MATTEO CROCETTI: My name is Matteo Crocetti. And here's my mate Bertrand Masure. Just a few word about my bio, before starting the lesson. I don't want to bore you about my 17 years experience in CAD/CAM software.

But the first time, I would like to say you is that I'm a Fusion 360 [INAUDIBLE]. And this could be enough, for the moment, for me. Bertrand, if you want to introduce yourself.

BERTRAND MASURE: So, good evening. So I am Bertrand Masure, a technical specialist in France, so mainly focused on data management, and product design suites, with the French partner in manufacturing. And today, I will try to help Matteo and yourself to understand our French and Italian accents. So, hopefully, you will get everything.

MATTEO CROCETTI: Merci, Bertrand.

BERTRAND MASURE: De nada.

MATTEO CROCETTI: De nada is Spanish, come on.

BERTRAND MASURE: That's the same.

MATTEO CROCETTI: Oh, the same, OK one more beer for me. OK, so the class summary-- as you already know, the class summary is centered into the Fusion workflow. We are going to discover two design workflow, one made for me, and one made for my colleague Bertrand.

And during this workflow, we are going to go over the most principal rule for playing with the freeform command as better as possible. This is the main summary I can give you.

So regarding the learning objectives, of course we will go into surf around the Fusion360 environment, touching, of course, the freeforms modeling, and the golden rule behind. And we'll take a overview also of the CAM environment, and the 3D printing environment as well.

So my first objective today is to boring you as less as possible with slide, and playing with the

software as much as possible. Regarding the workflow, just to take a look. My first workflow will be centered to this part.

This is an edge trimmer, where I will cover the modeling displaying a solution, adding more detail in the design environment, and ending my workflow into the Cam environment.

BERTRAND MASURE: And for my side, I will try to build with you [INAUDIBLE]. So the idea is to insert a mesh, and to fit the design to it. And I 3D print this part. So we will go deeper afterwards.

MATTEO CROCETTI: OK, let's start. First design workflow is my turn. As I already said, freeform engineering and manufacturing CAM. What does it mean? It mean that I will start with an environment where I will put animation into the [INAUDIBLE], into the background, starting with some freeform modeling.

And after that, I will try to engineer some detail to this plastic part, and the ending to the mold, because, of course, the plastic part we have to produce. And for producing, we need the mold and the CAM for removing the material. This is a standard process used today to [INAUDIBLE].

So let's have a look. We can start directly to the Fusion360 environment. OK, we are in. So the edge trimmer-- mainly, in this workspace I have the components that the design must contain inside.

And I already placed it into the background some scratch that the shape it will be. One more thing is missing. And I would like to add the batteries, because maybe we have the model, the socket. And only the battery is missing. As you already know, in Fusion we can refer the component simply drag and drop into the environment. And with this handle, with this triad, try to place into the correct way like this.

OK, let's hide the interface. So the interface, you already saw, is the A360 content. You know Fusion360 is managed by cloud storage .

We can start with the free form. So, mainly, the first rule inside the free form is that there is no rule. No rule because we can start with different starting shape. And the best things I look into this environment is that you can easily surf into the shape, simply moving face point and edges.

So for covering the piece I placed into the background, I start with the plane-- sorry, not here,

but in a different position. We have already create a construction plane. I put my geometry into the world, and the center.

Let me give a 300 millimeter plus 90 here, just for creating this simple plain. As for adding more detail, I will display to the face with seven face and five in the opposite direction. This is the first starting point I made for create the shape.

So double click. And the right click, and the complete plane, just for adding moving a little bit the plane for covering all my component. And now let's have an exploration of the shape.

For example, this part is symmetrical. Now I can add the role here, simply touching two different faces. And now we're able to move with the edit command the faces, and obtain something I want.

But let's have a look to covering the area where it is placed the model. I can simply select those faces. And I create an edit form again. Edit form is the best command you can use for editing the T-spline.

And now I put to the view in autographic view, because it's the best position for seeing what happen when I, for example, extrude the faces simple pushing the Add command, and dragging the arrow. So let's have a look to create the handle here. And what I would like to do is to add detail here, booking again in the auto view, and create another extrusion.

So also during the auto view, you can select some other faces and the more detail here, extruding the faces, and exploring how can be the shape. If you made the mistake, no worry. You can undo as you want.

Now, for create the connection, we use another command called Bridge. Bridge is able to connect the faces and the edges, simply defining the number of section you have to match with. And this is a rough part we obtain.

So now is not covering the images in the background. But now we are start to playing with these faces for obtain the matching to the images. Another rule you can play for obtain a very good surfaces is to try to keep the section as much as possible normal to the surface.

For doing that, double click on the section, and edit again. Edit again allow you to rotate, re-positioning the section again. So this is a work you have to do for trying to reach the images in the background.

Another rule you can play is try to move as much as possible, firstly a group of faces just for obtain more or less the shape you want, like here and here. After moving a group, move your attention to the single faces for adjust, here, are. Only at the end of your editing, move the edges and the point-- for example, here.

So this could be nice, for example, for the moment. And you can play around, whatever you want. For example, here we can select the side faces, and putting them into the top view, restrict the area here-- very simple rule, very good surface.

So, again, here, now we have to cover the lower part. And for doing that, I reuse the edit command. So I prefer to preselect the entity I want to edit. And after that, I come to the edit form.

Now I will add an extrusion, just because we can extrude the faces, and also edges. Pushing and holding the control, we can extrude the edges into the bottom. For doing that, again, auto view-- OK, could be enough.

Again, multiple selection, because I have to move this one here. And only at the end, move the point for create and covering in a very precise way the shape. Let's close the shape now.

Four closing this area, I'm going to use another time the Bridge command. Bridge command can bridge, as I already said, faces and edges. Now I want to bridge double click on these edges to another double click here.

And, again, I want to connect these two edges with the three faces. This is the result, very quickly. So the only part is missing now is to covering the batteries here.

So another rule you can play for check the quality of your display, or if you have freeform environment, is to switch sometime into the gauge visualization. For doing that, you can select the command into the toolbar, or simply push add plus one, or add plus two. That is able to you to see the gauge here.

I prefer add plus one. The gauge is the box that controlling the surfaces. So try to keep as much as possible smooth. For example, here I have a convex point that would be reflected to the surface again. For moving that, you can simple edit, as we already saw in the surface editing.

So here, I can select the edges, move into the autoview, and, for example, keep it a little bit

more smooth for keeping the curvature positive here. You can also switch for a see what happened into the surface. Again, if you want to modify the roundness in this area, you can select these faces as move a little bit in this direction.

OK, so the cage is very important to see what happen into the display. So another thing, I switch again into the control cage visualization, just because I want to cover the battery here. And for doing that, I select the faces in this area, and again into the front view [INAUDIBLE] multiple here.

Push out, and create an extrusion again. So now I need to create a plane here. No worry, that is a command, flatten. Flatten is able to you to select some faces, and say, I want to be parallel, for example, to the top plane selection. OK, what happened to the display?

Another modification here-- edit form. I want to englobe the wall volume here. OK, it could be good. OK, I don't want this rear face, because I will create the patch with another command. And for erase it, I can select and pushing cancel. They will be disappear.

So moving into the tail of the shape, I can manage something-- for example, here, for managing the round. So there are many, many workflow for create some control in this area. The first I prefer is to increase the geometry.

Increase means I want to create a sharp edge. The sharp edge is a very controlled way to manage the edges, because you can pull the round here using this solid compound. The solid command are very powerful. And you can manage better the round.

If you want to stay inside the spline, you can also increase the geometry, keep the round. And if you want to add more detail, maybe I can insert one more edges here using the Insert Edge. Insert Edge means I want more detail in both way in this area. And this could be the result.

OK, let me sharp this area, and increase because I prefer to add more detail with the round. And let me, for example, delete this edge. So as you can see, when I delete the edge, the shape change. OK, so this could be a good starting point for my design.

What happen? Exit from the freeform environment-- oh, sorry. I want to show one more things. Let me into the freeform environment. Crease this edge.

OK, exit from display environment. Now I'm able to, for example, manage these sharp edge with a solid round. Now we are in the solid modeling, and all is parametric.

As you can see here, there is the history tree. And you can playback and modify whatever you want. OK, what the next?

So I already created here the patches for the top and the bottom made by some surfaces. Let me show you, like this and that, just for create an intersection in this area. And I have to extend a little bit here the intersection.

But the final result could be this one-- so a very powerful tool you can play with and obtain a very quality surface. If you apply the rule I already said, you can control everything here in curvature continuity-- so very good.

AUDIENCE: [INAUDIBLE]

MATTEO
CROCETTI: Yeah, I use this [INAUDIBLE] boundary field. I select all the faces. And Fusion360 allow me to obtain the water-tight, solid body. After that, I put the round for the plastic, because as you know, the plastic need rounds.

And I create some pin here because of the switch. And I create the shell, the shell adding the stiffness to the surface. Splitting the part, this is the result. This is the standard plastic part.

Now what I have to do is to add more detailing, because I have the component to fix it inside here. So the component needs ribs and bosses. So those are very, very simple operation you can add are very simple, solid modeling feature.

And all is related to a history tree. So you can modify everything. And this could be the final result.

So now I have to produce the part. For producing the plastic part, I need the mold. And for creating a mold, I have to play with the solid modeling command. And with some Boolean operation, I can obtain this result.

So for create-- I just create this animation for you for let you know what happen in a mold for a plastic part. And here, you can see this part will be fixed into the molding machine that will inject plastic inside the cavity, because I create core and cavity that contain the final part. And the final part, the green part, can be modified. And, of course, all the mold's current cavity will be update.

So this is a steel. For removing this area, I need the CAM. And we have inside Fusion360. So what I have to do is--

AUDIENCE: [INAUDIBLE]

MATTEO CROCETTI: OK, removing material to the block, I can apply the shrinkage. After that I-- yeah, you can also apply different shrinkage in different axis.

AUDIENCE: [INAUDIBLE]

MATTEO CROCETTI: Yeah, yeah, of course this is an overview. I have to skip some phases. But, again, it's just to show that inside Fusion, we have all the tool for complete the product creation.

And now here we are. We are in the CAM environment with some strategy, I've created basic to the final result I have to obtain. This is part of the knowledge you have.

But in any case, here we can see the yellow part is the starting box, the starting material I put inside my milling machine. And after that, I create, for example, a rough removing material by layer, and some finishing like a scallop.

The scallop is a contour finishing made with a [INAUDIBLE] tool that is touching the surfaces and removing the material. I want to show you the final simulation, just because we want to be sure that there is no arrow in the programming of the tool button. And here we are.

Maybe we can skip the first, and going faster. OK, this is the removing material by layer. So the quality is not the best one, because I need more power for create high quality simulation. But if you want, if you have time, you can do. And moving forward, this is another finishing operation for obtain the mold.

OK, next one-- I want another contour. I want to show you the parallel is better. This is a touching movement between the final surface and the spherical tool I've used for make the removing. Of course, I keep the distance between the tool larger because of graphic visualization for your.

OK, and for this first workflow, I think is enough. I can pass the play to Bertrand.

BERTRAND MASURE: OK, so let's go for the second design process. So, first of all, we will use a freeform command, playing with a sketch as a first part, and doing with the freeform modeling some web design,

[INAUDIBLE] basically. And just to finish our part, I want to insert one of the mesh [INAUDIBLE] to be a fully scan in a Autodesk [INAUDIBLE].

So I just with mesh mixer cut one of my hands. And the idea is to fit this handle to my hand. The second step, it will be more engineer part, just to split my handle in three just to fit inside a 3D printer, add some motion in it just to inspect this handle with a body created from a material, and finish our workflow with the 3D printer.

OK, so live demo, that the way it works? So just to introduce to you the final design, I will play with the animation tab. So as you can see with Matteo, you can play with the camera, export some component, annotate it, and highlight the final designs that you want to build. So here we are.

So that's a final one. You will get three part the mesh to fit it. And that's the idea that we would like to go. So to do so, let's start from scratch.

I will play with the selection sets. It's quite good tool. So you can preselect body component edges. It's really good when you want to select precisely some edges. Go back to your autographic view.

And we will remove a lot with this kind of tool. So let's hide this component like this. And we can start now with the freeform tool. So before using the freeform tool, I want to create the profile.

So let's create a rectangle first in the form view like this. Let's say tabulation 180 [INAUDIBLE]. And here we are. So with the short command [INAUDIBLE], I can say, I want to fill it now. [INAUDIBLE] and say 35.

So as an [INAUDIBLE] user, you've got two colors. So my habit is coming back quite quickly now. And you can say, I want an angle here like this-- say, 20, and 110 for this one. And here you are.

So I will add some edge like this, just for fun, and to show you the next usage of the pipe command. So to do so with Fusion360, when you are in the script environment, you don't need any reference plane. You can use the move command to move your sketch like this.

And here you are. So I could go to the form view, say I want to move this to 35 degrees, and go precisely here, for example. So that's the idea, don't need anymore [INAUDIBLE] in the

script mode.

Now you can use the pipe command. So the pipe command, it's quite similar to the sweep command if you use a circle as a [INAUDIBLE]. So the idea is to select this [INAUDIBLE] selection, going to the smooth display, saying 30 millimeter.

And it's the benefit of this feature, because if you select this kind of line, now automatically it will manage the junction. So for my design, it's not necessary. So I will go just like this.

And let's try now to prepare the design. So I will delete some face. And I want to do a symmetry.

So to do so, here you see that's my origin here. So my design is not aligned with this. So, easily, I could select my edges like this.

And I want to set my preview form here, say OK. And now I am quite precise to say, OK, let's go there, scale a bit like this. And here you are.

So now I will play a lot with the symmetry. So you can do mirror duplicate as your first step. Say, I want this body, and this primitive here.

And here you've got a gap. So you've got [INAUDIBLE] 21. So you can say 20, for example. And it will merge the two side. So it's quite interesting.

So now to differentiate a symmetric part and a non-symmetric part, you've got a green edge here. So that's the idea. So now I would like to go to my design here. I will show you before, because maybe you do not understand my strong French accent. So I want to do this kind of stuff here, just for the finger, and get the curvature here with an arch. So that's the idea.

So to do so, I will select these two edges, and say, OK, let's create a selection set. That's the idea. Now I will use my sketches that I create here like this.

And I will say, let's go to look at these sketches. Like this, I am in the autographic view, following the words of Matteo. Say, Edit Form, and pre-select my edges.

Now, you've got different coordinate space-- so workspace, view space, and local space. If you're in the autographic view, you can use the view space quite easily. And like this, you are exactly in the direction of your sketch. So that's a really good trick. And you don't click everywhere.

For preset activity, it's quite good as well. So now let's do an arch. So if you are looking to here, I need a mid edge. So to do so, let's pre-select this and say, subdivide my faces.

So I will subdivide it. You've got different option. So let's be exact and say, I just want those edges. And now I can say, OK, I'm here.

I want to say, I want an arch. Select my sketches like this. And say, one point, two point, three point. I can say now match. I love this command, by the way.

And say, OK, let's go there. Targeted edges is this one. And here you are more or less.

So just to explain you the next step, I've got a little notification here. So I'll let you read. Maybe it's better.

But you will get some issue with the mirror command. So now my part become red. Nothing really special, but just to let you know it's this kind of feature used. So its relation is the symmetry.

If I want to explain you how it works, I will go to the edit form, say, OK, let's do it on the left side, saying, I want something like this. And you see the impact of the other side. And if I go in there, it's only moving on one side, not on the other. So you isolate the symmetry.

AUDIENCE: [INAUDIBLE]

BERTRAND MASURE: Because of the match edge, I suppose. So if you want to go back on track, you can clear the symmetry like this, and use a mirror internal one, and say, one face, two face. And your green edge is now back.

OK? So now I just want to distinguish the three parts that I would like to create. So let's say I want to select here T-spline edges. Double click, say I want to insert my edge, exact one with a symmetry on both side, let's say 35.

OK, repeat once again, this one will go to five. And you can double-click here, double-click here, and double-click here, and say, let's edit the form. So I will go back to the coordinate space, the [INAUDIBLE] one, and say, I want to scale it like this. So you've got the idea.

Now that's the trick that I give you just before. Go back to-- I want to modify these six edges. You can update your selection sets. Go back to your sketches to be autographic view. And

let's say I want to edit my form, say select this. And now you can precisely use the scale and move in the two direction like this.

So it's time to insert my mesh. It's time to insert my mesh. So to insert my mesh, I will insert an OBG-- so my end left, OK, like this.

It's coming. OK, here we are. So the little trick that you might know, it's when you want to move something really precisely, do not select it. So I will go back to my Selection [INAUDIBLE].

So if you select this one and say, move, your point will be there. So just cancel it. Say, I want to move that.

Go to your form view. And now I can be really precise to where I put my [INAUDIBLE]. So I want to set my [INAUDIBLE] here.

And like this, I'm quite precise in this direction like this. If you want to have more precise view, you can use the multiple view like this, and say, OK, so that's a synchronization [INAUDIBLE] view like this.

And you can say, OK, let's move it like this. And you can say, OK, I want to move my hands like this, and maybe a bit of like this. Good? So let's have a look on here. So maybe I can move a bit my hand. Sorry.

AUDIENCE:	[INAUDIBLE]
BERTRAND MASURE:	Pardon?
AUDIENCE:	[INAUDIBLE]
BERTRAND MASURE:	It's a mesh. It's a mesh. So you can convert the mesh as a T-Spline. But you will need to--
MATTEO CROCETTI:	Code mesh.
BERTRAND MASURE:	You will need to get OBG quad. So with--

AUDIENCE:	[INAUDIBLE]
BERTRAND MASURE:	It's here. It's here. So now it's time to fit my hands to this handle. So to do so, I will select my faces [? is related ?] to the symmetry, like this, and subdivide my part. So I will subdivide my part.
	And now it's time to use a pull command. So the pull command-- basically, you will say, OK, I want to select this mesh. And now I could say I want to select all these points.
	But the point is I would like to only select the visible one. So to do so, I will say, selections [INAUDIBLE] disable. But, OK, I could select all these points here. But behind my mesh, I could not.
	So the trick is to hide my mesh. Say, welcome to [INAUDIBLE], something like this. And now it's really easy to say, OK, I have to select all this point here.
	So let's say selection like this. I'll select like this. And now you see that my handle, it's becoming to fit to my hand.
MATTEO CROCETTI:	Yeah, so the reason why, you have more detailing dividing the faces. And this allow you to push every point in a very precise way to the [INAUDIBLE].
BERTRAND MASURE:	OK, so now I've got an open surface. So I will weather tight my design with the fill hole command-- so nothing special here. And here, you've got your handle.
	So now it's time to prepare our model to print it out in 3D. So I will create a sketch. So I'll go back to my sketch to split my body in three.
	So I will go back here. Say, let's do something like this. [INAUDIBLE] to avoid any, how you call that, snap to grid. So I will Control key.
	Next, go here and here. Here we can say construction. And let's mirror one, two, three. And the mirror line is here.
	So I'm using a close contour. It's only about getting three bodies. So if I'm going to the split body tool, I can split my sketches.
	And you see, if I didn't get a close contour, I will get one, two, three. But if I use the other one, I will get one, two, three, four, five. So I just use a close contour like this. And here you are.

You've got now three body-- OK, so one, two, and three.

Now it's time to create some sub-assembly in it. So it's quite easy with Fusion360. And that's pretty cool.

So you can create some component from bodies here. Get your component. And you can create the subcomponent.

So I will create the new component from this one, and move the other one like this. So just to get a better idea of the design now, you can add some material, if it wants to coming. OK, it's coming. So one color here, one color here, and here.

So now it's time to use some animation and assemble [INAUDIBLE] component. So if I'm going to see my part here, everything is fixed, or seems to be. But the side part are not rigid to my sub assembly.

So if I want to get a rigid [INAUDIBLE], use this feature here rigid [INAUDIBLE]. Include tri-component. And now that's one single part, like this.

One of the features I love to use it's [INAUDIBLE] joint. So [INAUDIBLE] joint will allow you to select this component, say I want to regulate this one to this one. And let's put my [INAUDIBLE] to my sketches, like this in the y-axis.

And now if I'm going through my join command, I can edit it like this, and say, edit the joint command-- minimum, maximum, and there you are. So I will say minus 30 like this-- sorry for that. And here I will say 20. And let's go back to zero degrees.

So like this, I can say, and I can play with my handle. So that's the idea of this workflow.

MATTEO CROCETTI: Well done.

BERTRAND MASURE: OK, so let's show up all the components. And if you want to have some more fun, you can say, let's animate the motor like this. And you can see the end result. It

So now it's time to print. We are on time?

MATTEO CROCETTI: Yeah, we are on time.

BERTRAND MASURE: OK, so maybe I can wonder a bit the thing here. We with Fusion360, it's a really cool tool because you can use a local performance on your design, on your workstation. So if you want to setup a bit the camera, you can say, 2,000, it's OK. But let's put a lens like this and use enable re-tracing.

So with this, you can say, OK, let's have a center of focus and a default here. And here you are. So you can use quite easily and have a end result

If you don't want to use your workstation, you've got the second location. It's to cloud credit. So if you do a cloud credit, you will use your credit.

So let's say I want to get rid of this one like this. You will get an estimation of your cloud credit. And you can wonder like this. I wonder. I close this, if you want.

And I can do other stuff. So that's print in 3D this handle. So to print it in 3D, I will select my handle here.

And I will be back in Print Studio quite soon. So Print Studio is really easy to use. You've got different step.

The first step, it's layout. So in the layout, you can move your part in the three directions. What I did-- scale it, and lay flat.

You can do other stuff. So if you create and you already prepared the three STLs of your part, you can say, OK, let's import my STL that I create here-- so all ends, right, main, and left, like this. And here you are.

I can print in one shot the three part. Why not? I can say, OK, I want to move this one like this. And let's put it like this.

[INAUDIBLE] it's coming from Fusion. So no issue there at all. And you can add after a while some support. So let's put some support here.

Let's add some support here. And let's add some support here. So that's the idea of the 3D printer. So we are using bigger ones than [INAUDIBLE]. It's more useful for this part.

And here, you've got some different options. So to use different option, as a stand out you've go draft stand out and best. If you want to use your own, you can say, I want to copy it, or

duplicate the profile.

Like this, you've got the possibility to say, OK, I want in the pattern some [INAUDIBLE] or [INAUDIBLE]. So I will use [INAUDIBLE] here. And let's-- I have to redo it.

So let's redo it. And let's say preview. So now we [INAUDIBLE] 3D print and create the [INAUDIBLE] work code. And we will get soon an estimation of the volume and the time of the 3D print.

And here we are. So that's--

MATTEO CROCETTI: Yeah. So the [INAUDIBLE] inside is very important for the weight of the component, of course, and the cost of the component at the end. We can take a look inside. OK, no, don't have time. OK, what is the [INAUDIBLE].

We can use our presentation. So we are at the conclusion of this class. What to take away?

Regarding the T-Spline, regarding the freeform, there are some rule you can apply for obtaining the best shape. First of all, when you start to making the adjustment, please try to select as much as possible a group of faces for create the rough shape. After that, moving to the refinement using the edges or group of edges. And only at the end of your design, try to use the point, because the point is a very localized modification. You can do it.

When you edit, try to stay as much as possible in the autographic view because the autographic view is the best way for you for achieve the modification, and look at what happened to the model. If you move the faces in through the eyes of view, you cannot see the real result. Of course, when you move your face, when you extrude faces or edges, please take a look to the gauge, because the gauge is the object that control your shape. And try to keep it as proportional as possible.

If you want to add more detail, you can insert edges. Or you can crease the edges for the round in the solid environment. Of course, when you model into the freeform, try to pay attention to the code face.

Try to don't use the [INAUDIBLE] face because of the continuity. And, of course, try to use and move the cross control edges use as much as possible normal to the face. At the end, try to manage at the add the start point only at the end of your modeling.

So with that, I think we have covered a lot of things, not only freeform inside Fusion in this class. But our team was try to take a work picture of the instrument. Now, if you have a question we are here. We are open to discuss with you.

BERTRAND MASURE: Any questions so far about the start point or the T point?

AUDIENCE: Yeah, where did you get that hand? Did you scan that yourself?

BERTRAND MASURE: Yes, Autodesk [INAUDIBLE], we will get fully scanned.

AUDIENCE: [INAUDIBLE]

MATTEO CROCETTI: Yeah, you can have different away to obtain--

BERTRAND MASURE: I want to use my data set and build some glasses with my head, because during [INAUDIBLE] I used to play with this kind of glass. But Matteo don't want to use this extreme. So I just find a way--

MATTEO CROCETTI: France guy, oh, come on. So there are some different away for obtain this kind of scan. The simple way is to use [INAUDIBLE] or Memento, for example. Once you do [INAUDIBLE], you can use your phone, create a lot of turnaround picture important to the application. And will obtain a 3D model-- of course, not so precise.

BERTRAND MASURE: In the end, I don't think it's the best way to do this, to be honest. The best way would be to use-- I don't know the name in English, but [INAUDIBLE]. And you will get after--

MATTEO CROCETTI: Yeah, yeah, the scan.

AUDIENCE: [INAUDIBLE] uploaded [INAUDIBLE].

MATTEO CROCETTI: OK.

AUDIENCE: I'm interested in the [INAUDIBLE] tooling process myself. So I was wondering if you had that list that you just showed online somewhere, like on YouTube, that I could go and refer to later.

MATTEO CROCETTI: You mean--

AUDIENCE: Basically, building your freeform model to the CAD process of making--

MATTEO CROCETTI: Yeah, so the question is regarding the tool for better produce some plastic model inside Fusion. At the moment, there is no tool inside Fusion like this. But, of course, we have more flow, as you already know.

And maybe in the future, there will be insert inside a little small part called MDF that allow you to-- and [INAUDIBLE] the critical part inside the plastic. For example, I had some bosses that increase a lot the thickness of the plastic. And there will be a problem during the plastics. Any questions, other?

AUDIENCE: Do we get to see the final rendering of the 3D print.

MATTEO CROCETTI: OK.

BERTRAND MASURE: It's arriving.

MATTEO CROCETTI: OK, zoom out, the honeycomb.

BERTRAND MASURE: [INAUDIBLE]

MATTEO CROCETTI: This is the tool part for the--

BERTRAND MASURE: So you've got three lines for the skin. And inside, it will be the honeycomb.

AUDIENCE: [INAUDIBLE]. [INAUDIBLE] 3D printing [INAUDIBLE] that does metal centerings. And they have made ears. And the ear had a honeycomb in the center. And then a thick, not honeycomb. It was a [INAUDIBLE] around the perimeter.

Is that control at this stage? Or do they have to model that honeycomb somehow?

	[INAUDIBLE]
MATTEO CROCETTI:	Yeah.
AUDIENCE:	So at some stage in here, you had a choice to say, I want the [INAUDIBLE] to be solid. But I want the middle to be [INAUDIBLE].
MATTEO CROCETTI:	Yeah, for doing that you need--
BERTRAND MASURE:	[INAUDIBLE] for honeycomb you want.
AUDIENCE:	So you can exclude some areas from your density [INAUDIBLE].
MATTEO CROCETTI:	I don't know, maybe no. At this stage, no.
AUDIENCE:	[INAUDIBLE]
MATTEO CROCETTI:	Yeah, but for our--
BERTRAND MASURE:	If you want to get more control, maybe Within should be the answer.
AUDIENCE:	Maybe what?
BERTRAND MASURE:	Within, Autodesk Within.
MATTEO CROCETTI:	Within is another-- you know? You already see? Yes, it's a program where you can say to fill the area inside the component with a different texture. And you can check with the fan. This is the important things, because if you subtract material inside, you have to be sure that it work.
BERTRAND MASURE:	But I think Autodesk Within, it's maybe one of the answer. Have a look. I think you will get better control than Spark.

MATTEO CROCETTI: OK, thank you very much for attending this class.