Finite Element modeling of Welding processes
Prof. Swarup Bag
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati

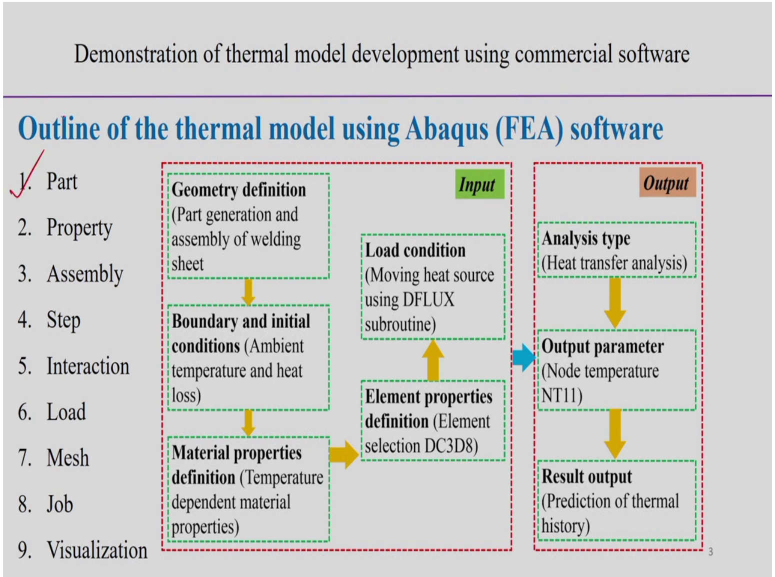
Module - 04
Application of FEM to model welding processes
Lecture - 24
Demonstration of thermal model development using commercial software

Hello everybody. So, far we have discussed the Application of the FEM to model different kind of the welding processes, and we have shown that what way we can implement the boundary condition that how to choose the governing equation. And mainly, we can see if we look into the thermal analysis, the only difference is that different welding processes the in terms of the boundary conditions.

But we are using a similar same governing equation so, maybe we can say that heat conduction equation we are using, but different welding processes that two aspect is different, one is in terms of the boundary condition and another in terms of the how implementing the heat source basically in this particular welding process.

Now, we will try to look into that how we can demonstrate the development of thermal model. Maybe we can say the temperature analysis using some sort of commercial software. I think most of the commercial software, they follow that we have already discussed that steps of FEM model the similar kind of the steps. But intuitively we can look into this one particular commercial software and we will able to know how this thermal model can be developed using this software.

(Refer Slide Time: 01:52)



Outline of the thermal model using the Abaqus that is the finite element software we can see that there are several steps. One is if we look into that it is a; it is already I mentioned here 1 is the selection of the part or maybe defining of the part, which is or basically we can say the selection of the domain, solution domain we are supposed to calculate the temperature distribution in this particular domain.

Then, 2nd property, property once we look into thermal analysis, then we should know what are the different thermal properties is required for the development of the model. So, if it is thermo mechanical model, then property related to the stress analysis property has to be defined. For example, Young's modulus, Poisson's ratio that kind of properties has to be defined along with the thermal properties in case of thermo-mechanical analysis.

After that, the next step is the assembly. So, assembly maybe part geometry all these things, then we have to follow the steps has to be defined there, different steps, how what are the different steps that the in part 4. Then, interaction of all these things and finally, application of the load and then, go for meshing, different meshing means discretization of the domain by following the different type of the elements.

In commercial software, you will be getting the different states of the element having some typical behavior or properties of a particular element or type of the elements that we have to choose and then, we can choose the meshing strategy also and maybe in selective position part, we can go for very fine meshing to capture the very high thermal temperature gradient and some part there may not be a much significant or there may not be much variation in the temperature distribution in that case we can give some kind of the coarse mesh. So, that is why the meshing strategy we can understand from this options this part 7.

Then, we do for job. This is the terminology we used in case of the this particular commercial software. Job means to do the analysis may be after which is we take some time to get this output. So, once we get the results, then we can visualize the result in different way and that may vary depending upon the one software to another software.

So, this basic steps is something like that we can see this geometry definition. So, first we define the geometry, or we define the solution domain, part generation and assembly on the welding sheet. May be suppose we want to join two different sheet, two different material, two different sheet or similar material we want to join so that part geometry has to be defined. So, with the proper dimension, what is the thickness, width and maybe length that thickness has to be defined here all information is required in the geometric definition.

Then, steps is the boundary and initial condition. So, initial condition means what is the condition required at the time t equal to 0 the before start of the solution. What are the condition, what are the temperature, whether these ambient temperature or is there any some kind of the already preheating temperature is there that has to be defined in the form of a initial temperature or initial condition.

Apart from that boundary condition can be defined in this cases, the what kind of boundary condition we are considering? Whether we are considering only the radiation or whether we are considering only the convection or we are considering both radiation and convection that kind of the boundary interaction we have to define. And even sometimes, the boundary interaction can be in the form of a for example, in the certain boundary, we can put as a particular temperature, constant temperature in that way also we can put the boundary condition.

Then, there is a need to define the material properties. Material properties it can be the either we can give the input as a constant material properties value or input can be given in the form of a temperature dependent material property. So, temperature dependent material properties can be given in the form of a tabulated form. For example, we extract the data from literature so, for example thermal conductivity of a particular material. So, thermal conductivity may vary depending upon the temperature.

So, therefore, at room temperature thermal conductivity and thermal conductivity are 1000 degree centigrade can be different. So, that the discrete data point has to be defined at the what is the temperature and what is the corresponding thermal conductivity thus can be considered as a input in the form of a material properties for a particular analysis.

Then, next step is the elemental property. So, selection of the element elemental property is there are so many cell element these other kind of elements are there. So, then we have to decide which particular element is basically do can do thermal analysis. So, there in any kind of commercial software, there is a list of the element type for example, their symbolic list.

For example, in this case DC3D8, these normally we use in case of the thermal analysis and it is it already defined what are the type of the elements, how many nodes are there in the particular elements, I think in this particular there are 8 nodes. So, that symbols can vary depending upon the software to software. And if you follow any kind of the commercial software, they are having some listed type of the elements.

And what is the limitation and applicable applicability of a particular type of the element that is well documented in any kind of the manual of a particular software. So, there we can go details, go in the about the types of the elements. Next is once we decide the types of the elements and then, next is the we define the loading condition; load condition.

Load condition means for example, in case of thermal analysis, the input in the form of a heat flux or maybe in case of volumetric heat flux also we can give for a moving heat source problem, but that kind of heat flux or moving heat source problem in that cases, probably we can some user subroutine is there.

So, we can write it is the small program in the user subroutine and that user subroutine can be interact with the mainstream software. So, the if that to perform particular task, there are different types of the user subroutine. This user subroutine is also well listed for a particular kind of software commercial software.

So, looking into this particular problem, we can choose the this particular user subroutine. For example, in the heat transfer analysis, if we assume specific to the welding process, we need to go through in details about the DFLUX subroutine. So, DFLUX subroutine, they are it is having some specific format that means there is some global variable interacting with the software that we have to understand what are the different global variability's associated with this particular subroutine.

So, once we understand what is the role of the different global variables particular subroutine, accordingly, we can make our own program to apply the heat flux or maybe what the heat flux in the sense that their distribution, shape, size all can be calculated or all can be small program can be written in this subroutine and then, subroutine will then interact with the software.

So, this way we can put the heat flux or maybe input load condition, we can say the load to the domain. So, these all are the input, these all we that in the category of the input, in a particular commercial software. Then, look into the output. Output in first is the analysis type, what type of analysis we are going for? For example, we can go only thermal analysis, we can go for only thermal mechanical analysis so, that different type analysis is basically is associated or may be involved in a particular software.

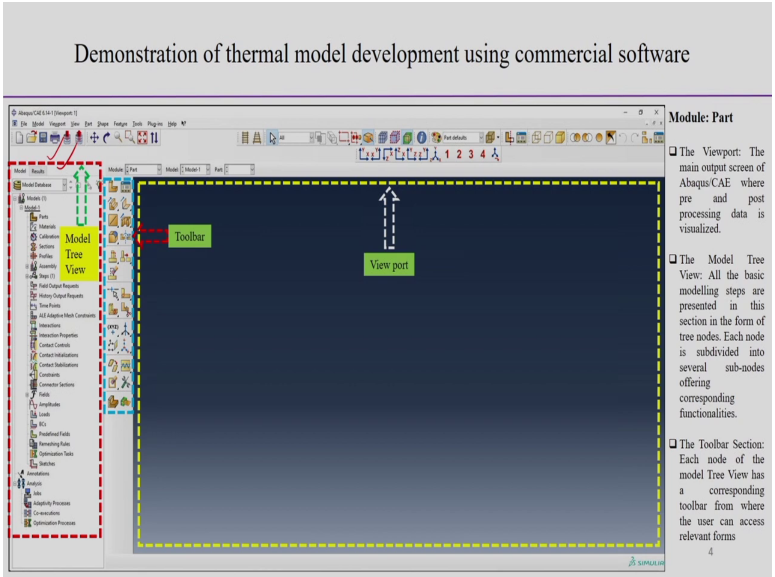
So, in this case, we can choose very the heat transfer analysis, we choose the heat transfer analysis. Then, heat transfer analysis after if we choose it and then we submit the task, then after some time we will be getting the result of this thing. Once the analysis completed, then from there we can look into what are the output parameters.

So, maybe output parameters for the thermal analysis is only the temperature. So, then we can see in particular domain, there is a output parameter are available, there we can choose, we can select which parameter output parameters we are looking for and then, we can extract the data also this as in term temperature or even we can directly plot this temperature distribution using that software also. We can see directly the what are the temperature distribution.

Then, result output. Prediction of thermal history output apart from the temperature output parameter so, then result output can also be. For example, we are interested to know one particular position, what is the time temperature profile? So, how the temperature varies with respect to time? So, that we can simply select this particular position and then, we can extract the data in the form of a time temperature data.

So, that is about that kind of output we can expect from a particular commercial software. So, definitely I am not going into much details the thermo-mechanical analysis because I think one of the module we have we will also discuss about the thermo-mechanical maybe analysis using this commercial software.

(Refer Slide Time: 10:51)



So, with this, we can start with this that how it looks like the this thing. If you see this, one is the this toolbar is there, the highlighted and another is the model tree view. So, this highlighted part is the model tree view. You see there are so many things are there on this is the model tree view, all has been defined in the different way.

And this thing maybe we can see that tool bar is also there, the model first model tree view, then we can look into the tool bars all the different kind of the tools to perform the specific task is there and the viewpoint. So, viewpoint is there. Here, you can see the geometry after obtaining running the a particular program, what are the results all these kind of things we can look into this view part view port, there you can see that this also highlighted here.

Now, module, this particular module part module. So, now, we are first we looking into the 1st module. If you remember, we are looking into the first module that this one module 1 part.

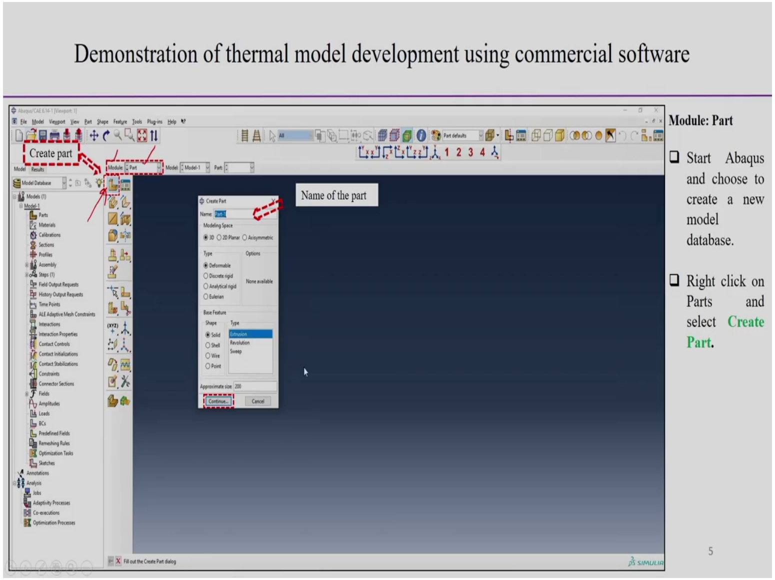
So, this part module we can see that the viewpoint, viewpoint view port basically, the main output screen of the Abaqus where the pre and post processing data is visualized basically. In the viewport, all the preprocessing before running of the analysis or after the analysis is done, all this pre and post processing this thing can be view, visualized in this view port.

Then is the model tree view: Model tree view here we can see this is the model tree view and this we have already mentioned this model tree view, all the basic modeling steps are presented here, you can see in this section in the form of a tree nodes. In the form of a tree nodes, we can see that each node is subdivided again there are into the several sub nodes offering the corresponding functionality.

So, this different subsection are having different kind of the functionalities that we can complete model tree view, we can get from here in this particular domain. Then, the tool bar section. Toolbar section the each node of the model tree view has a corresponding toolbar. So, each view of the model tree view, they are having they corresponds to one toolbar.

So, therefore, toolbar from where the user can access the relevant form. So, different kind of the forms, we can look into these toolbars, but this looking into this picture, if we go through in details the manuals, you will be able to know by looking into the symbol, then what is the task of this particular symbol that is well defined in any case of the this manual.

(Refer Slide Time: 13:24)



Now, once we do this, next step is the also we are continuing the part, then we next step will be the create the part. To create the part, once we look into this particular if you see there is it is also highlighted, this is the module and here you can see this is the part. Now, clicking here the create part.

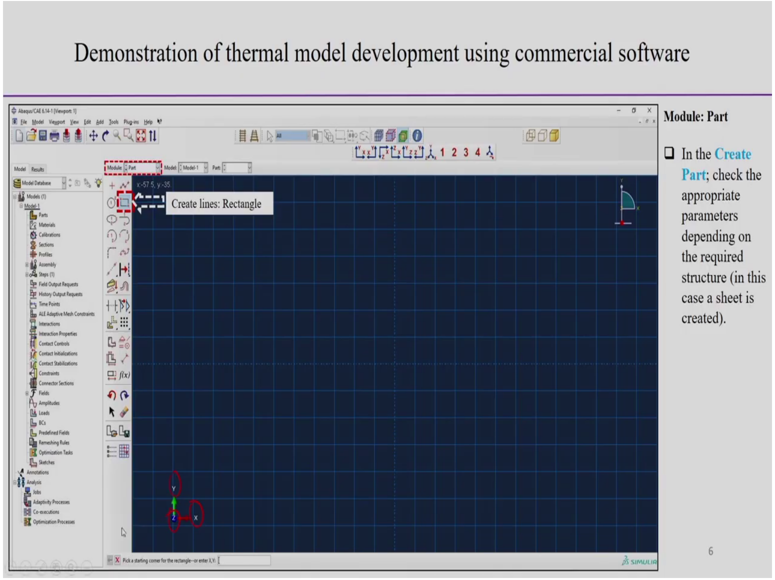
So, here this part if you mark it, then create the part so, here this display, the name of the part so, part 1, part 2 or some name we can give this particular part. Then, start Abaqus and choose to create a new model database. So, there we can see the for a model database and then, right click on the parts and select the create part.

So, here we can see, we can right click here and you can see the create part and you can see name of the part we can give it to the and there are so many options are there, the modeling space, then whether it is three-dimensional we want to do the analysis so, whether it is two-dimensional, whether it is axisymmetric problem that we can choose here also.

Then, type. This is the deformable, discrete rigid, analytical rigid, Eulerian approach or in this Eulerian model, we want to these things that all different types of the options are there and then, base features a maybe shape. Shape is the solid or the different type I am talking about the cell, cell wear type or point type. So, that shape has to be defined in these cases the in welding problem, normally, we choose the solid.

And then, type is a extrusion. May be extrusion in the sense that how we can create the geometry in this cases, the following the extrusion process this principle that excluded one direct particular direction is the same cross section will be projected will be extended one particular direction. So, that is why these cases we have the option, the different option we can create the part. Then, once we do this particular term they continue.

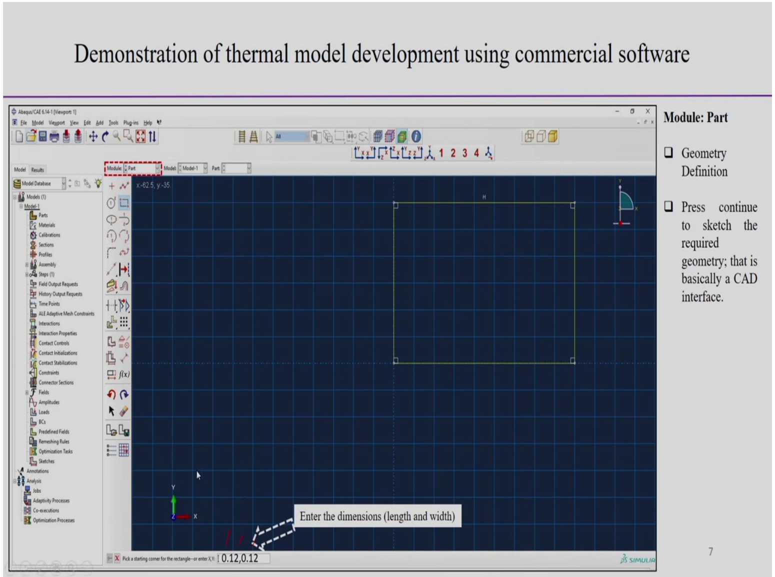
(Refer Slide Time: 15:17)



Then, create lines for example then we are creating the part, so, then if this part was the creating part. Then check the appropriate parameters depending on the required structure. In this case, a sheet is created. For example, we want to do the welding of the sheet.

So, therefore, in this case, it is important to create a sheet. So, then, the sheet having very small thickness, this create lines, it is a form of a rectangle. So, here we can see the different geometric shape are there also available so, in this, you can see that this Y axis, X axis and Z also though create lines.

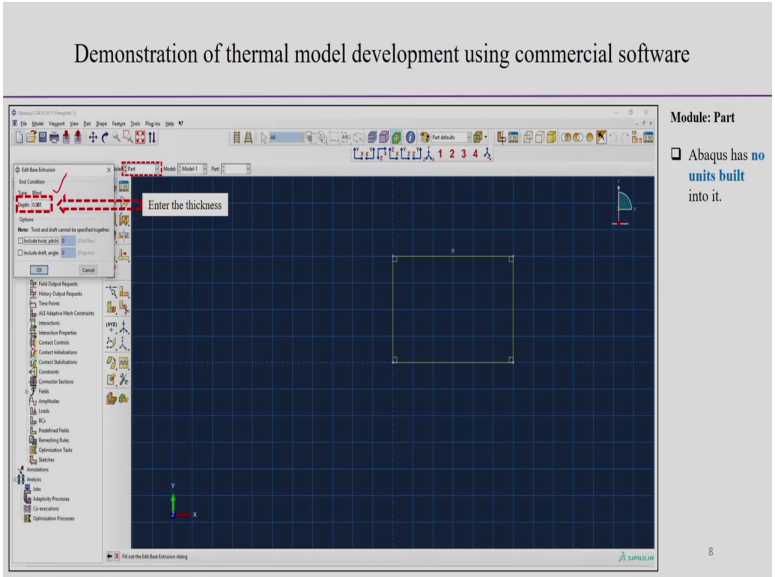
(Refer Slide Time: 15:59)



So, basically create lines means we can create the rectangle here. Now, enter the dimension, we select first the rectangle, then enter the dimension length and width. Here, we can put the enter the dimension 0.12 and 0.12 two-dimension that means, length and width we can give this thing.

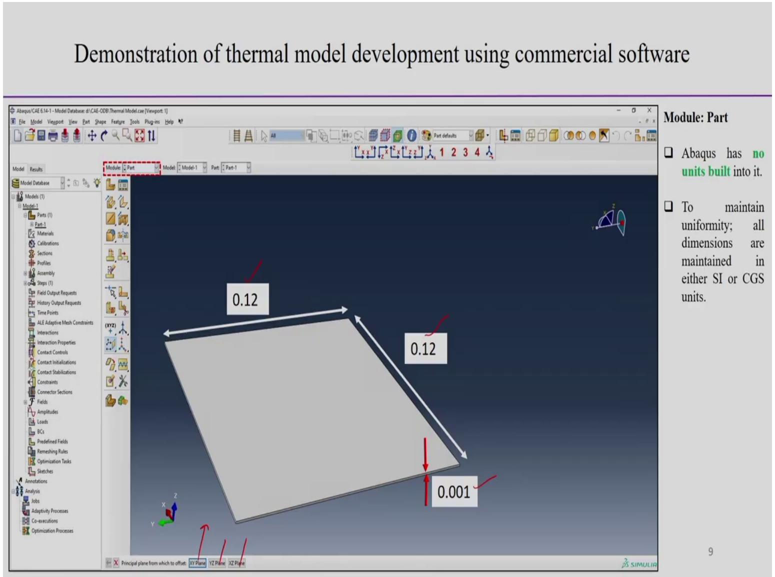
But remember in this cases, it is not following any kind of the units as a relative this value 0.12 and 0.12 that this created the rectangle, this kind of space. But so, geometry definition that press continue to sketch the required geometry that is basically a CAD interface. It is basically actually CAD interface to create the geometry. So, here, the length and time; width we define and then, we can create this particular space.

(Refer Slide Time: 16:42)



Now, enter the thickness. So, once we the rectangular shape surface we create, then we can part if you put the continue and then, we can pop with this edit the base extrusion. Here, we can see the depth we can define that we can, you can the highlighted part here we can see the depth is defined. So, here, we can see the enter the thickness here, we can put the desired thickness.

(Refer Slide Time: 17:08)

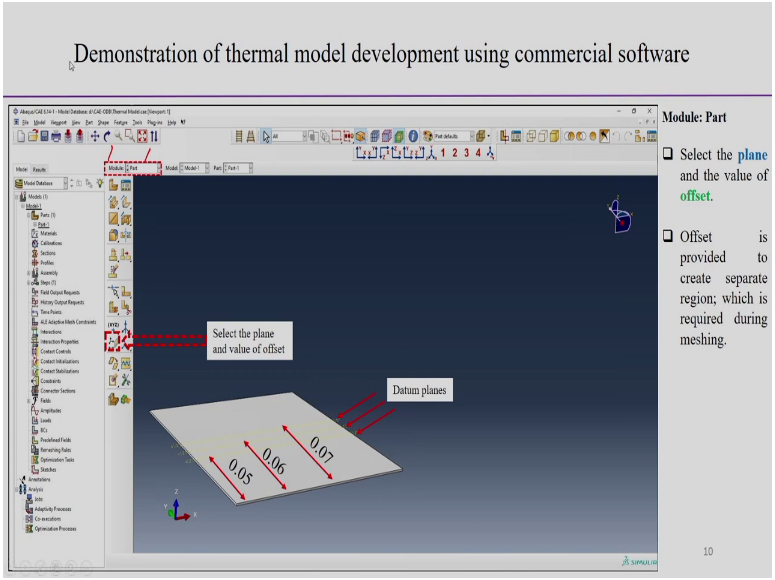


And then, we can create this Abaqus has no unit built into it to maintain the uniformity, all dimension are maintained either SI or CGS unit. So, therefore, we maintained particular unit, but it should be throughout the same unit. So, then 0.12 is the length, 12 is the width and the thickness is 0.001 so, this is the now we create the solid geometry. So, solid geometry having very small thickness in this particular case, the once the geometry has been created in this particular case.

So, we can see that if you see this highlighted also, if you see that this is the XY plane, it is actually showing the XY plane. If we want to project only on you want to see only the XY plane, then we can click on this particular, select this particular position or mouse click here, then we will be able to get the projected area XY.

Similarly, YZ plane, XZ plane that different view, it is possible to see for a three-dimensional object that we have the option to look into on different views also. So, this is the first step, we basically first we create the geometry to start the analysis. So, basically, other way you can say we can create the solution domain in the very first step.

(Refer Slide Time: 18:24)

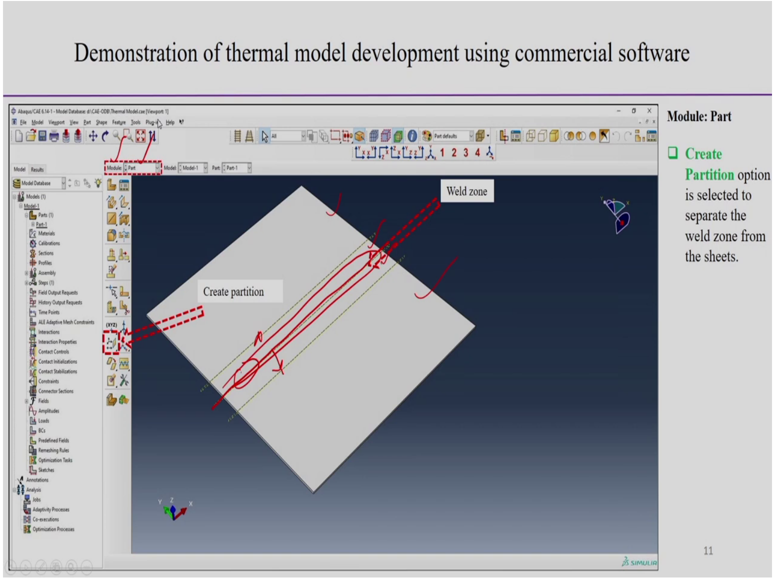


Next select the I think next till it is continue, if you remember it is highlighted, the module and the part module is basically, you are handling the part module. Now, select the plane and the value of the offset, we can put some sort of offset also. Offset is provided to create the separate zone and which is required during the meshing. So, here we can create the select the plane and the value of the offset.

So, we can see separate I think separate region we can create. For example, once we create the geometry, there are sometimes there is a need to identify the zone near about the weld central line. Because you want to create the very fine mesh near about the weld central line so, therefore, we have to define this particular zone.

So, that offset can be used that particular option can be used, select the plane and the value of the offset; such that we can create the separate zone through which you need the different kind of meshing, which is different from the other zone. So, that is why to identify this thing, select the plane and the value of the offset value we can put it and the with respect to the datum plane, we can find out create the region zone.

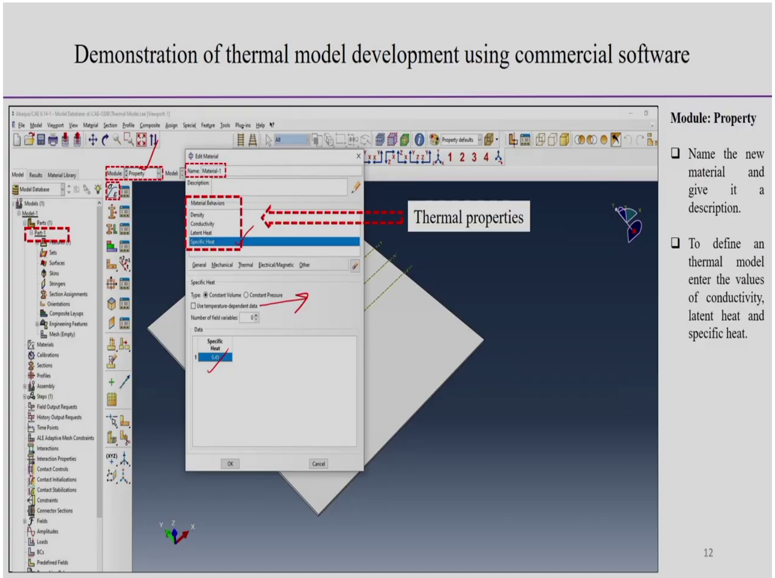
(Refer Slide Time: 19:31)



Now, create partition. So, this is the zone. Basically, this indicates the weld zone see this indicates the weld zone and this zone may be differentiate with respect to the other part. Now, create partition. So, create partition option is selected to separate the weld zone from the sheets. So, basically to separate out this weld zone so, this we define suppose we are moving the heat source from here so, this the, this can be the fusion zone and near about it can be the heat affected zone.

So, basically, we can say this is the weld zone particular case. And then, after that create the partition option to select to selected; selected to separate the zone from the sheet. So, that is we separate out this particular zone from the sheet. For that purpose, we can create the partition option.

(Refer Slide Time: 20:25)



Now, once we create partition option, then now next step is the defining the properties. Now, next module is the property. If you see now there is a change, now in previous one, we can see that module part; module part. So, part module was looking the very first step. We remember part module that once the part module is done then we can look into the module property. Here, we can look into that the property module.

So, property module is simply what way we can define the different properties for the particular analysis. So, once we select particular materials, some material library maybe there in the software, you can directly choose this particular material and we will get all the data or you can give the input your own properties value through this interface also even the software.

So, for example, in this case, the we edit the material property basically property and property we can we have seen that here we can see that property specific it is defined. And this is basically material properties and this all these density, conductivity, latent heat, specific all these particular material properties associated we can say these are the thermal properties.

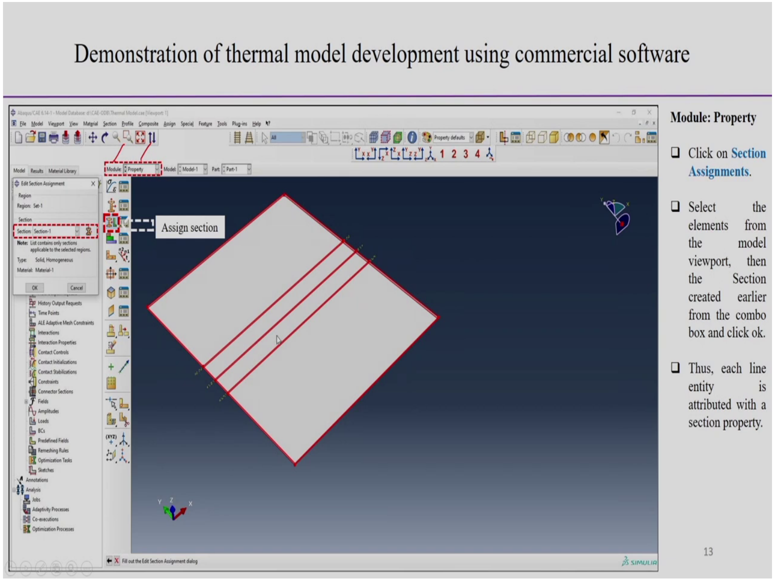
So, thermal properties we can see the type constant value for example, in this case, the constant volume type or constant pressure, specific heat ok. So, two types of the specific heat we can the we can CP or CV. So, constant at specific constant volume CV or we can see give the specific this value. So, here we can given the at constant volume, what is the specific heat value? Specific heat we can put the value here.

So, property, name the new material and give it to a description. So, we can change a particular, we can change the material 1 or something you can give some new material also whatever you want to give the input properties.

In that case, the define the thermal model, enter the values of the conductivity, then once we choose the specific heat, then accordingly you can choose the conductivity, you can choose the latent heat and density individually, you can put the specific values here also so, put this particular here.

And you can see also there is a option user use temperature dependent data. So, that option also we can if we click on this temperature dependent data. Then also you are you will be able to define the different temperature dependent properties that means, property value at the different temperature. So, that can also be done for this particular analysis.

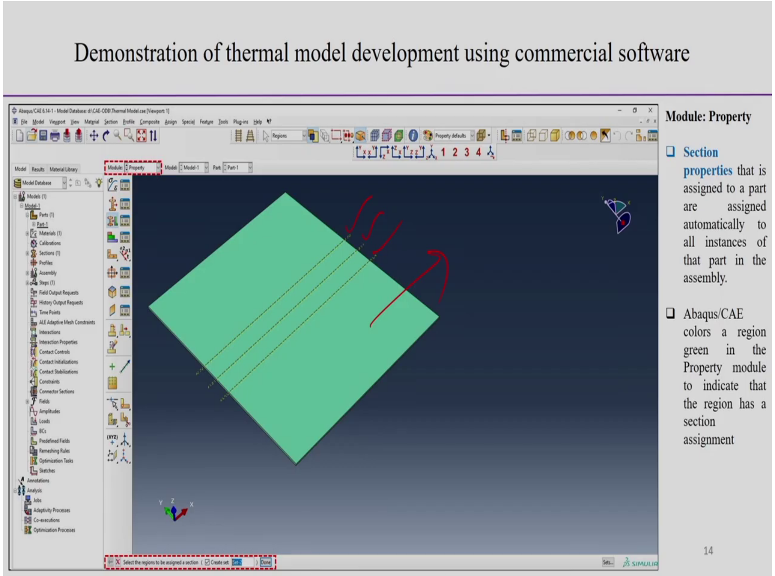
(Refer Slide Time: 22:54)



Now, till it is going on the module property and the click on the section assignments. So, then if we see the assign section so, it is a module property now, we can put the assign section. So, once we click this assign section, select this thing assign section. So, select the elements from the model view viewport and then, the selection; selection created earlier from the combo box and click ok. Once we put the selection, put the combo box here and keep on click on ok. Thus the each line entity is attributed with a six section property.

So, we can see that when we assigning the property, we can put the select section assignment, section assignment is basically this different setting, the different zone because we have we already created, we have already defined the different zone so, the weld zone and the remaining part. So, that could be highlighted when we see the assigned section here, we can see these are highlighted and this to define the different zone in this particular analysis.

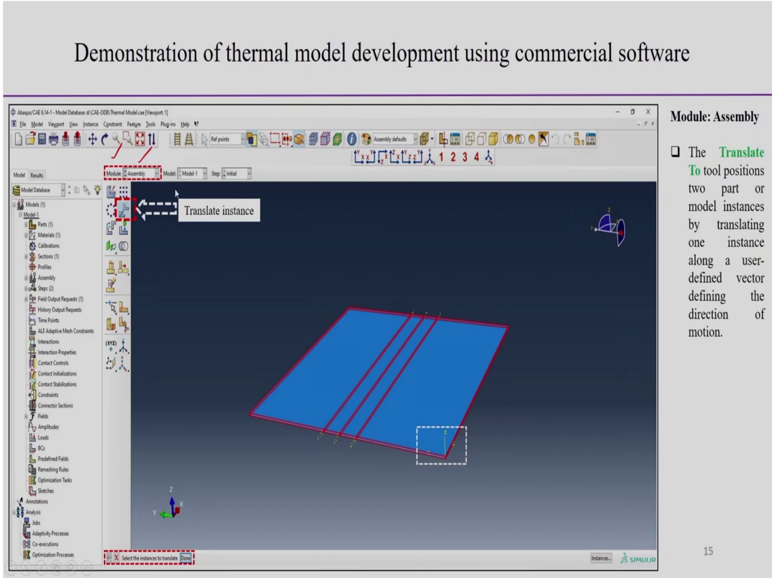
(Refer Slide Time: 24:06)



Now, selection properties, then that is assigned to a part are assigned automatically to all instances of that part in that assembly. So, basically, we can see that Abaqus colors a region green in the property module to indicate that the region has a section assignment. So, if you see region green in the this, if you see this type of marks are there.

So that indicate there are some section assignments and even color green in the property module to indicate that the region has a section assignment. So, this when the change is this green color, we can display that indicate the section assignment is there in this particular analysis.

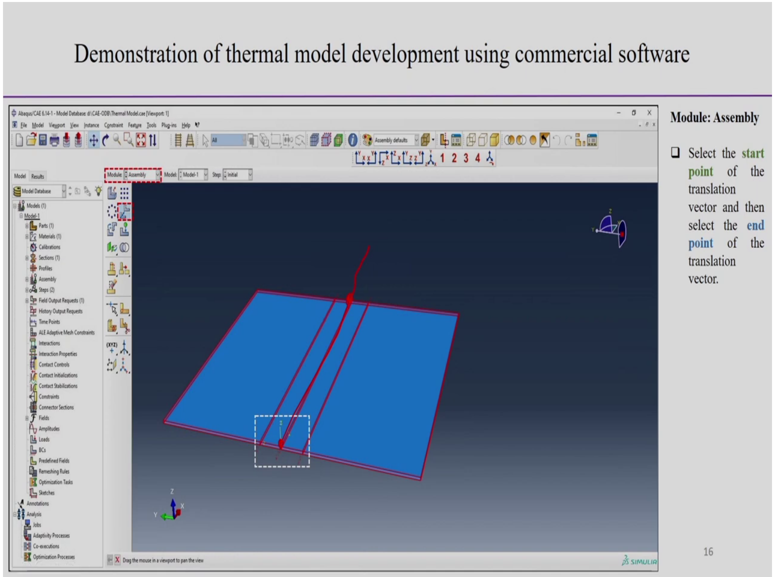
(Refer Slide Time: 24:49)



Now, translate the tool positions two part of the model instances by translating one instances along the user defined vector defining the direction of motion, ok. Now, we can put one particular direction of the motion, the translate in intense in using this particular option, we can decide the which direction heat source will be moving so, that option we have to look this thing see here, if you see.

Now, come to this point module assembly here, this assembly part is coming into this picture. So, then, here we have to define which direction the heat source is moving so, that can be the translate to tool position two part of the model instance by translating one instance along a user defined vector defining the direction of the motion. So, basically this now we come to this point of the assembly module.

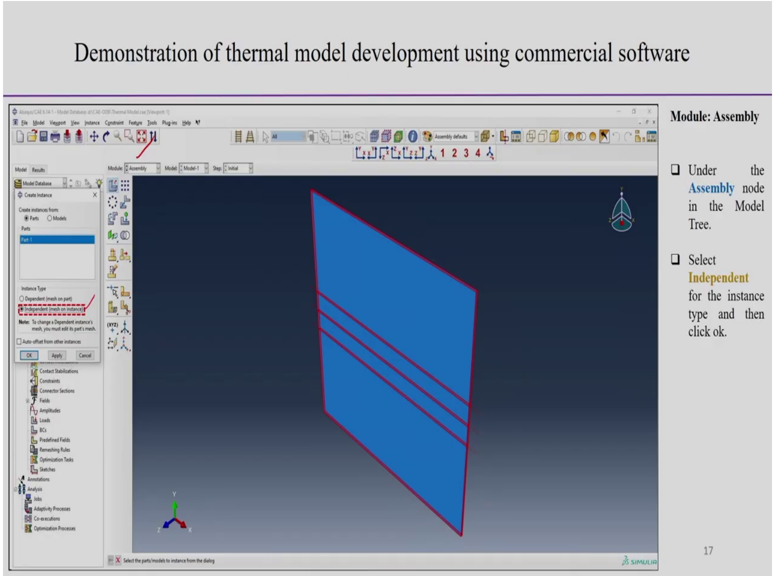
(Refer Slide Time: 25:43)



Now, assembly module, select the start point of the translation vector and then, select the end point of the translation vector. So, the heat source will be moving actually at the center point so, from here to here. So, translation motion we can put the select this point and select the end point; such that it will be automatically accommodated by the software.

The direction of the motion the heat source will be moving from this point to first point to the second point that is why we have to that actually necessary to do to defining the direction of the moving heat source in this particular case. So, that comes under the assembly module.

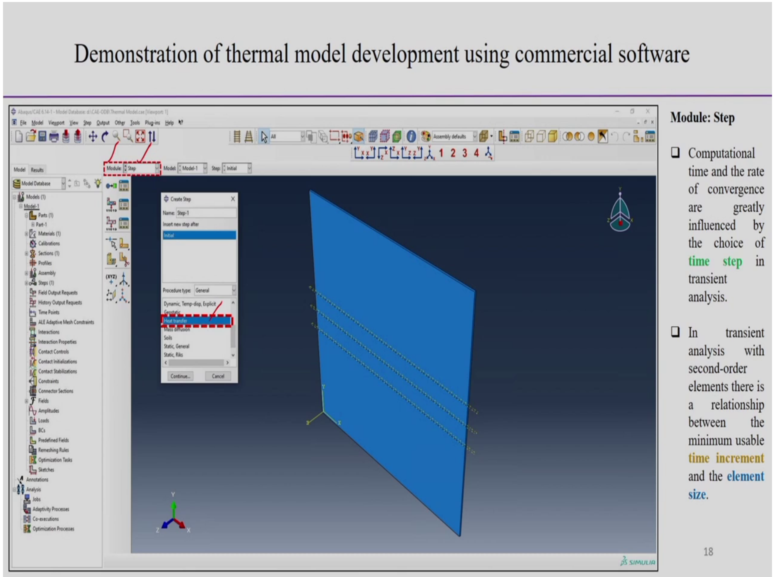
(Refer Slide Time: 26:25)



Now, this assembly module even I think till we are under assembly module, we can see the here the module assembly module. So, in this module, now we try to look into the under the assembly node in the model tree.

So, here we can see look into the assembly mode in the model tree also, there we can seek the select independent for the instance type; and then click ok. So, independent mesh on the interface in this particular, we can put the independent and then, we can put the click ok. Let us see how it works.

(Refer Slide Time: 27:03)

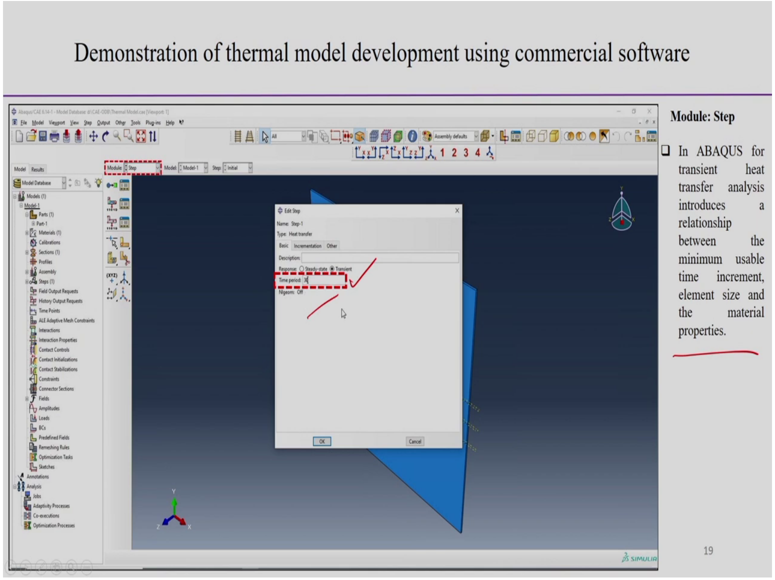


Now, here you can see. Now, we come to this next module that is the step module. The step module, the computational time and the rate of convergence definitely we know are greatly influenced by the choice of the different time step in the in this transient analysis. So, time step is very important part in the transient analysis.

Here in transient analysis with the second order elements, there is a relationship between the material properties and the what is the maximum value of the, minimum value of the time step is required are on in incremental time step required in an analysis process.

Now, here we can see that module and the we can in the modeling the step module so, step module, we can see the create step. We can see the create step that we have seen in the, we can choose this particular heat transfer analysis so, that means, initial step heat transfer analysis in this, for this particular geometry.

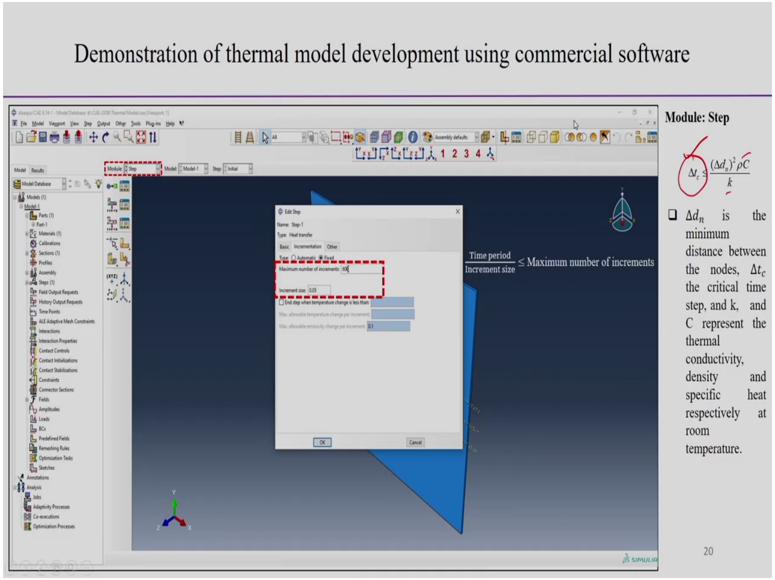
(Refer Slide Time: 27:58)



Now, there is a need to in the particular step, the edit step the in this particular module, the time period has to be defined. So, time period we can see the time period has to be defined here, let us see in ABAQUS, let us see the nodes.

In ABAQUS for transient heat transfer analysis, introduce a relationship between the minimum usable time increment, element size and the material properties. There are some relation the what can be the minimum incremental time step that can be relate to the material properties and may be mesh size also having some relation. So, here basically, time period has to be defined. So, in this edit step so, one may be we assume it is a time step for example, it is 30.

(Refer Slide Time: 28:42)

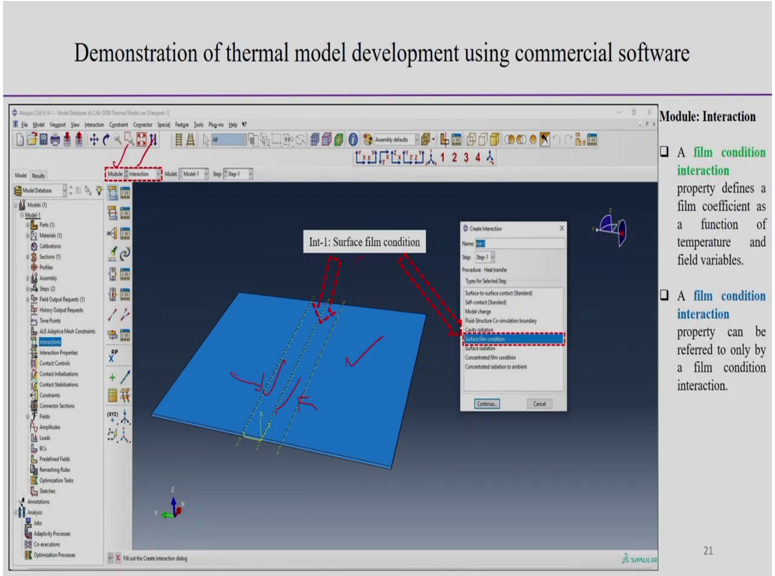


Now, see that what is the minimum time step t c is the del d n square rho C you see all the material properties are there. Del d n is the minimum distance between the nodes; between nodes. Therefore, delta times is the critical time step and k and c are represent thermal conductivity and specific heat; and the specific heat respectively and that is under the room temperature. So, then accordingly, we can decide the what may be the time step we can estimate this thing.

Now in this step where we can see the maximum number of increments. So, maybe each and every step and step by step, it will be calculating in transient analysis. So, how many number of increments is required that we can define or increment size also we can see that increment size can be 0 point some value we can give some 0.05; such that it will be accommodated.

For example, one time step there may be several other incremental steps can be there so, that incremental step can be very small also.

(Refer Slide Time: 29:40)



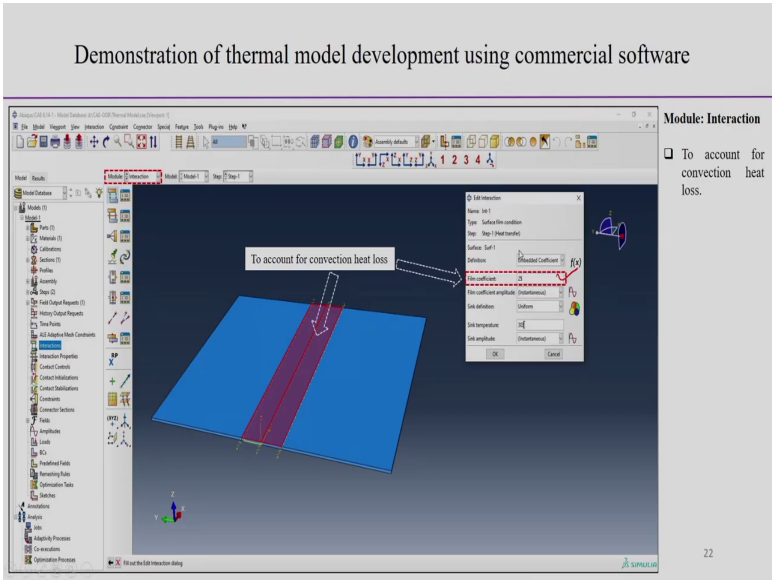
Now, this module is the interaction. So, interaction is required this module. So, module interaction how it works in this particular module interaction? Here you can see there is a in interaction 1, the surface film coefficient basically, the a film condition interaction property. Interaction property is actually defined on the surface in the terms of the film coefficient, which is as a function of temperature and field variables.

So, basically, this once so, we want to implement the heat transfer coefficient basically, in that is the comes under this particular module the interaction properties. Interaction properties, the film heat transfer coefficient normally or otherwise a convective heat transfer coefficient on the surface defined.

But in this particular welding problem, it is the that there are different surfaces we can put the different values of the heat transfer coefficients at the different surfaces that is why different surfaces in the sense that if we put this weld zone maybe different value we can put in this particular this the surface.

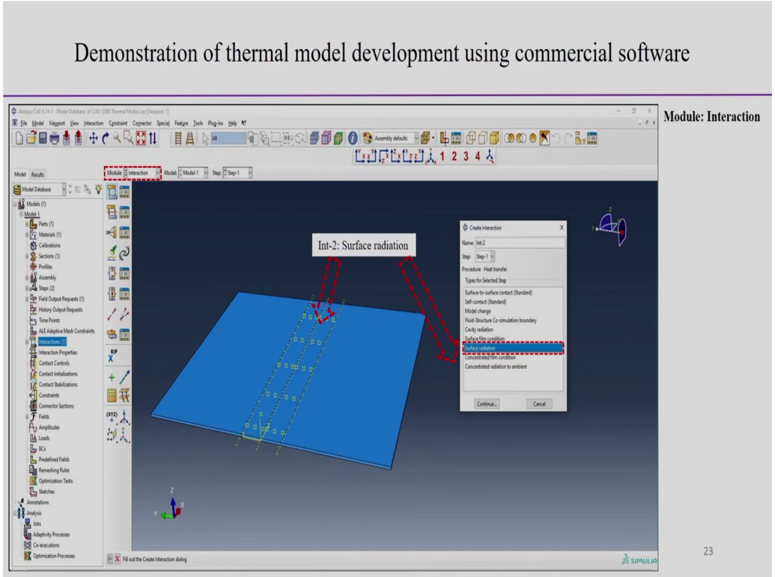
But here also, but that has to be separately has to be different zone has to be defined. So, then only we can one particular zone we can put the one particular value of the heat transfer coefficient. So, here put interaction 1, the surface film condition that surface film condition we can put here with the continue this thing.

(Refer Slide Time: 31:06)



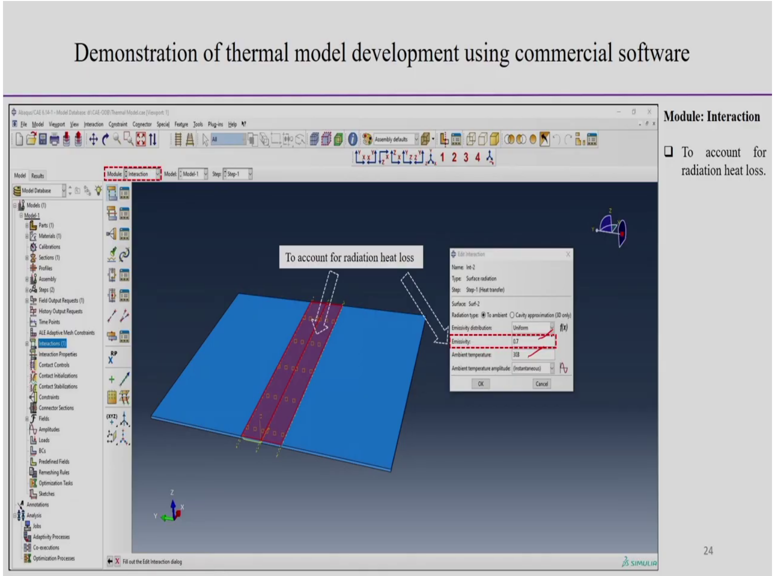
Then, we can see to account for the convection heat loss, here we can put the some value of the film coefficient, we can put the value of the film coefficient equal to 25 to account for the convective heat loss. That I can put there the film coefficient we can put this interaction.

(Refer Slide Time: 31:25)



Now, surface radiation; Now, once the film coefficient defined, then you basically you are considering the convective mode of the heat loss from the surface. Now, there is a radiative heat loss also. So, in that cases, the surface radiation also has to be incorporated so, then choose the step 1, one is the then we can see the surface radiation term. So, once we choose the surface radiation term to put the properties related to surface radiation term.

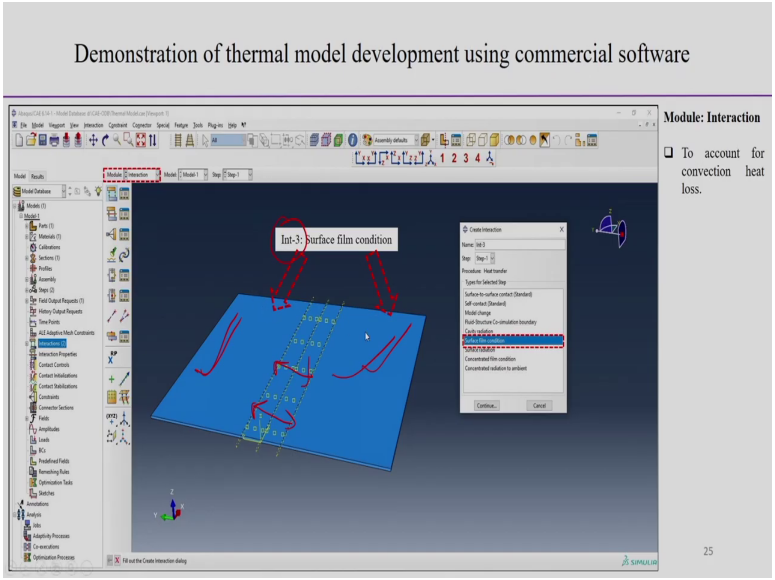
(Refer Slide Time: 31:50)



So, therefore, to account for the radiative heat loss on this particular defined zone on the surface. Then I can put the emissivity value and ambient temperature is already defined and maybe emissivity distribution, we can follow the emissivity distribution.

We can choose the uniform or we can any functional distribution can follow any functional form that can also be possible to incorporate here. So, to account for the radiation heat loss, we can put, we can define the emissivity value.

(Refer Slide Time: 32:22)

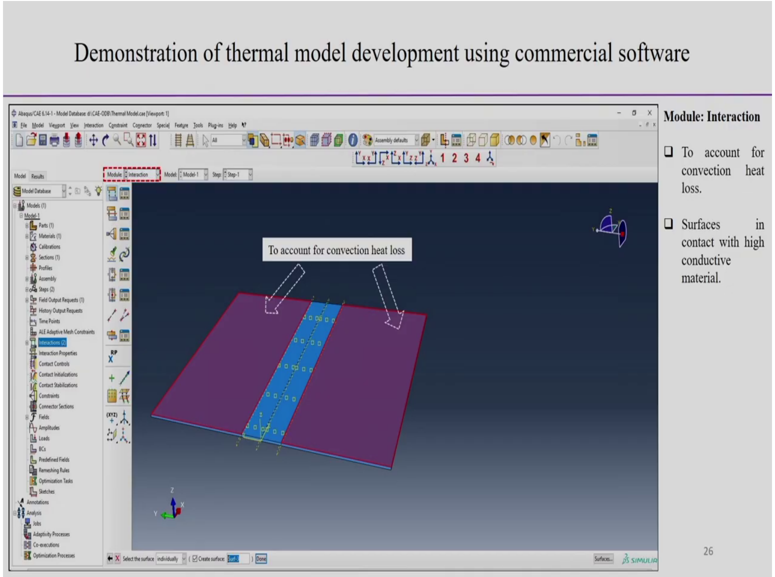


Now, surface film condition: So, surface film condition again, we come back to the surface film condition because this weld zone, this zone we have defined the film heat transfer coefficient as well as the for the radiative heat transfer, both the properties has defined already in this particular, but what should be the value in this particular zone?

There also we can see though we are defining Int 3, interaction 3 so, different interaction. So, if we look into that was the account to convection heat loss that is interaction 1 surface heat film condition, then interaction 2, surface radiation, then interaction 3, surface film condition.

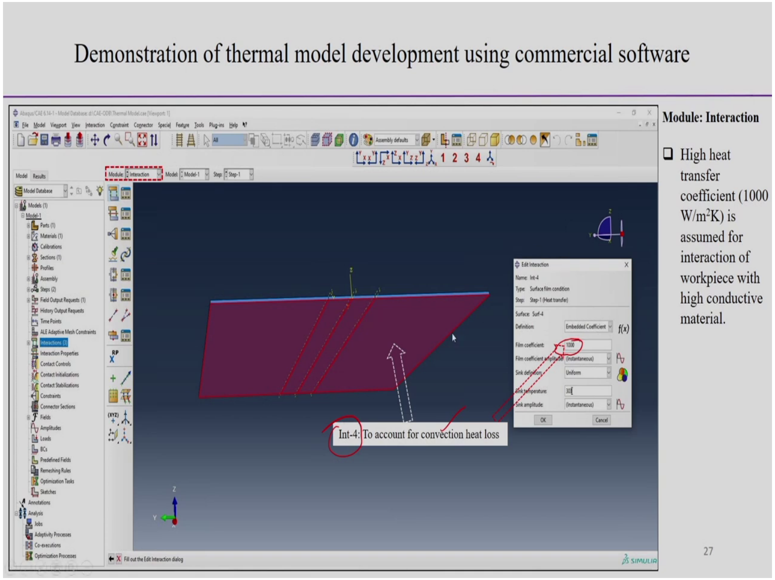
So, different interactive properties has to be defined. So, but the surface film condition are different zone this on this particular surface has to be defined, also film coefficient has to be defined. So, that is why we have separate out this particular part.

(Refer Slide Time: 33:13)



So, once we define the surface film condition, then we can go for the next step to account for the convective heat loss these two part. They are to account for the convective heat loss and the surfaces in contact with the high conductive material. So, that we can put account for the convective heat loss from this other surfaces.

(Refer Slide Time: 33:29)



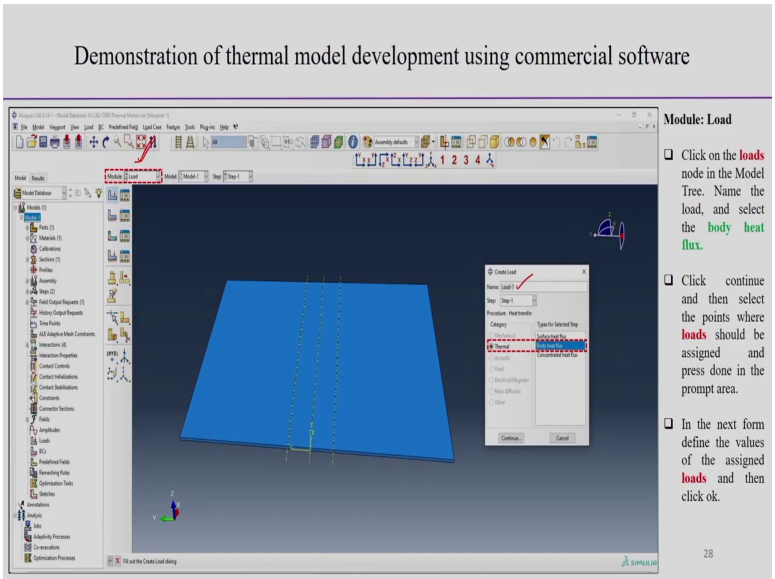
Now, in the bottom surface to account for the convection heat loss from the bottom surface for example, in the analysis from the bottom surface, we can put since the bottom surface is basically attached with the some sort of fixture during the actual learning process so, bottom surface heat transfer coefficient can be considered as very high value as compared to the conventional which is interaction surface to the year.

So, therefore, in this case, the embedded coefficient can be defined. You see that interaction 4 basically from the bottom surface, bottom surface to account for the convective heat loss, you can put the very high heat transfer coefficient 1000 so, that value has to be defined.

It means that once in this particular module interaction, you just choose the different interactive surface. So, different interaction 1, interaction 2 in that way we can define these different surfaces and the different surfaces we can put the different values of the heat transfer coefficient. So, that is the purpose of this interaction.

So, high heat transfer coefficient is assumed for interaction of the work-piece with basically high conductive material. So, basically if there is a very high conductive metal, you can work. If you use as an back plate or maybe as a fixture, then in that cases probably, we can put the very high value of the heat transfer coefficient to get the result. So, that is the interaction 4 to account for the convective heat loss.

(Refer Slide Time: 34:53)



Now, once it is now, next step is the once we defined all the boundary interaction is done, then next module is the load. So, basically application of the thermal load to this particular problem. So, thermal load here you can see click on the loads node of the model; the click on the load nodes of the model.

And there name basically model tree it is available, click on the loads mode on the in the model tree, then name the load, we have to give the name of the load. So, basically, there here you can see that load 1 and in this case name the load and then, select the body heat flux.

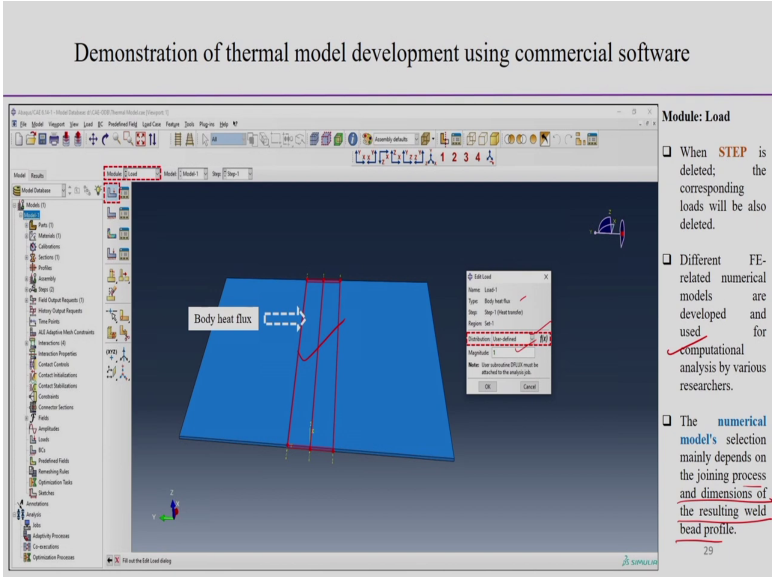
So, then in the if you see the thermal model, choosing the thermal model. So, if you see the thermal model, we can see that types of the selected steps, type of the selected thermal model what kind of the thermal load we can apply? It can be body heat flux that means, the heat flux is over the volume basically. So, throughout this body or some selective zone, we can put the heat flux, then the type of can be surface heat flux.

For example, in particular problem, there is you need to apply only on the surface flux. So, then you can choose the surface heat flux option. And maybe some cases the concentrated heat flux is required for a particular point so, therefore, we can choose the concentrated heat flux also.

Anyway, there are so many options are there that you have to look into this thing click continue and then, select the points where loads should be assigned. So, that means, once we have to continue and then, select the point so, particular points you have to select on the domain when the what amount of the intensity you can apply? In either in the body in the terms of the body heat flux or in terms of the surface heat flux, then the particular point if we select manually and then, apply the, define the what is the value of the flux that can be done also.

And in the next form, define the values of the assign loads and then click, ok. Definitely so, first step is the select the nodes, in this particular node, calculate the value of the heat flux is required apply the particular value of the heat flux and then, next step assign loads and then, click ok.

(Refer Slide Time: 37:00)



Once we do this putting click on in this particular case, we can see that we have defined this particular zone, the body heat flux is applied particular zone. So, then, body heat flux added load also we can see this particular option definition of the user defined magnitude 1 for example, user defined magnitude 1, then we can apply the that body heat flux that in the weld zone.

So, therefore, this particular load module the when the step is detected; the step is deleted, the corresponding loads will also be deleted. So, once we delete the steps, then corresponding loads will also be deleted, but different FE related numerical models are developed and used for the computational analysis by various that is true also, different FE related models.

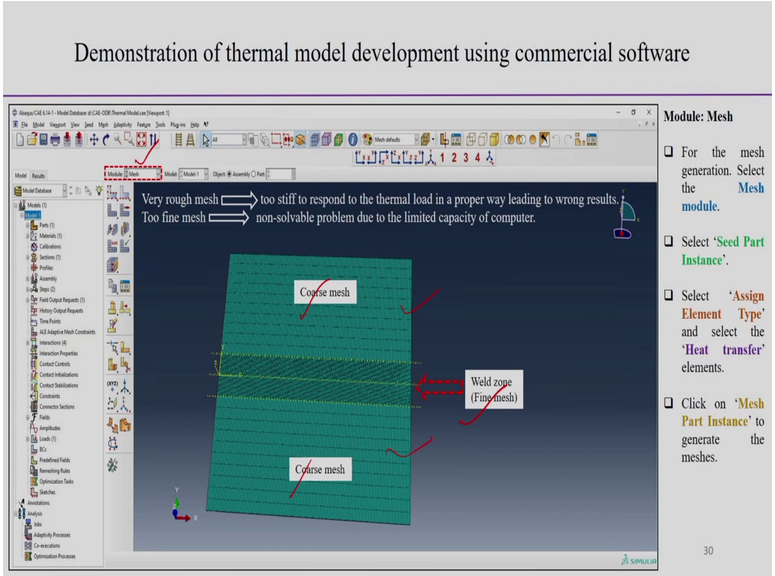
But the numerical model selection mainly depends on the joining processes and dimension of the resulting weld bead profile. So, that means, the point is that there are the manually we can option the user defined the value basically, it is a I think it is not a very good approach.

So, just you can check also that whether this how it works, what kind of results you are getting, but in these cases, you have to define each and every selective point, what is the value of the load flux, then accordingly, we will get the output. So, my point is that rather it is more easier to do particular this model development if you use some sort of the user subroutine.

Because user subroutine is more flexible to apply for the some sort of the volumetric heat flux or surface heat flux whatever is more easy to and that user subroutine can interact with these things, then no need to worry about this thing manually entering the value of the heat flux for a actual welding problem. So, that is the thing.

So, it is necessary to some extra to understand the different kind of the user subroutine and such that you will be able to apply the heat flux through the user subroutine and then, you can run the program also.

(Refer Slide Time: 38:57)



Now, this after application of the load once the load part is done then we look into the mesh that how the mesh can be done. For the mesh generation, select the mesh module. So, we see the module is the here is the module is the mesh module.

So, mesh module is the very rough mesh, the coarse mesh can be done the to stiff to represent to respond to the thermal load in a proper way leading to the a wrong result. So that means, we cannot put very big element so, then we will not be able to capture the change in the temperature or temperature gradient will be difficult or maybe you may not expect very good accurate results in this if we use the coarse mesh.

And other way also if you too much of very small mesh if you consider or very fine that is means size of the elements are very small, it means the very fine mesh if we consider, then it drastically increase the computational time also because in these cases very fine mesh also enhances the number total number of elements, total number of nodes.

So, once there is a total number of nodes increases, then it in it increases the linear system number of the linear system in the equation. So, therefore, then solving the linear system of the equation will be a huge tasks by this computer.

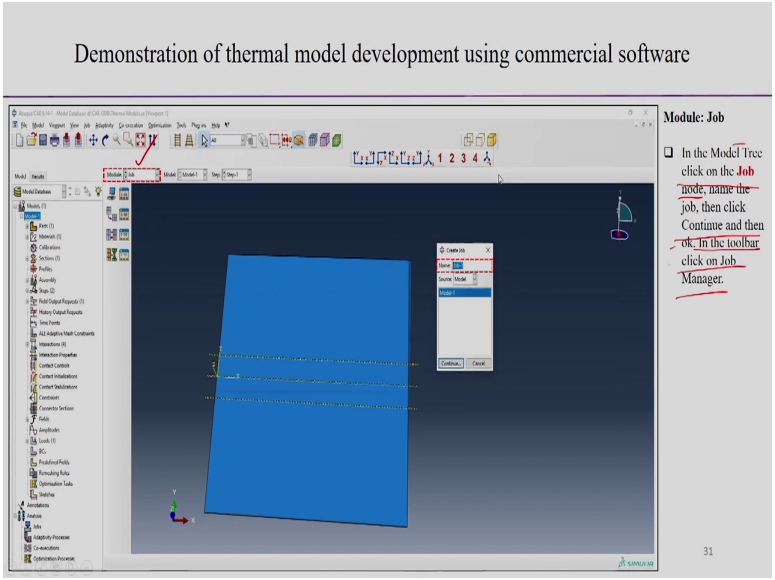
So, there are some compromise may be required or some optimized mesh is mesh is required to get a practical solution of a thermal problem associated with the welding process. So, that is why we have seen different domain also and the in the near about the weld zone, we can use the very fine mesh.

So, then, it is the drastic change in the temperature will be able to capture. If we consider the very fine mesh near the weld zone and that is we know also near the finite sheet is very high, temperature can also variation of the temperature also very high near about the weld zone.

So, therefore, in that part, we can consider the very fine mesh, but remaining part we can consider the coarse mesh. Then in that cases we optimize the total number of elements and nodes in this particular problem. Now, once for the generation of the mess, then select the seed part instance, then select assign element type, these are the steps basically select assign element type and the select the heat transfer elements in this case and click on the mesh part instance to generate the meshes.

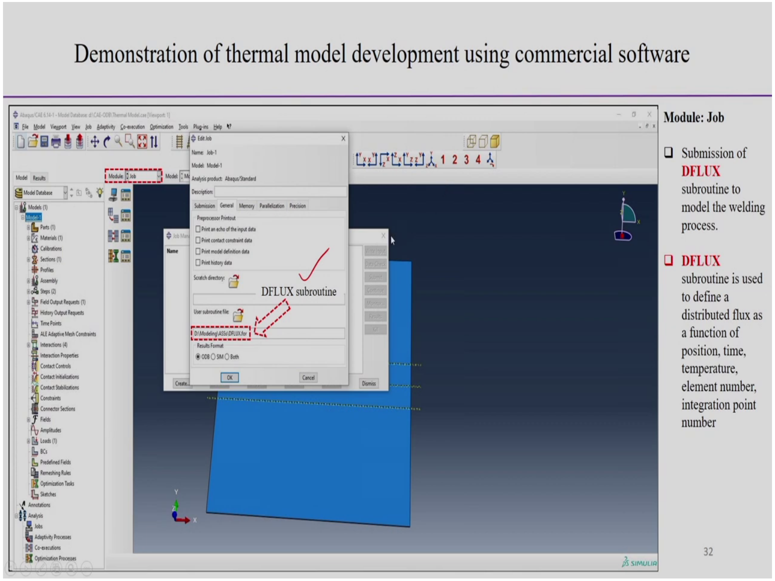
So, here I am not explicitly describe how automatic meshing all these thing meshing can be done all these thing that is not required at all, but looking more on the analysis type by choosing the particular governing equation. So, therefore, this we if we follow all these particular steps and if you see the fine mesh as well as the coarse mesh all the different zone, we can provide, we can meshing the or may be discretize the solution geometry.

(Refer Slide Time: 41:46)



So, once it is done, then next task is the basically job. So, job is this module is the job after mesh, the module is the job. So, job in the model tree click on the job node. So, in the model tree, there you can click on the job node and from there, then click continue and then ok so, continue ok in the toolbar, click on the job manager. See here, if you see name job module 1, we continue and ok and then, we can look into the job manager also.

(Refer Slide Time: 42:18)



So, in that case, we can see the job manager by submission of the in the job manager, there is a option also that already discussed that we can apply the heat flux, or the may be load through the DFLUX subroutine also. Here, there is a at this point when you are going to the module job, there is a option to look into the application of the user subroutine that is called the DFLUX subroutine.

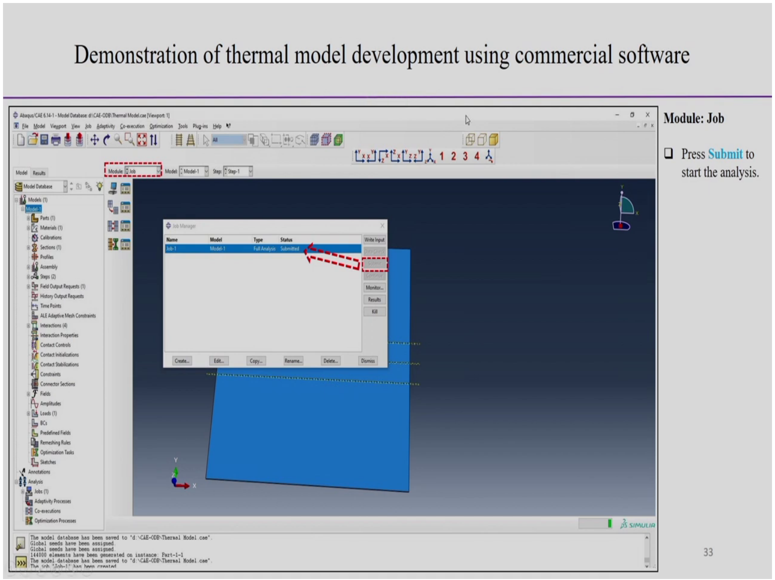
So, here, the modeling, here you can see the DFLUX subroutine can be incorporated here in the different way. So, that actually give you the user subroutine can interact during this thermal calculation.

So, the here we can put that module job, the interaction of the DFLUX subroutine. So, submission of the DFLUX once with the summation of the DFLUX subroutine to model the welding process DFLUX subroutine, but we have to before that we have to be ensure writing the small program in the DFLUX subroutine.

Then, DFLUX subroutine is used to define the distributed heat flux which is a function of position, time all can variation can be done stand even for temperature, element number, integration point number all kind of variation is there to define the DFLUX subroutine.

So, we have lots of flexibility actually to define the according to the problem, the kind of heat flux required in this particular any kind of problem. So, that is that kind of flexibility is there to define in user DFLUX user subroutine.

(Refer Slide Time: 43:50)

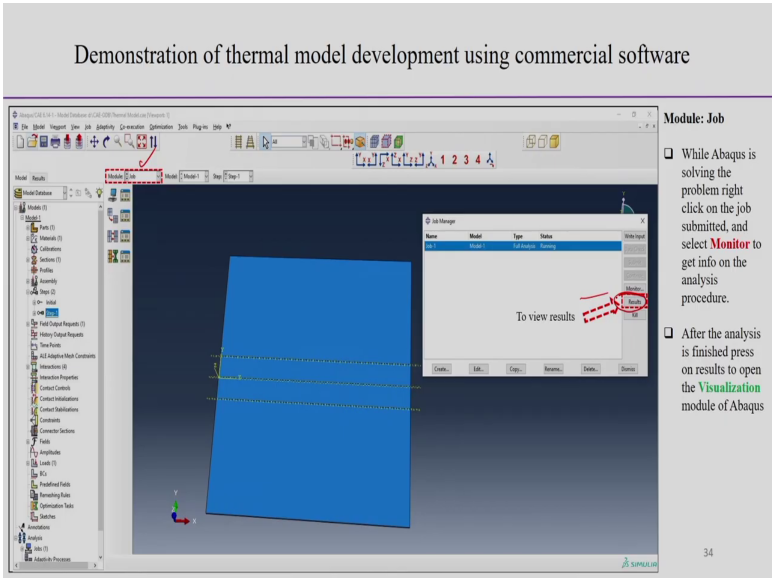


So, once it is done, job is done, then we can then submit the job. So, once we submit the job, press submit the job, then it start analysis the process starts. So, analysis means that then program will basically it will run the this particular problem and some finite time is required to get the results, results means the in the form of a temperature distribution in case of the thermal analysis.

So, once job submitted, then wait for some time to get the results or maybe in between you if you can see the what is the progress of the result. That means, if it is a moving heat source problem so, it starts from one position and moves on particular position.

So, then at which position heat source are there and then, accordingly, we can see the results in between, but you will be getting the actual all the results after completing of the total number of the time steps we have defined. After that time step, you will be getting the actual results. So, they have to wait till completion of the all the steps for this particular analysis.

(Refer Slide Time: 44:52)



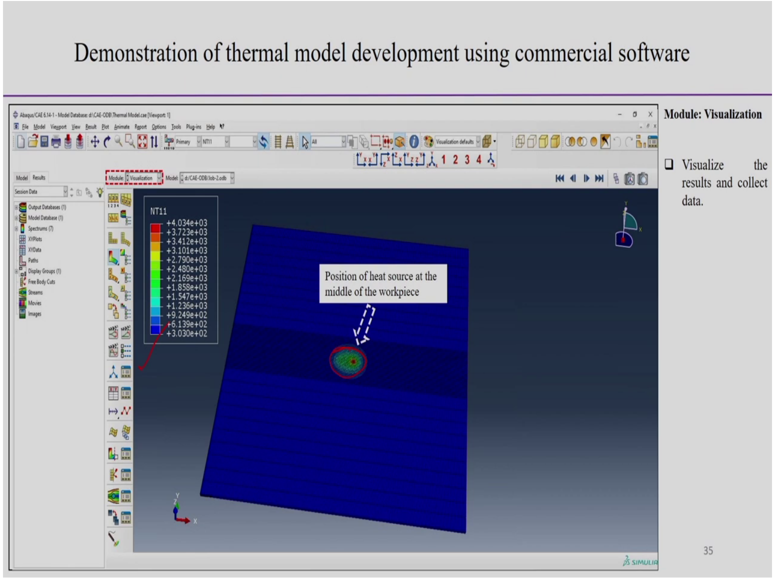
So, once it is done, then once results done, then we can see the visualization of the results. So, that means, that is also under the module job. So, under the module job, we can see that while Abaqus is solving the problem, right click on the job submitted, select the monitor to get information onto the analysis procedure.

So, even if you select the monitor also, see job manager even if you look into the select the monitor also. Then we can get the information of the analysis procedure it means that suppose one particular steps, what are the whether the solution is converging on particular steps or not or the solution is diverging or how many number of steps are required to get the converse solution.

All kind of the information basically if you during the running of the program to select the monitor, you can see all kind of the information. So, then, once the analysis is done, after the analysis is finished, press on the results to open the visualization model of the Abaqus.

So, once the analysis is done to see the results and then here you can see to view the results, if you select this particular option results. And then, there you can visualize the different kind of the results for this particular case.

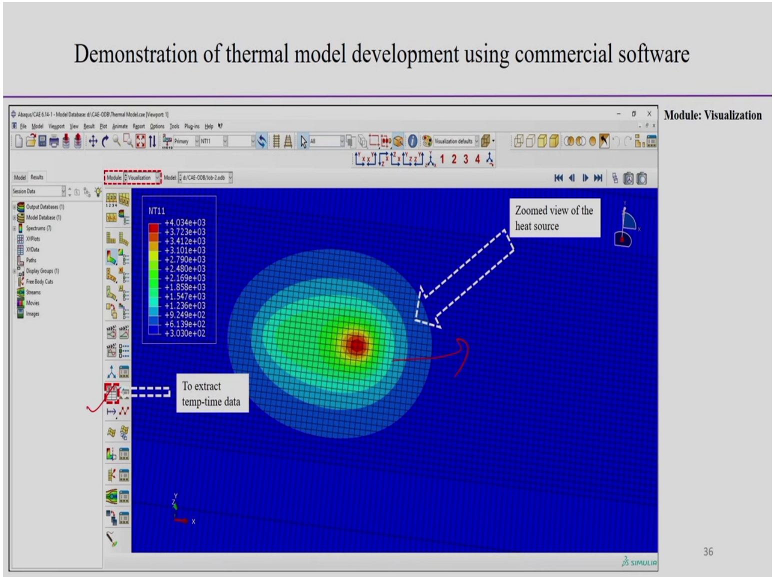
(Refer Slide Time: 46:01)



Now, here you can see some sort of visualization of the results and collection of the particular data once we finish the analysis process. So, here you can see the highlighted part, the temperature distribution we can see and in the welding process also, once it is moving one particular direction, the it is the consequently heating on the front part and at the same time there is subsequently, there are solidification also happens and gradually the cooling process also happened behind the weld pool.

So, that is way we can get the temperature distribution at any instant of time and then, we can get the color bar also, color bar indicates the distribution of the temperature for corresponding isotherm from here we can define. Then which is the molten zone, what is the heat affected zone just looking at the temperature. So, we can expect the visualization of the result in this particular form related to the welding process.

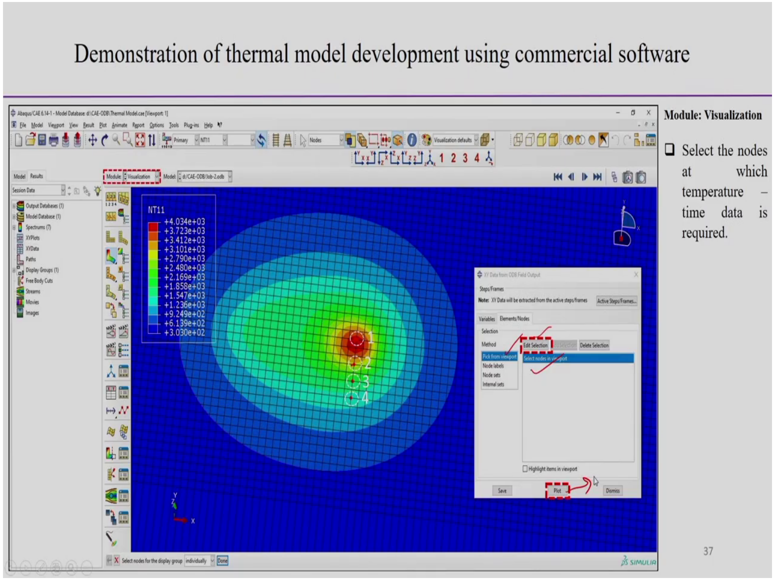
(Refer Slide Time: 46:52)



Now, you can look into zoom view also this thing. Here, you can see that we can choose the module is the visualization module. So, now after the job, then we can look into the visualization module if we follow, I can zoom view. You see this is the weld pool this is the typical weld pool profile we can get from the thermal analysis using the software.

And now, to extract the time temperature data, we can put this particular, select this particular part, there we will be able to extract what is the time versus temperature data. But how we can look into time versus temperature data? Let us look into that.

(Refer Slide Time: 47:27)

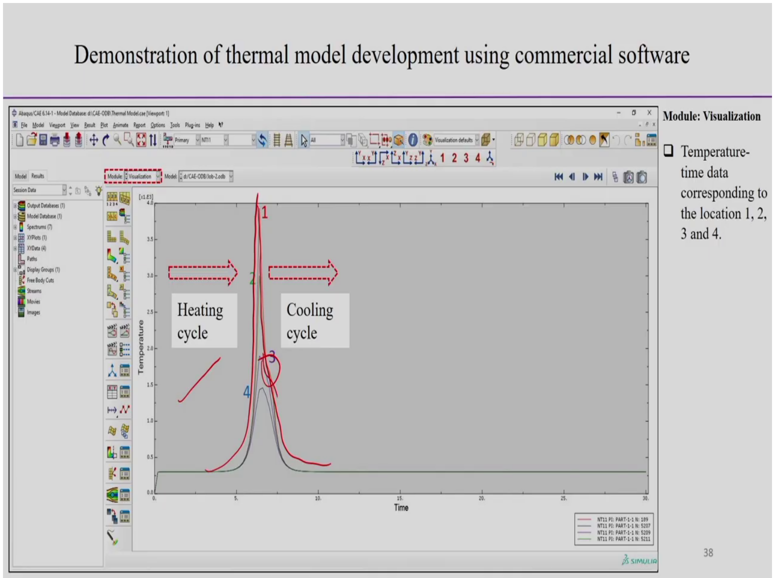


To select the nodes at which temperature time data is required for example, I can select the node for example, I want to select this particular node because the software having all the data point is the not at the particular node point. Even if you want to get the data and the in between the particular node point in the in between the node so, then it actually interpolate the data and we will be getting the results accordingly.

So, now, if we select this particular node for example, 1, 2, 3, 4 we want to know what is the value of time versus temperature for this selected nodes. Then once we select the node and the we have to select the step if you see that select the nodes at which the temperature time data is required.

So, edit pick from the viewport here, you can see the edit selection select nodes in the viewpoint and select nodes with the viewport. So, here we can see the select nodes at the viewports and then which is the plot.

(Refer Slide Time: 48:33)



So, once we select the viewport say 1, 2, 3, 4, the select this particular point, then plot this data, then we will be able to get that particular this time versus temperature data from this analysis. You see the point 1, this is the particular graph point 1, point 2 green color and point 3 and point 4 the different color we are getting the different kind of the profile.

So, it means that the heating cycle is gradually increasing the temperature, then gradually decreasing that is the cooling cycle during this and we can see there is a some bump is there also so, that may be if we incorporate the temperature dependent properties and may be latent heat of melting a solidification.

Then we can get some kind of the bump that we already explained this thing that because at the between the solidus and liquidus temperature, there may be some sort of the phase transformation happening. So, that is why we cannot get exactly the smooth profile in this particular zone.

So, that is way so, once we get the we can visualize the data that two-dimensional, then we can capture the two-dimensional or three-dimensional view along temperature distribution. Apart from that, if you want to time extract the time data for the data that is also possible to from the module visualization. So, that is way, we can get the results from the thermal analysis.

Now, this is the end of the module 4. I hope this will be helpful to start using some sort of the commercial software Finite element based commercial software to develop the process model associated with the welding process. This was the just a sample, presentation associated with only the thermal analysis. I hope it will be helpful for you.

So, thank you very much for your kind attention.