CEE 618 Scientific Parallel Computing (Lecture 11)

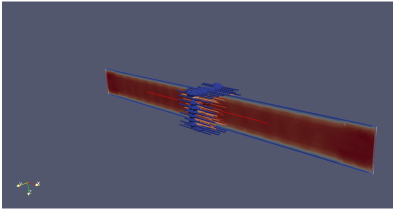
OpenFOAM – Pipe Flow & small Pool Fire 2D/3D

Albert S. Kim

Department of Civil and Environmental Engineering
University of Hawai'i at Manoa
2540 Dole Street, Holmes 383, Honolulu, Hawaii 96822

Table of Contents

Square Pipe

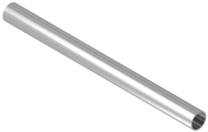


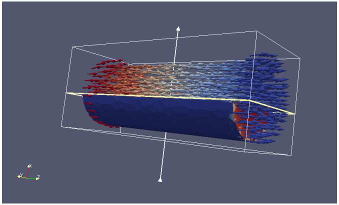
1. Analytic Solution
2. OpenFOAM CFD simulation

Square Pipe: geometry file

```
1 = 1;
Point(1) = {10, 1, 1, 1};
Point(2) = {10, -1, 1, 1};
Point(3) = {10, 1, -1, 1};
Point(4) = {10, -1, -1, 1};
Point(5) = {-10, 1, 1, 1};
Point(6) = {-10, 1, -1, 1};
Point(7) = {-10, -1, -1, 1};
Point(8) = {-10, -1, 1, 1};
Line(1) = {1, 5};
Line(2) = {5, 8};
Line(3) = {8, 2};
Line(4) = {2, 1};
Line(5) = {1, 3};
Line(6) = {5, 6};
Line(7) = {8, 7};
Line(8) = {2, 4};
Line(9) = {3, 6};
Line(10) = {6, 7};
Line(11) = {7, 4};
Line(12) = {4, 3};
```

Cylindrical Pipe



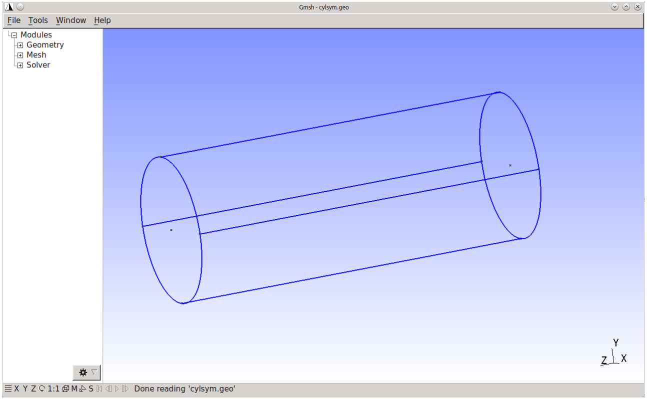


1. Analytic Solution
2. OpenFOAM CFD simulation

Cylindrical Pipe: geometry file

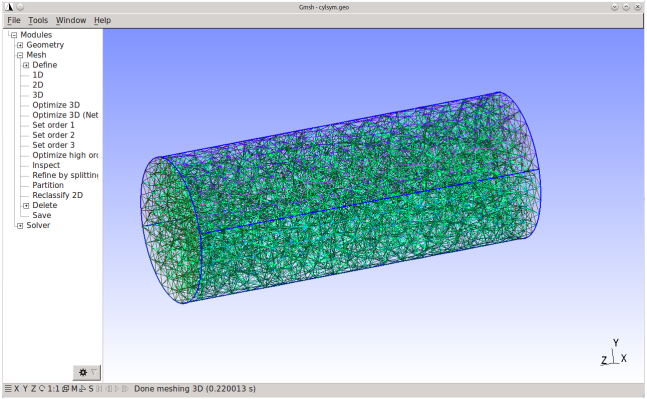
```
R = 2.0;
H = 10 ;

Point(1) = { 0, 0, 0 ,0.5};
Point(2) = { R, 0, 0 ,0.5};
Point(3) = { 0, R, 0 ,0.5};
Point(4) = {-R, 0, 0 ,0.5};
Point(5) = { 0,-R, 0 ,0.5};
Circle(1) = {2,1,3};
Circle(2) = {3,1,4};
Circle(3) = {4,1,5};
Circle(4) = {5,1,2};
Line Loop(5) = {1,2,3,4};
Plane Surface(6) = {5};
Extrude {0,0,H} {
  Surface{6};
}
Physical Surface("left") = {6};
Physical Surface("right") = {28};
Physical Surface("wall1") = {19};
Physical Surface("wall2") = {23};
```

Cylinder geometry



1. There are 6 surfaces.
2. Those are `wall1` – `wall4`, `left`, and `right`.

Cyinder mesh



Small Pool Fire 3D

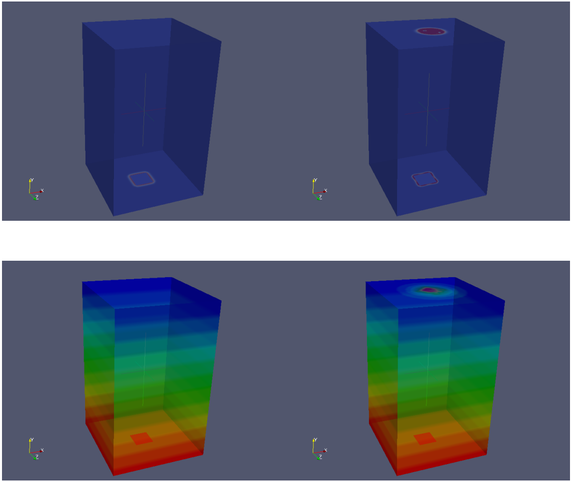


Figure: Temperature (first row) and Pressure (second row) Profiles: Initial (left) and Later (right)

Small Pool Fire 2D

Figure: Temperature (left) and pressure (right) profiles.

Note: Lab work

1. After you login fractal, make a directory 'smallPoolFire2D' under your OpenFOAM-Case directory.
2. Copy files from /opt/cee618s13/class11/fireFoam2D
   `cp␣/opt/cee618s13/class11/fireFoam2D/* ./`
3. To setup: `make`
4. Open system/controlDict, and change 'endTime' value from 3.0 to 1.0.
5. (For your homework, change to 5.0.)
6. To run: `make␣run`
7. This simulation will take approximately an hour.

Note: After you down load a zip file, change "0" directory to anything else such as "t0". This is because some configuration errors occur in Paraview: `mv␣0␣t0`

smallPoolFire2D: Makefile

```
OpenFOAMfireFOAM=~/OpenFOAM/OpenFOAM-2.1.1/tutorials/combustion/fir

all: copy
	touch smallPoolFire2D.foam

copy:
	cp -r $(OpenFOAMfireFOAM)/* ./

reset: clean
	rm -rf 0* 1* 2* 3* constant system processor* Allrun

clean:
	rm -f PBS-* *.foam *.msh *.geo

run:
	qsub fireFoam2Dsrl.pbs
```

fireFoam2Dsrl.pbs: PBS file for serial computation

```
#!/bin/bash
#PBS -l walltime=48:00:00
#PBS -q batch
#PBS -N PBS-fireFoam
#PBS -V
cd $PBS_O_WORKDIR
hostname
blockMesh
topoSet
createPatch -overwrite
date
time fireFoam
date
```

smallPoolFire2D: `Allrun` file

- blockMesh reads constant/polyMesh/blockMeshDict, generates the mesh and writes out the mesh data to points and faces, cells and boundary files in the same directory.
- topoSet and creatPatch are pre-processing actions.
- firFoam is a solver for this combustion problem using Large Eddy Simulation (LES) model developed for turbulence used in computational fluid dynamics.
- LES uses a filter (called LES filter). This filter is applied to a function $f(\boldsymbol{r}, t)$ and splits f into a filtered and sub-filtered parts. Governing equations are used separately for the two parts of the solution. Computation is faster than directly solving NS equation, i.e, direct numerical simulation (DNS).
- We will use this fireFoam2D case primarily for Paraview visualization. See HW11.