Engineering Drawing and Computer Graphics
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Lecture – 59
Solidworks (Contd.)

Hello everyone. Welcome to our online certification courses on Engineering Drawing and Computer Graphics. We are learning about a software tool named Solidworks to construct 3D drawings. In today's class, we will learn a little bit about how to make bolts and nuts, how to construct these threads. This is the first objective of what we will achieve

So, first, we will very quickly go through a simple schematic in terms of taking a stud and making a thread cut it on that. After that, we will go construct a metric 10 thread.

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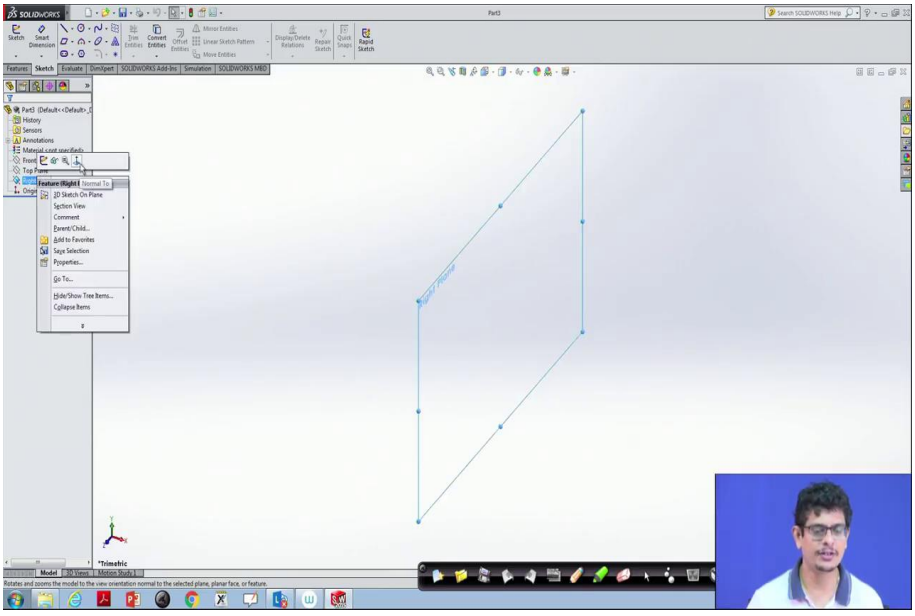

So, the first step is after double-clicking a new part, we will have a blank screen, then we pick one of the planes perhaps right plane normal to the right plane we will draw a stud is a circular cylinder we have.

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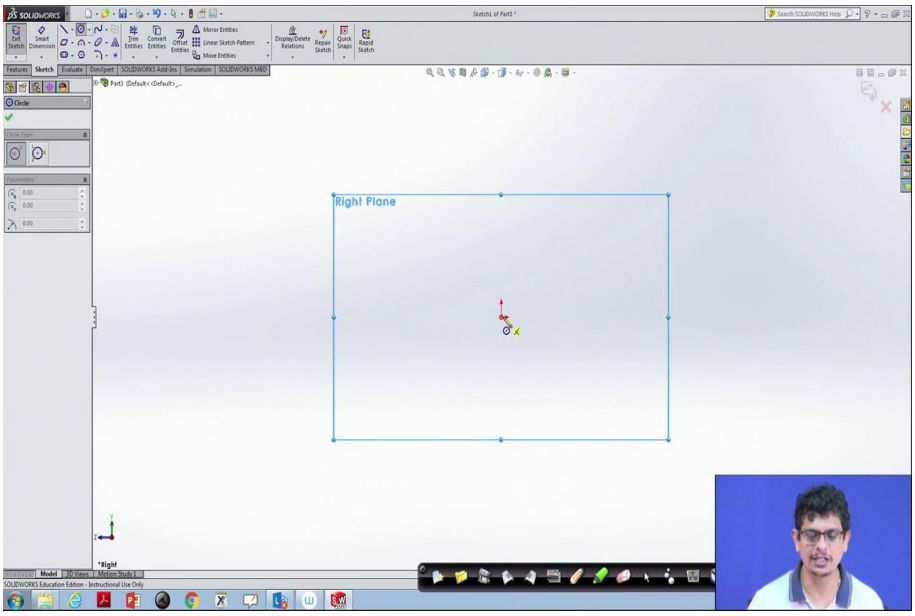

It can be some 5 millimeters radius, ok.

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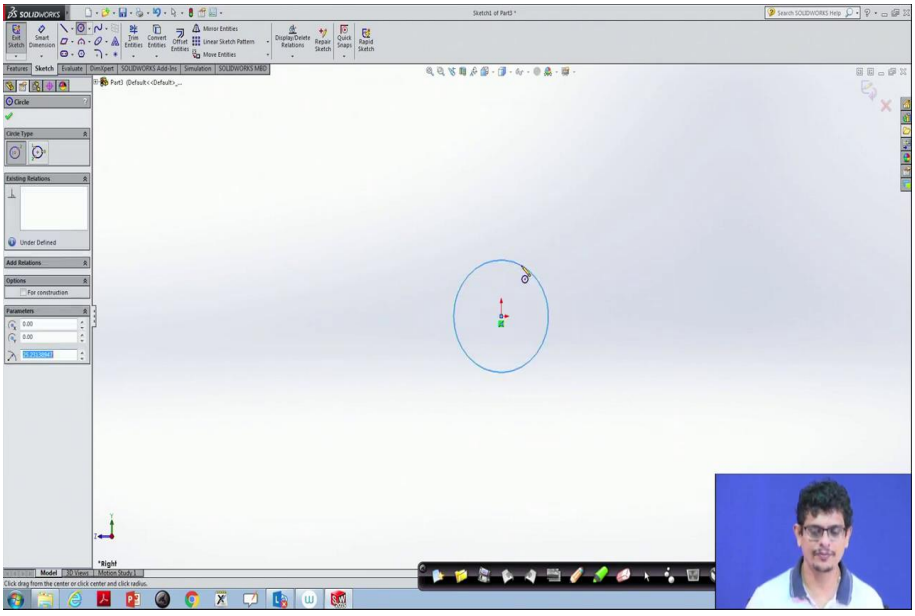

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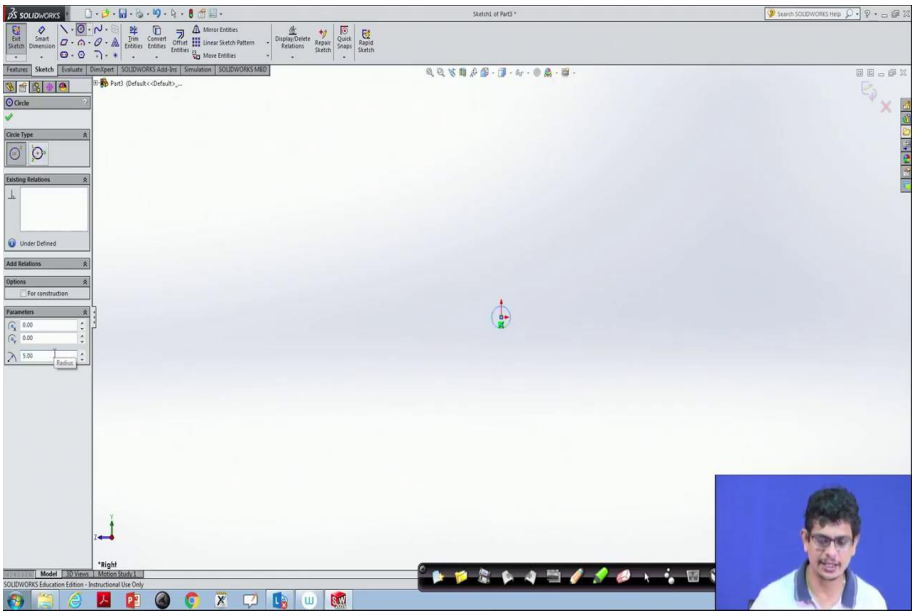

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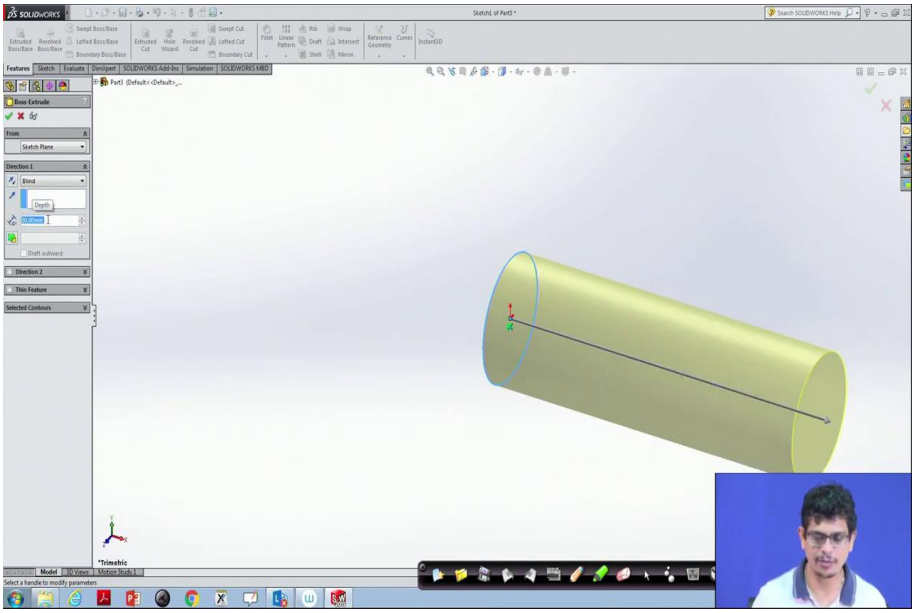

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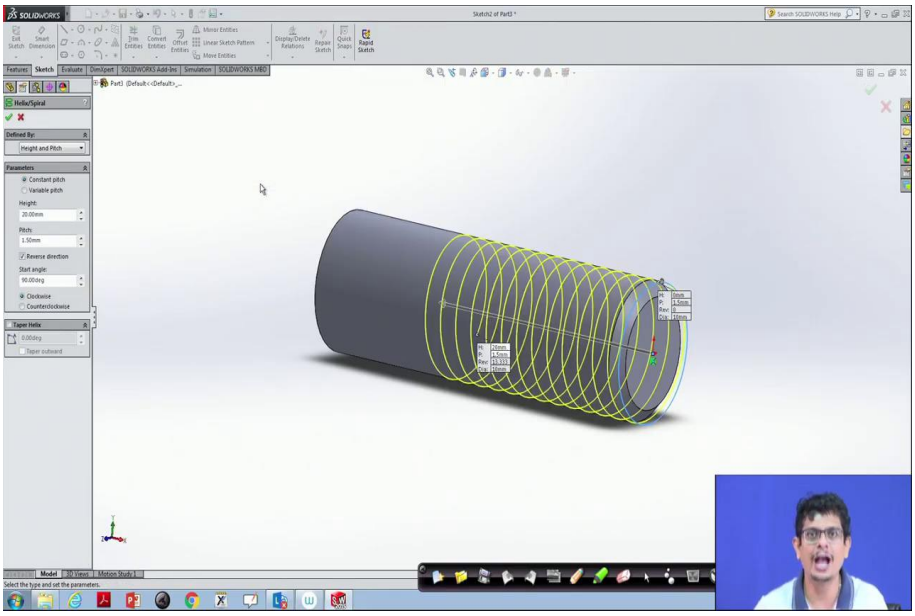

Now, this circle we extrude it in 3 dimensions may be making it some 30 mm. So, we have this cylinder on which we would like to have something like a threaded cut on it. So, usually, these studs are always be chamfered. So, this surface we will take it, go with the chamfering, maybe 1 mm chamfer with 45 degrees we make. When we do we will have that kind of chamfering on that solid surface.

To make thread cut first what we have to do is a helical path because these screws or threads are always be having a helical path about which a thread, a tool bit or cut goes knife, so that it removes the material and finally, we will have the thread. So, first for that what we do is we construct a helix around this shaft.

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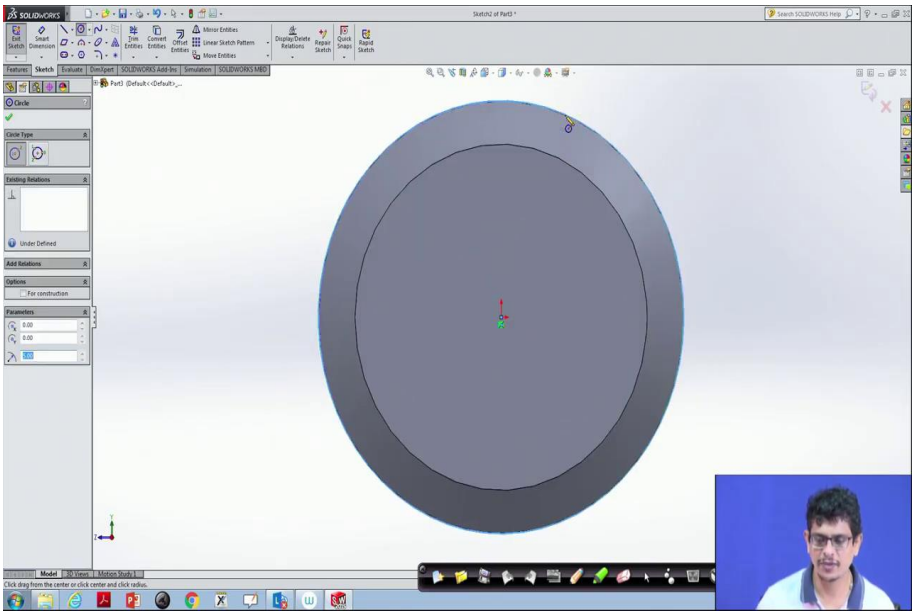

So, the first step is to construct helix we have to make a circle of same 5 meters radius. So, we have that circle. Around that circle, we would like to have a helical thread. For that purpose what we have to do go to insert on the top level there is a curve in that curve goes to helix spiral.

Now, we have such kind of thread. Because it is just practice as of now what we are going to do is we will have some helical path with height 20 mm; that means, the length of the helix what we are going to have is 20 mm pitch is around 1.5 mm, and this is a clockwise kind of helix. And this helix begins if we would like to update or change anything if you give a right-click there is a button to edit future. So, it is starting at 90 degrees.

So, helix begins at starting of this arrow, goes around the cylinder around which we want to construct this cut. So, the pitch is 1.5 mm and it goes all the way to 20 mm. And it begins at 90 degrees, and it is a clockwise uh thread we were constructing, so click ok. Once it is done go to the front plane where we can see this helix thing.

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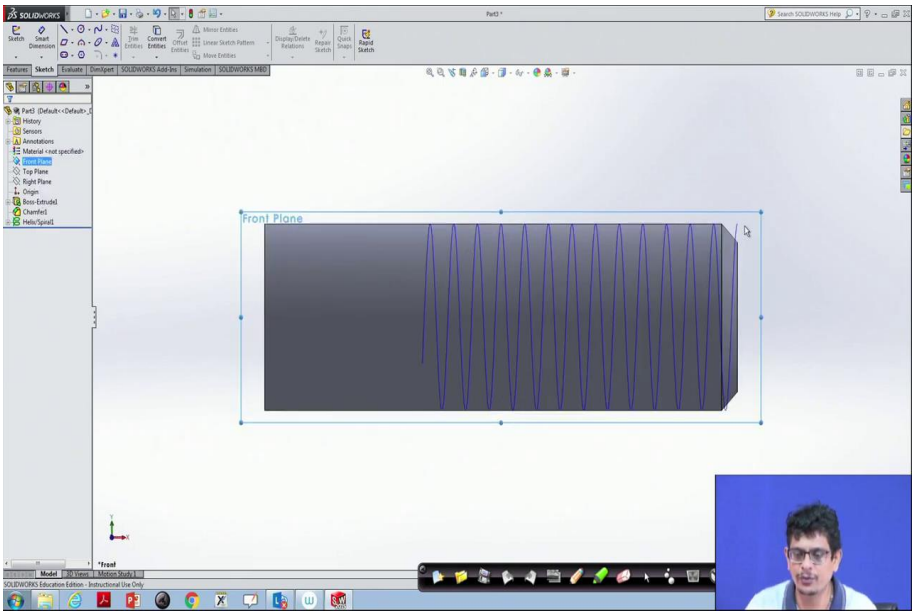

So, because it is a practice, we what we are going to take is a triangular knife, we take it to pass via this helix.

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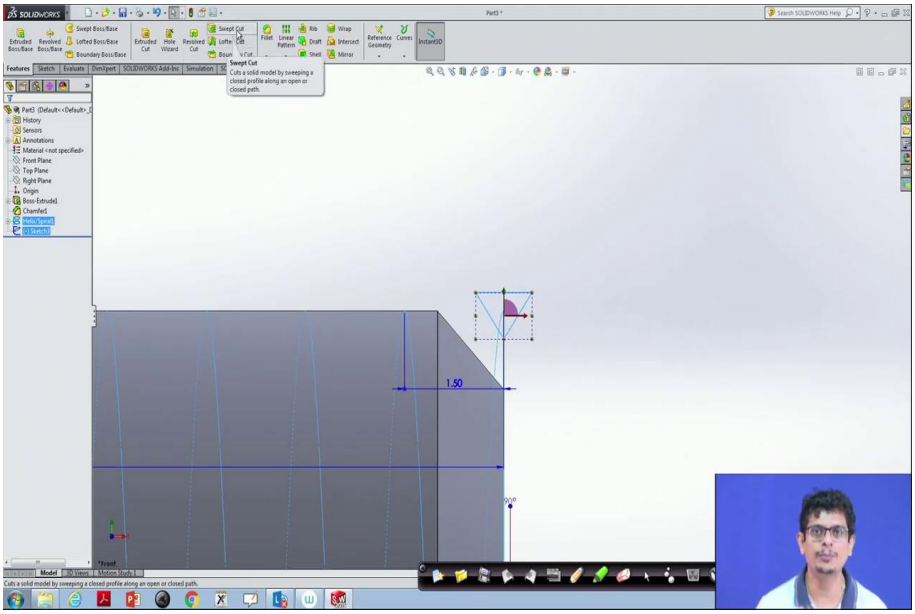

So, after constructing this triangle we arbitrarily place this position because it is just a practice to construct a thread. Once we have that triangle pick these edges come out of the sketch mode, then we will see sketch 3 and also helix.

Now, pick sketch 3 and also helix, both are highlighted. Now, see at the features level there is something like extruded boss revolves based extruded cut and swept cut.

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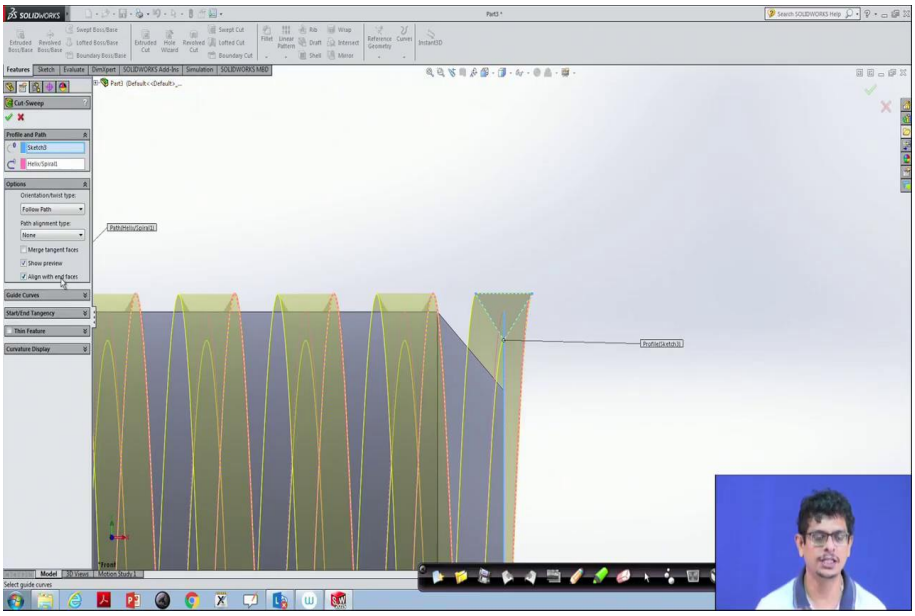

So, click swept cut then we will see something like a threaded portion passes. So, you have to pick both sketch 3, and also helix paths.

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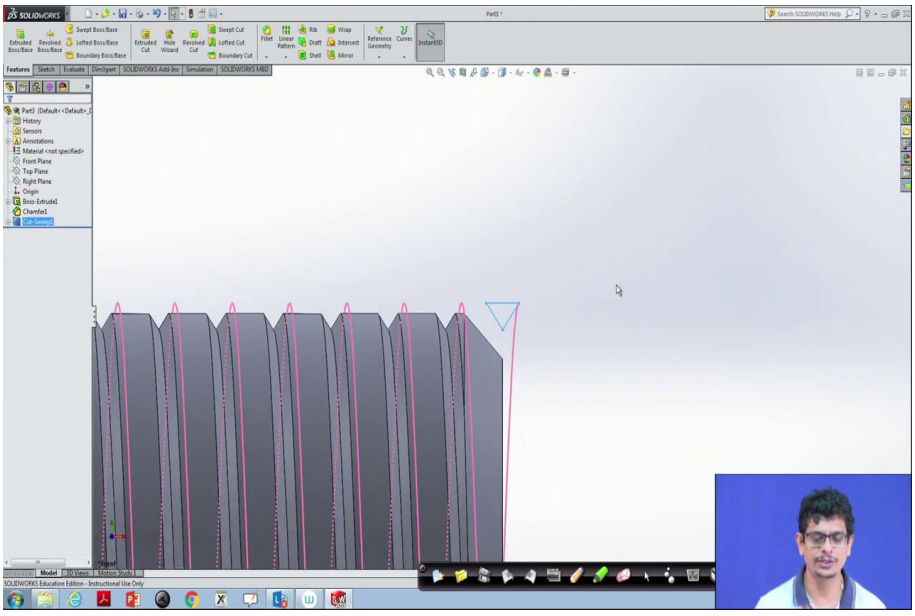

Once it is done go to swept cut it shows this kind of yellowish tint, then click ok, then you see the threaded portion.

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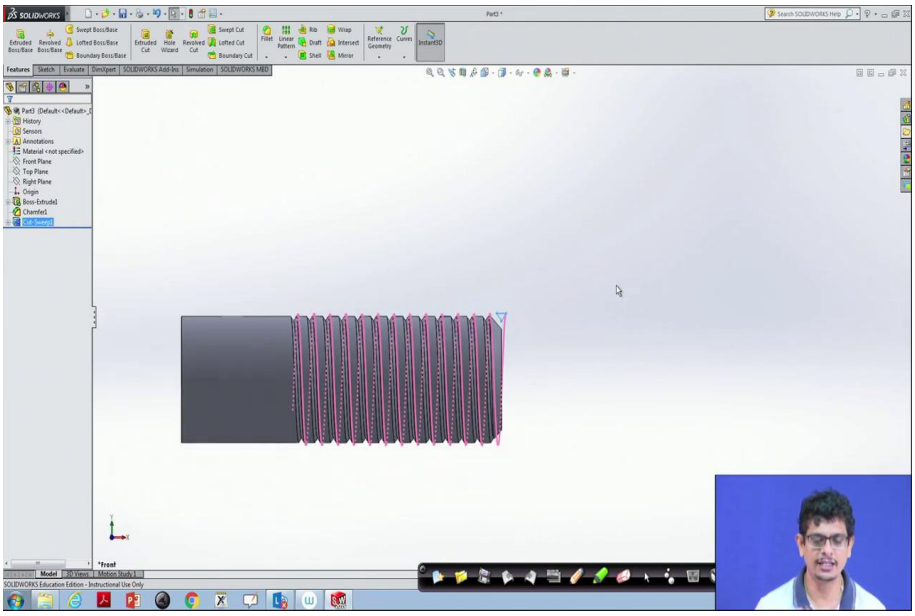

Now, we will construct a systematic metric 10 mm thread. So, usually, these 10 mm threads always are having something like a head portion, there is a stud portion, there are some threads some hexagonal kind of structures we will be having. So, we will go step by step construct that m 10 thread.

So, let us pick a right plane, normal to that right plane. First, we construct the head of a bolt. So, the head always is a circular one. This is a circular one of 16 mm we construct, 16 mm diameter; that means, 8 mm radius.

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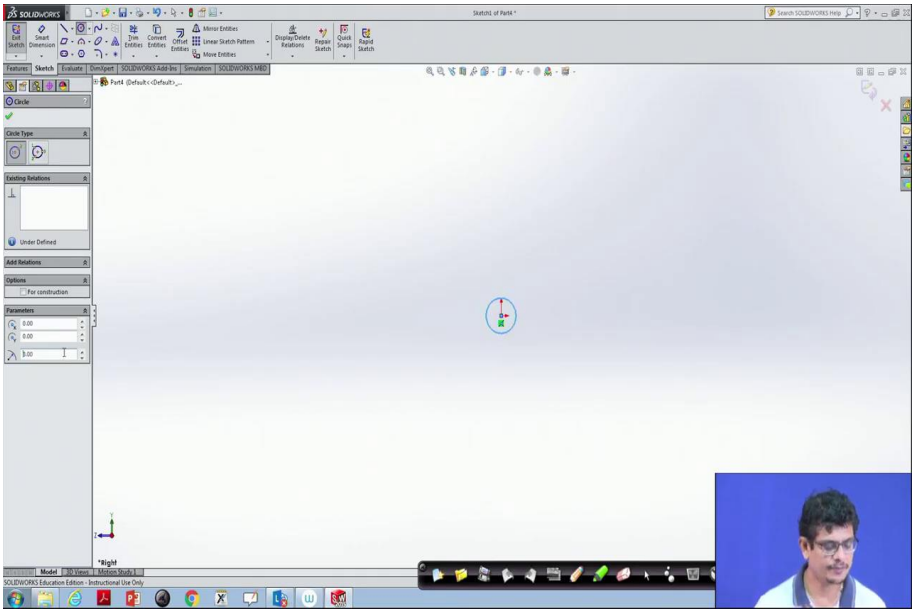

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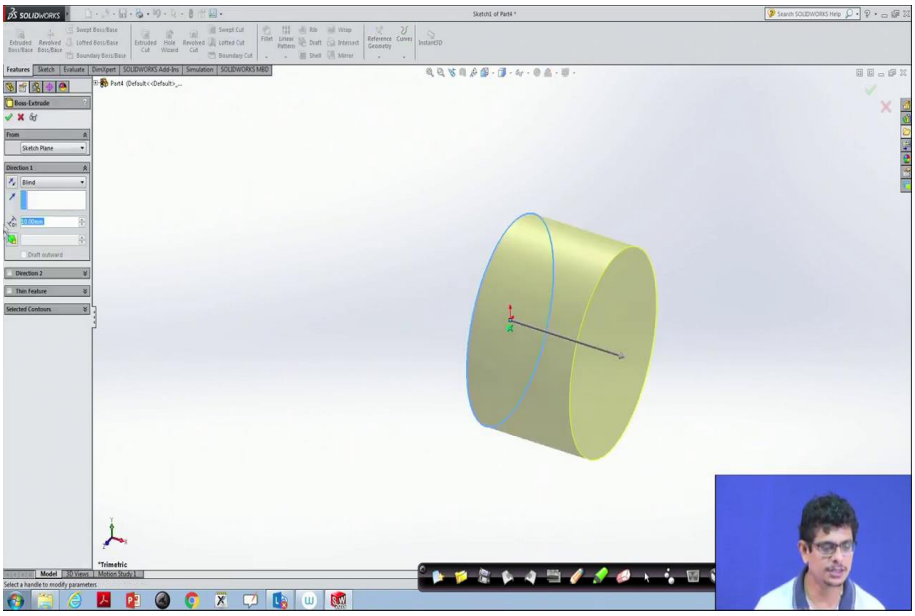

And that will have ahead of 10 mm into depth. So, what we will do is construct extruded boss it will be 10 mm head.

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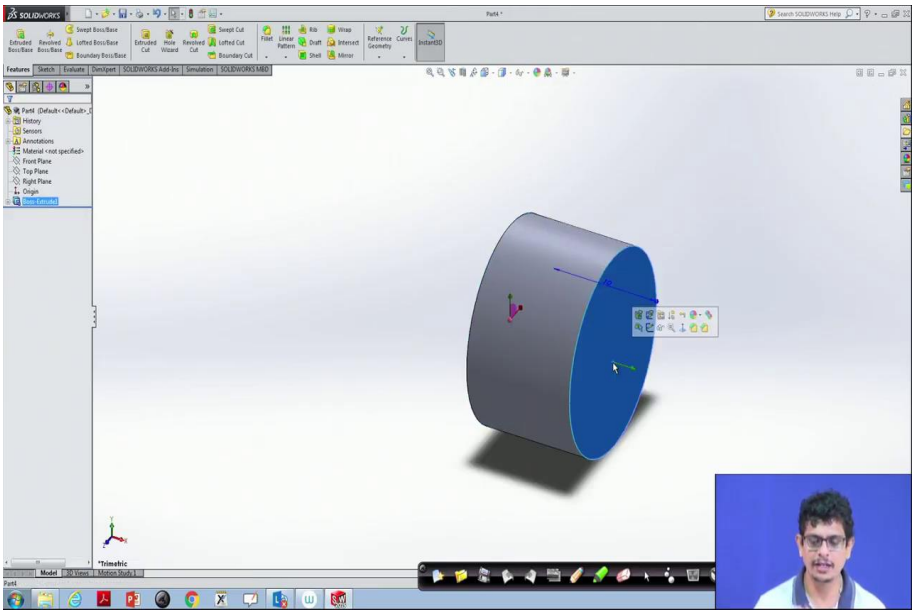

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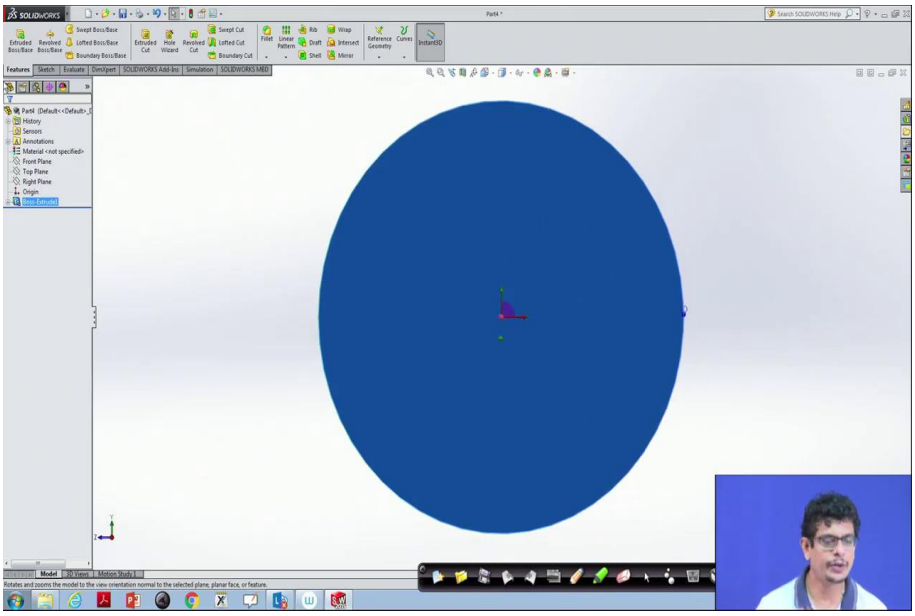

Now, on this normal to the surface we draw a hexagon, a hexagon 6 sides we will construct it.

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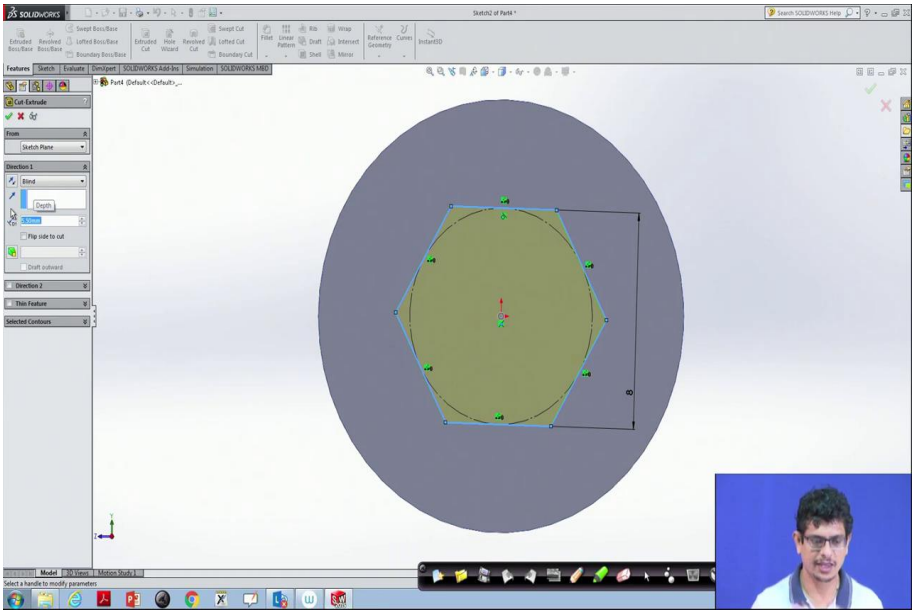

Then we use smart dimension, from here to here is supposed to be 8 mm. These are the standard numbers what you will get from data books. Whenever you manufacture a bolt or machine a bolt and try to prepare it there are some standard conventions like heights supposed to this much, length supposed to be this much and so on because of design calculation.

Like you calculate stresses, strains, how much stock you apply on that bolt, whether it can sustain, what kind of materials one has to use these kinds of things you will get it in machine design data books. So, based on those dimensions we are constructing this kind of bolt.

So, now we would like to have a hexagonal cut inside of that. So, let us pick this one, this one, all the 6 sides of the hexagon. Now, have an extruded cut. The depth of the cut we can have something like 5 mm, 5.5 mm.

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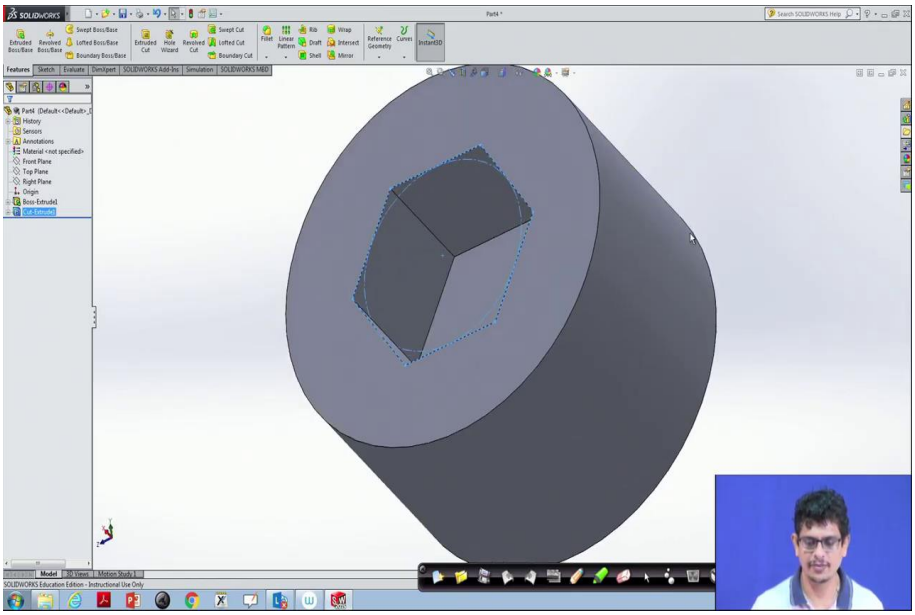

So, we have such kind of head portion. So, you will have l wrench kind of thing through which you will operate it.

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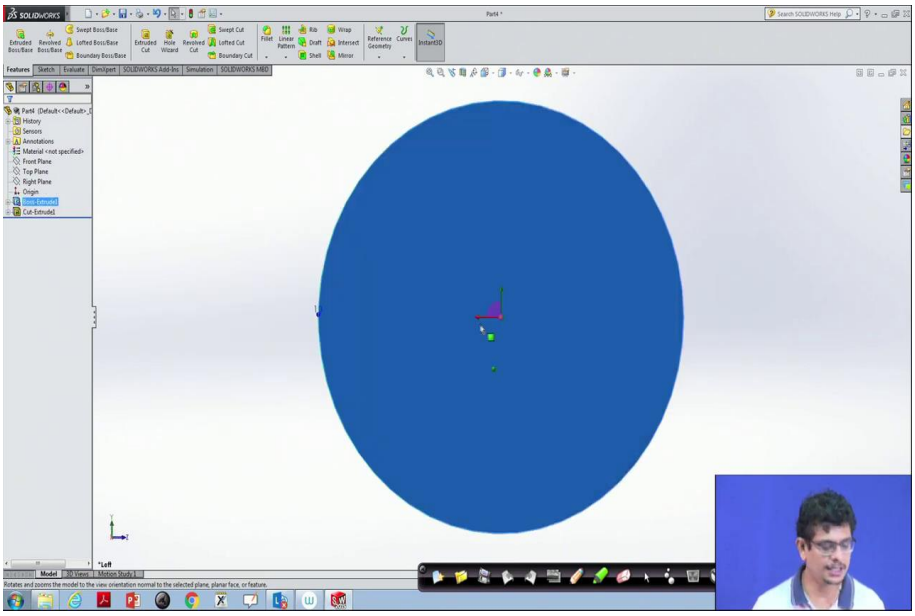

Now, here we would like to draw another circle, so that the stud portion of that bolt we can have it, which will be 25 mm in length and a circle of 10 mm diameter.

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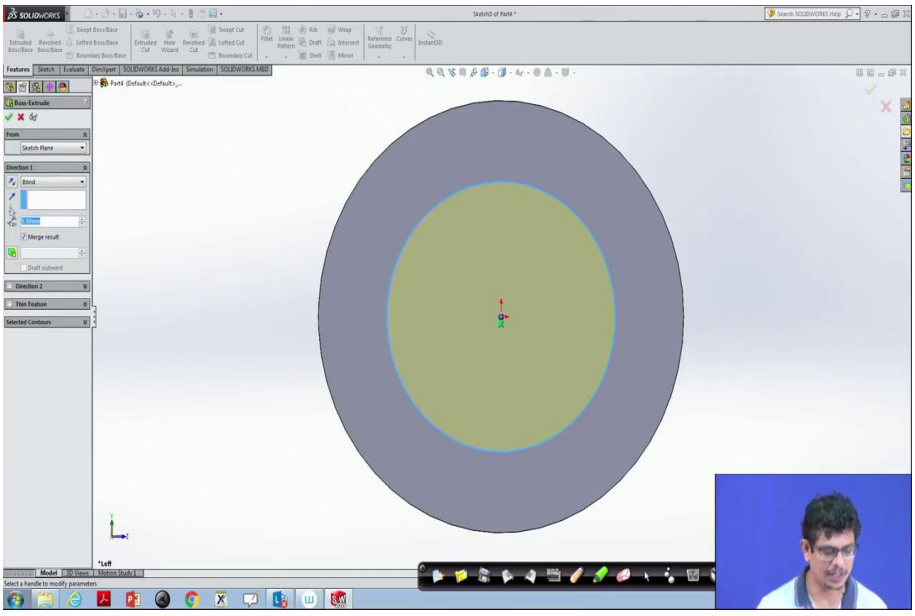

So, let us construct a 5 mm radius or 10 mm diameter. On this circle, we can extrude boss base up to 25 mm length.

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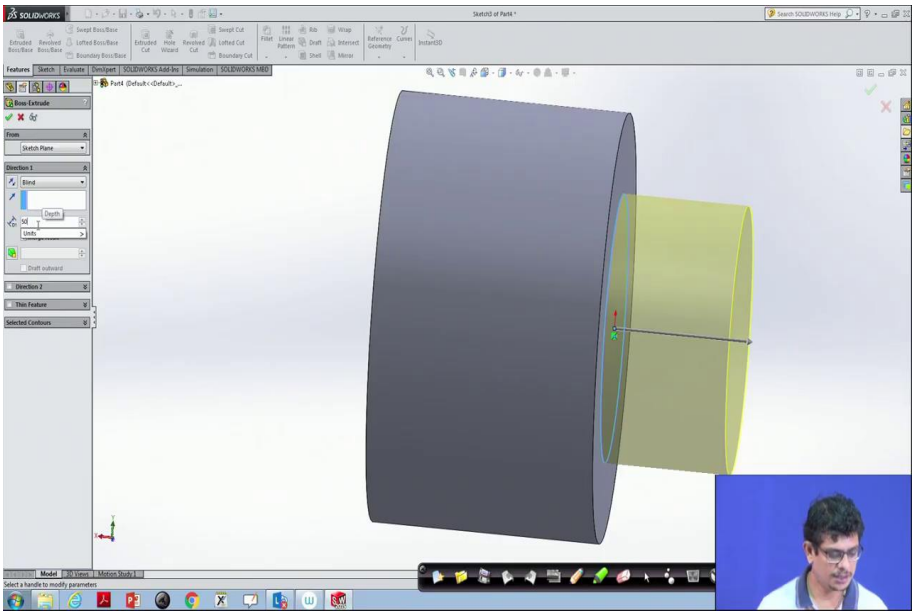

So, we have that bolt portion.

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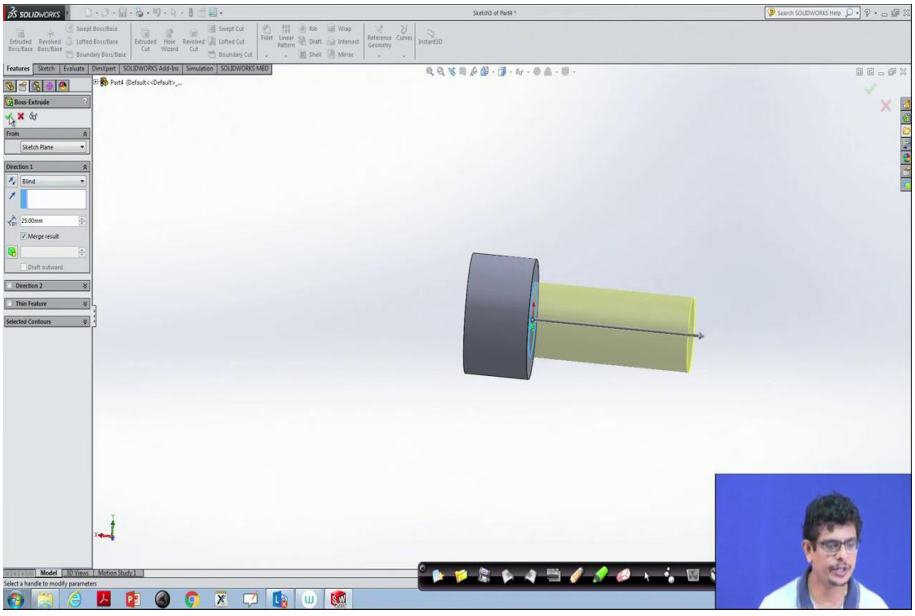

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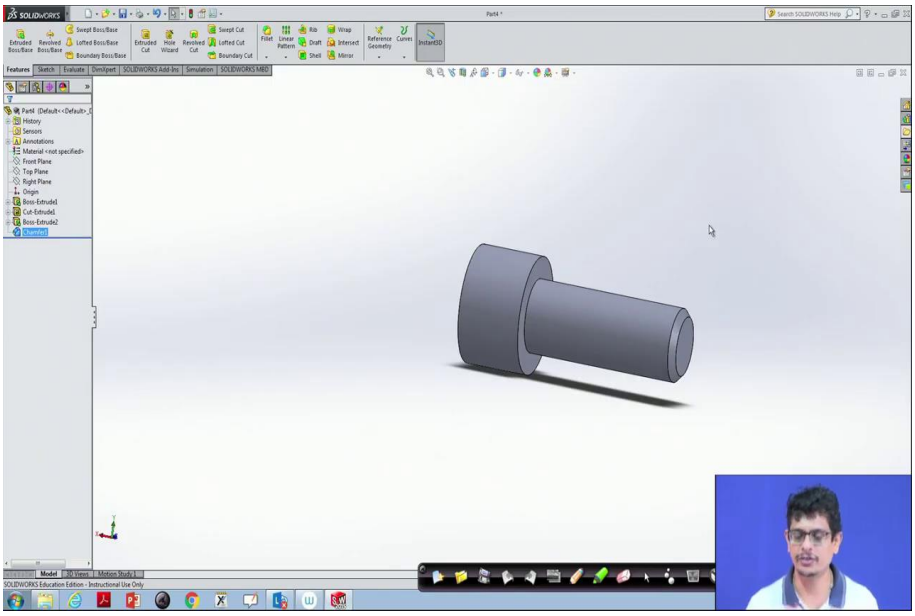

Now, usually, we give this the sharp corners we avoid it by going with chamfering this chamfering we can go with 1 mm with 45 degrees. So, the stud is done.

So, once this bolt is done what we can do is we make a helical path around this and a triangular kind of cut with a nice curvature and pass with match with this pitch and height thing and make a thread out of it.

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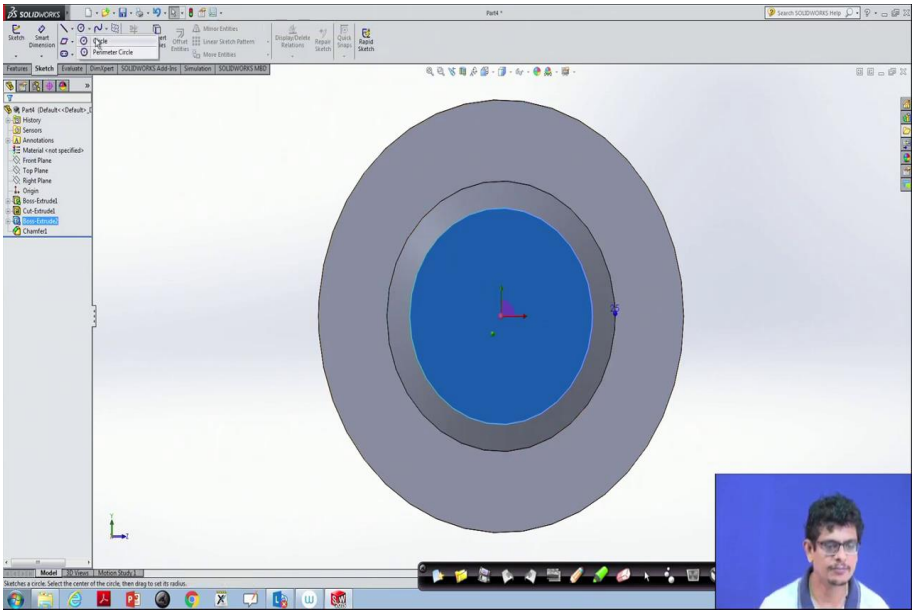

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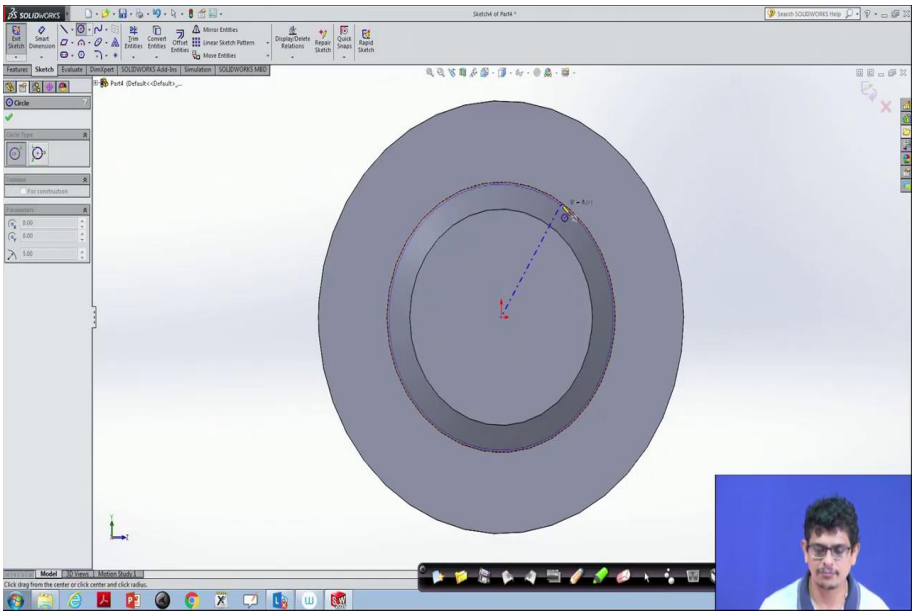

So, the first step is always normal to that construct one more circle matching with 5 mm radius.

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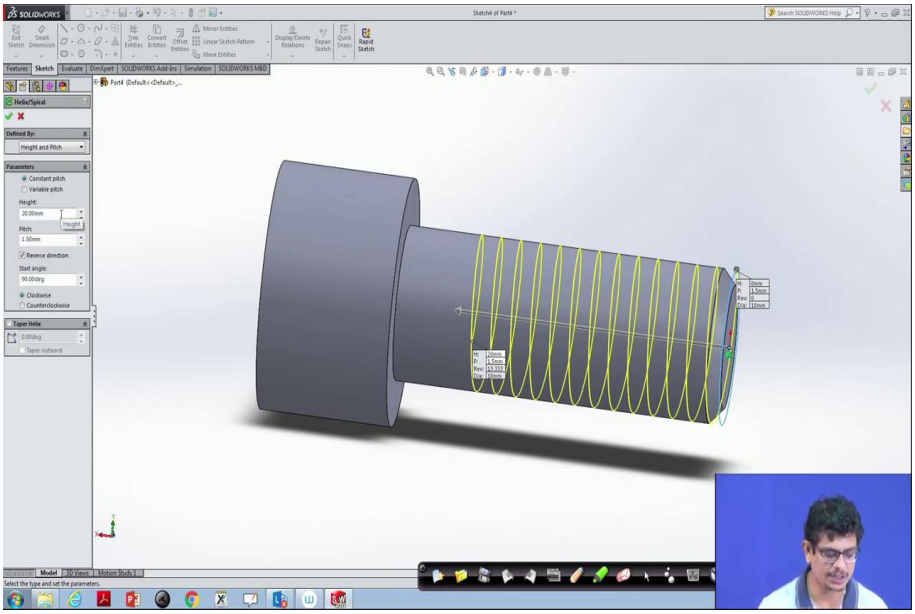

So, we have that 5 mm radius. Now, on that, we have to construct select this one go to insert on top of that curve helix. So, up to 20 mm we would like to have this thread and 1.5 mm pitch we can use. And it can begin at 90 degrees angle. So, that I can construct a triangle to make a tool to pass through that and click ok.

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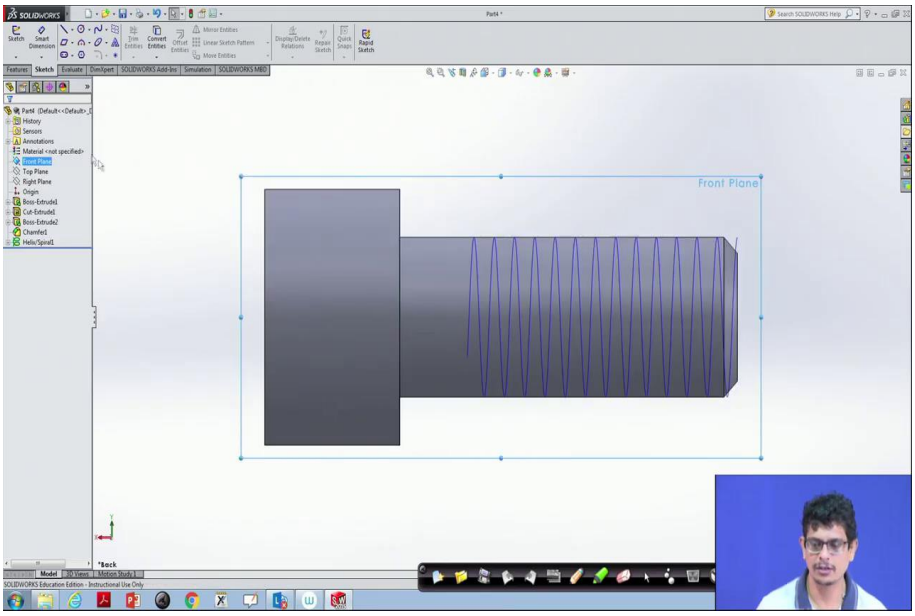

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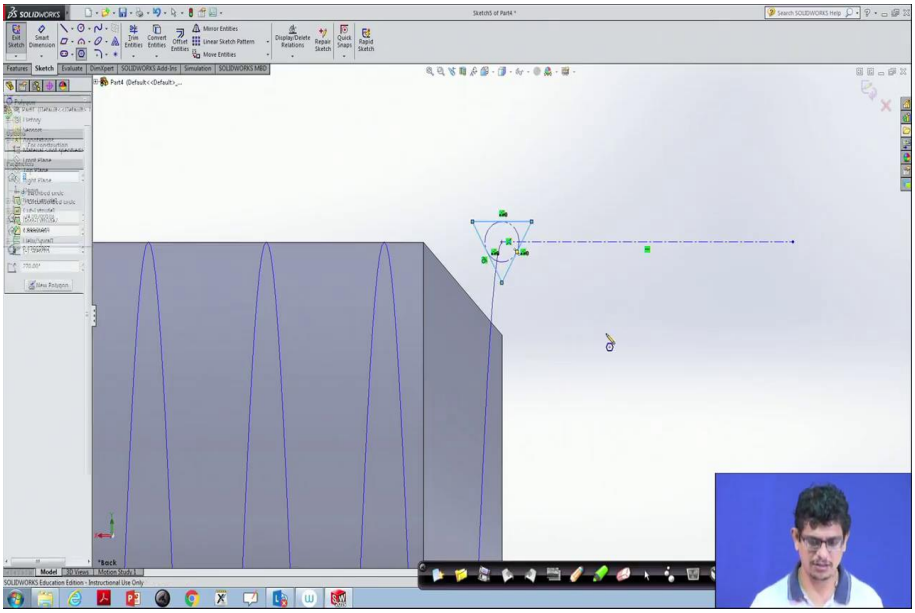

Once it is done go to frontal view, ok. Now, here zooming and here we make a centre line which passes through this point with 0 degrees. Let us look at the direction, ok.

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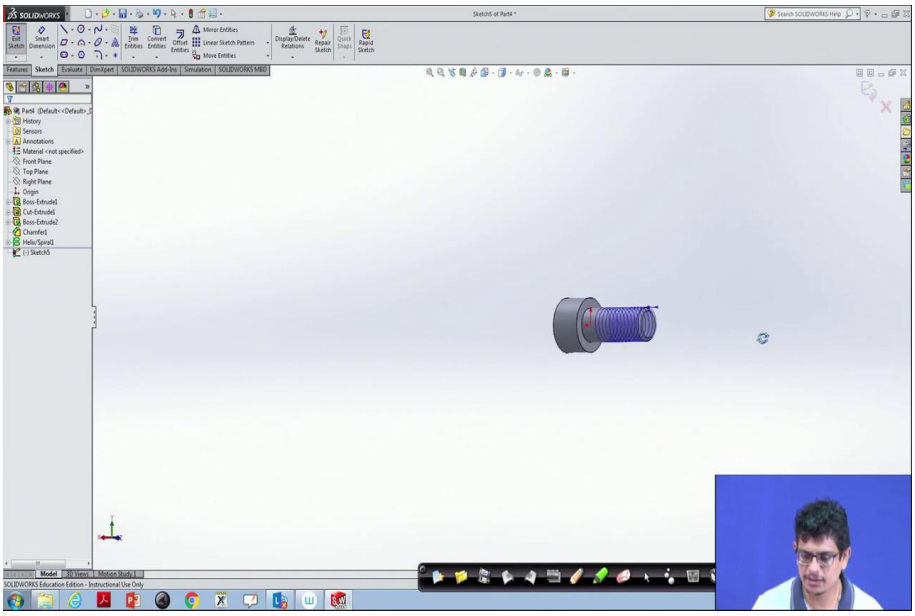

So, now again go to the frontal plane, ok, on that let us consider a triangle. So, use the triangle, tool bit always be in this downward direction, constructed remove this one, zoom in.

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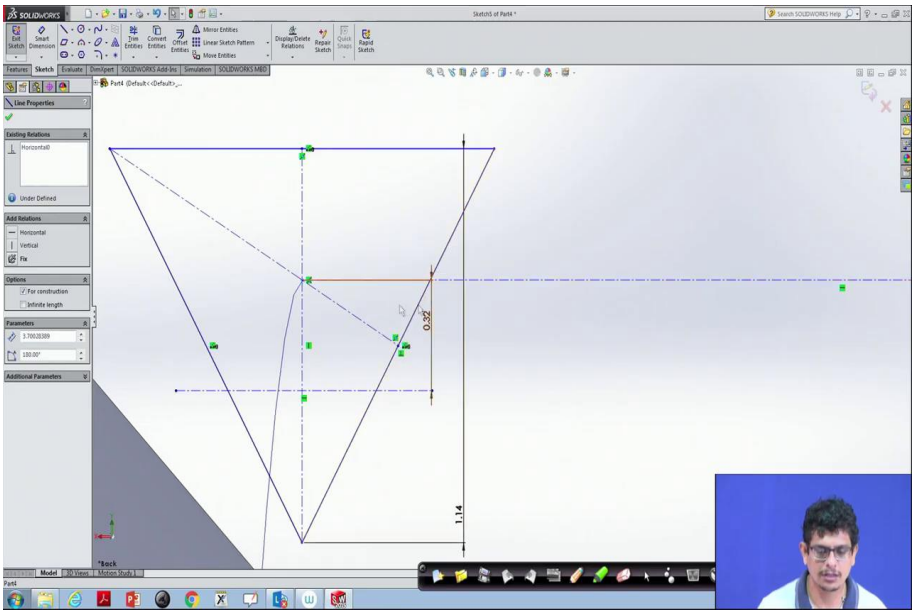

Now, this height from this edge to this one supposed to 1.14 the tool bit, so use smart dimension pick this one, pick that point, make sure that that will be 1.14 mm. And second thing let us have another centre line join the mid one and that is passing through this point, fine.

Now, we require something like an arc around that because we do not want this kind of sharp or tool bit, something like a blunted one we would like to have it so that a smooth curve we will be in a

portion to do. For that purpose what we will do? We construct one more centre line, ok because we require the centre of the curve, so this is the point.

So, what we will do is construct a centre line here in such a way that this line from this centroid will be of 0.32 mm.

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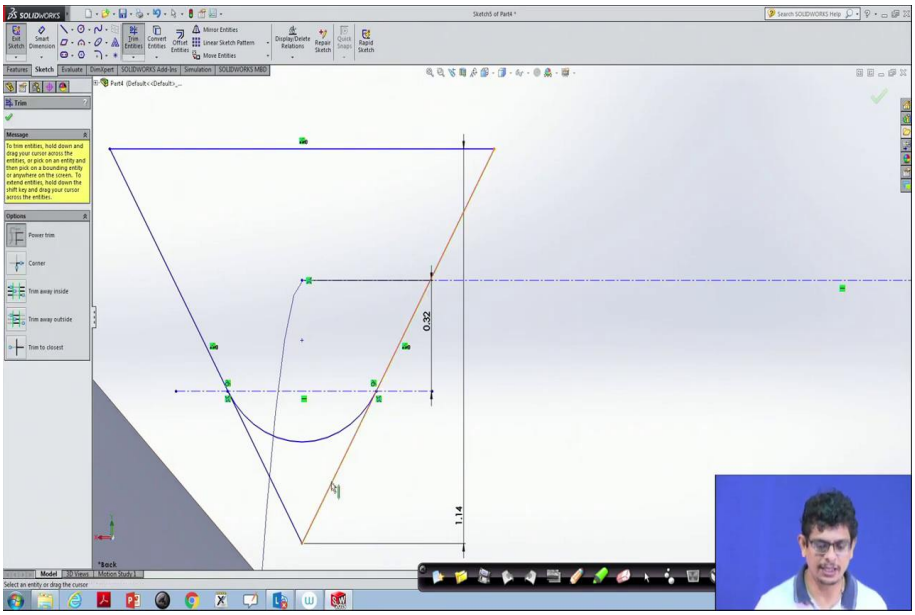

Once it is done we remove this line, ok. We can remove that line and also this one also we can remove. These are the points through which we would like to pass an arc. So, a 3 point arc pick that point, the second point and move it.

Now, this supposed to be tangent to this line. So, what we will do? Pick this one, pick this one, they are tangent to each other. Once it is done, ok. Similarly, pick this line they are tangent with each other. Once it is done, ok.

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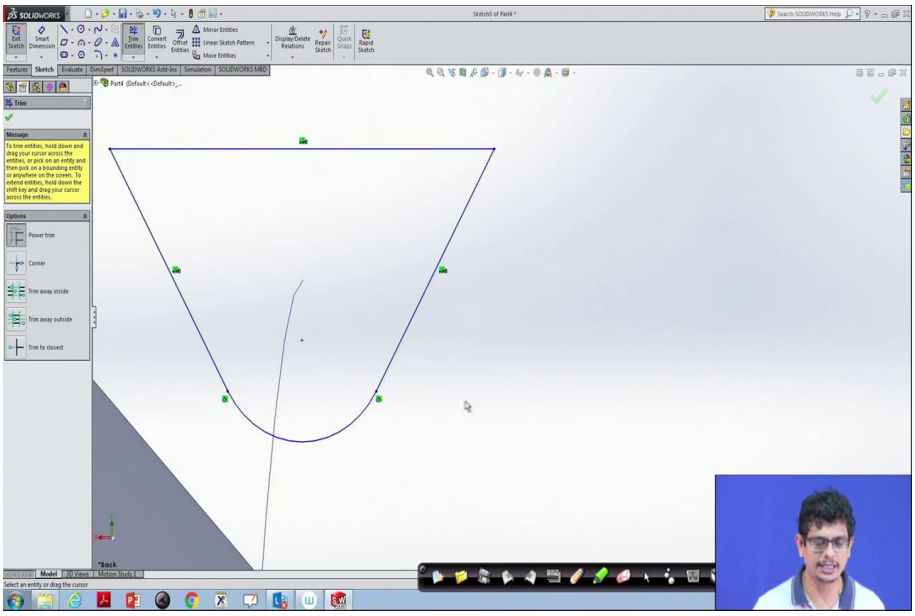

Now, what we will do? Go to trim entities, remove this part, remove this part and also we do not require this one also, this one. Similarly, we do not require these lines, then click ok.

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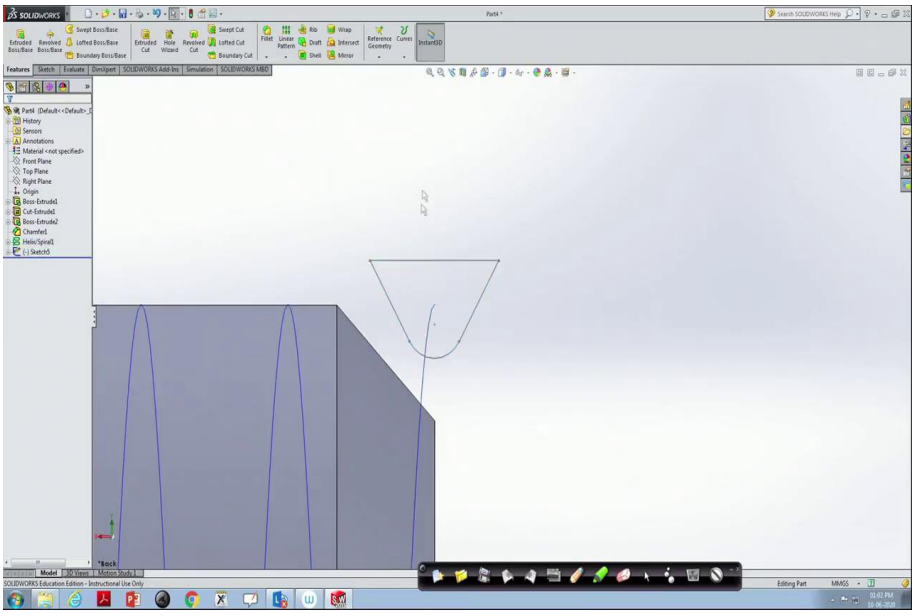

So, after coming out from sketch mode, we have to select this entire object.

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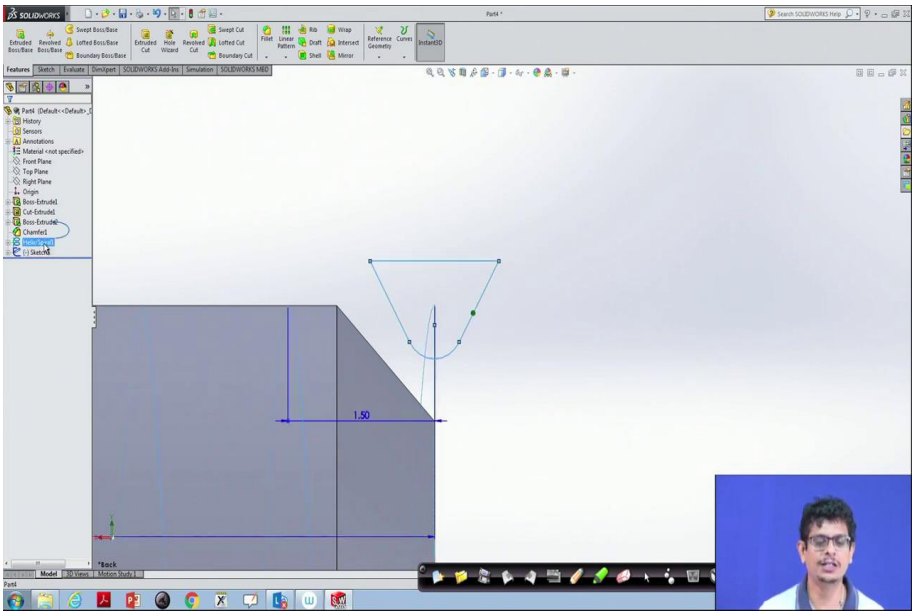

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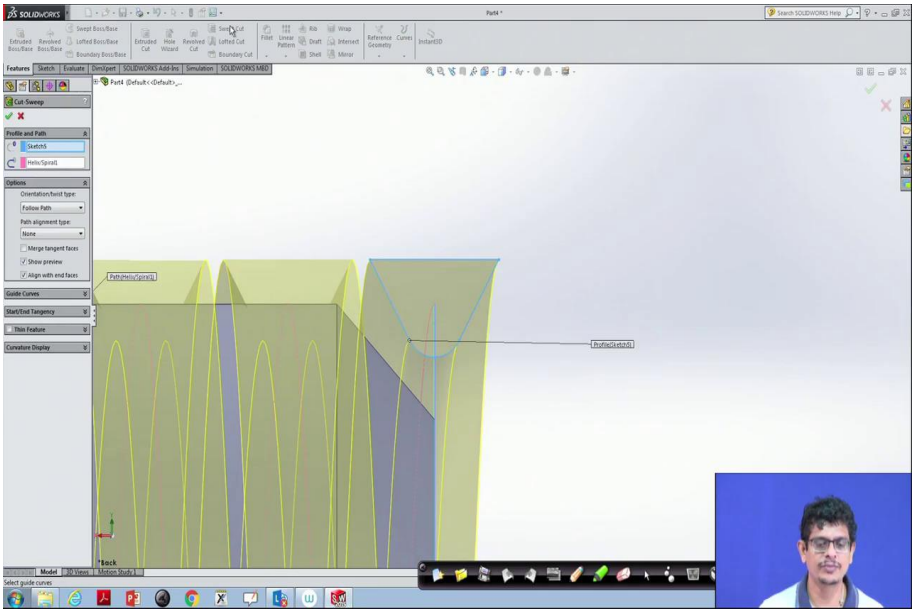

Then pick the helix path also, control, go to features, go to swept cut, then it shows you this entire thing something like revolving kind of portion, and then click ok. Then, you will have a thread in this way.

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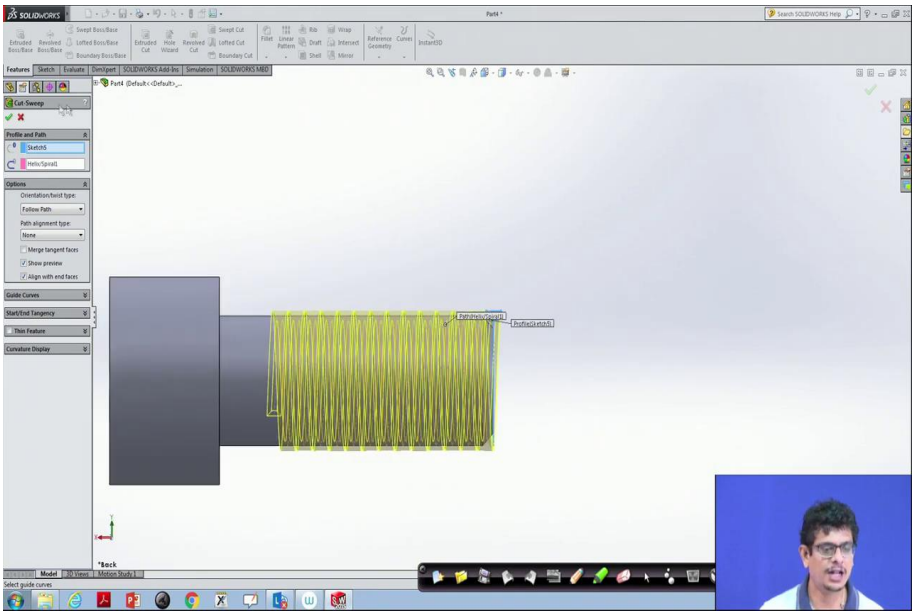

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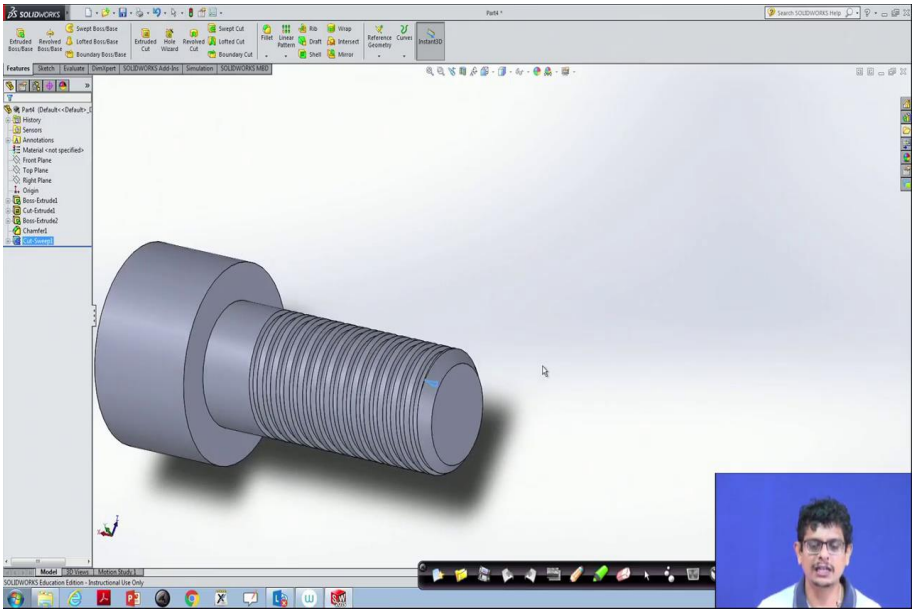

There are other aspects like aesthetic aspects we always have, how to smoothen this portion, whether we can have some other kind of triangular cut and other things.

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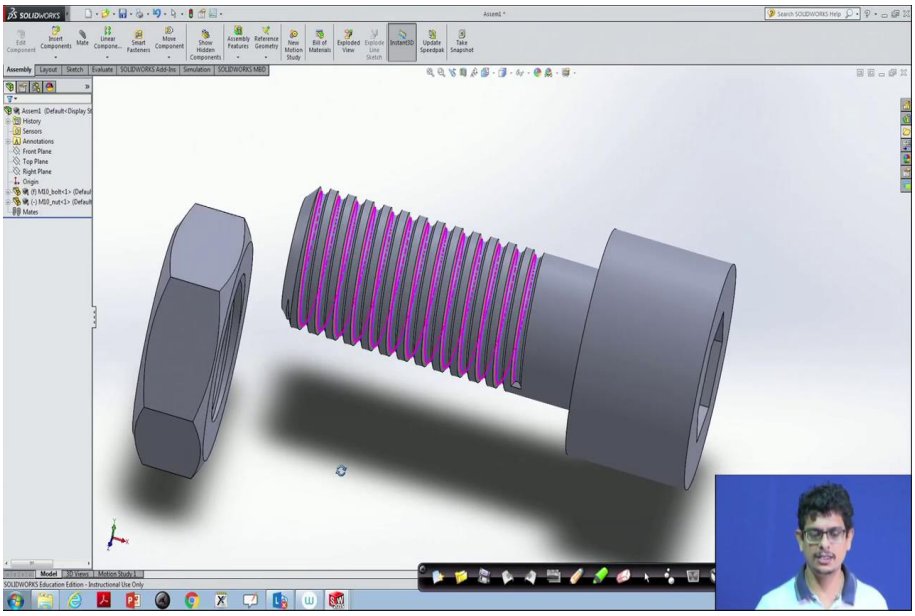

Let us look at different views of this bolt. So, the first thing what we will do is look at the isometric view. If it is a construction one we can remove that. So, this is the way isometric view looks like.

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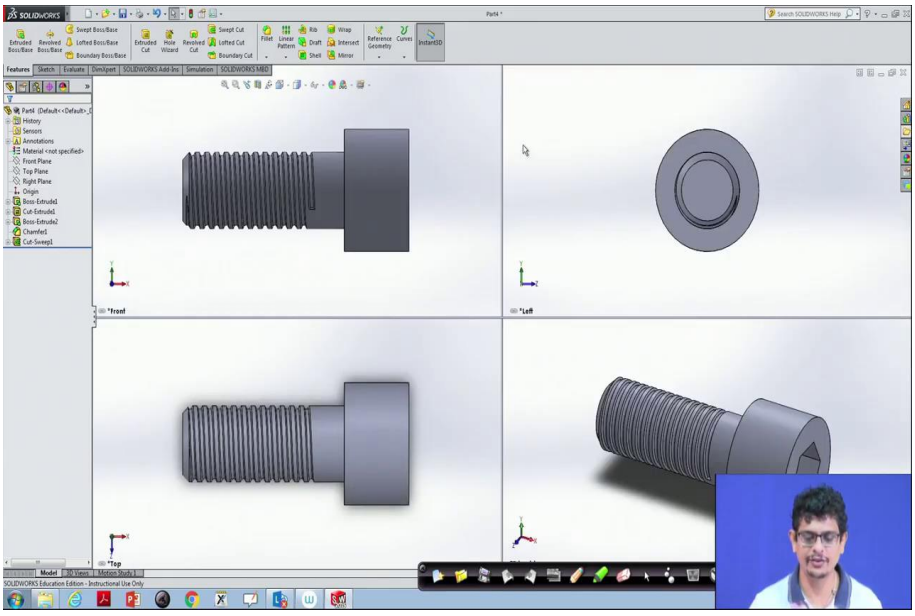

Now, look at 3 different views, something like the front view, the top view, the left side view and the isometric view.

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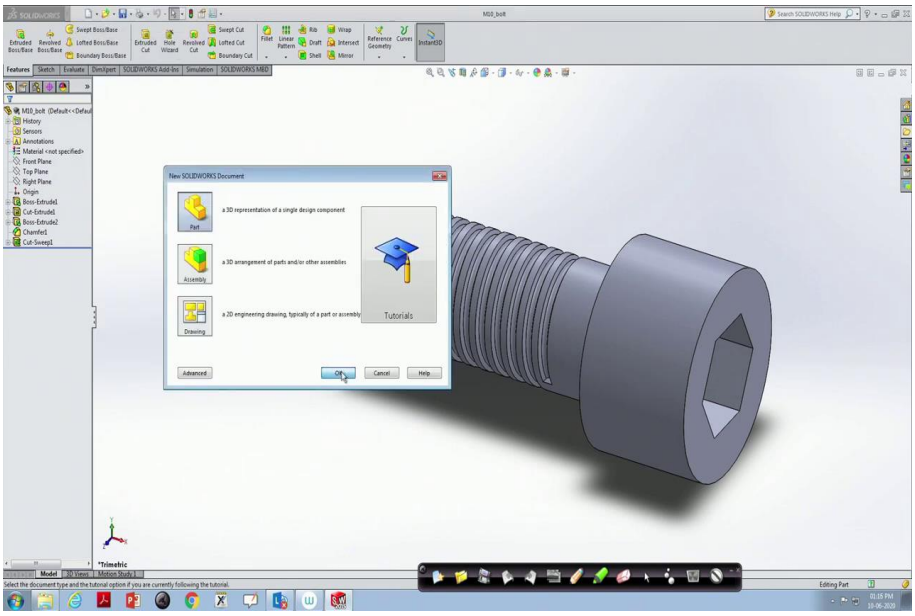

Now, we will construct a nut for this bolt. For that purpose what we do is go to new, open a part, ok.

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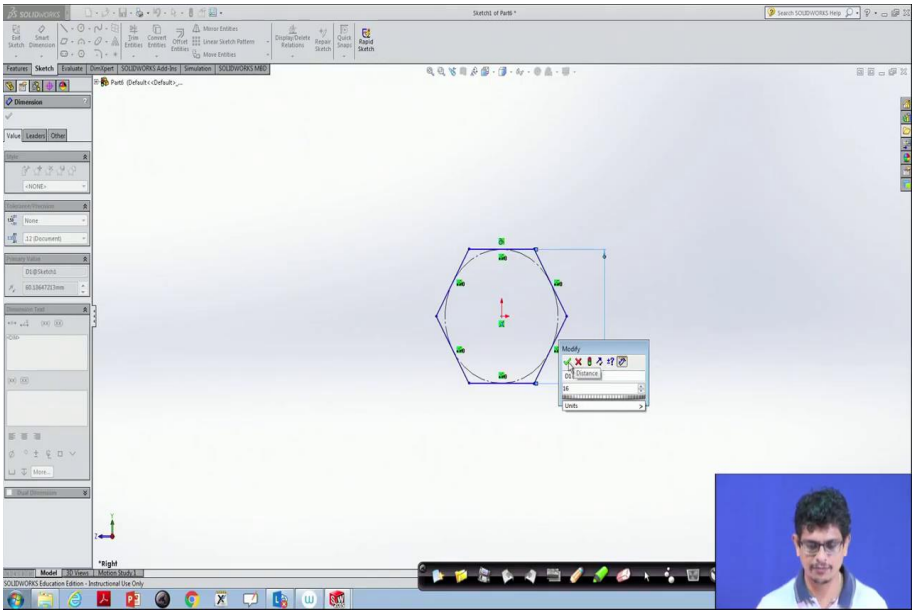

Now, on the right plane normal to that we want to have a hexagonal nut. So, for that purpose, we go to sketch, pick hexagon 6 units from here draw it. Use smart dimensions from here to here. We use same the head portion of that bolt whatever that height, we have the same 16 mm, we use it.

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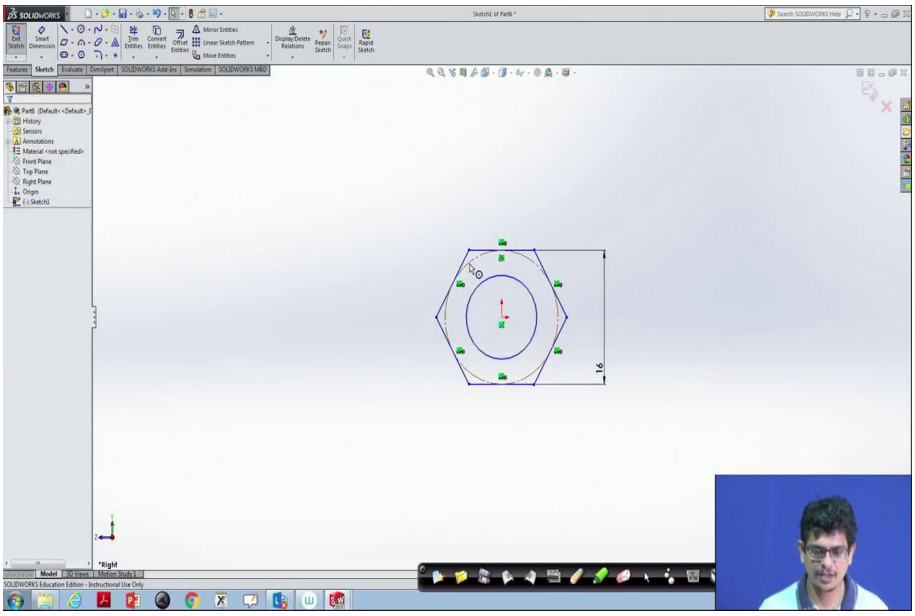

And we have to make a thread over that internal portion and we know that the stud has a diameter of 10 mm. So, what we make is a circular hole of 10 mm we make it, internally circle 10 mm diameter or 5 mm radius click, ok.

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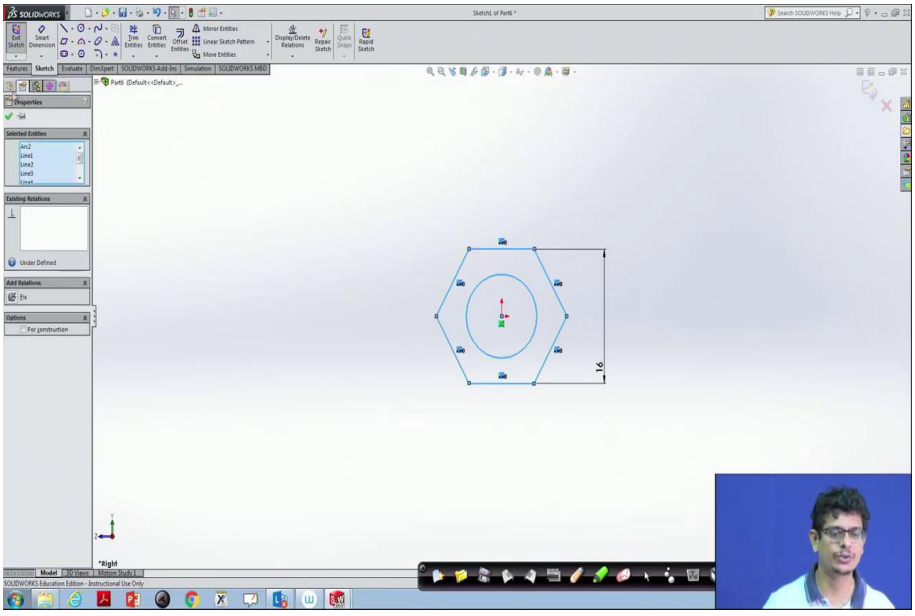

Now, we do not want this inscribed circle. Pick this one the lines by holding the control button.

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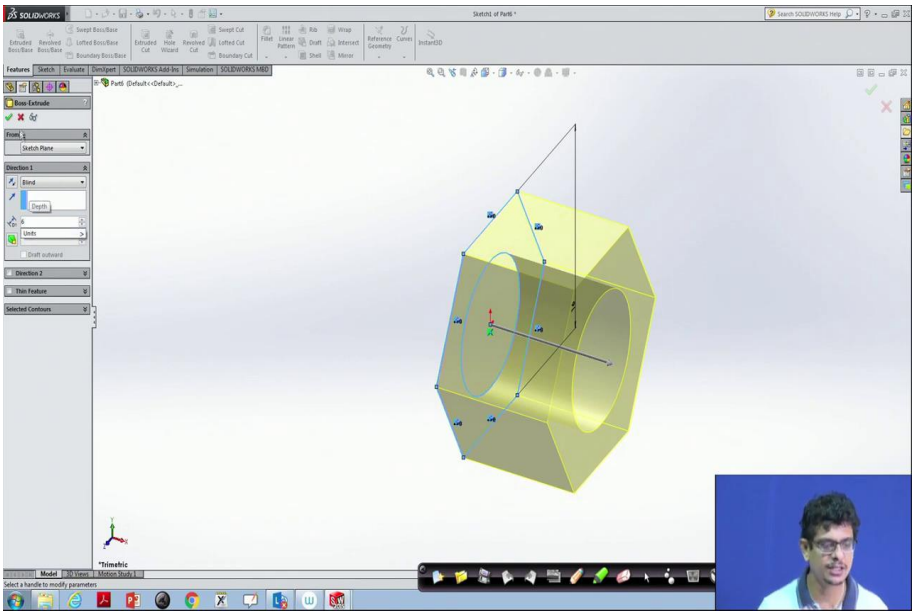

Now, this is the surface what we have picked. Go to features, extrude boss base. So, it creates a nut kind of a portion. So, let us use 6 mm as the width to construct this nut.

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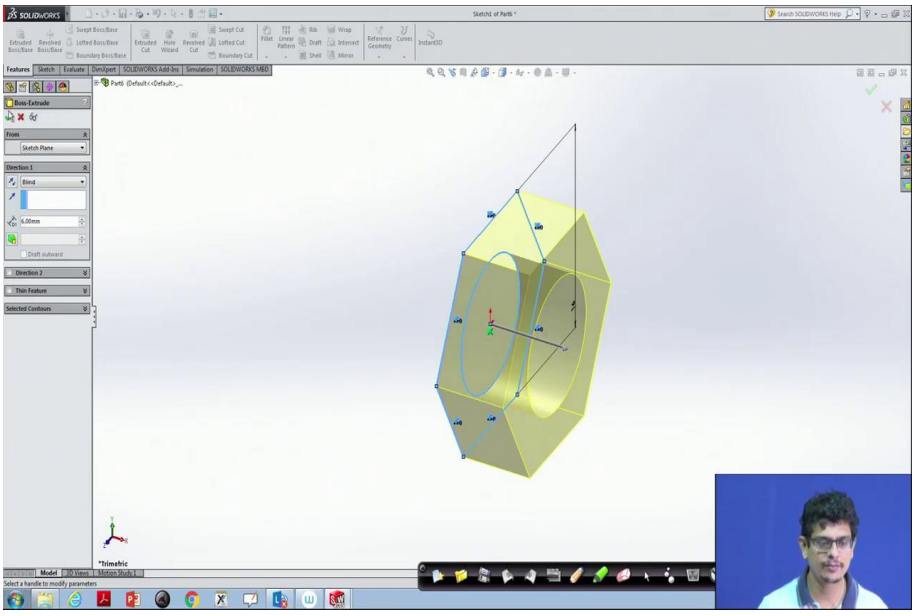

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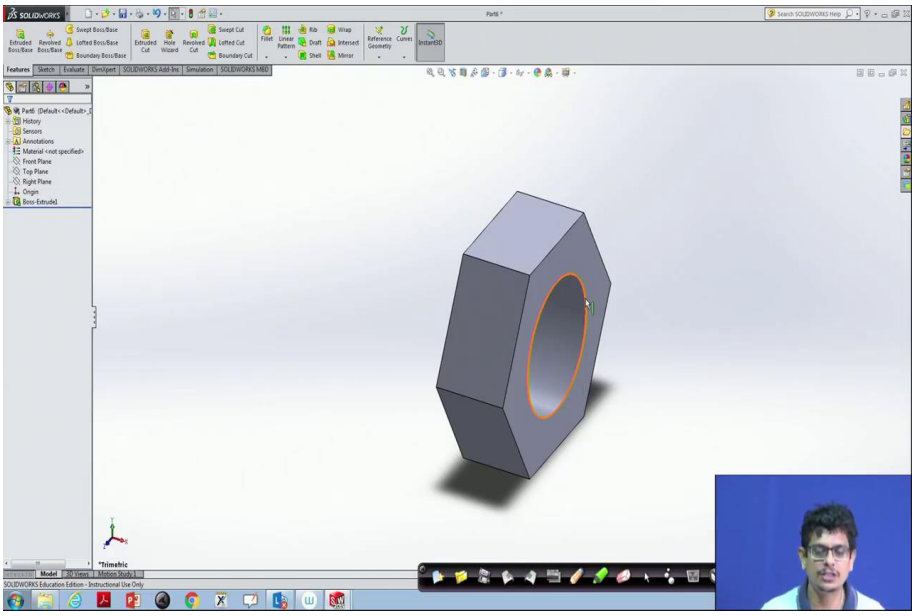

Now, we have that. We are not having this, I mean we would like to give something like a chamfer we can really construct that and so on, but now we will mainly focus on constructing the internal threat portion of that.

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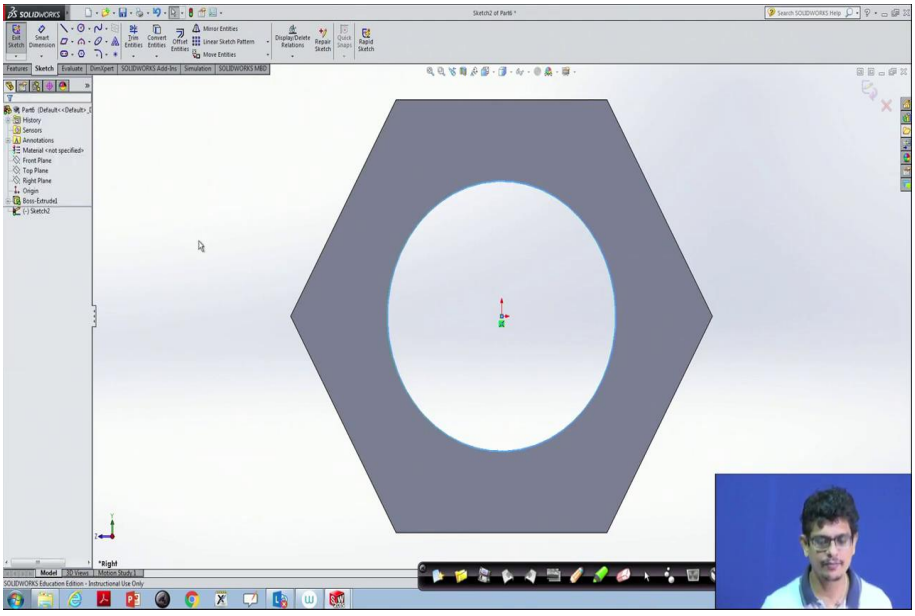

So, the first thing what we have to do is to go to the frontal right plane normal to that draw a circle, the first step for any helix construction is making a circle 5 mm, click ok.

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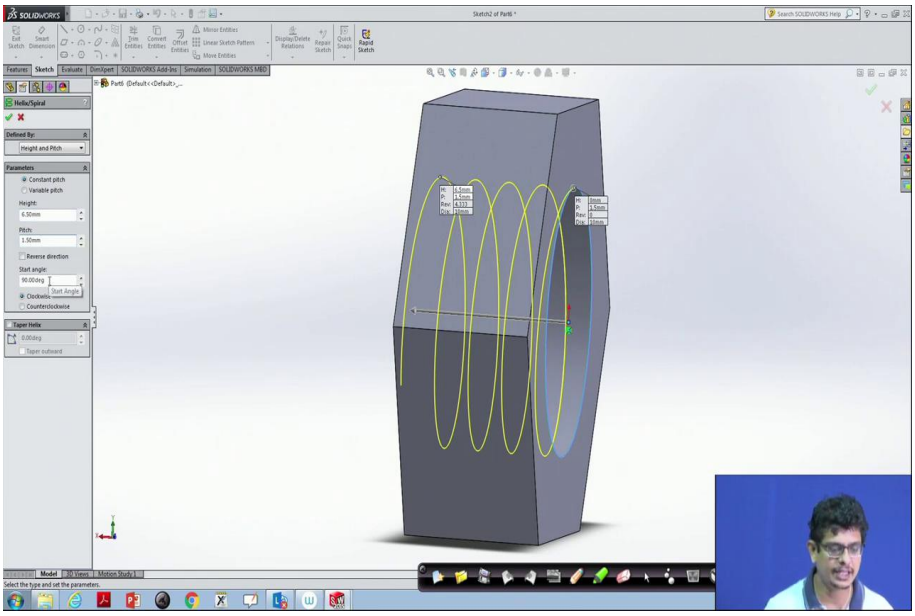

Then, visualize it in this way, we have the, pick that circle, then go to insert curve, helix spiral. So, we have the spiral curve where we can give height up to 6 mm. So, up to that level, we can have a cut. Maybe just make it 6.5, so that it smoothly comes out as a cut.

Pitch as usual 1.5 mm we are giving. And it is in the counterclockwise direction, and direction also fine clicks ok. So, internally we have this circle. And this helix begins from 90 degrees angle. Let us recheck it. So, go to edit feature it is 90 mm, 90 degrees, so click ok.

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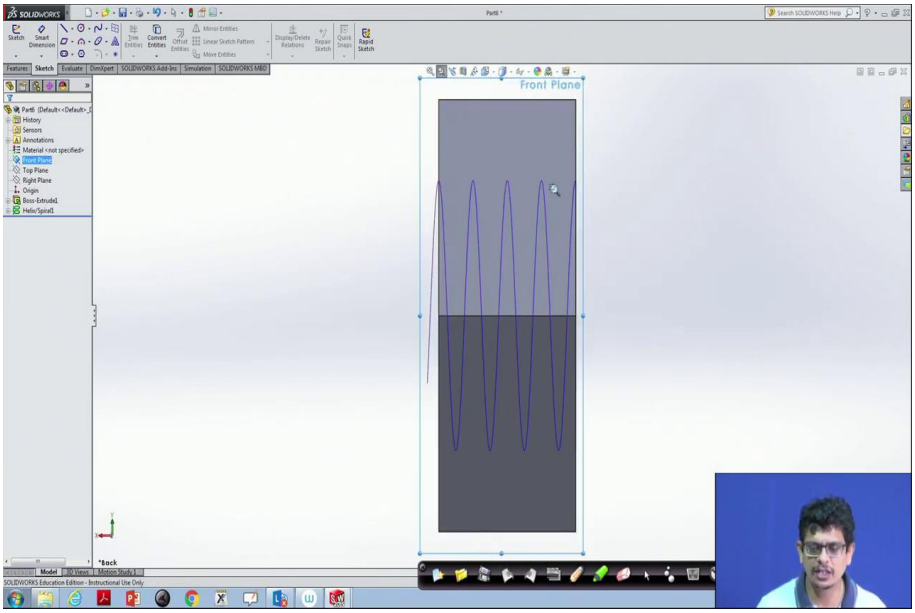

Now, go to the front plane normal to that. So, here we want to construct a triangular tool.

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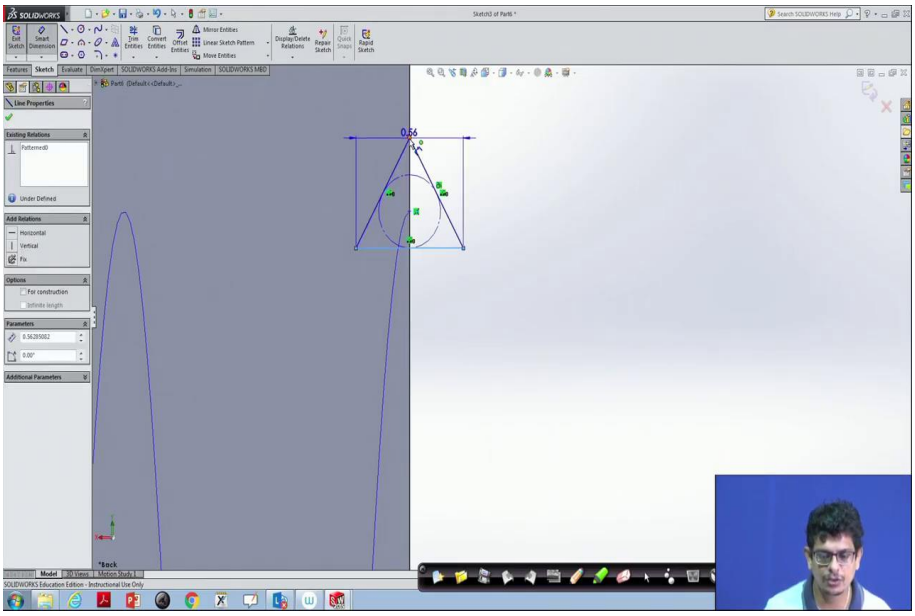

So, this is the portion where we would like to have, for that purpose we go to polygon 3 sides pick this one as the centre. Now, reverse this triangular tool cut because now we want to remove the top portion, so the triangle will be on the top side and make it in that way.

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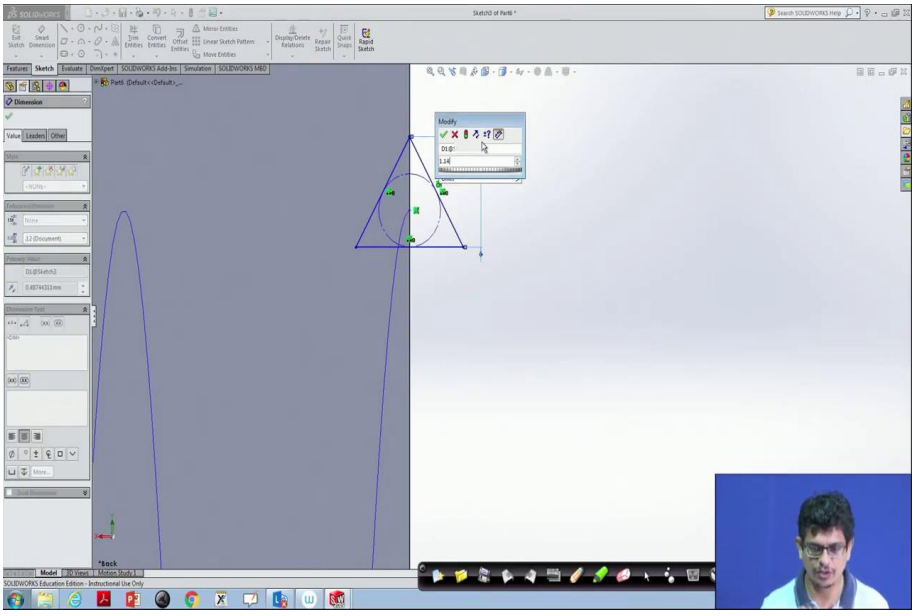

Now, this supposed to be 1.1 mm. This height to that point supposed to be 1.14 mm, so we have the triangle.

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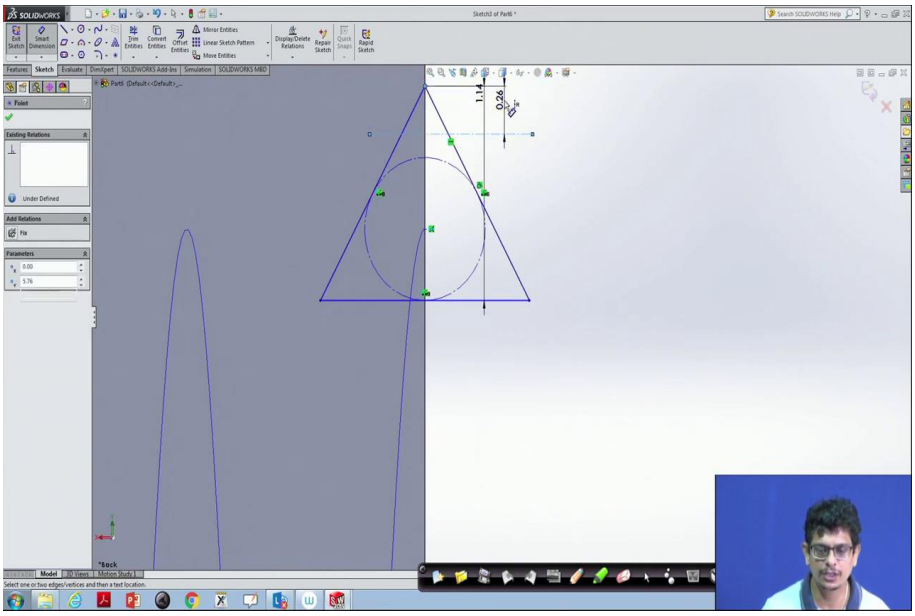

Then, we require 0.32 mm. So, let us consider a centre line in this way and use smart dimension from here to that first point it is 0.32 mm.

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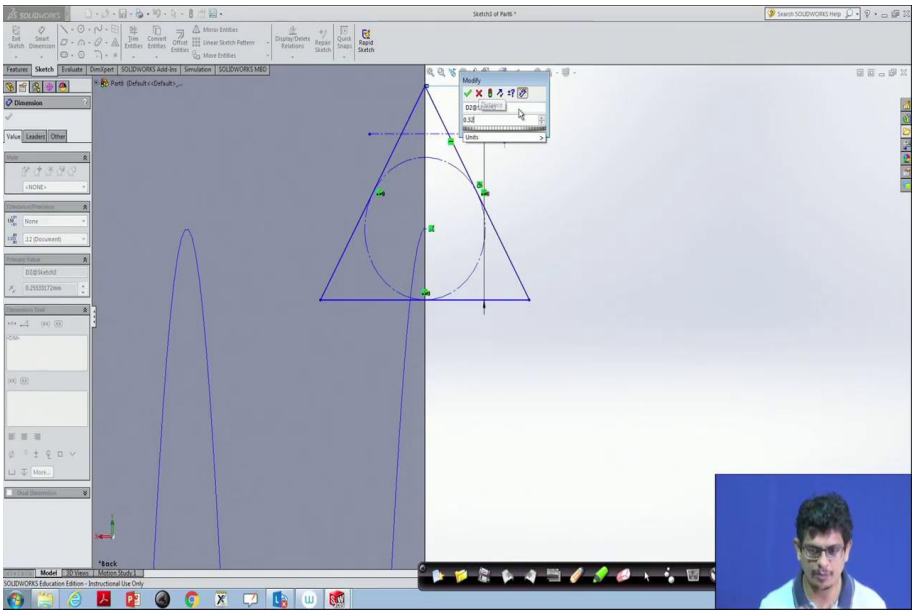

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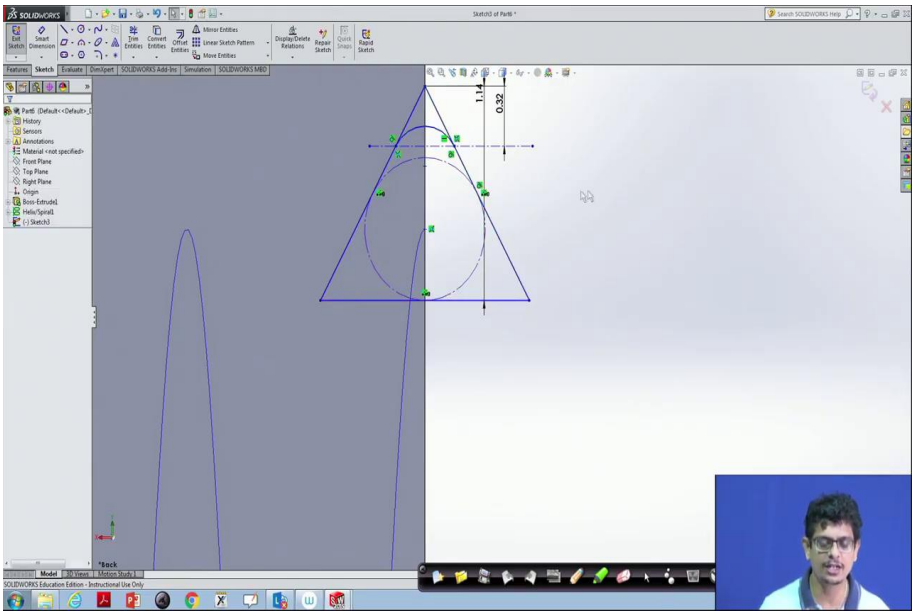

Once it is done, we make an arc 3 point arc from that point to that point with a certain radius.

Now, pick this one on that line make it tangent. Similarly, pick this curve line make it tangent. So, when you are picking the things you have to make keep hold of that control button.

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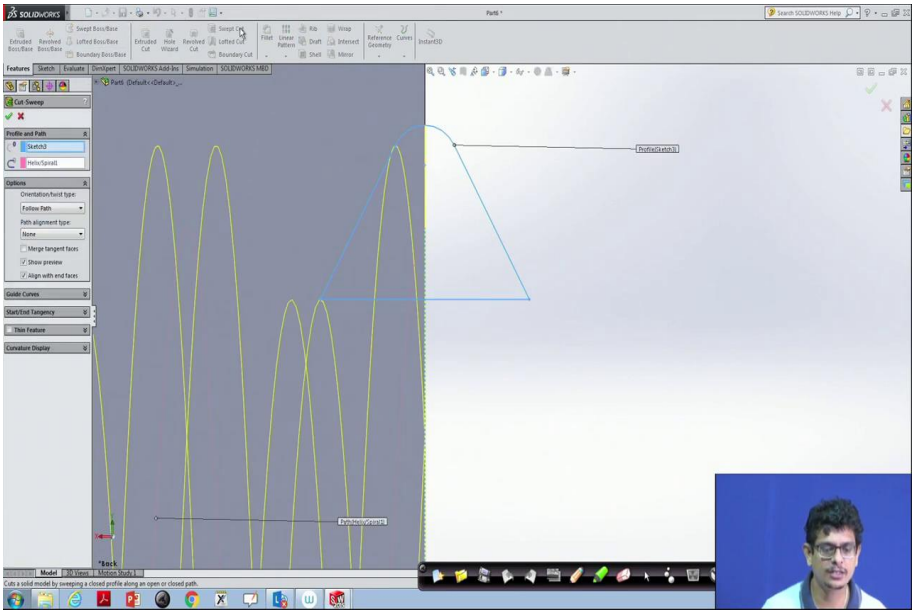

So, once it is done this triangular construction, we can internally remove this circle and we can use trim entities to remove that portion and this portion also. And we can remove this one these tiny portions. So, once it is done, click ok.

Now, we have this entire object first come out of sketch mode, then pick click hold select, all these buttons by clicking hold. Now, sketch 3 is selected, then control button, hold it, click helix, and also sketch, then go to features, use swept cut, click ok.

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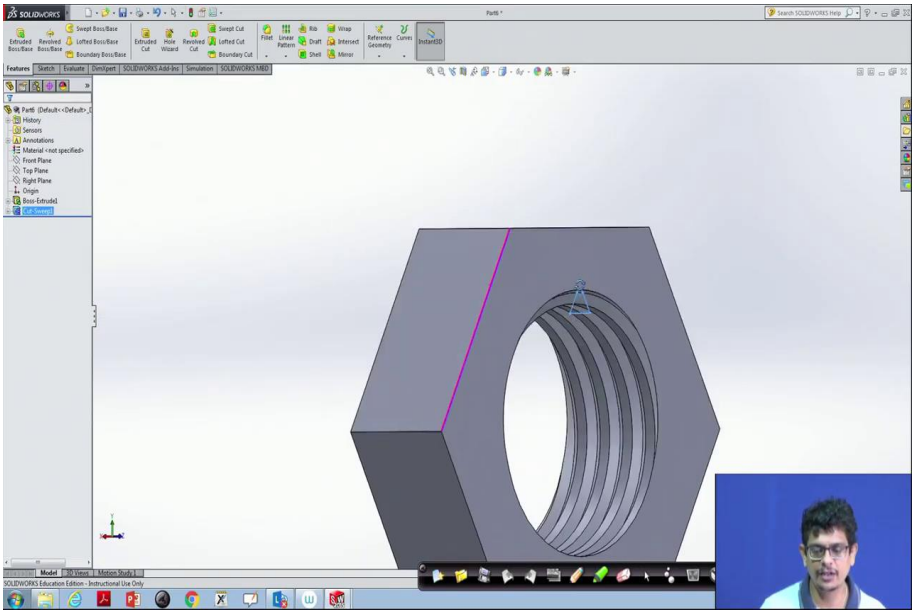

So, internally if you are seeing we have those threads in that nut. This is the way we construct it.

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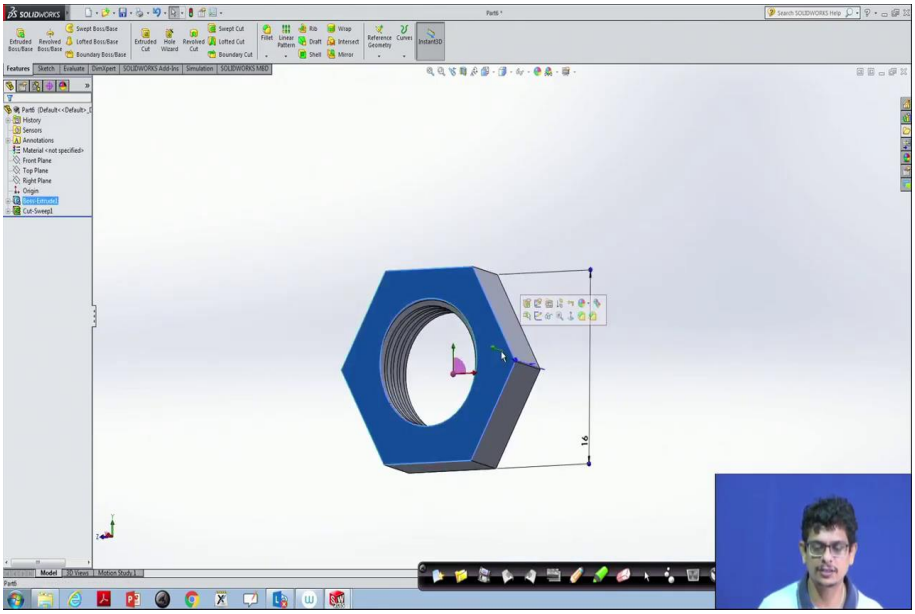

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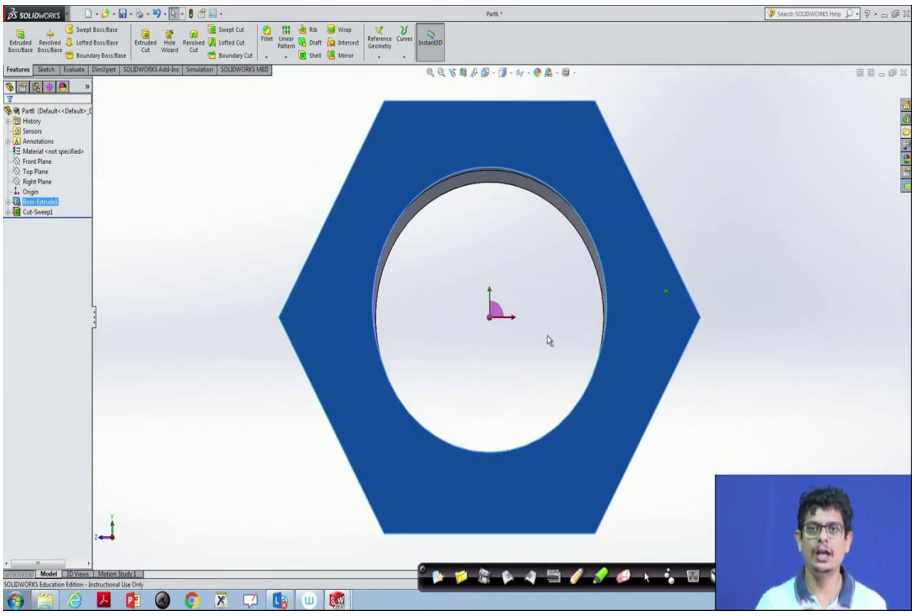

Sometimes what we do is on these nuts, we would like to have a little bit chamfer on the outer surface, we do not want a flat surface.

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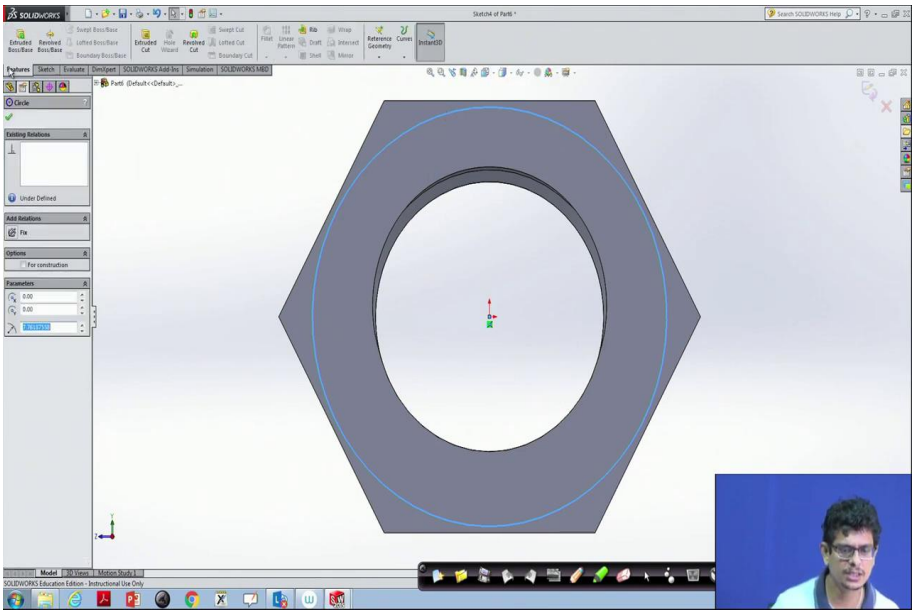

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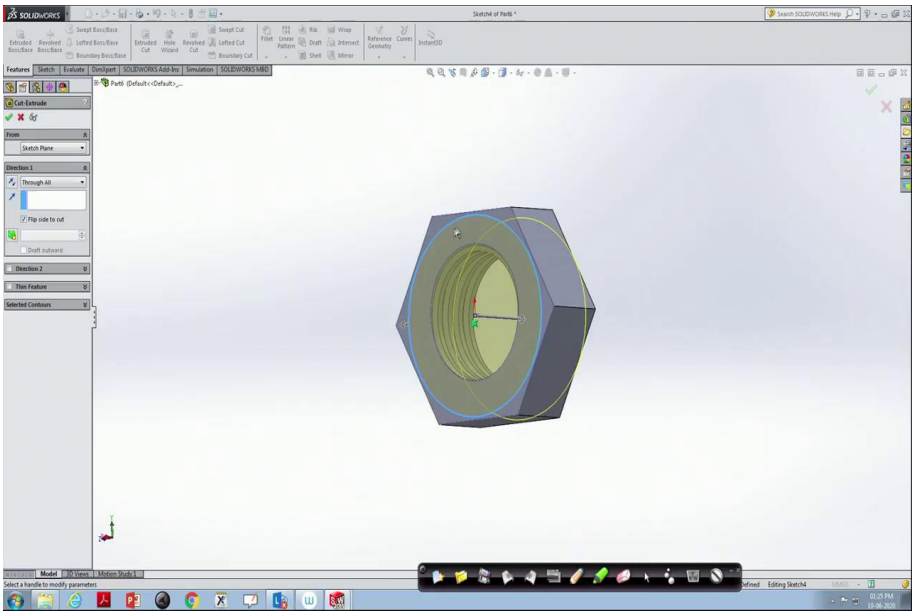

For that purpose, what we do is we go draw a circle of arbitrary size and pick that one go-to feature. After drawing that circle we go to features use extrude cut, in that extrude cut the direction we pick through all, then flip side to cut, so outside we are going to cut and we give a tapered angle like 60 degrees outwards, then click ok.

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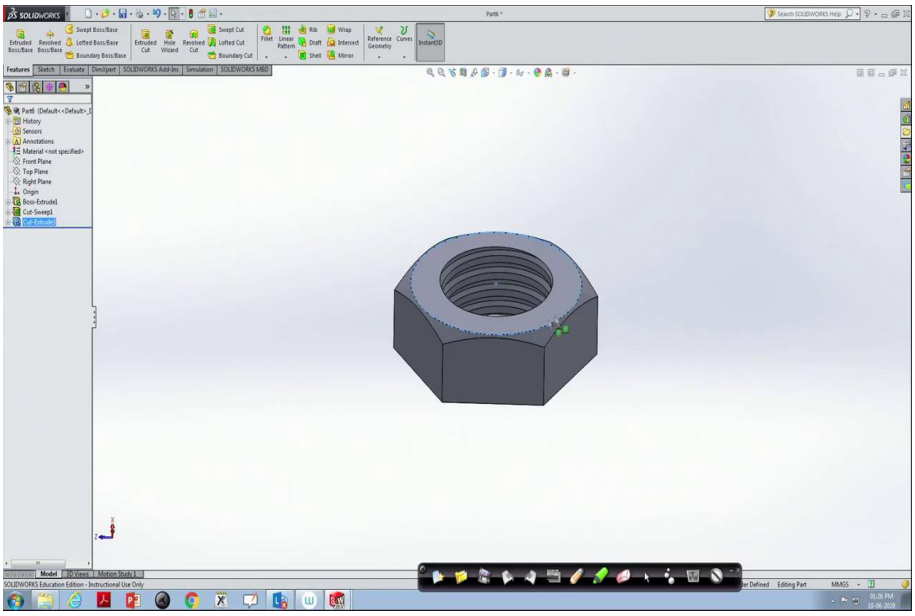

So, now you can see the top side is flat one, but these are slightly tapered. Similarly, we construct for this one also.

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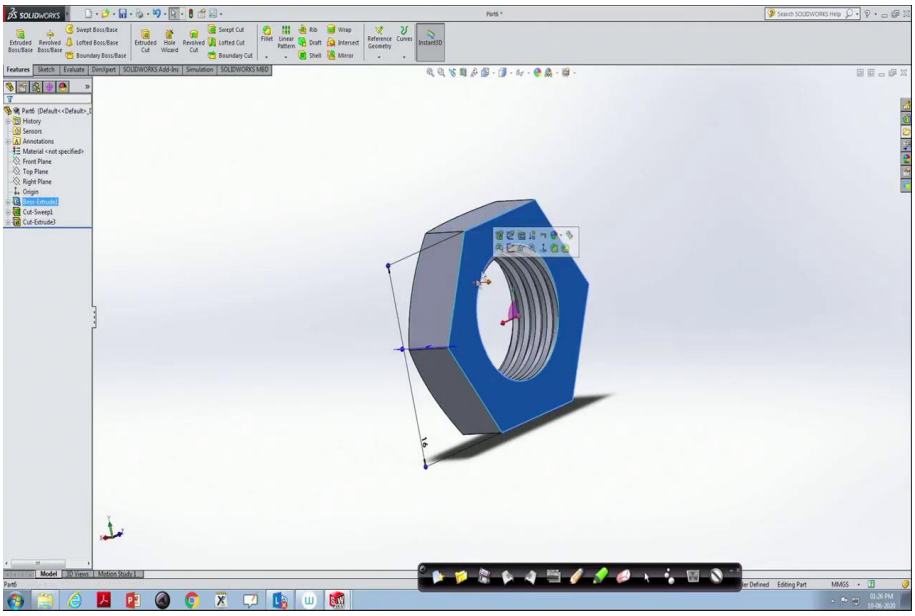

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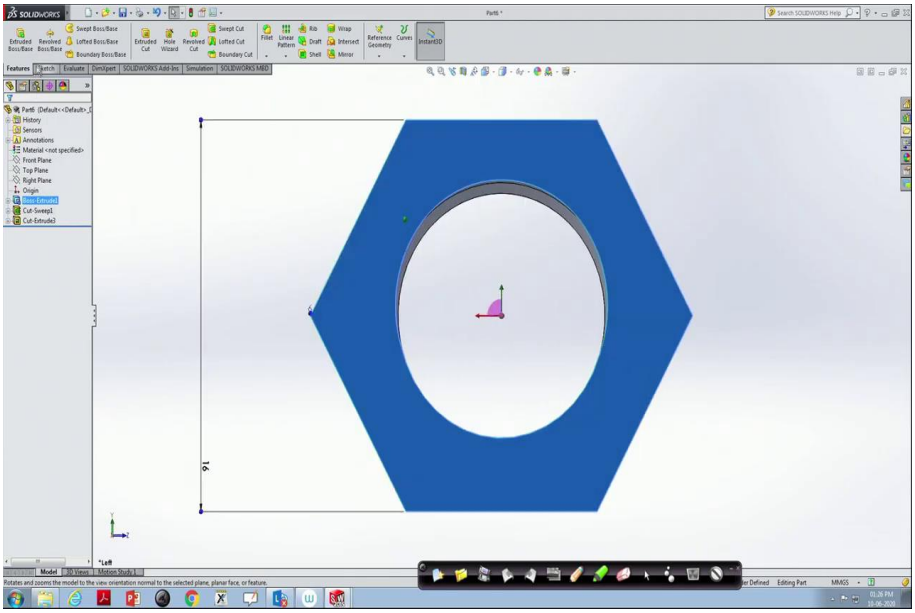

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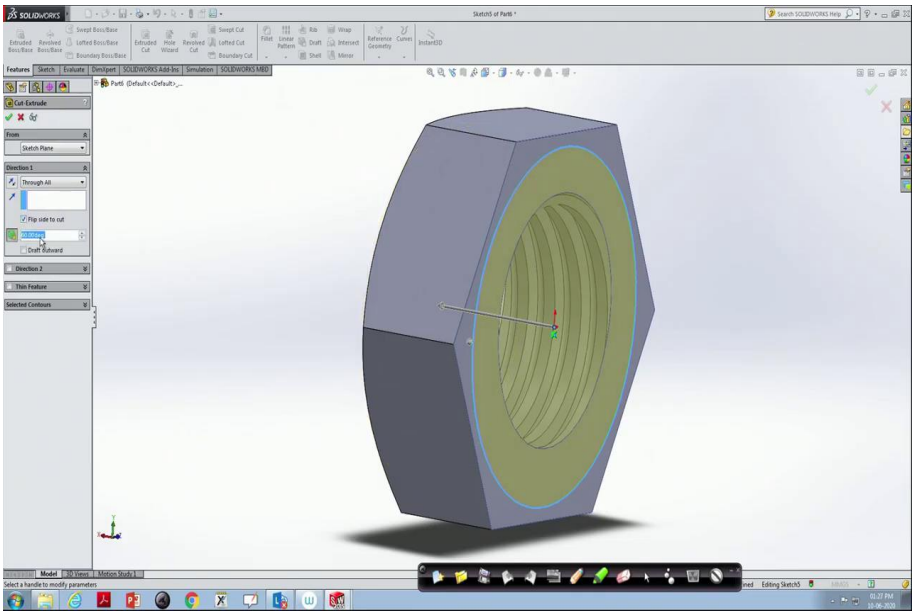

For this normal to it, draw a circle of the same radius, once done we pick this surface, go to features, extrude cut, instead of blind pick through all flip side to cut. It is 60 degrees angle we are going to have, click ok. So, the top portion we have removed.

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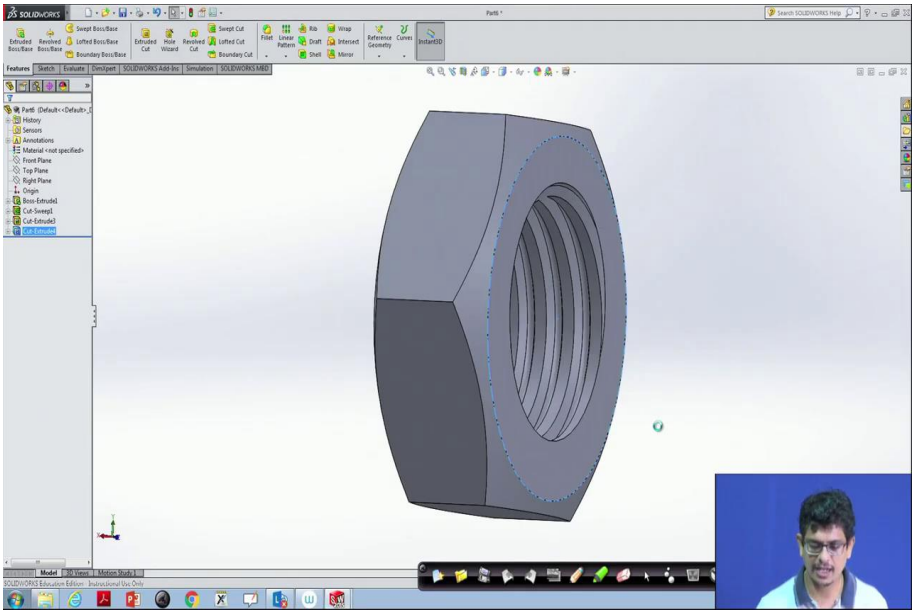

So, this only the portion where you can have a washer which one can fix it and these portions are always be having inclined cut.

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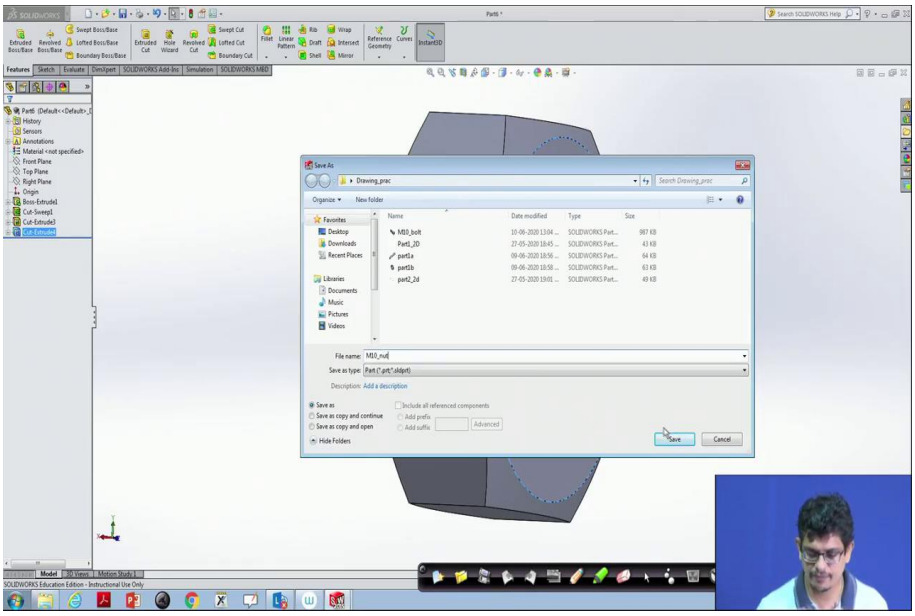

Now, let us save this drawing, save as M 10, now let us have nut and save.

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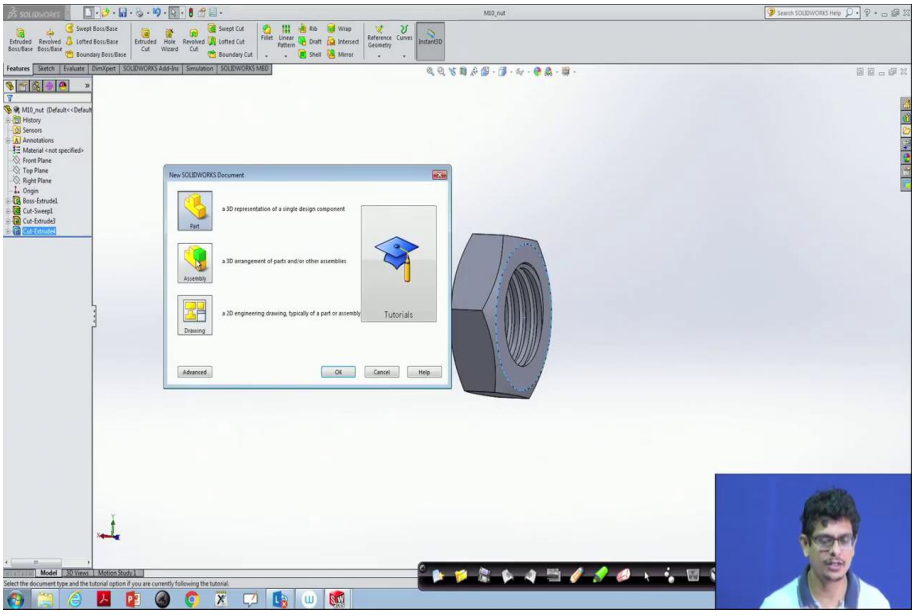

So, we have both nut and bolt, we can go with assembly drawing, assembly, ok, pick bolt nut both the things, drop it here, insert nut also and keep it. This is the way bolt and nut we constructed.

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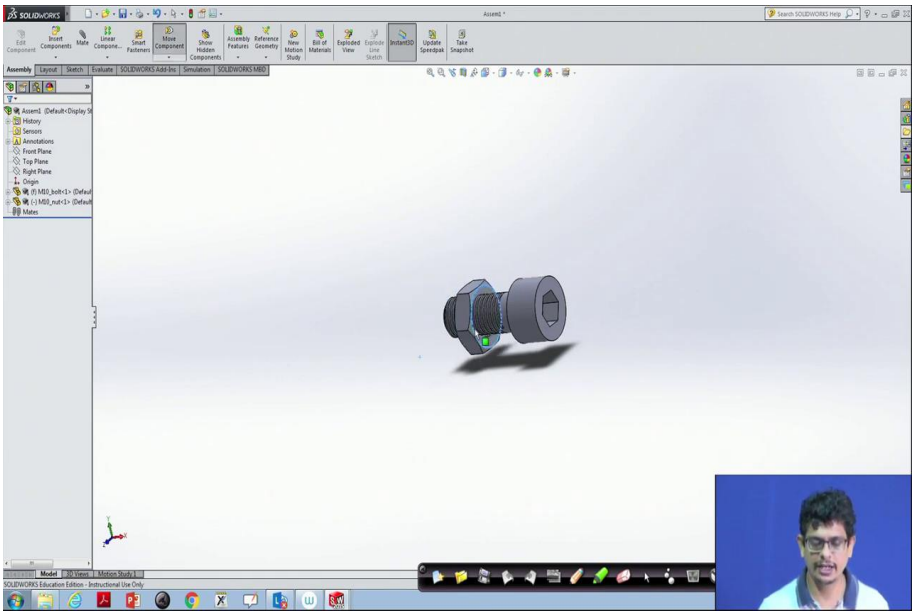

Now, we want to mate these surfaces. What we can do is take this one here and so on.

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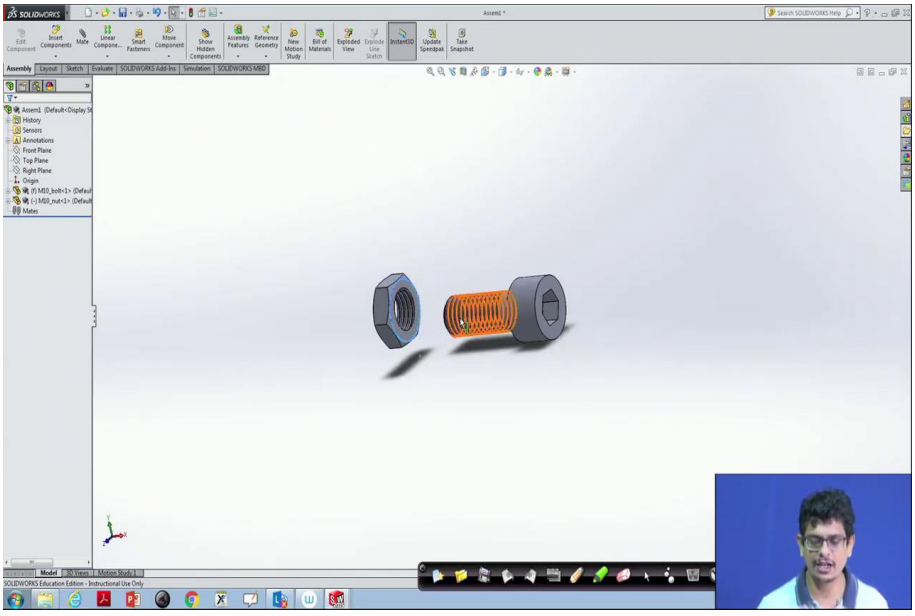

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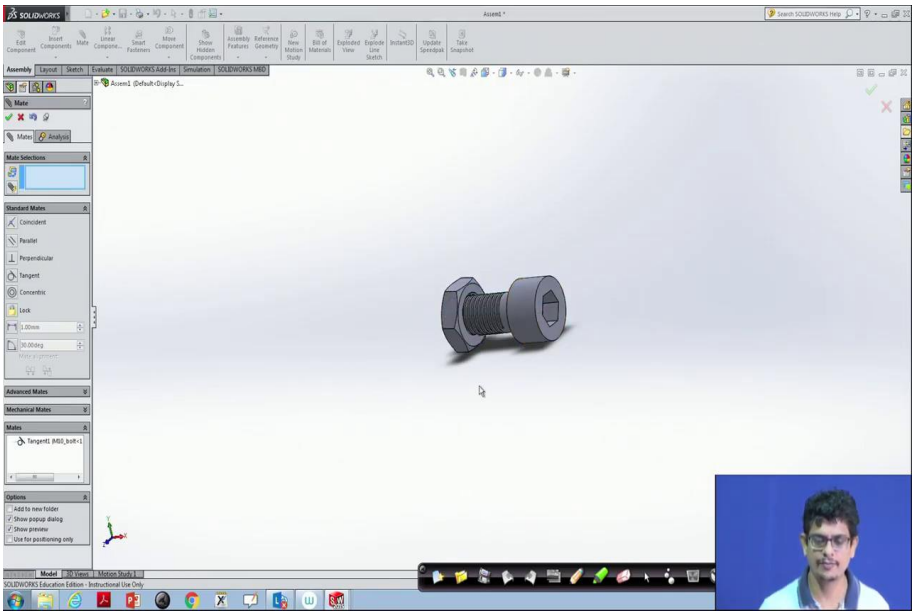

We can mate this spiral with that spiral so that it can be screwed otherwise surfaces are also we can mate it. Let us do something like mate this surface with that surface, and click ok.