NEXTfoam
open source CFD consulting

2014 전산유체공학회 춘계 학술대회

오픈 소스 CFD 프로그램 오픈폼(OpenFOAM)이 제공하는 다물리 해석을 위한 라이브러리

2013. 12. 05 김 병 윤
bykim@nextfoam.co.kr

목 차

- 개요
- 오픈폼의 물리 모델 제공 방식
- 오픈폼이 제공하는 물리적 모델
  - 열전달, 다상유동, 입자유동, 동적격자계, 유체-구조 연성 해석, 화학 반응, 전자기장
- 결론

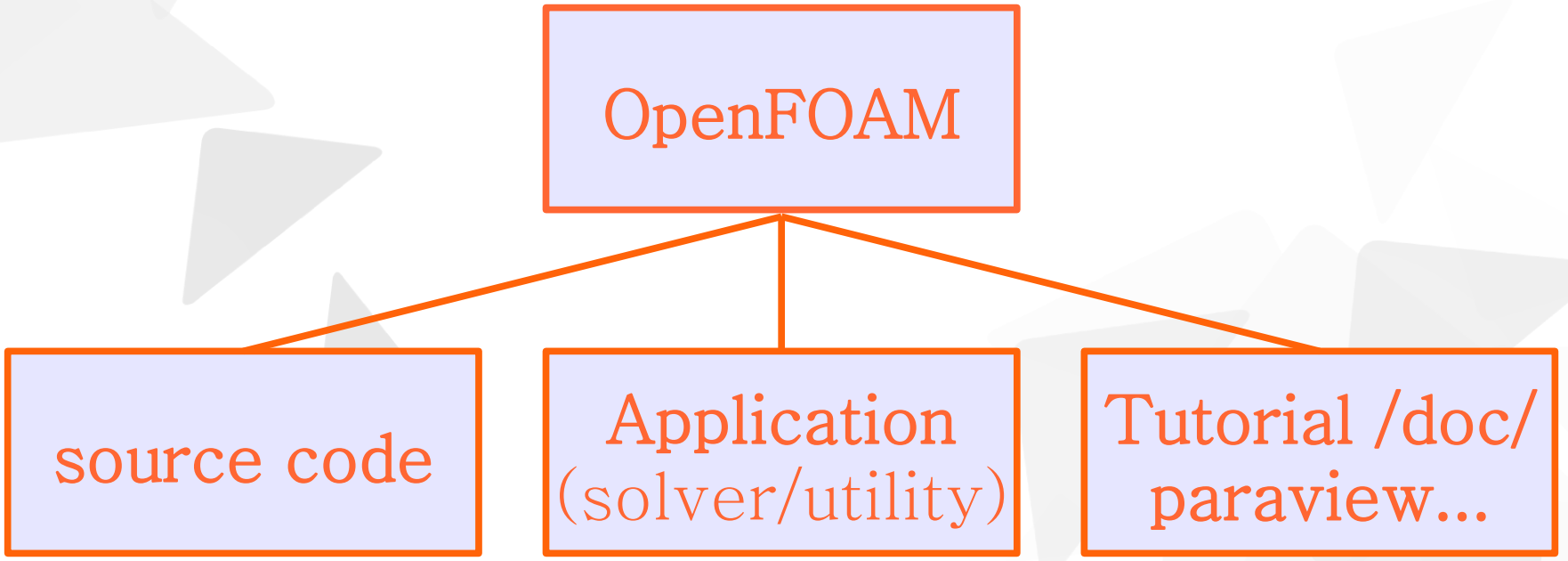
개요

- 본 발표의 목적
  - 오픈폼이 제공하는 다양한 물리 모델들을 소개
  - 오픈폼과 상용 코드의 차이점 소개
  - 다양한 분야의 활용 사례 소개

개요

- **Open Field Operation And Manipulation**
- Open Source CFD Toolbox

-실행파일(application)을 만들기 위한 C++ 프로그램들의 모음



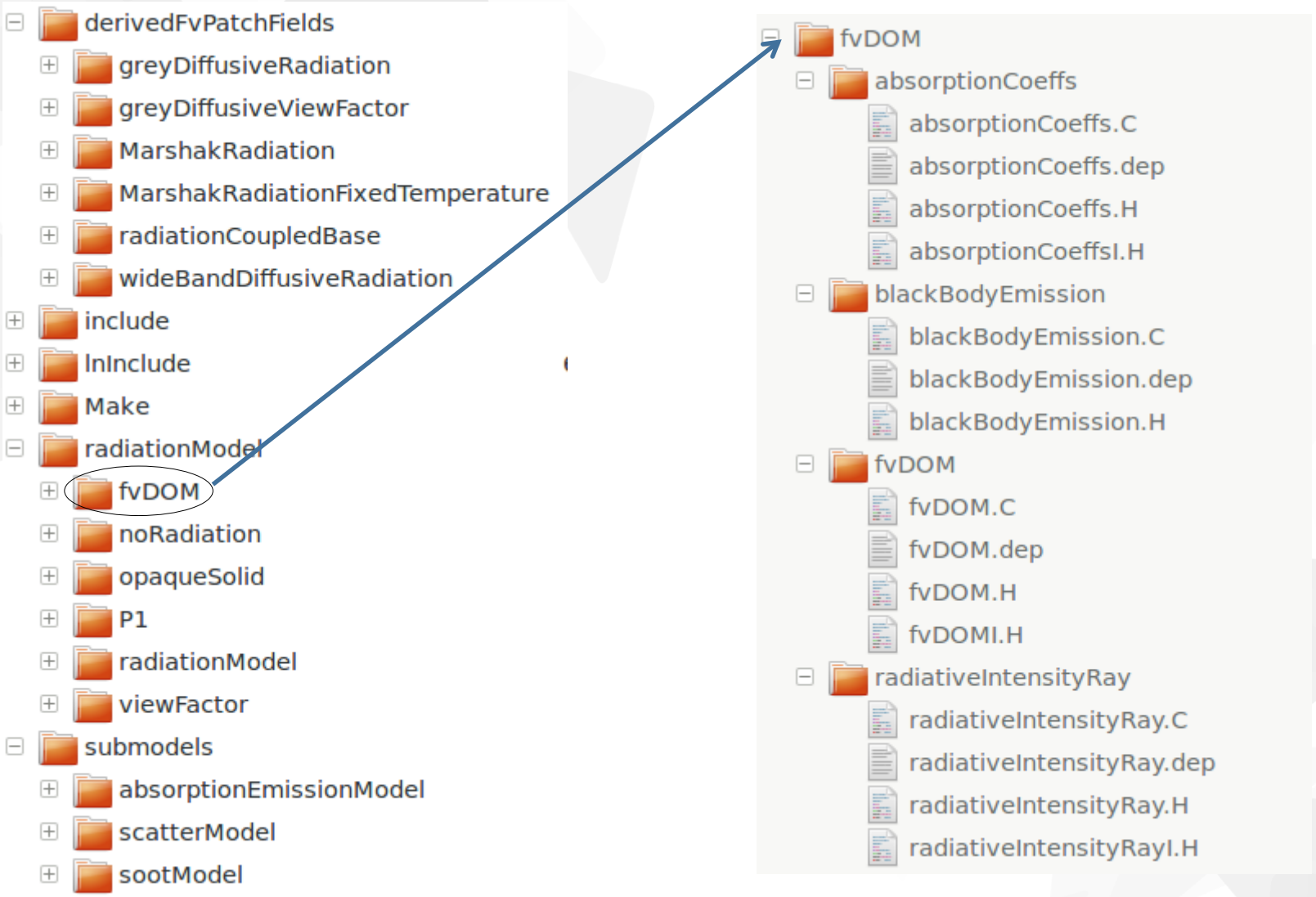
개요

- 오픈폼의 두 가지 버전
  - OpenFOAM Foundation(ESI) : OpenFOAM-2.3.0
  - OpenFOAM Extend Project : foam-extend-3.0, Jeju

오픈폼이 물리 모델 제공 방식

- 소스 코드(라이브러리)
- 유틸리티
- 솔버
- 경계조건

오픈폼이 물리 모델 제공 방식 - 소스 코드



오픈폼이 물리 모델 제공 방식 - 솔버

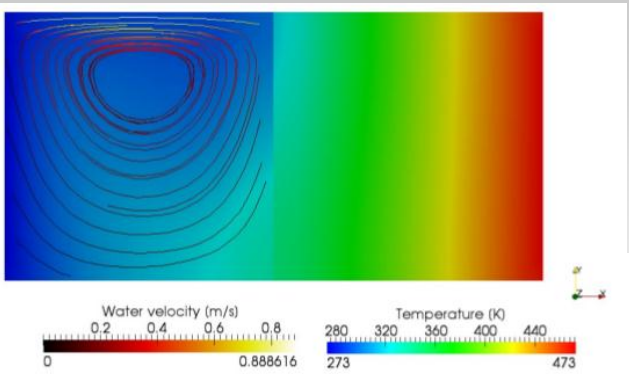
buoyantSimpleFoam.C

```
 ….
#include ...
#include "radiationModel.H"

int main(int argc, char *argv[])
{
    #include ...
    #include "createRadiationModel.H"

    simpleControl simple(mesh);

    while (simple.loop())    {
        {
            #include "UEqn.H"
            #include "EEqn.H"
            #include "pEqn.H"
        }
        turbulence->correct();
...
```

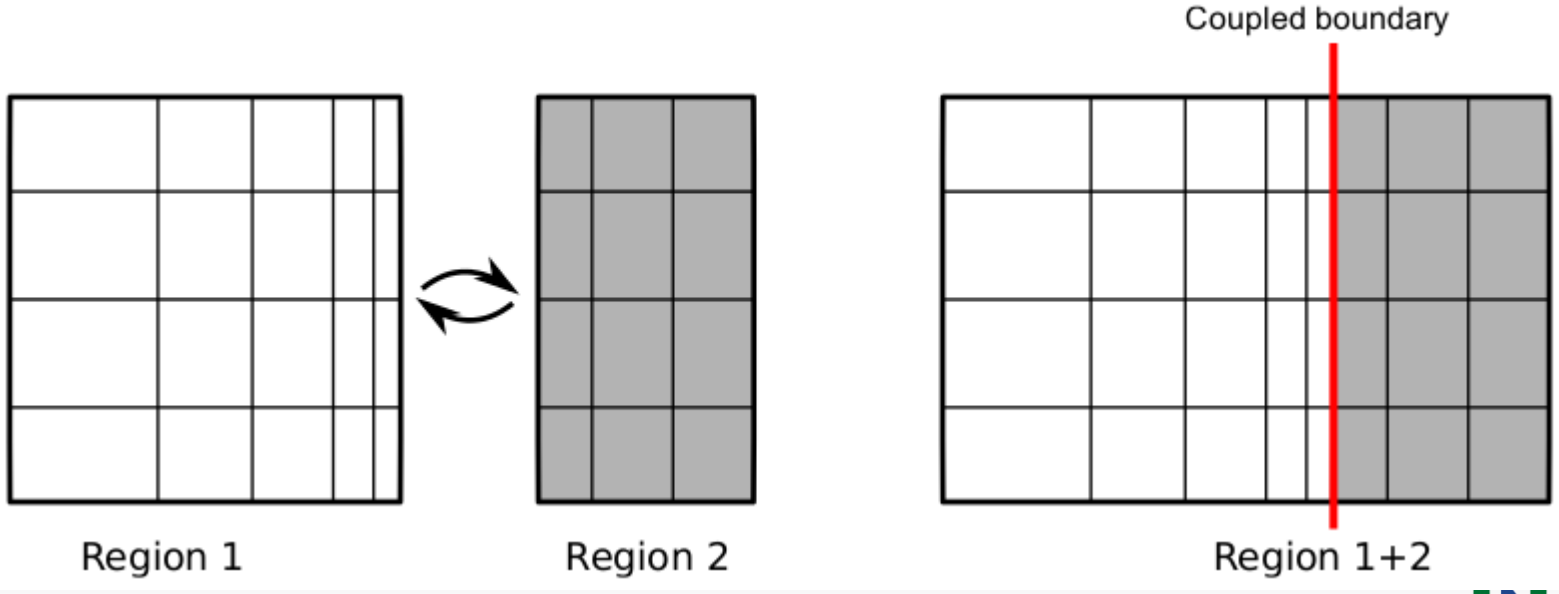
EEqn.H

```
 ….
fvScalarMatrix EEqn
(
    fvm::div(phi, he)
  + (
        fvc::div(phi, volScalarField("Ekp", 0.5*magSqr(U)
        + p/rho))
    )
  - fvm::laplacian(turbulence->alphaEff(), he)
 ==
    radiation->Sh(thermo)
  + fvOptions(rho, he)
);

EEqn.relax();
...
EEqn.solve();
...
radiation->correct();
```

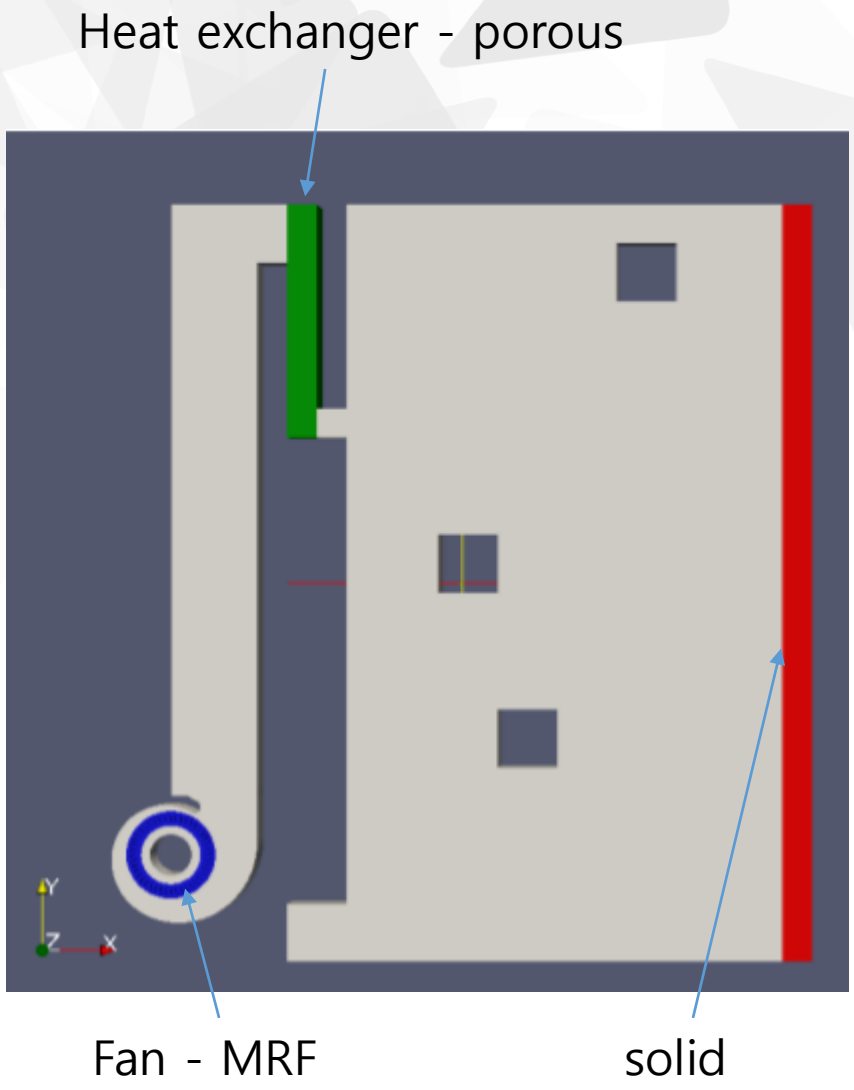
열전달

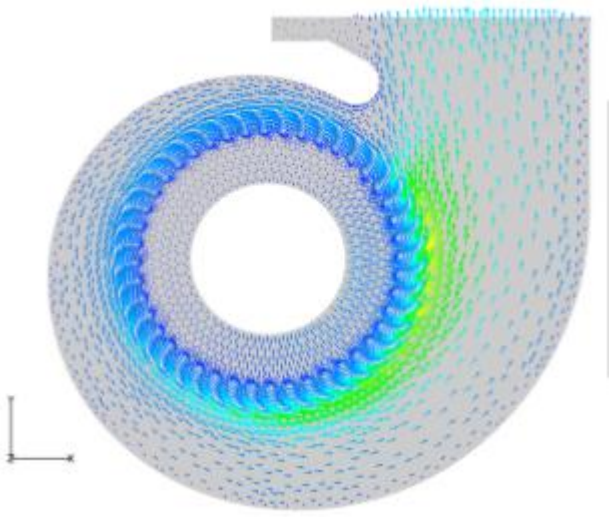


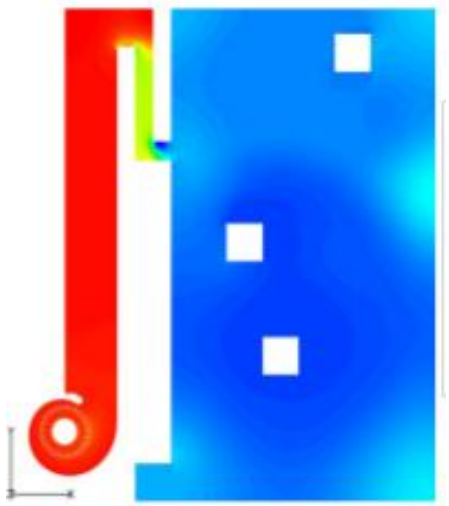
- 복합열전달(conjugated heat transfer)
  - Multi-region – fluid / solid
  - Segregated solver – iterative solution of region by region
  - Coupled solver – solve multiple region at once
    - block coupled solver, Jeju
- Thin wall

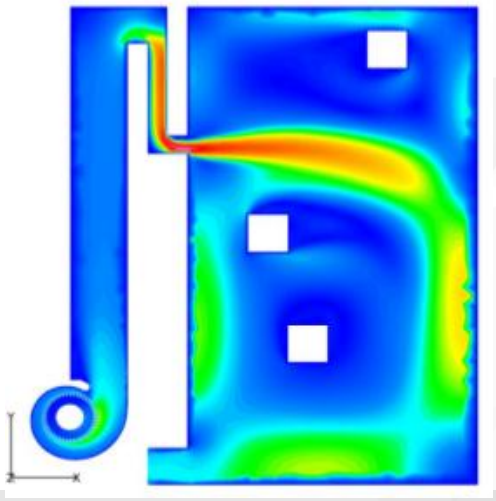


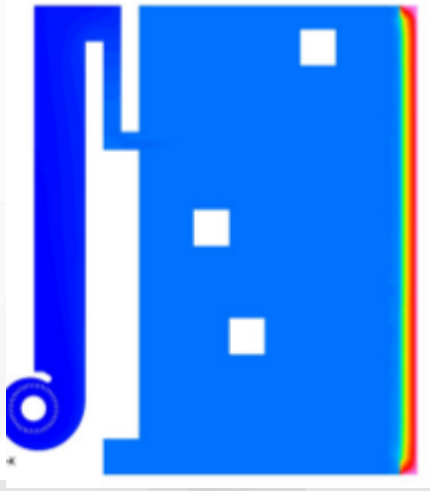
복합 열전달 해석 사례



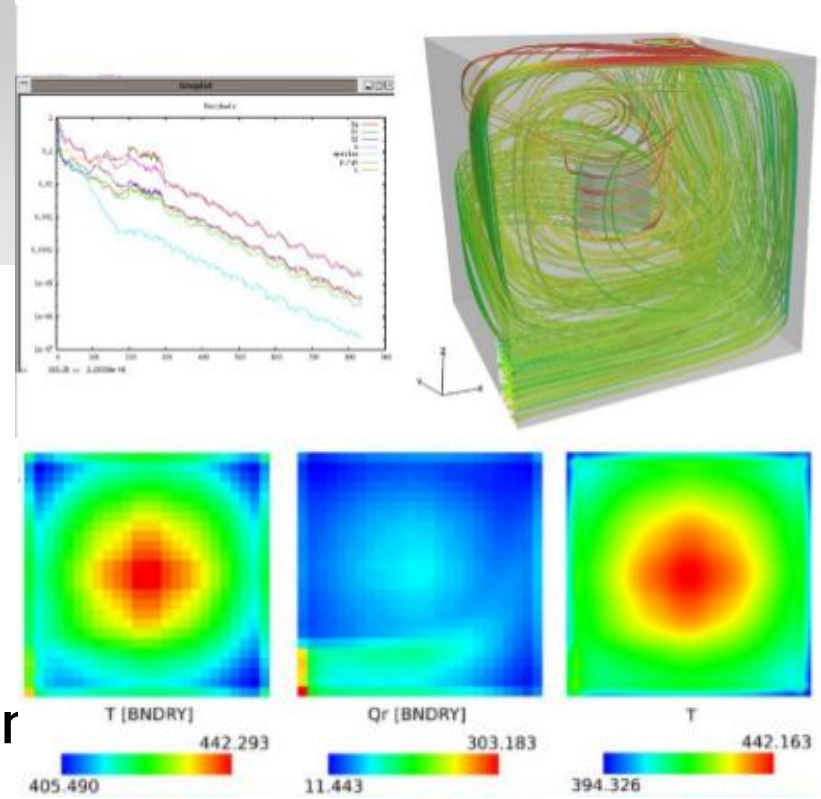








복사열전달



- 복사열전달
  - 복사열전달 모델
    - fvDOM, viewFactor, P1
  - Sub model
    - Scattering, soot, absorption/emissior
  - B.C.
- fvDOM model
  - Scattering, semi-transparent media, specular surface, banded gray
- viewFactor model
  - viewFactorGen utility
  - faceAgglomerate utility

열전달 표준 솔버

- buoyantBoussinesqPimpleFoam
- buoyantBoussinesqSimpleFoam
- buoyantPimpleFoam
- buoyantSimpleFoam
- thermoFoam
- chtMultiRegionFoam
- chtMultiRegionSimpleFoam
- conjugateHeatFoam - Jeju
- conjugateHeatSimpleFoam – Jeju

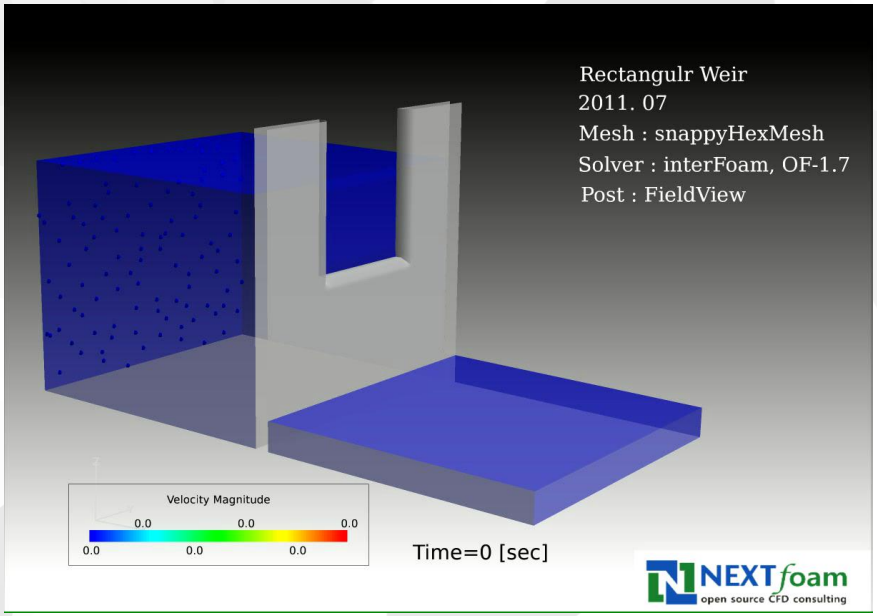
다상 유동

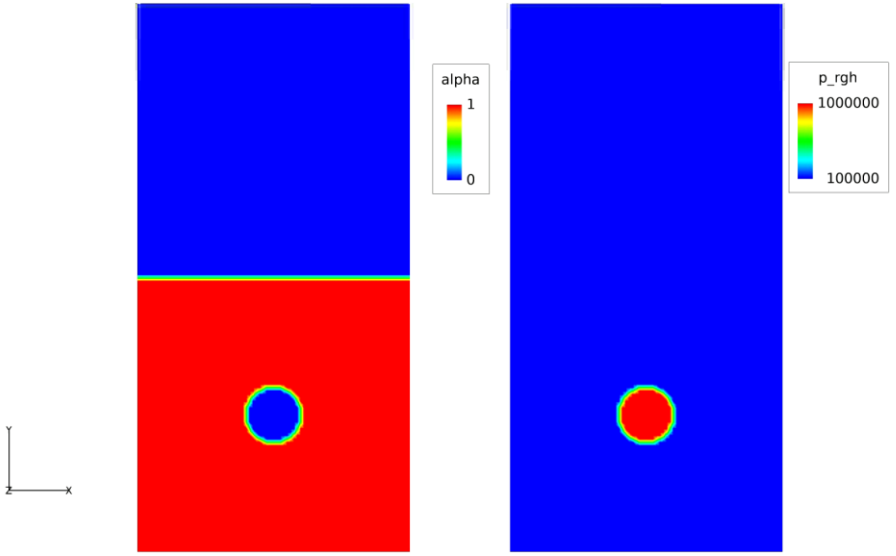
- 오픈폼이 지원하는 다상유동 해석 기능
  - VOF, Eulerian-Eulerian, mixture, DPM
- VOF
  - Explicit / semi-implicit solver
  - Compressible / incompressible
  - unsteady /steady(Local Time Step) solver
- Eulerian
  - Compressible / incompressible
  - Phase interaction sub model : Drag, virtual mass, heat transfer, lift, turbulence dispersion, wall lubrication
- Mixture
  - Phase change - cavitation

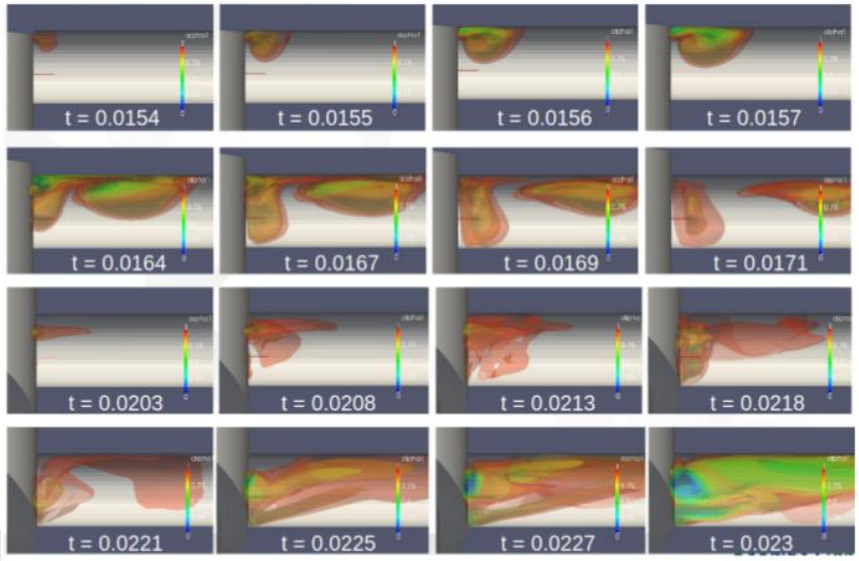
다상유동

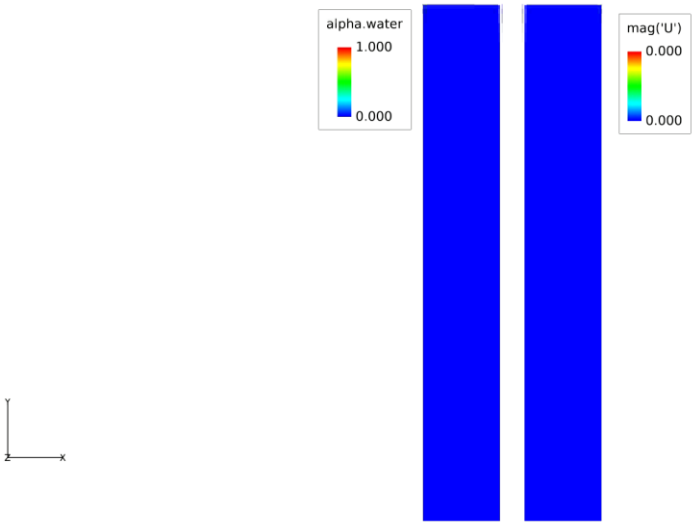
- 솔버
  - interFoam
  - interMixingFoam
  - interDyMFoam
  - LTSInterFoam
  - cavitatingFoam
  - cavitatingDyMFoam
  - interPhaseChangeFoam
  - interPhaseChangeDyMFoam
  - multiPhaseInterFoam
  - multiPhaseEulerFoam
  - twoPhaseEulerFoam
  - potentialFreeSurfaceFoam
  - settlingFoam
  - twoLiquidMixingFoam
  - barotropicCavitatingFoam - Jeju
  - bubbleFoam - Jeju

다상유동 해석 사례


Rectangulr Weir
2011. 07
Mesh : snappyHexMesh
Solver : interFoam, OF-1.7
Post : FieldView
Velocity Magnitude
0.0 0.0 0.0
0.0 0.0 0.0
Time=0 [sec]
NEXTfoam
open source CFD consulting


alpha
1
0
p_rgh
1000000
100000


t = 0.0154
t = 0.0155
t = 0.0156
t = 0.0157
t = 0.0164
t = 0.0167
t = 0.0169
t = 0.0171
t = 0.0203
t = 0.0208
t = 0.0213
t = 0.0218
t = 0.0221
t = 0.0225
t = 0.0227
t = 0.023


alpha.water
1.000
0.000
mag('U')
0.000
0.000

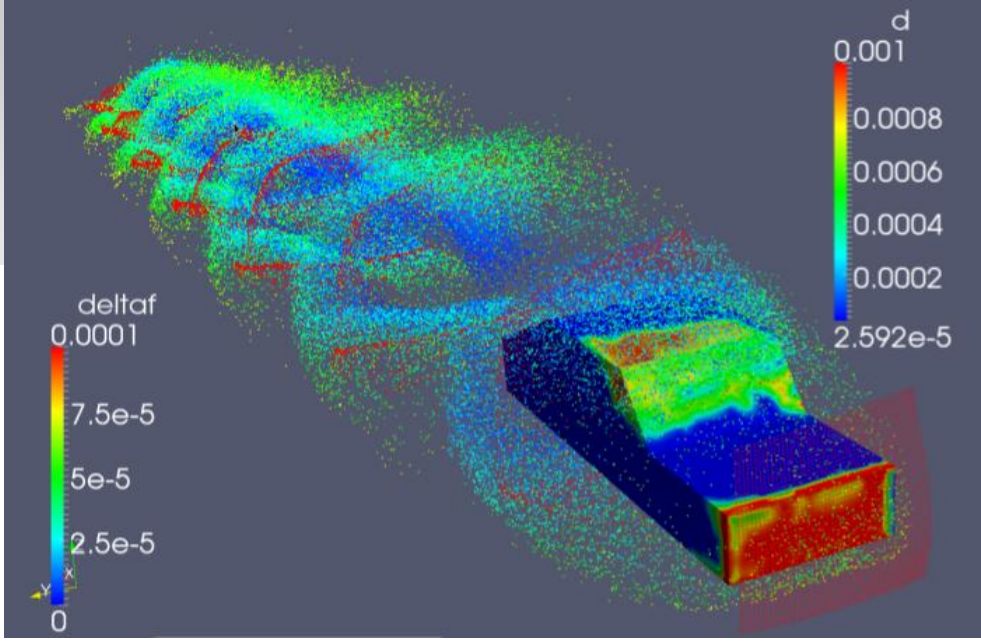
다상 유동 - DPM

- 분사 방법
  - Patch, cone, hollow cone, cell zone, lookup table, manual
- Atomization model
  - Blob sheet atomization, LISA
- Breakup model
  - TAB, ETAB, ReitzDiwakar, ReitzHRT, PilchErdman, SHF
- Distribution model
  - RosinRammler, uniform, exponential
- Drag model
  - WenYu, ErgunWenYu, PlessisMasliyah
- Wall interaction
  - rebound, stick, escape

다상 유동 - DPM

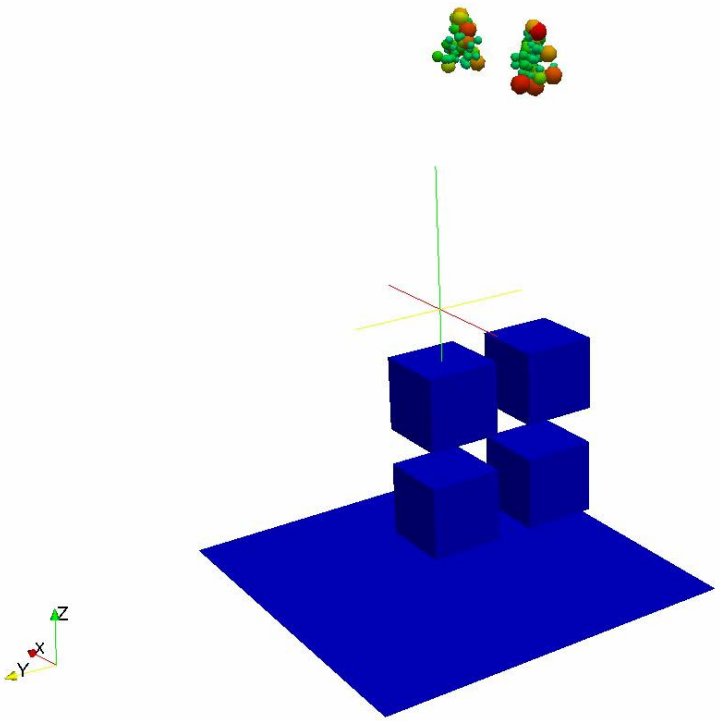
- RansMarshall heat transfer, devolatilisation model, phase change, stochastic collision model, surface film
- Dense particle flow model
  - Include the effect of particulate volume fraction on the continuous phase
  - Resolve particle-particle interaction
- MP-PIC(Multi-Phase Particle In Cell) method
  - For collisional exchange
  - Packing models
    - Inter-particle stress model
    - Collisional damping model
    - Collisional isotropy model

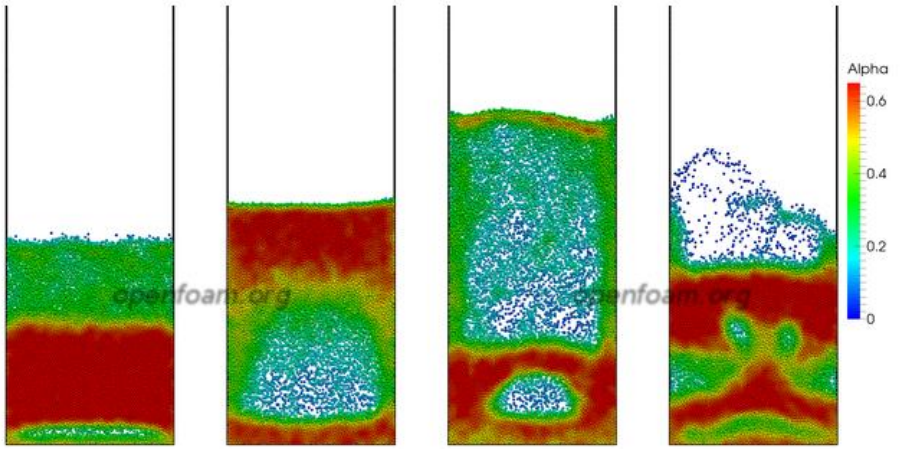
다상유동 – Surface Film

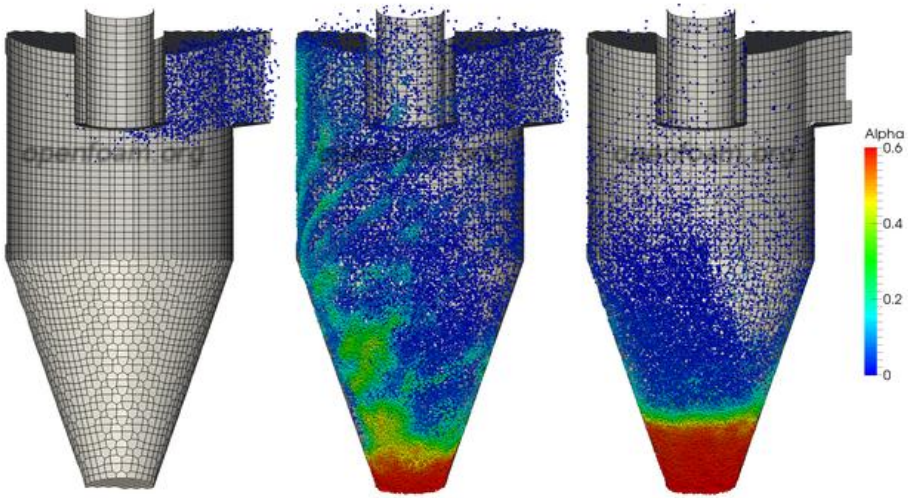


- Two approach
  - Multi-region approach
    - Create one volume mesh layer
    - FVM
  - FAM(Finite Area Method) approach - Jeju
    - Use surface mesh
- Thermodynamic models for density, Surface tension, thermal conductivity, specific heat capacity
- Viscosity model
- Film turbulence model
- Phase change : evaporation, solidification

DPM, Surface Film 해석 사례

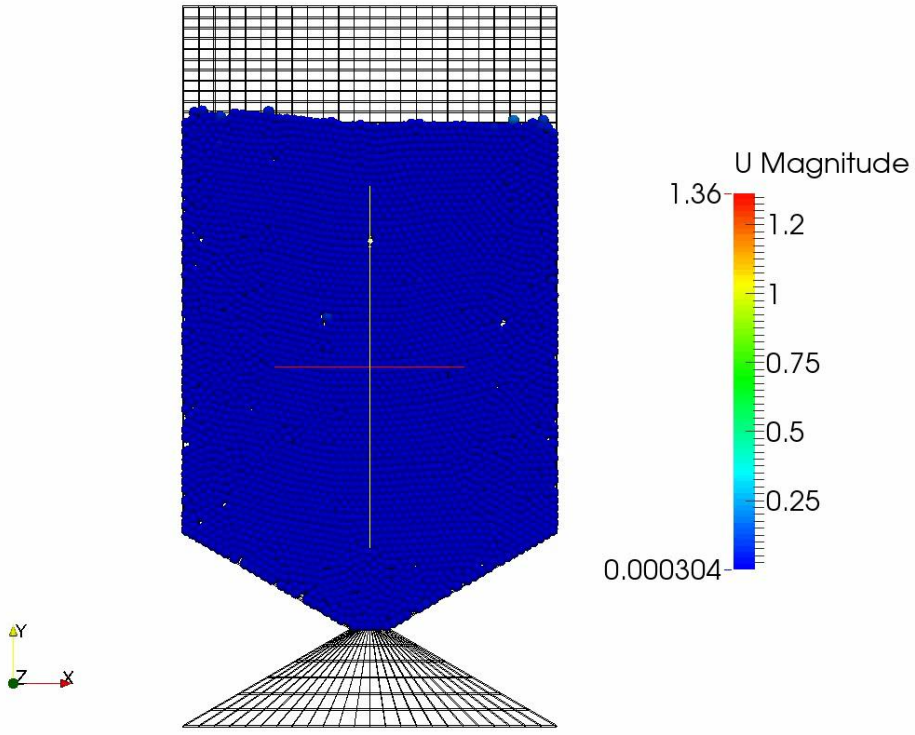


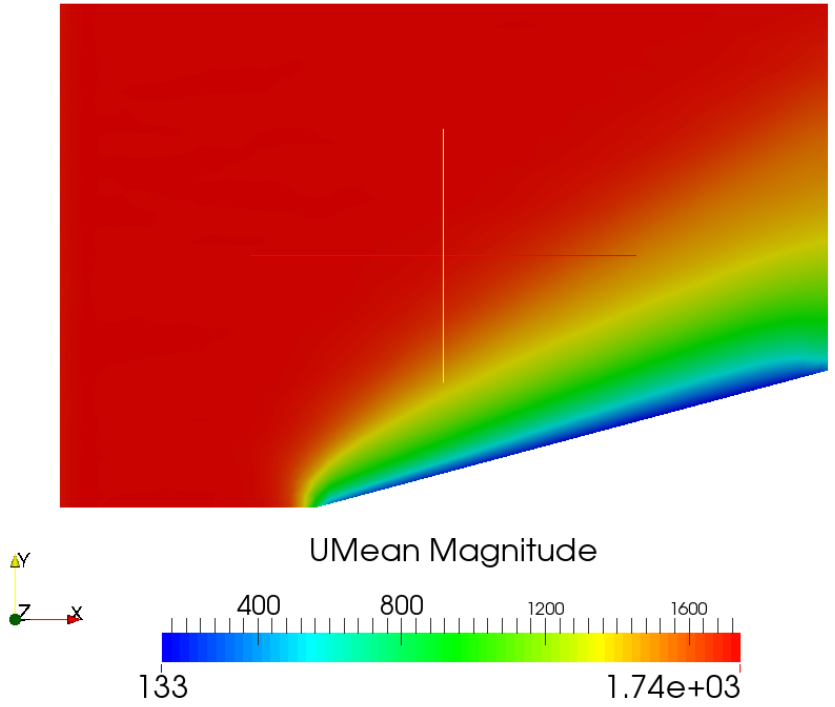




입자 유동

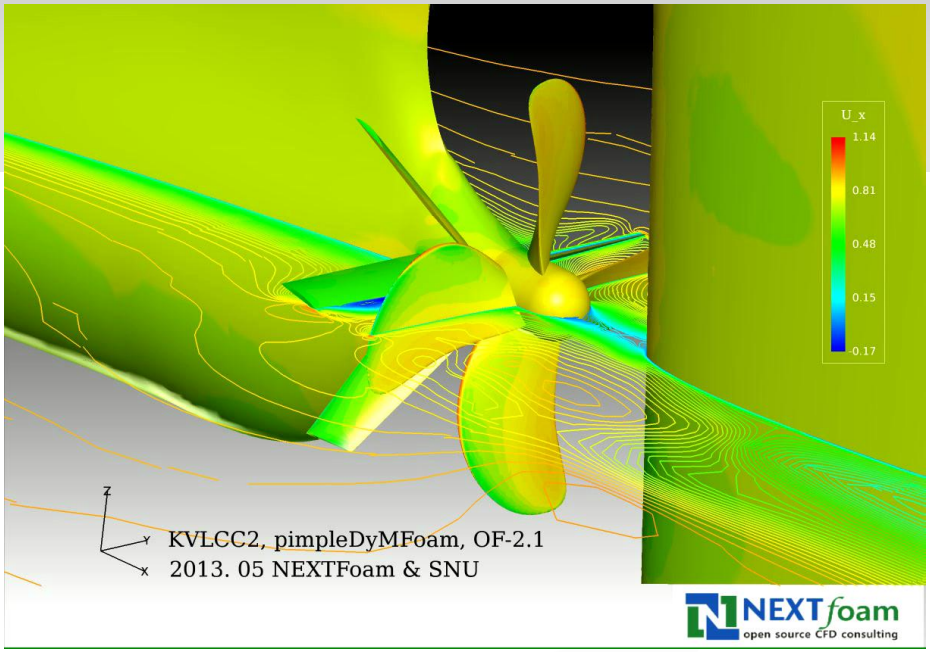
- MD(Molecular dynamics)
- DSMC(Direct Simulation Monte Carlo)
- DEM(Discrete Element Method)

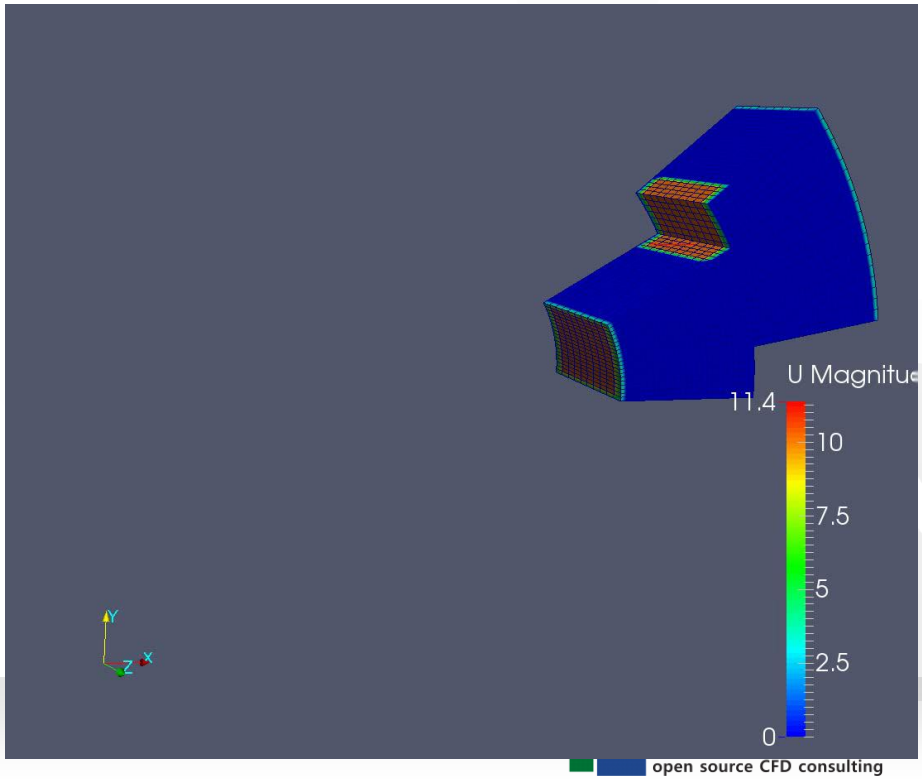




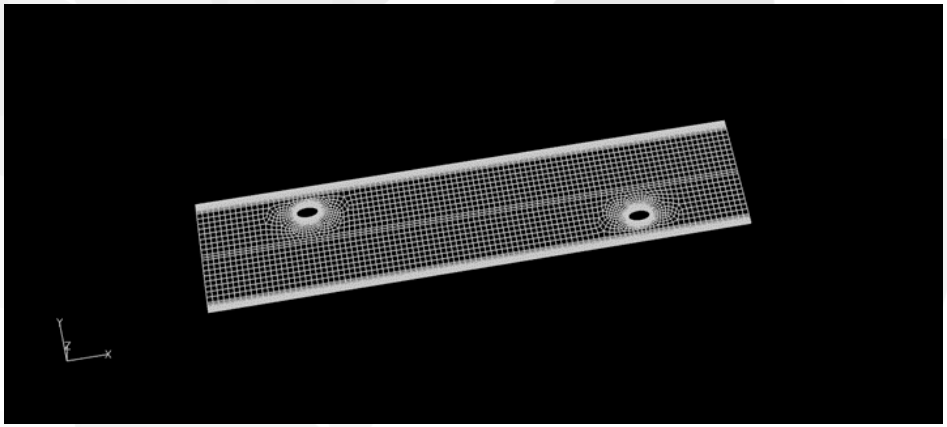
동적 격자계

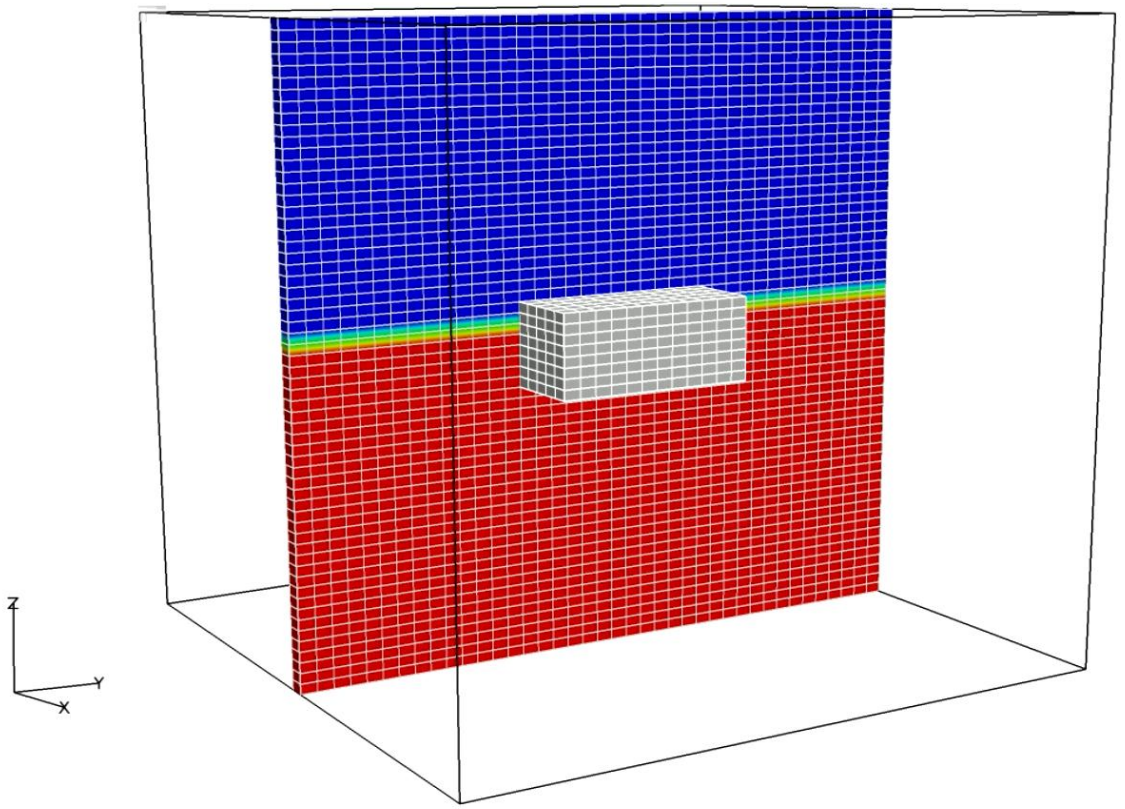
- Given motion / 6-DOF Motion
- Dynamic refinement/unrefinement
- 격자 변형 기법
  - Sliding mesh
  - Smoothing
  - Layering
  - Deformation
    - Tet FEM mesh deformation
    - RBF mesh deformation
    - Tetrahedral remeshing





동적 격자계





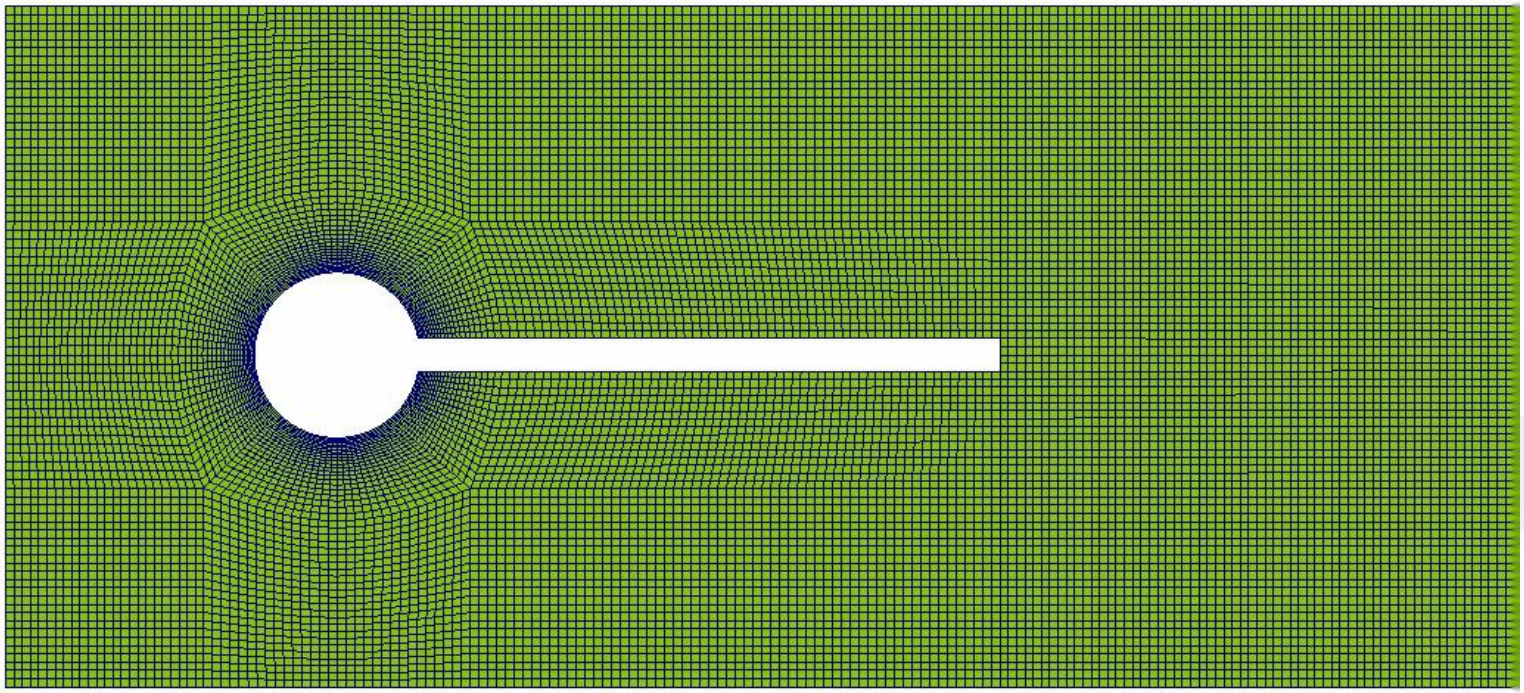
유체 구조 연성해석(FSI)

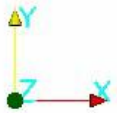
- FVM 구조해석 라이브러리
- Contact, crack, thermal model
- 구조해석 솔버

  - elasticAcpSolidFoam / elasticIncrAcpSolidFoam / elasticIncrSolidFoam / elasticNonLinIncrTLSolidFoam / elasticNonLinTLSolidFoam / elasticNonLinULSolidFoam / elasticOrthoAcpSolidFoam / elasticOrthoNonLinULSolidFoam / elasticOrthoSolidFoam / elasticPlasticSolidFoam / elasticPlasticNonLinTLSolidFoam / elasticPlasticNonLinULSolidFoam / elasticSolidFoam / elasticThermalSolidFoam / viscoElasticSolidFoam
- FSI solver : icoFsiElasticNonLinULSolidFoam

FSI – 해석 사례

- Solver : icoFsiElasticNonLinULSolidFoam
- Tight coupling using Aitken's under-relaxation method

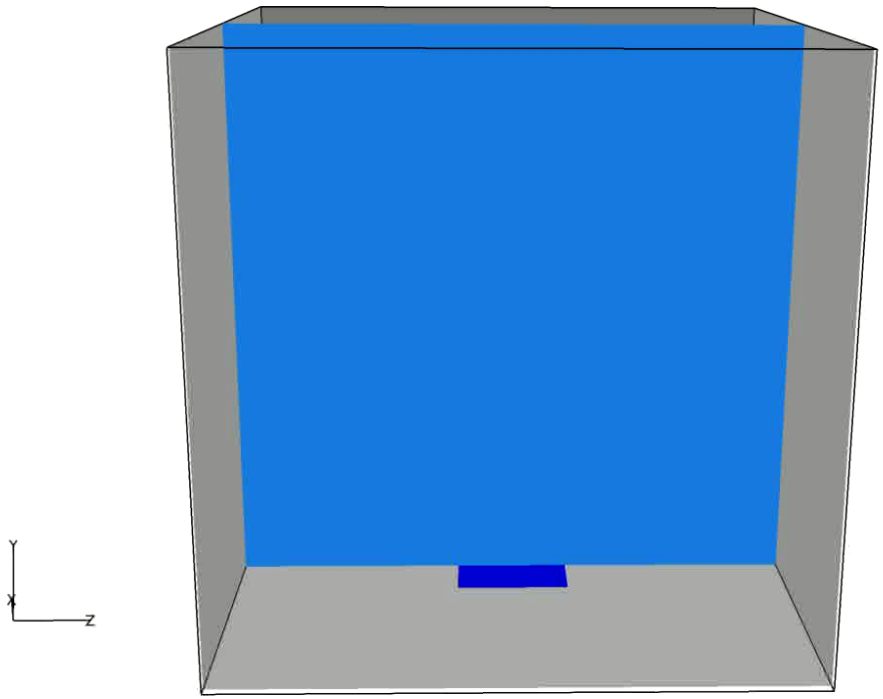




foam-3.0-extend tutorial, E=1e4

화학반응

- 연소 모델
  - Laminar, single step combustion, infinitely fast chemistry, partially stirred reactor, diffusion, flame surface density
- ODE solver
- Chemkin reader
- Surface reaction model
- Soot formation model
- Pyrolysis model
- ICE
  - Dynamic mesh
  - Ignition model
  - Post-processing utilities



전자기장

- Solvers
  - electrostaticFoam
  - magneticFoam
    - Magnetic scalar potentia의 poission 방정식을 계산
    - Magnetic field intensity H, magnetic flux density B를 계산
  - mhdFoam
    - 자기장이 주어진 영역을 지나는 비압축성 층류 솔버

맺음말

- 오픈폼은 다양한 물리 현상을 구현할 수 있는 라이브러리, 유틸리티, 경계조건, 솔버를 제공한다.
- 표준 솔버는 모든 기능들을 포함하지 않으며 사용자는 필요한 기능을 모아 자신의 솔버를 제작할 수 있다.
- 모델 및 솔버의 검증 및 개발시 공동연구 및 자료 공유가 중요하다.