Rankine Semi-infinite Body computation

Ramkumar

August 11, 2022

1 Problem Definition

In this work, Rankine semi-infinite body is computed using potential flow theory. The code was made as a custom application in OpenFOAM v2206 and the postprocessing is done using ParaView. The main aim of the work is to introduce the OpenFOAM programming powers to the users.

2 Governing equations

In this work, the potential flow equation containing a Uniform flow and a source positioned appropriately to form the semi-infinite body. The main equation that is used to compute streamfunction ψ is given below, Equation 1.

$$\psi = V_\infty \cdot r \cdot sin(\theta) + \frac{\Lambda}{2\pi}\theta \tag{1}$$

The corresponding velocities in polar coordinates is given at Equations 2 and 3 below.

$$V_r = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = V_\infty \cdot cos(\theta) + \frac{\Lambda}{2\pi r} \tag{2}$$

$$V_\theta = -\frac{\partial \phi}{\partial r} = -V_\infty \cdot sin(\theta) \tag{3}$$

A rectangular mesh was made using *blockMesh* utility in OpenFOAM and the cell centers in the mesh were used as the computation points for streamfunction ϕ and other variables. The radial distance r and azimuthal distance θ are calculated using the Equations 4 and 5, respectively.

$$r = \sqrt{\Delta x^2 + \Delta y^2} \tag{4}$$

$$\theta = arctan2(\Delta y, \Delta x) \tag{5}$$

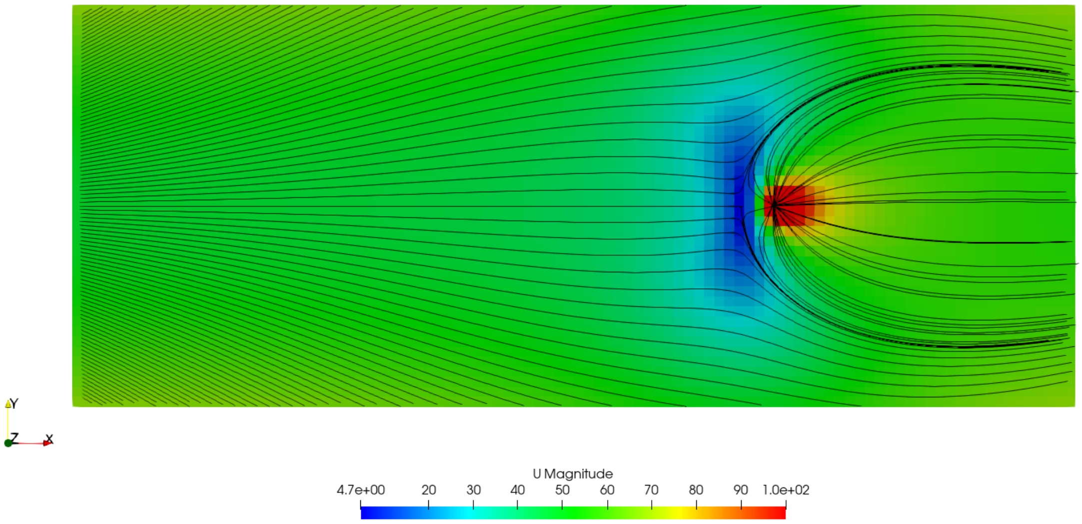

Figure 1: streamlines contour of semi-infinite body

The transformation equations used to convert polar velocities to cartesian ones is given at Equations 6 and 7.

$$U_x = V_r \cdot cos(\theta) - r \cdot V_\theta \cdot sin(\theta) \tag{6}$$
$$U_y = V_r \cdot sin(\theta) + r \cdot V_\theta \cdot cos(\theta) \tag{7}$$

3 Computation Methodology

The steps followed in OpenFOAM to generate the semi-infinite body are given below.

1. a new custom application was made in OpenFOAM v2206 and named as *semiInfiniteBody* using command *foamNewApp appName*.

2. totally 6 new volScalarFields and 1 volVectorField were created to store the corresponding fields computed using the equations mentioned in the Section 2.

3. a loop over cells is created using which the field values for each cell is computed and then at the end of loop the solution fields will be writen to the *0* folder in the case directory.

The result obtained using the OpenFOAM code is shown as a contour with streamlines that were made using *ParaView*, in Figure 1.

4 Instructions

The instruction to generate/execute the files present in this work is given below.

1. copy the contents of this entire root folder named *01_RankineSemiInfiniteBody* to a new directory.

2. go into the subfolder named *03_OpenFOAMCode* which contains another sub-subfolder named *semiInfiniteBody*, go into that using the terminal which is enabled with OpenFOAM environment.

3. in the terminal, type *wclean* and press enter, to clean any previous compilation files. Then type *wmake* and press enter, this will compile the application.

4. after compilation, cd to the directory named *02_testCase* and execute the command named *semiInfiniteBody*. this should compute the field values and store them in the *0* folder.

5. after computation, use *ParaView* for visualization of results.