Hello everyone, welcome to the 4th video lecture on the course of CFD using open foam. this lecture is dedicated to the essential governing laws and open foam implementation. this is Part 2 which is mainly based on computational heat convection. The course is brought to you through FOSSEE IIT Bombay. Along with me, Professor Janani Murallidharan is the coordinator for this course.

Coming to the outline of today's lecture. First, we will briefly recap the part one which was based on computational heat conduction. Then we will see what is the Convective heat transfer phenomena And then we will look into the governing laws and partial differential foamat for the convective phenomena. Later, we will see how a finite volume Discretization can be perfoamed for the computation for heat convection Which is usually implemented in open foam. Finally, we look at how open foam tries to solve the Governing equations through an illustrative example.

Coming to the recap of computational heat conduction, which was dealt in part one. We saw that, Conductive heat transfer mainly occurs due to Either vibration or collision of molecules in a substance. We also looked at the governing equations for Heat conduction Which was the unsteady part - Row CP into T. Combined with the Laplace foamat, which was K del Square T by Del X Square plus del Square T by Del Y Square. We also looked at how we can discretize this particular equation through finite volume methodology and land up with linear algebraic equation. After discussing the discretization technique, we also saw how the algebraic equations can be solved through an iterative scheme with implicit time stepping. Finally, we had looked into an illustrative problem of unsteady 2D heat conduction in a metallic slab.

Coming to today's discussions, Let us try to first understand what is convective heat transfer? So, if you consider a flow over a bike engine as shown in this figure. We see that there are a number of thin plates, which are Inserted above the engine. It can be seen in this particular figure. Now we see that the heat which is generated from the engine is carried away by the bulk motion of air as shown in this particular figure. Also at the surface, Contact between the Plate and the air, The heat transfer occurs due to Flow of heat due to conduction. That is mainly inside the plate. But whereas in the region adjacent to the plate, the heat transfer mainly occurs due to bulk motion of air molecules since the bike is in motion. Now what if the bike stopped and airspeed became 0? In this situation, the mode of heat transfer would be pure conduction. That is only due to the vibration of molecules inside the plate the heat transfer would occur. Hence, we can define convection as an energy transfer between fluid or solid interfaces in motion due to vibration of molecules which is conduction as well as bulk motion of fluid which is called advection, so convective heat Transfer is a combination of Conductive and advective heat transfer.

Let us look at some practical application where convective heat transfer is involved. If we look at the day and night breeze, We see that the land Has a very low specific heat capacity as compared to the water surface. Due to which the land gets warmer very quickly. And due to the Active heat transfer from the water body, cool sea breeze occurs during the daytime. During the night time The land gets Colder quickly, and the cold breeze moves from land to water. The water body so as to Keep the land At a particular temperature. Due to this The land Doesn't get warmer quickly during the daytime and it doesn't get colder during the night time. That is the main Concept which is involved in day and night breeze wherein heat transfer occurs due to convective mode. Now the second example is the body temperature regulation. Wherein the heat generated by cells and the shoes cells in the tissues are carried by blood, which moves through arteries and veins.

The variables which are involved in Convective heat transfer are Temperature and flow. Mainly the velocities involved with the flow. So, in pure conduction we had seen that only the temperature Was involved in the governing equation, whereas now we have flow which kind of affects the convective heat transfer. So, the velocity of the fluid also needs to be accounted for. Now let us try to look at the energy conservation involved in convection, as we saw in part one. The governing law can be extended as follows, over a time interval Delta T, the net amount of convicted thermal energy Entering the control volume plus the heat generated within the control volume Will be equal to the amount of increase of enthalpy stored within the control volume. Which is as shown in this control volume figure. We can see that along with the conduction heat flux we also have additional heat flux which is involved due to advection along the surfaces. This Arm is a deep represented by enthalpy Flux which is denoted in this figure. At X and X plus DX represent the heat flux along the X direction whereas Y and Y plus DY represents the heat fluxes along wide direction. Note that we are considering only 2D Convection for simplicity. The amount of increase of enthalpy stood within the control volume can be given by DOE by dot.

No, we can write the Conservation equation as conductive heat transfer which is entering the surface plus the net advective heat transfer which is entering the Control volume along with the heat Generation. Will be equal to the due by Dou T which is the rate of increase of energy. Now, Heat flux which is entering the control volume will be a combination of conduction and advection. And similarly for the Y direction. So, as seen in the last week, part one dealt with. Heat conduction we can see here that along with. 2QX by 2X dot Q by two Y we have additional 2 terms which are 2X by 2X and 2Y2Y. So, these are the two additional terms which are coming due to advective heat transfer. Now let us try to derive a continuous partial differential foamat for the. Convective energy conservation. So as we saw in part one, The conductive heat fluxes can be given in terms of primary temperature variable using the Fourier's law. Now when we go into the enthalpy flux which is given by X&Y, we see that it can be given using the mass flow rate which is happening across the control surface along with the temperature. Given by MCP into T. Long X direction M can be written as row into UX whereas along the Y direction it can be written as row into UI. Substituting these terms, we finally get a continuous partial differential equation. To solve conservation. Of energy. Let us look at. A simple Vandy domain and try to discretize the convective equation which we derived earlier. So, for simplicity it is assumed that there is no heat generation. During the process. We see that along with. The KD Square D by DX square we also have M into CP into DT by DX term which is involved. Since we had two oh by two X in the Energy conservation. Also, since we have. Assumed steady state foamulation. The rate of change of temperature is not present in these equations. If we integrate across the windy control volume and apply the ghost divergence theorem. We then land up with the. Integral equation as shown in the last line. So, once we do the integration, we finally see that. We land up with K into don't by Dou X along the East face, K into dot by Dou X along the West face along with that. Additionally, we get this row CP into UT along the East Face and UT along the West face. U represents the velocity in X direction here. This don't by Dou X along the E face can be anyways directly. Expanded using A central difference scheme as given in this equation. Now, we are left with evaluating you at the East face, The U at the east face, U at the West face, and T on the face. So, one of the preliminary options would be to Take an average of Adjacent control volumes to find the variable values at the face center. That is at East Face and West face.

These averaged variables do not work when the gradient of flow is very large. For example, assume that the flow is in this particular positive direction. And the predators are mainly affected by the upstream variables. So, the value of temperature at W&P affects more the temperature at E as compared to the temperature at East Control volume. Hence, we come up with advection schemes wherein. The temperature, or any variable at the face, is calculated as a weighted average of temperature along the downstream, upstream, and upstream of upstream direction. So, in the first case, wherein flow is in the positive direction, which is from left to right, we see that the downstream region corresponds to the ETH control volume, upstream corresponds to the beat control volume, and upstream of upstream corresponds to the. W control volume, so based on the temperature at these control volumes, we can get the temperature at the East face. A similar approximation can be done if the flow is in the. negative direction that is from right to left W. Now depending on the various advection schemes like second order upwind, first order upwind or Central difference or quadratic interpolation for kinematic convection which is quick scheme we can calculate this weight functions.

Now, let us try to look at the open foam format of equations and how open foam implements the convective heat transfer. For this we have to check scalar transport model which is available as a solver in open foam. For this you can go into the corresponding directory which is open foam your version, number applications, Solvers, BASIC and scalar transport foam. So, the scalar transport foam is a basic solver which is present in open foam. To know the implementation of the solver, you can go into scalar transport foam dot C file. Once this file is opened, you can see the following equations Which are embedded in the solver. Mainly you see fvm et of T Which represents the rate of change of temperature. Plus, you see Divergence of phi, T. Now the variable Phi here mainly represents our rho into U into cp. So that is mainly the phi term which is involved in this particular equation. Also you come up with Laplacian of K into T which is due to conductive heat transfer. And then you can give a source term, the heat generation term in the Rohs. So last Part that was the computational heat conduction we saw What is the relevance of FVM it tries to Discretize this equation using finite volume method and finally land up with the matrix of coefficients which is a PW, etcetera, so. The discretization and conversion into coefficient matrix is particularly done by this finite volume matrix. So this is the way in which Open foam tries to solve the Computational heat convection.We can try to non dimensionalise the equation and say that DT will be equal to K by row CP.

Now to check the solution methodology implemented in open foam, we have to go to a particular example. So we can see that using the tutorials basic scalar transport foam in that we will get one particular Example which is Pittz daily example. Now if we go on to the FV solution and FV schemes file, Present in the System folder, We then see the divergence schemes which corresponds to the Various advective schemes. Being implemented here, it is just a linear upwind scheme Which corresponds to a second order, a point which we had seen earlier. Also, in the Laplacian scheme, You can see that, A Gauss linear scheme is implemented where second order central difference will be considered. These were the discretization schemes For resolving the activation and the conduction. Now if we go on to see the solution schemes, we see that the. Linear algebraic equation here is solved using the by conjugate gradient method Which is an extension of conjugate method which we had seen in computational heat conduction.

If you look at an illustrative example of 2D unsteady state convection in a backward facing step, which is mainly called as the pitz daily example, We see that this particular example consists of a 2D channel with a a step which is present near the inlet. So the main intention is to study the variation of temperature inside the channel due to the variation of velocity. No, as a boundary condition we provide. Uniform inlet temperature of 1 units whereas the diffusivity which is defined as alpha which is equal to K by rho CP is defined as .01. And along with that we have zero gradient of temperature at the wall and outlet. For the velocities it is assumed that the steady state. Velocity values are already known, so the velocity values are already feeded into the Example. In general case we have to also solve the Navier Stokes equation to obtain velocity, but here for simplicity it is assumed that the velocities are already known. So to test this example, you can download the zip file given with the video and extract the files in the terminal. You can type the block mesh to generate the mesh file. Once the mesh file is generated, you can type the scalar transport foam command. Obtain the results and then the results can be finally checked using parafoil. So it was assumed that the velocities were already known. So this is the initial velocity contour initial as well as final since the steady state is assume. No using this particular velocity. The temperatures are calculated by solving the convective Equation. So it can be clearly seen that. That had the higher velocities have corresponded to the higher temperatures, as can be seen in this figure. So through this we can get an appreciation for convective phenomena.

The references used in today's presentation are as follows. Mainly the contents are obtained from introduction to computational fluid dynamics development application and analysis by Professor Atul Sharma and the Openfoam implementations are obtained from the One Foam website. Thank you for listening.