Computer Integrated Manufacturing

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Lecture 11

Geometric Modelling (part 2 of 2)

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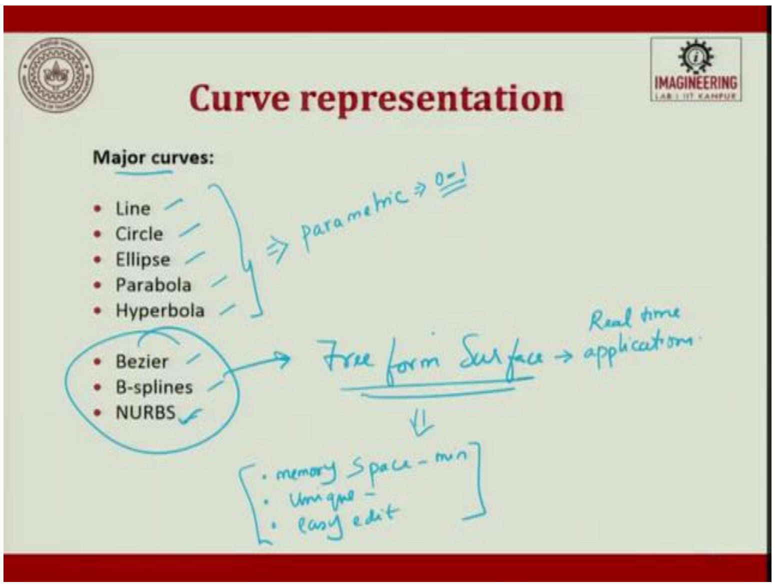

Now, let us continue our discussion on free form curved surfaces. If you go back and remember, we were looking at major curves. So, it was line, circle, ellipse, parabola, hyperbola, all these gave us a freedom of easily representing in parametric form. So, what is parametric form? We were also discussing where the value is taken between 0 to 1, so you establish a relationship with a parameter and that in turn is linked with X, Y and Z. So, life is not as easy as what it is look like.

So, now we will try to make little more complex free form surfaces, because lot of applications in real time comes from free form surfaces. Real time application you look at any car, you look a scooter, you look at human body, a bio-mimicking surface, you will see a complex free form surfaces.

So, now let us see whatever we have studied till now ruled surface, all these things will be used because we have to represent this free form surface with minimum space. I am talking about memory space, and as I told you earlier memory space should be minimum and it has to be unique representation all the time when you use a CAD and then it has to give me an easy edit. So, all these three things if you want to do it again free form surfaces have to be converted into parametric form. So, now let us focus on that and see more.

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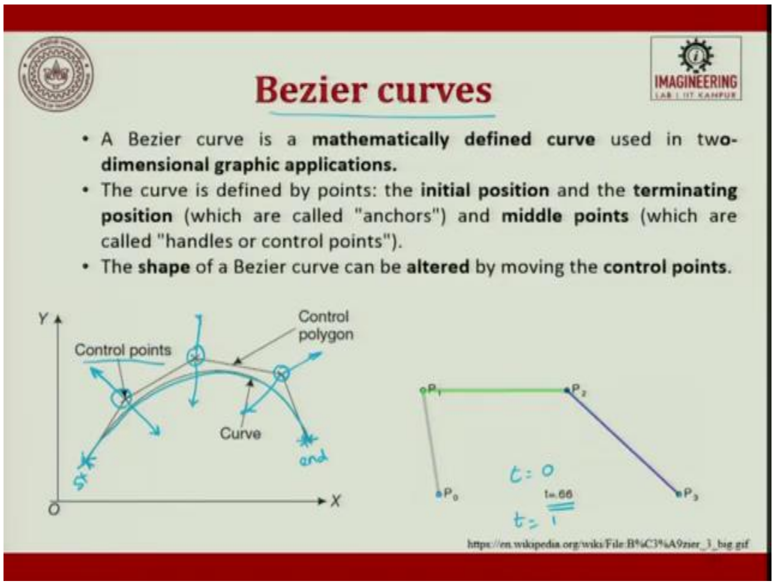

Bezier Curve falls under the free form surfaces. A Bezier Curve is a mathematically defined curve used in two dimensional graphic applications. So, the curve is defined by points, the initial point, and the terminating point, which are called anchors. And middle points which are called as handles or control points are used.

For example, if you want to draw this curve, you will have a start point and the endpoint that end point is called as terminating point, start point and endpoint and in between if you put start point and end point you can draw a straight line and you can. So, this is also a straight line which takes a different route in it. So, two points can be joined by n number of lines, okay.

Now, you have to define a curve there with using two points, so what you will do is, you will further define some more points in between the two points such that it tries to form a curve. Now, what are these points which you have defined? These points are called as control points, by pulling the control point left and right the curve shape changes. So, the shape of the Bezier curve can be altered by moving the control points.

So, it will start fixing start point and endpoint alone, rest of the points when you move left, right, up, down you can see there will be a variation in the complete curve. So, why are we trying to do it? Now, we are trying to go back to our parametric form. So, if I can establish this control points into a parametric value and then relate it, or link it with the start point, and end point, then this curves also becomes easy for representation.

So, here is a simulation which is stored, showed to you so P1, P3 are start point and end point. Now, you see there I am establishing various control points by just changing the parametric value from T equal to 0 to T equal to 1, you see that how beautiful it follows. Now, it is very clear, I have to use start point, end point and I will define control points, these control points again into parametric form such that I define a curve.

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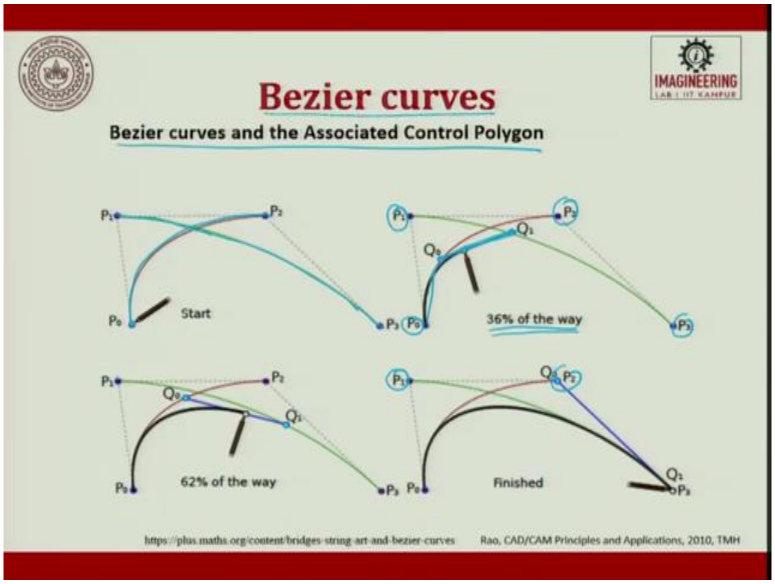

So, the Bezier curve and the associated control polygons, you look at it, the start point and then I tried okay, so, I tried to join start point P, P0 and P2 then P1 and P3. Now, 36 percent on its way if you see it will try to join this and this and then it will try to join this point and this point, and then it will start drawing an arch, okay. So, you see here it tries to draw along this when it goes it will try to define Q0 and Q1 points.

So, these are the control points, these are the start point, end point, and these are the points which are generated during the process, and then the curve tries to move along with it so that you get it.

So, 36 percent on its way you can see how the curve tries to take the profile. So, it is trying to link from this curve which was first started doing it with the P2.

Now, as in when it start moving, it starts moving and getting itself joined along with the line, or a curve, which comes from P1 to P3. So, you see here it follows 36 percent on its way, when we go 62 percent on its way you see that from P0 it has to go towards the destination this point and it is trying to be governed by P1 and P2, it tries to take the curve and finally it tries to join the curve.

This is how a Bezier curve and the associated control points are established, P1 and P2 are control points, Q knot and Q1 are during the motion which is trying to establish and then you will try to convert it into a straight line and that tries to connect between these two curves. Now, what is clear if I try to move P1 and or P2 or both you can see the complete curve gets dictated that means to say by changing one point the complete curve profile changes, that is what is the understanding you should have in this Bezier curve.

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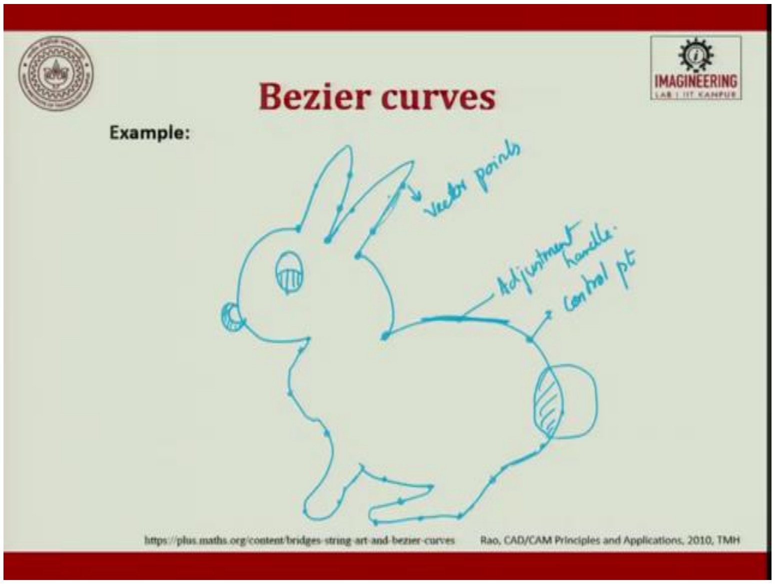

So, now let us look at an example for Bezier curve. Let me draw a cartoon figure so that you can try to understand much more. I hope it looks like a rabbit of course you have to have this symmetry. So, let me try that also. Okay, I hope it is a rabbit. So, now let us try to draw what are all the control points in this.

So, these are the adjustable handles okay, so this is a control point, these points are called as vector points okay and this one where there was a straight line which was coming up this is called as adjustment handle. So, if you draw a complex profile it looks like this, so these are the adjustable handle, and these are the control points. So, these are all the control points, this is a vector point.

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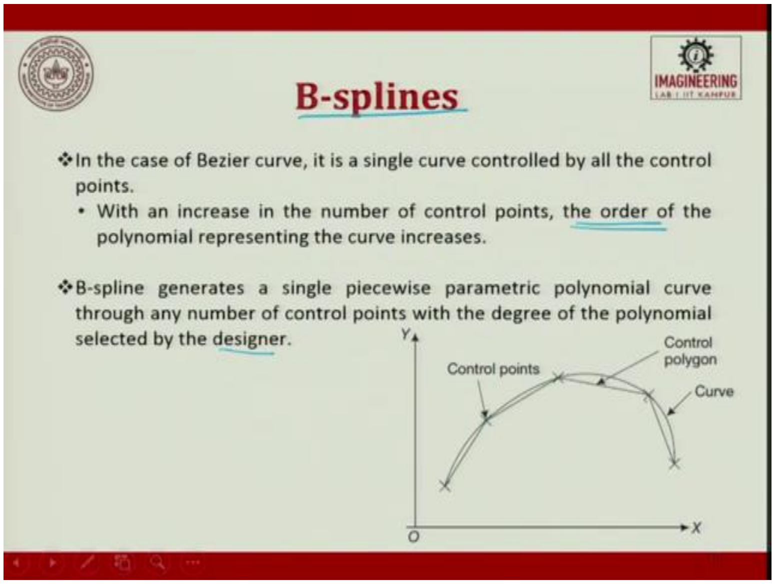

So, the next curve which we saw was B splines, I have set the limitation of Bezier curve also. What is the limitation of Bezier curve? If you try to move one point, the complete curve profile changes which you do not want to do. So, if you want to a very complex curve. So, then what you have to do is you should have so many start points, end points, so many control points in between such that you can try to draw a complex object.

But this tries to put me a restriction. The restriction is what? I am going to occupy lot of data because moment I say start point, end point, start point, end point, line, vertices all these things there has to be a table and then the relationship between these two also has to be established.

So, in that way Bezier had a limitation so that limitation has to be removed, so then came next curve is B spline. Next came B-spline in case of Bezier curve it is a single curve controlled by all the control points with an increase in number of control points the order of the polynomial representing the curve increases.

So more and more and more data points. B spline generates a single piecewise parametric polynomial curve through any number of control points with the degree of the polynomial selected by the designer. So, here in B spline we are trying to reduce the higher order terms and still try to draw the curve whatever we want. So, again here you will have a start point, end point and control points.

So, the next advancement even here the, when we try to move a curve the only difference between Bezier and B-spline is you try to fix the power the order of the curve, in B-spline you can fix it to the third order, and fourth order still you can try to draw a curve, but the equi-weightage to the control points were still a limitation.

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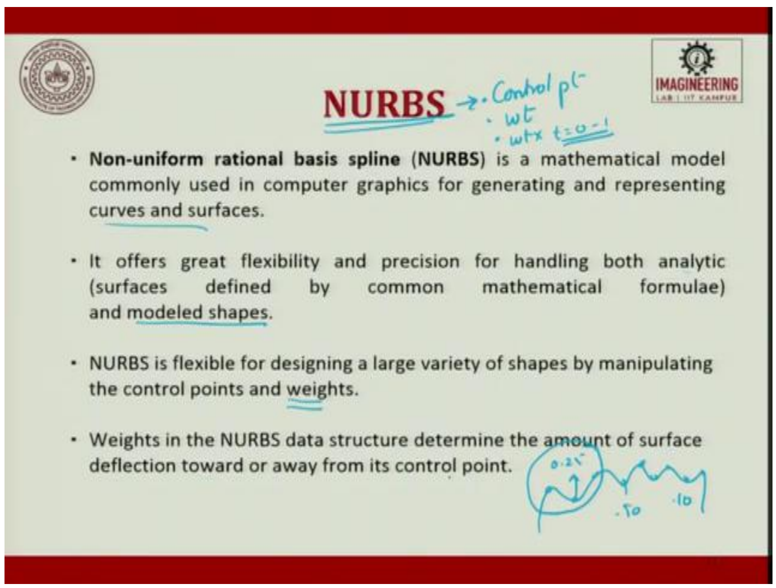

So, then came the next advancement call NURBS. In NURBS, NURBS you have it is expansion is non-uniform rational basic spline is a mathematical model commonly used in computer graphics for generating and representing curves and surfaces. Till now we were only, for the discussion point of view we were looking at 2D, please keep that in mind it was only 2D, when you try to draw a 3D surface it makes little more complex, fine.

The initial discussions whatever we had was a simple surface to be to be developed, so that was easy. So, when you have a free form surface it is much more difficult, it offers a greater flexibility nurbs and precision for handling both the analytical and modeled shapes. Nurbs is flexible for designers, designing a large variety of shapes by manipulating the control points and the weightages.

So, now what is the difference here with every control point, you try to define the weightage. So, the weightage will be multiplied with the parametric value t equal to 0 to 1 and then you try to tweak the curve as and when at which place it is required. So that means to say if you have a curve like this rather than defining every control point you just define few control points and start giving weightage saying that it takes 25 percent, it takes 50 percent, it takes 10 percent.

So, now depending upon the weightage when you try to move the curve only this patch is changed, rather than the entire curve. So, that is what is told along with the control point weightages, weights in NURBS data structure determine the amount of the surface deflection towards or away from the control point.

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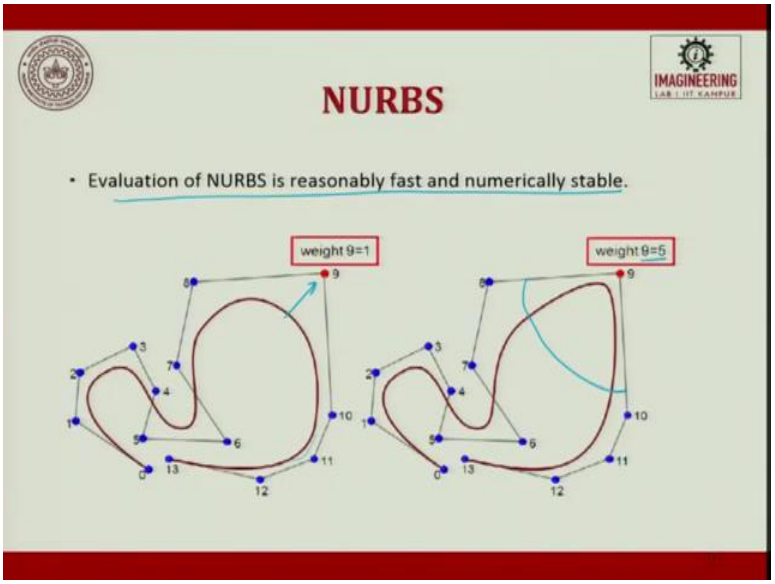

So, this is what is evaluation of NURBS is reasonably faster and numerically stable, so if weight 9 equal to 1. So, the curve looks like this okay, if the weight 9 is equal to 5 so then you can see how the curve changes with 9, when 9 equal to 1 the curve gives this wattage when 9 equal to 5 you see how the curve shifts only in this parts and rest of it is very stable. This is very important when we try to do geometric modeling of real surfaces, or real life surfaces.

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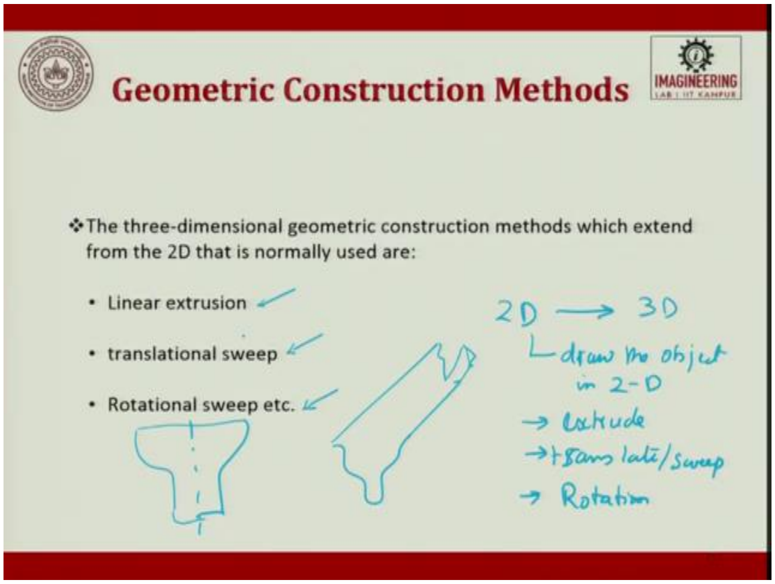

Moving on we are now getting into the topic of geometric construction methods. The three dimensional geometry construction methods which extends from 2D that is normally used, linear extrusion, transitional sweep, and rotational sweep. So, here what does it mean is till now what we saw was 2D.

Now, we are moving from 2D towards 3D. So, while moving in we are now trying to see and intermediate, so that intermediate step is draw the object in 2D and try to extrude that means to say pull the same object what you have drawn in 2D in the Z direction or try to translate. So, that means to say I have a channel made I wanted to extend a channel in this direction.

So, now what has happened is this is translational or sweep, translational sweep or if you want to do rotation, you can do rotation, for example, if you have axes symmetry part I have drawn here cup and then I try to say please rotate this cup by 360 degrees, then immediately this is now made, okay. So, this is an extension of 2D in only one direction or in some degrees. So, linear translation sweep.

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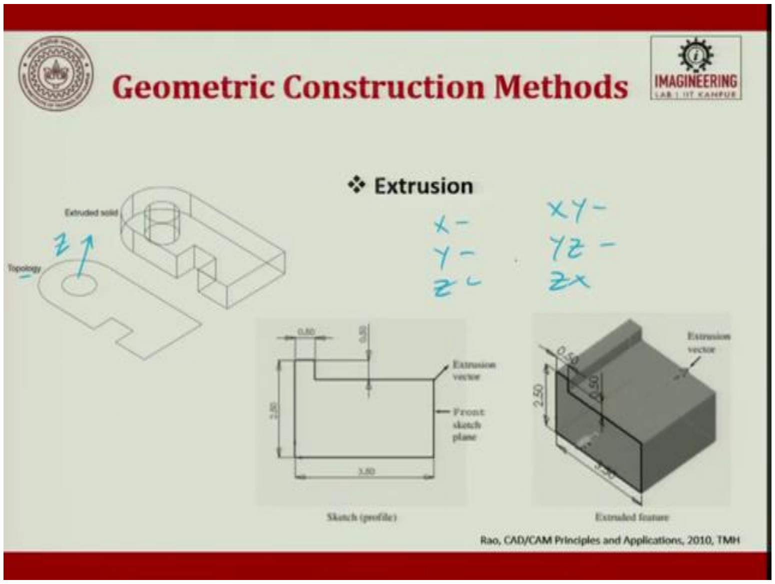

So, let us look at these examples whatever I have developed. The topology which is drawn in 2D is this, this is a 2D. Now, all I did was extruded in Z direction. So, now you get a 3D object. So, this is a 2D which is drawn I sweep it in this direction, you can do it in X plane, Y plane, Z plane, you can do it in XY plane, YZ plane, or XZ plane sweeping and extruding.

It is very easy because I have represented the 2D in a simple mathematical form. Now, I have to just extend the value in one direction, so this is extrusion which is done.

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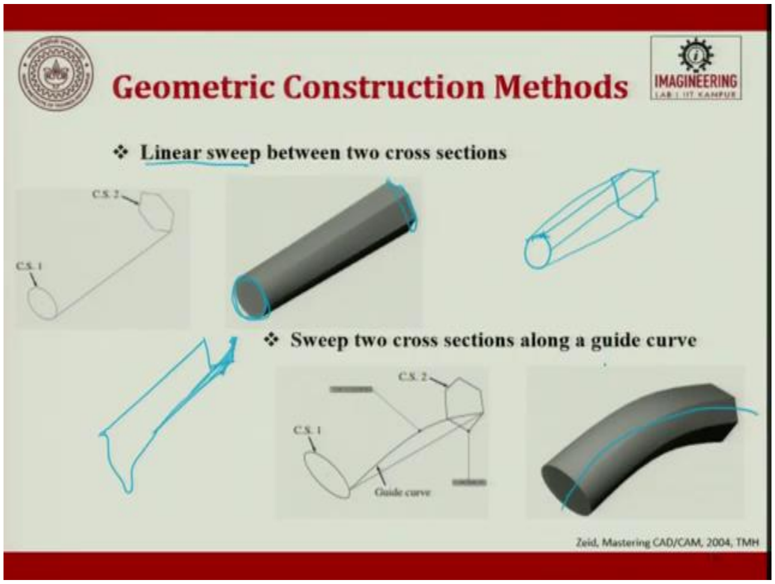

Next let us look at translation. So, I have a circle and this is a hexagon, now these two are joined now what I do is, I try to translate it with this. So, the circle is now joined with every edge of hexagon, so you can see here the cross section is circle here and cross section is hexagon here. Now, you have joined these two very easily.

If you want to do it line by line by line so then you have to choose a point, choose in the hexagon whatever it is and then you have to link, so which is going to be time consuming and it is not going to be so easy for you to locate the midpoints of hexagon as well as this. So, now by doing it I am able to get this done. So, if you want to draw pipes of varying cross sections and link it.

So, linear sweep is the operation which we use. As a told you again a channel attached to a rectangle here, you want to do that. I think it is very easy to do through linear sweep. Next is sweep, two cross sections along a guided curve so you have two cross sections and you have guided only the curve you have given and then it tries to join.

So, you can try it, because that was a straight line, this is a curved, now you look at it how complex jobs can be made easily just by using the translation and sweep. This was linear sweep, this was a sweep for a curved surface.

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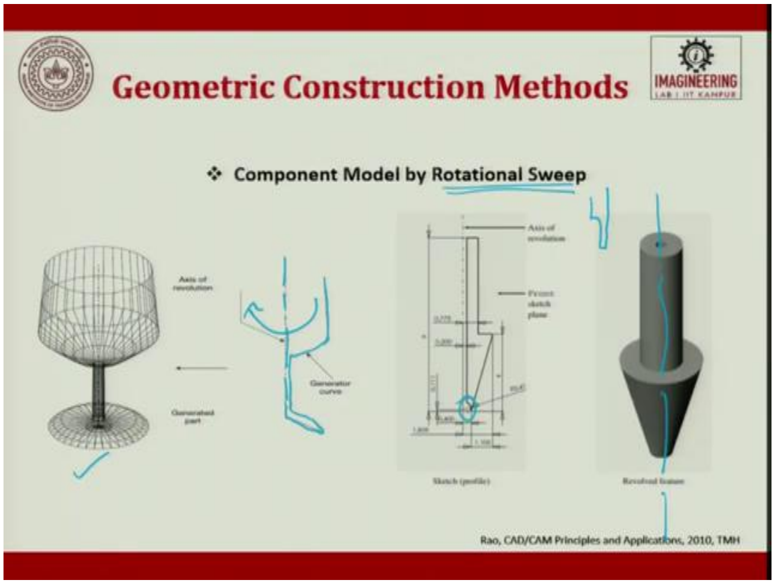

Now, let us look for rotation as I told you, you have just a drawn a wine glass a quarter of it right, and now what you say about this axis please rotate it by 180 degrees or 360 degrees, you get a profile of a cup. So, this is rotational sweep and see look at it this is a object again wherever there is an axes symmetry object you should try to use this option and start developing it. So, it will occupy minimum space and it is handy for you to edit also, it is unique, look at it here, how beautifully they have done.

So, this portion is like this, so if you want to sweep it around also you can sweep it and that is at the bottom you can get this done. So, you draw it in a zoomed manner and then you reduce it to normal scale and sweep it, or you sweep it and then you want to expand you can do. So, here this gives you a very easy way of constructing 3D solid from a 2D basic drawing.

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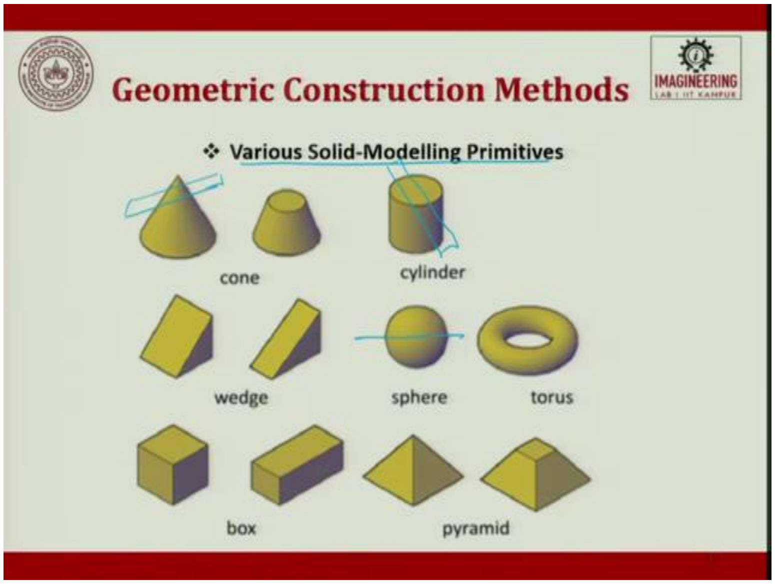

So, these are all the various common functions of solid, these are all what you create. Now, what is available in all the CAD software today is they have a standard library functions and they have put all the basic, basic solid models and if you want to link it with my previous lecture whatever I have discussed you draw a cone, you cut with a plane, you get whatever curves we have defined in the previous lecture is possible here.

So, you see here a cone is cut, you get a circle, then you draw a cylinder, you want to cut a cylinder in this you can get it done, you can get a wedge shaped, then you get a sphere, then you can come develop a hemisphere, you develop torus, cube, cuboid, pyramid, and a cut pyramid. So, these are the shapes which are already available in the library function of any 3 dimensional software, 3D software, you take and under solid modeling you have all these options.

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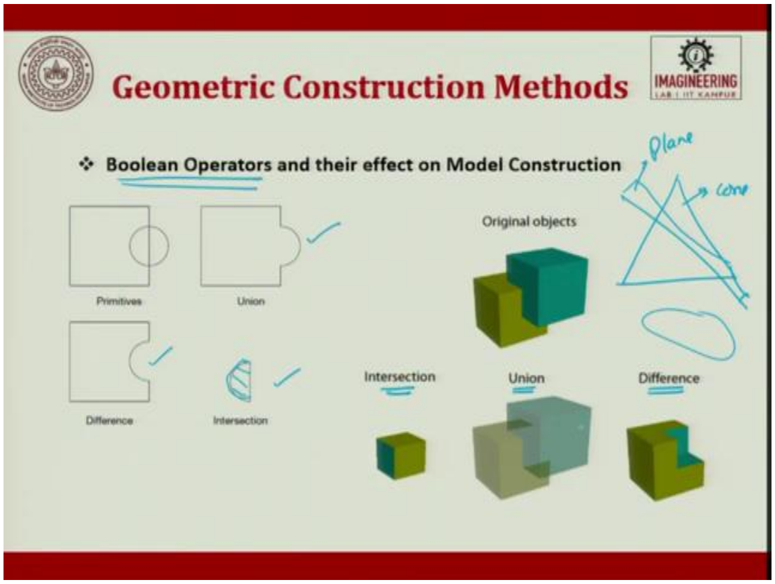

Now, you have all these objects, so now depending upon your requirement the easiest thing is going to be you start applying Boolean operations. Boolean operations are intersection, union and difference. Three are the A union B, A intersection B, and then you have a difference.

So, what this means to say is rather than drawing a component fully from scratch, you use this solid model primitives which are available and try to smartly use the Boolean operation and try to get the profile, or the feature whatever you want.

For example, if you wanted to generate an ellipse, all you have to do is pull a plane option in the model, pull a cone option and try to place it at some place and say A intersection, or A union whatever it is you try to define and then the profile what gets generated is going to be an ellipse. So, we try to think of using Boolean operations because mathematically Boolean operations are easy to store and edit.

So, primitive so you had a circle on a square, so then when you say A union B, you get this union this half goes away, when you say A intersection B, you try to get this portion, because this is the portion common to both, you say A difference B this is what you get. So, either you can choose this, choose this, chose this for your requirements.

So, that is the beauty of using Boolean operation. I had two original objects when I wanted to do the intersection of these two, I got this. When I do a union of these two I get this solid, when I try to do a difference of these two I get this solid. Now, by using this in a very efficient manner you can try to develop several of these objects.

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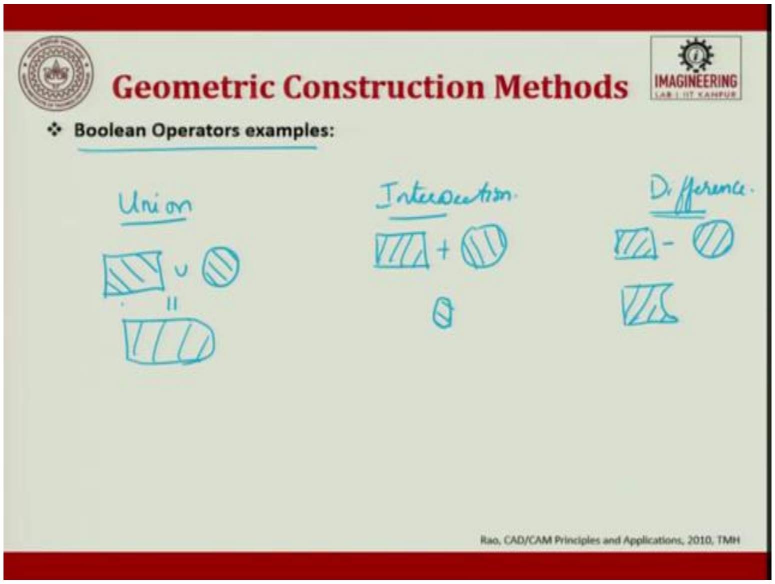

So, some more examples of Boolean. So, as I told you, you will have union, you will have intersection and then you will see difference. So, a union so let us assume this is a square and a circle, so what you get is going to be, this is the object which is getting generated. So, you try to get the intersection of it, so what you get is this portion alone.

So, difference is this minus this, so you try to get this portion alone. So, you can try to do any of these operations and try to develop whatever I have said here, I have given some more examples it can be a solid, it can be whatever it is.

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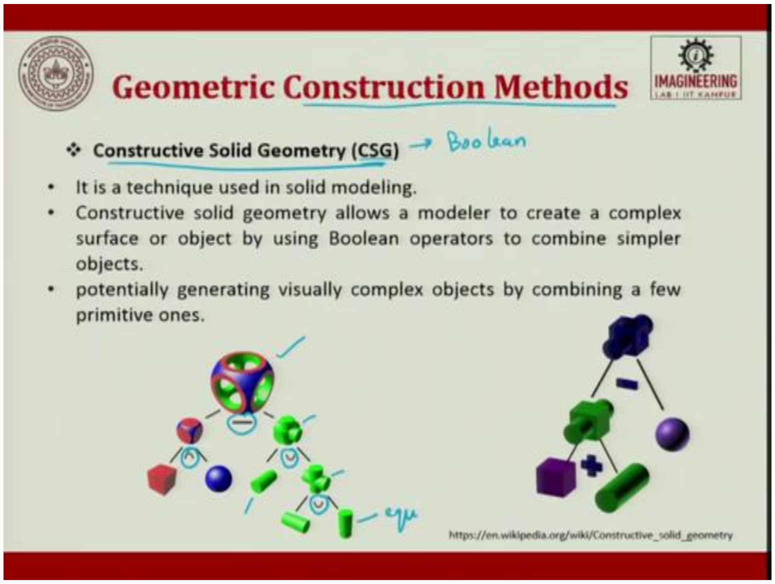

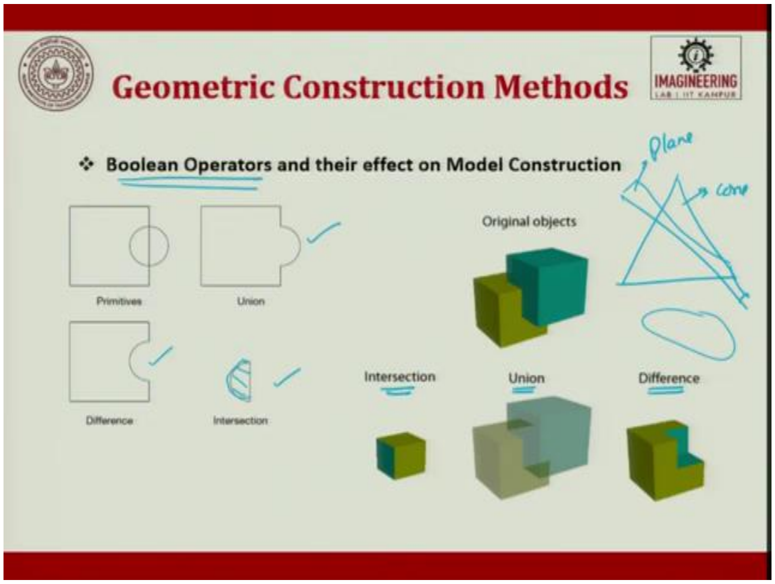

The next one is called as constructive solid geometry which is also used as part of construction method. So, constructive solid geometry means using these Boolean operations you can try to draw or store, or develop any 3 dimensional object from the solid model whatever it exist. This is a technique used in solid modeling, the construction solid geometry allows a modeler to create a complex surface, or object by using Boolean operation to combine simpler objects.

I know all these things can be defined by parametric equation, then all I do is I try to follow Boolean operations and get whatever I want to the final. So, there are two ways of achieving the same output, for example I can start with the cylinder-cylinder, cross cylinder-cylinder and then I can develop this, or I can start with this and then subtract it from this and then put this inside I try to get this object. Potentially generating visual complex objects by combining of fewer primitive once are possible by using constructive solid geometry.

So, you will have a cube, you will have a cylinder, when these two join, you get this, and then this has to be subtracted from this. So, finally you get an object like this. So, constructive solid geometry follows Boolean operation what we have discussed in the prior, so here solids are taken and these solids are removed, so that you try to get. So, whatever we have studied in the previous case these are an extension that extension leads to constructive solid geometry.

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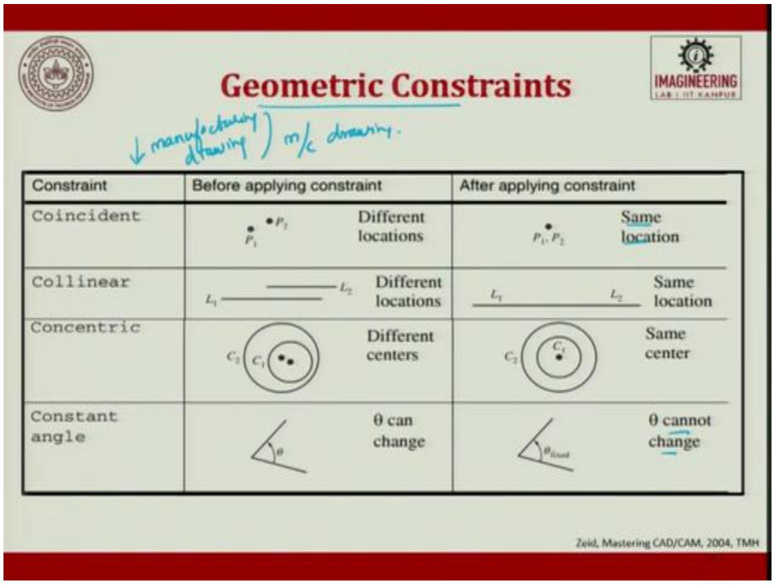

So, the constraints generally used, geometric constraints which we generally use are you constraints are coincident, collinear, concentric, and constant angle, these are some things which we follow very importantly when we try to develop a manufacturing drawing, manufacturing drawing. So, that we try to tell in terms of references or I can try to write is as machine drawing.

So, here before applying constraints you define two locations then after applying constraints you put both and then say same locations before applying constraints it will be like this and after applying constraints it is this. So, this is for coincident then this is for collinear, so you can have different locations at different locations, at the same location concentric at different locations concentric at the same location you can express this before applying constraints it will be something like this.

And then when you apply constraints it will exactly make sure it will join and it will draw, then constant angle is this theta can change. So, the theta is always fixed, you can put constraints and draw the geometry for your requirement this makes a standard so that you will not make mistakes while joining the geometric modeling.

It is almost like establishing in a excel file you are trying to established format 2 decimal and 3 decimal, moment you said 2 decimal it always takes to the rounding of and then it tries to add the value. It is almost the same feature here. If you have 2 points and if you do not apply constraints you can land up like this but if you say constraints it will go like this. So, this tries to help you in drawing.

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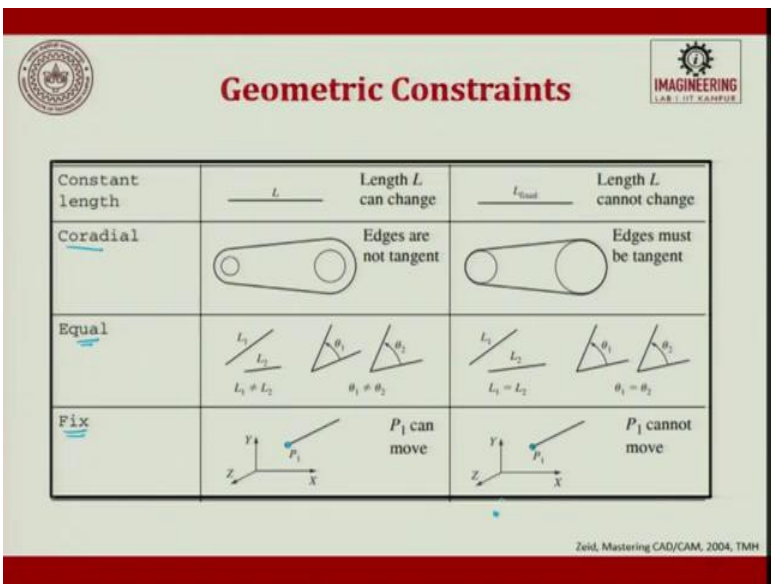

So, then, the other thing is constant length so the length wherever you draw is the length can change. So, when you put constraints, it can be fixed coradial, coradial can be there, edges are not tangent here but edges are tangent here when you say coradial, when you say equal, so L1 equal to L2 theta 1 equal to theta 2. So, all these things like fixed angle you can get this done.

Then the other thing is fixed so X, Y, Z point 1, is here P1 can move so P 11 you say fixed, it cannot move wherever you draw it will go and take this point P1 closest to this point and start doing it. In some of the commercial softwares you can, you not go exactly to dot to dot in that point and draw the next line you can go very close to it and then please choose the closest end point, so immediately it goes chooses it and then draws. So, those things are called geometric constraints which are used for helping you to draw drawings in easier fashion without error.

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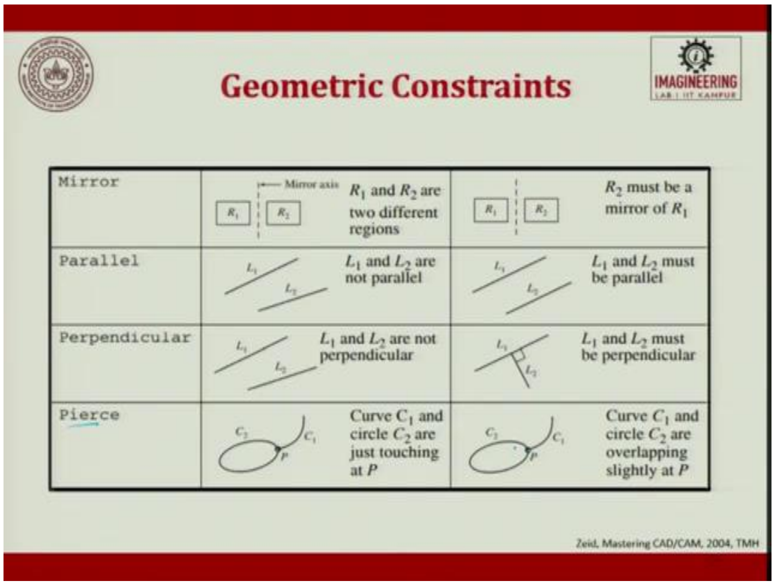

So, again mirroring can be done R1 and R2 are two different regions it is fixed, parallel, can be done. L1, L2 are not parallel it can be made parallel, perpendicular when I talked about line definition I told these things L1, L2 must be perpendicular then it is peers.

So, peers means it is curve C1 and circle C2 are touching a point at P, so curve C1 and circle C2 are overlapping slightly at the point P. So, this is constraint, this touching need not be when you zoom it and see it might not be touching with each other but here it will make sure it touches with that curve.

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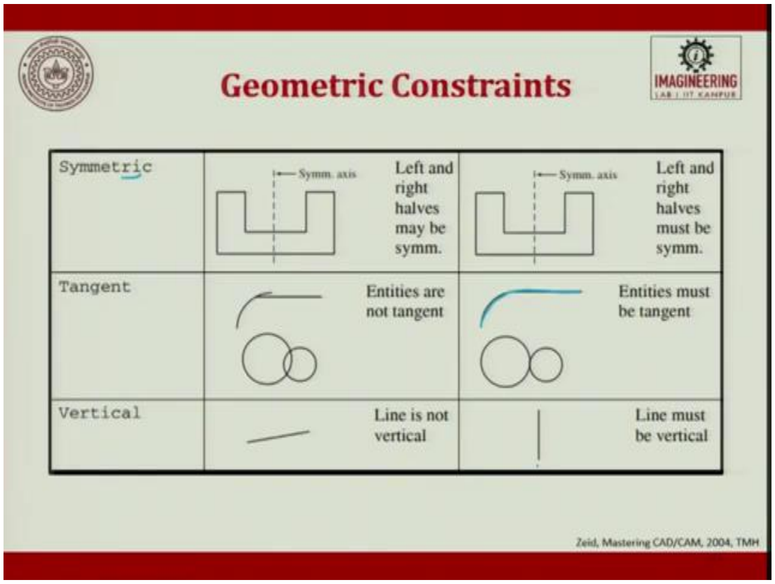

Some more constraints you can see symmetry left and right will be symmetry when you put constraints, tangent 2 you can see it will be exactly drawing the tangent. In CNC as I told you previously also if you want, you can write it by G code and M code otherwise with this geometry and the tangent you can start writing the program and the features can be understood accordingly.

These are the leads statements later when we use these drawings to recognize feature and automatically plan some processes. So, we use all these things for our betterment. So, vertical line need is not vertical whenever you draw a line it is vertical which it can be done.

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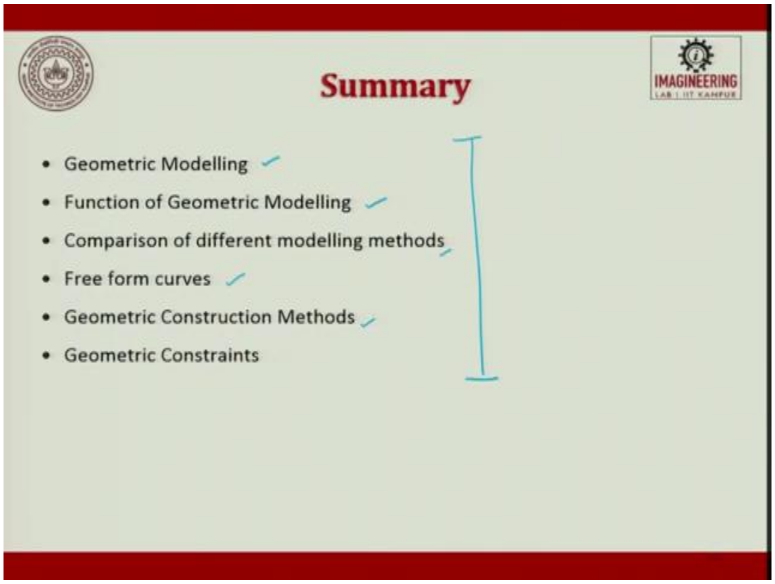

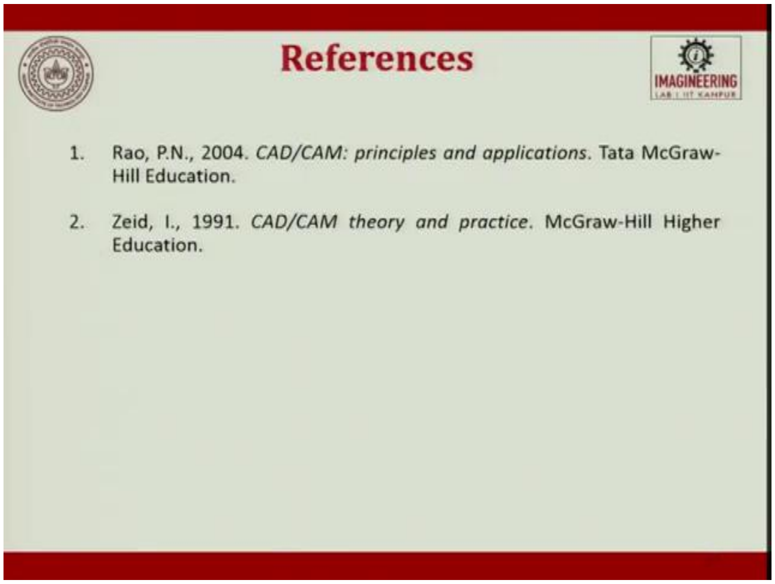

To summarize in the geometric modeling chapter what all did we see? We saw geometry models, we saw functions of geometric models, we saw comparison of different modeling methods, then we saw free form curves, then we saw constructive solid geometry and at the last we saw about constraints this will try to help you to draw the drawings without any error.

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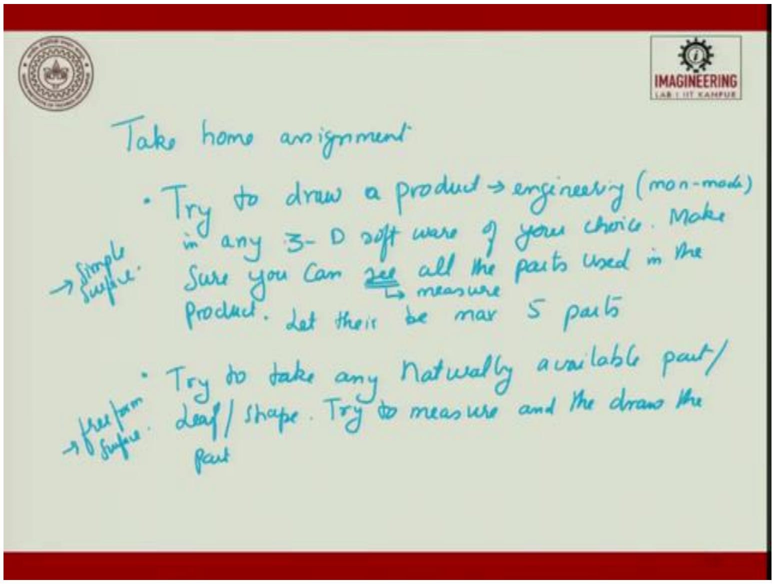

So, I would like to give you take home assignment, so take home assignment or home assignment, so I will just give you two assignments for you. Try to draw a product which is engineering because engineering means, I am making it man made in any software, any 3D software of your choice and make sure you can see all the parts used in the product.

So, that means to say you can see means you can measure okay, and here it is your engineering products. And let there be maximum 5 parts, okay next try to take any naturally available, naturally available part, it can be a leaf, it can be some shape try to measure and then draw the part. So, here you will see it will all be simple surfaces, here it will all be free form surfaces.

So, now the challenge is you have to measure and then draw, here you will measure and draw it will be very easy. I want you to do these two exercises such that you will try to enjoy what all we have studied till now in geometric modeling, Bezier, B spline everything you will do and when you start using any 3D software, CAD based software, you will start appreciating how did this CAD concept evolve and how really it is difficult for measuring and defining surfaces. Thank you.