

# Special Topics in CFD

DAY 7

## Convection problem in OpenFOAM

---

*Kumaresh*

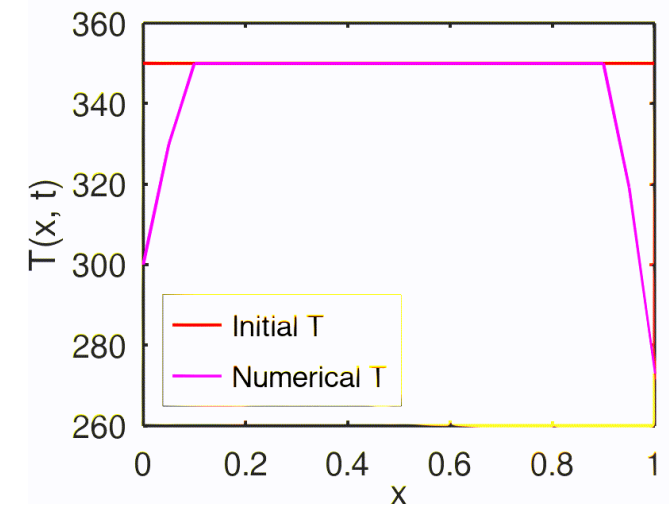
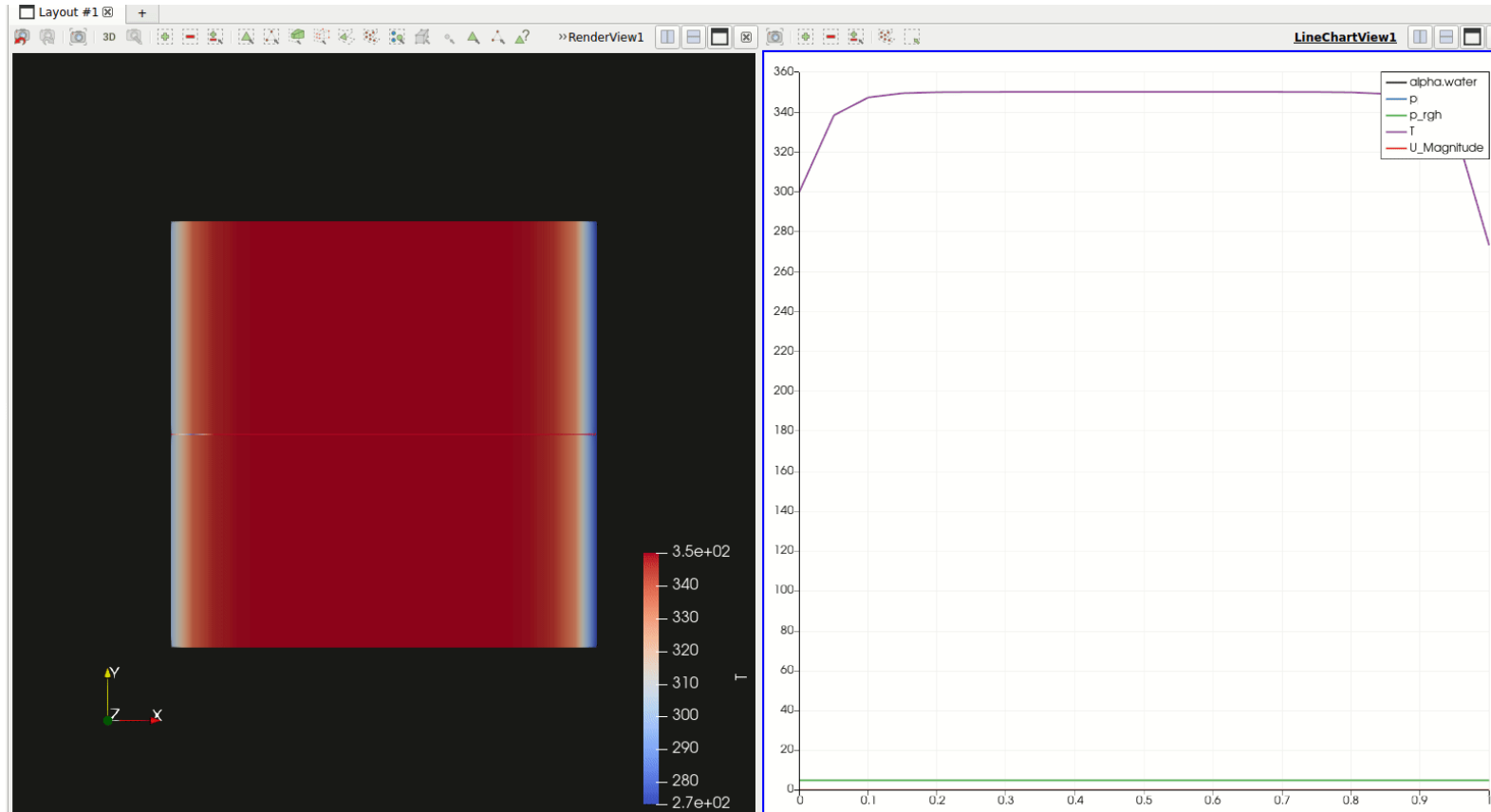
# Contents

➤ Explanation about Convection equation

➤ Exercise






# OpenFOAM: Numerical Solution to Diffusion Equation

Paraview – Post Processing tool in OpenFOAM



$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

# Extract files from GitHub

 Kumaresh0402 Add files via upload	
Name	Last commit message
 ..	
 DAY6-OpenFOAM_diffusion_equation.rar	Files
 DAY6_CFD.pdf	Add files via upload
 Readme.md	Update Readme.md
Readme.md	
Project 1: <a href="#">#7</a>	

# solver file → diffusionFoam/createFields.H

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

ComputationalThermalEngineering / DAY7-OpenFOAM\_diffusion\_equation / diffusionFoam / createFields.H

Kumaresh0402 Add files via upload

Code Blame 49 lines (43 loc) · 846 Bytes Code 55% faster with GitHub Copilot

```
1 Info<< "Reading field T\n" << endl;
2
3 volScalarField T
4 (
5     IObject
6     (
7         "T",
8         runTime.timeName(),
9         mesh,
10        IObject::MUST_READ,
11        IObject::AUTO_WRITE
12    ),
13    mesh
14 );
15
16
17 Info<< "Reading diffusivity DT\n" << endl;
18
19 volScalarField DT
20 (
21     IObject
22     (
23         "DT",
24         runTime.timeName(),
25         mesh,
26        IObject::READ_IF_PRESENT,
27        IObject::AUTO_WRITE
28    ),
29    mesh,
30    dimensionedScalar(dimViscosity, Zero)
31 );
```

Temperature “T” field is created

Diffusivity as a field created

# solver file → diffusionFoam/diffusionFoam.C

```
#include "fvCFD.H"
#include "fvOptions.H"
#include "simpleControl.H"

// ..... //

int main(int argc, char *argv[])
{
    argList::addNote
    (
        "Laplace equation solver for a scalar quantity."
    );

    #include "postProcess.H"

    #include "addCheckCaseOptions.H"
    #include "setRootCaseLists.H"
    #include "createTime.H"
    #include "createMesh.H"

    simpleControl simple(mesh);

    #include "createFields.H"

    // ..... //

    Info<< "\nCalculating temperature distribution\n" << endl;

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

        while (simple.correctNonOrthogonal())
        {
            fvScalarMatrix TEqn
            (
                fvm::ddt(T) - fvm::laplacian(DT, T)
                ==
                fvOptions(T)
            );
```

Necessary “header” files

Matrix is created to solve “FVM”

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

```
while (simple.correctNonOrthogonal())
{
    fvScalarMatrix TEqn
    (
        fvm::ddt(T) - fvm::laplacian(DT, T)
        ==
        fvOptions(T)
    );

    fvOptions.constrain(TEqn);
    TEqn.solve();
    fvOptions.correct(T);
}

#include "write.H"
runTime.write();

runTime.printExecutionTime(Info);
}

Info<< "End\n" << endl;

return 0;
}
```

# Compile diffusionFoam “solver”

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ ls
CASE1_diffusionFoam  diffusionFoam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ cd diffusionFoam/
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$ wmake
Making dependencies: diffusionFoam.C
g++ -std=c++11 -m64 -pthread -DOPENFOAM=2306 -DWM_DP -DWM_LABEL_SIZE=32 -Wall -Wextra -Wold-style-cast -Wnon-virtual-dtor -Wno-unused-parameter -Wno-invalid-offsetof -Wno-attributes -Wno-unknown-pragmas -
03 -DNoRepository -ftemplate-depth-100 -I/home/openfoam/OpenFOAM/openfoam/src/finiteVolume/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/meshTools/lnInclude -I/home/openfoam/Open
FOAM/openfoam/src/OpenFOAM/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/OSspecific/POSIX/lnInclude -fPIC -c diffusionFoam.C -o Make/linux64GccDPInt320pt/diffusionFoam.o
g++ -std=c++11 -m64 -pthread -DOPENFOAM=2306 -DWM_DP -DWM_LABEL_SIZE=32 -Wall -Wextra -Wold-style-cast -Wnon-virtual-dtor -Wno-unused-parameter -Wno-invalid-offsetof -Wno-attributes -Wno-unknown-pragmas -
03 -DNoRepository -ftemplate-depth-100 -I/home/openfoam/OpenFOAM/openfoam/src/finiteVolume/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/meshTools/lnInclude -I/home/openfoam/Open
FOAM/openfoam/src/OpenFOAM/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/OSspecific/POSIX/lnInclude -fPIC -Xlinker --add-needed -Xlinker --no-as-needed Make/linux64GccDPInt320pt/diffusionFoam.o -L/h
ome/openfoam/OpenFOAM/openfoam/platforms/linux64GccDPInt320pt/lib \
-lfiniteVolume -lfvOptions -lmeshTools -lOpenFOAM -ldl \
-lm -o /home/openfoam/OpenFOAM/openfoam-v2306/platforms/linux64GccDPInt320pt/bin/diffusionFoam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$
```



Compiled “diffusionFoam” solver

# case file

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

[ComputationalThermalEngineering](#) / [DAY7-OpenFOAM\\_diffusion\\_equation](#) / **CASE1\_diffusionFOAM** /



**Kumares0402** Delete DAY7-OpenFOAM\_diffusion\_equation/CASE1\_diffusionFOAM/system/as

Name	Last commit message
..	
0	Initial conditions Delete DAY7-OpenFOAM_diffu:
constant	Properties Delete DAY7-OpenFOAM_diffu:
system	Mesh, schemes, solvers, algorithm controls and tolerances for the implicit solvers. Delete DAY7-OpenFOAM_diffu:



# Compile the “case” file named – CASE1\_diffusionFOAM

```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$ cd ..
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ ls
CASE1_diffusionFOAM  diffusionFoam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ cd CASE1_diffusionFOAM/
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ blockMesh

/*
=====
\\  / F ield      | OpenFOAM: The Open Source CFD Toolbox
\\  / O peration  | Version: 2306
\\  / A nd        | Website: www.openfoam.com
\\  / M anipulation
=====
*/
Build : a6e826bd55-20230630 OPENFOAM=2306 version=v2306
Arch  : "LSB;label=32;scalar=64"
Exec  : blockMesh
Date  : Oct 23 2023
Time  : 16:04:30
Host  : openfoam
PID   : 80174
I/O   : uncollated
Case  : /home/openfoam/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM
nProcs : 1
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// *****
Create time

Creating block mesh from "system/blockMeshDict"
Creating block edges
No non-planar block faces defined
Creating topology blocks

Creating topology patches - from boundary section
Creating block mesh topology
```

1

```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ diffusionFoam

/*
=====
\\  / F ield      | OpenFOAM: The Open Source CFD Toolbox
\\  / O peration  | Version: 2306
\\  / A nd        | Website: www.openfoam.com
\\  / M anipulation
=====
*/
Build : a6e826bd55-20230630 OPENFOAM=2306 version=v2306
Arch  : "LSB;label=32;scalar=64"
Exec  : diffusionFoam
Date  : Oct 23 2023
Time  : 16:06:31
Host  : openfoam
PID   : 80220
I/O   : uncollated
Case  : /home/openfoam/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM
nProcs : 1
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// *****
Create time

Create mesh for time = 0

SIMPLE: no convergence criteria found. Calculations will run for 0.1 steps.

Reading field T

Reading diffusivity DT

No finite volume options present

Calculating temperature distribution

Time = 0.001

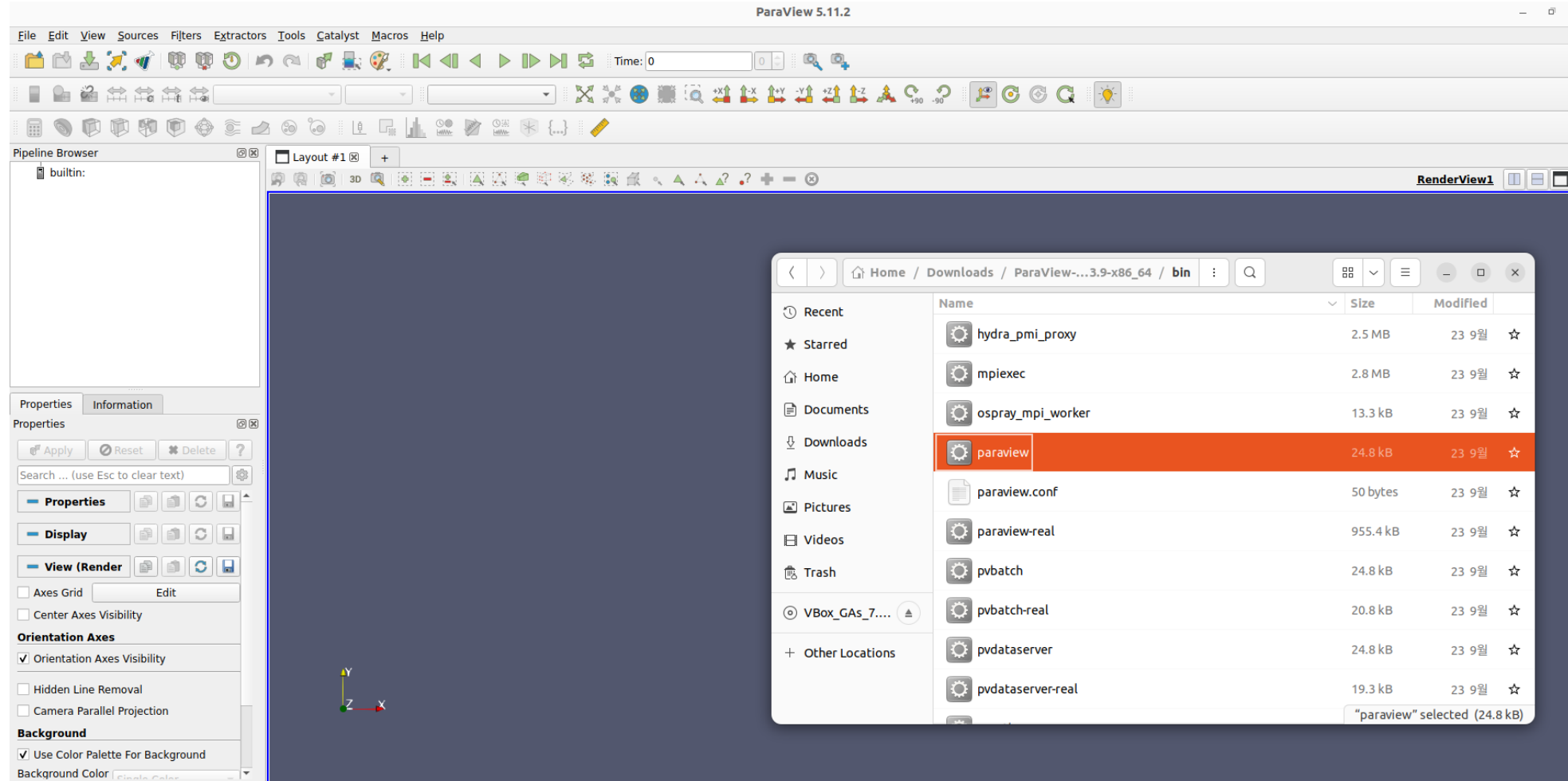
DICPCG: Solving for T, Initial residual = 1, Final residual = 1.90148e-16, No Iterations 1
DICPCG: Solving for T, Initial residual = 3.21798e-15, Final residual = 3.21798e-15, No Iterations 0
```

2

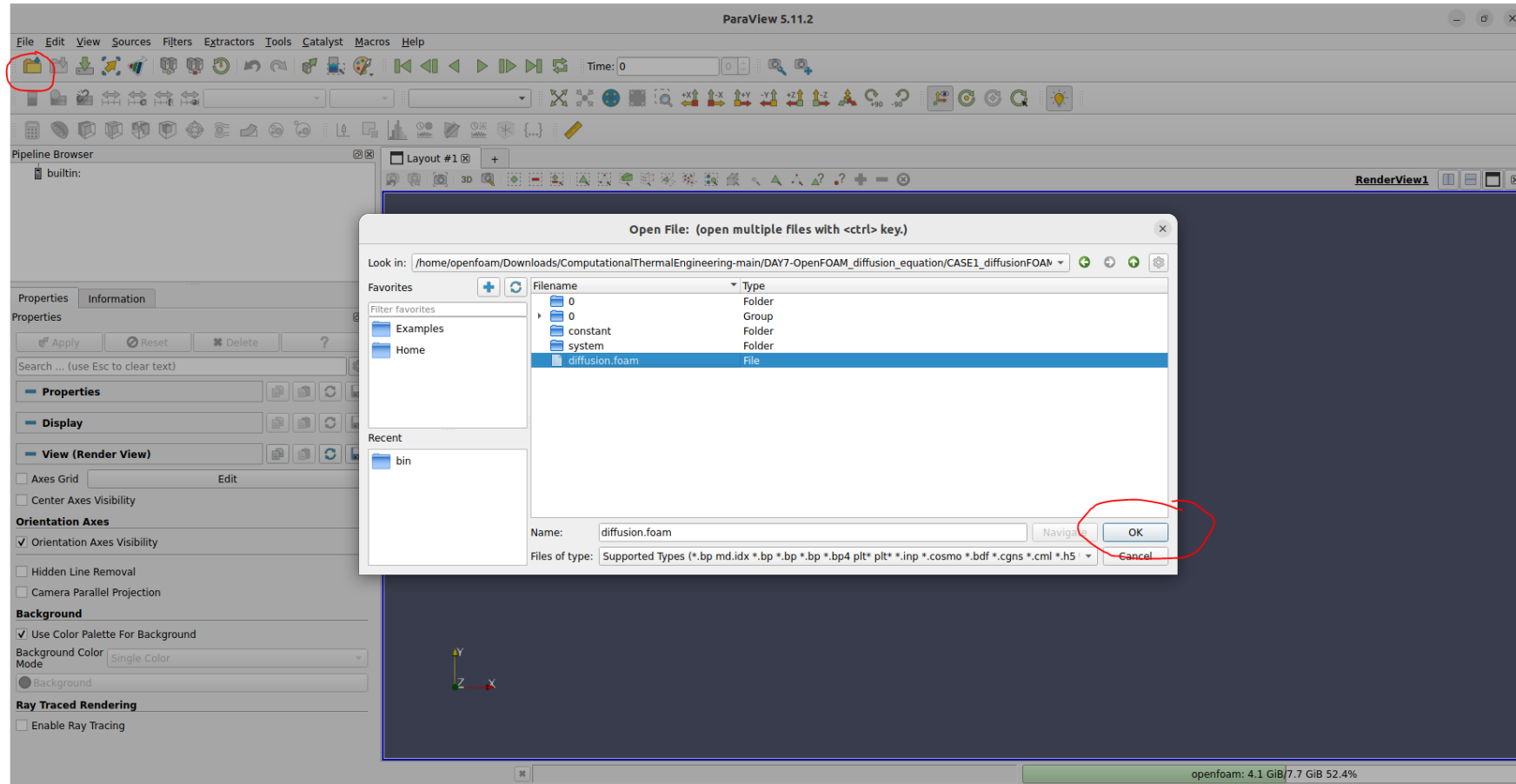
# Compile the “case” file named – CASE1\_diffusionFOAM

```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ ls  
0          0.004  0.008  0.012  0.016  0.02   0.024  0.028  0.032  0.036  0.04   0.044  0.048  0.052  0.056  0.06   0.064  0.068  0.072  0.076  0.08   0.084  0.088  0.092  0.096  0.1  
0.001  0.005  0.009  0.013  0.017  0.021  0.025  0.029  0.033  0.037  0.041  0.045  0.049  0.053  0.057  0.061  0.065  0.069  0.073  0.077  0.081  0.085  0.089  0.093  0.097  constant  
0.002  0.006  0.01   0.014  0.018  0.022  0.026  0.03   0.034  0.038  0.042  0.046  0.05   0.054  0.058  0.062  0.066  0.07   0.074  0.078  0.082  0.086  0.09   0.094  0.098  system  
0.003  0.007  0.011  0.015  0.019  0.023  0.027  0.031  0.035  0.039  0.043  0.047  0.051  0.055  0.059  0.063  0.067  0.071  0.075  0.079  0.083  0.087  0.091  0.095  0.099  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ touch diffusion.foam  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ ls  
0          0.004  0.008  0.012  0.016  0.02   0.024  0.028  0.032  0.036  0.04   0.044  0.048  0.052  0.056  0.06   0.064  0.068  0.072  0.076  0.08   0.084  0.088  0.092  0.096  0.1  
0.001  0.005  0.009  0.013  0.017  0.021  0.025  0.029  0.033  0.037  0.041  0.045  0.049  0.053  0.057  0.061  0.065  0.069  0.073  0.077  0.081  0.085  0.089  0.093  0.097  constant  
0.002  0.006  0.01   0.014  0.018  0.022  0.026  0.03   0.034  0.038  0.042  0.046  0.05   0.054  0.058  0.062  0.066  0.07   0.074  0.078  0.082  0.086  0.09   0.094  0.098  diffusion.foam  
0.003  0.007  0.011  0.015  0.019  0.023  0.027  0.031  0.035  0.039  0.043  0.047  0.051  0.055  0.059  0.063  0.067  0.071  0.075  0.079  0.083  0.087  0.091  0.095  0.099  system  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$  
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
```

Open “Paraview” → Used for **post-processing** your results



# Open the “case” in paraview



# Project – 1 (Diffusion Equation using OpenFOAM)

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$



Kumares0402 last week Maintainer

edited ...

Based on DAY 6 presentation, repeat all steps we discussed during the session

Make sure OpenFOAM is installed on your systems.  
Install ParaView.  
Copy solver and test case to the working directory.  
Build/compile the solver.  
Run the test case by using following commands:

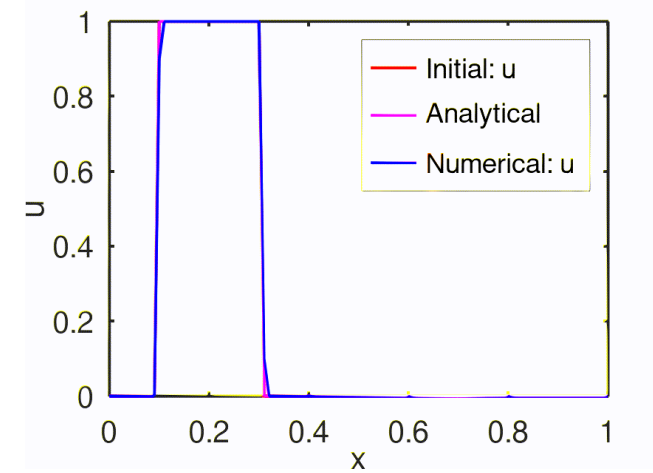
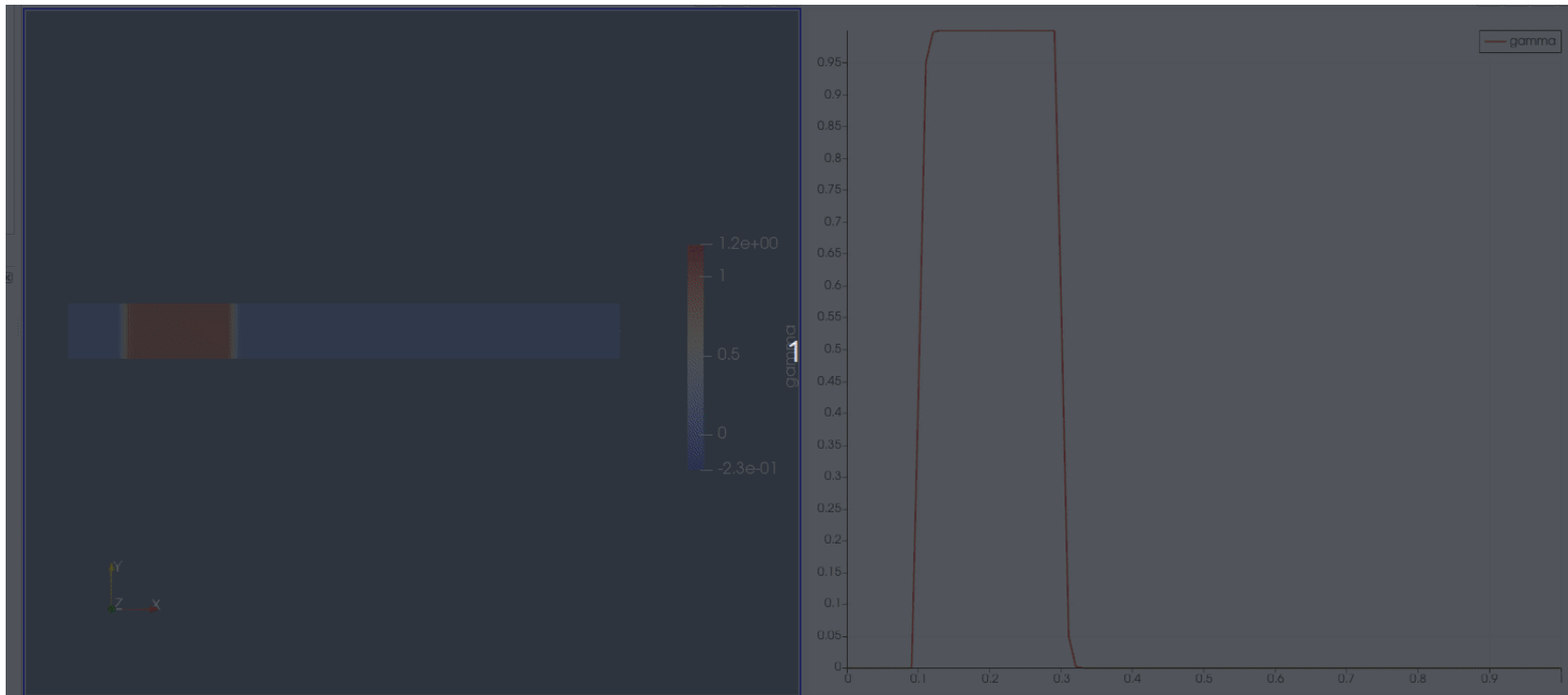
1. blockMesh
2. diffusionFoam
3. touch a.foam

Visualize the results.  
Share screenshots of results here with clear description





# OpenFOAM: Numerical Solution to Convection Equation

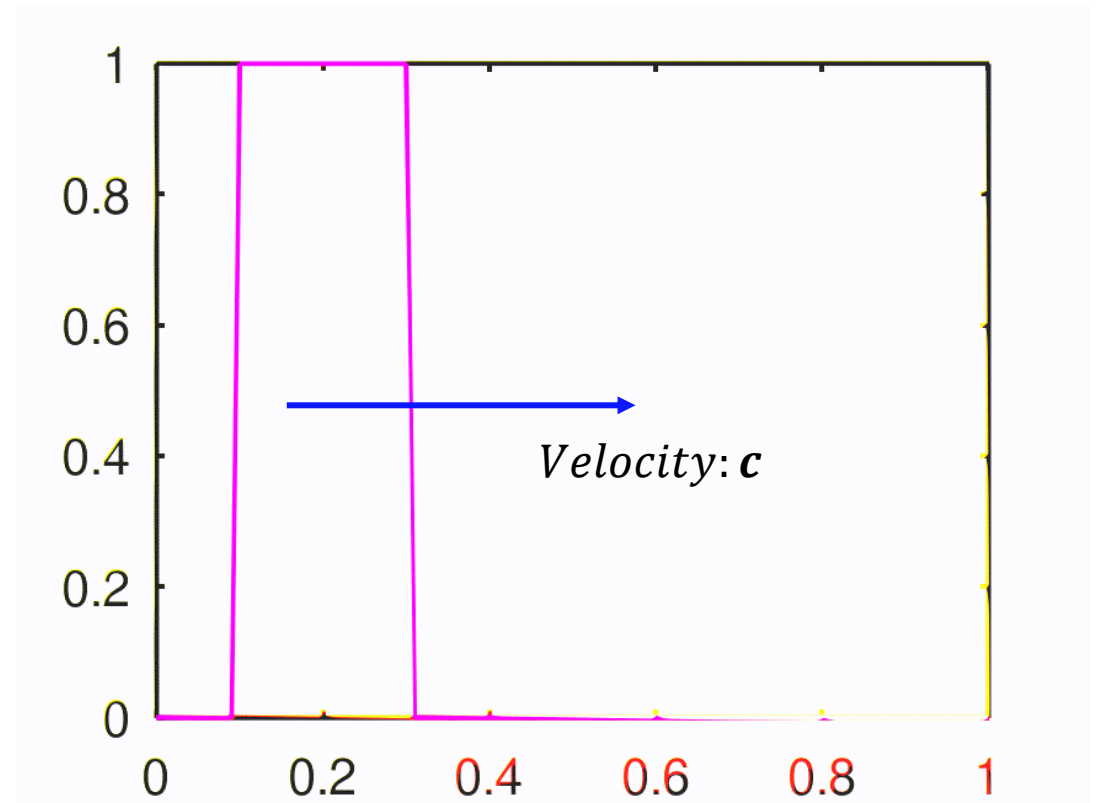
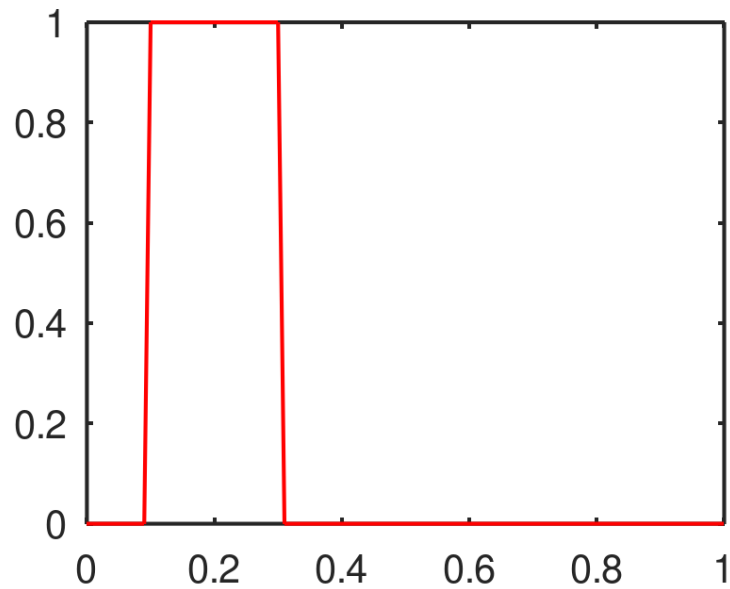


$$\frac{\partial u}{\partial t} + c \frac{\partial u}{\partial x} = 0$$

# Convection Equation

## Setting initial field

```
for i = 1 : length(x)
    if (x(i, 1) >= 0.1) && (x(i, 1) <= 0.3)
        u(i, 1) = 1;
    endif
end
```

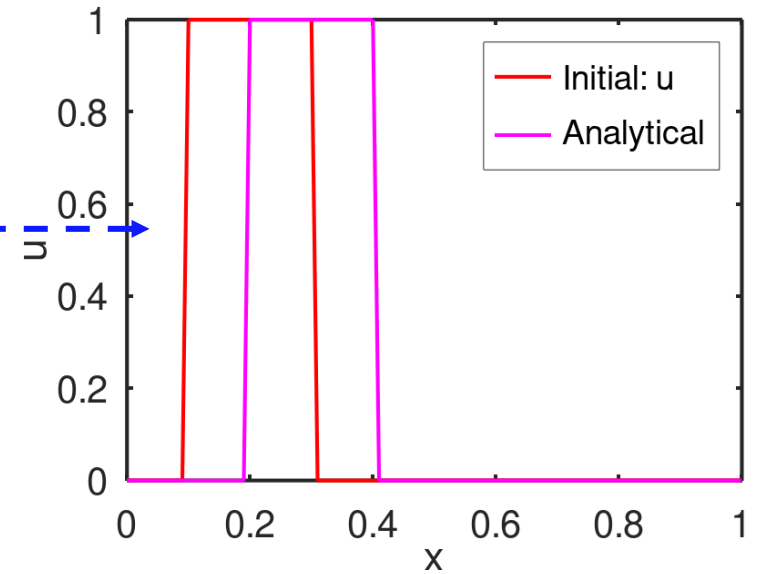
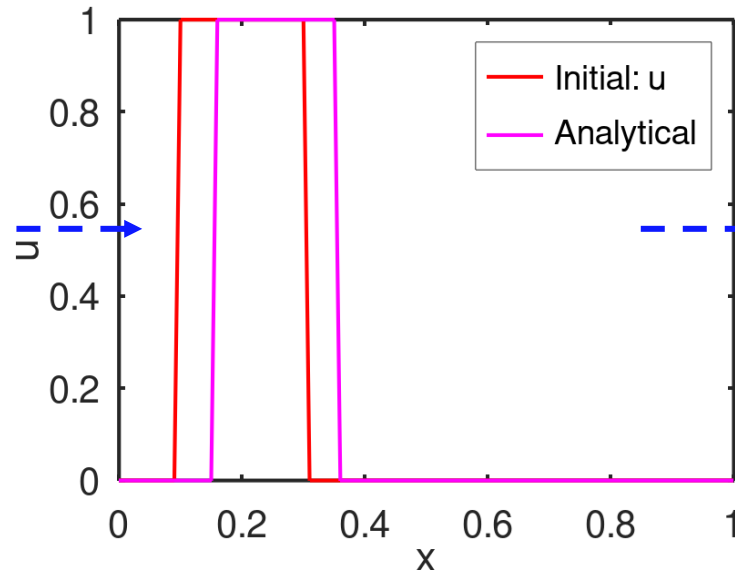
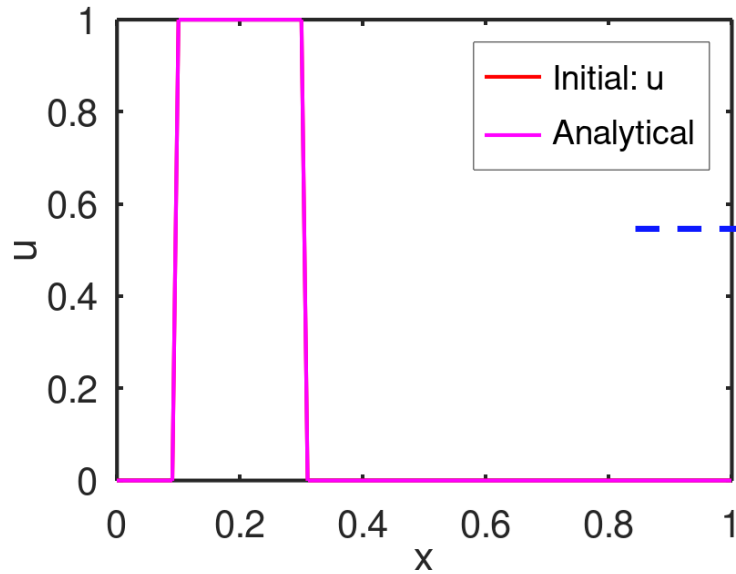




# Convection Equation

$$\frac{\partial u}{\partial t} + c \frac{\partial u}{\partial x} = 0 \quad \leftarrow \text{Advection equation}$$

```
for i = 1 : length(x)
    if (x(i, 1) >= 0.1+c*t) && (x(i, 1) <= 0.3+c*t)
        u_analytical(i, 1) = 1;
    endif
end
```

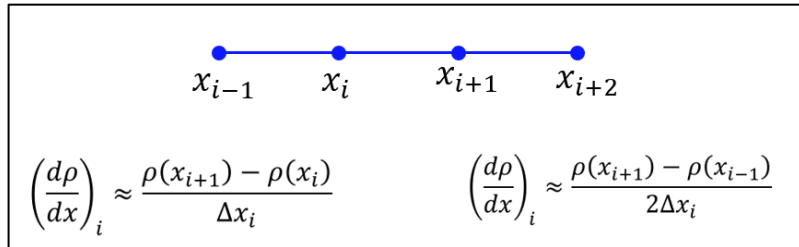


# Convection Equation

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

$$\frac{u_i^{n+1} - u_i^n}{\Delta t} + c \left( \frac{\partial u}{\partial x} \right)_i^n = 0$$





$$\left( \frac{d\rho}{dx} \right)_i \approx \frac{\rho(x_{i+1}) - \rho(x_i)}{\Delta x_i} \quad \left( \frac{d\rho}{dx} \right)_i \approx \frac{\rho(x_{i+1}) - \rho(x_{i-1}))}{2\Delta x_i}$$

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

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

Simple forward difference scheme

$\left( \frac{\partial u}{\partial x} \right)_i^n \approx \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x_i}$ 

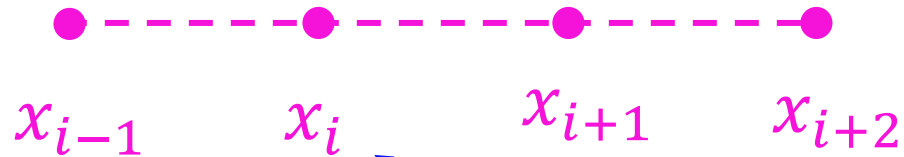
Central difference

(Explicit) First order - Forward Euler

*(only one unknown (n+1) with other knowns at n<sup>th</sup> node)  
→ conditionally stable*

# Convection Equation

Time level:  $n + 1$



Time level:  $n$



Information from right  
to left end

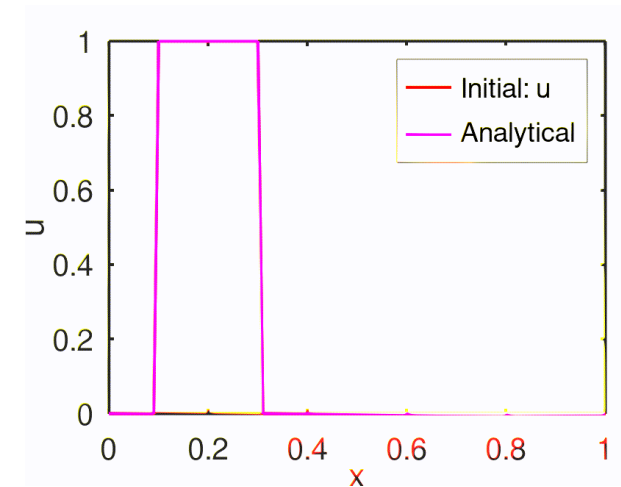
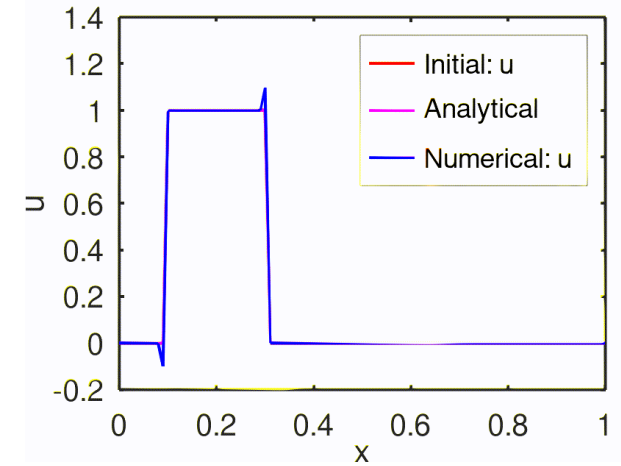
$$u_i^{n+1} = u_i^n - c\Delta t \left( \frac{\partial u}{\partial x} \right)_i^n$$

Simple forward difference scheme

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

Central difference

$$\left( \frac{\partial u}{\partial x} \right)_i^n \approx \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x_i}$$



# Convection Equation

Time level:  $n + 1$



Time level:  $n$

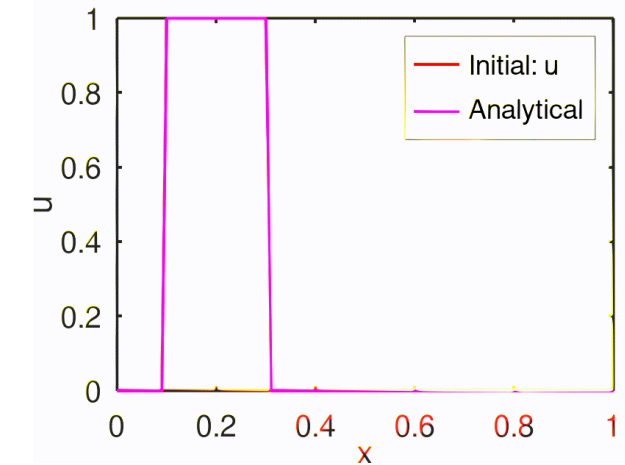
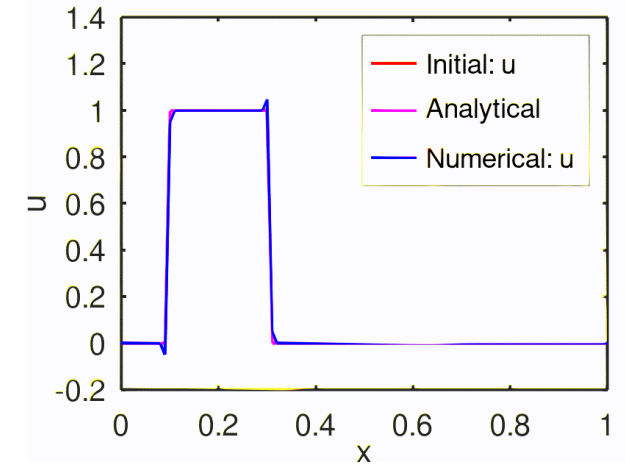


Information from left and right ends

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

$\left( \frac{\partial u}{\partial x} \right)_i^n \approx \frac{u_{i+1}^n - u_i^n}{\Delta x_i}$   
 Simple forward  
difference scheme

$\left( \frac{\partial u}{\partial x} \right)_i^n \approx \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x_i}$   
 Central difference

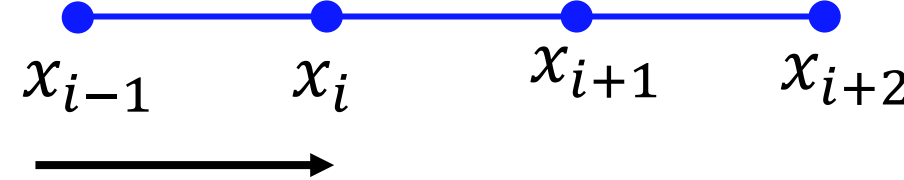


# Convection Equation

Time level:  $n + 1$



Time level:  $n$



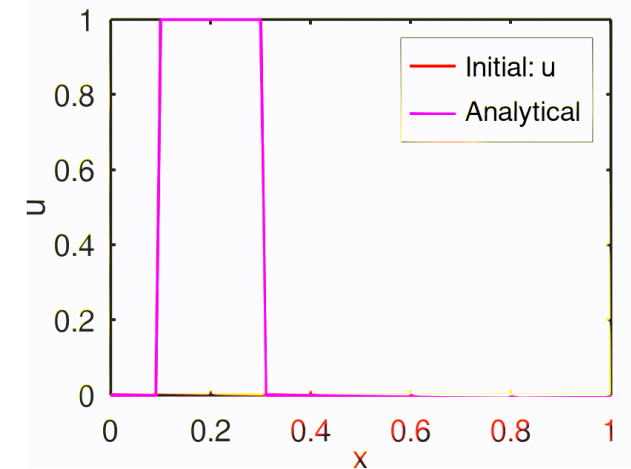
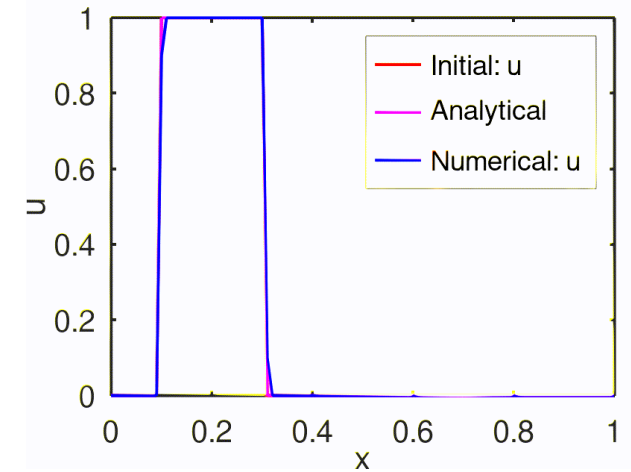
Information from left to right end  
Wind is flowing from left end (bird moves from left to right)

$$u_i^{n+1} = u_i^n - c\Delta t \left( \frac{\partial u}{\partial x} \right)_i^n \longrightarrow \left( \frac{\partial u}{\partial x} \right)_i^n \approx \frac{u_i^n - u_{i-1}^n}{\Delta x_i} \quad \text{Simple backward difference scheme}$$

**CFL = 0.1**       $CFL: \frac{c\Delta t}{\Delta x}$

**CFL < 1** → Numerically stable (conditionally stable based on the condition imposed by CFL) – EXPLICIT method

**CFL ≥ 1** → Numerically unstable



# Project – 2 (Convection Equation using OpenFOAM)

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



**Kumares0402** 8 minutes ago Maintainer

Based on DAY 7 presentation, repeat all steps we discussed during the session

Make sure OpenFOAM is installed on your systems.  
Install ParaView.  
Copy solver and test case to the working directory.  
Build/compile the solver.  
Run the test case by using following commands:

1. blockMesh
2. setFields
3. simpleConvection
4. touch a.foam

Visualize the results.  
Share screenshots of results here with clear description

