

Lab 1 GE05015: Linux and OpenFOAM

Modelling wind and dispersion in urban environments

Learning Objectives:

After this class, you will be able to:

- How to run openFOAM in your machines
- Install and run paraView

Activities:

This class is hands-on, so you will need to follow my steps with your own computer, so no activities will be carried out, but I will still ask you questions....

What we will learn might be very new to some, and not so new to others, if you have any doubt it is very important to solve it asap, so you don't get lost

Activities:

This class is hands-on, so you will need to follow my steps with your own computer, so no activities will be carried out, but I will still ask you questions....

What we will learn might be very new to some, and not so new to others, if you have any doubt it is very important to solve it asap, so you don't get lost

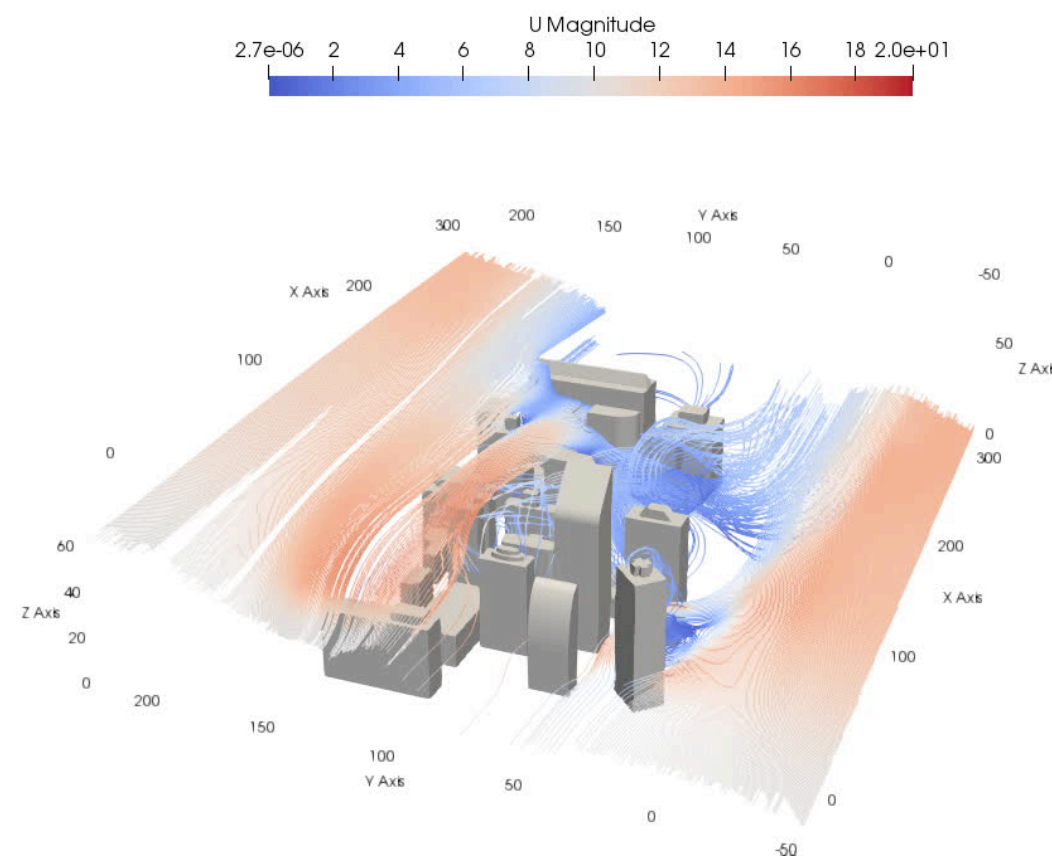
Remember you will use what I show you today, to run your own case for the assignment!!!

Outline of the class:

1. What is openFOAM?
2. openFOAM test case structure
3. How to run openFOAM test case
4. Visualise the results with paraView

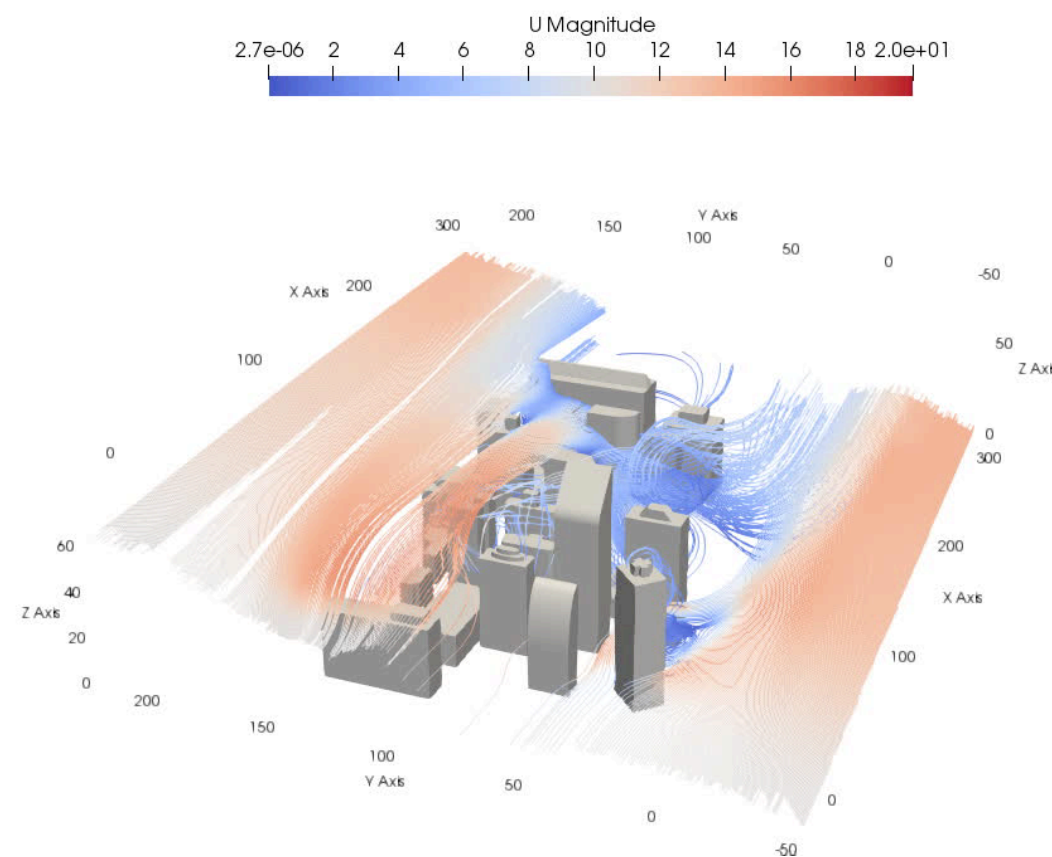
What is openFOAM?

- It is a collection of about 250 applications built from a collection of 100 software libraries (modules)
- Each application performs a specific task within a CFD workflow: snappyHexMesh will generate a mesh for complex geometry for example a car, while simpleFoam will then simulate steady-state, turbulent, incompressible flow around that car.



What is openFOAM?

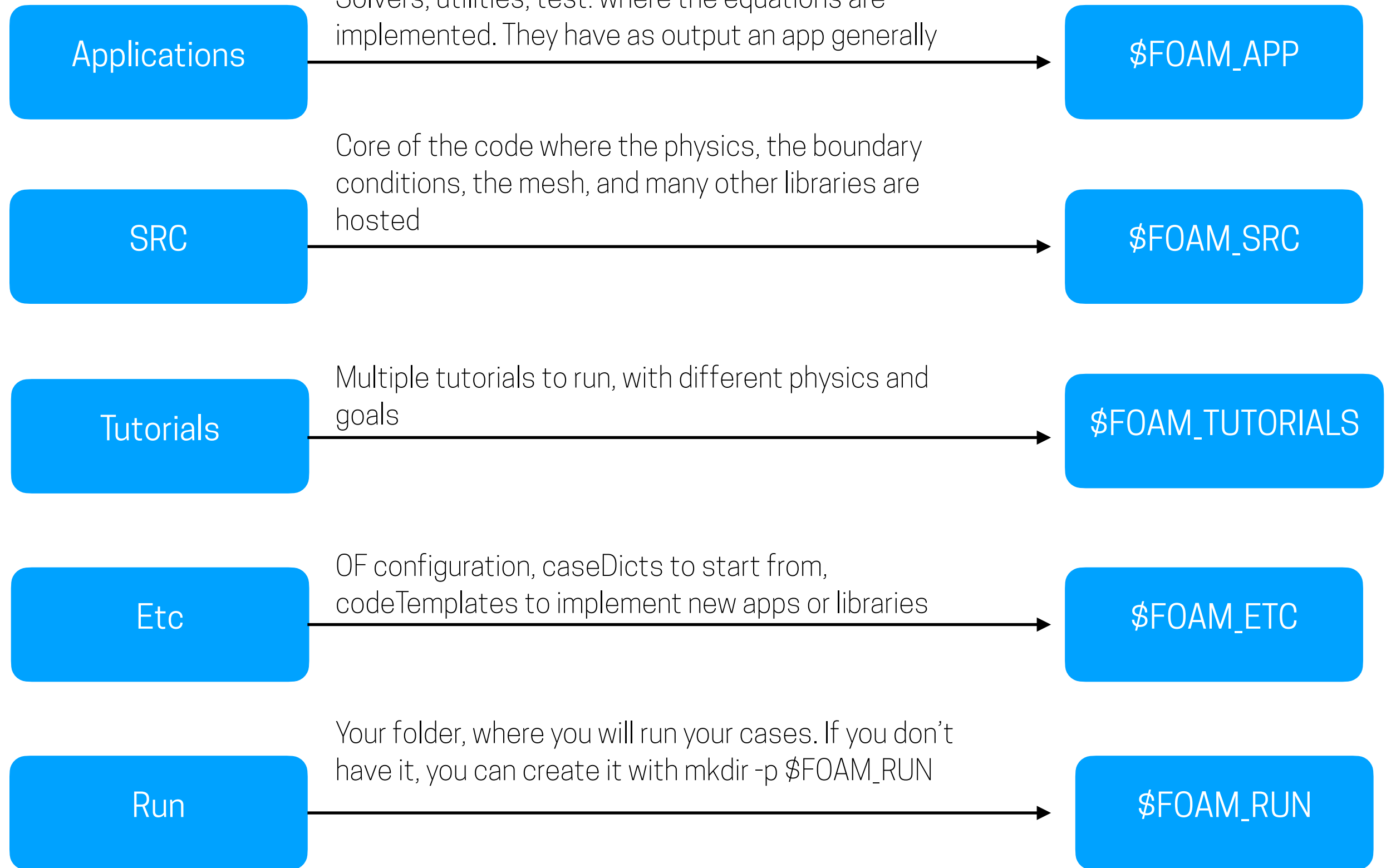
- It is a collection of about 250 applications built from a collection of 100 software libraries (modules)
- Each application performs a specific task within a CFD workflow: snappyHexMesh will generate a mesh for complex geometry for example a car, while simpleFoam will then simulate steady-state, turbulent, incompressible flow around that car.



openFOAM test case structure

Folders tree code:

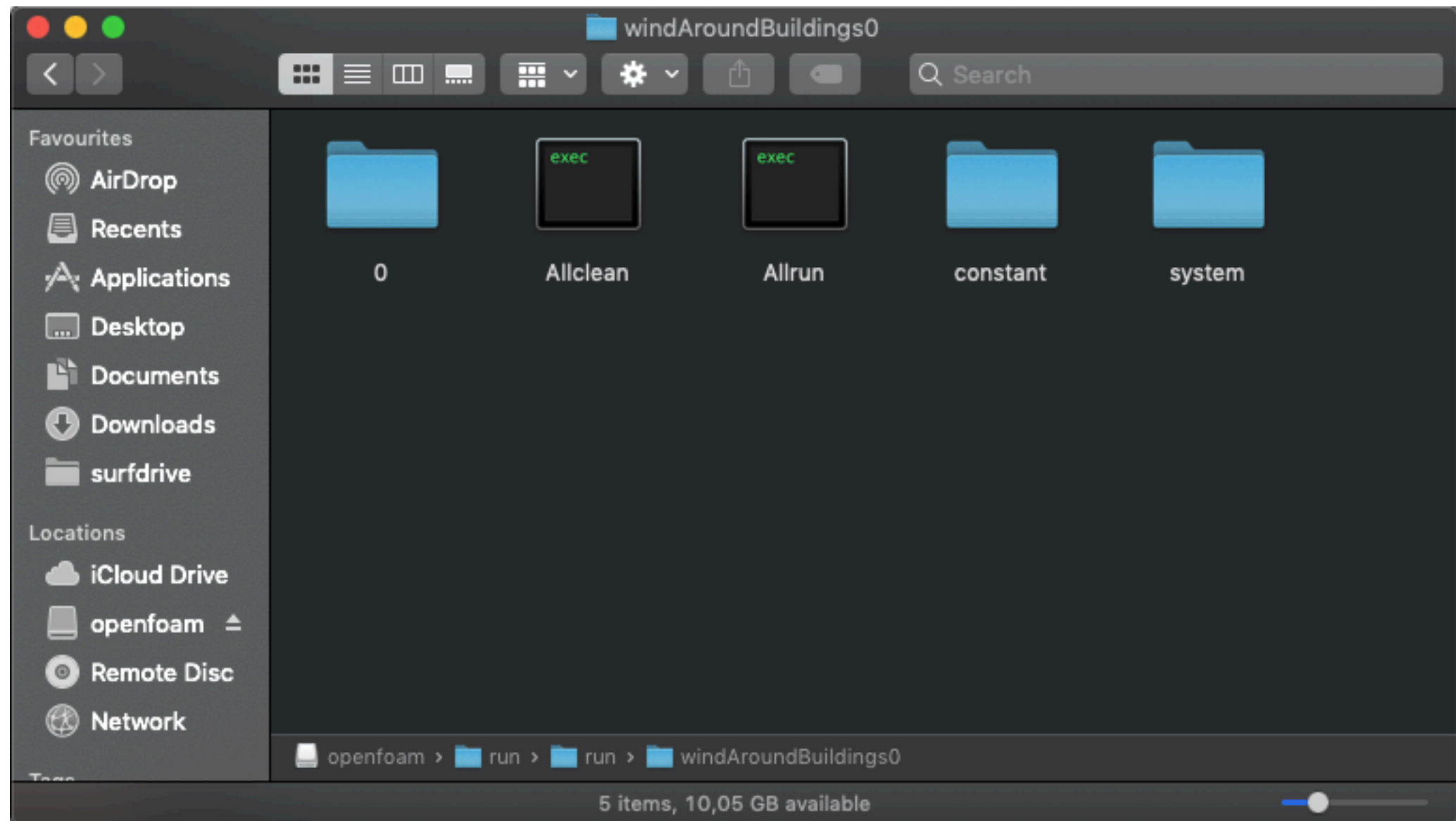
Symbolic path:



openFOAM test case structure

- Folders tree in case:

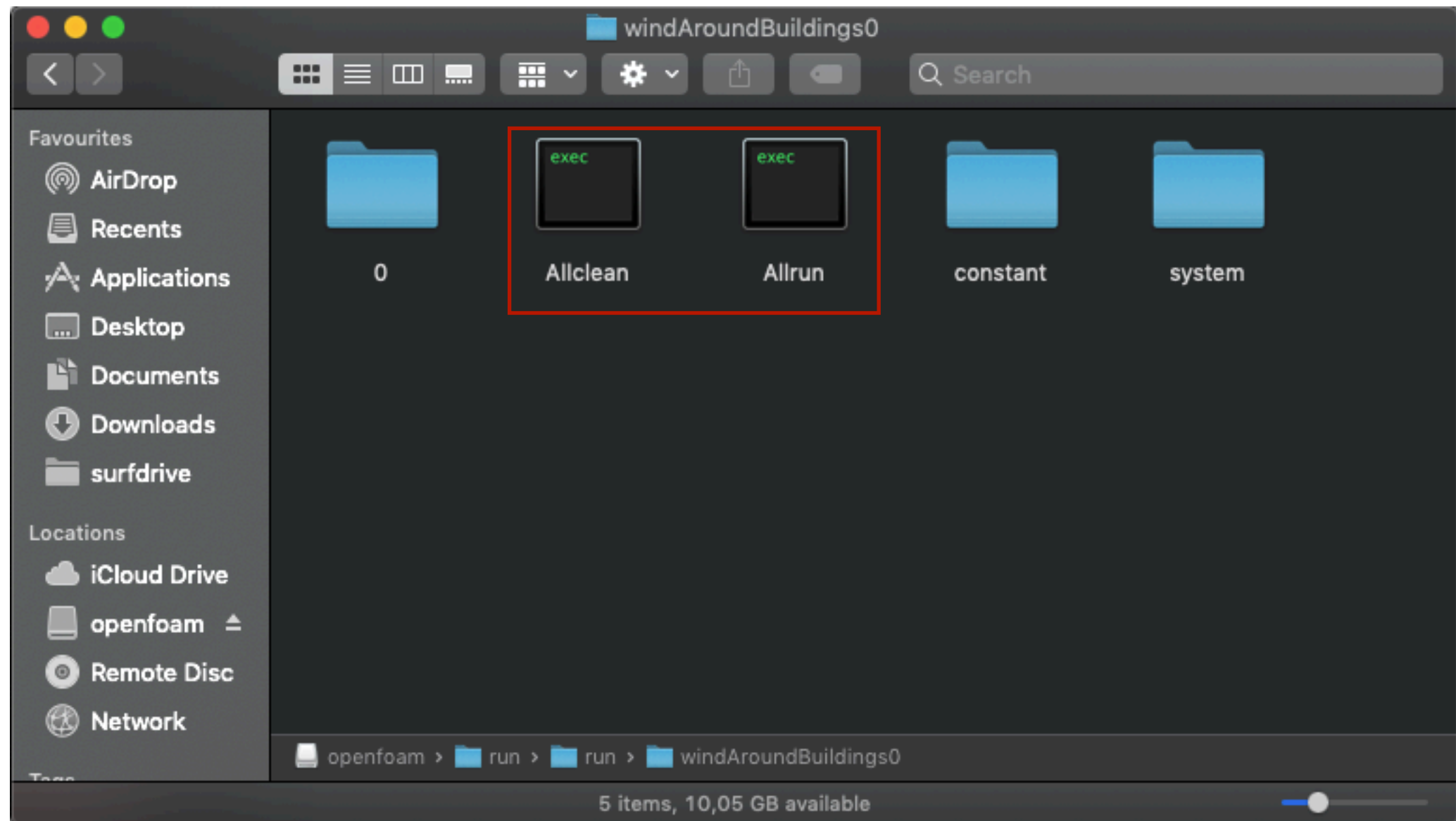
```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/windAroundBuildings $FOAM_RUN/
```



openFOAM test case structure

- Folders tree in case:

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/windAroundBuildings $FOAM_RUN/
```



openFOAM test case structure

- Allrun:

```
#!/bin/sh
cd ${0%/*} || exit 1    # run from this directory

# Source tutorial run functions
. $WM_PROJECT_DIR/bin/tools/RunFunctions

runApplication surfaceFeatures

runApplication blockMesh
runApplication snappyHexMesh -overwrite

runApplication $(getApplication)

#-----
```

openFOAM test case structure

- Allclean:

```
#!/bin/sh
cd ${0%/*} || exit 1    # run from this directory

# Source tutorial clean functions
. $WM_PROJECT_DIR/bin/tools/CleanFunctions

cleanCase

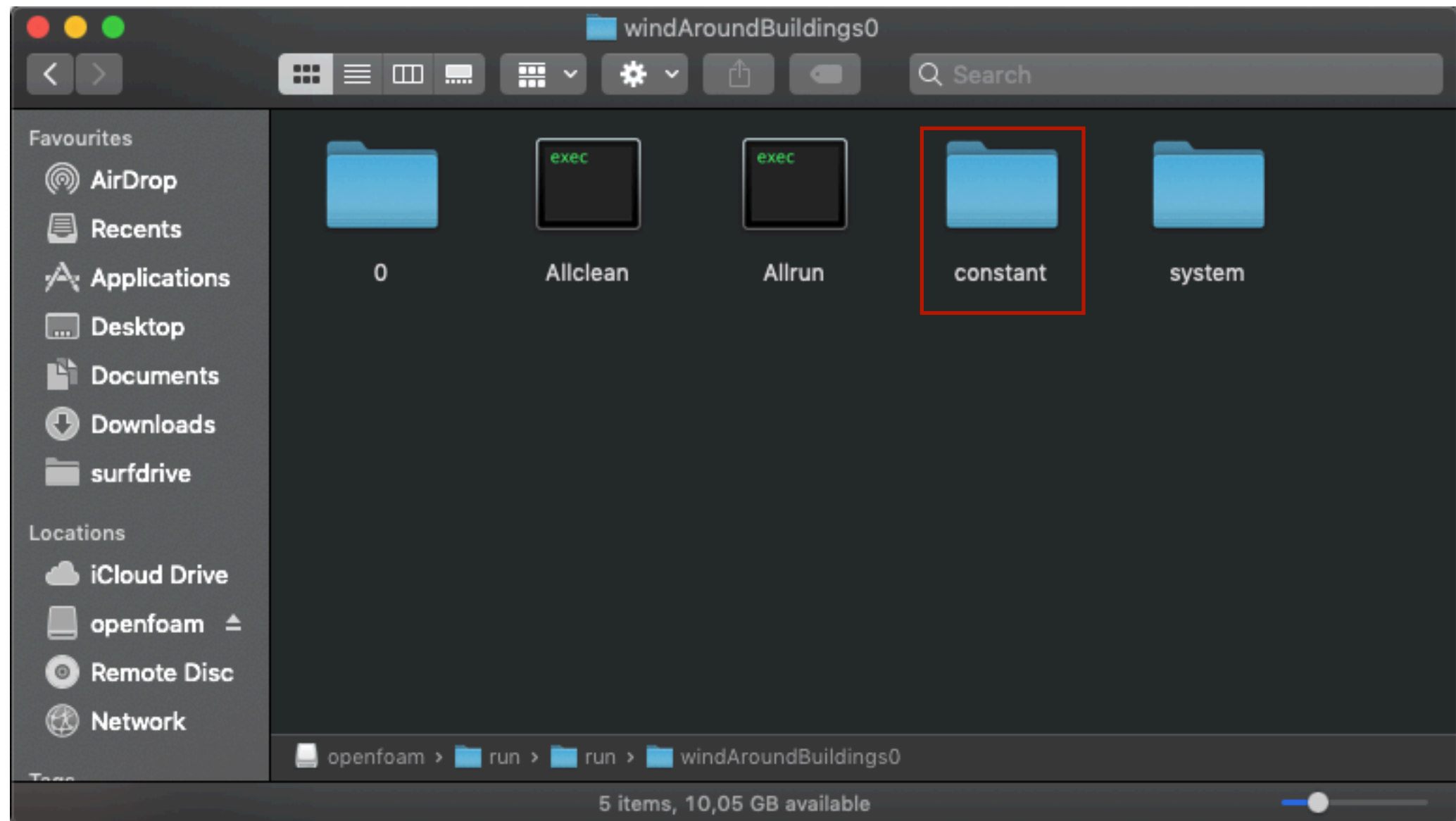
rm -rf constant/extendedFeatureEdgeMesh > /dev/null 2>&1
rm -f constant/triSurface/buildings.eMesh > /dev/null 2>&1
rm -f constant/polyMesh/boundary > /dev/null 2>&1

#-----
```

openFOAM test case structure

- Folders tree in case:

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/windAroundBuildings $FOAM_RUN/
```



openFOAM test case structure

- Constant:

Name	Date Modified
> geometry	13 Apr 2023 at 16:41
momentumTransport	13 Apr 2023 at 16:41
physicalProperties	13 Apr 2023 at 16:41

```
/*-----*- C++ -*-----*\n\n=====\n\\      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox\n\\      / O p e r a t i o n | Website: https://openfoam.org\n\\      / A n d             | Version: 10\n\\      / M a n i p u l a t i o n | \n\n/*-----*/\nFoamFile\n{\n    format      ascii;\n    class       dictionary;\n    object      physicalProperties;\n}\n\n// * * * * *\n\nviscosityModel  constant;\n\nnu              [0 2 -1 0 0 0 0] 1.5e-05;\n\n// *****\n~
```

openFOAM test case structure

- Constant:

Name	Date Modified
> geometry	13 Apr 2023 at 16:41
momentumTransport	13 Apr 2023 at 16:41
physicalProperties	13 Apr 2023 at 16:41

```
/*-----*- C++ -*-----*\
=====
\\      /  F i e l d      | OpenFOAM: The Open Source CFD Toolbox
\\      /  O peration      | Website:  https://openfoam.org
\\      /  A nd            | Version:  10
\\      /  M anipulation    |
\\*-----*/
FoamFile
{
    format      ascii;
    class       dictionary;
    object      RASProperties;
}
// *****

simulationType RAS;

RAS
{
    model        kEpsilon;

    turbulence    on;

    printCoeffs   on;
}
// *****
```


openFOAM test case structure

- Constant:

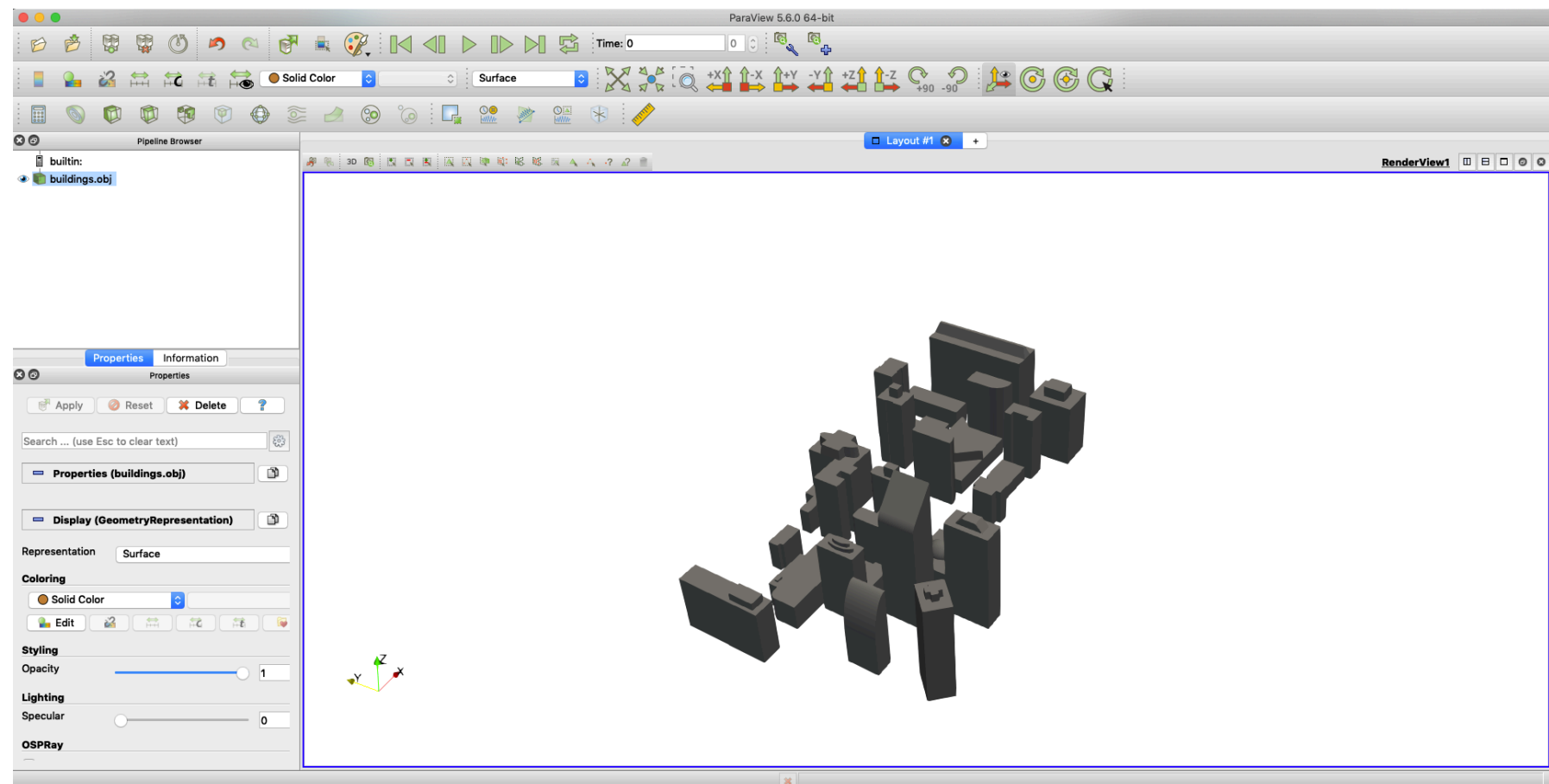
Geometry —>

Name	Date Modified
transportProperties	Yesterday at 17:11
triSurface	Yesterday at 17:11
turbulenceProperties	Yesterday at 17:11

Name	Date Modified
buildings.obj.gz	Yesterday at 17:11

`gzip -d buildings.obj.gz`

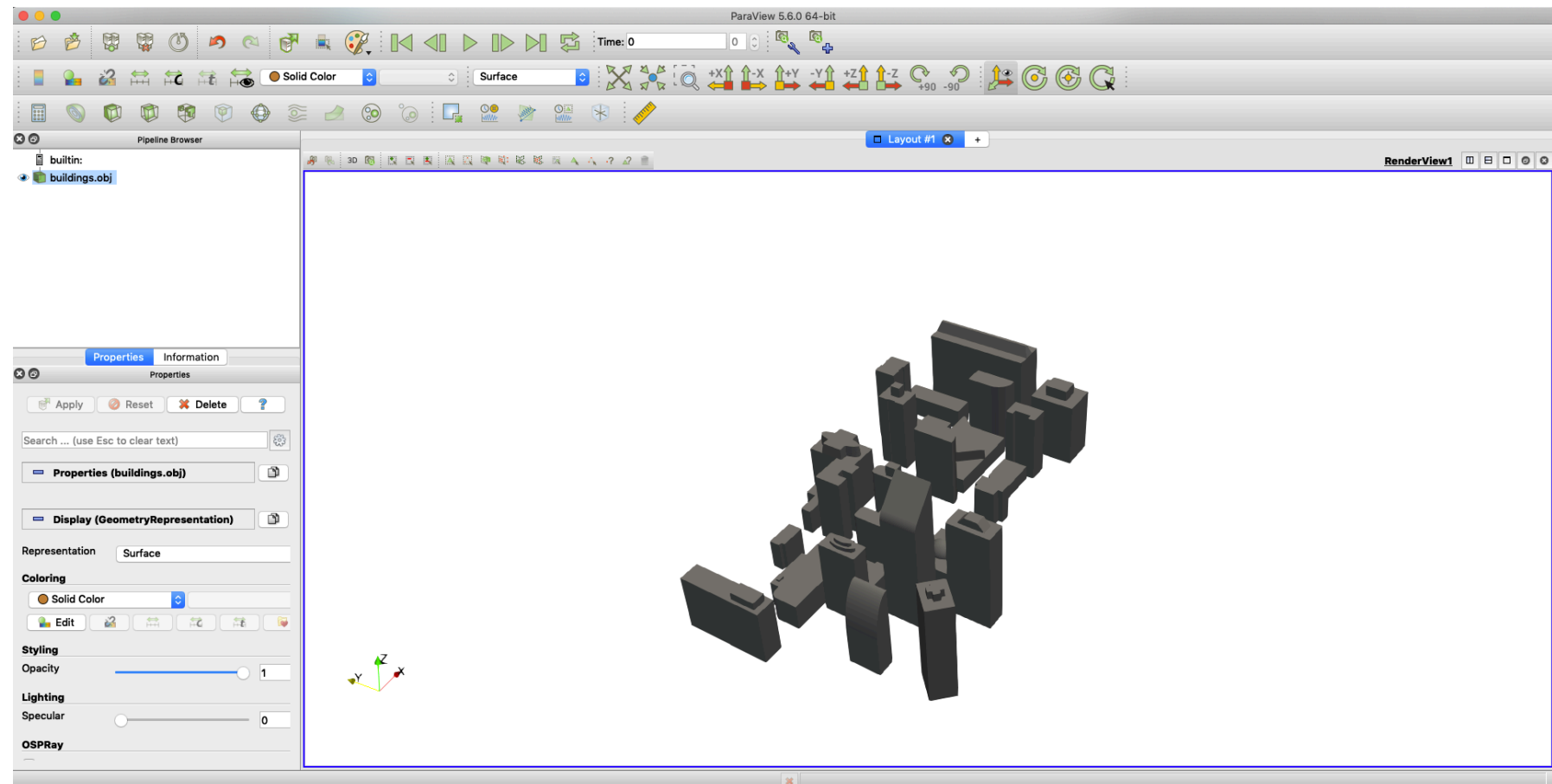
Open buildings.obj with Paraview



Let's install paraView now!

Go to: <https://www.paraview.org/download/>

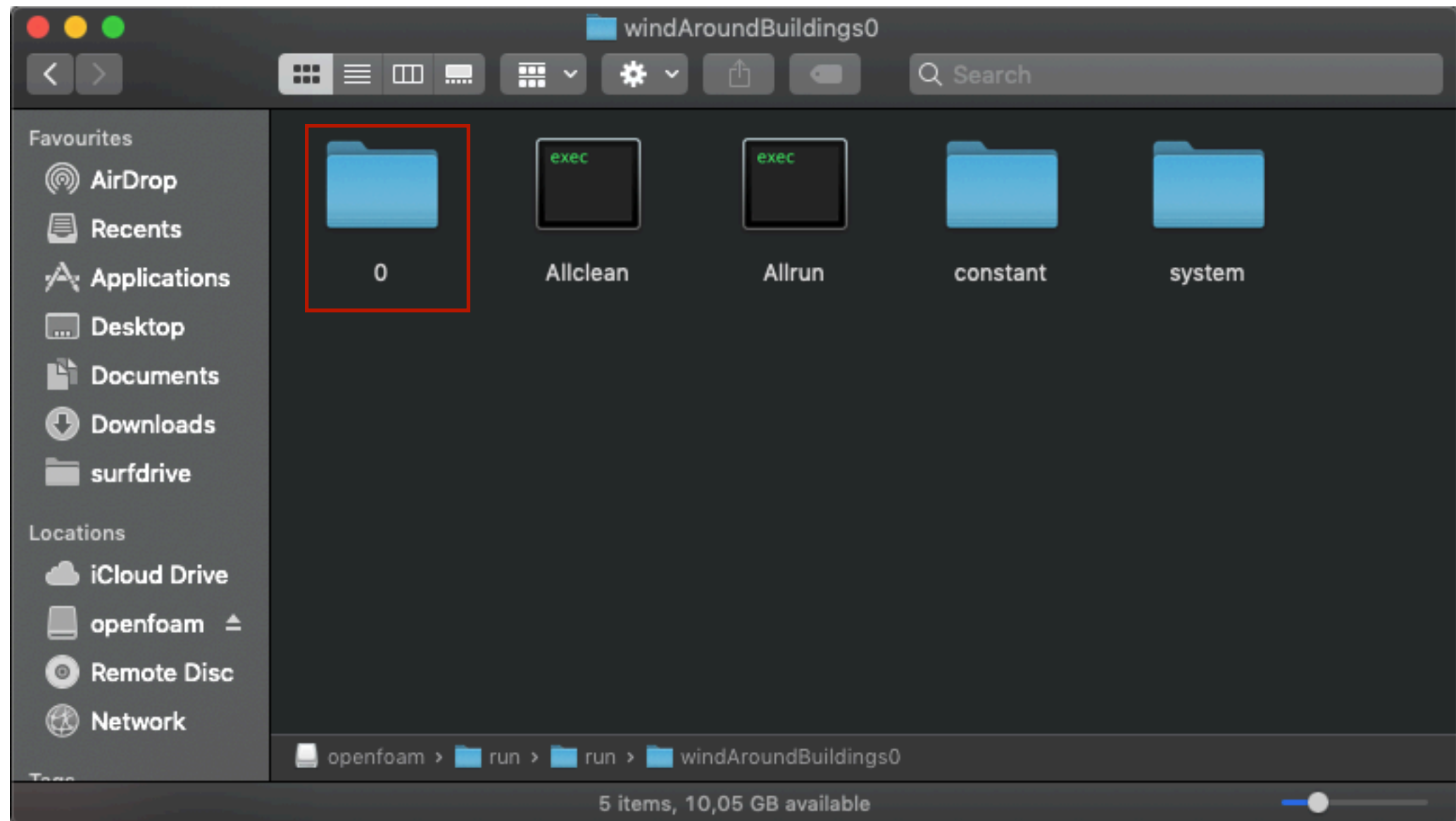
To visualise an openFOAM case you always need a dummy file with any name and .foam extension, like windAroundBuildings.foam



openFOAM test case structure

- Folders tree in case:

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/windAroundBuildings $FOAM_RUN/
```



openFOAM test case structure

- 0: contains the initial and boundary conditions for the fields to be modelled

The basic boundary conditions are:

- fixedValue: value of field is specified
- fixedGradient: value of the normal field gradient is specified
- zeroGradient: normal field gradient is null (0)
- calculated: patch field value is calculated from other patches
- mixed: mixed fixedValue/fixedGradient condition depending on a valueFraction ($0(\text{refGradient}) < \text{vf} < 1(\text{refValue})$)
- directionMixed: mixed conditions with tensorial valueFraction to allow different conditions in normal and tangential direction of a vector patch field (fixedValue in tangential, zeroGradient in normal direction)

simpleFoam -listScalarBCs -listVectorBCs

foamInfo totalPressure

openFOAM test case structure

- 0: U - velocity initial and boundary values:

Name	Date Modified
epsilon	Yesterday at 17:11
k	Yesterday at 17:11
nut	Yesterday at 17:11
p	Yesterday at 17:11
U	Yesterday at 17:11

```
// ***** //
20
21 Supposing you have Ubuntu 18.04 installed, we can download and run the pre-compiled distribution from OpenFOAM 7. Go to the
22 website where you can download the pre-compiled distribution and follow the instructions. Here are some screenshots from the website where you should
23 follow the instructions.
24 1[ScreenShot1](ScreenLinux1.png){ width=45% } 1[ScreenShot2](ScreenLinux2.png){ width=45% }
25 2[ScreenShot3](ScreenLinux3.png){ width=45% }
26
27 Keep in mind that you will need to complete the installation. Within the website there are a list of problems,
28 listed, please email me to report them, and we can go over the issues together.
29 In the materials you can find a video called: _InstallingOF7_Ubuntu.mp4_ where I show step by step how you install openfoam
30
31 boundaryField
32 {
33   inlet
34   {
35     type fixedValue;
36     value uniform $Uinlet;
37   }
38   **What to do if you can't install linux/openfoam**
39
40   outlet
41   {
42     type pressureInletOutletVelocity;
43     value uniform (0 0 0);
44   }
45   1[ScreenShot4](ParaviewDownload.png){ width=100% }
46
47 3.263 Using Paraview:
48 console
49 {
50   type terminal;
51   type R Markdown;
52 }
53
54 type license() for details.
55
56 Natural language support but running in an English locale
57
58 is a collaborative project with many contributors
59 type 'contributors()' for more information and
60 'citation()' on how to cite R or R packages in publications.
61
62 type 'demo()' for some demos, 'help()' for on-line help, or
63 'help.start()' for an HTML browser interface to help.
64 // ***** //
```

openFOAM test case structure

- 0: U - velocity initial and boundary values:

Name	Date Modified
 epsilon	Yesterday at 17:11
 k	Yesterday at 17:11
 nut	Yesterday at 17:11
 p	Yesterday at 17:11
 U	Yesterday at 17:11

```
// *****
20
21 Supposing you have Ubuntu 18.04 installed, we can download and run the pre-compiled distribution from OpenFOAM 7. Go to t
under downloads you choose Ubuntu and follow the instructions. Here are some screenshots from the website where you shoul
22
23 ![Screenshot1](ScreenLinux1.png){ width=45% } ![Screenshot2](ScreenLinux2.png){ width=45% }
dimensions [0 1 -1 0 0 0 0];
24
25
26
27 internalField uniform (0 0 0);
listed, please email me to report them, and we can go over the issues together.\
28 In the materials you can find a video called: _InstallingOF7_Ubuntu.mp4_ where I show step by step how you install openfo
boundaryField
29 **MAC users:**
{
30
31
32 At the moment mac has unix, which is similar to Linux, installing openfoam can be somehow complicated. So we will use docker i
33 1. First install docker for mac following the instructions here: https://docs.docker.com/docker-for-mac/
```

No.	Property	SI unit	USCS unit
1	Mass	kilogram (kg)	pound-mass (lbm)
2	Length	metre (m)	foot (ft)
3	Time	second (s)	second (s)
4	Temperature	Kelvin (K)	degree Rankine (°R)
5	Quantity	mole (mol)	mole (mol)
6	Current	ampere (A)	ampere (A)
7	Luminous intensity	candela (cd)	candela (cd)

```

is a collaborative project with many contributors
#includeEtc "caseDicts/setConstraintTypes"
type 'contributors()' for more information and
} citation()' on how to cite R or R packages in publications.

type 'demo()' for some demos, 'help()' for on-line help, or
//p.start()' for an html browser interface to help.

```

openFOAM test case structure

- 0: p - pressure initial and boundary values:

Name	Date Modified
epsilon	Yesterday at 17:11
k	Yesterday at 17:11
nut	Yesterday at 17:11
p	Yesterday at 17:11
U	Yesterday at 17:11

```
// *****
// Suppose you have Ubuntu 18.04 installed, we can download and run the pre-compiled distribution from OpenFOAM7 //
// under downloads you choose Ubuntu and follow the instructions. Here are some screenshots from the website where you sh
22
dimensions [0 2 -2 0 0 0 0]; ![Screenshot2](ScreenLinux2.png){ width=45% }
24 ![Screenshot3](ScreenLinux3.png){ width=45% }
25
internalField uniform 0;
27 Keep in mind you will need root privilege to complete the installation. Within the website there are a list of problem
listed, please email me to report them, and we can go over the issues together.\
boundaryField
{
29
30 **MAC users:**
31 inlet
32 {
33 Although mac has unix, which is similar to Linux, installing openfoam can be somehow complicated. So we will use docke
34 1. First install docker for mac following the instructions here: https://docs.docker.com/docker-for-mac/ \
35 2. Install openfoam following the instructions here: https://openfoam.org/download/7-macos/
36 }
37 In the materials you can find a video called: _InstallingOF7_MacDocker.mp4_ where I show step by step how you install
2.1.0.4.
37 outlet
38 {
39 **what to do if you can't install linux/openfoam**
40 Although the websites I point to provide information regarding common mistakes, often there are different
install the software before the lab, just email me and book an hour with me so we can check it out together.
41
42 # Using gamma:
43 Paraview is an open-source, multi-platform data-analysis and visualization application. You can install it in Windows,
download the version that corresponds to your operative system (right bottom in the screenshot shown here): https://www.paraview.org/en/download/
44 ![Screenshot4](ParaviewDownload.png){ width=100% }
45
3.263 Using Paraview:
3.263 console Terminal R Markdown
3.263
3.263 type 'license()' for full
3.263
3.263 Natural language support but running in an English locale
3.263
3.263 is a collaborative project with many contributors.
3.263 type 'contributors()' for more information and
3.263 'citation()' on how to cite R or R packages in publications.
3.263
3.263 type 'demo()' for some demos, 'help()' for on-line help, or
3.263 'help.start()' for an HTML browser interface to help.
3.263
3.263 *****
//
```

openFOAM test case structure

- 0: k - tke initial and boundary values:

Name	Date Modified
epsilon	Yesterday at 17:11
k	Yesterday at 17:11
nut	Yesterday at 17:11
...	Yesterday at 17:11
...	Yesterday at 17:11

```

// *****
kInlet      1.5;    // approx k = 1.5*(I*U)^2 ; I = 0.1
dimensions  [0 2 -2 0 0 0 0];
internalField uniform $kInlet;
boundaryField
{
    inlet
    {
        type      fixedValue;
        value      uniform $kInlet;
    }
    outlet
    {
        type      inletOutlet;
        inletValue uniform $kInlet;
        value      uniform $kInlet;
    }
    wall
    {
        type      kqRWallFunction;
        value      uniform $kInlet;
    }
}

#includeEtc "caseDicts/setConstraintTypes" types
}
// *****

```

In continuum mechanics, properties are represented in some chosen units, e. g. pressure in Pascals (N/m²). Algebraic operations must be performed on these particular, addition, subtraction and equality are only physically meaningful for particular dimensions. First implementing a meaningless operation, OpenFOAM attaches dimensions to the scalar. The I/O format for a *dimensionSet* is 7 scalars delimited by square brackets, e. g. [0 2 -2 0 0 0 0].

No.	Property	SI unit
1	Mass	kilogram (kg)
2	Length	metre (m)
3	Time	second (s)
4	Temperature	Kelvin (K)
5	Quantity	mole (mol)
6	Current	ampere (A)
7	Luminous intensity	candela (cd)

Table 4.2: Base units for SI and the United States Customary System (USCS). where each of the values corresponds to the power of each of the base units of mass, length, time, temperature, quantity, current and luminous intensity for the Systeme International (SI) and the United States Customary System (USCS). All that is required is that the *input data is correct for the chosen system of units*. That OpenFOAM requires some dimensioned physical constants, e. g. the Universal Gas Constant, is not a problem for the physical modelling. These dimensioned constants are specified in a *dimensionedScalar* file of the OpenFOAM installation (*\$WM_PROJECT_DIR/etc/constant*). Those wishing to use the USCS or any other system of units should modify these files.

Physical properties are typically specified with their associated dimensions. The example of a *dimensionedScalar* demonstrates:

```

// *****

```


openFOAM test case structure

- 0: ϵ - dissipation initial and boundary values:

Name	Date Modified
epsilon	Yesterday at 17:11
k	Yesterday at 17:11
nut	Yesterday at 17:11
p	Yesterday at 17:11
	Yesterday at 17:11

```
// ***** 4.2.6 Dimensional units ***** //

epsilonInlet 0.03; // Cmu^0.75 * k^1.5 / L ; L=10

dimensions      [0 2 -3 0 0 0 0];

internalField   uniform $epsilonInlet;

boundaryField
{
    inlet
    {
        type      fixedValue;
        value      uniform $epsilonInlet;
    }

    outlet
    {
        type      inletOutlet;
        inletValue uniform $epsilonInlet;
        value      uniform $epsilonInlet;
    }

    wall
    {
        type      epsilonWallFunction;
        value      uniform $epsilonInlet;
    }

    #includeEtc "caseDicts/setConstraintTypes" types
}

// ***** example of a dimensionedScalar demonstrates: ***** //
```


openFOAM test case structure

- 0: $\mathbf{v}_t$ - turbulent viscosity initial and boundary values:

Name	Date Modified
epsilon	Yesterday at 17:11
k	Yesterday at 17:11
nut	Yesterday at 17:11
p	Yesterday at 17:11
U	Yesterday at 17:11

```

// *****
dimensions      [0 2 -1 0 0 0 0];
internalField    uniform 0;
boundaryField
{
    inlet
    {
        type      calculated;
        value      uniform 0;
    }
    outlet
    {
        type      calculated;
        value      uniform 0;
    }
    wall
    {
        type      nutkWallFunction;
        value      uniform 0;
    }
}

#includeEtc "caseDicts/setConstraintTypes" types
}
// *****

```

particular, addition, subtraction and equality are only physically meaningful // p
 against implementing a meaningless operation, OpenFOAM attaches dimensions
 dimension checking on any tensor operation.
 The I/O format for a *dimensionSet* is 7 scalars delimited by square brackets, e.g.

[0 2 -1 0 0 0 0]

No.	Property	SI unit
1	Mass	kilogram (kg)
2	Length	metre (m)
3	Time	second (s)
4	Temperature	Kelvin (K)
5	Quantity	mole (mol)
6	Current	ampere (A)
7	Luminous intensity	candela (cd)

Table 4.2: Base units for SI and

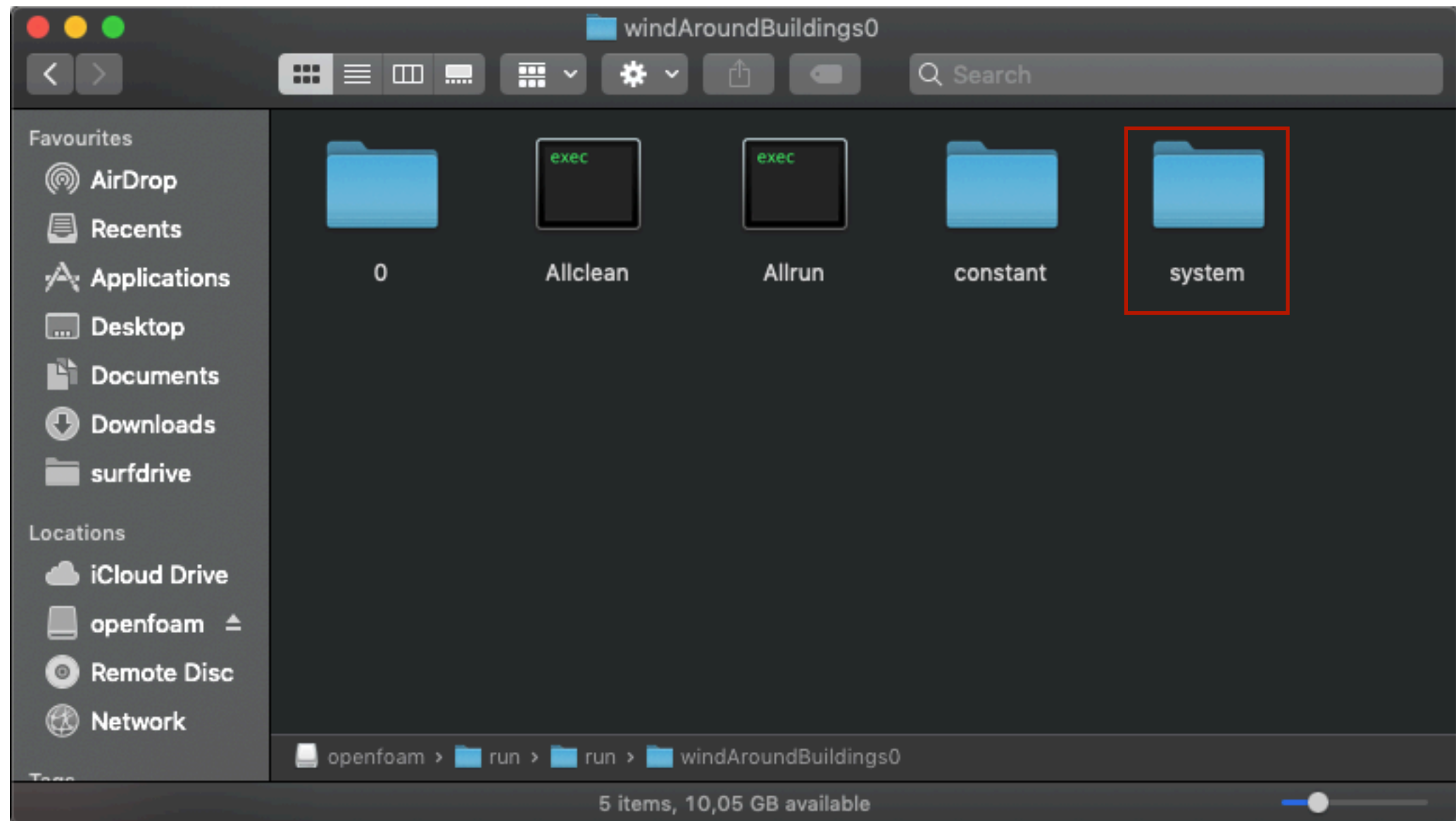
where each of the values corresponds to the power of each of the base units of m
 units for the Système International (SI) and the United States Customary System
 units. All that is required is that the *input data is correct for the chosen*
 that OpenFOAM requires some dimensioned physical constants, e.g. the Unive
 their physical modelling. These dimensioned constants are specified in a *Dimen*
caseDict file of the OpenFOAM installation (*\$WM_PROJECT_DIR/etc/contr*
 Those wishing to use the USCS or any other system of units should modify these

Physical properties are typically specified with their associated dimensions. Thes
 example of a *dimensionedScalar* demonstrates:

openFOAM test case structure

- Folders tree in case:

```
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/windAroundBuildings $FOAM_RUN/
```



openFOAM test case structure

- System - BlockMeshDict:

```
boundary
(
    inlet
    {
        type patch;
        faces
        (
            (0 3 7 4)
        );
    }

    outlet
    {
        type patch;
        faces
        (
            (1 5 6 2)
        );
    }

    ground
    {
        type wall;
        faces
        (
            (0 1 2 3)
        );
    }
);
```

```
frontAndBack
{
    type symmetry;
    faces
    (
        (0 4 5 1)
        (3 2 6 7)
        (4 7 6 5)
    );
}
```

Name	^	Date Modified
blockMeshDict		Yesterday at 17:11
controlDict		Yesterday at 17:11
fvSchemes		Yesterday at 17:11
fvSolution		Yesterday at 17:11
meshQualityDict		Yesterday at 17:11
snappyHexMeshDict		Yesterday at 17:11
surfaceFeaturesDict		Yesterday at 17:11

openFOAM test case structure

- System - BlockMeshDict:

Name	Date Modified
blockMeshDict	Yesterday at 17:11
controlDict	Yesterday at 17:11
fvSchemes	Yesterday at 17:11
fvSolution	Yesterday at 17:11
meshQualityDict	Yesterday at 17:11
snappyHexMeshDict	Yesterday at 17:11
surfaceFeaturesDict	Yesterday at 17:11

```
backgroundMesh
{
    xMin    -20; // L = 350
    xMax    330;
    yMin    -50; // L = 280
    yMax    230;
    zMin     0;
    zMax   140;
    xCells   25;
    yCells   20;
    zCells   10;
}
```

```
convertToMeters 1;
```

```
vertices
(
    ($:backgroundMesh.xMin $:backgroundMesh.yMin $:backgroundMesh.zMin)
    ($:backgroundMesh.xMax $:backgroundMesh.yMin $:backgroundMesh.zMin)
    ($:backgroundMesh.xMax $:backgroundMesh.yMax $:backgroundMesh.zMin)
    ($:backgroundMesh.xMin $:backgroundMesh.yMax $:backgroundMesh.zMin)
    ($:backgroundMesh.xMin $:backgroundMesh.yMin $:backgroundMesh.zMax)
    ($:backgroundMesh.xMax $:backgroundMesh.yMin $:backgroundMesh.zMax)
    ($:backgroundMesh.xMax $:backgroundMesh.yMax $:backgroundMesh.zMax)
    ($:backgroundMesh.xMin $:backgroundMesh.yMax $:backgroundMesh.zMax)
);
```

- *zeroGradient*: normal gradient of Q is zero.
- *calculated*: patch field Q calculated from other patch fields.
- *mixed*: mixed *fixedValue*/*fixedGradient* condition depending on $valueFraction = \begin{cases} 1 & \text{corresponds to } fixedValue \\ 0 & \text{corresponds to } fixedGradient \end{cases}$
- *directionMixed*: mixed condition with tensorial $valueFraction$ of a vector patch field, e.g. *fixedValue* in the tangential direction

5.2.3 Derived types

There are numerous more complex boundary conditions derived from *fixedValue*, where the value is calculated by a function or conditions derived from *mixed*/*directionMixed* switch between *fixedValue* and *fixedGradient*.

There are a number of ways the user can list the available boundary conditions using the *orBCs* utility being the quickest. The boundary conditions for scalar fields, e.g. *simpleFoam*, as follows.

In the following sections we will highlight some particular important

```
blocks
(
    hex (0 1 2 3 4 5 6 7)
    (
        $:backgroundMesh.xCells
        $:backgroundMesh.yCells
        $:backgroundMesh.zCells
    )
    simpleGrading (1 1 1)
);

edges
(
);
```

openFOAM test case structure

- System - SnappyHexMeshDict:

Name	Date Modified
blockMeshDict	Yesterday at 17:11
controlDict	Yesterday at 17:11
fvSchemes	Yesterday at 17:11
fvSolution	Yesterday at 17:11
meshQualityDict	Yesterday at 17:11
snappyHexMeshDict	Yesterday at 17:11
surfaceFeaturesDict	Yesterday at 17:11

```
/*----- C++ -----*/
//
// \ \ / \ Field
// \ \ / \ Operation
// \ \ / \ And
// \ \ / \ Manipulation
//
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       snappyHexMeshDict;
}

// *****

5.2.3 Derived types

#includeEtc "caseDicts/mesh/generation/snappyHexMeshDict.cfg"

castellatedMesh on;
snap           on;
addLayers      off;

geometry
{
    buildings
    {
        type triSurfaceMesh;
        file "buildings.obj";
    }

    refinementBox
    {
        type searchableBox;
        min ( 0 0 0);
        max (250 180 90);
    }
}
```

```
{
    features
    (
        { file "buildings.eMesh"; level 1; }
    );

    refinementSurfaces
    {
        buildings
        {
            level (3 3);
            patchInfo { type wall; }
        }
    }

    refinementRegions
    {
        refinementBox
        {
            mode inside;
            levels ((1E15 2));
        }
    }

    locationInMesh (1 1 1);
}
```

- System - surfaceFeaturesDict:

```

/*-----*-- C++ -*\
=====
\\      /  F i e l d
\\      /  O p e r a t i o n
\\      /  A n d
\\      /  M a n i p u l a t i o n
\\      /

/*-----*/

FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       surfaceFeaturesDict;

}

// * * * * *
5.2.3 Derived types

surfaces ("buildings.obj");

#includeEtc "caseDicts/surface/surfaceFeaturesDict.cfg"

// *****

```



- System - meshQualityDict:

```

/*-----*- C++ -*/
=====
\\      /  F i e l d
\\      /  O p e r a t i o n
\\      /  A n d
\\      /  M a n i p u l a t i o n
\\      /

FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       meshQualityDict;

    valueFraction = {
        1  corresponds to Q
        0  corresponds to Q
    };

    directionMixed mixed condition with tensor
    of a vector patch field, e.g. fixedValue in the

}

// * * * * *

```

```

-----*-- C++ -*\
=====
To: \Margo van der Heide | F ield | OpenFOAM: The Open Source CFD Toolbox
Cc: \ / | O peration | Website: https://openfoam.org
Dec: \ / | A nd | Version: 7
\ / | M anipulation |
\*-----*-/

From: Clara Garcia Sanchez - C.Garcia-Sanchez@tudelft.nl
maxNonOrtho 65;
Hi Margo,

maxBoundarySkewness 20; get a couple of power strips (stekkerdoos) for our classes.
Students will come with their own computer, so we want to make sure they can charge them if they need during class.
Do you know if we can find some of these around? Or should we buy them?

maxInternalSkewness 4;

Best,
maxConcave 80;
-----
Clara Garcia-Sanchez, PhD
// Minimum cell pyramid volume; case dependent
minVol 1e-13;
ADP Innovation Group, Architecture and Built Environment
University of Technology
Julianalaan 134 Delft 2628BL - The Netherlands
C.Garcia-Sanchez@tudelft.nl
minTetQuality -1e30;

// if >0 : preserve single cells with all points on the surface if the
// resulting volume after snapping (by approximation) is larger than
// minVolCollapseRatio times old volume (i.e. not collapsed to flat cell).
// If <0 : delete always.
//minVolCollapseRatio 0.5;

minArea -1;

minTwist 0.02;

minDeterminant 0.001;

minFaceWeight 0.05;

minVolRatio 0.01;

```

openFOAM test case structure

- System - fvSchemes:

Name	Date Modified
blockMeshDict	Yesterday at 17:11
controlDict	Yesterday at 17:11
fvSchemes	Yesterday at 17:11
fvSolution	Yesterday at 17:11
meshQualityDict	Yesterday at 17:11
snappyHexMeshDict	Yesterday at 17:11
surfaceFeaturesDict	Yesterday at 17:11

```
/*-----*-- C++ -*-----*\
=====
\\      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox
\\    /  O peration     | Website:  https://openfoam.org
\\  /    A nd           | Version:  7
\\ /      M anipulation  |
=====
\*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       fvSchemes;
}
// *****

addtSchemes
{
    default      steadyState;
}

gradSchemes
{
    default      Gauss linear;

    limited      cellLimited Gauss linear 1;
    grad(U)      $limited;
    grad(k)      $limited;
    grad(epsilon) $limited;
}

divSchemes
{
    default      none;

    div(phi,U)   bounded Gauss linearUpwind limited;
}
```

```
laplacianSchemes
{
    default      Gauss linear corrected;
}

interpolationSchemes
{
    default      linear;
}

snGradSchemes
{
    default      corrected;
}

wallDist
{
    method meshWave;
}

// *****
```


openFOAM test case structure

- System - fvSolution:

Name	Date Modified
blockMeshDict	Yesterday at 17:11
controlDict	Yesterday at 17:11
fvSchemes	Yesterday at 17:11
fvSolution	Yesterday at 17:11
meshQualityDict	Yesterday at 17:11
snappyHexMeshDict	Yesterday at 17:11
surfaceFeaturesDict	Yesterday at 17:11

```
/*-----*- C++ -*/
=====
\\ / Field | OpenFOAM: The Open Source CFD Toolbox
\\ / Operation | Website: https://openfoam.org
\\ / And | Version: 7
\\ / Manipulation |
/*-----*/

FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       fvSolution;
}
// *****

solvers
{
    p
    {
        solver      GAMG;
        smoother     GaussSeidel;
        tolerance    1e-6;
        relTol       0.1;
    }

    "(U|k|omega|epsilon)"
    {
        solver      smoothSolver;
        smoother     symGaussSeidel;
        tolerance    1e-6;
        relTol       0.1;
    }
}
```

```
SIMPLE
{
    residualControl
    {
        p          1e-4;
        U          1e-4;
        "(k|omega|epsilon)" 1e-4;
    }
    nNonOrthogonalCorrectors 0;
    pRefCell 0;
    pRefValue 0;
}

potentialFlow
{
    nNonOrthogonalCorrectors 10;
}

relaxationFactors
{
    fields
    {
        p          0.3;
    }
    equations
    {
        U          0.7;
        "(k|omega|epsilon)." 0.7;
    }
}
// *****
```

- System - controlDict:

```

/*-----*- C++ -*-----*/
=====
\\      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox
\\      / O p e r a t i o n | Website: https://openfoam.org
\\      / A n d             | Version: 7
\\      / M a n i p u l a t i o n |
/*-----*- C++ -*-----*/

FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       controlDict;
}
// * * * * *

application      simpleFoam;

startFrom        latestTime;

startTime        0;

stopAt           endTime;

endTime          400;

deltaT           1;

writeControl      timeStep;

writeInterval     50;

purgeWrite       0;

writeFormat       ascii;

```

```
writePrecision 6;

writeCompression off;

timeFormat      general;

timePrecision 6;

runTimeModifiable true;

// adjustTimeStep yes;

// maxCo      1;

functions
{
}
```

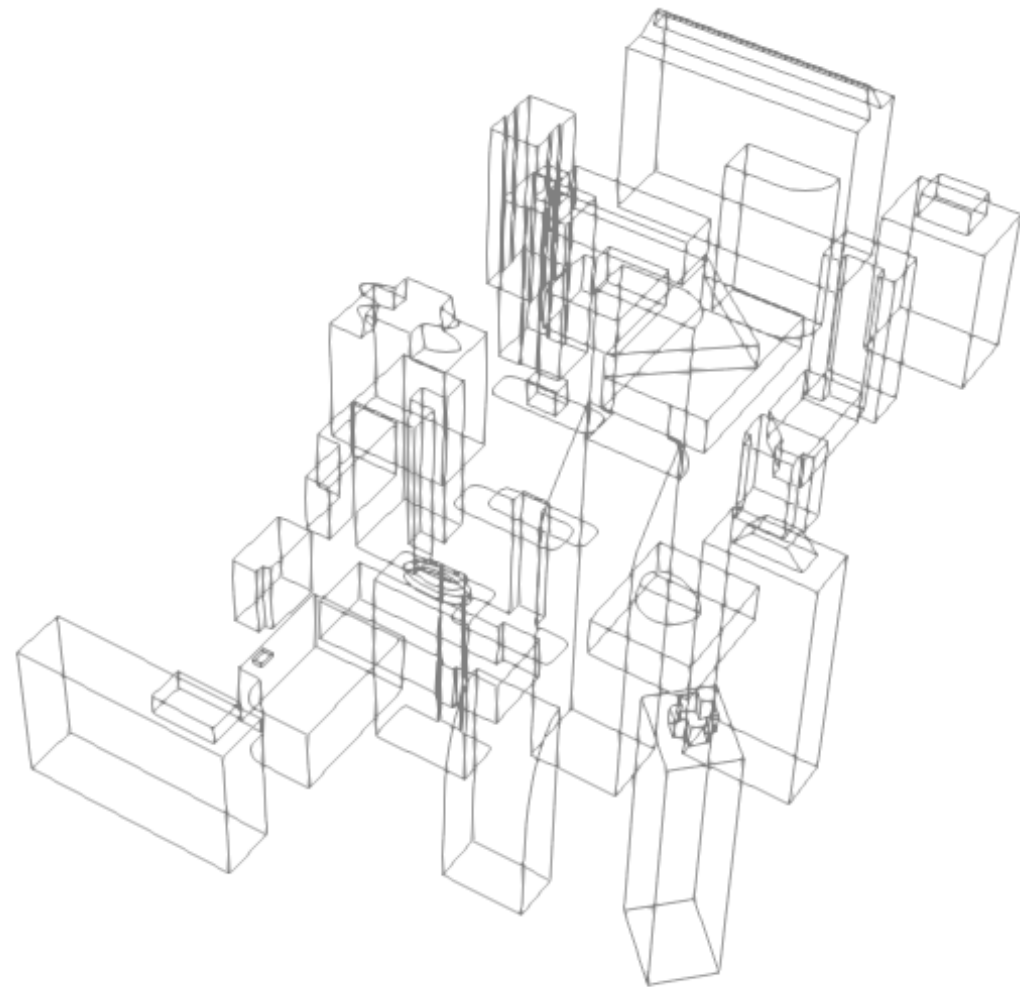
How to run openFOAM test case:

1. surfaceFeatures

- 2. blockMesh
- 3. snappyHexMesh
- 4. simpleFoam

Name	Date Modified
▶ extendedFeatureEdgeMesh	Today at 13:27
transportProperties	Yesterday at 17:11
▶ triSurface	Today at 13:06
turbulenceProperties	Yesterday at 17:11

With this command we can extract the edges, which can be used within snappyHexMesh, to help the snapping process and improve the quality of the mesh generation



How to run openFOAM test

1. surfaceFeatures

2. blockMesh

3. snappyHexMesh

4. simpleFoam

Name	^	Date Modified
▶ extendedFeatureEdgeMesh		Today at 13:30
▶ polyMesh		Today at 13:30
transportProperties		Yesterday at 17:11
▶ triSurface		Today at 13:06
turbulenceProperties		Yesterday at 17:11

Name	^	Date Modified
boundary		Today at 13:30
faces		Today at 13:30
neighbour		Today at 13:30
owner		Today at 13:30
points		Today at 13:30

```
// *****  
4  
(  
  inlet  
  {  
    type      patch;  
    nFaces    200;  
    startFace 14050;  
  }  
  outlet  
  {  
    type      patch;  
    nFaces    200;  
    startFace 14250;  
  }  
  ground  
  {  
    type      wall;  
    inGroups  List<word> 1(wall);  
    nFaces    500;  
    startFace 14450;  
  }  
  frontAndBack  
  {  
    type      symmetry;  
    inGroups  List<word> 1(symmetry);  
    nFaces    1000;  
    startFace 14950;  
  }  
)  
// *****
```

How to run openFOAM test

1. surfaceFeatures

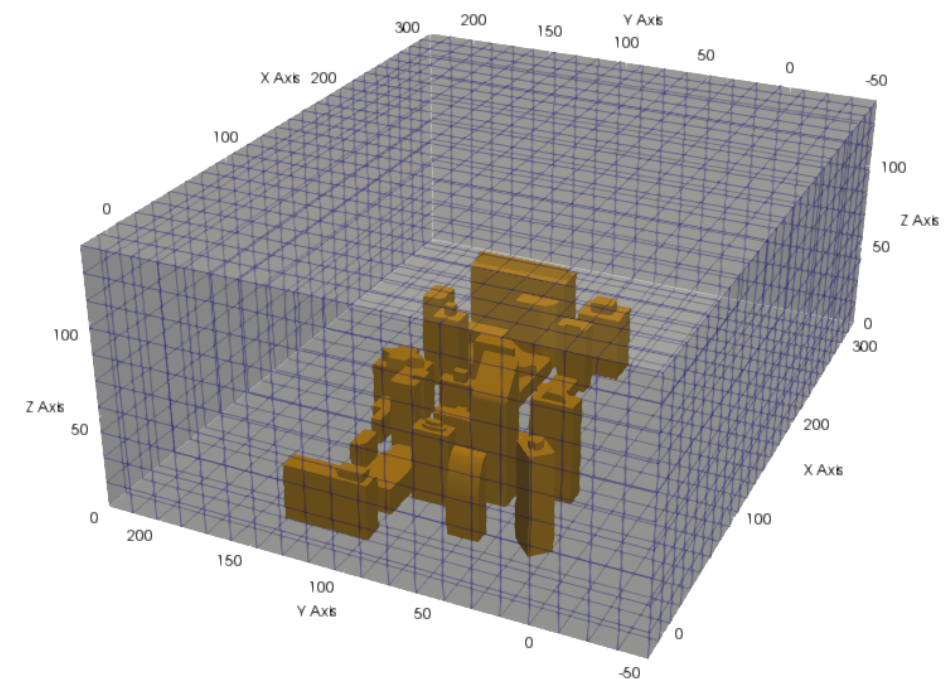
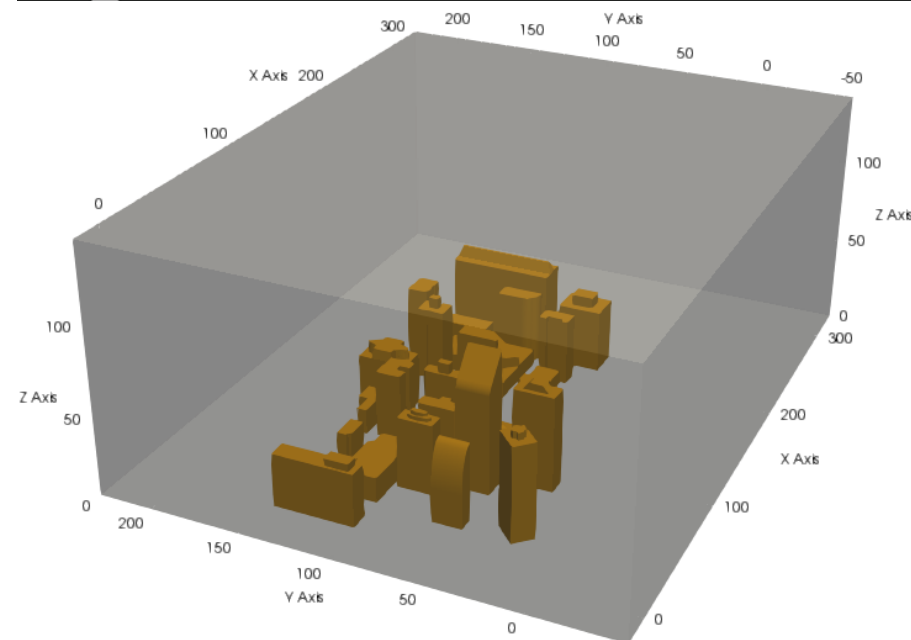
2. blockMesh

3. snappyHexMesh

4. simpleFoam

Name	Date Modified
▶ extendedFeatureEdgeMesh	Today at 13:30
▶ polyMesh	Today at 13:30
transportProperties	Yesterday at 17:11
▶ triSurface	Today at 13:06
turbulenceProperties	Yesterday at 17:11

Name	Date Modified
boundary	Today at 13:30
faces	Today at 13:30



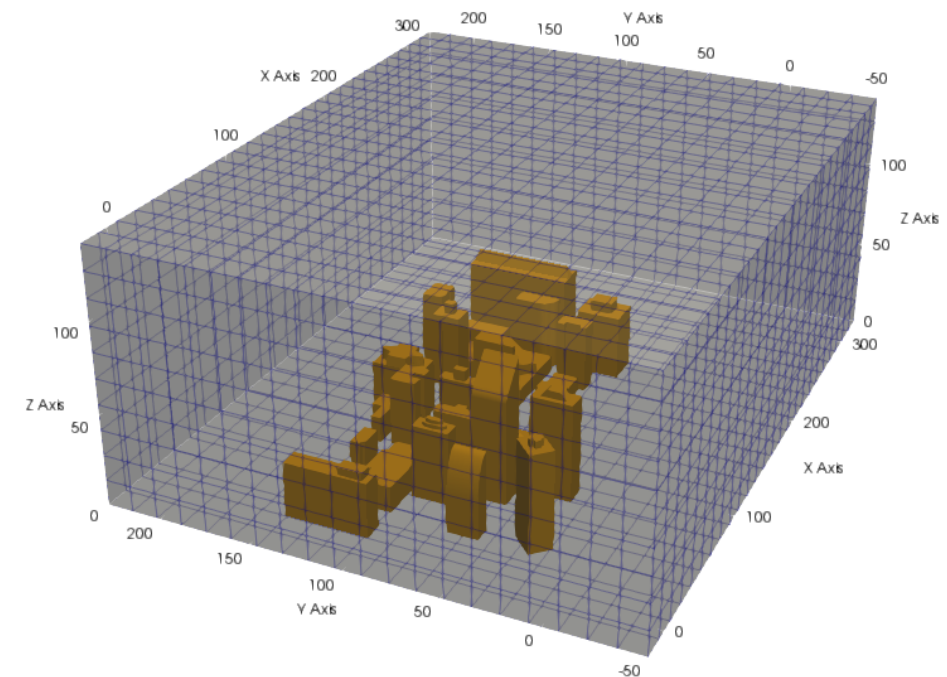
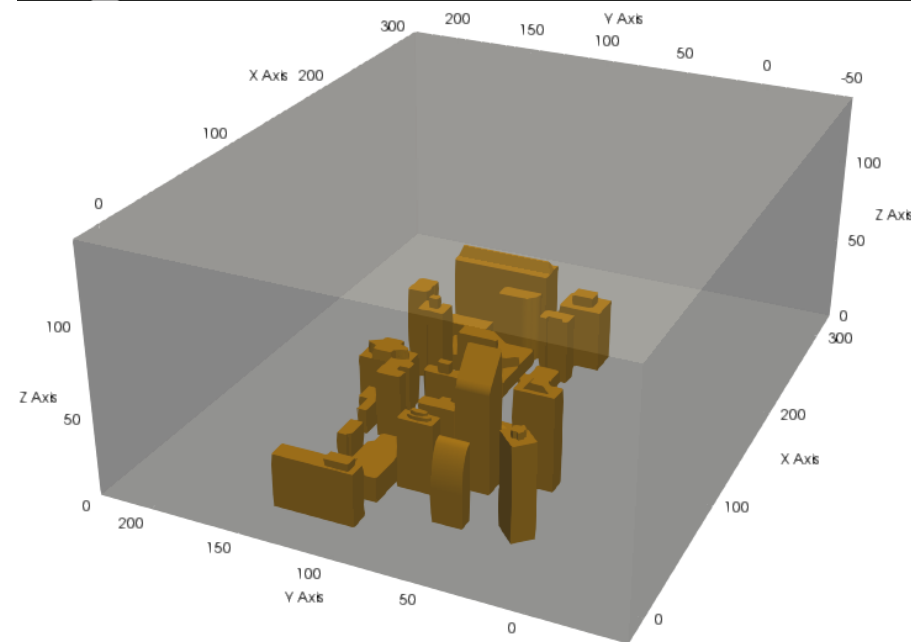
```
// *****  
4  
(  
  inlet  
  {  
    type            patch;  
    nFaces          200;  
    startFace       14050;  
  }  
  outlet  
  {  
    type            patch;  
    nFaces          200;  
    startFace       14250;  
  }  
  ground  
  {  
    type            wall;  
    inGroups        List<word> 1(wall);  
    nFaces          500;  
    startFace       14450;  
  }  
  frontAndBack  
  {  
    type            symmetry;  
    inGroups        List<word> 1(symmetry);  
    nFaces          1000;  
    startFace       14950;  
  }  
)  
// *****
```

How to run openFOAM test

1. surfaceFeatures
- 2. blockMesh**
3. snappyHexMesh
4. simpleFoam

Name	Date Modified
▶ extendedFeatureEdgeMesh	Today at 13:30
▶ polyMesh	Today at 13:30
transportProperties	Yesterday at 17:11
▶ triSurface	Today at 13:06
turbulenceProperties	Yesterday at 17:11

Name	Date Modified
boundary	Today at 13:30
faces	Today at 13:30



```
// *****  
4  
(  
  inlet  
  {  
    type            patch;  
    nFaces          200;  
    startFace       14050;  
  }  
  outlet  
  {  
    type            patch;  
    nFaces          200;  
    startFace       14250;  
  }  
  ground  
  {  
    type            wall;  
    inGroups        List<word> 1(wall);  
    nFaces          500;  
    startFace       14450;  
  }  
  frontAndBack  
  {  
    type            symmetry;  
    inGroups        List<word> 1(symmetry);  
    nFaces          1000;  
    startFace       14950;  
  }  
)  
// *****
```

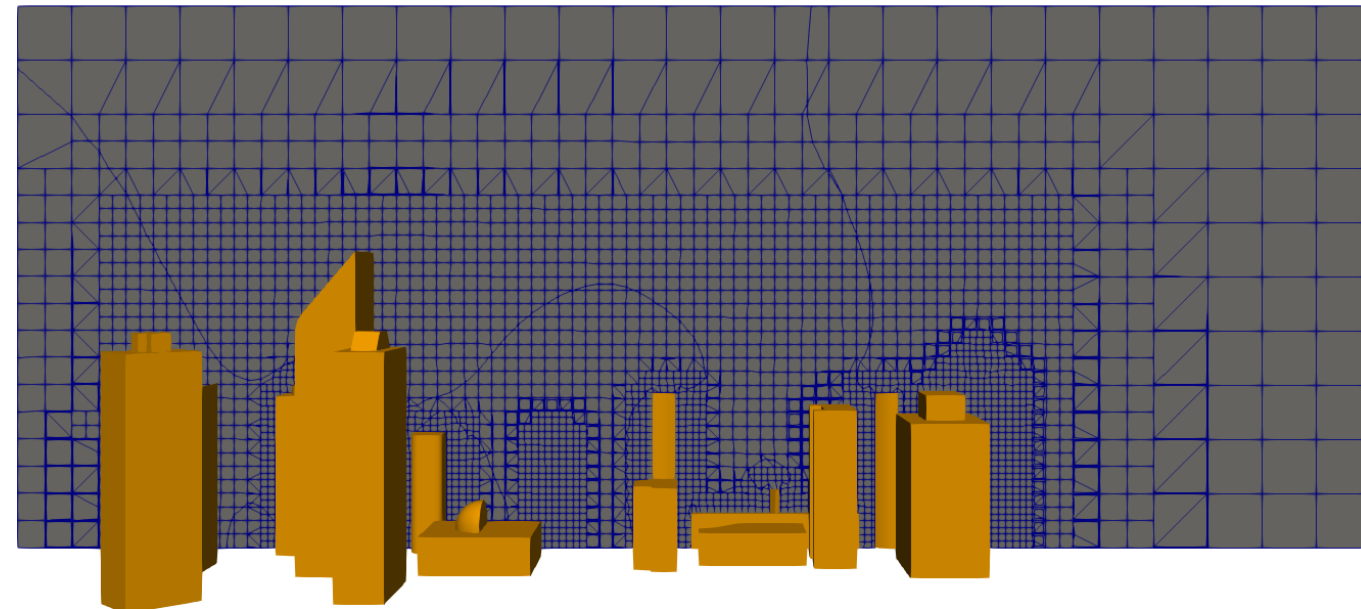
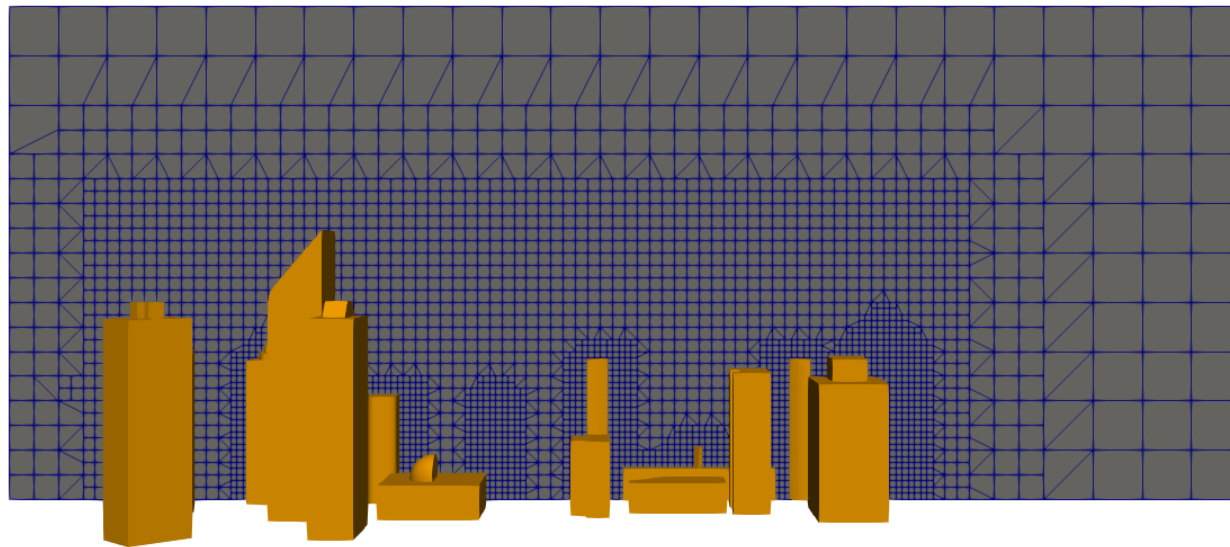

How to run openFOAM test case

Name	Date Modified
▶ 0	Today at 10:38
▶ 1	Today at 13:37
▶ 2	Today at 13:37
▶ Allclean	Today at 09:46
▶ Allrun	Yesterday at 17:11
▶ constant	Today at 13:30
▶ system	Today at 13:10
windAroundBuildings0.foam	Today at 13:16

1. surfaceFeatures
2. blockMesh
- 3. snappyHexMesh**
4. simpleFoam

```
#includeEtc "caseDicts/mesh/generation/snappyHexMeshDict.cfg"

castellatedMesh on;
snap on;
addLayers off;
```



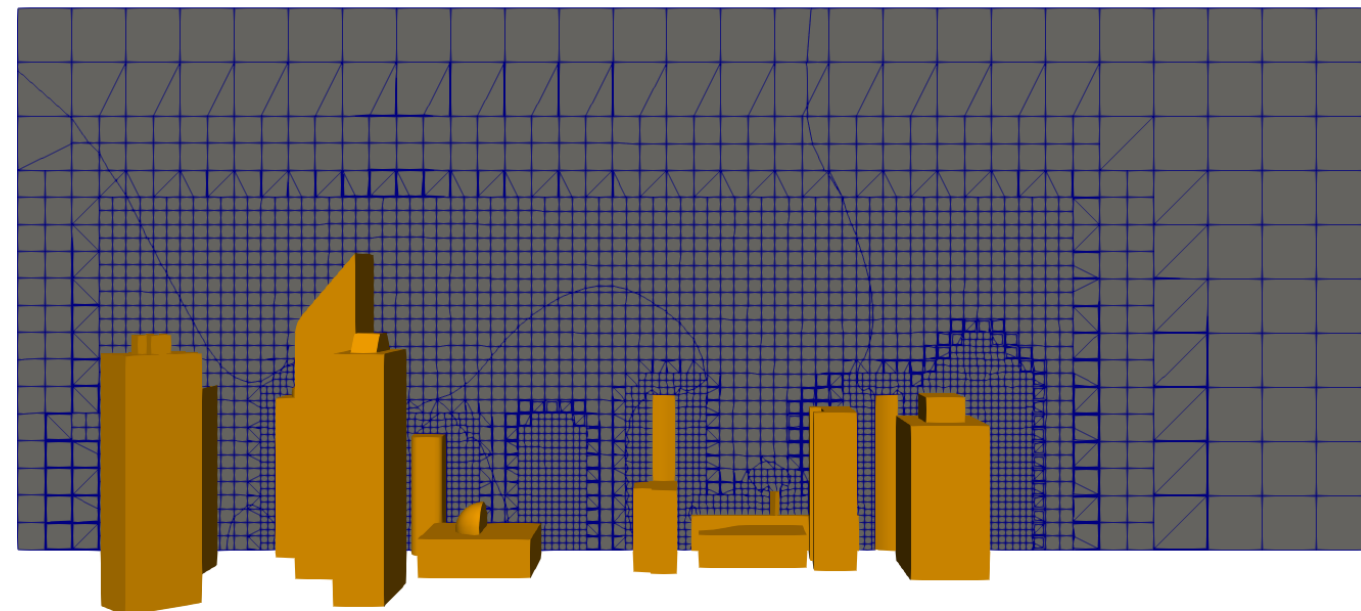
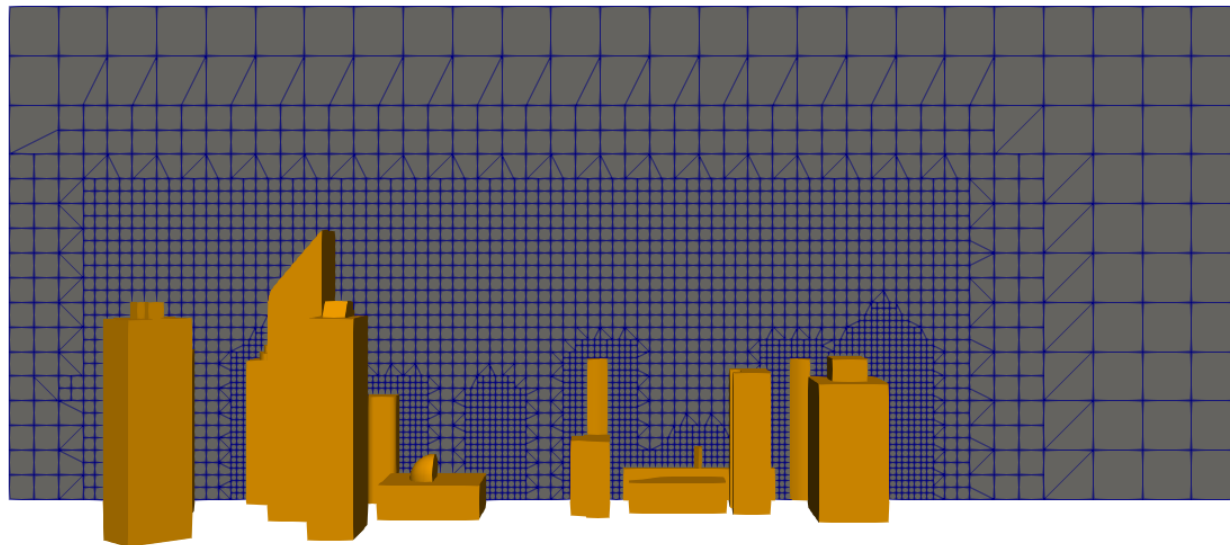
How to run openFOAM test case

Name	Date Modified
▶ 0	Today at 10:38
▶ 1	Today at 13:37
▶ 2	Today at 13:37
▶ Allclean	Today at 09:46
▶ Allrun	Yesterday at 17:11
▶ constant	Today at 13:30
▶ system	Today at 13:10
windAroundBuildings0.foam	Today at 13:16

1. surfaceFeatures
2. blockMesh
- 3. snappyHexMesh**
4. simpleFoam

```
#includeEtc "caseDicts/mesh/generation/snappyHexMeshDict.cfg"
```

1. castellatedMesh on;
2. snap on;
addLayers off;



How to run openFOAM test case

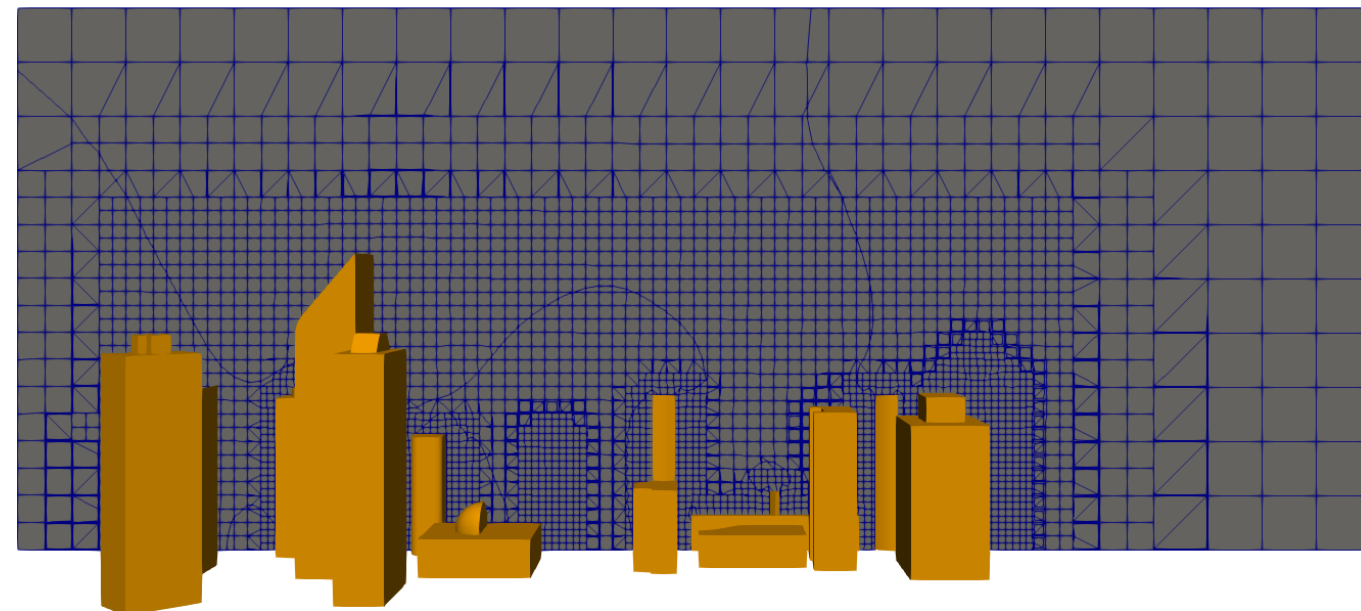
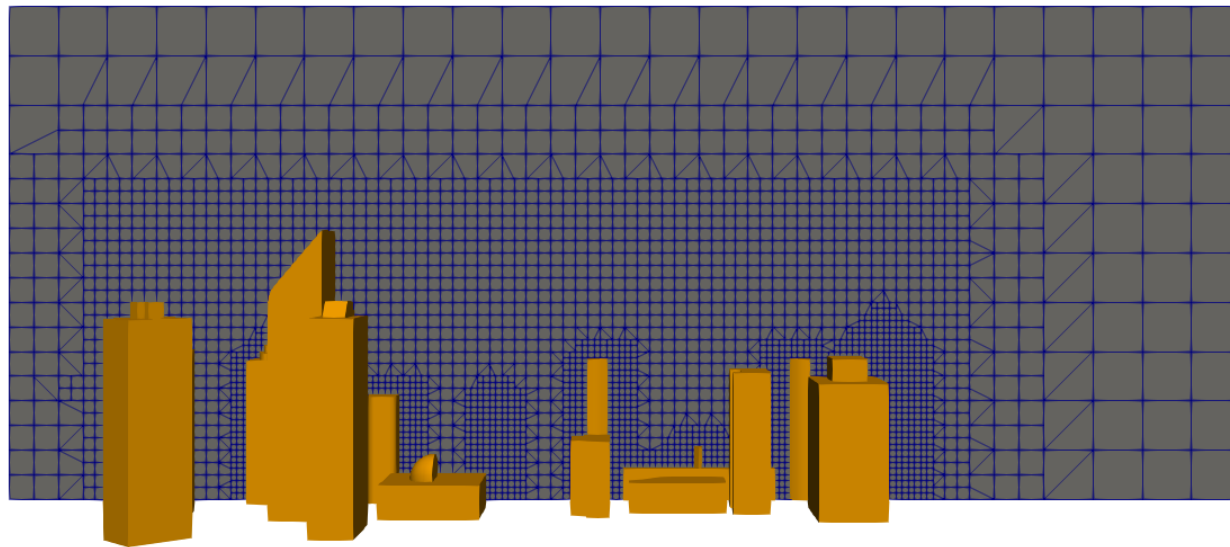
Name	Date Modified
▶ 0	Today at 10:38
▶ 1	Today at 13:37
▶ 2	Today at 13:37
▶ Allclean	Today at 09:46
▶ Allrun	Yesterday at 17:11
▶ constant	Today at 13:30
▶ system	Today at 13:10
windAroundBuildings0.foam	Today at 13:16

1. surfaceFeatures
2. blockMesh
- 3. snappyHexMesh**
4. simpleFoam

```
#includeEtc "caseDicts/mesh/generation/snappyHexMeshDict.cfg"
```

1. castellatedMesh on;
2. snap on;
addLayers off;

1. Castellation



How to run openFOAM test case

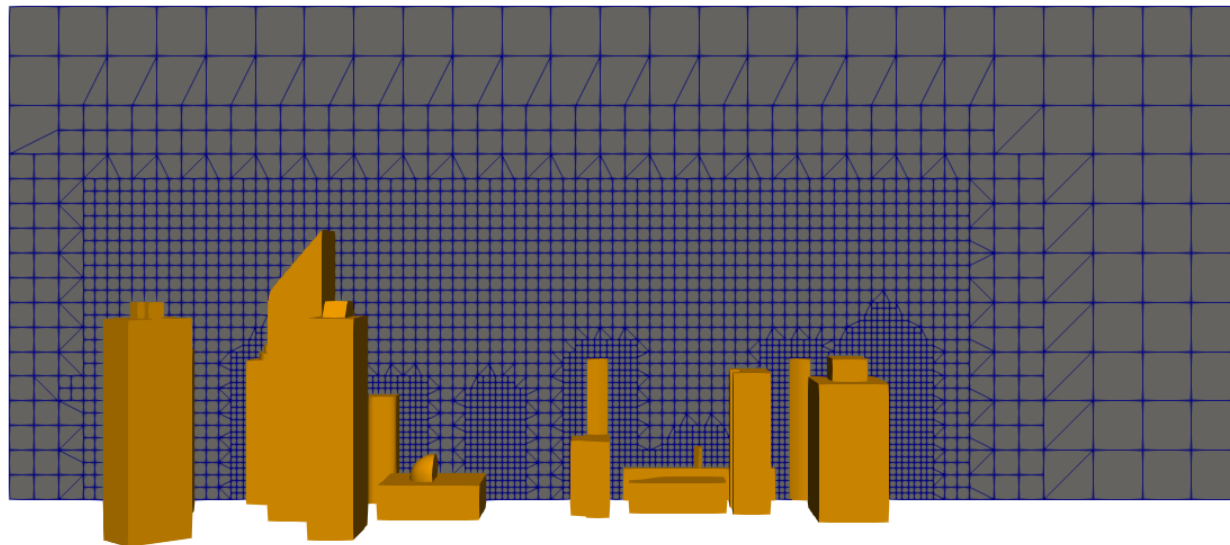
Name	Date Modified
▶ 0	Today at 10:38
▶ 1	Today at 13:37
▶ 2	Today at 13:37
▶ Allclean	Today at 09:46
▶ Allrun	Yesterday at 17:11
▶ constant	Today at 13:30
▶ system	Today at 13:10
windAroundBuildings0.foam	Today at 13:16

1. surfaceFeatures
2. blockMesh
- 3. snappyHexMesh**
4. simpleFoam

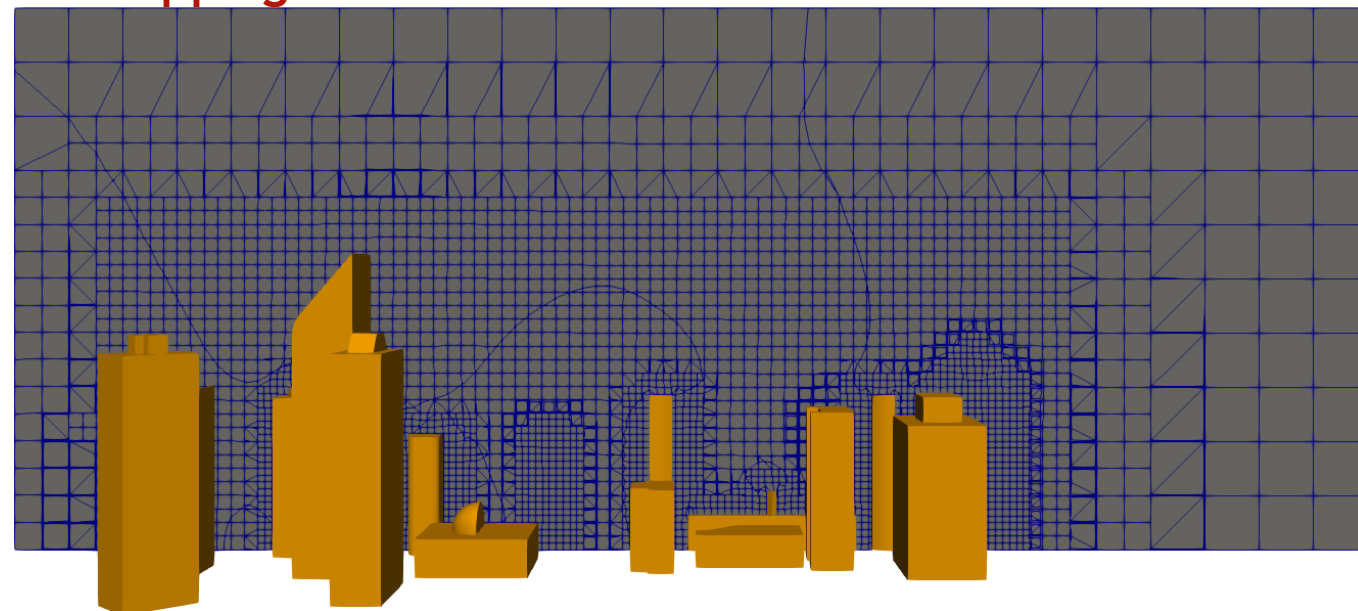
```
#includeEtc "caseDicts/mesh/generation/snappyHexMeshDict.cfg"
```

1. castellatedMesh on;
2. snap on;
addLayers off;

1. Castellation



2. Snapping



How to run openFOAM test case:

1. surfaceFeatures
2. blockMesh
3. snappyHexMesh
4. **simpleFoam**

```
SIMPLE
{
    residualControl
    {
        p            1e-4;
        U            1e-4;
        "(k|omega|epsilon)" 1e-4;
    }
    nNonOrthogonalCorrectors 0;
    pRefCell          0;
    pRefValue          0;
}
```

Name	Date Modified
▶ 0	Today at 10:38
▶ 1	Today at 13:37
▶ 2	Today at 13:45
▶ 52	Today at 13:46
▶ 102	Today at 13:46
▶ 152	Today at 13:47
▶ 202	Today at 13:47
▶ 252	Today at 13:48
▶ 302	Today at 13:48
▶ 352	Today at 13:49
▶ Allclean	Today at 09:46
▶ Allrun	Yesterday at 17:11
▶ constant	Today at 13:30
▶ system	Today at 13:45
▶ windAroundBuildings0.foam	Today at 13:16

```
Time = 399

smoothSolver: Solving for Ux, Initial residual = 0.000374315, Final residual = 2.13748e-05, No Iterations 2
smoothSolver: Solving for Uy, Initial residual = 0.00149301, Final residual = 8.74602e-05, No Iterations 2
smoothSolver: Solving for Uz, Initial residual = 0.00168795, Final residual = 0.000102205, No Iterations 2
GAMG: Solving for p, Initial residual = 0.0132326, Final residual = 0.000431114, No Iterations 2
time step continuity errors : sum local = 1.12228e-05, global = 1.23819e-06, cumulative = 0.00416481
smoothSolver: Solving for epsilon, Initial residual = 0.00286634, Final residual = 0.000159461, No Iterations 1
smoothSolver: Solving for k, Initial residual = 0.00206614, Final residual = 0.000165122, No Iterations 1
ExecutionTime = 241.25 s  ClockTime = 247 s

Time = 400

smoothSolver: Solving for Ux, Initial residual = 0.000373325, Final residual = 2.13718e-05, No Iterations 2
smoothSolver: Solving for Uy, Initial residual = 0.00148155, Final residual = 8.70915e-05, No Iterations 2
smoothSolver: Solving for Uz, Initial residual = 0.00167385, Final residual = 0.000101258, No Iterations 2
GAMG: Solving for p, Initial residual = 0.0132477, Final residual = 0.000427785, No Iterations 2
time step continuity errors : sum local = 1.11348e-05, global = 1.15567e-06, cumulative = 0.00416596
smoothSolver: Solving for epsilon, Initial residual = 0.00288552, Final residual = 0.000160044, No Iterations 1
smoothSolver: Solving for k, Initial residual = 0.00205397, Final residual = 0.000165605, No Iterations 1
ExecutionTime = 241.83 s  ClockTime = 248 s
```

How to run openFOAM test case:

1. surfaceFeatures
2. blockMesh
3. snappyHexMesh
4. **simpleFoam**

```
SIMPLE
{
    residualControl
    {
        p          1e-4;
        U          1e-4;
        "(k|omega|epsilon)" 1e-4;
    }
    nNonOrthogonalCorrectors 0;
    pRefCell        0;
    pRefValue        0;
}
```

```
// * * * * *
application      simpleFoam;
startFrom        latestTime;
startTime        0;
stopAt           endTime;
endTime          400;
deltaT           1;
writeControl      timeStep;
writeInterval     50;
```

Name	Date Modified
▶ 0	Today at 10:38
▶ 1	Today at 13:37
▶ 2	Today at 13:45
▶ 52	Today at 13:46
▶ 102	Today at 13:46
▶ 152	Today at 13:47
▶ 202	Today at 13:47
▶ 252	Today at 13:48
▶ 302	Today at 13:48
▶ 352	Today at 13:49
▶ Allclean	Today at 09:46
▶ Allrun	Yesterday at 17:11
▶ constant	Today at 13:30
▶ system	Today at 13:45
▶ windAroundBuildings0.foam	Today at 13:16

```
Time = 399
smoothSolver: Solving for Ux, Initial residual = 0.000374315, Final residual = 2.13748e-05, No Iterations 2
smoothSolver: Solving for Uy, Initial residual = 0.00149301, Final residual = 8.74602e-05, No Iterations 2
smoothSolver: Solving for Uz, Initial residual = 0.00168795, Final residual = 0.000102205, No Iterations 2
GAMG: Solving for p, Initial residual = 0.0132326, Final residual = 0.000431114, No Iterations 2
time step continuity errors : sum local = 1.12228e-05, global = 1.23819e-06, cumulative = 0.00416481
smoothSolver: Solving for epsilon, Initial residual = 0.00286634, Final residual = 0.000159461, No Iterations 1
smoothSolver: Solving for k, Initial residual = 0.00206614, Final residual = 0.000165122, No Iterations 1
ExecutionTime = 241.25 s  ClockTime = 247 s

Time = 400
smoothSolver: Solving for Ux, Initial residual = 0.000373325, Final residual = 2.13718e-05, No Iterations 2
smoothSolver: Solving for Uy, Initial residual = 0.00148155, Final residual = 8.70915e-05, No Iterations 2
smoothSolver: Solving for Uz, Initial residual = 0.00167385, Final residual = 0.000101258, No Iterations 2
GAMG: Solving for p, Initial residual = 0.0132477, Final residual = 0.000427785, No Iterations 2
time step continuity errors : sum local = 1.11348e-05, global = 1.15567e-06, cumulative = 0.00416596
smoothSolver: Solving for epsilon, Initial residual = 0.00288552, Final residual = 0.000160044, No Iterations 1
smoothSolver: Solving for k, Initial residual = 0.00205397, Final residual = 0.000165605, No Iterations 1
ExecutionTime = 241.83 s  ClockTime = 248 s
```

How to run openFOAM test case:

1. surfaceFeatures
2. blockMesh
3. snappyHexMesh

4. simpleFoam IN PARALLEL

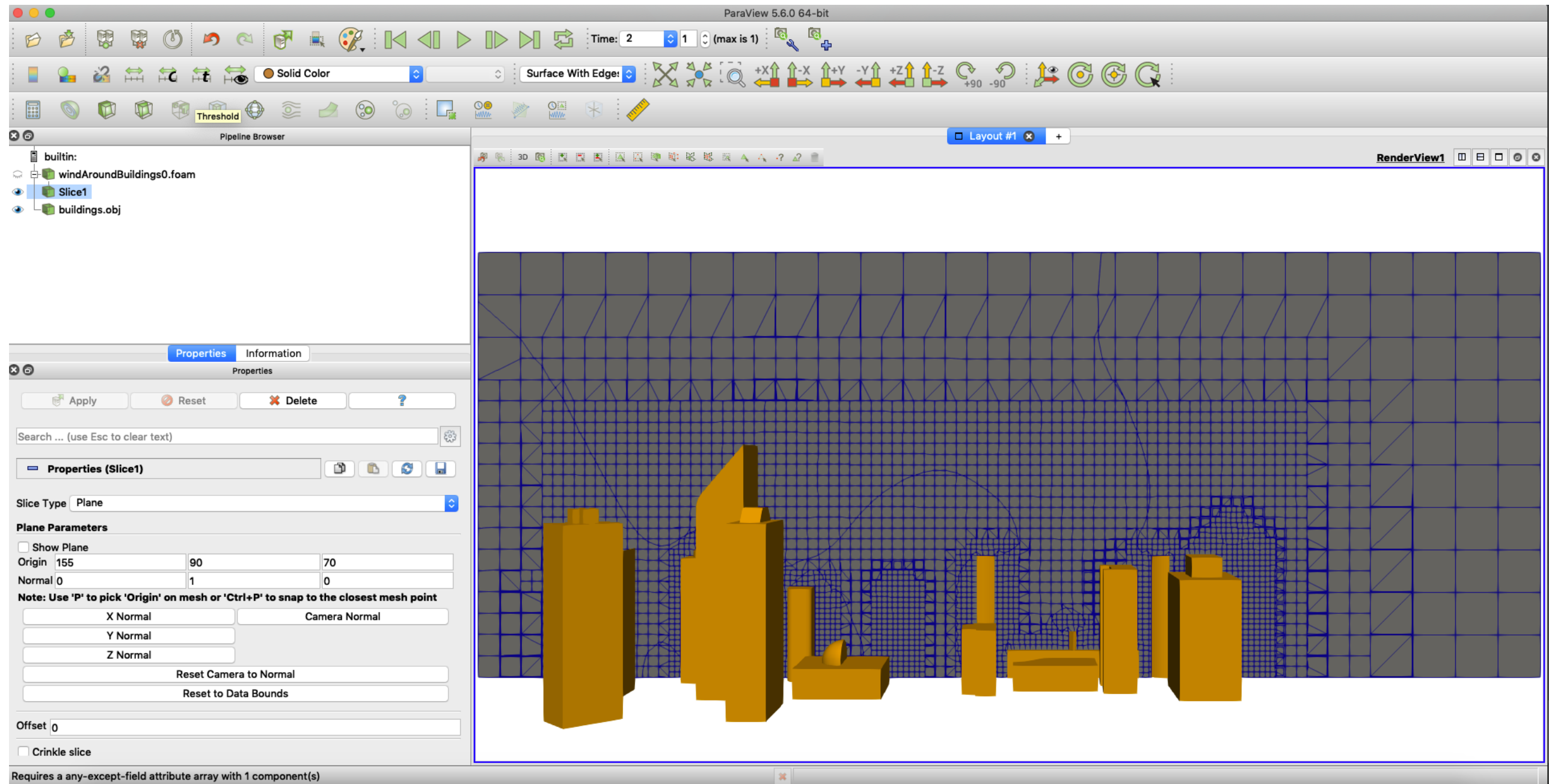
```
cp $FOAM_ETC/caseDicts/preProcessing/decomposeParDict system/
```

```
mpirun -np 4 simpleFoam -parallel >& log.SF &
```

```
tail -f log.SF
```

```
reconstructPar -latestTime
```


■ Paraview:



<https://www.paraview.org/Wiki/images/6/6e/ParaViewTutorial54.pdf>

Activities before next class

- Go to: <https://www.paraview.org/Wiki/images/6/6e/ParaViewTutorial54.pdf>
- Make the different plots suggested in the tutorial
- Run the case switching off turbulence
- Run the case increasing the **ν** (viscosity) to honey level (2000) or oil level (56.2)
- What happens if you change endTime in controlDict