Sunglasses Rhino Tutorial
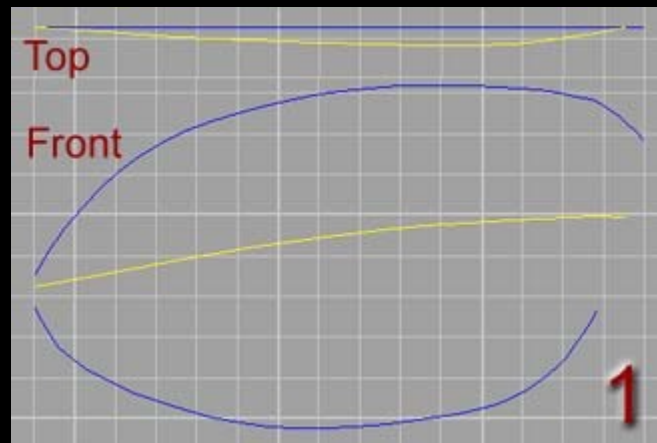
This is a tutorial on one method of modeling a pair of sunglasses with Rhino. This was a project inspired by the Renderosity Rhino3D Forum's Challenge of the Month for April 2000. This is by no means the only, or best, way of modeling this item, it is just the way I happened to go about it. This page has many graphics, so be warned, it may take a while to load.

Step 1 - Draw the profile of the lens. The centre (yellow) line describes the curvature of the lens. Loft these three curves to form a surface.
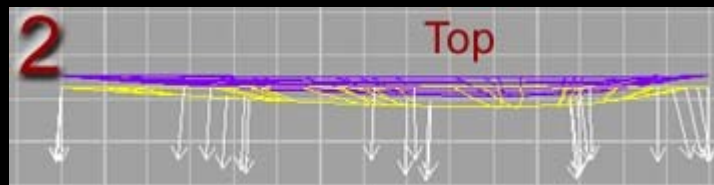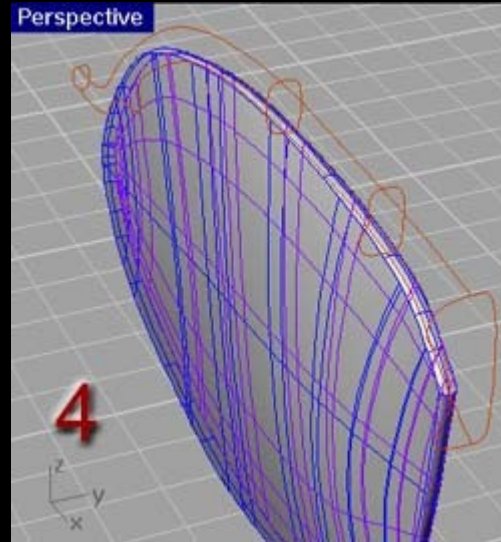

Step 2 - Use the Surface>Offset command to create a copy of the lens surface from step 1 that parallels it. Use the Analyse>Direction command to check that the normals face out on the outer lens and in on the inner lens. Reverse the directions if necessary.
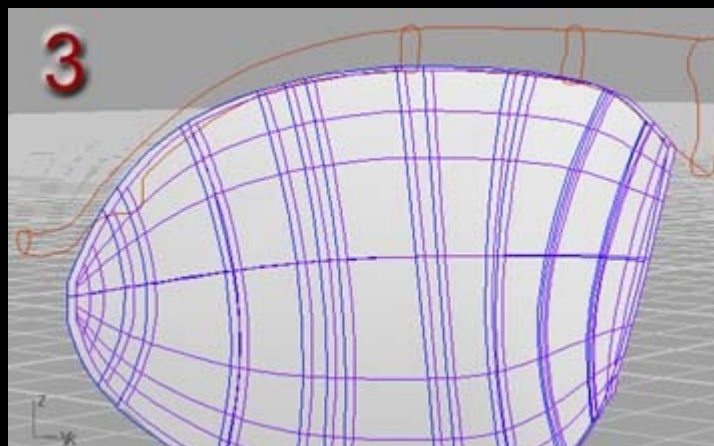

Step 3 - Draw two curves that define the top and bottom profiles of the front of the

frame. Note we are only going build one half of the glasses at a time. Since the glasses are symmetric about the centre we can easily mirror the other half when the first half is complete. Also draw cross-section curves of the front of the frame along the profile curves. Use the Surface>Sweep 2 Rails command to form the frame.

Step 4 - There needs to be a surface joining the two lens surfaces. Use the Surface>Blend command to create this. Join it to the front edge. The inner surface cannot be joined to this edge so that it can be assigned different properties in the rendering application (such as refraction).
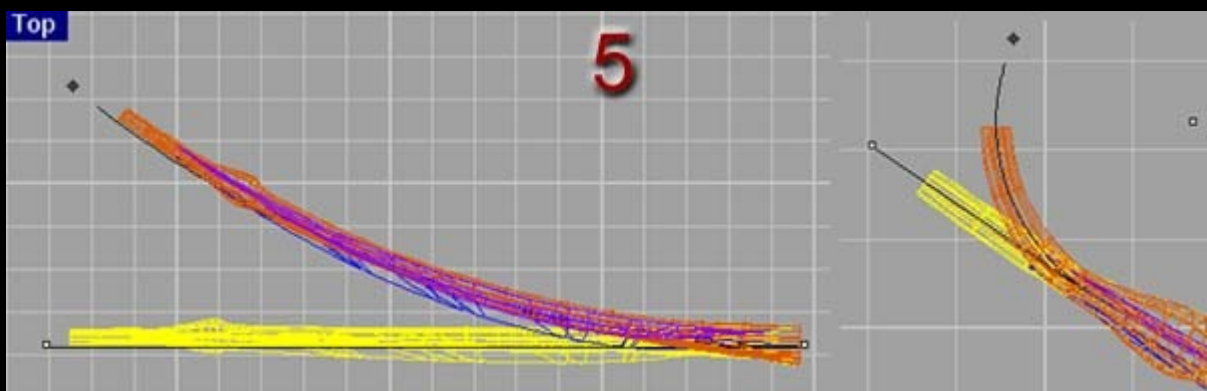

Step 5 - Select all the objects created so far and use the Transform>Bend command to create the smooth curve of the front of the sunglasses. Then use the same command, but select just the frame and set the origin point near the end of the lens, and bend the end of the frame so it is perpendicular to the Y plane.
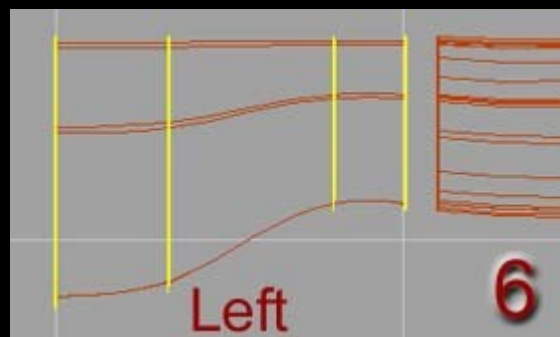

Step 6 - Duplicate the curve at the end of the frame. Move and scale this curve

along the Y plane a few times. Loft the curves then blend the two surfaces together to complete the frame up to the where the hinge will be. Make sure to join all the frame surfaces together. This step could have been combined with the previous couple of steps if I had thought ahead to have designed the area around the hinges. This part could have been part of the 2 rail sweep.
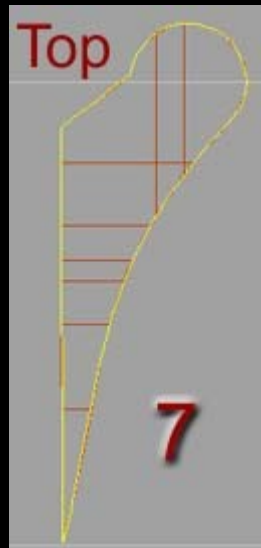

Step 7 - Draw the profile of the hinge in the top viewport. Use the Solid>Extrude Planar Curve command to create the first part of the hinge.
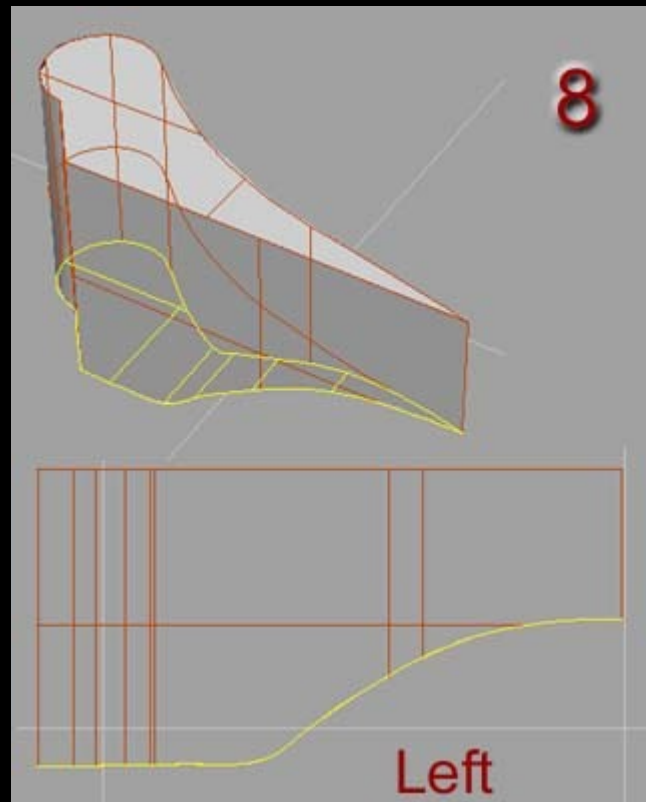

Step 8 - In order to match the lower profile of the frame, the bottom face of the hinge needs to be curved. Draw this profile from the side and use Surface>Extrude>Straight to make a surface we will use for trimming. First use this surface to trim the hinge, then use the hinge to trim the surface. Join all remaining hinge surfaces together.

Step 9 - Mirror the hinge to create a copy with the center of each hinges pivot point overlapping.
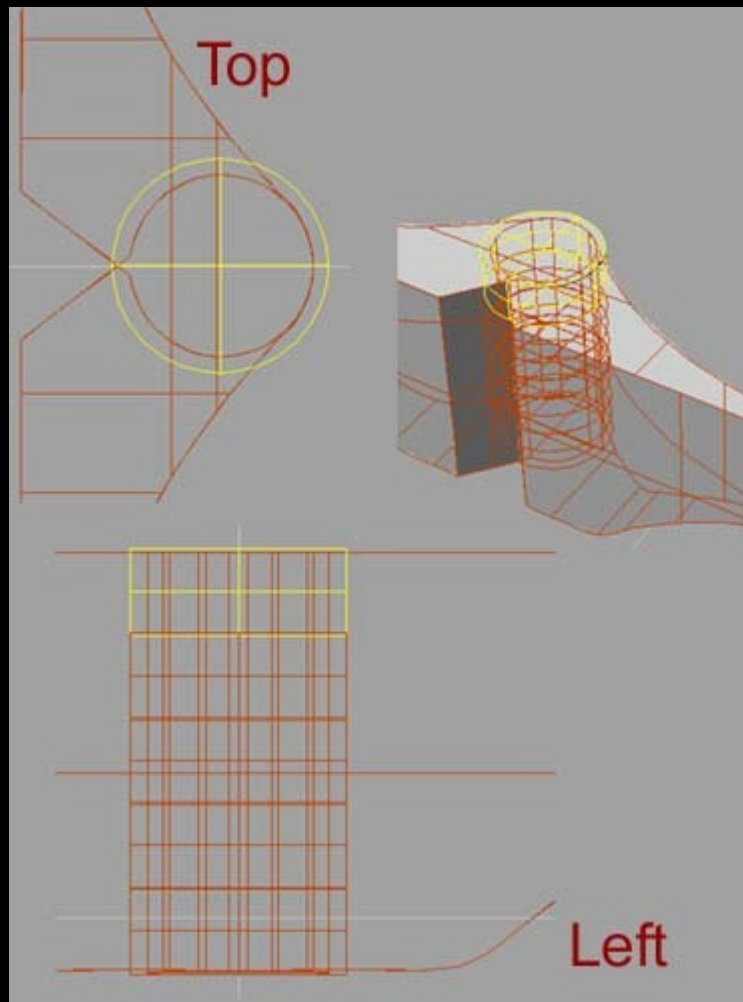
Step 10 - Create a cylinder just larger than the overlapping pivot point. Use the Transform>Copy command to create 4 additional cylinders lined up vertically and overlapping each other by a small amount.
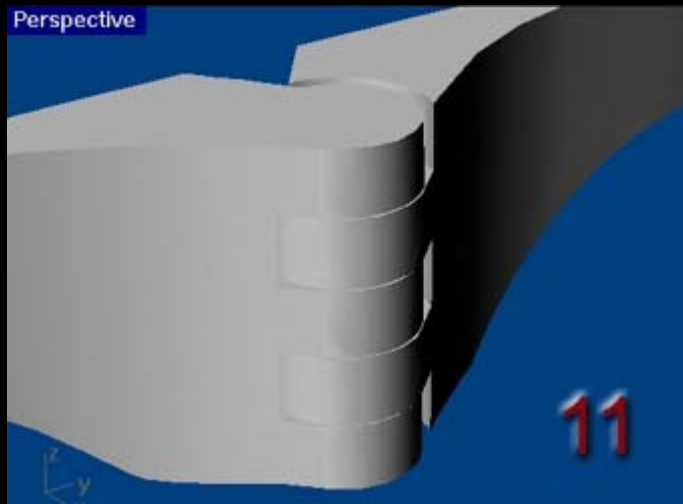
Step 11 - Use Boolean Difference to subtract every alternate cylinder from each hinge to create the fingers of the joint.

Step 12 - Create an additional cylinder slightly smaller in diameter than the cylinders you just used. Use this to boolean difference out a depression in the top pf the hinge.


Step 13 - Now here's a funky step. We're creating a cutting object to make our Phillips head screw-head. Make two rectangular cubes (solids) smaller in both height and diameter of the depression in the top of the hinge. Also create an inverted tapered cone with the top diameter slightly larger than the two rectangular solids and the lower diameter almost as small as the intersection of the two

rectangles. Now use Boolean Intersect to trim off the bottom of the two rectangular solids.


Step 14 - Create another cylinder that just fits within the depression in the top of the hinge. Position the cutting object from step 13 in it. Use Boolean Difference to subtract the cutting object from the cylinder. Use Solid>Fillet Edge to round the outside edge of the cylinder.
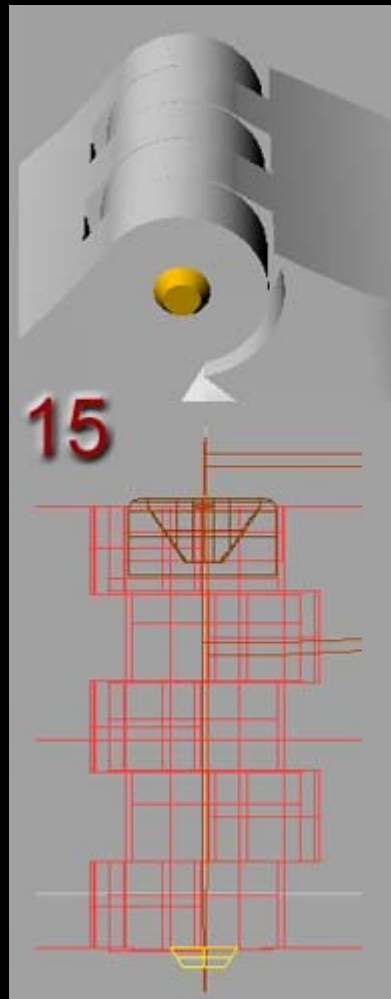

Step 15 - Finish off our ridiculously detailed screw (OK, maybe not, we could have added threads!) by positioning a small tapered cone at the bottom of the hinge to

simulate the end of the screw protuding from it's hole.
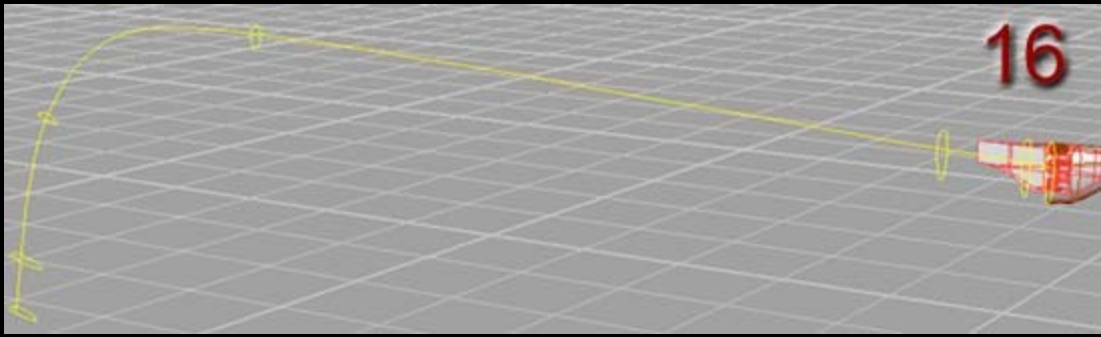
Step 16 - The next part to work on is the arm. Draw the general shape of the arm and postion cross sections along the length of it. It's easiest to copy the cross-section from the frame immediately on the opposite side of the hinge them move it and scale it along the length of the arm as appropriate. The best results are obtained when each cross-section has the same number of control points. Use Surface>Sweep 1 Rail to create the arm. I rebuilt mine with 30 control points in both the U and V directions to fix a small defect that appeared once swept (don't know why it appeared, but this made it go away).
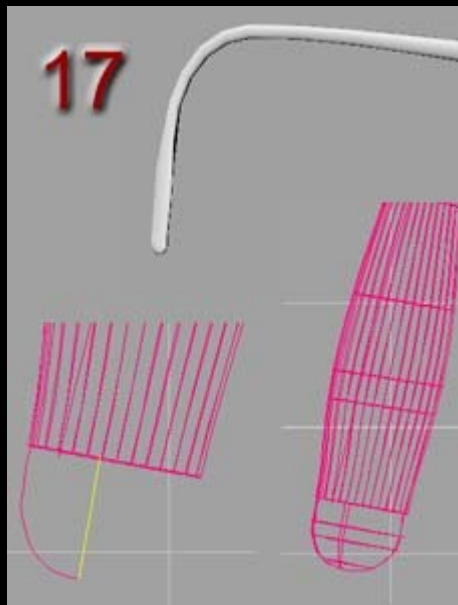
Step 17 - The end of the arm/ear-piece was created by drawing the profile of half the curve and rail revolving it using the end curve of the arm as the rail. This was then joined to the arm.
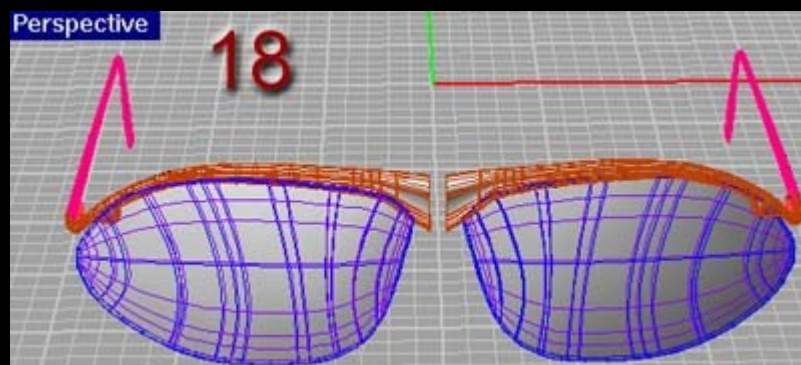
**Step 18** - Mirror all objects created so far, leaving a small gap between the halves of the glasses.
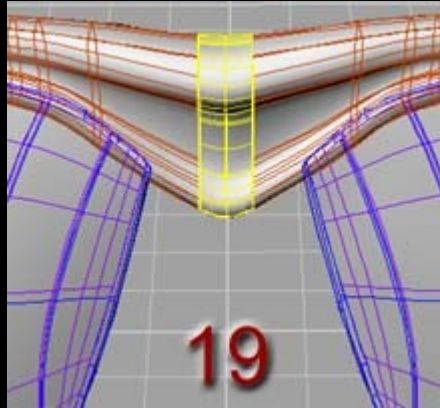
Step 19 - In order to get a smooth transition between each side, use Surface>Blend to join them (and don't forget to actually join this new surface to both halves of the frame.


Step 20 - Now to work on the detailing of the bridge and nose pads. To make the arm across the bridge draw a curve and an ellipse at 90 degrees to the curve. I find it sometimes easiest to make the curve by drawing with the polyline tool that makes sharp 90 degree corners then filleting the corner with the Curve>Fillet command. Rail revolve the curve around the ellipse.
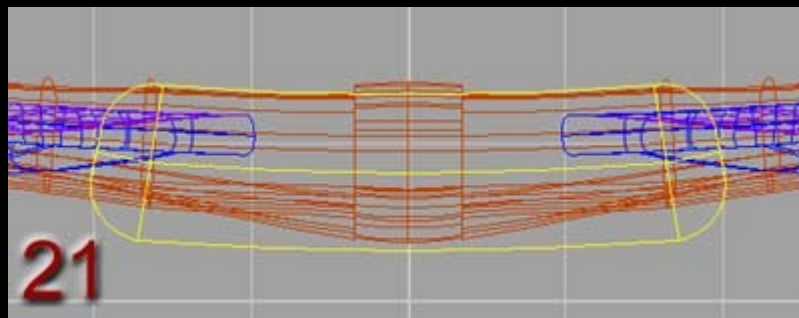
Step 21 - In the top viewport use the Transform>Bend command, setting the origin point to the centre of the revolved surface and bending each end slightly inwards.
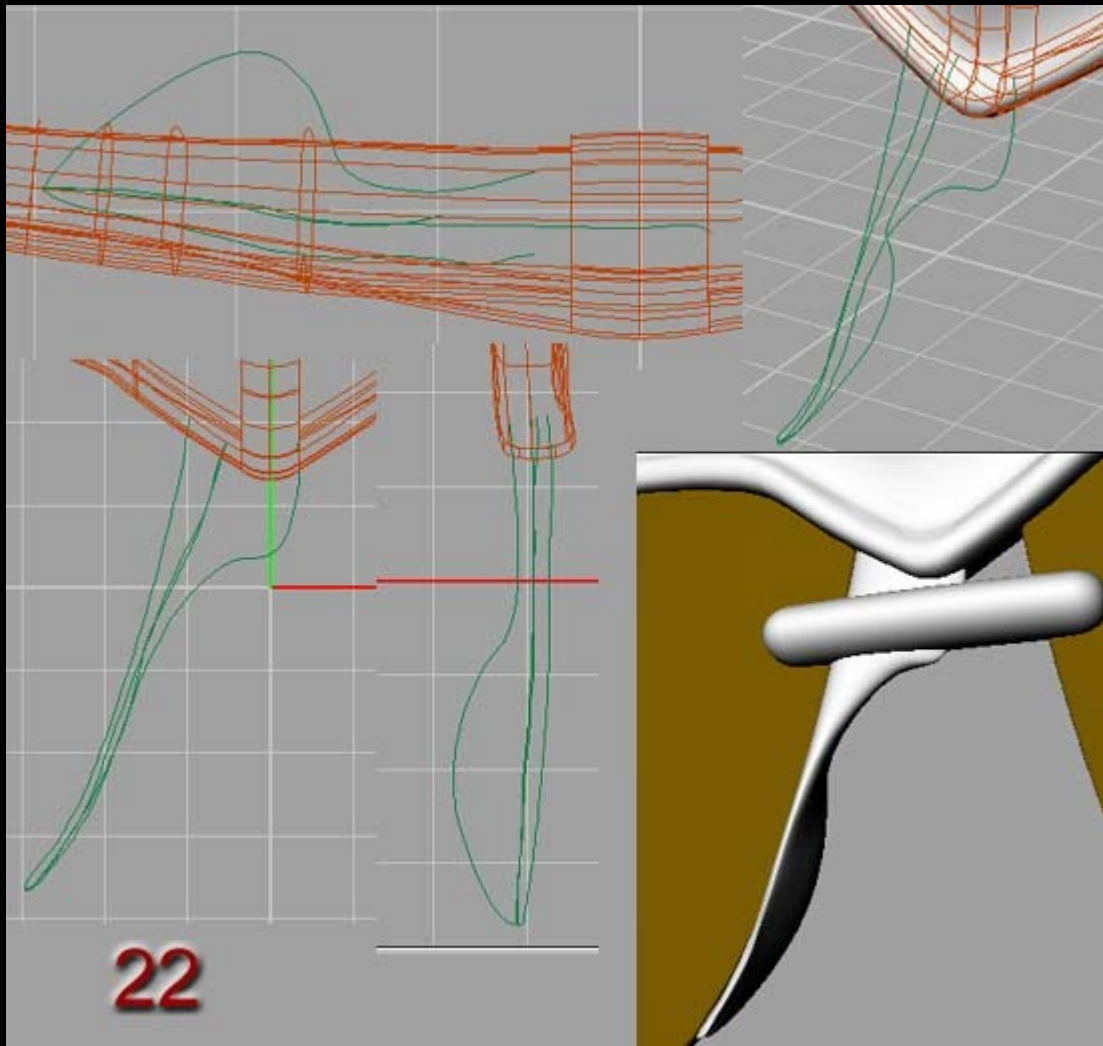
Step 22 - Create 4 curves that define the edges of the nose-pad piece. Loft these to form the nose-pad surface and piece across the bridge. If it doesn't loft properly the first time try selecting the curves in a different order.
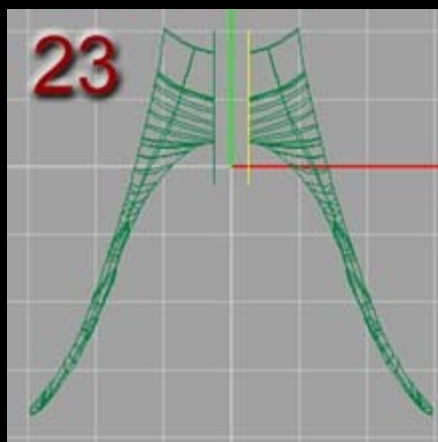
Step 23 - Trim this surface just short of the centre then mirror it to create the other nose-pad.

Step 24 - Use blend surface to make a smooth transition between the two trimmed edges of the nose-pads. If this doesn't work (I had one side reversed causing a twist in the blend) create a small straight polyline between the top edge of each trimmed edge then use Surface>Sweep 2 Rail to sweep the line along both trimmed edges to create the surface. Join the nose-pad surfaces together. Finally, be sure to put each section that will require a different texture on a separate layer to allow texturing in any other rendering application.
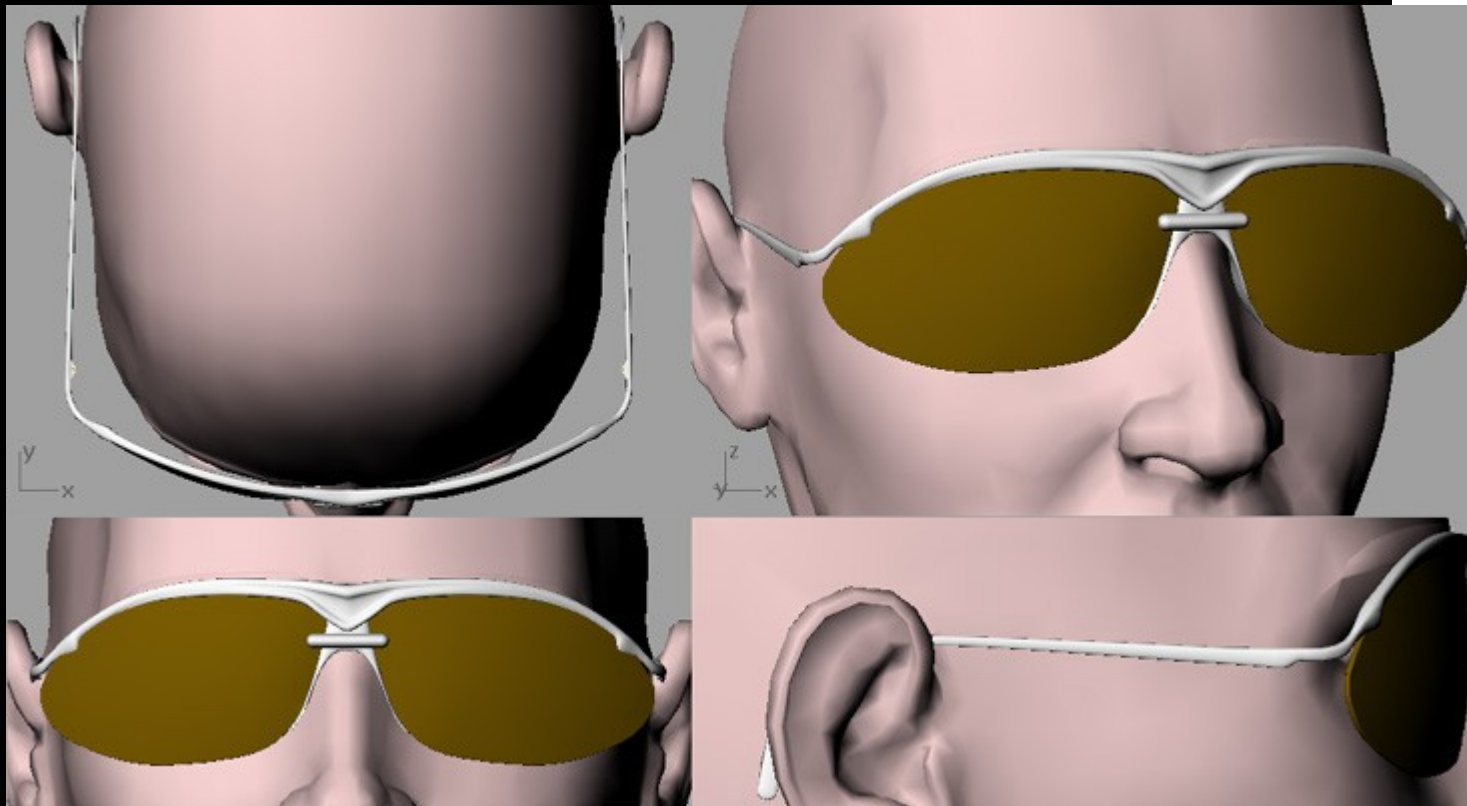
Here's the final sunglasses model OpenGL previewed on a standard Poser head.