LA-UR-24-26805

Approved for public release; distribution is unlimited.

Title: HOSS-FSIS example: Flow in a pipe

Author(s): Euser, Bryan Jeffry
Padilla, Angel Melissa
Rougier, Esteban
Lei, Zhou
Knight, Earl E.

Intended for: Training material for external customers

Issued: 2024-07-08

HOSS-FSIS example: Flow in a pipe

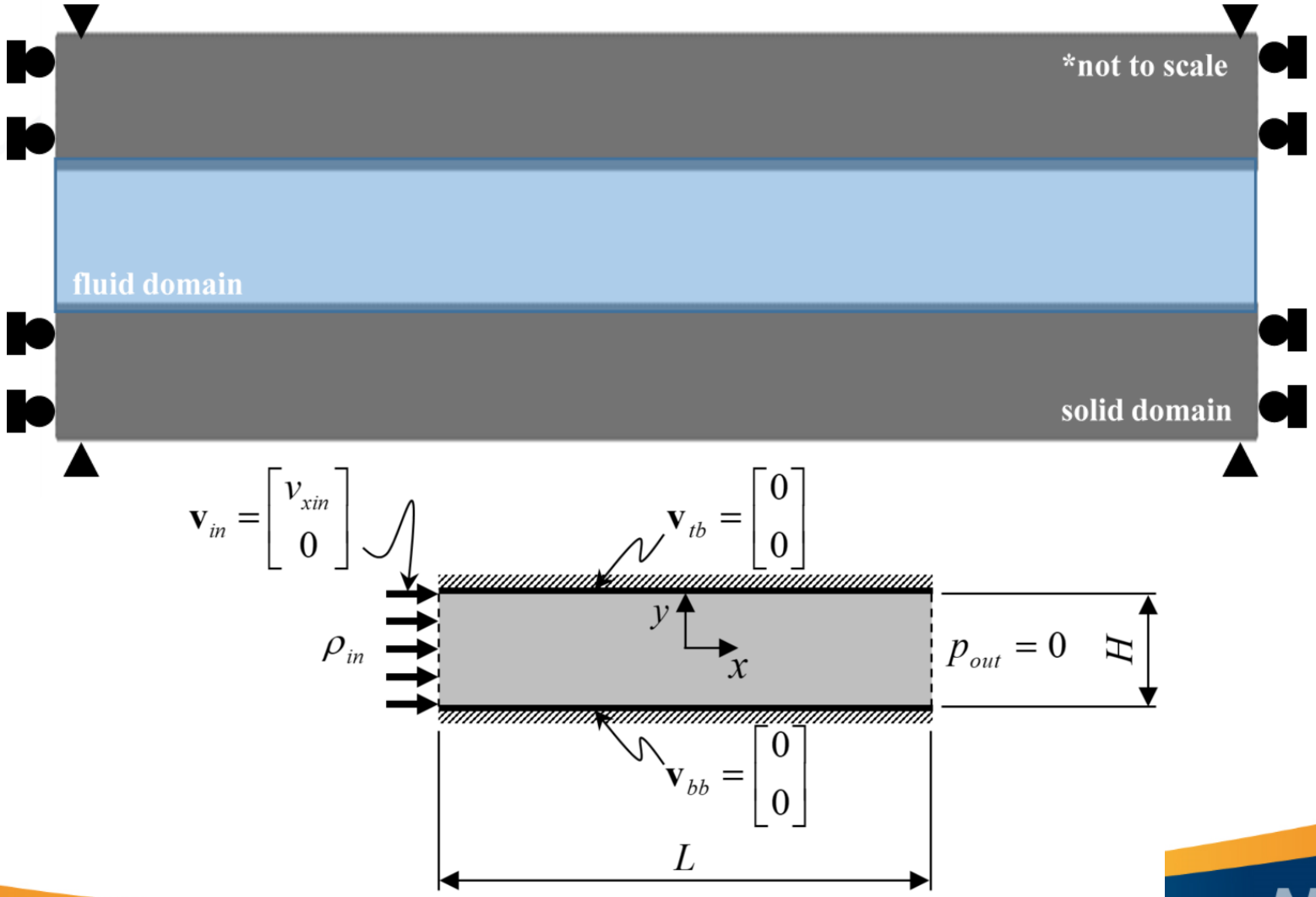
Bryan Euser, Angel Padilla, Esteban Rougier, Zhou Lei, Earl Knight

July 2024

Operated Triad National Security, LLC for the U.S. Department of Energy's NNSA

LA-UR-20-29964

Create input files that represent two infinite plates with water flowing in between them

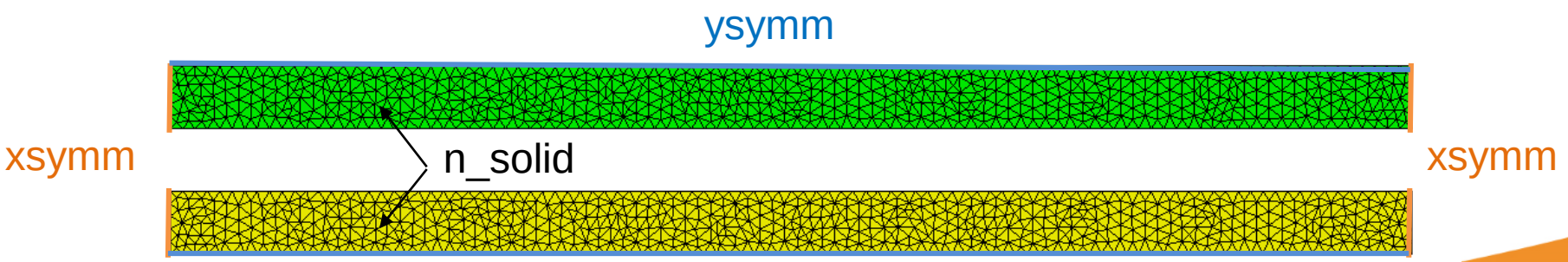


Setting up the Solid Components in Cubit

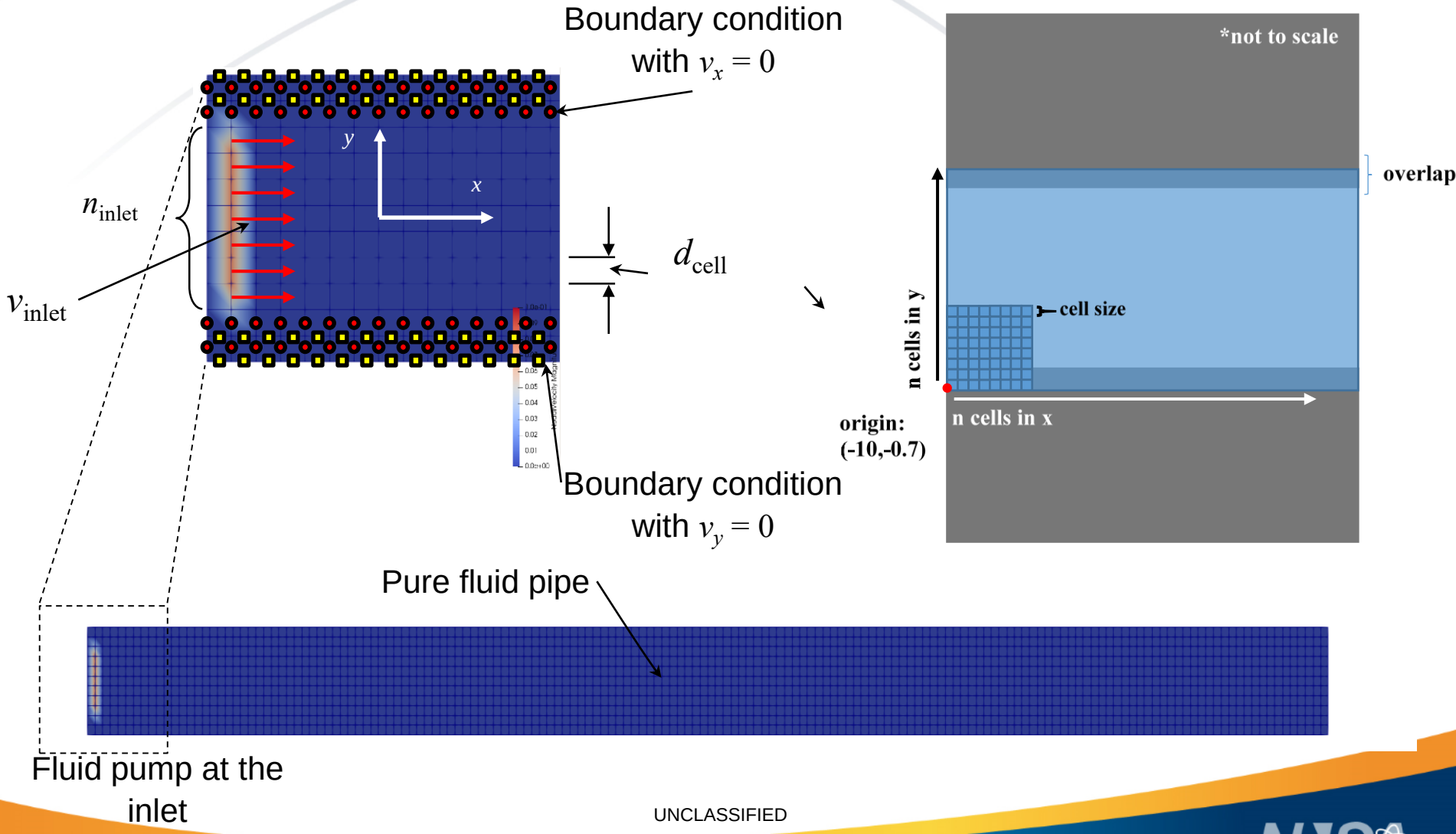
```
reset
create surface rectangle width 20 height 1 zplane
move vertex 3 location x -10 y 0.5
surface 1 copy reflect y

surface all scheme tridelaunay
tridelaunay point placement gq
surface all size 0.2
mesh surface all
nodeset 1 vertex 1 2 5 6
nodeset 1 name "fixed"
nodeset 2 curve 2 4 6 8
nodeset 2 name "xsymm"
nodeset 3 curve 1 5
nodeset 3 name "ysymm"
block 1 surface all
block 1 name "n_solid"
export abaqus"mesh.inp" overwrite
```

It is important to note that only the solid components of the simulation are defined in cubit. The fluid grid will be defined later.

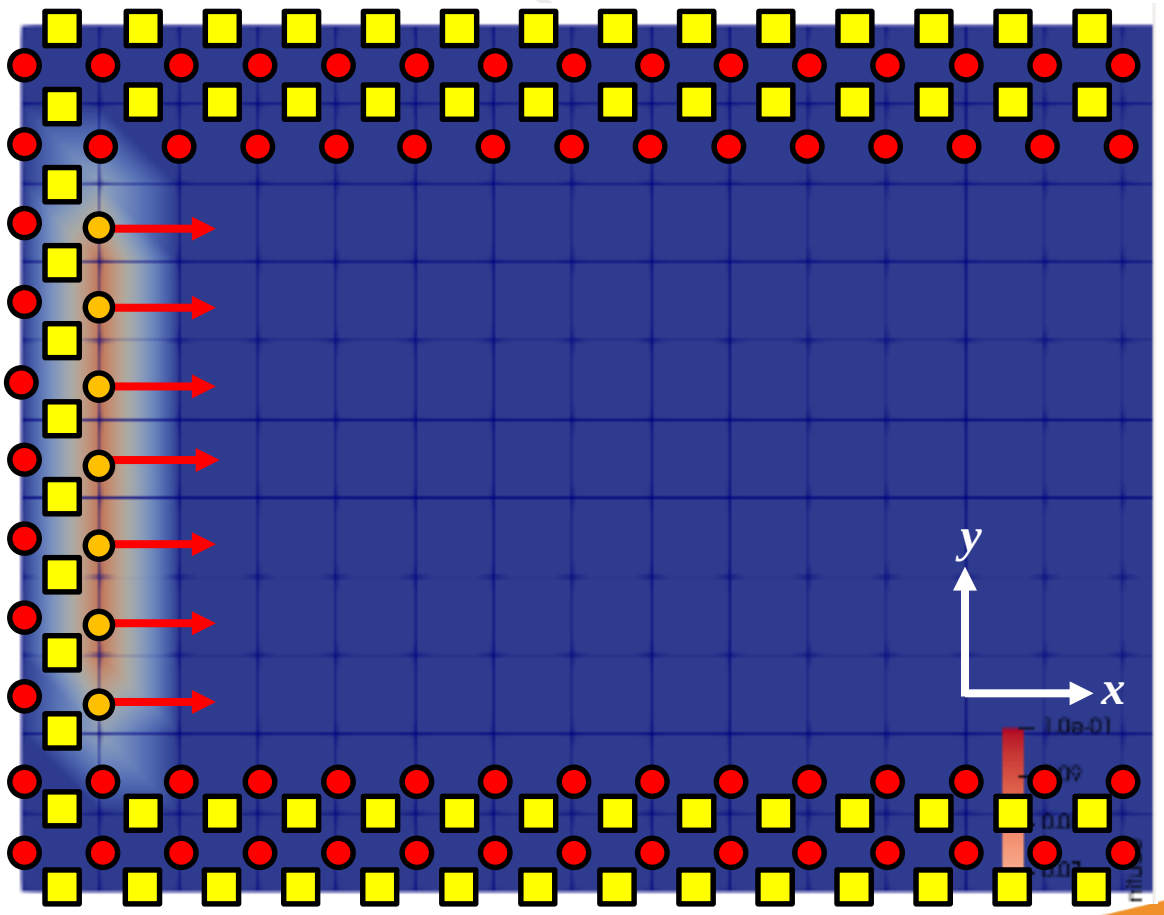


Visualizing the FSIS set up



Looking a little closer… we need to define the boundary conditions for the fluid system

- We need to define a single cell group:
  - FLCGBorder
- In addition, we will have 3 node groups:
  - FLNGVyBorder
  - FLNGVxBorder
  - FLNGVxPump
- These groups will be included in the **fmesh.dat** file
- The fmesh.dat is used to create our fluid simulation domain



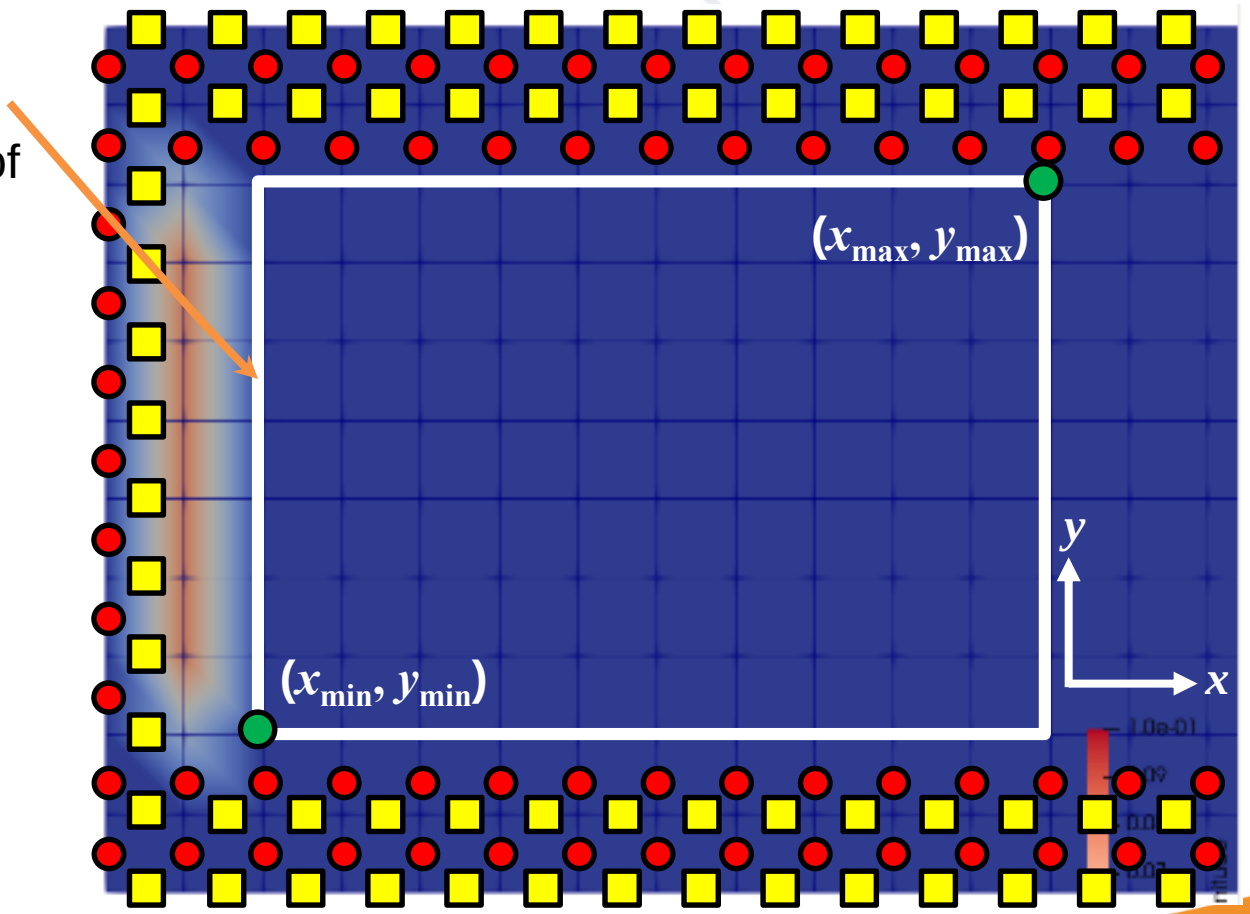
Logic for defining cell/node groups

- Rectangles define what cells/nodes to included in groups we want to define.
  - Defined by a pair of (x,y) coordinates

- Inside rectangle?
  - flag = 1
- Outside rectangle?
  - flag = 0

- x-nodes?
  - flag = 1
- y-nodes?
  - flag = 2

note* All of this logic extends to 3D problems



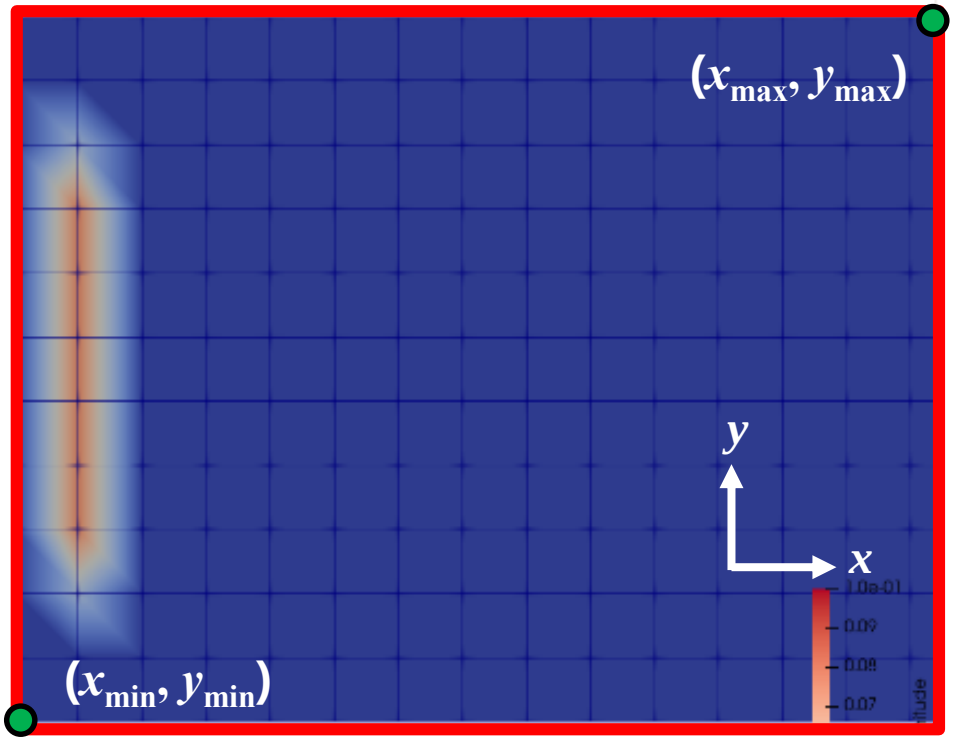
fmesh.dat – begin by defining origin, cell size, and number

```
<Dimension>
    2
</Dimension>

<RegularMeshDomain> FLUIDGROUP
    MeshOriginCoordinateX -10.0
    MeshOriginCoordinateY -0.7
    CellSize 0.1
    NumberOfCellsInX  200
    NumberOfCellsInY  14
</RegularMeshDomain>
```

Tells mesh generator the size of domain and total number of cells

Origin is the minimum x- and y-coordinates

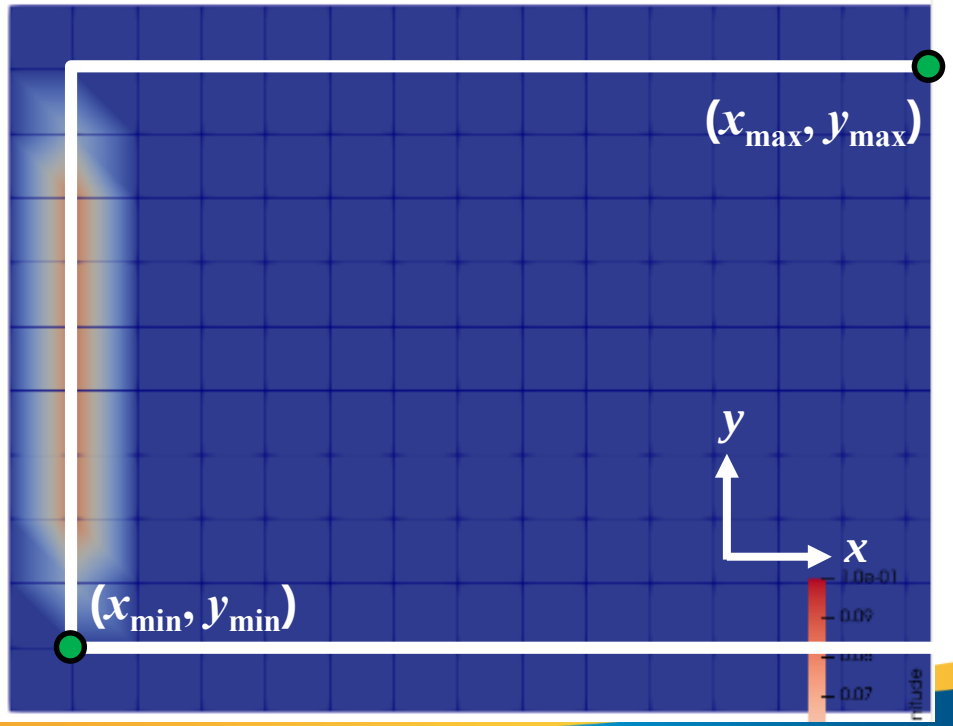


fmesh.dat – cell group

```
<FLCellGroupBC> FLCGBorder
    IfInside 0
    Corner1CoordinateX -9.9
    Corner1CoordinateY -0.6
    Corner2CoordinateX +9.9
    Corner2CoordinateY +0.6
</FLCellGroupBC> FLCGBorder
```

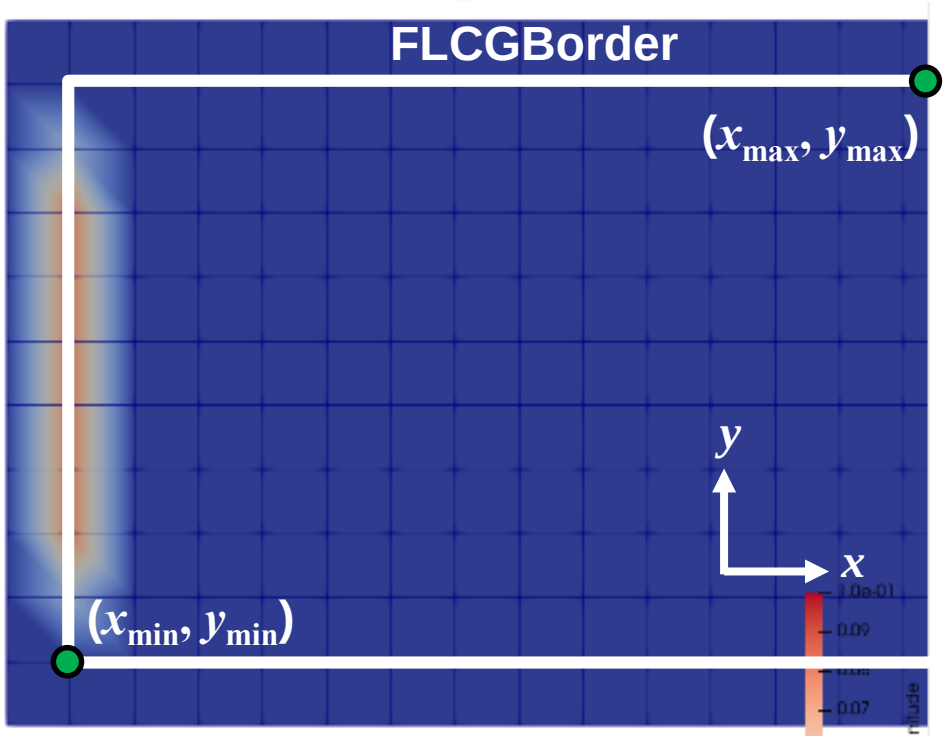
Inside rectangle? flag = 1
Outside rectangle? flag = 0

FLCGBorder



fmesh.dat – cell group

- You will always have at least one group (e.g., FLCGBorder).
- The border cells act as a reservoir where fluid can either enter or exit the system.
- The ‘main’ cells (those inside the border) are where the computation will take place.
- One can define variables such as density, specific energy, etc.



Operated Triad National Security, LLC for the U.S. Department of Energy's NNSA

LA-UR-20-29964

fmesh.dat – node groups

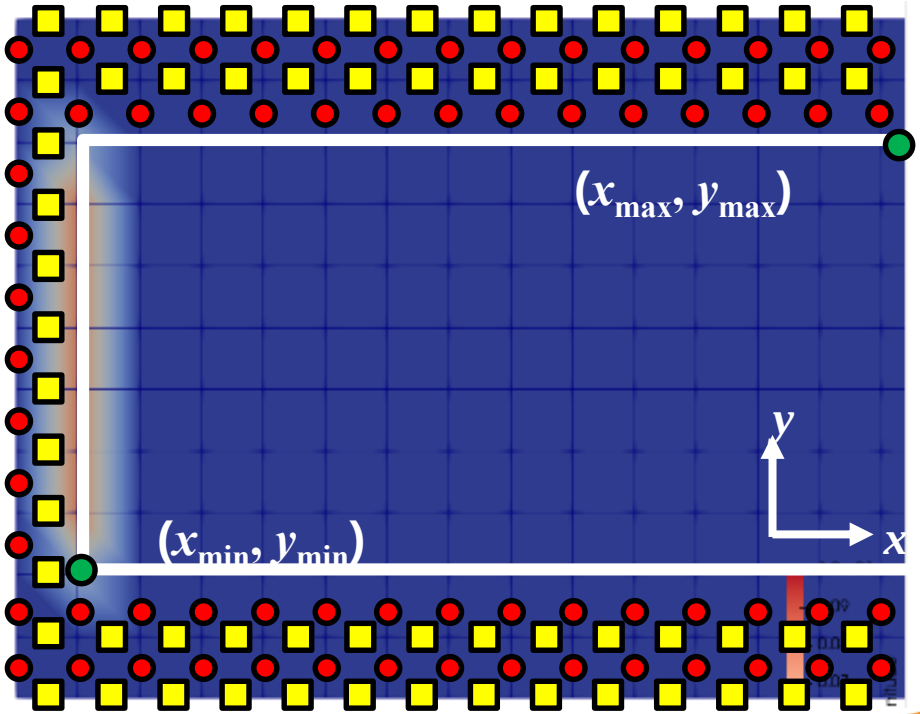
```
<VelocityNodeGroup> FLNGVxPump
    IfInside 1
    VelocityNodeType 1
    Corner1CoordinateX -9.95
    Corner1CoordinateY -0.45
    Corner2CoordinateX -9.85
    Corner2CoordinateY +0.55
</VelocityNodeGroup> FLNGVxPump

<VelocityNodeGroup> FLNGVxBorder
    IfInside 0
    VelocityNodeType 1
    Corner1CoordinateX -9.91
    Corner1CoordinateY -0.59
    Corner2CoordinateX +9.91
    Corner2CoordinateY +0.59
</VelocityNodeGroup> FLNGVxBorder

<VelocityNodeGroup> FLNGVyBorder
    IfInside 0
    VelocityNodeType 2
    Corner1CoordinateX -9.89
    Corner1CoordinateY -0.59
    Corner2CoordinateX +9.89
    Corner2CoordinateY +0.59
</VelocityNodeGroup> FLNGVyBorder
```

Inside rectangle? flag = 1
Outside rectangle? flag = 0

***x*-dir. nodes ? flag = 1**
***y*-dir. nodes? flag = 2**



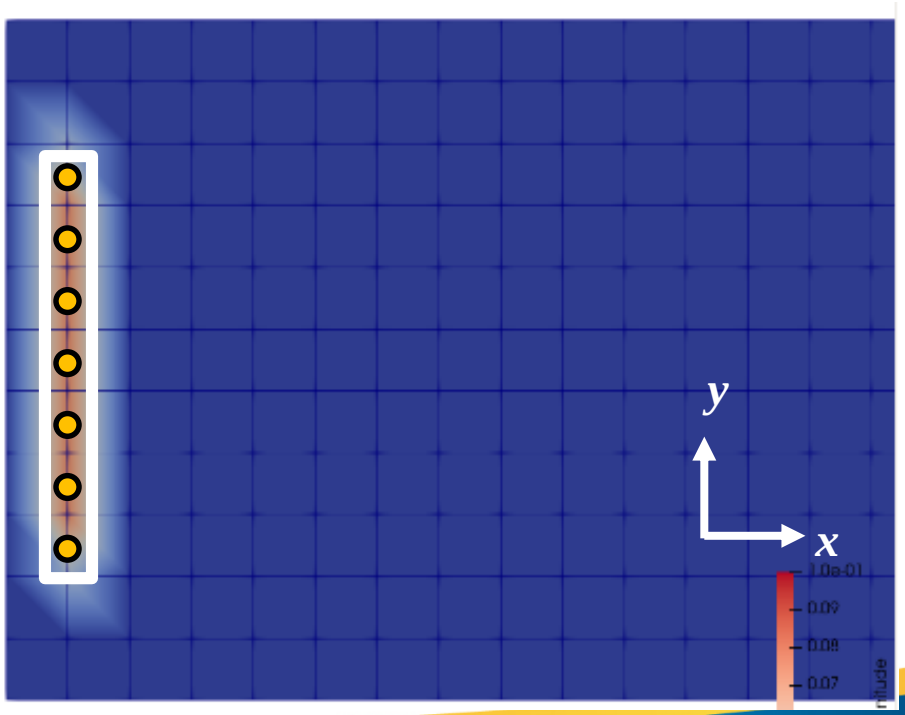
UNCLASSIFIED

fmesh.dat – node groups (cont.)

```
<VelocityNodeGroup> FLNGVxPump
    IfInside 1
    VelocityNodeType 1
    Corner1CoordinateX -9.95
    Corner1CoordinateY -0.45
    Corner2CoordinateX -9.85
    Corner2CoordinateY +0.55
</VelocityNodeGroup> FLNGVxPump
```

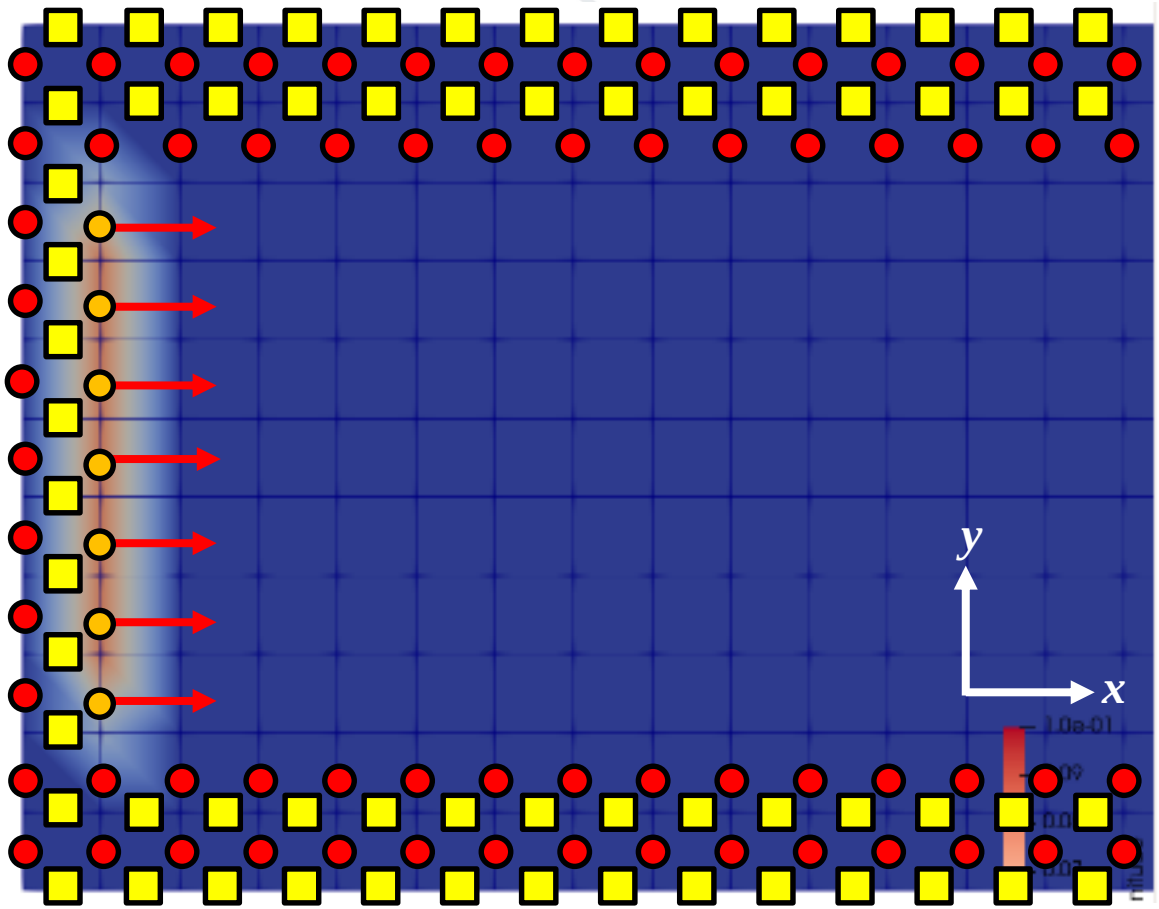
Inside rectangle? flag = 1
Outside rectangle? flag = 0

***x*-dir. nodes ? flag = 1**
***y*-dir. nodes? flag = 2**

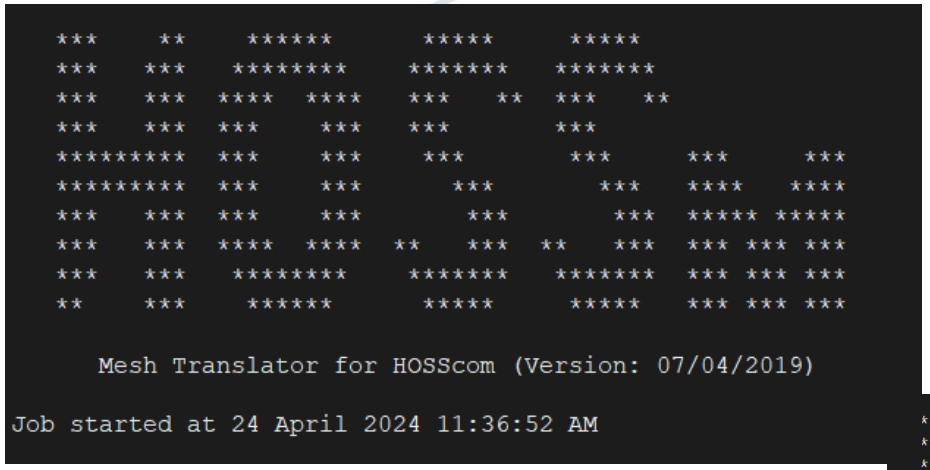


Node groups

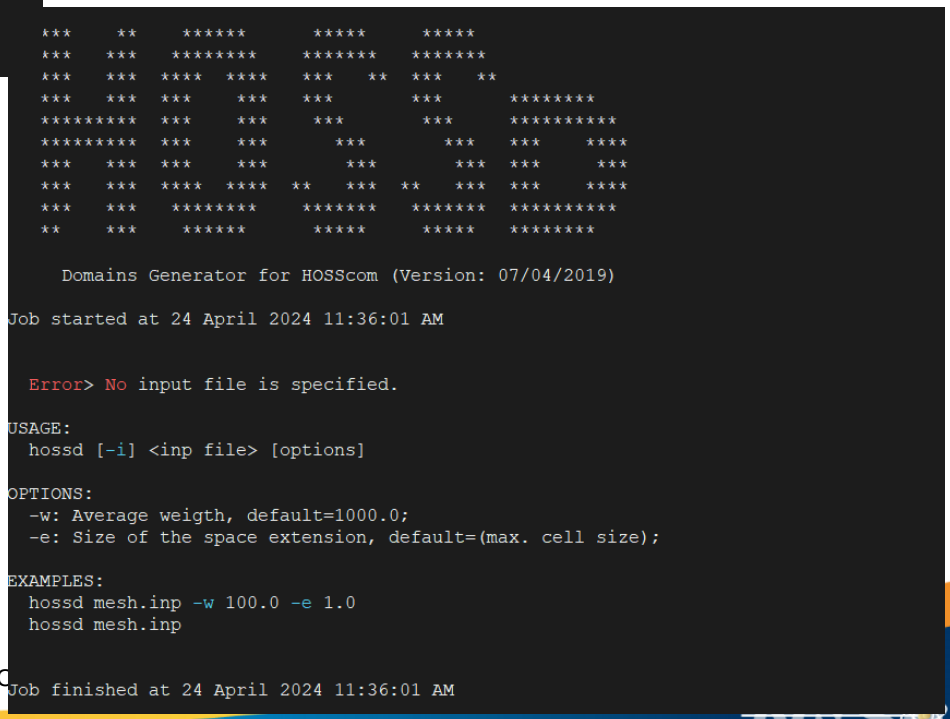
- The behavior of these is identical to that of node groups for solids:
  - One can define the time-history behavior of the fluid
- For this example, the v_x and v_y nodes associated with the border are set to 0 for all time
- Mass is pumped into the system by the v_x pump nodes @ a constant rate



Generate the *MPIDomains.input* and *mesh.input* using executables *hossd* and *hossm*



The next slide breaks down what each executable outputs and what input you use with the executables.



Operated Triad National Security, LLC for the U.S. Department of Energy's NNSA

LA-UR-20-29964

HOSS Executable and Their Outputs

For Solid Only:

Path	Command	Output
/usr/projects/packages/lei/tools/**hossd**	**HOSSd** mesh.inp	MPIDomains.input
usr/projects/packages/lei/tools/**hossm**	**HOSSm** mesh.inp	mesh.input

For Fluid and Solid:

Path	Command	Output
/usr/projects/packages/lei/tools/**hossd**	**HOSSd** mesh.inp	MPIDomains.input (only need Maximum Buffer Zone Size and Domain Initial Buffer Zone from MPIDomains.input file)
/usr/projects/packages/erougier/HOSS_FSIS_2020/chintel/**HOSSm**	**HOSSm** mesh.inp --fsis	mesh.input (with fsis tag)
/usr/projects/packages/erougier/HOSS_FSIS_2020/chintel/**HOSSfmNoCubit**	**HOSSfm** fmesh.dat --vtu	fmeshgroups.input, fmesh.input (the 'vtu' tag also outputs FMesh.vtu)
N/A: Place script in your geometry folder	Run GenerateMPIDomainsFluid.py script	MPIDomains.input (these are manually built with the solid and fluid domain size and Maximum Buffer Zone Size and Domain Initial Buffer Zone from HOSSd MPIDomains.input)

Generate MPI domains using the python script ‘GenerateMPIDomainsFluid.py’

- Generating MPI domains for problems involving the FSIS require two steps now:
  - First, the user must use the **hossd** executable on the mesh.inp associated from the solid.
  - This will provide the user with values for ‘Maximum Buffer Zone Size’ and ‘Domain Initial Buffer Zone’.
  - Once obtained, the user will use the python script to create the MPI domains.

- This example includes a solid domain that does not interact with the fluid (so the code can run).

- One then defines:
  - A pair of coordinates that define opposing sides of a rectangle the encompasses the fluid domain (dmincx, dmincy, etc.)
  - The maximum buffer zone size (dbuffmax)
  - The domain initial buffer zone size (dbuffmin)
  - The numbers of MPI domains they would like in the *x*, *y*, and *z*-directions (ncx, ncy, ncz) in the script.

Now piece together the *xxx.input* file

- Define the `<Problem>` and `<Solver>` contexts

Problem Context

```
<Problem>
    Problem Title  = "2DPoiseuilleFlow"
    Problem_Description = "2DPoiseuilleFlow"
    CycleLimit = 500001
    StopTime  = 10000000.0e+0 s
    InitialTimeStep = 5.0000e-06
    MessageFreq = 10000
    OutSolidFreq = 10000
    OutFluidFreq = 10000
    OutputScaleValueFreq = 200
    MemoryFactor = 1.0
    DebugFlag = 0
  . . . (some developer commands, disregard)
</Problem>

<Solver>
   <SKF_2D3CST>  SolidSolverS1
       Cell_Group_Name     =   "N_SOLID"
    </SKF_2D3CST>

    <FKF_2D4CSQ>  FSolver_1
       Cell_Group_Name     =   "FLUIDGROUP"
       NumberOfPhases = 1
       MinimumSpecificInternalEnergy = 1.0e-03
    </FKF_2D4CSQ></Solver>
```

Problem title/description

Number of time steps

Time step

Message and results output frequency

Solver Context

2-D 4-noded constant strain quadrilateral

FKF_2D4CSQ Subcontext

Continue to piece together the *xxx.input* file…

- Define the <Materials> contexts

```
<Materials>
  <EOS_HossMatLib_Elastic> EOS
      Cell_Group_Name = "FLUIDGROUP"
      5 1 1 1 1 1
      !-----------------------------------
      Bulk Modulus       =   2.0e+09
      Density            =   1.000e+03
      SpecificEnergy     =   0.0
      SpeedOfSound       =   1.4000e+03
      PhaseId            =   0
   </EOS_HossMatLib_Elastic>
  <VIS_HossMatLib_Newtonian> VisFLCG
      Cell_Group_Name = "FLUIDGROUP"
      3 1 1 1
      !-----------------------------------
      Lambda             = -5.93e-04  !Pa*s
      Viscosity          =  8.90e-04   !Pa*s
      Density            =  1.000e+03
      PhaseId            =  0
   </VIS_HossMatLib_Newtonian>
```

Materials Context

Fluid EOS

Cell group defined in the geometry creation stage

Material properties

Viscous behavior of the fluid

Munjiza viscosities

Continue to piece together the *xxx.input* file…

- Define the `<Materials>` contexts

Cell group defined in the geometry creation stage

```
<DMP_HossMatLib_2DFluid> DMPFLCG
    Cell_Group_Name = "FLUIDGROUP"
    6 1 1 1 1 1 1
    !-----------------------------------
    a                   =  0.1
    b                   =  0.75
    Density             =  1.000e+03
    SpecificEnergy      =  2.0674763e+05
    SpeedOfSound        =  1.4000e+03
    PhaseId             =  0
  </DMP_HossMatLib_2DFluid>
<Materials>
```

Damping behavior of the fluid

Material properties

Piecing together the *xxx.input* file continued…

Include Context

```
<INCLUDE>
  FileName =  GEOMETRY/MPIDomains.input
  Format = ASCII
</INCLUDE>

<INCLUDE>
  FileName = GEOMETRY/mesh.input
  Format = ASCII
</INCLUDE>
<INCLUDE>
  FileName = GEOMETRY/fmesh.input
  Format = ASCII
</INCLUDE>

<INCLUDE>
  FileName=bc.input
  Format=ASCII
</INCLUDE>

<FLSENSORout>
  Number of Sensors = 2
  Sensor Coordinates =
    1  0.500000      0.000000
    2  19.50000      0.000000
</FLSENSORout>
```

File that defines the MPI Domains

File that describes the fluid mesh

File to describes the boundary conditions

Sensors for recording behavior at specified points in the fluid

UNCLASSIFIED

Defining the boundary conditions in the *bc.input* file

```
<Boundary Conditions>
 <FLNodalVelocity> FLNGVxBorder
    Group = FLNGVxBorder
    Is_Velocity_Fixed = 1
    Velocity_Fixed = 0.0
    NPointFactor = 2
    InitialTimeFactor = 0.0
    FinalTimeFactor = 0.0
    DefaultValueLeftFactor = 0.0
    DefaultValueRightFactor = 0.0
    Factor = 0.0 0.0
  </FLNodalVelocity>

  <FLNodalVelocity> FLNGVyBorder
    Group = FLNGVyBorder
    Is_Velocity_Fixed = 1
    Velocity_Fixed = 0.0
    NPointFactor = 2
    InitialTimeFactor = 0.0
    FinalTimeFactor = 0.0
    DefaultValueLeftFactor = 0.0
    DefaultValueRightFactor = 0.0
    Factor = 0.0 0.0
  </FLNodalVelocity>
<Boundary Conditions>
```

Defined in the geometry creation

Is the velocity fixed?
flag = 1
Is the velocity free?
flag = 0

Velocities applied to nodeset

Defining the boundary conditions in the *bc.input* file (cont.)

```
<Boundary Conditions>
 <FLNodalVelocity> FLNGVxPump
    Group = FLNGVxPump
    Is_Velocity_Fixed = 1
    Velocity_Fixed = 5.0
    NPointFactor = 2
    InitialTimeFactor = 0.0
    FinalTimeFactor = 0.0
    DefaultValueLeftFactor = 1.0
    DefaultValueRightFactor = 1.0
    Factor = 1.0 1.0
  </FLNodalVelocity>
<Boundary Conditions>
```

Defined in the geometry creation

Is the velocity fixed?
flag = 1
Is the velocity free?
flag = 0

Inlet 'pump' velocity

Describes the behavior of the velocity as a function of time.

As defined, the velocity in this context is constant for all time

LA-UR-20-29964

Defining the boundary conditions in the *bc.input* file (cont.)

```
<Boundary Conditions>
  <FLDensity> DFLCGBorder
    Group = FLCGBorder
    Is_Density_Fixed = 1
    Density_Fixed = 1.0e+03
    Number_Of_Factor_Points = 2
    InitialTimeFactor = 0.0
    FinalTimeFactor = 0.0
    DefaultValueLeftFactor = 1.0
    DefaultValueRightFactor = 1.0
    Factor = 1.0 1.0
  </FLDensity>
<Boundary Conditions>
```

Defined in the geometry creation

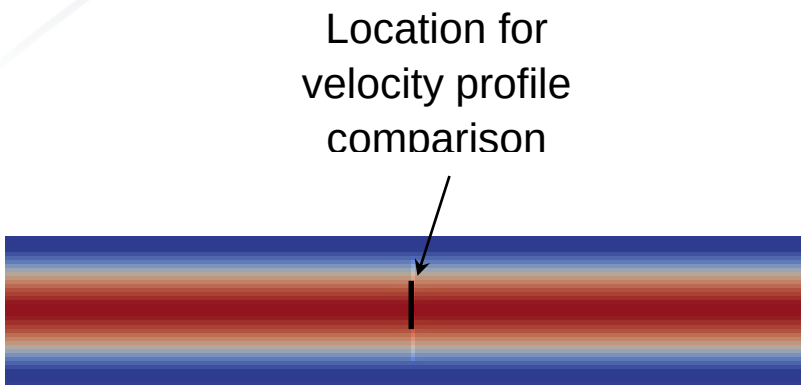
Is the velocity fixed?
flag = 1
Is the velocity free?
flag = 0

Fluid density of the border cells

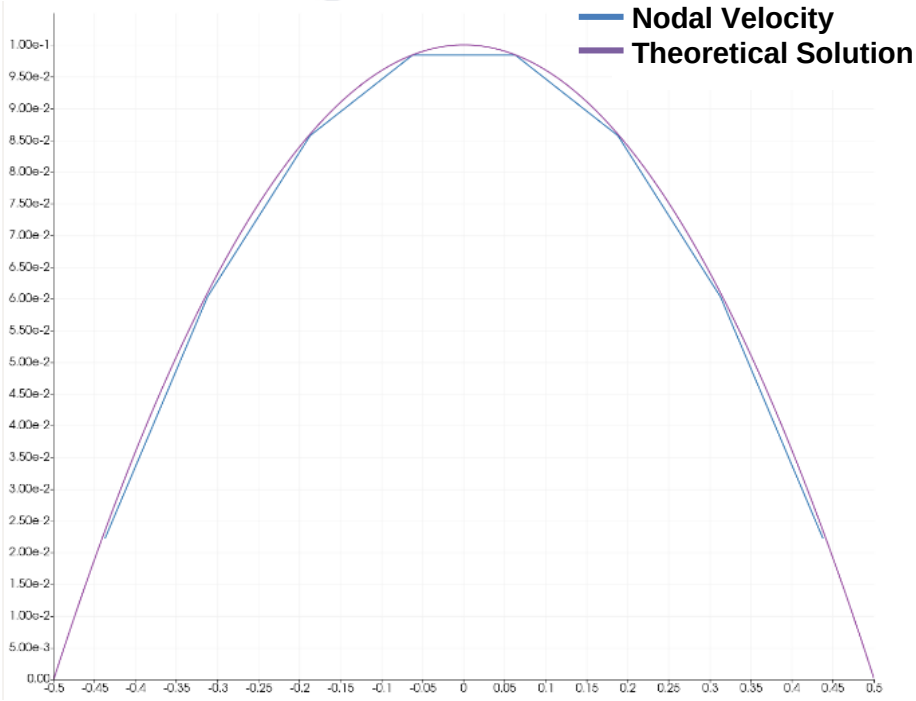
Describes the behavior of the density as a function of time.

As defined, the density in this context is constant for all time

Run the executable… once finished, the results can be visualized with Paraview



Velocity contour along the pipe



Velocity profile along a vertical line located at the middle of the pipe

LA-UR-20-29964

The End!
Any questions?

Operated Triad National Security, LLC for the U.S. Department of Energy's NNSA

LA-UR-20-29964