Introduction to 2018 OpenFOAM® Release

1 Nov 2018
Geon-Hong Kim
Engineer, Ph.D.
Hyundai Heavy Industries Co., Ltd.
NINANO COMPANY Inc.

NINANO COMPANY

Special Thanks to…

Hyundai Heavy Industries Co., Ltd.

11개월의 휴직을 승인하여 다양한 활동과 경험을 할 수 있도록 배려해 주신 점에 대해 진심으로 감사드립니다.

그리고 지금도 저를 대신하여 업무를 수행하고 있을 연구원들에게 심심한 사과의 말씀 전합니다.

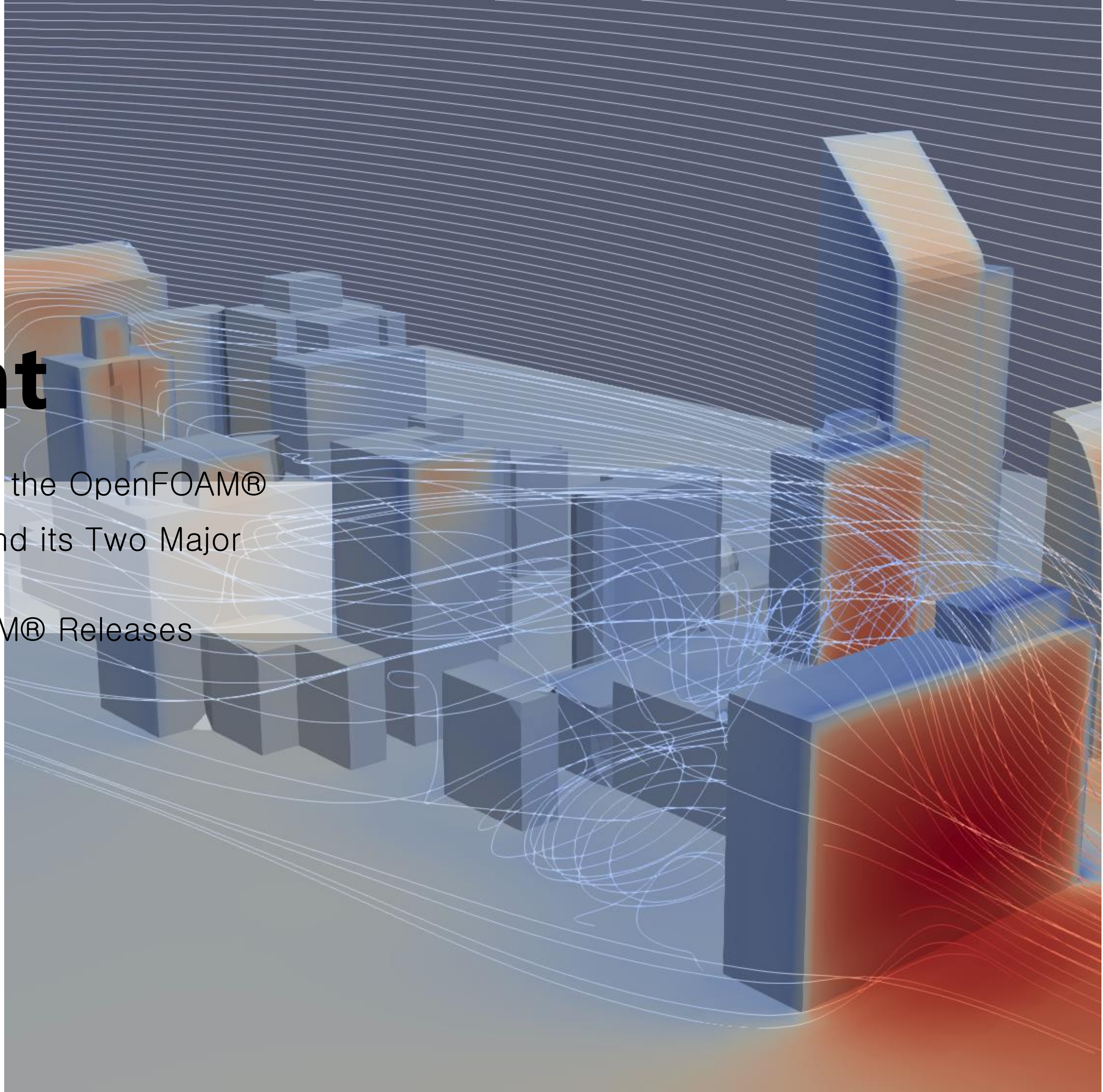
2018 7th OKUCC

— Content

01 Short History of the OpenFOAM®

02 OpenFOAM® and its Two Major Folks

03 2018 OpenFOAM® Releases



2018 7th OKUCC

Short History of the OpenFOAM®

OpenFOAM® Chronicles

1989

FOAM

The original FOAM software was created by Henry Weller.

2000

Nabla ltd.

H. Jasak and H. Weller started a company called Nabla ltd. and was doing all FOAM development.

2004

OpenFOAM

The FOAM was modified, improved and released as open-source by OpenCFD (10 Dec 2014 - OpenFOAM-1.0)


International OpenFOAM Workshop


Hrvoje Jasak

2011

SGI and OpenFOAM Foundation

SGI bought OpenCFD and the OpenFOAM Foundation was created.

2012

ESI

OpenCFD was bought by ESI in 2012. Later, ESI released OpenFOAM+

2015

OpenFOAM Foundation

In 2014, Henry Weller left OpenCFD/ESI and remains as director of the Foundation. Development continues by the Foundation.

Major Contributors

A. Henry Weller
B. Charlie Hill
C. Hrvoje Jasak
D. Chris Greenshields
E. David Gosman

NINANO COMPANY

OpenFOAM-1.0 Contributors

Henry Weller
Hrvoje Jasak
Chris Greenshields
Mattijs Janssens
Niklas Nordin
Eugene De Villiers
Gavin Tabor
Zeljko Tukovic
Tommaso Lucchini
David Hill
Niklas Wikstrom
Hilary Spencer
Andy Heather
Henrik Rusche

Eugene De Villiers

Managing Director
Engys Ltd.

Gavin Tabor

Associate Professor
University of Exeter

Henrik Rusche

Wikki Ltd.

— The beginning of the story…

In the same way that Spalding's group eventually spawned multiple CFD codes including TEACH, the current leading Open Source CFD code FOAM (now OpenFOAM) was developed by Henry Weller during his time in David Gosman's research team.

The first lines of FOAM were written by a guy called Charlie Hill as a part of his PhD into computer graphics and presentation of CFD results on modern workstations in early 1990s in prof. Gosman's group. The code was converted/developed into a basic CFD code in late 1993 and the first-ever simulation was a shedding flow around a cylinder in December 1993.

OpenFOAM® and its Two Major Folks

Major Development Groups

OpenFOAM

www.openfoam.org
Supported by OpenFOAM Foundation
Latest release: v6

OpenFOAM+

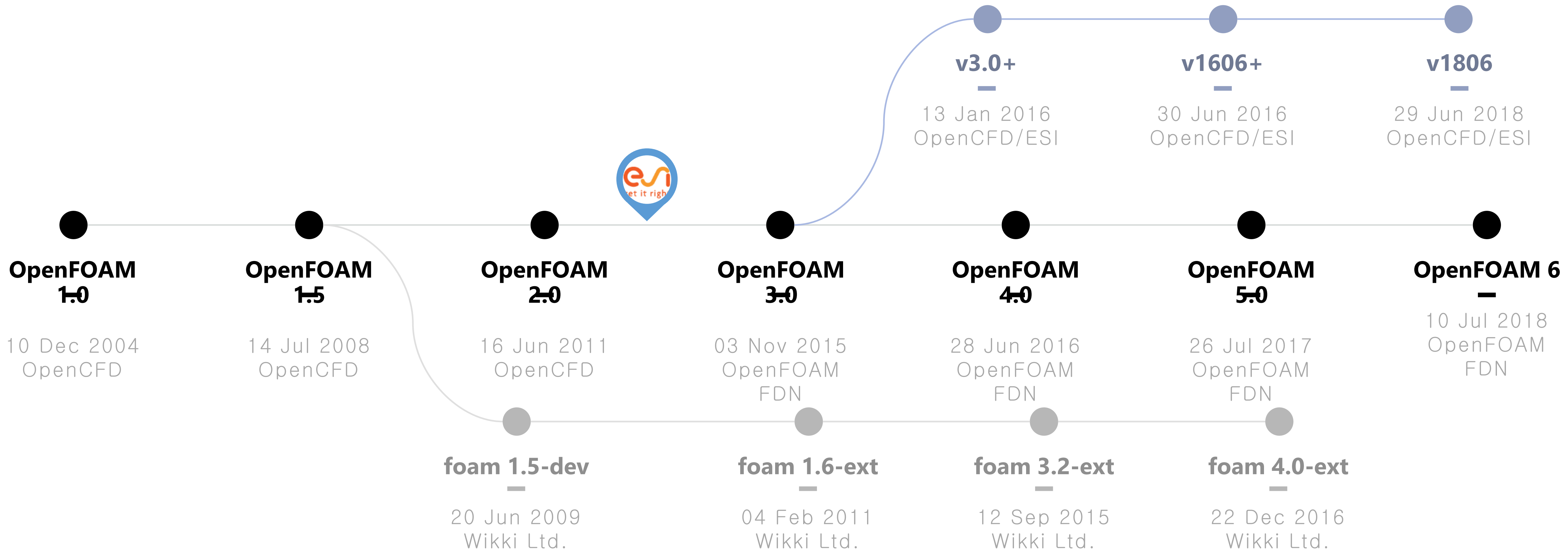
www.openfoam.com
Supported by ESI group
Latest release: v1806

foam-extend

foam-extend.fsb.de
Supported by Wikki
Latest release: 4.0

Release Map

Release of the OpenFOAM® by three major development groups



OpenFOAM vs foam-extend

Why was the foam-extend separated from the mainstream of the OpenFOAM?

Hrvoje Jasak

Co-author of OpenFOAM (known as)

I started working on the code around September 1993, firstly to have visualisation capability for an old heap of Fortran and then I switched (with Henry) to FOAM for my PhD work.

All the basic development stuff happened in the next 3-4 years and Jasak and Weller carried on working for full 11 years, developing this code together. The file signatures aren't really representative because up to about 2001 all headers named Weller as the author, even if he did not write them.

In 2000, Jasak and Weller started a company called Nabla Ltd which lasted until 2006 and was doing ALL FOAM development – I was the technical director. The estimate of the code base authorship at this point (Sep/2000) was 80% Weller and 20% Jasak – which I think was fair. At the end of Nabla, the estimate for the code base, with signatures etc was 60% Weller, 35% Jasak and 5% other authors (we had eg. FoamX), which was again fair.

With the start of OpenCFD, Weller and Greenshields start pretending nothing of this ever happened and the code "just appeared out of nowhere". The file signatures from Jasak are deleted from the code (see GPL violation) and OpenCFD claims ownership – which they do not actually hold.

2018 7th OKUCC

International OpenFOAM Workshop

Annual Event
Supported by Wikki Ltd.
Since 2006 in Zagreb, Croatia
openfoamworkshop.org

OpenFOAM Community Conferences

OpenFOAM Conference

Annual Event
Supported by ESI Group
Since 2013 in Frankfurt, Germany
www.esi-group.com

2018 OpenFOAM Releases

OPENFOAM 6

10 Jul 2018
Main stream of the OpenFOAM development
Core Team : Henry Weller, Chris Greenshields, Will Bainbridge

OpenFOAM® v1806

29 Jun 2018
The primary folk of the OpenFOAM development
Supported by ESI-OpenCFD

OPENFOAM 6

Key developments and features

Conjugate Heat Transfer improved usability	Rotating/Sliding Geometries robust AMI	Particle Tracking optimized/improved	Reacting Multiphase faster
Additional Models wave, turbulence etc.	New Boundary Conditions new freestream BCs	Function Objects ddt, scale	Further Tools foamInfo/foamGet

Boundary Conditions

New freestream BCs

—

Type freestream for velocity is replaced to freestreamVelocity and the freestream pressure requests the freestreamValue for freestreamPressure boundary condition.

The new version for the pressure is an outlet–inlet condition that uses the velocity orientation to continuously blend between zero gradient for normal inlet and fixed value for normal outlet flow

8 tutorials/incompressible/simpleFoam/airFoil2D/0/U

```diff
@@ -22,14 +22,14 @@ boundaryField
 {
     inlet
     {
-        type            freestream;
-        freestreamValue uniform (25.75 3.62 0);
+        type            freestreamVelocity;
+        freestreamValue $internalField;
     }

     outlet
     {
-        type            freestream;
-        freestreamValue uniform (25.75 3.62 0);
+        type            freestreamVelocity;
+        freestreamValue $internalField;
     }

     walls
```

2 tutorials/incompressible/simpleFoam/airFoil2D/0/p

```diff
@@ -23,11 +23,13 @@ boundaryField
     inlet
     {
         type            freestreamPressure;
+        freestreamValue $internalField;
     }

     outlet
     {
         type            freestreamPressure;
+        freestreamValue $internalField;
     }

     walls
```

```diff
@@ -64,14 +64,39 @@ int main(int argc, char *argv[])

     while (runTime.run())
     {
-        #include "readTimeControls.H"
+        #include "readControls.H"
         #include "CourantNo.H"
         #include "setDeltaT.H"

         runTime++;

         Info<< "Time = " << runTime.timeName() << nl << endl;

+        mesh.update();
+
+        #include "updateUf.H"
+
+        if (mesh.changing())
+        {
+            MRF.update();
+
+            if (correctPhi)
+            {
+                // Calculate absolute flux from the mapped surface velocity
+                phi = mesh.Sf() & Uf();
+
+                #include "correctPhi.H"
+
+                // Make the flux relative to the mesh motion
+                fvc::makeRelative(phi, U);
+            }
+
+            if (checkMeshCourantNo)
+            {
+                #include "meshCourantNo.H"
+            }
+        }
+
         // --- Pressure-velocity PIMPLE corrector loop
         while (pimple.loop())
         {
```

pimpleFoam.C

Meshes

Deprecated DyM Solvers

The dynamic mesh functionality in $DyMsolver has been merged into $solver and the $DyMsolver tutorials moved into the $solver tutorials directory.

One should specify staticFvMesh as the dynamicFvMesh in 'constant/dynamicMeshDict' for running a static case.

2018 7th OKUCC

Marine/Waves

New stokes5 and solitary wave models

Stokes 5th order wave model was added as well as solitary wave model of Dean and Dalrymple.

The generic base class for waves, waveModel has been modified (simplified).

2018 7th OKUCC

```
+ /*---------------------------------------------------------------------------*\
+   =========                 |
+   \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
+    \\    /   O peration     |
+     \\  /    A nd           | Copyright (C) 2017 OpenFOAM Foundation
+      \\/     M anipulation  |
+ -------------------------------------------------------------------------------
+ License
+     This file is part of OpenFOAM.
+
+     OpenFOAM is free software: you can redistribute it and/or modify it
+     under the terms of the GNU General Public License as published by
+     the Free Software Foundation, either version 3 of the License, or
+     (at your option) any later version.
+
+     OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
+     ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
+     FITNESS FOR A PARTICULAR PURPOSE.  See the GNU General Public License
+     for more details.
+
+     You should have received a copy of the GNU General Public License
+     along with OpenFOAM.  If not, see <http://www.gnu.org/licenses/>.
+
+ Class
+     Foam::waveModels::Stokes5
+
+ Description
+     Fifth-order wave model.
+
+     Reference:
+     \verbatim
+         "A Fifth Order Stokes Theory for Steady Waves"
+         J D Fenton
+         Journal of Waterway, Port, Coastal, and Ocean Engineering (1985),
+         Volume 111, Issue 2, Pages 216-234
+     \endverbatim
+
+ SourceFiles
+     Stokes5.C
+
+ \*---------------------------------------------------------------------------*/
+
+ #ifndef Stokes5_H
+ #define Stokes5_H
+
+ #include "Stokes2.H"
+
```

```
@@ -0,0 +1,218 @@
+ #!/bin/sh
+ #------------------------------------------------------------------------------
+ # =========                 |
+ # \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
+ #  \\    /   O peration     |
+ #   \\  /    A nd           | Copyright (C) 2018 OpenFOAM Foundation
+ #    \\/     M anipulation  |
+ #-------------------------------------------------------------------------------
+ # License
+ #     This file is part of OpenFOAM.
+ #
+ #     OpenFOAM is free software: you can redistribute it and/or modify it
+ #     under the terms of the GNU General Public License as published by
+ #     the Free Software Foundation, either version 3 of the License, or
+ #     (at your option) any later version.
+ #
+ #     OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
+ #     ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
+ #     FITNESS FOR A PARTICULAR PURPOSE.  See the GNU General Public License
+ #     for more details.
+ #
+ #     You should have received a copy of the GNU General Public License
+ #     along with OpenFOAM.  If not, see <http://www.gnu.org/licenses/>.
+ #
+ # Script
+ #     foamGet
+ #
+ # Description
+ #     Finds an example OpenFOAM case dictionary in $FOAM_ETC/caseDicts and
+ #     copies it into the respective case directory.
+ #
+ #-------------------------------------------------------------------------------
+ usage() {
+     cat<<USAGE
+
+ Usage: ${0##*/} [OPTIONS] <file>
+ options:
+   -case   | -c <dir>  specify case directory (default = local dir)
+   -ext    | -e <ext>  specify file extension, e.g -e cfg for files with ".cfg"
+   -help   | -h        print the usage
+   -no-ext | -n        specify file without extension
+   -target | -t <dir>  specify target directory (default = system)
+
```

New Scripts

foamGet to copy a configuration file

Uses sample configuration files in $FOAM_ETC/caseDicts, including utility configuration files and packaged function objects.

Files are copied into the system directory by default, otherwise a different target directory can be specified with –target|–t option.

OpenFOAM v6

The foamInfo and foamGet tools

CFD **Direct**

http://cfd.direct

OpenFOAM v1806

Key developments and features

Pre-processing new and improved	Numerics stabilisation	Solvers laser melting	Physical Models phase & mass models
Boundary Conditions fan, irregular waves	Post-processing Catalyst, sampling	Documentation solvers, fvOption	Bug Fixes snappyHexMesh

Consolidation of Moving Mesh Solvers

Moving mesh functionality has been incorporated into many of the static mesh solver applications from earlier releases

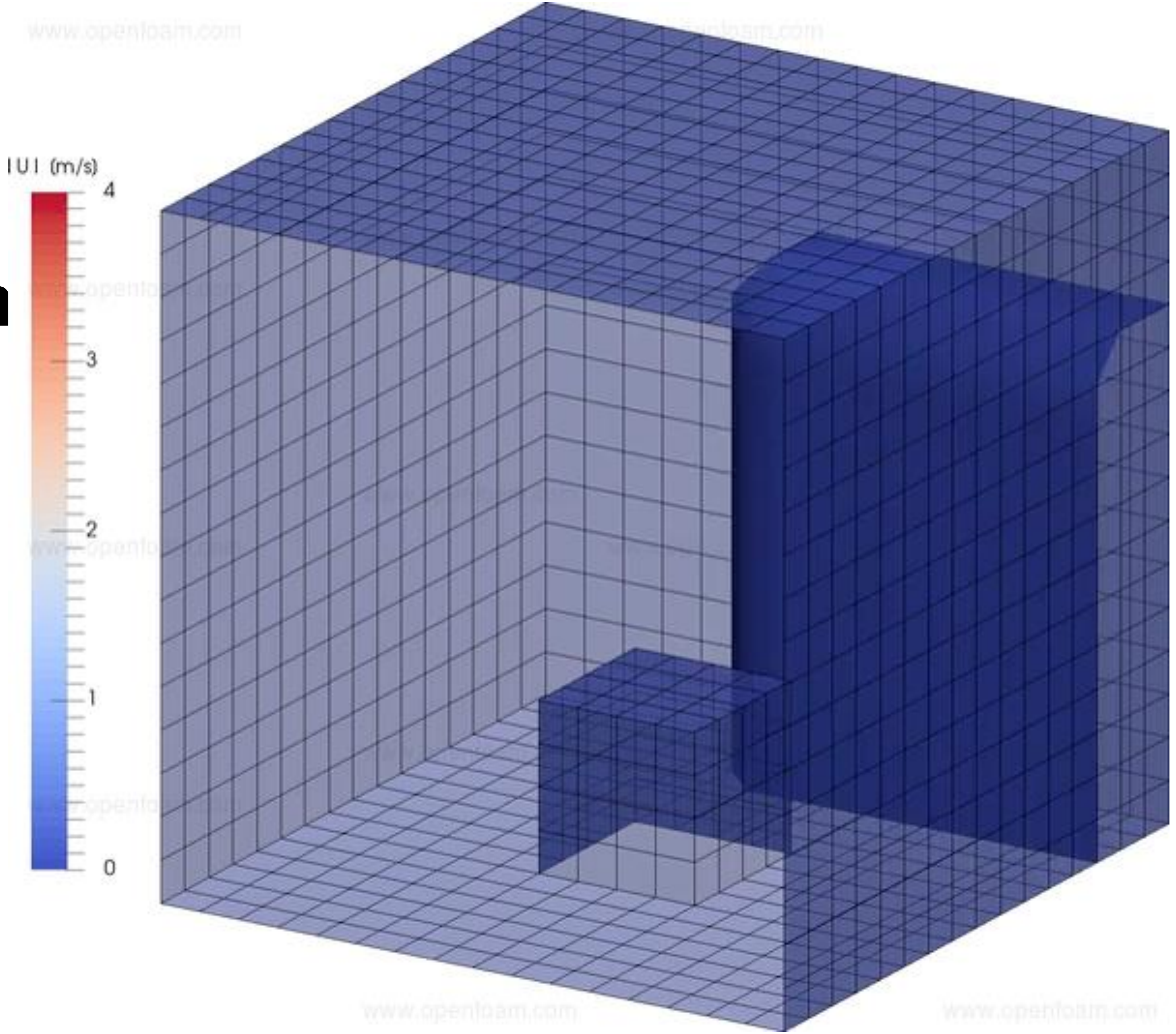
Old solver	New solver
pimpleDyMFoam	*pimpleFoam*
rhoPimpleDyMFoam	*rhoPimpleFoam*
interDyMFoam	*interFoam*
multiphaseInterDyMFoam	*multiphaseInterFoam*

NINANO COMPANY

Extended interIsoFoam solver

—

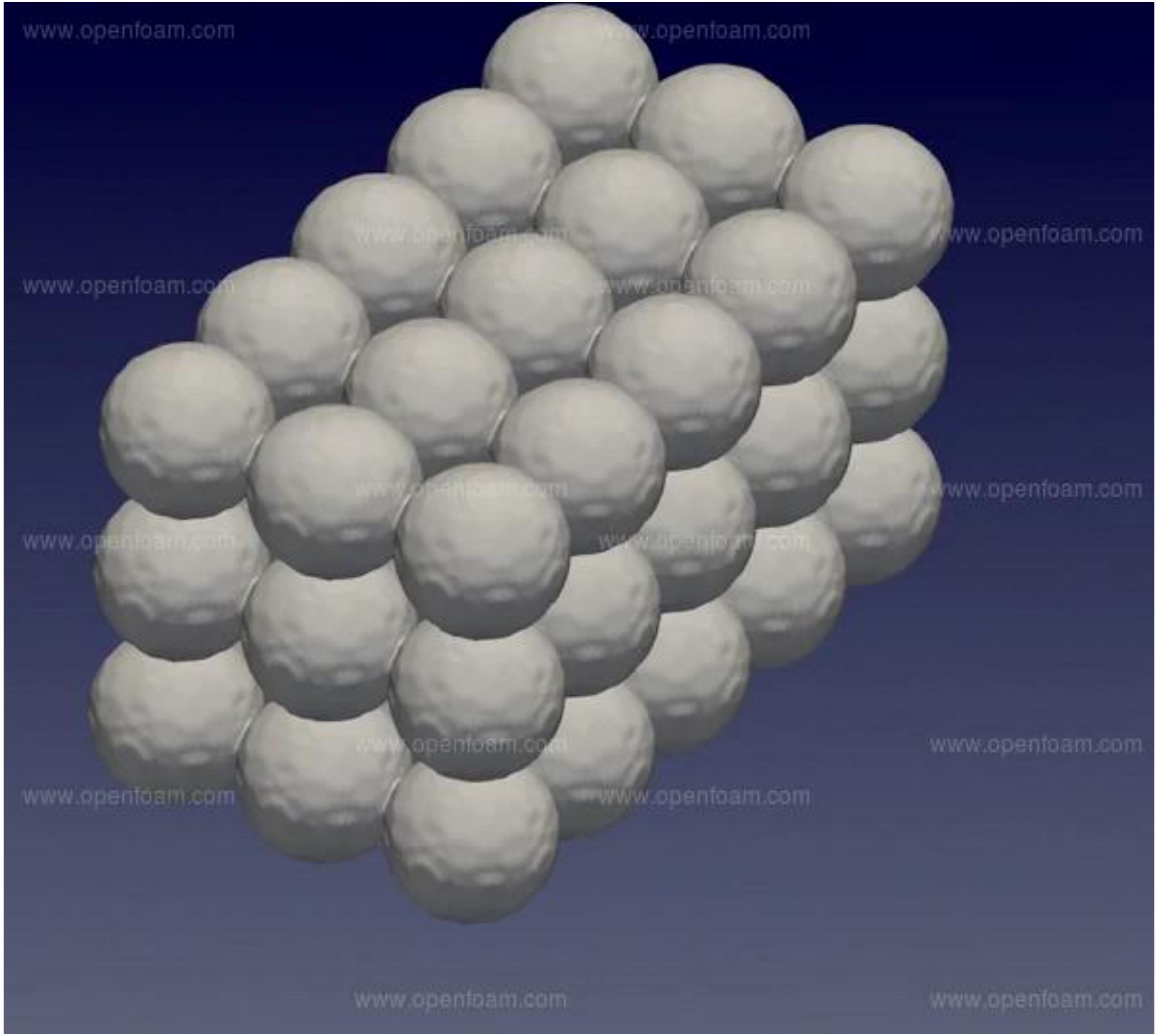
The interIsoFoam solver and its core isoAdvector library have been extended to work with dynamic meshes



NINANO COMPANY

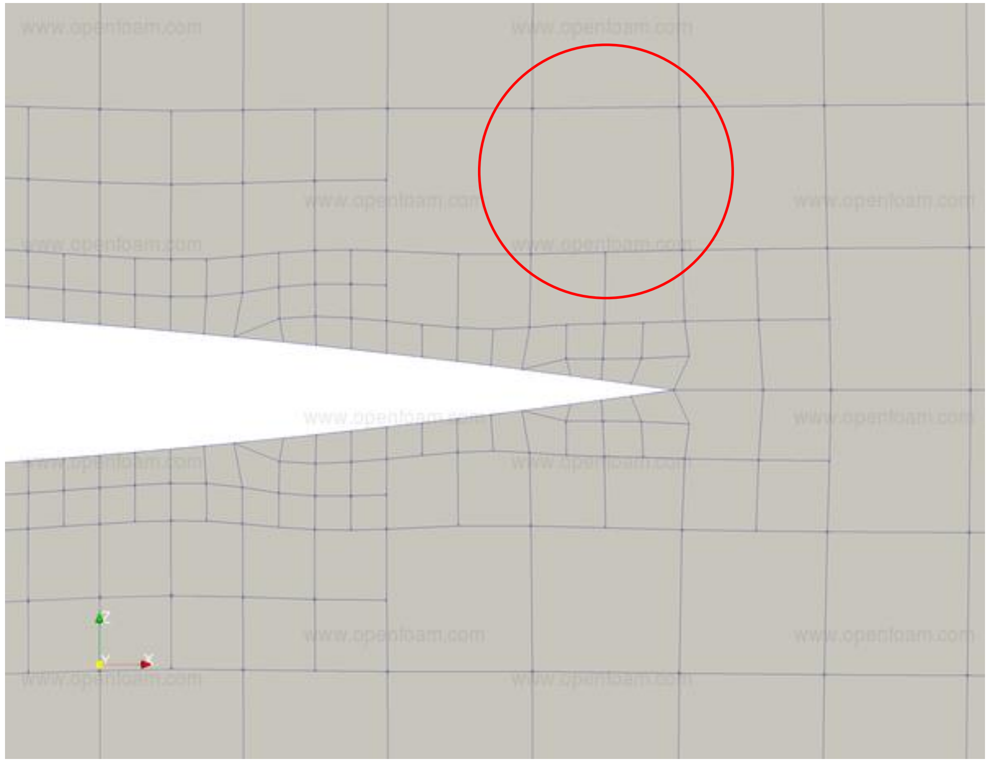
icoReacting-Multiphase-InterFoam

A multi-phase, multi-component incompressible solver based on a Volume Of Fluid (VOF) method with per-phase choice of thermodynamics model (sharing pressure and temperature).

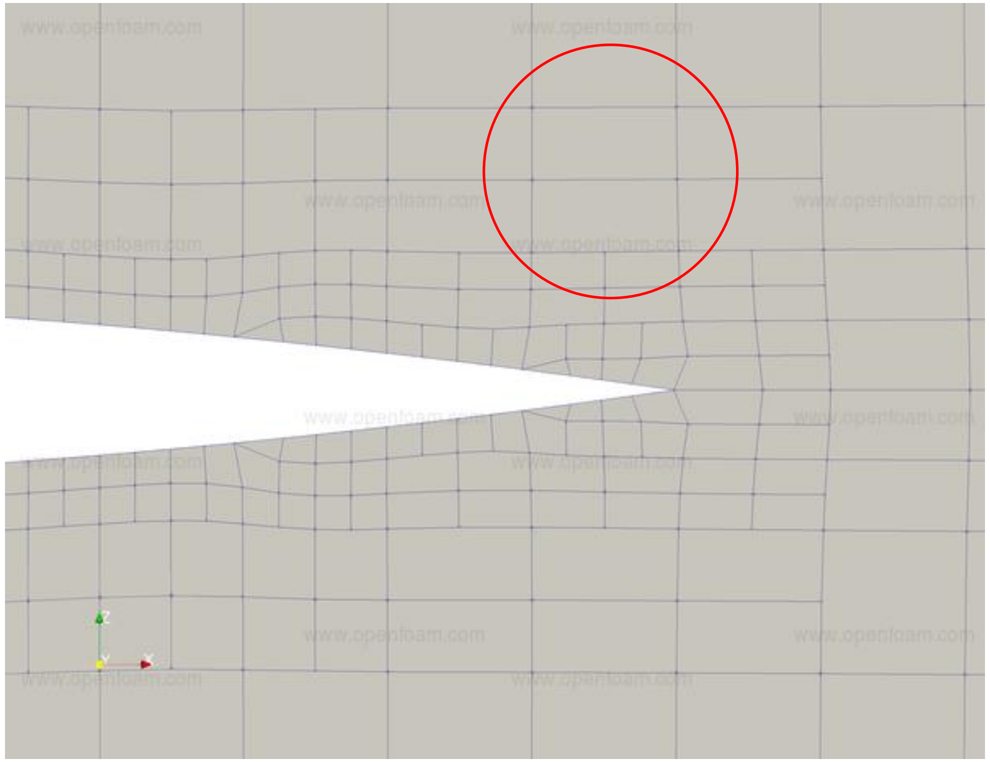


snappyHexMesh

snappyHexMesh supports additional directional refinement inside refinementRegions

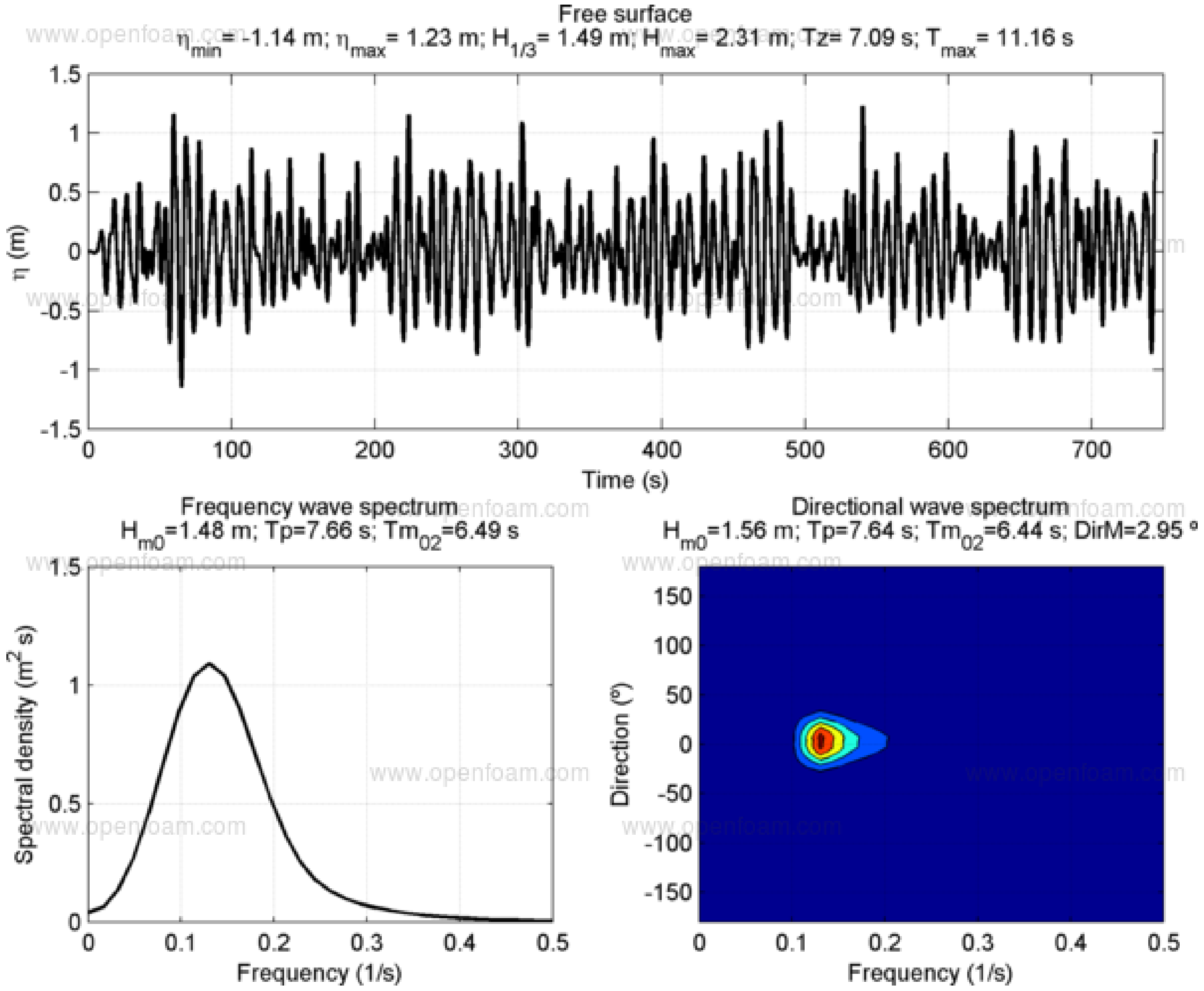


w/o directional refinement



w/ directional refinement

2018 7th OKUCC



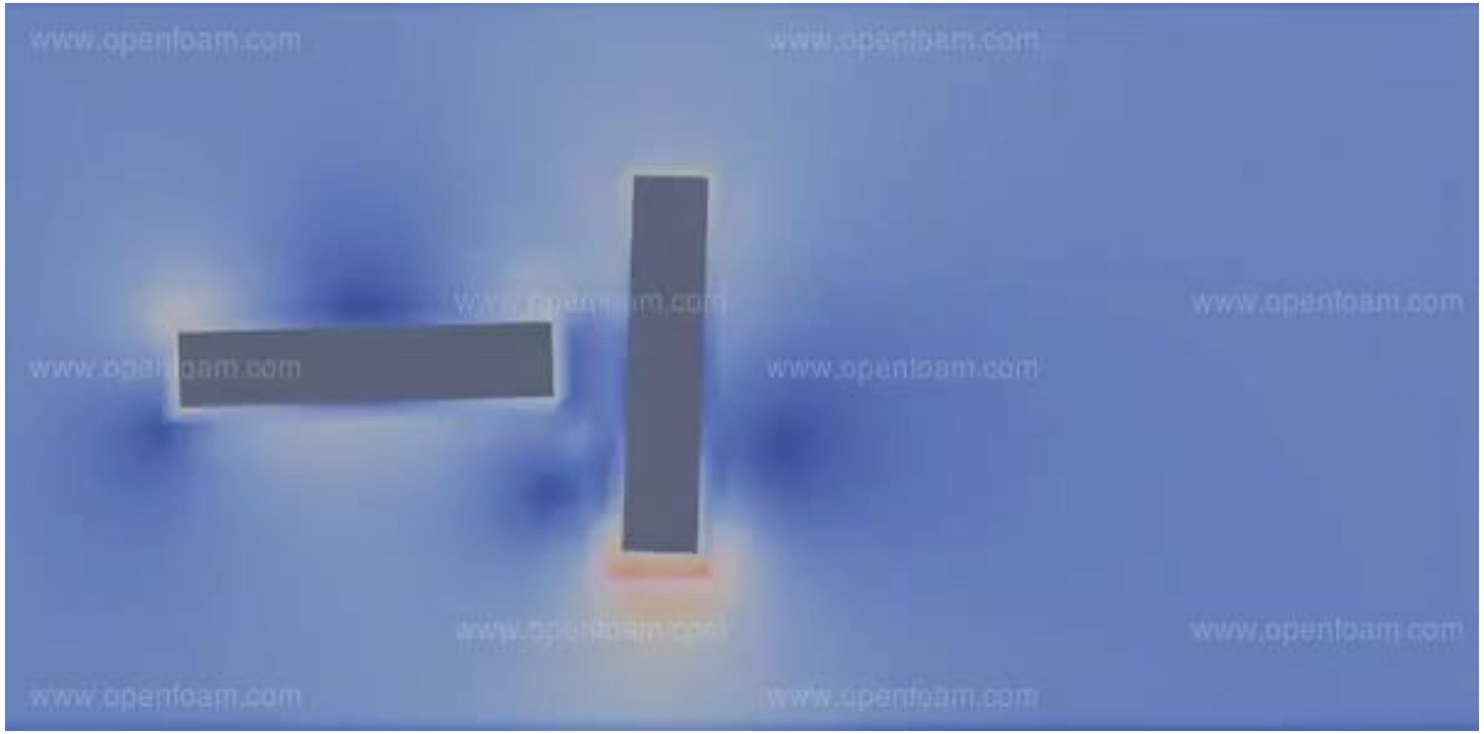
Irregular Waves

A new irregular wave model based on the frequency-direction spectrum has been added to the suite of available wave models

ParaView Catalyst

Catalyst adds in-situ and live visualisation capabilities to arbitrary OpenFOAM simulations. Rather than post-processing at the end of a simulation, it is now possible to harness the capabilities of ParaView and generate visualisation results simultaneous to the simulation.

The scripts for the visualisation pipelines can be created interactively using the ParaView GUI



Thank you.