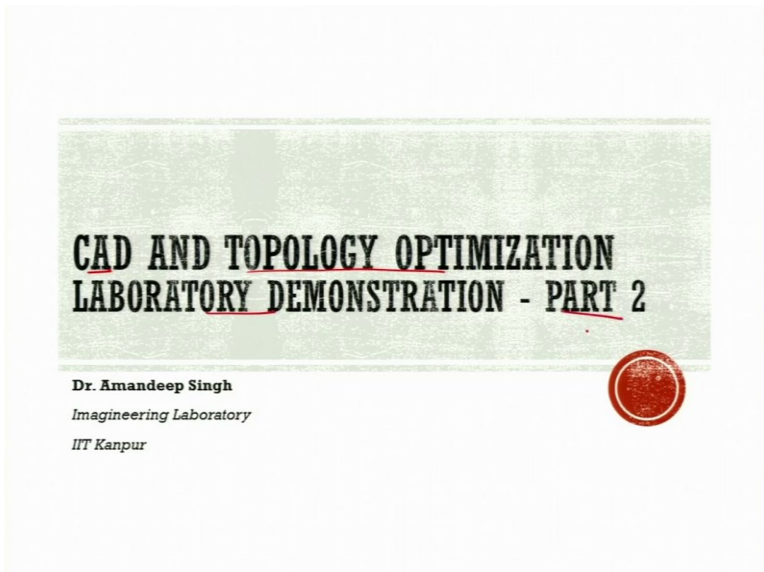
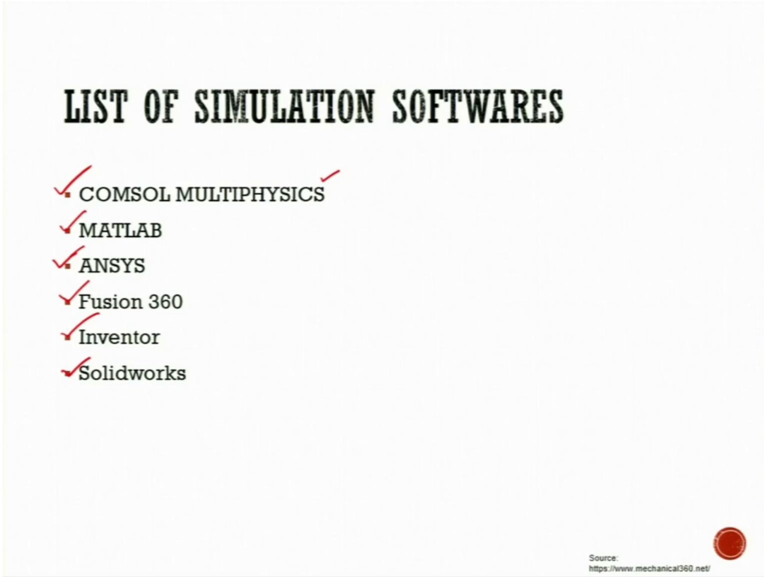
Metal Additive Manufacturing
Professor Doctor J Ramkumar
Professor Doctor Amandeep Singh
Department of Mechanical Engineering and Design
Indian Institute of Technology, Kanpur
Lecture 35
CAD and Topology Optimization Laboratory Demonstration 2

(Refer Slide Time: 00:18)



This is the second part of the lecture on CAD and Topology Optimization Laboratory Demonstration. We discussed about the CAD design. We did design using Solidworks software in the previous lecture. We developed arms, I have just made further steps on those arms, three arms in the three different directions at three different angles, < 45^0, exactly 45^0 and > 45^0. Also, I have made flanges over the ends of those arms. Those, I will try to demonstrate using the Ansys software for topology optimization.

(Refer Slide Time: 00:55)



Before that let us have a look over the different simulation softwares which are available. For example, COMSOL MULTIPHYSICS. COMSOL and Ansys are two major softwares which are used. COMSOL use MATLAB as a space. So, if we suppose, we need to use very general physics models and save time in general, because computational COMSOL takes time. So, Ansys is recommended. And if we have the best access to physics and we understand the MATLAB, the coding in detail, COMSOL can also be used.

So, Ansys is generally used for volumetric simulations, COMSOL can also do volumetric simulations with some different or separate methodologies. But it is further extension to go deep to design, to have deep research on the components as well. Now, COMSOL provides the finite element analysis, that is Multiphysics and solvable simulations of the partial differential equations in general.

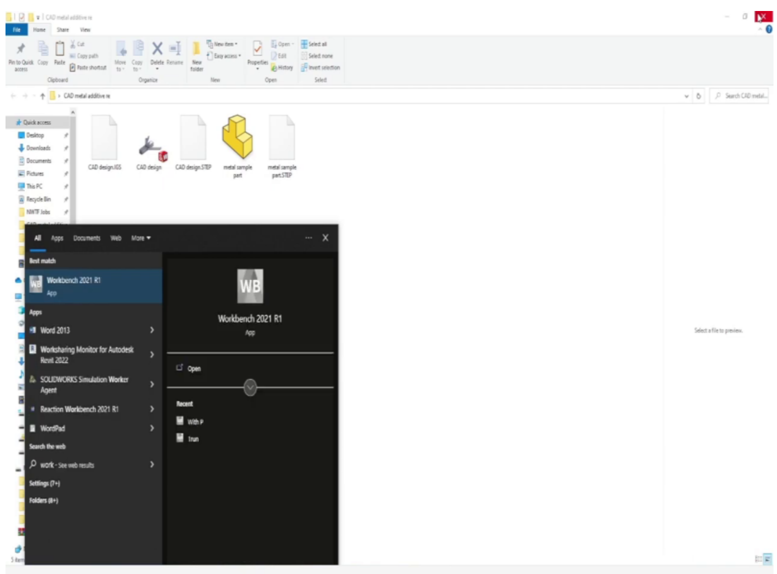
And these are physics-based interfaces which provide the mechanical, electrical, fluid, chemical applications. Similarly, MATLAB uses math works and it was created basically for math works and using the math works and numerical computing only and small programming language, it has a toolbox to provide simulations and modeling capacities.

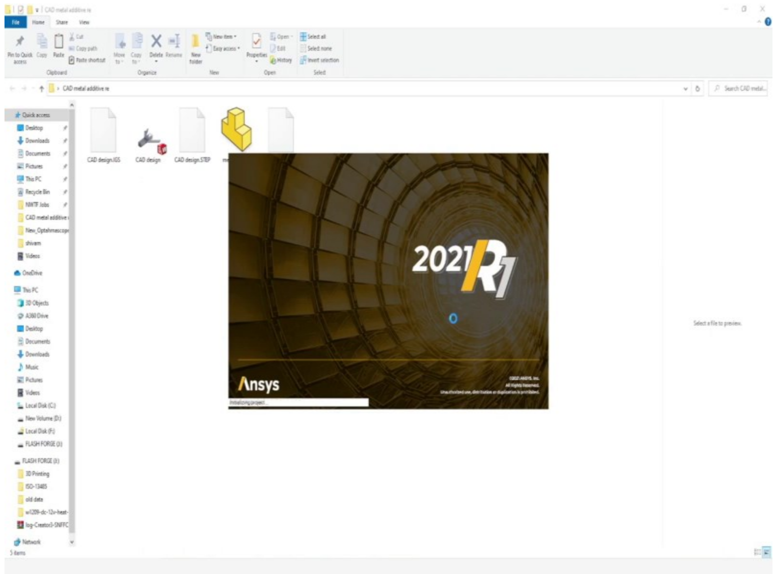
Then we have Ansys. Ansys is basically mechanical finite element analysis software and is used for computer models as I just said, the volumetric models for different structures, for electronics, machine components for analyzing different parameters for example, strength, toughness, elasticity, temperature distribution, in computational fluid dynamics, fluid flow, electromagnetism. So, all different features could be seen here in Ansys.

Next comes Fusion 360. As I told in the previous lecture as well here also, small simulations which can be used for machining, manufacturing, small industrial designs, that could also be conducted. Then to create tool paths for computer aided manufacturing, CAM. This is also used. Then we have Inventor. Since Autodesk has expanded from their base home, this provides another software support, Inventor, which helps in simulation, animation, then photorealistic rendering as well. These are all taken care in Inventor.

Then we have Solidworks simulation as well, in which again, finite element analysis to predict the real-world behavior of a product, is virtually designed. So, it also provides the portfolios in linear, non-linear, static, dynamic, different kinds of analysis that could be used. So, now I will come to the Ansys software. I will try to import the file that was generated using the CAD in Solidworks and try to have a small demonstration here.

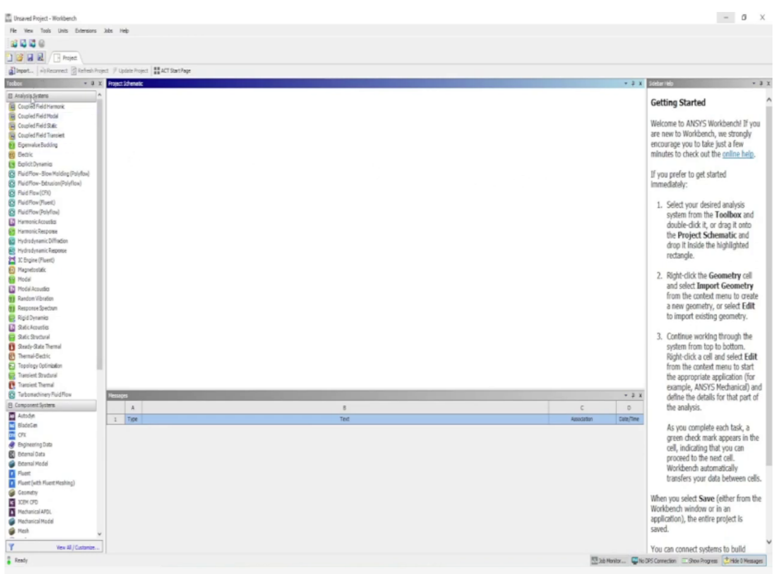
(Refer Slide Time: 04:04)

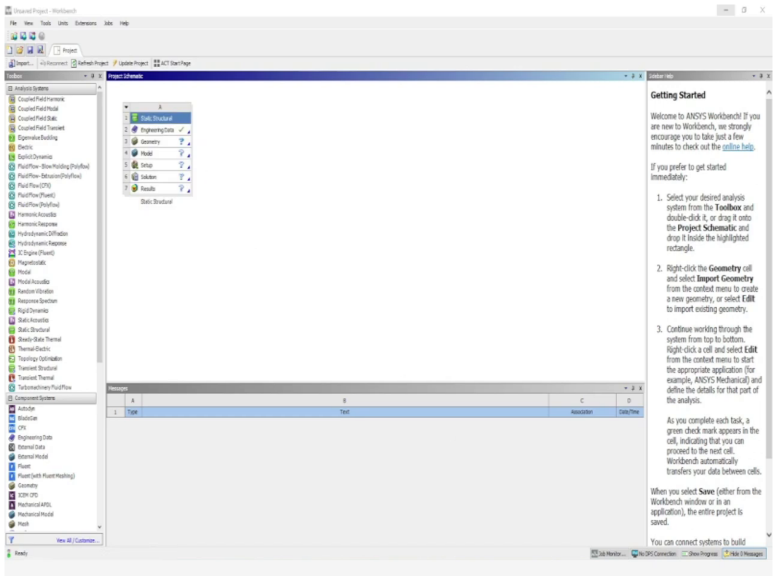


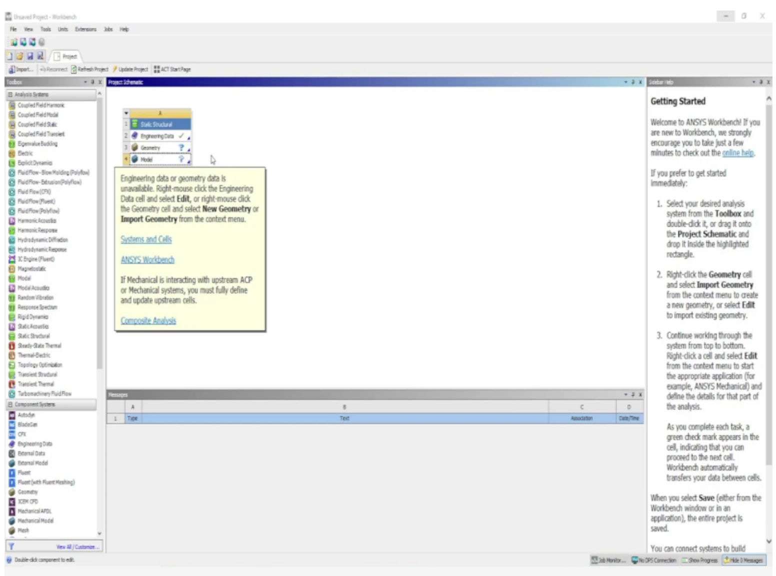


Since, now the CAD model was ready, we are opening now the Ansys software to understand the capacity of the model that is generated there. So, we can do small static analysis. So, I am just typing here Workbench. Workbench 2021 R1 is the Ansys software that would be used. So, different modules, selection of materials, for designing, as well as, different simulation processes that we can do in this software.

(Refer Slide Time: 04:39)







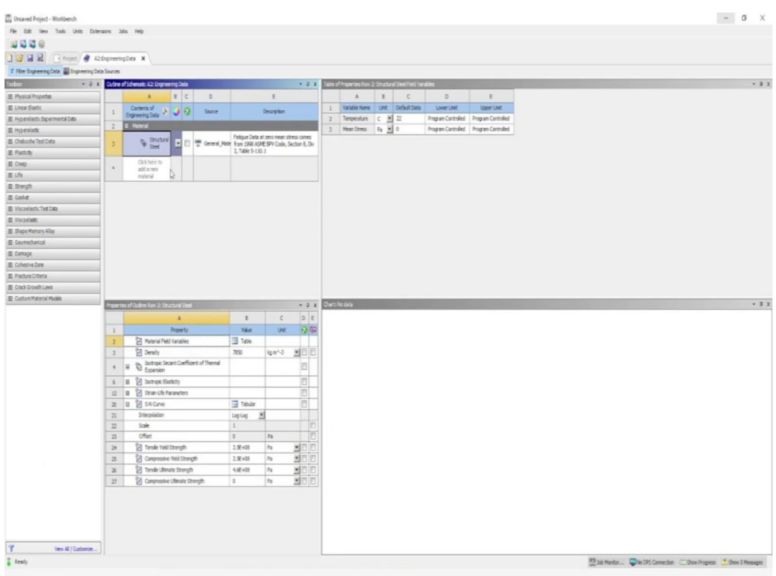
So, this is a software interface. You have the workplace area here; we have the left windowpane. So, this is project schematic. We have messages down there, if some messages, they keep on giving, regarding the information on the steps that we conduct here. Now, it has opened. So, this is the main interface in which we can have left pane here and the workspace here.

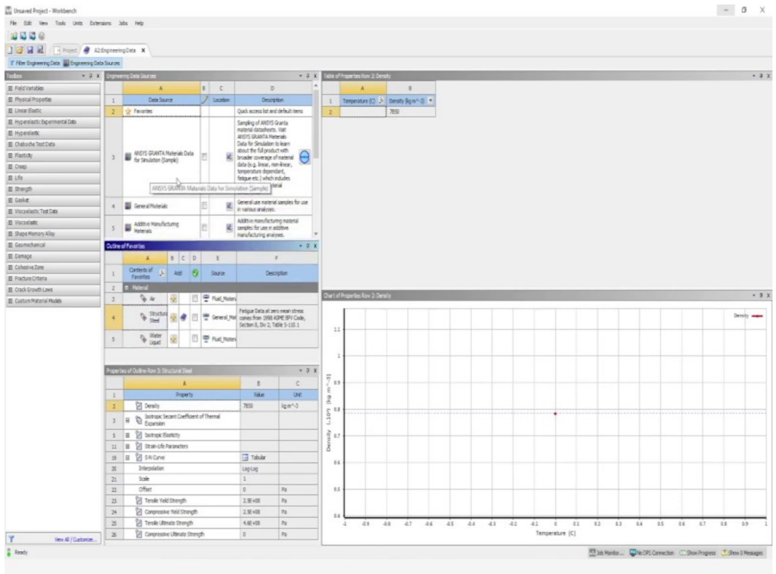
So, in the left pane, you can see the analysis systems, that is the coupled field, so, random vibrations, general acoustics, different kinds of models could be built here. So, CFT could be used, so fluent with the machine, we can use the geometry here. The component systems. Then IGM, CFT systems, the static structure could also be used for static analysis.

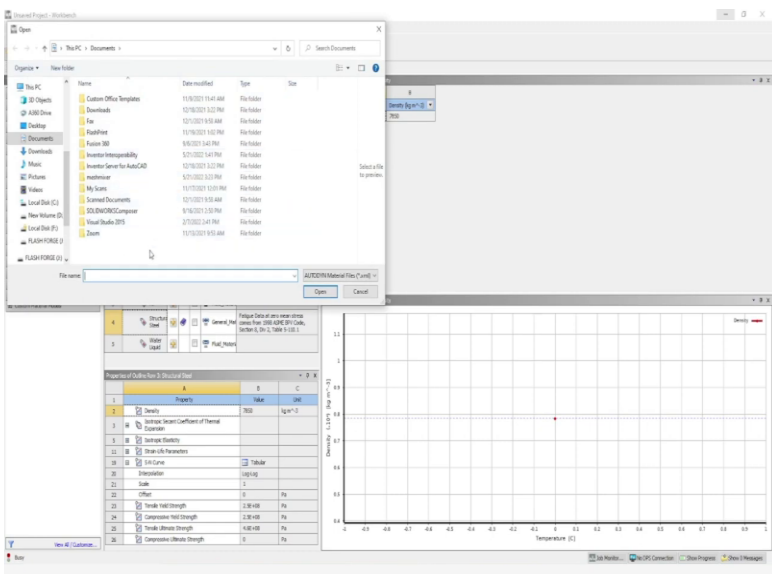
Then explicit dynamics would also be used for the non-linear dynamics if we suppose have something here like that. Then transient thermal analysis could also be taken. So, turbo machinery, fluid flow, in the fluid dynamics. So, different kinds of the models could be used depending upon our requirement. And our learning in that software, learning and using that module, giving the right input is most important here.

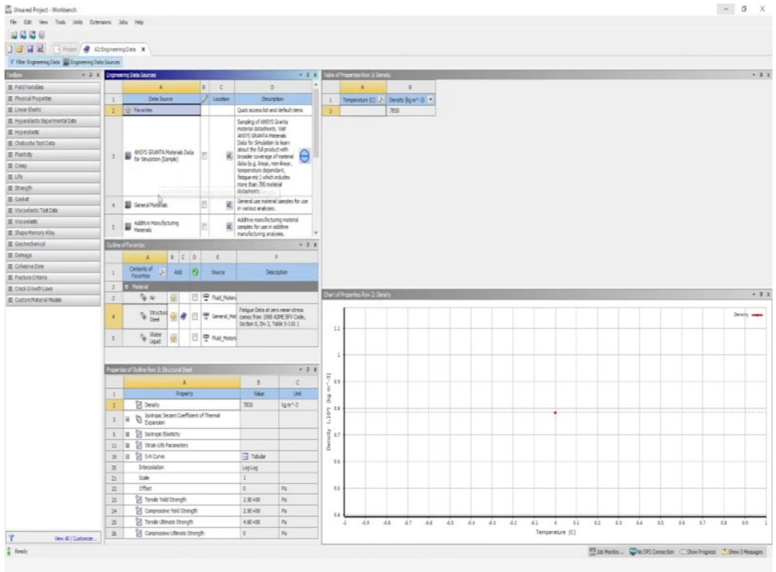
So, we will now have a tree for the simulation that we are going to conduct. So, this is the tree. In this tree, we have static structure, we have different options available like the engineering data. So, we can specify the material, linear non-linear, inputs geometry, so geometries which are made in general works, that can be induced into this interface. So, it shows that no data is specified, or we need to add it still in the model. Then we can add and edit model, we can add new geometry, we can import geometry from the context menu. So, then, in setup solutions and results could also be taken from here.

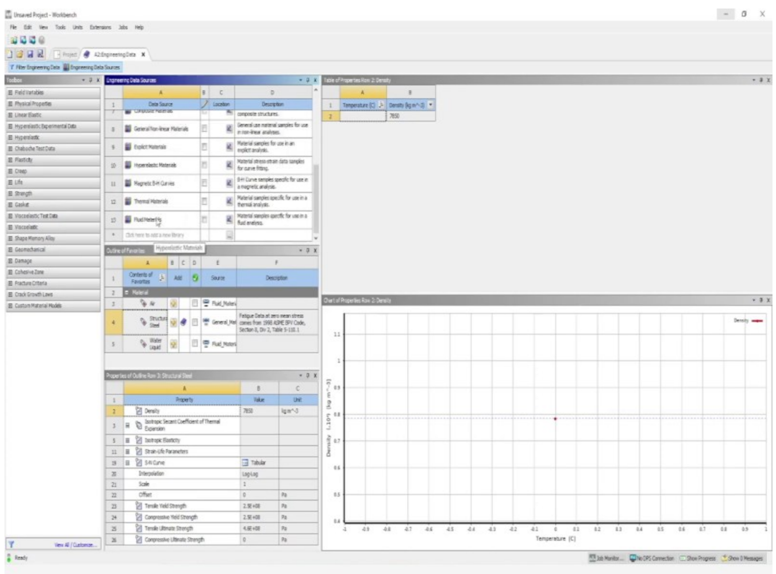
(Refer Slide Time: 07:14)

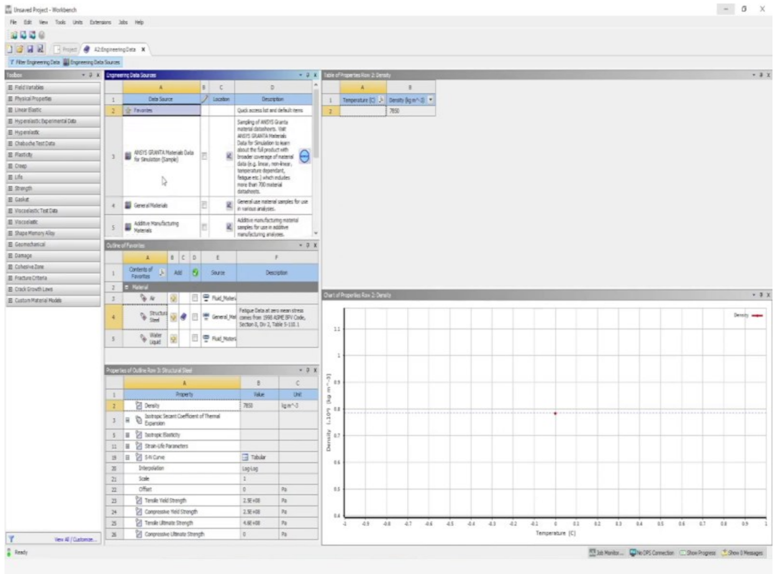


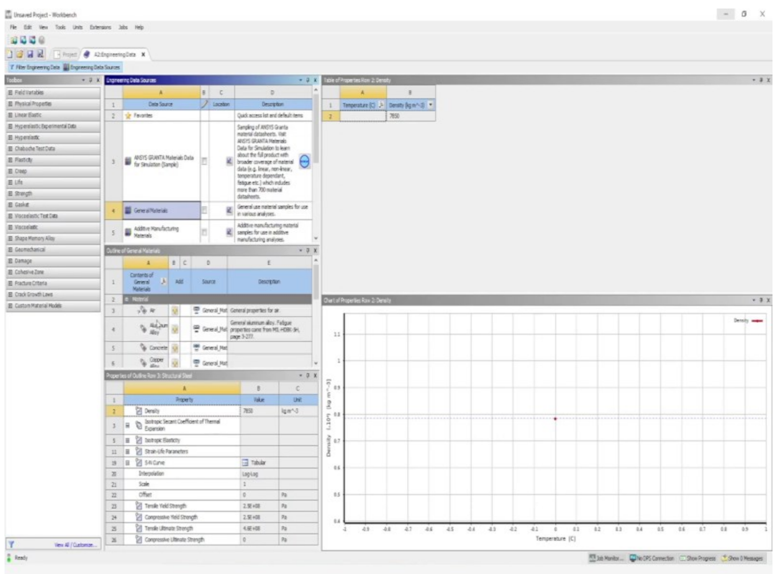


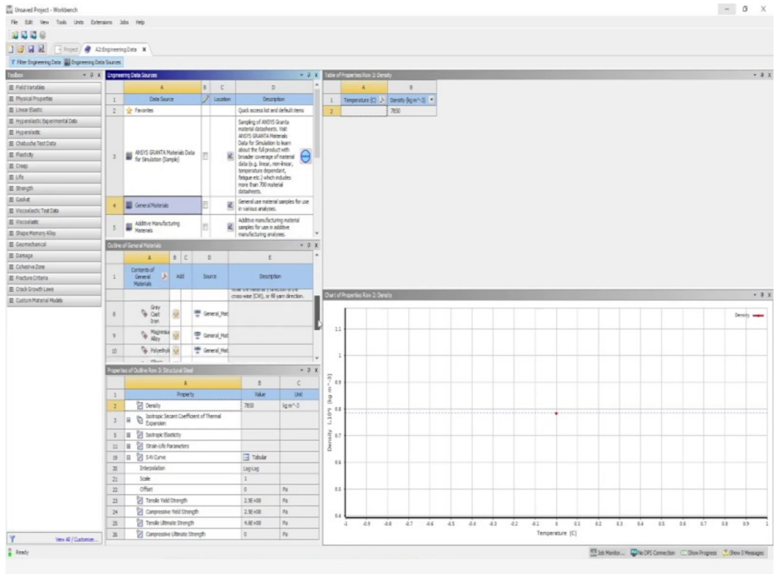


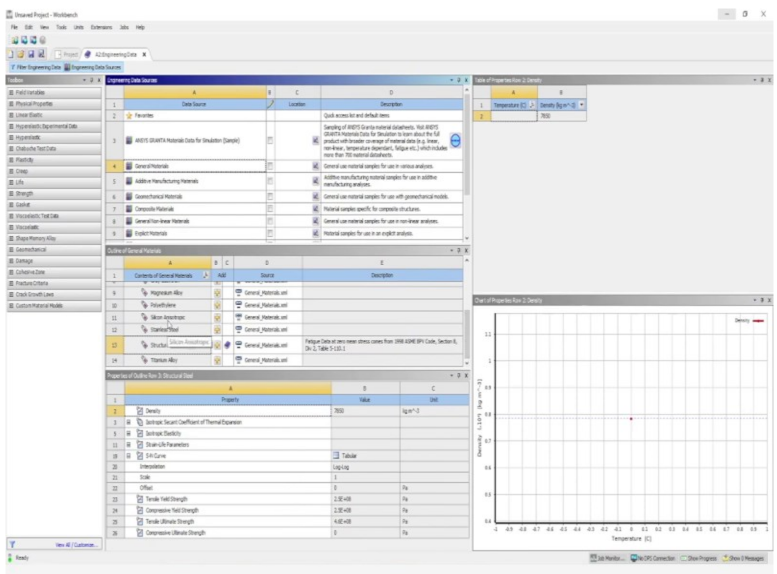


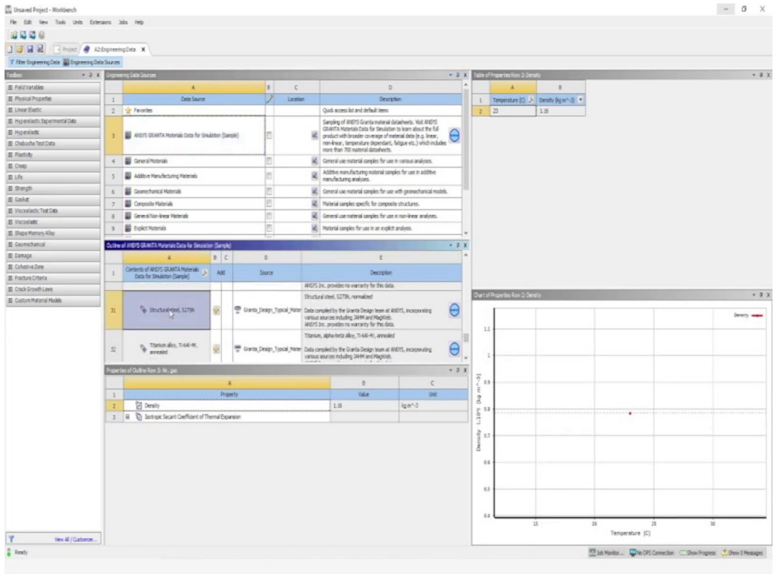


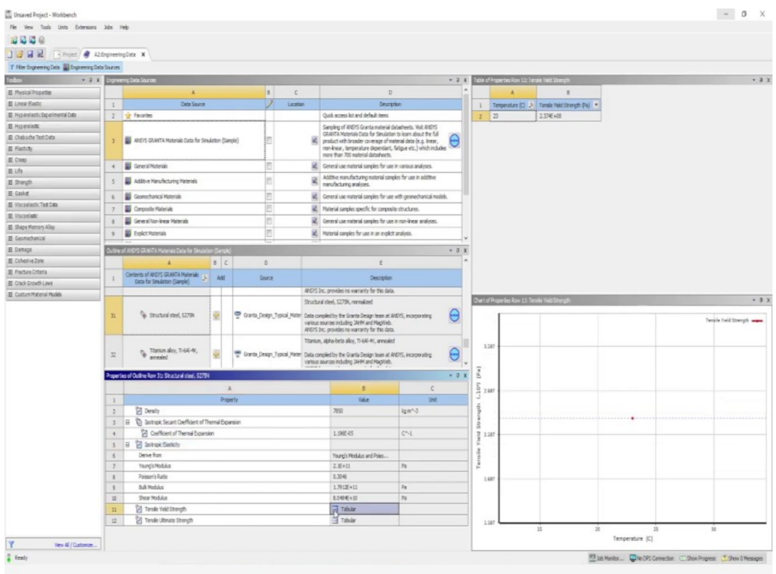


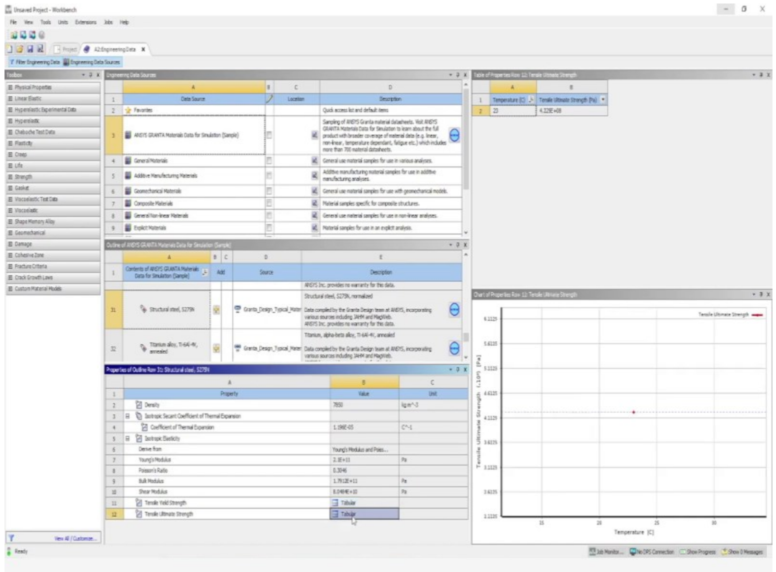


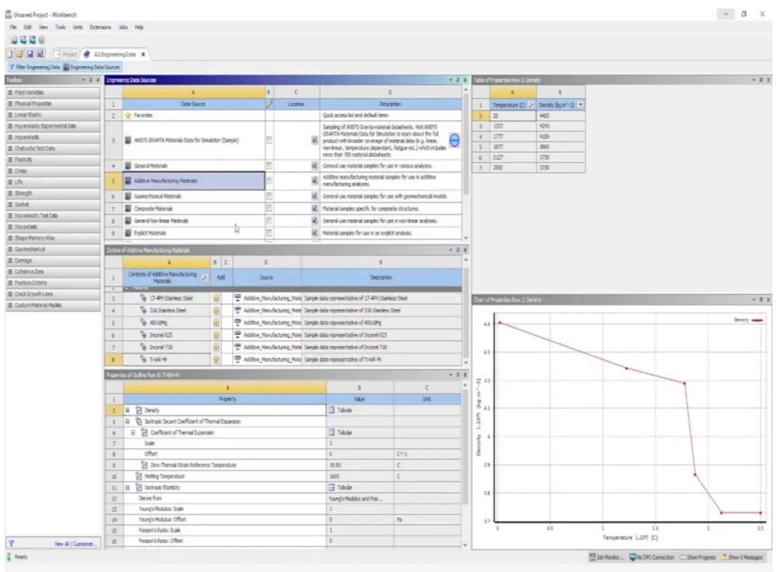


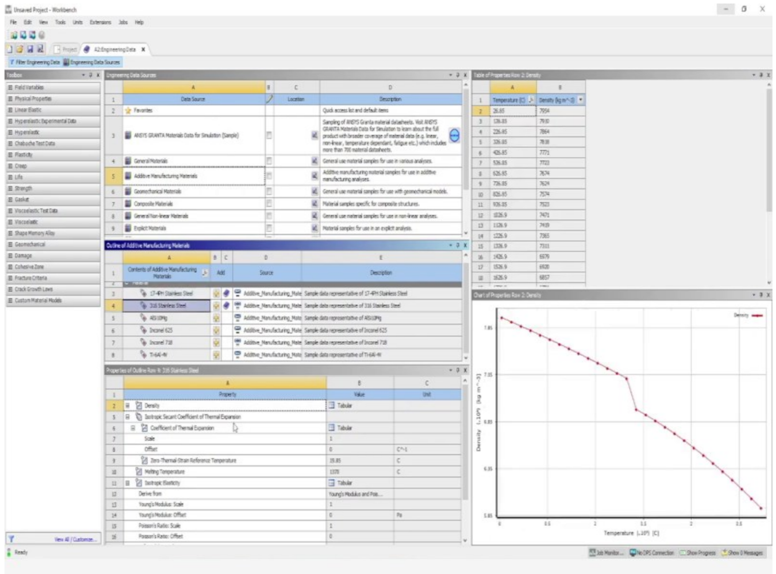


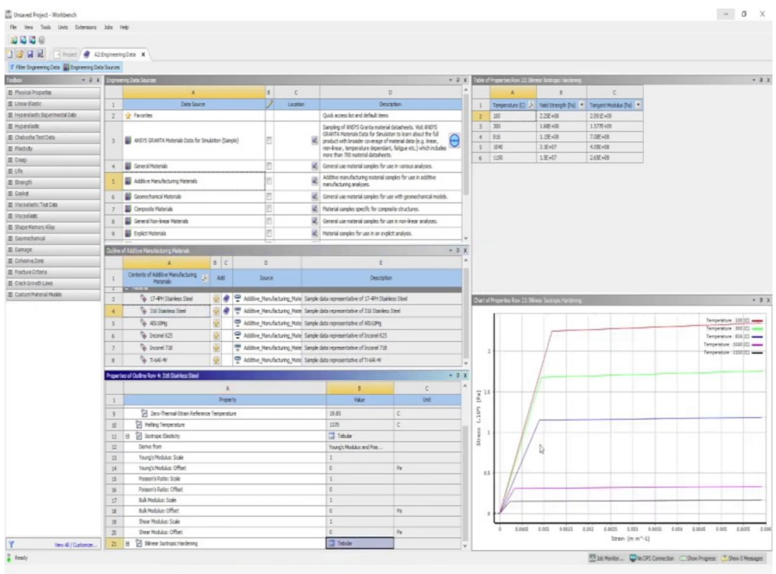


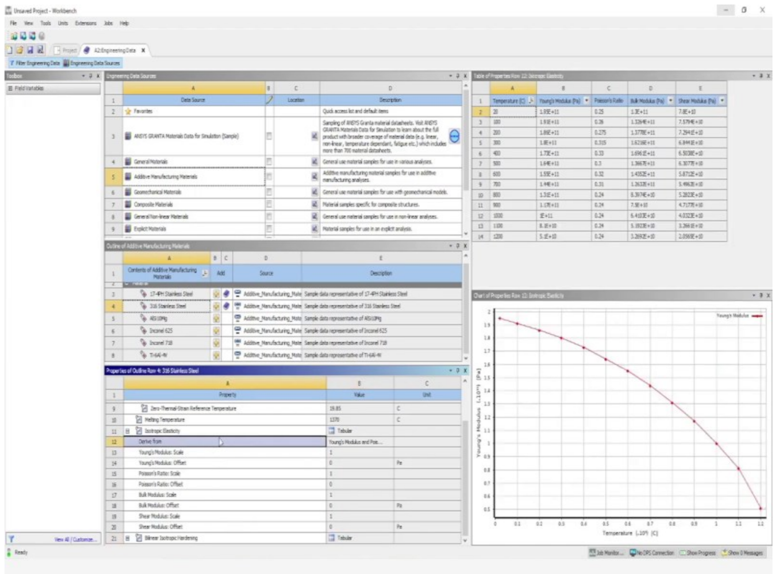












First, we come to the engineering data. To add the material here. In our library here, we can have different property, physical properties, linear, elastic, hyper elastic, then plasticity, creep life. Different kinds of the toolboxes are here from which we can put the features, the parameters that we are going to use or that we are going to fix as a range or as a fixed or constant.

So, in engineering data sources, Ansys Granta materials data has already been defined here. So, some materials in additive manufacturing would be here, some might not be here because materials are under development or so. So, we can add more materials and add new materials or new database here. So, this is option is also available. So, general materials and Ansys Granta materials, both could be used.

Then, geo-mechanical materials, composite materials, general non-linear materials, explicit materials, thermal, fluid, different kinds of materials could be used. So, if I select the general materials, it will now show the list of the materials which are general. So, aluminum, concrete, copper, stainless steel, gray cast iron, then magnesium alloys. So, many materials are here, and even polygon material, like the polyethylene, those are also here. So, to extend it further, I also see the detail. So, silicon anisotropic, stainless steel, structural steel.

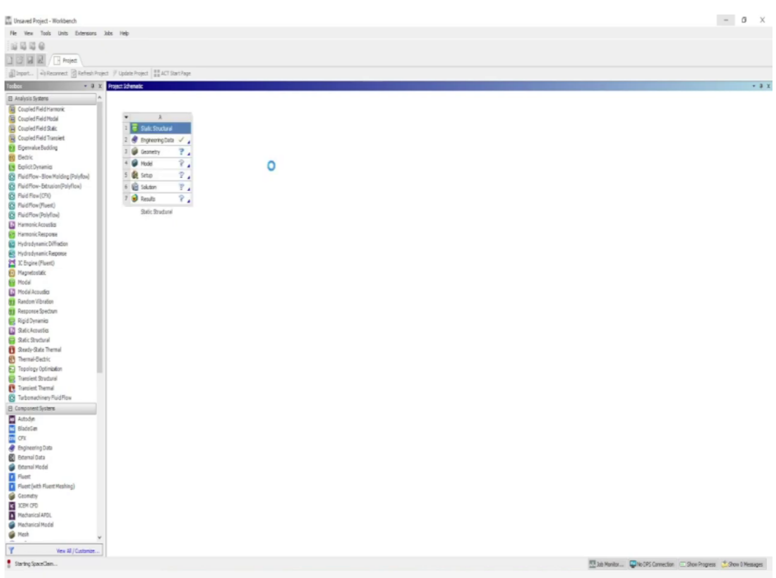
So, if we select here, it will show that it is added. So, this is a plus sign here. So, the general material. Ansys Granta material, it is showing the materials which are available in the Ansys base. So, structural steel, if I select structural steel, you can see the density, the properties of that material. Then coefficient of thermal expansion, then Young's modulus, Possion's ratio, bulk modulus, all those values can be seen here. Then, tensile yield strength.

So, for non-linear, it is in a tabular form. For the linear one, it is already a constant value, put here at the right, top here. So, for a tabular data, it will be a range. That is for non-linear. This can be taken for or put from the library. So, we can also add some material from here. So, if we wish to add, I will just click plus here, and this material will be added. So, this is added to the directory.

So, from this library, it will show that the book icon is here. That means it is added to our engineering data. You can also insert copper. We can select additive manufacturing materials. So, we are using 17 4 H stainless steel, which I have added. Then stainless-steel general. So, out of these two materials, stainless steel, I selected. So, these are the properties of stainless steel. So, coefficient of thermal expansion, everything is given here.

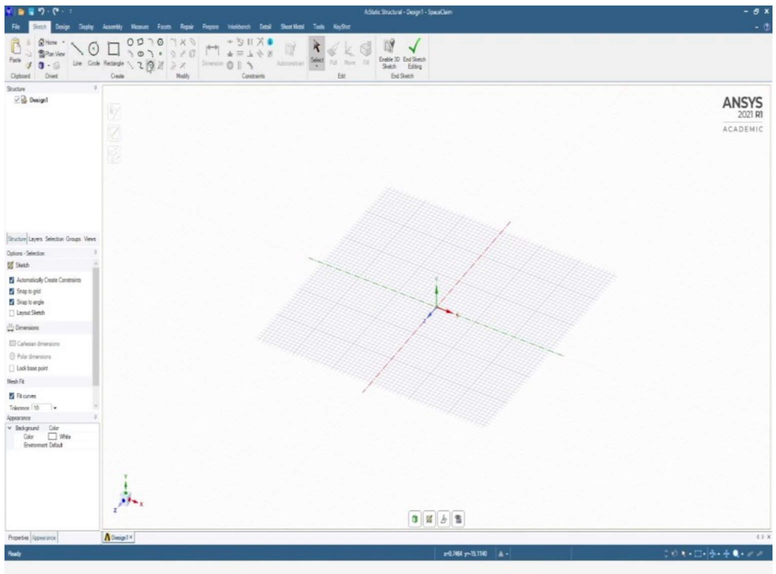
So, this is also table giving the comparison of the temperature and density, at different temperatures, how the densities behave. So, this is table given for the temperature and strengths. That is, the modulus, Young's modulus and the yield strength. Now, this is temperature versus Young's modulus, Poisson's ratio, bulk modulus, shear modulus.

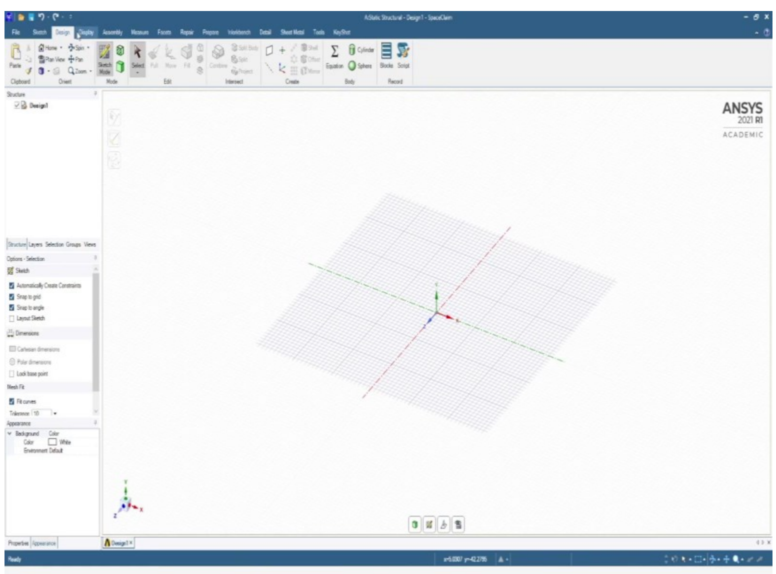
(Refer Slide Time: 11:49)

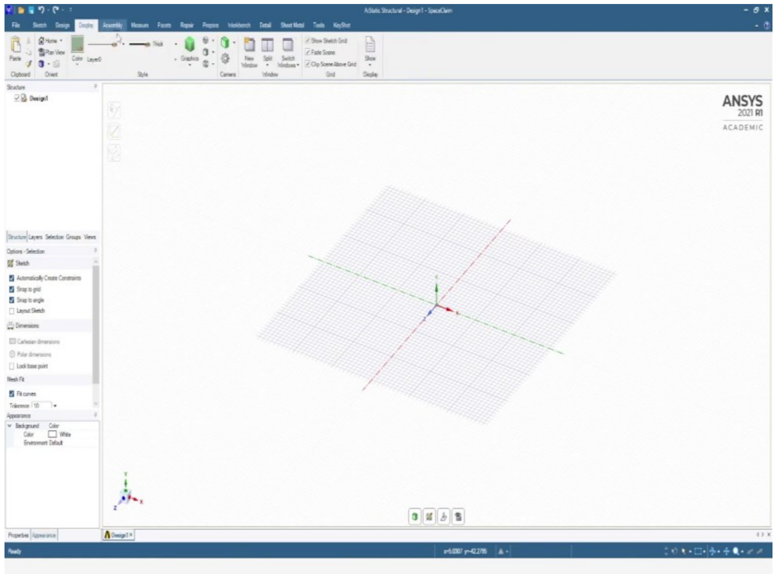


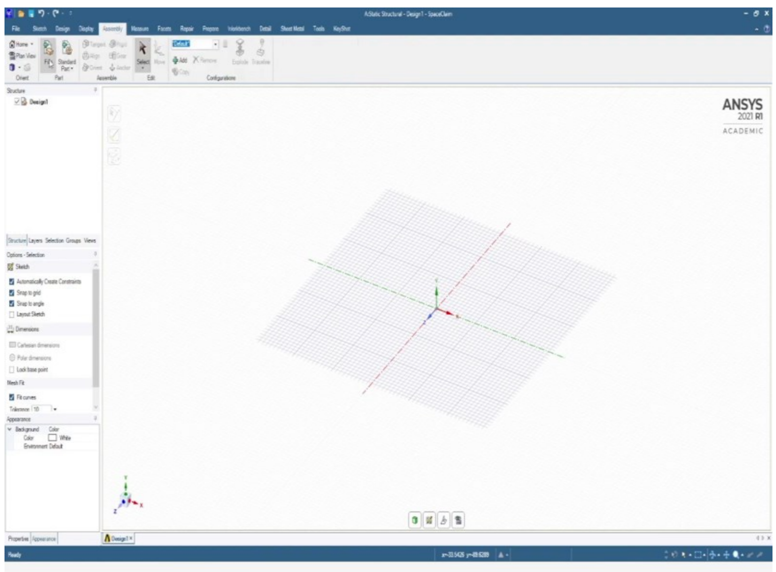
So, this material is selected as stainless steel. Now, we need to select, or import the data here, that is the geometry. So, the CAD, that is developed there, could be transferred here, or imported here. So, then we will have this static structure analysis over it. It is still opening. It takes time in loading. So, let me quickly come to the point when it is loaded.

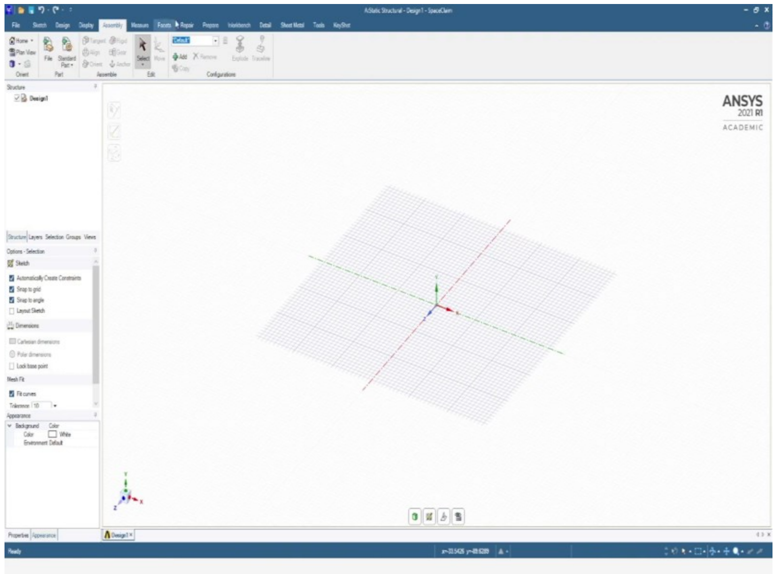
(Refer Slide Time: 12:15)

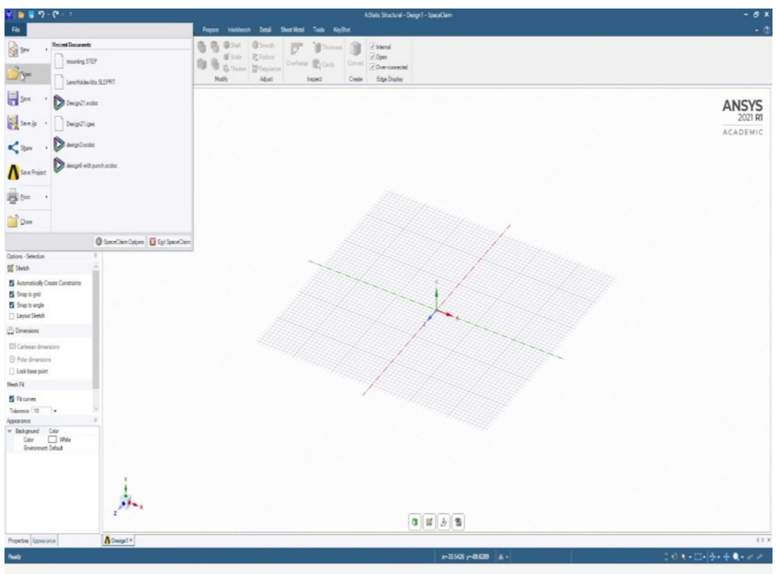





ANSYS
2021 R1
ACADEMIC

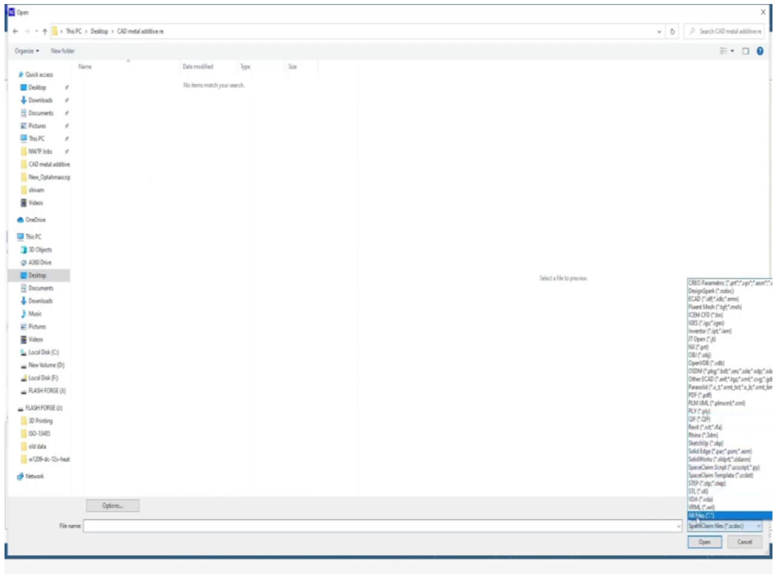

ANSYS
2021 R1
ACADEMIC

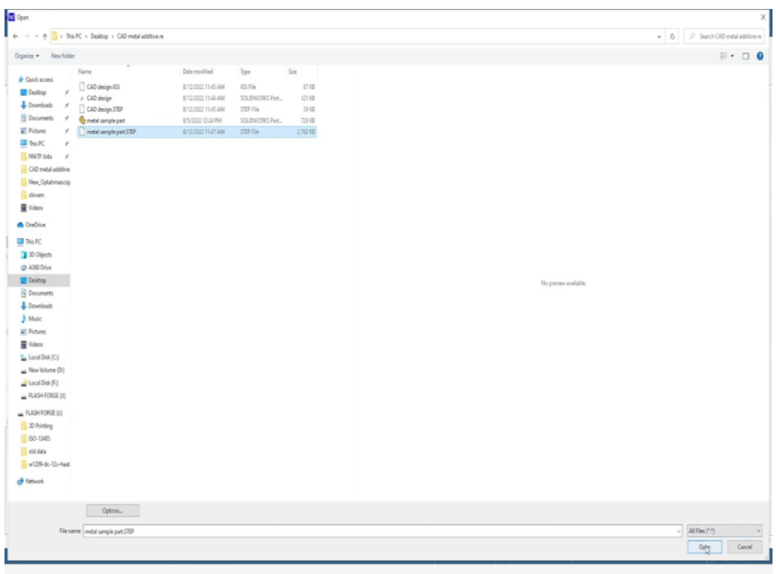




So, this is the page for the Ansys analysis workspace. So, similarly to the CAD, we have the tabs and the ribbons here, then a clipboard, orient, then sketch ribbons here. We can design, display. Display tab, you can see the style ribbon. In the assembly, we have assembled edited part, there is file or standard part. Then, we can measure it and can see the facets of it.

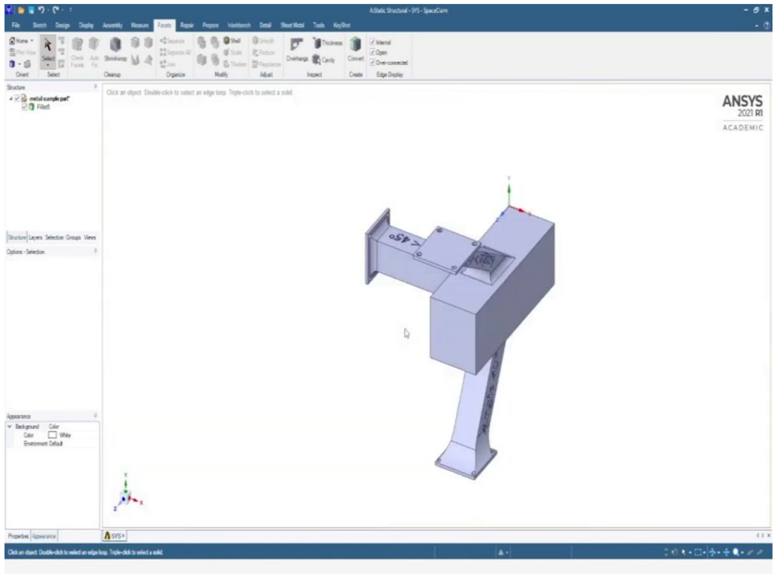
(Refer Slide Time: 13:01)

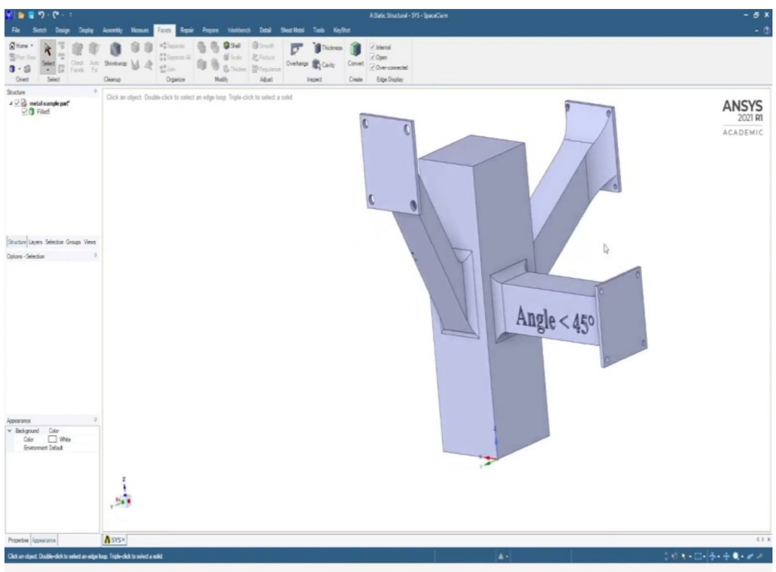


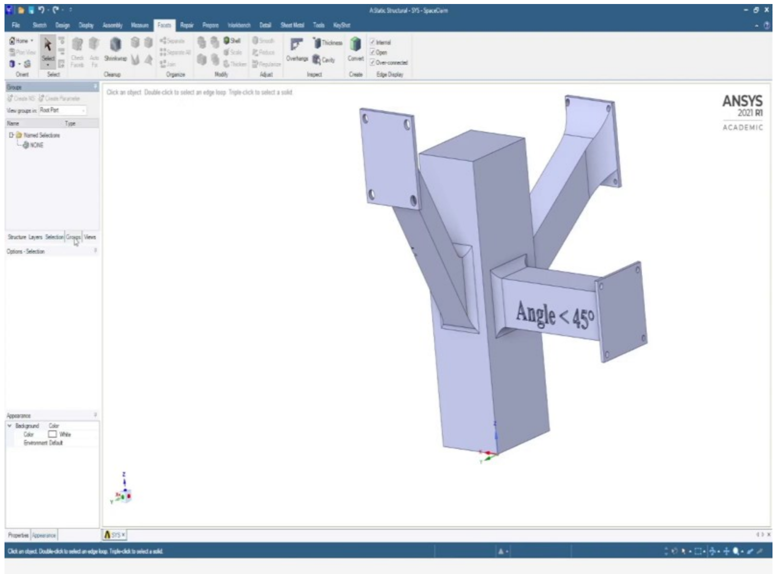


So, let me try to put the file from this saved folder here. It is kept in the desktop. So, all files, then metal sample part. Step format. That is imported here.

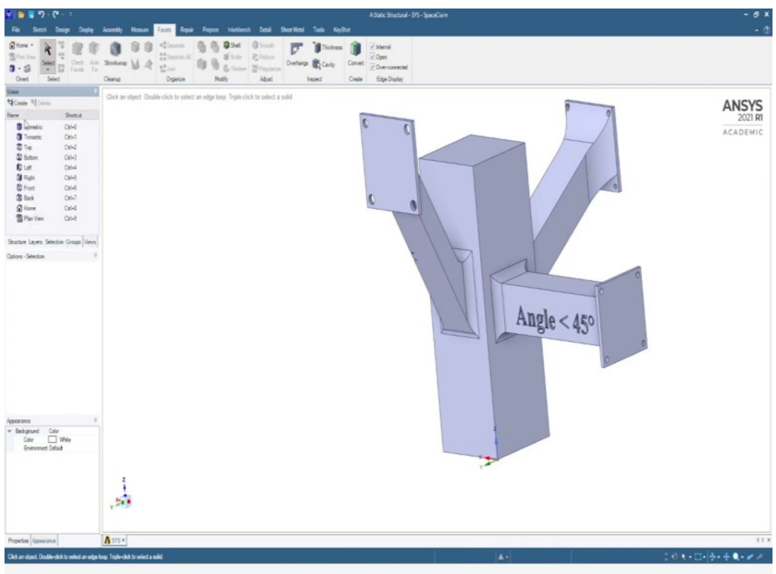
(Refer Slide Time: 13:20)



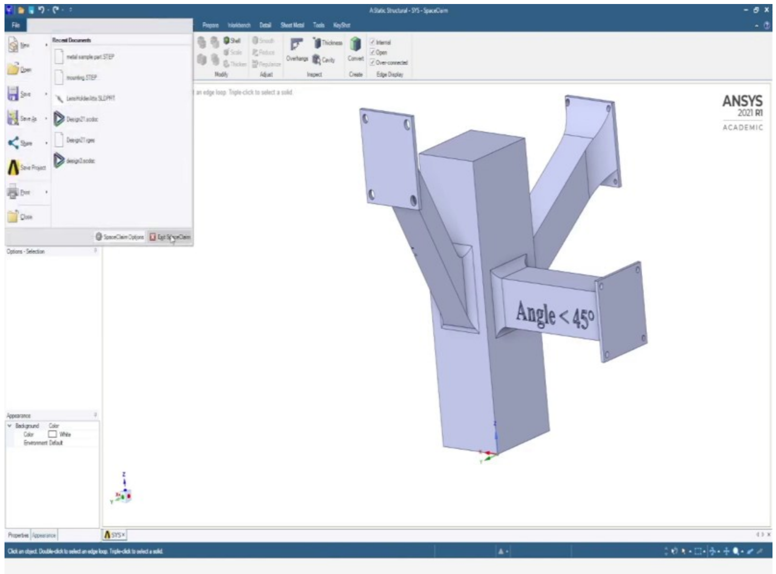




ANSYS
2021 R1
ACADEMIC
Angle < 45°

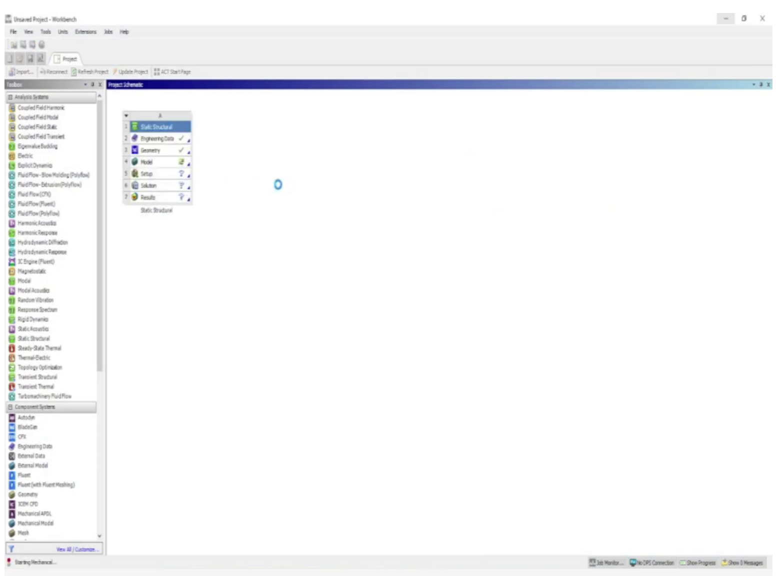


ANSYS
2021 R1
ACADEMIC
Angle < 45°



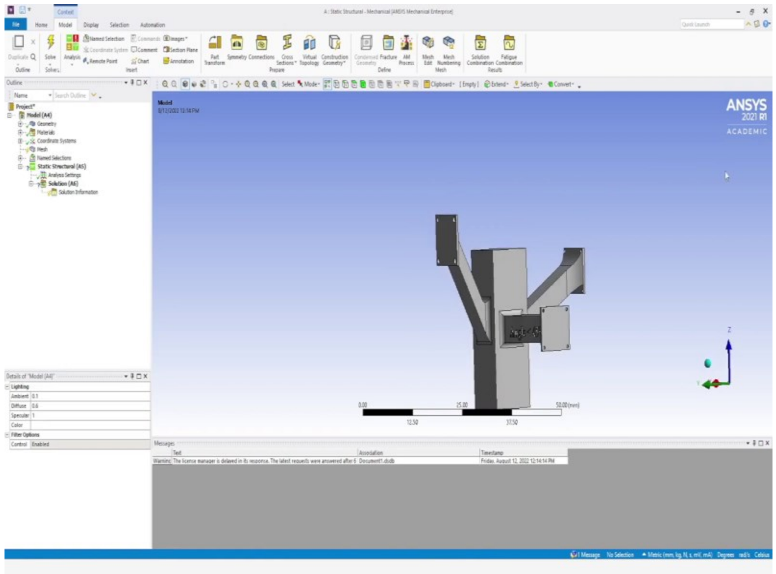
So, after importing, you can see, this is the design, which I modified from the one that I just showed you in the demonstration in the previous lecture, just to add a few flanges and to just make the angle, that is, 90-degree angle, as also square. So, if we need to amend and enter, so we can have different views here. We exit after importing and say ok.

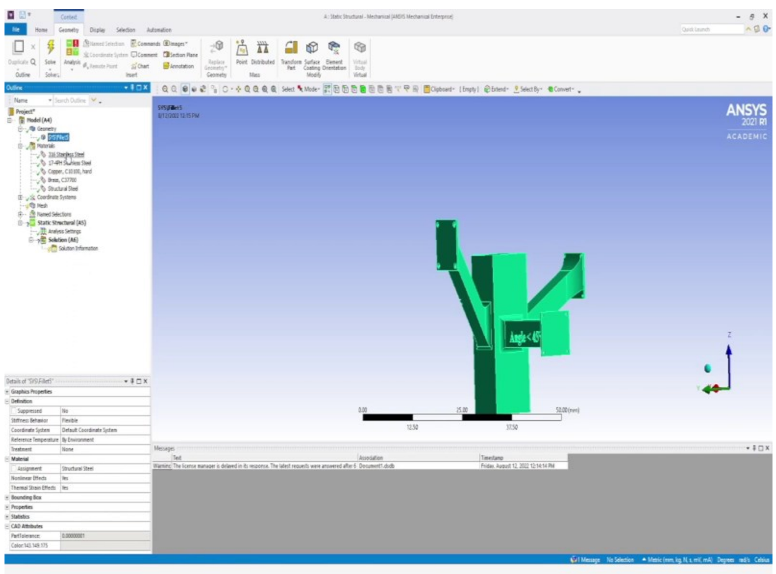
(Refer Slide Time: 14:04)



Then, we again come to the model here. The model option to give the boundary conditions, we need to specify what boundary conditions or based upon what different restrictions, our model is going to be developed.

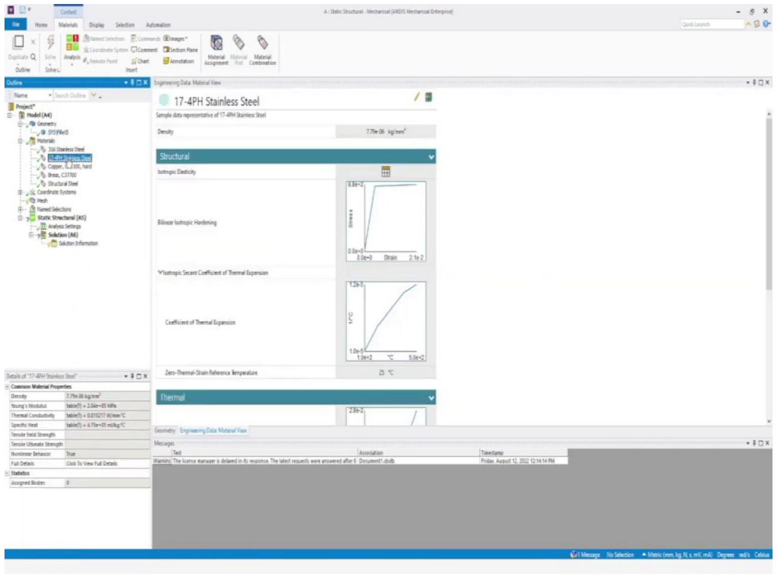
(Refer Slide Time: 14:22)

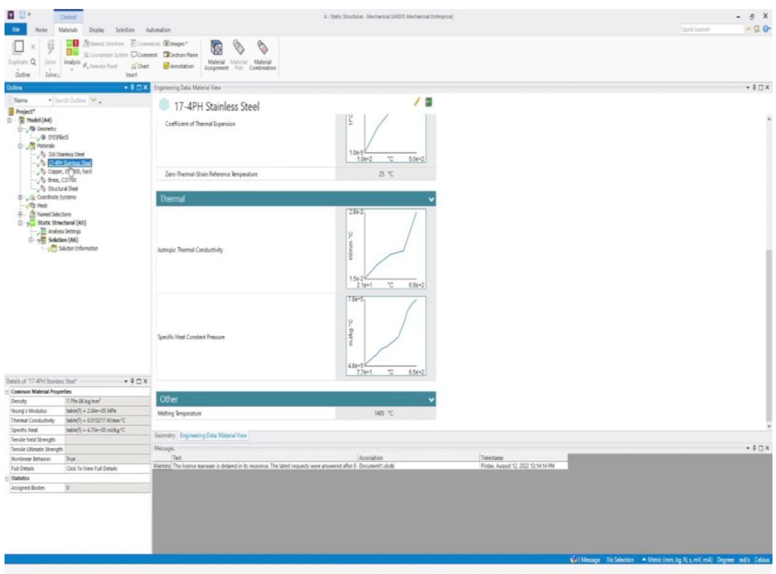


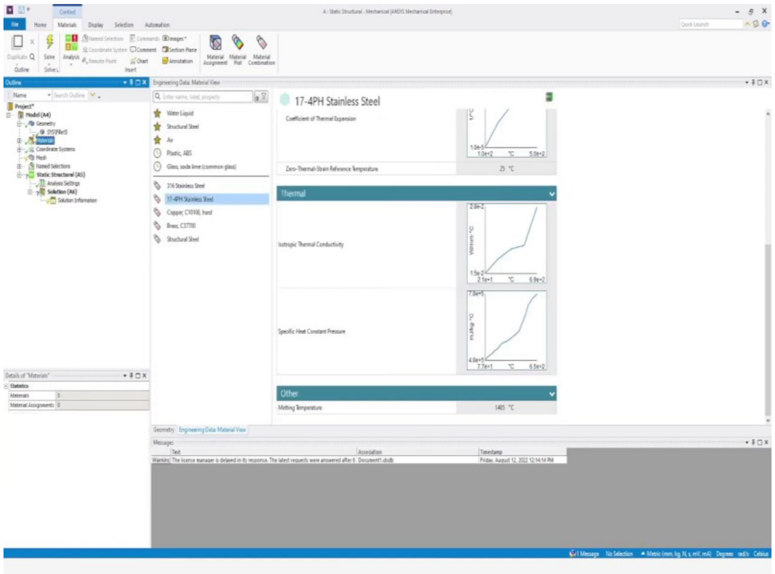


So, after opening, this was the model, model page, which is now opened. So, in the model page, we can see system, and we can also see the materials that we have added from the library. 316 stainless steel and 174 H stainless steel.

(Refer Slide Time: 14:47)

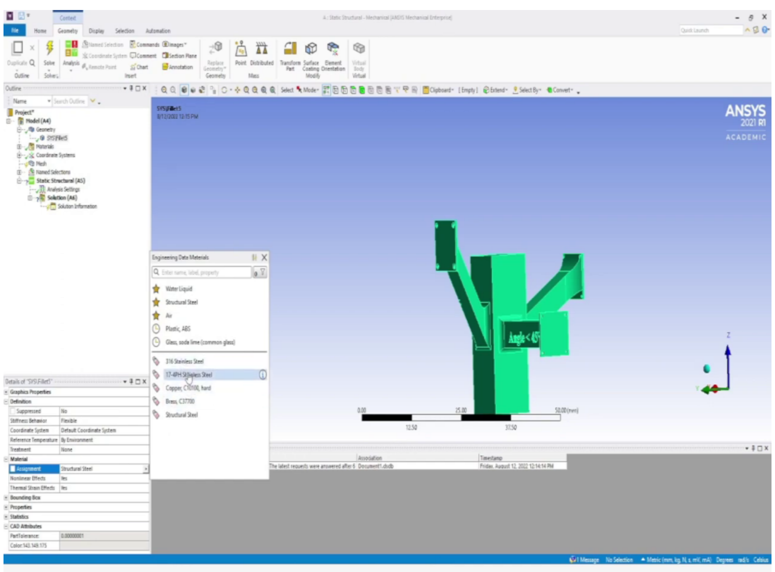


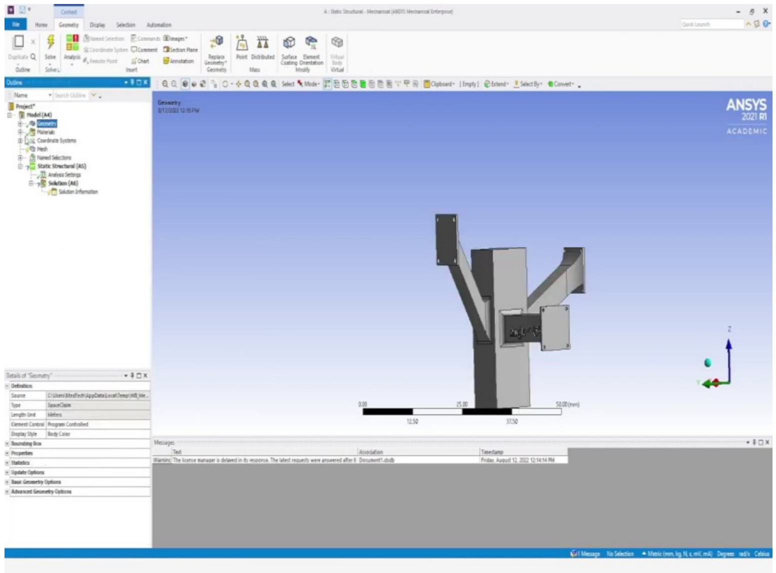


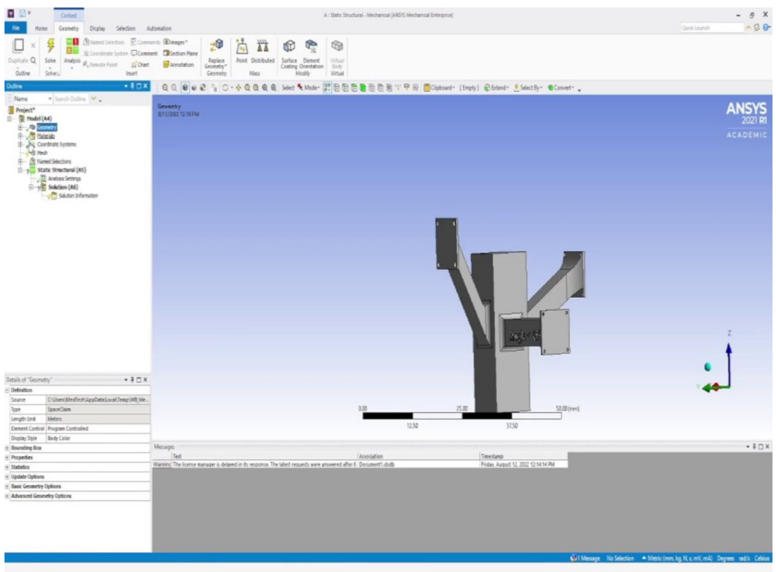


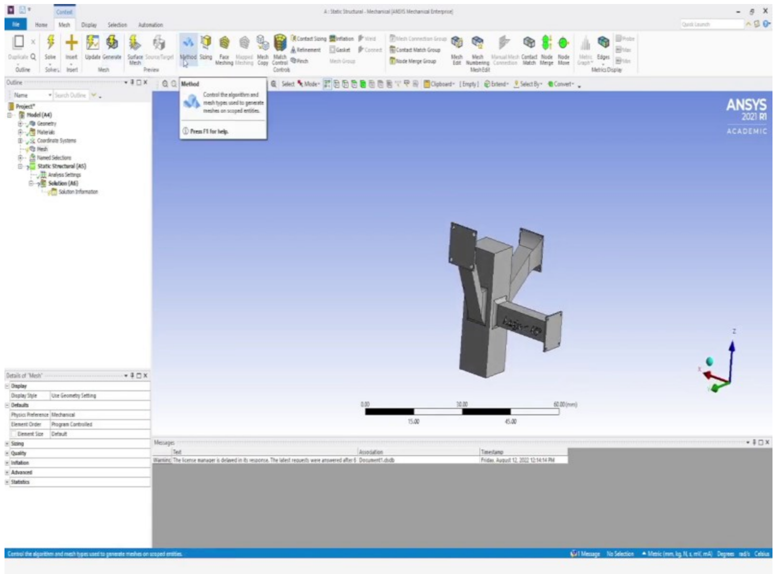
So, we will select 17 4 stainless steel. So, you can see the properties here, that could also be again seen. It is also showing, the graph of the different modulus, different mechanical properties.

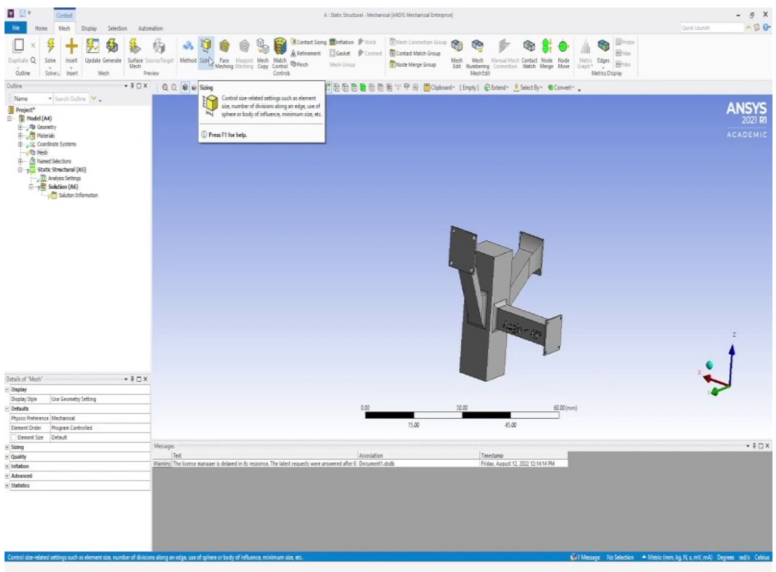
(Refer Slide Time: 15:06)

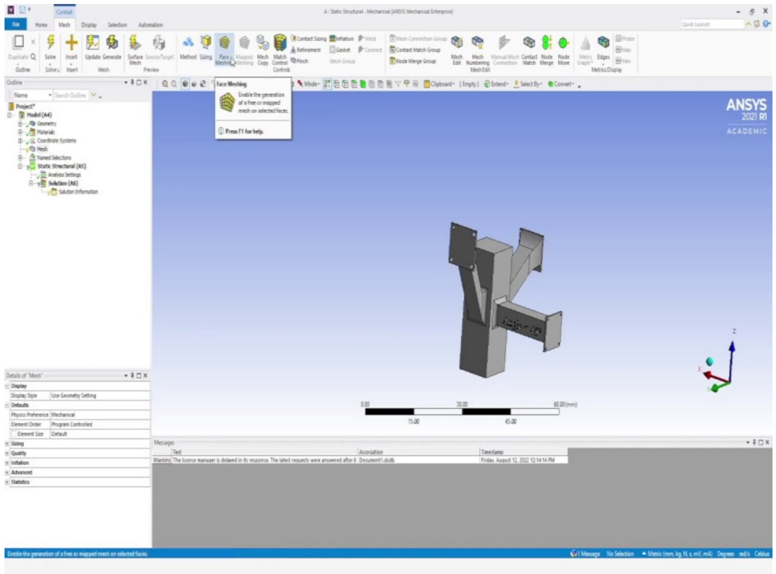


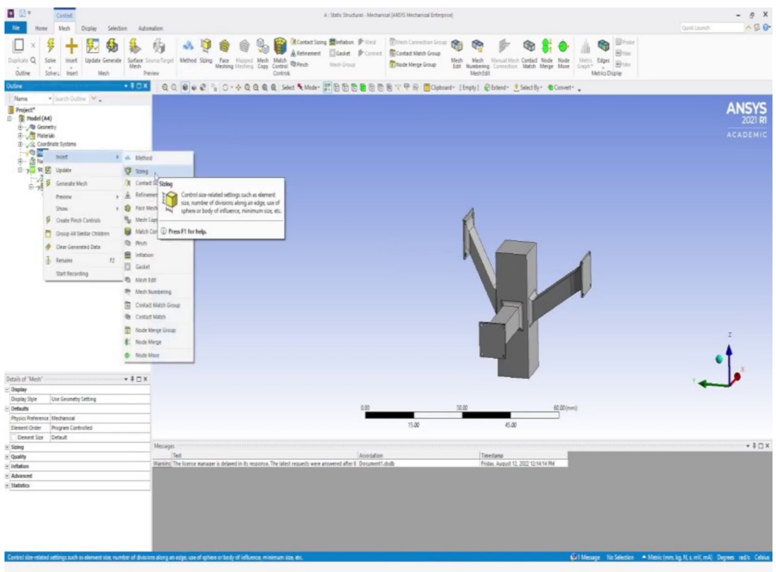


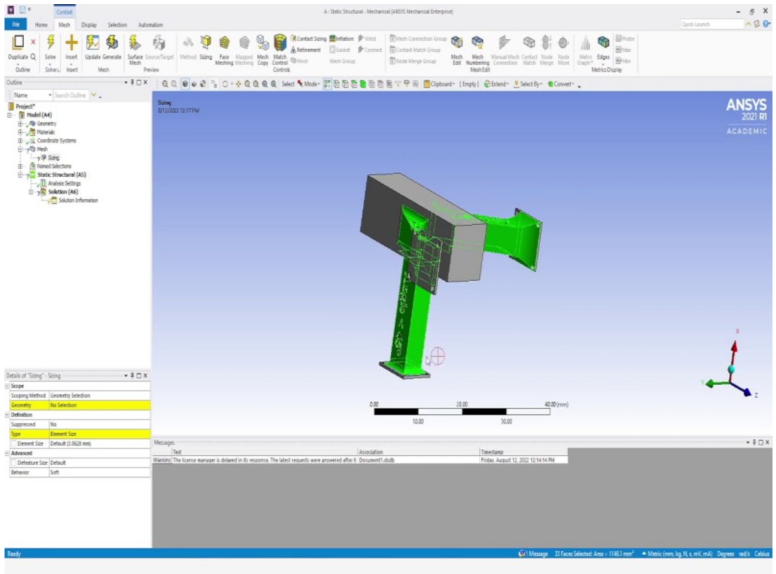


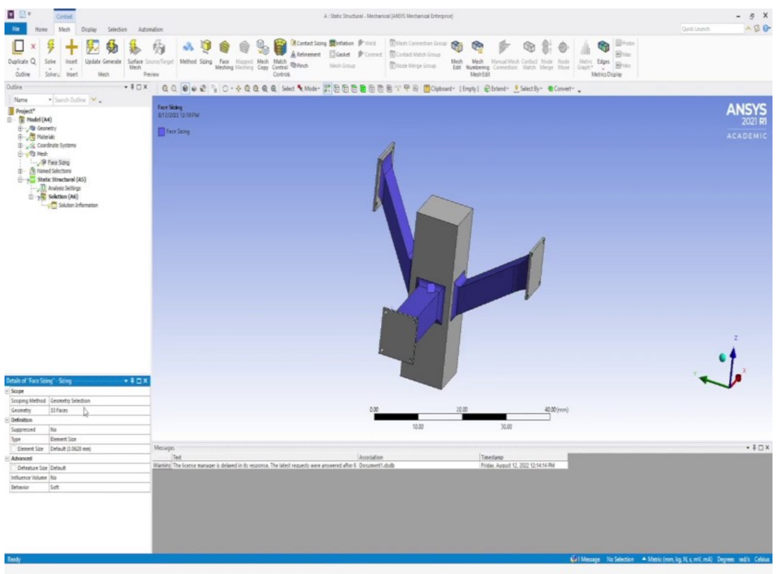


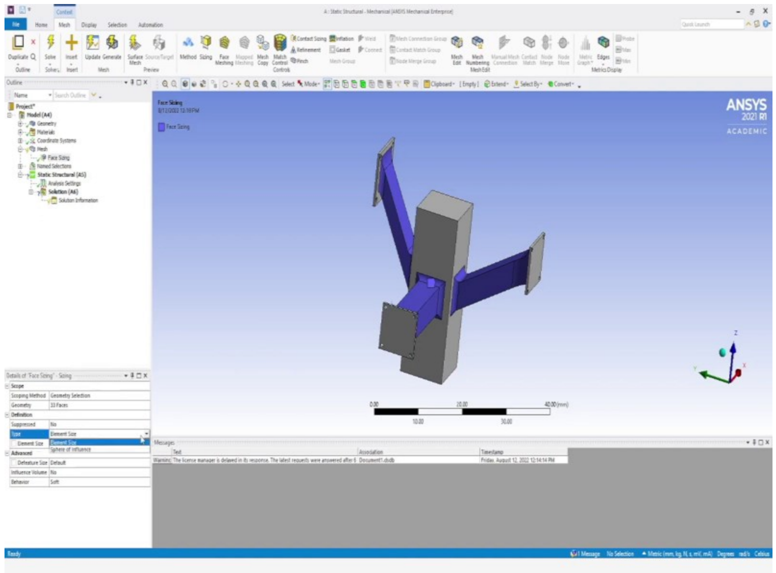


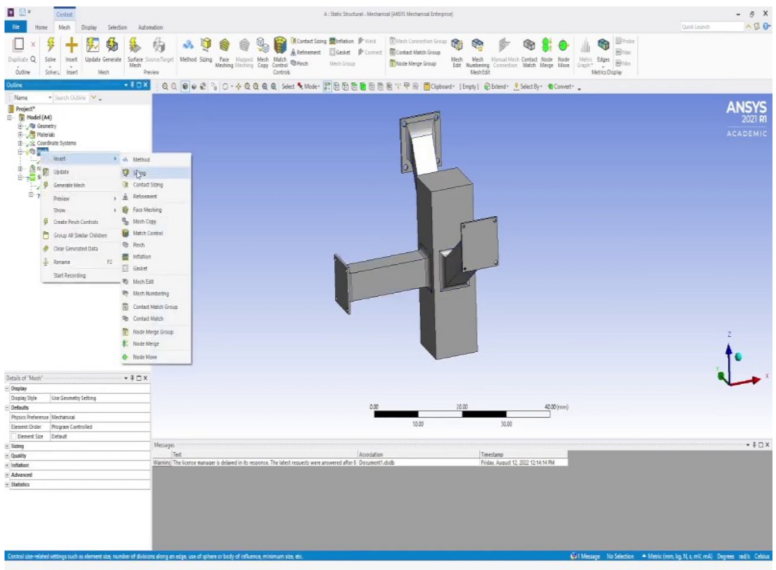

ANSYS
2021 R1
ACADEMIC

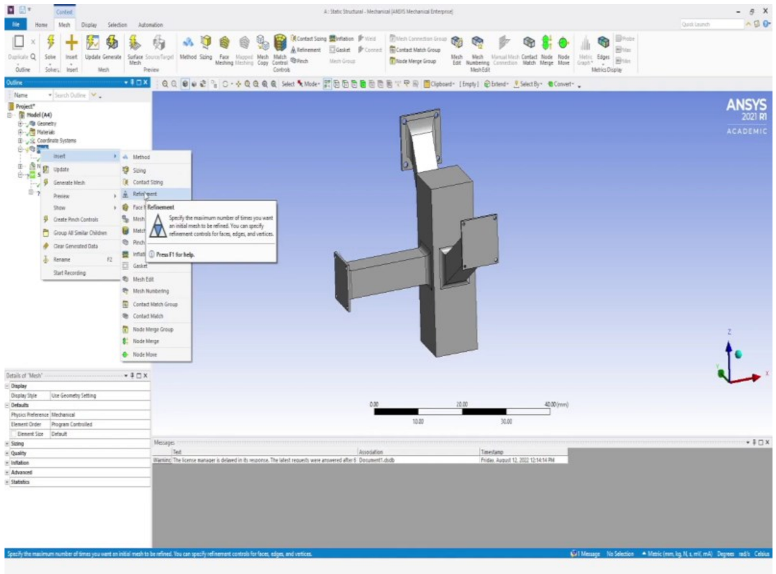

ANSYS
2021 R1
ACADEMIC

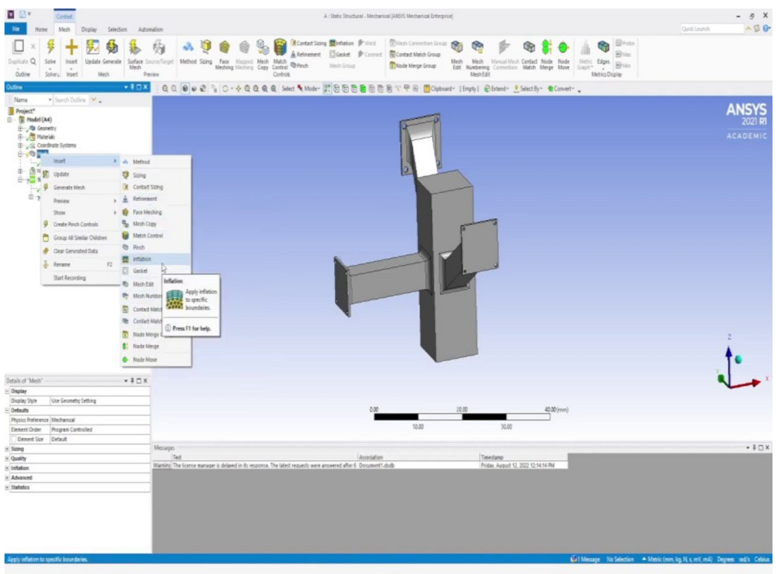


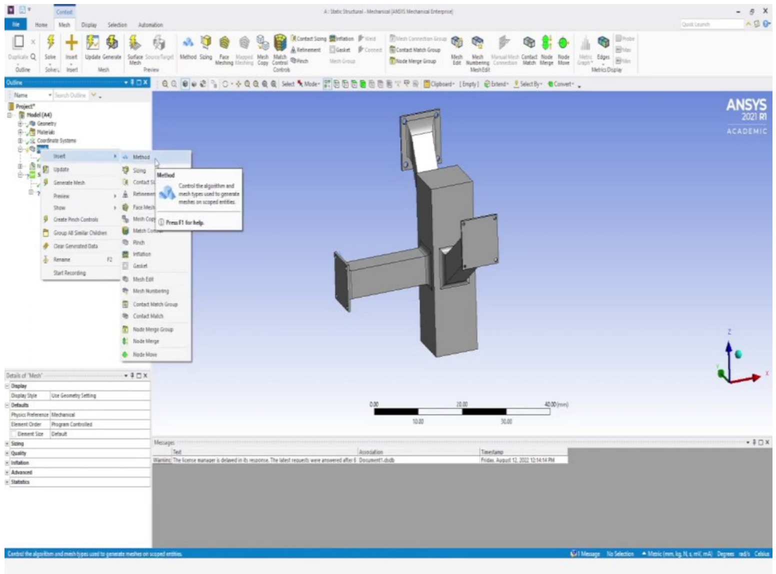


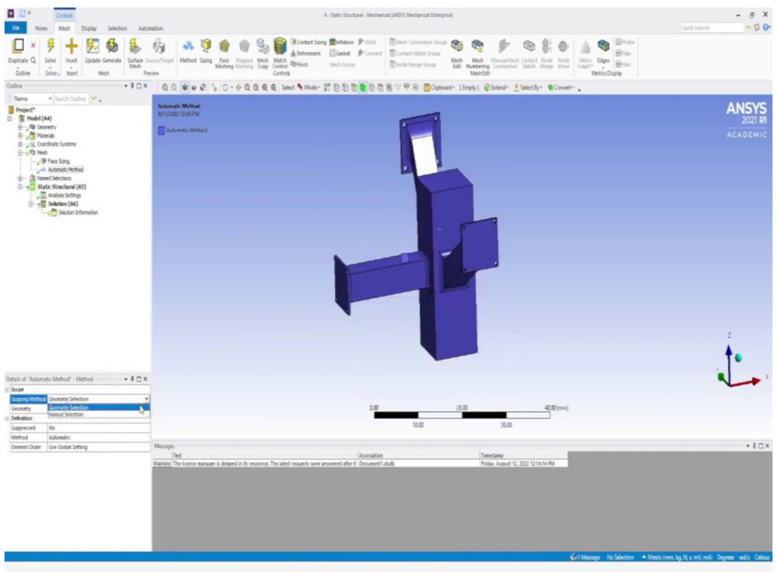

ANSYS
2021 R1
ACADEMIC

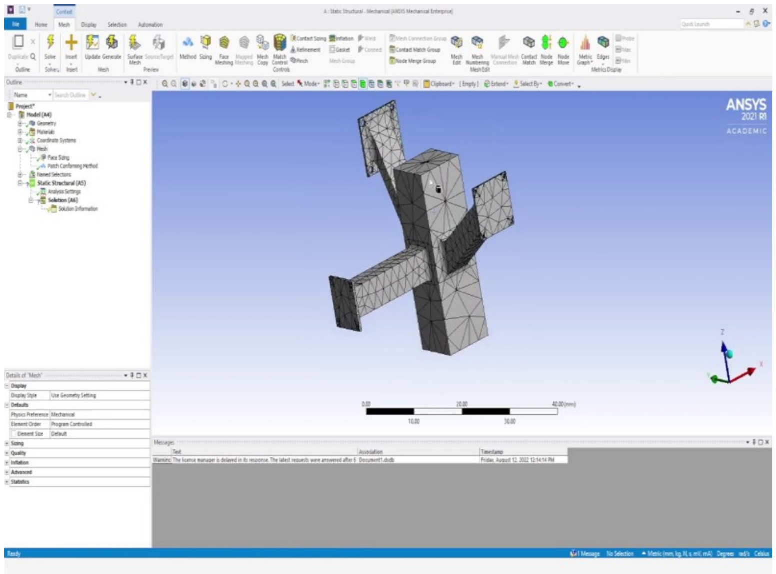

ANSYS
2021 R1
ACADEMIC

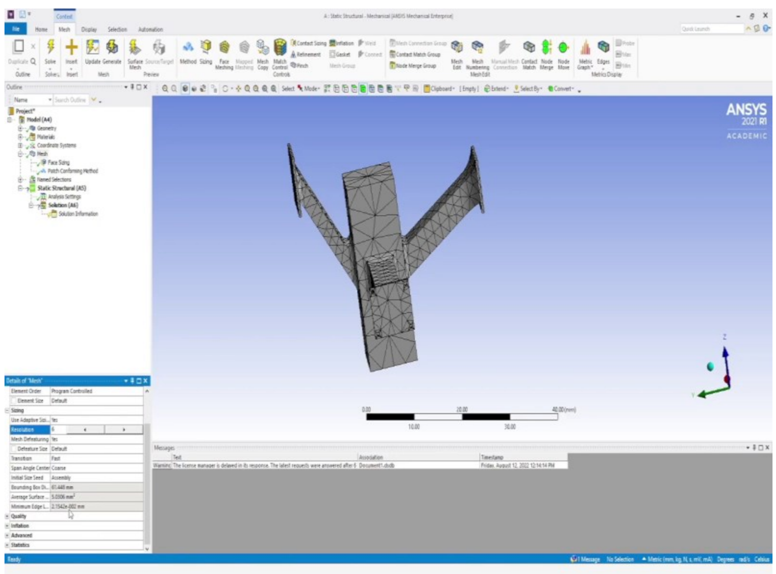

ANSYS
2021 R1
ACADEMIC
Refinement
Press F1 for help.
Specify the maximum number of times you want an initial mesh to be refined. You can specify refinement controls for faces, edges, and vertices.

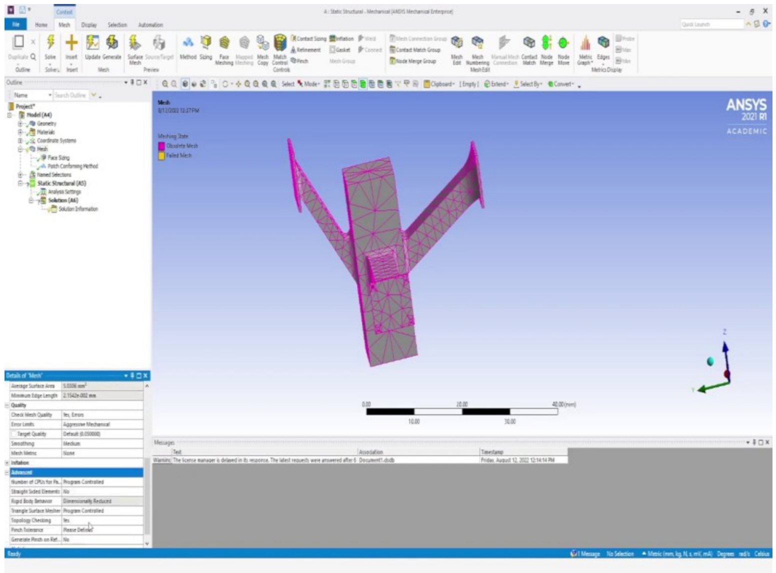

ANSYS
2021 R1
ACADEMIC
Inflation
Apply inflation to specific boundaries.
Press F1 for help.
Apply inflation to specific boundaries.

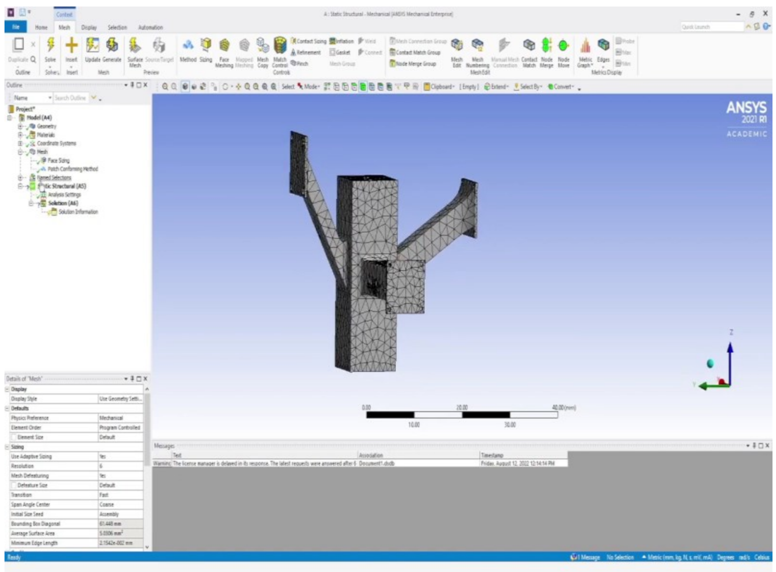

ANSYS
2021 R1
ACADEMIC

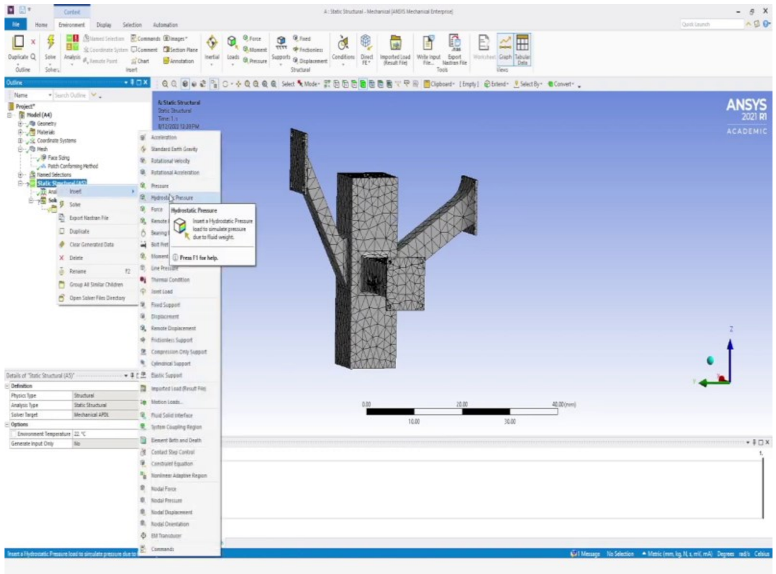

ANSYS
2021 R1
ACADEMIC

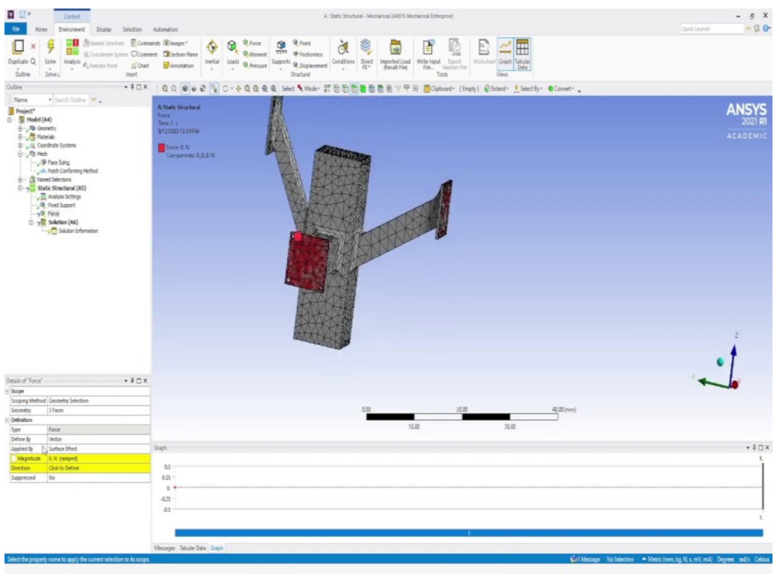


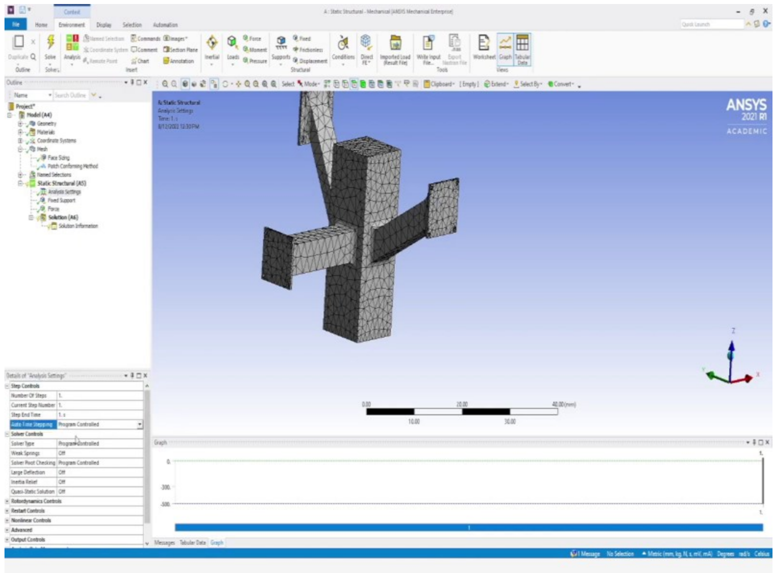


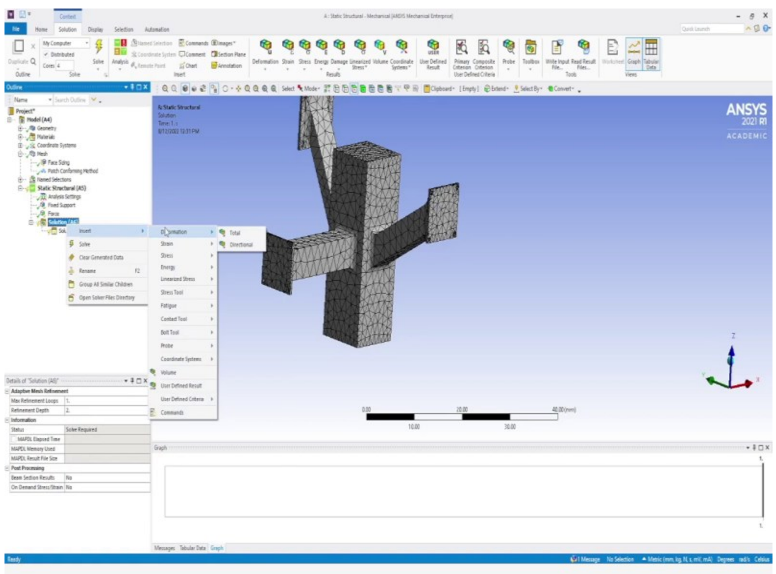

ANSYS
2021 R1
ACADEMIC

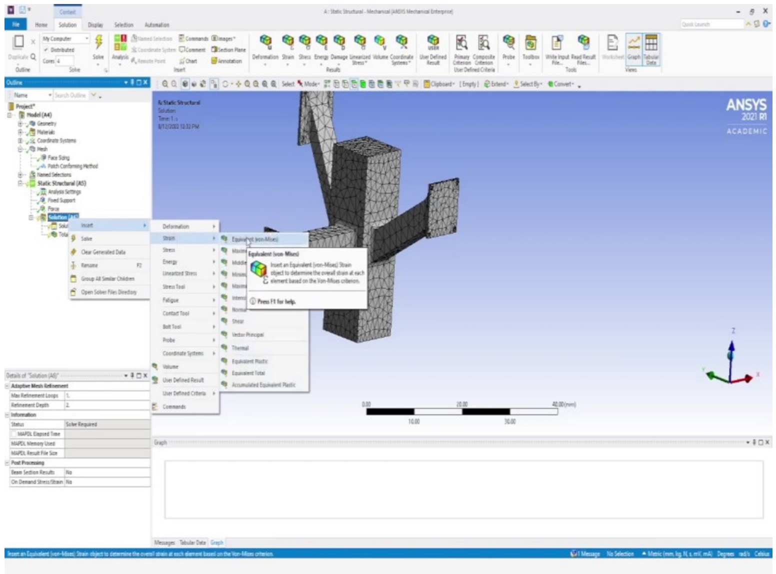

ANSYS
2021 R1
ACADEMIC

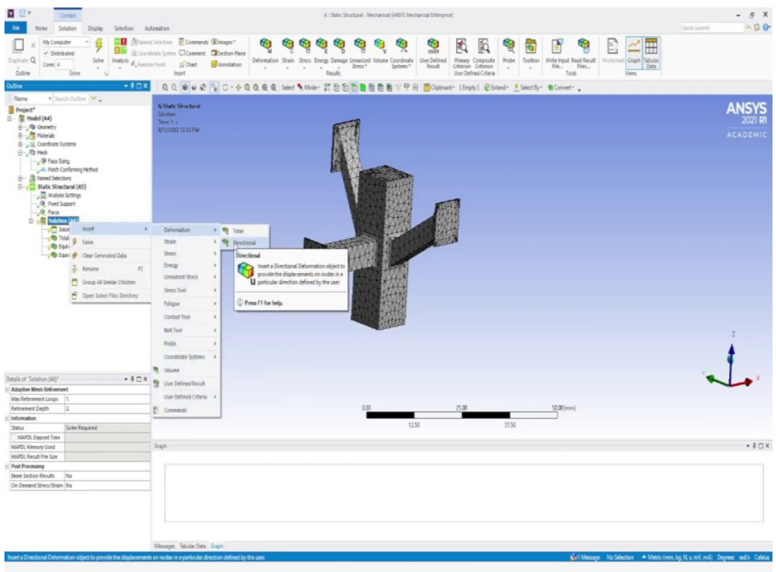

ANSYS
2021 R1
ACADEMIC

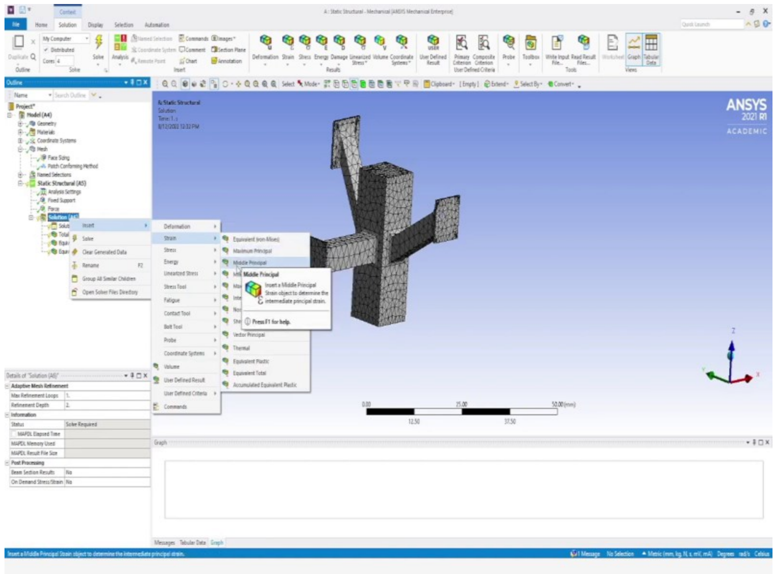

ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC



So, we can select from material assignment, 17 4 pH stainless steel. Based upon this material, we will try to see how the stress concentration flows. So, now we can come to mesh. I will now generate a mesh over it. I will just click mesh here. To have a structural analysis, this is one of the most important steps, to create mesh.

So, mesh type could be taken from method, from sizing, we can control the size related settings such as element size, number of divisions along an edge, use of different shapes such as sphere or body of influence. So, that means the sizing of the mesh must be in such a way that it cannot be so dense, it cannot be so less. So, a very dense mesh would take much time, very less would not give us the correct results.

Then face meshing is there, that enables the generation of the meshing at the face itself. So, we will just provide the method that is slicing. And select the geometry here. So, the main criteria that is being taken here is that the joints of these arms have to be strong. So, from here and the joining surface of the arm, this body. Not only surface, the complete solid body, if we select all surfaces, it will show the complete body.

So, angle less than 45, that is 0 degree. Then, angle more than 45 and equal to 45, all the joints there, and the surfaces. So, this is the intersectable point, this is being selected. Why not selective flanges as well. So, this whole extruded portion would be not tested because if maximum, extra material is there or the flange as well, that would also be removed.

So, we cannot define the element size here, in which, we can give the size the element. Here, it is given as 2 mm. So, now, in the meshing, we can provide the sizing, contact sizing. So, contact sizing means we can create elements, just like the faces or face to face or face to edge kind of contact region. So, like cylinder-piston assembly or cam-follower assembly, the contact sizing is there.

So, we can further refine or specify the maximum number of times we want the initial mesh to be redefined. So, different options could be taken. To keep it quite simple and understand the meshing process, I will just provide a method and apply. I will select the whole geometry, and selection control method.

We can say what kind of elements do I need. So, it will be tetrahedral, hexagonal, what kind of systems, what kind of the element order do we need to show. So, we will set it tetrahedrons here. And I will start the meshing and then generate a mesh. So, you can see the meshing here. So, other than the center part is less dense, and the arms are little denser.

So, we can also select the resolution. Resolution can be increased or decreased to provide specific mesh density at specific points that we need to be more calculative. So, after the meshing is completed, we can see the number of nodes and number of elements which are generated. So, it is showing that around 20, 25,000 nodes and around 15,000 elements are generated in this mesh.

So, we can also update after changing the resolution. So, because the resolution is now increased, mesh would become denser. So, you can see the meshes become much denser than we had the previous. So, now, in static structure, we will try to insert the criteria. We can show, express acceleration, gravity rotation, hydraulic pressure, the thermal analysis, friction support, then movement line pressure, what kind of criteria are we going to pick here.

So, one of the criteria that could be selected to see whether, how the material would behave and how the stress concentration would flow if we tried to provide a load over it. Initially, we will see the fixed supports. So, if I fix this support, this part is not fixed. For example, let me say this part is fixed way body or the ground. So, we are going to select the geometries now, from where we wish to put a force.

Then we are going to apply, we are going to put forces in these faces, on these surfaces of the flanges. We are going to apply the forces on the surfaces of this flanges. And we can select the coordinate system in a global coordinate system. We have different, you can see, x-component, y-component, z-component. So, since we must provide it, the force in the z-direction, an active force has to be applied.

The direction of the load must come downwards. So, I will put minus here, -500 Newton of the load is applied here. So, whenever these loads are selected, that is, again the basic engineering, we always see the factor of safety, the use of the component. So, that is all the skill of the operator again, as we discussed in reverse engineering as well. So, the physics or the engineering fundamentals must be very clear.

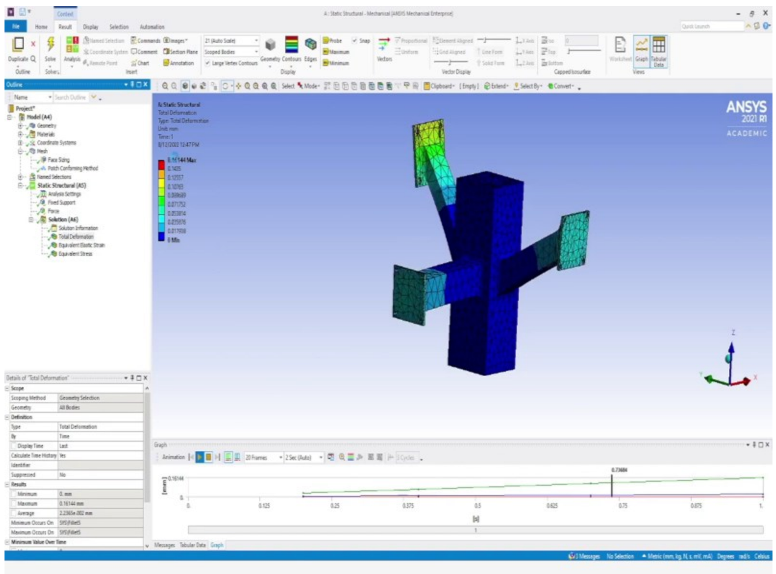
Now, you can see here in the mesh, the features, number of steps, one these as end time and start time for this. So, program controller, the solver type is again program control, direct or iterative. So, we can select iterative or program controls all over time. So, large deflection, industrial relief, these are all kept off. So, for the non-linear controls as well, we have set the program control. That is, whatever data we put, just take it accordingly. It will not assume any things here.

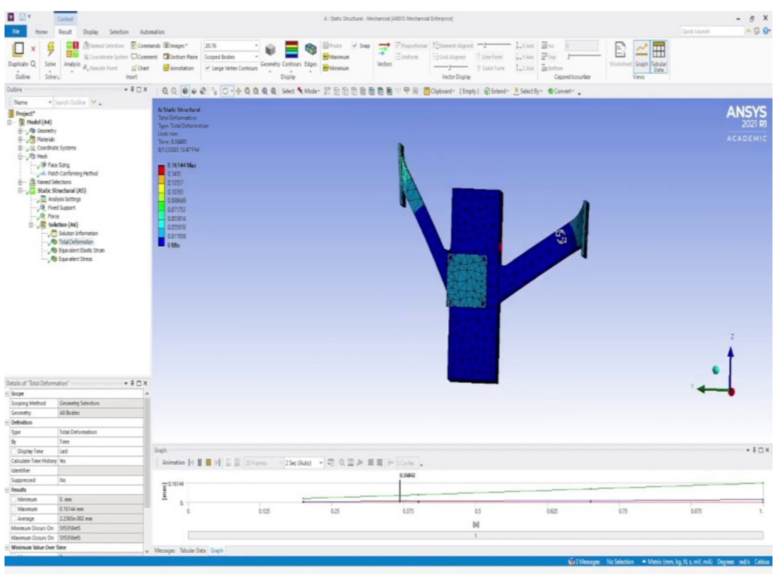
So, it is taking the values taken in the program. So, then we select solution, what kind of solutions do we need to check here. It could be deformation, stress, then total deformation energy, strain. So, we can say total deformation, we can say equivalent stress, then equilateral elastic strain is taken. The total deformation is to be seen, and equivalent stress is selected. Deformation, strain, stress.

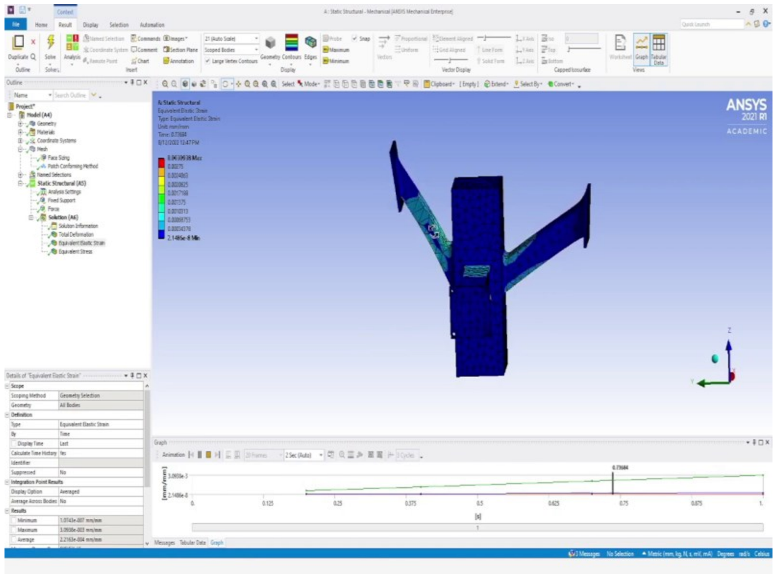
We could also have picked maximum principle, the middle principle, minimum principle, maximum shear, different kinds of strains could have been selected. Now, in the stress itself also, equivalent stress is selected, but we could again have selected the intensity, normal shear, the membrane stress. So, similarly, different other solution, responses that we need to test, that could have been selected here.

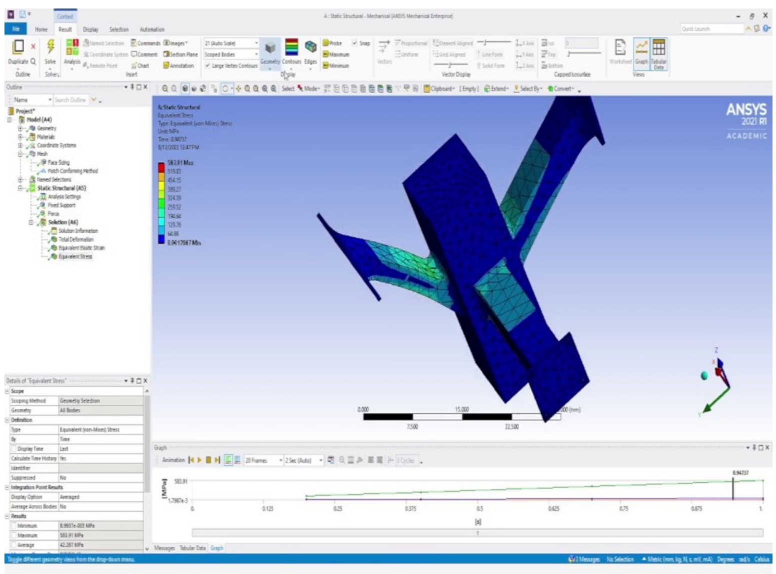
So, if we need the solution, now the solver required status is solver required. And we try to solve the problem here. So, it will take its time. So, this is, you can see in the left status bar here, it is showing that. So, now it is started. We can see here, the simulation is 10 percent run, it is holding the mathematical model here, 12 percent, 13 percent. So, it will take its time. Since it is now 15 percent and the simulation, this mesh solution that we have seen that, there is a body that is fixed and the load is applied from the top. And I will just fast forward it to see how the stress concentration flows. And I know that it flows like this. So, this is now fast forwarded.

(Refer Slide Time: 26:29)





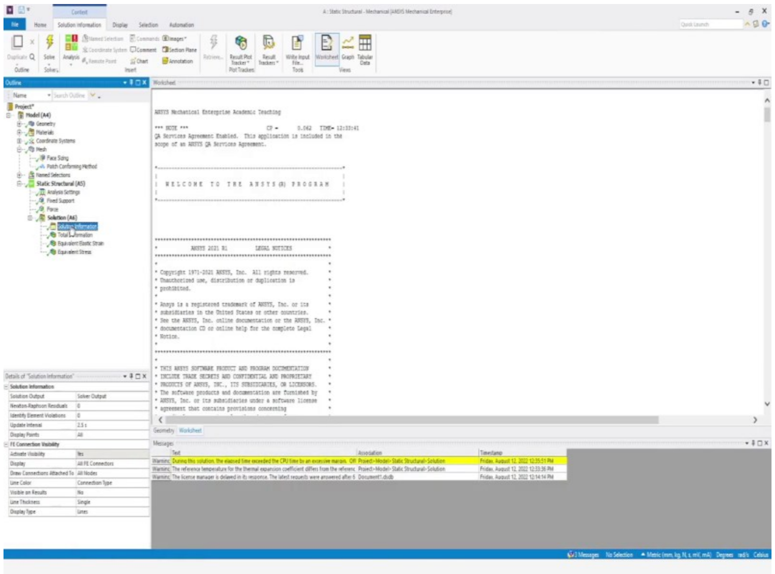


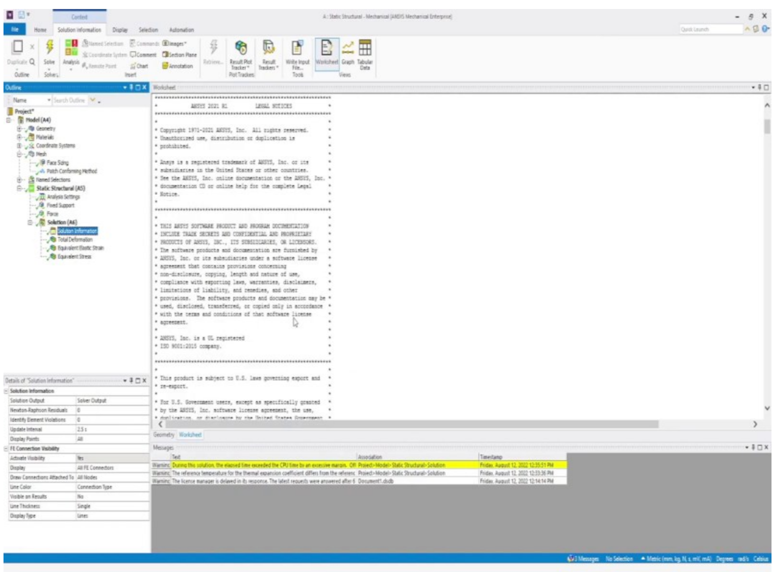


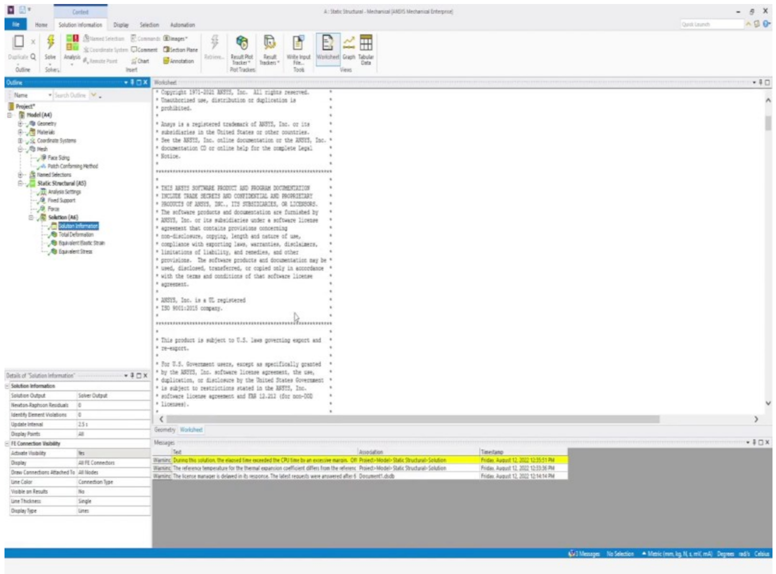
So, we have got the solution information. For the total information, you can see the animation of the total deformation here. It is showing how the load is applied on the body and how the stress concentration is flowing. It is a static structural flow. So, this is, and then total deformation was shown. Then we have the equivalent elastic strain that is being shown here.

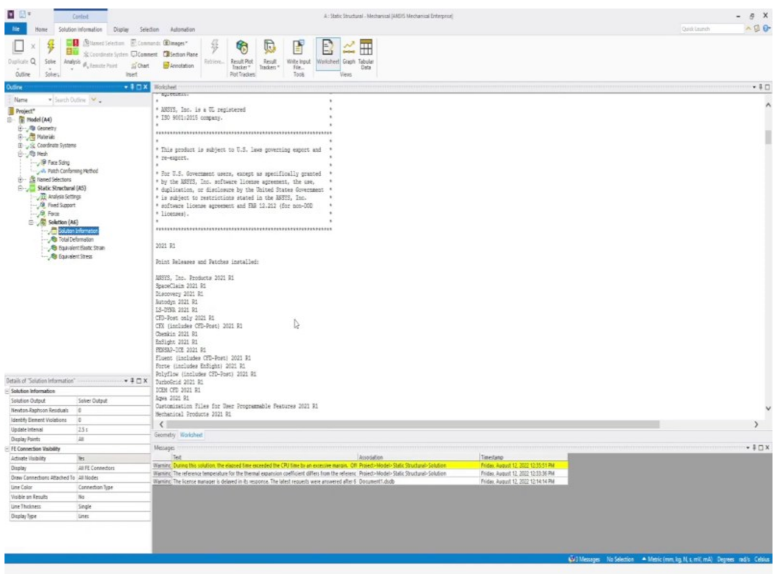
So, as you can see at the fixed part, the strains are maximum. So, these are all cantilevers, a fixed point of the cantilever, that is receiving the maximum stress. So, let me try to make it less dense, the contours could be made smooth here. The colors could be made smooth. So, it is showing that around 400 Mega Pascal of the stresses there.

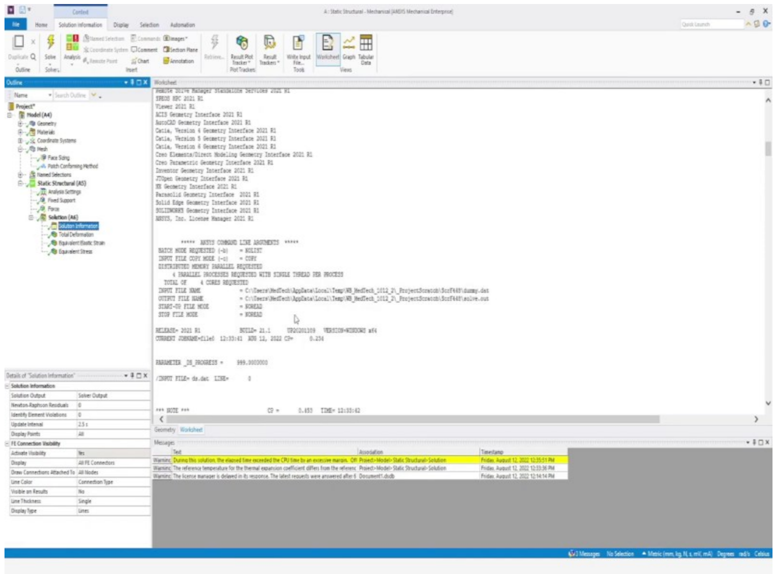
(Refer Slide Time: 27:43)

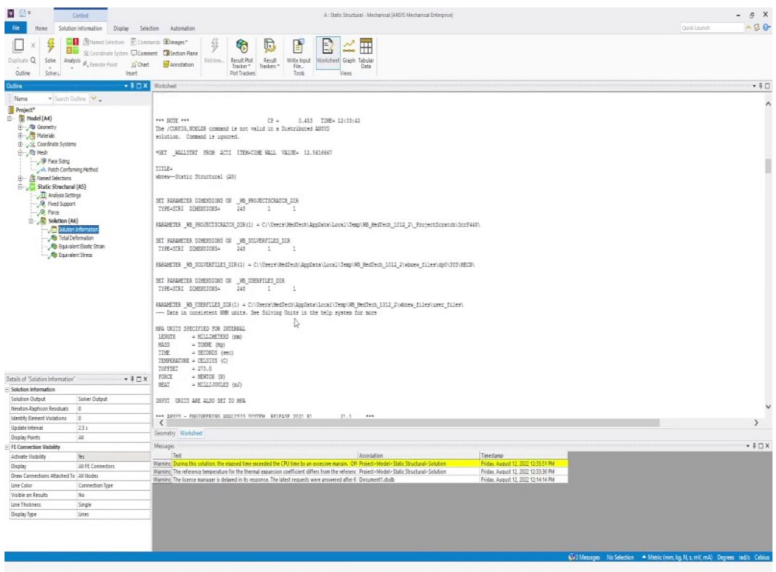


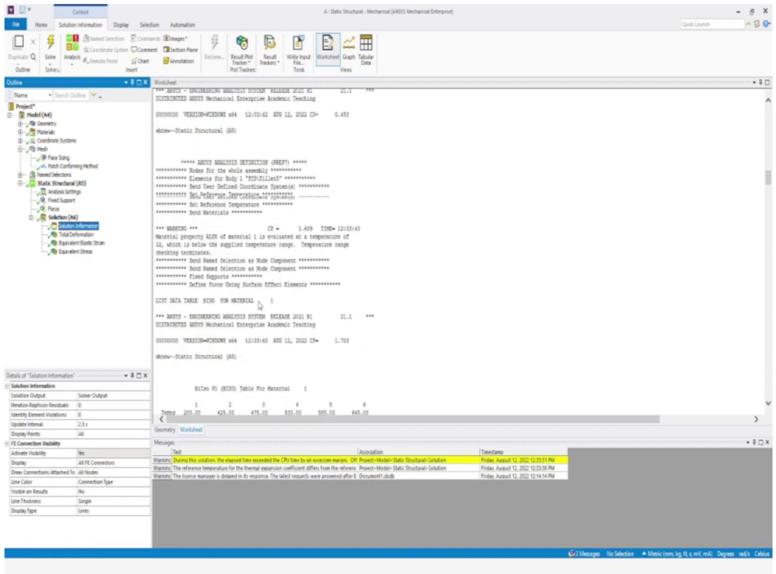


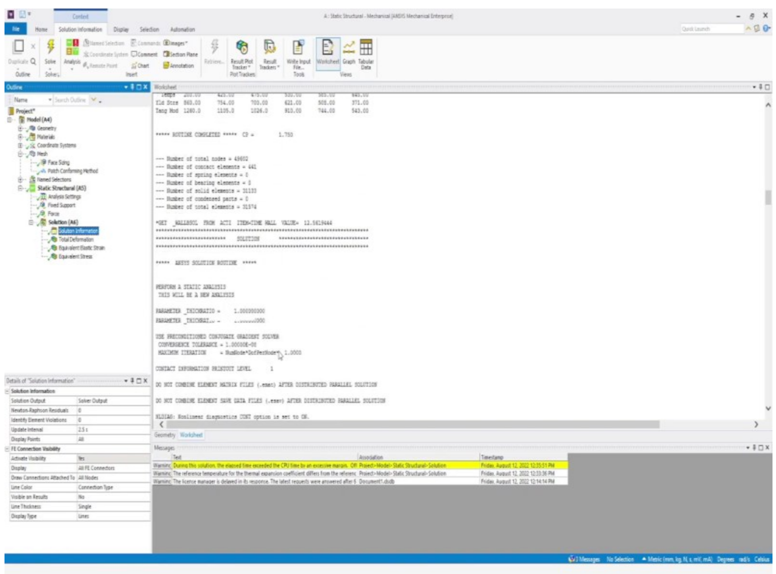


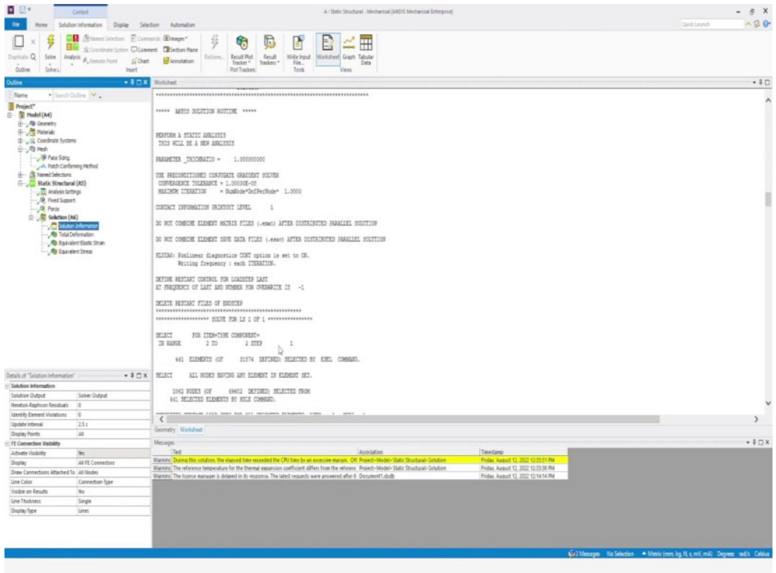


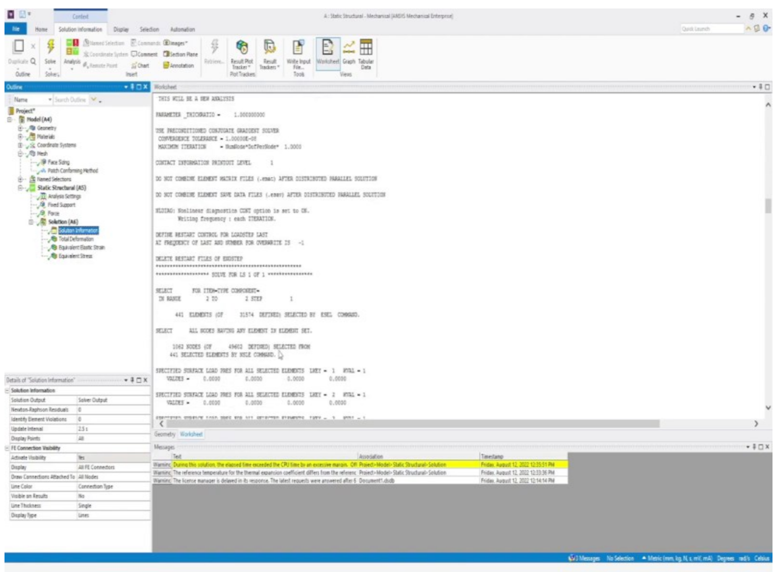


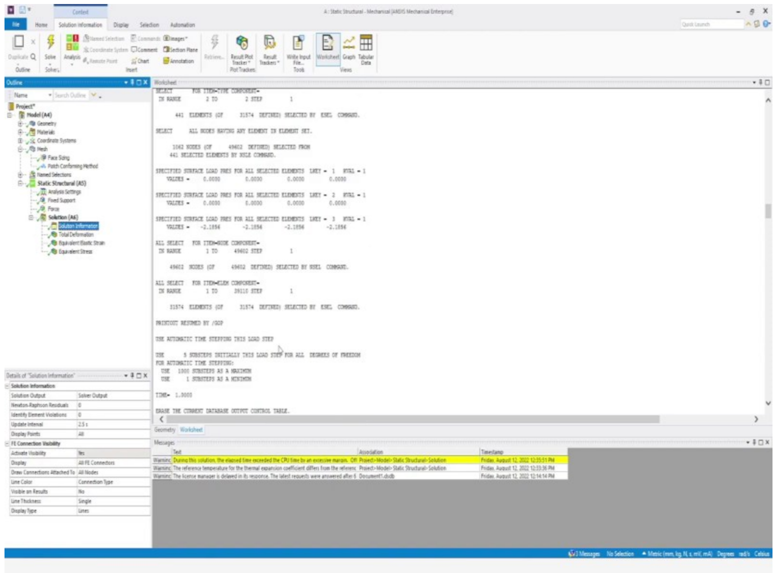


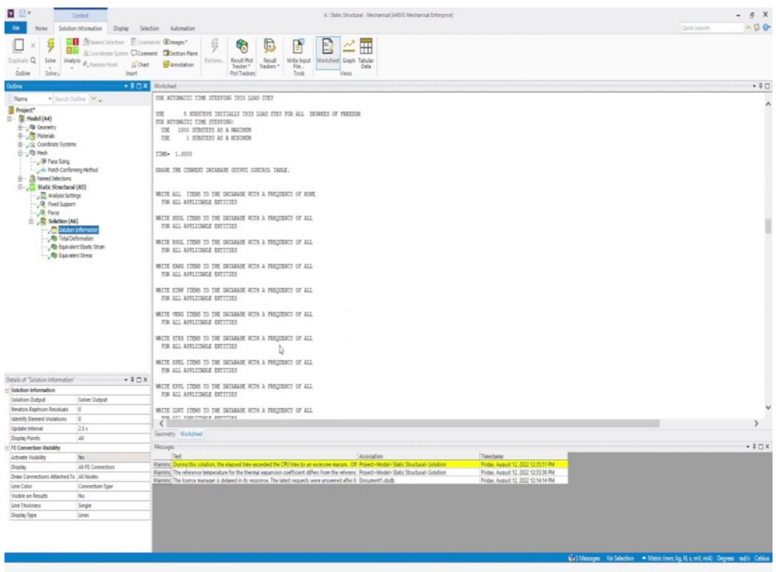


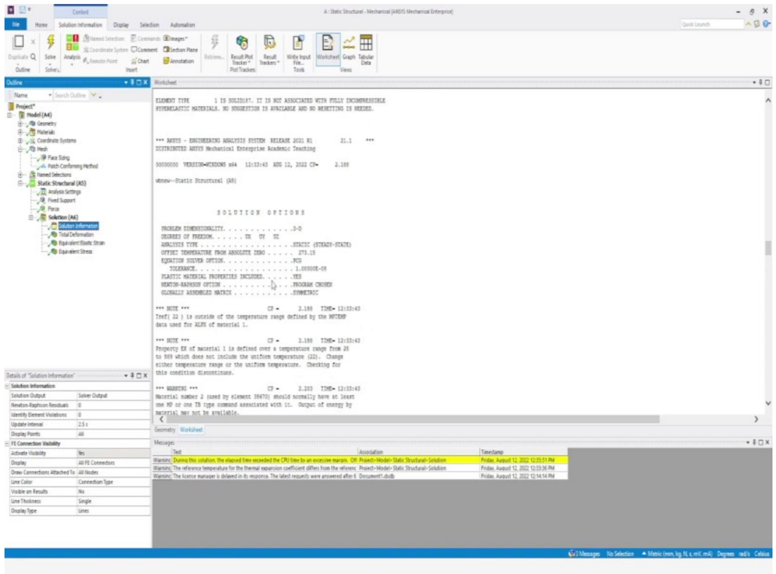


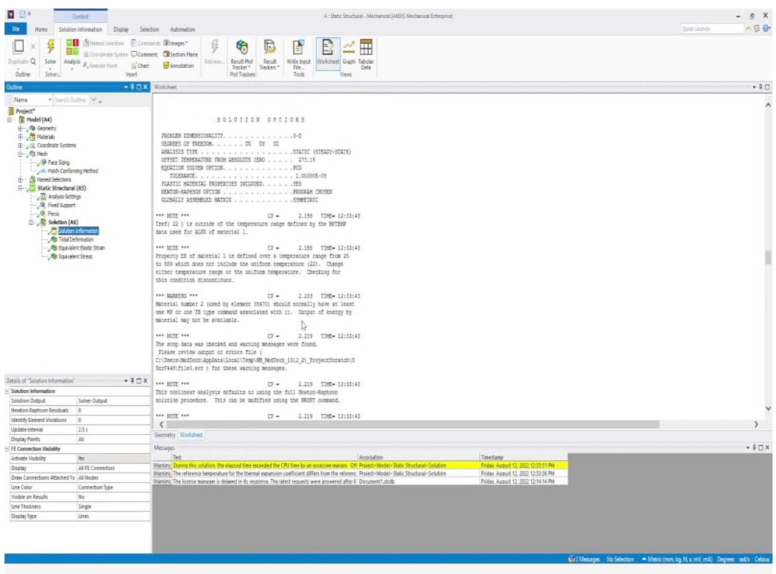


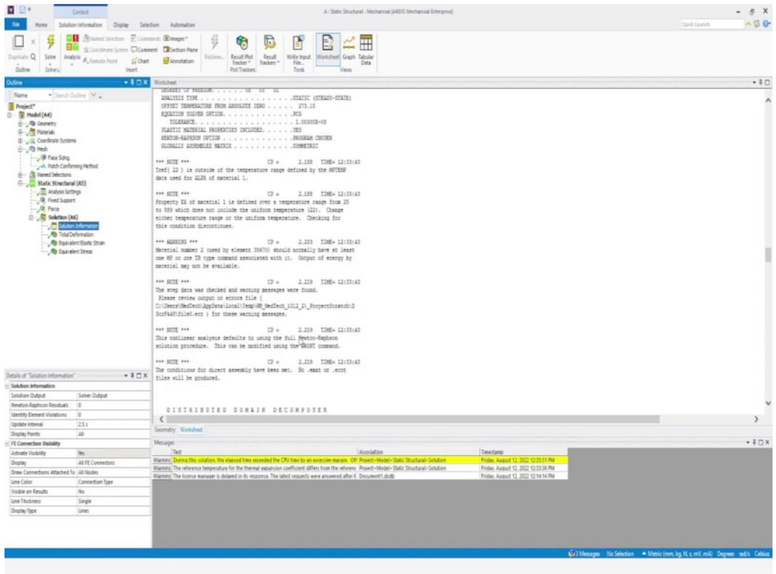


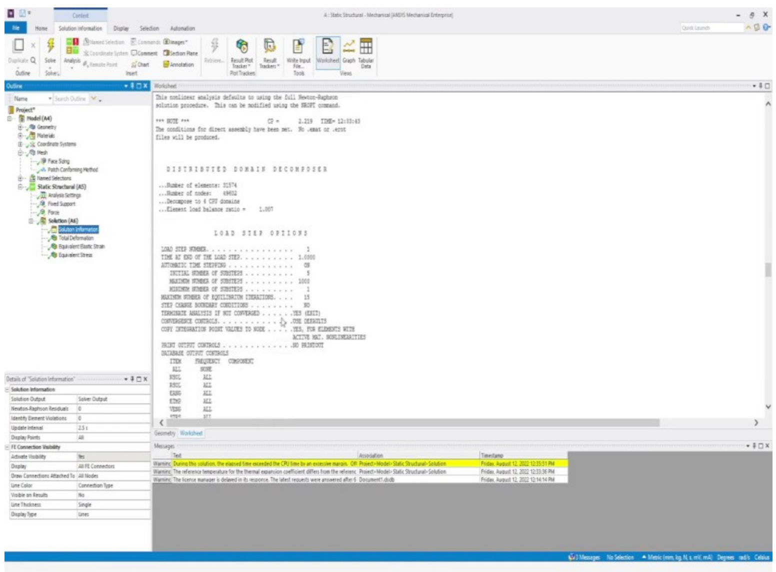


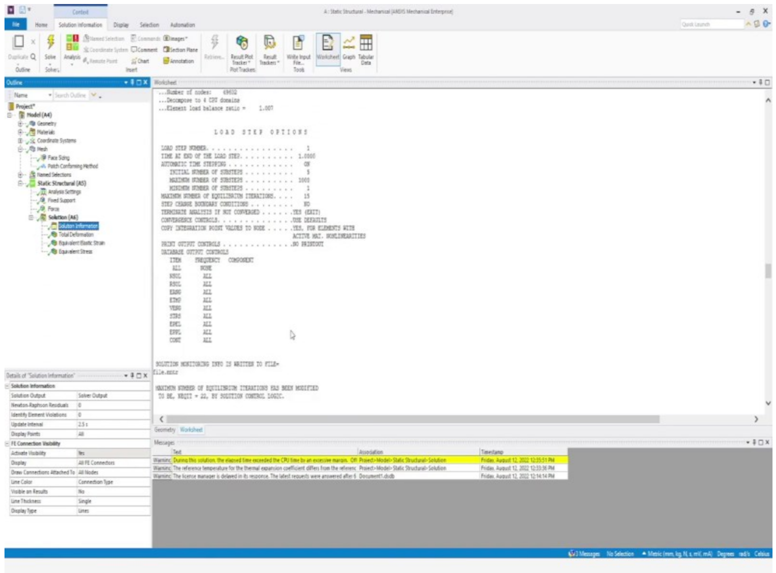


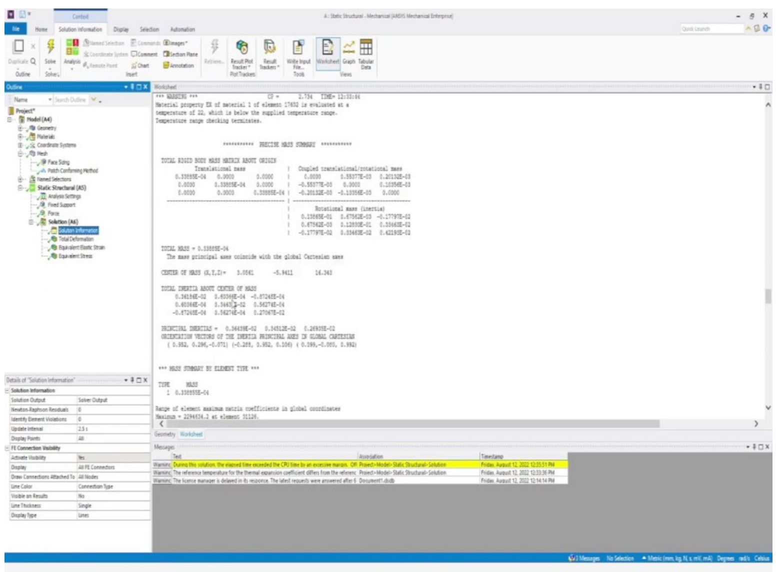


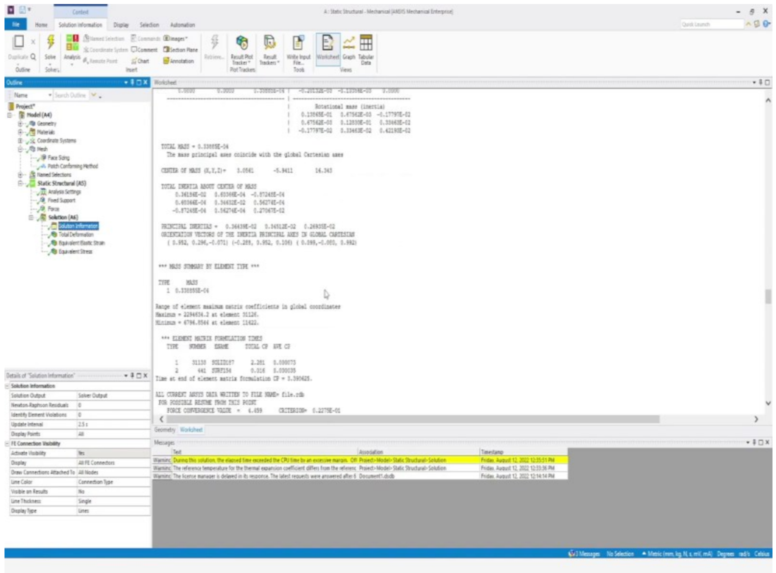


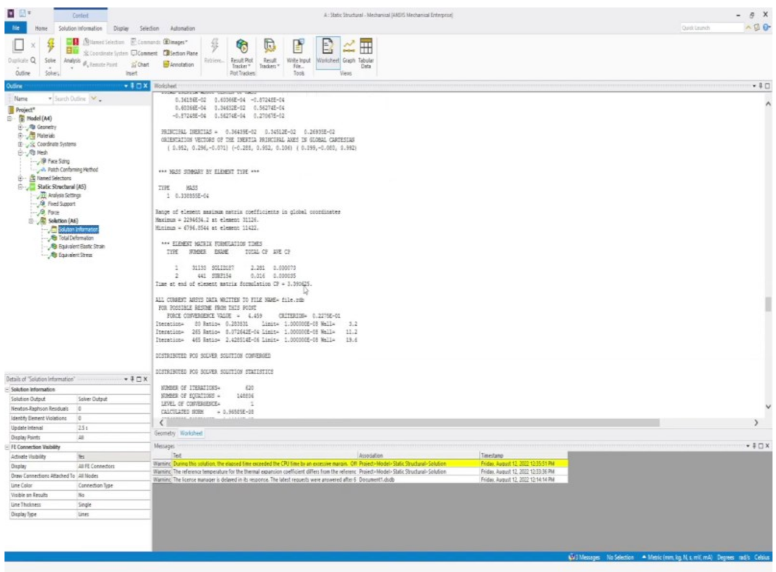

A : Static Structural - Mechanical [ANSYS Mechanical Enterprise]

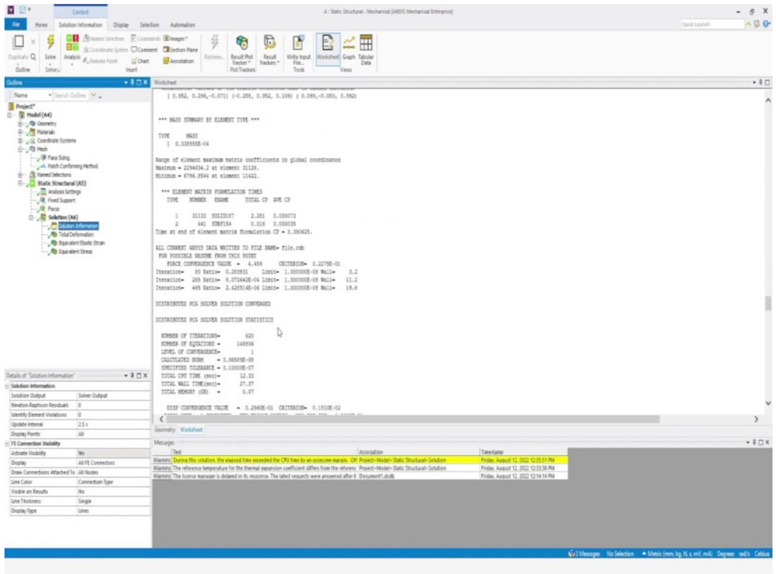

A : Static Structural - Mechanical [ANSYS Mechanical Enterprise]

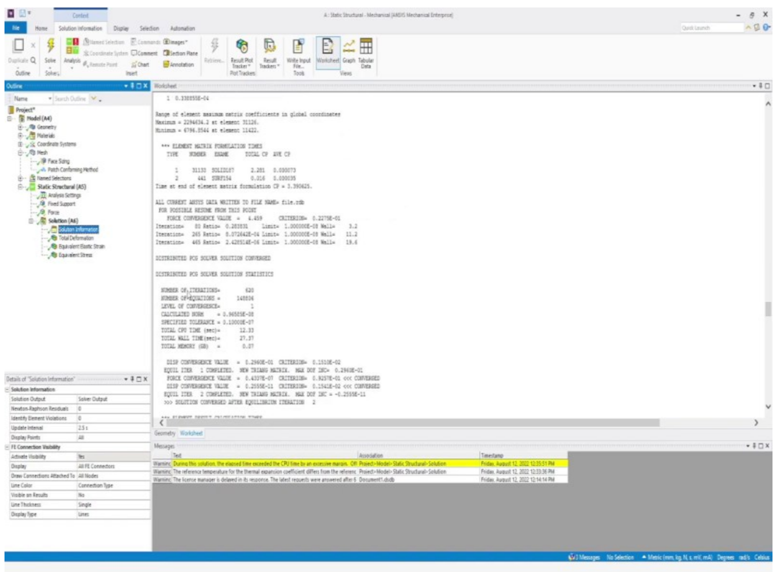


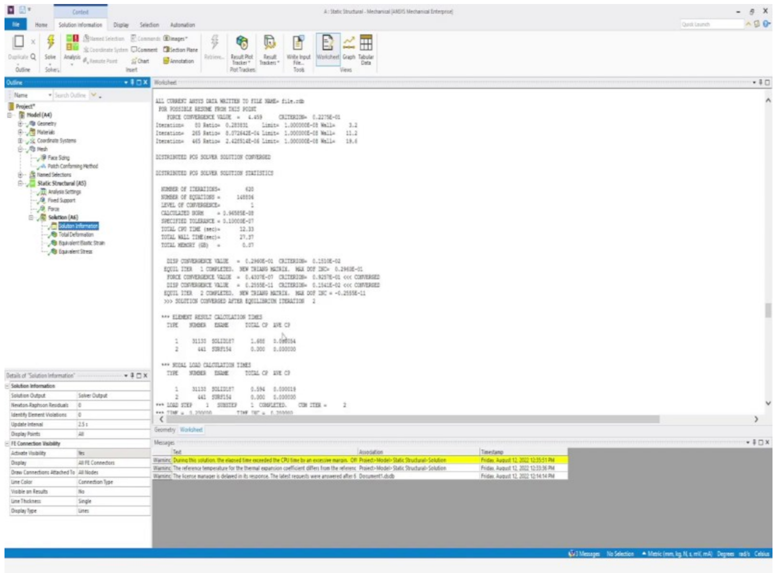


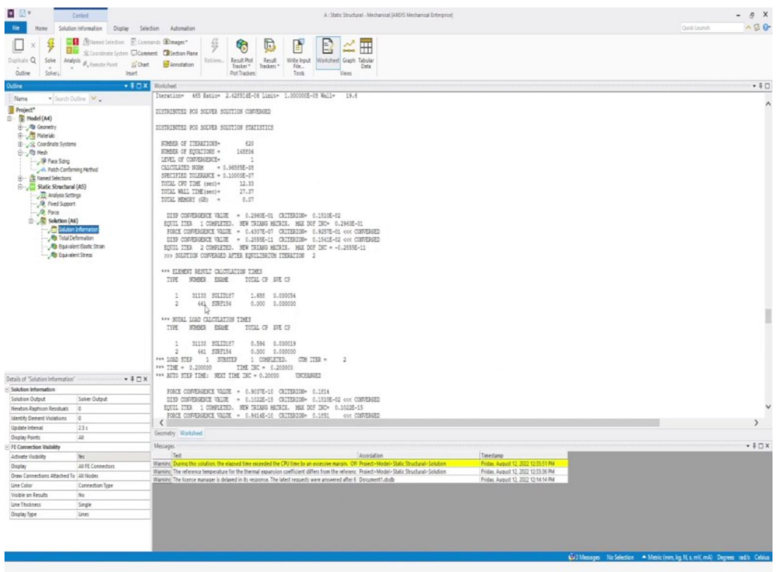


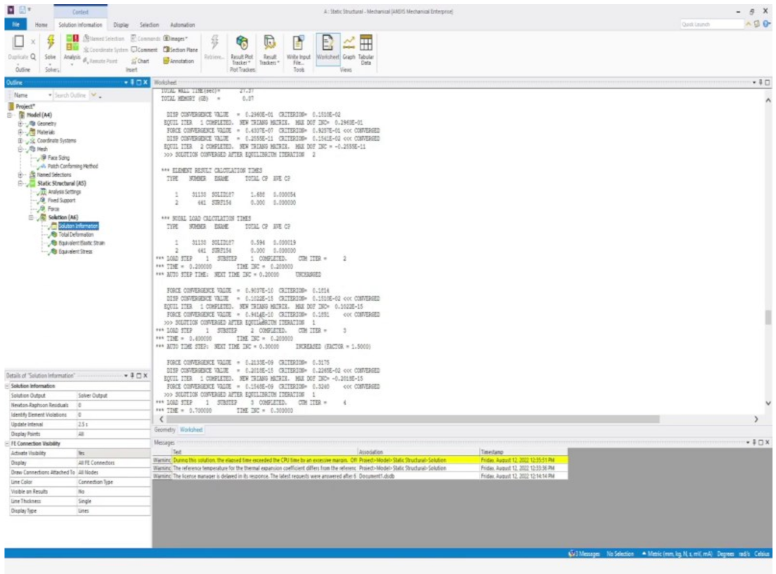


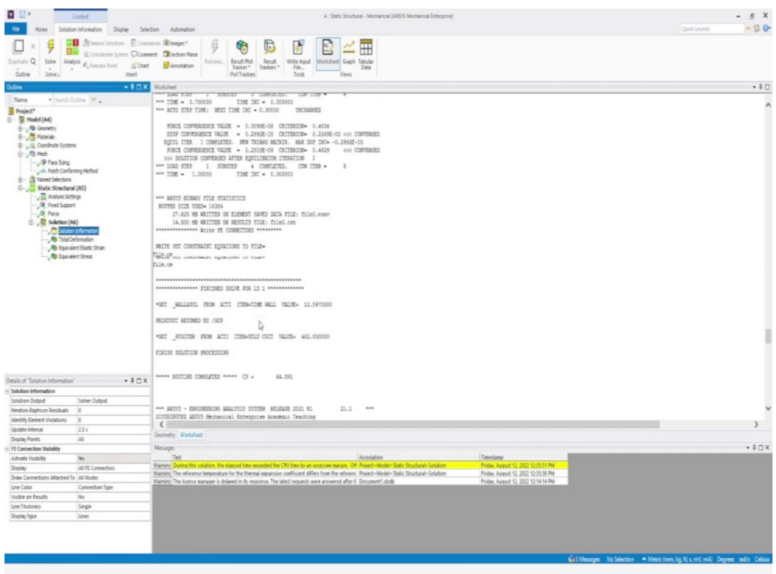


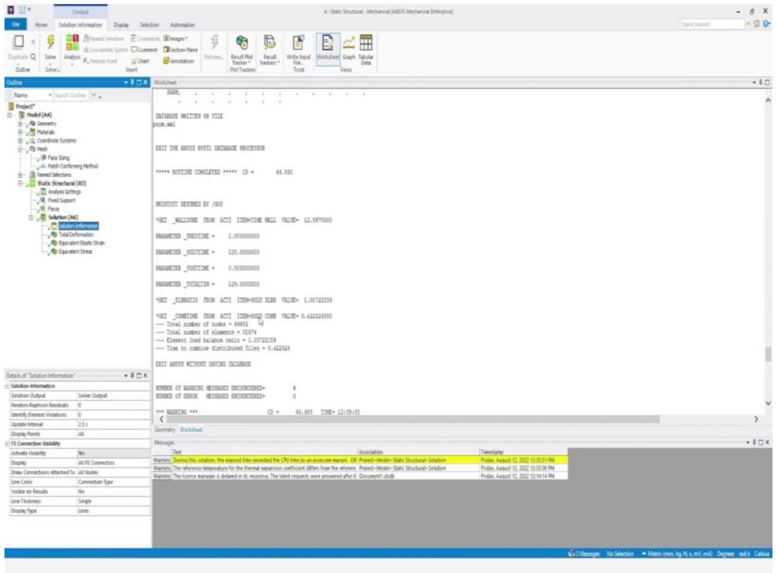


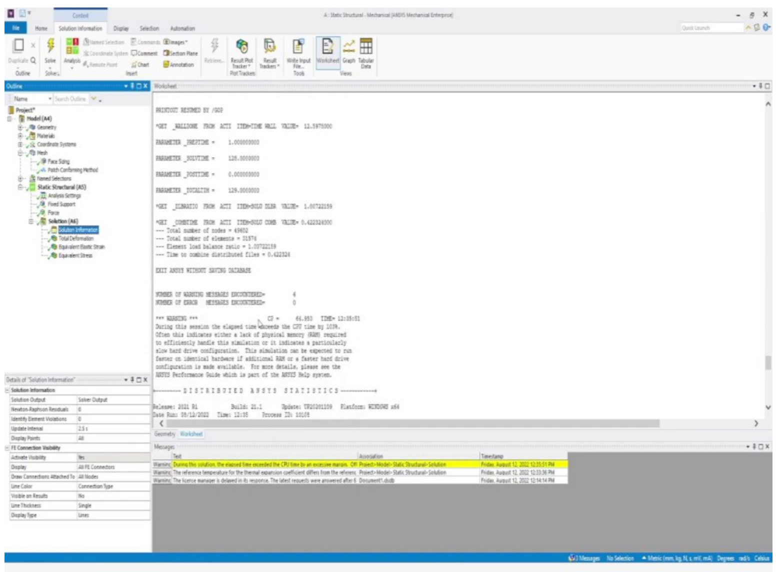


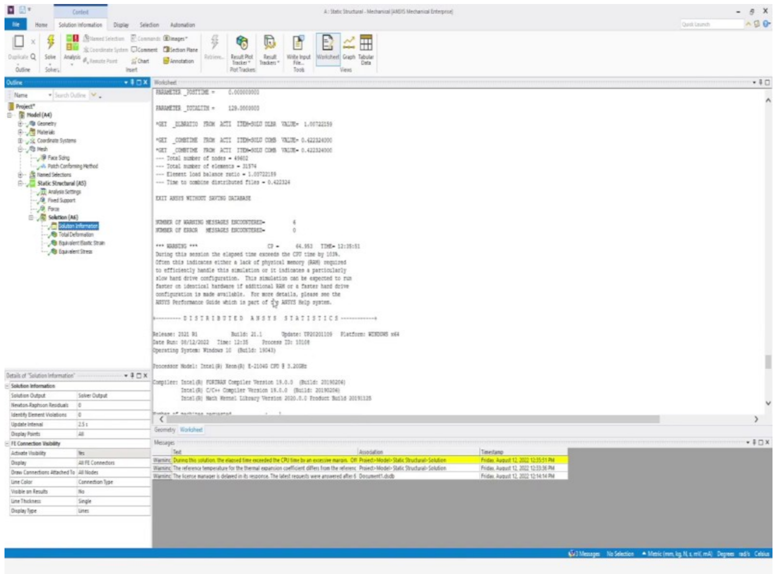


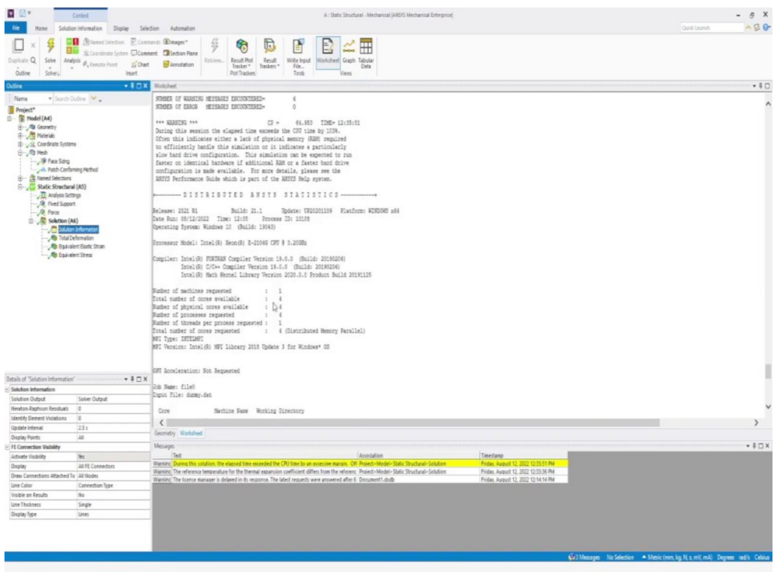


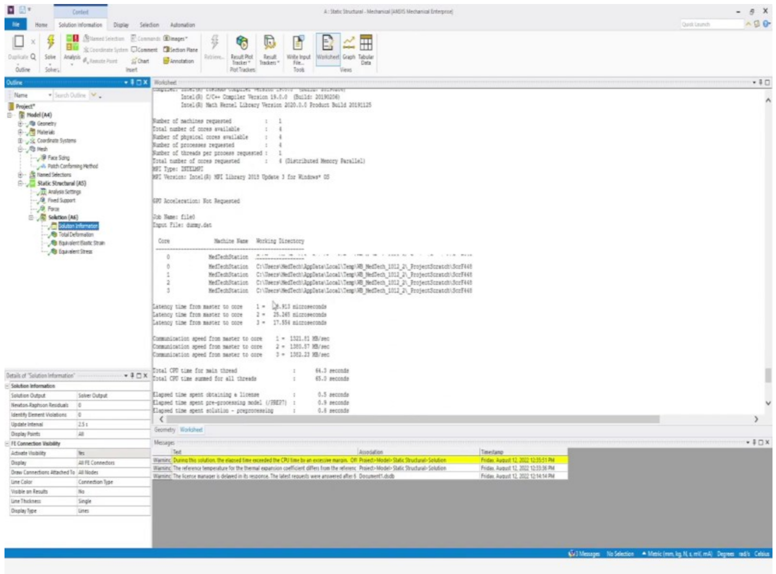


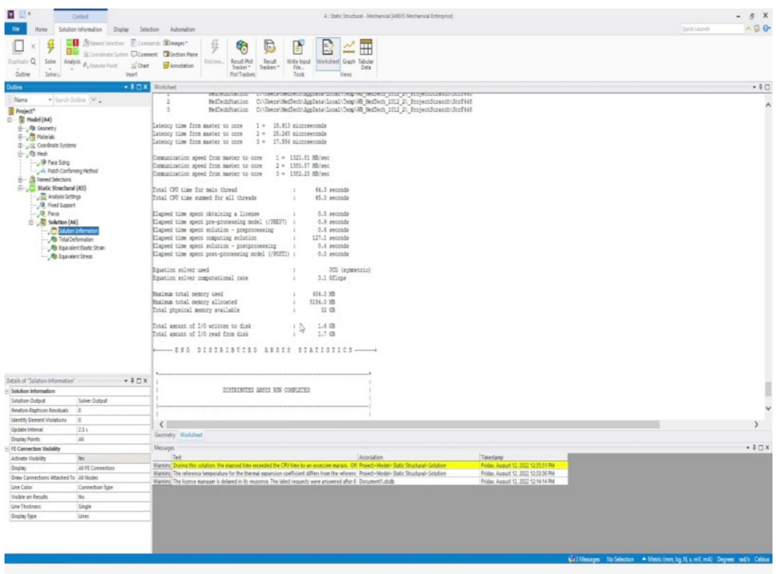










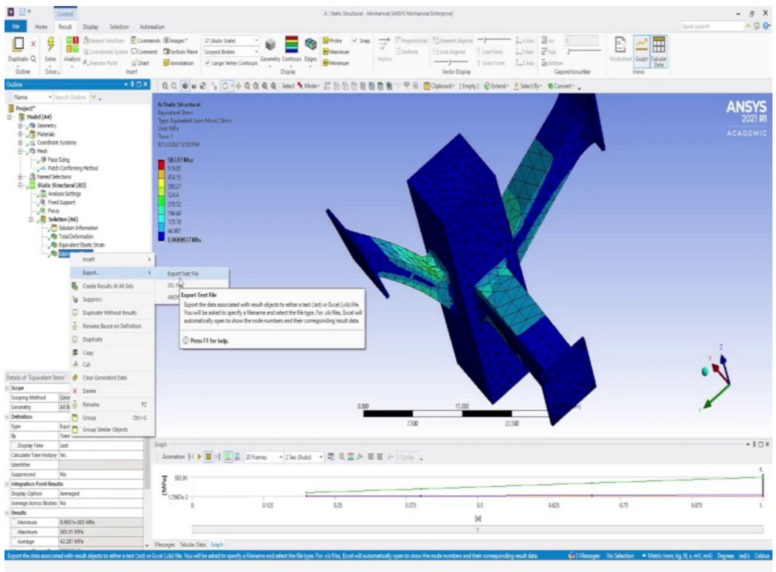


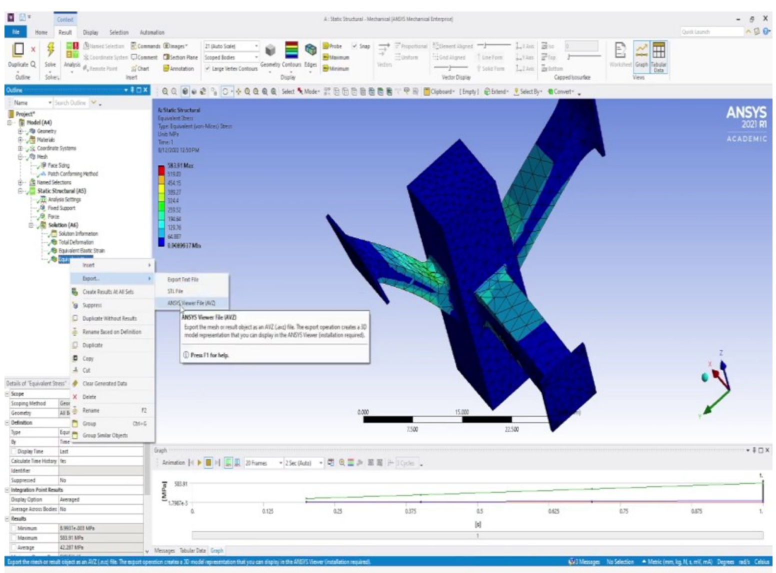
So, also, the report is generated in the text form, where we can see the outputs here. You can see this temperature values, the yield strength, the tangential modulus for a different node it is given, and 200, 425 these all values are given here. So, iterations on different nodes and elements is given here.

So, these are solution options, static steady state solution is given, 273, resistivity domain decomposer, the solution setting for the boundary condition is given. Also, we will get the metrics for precise mass summary. The total number of iterations, that is here, are 620, total number of equations which are taken are around 1.4 lakh. It is time, it is telling that total time it has taken is 27 seconds.

Then force convergence values are also given, displacements convergence values. So, this detailed or the, I would say, comprehensive data is not generally required. Majorly we require the value of the minimum and the maximum stress concentration, that is what we call the elastic strain value or equivalent stress, at or where does the component fail. So, this is the result of the simulation analysis that we can see here.

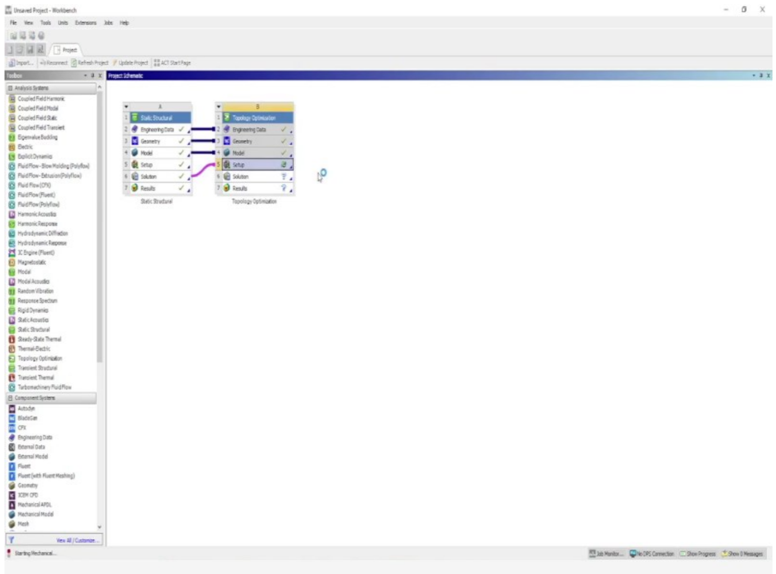
(Refer Slide Time: 29:58)





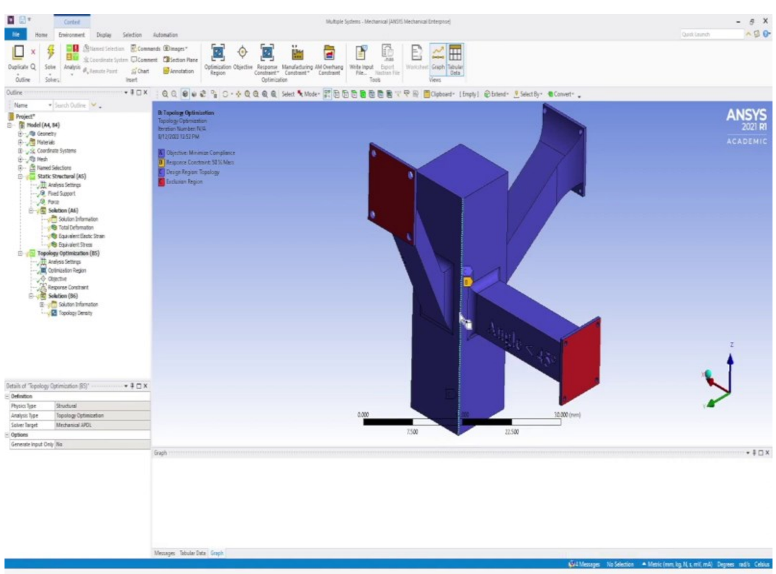
So, also, we can export this file in the STL format to get the 3D printing or for topology optimization.

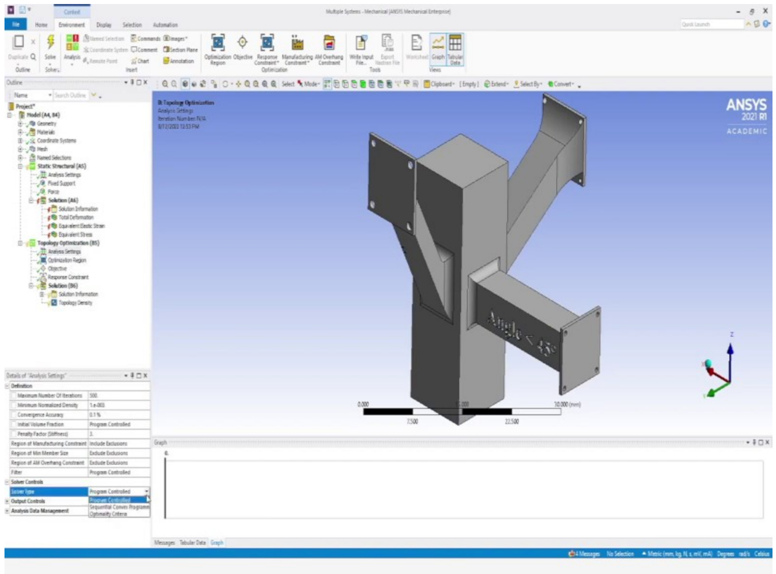
(Refer Slide Time: 30:15)

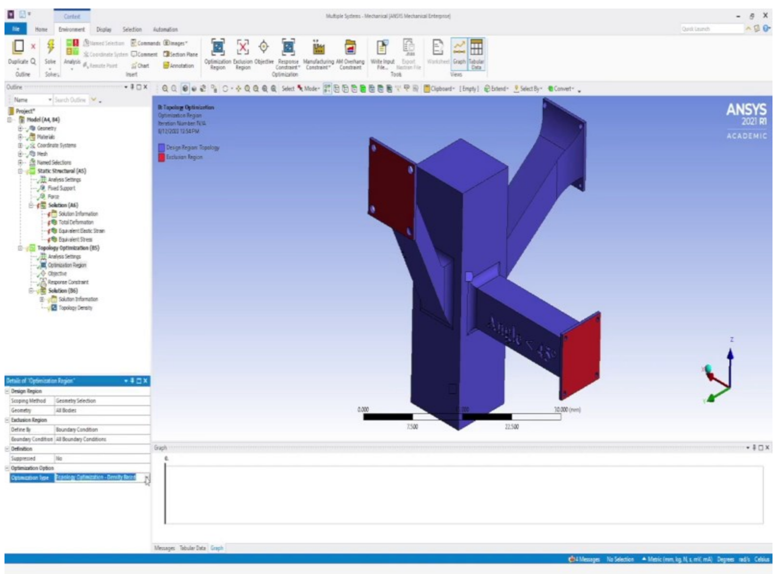


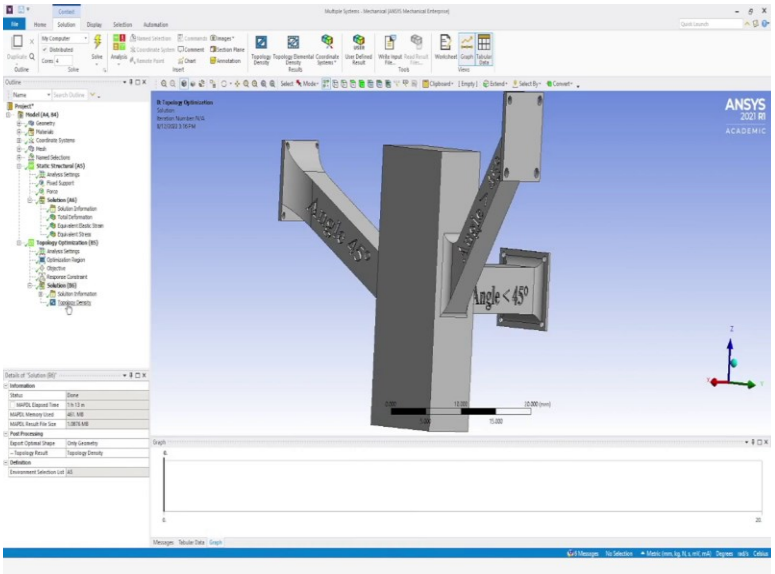
So, we can put the same product in for topology optimization. So, we will connect this solution, that is the static structure to the topology optimization. So, you have selected topology optimization from the left pane, and we will just connect the solution to this setup. You can see, these two packages are not connected. So, all engineering data, geometry, model, setup solution, results are now connected to the topology optimization. A small pop-up window here. So, let us see the setup.

(Refer Slide Time: 30:59)




ANSYS
2021 R1
ACADEMIC

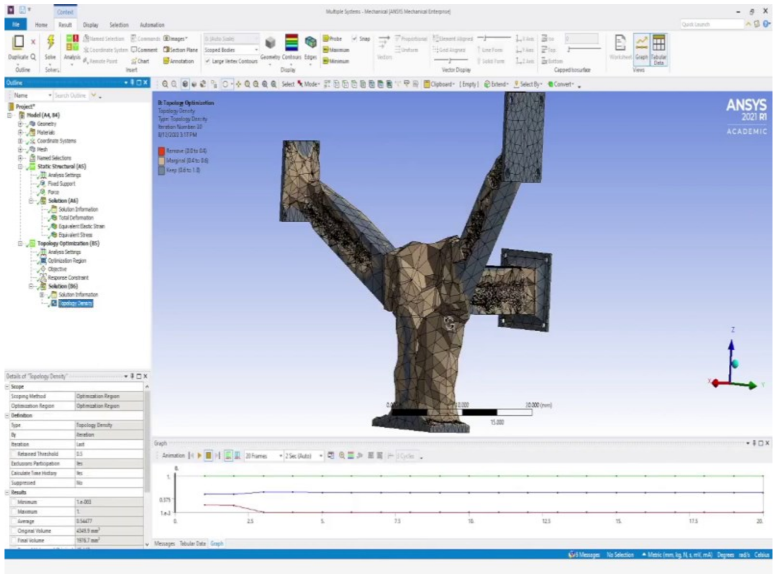

ANSYS
2021 R1
ACADEMIC

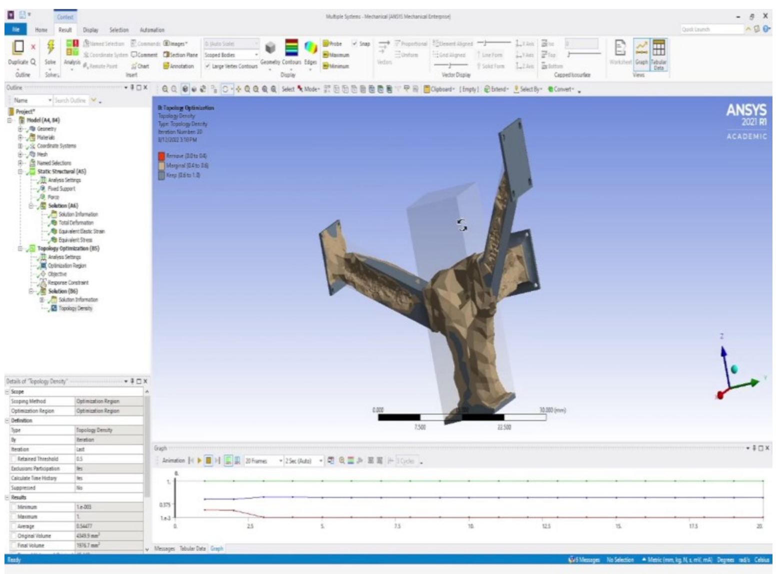


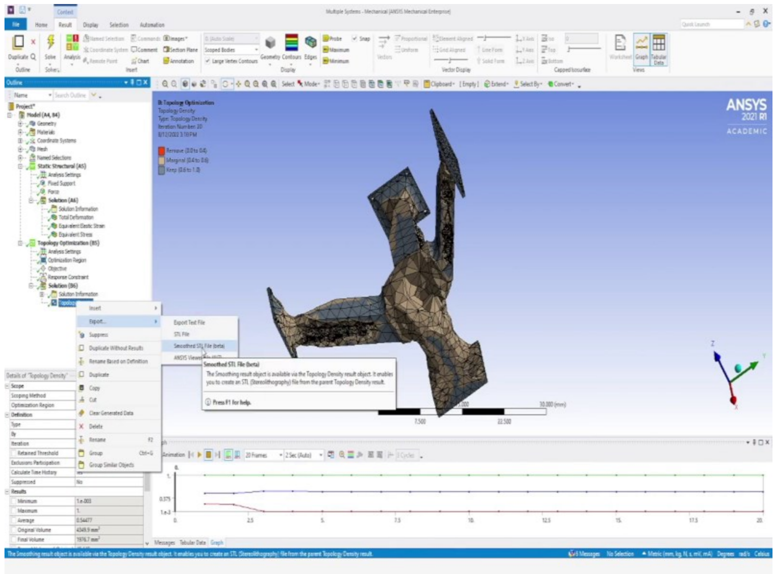
So, this is the model or the base that we are going to use. And we are going to optimize the topology. So, after starting the topology optimization, once we open this window, or this workspace for topology optimization, the target is mechanical APDL. APDL is Ansys Parametric Design Language. The optimality criteria are being selected here. Number of iterations, it is doing 500. We can say, okay only 100 iterations are required because this is just for demonstration only. So, in optimization, a region can be put here. There are three, again these three arms could be selected. So, the whole body could be picked for optimization.

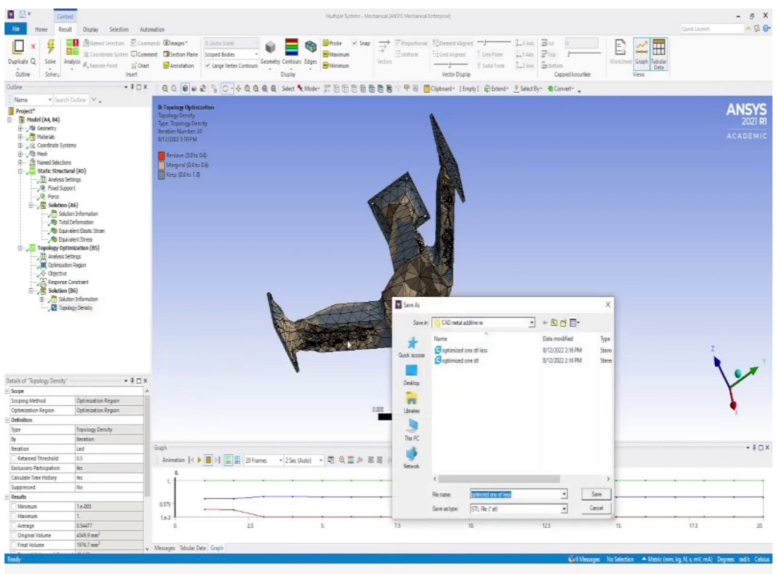
For the boundary conditions, we can select all loads or all boundary conditions. Then about the optimization it is asking, using mass, density, then lattice optimization, space optimization, then level set-based optimization, we will just say density based. So, unwanted of the extra material where the stress was almost 0 or there is no stress, no strain being applied to that material, that part of the body, it will just show that material could be removed from here. For the current design, let us try to conduct it. So, we can say solve. Now, again it will take time. You can again see the status bar down here. It is taking time. So, I will just fast forward it to see the solution of the topology optimization. Let us see the topological density here.

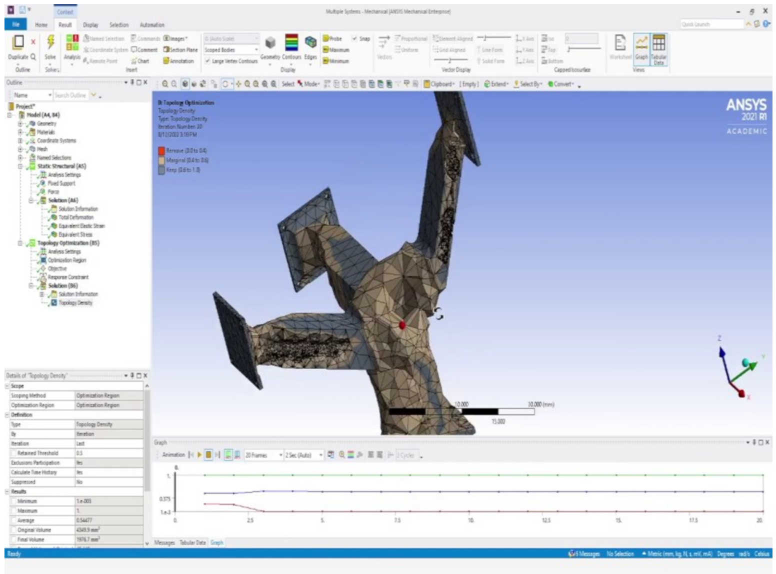
(Refer Slide Time: 33:07)






ANSYS
2021 R1
ACADEMIC

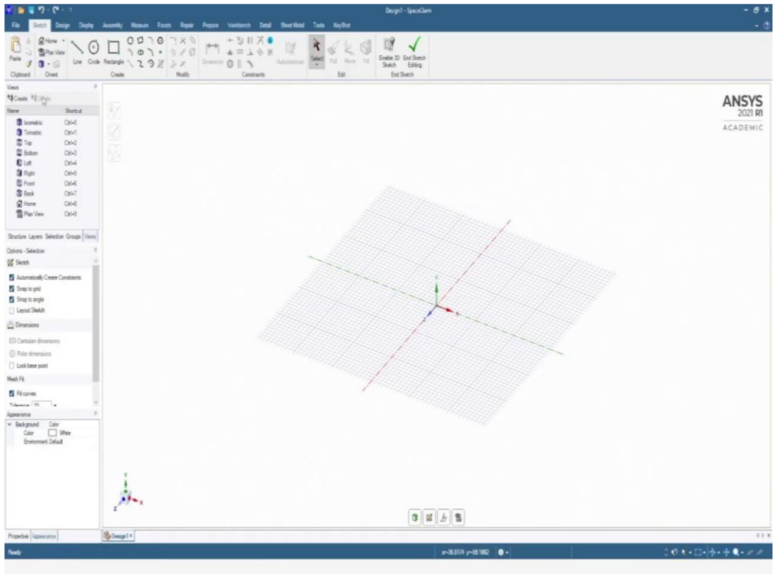

ANSYS
2021 R1
ACADEMIC

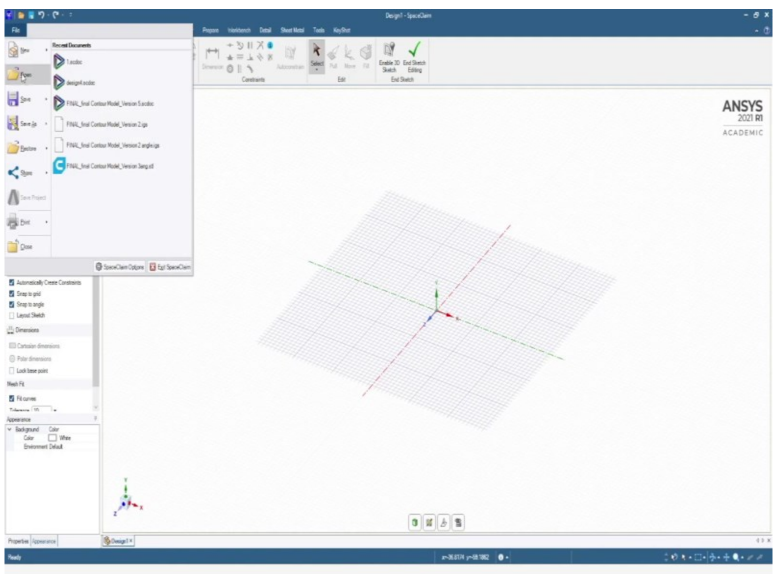


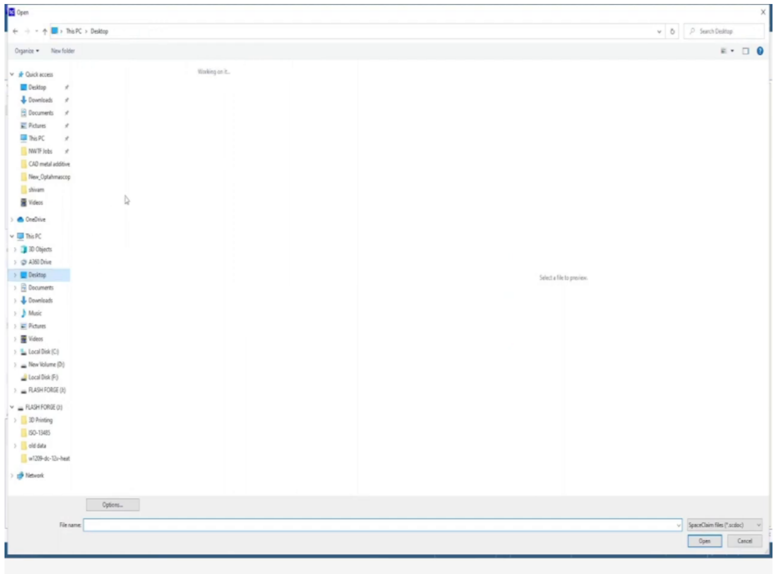
So, this is how it shows that the material could be there, or the component could be there. So, wherever the material is not required, it has removed the material from that part. So, that was the base model. And this one is optimized based upon the density. So, it can be reduced around 50% of the material.

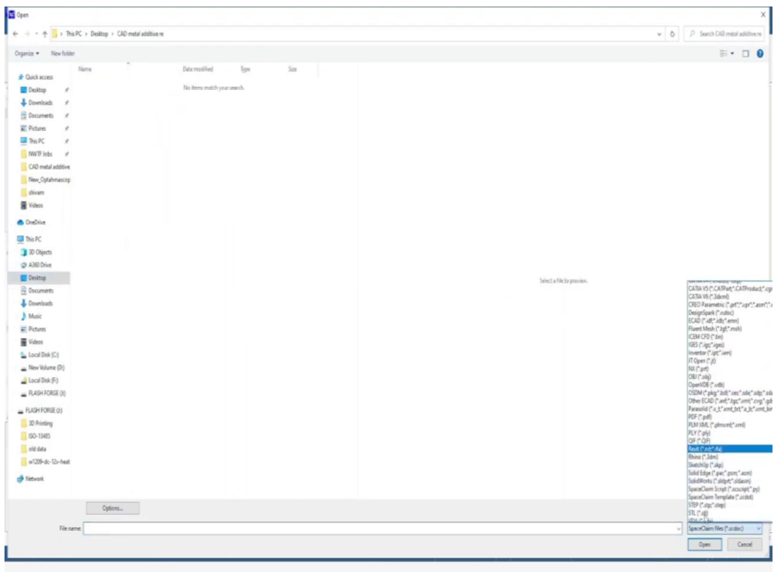
So, it means that wherever the stress or the strain is more, the material is more, or the thickness of the material is high. Wherever the stress is lesser, the material is either taken off or it is lesser. So, to put this component in additive manufacturing, it would be very tough to have these abstract or these, I would say, almost organic features to be developed. So, that is one of the reasons that we must little clean the surfaces or the volume as well. So, we can just export this file in the data first, for 3D printing. That is in STL file. So, this is now the STL file. That is, that we can bring here to smoothen the surfaces.

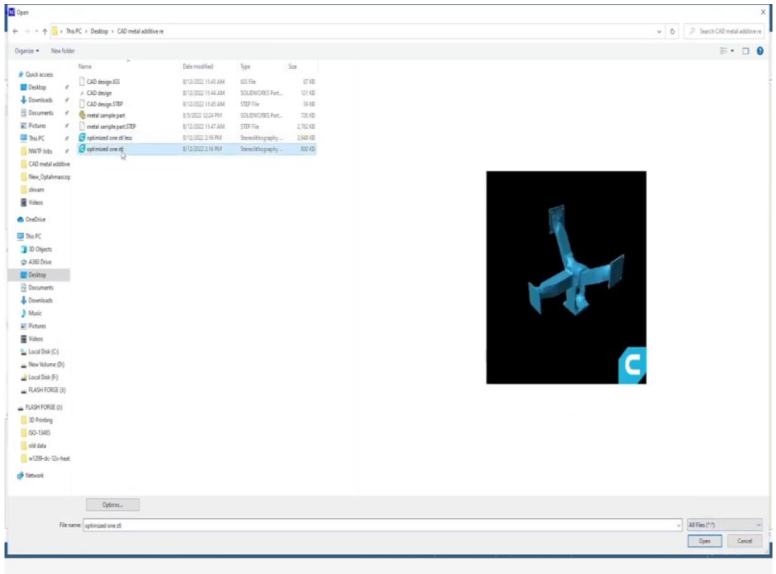
(Refer Slide Time: 34:46)

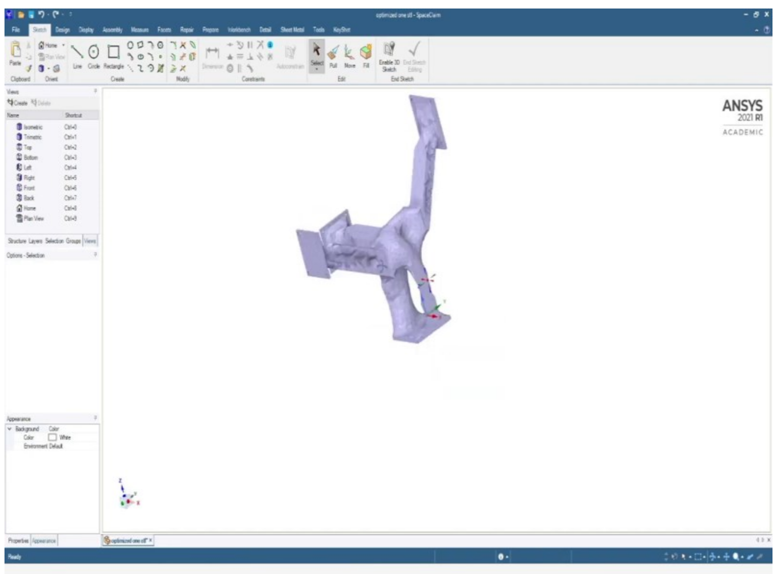


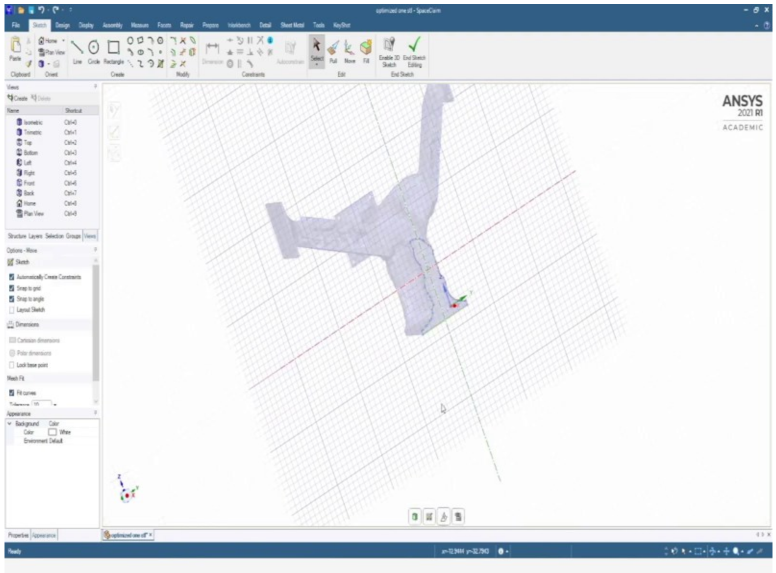


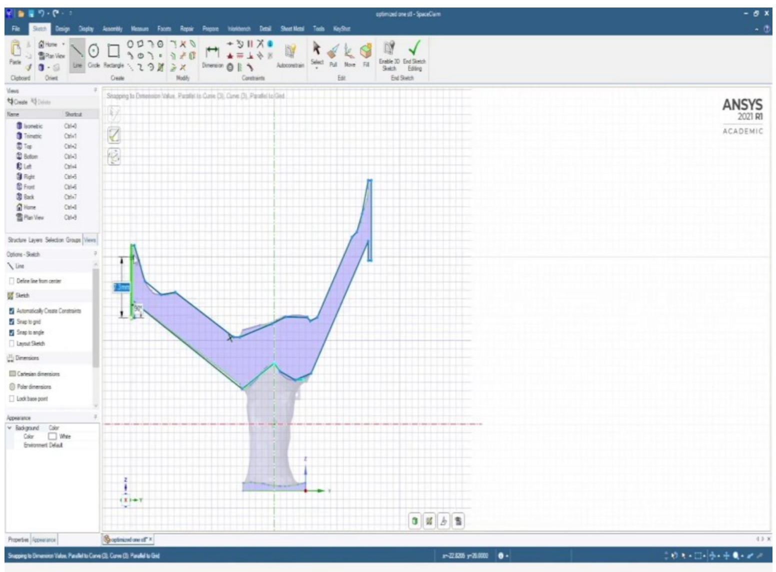


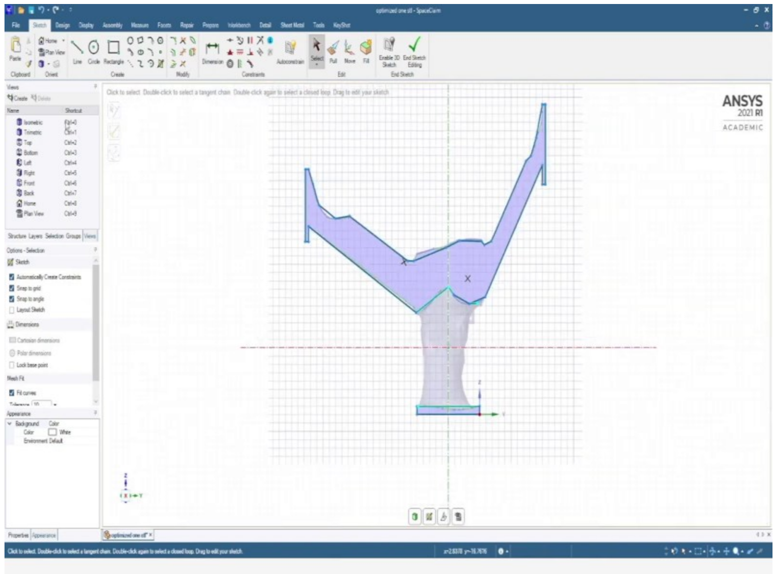


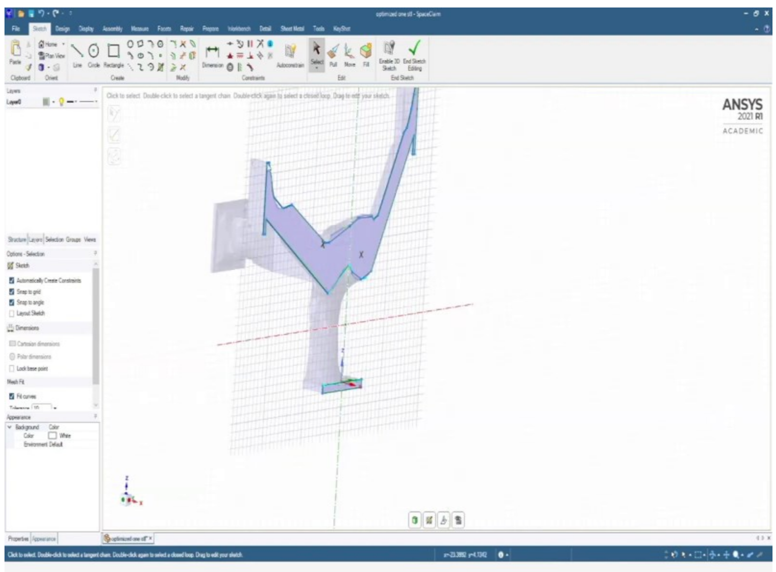


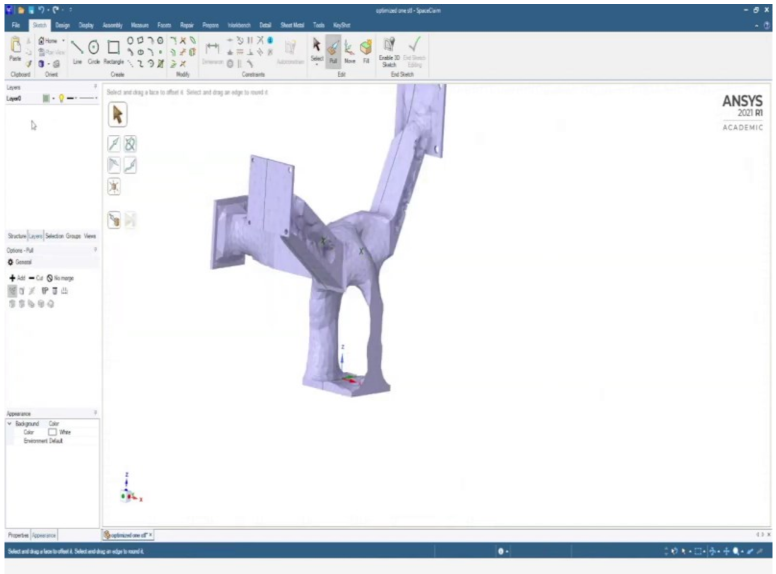

ANSYS
2021 R1
ACADEMIC

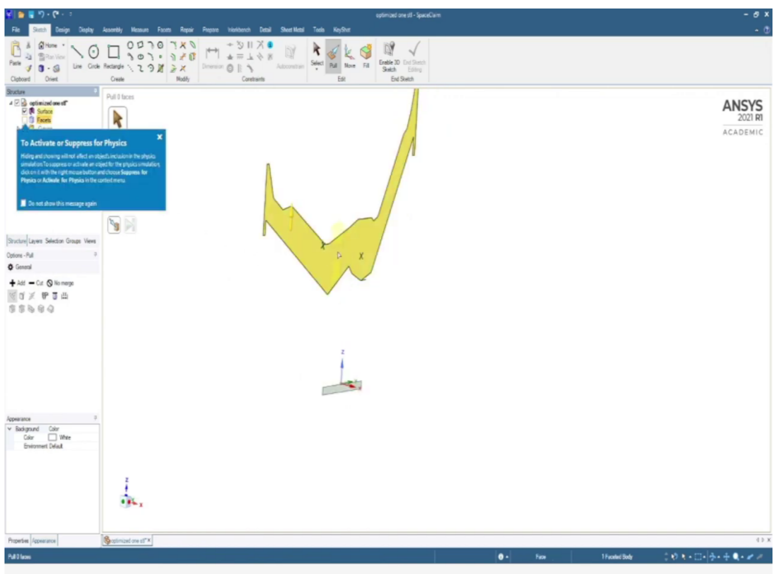

ANSYS
2021 R1
ACADEMIC

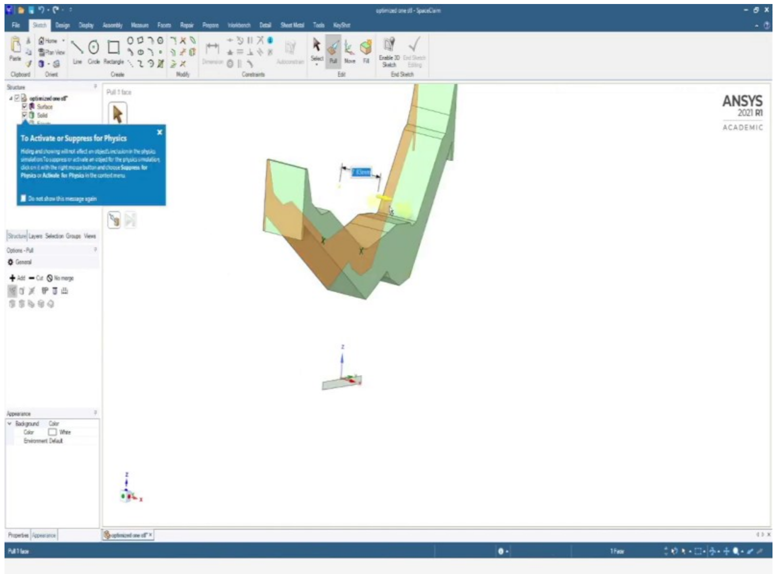

ANSYS
2021 R1
ACADEMIC

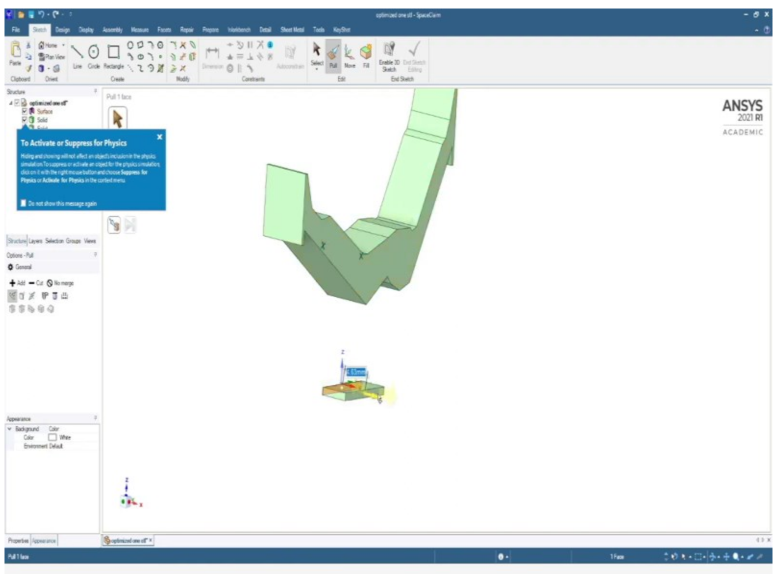

ANSYS
2021 R1
ACADEMIC

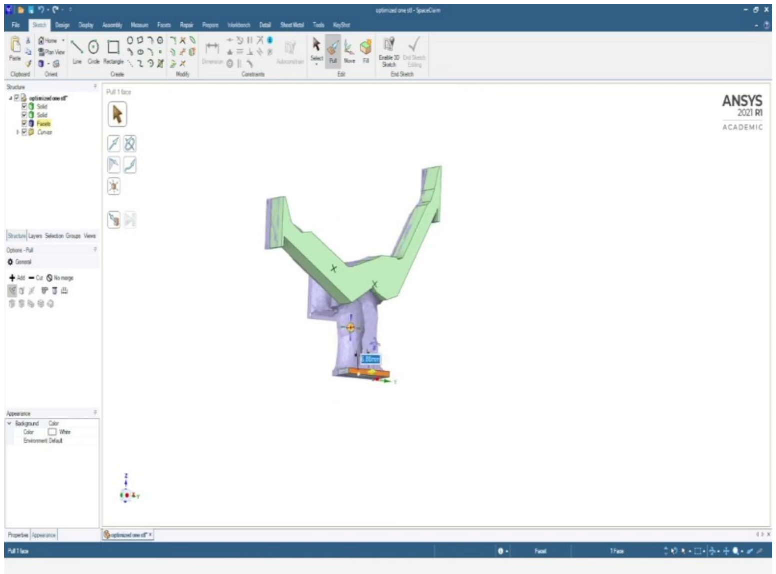

ANSYS
2021 R1
ACADEMIC

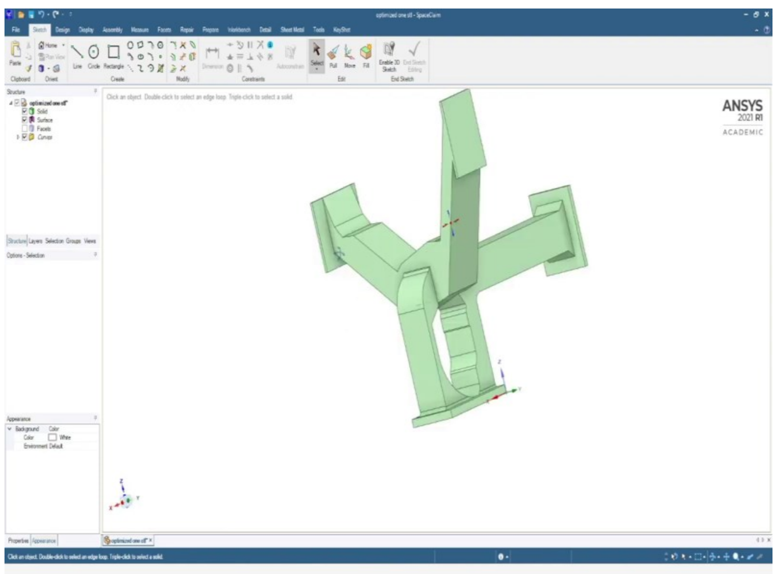

ANSYS
2021 R1
ACADEMIC

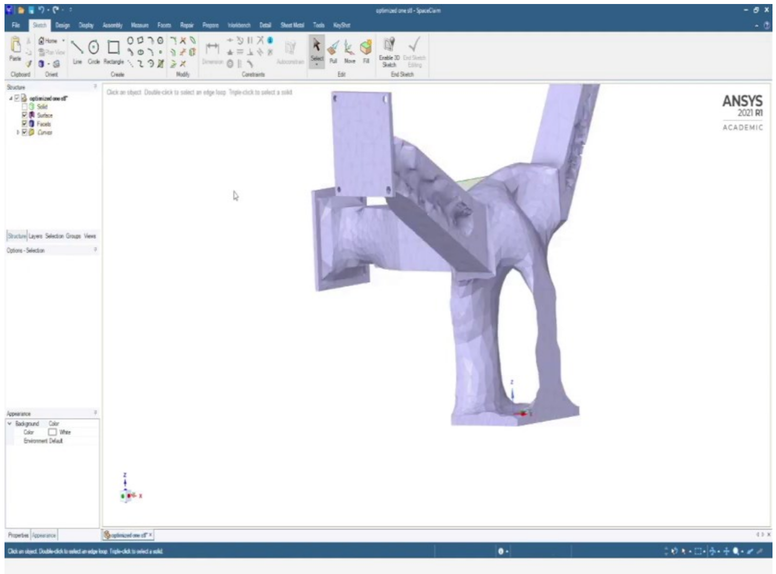

ANSYS
2021 R1
ACADEMIC
To Activate or Suppress for Physics
Do not show this message again

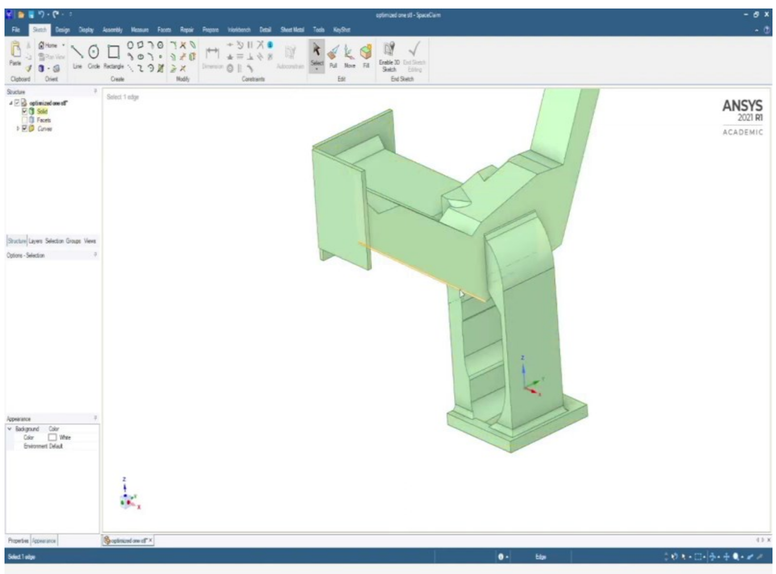

ANSYS
2021 R1
ACADEMIC

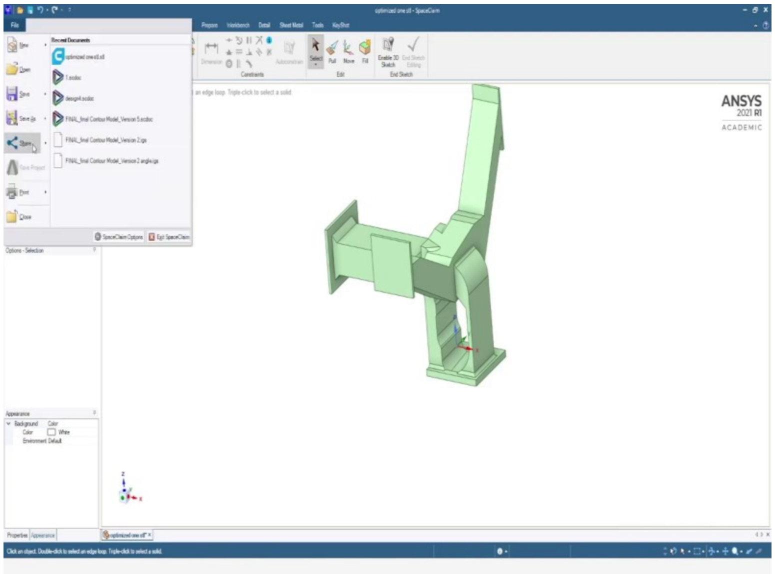

ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC


ANSYS
2021 R1
ACADEMIC



Once STL file is saved, we can then bring that. You open it and bring it to the Ansys face here. So, now, we can optimize, we can just smoothen these surfaces. So, one of the ways to manually smoothen them is to have or to put the manual sketches over it. So, we can just take the surface. This is surface, wherever the surface connects. This is again like the reverse engineering. We can select these straight lines and try to make or connect. So, I am just doing it very quickly. So, we can spend more time to cover the curves here, we can provide small round surfaces as well. We can provide small arcs as well, but it depends upon the machine capability as well.

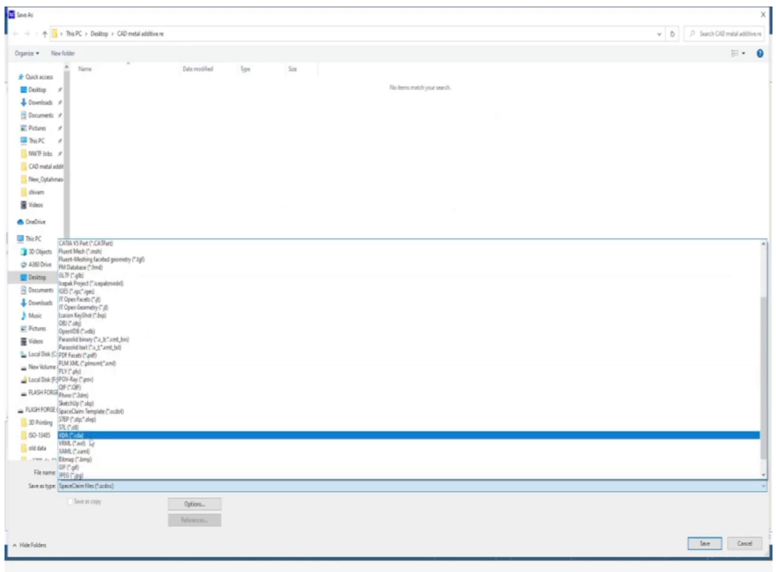
For instance, the machine resolution or the minimum component or the feature size that could be developed in my printer, if that is 50μ, and if I provide a radius lesser than that, that will not be able to be produced here in additive manufacturing. So, based upon the machine capability as well, sometimes this must be manually changed.

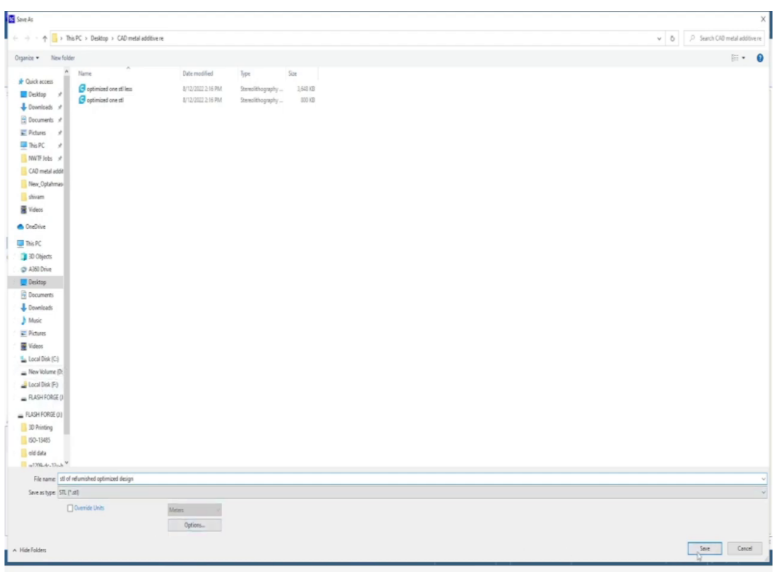
And in the similar way, now we have just generated a 2D surface jet on this plane, the intersection plane. So, similarly, a 2D surface from the bottom line, a rectangle is generated. So, let us now try to pull it. The main body, this is 2D surface. This will be pulled out. This is now pulled out; this also is pulled out to match with the optimized material structure.

So, because it was starting from the center, it could be pulled from both the sides. So, this is only the steps that we are showing here. So, it will take time to overall fit the surface to the structural body that we have got. So, we have done that, and we have come up to this level.

But this is optimized 3D printable system that we have got from here, 3D printable component. But this has majorly rectangular surfaces. So, we can make it smoother while spending some more time because we are doing it all manually. This, we can also show the facets. So, we can view or unview the things or the bodies which are here in the system. So, this is whole body, how close it is to the structural body and how different it is from the basic model that was generated, that is being shown here.

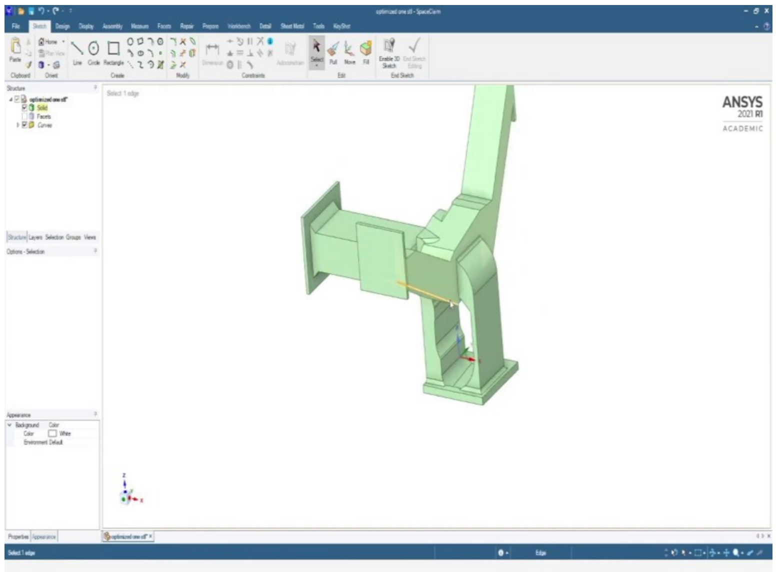
(Refer Slide Time: 38:47)





So, we can now save the file for final 3D printing. So, that is obviously the STL format. We will name the component as STL of refurbished CAD model, STL of refurbished optimized design. We will give some name here, file name, STL of refurbished optimal design.

(Refer Slide Time: 39:24)



And this model is ready now for 3D printing. So, this is how we have the topology optimization conducted on a very simple kind of structure. We developed a CAD model, we try to develop the mesh using Ansys software. We tried to add the static deformation model out of it. Then, we transfer that model for topology optimization, tried to have the material only wherever is required, smoothen that.

This was very broad presentation. Ansys softwares and COMSOL softwares need the understanding of engineering fundamentals to a great extent. Then using the software, as I said before as well, these are always GIGO, Garbage In, Garbage Out. I fast forwarded many steps because it takes time for the simulations to happen, for the optimization to happen.

With this, the demonstration on the CAD and laboratory demonstration for metal additive manufacturing is completed. Definitely, you can come up with the questions in the forum. If you need any more scientific input or need any more details, we would be happy, we will try our best to cater to all of them. Thank you.