

Introduction to Computational Fluid Dynamics using OpenFOAM and Octave

Dr. Lakshman Anumolu (Sr. Research Engineer)

Kumaresh Selvakumar (PhD candidate)

(Session-12)

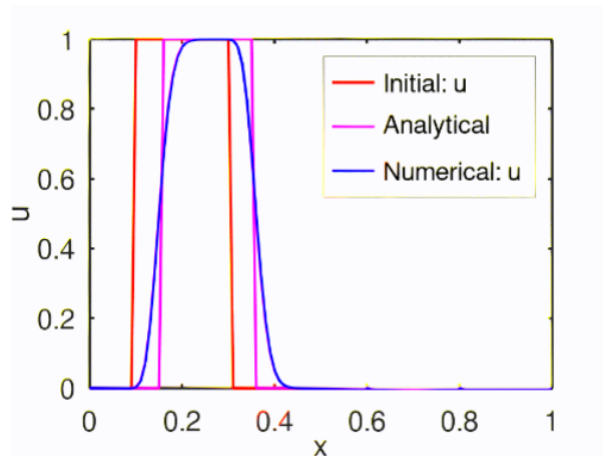
Instructions: Mon, Wed, Thu (5:30PM-6:30PM IST)

Query session: Sundays 8AM-8:30AM IST

Quick Recap

What Did We Discuss?

Numerical Solution to Convection Equation



$$c = 0.1; \Delta x = 0.01; \Delta t = 0.01 \quad CFL: \frac{c\Delta t}{\Delta x} = 0.1$$

$$u_i^{n+1} = u_i^n - c\Delta t \left(\frac{\partial u}{\partial x} \right)_i^n$$

$\left(\frac{\partial u}{\partial x} \right)_i^n \approx \frac{u_i^n - u_{i-1}^n}{\Delta x_i}$ Upwind

```
1  #include <iostream>
2  #include <vector>
3  #include <cmath>
4
5  std::vector<float> compute_rate_of_convergence(std::vector<float> errors, std::vector<float> dx) {
6      std::vector<float> roc;
7
8      for (int i = 0; i < errors.size()-1; ++i) {
9          float r = log(errors[i]/errors[i+1])/log(dx[i]/dx[i+1]);
10
11         roc.push_back(r);
12     }
13
14     return roc;
15 }
16
17 int main()
18 {
19     std::vector<float> errors({0.16, 0.0775, 0.038125});
20     std::vector<float> dx({0.1, 0.05, 0.025});
21
22     std::vector<float> roc = compute_rate_of_convergence(errors, dx);
23
24     for (int i = 0; i < roc.size(); ++i) {
25         std::cout << roc[i] << std::endl;
26     }
27
28     return 0;
29 }
```

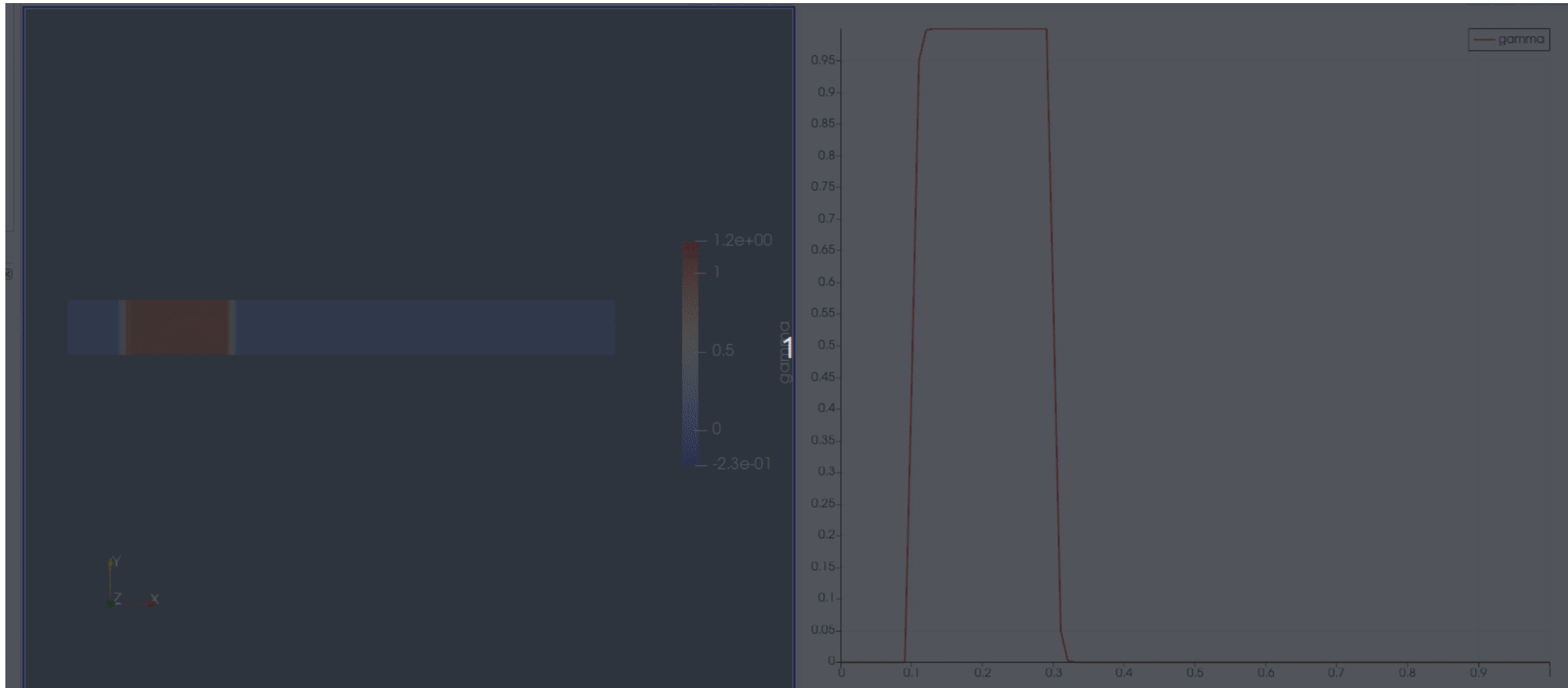
Current Session

Overview

- **OpenFOAM:** Numerical Solution to convection Equation
- Introduction to C++ for OpenFOAM (contd.)

Numerical Solution to Convection Equation

$$\frac{\partial \gamma}{\partial t} + c \frac{\partial \gamma}{\partial x} = 0$$



Numerical Solution to Convection Equation

$$\frac{\partial \gamma}{\partial t} + c \frac{\partial \gamma}{\partial x} = 0$$

$$\frac{\partial \gamma}{\partial t} + \frac{\partial c\gamma}{\partial x} = 0$$

$$\frac{\partial \gamma}{\partial t} + \nabla \cdot (c\gamma) = 0$$

Introduction to C++ for OpenFOAM

a12_roc.cpp

Next Session

- Finite volume method to solve convection-diffusion equation in OpenFOAM
- Introduction to C++ for OpenFOAM (Contd.)

Thank you