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Subject: Creating balljoints in SketchUp

Posted by [Drawn-SteelHero](#) on Thu, 11 Aug 2011 22:04:59 GMT

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My previous experience with SketchUp has been... sketchy, at best, and I've never really stayed with it long enough to get good at it. But after talking to Asphalt and trying to solve his ball joint issue, I took another crack, and actually managed to build what (theoretically, at least) should be a printable, working ball and socket. A lot of the mechanics of it is stuff I've learned through making AutoCAD models, but I'd not got this far with SketchUp before.

Surprisingly, very few special tools are needed. although I did make use of the RoundCorner plugin to fillet the corner of the socket.

I've worked around making a ball of 4mm diameter, using a shaft of 3mm diameter, since that's what I use most for my own models. However, since SketchUp doesn't like working at such small scales, I've scaled it up by 100; I'd scale it back down on completion.

First off, make the male part of the joint. I've done this here by essentially drawing a cross-section of it. I've included a shot with all the dimensions I used. Once the cross-section is done (remember to clean out any unnecessary lines), create a circle perpendicular to the cross-section. This will form the path through which it'll be lathed.

(NOTE: All the pics here are clickable thumbnails, which link to larger images for clarity.)

Select the circle, pick FollowMe under the Tools menu, then click on the cross-section. The flat shape will revolve round the circle creating the male part of the joint.

The socket will be based at the centre of a 600mm cube, but it needs to have an open side, which is easiest to do by cutting the piece in half. So draw a 300x600 rectangle on the base plane, and Pull it up by 600. Then draw a new cross-section on the square face of the box. This is exactly the same as the previous one, except I've given the shaft piece a 320mm diameter. This means that the shaft on the ball is slightly narrower than the channel housing it, and it'll revolve more cleanly when moved.

Again, clean up your cross-section, draw a perpendicular circle, and use the FollowMe command just as you did to create the male joint.

Delete all the geometry sticking out of the box, and you'll have a nicely recessed socket! Now Pull

the face out by 300mm, and you'll have the 600mm cube we were after.

Now we want to round off the edge of the cube, so that it can rotate freely without its corners colliding with the adjoining component. This is where the RoundCorner plugin comes in handy. I've chosen a fillet radius of 250mm, since I usually have my components a millimetre or so apart anyway. If you don't, then you'll probably want to go for 300mm. Either way, put in your radius, click the bottom two edges, and then click outside the box to confirm it. Bosh.

That'll probably leave some odd bits of geometry, so you'll want to erase any lines sticking out and fill in any gaps you get.

UPDATE: At the time of writing, I achieved this cleaning up using a long-handed process of deleting any extraneous geometry, leaving several gaps in the mesh, then drawing in new faces to fill them in. I now know this was unnecessary.

Instead, highlight the whole socket, right-click, and select "Intersect Faces > By Selection". This automatically generates dividing lines wherever panels intersect, so any protruding geometry can be deleted much more cleanly.

And I decided to Soften the edges round the curves, just to make it look nicer. But you probably don't need to do that.

Now you've got a pretty decent socket, but when you try to install the ball, you'll find there's nothing stopping it from coming out unexpectedly. You need something inside the entrance small enough to let the ball in, but large enough to stop it coming out without actually pulling it out. Through my own trial and error, I've found that for my 4mm sockets, I'm best off adding "nubbins" (that's my own technical term) that protrude by 0.3-0.5mm each side. Any smaller won't keep the ball in, but much larger won't let it in.

To that end, I've here created a 70mm radius sphere, and copied it 240mm to each side of the socket.

Now to install the nubbins. Move the spheres forward by 125mm, then up by 500, level with the centre of the socket.

Now just select the socket and spheres, and Intersect Selection (Edit menu). Once you've erased the bits of sphere protruding on the outside of the cube, you're golden!

Once I finished those, I turned the socket into a Group, and did the same with the ball, so they don't merge when moving them together.

Hopefully that's of some help to those of you trying to design moving parts.

Hopefully it's of some help to me - I might actually try more SketchUp modelling now...

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Asphalt](#) on Thu, 11 Aug 2011 22:36:30 GMT  
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Now to give this one a try. Hopefully I don't get any weird stuff flying off the surface of the form like I usually do.

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Asphalt](#) on Thu, 11 Aug 2011 22:44:43 GMT  
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Yeah lots of unwanted geometry, any idea what I did wrong?

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#### File Attachments

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1) [unwanted geometry.JPG](#), downloaded 82 times

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Drawn-SteelHero](#) on Thu, 11 Aug 2011 22:58:07 GMT  
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Blimey, that's weird. It looks like it tried to follow a pretty screwy path. Did you have the circle (the one you were using as a path) lined up properly relative to the cross-section? That's all I can think of offhand.

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Asphalt](#) on Fri, 12 Aug 2011 00:27:21 GMT  
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I thought I did.  
perhaps not.

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Drawn-SteelHero](#) on Fri, 12 Aug 2011 00:56:05 GMT  
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It can be difficult to be sure, I know; it looks like it's moving in one plane, but it turns out to be in another.

You can make certain of your planes by drawing a small cube somewhere and using its faces as plane references. Before you go to draw, say, a circle, hover over the face you want the circle to be parallel to, and hold Shift. That will lock the reference plane, so when you click to start the circle, it'll be facing the right direction.

That make sense..?

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Asphalt](#) on Fri, 12 Aug 2011 01:51:50 GMT  
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Yep, did that. My guess is I was a little off center somewhere.

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Subject: Re: Creating balljoints in SketchUp  
Posted by [Drawn-SteelHero](#) on Mon, 15 Aug 2011 21:12:02 GMT  
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I've just had a realisation that's made my life a little easier when using the RoundCorner on odd shapes, like the socket here.

When I built the socket for this tutorial, after filleting the corner (so the component has clearance to swivel in more directions), I was left with a bunch of extraneous geometry that I had to tidy up manually. This basically consisted of deleting whatever stuck out, and drawing in some extra faces to fill in the resulting gaps.

I've since realised this was unnecessary, and I could have done a much neater job, much more quickly.

Once you've filleted the socket with RoundCorner and got that unsightly geometry, highlight the whole socket, right click on it, and select "Intersect Faces > By Selection". This will automatically generate lines wherever planes intersect, so that you can delete the extra geometry simply by viewing it edge-on and removing whatever sticks out.

I'll edit this info into the first post as well..

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