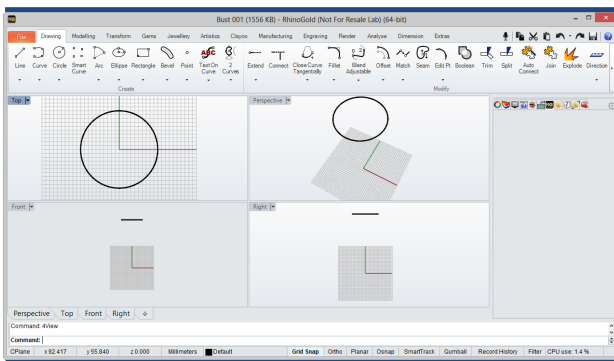




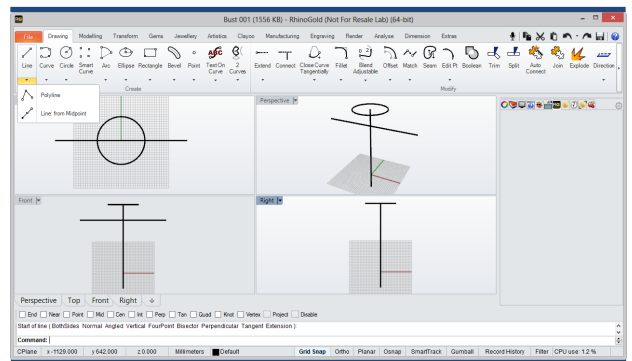
## Bust

In this tutorial we will try out some of the most useful commands in RhinoGold. Powerful tools such as Network, Revol, Loft and Offset Surface.



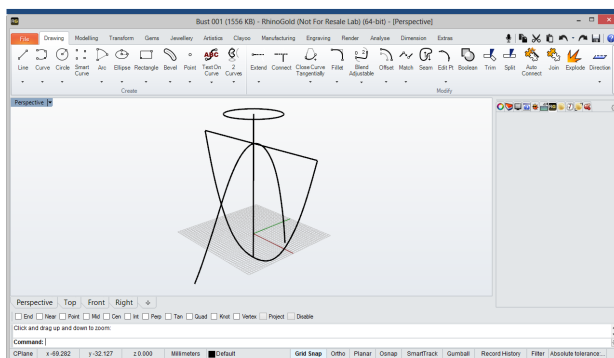
### 1 Circle

Under the Drawing tab with the circle tool make a circle with a diameter of 100mm and move it 220mm along the Z axis.



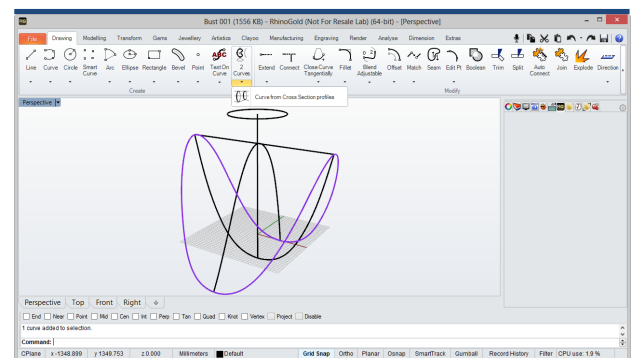
### 2 Line

Still in the Drawing tab with the Line command in the front view make a line with a length of 270mm starting from the center of the circle and another one with a length of 280mm drawn 50mm under the circle as seen in the image above. These will be our support lines for the next steps.



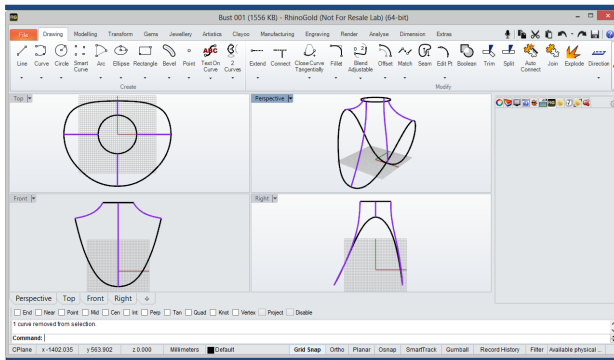
### 3 Curve

Now, with the curve tool under the Drawing tab make two arcs, one in the front view and another in the right view using the previous lines as reference.



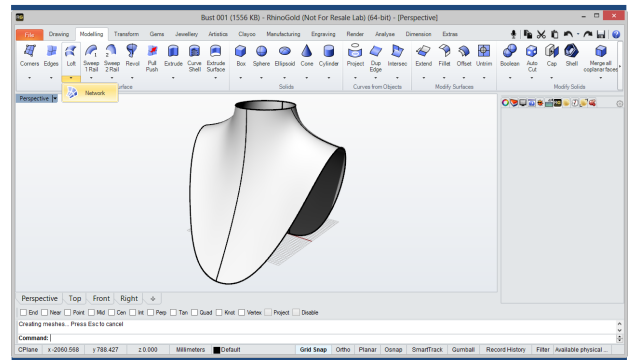
### 4 Curve from Two Views

In the same tab select both curves created in the previous step and choose the 2Curves tool to create another one.



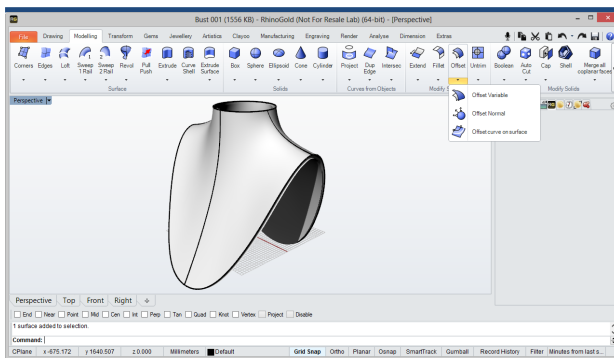
## 5 Curve

Now we can create four curves under the Drawing tab as seen in the image above.



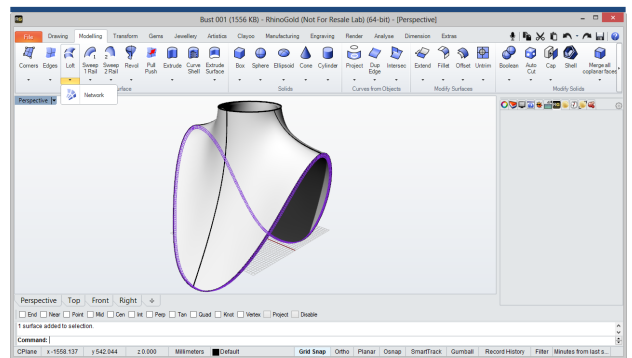
## 6 Network Surface

Under the Modelling tab use the Network tool use all the curves to create a surface for the Bust.



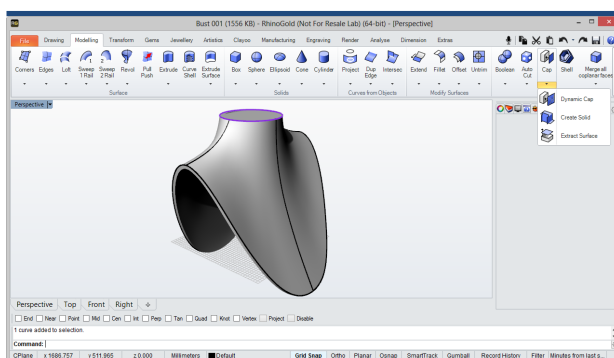
## 7 Offset Surface

We can then select the surface. Under the Modelling tab choose the Offset Surface Tool with a 6mm distance from the original to the inner side.



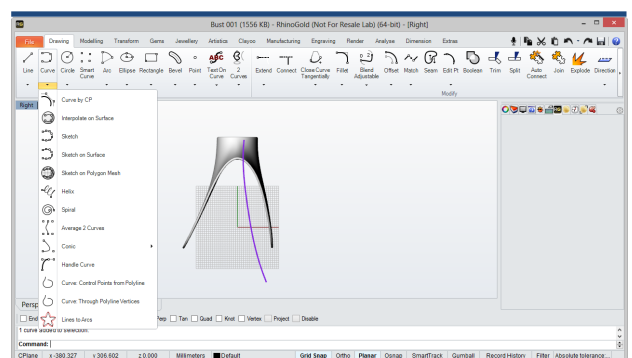
## 8 Loft Surface

Now it's time to run the Loft Surface command to make a section between the two surfaces, for that, select the out sides of each one. Then, use the Join tool to unite them.



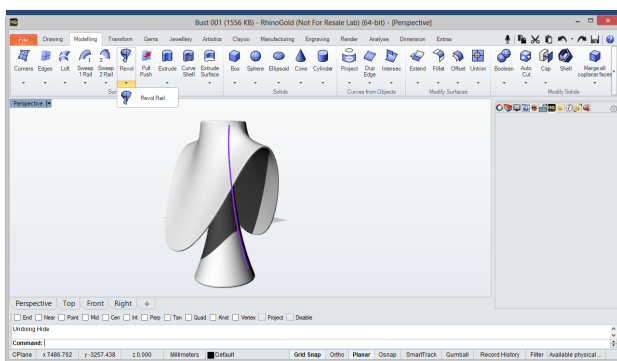
## 9 Cap

The next step under the Modelling tab consists of the creation of top surface of the bust. For that we use the Cap tool.



## 10 Curve

Now we can make a curve in the right view as you can see in the image above. This will be the curve to create the base.



## 11

### Revol Surface

Under the Modeling tab, with the Revol tool, select the curve created in the previous step and define an angle of 360° then cap the top and the bottom of this section. We can then use the Boolean Union tool to unite the parts as one single object.

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