



# Introduction to 2018 OpenFOAM® Release

---

1 Nov 2018

Geon-Hong Kim  
Engineer, Ph.D.

Hyundai Heavy Industries Co., Ltd.  
NINANO COMPANY Inc.

# 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 연구원  
들에게 심심한 사과의 말씀  
전합니다.



도크없이 땅에서  
배를 만들 수 없다!

## 해봤어?

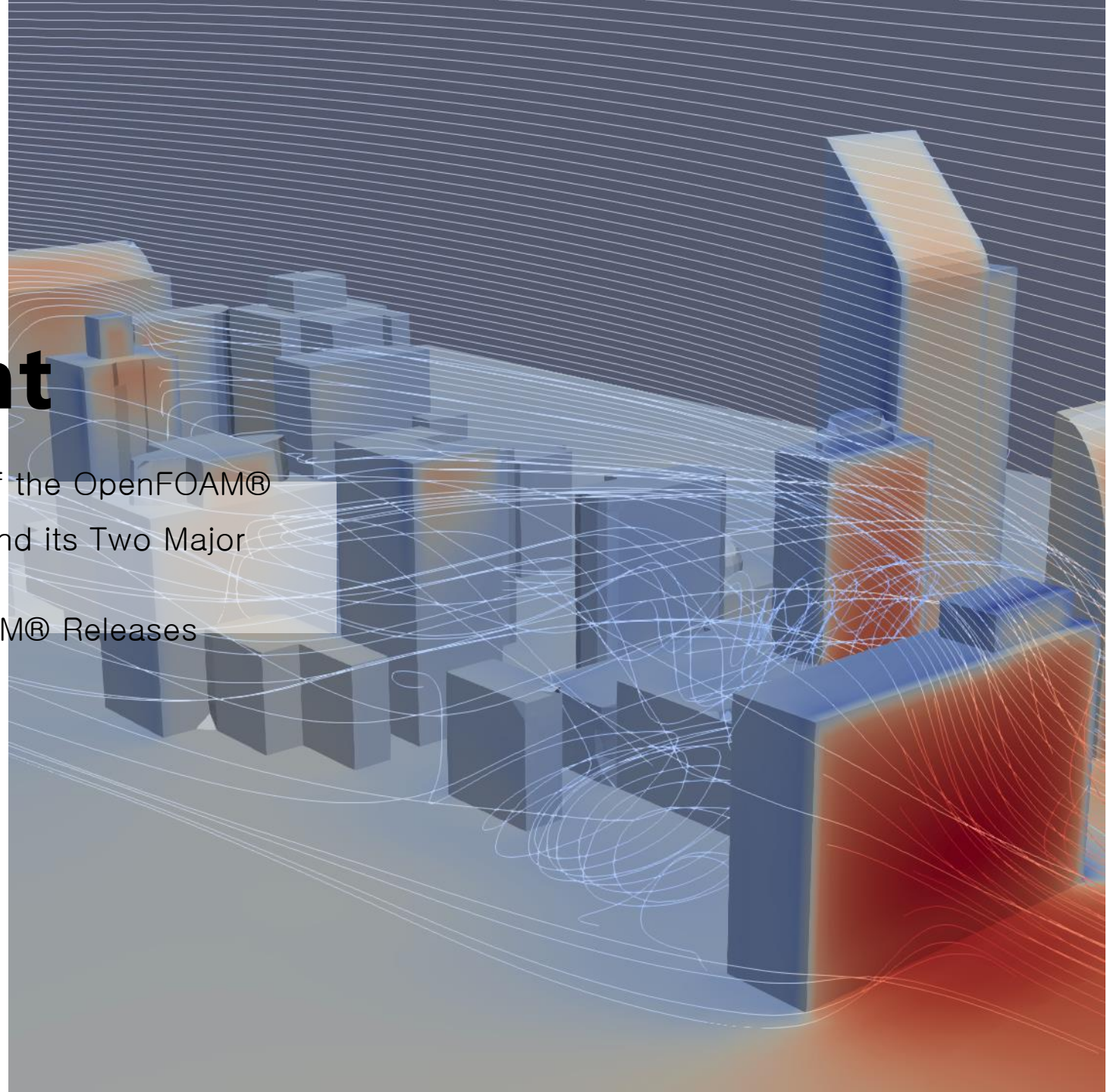
도크없이 배를 만들 수 있을까?  
배는 도크 내에서 지어야 한다는 통념을 깨고, 세계 최초로 육상건조공법을 실험해낸 현대중공업 -  
불가능을 가능으로 바꾸는 창의적 도전정신과 첨단기술로  
조선강국 대한민국을 이끌어가고 있습니다.

미래를 개척하는  
**현대중공업**



# — Content

- 01 Short History of the OpenFOAM®
- 02 OpenFOAM® and its Two Major Folks
- 03 2018 OpenFOAM® Releases



# **Short History of the OpenFOAM®**

# — OpenFOAM® Chronicles



2018 7th OKUCC

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)





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

```
eld divU(fvc::div(fvc::absolute
nsorField> tgradU = fvc::grad(U)
rField G(this->GName(), nut*(tgr
clear());

date epsilon and G at the wall
on_.boundaryField().updateCoeffs(

Dissipation equation
p<fvScalarMatrix> epsEqn

fvm::ddt(alpha, rho, epsilon_)
+ fvm::div(alphaRhoPhi, epsilon_)
- fvm::laplacian(alpha*rho*Depsil
==
C1_*alpha*rho*G*epsil
```

A

B



D



C

E

# OpenFOAM-1.0 Contributors

2018 7<sup>th</sup> OKUCC

—  
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.



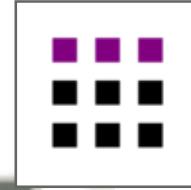
# — The beginning of the story...

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](http://www.openfoam.org)

Supported by OpenFOAM  
Foundation

Latest release: v6



## **OpenFOAM+**

---

[www.openfoam.com](http://www.openfoam.com)

Supported by ESI group

Latest release: v1806



## **foam-extend**

---

[foam-extend.fsb.de](http://foam-extend.fsb.de)

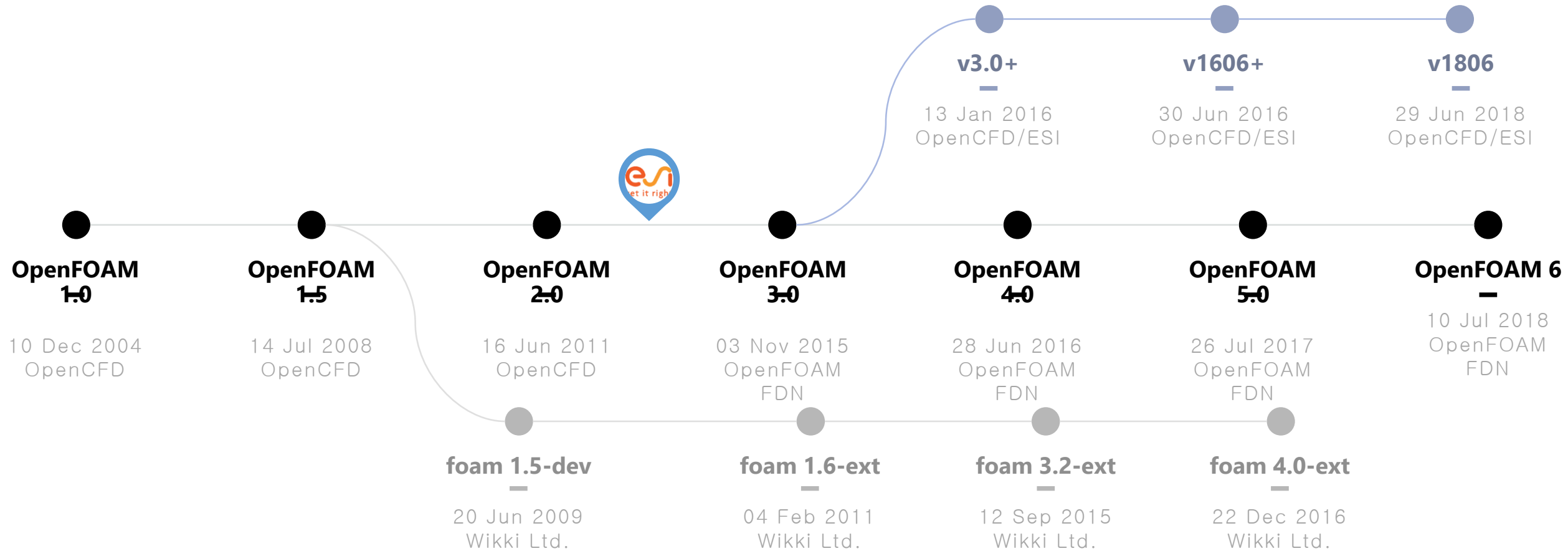
Supported by Wikki

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**

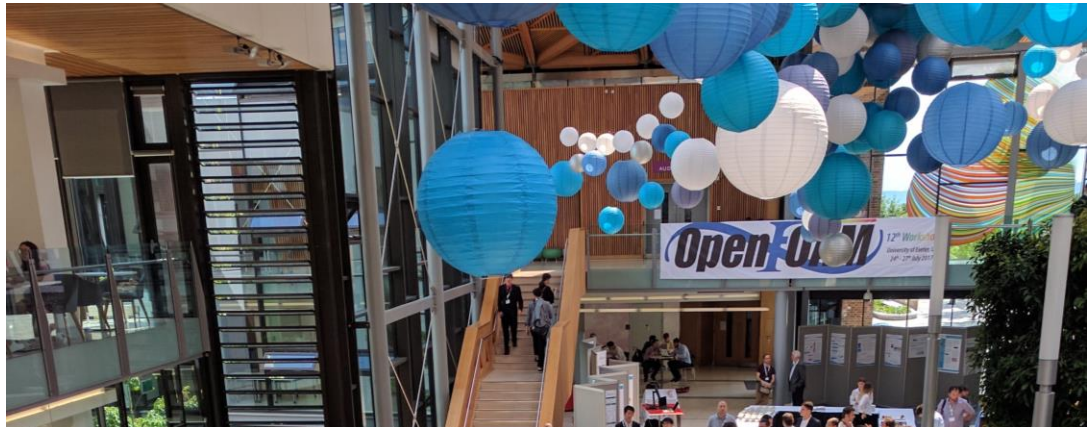
—  
Co-author of OpenFOAM (known as)

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.



**International OpenFOAM Workshop**  
Annual Event  
Supported by Wikki Ltd.  
Since 2006 in Zagreb, Croatia  
[openfoamworkshop.org](http://openfoamworkshop.org)

**OpenFOAM  
Community  
Conferences**



**OpenFOAM Conference**  
Annual Event  
Supported by ESI Group  
Since 2013 in Frankfurt, Germany  
[www.esi-group.com](http://www.esi-group.com)





# **20<sup>18</sup> OpenFOAM Releases**

# OPENFOAM 6

FLUID SIMULATION  
SOLID FOUNDATION  
<https://openfoam.org/6>



Read More



# OpenFOAM® v1806

29 Jun 2018

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

# OPENFOAM 6

10 Jul 2018

Main stream of the OpenFOAM development

Core Team : Henry Weller, Chris Greenshields, Will Bainbridge

OpenFOAM

The open source CFD toolbox

[Home](#) [Products](#) [Services](#) [Download](#) [Code](#) [Documentation](#) [Community](#) [News](#)

[About us](#) [Contact](#) [Jobs](#) [Legal](#)

OpenFOAM v1806

[Release Summary](#)  
[Pre-processing](#)  
[Numerics](#)  
[Solver and physics](#)  
[Boundary conditions](#)  
[Post-processing](#)  
[Documentation](#)  
[Build system](#)  
[Community](#)  
[Notable bug-fixes](#)

## OpenCFD Release OpenFOAM® v1806

29/06/2018

OpenCFD is pleased to announce the June 2018 release of OpenFOAM® v1806. This release extends [OpenFOAM-v1712](#) features across many areas of the code. The new functionality represents development sponsored by OpenCFDs customers, internally funded developments, and integration of features and changes from the [OpenFOAM community](#). OpenFOAM is distributed by OpenCFD under the [GPL license](#) as:

- [Source code](#) to be compiled on any Linux system
- Pre-compiled binary installation [for Linux systems](#)
- Pre-compiled binary installation [for Mac OS X systems](#)
- Windows installer
- [MS Windows installer](#)
- Bash on Ubuntu on Windows [for MS Windows 10](#)

Please refer to the [download instructions](#) to obtain the code.

The development [repositories](#) are publicly available. These repositories are regularly updated with bug fixes and new functionality.

[FOLLOW US ON](#)

# OPENFOAM 6

—

Key developments and features

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




# 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

```

@@ -22,14 +22,14 @@ boundaryField
{
    22 22 {
    23 23     inlet
    24 24     {
    25 -         type            freestream;
    26 -         freestreamValue  uniform (25.75 3.62 0);
    25 +         type            freestreamVelocity;
    26 +         freestreamValue  $internalField;
    27 27     }
    28 28
    29 29     outlet
    30 30     {
    31 -         type            freestream;
    32 -         freestreamValue  uniform (25.75 3.62 0);
    31 +         type            freestreamVelocity;
    32 +         freestreamValue  $internalField;
    33 33     }
    34 34
    35 35     walls

```

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

```

@@ -23,11 +23,13 @@ boundaryField
{
    23 23     inlet
    24 24     {
    25 25         type            freestreamPressure;
    26 +         freestreamValue  $internalField;
    26 27     }
    27 28
    28 29     outlet
    29 30     {
    30 31         type            freestreamPressure;
    32 +         freestreamValue  $internalField;
    31 33     }
    32 34
    33 35     walls

```

```
64
65     while (runTime.run())
66     {
67         - #include "readTimeControls.H"
68         + #include "readControls.H"
69         #include "CourantNo.H"
70         #include "setDeltaT.H"
71
72         runTime++;
73
74         Info<< "Time = " << runTime.timeName() << nl << endl;
75
76         + mesh.update();
77         +
78         + #include "updateUf.H"
79         +
80         + if (mesh.changing())
81         + {
82         +     MRF.update();
83         +
84         +     if (correctPhi)
85         +     {
86         +         // Calculate absolute flux from the mapped surface velocity
87         +         phi = mesh.Sf() & Uf();
88         +
89         +         #include "correctPhi.H"
90         +
91         +         // Make the flux relative to the mesh motion
92         +         fvc::makeRelative(phi, U);
93         +     }
94         +
95         +     if (checkMeshCourantNo)
96         +     {
97         +         #include "meshCourantNo.H"
98         +     }
99         +
100        // --- Pressure-velocity PIMPLE corrector loop
101        while (pimple.loop())
102        {
```

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.

# Marine/Waves

## New stokes5 and solitary wave models

Stokes 5<sup>th</sup> 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).

```

1  + /*-----*\
2  + ===== |
3  +  \ \    /  F ield      | OpenFOAM: The Open Source CFD Toolbox
4  +  \ \    /  O peration  |
5  +  \ \    /  A nd        | Copyright (C) 2017 OpenFOAM Foundation
6  +   \ \ /   M anipulation |
7  + -----*\
8  + License
9  +   This file is part of OpenFOAM.
10 +
11 +   OpenFOAM is free software: you can redistribute it and/or modify it
12 +   under the terms of the GNU General Public License as published by
13 +   the Free Software Foundation, either version 3 of the License, or
14 +   (at your option) any later version.
15 +
16 +   OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
17 +   ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
18 +   FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License
19 +   for more details.
20 +
21 +   You should have received a copy of the GNU General Public License
22 +   along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>.
23 +
24 + Class
25 +   Foam::waveModels::Stokes5
26 +
27 + Description
28 +   Fifth-order wave model.
29 +
30 + Reference:
31 +   \verbatim
32 +       "A Fifth Order Stokes Theory for Steady Waves"
33 +       J D Fenton
34 +       Journal of Waterway, Port, Coastal, and Ocean Engineering (1985),
35 +       Volume 111, Issue 2, Pages 216-234
36 +   \endverbatim
37 +
38 + SourceFiles
39 +   Stokes5.C
40 +
41 + \*-----*/
42 +
43 + #ifndef Stokes5_H
44 + #define Stokes5_H
45 +
46 + #include "Stokes2.H"
47 +

```



```

..      @@ -0,0 +1,218 @@
1  + #!/bin/sh
2  + #-----
3  + # ===== |
4  + # \\      / F ield      | OpenFOAM: The Open Source CFD Toolbox
5  + # \\      / O peration  |
6  + # \\      / A nd        | Copyright (C) 2018 OpenFOAM Foundation
7  + #   \\    M anipulation |
8  + #-----
9  + # License
10 + #   This file is part of OpenFOAM.
11 + #
12 + #   OpenFOAM is free software: you can redistribute it and/or modify it
13 + #   under the terms of the GNU General Public License as published by
14 + #   the Free Software Foundation, either version 3 of the License, or
15 + #   (at your option) any later version.
16 + #
17 + #   OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
18 + #   ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
19 + #   FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License
20 + #   for more details.
21 + #
22 + #   You should have received a copy of the GNU General Public License
23 + #   along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>.
24 + #
25 + # Script
26 + #   foamGet
27 + #
28 + # Description
29 + #   Finds an example OpenFOAM case dictionary in $FOAM_ETC/caseDicts and
30 + #   copies it into the respective case directory.
31 + #
32 + #-----
33 + usage() {
34 +     cat<<USAGE
35 +
36 + Usage: ${0##*/} [OPTIONS] <file>
37 + options:
38 +   -case | -c <dir> specify case directory (default = local dir)
39 +   -ext  | -e <ext> specify file extension, e.g -e cfg for files with ".cfg"
40 +   -help | -h      print the usage
41 +   -no-ext | -n     specify file without extension
42 +   -target | -t <dir> specify target directory (default = system)
43 +

```

# 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

<p>Pre- processing</p> <p>new and improved</p>	<p>Numerics</p> <p>stabilisation</p>	<p>Solvers</p> <p>laser melting</p>	<p>Physical Models</p> <p>phase &amp; mass models</p>
<p>Boundary Conditions</p> <p>fan, irregular waves</p>	<p>Post- processing</p> <p>Catalyst, sampling</p>	<p>Documentatio n</p> <p>solvers, fvOption</p>	<p>Bug Fixes</p> <p>snappyHexMesh</p>

# Consolidation of Moving Mesh Solvers

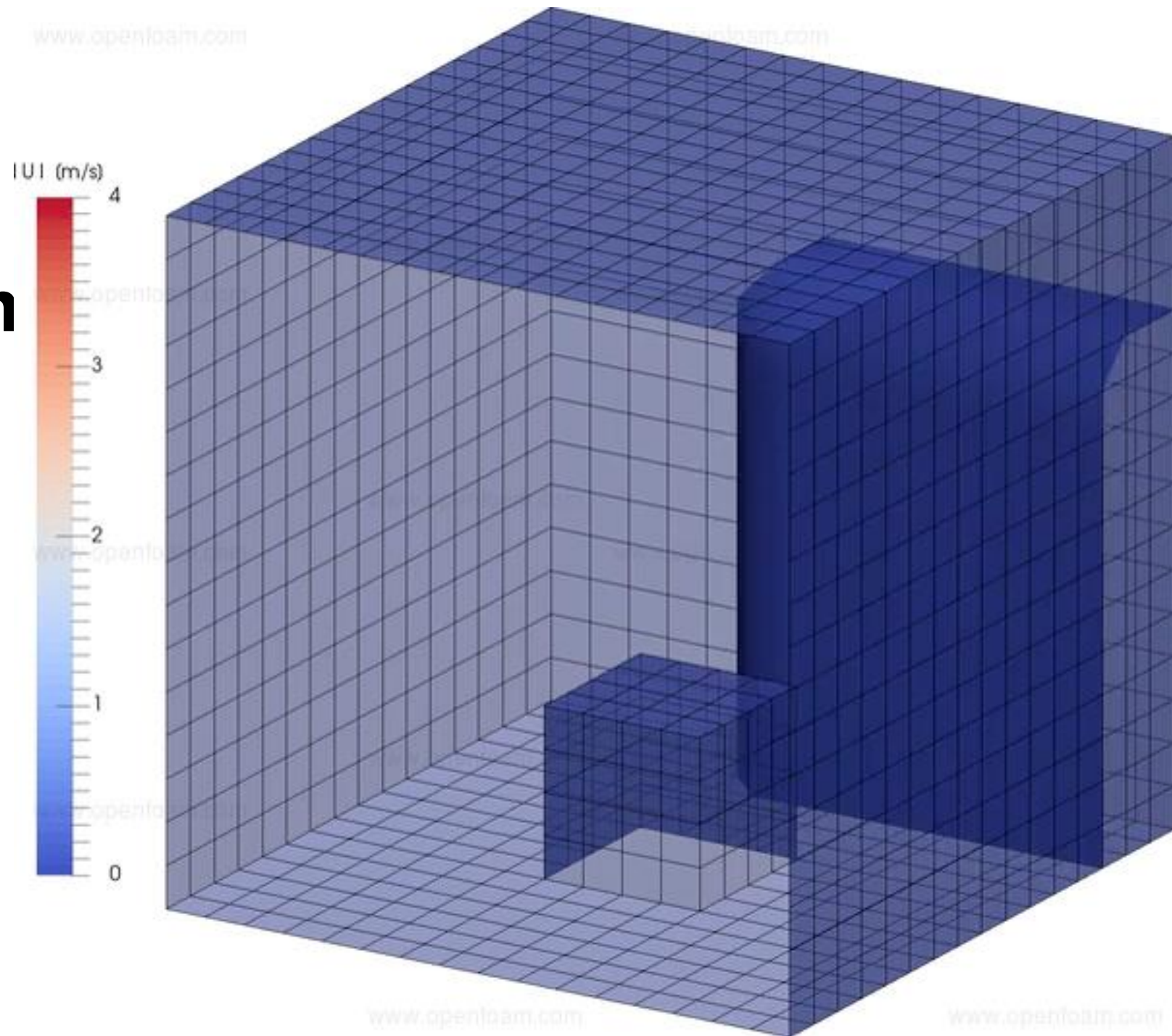
—  
Moving mesh functionality has been incorporated into many of the static mesh solver applications from earlier releases

Old solver	New solver
<i>pimpleDyMFoam</i>	<i>pimpleFoam</i>
<i>rhoPimpleDyMFoam</i>	<i>rhoPimpleFoam</i>
<i>interDyMFoam</i>	<i>interFoam</i>
<i>multiphaseInterDyMFoam</i>	<i>multiphaseInterFoam</i>



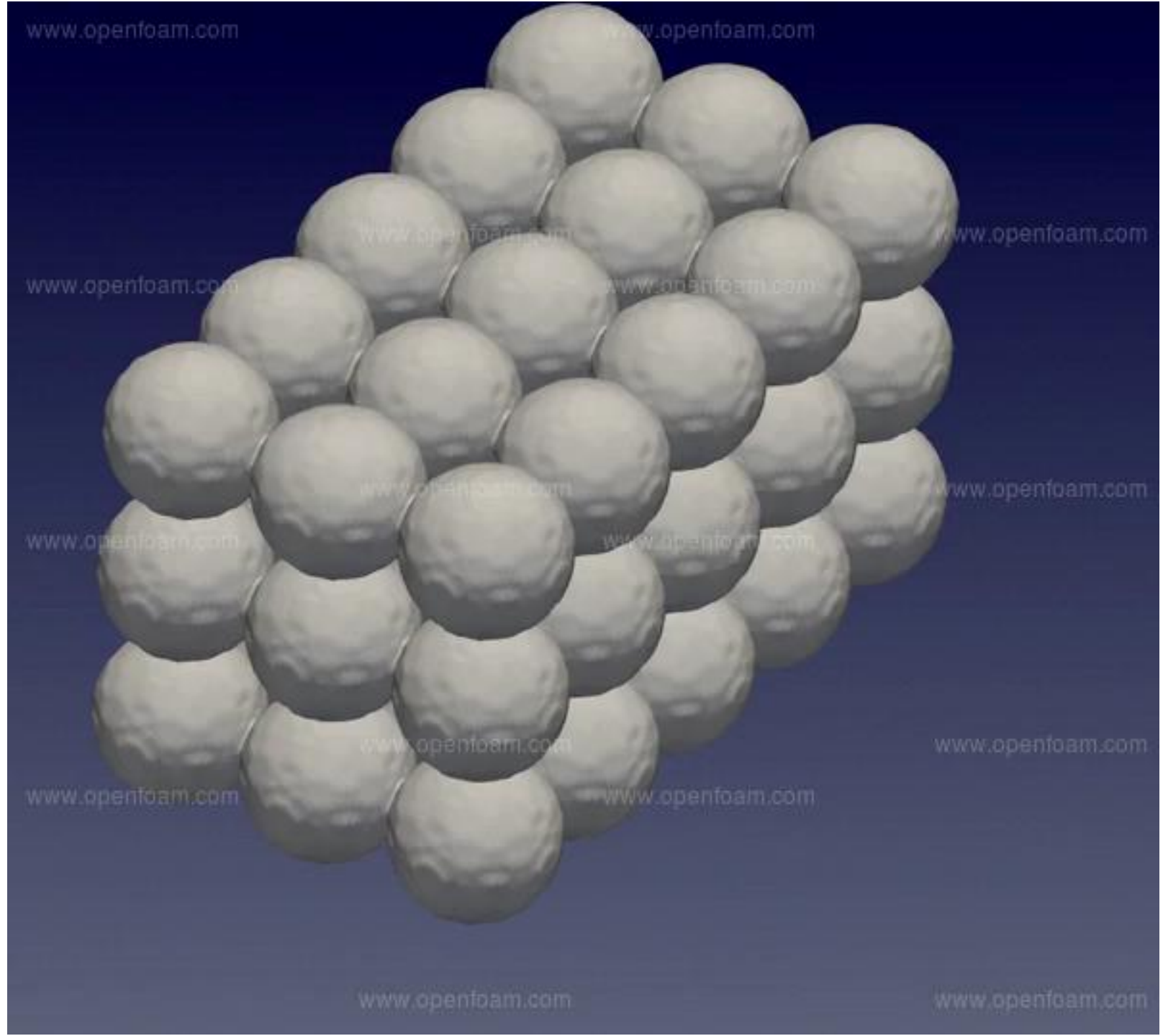
# Extended interIsoFoam solver

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



# icoReacting- Multiphase- InterFoam

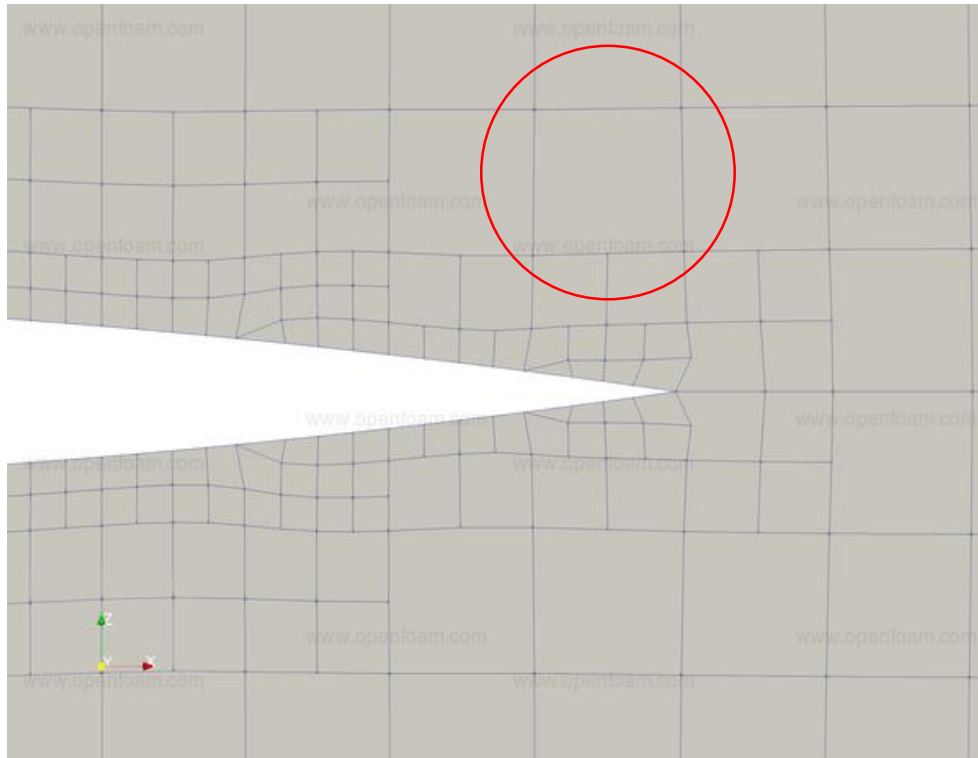
—  
A multi-phase, multi-  
component  
incompressible solver  
based on a Volume Of  
Fluid (VOF) method  
with per-phase choice  
of thermodynamics  
model (sharing  
pressure and  
temperature).



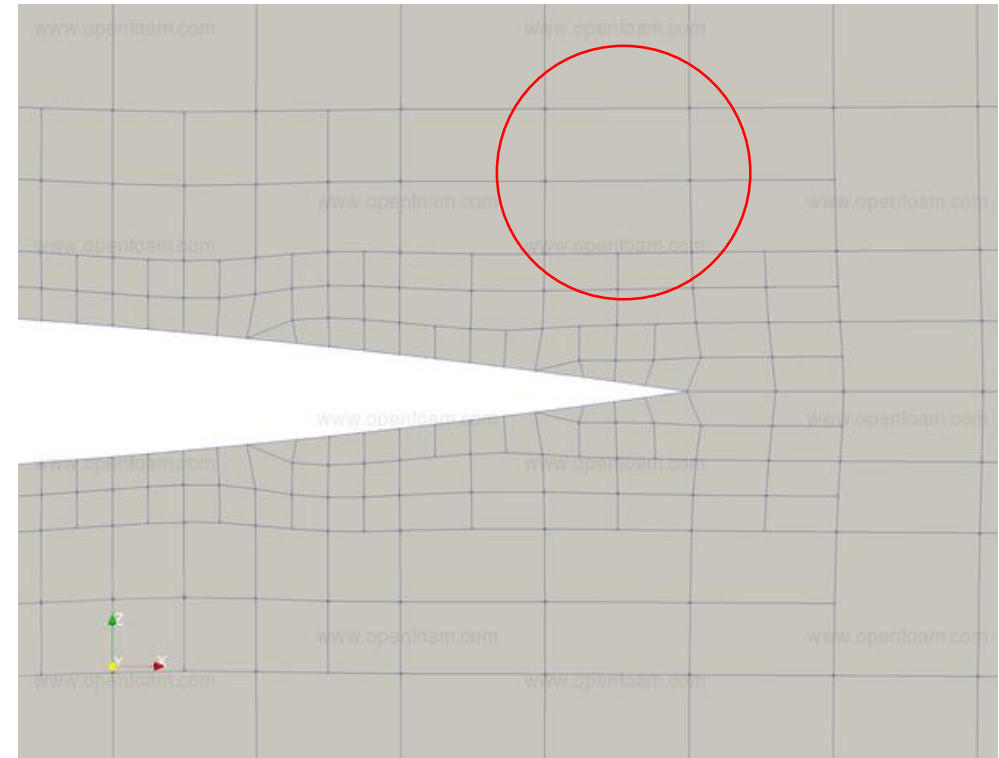
# snappyHexMesh

—

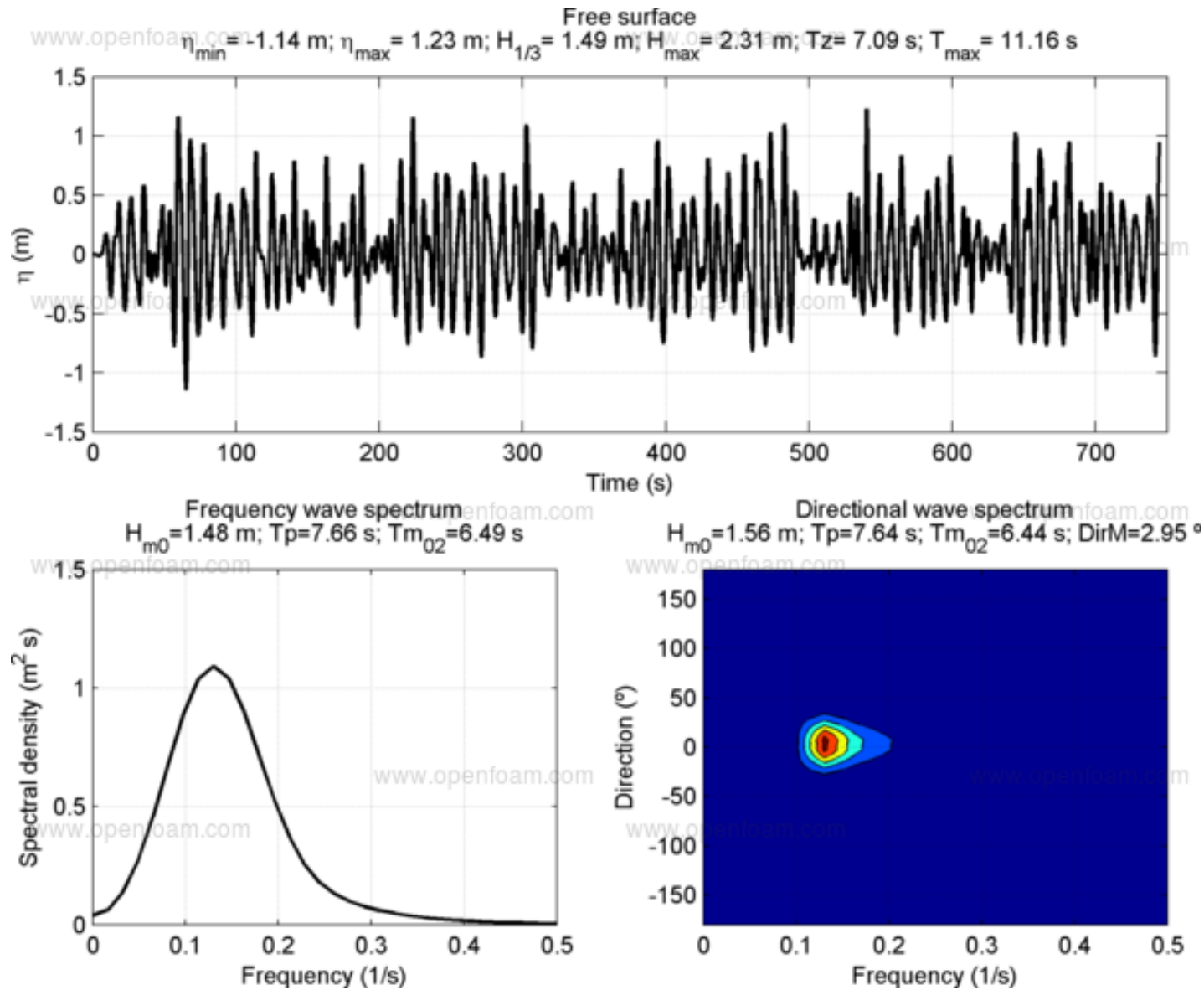
snappyHexMesh supports additional directional refinement inside refinementRegions



w/o directional refinement



w/ directional refinement



# Irregular Waves

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

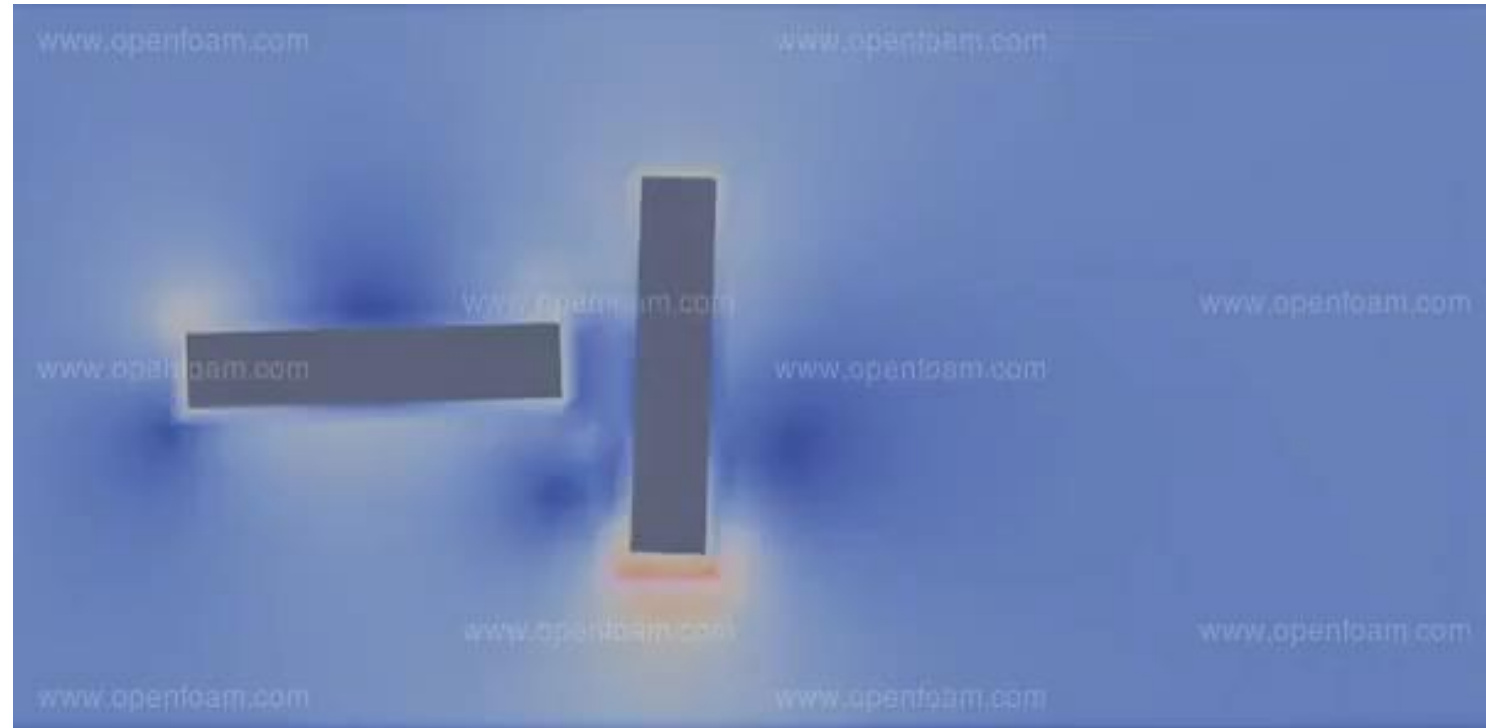


# ParaView Catalyst

—

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.**