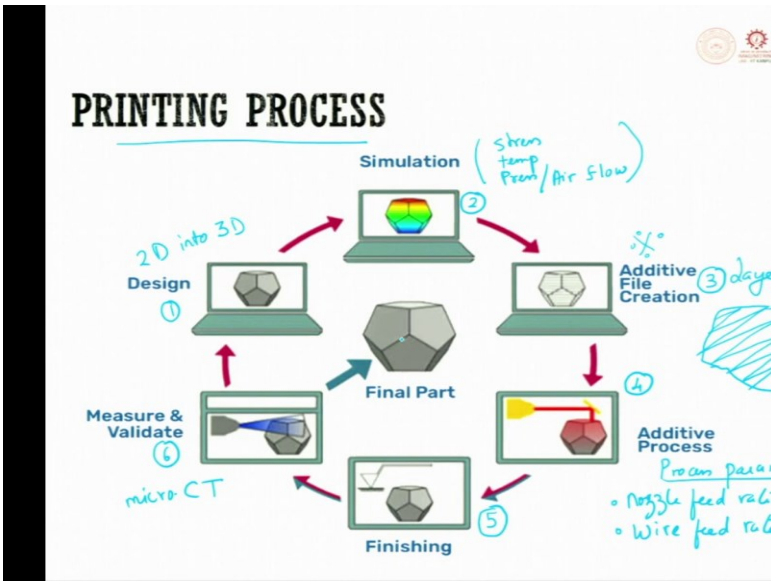
Metal Additive Manufacturing
Prof. Janakranjan Ramkumar
Prof. Amandeep Singh Oberoi
Department of Mechanical Engineering and Design
Indian Institute of Technology, Kanpur
Lecture 06
CAD for Additive Manufacturing

Welcome to the next lecture in our course, Additive Manufacturing, we will see the involvement of CAD in additive manufacturing. When we talked about additive manufacturing, we said in the introductory lectures we said that it is used to reduce the product life cycle time. So, when you say it is used to reduce the product lifecycle time apart from empathy study, definition, ideation, prototyping is also one study where we spent a lot of time in order to reduce it people started using rapid prototyping. For rapid prototyping, the initial point is to make CAD. So, in this lecture, we will see CAD for additive manufacturing.

(Refer Slide Time: 01:07)

We will be covering the following topics Printing process, then various CAD file formats, CAD CAM software's, Modelling and Data processing, issues which are related to STL file format, AFM file format, Slicing, Design consideration and Machine Setup, we will try to cover all these topics in brief. So, that end of the lecture you get to know what is the importance of CAD and how CAD is used in Additive Manufacturing.

(Refer Slide Time: 01:44)



If we look at the printing process, it goes like this. So, the first one is the Designing phase. So, designing phases where you try to visualize an idea, convert your 2D idea into a 3D idea and whatever you had a 3D idea try to draw that idea on the screen and where in which you try to develop a 3D model which is water tight, underline the word water tight because whatever is the CAD information you give at the design stage that will be further processed and you will try to make the additive manufacturing process.

So, design is a place where in which you convert 2D into a 3D airtight model, whatever you have done in 2D into 3D model, now, that CAD you use it for simulation, you can do stress simulation, you can do temperature simulation, you can do pressure or whatever or air flow simulations.

So, you try to do any one of the simulation and try to do optimization at the simulation stage trying to reduce the amount of material then transfer this file into additive file creation. Why am I saying additive file creation means because you have created a CAD model. Now, that CAD model is optimized, the next step to that optimization is CAD file creation, from there you are going to send it to additive manufacturing process.

So, now converting a virtual data into a realistic data has to happen. So, now at this stage you try to convert whatever is your CAD model into machine understandable manner. So, here you convert the CAD file into layer by layer-by-layer file, this layer by layer-by-layer information will be taken by the machine and it will try to develop.

So, in the third stage you are trying to create layer information. One-layer information will have a boundary and inside a boundary you will have hatches. Now, you will have to fill up all these information. So, here you will try to have your layer and then inside a layer what all has to be done hatch patterns, all these things are created in addittive file creation.

That is the error which can happen always at this stage. Whatever is the error happening at this stage will get amplified in the further stages. So, this is a very critical operation, whatever you get here, it converts a CAD into machine understandable figures. So, now from here you will try to fed into the machine where in which you will try to give process parameters. Today, you have simulation software, which can convert layer information into process parameters say for example, the movement of the nozzle, feed rate, the wire feed rate, the powder feed rate.

So, all these things are processed parameters, which also are getting created automatically at this additive file creation stage. Then it is pushed into Additive Manufacturing process stage, this is the fourth stage. Whatever you create as an additive file here, that file is now printed into layer-by-layer information and then it is accumulated.

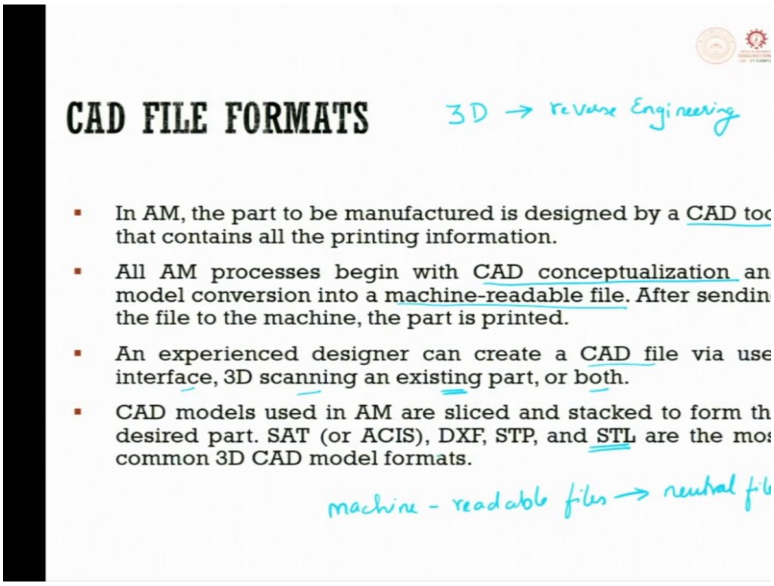
So, again here whatever was the curved surface is now converted into straight line surfaces. So, that means to say you will have facets now, these facets are going to be with some approximation to the curved surface. So, here you will have some amount of small error and you might have to do some finishing to make the product customer worthy.

So, here what we do, we take the product and then we do secondary processing or post processing on the Additive Manufactured part. So, here we try to smoothen the surface, try to remove the supporting structure, try to give a coating for it, try to add colour for it and try sometimes even do anneal the sample such that it is getting stress relieved. So, those things are called as finishing process.

After the finishing process, the sample will be sent for validation. So, whatever you have made, is it according to the dimensions whatever you wanted and does it have any defect. So, here we try to use optical noncontact techniques to measure the feature dimensions and sometimes we also use CT, micro–CT Computer Tomography to find out internally is there any voids.

So, this will be the sixth stage. So, this entire thing is called as printing process. In the printing process, the steps involved are design, simulation, additive file creation, additive process, then finishing. Finally, it is measure and validate, moment it is done then finally, what you get is a finished part. So, all these steps have to be done when you try to use any metal additive manufacturing machine or process if it is plastic, ceramic or metal the process is going to be the same.

(Refer Slide Time: 08:22)

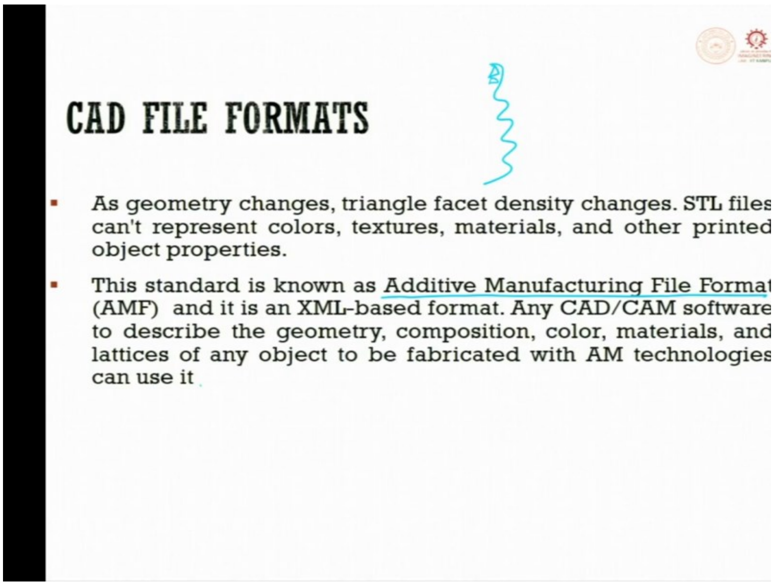


So, CAD file format in Additive Manufacturing the part to be manufactured is designed by a CAD tool, Computer Aided Design that contains all the printing information. All AM processes begin with a CAD conceptualization and model conversion into your machine-readable file. Conceptualization, model conversion concept to Model and Model to machine readable files, after sending the file to the machine, the part is printed. What is machine readable file? These machine-readable files are nothing but neutral files.

These neutral files are used to convert CAD model into machine readable file after sending the file to the machine. The part is printed and experienced designer can create CAD file via user interface 3D scanning and existing part or both. So, what is this 3D scanning? 3D scanning here is a part of reverse engineering. In reverse engineering, we have an existing part. We try to create information with the existing part by a scanning technique. If you look at it, rather than validate it, use it only for measurements using light or using x ray for measuring the dimension of the part is nothing but reverse engineering. So, that is what is taught here.

So, 3D scanning and existing part or you can have a combination of these two, the CAD files used in additive manufacturing are sliced and start to form the desired part SAT, DXF, STP and STL are most commonly used for 3D CAD model formats. So, from the CAD we try to convert it into tessellations, tessellations are nothing but *.STL. So, *.STL files are called as neutral files. CAD model used in AM are sliced and stacked to form a desired part. So, these are all the different file formats which are available.

(Refer Slide Time: 10:52)



STL which is very commonly used is nothing, but stands for Stereolithography. It is the most common AM format, this file format specifies ASCII or binary representation of 2D and 3D objects surface geometry. The only thing which is missing in STL is it will not have any colour or texture details.

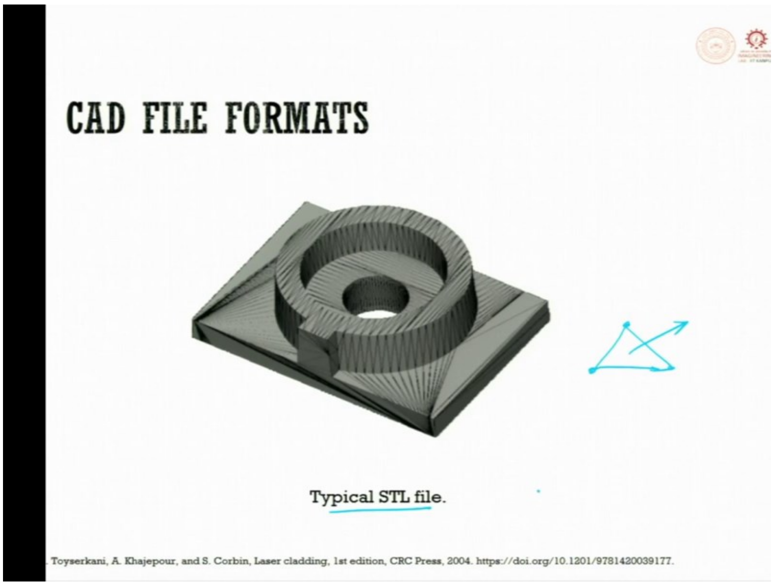
So, today we look forward for other higher neutral files, which will try to convert CAD into machine readable format. So, this will always be *.STL. But this is very important point this will not have colour information and texture information. The STL file containing triangle face is identified by a unit normal and using a three-dimensional Cartesian coordinate system.

So, what we are trying to say is we try to convert the surface curved surface into triangles and a normal. So, each vertex will have coordinate points. So, this will have x,y,z data; x,y,z and this will also be x,y,z data. So, it is x1, x2, x3; y1, y2, y3 and z1, z2, z3 and then they have a normal. This is what the STL file contains a facet or triangle facet which is the simplest facet and easy to represent and compute.

So, triangle facet is identified by a unit normal with three vertices using three-dimensional Cartesian coordinate system. Two nonstandard variations of the binary STL file can be used to include colour information. As geometry changes, the triangle facet density changes, that means to say if you have a curved object then the number of triangles whatever it is the density will be very large.

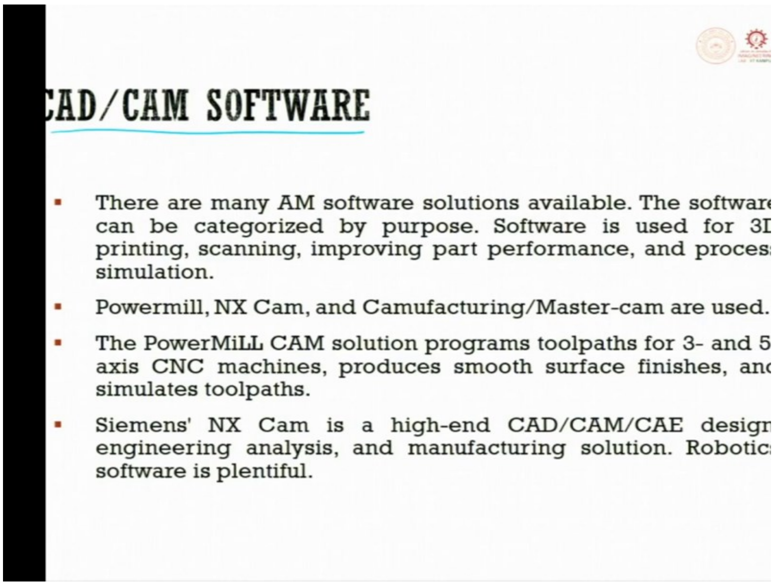
As the geometry changes, triangle facet density changes. STL file cannot represent colour, texture material and other Print Object Properties. Today we are using this standard which is known as Advanced Manufacturing file format. AFM is an XML based format this tries to give all the other details like texture, colour, material, etc. Any CAD CAM software to describe the geometry, composition, colour, material and the lattices of any object to be fabricated with AM technologies can be used to create Additive Manufacturing file formats.

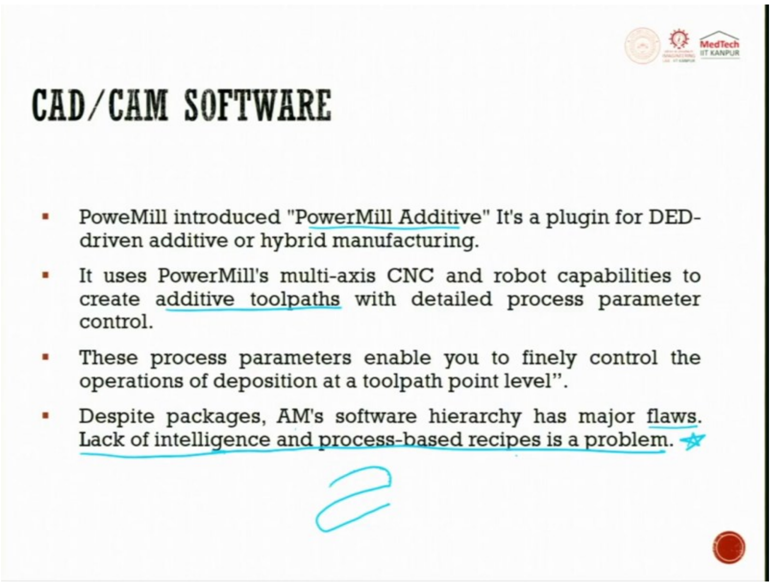
(Refer Slide Time: 13:53)



This is a typical CAD drawing. In this, CAD drawing have been converted into STL files, you can see here that these the object is converted into several soft small facets and these facets are linked. So, each facet finally if you look in detail, it will be like this we will have three vertices and a normal. So, this is a STL file when you store the file a *.STL, *.AMF then you will get all this information for STL files.

(Refer Slide Time: 14:27)





So, what are all the softwares which are there? The CAD CAM software- there are many AM software solutions available today. The software can be categorized by purpose. Software is used for 3D printing, scanning, improving part performance and process simulation. Power mill, NX Cam and Cam manufacture, Master Cam are generally used as CAD CAM softwares for Additive Manufacturing.

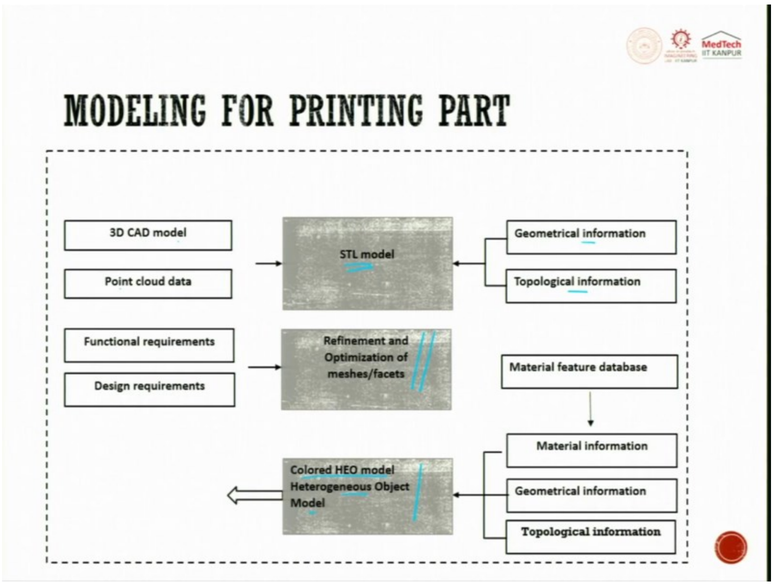
Power Mill cam Solution Program programs the tool path for three and five axis CNC machines to produce a smooth surface finish and simulates path toolpath. The same thing you can use here. Siemens NX cam is a high-end CAD CAM CAE and engineering analysis software, which also has manufacturing solutions in it.

Robotic software is also in plenty. Today what has happened all the softwares which were initially used for machining are getting converted for additive manufacturing or they are trying to have add on modules for additive manufacturing. Power mill introduced power mill additive. It is a plugin for DED driven additive or hybrid manufacturing. It uses power mill multi axis CNC and the robot capabilities to create additive toolpath with detailed process parameter control. So, when the toolpath means when the nozzle goes around in hitting the laser on the powder bed, so, you will see some patterns are getting created.

So, those things are called as additive toolpath these process parameters enable us or you to finally control the operation of the deposition at the tool path point level. So, it is like this the laser will move and it will try to create all along the path despite packages, additive manufacturing software hierarchies have major flaws, lack of intelligence and the process-based recipe is even now, a problem. This is something which you should know please don't think whatever software or simulation you do, it tries to give you readymade information, still it lacks intelligence.

So, today that is what people are started using machine learning and deep learning in additive manufacturing. People started using process based recipes, rather than titanium powder if I use aluminium powder, what happens to the process parameters that is what is called as process based recipes.

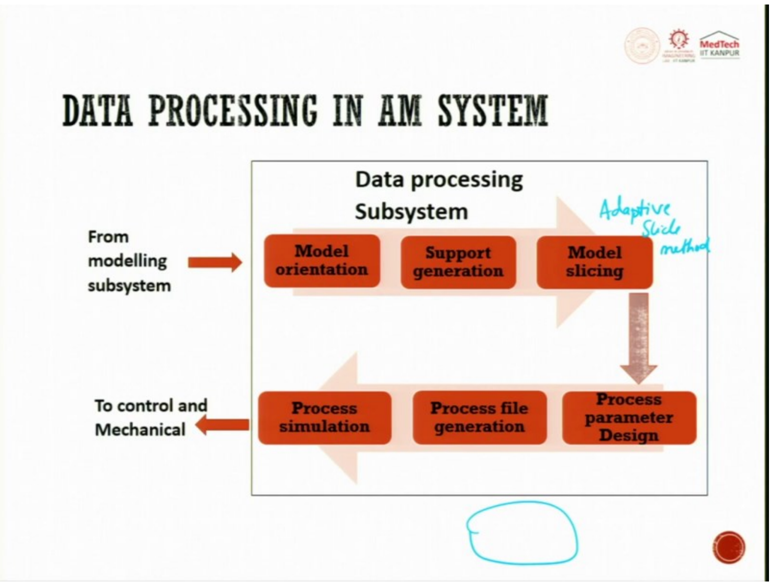
(Refer Slide Time: 17:00)



Modelling for printed parts you will have 3D CAD model you can also extract it from point cloud data, this point cloud data comes from the reverse engineering where in which you use a laser and you try to scan the details. So that is point cloud data. So, that is given to an STL model. So, you try to add to this model geometric information and topological information. These geometrical information means dimensional, and topological means on the surface roughness and other things. So, you try to add these things to STL file, then what you do is we put functional requirements and design requirements then what we do we try to do refinement and optimization of mesh and facet, because the STL file will have a lot of facets.

So, now what we do is we try to look at material feature database and that gives material information. Again, geometric information, and topological information is fed into coloured HEO model heterogeneous model for printing. So, this is how the modelling of a printed part happens, you will have STL file you will have refinement and optimization of mesh and then you will also have coloured HEO model heterogeneous object model.

(Refer Slide Time: 18:15)



When you look into the data processing in AM systems, from the model subsystem you will try to have model orientation, then support generation, then you will have model slicing, then process parameter, design process, file generation, process simulation and then you try to get the final part to control and mechanical.

So, here what does it mean is when you want to develop any part for example, I wanted to develop my hand. So, should I keep my hand like this and print, or should I keep my hand like this and print the object, should it be oriented vertical or horizontal? When I do it vertical, you will have more layers. When I do horizontal, you will have less layers. When I do vertical, you have to support my hand because it is a free hanging structure, when you do horizontal you do not have to support so much.

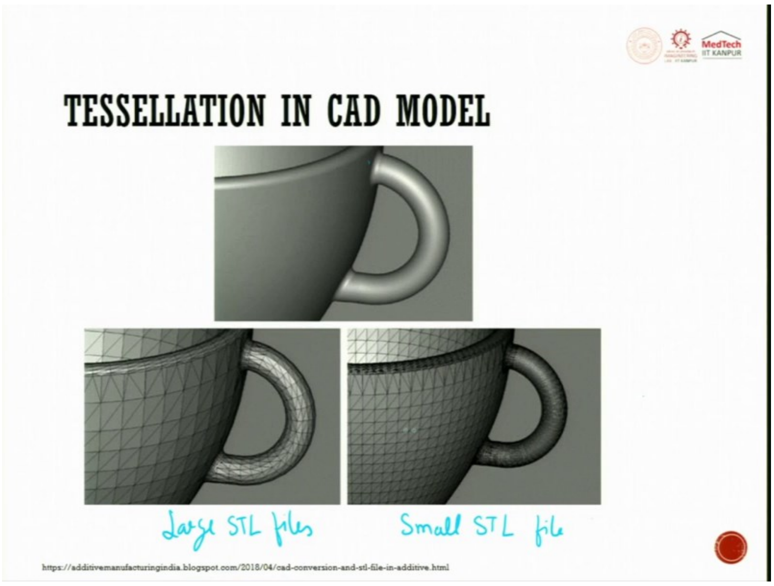
So, now what does it happen is you try to decide which orientation I should keep the path such that I try to get the best roughness and best strength. So, model orientation is first thing, next is for this developed model, if I wanted to have a supporting structure, so that the free hanging projections does not sag down. So, I would like to support them.

So, support generation is the next step. The third step will be for the support, develop the model how am I going to create layers? So, the modelling of layers. How am I going to slice them? Is it going to be as thin as possible or is it going to be as heavy, that means to say as thick as possible? When I wanted to have lots of micro feature details, I go for very small slicing thickness. When I wanted to have a slightly larger, I go for a thicker slice.

So, something called as adaptive slicing methods are today talked about. Adopt adaptive slicing methods. So, here what does it mean is wherever there is a complex feature wherever there is a change in geometry more than x value in the CAD model, it will try to automatically slice to have a thicker or thinner thing. So, that is model slicing.

From the model slicing, it is only one layer you have sliced now, in that layer how are we going to fill up details, so, that is called as process parameter design, then process file generation after you do the process generation, then you have a process file which tries to do switch ON switch off and then you try to do a simulation. So, these are the steps which are involved in data processing in Additive Manufacturing.

(Refer Slide Time: 20:42)

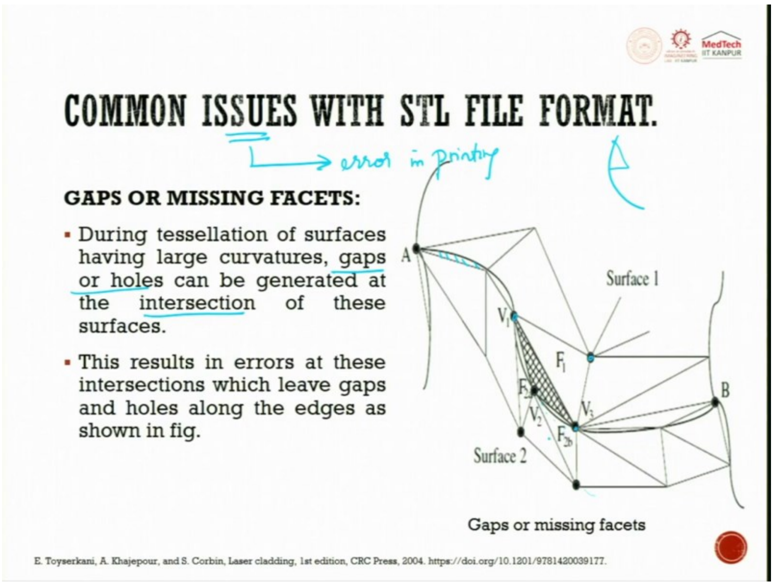


So, the steps which are involved as I was telling you, the second step is tessellation. So, tessellation in CAD, so, this is a cup which is drawn in CAD, but now, after that you wanted to convert this information into adaptive machine control file or adaptive machining process file or something like that. So, there what you do is you try to convert this CAD model into tessellations.

When I talk about tessellations, I try to convert this information into triangles, when the triangle size is large, the surface roughness is very high when the triangles surface is small then you will try to have a smoother surface as compared to this the interesting part when you compare these two this will try to have more information and as compared to that of this.

So, here you will have more triangles, good surface finish, you will have good product control with smaller tessellation triangles as compared to that of larger. So, this shows large tessellation files. These are small, I am talking about triangles size STL file with respect to all triangles, but if you try to see the volume this will be the large as compared to that of this.

(Refer Slide Time: 22:04)

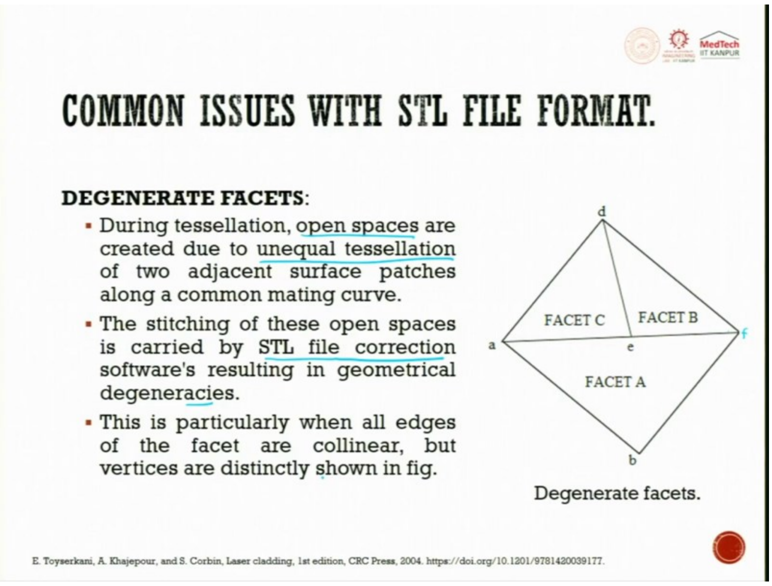


Some issues with STL file are though it might look very easy you have a curved surface and you will try to create facet it is very easy, but when you start looking at it but there are small issues which come that will lead to error in printing. So, let us understand these small issues. So that, when you start using this CAD, when you start developing STL; today of course STL files are auto generated, you can also understand where exactly the mistake is and you can try to correct it.

So, there are generally gaps or missing facets, if you see this is a curved surface. This curved surface is divided into several small triangles, you see here this is a curved surface, the black dark one is a curved surface and you see a triangle here. So, there is a missing information in this place or there is a gap in this place or there are triangles sharp edges and this information is missing. So, gaps are missing facets and is one of the biggest problems when we tried to create STL files from the given CAD drawing. During tessellation of surfaces having large curves, gaps or holes can be generated at the intersection of these surfaces, you can have gaps or holes, so intersection.

So, this will try to give you a gap there and the system will not know what to do here either it will fill with material or it will not even fill with material and go away. This results in error at these intersections, which leaves gaps and holes along the edges as shown in the figure. So, you see here this is a gap which gets generated when you try to create this. So, you see here vertex, one vertex to vertex three, phase one, phase two, phase two A phase two B.

(Refer Slide Time: 24:06)

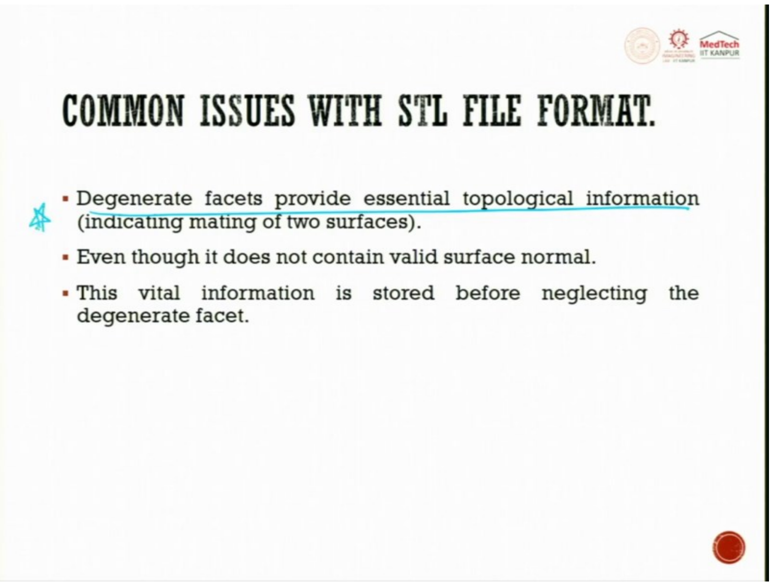


Next is you can have degenerated facets, Degenerated facets means during tessellation, your open space are created due to unequal tessellations. Open spaces have to adjust and surface patches along your common mating surface, you will have mating surface, you will have unequal tessellation stitching of these open space carried out by STL file correction software resulting in geometrical degeneracy.

So, degeneracy means having more information. So, you will have a, e, d has one edge d, e and f has one edge. So, these edges are common for this b, a, a, b and f. So, you will see that there are common edges. This is a redundancy they get could have taken it as a single triangle. Now it is taking it as two triangles.

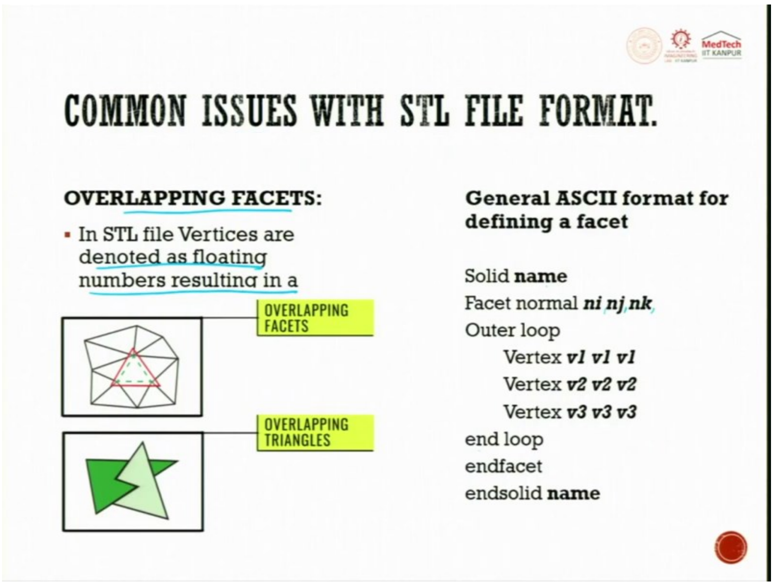
So, the stitching of these open space is carried out by STL file correction, there is something called as a STL file correction or there is something called as a patch working. So, wherever there is an error while running the test after tessellation they will run it before layer information generation, you will try to figure out there will be a lot of errors, these errors are either manually cleared or there is a software which converts all these things and it tries to remove the error. When it tries to remove them, it creates a lot of degeneracies. This is particular when all edges of the facet are collinear. But what he says are distinctly shown in the figure they are all collinear and distinctly shown in the figure, this is a degenerated facet.

(Refer Slide Time: 25:44)



Degenerated facets provide essential topological information indicating meeting of two surfaces even though it does not contain valid surface normal. This vital information is stored before neglecting the degenerated facet. So, degenerated facet provides essential topological information. So, this is a valid point.

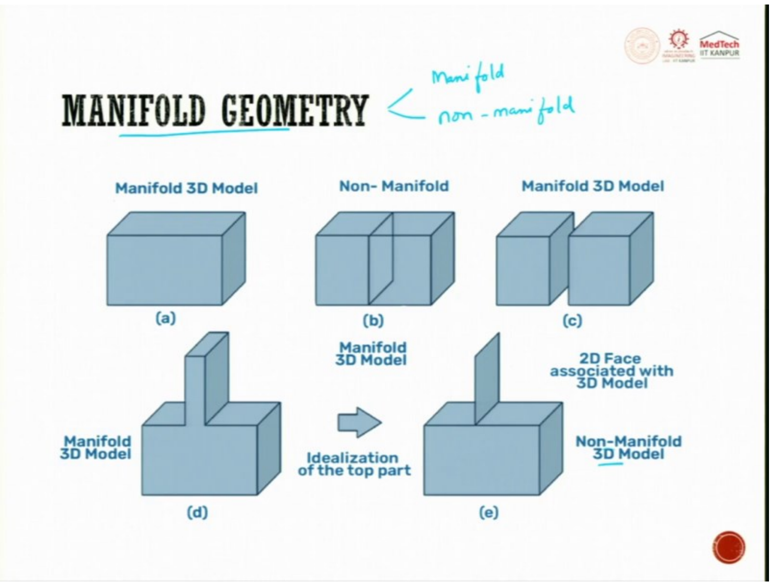
(Refer Slide Time: 26:09)



There can be other issue of overlapping facets. For example, the STL file vertices are denoted as floating numbers resulting in overlap facets. So, if you see here, generally ASCII format defines a facet solid name facet n_i, n_j , n_k over the outer loop is v1x,y,z, v2x,y,z then v3x,y,z. So, end of loop this is how the ASCII format file looks like for a facet.

So, when we have a rounding of a floating number, resulting in an overlap of faces or you can try to have overlapping of triangles. So, there is one facet like this, the other triangular facet is like this, they try to intersect each other which cannot happen in reality. So, these are overlapping facets, the earlier was degeneration, there we saw a missing gap and now we see overlapping facets. All these things are problems or issues which are created while creating STL format.

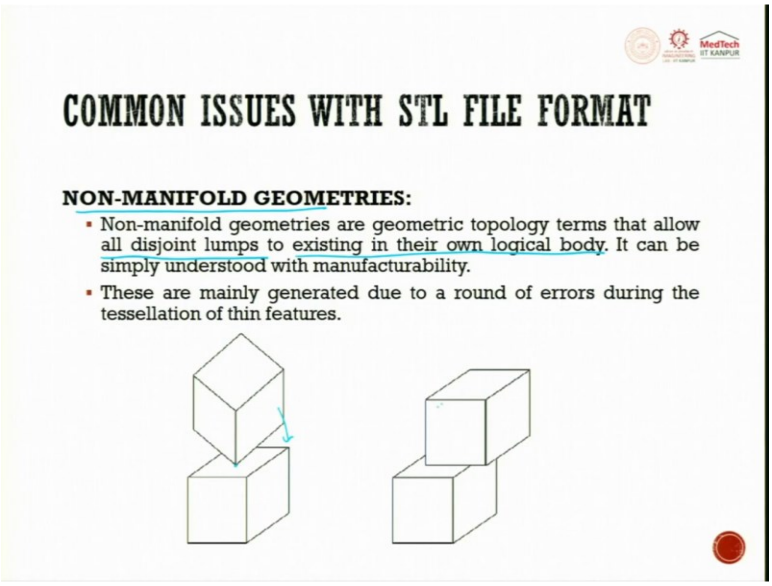
(Refer Slide Time: 27:21)



There is another interesting thing in CAD, which is manifold geometry. So, in manifold geometry you have manifold 3D model. So, this is a manifold 3D model, this is a non-manifold 3D model. So, you can have manifolds so you split it down and then you have manifold 3D model like this. So, then in ideation on the top place, you will have something like this. So, 2D phase is associated with a non-manifold 3D model.

So, you cannot have a 3D model where in which you can have one plane as 2D on this. So, this will try to say a non-manifold model. So, in geometry, there is something called as manifold and non-manifold models. You can see non-manifold here, because in this course, we cannot get more deeper into CAD. So, manifold and non-manifold models are there and when you start working with 3D software CAD, so you will see this manifold and non-manifold.

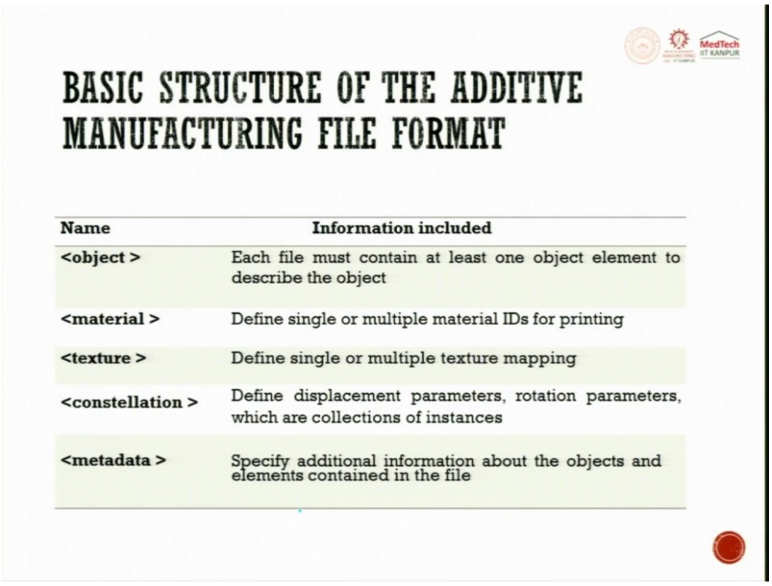
(Refer Slide Time: 28:27)



So, what are non-manifold geometries? Non-manifold geometries are geometrical topological terms that allow all disjoining lumps to exist in their own logical body, it can be simply understood with manufacturability.

So, here all disjoining lumps to exist in their own logical body are mainly generated due to rounding of error during the tessellation of thin features. So, you can have your figure like this, when you try to develop a figure like this, it is very difficult for the point to rest. So instead of this if you can rotate it and print it like this, then this will try to give a valid object.

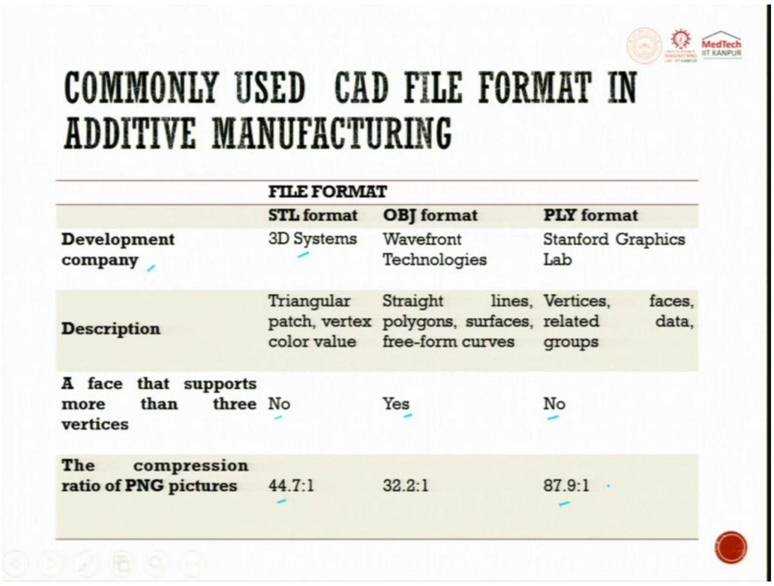
(Refer Slide Time: 29:21)



Name	Information included
<object >	Each file must contain at least one object element to describe the object
<material >	Define single or multiple material IDs for printing
<texture >	Define single or multiple texture mapping
<constellation >	Define displacement parameters, rotation parameters, which are collections of instances
<metadata >	Specify additional information about the objects and elements contained in the file

So, the basic structure of Additive Manufacturing file format is going to be object, material, texture, constellation and meta data. So, object in each file must contain at least one object element to describe the object. Material will be defined by simple or multiple material IDs for printing. Texture defines single or multiple texture mapping. Constellation define displacement parameter, rotation parameter which are collection of instances. Meta data is simplified additional information about the objects and elements containing in the file.

(Refer Slide Time: 30:05)



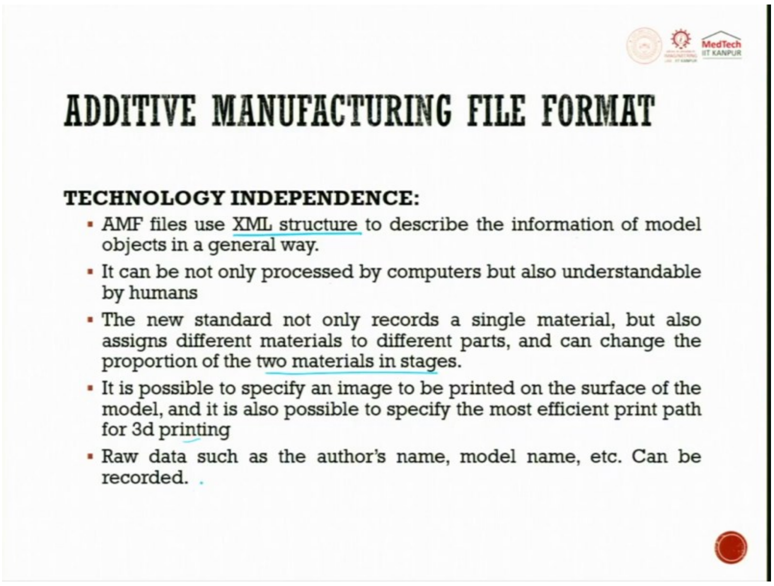
COMMONLY USED CAD FILE FORMAT IN ADDITIVE MANUFACTURING

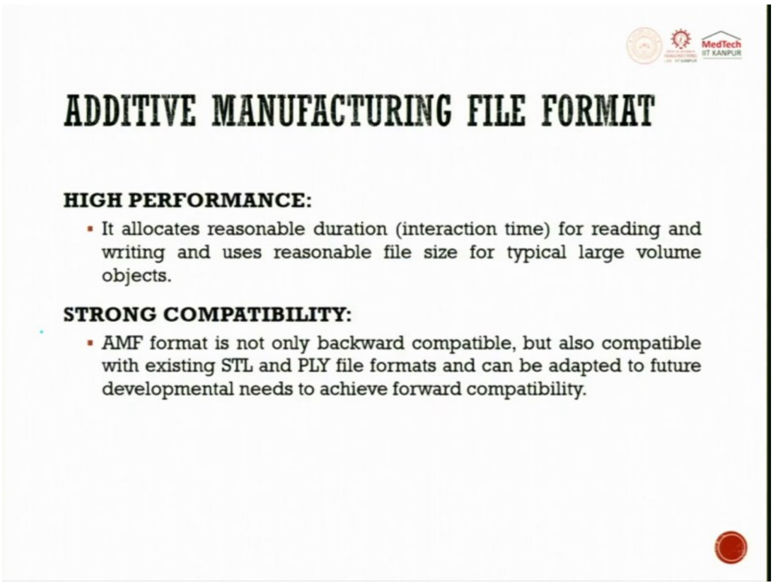
	FILE FORMAT		
	STL format	OBJ format	PLY format
Development company	3D Systems	Wavefront Technologies	Stanford Graphics Lab
Description	Triangular patch, vertex color value	Straight lines, polygons, surfaces, free-form curves	Vertices, faces, related data, groups
A face that supports more than three vertices	No	Yes	No
The compression ratio of PNG pictures	44.7:1	32.2:1	87.9:1

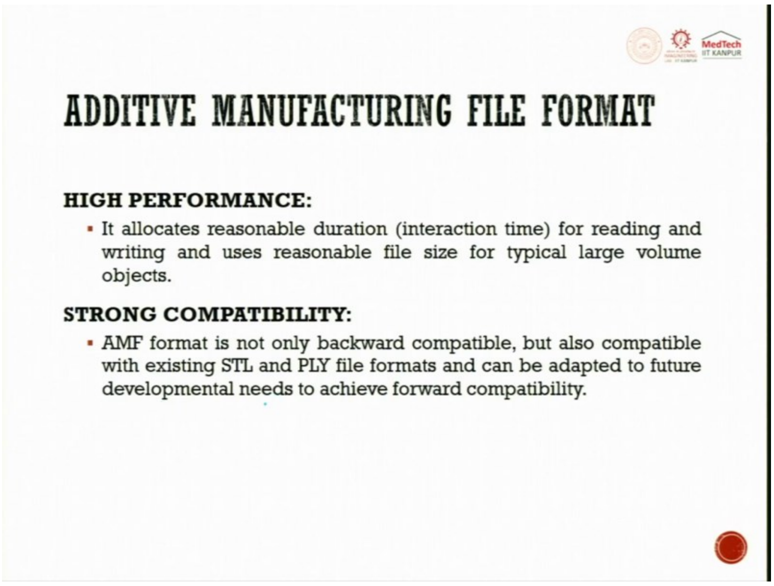
Commonly used CAD file formats in Additive Manufacturing developed by the company 3D systems is STL format. OBJ format is by wavefront technologies and the PLY format is by Stanford graphic lab. We use triangle patches, and vertices colour values in STL format, when you go to OBJ we use straight polygon and freeform surfaces, Straight Line, Polygon surface and freeform curves. When we talk about PLY format, vertices, faces related data and groups are used.

A face that supports more than more than three vertices which will never happen in STL which will happen in OPG and which will not happen in PLY the maximum compression ratio of a PNG file is 44.7:1 in STL, in OBJ it is going to be 33.2:1 and 87.9:1 in PLY. Why is this very important because this will try to talk about how much size can you reduce it such that you can occupy less space in your CAD software's.

(Refer Slide Time: 31:08)







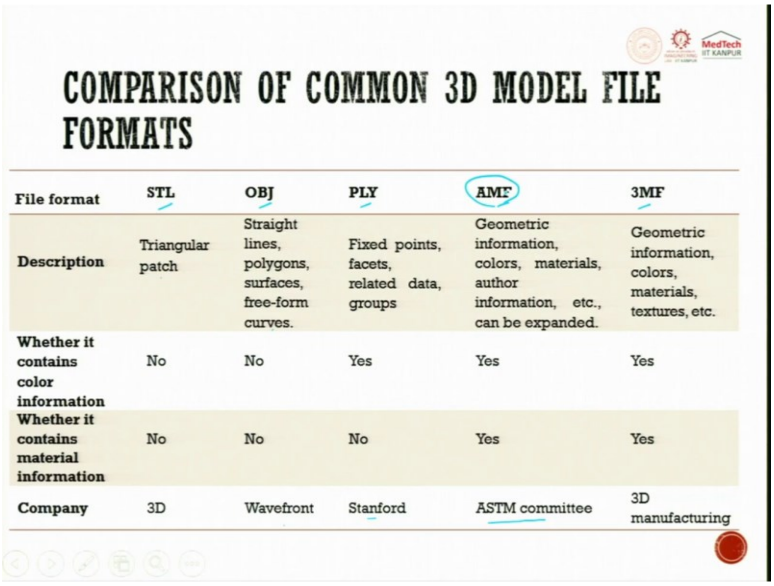
In terms of technology independency, AMF, the Additive Manufacturing files use XML structure to describe the information of model objects in a general way, it can be not only processed by computer but also understood by human XML format structure. The new standard not only records a single material, but also assigns different materials to different parts and can change the proportion to the two materials in stage for example, if you want to mix two materials, you can try to decide in that technology independency you can do.

It is possible to specify an image to be printed on a surface of the model and it is also possible to specify most effective print path for 3D printing. Raw data such as author's name, model name, etc. can also be recorded in the technology independency. Additive manufacturing file format has to be simple or the simplicity matters very much for ease of understanding an application, the file format should be able to read and edit in a simple text viewer and the same information should not be stored in multiple places in the file.

So, the additive manufacturing file format should be as simple as possible, it should simply text view that your information should not be stored in multiple places in a single file, it should be extendable that means to say you have made an object just by adding some module you will be able to make a larger one. So, as the complexity of the size of the component increases, it can be expanded with the increasing resolution and accuracy of the manufacturing including equipment including large array capable of handling the same object complex repetitive internal features, smooth surface with fine print resolution and many components for printing that are placed in the ideal fill material package.

So, extendibility is also very important. 1 is simplicity, 2 is extendable. Today in manufacturing, we always look for this extendibility. Third is high performance, it allocates reasonable duration for reading and writing and use reasonable file size for typically large volume objects. So, try to have a large object in a small space that is what is high performance. Strong compatibility, AMF format is not only backward compatible, but also compatible with the existing STL and PLY file format and can be adapted to further development needs to achieve forward complexity.

(Refer Slide Time: 33:52)



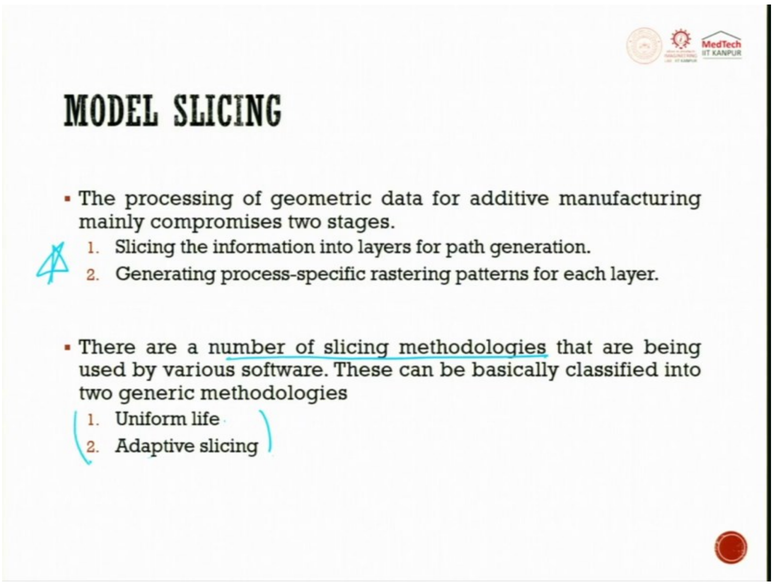
COMPARISON OF COMMON 3D MODEL FILE FORMATS

File format	STL	OBJ	PLY	AMF	3MF
Description	Triangular patch	Straight lines, polygons, surfaces, free-form curves.	Fixed points, facets, related data, groups	Geometric information, colors, materials, author information, etc., can be expanded.	Geometric information, colors, materials, textures, etc.
Whether it contains color information	No	No	Yes	Yes	Yes
Whether it contains material information	No	No	No	Yes	Yes
Company	3D	Wavefront	Stanford	ASTM committee	3D manufacturing

When we look into the comparison of 3D model file formats, these are different file formats STL OBJ, PLY, AMF and 3MF. So, it is STL, object, PLY an advanced manufacturing format or Additive Manufacturing format, 3D manufacturing format. So, here description of STL is triangular patches, OBJ is straight line, polygon surface, freeform surface; PLY fixed point, facet, related data groups, AMF is geometric information, colour, material, author information etc can be expanded, 3MF is geometric information, colour, material, texture etc.

Whether it contains colour or not? STL No, OBJ No, PLY Yes, AMF Yes and 3MF Yes. So, this colour information is very important when you are trying to build a three-dimensional topological dominated information for example, hilly terrain. Whether it contains material information STL NO, object NO, PLY No, AMF Yes, 3MF Yes, they have object they have material information also. Companies which follow STL is 3D systems, OBJ is wavefront, PLY is Stanford, AMF is ASTM committee and 3D manufacturing is 3MF.

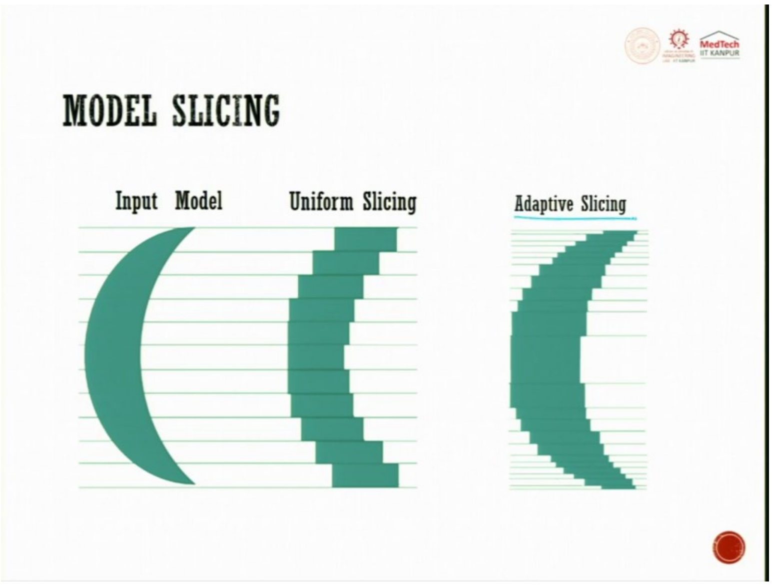
(Refer Slide Time: 35:18)



Now, let us move from STL into slicing model. The process of slicing of geometric data for additive manufacturing mainly composes of two stages. One is slicing the information into layer for path generation, next one is creating process specific rastering pattern for each layer, these two are very important, when we are trying to do slicing, you will have these two till now whatever we studied about STL everything is done.

So, STL means it is facet information converting from CAD to machine readable facet format. So, it can be triangle it can be straight line it can be fixed point it can be geometric information, it can be geometric information colour, there are a number of slicing methodologies that are being used for various software, this can be basically classified into two generic methodology, uniform life adaptive slicing. So, number of slicing methodology we are discussing are the two important ones- uniform and adaptive.

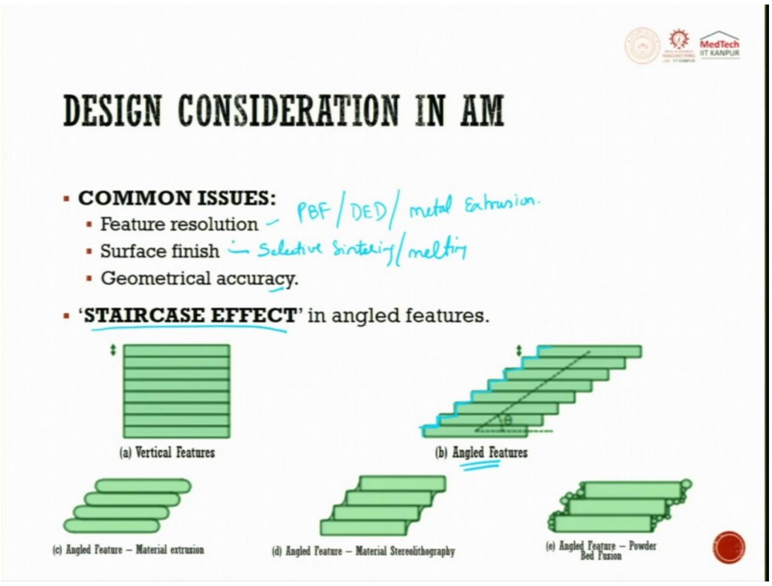
(Refer Slide Time: 36:20)



So, this is the input one this is uniform slicing. So, here whatever might be the curve change, I tried to slice it in a very uniform manner. So, you get a uniform slice, but when we try to do adaptive slicing, wherever there is a change in geometry more than a particular sub slope change, we try to slice it, we try to fix that slope angle and if it crosses that slope angle immediately a slice is drawn.

So, this is called as adaptive slicing. So, input model, uniform slicing and adaptive slicing. The two types of slicing which we saw are uniform and adaptive slicing. So, this is also very important, when we try to slice the information in one layer fastest outer geometry next is from inside the outer geometry What are you going to fill in.

(Refer Slide Time: 37:09)



So, design concentrations in AM. The common issues always which comes when we use additive manufacturing is feature resolution, how accurate you try to develop a feature matters, the feature resolution is very important. When we decide which process to use, when we use powder bed fusion or when we use DED or when we use extrusion metal extrusion, we always look for the one which gives a better size, better shape accuracy, and what is the smallest feature size it can do that is feature resolution.

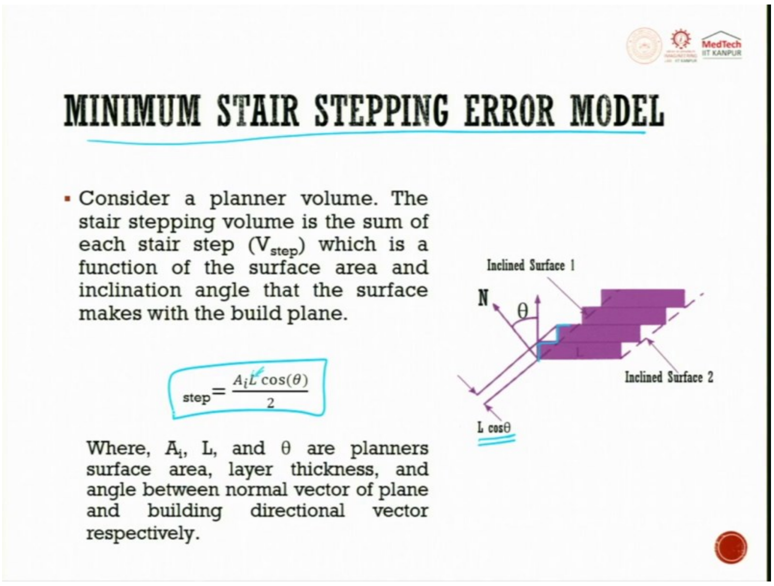
Next one is roughness. Surface roughness is because here when we talk about metals, it is more always like selective sintering or it will be selective melting. When we try to sinter or when we try to melt, it is basically the metal powders are going to join with each other. And once it joins, it makes a surface very rough.

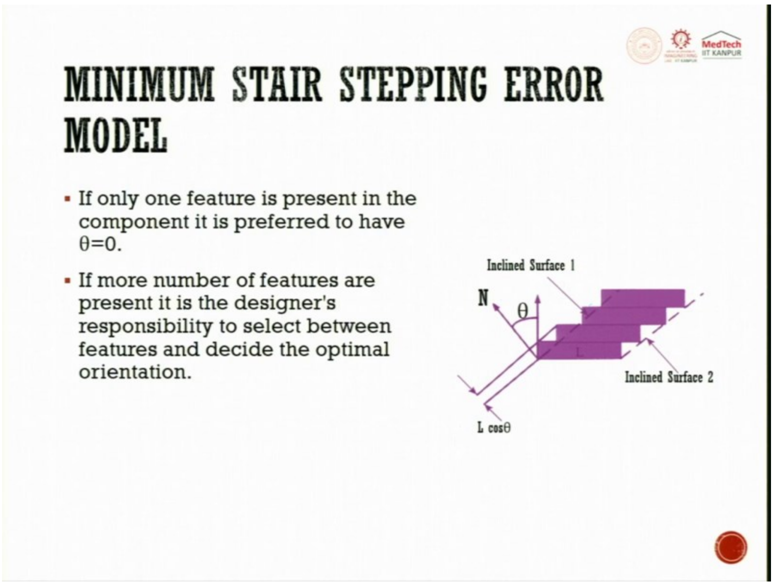
So, how to get a better surface because after doing additive manufacturing process, as far as metal is concerned, finishing is a very big challenge. The last one is geometrical accuracy. We are trying to talk about not dimensional accuracy, but geometrical accuracy, where in which we involve 3d geometry. And we also look for cylindricity, conicity, runout all these things we try to look at while developing this.

So, geometrical accuracy is the other important thing. The most important problem, which we face when we do layer by layer additive manufacturing is the stair case effect. Stair case of this is a vertical feature. This is an angular feature. This is an angular feature. So, now if you see this is called as a stair case effect, what you wanted was your curved surface, but what you got generated is a stair case effect.

So, this is for an angle feature. If you want a vertical feature, it is very easy. When you have it slightly at an angle, then you get this stair case effect. So, you have angular feature material extrusion, you have angular feature materials stereolithography, you have angular feature, bed fusion powder. So anywhere and everywhere, you will always have a stair case effect. This depends or dictates the feature resolution, which was taught here.

(Refer Slide Time: 39:41)





How do we calculate the Minimum Stair Stepping Error model? We are trying to calculate the staircase effect. So, staircase consider a planar volume, the stair stepping volume is the sum of each stair. Each stair step, which is a function of surface area and inclination that surface makes with the build a plane. So, this is the staircase.

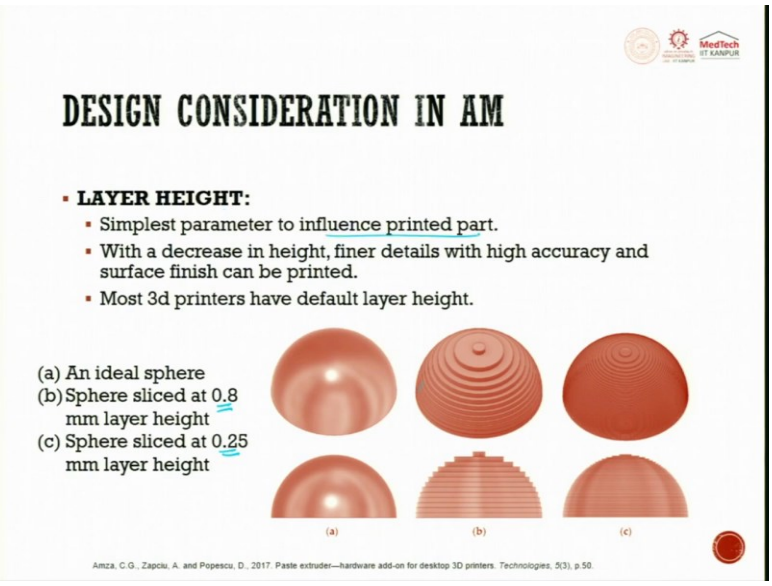
And here, if you want to find out the thickness, it is L cosθ, and N is point normal and θ is along the line. This angle becomes θ. These are inclined surfaces.

$$\text{step} = \frac{A_i L \cos(\theta)}{2}$$

Where, A_i, L, and θ are planar surface area, layer thickness, and angle between normal vector of plane and building directional vector respectively So, this is a very important formula. You have to know, you can calculate. So, by this, if you want to reduce the stair case effect, you reduce the step size. So, you see here, L is nothing but the layer thickness. It has a direct influence.

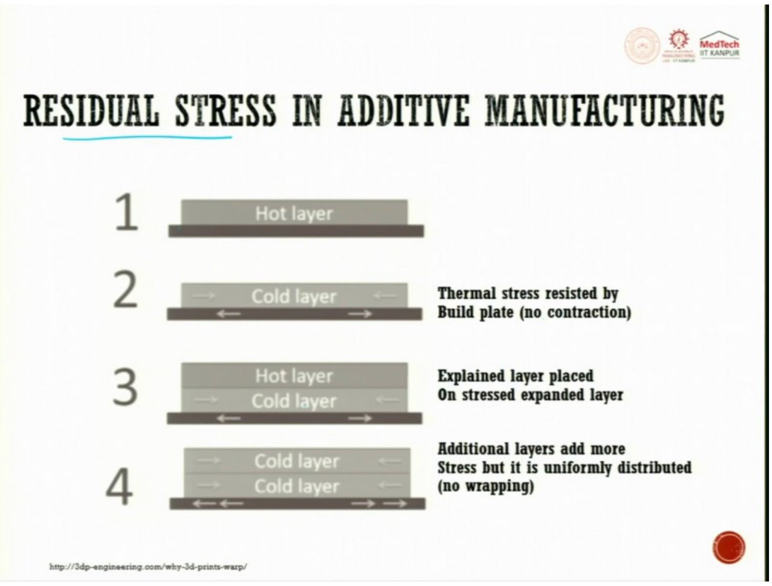
If only one feature is present in the component, it is perfect to have θ=0. If more number of features are present in the designers, it is that designers responsibility to select between the features and decide the optimum orientation. This Stair case effect is inbound. If you want to reduce the stair case effect, what we do is we try to orient the object in such a way such that the theta is reduced. So, we try to know. So, this is where you see, we started initially with orientation of the object.

(Refer Slide Time: 41:45)



So, layer, height, are simplest parameter to influence printed parameters. First, we have to see, this is an ideal sphere if the sphere is sliced at 0.8-mm thickness, you will get this. If your sphere 0.25 mm, you will get this. So, when we decreased with a decrease in height, finer details with higher accuracy and surface finish can be obtained, 0.25mm. When the thickness is high, you will have a rough surface. Surface is lost. Accuracy is also lost. So, layer thickness plays a very important role as far the influence of printing.

(Refer Slide Time: 42:28)

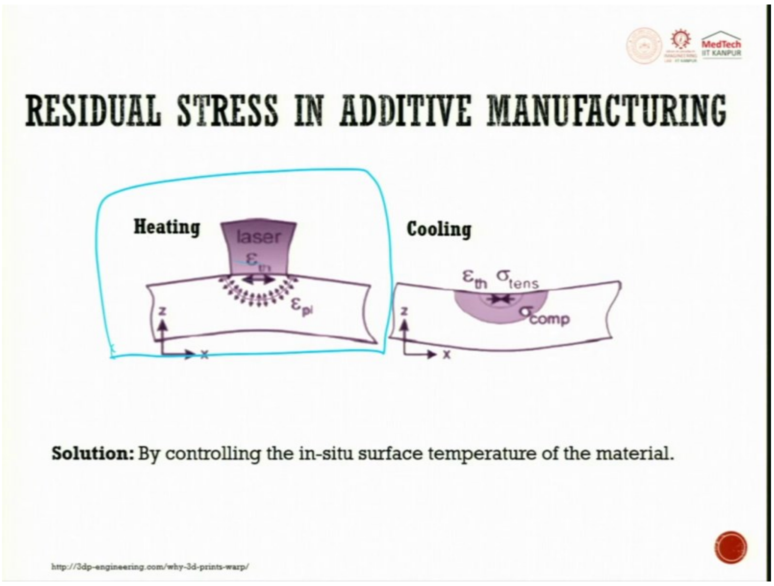


So, there is always residual stress, which is getting imbed in the part while printing through additive manufacturing, especially polymer, as well as metal is concerned. So, what we do is we always have a hot layer. So, there is a hot layer we keep, and then we also try to have cold layer.

So, we try to have a hot layer, cold layer. So here, the thermal stresses resist by the built plate. No contraction will happen. So, this is coldest contraction. This is expansion. So, you get a very good surface. So, we explain layer placed on stressed, expanded layer. So, you can see this and you can also have additional layer add more stresses, but it is uniformly distributed. No wrapping happens when you have this cold layer.

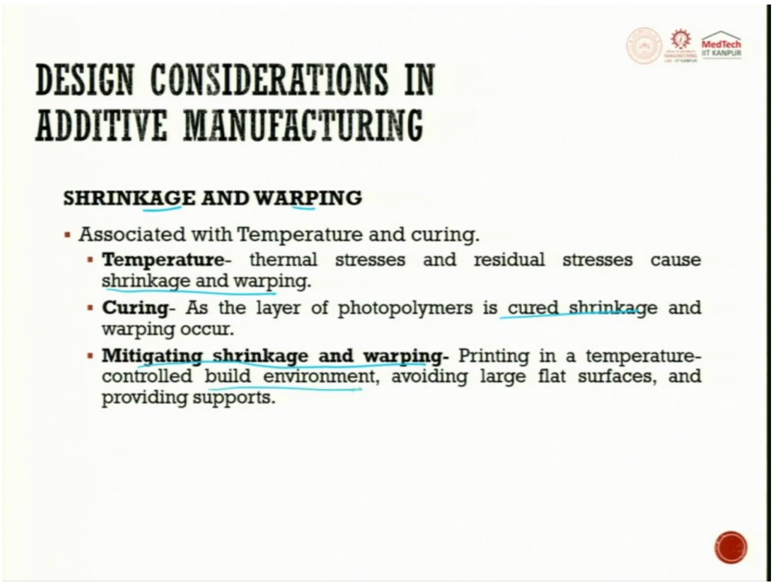
So, that means to say, after you do one layer, you will do sometimes so that the layer gets solidified. So, these are the places where residual stresses can be created. This is tensile. This is compressive between these two there will always be a huge stress variation. So, there will be a demarcation here. The residual stresses in additive manufacturing, especially in metal. Additive manufacturing is a very dominant quality issue creator.

(Refer Slide Time: 43:42)



When we try to heat a surface using a laser, you can see that lot of thermal stresses are created, and these are tensile stresses when we cool it, then this gets shrunk and it tries to introduce air compressive stress just by controlling the NC to surface temperature of the material that is laser adding and cooling and the super and the purify say, for example, you have an object which is done, and I put it inside a container. And in that container, I try to heat it. So, then the difference will not be very high. So, the tensile stresses created will not be very high.

(Refer Slide Time: 44:18)

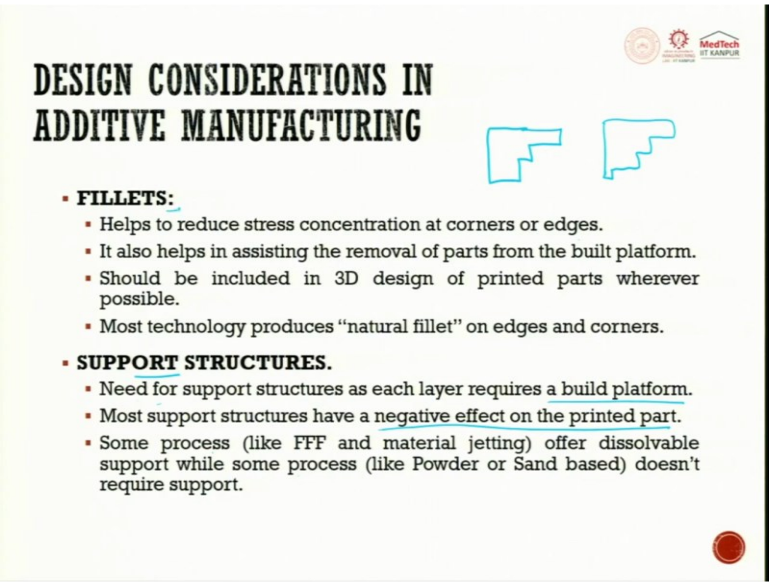


So, when we try to design additive manufacturing, some of the challenges are shrinkage and warping. They both are associated with temperature. So, we would like to look at temperature curing, mitigating shrinkage and warping temperature, thermal stresses, and residual stresses causes shrinkage and warping, which is a very common feature.

So, when we try to decide the layer thickness, you try to have enough layer such that the shrinkage and warpage does not happen in one go, curing as the layer of a photo polymer is cured. There is a shrinkage which happens, which is also true. When we start doing selective laser melting, there is a curing, and there is a shrinkage process, which is attached, mitigating shrinkage and warping printing in a temperature-controlled build environment, avoiding large surface flat surfaces and providing supports will try to reduce the shrinkage and warping very large surface.

Say, for example, my hand is a very large surface. If I do it, it is very small. So, I can try to reduce the warping, and the build environment, the complete object, where it is getting built can be sealed in a container and the temperature can be maintained, or the bed can be maintained with that temperature.

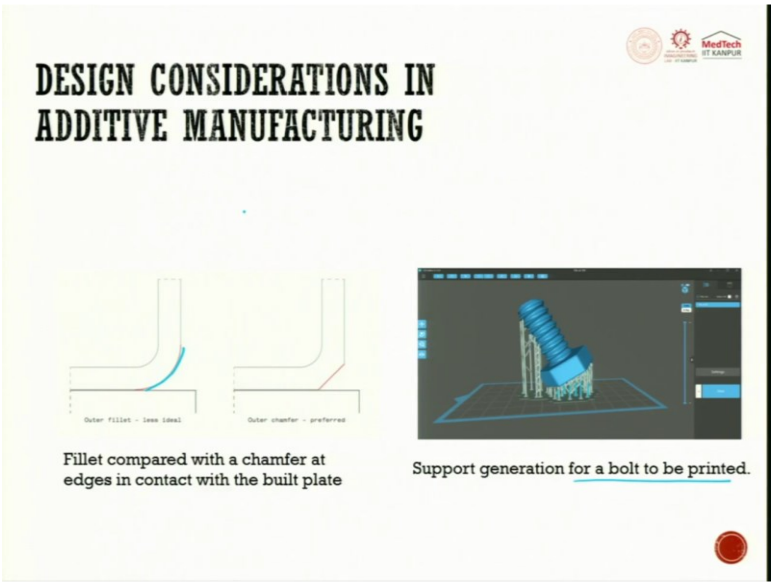
(Refer Slide Time: 45:40)



You can always try to give filleting. So, filleting means sharp edges you try to avoid. So, rather than sharp edges, just we try to give corner very smooth. And then we try to do something like this. So, filleting helps to reduce the stress concentration at corners or the edges. It also helps in assisting the removal of part from the built platform. So, trying to produce filleting and support should be introduced in 3d design for printing parts, whatever possible, most technological produces natural fillet on edges and corners support structure, which is always given to support the overhanging.

The support structure is needed, as each layer requires a build platform. Most support structure have a negative effect on printed part. Once it is supported, it does not. Some processes offer disallowable support while some process does not require support at all. So, you will always look for whether there is a need for support structure, whether there is a need for fillet, and then we start doing it.

(Refer Slide Time: 46:49)



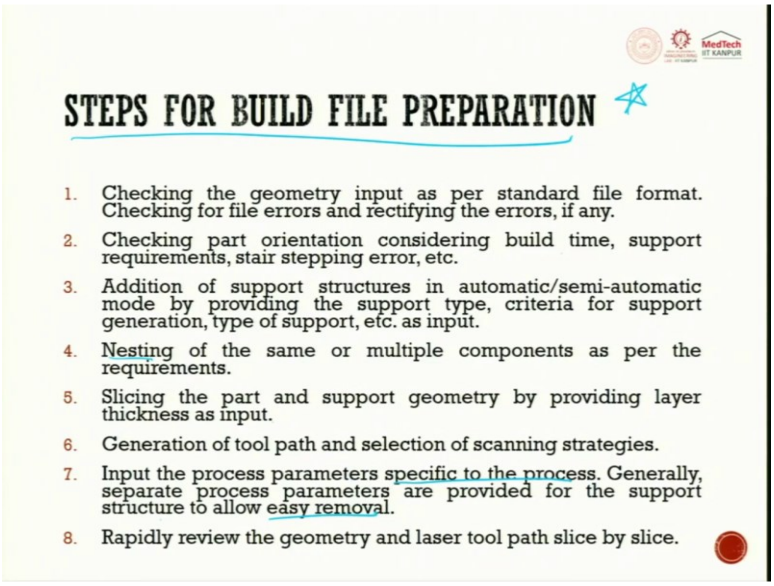
So, this is a typical fillet, which is given. So, fillet compared with a chamfer at the edge in contact with a built plate. This is a built plate, fillet. So here, the support generation for your bolt to be printed. So, these are all support because bolt, if you do it like this, then the bottom surface, you will never get it. So, you do it at an angle. So, these support structures will be dissolved, and then you get a bolt. So now, we have been looking into all these, the important things, which is design consideration, feature resolution surface finish, geometric accuracy.

These are very important for this, there will be a stair case effect. We saw how to calculate the step size, then the layer height. Then we talked about thermal induced stresses, then we talked about, curing and the mitigating shrinkage, filleting and supporting. These are some of the design considerations, which you have to do when we try to do additive manufactured part.

(Refer Slide Time: 47:50)

Some of the commercially available softwares for simulation are RPS, Digi max, RP-photonic, Additive Lab, Procast arena, Auto mode and Sim process. These are all some of these softwares where you can do simulation and try to see the part, how is it getting printed?

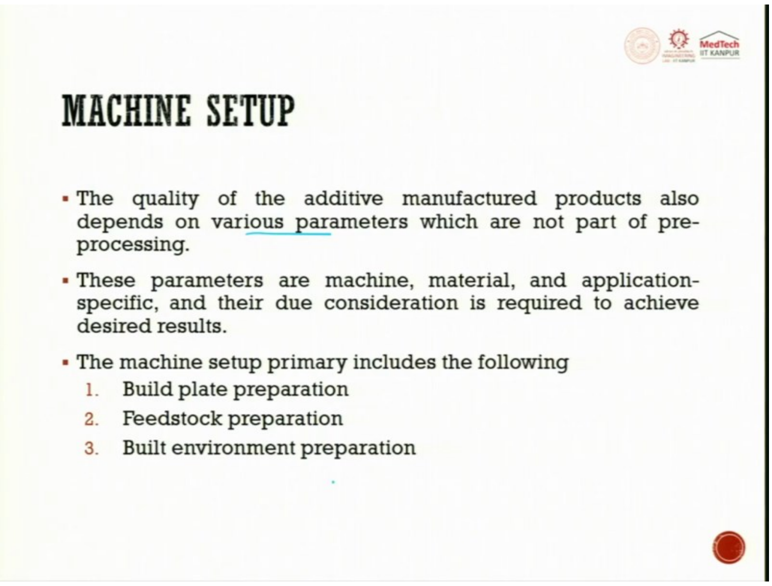
(Refer Slide Time: 48:11)



The steps for building file preparation- check the geometric input as per the standard file format, check for file error and rectify the error if there is any, check for the part orientation consideration, build time, support requirements, stair case stepping error, etc. This is very important in your course, addition of support structure in automatic and semiautomatic mode by providing the support type criteria for support generation type of support, etc. as input nesting of the same are multiple components as per the requirement.

Nesting means trying to build more components in one shot, slicing the part and support geometry by providing layer thickness as input. We saw the effect of layer, thickness, height, generation of tool path, and selection of scanning strategies, input the process parameter specific to the process. Generally separate process parameters are provided for support structures to allow easy removal. So, input the process parameters specific to the process. Last, rapidly review the geometry and the laser tool part slice by slice.

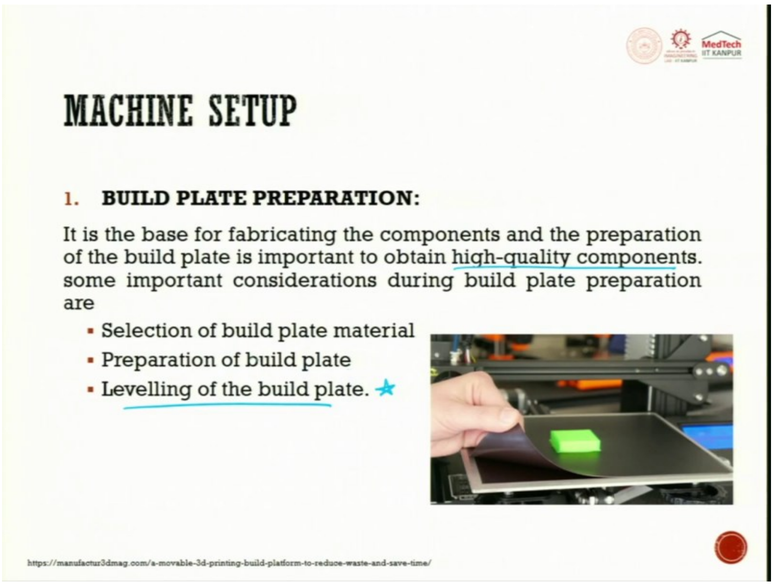
(Refer Slide Time: 49:24)



When it goes to the machine setup, the quality of the additive manufactured product also depends on various process parameters and not on part pre-processed preparation. These parameters are machine, material, application specific, and their due consideration is required to achieve the results.

The machine set up primarily includes build plate preparation, feed stock preparation, built environment preparation plate. And then next is feed stock, feed stock is the powder here. Then build development. Environmental is the jacket, whatever is built around the platform.

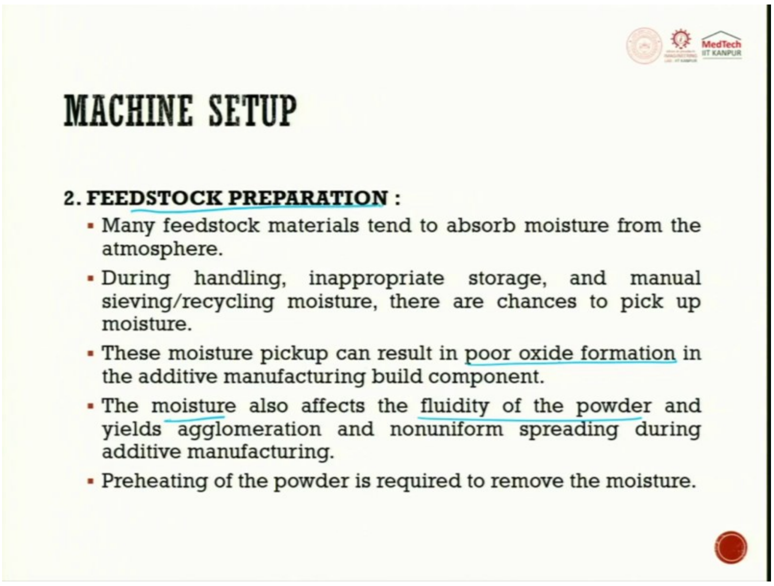
(Refer Slide Time: 49:58)



So, this is build plate preparation. So, this is the build plate. When we do DED, build plate is important, even when you do material extrusion. It is the base of the fabricating, the component, and the preparation of the build plate is important to obtain high quality components.

Some important considerations are selection of build plate, material, preparation of build plate and levelling of this is very important. Recently, we, missed out this levelling and we lost lot of money because we did not level the bed before doing the SLS process. So, preparation of the build plate and selection of the build plate are very important.

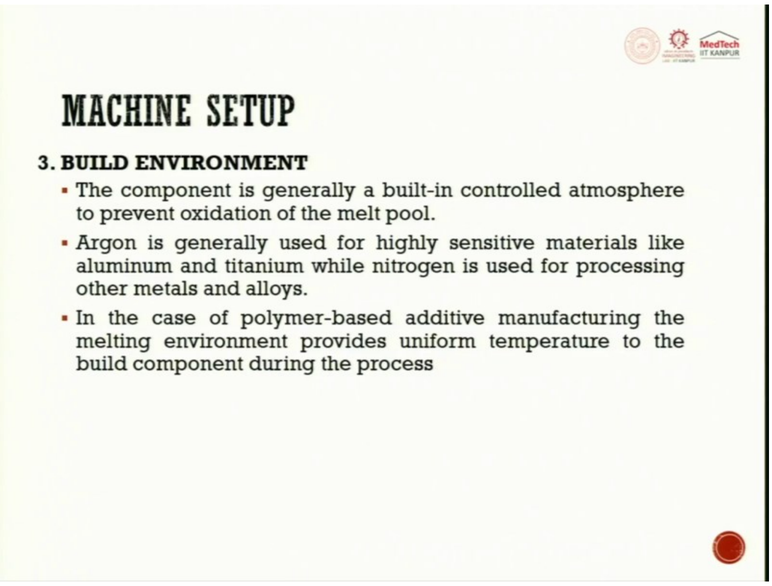
(Refer Slide Time: 50:39)



Then feedstock preparation, many feedstock materials tend to absorb moisture. So, we have to be careful. Even the metal powder can absorb moisture during handling inappropriate, storing and manual sieving recycling moisture, there are chances to pick up the moisture so they all can happen. And once the moisture is picked up, when we do SLS or SLM process, there is a problem of generating porosity.

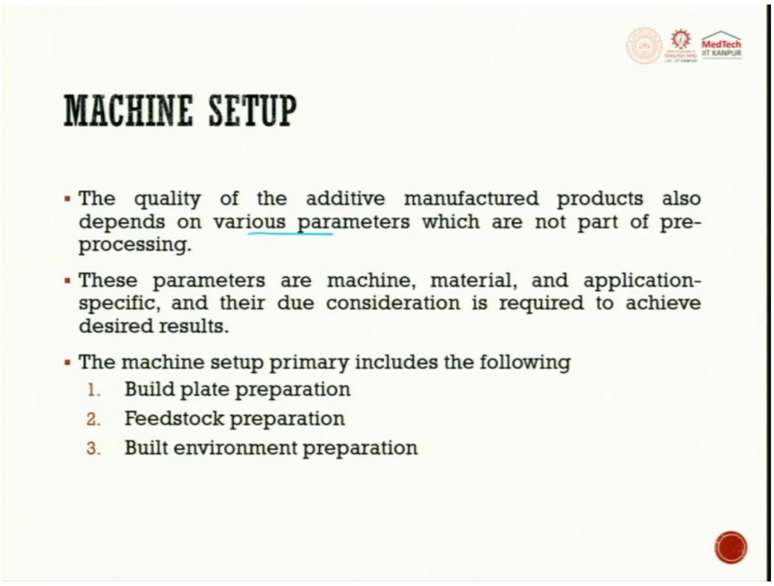
This moisture picked up can result in poor oxidation formation in the additive manufacturing build components. The moisture can affect the fluidity of the powder. So, the flowing of the powder through the nozzle, that can be before it is getting used by a recoated blade, the fluid, the moisture can affect the fluidity of the powder and yield agglomeration and non-uniform spreading during the additive manufacturing, preheating of powder is required to remove the moisture. So, these are all feed stock preparation, which is to be done before the process.

(Refer Slide Time: 51:40)



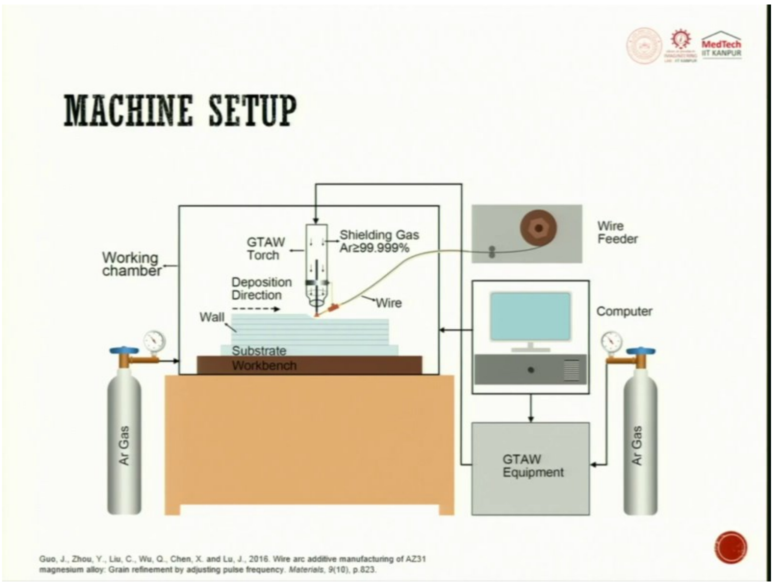
Build environment. The component is generally a built-in controlled atmosphere to prevent oxidation of the melt pool. Argon is generally used for highly sensitive material like aluminium and titanium while nitrogen is used for processing of other metals and alloys. In case of polymer based additive manufacturing, melting environment provides uniform temperature to the build component during the process. So, build environment is also very important.

(Refer Slide Time: 52:14)



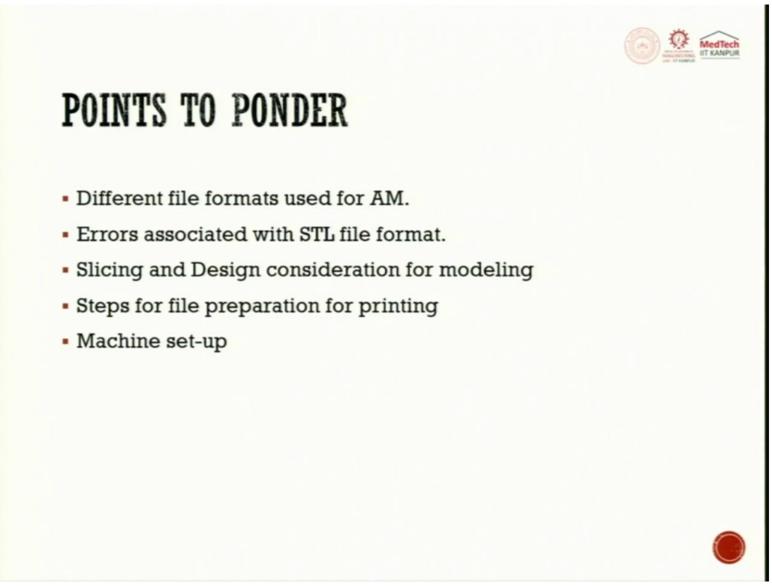
Three important steps in machining setup is build plate preparation, feed stock preparation, and built environmental preparation.

(Refer Slide Time: 58:24)



So, this is the machine set up. You can have a wire which comes here. You can have a shielded gas. This is for wire, DED. You can have powder. So, you will have argon where in which they maintain the working chamber temperature and pressure such that the oxidation does not happen.

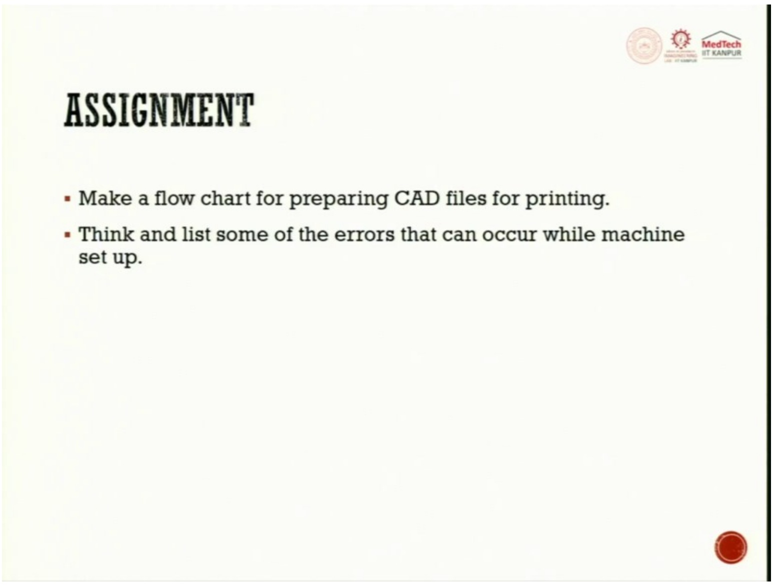
(Refer Slide Time: 52:54)



So, now to the end of this lecture points to ponder, what are the different types of file formats we use in AM? We used STL, OBJ, PLY, AMF, 3MF. So, what are all the different errors which are associated? So, errors are associated while creating tessellations. So, you can have degeneracy missing file, all these things overlapping, Slicing and design consideration for modelling.

So, design consideration, we saw so many design considerations. Next steps for final preparation of printing. We saw three steps and the last one is machine setup. We have clearly understood how do we convert your CAD file into your machine-readable file and what are all the issues which come while conversion and after it is converted, when you start processing, what are all the difficulties you face and how will you remove those difficulties by doing post processing and by valid inspection.

(Refer Slide Time: 53:50)



The assignments are two. Again, it need not be submitted, make a flow chart for preparing CAD file for printing. Think and list some of the errors that can occur while machining set up is not done properly. I said one is levelling error. So, like that you should sit down and list down all the things possible from the understanding of this course. Thank you so much.