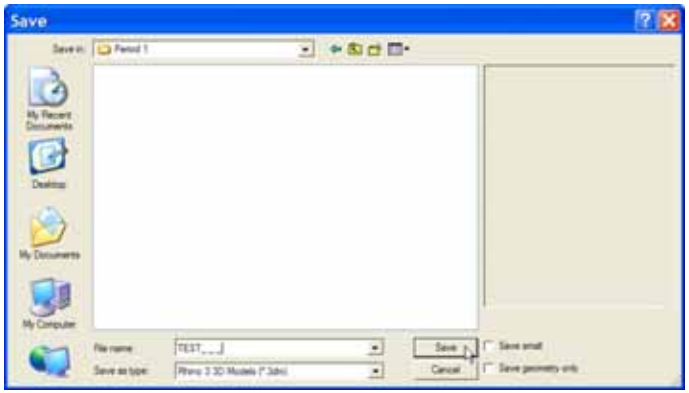
This should show in the Save in box.

Double click on the period you are in.

This should show in the Save in box.

Type, in the File name box:

TEST_ _ _ (the spaces are for your initials) i.e.: TESTSIM
Pick Save to finish the command.



Opening a drawing

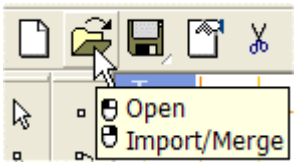
You will open the drawing that you just saved.

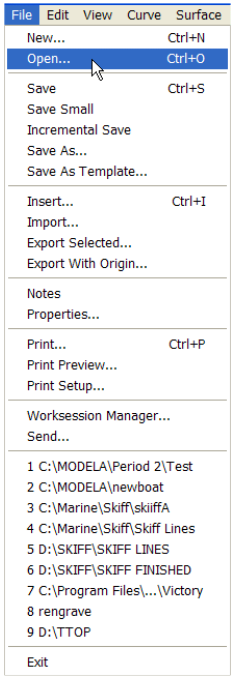
To open a drawing:

File - Open

or

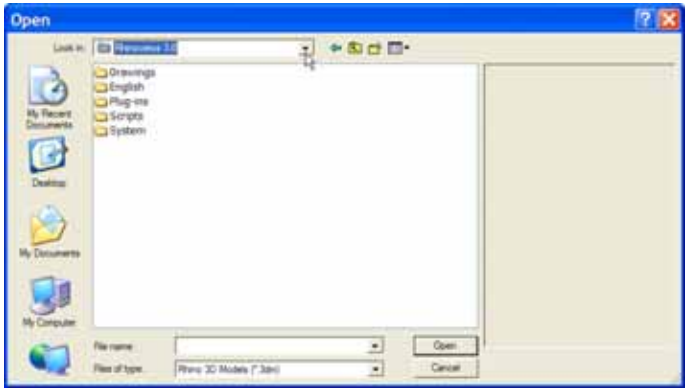
Pick the Open icon from the top tool bar.





If the save changes dialogue box appears, **select No**.

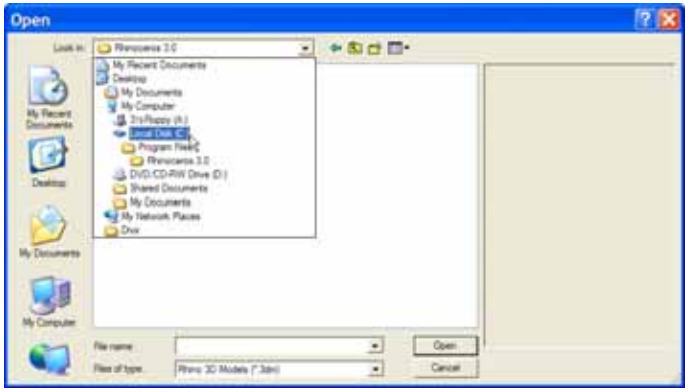
In the dialogue box, **pick the Arrow** across from the Look in box.



In the dialogue box, **pick the Arrow** across from the Look in box.

Click on C:

This should show in the Look in box.



Double click on Modela

Modela should show in the Look in box.

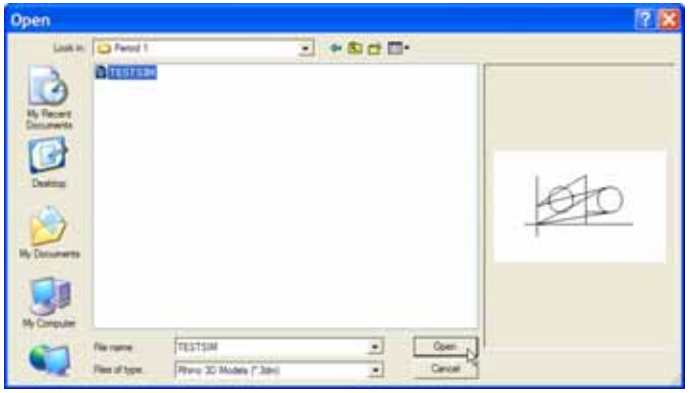
Double click on the period you are in.

This should show in the Save in box.

Pick on the desired file name.

In this case:

TEST_ _ _ (the spaces are your initials)



Pick Open

Selecting Geometry

When you want to edit any of your Rhino geometry, you must select it. There are three main ways geometry is selected.

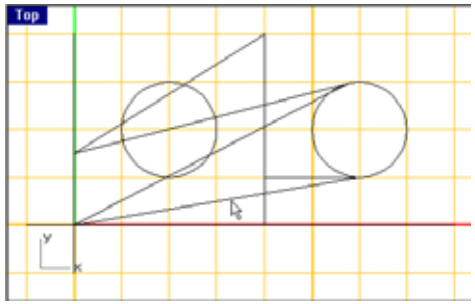
1) By Picking

2) Window

3) Crossing

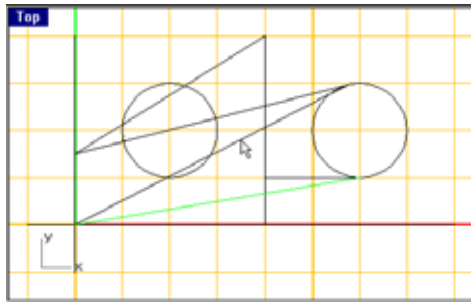
You will use the TEST drawing that you just opened to practice.

By Picking



At a command prompt, **pick on the line as displayed**.

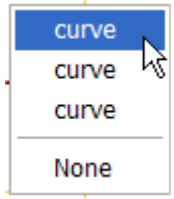
It will change color, showing that is selected.



Hold the Shift key down and select other geometry.

Press the Esc key to de-select everything.

If you pick near two or more objects, the box will appear listing the possible geometry choices. The highlighted geometry in the box is also highlighted on the screen.

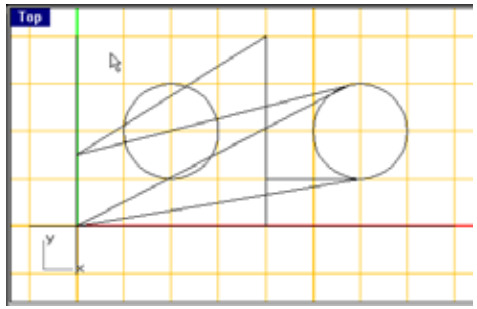


Pick when the desired geometry is highlighted.

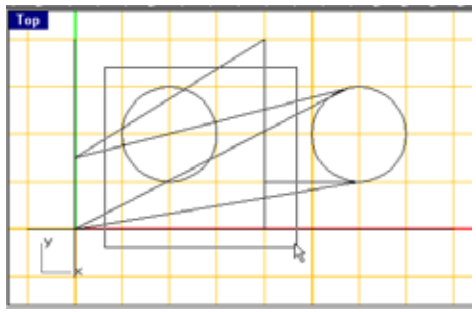
By Window

Using Window you can select multiple pieces of geometry quickly. Window selects any geometry that is completely inside of the window.

Pick in empty space as displayed.



Move to the right and down, in this case and **pick as displayed**.

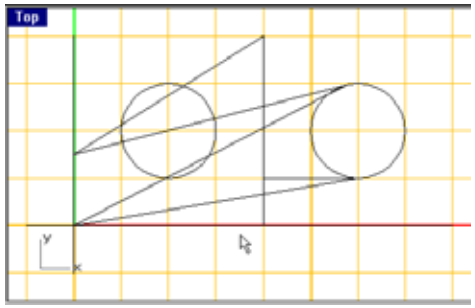


Only the circle is selected, as it is the only thing completely in the window.

Press the Esc key to de-select everything.

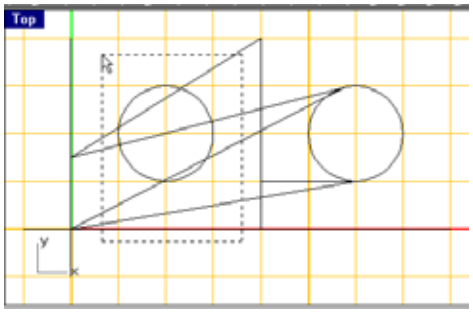
By Crossing

Crossing works very much like window but it selects everything that it crosses and everything inside of it. Instead of picking left to right, crossing goes right to left.



Pick in empty space as displayed.

Move to the left, in this case, and up and **pick as displayed**.

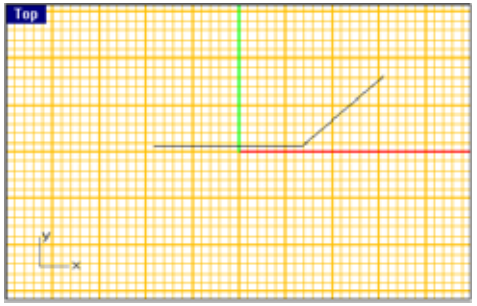


Many more items are selected than with window.

UNDO REDO and U

Undo cancels the work you did in the last command. Many times a user will use Undo at the wrong time. Redo reverses Undo but it must be used immediately after Undo command.

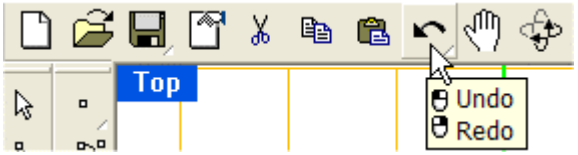
1) **Start** a new drawing with 4 Viewports.



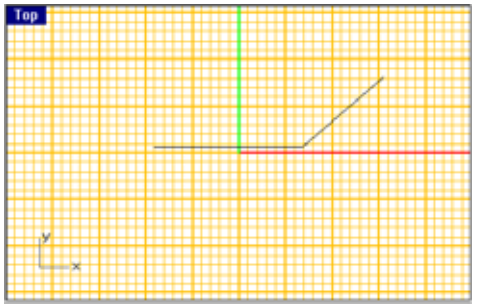
Using the Line Segments command, draw the following and press Enter to end the command. (you should be at the command prompt)

Left click on the Undo/Redo icon on the top toolbar.

The command will be undone and the lines will disappear.



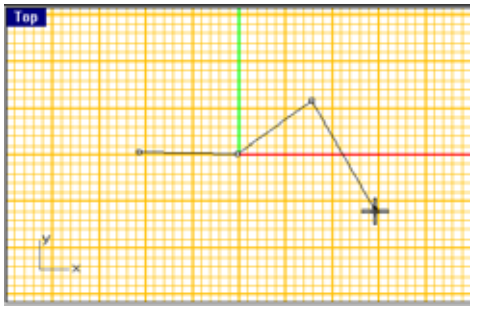
2) Without running any other command, **Right click on the Undo/Redo** icon to run Redo. The line segments will reappear as the Undo has been reversed.



From a command prompt, typing U will run the Undo command. You must type out Redo.

U

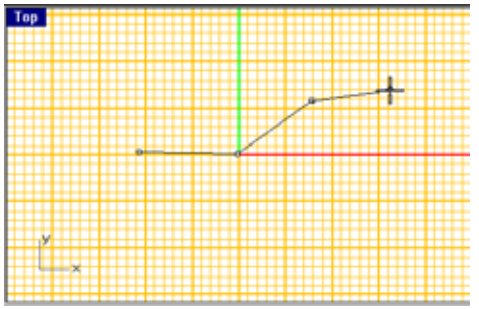
U is a very useful option when drawing line segments and curves. If you draw a line segment incorrectly it will let you back up one segment and leave you in the command.



1) Using the Line Segments command, draw the following and **STAY in the command.**

Type U and Enter. (the U is not case sensitive).

The last segment is eliminated and you can redraw it.



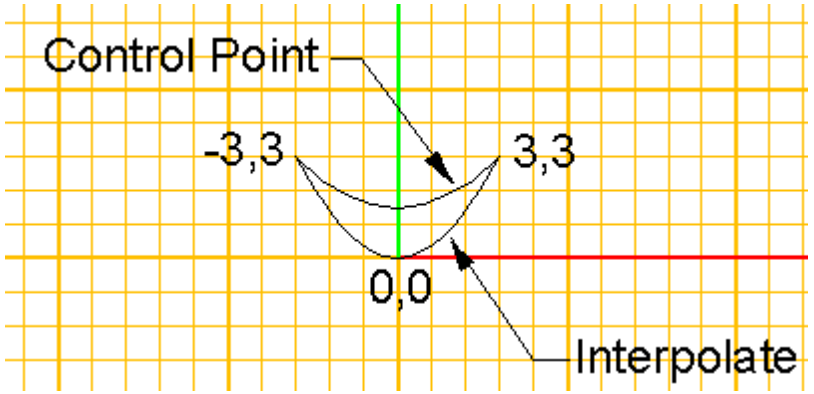
You can U back as many times as there are segments or points picked in the case of curves.

Drawing Curves

You will be using the Rhino curve commands often in the modeling process. The two Free-form curve commands you will use the most are Control Points and Interpolate Points.

For the beginner it will seem like Interpolate Points is a better command than Control Points. The reason for this is, Interpolate Points runs a curve right through where you pick. Control points forms a curve through the beginning and end points but uses the intermediate points as gravity and does not cross through them.

The example uses the same coordinates to draw both a Control Point and an Interpolate Point curve. From this it looks like Interpolate is a easier command to use.



However, in most cases, Control Points is better as it automatically creates a smooth curve between pick points. Interpolate creates bumps in most complex curves. Use Control Point whenever possible.

1) Start a new drawing with the **Millimeters template**.

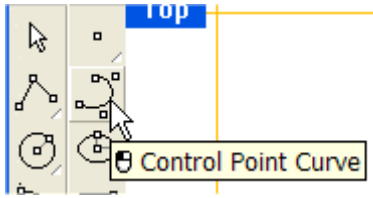
Control Points Curve

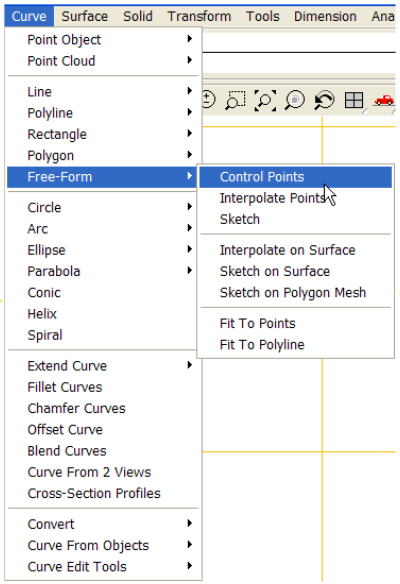
1) Start the Control Points Curve command.

Curve – Free Form – Control Points

or

Pick the Control Points icon from the tool bar.





In the Top Viewport:

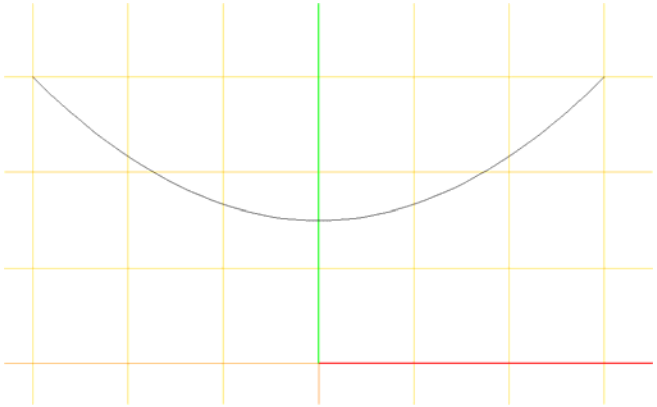
Start of curve (Degree=3): **-3,3 Enter**

Next point (Degree=3 Undo): **0,0 Enter**

Next point. Press Enter when done (Degree=3 Undo): **3,3 Enter**

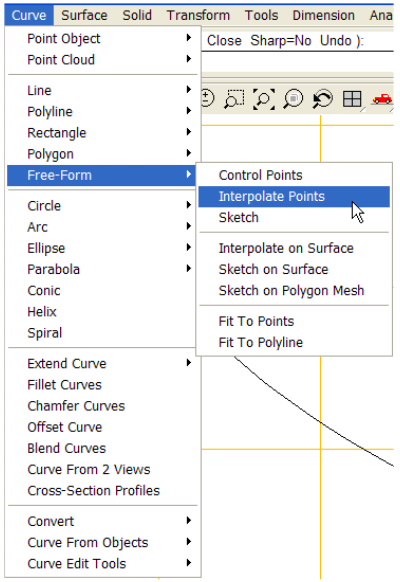
Next point. Press Enter when done (Degree=3 Close Sharp=No Undo): **Enter**

Result



Interpolate Points

2) Start the Interpolate Points Curve command.

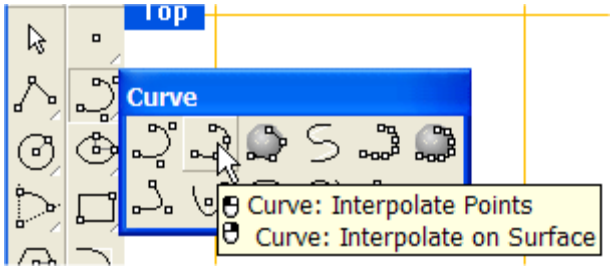


Curve – Free Form – Interpolate Points

or

Pick the Interpolate Points command from the toolbar by:

Right click on the Control Points Curve icon and select **the Interpolate Points Curve command**.



In the Top Viewport:

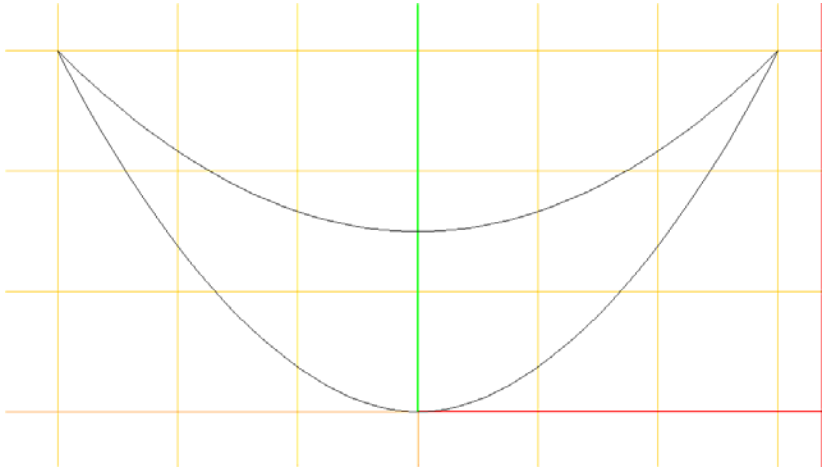
Start of curve (Degree=3 Knots=SqrtChord StartTangent): **-3,3 Enter**

Next point (Degree=3 Knots=SqrtChord EndTangent Undo): **0,0 Enter**

Next point. Press Enter when done (Degree=3 Knots=SqrtChord EndTangent Undo): **3,3 Enter**

Next point. Press Enter when done (Degree=3 Knots=SqrtChord EndTangent Close Sharp=No Undo): **Enter**

Result



This Introduction covers the basic information you need to be successful in completing the following lessons. Review any information that you need to before continuing on to the following lessons.

2

Bottle - Rhinoceros

In this first Rhino design lesson you will be modeling a bottle using a two-rail sweep. This is a very useful and fairly simple process that can be applied to create a variety of other shapes. First, you will be building a template that you will use in creating all of the parts in this series.

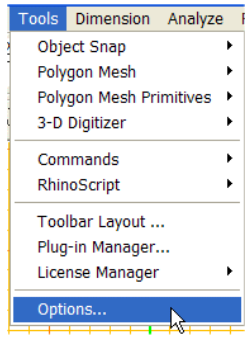
It is assumed that you have completed and understand the information presented in Lesson 1. IF NOT, please review Lesson 1 – Rhinoceros Basics.

Creating a Template

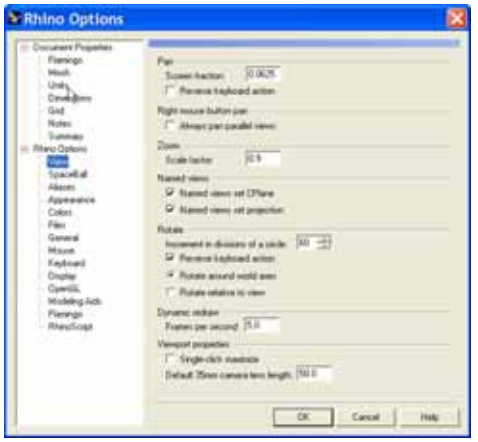
A template is a planned environment in which you can model. Some of the items that can be set are Units, Layers, Grid, Snap and the area of the drawing space.

1) Start Rhino.

2) Select the Tools pull down menu and **Options**.

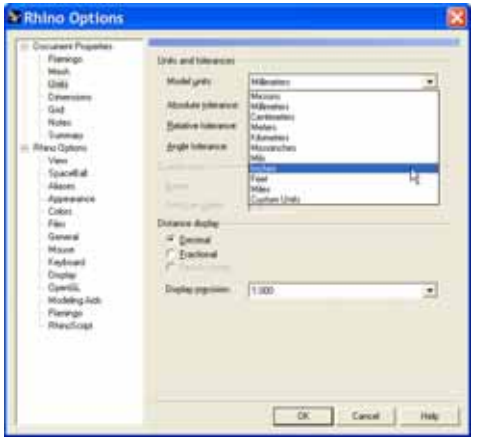


3) From the list on the left, **pick Units**

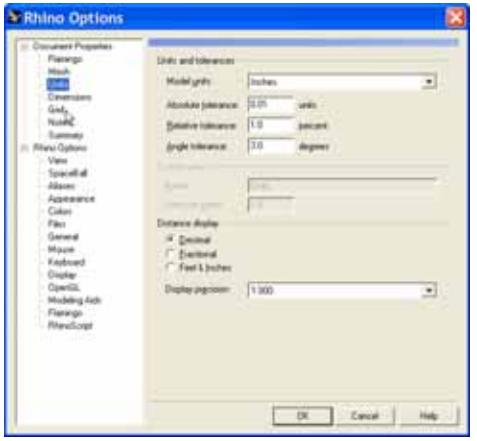


4) Pick the arrow across from **Model units**.

5) Select Inches from the list.

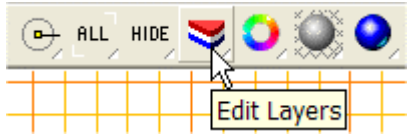


6) Select Grid from the list on the left.

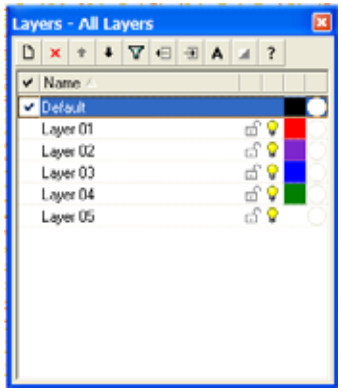


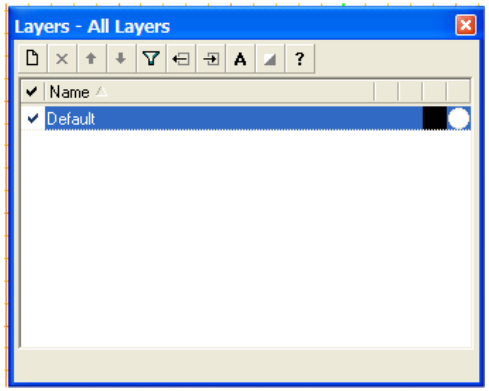
7) Set the dialogue box on the right as displayed, if it is not already, and **pick OK**.

8) Pick the Edit Layer icon.



And the Layer dialogue box is displayed

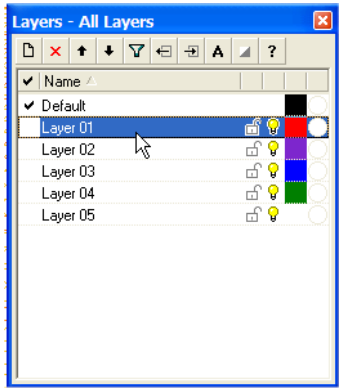




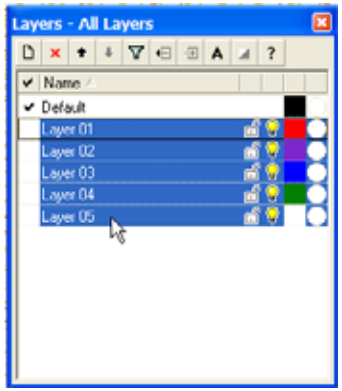
It may come up with only the default layer or with a series of layers.

If it comes up with multiple layers do the following:

Select Layer 01 by clicking on the name.

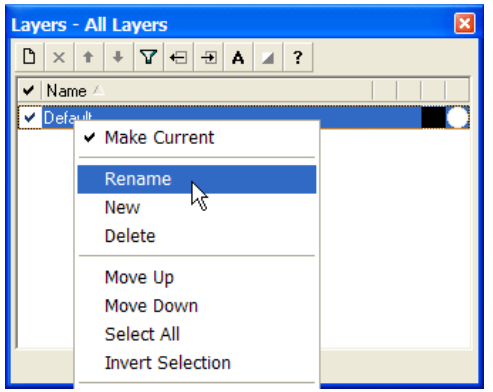


Hold the Shift Key on the keyboard down and **select the last layer** on the list, so all but the Default layer is highlighted.

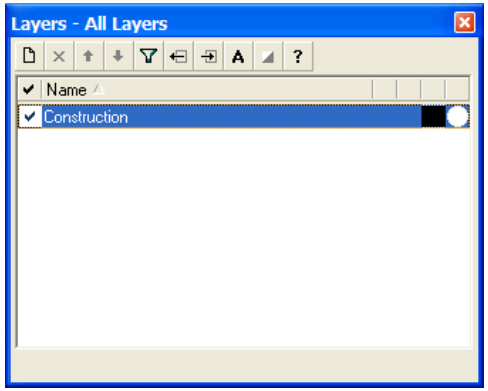


Press the Delete key on the keyboard to erase all but the Default layer.

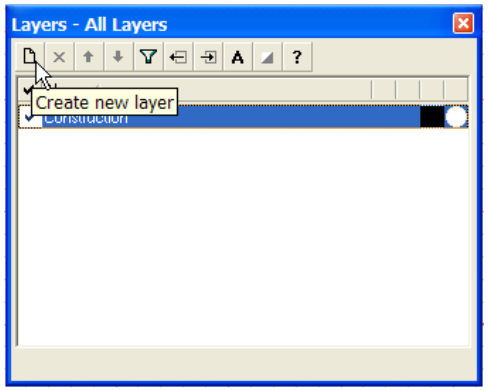
9) Right click on the Default layer name and **select** Rename.



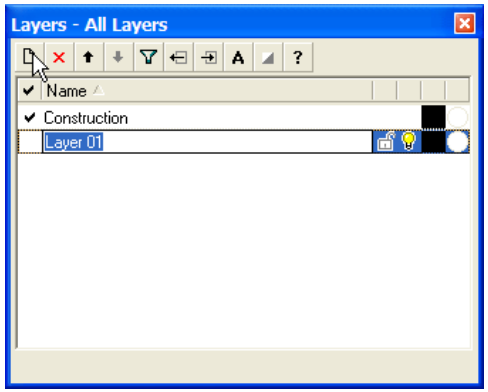
10) Type Construction for the new layer name and **press Enter** on the keyboard.



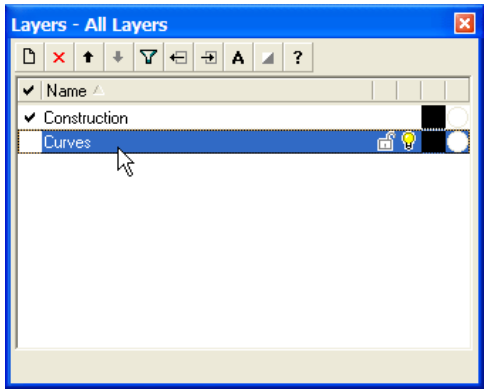
11) Select the Create new layer icon in the Layer dialogue box.



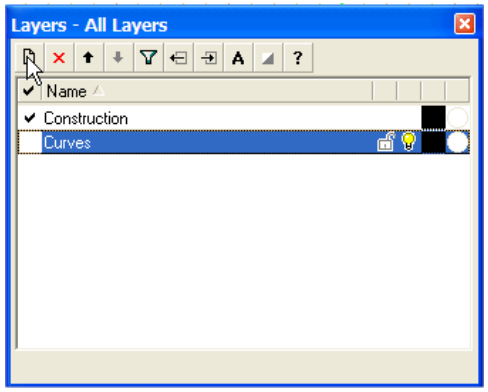
12) A new layer name will appear which will be changed:



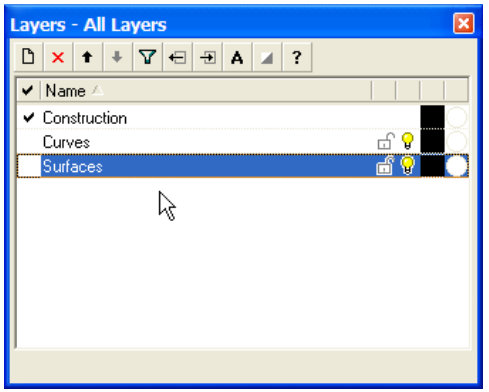
Type Curves for the new layer name and **press Enter**



13) Select the Create new layer icon again.

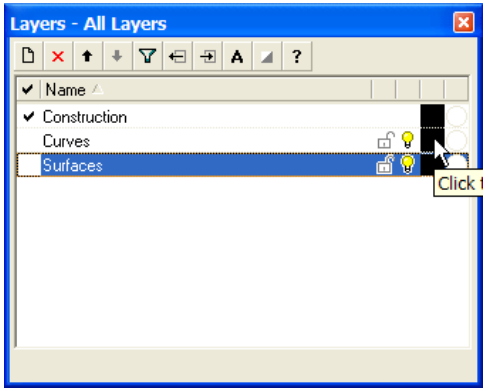


Rename the new layer: **Surfaces** and **press Enter**.



14) All of the layer colors are black. The colors for layers Curves and Surfaces will be changed.

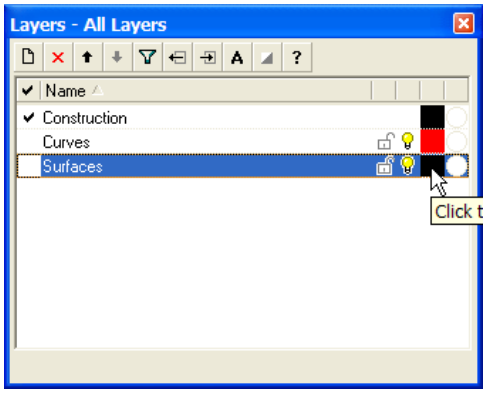
Select the color block across from Curves.



In the Select Color dialogue box:

Click Red and **pick OK** at the bottom of the dialogue box.

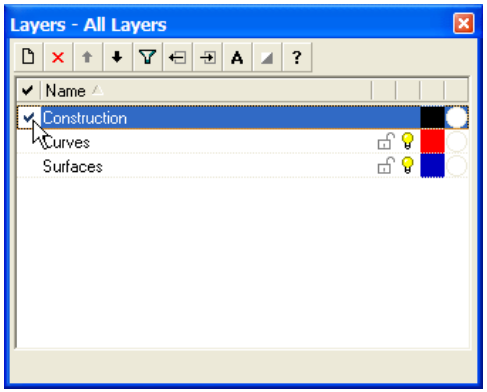
Select the color box across from Surfaces.



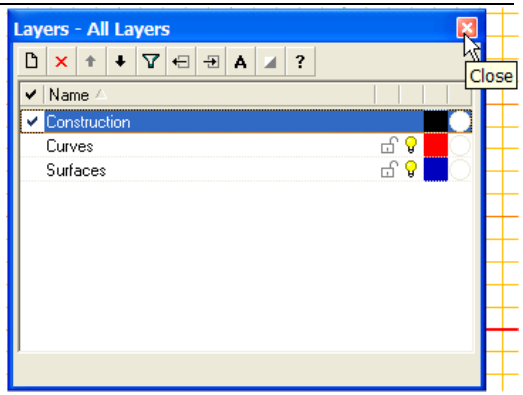
In the Select Color dialogue box:

Click Dark Blue and **pick OK** at the bottom of the dialogue box.

15) Click the Make Layer Current box to the left of Construction.

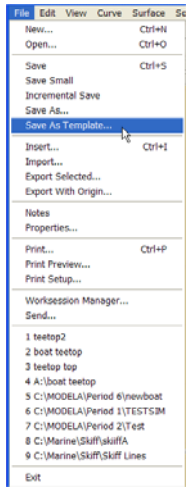


16) To close the Layer dialogue box, **pick the X** in the upper right corner.



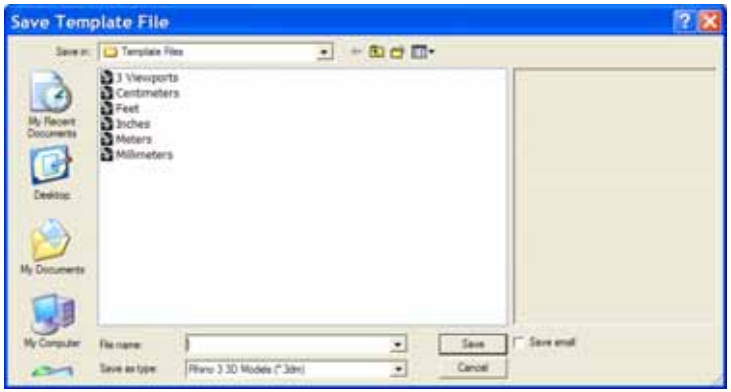
The next steps save the template for future use.

17) Select File and **Save As Template…**

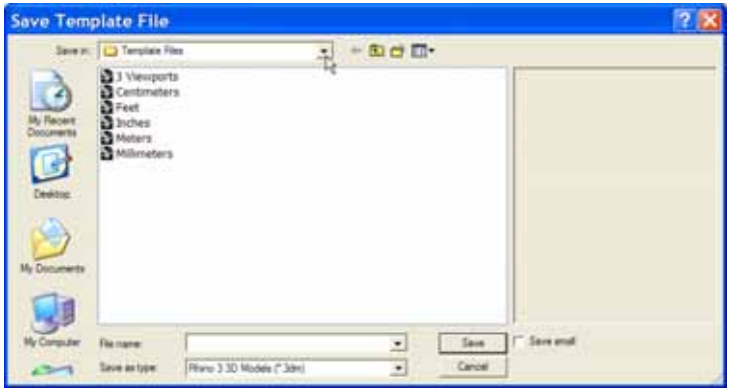


The Save Template File dialogue box appears with Template Files as the “Save in” directory. This is the standard template File location.

Your template file is going to be saved in your period directory.



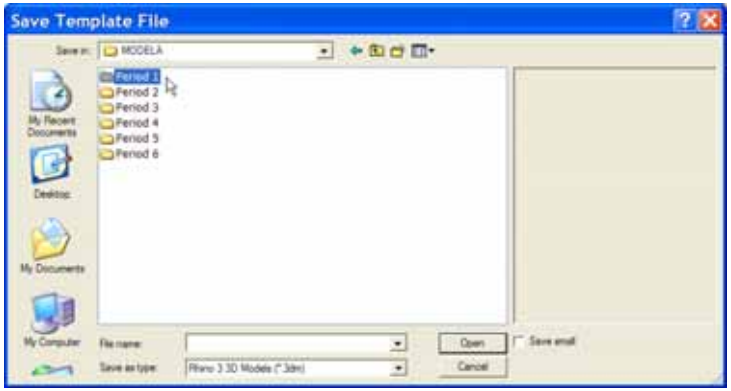
18) Select the arrow across from Template Files.



19) From the listing, **pick C: .**

20) Double click the Modela directory.

21) Double click your class period directory.



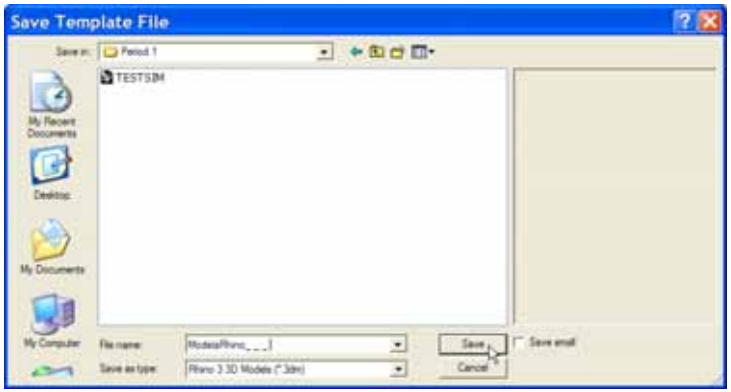
22) For the file name input:

ModelaRhino_ _ _

The spaces are for your initials.

i.e. ModelaRhinoSIM

Pick Save.



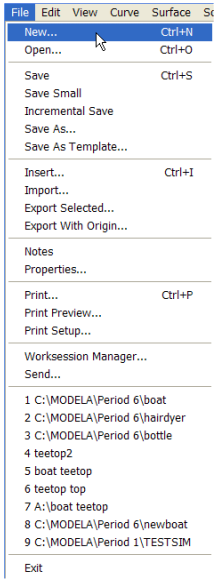
23) Exit Rhino

Creating the Bottle

1) Start up Rhino.

The next few steps call up the template file.

2) Select the File pull down menu and **New…**



or

Select the **New** icon from the toolbar

The Open Template File dialogue box appears with Template Files as the "Look in" directory. This is the standard template file location.

Your template file is in your period directory.

3) Select the arrow across from Template Files.

4) From the listing, **pick C: .**

5) Double click the Modela directory.

6) Pick your class period directory.

7) Select your template file:

ModelaRhino_ _ _

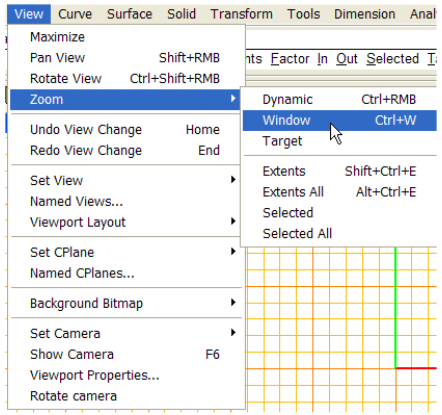
The spaces are for your initials.

i.e. MODELARHINOSIM

Pick Open.

Rhino is now using your template file.

8) **In the top viewport**, zoom window into an area as displayed.

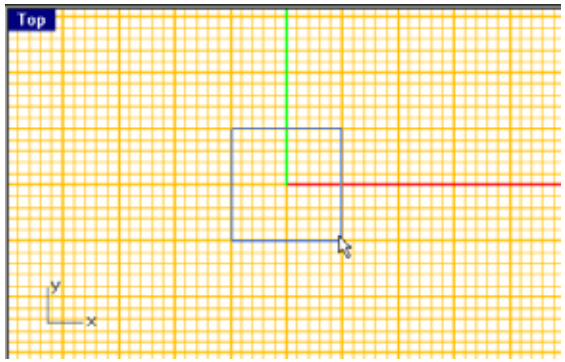


View – Zoom – Window

or

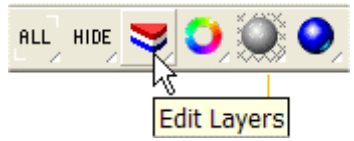
Pick the Zoom Window icon from the top tool bar.

Select two points, forming a window as displayed.



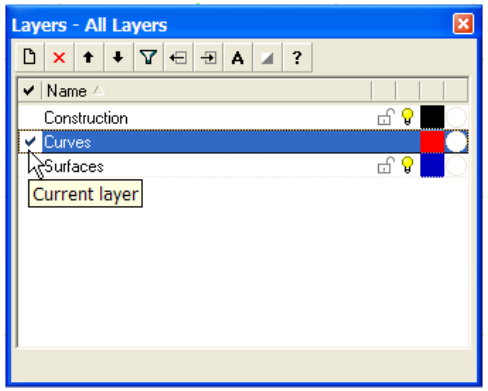
9) To see the layer:

Pick the Layer Edit icon from the toolbar.



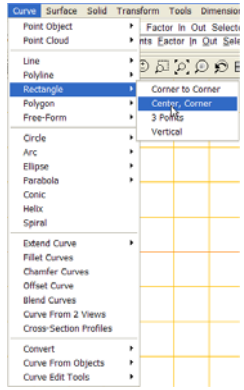
Set the Curves layer as current.

Pick the X in the upper right corner of the Layers dialogue box to close it.



10) To draw the first piece of geometry, a rounded rectangle for the base of the bottle, **select:**

Curve – Rectangle – Center, Corner



With the **Top viewport active**:

Center of rectangle (Rounded): **R Enter**

(This sets the command to create a rounded end rectangle.)

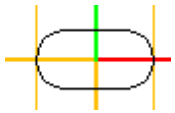
Center of rectangle: **0 Enter**

Corner or length: **2 Enter**

Width. Press Enter to use length: **1 Enter**

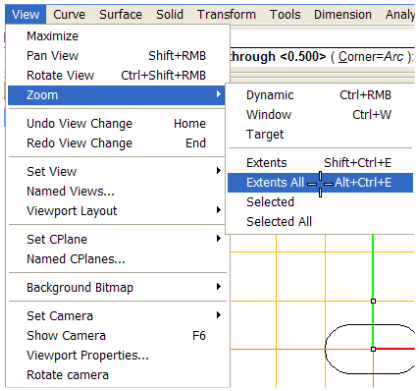
Radius or point for rounded corner to pass through <0.500> (Corner=Arc): **Enter**

Result



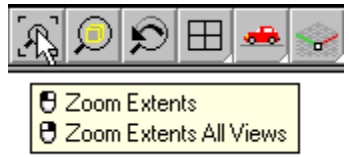
11) To zoom into the rectangle, in all views, **Zoom Extents All**.

View – Zoom – Extents All

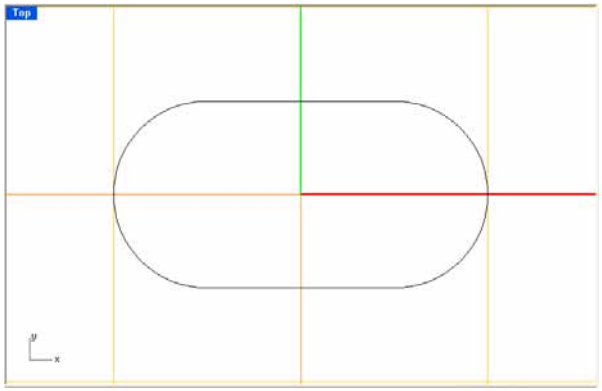


or

Right pick the Zoom Extents All icon from the top tool bar.



Result

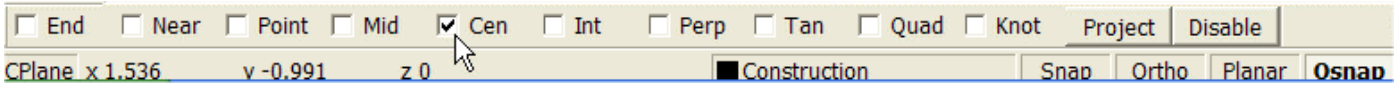


12) The circle forming the mouth of the bottle will be created next.

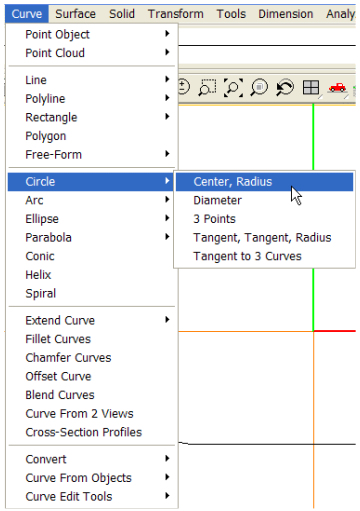
The Center Object snap needs to be turned on.

If the Osnap box is not displayed, pick Osnap from the bottom of the graphics screen.

Set the Osnap box so that **only** the **Center (Cen) is activated**.



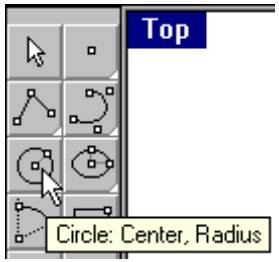
Start the Center, Radius circle command.



Curve – Circle – Center, Radius

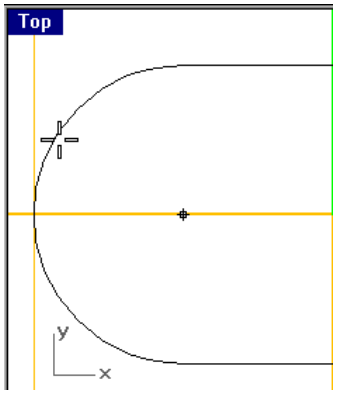
or

Pick the Circle command from the Draw toolbar from the left side of the graphics screen.



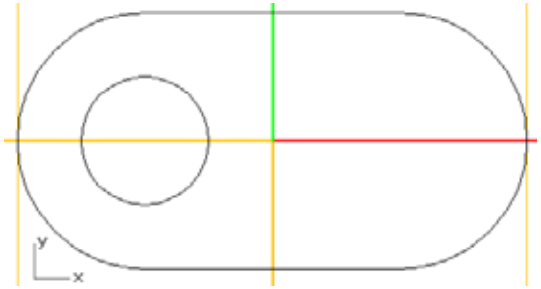
Center of circle (Deformable Vertical Diameter 3Point Tangent AroundCurve):

Pick as displayed

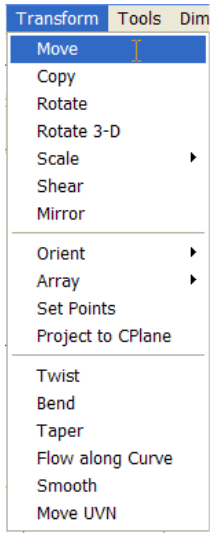


Radius <1> (Diameter): **.25 Enter**

Result



13) The circle is on the same plane as the rectangle and needs to be moved to form the top of the bottle.



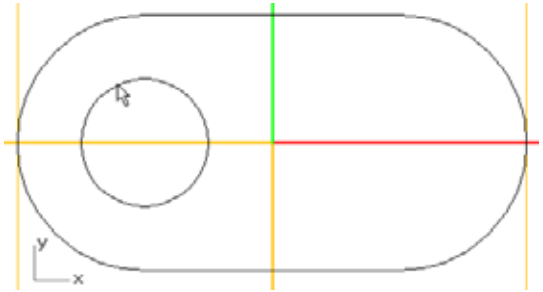
Transform – Move

or

type: M Enter from a command prompt.

In the **Top Viewport**:

Select objects to move: **Pick the circle**



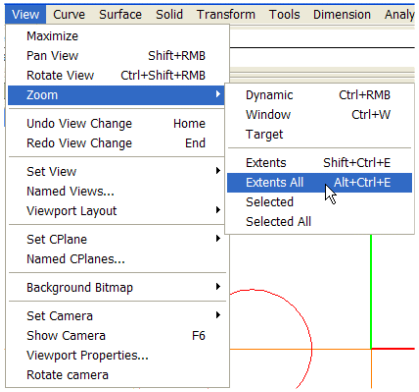
Select objects to move. Press Enter when done: **Enter**

Point to move from (Vertical=No):

With the **Center Osnap on, pick the circle**

Point to move to: **R0,0,5 Enter**

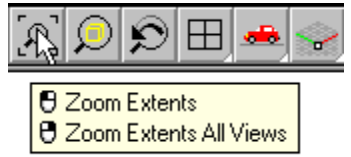
The results will be hard to see until you zoom extents.



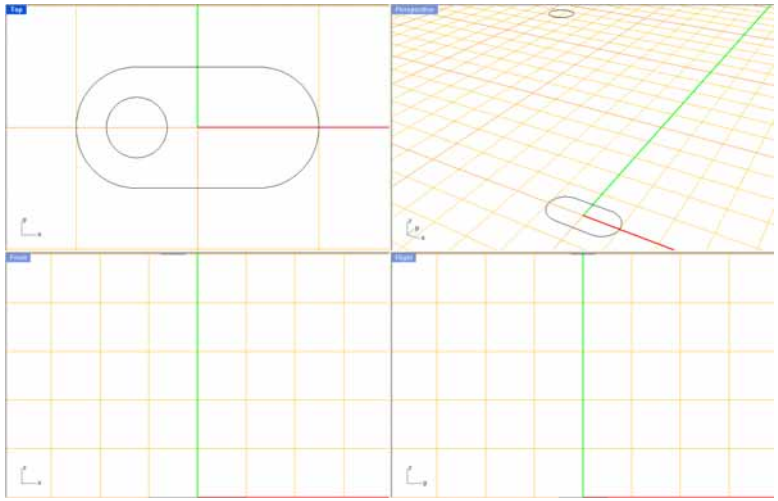
View – Zoom – Extents All

or

Right pick the Zoom Extents All icon from the top tool bar.



Result (you can see the circle above the rectangle most easily in the Perspective view)



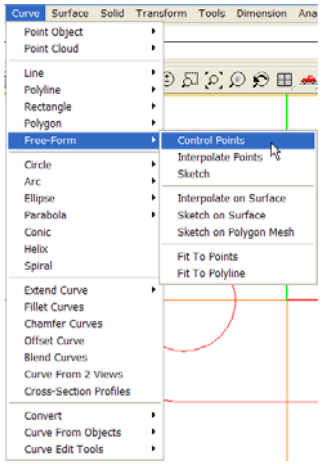
The rail curves are the next to be created. The Control Point Curve command will be used to draw this geometry.

14) The left curve will be the first to be created.

Set the Osnap box so only Quadrant (Quad) is on.

Start the Control Point Curve command:

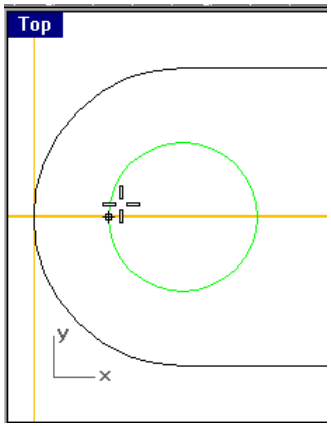
Curve – Free-form - Control Points



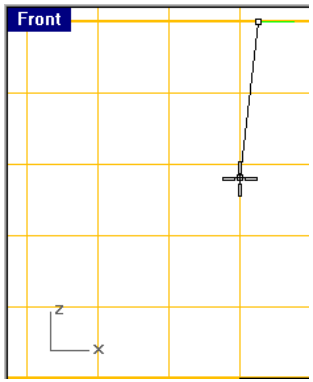
or

Pick the Control Point Curve command from the Draw toolbar from the left side of the graphics screen.

In the Top viewport, pick the circle quadrant as displayed.

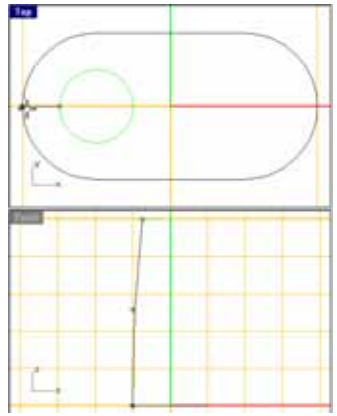


Move to the Front viewport.



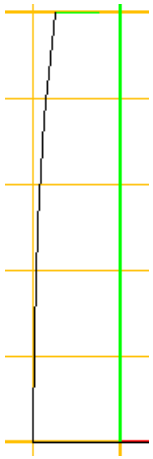
Pick as displayed, directly above the left end of the bottom rectangle.

Move to the Top viewport.



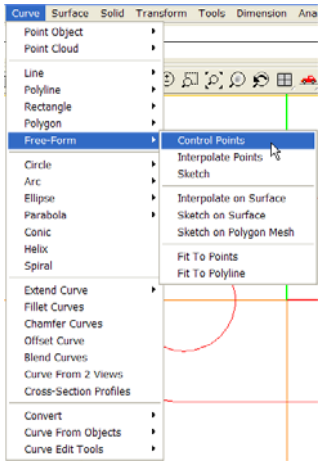
Pick the left quadrant of the rectangle as displayed.

Result



Press Enter to complete the command.

15) To create the right rail curve for the bottle:

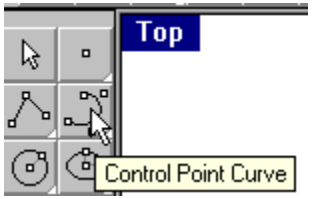


The Osnap box should still be set so that only Quadrant (Quad) is on.

Start the Control Point Curve command:

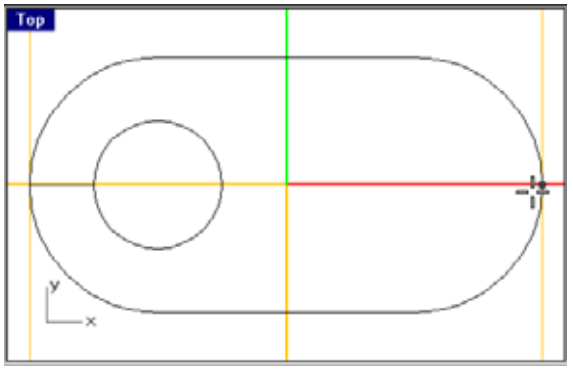
Curve – Free-form - Control Points

or



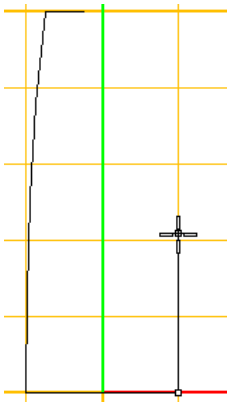
Select the Control Point Curve command from the Draw toolbar from the left side of the graphics screen.

In the Top Viewport, pick the right quadrant of the rectangle as displayed.

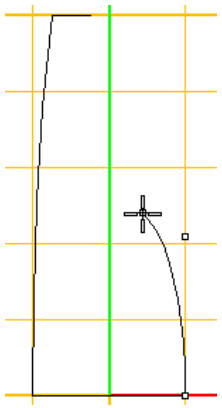


Move to the Front Viewport.

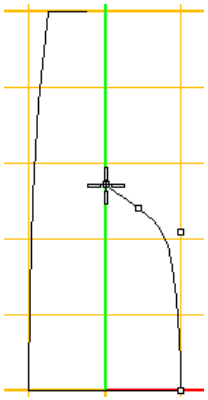
Hold the shift key down, which will cause Ortho to be on for the pick, and **pick as displayed**.



Pick as displayed.

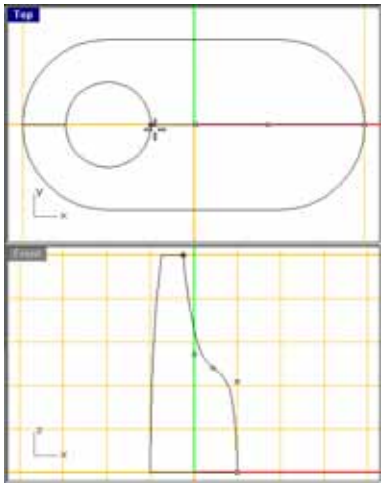


Pick as displayed.



Move to the Top Viewport.

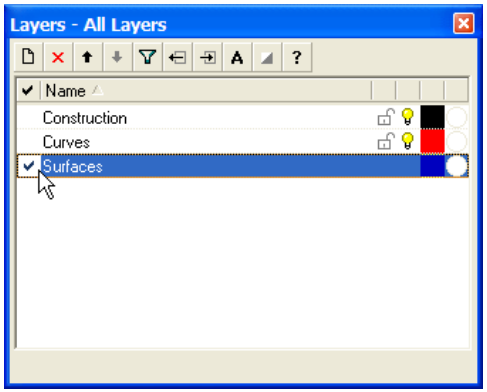
Pick on the right quadrant of the circle as displayed.



Press Enter to complete the command.

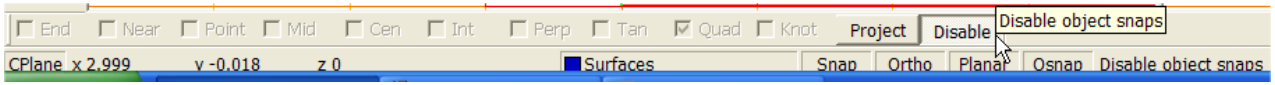
The basic geometry is complete and ready for surface creation.

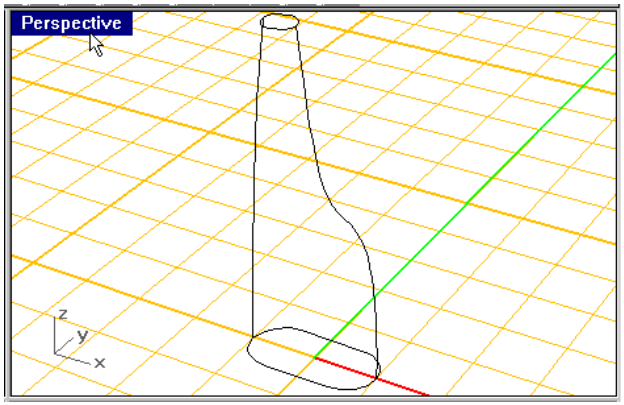
16) Change the current layer to **Surfaces**. Then **close** the Layer dialogue box.



17) To create the main surface of the bottle:

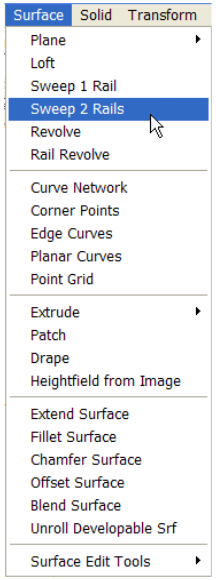
Disable the Osnaps





Double click on the word Perspective in the Perspective Viewport.
(This will make this viewport cover the complete graphics screen)

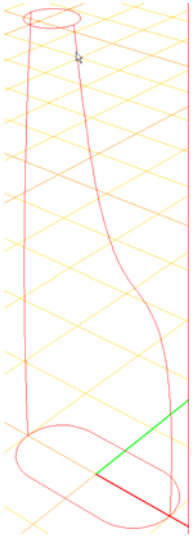
Start the Sweep 2 Rails command.



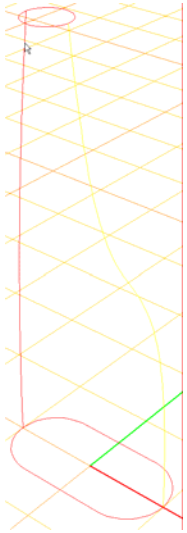
Surface – Sweep 2 Rails

READ THIS!!!!!!!!!

Make the next two picks on the same side of the objects as displayed or the surface will be twisted.

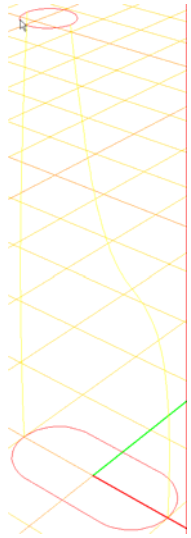


Select 2 Rail curve: **Pick the left Control Point curve** as displayed.

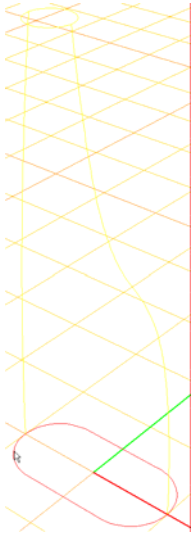


Select 2 Rail curve: **Pick the right Control Point curve as displayed**

Select cross-section curves (Point): **Pick the top circle as displayed**

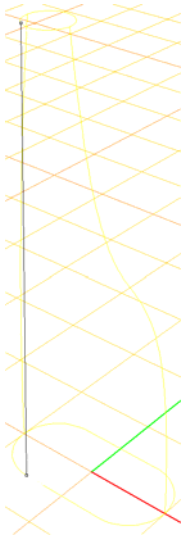


Select cross-section curves (Point): **Pick the bottom arc as displayed**



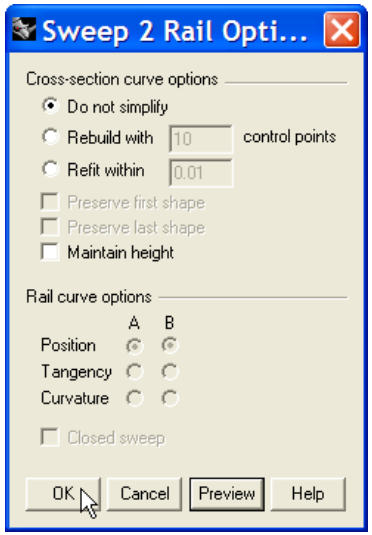
Press Enter to finish the cross section selection.

The Seam prompt will appear. The seam is the edge where the two edges of the surface wrapped around the bottle meet.

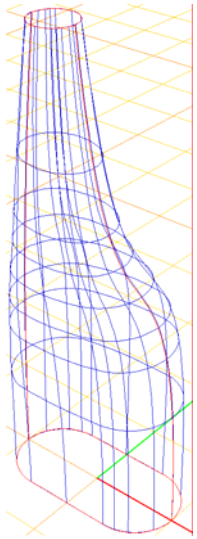


Select Seam Point to adjust. Press Enter when done (FlipDirection Automatic Natural): **Enter**

With the Sweep 2 Rail Option dialogue box set as displayed, **select OK**.



Result



18) To see the bottle shaded:

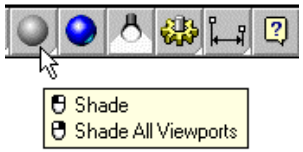
Pick the Shade icon on the top tool bar.

(it is the gray sphere)

Result

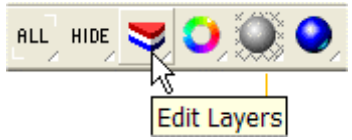
(Notice the top of the bottle is open)

Right Click on the Shade Icon to end the shading.



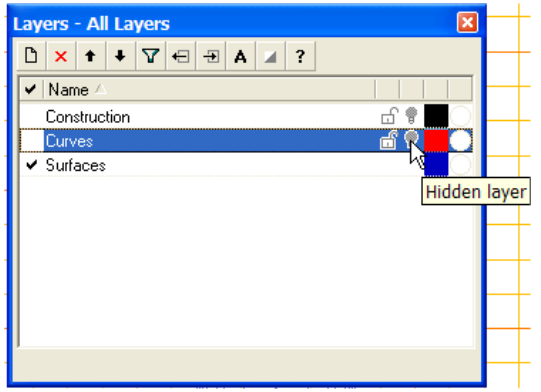
19) Freeze the Construction and Curves layers.

If the Layer dialogue box is not open, **pick the Edit Layers** icon from the toolbar.



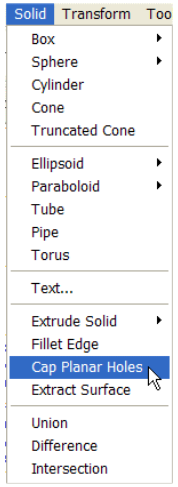
In the Layers dialogue box **freeze the Construction and Curves** layers by **picking on the light bulb** in the desired layer row.

The bulbs will turn gray when the layer is frozen.



Close the Layer dialogue box.

20) The top and bottom of the bottle are open. For the machining process in the next lesson they need to be closed.



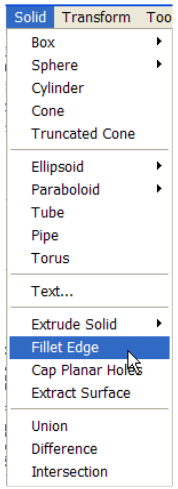
Solid – Cap Planar Holes

Pick anywhere on the bottle.

Press Enter to complete the command.

This turns the bottle from a surface model to a solids model.

21) A fillet needs to be places around the bottom edge of the bottle.



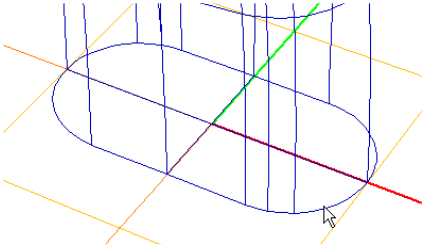
Solid – Fillet Edge

Select edges to fillet (Radius=1): **.125 Enter**

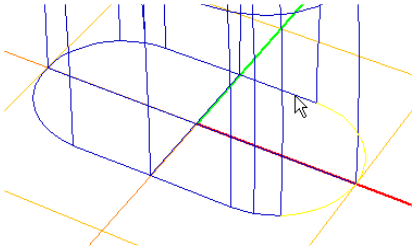
It will take 6 picks to select all of the edge geometry at the bottom of the bottle. Pick the edges in adjoining order, do not skip around.

If you make a mistake, press ESC and start the command over.

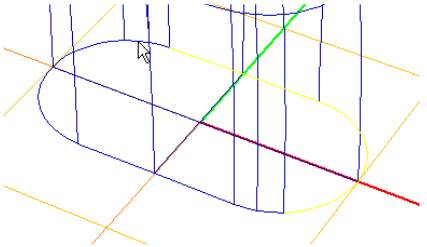
Select edges to fillet. Press Enter when done (Radius=0.125): **Pick as displayed**



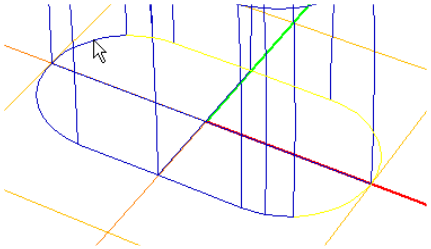
Select edges to fillet. Press Enter when done (Radius=0.125): **Pick as displayed**



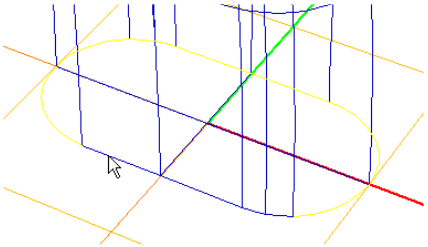
Select edges to fillet. Press Enter when done (Radius=0.125): **Pick as displayed**



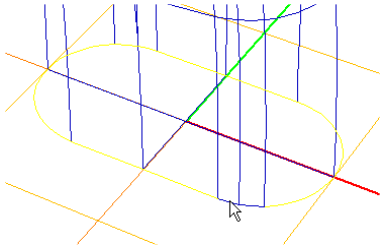
Select edges to fillet. Press Enter when done (Radius=0.125): **Pick as displayed**



Select edges to fillet. Press Enter when done (Radius=0.125): **Pick as displayed**

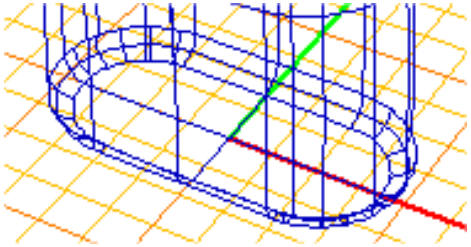


Select edges to fillet. Press Enter when done (Radius=0.125): **Pick as displayed**



Select edges to fillet. Press Enter when done (Radius=0.125): **Press Enter** to finish the selection process.

After some processing time, the fillets will appear as displayed.



22) To see the completed shaded bottle:

Pick the Shade icon on the top tool bar.

(it is the gray sphere)

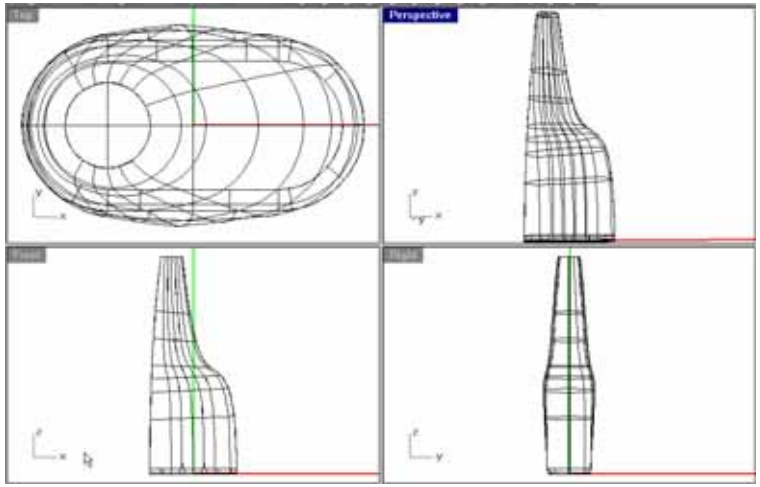
Result

Holding down the right mouse button and moving the mouse will rotate your view of the object.

Right click on the Shade icon to end the shading.

23) To return to four viewports, **double click on the word Perspective** in the Perspective Viewport.

Result

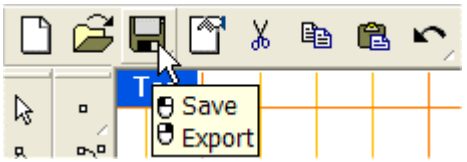


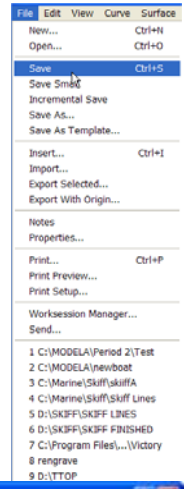
24) To save the bottle:

File – Save

or

Pick the Save icon from the top tool bar.

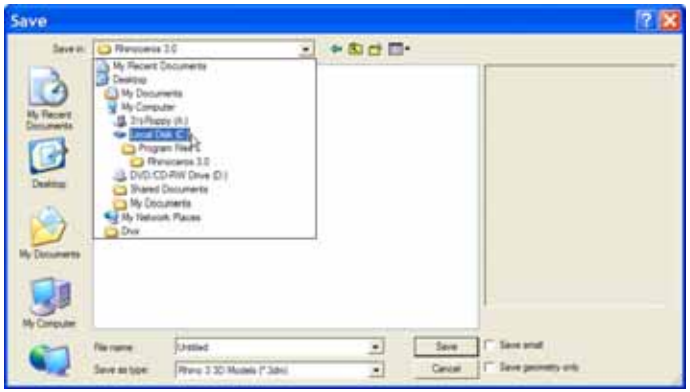




In the dialogue box, **pick the Arrow** across from the Save in box.

Click on C:

This should show in the Save in box.



Double click on MODELA

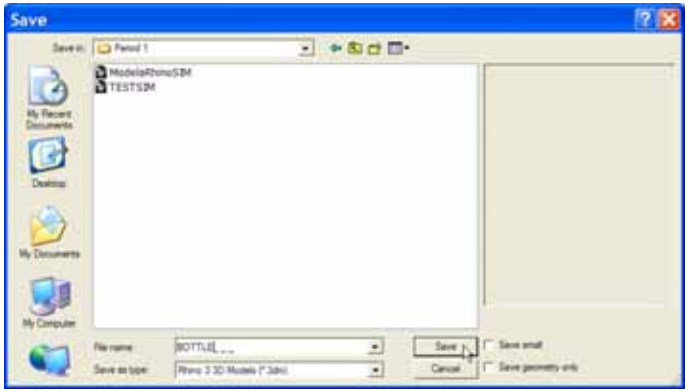
This should show in the Save in box.

Double click on the period you are in.

This should show in the Save in box.

Type, in the File name box:

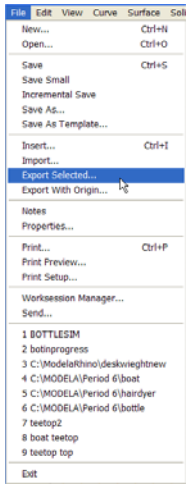
BOTTLE_ _ _ (the spaces are for your initials) i.e.: BOTTLESIM
Pick Save to finish the command.



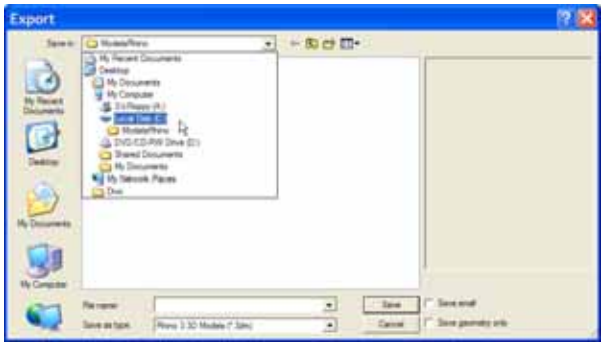
25) To create a file to send to the machining software (Modela Player):

From a command prompt, **pick the bottle** so that it is highlighted.

File – Export Selected…



In the dialogue box, **pick the Arrow** across from the Save in box and **select C:**.



Double click on MODELA

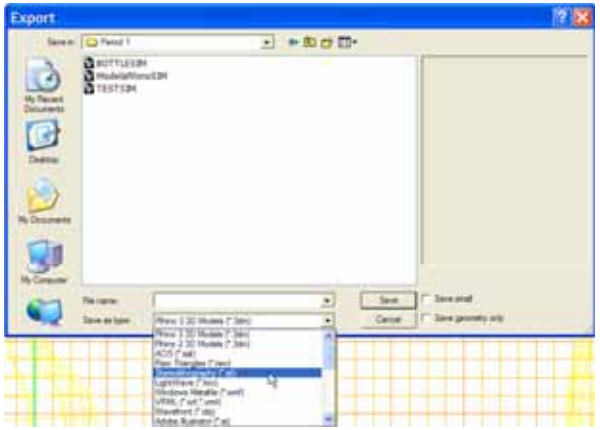
This should show in the Save in box.

Double click on the period you are in.

This should show in the Save in box.

At the bottom of the dialogue box, pick on the Save as type: bar and select Stereolithography *.stl

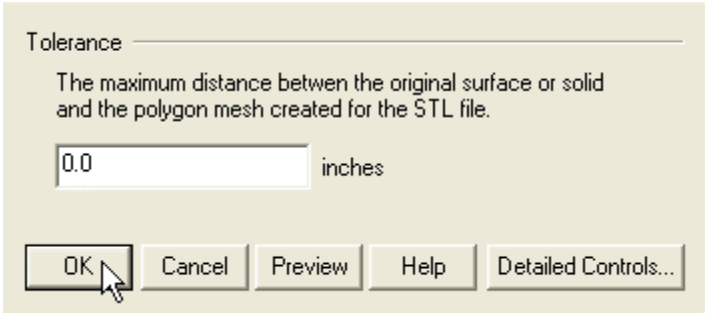
A stereolithography file is an export file type used for solid objects. These are models that are completely enclosed.



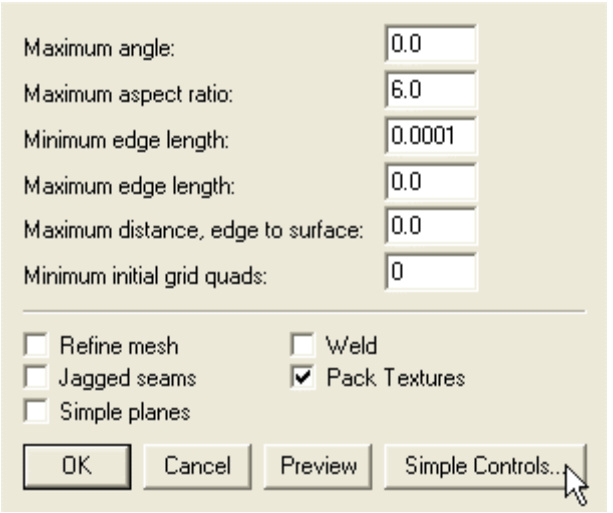
Type, in the File name box:

BOTTLE_ _ _ (the spaces are for your initials) ie: BOTTLESIM
Pick Save to finish the command.

The STL Tolerance dialogue box should be **set as follows, then pick OK**.

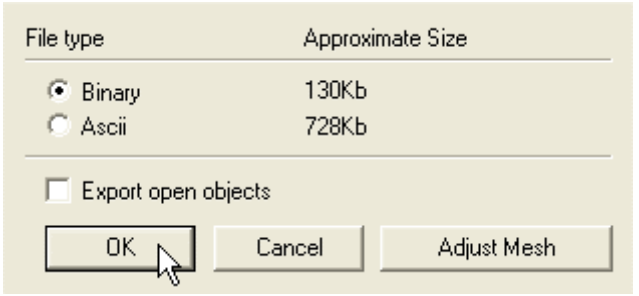


If the Detailed Tolerance dialogue box is displayed, **set it as displayed and pick Simple Controls button**.



You will then need to **pick OK** in the Simple Tolerance dialogue box

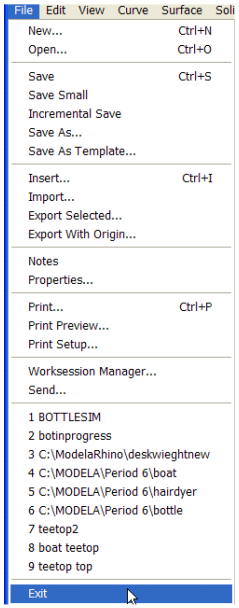
With the **Export open objects box unchecked, pick OK.**



Depending on the speed of the computer it could take upwards of two minutes to complete the export process.

26) Close Rhino.

File - Exit

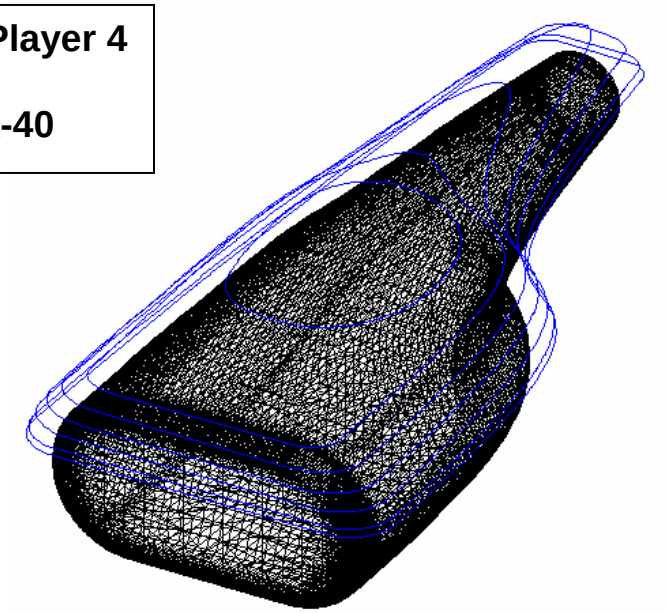


In the next lesson you will learn how to create the toolpaths to mill the bottle and machine it.

3

Modela Player 4

MDX-40



Bottle Modela Player 4

The Roland Modela Player 4 is used to created toolpaths so that your Rhino models can be prototyped. There are two machining processes (toolpaths) we are going to use in Modela Player 4. First, is Roughing which removes most of the material, forming the rough shape of the model. The Finishing toolpath creates the final, smooth shape. There are other toolpath options, but we will limit our use to roughing and finishing for this model.

All of the models in this curriculum will be cut from machining wax. Machining wax has a much higher melting temperature than candle wax so the tool bit will not melt it during the machining process. You can get the machining wax from your instructor.

The machining time for the two tool paths for the bottle, using a Roland MDX-40 is about 40 minutes. This time could vary if you are using one of the other Roland machines. This does not include setup time for either the MDX-40 or creating the tool paths in Modela Player 4. Be sure you have plenty of time to machine the part before sending the tool paths to the machine.

Always be safe when using a machine tool. Don't waste time but do not hurry. Think safe!!!!

The models we are cutting are small because it makes learning the process economical and affordable. The MDX-40 has a working envelope of 12 x 12 inches. It is very easy to adjust the size of the Rhino model inside of Modela Player 4 to accommodate larger stock and larger tools.

Below is a flow chart of the process of machining the model with the Modela Player 4 and your Roland milling machine. Be sure to notice the 1st and 2nd numbering. These need to be done in order to properly machine your work and protect your milling machine. The Roughing toolpath, which cuts away the majority of the material, must be done before the Finishing toolpath.

Modela Player 4

1st … Roughing tool path

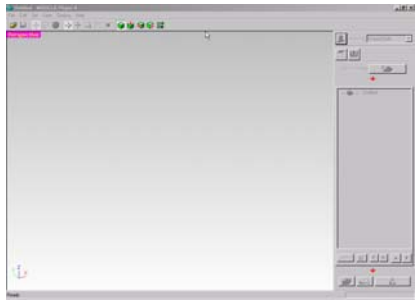
2nd … Finishing tool path

Processes:	Details:
Model Button 	>Set finished size. Set the Origin.
Cutting Surfaces Button 	>Select number of surfaces to machine.
Modeling Form Button 	>Set the depth and the cutting area
New Process Button 	>Create tool paths: One Roughing & One Finishing
Preview Button 	>Create a tool path simulation & cutting time est.
Cut Button 	>Sends the file to the MDX40 and begins cutting.

1) Start the Modela Player 4

Pick the Modela Player 4 icon from the desktop.

Modela Player 4 will open……..

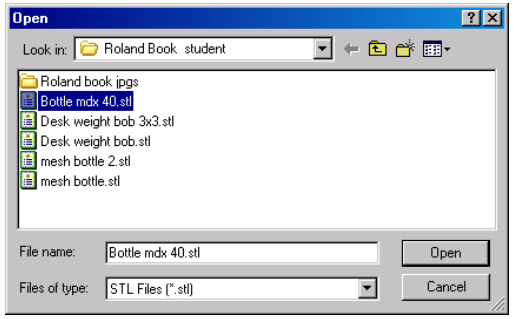


2) To open the bottle stl file:

File – Open –

Pick File from the pull down menu in the top left of the screen and

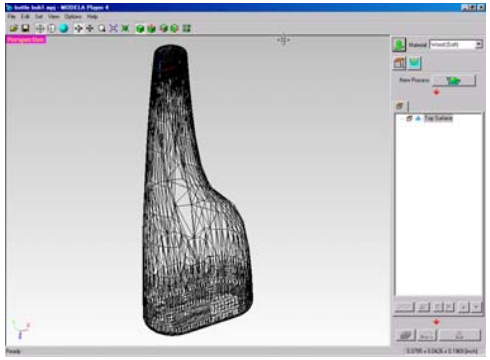
Select Open…



Select the file **BOTTLE mdx40.stl**

and **pick Open**

Result



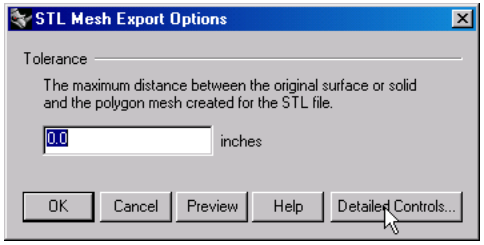
Imported model quality: (the stereolithography .stl file)

If the quality of the imported model is not fine enough for you, go back to Rhino and export the stl file again.

Open the model in Rhino.

Select the bottle model and go to **File, Export Selected, File Type, stl**

Adjust the STL Mest Export Options as follows:

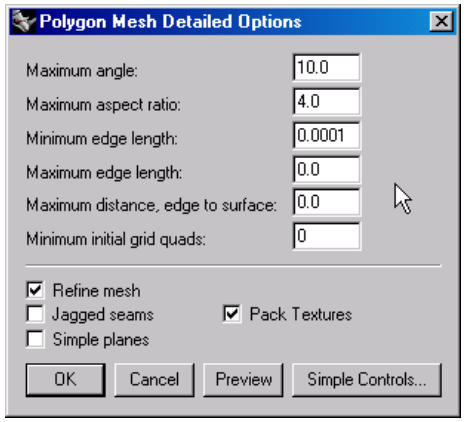


Click Detailed Controls:

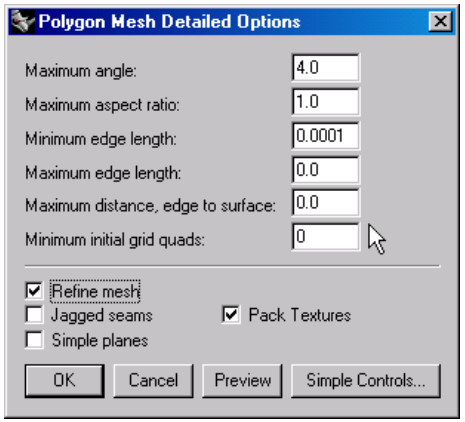
Change the settings as follows:

Export the model again as an stl file.
Experiment with the Polygon Mesh options
to get the desired results when opened in Modela 4.

Preview before selecting OK. The finer, more exact the mesh,
The larger the stl file will be.

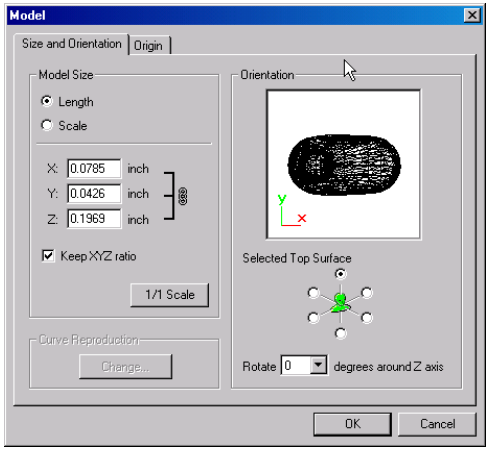


Medium quality mesh



Fine, more exact mesh

3) You also have to take into consideration the orientation (direction) of the part and the final size of the part. The mill is going to cut the part from the top of the screen down. In the current orientation, the shape would not be well defined and not recognizable as a bottle.

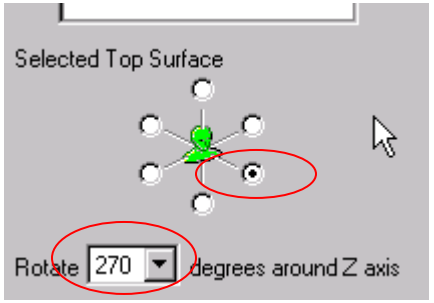


The orientation of the bottle needs to be changed, so that the side of the bottle is facing up.

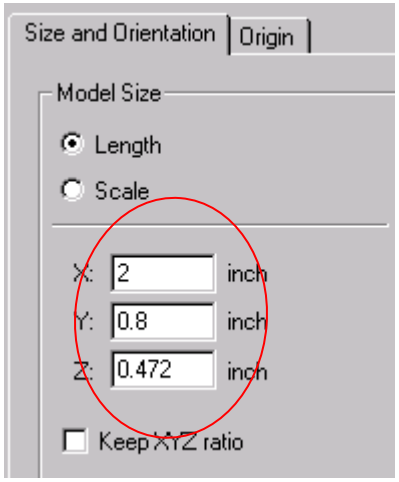
The Finished size of the bottle needs to be set.

To change the direction (orientation) of the part:

Pick the Model button. 



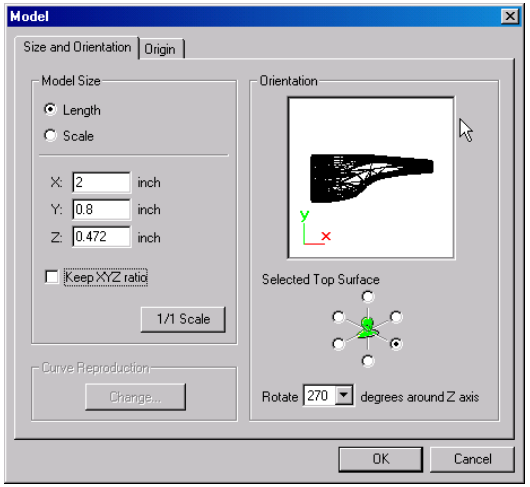
Select the "from Front" radio button, as displayed to flip the bottle to the correct orientation. It is the bottom right button. **Rotate the bottle 270** around the Z for the correct view.



Set the finished size of the model. We are choosing 2 x .8 x .472, which will easily fit our wax block.

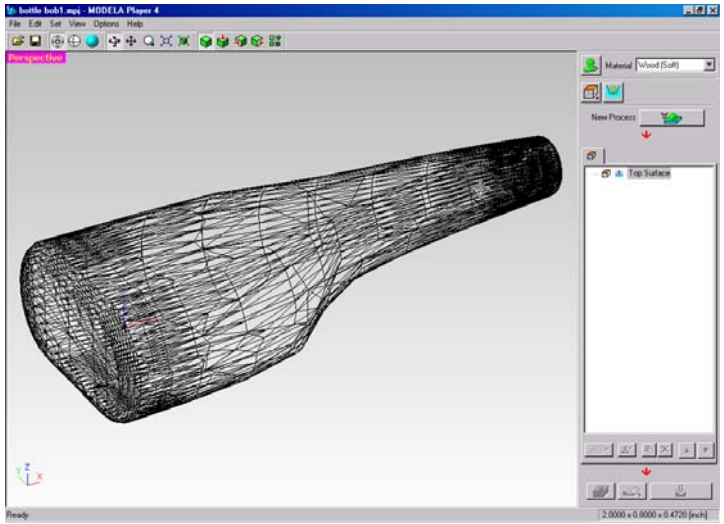
If you have a larger wax block this is where you change the size of the model to fit it. Remember that increasing the size of the model increases the machining time. These introductory exercises are designed so they can be machined in one class period.

The Model dialogue box.



With the finished size and orientation (direction) set.

Result



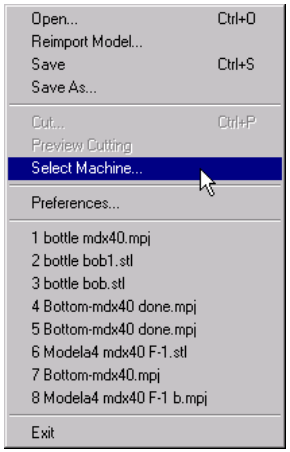
4) The Roland MDX-40 that you will be using, needs to be set as the current machine.

Always check these steps every time before you use the machine, even if you think they are set properly.

If necessary, change the machine type:

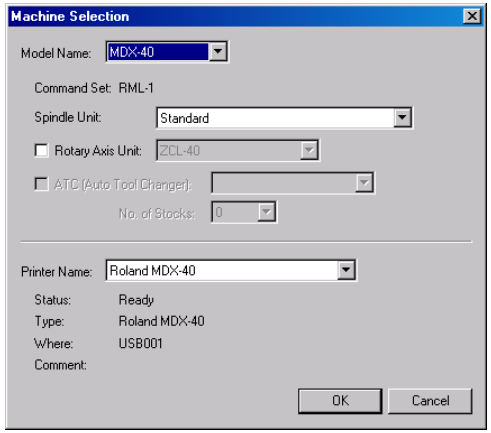
Pick the File pull down menu and

Select Machine…



Scroll through the list and **select** the MDX-40.

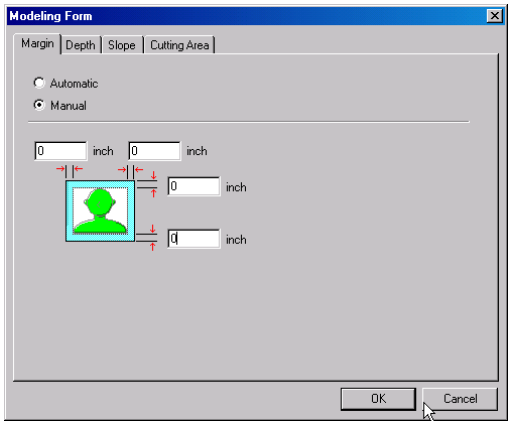
Pick OK.



5) The margin, depth, and cutting area need to be set. Set the number of surfaces to be cut. In this case we will cut one surface.

Select one surface with this button: 

Pick the Modeling Form button. 



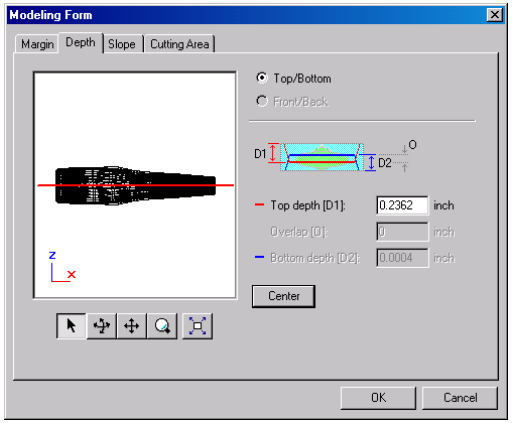
Select the Margin Tab. Set the margin to 0"

In the next exercise, we will put in a margin. You will then see the value of a margin.

Select the Depth tab.

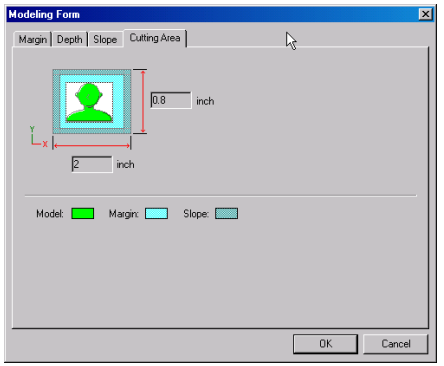
Another set up factor is maximum depth of cut. This will always have to be set when you change the part size as was done earlier in this example. Depth of cut will also needs to be considered on symmetrical parts such as a sphere. For most symmetrical parts the depth should be set at Center.

Select the Center button. 



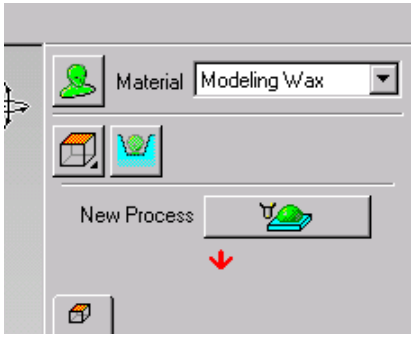
Since this is a symmetrical part, picking Center will move the cutting depth to the middle of the part. Cutting any deeper into a shape, such as a sphere, will give straight sides after the center point.

Select the Cutting Area tab. Verify the correct size of the finished model.



6) The material to be machined needs to be checked and/or set.

Note: You will be using Modeling Wax for all of the parts in this series.

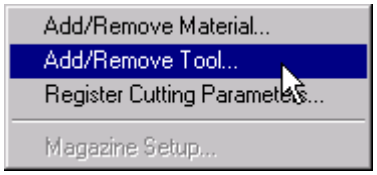


If the material is not set to Modeling Wax:

Select as shown on the Modela screen.

Modeling Wax is the displayed material in this example. The material displayed is the last one used.

7) Mill cutter type and size also needs to be added if necessary.



From the Options Menu, Select……

There are two main types of cutters or tools used in machining:

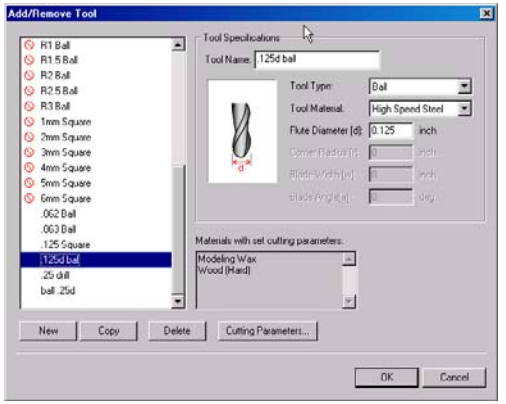
A) End mills

B) Ball nose end mills

End mills are flat on the end and are used to machine objects with flat horizontal surfaces and vertical sides.

Ball nose mills are used to machine curved complex shapes. Using a Flat end mill on these kinds of shapes will cause gouging and incorrect part size.

We use a .125” diameter Ball Nose mill for all of the Rhino parts in this set of lessons.



Select the .125d ball mill as displayed……

Cutter diameter is determined by various factors:

A) Complexity of the part. Complex parts with lots of complex shapes need smaller bits to machine the parts accurately.

B) You want to use the largest mill tool possible to use the least amount of time machining.

C) The mills tools you have and the sizes the machine you have will accept.

For the bottle and all of the other parts in this series, we will be using a .125” (1/8”) ball mill.

.

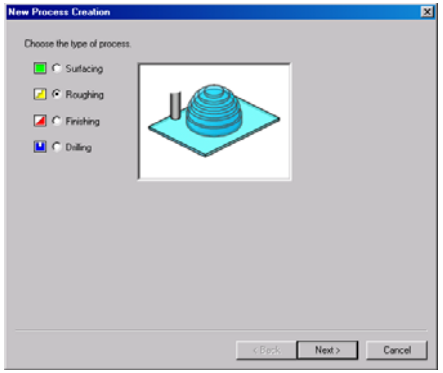
8) We will cut the part in a two-step process. The processes are often called toolpaths. **Roughing** takes away the bulk of the material and **Finishing** which gives the part a finished size and creates a smooth finish. There are additional toolpaths available, but we will concentrate on Roughing and Finishing in this series.

Roughing must be cut first followed by Finishing. Finishing without a roughing cut first can **damage** the milling machine.

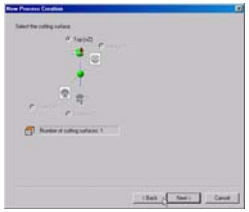
To create the Roughing tool path:

Select the New process Button: 

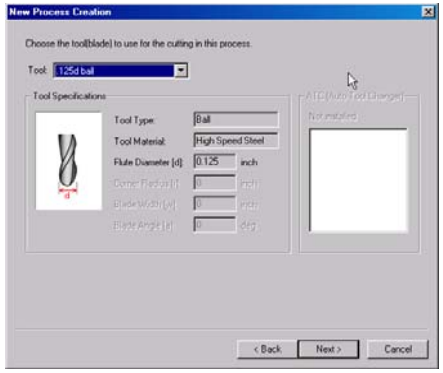
Select Roughing:



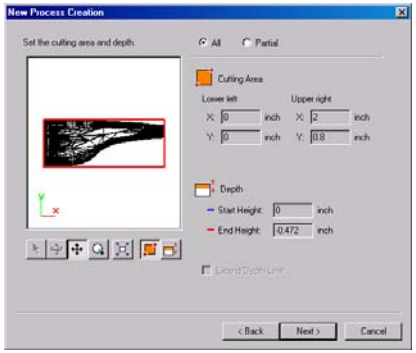
Because we have selected one side cutting, the Top is selected.

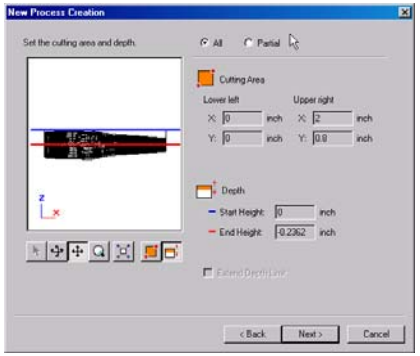


Select the .125d ball mill ……



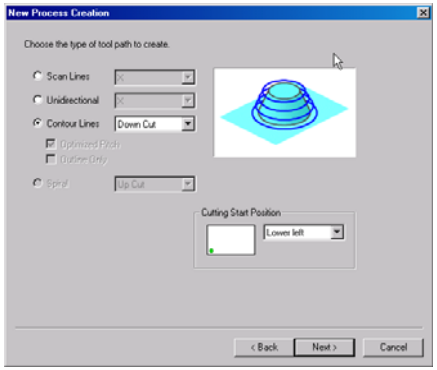
Check the cutting area and depth ……
The ‘All’ buttons should display the correct settings.





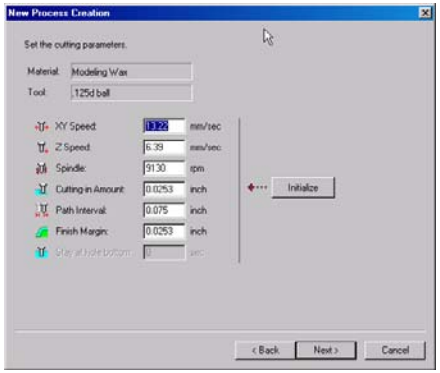
Roughing should be set to “Contour Lines Down Cut”. Contouring is a very efficient method of removing large amounts of material.

It cuts out the object in elevations.



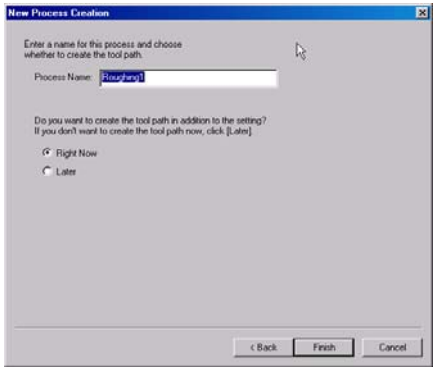
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

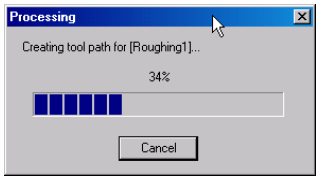


Process Name is listed as **Roughing 1**. It is a good idea to generate the tool path “Right Now” because you’ll be able to see the results before sending the tool path to the machine.

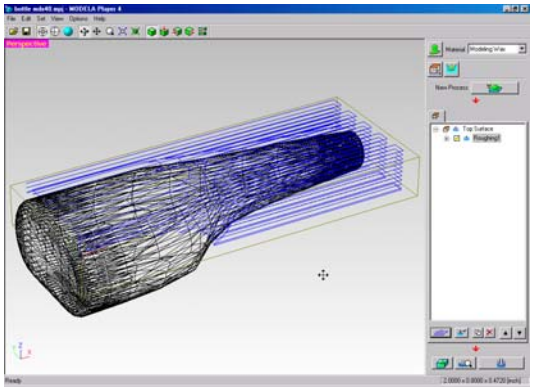
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.

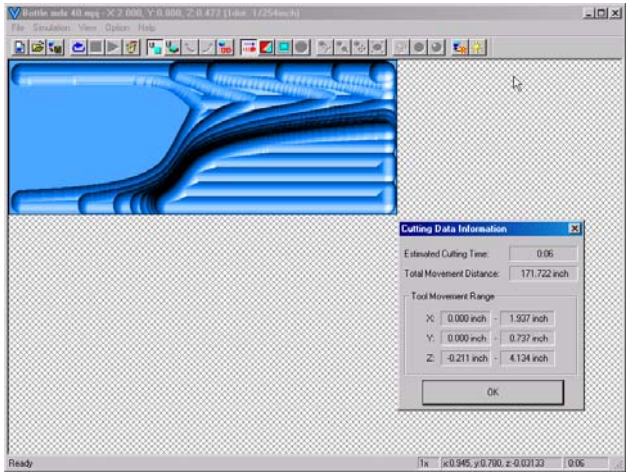


Roughing Tool Path.



9) In Modela Player 4 there is a built-in tool path simulator. It provides an accurate representation of the tool path being cut and a time estimate for its completion.

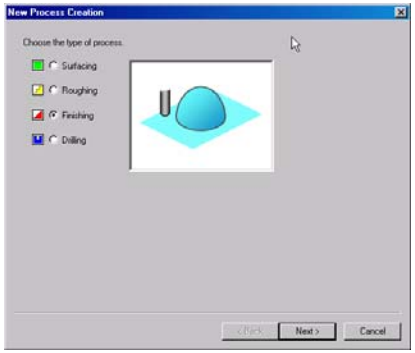
After the tool path is processed, **press the Preview Cutting button**. 



Virtual Modela automatically starts which displays the simulation and shows the cutting time.

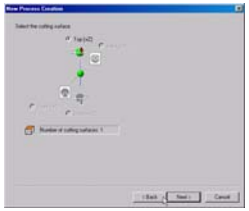
To create the Finishing tool path:

Select the New process Button: 

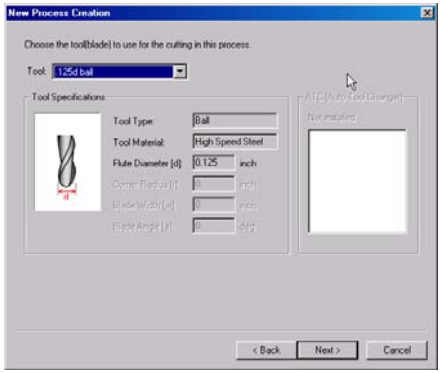


Select Finishing:

Because we have selected One-Side cutting, the Top is selected.

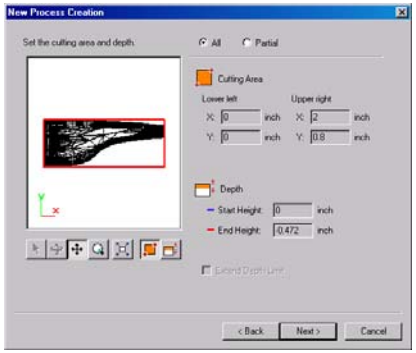


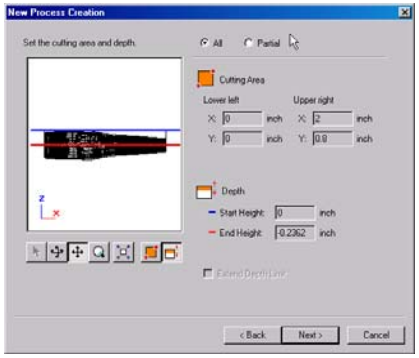
Select the .125d ball mill ……



Check the cutting area and depth …….

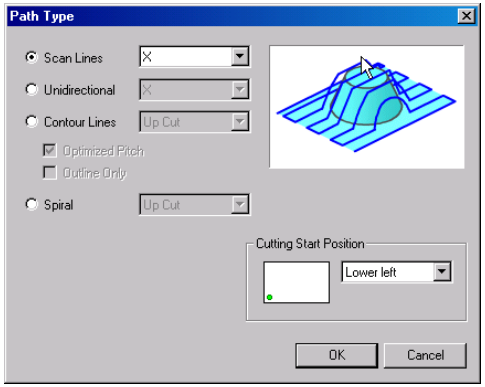
The ‘All’ buttons should display the correct settings.





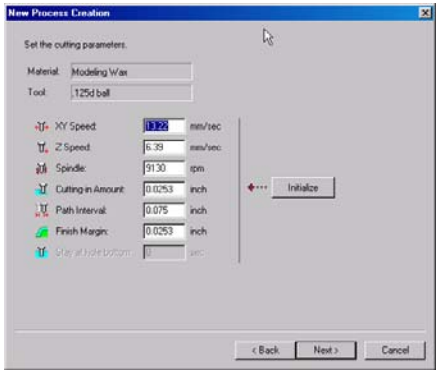
The **Finish** cutting direction can be set to cut in the X, Y, or both directions. These are lines cut along the chosen axis. Cutting both X and Y takes twice the time but gives a better finish.

Set the Finish cutting direction to Scan Lines X direction.



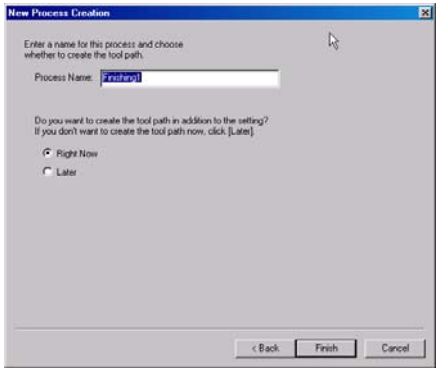
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

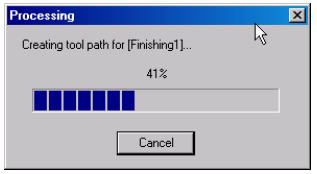


Process Name is listed as **Finishing 1**. Generate the tool path **"Right Now"** so you can see the results before sending the tool path to the machine for cutting.

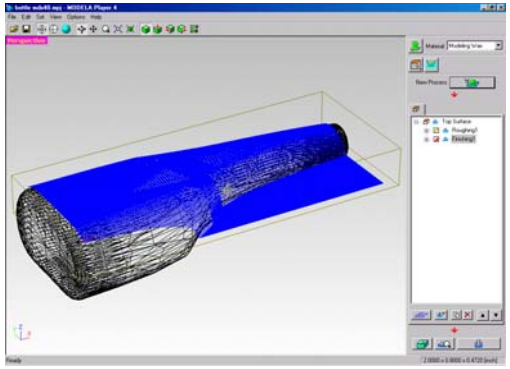
The **"Later"** option: Depending on the complexity of the part and the speed of your computer, tool paths can take extra time to generate. Use 'later' if time is tight.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.

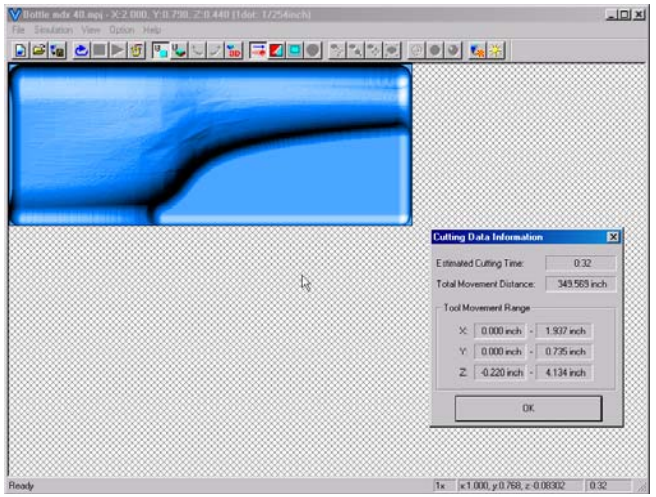


Finishing Tool Path.



10) In Modela Player 4 there is a built-in tool path simulator. It provides an accurate representation of the tool path being cut and a time estimate for its completion.

After the tool path is processed, **press the Preview Cutting button**. 



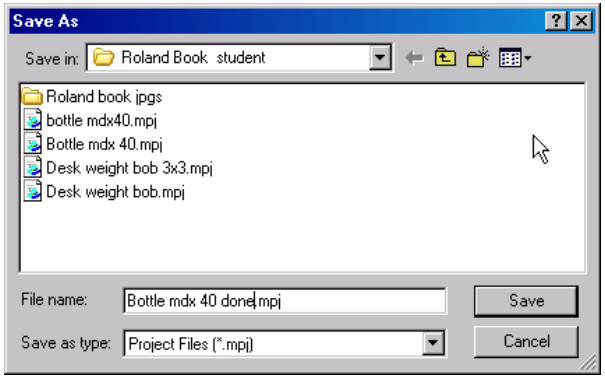
Virtual Modela automatically starts which displays the simulation and shows the cutting time.

11) To save the part as a Modela file:

Pick the File pull down menu and Save.

The file name is the same as the one that was originally imported. Only the file extension is changed to .mdj

This is often a good time to make a second save too. Simply change part of the file-name to create a second file.



Pick Save As. **Type in the name** . 'Bottle mdx 40 done'
or another file name of your choice.

Result: There are now two files. In case one is damaged, there is a backup.

NOTE: If you do not have at least 45 minutes in the class period, do not continue to the milling steps. The average machining time for the bottle is 40 minutes. It is advantageous to complete both toolpaths without turning off the milling machine.

Step 14 explains how to exit Modela Player 4 if you do not have enough time at this point. There are instructions, at the end of the lesson, on how to open a saved Modela file.

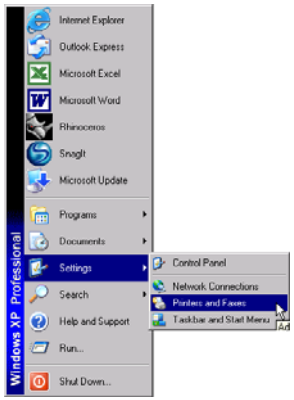
12) To machine the part on the MDX 40:
Highlight the Tool Path to be machined.

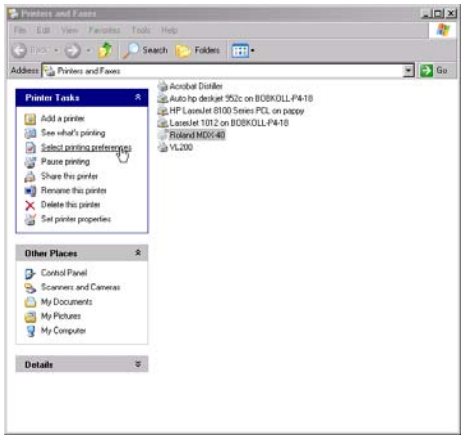
In this case, **Highlight both tool paths**

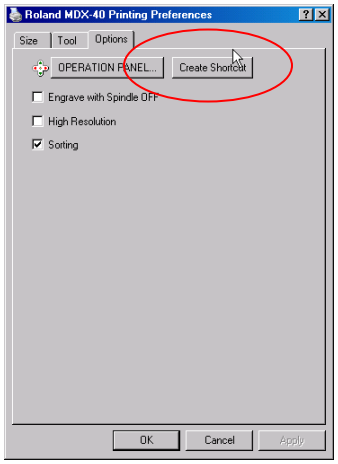
Select the Cutting/No Cutting button: 

13) MDX-40 control panel:
Place a shortcut to the MDX 40 control panel on your desktop. This gives you access to setting the cutter X, Y, Z origin.

To create the shortcut, Select Create Shortcut in the printing preferences for the Roland mdx40:



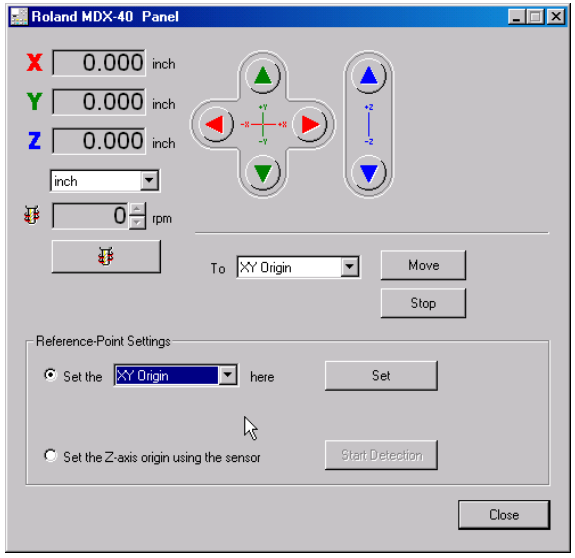




Using the mouse and the blue, red and green arrows on the control panel, move the cutter to the proper location on the wax block.

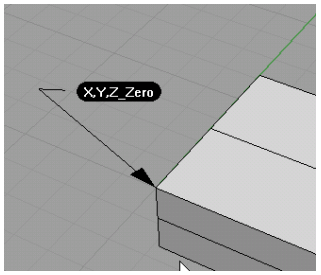
Set the Z origin, Set the Y origin, Set the X origin.

The MDX-40 ‘remembers’ the origin after it is set. If you have to restart your cutting it is easy to move the cutter to this location for re-starting if necessary.



One method to 'accurately' Set the X,Y,Z Origin at the corner of the wax stock block:

Set the X,Y,Z origin so it is at the left hand corner of the model……..

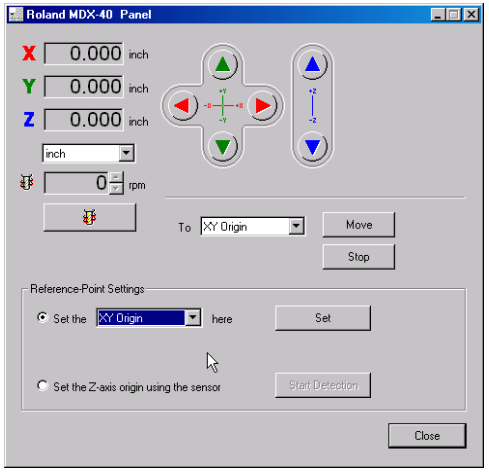


1 Set the Z origin at the top surface of the model by moving the cutter (with the mouse and the arrows in the control panel) so it touches the top of the block. Use a piece of notebook paper, approximately .003" thick. Move the cutter down until there is friction between the cutter and the block. The cutter at this point is .003 above the block. **Set Z origin here.** (temporary Z)

2 Set the X origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the left side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the X Origin here.** (temporary X)

3 Set the Y origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the front side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the Y Origin here.** (temporary Y)

IMPORTANT: The XYZ readouts read 0.000 inches.
You need to change the position of the tool so the XY center of the tool is exactly at the corner of the block. The XY is currently one-half the tool diameter (.0625) away from the correct origin. Also, remember that our tool is .003 above the block.



With the mouse and the arrows in the control panel:

z… Move the cutter so the Z reads: Z -.003" **Set Z origin here**
x… Move the cutter so the X reads: X .0625" **Set X origin here**
y… Move the cutter so the Y reads: Y .0625" **Set Y origin here**

Select the Cut button: Sends the tool path files to the MDX-40 and begins to cut the part.

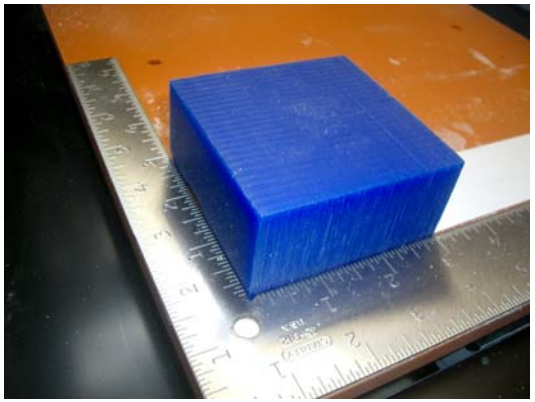
Remember: Highlight the Roughing and Finishing tool paths and select the Cut/NoCut button so the desired tool path file will be sent to the MDX40. This button controls which tool path is or is cut by the machine when the Cut button is pressed.

As your machine begins cutting:

To Pause: Press the View button on the front of the MDX 40. The machine will move the cutter to a safe Z height and move the XY platform for a closer view.
>>>>**Caution:** Do not open the cover until the machine stops moving<<<<
To Resume: The machine will begin cutting at the exact location where it was when the View button was pressed.

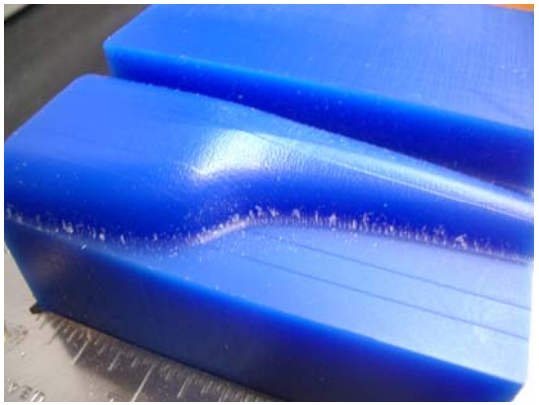
Do not leave the milling machine unattended when it is running!!!

Follow all standard safety procedures while operating the mill.



Wax block Ready to Cut

Machined Bottle



14) To exit the Modela Player 4:

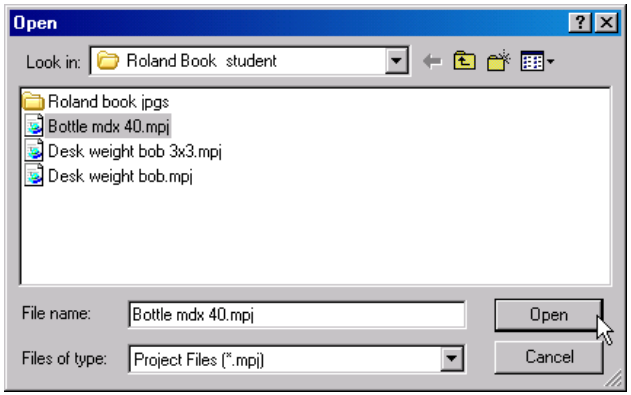
Pick the X in the top right corner of the Modela screen. When the Save dialogue box displays, select Yes to save the current file. The file retains the original file name except the extension changes to .mpj

To open a saved Modela File:

With Modela Player 4 started:

Select the File pull down menu.

Select the File…… Bottle mdx 40.mpj

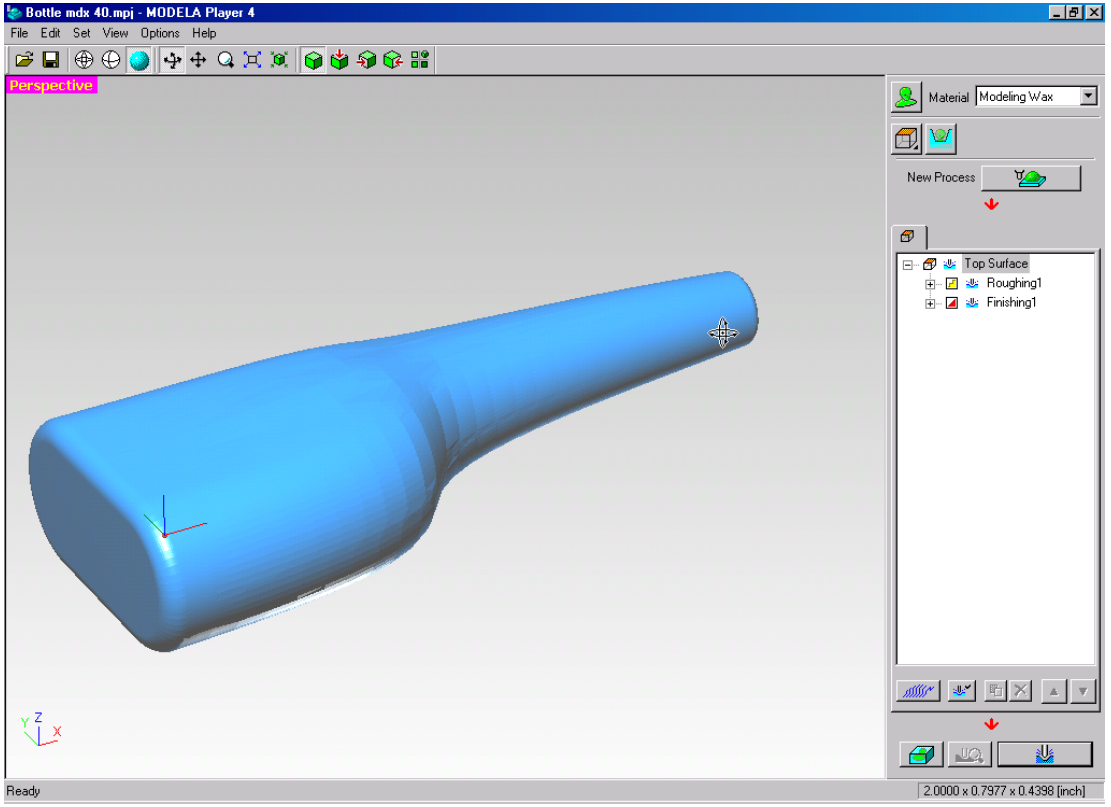


To easily recognize a Modela 4 file, look for the following icon In front of the file name:

Double clicking on this file name will also start Modela Player 4 and open the file.

Pick Open.

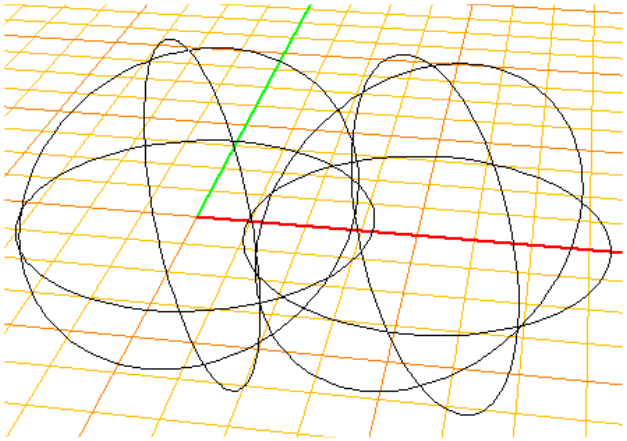
The file will open in Modela Player 4.

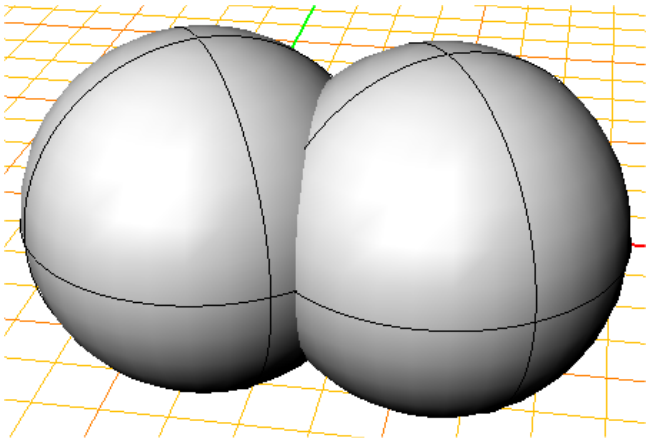


4 Desk Weight - Rhinoceros

The focus of this lesson will be creating models using solids as compared to the bottle which was designed with surfaces. Unlike surfaces, which are the outside skin of the model, a solids model is solid material completely through the object. Solids can be made by extruding curves or surfaces and by using primitives. Spheres, Cones, Boxes and Ellipsoids are examples of solid primitives. Primitives are edited with Boolean Commands. The main three are Union, Intersection and Difference. The following displays an example of each.

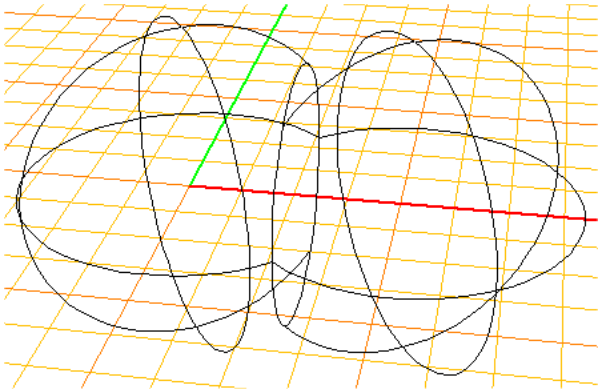
Original Object; two spheres





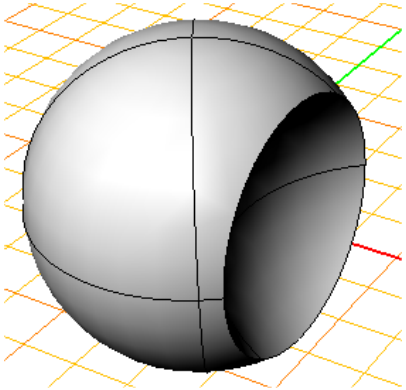
Union

Note: The shaded object would look the same as the original.



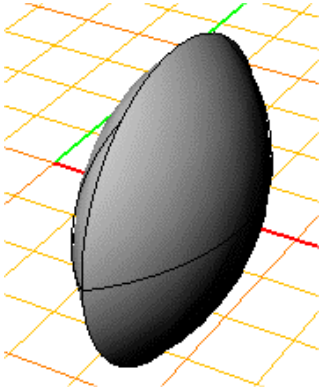
Difference

Note: The right sphere was subtracted from the first.
(the left object was selected first in the command)

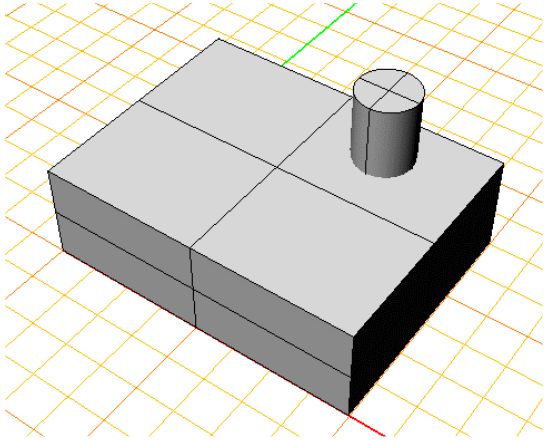


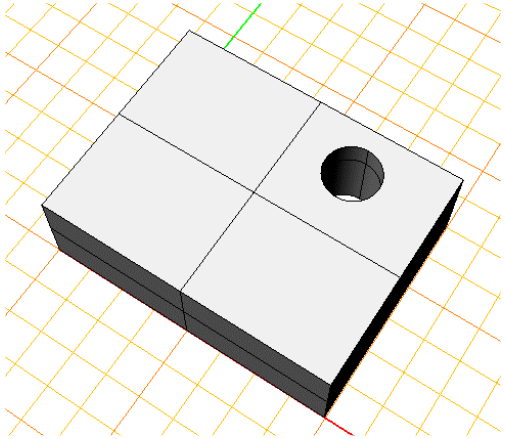
Intersection

Note: The intersection is the portion of the two objects that existed in the same space.



A common operation is to put a hole in a solid. Displayed is a box with a cylinder through it. To create the hole, the cylinder is subtracted from the box using the Difference command.



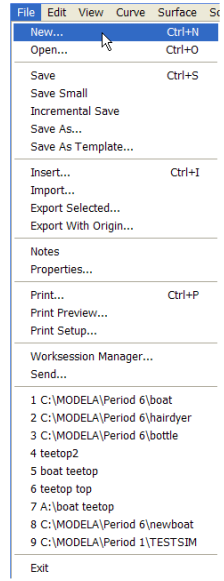


Note:
This lesson will be the last in Rhino to explain in detail how to open the template file, change layers, saving the model and creating a stereolithography file. Be sure you know how to accomplish these actions after completing this lesson.

Creating the Desk Weight

1) Start up Rhino.

The next few steps call up the template file.



2) Select the File pull down menu and **New…**

or

The **New** icon from the toolbar

The Open Template File dialogue box appears with Template Files as the “Look in” directory. This is the standard template file location.

Your template file is in your period directory.

3) Select the arrow across from Template Files.

4) From the listing, **pick C:**.

5) Double click the Modela directory.

6) Double click your class period directory.

7) Select your template file:

ModelaRhino_ _ _

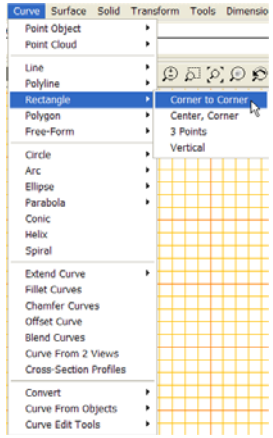
The spaces are for your initials.

i.e. MODELARHINOSIM

Pick Open.

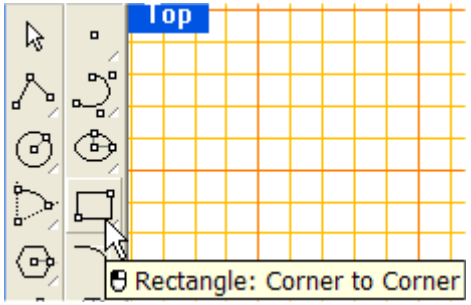
Rhino is now using your template file.

8) Start the **Rectangle command**.



Curve – Rectangle – Corner to Corner

or



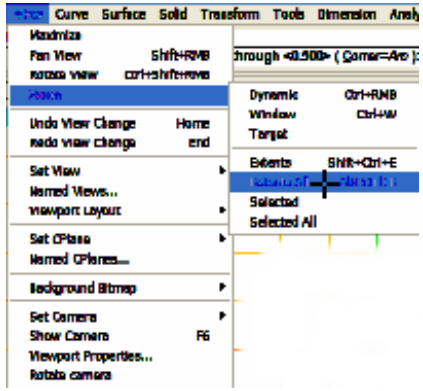
Pick the Rectangle icon from the tool bar.

In the Right viewport:

First corner of rectangle (3Point Vertical Center Rounded): **0 Enter**

Other corner or length: **1,0.4 Enter**

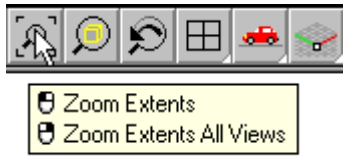
To zoom into the rectangle, in all views, select **Zoom Extents All**.



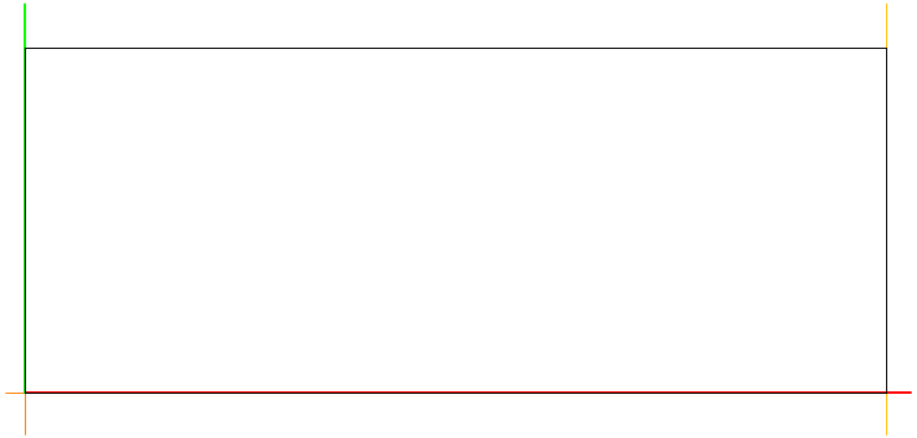
View – Zoom – Extents All

or

Right pick the Zoom Extents All icon from the top tool bar.

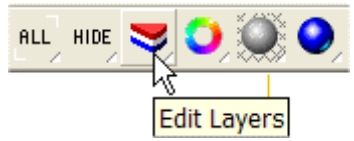


Result



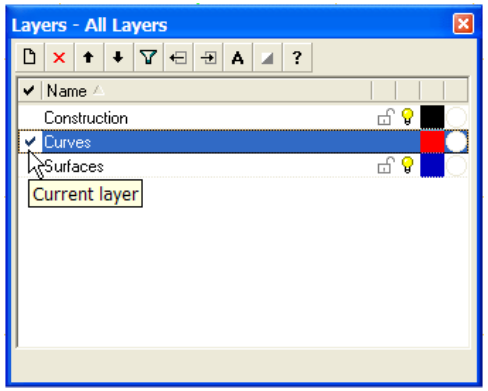
9) To see the layer:

Pick the Layer Edit icon from the toolbar.



Set the **Curves** layer as current.

Close the Layer dialogue box.

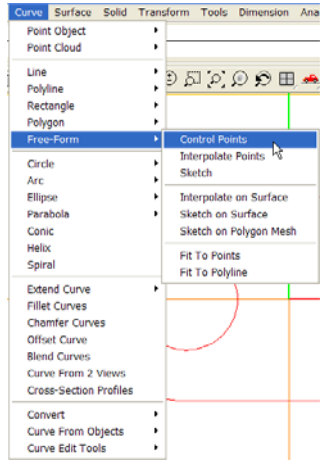


10) This step creates the outline for the end of the desk weight.

Set the Osnap box so only Near and End are on.

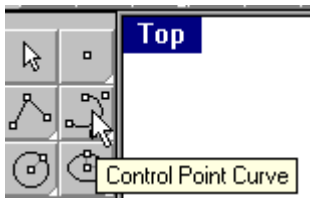
Start the Control Point Curve command:

Curve – Free-form - Control points

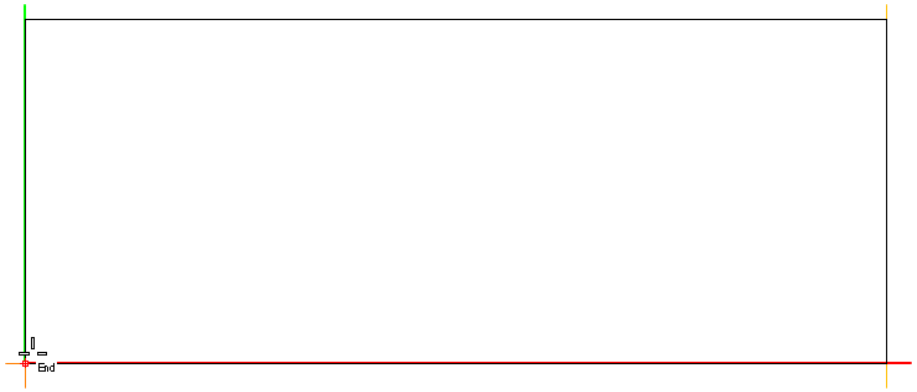


or

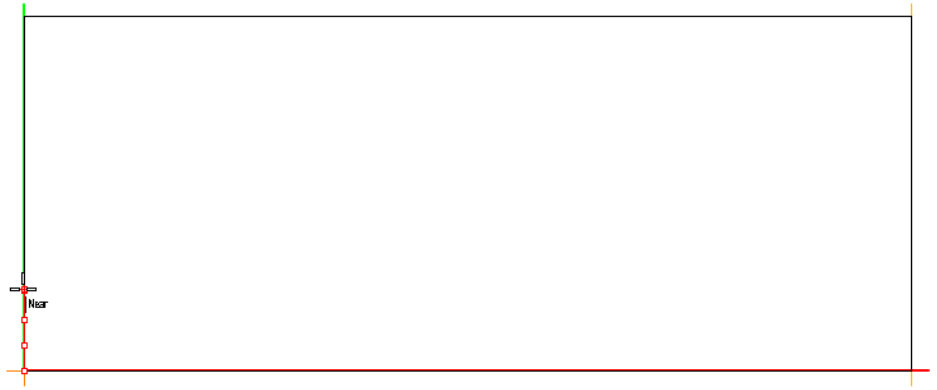
Pick the Control Point Curve command from the Draw toolbar from the left side of the graphics screen.



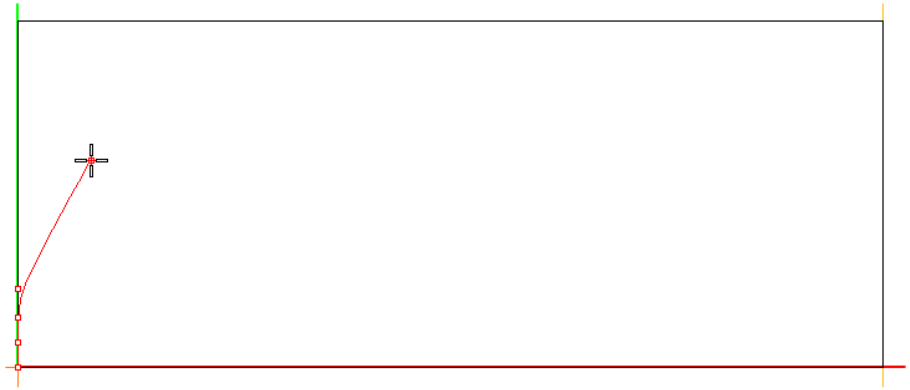
Start the curve, **by picking the lower left corner** of the rectangle.



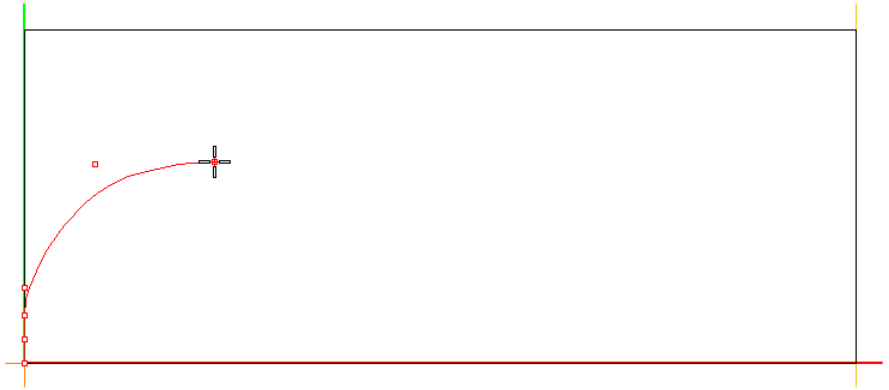
Make three more picks per the illustration on the vertical left side of the rectangle.



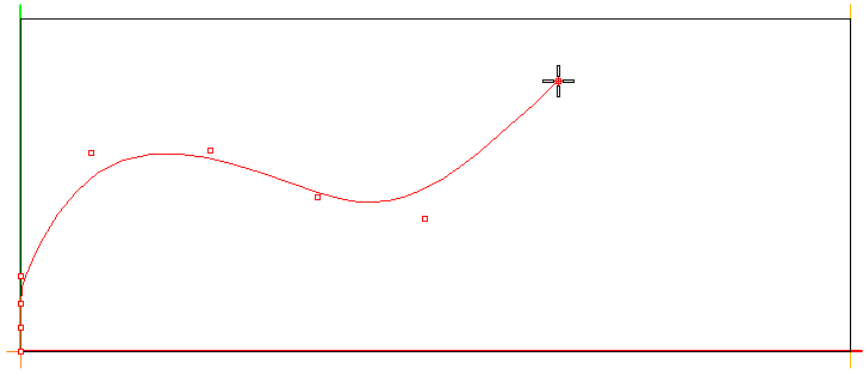
Pick as displayed.



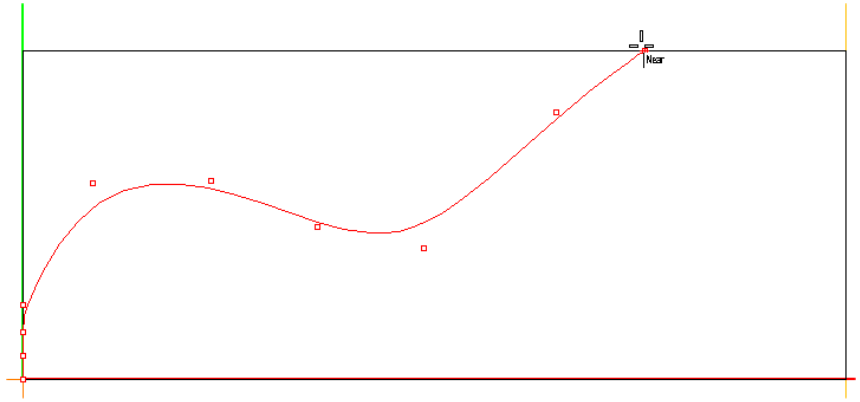
Pick as displayed.



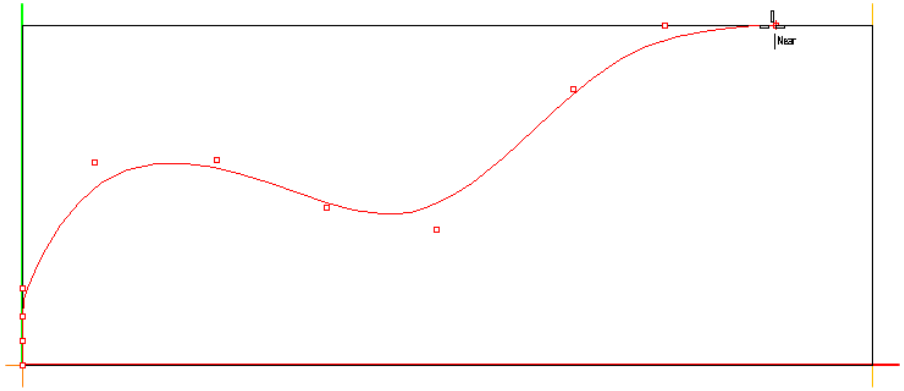
Make three picks as displayed.



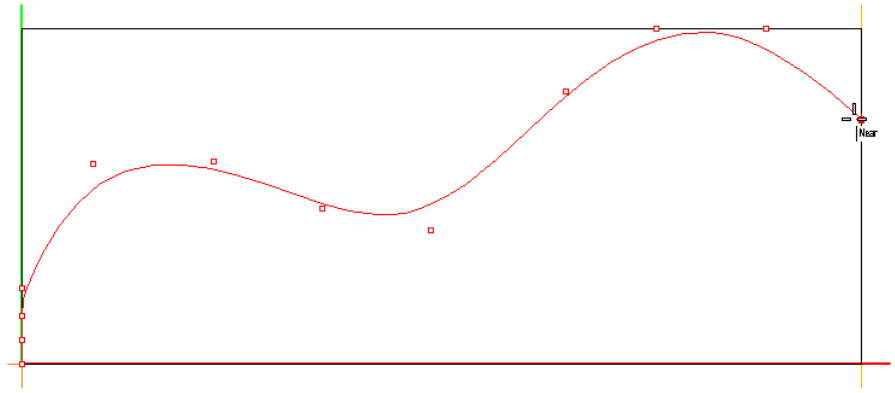
Pick on the top of the rectangle.



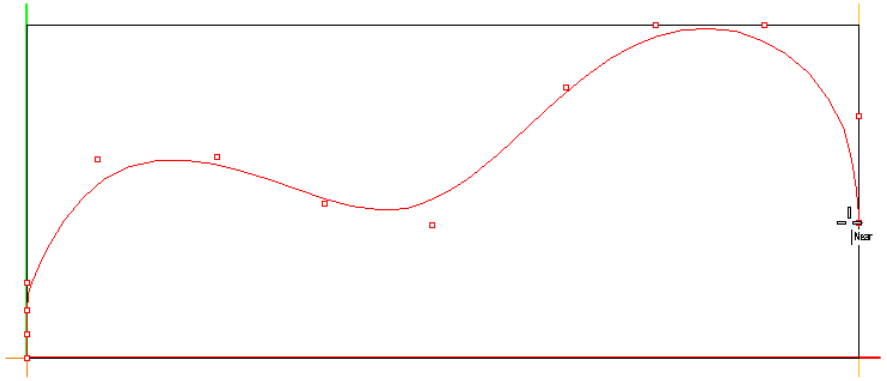
Pick, again, on top **of the rectangle as displayed**.



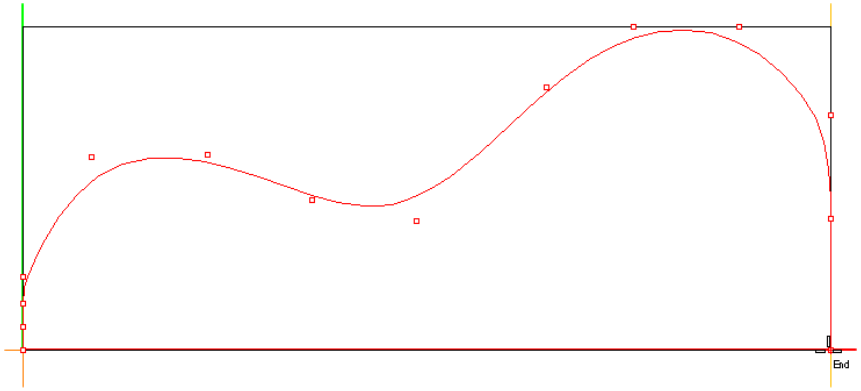
Pick on the right side of the rectangle.



Pick, again, **on the right side** of the rectangle.



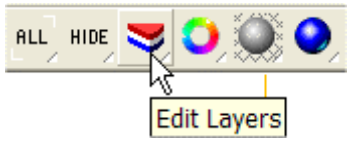
Pick on the lower right corner of the rectangle.

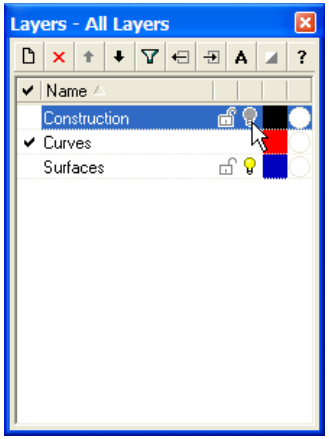


Press Enter to end the command.

Note: If your curve does not look close to the one displayed, delete and draw the curve again.

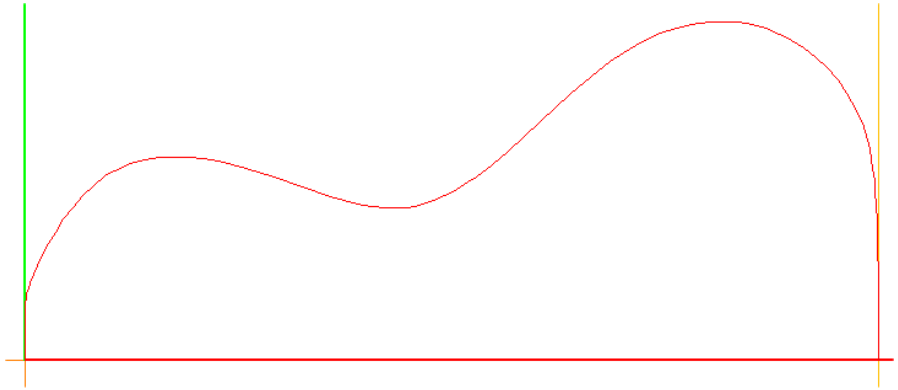
11) Open the Layer command and freeze the Construction layer.





Close the Layer dialogue box after setting the layer.

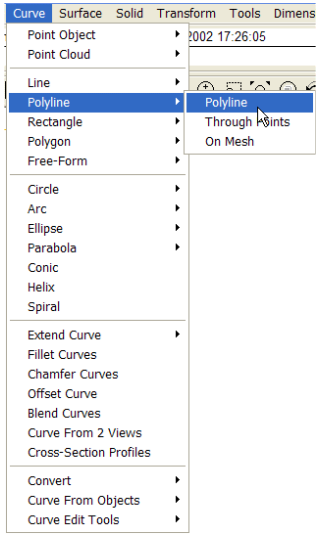
Result



12) Even though it looks like there is a line along the bottom of the curve, there is not. What you are seeing is the X axis line. The curve needs to be closed to extrude properly.

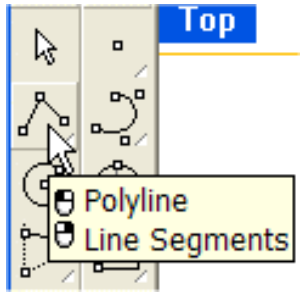
Start the polyline command:

Curve – Polyline - Polyline

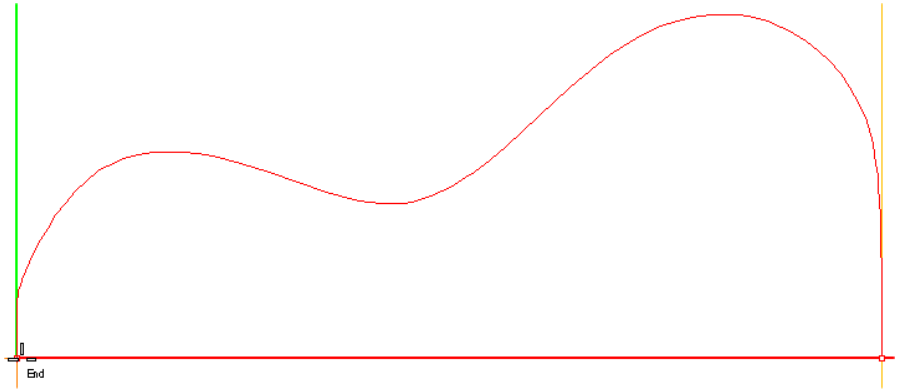


or

Pick the Polyline command from the toolbar.



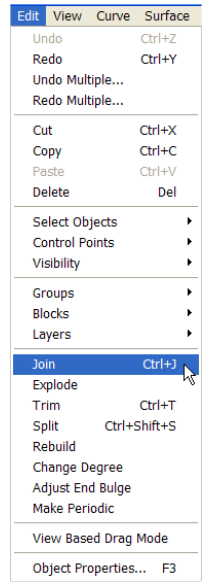
With End Osnap on, **pick each end of the bottom of the curve**.



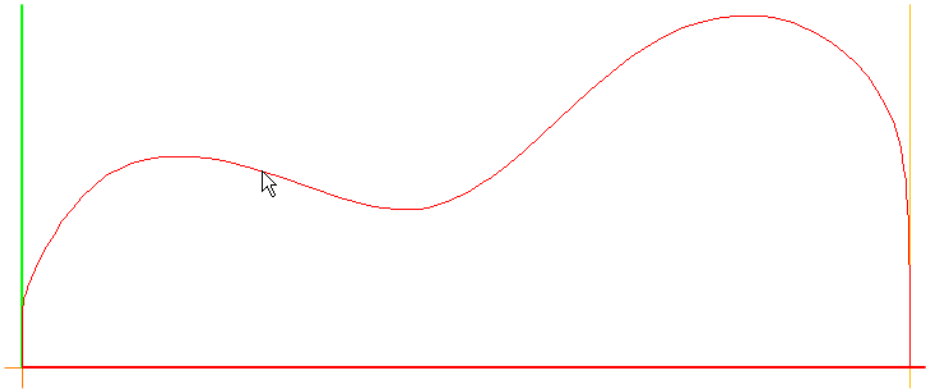
Press Enter to end the command.

13) The curve and the bottom line need to be joined.

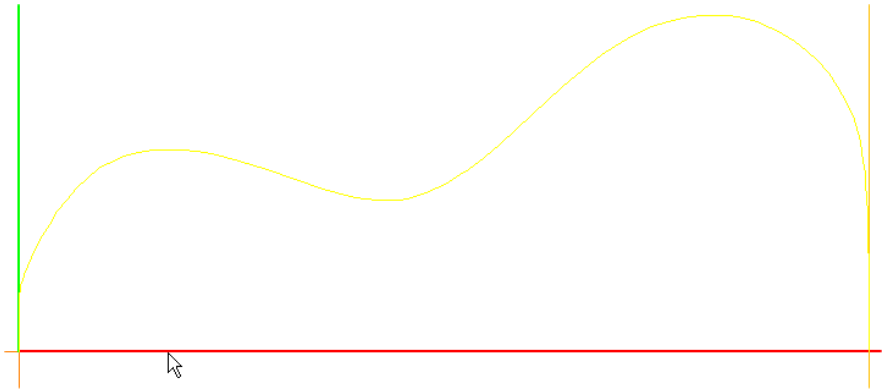
Edit – Join



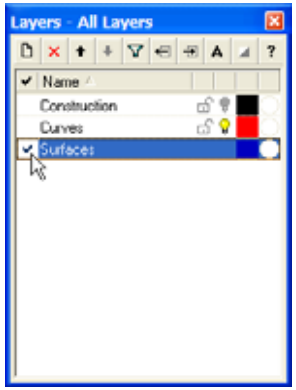
Pick the Curve.



Pick the bottom line.

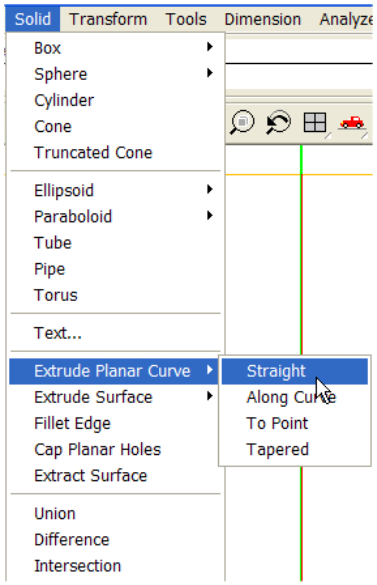


14) **Set the layer to Surfaces**.



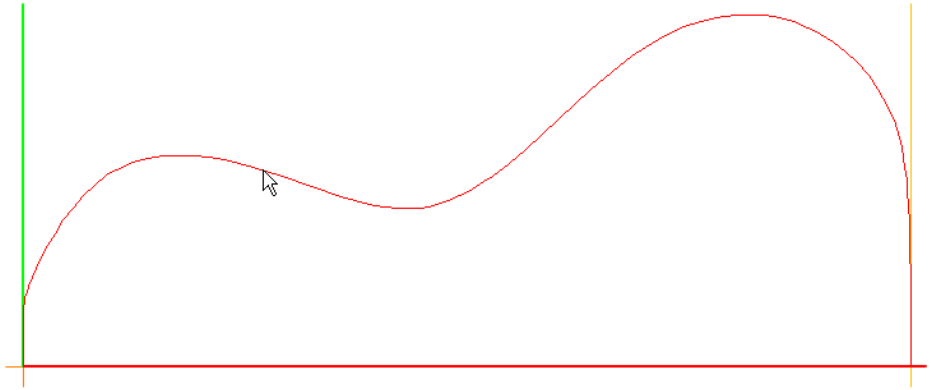
Close the Layer dialogue box after setting the layer.

15) To create the solid for the base of the desk weight:



Solid – Extrude Planar Curve - Straight

Select curves to extrude (DeleteInput=No): **Pick the Curve**

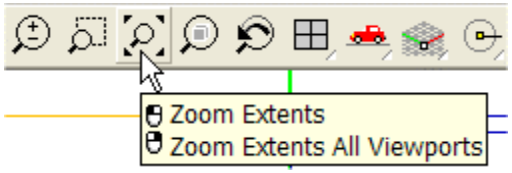


Select curves to extrude. Press Enter when done (DeleteInput=No): **Enter**

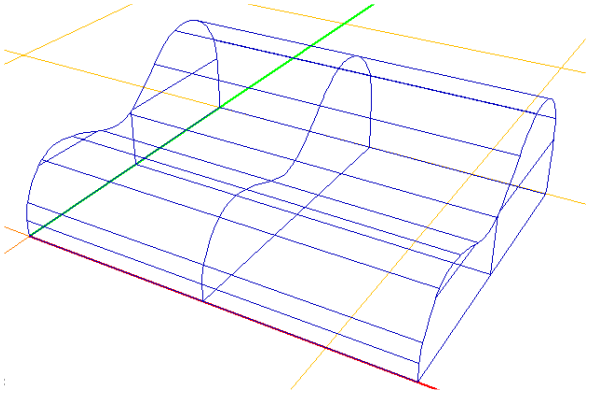
Extrusion distance (Direction BothSides=No Cap=Yes Mode=Straight DeleteInput=No): **1.25 Enter**

To see the results:

Right click on the Zoom Extents icon.

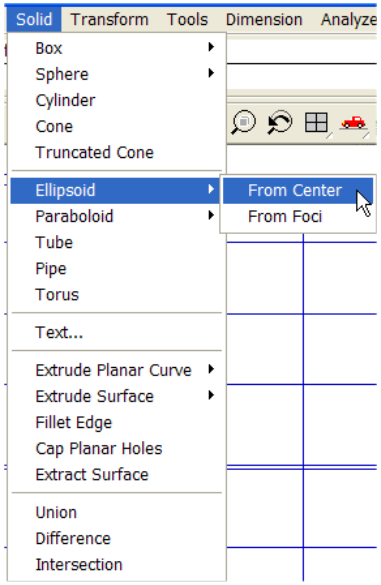


Result



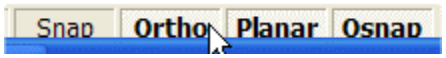
16) Adding the ellipsoid in the center of the desk weight is the next step.

Solid – Ellipsoid – From Center



In the Top viewport:

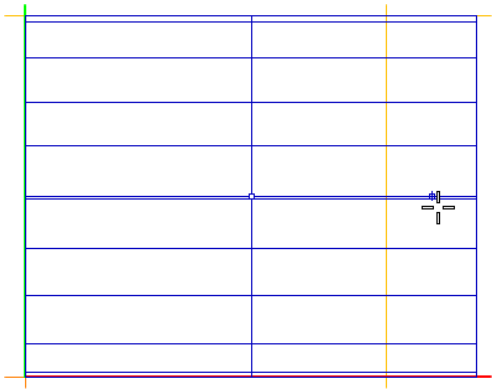
With **Ortho on**:



Ellipsoid center (Corner Diameter FromFoci AroundCurve): **.625,.5,.15 Enter**

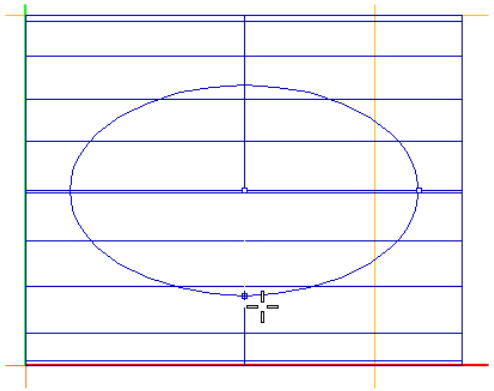
End of first axis (Corner): **.5 Enter**

End of first axis (Corner): **Pick to the right as displayed**



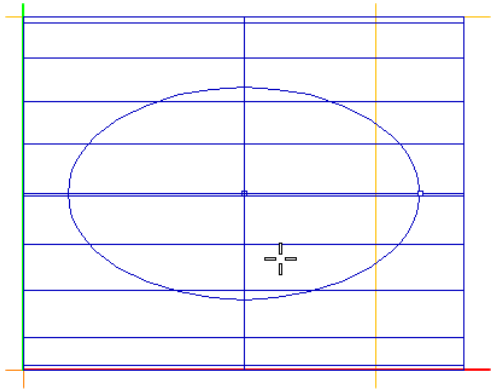
End of second axis: **.3 Enter**

End of second axis: **Pick as displayed**

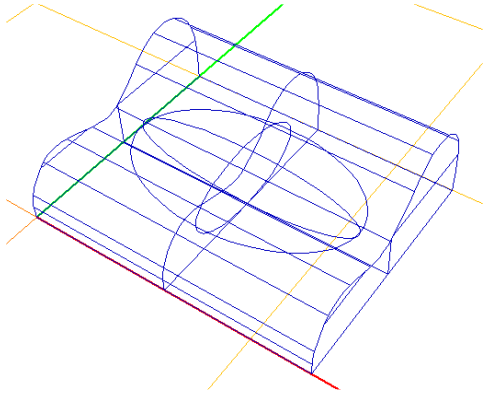


End of third axis: **.15 Enter**

End of third axis: **Pick as displayed**



Result



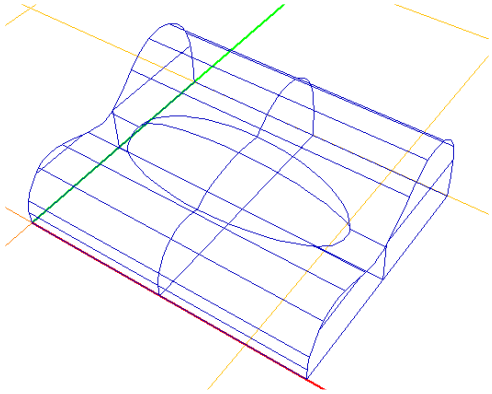
17) The two solids are to be joined into one with the Union command.

Solid - Union

Select surfaces or polysurfaces to union: **Pick one of the objects**

Select surfaces or polysurfaces to union. Press Enter when done: **Pick the other object and press Enter**

Result



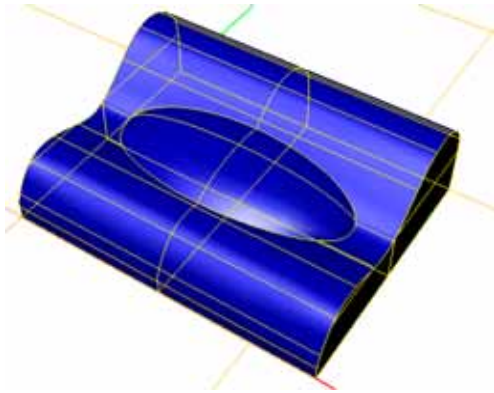
18) To see the completed shaded desk weight:

With Perspective active:

Pick the Shade icon on the top tool bar.

(it is the gray sphere)

Result



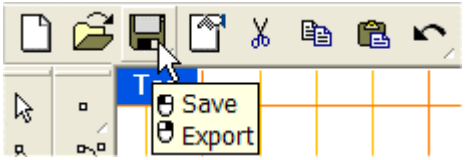
19) To save the desk weight:

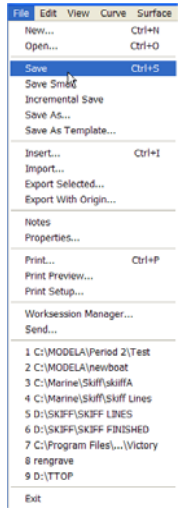
1) To save the desk weight:

File – Save

or

Pick the Save icon from the top tool bar.

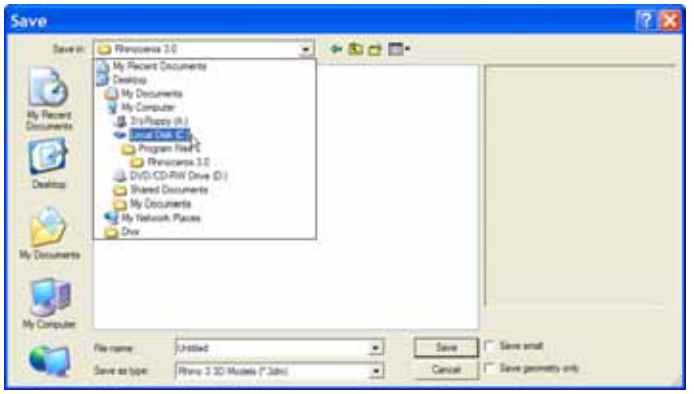




In the dialogue box, **pick the Arrow** across from the Save in box.

Click on C:

This should show in the Save in box.



Double click on MODELA

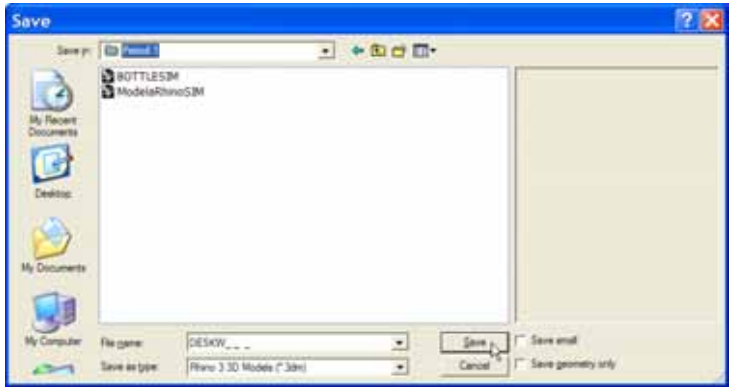
This should show in the Save in box.

Double click on the period you are in.

This should show in the Save in box.

Type, in the File name box:

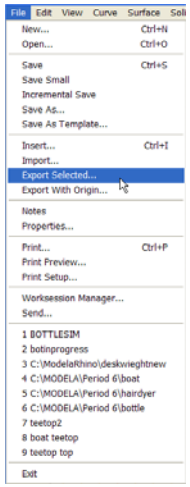
DESKW_ _ _ (the spaces are for your initials) i.e.: DESKWSIM
Pick Save to finish the command.



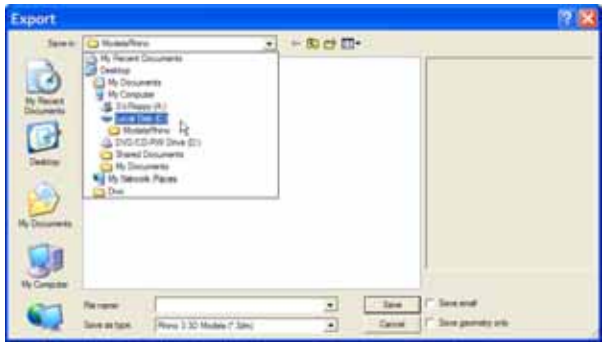
20) To create a file to send to the machining software (Modela Player):

From a command prompt, **pick the bottle** so that it is highlighted.

File – Export Selected…



In the dialogue box, **pick the Arrow** across from the Save in box and **select C:**.



Double click on MODELA

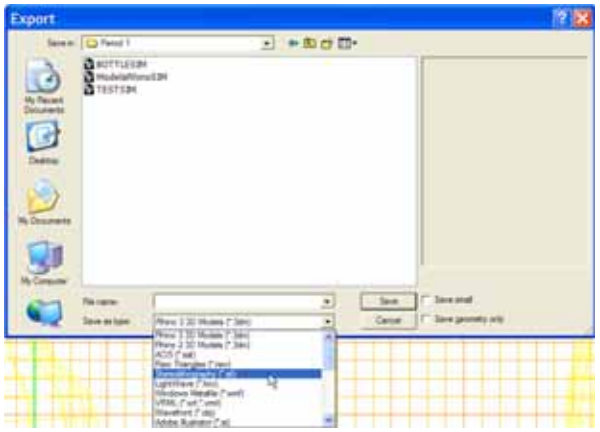
This should show in the Save in box.

Double click on the period you are in.

This should show in the Save in box.

At the bottom of the dialogue box, pick on the Save as type: bar and select Stereolithography *.stl

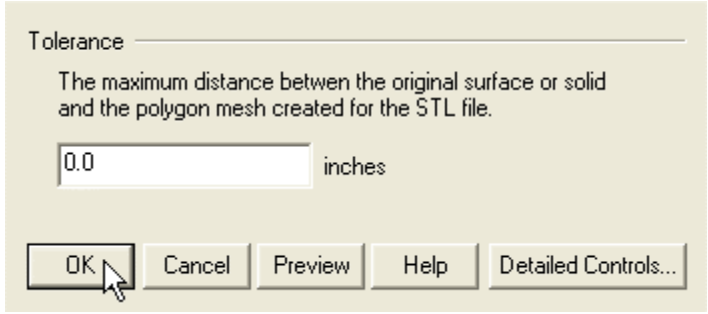
A stereolithography file is an export file type used for solid objects. These are models that are completely enclosed.



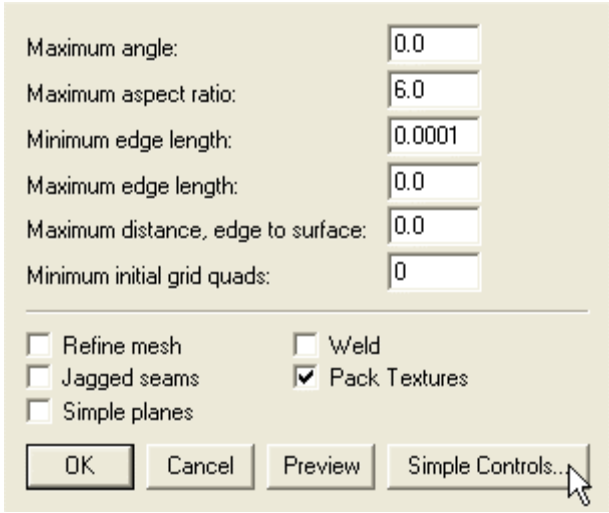
Type, in the File name box:

DESKW_ _ _ (the spaces are for your initials) ie: DESKWSIM
Pick Save to finish the command.

The STL Tolerance dialogue box should be **set as follows, then pick OK**.



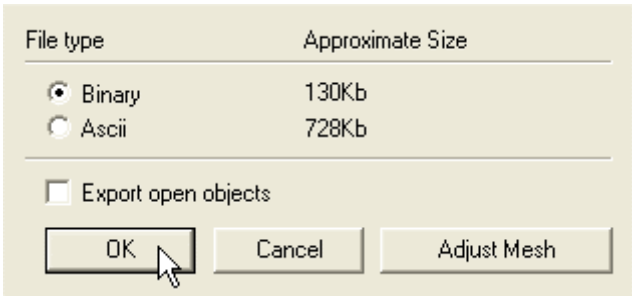
If the Detailed Tolerance dialogue box is displayed, **set it as displayed and pick Simple Controls button**.



You will then need to **pick OK** in the Simple Tolerance dialogue box

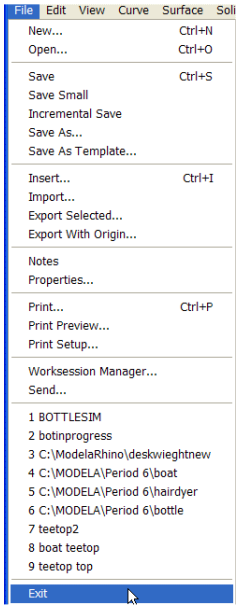
With the **Export open objects unchecked, pick OK.**

Depending on the speed of the computer it could take upwards to two minutes to complete the export process.



26) Close Rhino.

File - Exit



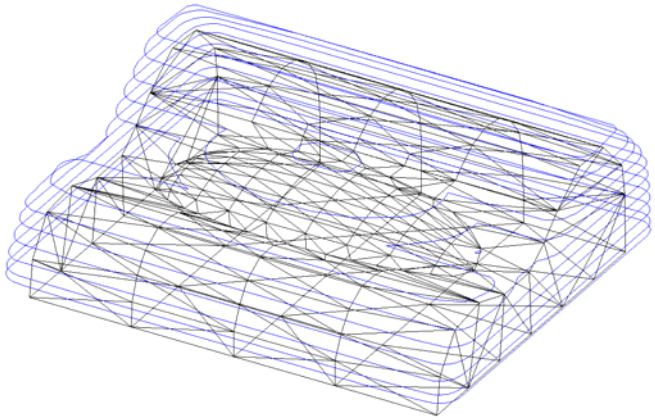
In the next lesson you will create the toolpaths to machine the desk weight.

5

Modela Player 4

MDX-40

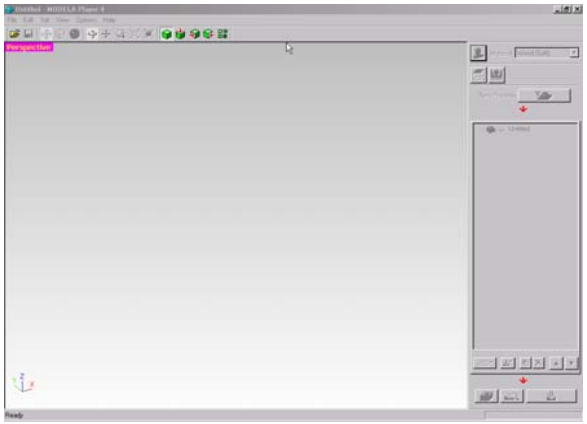
Desk Weight Modela



1) Start the Modela Player 4

Pick the Modela Player 4 icon from the desktop.

Modela Player 4 will open……..



Below is a flow chart of the process of machining the model with the Modela Player 4 and your Roland milling machine. Be sure to notice the 1st and 2nd numbering. These need to be done in order to properly machine your work and protect your milling machine. The Roughing toolpath, which cuts away the majority of the material, must be done before the Finishing toolpath.

Modela Player 4

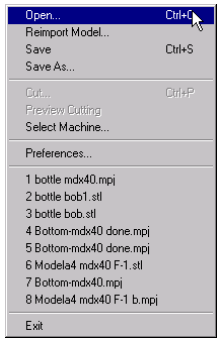
1st … Roughing tool path

2nd … Finishing tool path

Processes:	Details:
Model Button 	>Set finished size. Set the Origin.
Cutting Surfaces Button 	>Select number of surfaces to machine.
Modeling Form Button 	>Set the depth and the cutting area
New Process Button 	>Create tool paths: One Roughing & One Finishing
Preview Button 	>Create a tool path simulation & cutting time est.
Cut Button 	>Sends the file to the MDX40 and begins cutting.

2) To open the Desk Weight stl file:

File – Open –

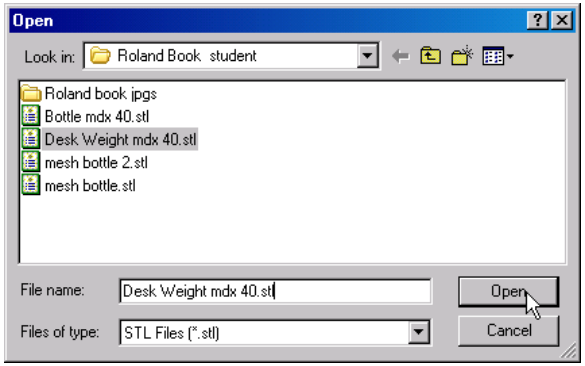


Pick File from the pull down menu in the top left of the screen and

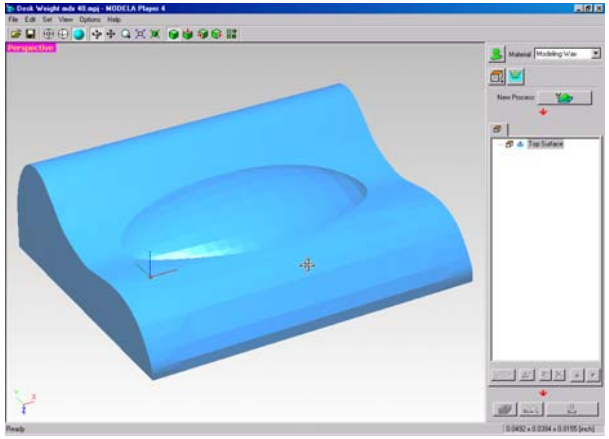
Select Open…

Select the file **Desk Weight mdx 40.stl**

and **pick Open**



Result



Imported model quality:

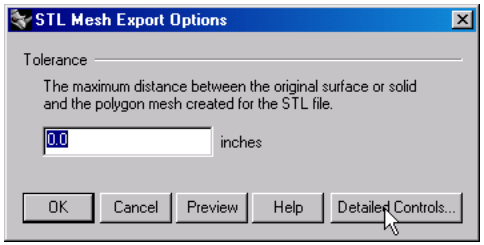
If the quality of the imported model is not fine enough for you, go back to Rhino and export the stl file again.

Open the model in Rhino.

Select the model and go to **File, Export Selected, File Type, stl**

Adjust the STL Mest Export Options as follows:

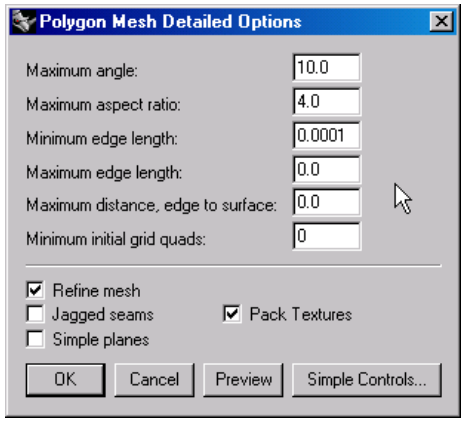
Click Detailed Controls:



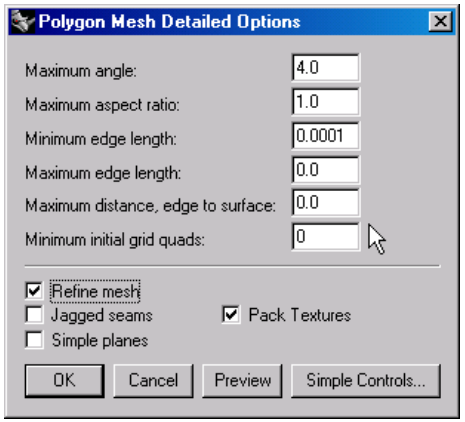
Change the settings as follows:

Export the model again as an stl file.
Experiment with the Polygon Mesh options
to get the desired results when opened in Modela 4.

Preview before selecting OK. The finer, more exact the mesh,
The larger the stl file will be.



Medium quality mesh

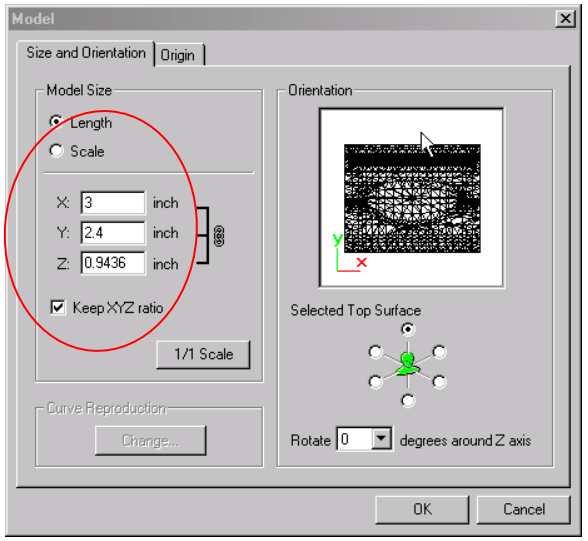


Fine, more exact mesh

3) You also have to take into consideration the orientation (direction) of the part and the final size of the part. The mill is going to cut the part from the top of the screen down. For the Desk Weight, the orientation does NOT need changing. The model was created with the side that will be machined facing up. For this reason we do not need to change the orientation of the model.

But, we will need to set the finished size of the machined part.

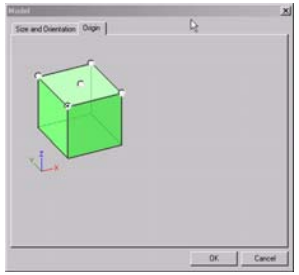
Pick the Model button.



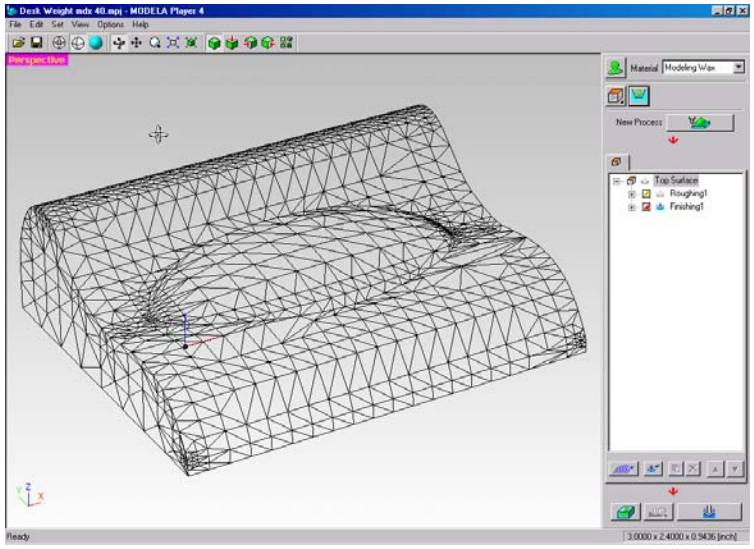
Set the finished size of the model. 3 x 2.4 x .9436, which will easily fit our wax block.

Select the "TOP Surface" radio button, as displayed because we do not need to rotate the model to orient it.

The origin button displays the xyz origin to be used for cutting the model.



Result



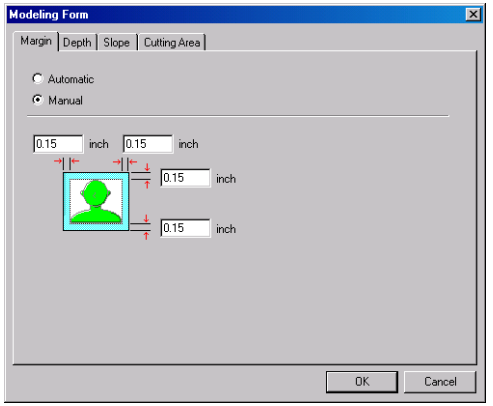
Set the number of surfaces to be cut. **Select one surface** with this button:

The Model Form Button:

4) The **Margin, depth, and cutting area** need to be set.

Select the Margin Tab.

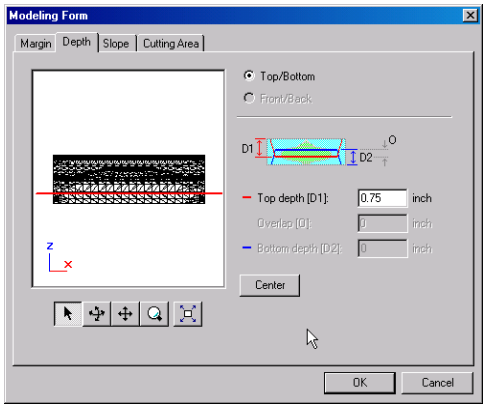
Set the margin to .15” on all edges of the model. This will cause the cutter to cut all the way to and past the edges of the block. ……..



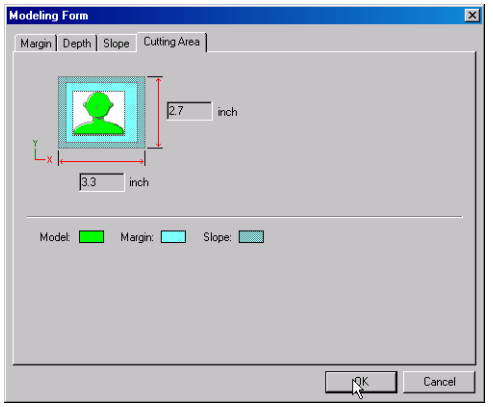
Select the Depth tab.

Set the depth to .75” This depth is chosen to cut near but not through the bottom of the model. If you go back to the Rhino Model and scale the model to its full size, you will be able to determine why .75 was chosen. In Modela 4, the depth is .946 inches. Setting the cutting depth to less than .946 will keep us from cutting into the machine base.

Note: If there is ever a situation when you have to completely cut through your model, simply attach the stock material to a sacrificial base plate that is safe to cut into.



Select the Cutting Area tab. Notice that the cutting area is .3 (.15+.15) larger than the original model. This is because of the margin we added to the model.



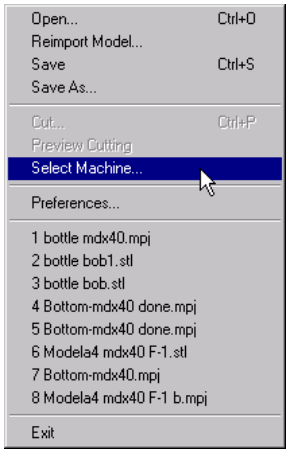
5) The Roland MDX-40 that you will be using, needs to be set as the current machine, if it is not already.

Always check these steps every time before you use the machine, even if you think they are set properly.

If necessary, change the machine type:

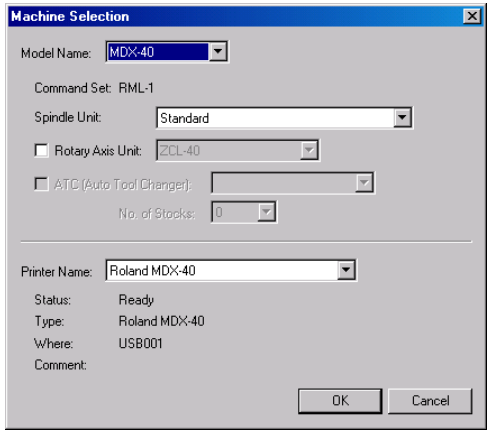
Pick the File pull down menu and

Select Machine…

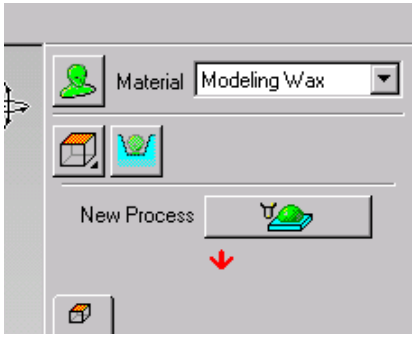


Scroll through the list and **select** the MDX-40.

Pick OK.



6) The material to be machined needs to be checked and/or set.



.

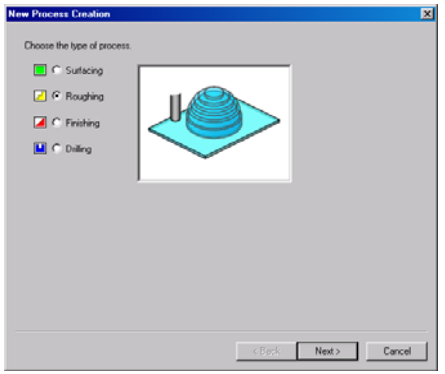
7) We will cut the part in a two step process. The processes are often called toolpaths. **Roughing** takes away the bulk of the material and **Finishing** which gives the part a finished size and creates a smooth finish. There are additional toolpaths available, but we will concentrate on Roughing and Finishing in this series.

Roughing must be cut first followed by Finishing. Finishing without a roughing cut first can **damage** the milling machine.

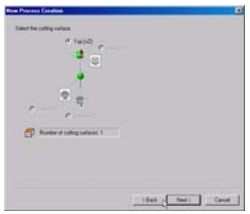
To create the Roughing tool path:

Select the New process Button: 

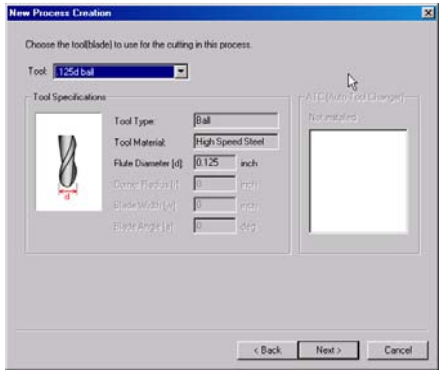
Select Roughing:



Because we have selected One side cutting, the Top is selected.



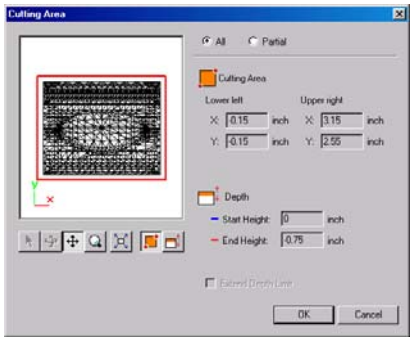
For all of the other parts in this series, we will be using a .125” (1/8”) ball mill.

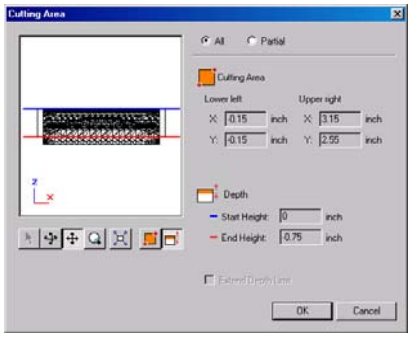


Select the .125d ball mill …….

Check the cutting area and depth …….
The ‘All’ buttons should display the correct settings.

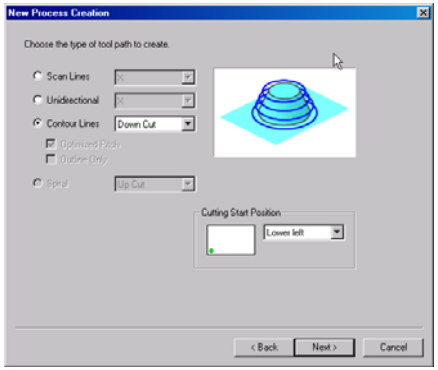
Notice the Margin in the X,Y view. This can be changed if desired by selecting the ‘Partial’ button and making desired changes.





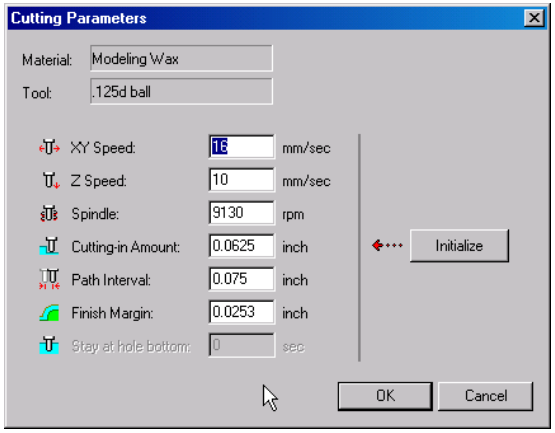
Roughing should be set to “Contour Lines Down Cut”. Contouring is a very efficient method of removing large amounts of material.

It cuts out the object in elevations.



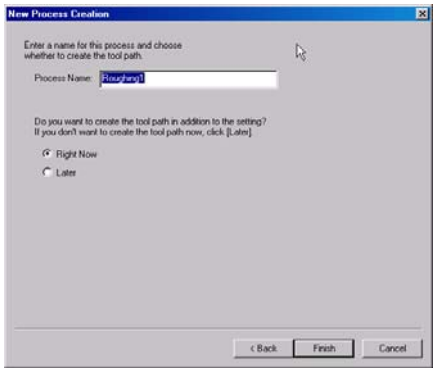
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

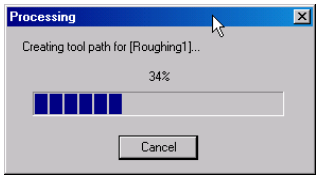


Process Name is listed as **Roughing 1**. It is a good idea to generate the tool path “Right Now” because you’ll be able to see the results before sending the tool path to the machine.

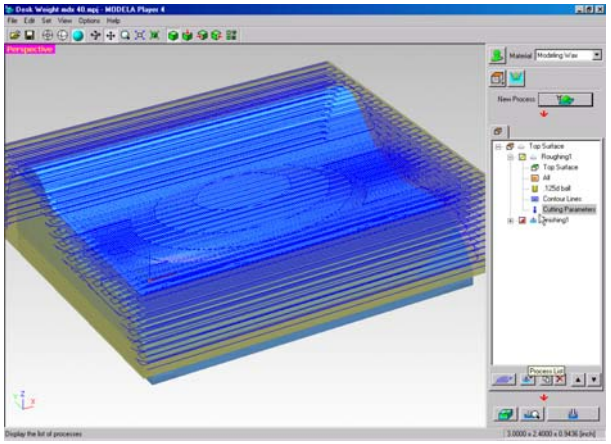
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.



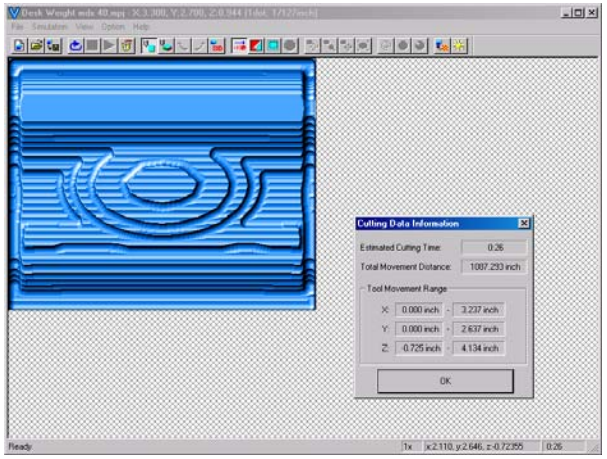
Roughing Tool Path.



8) In Modela 4 there is a built-in tool path simulator. It provides an accurate representation of the tool path being cut and a time estimate for its completion.

After the tool path is processed, **press the Preview Cutting button**. 

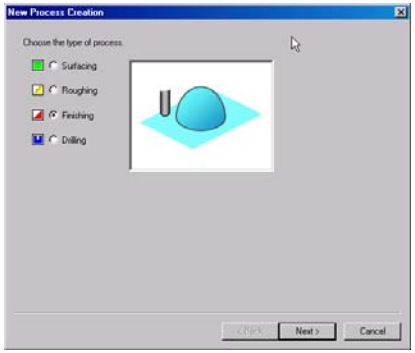
Virtual Modela automatically starts which displays the simulation and shows the cutting time.



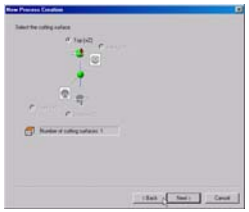
To create the Finishing tool path:

Select the New process Button: 

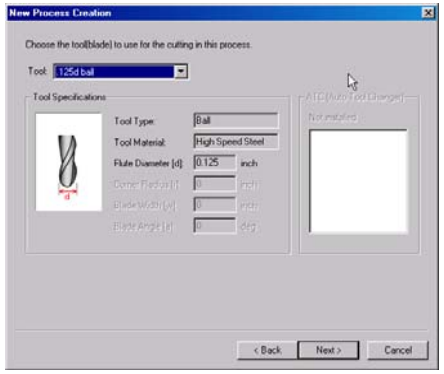
Select Finishing:



Because we have selected One-Side cutting, the Top is selected.

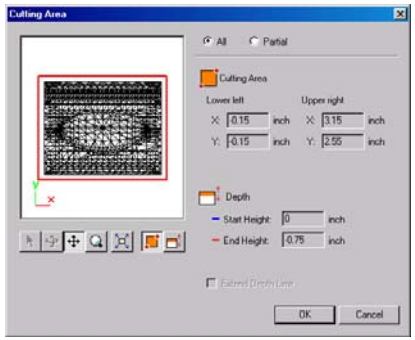


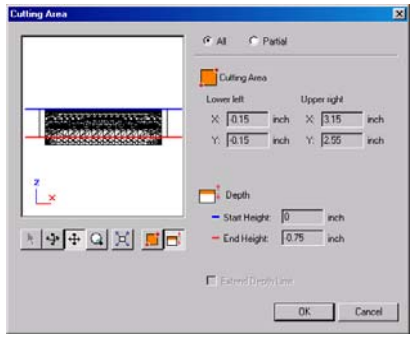
Select the .125d ball mill …….



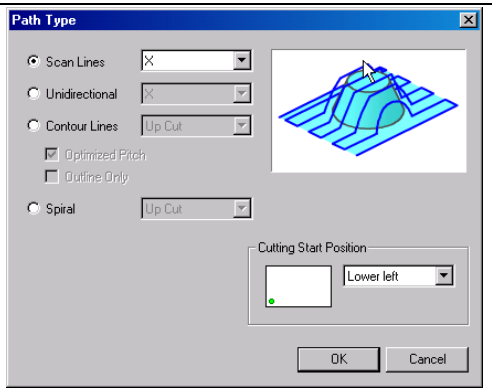
Check the cutting area and depth ……

The ‘All’ buttons should display the correct settings.





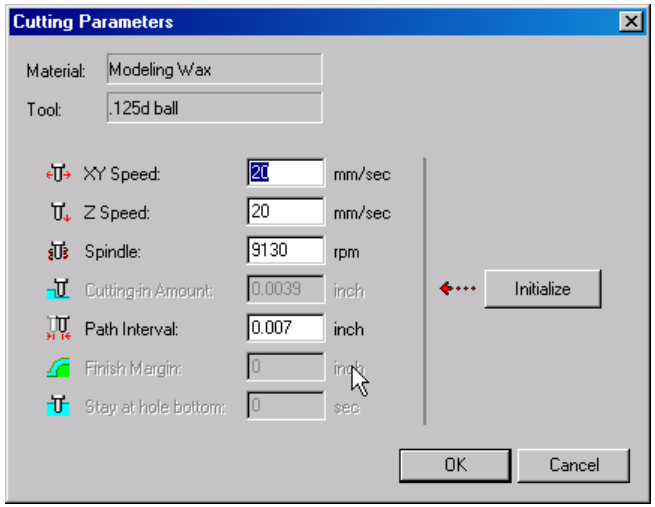
The **Finish** cutting direction can be set to cut in the X, Y, or both directions. These are lines cut along the chosen axis. Cutting both X and Y takes twice the time but gives a better finish.



Set the Finish cutting direction to Scan Lines X direction.

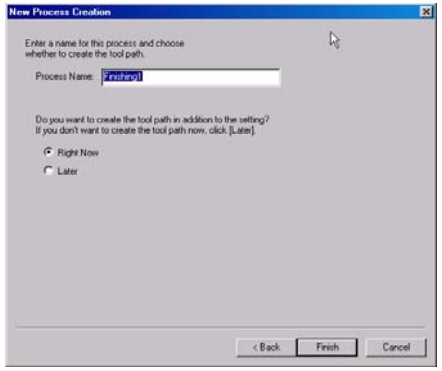
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

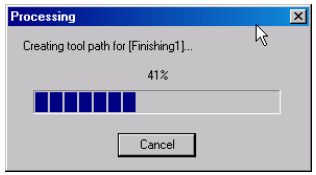


Process Name is listed as **Finishing 1**. It is a good idea to generate the tool path “Right Now” so you can see the results before sending the tool path to the machine.

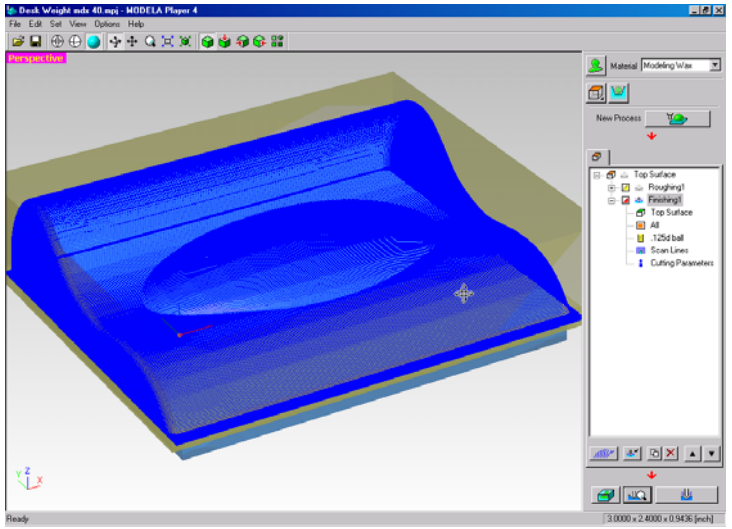
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.



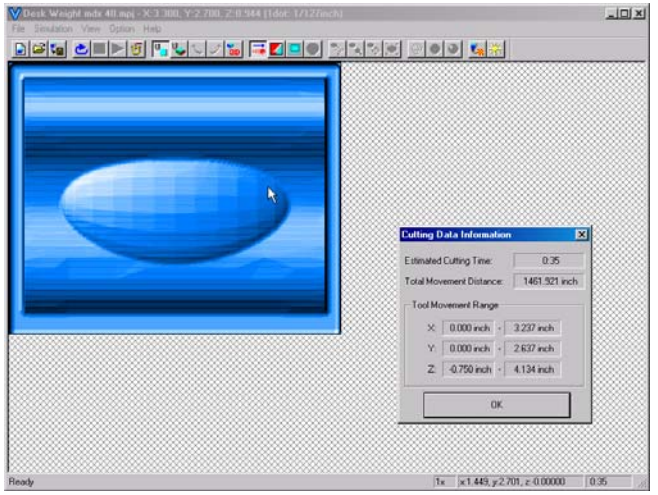
Finishing Tool Path.



9) In Modela Player 4 there is a built-in tool path simulator. It provides an accurate representation of the tool path being cut and a time estimate for its completion.

After the tool path is processed, **press the Preview Cutting button**. 

Virtual Modela automatically starts which displays the simulation and shows the cutting time.



10) To save the part as a Modela file:

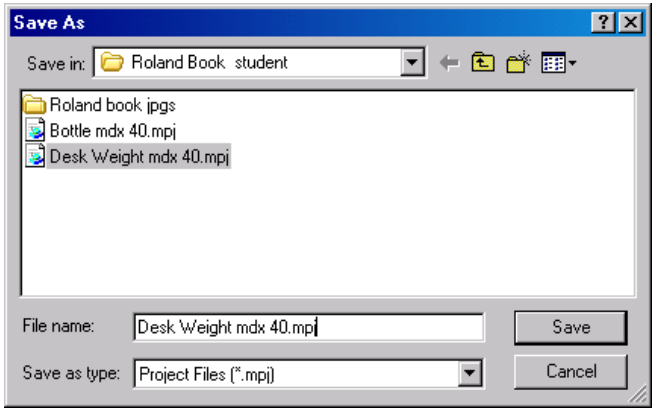
Pick the File pull down menu and Save.

The file name is the same as the one that was originally imported. Only the file extension is changed to .mdj

This is often a good time to make a second save too. Simply change part of the file-name to create a second file.

Pick Save As. **Type in the name** . 'Desk Weight mdx 40 done'
or another file name of your choice.

Result: There are now two files. In case one is damaged, there is a backup.

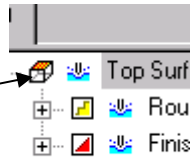


NOTE: If you do not have at least 55 minutes in the class period, do not continue to the milling steps. The average machining time for the Desk Weight is 50 minutes. Both toolpaths need to be completed without turning off the milling machine.

Step 13 explains how to exit Modela Player 4 if you do not have enough time at this point. There are instructions, at the end of the lesson, on how to open a saved Modela file.

11) To machine the part on the MDX 40:
Highlight the Tool Path to be machined.

In this case, **Highlight both tool paths**

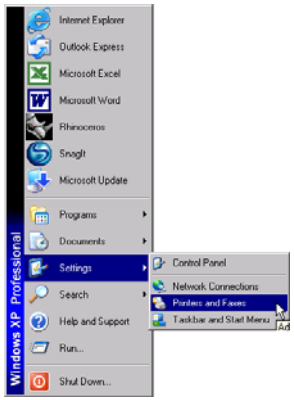


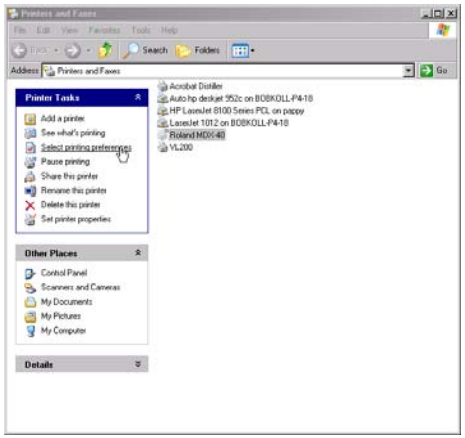
Select the Cutting/No Cutting button:

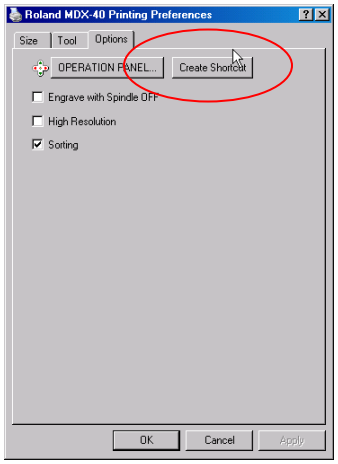
12) MDX-40 control panel:
Place a shortcut to the MDX 40 control panel on your desktop.

This gives you access to setting the cutter X, Y, Z origin. This step is Required before pressing the Cut button.

To create the shortcut, Select Create Shortcut in the printing preferences for the Roland mdx40:



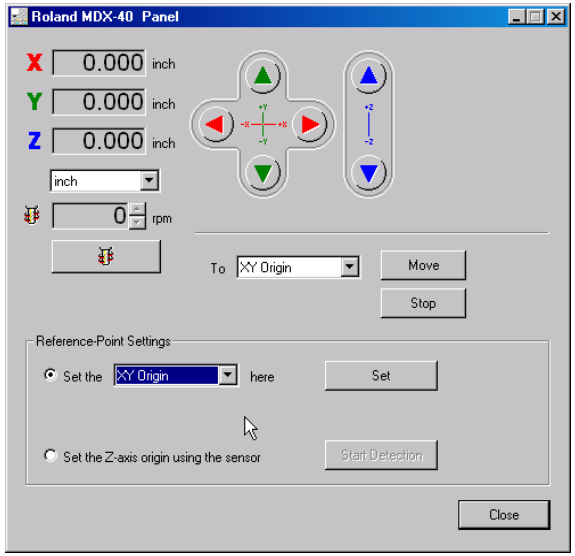




With the mouse and the arrows in the control panel, move the cutter to the proper location on the wax block.

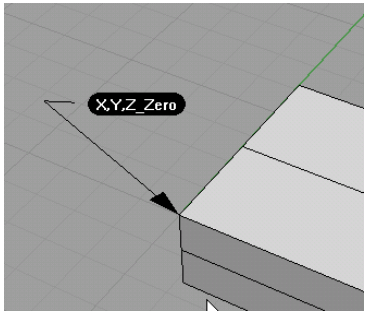
Set the Z origin, Set the Y origin, Set the X origin.

The MDX-40 ‘remembers’ the origin after it is set. If you have to restart your cutting it is easy to move the cutter to this location for re-starting if necessary.



One method to 'accurately' Set the X,Y,Z Origin at the corner of the wax stock block:

Set the X,Y,Z origin so it is at the left hand corner of the model……..

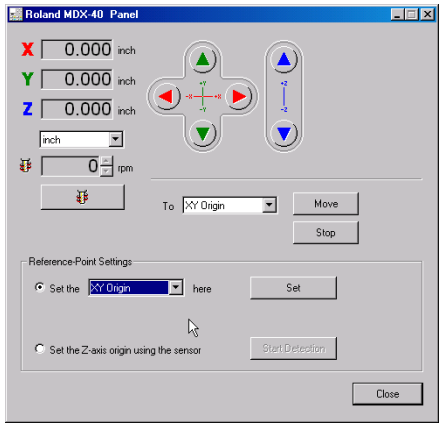


1 Set the Z origin at the top surface of the model by moving the cutter (with the mouse and the arrows in the control panel) so it touches the top of the block. Use a piece of notebook paper, approximately .003" thick. Move the cutter down until there is friction between the cutter and the block. The cutter at this point is .003 above the block. **Set Z origin here.** (temporary Z)

2 Set the X origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the left side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the X Origin here.** (temporary X)

3 Set the Y origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the front side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the Y Origin here.** (temporary)

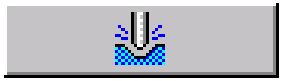
IMPORTANT: The XYZ readouts read 0.000 inches.
You need to change the position of the tool so the XY center of the tool is exactly at the corner of the block. The XY is currently one-half the tool diameter (.0625) away from the correct origin. Also, remember that our tool is .003 above the block.



With the mouse and the arrows in the control panel:

z… Move the cutter so the Z reads: Z -.003" **Set Z origin here**
x… Move the cutter so the X reads: X .0625" **Set X origin here**
y… Move the cutter so the Y reads: Y .0625" **Set Y origin here**

Select the Cut button: Sends the tool path files to the MDX-40 and begins to cut the part.



Remember: Highlight the Roughing and Finishing tool paths and select the Cut/NoCut button. This is button is used to control whether or not each tool path file will be cut by the the MDX40.

As your machine begins cutting:

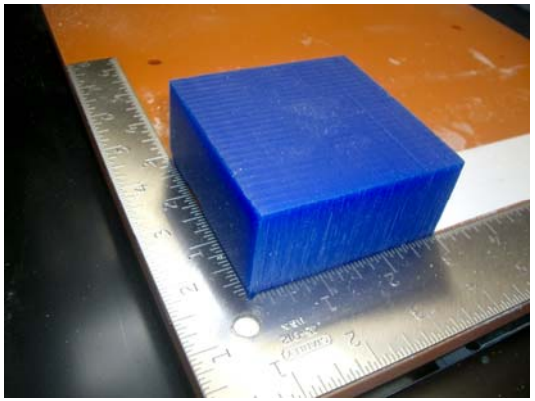
To Pause: Press the View button on the front of the MDX 40. The machine will move the cutter to a safe Z height and move the XY platform for a closer view.

>>>>**Caution:** Do not open the cover until the machine stops moving<<<<

To Resume: The machine will begin cutting at the exact location where it was when the View button was pressed.

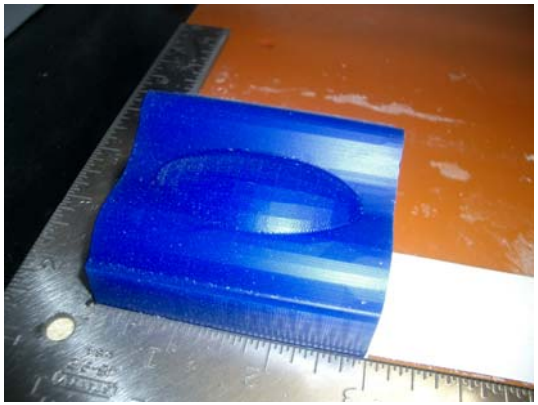
Do not leave the milling machine unattended when it is running!!!

Follow all standard safety procedures while operating the mill.



Wax block Ready to Cut

Machined Model



13) To exit the Modela Player 4:

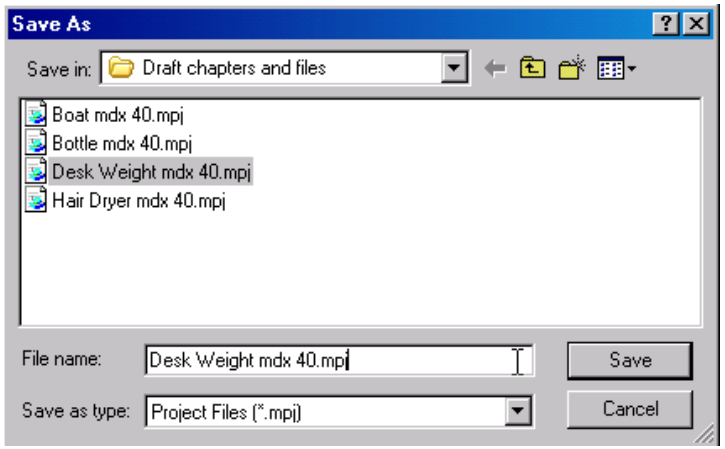
Pick the X in the upper top right corner of the Modela screen. When the Save dialogue box opens, **select Yes** to save the current file. The file retains the original file name except the extension changes to .mpj

To open a saved Modela Player 4 File:

With Modela Player 4 started:

Select the File pull down menu.

Select the File…… Desk Weight mdx 40.mpj

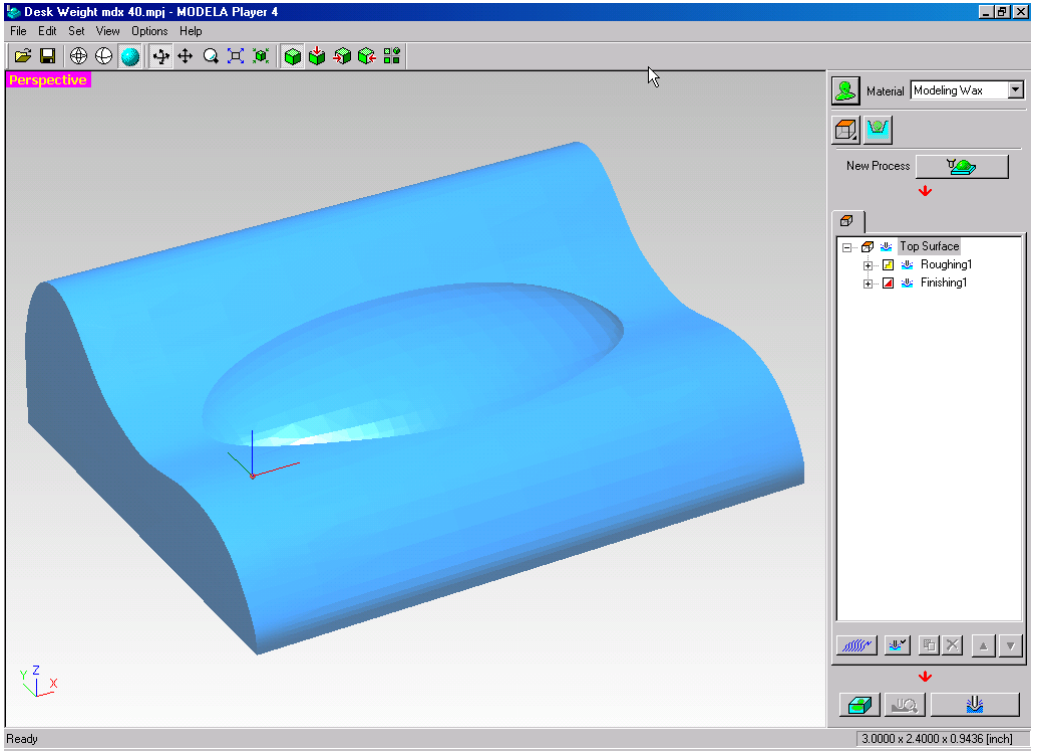


To easily recognize a Modela Player 4 file, look for the following icon In front of the file name:

Double clicking on this file name will also start Modela Player 4 and open the file.

Pick Open.

The file will open in Modela Player 4.



6

Hair Dryer - Rhinoceros

The hair dryer is a good example of creating a fairly complex model from some fairly simple geometry. Remember that operations that have been cover in the last two models such as opening the template, changing layers, etc. are not explained in detail in this lesson. It is assumed you have mastered these skills. If not, refer to the earlier lessons.

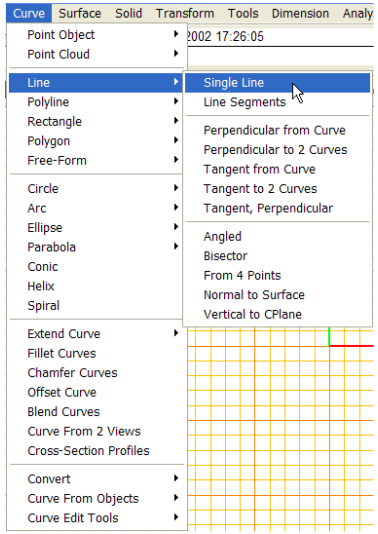
1) Start Rhino

2) Start a new drawing using the template created for these lessons.

3) Check to see that the **current layer is Construction**.

4) The lines for the tube of the hair dryer are the first to be created.

Note: Be sure to look at the number inputs very carefully as to not confuse commas with decimal points. Remember a 0 in front of a decimal point is not required. Example 0.25 can be input as .25.



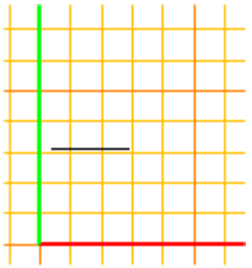
In the Top viewport:

Curve – Line – Single Line

Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **0.375,3.125 Enter**

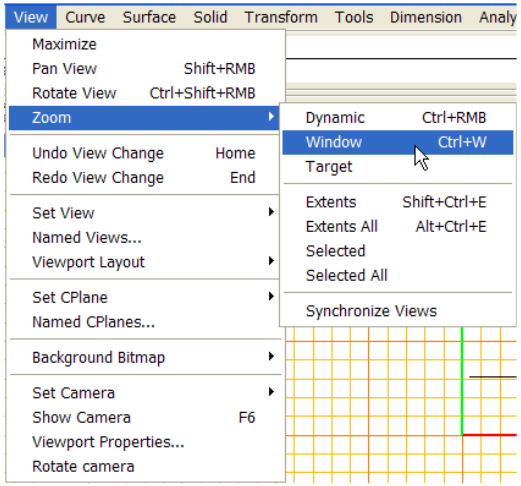
End of line (BothSides): **R2.5<0 Enter**

Result



5) Zoom window as displayed in the Top viewport.

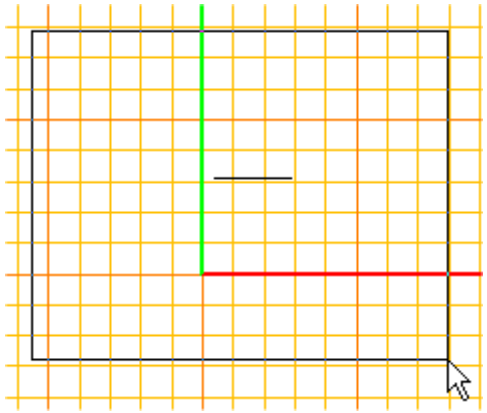
View – Zoom – Window



or

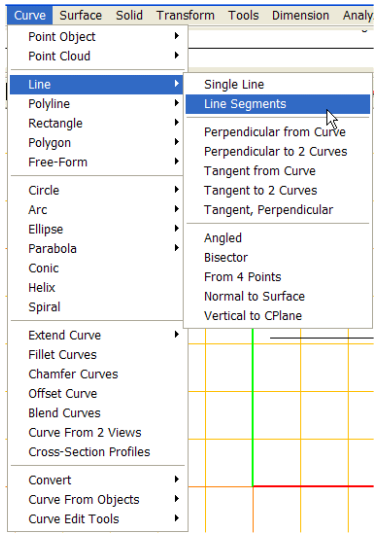
Pick the Zoom Window icon from the top tool bar.

Window placement



6) In this step, some of the geometry needed to create the handle will be drawn.

Curve - Line - Line Segments

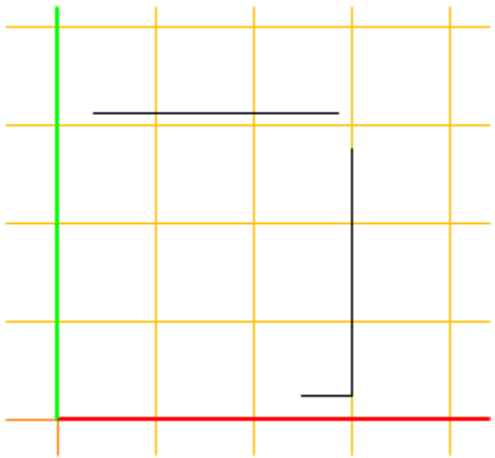


Start of line: **3,2.75 Enter**

End of line (Undo): **R2.5<270 Enter**

End of line. Press Enter when done (Undo): **R0.5<180 Enter**

End of line. Press Enter when done (Close Undo): **Enter**

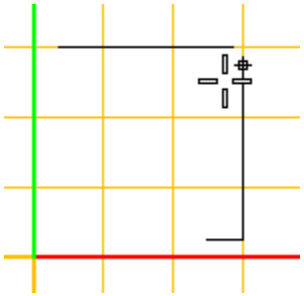


7) Run the **Zoom Extents All** command so you can better see the geometry.

The next few steps will create the geometry for the body of the dryer.

8) Set the **Osnap with End on only**.

9) To create the rotation line for the center of the body:



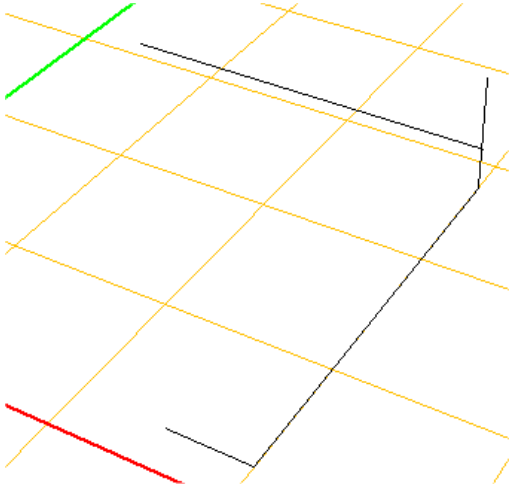
In the Top Viewport:

Curve – Line – Single Line

Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **Pick the top end of the vertical line as displayed**

End of line (BothSides): **R0,0,0.75 Enter**

Result (shown from the Perspective viewport)

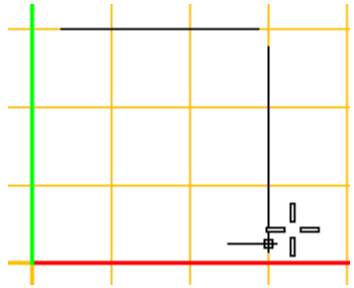


10) If you have not picked any other commands press **Enter** to return to the last command

or

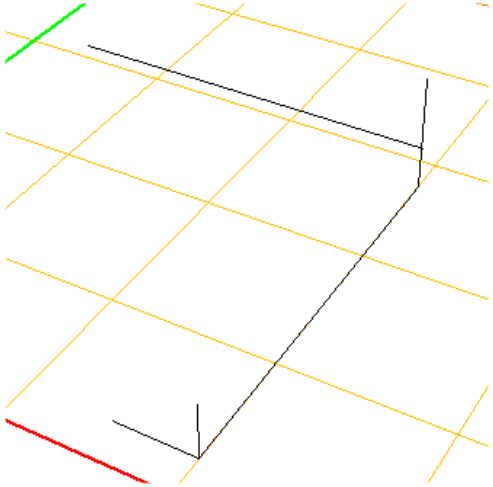
Curve - Line - Single Line

Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **Pick the bottom end of the vertical line as displayed**



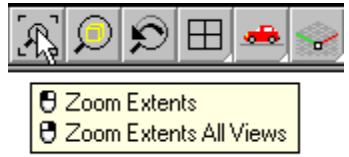
End of line (BothSides): **R0,0,0.35 Enter**

Result (shown from the Perspective viewport)

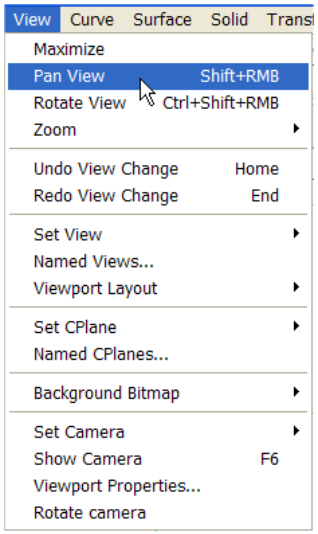


11) The view in the Front view port needs to be adjusted for step 10.

Run the **Zoom Extents All command**.



Pick in the **Front viewport** to make it active.



The view needs to be panned so the next step can be viewed.

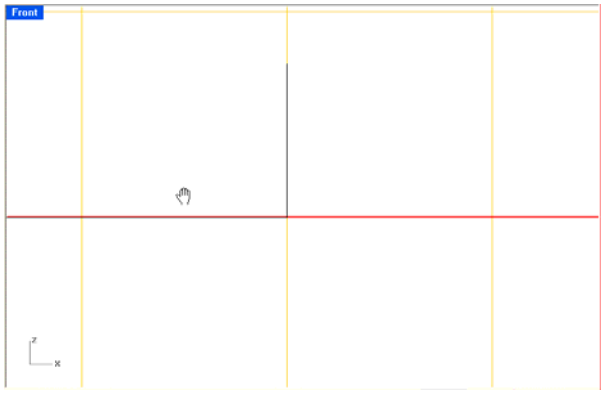
View – Pan View

or

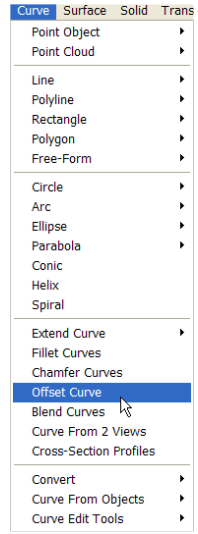
Pick the Pan icon from the top tool bar.

Pick and hold the left mouse button towards the right side of the Front viewport. **Drag to the left** about 1/2 of the way across and **release** the mouse button.

The result is displayed.

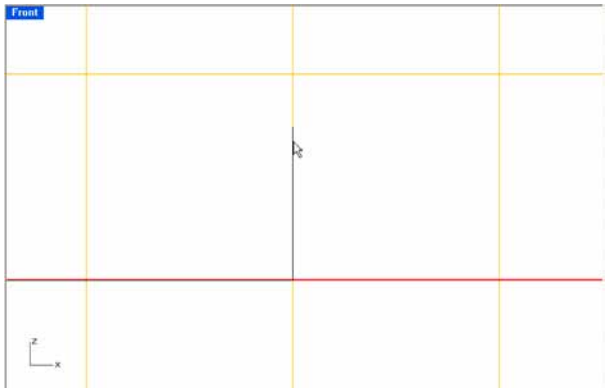


12) The vertical line in the Front viewport needs to be offset to be used to create the curve for the body of the hair dryer.

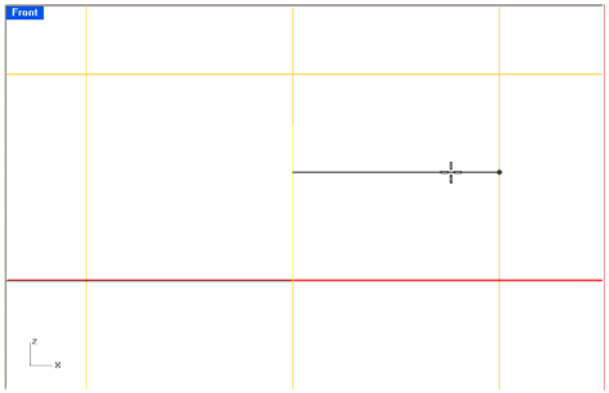


Curve – Offset Curve

Select curve to offset (Distance=1 Corner=Sharp ThroughPoint Tolerance):1
Select curve to offset (Distance=1 Corner=Sharp ThroughPoint Tolerance):**Pick the vertical line in the Front viewport**

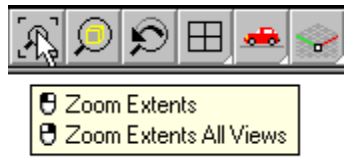


Through point (Distance=1 Corner=Sharp ThroughPoint Tolerance):**Select to the right as the side to offset**

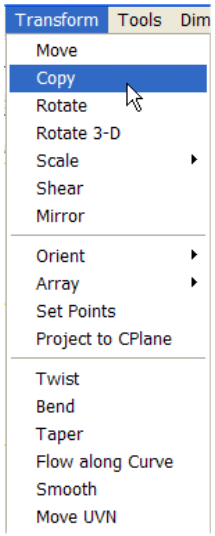


Result

13) Run the **Zoom Extents All command** so you can see all of the geometry

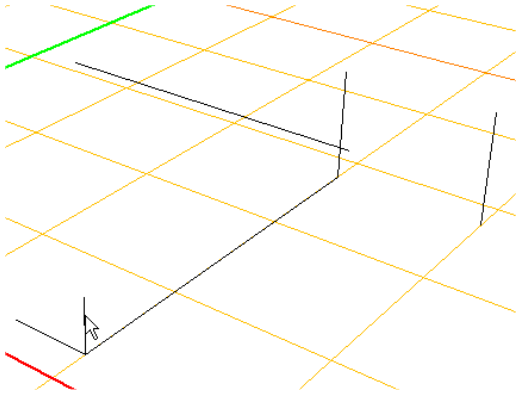


14) A copy of the vertical line for the handle is required at the other end of the horizontal handle line.

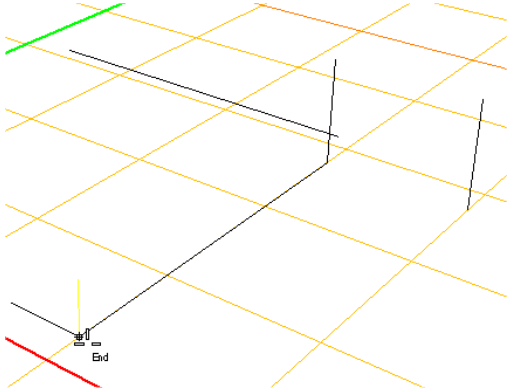


Transform - Copy

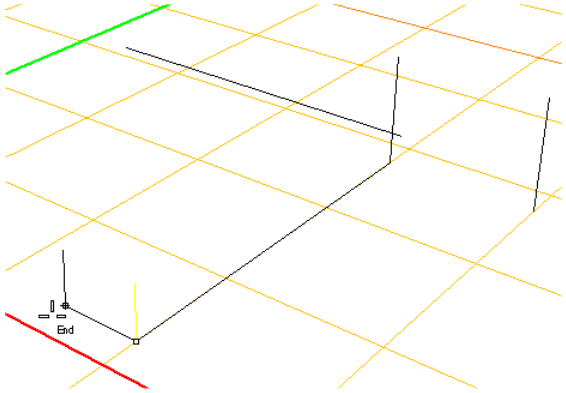
Select objects to copy: **Pick as displayed** in the Perspective viewport.



Select objects to copy. Press Enter when done: **Enter**

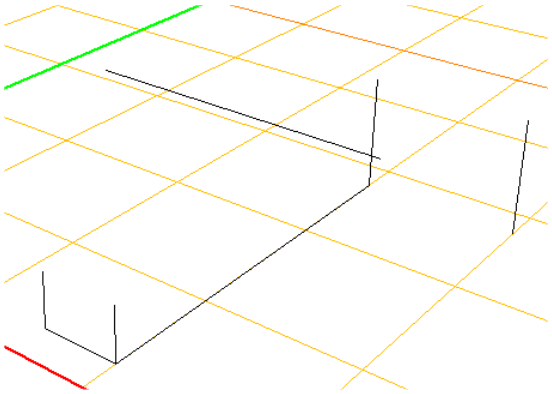


Point to copy from (Vertical=No InPlace): **Pick as displayed**



Point to copy to: **Pick as displayed**

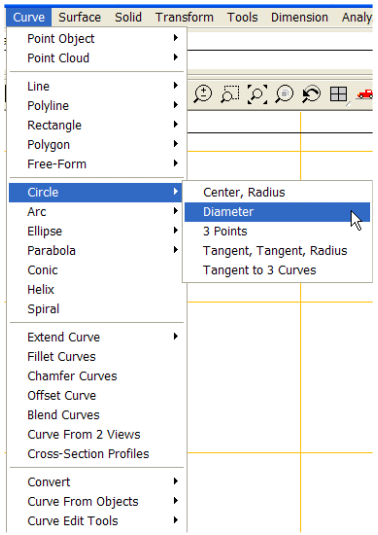
Point to copy to: **Enter**



Result

15) Change the layer to Curves.

16) To draw the circle for the tube of the hair dryer:

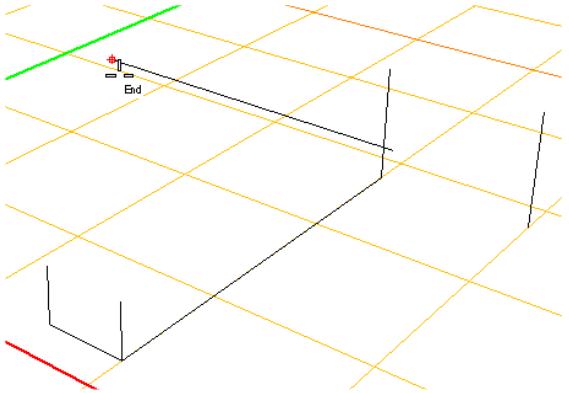


Curve - Circle - Center, Radius

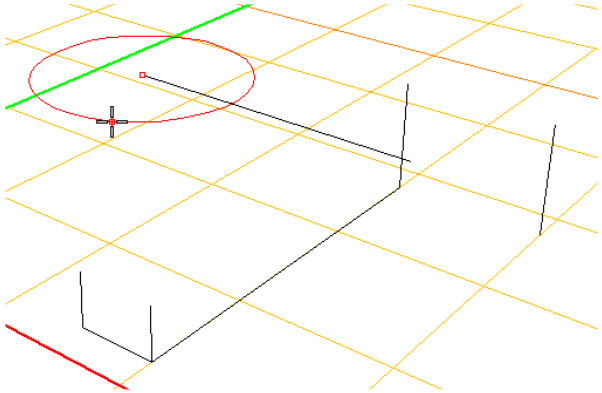
In the Perspective Viewport:

With **End** Osnap on only:

Center of circle (Deformable Vertical Diameter 3Point Tangent AroundCurve):
Pick as displayed

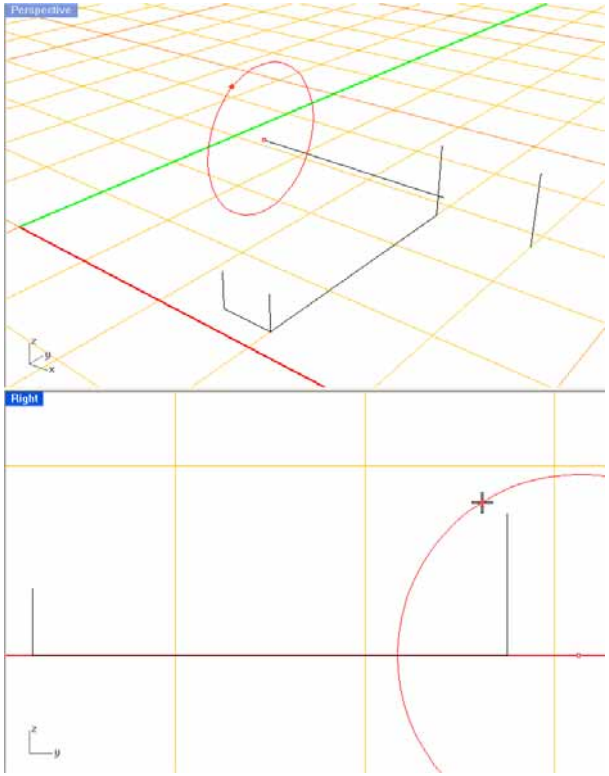


Move the mouse but **do not** make a pick.



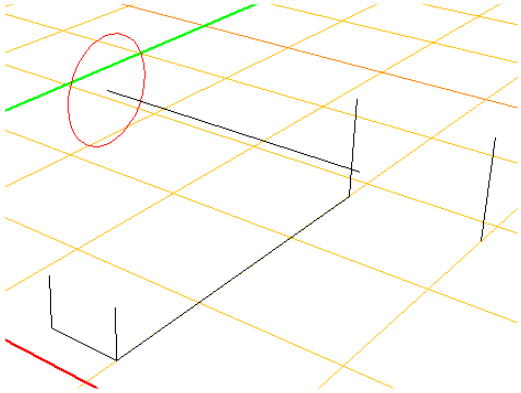
You can see the circle is not perpendicular to the line that you picked.

Move the pointer into the Right viewport. The circle is now perpendicular to the line.



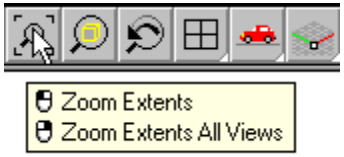
Radius <1> (Diameter): **.5 Enter**

Result

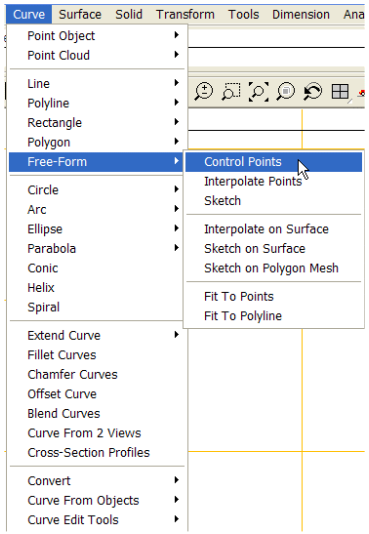


17) The body and handle curves are the next geometry to be created.

Run the Zoom Extents All command.

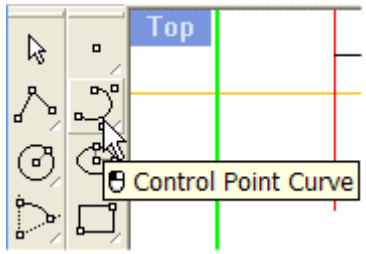


Curve – Free-Form – Control Points



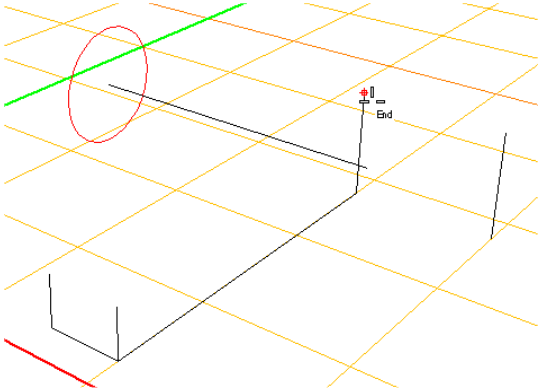
or

Pick the Control Point Curve command from the toolbar.

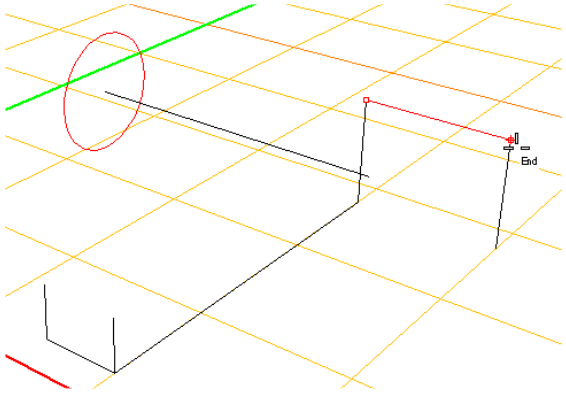


In the Perspective Viewport:

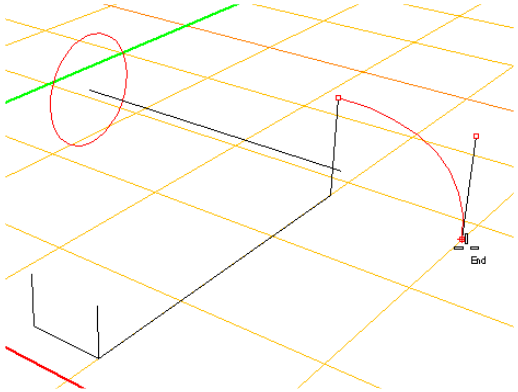
Start of curve (Degree=3): **Pick as displayed**



Next point (Degree=3 Undo): **Pick as displayed**

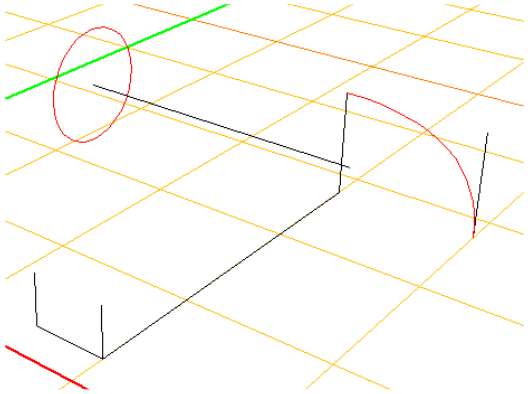


Next point. Press Enter when done (Degree=3 Undo): **Pick as displayed**



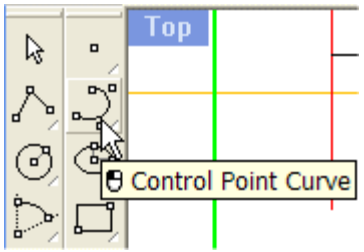
Next point. Press Enter when done (Degree=3 Close Sharp=No Undo): **Enter** to end the command

Result

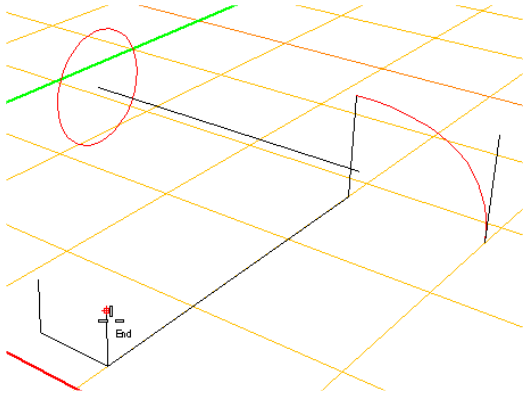


18) The same process needs to be done to create the handle curve.

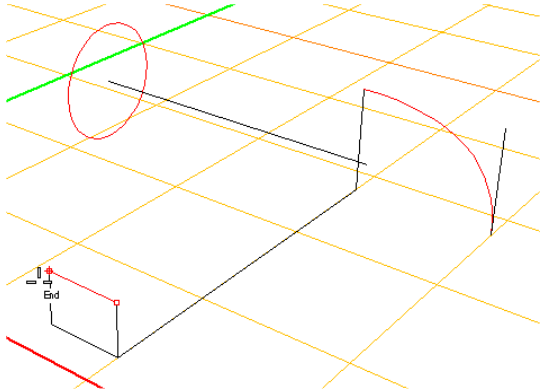
Curve – Free-Form – Control Points



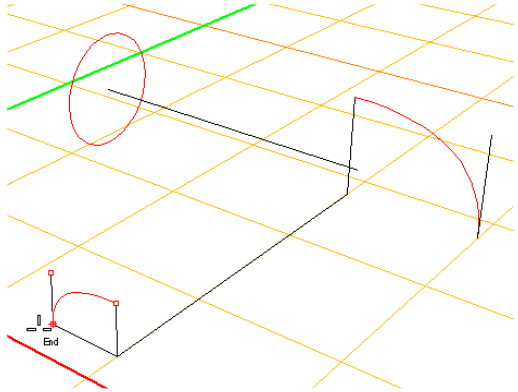
Start of curve (Degree=3): **Pick as displayed**



Next point (Degree=3 Undo): **Pick as displayed**

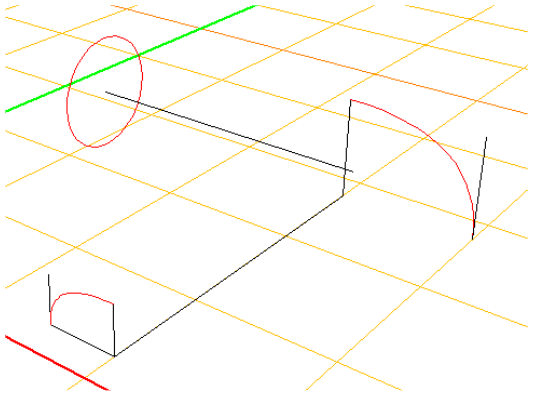


Next point. Press Enter when done (Degree=3 Undo): **Pick as displayed**



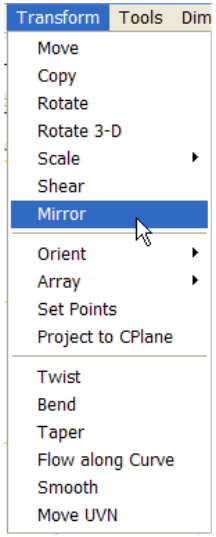
Next point. Press Enter when done (Degree=3 Close Sharp=No Undo): **Enter** to complete the command

Result

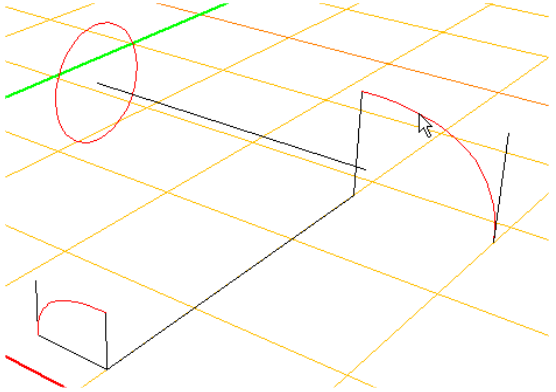


19) Both the handle and the body curves need to be mirrored to create the geometry necessary to form proper surfaces.

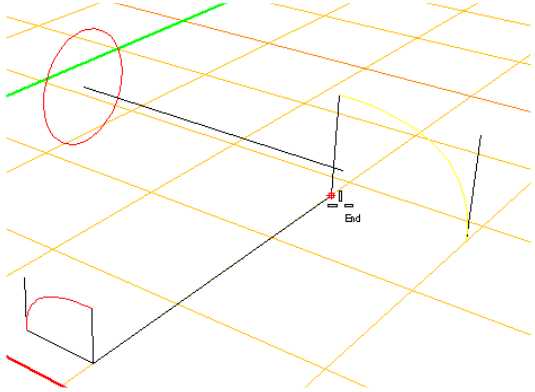
Transform - Mirror



Select objects to mirror: **Pick as displayed** in the Perspective viewport

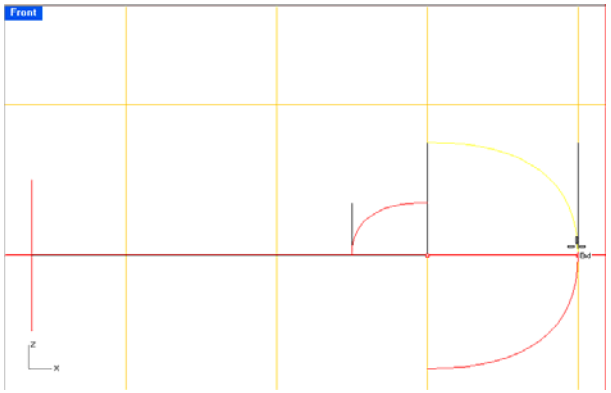


Select objects to mirror. Press Enter when done: **Enter**



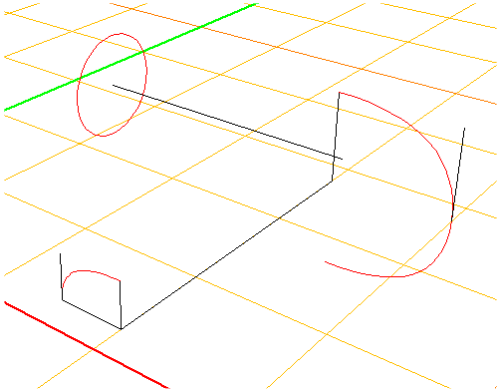
Start of mirror plane (Copy=Yes): **Pick as displayed**

Move to the Front viewport:



End of mirror plane (Copy=Yes): **Pick as displayed**

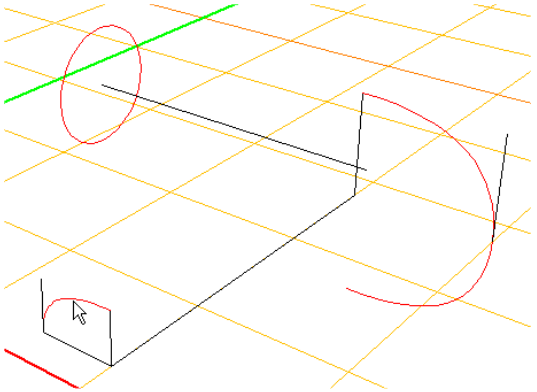
Result



To mirror the handle curve:

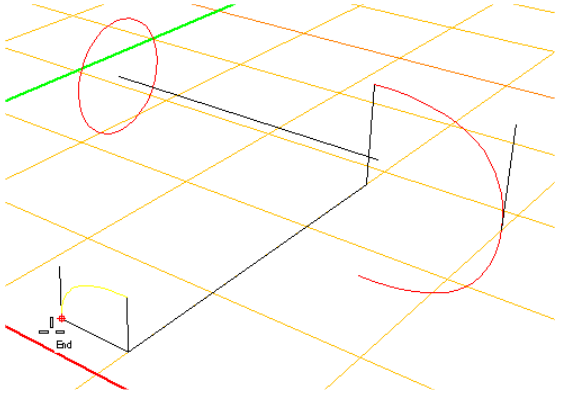
Transform - Mirror

Select objects to mirror: **Pick as displayed** in the **Perspective viewport**.

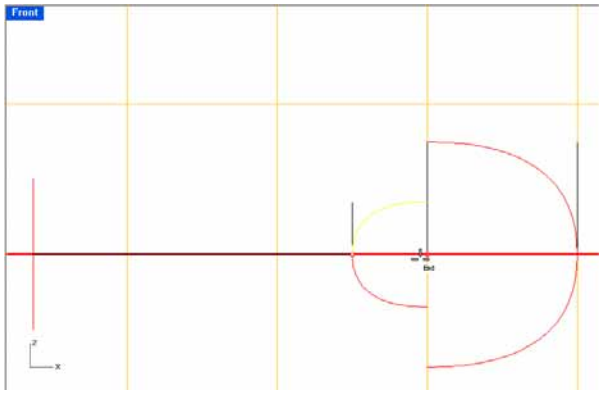


Select objects to mirror. Press Enter when done: **Enter**

Start of mirror plane (Copy=Yes): **Pick as displayed**

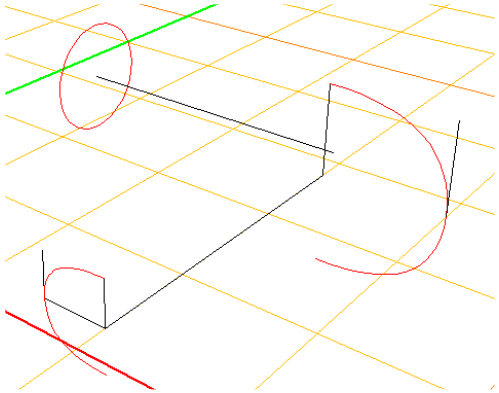


Move to the Front viewport:



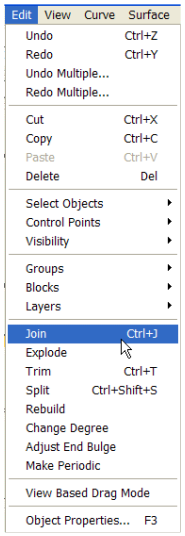
End of mirror plane (Copy=Yes): **Pick as displayed**

Result

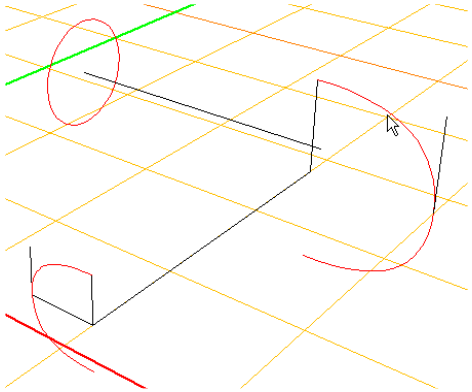


20) The two body curves and the handle curves need to be joined together.

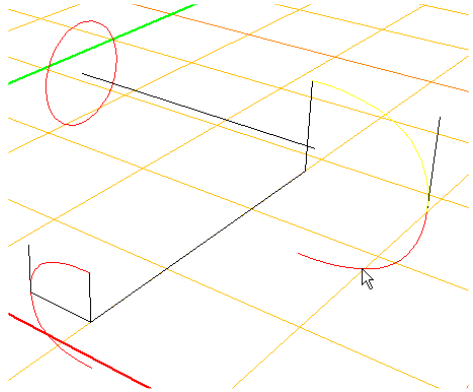
Edit - Join



Select object for join: **Pick as displayed**



Select curve to join: **Pick as displayed**

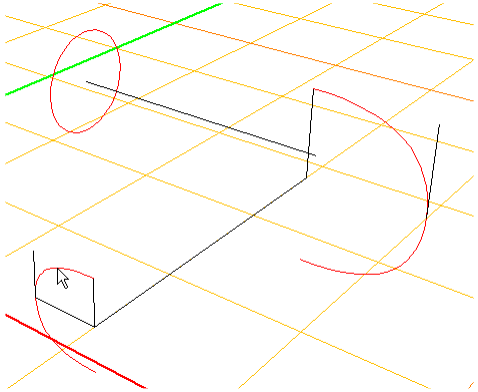


Select curve to join: **Enter** to complete the command

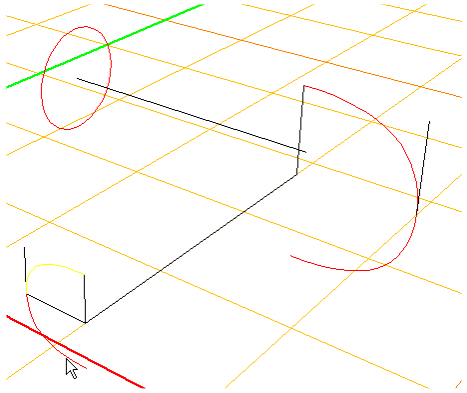
2 curves joined into 1.

Edit - Join

Select object for join: **Pick as displayed**



Select curve to join: **Pick as displayed**



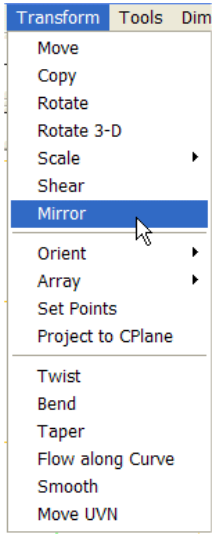
Select curve to join: **Enter** to complete the command

2 curves joined into 1.

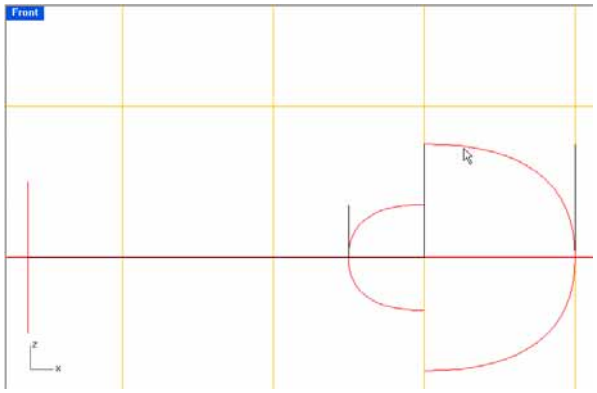
21) Both body and handle geometry need to be mirrored and joined closed shapes.

With only the End Osnap on.

Transform – Mirror

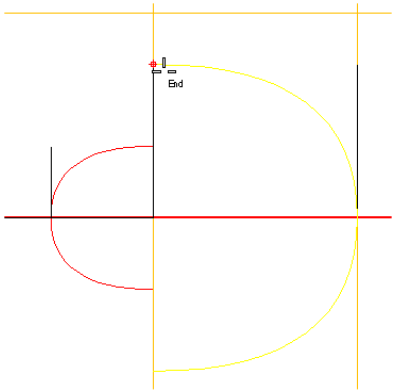


Select objects to mirror: **Pick as displayed** in the Front viewport

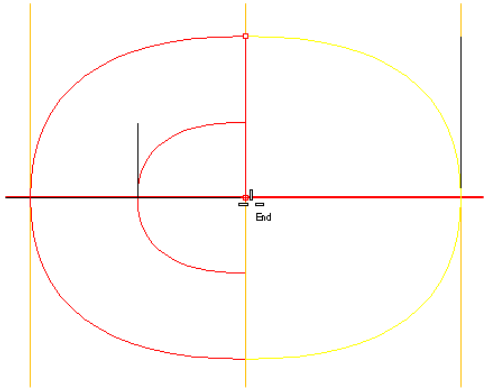


Select objects to mirror. Press Enter when done: **Enter**

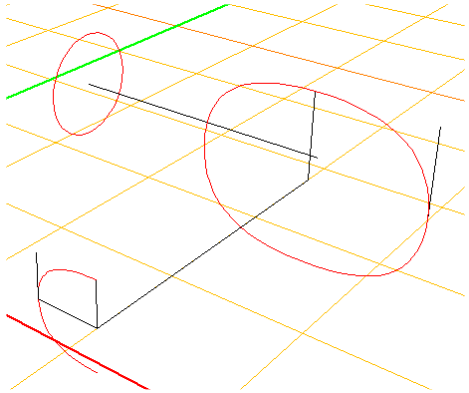
Start of mirror plane (Copy=Yes): **Pick as displayed**



End of mirror plane (Copy=Yes): **Pick as displayed**

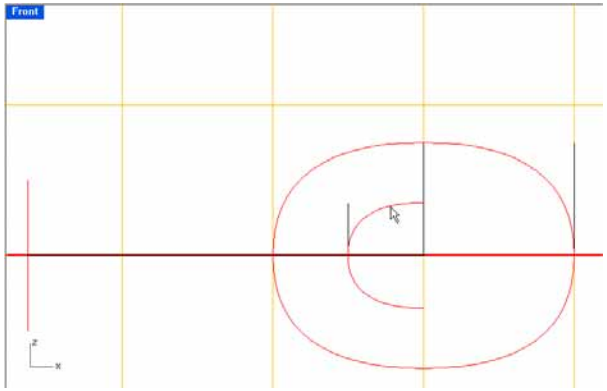


Result



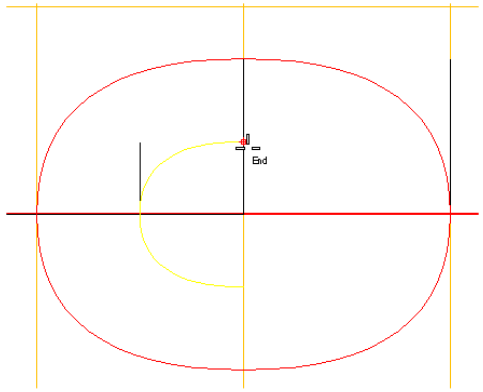
Transform – Mirror

Select objects to mirror: **Pick as displayed** in the Front viewport

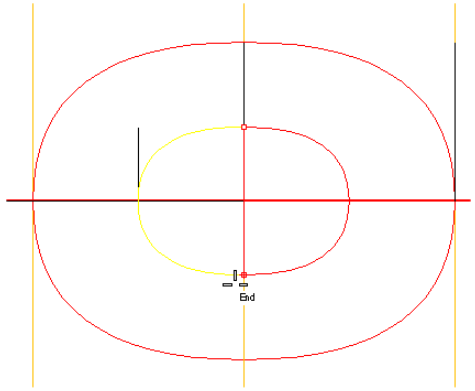


Select objects to mirror. Press Enter when done: **Enter**

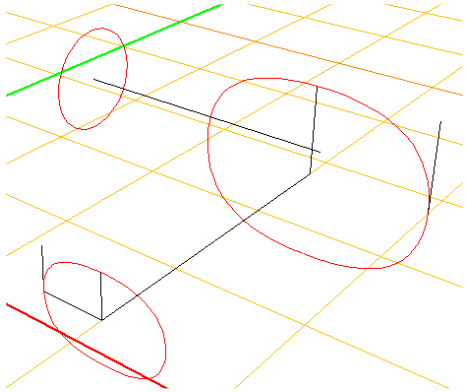
Start of mirror plane (Copy=Yes): **Pick as displayed**



End of mirror plane (Copy=Yes): **Pick as displayed**



Result

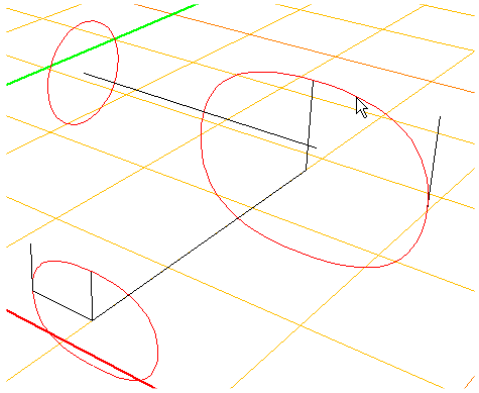


22) The halves of the body and handle need to be joined.

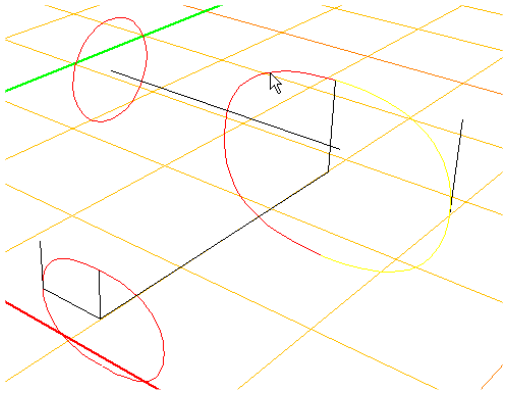
To join the body curves:

Edit - Join

Select object to join: **Pick as displayed**



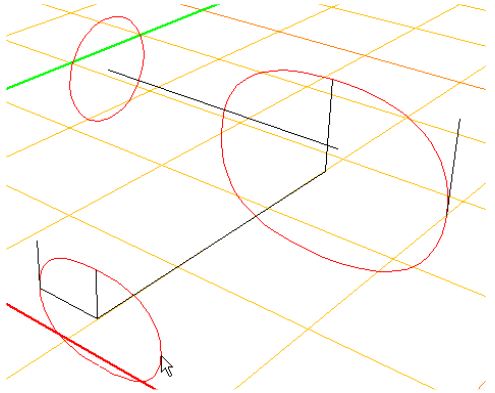
Select next object to join: **Pick as displayed**



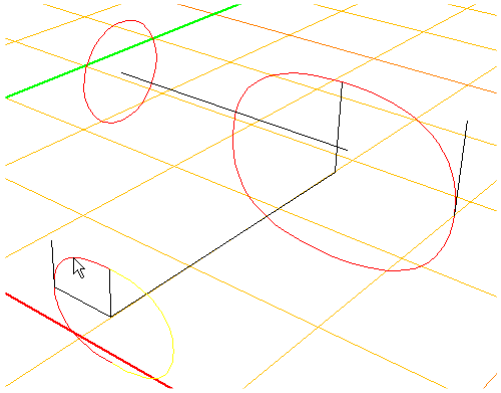
To join the handle curves:

Edit - Join

Select object to join: **Pick as displayed**



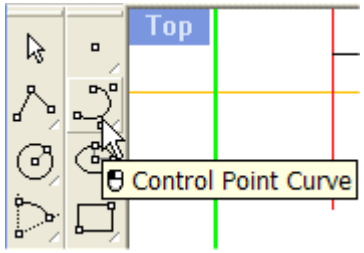
Select next object to join: **Pick as displayed**



23) The next step is to create the curve to sweep the handle.

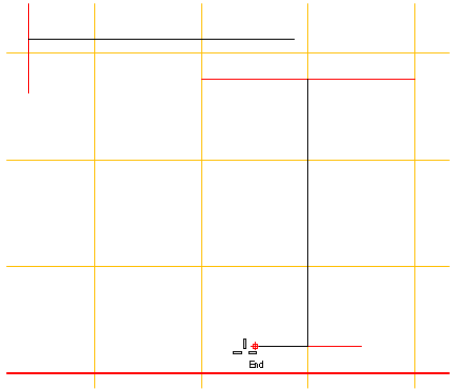
Turn **ORTHO** off, if it is on. **Only End Osnap should be on.**

To create the curve:

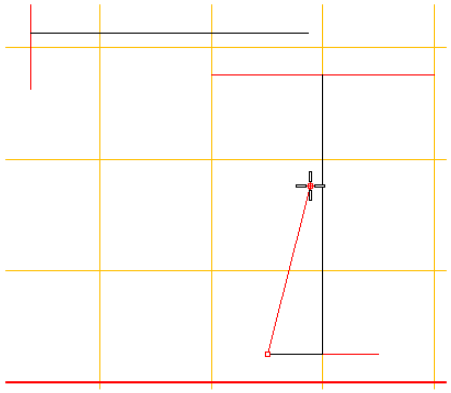


Curve – Free-Form – Control Points

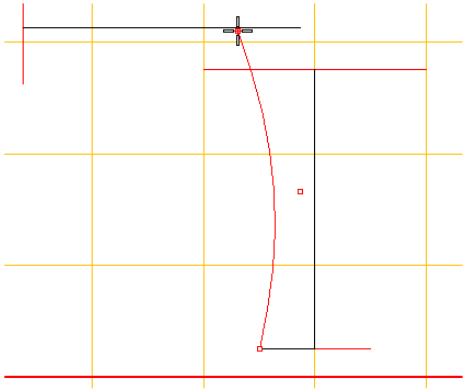
Start of curve (Degree=3): **Pick as displayed**



Next point (Degree=3 Undo): **Pick as displayed**

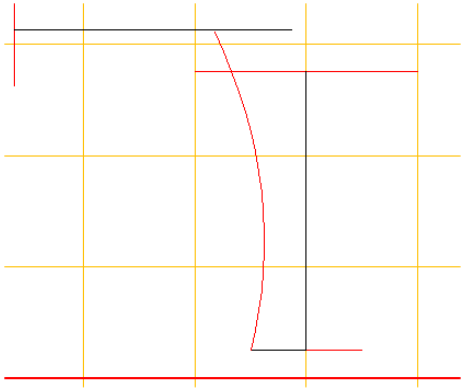


Next point. Press Enter when done (Degree=3 Undo): **Pick as displayed**



Next point. Press Enter when done (Degree=3 Close Sharp=No Undo): **Enter** to complete the command

Result

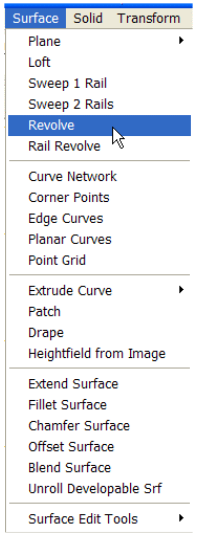


The line and curve geometry is complete and the surfaces can be created.

24) The first surface that will be created is the body. It will be a revolved surface.

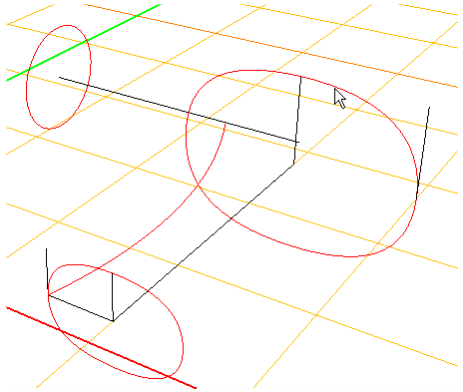
Set the Layer to Surfaces.

Surface - Revolve

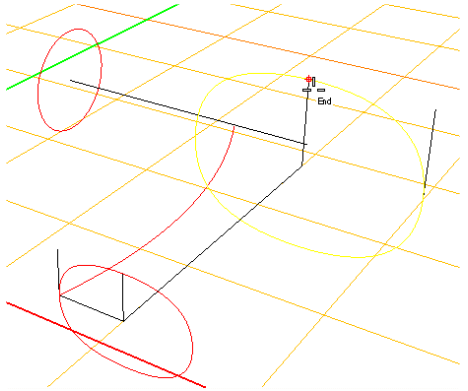


Select curves to revolve: **Pick as displayed**

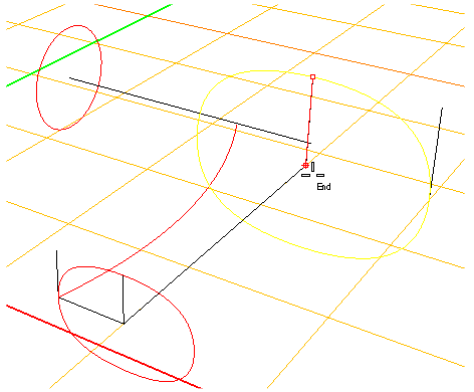
Select curves to revolve. Press Enter when done: **Enter**



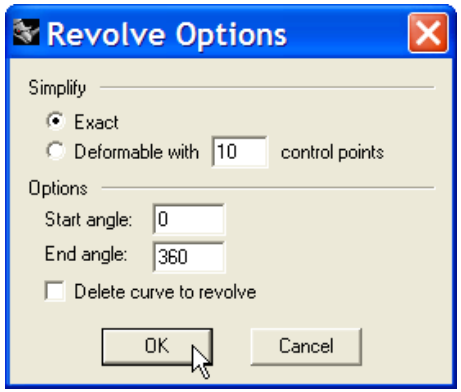
Start of revolve axis: **Pick as displayed**



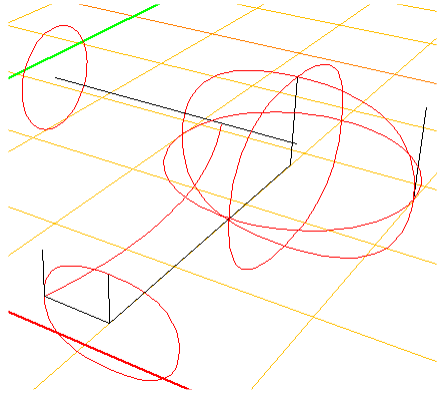
End of revolve axis: **Pick as displayed**



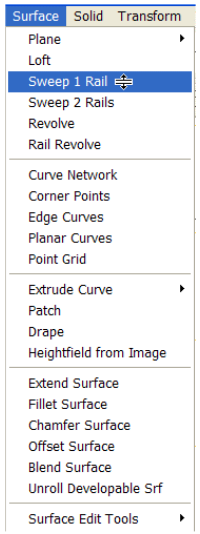
Set the Revolve Options dialogue box as displayed and **select OK**.



Result

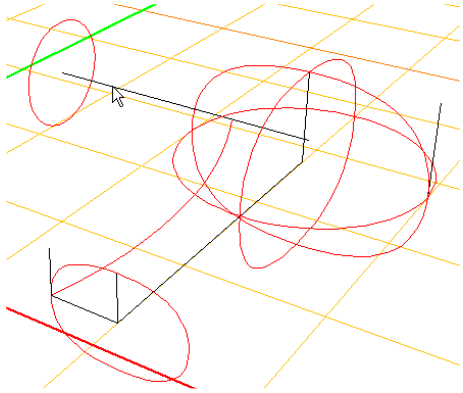


25) To create the air tube with a Sweep:



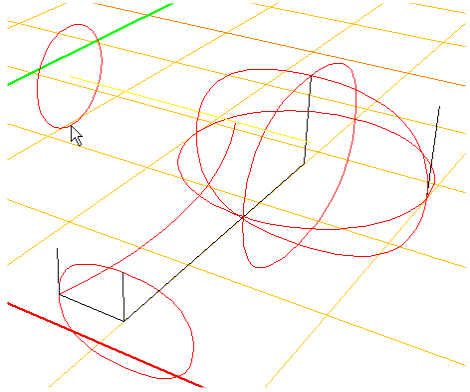
Surface - Sweep 1 Rail

Select rail curve: **Pick as displayed**

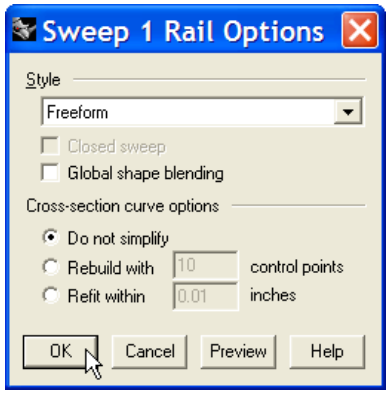


Select cross section curves (Point): **Pick as displayed**

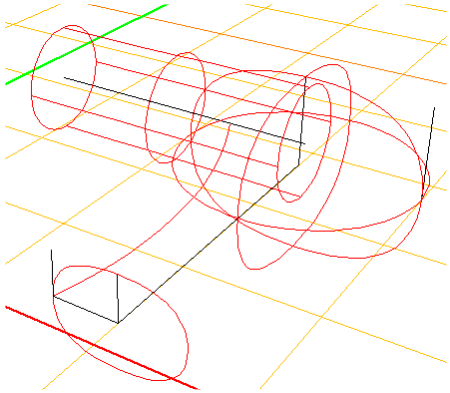
Select cross section curves. Press Enter when done (Point): **Enter**



Set the Sweep options as displayed and **select OK**.

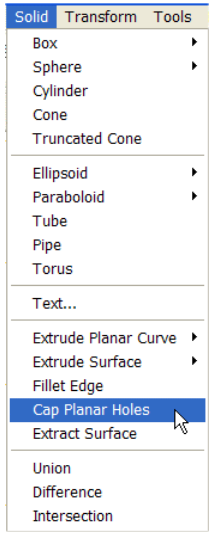


Result

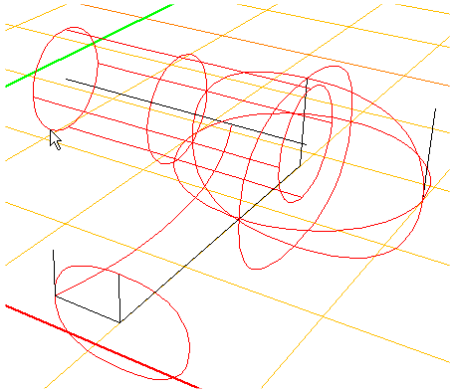


26) The air tube ends are open. To turn the tube into a solid they need the ends capped.

Solid - Cap Planar Holes

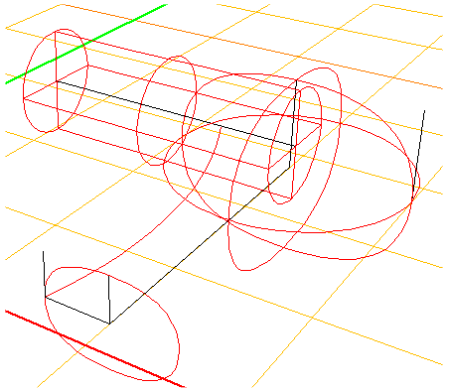


Select surfaces or polysurfaces to cap: **Pick as displayed**



Select surfaces or polysurfaces to cap. Press Enter when done: **Enter**
Created 2 end caps on one object.

Result

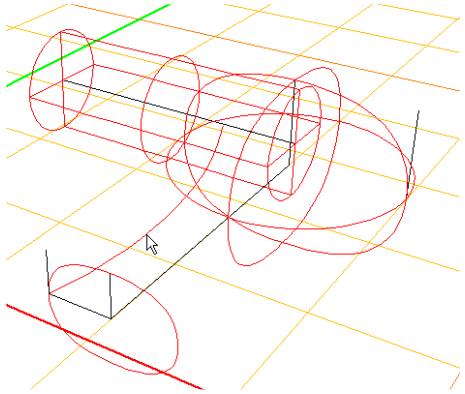


27) The handle will also be created with Sweep along a path.

To create the handle:

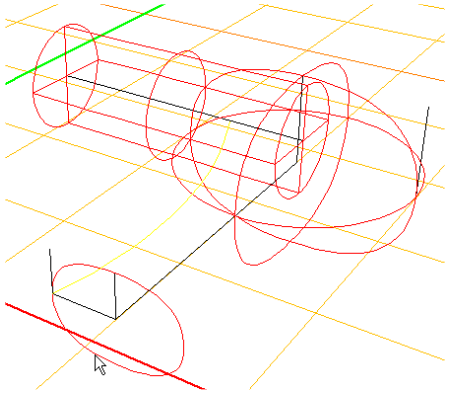
Surface - Sweep 1 Rail

Select rail curve: **Pick as displayed**



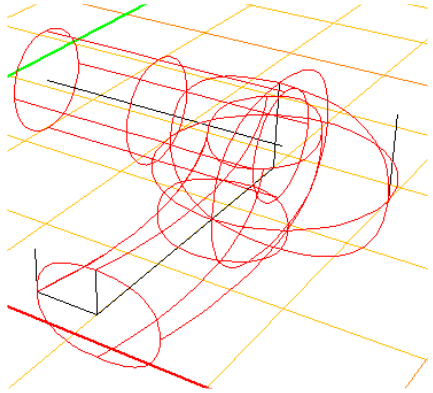
Select cross section curves (Point): **Pick as displayed**

Select cross section curves. Press Enter when done (Point): **Enter**



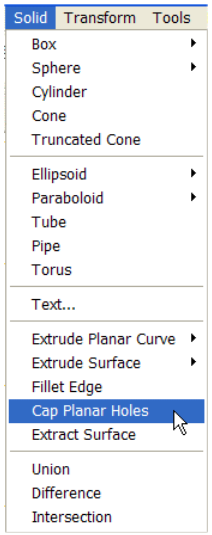
Set the Sweep options as displayed and **select OK**.

Result

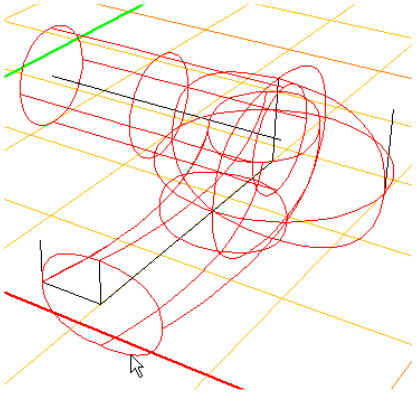


28) The handle ends need to be capped.

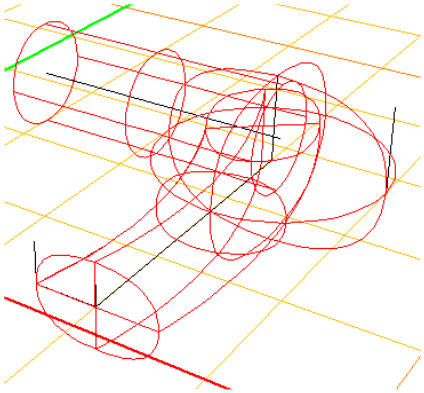
Solid - Cap Planar Holes



Select surfaces or polysurfaces to cap: **Pick as displayed**



Select surfaces or polysurfaces to cap. Press Enter when done: **Enter**
Created 2 end caps on one object.

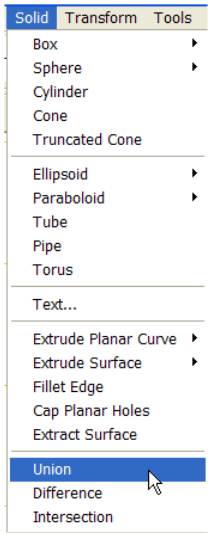


Result

29) Turn OFF all of the layers **except Surfaces**.

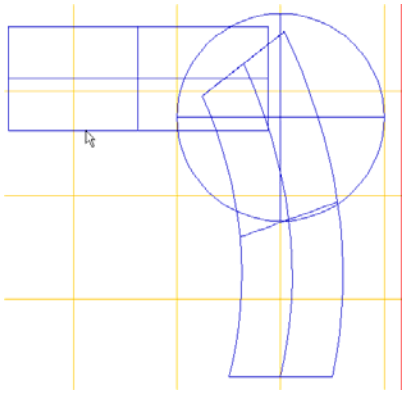
30) The three parts of the hair dryer need to be joined together.

To join the three parts together:

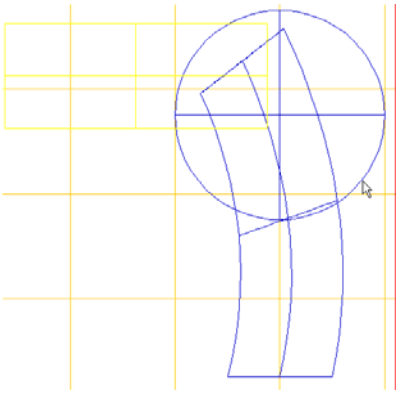


Solid – Union

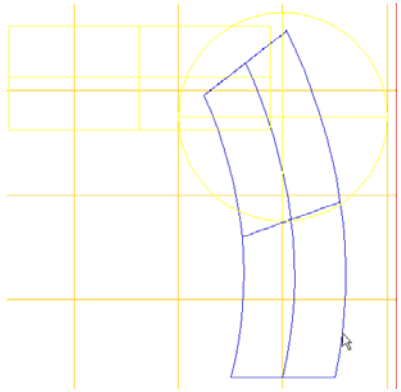
Select surfaces or polysurfaces to union: **Pick as displayed**



Select surfaces or polysurfaces to union: **Pick as displayed**

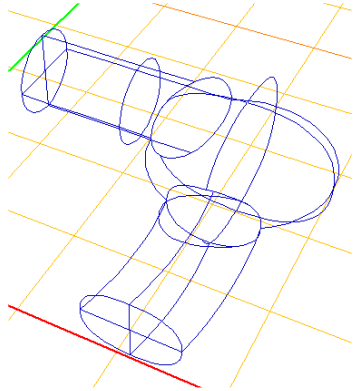


Select surfaces or polysurfaces to union. Press Enter when done: **Pick as displayed**

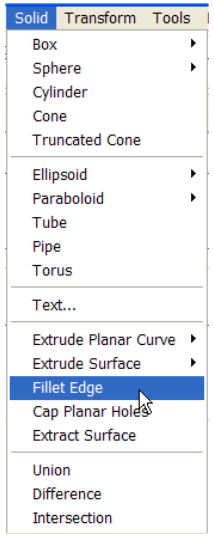


Select surfaces or polysurfaces to union. Press Enter when done: **Enter**
Boolean union in progress... Press Esc to cancel

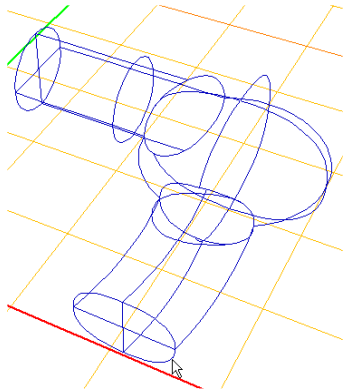
Result



31) The final step in creating the hair dryer is adding the fillets for the end of the handle and at the intersection of the handle and air tube to the body.



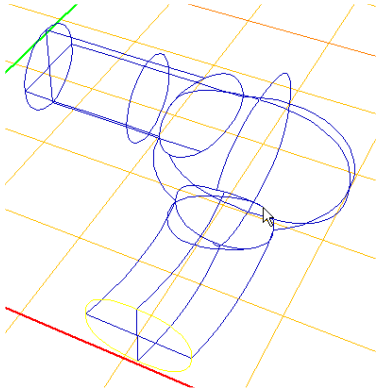
Solid - Fillet Edge



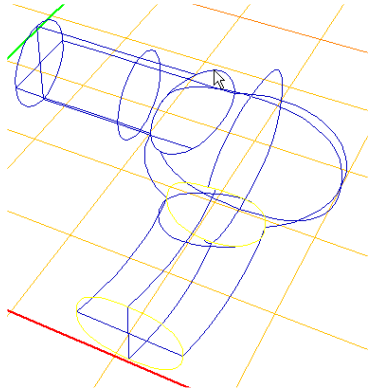
Select edges to fillet (Radius=1):**0.15 Enter**

Select edges to fillet (Radius=0.15): **Pick as displayed**

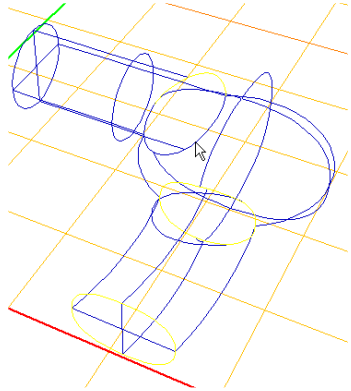
Select edges to fillet. Press Enter when done (Radius=0.15): **Pick as displayed**



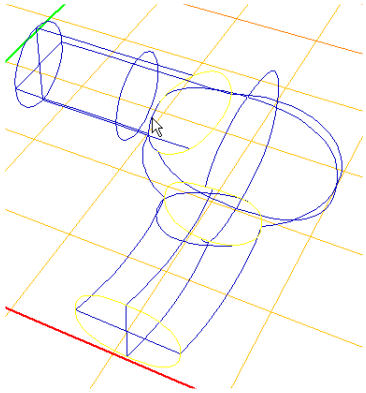
Select edges to fillet. Press Enter when done (Radius=0.15): **Pick as displayed**



Select edges to fillet. Press Enter when done (Radius=0.15): **Pick as displayed**

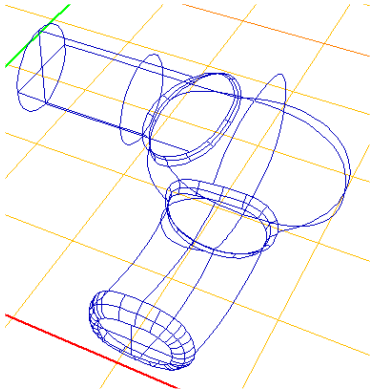


Select edges to fillet. Press Enter when done (Radius=0.15): **Pick as displayed**



Select edges to fillet. Press Enter when done (Radius=0.15): **Enter**
Creating Fillets.... Press Esc to cancel

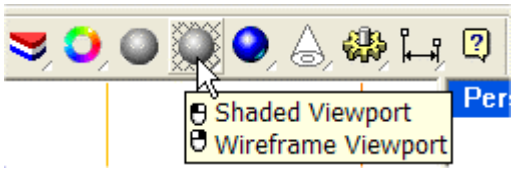
Result



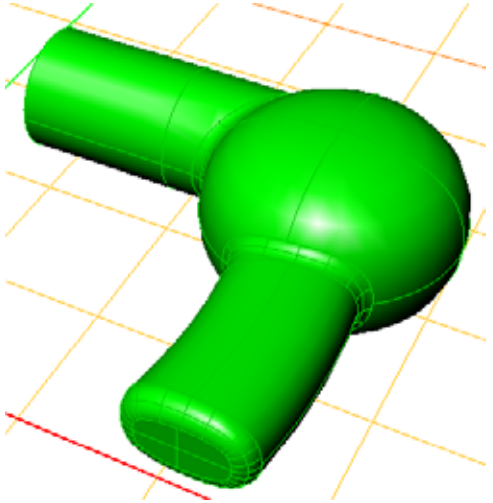
32) To shade the model:

Have **the Perspective viewport active** (any view will render)

Pick the Shade icon on the top tool bar.



Result



33) Save the hair dryer model.

Name the model:

HD_ _ _ The spaces are for your initials i.e. HDSIM

Save it to your period directory

34) Export the model as a stereolithography file for the Modela software.

From a command prompt, **pick the hair dryer** so that it is highlighted.

File – Export Selected…

Name the model:

HD_ _ _ The spaces are for your initials i.e. HDSIM

Save it to your period directory

35) Exit Rhino.

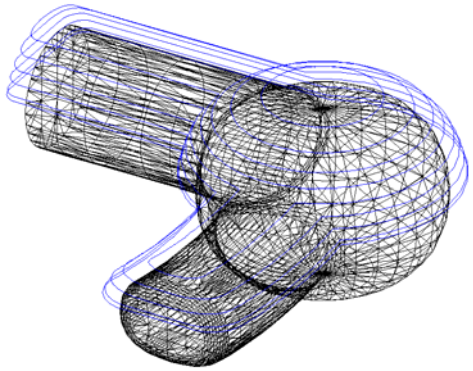
In the next lesson you will learn to create the tool paths and machine the hair dryer.

7

Modela Player 4

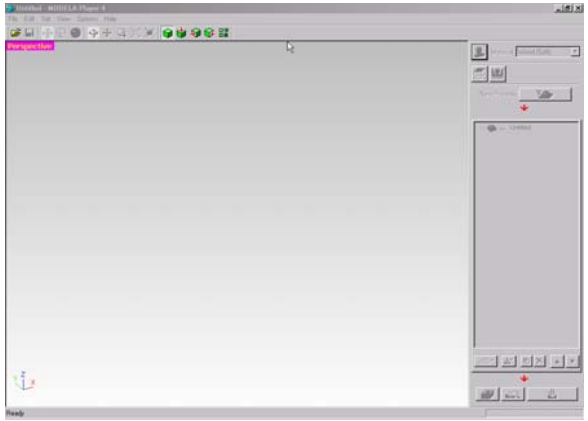
MDX-40

Hair Dryer
Modela Player 4



1) Start the Modela Player 4

Pick the Modela Player 4 icon from the desktop.



Modela Player 4 will open……..

Below is a flow chart of the process of machining the model with the Modela Player 4 and your Roland milling machine. Be sure to notice the 1st and 2nd numbering. These need to be done in order to properly machine your work and protect your milling machine. The Roughing toolpath, which cuts away the majority of the material, must be done before the Finishing toolpath.

Modela Player 4

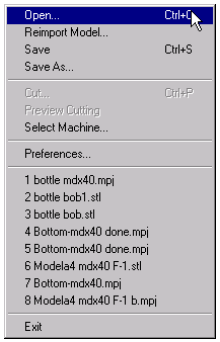
1st … Roughing tool path

2nd … Finishing tool path

Processes:	Details:
Model Button 	>Set finished size. Set the Origin.
Cutting Surfaces Button 	>Select number of surfaces to machine.
Modeling Form Button 	>Set the depth and the cutting area
New Process Button 	>Create tool paths: One Roughing & One Finishing
Preview Button 	>Create a tool path simulation & cutting time est.
Cut Button 	>Sends the file to the MDX40 and begins cutting.

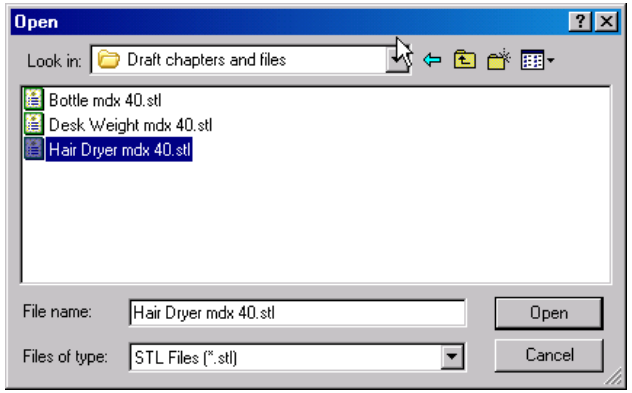
2) To open the hair dryer stl file:

File – Open –



Pick File from the pull down menu in the top left of the screen and

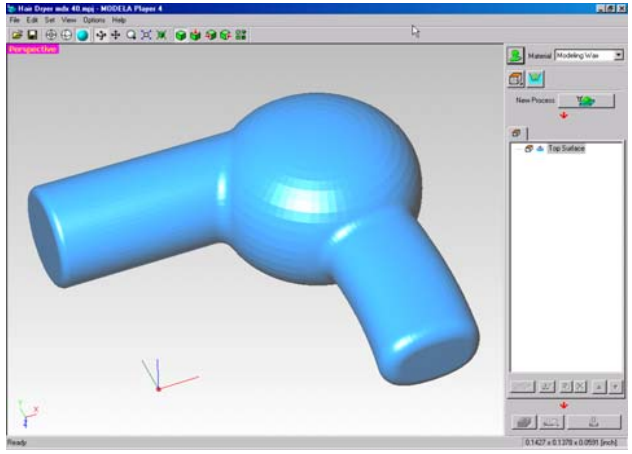
Select Open…



Select the file **Hair Dryer mdx 40.stl**

and **pick Open**

Result



Imported model quality:

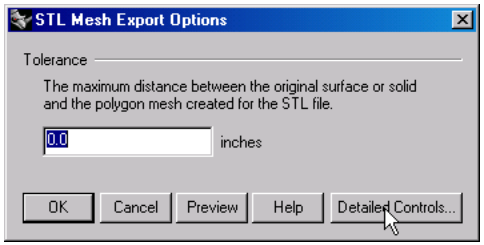
If the quality of the imported model is not fine enough for you, go back to Rhino and export the stl file again.

Open the model in Rhino.

Select the model and go to **File, Export Selected, File Type, stl**

Adjust the STL Mest Export Options as follows:

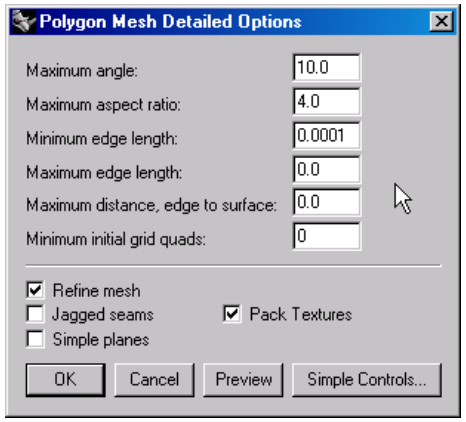
Click Detailed Controls:



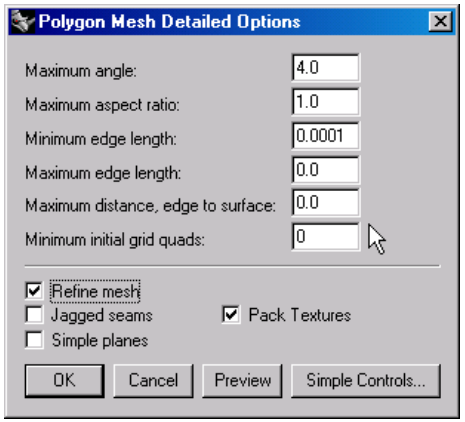
Change the settings as follows:

Export the model again as an stl file.
Experiment with the Polygon Mesh options
to get the desired results when opened in Modela 4.

Preview before selecting OK. The finer, more exact the mesh,
The larger the stl file will be.



Medium quality mesh

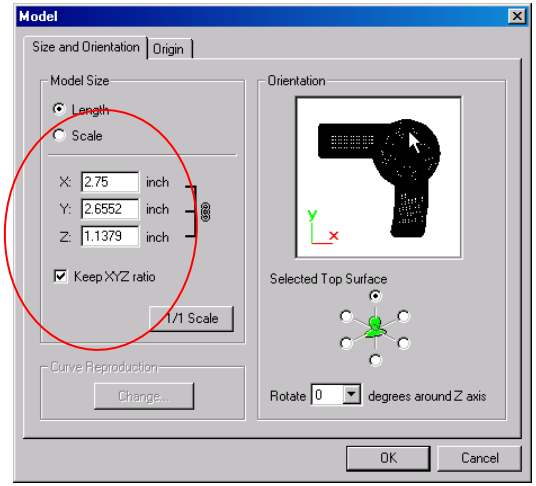


Fine, more exact mesh

3) You also have to take into consideration the orientation (direction) of the part and the final size of the part. The mill is going to cut the part from the top of the screen down. For the hair dryer, the orientation does NOT need changing. The model was created with the side that will be machined facing up. For this reason we do not need to change the orientation of the model.

But, we will need to set the finished size of the machined part.

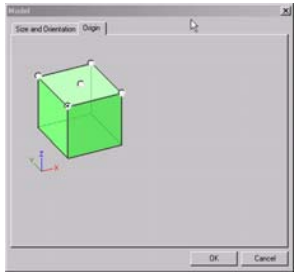
Pick the Model button.



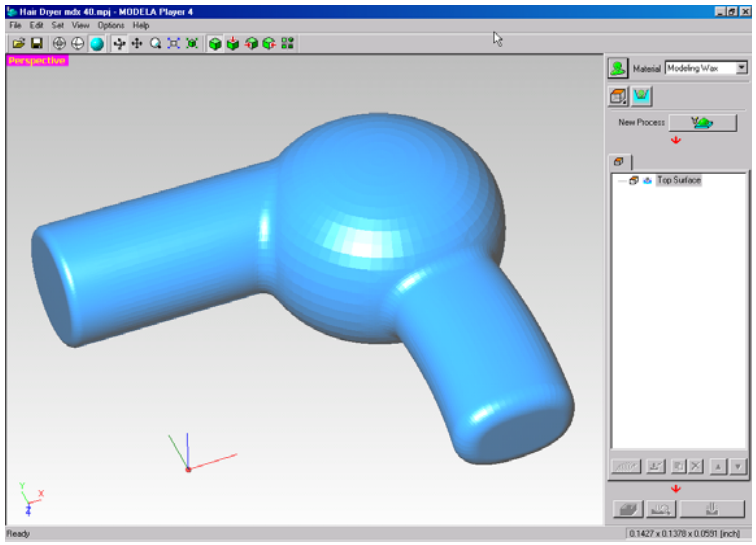
Set the finished size of the model. 2.75 x 2.65 x 1.14, which will easily fit our wax block.

Select the "TOP Surface" radio button, as displayed because we do Not need to rotate the model to orient it.

The origin button displays the xyz origin to be used for cutting the model.



Result



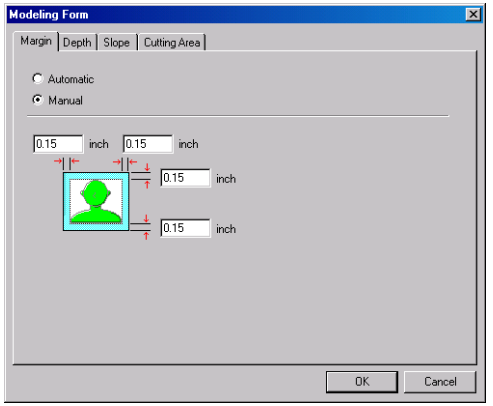
Set the number of surfaces to be cut. **Select one surface** with this button:

The Model Form Button:

4) The **Margin, depth, and cutting area** need to be set.

Select the Margin Tab.

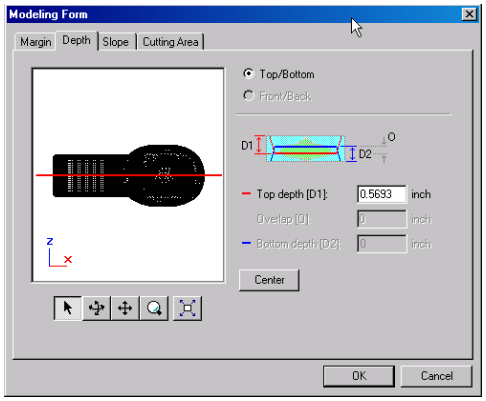
Set the margin to .15” on all edges of the model. This will cause the cutter to cut all the way to and past the edges of the block. ……..



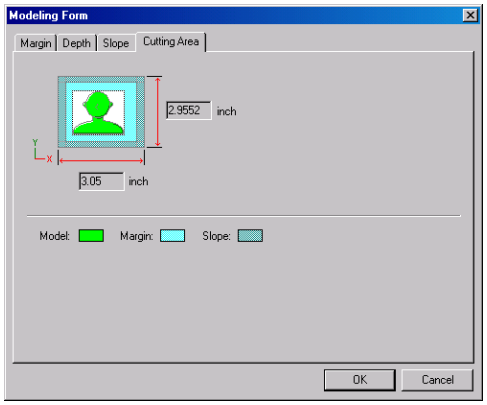
Select the Depth tab.

For most symmetrical parts the depth should be set at Center. Since this is a symmetrical part, picking Center will move the cutting depth to the middle of the part. Cutting any deeper into a shape, such as a sphere, will give straight sides after the center point.

Select the Center button.



Select the Cutting Area tab. Notice that the cutting area is .3 (.15 +.15) larger than the original model size. This is because of the margin we added to the model.



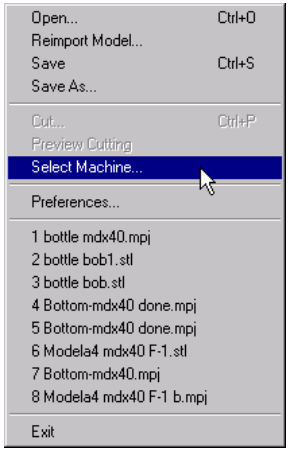
5) The Roland MDX-40 that you will be using, needs to be set as the current machine, if it is not already.

Always check these steps every time before you use the machine, even if you think they are set properly.

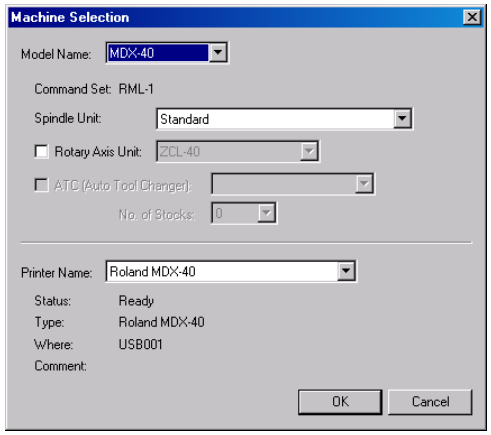
If necessary, change the machine type:

Pick the File pull down menu and

Select Machine…

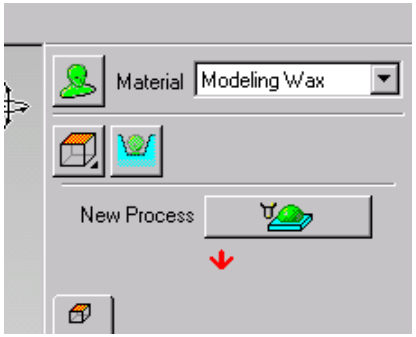


Scroll through the list and **select** the MDX-40.



Pick OK.

6) The material to be machined needs to be checked and/or set.



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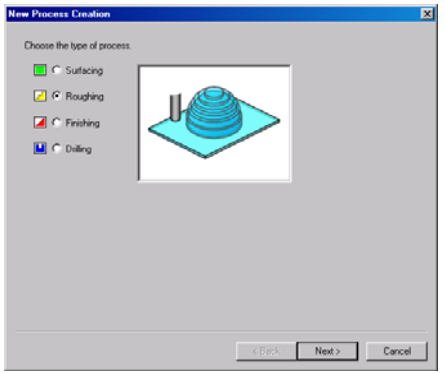
7) We will cut the part in a two step process. The processes are often called tool paths. **Roughing** takes away the bulk of the material and **Finishing** which gives the part a finished size and creates a smooth finish. There are additional tool paths available, but we will concentrate on Roughing and Finishing in this series.

Roughing must be cut first followed by Finishing. Finishing without a roughing cut first can **damage** the milling machine.

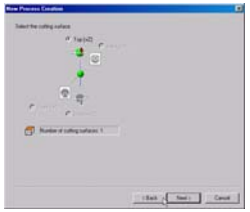
To create the Roughing tool path:

Select the New process Button: 

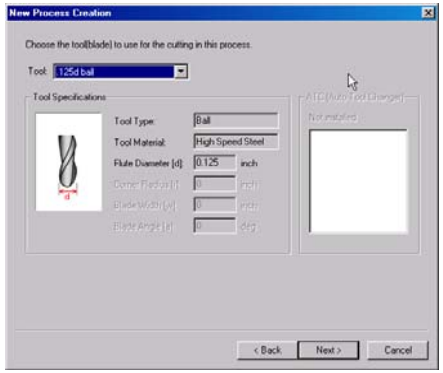
Select Roughing:



Because we have selected One side cutting, the Top is selected.



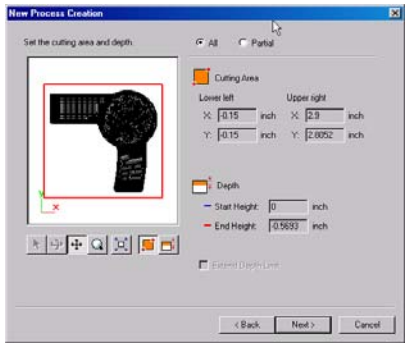
For all of the other parts in this series, we will be using a .125” (1/8”) ball mill.

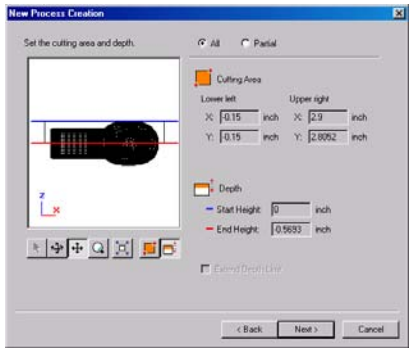


Select the .125d ball mill …….

Check the cutting area and depth …….
The ‘All’ buttons should display the correct settings.

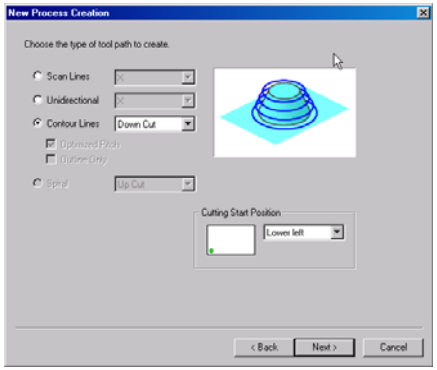
Notice the Margin in the X,Y view. This can be changed if desired by selecting the ‘Partial’ button and making desired changes.





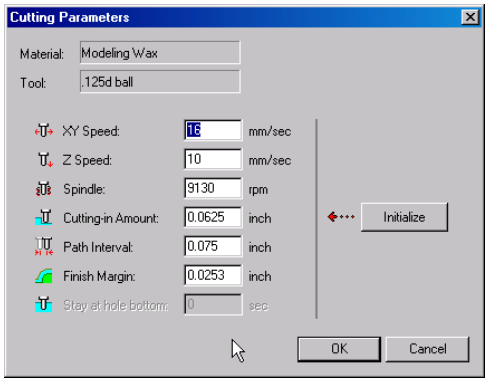
Roughing should be set to “Contour Lines Down Cut”. Contouring is a very efficient method of removing large amounts of material.

It cuts out the object in elevations.



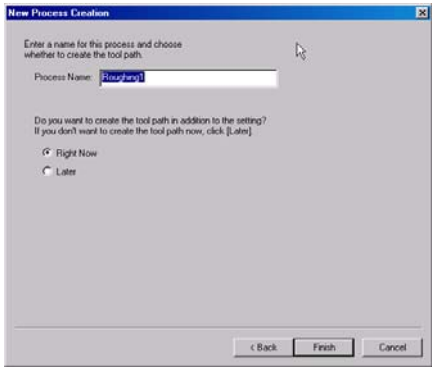
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

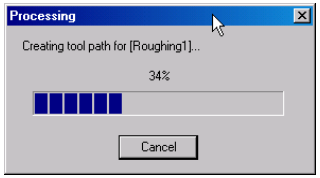


Process Name is listed as **Roughing 1**. It is a good idea to generate the tool path “Right Now” because you’ll be able to see the results before sending the tool path to the machine.

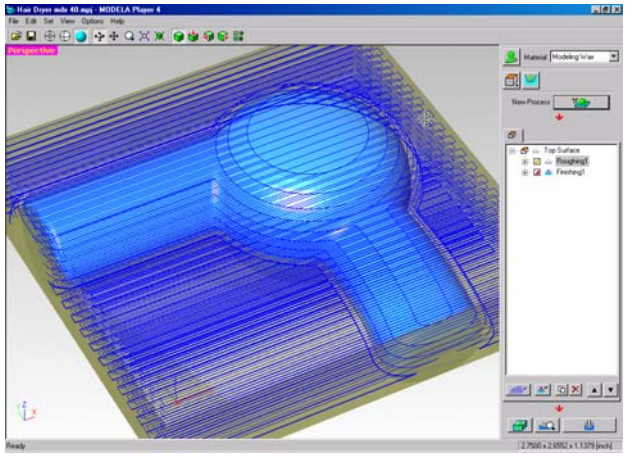
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.



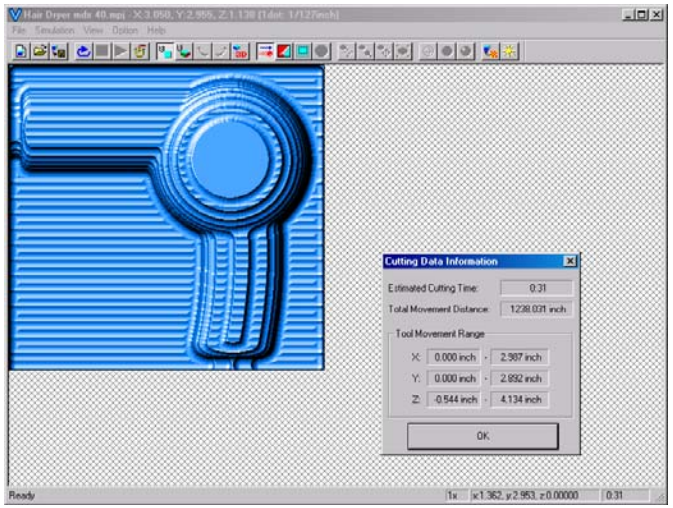
Roughing Tool Path.



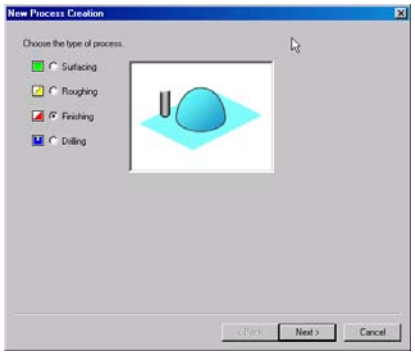
8) In Modela 4 there is a built-in tool path simulator. It provides an accurate representation of the tool path being cut and a time estimate for its completion.

After the tool path is processed, **press the Preview Cutting button**. 

Virtual Modela automatically starts which displays the simulation and shows the cutting time.



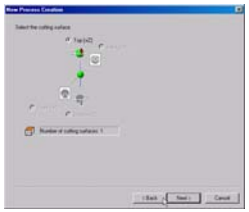
To create the Finishing tool path:



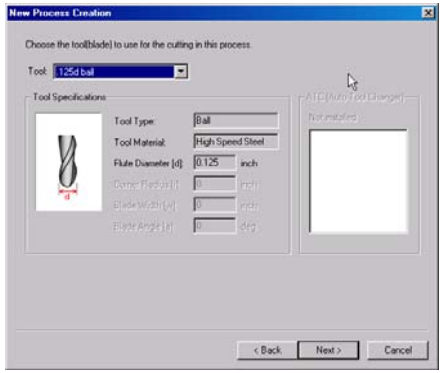
Select the New process Button: 

Select Finishing:

Because we have selected One-Side cutting, the Top is selected.

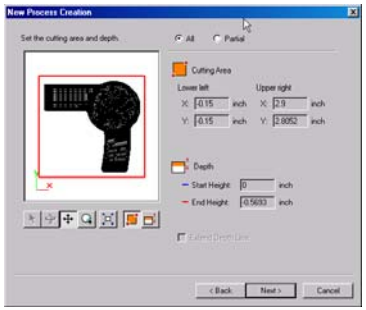


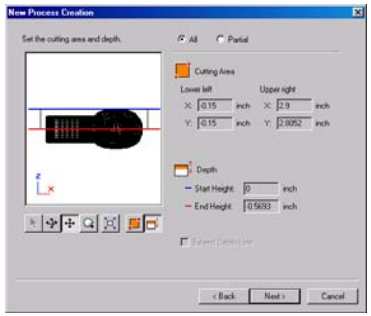
Select the .125d ball mill …….



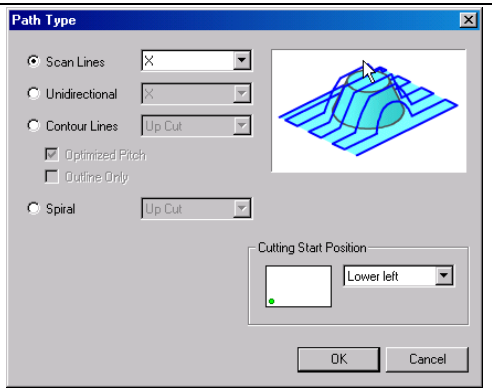
Check the cutting area and depth …….

The ‘All’ buttons should display the correct settings.





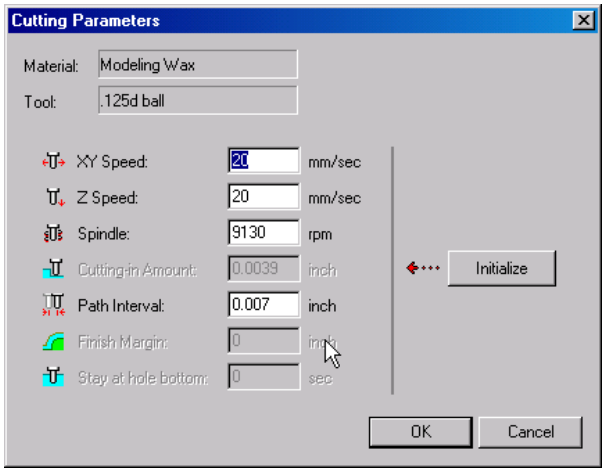
The **Finish** cutting direction can be set to cut in the X, Y, or both directions. These are lines cut along the chosen axis. Cutting both X and Y takes twice the time but gives a better finish.



Set the Finish cutting direction to Scan Lines X direction.

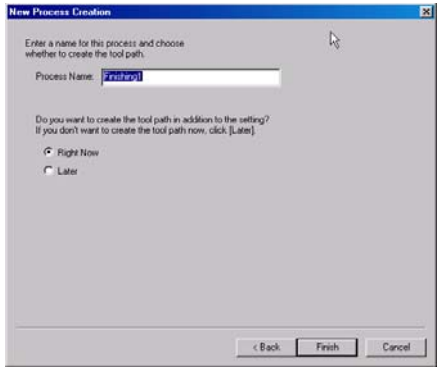
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

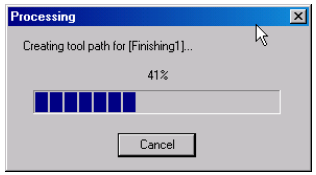


Process Name is listed as **Finishing 1**. It is a good idea to generate the tool path “Right Now” so you can see the results before sending the tool path to the machine.

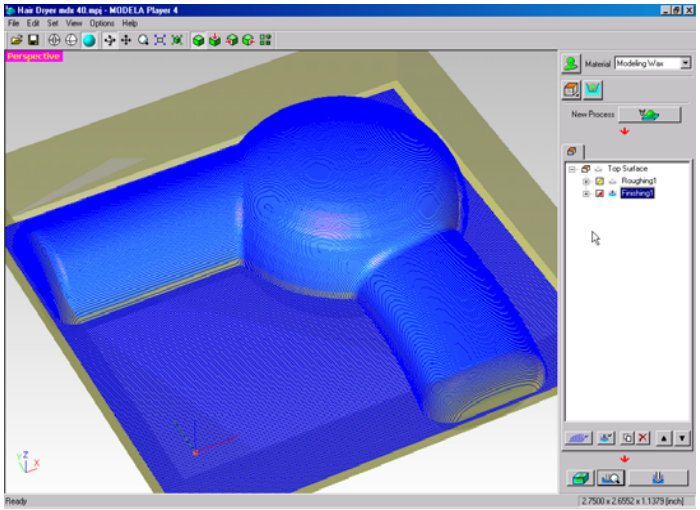
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.

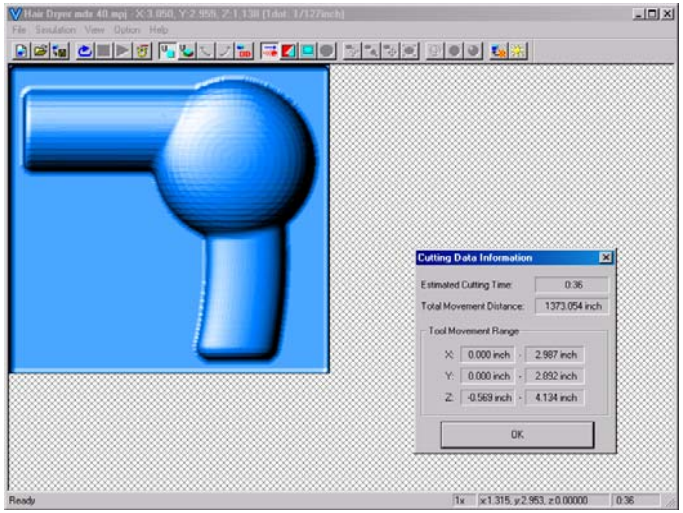


Finishing Tool Path.



9) In Modela Player 4 there is a built-in tool path simulator. It provides an accurate representation of the tool path being cut and a time estimate for its completion.

After the tool path is processed, **press the Preview Cutting button**. 

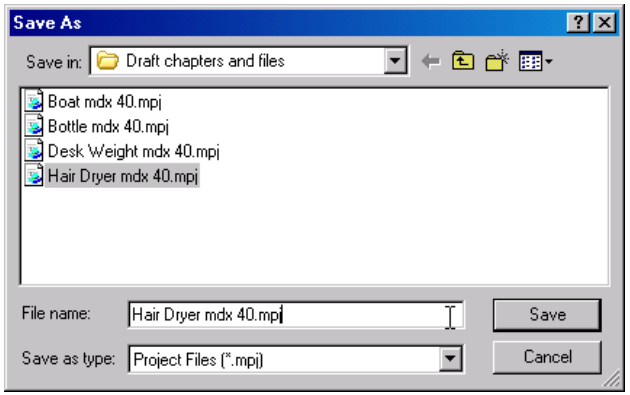


Virtual Modela automatically starts which displays the simulation and shows the cutting time.

10) To save the part as a Modela file:

Pick the File pull down menu and Save.

The file name is the same as the one that was originally imported. Only the file extension is changed to .mdj



This is often a good time to make a second save too. Simply change part of the file-name to create a second file.

Pick Save As. **Type in the name** . 'Hair Dryer mdx 40 done'
or another file name of your choice.

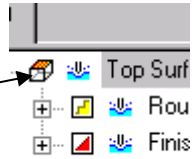
Result: There are now two files. In case one is damaged, there is a backup.

NOTE: If you do not have at least 70 minutes in the class period, do not continue to the milling steps. The average machining time for the Desk Weight is 66 minutes. Both tool paths need to be completed without turning off the milling machine.

Step 13 explains how to exit Modela Player 4 if you do not have enough time at this point. There are instructions, at the end of the lesson, on how to open a saved Modela file.

11) To machine the part on the MDX 40:
Highlight the Tool Path to be machined.

In this case, **Highlight both tool paths**

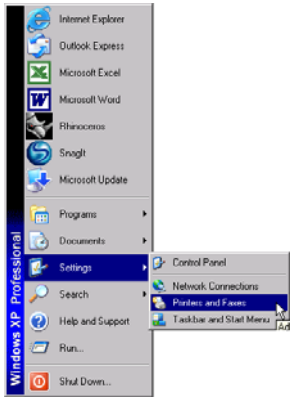


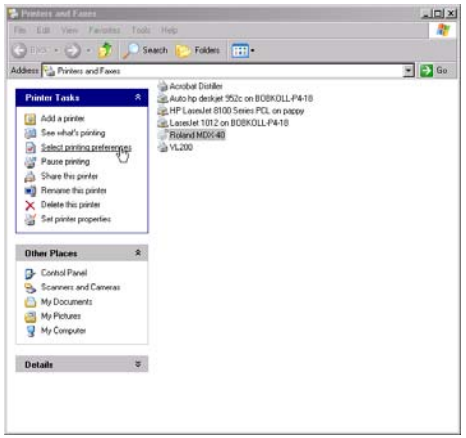
Select the Cutting/No Cutting button:

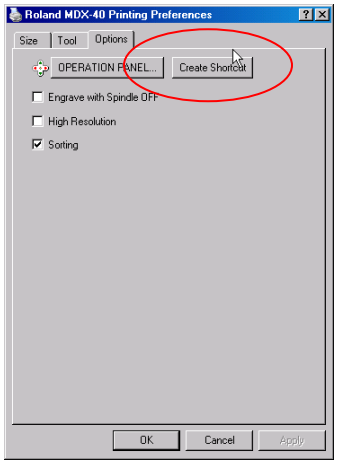
12) MDX-40 control panel:
Place a shortcut to the MDX 40 control panel on your desktop.

This gives you access to setting the cutter X, Y, Z origin. This step is Required before pressing the Cut button.

To create the shortcut, Select Create Shortcut in the printing preferences for the Roland mdx40:



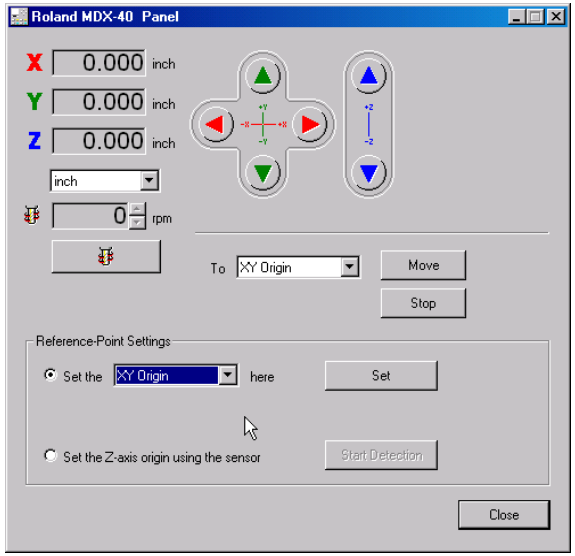




With the mouse and the arrows in the control panel, move the cutter to the proper location on the wax block.

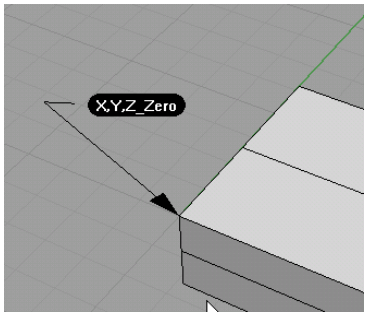
Set the Z origin, Set the Y origin, Set the X origin.

The MDX-40 ‘remembers’ the origin after it is set. If you have to restart your cutting it is easy to move the cutter to this location for re-starting if necessary.



One method to 'accurately' Set the X,Y,Z Origin at the corner of the wax stock block:

Set the X,Y,Z origin so it is at the left hand corner of the model……..

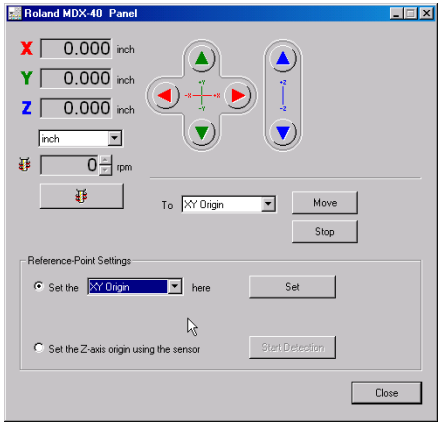


1 Set the Z origin at the top surface of the model by moving the cutter (with the mouse and the arrows in the control panel) so it touches the top of the block. Use a piece of notebook paper, approximately .003" thick. Move the cutter down until there is friction between the cutter and the block. The cutter at this point is .003 above the block. **Set Z origin here.** (temporary Z)

2 Set the X origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the left side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the X Origin here.** (temporary X)

3 Set the Y origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the front side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the Y Origin here.** (temporary)

IMPORTANT: The XYZ readouts read 0.000 inches.
You need to change the position of the tool so the XY center of the tool is exactly at the corner of the block. The XY is currently one-half the tool diameter (.0625) away from the correct origin. Also, remember that our tool is .003 above the block.



With the mouse and the arrows in the control panel:

z… Move the cutter so the Z reads: Z -.003" **Set Z origin here**
x… Move the cutter so the X reads: X .0625" **Set X origin here**
y… Move the cutter so the Y reads: Y .0625" **Set Y origin here**

Select the Cut button: Sends the tool path files to the MDX-40 and begins to cut the part.

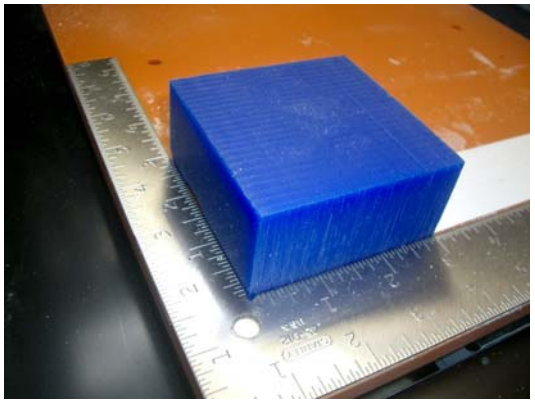
Remember: Highlight the Roughing and Finishing tool paths and select the Cut/NoCut button. This is button is used to control whether or not each tool path file will be cut by the the MDX40.

As your machine begins cutting:

To Pause: Press the View button on the front of the MDX 40. The machine will move the cutter to a safe Z height and move the XY platform for a closer view.
>>>>**Caution:** Do not open the cover until the machine stops moving<<<<
To Resume: The machine will begin cutting at the exact location where it was when the View button was pressed.

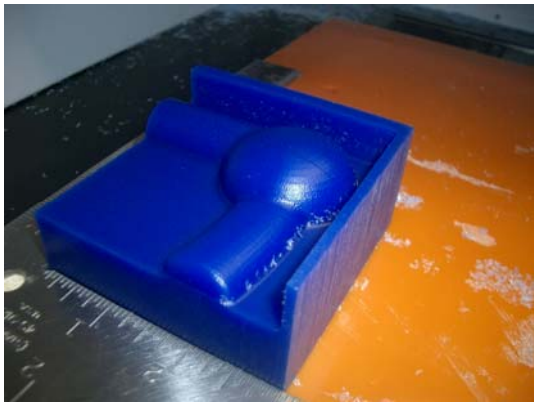
Do not leave the milling machine unattended when it is running!!!

Follow all standard safety procedures while operating the mill.



Wax block Ready to Cut

Machined Model



13) To exit the Modela Player 4:

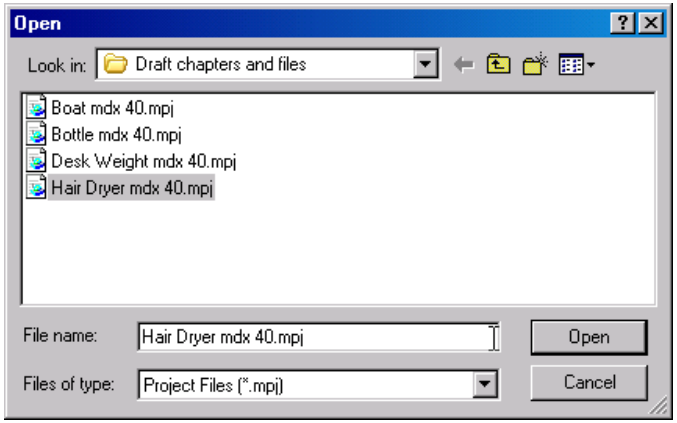
Pick the X in the upper top right corner of the Modela screen. When the Save dialogue box opens, **select Yes** to save the current file. The file retains the original file name except the extension changes to .mpj

To open a saved Modela Player 4 File:

With Modela Player 4 started:

Select the File pull down menu.

Select the File…… Hair Dryer mdx 40.mpj

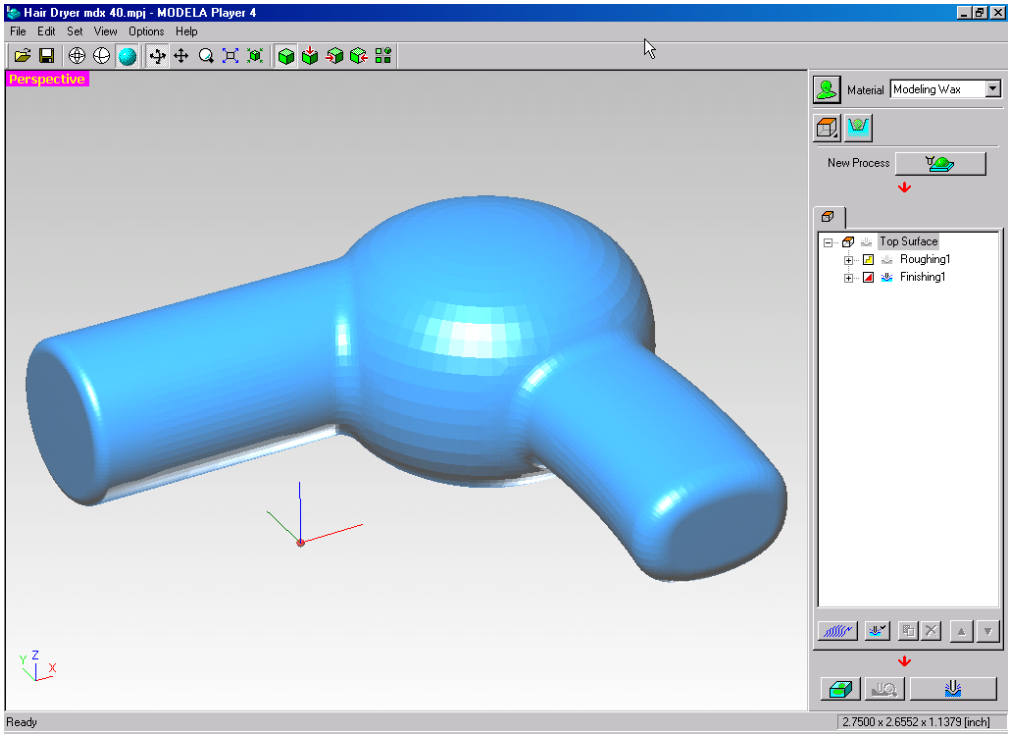


To easily recognize a Modela Player 4 file, look for the following icon In front of the file name:

Double clicking on this file name will also start Modela Player 4 and open the file.

Pick Open.

The file will open in Modela Player 4.



8

Boat - Rhinoceros

In this exercise you will be creating the geometry, and surfaces to model a rowboat.

It should be noted, that designing a boat is a complex process. The creation of this rowboat is very simplified as compared with truly designing a boat.

Creating the Line Geometry

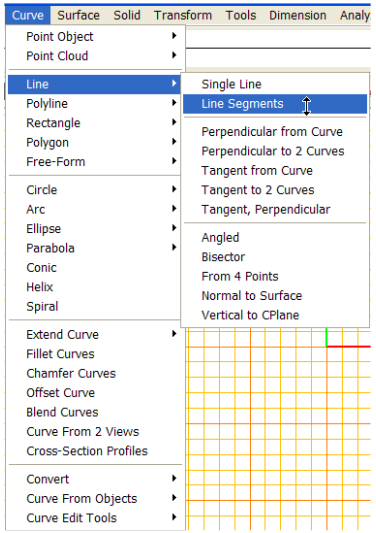
The geometry for the rowboat will be created in three general steps. The first will be a set of lines that will be used to construct the curves for the boat. After the curves, surfaces will be created to complete the boat.

1) Start Rhino

2) Start a new drawing using the template created for these lessons.

3) Check to see that the **current layer is Construction**.

4) The lines along the center (profile) of the rowboat will be created first.



In the FRONT Viewport:

Curve - **Line** - **Line Segments**

Start of line: **0,0.5 Enter**

End of line (Undo): **0 Enter**

End of line. Press Enter when done (Undo): **R2.5<0 Enter**

End of line. Press Enter when done (Close Undo): **R0.43<90 Enter**

End of line. Press Enter when done (Close Undo): **Enter**

5) To view the geometry more closely select:

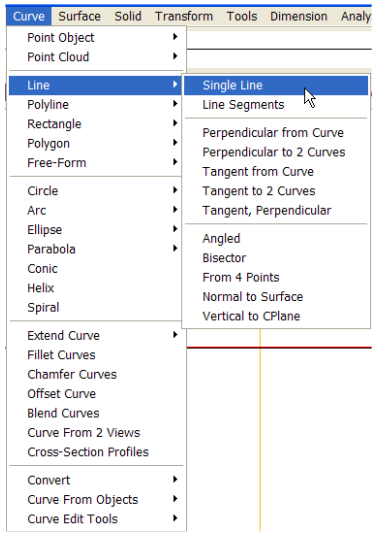
View - Zoom - Extents All

Result

6) One more vertical line needs to be added to help construct the middle cross section of the rowboat.

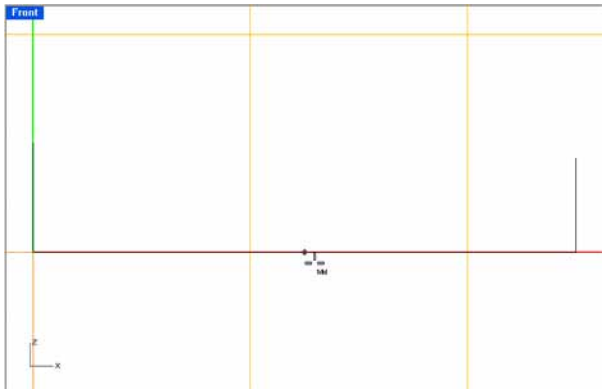
Turn on Mid Osnap only.

Curve – Line – Single Line



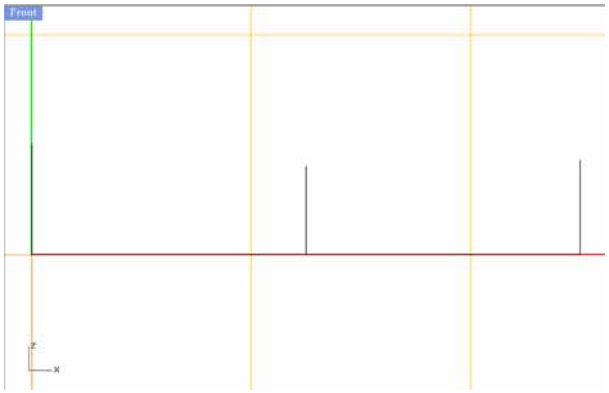
In the Front Viewport:

Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides):**Pick with Mid Osnap as displayed**



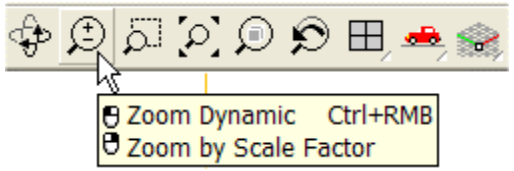
End of line (BothSides): **R0.4<90 Enter**

Result

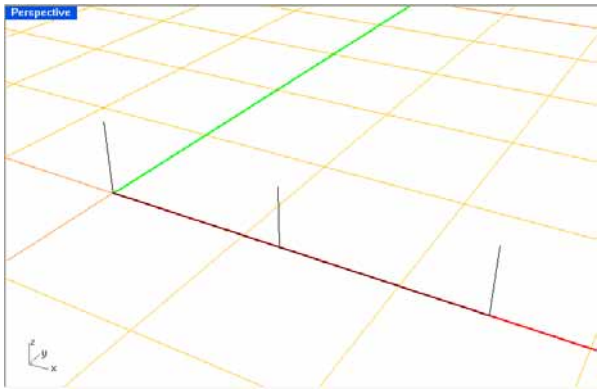


7) To define the width of the boat, horizontal lines need to be placed on the right two vertical lines.

Using Zoom Dynamic, fit all of the geometry, as displayed, in the Perspective view.



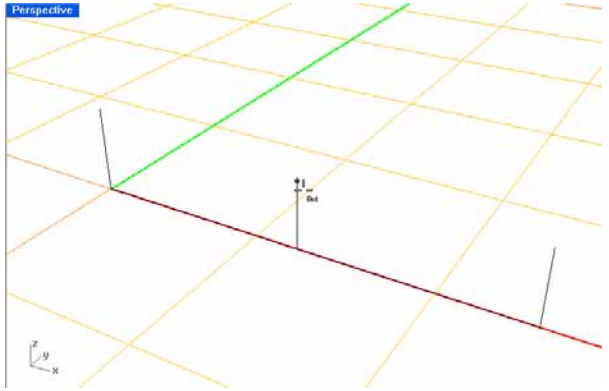
Right click on the icon, pick and hold in the Perspective viewport and drag down.



With the End Osnap on only:

Curve - Line – Single Line

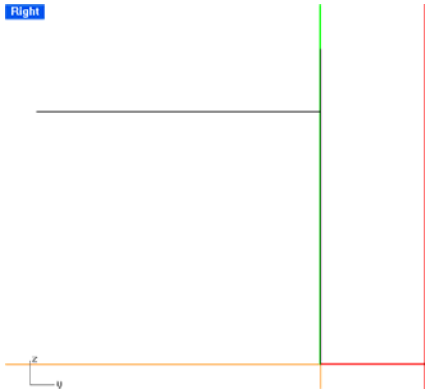
Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **Pick in the Perspective viewport as displayed**



Move the Right viewport:

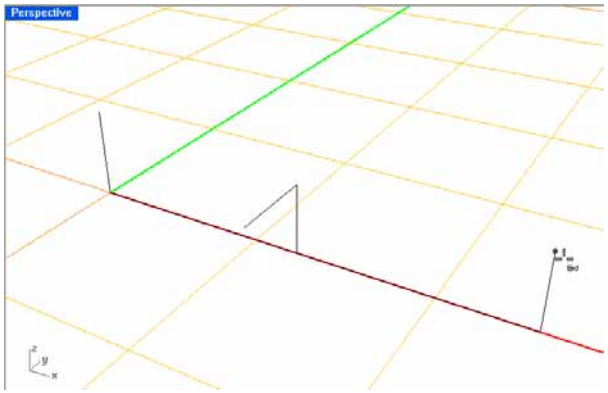
End of line (BothSides): **R0.45<180 Enter**

Result



Curve - Line – Single Line

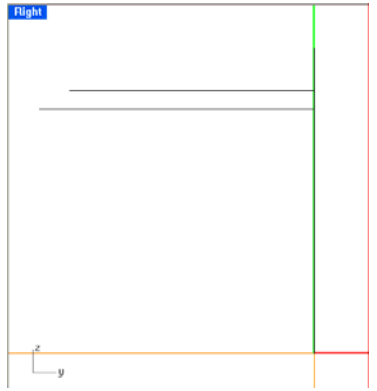
Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **Pick in the Perspective viewport as displayed**



Move the Right viewport:

End of line (BothSides): **R0.4<180 Enter**

Result

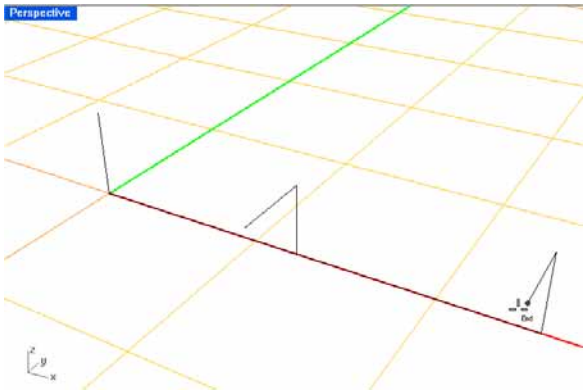


8) One more construction line needs to be added.

Turn on End Osnap only.

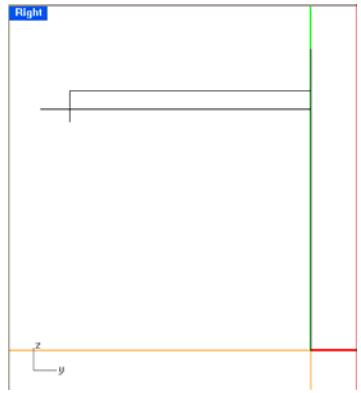
Curve – Line – Single Line

Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **Pick in the Perspective viewport as displayed**



Move the Right viewport:

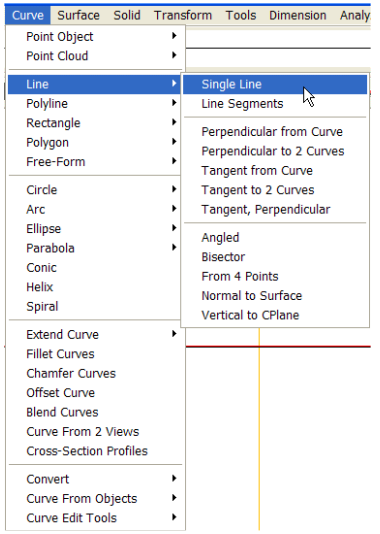
End of line (BothSides): **R0.05<270 Enter**



Result

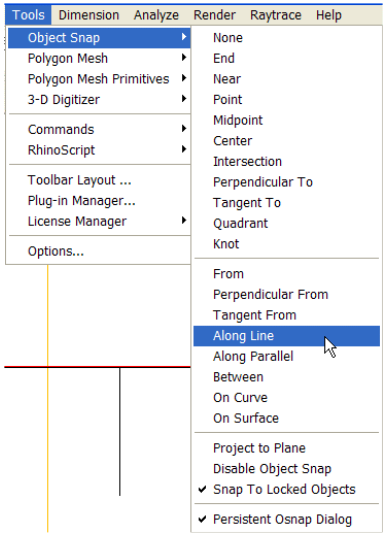
9) One more line needs to be added to define the bottom of the transom.

Curve – Line – Single Line

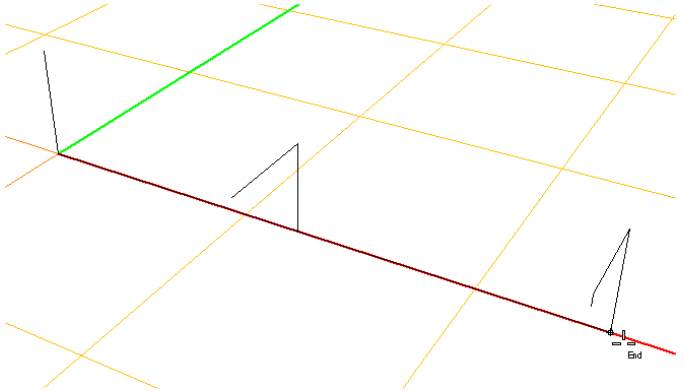


For this step we are going to use a custom Osnap selected from the Tools pull down menu.

Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **Tools – Osnap – Along Line**

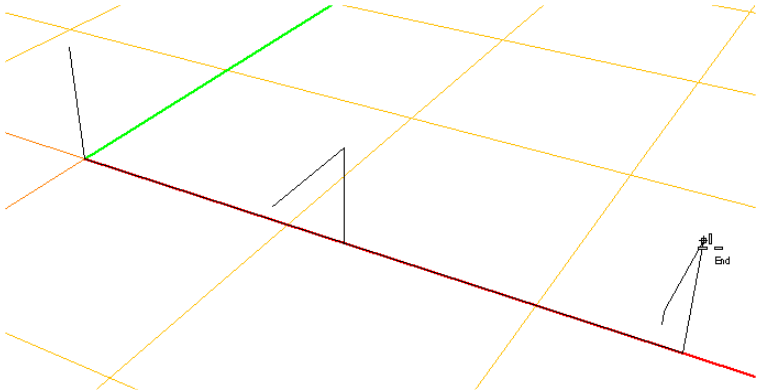


Start of tracking line: **Pick as displayed**



End of tracking line: **Pick as displayed**

Leave the cursor at the top of the line as this determines whether the next input goes up or down.

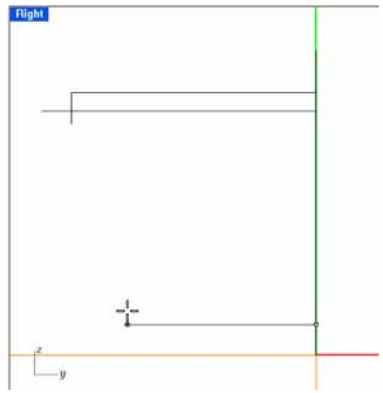


Start of line (Normal Angled Vertical FourPoint Bisector Perpendicular Tangent Extension BothSides): **0.05 Enter**
(This is the distance up from the first pick where the line will start)

Move the Right Viewport.

Turn Ortho on.

End of line (BothSides):**Pick as displayed**



Turn Ortho off.

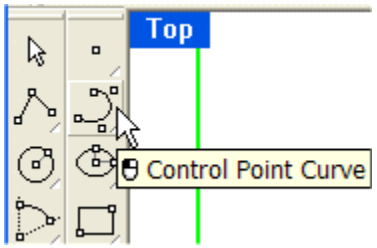
Creating the Curves

Note: If a curve created in the following steps does not come out like you want, just delete it and draw it over.

10) Change the Layer to Curves.

11) The stem for the bow (front) of the boat will be created first.

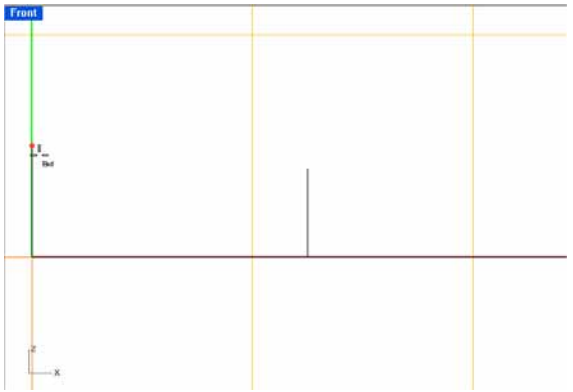
Curve – Free-Form - Control Points



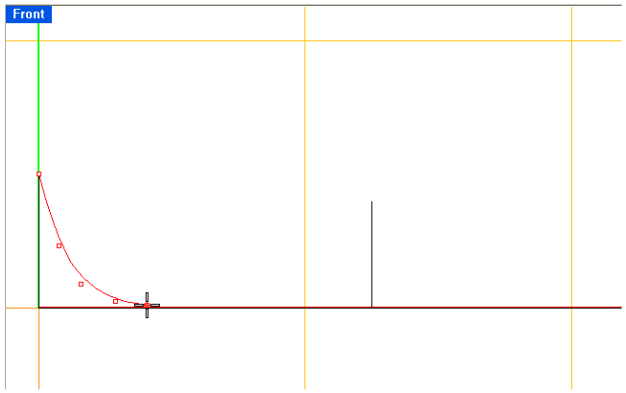
In the **Front Viewport:**

Turn on End Osnap only and be sure **Ortho is off**.

Pick as displayed.

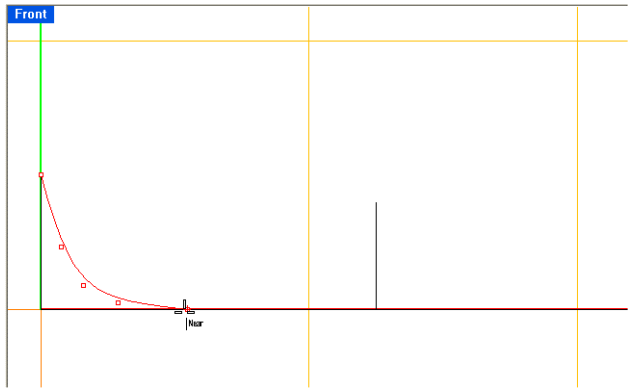


Make three picks as displayed.



Turn on Near Osnap.

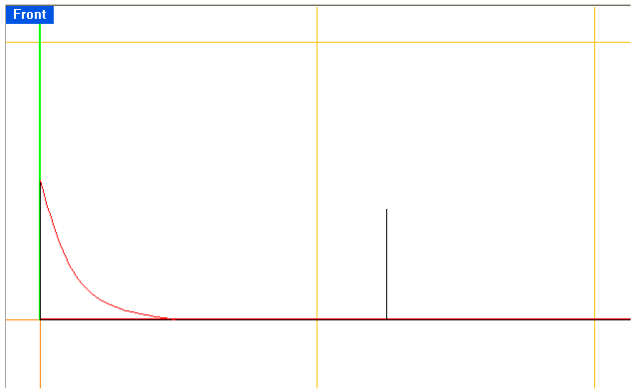
For the last pick use a **Near** Osnap to **pick** the bottom construction line.



Press Enter to complete the command.

Result

If your curve is not close to the one displayed, delete and repeat the step.



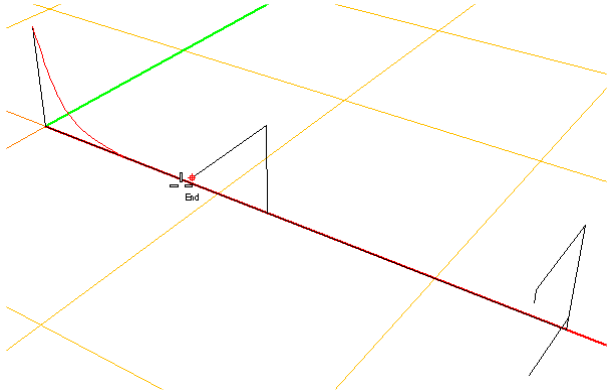
12) The mid body curve is next.

Make sure Planar is ON!!!!
(The Planar button at the bottom of the graphics screen)

Have End Osnap on only.

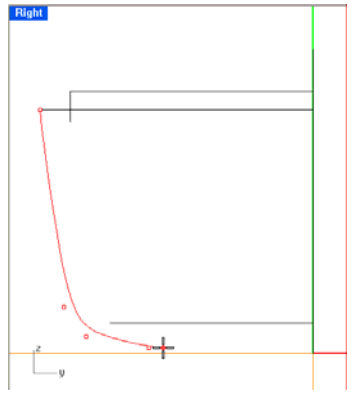
Curve – Free-Form - Control Points

In the **Perspective Viewport:**



Pick as displayed.

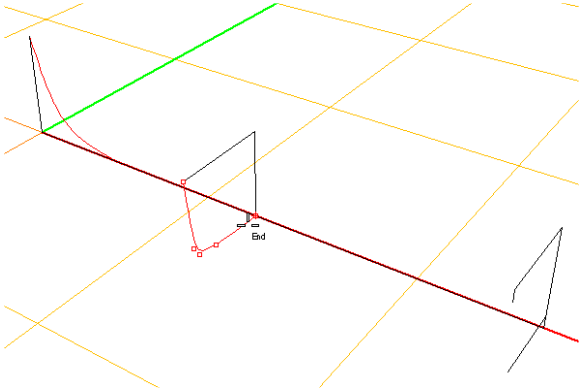
Move to the Right viewport.



Make the following three picks.

To make the final pick correctly it has to be done in the Perspective viewport or the osnap will grab the front end of the lower construction line.

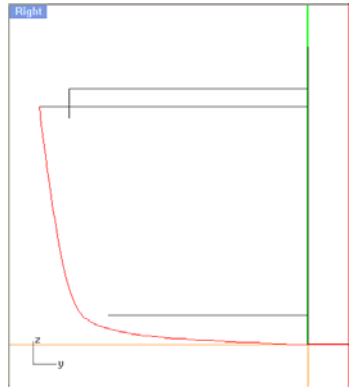
Move to the Perspective viewport.



Pick as displayed.

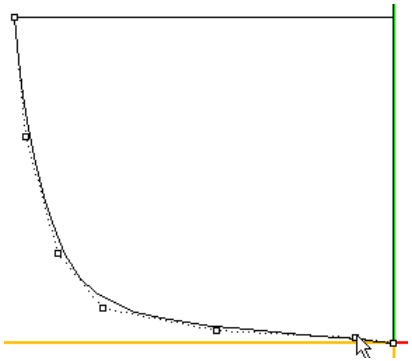
Press Enter to complete the command.

Result



13) The curve may need some editing to smooth out any bumps or dips. If your curve is smooth, skip this step.

There is a dip in the example curve right before it hits the vertical line.



To fix this:

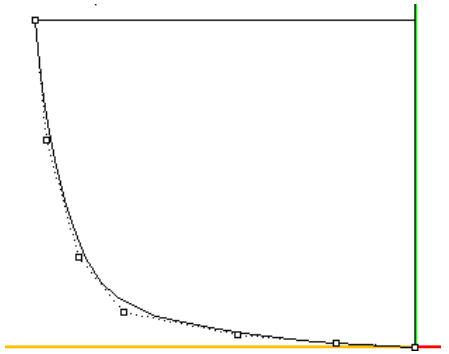
From a command prompt, **pick the curve**.

Edit - Point Editing - Control Points On

Control points will be displayed on the curve. The pointer is on the control point that needs to be edited in this example.

Pick on the Control Point, hold the mouse button down and drag the control point. Do this until the curve is fair (smooth). This may need to be done to more than one control point.

Note: Snap and Ortho should be off.



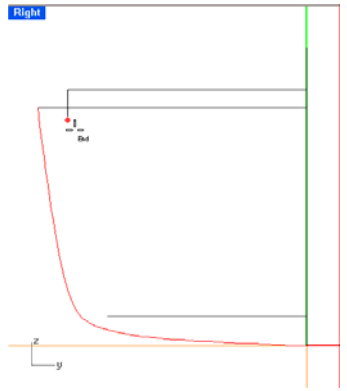
To turn the control points off:

Edit - Point Editing - Points Off

14) The transom curves (back of the boat) are the last to be constructed.

Curve – Free-Form - Control Points

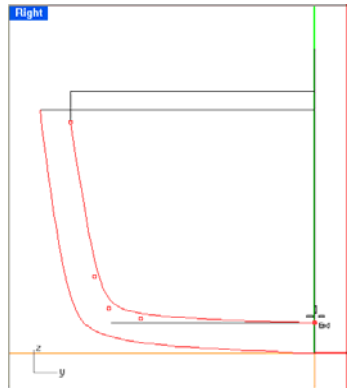
In the **Right Viewport:**



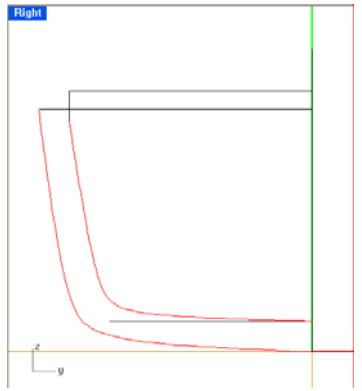
With Ortho off and End Osnap on only:
Pick as displayed.

Disable Osnap for the next **three picks**.

Turn on Osnap and **make the last pick.**



Result



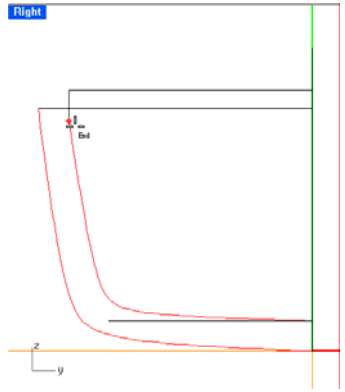
Redraw or Edit the curve, with controls points, if necessary, as in step 10.

15) The curve for the top of the transom is the next to be created.

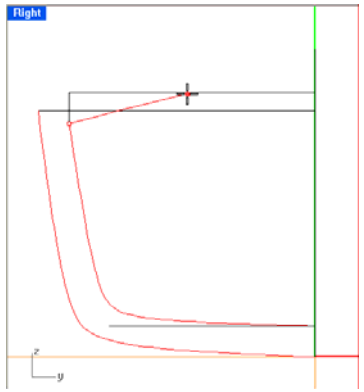
Curve – Free-Form - Control Points

In the **Right Viewport:**

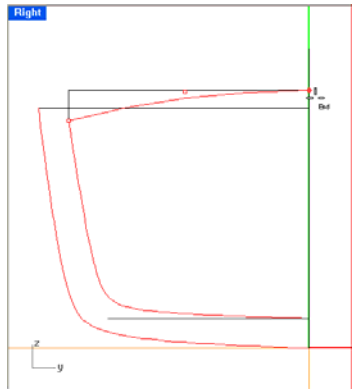
With an **end Osnap, pick** as displayed.



Pick as displayed.

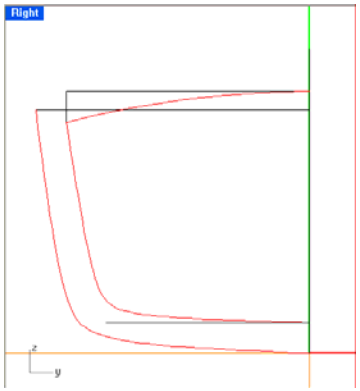


Pick as displayed.



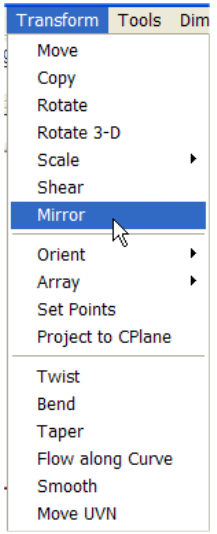
Press Enter to end the command.

Result

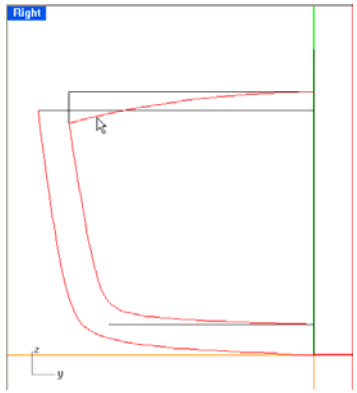


16) The two transom curves need to be mirrored so the transom surface can be created.

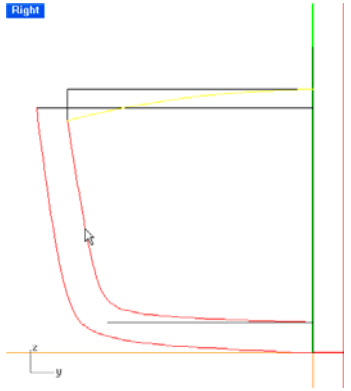
Transform - Mirror



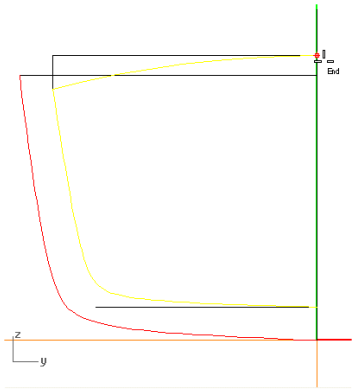
Select objects to mirror: **Pick as displayed**



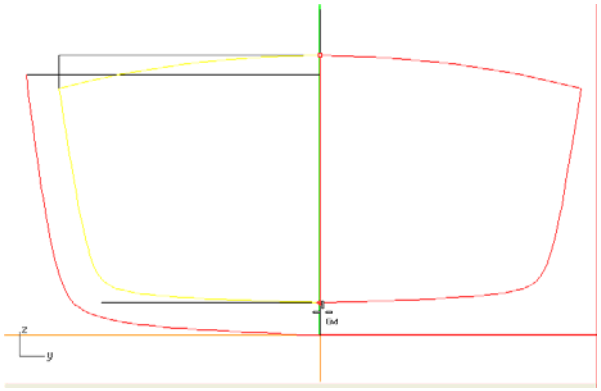
Select objects to mirror. Press Enter when done: **Pick as displayed**



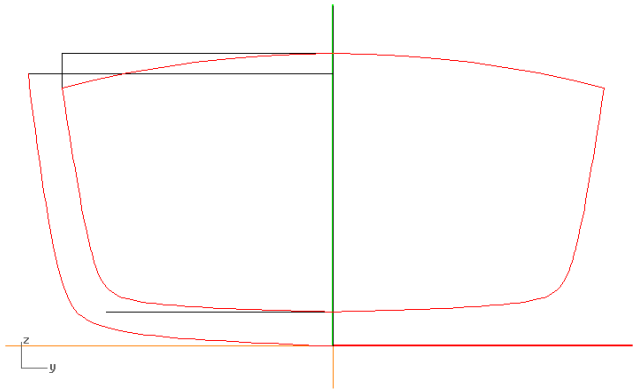
Start of mirror plane (Copy=Yes): **Pick as displayed**



End of mirror plane (Copy=Yes): **Pick as displayed**



Result



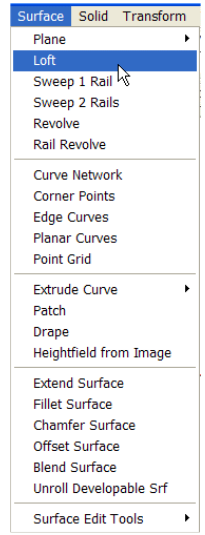
The curves for the boat are now complete.

Creating the Surfaces

Creating the surfaces for the boat will complete the project.

17) Set the Layer to Surfaces and **turn off the Construction layer**.

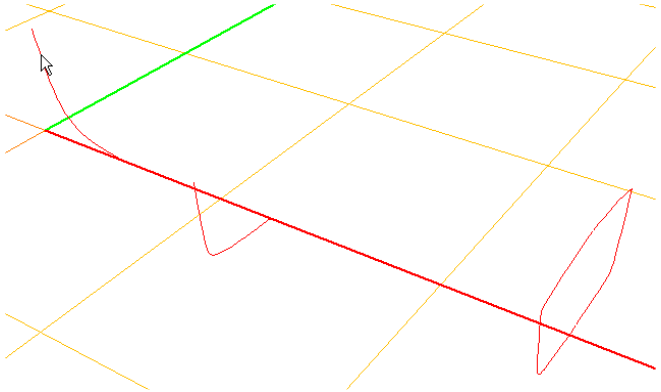
18) To create the surface for one side of the boat:



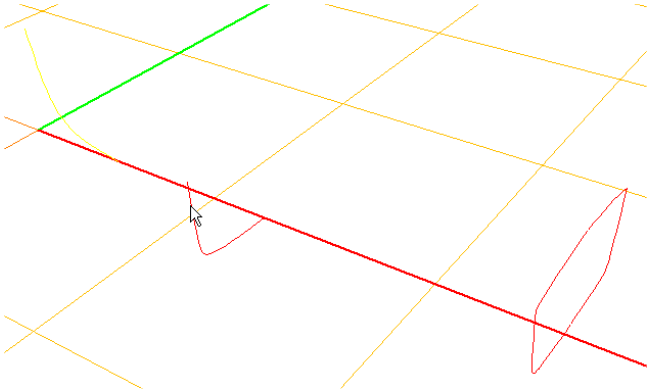
Surface – Loft

When selecting loft curves you must pick on the same half of the curve or the surface will be twisted and will look like a bow tie.

Select curves to loft (Point): **Pick as displayed**

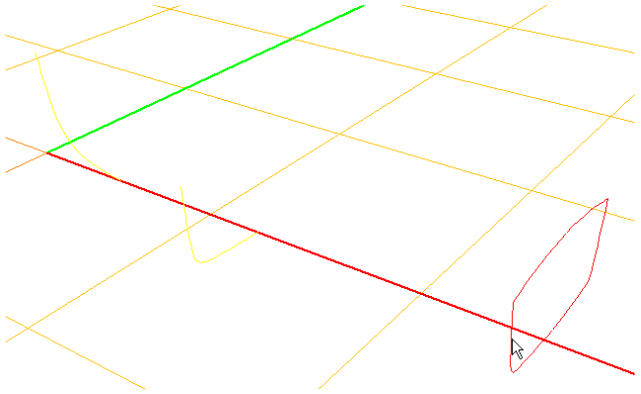


Select curves to loft. Press Enter when done (Point): **Pick as displayed**

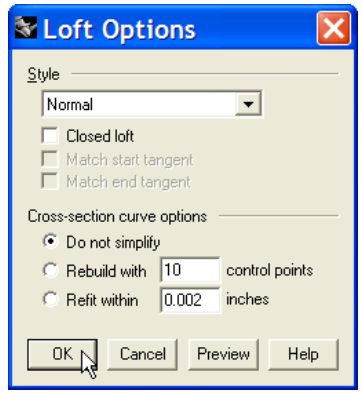


Select curves to loft. Press Enter when done (Point): **Pick as displayed**

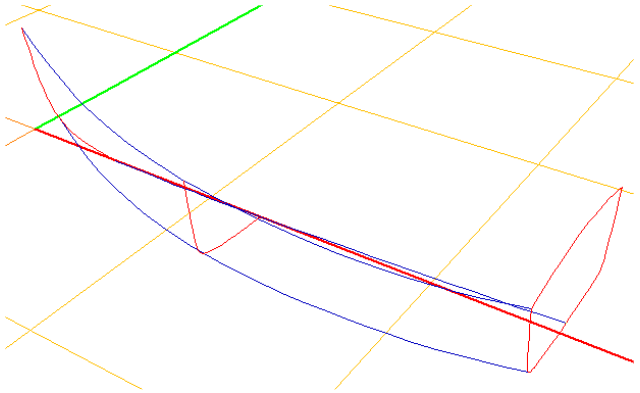
Select curves to loft. Press Enter when done (Point): **Enter**



Set the dialogue box as follows and **pick OK**.



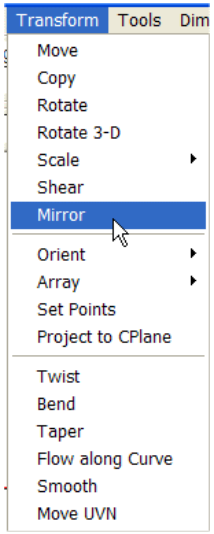
Result



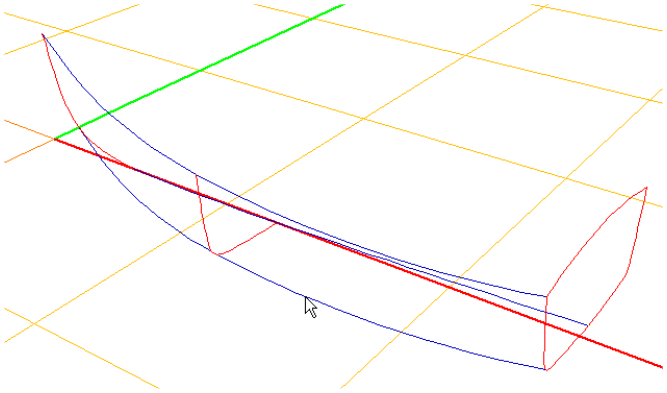
If the surface does not look correct, delete and repeat the step.

19) The surface can be mirrored to create the other half of the hull.

Transform – Mirror

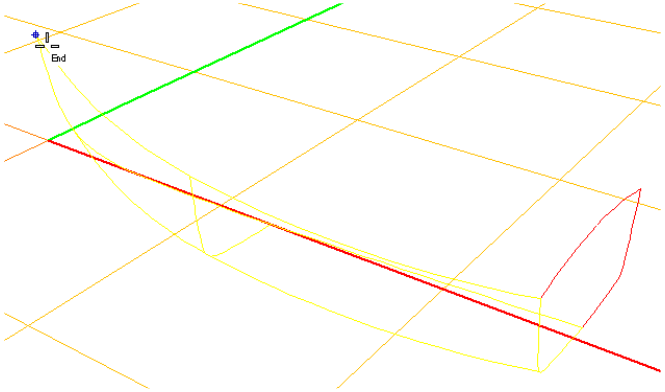


Select objects to mirror: **Pick as displayed**



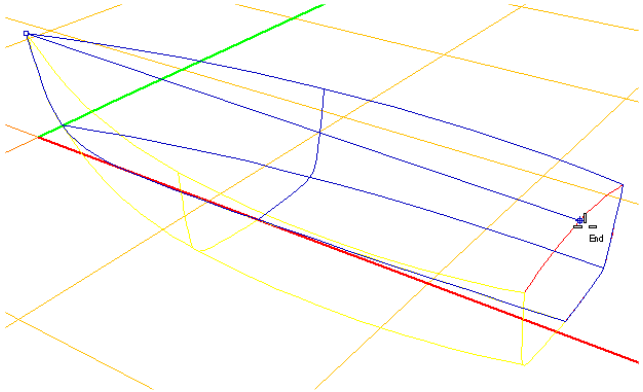
Select objects to mirror. Press Enter when done: **Enter**

Start of mirror plane (Copy=Yes): **Pick as displayed**

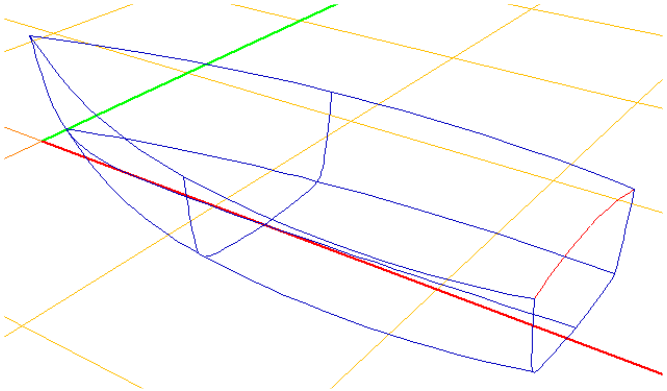


Have End Osnap on:

End of mirror plane (Copy=Yes): **Pick as displayed**

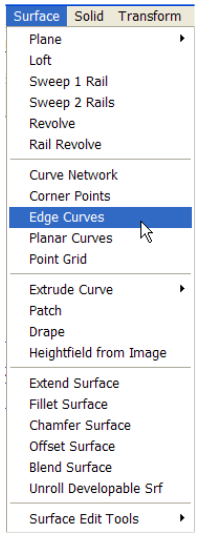


Result

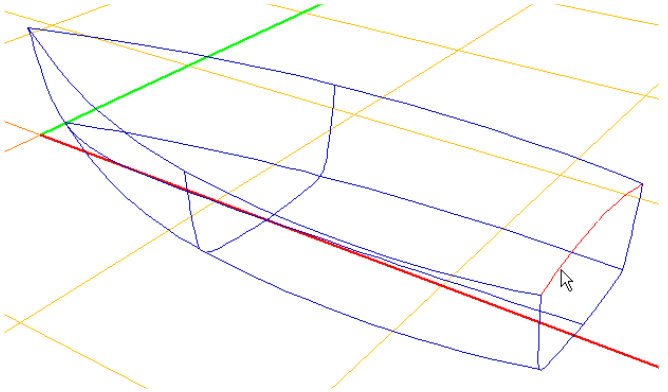


20) Creating the transom surface is the next step.

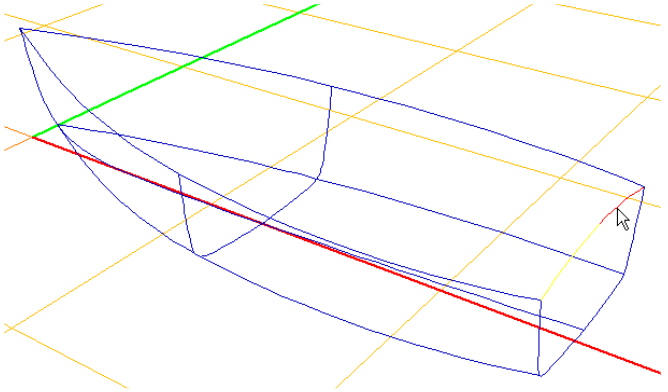
Surface – Edge Curves



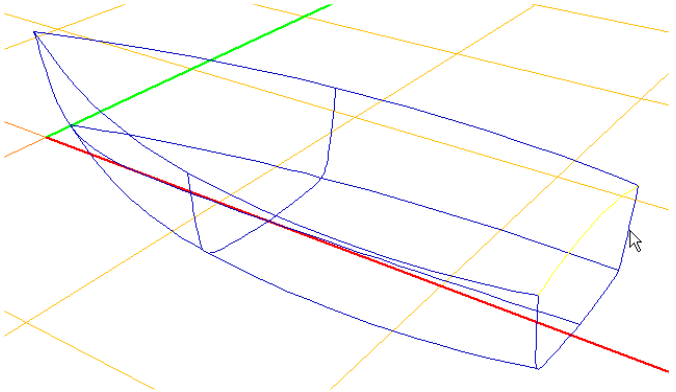
Pick as displayed



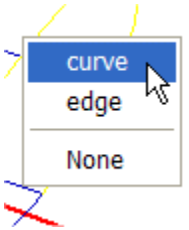
Pick as displayed



Pick as displayed

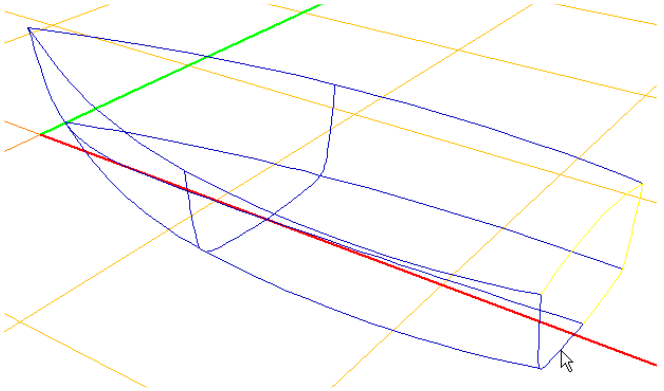


Because there is a surface edge and a curve line on top of each curve there are two possible selections.

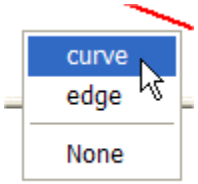


Pick Curve from the dialogue box.

Pick as displayed

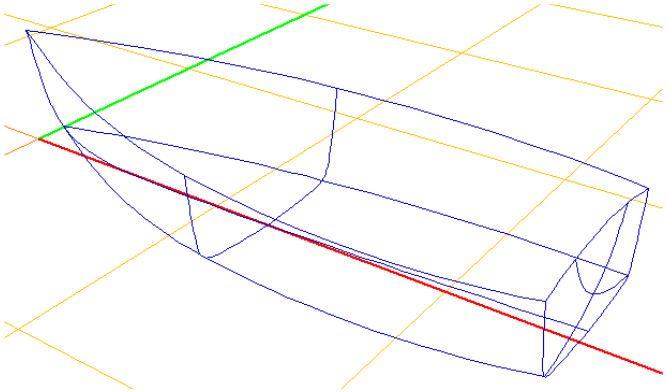


Because there is a surface edge and a curve line on top of each there are two possible selections.



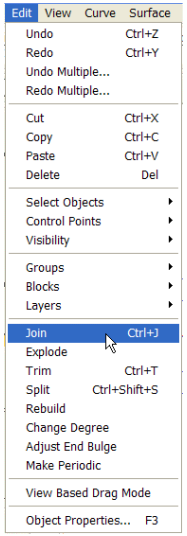
Pick Curve from the dialogue box.

Result



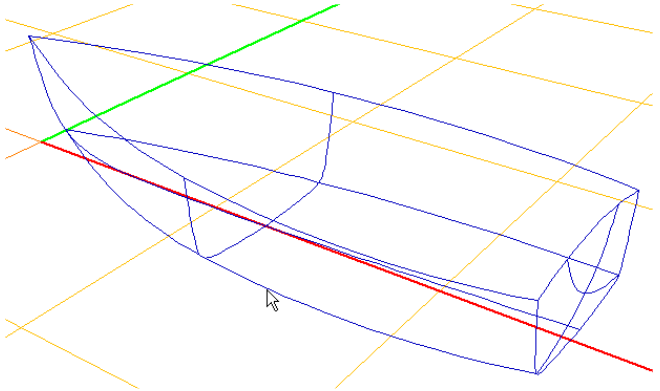
21) Turn off the Curves layer.

22) It will easier to export the surfaces it they are joined together.

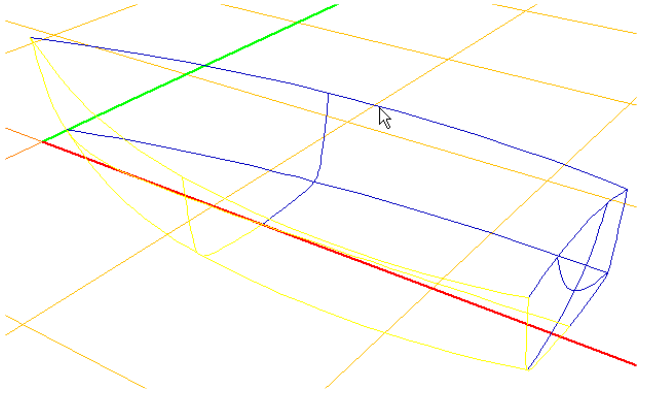


Edit – Join

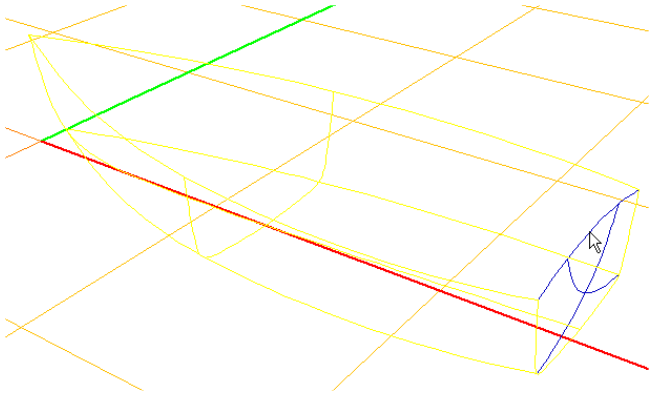
Select object for join: **Pick as displayed**



Select surface or polysurface to join: **Pick as displayed**



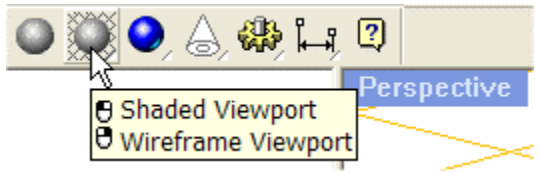
Select surface or polysurface to join: **Pick as displayed**



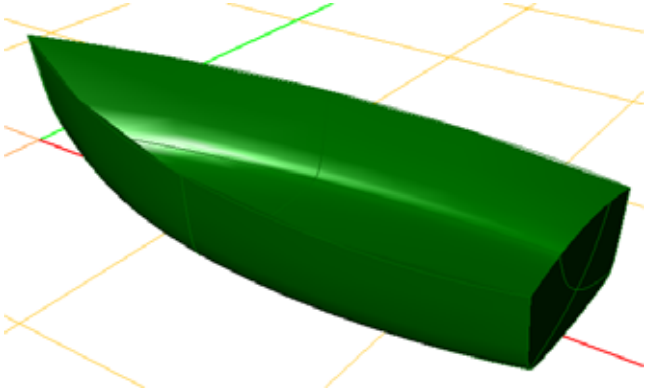
Select surface or polysurface to join: **Enter**
(**You will see the following messages displayed**:)
Joining surfaces.
Surface join in progress... Press Esc to cancel
3 surfaces or polysurfaces joined into 1.

23) To see the boat shaded:

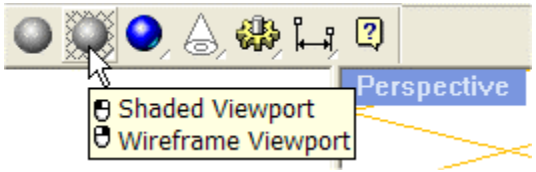
With the Perspective viewport active, **pick as displayed** on the Shade icon from the top toolbar.



Result



Right click on the Shade icon to remove the shading.



24) Save the boat to your period directory.

Call the file:

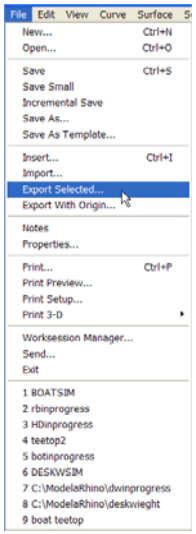
BOAT_ _ _ The spaces are for your initials: i.e. BOATSIM

25) The next step is to export the file for the Modela software**.**

Because the boat is not a completely enclosed model it cannot be exported as a stereo-lithography file. Instead it will be exported as a Drawing Exchange File (dxf), which can export open surfaces.

Pick the boat in the Top viewport.

File – Export Selected…

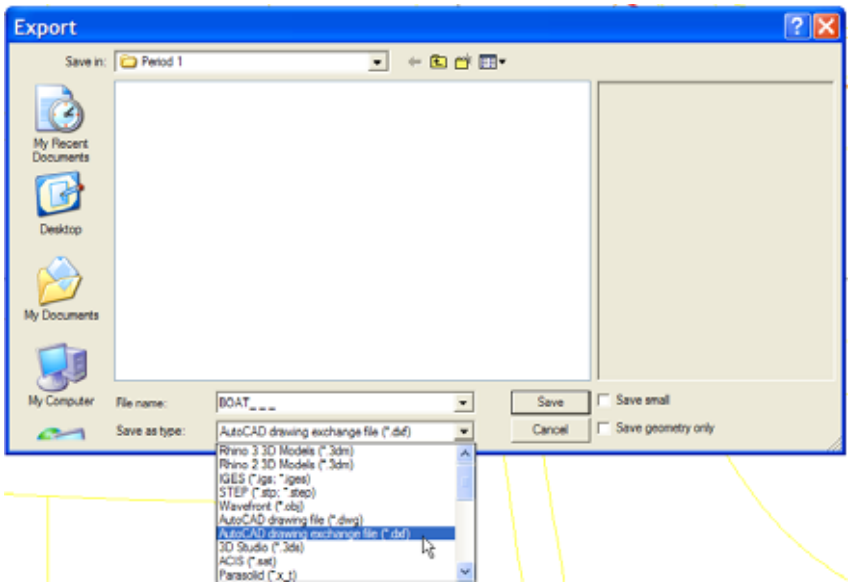


For the File name input:

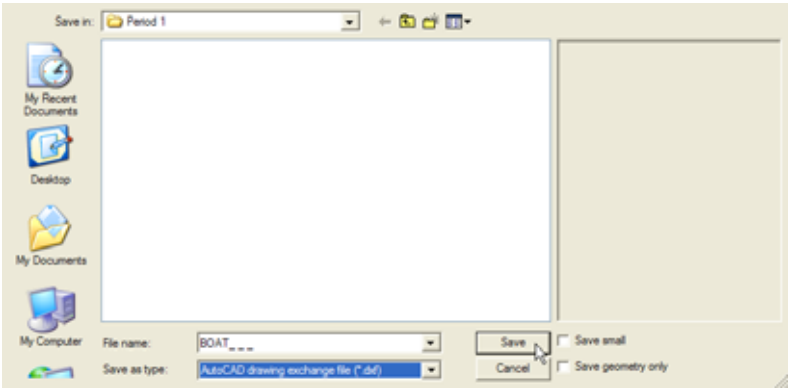
BOAT_ _ _ The spaces are for your initials
i.e. BOATSIM

Set the “Save as Type” to:

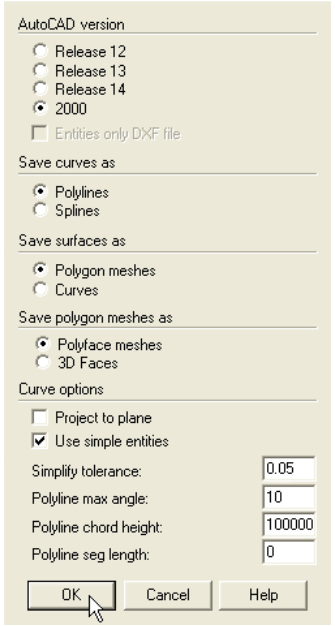
AutoCAD drawing exchange file (*.dxf)



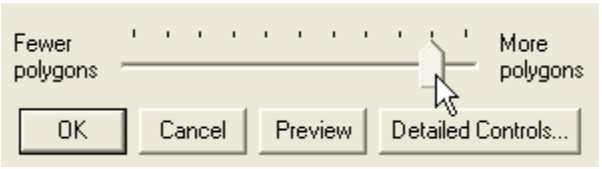
Pick Save as displayed.



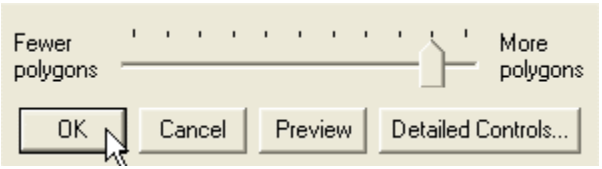
Set the dialogue box as displayed and **pick OK**.



Move the slider as displayed by picking, holding and dragging.



Pick OK



26) Exit Rhino

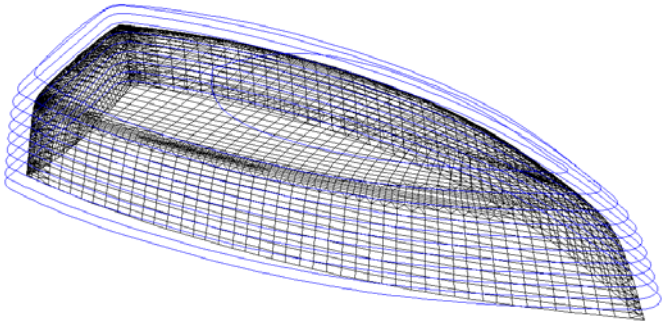
In the next lesson, you will learn to create the toolpaths for machining the boat.

9

Modela Player 4

MDX-40

Boat Modela Player 4

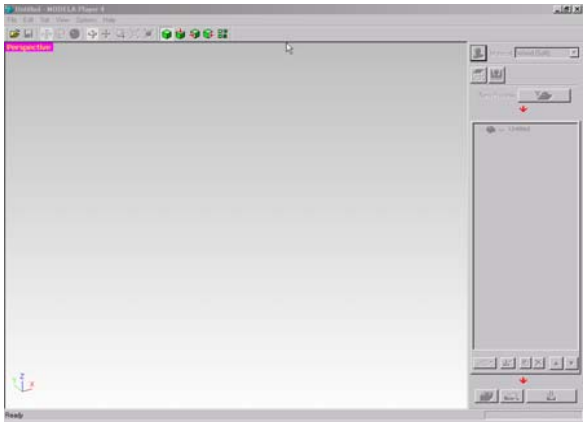


These steps will set up and machine the boat. The model consists of a polysurface consisting of 3 surfaces. It is not a closed object like the previous models. Export the model as a .dxf file. You can still export it as an .stl file but you must check the “export open objects” box when saving the file in Rhino. Total machining time for the boat is about 40 minutes.

1) Start the Modela Player 4

Pick the Modela Player 4 icon from the desktop.

Modela Player 4 will open……..



Below is a flow chart of the process of machining the model with the Modela Player 4 and your Roland milling machine. Be sure to notice the 1st and 2nd numbering. These need to be done in order to properly machine your work and protect your milling machine. The Roughing toolpath, which cuts away the majority of the material, must be done before the Finishing toolpath.

Modela Player 4

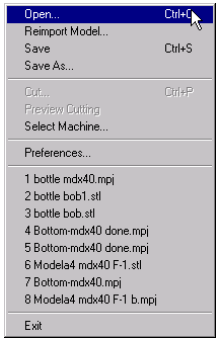
1st … Roughing tool path

2nd … Finishing tool path

Processes:	Details:
Model Button 	>Set finished size. Set the Origin.
Cutting Surfaces Button 	>Select number of surfaces to machine.
Modeling Form Button 	>Set the depth and the cutting area
New Process Button 	>Create tool paths: One Roughing & One Finishing
Preview Button 	>Create a tool path simulation & cutting time est.
Cut Button 	>Sends the file to the MDX40 and begins cutting.

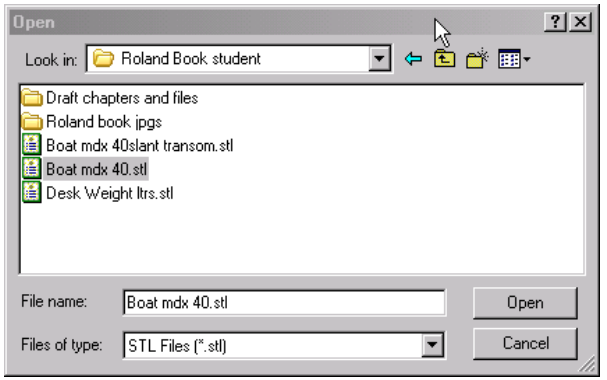
2) To open the boat dxf file:

File – Open –



Pick File from the pull down menu in the top left of the screen and

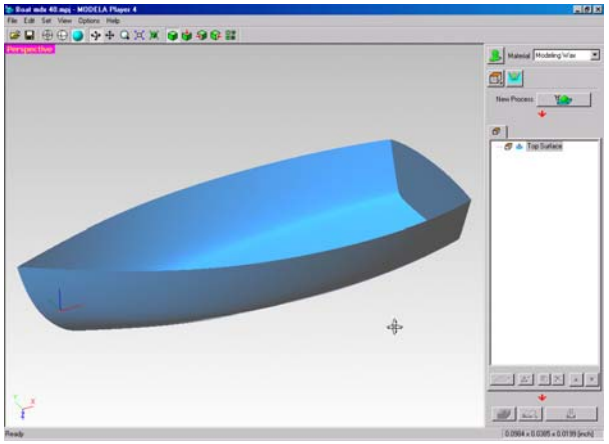
Select Open…



Select the file **Boat mdx 40.dxf**

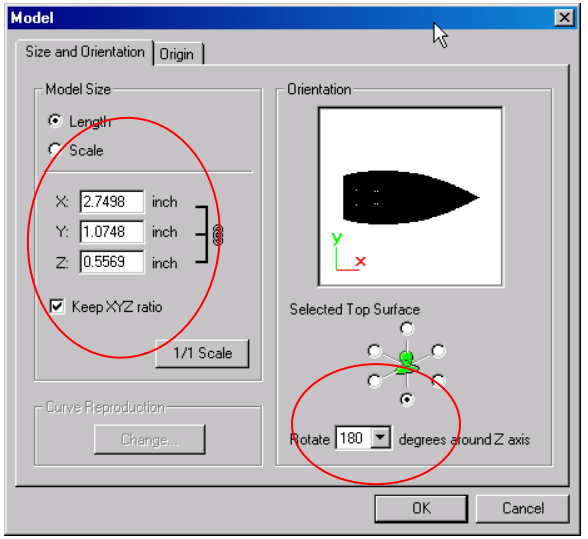
and **pick Open**

Result



3) You also have to take into consideration the orientation (direction) of the part and the final size of the part. The mill is going to cut the part from the top of the screen down. For the Boat, the orientation needs changing. The model was created with the side that will be machined facing down. For this reason we need to change the orientation of the model. Turn the boat over. But, we will need to set the finished size of the machined part.

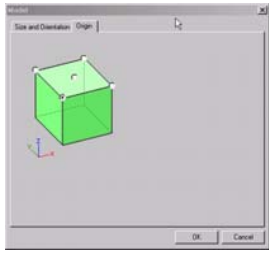
Pick the Model button.



Select the "Bottom Surface" radio button, as displayed because we need to invert the model to orient it so we are cutting the bottom of the boat.

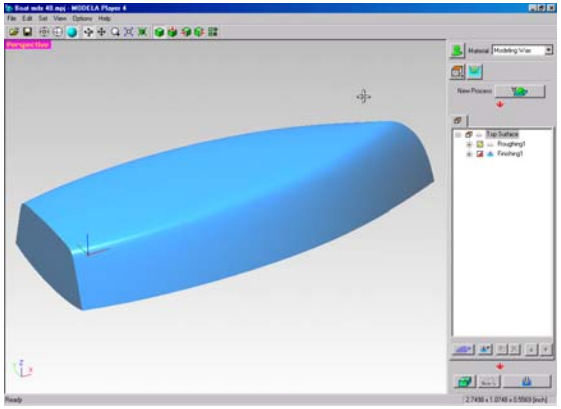
Rotate the model 180 degrees. Modeler's choice; not required.

Set the finished size of the model to 2.75 x 1.07 x .56, which will easily fit our wax block.



The origin button displays the xyz origin to be used for cutting the model.

Result



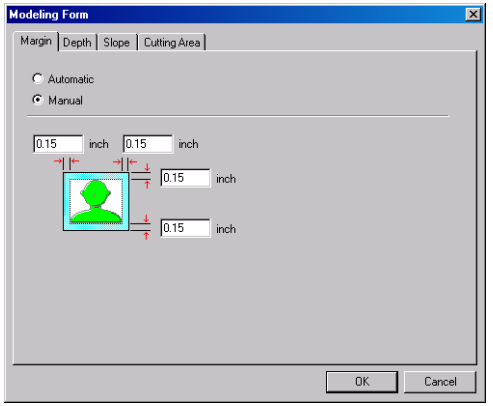
Set the number of surfaces to be cut. **Select one surface** with this button:

The Model Form Button:

4) The **Margin, depth, and cutting area** need to be set.

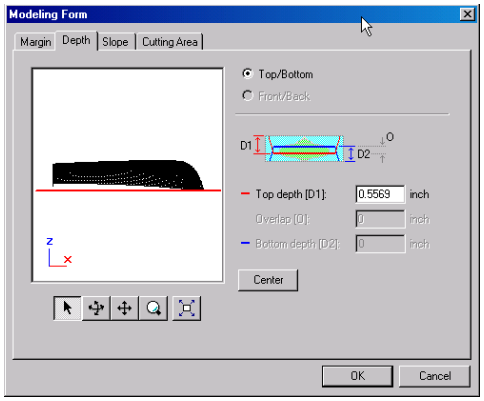
Select the Margin Tab.

Set the margin to .15” on all edges of the model. This will cause the cutter to cut all the way to and past the edges of the model. ……..

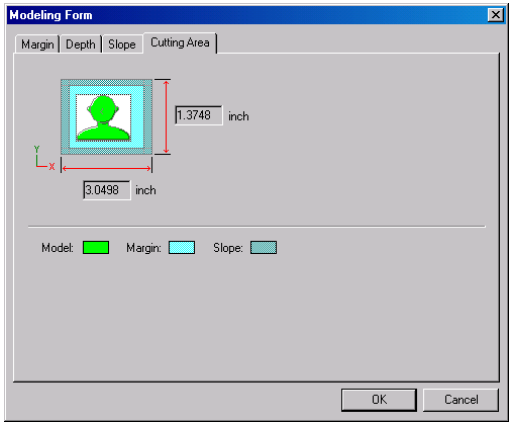


Select the Depth tab.

Set the depth to .5569” This is the full depth of the model boat.



Select the Cutting Area tab. Notice that the cutting area is .3 (.15 +.15) larger than the original model size. This is because of the margin we added to the model.



5) The Roland MDX-40 that you will be using, needs to be set as the current machine, if it is not already.

6) The material to be machined needs to be checked and/or set.

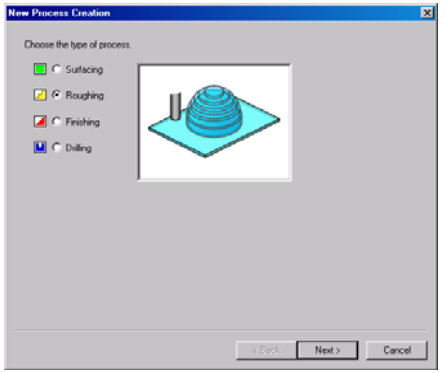
7) We will cut the part in a two step process. **Roughing** takes away the bulk of the material. **Finishing** gives the part a finished size and creates a smooth finish.

Roughing must be cut first followed by Finishing. Finishing without a roughing cut first can **damage** the milling machine.

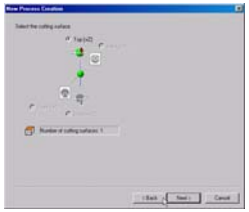
To create the Roughing tool path:

Select the New process Button: 

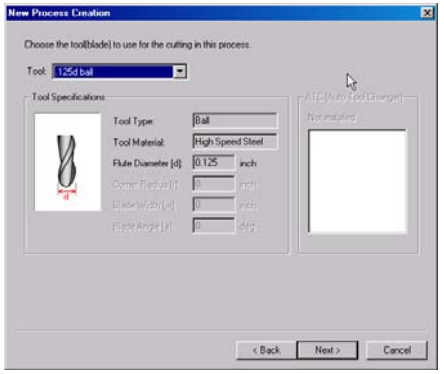
Select Roughing:



Because we have selected One side cutting, the Top is selected.



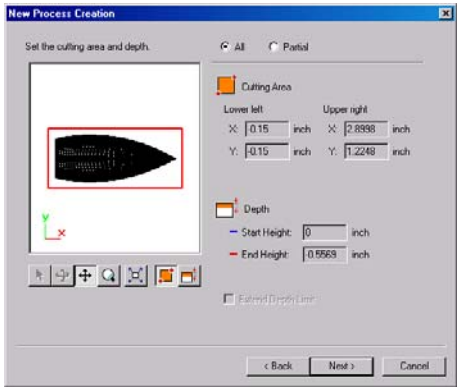
For all of the other parts in this series, we will be using a .125” (1/8”) ball mill.

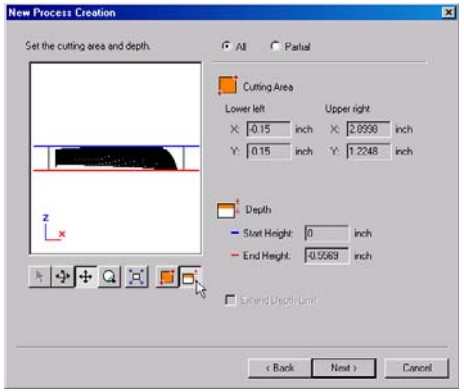


Select the .125d ball mill …….

Check the cutting area and depth ……
The ‘All’ buttons should display the correct settings.

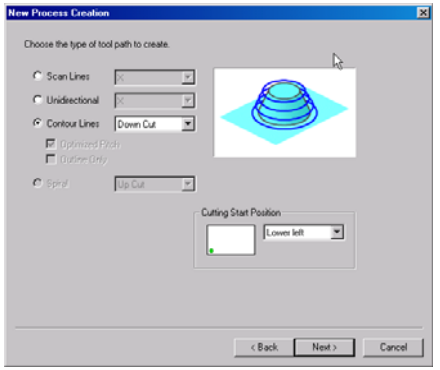
Notice the Margin in the X,Y view. This can be changed if desired by selecting the ‘Partial’ button and making desired changes.





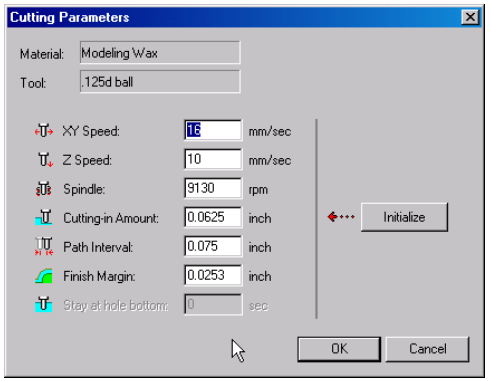
Roughing should be set to “Contour Lines Down Cut”. Contouring is a very efficient method of removing large amounts of material.

It cuts out the object in elevations.



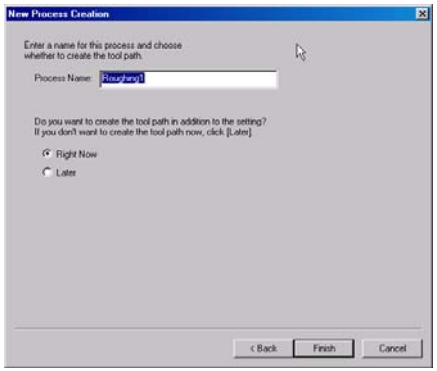
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

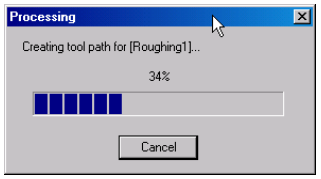


Process Name is listed as **Roughing 1**. It is a good idea to generate the tool path “Right Now” because you’ll be able to see the results before sending the tool path to the machine.

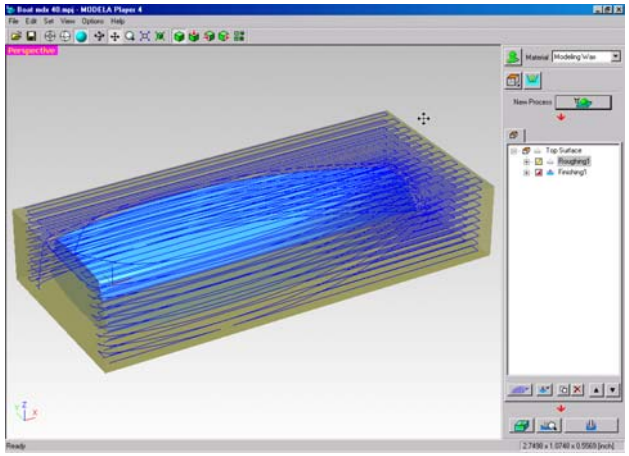
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.



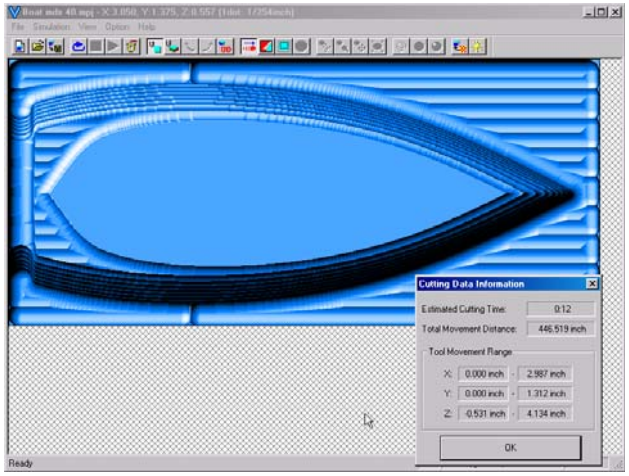
Roughing Tool Path.



8) In Modela 4 there is a built-in tool path simulator.

After the tool path is processed, **press the Preview Cutting button**. 

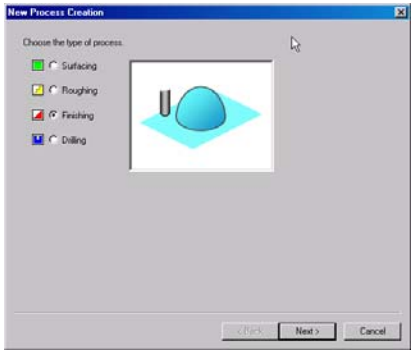
Virtual Modela automatically starts which displays the simulation and shows the cutting time.



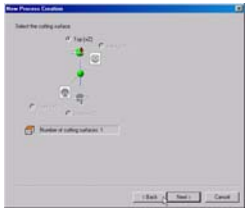
To create the Finishing tool path:

Select the New process Button: 

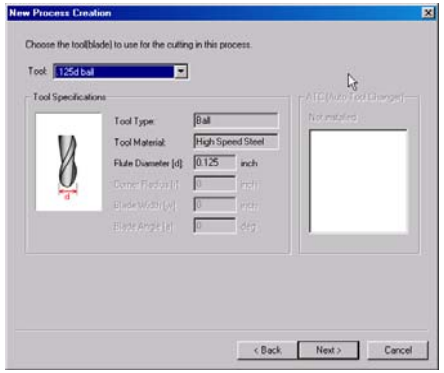
Select Finishing:



Because we have selected One-Side cutting, the Top is selected.

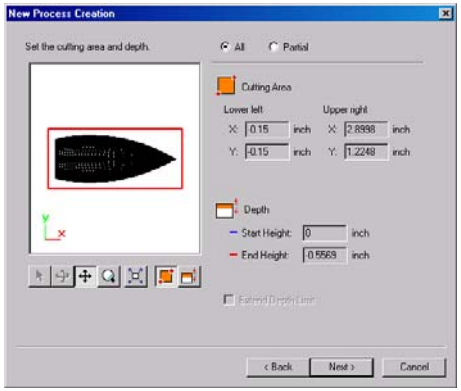


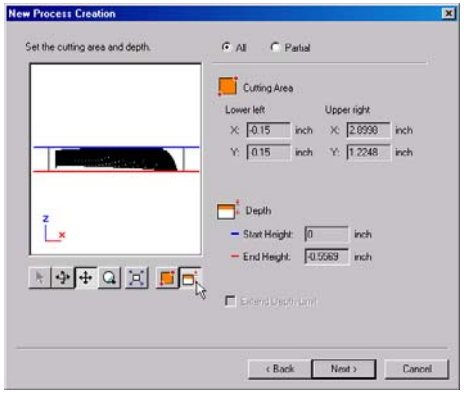
Select the .125d ball mill …….



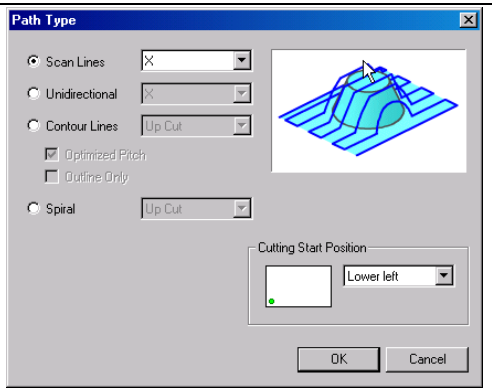
Check the cutting area and depth …….

The ‘All’ buttons should display the correct settings.





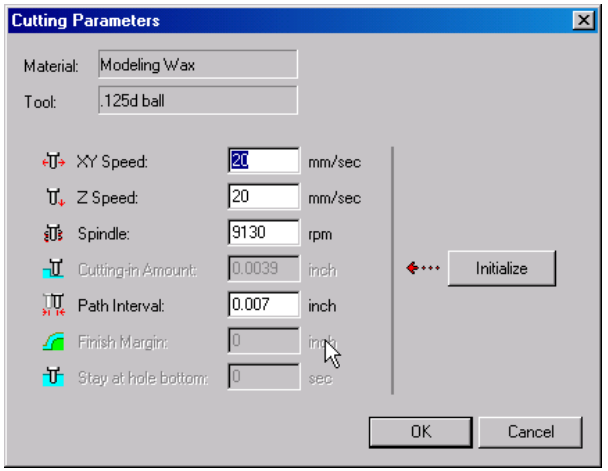
The **Finish** cutting direction can be set to cut in the X, Y, or both directions. These are lines cut along the chosen axis. Cutting both X and Y takes twice the time but gives a better finish.



Set the Finish cutting direction to Scan Lines X direction.

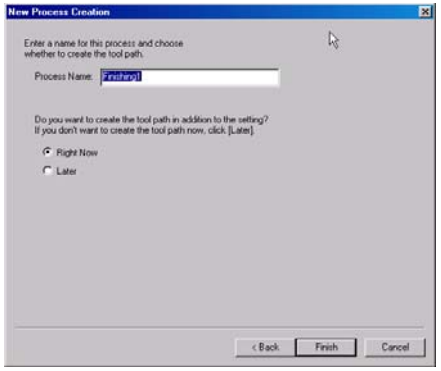
Cutting Parameters. The default settings are based on the material being cut and the cutting tool selected. In this case it is modeling wax and a .125 ball mill.

Change the settings to match the screen at the right…….. (if necessary)

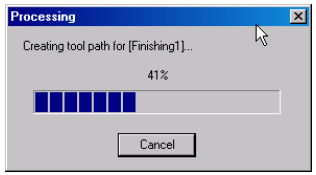


Process Name is listed as **Finishing 1**. It is a good idea to generate the tool path “Right Now” so you can see the results before sending the tool path to the machine.

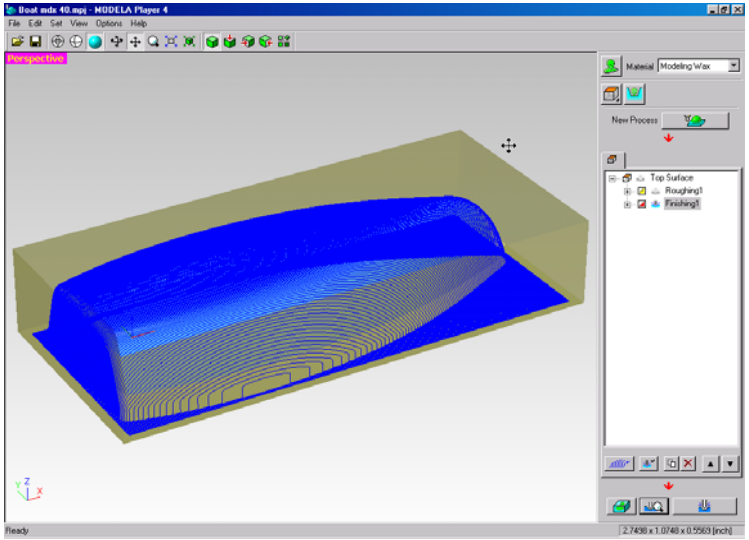
Depending on the complexity of the part and the speed of your computer, this may take extra time to generate. For this reason, the ‘Later’ option can be chosen.



You will see the dialogue box at the right as Modela Player 4 generates the tool path file.



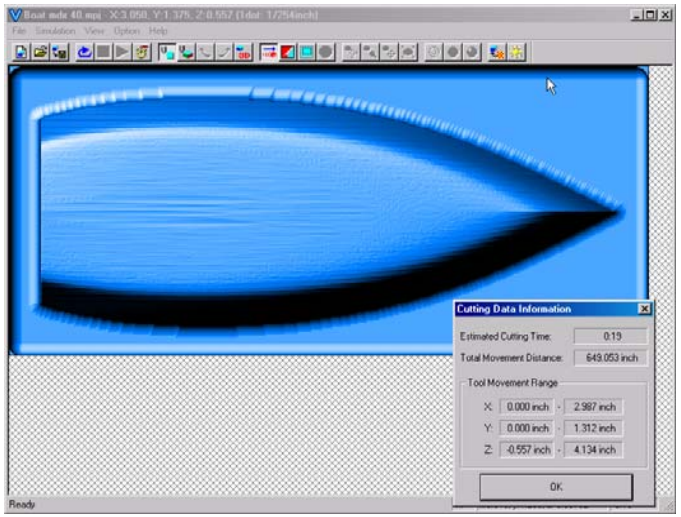
Finishing Tool Path.



9) In Modela Player 4 there is a built-in tool path simulator.

After the tool path is processed, **press the Preview Cutting button**. 

Virtual Modela automatically starts which displays the simulation and shows the cutting time.



10) To save the part as a Modela file:

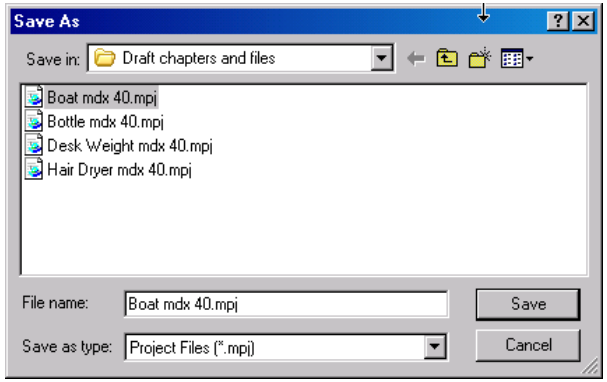
Pick the File pull down menu and Save.

The file name is the same as the one that was originally imported. Only the file extension is changed to .mdj

This is often a good time to make a second save too. Simply change part of the file-name to create a second file.

Pick Save As. **Type in the name** . 'Boat mdx 40 done'
or another file name of your choice.

Result: There are now two files. In case one is damaged, there is a backup.

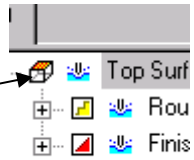


NOTE: If you do not have at least 40 minutes in the class period, do not continue to the milling steps. The average machining time for the Boat is 35 minutes. Both tool paths need to be completed without turning off the milling machine.

Step 13 explains how to exit Modela Player 4 if you do not have enough time at this point. There are instructions, at the end of the lesson, on how to open a saved Modela file.

11) To machine the part on the MDX 40:
Highlight the Tool Path to be machined.

In this case, **Highlight both tool paths**

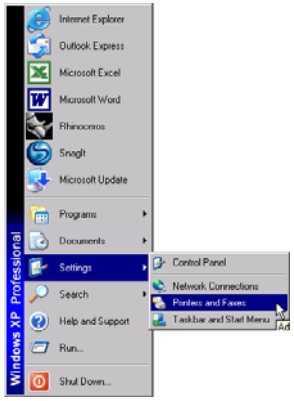


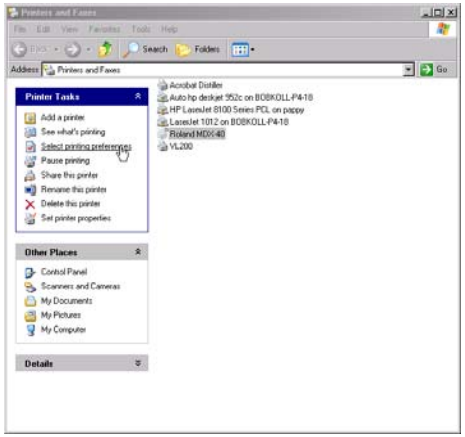
Select the Cutting/No Cutting button:

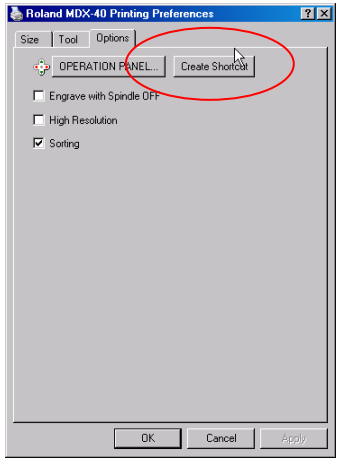
12) MDX-40 control panel:
Place a shortcut to the MDX 40 control panel on your desktop.

This gives you access to setting the cutter X, Y, Z origin. This step is Required before pressing the Cut button.

To create the shortcut, Select Create Shortcut in the printing preferences for the Roland mdx40:



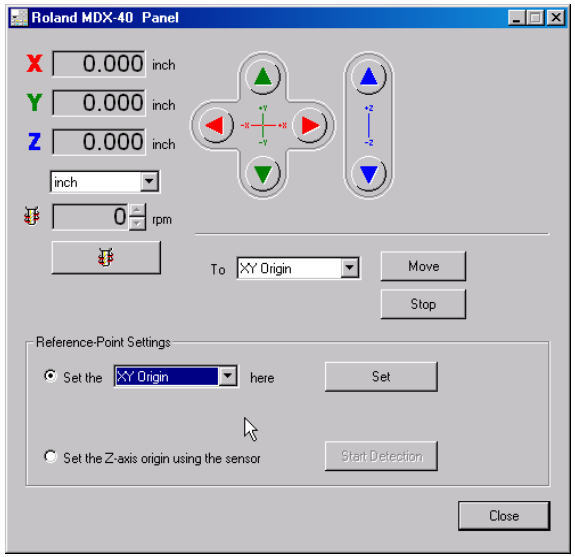




With the mouse and the arrows in the control panel, move the cutter to the proper location on the wax block.

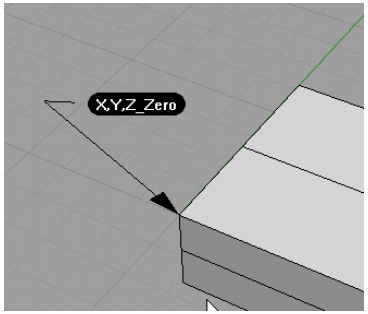
Set the Z origin, Set the Y origin, Set the X origin.

The MDX-40 ‘remembers’ the origin after it is set. If you have to restart your cutting it is easy to move the cutter to this location for re-starting if necessary.



One method to 'accurately' Set the X,Y,Z Origin at the corner of the wax stock block:

Set the X,Y,Z origin so it is at the left hand corner of the model……..

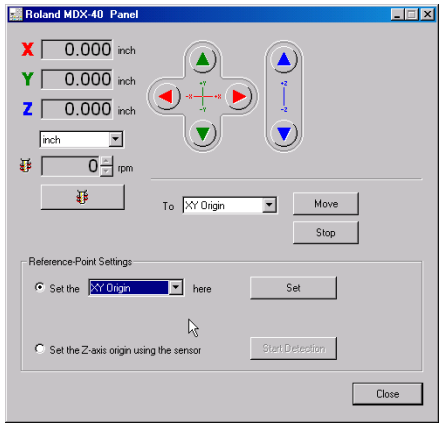


1 Set the Z origin at the top surface of the model by moving the cutter (with the mouse and the arrows in the control panel) so it touches the top of the block. Use a piece of notebook paper, approximately .003" thick. Move the cutter down until there is friction between the cutter and the block. The cutter at this point is .003 above the block. **Set Z origin here.** (temporary Z)

2 Set the X origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the left side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the X Origin here.** (temporary X)

3 Set the Y origin at the left side of the block. With the mouse and the arrows in the control panel, move the cutter so it touches the front side of the block. Put a piece of paper between the cutter and the block. Move the cutter until you feel friction between the side of the cutter and the end of the block. **Set the Y Origin here.** (temporary)

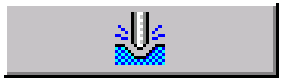
IMPORTANT: The XYZ readouts read 0.000 inches.
You need to change the position of the tool so the XY center of the tool is exactly at the corner of the block. The XY is currently one-half the tool diameter (.0625) away from the correct origin. Also, remember that our tool is .003 above the block.



With the mouse and the arrows in the control panel:

z… Move the cutter so the Z reads: Z -.003" — Set Z origin here
x… Move the cutter so the X reads: X .0625" — Set X origin here
y… Move the cutter so the Y reads: Y .0625" — Set Y origin here

Select the Cut button: Sends the tool path files to the MDX-40 and begins to cut the part.



Remember: Highlight the Roughing and Finishing tool paths and select the Cut/NoCut button. This is button is used to control whether or not each tool path file will be cut by the the MDX40.

As your machine begins cutting:

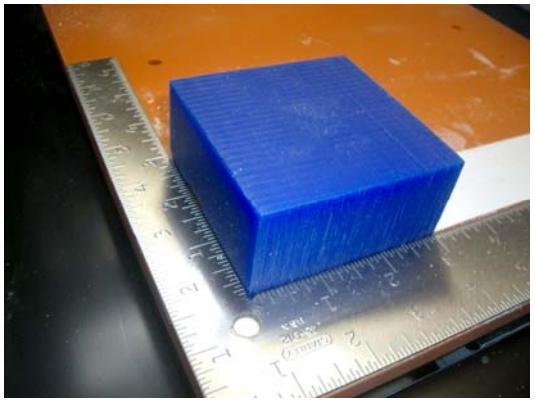
To Pause: Press the View button on the front of the MDX 40. The machine will move the cutter to a safe Z height and move the XY platform for a closer view.

>>>>**Caution:** Do not open the cover until the machine stops moving<<<<

To Resume: The machine will begin cutting at the exact location where it was when the View button was pressed.

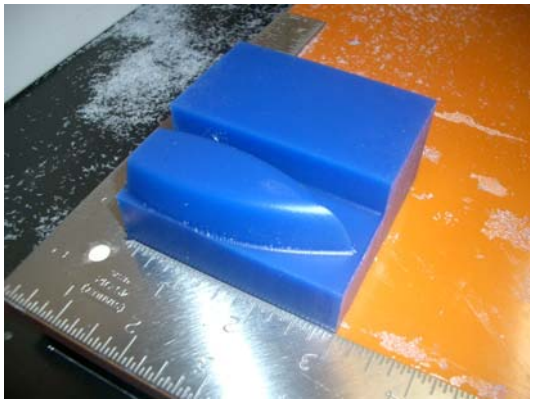
Do not leave the milling machine unattended when it is running!!!

Follow all standard safety procedures while operating the mill.



Wax block Ready to Cut

Machined Model



13) To exit the Modela Player 4:

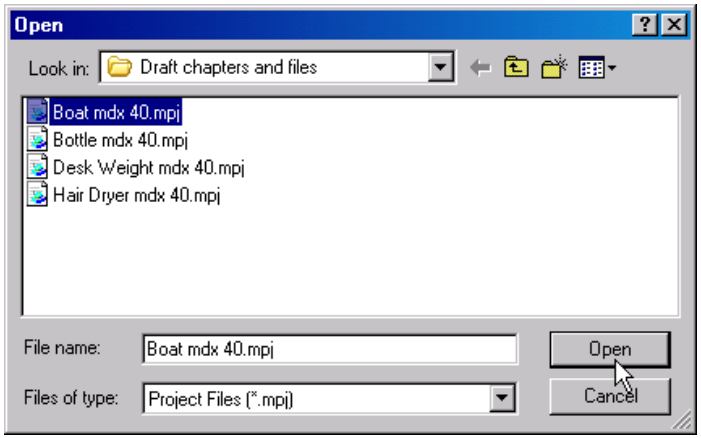
Pick the X in the upper top right corner of the Modela screen. When the Save dialogue box opens, **select Yes** to save the current file. The file retains the original file name except the extension changes to .mpj

To open a saved Modela Player 4 File:

With Modela Player 4 started:

Select the File pull down menu.

Select the File…… Boat mdx 40.mpj

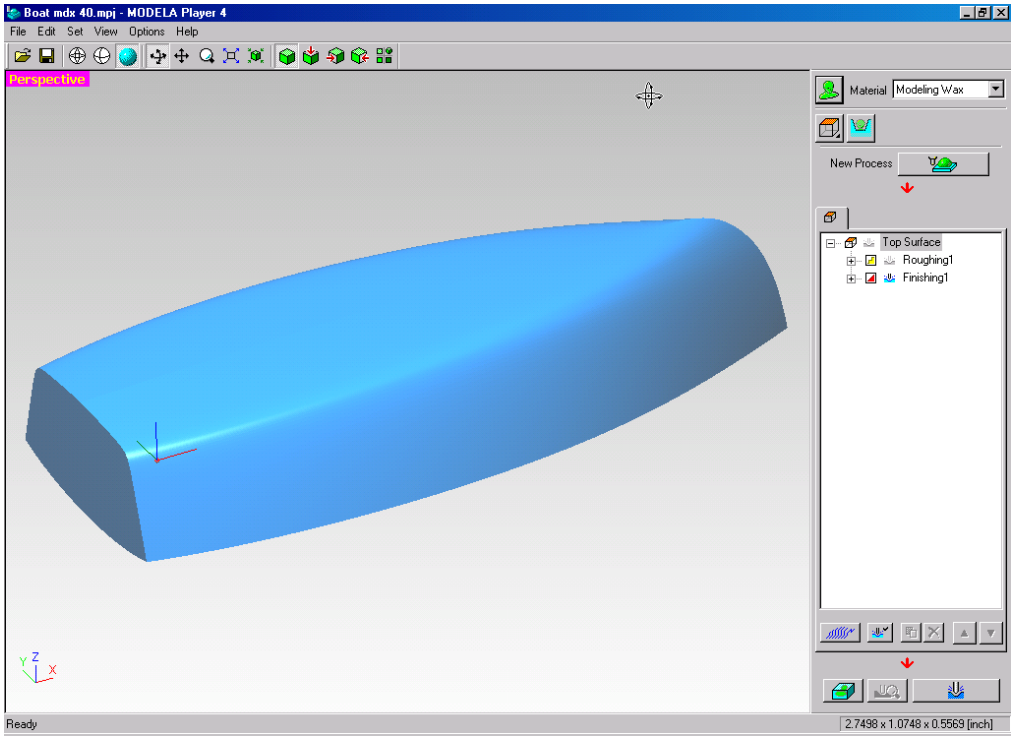


To easily recognize a Modela Player 4 file, look for the following icon In front of the file name:

Double clicking on this file name will also start Modela Player 4 and open the file.

Pick Open.

The file will open in Modela Player 4.



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