Modeling a car with Rhinoceros 3.0
By
Romain Chevalier aka The Rain Knight
trk@melix.net

In this tutorial, you’ll first learn to use Rhinoceros. If you used to work on 3dsmax like me, you’ll be first a bit disturbed by the graphic interface, but with time, you’ll see it’s very powerful. In the end when coming back to max, you’d just want to type in your commands name…
The following is about modeling a non existing car. I worked with Audi TT blueprints and tried to think about the next generation in a way. This way, you could do your own model and it will be more fun than following a classic tutorial. There are 14 lessons in the tutorial, some are longer than others. The whole modeling is considered, even the export to 3dsmax for rendering.
If you have any question, email me

Good luck!
The Rain Knight.

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Stage 1: General configuration, setting up blueprints

Blueprints cropping (Photoshop):

First thing is to work on blueprints. If you already searched how to set your blueprints in Rhino, you found the suurland method (which is pretttty long) or the tutorials did not mention how to do this.

My method is simple: Cropping the blueprints with Photoshop, noting their size and placing them in Rhino by entering planes corners coordinates.

The harder is to crop your four views: front, back, side and top with only 3 dimensions: L x l x h.

I personally do the following:
- cut and crop the front and back view. Check that you have the same dimensions l x h. (Use "Canvas Size" if not)
- cut the side view and set the height (which is known) to h ("Canvas Size")
- center the side view to fit the canvas size.
- crop it in its length. You have know the missing dimension: L
- Now you can cut the top view, use Canvas Size with L and l, and finally center the view in the canvas.

Keep these three value somewhere in mind (L, l and h) L=652, l=298, H=219

Close Photoshop and open Rhino. If you never did...what are you doing here??!

Go to Dimensions/Properties. In the Unit Tab, use the following:

Model Units: Millimeters
Absolute tolerance: 0.01 units
Relative tolerance: 1 percent
Angle tolerance: 3

Don't change the other values.

Setting up blueprints

Left Click in the top view.
Go in View/Background Bitmap/Place. Search the directory where you placed your fours views and obviously select the top one.
Then your cursor is now a cross and in the Prompt line is asked: "First Corner? "
Enter -65.2,-29.8
Then when "Second Corner or Length?" is asked enter 130.4.
You have now your top view in correct position.
I divided the original value by 10 to fit the Rhino workspace better.

Know place the other view by yourself using the same method:
For the side view enter -65.2,0 then 130.4
For the back (right) and front (left) views enter -21.9,0 then 43.8.
Note that you obtain the view you want by right clicking the name of the view, then Select "Set View" and choose the good one.

Note that you can hide/Show the bitmaps by using View/Background Bitmap/Hide or Show command.

Rhino Tip: right clicking both validate or repeat the last command used.

Other important Tip: you can directly enter the name of the command you want to use in the prompt line. Sometimes it's faster than searching the command button. In the tutorial I'll always give you the name of the command to use, in this way: _Command. But actually you don't need to enter the "_".

Save your work by doing File/Incremental Save. This way you could get back in previous files if necessary.
So here is what you should have now in your workspace:
Stage 2: Basic Shapes

Let's go modeling now!

In the front view (double clicking on the name of the view toggle full screen), trace a curve by using the _InterpCrv function (right click on the curve button, then second icon)

In the view click three times to define the curve points. Actually a nurbs curve has more controls points than those you defined by clicking. It depends on his degree (as in mathematics :)). By default the degree is 3 (see the prompt). When tracing a curve by clicking 2 times, we obtain 3 "edges" and 4 controls points, etc...

Then we are going to move theses points to get the right shape. Click on the icon representing a curve with points on it:
You just see now points that are on the curve. When using the icon that is just right of the one used before, you obtain points that are not on the curve.

Depending on what you want to do, choose the best one.

Move the points in the side view:
Then in the top view:

You surely noticed that curves are always drawn is the world plane. Later in this tutorial we'll use the Snap to draw curves anywhere in space.

Now we build the symmetrical curve by doing the following:

Right click on the icon representing a moving square, and then click on this icon

Or just enter _Mirror

When asked for first point, enter "0" (the origin) then by pressing the Shift key,

You'll constraint your mouse moves in the perpendicular planes containing the origin point. In the top view for example, click anywhere in the plane to build the symmetrical curve.

Ok now we are going to build few others curve, using the snap option.

Click on the Osnap button in the bottom tab. A small popup (Rhino 2.0) or an additional tab (Rhino 3.0) appears.
you have quotes to define where to snap, one quote to on/off toggle the snap, and the "Project" quote to snap but trace your curve in World Planes (very useful believe me :))

Ok toggle the "End" on and trace 2 curves (_InterpCrv) joining the ends of the first build curves.

Edit the points as before but in a symmetric way. (Remember the Shift key influence)

Use a non uniform scale (_Scale1D) to move the points away from the center points.
Ok when satisfied with your curves, you can build your first surface!
Choose the _NetWorkSrf command (enter it or get the icon by right clicking then left clicking on
Select the four curves we have and validate (right click). Don't consider the pop up opening, we'll work on it later, so just validate again and there you have your first surface.

In the side view build now the body line with 4 or 5 clicks; with the Snap on you snapped the rear corner. Move the points in the top view.
Do the same thing with the bottom line.

We build now a 2rails surface (_Sweep2). Or here
The prompt line asks for "rails". Click the two new curves. It asks then for "crosssections". Click the first curve you built. A pop up ask you to choose between the curve and the Edge. Choose Edge. Remember that a surface edge can always be used a construction curve, and is sometimes different from the curve that has been used to build it.

You have your 2rails surface now

In the side view, trace a curve (don't snap) that will limit the side surface:

We are going to split the surface with the curve. You noticed that the curve is in the world plane. By working in this view the surface will be split anyway.

Select the surface, and enter _Split or click on this
In the prompt you are asked to select the cutting objects. Click on the curve you built before and validate. The surface is now split into 2 parts. Don't delete the small part in front. Hide it (click on the wheel button of your mouse, then left click on Hide). Do the symmetrical surface.

Now we build the front bumper base surface by doing the same thing we did for the rear. Trace 2 _InterpCrv with the Snap on (End) and then move their control points.
A new _NetworkSrf. This time be sure (when pop up asks) that all the edges are on "Position"

Ok then trace a new _InterCrv in the top view, with the Snap (Near) on.

This curve has not enough points to be well defined.
So enter _Rebuild and then select the curve. When pop up comes out, you have many parameters you can change. Let the degree on 3, but raise the control points number to 6 for example. Validate.

You see now that your curve has more points. Move them to have a nice shape.

Advice: when moving controls points turn the Snap off...

Build a _NetworkSrf by using the four limiting curves/edges. Even if the front and rear curves are longer, the Network Constructor will limit them by himself. Very practical! Again let all the edge on "Position" in the pop up.

This last pick was done in the "OpenG1 mode", which is obtained by clicking the Opengl icon in the "mouse wheel button pop up".

Build the symmetrical surface.
Stage 3: Basic Shapes (part 2)

Now let’s move on the roof shapes.
In the top view, trace the curve that fit the bottom of the windows

Trim the last built surface with this curve (_Split) and then Hide.

Then in the left or right view, draw the curve that fit the curvature of the windows

To help you, you can use the snap in Project mode.
Build a 1Rail surface (\texttt{Sweep1}). As a rail use the last curve you build and as unique crosssection the edge of the last surface you split.

You got now something ugly like this:
Don't worry. We are up to split it. In the side view, trace a curve that fit the upper limit of the TT windows.
Then split:

Do the symmetric and then trace the curve limiting the roof:

Let's build now the top of the rear: this part will be useful to build the roof afterwards. Do the following to get perfect shapes.

Draw a curve like this (use the snap):
By editing points, try to fit the curvature of the top edge of the rear surface (the first surface we build).

Create then a _NetworkSrf using the 3 edges (be sure to choose edges) and the curve. Use the "Tangency" on the two edges that are on the side, and "Position" on the curve and on the rear edge.

Now in the top view trace the curve that you want to use to limit the roof.
Then split the surface with this curve

I hope you understood that this method is really good. You could have built the curve, then build a Network.

But it would not have been obvious you have tangency on the sides.

Remember that building surface and trim then is always better than trying to build final surfaces directly...

Then to make the roof, draw additional curves like these ones:
Then use a _NetworkSrf with all the limiting curves and with the one in the middle too.

We need two surfaces to build the roof arches.

If you try to build a NetworkSrf directly you'll understand that the bottom curve is not usable this way.

Use the _DupEdge command
When asked, click on the upper edge of the window surface.

Then on the bottom one:

You now have 2 curves where you had just edges.
Split the bottom curve with the upper one (do the same procedure than when splitting surfaces)

Now hide/show surfaces like in the picture below

You can now build the networksrf. Specify tangency wherever it's possible.

Depending on the quality of your curves, you'll get nice surfaces or not!
Secret of nurbs is here...
When you understand the method, you can do it again and again, till you get the perfect shape!
Stage 4: Tweaking the front part

Let's build the hood now. We'll use a _NetworkSrf.

We first draw a Blend Crv (_Blend) between the edge right on the bottom of what's going to be a windshield. When asked click on the end of the first edge, then on the end of the opposite edge.

You Get that:

The _NetworkSrf is now just piece of cake!
Choose tangency on the side but only position on the edge of the bumper.
Ok now you think the angle between hood and bumper is not nice? Me too!
So let's smooth it.
Use the "Split at Isocurve" command (right click on split icon or Surface/Surface Edit Tools/Split at Isocurve)
Then click on the surface like below

A line appears attached to your mouse and follow the surface. When we click, we just split the surface into to parts.
So if your line is in the same direction as mine, it's ok. If not, enter "V" or "U" to toggle the direction. U and V are the coordinates of a point on a surface, whereas X, Y and Z are the coordinates in world space. So a "U isoline" is a surface line where the coordinate u is constant.

Ok when the isoline is on the right place to begin the future "fillet" click.
Hide the little surface you create by splitting.

Do the same on the "roof" part, with the Snap On (End) to split exactly in the continuity of the near surface. Endly do the same on the opposite surface, with the snap on once again. Then Split the bumper surface the same way, and with nearly the same amount.

Ok now we can't use a Blend surface yet. Use the _Blend command to create the curve like below

Then, in the side view, split the side surface of the car with this curve.
Repeat the same operation on the other side.
Now we could almost build the surface but you'll notice that we
Have three edges facing only one. So that a little problem.
Because we absolutely want to have the tangency with the upper edges, we are going to build three
Network surfaces.

So use the _Extend command. At first prompt, just validate.
Now click on the edge between hood and near side surfaces and then move your mouse to the front. The
curve extends itself to front. Click again to end the line. The length has no importance since we are going
to use them to cut the bumper surface.

Do the same on other side. And then cut the bumper with this line (in the top view)
Now just build two blend curves (_Blend) like below:

Ok now everything ready for building NetworkSrfs...

Do the two exterior ones. Specify tangency on the two long edges

Do the last one in the center and specify tangency on all edges!

There might be some faster method, like doing everything as soon as side surface is built. In all cases everything’s clean now.
Now it's time to do some cutting to see where we're going.

Cut wheel arches:

And why not lights, grill etc...
Stage 5: Volumes Sculpture

In this part we are going to change the hood shape, and deform the side surface to have nice lights plays on it. This second operation is not so far as editing a mesh with 3dsmax for example, but with additional rules.

The Hood:

On the top view, draw two curve (with Snap Project on) by snapping on the reds points (next figure). Side curve is corresponding to the hood limit, whereas the inner line will be used to split the actual surface a rebuild a new one.

Be sure you snapped to the marked points because if you don't you'll have many little surfaces that are not good to work with afterwards.

Split the big surface (the one that runs from front to rear) with the inner curve. Build its symmetric. (We don't use the other curve yet, but it is useful helping to build the first one)

Draw a line (with Snap on NO projection) joining the two news corners you formed
Rebuild it with more Control points as below
In the left view move the side points to give an incurved shape there:

You can now rebuild a New Network surface with the 3 edges and this curve. Ask for continuity only on the front edge.
Sides Sculpture

I'd like to give to the side some shape they don't have. They are really flat. I could have done it with an additional cross section when building the 2 rails surface, but I want to show another method, which could be named "sculpting".

Here I will deform the surface only between the wheel arches. Since its nurbs, the whole surface will be deformed a bit but I don't care.

The Principe is that every isoline of the surface (we talk about that is lesson 4) is editable as a curve is, moving its control points.

Select the side surface, click on the mouse wheel, and then left click on the upper right icon. Controls points appear! Note that for surface there is only one type of points (those are not on the surface).

Ok you can move those points in every direction, deforming the surface.

BUT, THERE ARE FEW RULES TO RESPECT:

- Don't move the points that are on the border. If not you'll loose the continuity with surrounding surfaces.

- The Split Edges are not borders! If you move points, the surface change and the splitting curve re-project on the new surface. Just try and you'll see.

Here we are just "snapping" the five points that are on the second row (from the bottom) and between the wheel arch, and moving them a bit to the inner side of the car. The wheel arch edges are moving a bit, but that's not a problem, since we got no surface here for now. (Construction ordering is very important so)
Adding the side skirts

This can be easily done by adding small curve, editing them like the red marks on the next figure, and building _Sweep2 using these curves as rail and body edges as crosssection.
Stage 6: Wheel arches

Use the _Offset tool (right click on the "filet curve" tool, then offset)

In the side view, pick the edge of the actual wheel arch

Actually it's important to choose the view you're working in when doing such operations. Here in the side we're doing an offset in "the view plane"
Enter 3 as the offset value and move the mouse inside the arch. You'll see a white arrow which length correspond to the value you entered. Left click to build the curve.
You get something like this:
I toggled the control points on. You note there are as many points on the native curve as on the offset curve. If there are too many points according to you to edit it properly, rebuild it with fewer points. You can then move these points to adjust the curve.

We must have something nice and regular; with a shorter distance (between curves) on the side than on the top (this is true on all views). Check that you have something nice on all views:

Build the two curves that close the arch using the snap (end) option.
I choose not to edit them. If the result is not convenient to you, you can edit them.

We are now going to build the surface. Use a NetworkSrf and choose tangency as below (btw that the only edge on which you can obviously have it!)

You can see that A and C (on my fig) curves are not tangents to the side surface. However, I’m asking for tangency on D. That’s normal. The tangency is a harder constraint that position, and if you look nearer after you built the surface, you'll see that the surface don't follow short edges near the side surface in order to be tangent it.

Now we can simply add the too little surface by doing 2 Sweep1, using the blue edges as rails (be careful to pick edge and not curves, since as said before, they are not the same...) and red edges as cross sections. (Previous fig)
You we are now building the last short surface that finish the wheel arch, as on the actual TT. Do a _Offset again, with a 0.8 value. Have in mind to work in the side view, as previously mentioned.

Then (without editing it) build a _Loft Surface between the native edge and this offset curve. For the two little surfaces, do 2 _Sweep1 again.

Do the same thing on the rear wheel arch. We have now a nice body volume. Harder is done now.
**Tip:** When you build 1rail surfaces (Sweep1) in this lesson, sometime the final shape is not nice. Duplicate interior edge of your surface, delete surface, and then edit the curve (you'll maybe have to rebuild it) Finally rebuild a 2 rails (Sweep2) with this new curve as second rail.
Stage 7: Choosing final design and splitting

This part is important but not hard. It's the most pleasant thing to do when modeling a car in Rhino. You worked hard to build nice volumes and surfaces. You are now going to split them to create different body pieces of the car. This phase is often long, but I'll be short. It's up to you to choose your design. I posted views of the design I choose. You can do the same or not...

Practically speaking, work in projection, i.e. draw curves in the view planes then project them on surface (don't split) just to appreciate how it is going. The projection tool is the following:

Be careful, as for the offset tool, to be in the right view before projecting. The projection occurs perpendicularly to the view, as for the splitting process.

Once some curves projected, to draw new curve that use the previous ones, use the Snap Option in Projection (Project toggled on). Then use the projection tool again...Delete and draw again till you're satisfied. I'm using often the _Extend tool (you already used it):

It is very practical and has many option, for example you can extend a curve while keeping the curvature at his end.
Read the prompt line, choose and try. Remember again the quality of the curves is actually doing the quality of the final model.

So take your time and get inspired:
Once satisfied, split the surfaces. If you used the projection tool properly, you can split directly in the Perspective view, since curves are already on surface. You can do it in the other views too. Note that, when splitting a surface, splitting objects do not need to be unique: there's no need to join curves before splitting.

Once you split all the surfaces, move each set of piece in his own layer (in the Object Properties) You are now ready to work on each piece separately, and without thinking of the surrounding ones.
I personally changed the headlights again. They were not high enough I think.
These new ones are almost the same as on the actual TT but they fit the body line better.
Stage 8: Fillets everywhere!

We are going to do hardest part maybe: filleting edges. For example the "body line" must not be so "edgy". We have to create a small round surface over the edge. Mathematically the software use the value entered on keyboard to calculate a pipe (which radius is the value you entered) tangent to both surfaces in the concerned edge region. It splits then the 3 surfaces with each other. Then it deletes useless parts.

However, this operation may fail. We are going to see how to get a fillet manually, that is as good as an automatic one.

Work now of the front wings layer. Hide one of the two. We are going to work on the other. Select all surfaces:

And join them (_Join)

After some time, the prompt line indicates that 7 surfaces have been joined properly. Good for us.

Use now the _FilletEdge command. This tool is only usable on polysurfaces edges, remember that.
Enter 0.5 as value for it, and then select the edge you want to smooth:

You can already realize that the edge may not be in one single part. Click then all the edges in order to have the full length you want to smooth. Sometimes, Rhino ask you to pick between 2 edges whereas there's only one. In this case the joining failed and so no fillet will be built.

OK so after picking all edges, right click and wait.
If everything works, well for you!
If not that was my case, because of "dirty splits" I think), read the following.

Unjoin the polysurface with the _Explode tool:

We are going to use a pipe to split surface, then create the blend surfaces between the two edges.

Ok let's create first the line that will support the pipe.
With the _DupEdge select the border edges of the bottom surface (you can hide other one to make it easier)
Validate, and then Join these curves together (_Join). We got the support line for the pipe. The question is: What radius do we use for it? 0.5? Certainly not! We need to have a reference.

Toggle doors layer on. Join the 2 surfaces

Then do the fillet between them (it's working I hope. for me it was, since these two surfaces are simple)
Ok back to the wing. Use the _Pipe command:

Then select the curve we build before near to the door polysurface. You now have a circle that follow your mouse pointer and that will define the pipe section.
Get close as possible of the door polysurface and when intersection between your circle and wing surface fit the fillet edges of the door polysurface, click. Validate for second section radius. The chosen value might be around 0.17 (see the prompt).

You got a pipe now:

We can't use it now for splitting because at his ends it is not long enough. We are going to make it longer. Select the pipe and explode it (_Explode).

You got now 3 surfaces instead of a polysurface.
Zoom to the ends of the pipe and delete the two cap surfaces.

Then use the _ExtendSrf command:

Enter a value for the extend coefficient (2 for example), then click on the extreme edge of the pipe. Do the same operation on the other side.
You got now a tube surface that goes far beyond the border of surrounding surfaces:

Now split the concerned surfaces with pipe surfaces. Delete the tube and the small split surfaces.

Ok let's fill the hole now.

Zoom to the ends of the hole and use the _Blend command
Pick then the two edges on the near surfaces. A blend curve is created.

Do the same on other end.

We got a 4 sided region but we can't build the Network since the upper edge is not unique. As before use the _DupEdge command to build a curve that will be our fourth edge. You can now use a network surface using the bottom edges and the 3 curves. Join all the surfaces together. You got a nice smooth edge now.
You’d had surely noticed that, since we can use tangency on upper curve, the blend surface is not tangent to the upper surfaces. But it's not so far. (You can try to build additional crosssections or use the BlendSrf command, but it's complicated to; try it.)

This method is always working when a fillet operation fails. It's pretty long and give rather good results (don't forget we are doing art here, not industrial model!)

Ok now filet the edges on rear surfaces (for me it works).
You just have to select all edges at the same time before you validate.
If everything works, Rhino create a small triangle surface at the junction between the 3 fillets. Perfect!
Stage 9: Front bumper details

Toggle the front bumper layer on and toggle the others off.

We are going to build the lateral airintake that contains fog light too.

First split the concerned surface with the curve you project on it.

Keep the splitted part and hide it.
In the left or right view, Extrude the border you just created (choose edge rather than curve)

Join the first surface with the new ones. The fillet the yellow edges (see next picture)

I choose 0.5 for fillet radius. Try, cancel, and build again to find a good value (depending on your shape, and on what you want to do)
Show the surface you hide at first step and hide what you just build.

In left or right view, draw box curves for the grill, and a circle for the fog light (if it has not been done yet)
Make a copy of the surface you have (one will be split to do the grill, and the other will be the black surface that cap the hole) and hide it.

Split the surface with the circle, then split the "non circle surface" with the boxes. Delete the surfaces that will be holes, and move the grill surface a bit backwards.

Extrude the upper and bottom edges of the grill surface, and the edge of the fog light surface.
Join the surfaces (in the end you have 7 polysurfaces = 6 grills and one fog light)

Then do the fillets on edges you want.

You can do exactly the same operation with the central air intake. I use the following curves:
And that's what I have with some work:

The method is simple. Don't forget to keep always a surface (make a copy Crtl+C Crtl+V) before splitting it. It's always useful since for example on the central air intake, the surface is used 3 times (once for vertical grills, once for horizontal, and once for plate support surface)

Ok now let's do the little air intake I designed just under the central air intake.

Hide everything that's useless. Keep the surfaces and the curve:
Split the two surfaces with the curve and hide the upper split part.

Use the _OffsetSrf command on the bottom split part. White arrows appear on the surface, showing you the value (the length of arrows) and the directions of offset operation. Enter a value (0.5 for example) and click on FlipAll in the prompt to flip arrow direction if they are not pointing backwards.

Validate to build the surface.
Hide the parent surface.

Build a _Loft surface using the yellow curves (next picture). You get the surface that is on next figure too.

Do the same operation on the other side.

Then build a _Sweep1 surface using the big edge as rail and the edges of the loft surfaces as cross sections
And finally extrude the curve backwards:

Join everything and you're done!
Stage 10: Adjusting body parts

Ok if you're at this stage, you know Rhino a lot better no :D.
Concerning body parts, we have done the funny work!
Here is coming a pretty long stage, not very hard, but repetitive and boring, since Rhino don't have some useful command that other professional software have (Auto Studio for example).

First we have to do small spaces that exist between moving parts on a car: doors, hood, fuel tank opening, etc...

I show you the technique on the door:

Isolate the door layer. Use the _Pipe command and build a pipe on one edge of the door. I use 0.12 for the Radius but you can choose want you want. You are going to trim the door only, so the size of the space will depend on the pipe radius.

As we done in stage 8, extend the pipe on both of these ends after you explode it and delete cap surfaces. Extend factor of 2 is enough.
Use the _Intersect command to create the intersection curve between door surface and pipe surface.

Don't split now. This way you can see if the radius is good or not, and in some case, this curve will not be the splitting curve.

Use the _Blend command to create this little junction curve. Click the two other intersection curves when asked.
Next images show were I’ve repeated the operation, which I familiarly call "TubeOffsetCrvOnSrf" :D. I hope this will be integrated in Rhino 4.0. Just having to enter the radius...

The door:

The hood:
If you’re satisfied, you can split surfaces with these curves and delete small surfaces. (Since you can only split surface with curve and not polysurface, explode them before splitting and Join them afterwards).

We are now going to give some thickness to the body parts. I show you the technique on a small and simple part, but his operation is often long and hard to realize.

Isolate the front bumper part and split the small surface that cap the light washing system:
Use this tool now (don't remember the name, but you'll find it in the line pop up))

Pick the small surface, be sure the Osnap is on, and click on the corner:

You have now a white line that follows your mouse, perpendicularly to the surface. Enter the value 0.1 on keyboard and validate. Finally click on the inner side of the car.
Repeat the operation on all edge end of the part (Here we have only two).

Build a _Sweep1 surface using the part edge as rail and the two short lines we just built as crosssections.

Do the same operation on the other edge:

Join the 3 surfaces and _FilletEdge of 0.05 on all edges (here we have 4).
And here is a nice surface with round borders!
Once again there is no build in command to build a surface perpendicularly to another (there is one in the bonus tool set, but its crap! believe me).
Ok now you have to do this on all pieces!
This is fairly long and sometimes fails or let small holes in some places.
We'll see in next lessons how to get rid of them.
Stage 11 : Finishing fillets

Here are some tips to fill holes that are often created when you fillet your polysurfaces.

First of all are 2 examples you will see very often.

The first one is very easy to fill. It appends when a corner is much larger than 90º:

Here you'll just need a Network. Don't forget to join the two surfaces afterwards.

Second case, occurs sometimes. The method presented here is general.

Remember in stage 9 we built the small air intake in the bottom of the front bumper.
You may have not built it, but look at the method.

After you joined everything, you want to fillet the edges that are yellow on the next pic (and the symmetrical ones of course):

But when looking nearer in the corner, you'll see the following:

That's normal...Rhino don't want to be mean with you :)
Solution:

1- Create a Blend curve from edge 1 to edge 2

2- Explode the polysurface to split the concerned surface with this blendcurve
3 - Build a Network with the 4edges we have now. Specify tangency everywhere.

4- Join everything and you are done:

Here is a new problem to fix. I realize it only now. However it's not a small fillet but a big one.

You certainly succeeded in building the fillet between backdoor and roof:
The fillet that continues till the lateral window is a bit tricky. If you try to do it automatically, you'll get something bad in the corner near the window. I personally don't want such a surface.

So here is a method to build a nice fillet.

With the _Pipe command, build a pipe on the concerned edge. Give a radius value on the left that match the existing fillet radius (0.16 for me) and 0 on the other side.

Explode the pipe, delete the cap surface and _ExtendSrf it (2.0 as coefficient is enough)
With the _Intersect command, create the intersection curves between the pipe and the two other surfaces.

Split the bottom one with the curve you just build (you could have split it with the pipe actually).

For the other one, split it by isoline (right click on the split button), choose the correct direction (u or v) and pick the left end of the intersection curve we just built.
After deleting the trims:

On the left build now a _Blend curve.
On the right side toggle the window surface visible:

We are now using a diagnostic tool that is very useful: _ShowEdges

Pick the bottom surface and choose nakededges

You are now seeing the boundaries edges of the selected surface, and the points that limit the edges. When zooming near to the window, you realize that the trimming we’ve generated 2 edges.
This problem will not be good for us to build a nice network surface with the tangency on this edge. Ok let's try to fix that. Use the _MergeEdge command and pick the small edge we don't want to have:

A pop up appears asking you to choose which edges you want to merge. Choose the left side (it's highlighted)

And that's fixed!

Check with the _ShowEdges command that everything's fine:
Alright! Keep in mind that you can perform the opposite operation with the _SplitEdge command. Concerning the MergeEdge command, it's not always working as you can imagine. The edges have to be "continuous a second order" somehow to be merged, that's to say: edges that form corners can't be merged...

Finally build the NetworkSrf:

Specify tangency on the concerned edges as below: I use "Loose" on edge C to have a nice tangency on edge B.
Note that near the window, the surface is not following the window edge since we used Loose and not Position for the network.

This way the tangency is nicer! Compare the 2 cases and you'll see.
Stage 12: Rear view mirror and door handles

The rear view mirror will be the TT one. But you can use the method for other shapes.

The difficulty here is the fact that the surface can't be built with a Network or a Revolve command.

We are going to use a sphere primitive and deform it;

Split it on 2 of its isolines

Show the control points (wheel mouse click, and then left click in the upper right corner icon)
In my case I rebuild the surface to increase the number of control points (increased the V value):

Use the blueprints in all views and take a look in the following pics to help you.
When done fill the hole with a patch surface:

With an offset curve (Offset) with 0.2 values in my case:
Split the patch surface

Keep the inner part to do the glass.

Finish the volume by creating fillets. I extruded the inner boundary a bit to make 2 fillets. The "foot" is simply done using a cone primitive. I trimmed it with the sphere and with the door. Everything is joined and fillets are working automatically.

For the glass, use an Offset curve again to shrink it a bit. Some extrusion on the boundary, join the 2 surfaces, do a fillet a you are done.
Move it finally a bit inside the sphere.

The door handle:

To make the hole in the door, I just used a sphere. Split the door with it and split the sphere with the door. You can use Boolean but, since we have opened polysurfaces, result is not predictable.

Small filets are working fine.

For the handle itself, I used a box I split many times. The fillets are simple here, too.
Finally a cylinder with a squary hole:
Stage 13: Lights

Front lights:

I began with doing the black joint that you can see on my renders. I simply used _Pipe on border edges, built the intersection curves between pipes and the light surface with _Intersect, trimmed these curves with each other to have nice corners; finally split the surface with these curves joined together. I also "thickened" the surface, as you've done on body parts.

First give some thickness on the inside of the joint. Use the same technique as for body parts (Normal line to the surface, then Sweep1 surfaces)

Now _Extrude the new border from the front view:
In this front view draw the line that will limit the turning indicator, and 2 circles for the spotlights.

Once you've made a copy of your light glass surface, trim it with the previous line, the _Extrude and _Fillet:
Now we simply cap with a line (snap on surfaces end on the right and somewhere in the middle on the edge on the left):

And with 2 _NetworkSrf. Ask for tangency for the second surface on the line we build before.
Trim these 2 surface with the circles you drawn before, _Extrude and _Fillet:

Spotlights are simple geometry:

One filleted tube, torus, one trimmed sphere (with a circle) for the lens, and filleted cylinder (fillet radius = cylinder radius)
Rear lights

first get the border of your surface (_DupBorder) and project it on the CPlane (use the _ProjectToCPlane in the rear view). Tell you don't want to keep the original curve:
For these kind of curve work, always project before you work on it. If not, curve fillet won’t work...

Do a 0.3 Offset inside the curve. Corners are screwed but we don’t mind.

Draw now the curves that will limit different parts in the lights. Draw one curve each time.

Use the offset command again, will a 0.15 value. Use it twice for each curve, on each side
Delete the first curves, and then build fillets in the corners with the radius of your choice. You can choose to join curve while building fillets. (Prompt option)

Duplicate the light surface, move it a bit backwards "in" the car, and trim it with the curves we just built. _Extrude the borders:
In the side view, draw curves that fit the shape you want to give to the inner reflective surfaces of the light. Usually these surfaces are spherical or pseudo-cylindrical in order to reflect light coming from the center bulb.

Build some Sweep1 with these crosssections (build the rails in the top view)
Trim these surfaces with the extruded surfaces. I also personally trimmed them with circles to put LEDs in it.
Stage 14: Exporting 3ds/igs

While you’re modeling in Rhino or once everything finished, you’d surely like to render that all, but not necessarily with rhino. I personally render my models with 3dsmax. Here is the way to export your rhino files into Max. Rhino offers lots of exporting formats because he’s largely used by industrial designers all around the world.

We are going to learn how to go from Rhino into max using the 3ds format (mesh) or the igs format (nurbs). I personally use the 3ds now but I’ll show you the good side of the igs export.

3ds export:

In rhino, keep only the part you want to export. You should export each part alone, because the mesh preview is pretty long depending on the power of your computer. Moreover, parameters may change from one part to another. Finally, sometimes you don't want the same level of detail on all parts.

I picked the front fender:

Right click on the Save Icon or choose Export in the File menu. A window appears, asking for a file name and for a format. Choose the 3ds format:
In the pop up that follows, click "Details controls"

Parameters are rhino defaults parameters (not on my pic!)

- **Maximum angle**: maximum angle between 2 triangles
- **Maximum aspect ratio**: maximum ratio between longer and shorter length of a triangle
- **Minimum edge length**: Minimum edge length of a triangle
- **Maximum edge length**: Maximum edge length of a triangle
- **Maximum distance, edge to surface**: maximum distance between a triangle edge and the mathematical theory nurbs surface
- **Maximum initial grid quads**: size of the initial quad grill that Rhino put in each surface before mesher the seams.

Other parameters are to be checked or not:
Refine mesh: always check. Allow the mesher to work properly on seams.

Jagged seams: do not check. If you check it, each surface is independently meshed. Seams are off.

Simple planes: do not check. Ignore it.

Weld: check or not. I never saw any difference in Max.

Pack textures: do not check. Ignore it.

Let's get back a bit on nurbs theory. Nurbs are mathematical objects. They can only be represented by small faces. So mesh is used. We you calculate an approximation of the surface by a mesh, we always do an Error. The smaller this error is, the better the mesh looks like the perfect surface. When the Error tends to zero, the mesh get dense and tends to the mathematical surface. The Maximum distance, edge to surface represents this Error in a way. Think of the representation of a sphere in max. When you increase the number of faces, the sphere become more and more "round", and the distance between the faces and the perfect sphere decreases. It works the same for nurbs surfaces. If rhino mesher were perfect, this parameter would be enough to mesh whatever the part you want. But you often have to play with the other to have something nice. Try this:

Click on Preview, wait and see the meshed surface:
You can move, pan, and zoom to inspect the geometry

In some places, Rhino will do this kind of crappy mesh (I never understood why because there is no surface variation here...)

To get rid of that, you can decrease the "Maximum distance, edge to surface" parameter, or play with the other ones. I often use the last one and put a value of 50 for example. Sometimes you'll have to use the "Minimum edge length" to avoid the numerous little faces that Rhino like to create in some places...
Don't be scared of preview calcul time. It can be pretty long with complexes polysurfaces. Check the Prompt line to see how many faces you got.

| Meshing . Press Esc to cancel |
| Created a mesh with 11736 points and 12751 polygons. |
| Meshing . Press Esc to cancel |
| **Click OK to keep mesh** |

Depending on your goals, choose a maximum value for the number of faces. But whatever the face number, try to keep it under 65000. The problem is that Rhino counts quads as 1 face instead of 2 like max...

In 3ds files, maximum face count for a piece is 65536. If you go over this number, the part will be split into 2 parts, in a crappy way. So (and with the quad issue it's not easy) try to remain under the 50000 faces, which is already a big part. If not you will have more than 1 million face for the whole car, which is not really practical to work with afterwards.

Here is a nice export re imported in max:

![Image of a 3D model](image)

Apply a material with a non zero specular level to it to check it. Export it again in Rhino if you're not satisfied with it. Keep Rhino and Max opened and jump from one to another to find the perfect export parameters.

When you found the good parameters, you will notice this in all cases:
What were surfaces seams in Rhino is not welded in Max!

What you must not do: select all vertexes and weld everything. You'll get some mess up areas because some triangles are deformed and because shading problems.
What you must do: select the big surfaces in the volume sub-object mode, and Hide the other (Invert selection and Hide it). Select the regions of vertex you want to weld. Weld them using a small value, since they are exactly at the same place.

After that the surface should be ok. Good luck for the whole car...
iges export:

When exporting in Rhino, choose the igs format:

In the pop up, choose Details controls, then 3dsmax 5.0 in the first item. Check that Iges tolerance is the same as your Rhino global tolerance, which is 0.01 if you haven’t changed it.

Import in Max.
You have now nurbs object in max. Check the modifier stack for sub objects:

In Surface sub object mode, you have this menu:
The mesher works a bit like the Rhino mesher, but with few parameters only.
You'll notice there are two configurations: one for the viewport, and the other for the Render
This is one advantage of the igs export, that allow light meshes in viewport to work easily and nice
renders.

Don't change the parameters of the viewport configuration and switch to render configuration.
You have 3 presets. You can try the 3 ones and look at the difference when rendering.
You should see the difference when you zoom near to the surface.
Be aware that "low" in the render is already more accurate that the "high" in the viewport...

So even with the "High" preset in the render mode, you see holes between surfaces:
You have to increase the "Merge" parameter.

With the default 0.01, this is what I get
Increasing now to 0.5 is better:

The problem with nurbs is that you don't really control the number of faces you have.

One can see that here I’ve used 58000 face with the nurbs, whereas I only had 15000 in 3ds format.
I personally think that both techniques are equivalent. Igs is faster to work with but generates often heavy files. So if you have a powerful configuration, choose igs export. 3ds export is good because you control the mesh, but the export phase is pretty long, and the post work in max too...

For every question about this lesson or about the tutorial, email me. Good luck.

The Rain Knight 2005 (c).