



IHCantabria

UNIVERSIDAD DE CANTABRIA

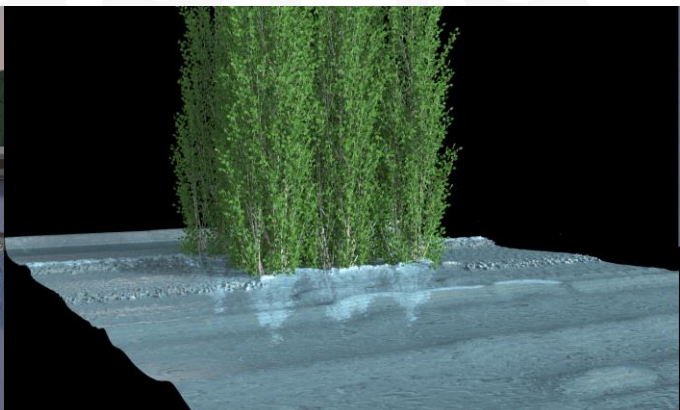
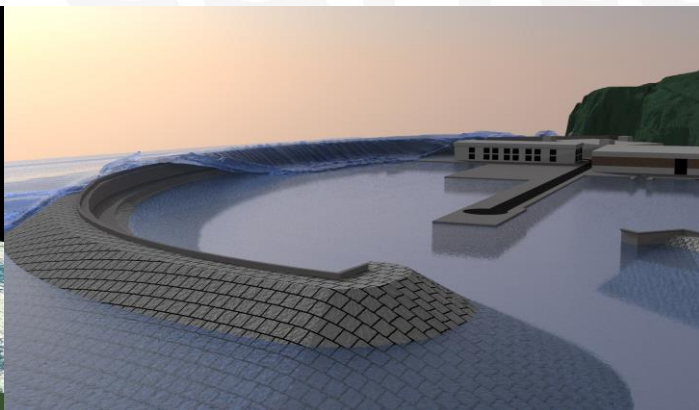
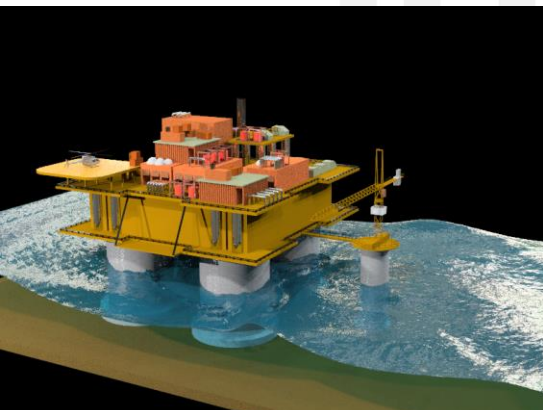
R+D+i for a Sustainable Development

(IHFOAM GUI)

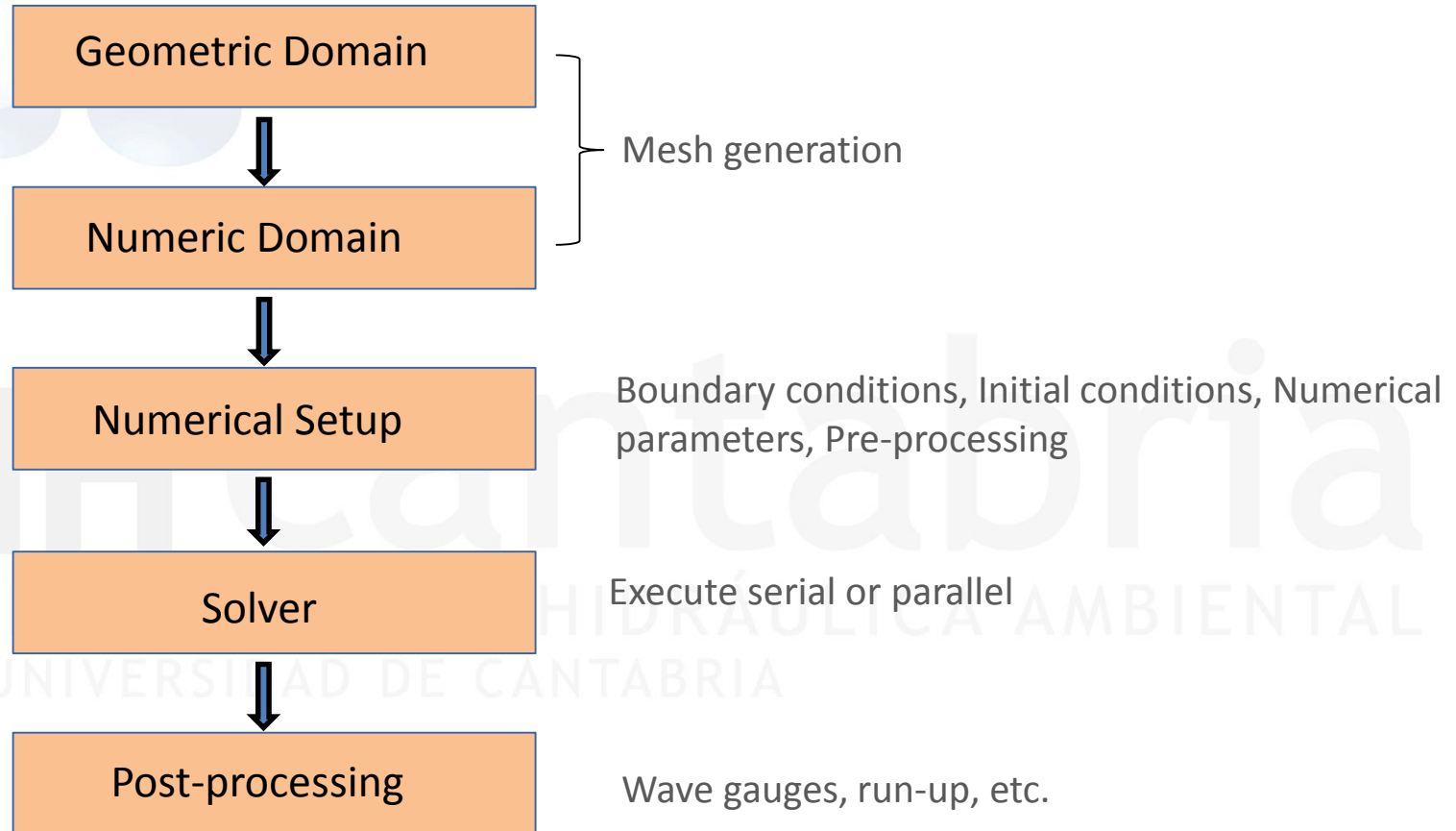
IHFOAM applied to Coastal Engineering

Regular waves interaction with a rubble-mound breakwater

Gabriel Barajas, Javier L. Lara, María Maza, Alejandro Gonzalez



OpenFOAM workflow



OpenFOAM case

0

- alpha.water
- p_rgh
- U
- porosityIndex

- k
- epsilon
- nut

constant

- g
- transportProperties
- turbulenceProperties
- porosityDict

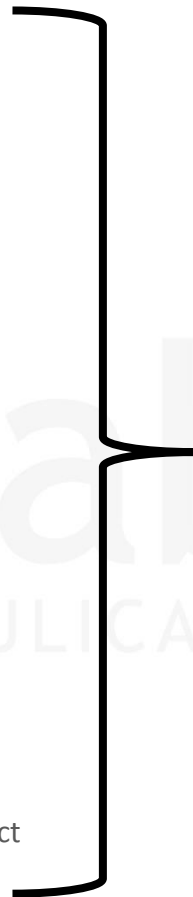
- waveProperties

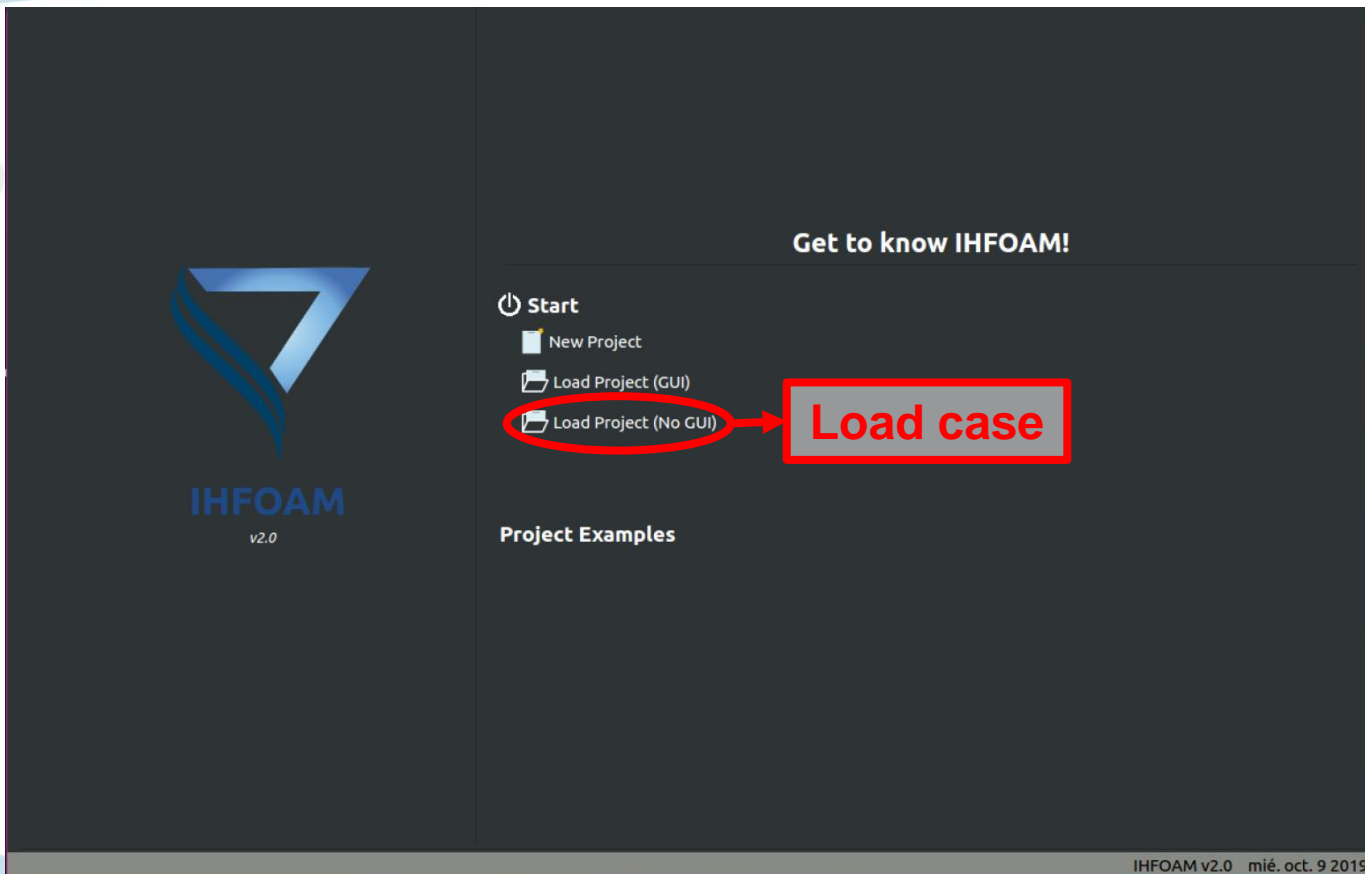
system

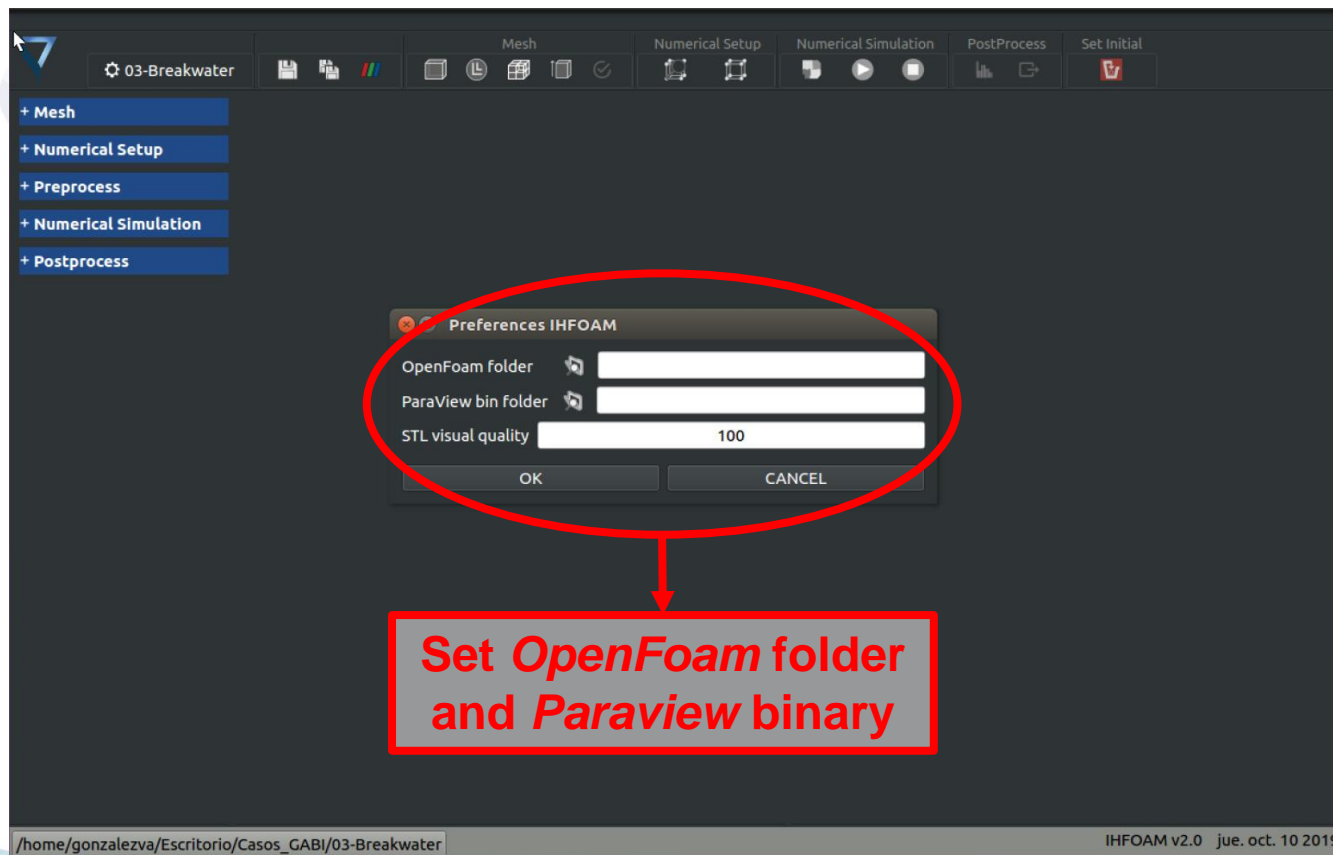
- blockMeshDict
- setFieldsDict
- snappyHexMeshDict
- extrudeMeshDict

- fvSchemes
- fvSolution
- decomposeParDict
- controlDict

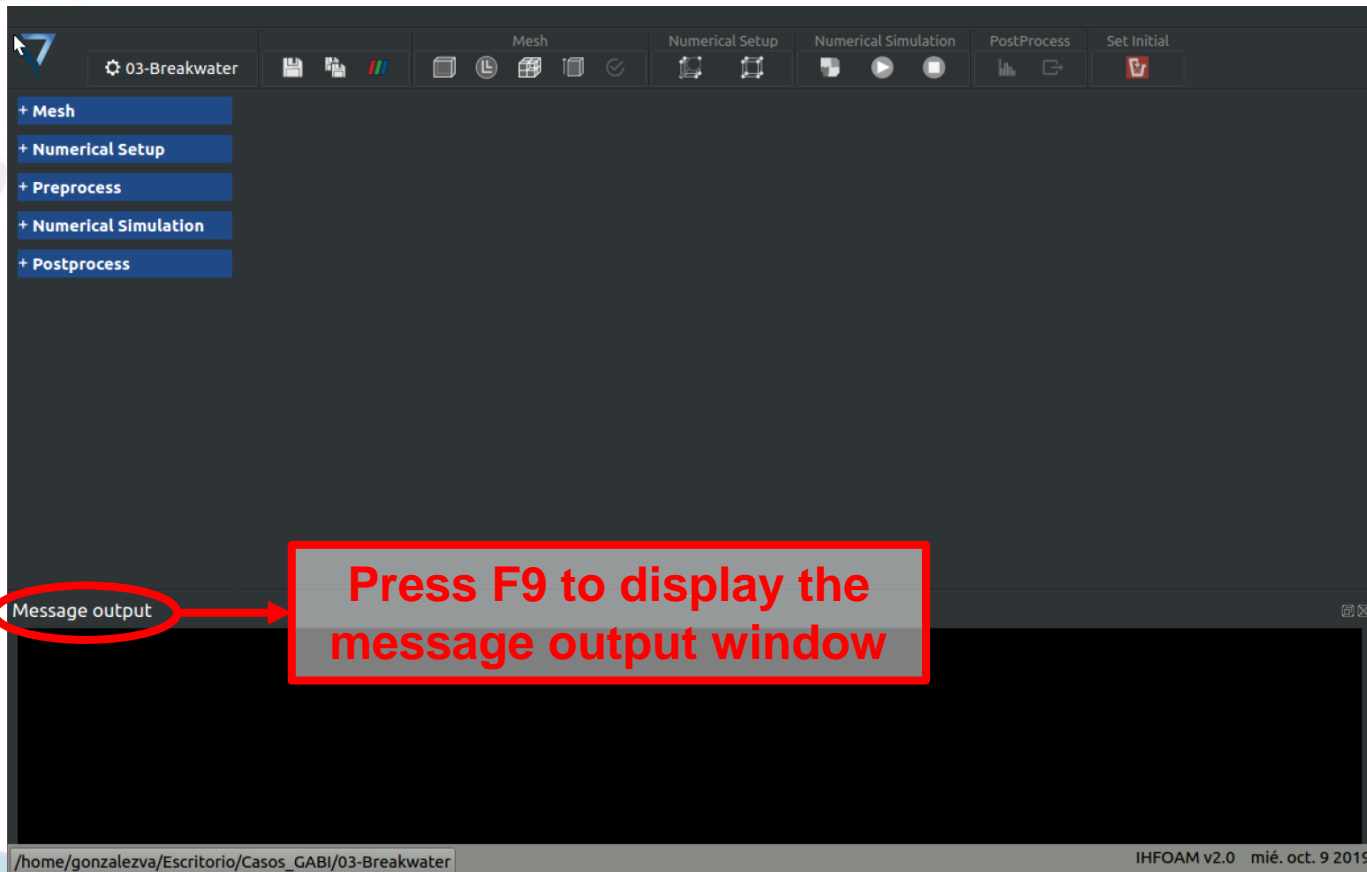
IHFOAM GUI





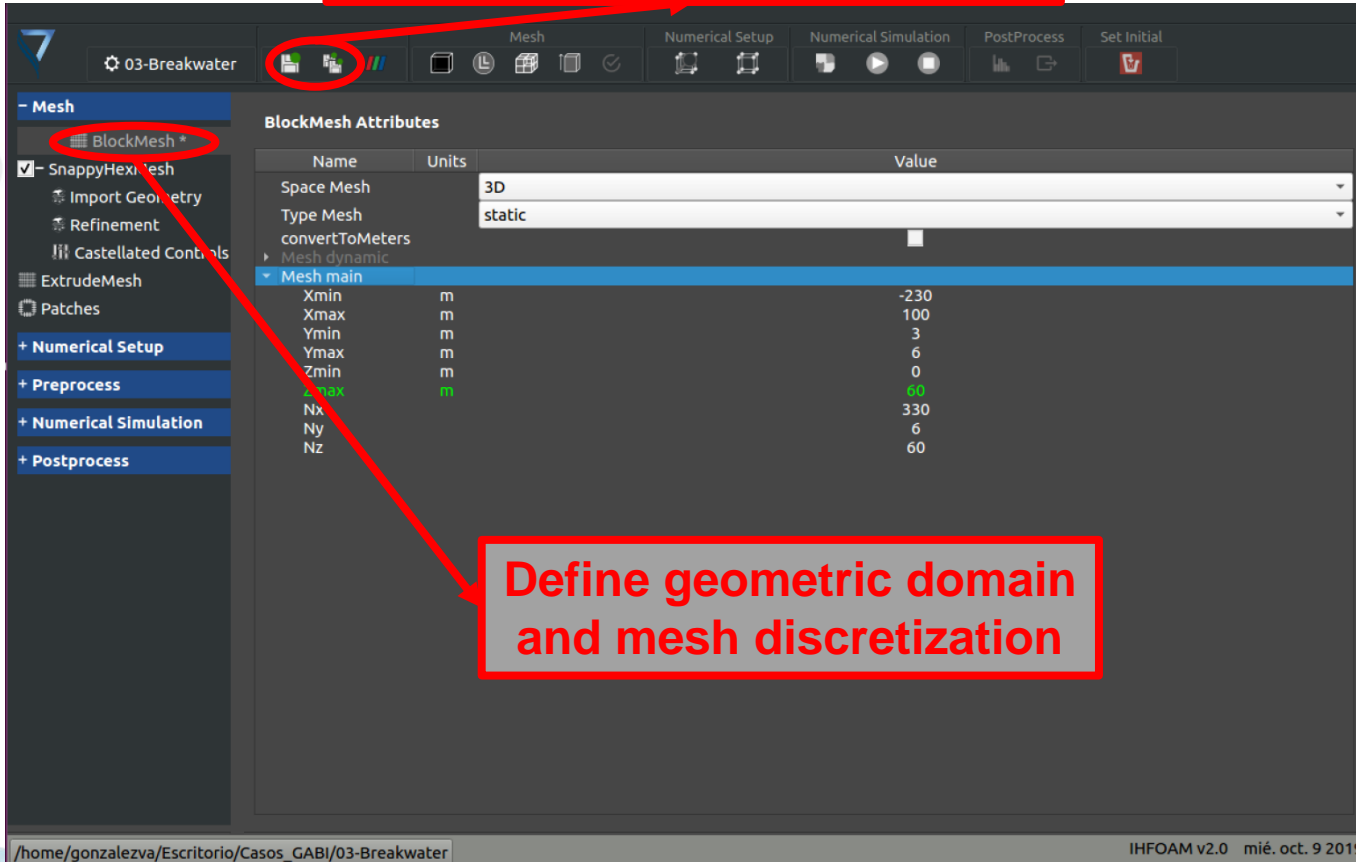


**Set OpenFoam folder
and Paraview binary**



Save single pannel modified
or save all pannels modified.

Regular waves interaction with a rubble-mound breakwater



BlockMesh Attributes

Name	Units	Value
Space Mesh		3D
Type Mesh		static
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	-230
Xmax	m	100
Ymin	m	3
Ymax	m	6
Zmin	m	0
Zmax	m	60
Nx		330
Ny		6
Nz		60

Define geometric domain and mesh discretization

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

blockMesh button

The screenshot shows the IHFOAM v2.0 software interface. The top menu bar includes 'Mesh', 'Numerical Setup', 'Numerical Simulation', 'PostProcess', and 'Set Initial'. The 'Mesh' menu is open, showing options like 'BlockMesh', 'SnappyHexMesh', 'Import Geometry', 'Refinement', 'Castellated Controls', 'ExtrudeMesh', 'Patches', 'Numerical Setup', 'Preprocess', 'Numerical Simulation', and 'Postprocess'. The 'BlockMesh' button is highlighted with a red circle and an arrow pointing to it from the text 'blockMesh button'.

The 'BlockMesh Attributes' panel is visible, showing a table of attributes:

Name	Units	Value
Space Mesh		3D
Type Mesh		static
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	-230
Xmax	m	100
Ymin	m	3
Ymax		
Zmin		
Zmax		
Nx		
Ny		
Nz		

An IHFOAM dialog box is open in the center, displaying an error message: 'OK!! rm -rf constant/polyMesh/ && blockMesh End'. The dialog box has an 'OK' button. A red circle and an arrow point from the text 'IHFOAM dialog box' to the dialog box.

The 'Output' panel on the right shows a 3D visualization of the mesh. The 'Domain Visor' tab is active, showing a 3D view of the mesh with dimensions. The 'Result' tab is also visible.

The 'Message output' panel at the bottom shows the following text:

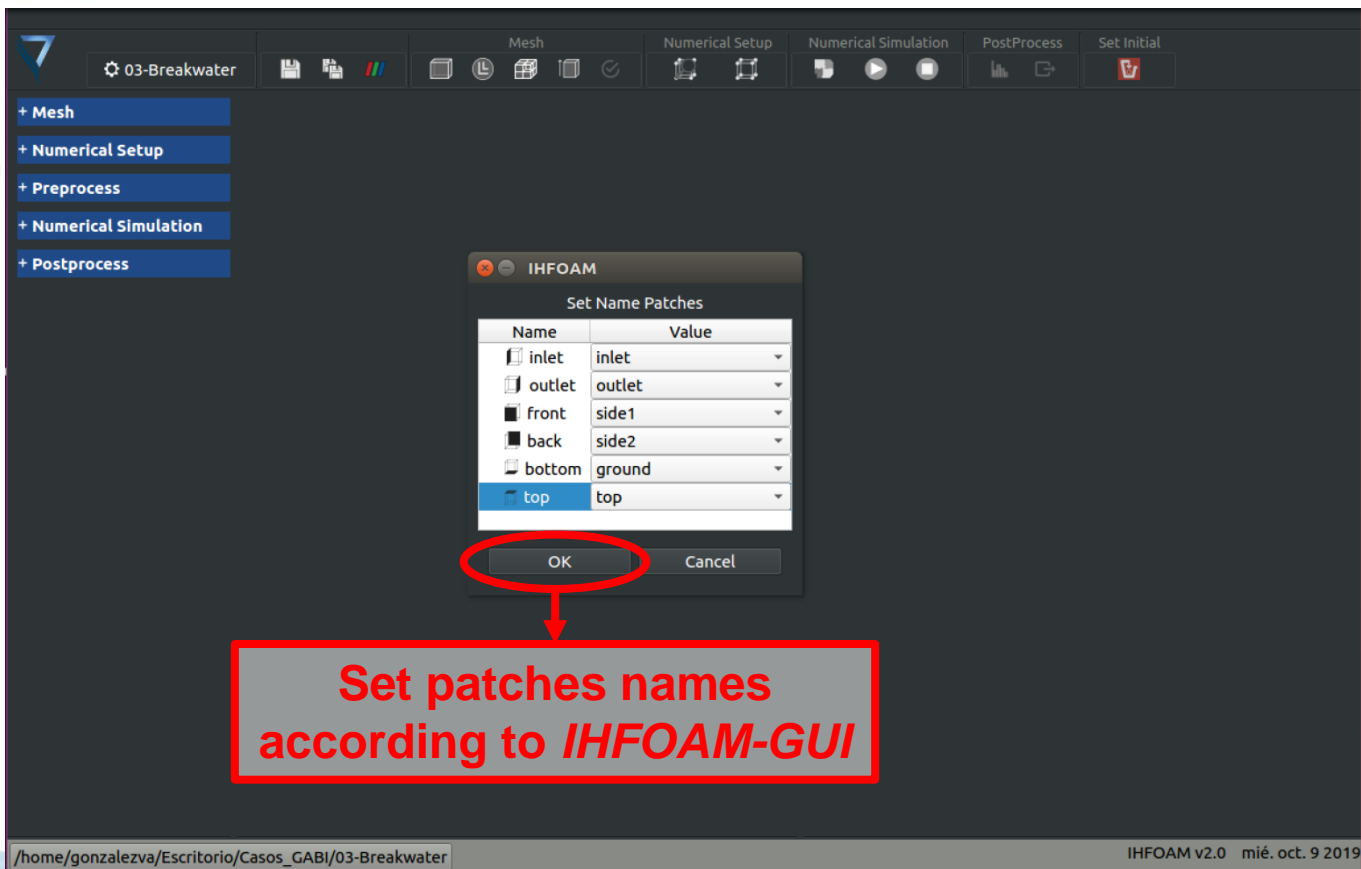
```

patch 0 (start: 334260 size: 360) name: inlet
patch 1 (start: 334620 size: 360) name: outlet
patch 2 (start: 334980 size: 1980) name: ground
patch 3 (start: 336960 size: 1980) name: top
patch 4 (start: 338940 size: 19800) name: side1
patch 5 (start: 358740 size: 19800) name: side2

End

```

The status bar at the bottom shows the file path: '/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater' and the version: 'IHFOAM v2.0 mié. oct. 9 2019'.

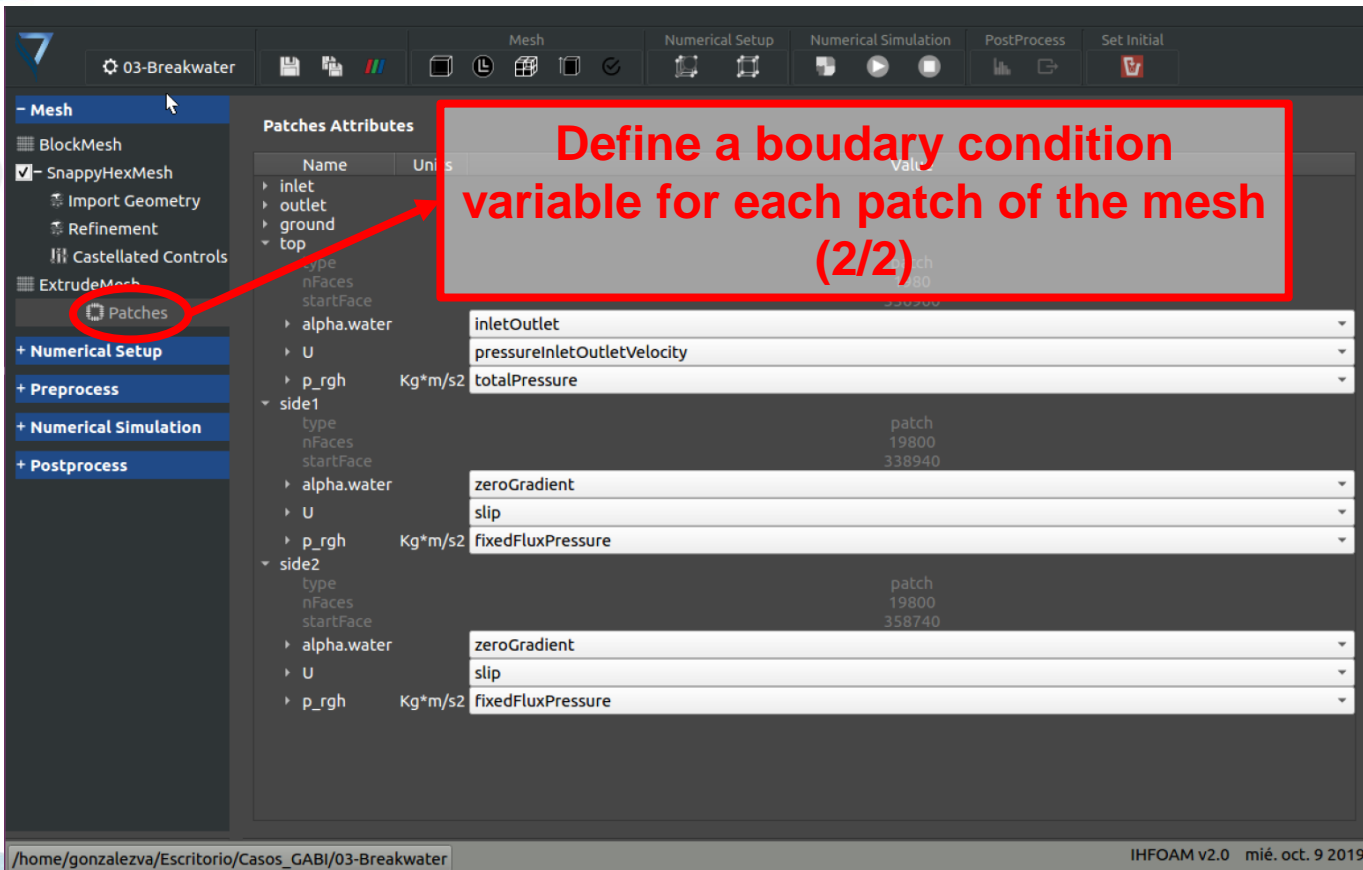


Define a boundary condition variable for each patch of the mesh (1/2)

Name	Units	Value
inlet		
type		
nFaces		360
startFace		334620
alpha.water		waveAlpha
U		waveVelocity
p_rgh	Kg*m/s2	fixedFluxPressure
outlet		
type		patch
nFaces		360
startFace		334620
alpha.water		zeroGradient
U		waveVelocity
p_rgh	Kg*m/s2	fixedFluxPressure
ground		
type		wall
inGroups		1
nFaces		1980
startFace		334980
alpha.water		zeroGradient
U		fixedValue
p_rgh	Kg*m/s2	fixedFluxPressure
top		
side1		
side2		

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

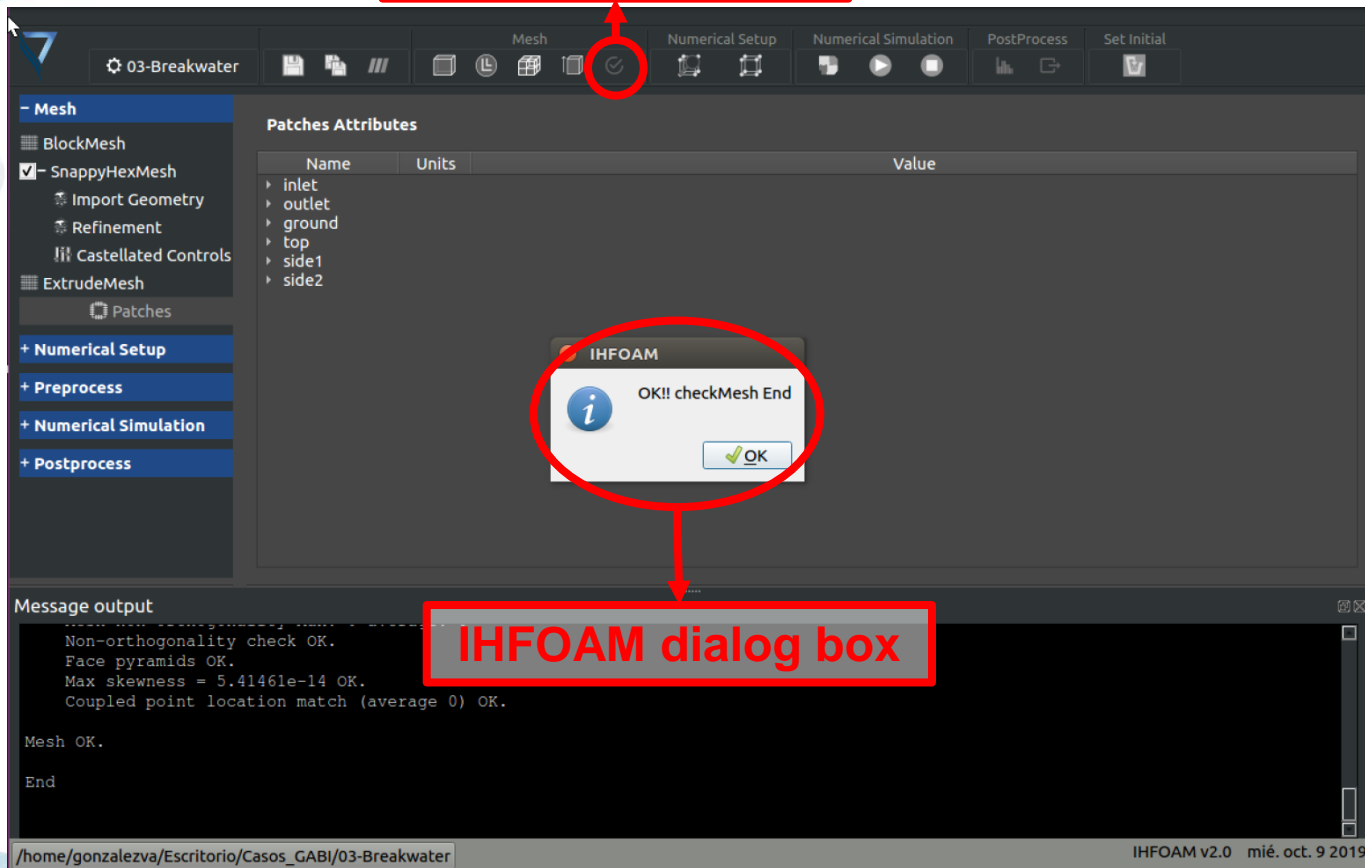
IHFOAM v2.0 mié. oct. 9 2019



Define a boundary condition variable for each patch of the mesh (2/2)

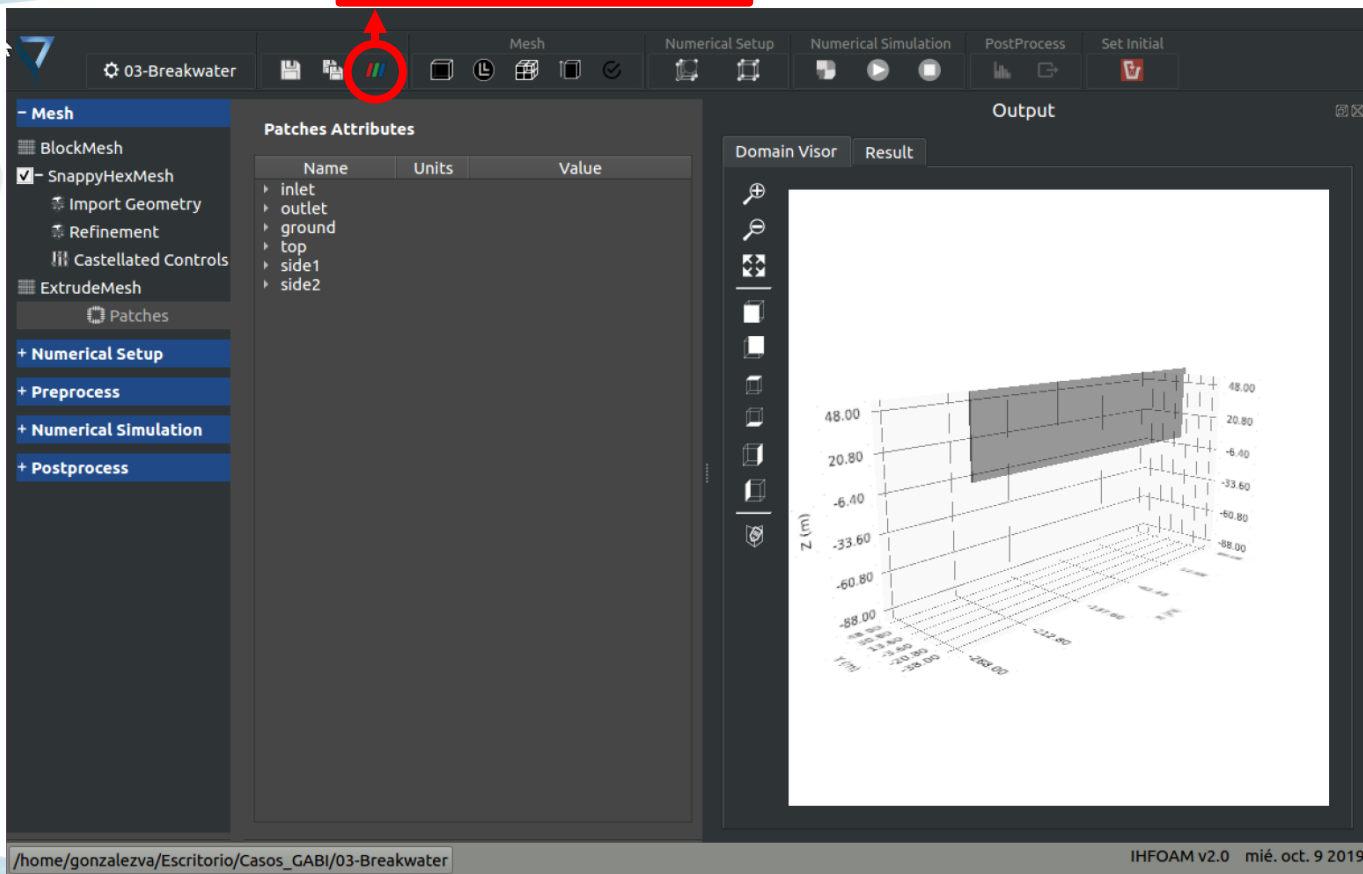
Name	Units	Value
inlet		inletOutlet
outlet		pressureInletOutletVelocity
ground		totalPressure
top		zeroGradient
alpha.water		slip
U		fixedFluxPressure
p_rgh	Kg*m/s2	fixedFluxPressure
side1		zeroGradient
side2		slip
alpha.water		fixedFluxPressure
U		zeroGradient
p_rgh	Kg*m/s2	fixedFluxPressure

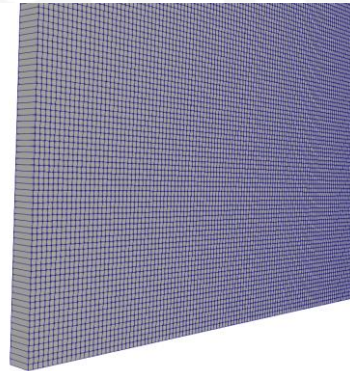
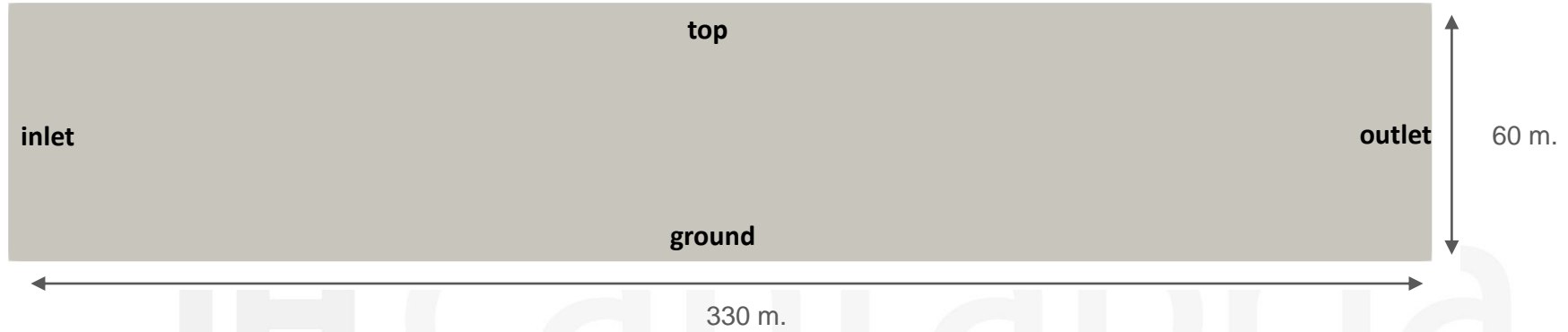
checkMesh button

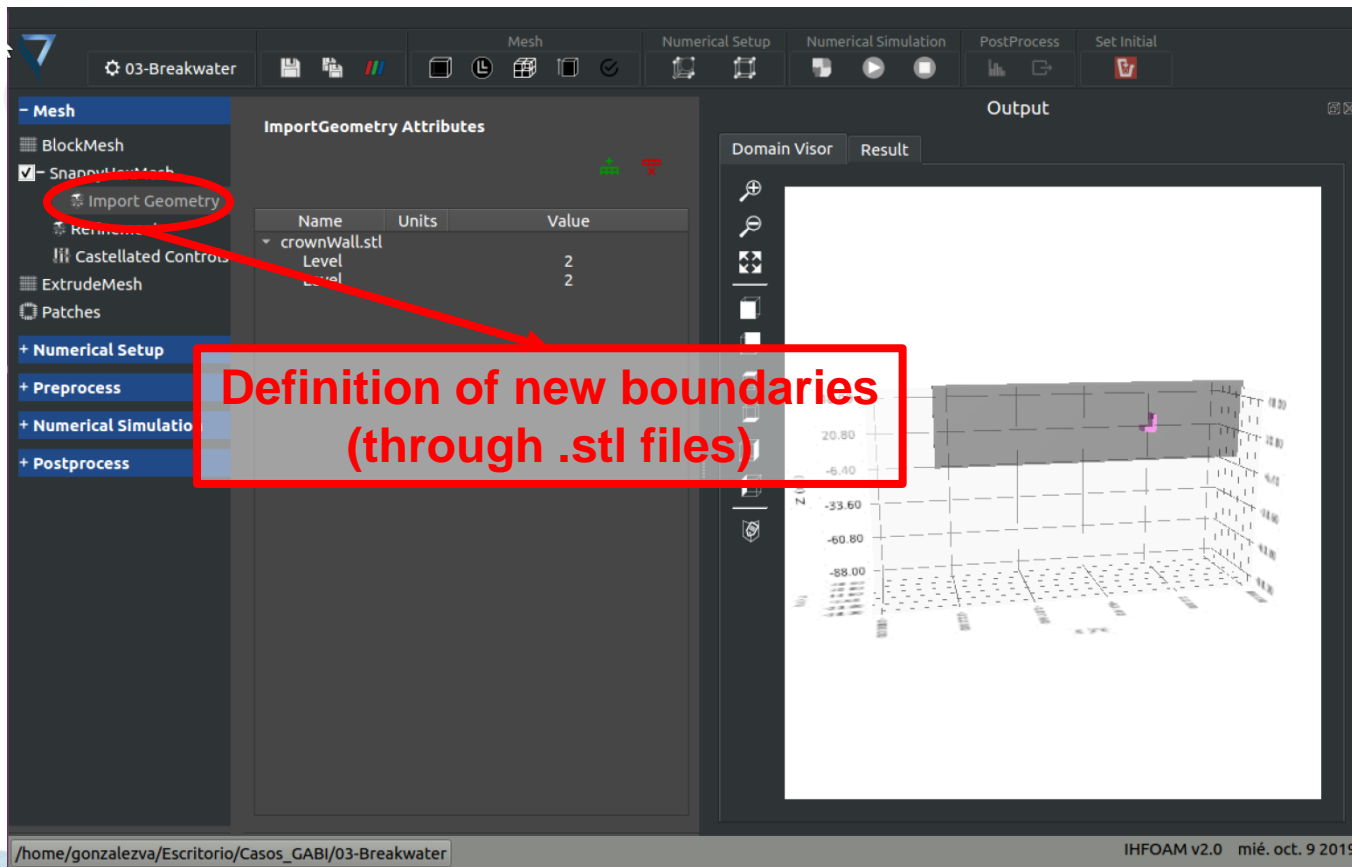


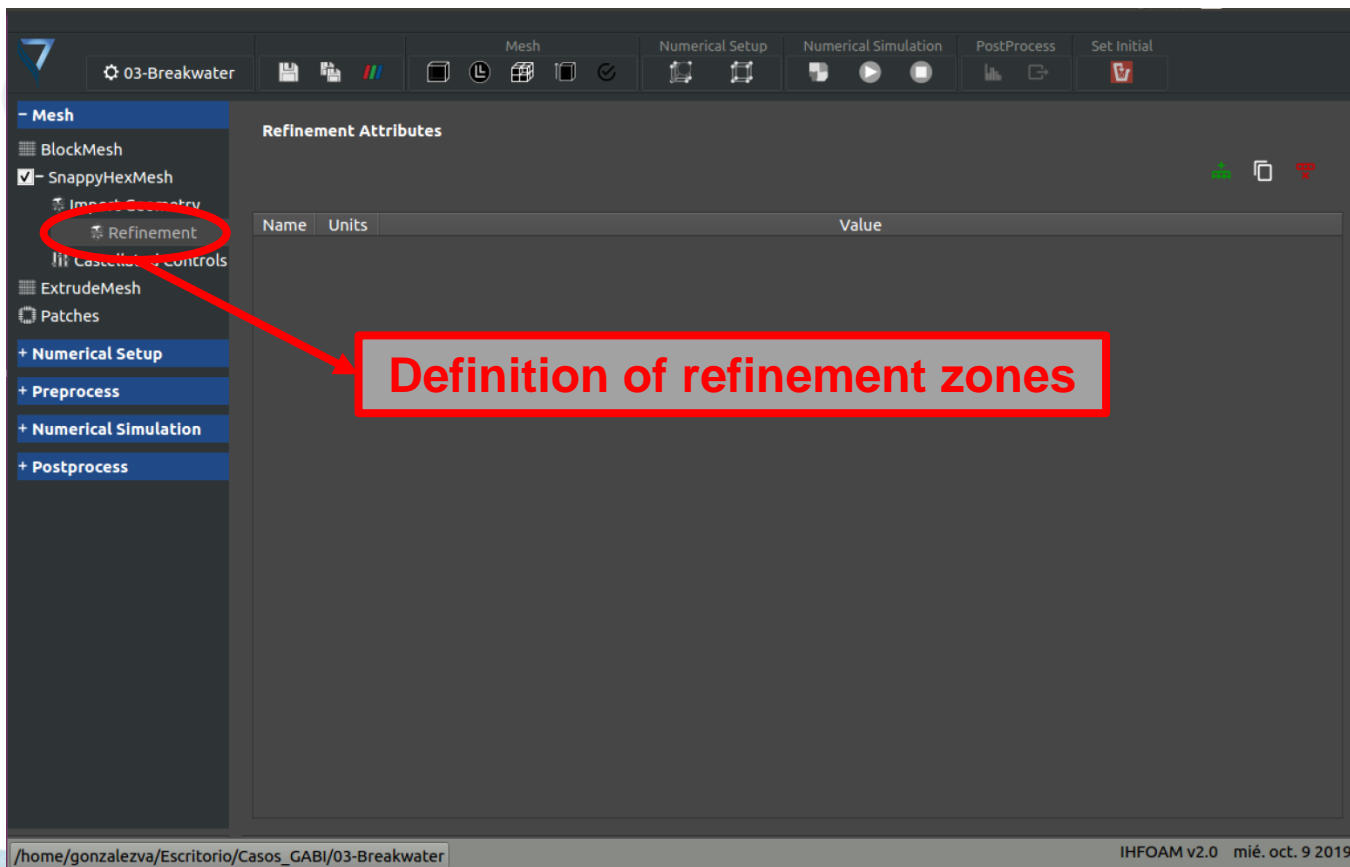
IHFOAM dialog box

Paraview button









The screenshot displays the IHFOAM v2.0 software interface. The left sidebar shows a tree view with the following items: Mesh, BlockMesh, SnappyHexMesh (checked), Import Geometry, Refinement, Castellated Controls (circled in red), ExtrudeMesh, Patches, Numerical Setup, Preprocess, Numerical Simulation, and Postprocess. The main panel is titled 'CastellatedMeshControls Attributes' and contains a table of parameters.

Name	Units	Value
maxLocalCells		100000
maxGlobalCells		3000000
minRefinementCells		0
nCellsBetweenLevels		2
resolveFeatureAngles		0
allowFreeStandingZoneFaces		<input checked="" type="checkbox"/>
locationInMesh		
X	m	-100
Y	m	5
Z	m	30

A red arrow points from the 'Castellated Controls' item in the sidebar to a red-bordered box containing the text 'Meshing parameters'.

03-Breakwater

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

ia
NTAL

snappyHexMesh button

The screenshot shows the IHFOAM v2.0 software interface. The top toolbar contains several icons, with the 'snappyHexMesh' icon (a cube with a grid) circled in red. Below the toolbar, the 'Mesh' tab is active, and the 'CastellatedMeshControls Attributes' table is visible. The table lists various attributes and their values:

Name	Units	Value
maxLocalCells		100000
maxGlobalCells		3000000
minRefinementCells		0
nCellsBetweenLevels		2
resolveFeatureAngles		0
allowFreeStandingZoneFaces		<input checked="" type="checkbox"/>
locationInMesh		
X	m	-100
Y	m	5
Z		30

An 'IHFOAM' dialog box is overlaid on the interface, containing the text 'OK!! snappyHexMesh -overwrite End' and an 'OK' button. The dialog box is also circled in red. Below the dialog box, the 'Message output' window shows the following text:

```

faces with skewness > 4 (internal) or 20 (boundary) : 0
faces with interpolation weights (0..1) < 0.05 : 0
faces with volume ratio of neighbour cells < 0.01 : 0
faces with face twist < 0.05 : 0
faces on cells with determinant < 0.001 : 0
Finished meshing without any errors
Finished meshing in = 26.4 s.
End
  
```

The status bar at the bottom of the window displays the path '/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater' and the version 'IHFOAM v2.0' with the date 'mié. oct. 9 2019'.

IHFOAM dialog box

ia
NTAL

extrudeMesh button

The screenshot shows the IHFOAM v2.0 software interface. The top toolbar has several tabs: Mesh, Numerical Setup, Numerical Simulation, PostProcess, and Set Initial. The 'Mesh' tab is active, and the 'extrudeMesh' button is circled in red. Below the toolbar, the 'CastellatedMeshControls Attributes' panel is visible, showing a table of attributes and their values. A dialog box titled 'IHFOAM' with the message 'OK!! extrudeMesh End' is open, and it is also circled in red. The bottom panel shows the 'Message output' log.

Name	Units	Value
maxLocalCells		100000
maxGlobalCells		3000000
minRefinementCells		0
nCellsBetweenLevels		2
resolveFeatureAngles		0
allowFreeStandingZoneFaces		<input checked="" type="checkbox"/>
locationInMesh		
X	m	-100
Y	m	5
Z	m	30

Message output

```

Not visited      : 0
Not collapsed    : 104735
Collapsed        : 0

Collapsing 0 cells
Writing mesh to "/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater/constant/region0"

End
  
```

IHFOAM dialog box

Define a boundary condition variable for each new patch of the mesh

Name	Units	Value
inlet		
outlet		
ground		
top		
side1		
side2		
rownWall		
type		
inGroups		
nFaces		800
startFace		266318
alpha.water		zeroGradient
U		fixedValue
p_rgh	Kg*m/s2	fixedFluxPressure

03-Breakwater

Mesh

Numerical Setup

Numerical Simulation

PostProcess

Set Initial

- Mesh

BlockMesh

SnappyHexMesh

Import Geometry

Refinement

Castellated Controls

ExtrudeMesh

Patches

+ Numerical Setup

+ Preprocess

+ Numerical Simulation

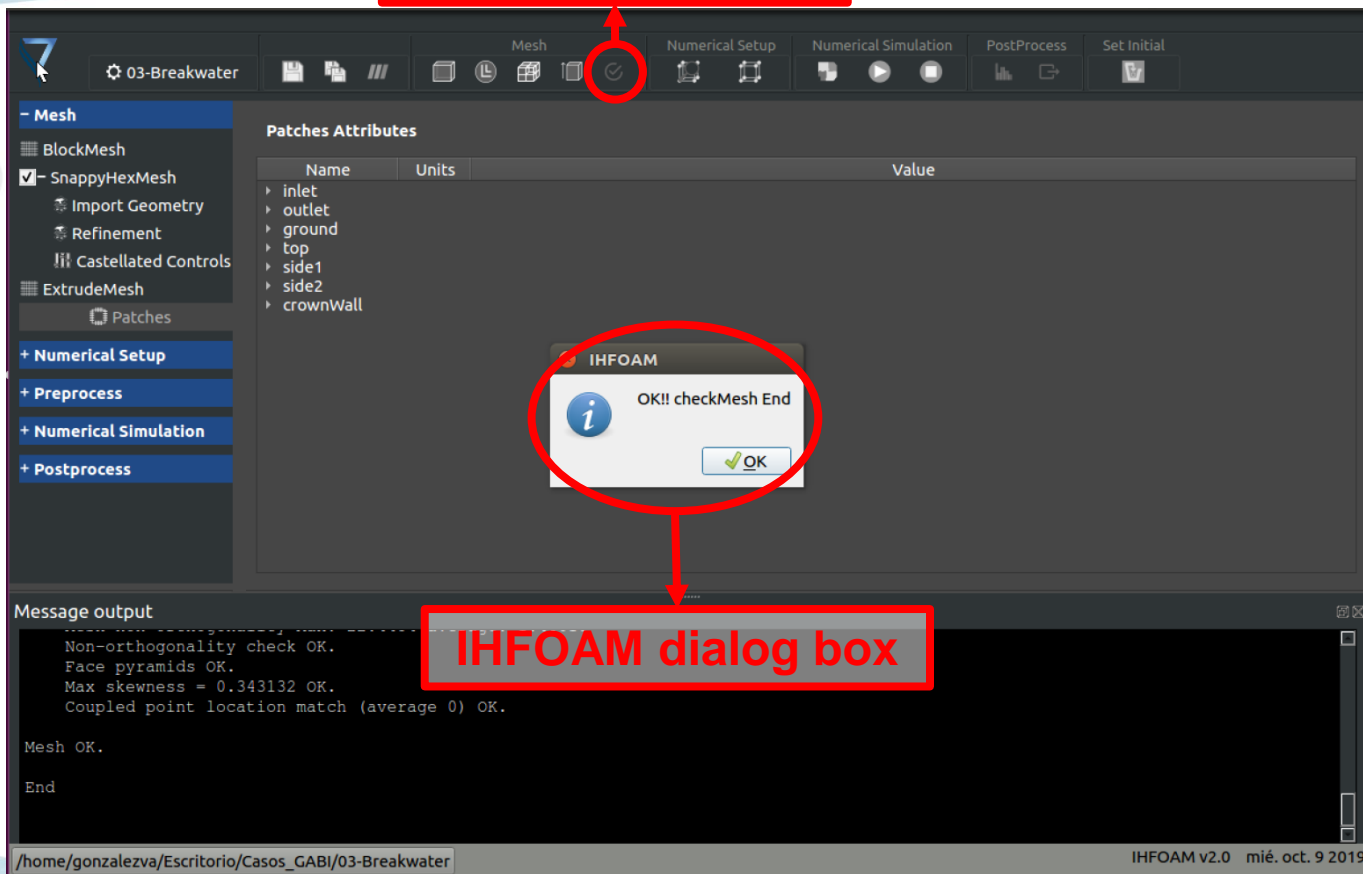
+ Postprocess

Patches Attributes

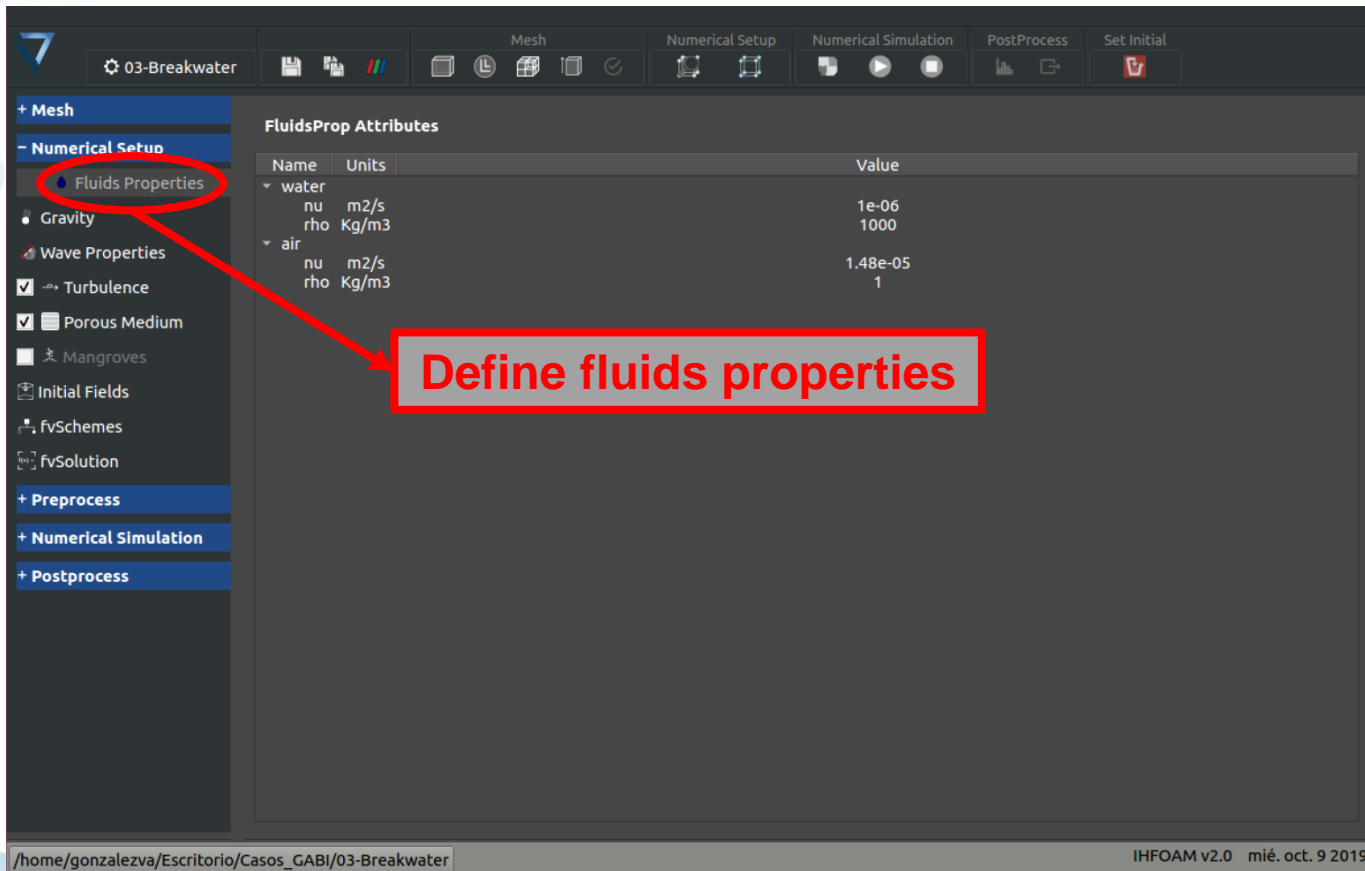
/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

checkMesh button



IHFOAM dialog box



The screenshot displays the IHFOAM v2.0 software interface. The left sidebar shows the 'Numerical Setup' tab selected, with 'Fluids Properties' highlighted. The main panel, titled 'FluidsProp Attributes', contains a table of fluid properties for water and air. A red arrow points from the 'Fluids Properties' menu item to a red box with the text 'Define fluids properties'.

Name	Units	Value
water		
nu	m ² /s	1e-06
rho	Kg/m ³	1000
air		
nu	m ² /s	1.48e-05
rho	Kg/m ³	1

Define fluids properties

The screenshot displays the IHFOAM v2.0 software interface. The left sidebar shows a tree view with categories: Mesh, Numerical Setup, Wave Properties, Initial Fields, FvSchemes, FvSolution, Preprocess, Numerical Simulation, and Postprocess. Under 'Numerical Setup', 'Fluids Properties' is expanded, and 'Gravity' is selected and circled in red. A red arrow points from this selection to a red-bordered box containing the text 'Define gravity'. The main panel shows the 'Gravity Attributes' table with the following data:

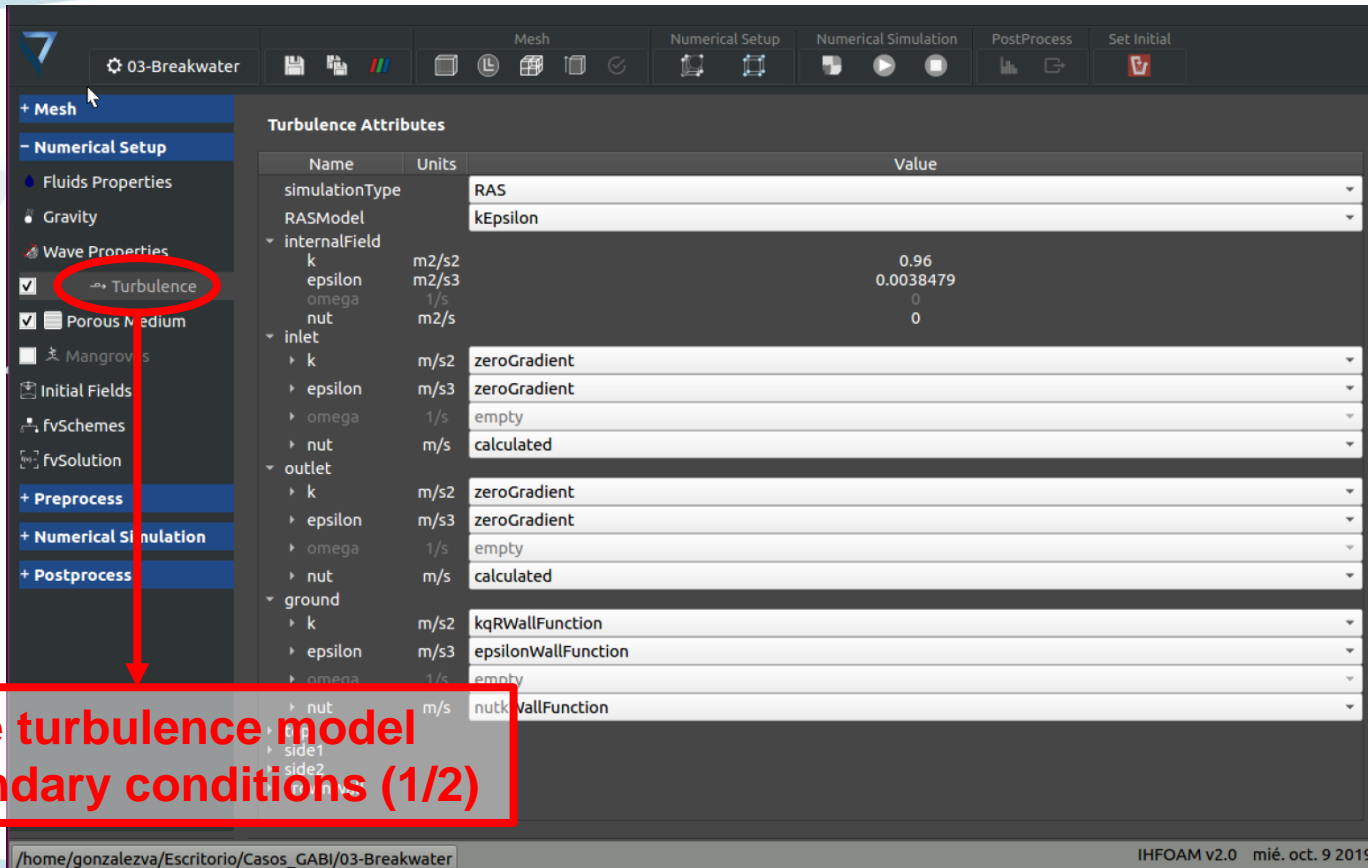
Name	Units	Value
gx	m/s ²	0
gy	m/s ²	0
gz	m/s ²	-9.81

The bottom status bar shows the file path: /home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater and the version/date: IHFOAM v2.0 mié. oct. 9 2019.

WaveProp Attributes

Name	Units	Value
waveGeneration		<input checked="" type="checkbox"/>
patch		inlet
waveModel		StokesV
wavePeriod	s	14
waveHeight	m	10
waveAngle	degree	0
nPaddle		1
activeAbsorption		<input checked="" type="checkbox"/>
rampTime	s	14
restart		<input type="checkbox"/>
waterDepth		<input type="checkbox"/>
waveAbsorption		<input checked="" type="checkbox"/>
patch		outlet
waveModel		shallowWaterAbsorption
nPaddle		1

Define wave properties (generation and absorption)



Turbulence Attributes

Name	Units	Value
simulationType		RAS
RASModel		kEpsilon
internalField		
k	m2/s2	0.96
epsilon	m2/s3	0.0038479
omega	1/s	0
nut	m2/s	0
inlet		
k	m/s2	zeroGradient
epsilon	m/s3	zeroGradient
omega	1/s	empty
nut	m/s	calculated
outlet		
k	m/s2	zeroGradient
epsilon	m/s3	zeroGradient
omega	1/s	empty
nut	m/s	calculated
ground		
k	m/s2	kqRWallFunction
epsilon	m/s3	epsilonWallFunction
omega	1/s	empty
nut	m/s	nutkWallFunction
side1		
side2		

Define turbulence model and boundary conditions (1/2)

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

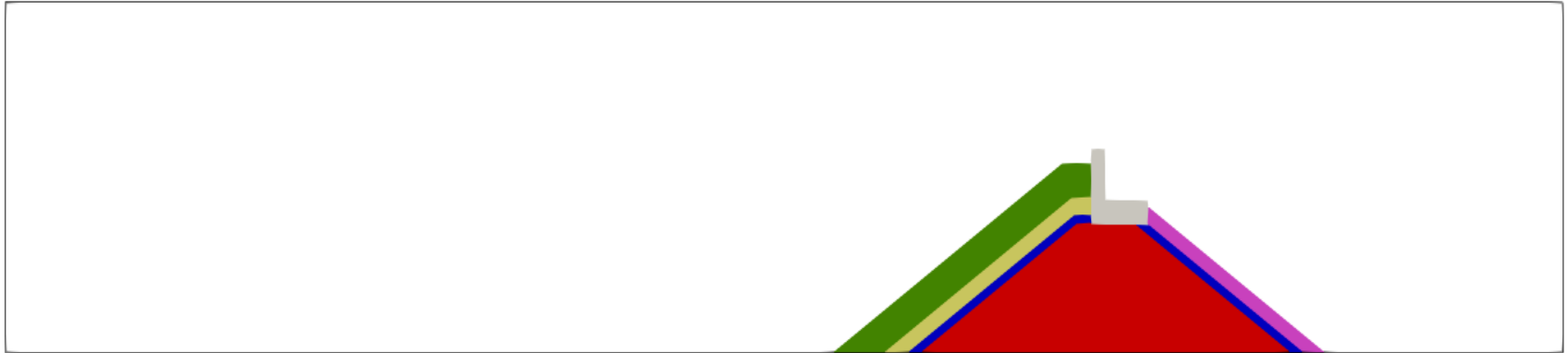
IHFOAM v2.0 mié. oct. 9 2019

Turbulence Attributes

Name	Units	Value
RASModel		kEpsilon
internalField		
inlet		
outlet		
ground		
top		
k	m/s ²	inletOutlet
epsilon	m/s ³	inletOutlet
omega	1/s	empty
nut	m/s	calculated
side1		
k	m/s ²	zeroGradient
epsilon	m/s ³	zeroGradient
omega	1/s	empty
nut	m/s	calculated
side2		
k	m/s ²	zeroGradient
epsilon	m/s ³	zeroGradient
omega	1/s	empty
nut	m/s	calculated
groundWall		
k	m/s ²	kqRWallFunction
epsilon	m/s ³	epsilonWallFunction
omega	1/s	empty
nut	m/s	nutkWallFunction

Define turbulence model and boundary conditions (2/2)

- Define, create and name the rubble mound elements (using Autocad, Rhino, etc.).



UNIVERSIDAD DE CANTABRIA

03-Breakwater

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

+ Mesh

- Numerical Setup

Fluids Properties

Gravity

Wave Properties

☒ Turbulence

☒ Porous Medium

Mangroves

Initial Fields

FvSchemes

FvSolution

+ Preprocess

+ Numerical Simulation

+ Postprocess

Porous Attributes

Name	Units	Value
core.stl		
D50		0.3
a		200
b		0.8
c		0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
filter.stl		
D50		0.4
a		200
b		1
c		0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
cubes_6tn.stl		
D50		1.37
a		200
b		1.1
c		0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
cubes_65tn.stl		
D50		1.37
a		200
b		1.1
c		0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10

Output

Domain Visor Result

Define porous layers (1/2)

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

Numerical Setup

- Fluids Properties
- Gravity
- Wave Properties
- ☒ Turbulence
- ☒ **Porous Medium**
- Mangroves
- Initial Fields
- FvSchemes
- FvSolution
- Preprocess
- Numerical Simulation
- Postprocess

Porous Attributes

Name	Units	Value
core.stl		
filter.stl		
cubes_6tn.stl		
cubes_65tn.stl		
D50		3
a		200
b		1.1
c		0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
rearArmourLayer.stl		
D50		0.6
a		200
b		1
c		0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10

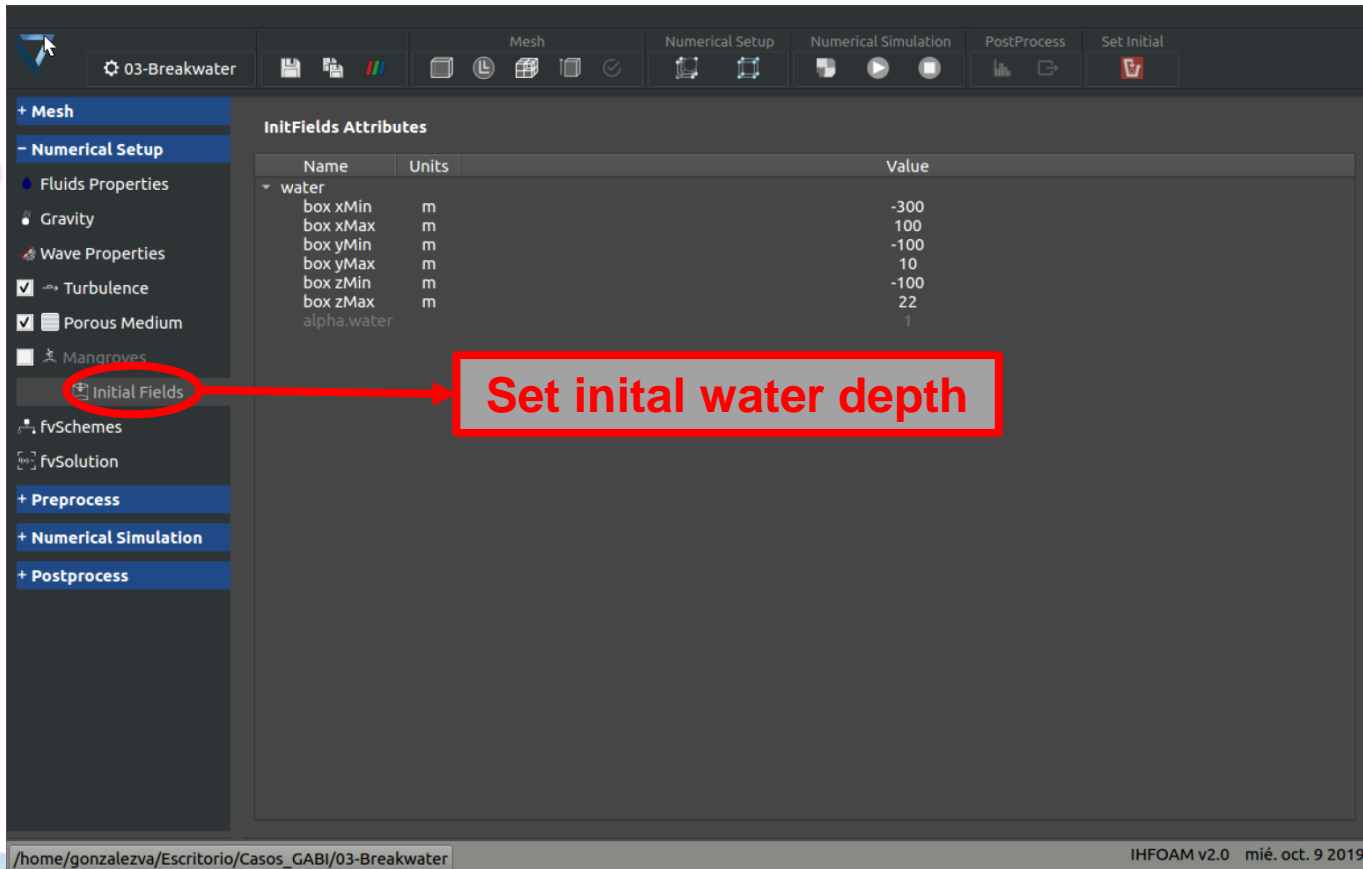
Domain Visor

3D visualization of the breakwater structure showing the rubble-mound breakwater and the surrounding water domain.

Define porous layers (2/2)

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019



03-Breakwater

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

+ Mesh

- Numerical Setup

- Fluids Properties
 - Gravity
 - Wave Properties
 - ☒ Turbulence
 - ☒ Porous Medium
 - ☐ Mangroves
 - Initial Fields**
- FvSchemes
- FvSolution

+ Preprocess

+ Numerical Simulation

+ Postprocess

InitFields Attributes

Name	Units	Value
water		
box xMin	m	-300
box xMax	m	100
box yMin	m	-100
box yMax	m	10
box zMin	m	-100
box zMax	m	22
alpha.water		1

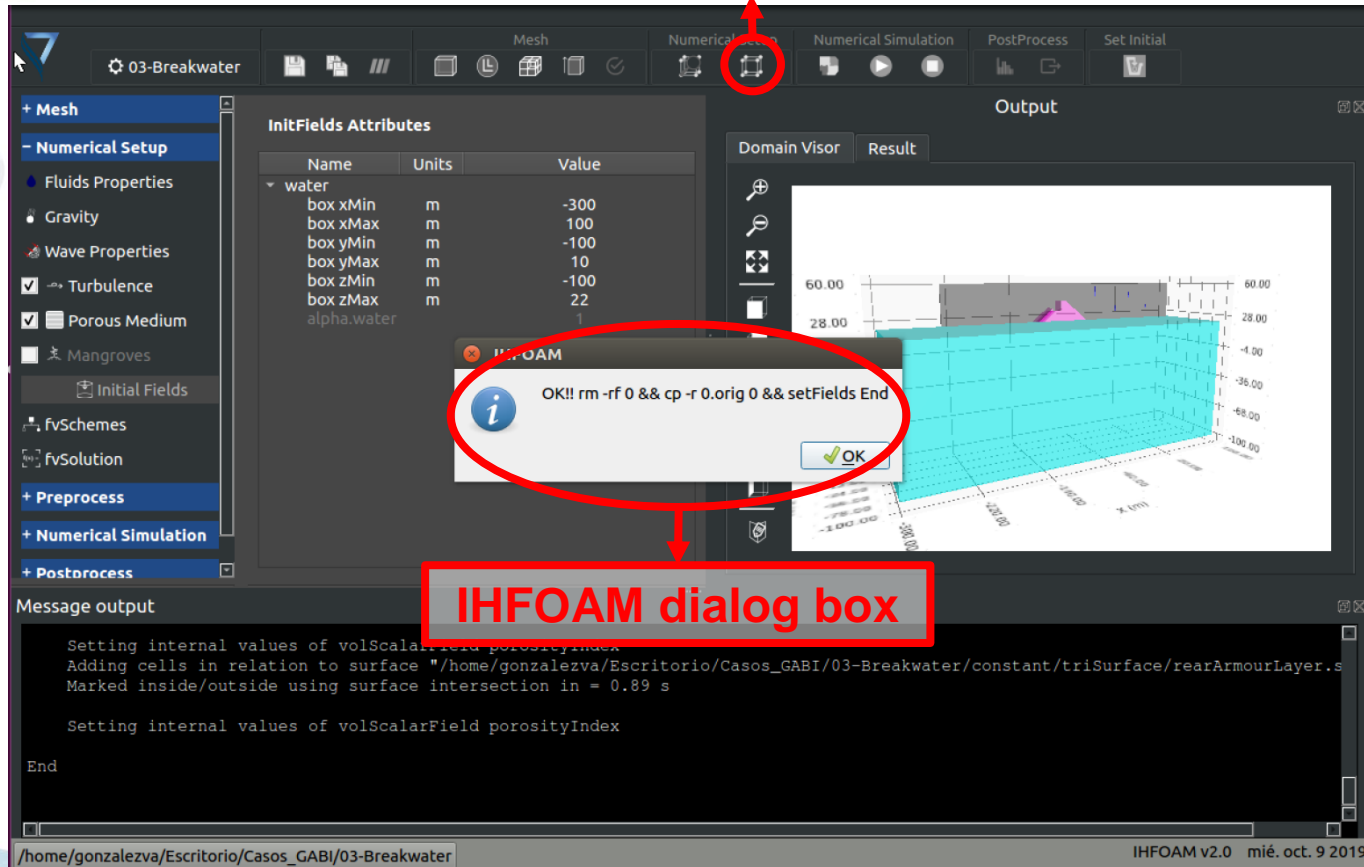
Set initial water depth

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

ia
NTAL

setFields button



The screenshot displays the IHFOAM v2.0 software interface. The top toolbar contains several buttons, with the 'setFields' button (represented by a cube icon) highlighted by a red circle and labeled 'setFields button'. Below the toolbar, the 'Numerical Setup' tab is active, showing the 'InitFields Attributes' table. The table lists various attributes for the 'water' domain, including box dimensions and alpha values. A red circle highlights the 'IHFOAM' dialog box that appears after clicking the 'setFields' button. The dialog box contains the text 'OK!! rm -rf 0 && cp -r 0.orig 0 && setFields End' and an 'OK' button. The background shows a 3D visualization of the domain with a breakwater structure. The bottom status bar indicates the file path and version information.

Name	Units	Value
water		
box xMin	m	-300
box xMax	m	100
box yMin	m	-100
box yMax	m	10
box zMin	m	-100
box zMax	m	22
alpha.water		1

OK!! rm -rf 0 && cp -r 0.orig 0 && setFields End

Message output

```
Setting internal values of volScalarField porosityIndex
Adding cells in relation to surface "/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater/constant/triSurface/rearArmourLayer.s
Marked inside/outside using surface intersection in = 0.89 s

Setting internal values of volScalarField porosityIndex

End
```

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

Paraview button

03-Breakwater

InitFields

Mesh

Numerical Setup

Numerical Simulation

PostProcess

Set Initial

+ Mesh

- Numerical Setup

Fluids Properties

Gravity

Wave Properties

☒ Turbulence

☒ Porous Medium

☐ Mangroves

Initial Fields

fvSchemes

fvSolution

+ Preprocess

+ Numerical Simulation

+ Postprocess

InitFields Attributes

Name	Units	Value
water		
box xMin	m	-300
box xMax	m	100
box yMin	m	-100
box yMax	m	10
box zMin	m	-100
box zMax	m	22
alpha.water		1

Domain Visor

Result

60.00

28.00

-1.00

-36.00

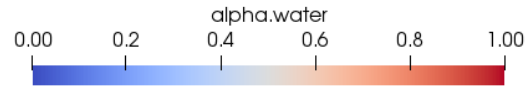
-68.00

-100.00

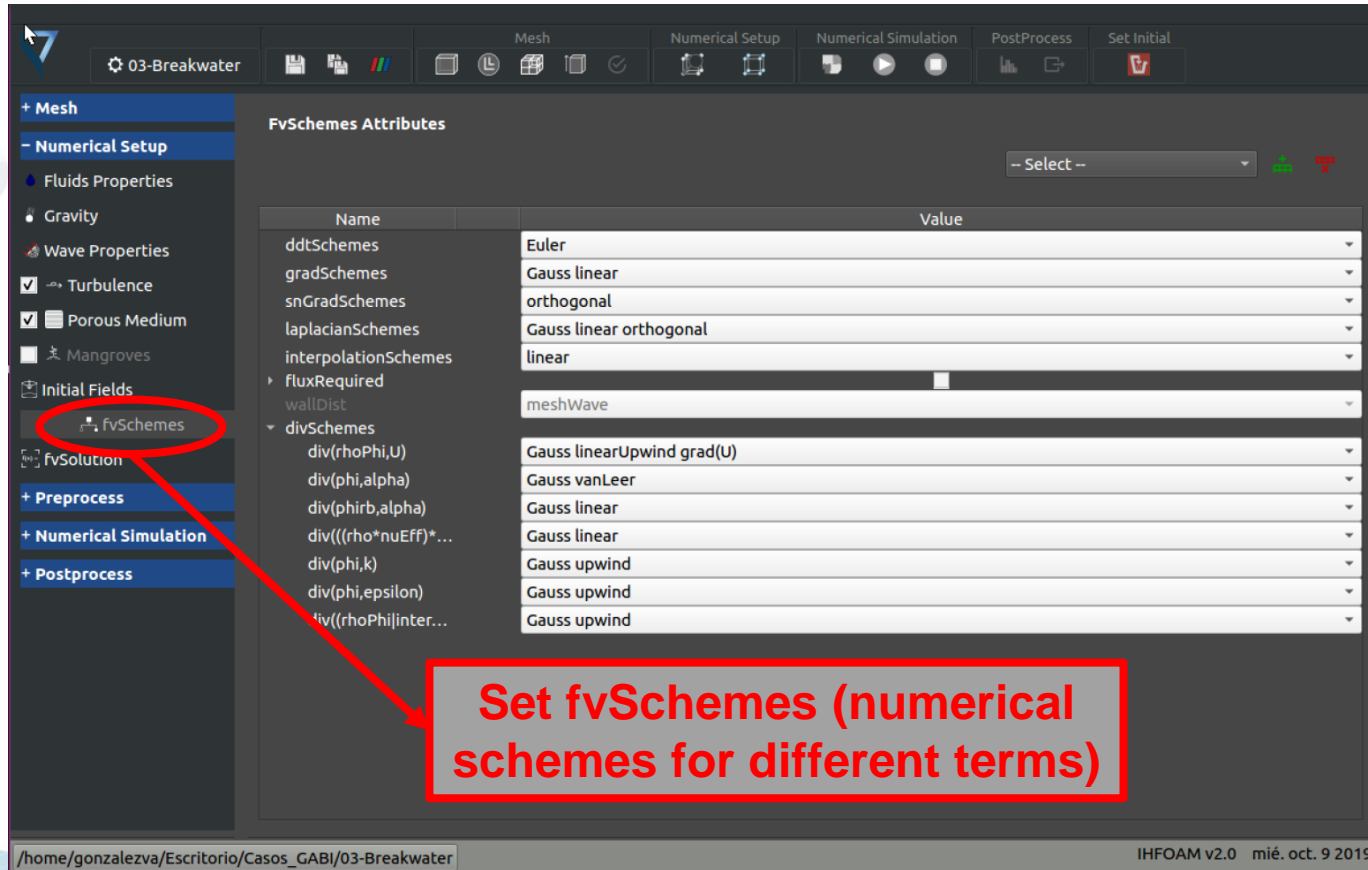
Z (m)

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019



INSTITUTO DE HIDRAULICA AMBIENTAL
UNIVERSIDAD DE CANTABRIA



The screenshot shows the OpenFOAM GUI with the 'FvSchemes Attributes' panel. The left sidebar lists various settings, with 'fvSchemes' highlighted by a red circle. A red arrow points from this circle to a red-bordered box containing the text 'Set fvSchemes (numerical schemes for different terms)'.

Name	Value
ddtSchemes	Euler
gradSchemes	Gauss linear
snGradSchemes	orthogonal
laplacianSchemes	Gauss linear orthogonal
interpolationSchemes	linear
fluxRequired	
wallDist	meshWave
divSchemes	
div(rhoPhi,U)	Gauss linearUpwind grad(U)
div(phi,alpha)	Gauss vanLeer
div(phi,b,alpha)	Gauss linear
div(((rho*nuEff)*...)	Gauss linear
div(phi,k)	Gauss upwind
div(phi,epsilon)	Gauss upwind
div((rhoPhi)inter...	Gauss upwind

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

Set fvSolution (equations solvers, tolerances and algorithms) (1/2)

The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view with the following items: + Mesh, - Numerical Setup (selected), Fluids Properties, Gravity, Wave Properties, Turbulence (checked), Porous Medium (checked), Mangroves, Initial Fields, fvSchemes, and Preprocess. The 'fvSchemes' item is circled in red, and a red arrow points from it to a red box containing the text 'Set fvSolution (equations solvers, tolerances and algorithms) (1/2)'. The main panel displays the 'fvSolution Attributes' for the 'fvSolution' scheme. The attributes are organized into a table with columns for the attribute name and its value.

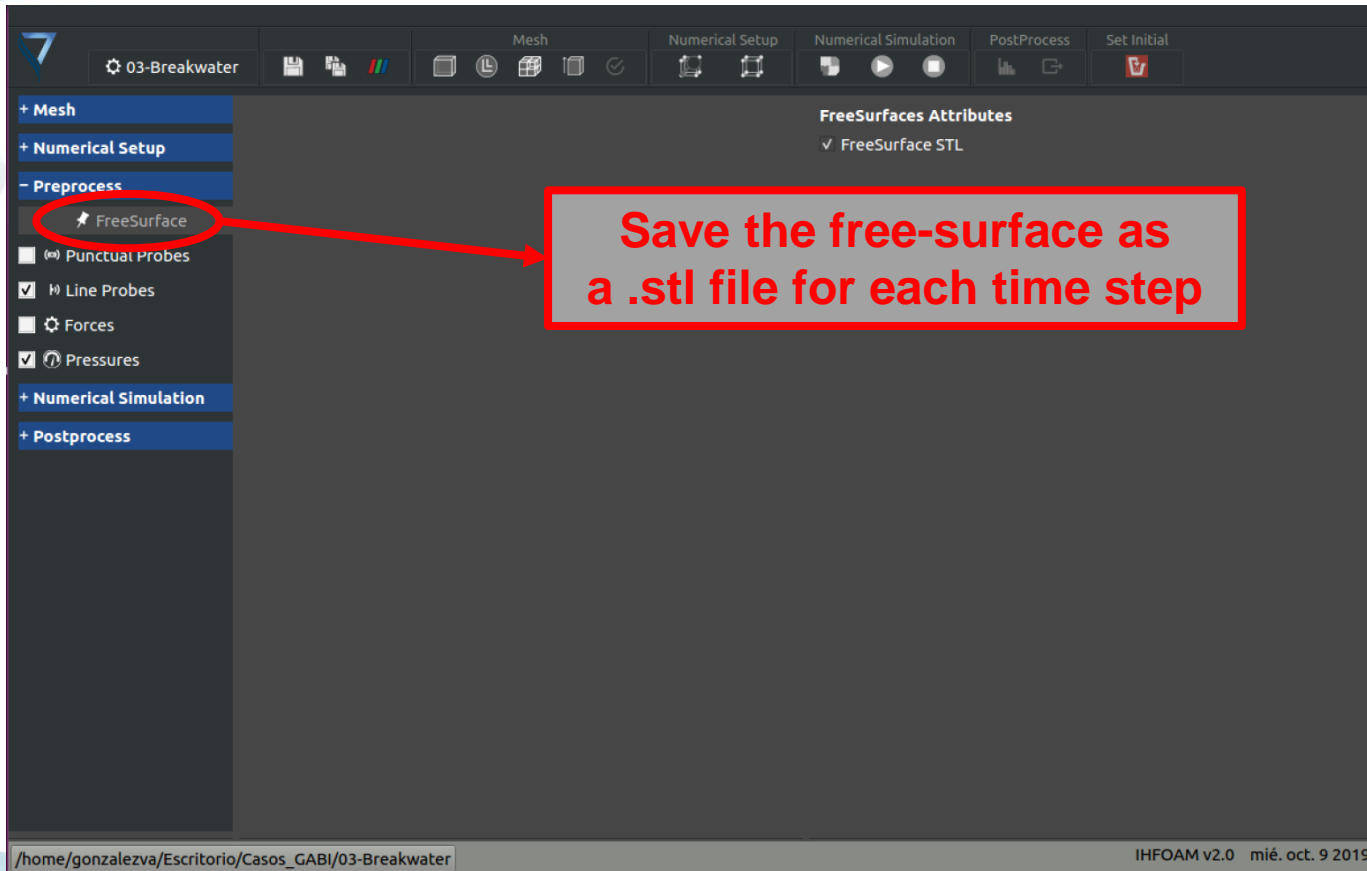
Attribute	Value
momentumPredictor	no
nCorrectors	3
nNonOrthogonalCorrectors	0
alpha.water.*	
alphaOuterCorrectors	
cAlpha	1
nAlphaCorr	1
nAlphaSubCycles	2
pcorr	
solver	PCG
preconditioner	DIC
tolerance	1e-06
relTol	0.1
pcorrFinal	
solver	PCG
preconditioner	DIC
tolerance	1e-07
relTol	0
p_rgh	
solver	PCG
preconditioner	DIC
tolerance	1e-06
relTol	0.1
p_rghFinal	
"(U k epsilon)"	
"(U k epsilon)Final"	

The bottom status bar shows the file path: /home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater and the version: IHFOAM v2.0 mié. oct. 9 2019.

The screenshot displays the IHFOAM v2.0 software interface. The left sidebar shows the 'FvSchemes' menu item circled in red. A red arrow points from this menu item to a red-bordered box containing the text: 'Set fvSolution (equations solvers, tolerances and algorithms) (1/2)'. The main panel shows the 'fvSolution' configuration for the 'p_rghFinal' field. The configuration is as follows:

Field	Property	Value
p_rghFinal	solver	PCG
	preconditioner	DIC
	tolerance	1e-07
	relTol	0
"(U k epsilon)"	solver	PBiCG
	preconditioner	DILU
	tolerance	1e-06
	relTol	0.1
"(U k epsilon)Final"	solver	PBiCG
	preconditioner	DILU
	tolerance	1e-07
	relTol	0

The bottom status bar shows the file path: /home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater and the version: IHFOAM v2.0, dated mié. oct. 9 2019.



03-Breakwater

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

+ Mesh

+ Numerical Setup

- Preprocess

FreeSurface

☐ Punctual Probes

☒ **Line Probes**

☐ Forces

☒ Pressures

+ Numerical Simulation

+ Postprocess

LineProbes Attributes

ID		X (m)	Y (m)	Zmin (m)	Zmax (m)	nPoints
1	line1	-100	2	0	60	1001
2	line2	-75	2	0	60	1001
3	line3	-50	2	0	60	1001
4	line4	-25	2	35	60	1001
5	line5	1	2	26	60	1001
6	line6	8	2	0	60	1001

Save Fields

☒ VOF

☐ Velocity U (m/s)

☐ Pressure P (Pa)

☐ Turbulent Kinematic Energy k (m2/s2)

☐ Dissipation Rate EPSILON (m2/s3)

☐ Specific Dissipation Rate W (1/s)

☐ Eddy Viscosity Mu* (m2/s)

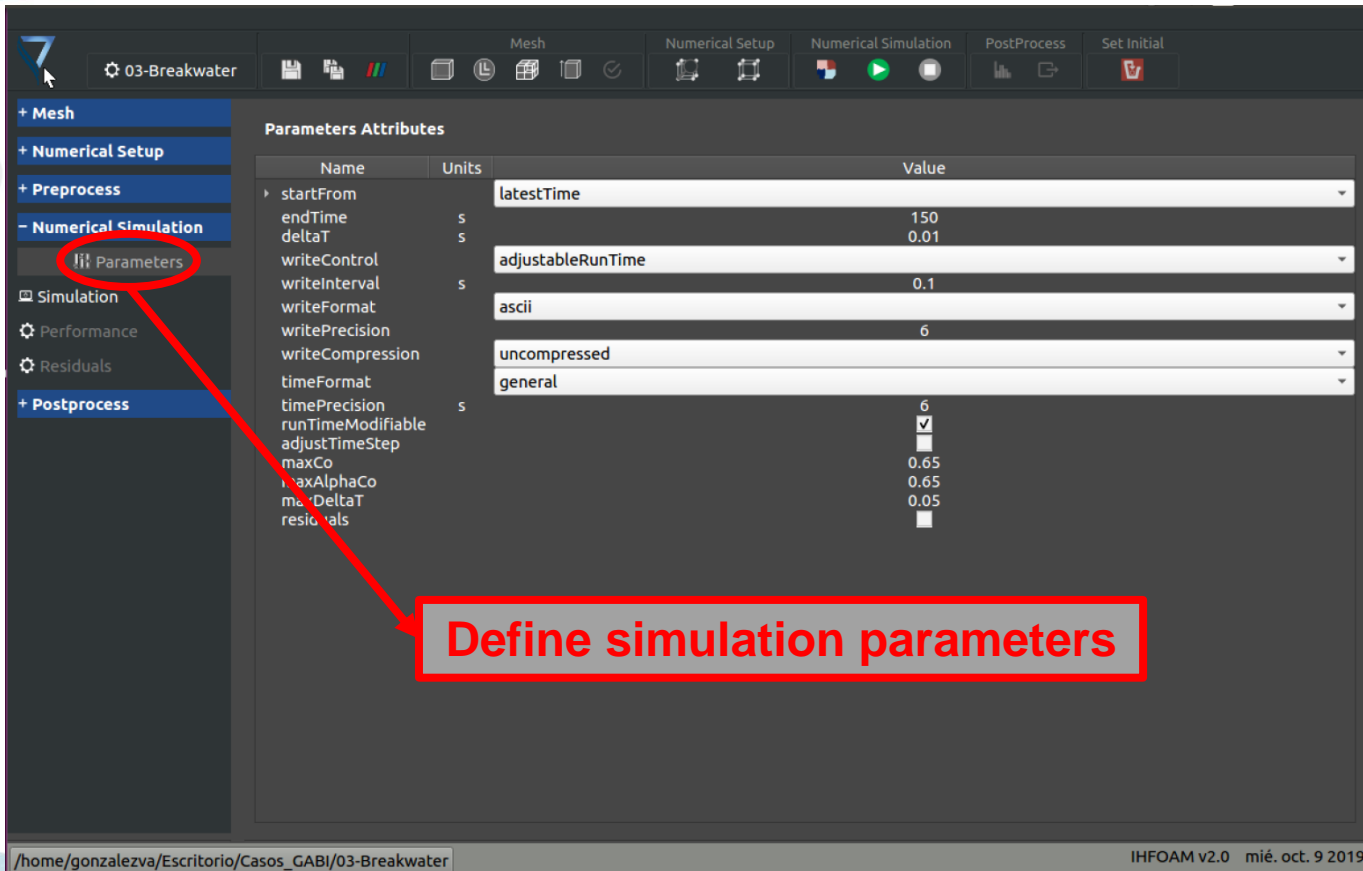
Output

Domain Visor Result

Set free-surface gauges

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019



The screenshot displays the IHFOAM v2.0 software interface. The left sidebar shows a tree view with the following items: + Mesh, + Numerical Setup, + Preprocess, - Numerical Simulation (expanded), Parameters (highlighted with a red circle), Simulation, Performance, Residuals, and + Postprocess. The main panel is titled 'Parameters Attributes' and contains a table of simulation parameters.

Name	Units	Value
startFrom		latestTime
endTime	s	150
deltaT	s	0.01
writeControl		adjustableRunTime
writeInterval	s	0.1
writeFormat		ascii
writePrecision		6
writeCompression		uncompressed
timeFormat		general
timePrecision	s	6
runTimeModifiable		<input checked="" type="checkbox"/>
adjustTimeStep		<input type="checkbox"/>
maxCo		0.65
maxAlphaCo		0.65
maxDeltaT		0.05
residuals		<input type="checkbox"/>

A red arrow points from the 'Parameters' option in the sidebar to a red-bordered box containing the text: **Define simulation parameters**

At the bottom of the window, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater and the version is IHFOAM v2.0, dated mié. oct. 9 2019.

decomposePar button

Run/Execute button

cleanCase button

Simulation

Run OpenFOAM case

The screenshot displays the OpenFOAM graphical user interface. On the left, a sidebar contains a tree view with categories like Mesh, Numerical Setup, Preprocess, Numerical Simulation, Parameters, Performance, Residuals, and Postprocess. The 'Simulation' category is highlighted. The main window shows 'Simulation Attributes' with a table of settings. The top toolbar includes icons for file operations, meshing, and simulation control. Red boxes and arrows highlight specific features: 'decomposePar button' points to the 'decomposePar' icon in the toolbar; 'Run/Execute button' points to the green play button; 'cleanCase button' points to the trash icon; 'Simulation' points to the 'Simulation' item in the sidebar; and 'Run OpenFOAM case' points to a button at the bottom of the main window.

Name	Units	Value
Numerical Solver		ihFoam
Execution Type		Parallel
method		hierarchical
Decompose X		2
Decompose Y		1
Decompose Z		1
numberOfSubdomains		2

/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

decomposePar button

The screenshot displays the IHFOAM software interface. The top toolbar includes buttons for Mesh, Numerical Setup, Numerical Simulation, PostProcess, and Set Initial. The 'Numerical Simulation' button is highlighted with a red circle and an arrow pointing to it from the 'decomposePar button' label. The left sidebar shows a tree view with categories like Mesh, Numerical Setup, Preprocess, Numerical Simulation, Parameters, Simulation, Performance, Residuals, and Postprocess. The 'Numerical Simulation' category is expanded, showing a table of simulation attributes.

Name	Units	Value
Numerical Solver		ihFoam
Execution Type		Parallel
method		hierarchical
Decompose X		2
Decompose Y		1
Decompose Z		1
numberOfSubdomains		4

A red circle highlights a message box that appears after clicking the 'decomposePar' button. The message box contains the text 'OK!! rm -rf processor* && decomposePar End' and an 'OK' button. An arrow points from this message box to a red box containing the text 'Case decomposed correctly'.

The 'Message output' panel at the bottom shows the following text:

```
Max number of faces between processors: 216 (0% of total faces)
Time = 0
Processor 0: field transfer
Processor 1: field transfer
End
```

Case decomposed correctly

Run/Execute button

The screenshot displays the IHFOAM v2.0 software interface. The top toolbar includes buttons for Mesh, Numerical Setup, Numerical Simulation, PostProcess, and Set Initial. The 'Numerical Simulation' button is highlighted with a red circle and an arrow pointing to it from the 'Run/Execute button' label. The left sidebar shows a tree view with 'Mesh', 'Numerical Setup', 'Preprocess', 'Numerical Simulation', 'Parameters', 'Simulation', 'Performance', 'Residuals', and 'Postprocess'. The 'Simulation' section is expanded, showing 'Simulation Attributes' with a table:

Name	Units	Value
Numerical Solver		ihFoam
Execution Type		Parallel

The 'Output' panel on the right shows a 3D visualization of the breakwater structure. A red circle highlights a message box that says 'OK!! mpirun -np 2 ihFoam -parallel End'. Below this, the 'Message output' panel shows the following text:

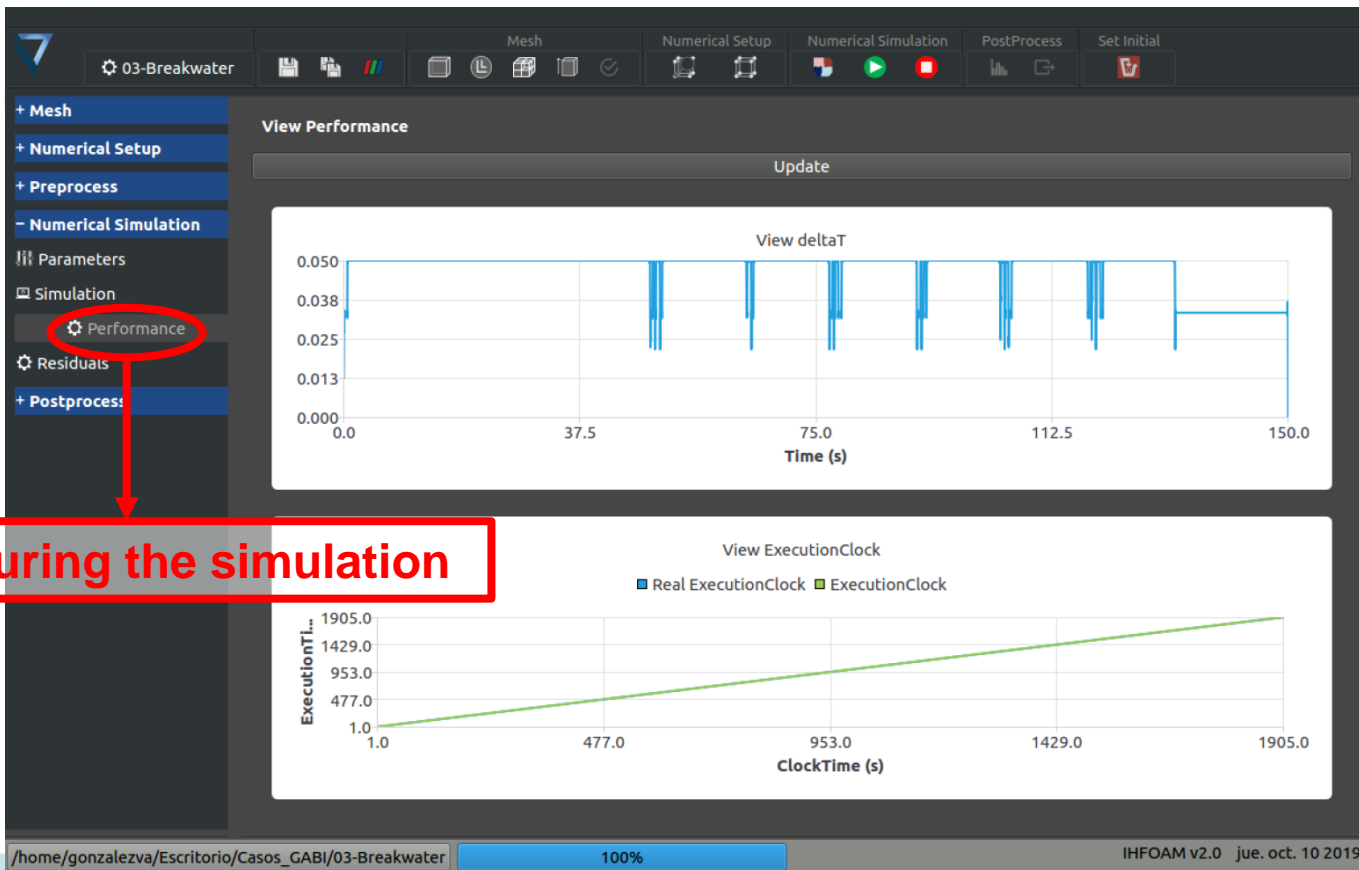
```

DICPCG: Solving for p_rgh, Initial residual = 2.7141e-05, Final residual = 9.20209e-08, No iterations 65
time step continuity errors : sum local = 1.99171e-07, global = -2.45762e-08, cumulative = 0.000999367
DILUPBiCG: Solving for epsilon, Initial residual = 0.00301186, Final residual = 1.44779e-08, No iterations 5
DILUPBiCG: Solving for k, Initial residual = 0.00156239, Final residual = 6.16755e-08, No iterations 5
ExecutionTime = 1902.88 s, ClockTime = 1905 s
End
Finalising parallel run
  
```

A red circle highlights the 'End' line in the message output. A red arrow points from the 'OK!! mpirun -np 2 ihFoam -parallel End' message box to the 'Case correctly executed!!' label. The status bar at the bottom shows the file path '/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater', the progress '100%', and the version 'IHFOAM v2.0' dated 'jue. oct. 10 2019'.

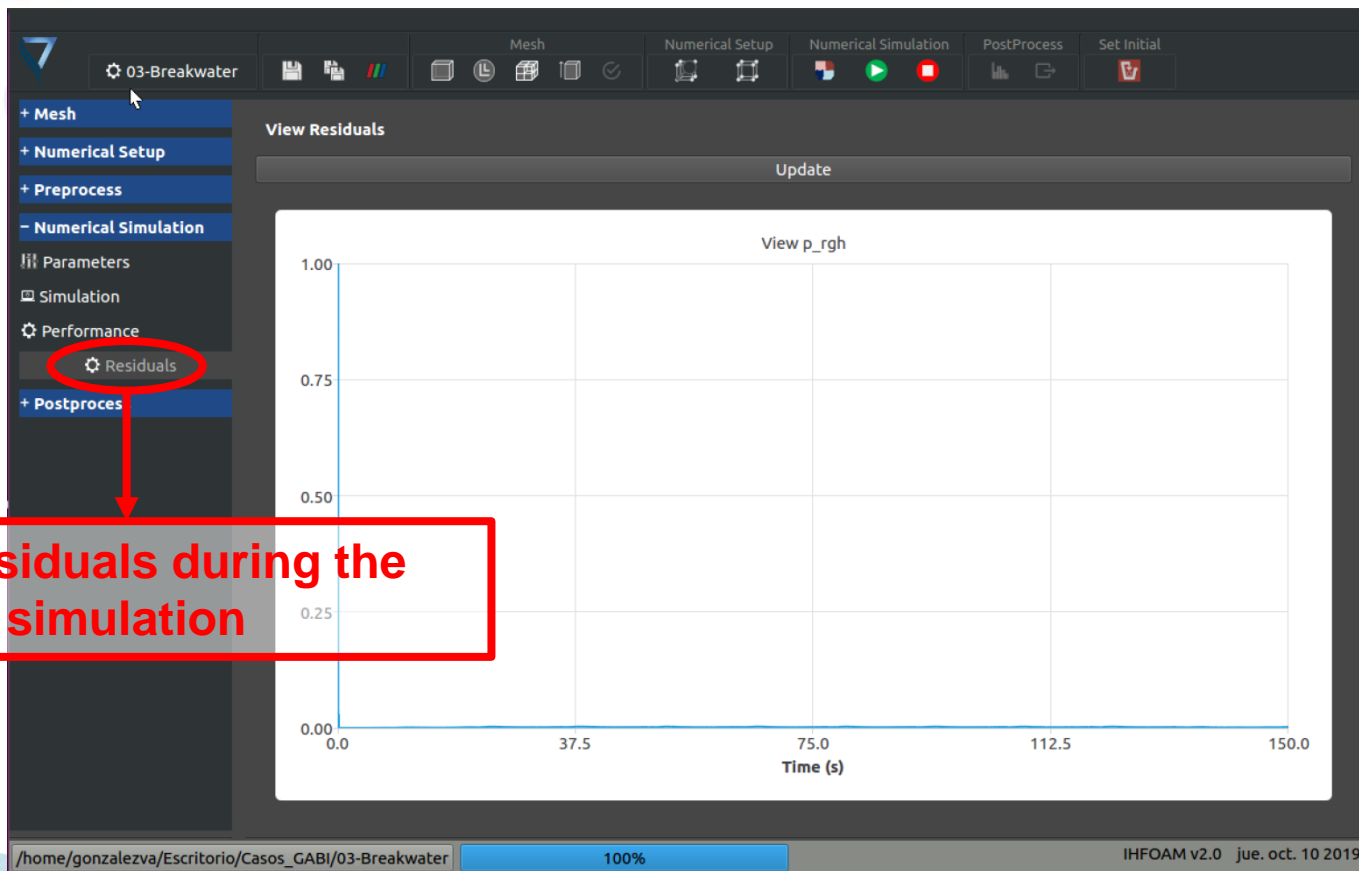
Case correctly executed!!

End

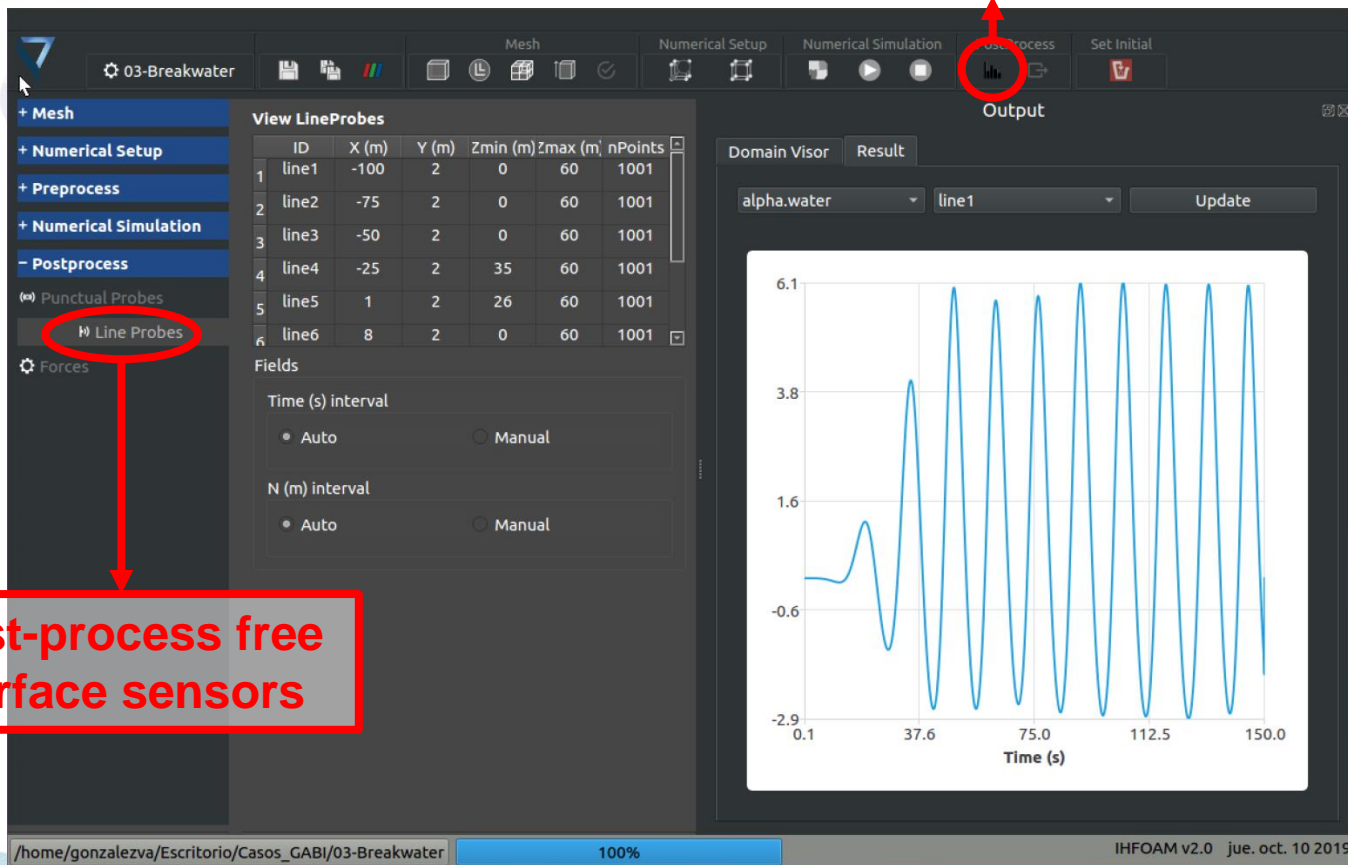


plot dt during the simulation

ia
NTAL

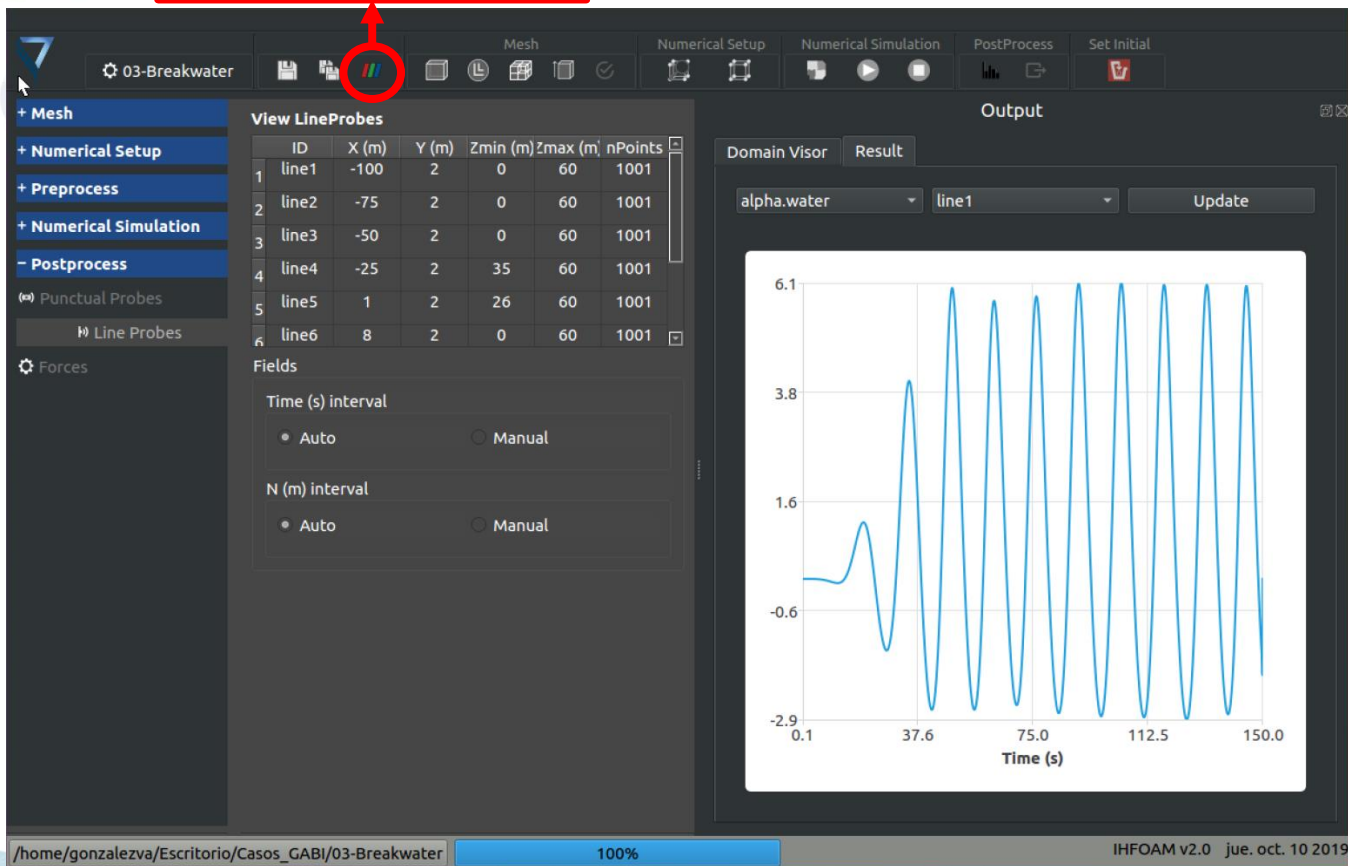


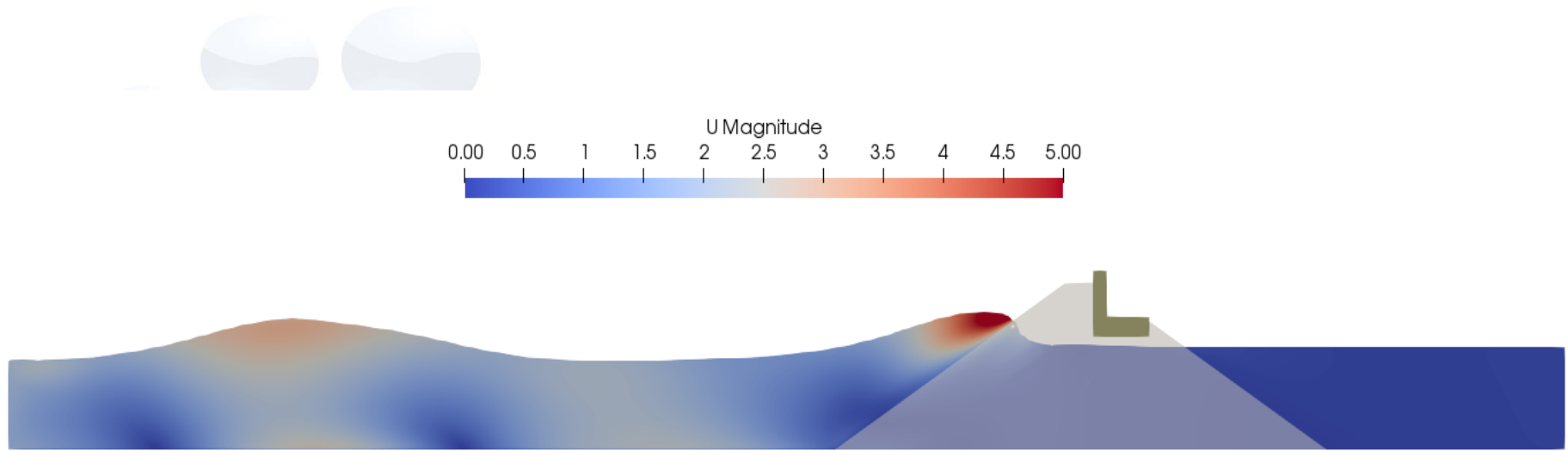
plot free surface sensors



Post-process free surface sensors

Paraview button

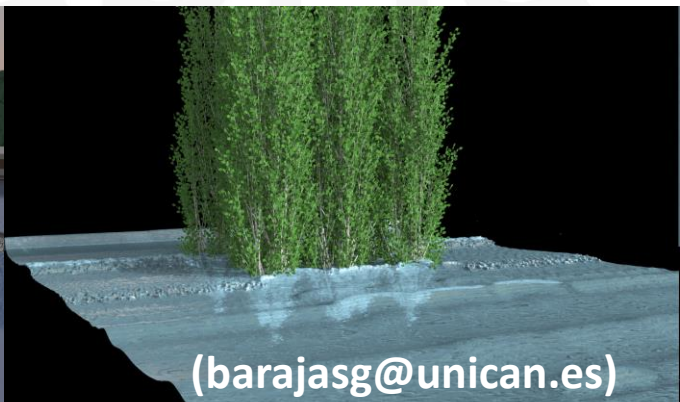
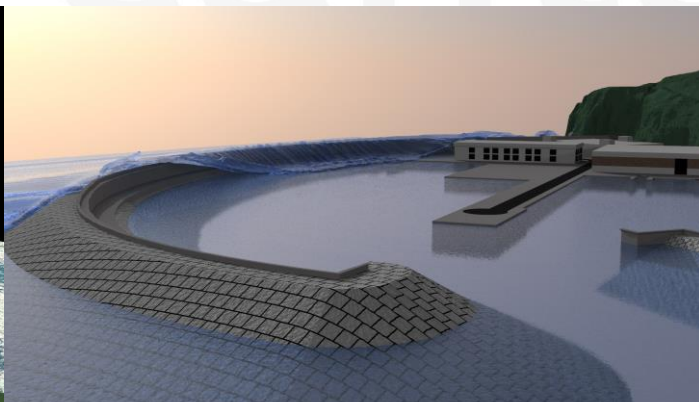
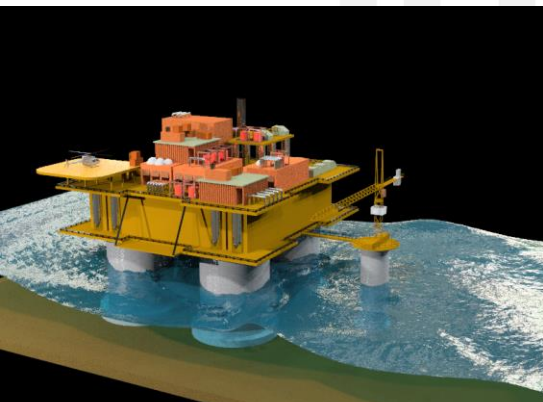




UNIVERSIDAD DE CANTABRIA



Gabriel Barajas, Javier L. Lara, María Maza, Alejandro Gonzalez



(barajasg@unican.es)