OpenFOAM Analysis of CANDU-6 Moderator Circulation

S. M. Chang*¶, H. T. Kim**, K. L. Jeong***
* Kunsan National University
** KAERI, ***NEXT foam

Contents

Objective of Research

3-D Analysis with CFD(Computational Fluid Dynamics code and consideration of its feasibility in the moderator system of heavy-water CANDU reactors: Calandria tank

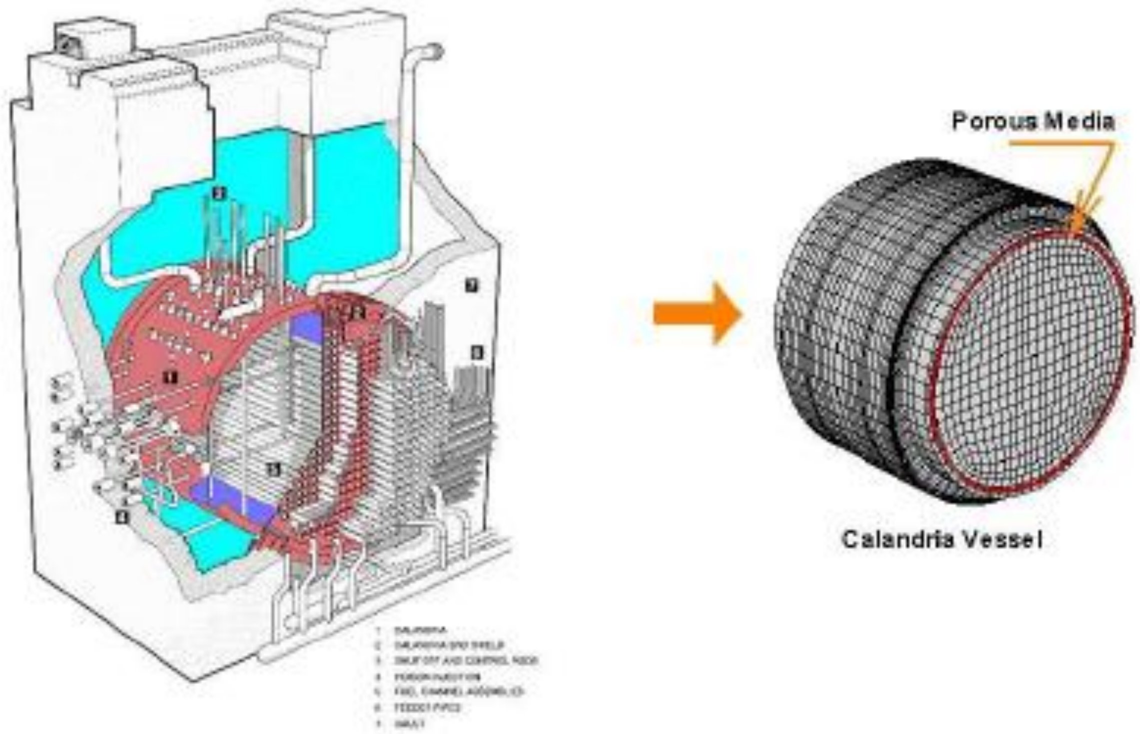
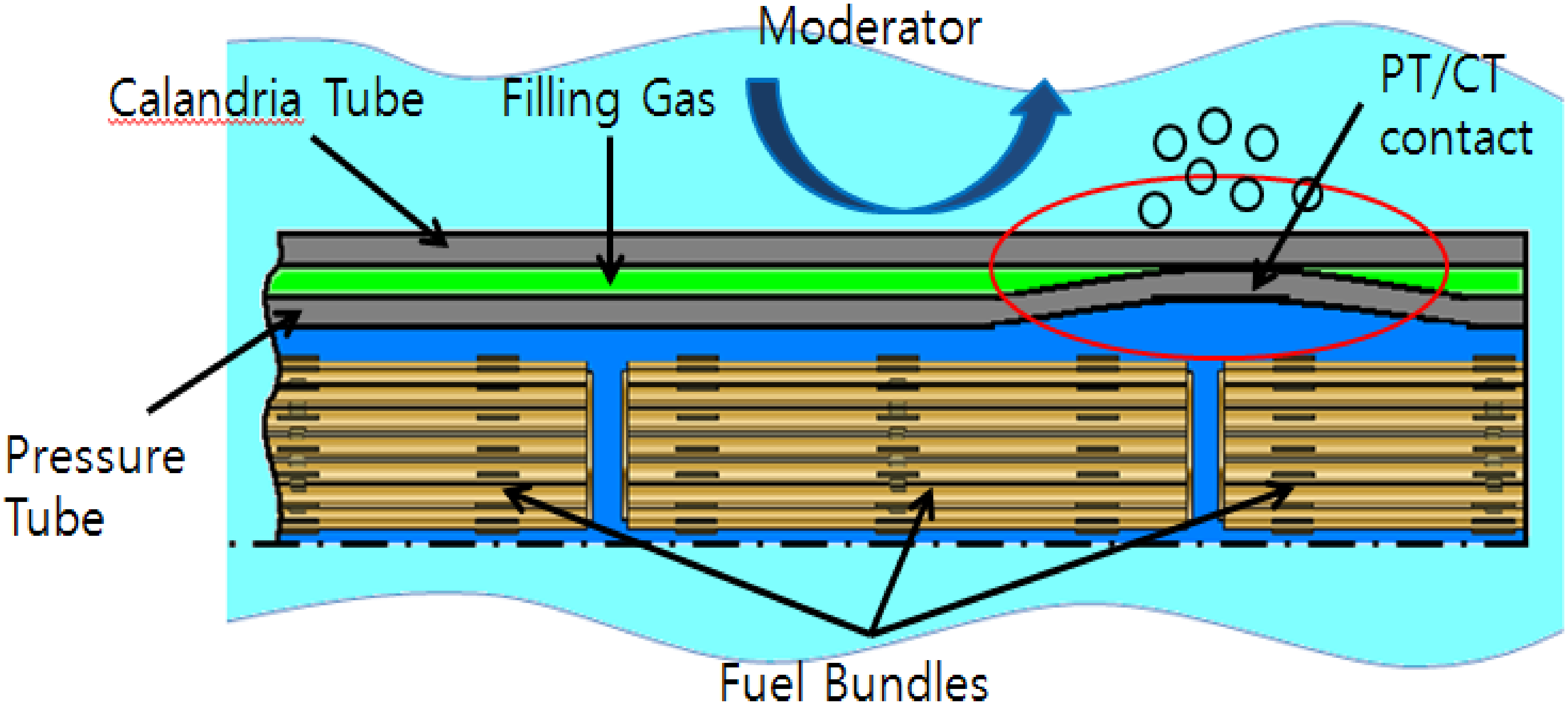


그림 1. CANDU-6형 원자로와 나비형(Butterfly-Shaped) 격자구조

Introduction: Calandria Tubes



LOCA: Loss Of Coolant Accident
CHF: Critical Heat Flux
Codes: MODTRUC/MODTRUC_CLAS, CATHENA, CUPID

Key Physics

Primary Physical Phenomena

1) Incompressible flow solver + turbulence model
2) Heat transfer: energy equation
3) Buoyancy term: source terms in N-S equation
4) Porous media: modeling of porosity, permeability
5) Heat source model: energy equation

Boundary conditions

No-slip (wall), adiabatic, inlet/outlet, ...

Governing Equations

$$\nabla \cdot \mathbf{V} = 0$$

$$\rho \left\{ \frac{\partial \mathbf{V}}{\partial t} + \left(\mathbf{V} \cdot \nabla \right) \mathbf{V} \right\} = -\nabla p + \rho \mathbf{g} + \mu \nabla^2 \mathbf{V} + \rho_0 \mathbf{g} \beta \left(T_0 - T \right)$$

$$\rho C_p \left\{ \frac{\partial T}{\partial t} + \left(\mathbf{V} \cdot \nabla \right) T \right\} = \sigma \nabla^2 T + Q_s$$

$$\rho \left\{ \frac{\partial k}{\partial t} + \left(\mathbf{V} \cdot \nabla \right) k \right\} = \frac{\partial}{\partial x_j} \left\{ \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right\} + P_k + P_b - \rho \varepsilon - Y_M + S_k$$

$$\rho \left\{ \frac{\partial \varepsilon}{\partial t} + \left(\mathbf{V} \cdot \nabla \right) \varepsilon \right\} = \frac{\partial}{\partial x_j} \left\{ \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right\} + C_{1\varepsilon} \frac{\varepsilon}{k} \left(P_k + C_{3\varepsilon} P_b \right) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad C_{1\varepsilon} = 1.44,\ C_{2\varepsilon} = 1.92,\ C_\mu = 0.09,\ \sigma_k = 1.0,\ \sigma_\varepsilon = 1.3$$

Solvers

Table 1. OpenFOAM solvers used in the present computation.

Types	Name of solver	Related physics
Incompressible (isothermal)	(porousSimpleFoam) simpleFoam/pimpleFoam	(Prorous media) Steady/Unsteady-state Incompressible Turbulent (model)
Heat Transfer	buoyantBoussinesqSimpleFoam/ buoyantBoussinesqPimpleFoam	Steady/Unsteady-state Buoyant: free convection Incompressible Turbulent (model)

OpenFOAM: Folder Structure

- **0**
  - **U, p, k, epsilon, R, nuTilda, nut** … I/C, B/C
- **constant**
  - **polyMesh** … Mesh info.
  - **RASProperties**
    transportProperties … Turb. model, Mat. property
- **system**
  - **controlDict**
    decomposeParDict
    fvSchemes
    fvSolution … Solver ctrl, Num. coeff., Scheme, Tolerance, etc.
- ***.msh**

blockMeshDict

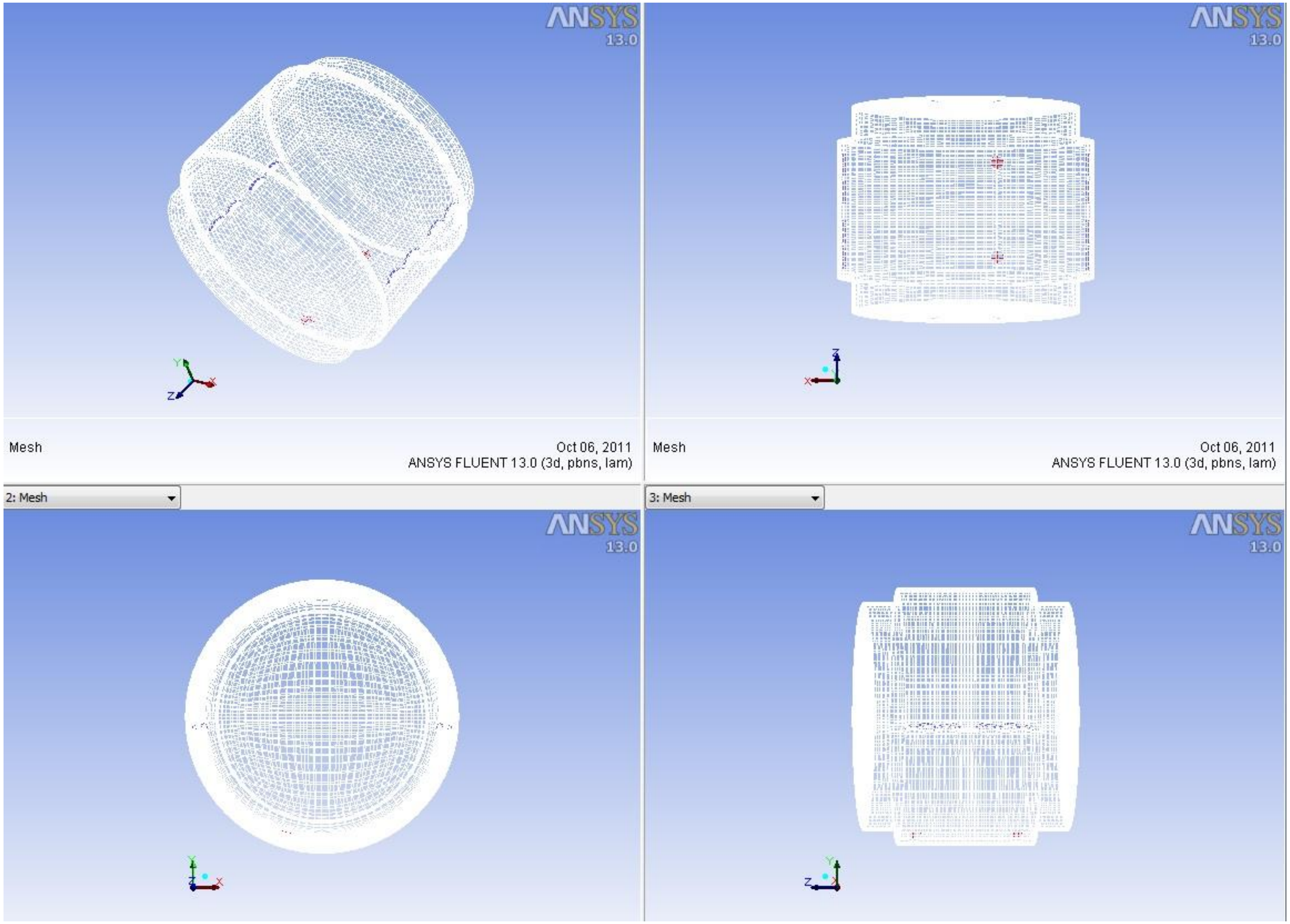
```
#include <stdio.h>
#include <math.h>

#define R                          (.03302/2.)
#define         L                  (.0714)

#define         Imax               28
#define         Jmax               4
#define IMAX                       (Imax*3)
#define JMAX                       (Jmax*3)

#define         Per                ((IMAX+1)*(JMAX+1))
#define         Nx                 10
#define         Ny                 10

#define         Width              .2              // Width of Channel

#define         PI                 3.14159265      // Circumference ratio

main()
{
                int                                i, j, k, is, js, nc, Nc;
                double             x, y, z, Xc, Yc;
                FILE               *out;

                out                = fopen("blockMeshDict.dat", "wt");

                // Header
                fprintf(out, "FoamFile ₩n{ ₩n       version 2.0; ₩n       format ascii; ₩n");
                fprintf(out, "       class dictionary; ₩n       object blockMeshDict; ₩n} ₩n₩n");

                fprintf(out, "convertToMeters 1;₩n₩n");

                // Vertices
                fprintf(out, "vertices ₩n( ₩n");
                for(k=0; k<=1; k++)
                {
                                   if(k==0)        z = 0.;
                                   if(k==1)        z = Width;

                                   y = 0.;
                                   for(j=0; j<=JMAX; j++)
                                   {
                                   ........................................................................................
```

```
FoamFile
{
        version         2.0;
        format          ascii;
        class           dictionary;
        object          blockMeshDict;
}

convertToMeters 1;

vertices
(
        (0.000000 0.000000 0.000000)
        (0.024026 0.000000 0.000000)
        (0.047374 0.000000 0.000000)
        (0.071400 0.000000 0.000000)
        (0.095426 0.000000 0.000000)
        (0.118774 0.000000 0.000000)
        (0.142800 0.000000 0.000000)
        (0.166826 0.000000 0.000000)
        (0.190174 0.000000 0.000000)
```

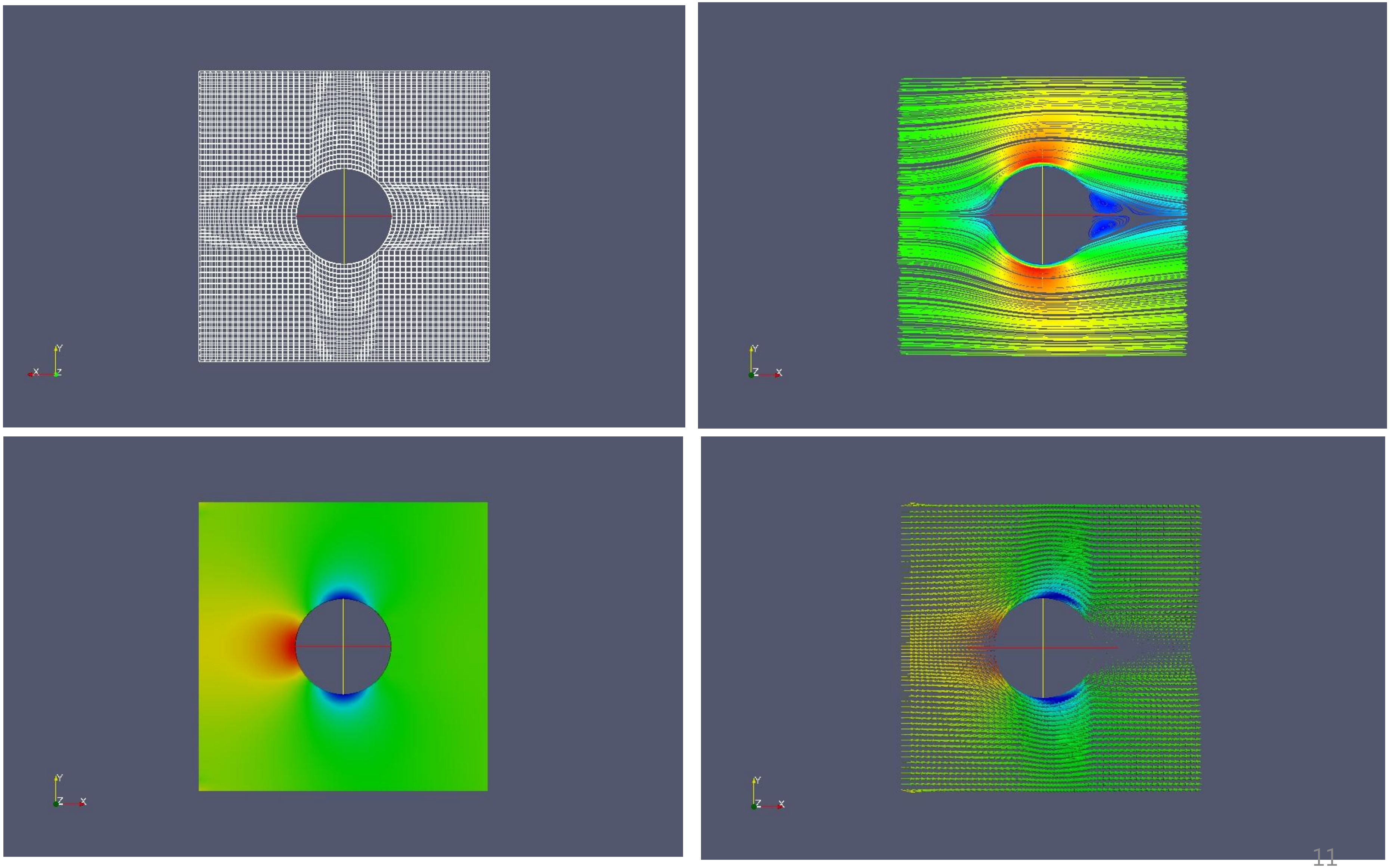
Alternative: Transform of Grids

Transformation of FLUENT Grids:

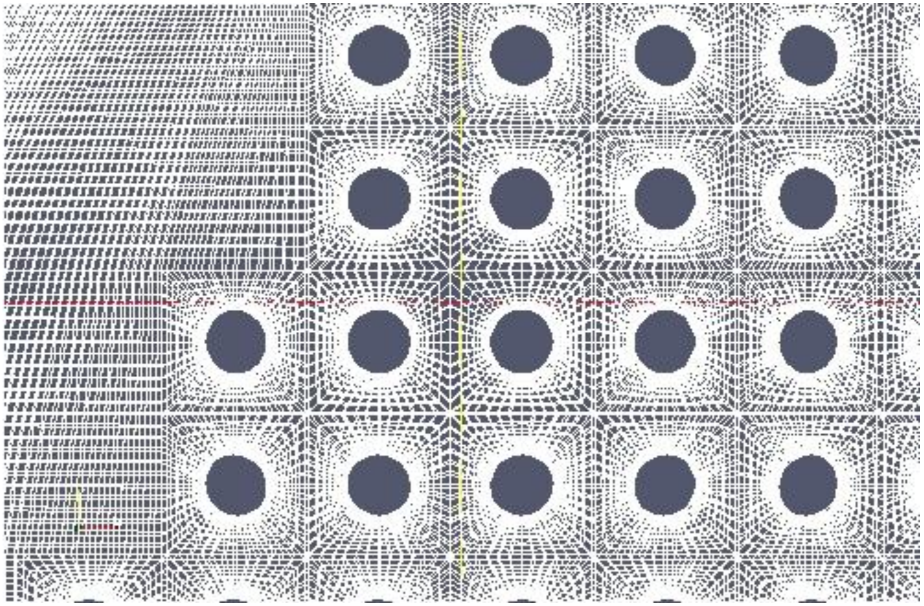
CFX (*.gtm) -> *.msh file -> fluentMeshToFoam

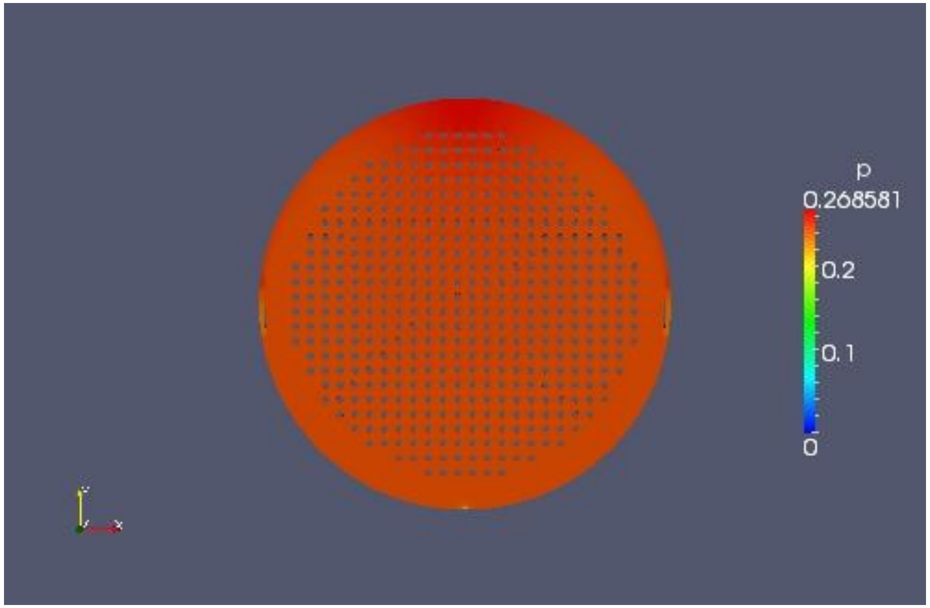


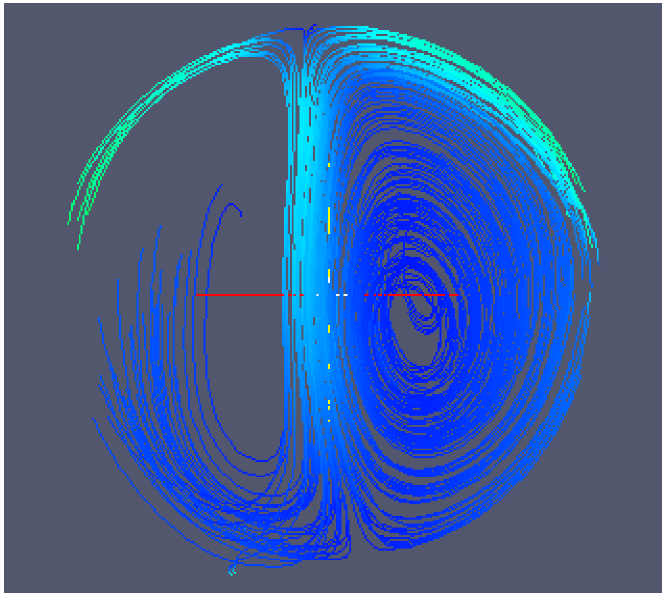
Example 1: Single Cylinder

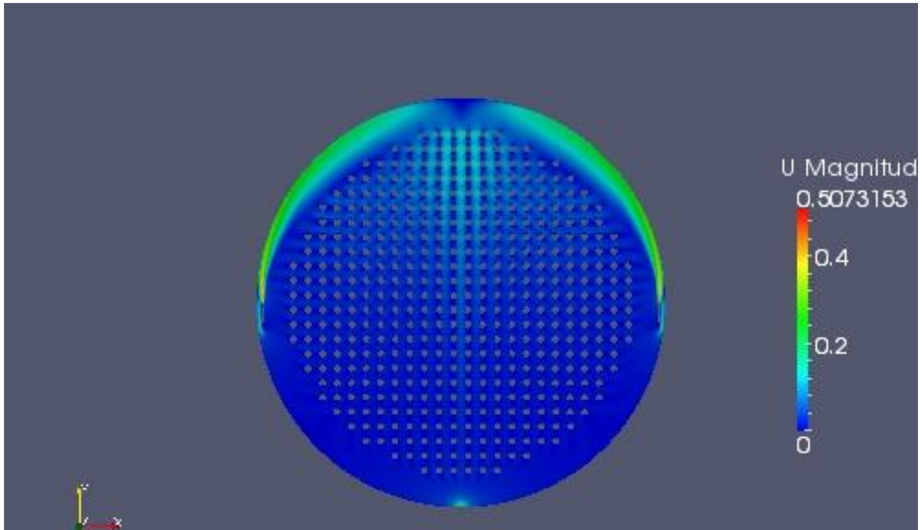


Example 2: 2-D Computation

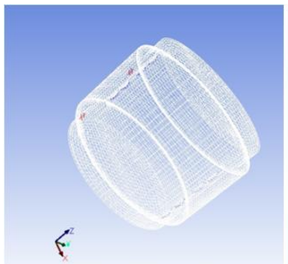


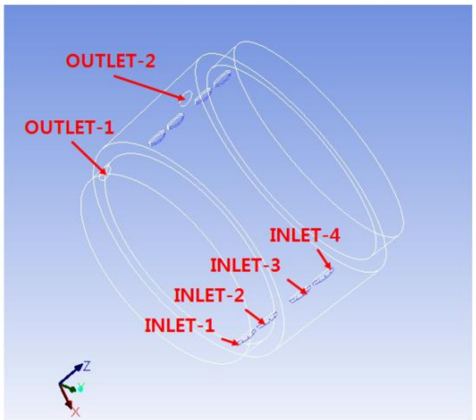


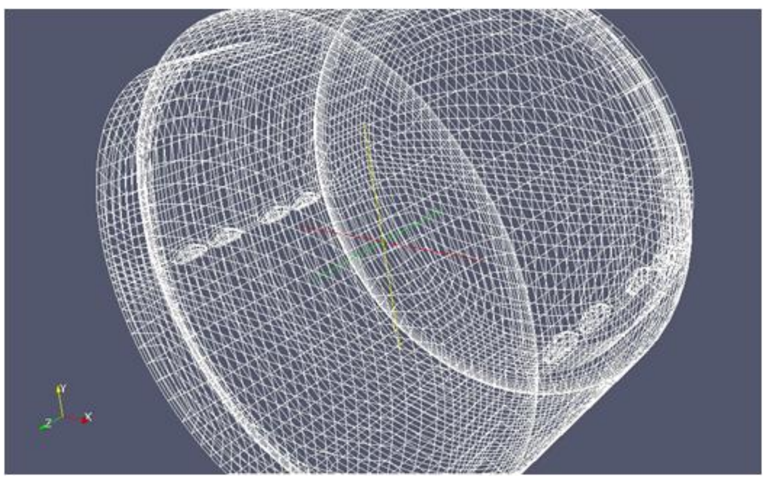




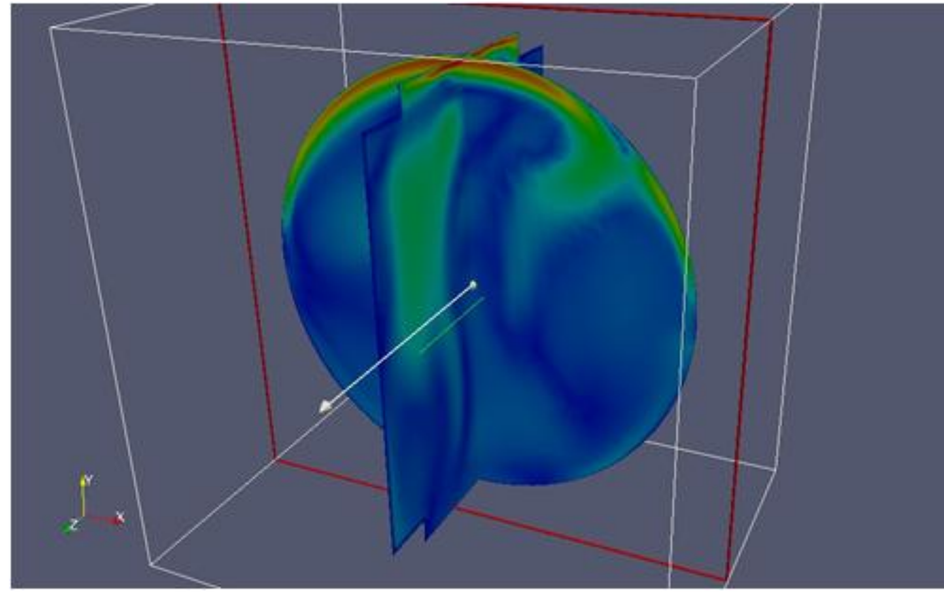
Example 3: 3-D Computation

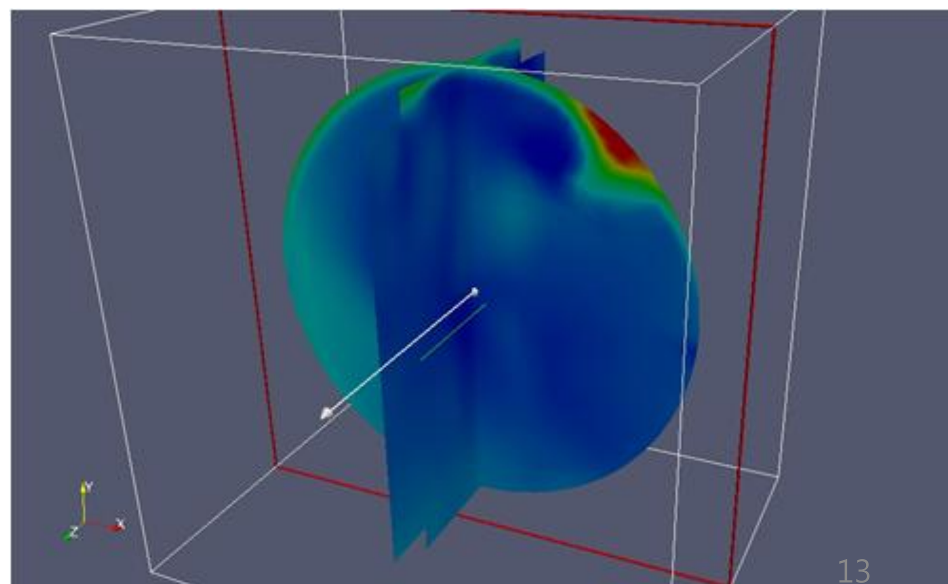






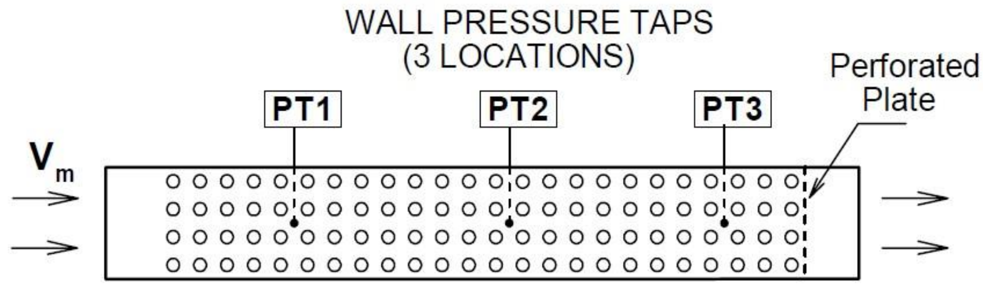
Velocity





Pressure

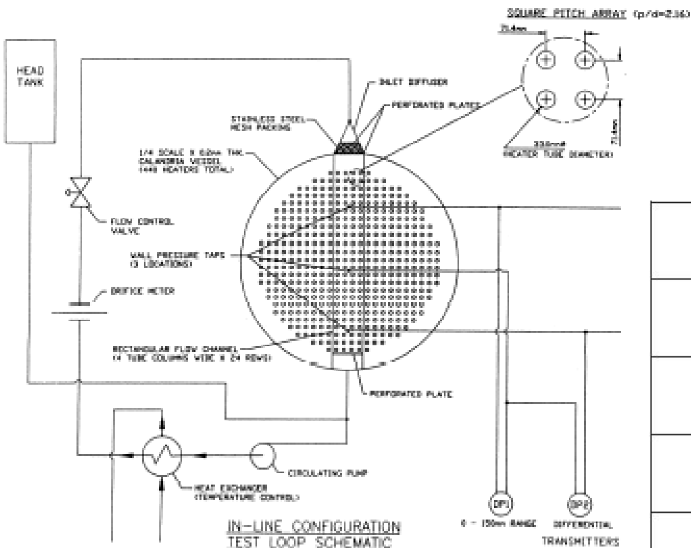
Benchmark: STERN Lab. Exp.



(a) In-Line Configuration

Table I Comparison of Measured and Predicted Pressure Drops

		STERN T.P. 326	STERN T.P. 299	STERN T.P. 306
Mass Flow [kg/s]		3.089	3.904	5.734
Temp. [^{o}C]		39.5	63.6	79.8
V_m [m/s]		0.054	0.070	0.103
Reynolds Number		2746	5237	9392
ΔP [Pa]	Measurement, **A**	28.2	41.3	78.7
	CFX-4, **B**	27.6	41.2	79.3
	MODTURC	30.5	44.9	87.3
	| **A** – **B** | / **A**	2.13 %	0.242 %	0.762 %



(a) 압력강하 측정을 위한 실험장비 (Hadaller et al. 1996

(b) 검증을 위한 계산 영역

Material Properties

Case	Vm [m/s]	Density [kg/m^3]	Viscosity [kg/(m s)]	Re_d
1	0.054	992.25	0.000653	2,709
2	0.070	981.00	0.000440	5,153
3	0.103	971.60	0.000355	9,308

Dia. of tube: 0.03302 m
X = 2 m
Y = 0.02856 m
Z = 0.2 m

Boundary Condition / Grids

Inlet Boundary Condition: fix volume flow rate

Turbulence Intensity: assume 5%

$$k = \frac{1}{2}\left\|\mathbf{V}'\right\|^2$$

$$\varepsilon = \frac{C_\mu^{0.75} k^{1.5}}{l}$$

Outlet Boundary Condition: const pressure

Wall Boundary Condition: no slip

Treatment of Porosity: Theory

N-S (Momentum) Equation:

$$\frac{\partial}{\partial t}(\gamma \rho u_i) + u_j \frac{\partial}{\partial x_j}(\rho u_i) = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{i,j}}{\partial x_j} + S_i$$

Porosity

Darcy-Forchheimer Equation:

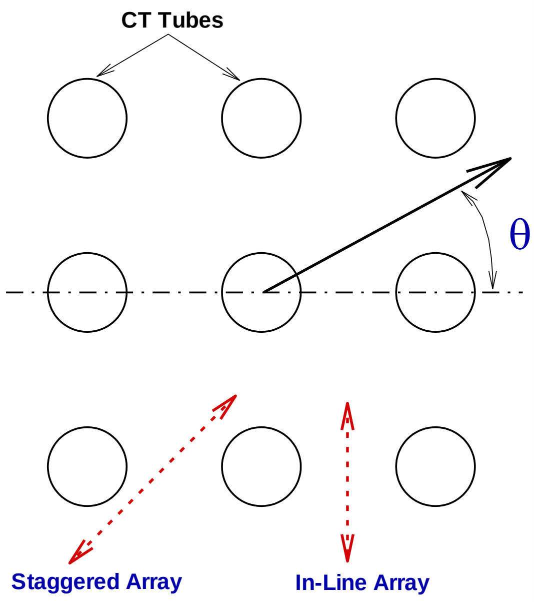
$$S_i = -\left(\mu D_{i,j} + \frac{1}{2} \rho \left| u_{kk} \right| F_{i,j} \right) u_i$$

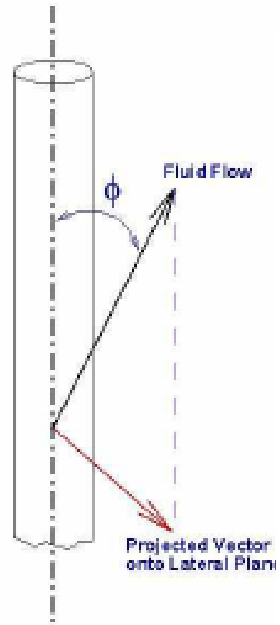
$$\frac{\Delta p}{\Delta L} = \frac{\mu}{K} U_\infty + c \frac{1}{2} \rho U_\infty^2$$

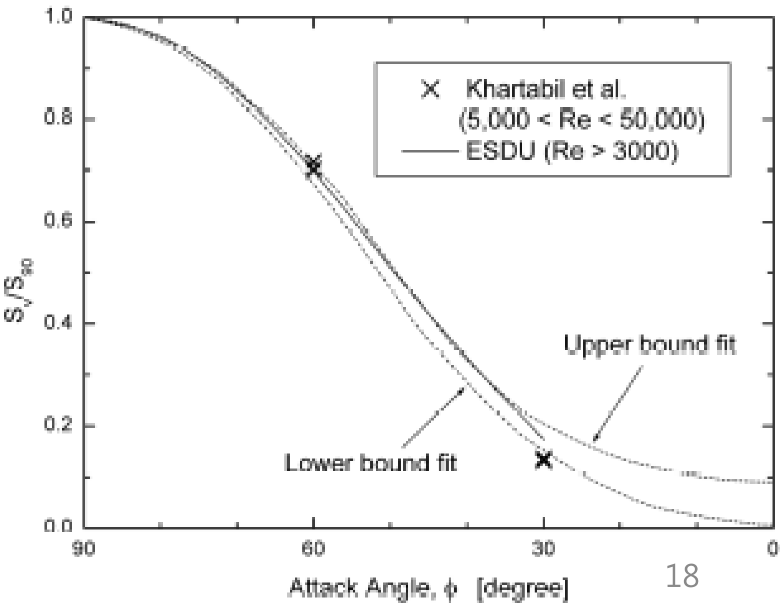
Correlation: Experiment

$$\frac{\Delta p}{\Delta L} = \frac{PLC}{p\cos\theta}\frac{1}{2}\rho\left(\gamma V\right)^2$$

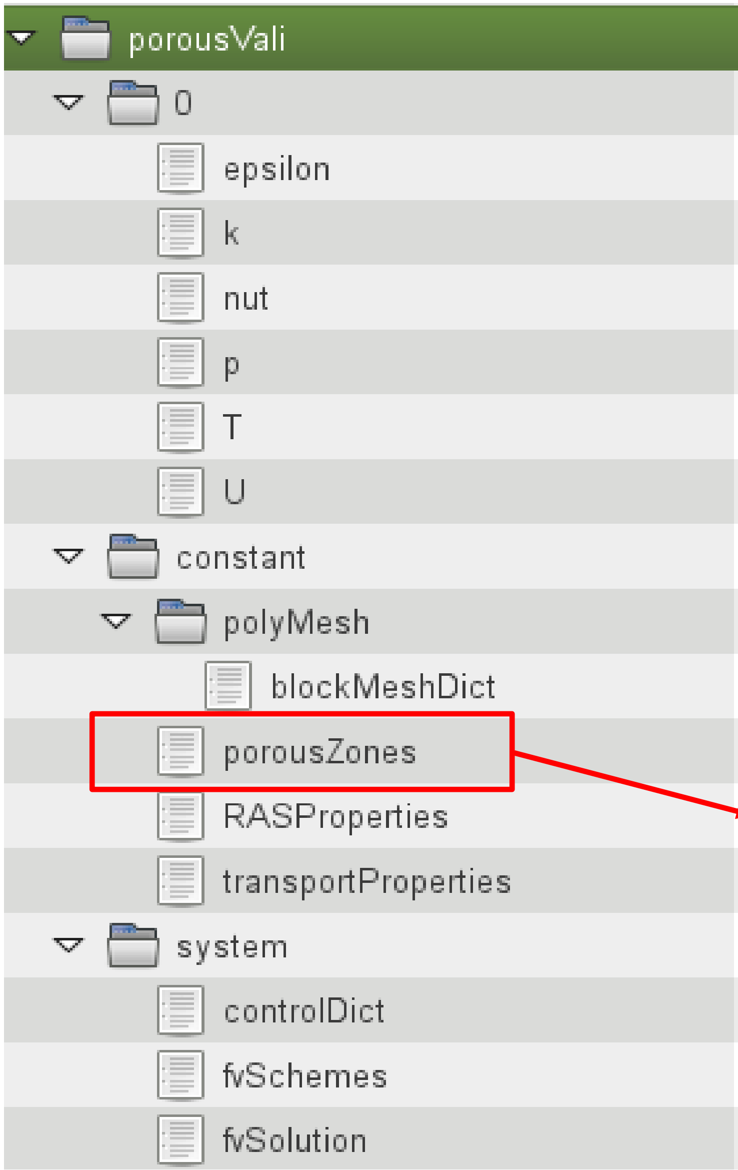
$$\mathrm{Re} = \frac{\rho\left(\gamma V\right)d}{\mu} = \mathrm{Re}_d \qquad PLC = 4.54\,\mathrm{Re}^{-0.172}$$







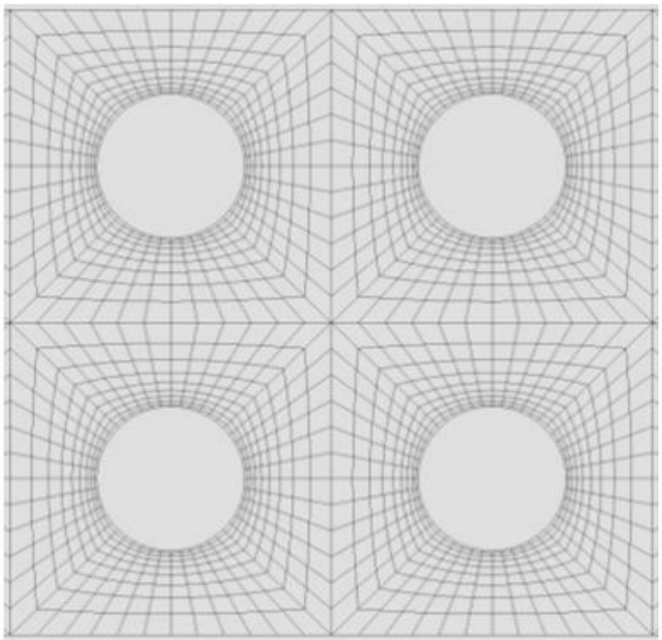
Implementation: OpenFOAM



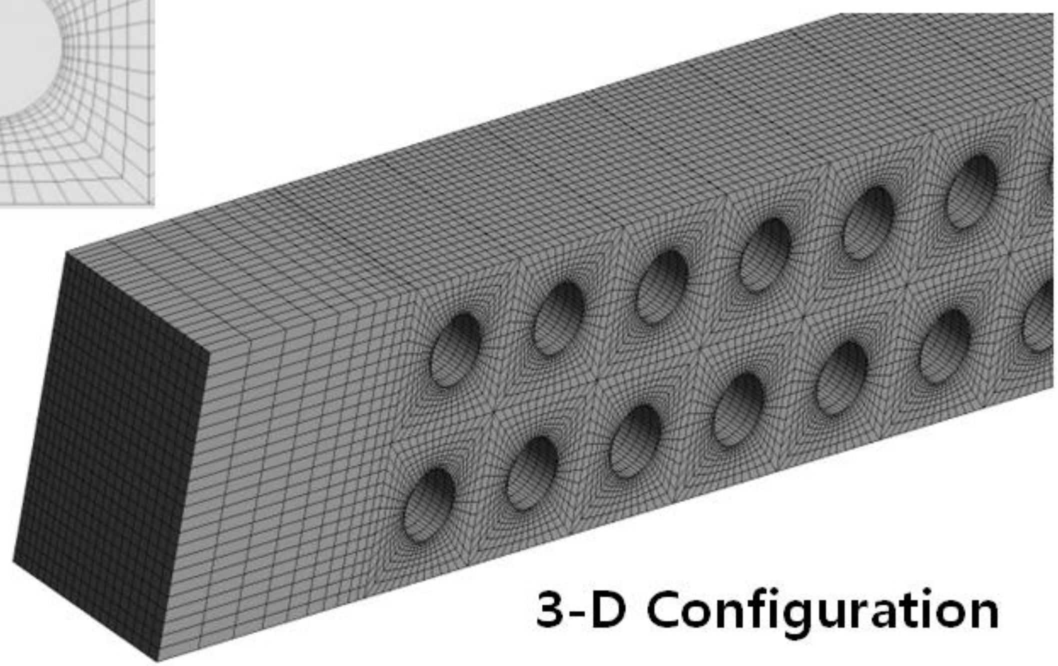
$$\frac{\Delta p}{\Delta L} = \frac{\mu}{K} U_\infty + c \frac{1}{2} \rho U_\infty^2$$

```
1
(
        porous
        {
                coordinateSystem
                {
                        e1 ( 1 0 0);
                        e2 ( 0 1 0);
                }
                Darcy
                {
                        d d[0 -2 0 0 0 0 0] (2.5e10 2.5e10 0)
                        f f[0 -1 0 0 0 0 0] (700 700 0)
                }
        }
)
```

3-D Grids System

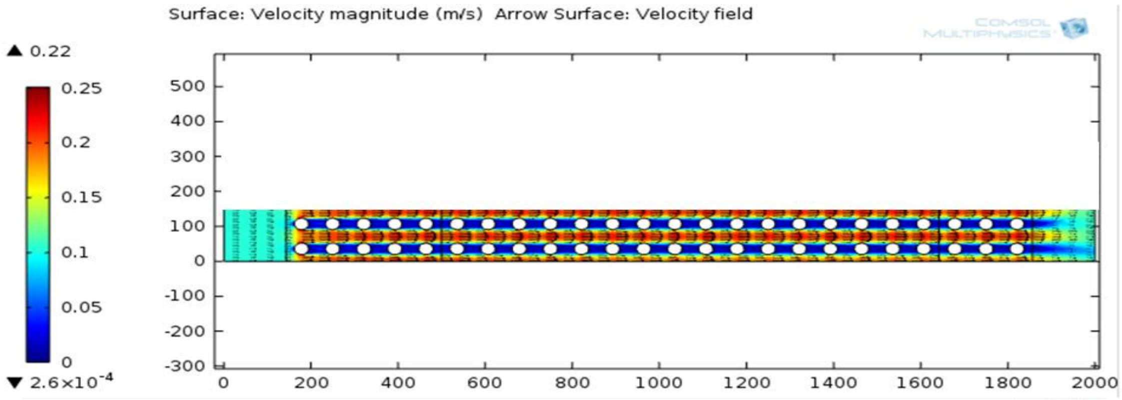


Tube Channel

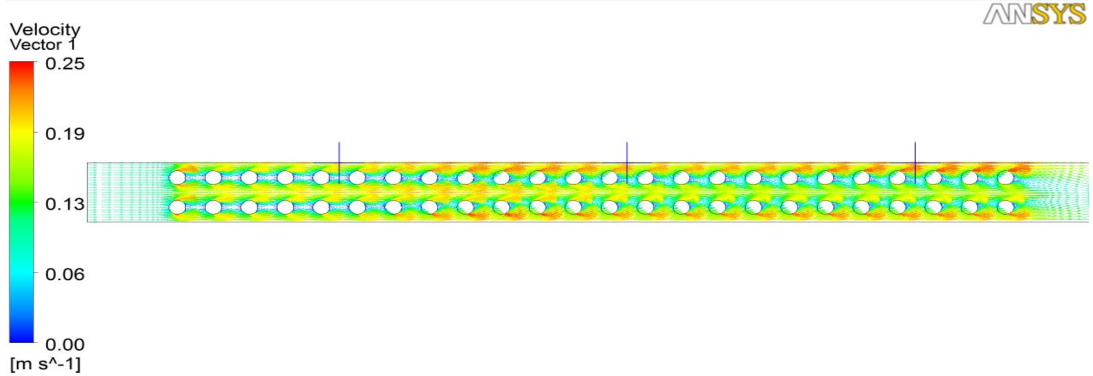


3-D Configuration

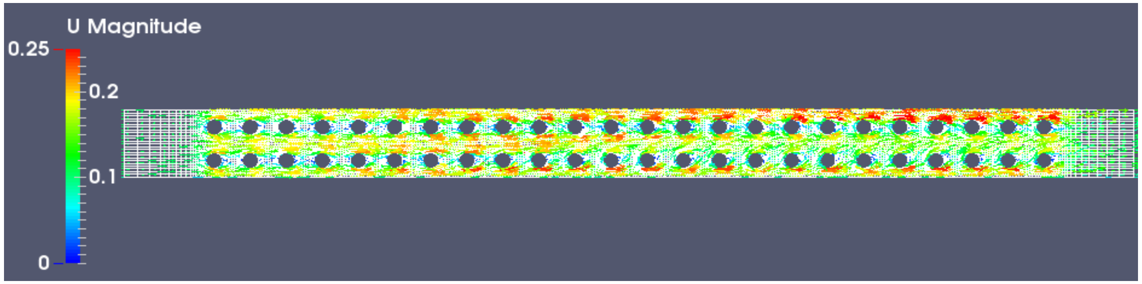
Comparison of Result



COMSOL Multiphysics

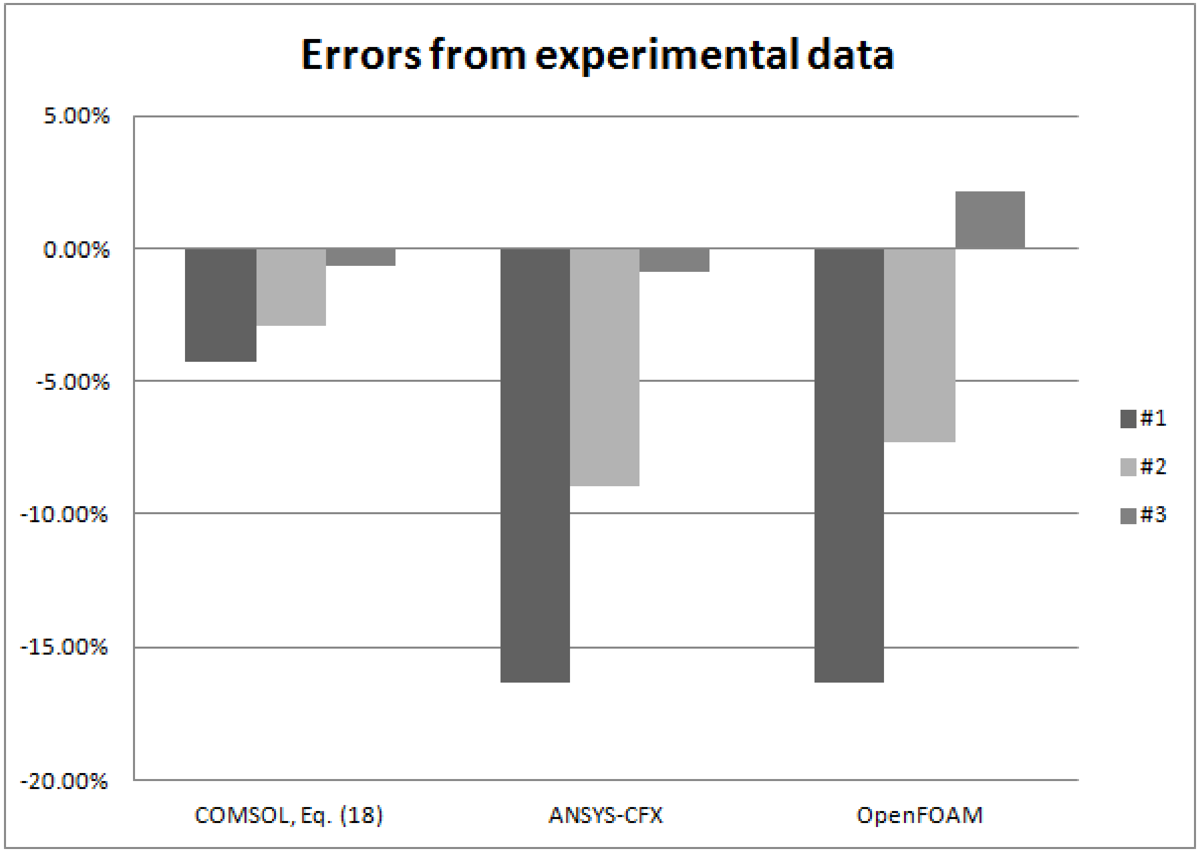


ANSYS-CFX



OpenFOAM

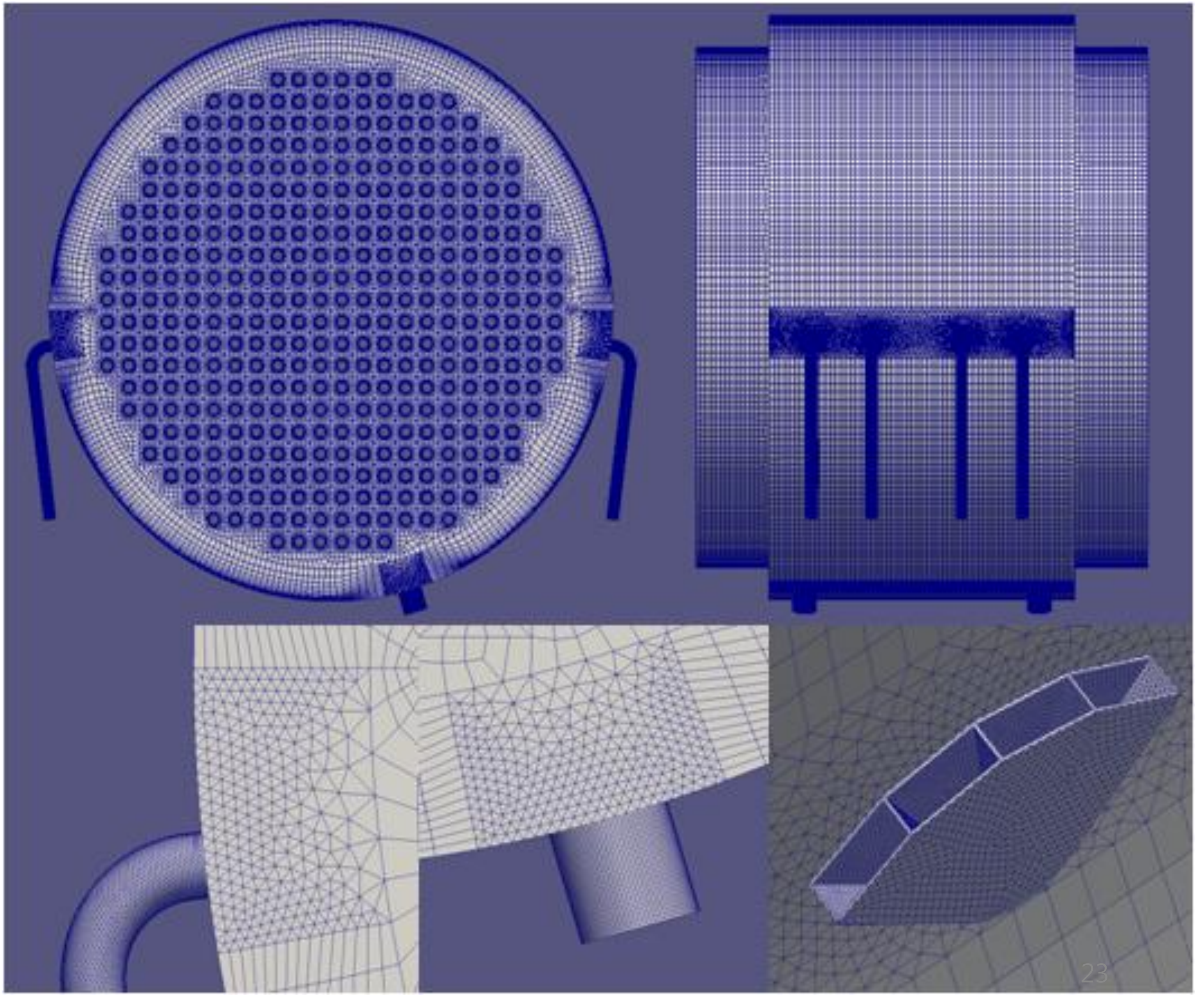
Comparison of Results



Cause of error:

1) **Turbulence Intensity:**
   - **Intake Turbulence**
   - **Surface Roughness**
2) **3-D Effect:**
   - **Axial Flow along the Cylinder Surface**
   - **Relaxation of Flow Instability**

Full 3-D CANDU-6 Grids



Total = 6.74 M
Hexa = 5.11 M
Pyramid = 0.013 M
Tetra = 1.62 M

Boundary/Initial Conditions

- Massflow rate at each inlet = 127.4 kg/s.
  (Total Massflow = 1,019 kg/s)
- No-slip at the wall.

- Outlet pressure: fixed, & zero-gradient otherwise.

- Inlet temperature = 47.3 degC.
- Adiabatic at the wall.

- Initially stationary & isothermal.

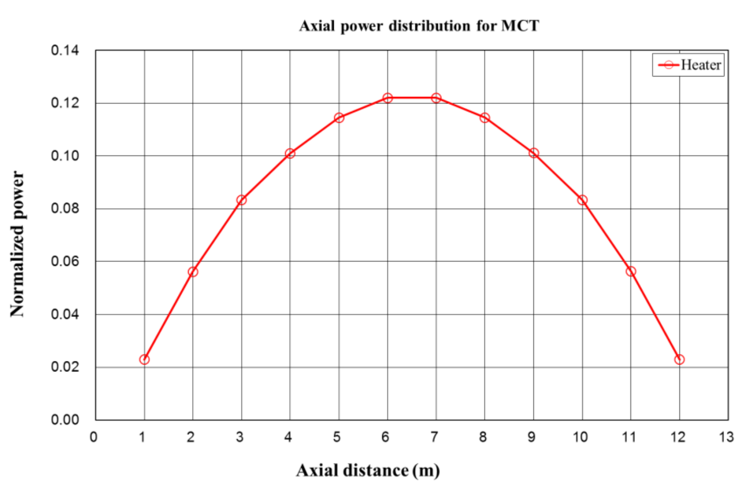
Modeling of the Heat Source

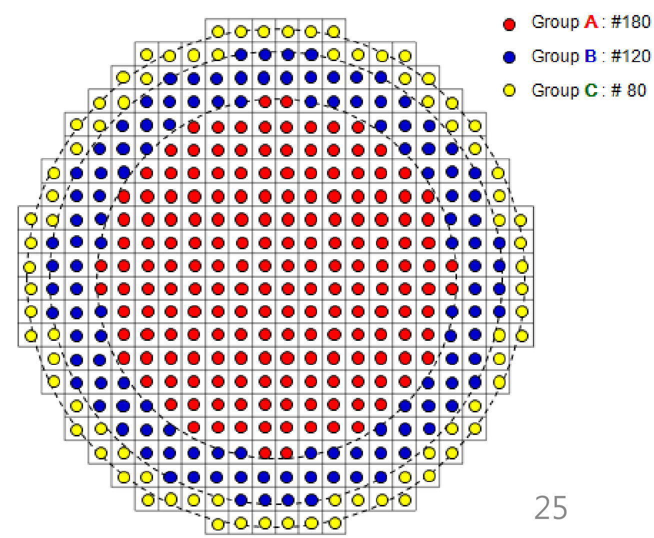
Total Thermal Power = 100 MW
Correction Factor for the Volume of Tube Bundle
= 1.089

$$Q_s(r,z)=Q_s\,f_r(r)\,f_z(z)$$

$$f_r(r)=0.94588-0.01989\,r+0.0995\,r^2-0.03888\,r^3-0.00256\,r^4\ (0.0\le r\le 3.38\ m)$$

$$f_z(z)=1.0-0.1111\,z^2\ (-3.0\le z\le 3.0\ m)$$



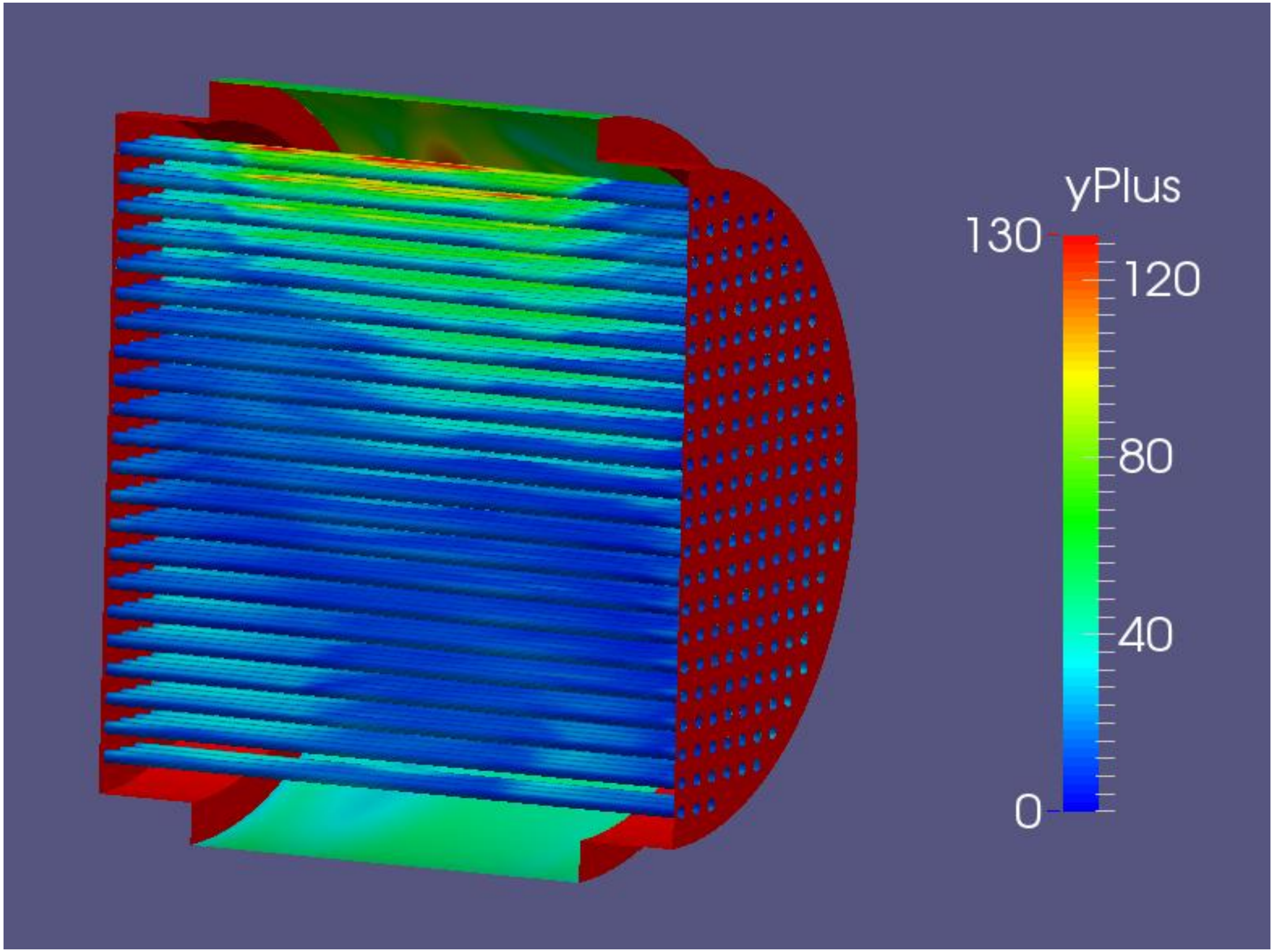


Implementation with OpenFOAM

File Name: fvOption

```
energySource1
{
	type scalarCodedSource;
	active true;
	selectionMode all;
	#{
		...
		forAll(cc,cellI)
		{
			...		//define the source terms
		}
	#};
}
```

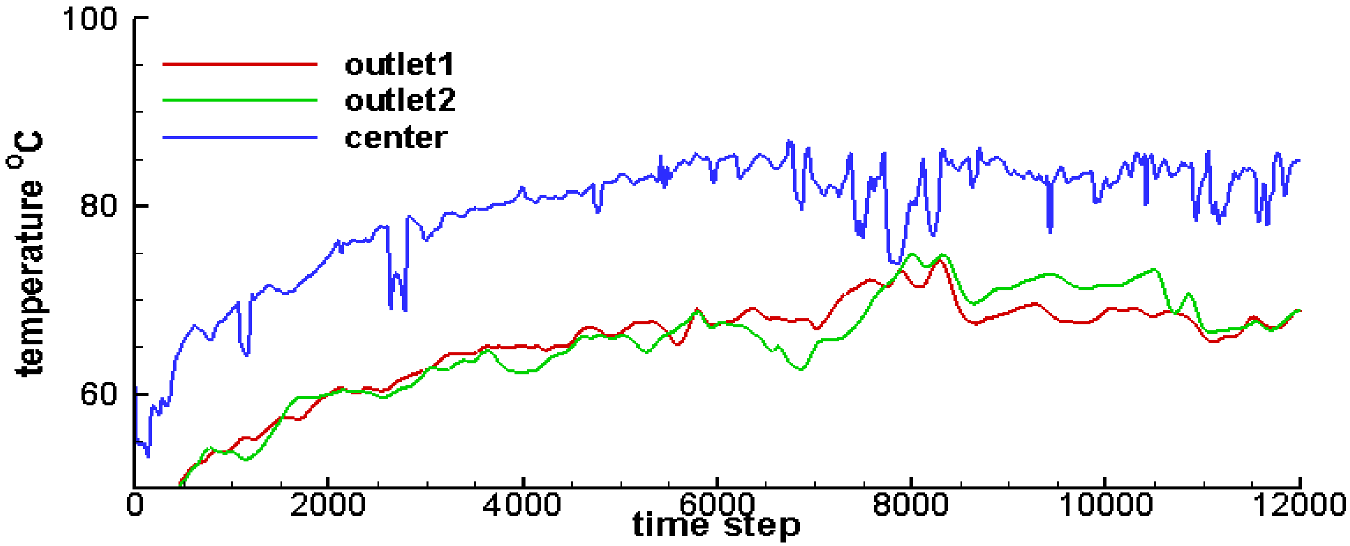
Turbulence Scale



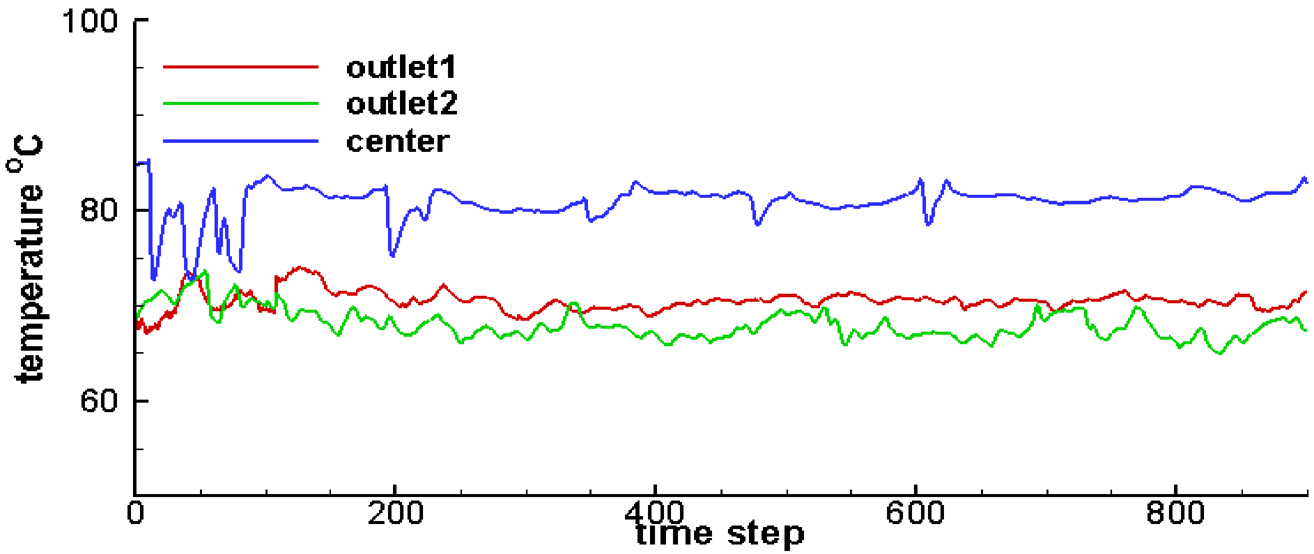
At 935 s,
y^+ <80

Temperature at the Inlet/Outlet

Stage 1

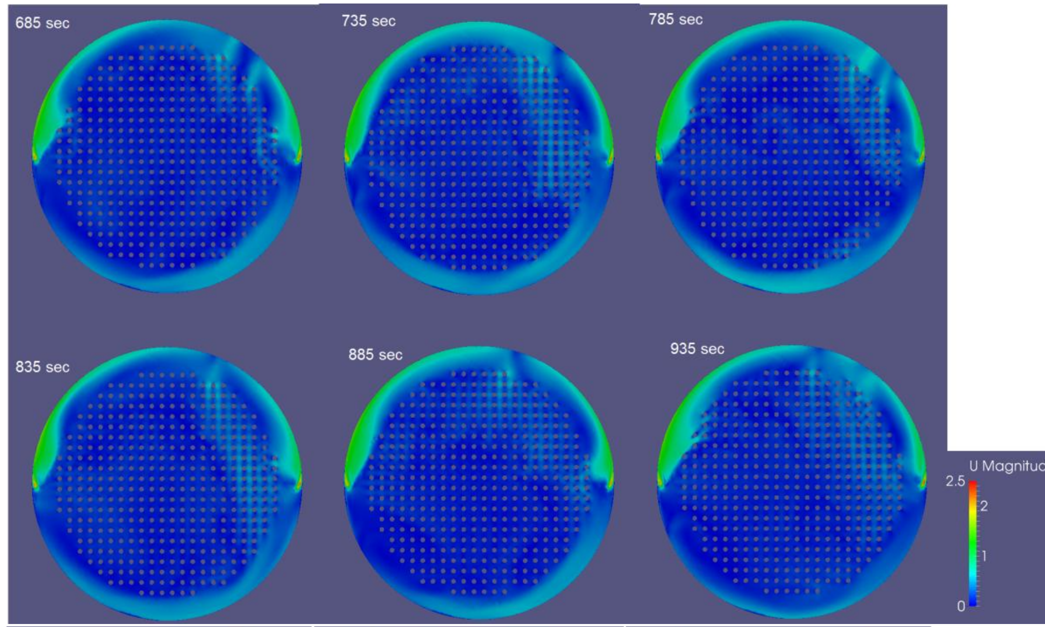


Stage 2

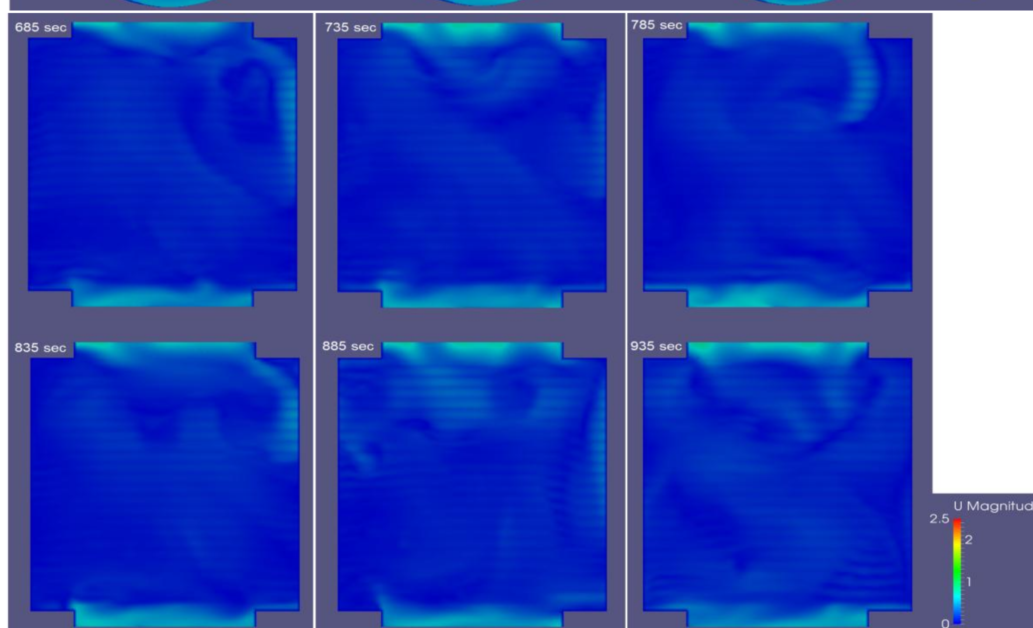


Velocity Profiles

Cross-sectional

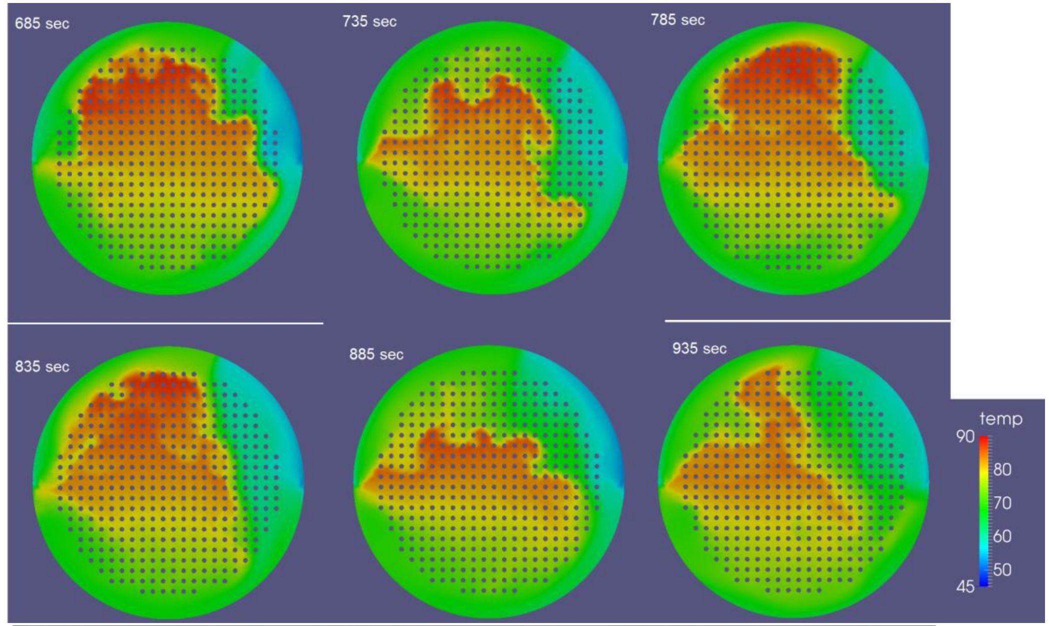


Axial

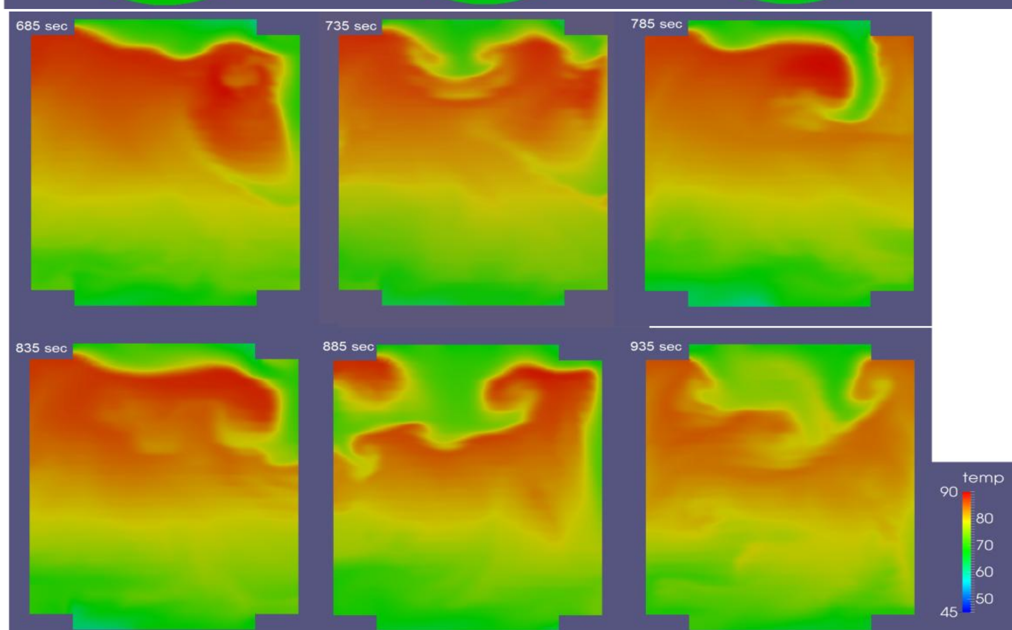


Temperature Profiles

Cross-sectional



Axial



Animations

Summary and Conclusion

1) Computation with OpenFOAM: gives us a new horizon in potential. Simulation of CANDU-6 with open source codes may be possible with full scale in the near future.

2) Validation of STERN Lab. Exp. with OpenFOAM:
   - Cause of Error: turbulence intensity, surface roughness, 3-D effect, and grid dependency, etc.

3) Simulation of Full-scale CANDU-6 Moderator:
   - Two-stage computation/pseudo-steady
   - Max. temperature: 89 degC

Q & A

Thank you for your attention!