Task 2.

Sub-tasks. It is recommended to create different projects for different sub-tasks.

1. **(3 points)** Add lines in the code to also calculate within the code the analytical solution for the laminar channel flow (see Task 1 for the analytical solution formula). Write an additional column with the analytical solution in the file OUTPUT.DAT. Plot both velocity profiles, calculated and analytical, on a single plot and compare the accuracy of simulations.

2. **(3 points)** Change the grid distribution along the y-axis, but keep the number of grid nodes the same. Make a plot where you compare initially calculated velocity, analytical solution, and the solution obtained with a new grid.

3. **(3 points)** Using the grid from sub-task 2, reduce the number of nodes in that grid 2, 5, and 10 times. Then, conduct simulations on these 3 grids and add the results to the previous plot. Now you should have 6 lines on a single plot from several files. It is highly recommended that you change the name of your OUTPUT files as Excel and you can confuse them. Same applies to the Residual files. Also notice how the number of steps required for the solutions to converge changes depending on a grid.

So, the good practice is to reflect your changes in output files and work in different directories/projects

4. **(3 points)** Now, a grid with uniform distribution of the grid nodes on the y-axis. You can start from the initial grid number and again reduce it 2, 5, and 10 time. You can choose your own algorithm for reducing the number of nodes too. For this task, you will get 4 outputs. Compare them with the initial grid solution and the analytical solution on the same plot. This plot will also have 6 lines. Same applies to the Residual files. Also notice how the number of steps required for the solutions to converge changes depending on a grid.

5. **(4 points) Analyze the two plots and write down your conclusions on how the nodes distribution and their number affect the solution. Do the same for the residuals. How do they change with the grid modifications? How the number of steps required for the solutions to converge changes with the grid modification. The report should not exceed 2 pages!**

6. **(3 points)** Return back to the code with the initial grid. The grid node distribution is written in the OUTPUT.DAT file. Add lines in the code that will open your initial OUTPUT.DAT file and read the grid node distribution from there instead of generating a grid within the code. Instead of deleting original lines in the code relevant to the initial grid generation, make them comments.

If you do not change the OUTPUT.DAT file name during this exercise, either you will modify unintentionally the original file from which you read the grid or the code will not run if you have not closed it before writing in this again.

7. **(3 points)** Return back to the code with the initial grid. The current setting is for the laminar flow. Change the flow Reynolds number to 2000 (transitional to turbulent) and then to 20000 (turbulent flow). Compare the results with the initial grid result.

8. **(3 points)** Increase and decrease the flow viscosity. How does it affect the pressure drop in the flow? Conduct simulations with these new flow viscosities and compare the results with those obtained with the initial viscosity value.

Add to the output file two lines above the columns. In the first line, write the values of viscosity, Reynolds number, and the pressure drop. In the second line, the number of grid points.

9. **Additional 5 points:** Modify the code to reverse the y-axis direction. Do not forget that reversing the axis direction requires reversing the boundary conditions. Check the subroutines for the implemented boundary conditions, not only the main code.