Metal Additive Manufacturing
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Indian Institute of Technology, Kanpur
Lecture 33
Reverse Engineering in MAM
Laboratory Demonstration-II

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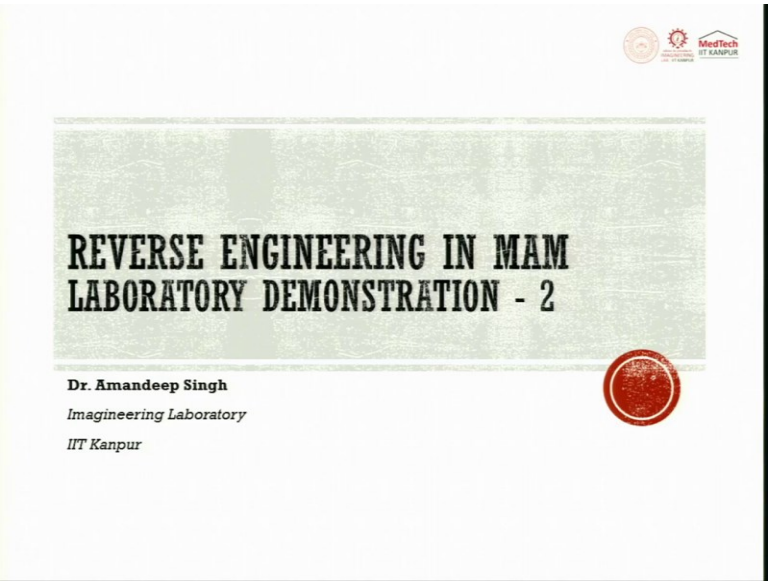

Welcome to the 4th lecture of the 9th week of the course metal additive manufacturing. We have discussed about reverse engineering in this week. And this is the last and the 4th lecture, where I am going to take the laboratory demonstration the second part, where I am going to take you to the design X Geomagic software.

Majorly there are 4 steps in data capturing using 3D scanner, number one, creating the project, number two, alignment with a rotary table, the synchronization of the rotary table and the camera, and the software that is to be made sure, then we capture the data actually, we start the scanning process, and then we clean the data that is we edit the data that has been obtained using the scanner using the structured light that we have got in the previous lecture.

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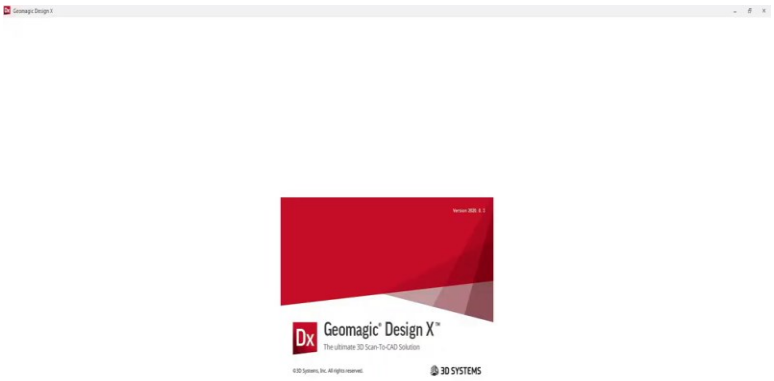

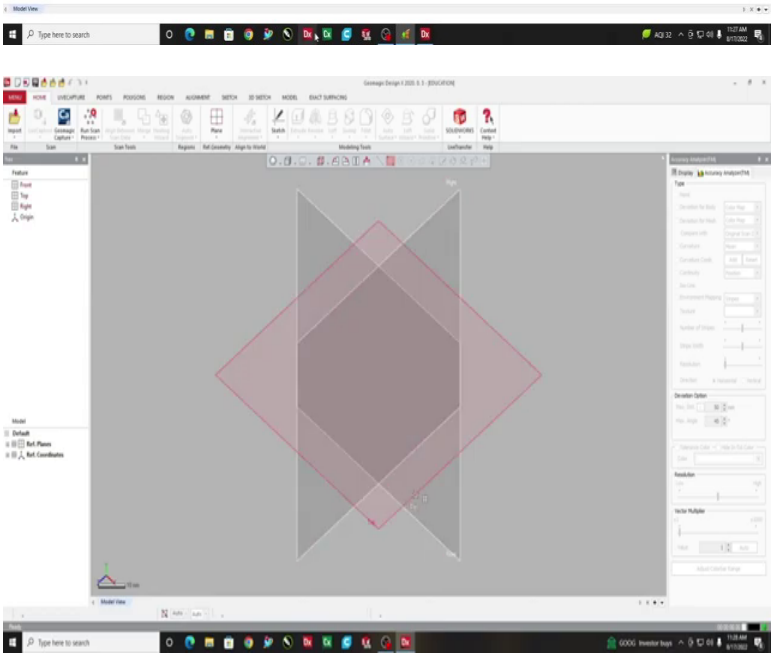

Now, we will start our software which is Geomagic. So, I am going to start the file, I am going to open the Geomagic design X software, if I click on the Geomagic icon on the desktop, it will start opening the software.

So, if I get any scan model, if I have any scan model, I have to now create a solid model out of that. For that part it could be taken care here in the Geomagic, where I will try to create a certain model, it has certain steps that is bringing the data or bringing the file or importing the file that is scanned using the scanner then trying to put that file in the layout of my work area in the

software then we have different tabs here in the software in the right side sequence see there is an analyzer that is the accuracy analyzer.

So, each time and again we need to click and see whether how accurate our obtained model is to the actual model. So, accuracy analyzer will tell us that based upon the color patterns, so, it will just project the obtained model on the colors that how exactly it is matching. So, there are different uses of the different tabs which are available here for example, home live capture points polygons.

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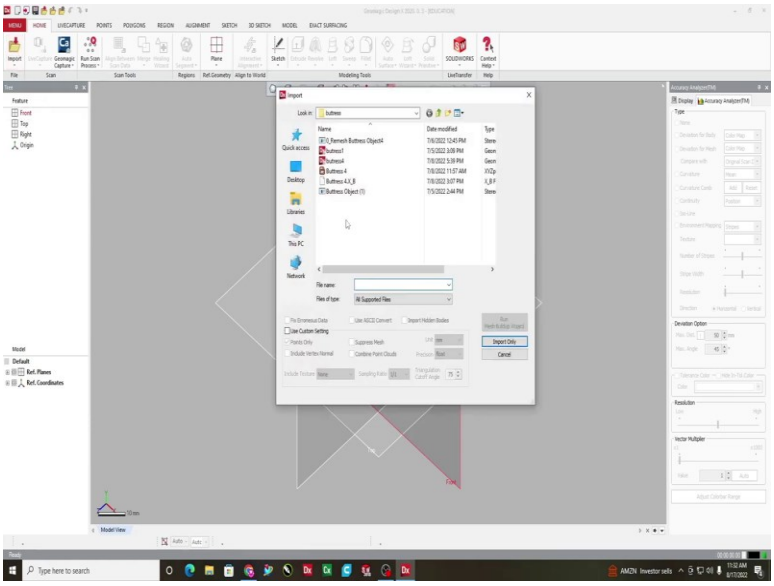

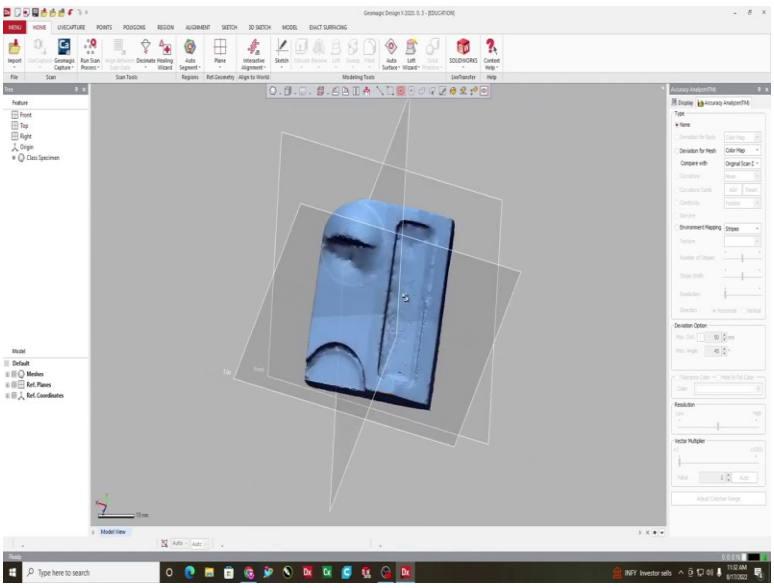

So, from the import option, I try to get the file imported into it. So, in the class specimen folder, there is a file kept as class specimen, it is in the .STL format, I need to put the file in this folder by importing the file using this import button. So, the file is imported here.

So, we can see that this is the model that is obtained as far as possible, we have tried to clean it there itself while scanning all the unrequired or unwanted points were all deleted, but still it has certain issues like the surface finish like the edges are quite distorted, this is the way how the data is captured there.

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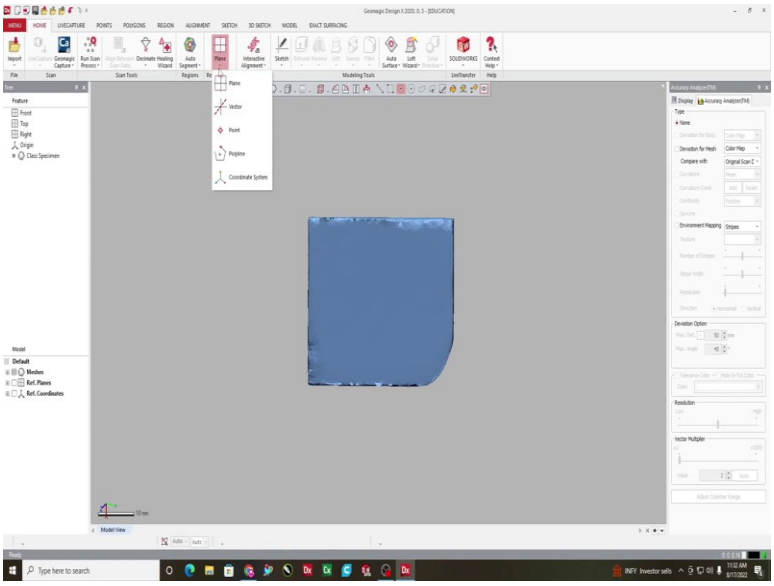

So, using the dimensions, or using the original dimensions or the help of the software tools we will try to make it as soft or as clean as possible. So, there is add plane, if I first need to add a plane here, so I will select 1 2 3 4 points and I will generate a plane here this will become a base plane so, based upon which the object will be created over it. So, there is a blue line on outside the boundary of the object, well this is a virtual plane now.

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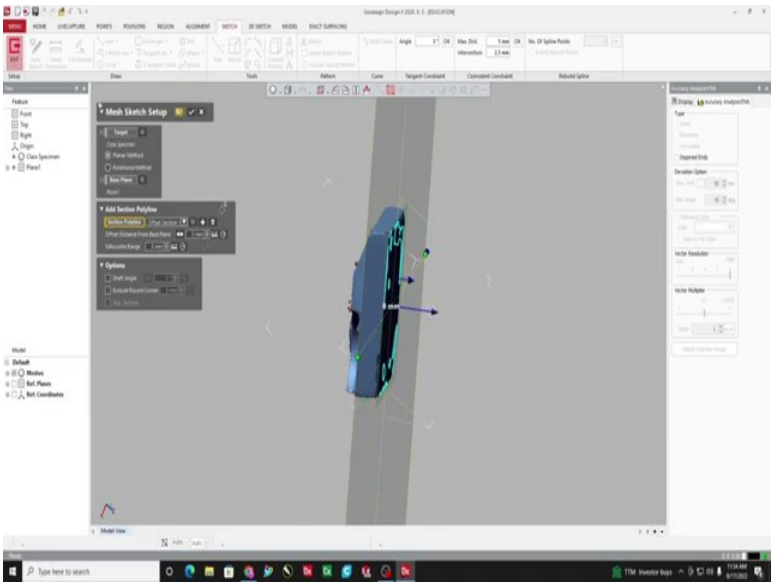

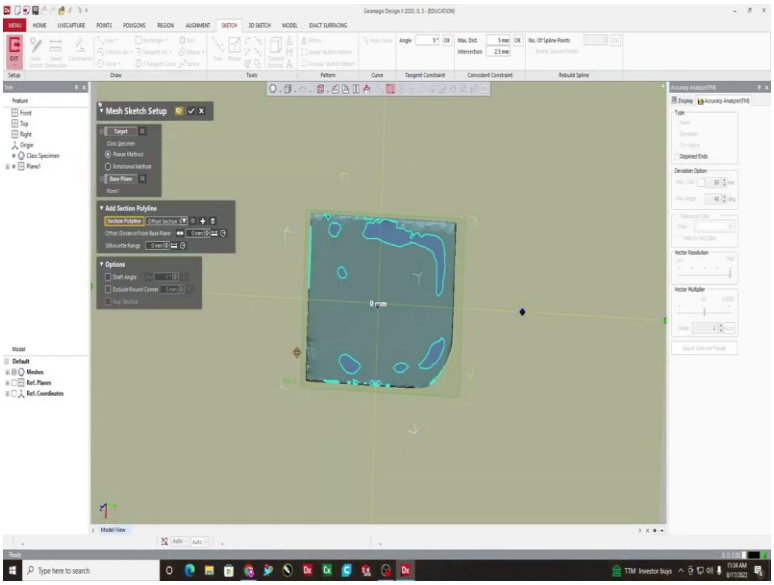

So, now I go to the sketch menu bar. Now, in this mesh, sketch there on the left of the ribbon. So, if I click it, it will show mesh sketch and sketches. It will now help us to create the outline of the virtual plane, virtual outline would be seen when we try to click it, it will be now in this form.

Now, you can see when mess sketch setup is opened, it is showing the virtual plane here. So, it is showing that how closely we can move the polygon sections here. You can see these dotted blue

lines here that is the intersection of the touch area, contact area of the virtual plane with the object.

So, it must just touch the boundary. So, now you can see this purple line, this purple line is showing the object outline. So, there are different models down here, meshes sketches references. So, we can hide and unhide the points. So, once it is opened, you can see all the ribbons are now active, these are enabled in auto sketch also we can count.

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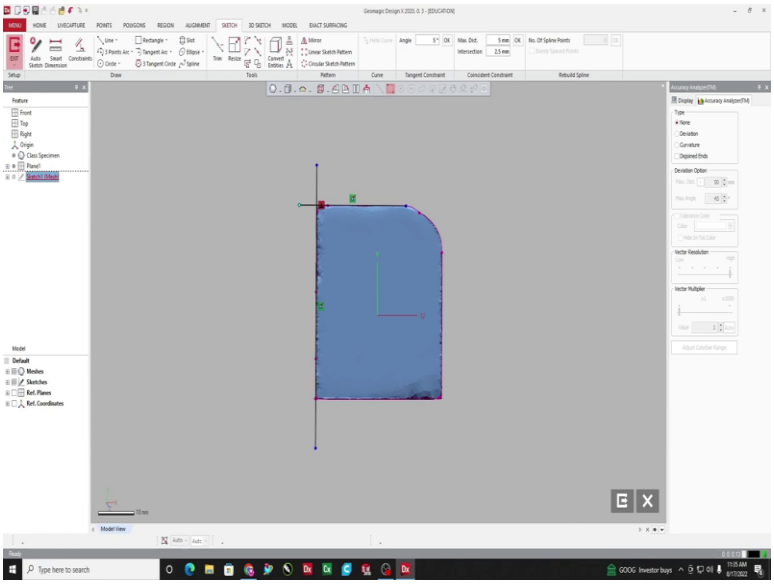

So, we can just try to make the line as straight as possible. So, we select this line set there, I just extended it so that I make sure that the line is properly straightened. So, the nodes are extended beyond the boundaries of the object. So, this line is now put here, now, I need to create its constraints, I will make another perpendicular line to it. So, again, I fixed line, the second line, is fixed now the nodes of the second lines are also extended. So, we have got one straight line one horizontal and one vertical line.

So, using this procedure only we try to put the lines in the perpendicular and to the horizontal of the one line that we started in the beginning, which we picked the longer edge of the object here. So, in this case, we then got the whole base of the object, this is using the similar methodology, we tried to create the side surfaces at the top surface of the model, then the dome, the cut pattern,

the dome, dome cavity, all those would be made further for those, we can create a tool over it depending upon the angle of the pattern that we have, we can generate a surface over it.

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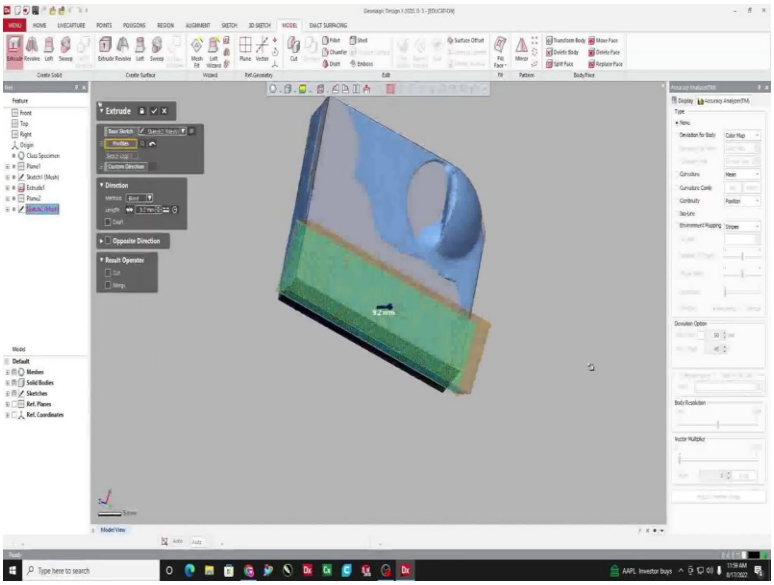

For example, this so we have created. Say there is an extended volume, over that is kept over the body that is kept at an angle for example, this is 30-degree angle or so. So, it is created now this object that is yellow color that will act as a tool, it will cut the object and try to make it straight.

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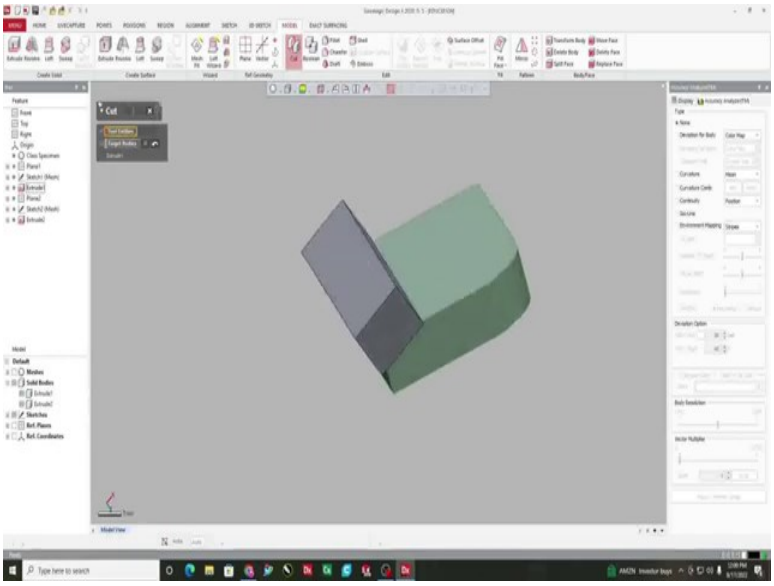

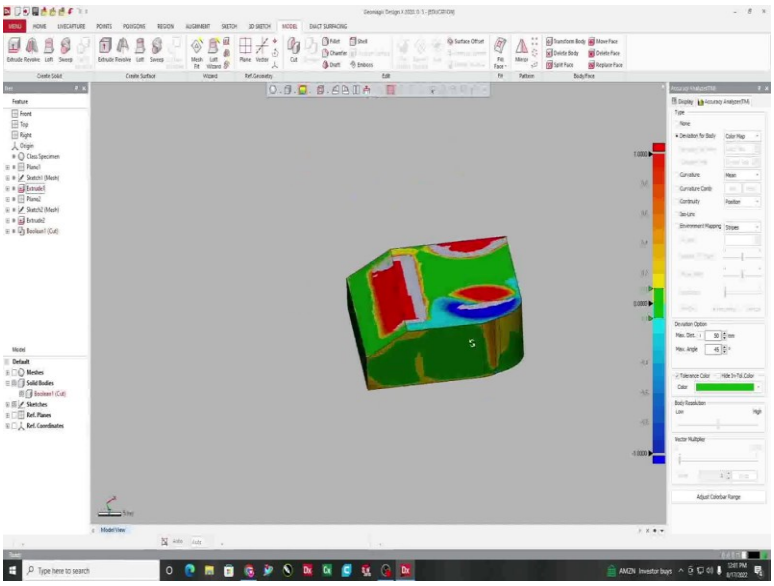

You can see in the model, we say cut, we select in the model tab, we can select cut. So, this is the body that is created, the top and the surfaces are created. Now, the second part we have can regenerate your surface here, when we extrude it, we get it to kind of the body here, that tool why I am calling it tool, because this I will use to cut the portion of my object. So, this is a target body, green one is the target body.

Now, another one is a tool body, now green is the target body so that will, just try to cut it there, there are 3 options which are asked here, merge, cut, intersect, so I need to cut it. So, using the creative body that we will just use to create the angle over here we have cut our original body. We have cut our original body and now using the accuracy analyzer, we can see that maximum part is green.

So, what is this x accuracy analyzer telling, the green part is telling that this is closest to the original body, yes, there are red portions that means there are no material there, there are potions which are blue or so that means the upper side the material is extended there.

So, in this case, you can see the colors vary from blue to red, in between we have green, red means cavity at the top and down there it is blue that means the material is access over there. so, it is cutting the material there because we have not created dome. So, wherever dome was, it is showing blue color wherever dome cavity is there, no material that is showing red here.

So, it is almost complete we have tried to make it as close as possible, that we have tried to make it very close to the final object that we need.

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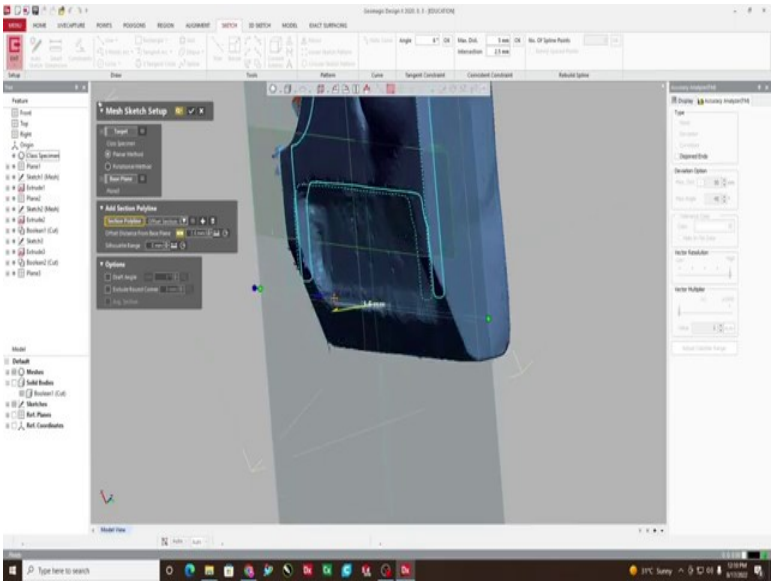

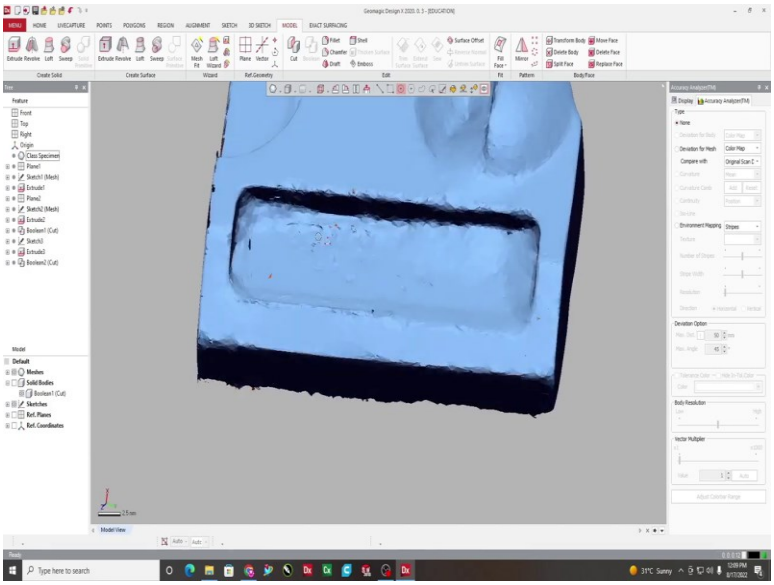

So, this is the object this is the data that we have got from the camera. So, still there is a cavity we need to create a plane here. Then we can create this pocket here. So, again, the red zone means there is no material. So, I will create a virtual plane here. So, this will be a third plane. So, using the plane menu, we will create a plane here. So, we come to the plane menu in the model,

the plane ribbon, in the model, we select plane, then we select or in the sketch itself, we can select mesh sketch setup.

So, using the sketch menu, or mesh sketch setup, again, we can try to align it we can see, so where the plane just touches my virtual plane, the plane would be there. So, I am trying to just see okay this is complete almost closest alignment with the upper surface of the body.

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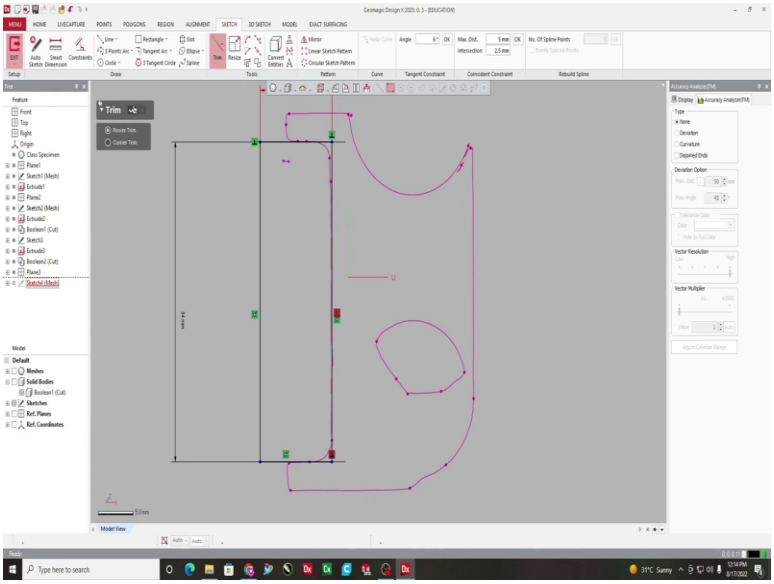

So, this is again, a plane is being created at the top. Again, as we did for the bottom plane, I am just showing it again. So, we can use the line options or the sketch options. So, in the sketch, I go to the ribbon where line is here, if I create a line here, I create another line that is parallel to this line.

So, these two lines are already generated here. So, we can make them parallel to each other. So, the lines are to be made constant, the lines are made parallel to each other. So, again, I come to the line command here. So, it is now being made fixed here. So, it is asking vertical or horizontal, so both the lines are now selected we select parallel. So, both the lines will be parallel to each other now.

So, the nodes of the lines, that is the end of the line is a line segment which has fixed ends here. So, I am just extending them so that I see them clearly. So, to put the pocket, the actual depth of

the pocket is to be taken which was seen when we touched the virtual plane there that was around 3 mm.

So, again, I put the constraints here to put the pocket I need to have the boundary of the pocket where the pocket is required, I now put the lines here they are perpendicular to the line that is generated on the top. So, this is made perpendicular to it, I have just extended it so that I get it clear again a similar line is created between the two nodes up there. So, these are again to be accepted, this dimension could also be seen what the actual distance is, it is yet tilty, it is not parallel that is why the dimensions were not coming correct.

So, I again make the small line that is up there perpendicular to the adjacent line should be these two lines now become parallel now the dimensions have come. Now, these dimensions it is which was showing in points, but it is now put what we were gone about 24 mm, these two lines are now perpendicular, and the lines within each dimension given are parallel to each other.

Now, we will create the width so we can see one line. So, to fix the width, we will also just create a line here we will fix the dimension 10.2 mm. So, the field or the extension of the line is again made, and we create them. Now, we can trim the line from the same tools ribbon here in the sketch tab. So, we are trimming it, the unwanted part is all trimmed.

So again, we will come to model options. It is the model tab. We will try to extrude so that we create one tool, and the main body remains as our material that is cut and we get this pocket in the model.

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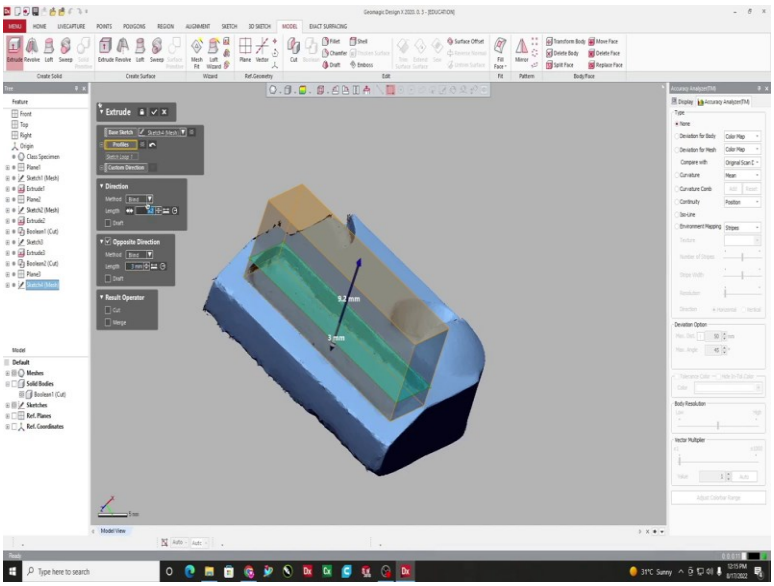

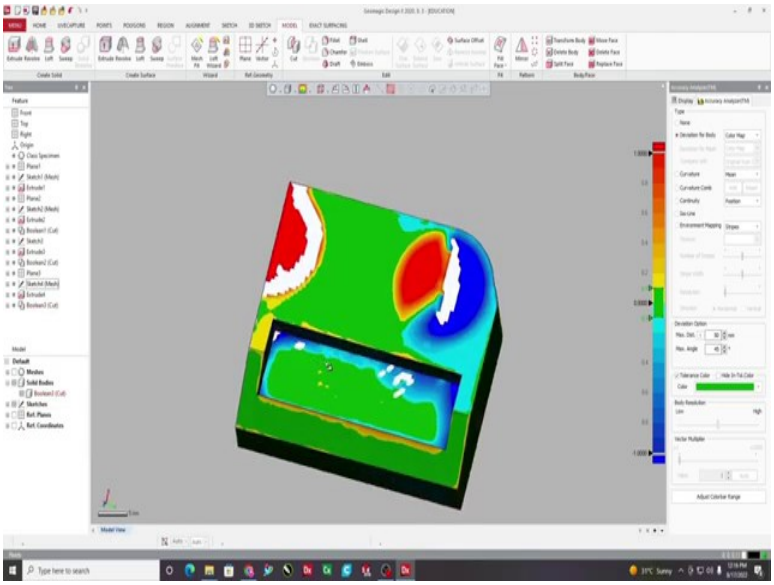

So, now this is created. In the model, it is extruded out and down there its side it is 3 mm. So it becomes my tool, my base objects becomes the material and this is, this will now help us to cut the pocket, but its pocket definitely does not show the fillets or the curved edges so if we now perform the Boolean operations over it so we can again cut it so we will cut it, it is showing right intersect, the tool merge cut intersect I just select tool, select the material, previously the options were swept through it just cut. The tool was selected as the material and the materials selected as tool.

So, now it is corrected. So, now again there is green zone showing what the gotten captured data using the scanner and the data that is required how it is showing closer to it this only gave us the dimension that the depth is 3 mm while putting the virtual plane there.

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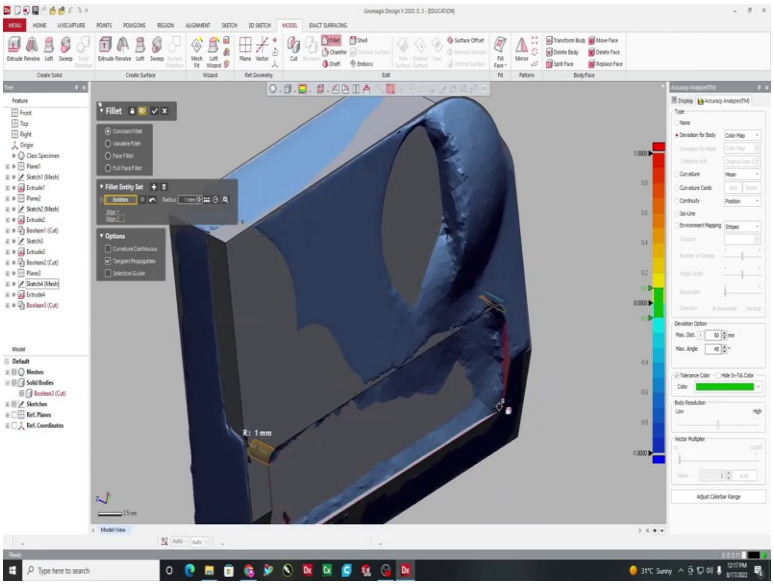

Now, we can put the fillets here, the fillets are also calculated from the data or from the captured data. So, we will put 1 mm fillet here, each fillet is 1 mm, while putting the dots there in the original data that is captured, those fillets are put in the all 4 corners, here all 4 edges, so we can hide them as sometimes the corner is not, we are not able to select, so now we can select the corner, we have hidden the original object.

So, these 4 fillets are now put here all with 1 mm, these pockets are not generated, these fillets are generated, and the pocket is now complete. The maximum part you can see is showing the green zone using the accuracy analyzer time and again we are able to find how close our object is to the captured data.

Now, we are now left with these slots here and the dome so again, I will go to the model tab the reference geometry ribbon and select the plane. We will add a plane here, I will select 3 points here and put a plane here again I will come to the sketch options that is sketched tab, I will again create mesh sketch setup.

So, this is a virtual plane again. So, this will also again give me the pink dotted boundary or the purple dotted boundary so I will try to see where the surface is required. Now, we have got this line here. So, we will now hide our mesh, mesh that is the captured data.

Now, this is the circle that is to be generated. Which option is suitable to generate a circle, there are certain options available we can create a circle directly while putting radius or using 3 points or 3 R's so we will select circle so we can create circle using this, we have now generated a circle again we will have the mesh body, we will have the dimension of the depth of the slot from here where circular slot and we will have the solid body, this circle will now again be used as a tool so we will extrude it and create a tool out of it.

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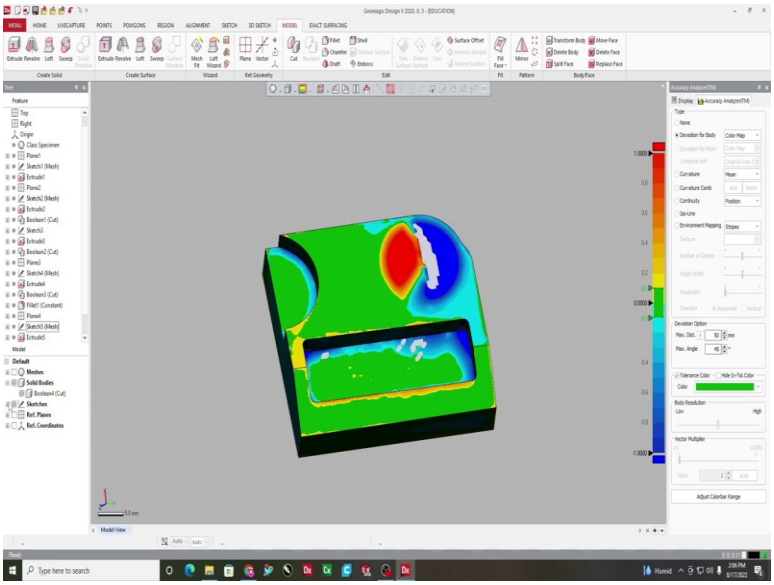

So, I exit again I come to model option this is asking to create a tool out of it or create a tool means I am creating another body, the depth is again 3 mm here. So, we come to extruded part, we operate the Boolean operation here, we select the tool and the target body.

So, now it is cut extruded. The target body and tool body again it is coming closer to what the CAD model would look like or should look like. Now, this is the mesh body that we have got. Mesh body means the captured data that we have got from the scanner.

So, the CAD model now is when it is projected over it so, we can see the maximum part is green it is close to what we have got in the data. So, now only the red portion and the blue portion is left that is the dome.

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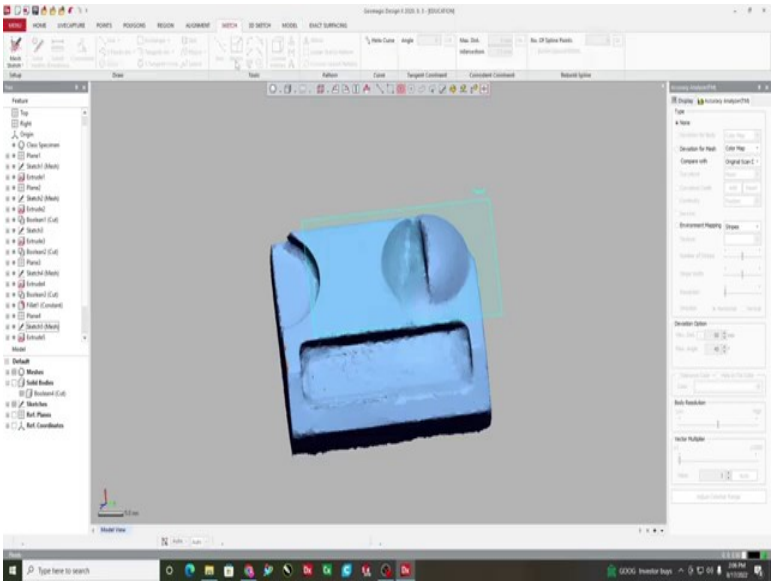

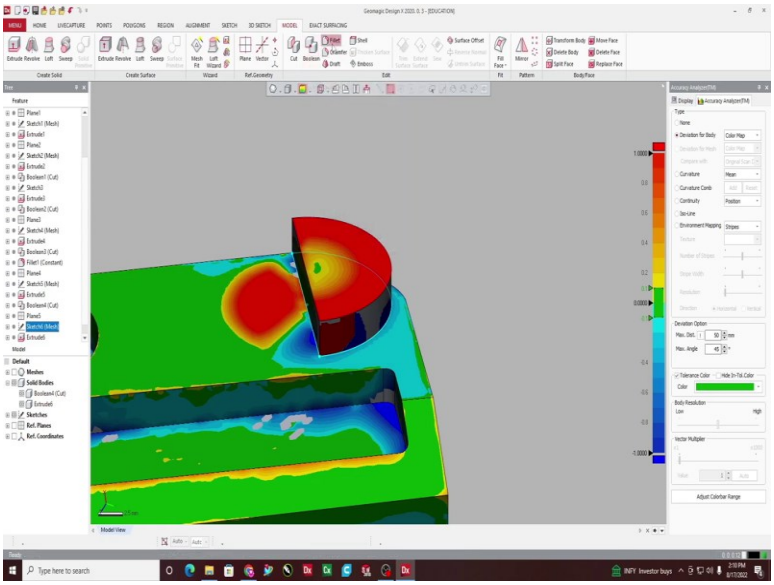

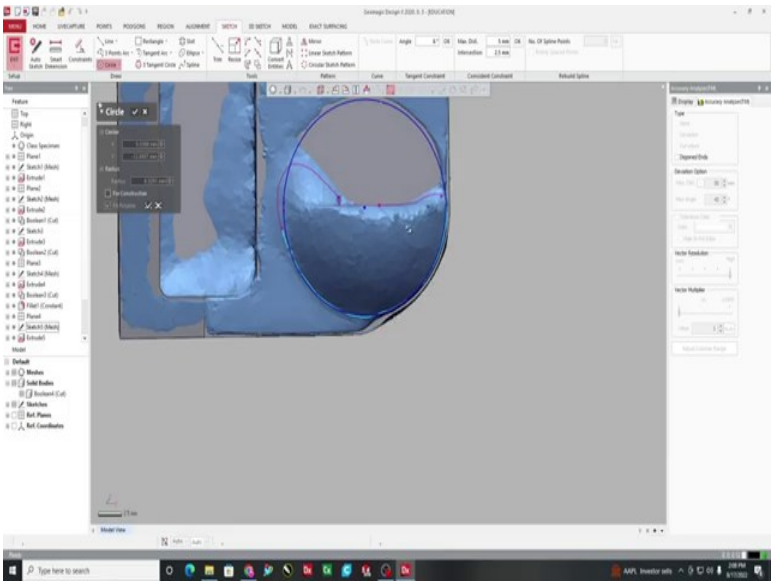

So, this is my base object. So, now I will create the dome for it, cavity dome and the projected for the protruded dome here. So, I will again create a virtual plane to see whether how to see the circularity of the dome. So, mesh sketch up setup will give me this line boundary, this line boundary would be the outer boundary of the dome so it actually looks like it is a half of the total circle, it is semicircle, so we have created this kind of the semicircle body we will try to see how to generate a circle out of it and we will try to generate the dome so number one is circularity, number two the dome straightness of the diameter of the circle.

So, we have just selected circle from this sketch, so we will just try to make a circle using 3 points here, we selected 3 points in trying to fix a circle this is always the skill of the operator who is trying to do this.

So, we are trying to generate this body. So, we have created this circle. So, it is divided into half part here. So, using a fully rebuilt body, the project-oriented user interface is always there. So, half of this has to be dome, half of this has to be cavity, it depends upon the sometimes we also take the photograph of the object, and we took them put them together, this helps the user to have better results.

The softwares are always created to make it as simple as possible. So, it requires little help to set up parameters to speed up the scanning by letting the user focus on the task that is there in the hand.

So, we can see now that it is connected the circle that is generated, that is connected to the base model here, I am trying to create the diameter out of it. Yes, the diameter is almost matching, I am just trying to create a line on the purple boundary of the object that I have got using the virtual scale which is again fixed it is extended, so that we try to intersect it. So, then the remaining part is not trimmed.

So, we are now divided into two parts, because creating a single circle would have been difficult to create separate dome and cavity dome. So, both will be taken separately. So, half circle is generated, or the semicircle is generated. So, now using the accuracy analyzer, we will see the height of the dome. So, taking the height of the dome and the circularity, we must keep matching it using the accuracy analyzer.

First, you can see where it touches, when it is just touches you can see a small green part in the center. So, that means it is exactly matching the mesh body, and it is a mesh body captured data. So, this is the height of the dome that is there that we have got because it was just going to touch the green portion there at the center.

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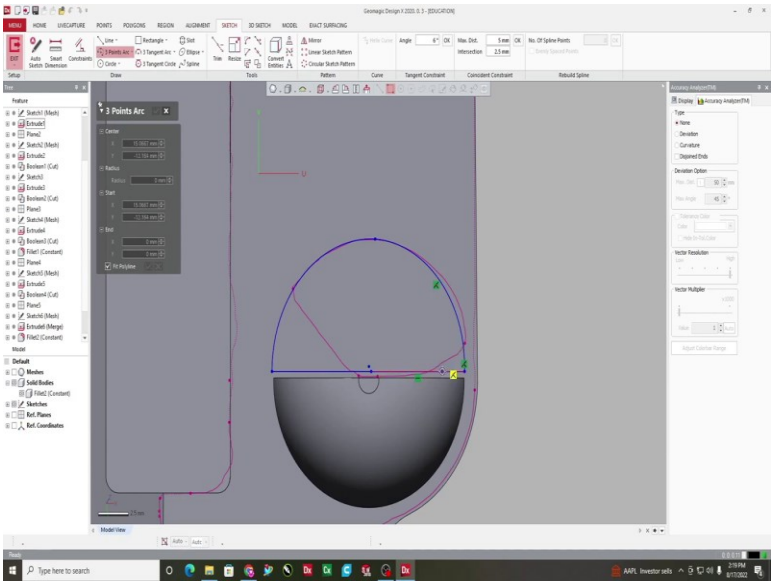

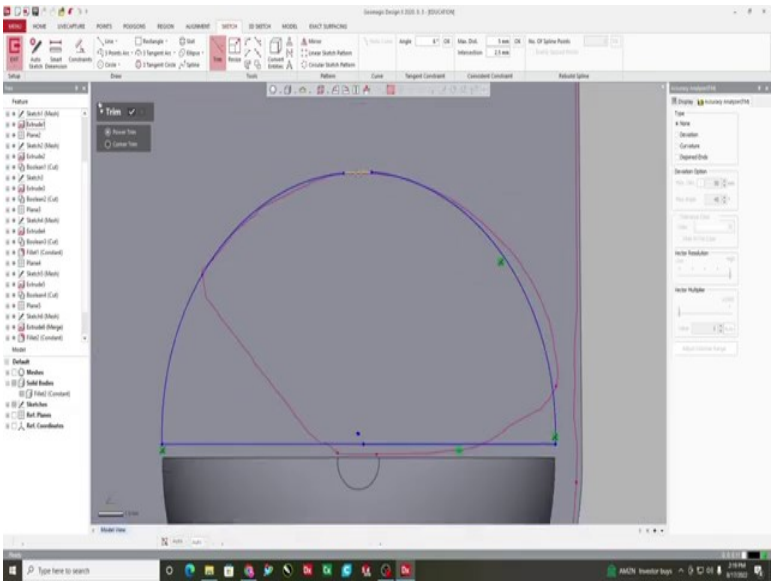

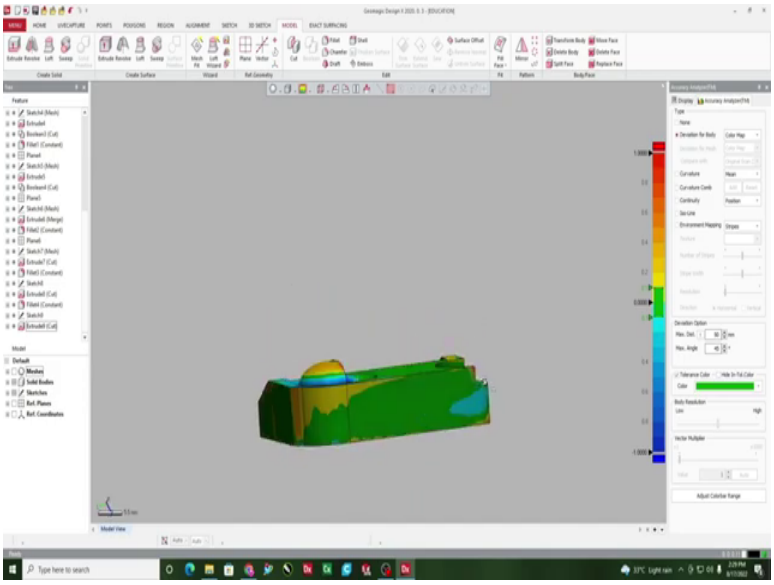

Now, we need to create the circularity or the curvature or the top of it so, the camera lens that we used had 25 mm focal length. So, that is why it has given the great or the close data that we have caught. So, we have created the dome while keep on matching the curvature to the mesh body and getting the green part as close as possible, there are certain small points that we need to take care when we try to scan like the temperature of the room must not be more than 32^0 and the field of view is within 760 mm, so these points we need to be taken care of.

So, now I need to create the dome cavity. I will again add a plane, I will select a plane, I will pick multiple points on the plane, on the top of that we will create our plane, now I will again create a plane on the surface, this plane is generated but this is plane 6, it is also showing the

number of the plane, it keeps on adding the commands that we are the planes that we are generating.

So, now, this is the plane that is generated and, we have got the outer line of the cavity. So, again I select the circle option there I try to match it here, it is also not showing the full circle now if using this I need to generate the cavity, so there is small misalignment.

So, outer pink color or the purple color is not to be adjusted according to. It is the circle that is generated here you can see it is larger than the upper dome, so we need to connect these two, the dome and the cavity dome, these were a complete full circle. So, the cycle that we generated is not working properly as per our requirement.

So, through using 3 points we are trying to match the circle as close as possible. The small hiccups too come when we try to scan something because this is scan data so let me try to put this one circle, so I will try to cut the extended circle part so we will cut the cavity so I will trim the other portion that is not required. So, this part is the upper dome and the other part we create now is the cavity dome.

Using trim options, now, we will try to exit the trim and we will try to make the circle as close to the upper dome. So, through this right edge of the upper dome also I will try to see what the issue is, sometimes when the edges are selected that the circular edge does not match perfectly with the virtual plane because the edge is distorted so, because of the distorted edge at the end or the distorted circular edge it is going the largest circle.

So, we need to now match it properly and it was not the exact diameter, yes now the problem is sorted out the circle we try to generate was not the exact diameter, there is a small gap in the center so, we have created this circle over it, semicircle over it so, again the depth is 3 mm again the dimension is 3 mm here the depth is 3 mm so, pocket circle, the dome, everything is 3 mm in this object.

So, since we need to cut it down, we will again come to the extrude command and put cut extrude so, we will get a cavity here using this circular cavity, we need to create the replica of the dome, exact replica we do not know but we will try to match it with the mesh object and try to see when is the maximum green color coming.

So, fillets are also to be put here down there. So, 7.5 is the fillet radius that is to be generated here. So, it is just 7.5 mm radius fillets size here now, the fillet is given here, and we have created the dome cavity also here. Now, the object of the solid body that we have generated is only always check through accuracy analyzer. We just noticed that the dome circles were not matching because we thought of that as the exact diameter, but it was not the exact diameter when we see.

So, you can see maximum of this is green. But still, we are not able to get the perfect dome. Obviously, it is not a 1 hour or half an hour job, the operator has to work on it multiple times it is sometimes too small, manual cut in the curvature in the fillet to make sure that using accuracy analyzer, we keep on obtaining the model that is closest to the mesh.

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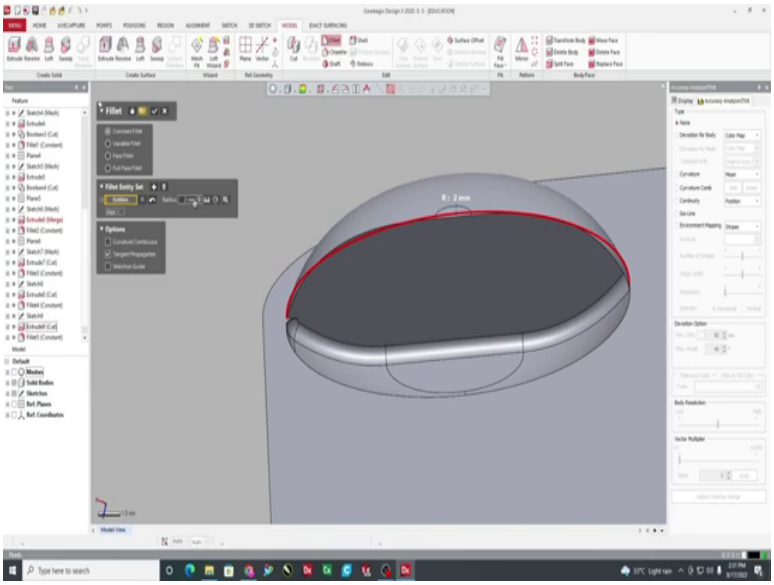

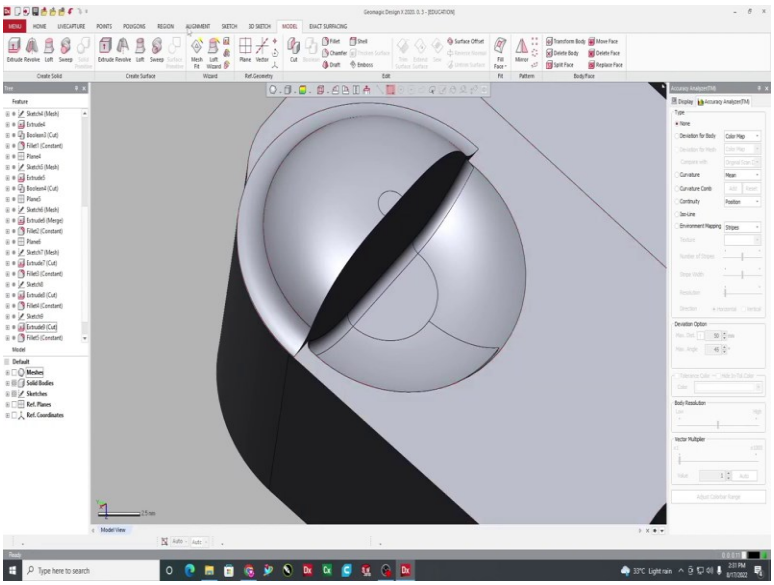

Now, we are generating fillet down there. So, small fillets which are observed from the mesh 0.7 mm fillet is being given here. Now, the fillet is now made 1 mm here or 0.5 mm so, only small approximations are being taken then we will match it with the obtained mesh. So, this fillet is also to be given on the edge at the back of the dome. Again, this is given as 1 mm.

Sometimes, we have the freedom to make the product look a little better when we try to print it or sometimes the fillet's requirements are always there to have the good joining in the 3D printing as well. So, those fillets are also put accordingly. So, just to make it look better only the fillets size is put in larger.

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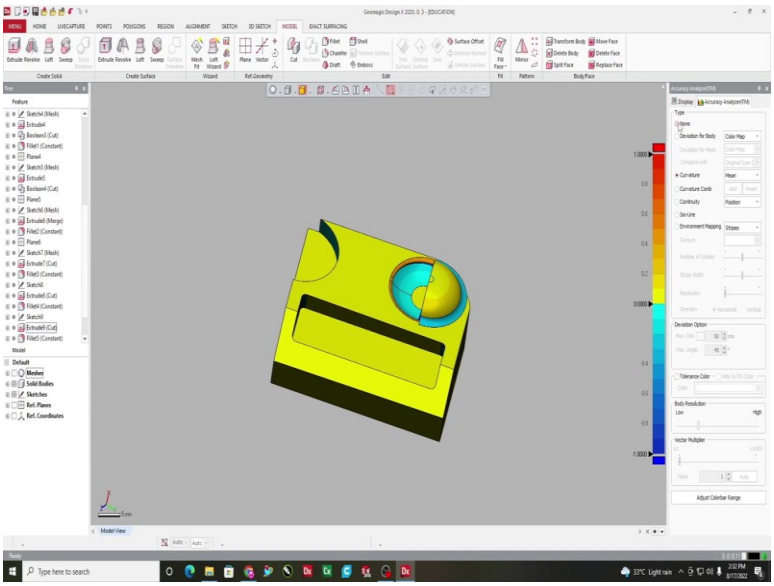

So, the blue line, which is showing fillets now, using accuracy analyzer, it is showing that it is a good match, so this is the scan part that is the mesh body we have the dome, the dome cavity, the cut extruded part, the pocket, so, this is a solid model.

We have now obtained the solid body so, sometimes the design of the object is available, we need to just compare it with the existing design, sometimes the design is to be generated. So, this object that we selected was already distorted to just to show you that what kind of challenges could come when we try to obtain the data, we have selected this model here.

Now, we are trying to the save this model, say any of these formats it could be saved. Step format or so, at any it is being saved in the IGES format. So, again it is being named as class specimen IGES. So, as per requirement it is saved. So, in this way we have obtained the solid model which is close to the mesh model that was generated. There are multiple other things that could be thought of that could have been missed in this lecture.

Though the learning of the 3D printing process, I have just shown you the major steps in data capture, the data cleaning during the capture itself and the alignment, the selection of certain parts when we start the creating of plane or so how do we cut extrude, general just like we work in the other solid model developing we will try to work on the data cleaning or data editing using the software Geomagic design X.

We will try to keep it matching using accuracy analyzer and try to see how close it is to the obtain mesh object. This is how reverse engineering works. There were many things that could have been missed in this lecture in a way multiple commands, the multiple options in different tabs. There are multiple other small intricacies in different ribbons or selecting dimensions. The purpose of this lecture was just to have a brief introduction to how does editing and data capture happens. Let us meet in the next week. Thank you.