1: Good afternoon. I am Laura Pla Olea and I am going to present part of the work done in my study on the computational resolution of conservation equations.

2: The presentation is divided in different sections. First of all I am going to explain the objectives of the study and the methodology followed, and then I am going to do a brief introduction on what are the conservation equations and numerical methods. Next, I am going to show you the results obtained in the resolution on some basic cases and finally, the results of an application case studied.

3: The main objective of the study was to learn more about computational methods, because I think it is an important subject and I thought I didn’t know a lot about it. A second objective was to apply these methods to a specific case in order to study it. To do so, the methodology was quite simple. First I started with the resolution of some basic cases in order to learn the basis on numerical methods, and then these were applied to the application chosen. In both cases the process followed was the same. First it was necessary to identify the equations needed in the resolution of the problem – what needs to be solved -, and then the numerical method that should be applied in each case. Finally, the simulation code had to be developed. This part may look like the simplest one but it actually took a lot of time.

4: OK. So, the conservation equations are the set of equations that we see here. The conservation of mass, momentum and energy equations are the mathematical expressions of fundamental physical principles: mass is conserved, Newton’s second law and that energy is also conserved. They are the basis of fluid dynamics and can be seen as particular cases of the generic convection-diffusion equation that we see here. The only problem of these equations is that they don’t have an analytical solution except for some specific simple cases. So, in order to solve them it is necessary to use numerical methods.

5: The basis of numerical methods is to divide the domain that has to be studied in different pieces. Instead of calculating the unknowns with a continuous function in the whole domain, they are obtained in a finite number of points named the grid points. The first step to use these methods is to divide the domain with a process named discretization. There are several ways to do it, but the one used in this study is the finite volume method, in which the domain is divided in different control volumes, as we see in the picture. These nodes are the points in which the variables are going to be calculated. This is the spatial discretization, but it is also necessary to discretize over time. This is done dividing the domain in time steps and calculating the values in each of them. When the variables of one time step are obtained we move on to calculate the next one.

6: Starting with the basic cases, one of the first problems I solved was the Smith-Hutton problem. In this case, we have a rod with a given velocity field and we have to obtain the distribution of the variable phi in it. These three lines that we see here are the boundary conditions of this variable, the values that it takes in the boundary.

7: In this case, the equation that has to be solved is only the convection-diffusion equation, and the only unknown is the variable phi. The mesh used in this problem is the one we see here, it is very similar to the one I described previously.

8: Looking at the results obtained, when the ratio between the density and the diffusion coefficient is low, the main transport method is diffusion, which tends to uniformly distribute the properties of the inlet to the whole domain, but as this ratio increases the velocity field is what determines the distribution of phi.

9: However, the only problem of the Smith-Hutton problem is that it is not a realistic case, because usually the velocity field is the unknown that has to be calculated. This is the reason why the driven cavity problem was proposed. In this case we have a flow inside a cavity and suddenly the top wall starts to move, originating a motion inside of the cavity.

10: In this case the equations to be solved are the mass and momentum equations. However, if we look at the momentum equation, in order to obtain the velocity field it is also necessary to calculate the distribution of pressure inside of the cavity. Even more, these two variables are linked in the same equation. Fortunately, there is an algorithm that allows the resolution of this equation step by step: the fractional step method.

11: According to this method, it is possible to obtain the pressure just with the velocity field in the previous instant of time and then calculate the velocity in the next time step. However, if we look at the expression of the velocity, it is necessary to calculate the pressure gradient. So, in the mesh that we see here, in order to obtain this velocity, we have to calculate this pressure minus this pressure, meaning that there is a pressure node in the middle that we are not taking into account, and this node can lead to a lot of error. In order to solve it, staggered meshes are used.

12: These meshes are just a complicated way of saying that we have one mesh for each unknown. In this case, in black we have the mesh of the pressure, the points in which the pressure is calculated. The green arrows show the points in which the horizontal velocity is obtained and finally the purple arrows indicate the nodes in which the vertical velocity is computed. If we look at the previous expression and we calculate it in this mesh we see that, for example, to obtain this velocity we have to compute this pressure minus this pressure, so there is no extra node in the middle that has to be taken into account.

13: And these are the results of the driven cavity problem. In the slide we see the streamlines for different Reynolds numbers. As we see, for low Reynolds numbers, when the viscosity is high, only the fluid close to the top wall tends to rotate, but as the Reynolds number increases the rotation extends to the whole cavity because the centre of rotation is displace to the middle and also some other vortices appear in the corners of the cavity.

14: Finally, the only equation that is left is the energy equation, and it is introduced in the differentially heated problem. In this case the movement in the cavity is originated by the difference of temperatures between the left and right walls of the cavity. The fluid near the hot wall has low density and moves upwards and the flow near the cold cavity with higher density moves downwards, originating the rotation that we see in the picture.

15: Again, it is necessary to solve the mass and momentum equations, but we also have to add the energy equation in order to obtain the temperature field. This distribution of temperature will generate the difference in densities that will cause the motion. This is the reason why in this case the gravitational term of the conservation equation cannot have a constant density, has to take into account some compressibility, and this is done with the Boussinesq approximation.

16: Solving the equations the results obtained are the ones we see here. In the left we see the temperature field and in the right the streamlines. As we expected, in the distribution of temperature the hot fluid is in the top and the cold fluid is in the bottom. As for the streamlines, we see that the fluid rotates around the centre of the cavity, but as the Rayleigh number increases this centre of rotation stretches and is divided in different vortices.

17: Finally, the application case: the square cylinder problem. As we can see, it consists in a channel that can be a wind tunnel and a cylinder in the middle of it. The main difference between this case and the previous ones is that in the other cases we had the flow inside a cavity and now we have an external flow that moves around something. This means that we have to tell the simulation that there is something in the middle, and it is done treating the cylinder as a wall. The velocity in the cylinder is equal to zero and the flow cannot pass through it.

18: As for the resolution it is exactly the same that in the case of the driven cavity problem.

19: The results obtained in this application depend on the Reynolds number. As we can see, for low Reynolds numbers, the fluid simply avoids the cylinder but flows around it, with no separation. However, for Reynolds numbers higher than 5 it is detached and forms a recirculation bubble in the rear side of the cylinder. This is the two vortices that we see here. The length of this bubble increases with the Reynolds number.

20: If we calculate this recirculation length, the results of the simulation are pictured here as purple points. In green we see the plot of this expression, which is an empirical expression that describes the dependence of the length on the Reynolds number. As we can see, the results are very similar except for the cases of Reynolds equals 1 and 3 but these are out of the range of application of the equation and are not taken into account.

21: As a conclusion of the work done, I would say that I have learned a lot on numerical methods, not only with the things that are in this presentation, but also in a lot of other aspects. I still have a lot of room to improve, though. And regarding the simulations, I have to say that the results are realistic and pretty accurate, which I am satisfied with. However, the one complaint I have is that the application case is only valid for steady solutions that is Reynolds numbers up to 60 more or less, which means that there is also some future work that can be done in order to increase the range of solutions.

22: And that is all that I have to say. Thank you for your attention.