12th OpenFOAM Study Meeting for beginner

-What I learned since previous meeting-

Twitter ID:oga_shin
小縣信也
ogata shinya

Contents

1. 6th OpenFOAM Workshop
    - Brief report
    - Result of inviting Jasak to our Study Group
    - swak4FOAM+OpenFOAMvr2.0.0
    - Blender+Python

3. How to make equal interval vector in Paraview

4. How to understand “g & mesh.C()”

5. Encoding software “ffmpeg”
   how to encode single pictures into movie

6. Recommending books

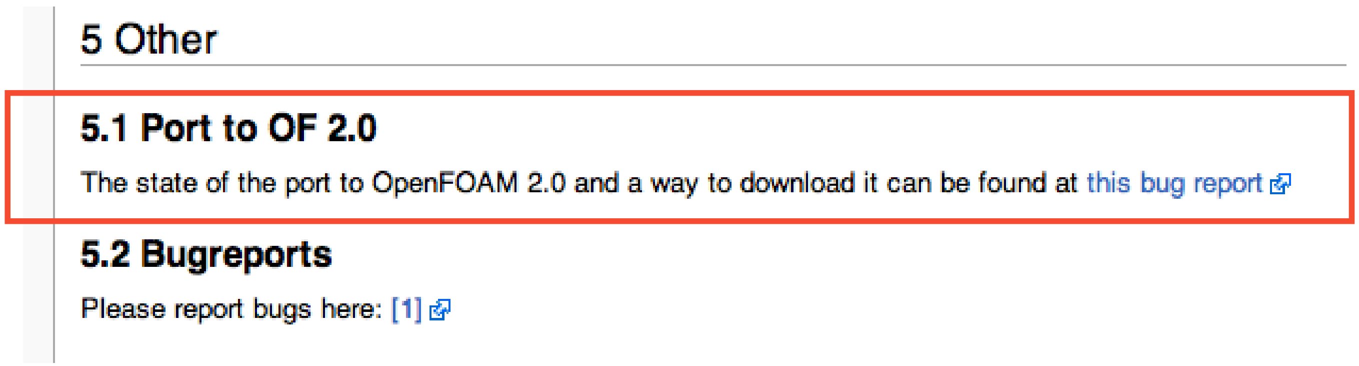
swak4FOAM + OpenFOAMvr2.0.x

Is it possible to use swak4FOAM with OpenFOAMvr2.0.0?

•

•

•



swak4FOAM + OpenFOAMvr2.0.x

We couldn't compile swak4FOAM with OpenFOAMvr2.0.0 at the time OF2.0.0 was released

Summary	0000073: swak4foam does not compile with OF-2.0.0.
Description	swak4foam for OpenFOAM 1.7. does not compile with C in the swak4Foam directory. My swak4foam version is http://openfoam-extend.svn.sourceforge.net/viewvc/op

Bernhard F.W. Gschaider improved S4F, now we probably can do it. But how?

(0000190) **Bgschaid** (administrator) 2011-06-27 12:14	Yep. Am aware of this (thanks anyway for the report). A first stab at the port can be found in the branch port_2.0.x of the mercurial archive: hg clone http://openfoam-extend.hg.sourceforge.net:8000/hgroot/openfoam-extend/swak4Foam [^] cd swak4Foam hg update port_2.0.x (if you already have that cloned you only need to pull and then update) It compiles parts: I see the binaries for groovyBC and funkySetFields but I didn't test them (feedback appreciated as I'm currently on the road and have little time to test this) What doesn't compile are the simpleFunctionObjectsand the stuff that is based on it. I'll do that in the next days (SFO will propably become a part of swak)
(0000198) **Bgschaid** (administrator) 2011-07-06 21:21	Now the whole swak-Stuff (except of course finiteArea) compiles and all the Examples can be started but I havn't compared the results to the old version. SimpleFunctionObjects are now a part of swak This has been pushed to the Mercurial-repository. I will have a look at the simulation results when I find the time. Once I'm convinced that differences to the 1.7-results are due to changes in the Upstream-solvers and not swak I will announce the port publicly. Feedback will speed up that process

swak4FOAM + OpenFOAMvr2.0.x

Hot to complile SW4F with OF2.0.0

- to clone SW4F-source in Mercurial-repository
- to update cloned directory
- to Allwmake as usual.

```
hg clone http://openfoam-extend.hg.sourceforge.net:8000/hgroot/openfoam-extend/swak4Foam [^]
cd swak4Foam
hg update port_2.0.x耀
```

I succeeded to run SW4F example case.
I have one question, that is Does [^] mean swak4FOAM directory??

6th OpenFOAM Workshop

One of training sessions I took

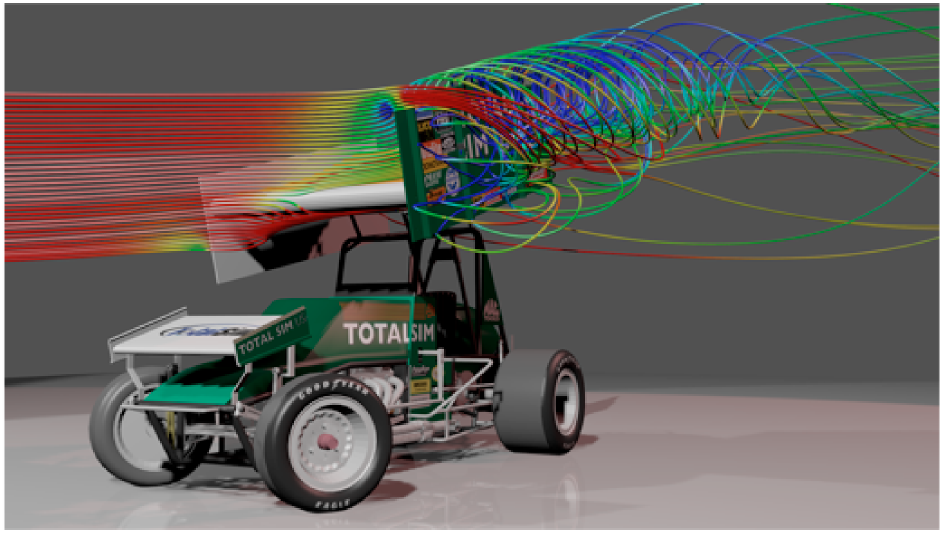
Title: Using Blender with OpenFOAM to produce high quality renderings

Instructor: Matt Cragun, Totalsim Inc.

Description: Blender is an open source 3D modeling program that has the capability to produce high-quality 3D data renderings. This class will provide instructions on Blender basics as well as importing and rendering models in both vrml, stl, and VTK formats. Topics will include basics of Blender navigation, materials, rendering, and python scripting with the VTK toolset from inside Blender. Participants will receive a copy of the presentation, a sample dataset, and a copy of Blender 2.49 and Blender 2.50. It is recommended that students be familiar with VTK or Paraview; as well, some experience with python scripting would be useful.

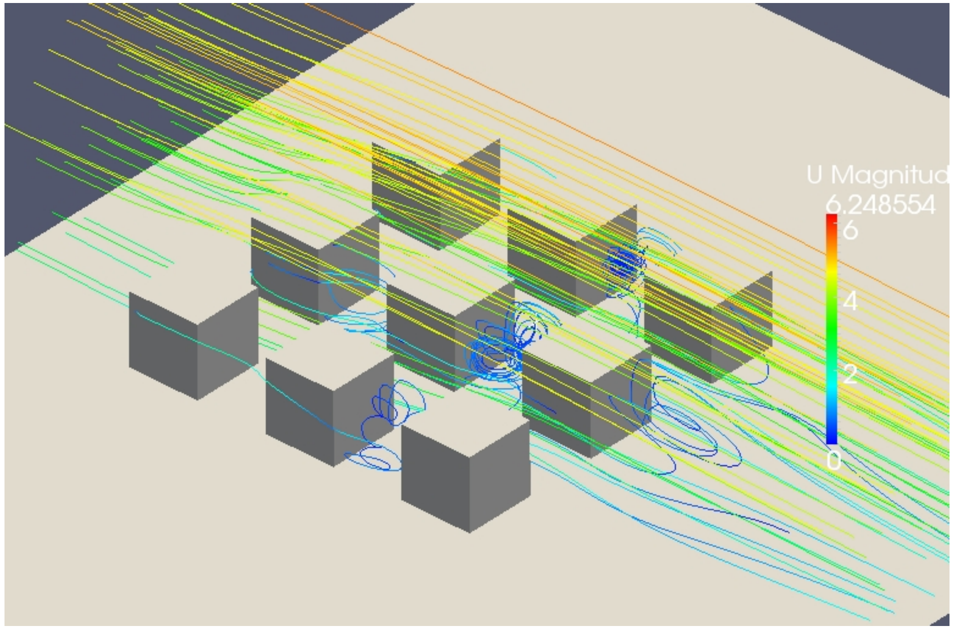
Duration: 90 minutes

Date and Location: 13 June 2011, Penn Sta

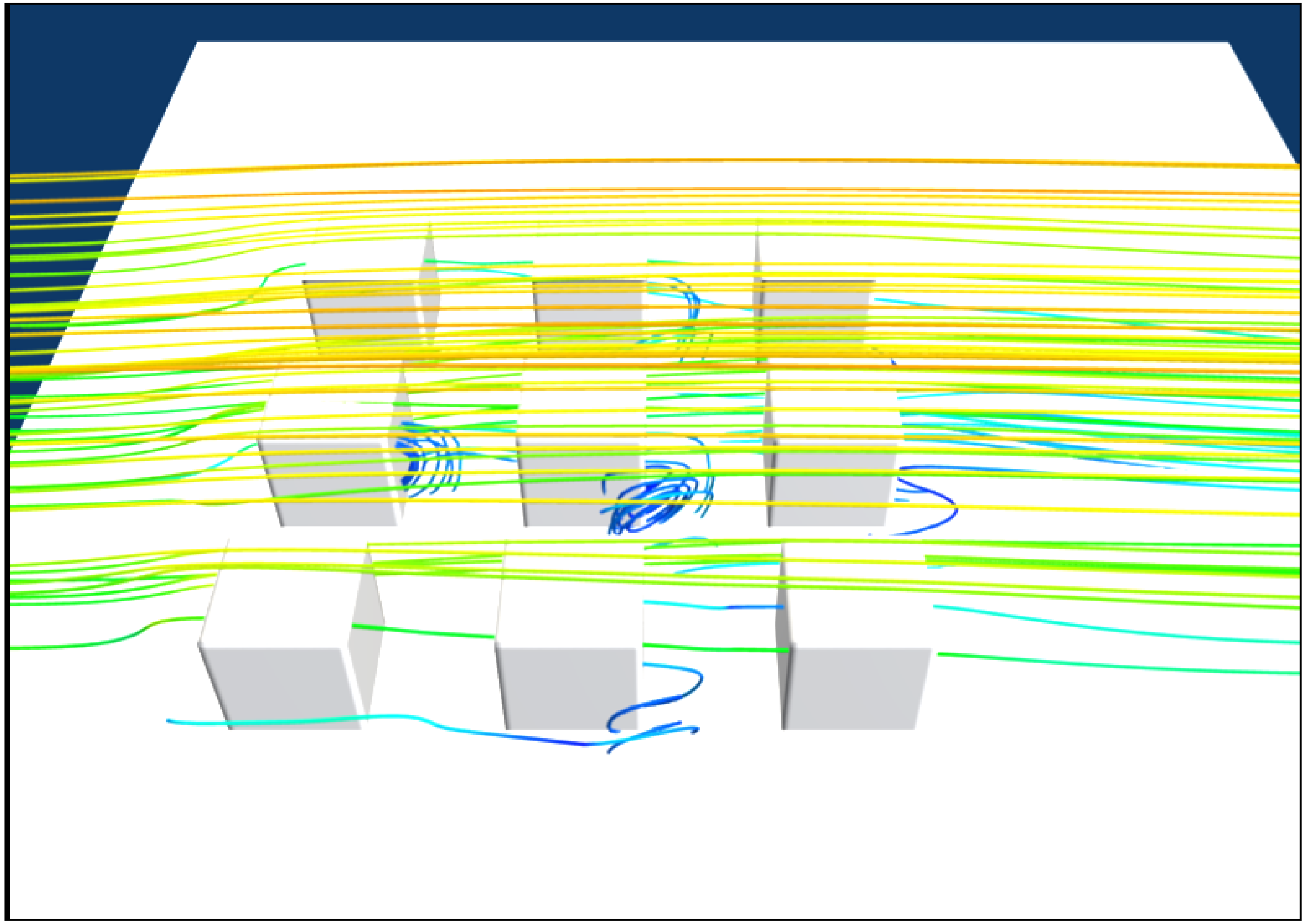


How to render the OF result with Blender

1.make stream line and tube filiter in paraview
2.export it as VRML file format
3.import the VRML file as X3D&VRML97
4.select stream line object
5.click the button for material context
6.turn on 'Vcol paint'
7.render

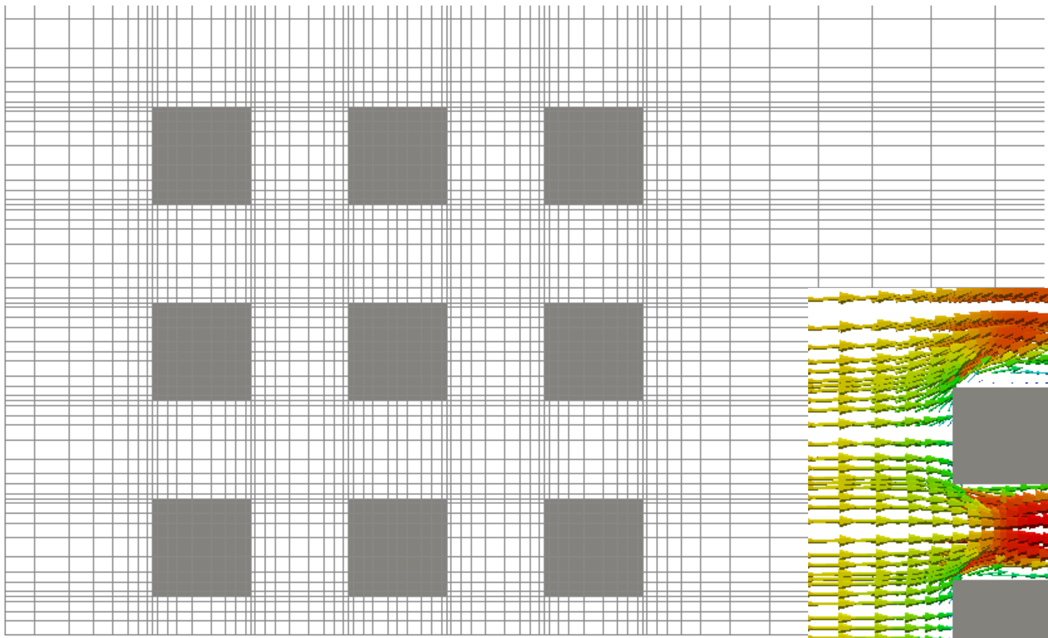


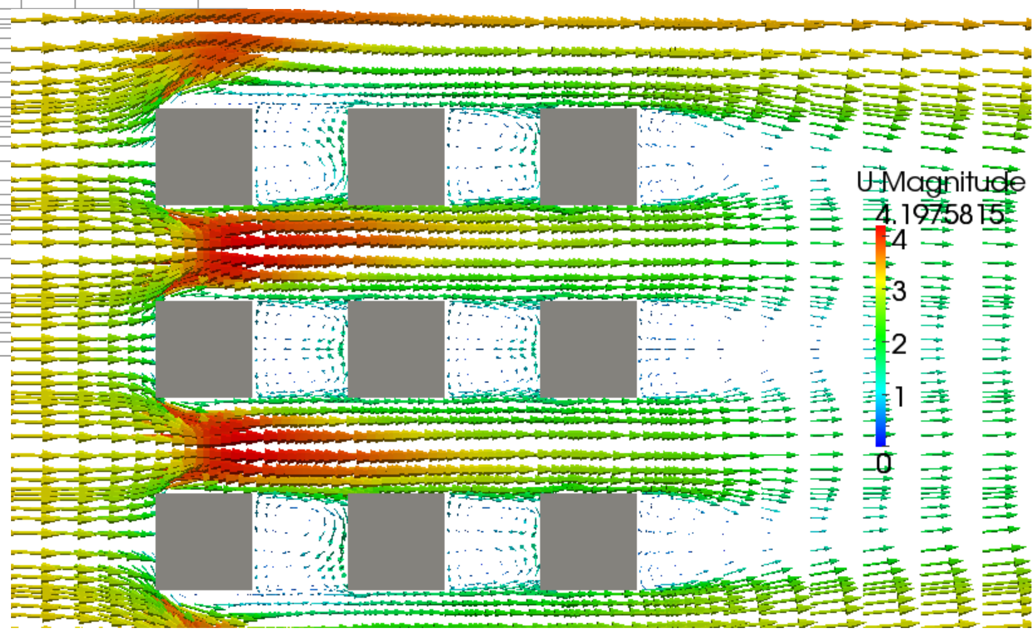
Rendered picture in Blender



How to make equal interval vector in Paraview

In Paraview, arrows are made on the cross point of grid.
If grid is not equal interval , arrows are not equal interval.
One of the solutions is using Resample With Dateset filter.





How to make equal interval vector in Paraview

How to use “Resample With Dataset” filter.

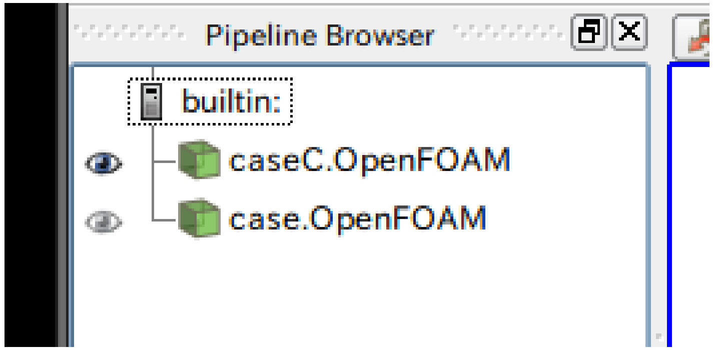
1.duplicate the OpenFOAM case directory.
2.make blockMeshDict in duplicated-case/constant/polymesh
3.modify the file as you want.
 The division number will define the interval of Vector in Paraview.
4.blockMesh & cellSet
5.make dummy file “case.OpenFOAM” under the duplicated case directory to read the mesh data in Paraview later.

```
vertices
(    (-1 -1.5 0)
     (1.5 -1.5 0)
     (1.5 1.5 0)
     (-1 1.5 0)
     (-1 -1.5 1.8)
     (1.5 -1.5 1.8)
     (1.5 1.5 1.8)
     (-1 1.5 1.8)     );

                              division number
blocks
(  hex (0 1 2 3 4 5 6 7) (100 100 100) simpleGrading (1 1 1) );
```

How to make equal interval vector in Paraview

5.move to original case directory and type “paraFoam”
6.import “case.OpenFOAM” under the duplicated case directory.
Now, you have two source in pipelin of paraview.

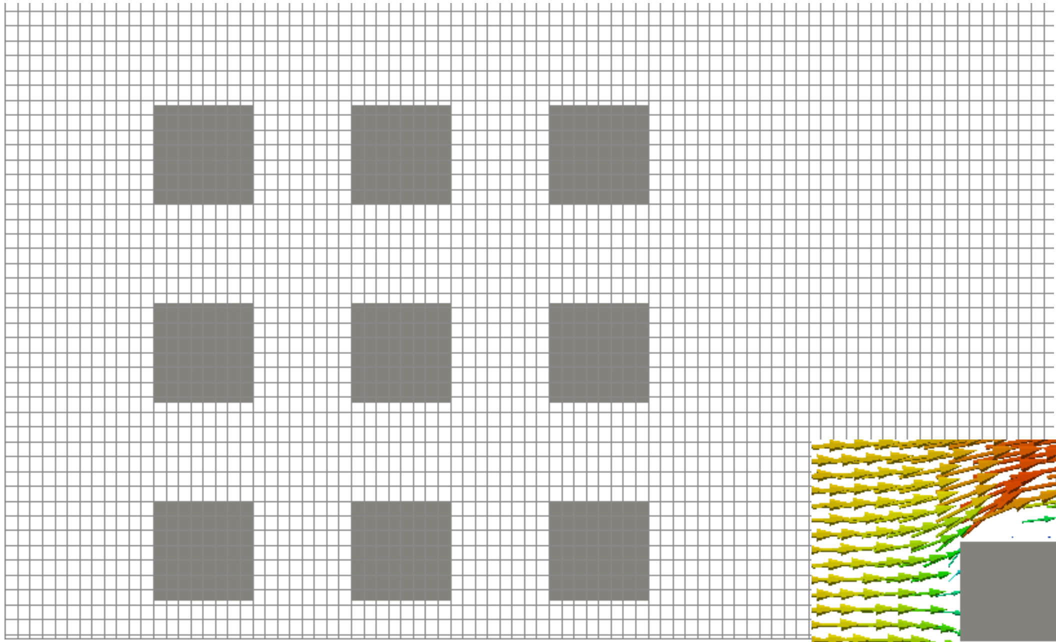


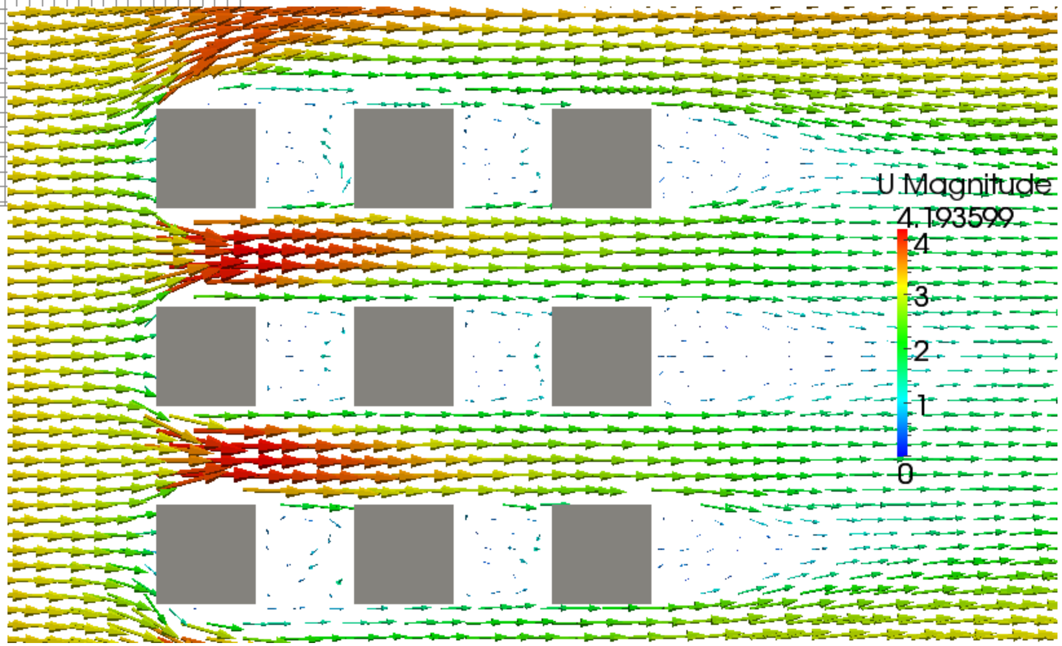
7.choose “Resample With Dataset” filter
Input: original data Source:duplicated data

Then, original data will be mapped on the new grid which was defined on blockMeshDict.

How to make equal interval vector in Paraview

Results of “resample with dataset”





buoyantBoussinesqPimpleFOAM/createFields.H

```
Info<< "Calculating field g.h\n" << endl;
  volScalarField gh("gh", g & mesh.C());
  surfaceScalarField ghf("ghf", g & mesh.Cf());
```

g & mesh.C()

Inner product
Programmers guide

Vector of cell center

buoyantBoussinesqPimpleFOAM.C

```
#include "fvCFD.H"
#include "singlePhaseTransportModel.H"
#include "RASModel.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

int main(int argc, char *argv[])
{
    #include "setRootCase.H"
    #include "createTime.H"
    #include "createMesh.H"
    #include "readGravitationalAcceleration.H"
    #include "createFields.H"
    #include "initContinuityErrs.H"
    #include "readTimeControls.H"
    #include "CourantNo.H"
    #include "setInitialDeltaT.H"

    // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
```

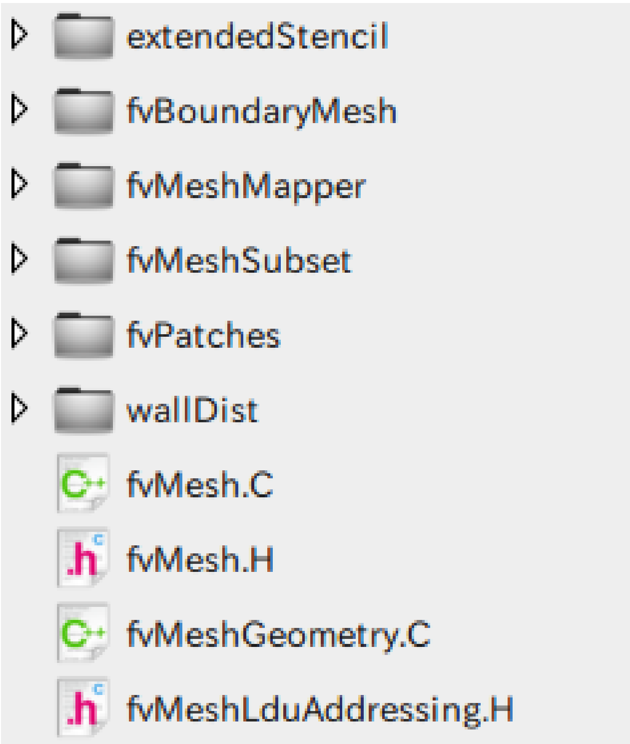
createMesh.H

```
Foam::fvMesh mesh
  (
     Foam::IOobject
     (
        Foam::fvMesh::defaultRegion,
        runTime.timeName(),
        runTime,
        Foam::IOobject::MUST_READ
     )
  );
```

find -name fvMesh*
Then,
src/finite/fvMesh/fvMesh.C
src/finite/fvMesh/fvMesh.H
• • •

src/finite/fvMesh

src/finite/fvMesh/



grep -rn ‘magsf’ * →

```
ogashin@node12:/opt/openfoam170/src/finiteVolume/fvMesh$ grep -rn
'magSf' *
fvMesh.C:77:    deleteDemandDrivenData(magSfPtr_);
fvMesh.C:161:    magSfPtr_(NULL),
fvMesh.C:253:    magSfPtr_(NULL),
fvMesh.C:283:    magSfPtr_(NULL),
fvMesh.H:111:        mutable surfaceScalarField* magSfPtr_;
fvMesh.H:281:            const surfaceScalarField& magSf() const;
fvMeshGeometry.C:87:    if (magSfPtr_)
fvMeshGeometry.C:97:    magSfPtr_ = new surfaceScalarField
fvMeshGeometry.C:101:            "magSf",
fvMeshGeometry.C:355:const surfaceScalarField& fvMesh::magSf() const
fvMeshGeometry.C:357:    if (!magSfPtr_)
fvMeshGeometry.C:362:    return *magSfPtr_;
fvPatches/fvPatch/fvPatch.C:128:    return Sf()/magSf();
fvPatches/fvPatch/fvPatch.C:138:const scalarField& fvPatch::magSf() const
fvPatches/fvPatch/fvPatch.C:140:    return
boundaryMesh().mesh().magSf().boundaryField()[index()];
fvPatches/fvPatch/fvPatch.H:208:            const scalarField& magSf() const;
fvPatches/constraint/cyclic/cyclicFvPatch.C:46:    const scalarField& magFa =
magSf();
wallDist/nearWallDistNoSearch.C:54:                =
mesh_.magSf().boundaryField()[patchI];
wallDist/reflectionVectors.C:66:
/mesh.magSf().boundaryField()[patchi];
```

fvMeshGeometory.C

```
const surfaceScalarField& fvMesh::magSf() const
{
    if (!magSfPtr_)
    {
        makeMagSf();
    }

    return *magSfPtr_;
}


const volVectorField& fvMesh::C() const
{
    if (!CPtr_)
    {
        makeC();
    }

    return *CPtr_;
}
```

Encoding software “ffmpeg”

ffmpeg
http://www.ffmpeg.org/

- free software
- able to encode single pictures into movie
  e.g paraview animation
- how to use
  1. prepare sequenitial picture
     test.0001.png
     test.0002.png
     .
     .
  2. move to the directory in terminal
  3. Type follwing command
     <normal speed>
     ffmpeg -sameq -i test.%4d.png test.mp4

     <Ten times faster>
     ffmpeg -sameq -r 10 -i test.%4d.png test.mp4