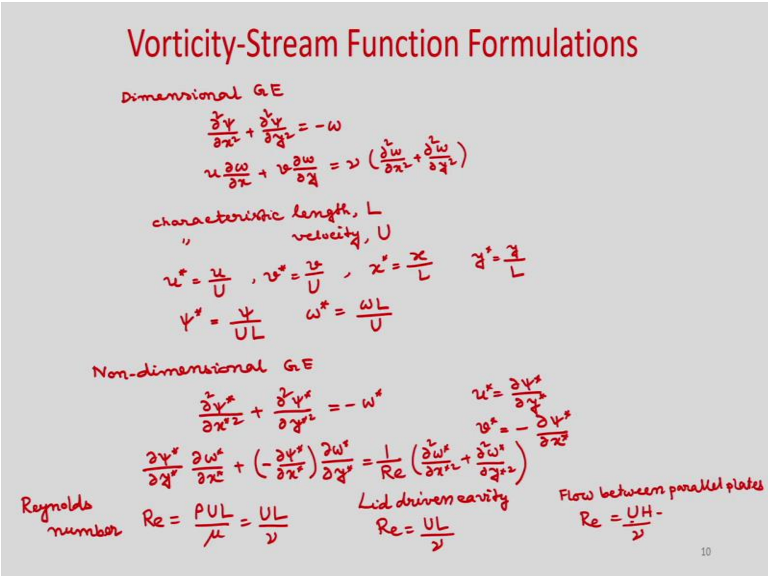
Computational Fluid Dynamics for Incompressible Flows
Professor. Amaresh Dalal
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati
Lecture 3
Solutions of vorticity-stream function equations

We have written the governing equation in dimensional form. So, now let us write the Stream Function and Vorticity Equation in non-dimensional form.

(Refer Slide Time: 00:43)



So, our governing equations in dimensional form, dimensional governing equation we have written, del 2 psi by del x square plus del to psi by del y square is equal to minus omega and u del omega by del x plus v del omega by del y is equal to nu del 2 omega by del x square plus del 2 omega by del y square.

So, we will use this non-dimensional parameters as say characteristic length, we will take as a L and characteristic velocity will take as U capital U. So, you can write the non-dimensional form in star quantities. So, u star is the non-dimensional velocity. You can write u by capital U, v star is non-dimensional v velocity. So, v by characteristic velocity U. Similarly, x star is the length, so x by L. y star is equal to y by L and your psi star you can write as, psi by U into L and omega star you can write omega L by U.

So, these are the non-dimensional parameters. So, these now you substitute it in the dimensional equation and rearrange it, you will get the non-dimensional form of the governing equation. So, in non-dimensional form of governing equation. So, I am not going to derive just I am writing del 2 psi star by del x star square, plus del 2 psi star by del y star square is equal to minus omega star. So, these are non-dimensional psi and non-dimensional omega and similarly u star and u star is nothing but your, u star is nothing but del psi star by del y and v star is equal to minus del psi star by del x star. So, this is also star.

So, you can write u as del psi star by del y star, del omega by del x. So, this is also star. Because non-dimensional equation and v star is minus del psi star by del x star and del omega star by del y star and you will get the non-dimensional number as Reynolds number. 1 by Re del 2 omega star by del x star square plus del 2 omega star by del y star square. So, this is the vorticity equation in non-dimensional form and in the right hand side the diffusion coefficient you are getting as 1 by Re.

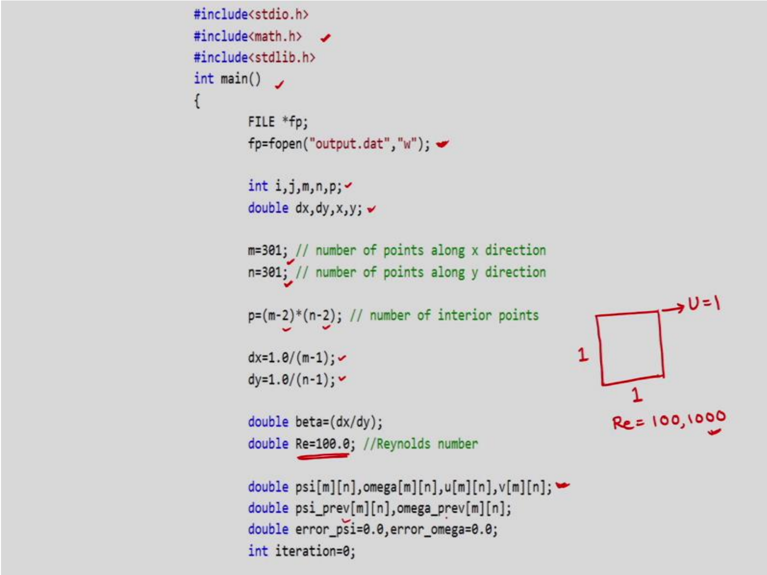
So, earlier it was nu which was kinematic viscosity. Now, using non-dimensional parameters now you are getting 1 by Re. Re is equal to Reynolds number is equal to here you are getting rho UL by mu or UL by nu. U is the characteristic velocity, L is the characteristic length and nu is the kinematic viscosity and you know what is Reynolds number, Reynolds number is the ratio of inertia force to viscous force. So, this is your Reynolds number.

So, now for this lid driven cavity problem will take the characteristic length as L side of the cavity and characteristic velocity will take as the lid velocity U. So, with that if you non-dimensionalise it. So, you will get this non-dimensional equation and if you discretize it, you will get the discretize form of the governing equation. Only difference will be that wherever nu was there, you just put 1 by Re. Then you will get in non-dimensional form these discretized equation and when we will consider the flow between parallel plates.

So, that time I will take each height is the non-dimensional, characteristic length and inlet velocity U as characteristic velocity. So, accordingly we will get the Reynolds number. So, in the Reynolds number for lid driven cavity problem. Lid driven cavity, Reynolds number is your UL by nu, where U is your lid velocity and L is the side of the cavity and for flow between parallel plates, flow between parallel plates. So, you will get Reynolds number as UH by nu, where H is the height of the, or the distance between two plates and U is the inlet velocity.

So, now we will show the code of this lid driven cavity problem. So, whatever we discretized the equation as well as the boundary condition. So, that how you can write the code that I will show, then I will show some results.

(Refer Slide Time: 07:11)

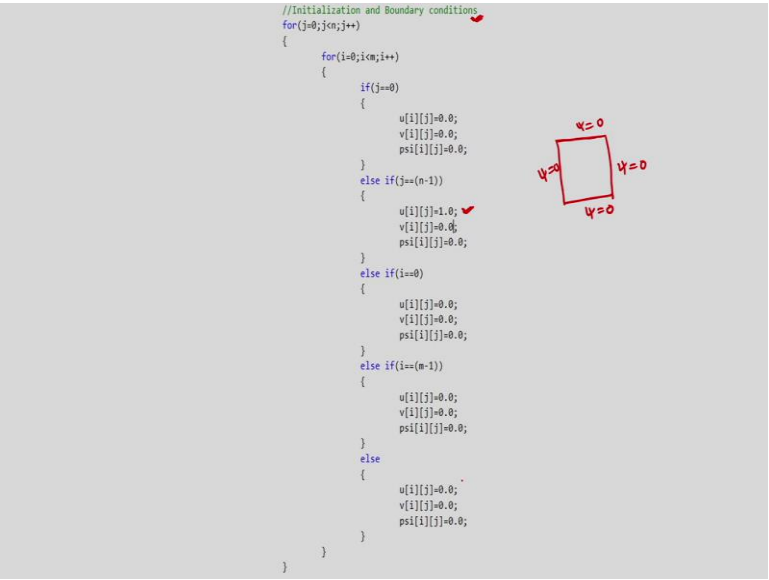


So, this is the code you can see. So, this is general header file you just include it. So, this is the main program for writing the output we are opening one output file. So, some integer i, j, if, m, n, p we are just defining double dx, dy, x, y. We are taking number of points along x direction, 301 and number of points along y direction 301.

So, total number of interior points will be m minus 2 and n minus 2 and delta x, delta y will be 1 by m minus 1 and 1 minus n minus 1. So, we are taking lid driven cavity and this is your velocity U and we are taking in non-dimensional form. So, we are taking 1 as side and U is equal to 1. So, your U is equal to 1. So, accordingly we have solved the problem for Reynolds number. 100, 1000, 400. So, this we have taken.

So, now we can see beta just delta x by delta y you have defined and Reynolds number 100 we are giving the input. So, psi omega u, v this just added we have defined here. We are using previous value of psi and omega just to calculate the error.

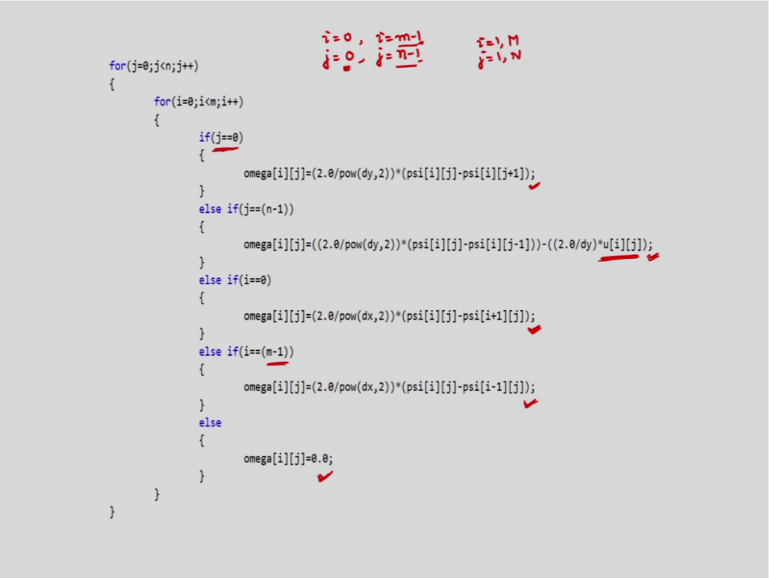
(Refer Slide Time: 08:42)



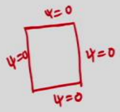
And now we are initialising at the interior point and that the bounded we are writing the boundary conditions for velocity and the stream function. So, now we are giving the boundary conditions for U and psi. So, this U and psi we are giving after that from psi we will calculate the boundary conditions for the vorticity.

So, this you can see, so all over we are giving 0. Because all the, in all four walls we have put psi equal to 0, if you remember. So, psi is equal to 0, so psi is equal to 0. So, that we are putting and on the top wall, you have velocity 1. U i, j is equal to 1 that we are defining and accordingly all everywhere it is 0.

(Refer Slide Time: 09:30)



```
//Initialization and Boundary conditions
for(j=0;j<n;j++)
{
        for(i=0;i<m;i++)
        {
                if(j==0)
                {
                        u[i][j]=0.0;
                        v[i][j]=0.0;
                        psi[i][j]=0.0;
                }
                else if(j==(n-1))
                {
                        u[i][j]=1.0;
                        v[i][j]=0.0;
                        psi[i][j]=0.0;
                }
                else if(i==0)
                {
                        u[i][j]=0.0;
                        v[i][j]=0.0;
                        psi[i][j]=0.0;
                }
                else if(i==(m-1))
                {
                        u[i][j]=0.0;
                        v[i][j]=0.0;
                        psi[i][j]=0.0;
                }
                else
                {
                        u[i][j]=0.0;
                        v[i][j]=0.0;
                        psi[i][j]=0.0;
                }
        }
}
```

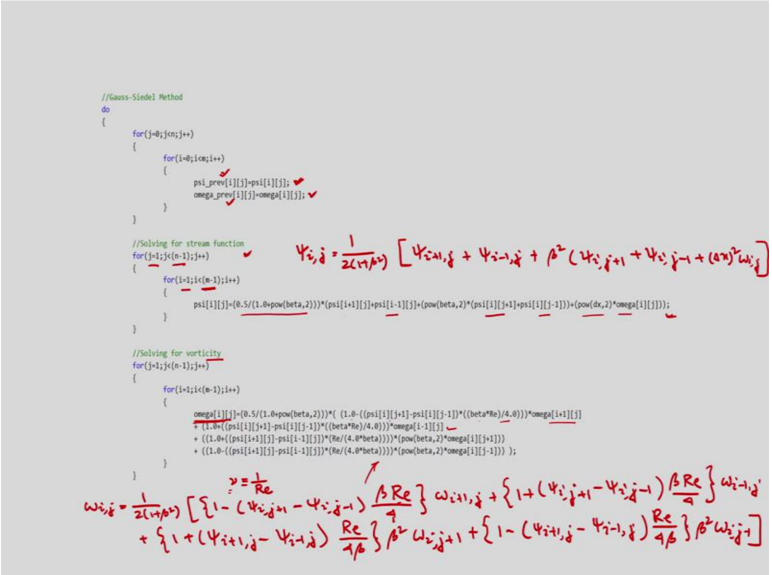


And all interior points also we are putting 0. So, you can see so all interior points psi also we are putting 0 and for omega also whatever boundary condition today we have derived. So, those boundary conditions just we are writing here and before starting the solution; we are applying the boundary condition from the u and psi boundary conditions. So, you can see that from psi we are just writing j is equal to 0. So, you remember here we are starting from j is equal to 0 to, j is equal to n minus 1, n minus 1 and i is equal to 0 to i is equal to m minus 1.

Because during derivation we have written i is equal to 1 to j is equal to m, i is equal to 1 to m and j is equal to 1 to n, as you are starting from i and j 0. So, just minus 1 we have done from the i max and j max. So, it is m minus 1 and n minus 1. So, you are see that j is equal to 0, j is equal to 0 is the bottom wall.

So, you are putting it and this is your j is equal to n minus 1, j is equal to n minus 1. So, you can see n minus 1 means it is on the top wall and top wall you have velocity u. So, this is the u i, j. So, u i, j is 1 in this case, so this we have written and for i is equal to 0, i is equal to 0 means you have left wall. So, this is the left wall omega boundary condition and i is equal to m minus 1 it is right wall boundary condition and all interior point omega is equal to 0.So, these are the boundary condition and initialisation, we have done before going to the solution.

(Refer Slide Time: 11:14)



So, now we have used Gauss-Seidel method. So, just only one unknown omega i, j and psi i, j and we have used the best available value of psi and omega while calculating this omega i, j and psi i, j. So, for calculating the error you can see that we are storing the psi i, j and omega i, j in the previous value and then we are solving for stream function. So, for all interior points. So, j is equal to 1 to n minus 1, i is equal to m minus 1.

So, this interior points, so psi i, j. So, this is the, if you remember we have discretized equation psi i, j is equal to 1 by 2 into 1 plus beta square, psi i plus 1 j plus psi i minus 1 j plus beta square, psi i, j plus 1 plus psi i, j minus 1 plus delta x square omega i, j. So, this is the equation discretize equation. So, that we have written. You see this is your 1 by 2 1 plus beta square psi i plus 1 j plus psi i minus 1 j plus beta square psi i j plus 1 psi i j minus 1 and plus delta x square omega i j.

So, this we are calculating the psi value at all interior points and you can see that these, these are coupled equation, the vorticity equation as well as your stream function equations. So, after converging it, after solving for all interior points. Then we are solving immediately this vorticity at all interior points. Before going to the next iteration, so you can see this also we are solving for vorticity. So, in the vorticity you see omega, so this is discretized equation if you remember.

So, whatever we have derived, but here nu just you put 1 by Re, whatever equation we have written. So, nu is equal to 1 by Re if you put, then you will get this equation as omega i j is equal to 1 by 2 into 1 plus beta square and 1 minus psi i j plus 1 minus psi i j minus 1 beta by 4 nu. So, nu 1 by Re we are putting. So, this is your equation omega I plus 1, j plus 1 plus psi i j plus 1 minus psi i j minus 1 beta into Re by 4 omega i minus 1 j plus 1 plus psi i plus 1 j minus psi i minus 1 j.

Then Reynolds number by 4 beta beta square omega i j plus 1 plus 1 minus psi i plus 1 j minus psi i minus 1 j Reynolds number by 4 beta beta square omega i j minus 1. So, this is the discretized equation in non-dimensional form. So, Reynolds number just wherever nu was there we have put 1 by Re. So, you can see that we have written. So, you see 1 minus psi i j plus 1 minus psi i j minus 1 beta into Re by 4 into omega i plus 1 j. Similarly, this is for omega I minus 1 j.

So, all these we have just written the, for all interior points and we are calculating omega i j. So, in single loop, now we have calculated the psi i j and omega i j. Because both are coupled.

(Refer Slide Time: 15:27)

```
//Update vorticity at boundaries
for(j=0;j<n;j++)
{
	for(i=0;i<m;i++)
	{
		if(j==0)
		{
			omega[i][j]=(2.0/pow(dy,2))*(psi[i][j]-psi[i][j+1]);
		}
		else if(j==(n-1))
		{
			omega[i][j]=((2.0/pow(dy,2))*(psi[i][j]-psi[i][j-1]))-((2.0/dy)*u[i][j]);
		}
		else if(i==0)
		{
			omega[i][j]=(2.0/pow(dx,2))*(psi[i][j]-psi[i+1][j]);
		}
		else if(i==(m-1))
		{
			omega[i][j]=(2.0/pow(dx,2))*(psi[i][j]-psi[i-1][j]);
		}
	}
}
```

So, next we are actually applying the vorticity boundary condition again. Because psi is applied. So, omega boundary condition will be updated. So, we have updated the boundary conditions.

(Refer Slide Time: 15:38)

```
        //Error calculation
        error_psi=0.0;
        error_omega=0.0;

        for(j=1;j<(n-1);j++)
        {
                for(i=1;i<(m-1);i++)
                {
                        error_psi=error_psi+pow((psi[i][j]-psi_prev[i][j]),2.0);
                        error_omega=error_omega+pow((omega[i][j]-omega_prev[i][j]),2.0);
                }
        }

        error_psi=sqrt(error_psi/p);
        error_omega=sqrt(error_omega/p);

        printf("iteration=%d\t",iteration);
        printf("error_psi=%.10lf\terror_omega=%.10lf\n",error_psi,error_omega);
        iteration++;
}
while(error_psi>1.0e-8 || error_omega>1.0e-8);

//Update velocities u and v
for(j=1;j<(n-1);j++)
{
        for(i=1;i<(m-1);i++)
        {
                u[i][j]=(0.5/dy)*(psi[i][j+1]-psi[i][j-1]);
                v[i][j]=(-0.5/dx)*(psi[i+1][j]-psi[i-1][j]);
        }
}
```

$\varepsilon_\psi = 10^{-8}$

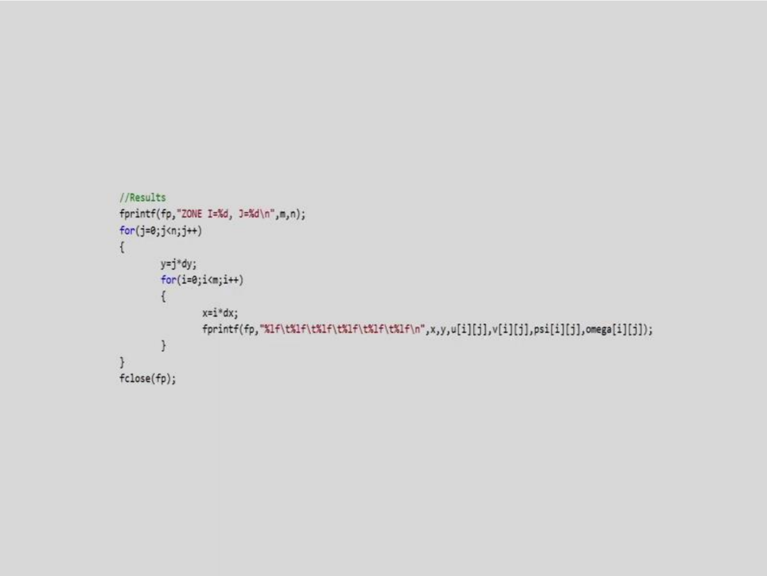
$\varepsilon_\omega = 10^{-8}$

Then we have calculated the error. So, error psi is equal to error psi plus psi i j minus psi previous i j whole square. So, this is the summation over all the grid points we are doing. So, then after coming out of this loop just we are dividing by total number of grid points and we are making the square root and error for psi and error for omega we are calculating, also we are calculating the iteration, we are printing error psi, error omega with iteration and going to the next iteration, then we are checking the condition.

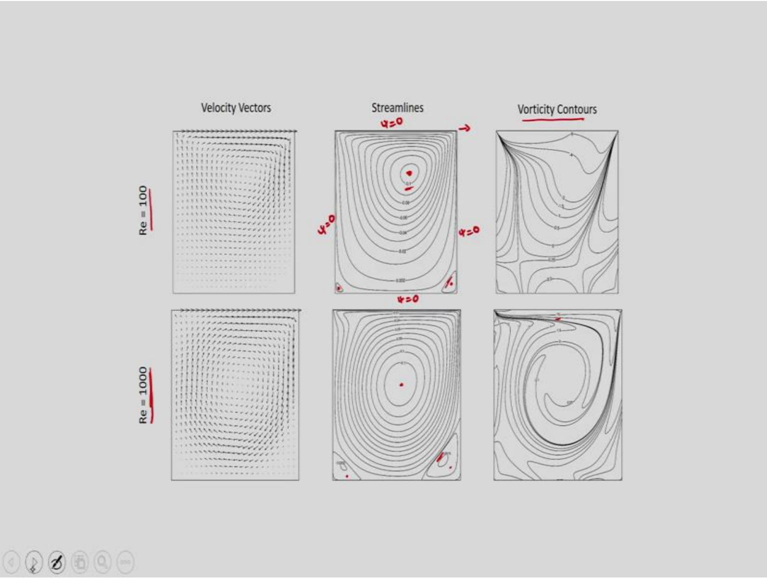
So, this error psi and error omega both if it is satisfied 10 to the power minus 8. Then it will actually it is greater than 10 to the power minus 8. Error for psi and error for omega, so if it is greater than this. Then this iteration will continue if it goes below 10 to the power minus 8, then it will come out of the loop.

That means, your solution has been converged. So, once the solution is converged, then you can calculate the u, v at all points including boundary points you know. So, all interior points you calculate.

(Refer Slide Time: 16:59)



```
//Results
fprintf(fp,"ZONE I=%d, J=%d\n",m,n);
for(j=0;j<n;j++)
{
        y=j*dy;
        for(i=0;i<m;i++)
        {
                x=i*dx;
                fprintf(fp,"%lf\t%lf\t%lf\t%lf\t%lf\t%lf\n",x,y,u[i][j],v[i][j],psi[i][j],omega[i][j]);
        }
}
fclose(fp);
```



And you print this x, y, u, v, psi, omega to just plot it. So, now you can see the velocity vectors, streamlines and vorticity contours for different Reynolds number. So, Reynolds number 100 and Reynolds number 1000. So, these are the velocity vector u, v we have calculated.

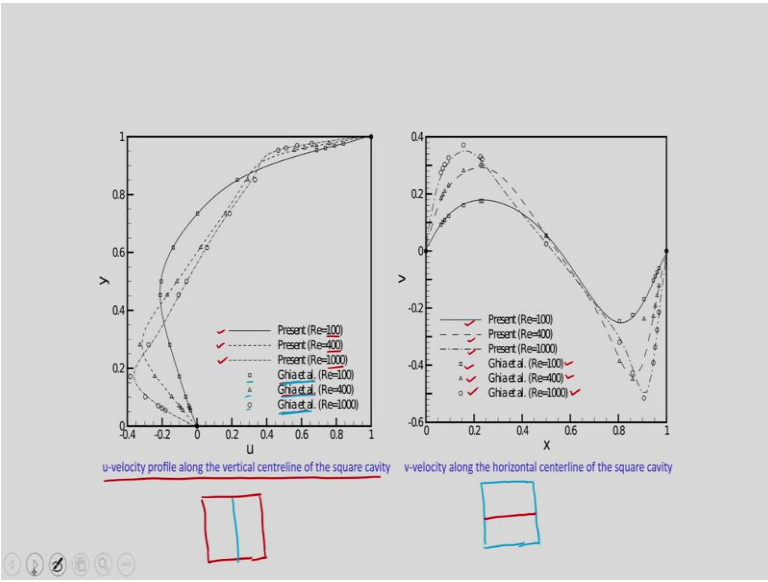
So, this u, v from u, v we have shown the velocity vector. These are the stream function contour actually. So, these are stream function contours. So, you can see, so we have given 0 at the wall. So, 0 at the wall we have given, so accordingly you can see that this value is increasing 0.002, 0.02, 0.2, 0.4, 0.6, 0.8, it is actually, so you can see psi is equal to 0 we have put in all walls.

So, this psi is decreasing minus 0.002, minus 0.02, minus 0.04 and minus 0.1 and now you can see that once the lid is moving this side. So, this is your velocity vector, so 1 primary vertex is generated here. This is your primary vertex, so it is primary vertex is generated and some secondary vertexes are also generated at the corners. If you go Reynolds number 1000, so the primary vertex centre has been shifted downward direction and there you can see this secondary vertex size has been increased.

So, here it was like this you can see, but here the size is increased and this is the vorticity contour. So, obviously you can see near to the wall you have maximum vorticity. The value you can see, you have maximum vorticity near to the wall. Because velocity gradients are more near to the wall.

So, you will get more vorticity near to the wall. So, this way you can calculate the stream function, contour as well as the vorticity contour and velocity vector and now you got the result. So, to know whether the results are correct or not. We have compared these results with some benchmark solutions.

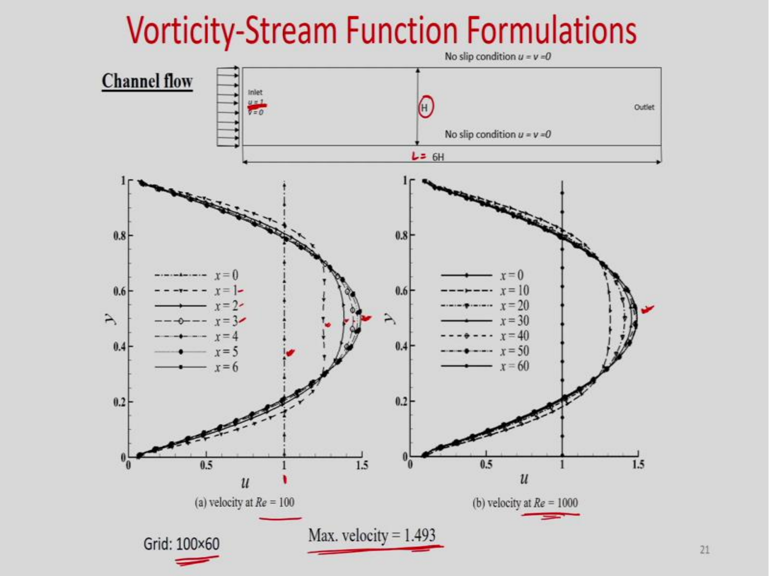
(Refer Slide Time: 19:18)



So, there is a paper Ghia et al, so with that we have compared our results. So, you can see that present case, we have taken 100, 400 and 1000 and these are the lines in our simulations. So, these are the u-velocity profile along the vertical centreline of the square cavity. So, if you have square cavity here, so you are calculating the velocity along the vertical centreline. So, along these you are actually plotting the U-velocity. So, that you can see that our results are these lines and Ghia et al which actually is also numerical solutions.

So, that we have put with the symbols. So, you can see that these our results matches well with the literature. Here, also you have compared v-velocity, v-velocity we have compared along horizontal centreline. If it is the cavity and along horizontal centreline, we have compared the v-velocity. So, our results with lines and Ghia et al with symbols for different Reynolds numbers and you can see how the velocity profile looks like. So, these are some comparison with the literature.

(Refer Slide Time: 20:39)



Next, we have solved the channel flow. So, can see, so here L is equal to 6 H we have considered. H is the distance between two plates and u is equal to 1 we have given non-dimensional velocity and H we have taken in the Reynolds number. So, we have used grid 100 by 60 and now once you solve and with the, if you solve the vorticity discretize equation and the stream function discretize equation. Along with the boundary condition what we discussed for this channel flow.

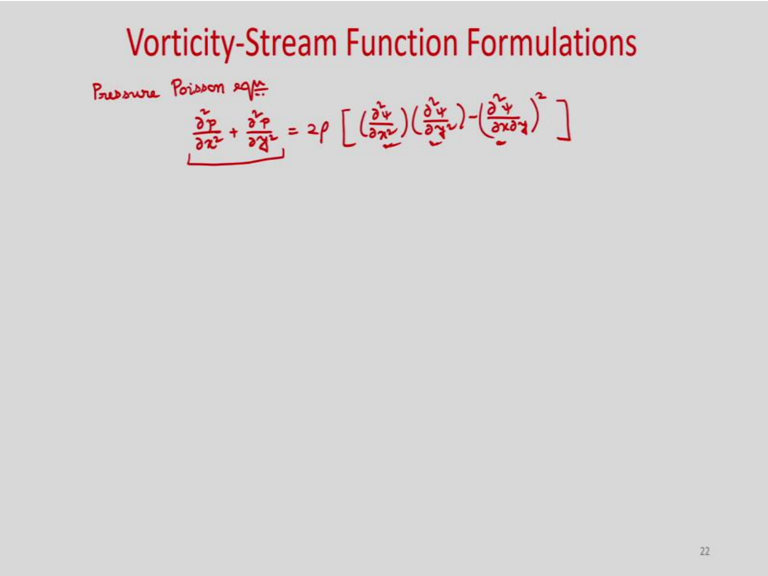
Then if you solve this vorticity and stream function, then you will get the velocity profile inside the channel. So, these are the velocity profiles I have shown. So, obviously at the you can see that these are x equal to 0. So, that means it is inlet velocity condition and 1 we have given and as your x axial direction if you proceed then obviously it is developing. You can see that these are developing; developing velocity profile. Then once it becomes it comes to the fully developed regions.

So, it becomes fully developed and in fully developed regions you know maximum velocity will be the 1.5 times the average velocity. So, that we are getting, so your inlet, average velocity is 1, average velocity is 1, so obviously the maximum velocity will be 1.5 times. So, in our numerical simulations, we are getting 1.493. So, you can see that here is the maximum velocity and it is parabolic in nature and for different Reynolds number we have simulated and the velocity is becoming almost 1.5 times the average velocity.

So, also you can plot the stream function for this. So, all will be almost parallel except the near to the inlet. So, this we have solved using stream function vorticity equation. So, you can see that it is this vorticity-stream function formulation is very simple to write the code and solve the problem and but it is limited to two dimensional problems. But if you want to calculate the shear stress, if you want to calculate the pressure difference or the drag coefficient or lift coefficients. Then you cannot do just after solving this vorticity-stream function.

You need also the pressure distribution, but as we derive, while deriving the vorticity transport equation. We have actually, the pressure term is avoided, so we cannot directly calculate the pressure from these equations. So, but some if you really want to calculate the drag coefficient and lift coefficient, then you need to solve by Pressure Poisson equation.

(Refer Slide Time: 24:00)



So, this Pressure Poisson equation will look like this but we are not deriving it here, but if you want to calculate, then for this two dimensional situation your pressure person equation will be, del 2 p by del x square plus del 2 p by del y square. So, from the momentum equation you can actually calculate it.

So, it will be del 2 psi by del x square and del to psi by del y square minus del 2 psi by del x del y square. So, this is the Pressure Poisson equation, so this equation is once the size then need to calculate these gradients and you can solve for this pressure equation. So, this is the limitation of using vorticity-stream function formulations.

Today we have derived the boundary conditions for stream function as well as the vorticity from the known velocity boundary conditions. Then we have shown the code for lid driven cavity and the boundary conditions, we have derived for both lid driven cavity problem and the channel flow and then we have shown a code for solving the lid driven cavity problem and we have shown the results for different Reynolds numbers, for this lid driven cavity problem. We solved the governing equation in non-dimensional form and the Reynolds number comes into picture.

So, here for different Reynolds number, we have solved a lid driven cavity problem and the results we have compared with the literature and the results matches well with the literature. Then we solved the channel flow. So, in channel flow also we have shown that the fully developed condition, the velocity profile becomes parabolic and maximum velocity at the centreline, you will get 1.5 times the average velocity, when it becomes fully developed. Thank you.