Mix meshes using Symvol

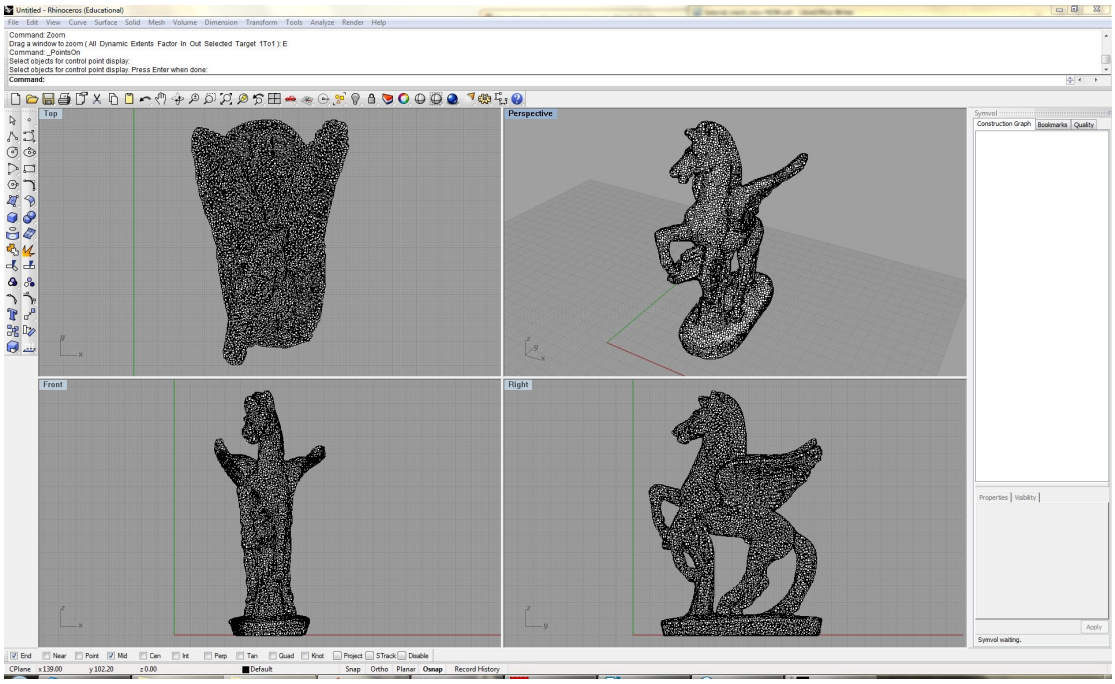
All the steps below can be followed using the exact parameter values in the corresponding 3dm and vol files named meshmix_stx, where x is the step number.

The original mesh models are available in the data archive. The "fertility" model is provided courtesy of UU by the AIM@SHAPE Shape Repository. The "pegasus" model is provided courtesy of CNR IMATI by the AIM@SHAPE Shape Repository.

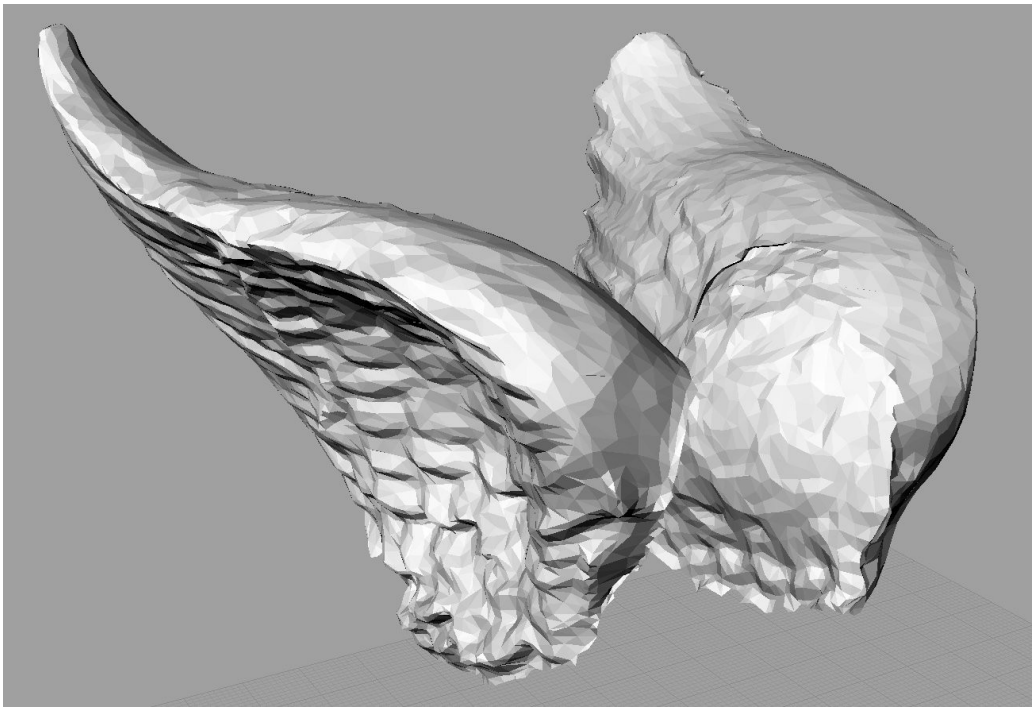
TIP: We recommend whenever using Symvol that you work in Flat Shaded mode. This will render the the objects in their true form, without any rendering tricks applied, which can be deceiving. This is particularly important when working with mesh data imported via vMesh or when exporting meshes.

Step 1

Import the test file Pegasus.stl. by using the standard Rhino File > Import command. Press F10 to turn on control point display and select the Pegasus mesh.



Double click on Perspective view and begin removing all the control points to isolate just the wings. The end result will look something like this:



For more control later when mixing the wings with another mesh, separate the wings into 2 meshes with Rhino's Mesh > Mesh Repair Tools > Split Disjointed Mesh, then select the mesh.

Step 2

Select vMesh from the Volume menu or type the vMesh command and click on one of the now separate wings. vMesh will effectively run the command vMeshRepair first, which will create a first class volume (3 manifold volume).

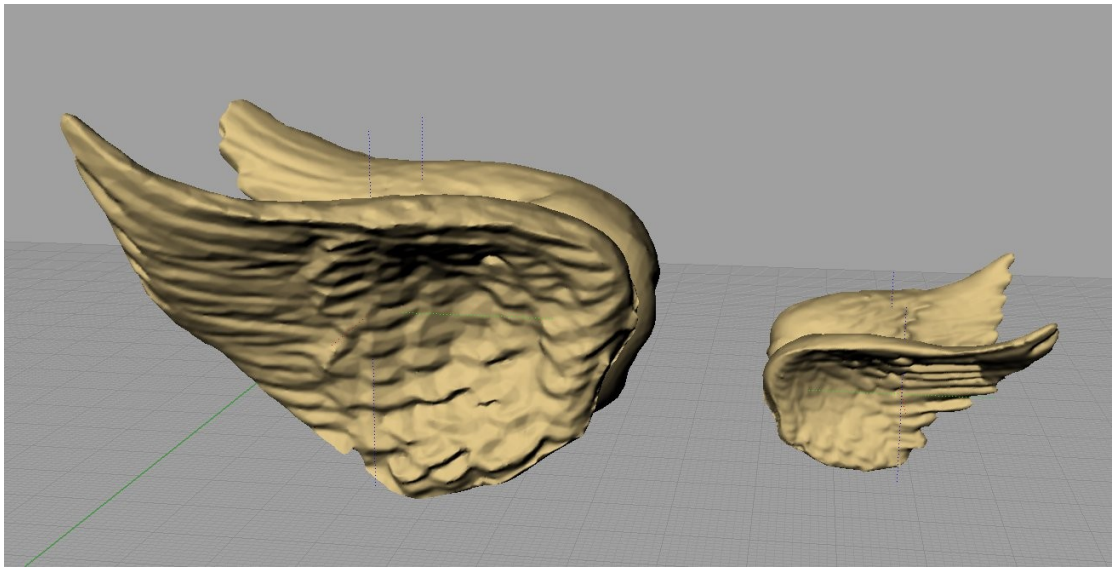
Select Volume when prompted to *Select mode*. For the prompt *Minimum edge length*, press the space bar to select the default value. For the most part this default will be in range to get proper mesh repair. If you enter a different value it is usually best to choose a smaller value than the system default.

Select Rounded at the *Hole repair method* prompt. Select True at the prompt *Accept repair*. Note that the mesh and the created volume overlap in space. Delete the mesh.

Repeat this sequence for the other wing.

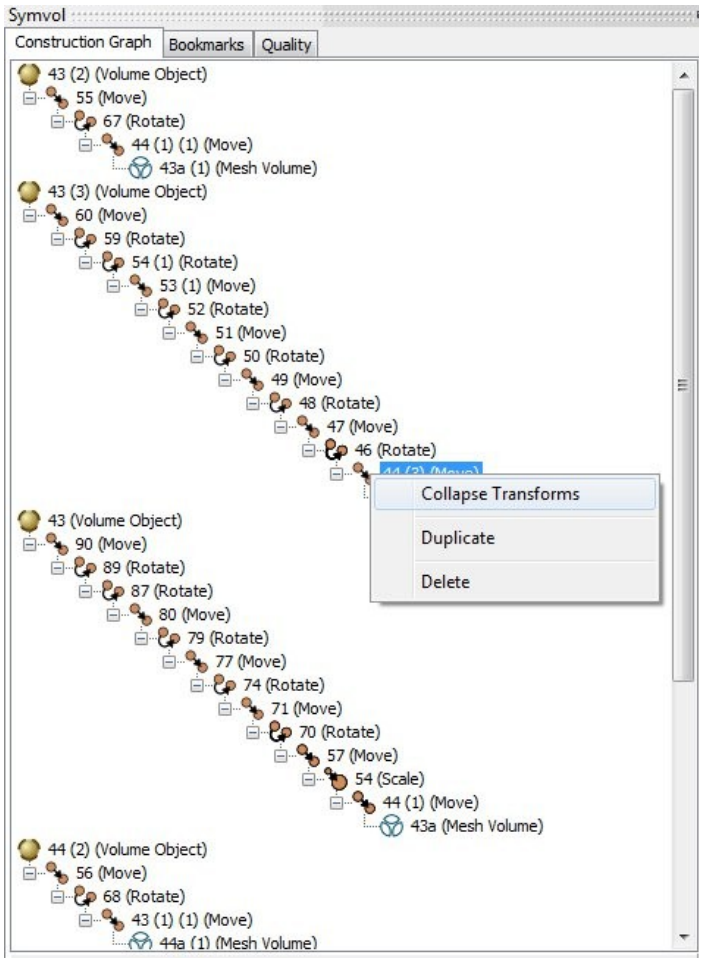
You can change how the hole is closed during repair by adding triangles to connect the blank space. This will create less of a bulge in the hole.

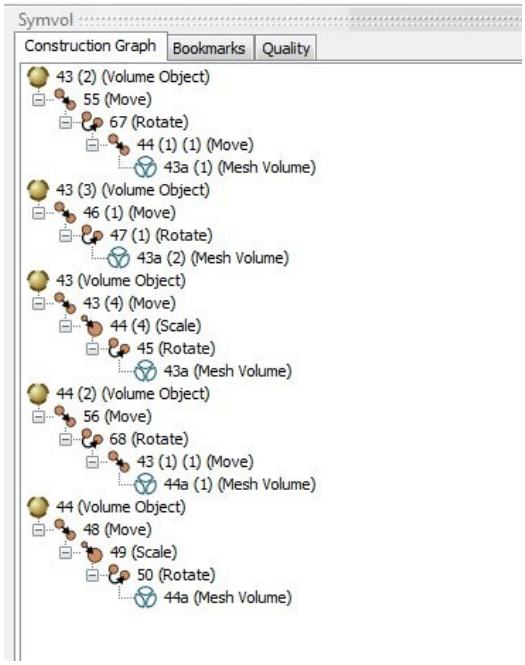
Copy, scale (50%) and rotate the wings using the standard Rhino commands to obtain a smaller set of wings.



Select one set of wings, then vUnion from the Volume menu or type the vUnion command. Repeat for the other set of wings.

TIP: If you find that your construction tree has become too wide with nested operations, take advantage of the right click, Collapse Transforms. This command will try to collapse all like-transforms into one operation.



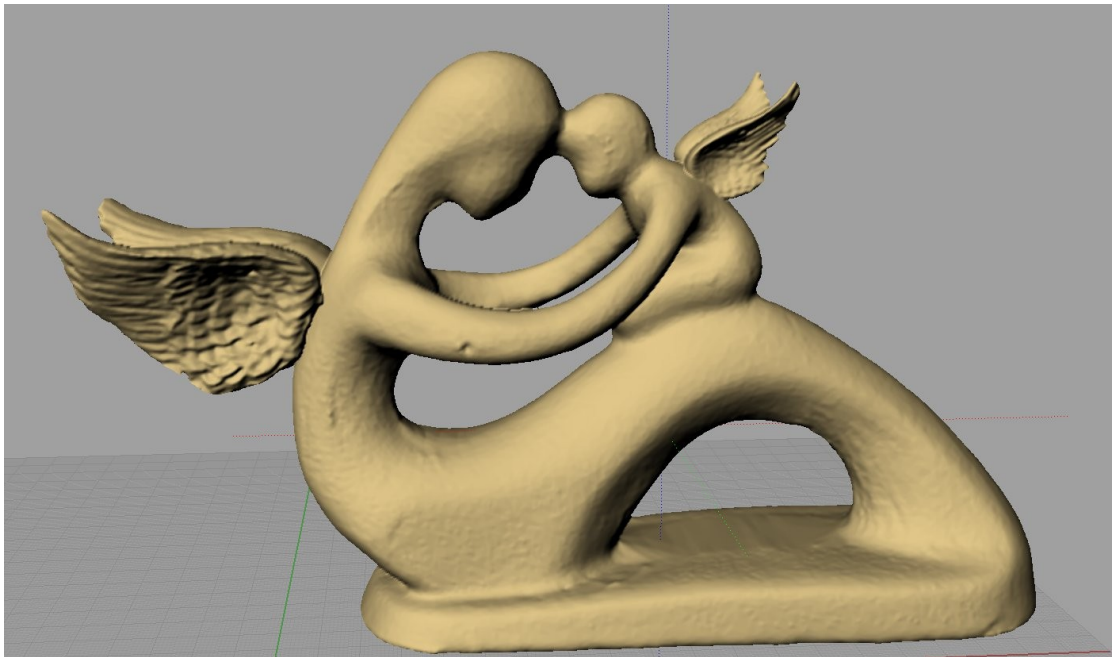


Step 3

Select vMesh from the Volume menu or type the vMesh command. Select External when prompted to *Select source objects*. Open the fertility.stl file: http:uformia.com/tutorials/fertility.stl

Select Volume when prompted to *Select mode* and hit the space bar at the prompt Minimum edge length.

Position both the larger set of wings on the woman and the smaller set of wings on the child.



In order to apply only one blend operation, we will first union the wings together as one object, before applying the union to the wings with the fertility volume.

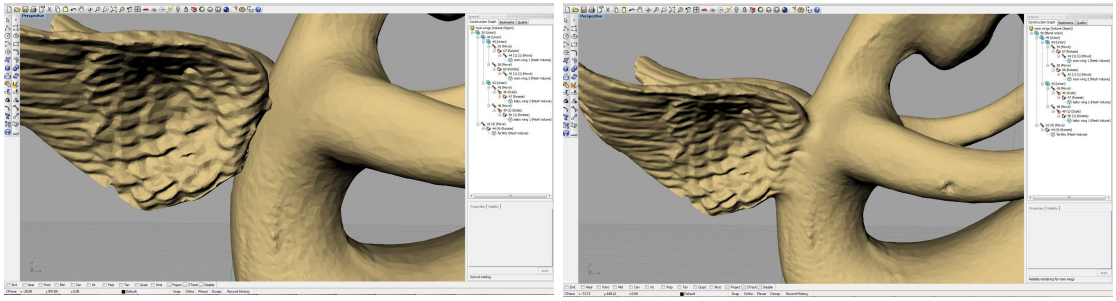
Select both sets of the wings and then select vUnion from the Volume menu or type the vUnion command. Select the wings and the fertitlity volume then select vUnion from the Volume menu or type the vUnion command.

Step 4

To smooth the interaction between the wings and the fertility model, we will replace the top level pure union with a blending union. In the construction graph, right click on the top level union and select Blend ('50 Blend Union' if you are using the provided .vol or .3dm files).

Note, the blending union is computationally more intensive than a union. Depending on your system, it could take a little longer to render.

With the same top level union (now blend union) selected, click on the properties tab and change the displacement to 10. Compare the sharp edge of the wing attachment with union and the smooth transition with the blending union.



Step 5

Generate a new mesh using vExportMesh. Before instantiating the command look closely at your model to determine what *X,Y and Z Precision* to use. In other words, what is the smallest detail you want to capture. For example using the grid which is a unit of 1, look to see the size of one of your smallest features. If it is 1/2 of the grid, then you will want to use at least .5, but to be safe probably a bit lower, like .2. Of course the smaller the X,Y,Z Precision, the larger the mesh and the larger the resulting file.

For this model, use .3 for the *X,Y,Z Precision*. Select True for *Vertex shading* and hit the space bar for *Decimation*. Select STL for *Target* and hit the space bar to select the default at the prompt for *Set slicing Bounding box*.

Note the time estimation in the command line. Depending on the X,Y,Z Precision, it will take a few minutes to execute the command.

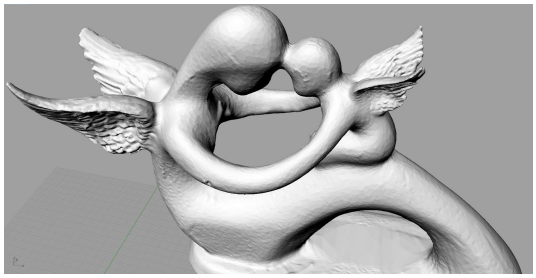
The generated mesh meshmix.stl is available here:
http://uformia.com/support/tutorials/meshmix.stl

Generating slices is also an option via the vExportSlices command. Note that in its current state, and depending on the model and your system it can take 20-30+ minutes to generate slices. We are working on speeding up this process.

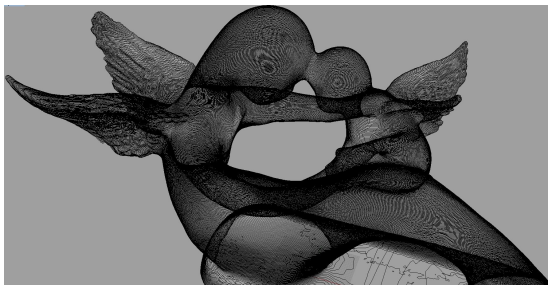
To generate slices, use the vExportSlices, and once again for this model use .3 for the *X,Y,Z Precision* and hit the space bar to select the default at the prompt for *Set slicing Bounding box* and hit the space bar at the prompt *Set slicing base.*

The generated mesh is here:
http:uformia.com/tutorials/mesh_mix/meshmix.slc

Compare the sizes of two files. On average, the STL file size is 4-5 times bigger than the SLC file size.



STL



Slices

Have fun!