Introduction to modeling free-form surfaces with Rhinoceros

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1. Introduction

The objective of this introduction is to explain step-by-step several techniques to model free-form surfaces, mainly free-form surfaces for buildings – free-form surfaces like those found in the Guggenheim Museum in Bilbao.

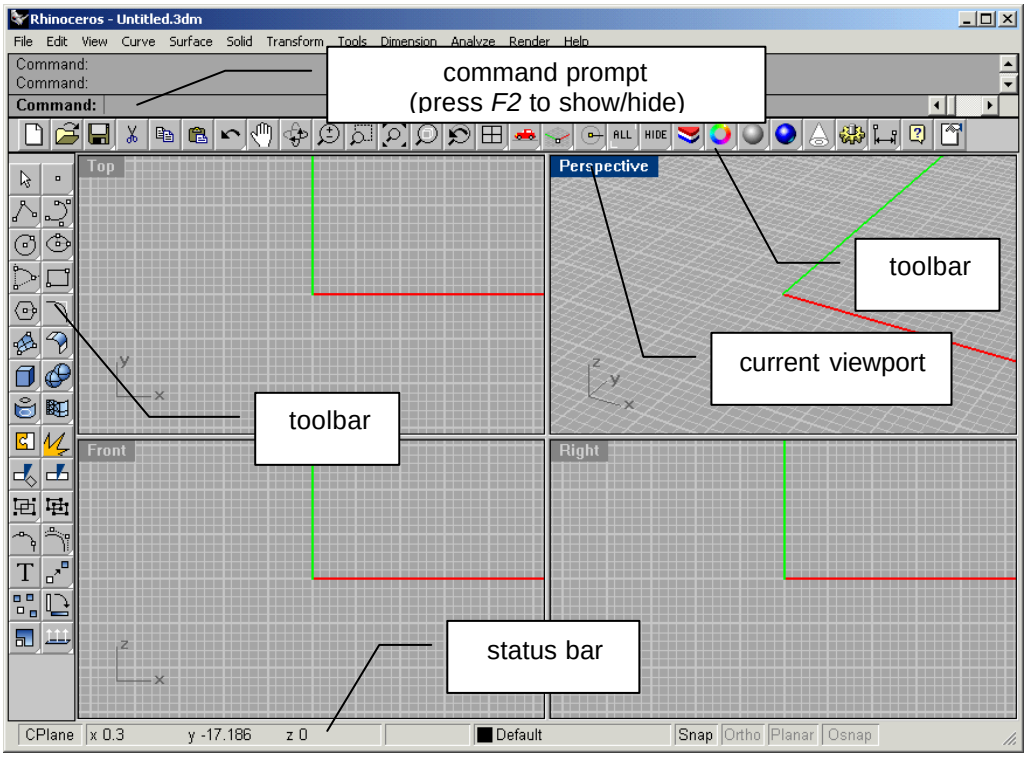
The modeling program used is Rhinoceros, version 2.0. The main reasons for this version instead of version 1.0 are:

- better visual support for object snaps
- better render capabilities (support for OpenGL)
- better unit model – supports now AutoCAD 2000 units and scale
- support for AutoCAD 2000 .dwg files (still unable to load AutoCAD solids)
- scripting (VBScript and JScript support)
- plug-ins support

For an introduction to basic commands used here see *Getting Started* in Rhinoceros *Help* menu (in Contents click *tutorials* and then click *introduction*).

For more options or more detail in a given command don't hesitate to search for it in the *Help Topics* from the *Help* menu in Rhinoceros – or press *F1*.

This is how Rhinoceros 2.0 looks like:



Rhinoceros 2.0 interface

2. Keyboard shortcuts

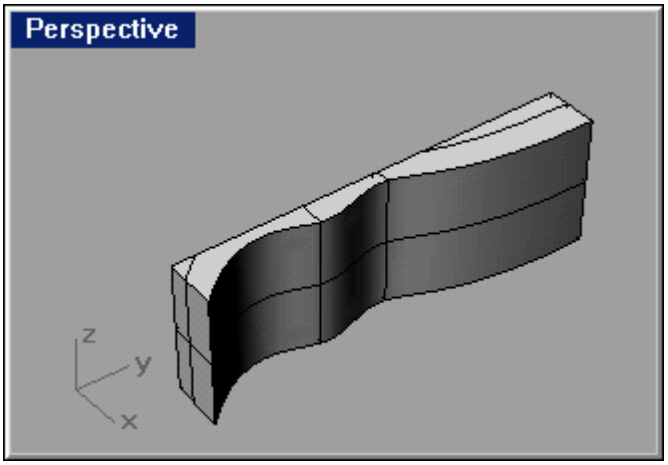
Some of these keyboard shortcuts can be customized from *Options* in the *Tools* menu. These shortcuts can also be found in Rhinoceros help.

F1	Help
F2	Command History
F3	Properties
F7	Show Grid and/or Show Grid Axes
F8	Ortho
F9	Snap
F10	Points On
F11	Points Off
F12	Properties
Ctrl + A	Select all
Ctrl + S	Save
Ctrl + Z	Undo
Ctrl + Y	Redo

3. Modeling Technique I

3.1. Objective

Model a very simple wall where the exterior is a free-form surface and the inside is like a regular wall, flat and regular.

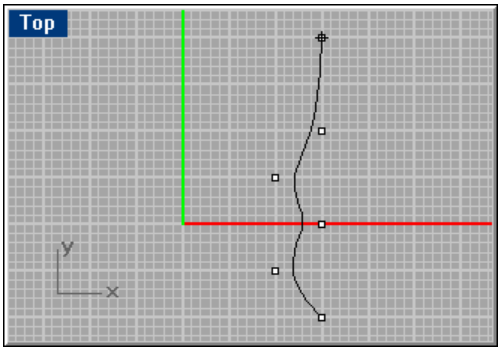


Final result after applying this technique

3.2. Step-by-Step

1. Create the curve that represents the wall shape

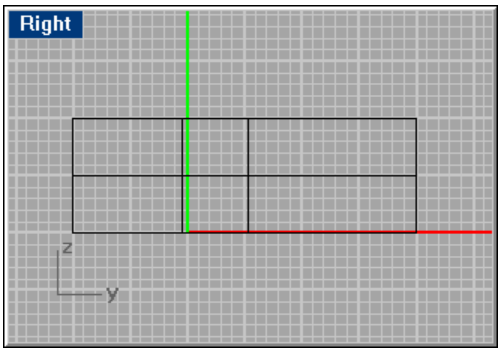
1.1. Left click in  (*Curve* command) to create a curve in the Top viewport and follow instructions in the command prompt



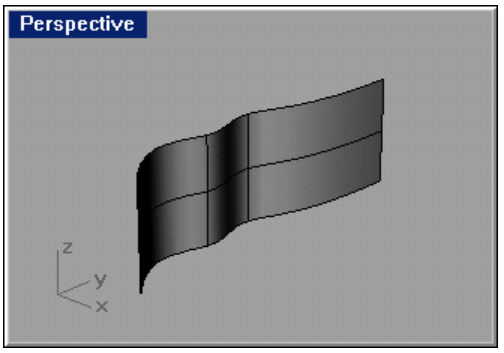
Define the curve with the help of control points (or any other curve command)

2. Extrude the curve to create a surface (this step is necessary to allow us to extrude the surface and do a solid by extruding this surface is this final solid that will be the wall)

2.1. Right click in  and then left click in  (*Extrude* command)



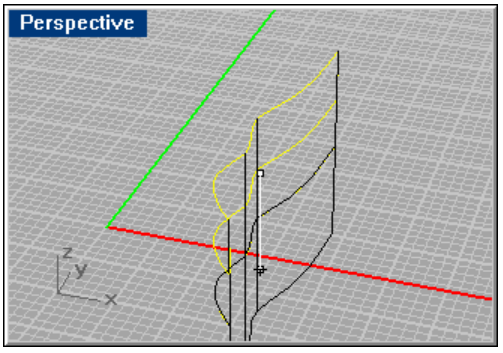
Result of extrusion of a curve into a surface (1)



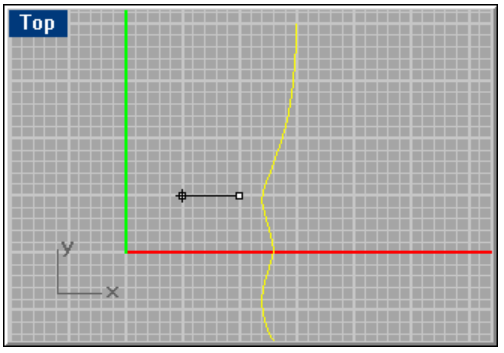
Result of extrusion of a curve into a surface (2)

3. Extrude the surface to give thickness to the wall

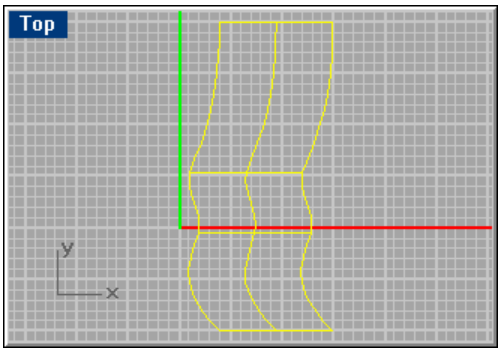
3.1. Right click in  and then left click in  (*ExtrudeSrf* command) it's very probable that the direction of extrusion is incorrect. To correct this press 'D' key and choose a new vector representing the direction of extrusion (do not forget that this key pressed is to invoke an option in the extrusion surface command always check command prompt for options)



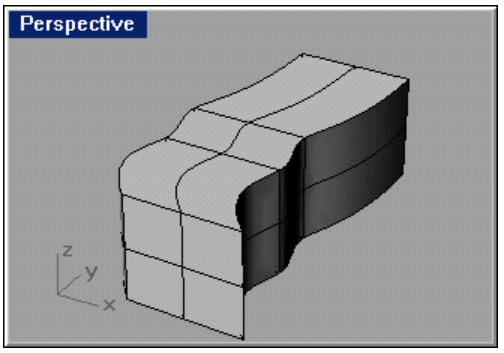
Incorrect extrusion direction



Defining a new vector for extrusion is like draw a line, the endpoints define the line and its direction the direction of extrusion



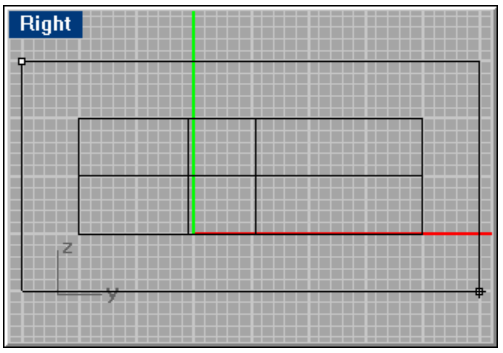
Result of extrusion (1)



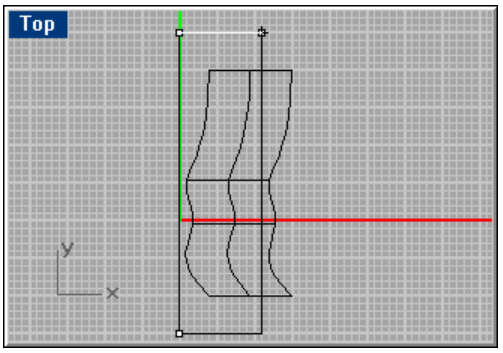
Result of extrusion (2)

4. Create a box big enough to partially cover the resulting solid from extrusion

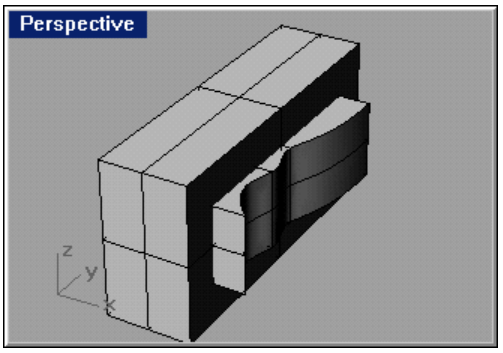
4.1. Left click in  (*Box* command) and define the box with the help of Right viewport first and the with the help and Top viewport



Defining the box with Right viewport (1)



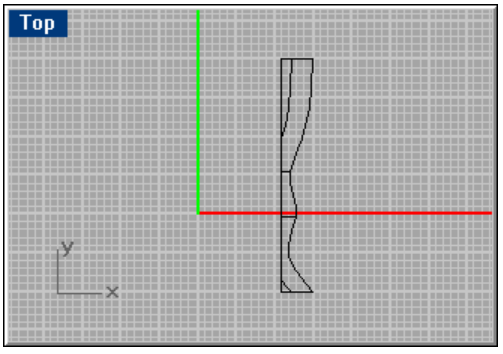
Defining the box with Top viewport (2)



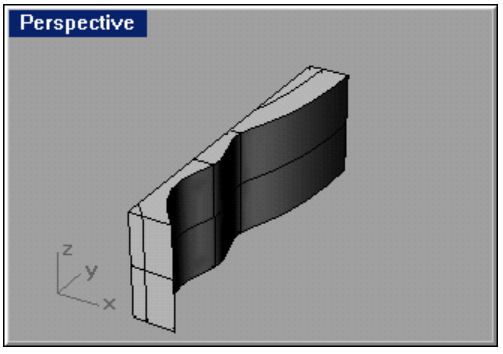
Result from drawing the box

5. Subtract the box from the initial solid

5.1. Right click in  and then left click in  (*BooleanDifference* command) the order in which we select the first and second surface or polysurface (a solid in Rhinoceros) determines the final result. In this step we want to subtract from the initial solid the box defined in step 4 see the command prompt for auxiliary help when select the objects



Final result after choose first the initial solid and then the box (1)



Final result after choose first the initial solid and then the box (2)

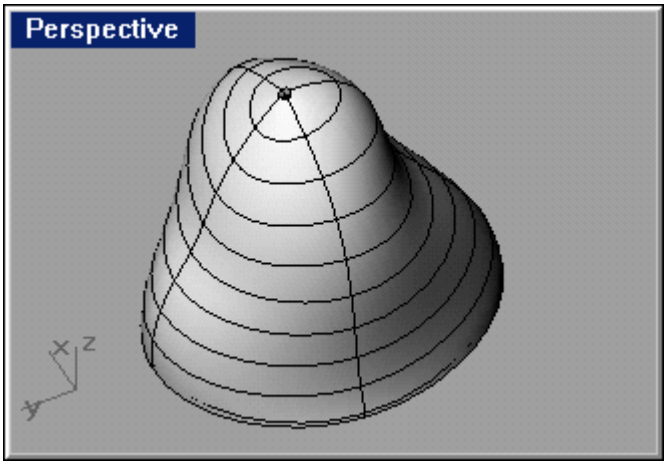
3.3. Notes

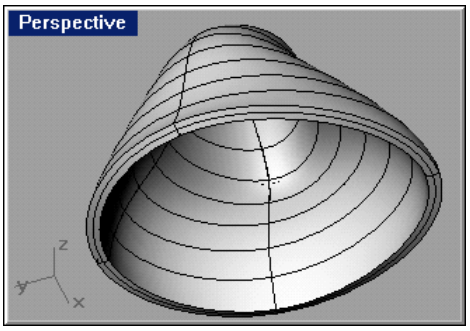
- **Save your work regularly**
- By waiting a brief moment above an icon without moving the mouse cursor we can see the command name that represent by click it with left and/or right mouse button
- There are three ways to start a command: write the command (in the command prompt), choose it from a menu (or sub-menu) or click in the respective icon from the toolbar (or sub-toolbar)
- We can use the command *What* to visualize several object properties *What* command give us is a verbose response in the command area. Because the size of this area is very small for this command we should press *F2* in order to view the properties in a more clear way (press again *F2* to hide command history).
- We can also view an object properties in a more sophisticated way by left click in  (*Properties* command). Here we can control several things like the object names and layer.
- In Rhinoceros the solids are called PolySurfaces this means that we can create a box given six surfaces and join all together. With the same idea in mind we can explode a box to get six planes that defines the box (Tip: try it out First draw a box with  (*Box* command). Then explode it with  (*Explode* command). Finally with  (*Join* command) join all six planes)
- Note that a solid in the sense of Rhinoceros can be open Repeat the previous note but now with only five planes. The result is still a PolySurface that it's an open one we can see this with *What* command or with  (*Properties* command)
- When a given command can't support PolySufaces we can explode it, edit it or do what we want it and join them altogether again.

4. Modeling Technique II

4.1. Objective

Imagine a surface like a dome, but more irregular. Because this is a surface there is no thickness and for several technical designs this wouldn't do. To solve this we must apply some thickness to the surface.



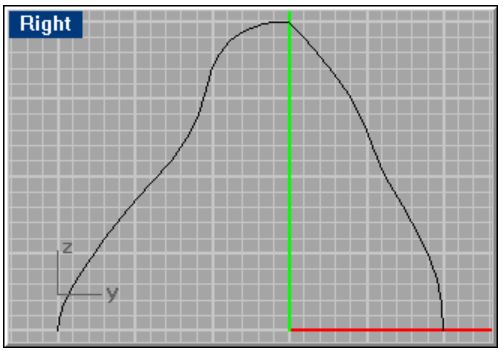


Final result after applying this technique

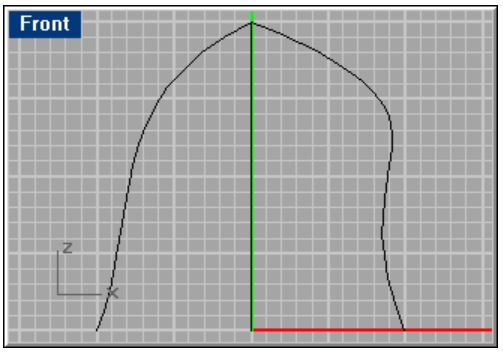
4.2. Step-by-Step

1. Create the surface with the help of curves

   1.1. With the help of  (*Curve* command) draw two curves in Right viewport and other two in the Front viewport – these lines serve as guidelines to lofting, the next step

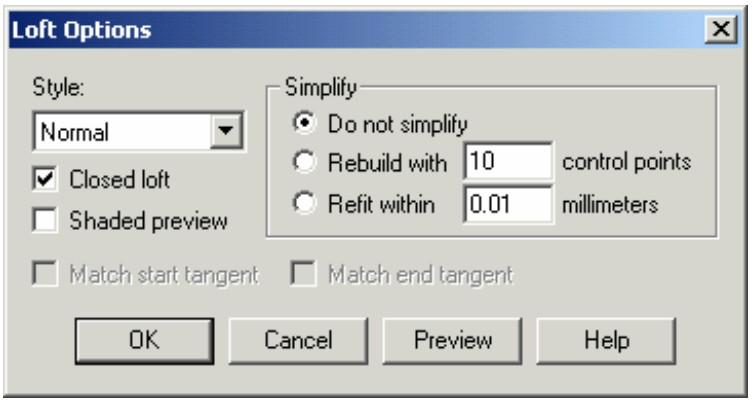


Curves in the Right viewport

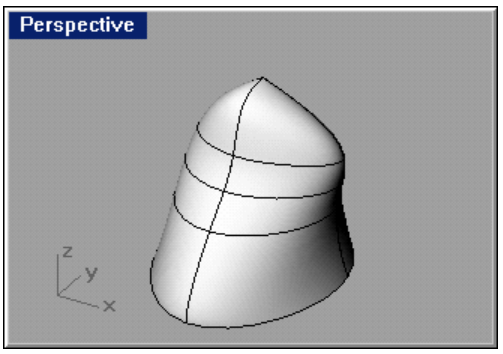


Curves in the Front viewport

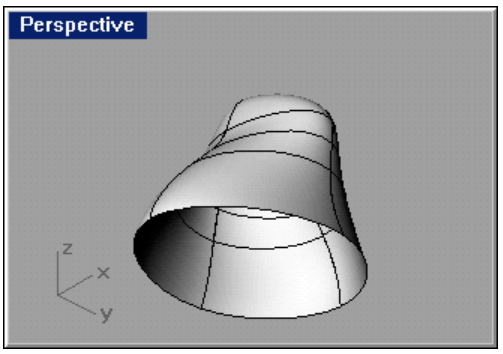
1.2. Right click in  and then left click in  (*Loft* command) to create the surface (in loft options choose closed loft for best results – try the other options for your taste and/or purposes)



Loft options after selecting curves and pressing Enter



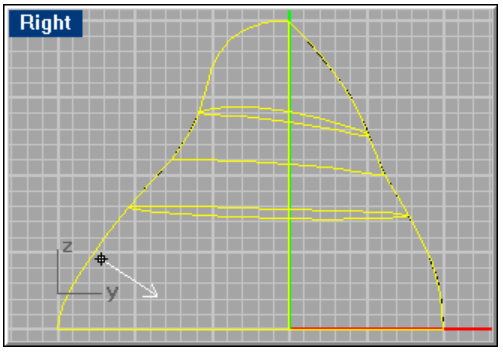
Result of lofting (1)



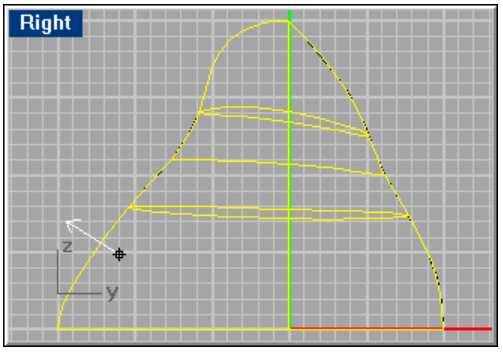
Result of lofting (2)

2. Now we only need to add the thickness required (beware that if we have very closed angle in any part of the surface and the thickness will be defined from the outside to the inside we must choose a wise thickness or the results could be catastrophic – maybe it's better to define the surface and then add the thickness from the inside to the outside)

2.1. Right click in  and then left click in  (*OffsetSrf* command) and choose the direction of the offset, just click inside the surface in any viewport to flip the direction

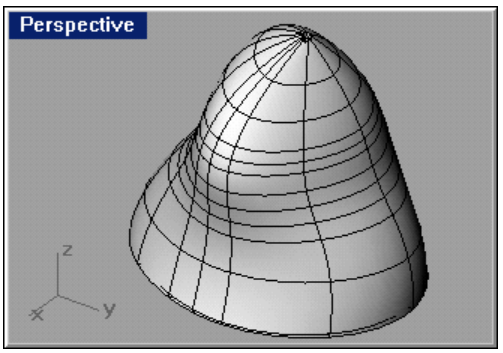


Flipping the direction to offset occur (1)

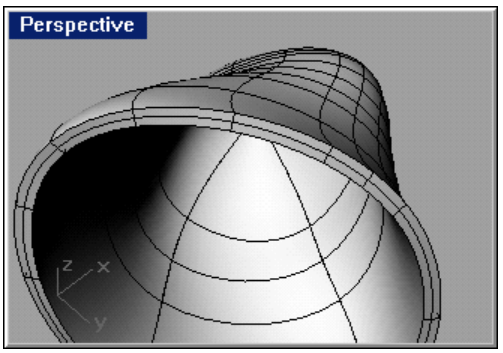


Flipping the direction to offset occur – just click in the surface (2)

2.2. Still in the *OffsetSrf* command press *S* key to say to Rhinoceros to offset the surface has a solid. Now write the thickness desired and then *Enter* (all these instructions are presented in the command prompt just follow them)



Final result (1)



Final result (2)

4.3. Notes

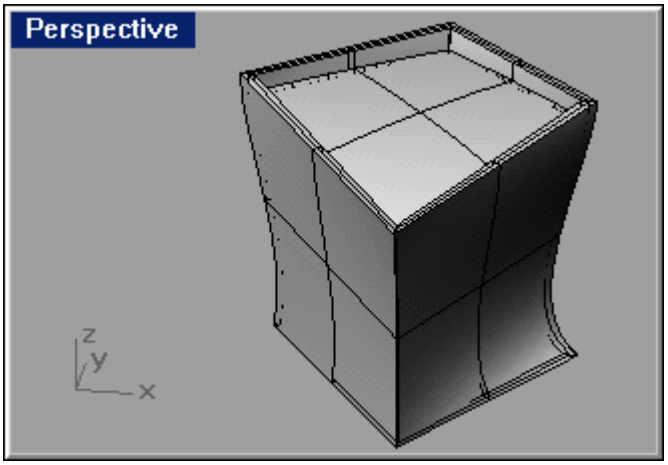
- Keep in mind that we are dealing with computers and because of that we can always do something in several ways, some more correct than others, some more complex than others (for example, there are a few commands to draw a curve in Rhonceros. In this introduction we create curves by providing the control points by using the *Curve* command because this serve best our purposes)
- We can edit an surface by left click in  (*PtOn* command) (Tip: hide unnecessary control points to simplify points selection and editing)
- Left click in 

until the sub-toolbar appears and then left click in  (*HidePt* command) to hide the control points that is desired.
- Note that the commands for curves/lines like trim, offset, etc work in similar ways with surfaces
- Don't forget to follow the instructions in the command prompt
- In the *Loft* command if the results are disastrous check the order in which you choose the lines that are the base of lofting – the order matters

5. Modeling Technique III

5.1. Objective

Here we will model something like we can find in the Guggenheim museum in Bilbao.

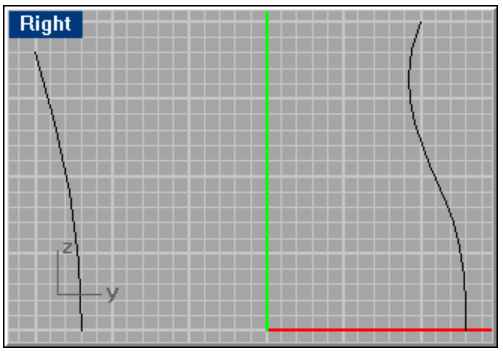


Final result after applying this technique

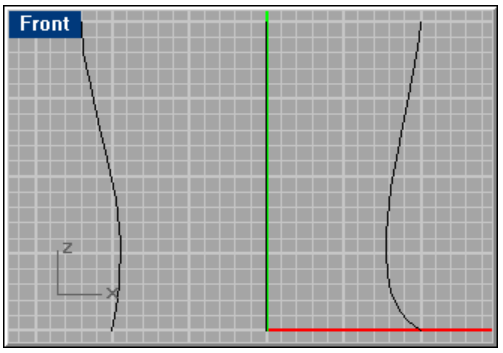
5.2. Step-by-Step

1. Create the curves that will define the several walls (in this case only four walls, but more could be defined)

1.1. With the help of  (*Curve* command) draw two curves in Right viewport and other two in the Front viewport – these lines serve as guidelines to lofting, the next step



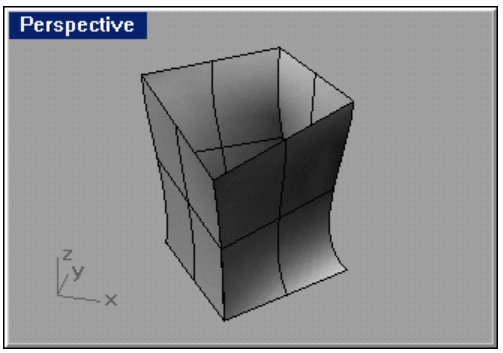
Curves in the Right viewport



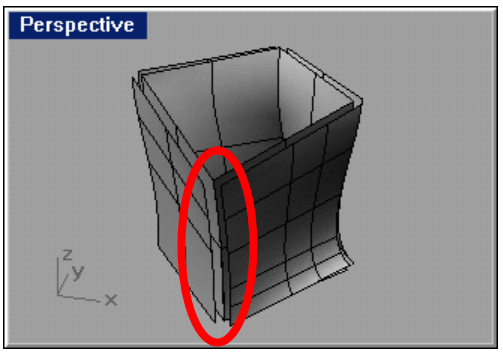
Curves in the Front viewport

2. In the previous technique we use the  (*OffsetSrf* command) to add the desired thickness to the model but this wouldn't work for the model show in 5.1

   2.1. Use  (*Loft* command), has in the previous technique, and in the loft options dialog in the *Style* choose *Straight Sections*. The result for a surface is good but to define thickness with *OffsetSrfc* command is very bad



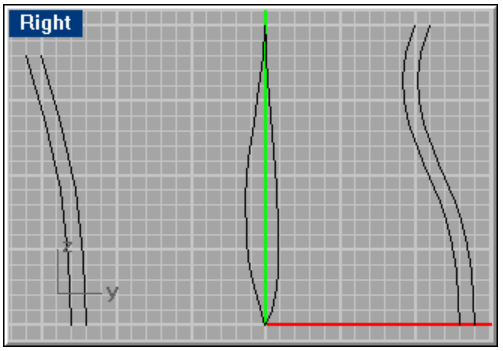
Result of lofting



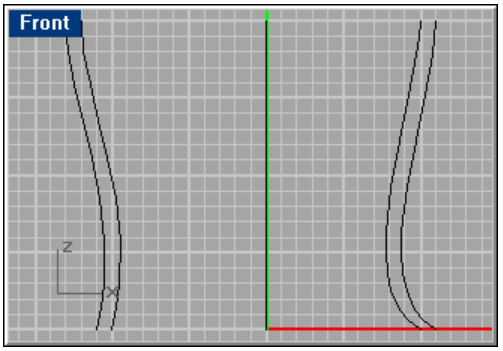
Result of the *OffsetSrfc* command – the outside surface edges don't match. If the offset was done outside to inside then the surfaces would intersect

3. In order to prevent the bad results given in step 2 we will add thickness in another way

   3.1. Copy (with  (*Copy* command)) the lines draw in step 1 one unit (or another small amount) from inside to the outside, both in the *Right* and *Front* viewport



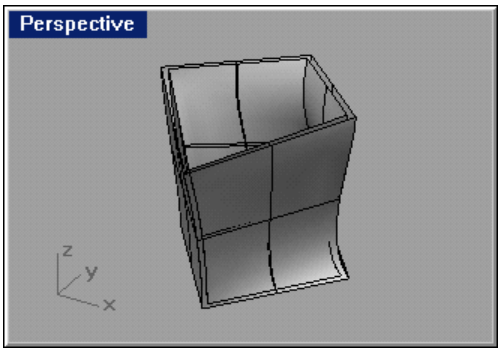
Curves in the Right viewport



Curves in the Front viewport

4. Create the insider and outsider surface

   4.1. Use  (*Loft* command) twice, with *Straight Sections* selected in the *Style* from the lof options dialog, to get two *PolySurfaces*



Result of lofting with straight sections – two *Polysurfaces*

5. Separate the *PolySurfaces* into several surfaces to define the walls correctly

   5.1. Use  (*Explode* command) in both *PolySurfaces* to get eight individual surfaces

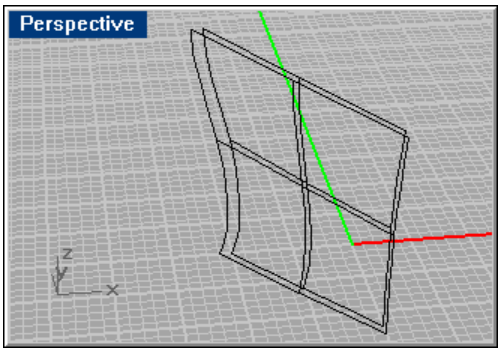
6. Form the walls as solids (or *Polysurfaces* in Rhinoceros)

   6.1. To avoid a more complex scene, hide all but the two surfaces that are needed. Do this by left click in 

(*Hide* command) – to show all objects right click in 

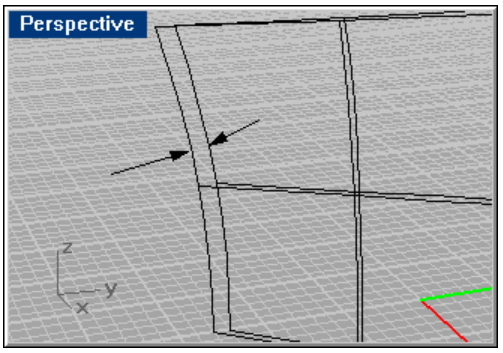
. For more options when select an object right click in 

and search for the right icon in this sub-toolbar to select a given type of objects

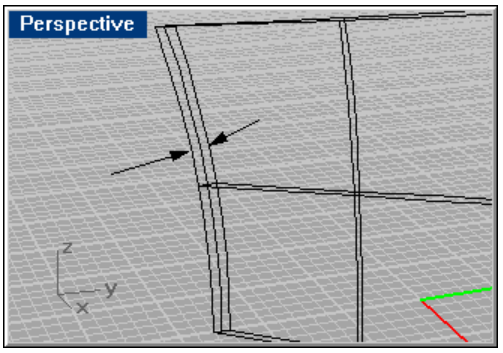


Result of hiding all the objects but the two that matters for now

6.2. Use  (*Loft* command) to create the borders that are missing (four borders, four lofts). The lines that we choose are the surface edges

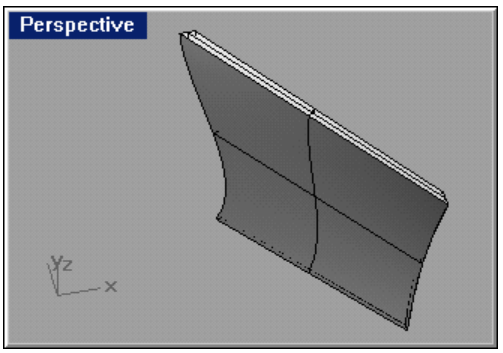


Missing surfaces to achieve a solid with *Join* command



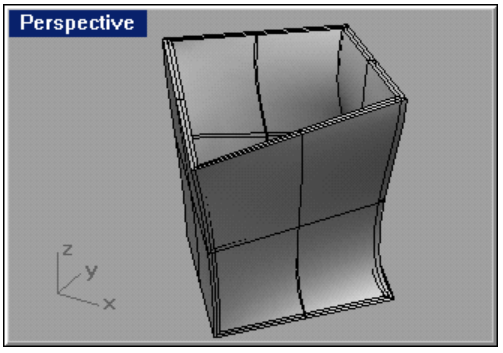
Results after do the lofting in the left border (keep the *Straight Sections* selected in the *Style* and also the *Do not simplify* in the *Simplify* to achieve best results).

6.3. Use  (*Join* command) to create the *PolySurface*



Result of lofting all edges an joining them together – one side of the structure

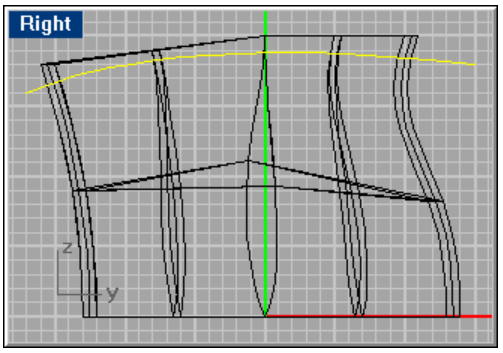
7. Repeat step 6 for the walls remaining



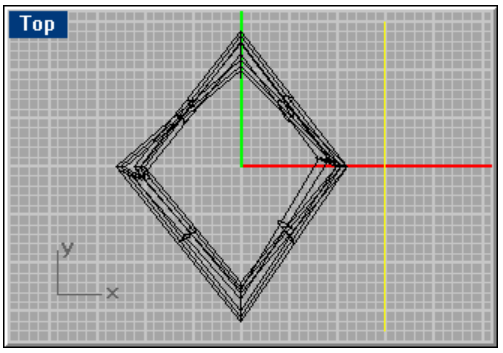
Result after lofting and joining all the walls

8. Draw the ceiling – the idea is that from a given view (in this case from *Right* viewport) the structure is "cut" with a imaginary curve that defines the ceiling

8.1. Draw the curve that represents where the struct would be "cut" (in the Right viewport) with  (*Curve* command)

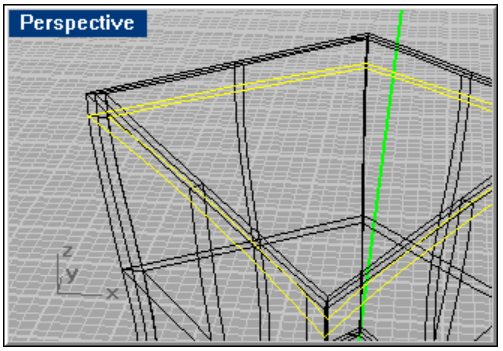


Curve that represents the ceiling cutting the structure – for the outside surface



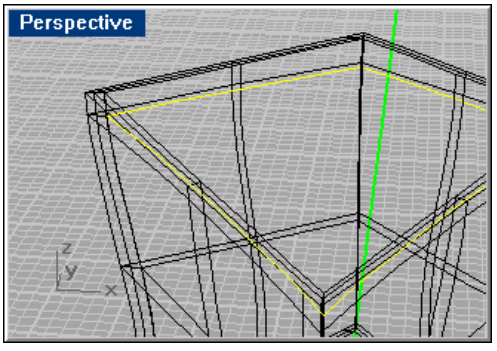
The size and position of the curve influence the success of the next step – the curve should be big enough to include all the structure and should be all in one side of the structure

8.2. Project the curve to the structure using  (*Project* command). This should be done in the *Right* viewport – this is important, see help for more details (search for project)

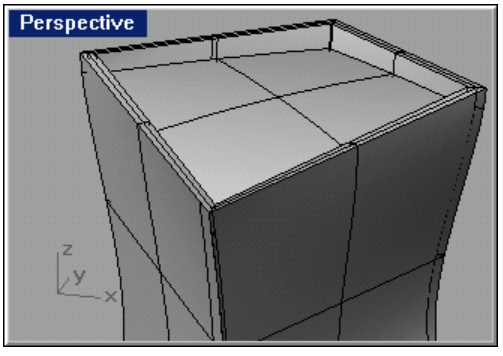


Result from projecting the curve into the structure – the project command was done in the *Right* viewport. As you can see there are 2 curves project for each wall

8.3. Create the surface with only the curves from inside and with right click in  and then left click in  (*EdgeSrf* command)



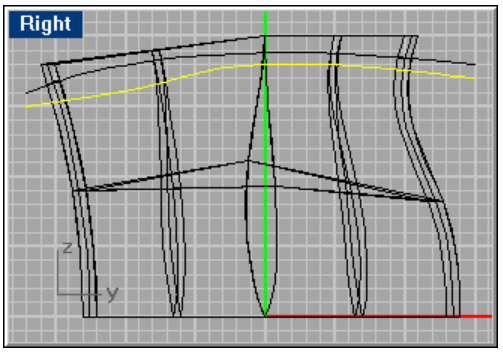
Curves selected to define the ceiling

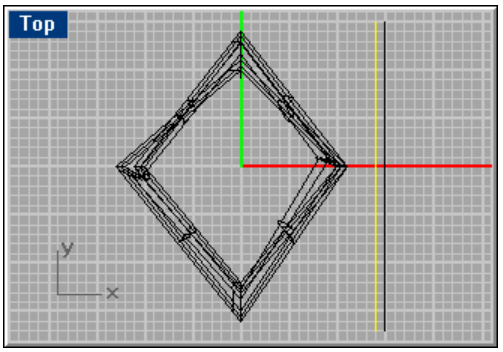


Result from *EdgeSrf* command

9. Now, we only need another surface to add some thickness to the ceiling in the next step. The idea is just like in step 8

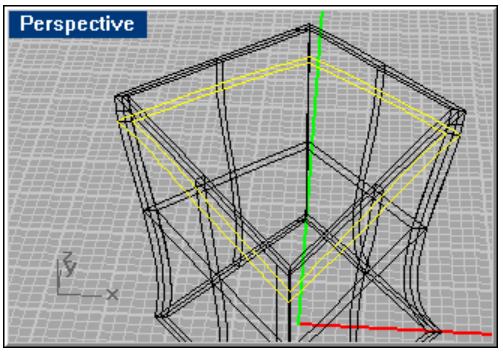
9.1. Draw another curve to represent the thickness of the ceiling (with  (*Curve* command))





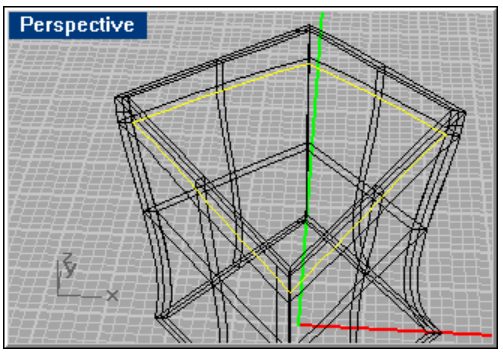
Curve that represents the ceiling cutting the structure – for the inside surface

9.2. Project the curve as done in step 8.2

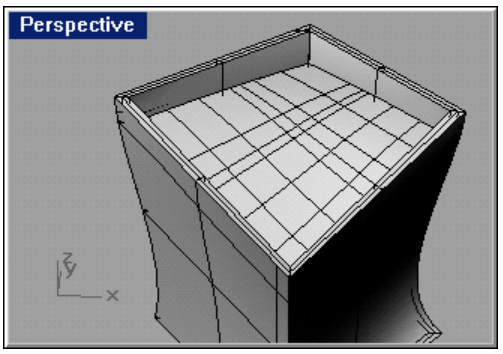


Result from projecting

9.3. Create the surface as in the step 8.3 (only the curves inside)



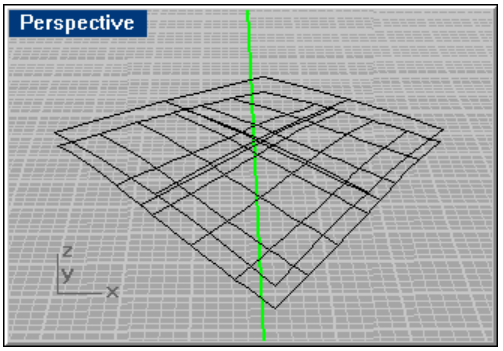
Curves selected to define the ceiling thickness

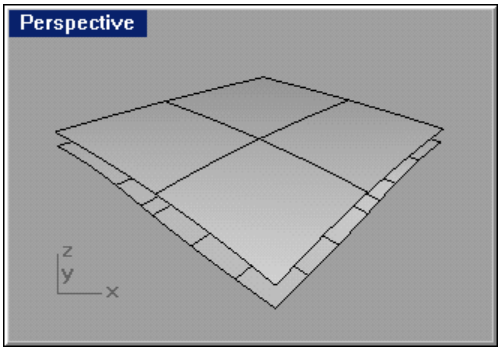


Result from *EdgeSrf* command with second projected curves

10. What is missing is the thickness in the ceiling

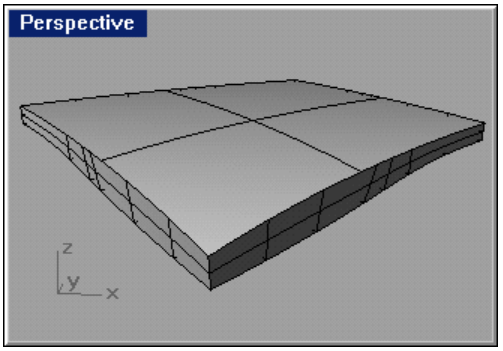
10.1. To achieve the thickness just do has in step 6 – hide unnecessary objects





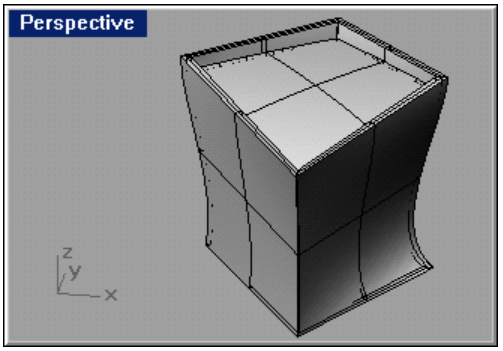
Ceiling surfaces

10.2. Use Loft command in all four surface edges and then join all the six surfaces (like in steps 6.2 and 6.3)

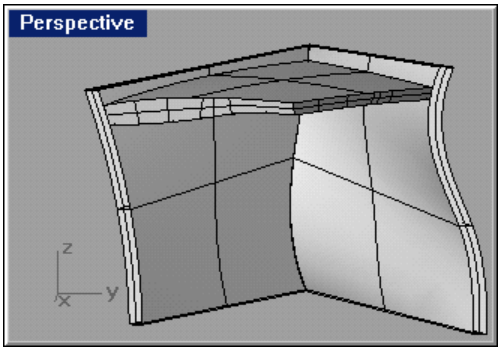


Resulting ceiling

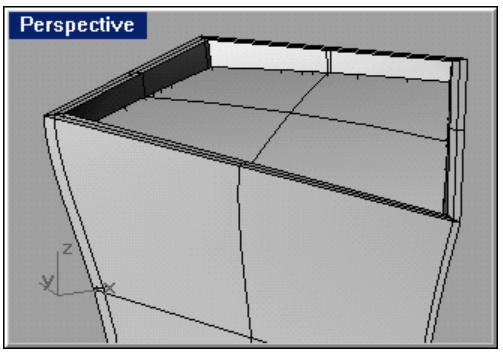
11. Now see the results



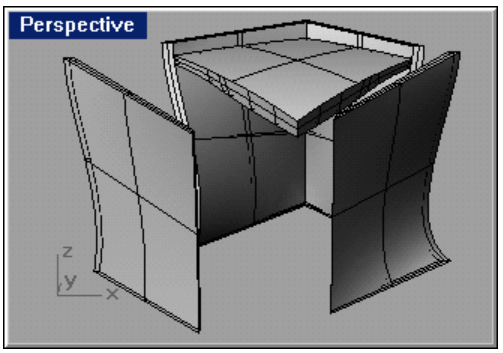
Final result (1)



Final result (2)



Final result (3)



Final result (4)

5.3. Notes

- Use layers like in AutoCAD to auxiliary in drawing
- In Rhinoceros help is written “Use Join (…) so the rendering and meshing won't have any cracks in it and will match up exactly.”
- A surface can be edit like a curve can be – just use  (*PtOn* command) to show the control points for the surface (Tip: hide unnecessary control points – this saves you and your computer)
- This technique should also work with more (or three) surfaces
- In step 10.2 if the edge, when selecting the surface edge for lofting, can

6. Final words

For any questions, recommendations, corrections or anything else, you can contact me at rcco@mega.ist.utl.pt (alternatively drigoor@yahoo.com).