

HyperCFD
Fertinaz Yazılım Ltd.

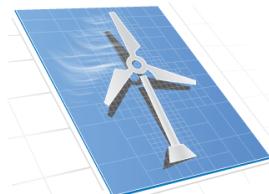
OpenFOAM Workshop Teknopark İstanbul

02 Capabilities
January 2017



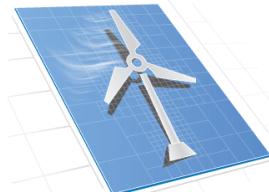
HyperCFD – Disclaimer

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OpenFOAM and OpenCFD trademarks.



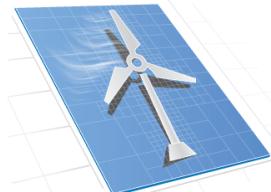
HyperCFD – OpenFOAM: Some Tricks

- OpenFOAM runs on Linux so make your life easier and use aliases, shell scripts:
 - Type `run` instead of `cd ~/OpenFOAM/OpenFOAM-4.0.x/username-4.0.x/run`
 - See `~/OpenFOAM/OpenFOAM-4.0.x/etc/config.sh` for all aliases
- OpenFOAM comes with lots of useful functions:
 - Compute y^+ with `simpleFoam -postprocess -func -yPlus`
 - In the older versions it was `yPlusRAS` or `yPlusLES`
 - Type `util` to see utilities and check them
- OpenFOAM supports parallel processing:
 - `mpirun -np 4 snappyHexMesh -parallel`
 - `mpirun -hosts myhostfile simpleFoam -parallel`



HyperCFD – OpenFOAM: Some Tricks

- Grep schemes, values, errors etc.:
 - Type `src` and then `grep -rn "divDevRhoRef"` *
- Also there is an OpenFOAM function `foamSearch`:
 - default ddtSchemes entries in the fvSchemes in heat transfer tutorials
 - `foamSearch -c $WM_PROJECT_DIR/tutorials/heatTransfer ddtSchemes.default fvSchemes`
- Use sed and awk
 - Change an existing keyword to mykeyword
 - `sed -i s/keyword/mykeyword/g sourceFile.C`
 - Replaces all occurrences in the specified file



HyperCFD – OpenFOAM: Some Tricks

- Use banana rule when list of available keys are needed.
 - Assume that you want to change a smoother in one of the linear solvers
 - Therefore you want to see a list of options
 - Banana is just a funny convention here. Technically you can use any word

```
fertinaz@fertinaz-M4800: ~/OpenFOAM/OpenFOAM-4.0/tutorials/incompressible/pisoFoam/ras/cavity
solvers
{
    p
    {
        solver      GAMG;
        tolerance   1e-06;
        relTol     0.1;
        smoother   banana;
    }

    pFinal
    {
        $p;
        tolerance   1e-06;
        relTol     0;
    }

    "(U|k|epsilon|omega|R|nuTilda)"
    {
        solver      smoothSolver;
        smoother   GaussSeidel;
        tolerance   1e-05;
        relTol     0;
    }
}
```

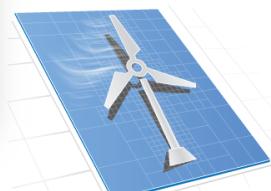
```
fertinaz@fertinaz-M4800: ~/OpenFOAM/OpenFOAM-4.0/tutorials/incompressible/pisoFoam/ras/cavity
Valid symmetric matrix smoothers are :

6
(
DIC
DICGaussSeidel
FDIC
GaussSeidel
nonBlockingGaussSeidel
symGaussSeidel
)

file: /home/fertinaz/OpenFOAM/OpenFOAM-4.0/tutorials/incompressible/pisoFoam/ras/cavity/system/fvSolution.solvers.p from line 22 to line 25.

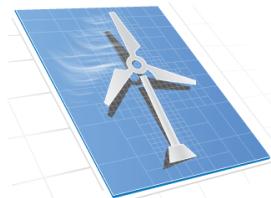
From function static Foam::autoPtr<Foam::lduMatrix::smoother> Foam::lduMatrix::smoother::New(const Foam::word&, const Foam::lduMatrix&, const Foam::FieldField<Foam::Field, double>&, const Foam::FieldField<Foam::Field, double>&, const lduInterfaceFieldPtrsList&, const Foam::dictionary&)
in file matrices/LduMatrix/LduMatrix/LduMatrixSmoothen.C at line 94.

FOAM exiting
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/tutorials/incompressible/pisoFoam/ras/cavity$
```



HyperCFD – OpenFOAM: Some Tricks

- You can use trick in many cases:
- Initial conditions and Boundary conditions
- Discretization schemes
- Linear solvers, preconditioners and smoothers
- Simulation settings
- Post-processing options

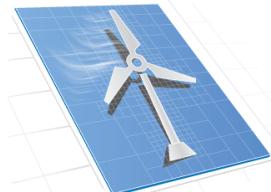


HyperCFD – OpenFOAM Utilities and Capabilities

- Some pre-processing tools comes with OpenFOAM
- Surface mesh manipulation

```
fertinaz@fertinaz-M4800:~$ surface
surfaceAdd          surfaceFeatureExtract    surfaceMeshImport      surfaceSplitByTopology
surfaceAutoPatch    surfaceFind             surfaceMeshInfo        surfaceSplitNonManifolds
surfaceBooleanFeatures surfaceHookUp          surfaceMeshTriangulate surfaceSubset
surfaceCheck         surfaceInertia          surfaceOrient          surfaceToPatch
surfaceClean         surfaceLambdaMuSmooth   surfacePointMerge     surfaceTransformPoints
surfaceCoarsen       surfaceMeshConvert    surfaceRedistributePar surfaceRefineRedGreen
surfaceConvert       surfaceMeshConvertTesting surfaceSplitByPatch
surfaceFeatureConvert surfaceMeshExport
fertinaz@fertinaz-M4800:~$ surface
```

- You can apply lots of operations:
 - Surface transformation
 - Surface conversion
- You can also create zones of mesh elements
 - cellSet, cellZoneSet, faceZoneSet, ...



HyperCFD – OpenFOAM Utilities and Capabilities

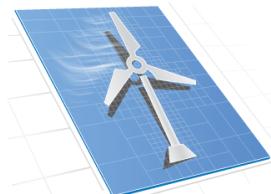
- surfaceCheck example

```
s.coarse$ surfaceCheck constant/triSurface/terrain.ofw.stl
/*
| ====== |
| \ \ / F ield | OpenFOAM: The Open Source CFD Toolbox
| \ \ / O peration | Version: 4.0
| \ \ / A nd | Web: www.OpenFOAM.org
| \ \ / M anipulation |
\*-----*/
Build : 4.0
Exec  : surfaceCheck constant/triSurface/terrain.ofw.stl
Date  : Jan 16 2017
Time  : 14:49:15
Host   : "fertinaz-M4800"
PID    : 22774
Case   : /home/fertinaz/OpenFOAM/fertinaz-4.0/run/turbineSiting.tests/sancak_yah
coarse
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Allowing user-supplied system call operations

// * * * * *
Reading surface from "constant/triSurface/terrain.ofw.stl" ...

Statistics:
Triangles   : 1568664
Vertices    : 388024
Bounding Box : (707000 4.197e+06 480) (743000 4.233e+06 2480)

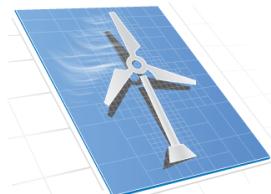
Region  Size
----- -----
```



HyperCFD – OpenFOAM Utilities and Capabilities

- Mesh manipulation and conversion

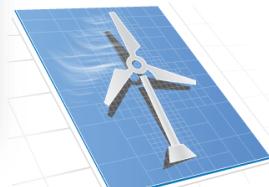
```
fertinaz@fertinaz-M4800: ~/OpenFOAM/OpenFOAM-4.0/applications/utilities/mesh/conversion
fertinaz@fertinaz-M4800:~$ OF40
fertinaz@fertinaz-M4800:~$ cd $FOAM
$FOAM_APP      $FOAM_INST_DIR    $FOAM_RUN        $FOAM_SITE_LIBBIN  $FOAM_USER_APPBIN
$FOAM_APPBIN   $FOAM_JOB_DIR     $FOAM_SETTINGS   $FOAM_SOLVERS    $FOAM_USER_LIBBIN
$FOAM_ETC       $FOAM_LIBBIN     $FOAM_SIGFPE     $FOAM_SRC       $FOAM_UTILITIES
$FOAM_EXT_LIBBIN $FOAM_MPI      $FOAM_SITE_APPBIN $FOAM_TUTORIALS $FOAM_HEX_MESH
fertinaz@fertinaz-M4800:~$ cd $FOAM_UTILITIES
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities$ ls
mesh  miscellaneous  parallelProcessing  postProcessing  preProcessing  surface  thermophysical
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities$ cd mesh/
advanced/  conversion/  generation/  manipulation/
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities$ cd mesh/conversion/
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities/mesh/conversion$ ls
ansysToFoam      fluentMeshToFoam  gambitToFoam    mshToFoam      sammToFoam      vtkUnstructuredToFoam
cfx4ToFoam       foamMeshToFluent gmshToFoam     netgenNeutralToFoam star3ToFoam     writeMeshObj
datToFoam        foamToStarMesh   ideasUnvToFoam Optional      star4ToFoam
fluent3DMeshToFoam foamToSurface   kivaToFoam    plot3dToFoam   tetgenToFoam
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities/mesh/conversion$
```



HyperCFD – OpenFOAM Utilities and Capabilities

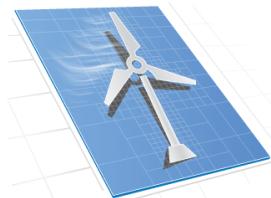
- Sample mesh conversion from Fluent: fluent3DMeshToFoam kaplan_ascii.msh

```
1 /*-----*\n2 | ======\n3 | \\" / F ield | OpenFOAM: The Open Source CFD Toolbox\n4 | \\" / O peration | Version: 2.2.x\n5 | \\" / A nd | Web: www.OpenFOAM.org\n6 | \\"/ M anipulation |\n7 *-\n8 Build : 2.2.x-d1a855ac2674\n9 Exec  : fluent3DMeshToFoam kaplan_ascii.msh\n10 Date  : Jun 03 2014\n11 Time  : 16:24:09\n12 Host   : extinazz-workstation\n13 PID    : 17552\n14 Case   : /home/extinazz/OpenFOAM/extinazz-2.2.x/run/ImportAnsysMesh3D-Kaplan\n15 nProcs : 1\n16 sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).\n17 fileModificationChecking : Monitoring run-time modified files using timeStampMaster\n18 allowSystemOperations : Disallowing user-supplied system call operations\n19\n20 // * * * * *\n21 Create time\n22\n23 Dimension of grid: 3\n24 Number of points: 243762\n25 Number of faces: 1968409\n26 Number of cells: 911810\n27 PointGroup: 9 start: 0 end: 213878. Reading points...done.\n28 PointGroup: 10 start: 213879 end: 243761. Reading points...done.\n29 FaceGroup: 1 start: 0 end: 1082. Reading uniform faces...done.\n30 FaceGroup: 2 start: 1083 end: 4445. Reading uniform faces...done.\n31 FaceGroup: 3 start: 4446 end: 1775879. Reading mixed faces...done.\n32 FaceGroup: 4 start: 1775880 end: 1874583. Reading uniform faces...done.\n33 FaceGroup: 5 start: 1874584 end: 1908650. Reading uniform faces...done.\n34 FaceGroup: 11 start: 1908651 end: 1954614. Reading uniform faces...done.\n35 FaceGroup: 12 start: 1954615 end: 1954650. Reading uniform faces...done.\n36 FaceGroup: 13 start: 1954651 end: 1955871. Reading uniform faces...done.\n37 FaceGroup: 14 start: 1955872 end: 1966909. Reading uniform faces...done.\n38 FaceGroup: 15 start: 1966910 end: 1968372. Reading uniform faces...done.\n39 FaceGroup: 16 start: 1968373 end: 1968408. Reading uniform faces...done.\n40 CellGroup: 6 start: 0 end: 843623 type: 1\n41 CellGroup: 7 start: 843624 end: 894130 type: 1\n42 CellGroup: 8 start: 894131 end: 911809 type: 1\n43 Zone: 1 name: interior-wake_adoped_kaplan_series-split1_1_-wake_adoped_kaplan_series-split1_3_ type: interior. Reading zone data...done.\n44 Zone: 2 name: interior-wake_adoped_kaplan_series-split1_1_-wake_adoped_kaplan_series-split1_2_ type: interior. Reading zone data...done.\n45 Zone: 3 name: interior-wake_adoped_kaplan_series-split1_1_ type: interior. Reading zone data...done.\n46 Zone: 4 name: interior-wake_adoped_kaplan_series-split1_2_ type: interior. Reading zone data...done.\n47 Zone: 5 name: interior-wake_adoped_kaplan_series-split1_3_ type: interior. Reading zone data...done.\n48 Zone: 6 name: wake_adoped_kaplan_series-split1_1_ type: fluid. Reading zone data...done.\n49 Zone: 7 name: wake_adoped_kaplan_series-split1_2_ type: fluid. Reading zone data...done.\n50 Zone: 8 name: wake_adoped_kaplan_series-split1_3_ type: fluid. Reading zone data...done.\n51 Zone: 11 name: propeller type: wall. Reading zone data...done.\n52 Zone: 12 name: inlet type: velocity-inlet. Reading zone data...done.\n53 Zone: 13 name: stationary_in type: wall. Reading zone data...done.\n54 Zone: 14 name: rotating type: wall. Reading zone data...done.\n55 Zone: 15 name: stationary_out type: wall. Reading zone data...done.\n56 Zone: 16 name: outlet type: pressure-outlet. Reading zone data...done.\n57\n58 FINISHED LEXING\n59\n60 Creating patch 0 for zone: 11 name: propeller type: wall\n61 Creating patch 1 for zone: 12 name: inlet type: velocity-inlet\n62 Creating patch 2 for zone: 13 name: stationary_in type: wall\n63 Creating patch 3 for zone: 14 name: rotating type: wall\n64 Creating patch 4 for zone: 15 name: stationary_out type: wall\n65 Creating patch 5 for zone: 16 name: outlet type: pressure-outlet\n66 Creating cellZone 0 name: wake_adoped_kaplan_series-split1_1_ type: fluid\n67 Creating cellZone 1 name: wake_adoped_kaplan_series-split1_2_ type: fluid\n68 Creating cellZone 2 name: wake_adoped_kaplan_series-split1_3_ type: fluid\n69 Creating faceZone 0 name: interior-wake_adoped_kaplan_series-split1_1_-wake_adoped_kaplan_series-split1_3_ type: interior\n70 Creating faceZone 1 name: interior-wake_adoped_kaplan_series-split1_1_-wake_adoped_kaplan_series-split1_2_ type: interior\n71 Creating faceZone 2 name: interior-wake_adoped_kaplan_series-split1_1_ type: interior\n72 Creating faceZone 3 name: interior-wake_adoped_kaplan_series-split1_2_ type: interior\n73 Creating faceZone 4 name: interior-wake_adoped_kaplan_series-split1_3_ type: interior\n74 faceZone from Fluent indices: 0 to: 1082 type: interior\n75 faceZone from Fluent indices: 1083 to: 4445 type: interior\n76 faceZone from Fluent indices: 4446 to: 1775879 type: interior\n77 faceZone from Fluent indices: 1775880 to: 1874583 type: interior\n78 faceZone from Fluent indices: 1874584 to: 1908650 type: interior\n79 patch 0 from Fluent indices: 1908651 to: 1954614 type: wall\n80 patch 1 from Fluent indices: 1954615 to: 1954650 type: velocity-inlet\n81 patch 2 from Fluent indices: 1954651 to: 1955871 type: wall\n82 patch 3 from Fluent indices: 1955872 to: 1966909 type: wall\n83 patch 4 from Fluent indices: 1966910 to: 1968372 type: wall\n84 patch 5 from Fluent indices: 1968373 to: 1968408 type: pressure-outlet\n85\n86 Writing mesh to "/home/extinazz/OpenFOAM/extinazz-2.2.x/run/ImportAnsysMesh3D-Kaplan/constant/region0"\n87\n88 End
```



HyperCFD – OpenFOAM Utilities and Capabilities

- Some postprocessing tools comes with OpenFOAM
- Application: `Pe`
Calculates the Peclet number Pe from the flux ϕ and writes the maximum value
- Application: `vorticity`
Calculates and writes the vorticity of velocity field \mathbf{U}
- Application: `sample`
Creates sampling points/regions using interpolation on a given set

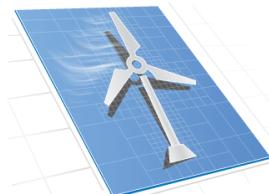


HyperCFD – OpenFOAM Third Party Libraries

- Some of the Third party code helps a lot. To name a few:
- pyFoam: Very handy Python library. It provides lots of scripts to prepare cases and postprocessing. I use it mostly to plot residuals during run-time.

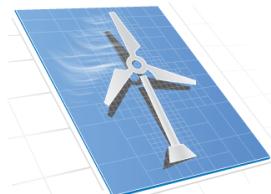
pyFoamPlotWatcher.py log.simpleFoam

```
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities/postProcessing/postProcess$ pyFoam
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities/postProcessing/postProcess$ pyFoam
pyFoamAddCaseDataToDatabase.py      pyFoamDecompose.py          pyFoamPrintData2DStatistics.py
pyFoamAddEmptyBoundary.py          pyFoamDisplayBlockMesh.py    pyFoamPVLoadState.py
pyFoamAPoMaFoXiiQt.py             pyFoamDumpConfiguration.py pyFoamPVSnapshot.py
pyFoamBench.py                    pyFoamDumpRunDatabaseToCSV.py pyFoamReadDictionary.py
pyFoamBuildHelper.py              pyFoamEchoDictionary.py    pyFoamRedoPlot.py
pyFoamCaseBuilder.py              pyFoamEchoPickledApplicationData.py pyFoamRunAtMultipleTimes.py
pyFoamCaseReport.py               pyFoamExecute.py          pyFoamRunner.py
pyFoamChangeBoundaryName.py       pyFoamFromTemplate.py     pyFoamSamplePlot.py
pyFoamChangeBoundaryType.py       pyFoamInitVCSCase.py     pyFoamSGECommand.py
pyFoamClearBoundaryValue.py       pyFoamIPythonNotebook.py   pyFoamStandardLogAnalyzer.py
pyFoamClearCase.py                pyFoamJoinCSV.py         pyFoamSteadyRunner.py
pyFoamClearEmptyBoundaries.py    pyFoamListCases.py        pyFoamSTLUtility.py
pyFoamClearInternalField.py       pyFoamMeshUtilityRunner.py pyFoamSurfacePlot.py
pyFoamCloneCase.py                pyFoamMetaServer.py       pyFoamSymlinkToFile.py
pyFoamClusterTester.py            pyFoamModifyGGIBoundary.py pyFoamTestConfiguration.py
pyFoamComparator.py               pyFoamNetList.py         pyFoamTimelinePlot.py
pyFoamCompareDictionary.py       pyFoamNetShell.py        pyFoamUpdateDictionary.py
pyFoamCompressCaseFiles.py       pyFoamPackCase.py        pyFoamUpgradeDictionariesTo17.py
pyFoamConvertToCSV.py             pyFoamPlotRunner.py       pyFoamUpgradeDictionariesTo20.py
pyFoamCopyLastToFirst.py          pyFoamPlotWatcher.py     pyFoamUtilityRunner.py
pyFoamCreateBoundaryPatches.py   pyFoamPotentialRunner.py   pyFoamVersion.py
pyFoamCreateModuleFile.py         pyFoamPrepareCase.py     pyFoamWriteDictionary.py
fertinaz@fertinaz-M4800:~/OpenFOAM/OpenFOAM-4.0/applications/utilities/postProcessing/postProcess$ pyFoam
```



HyperCFD – OpenFOAM Third Party Libraries

- Some of the Third party code helps a lot. To name a few:
- Paraview and VTK:
 - VTK is a C++ library just like OpenFOAM. It is designed for scientific visualization and provides lots of filters for visualization.
 - Example: In my own project, I apply surface triangulation to the point clouds using Delaunay triangulation algorithm that comes with VTK. Then snappy snaps .stl from the computational grid.
 - OpenFOAM outputs are supported by ParaView which is a GUI developed using Qt and VTK. It is the most natural way of visualizing OpenFOAM applications. In the ThirdParty directory there is also a wrap of ParaView which is invoked by `paraFoam` command.
- swak4Foam: SWiss Army Knife for OpenFOAM
 - Ideal for parameterizing IC's and BC's
 - Example: Wave generation at the inlets



HyperCFD – OpenFOAM Utilities and Capabilities

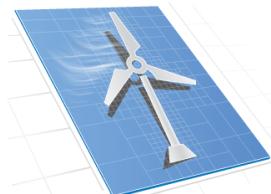
- Lets visualize bad mesh elements using some OpenFOAM commands and ParaView
- Run `checkMesh` to control the quality of the mesh

```
Checking geometry...
Overall domain bounding box (700000 4.195e+06 750) (750000 4.235e+06 6000)
Mesh has 3 geometric (non-empty/wedge) directions (1 1 1)
Mesh has 3 solution (non-empty) directions (1 1 1)
Boundary openness (1.25094e-17 5.17758e-17 0) OK.
Max cell openness = 3.2905e-16 OK.
Max aspect ratio = 15.2084 OK.
Minimum face area = 123.6. Maximum face area = 202468. Face area magnitudes OK.
Min volume = 33210.3. Max volume = 2.89489e+07. Total volume = 8.83394e+12. Cell volumes OK.
Mesh non-orthogonality Max: 65.3141 average: 17.6789
Non-orthogonality check OK.
Face pyramids OK.

***Max skewness = 6.85769, 224 highly skew faces detected which may impair the quality of the results
<<Writing 224 skew faces to set skewFaces
Coupled point location match (average 0) OK.

Failed 1 mesh checks.

End
```



HyperCFD – OpenFOAM Utilities and Capabilities

- It complained about the number of skewed cells
- So let's detect their locations and visualize them
 - So lets send them to a VTK file so that ParaView can read them

```
foamToVTK -faceSet skewFaces
```
 - Output is written to the following file:
`/VTK/skewFaces/skewFaces_0.vtk`
- You can now open this file using ParaView

