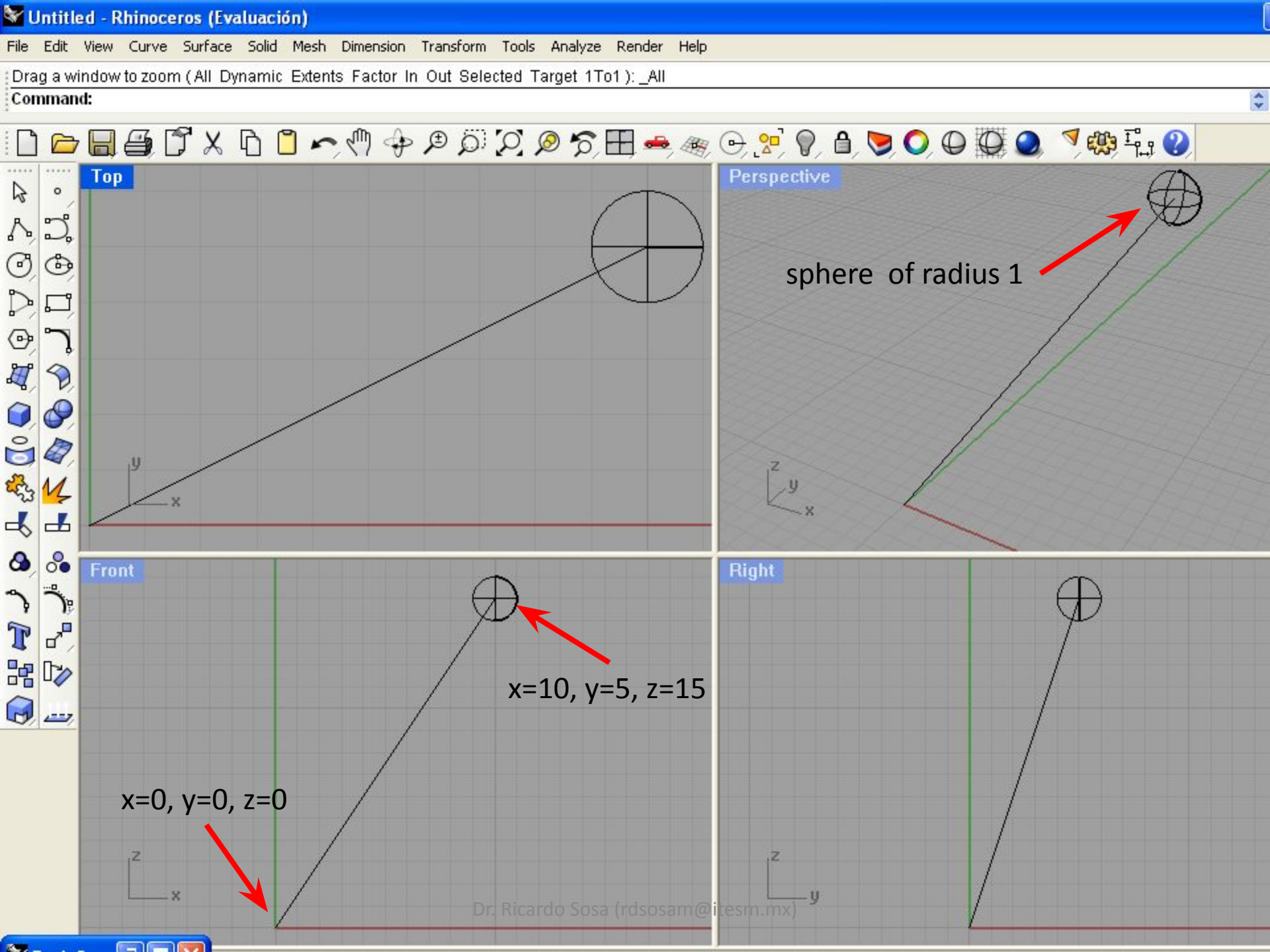
RhinoScript 101 Creativity

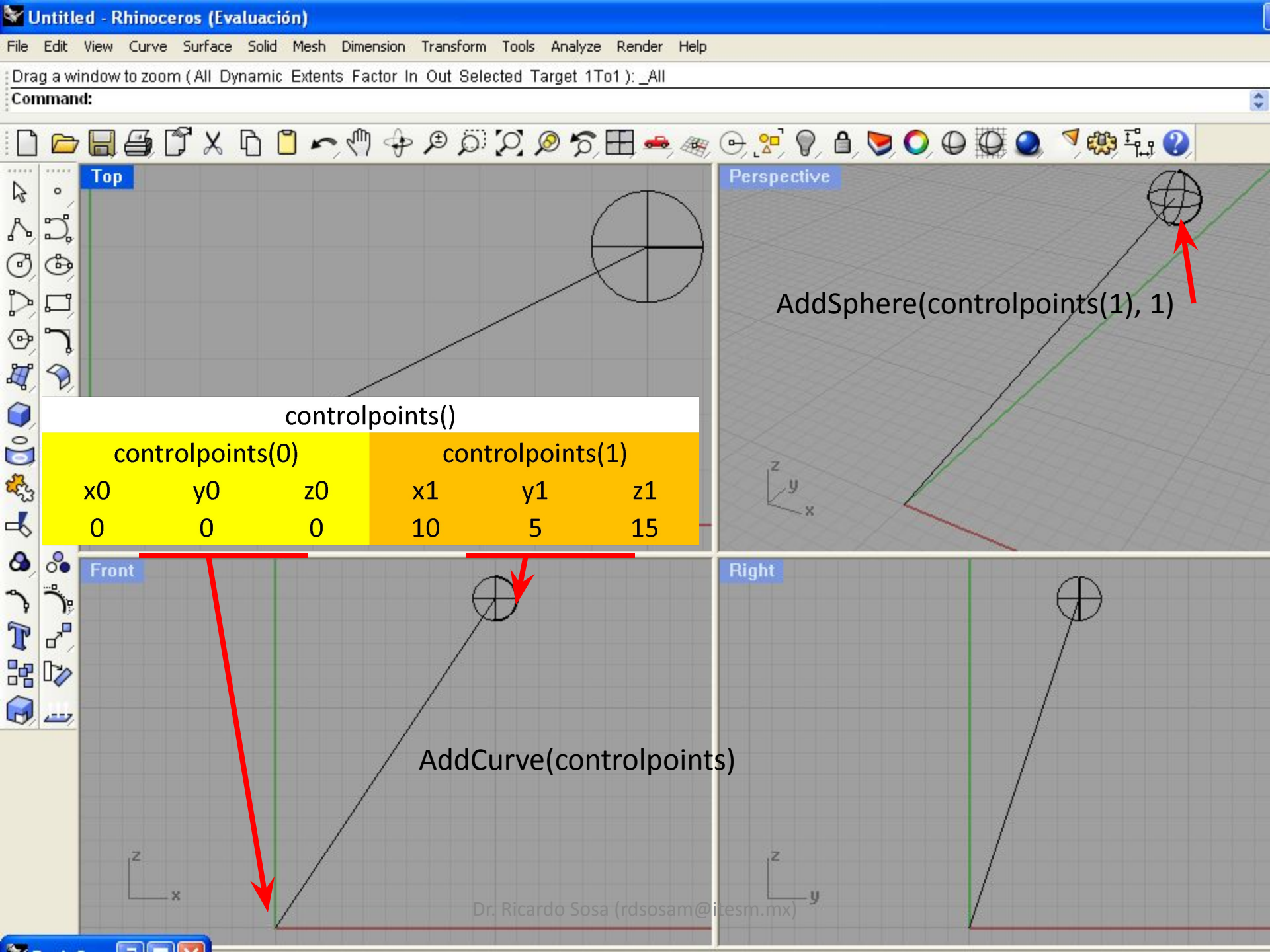
"or how to do interesting things that are not easy to do with the mouse"

Dr. Ricardo Sosa (rdsosam@itesm.mx)

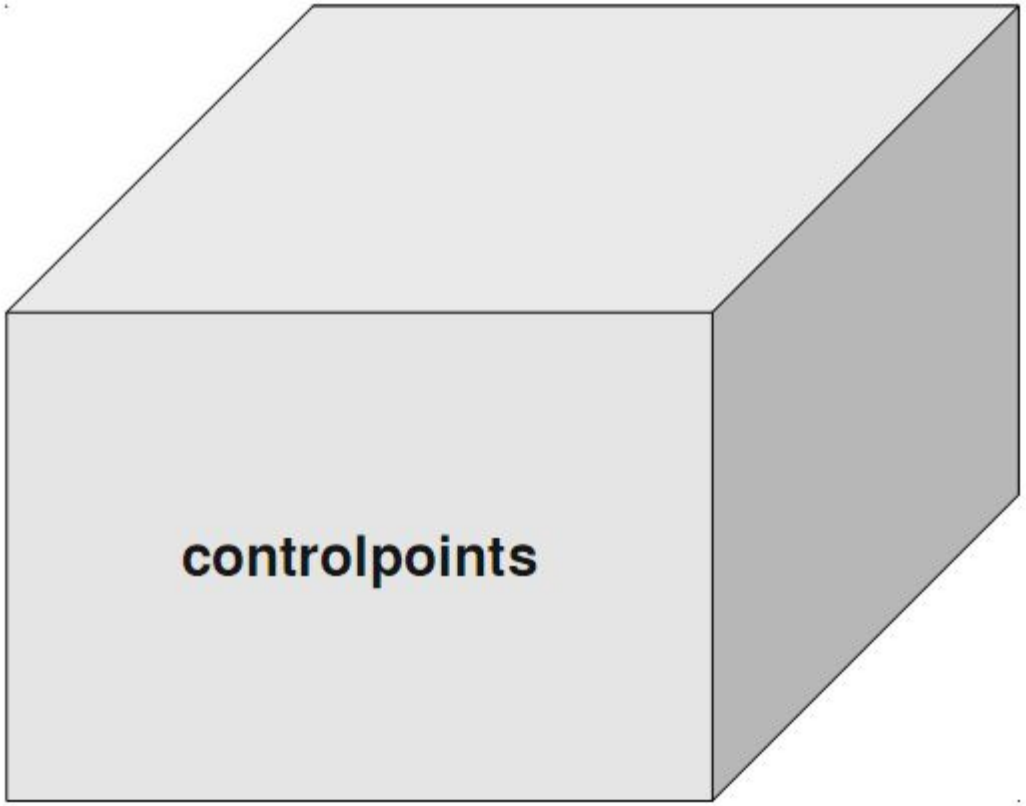
Nevermind the code…

```
' Copy And paste this code In your RhinoScript Editor (Tools □ RhinoScript □ Edit…)
' This Is a basic script To draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start With it
DrawCurve  ' this tells the program what subroutine To run

Sub DrawCurve ' this Is the code To Execute when “DrawCurve” Is called above
    Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
    Dim controlpoints(1)  ' controlpoints is an array of 3-D points (see next slide)

    controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
    controlpoints(1) = Array(10,5,15) ' x = 10, y = 5, z = 15

    Rhino.Print ( "Curve ID: " + Rhino.AddCurve(controlpoints) ) ' this draws the curve and prints its I.D.
    Rhino.Print ( "Sphere ID: " + Rhino.AddSphere (controlpoints(1), 1) ) ' this draws a sphere and prints its I.D.

    Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
 Rhino.ZoomExtents ' and this adjusts the zoom level

End Sub ' this is the end of the "DrawCurve" subroutine
```

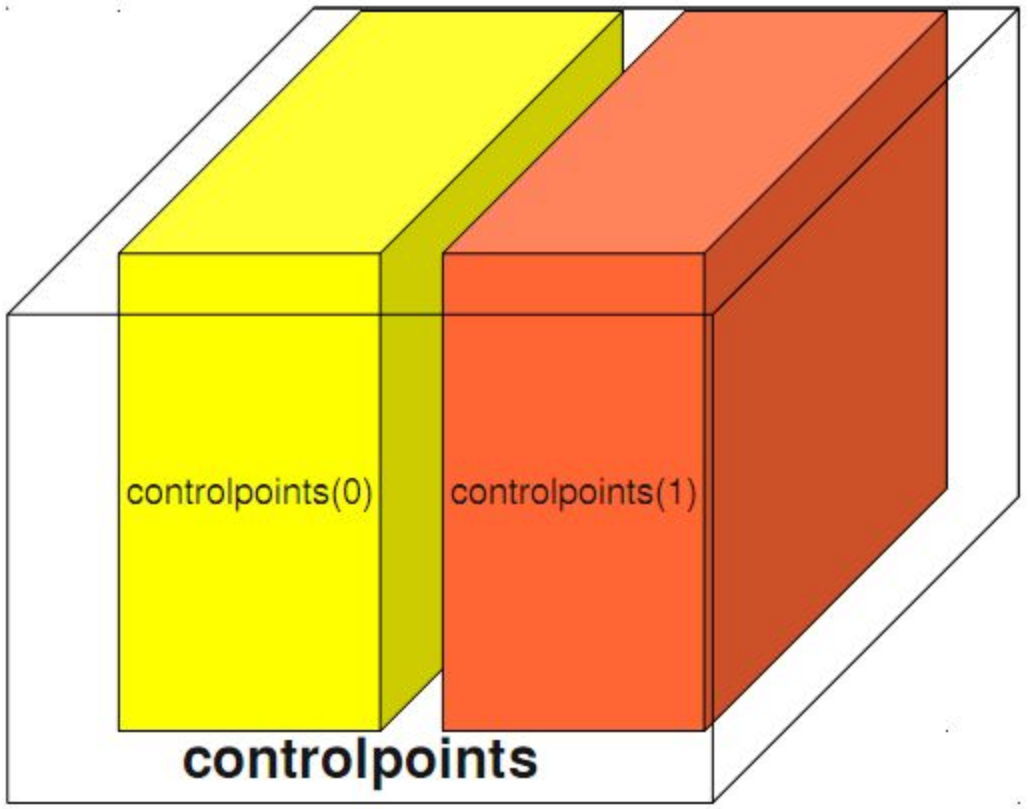

Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Drag a window to zoom (All Dynamic Extents Factor In Out Selected Target 1To1): _All
Command:
Top
Perspective
sphere of radius 1
Front
x=10, y=5, z=15
x=0, y=0, z=0
Right
Dr. Ricardo Sosa (rdsosam@itesm.mx)


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Drag a window to zoom (All Dynamic Extents Factor In Out Selected Target 1To1): _All
Command:
Top
Perspective
AddSphere(controlpoints(1), 1)
controlpoints()
controlpoints(0)
x0 y0 z0
0 0 0
controlpoints(1)
x1 y1 z1
10 5 15
Front
AddCurve(controlpoints)
Right
Dr. Ricardo Sosa (rdsosam@itesm.mx)

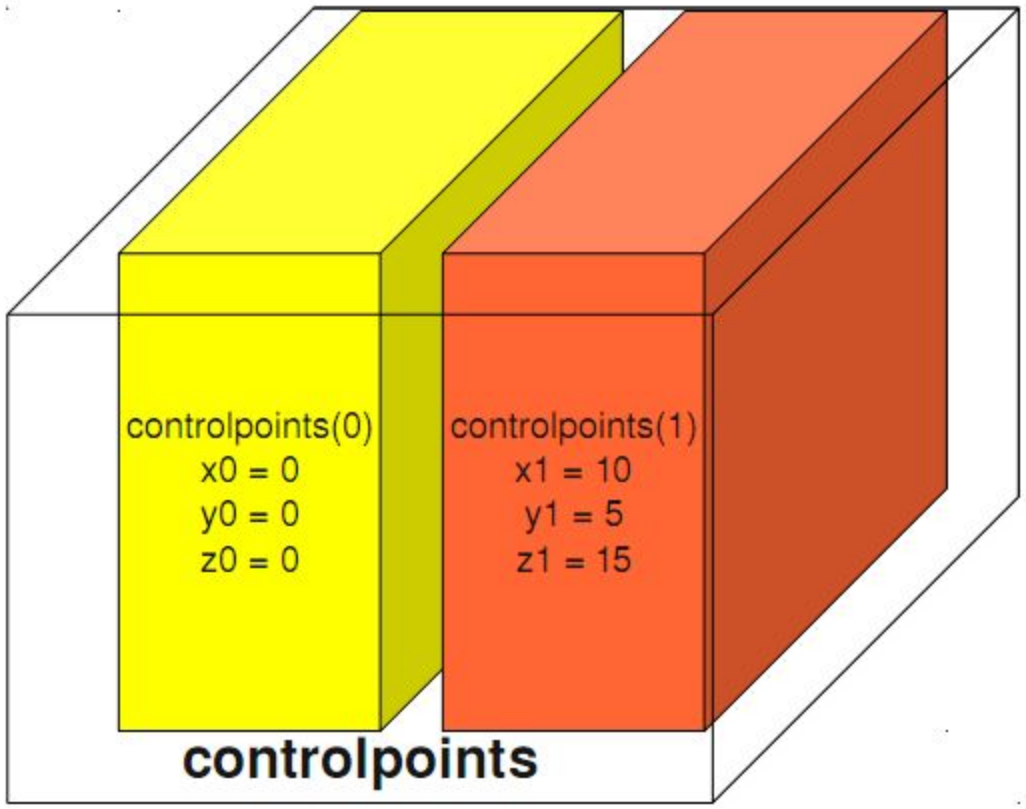
Hold on! What's an "array"?



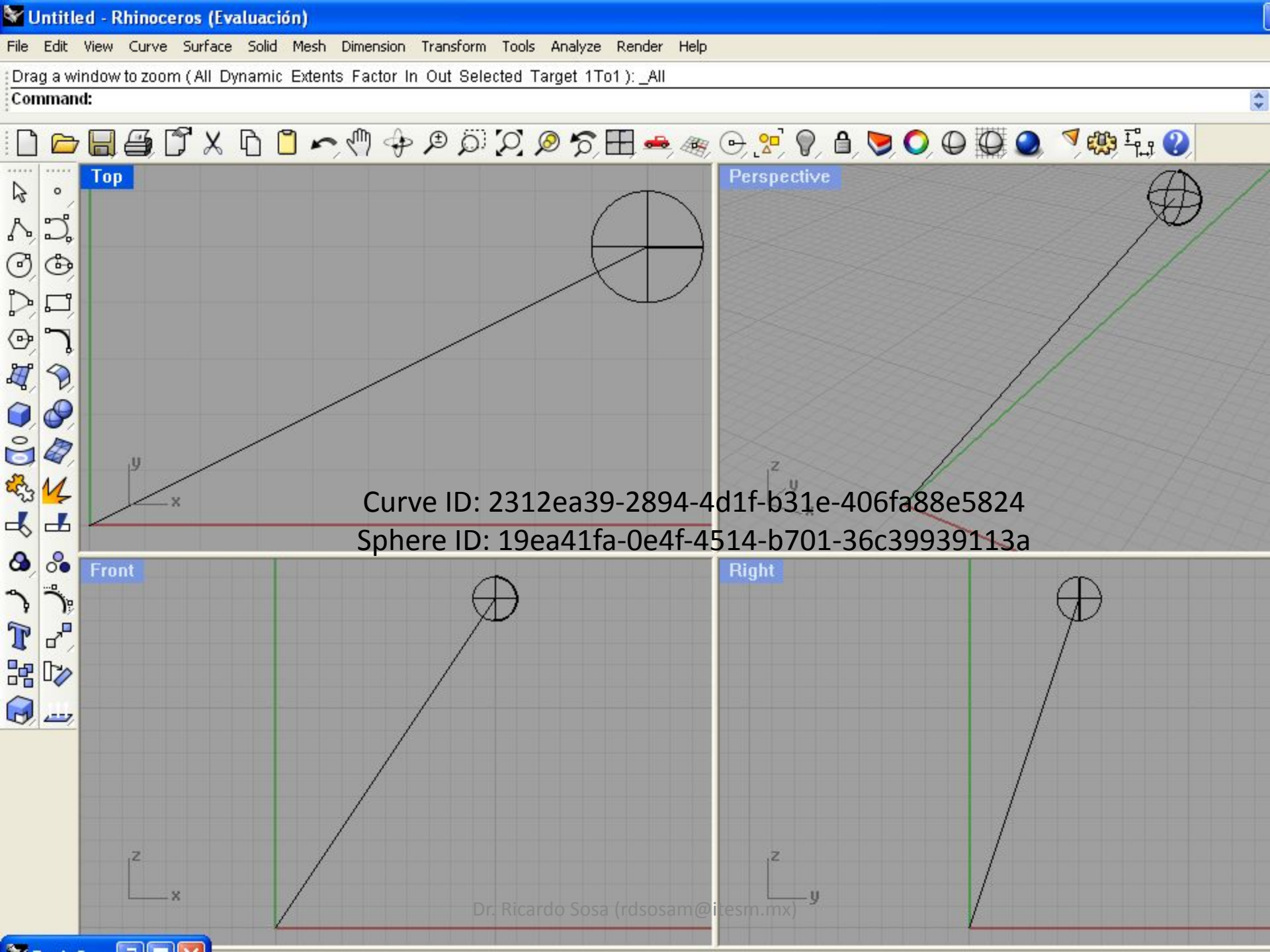
An array is like a box



A box that holds 3D coordinates

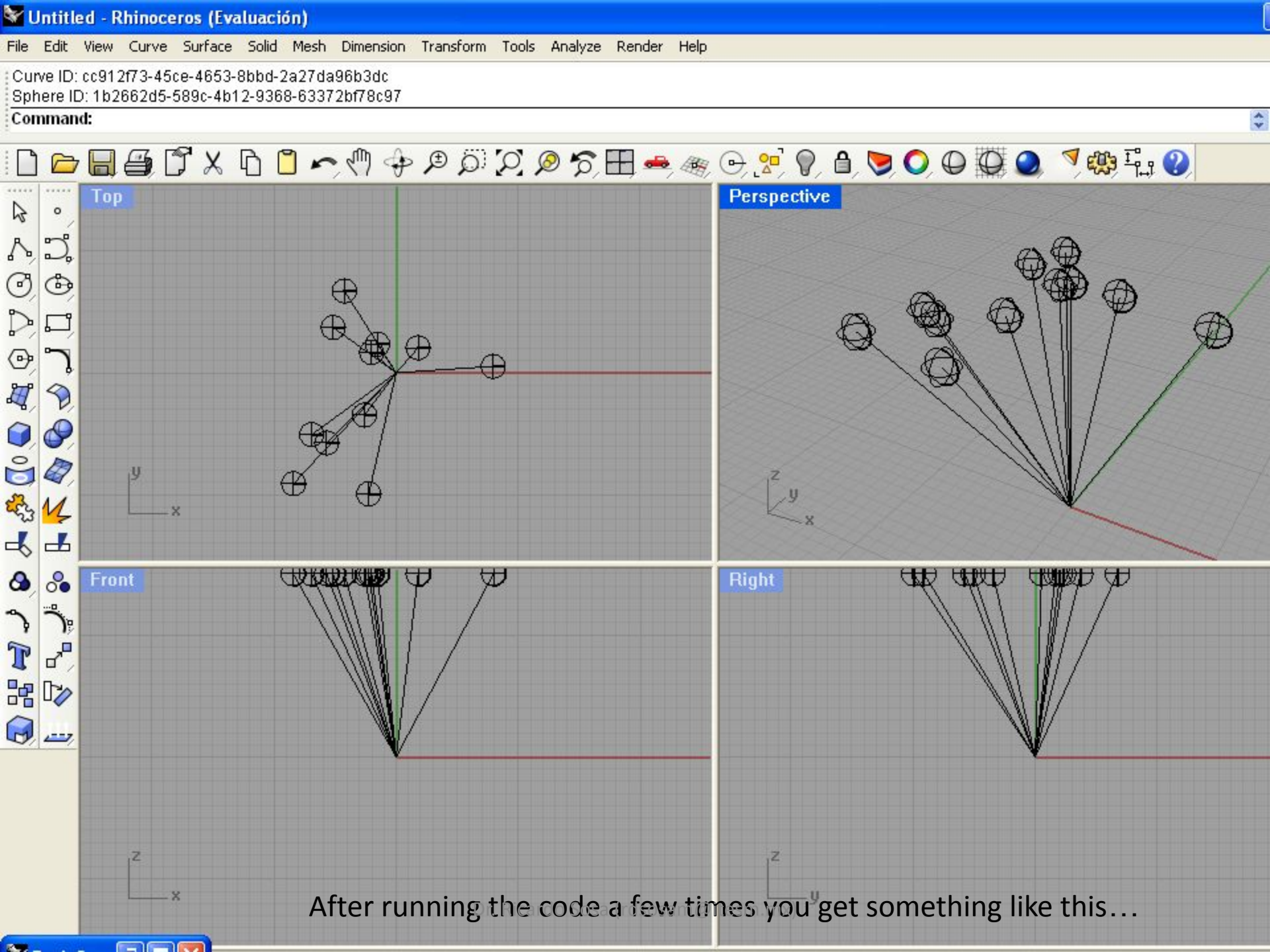


```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when "DrawCurve" is called above
    Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
    Dim controlpoints(1)  ' controlpoints is an array of 3-D points (see next slide)

    controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
    controlpoints(1) = Array(10,5,15) ' x = 10, y = 5, z = 15

    Rhino.Print ( "Curve ID: " + Rhino.AddCurve(controlpoints) ) ' this draws the curve and prints its I.D.
    Rhino.Print ( "Sphere ID: " + Rhino.AddSphere (controlpoints(1), 1) ) ' this draws a sphere and its I.D.

    Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
 Rhino.ZoomExtents ' and this adjusts the zoom level

End Sub ' this is the end of the "DrawCurve" subroutine
```


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Drag a window to zoom (All Dynamic Extents Factor In Out Selected Target 1To1): _All
Command:
Top
Perspective
Front
Right
Curve ID: 2312ea39-2894-4d1f-b31e-406fa88e5824
Sphere ID: 19ea41fa-0e4f-4514-b701-36c39939113a
Dr. Ricardo Sosa (rdsosam@itesm.mx)

Now some randomness…

```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when “DrawCurve” is called above
      Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
      Dim controlpoints(1)  ' controlpoints is an array of 3-D points (see next slide)
      controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
      controlpoints(1) = Array(randomBetween(-10,10),randomBetween(-10,10),15) ' x = random, y = random, z = 15
      Rhino.Print ( "Curve ID: " + Rhino.AddCurve(controlpoints) ) ' this draws the curve and prints its I.D.
      Rhino.Print ( "Sphere ID: " + Rhino.AddSphere (controlpoints(1), 1) ) ' this draws a sphere and its I.D.
      Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
  Rhino.ZoomExtents ' and this adjusts the zoom level
End Sub ' this is the end of the "DrawCurve" subroutine

Function randomBetween(min,max)  ' this is the code to generate random numbers between limits
    randomBetween = Rnd*(max-min)+min  ' returns a random number between the limits specified
End Function ' end of the randomness function
```


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Curve ID: cc912f73-45ce-4653-8bbd-2a27da96b3dc
Sphere ID: 1b2662d5-589c-4b12-9368-63372bf78c97
Command:
Top
Perspective
Front
Right
After running the code a few times you get something like this…

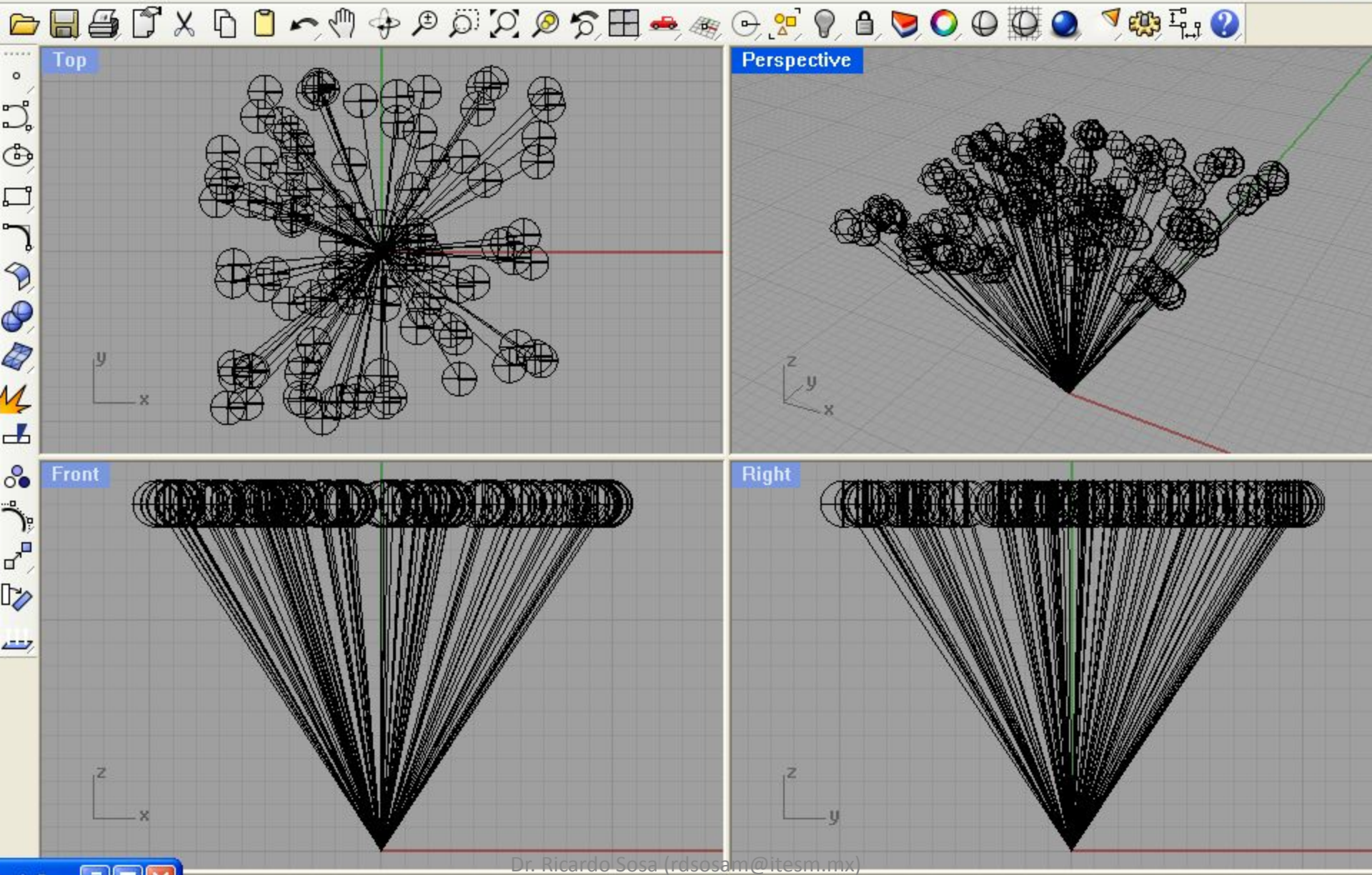
Now some recursion…

```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when “DrawCurve” is called above
        Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
        Dim controlpoints(1)  ' controlpoints is an array of 3-D points (see next slide)
        Dim i
        For i=0 To 100
          controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
          controlpoints(1) = Array(randomBetween(-10,10),randomBetween(-10,10),15) ' x = random, y = random, z = 15
          Rhino.AddCurve controlpoints ' this draws the curve
          Rhino.AddSphere controlpoints(1), 1  ' this draws a sphere
        Next
        Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
  Rhino.ZoomExtents ' and this adjusts the zoom level
End Sub ' this is the end of the "DrawCurve" subroutine

Function randomBetween(min,max)  ' this is the code to generate random numbers between limits
    randomBetween = Rnd*(max-min)+min  ' returns a random number between the limits specified
End Function ' end of the randomness function
```

Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help

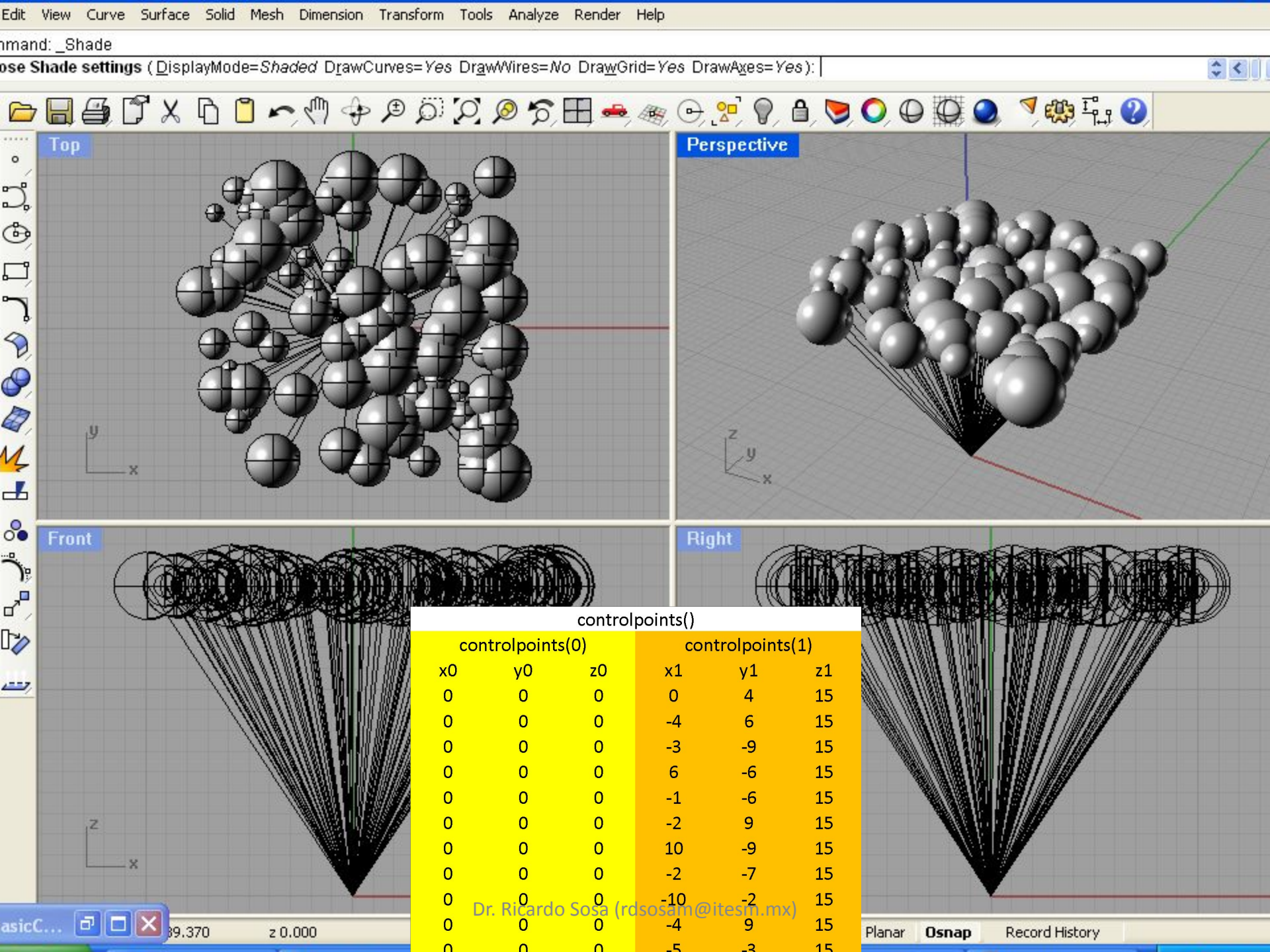
g a window to zoom (All Dynamic Extents Factor In Out Selected Target 1To1): _All
ose option (Extents Selected 1To1): _Extents

mand:



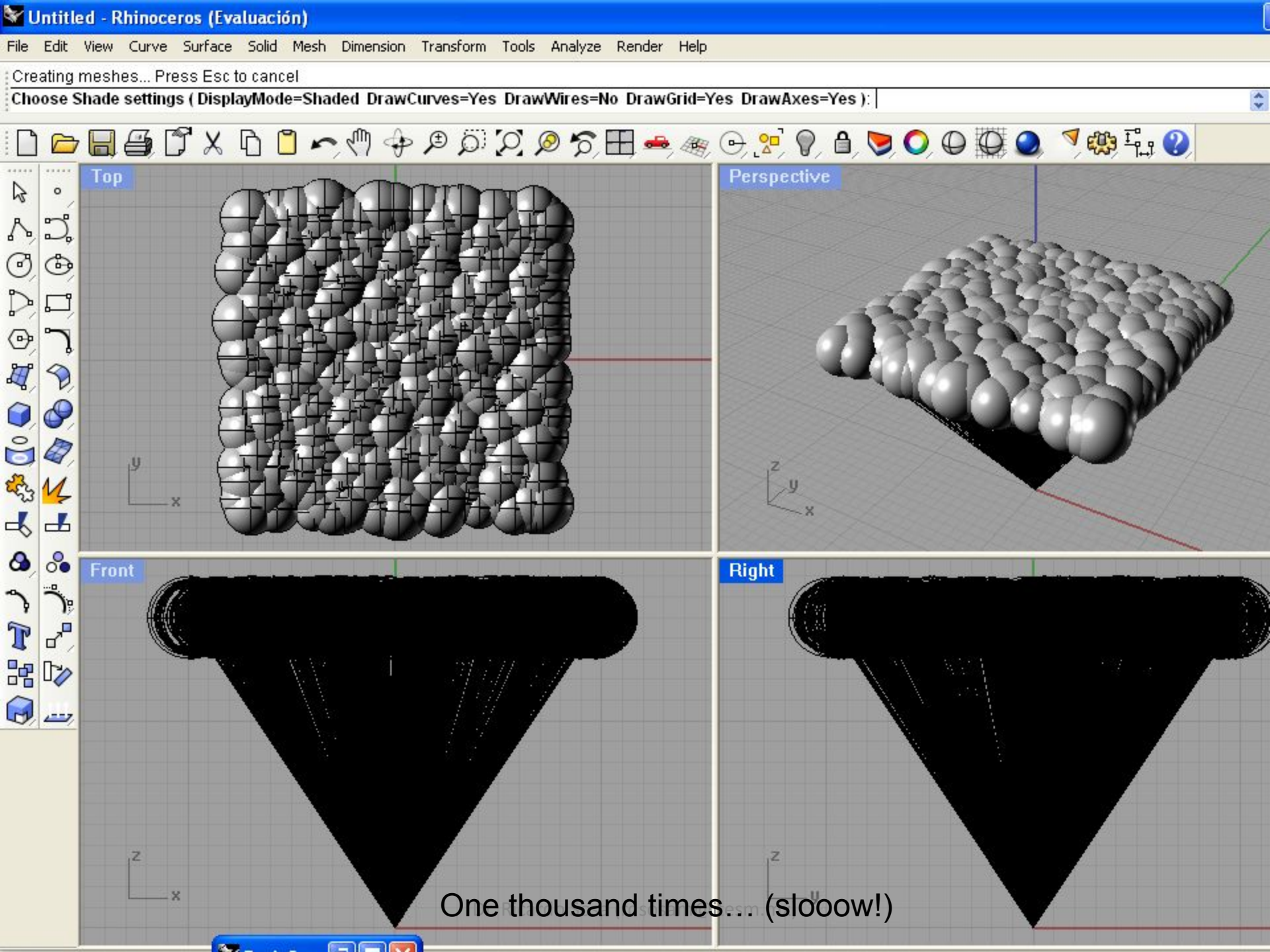
asicC...

22.390 z 0.000 Default Snap Ortho Planar Osnap Record History



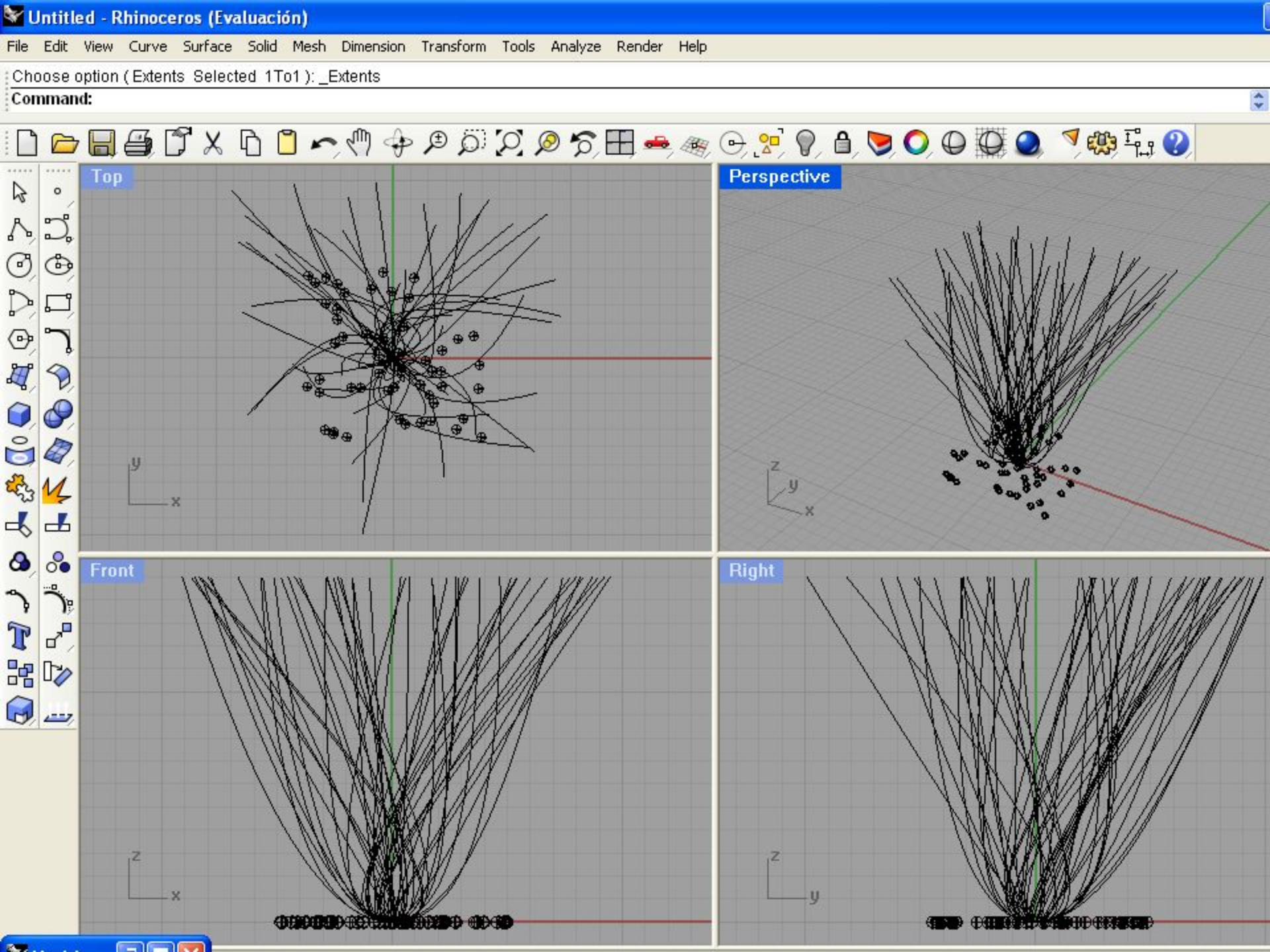
controlpoints()

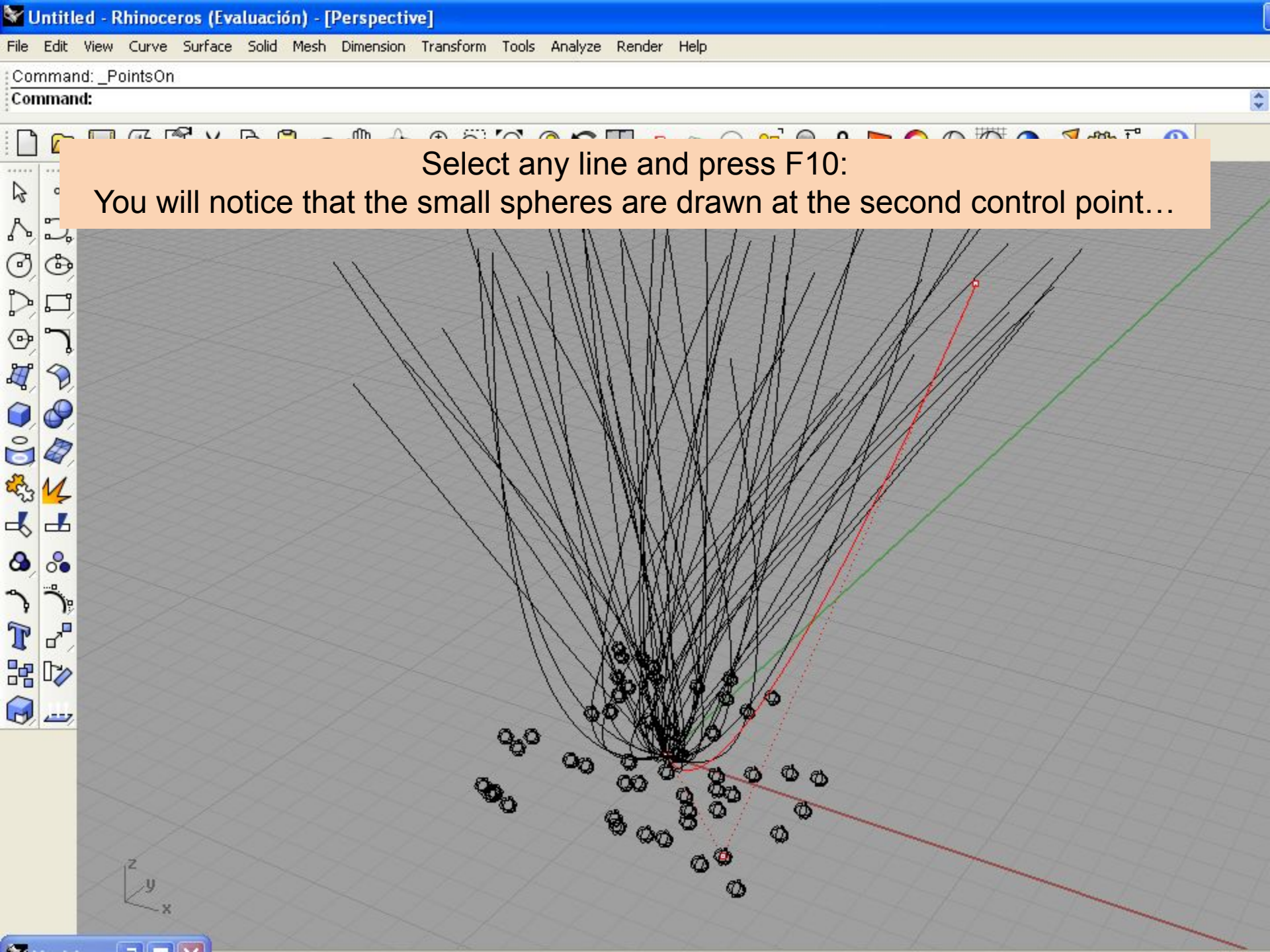
controlpoints(0)			controlpoints(1)		
x0	y0	z0	x1	y1	z1
0	0	0	0	4	15
0	0	0	-4	6	15
0	0	0	-3	-9	15
0	0	0	6	-6	15
0	0	0	-1	-6	15
0	0	0	-2	9	15
0	0	0	10	-9	15
0	0	0	-2	-7	15
0	0	0	-10	-2	15
0	0	0	-4	9	15
0	0	0	-5	-3	15


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Creating meshes... Press Esc to cancel
Choose Shade settings (DisplayMode=Shaded DrawCurves=Yes DrawWires=No DrawGrid=Yes DrawAxes=Yes):
Top
Perspective
Front
Right
One thousand times… (slooow!)

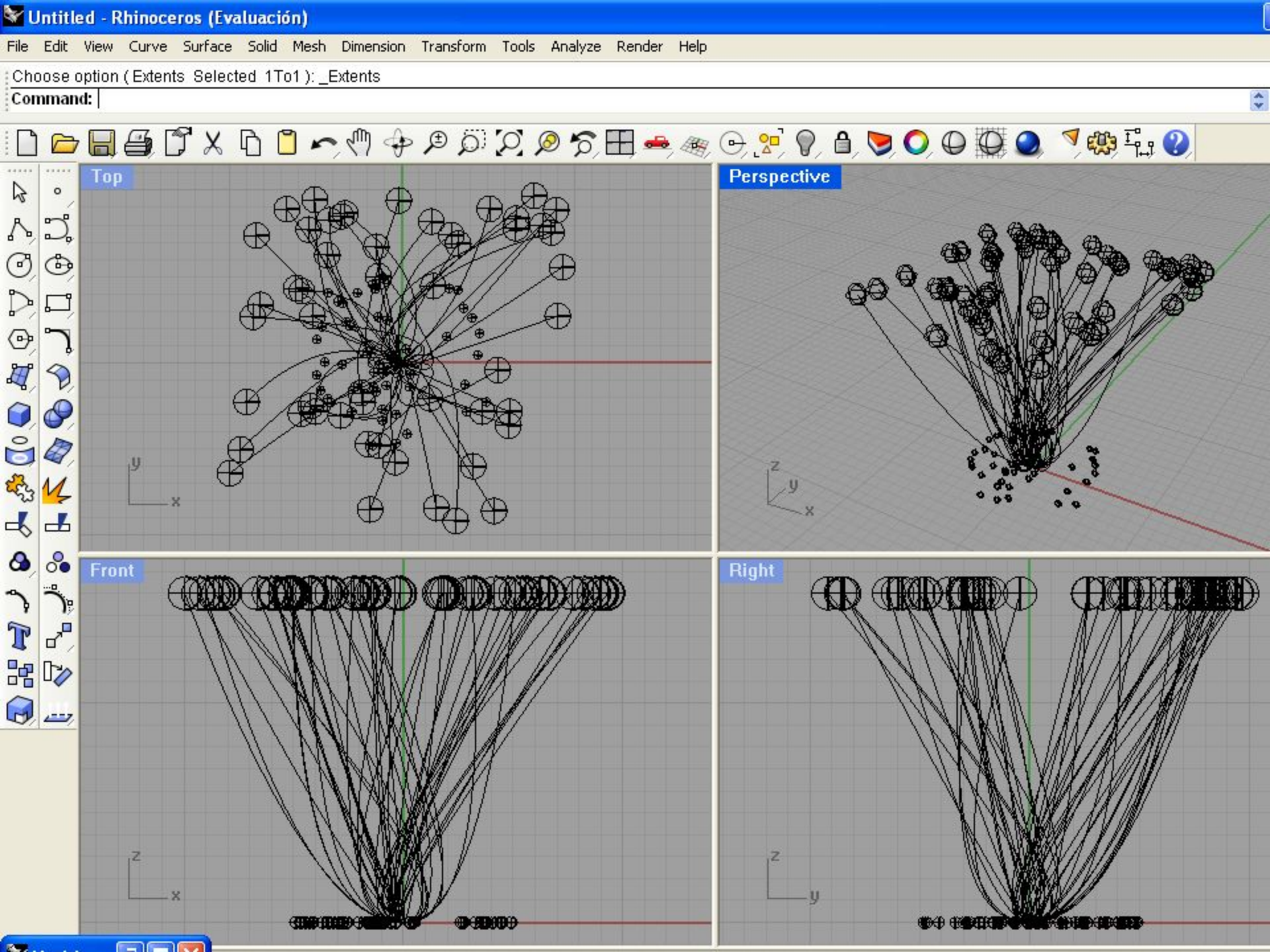
More interesting curves…

```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when “DrawCurve” is called above
        Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
        Dim controlpoints(2), i  ' controlpoints is an array of 3-D points (see next slide)
        For i=0 To 50
          controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
          controlpoints(1) = Array(randomBetween(-5,5),randomBetween(-5,5),0)
          controlpoints(2) = Array(randomBetween(-10,10),randomBetween(-10,10),15)
          Rhino.AddCurve controlpoints, 2 ' this draws the curve of two degrees now
          Rhino.AddSphere controlpoints(1), 0.25  ' this draws a small sphere at second point
        Next
        Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
  Rhino.ZoomExtents ' and this adjusts the zoom level
End Sub ' this is the end of the "DrawCurve" subroutine

Function randomBetween(min,max)  ' this is the code to generate random numbers between limits
    randomBetween = Rnd*(max-min)+min  ' returns a random number between the limits specified
End Function ' end of the randomness function
```


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Choose option (Extents Selected 1To1): _Extents
Command:
Top
Perspective
Front
Right


Untitled - Rhinoceros (Evaluación) - [Perspective]
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command: _PointsOn
Command:
Select any line and press F10:
You will notice that the small spheres are drawn at the second control point…

```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when “DrawCurve” is called above
      Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
      Dim controlpoints(2), i  ' controlpoints is an array of 3-D points (see next slide)
      For i=0 To 50
        controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
        controlpoints(1) = Array(randomBetween(-5,5),randomBetween(-5,5),0)
        controlpoints(2) = Array(randomBetween(-10,10),randomBetween(-10,10),15)
        Rhino.AddCurve controlpoints, 2 ' this draws the curve
        Rhino.AddSphere controlpoints(1), 0.25  ' this draws a small sphere at second point
        Rhino.AddSphere controlpoints(2), 0.75  ' this draws a big sphere at third point
      Next
      Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
  Rhino.ZoomExtents ' and this adjusts the zoom level
End Sub ' this is the end of the "DrawCurve" subroutine

Function randomBetween(min,max)  ' this is the code to generate random numbers between limits
    randomBetween = Rnd*(max-min)+min  ' returns a random number between the limits specified
End Function ' end of the randomness function
```


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Choose option (Extents Selected 1To1): _Extents
Command:
Top
Perspective
Front
Right

Time for a challenge…

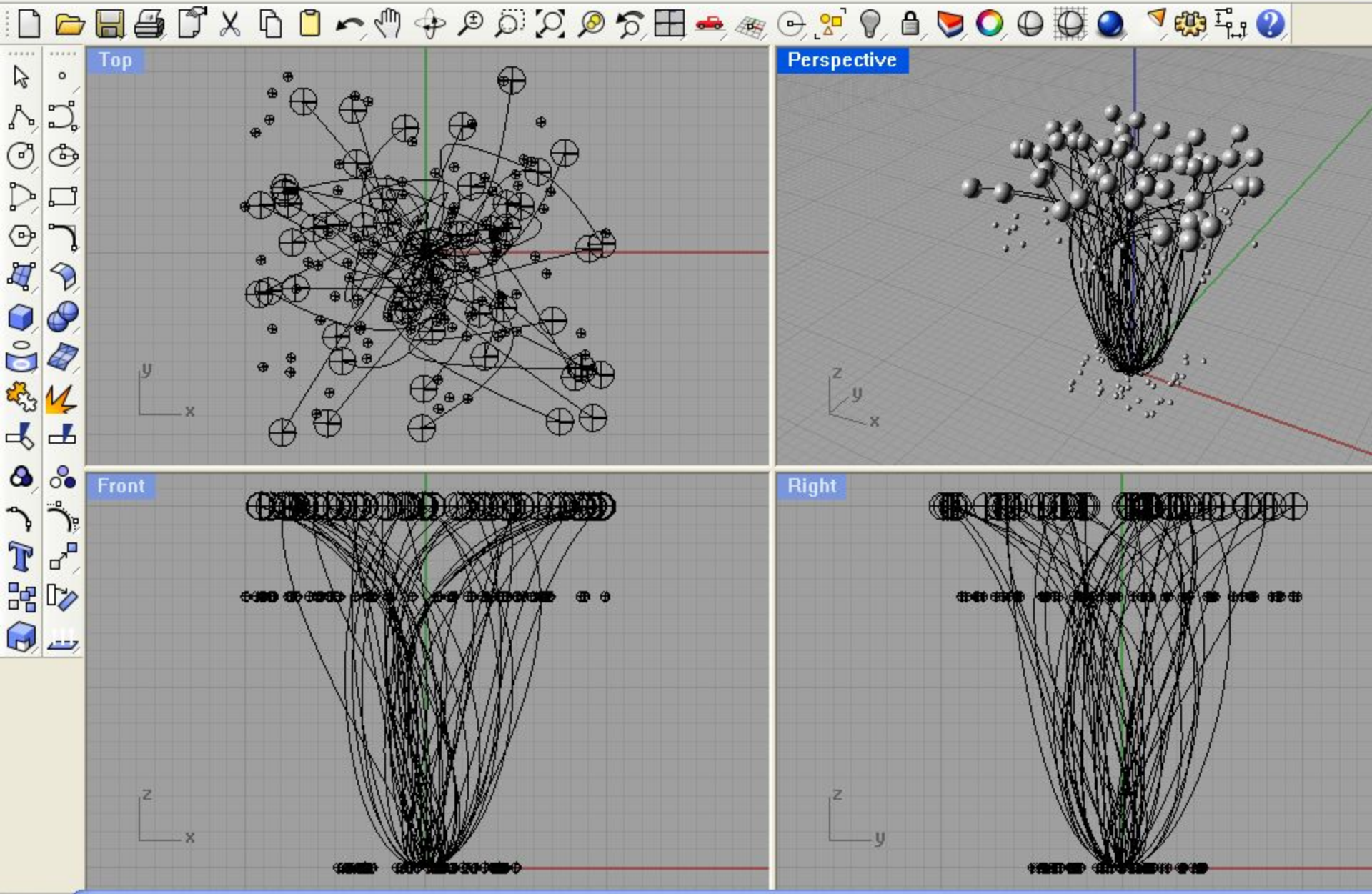
How do you achieve the following?

Untitled - Rhinoceros (Evaluación)

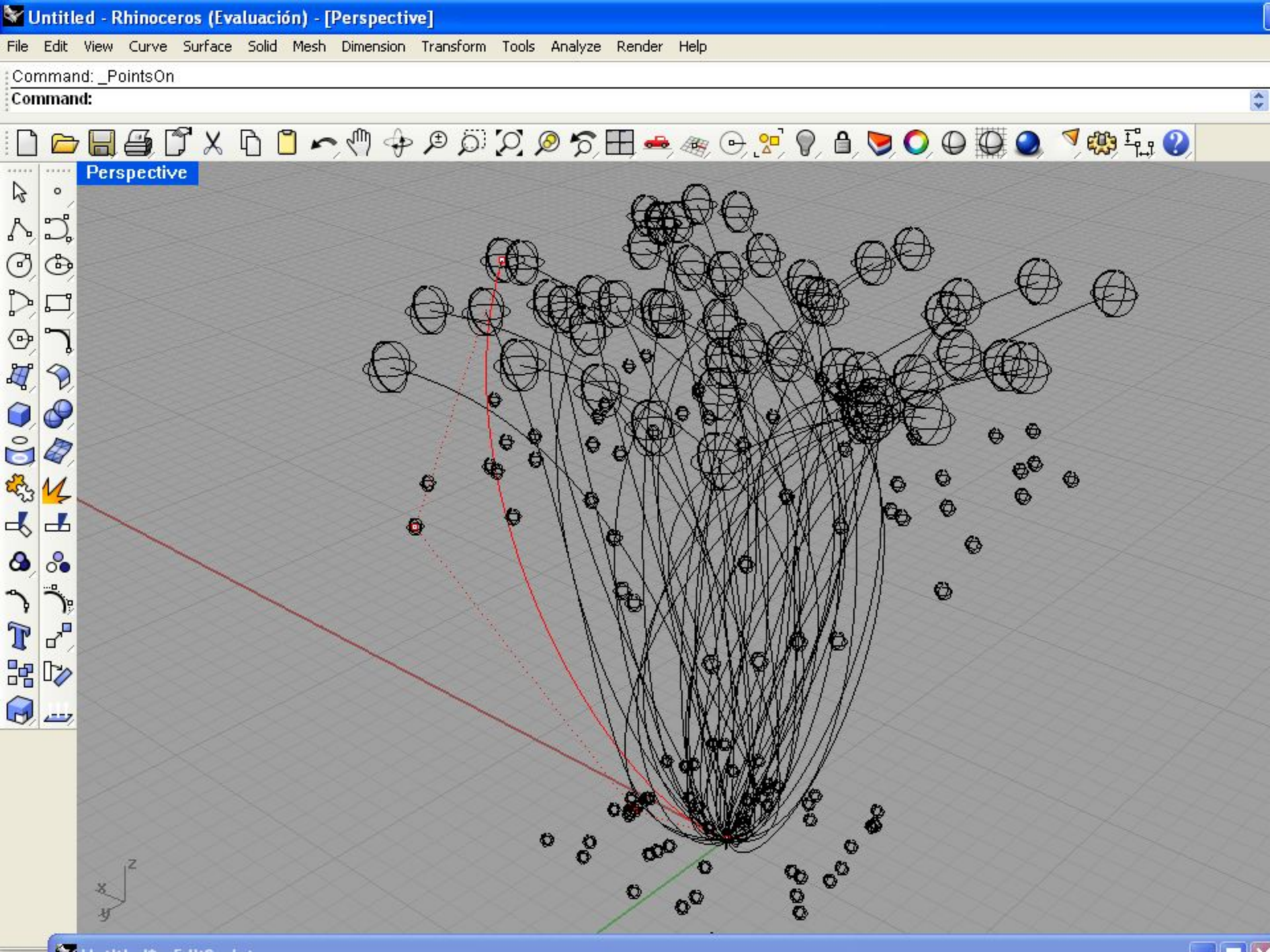
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help

Creating meshes... Press Esc to cancel

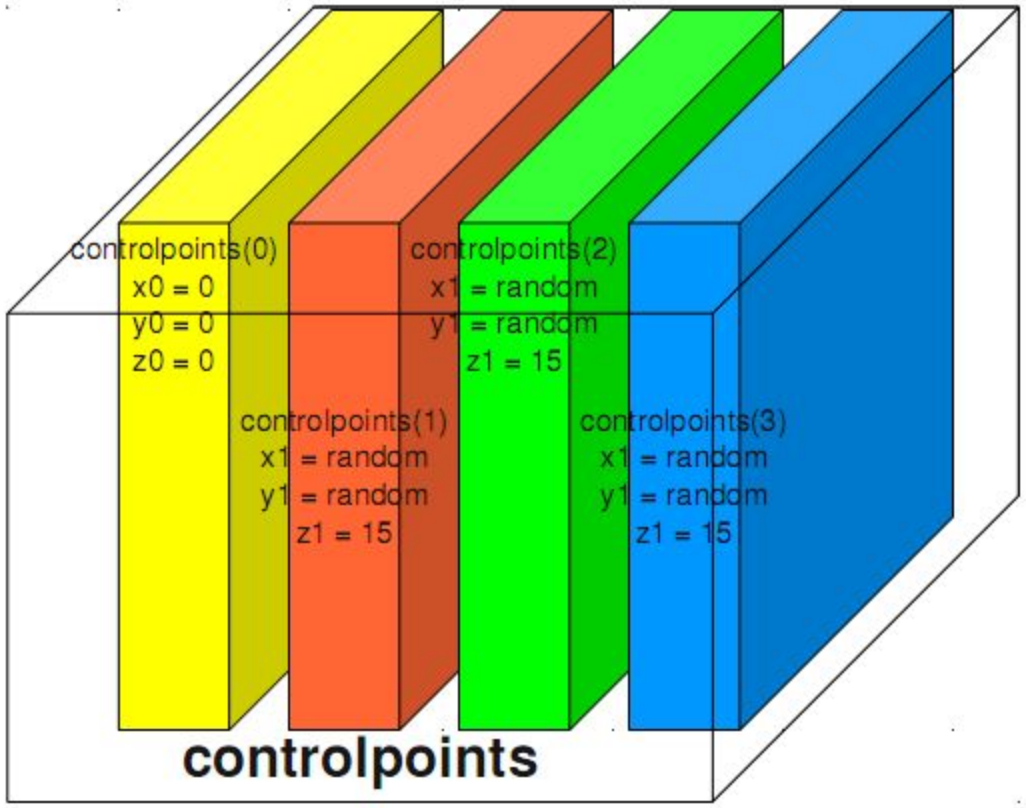
Choose Shade settings (DisplayMode=*Shaded* DrawCurves=*Yes* DrawWires=*No* DrawGrid=*Yes* DrawAxes=*Yes*):



```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when “DrawCurve” is called above
      Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
      Dim controlpoints(3), i  ' controlpoints is an array of 3-D points (see next slide)
      For i=0 To 50
        controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
        controlpoints(1) = Array(randomBetween(-5,5),randomBetween(-5,5),0)
        controlpoints(2) = Array(randomBetween(-10,10),randomBetween(-10,10),15)
        controlpoints(3) = Array(randomBetween(-10,10),randomBetween(-10,10),20)
        Rhino.AddCurve controlpoints, 3 ' this draws the curve
        Rhino.AddSphere controlpoints(1), 0.25  ' this draws a small sphere at second point
        Rhino.AddSphere controlpoints(2), 0.25  ' this draws a big sphere at third point
        Rhino.AddSphere controlpoints(3), 0.75  ' this draws a big sphere at third point
      Next
      Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
  Rhino.ZoomExtents ' and this adjusts the zoom level
End Sub ' this is the end of the "DrawCurve" subroutine

Function randomBetween(min,max)  ' this is the code to generate random numbers between limits
    randomBetween = Rnd*(max-min)+min  ' returns a random number between the limits specified
End Function ' end of the randomness function
```


Untitled - Rhinoceros (Evaluación) - [Perspective]
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command: _PointsOn
Command:
Perspective

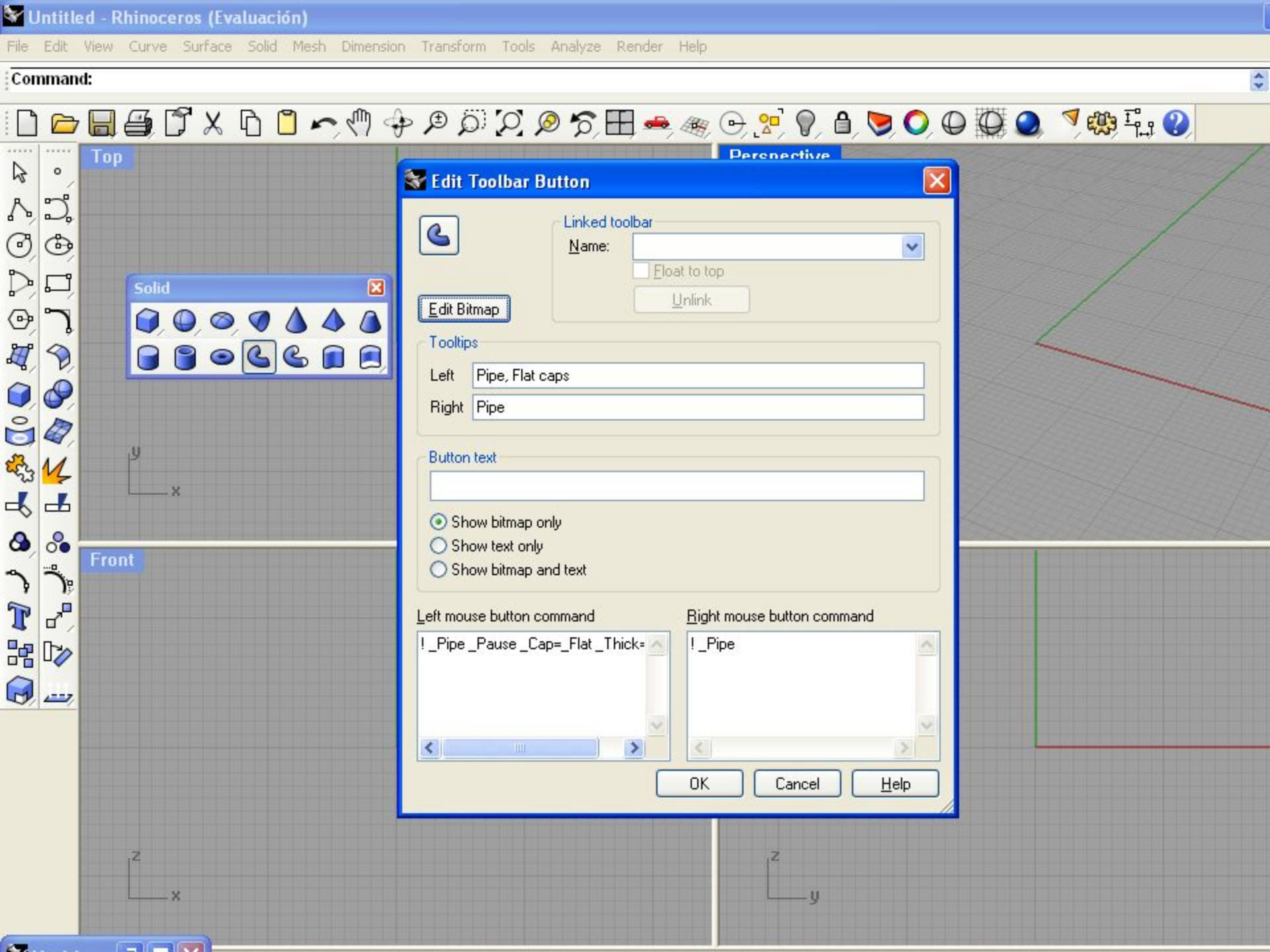
Add another set of coordinates…



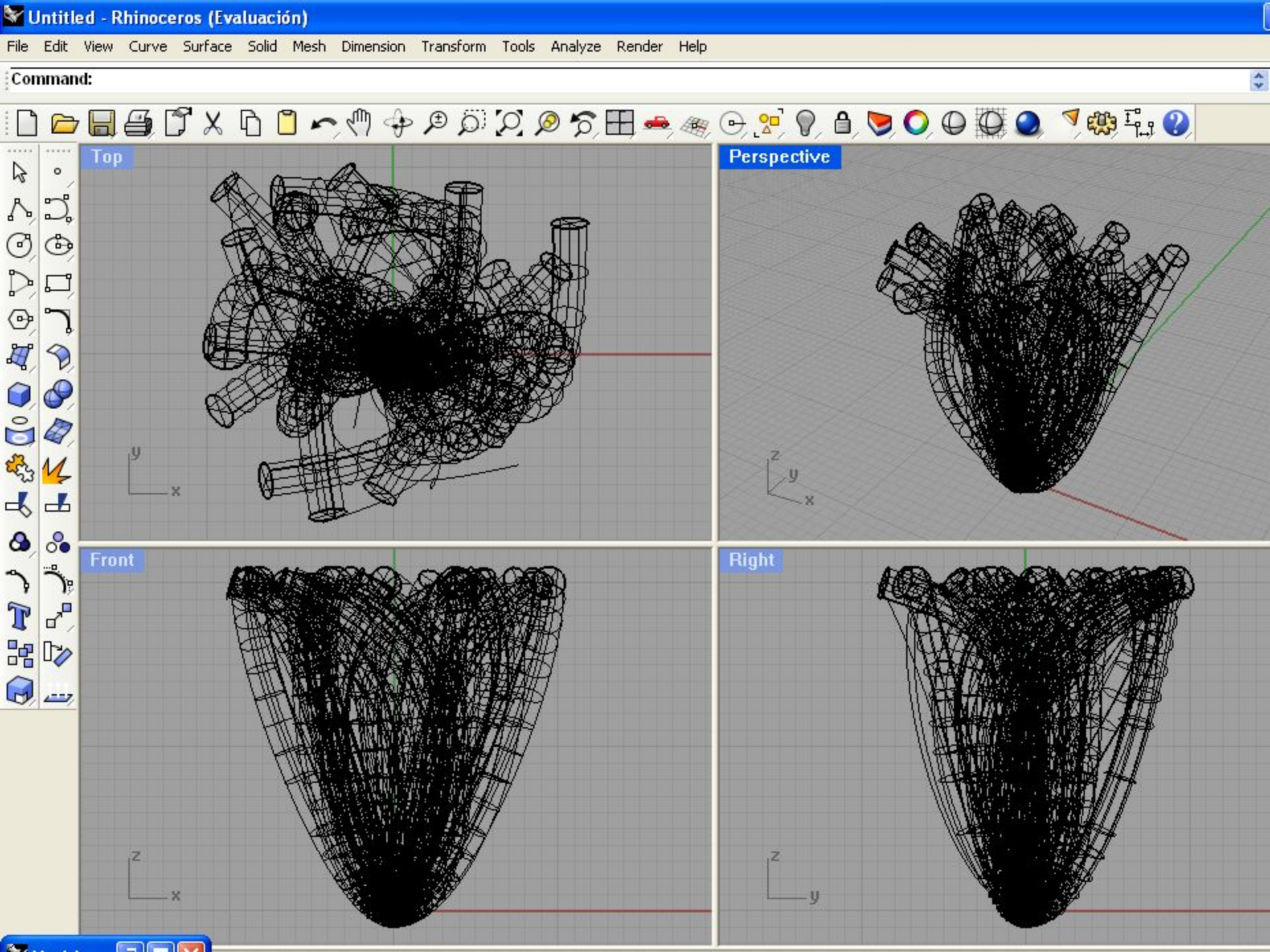
Rhino.Command "anycommand"

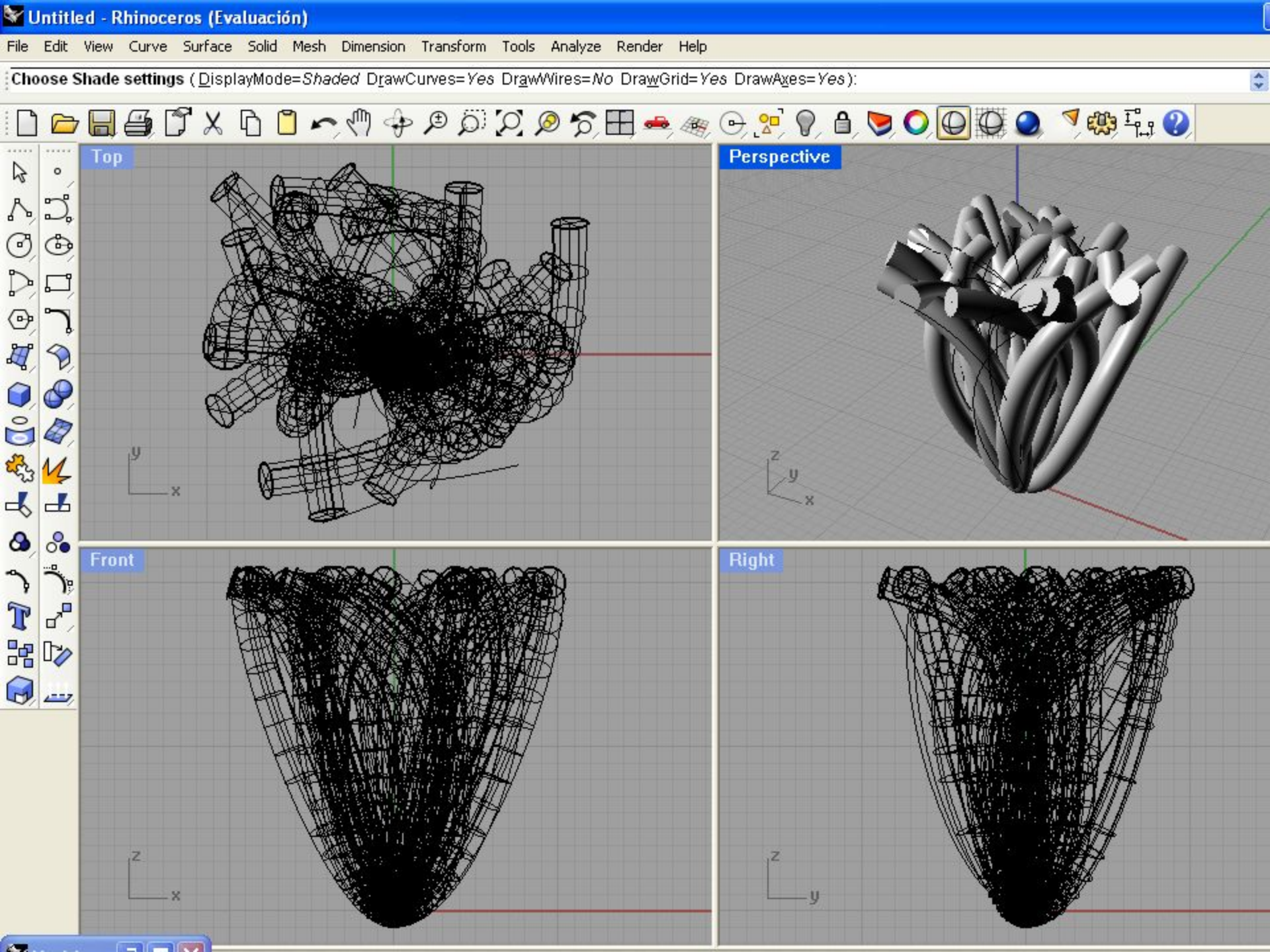
Shift + Rightclick any tool icon to see its _Command

Dr. Ricardo Sosa (rdsosam@itesm.mx)


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command:
Top
Perspective
Front
Solid
Edit Toolbar Button
Linked toolbar
Name:
Float to top
Unlink
Edit Bitmap
Tooltips
Left
Pipe, Flat caps
Right
Pipe
Button text
Show bitmap only
Show text only
Show bitmap and text
Left mouse button command
! _Pipe _Pause _Cap=_Flat _Thick=
Right mouse button command
! _Pipe
OK
Cancel
Help

```
' Copy and paste this code in your RhinoScript Editor (Tools  RhinoScript  Edit…)
' This is a basic script to draw a curve with fixed coordinates (Not very useful, but a good starting point)

Option Explicit ' nevermind this, just make sure that your scripts always start with it
DrawCurve  ' this tells the program what subroutine to run

Sub DrawCurve ' this is the code to run when “DrawCurve” is called above
      Call Rhino.enableRedraw(False) ' nevermind this, it speeds up the execution of the code
      Dim controlpoints(3), i  ' controlpoints is an array of 3-D points (see next slide)
      Dim strCmd, curveID
      For i=0 To 50
        controlpoints(0) = Array(0,0,0) ' x = 0, y = 0, z = 0
        controlpoints(1) = Array(randomBetween(-5,5),randomBetween(-5,5),0)
        controlpoints(2) = Array(randomBetween(-10,10),randomBetween(-10,10),15)
        controlpoints(3) = Array(randomBetween(-10,10),randomBetween(-10,10),20)
        curveID = Rhino.AddCurve(controlpoints, 3) ' this draws the curve
        Rhino.SelectObject(curveID)
        Rhino.Command "_Pipe " & 1.0 & " Enter "  & 1.0 & " Enter"
      Next
      Call Rhino.enableRedraw(True) ' nevermind this, it refreshes the screen
  Rhino.ZoomExtents ' and this adjusts the zoom level
End Sub ' this is the end of the "DrawCurve" subroutine

Function randomBetween(min,max)  ' this is the code to generate random numbers between limits
    randomBetween = Rnd*(max-min)+min  ' returns a random number between the limits specified
End Function ' end of the randomness function
```


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Command:
Top
Perspective
Front
Right


Untitled - Rhinoceros (Evaluación)
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Help
Choose Shade settings (DisplayMode=Shaded DrawCurves=Yes DrawWires=No DrawGrid=Yes DrawAxes=Yes):
Top
Perspective
Front
Right

RhinoScript

Ocultar Atrás Adelante Imprimir

Contenido Índice Búsqueda

Escriba la palabra clave a buscar:

Command

- Command
- CommandHistory
- CompareGeometry
- ConvertCurveToPolyline
- CopyMaterial
- CopyObject
- CopyObjects
- CopyObjectsToLayer
- CosH
- CreatePreviewImage
- CreateShortcut
- CreateSolid
- CullDuplicateNumbers
- CullDuplicatePoints
- CullDuplicateStrings
- CurrentDetail
- CurrentDimStyle
- CurrentHatchPattern
- CurrentLayer
- CurrentModelInfo
- CurrentPrinter
- CurrentView
- Curve Methods
- CurveArcLengthPoint
- CurveArea
- CurveAreaCentroid
- CurveArrows
- CurveBooleanDifference
- CurveBooleanIntersection
- CurveBooleanUnion
- CurveBrepIntersect
- CurveClosestObject
- CurveClosestPoint
- CurveContourPoints
- CurveCurvature
- CurveCurveIntersection
- CurveDegree
- CurveDeviation
- CurveDim
- CurveDirectionsMatch
- CurveDiscontinuity

Mostrar

Command

Runs a Rhino command script. All Rhino commands can be used in command scripts. The command can be a build-in Rhino command or a command that is provided by a 3rd party plug-in.

Write command scripts just as you would type the command sequence at the command line. A space betwee characters or a new line act like pressing <Enter> at the command line. For more information on writing command scripts, see "Scripting" in the Rhino help file.

Note, this method is designed to run one command and one command only. Do not combine multiple Rhino commands into a single call to this method. For example:

WRONG:

```
Rhino.Command "_Line _SelLast _Invert"
```

CORRECT:

```
Rhino.Command "_Line"
Rhino.Command "_SelLast"
Rhino.Command "_Invert"
```

Also, the exclamation point and space character (!) combination used by button macros and batch-driven scripts to cancel the previous command is not valid. For example:

WRONG:

```
Rhino.Command "! _Line _Pause _Pause"
```

CORRECT:

```
Rhino.Command "_Line _Pause _Pause"
```

After the command script has run, you can obtain the identifiers of most recently created or changed object calling LastCreatedObjects.

Syntax

```
Rhino.Command (strCommand [, blnEcho])
```

After you add a curve, select it with:

Rhino.SelectObject(curveID)

Then apply the command:

Rhino.Command "_Pipe " & 1.0 & " Enter " & 1.0 & " Enter"

Due next class…

Do something interesting of your own!