LA-UR-24-26807
Approved for public release; distribution is unlimited.

Title: HOSS-FSIS example: Energetic Event

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

Intended for: Training material for external customers

Issued: 2024-07-08

HOSS-FSIS example: Energetic Event

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

July 2024

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

LA-UR-20-29965

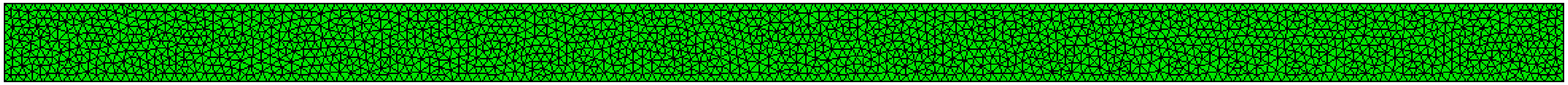
Create input files that represent an energetic event that interacts with a fixed solid

LA-UR-20-29965

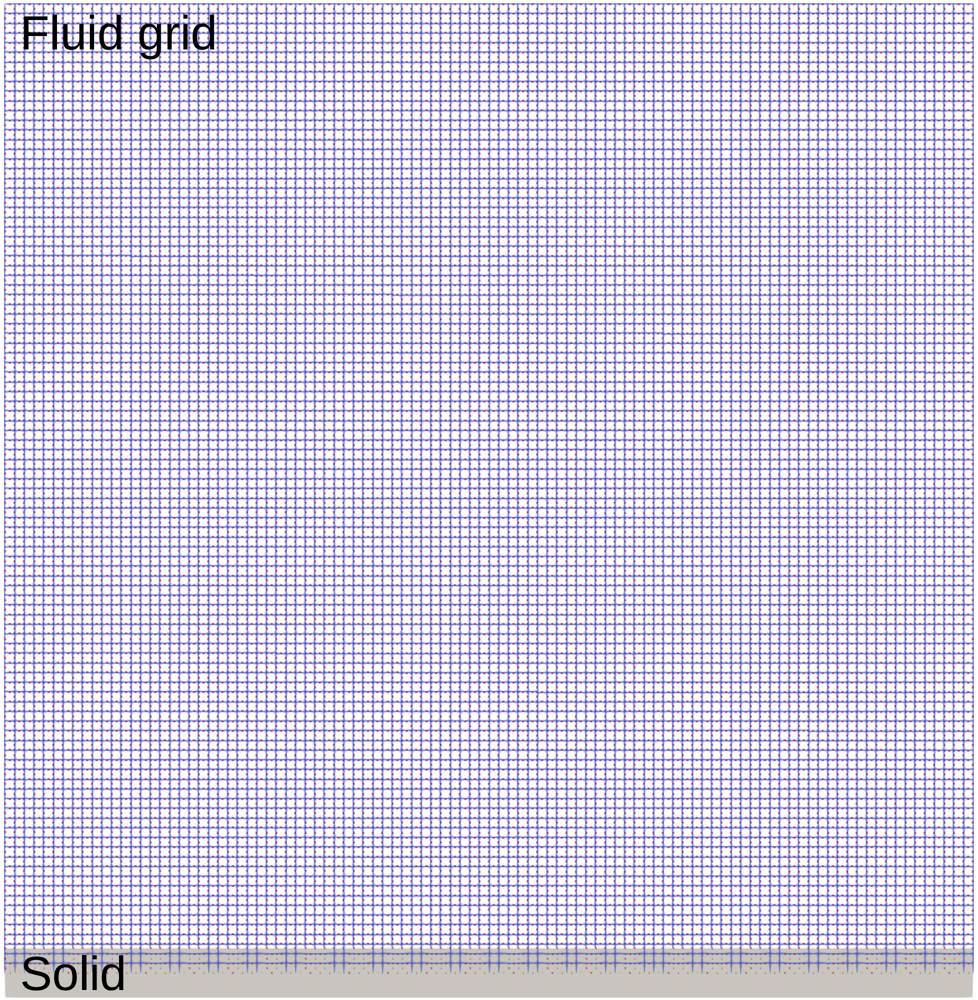
Create the solid geometry and mesh in Cubit

```
reset

create surface rectangle width 40.0 height 2.0
zplane
move vertex 3 y {-1.0}

surface all scheme tridelaunay
tridelaunay point placement gq
surface all size {1.0/4}
mesh surface all

surface all smooth scheme edge length
smooth surface all

# Nodeset & Sideset
nodeset 1 surface 1
nodeset 1 name 'fixed'

block 1 surface 1
block 1 name 'n_rock'

# export
set abaqus precision 15
export abaqus "mesh.inp"  all overwrite  everything
```

This solid mesh is a simple strip of rock material



Visualizing the FSIS set up



- The fluid is a uniform grid which must created using the fmesh.dat file.
- There should be a minimum of a 4 cell overlap between the fluid mesh and solid mesh to ensure proper fluid-solid interactions.

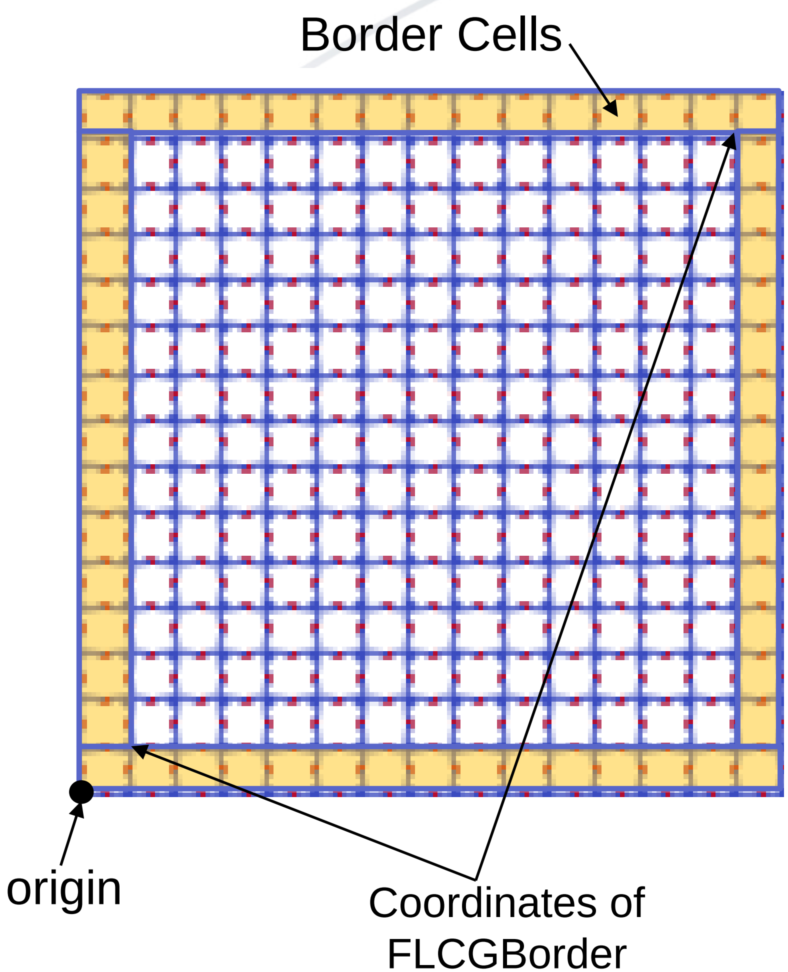
fmesh.dat

- Defines the fluid group by specifying border cells
- Defines node groups for the v_x and v_y nodes associated with the border

```
<Dimension>
          2
</Dimension>

<RegularMeshDomain> FLUIDGROUP
    MeshOriginCoordinateX -2.000000E+01
    MeshOriginCoordinateY -1.000000E+00
    CellSize 0.2
    NumberOfCellsInX  200
    NumberOfCellsInY  200
</RegularMeshDomain>

<FLCellGroupBC> FLCGBorder
    IfInside 0
    Corner1CoordinateX -1.980000E+01
    Corner1CoordinateY -8.000000E-01
    Corner2CoordinateX 1.980000E+01
    Corner2CoordinateY 3.880000E+01
</FLCellGroupBC> FLCGBorder
```

```
<VelocityNodeGroup> FLNGVxBorder
    IfInside 0
    VelocityNodeType 1
    Corner1CoordinateX -1.982000E+01
    Corner1CoordinateY -7.800000E-01
    Corner2CoordinateX 1.978000E+01
    Corner2CoordinateY 3.882000E+01
</VelocityNodeGroup> FLNGVxBorder

<VelocityNodeGroup> FLNGVyBorder
    IfInside 0
    VelocityNodeType 2
    Corner1CoordinateX -1.982000E+01
    Corner1CoordinateY -7.800000E-01
    Corner2CoordinateX 1.978000E+01
    Corner2CoordinateY 3.882000E+01
</VelocityNodeGroup> FLNGVyBorder
```

Making the fmesh.dat




```
<Dimension>
           2
</Dimension>

<RegularMeshDomain> FLUIDGROUP
    MeshOriginCoordinateX -2.000000E+01
    MeshOriginCoordinateY -1.000000E+00
    CellSize 0.2
    NumberOfCellsInX  200
    NumberOfCellsInY  200
</RegularMeshDomain>

<FLCellGroupBC> FLCGBorder
    IfInside 0
    Corner1CoordinateX -1.980000E+01
    Corner1CoordinateY -8.000000E-01
    Corner2CoordinateX 1.980000E+01
    Corner2CoordinateY 3.880000E+01
</FLCellGroupBC> FLCGBorder
```

(x,y) coordinates of the origin

Size of the fluid cells

Number of cells in each direction.

LA-UR-20-29965

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

```
 ***    **    ******      *****     *****
 ***   ***   ********    *******   *******
 ***   ***  ****  ****   ***   **  ***   **
 ***   ***  ***    ***   ***       ***
 *********  ***    ***    ***       ***     ***     ***
 *********  ***    ***      ***       ***   ****   ****
 ***   ***  ***    ***       ***       ***  ***** *****
 ***   ***  ****  ****  **   ***  **   ***  *** *** ***
 ***   ***   ********    *******   *******  *** *** ***
 **    ***    ******      *****     *****   *** *** ***

    Mesh Translator for HOSScom (Version: 07/04/2019)

Job started at 24 April 2024 11:36:52 AM
```

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

```
***   **   ******     *****    *****
***  ***  ********   *******  *******
***  ***  ****  ****  ***  **  ***  **
***  ***  ***   ***  ***      ***      ********
********* ***   ***   ***      ***     **********
********* ***   ***     ***      ***   ***   ****
***  ***  ***   ***      ***      ***  ***    ***
***  ***  ****  **** **  ***  **  ***  ***   ****
***  ***  ********   *******  *******  **********
**   ***   ******     *****    *****   ********

  Domains Generator for HOSScom (Version: 07/04/2019)

Job started at 24 April 2024 11:36:01 AM


 Error> No input file is specified.

USAGE:
 hossd [-i] <inp file> [options]

OPTIONS:
 -w: Average weigth, default=1000.0;
 -e: Size of the space extension, default=(max. cell size);

EXAMPLES:
 hossd mesh.inp -w 100.0 -e 1.0
 hossd mesh.inp


Job finished at 24 April 2024 11:36:01 AM
```

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 (see next slide for more info)

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

- Generating MPI domains for problems involving the FSIS require two steps:
  - 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’.

- From the GenerateMPIDomainsFluid.py, one then defines:
  - A pair of coordinates that define opposing sides of a rectangle the encompasses the fluid domain (dmincx, dmincy, etc.)
  - The numbers of MPI domains they would like in the *x*, *y*, and *z*-directions (ncx, ncy, ncz) in the script. The total number of MPI domains should follow the rule of 6000 fluid cells per domain or 2000 solid elements per domain, whichever is greater.
  - The maximum buffer zone size (dbuffmax)
  - The domain initial buffer zone size (dbuffmin)

```
import sys


#*************************************
#*************************************
#          INPUT DATA
#*************************************
#*************************************

dmincx =  -20.5
dmincy =  -2.5
dmincz = -20.0
dmaxcx =   20.5
dmaxcy =   40.5
dmaxcz = +20.0

ncx = 3
ncy = 3
ncz = 1
nMPI = ncx*ncy*ncz

ddx = (dmaxcx-dmincx) / float(ncx)
ddy = (dmaxcy-dmincy) / float(ncy)
ddz = (dmaxcz-dmincz) / float(ncz)

ifAutoBuffer = 1
nintervals = 50
dbuffmax = 0.019851
dbuffmin = 7.07111975582039e-01
```

Now piece together the *xxx.input* file

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

```
<Problem>
    Problem Title  = "2DEnergeticEvent"
    Problem_Description = "2DEnergeticEvent"
    CycleLimit = 15001
    StopTime  = 1000.0e+0 s
    InitialTimeStep = 1.0e-07
    MessageFreq = 1000
    OutSolidFreq = 1000
    OutFluidFreq = 1000
    OutputScaleValueFreq = 200
    MemoryFactor = 1.0

    . . . (developer commands, ignore)
</Problem>

<Solver>
  <SKF_2D3CST>  SolidSolverS1
      Cell_Group_Name     =   "N_ROCK"
  </SKF_2D3CST>
  <FKF_2D4CSQ>  FSolver_1
      Cell_Group_Name     =   "FLUIDGROUP"
  </FKF_2D4CSQ>
</Solver>
```

Problem title/description

Number of time steps

Time step

Message and results output frequency

Solver Context

FKF_2D4CSQ Subcontext

Now piece together the *xxx.input* file

- Define the <Initial_Conditions> context

```
<Initial_Conditions>
  <FLInitialSpecificEnergy> MyInitSpecEnergy
    ShapeType = CIRCLE
    Is_Specific_Energy_Fixed = 1
    NumberOfPhases = 1
    PhaseId = 0
    Symmetry_Flag_Direction_1 = -1
    Symmetry_Flag_Direction_2 = 1
    Specific_Energy_Fixed = 2.04E11
    Center_Coordinate_X = 19.6
    Center_Coordinate_Y =  0.2
    Unit_Vector_1_Coordinate_X = 1.0
    Unit_Vector_1_Coordinate_Y = 0.0
    Dimension_In_Direction_1 = 2.0
    Dimension_In_Direction_2 = 1.0
  </FLInitialSpecificEnergy>
</Initial_Conditions>
```

Initial_Conditions Context

Initial spec. energy context

Shape? Circle or rectangle

Symmetry flags

Initial spec. energy

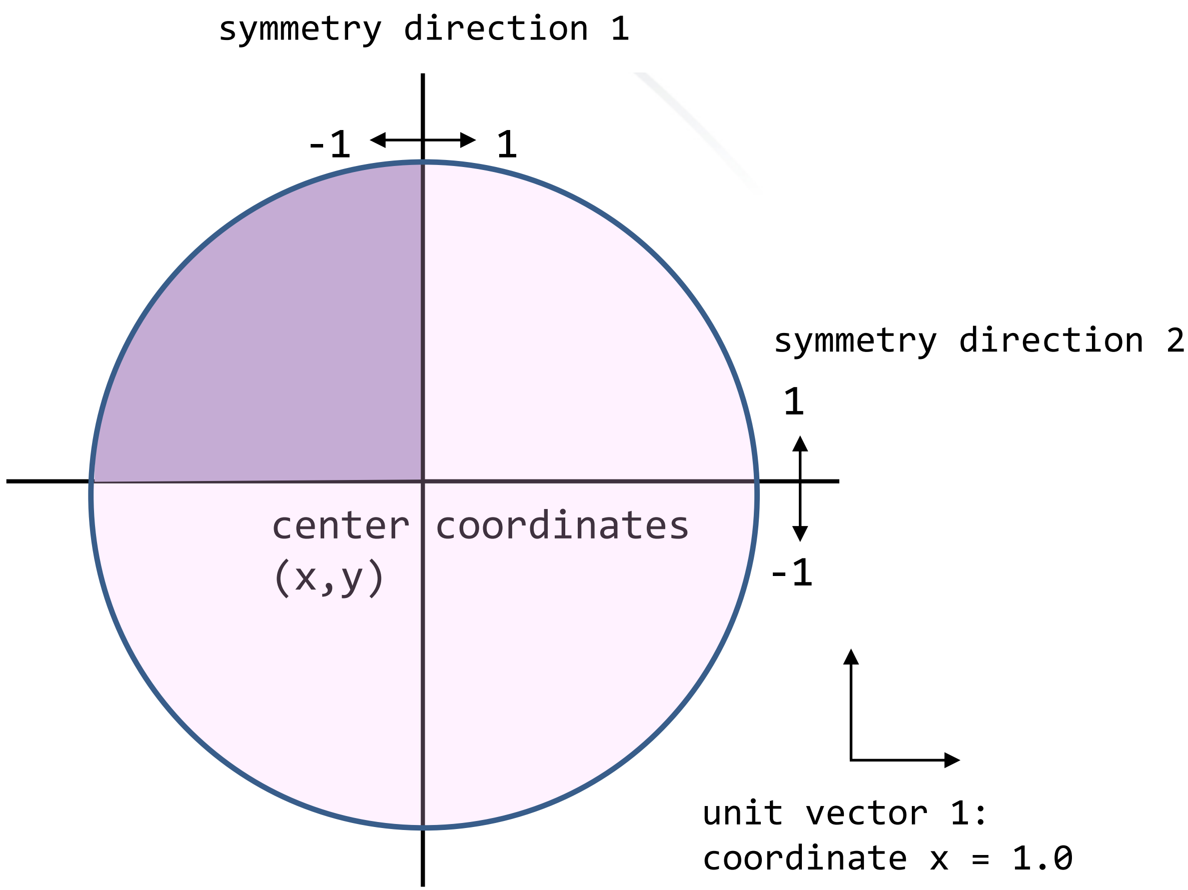
Coordinates of the center of the shape

Unit vectors to specify the local orientation of the shape

Specify the size of the shape

Explaining the <Initial_Conditions> context

```
<Initial_Conditions>
  <FLInitialSpecificEnergy>
    ShapeType = CIRCLE
    Is_Specific_Energy_Fixed = 1
    NumberOfPhases = 1
    PhaseId = 0
    Symmetry_Flag_Direction_1 = -1
    Symmetry_Flag_Direction_2 = 1
    Specific_Energy_Fixed = 2.04E11
    Center_Coordinate_X = 19.8
    Center_Coordinate_Y =  0.2
    Unit_Vector_1_Coordinate_X = 1.0
    Unit_Vector_1_Coordinate_Y = 0.0
    Dimension_In_Direction_1 = 2.0
    Dimension_In_Direction_2 = 1.0
  </FLInitialSpecificEnergy>
</Initial_Conditions>
```



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

- Define the `<Materials>` contexts

Elastic solid context

```
<Materials>
 <MEL_HossMatLib_2DSolid> Rock
     !----------------------------------!
     !        2D Plane Strain           !
     !----------------------------------!
     ! Young's Modulus:  1.9418000e+09
     ! Poisson's ratio:  3.3000000e-01
     ! Element Size:     1.0000000e+00
     ! Density:          2.0000000e+03
     !----------------------------------
     Cell_Group_Name = "N_Rock"
     13 1 1 1 1 1 1 1 1 1 1 1 1 1
     !----------------------------------
     Density            =  2.0000000e+03
     MunjizaConstantV   =  1.4170588e+09
     MunjizaConstant1   =  1.4600000e+09
     MunjizaConstant2   =  1.4600000e+09
     MunjizaConstant12 =  1.4600000e+09
     !----------------------------------
     MunjizaViscosityV =  0.0000000e+00
     MunjizaViscosity1 =  0.0000000e+00
     MunjizaViscosity2 =  0.0000000e+00
     MunjizaViscosity12=  0.0000000e+00
     !----------------------------------
     dmvi               =  0.0
     dm1i               =  0.0
     dm2i               =  0.0
     dm12i              =  0.0
  </MEL_HossMatLib_2DSolid>
<Materials>
```

This context exists for completeness. We assume the solid is completely rigid in this simulation.

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

- Define the `<Materials>` contexts

Fluid EOS contexts

The fluid behaves as an ideal gas in this simulation

```
<Materials>
 <EOS_HossMatLib_IdealGas> EOSFLCG
     Cell_Group_Name = "FLUIDGROUP"
     8 1 1 1 1 1 1 1 1
     !---------------------------------
     Density            =  1.2250000e+00
     SpecificEnergy     =  2.0674763e+05
     InitialTemperature=  0.0000000e+00
     InitialInternalEnergy =  0.0000000e+00
     InitialPressure        =  0.0000000e+00
     SpecificGasConstant    =  287.058 ! J/kg K
     SpecificHeatConstVol   =  717.5   ! J/kg K
     PhaseId                = 0
 </EOS_HossMatLib_IdealGas>
```

Density of air

Specific energy of air

gas constant

spec. heat

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>
```

File that defines the MPI Domains

File that describes the solid mesh

File that describes the fluid mesh

File to describes the boundary conditions

LA-UR-20-29965

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

Los Alamos NATIONAL LABORATORY EST.1943

This context defines establishes the interaction between the fluid and solid

```
<FSISolver> MyFSISolver1
   SCellGroupName = N_ROCK
   FCellGroupName = FLUIDGROUP
   Is_Solver_Active = 1
   IfQuadraticSearch = 0
   FSISolverType = 0
   Penalty = 1.0e14
 </FSISolver>
```

Defined in the geometry creation

Is the solver active? yes (1) or no (0)

FSISolverType 0 is used when the fluid cells are much smaller than the solid mesh. FSISolverType 1 is used when the fluid cells are larger or similar to the size of the solid mesh.

Penalty for the fluid-solid interaction

Note: The penalty defined in this context does not affect the time step of the simulation

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

```
<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>
```

Defined in the geometry creation

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

Velocities applied to nodeset

As defined, *x*- and *y*-velocities in the border nodes are zero for all time

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

```
<Boundary Conditions>
 <NodalVelocity> fixed
   Group = "FIXED"

   Is_Velocity_X_Fixed = 1
   Is_Velocity_Y_Fixed = 1

   Velocity_Fixed_X = 0.0
   Velocity_Fixed_Y = 0.0

   NPointFactorX = 2
   NPointFactorY = 2
   InitialTimeFactorX = 0.0
   InitialTimeFactorY = 0.0
   FinalTimeFactorX = 1.0
   FinalTimeFactorY = 1.0
   DefaultValueLeftFactorX = 0.0
   DefaultValueLeftFactorY = 0.0
   DefaultValueRightFactorX = 1.0
   DefaultValueRightFactorY = 1.0
   FactorX = 1.0 , 1.0
   FactorY = 1.0 , 1.0
 </NodalVelocity>
<Boundary Conditions>
```

Defined in the geometry creation

Is the velocity fixed? yes (1) or no (0)

Velocities applied to nodeset

As defined, the solid material is completely fixed for all time

LA-UR-20-29965

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

```
<Boundary Conditions>
<FLDensity> FLCGBorder
    Group = FLCGBorder
    Is_Density_Fixed = 1
    Density_Fixed = 1.225
    Number_Of_Factor_Points = 2
    InitialTimeFactor = 0.0
    FinalTimeFactor = 0.0
    DefaultValueLeftFactor = 1.0
    DefaultValueRightFactor = 1.0
    Factor = 1.0 1.0
  </FLDensity>

  <FLSpecificEnergy> FLCGBorder
    Group = FLCGBorder
    Is_SpecificEnergy_Fixed = 1
    Specific_Energy_Fixed = 2.0674763e+05
    Number_Of_Factor_Points = 2
    InitialTimeFactor = 0.0
    FinalTimeFactor = 0.0
    DefaultValueLeftFactor = 1.0
    DefaultValueRightFactor = 1.0
    Factor = 1.0 1.0
  </FLSpecificEnergy>
<Boundary Conditions>
```

Defined in the geometry creation

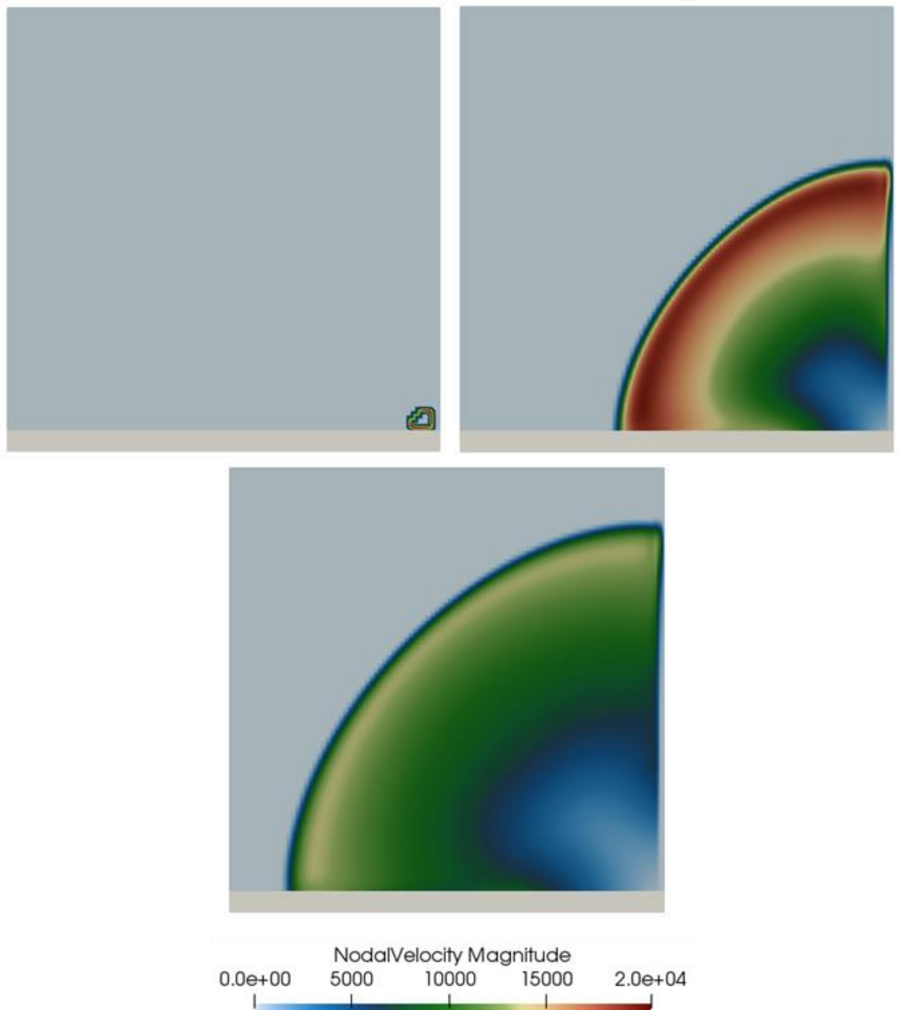
Is the velocity fixed? flag = 1
Is the velocity free? flag = 0

Fluid density of the border cells

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

As defined, the density and specific energy in the border cells are constant for all time

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



UNCLASSIFIED

The End!
Any questions?

LA-UR-20-29965