PAUL MUNFORD: Good morning, everybody. Thank you very much for coming along. This is Thursday morning, so this is the last day. So it's really good to see so many people here.

OK. So this class is Complex Typology Class A Surface Modeling with Autodesk Inventor. Just to quickly do a class summary, so this is from hack and whack to planned and perfect complex topology. And learn the objectives here, how to use the language of curvature continuity with confidence, discover some of those hidden away software surfacing tools, building complex topology from individual surface patches, and converting surfaces into a solid model.

But on top of that, what I really want you guys to get out of today is the confidence to go ahead and play with this stuff. This is only an hour and a half class. I really can't cover everything about surfacing. But I want you to have a bit more of a theory so you can go and try these things out and practice and then give me some feedback, and let me know how you get on.

So just to let you know a bit more who I am, My name's Paul Munford. I'm an application engineer with a company called Graitec. We're an Autodesk reseller in the UK. And I've just been to Graitec for about three months. I actually just started off on the tools as a carpenter and building scenery for TV and film.

I did that for eight years, and now I moved into the drawing office and learned Autocad, and spent 10 years with Autocad and Inventor in the construction industry doing woodworking trades, joinery, and mill work, and that kind of stuff, and the last three years as a CAD manager. And I'm a big CAD geek. In my spare time I write for *AUGIworld* Magazine, *Develop3D* magazine. And I just finished a book, *Mastering Events 2016* for Wiley, which comes out in December.

And I've also done some training courses for Infinite Skills. And in fact, this class seem like a father class of Infinite Skills. I'm just trying to pack it all down into an hour and a half to give you a good indicator of it.

I'm going to try and do as much theory as I can in the first half of this class. And the theory really is applicable to any software. So whether you use Inventor, whether you use other software tools, hopefully you'll get something out of that. Second half of the class is going to be a bit more demo.

So I'll be up in the machine. I'll be showing some stuff inside inventor. I'm happy to keep it a bit of a dialog. So if I do you go fast, or if you want to see me do something again, then just shout out.

OK. So what is surfacing? So surfacing modeling is creating kind of more organic shapes, industrial design. And it's a method of modeling using surface patches. We create one surface at a time. And a surface, what is a surface?

A surface can be considered to be an infinitely thin skin stretched between boundary geometry. Typically surfaces are open, but we can close them off once. Once we close them off, we can turn them into solids. So a solid is an enclosed volume completely surrounded by faces that can then have Information attached to it, like weight or mass and that kind of thing.

I also want you to define difference between prismatic and nerve surfaces. So prismatic surfaces are pretty much built out of lines and arcs. And prismatic modeling is what most people do with [INAUDIBLE]. It's a standard thing. So it's rectangles, circles, pyramids, that kind of stuff.

The last example over here would be considered a nurb surface. So a nurb surface is a surface built from splines. So why inventor? Well, I have to say, if you're really interested in surface modeling, you don't use inventor.

You'll need something like Alias. This class is really aimed at people who either don't have access to something like an Alias, or don't have time to learn it, or who pretty much 95% of the time you're very confident modeling with Inventor. And it's just that last 5%. It's just that kind of, when you get something a bit more complex, you're not quite sure where to start, which is why I want to cover so much of the theory because I really want to give you guys a sort of grounding, and how do I start, where do I start? So we use Inventor because that's what we got.

OK. So let's begin with a bit of theory about surface continuity. So when we're doing organic shapes, it's all about the smoothness, particularly where the shapes join at the seams. And where they join, that's called surface continuity. And that's the first thing they're going to look at and try to understand.

So curvature continuity is measured in units of g, which I believe just stands for geometry. So typically, if two lines aren't circles, if they don't touch, then there's no g. There's no continuity between them at all. And I think of that like this.

You can have beautiful curves, they can look really great, but if they don't touch, you got no continuity. So the first degree of curvature continuity is touching. Usually, we're talking about G0, we're talking about something that comes to a sharp point. And you can see here the constraint we would use in Inventor to create 2D curvature continuity between two curves, would be a coincident constraint.

And I kind of think of that like this. So it's usually sharp curves, the kind of stuff you'd get with your prismatic modeling. Now, as we go through curvature continuity, what we're doing is we're matching information between the two curves. So we have to have touching to be able to have curvature continuity.

But as we start to match, we start to match properties. So the first property we're matching would be the Tangent Property. So the two curves are tangent to each other. And we call that G1. So one property is matching between the two curves.

So I think of that like the Batman car. This is in the Michael Keaton Batman, when the grappling hook comes out the side of the car, and grabs the lamppost, and it spins 390 degrees. So G1 is usually considered to be kind of engineering fillets.

So when you put a fillet on the sharp edge, and you get that tangent continuity between the sharp edge and the fillet, that's G1 continuity. The next one we would match up is curvature. So what we're matching here is we're matching the degree of curvature at the point where the two things meet.

OK. So if we had two arcs, the curvature would be consistent. In this case, we're doing spline. So the curvature can change along the degree of the spline. But at the point they meet, we got the same curvature. And just to backtrack slightly, I've got a tangent constraint on the last one. This one I'm using a G2 constraint.

OK? I've turned on the curvature cone here so you can see the degree of curvature on these two splines. And we know the degree of curvature is the same because the height right here is the same. So where those two curvature cones come in, we got the same degree of curvature. And I think of that like this.

This is like a race car with smooth curves coming in and out, smooth accelerations that goes around the corners. And this car's from *Battle of the Planets*. I used to watch them as a kid.

And I was surprised to find it's actually called a G2 car. So that was pretty cool.

So the last degree that we're going to talk about today is acceleration. Now actually, you could continue matching properties. But realistically, G3 is as far as you would usually go in surfacing. And in fact, there is no G3 constraint in Inventor. So we can coax G3 out of inventor by careful manipulation. But it's not a standard out of the box thing.

So G3 matches. It's touching, it's tangent, it's got the same curvature where the two curves meet, and we have the same acceleration. And acceleration can be thought of the degree of changing curvature. So right here we start off with not much curvature where it's pretty straight.

And as we go up we got more and more curvature. And here we can see that the curvature cone graph here is tangent at the top as well as the two lines being tangent. Now, that tells us we've got G3 curvature continuity. And again, the only way you can get that is by manipulating that spline until you get that curvature continuity.

So I think about this. This is super smooth. This is as smooth as you can get. Now typically, you only really want G3 on some of your most important surfaces. So it's not something you're going to be doing all the time. But you can get it.

OK. So that was 2D curvature continuity. But the same rules apply with 3D curvature continuity. So we can see here again, G nothing. They're not touching. There is no continuity. G0 G-0 is typically touching but shark. G1 will be tangent. G2 is tangent plus curvature. And G3 is tagent plus curvature plus acceleration.

OK? So that's how we understand whether we've got smooth curvature in our model or not. OK. So I'm going to move onto another important concept. This is topology versus Geometry. So topology can be thought of as a number of faces and edges there are in your model. Geometry is the shape.

So on the left-hand side, I've got two surface models that have the same topology, one face with two seams there, one seam on the bottom and a seam up the side. On the right-hand side, I've got two models with the same geometry but different topology. So one has this seam going up the side and one is seamless.

OK. So we all know from the normal prismatic modeling that we do, that the same shape can be built in a number of different ways. So the way we describe that is using a number of different kinds of topology. And given the same shape, we probably will go-- probably in this room we'll all have a different approach. But it's worth thinking about your topology before we start the model.

Here's another important concept. And this is all stuff that I'll be using as I go through because I just want to make sure that there's a level of knowledge in the room. So I apologize if you know this stuff. But I can't assume that everyone knows it, so I'm going to just step through some of these words.

So this is surface normals. So the normal direction of a surface is really it's front and its back. And the normal direction can be thought of as a vector. A vector's coming off perpendicular to a surface. Now it's a little bit easier to see with work planes. So you may have noticed work planes have a distinctive, like usually an orange side, and the back side of it's blue.

So the orange side would be the face side. That's the normal direction of that work plane. Surfaces in inventor don't usually have an indicator of which side is the normal, which side is the face. You can usually tell. For example, if you thicken something, the direction it goes in is the normal direction. If you extrude some geometry, then it extrudes, that's the normal direction of the work plane that the sketch was hosted on.

OK? So again, it's just worth thinking about normals. It's worth thinking about how your faces, which normal direction you've got in your faces becomes important when we stitch them together later. OK. So this concept is called isoparms. Now, what you don't see in Inventor is you don't see how Inventor calculates surfaces. So I've tried to indicate that here using this analysis, this curvature cone analysis.

Isoparms are a grid. So inventor actually mathematically calculates surfaces as a grid. And it's a mesh square. OK? So this grid-- like in Autocad, you'd have Cartesian coordinates, x and y-coordinates. This grid for isoparms are known as Us and Vs. It's just the U and V direction of this grid.

But what's important to recognize is that inventor will always use a four-sided patch, so always a four-sided patch. So on the left-hand side here, we have a surface with a bit of a problem. You can see some of these patches don't have four sides. But in fact, they do. They are diverging into a single point.

That single point is known as a singularity because we have an edge in there, and everything calculated about that edge is being calculated as a 0 value. And not just Inventor, but any CAD system has a problem doing multiplication division of 0.

So you can build surfaces with singularities. And if that's your only surface in your model, that's great. But if you try and build more surfaces of that, you may have an issue. So I'll show you some techniques later to get around that. So that's isoparms.

OK. So there are a few things, a few phrases I'll be using I want to make sure everybody knew. So I'm going to talk briefly about patch layout. So when we're looking at something we'd like to build using surfacing, some we got to get smooth, surface continuity, the first thing we can think about is the theoretical sharp edge. So we can imagine when we take a square shape, and we apply a filet to it, the theoretical sharp edge is where the edges would have met before we applied the fillet.

So if it's something like this, the theoretical sharp edge would be somewhere right there. Now typically, it's a good idea to build to the sharp edge and then trim it back. So we got the edges, the theoretical sharp edge where they would meet. And then we can think about the blend line, so if we did a blend across that sharp edge, so we trim the surface back and apply a blend in there, where those blends need to start and finish.

OK? So one thing I struggle with in the past from the surface modeling is trying to put the blend lines in first, build the surface patch, and then blend. And that's just quite tricky to do. So I recommend you build to those theoretical sharp edges, then trim them back, then apply the blend in there.

So one more thing to think about after that, so typically you'd have three types of patches. So I'd say this is a primary patch. This is the first patch we're going to build. And we can build this without reference to anything else.

So typically, this might be a couple splines that are swept. And we could trim it back to make it that shape. Next, we'd have secondary patches. So these we've drawn up to that theoretical sharp edge. So secondary patches are patches that you have to build using existing geometry. You can't build them first, because you need to build a primary patch first.

And then the last set of patches would be the tertiary patches, which are usually those blends. So when we've trimmed them back, then we blend between the two. And there's one more, construction surfaces.

So one limitation in inventor, if we create a line in a sketch, and extrude that line, we can't apply any continuity or tangency to the sketch plane. So we'd have to apply a surface going in the other direction first and use that as a construction surface. So it's a surface that may never appear in your model. And you'll just put it right in there so you can use it to create tangency.

And one last tip on thinking about your patches and planning your model is flow. So it's keeping your loft so your sweep's flowing in the same direction. So again, those isoparms are all flowing in the same direction. So they match up nicely.

If you sweep things in different directions, again, you can have issues trying to get continuity. So it doesn't really matter whether you go one way or whether you go the other way, as long as it's consistent. So that's just a tip, help you out there.

OK, so creating surfaces. So how do we create a surface with Autodesk Inventor? Well actually, you'll find the surface option just about everywhere. It's not the standard. If you create a closed profile with Inventor, it will create a solid. And a solid really could be thought of as a short cut. It's a collection of surfaces that have already been trimmed and stitched together into a solid.

If we have an open profile, that in most cases Inventor will by default pick the surface option, not always, but most of the time. But you can override it. You can choose it yourself. And then you can create your surfaces. So most commands here you'll find here have got a surface option in it. So you have to make sure you pick it.

The thick and offset one, just take a look at that one, kind of come back and talk about that in a second. So that's it. You found them everywhere. Editing surfaces, well, most of the surface edit tools are in the surface panel of the modeling tab. So we have the trim surface. So that allows us to trim a surface to a sketch or to another surface.

Split face. So split face works on surfaces too. And don't forget that a surface creates a surface body in your model. So you can use body commands as well. Thick and offset we just scoped out earlier, could be thought of as copying a face. So if you thick and offset a face, choose a surface option, set it to 0, then you'll [INAUDIBLE] and copy that face.

Now, do have copy object in Inventor. But if you copy object, its non-associative to the underlying geometry. So when you change the underlying geometry, that surface won't update. If you use a thicker option, you can create a copy of a face that will update to the surface geometry underneath. So it's not something you would see. It's not intuitive, but it's a little tip.

So again, a surface will create a surface body. And you can operate on that body with some of the body commands. So if you want to move a surface, you can move a surface with the Move Body command. You just have to make sure you pick the body and not the surface. The Extend command-- so I go back in some of these tools here in the surface panel. The Extend command allows you to extend the surface. So you can select the edges and extend it. It's like untrimming it. So it's the opposite of Trim.

And then Rule Surface, so Rule Surface is new in 2016. I don't know if you have seen this one. So a Rue Surface allows you to select an edge of an existing surface and then create a surface of it off it that can be tangent or can be perpendicular to the edge. So that saves your hand to do it, like a line, and a sweep, and a 3D sweep around a curve. So that's a really helpful tool.

And the last one that could be thought of as a surface in too is Delete face. Now, one of things you'll notice, if you create a solid body in Inventor, you'll see a new body appear in the tree. If you delete a face, you'll see that body turn from a body into a surface body.

So by deleting a face, it's now open. It's no longer an enclosed volume. So even though it's gray, it's a surface. It's not a solid anymore.

Surfacing continuity tools. So again, these are kind of hidden away. A lot of people don't know they're there.

But in fact, in a lot of places, if you click on this little fellow at the end here, you'll have options to create curvature continuity between curves. And again, you'll notice we have G1, and we have G2. There is no G3. So if you want to create a G3 super smooth curvature continuity between services, it's all about how you create those splines that you then are going to build the services from.

OK. So that is a lot of the theory I want you to talk about. So it would helped you plan a more complex model. You can look at it in terms of its sharp edges, in terms of where you're going to put your blend lines in, in terms of the blends you're going to use, and some of the things that you think about in terms of curvature continuity.

So I'm now going to jump up here and start demoing stuff. OK. So there are two kinds of splines at Autodesk Inventor. There's the interpolation spline, and more recently we've got the CV spline. So I'm going to show you how these two things work.

So I will double click into this sketch. And you'll find the Interpolation Spline command up here tucked on the line. This is 2016. Sometimes the UI elements move around a bit. So I apologize if you're in a previous issue of Inventor. But this is 2016.

So I'm going to create an interpolation spline. And here's a tip. I recommend that when you create interpolation splines, you make it as simple as possible. So interpolation spline will create a curve between two points. Now we've created this curve, actually, we can apply curvature continuity to it using constraints because I'm using the G2 constraint there. And that is the best curve that Inventor can give me between those two original straight lines.

So I don't need to add anything else in there to get a nice blended curve. If you did want to add an additional fit point, just right click and choose Insert Point. There we go. And that will allow you to insert an additional point in there. And now you have a point that you can manipulate and move around to change the curvature.

You also have the option, if we right click somewhere near that point, we choose Activate Handle. When we activate the handle, that means we've got this handle that we can grab and pull and manipulate and push and pull around. And you can also dimension that point. You can dimension that handle.

Typically we do surface modeling, it's kind of underdimensioned or underconstrained, just because if you try to constrain every fit point in the spline you'd be there for a long time. So that is a lot different prismatic model. We like to lock everything down.

We also have the option here if we right click again to get a curvature handle. And the curvature handle allows us to adjust the degree of curvature around that point. And both of these I'd say can be dimensioned. This one is dimensioned. You can see it's a unitless value. So the highest that can be is 1, is a percentage of 1.

The curvature handle isn't just a radius. You've got to turn off the curvature. So here's a helpful thing. Sometimes when I'm working with interpolation splines, I can get out of control. So I just wanted to show you there's an option here to reset the handle or reset all the handles. And if we click that, it just goes back to the best degree of curvature we can get again.

One more option we have, if we look at the display curvature, so this is the curvature cone graph that I showed you earlier. So if we click on this, we can see what kind of curvature we've got. And we can begin to understand these relationships between the existing lines and the line that we've got. So we manipulate this to try and get a smoother curvature looking our curvature cone graph to see what we've got.

One other option you've got here is to choose Apply Intention. And if we choose Apply Intention and we bring out this little dialogue, we can use to adjust the degree of tension in that spline and smooth it off a little bit.

But OK. So interpolation splines are what we've always had in Inventor. They can be kind of tricky to work with. I'd recommend that you have as little or fewer fit points as possible to give you the nicest curve. But we do have a relevant ability of newcomer, the control vertex spline, which I also have to show you.

This time when we create points the curve doesn't go through the points we created. Each of these points is known as a CV or control vertex. And our curve is always going to be tangent to the line that comes off it. So by moving the point, we can adjust the degree of curvature, which is a little bit easier to deal with. We can add additional points, and again, just by right click to use this time instead of vertex.

And what you'll notice is when I click to insert a vertex, it's basically dividing that. It's subdividing each of those lines. OK. So those lines are called holes. So you have one CV and two holes to adjust the degree of curvature in this CV spline.

And if we click, we now have two CVs and three holes. If we were to add another one, it will subdivide again. And that means you got more points to adjust the acceleration of that curve in different places. So that's way simpler. I want to show you a couple of slight more practical example about how you might use that.

So I've dropped in here a sketch. So this is something we're using as reference to begin our surface model. So which spine we use in which case? Well, one of the nice things about an interpolation spline is an interpolation spline will create a closed loop. So if I work my way around this sketch using it as reference, when I hover over the closing point-- I don't know if you can see that there-- as I hover over the closing point, it creates a closed loop.

I'll hollow out that, see if you can see it. So we've got a completely closed loop of geometry. So interpolation spines do that really well. And that will maintain a G2 curvature. That will maintain a G2 curvature through each of those points.

Now, where I suggest you use a CV spline would be working on the outside of this shape. So you can have as many fit points in as you like, use those fit points to adjust until you get the kind of curve you're looking for. And then you will need to add additional curves for something like this.

And what we'll then do is apply our curvature continuity. So I'm using a G2 constraint to apply continuity. And then if I select both of these curves, right click, and show curvature, we can see that's actually pretty good curvature. I'm just going to go to select on the line again and choose Setup Curvature Display.

And that brings up a little dialog that allows us to adjust the height, just to help us see what's going on. As you can see, as I've adjusted the curvature here, it adjusted the curvature to the next one. So what we're looking for is really good tangency here. And that shows we've got the same degree of acceleration going in across this curve and across this curve.

So that's 2D, CV, and interpolation splines. I'm now going to jump in and talk about 3D splines. So in this example, I've created two sketches, the 2D sketches on different planes. But what I would like to do is create a blend that goes between the two. And I'll do that with the 3D spline.

So we start with a 3D sketch up here. Choose 3D Sketch. And I'm just going to create a control vertex spline between this point. I'm going to add [INAUDIBLE] between this point and this point. Choose Create.

OK? Now, in order to get that to be curvature continuous, I'm adding git points in there. So I'm going to choose Insert Vertex and just add some vertices in. Clicking it each time is subdividing. I'm going to add in three vertices. Now, although Inventor is automatically projected through this point for me and this point for me, in order to apply curvature continuity to this line and this line. I'll need to bring them into this 3D sketch. I'm going to use that as a tool called Include Geometry.

So in 2D we would have Project Geometry. But in 2D it's called Include. So our choose include this line, include this line, and it projects them into my 3D sketch. Now I can use the same continuity tool. So it's the curvature continuity constraint between this line and this line this line and this line. And that's given me the best blend I can get between those two curves.

But once again, if I want to control that a bit more I can turn on Display Curvature. And I'll display the curvature for those two curves as well. And you see I've got quite a good acceleration going from this curve to this one. This one's maybe a bit sharp. So then you could begin to use these CV points to adjust that curvature. See if you can get a slightly better result.

So we can use 3D splines to build these theoretical sharp edges. But typically-- just wanted to show you one more thing here, actually. So again, I just wanted to show you an example of how you might use an interpolation spine, again, for using a closed surface. So if I create a new 3D sketch here, once again, I've got some geometry underneath. This is 2D geometry sketched on two separate planes.

So this time I'm going to use my interpolation spline. And I'll start at one point, come down to the next, back up, back down, then I hover over that last point. Again, you can see it creates a closed loop with G2 curvature continuity built in. So that gives me my Pringle, which we could then use. We could then create a patch, then thicken it, and that would give us our surface model.

Here's a thing. Tell me how well that shows up in the projector. You can kind of see when you have a surface in a solid overlaid on top of each other.

You can see the isoparms there. OK? So along the Inventor is creating real geometry in the background. It actually uses the mesh, that mesh that it uses in the background to create the visuals on the screen. So you can kind of see those isoparms as yo go through.

OK. OK. So we can use 3D splines. But typically it can be quite hard to define a 3D spine in the middle of all that space. So typically, we would use 2D sketches. We want to map them onto a 3D surface. And again, there is some more tools in the 3D sketch environment. I want to just walk you through so we know what they are to do.

So if I'm going to project these curves down onto the surface, once again, I'm going to start with a 3D sketch. We're going to be looking at these tools here, projectiles. So the first one I'm going to choose is projector surface. I'm just going to choose the curves I want to project. I'll choose the face first, then choose the curves that I want to project.

And finally, it's asking me for direction. By default it's just pick the ends of that axis in my model. But I'm going to choose this axis because I want it to be tangent to the curve underneath, make sure I've picked that curve. Choose Apply. So you can see how it's projected onto that surface in that direction, using that direction as a vector.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: I have to find that vector by placing a sketch point in the curve that I used to create that surface, and then using sketch point and the face of the surface itself to create the vector. So the question was then, do you have to create that vector yourself or can you use the normal direction of the surface? Actually, I'll show you the next tool It might help.

So this time we're going to look at the projector closest point, which I guess would be the normal. So once again, choose the face, choose the curves we want to project. This time there is no direction option. Inventors going to work out for us. Choose Apply. Let me just reset this so we can see this a bit better.

Now this time it's projected this curve to the closest point, projected this 2D sketch to the closest point you can find on the curved surface. So if I create a couple of axis it might help us just to see that a bit better. Create and axis here. Create an axis here.

So can you see that? Can you say it's just picked the closest point on that curve? So that means it has distorted our 2D sketch quite a bit. So there's one more option we can look at. Double click back into the Sketch Tool Editor.

The final option is the Wrap To Surface, so same sequence. Choose the surface, choose the curves, choose Apply. So it's projected that curve on the surface. But this time if I take a measurement here, I can see that this line is 25 millimeters. And if I measure the same line down here making sure I get the line and not the radius, you see that's also 25 millimeters.

So depending on which Project Curve Tool we choose would depend whether we preserve that geometry or not. OK? So by Wrap to Surface, we get exactly the same shape. It's just projected onto that surface.

So these tools are really useful for be able to build up sketches and have a little bit more control rather than using a 3D spline and kind of having to pick points in space and guess.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: In the last case, that's a good question. It is going down. But you can adjust the position down here by adjusting the position in this one. So I guess you'd move that around to get where you need to be.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: The question is which direction is the last one going? And it looks like it's going to the z of that one. OK. The last one I want to show you the Intersection Curve Tool. So this is an extremely useful tool. So once again, I've got two 2D sketches. But what I really want, I really want the combination of both. And I don't have to project surfaces to create that.

So once again, I'm going to start a new 3D sketch. I'm going to use this tool here, Intersection Curve. So I use Intersection Curve, choose one geometry, choose the other geometry, choose OK, and it gives me the 3D equivalent of them both, this kind of mapping one curve onto the other curve.

So that's an extremely useful tool, being able to create your 3D curve for your theoretical sharp edge. OK. Any other questions about 3D curves?

OK. So surface analysis, so once we created some surfaces-- so we've created our 2D curves. We projected surface geometry off them. We want to see that that seam in 3D has good curvature continuity. How do we tell?

Well, there are some tools inside Inventor we can use for analyzing curvature continuity. Now, I'm only going to show you the two main ones that I think you'll use most just because we're a little bit limited on time. Let's just shut some of these down that we're finished with. So the first thing I recommend-- this is something I like to do when I'm doing surface modeling-- is I light to use this silver appearance.

So it's up here under Appearances, Silver. And the reason I like doing that is because you can actually see pretty well as you work whether you've got any issues with curvature continuity. As you're spinning your model around, you can kind of see where you've got nice, sharp edges and where you got good blends.

But if you did want to do something a little bit more precise, you'll find in the Inspect tab you'll find your analysis tools here. So we're going to look at Zebra Analysis and Curvature Analysis. Draft Analysis is particularly useful if you're doing plastics, allows you to detect when you've got any undercuts. Surface Analysis is another way of looking at the curvature continuity. And Section Analysis is particularly good, again, if you're doing plastic parts to detect whether you've got any problems with thin walls anywhere, making sure you've got consistent wall thickness.

But for today, I'm just going to to show you the first two. So Zebra Analysis, when we pick that, it's going to give us some options. But I'm just going to say, OK. And what that does is project these black and white stripes all over our model. And what that allows us to do is to look at the curvature continuity across this model and see if we have any issues.

So we can see if we just shift to make sure I'm on Slate Faces and Edges. This is basically two main patches. I've got the primary patch here, which I've then trimmed back and add a second repatch in.

And then at this point I've asked for curvature continuity. And as I look across here, there's a little bit of a break there. But actually, it's pretty good. We can see how smooth these type of stripes are cross this joint, across this seam. Right here, where I was looking for a style increase, I can see there's definitely a break there.

So it's a visual tool, the zebra stripes are a visual tool if you inspect your model and see whether you've got any kind of issues there. Just a little hint, it's not too obvious. If you right click here, you can turn off the visibility analysis and turn it back on again.

The other option one to show is the curvature. So the Curvature Tool is right here. So this is the 3D equivalent of that 2D curvature tool we looked at with our splines. So once again, we would choose a face or a couple of faces we want to work with. I'll just pick the defaults and choose OK.

So this is going to project onto their curvature cones. And the distance apart they all depends which option which chose. That allows us to see the degree of acceleration we've got going across our model.

So in this circular feature on the top, you can see the acceleration is pretty much the same the whole way around that curve because it's a circle. So acceleration is the same. It's a constant radius. We can see down the front here the acceleration changes. We've got more curvature at side. It flattens off across the top, and then we have nice curvature again.

We've got good curvature in the seams. That's nice. We do have a bit of a wobble in the front.

So you can see here that's maybe not as smooth as we could get it. But it rises right down the front there. I'm cool with that.

So there's a couple of tools that you can use to inspect your surfaces after you've created them. So you can kind of analyze whether it's as smooth as you would like it to be. It can be quite deceiving when you're looking at something on a computer model. It can look pretty smooth. If you try to 3-D print it, you might find it's not as smooth as you thought.

OK. So I'm moving on. What I would like to do next, what I've called surfacing gotchas. So there are some slightly weird things that you might need to deal with.

Actually, they're not weird. Inventor does it's geometry in a different way. If you don't understand it, it can look weird. So I wanted to give you a few hints and tips about how you might handle some of these issues.

So the first one I want to look at is high curvature. So we can say that this line here, this straight line we could say has low curvature. Or you could even say it's got no curvature. Unless your math geek, and it's probably got infinite curvature. That's not the way we're looking at it.

This line here definitely has low curvature. OK? So this one here is definitely high curvature. So where that could give is an issue is if we try to do a sweep around that curve. So I'll just show this. I'm going to sweep.

I'll choose the Surface option. I'll choose my profile. I'll choose my path. I'll choose OK, and it follows. Now, how many people know what happens if I click on the red text?

Cool. Here's a tip. It shows you a preview. Who knew that was there? I didn't know that was there. Nobody ever showed me that when I started learning Inventor.

Now, if you try and use your mouse, you can't navigate this. But if you're using a 3D connection control like I am, you can keep working on the model so you can see. The reason that hasn't built is because that curve is self-intersecting because of the high curvature. So now we know that's the issue. We can work around it.

And the work around here was to do it as a loft instead. So I had to replicate that 3D sketch on the other side and then do a loft from that edge. So there's a loft from this curve to this curve using the previous surfaces [INAUDIBLE].

So we do them with high curvature, you may need to think outside the box a bit and do something which is maybe not the most intuitive option. But that little tip there about clicking on the red text is a great diagnosis tool to help you realize what the issue is.

OK. So next one I want to talk about its near tangency. So in each of these surfaces I've created two surfaces, which are then joined together. And we can see here these two surfaces, it's pretty clear these are tangent where they join on this seam. And it's pretty clear here these are not tangent.

So this would be a G2. This would be a G1. So you've got this sharp seam where they join. This one here, it's a little bit tricky to see whether that's tangent or not. So in order to try and find that out-- so I did some curvature analysis on this. And I can see here that these two curves have got G3 curvature continuity.

So the curvature cones are telling me that where these two join on the seam, the curvature cones are the same length. So we have the same curve. And they're nice and tangent across the top as well as the curves themselves being tangent. So the curvature cones tell me I've got good curvature there.

We can see here on the one that's not tangent, we can see that although the curvature cones are the same length-- so we've got the same curvature-- they're not actually touching so we can see that we don't have that curvature continuity there. Now, we can also see on this one, now that we've done the curvature analysis, we can see that we don't have tangency because these two lines are not projected in the same direction.

But there is a slightly easier way to detecting near tangency. And that's just by selecting on the seam. So if I select on this seam, nothing. If I select on this seam, Inventor says, OK. What do you want to do? Do you want to do a fillet or do you want to do a chamfer?

If I select on this one, it offers me the same options. Do I want a fillet? Do I want a chamfer? So I know that that's near tangent.

OK. So if you ever get some slightly weird things going on in your model, and you think you've got good coach continuity, when you try and do your analysis, it doesn't look so great, maybe just select on that and see whether you've actually got tangency there or not. OK. Moving on to sliver faces.

So a sliver face, I'm going to quickly go into Shaded With Edges View. So sliver faces are typically more of an issue when you're dealing with imported geometry. So if you bring in geometry that's in a neutral file format or geometry from another CAD system that comes in as a surface, sometimes you have this little issue with sliver faces.

So if I zoom right in here-- so I've actually built this sliver face. And obviously, I recommend if you're building it yourself, just don't build sliver faces in. But I've had to build it to show you.

Now, the issue with the sliver face, just like the singularity we looked at earlier, it's about curvature degradation. So this tiny little surface, inventor is trying to do a lot of math within a very small area. And it's struggling. And that's why it doesn't like sliver faces.

But once you've detected you've got a sliver face, it's actually quite easy to fix. That's where my Delete Face Tool would come in. So I can just delete this face. But I'm going to use the Heal Option. And when I use the Heal Option and choose OK, Inventor just takes the face out and gives me the best drawing between the other two. So it's pretty straight forward.

But it's that Heal Option again, you won't even notice it's there. But the Heal Option's a really useful tool. OK. So the last issue I want to look at, let's go a bit more into singularities.

OK. So I would like to build a triangular patch. And I've created my theoretical sharp edges. So now I'm ready to create a surface. So where I'm going to do that, I'm just trying to line this up. I'm sorry if this makes you feel seasick by the way. Watching somebody else use a 3D controller can be a bit-- I'm going to create a surface using the Patch Tool.

So the Patch Tool just allows me to select some geometry. And then it will give me the best surface that it can. And again, this is a new feature in 2016. I don't know if you've noticed this. If you select an open geometry, it will give you an untrimmed patch. It will give you the best patch that they can, which is great, because in previous versions of Inventor, if you gave it an unclosed loop, it would just follow.

Now, it will always give you something, which is good. But if I give it the closed loop, it will give me the best patch it can. So I choose OK. That's my surface, and that's great.

But actually, if I look at that, so if I look at the curvature cone graph for that, you can see that what Inventor has done is created a trimmed surface for me. So you can see I've got no convergence here. So I've got no surface degradation at the edges.

But in a closer patch, Inventor will always create a square surface with those rectangular UV isoparms and then trim it back to the shape. OK, so that's kind of handy. But you can see we haven't had much control over that. And towards the edges we've got these weird sort of crunchy things going on.

That might give us a bit of an issue. If we try to build more patches off that and maintain curvature continuity, maybe that first patch, that primary patch isn't as good as we could get it. So how else could we do that patch?

Well, another way you could do is doing it with a loft. So a loft, again, with the surface option, we'll choose our first edge, go back, click to add another edge. But in this case, we'll pick a point. So that gives us a loft from an edge to a point.

And then we'll come over and click in the rails, and we'll choose that as our rails and choose OK. So loft to point is another great way of creating surfaces. But if we look at the graph again for this one, this is even worse because now we have got this diverge. We have got this singularity. Diverges got surface degradation going on there. Everything's coming to this point.

Now, just to demonstrate why that might be an issue, if I go to the Extend Tool in our surface tools, so I choose Extend, if I select this edge, you can see it will extend the edge quite happily-- not a problem there. As soon as I try to select one of these, nothing. They can't do it because it can't work out what's going on in that corner.

So once again, if you've got issues trying to extend a surface, that might be why, and you might want to think of another way of doing it. So what I wanted to introduce here is the concept of overbuilding surfaces. So what I'm going to do is turn on my original surface, turn on my original boundary, get rid of my 3D sketches.

So what I want is the best combination between the boundary patch that created a nice surface. And then that 3D sketch that I created is going to give me my theoretical sharp edges. So I'm going to do that by overbuilding the surface. This surface is bigger than I need it to be. It's bigger than the geometry I'm trying to define.

But then what I'll do is use the Trim Surface Command right here. And I'll trim up the surface using my 2D sketch as a cutting tool. Now let's take a look at the curvature continuity on that one. That's much better.

Let me just adjust that a bit. Yeah. Good. So we got no degradation. We've got no singularity. We have got four-sided patches, again, that have been trimmed back. So this is an important sort of concept when surface modeling of overbuilding surfaces and then trimming them back.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: We can extend this one. That's a good point. Let me show you. So that extends really nicely. OK. So Inventor can work out the math on that one, no problem. To an extent, these singularities are a problem whichever CAD package you use. So it doesn't matter whether you're using Inventor, or Alias, or [INAUDIBLE], or [INAUDIBLE], or anything else. Try and not to build any singularities is really important.

OK. What are we going to talk about next? Converting surfaces to solids, very good. So we have looked at the model we're trying to build.

We have thought about where we want those theoretical sharp edges to be. We have defined those sharp edges in 2D or 3D space. We've overbuilt our first all primary patches, trimmed them back using the geometry we had, and then we've built in some blends using probably the Loft Tool or the Sweep Tool.

OK. So now we have all our surface patches. What do we do next? How do we convert that from a surface model into a solid [INAUDIBLE] is what we'd like to talk about next. There are three options I'm going to talk you through.

OK. So the first option is fairly straightforward. I'll use the Thicken Command. And I'll just thicken this surface. OK. So that's built my solid. Fantastic. That's the first way you can do these things. And I just wanted to show you briefly, if I use the Shell Tool on the solid I've got right here-- let's just scoot that out of the way.

So the difference between using Shell and using Thicken, is that Shell maintains this face here as it was, whereas Thicken gives as a face that is perpendicular to the curve. So if you do plastic parts, you probably want this, I'm guessing. OK? So you may want to build an enclosed volume and then Shell it out afterwards.

If you're doing sheet metal, you need this. You have to have this, otherwise Inventor can't flatten it. OK, so just a couple of different options there. The main two options we would use are Sculpt and Trim.

So if I use the Sculpt Tool, what that does is-- I kind of think of it like filling a bag full of concrete. So it allows you to select a whole load of surfaces, planes, anything you've got. And if it finds an enclosed volume in that, it will create a solid that represents that enclosed volume.

So if I select this surface here and say OK, that's failed. That was deliberate because it doesn't enclose a volume. I've got an open face here. So I've closed that off using the Patch tool.

Now when a choose Sculpt I choose the Patch and the surface and straight away you can see I've got a solid. So the Sculpt Tool is one option for creating a solid form, a collection of surfaces. But you must have that enclosed volume. The other option is Stitch.

So we see Stitch right here. And Stitch allows us to collect a number of surfaces. When we choose Apply, it turns into a solid. So the question is, does Stitch-- if there are any sort of continuity gaps between the surfaces, would Stitch close the gaps up? And yes. Well done. You have a tolerance there.

So if there was a slight gap less than 0.254 millimeter, it would bring the seams together and close it off. So if you are trying to stitch again, if you bought in somebody else's surface model that you're working with and you want to create a solid from a surface model, and you try and stitch the whole thing together, you might want to adjust that tolerance.

It does have some analysis in there. So it can show you the edges that are a problem. But that's not something I was going to go through today. Excuse me, everybody. Sorry.

OK. Converting surfaces to solids. Now, I wanted to walk you through some examples of how you might bring all this together. So I'm going to start by showing you how I built that style increase and that mouse shaped thing we looked at earlier. So I have here my surface I've created for that main primary surface patch.

And I have a sketch here that I'm going to use to trim that surface back. So I'll use the Trim Command, select my geometry, trim out the bit I don't want, and choose OK. So that gives me the shape that I want to use. So I'm going to reuse that same sketch.

I'll create a loft, once again, with the surface condition. And I'll loft from this edge. And I want to just pull as to make sure I've got the edge and not the face. Then I can choose the edge I want to loft it to, the final edge.

Again, just pull this in to make sure I've got the edge. And I find I can add some rails. So I can use this edge as a rail and I can use this edge as a rail. And what happens is it fails. And again, I set out deliberately to show you something.

So when you're creating loft, Inventor is really bad a using a rail that goes beyond the edge. I'm going to jump down, try and point to this. So this curve here I want to use as a rail, it goes beyond the point where the loft started, and Inventor doesn't like that.

So we need to fix that first before we can use that edge. So what we'll do is we'll use the Split Face Command. I'll use the same geometry, and I'll just split this face right here. So I'll split that face in two, choose OK. And if I just switch into Shaded With Edges View so you can see-- OK. So now I have a seam between those two faces.

So let's try that again. Let's see whether I get a better result this time. So a 3D model, loft, pick my first edge, pick the curve I want in the middle there, pick the last edge, pick my rails. Oh, it's already decided it doesn't want to-- oh, OK. Got it. OK, whew. Nobody likes a demo that goes wrong.

OK. So now I have my loft that goes the whole way around. And there's just one last thing I need to show you. So in the loft, to set our surface continuity, we need to switch to the Conditions Tab. And this is where we can choose the conditions that we want. So our first edge condition, we want to be G2s to get as smooth as possible.

And our last edge condition, we want to be smooth G2. These last two are the rails. In this case, I don't want to set any curvature continuity between the rails because I'm looking for that nice sharp style increase. So I'll choose OK there, stitch the whole lot together to create my solid.

I'll just switch back into Shaded Without Edges. And you can see straight away we got good curvature continuity here. We can obviously see the drawing. But we we've got this nice sharp styling edge at the back.

OK. So that's just one example of how you might create a style increase. And I just wanted to show you there about lofts and rails, just make that point that the rail needs to stop at the geometry you're lofting. OK. Good. So I want to move on here to talk about end caps.

OK. So capping off a surface, again, is more complicated than you might think. So I'm going to show you a few different options the kind of ways you could cap off a surface. So the first one I'll show you is using the Patch Command.

So we create a patch. Now, I'm using these edges. So I've built construction edges, construction surfaces, so I can choose the edge of a surface. If I chose a sketch, I could not set curvature continuity.

So I've had to build these constructions surfaces that maybe now are going to be part of my model A run. But they allow me to apply curvature continuity. So I create this patch, and I'll choose my curvature continuity condition. I'll choose the G2 condition again.

So that gives me a really nice little patch. That's good. It looks nice. But I wanted that to be a bit bigger. So how do I control that? You can control the weighting here. And again, the weighting will be a percentage of 1.

I think-- yeah. So you can put a value in there, which will change that patch. Now, if you're just looking for a nice patch and not too worried about the dimension, that's fine. But if you were to precisely control the height of that patch, it's a bit tricky. If you're good at math, you might do it. So I'm going to choose-- maybe I'll choose a loft to point instead.

So let's say our loft to point would work, again, making sure I pick the edge and not the surface face itself. Click back in, choose my point. Now, this time by default, the loft will create that pointy shape, which is not what I want. So let me just edit this.

So I go back to my Conditions Tab again. So I can see for the first condition I can set a condition relative to that construction surface. So I'll choose my G2 condition. So that sorted out my continuity at the bottom. You can still see that the points may be a bit pointy. So that can set the condition here too and set it to be tangent to that point.

That gives me a slightly nicer curve. And again, you can adjust the waiting of that to try to give you a slightly better curve. Now, this is not a percentage of 1 this time. You can put anything you like in there. So that gives me better curvature. And I have a bit more control over the height.

But if I look at this with my analysis, so once again, you can see on the left our boundary patch has created a rectangular patch, is then trimmed back. So that's pretty good. My loft to point though has created a loft with a bunch of singularities going out to the point.

That's OK, but my ability to keep working with that, and I'm not happy with it. So the last way of creating an end cap that I just wanted to show would be to create a loft again, but this time choosing the surface option. I'm going to start with this edge, click to add, chose my next curve, click to add, choose the last edge, again, just a pause, pick up the edge.

Then for my rails, I'll use this rail here. So that has me a high degree of control over that shape. I can make it anything I like.

And once again, I can use my conditions to set the start and finish condition to G2, give me the best curvature continuity possible. So end caps more to it than you might think. But I just wanted to show you some examples there of how you might cap off the end of a shape in order to produce some of the nice smooth curve that we can stitch together and turn into something later.

OK. What's the next thing I want to show you? OK. So I have gone through the main things I wanted to go through about the kind of language we use in surfacing, some of the theories to think about using overbuilt trim surfaces, theoretical sharp edges. And now I just want to step through some models I've created to show you some examples of the kind of things you might use surface modeling for.

Now, this is something you think, that's a fairly easy, straightforward model. How complicated could it get? But actually, that was more complicated than you might think. So just to walk you through some of these things, So the first thing I started with was a sketch. So maybe a designer came in, gave me a sketch of what he was looking for, and I just put these in the background to work from.

What I like to do with nearly all my models, whether done prismatic or with surfacing, is start with a layout sketch. OK. So anything that I want to define, any dimensions I want to define, any points, any datums I want to define, are put in my layout sketch. And then later on I'll refer any sketches to this one. So I'm not building lots of linked sketches, one sketch linked to another, linked to another, linked to another. And if I pick one in the middle, it deletes it, it breaks my sequence.

So I just put one at the top and point them over to that same one. And I can delete any sketch I want later on, and it's not going to affect any others. So I always start with the layout sketch. Then I put some sketches in to represent the bowl.

And this is the curve with the bowl there. Then I started working on-- using my intersection geometry is turn these on. So I've intersected the difference between this curve and these curves to get my theoretical sharp edge, that double curved edge that goes away around the bowl.

And next thing I've done here is create a surface going this way with another curve on it. So just like the end cap, that's all the geometry I've built to get a nice, smooth, continuous bowl shape. And [INAUDIBLE] there's a loft there. So I have to tun all these off to create the bowl at the end of the spoon there.

Now, here's a tricky thing. In order to get the bowl to meet with the handle-- because I couldn't use that bowl as a rail, because the rails need to fit where the loft is-- I then have to copy that face and split it so I can make this edge. So I wanted to use that as a rail in my loft. And I needed just that edge, not the whole bowl. So I had to split it like this.

So then I could create the sketch for the handle with a profile, with a loft. And I was able to loft it down to the edge and then get curvature continuity between the handle and the bowl. So again, this is more complicated than you might think. It's just a spoon, for goodness sake, but a lot of work-- and having to split these faces up in order to get that edge for the curvature continuity.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: Sorry. What was the question?

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: Yes. That's exactly it. I actually then went in and trimmed that surface back-- sorry, sorry-- there to create the shape of the neck there. So I trimmed this main surface back. And then to merge the surfaces back together again, it was a stitch. So it was a stitch of those two surfaces.

But this is one that taught me something about direction. So when I started this off, you'll see that I created the bowl lofting in this direction. So lofting in the same direction I was going to create the handle from to begin with, it seemed more intuitive to me to sweep the bowl that way. But what it did was it created a singularity where I wanted to match in my handle. OK

So this tip of keeping your lofts going in the same direction-- now, I do have singularities here in this bowl. But they're here at the side of the bowl we're I'm not building anything else. OK? So it's fine to have them there because I'm not worried.

And then because my stitch works successfully, I could then thicken up all of these to create my spoon shape. And again, where I didn't have the continuity right there, what I ended up with was a break on the back of here because I couldn't get continuity. So just like my curvature cones, had a little break in the back.

And on we go. Good. I'll just show you maybe one or two more, and then we'll have some questions at the end. So this is a kitchen knife. This is actually a kitchen knife. My mom bought for me when I left home. So it's something I'm very familiar with. I've had it for years and years and years.

OK. So we have here primary surfaces. We have an end cap. And then we have some more primary surfaces down at this end. And finally, we have our blend. So this is actually our second resurfaced blending from this one to this one.

So just to roll it back and show you some the highlights from this one, so again, I started with an overall sketch. So I could use that to define some datums. I actually started here with a sketch of the blade, which I'm going to work out first.

I thought that would be the easy bit. But I'll show you, actually, that was a bit tricky as well. And then this one makes use of-- sorry. This one makes use of interpolation splines because it's a closed loop. So rather than CV splines, this is when I use interpolation splines for to get that closed loop all the way through and maintain that G2 continuity the whole way around the loop.

So we have a bit of work to define some of the shapes of the handle. So we could loft. And then begin lofting through those primary surfaces. So I have a primary surface, creating some more 3D sketches using 2D and on the intersection curve between that 2D sketch and the curve I've put in there. So we've created our next surface patch. And they're capping off.

So capping off the two ends, this end is sharp because we want to start increase there. This end's smooth. And that's one place where I use that technique of lofting from this age, lofting across the top and back down again. So did say I wanted to show you the blade as well.

So I thought the blade would be prismatic. I thought it would be pretty straightforward to do because it's just triangular. So I thought I'd just be able to extrude that. Extrude that in two directions, so extrude one profile, extrude the other profile. I thought it would be done.

But I didn't realize was that this curve here is actually got double curvature. So it's flat across the top, curves in this direction, and curves in this direction. So even though that looks like a fairly prismatic shape-- actually, I used the same surfacing tools to create those two surfaces, trim them back to the blade line, and then stitch the whole lot together. So I just want to show you that as an example of the fact you can use these tools to create complex geometry even if you're not doing curvature continuity, even if it actually [INAUDIBLE], It's kind of sharp like this.

OK. We have got 10 more minutes left. So I will open this one up as an example of what you can achieve. But we'll stop and have a Q&A instead. OK. Do we have any questions? Yes, sir.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: So Stitch, the edges need to meet, and it will stitch together at the seam where the edges meet. Sculpt will allow you to have overlapping edges. So I could have capped that dome off with a work plane and it would have worked. It would have found the enclosed volume. Is that helpful?

AUDIENCE: Yeah.

PAUL MUNFORD: OK. Anybody else? Yes, sir.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: OK. So the question was, could I show the curvature analysis on the lofted end cap? Yes. You do get singularities at these corners. But in that case, again, I was kind of happy because you can see I've got nice curvature going in there. And I was pretty happy when I did my analysis that that would look OK.

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: So would I recommend that as the default de facto precision capping. Well, I'd say the least work is the boundary patch. But you got the least control. So the loft to guardrails has the most control. The loft to point has some control because you control the height of it. B you don't really control the tangency.

So the loft with guardrails has the most control. So it really just depends on how much control you want, how precise you need that to be. So if you're doing something where you just felt you were going to create a shape, 3D print it, and see how it felt, you might not be that worried. But if you had a high degree of tolerance where you want to just know that curve was correct, that's why I'd recommend you use the last version. Any other questions? Sir?

AUDIENCE: [INAUDIBLE]

PAUL MUNFORD: So the question was are there any sheet material tools in Inventor you can use for developing surfaces? And the answer is yes. There is a sheet metal plug-in that comes default out of the box. But sheet metal will only unfold surfaces that are prismatic.

So if it's based on a cylindrical curve or a cone shaped curve, it will develop them because it's developed them for being rolled or flat or brake pressed. But it won't do-- for example, if you were to do fabric, it won't develop stuff. For example, if you're doing fabric on a chair or cushion-- yeah, so you've got curvature in two directions at the same time. It won't develop those. There is a plug-in for Fusion 360 that somebody's developed that would do that sort of thing. Any other questions?

AUDIENCE: Are these examples covered in the book?

PAUL MUNFORD: These aren't covered in the book because the book's generic. The book covers everything Inventor. The book's this thick. It was written for the last eight years by a guy called Curtis Waguespack. He's done some amazing work. It's really good stuff. But the surfacing one was a video tutorial.

I've put most of what we've covered in the handout. But it's quite brief. So one of the things is when I present this stuff, I've got an hour and a half. So I wouldn't tell you it's possible. But I guarantee you won't learn it all in this one session. But if you know it's possible, you'll find your way back there.

Any other questions? So let me just finish up then. So this is a call to action from me. So I now work for an authoress reseller. But I've only been working there for the past three months. Actually, I've been presenting it a year, for the last four years. The first day I went to was 2005.

And for me, it's always been about-- OK. That's what I meant to say. I'm not up here because I'm the best qualified. That's certainly not the case. I'm not up here because I'm the most intelligent guy in the room. And that's not the case, definitely not either.

I'm up here because Autodesk University said, have you got something to share with people? I thought, you know what? I have. I got something I want to share.

For me, that's the power of Autodesk University. There is probably no problem you've had with your CAD system. There isn't somebody else in this room that's had the same problem and probably has an answer for you.

So Autodesk University is all about networking. It's all about making those relationships with people that will help you when you get back to the office. There will be people you can ask. You can pick their brains.

So my challenge for you is that I think I want to see one of you guys up here. So I think you've all got something to share. You've all got a story to tell.

So next year, you're turn. Come up and present. Thanks you very much. Please don't forget to fill our your evaluation forms.

[APPLAUSE]