JOHNSON SHIUE: Hello, everybody. My name is Johnson Shiue. I'm a QA engineer at Autodesk for almost 20 years. Never ending but chasing endeavor. And then, today's topic-- welcome to my class. Today's topic is reverse engineering modeling techniques. I'm not teaching you guys how to steal people's data. That's wrong. That's plain wrong. But I'm talking about if you have some partial data to work with, how do you make sense out of it in different formats.

And a little bit about myself. I grew up in Taiwan, and I came to US in 95, and then got a degree in mechanical engineering from Purdue University. Any boilermakers here? That's OK. They're not doing that well in football anyways. Yeah, so-- and then I joined Autodesk soon after school and then ever since.

So I learn a lot from you guys. You know, I know a lot of you guys, you guys are power users. And to be honest, I am a little bit overwhelmed right now, because I feel like I'm a college student teaching a course to a bunch of professors. So if I'm not making sense to you, just bear with me.

So today's summary, the class structure is like this. We first talk about 2D to 3D. So you have some partial 2D data, and then how you leverage to 3D. And then we will jump into Mesh. And Mesh, traditionally, in Inventor 2016 and earlier, Inventor couldn't do much about Mesh. It's just a graphical object.

There's no sense. There's no knowledge. Just it's there. It looks pretty. But starting from 2017, [INAUDIBLE] has the ability to recognize [INAUDIBLE] even fit some faces, so that makes the workflow a little bit more interesting. And the third one is 3D Breps, so 2- 3D, so it's about you get some data in neutral format, like [INAUDIBLE] or SAT or [? IJES ?] or even SOLIDWORKS. And then how do you find out the design? Then how you recreate data? And then each section will follow by some tips and tricks. And then at the end, we'll do a Q&A.

So let's jump into 2D to 3D. So in Inventor actually there are quite a few 2D to 3D workflows false as many of you have used it and known for a long time. So basically you could import points from the [? export ?] table. So somehow, let's say if you have a table coordinates and x, y and z. And you put it very nicely in a table, and you can import it. And then I will show you in a demo.

And you can import 2D and 3D wires. And then they can be 2D sketched, 3D sketched. And then you can use-- there's an app tool called 2D to 3D tool. Have anybody use 2D to 3D tool? Yeah. And then we have this [INAUDIBLE] starting from 2016, I believe. And I will show you in an example.

Then the paper 2D drawing. Still, I think we still heard from our user worldwide, from time to time, they have some scan, maybe PDF, 2D drawings. And they still want to leverage those drawings. And then Inventor itself couldn't handle that type of data easily. But AutoCAD has a very nice tool called Raster Design. So if you want more information, you can go to Autodesk.com and then find out more information about that.

So what it does is basically it's kind of like a mapping tool. So because a lot of-- traditionally a lot of maps are in paper form or in scan form. And this Raster Design help you kind of track those lines and make sense out of it. And you can convert to poly lines or curve. And then you can go back to import wire and then make sense out of it. So I will show you each of the workflow in the Mentor.

OK so here is a typical example of import points to a 2D sketch. And the workflow is like this. So you got to start a part. And then if you are importing 2D point, you have to create a sketch. And then the command is called point here. And you select the Excel table containing the coordinate system.

In the options, actually you got some options, you can just import points as is. You know, each point, they are unconstrained. Or you can connect points as lines, you know. So basically, the first point will connect to the second one, second one connect to the third one. Bop, bop, bop. Or you can create a [? spline. ?] So I don't think you could do a lot with this type of technique, but it's an option, you know. If you have partial data, at least you can bring the points into Mentor and then do something.

And then 3D point, it's the same thing. So essentially you just need to create a 3D sketch. And then you go-- the same command is available there. And the option-- the options the same. And feel free to interrupt me if you have any question or you want to understand something more.

And then for wires and I mean this workflow has been there for a long, long time. And then sometimes you just got partial data, like this. And then you import as 3D sketch. But you could create some boundary patch, and then create some geometry out of it. And then you can connect it. like this one is kind of a very simple tube, and then you can find a center points easily, and then you can rebuild a path.

Like this one is much more complicated. You know, you got essentially a 3D model in wireframe form. And then you can import it, a 3D wire. And it's very hard to make sense out of it, to be honest, unless you know the design well already. But you still could. You could use a binary patch to build out patches. And then make sense out of it.

And for the 2D to 3D tool I was just mentioning. The workflow is like this. You have to import DWG file in here. So let me walk through the workflow real quick. So what you need to do is simply just open a DW file. And in here, in option, you have to select import.

So if you just keep it as default open, it's going to open itself in Inventor, just like this. So it's like a viewer, you can view DWG, but you cannot do much about it. So but now we are doing the import workflow. So let's just open it, and import it.

And you will go through a couple of dialogue options. You know, I think some of you are very familiar with this. And if you deal with AutoCAD data a lot. And then it depends on the size of the drawing, you know. It can take awhile to do the translation. But in this case, it is pretty simple.

And then you've got a bunch of sketches. Let me zoom. OK. Oh, sorry. I don't know what's happening. OK, sorry. So what you're seeing is-- it's the DWG geometry imported to Inventor. but they are not associative, meaning if you make a change in your DWG file, this won't update. You know, they stay the same.

And then you see on the left-hand side in the browser, each layer is being mapped as a individual sketch. So that's the workflow. And then what 2D and 3D tools help is here. There is some usability issue with the tool actually, but-- so essentially, you can only handle one unconsumed sketch at a time.

So like right now, if you have more than one, and the panel here, it's now lit up. It's the 2D and 3D-- because he couldn't handle it. So let's say I delete those. And I guess I have to save it. Now there's a little bit usability. Because it's assuming you always bring in one sketch at a time. So now it lit up.

So what-- there's some nice things about the tool, which kind of explain why it's kind of popular in app store. So what it does is, if you click on the base view command, it show you like a box.

And then you can select the orientation where you want to place that part of your sketch you are about to select.

So let's say-- and that's the so-called base view. And then you can start picking geometry. You can window select too, I think. And then immediately it kind of just transport that sketch to the front face, the face you select it. And then can keep doing it for projected view.

But the problem with this tool is, first, it is not associative, right? So you have to bring in the data. And there are some usability issue. And second of all, you can only do orthogonal views. So it's a base, top side, you know. That's sort of thing.

Like this case, you have an auxiliary view, you know. So it's not projected orthogonally. And you kind of run out of option in this case. But for simple thing, it does really well. And is something you guys can consider.

So and then I am going to introduce the DWG underlay workflow, which is starting to-- we starting to support in 2016. So you also need to start with an empty part. And then, you don't need to create a sketch, because the AutoCAD geometry it's not treated as a Sketch. It's kind of like a transparency in your model space.

So click on the import. And first you have to assign which plane you want to place that transparency to. And then you give an origin. And immediately you will bring in the data. So, and actually, based on our internal testing, this is really, really, fast because it doesn't involve full translation of the geometry.

So if you have tons of drawings, tons of sheet, this is a good tool, a good workflow to try out. But you will ask, then what? I mean, what does this mean? And the nice thing is, you could turn off some layer you don't want.

So for example, you bring in six, seven layers. But you only care about that geometry. And then you can easily manage that. And also you can crop. So, for example, you only care about-- you only want to work on this area. And crop it for you. So it's-- yeah, it's a lot better than the 2D to 3D workflow I show you.

But it depends. And another thing I forgot to mention, this workflow is fully associative. So if somehow the DW-- you make some change to a DWG file, it will update. So the part will know, hey, something has changed. And then you have to update.

AUDIENCE: [INAUDIBLE]

JOHNSON SHIUE: Not yet. So, yeah, so you will still need to build a sketch. And then you have to use a command called project DWG file-- project DWG underlay. And then you get projected. OK, and then they got projected, and then now you can use it. So that that's the underlying workflow.

So I mean, I just quickly walk through these workflows. And one thing to keep in mind is, I think, is probably kind of cliche. You need to understand your unit. You know, so it's inch, or millimeter, or centimeter. And then you need to understand your scale. Because DWG files, the scale is a little ambiguous. It's not true geometry scale.

So you could have a line, although it's not two-dimension, it's still dimension certain way. So you have to understand that in order to fully leverage in these workflows.

So we ran through these workflows real quick already. And then I covered the deficiency in [INAUDIBLE]. And then, yeah, the tips and tricks. So just like I said, you have to understand the unit, the scale. And also, you like to work on the base view first. And yeah, I mean, the first one is another no-brainer. You have to understand the data. You know, where do you start with? Otherwise you get into a situation that you don't know how this part should be positioned or oriented.

And then the last one is kind of interesting. Because Inventory's drawing is fully associative. So when you rebuild the 3D geometry, you can easily create a 2D drawing for verification purpose. So you can simply create a 3D view, and then you can match. See whether or not, hey, you missed something. You know, that's a pretty interesting tool. I mean interesting workflow.

So you don't have to kind of guess, hey, did I miss something here? Did I forget about certain feature? So any questions so far?

Now we're jumping to the Raster Mesh Data to 3D. So like I said, in 2016, Inventor 2016 and earlier, there's not much Inventor can do with this type of data. Because Inventor didn't have the capability to recognize the points and edges on those meshes. But now in 2017, we did enable that behavior. So you can do a lot more than before.

And the relevant workflows are here, so you can attach 2D images. So for example, you have some pictures of the parts you try to model. So you can, I mean, it's a little bit similar to what I mention. Raster design thing.

And then you can have the so-called Brep-convert mesh. So meaning the source mesh data is [? cap, ?] is a cap date. So you have faces, have bodies. But somebody just exported as a mesh. And that type of data is very nice, because it's more uniform, and the data is always complete. And it's much easier to handle.

And lastly, you got scanned data. So you got laser scanner, and then you do point cloud. But you can easily convert to mesh. And sometimes depends on the accuracy, You have, you could have missing data. And then, but because of 2017 enhanced ability, you could do a little bit more.

And I want to touch on the mesh model here. Look at this-- the model on the corner. Actually it looks round. But actually if you look closely, it's not round. You know, it's a mesh. It's always kind of either triangular or quadrilateral. So it is-- so there's no circle in mesh. I mean, that just FYI.

And also, what does raster, what does Brep, what does a vector mean? So what raster data is, is the data's quality or data's definition varies with this resolution. So it doesn't have a sense of a line or a curve or a body. So essentially if a line is pretty wide, you can say it's a phase or you have different meaning. But for vector data, like the CAD data, there's no ambiguity there. You know, a line's a lines. it's from a to b, and it doesn't matter how you represent it. So internally the data is bad.

OK, so this is a workflow. It's available In Inventor for a long, long time too. So essentially you can position some images on 2D-- on work plans. And then you can sort of model around it.

And then here's the mesh workflow. And I'll run through some demos real quick. And this is just highlight that the mesh data doesn't have sense of scale and unit. So you will, as the person deal with the data, you will have to determine. So what type of unit you are going to apply.

So like this case, you are importing something. The bounding box data tell you roughly the size. And then you have to attach a unit, and make sure that makes-- make sure it's consistent with your understanding. So that's-- now rung through those workflow real quick.

So here is the part. We saw three images. They are orthogonal to each other. And then, you could do a little bit. And also, if you have that Raster Design tool, AutoCAD Raster Design, you can convert some of the edges, the lines, to curves.

And here's an example of the so-called Brep convert mesh. Yeah, it's very nice, and it's very uniform. And let me turn on the edges. And you can see, yesh, those are mesh edges.

And then what can you do with it? Let me show you real quick. So now in Inventor, you can fit mesh face command. So what this does, is it try to detect those meshes, and then try to figure out-- are the three points on the same plane? Depends on the tolerance you decide, they decide, so-- but you probably don't want a very tight tolerance because it depends on your mesh quality actually.

So like this type, you can assign very high tolerance, very tight. Yeah, but for others, I will show you real quick. It may not be suitable. So the default type output is just-- let Inventor figure out what kind of face this means. But like this one, it's [INAUDIBLE] it's a plane. So it immediately recognize, and then it give you a plain surface.

And then you can also create a sketch on plane or printed mesh. And then you can start drawing data. And then you can count string. Or you can even project those points or circles.

So Inventor recognize it, although it's a mesh, but it can make a little sense out of it. But what if you-- I mean, this is-- the mesh I just show you is really nice, uniform. But what if you got date like this? It's a [INAUDIBLE] data. And there's a lot of imperfection. And what can you do with it?

So now I'm switching to a tool. It's called Autodesk Mesh Mixer. It's a free tool, available at Autodesk.com. Of And feel free to download to try. Very cool, very powerful tool. Initially it was positioned to-- for like a consumer product application. So if you want to Revit with a doggy hat, and you can a little bit kind of mixing, you know. And then print it. You know, yeah, it's kind of fun.

But I found out, actually it has some industrial application and potential too. So I will run through a few things. And you will probably understand what I'm talking about. So you will see, there's just a lot of-- I mean, I don't know how to feel about this data. I mean, there's not much you can do. I mean, if you bring this exact as is in Inventor, you can just view it.

But the Mesh Mixer can kind of help, kind of eliminate some imperfection. First, you could kind of-- you could create the so-called face group. So what it does is try to analyze the edged deviation, the angular deviation, from the neighbors. And then it give-- assign colors.

So in this case, you saw those very colorful, small pieces. Those are the imperfections, and then they go in and out like that. They kind of flip-flop quite a bit. And it gives you a sense of your data quality and then what you need to do to make it better.

So you can first assign a group. And then you can separate [INAUDIBLE]. So this tool is really cool. I mean, this command is really cool that it help you categorize non-continuous meshes. So what it does is it just separate each individual mesh lumps. And then they sort by-- you will see this object browser.

The tool essentially sort by size. So if you don't care about the small faces, you could simply delete them. And then you just keep those big one you care about. And then what do you do with it? OK, yeah, it's kind of-- we're sort of getting there. But it's still- the imperfections is still there. They're are holes. And yes.

And then you can try this. You can sculpting. So there are some tools, allowing you to kind of smoothen, smoothen the data. So it give you a lot of choices, and you can explore as you wish. One thing I find out is it really depends on your model size. You have to kind of trial and error. And then change the brush size. And then-- and different refinement options, you know.

And let me show you real quick what these all mean. And you see, I-- anywhere my brush goes, it is kind of doing-- it just smoothens out those imperfection. you can even heal the hole. But one thing you have to keep in mind. Actually is altering your mesh. You know, what it's doing is altering your mesh. Although it's making [INAUDIBLE], it's not the data you-- it's not the original raw data you got. So that's something you have to keep in mind.

But when you work with this type of data, your tolerance is already really loose. You know, you cannot ask for really-- yes, it is for reference only. So-- and I can run the face group command again, and you will see the difference.

So immediately, it smoother now. It was really shaky. I got goosebumps when I see it. now look at the data. I didn't, I mean, I didn't spend much time. But you could work on it, and then kind of make sense out of it.

And then this is a good tool for preprocssing. And then you can bring the data into Inventor, and then use the fit mesh face command or sketch and command, try to rebuild some data. Any questions so far? OK.

And then, just the process I was talking about. The mesh mixer. So you can import a mesh, and then you separate them into different shells, and you remove the unwanted shells. And then you can create a face group, and kind of find out where the imperfections are. And you use the scope command to smoothen the imperfection.

And then, it can be an [? iterative ?] process. You don't expect you do it in one shot. But at least you have some options. And then you can bring it to Inventor, and start building the geometry.

And the tips and trick about these workflow is you've got to have an expectation about your tolerance. You know, I mean, you are dealing with imperfect data, so the tolerance couldn't be too tight. And then again, the bread and butter, the unit, is also the same thing. It's even more important. Because the Raster data doesn't have a unit. So I want to make sure you understand the unit.

And the simplify meshes, you know, you want to recognize those surfaces, and then you want to identify features. And the last one, it's also kind of like the technique I mentioned in the 2D to 3D workflow. You could actually do a kind of round trip validation. So when you build, you build your mesh data-- you build geometry from your mesh data. You can also output as a mesh. And then you can bring in to a mesh mix or do some comparison, you know. Just say how far you are, how close you are.

Now we are switching gear to the 3D Brep to a 3D feature workflows. So this technique, yeah, it's also very old. I mean, as old as Inventor or any 3D feature [INAUDIBLE] system. So on the right-hand side is just an engineer part. You got it somewhere, and that you want to rebuild. Maybe your vendor didn't give you a complete data. You need to rebuild it from scratch. Or maybe your employee left. You don't know what he did. And then you have to redo it.

And the key about rebuilding these type of data is, you have to recognize the design [? intent, ?] and apply the right feature. So this is a very simple question. I mean, in Inventor, there are so many commands to do cylinder. And cylinder, it's a pretty simple mathematic definition. You've got a radius, you've got height. And that is very simple.

But in Inventor, there are tons command capable of creating cylinder. But which one to choose? It's, I mean, as a designer, you have to understand your design intent. And also, you have to understand how this feature is going to be leveraged in the future.

Like, OK, so if you have a hole. You want to create a whole. So it's more like a extruding cut. But maybe you want it-- you want to be more specific. You want it according-- you want the whole to obey certain standard, then you want a standardized whole.

But what if you just want a partial whole, meaning this whole, the angle can change. Then you have to use revolve, you know. So what I'm trying to say is rebuilding the data, it's not about just mimicking the geometry. You have to capture the design intent, and also you have to understand how this feature will help you. What kind of engineering operation you want to do with these feature down the road.

And I'll show you a few examples of 3D Brep to 3D geometry. So how do we start? So basically it's about simplification. You have to simplify the data. You know the prior example I show? I mean, there are so many features. I mean, where do you start? It's very overwhelming.

Do you start from those pockets? Or do you start from the holes? I mean, so you have to simplify in order to keep working. And also you want to divide into a region area. And also you want to find out if the area, or the piece of geometry, is analytical. If it's a plane, don't use [INAUDIBLE]. If it's a circle, don't use other type of geometry. You know, keep it simple.

Because a lot of time, I have seen a lot of models from our customers. They literally interpret the data. But actually, to a certain degree, they misinterpret the design intent. And then make the model unnecessary complicated. And then, recognize the design intent like I stressed before. And then you can rebuild the geometry.

So now let's do some demo. So here's the part I show on first slide too. And then we do have an app tool, or feature recognition. So that's free for subscription customer. How many of you are subscribers?

Yes, OK, so this is a tool, a small add-in. And then you can enable in your part environment. So every time you open up a part with no features, just the base solid, it will ask you whether or not you want to run feature recognition.

If you want, and then you can automatically recognize all the feature, or you can do one-by-one. And it's a little bit trial and error though. Because from Inventor perspective, yeah, Inventor see a lot of cylinders and planes, you know. So as a designer, you will have to tell

Inventor what you really want, and then Inventor can help you do that.

So that-- and what I mean by simplify the geometry, it's like, there are so many [? fillets ?] here. And let me turn on the edge view. And you make it really hard to work with this type of data. So a command called delete face is really handy. So this command has been in Inventor for awhile. And what it does is, it can help you delete the geometry, and also re-intersect the underlying geometry.

So let me show you real quick. So you have to turn around this [INAUDIBLE] command. And then you just remove that [? fillet, ?] the detail. And you can keep doing it. But if there is just too many, and then in this particular case, actually the key point is to get the base profile. And also those wholes and those cutouts. In the pocket, actually, it's lightweight feature. So they are kind of secondary importance.

And what you can do is actually like this. You simply create a work plan. And that's a lot simpler, so you can focus on those wholes, and then recreate the geometry much easier.

And then if you started to work on those lightweight areas, and you can delete a scope, and you can restore it. Here's another simple example. How to find a path. This is really simple. I think everybody can do this.

So you-- it's a sweep. So essentially you just need to create axes. And then those intersection-- I mean, when you can trace those work axes and recreate the path.

But what about this? Is a spline tube? What do you do? There's no work geometry to work with. And how do you find out the path? Here's the command. It's pretty handy. It's called rule service.

And usually in a spline geometry, you would have a seam. You would have a seam edge. But you have to turn on the edge display in shaded mode. Otherwise you don't see them. But they're there.

So you can create a rule surface. And rule surface, it's basically kind of like a sweep or extruded surface along a certain direction. In this case, you want it to go in there. And then essentially you just need the half of the tube diameter, which is the radius. Let me show you in wireframe mode, it's easier.

And the rule surface actually just give you the path. So the other side of the edge give you the path. So this tools is very handy in the sense that-- because Inventor doesn't have the capability of offsetting curves. But the tool can help you achieve that.

Then here is a more complicated model. And how do we find-- how do we rebuild geometry like this? I am not going to rebuild the whole thing here, because it will take a while. But I will point out some key points.

So this is a pump component. And so there are blades around it. And then there's a hole. And then there's a bottom. But what you can tell is-- actually the most difficult part here is how you recreate this blade. Not a pattern, because once you get one, you get all.

And this one is not that hard either. Because this one is probably just an extrusion, so we can measure. So it tell you it's 1.5 millimeter. That's easy. But this one is tricky, but how do we do that? So basically we just apply the same rule. You have to simplify it. This is just way too complicated. There's not much you can do with the data directly.

And you start deleting face, restoring the geometry. You know, you want to remove those small [? filett ?] faces. So now it's a lot better. You can make more sense out of this. But then how do you create the sweep? I mean, still it looks like sweep, it look like [INAUDIBLE], you know. It's hard to tell.

Now actually, what you can do is, you want to find out how these are created. You know, so this is a [? planer ?] face. The way to detect analytical geometry is quite interesting. You can just try to create a work plane. If its plane, it's a plant. If it's not, it's not a plane. So this one is a plane.

And what about these? I mean they look like a section of some sort. But we're not sure if they are on the same plane. So what we can do is we can just create one, and see if the other one, see it the other point is on the same plane. And then I can do a measurement with the point. So it's zero.

So that mean that particular section is on a plane. And you can keep doing that, and then you will find out this one is also a plane. But still, it doesn't resolve the mystery. I mean, how do you find out? You got section, but you don't have the path.

So you can project those edges to those planes. So you kind of get a better sense of what you're dealing with. And here's the key. Now you've got-- you got the edges. And then they, I mean this is by experience though. I don't think I can tell you a technique to find out, oh, these lines can be projected to a plane.

But let me-- so let's just look at the sketch real quick Let me hide those work planes so they don't clutter my view. And then let me turn on the wire frame. And what I'm looking at is, OK, so how this sweep was created. So it kind of turn, make a turn. It goes straight, and then it kind of make a turn.

And they just, I mean, I don't know about you, just looked kind of parallel to me. It's just two parallel curve go that way. And then what I did was, I just, then I project those edges to a sketch.

So the mystery is solved. So essentially the designer, basically he create this parallel path. This parallel path, and then actually, I think this, the mid path is the one he used to sweep it. And then he basically just project back to the base. Is actually it's [? torus ?] to create that 3D sketch path.

So this example show you, actually you could kind of find out the original design intent by checking, by understanding the geometry. By checking the analytical-- whether or not the geometry is analytical.

And the tips and trick about this is, yeah, I mean, I show you some tricks about finding the sweep path. And then you can use delete face to restore the geometry. And you can use the rule surface to create more geometry like sweep path and other surfaces.

And the key is, yeah, you want to keep it simple. Because a lot of time you got the geometry, it's just overly complicated. And if you go build on the complicated data, you just make things more complicated and out of control.

And then they are, I mean, I show you the little tool subscription users have access to. It's the feature recognition, and that can help a little bit. And in 2017 or 16, we have this curve-on surface. So you can draw a curve directly on any curvy surface. So that can help also to rebuild some data.

Now we are entering the last portion of the demo, is the sheet metal and thin part. So the sheet metal part is basically you either got a folded part or you got a flat pattern. But when you have a flat pattern, you probably have it. You know, you just need to fold it back where you want.

But the thing is, like this, if you have a flat pattern like this, and then you have taps around it, what you want to do is, you don't want [? to touch ?] those taps. You want to create those main shape first, and make sure you can fold them properly. And then you can rebuild those taps much easier.

Basically postpone that process later. Just don't do those details. So I keep telling-- yeah, keep it simple. You know, focus on the main shape. Don't worry about the detail. Because parametric slide modeling, the beauty is you can always go back. You can always add. And it's history based. And everything is in the history. And I will show you a demo real quick.

So like this part. This part came from [INAUDIBLE]. So to build this, you can simply do that. You know, you-- I always want to create those work plans defining the boundary. So it's either inside or outside. So basically, you want to identify the A side faces, and then you want to get a clear understanding of your boundary.

So what you-- how big you need to deal with. Something like that. And then you just keep working on it, keep creating those geometry. And then you can create-- starting creating faces, flanges like that.

But the other option you can choose is basically just use the reg edit. So for example, if you want to change this flange angle-- and you can use the reg edit command. And then you can modify the angle without recreating that geometry. So that's a tool you can consider using. It's called the reg edit.

And what if you have a thin wall part like this? And how do you recreate it? So first, you have to convert it to sheet metal part. And there's a kind of implicit command here. Inventor help you find out the thickness. So you just select the face. It automatically populate the thickness for you.

It's a thin-wall part, but you have to watch out on-- I mean, turn on the edges display. Because the bottom is a planer face. Apparently it was done by a shell. And actually, this may not be desirable for sheet metal part. Because the angle is not straight, not perpendicular. So I can do small measurement here.

So it's 95. something. It's not 90 degree. But how do you deal with it? Because there's an interesting technique you can try. It's kind of double thickening. So essentially, you just thicken in from outside and also from inside.

I think I select all. And then you just need to do that one more time outside Oh, sorry. Yeah, so that will ensure the detail face, the side face is 90 degree. And we can check real quick. Yeah, it's 90 degrees now.

But still, how do we deal with this type of solid? So you will have to rip the corners. And what I like to do usually is make the corner really big. And what this buy me is, you wouldn't have colliding geometry down the road. Because you can change the-- this is a parametric model, so you can change the gap later, you know.

But to do it first, you want to do something like this. You know, just try to create a really big gap. And then you can start creating [? bins ?] one by one. So this way, it doesn't like the bend, and then we can even rip it. We can use the rip command to do that.

Make a bigger rip. Something like that. And then you can clean up those detail face later. Yeah. So the mantra here is you don't want to worry about the detail too early. And then you focus on the main shape. And then you want to-- you want to use thicken to ensure that the detail face is at a right angle.

And then you can create a multi-solid in sheet metal. I'm not sure if you guys know about that. Starting from 2016, sheet metal support multi-solid body. And the conclusion of today's class is the fidelity of your reverse engineering data, it really depends on your source data. So if your source data is really, really poor, there's not much you can do.

And then I did introduce a lot of, kind of like round trip verification process. But you want to watch out precision lost during the process. So you want to make sure you understand how much you lose, and then you are not deviating too much. Inventor does offer some reverse engineering capability.

And then the last point is, because you know these techniques-- actually, you might want to understand how your data will be consumed by others. So that can help you kind of keep your data integrity.

So any questions? So the last one before we go. These are credit. I think, this presentation-- without you guys, without Inventor users worldwide, there is no presentation today. And then I just want to thank you guys, thank all the Inventor users worldwide. And also, the Inventor team. AutoCAD team and Autodesk sheet manager team. And without them, we don't have a presentation, and we don't have the capability. So we have the team members here. Let's give them a round of applause.

Thank you, and then I'll be at--