Project Report for CH603

Parallel plate flow - A simulation and analytical study

Mayank Chopra

MTech Ist year

`2022chm1007@iitrpr.ac.in`

Department of Chemical engineering

April 25, 2023

Abstract

Flow between parallel plates is simulated on openFOAM (solved using the nonNewtonianIcoFoam solver) for various conditions (as mentioned in project specifications) and thereby checked using analytical results obtained after solving the momentum equations.Analytical solutions of part 1 is split into two as the gradients in the two halves are not same hence the momentum equations change and thereby we see small modification in the final analytical result for upper and lower half.

Contents

1 Part 1

For part 1 , we made the geometry using blockMesh by modifying 'blockMeshDict' file in 'system' folder wherein we specified the inlet , outlet , walls and frontandBack faces for the parallel plate geometry.After that , we modified for n (0.5,1,1.5) for power law fluid in 'transportProperties' file in 'constant' folder and the 'p' and 'U' files in '0' folder appropriately as shown below (for inlet, P value is taken to be uniform and equal to 100 as we are given the value of P as 2 Pa/m and for 50m , P = 2*50 = 100 and for density = 1 , p = 100 Pa/kg-m-3):-

```
boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 100;
    }

    outlet
    {
        type            fixedValue;
        value           uniform 0;
    }

    walls
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

Figure 1:'p' file for part 1.

```
boundaryField
{
    inlet
    {
        type            zeroGradient;
    }

    outlet
    {
        type            zeroGradient;
    }

    walls
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    frontAndBack
    {
        type            empty;
    }
}
```

Figure 2:'U' file for part 1.

The final result obtained after comparing and plotting the values of velocity in the x-direction for n = 0.5 is:-

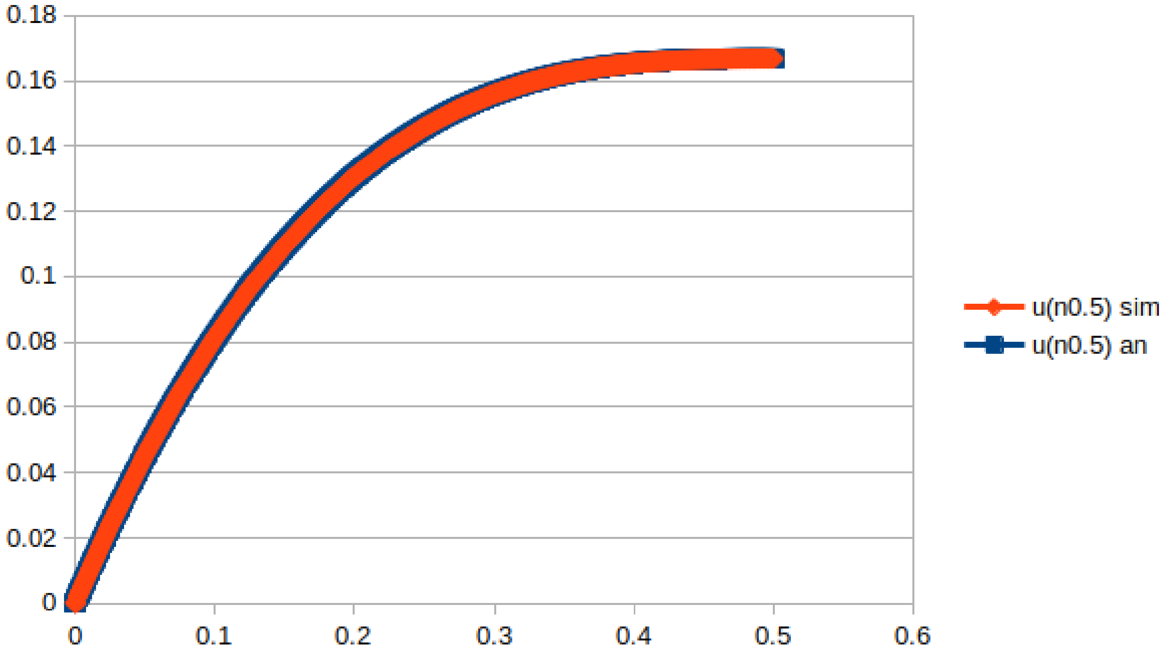

Figure 3:Velocity profile from $0 \leq y < 0.5$ for $n = 0.5$.

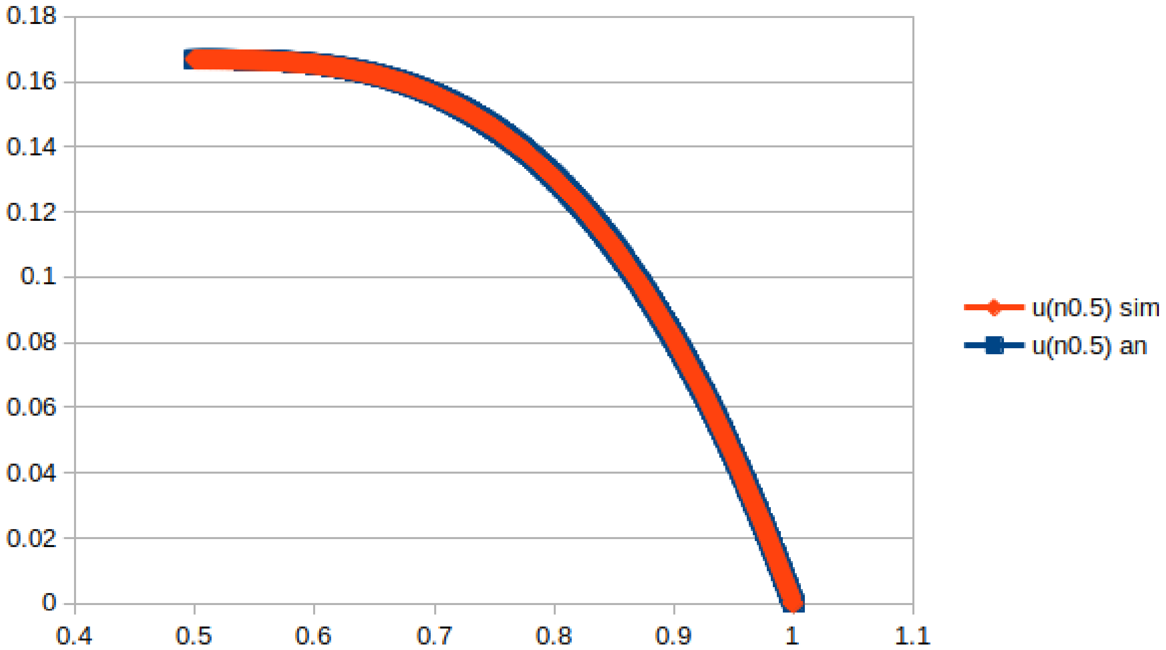

Figure 4:Velocity profile from $0.5 \leq y < 1$ for $n = 0.5$.

The final result obtained after comparing and plotting the values of velocity in the x-direction for n = 1 is:-

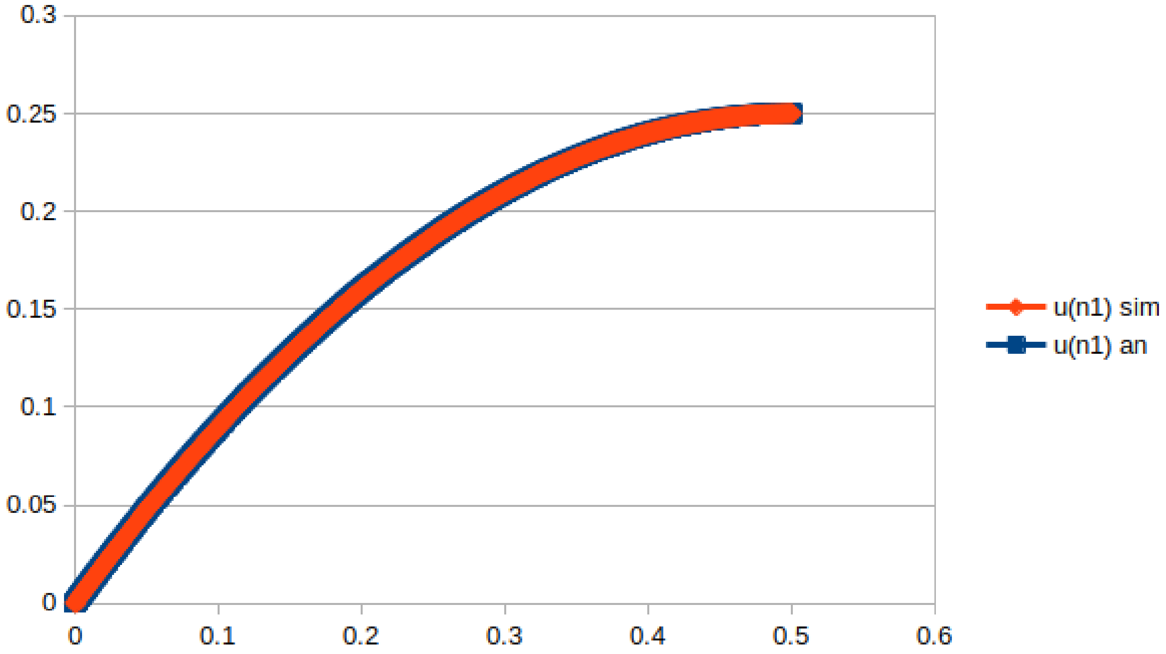

Figure 5:Velocity profile from $0 \leq y < 0.5$ for $n = 1$.

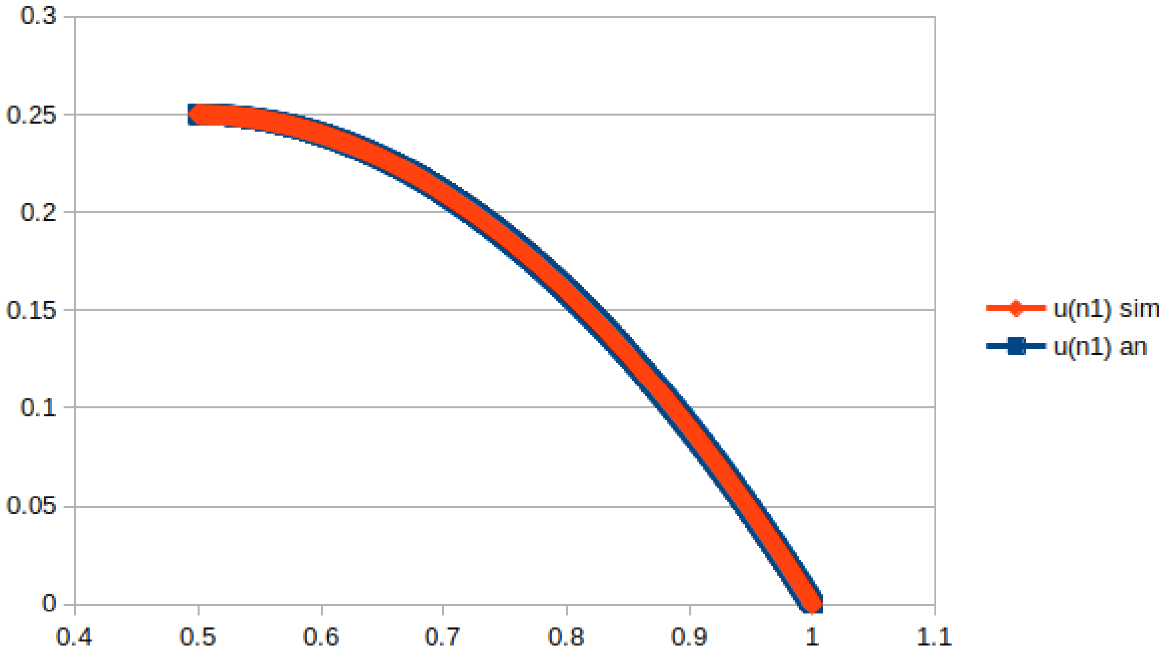

Figure 6:Velocity profile from $0.5 \leq y < 1$ for $n = 1$.

The final result obtained after comparing and plotting the values of velocity in the x-direction for n = 1.5 is:-

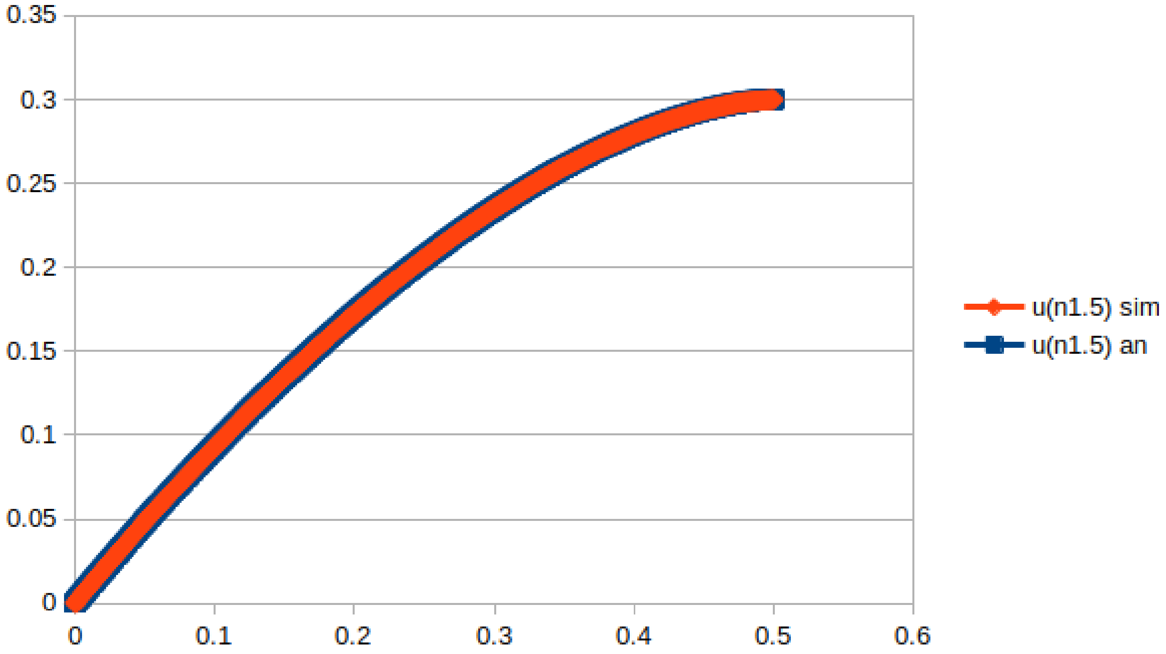

Figure 7:Velocity profile from $0 \leq y < 0.5$ for $n = 1.5$.

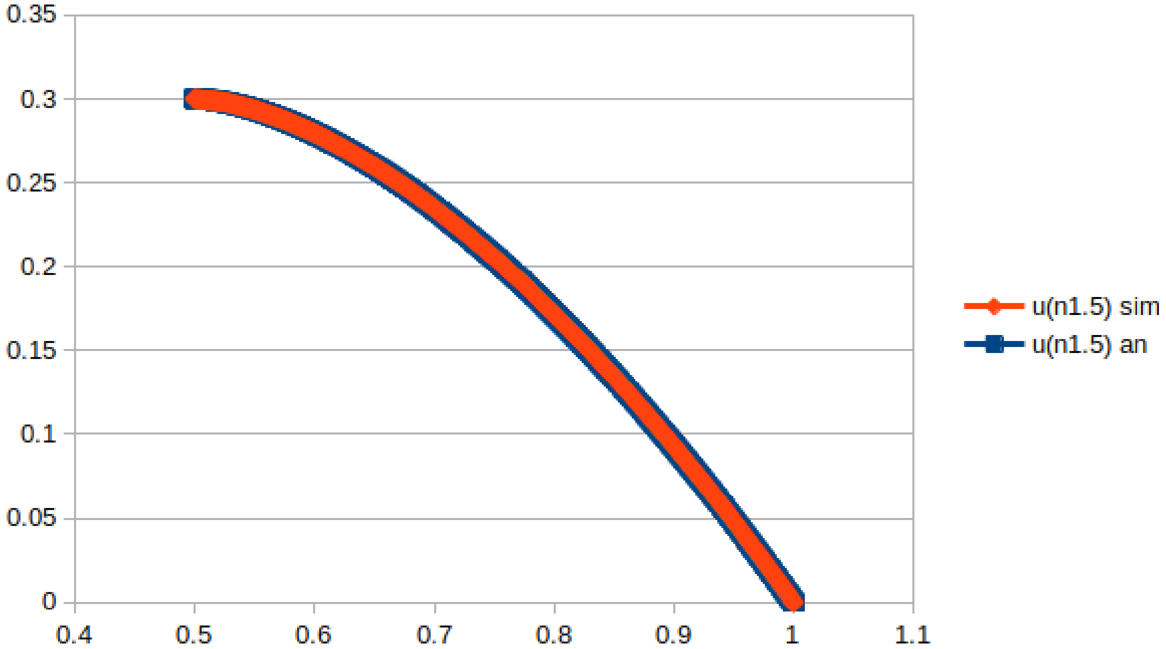

Figure 8:Velocity profile from $0.5 \leq y < 1$ for $n = 1.5$.

2 Part 2

For part 2 , we again made the geometry using blockMesh by modifying 'blockMeshDict' file in 'system' folder wherein we specified the inlet , outlet , upperPlate , lowerPlate and frontandBack faces for the parallel plate geometry(We specify upperPlate and lowerPlate specifically as both have different boundary conditions).After that , we modified for n (0.5,1,1.5) for power law fluid in 'transportProperties' file in 'constant' folder and the 'p' and 'U' files in '0' folder appropriately as shown below:-

```
boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 0;
    }

    outlet
    {
        type            fixedValue;
        value           uniform 0;
    }

    upperPlate
    {
        type            zeroGradient;
    }
    lowerPlate
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

Figure 9:'p' file for part 2.

```
boundaryField
{
    inlet
    {
        type            zeroGradient;
    }

    outlet
    {
        type            zeroGradient;
    }

    upperPlate
    {
        type            fixedValue;
        value           uniform (1 0 0);
    }
    lowerPlate
    {
        type            fixedValue;
        value           uniform (-1 0 0);
    }
    frontAndBack
    {
        type            empty;
    }
}
```

Figure 10:'U' file for part 2.

The final result obtained after comparing and plotting the values of velocity in the x-direction for n = 0.5 is:-

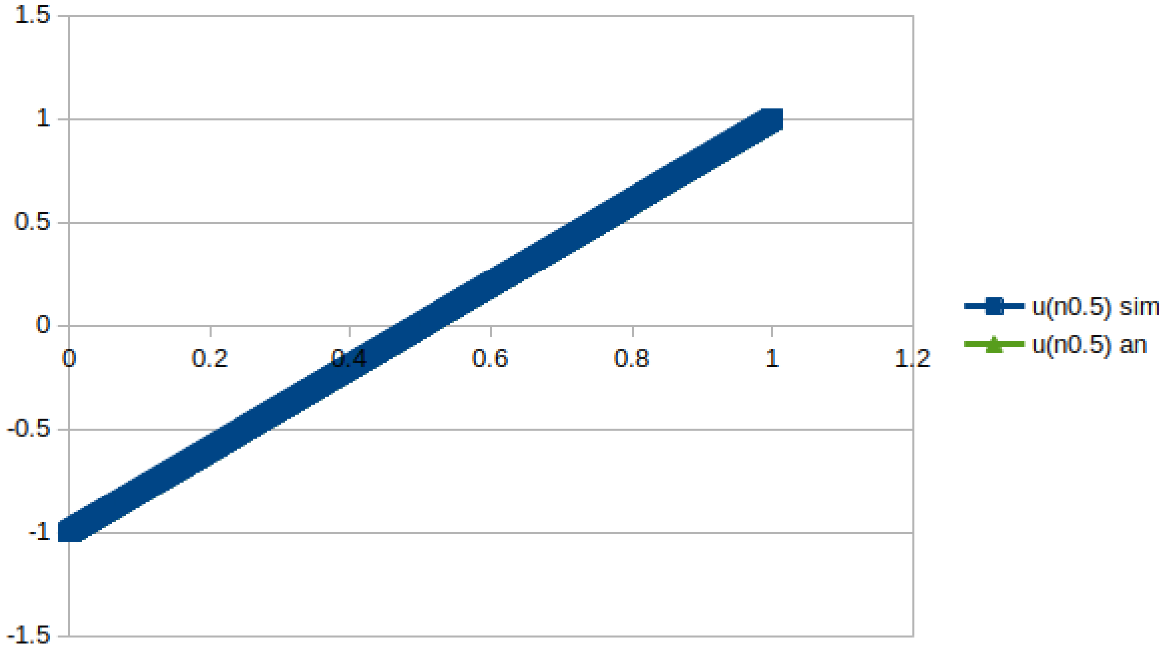

Figure 11:Velocity profile from $0 \leq y \leq 1$ for $n = 0.5$.

The final result obtained after comparing and plotting the values of velocity in the x-direction for n = 1 is:-

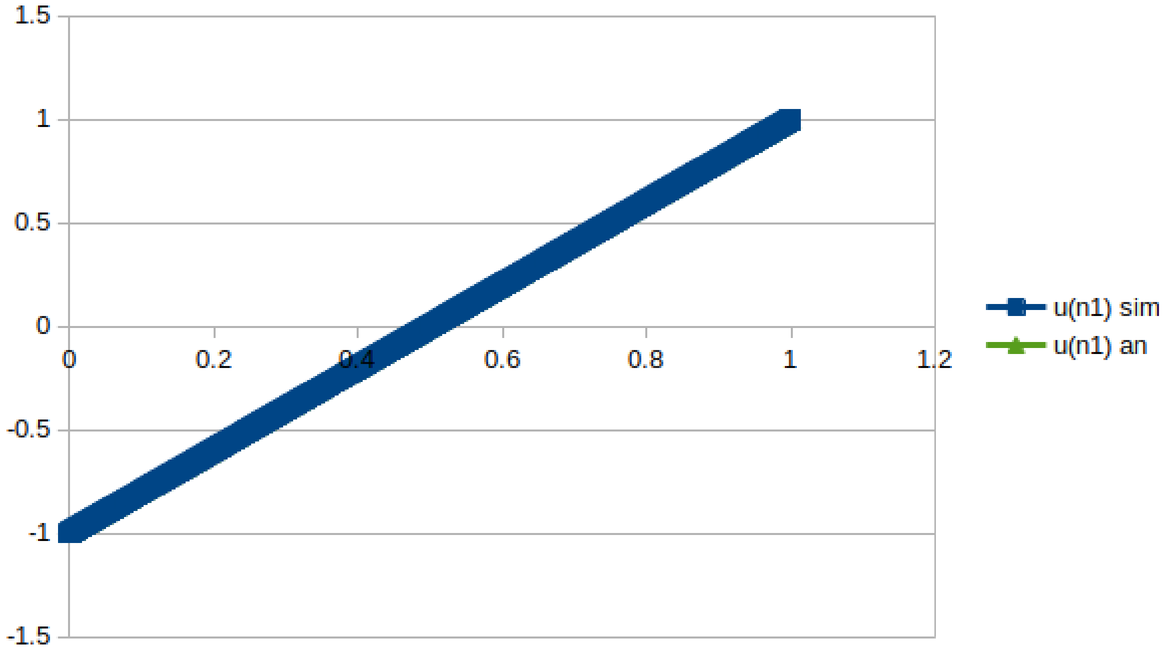

Figure 12:Velocity profile from $0 \leq y \leq 1$ for $n = 1$.

The final result obtained after comparing and plotting the values of velocity in the x-direction for n = 1.5 is:-

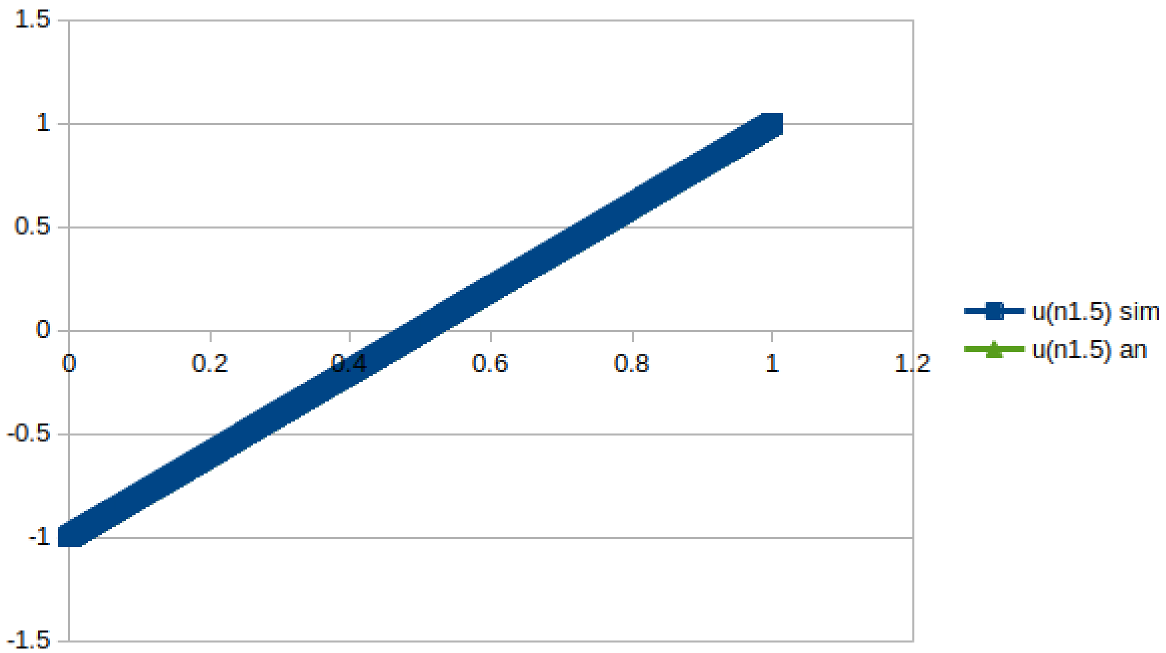

Figure 13:Velocity profile from $0 \leq y \leq 1$ for $n = 1, 5$.

The data used for plotting the above curves is attatched in the deadline as a .CSV file.

3 Part 3 (Misc.)

I have tried to simulate and analytically derive the expression for flow of a casson fluid (in my case , i have chosen blood and appropriately put in the parameters for it) for a pressure driven flow (similar to Part 1) much like how blood flows in arteries but for parallel plate geometry. For this study , i have also modified the pressure gradient to be that of the mean arterial pressure value [1] (MAP) for systolic and diastolic pressure of 120/80 (considered to be normal [2]) to find that the pressure = 93.33 mmHg and after converting into openFOAM units , it comes out to be 93.33 * 13.6 * 9.81 / 1060 (density of blood) = 11.746901207 Pa/kg-m-3.

For part 3 , we made the geometry using blockMesh by modifying 'blockMeshDict' file in 'system' folder wherein we specified the inlet , outlet , walls and frontandBack faces for the parallel plate geometry.After that , we modified model for casson fluid in 'transportProperties' file in 'constant' folder and the 'p' and 'U' files in '0' folder appropriately as shown below(The 'U' file remains the same as before for part 1):-

```
boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 11.746901207;
    }

    outlet
    {
        type            fixedValue;
        value           uniform 0;
    }

    walls
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

Figure 14:'p' file for part 3.

```
transportModel Casson;

CassonCoeffs
{
    m         [ 0 2 -1 0 0 0 0 ] 3.934986e-6;
    tau0      [ 0 2 -2 0 0 0 0 ] 2.9032e-6;
    nuMax     [ 0 2 -1 0 0 0 0 ] 13.3333e-6;
    nuMin     [ 0 2 -1 0 0 0 0 ] 3.9047e-6;
}
```

Figure 15:'transportProperties' file (containing values pertaining to blood) for part 3.

The final result obtained after comparing and plotting the values of velocity in the x-direction for blood is:-

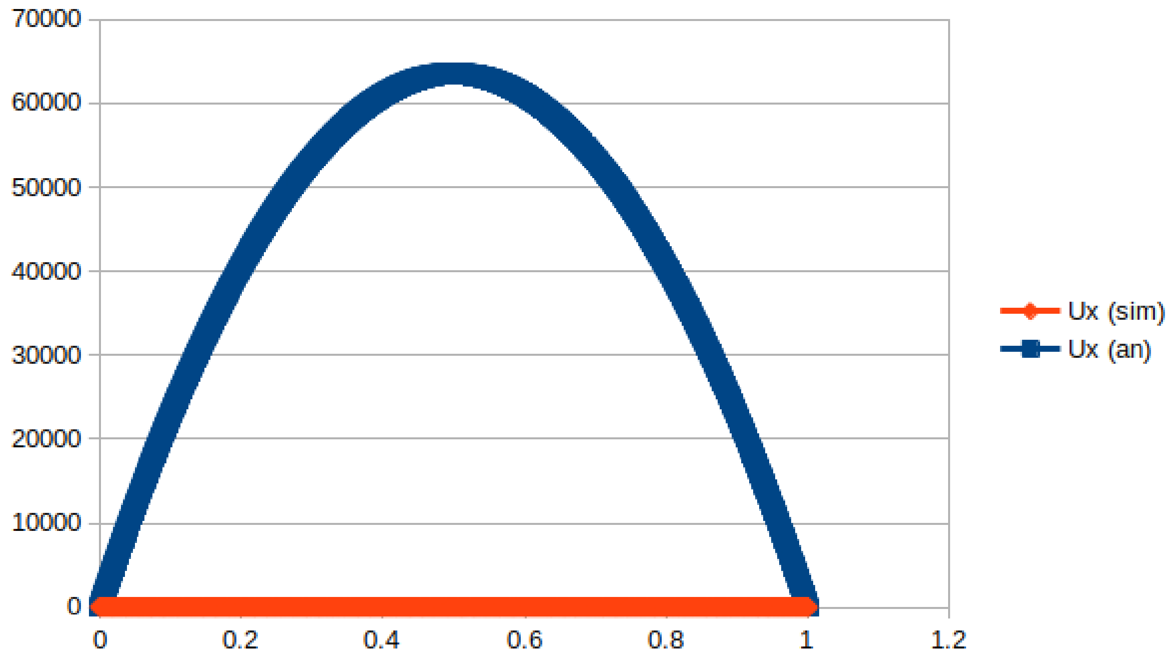

Figure 16:Velocity profile from $0 \leq y \leq 1$ for blood (analytical and simulated).

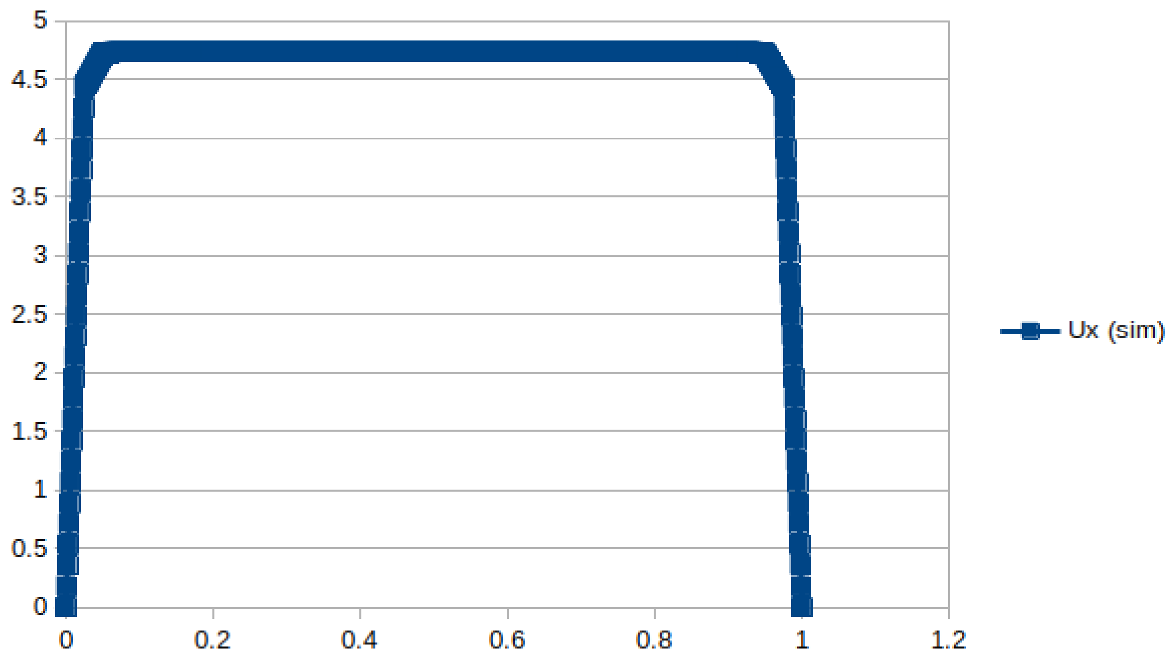

Figure 17:Velocity profile from $0 \leq y \leq 1$ for blood (simulated).

4 Results and discussion

Analytical and simulation results agree with each other for part 1 and part 2 while the analytical and simulation results for part 3 don't agree and the reason might be the assumptions that are taken while deriving the analytical solution to get to the final expression wherein certain terms are neglected by virtue of the order of magnitude difference but clearly , that is a wrong assumption as can be seen by the magnitude of the x-velocity for the analytical solution being very unreasonable .

5 Conclusion

We were able to analytically get an expression for the x-velocity in between parallel plates and compare those values with the simulated results (simulated using openFOAM) for different conditions as specified in the problem statement. The analytical solutions are provided as hardcopy and the files containing data are provided with the report.

References

[1] D. DeMers and D. Wachs, "Physiology, mean arterial pressure," in *StatPearls [Internet]*, StatPearls Publishing, 2021.

[2] T. F. Lüscher, "What is a normal blood pressure?," *European heart journal*, vol. 39, no. 24, pp. 2233–2240, 2018.