Rapid Manufacturing
Prof. J. Ramkumar
Dr. Amandeep Singh Oberoi
Department of Mechanical Engineering & Design Program
Department of Mechanical Engineering
Indian Institute of Technology, Kanpur

Lecture - 37
Rapid Product Development, CAD (Part 1 of 3)

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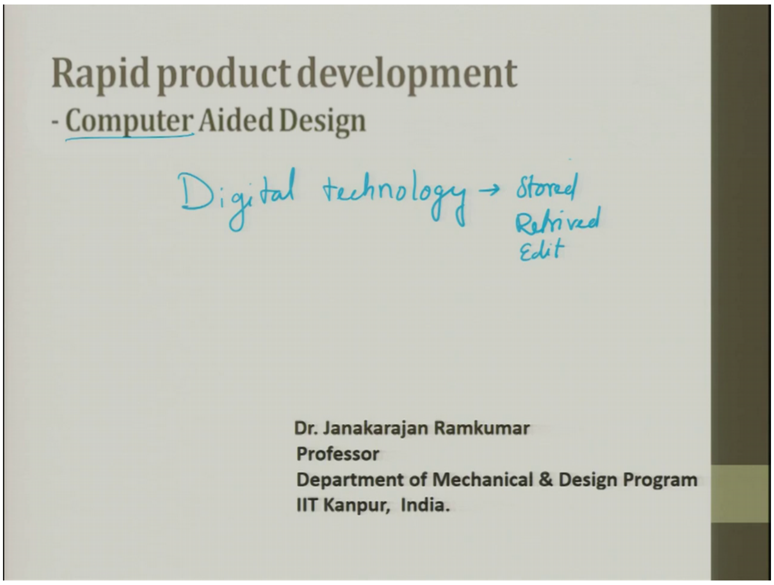

So, welcome to the lecture on Computer Aided Design and Manufacturing. See these computers played a major role in bringing all the informations' to digital form. Today we do not call it as CAD or CAM; we call it as digital manufacturing. This digital gave a big boon to the manufacturing; one in terms of creating objects whatever we want, two - the created objects could be manufactured just by transferring the digital data from CAD to manufacturing environment. Let it be a rapid photo typing, let it be a rapid manufacturing, let it be measurement, let it be machining. So, all these places a single data could be exported, imported, and people could use it for producing the required output.

So, computer played a very, very major role. I would reframe the digital technology played a very major role. So, the data could be stored; the data could be retrieved; the data could be easily edited too. And once you start doing this, and you have it at a common place, then all other people when they retrieve, when they use the data, they take it from this data base. So, this data base in the digital form became a center core where everybody use the center core for their application, and then they could edit, correct, save, store, latest version change, everything they could do at their end, and the data could be transferred. So, computer played a very, very important role.

When we talk about engineer's language is drawing. So, initially this computer, there were software's develop for drafting; and later came the additional knowledge which we started doing on analysis and optimization. So, today it has become computer aided design, and the same data could be used for manufacturing. So, computer aided design and manufacturing will be this lecture we will try to study more. The rapid manufacturing whatever we are talking in this course could happen only because there was a provision for taking reverse engineering data or there was a provision for starting the data forward direction from CAD files. So, this made a big change. So, in this lecture, we will see some introduction.

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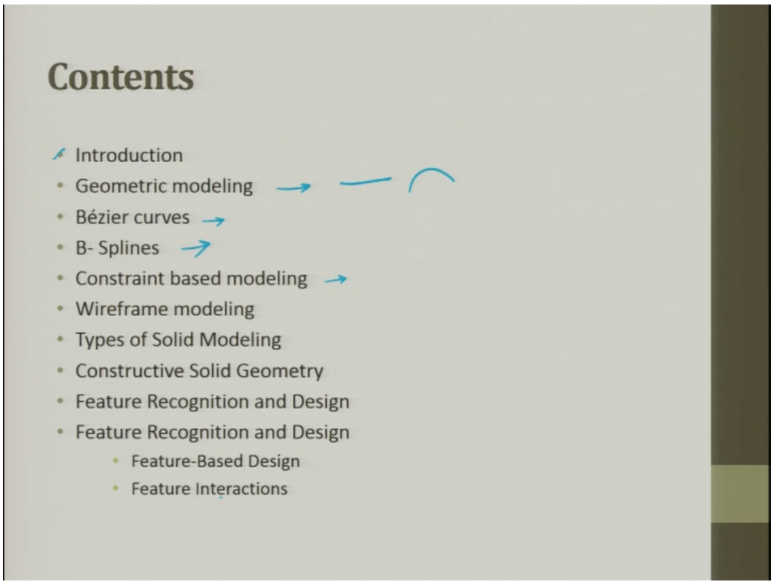

Followed by introduction we will try to see geometric modeling. So, if I know to draw a line, and if I know to draw a curve digitally, then there solves all the matter. So, if whatever complex objects I have, if I could discretize into lines and curves, then I am in the business. So, geometric modeling could be done, then followed by it we have Bezier curves. So, first we will see how to draw lines, and then we will see how to draw curves.

So, Bezier curve, B-Splines, then we have constraint based modeling, we will see wireframe modeling, then types of solid model, constructive solid geometry, feature recognition and design, then feature recognition and design, wherein which we saw future feature based design and feature interaction. So, these are the contents which we will try to cover in this lecture on CAD and CAM.

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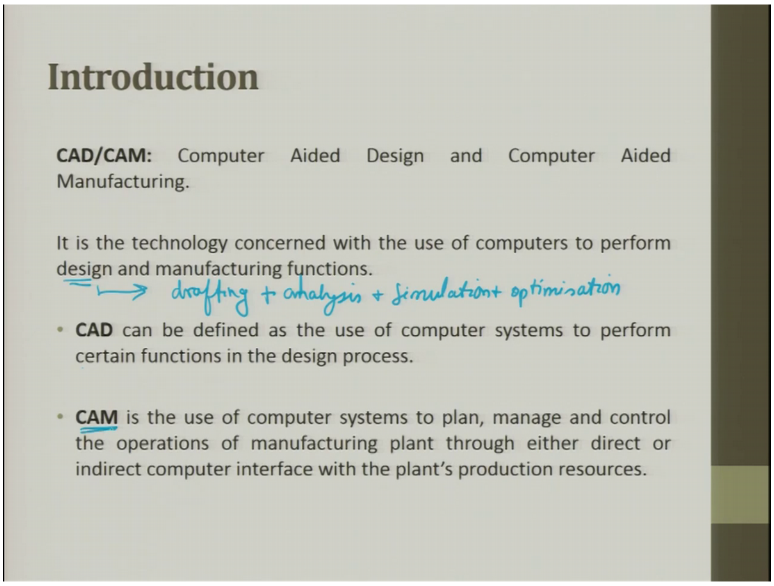

CAD and CAM are nothing but Computer Aided Design and Computer Aided Manufacturing. It is the technology concerned with the use of computers to perform design and manufacturing function. So, this design when we said as I said earlier also, it will involve drafting. On top of drafting, you will you will have analysis, simulation and optimization, this we will do in this designing. So, all these things could happen software's could talk everything could happen because the data was in digital form. It can be an ASCII, it can be in binary, if you want to read, if I want to read it will be in ASCII, if machine has to read it will be predominantly binary.

So, manufacturing function and manufacturing procedure. The CAD can be defined as the use of computer system to perform certain functions in the design process. CAM is the use of computer system to plan, manage and control the operations of manufacturing plant through either direct or indirect computer interface with the plant's production resources, this is CAM.

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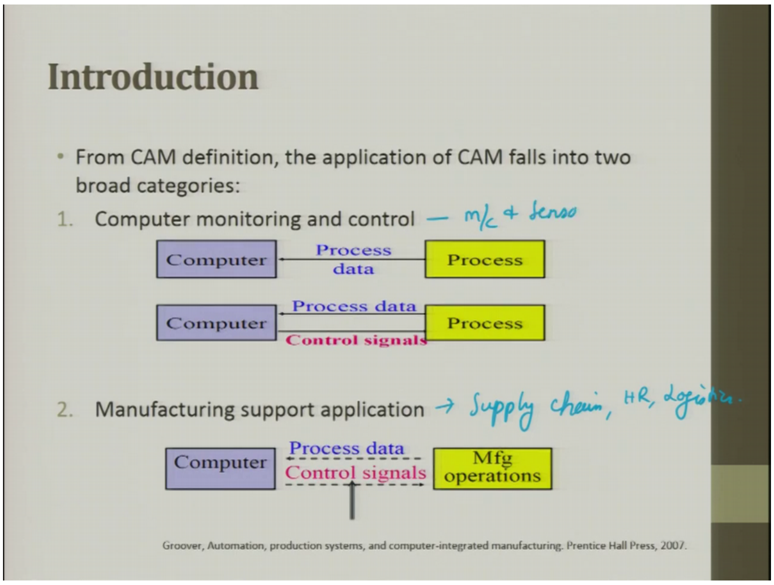

For CAM definition the application of CAM falls in two broad categories; one is computer monitoring and controlling, and the other one is manufacturing support application. Computer, process data, you have a process. Computer, process data, and control signals, this is the advancement. So, first is computer, process, the processing data is flowing. Computer, process, process data, and control signals are there. So, computer, monitoring and control, and manufacturing support application. So, in manufacturing support application, you will have computers and manufacturing operations. So, process data and signals will flow here. So, this is manufacturing operations.

So, the CAM is divided into two categories, computer monitoring and control, manufacturing support applications. When we talk about manufacturing support applications, it talks about supply chain and other things supply chain, HR, logistics. When we talk about computer monitoring and control, it talks about machine and sensors for online monitoring.

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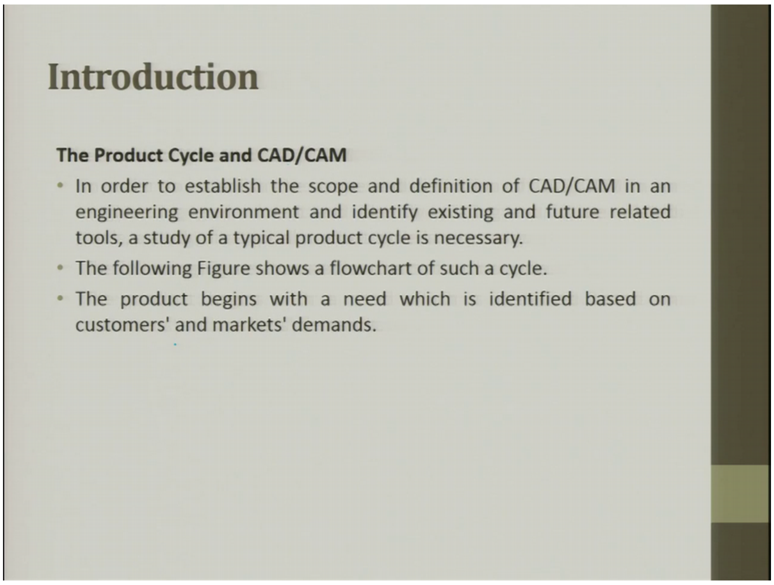

The product life cycle and CAD and CAM; in order to establish the scope and the definition of CAD CAM in the engineering environment and identifying existence and future related tools, a study of a typical product cycle is necessary. We will see the product cycle in the next graph. The following figure the next shows the flow chart of such a cycle. The product begins with the need which is identified based on the customer and market demand.

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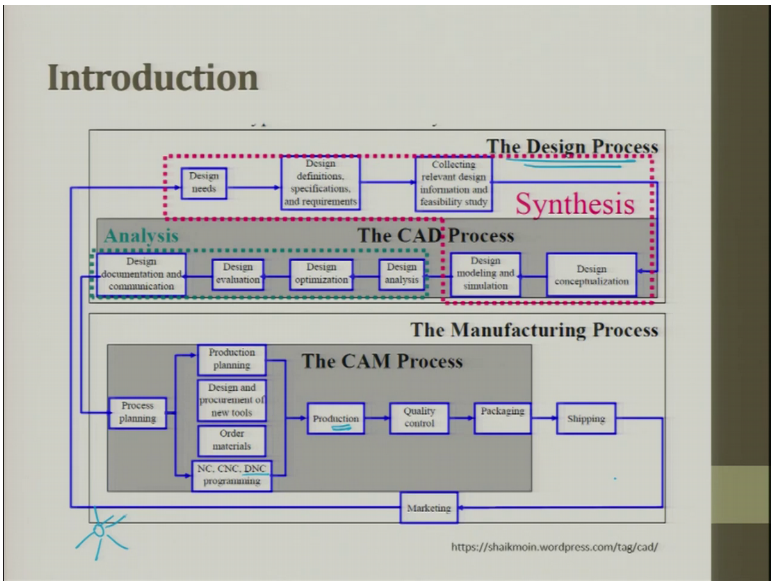

So, this is the graph or the schematic representation of the complete setup. So, here you can see, there are two parts; one is design part, the other one is manufacturing part. And here you can see here CAD process, and then you have synthesis here. So, this is analysis. So, the CAD process, in CAD process, what do we have is, we will have designer need, and then design specification, definition, specification and requirements, and then collection of relevant design informations and feasible studies. After this comes the design conceptualization. Then design conceptualization to design modeling and simulation, this leads to design analysis, design optimization, design evaluation and design documentation.

So, you can see this, these are all part of the design process; customer need, from the customer need you put a spec; from after putting the spec, you do the state of the art. State, after doing the state of the art, you develop concepts; these concepts are all these are all proof of concepts, and you can have functional add at functional requirements added to it. Then what you do is you try to optimize the developed POC such that it can go to the market. Once you try to do that, you will also try to have analysis, optimization, evaluation. So, here it is only modeling and simulation. Once you simulate, then you optimize.

When it comes to the manufacturing point of you, so the data from the documentation goes to process plan. In the process plan, it is divided into four; one is production process, design and procurement, order material, and NC, CNC, DNC. DNC is Distributed Numerical Control where in which you have a main server and you have several CNC machines attached to the server. Every time the information is pulled out from the server and then it is executed in the CNC machines. And here if you have a long program it is done. And if it goes line by line or segment by segment if it goes, then it is called as drip irrigation; drip numerical control. So, but what here we are talking about is direct numerical control, then all these things put together in a capsule come to its production. Once you produce, you have to check for the quality. After you check for the quality, you say yes, it gets into packaging. Moments it gets into packaging, it gets into shipping. Shipping it goes to the market and then you try to complete the entire cycle. This is the complete product life cycle for CAD and CAM.

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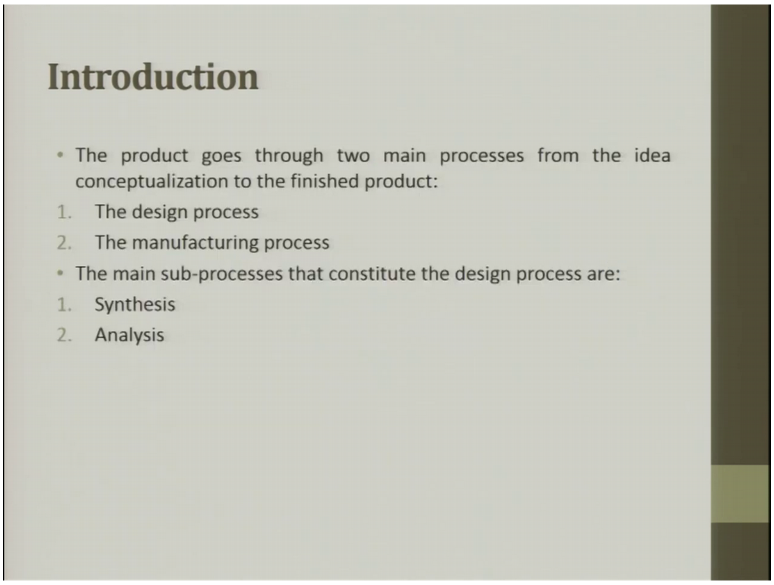

The product goes through two main processes from the idea conceptualization to the finished product, design process, manufacturing process. The main sub process that constitute the design process are synthesizing and analyzing. Synthesizing is you have humpy number of data. So, this data has to be genuinely looked into, and you remove the noise, you then consolidate the data into an information that is synthesizing. Synthesizing of a design is a very tedious process. Once you synthesize, then whatever you get you start analyzing what you do did in the synthesizes. So, these two are the subgroups. If you go to the previous figure, you can see synthesizing cycle. Synthesizing means this all process is you have collected data, and then you convert the data into information, and then into knowledge.

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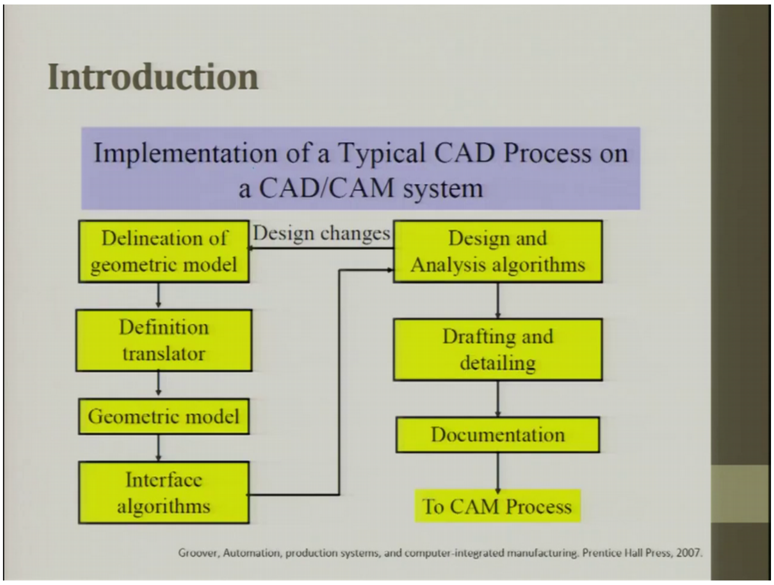

The implementation of a typical CAD process on a CAD/CAM system, it involves delineation of geometric model. Then you define translators, then you try to go for a geometric model, then you have an interface algorithm. So, here this interface algorithm goes to design and analysis algorithm. So, here design changes can happen between these two, then cams drafting and detailing documentation, and finally a CAM process. When we try to implement a implementation of a typically CAD process on a CAD/CAM system, these are the steps which are involved.

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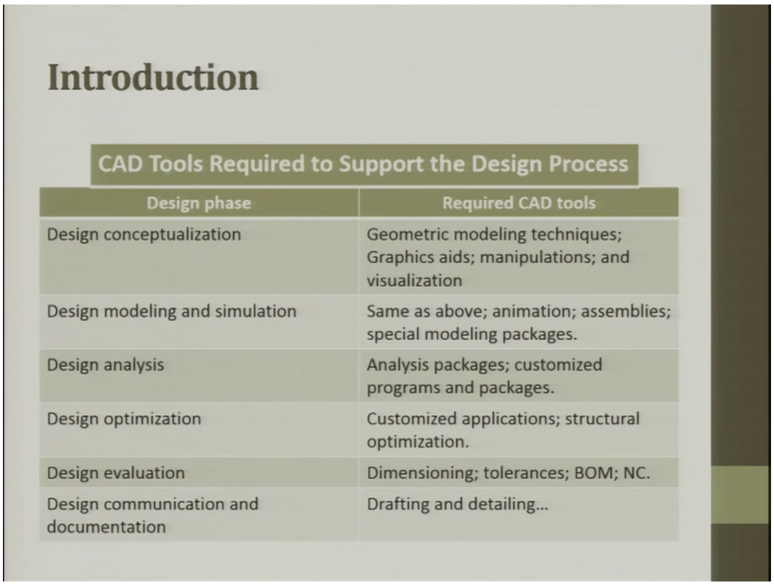

So, the CAD tool requirement required to support the designing process. These are the tools a design conceptualization, we need a geometric modeling technique, a graphic aid, manipulation and visualization, these are the requirements for the CAD tool. When design modeling and simulation has to be done same as above, you will have animation more assembly and special modeling packages.

Design analysis - analysis package, customized program and packages. Design optimization - customized application and structural optimization can be done, these are the requirement of the CAD tools. Design evaluation for dimension and tolerance. So, there is a big scope in dimensioning and tolerancing, and bill of materials and numerical control. Design communication and documentation will be drafting and detailing will be there. So, these are the design phase and these are the tools which are involved in the design phase or these are the tools required for CAD implementation.

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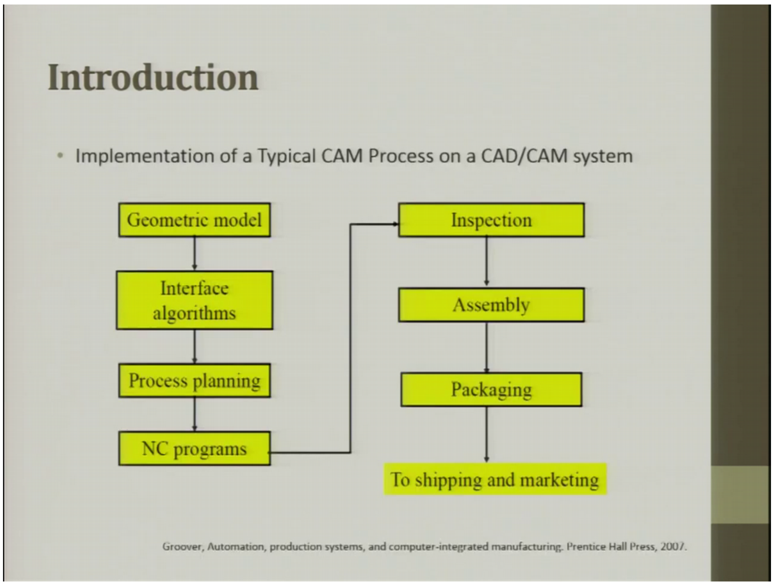

When we have to implement a CAM process? CAM process is whatever you have developed in the CAD, you do a feature recognition, each feature has a process attach to it, then this process are chosen, the machine is activated. So, the machine knows what, what it has to do like what process parameter, what is the tool path and then it executes along with it to get an output is the CAM process. So, geometric modeling, interface algorithm, process planning is done, NC programs are generated, then you inspect, assemble, package and ship. So, these are the implementation of typical CAM process on a CAD CAM system.

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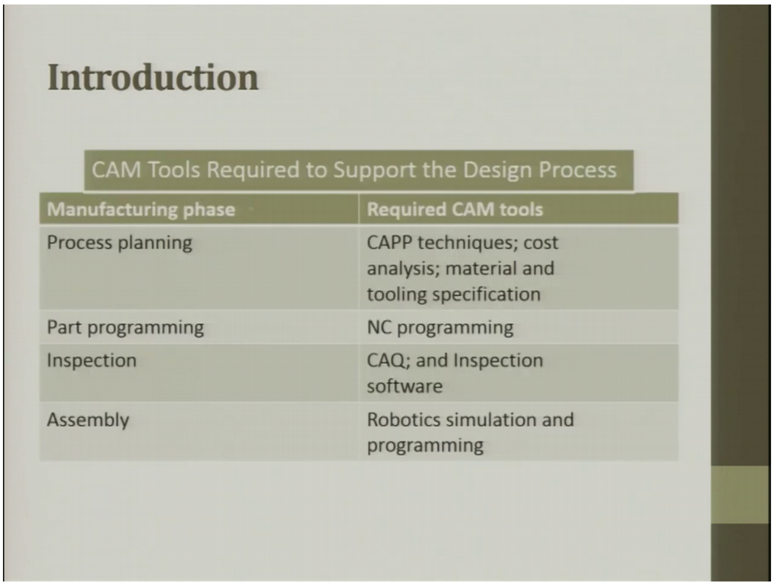

In the same tool, what we saw in CAD; if you want to do a CAM tool requirement to support the design process, manufacturing phase will be a planning process planning, part programming inspection and assembly. So, in process planning, we have computer aided process planning. So, computer, recognizes the feature, generates the process plan and tells you costing analysis which can be taken from CAD, material and tool specifications, these are all part of process plan. Then part programming - numerical programming is done; inspection - computer aided quality control and inspection software; assembly - robotics simulation and programming or some of the CAM tools which are used for CAM environment.

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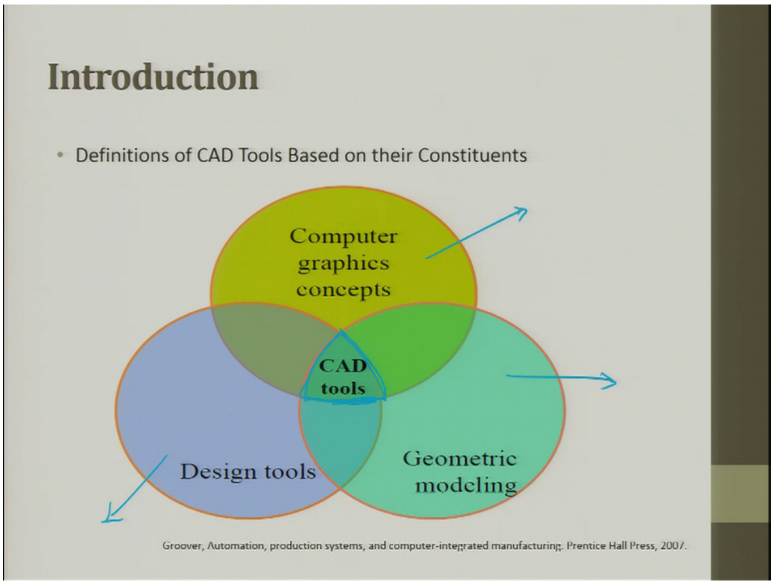

So, if you look at it computer graphic and concept, geometric modeling is the separate circle, and design tool is the separate circle. This can happen independent, independent, independent, this is just which a computer alone, this is the model alone, and this is the design alone. So, if you see CAD tool is a combination of A intersection B intersection C. It takes a common zone of the three things in the CAD tool. So, takes the position here. So, in the CAD tool, that means, to say you will have computer graphic concept, you will have geometric model, you will have design tools.

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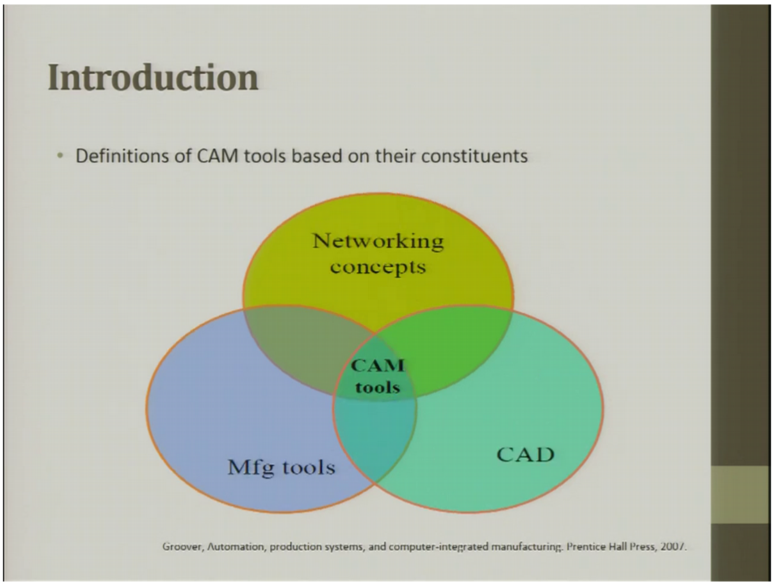

When we talk about a CAM tool based on constituent, networking concept, CAD and manufacturing, all these things meet at a point that is called as a CAM tool, all the CAM tools fall here. So, you should have a networking; networking concept between two machines today we have wireless, wired. So, networking, then you should a CAD under machine tool. Machine tool or manufacturing tool is the machine basically we are talking about.

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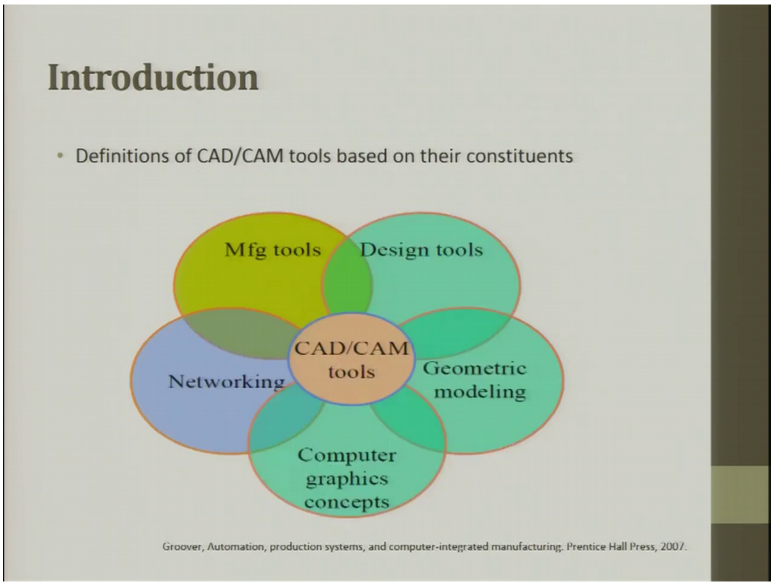

So, when we talk about CAD/CAM tool based on the constituents, it is design tool, manufacturing tool, networking, computer graphics, geometry, all of them needs at a point which calls as CAD/CAM and tools.

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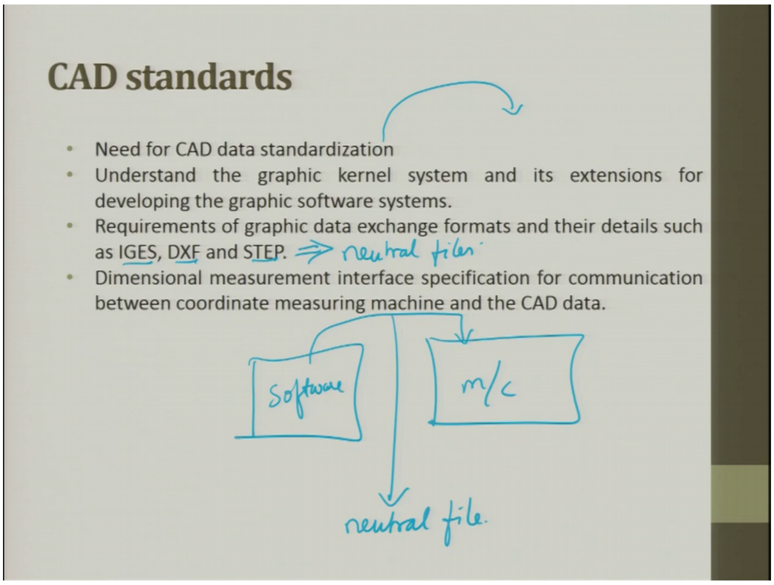

CAD standards; need for CAD data standardization is very important, so that this CAD data whatever gets develop can be transferred to another system in a unified manner. So, there is a need for CAD data standardization. Understanding the graphic kernel system and its extension for developing the graphic software systems so, you should know what is the minimum hardware required to run the software. Requirements of graphic data exchange format and their details in IGES, DXF and STEP, these things are called as neutral files. What are neutral files? You have a system, you have a software, you have another software or a machine. So, in this if you want to transfer the data to this machine without any seamlessly without any inference, so then this is called as a neutral file.

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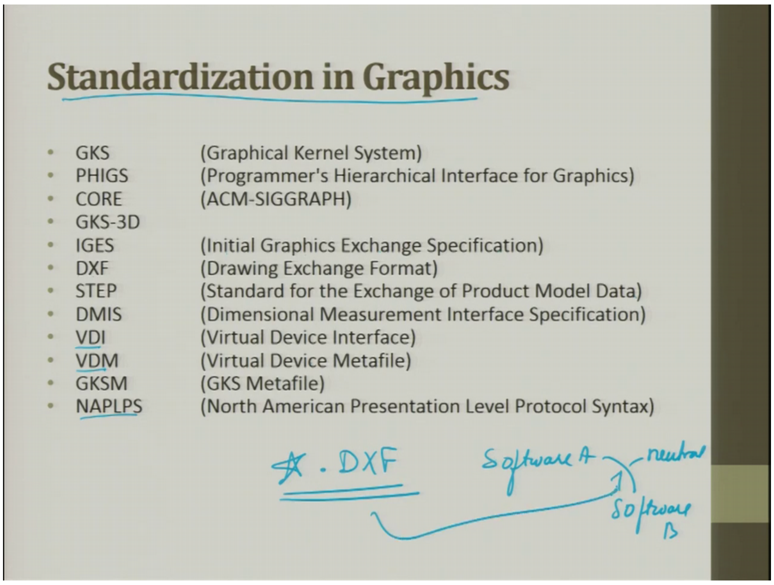

So, these are some of the standardization in the graphics, GKS - Graphical Kernel System. You also have a IGES - Initial Graphic Exchange Specification. DXF - Drawing Exchange Format, standard STEP - Standard for The Exchange of Product Model Data and we have VDI - Virtual Device Interface, VDM - Virtual Device Metafile, and nap NAPLPS - North American Presentation Level Protocol Syntax. So, all these things are standard sorry neutral files which are use to transfer a graphic data into your manufacturing data.

So, when you when you draw a component, after finishing the component you can store the extensions star dot DXF. So, now, this becomes a neutral file such that when you move from one software; software A to software B to process the data, then you need a neutral file. So, all these things are part of neutral file.

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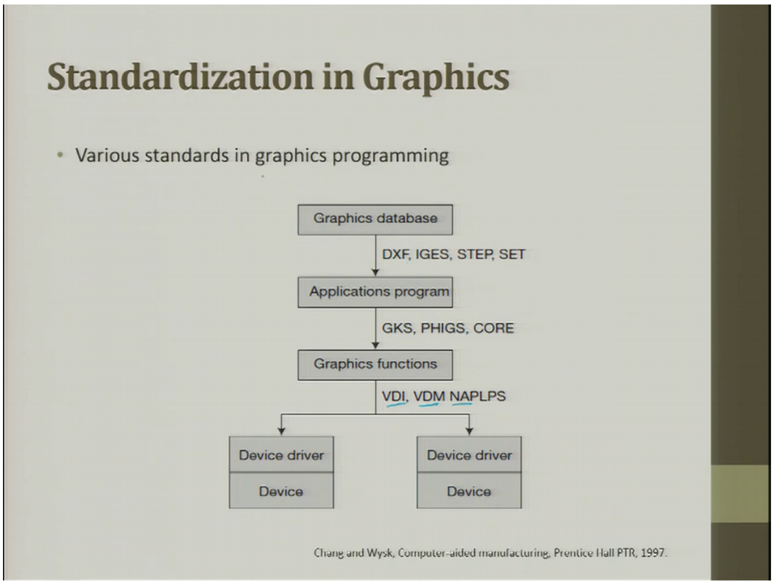

Various standards in graphic programming; so you have graphic data base, where in which you use this neutral file DXF, IGES, STEP and SET. Then we have application programs where in which GKS, PHIGS, and then CORE you have. We have graphic functions which are VDI, VDM and NAPLPS. So, then this will be further divided into device driver, you can have two different devices which will all get linked to get the graphical program.

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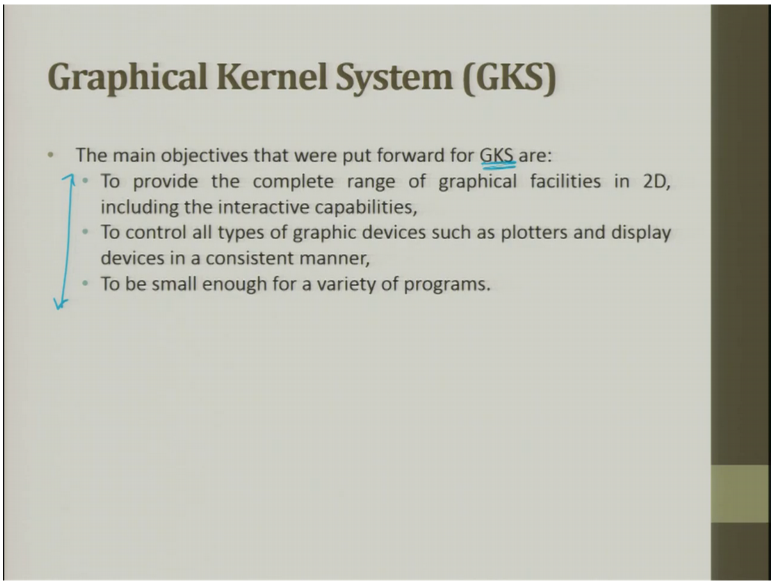

GKS which is Graphical Kernel System: the main objective that were put forward for GKS are to provide a complete range of graphical facilities in 2D, including the interactive capability. You can edit, move and do to control all types of graphic devices such as plotter and display devices in a consistent manner. So, this was the requirement for GKS. GKS again is a transfer of data from a CAD system to the device. To be small enough for a variety of programs. So, these are the requirements which was put, so that the data can be transferred from a CAD file form a CAD system to a plotter or to another accessory or to a machine.

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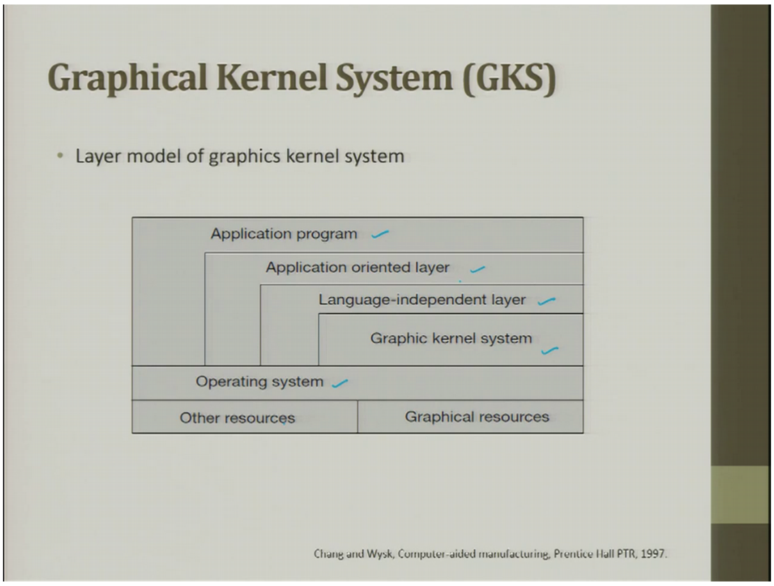

The layer model of graphic kernel system is you have application programming, then application oriented layer, then language-independent layer, then graphic kernel system. You have an operating system which is at the base. In the operating system, again you have to other resources and graphical resources. So, these two are added to the operating system, then graphic kernel system, language independent layer, application oriented layer and application program.

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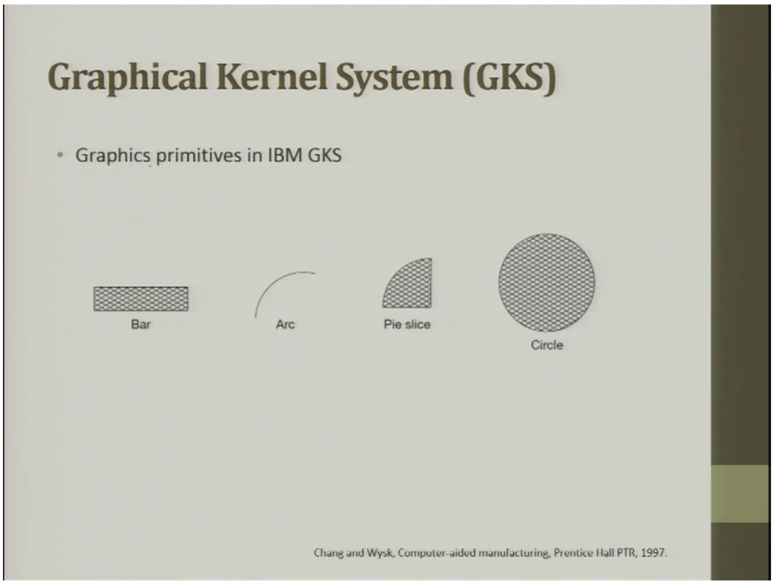

Graphic primitives in IBM GKS are bar, arc, pie slice and circle, these are all primitives which are already available in the graphic kernel system.

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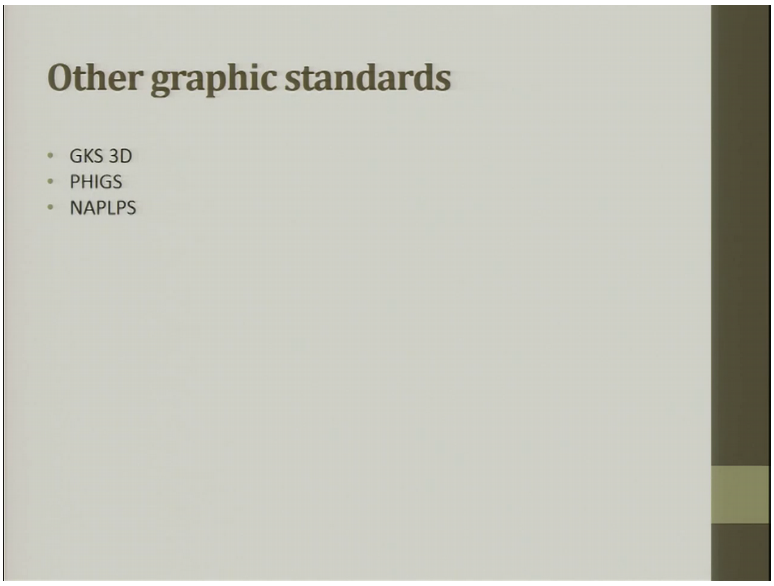

Other graphic standards are GKS 3D, PHIGS and NAPLPS.

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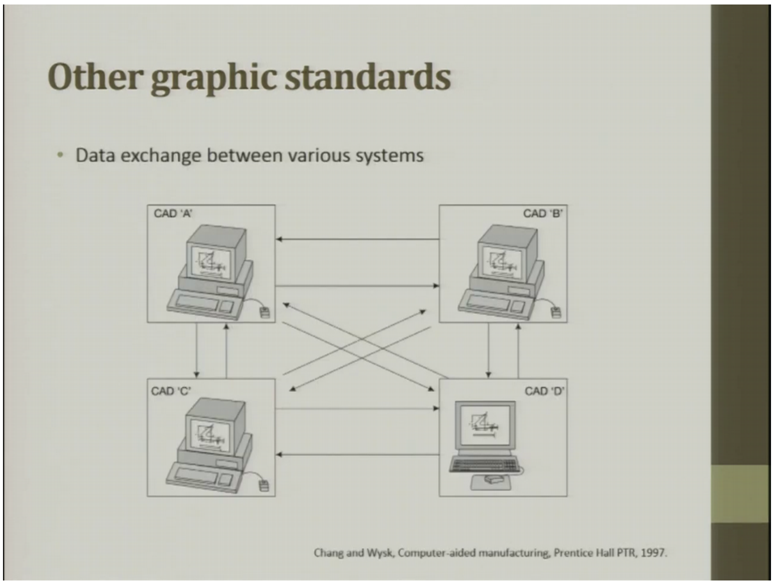

So, the data exchange between various system is CAD A to CAD B, CAD B to CAD C, CAD C to CAD D. So, you can have AB, BC, CD, DA. So, you can see all these things are linked. So, the data exchange between the various systems is very important; otherwise in manufacturing what will happen is you will have islands of infirm automation, but all these automations cannot talk to each other. If it cannot talk to each other, then there is a big difficulty, every time the data has to be transferred in to the other system manually, and then you start doing some editing process.

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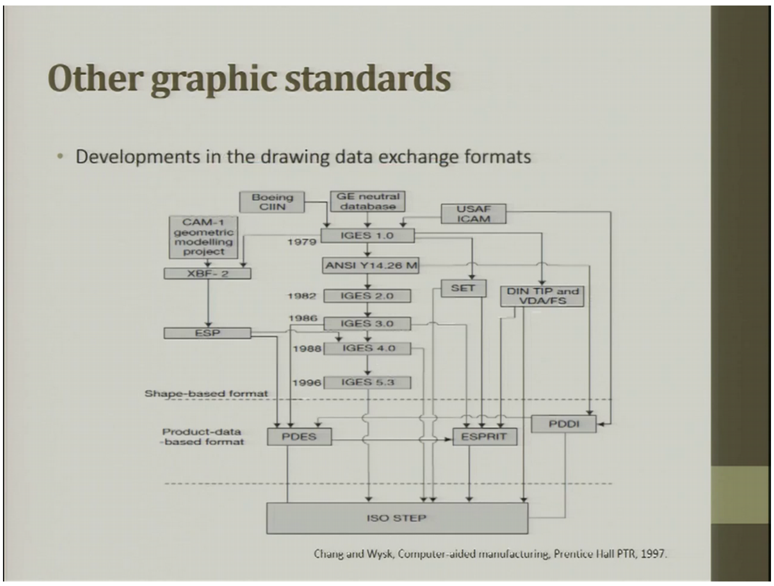

So, development in the drawing data exchange format these were the these are the years when it was started. And here are the contributions made and this is how the evaluation happened. So, IGES 1.0 came, then ANSY SI Y 14.26 M came. IGES 2 protocol, IGES 3 protocol, 4 protocol 5.3 protocol which came in new it is 1996. And this let to ISO STEP. And in between if you see IGES 3.0 lead to PDES which is nothing but product- data-based format PDES, and PDDI, they all came into existence. So, here if you talk about it is shape based format drawing initial shape based format, then it is product data based format. And then finally, from all these evolution, today we talk about STEP, which is used where in which the process details are also attached with the feature whatever is getting generated.

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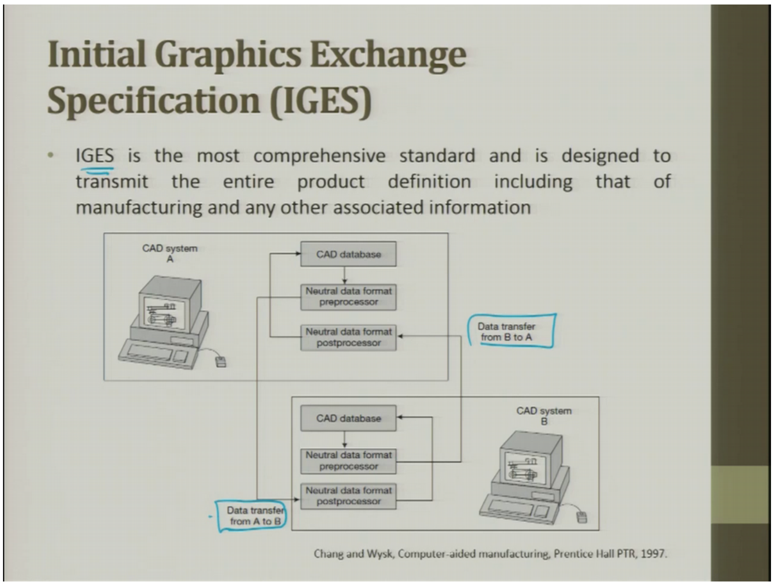

IGES is the most comprehensive standard and is designed to transmit the entire product definition including that of a manufacturing and any other associated information. So, IGES, again a neutral file, DXF, IGES, GKS all are neutral files. CAD system, CAD data base, neutral file format, you need a preprocessor, you need a postprocessor. First what all has to be changed it preprocessed. Once the process data is done, then again it has to be post processed to get the requirements. So, for every CAD system, you will have the same. This is CAD B. You had a CAD data base. You had a neutral data format preprocessor. So, this gets into the postprocessor, then the neutral data format of the this is there. So, this in turned will be linked to the CAD database ok. So, the data transfer from A to B is there. So, it happens from a to b or the data transfer happens from B to A.

The only difference is if you say B to A neutral data format preprocessed gets into the post processor, so that the data is understood by the other CAD system in a very logical manner.

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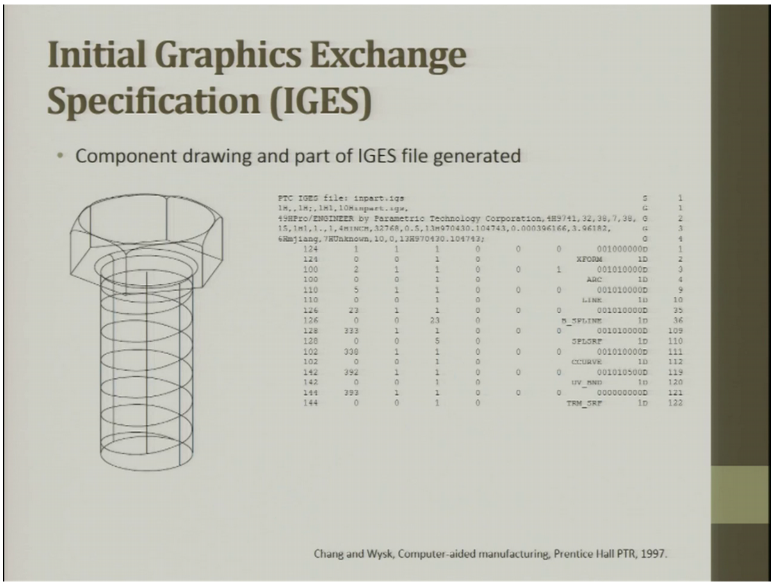

So, this is how an IGES file looks like. A component which is here bolt with a threading with a head, component drawing at a part of IGES file generator, you will see stake of data's the column of data's where in which it talks about an arc, line, b_spline, and then you have a curve, you have a other surfaces. So, this is how the output looks like of IGES which is preprocessed, post processed such that the other system can understand.

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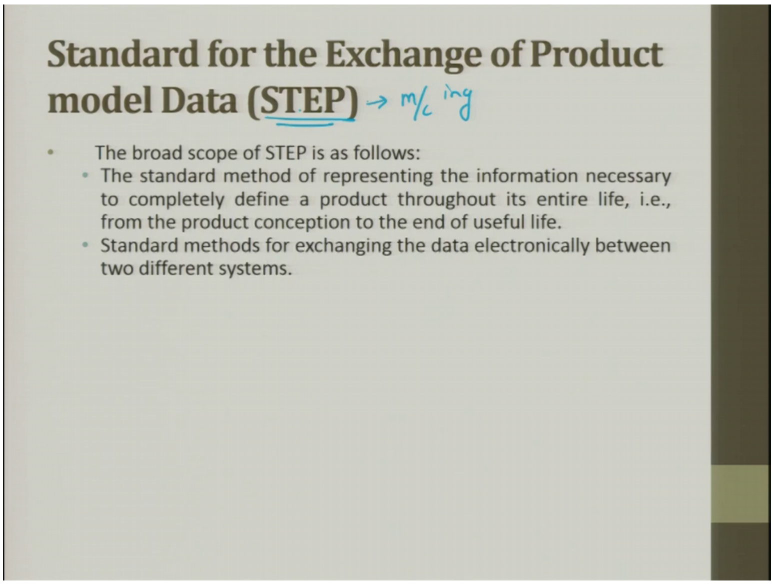

The latest standard which we saw ISO, STEP, STEP is nothing but standard for the exchange of product model data. So, today we are not trying to transfer the data from one CAD system to the other as individual features or individual entities, but we are trying to completely shift it like a product itself. Like a writing a small library function a library function is already written. In this library function, all the time you just put some parameters and you change the requirements whatever you want. For example, you have a library function of making a bolt. So, then all you have to say is what is the pitch you need, what is the length you need, what is the head you need, all those things. So, then the system itself generates.

So, STEP, what will happen is you will have other details like manufacturing tolerancing and other details will also be attached along with the entity details. A broad scope of STEP is as follows. The standard method of representing the information necessary to completely define a product throughout its entire life that is from the product conception to the end of the life can be done by STEP. The standard method of exchanging of data electronically between two difference systems is done using the STEP.

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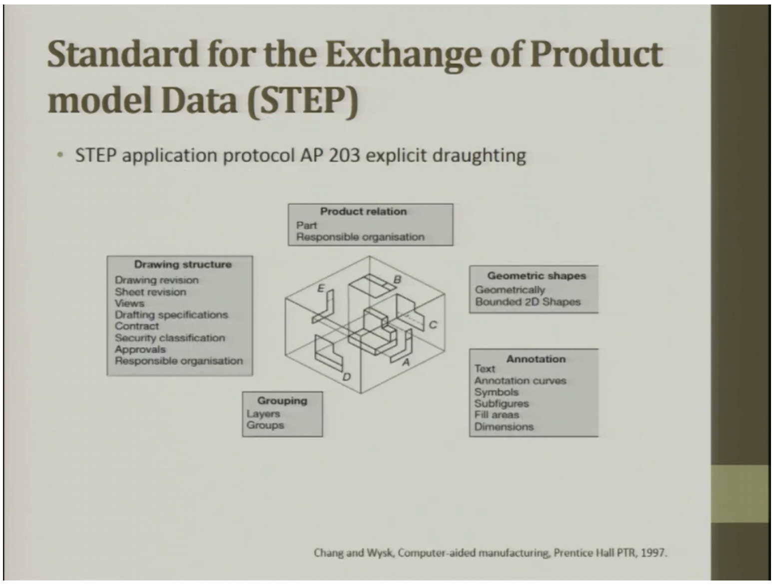

The STEP application protocol AP 203 explicitly explicit for draughting. So, you can see here product relationship part responsible organization. So, A, B, C, D, E, you look at a part. So, you will get projector in different flames. So, you get B, C, A, D, E all are projections of the same object into multiple walls or multiple phases. So, you have so many informations getting out of STEP. So, product realization relation, part responsible organization, then geometric say geometrically bounded 2D shapes, then annotation text, annotation curve, symbol, subfigure, filling area, dimensions, then we have grouping of them. Then we have drawing structures, drawing revision, sheet revision, view, drafting specification, contract, security classification, approvals and responsible organization.

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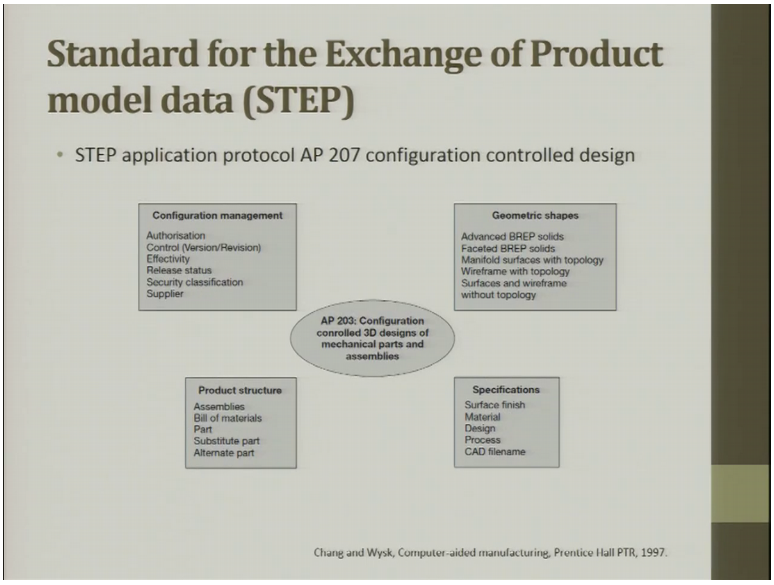

When we talk about controlling controlled design, AP 207 come into existence you will have configuration management shape. So, here you can see advanced BREP. We will see what is BREP as an when we go down. Then we see product structure, we see simplification. Today when we talk about the softwares which are ERP softwares where in which product structure bill of materials, all these things can be attached directly to the CAD file, and you can extract and then further information can be send to different, different departments of the manufacturing. So, STEP application protocol AP 207 configuration, control design is this.

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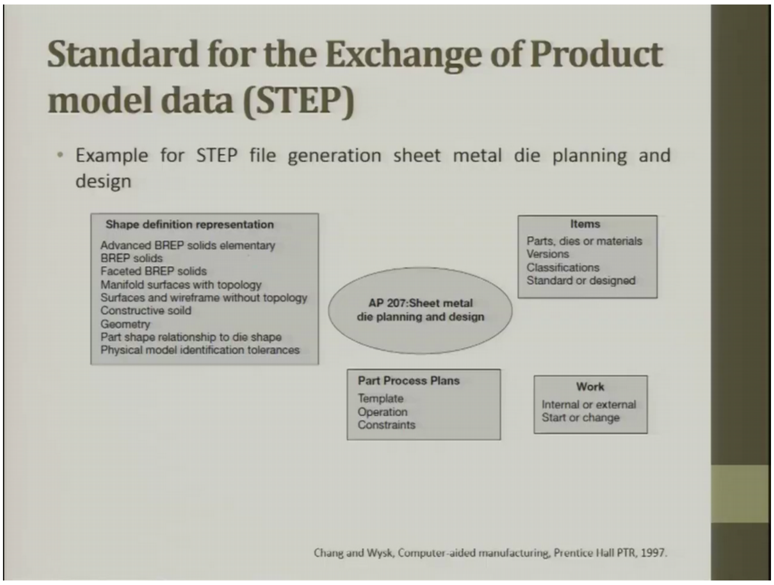

Here is another example for STEP file generation sheet metal die planning and design. So, again shape definition item, and then you will have process plan and then you will have whether the work is internal, external start or change.

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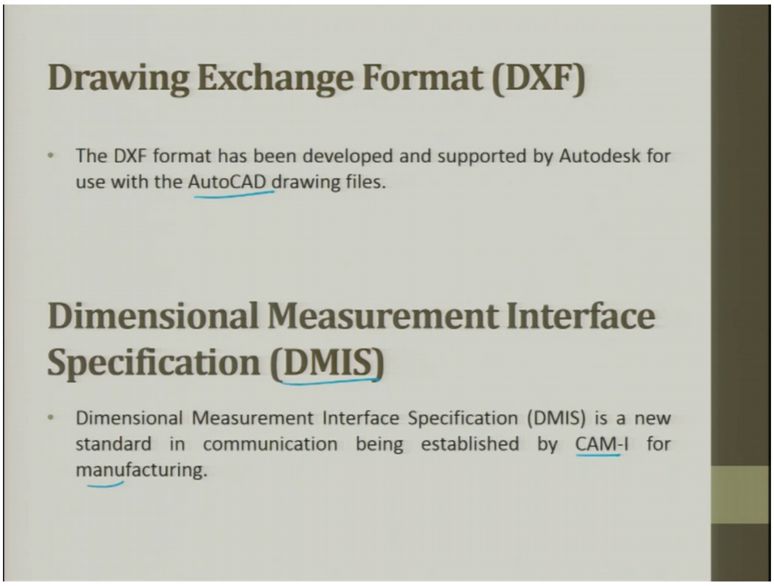

DXF we saw which is used by AutoCAD. The dimension measurements interface specification - DMIS is a new standard of communication being established by CAM-Ifor manufacturing. So, now, initially DXF and IGES is only to transfer the data from one CAD file to other CAD file, but moment STEP came into existence it is used to transfer a CAD data to a CAM environment and with more and the protocol is more and more and more focus towards manufacturing where in which dimensions, tolerances, the product life cycle everything can be done.

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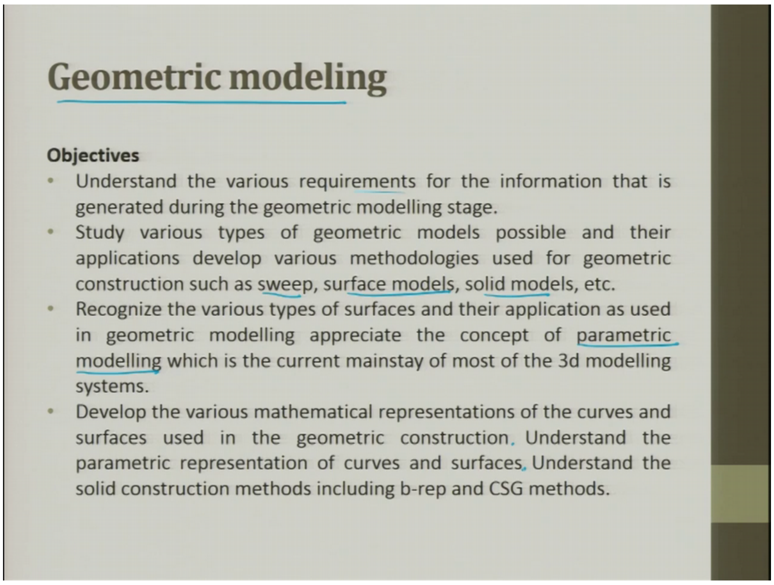

So, now let us move to a new topic which is on geometric modeling. How do I draw a part in a CAD file, that is geometric modeling, so I will try to model a part in a CAD file. Objective: here to understand the various requirements for the information that is generated during the geometric modeling stage. Study various types of geometric modeling possible and their application develop various methodologies used for geometric construction such as sweep, surface modeling and solid modeling. Recognize the various types of surfaces and their application as used in geometric modeling appreciates the concept of parametric modeling, parametric modeling which is very important we will see later which is the current mainstay of most of the 3d modeling systems. Develop the various mathematical representations of curve and surfaces used in the geometric construction. Understand the parametric representation of curves and surfaces, and then finally, understand the solid construction by including b-rep and CSG.

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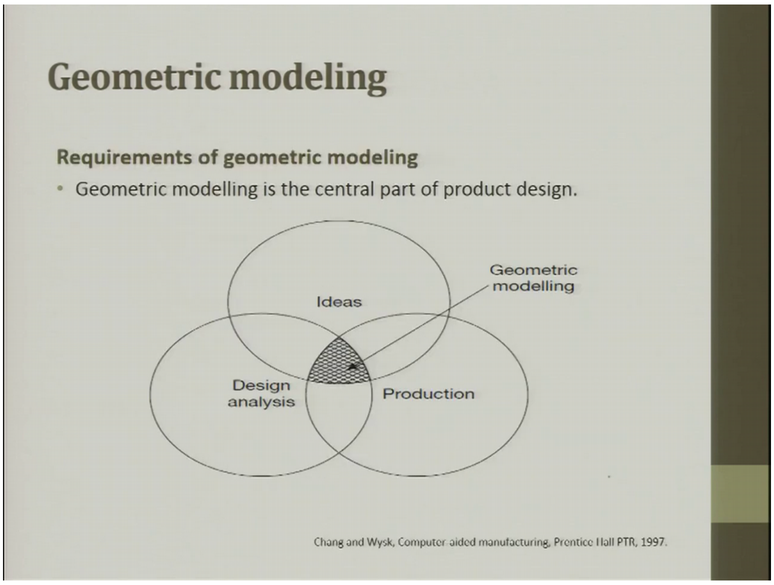

When we talk about geometric modeling, the requirement for the geometric modeling is you have to have an idea, you have to have a production, you have to have a design analysis. The center part is the geometric modeling. The geometric modeling is the center part of the product design.

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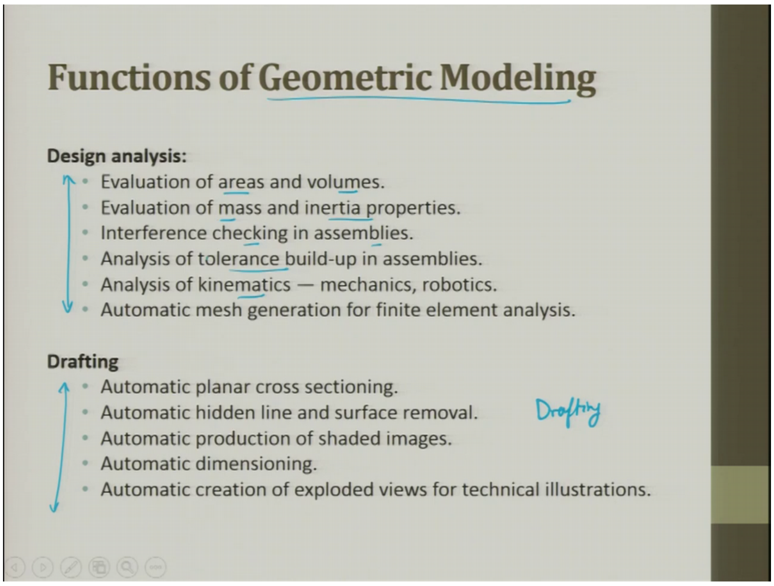

What are the functions of a geometric modeling? In the design analysis it is used to for evaluation of area volume, mass inertial properties, checking in assembly, tolerance building in assembly, analysis for kinematics that is mechanics and robots; it is also use for auto mesh generation for finite element analysis. All these things are done under design states design analysis. Then when we start using this for drafting geometric modeling for drafting, we automatic planar cross sections we can get, automatic hidden line and surface models, automatic production of shaded images, automatic dimensions and automatic creation of exploded view for the technical illustration.

All these things are functions of a geometric model. Here it is more of drafting, so that is only drawing that is why you see cross section you see creation of exploded views and all those things. Here if you see area, volume, CG, mass, all these things can be evaluated from the design analysis.

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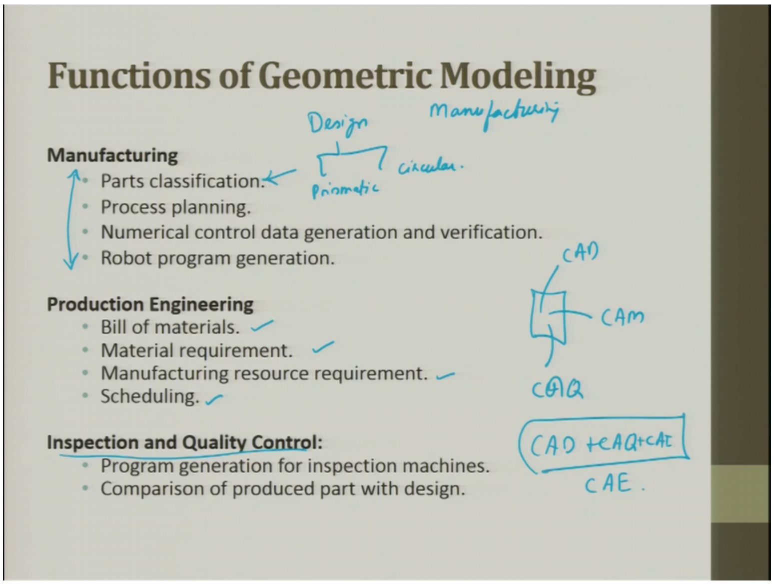

When you talk from the manufacturing, the geometric model is used for part classification. What is part classification? Part classification can be done based on the design similarity or in the manufacturing similarity. Design again can be divided into prismatic, and it can be divided into circular. So, there basically if you see later we will say about group technology. If I wanted to classify the part and give a part number to it, the geometric modeling is the very important role that is the input data. Then process planning, numerical control data generation and verification, and robot part generation. So, these are the functions of geometric modeling in manufacturing.

When we talk about production engineering, bill of materials, materials requirement, manufacturing resource requirements and scheduling, all these things are functions of geometric modeling. And finally, when we talk about inspections and quality control, the program generation for inspection machines and comparison of produced part with the design are part of inspection and quality control.

Today, what we talk about use the same center data what you have created for CAD to CAM, CAM to CAQ - computer aided quality control you check. So, now what we talk about is we talk about CAD, CAQ, and all this whatever it is CAI, it is all leading towards CAE, CAE - Computer Aided Engineering.

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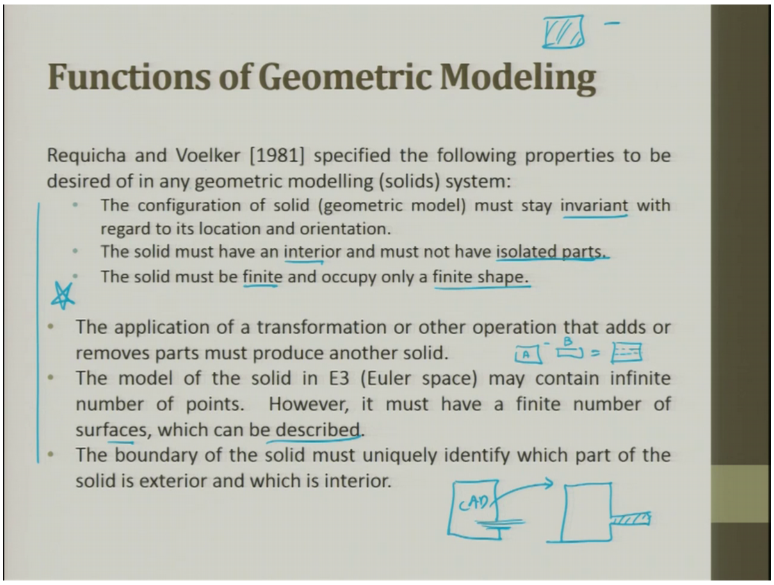

The Voelker specified in the following properties to be desired for or desired of in any geometric modeling system. The configuration of the solid may stay invariant with regard to its location or orientation. Whatever orientation you take, whatever location you take, you view from the front, you view from the top the geometric models should have a consistent figure, there should not be any variance. The solid model have an interior and must not have isolated parts, must have an interior solid must have an interior and must not have a isolated part. For example, you cannot have something like this and say these two are these two are interlinked, no.

The solid must be finite and occupy only a finite space. So, if this it should it should be a close to volume, and it should occupy a finite space that is why if you go back and see if you can do mass calculation, inertia calculation, volume calculation, CG calculation, all these things could be done it must have a finite and occupy only a finite shape. The application of transformation or other operations that adds or removes part must produce another solids. So, if you have a solid like this and you have a cylinder like this, you say A minus B, you should then you should get something like this an object with a whole inside. A model of the solid in Euler space may contain infinite number of points. However, it must have a finite number of surfaces which can be described in this, all these points are very very important to validate an object or to put it as functions of a geometric model.

The boundaries of the solid must uniquely identify which part of the solid is interior and which part is exterior. All these things are very, very important for ask in these curves, why, the CAD whatever you create if it does not follow the functions of a geometric model, then this data will be transferred to the rapid photo typing machine. And the rapid photo typing machine, it will start executing. Moment it executes if it does not identify interior and exterior, it is too difficult for it to build a part.

Second thing if it does not form a finite volume for example, if you say that I have a square like this, this data is given to rapid photo typing, this is a CAD data. When it goes to rapid photo typing it will try to generate a part like this, because here since it is not closed, it takes rest of everything is have is a valid object. It continuous and makes this projection as part of the rapid photo typing process. So, geometric modeling is very very useful, the functions of a geometric modeling is very useful or the properties of the geometric modeling is very useful as far as rapid manufacturing is concern.

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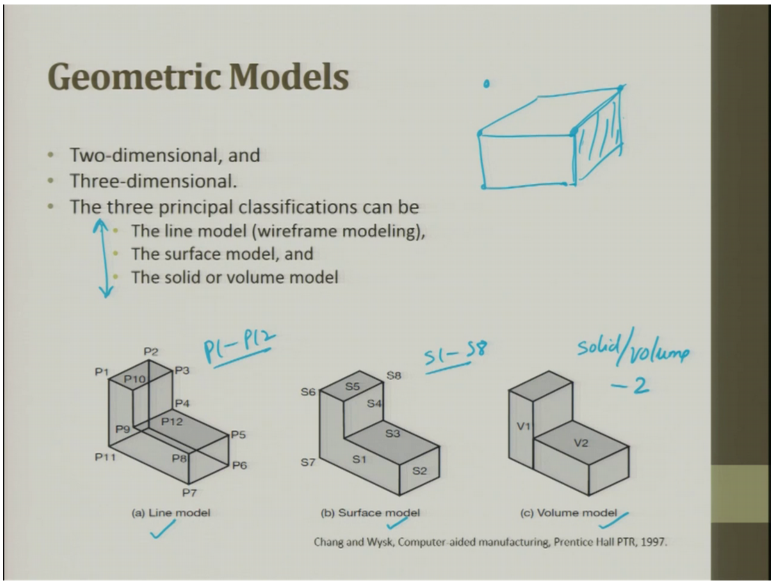

Any mistake here when it is taken to the next state gets amplified. So, geometric modeling can be two-dimensional, three-dimensional. When we talk about three-dimensional principal classification, they fall under line, surface and solid. So, if you look at geometric modeling, I have a dot, dot under dot leading to a line, dot under dot vertices, vertices to edge, edge right, these are edges, and these are faces and several of these faces joint together to form a solid. So, point, point to line, line to surface, surface to solid. So, these are the three principal classifications in geometric modeling. So, you can see here P 1 all these things are points P 1, P 2, P 3, P 4, P 5, 6, 7, 8, 9, 10, 11, 11, 12, so these are so P 1 to P 12 are the points. So, these points are joined by a line.

So, now, these points and lines join together to form a surface. So, you had P 1 to P 12 now you have S 1 to S 8. So, S 1 to S 8 are surfaces S 1, S 2, S 3, S 4, S 5, S 6 at the back side S 7 at the bottom and S 8 at the other side. Now, this surfaces from 12 points 8; it is now converted into volume or a solid. So, you have a solid or a volume which is only two, that means, to say point, line gets converted into surface, several surfaces or faces put together jointly to form a volume or a solid. So, this is line model, this is surface model, and this is volume model. So, in geometric modeling, you will have all the three possible things.

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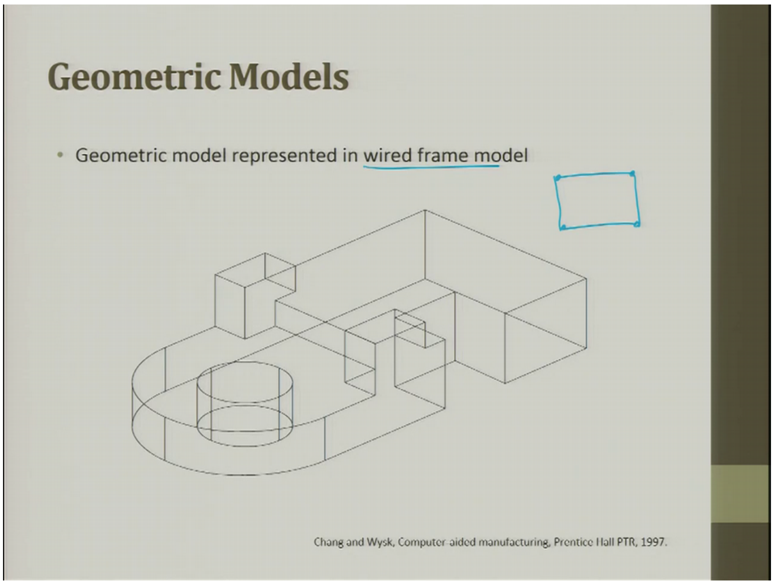

In geometric modeling when you represented in a wired frame. So, what is the wired frame? Wired fame is nothing but you had points and these points are joined by a wire, and this representation tries to give for a solid. But unfortunately when you use wire frame model, it gives lot of misinterpretations, and it can also give you a solid which is not very valid ok. So, the, but it occupies a very less space. So, editing is editing and storing becomes little easy, but the interpretation representation, it is not ambiguous or I would put it like this it is ambiguous the representation is ambiguous.

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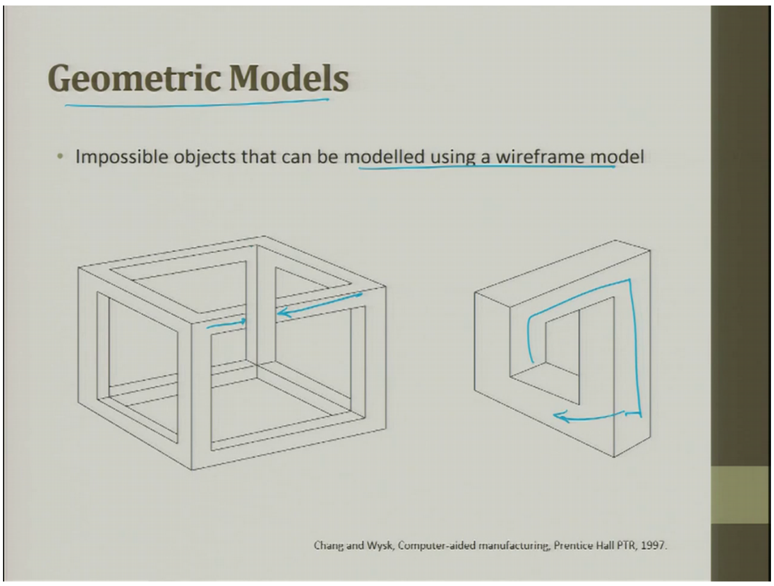

Impossible objects can be modeled using this wireframe. So, you see here. So, this is an object which is like this you have to see this. And here if you see the block or the beam goes behind the column, so geometric modeling wireframe model can be can be used to model impossible objects or it leads to ambiguity, so that is why wire frame is not exhaustibly used.

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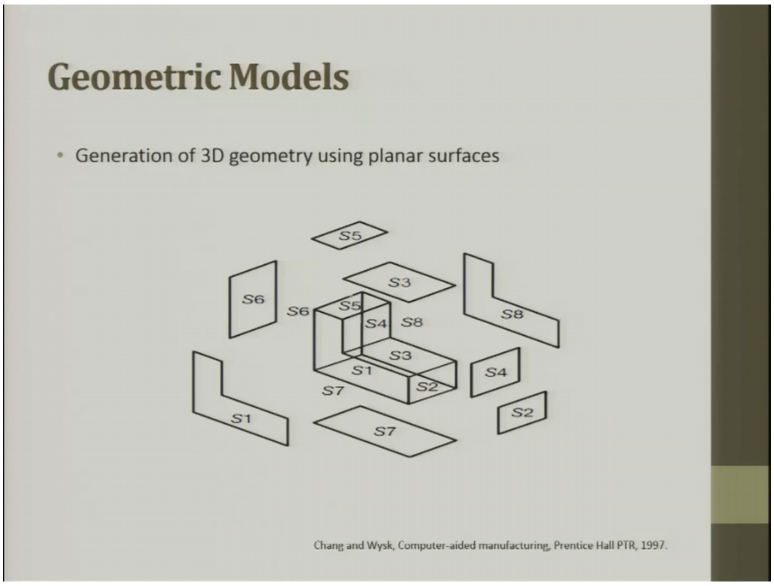

Generation of 3D geometry using planer surfaces; you have so many surfaces, so these surfaces joint together to form a solid.

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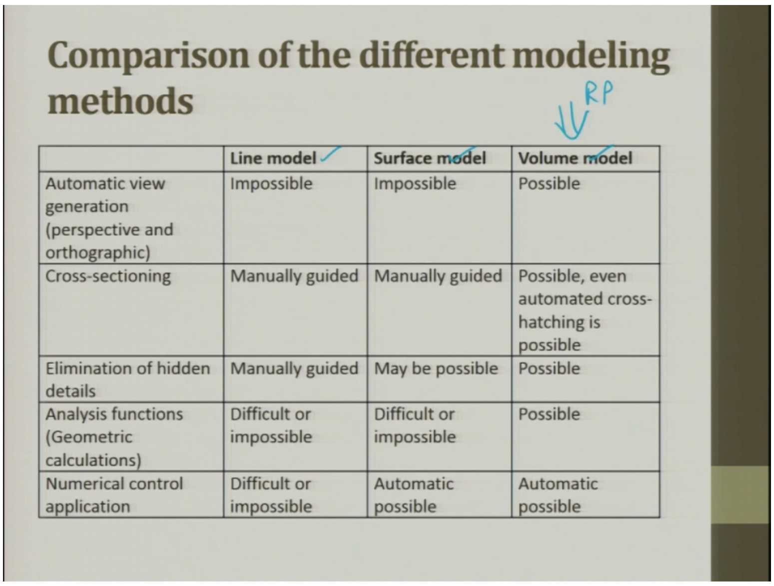

	Line model	Surface model	Volume model
Automatic view generation (perspective and orthographic)	Impossible	Impossible	Possible
Cross-sectioning	Manually guided	Manually guided	Possible, even automated cross-hatching is possible
Elimination of hidden details	Manually guided	May be possible	Possible
Analysis functions (Geometric calculations)	Difficult or impossible	Difficult or impossible	Possible
Numerical control application	Difficult or impossible	Automatic possible	Automatic possible

So, comparing the three different model, line model, surface model and solid model, what we want as far as rapid manufacturing; rapid photo typing is concerned is only volume model, that means, to say solid model. We are least bothered about line and surfaces ok. Automatic view generation - impossible in line model, impossible in surface model, possible in volume model; perspective and orthographic is possible only in volume. Cross section if you want to do, it can it can manually you can guided. So, surface also manually guided, but volume possible even automated cross cross-hatching is possible as far as volume is concerned.

Elimination of hidden details, it is manually guided, here may be possible, here it is [FL] possible. Analysis function difficult, because it is line, and then surface also it is difficult, but here it is possible. Numerical control applications difficult or impossible, here it is automatically possible, here it is automatically possible. So, what we need for RP is RP, RM additive manufacturing is volume model.

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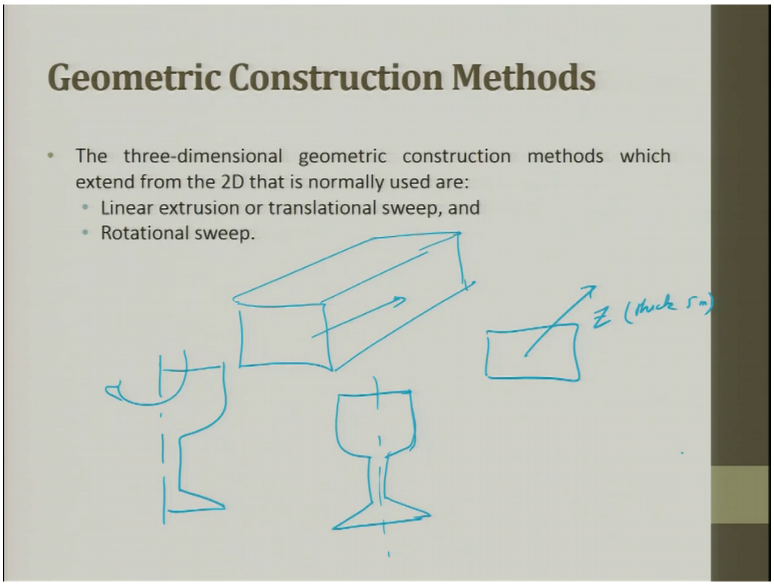

The three-dimensional geometric construction method which extends from 2D that is normally used are linear extrusion and translation or sweep. What are we trying to say is, suppose I draw a square and then I say please extrude the square along this line, now you get a channel. So, if I want to represent this, I can represent it is as a surface. And then I say in this direction is an please try to give a thickness of 5 millimeters or 5 meters whatever it is. So, this is line, extrusion of the square or a rectangle.

Translation sweep, this is also called as sweep, line extrusion or sweep. If you want to do a rotation sweep, so this object when you have a rotational sweep tries to converted into a into a full solid object. This is rotational sweep, so the 3D three-dimensional geometric construction method which extended from 2D. See these first we did only 2D point line and then curve. Now, what is happening from 2D we did some simple operations like extrusion, and rotation, rotational sweep. So, we now what we did, we have started representing 3D objects in 2D form by just giving some small variations that is translation sweep or rotational sweep.

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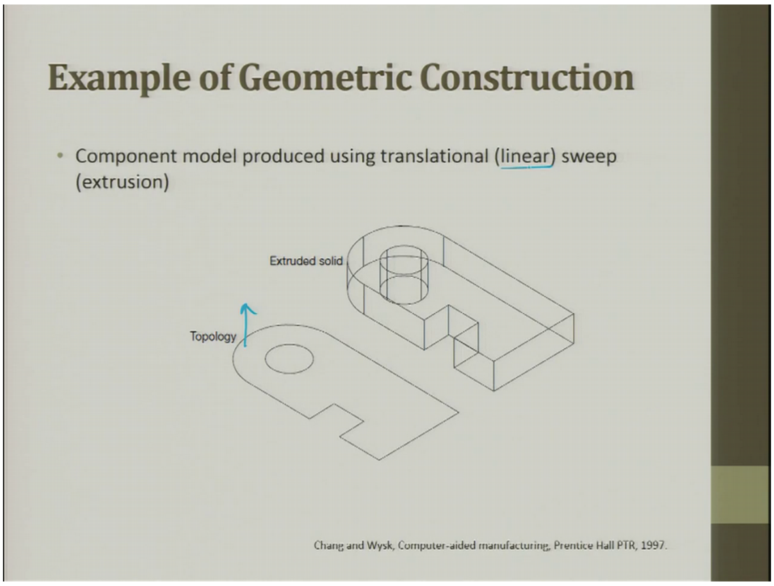

The geometric model produce, so you can see here this is a topology you say please extruded in this direction. So, you get a model. So, linear model and sweep extrusion was tried.

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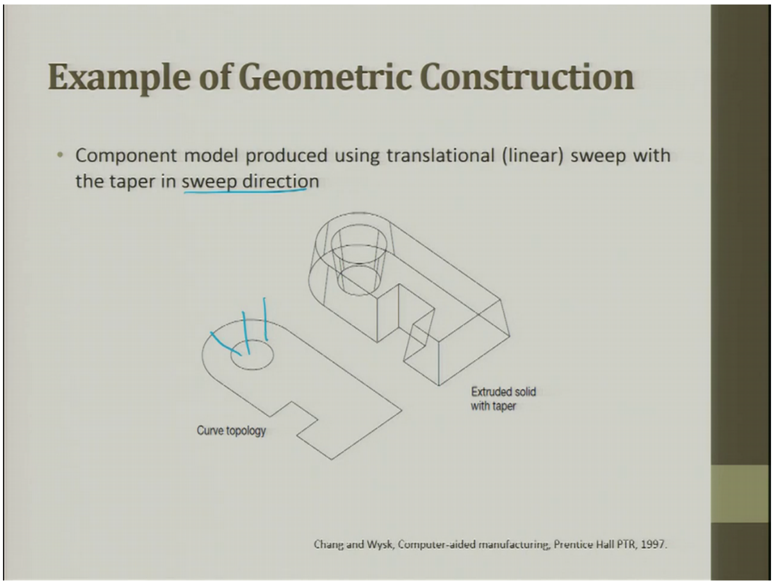

The component sweep produced using translation sweep with taper in the sweeping direction is also possible. You can say this way, so extruded solid with the taper.

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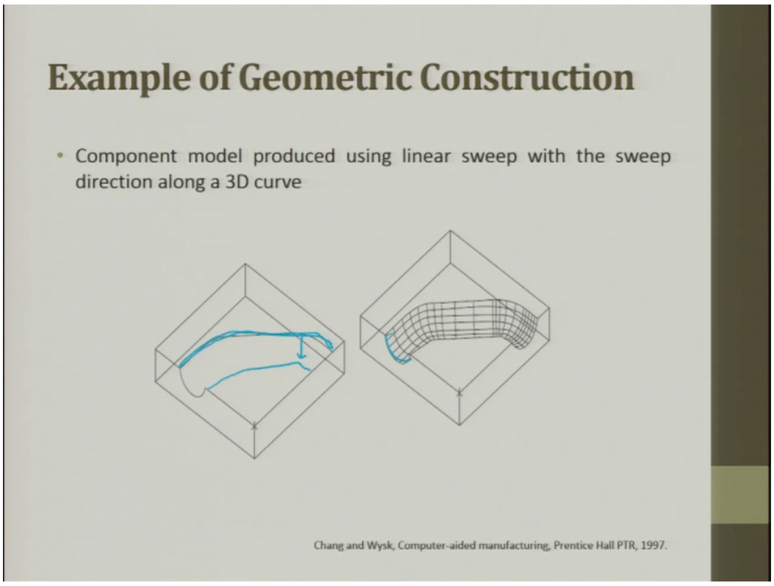

You can see here how did the so this is a curve produced with linear sweep with the sweep direction along the curved surface. So, you can get this is a sweep, and then you say please sweep it in this direction. So, you try to get this. So, basically if you see I define 2D and I get a 3D by doing some operational operations on the object.

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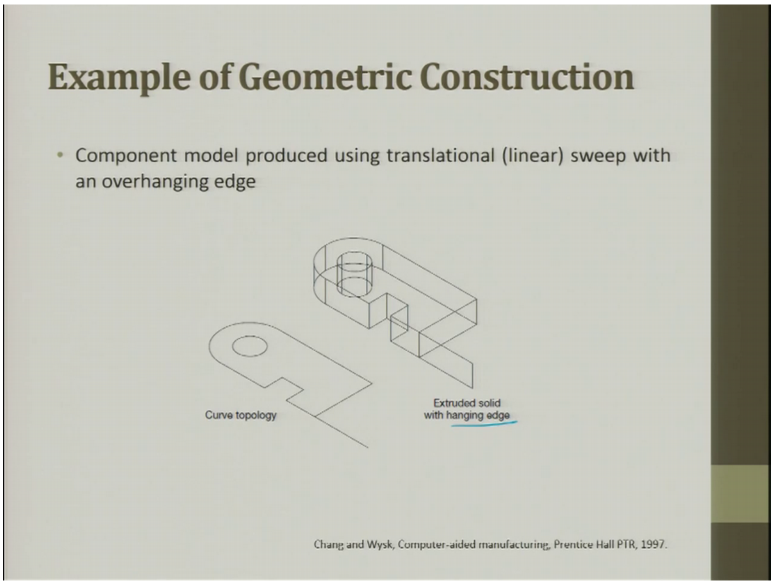

Component model producing using translation sweep with a overhanging edge. You can also try to make an overhanging edge, it is like opening of a lid. So, extruded solid have the hanging edge, we can also get back. So, 2D if I could put it in a crude sense it is 2D, two and half, then 3D. So, 2D to 3D, you are making, and here in between translation sweep and rotation sweep you are giving. So, this I have already dealt.

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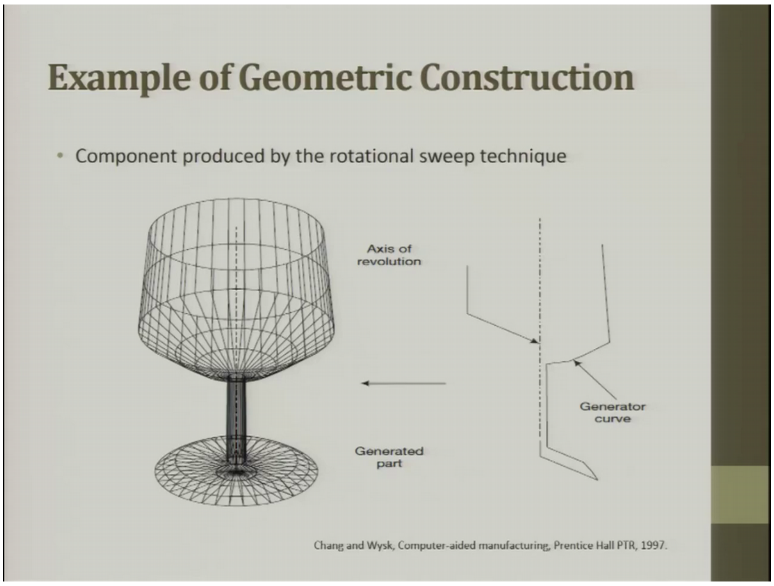

So, the rotation is done about an axis.

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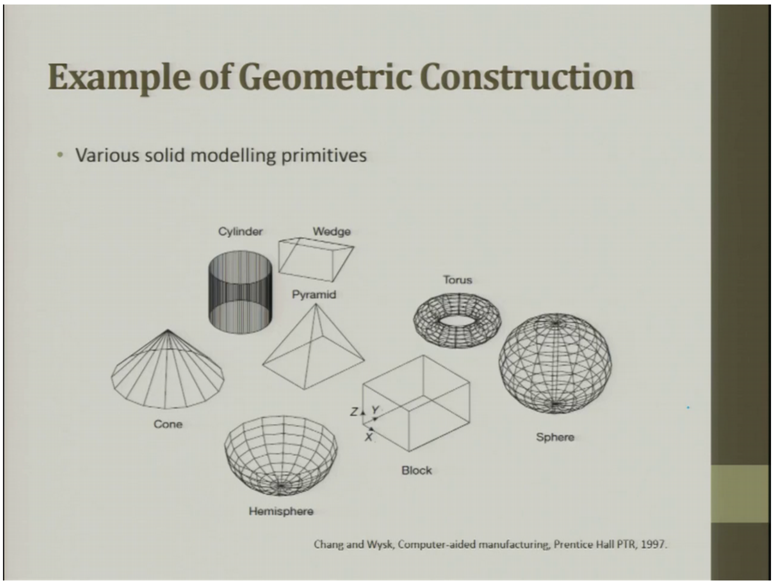

So, these are various solid primitives which are available. If you wanted to understand little bit more, I will put it in such a way such all theses library functions are already available inside the CAD. And today all these things are available in the tool box of even power point. So, all these surfaces are there. All you do is you define the diameter, you define the height to get a cone. You define the cylinder diameter and then you say along the z direction you sweep or you extrude, you translate, so then you get a cylinder. You there is a predefined library function for the wedge. So, all you have to give is give those the parameter, so you get an edge. Wedge to pyramid you can get. Hemisphere you can get; cube or a block you can get; torus you can do, and then you can also get hemisphere.

So, these are some of the solids you see it is how interestingly going, line, model, then you went to surface model, solid model. When you first what you could do you could do 2D, and now you wanted into 3D. So, first 2D, there were small variations like translation and sweep. Now, next stage what you have started doing is we have started making standard primitives which are available where in which a program is written and stored. So, all you have to do is call that primitive and start giving this things, so you get a three-dimensional object. See look at the evolution how it happened in the CAD.

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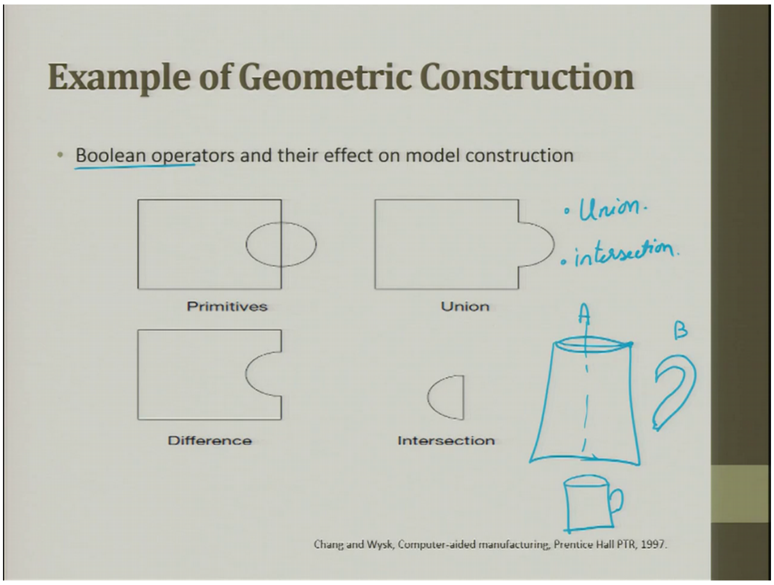

And now with that evolution, people were people were little more smarter, they started adding the Boolean operation. So, what is the Boolean operation? You will have a add right, you will have union or I will put you will have union or intersection; two Booleans. You will have union and you will have intersection. These two Boolean operations are now done. So, now, what is the advantage if you wanted to develop two different objects, and these two different objects are to be connected for example, I have a cup, I have a cup and then I have to attach a handle. So, how do I do? I try to do B, A, and then what I get will be A union B. So, this is A union B operation. So, you see after creating a simple models of 3D, people started using Boolean functions. These Boolean functions are used to develop 3D objects in a more creative manner.

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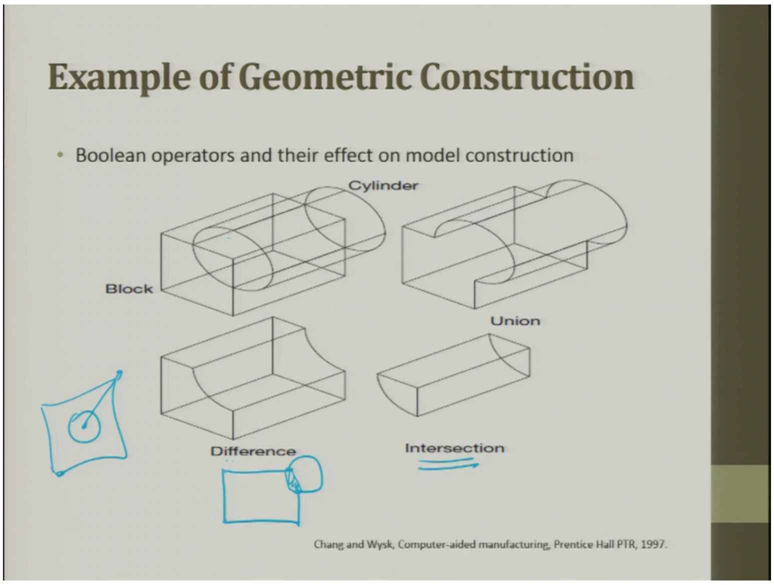

So, you can see here the Boolean operation and their effect on the model. You had a block; you had a cylinder. If you say subtract the cylinder from the block, you will try to get a side view will be like this object. If you wanted to join these two, then it will it will just form a single object. If you move the cylinder to the edge, and then if you do an union, then it will become something like this, something like this. And now you say I want only this portion to be taken or removed. So, then what do I do is I try to put the option of in difference, I get the difference, and then I also try to say which is intersecting get back portion an I am able to get. Now, with these operations, you can try to make several primitives do Boolean operations like union difference or intersection and develop different, different geometries.

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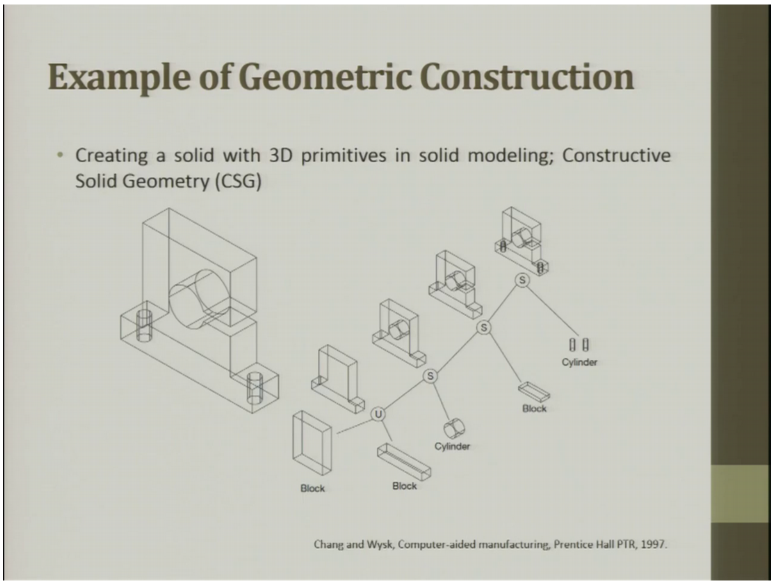

And the creation of a solid with 3D primitives in solid modeling; the constructive solid geometry. So, you can see here CSG is a Constructive Solid Geometry. So, first what did you do; you used the block. So, in this block, you union to one more block you got this. And now you have put a cylinder, you have given the locate for a cylinder, then you say subtract the cylinder. So, subtract, so you got this object. Then what you did, you created a slot, and then you said please subtract it from the block given location, so then this was generator. And then finally, what you did you started putting cylinders and this was put here, and then you start subtracting it from the solid.

So, till now it is a solid. So, you see a solid, solid, solid. And now what did you put two cylinders and put it there and say subtract it. So, now, you see a complex geometry can be created through several simple geometries and by doing the Boolean operation. Constructive solid geometry is follows this route. And again here it is also interesting to note should I follow this route alone, not necessary, these are the objects these are the operations you can do this as first and then you can start going next whatever it is. So, there is not a unique representation of the union and subtraction, but ultimate object will be the same.

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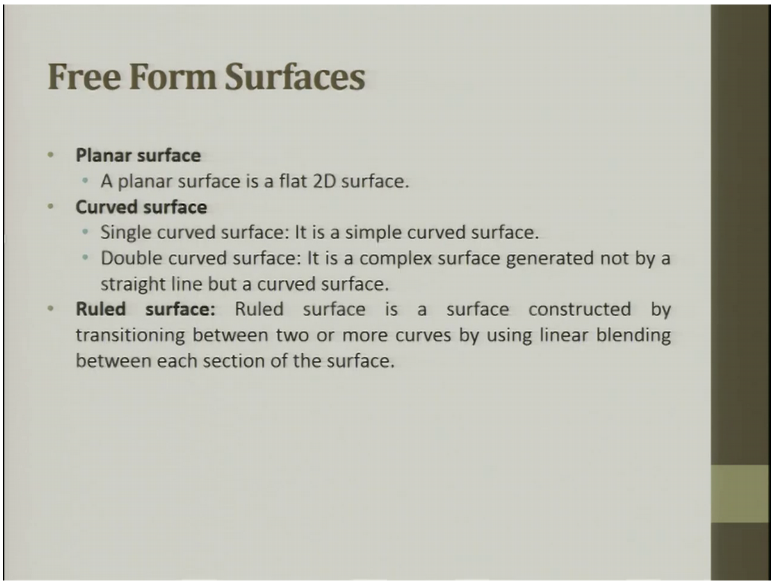

So, till now what we saw was all about engineering surfaces, where it is all easy to draw easy to manufacture. But what happens in real time or what is required in real time application is free form surfaces. So, free form surfaces to construct you need to have more sophistication than whatever was existing till now. So, here again we will follow 2D, we will move to 3D. So, first we will talk about planar surfaces. In planar surfaces, it is a flat 2D surface, then you have a this we saw.

Then next one is curved surface, single curved surface that is a simple curved surface; then double curved surface which is complex surfaces generated not by a straight line, but a curved object. Then ruled surface is ruled surface is a surface constructed by transitioning between two or more curves by using linear blending between each section of the curve. So, ruled surfaces we will see what is the ruled surfaces in the next coming slides. So, these are used for generating free form surfaces. Curve, single curve, double curve, then we went to ruled surface.

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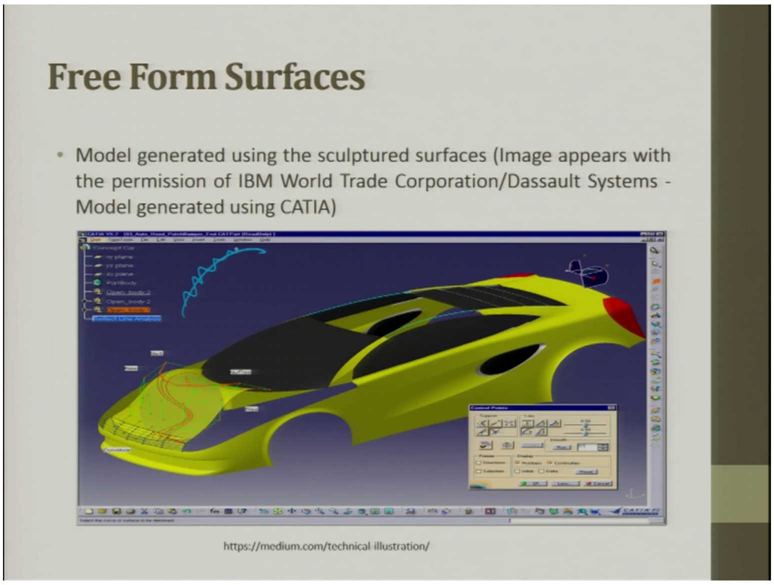

Today, you see in reality it is all free form surfaces. Model generated using sculptured surface this is what is this is energy efficient, the drag is reduced and then here if you see here no edge no surface is straight flat, it is all curve, complex curves are there, and there is lot of other inside a curve itself you have many variation facets.

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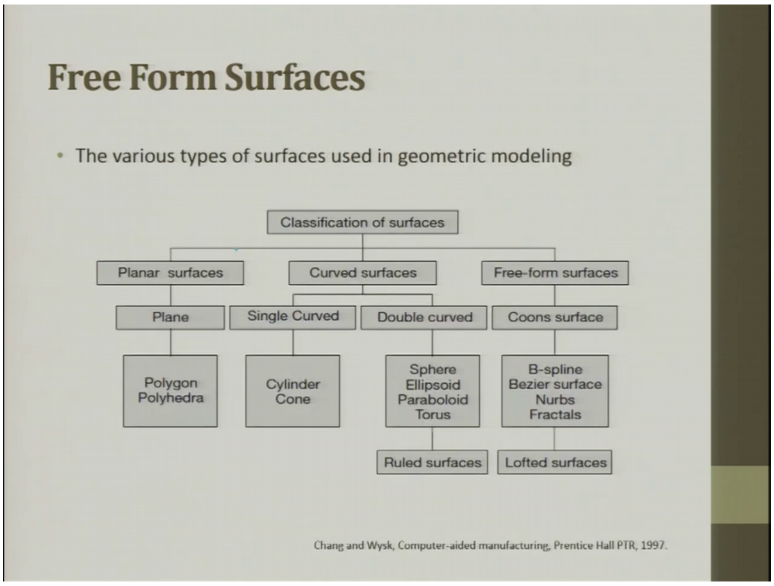

This is done by a CATIA software. When we talk about free form surfaces in a nutshell, these are the various types of surfaces in the geometry. So, we will have planar surface, curved surface and free form surfaces. In planar surfaces, you will have planes, these planes are done by polygons and polyhedral. When we talk about curves, there will be single curve, double curve. Cylinder, cone are part of single curve. Double curve is ellipse, ellipsoid, sphere, paraboloid and torus, these are all double curves. And then you will have ruled surfaces from here it goes down to ruled surfaces.

When we talk about free form surfaces, you will have coons surface you have B-spline, Bezier, Nurbs, and fractals. So, this is very important, and this is what is the big thing which CAD gave it to the manufacturing community. How do I generate free form surfaces? So, by using coon surface in coon surface, we had B-spline, Bezier, Nurbs and fractal, all the nature follows the fractal design, and finally, the lofted surface, ruled surface and lofted surface.

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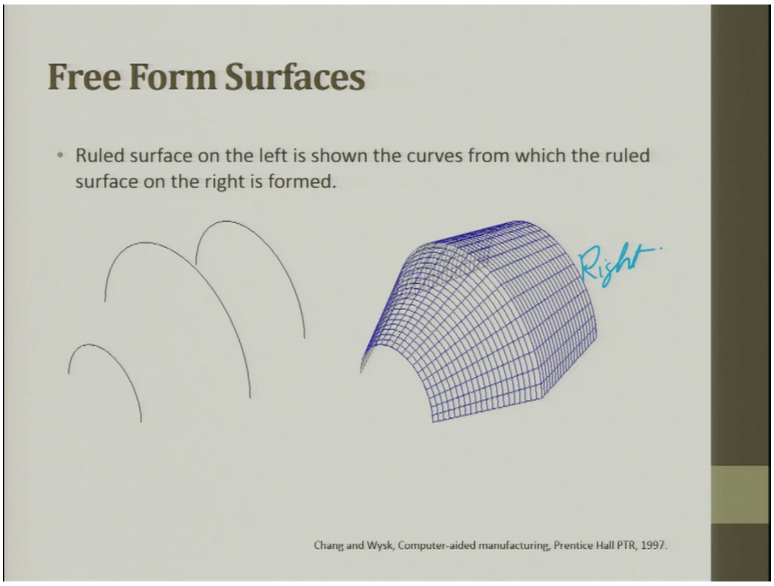

This is a example for ruled surface. Ruled surface on the left is shown the curves from which the ruled surface on the right is form; on the left is shown the curves. These are the curves from which the ruled surface on the right is formed. This is the right side form.

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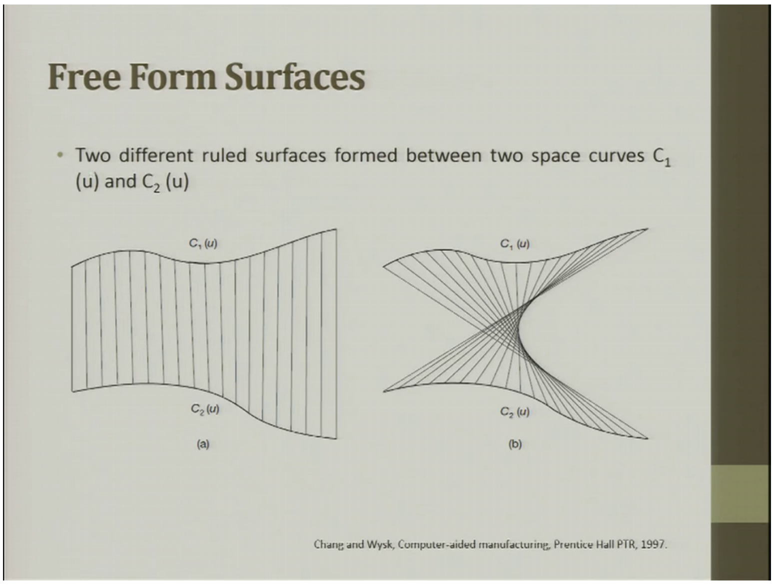

So, when we have two ruled surfaces formed between two space curves c 1 and c 2 with you as a control point or the vector. So, we will see c 1, c 2, u and u. And now you see when we try to go in the other direction, you will try to get a surface like this c 1 and c 2. So, two different rule surfaces formed by two space curves c 1 and c 2, both curves, the all you have to do is we the way you try to specify you can try to generate two different surfaces.

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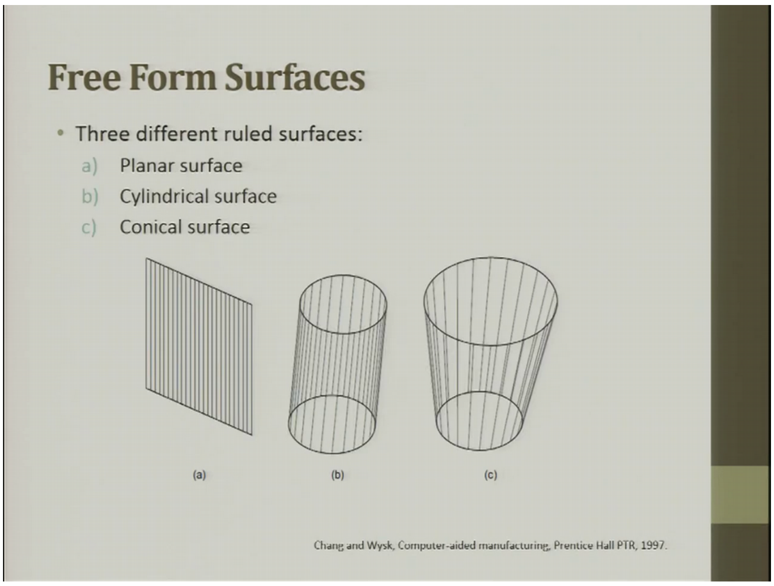

Next one these are three different ruled surface, planar surface, cylindrical surface and conical surface.

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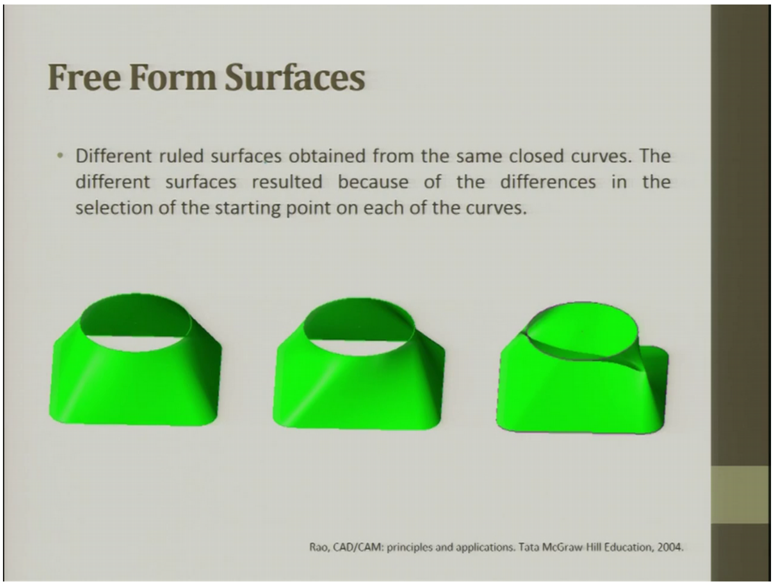

So, different rule surface obtain from the same closed curve. The different surface results because of the difference in the selection of the start point on each side of the curve. So, just by changing the start point you get different, different, different curves. So, ruled surface, you will have three, which is planar surface cylindrical surface and conical surface and here just by changing the starting point, you see you will get three different types of curves or surfaces.

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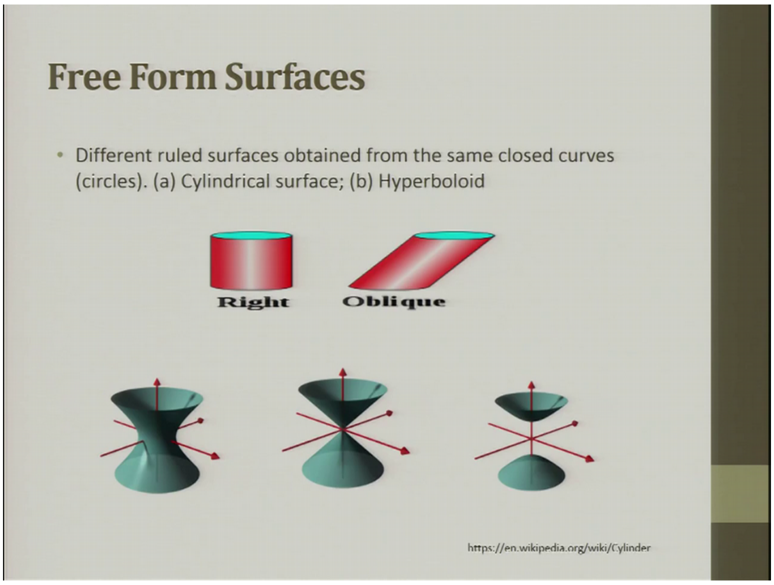

So, different ruled surface obtained from the same closed curve, this is right, and this is oblique. So, you can get, you can get any three different types of surfaces. It is different ruled surface obtained from the same closed curve, circle, cylindrical surface you can get, hyperboloid surface also you can get. So, we would stop here, we will continue in the next class.

Thank you very much.