Computer Aided Engineering (CAE)

Tutorials

For Fourth Year Students

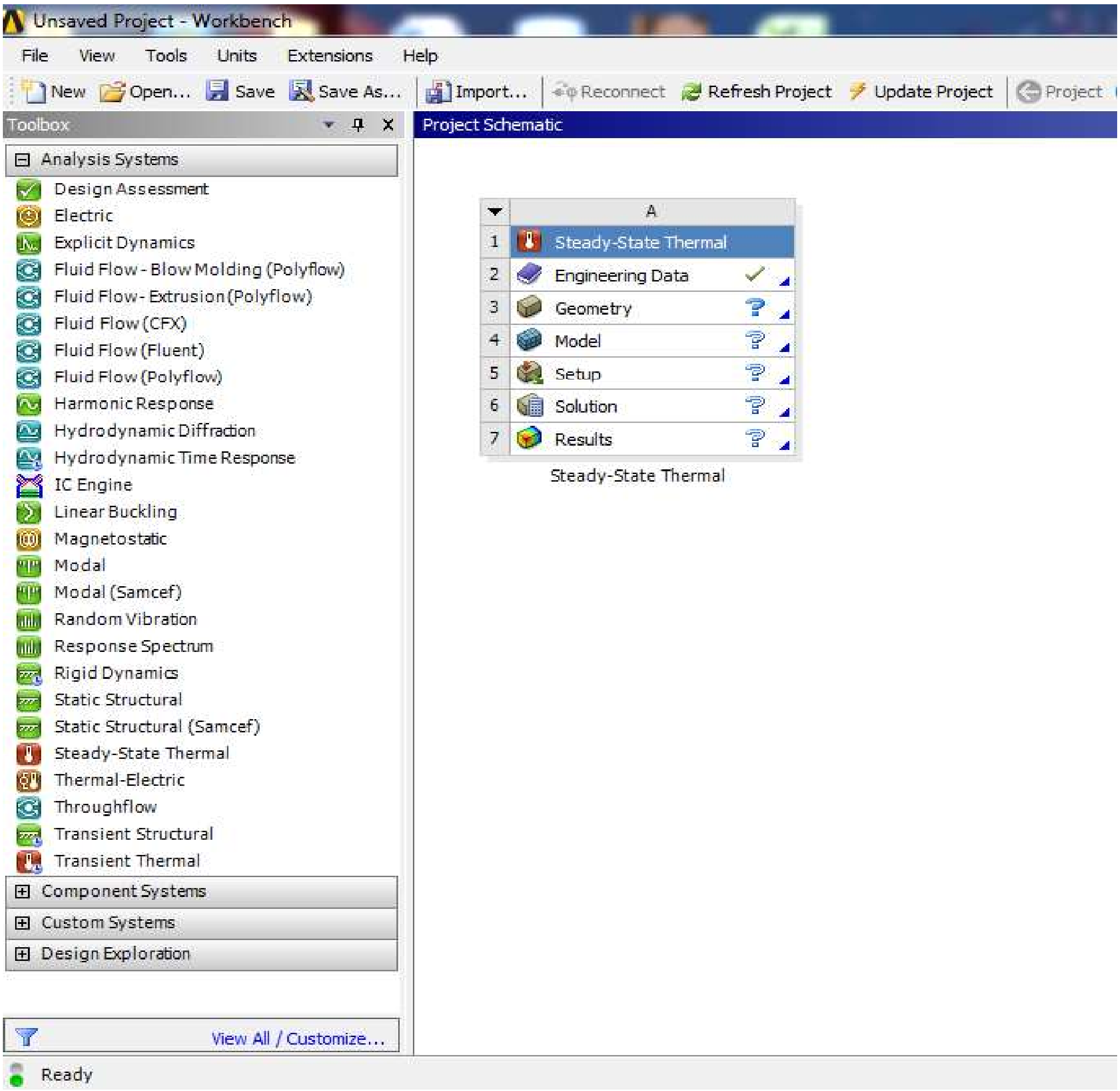
Mechanical engineering Department

2017–2018

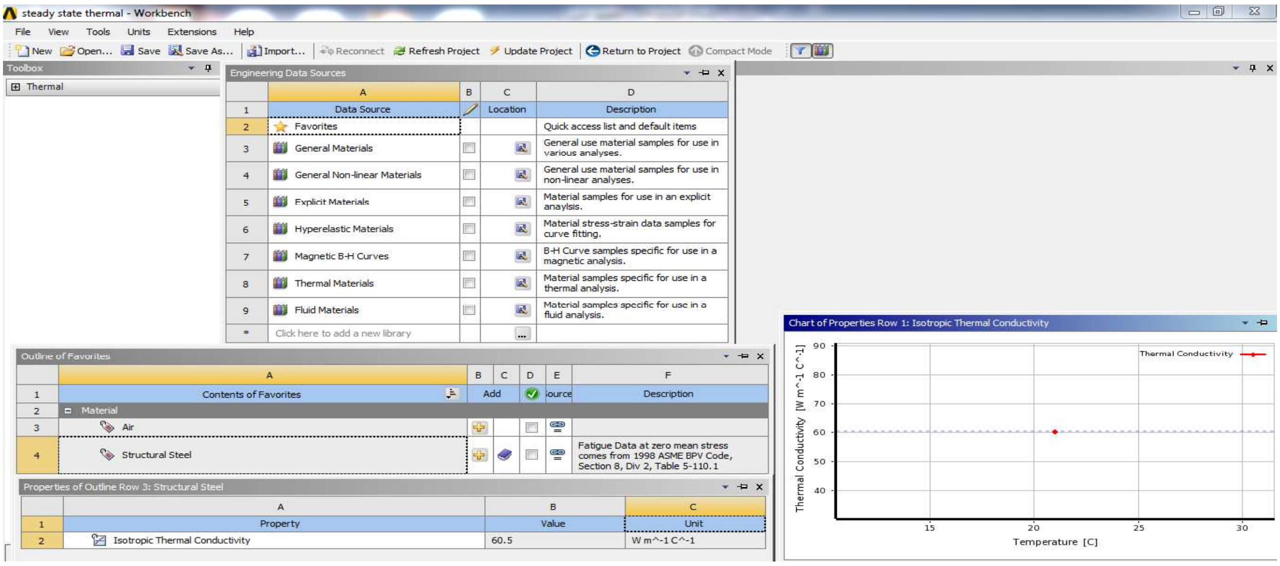
Tutorial One

Steady State Thermal

Select *(Steady State Thermal)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping the system on the workplace as shown in the following figure:

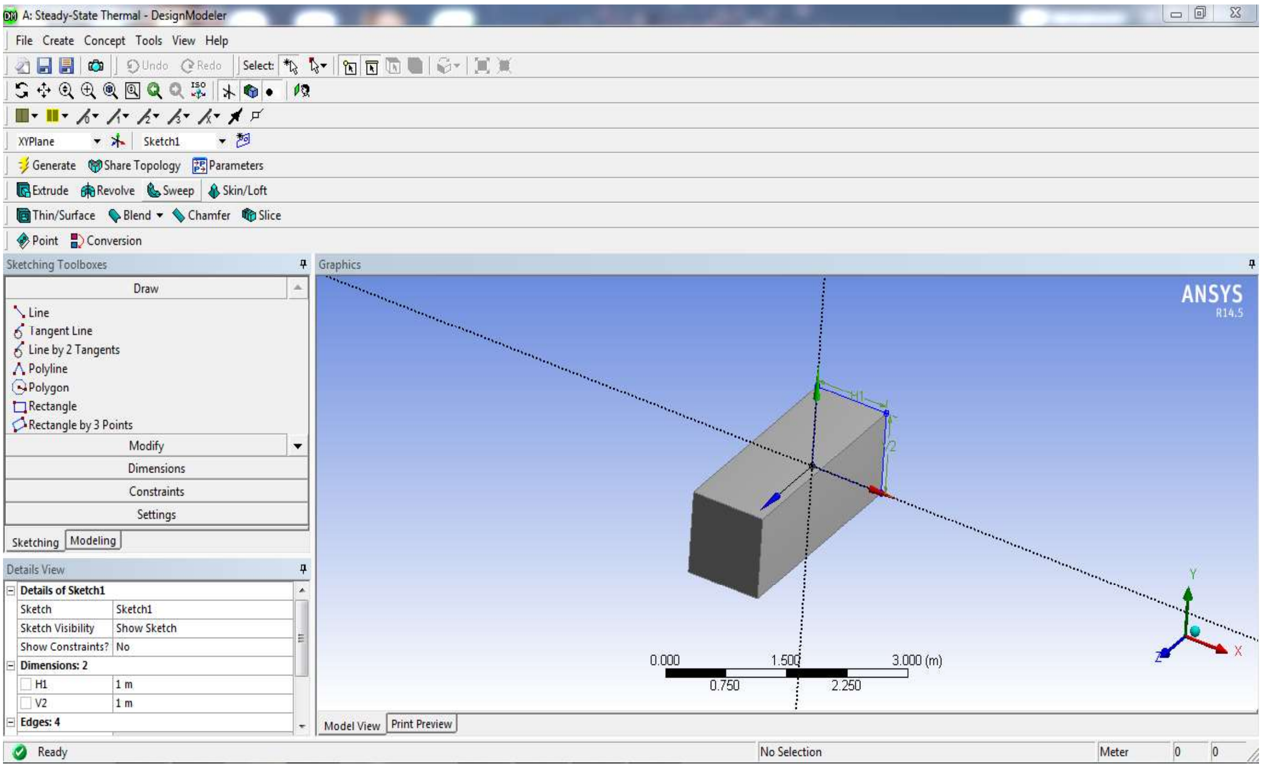


The next step is to choose material for the engineering test from the materials library. Double clicking on the  in the project that just was created spawns a new window. There you can choose the material to be tested, as shown in the following figure:

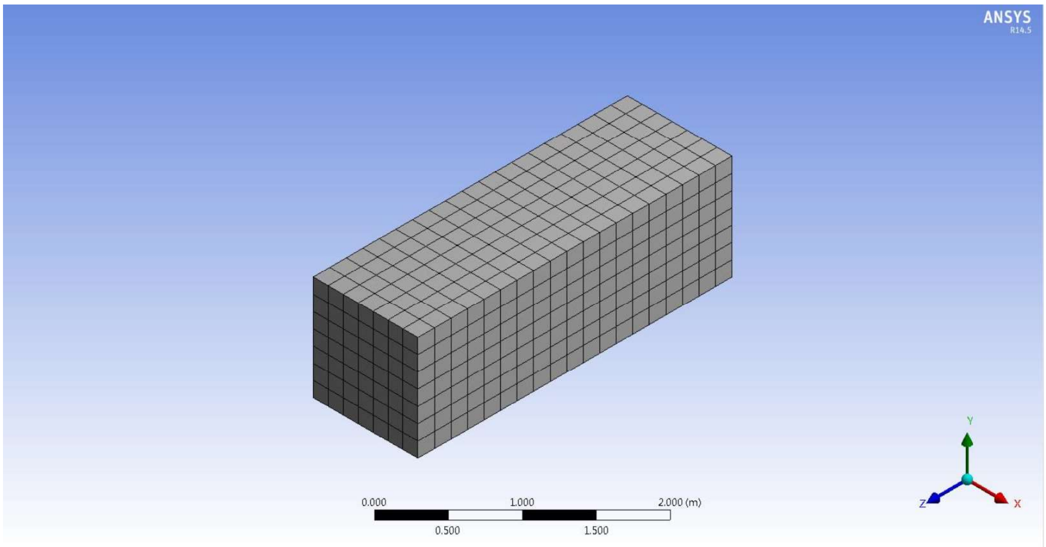


However, if the material was not found within the library it can be added by clicking on (*add material*) and entering the properties of the new material in a new table for each property.

The next step is to design the model in the form of a parallelogram in dimensions (*1× 1×3 m*) by using (*Design Modular*). The final model is shown in the following figure:

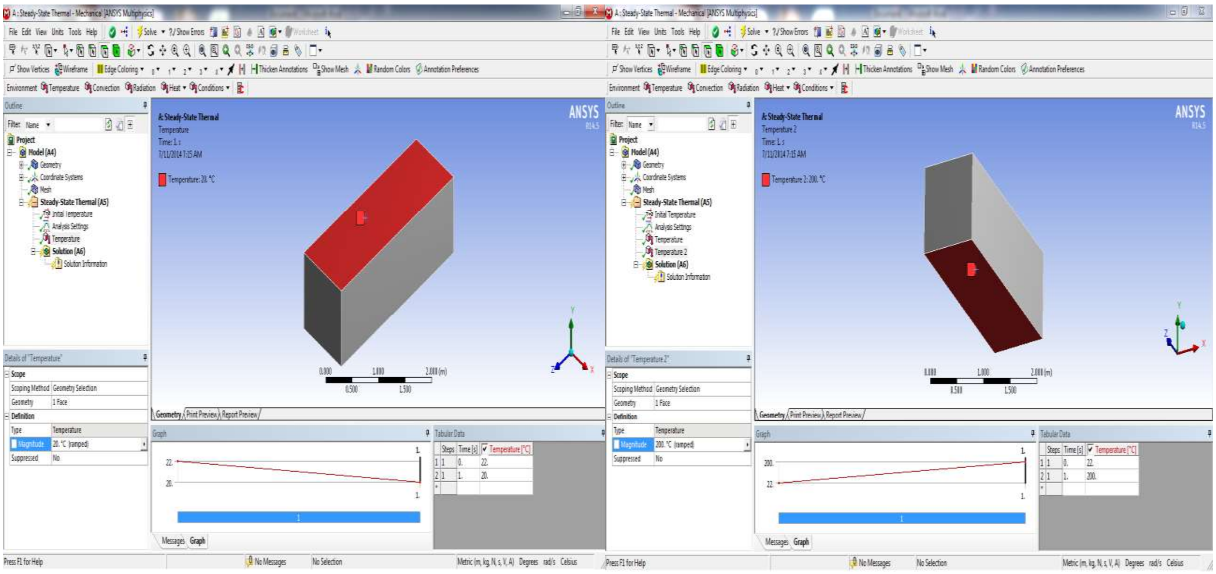


The next step is to create the mesh, the mesh type and size of the cell can be controlled to suit the situation to be solved, and as shown in the following figure:



Next, we apply thermal loads (temperature, convection heat transfer, radiation heat transfer etc.) as shown figure below:

In this example the loads are temperatures on the top and bottom surfaces, the temperature adjusted to the top cold surface is (*20 C*) and the bottom hot surface is (*200 C*), the other surfaces are insulated.

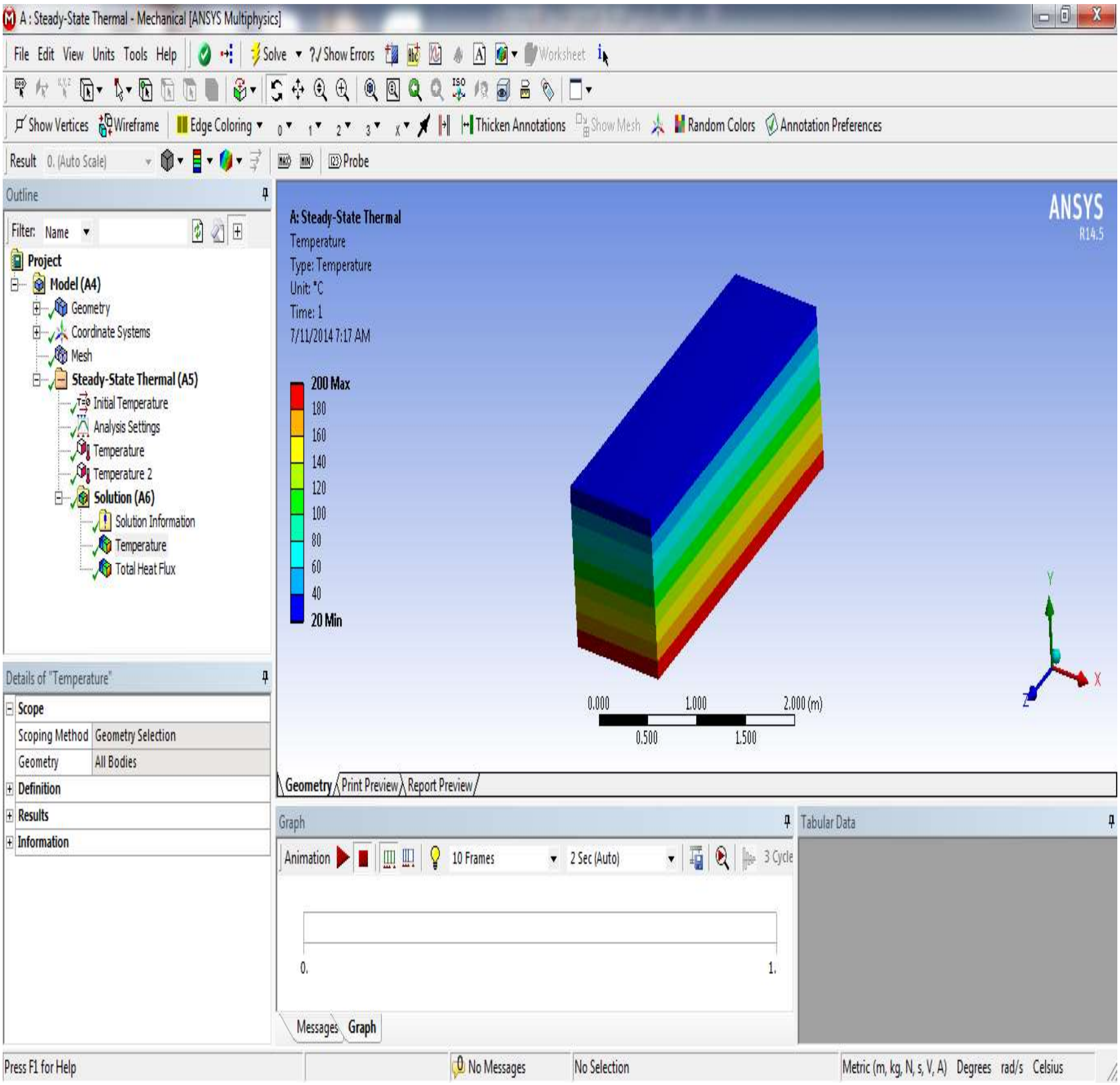


After setting boundary conditions, the case will be solved by clicking on (*Solve*). The unknowns will be calculated, which are the temperatures for the current application. Only energy equation will be used in this problem.

After that is the solution we can accept the results to be reviewed and that

can be temperature, heat flux etc as shown below:

Where you can choose any of the results to be reviewed, the following figure shows the review of the temperature distribution of the current example:



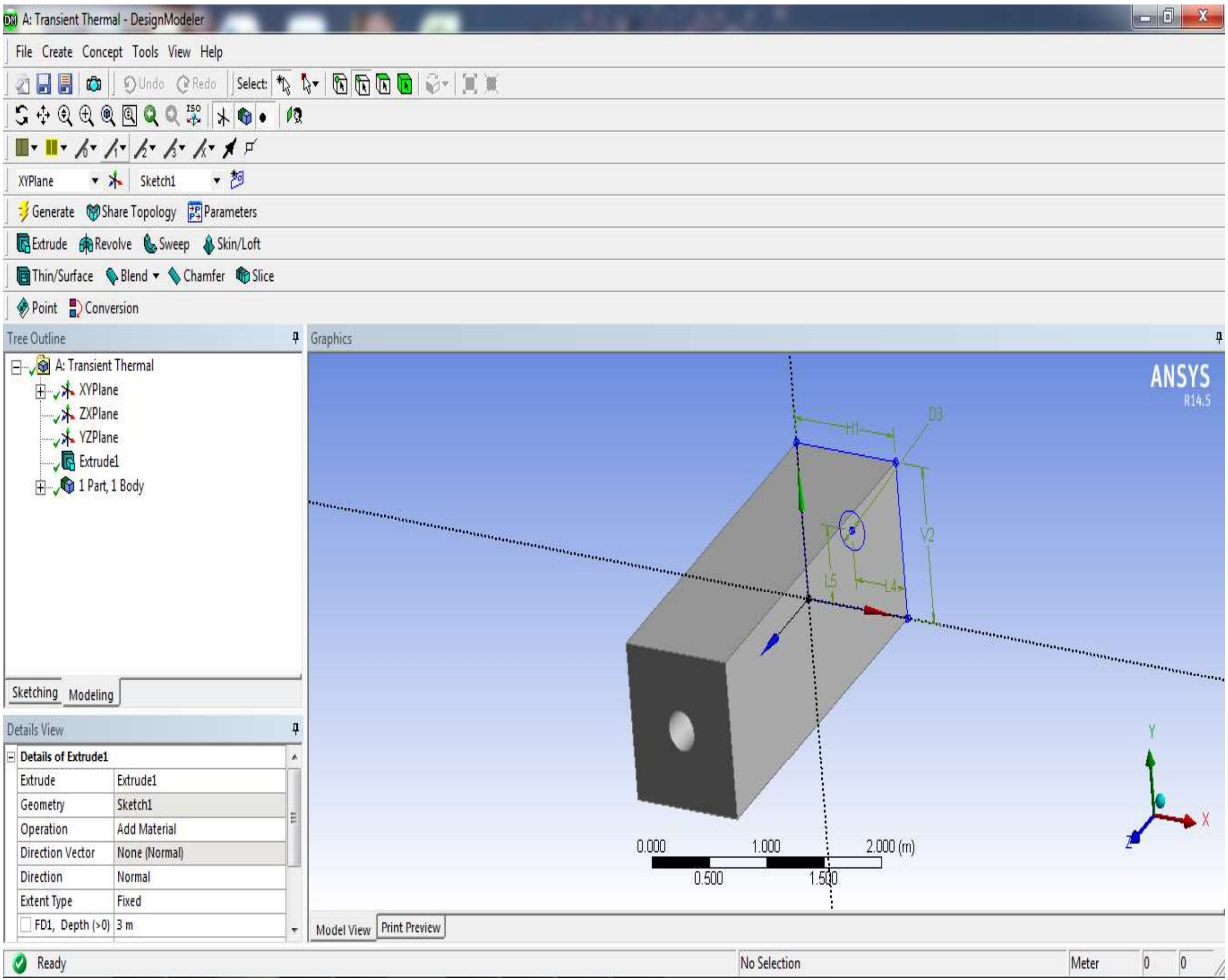
As shown in the video and our book.

Tutorial Two

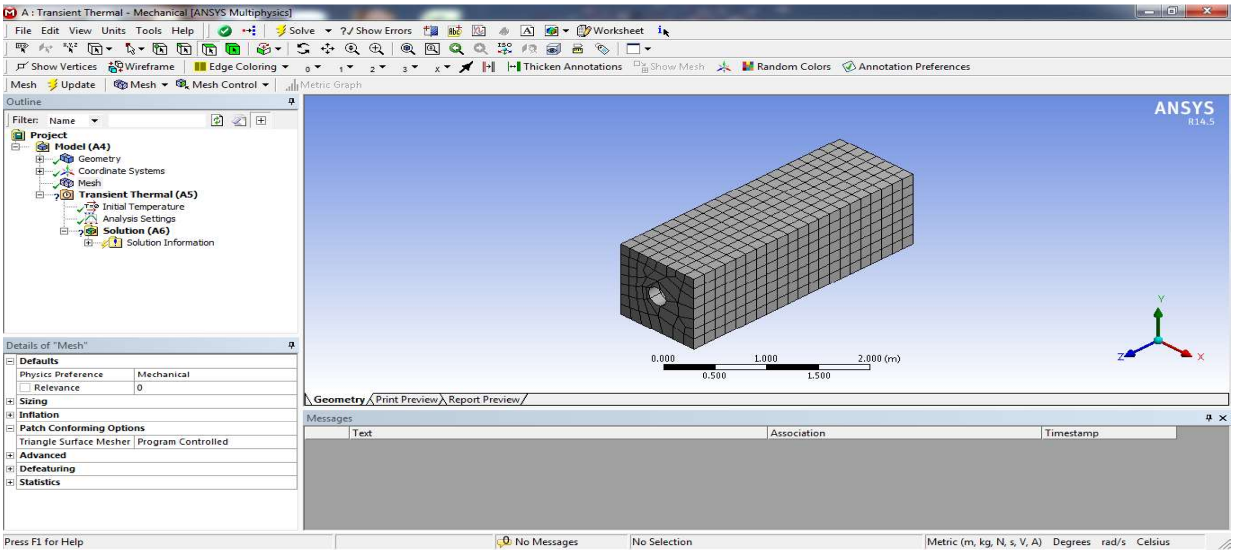
Transient Thermal

Selected Analysis System *(Transient Thermal)* from the main menu of *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

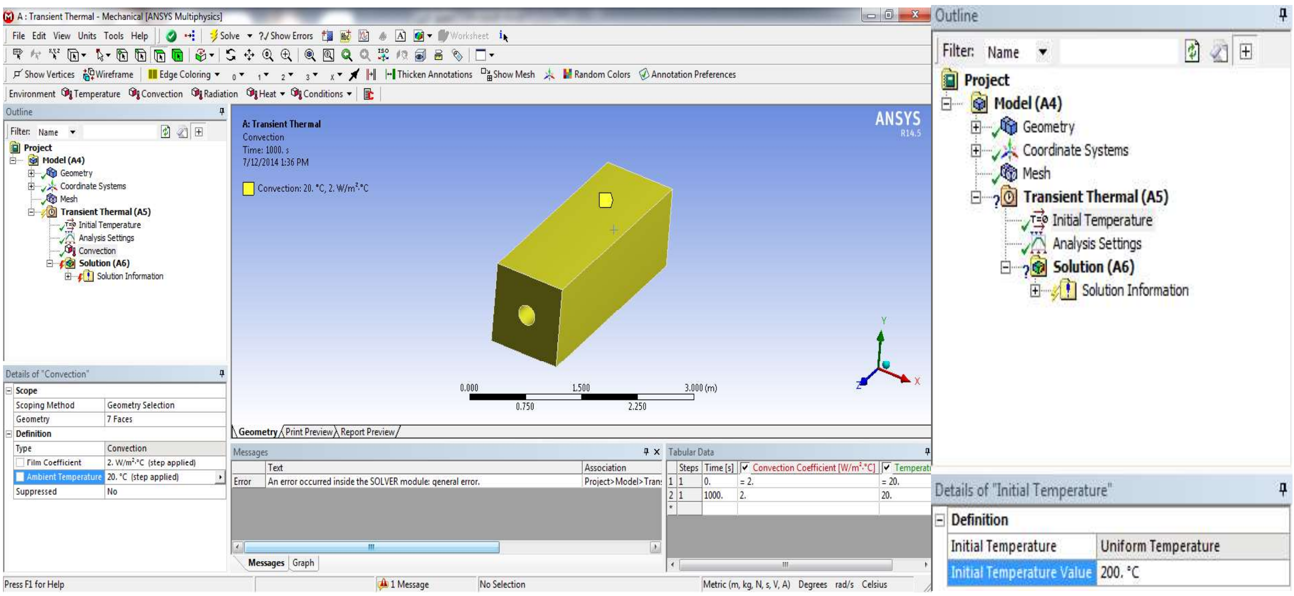
Then design model in the form of a holed parallelogram in dimensions (*1× 1 ×3 m*) and the hole diameter is (*m0.25*) by using (*Design Modular)*as shown in the following figure:



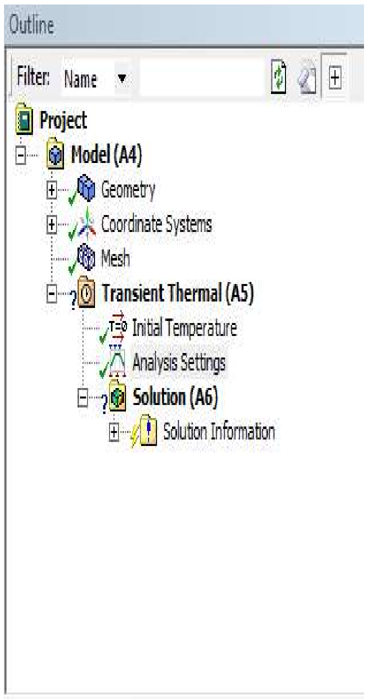
Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved, and as shown in the following figure:

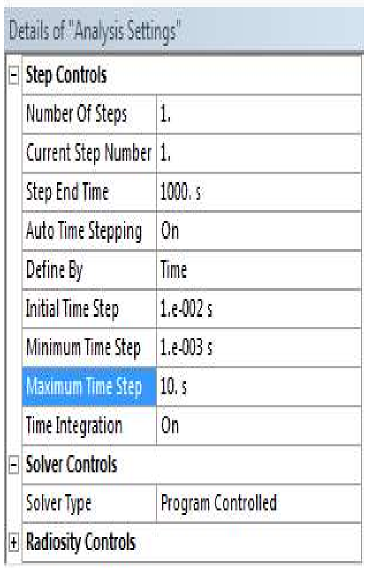


After making the mesh, loads are applied: The loads in this application are thermal loads (temperature, convection heat transfer, radiation heat transfer etc). In the present example has been set status as a loss of temperature to the surrounding from external surfaces of body where the heat transfer coefficient (*2 W / mm. C*) either ambient temperature was (*20 C*) and initial temperature was (*200 C*).



In such systems, the transition, the time enters influential factor as the case be variable with time in the current system, the body temperature is not constant relative to time, but are variable with time, so it must adjust the total time for the transition, as well as the greatest time of the step as shown in the following figure:

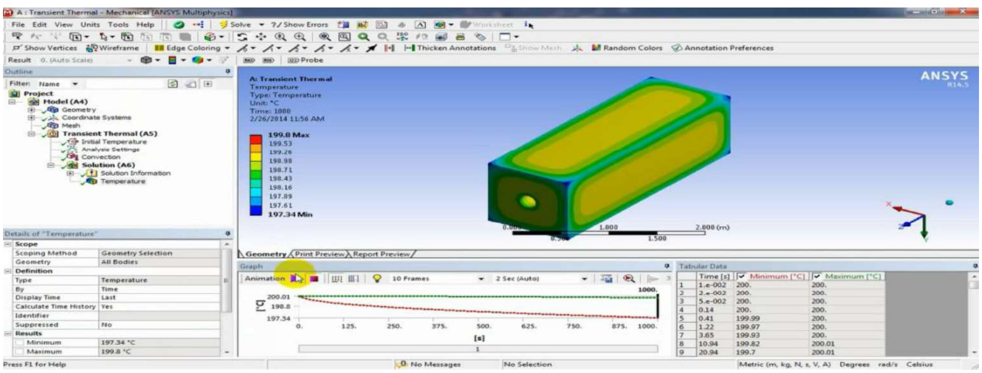




After adjusting marginal circumstances where the case is resolved to give it a solution (*Solve*) then the solution will be unknowns to be calculated, which is the temperature change with time for the current application, either equations that covers this application are the energy equation.

After that is the solution we can accept the results to be reviewed and that can be temperature, heat flux or other.

Where you can choose any of the results to be reviewed, the figure below shows the review of the temperature distribution of the current example.



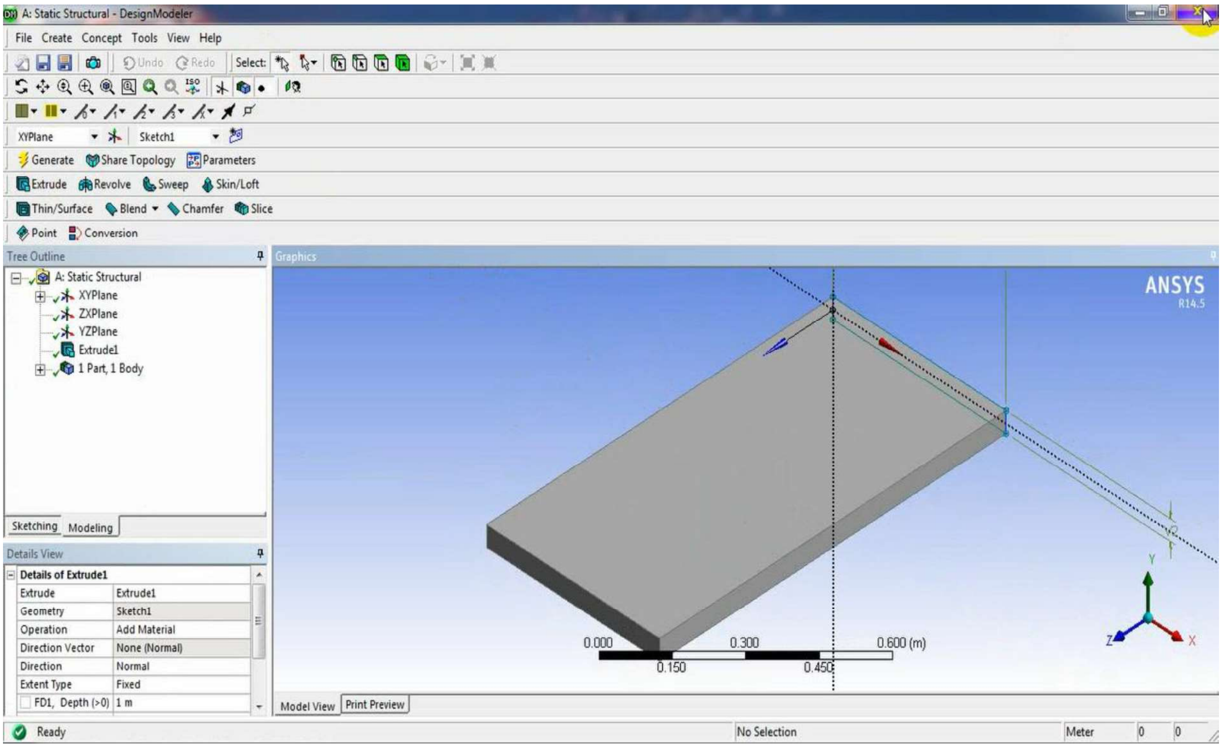
As shown in the video and our book.

Tutorial Three

Static Structure

Select Analysis System *(Static Structure)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

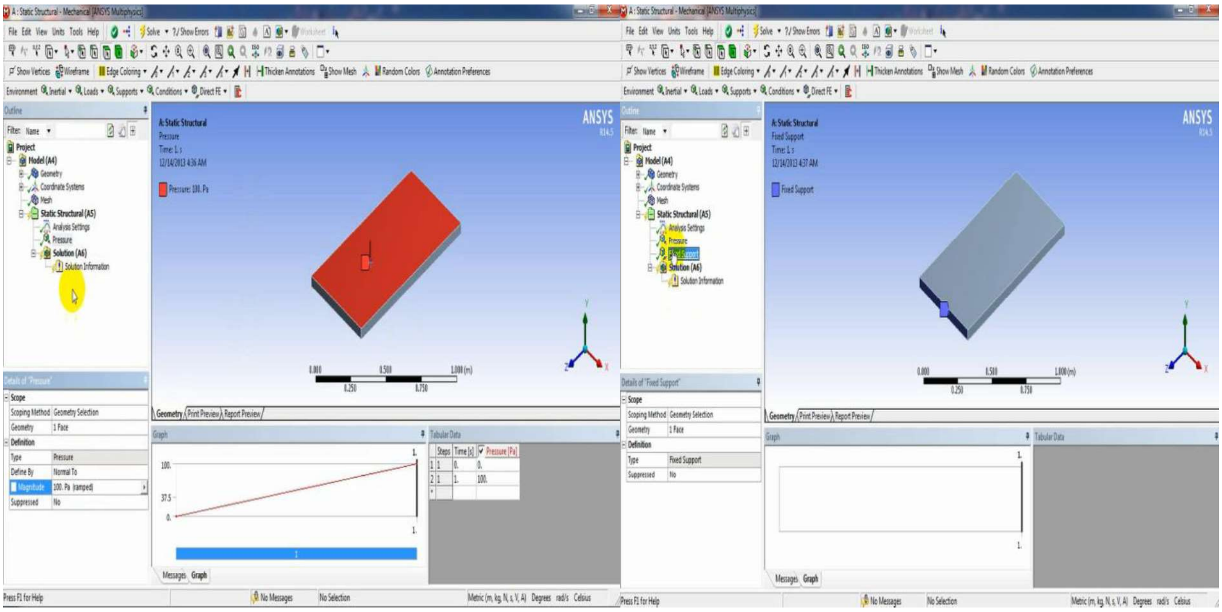
Then design model in the form of a plate in dimensions ($1 \times 0.5 \times 0.05$ m) by using (*Design Modular*) as is shown in figure follow:



Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved.

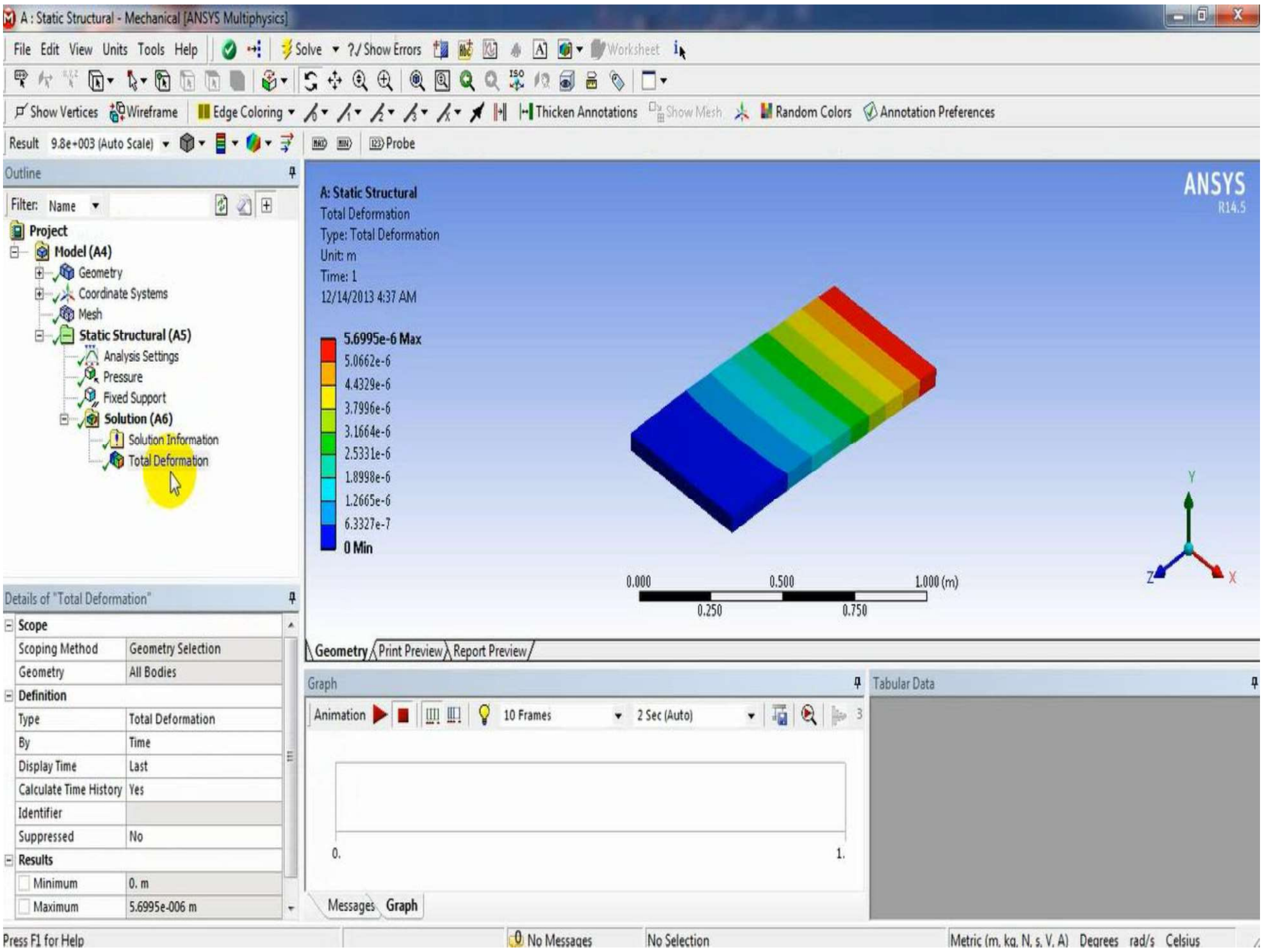
After make the mesh are applied loads where the loads in this application are (force, torque, pressure etc) as shown in bar follow:

In the present example the load was a pressure on the upper surface in magnitiude (*1000 pa*), and the body is support from one of those end as the following figures, which represent the boundary conditions of the body:



After adjusting boundary conditions where the case is resolved to give it a solution (*Solve*) then the solution will be for the unknowns to be computed which is displacement resulting from the applied loads on the body in three direction for the current application.

After that is the solution we can accept the results to be reviewed and that can be deformation , stress , strain etc. the following figure shows the results of the current deformation for present example:



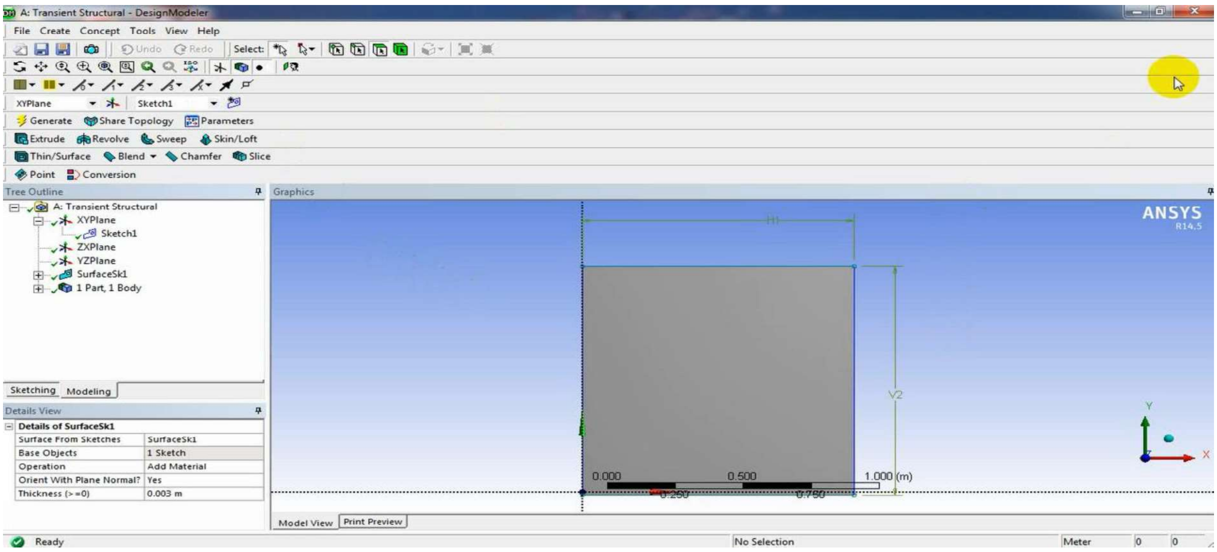
As shown in the video and our book.

Tutorial Four

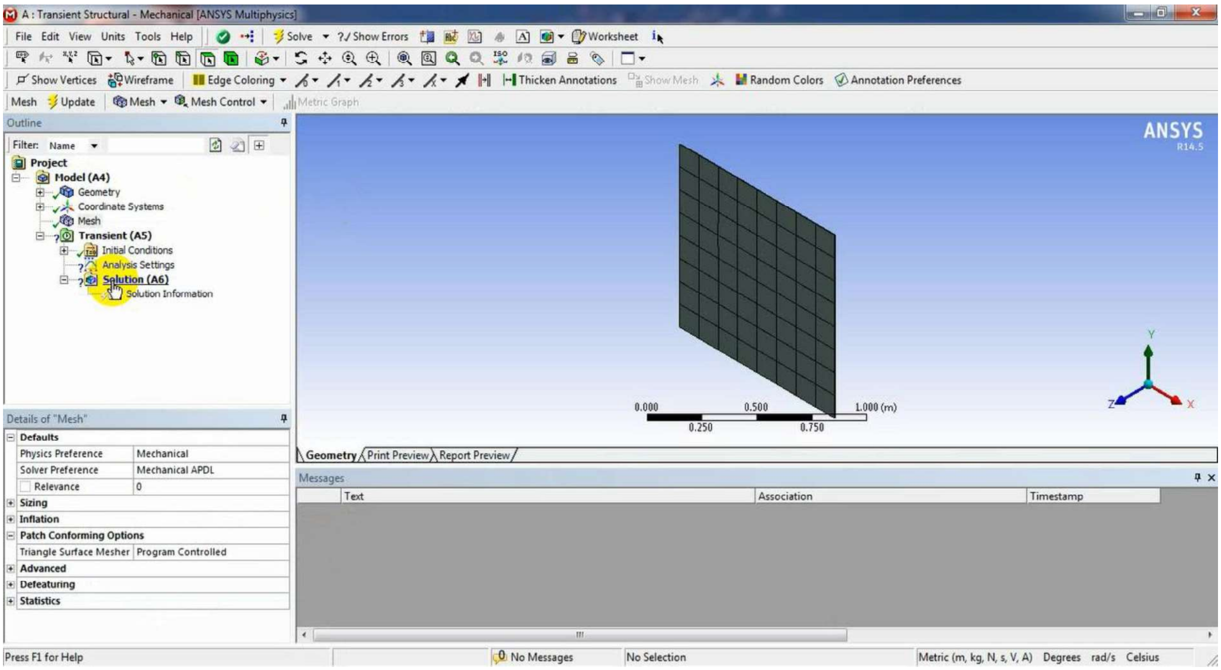
Transient Structure

Select Analysis System *(Transient Structure)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

Then design model by drive surface from sketch in dimensions (*1×1 m*) and thickness (*0.003 mm*) by using (*Design Modular*) as shown in figure bellow:



Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved as shown in figure bellow:



After making the mesh, loads are applied where the loads in this

application are (force, torque, pressure etc) as shown in bar follow:

Since the system was transient a time enters influential factor, the loads inflicted on the body are variable with time, so it must adjust the total time for the transition, as well as the greatest time of the step.

After adjusting boundary conditions where the case is resolved to give it a solution (*Solve*) then the solution will be unknowns to be computed which is displacement resulting from the applied loads on the body in three direction for the current application.

After that is the solution we can accept the results to be reviewed and that can be deformation , stress , strain etc.

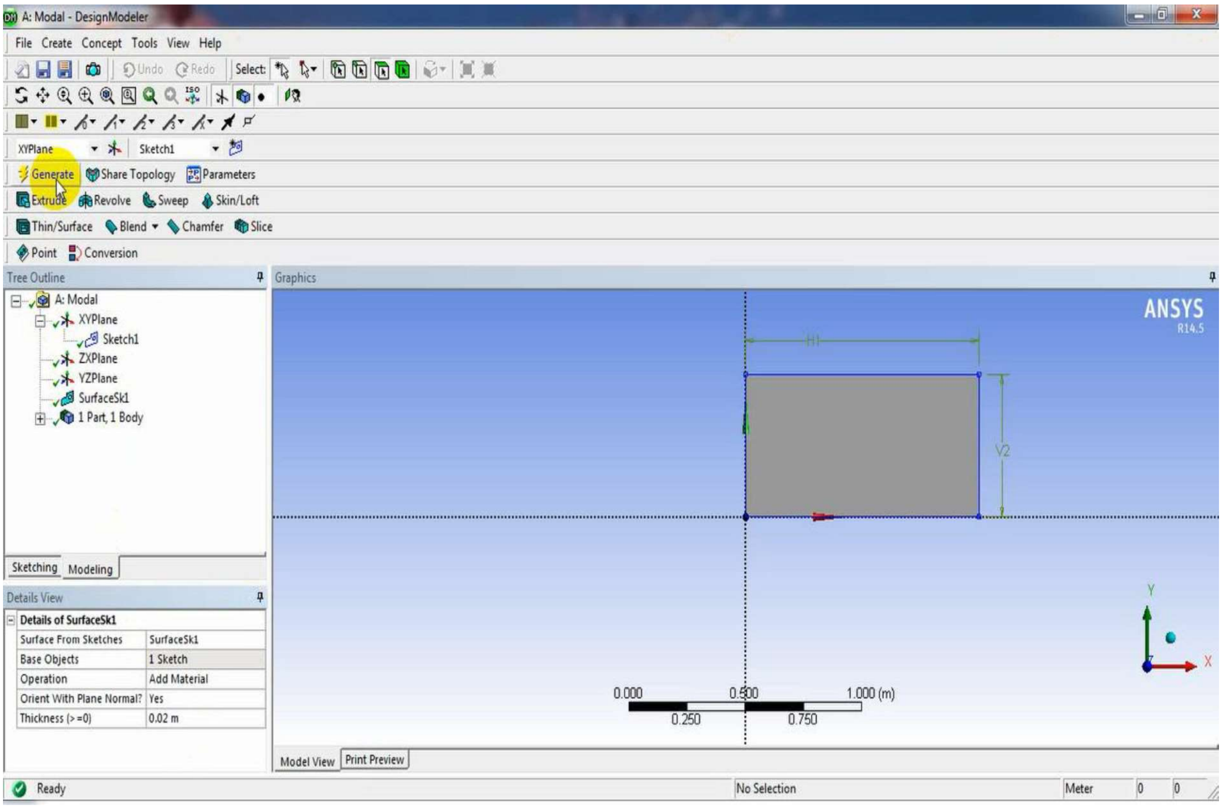
As shown in the video and our book.

Tutorial Five

Model

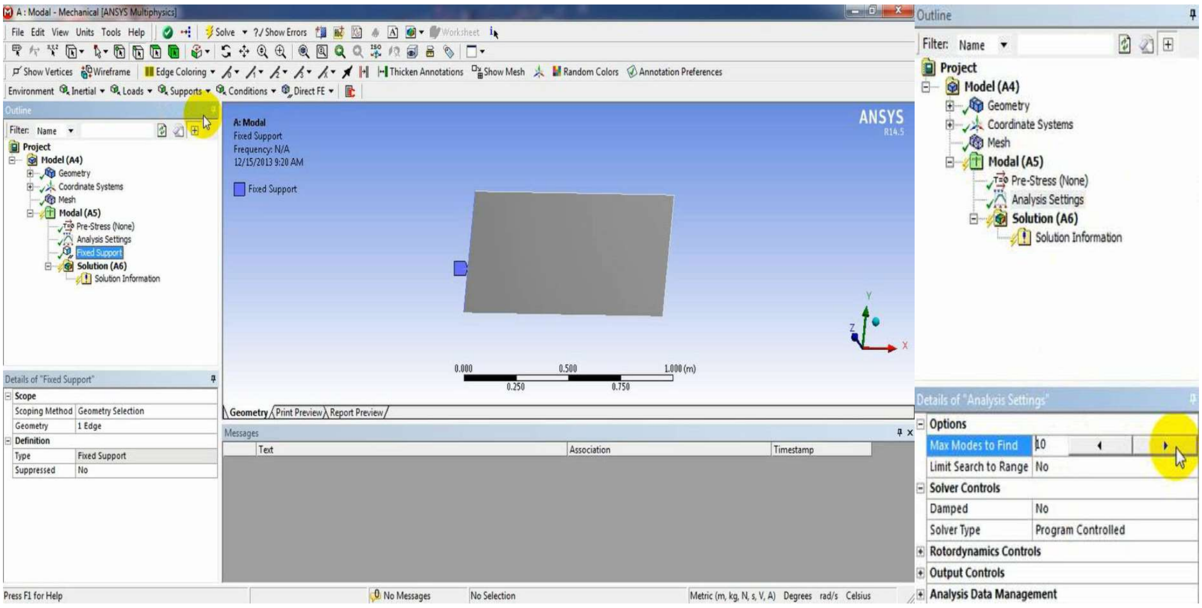
Select Analysis System *(Model)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

Then design model in the form of rectangular surface in dimensions (*1×0.5 m*) and thickness (*0.02 m*) by using (*Design Modular*) as shown in figure bellow:



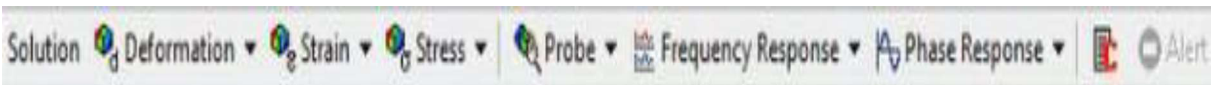
Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved.

After make the mesh a body is support from one of those ends and adjusts the number of models (*6*) to (*10*) model to see more of free vibrations that the body can vibrate as shown in the following:

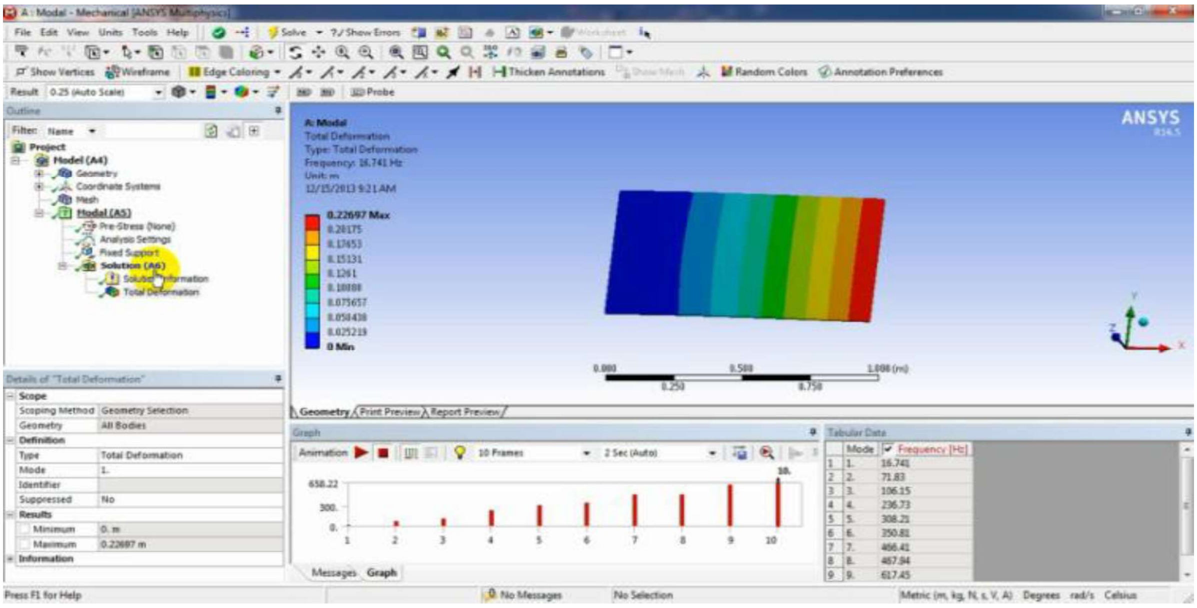


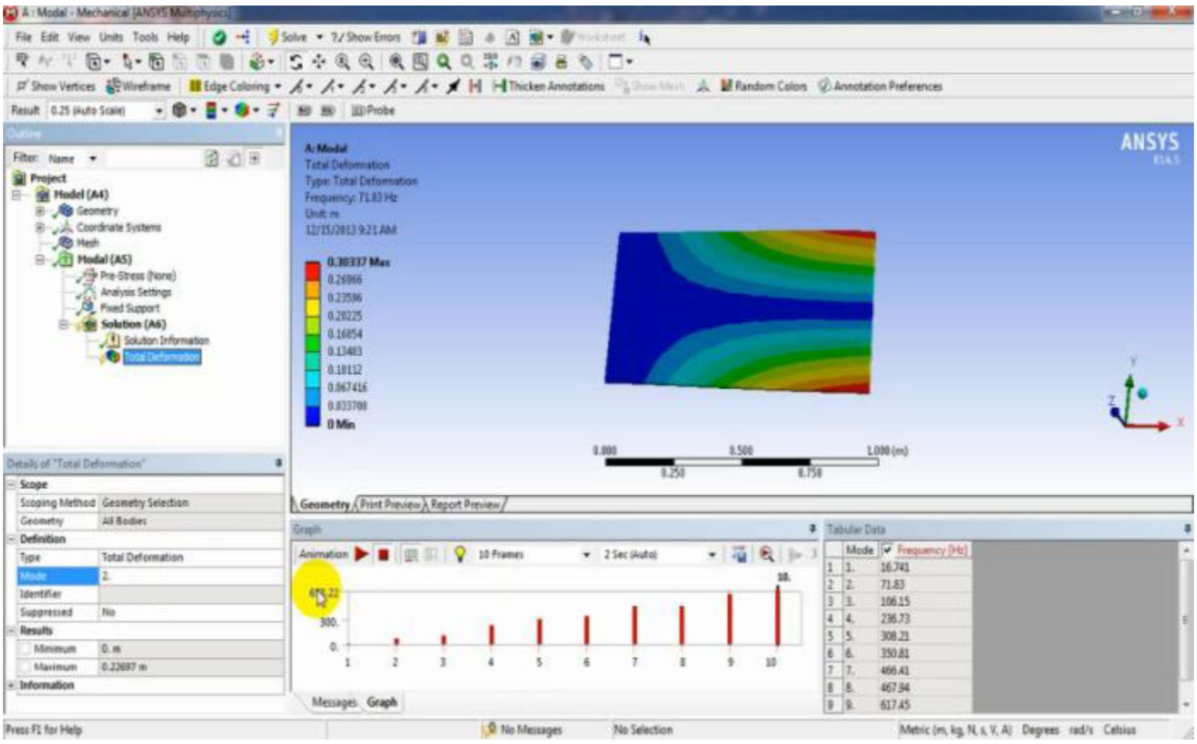
Then the case is resolved, where we give it a solution (Solve) then the solution will be unknowns to be computed which are free vibrations (ie, without applying any external force) for the current application.

After that is the solution. we can accept the results to be reviewed and that can be deformation, stress strain etc, as shown in bar below:



For example, the following figures are showing the deformation of the first and second model :





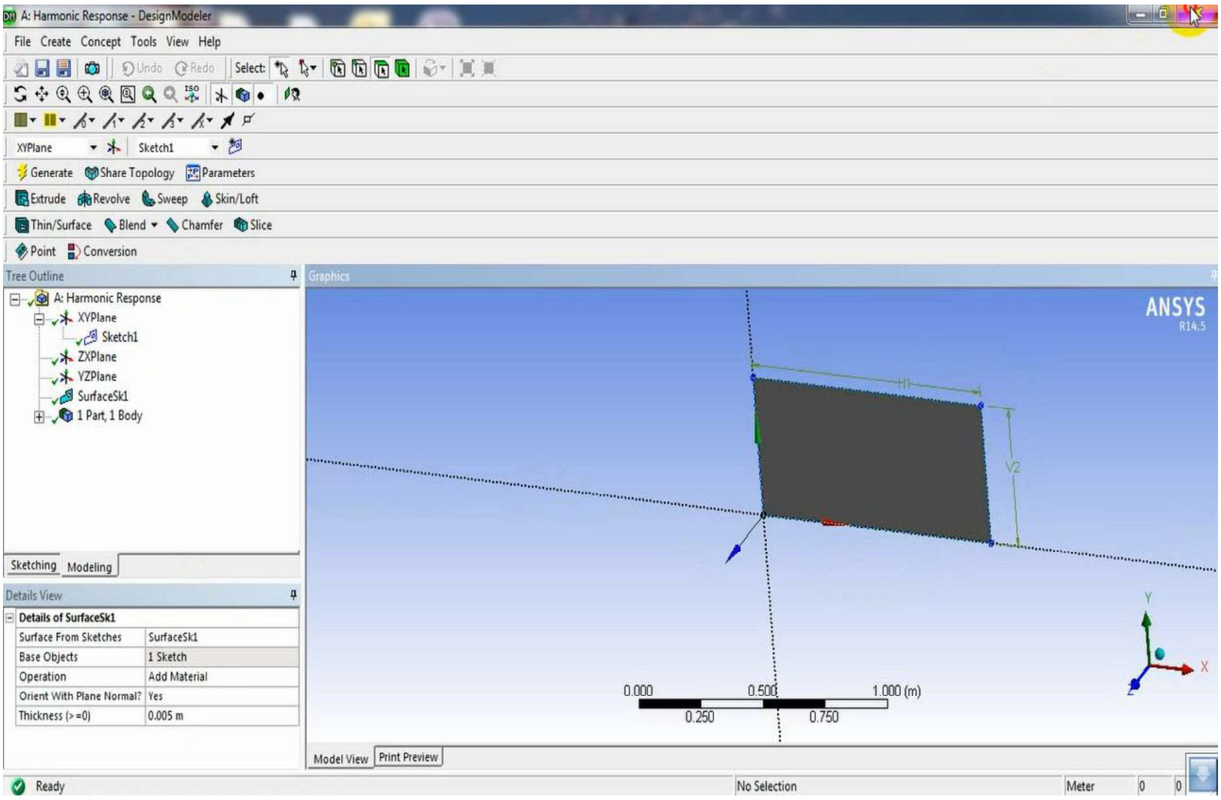
As shown in the video and our book.

Tutorial Six

Harmonic Response

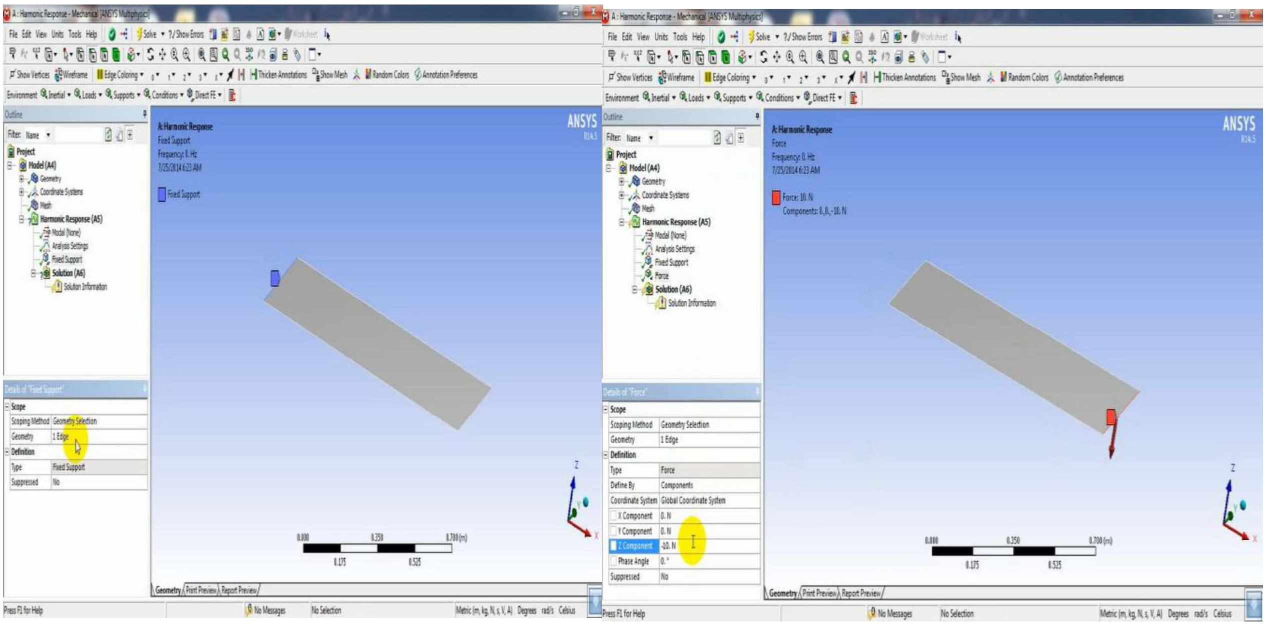
Select Analysis System *(Harmonic Response)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

Then design model in the form of rectangular surface in dimensions (*1×0.5 m*) and thickness (*0.005 m*) by using (*Design Modular*) as shown in figure bellow:

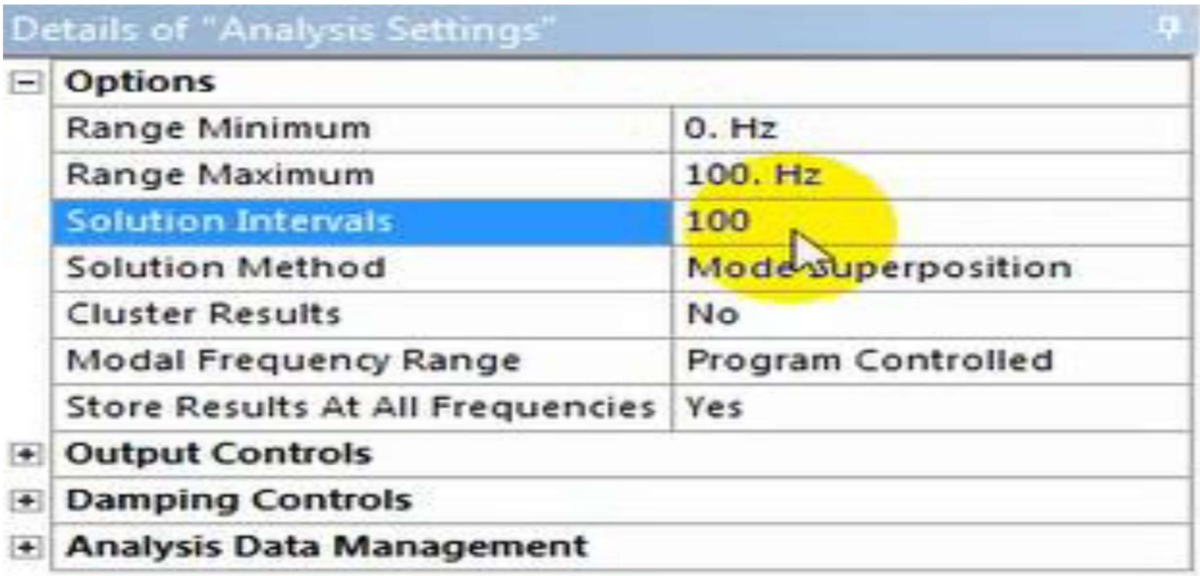


Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved.

After make the mesh a body is support from one of those ends and applied force on the other end in magnitude (*10 N*) in phase angle (*0 deg*) as shown in figures bellow:



As well as the analysis is adjusted with respect to the frequency where the frequency range is set to be the test then where it was set for the current model at the lowest frequencies (*0 Hz*) and a higher frequency range (*100 Hz*) as shown in the following figure:

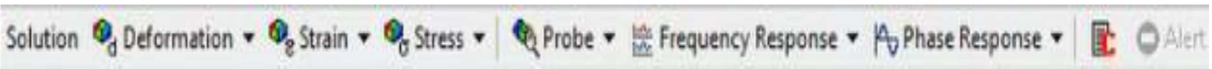


Details of "Analysis Settings"

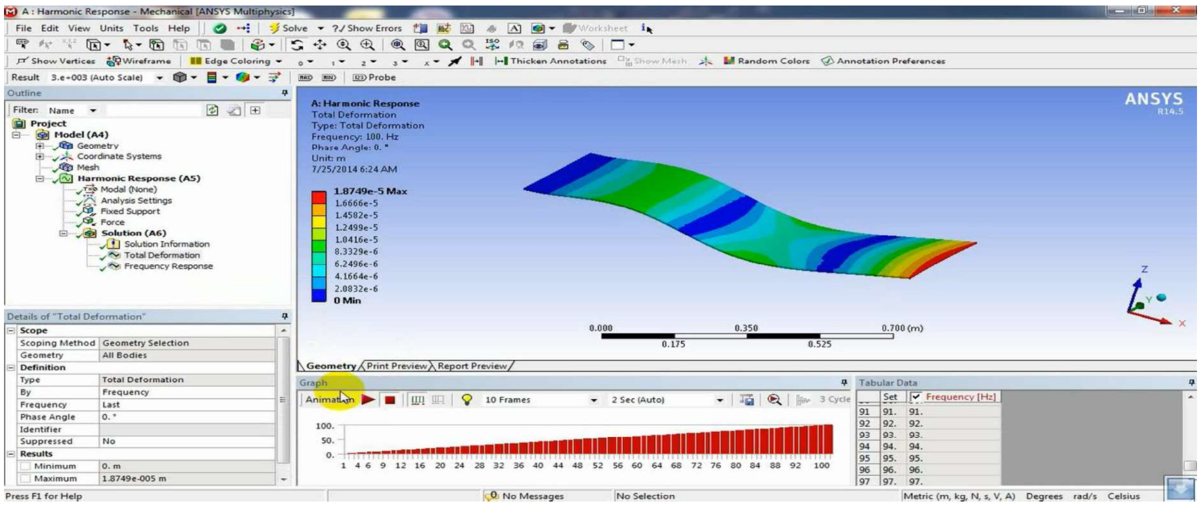
Options	
Range Minimum	0. Hz
Range Maximum	100. Hz
Solution Intervals	100
Solution Method	Mode Superposition
Cluster Results	No
Modal Frequency Range	Program Controlled
Store Results At All Frequencies	Yes
Output Controls	
Damping Controls	
Analysis Data Management	

Then the case is resolved, where we give it a solution (*Solve*) then the solution will be unknowns to be computed which it forced vibrations (under applied any external force) for the current application.

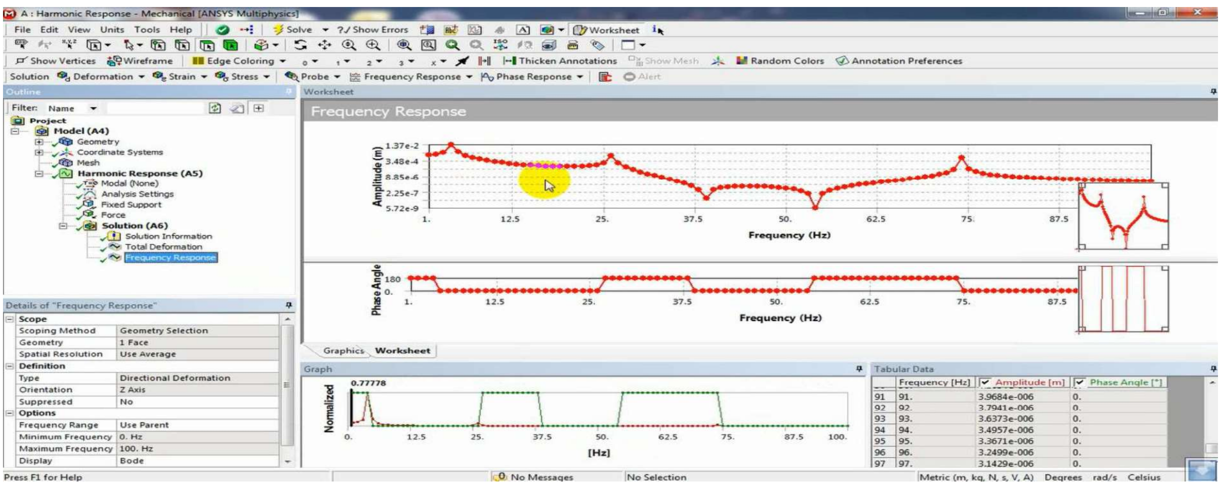
After that is the solution we can accept the results to be reviewed and that can be deformation, stress strain etc., as shown in bar below:



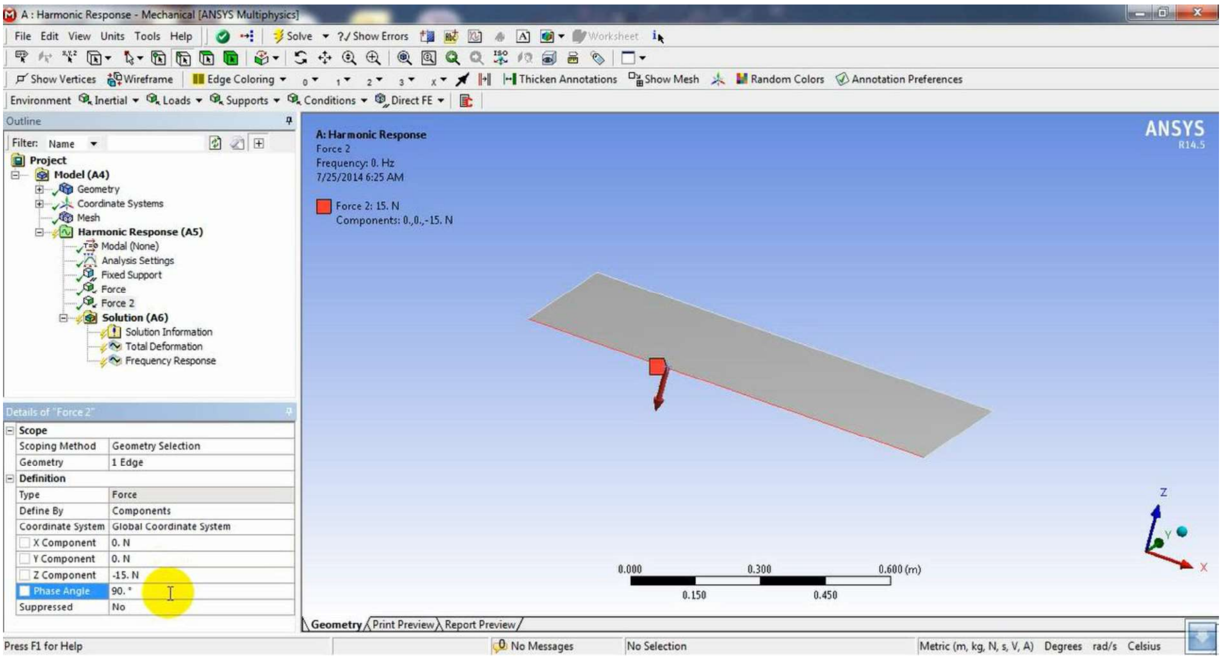
The following figure shows the results of the deformation of the current model under the applied force:



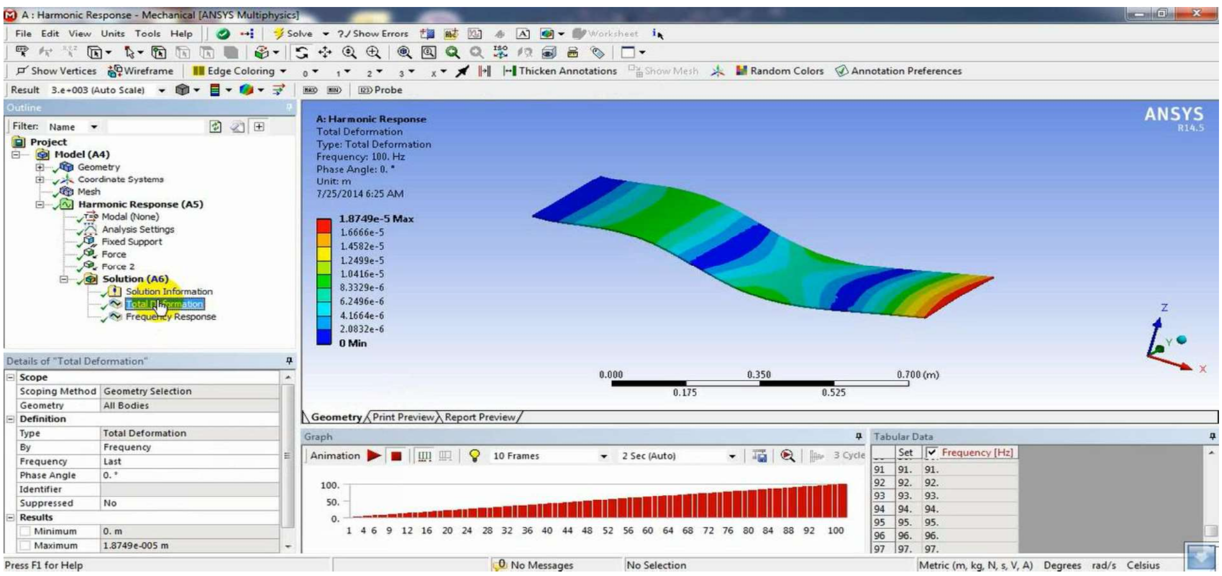
The following figure shows the response frequencies of the body under the influence of force:



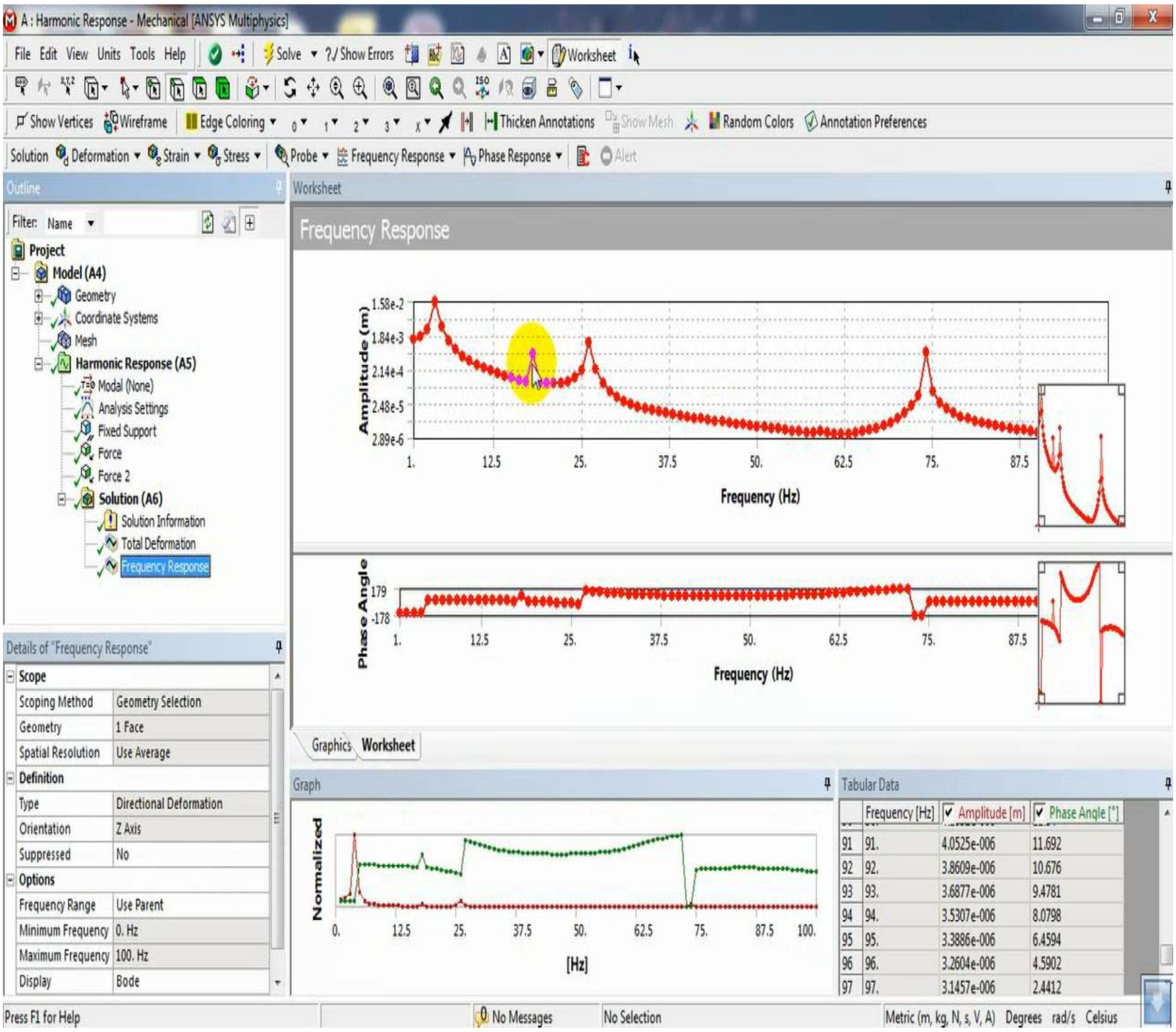
Also we can apply many of the forces on the body and through different phases and note the change in deformation and response frequencies for these loads For example when applied another force on the other edge in magnitude (*15 N*) and in phase angle (*90 deg*) as shown in the following figure:



As a result, the value of the deformation and fregucncy respond will change, in the following figure shows the results of the deformation inflicted as a result of force:



The following figure shows the results of the frequency response of the forces inflicted:

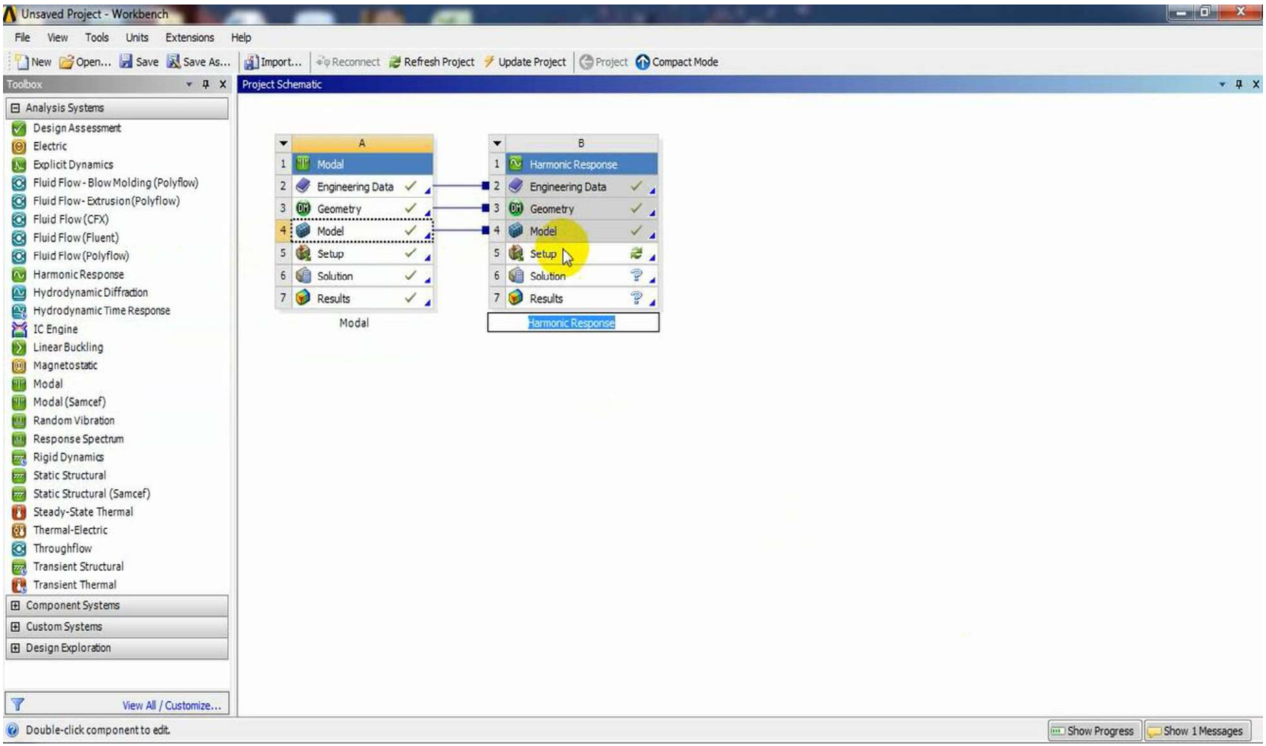


As shown in the video and our book.

Tutorial Seven

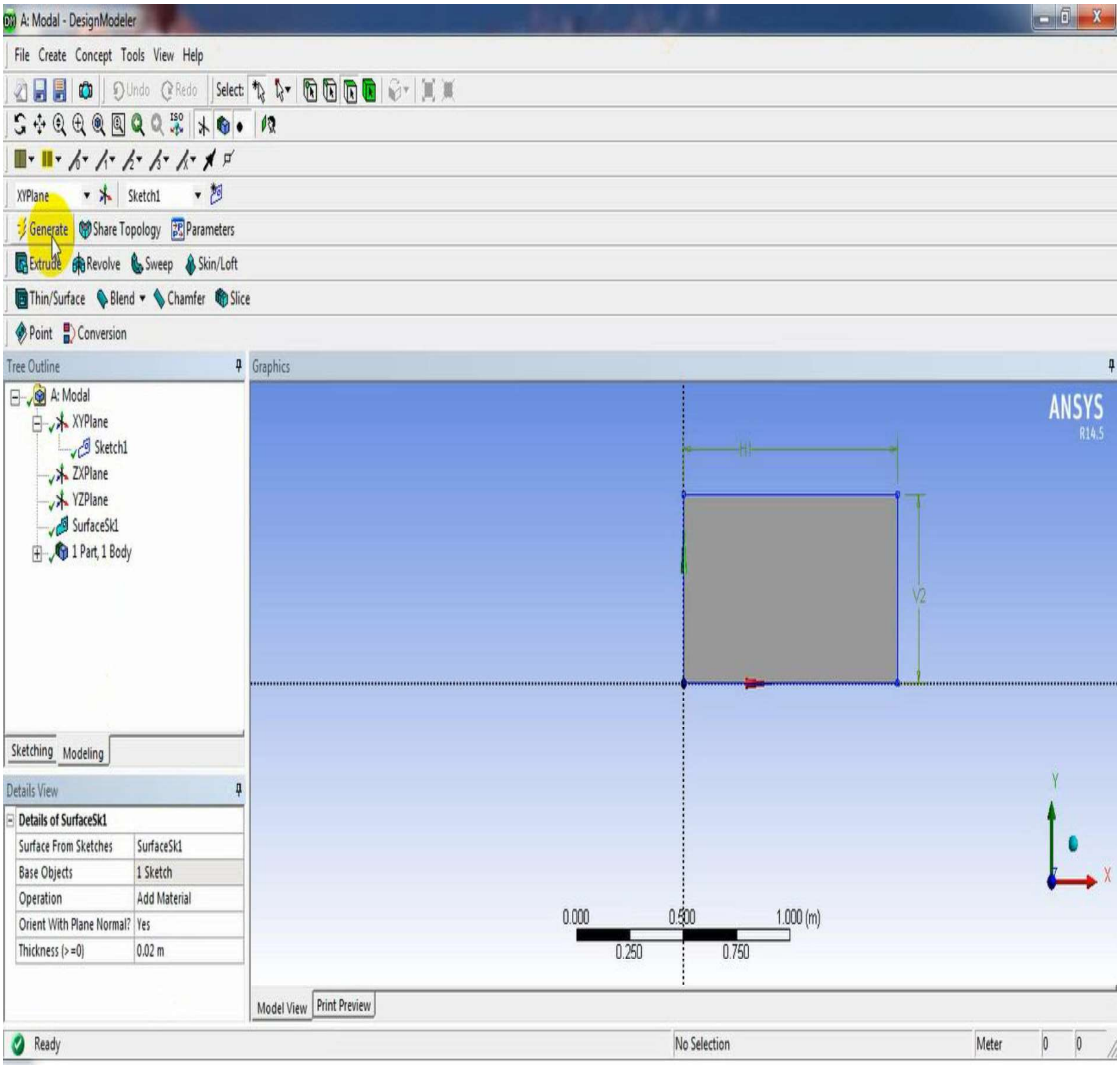
Model + Harmonic response

Select Analysis System *(Model)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then be select (*Harmonic Response*) from the analysis system and by dragging and dropping on the solution of the first analysis system (*Model*), it is linking the (*Engineering Data + Geometry + Model*), as is shown in the following figure:



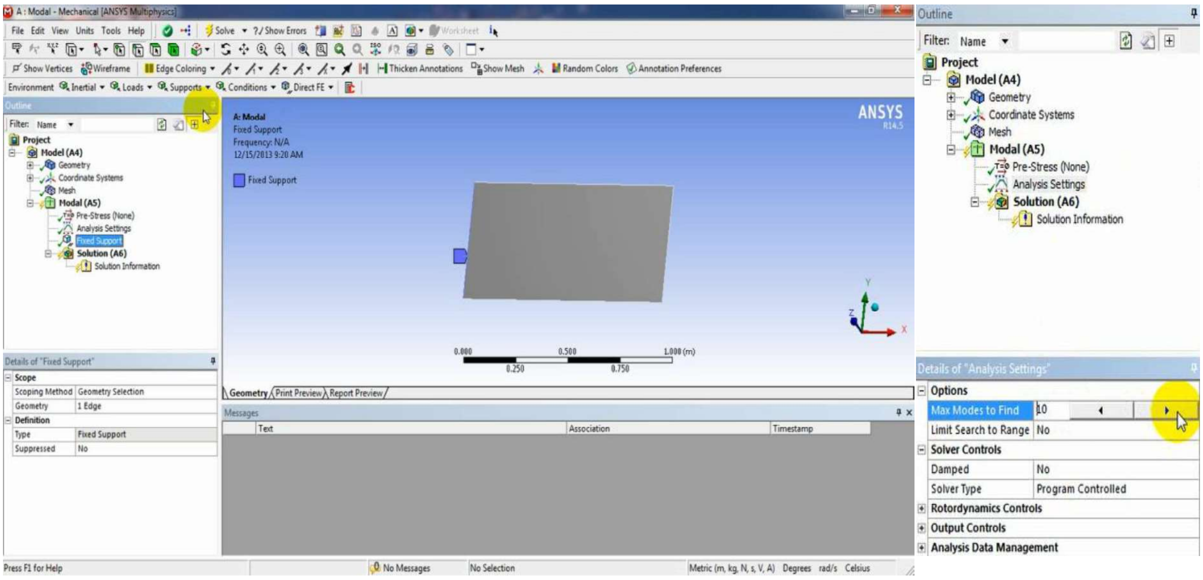
And then the test material is selected.

Then design model in the form of rectangular surface in dimensions (1×0.5 m) and thickness (0.02 m) by using (*Design Modular*) as shown in figure bellow:



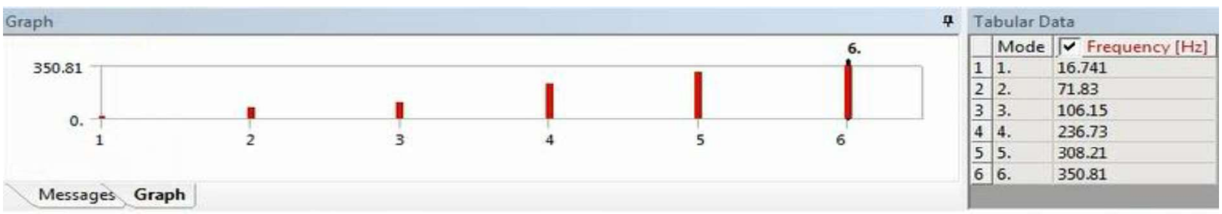
Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved.

After making the mesh a body is support from one of those ends and adjusts the number of models (*6*) to (*10*) model to see more of free vibrations that the body can vibrate as shown in the following:



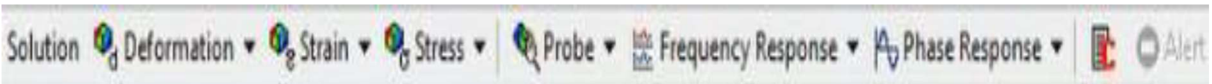
Then the case is resolved, where we give it a solution (Solve) then the solution will be unknowns to be computed which are free vibrations (ie, without applied any external force) for the current application.

After the completion of the solution we will get the range of frequencies that for the body and in the number of models that have been set as shown in the following:

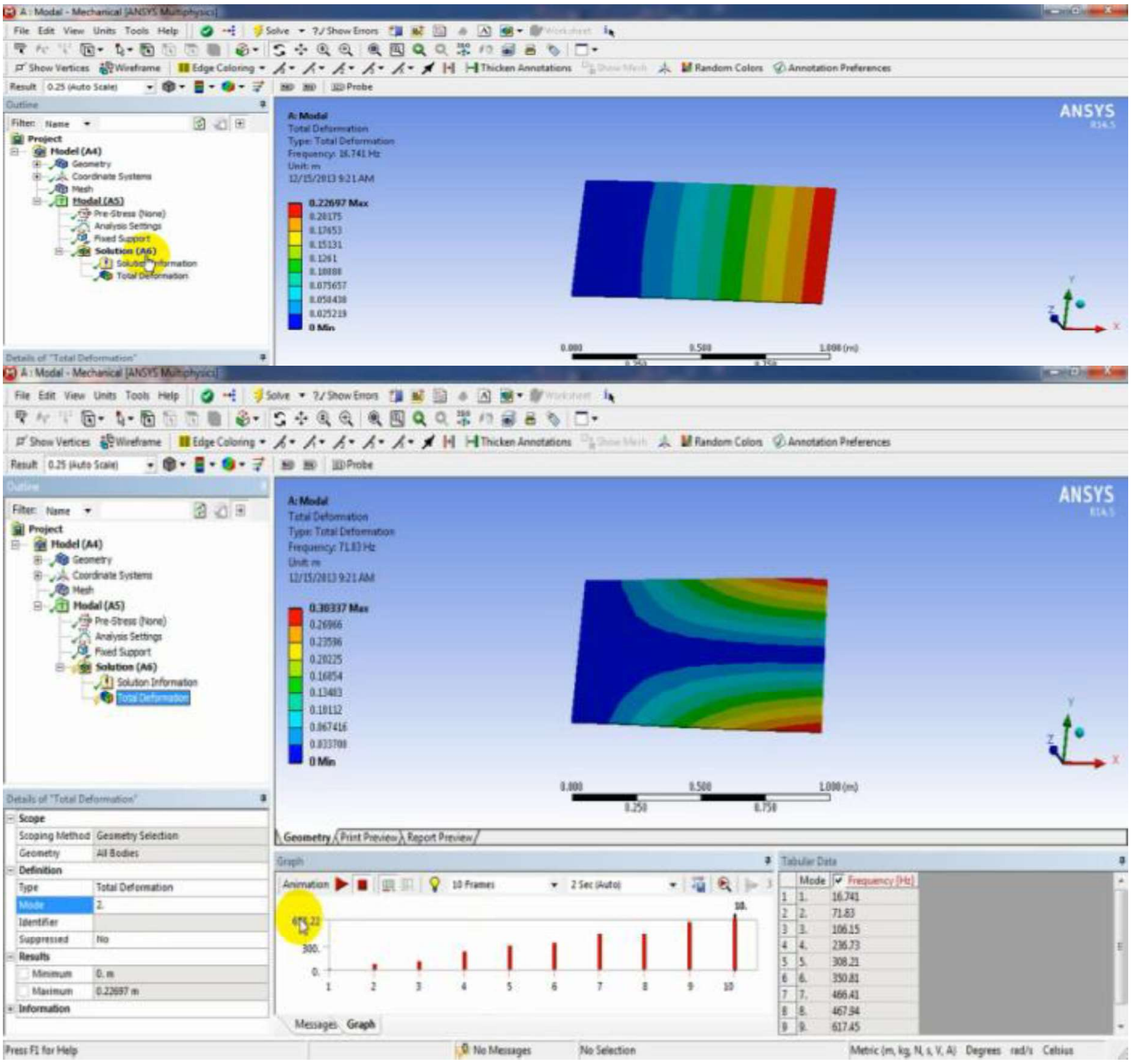


Where these frequencies can be adopted when the forced frequency to find out points overlap between free and forced vibrations which at that occur the resonance phenomenon in the body and lead to the breakdown of the body.

After that is the solution we can accept the results to be reviewed and that can be deformation, stress strain etc, as shown in bar below:

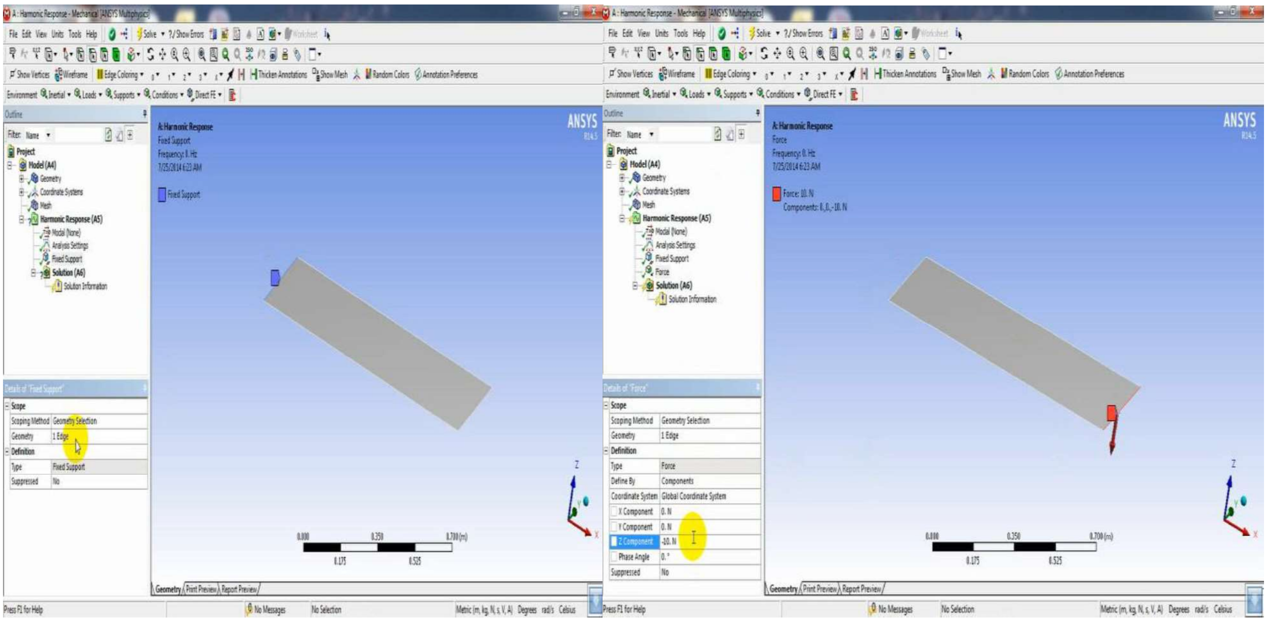


For example, the following figures are showing the deformation of the first and second model :

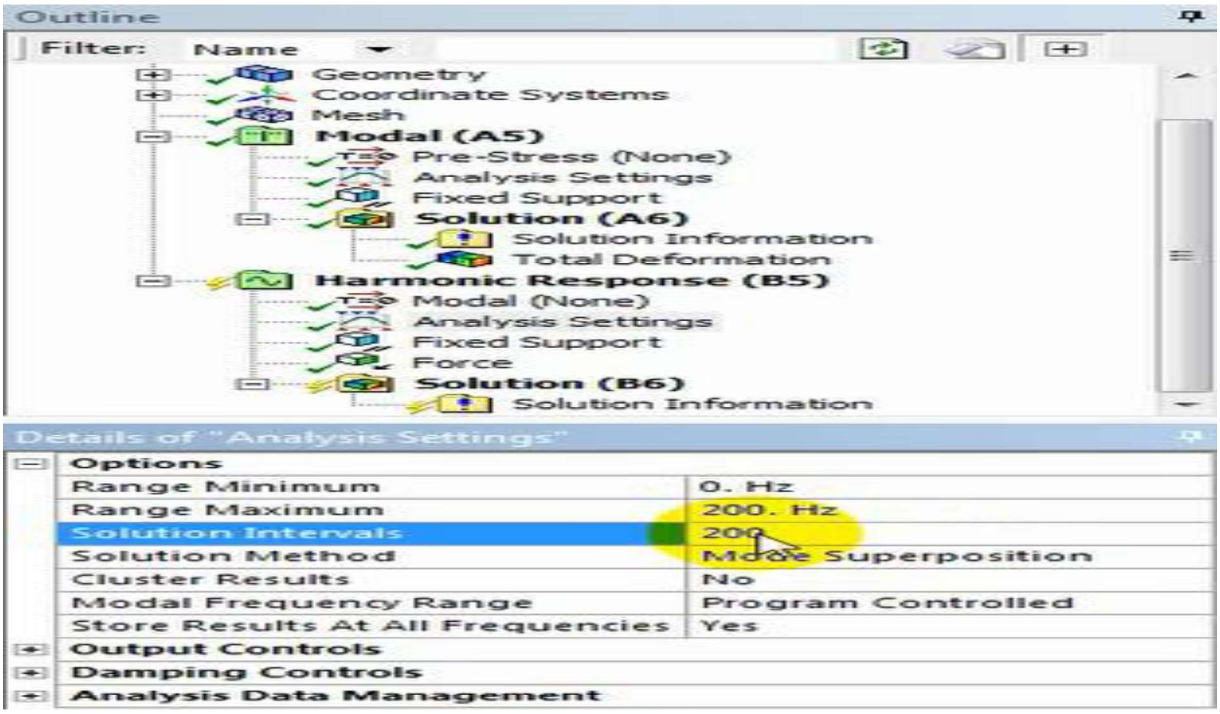


After solving the first part, which is the (*Model*) now going to the second part, which is the (*Harmonic Response*) where in this part starts with (*Setup*), where, as mentioned above, the sections linked in the (*Engineering Data + Geometry + Model*).

Where supported the body from one ends and applied load on the other end in magnitude (*-10 N*) and phase angle (*0 deg*), as shown in the following:

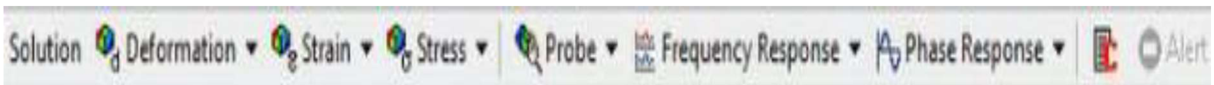


also adjusted analysis with respect to the frequency where they are set the frequency range to be test then that is within the range that we got from the first system solution (*Model*) that begin with (*16.741 Hz*) and ending with (*350.81 Hz*) to the first model, where we choose the extent of involving more than one frequency to see the points of overlap between free and force frequency, for example, we can adjust the range for the current model at the lowest frequencies (*0 Hz*) and a higher frequency range (*200 Hz*) as shown in the following figure:

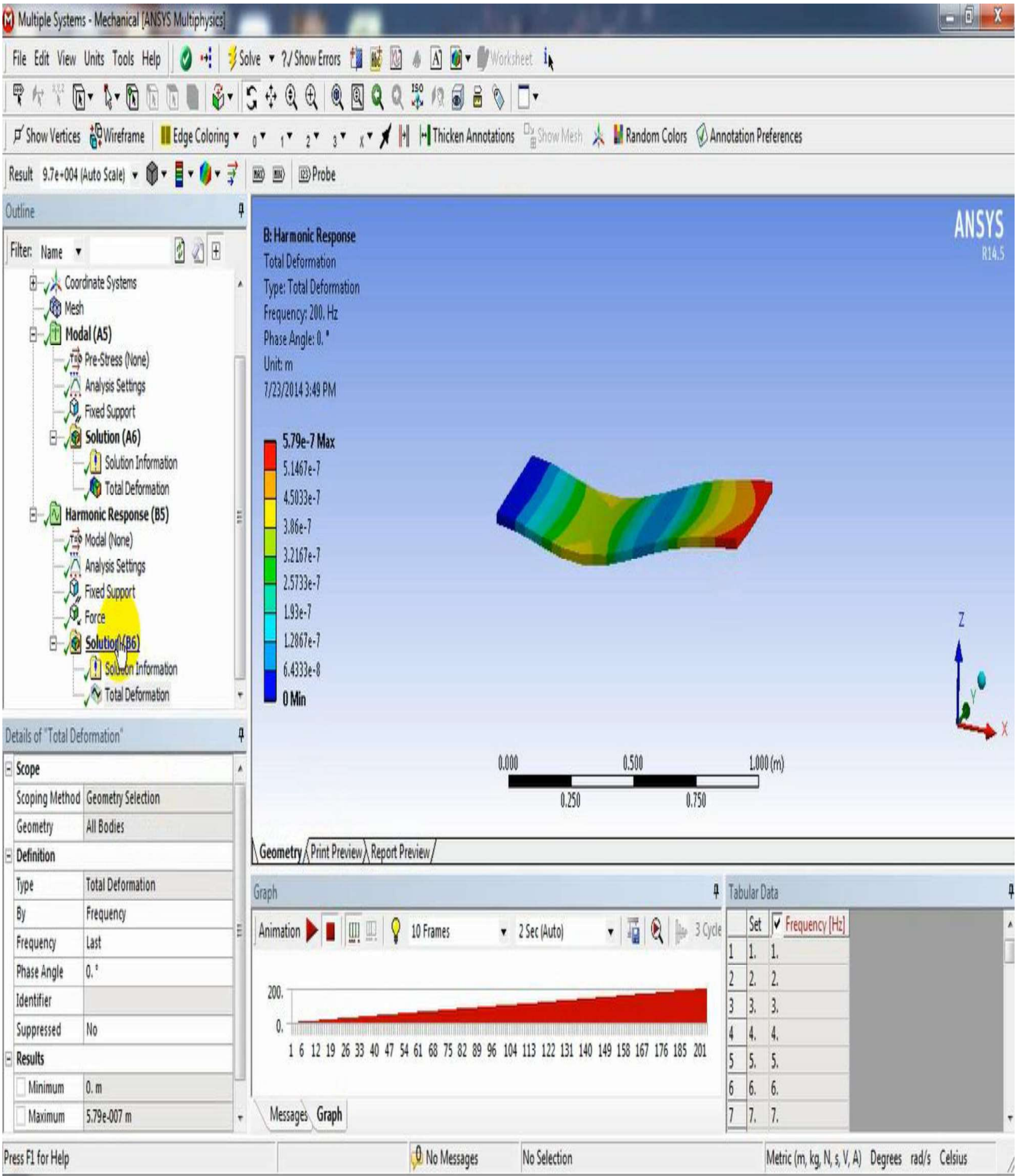


Where enters in this range natural frequency of the free first (*16.741 Hz*) and second (*71.83 Hz*) and third (*106.15 Hz*) and is then solve the case where we give it a solution (*Solve*) will then be unknowns to be calculated, which are force vibrations (i.e., under the applied external force) for the current application.

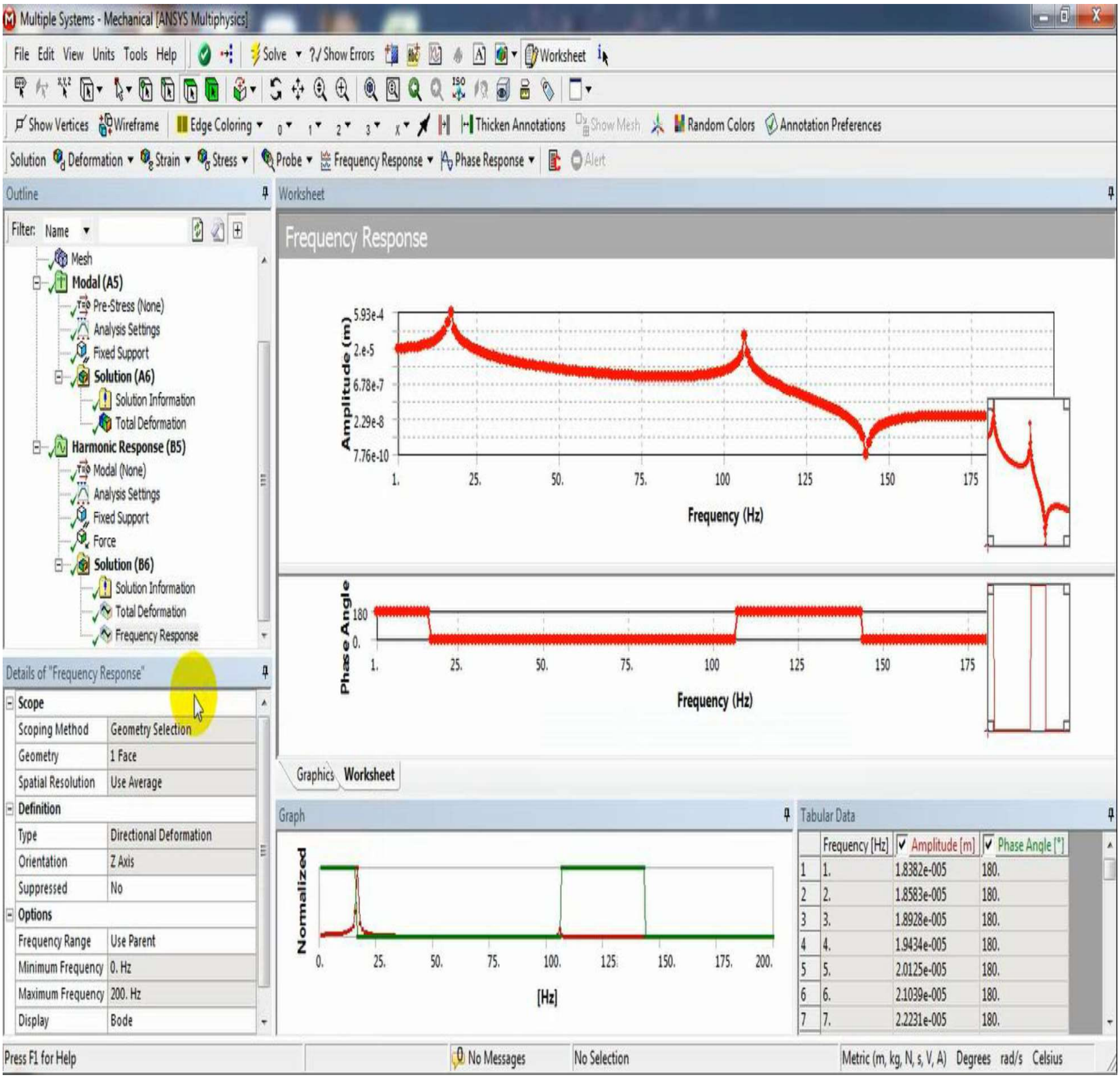
After that is the solution we can accept the results to be reviewed and that can be deformation, stress strain etc, as shown in bar below:



The following figure shows the results of the deformation of the current model under the applied force:



The following figure shows the response frequencies of the body under the influence of force and points overlay, which represents the highest value of the Amplitude:



As shown in the video and our book.

Tutorial Eight

Static Structure + Linear Buckling

Select Analysis System *(Static Structure)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then be select (*Linear Buckling*) from the analysis system and by dragging and dropping on the solution of the first analysis system (*Static Structure*), it is linking the (*Engineering Data + Geometry + Model + Solution with Setup*), as is shown in the following

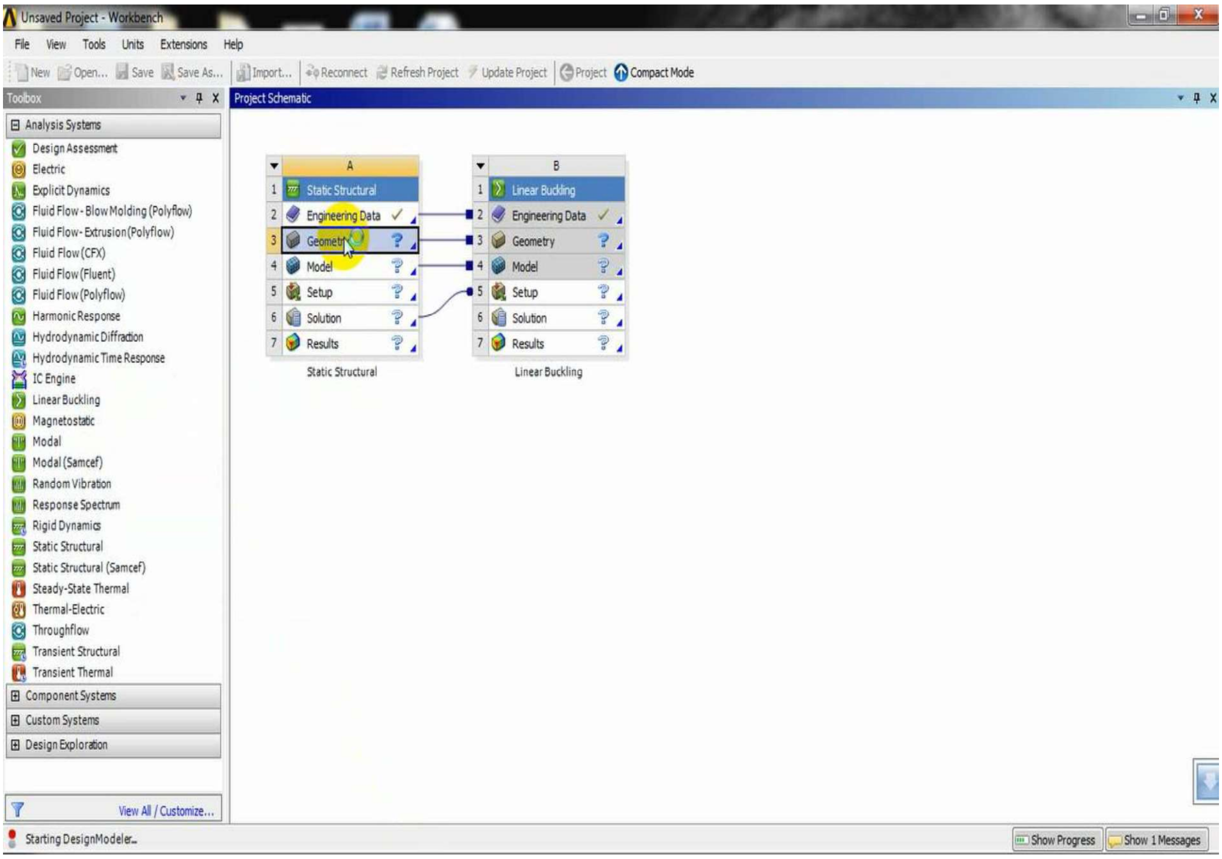
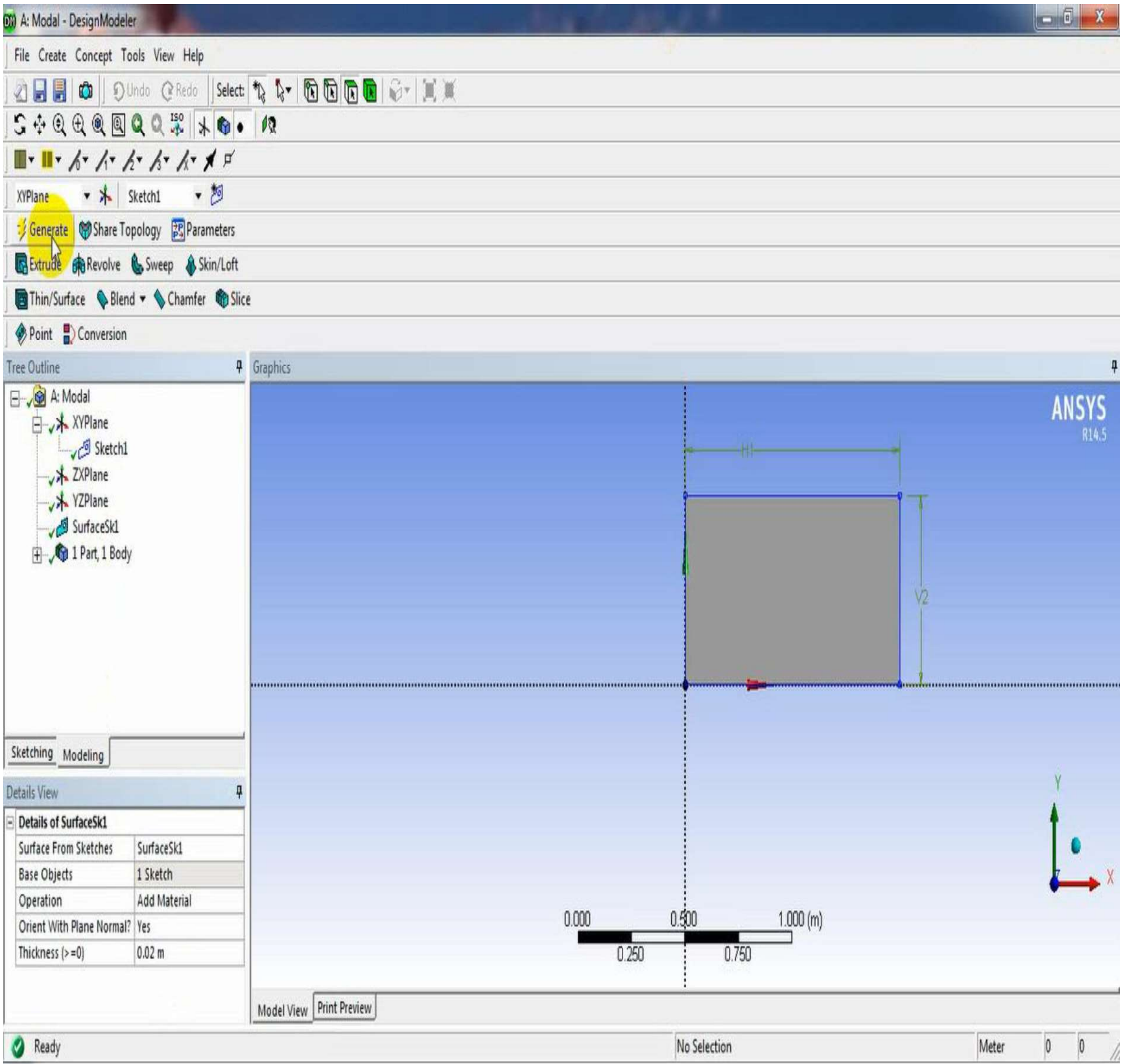


figure:

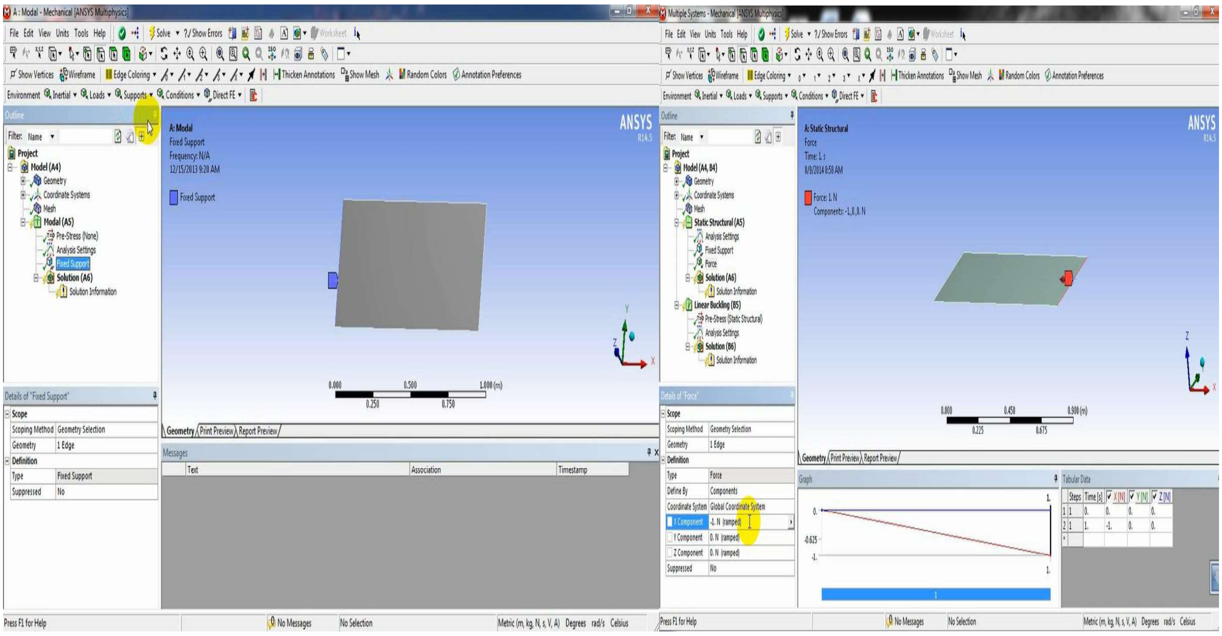
And then the test material is selected.

Then design model in the form of rectangular surface dimensions (*0.5 × 1 m*) and thickness (*0.02 m)* by using (*Design Modular*) as is shown in figure below:



Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved.

After make the mesh are support the body from one end and applied the compression force with value (*-1 N*) on the corresponding end as shown figure below:

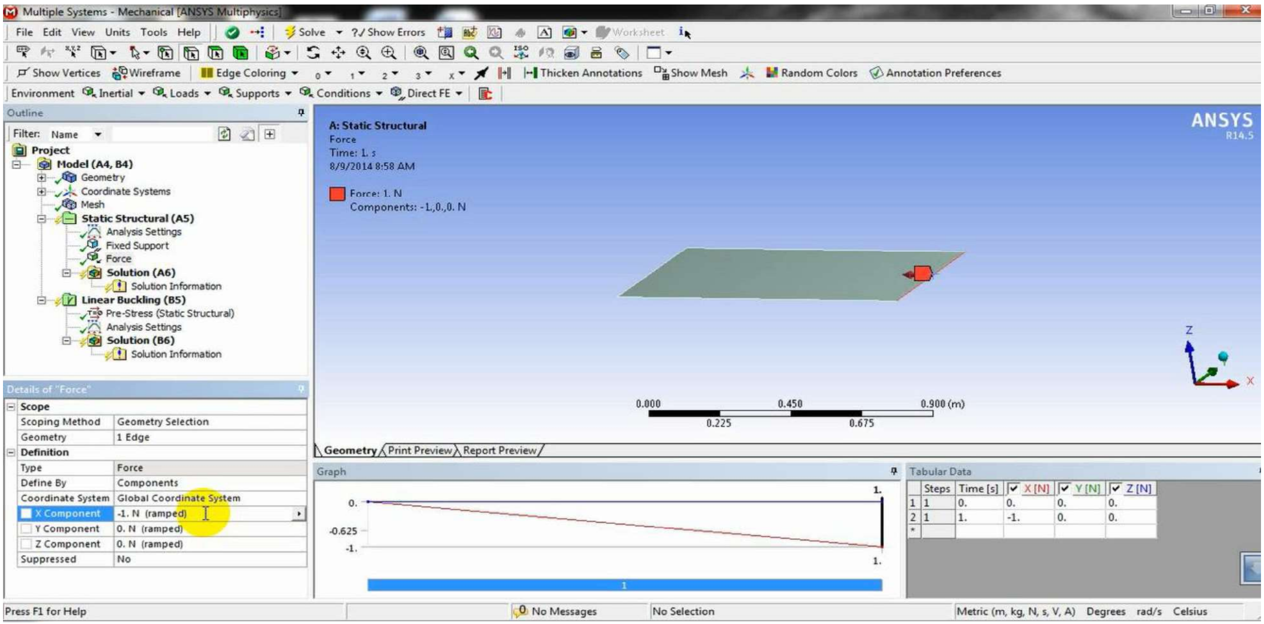


And then is solved case where we give it a solution (*Solve*) will then be solved unknowns to be computed with respect to the Static Structure, and then go to the Linear Buckling and give it Solve as this system does not contain any set additional terms depends adopt the Static Structure adjust.

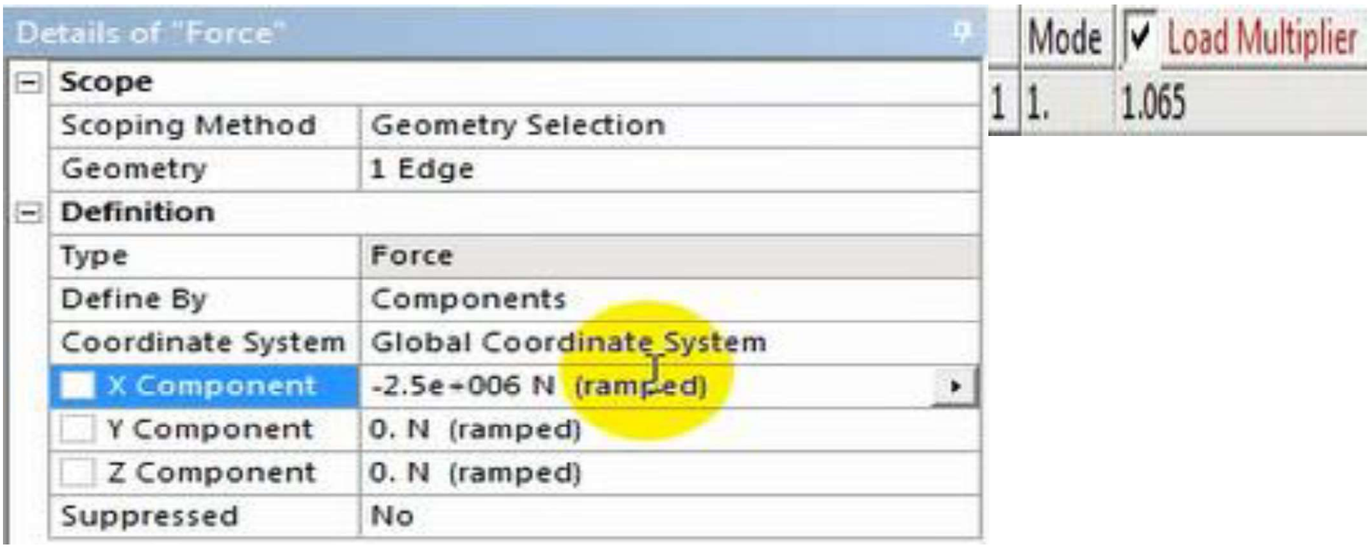
After the solution completion note result of solving the (*Linear Buckling*) If the value of the (*load multiplier*) is greater than (*1*) it does not have (*Buckling*) But if the value of the (*load multiplier*) is less than (*1*) is found (*Buckling*) where the (*load multiplier*) represents the ratio between the greatest allowable force to applied force on the body:

$$\text{Load multiplier} = \frac{F\ critical}{F\ actual}$$

To know the greatest allowable force we take the applied force on the body equal to (*1*) then it will be the (*load multiplier*) is equal to the greatest allowable force (*F critical*) on the body as shown in the following figure:



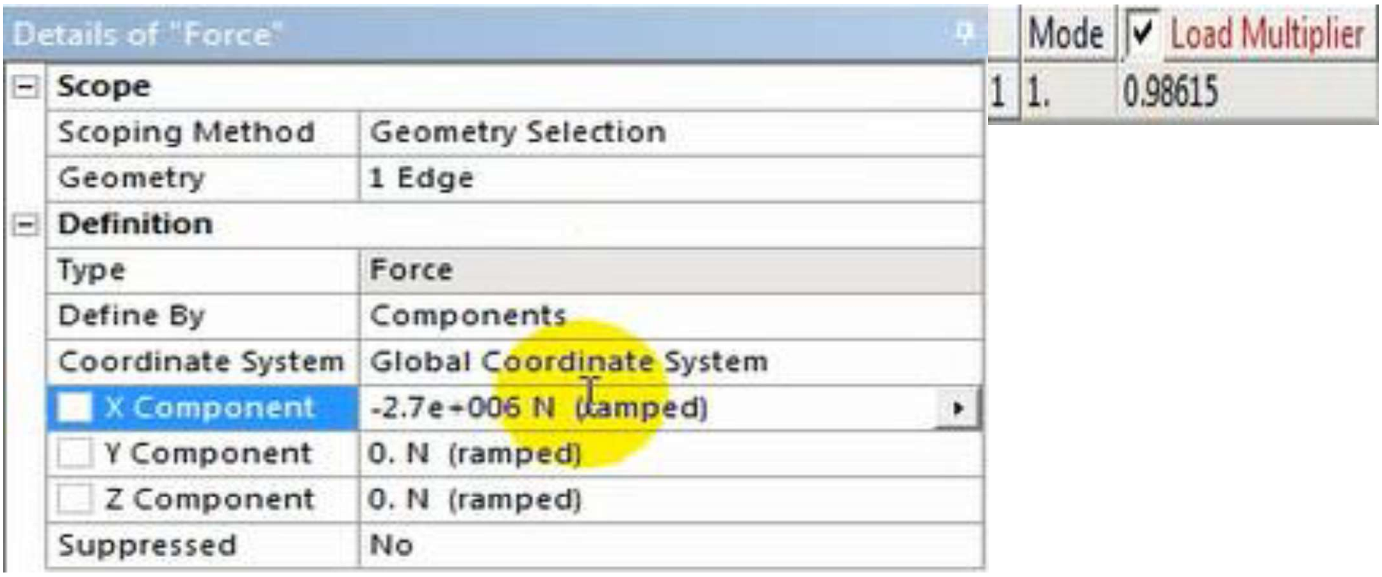
After knowing the greatest allowable force of the body is adjusting the force to suit the load multiplier for the model, When applied force with value (*2.5 e6*) which is less than the greatest allowable force on the body, which is the F critical (*2.6626 e6*), the value of the (*load multiplier*) was (*1.065*), which meaning there does not have a (*buckling*), either when applied force with value (*2.7 e6*), the largest of the greatest allowable force on the body, the value of the (*load multiplier*) was (*0.98615*) then it will happen(*buckling*)as shown in the following figure:



Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	-2.5e+006 N (ramped)
Y Component	0. N (ramped)
Z Component	0. N (ramped)
Suppressed	No

	Mode	Load Multiplier
1	1.	1.065



Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	-2.7e+006 N (ramped)
Y Component	0. N (ramped)
Z Component	0. N (ramped)
Suppressed	No

	Mode	Load Multiplier
1	1.	0.98615

As shown in the video and our book.

Tutorial Nine

Steady State Thermal + Static Structure Interaction

Select Analysis System *(Steady State Thermal)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then be select (*Static Structure)* from the analysis system and by dragging and dropping on the solution of the first analysis system (*Steady State Thermal*), it is linking the (*Engineering Data + Geometry + Model + Solution with Setup*), as is shown in the following

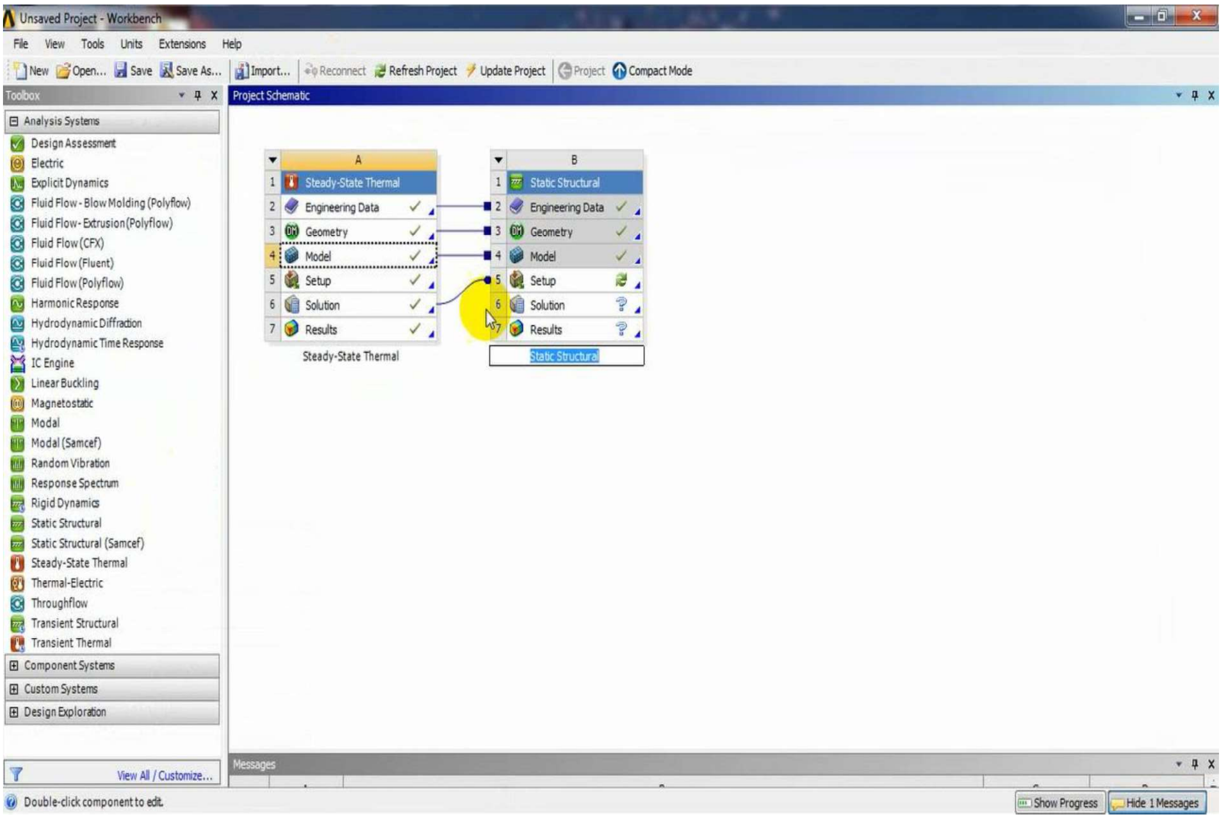
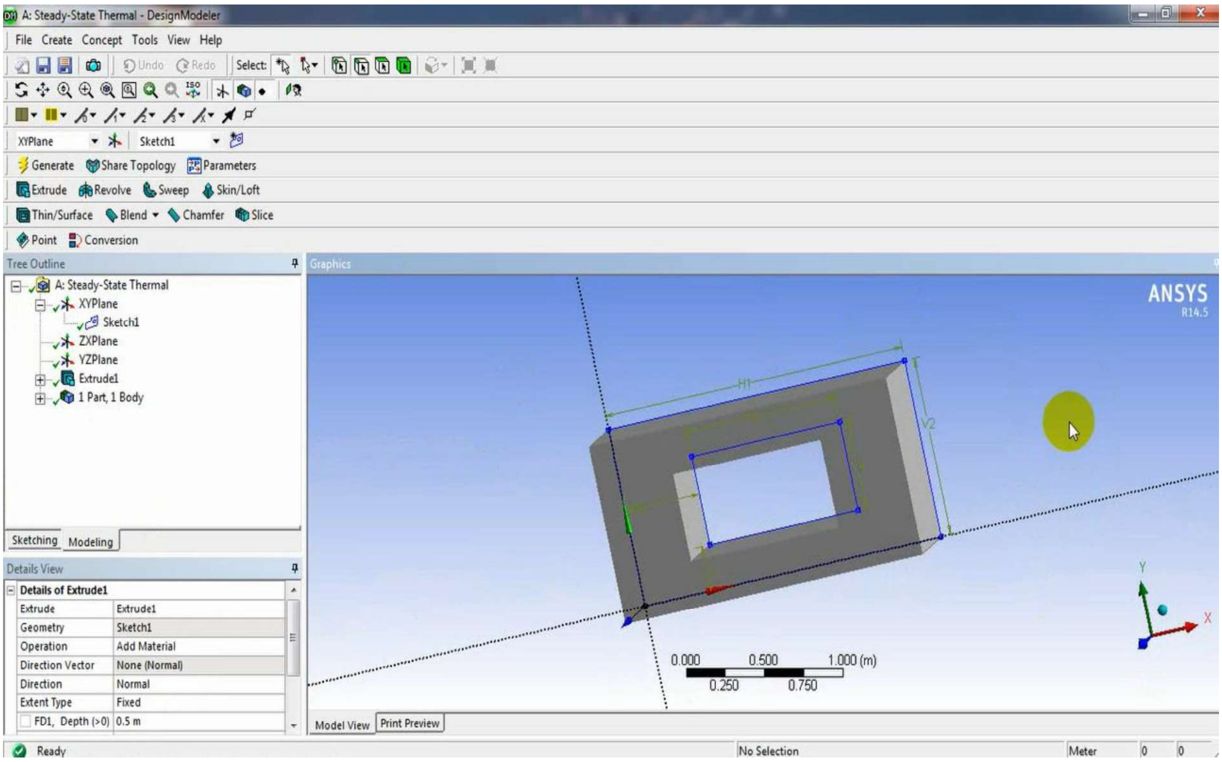


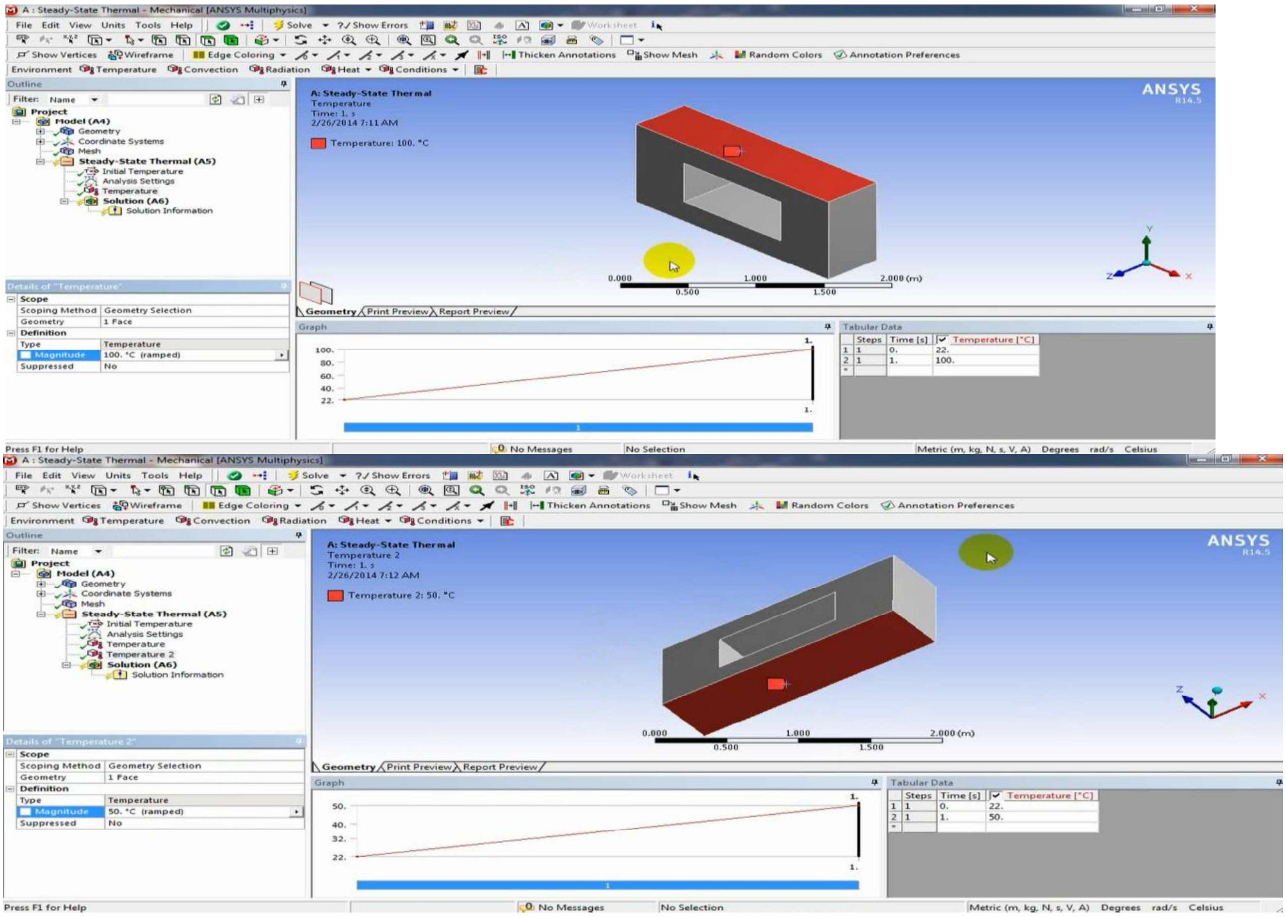
figure:

And then the test material is selected.

Then design model in the form of hollow rectangular with outer dimensions (*2×1 m*), inner dimensions (*1×0.5 m*) and thickness (*0.05 m)* by using (*Design Modular*) as is shown in following figure:

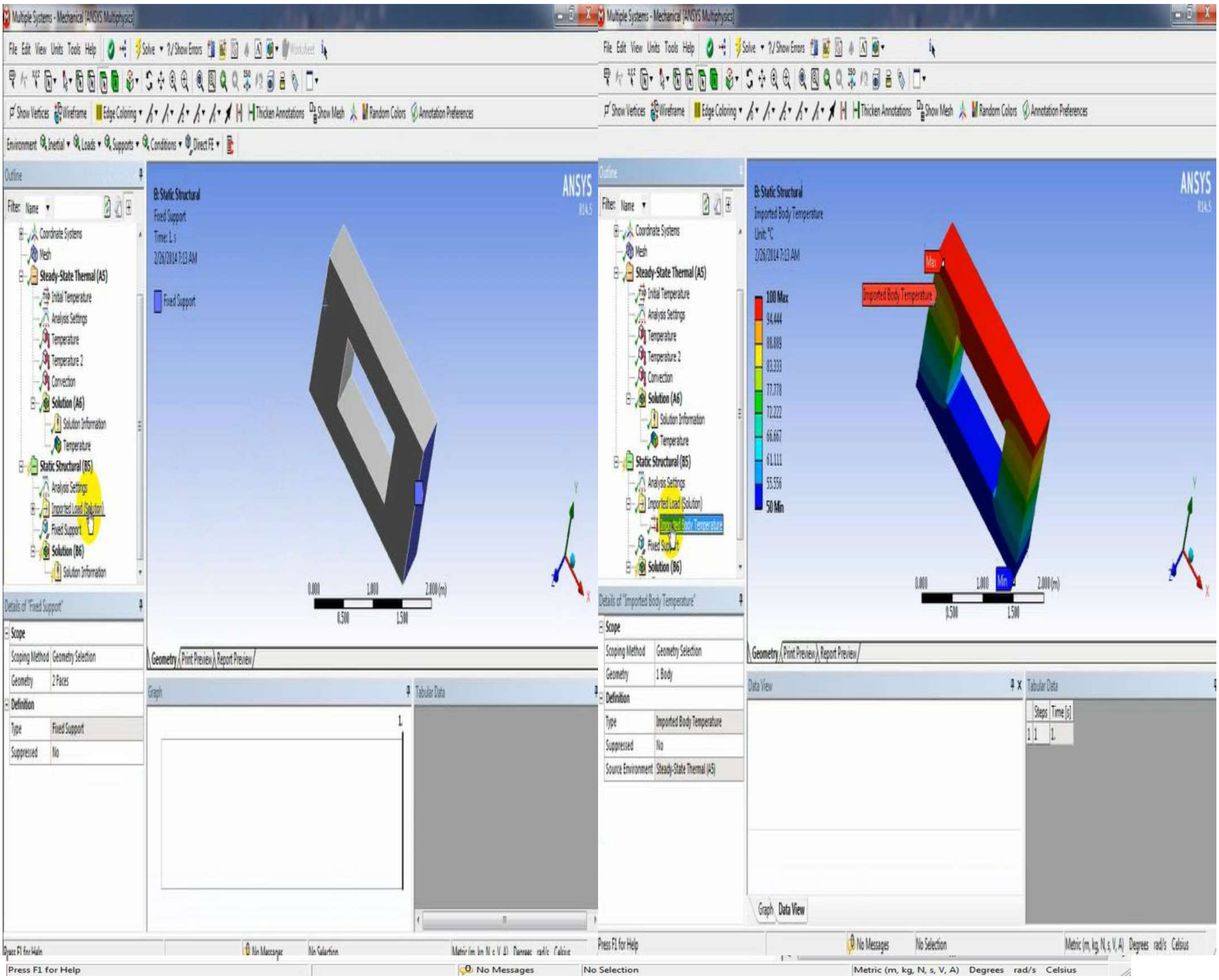


Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved. After make the mesh are applied loads on the body where the loads in this analysis are thermal loads (temperature, convection heat transfer, radiation heat transfer etc), In the present example the loads are temperatures on the top and bottom surfaces, where the temperature adjusted to the top cold surface is (*100 C*) and the bottom hot surface is (*50 C*), where the all other surfaces are losses the heat to the environment in by convection where the heat transfer coefficient is($5\ w/m^2.s$) and ambient temperature is ($40C$), as shown in

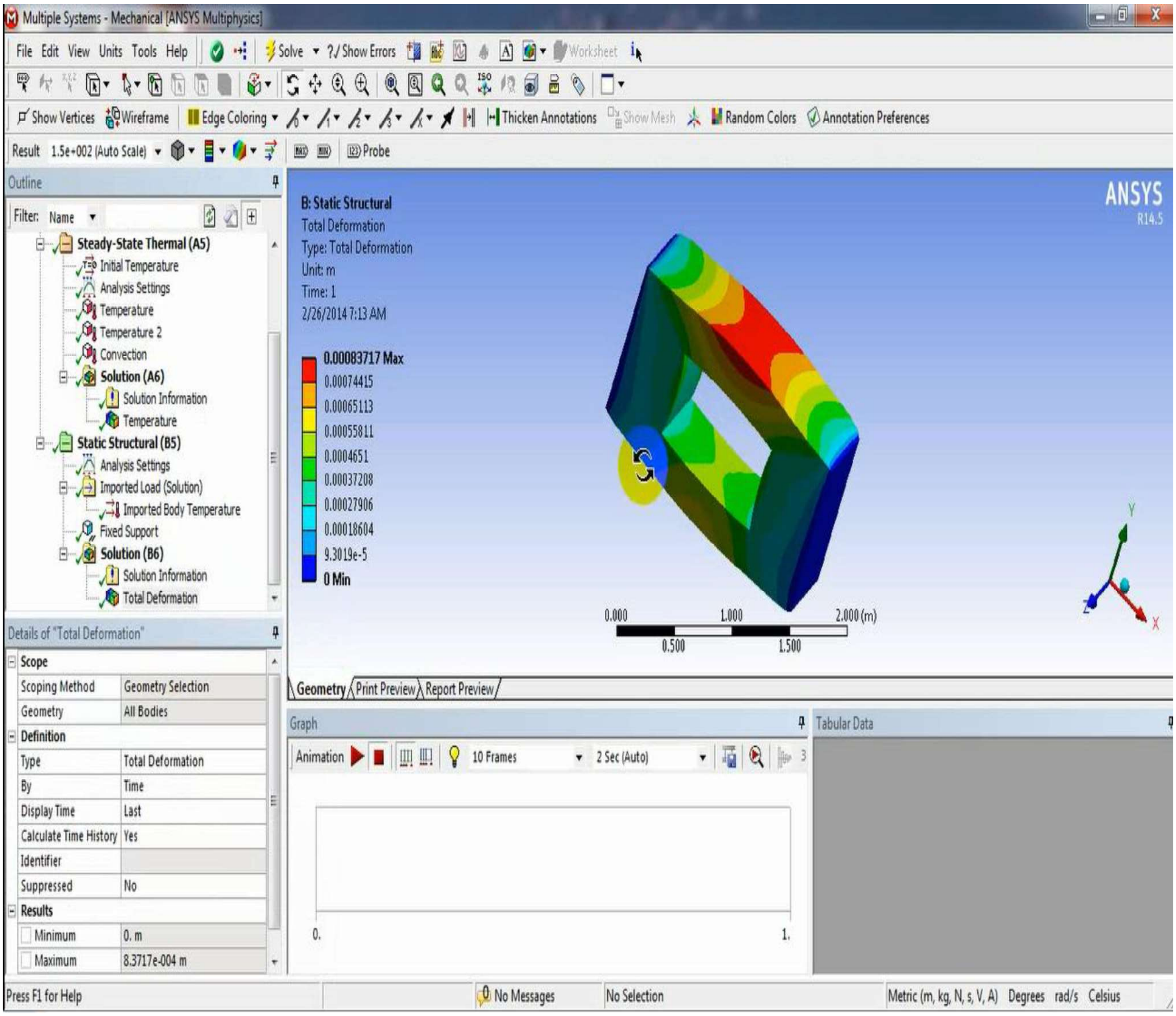


the following figure:

After setting boundary condition where solve and view the result the case respected to steady state thermal analysis system, then transient to the second analysis system is static structure where applied mechanical load for the case in this analysis system and then import the thermal load from the first analysis system (*Steady State Thermal*) in the present case are support the body from the tow ends and import and applied the thermal load from steady state thermal analysis system as shown in the following figure:



After applied loads are solving the case and view the results for example the following figure shows the deformation that produced from the loads:



As shown in the video and our book.

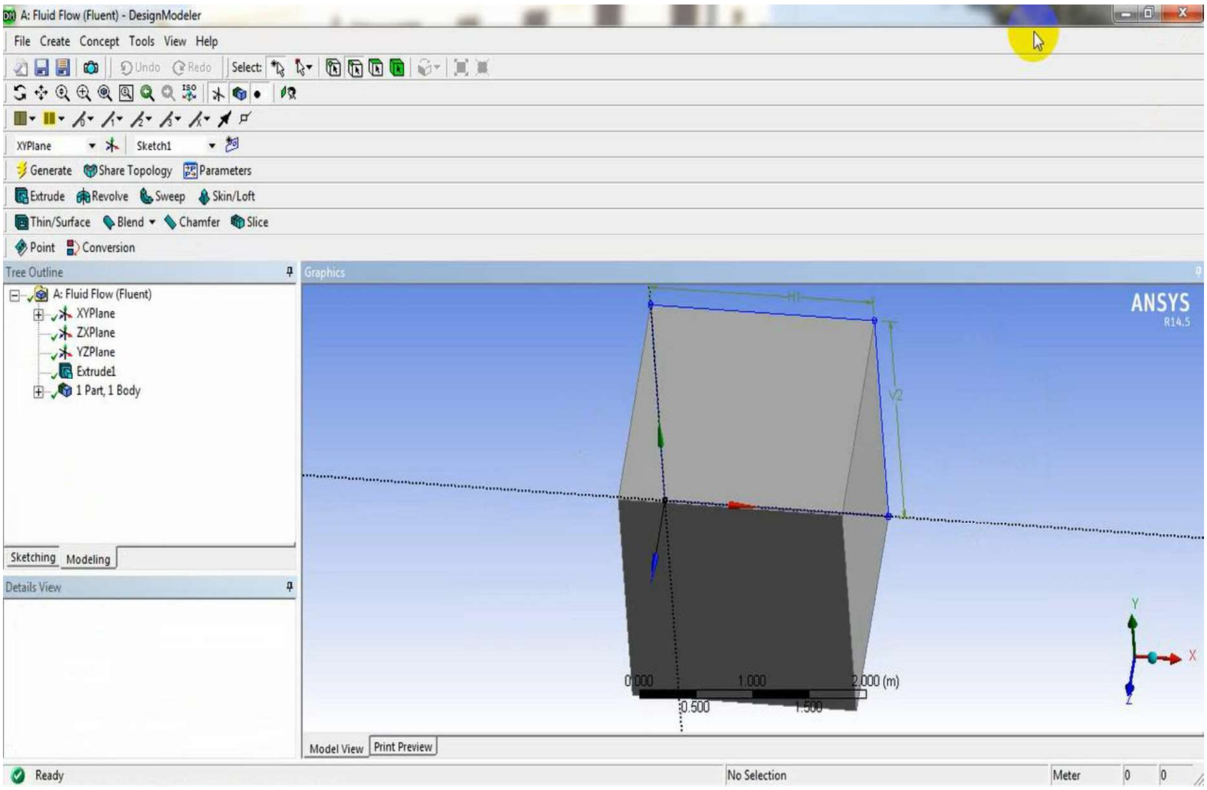
Tutorial Ten

Fluid Flow (Fluent)

Free Convection

Select Analysis System *(Fluid Flow Fluent)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

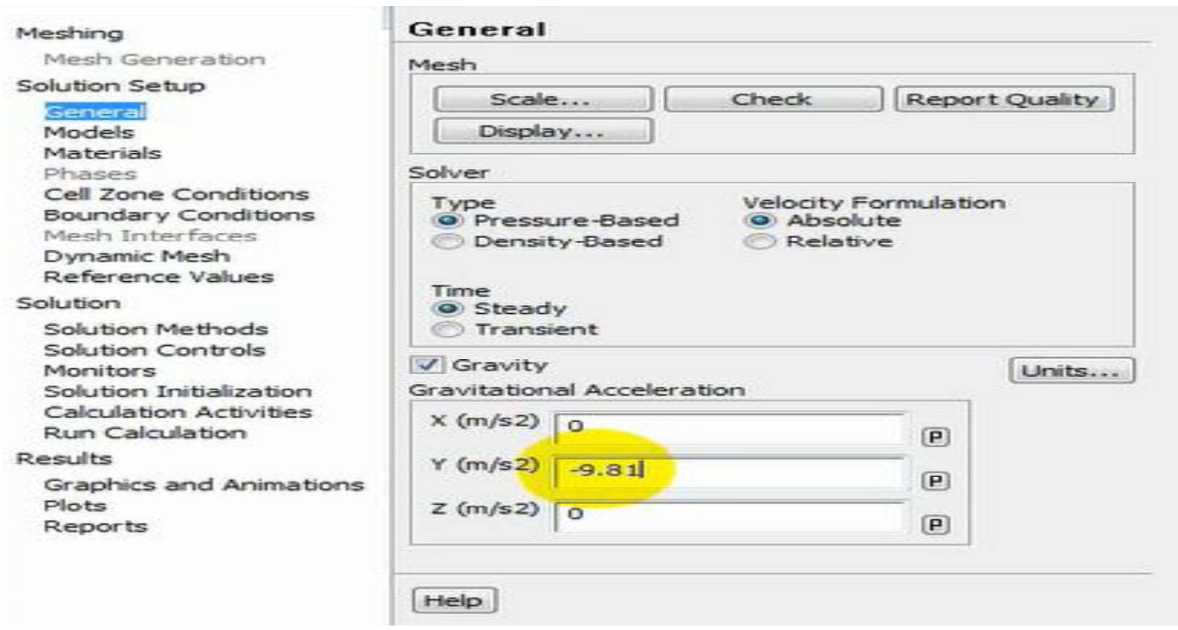
Then design model in the form of cube in dimension (1×1×1 *m*) by using (*Design Modular*) So as to simulate the process of heat transfer by convection for the present model, as shown in the following figure:



Then making the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved, As well as naming the surfaces that will be set later example, for the present example are named the upper surface as (*Cold Surface*) and the lower surface (*Hot Surface*) where the other surface are insulated.

After make the mesh and obtain the required number of slides are going to the next step, which is to open the program (*Fluent*).

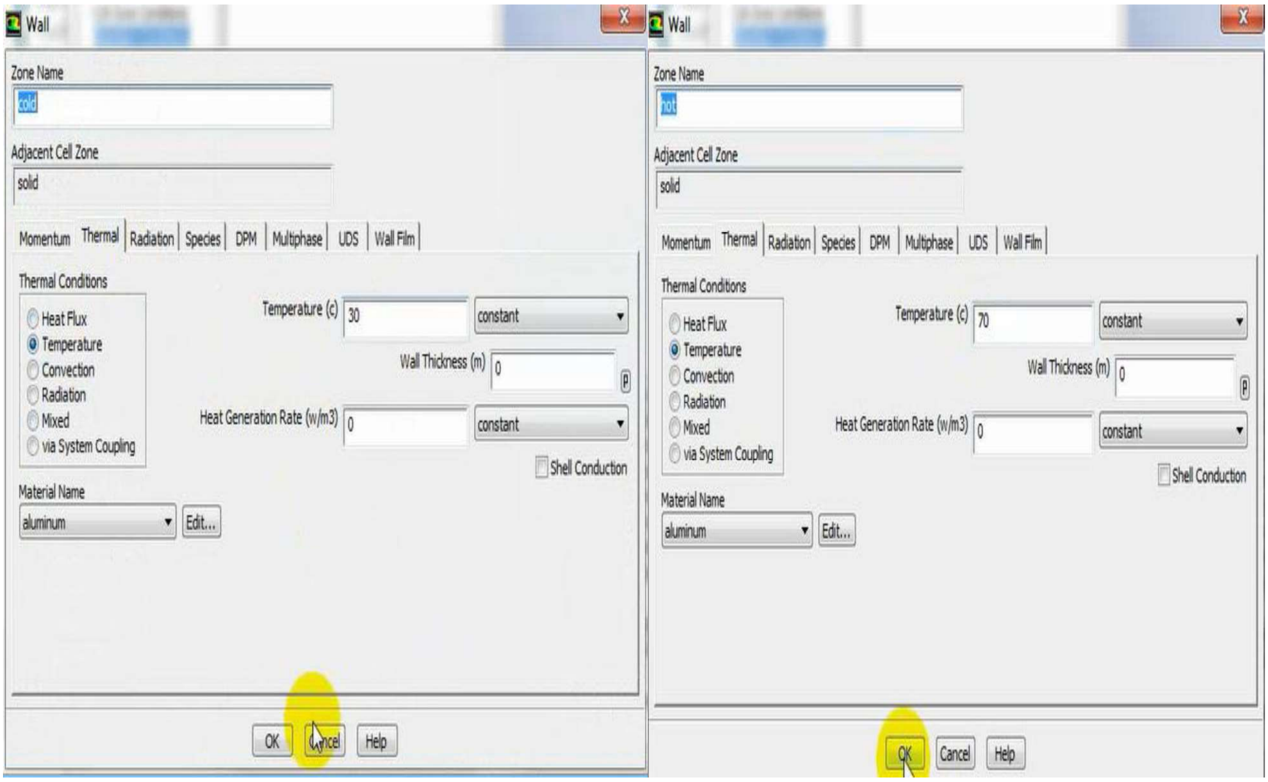
Where in this program settings must be adjusted (*General*), which adjust the type of solution which is adjusted units and accelerate where they are pointing the (*Gravity*) and adjust the value of the accelerating in the direction that act in it, as the acceleration of the influential in the current model is the ground accelerate where it is adjust the value of accelerating ($9.81\ m/s^2$) as shown in the following:



And where set the models light for the present model are activation (*energy equation*) only.

And where set the working material, for the present model the working material is air; it is selected and gives it (*Change / Create*) to be dependence by the program.

Then adjust the conditions are (*boundary conditions*) i.e. (applied loads) where they are applied loads depending on condition of model. For the present model we set temperature of upper surface with value (*30 C*) and temperature of lower surface with value (*70 C*), and the other surfaces are isolated and there is no any load on it as shown below:

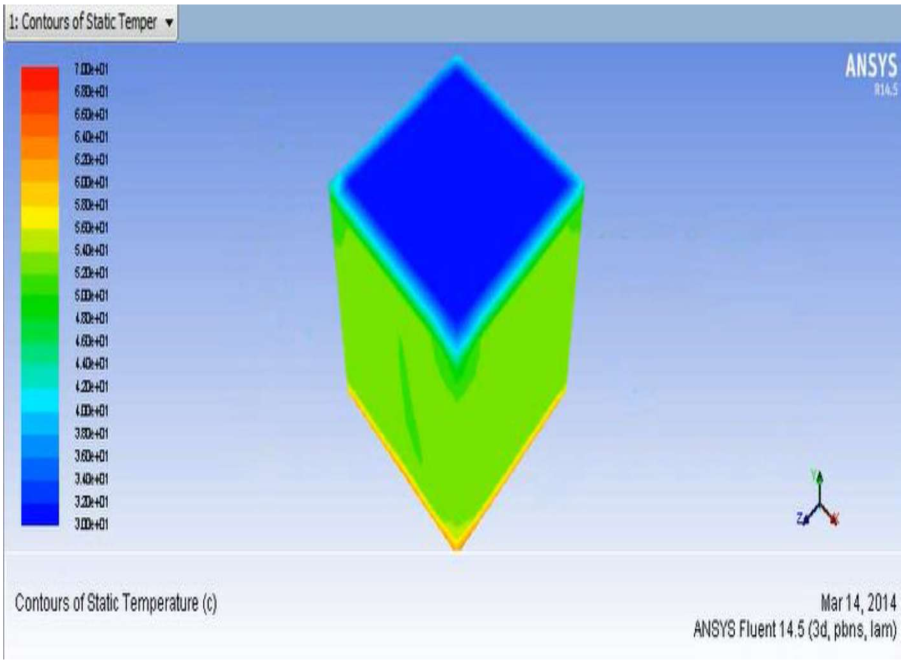


And after setting the boundary conditions are initialized solution(*Solution Initialization*)where we choose create a standard (*Standard Initialization*) can choose the configuration of any surface we want it to be as if the cold surface .

After that the solution is initialized is solution the case where we choose (*Run Calculation*) and give the number of times the correction with respect to the present example have been set number of iteration (*100*) and then click (*Calculate*) .

After the solution complete the program gives us message that the solution complete. After that we can reviewing the results it can be possible (*Graphics and Animation*) or (*Plots*) or (*Reports*).Relative to (*Graphics and Animation*) which can be possible (*Vectors*) or (*Contours*) .

The following figure shows (Contours) of temperature for the body:



As shown in the video and our book.

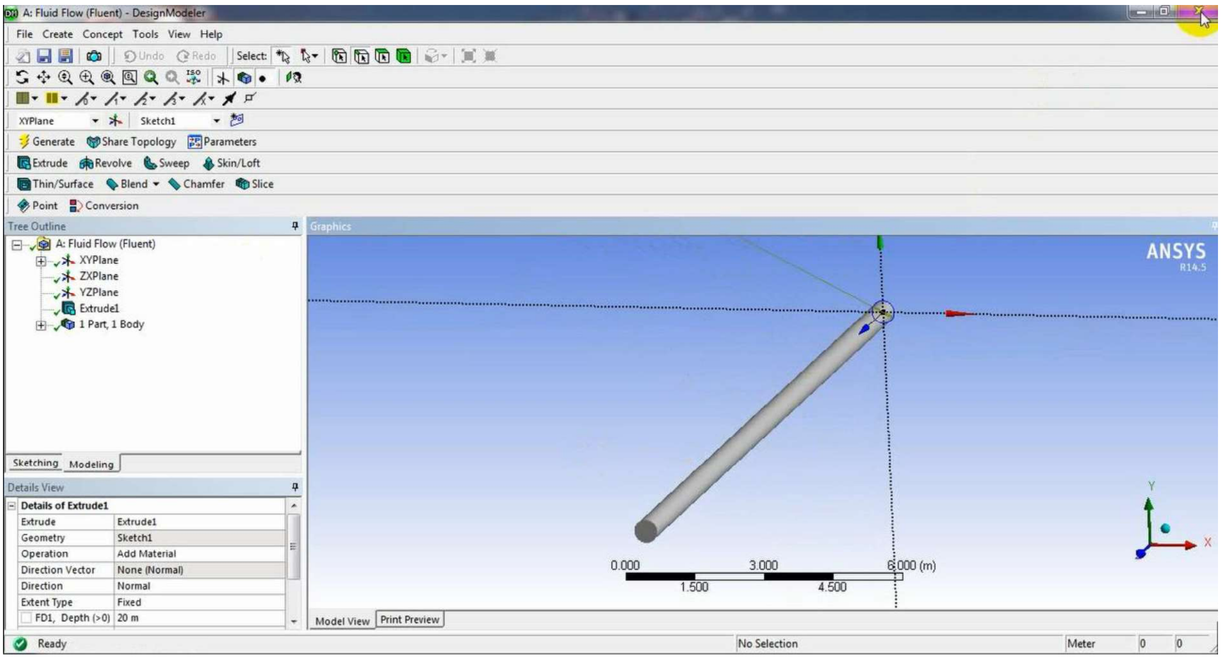
Tutorial Eleven

Fluid Flow (Fluent)

Laminar Flow

Select Analysis System *(Fluid Flow Fluent)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

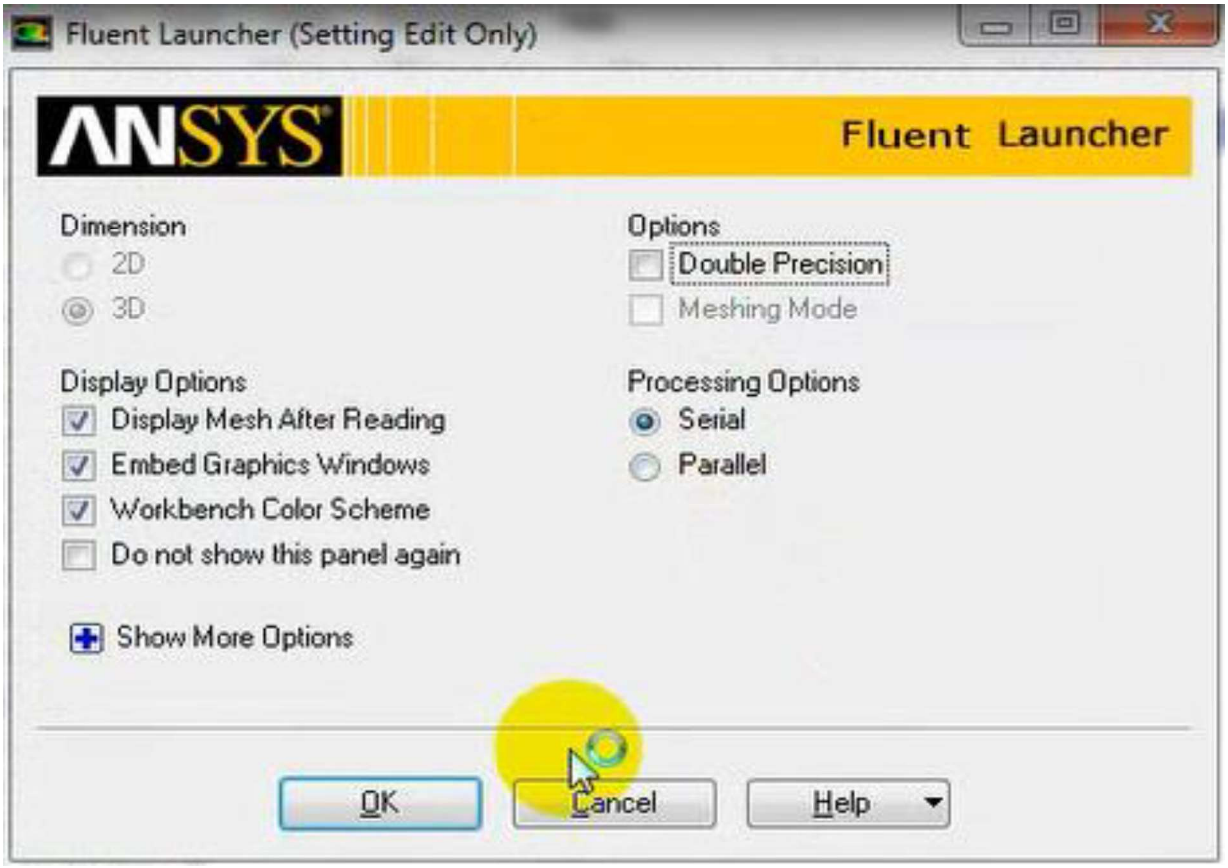
Then design model in the form of a pipe with diameter (*0.25 m*) and length (*20 m*) by using (*Design Modular*) So as to simulate the flow of air inside the pipe as shown in the following figure:



Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved, As well as naming the surfaces that will be set later example for an example of the current surface are named on the entry and exit them (*inlet*) and (*outlet*).

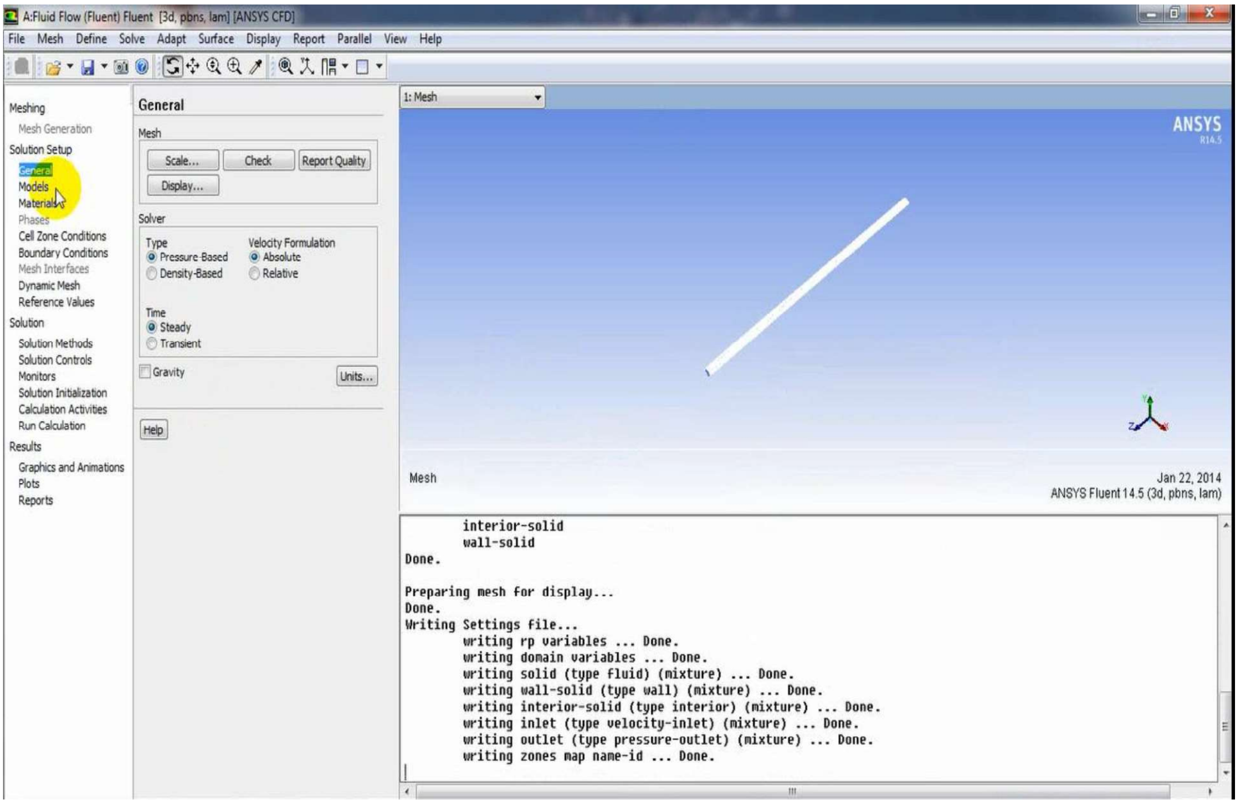
After making the mesh and obtaining the required number of slides are going to the next step, which is to open the program (*Fluent*) it is worth that the program (*Fluent*) and (*CFX*) Two of the programs that are used to simulate the movement of fluids and heat transfer processes and background are different from other programs within the existing (*Analysis System*) As will be reviewed later.

When you double click on the (*Setup*) will be opening the program window as shown below:

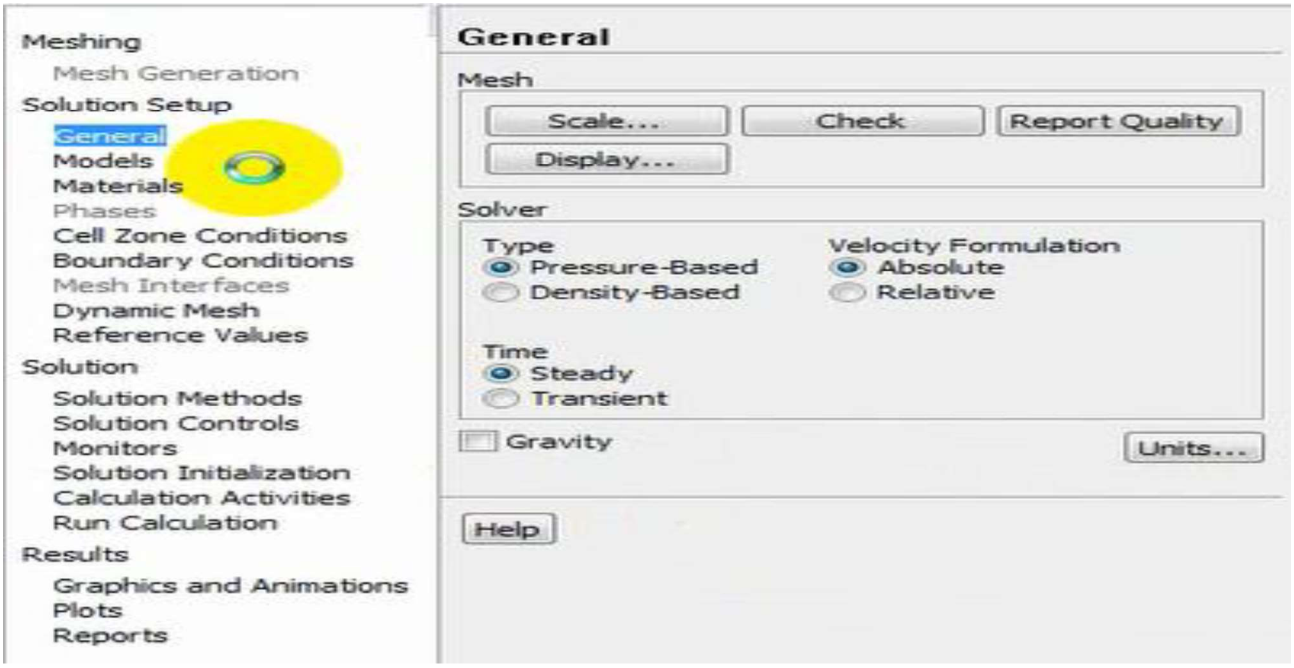


It was found window dialogue program choices available to the program where we can choose the dimensions between double and triple as well as precision where it can choose the accuracy doubling by choose (*Double precision*) and also can choose the number of processors that will be used to resolve through the selection of (*Parallel*) where it can do a number processors according to what is available on the computer that will be working on them where they can activate a single processor if the computer is used with a processor (*CORE i3*) while three processors can be activated if the computer is used (*CORE i5*) and so on.

And by clicking on the *OK* button will open the program window as shown below:



Where the window showing the model and the main menus which they can adjust the model, solved and review the desired results through it, where they are set through (*Model*)List (*Solution Setup*) as shown in the following:



Where this list begins in General setting (*General*), which set the type of solution and if the case is steady state or transition, as well as set the units and whether the case depends on the acceleration or not if they rely on accelerating are pointing the (*Gravity*) and set the value of the accelerating in the direction that acted in it.

And then adjust the models (*Models*) which can be set by:

The flow if it in (*Single phase*) or more then that (*Multiphase*).

Energy equation (*Energy Equation*), where it is activated that there was a heat transfer, with respect to the current case no heat transfer and therefore the situation remains *OFF*.

The type of flow if the flow (*Laminar*) or (*Turbulent*), for the present model the flow was laminar and for that the set remain as is assumed by the program.

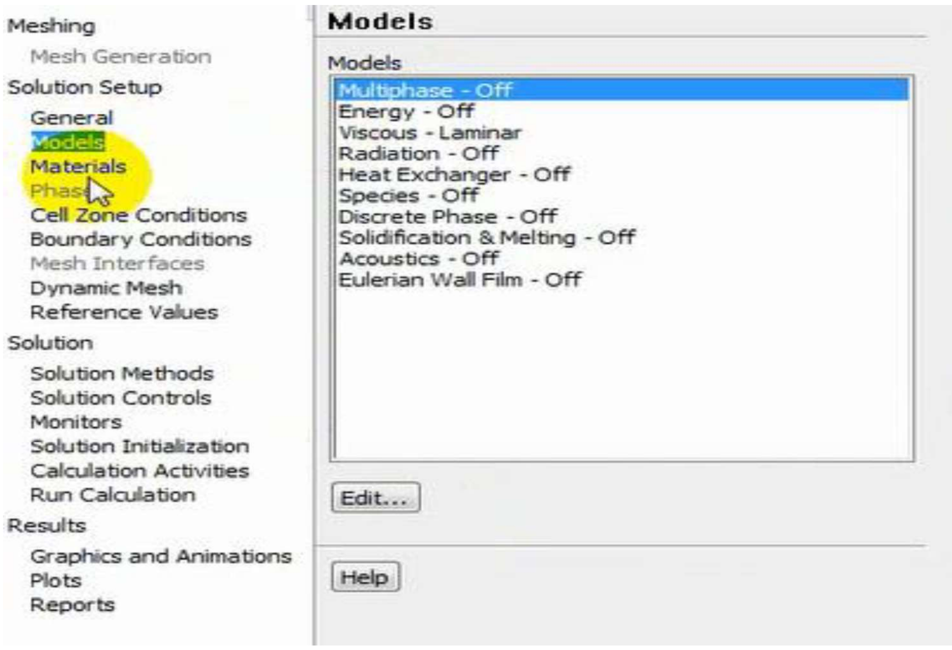
Radiation: Is active if there heat transfer by radiation from the body.

Heat Exchanger: Is there a heat exchanger where it is set through it.

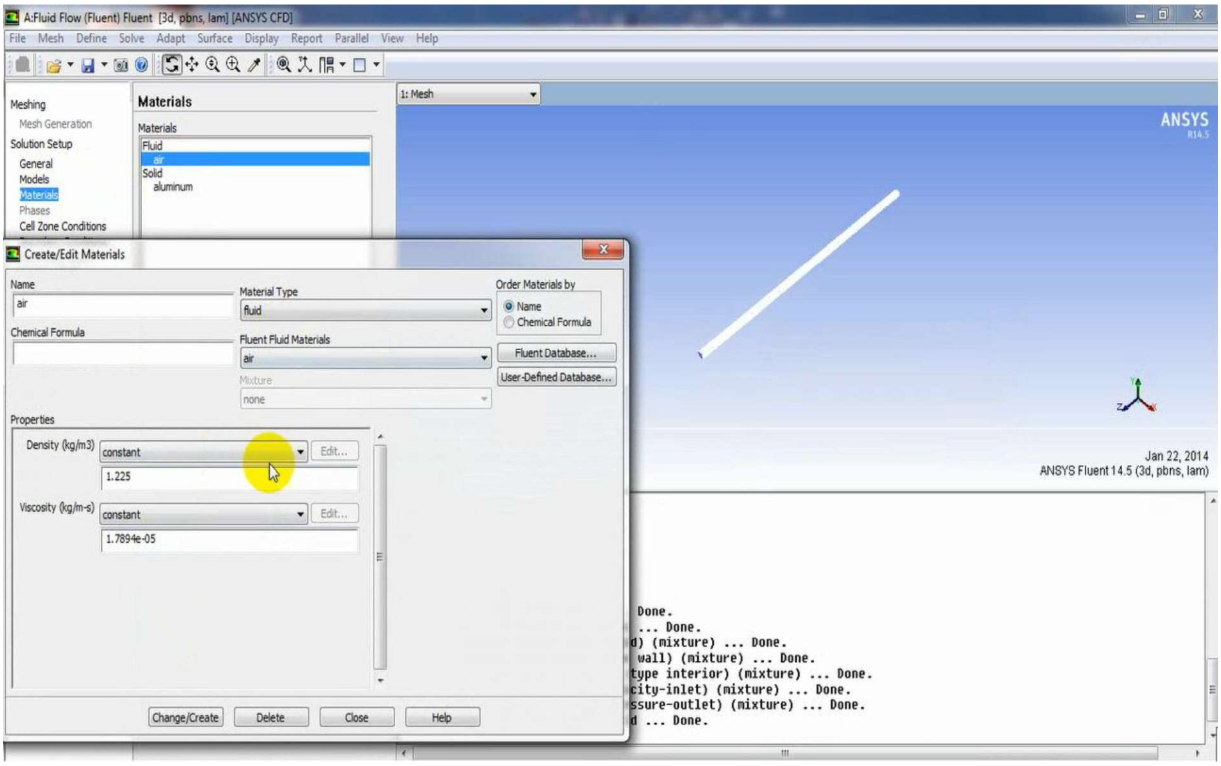
Discrete Phase: Is there a run for the two phases separate where they are set through it.

Solidification & Melting: Is there hardening or melting where they are set through it.

Acoustics: Is there the voices are caused by the model is set through it. as shown the following figure:



After set list of models is set to the type of material to work through the list (*Material*) as shown below:

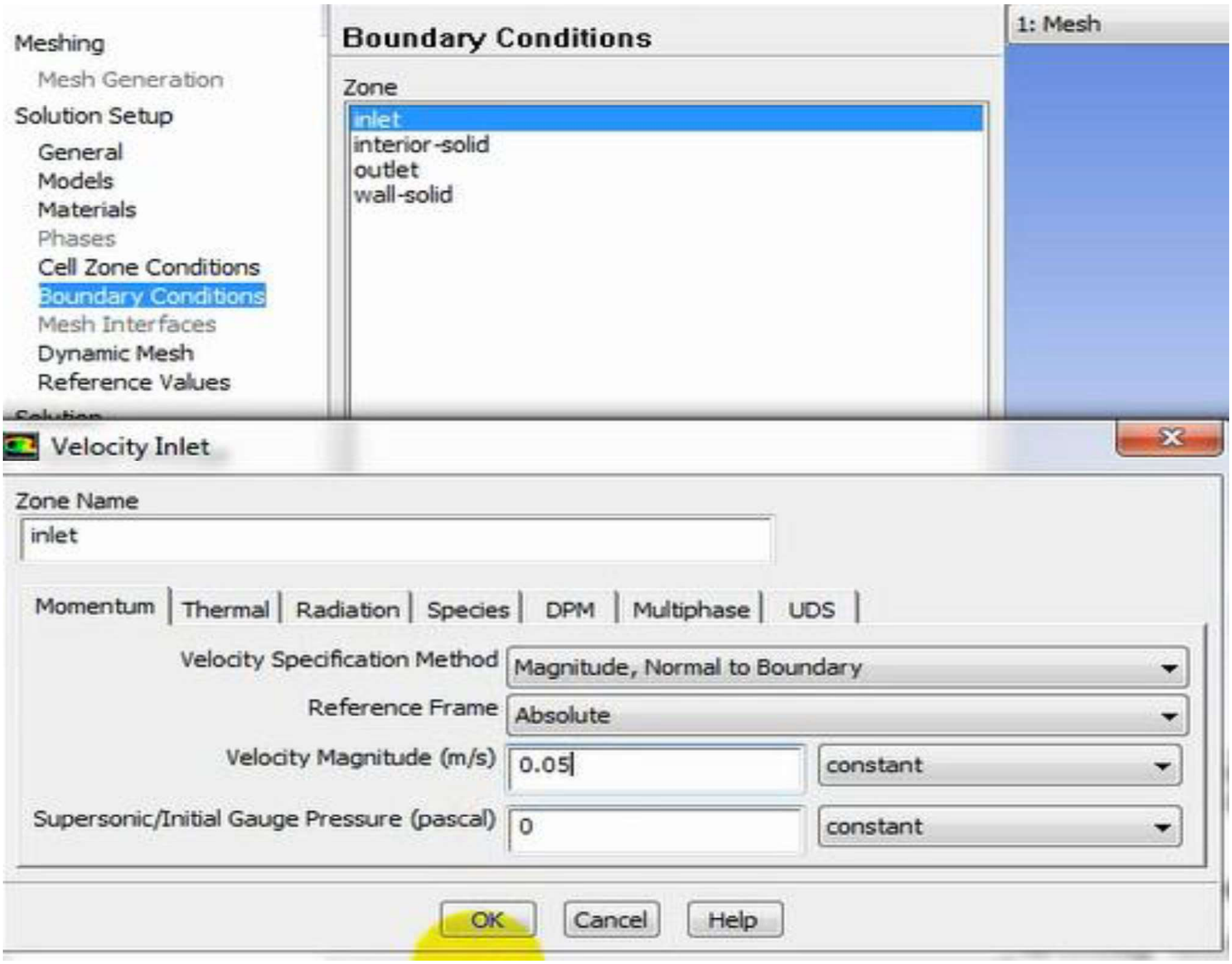


For the present model, the work material is air; it is selected and gives it (*Change / Create*) to be dependence by the program.

But if the material is non-air as if they are water or oil are selected through (*Fluent Database*).

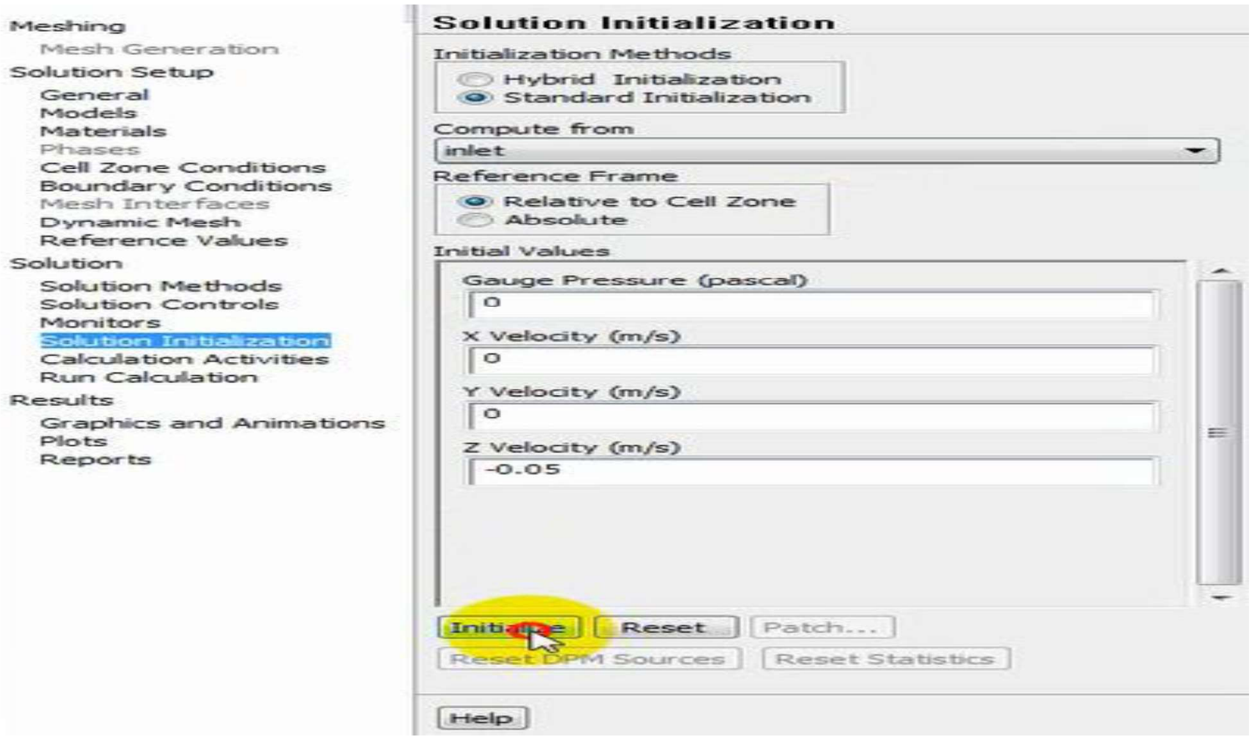
Then adjust the conditions are (*boundary conditions*) i.e. (applied loads) where they are applied loads depending on condition of model.

Where has been set conditions for the current example is the definition of surface access that entry velocity (*velocity inlet*) with velocity magnitude (*0.05 m*) as shown in the following figure:

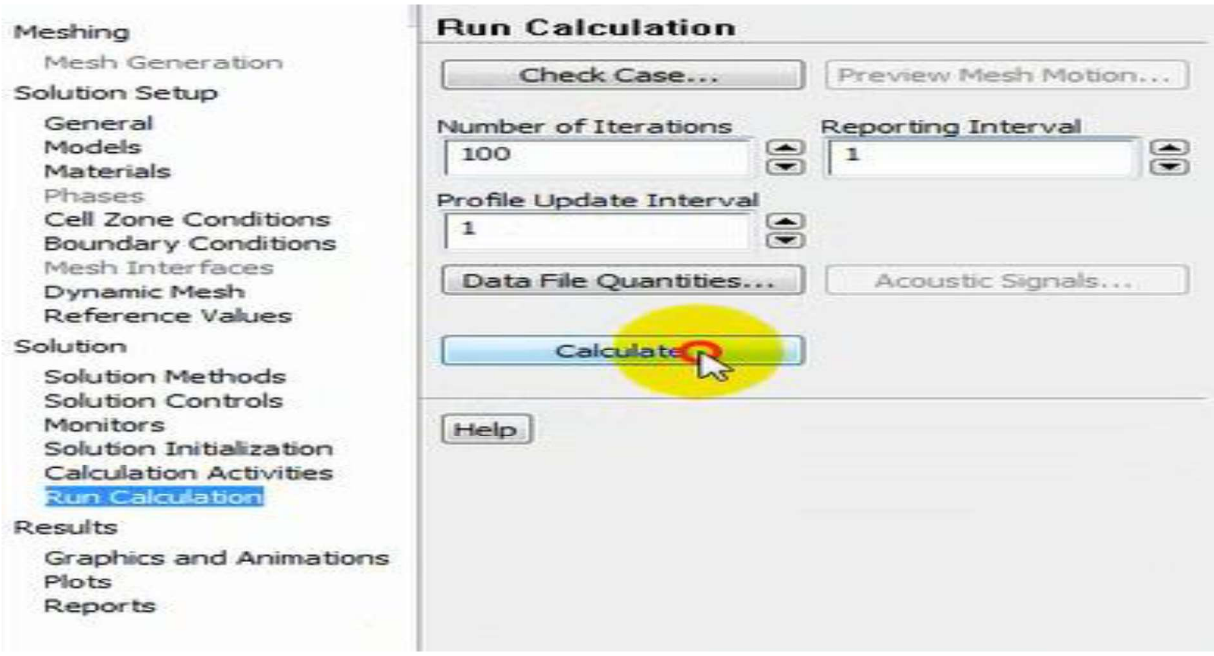


And defined exit surface (*outlet flow*), and the other surfaces are isolated and there is no any load on it.

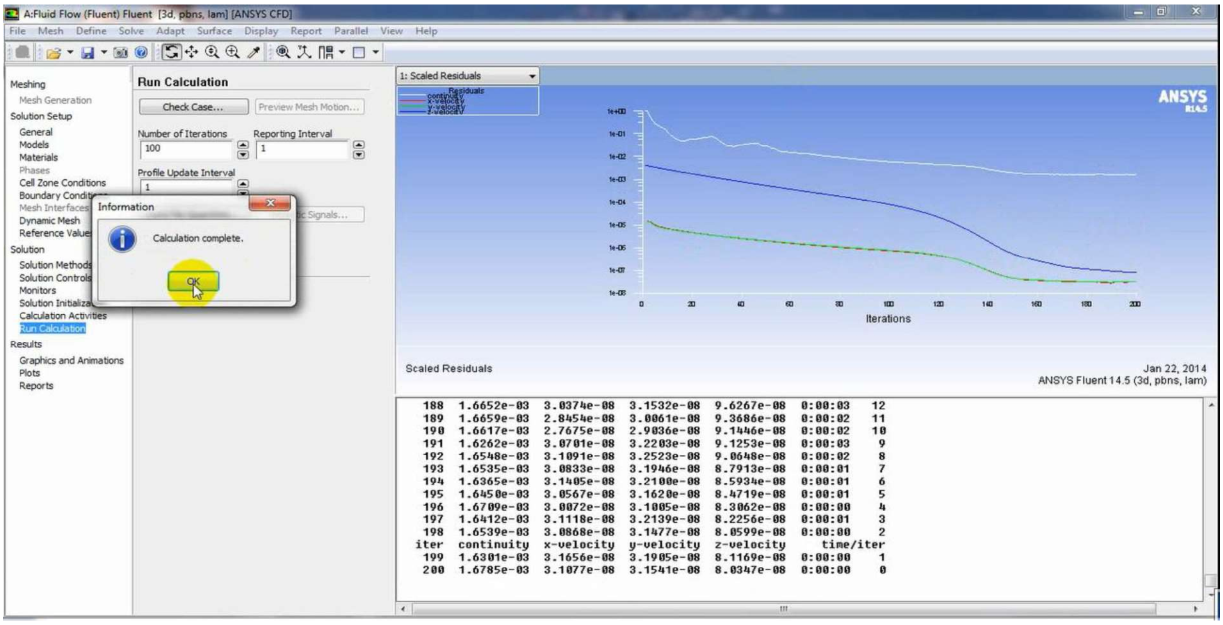
And after setting the boundary conditions are initialized solution(*Solution Initialization*)where we choose create a standard (*Standard Initialization*) can choose the configuration of any surface we want it to be as if the surface of the entry (*Inlet*) and as shown in the following:



After that the solution is initialized is solution the case where we choose (*Run Calculation*) and give the number of times the correction with respect to the present example have been set number of iteration (*100*) and then give (*Calculate*) as shown in the following figure:

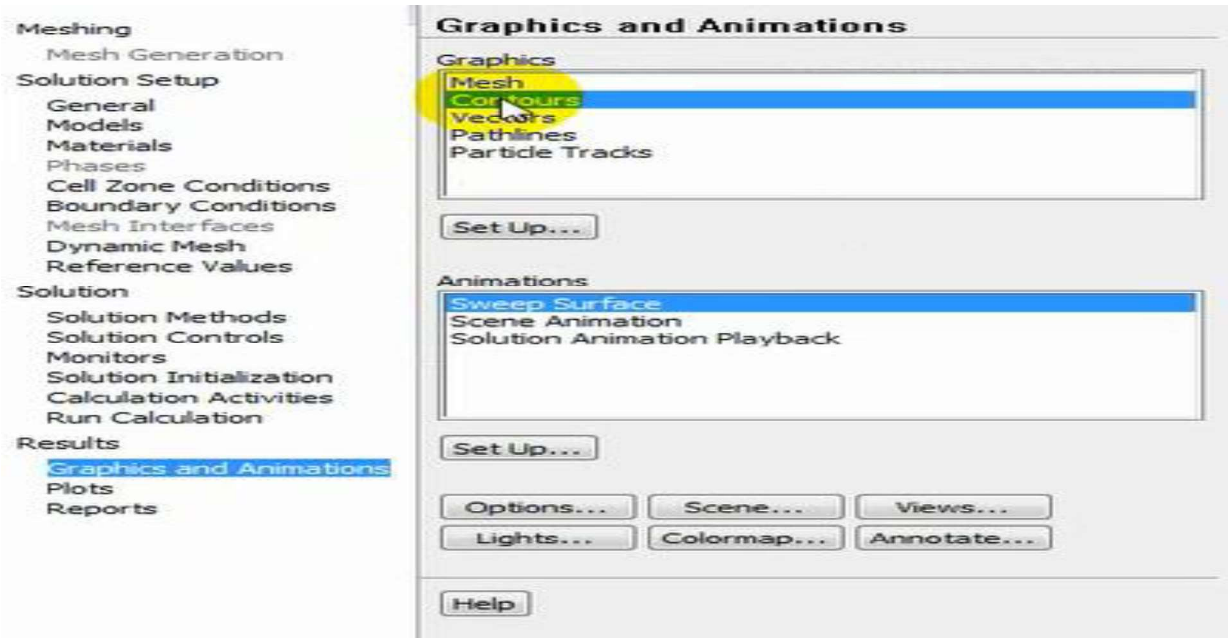


After the solution complete the program gives us message that the solution complete as shown in the following figure:

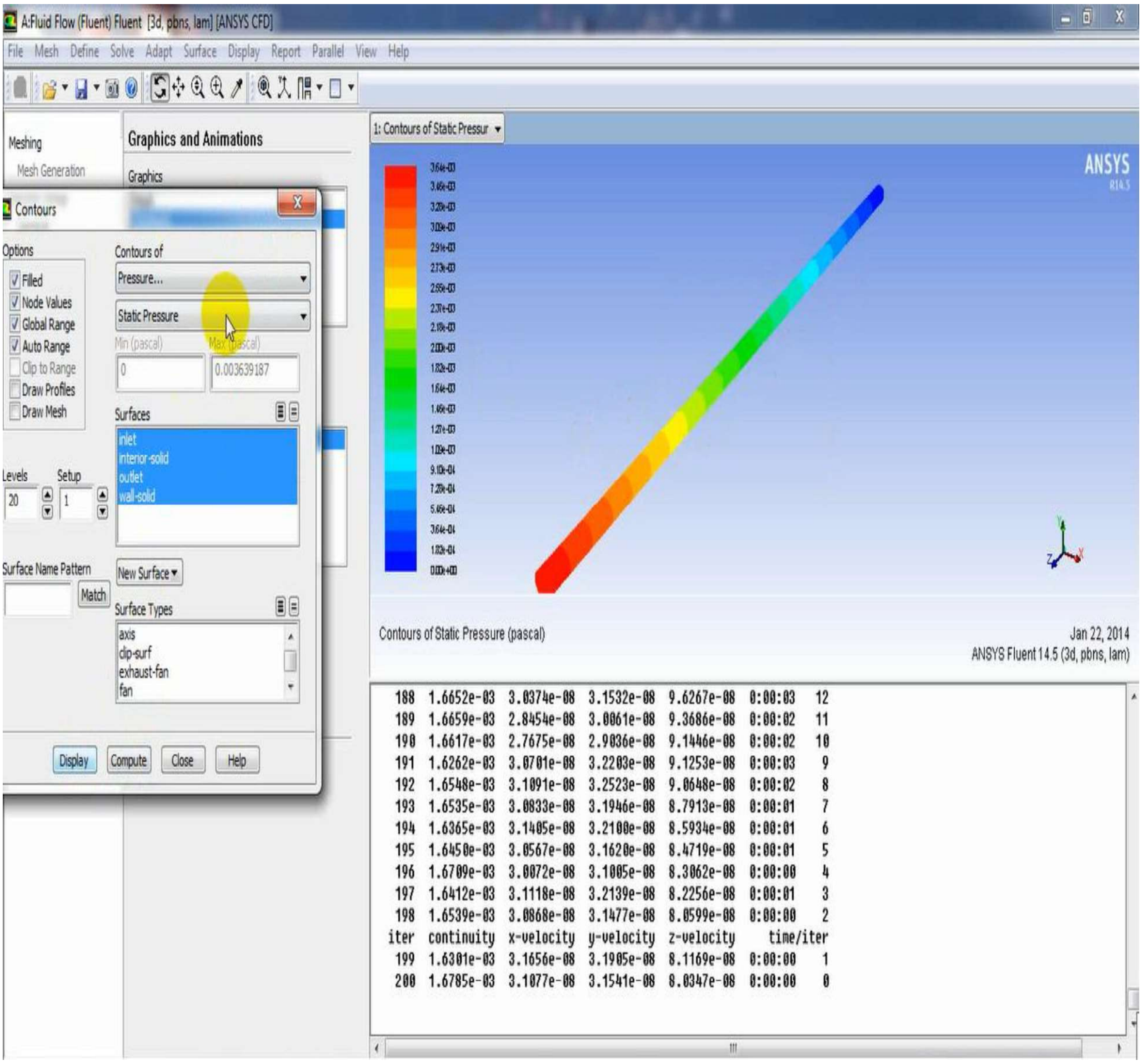


After that we can reviewing the results it can be possible (*Graphics and Animation*) or (*Plots*) or (*Reports*).

Relative to (*Graphics and Animation*) which can be possible (*Vectors*) or (*Contours*) and as shown below:



The following figure shows (Contours) pressure for the body:



As shown in the video and our book.

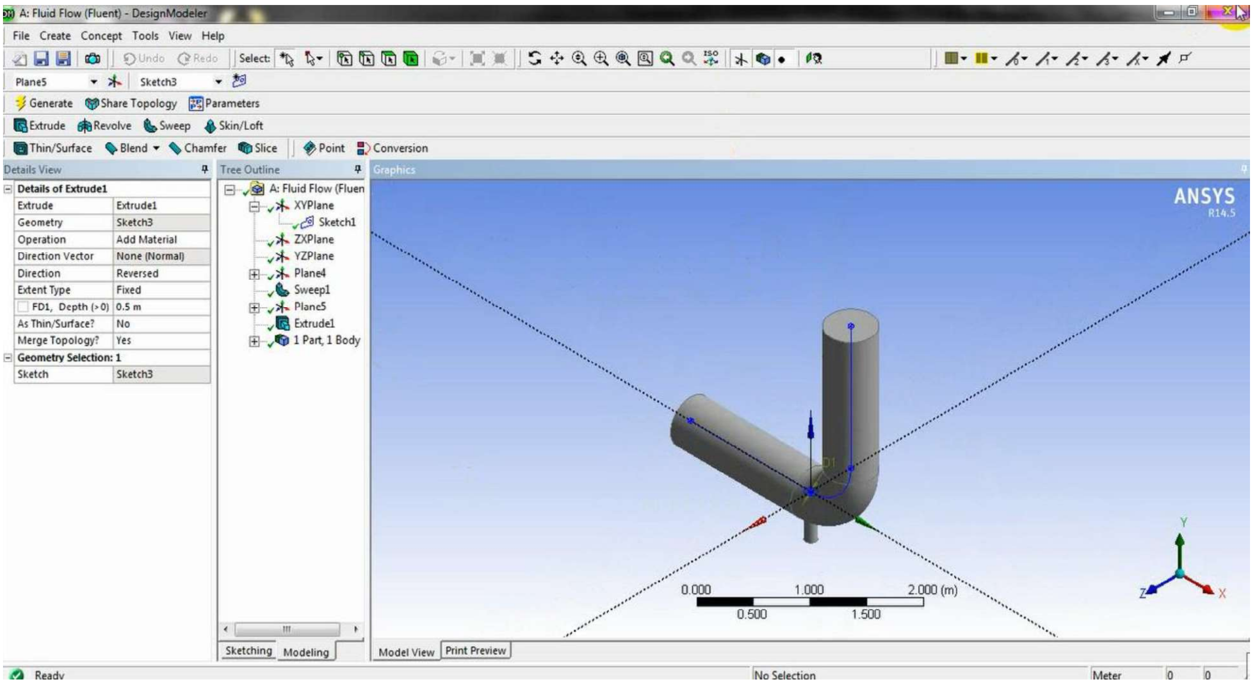
Tutorial Twelve

Fluid Flow (Fluent)

Turbulent Flow

Select Analysis System *(Fluid Flow Fluent)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

Then design model in the form of elbow with diameter (*0.5 m*) and arm length (*2 m*) as main stream and inject stream with diameter (*0.125 m*) and length (*0.5 m*)by using (*Design Modular*) So as to simulate the flow of air inside the pipe as shown in the following figure:



Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved, As well as naming the surfaces that will be set later example for the present example are named the two entry surface the first entry to main stream with name (*inlet1*) and the second for the inject stream with name (*inlet2*) and also name the exit with name (*outlet*).

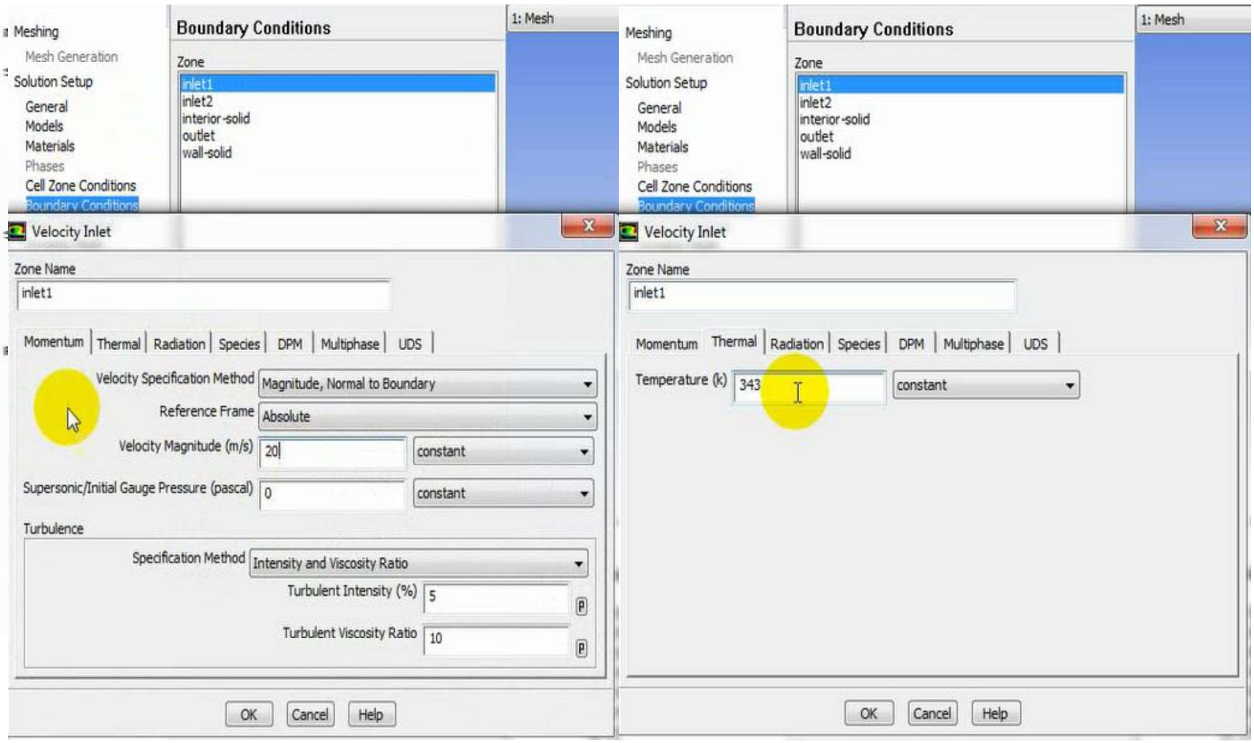
After making the mesh and obtaining the required number of slides, the next step is to open the program (*Fluent*).

Where we set the present model with activation (*energy equation*) and choose the flow type as (*turbulent flow*) and the type of turbulent is (*k-e*).

For the present model, the work material is air; it is selected and gives it (*Change / Create*) to be dependence by the program.

Then adjust the conditions are (*boundary conditions*) i.e. (applied loads) where they are applied loads depending on condition of model. For the present model we set the velocity for the (*Inlet1*) with value (*20 m/s*) and temperature (*343* k) for the velocity for (*Inlet2*) with value (*10 m/s*) and temperature (*300 k*) as shown below:

Now the exit surface is defined (*Outlet flow*), and the other surfaces are isolated and there is no any load on them.

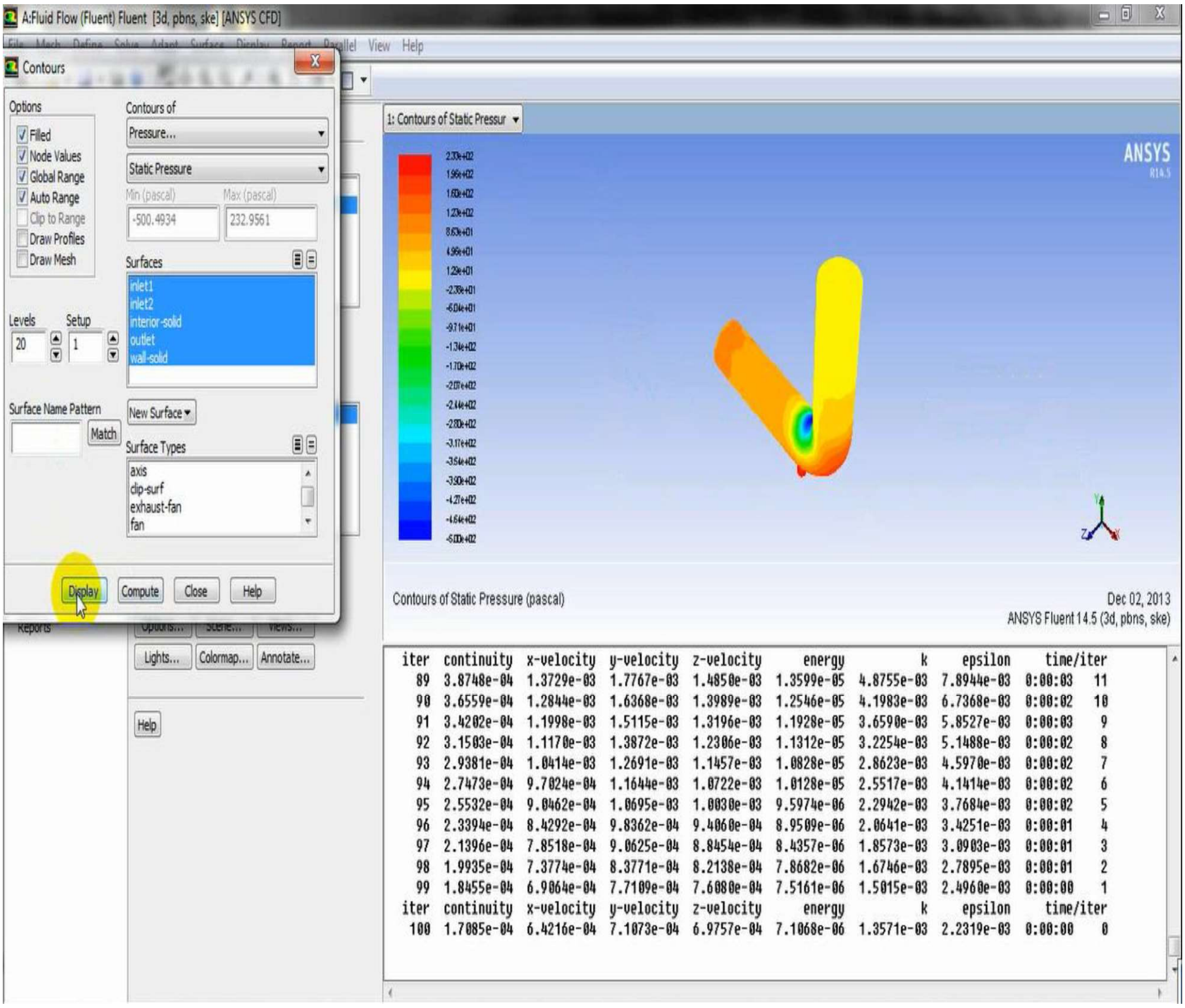


And after setting the boundary conditions are initialized solution(*Solution Initialization*)where we choose create a standard (*Standard Initialization*) can choose the configuration of any surface we want it to be as if the surface of the entry (*Inlet*) and as shown in the following:

After that the solution is initialized. the case where we choose (*Run Calculation*) and give the number of times the correction with respect to the present example have been set number of iteration (*100*) and then give (*Calculate*) .

After the solution complete the program gives us message that the solution complete. After that we can reviewing the results it can be possible (*Graphics and Animation*) or (*Plots*) or (*Reports*).Relative to (*Graphics and Animation*) which can be possible (*Vectors*) or (*Contours*) .

The following figure shows (*Contours*) pressure for the body:

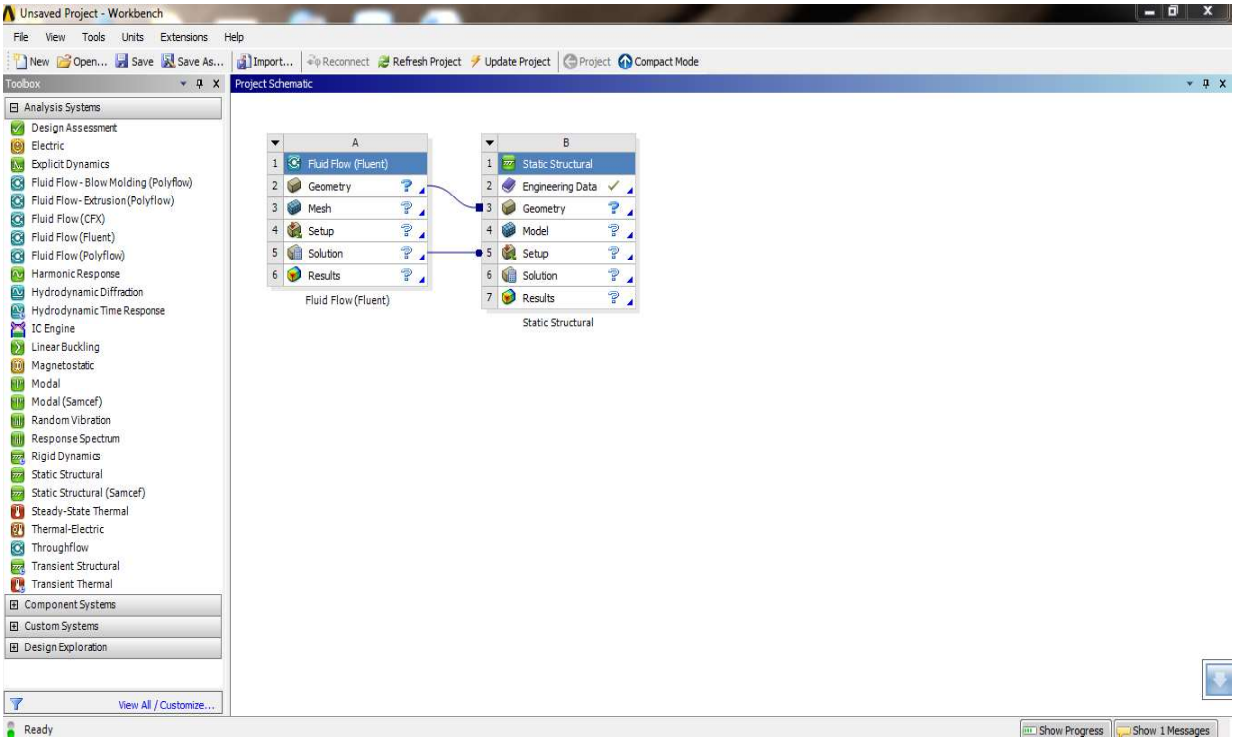


As shown in the video and our book.

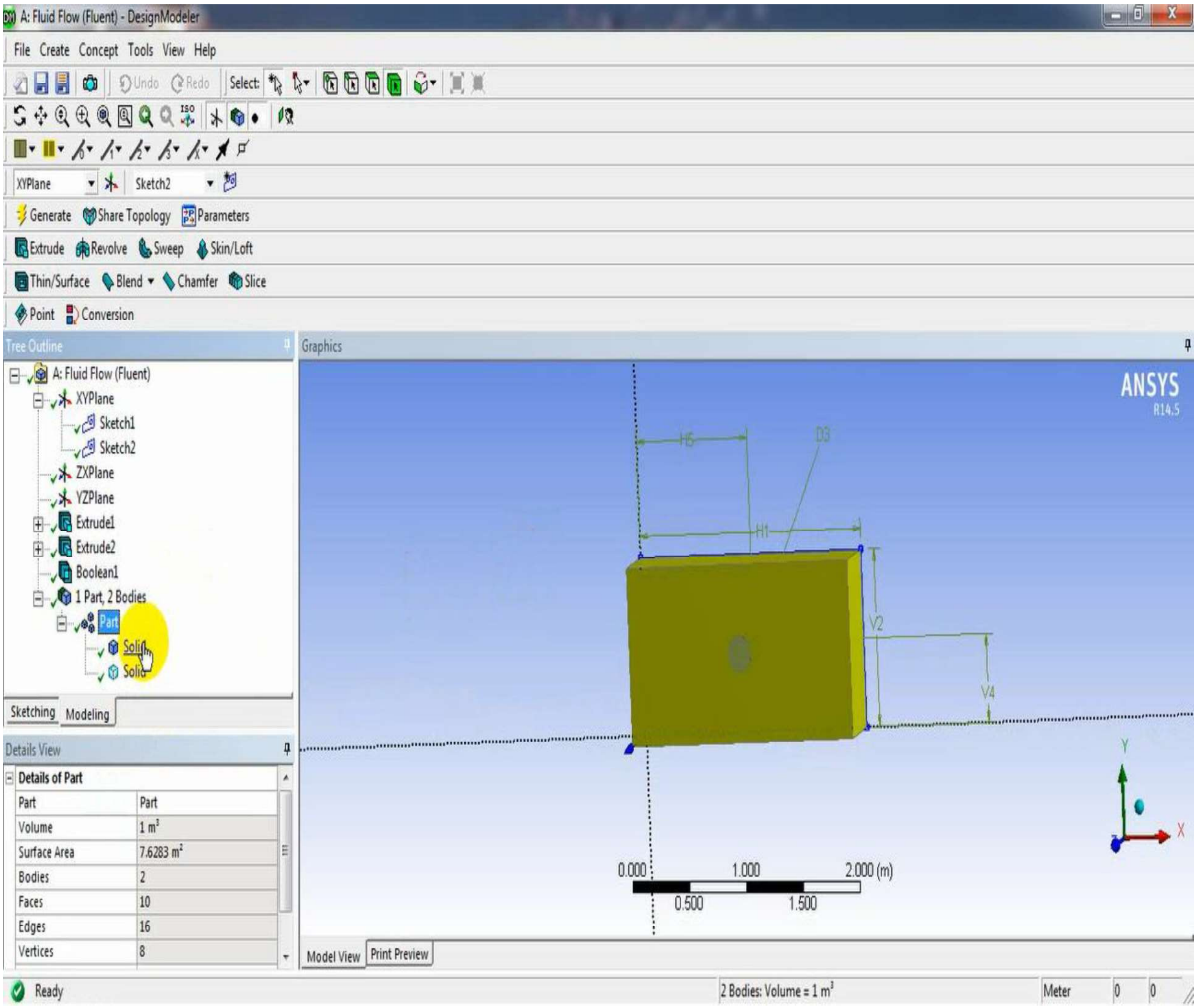
Tutorial Thirteen

Fluid Flow (Fluent) + Static Structure Interaction

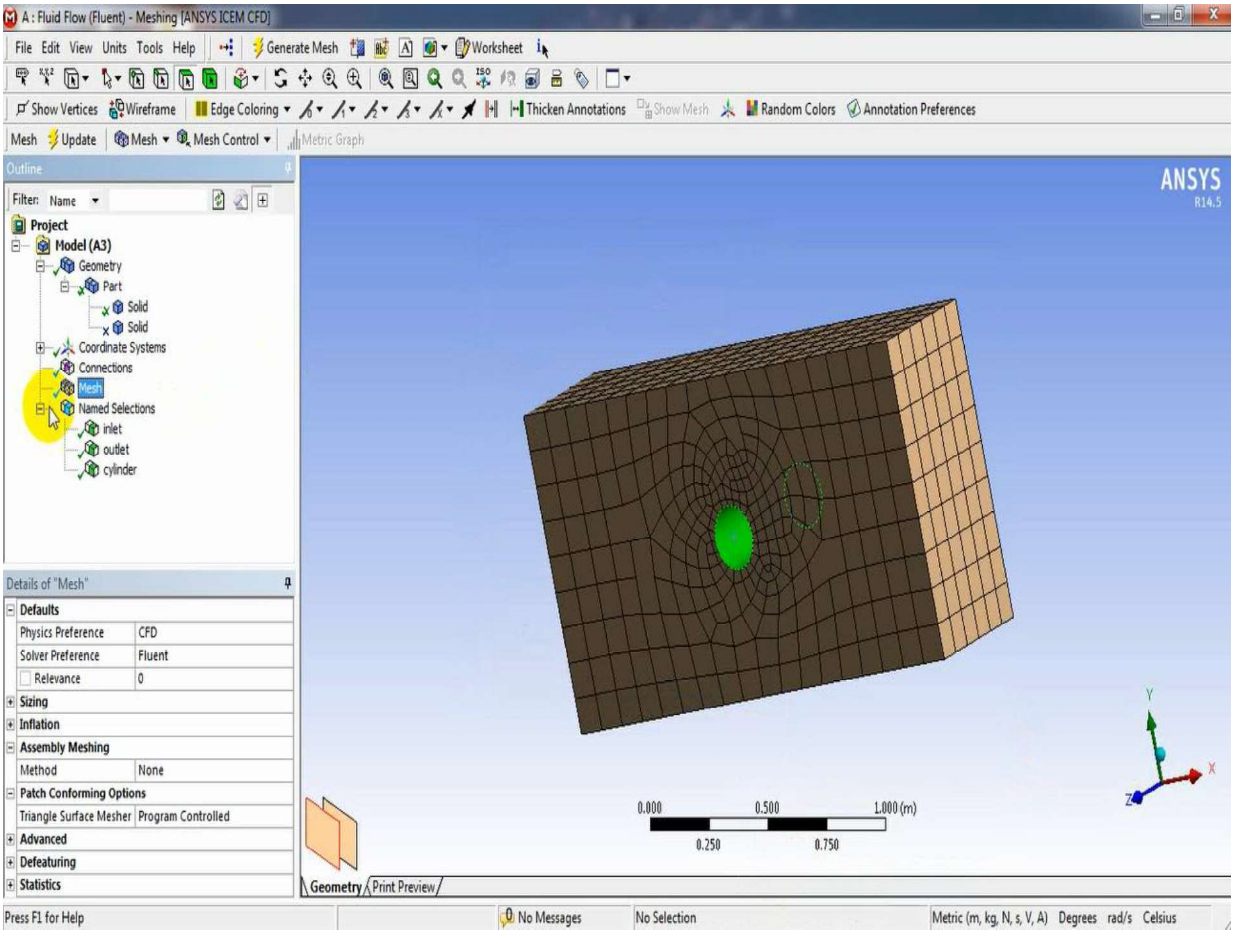
Select Analysis System *(Fluid Flow Fluent)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then be select (*Static Structure*) from the analysis system and by dragging and dropping on the solution of the first analysis system (*Fluid Flow Fluent*), it is linking the (*Geometry + Solution with Setup*), as is shown in the following figure:



Then design model in the form of rectangular with dimensions (*2×1 m*) and extrude (*0.5 m*) which as flow geometry and then create cylinder in diameter (*0.2 m*) as structure geometry by using (*Design Modular*) and then subtract the two geometry by using Boolean feature as is shown in follow figure:



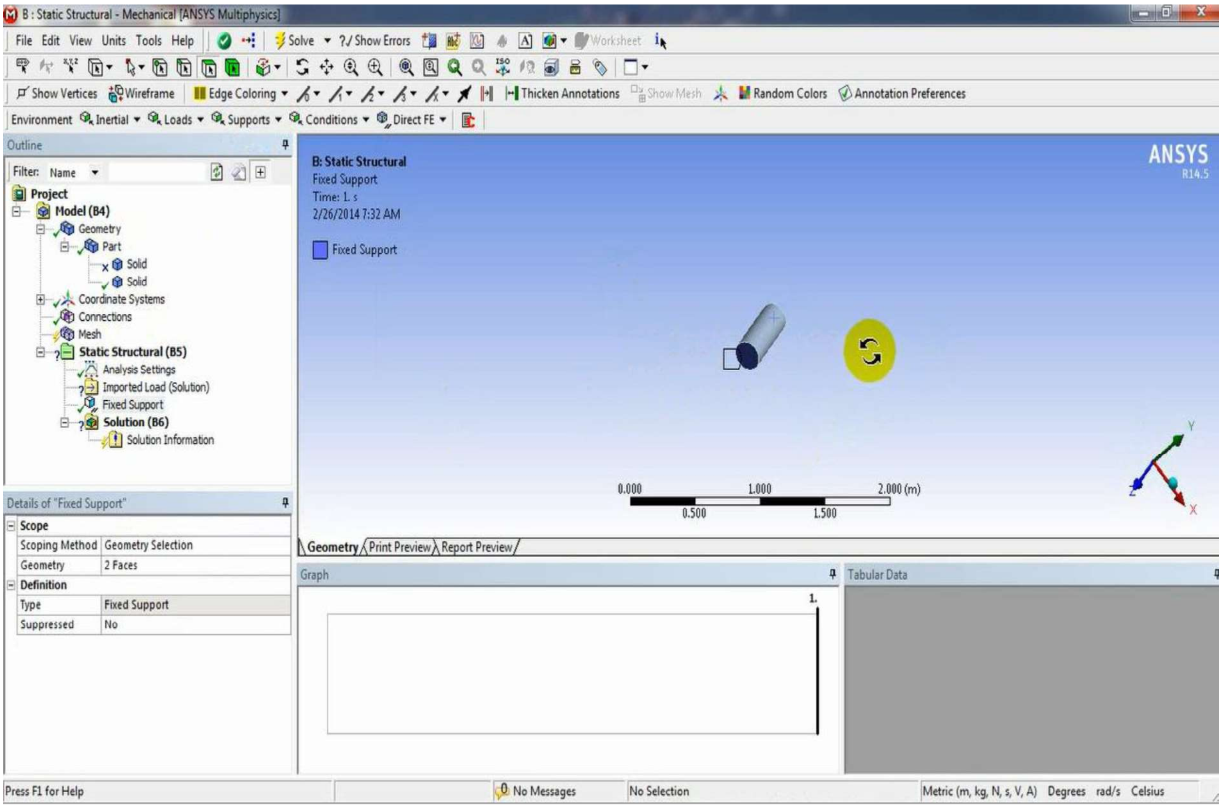
After the geometry is created, go to the next step which is mesh generation for the geometry where in this step we must suppress the geometry for the (*Static Structure*) and make mesh for the (*Fluid Flow Fluent*) geometry and named the surfaces which will setup it in the boundary condition in the the setup step, these surfaces are (*inlet*),(*outlet*) and the surface of cylinder which act the intersection surface between the flow and structure this surface nemed (*cylinder*) which allow in import the load on the cylinder from (*Fluid Flow Fluent*) when set the load of (*Static Structure*) analysis system,as shown in the following figure:



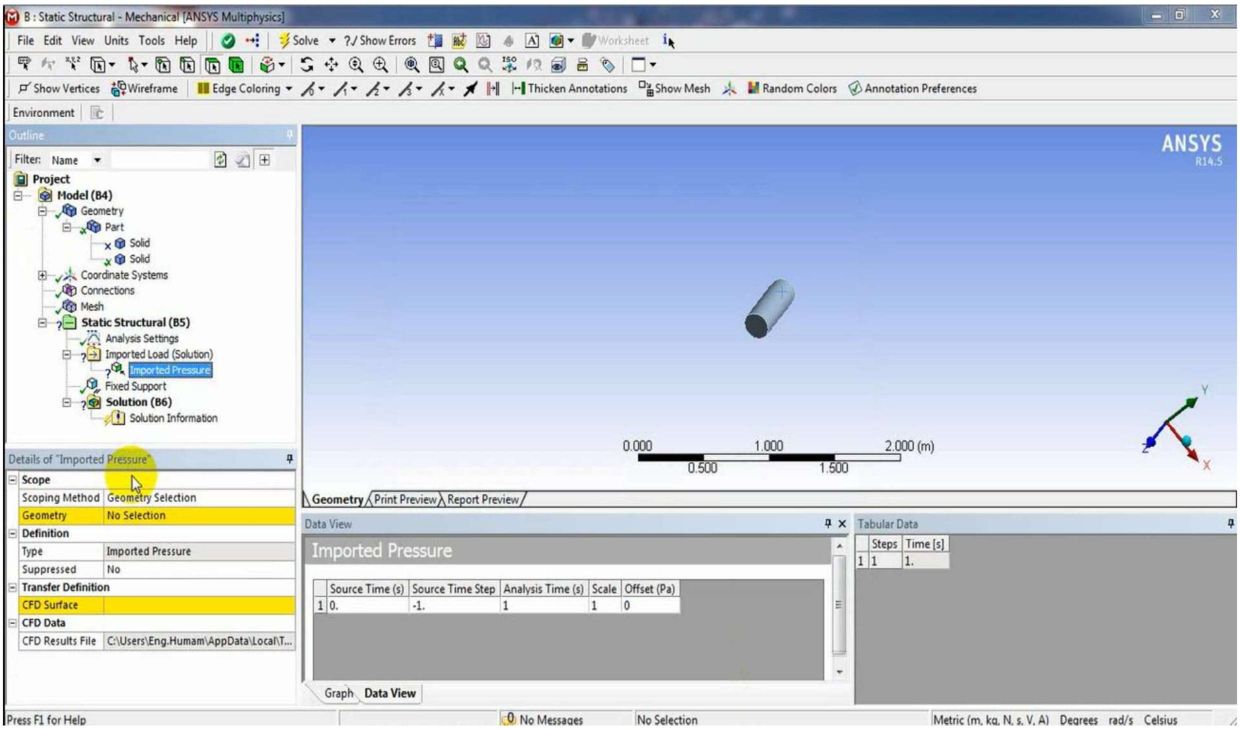
After the mesh done is set the boundary condition in the setup step where open the window of fluent and begin in setup where choose the flow type is (*Turbulent K-e*) and the work material is (*Air*) and then setup the boundary condition for the inlet surface which is set as (*Velocity Inlet*) in magnitude (*10 m/s*) and set the outlet surface as outlet flow or pressure outlet the cylinder surface are remain wall as default setting and then

(*Initialization*) the solution from the (*Inlet Surface*) and then make(*Run*) after that the solution was completely and can review the required results.

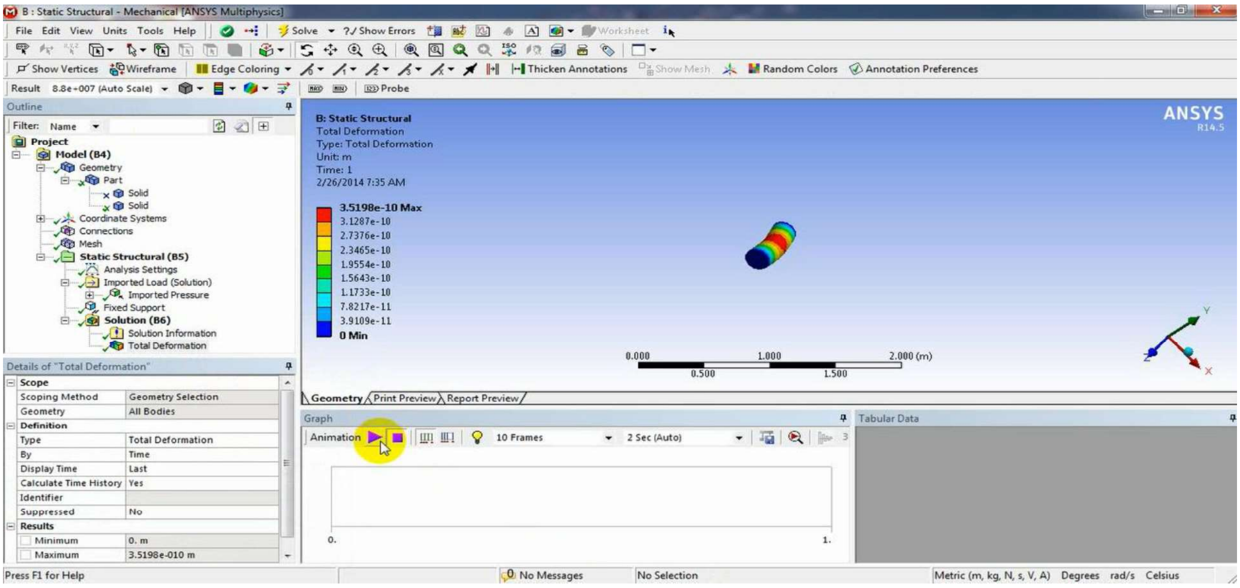
After that we go the (*Static Structure*) where active the Structure geometry and suppress the fluent geometry and then make mesh and setup the boundary condition which is fixed support from two side as shown in the figure:



And import the pressure load from the (*CFD*) surface which is the cylinder wall and applied it on the outer wall of cylinder structure as shown in the figure:



And then solve the case and review the required results such as deformation, stress, strain etc. the following figure show the deformation result:



As shown in the video and our book.

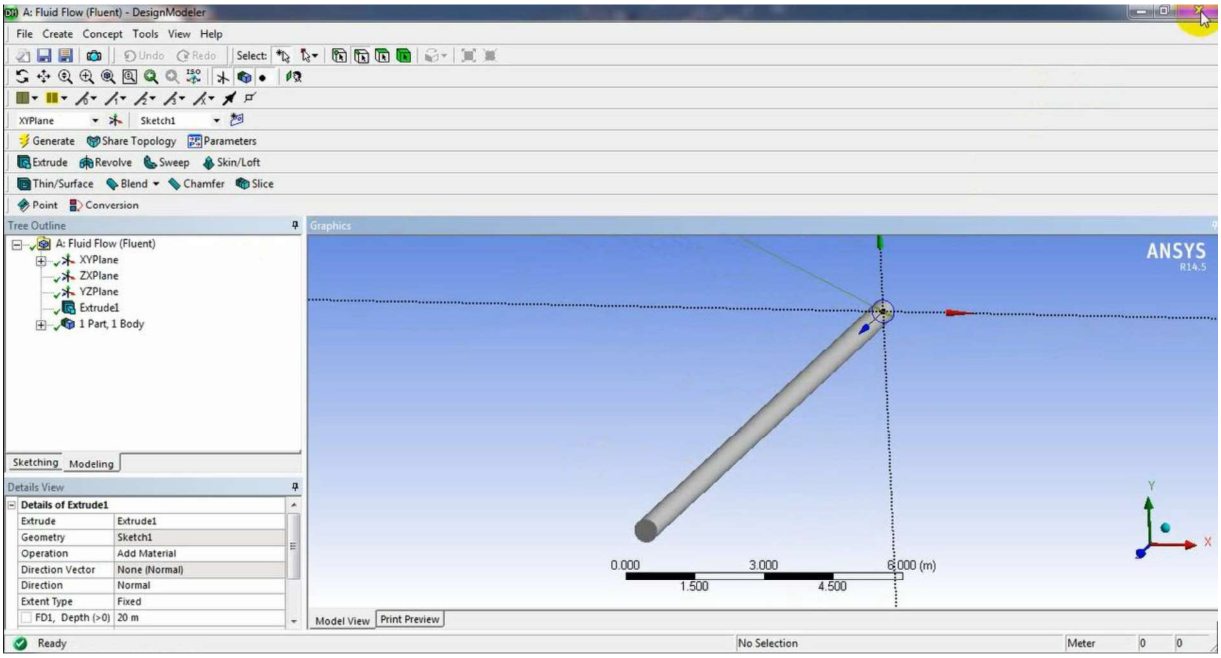
Tutorial Fourteen

Fluid Flow (CFX)

Laminar Flow

Select Analysis System *(Fluid Flow Fluent)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then the test material is selected.

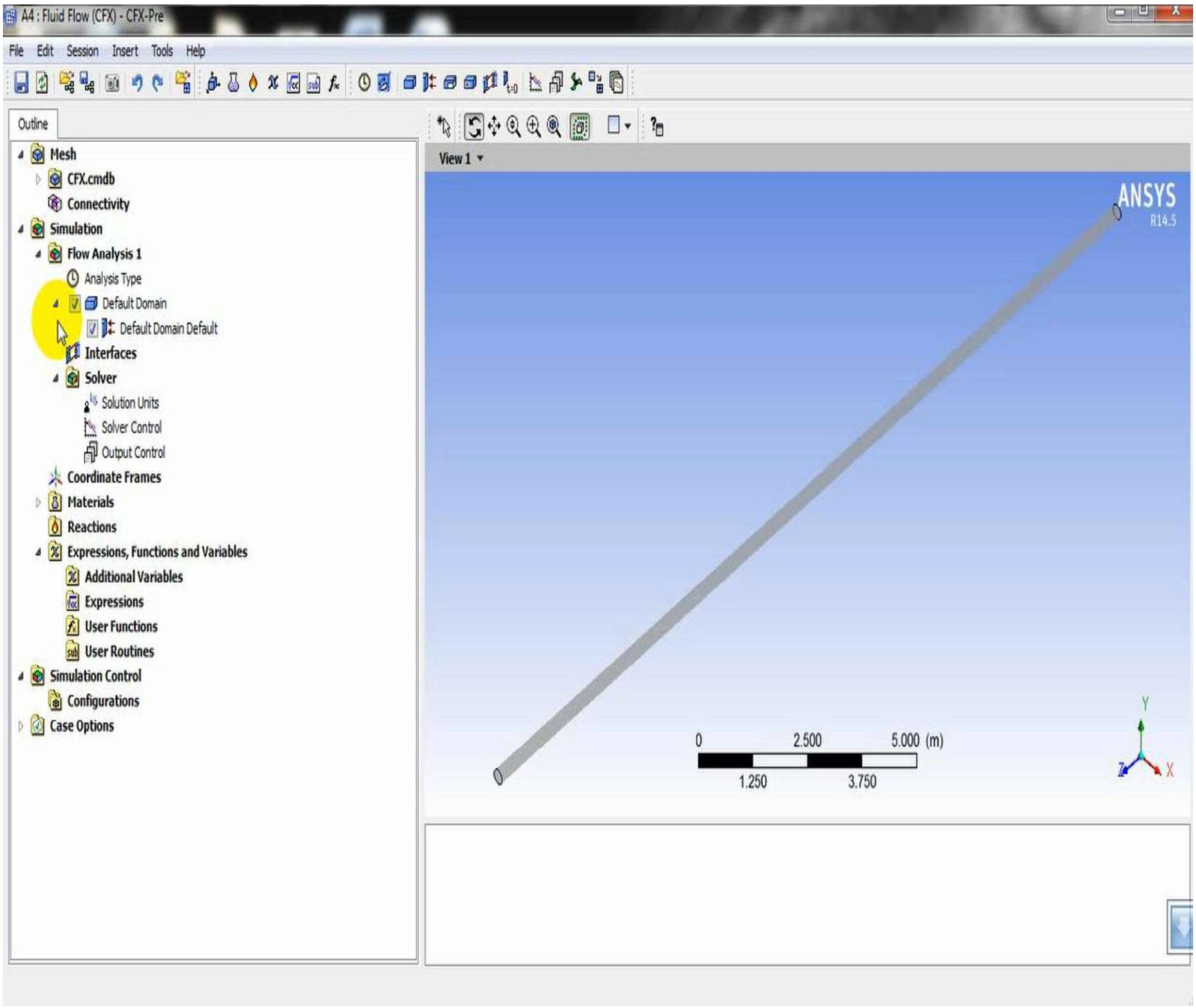
Then design model in the form of a pipe with diameter (*0.25 m*) and length (*20 m*) by using (*Design Modular*) So as to simulate the flow of air inside the pipe as shown in the following figure:



Then make the mesh where the mesh type and size of the cell can be controlled to suit the situation to be solved, As well as naming the surfaces that will be set later example for an example of the current surface are named on the entry and exit them (*inlet*) and (*outlet*).

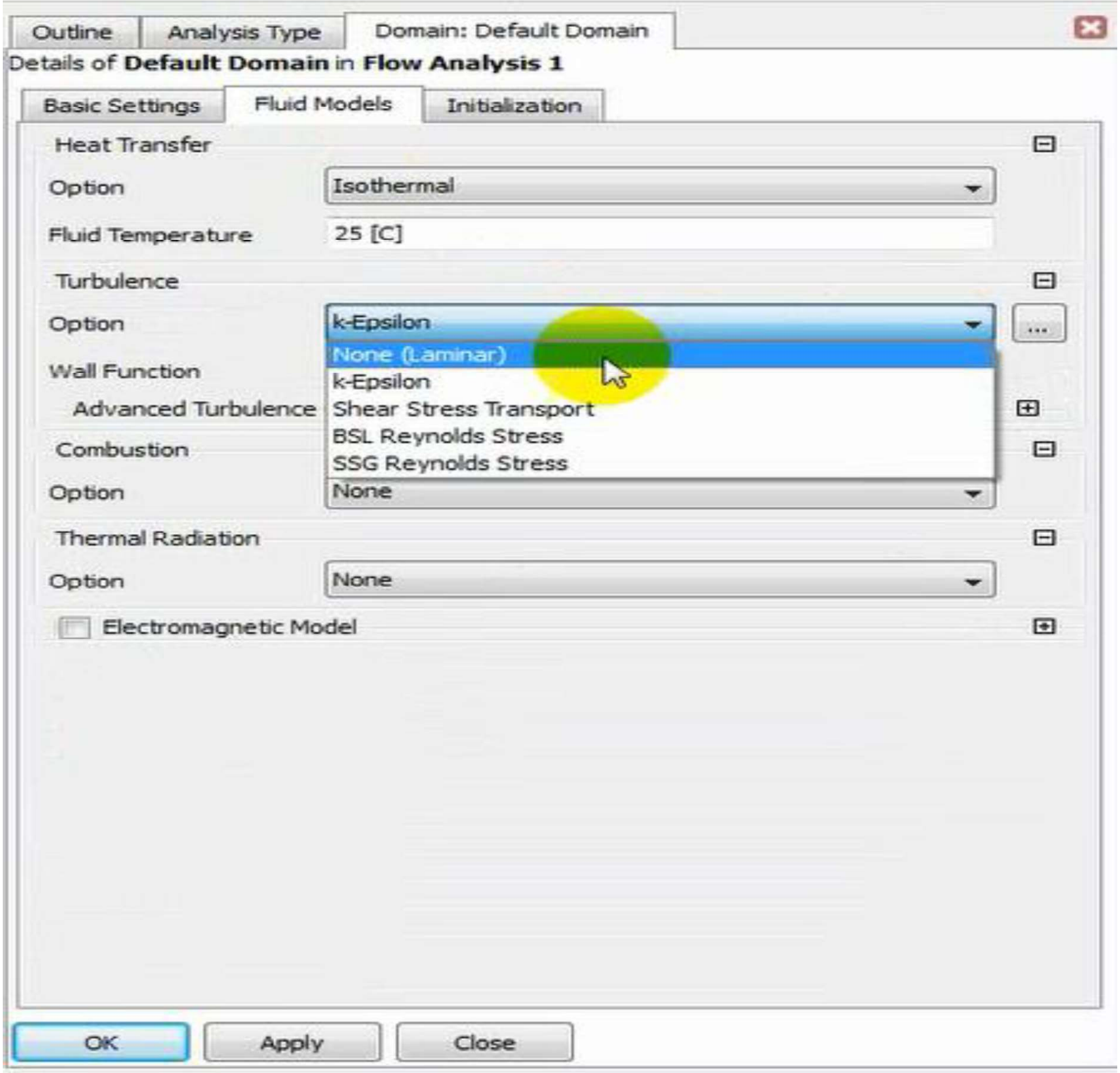
After making the mesh and obtaining the required number of slides, the next step is to open the program (*CFX*).

When you click twice on the (*Setup*) will be opening the program window as shown below:



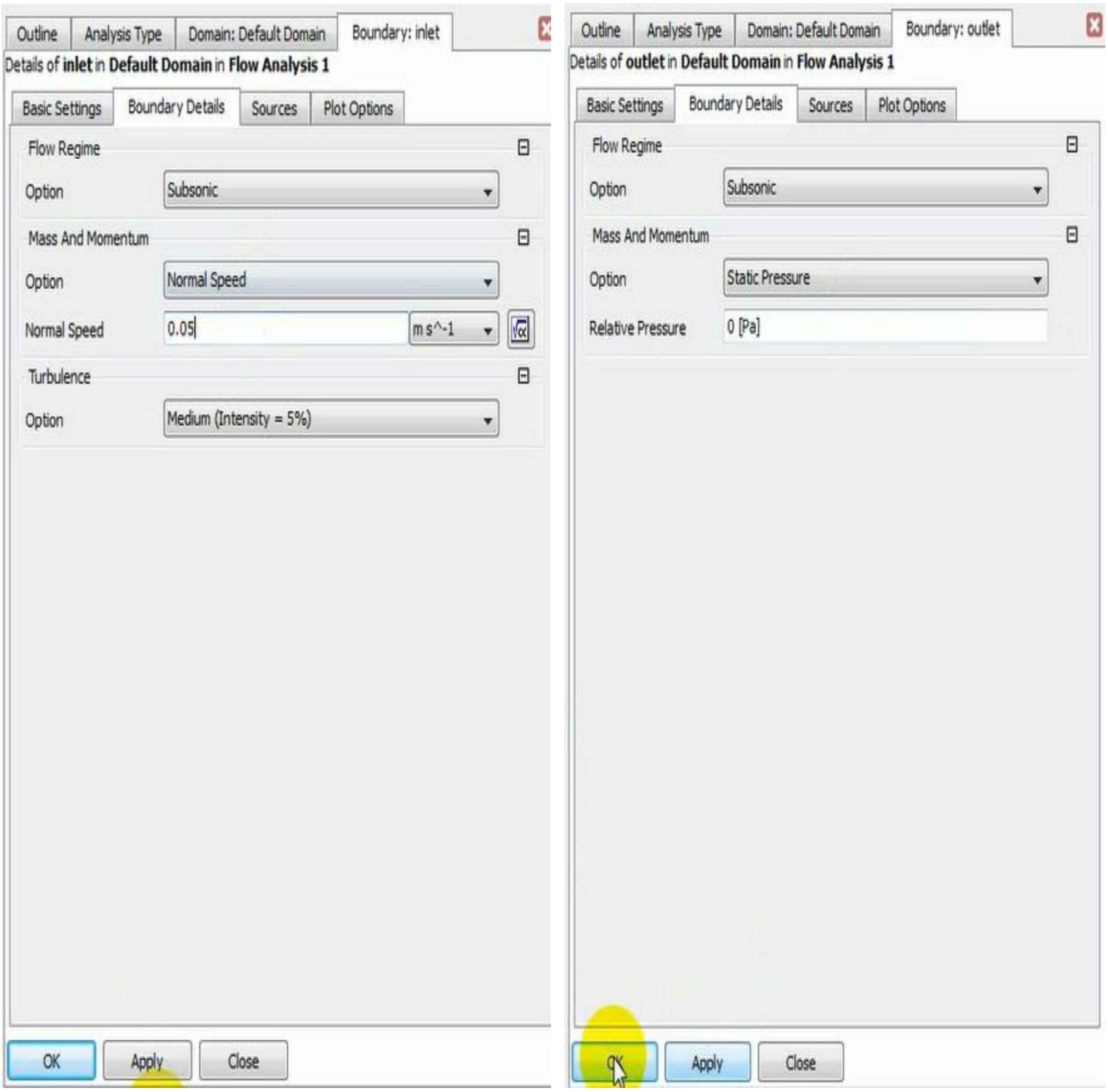
Where the figure shows the program window and the program main list, the program is one of the programs that are used to simulate the fluids flow and heat transfer processes.

we can adjust the model through the window (*Outline*) which are selected in places if there is a heat transfer, as well as whether or not the type of flow laminar or turbulent as show through the following window:



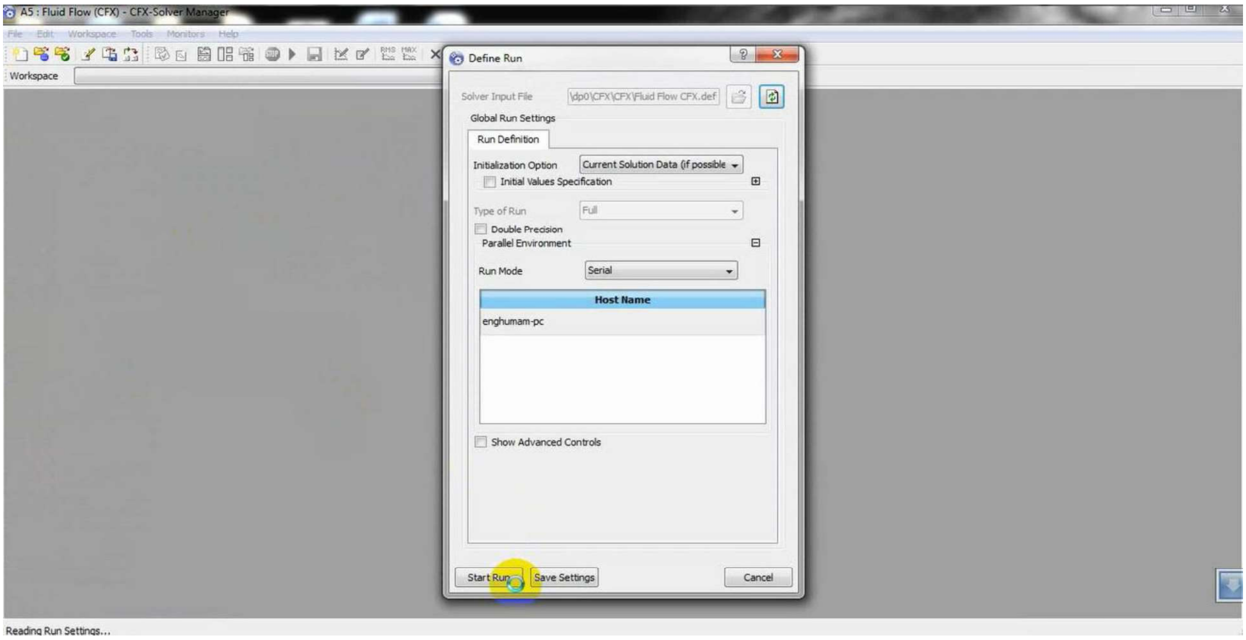
for the present example, the model does not contain the heat transfer and (*Laminer*) flow type.

It is then adjust the boundary conditions where the boundary conditions for the current example is the entery flow where defined as (*Velocity Inlet*) with value (*0.05 m / s*), either outlet flow is defined as (*outlet*) and chose the static pressure type with value (*0 pa*) as shown in the following figure:

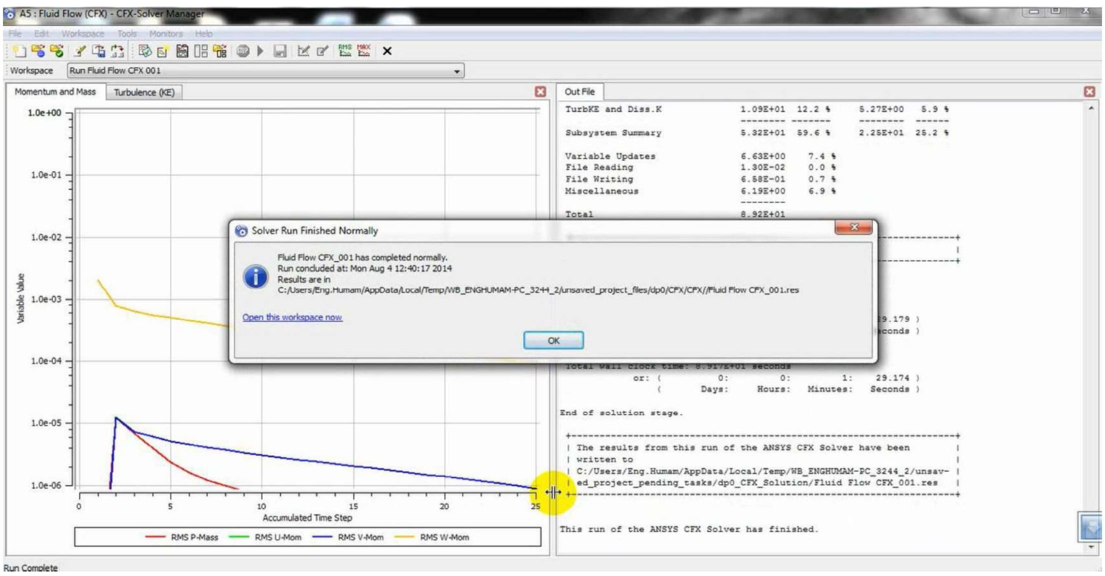


After that is set boundary conditions are initialized solution by given it (*Initialization*).

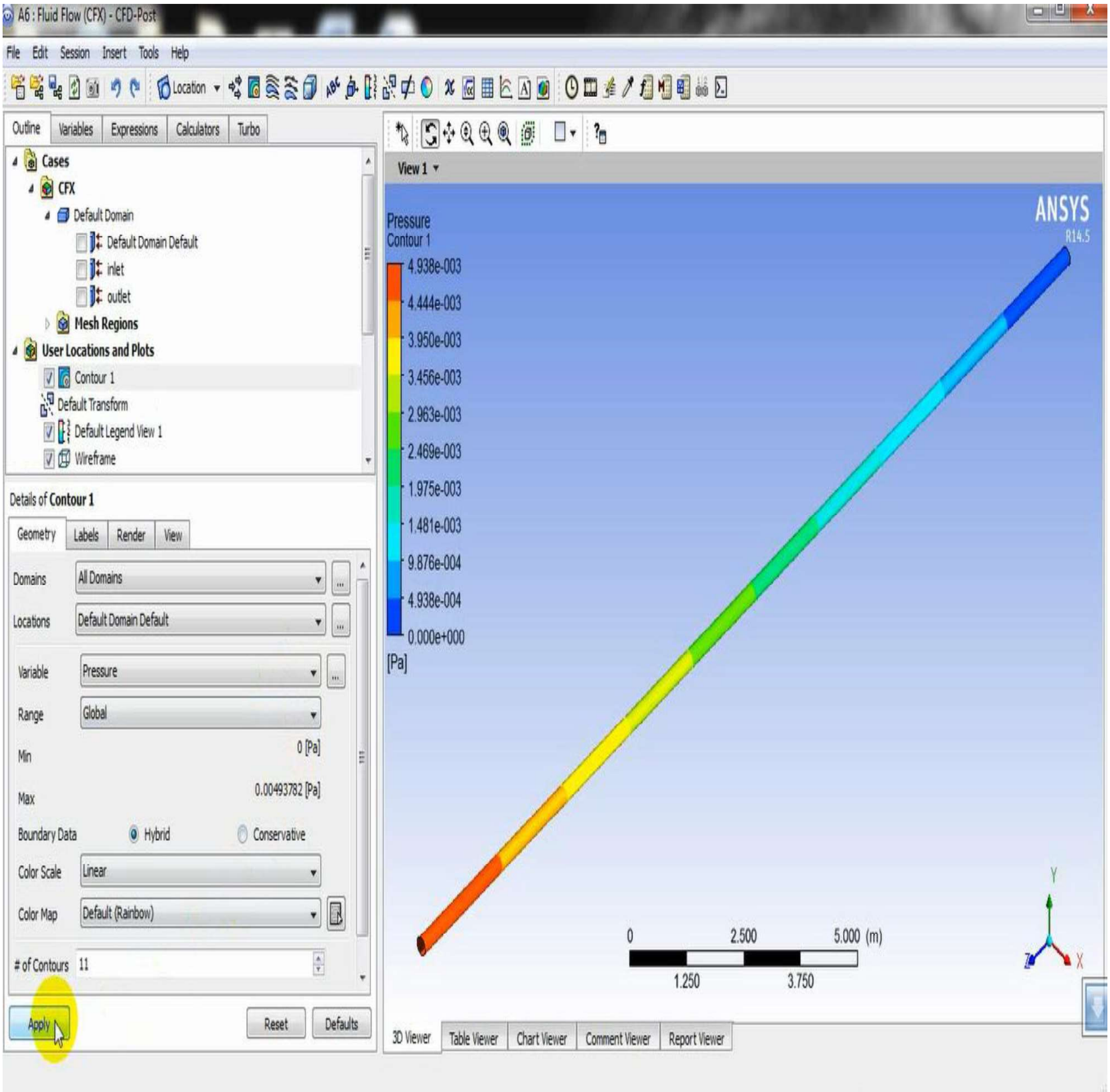
Then you are going to the next step was a solution by giving it the beginning of the solution (*Start Run*) as shown in the following:



After complete solution that gives the program a message on it as shown in the following:



After that we can review the results for the current example the following figure represents the pressure distribution:

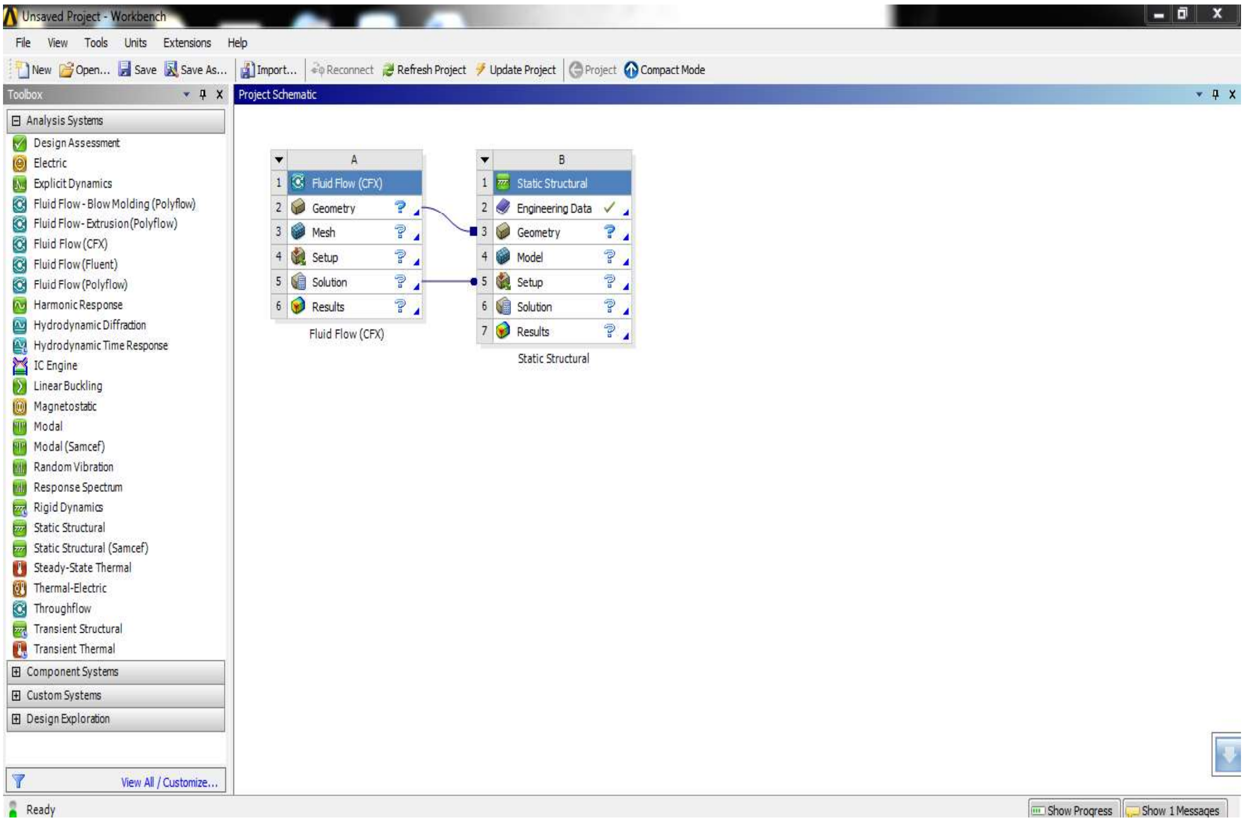


As shown in the video and our book.

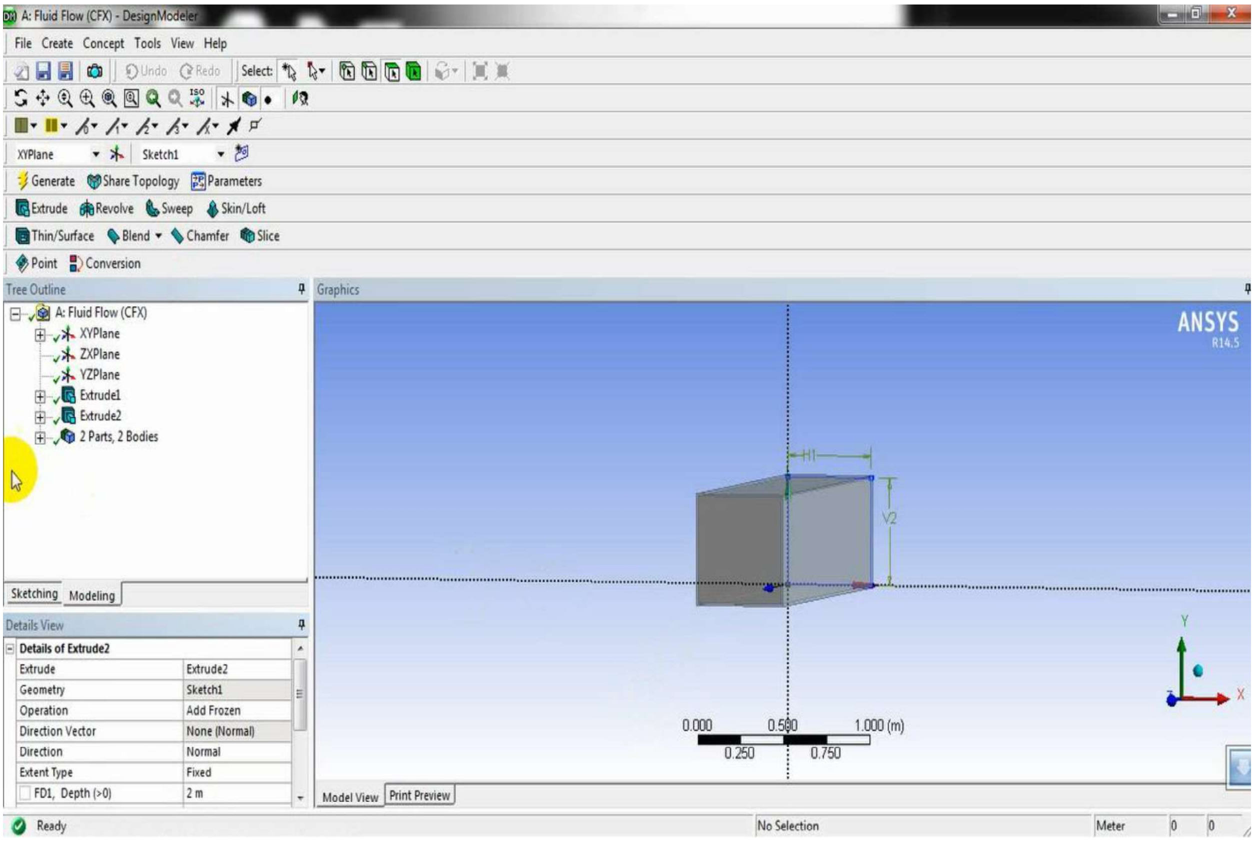
Tutorial Fifteen

Fluid Flow (CFX) + Static Structure Interaction

Select Analysis System *(Fluid Flow Fluent)* from the main menu of the *(Analysis System)* by double clicking on the system or by dragging and dropping on the workplace, and then select (*Static Structure*) from the analysis system and by dragging and dropping on the solution of the first analysis system (*Fluid Flow Fluent*), it is linking the (*Geometry + Solution with Setup*), as is shown in the following figure:



Then design model in the form of square duct with dimensions (*0.5×0.5 m*) and extrude (*2 m*) which as flow geometry and then create cover in thickness (*0.01 m*) as structure geometry by using (*Design Modular*) and then subtract the two geometry by using Boolean feature as shown in the following figure:

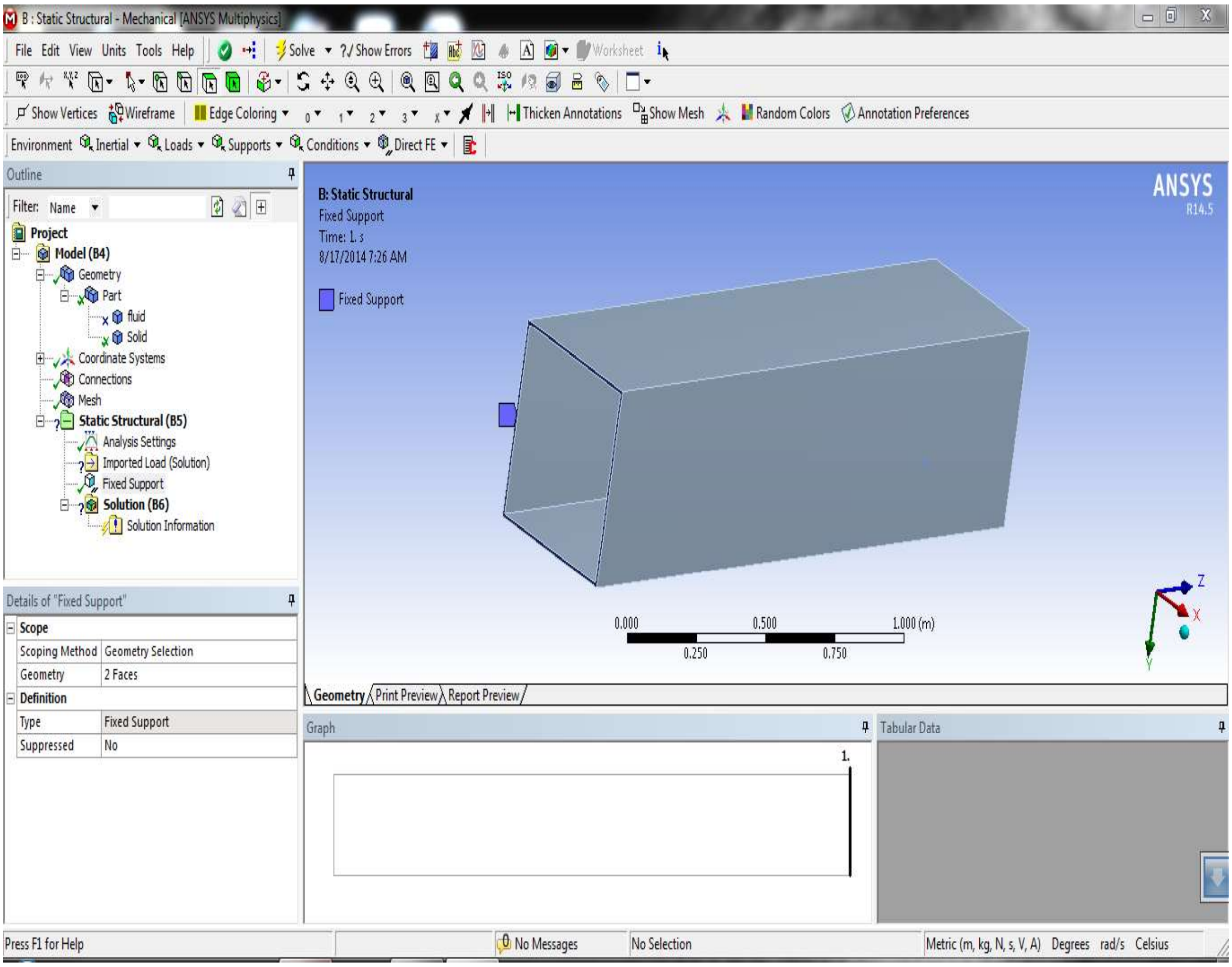


And then suppress the geometry of structure and mesh the fluid geometry and named the (*Inlet*) and (*Outlet*) surfaces and selected the all the other surfaces and named (*Wall*) to allowed in import the (*CFD*) load in set the structure system.

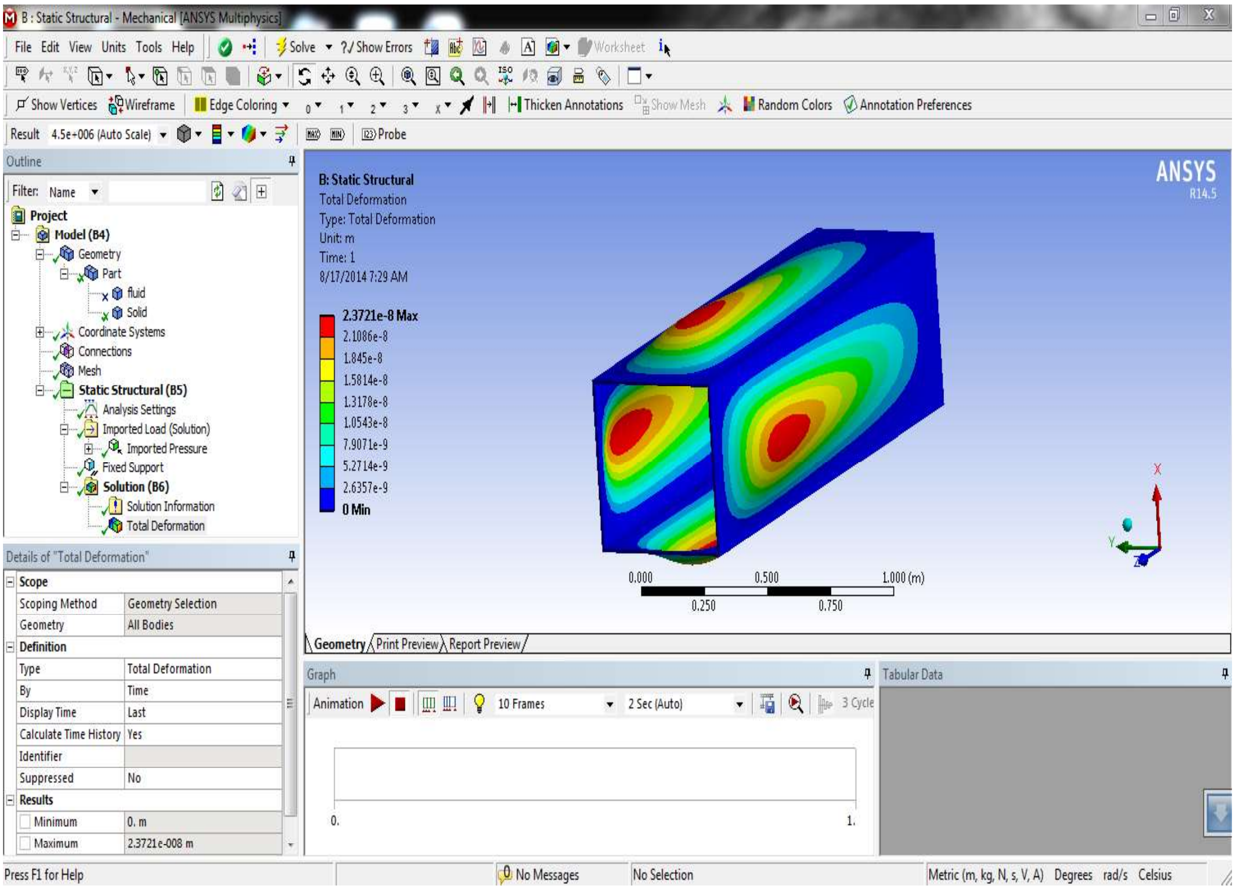
After that open the program window and adjusted the current case where the flow type is (*Turbulent K–e*) and the work material is (*Air*) and setting the boundary condition for the geometry where set the inlet surface as

(*Velocity Inlet*) with value (*10 m/s*) and the outer surface as (*Static Pressure*) with value (*0 Pas*) and the other surfaces as (*Wall*), and then (*Initialization*) the solution and make (*Run*) after that the solution was completed and can review any results required.

And then go to the next step which is set the static structure system where begin with geometry where suppress the flow geometry and mesh the structure geometry and then in load condition is support the geometry from the two ends and by selected it and make (*Fixed Support*):



And then import load where this imported load in this case is pressure load which is the flow applied on the structure and when imported this load are applied on the interior surfaces of structure geometry and import from the (*wall CFD surface*), and then (*Solve*) the case and review the required results, the following figure show the deformation produced from applied the flow pressure on the duct:



As shown in the video and our book.