FOSSEE, IIT Bombay
OpenFOAM Case Study Project
Mary 29, 2021

CFD Simulation of flow through pipe: A comparative study on cyclic and inlet-outlet type boundary conditions in OpenFOAM

Nikhil Sharma

Department of Mechanical Engineering
Malaviya National Institute of Technology Jaipur, Rajasthan India

Abstract

The objective of this project is to understand cyclic boundary condition and use it to simulate laminar and turbulent flow through a cylindrical pipe of constant cross section using the open source CFD package OpenFOAM. In this project, flow through a periodic pipe is simulated using pimpleFoam solver and the obtained results are compared with those of a full length pipe and available analytical results. Cyclic boundary condition is observed to significantly reduce the computational effort and the results obtained are found to be in very close agreement with full length pipe.

1. Introduction

Cyclic or periodic boundary conditions are often used in industrial CFD simulations to save computational cost. It treats two selected boundary faces as if they are physically connected. The flow exiting from one face then enters the other face. Mathematically, flux across the two cyclic faces is same in magnitude but opposite in sign. There are two types of cyclic boundary conditions: Translational and Rotational. Translational cyclic boundaries are frequently encountered when simulating heat exchangers and long pipes while rotational periodic boundaries may be used in turbomachinery applications[3].

2. Problem Statement

The objectives of this project are

- To solve for a laminar pipe flow through a pipe using cyclic boundary conditions and compare the results with that of a full length pipe and analytical results obtained using the Hagen–Poiseuille equation.

- To solve for a turbulent flow (using standard k-ϵ model) through a pipe using cyclic boundary conditions and compare the results with that of a full length pipe.
- To solve for the same turbulent flow using different turbulence models (realizable k-ϵ, k-ω and k-ω SST) and compare the results with each other.

When using cyclic boundary conditions, a source term is needed to drive the flow. To add this source term without modifying the source code *fVOptions* functionality is used. This functionality can be used with solvers that deal with advanced modeling capabilities. However, icoFoam is a very basic solver without any modelling capabilities and can not be used in our case. Hence, in our case pimpleFoam solver has been used for both laminar and turbulent flows.

3. Governing Equations

pimpleFoam equations

Following governing equations are solved by the pimpleFoam solver:

$$\nabla.U = 0 \tag{1}$$

$$\frac{\partial U}{\partial t} + \nabla.(U \otimes U) - \nabla.\mathbf{R}^{\text{eff}} = -\nabla p + \mathbf{S}_u \tag{2}$$

Here, $\mathbf{R}^{\text{eff}}$ is the stress tensor and $\mathbf{S}_u$ is the momentum source.

Hagen–Poiseuille equation

The Hagen–Poiseuille equation for a laminar flow through pipe is:

$$\Delta p = \frac{8\mu LQ}{\pi R^4} \tag{3}$$

Here, Δp is the pressure difference between the two ends, L is the length of pipe, μ is the dynamic viscosity, Q is the volumetric flow rate and R is the pipe radius.

Equations for Cyclic boundary

For a fully developed flow, the velocity field repeats itself in a succession of cross sections that are separated from each other by the period length(L) . For a duct/pipe with variable cross section, L is a parameter based on the periodically varying cross section. For a pipe having uniform cross section, any value of this period of variation can be taken as velocity field does not change sufficiently far from the inlet (after the entrance region).
The pressure field however does not show similar type of periodic behaviour. For a fully developed

flow far from its inlet, pressure drop(Δp) exhibits a periodic behaviour as shown in equation 4 .

$$p(x) - p(x+L) = p(x+L) - p(x+2L) = p(x+2L) - p(x+3L) = ... \tag{4}$$

here x is the flow direction

Hence the pressure field can be divided into two components:

$$\begin{aligned} p(x) &= \beta x + \tilde{p}(x) \\ \beta &= \frac{p(x) - p(x+L)}{L} \end{aligned} \tag{5}$$

In equation 5, the term βx relates to the global mass flow and $\tilde{p}(x)$ is known as the reduced pressure responsible for local motion and shows the same type of periodic behaviour as velocity field[4].

4. Simulation Procedure

Setting up a case in OpenFOAM requires three directories namely *0*, *constant* and *system*. Simulation is run by typing the commands in terminal. Transient, incompressible, turbulent solver pimpleFoam is used for the simulations. Table 1 below shows the different parameters used in the present case.

	Unit	Laminar	Turbulent
Density(ρ)	kgm^{-3}	996	996
Dynamic viscosity(μ)	$kgm^{-1}s^{-1}$	$7.98 * 10^{-4}$	$7.98 * 10^{-4}$
Reynolds Number	-	2100	10^5
Mean velocity	ms^{-1}	$8.4 * 10^{-2}$	4.01
Radius of the pipe	m	0.01	0.01
Length of the pipe	m	3	2

Table 1: Different parameters used in the simulations

4.1 Geometry and Mesh

Figure 1 shows the computational domain. As the flow is axis-symmetric simulations are performed on a sector of the cross section of pipe used which basically is a 2D geometry with wedge type boundary condition imposed on planes in swirl direction. The sector in the geometry is of angle 3°. A structured mesh having only hexahedra cells is used. Meshing was done using *blockMesh* utility. The mesh has been refined such that the first cell near the wall lies in the log-law region. The length of the pipe is taken to be more than the entrance length. For laminar flow at 2100 Reynolds Number, entrance length is calculated to be 2.52m and for turbulent flow at 10^5 Reynolds Number, the entrance length is calculated to be 0.59m. The entrance length was calculated using equation 6.

$$L_e/D = 0.06 * Re, \text{ for laminar flow}$$
$$L_e/D = 4.4 * Re^{1/6}, \text{ for turbulent flow} \quad (6)$$

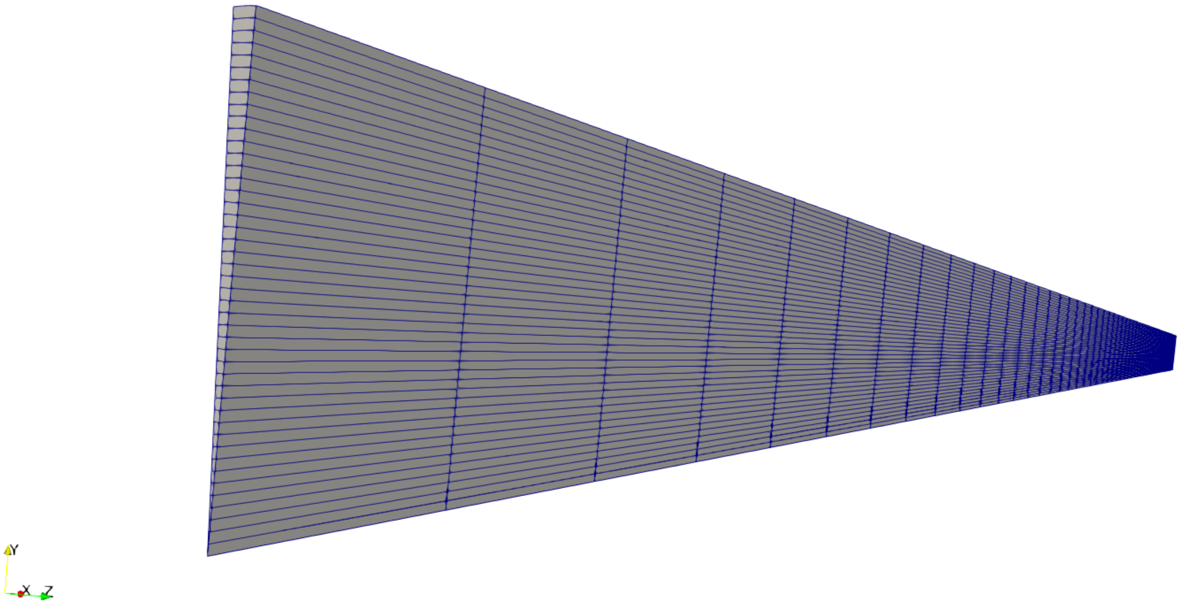

Figure 1: Computational domain

4.2 Boundary Conditions

Boundary conditions used in laminar flow through a full length pipe are as follows:

- **U**

inlet :	fixedValue
outlet:	zeroGradient
wall :	noSlip
wedgeFront :	wedge
wedgeBack :	wedge

- **p**

inlet :	zeroGradient
outlet:	fixedValue
wall :	zeroGradient
wedgeFront :	wedge
wedgeBack :	wedge

Boundary conditions used in turbulent flow through a full length pipe are as follows:

- **U**

inlet :	fixedValue
outlet:	zeroGradient
wall :	noSlip
wedgeFront :	wedge
wedgeBack :	wedge

- **epsilon**

inlet :	turbulentMixingLengthDissipationRateInlet
outlet:	zeroGradient
wall :	epsilonWallFunction
wedgeFront :	wedge
wedgeBack :	wedge

- **p**

inlet :	zeroGradient
outlet:	fixedValue
wall :	zeroGradient
wedgeFront :	wedge
wedgeBack :	wedge

- **nut**

inlet :	fixedValue
outlet:	zeroGradient
wall :	nutKWallFunction
wedgeFront :	wedge
wedgeBack :	wedge

- **k**

inlet :	turbulentIntensityKineticEnergyInlet
outlet:	zeroGradient
wall :	kqRWallFunction
wedgeFront :	wedge
wedgeBack :	wedge

For the flow through cyclic pipe, boundary conditions at the coupled faces(inlet and outlet) are specified to be of cyclic type. The value of velocity, pressure or any other parameter need not to be specified at the coupled boundary faces. Boundary conditions for all the other parameters at all the other boundary faces the are same as that in a full length pipe.

4.3 Solver

We need to analyze a transient, turbulent flow with cyclic boundary imposed. To do so we have used pimpleFoam solver which utilizes the **PIMPLE** algorithm which is a combination of **PISO** (Pressure Implicit with Splitting of Operator) and **SIMPLE** (Semi-Implicit Method for Pressure-Linked Equations).

The cyclic boundary condition is used to reduce computational cost when the geometry and expected flow profiles are periodically repeating in nature. It treats two boundary patches as if they are physically connected to each other, thus there is no driving force for the flow. An additional source term is needed to specify momentum source. An explicit mean pressure gradient is imposed to obtain specified bulk velocity. OpenFOAM implementation of the same is shown in listings 1 and 2.

```
tmp<fvVectorMatrix> tUEqn
(
    fvm::ddt(U) + fvm::div(phi, U)
  + MRF.DDt(U)
  + turbulence->divDevSigma(U)
 ==
    fvOptions(U)
);
```

Listing 1: Discretization of momentum equation(UEqn.H)

```
    scalar gradP = gradP0_ + dGradP_;

    UIndirectList<vector>(Su, cells_) = flowDir_*gradP;

    eqn += Su;
```

Listing 2: Addition of source term to discretized equation(meanVelocityForce.C)

fVOptions utility is used to give a single region momentum source namely "*meanVelocityForce*". This utility calculates a momentum source so that the volume averaged velocity in the selected computational domain (either all the cells or selected cellsets or cellzones) reaches the desired mean velocity (Ubar). All the required parameters are stored in *fvOptions* file which is contained in the *constant* directory. All entries of *fvOptions* file used in the present case are shown below in listing 3.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      fvOptions;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

momentumSource
{
    type            meanVelocityForce;
    active          yes;

    meanVelocityForceCoeffs
    {
        selectionMode   all;                    // Apply force to all cells
        fields          (U);                    // Name of velocity field
        Ubar            (4.01 0 0);             // Desired mean velocity
       relaxation       1;                    // Optional relaxation factor
    }
}
```

Listing 3: constant/fvOptions

The step by step procedure of solving the momentum equation(UEqn.H) with cyclic boundary is shown as a flowchart in figure 2.

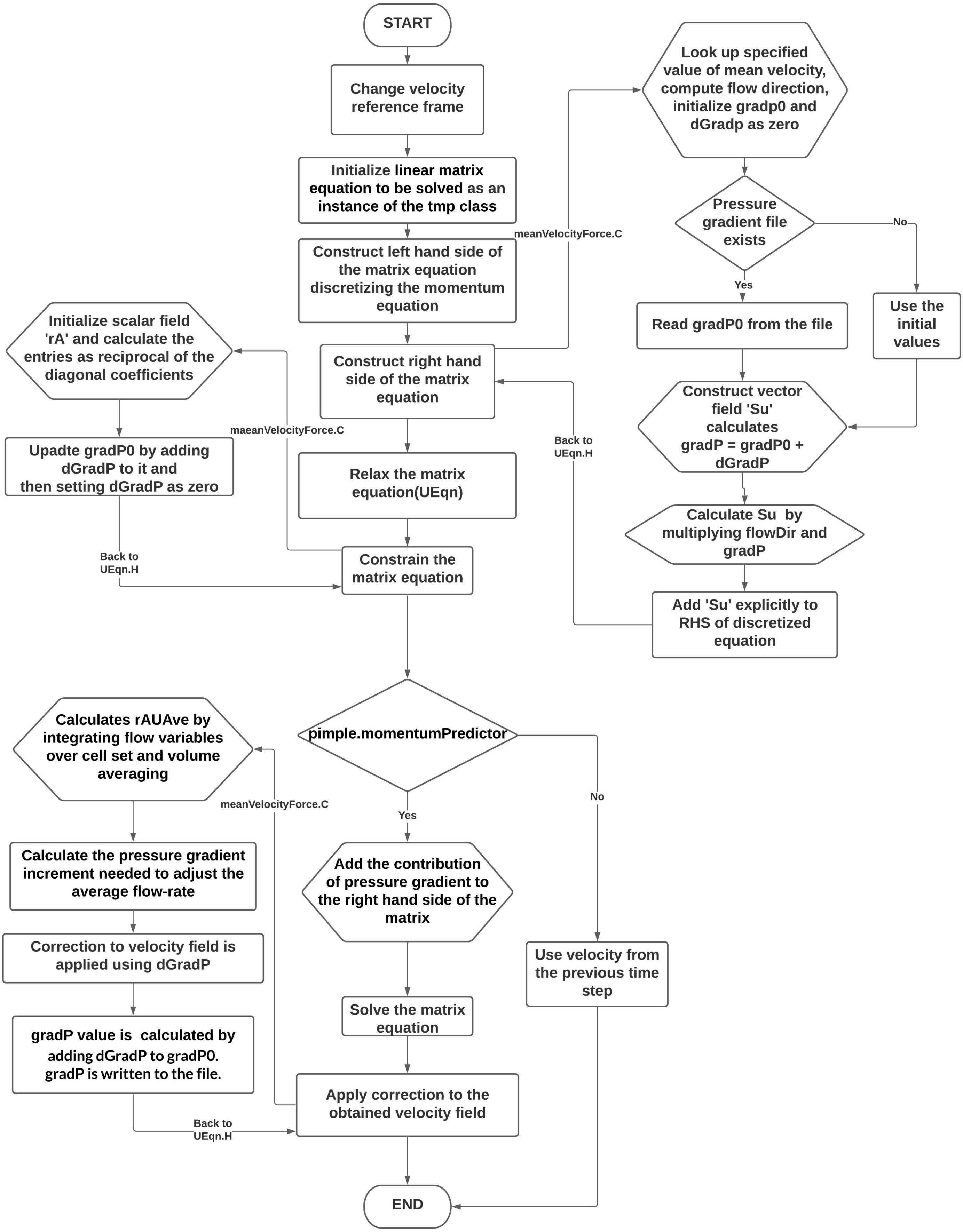

Figure 2: Flowchart of UEqn.H

5. Results and Discussion

5.1 Laminar flow through pipe

Velocity profiles for the fully developed laminar flow obtained using full length and cyclic pipe are compared with each other in figure 3. Both the velocity profiles are very close to each other. Comparison of the CFD and analytical results is shown in table 2. Pressure gradient[1] for cyclic boundary was obtained from the "momentumSourceProperties" file and for the full length pipe average pressure gradient was calculated. The CFD results closely match the analytical results.

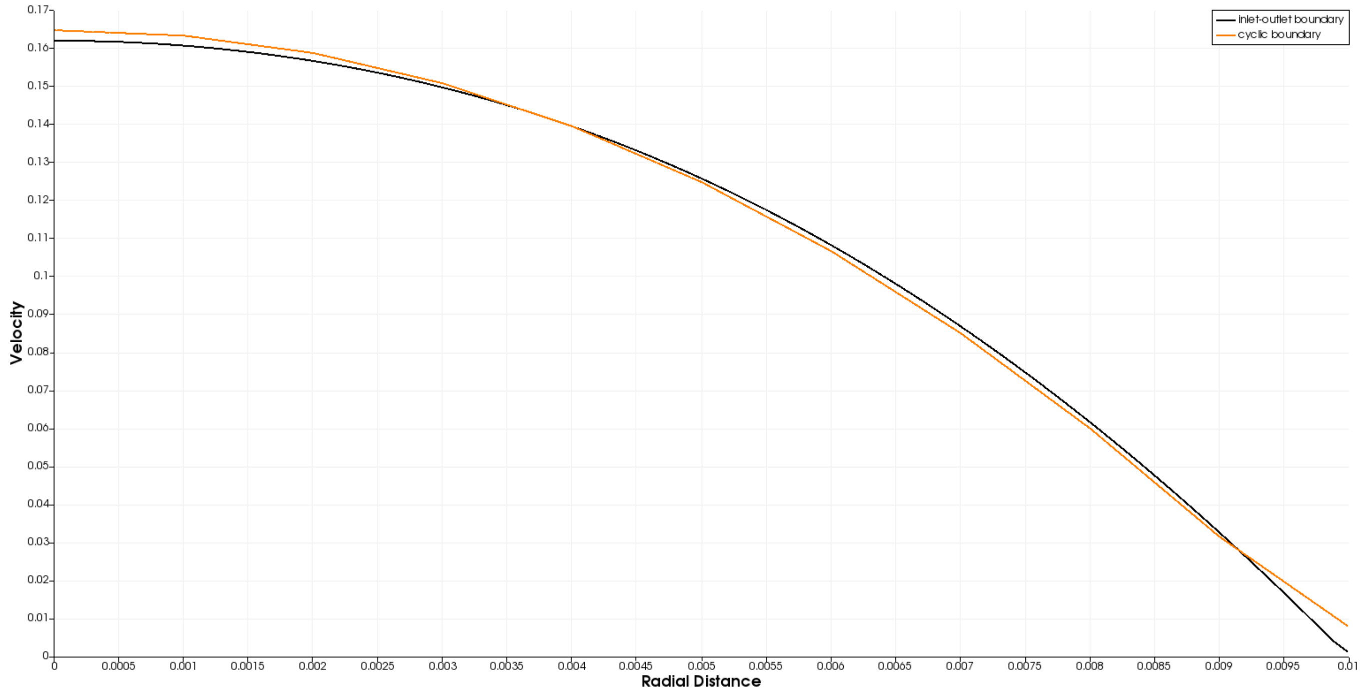

Figure 3: Velocity profile for laminar flow

	Analytical	Full length pipe	Cyclic pipe
Maximum velocity (ms^{-1})	0.168	0.162	0.164
Volume flow rate (m^3s^{-1})	$2.638 * 10^{-5}$	$2.640 * 10^{-5}$	$2.640 * 10^{-5}$
Pressure gradient (Pam^{-1})	5.369	5.3710	5.375

Table 2: Comparison of the CFD and analytical results

[1] Incompressible solvers in OpenFOAM such as pimpleFoam solve for kinematic pressure(m^2s^{-2}), to obtain the pressure field in Pascals it must be multiplied with fluid density

5.2 Turbulent flow through pipe

Figures 4, 5 and 6 below show the velocity profile, epsilon profile and kinetic energy profile respectively.

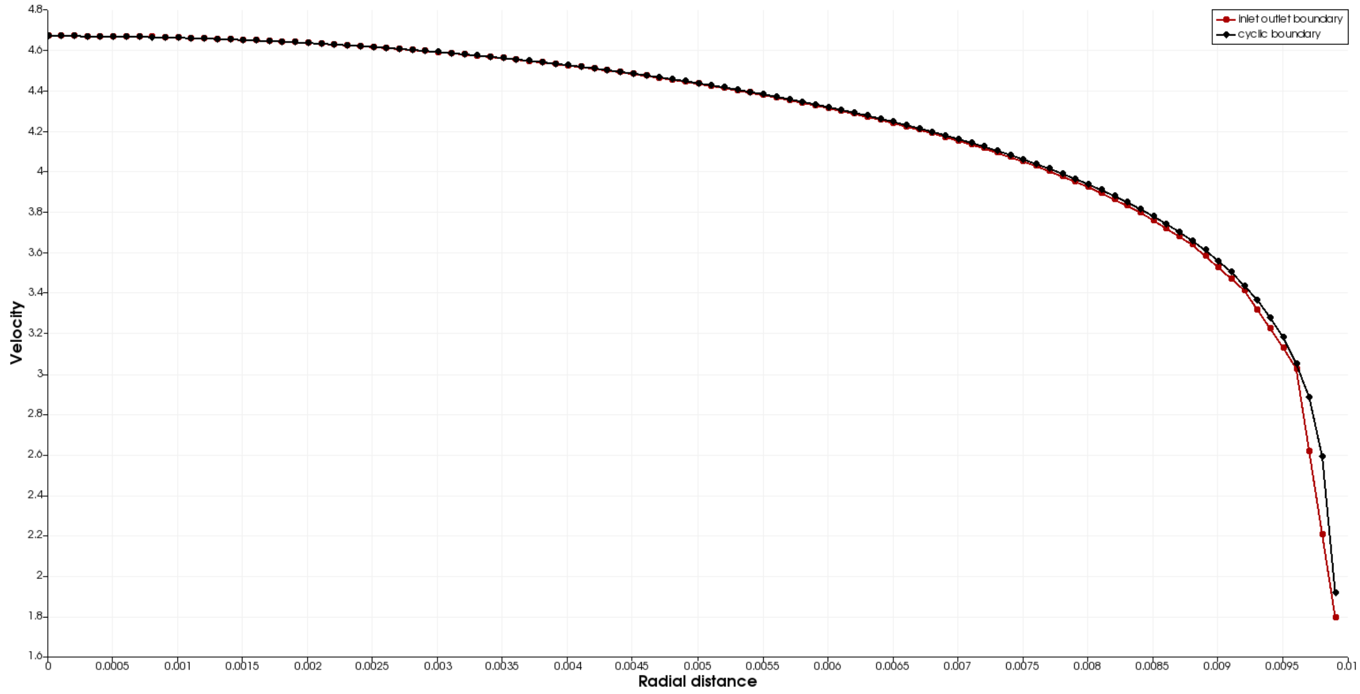

Figure 4: Velocity profile

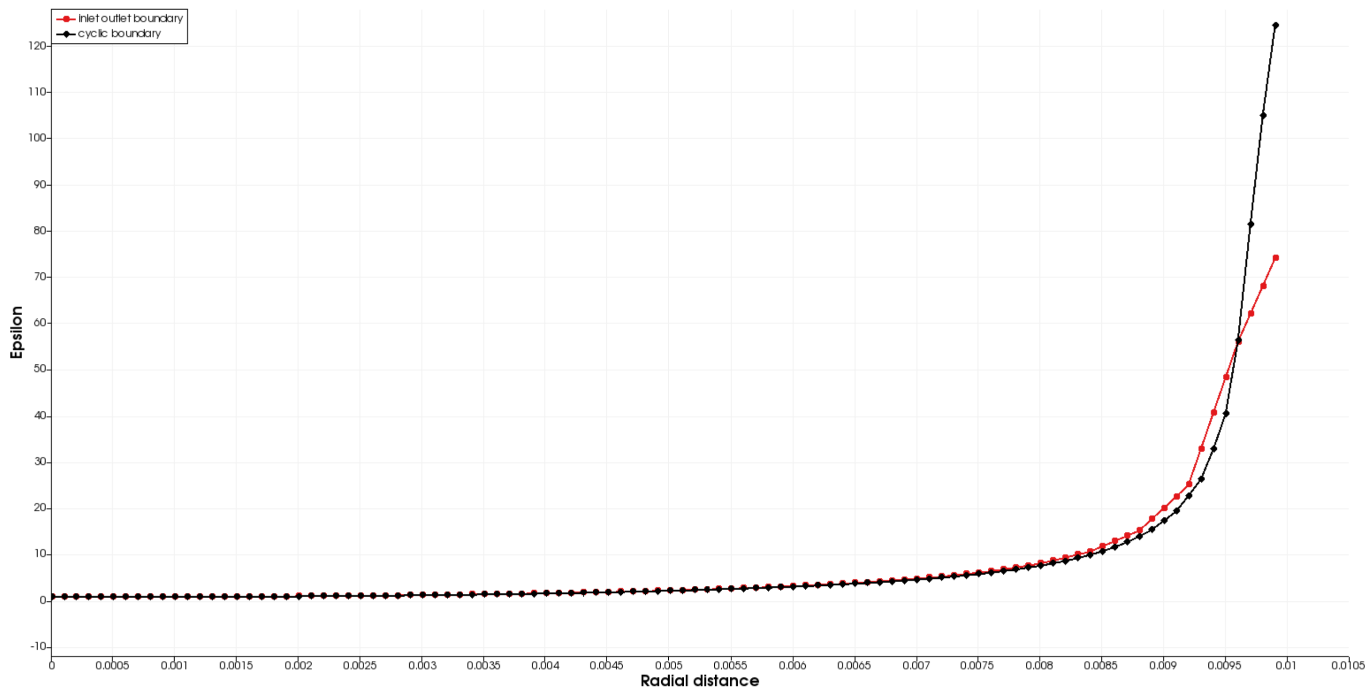

Figure 5: Epsilon profile

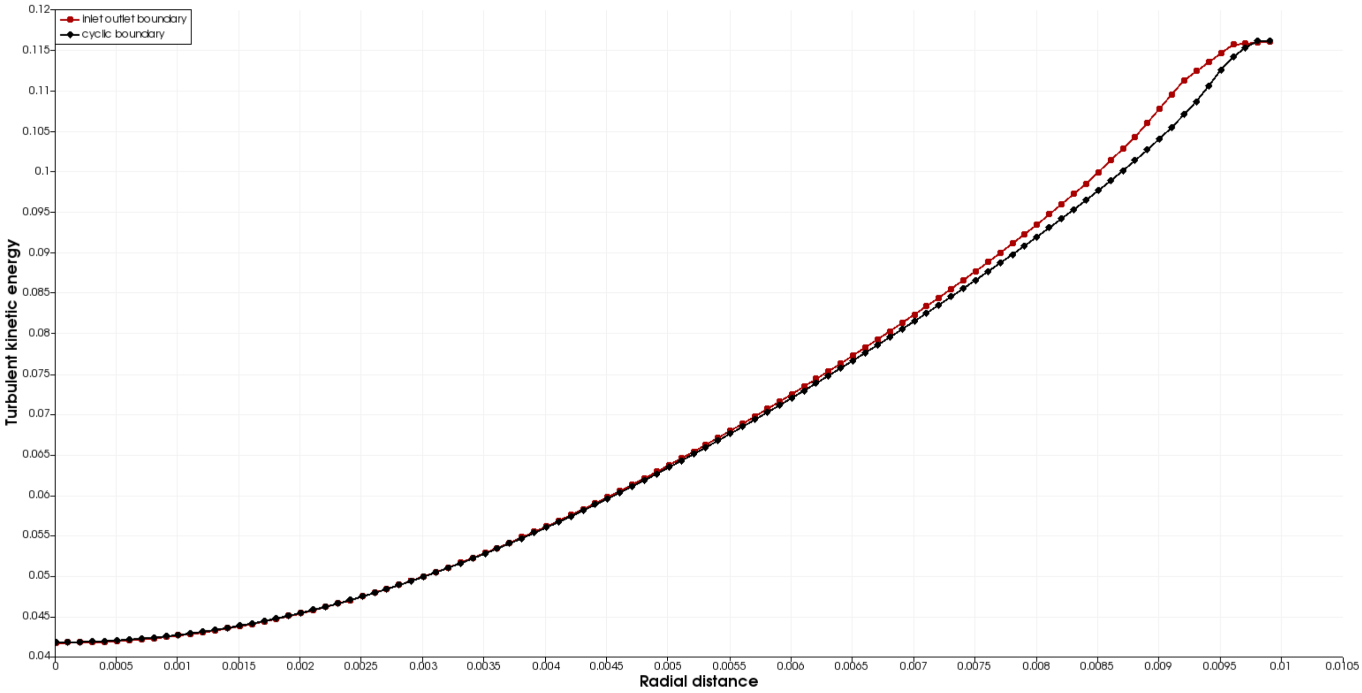

Figure 6: TKE profile

Table 3 below shows the average value of various parameters at different cross sections of full length pipe. From the table it is seen that the parameters k and ϵ show same kind of periodic behaviour as velocity in the fully developed region. The pressure field exhibits the type of behaviour as explained in equation 4. The average pressure gradient

- in full length pipe = 7162.634 Pa/m(**average pressure gradient through full length of pipe**)
- in fully developed region of full length pipe = 6958.244 Pa/m(**average pressure gradient between x=1.6 and x=2**)
- in cyclic pipe = 6945.924 Pa/m(**read from "momentumSourceProperties" file**)

Distance from the inlet	epsilon	k	p(kinematic)	Pressure drop	Gradient
0	6.41575	0.0638408	14.3828		
0.4	12.8026	0.0712957	11.1777	3.2051	8.01275
0.8	13.5405	0.0844199	8.41942	2.75828	6.8957
1.2	13.4208	0.0825549	5.62529	2.79413	6.985325
1.6	13.4245	0.0825803	2.82941	2.79588	6.9897
2	13.4268	0.0826236	0.0349342	2.7944758	6.9861895

Table 3: Average values at different cross sections from the inlet

The pressure drop in the region close to inlet is higher than the far from inlet region. The higher pressure drop in region close to inlet is due to the entrance region loss in pipe. Sufficiently far from the inlet (in the fully developed region), the effect of entrance loss fades away. In the cyclic pipe, where the solver solves for fully developed region and the effects of entrance region loss are not observed. Thus, differences in value of pressure gradient are observed.

It can be seen from the OpenFoam implementation of momentum equation with added source term(listings 1 and 2) that pressure field in equation with cyclic boundary represents **reduced pressure** which is responsible for local motion while the source term(**Su**) corresponds to "β" which is responsible for global mass flow. This reduced pressure exhibits cyclic behaviour similar to velocity profile.

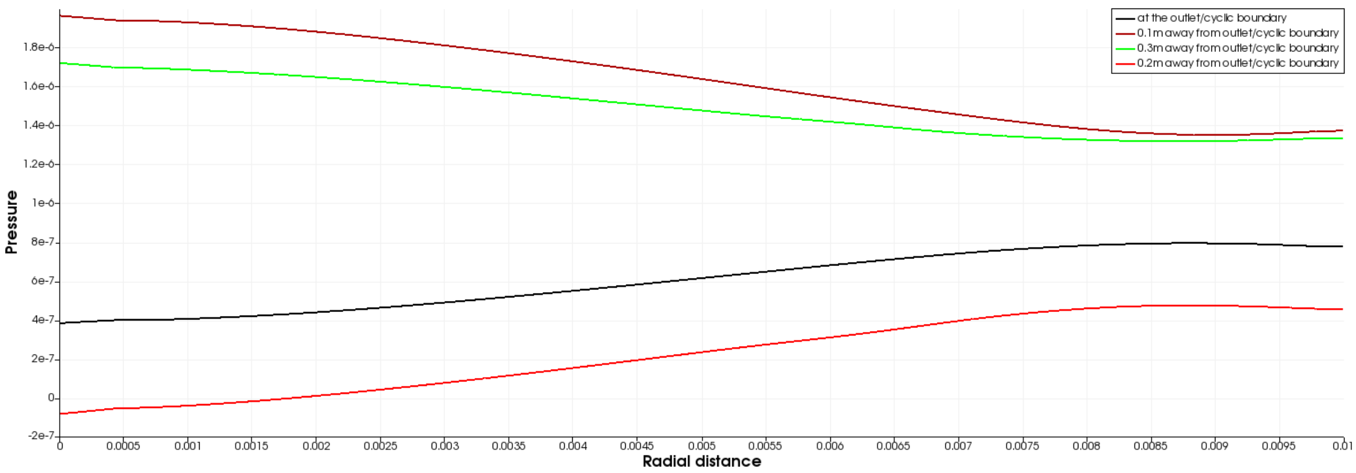

Figure 7: Variation of reduced pressure with the radial distance

In figure 7 reduced pressure at different cross sections of the cyclic pipe is plotted against the radial distance. The difference in the values of reduced pressure at different cross sections is of the order 10^{-7}, which can be neglected. Hence, the pressure field is observed to show periodic behaviour.

5.3 Comparison with other turbulence models

The turbulent flow through pipe in section 5.2 was solved again using different turbulence models. Velocity profiles for the different models seemed to be very close to each other. Figures 8, 9 and 10 show the plot of TKE, epsilon and omega obtained using cyclic boundary and different turbulence models.

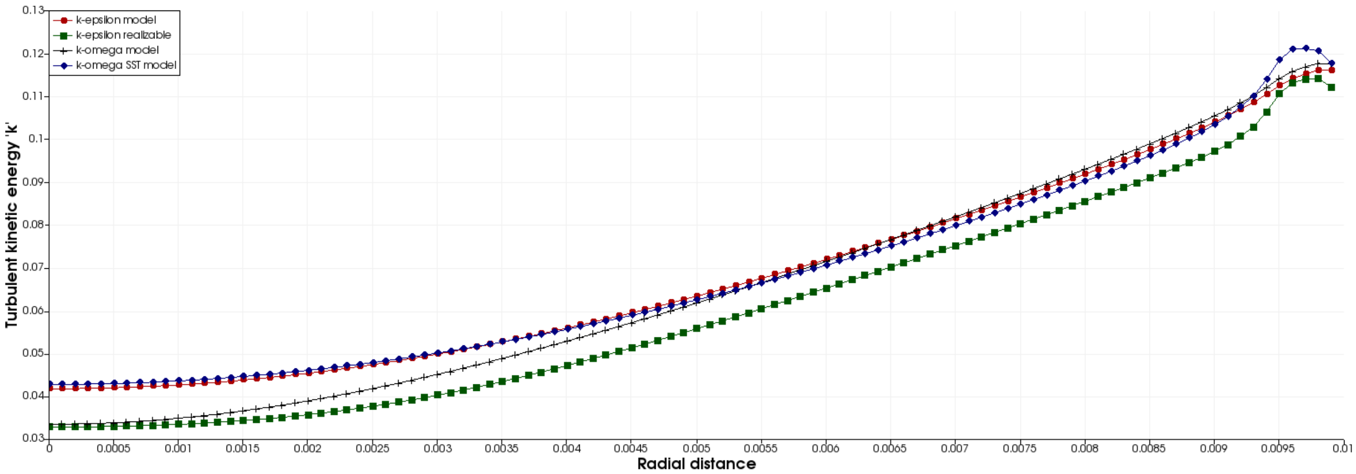

Figure 8: TKE plot for different turbulence models

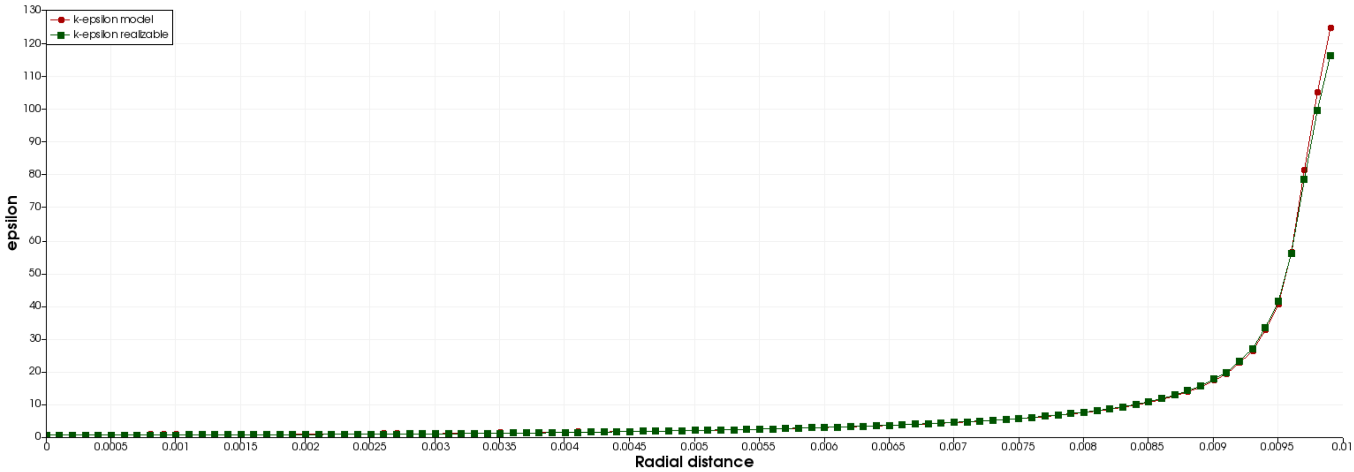

Figure 9: epsilon plot for different turbulence models

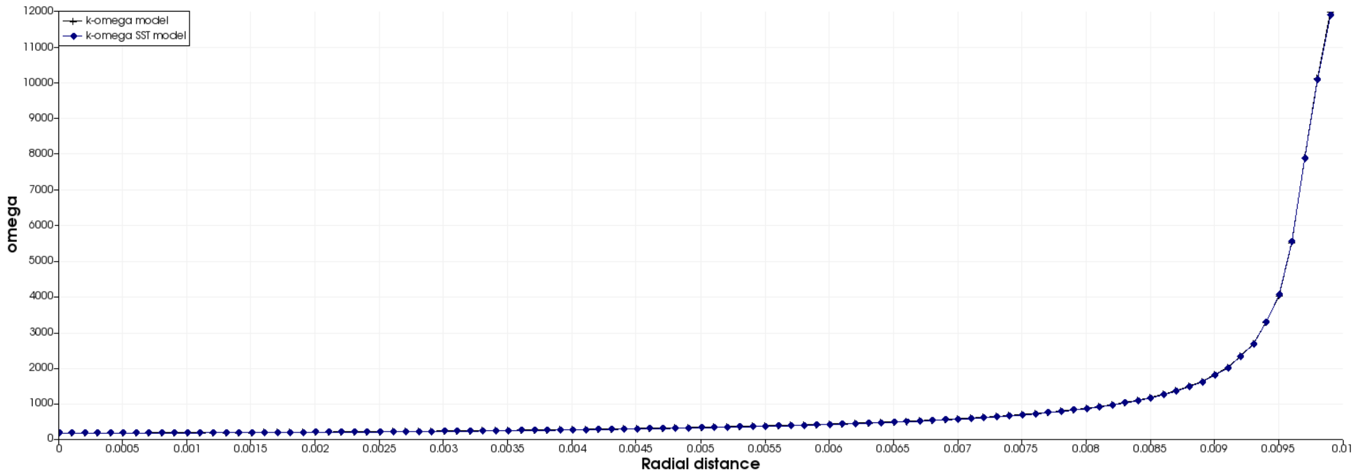

Figure 10: omega plot for different turbulence models

TKE predictions obtained using standard k-ϵ and k-ω SST are not much different in far from the wall region as the SST model uses a blend of k-ϵ and k-ω.The pressure gradient values obtained using different turbulence models are shown above in table 4.

Model	Pressure gradient(Pa/m) in pipe			Error(%) in **(A)**	
	(A) Cyclic	**(B)** Full length	**(C)** Fully developed region	Wrt. **(B)**	Wrt. **(C)**
Standard k-ϵ	6945.924	7162.634	6958.155	3.02	0.17
Realizable k-ϵ	6475.962	6866.971	6527.784	5.69	0.78
Standard k-ω	7021.302	7342.362	7077.650	4.37	0.78
k-ω SST	6807.809	7026.282	6830.916	3.10	0.34

Table 4: Pressure gradient for different turbulence models

From the table, it is seen that for different turbulence models pressure gradients predicted using cyclic boundary conditions closely match the full length pipe results. Full length pipe predicts slightly higher values of pressure gradient due to entrance losses. These losses are not accounted for cyclic boundaries, the same can be verified from table 3 by looking at low values of percentage error between pressure gradient in cyclic pipe and fully developed region of full length pipe.

Conclusions

By simulating flow through this cyclic pipe, it is seen that cyclic boundaries can accurately predict the flow profiles and significantly reduce the computational effort as it is difficult to capture fine details around the boundary layer and describe the gross motion at the same time. Cyclic boundaries are thus used, which considers the flow profiles to be fully developed and avoids the problem of solving the entrance region.

For a full length pipe, pressure drop and pressure gradients are high in the entrance region which increases overall pressure gradient. This increase in pressure gradient becomes negligible if length of the pipe is increased. In a cyclic pipe, this increase in pressure gradient is not accounted for as it considers pipe to be infinitely long. Also, when using cyclic boundaries the values of TKE dissipation rate are significantly large near the wall . This larger value implies higher shear stress and hence higher pressure drop. Thus the pressure gradient values obtained using cyclic and full length pipe slightly differ from each other and the difference is well within acceptable limits.

Upon comparing results obtained using different turbulence models it can be concluded that the velocity results are less sensitive to the choice of turbulence model while pressure field is very sensitive to choice of turbulence model. Also it is seen that the differences caused by choice of turbulence model are very well captured when using cyclic boundaries.

References

[1] C. J. Greenshields et al. Openfoam user guide. *OpenFOAM Foundation Ltd, version*, 3(1):47, 2015.

[2] G. Holzinger. Openfoam a little user-manual. *CD-Laboratory-Particulate Flow Modelling, Johannes Keplper University: Linz, Austria*, 2015.

[3] J. Murthy and S. Mathur. Periodic flow and heat transfer using unstructured meshes. *International journal for numerical methods in fluids*, 25(6):659–677, 1997.

[4] S. Patankar, C. Liu, and E. Sparrow. Fully developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area. 1977.