



IHCantabria

UNIVERSIDAD DE CANTABRIA

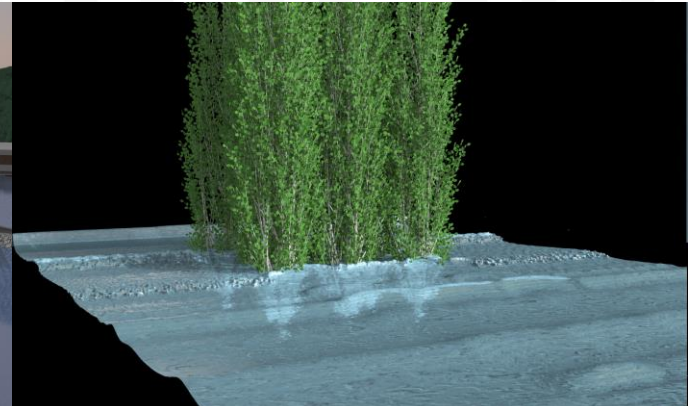
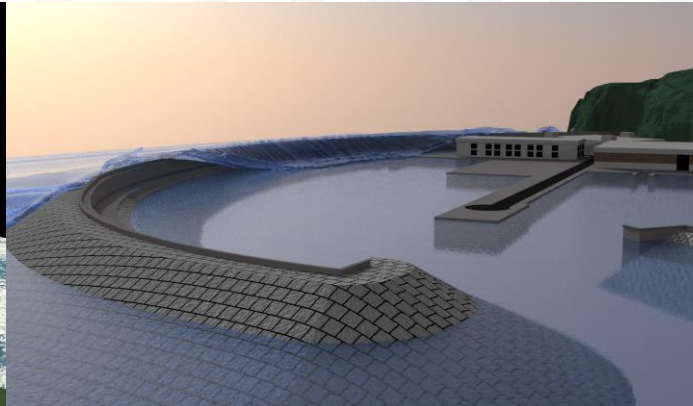
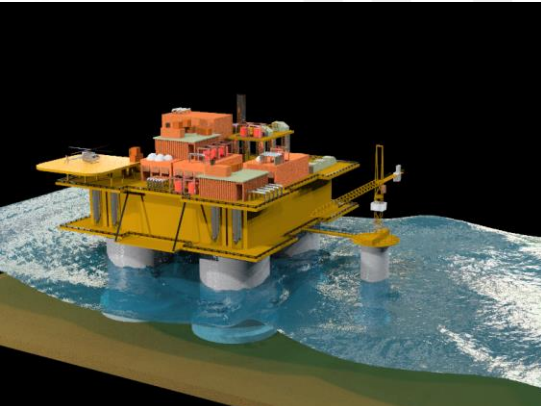
R+D+i for a Sustainable Development

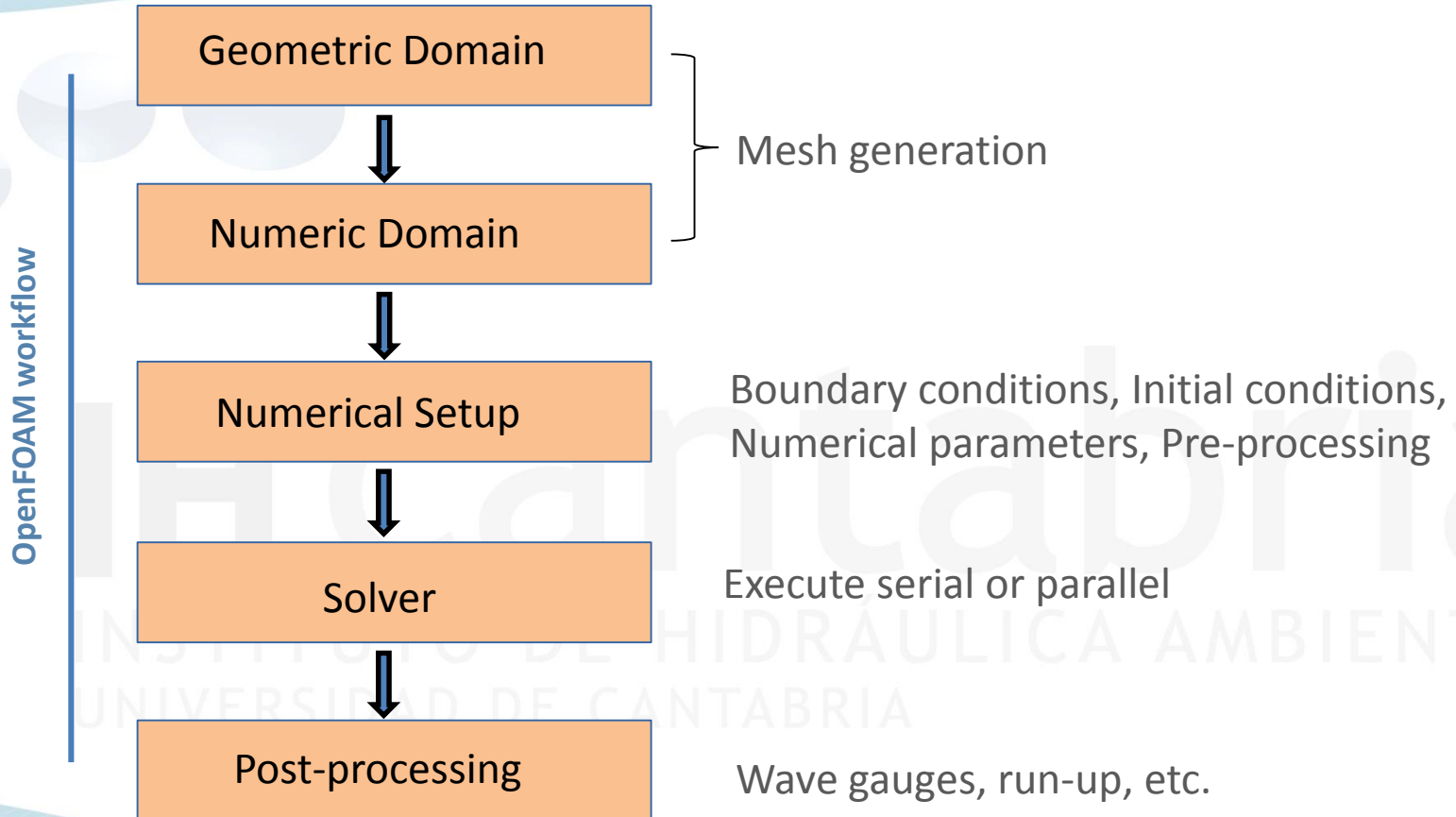
(IHFOAM GUI)

IHFOAM applied to Coastal Engineering

Regular waves in empty basin (3D)

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





OpenFOAM case

0

- alpha.water
- p_rgh
- U

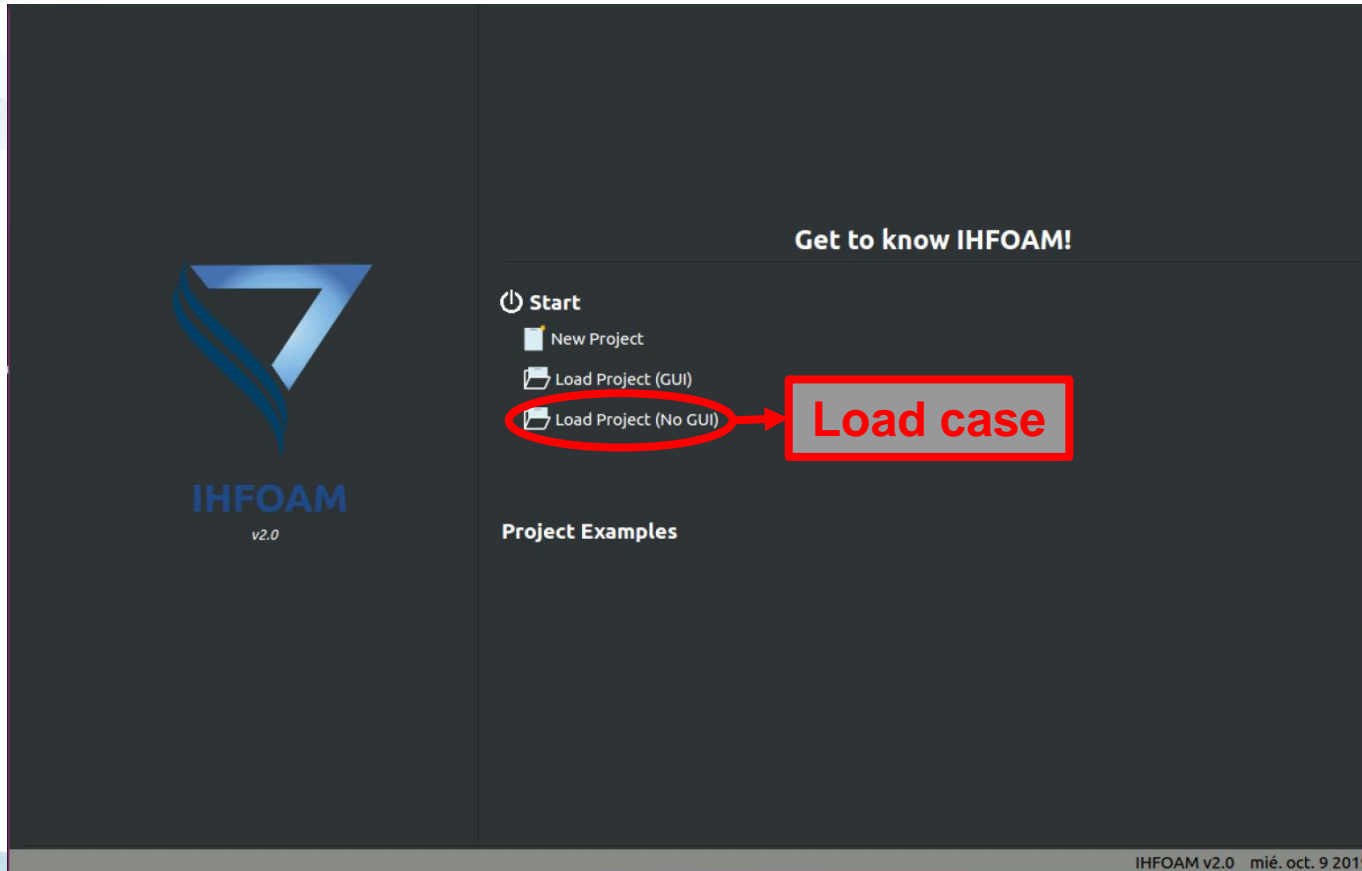
constant

- g
- transportProperties
- turbulenceProperties
- waveProperties

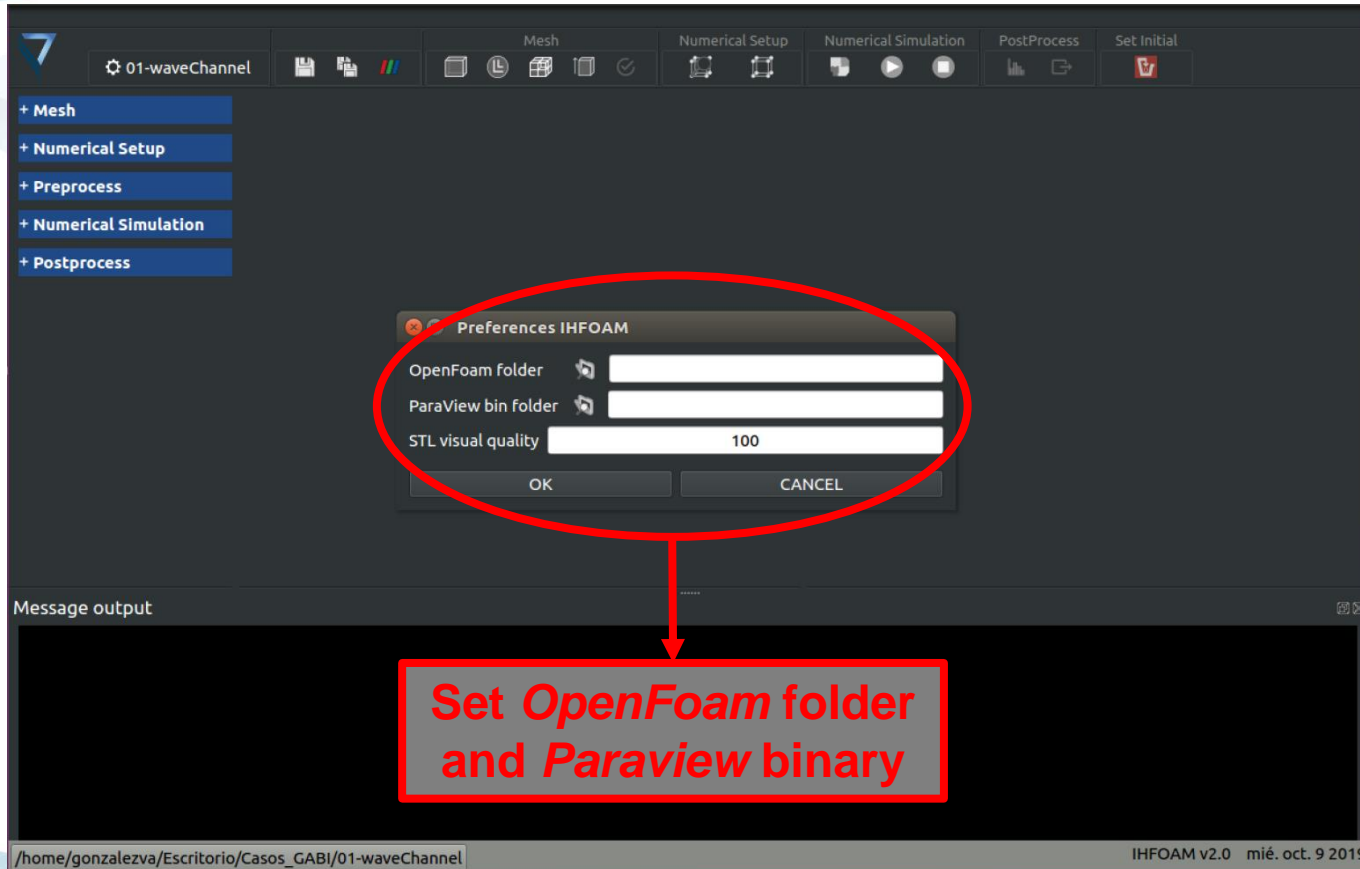
system

- blockMeshDict
- setFieldsDict
- decomposeParDict
- fvSchemes
- fvSolution
- controlDict

IHFOAM GUI

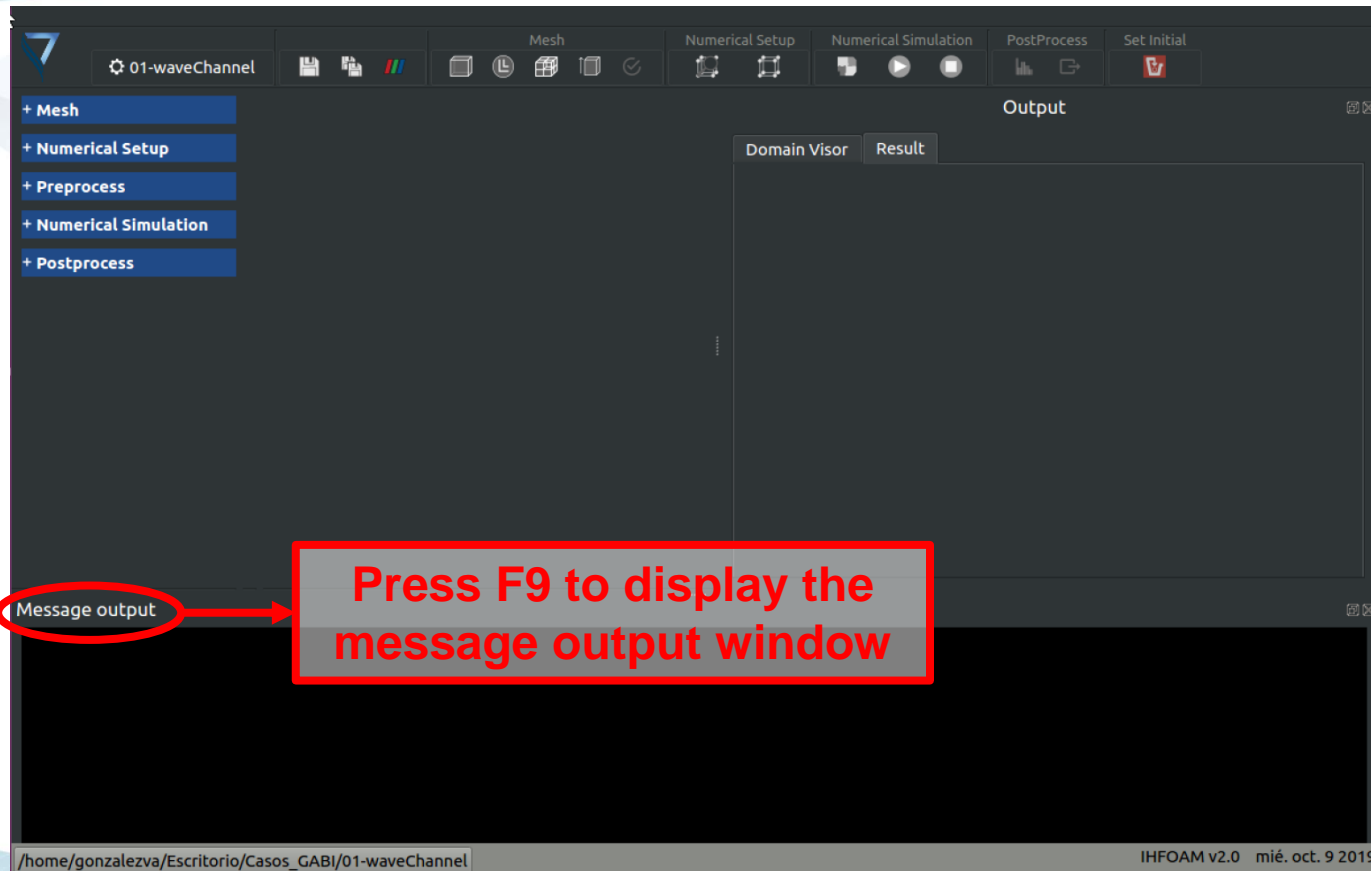


ia
ENTAL



**Set *OpenFoam* folder
and *Paraview* binary**

ia
ENTAL



**Save single pannel modified
or save all pannels modified.**

**Regular waves in empty
basin (3D)**

Mesh

- BlockMesh ***
- SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- ExtrudeMesh
- Patches
- + Numerical Setup
- + Preprocess
- + Numerical Simulation
- + Postprocess

BlockMesh Attributes

Name	Units	Value
Space Mesh		3D
Type Mesh		static
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	0
Xmax	m	10
Ymin	m	0
Ymax	m	2
Zmin	m	0
Zmax	m	1.4
Nx		250
Ny		50
Nz		70

**Define geometric domain
and mesh discretization**

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL

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, and the 'blockMesh' button is highlighted with a red circle and an arrow. Below the menu, the 'BlockMesh Attributes' panel is visible, showing a table of attributes for the 'Mesh main' block. A red circle highlights an error dialog box that appears when the 'blockMesh' button is clicked. The dialog box contains the text 'OK!! rm -rf constant/polyMesh/ && blockMesh End' and an 'OK' button. Below the dialog box, the 'Message output' panel shows the results of the blockMesh command, including the number of cells and faces for each patch.

BlockMesh Attributes

Name	Units	Value
Space Mesh		3D
Type Mesh		static
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	0
Xmax	m	10
Ymin	m	0
Ymax	m	2
Zmi		
Zma		
Nx		
Ny		
Nz		

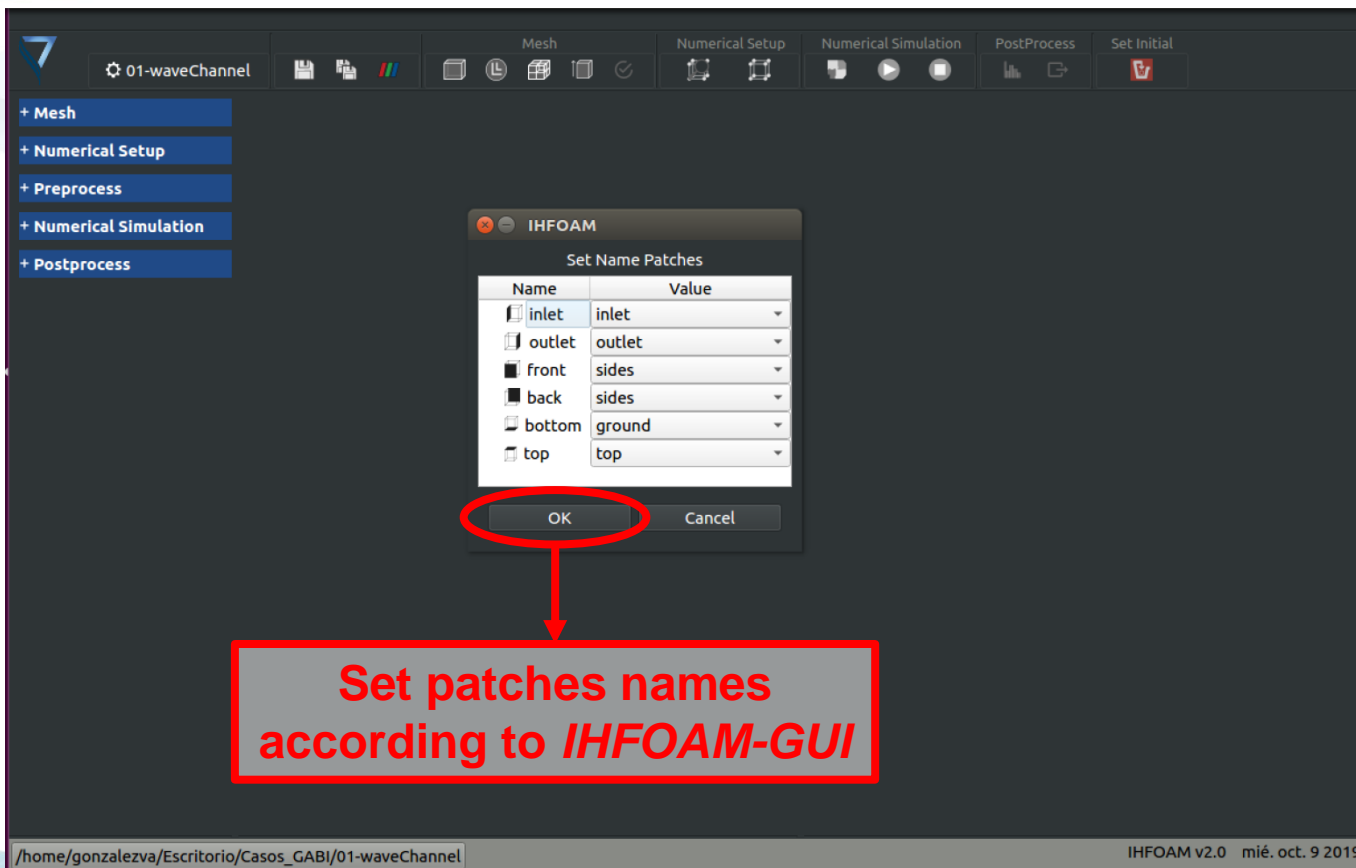
Message output

```

-----
patch 0 (start: 2591500 size: 3500) name: inlet
patch 1 (start: 2595000 size: 3500) name: outlet
patch 2 (start: 2598500 size: 12500) name: ground
patch 3 (start: 2611000 size: 12500) name: top
patch 4 (start: 2623500 size: 35000) name: sides

End
  
```

IHFOAM dialog box



ia
ENTAL

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

Patches Attributes

Name	Units	Value
inlet		
type		patch
nFaces		360
startFace		334260
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
nGroups		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

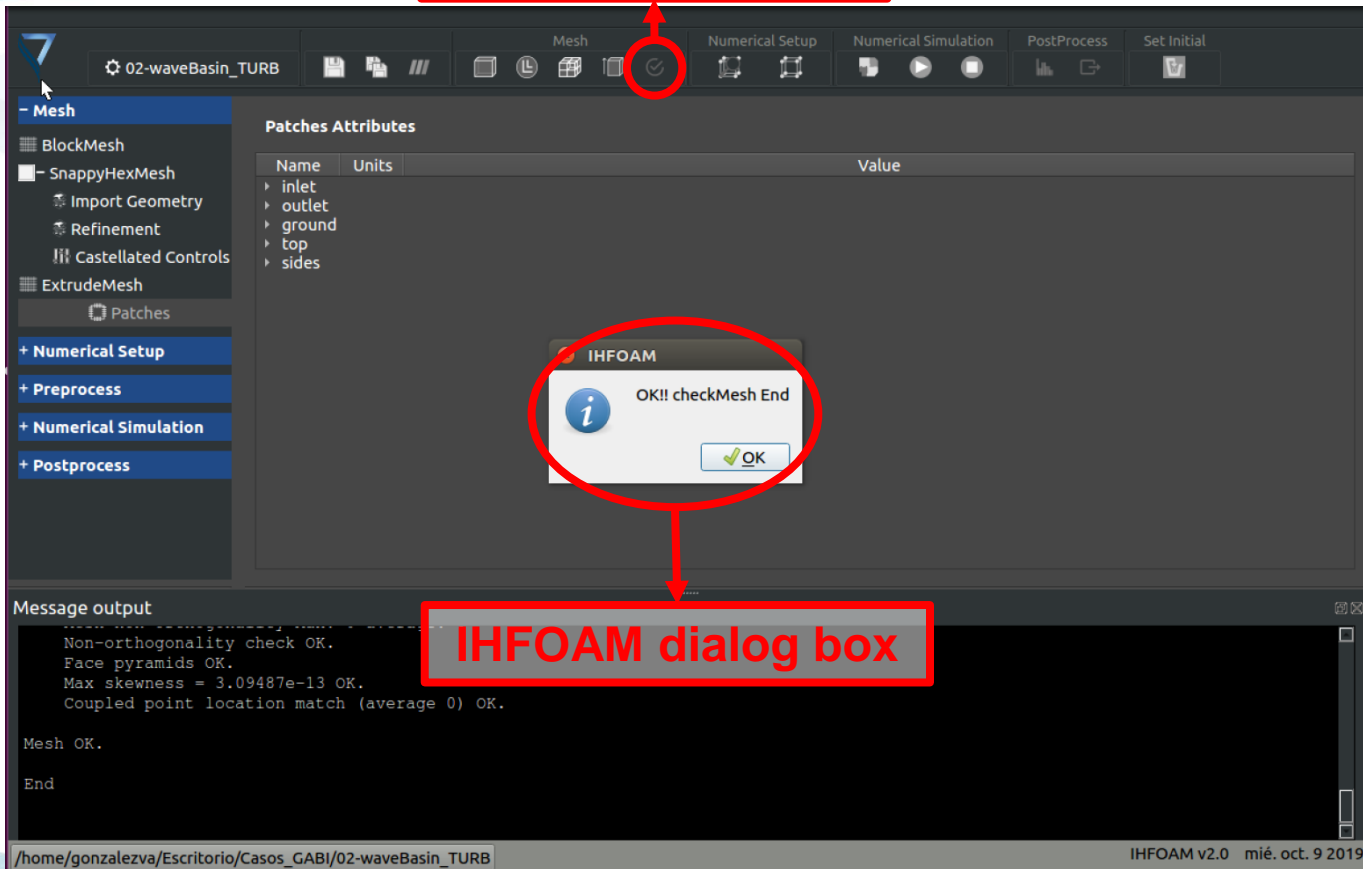
Define a boundary condition variable for each patch of the mesh (1/2)

Name	Units	Value
inlet		
outlet		
ground		
top		
type		patch
nFaces		1980
startFace		336960
alpha.water		inletOutlet
U		pressureInletOutletVelocity
p_rgh	Kg*m/s2	totalPressure
side1		
type		patch
nFaces		19800
startFace		338940
alpha.water		zeroGradient
U		slip
p_rgh	Kg*m/s2	fixedFluxPressure
side2		
type		patch
nFaces		19800
startFace		358740
alpha.water		zeroGradient
U		slip
p_rgh	Kg*m/s2	fixedFluxPressure

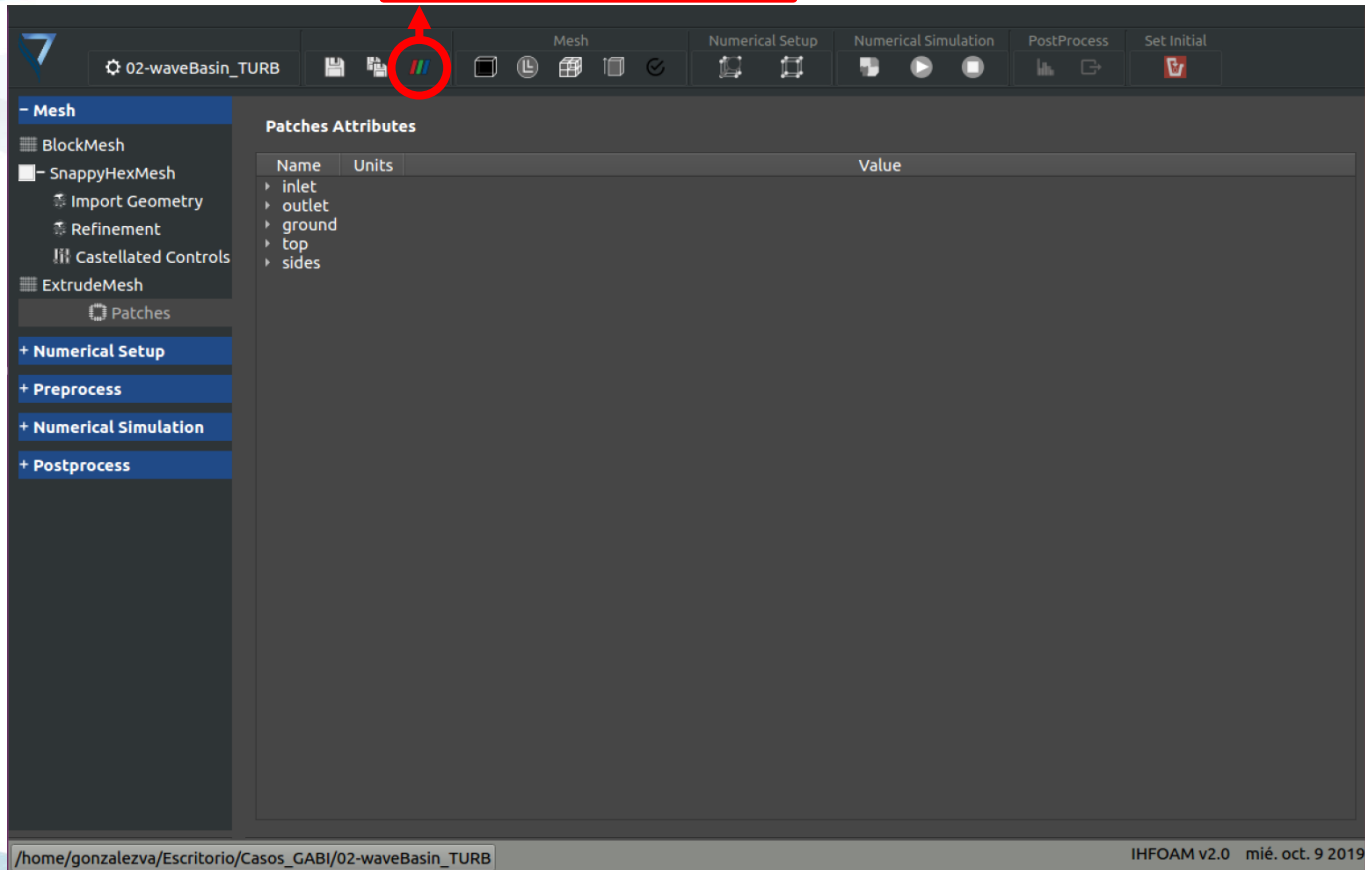
/home/gonzalezva/Escritorio/Casos_GABI/03-Breakwater

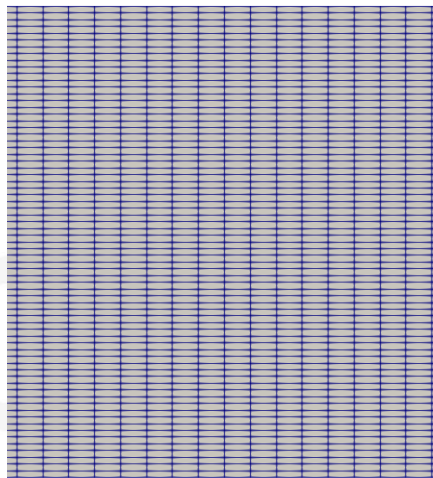
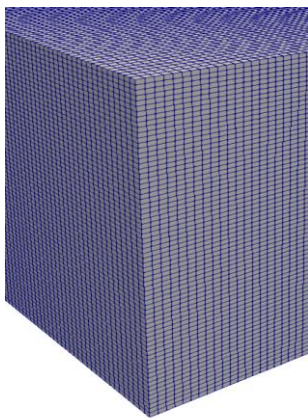
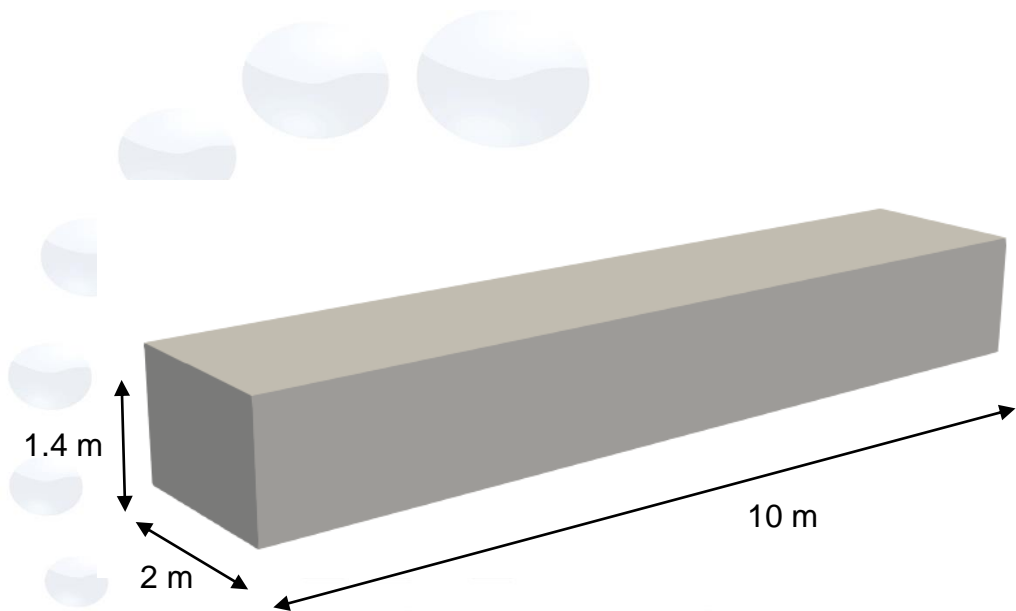
IHFOAM v2.0 mié. oct. 9 2019

checkMesh button



Paraview button





INSTITUTO DE HIDRÁULICA AMBIENTAL
UNIVERSIDAD DE CANTABRIA

The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view with the following items: + Mesh, - Numerical Setup (expanded), Gravity, Wave Properties, ☒ Turbulence, ☐ Porous Medium, ☐ Mangroves, Initial Fields, fvSchemes, fvSolution, + Preprocess, + Numerical Simulation, and + Postprocess. The 'Fluids Properties' item under 'Numerical Setup' is circled in red. A red arrow points from this circle to a red-bordered box containing the text 'Define fluids properties'. The main panel displays the 'FluidsProp Attributes' table.

Name	Units	Value
▼ water		
nu	m2/s	1e-06
rho	Kg/m3	1000
▼ air		
nu	m2/s	1.48e-05
rho	Kg/m3	1

At the bottom of the window, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB and the version is IHFOAM v2.0, dated mié. oct. 9 2019.

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

Name	Units	Value
gx	m/s2	0
gy	m/s2	0
gz	m/s2	-9.81

The bottom status bar shows the file path: /home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB and the version/date: IHFOAM v2.0 mié. oct. 9 2019.

ia
ENTAL

WaveProp Attributes

Name	Units	Value
WaveTheoryRange		
patch		inlet
waveModel		cnoidal
wavePeriod	s	2
waveHeight	m	0.1
waveAngle	degree	45
nPaddle		5
activeAbsorption		<input checked="" type="checkbox"/>
rampTime	s	2
restart		<input type="checkbox"/>
waterDepth		<input type="checkbox"/>
patch		outlet
waveModel		shallowWaterAbsorption
nPaddle		5

Define wave properties (generation and absorption)

ia
ENTAL

Define turbulence model and boundary conditions (1/2)

Name	Units	Value
simulationType		RAS
RASModel		kEpsilon
internalField		
k	m2/s2	0.00135
epsilon	m2/s3	1.116e-05
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

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 jue. oct. 10 2019

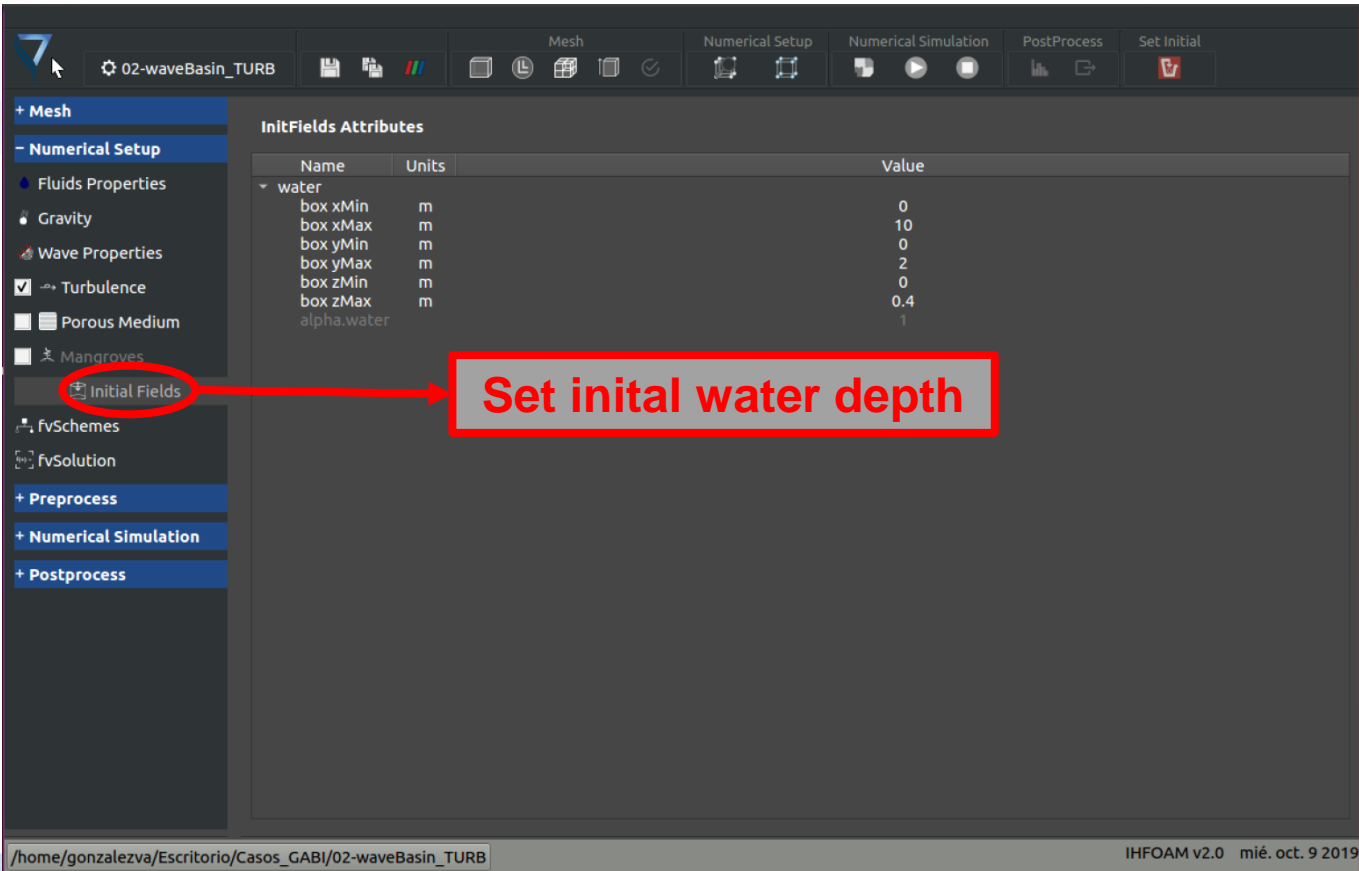
Turbulence Attributes

Name	Units	Value
simulationType		RAS
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
sides		
k	m/s ²	slip
epsilon	m/s ³	slip
omega	1/s	empty
nut	m/s	calculated
bottom		

Define turbulence model and boundary conditions (2/2)

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 jue. oct. 10 2019

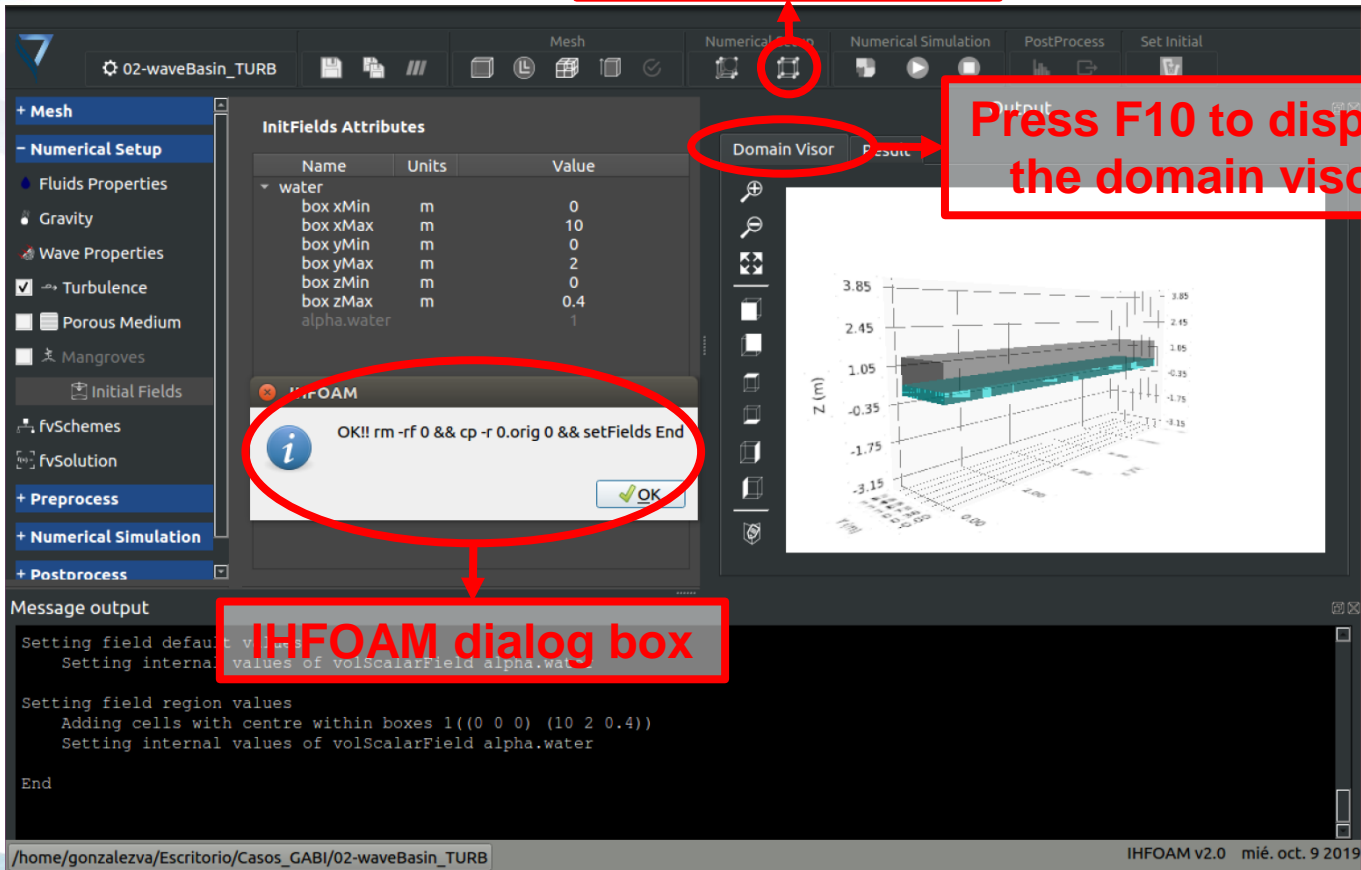


The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view with the following items: + Mesh, - Numerical Setup (expanded), Fluids Properties, Gravity, Wave Properties, ☒ Turbulence, Porous Medium, Mangroves, **Initial Fields** (circled in red), fvSchemes, fvSolution, + Preprocess, + Numerical Simulation, and + Postprocess. The main panel displays the 'InitFields Attributes' table.

Name	Units	Value
water		
box xMin	m	0
box xMax	m	10
box yMin	m	0
box yMax	m	2
box zMin	m	0
box zMax	m	0.4
alpha.water		1

A red-bordered box with the text "Set initial water depth" is positioned to the right of the 'Initial Fields' option in the sidebar, with a red arrow pointing from the option to the box.

At the bottom of the interface, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB and the version is IHFOAM v2.0, dated mié. oct. 9 2019.



Paraview button

InitFields Attributes

Name	Units	Value
water		
box xMin	m	0
box xMax	m	10
box yMin	m	0
box yMax	m	2
box zMin	m	0
box zMax	m	0.4
alpha.water		1

Output

Domain Visor Result

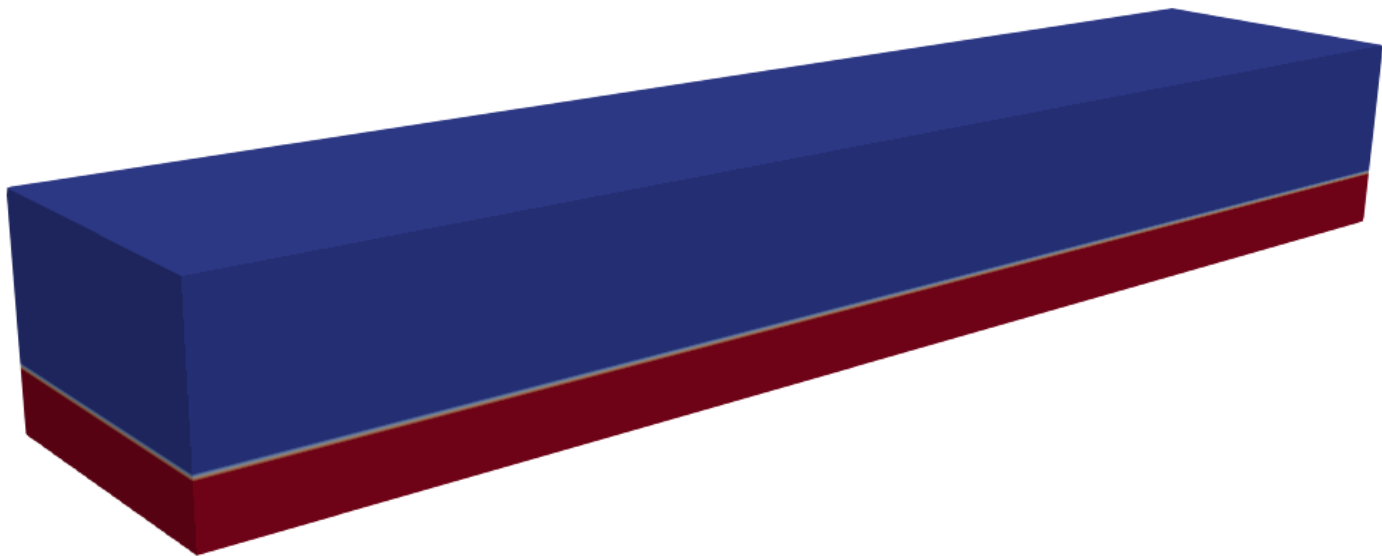
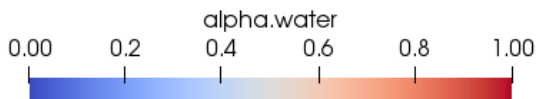
3.85 2.45 1.05 -0.35 -1.75 -3.15

z (m)

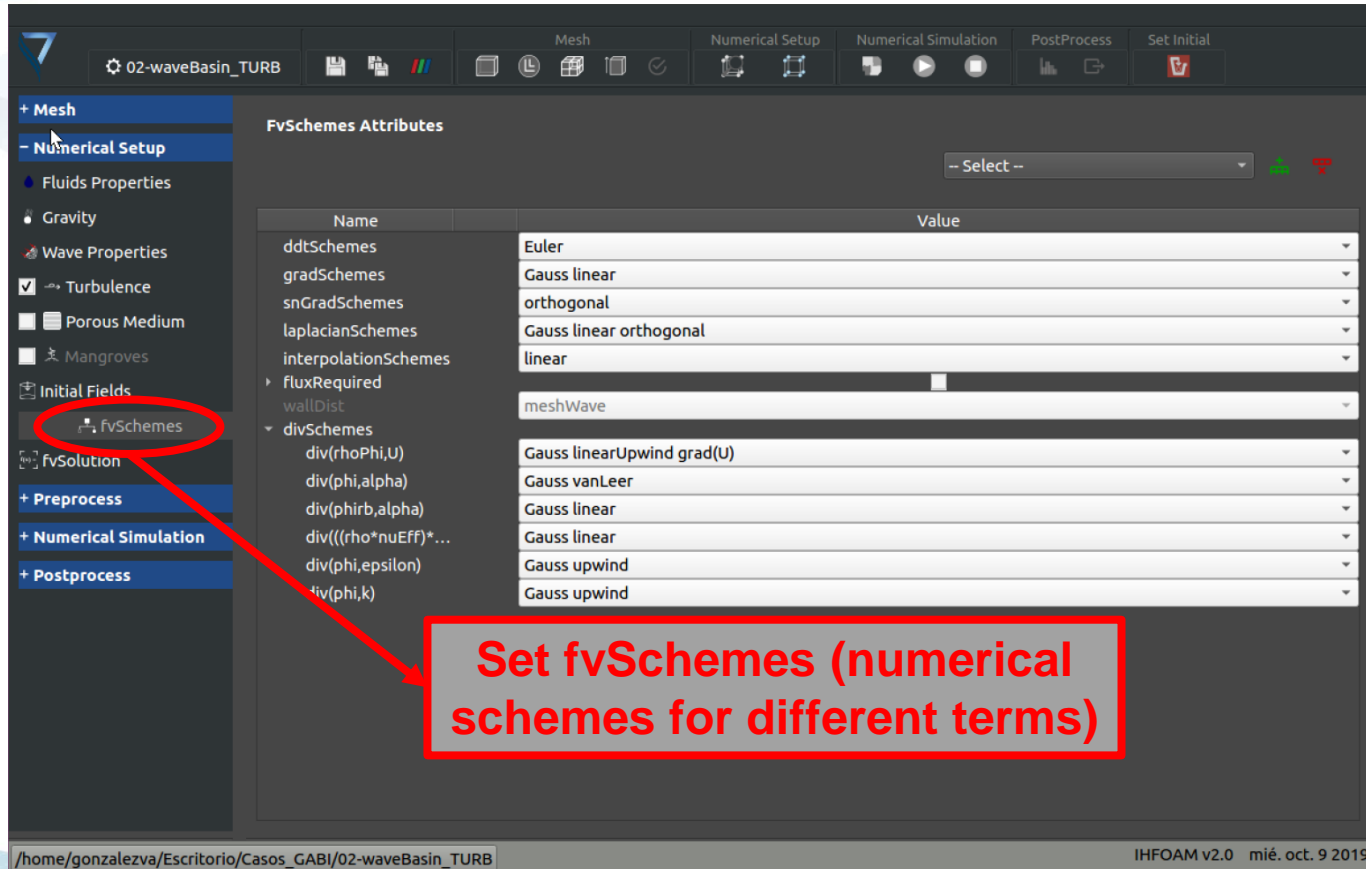
3.85 2.45 1.05 -0.35 -1.75 -3.15

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 mié. oct. 9 2019



a
ITAL



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, Mangroves, Initial Fields, FvSchemes (highlighted with a red circle), fvSolution, + Preprocess, + Numerical Simulation, and + Postprocess. The main panel displays the 'FvSchemes Attributes' for the case '02-waveBasin_TURB'. It features a table with columns 'Name' and 'Value'. A red-bordered box with the text 'Set fvSchemes (numerical schemes for different terms)' is overlaid on the bottom right of the interface, with a red arrow pointing from the 'FvSchemes' item in the sidebar to it.

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,epsilon)	Gauss upwind
div(phi,k)	Gauss upwind

Set fvSchemes (numerical schemes for different terms)

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 mié. oct. 9 2019

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

Name	Value
PIMPLE	
momentumPredictor	no
nCorrectors	2
nNonOrthogonalCorr...	0
alpha.water.*	
alphaOuterCorrectors	
cAlpha	1
nAlphaCorr	1
nAlphaSubCycles	3
p.corr.*	
solver	PCG
preconditioner	DIC
tolerance	1e-06
relTol	0
p_rgh	
solver	PCG
preconditioner	DIC
tolerance	1e-06
relTol	0.1
p_rghFinal	
solver	PCG
preconditioner	DIC
tolerance	1e-07
relTol	0
"(U k epsilon omega)"	
"(U k epsilon omega)Final"	

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

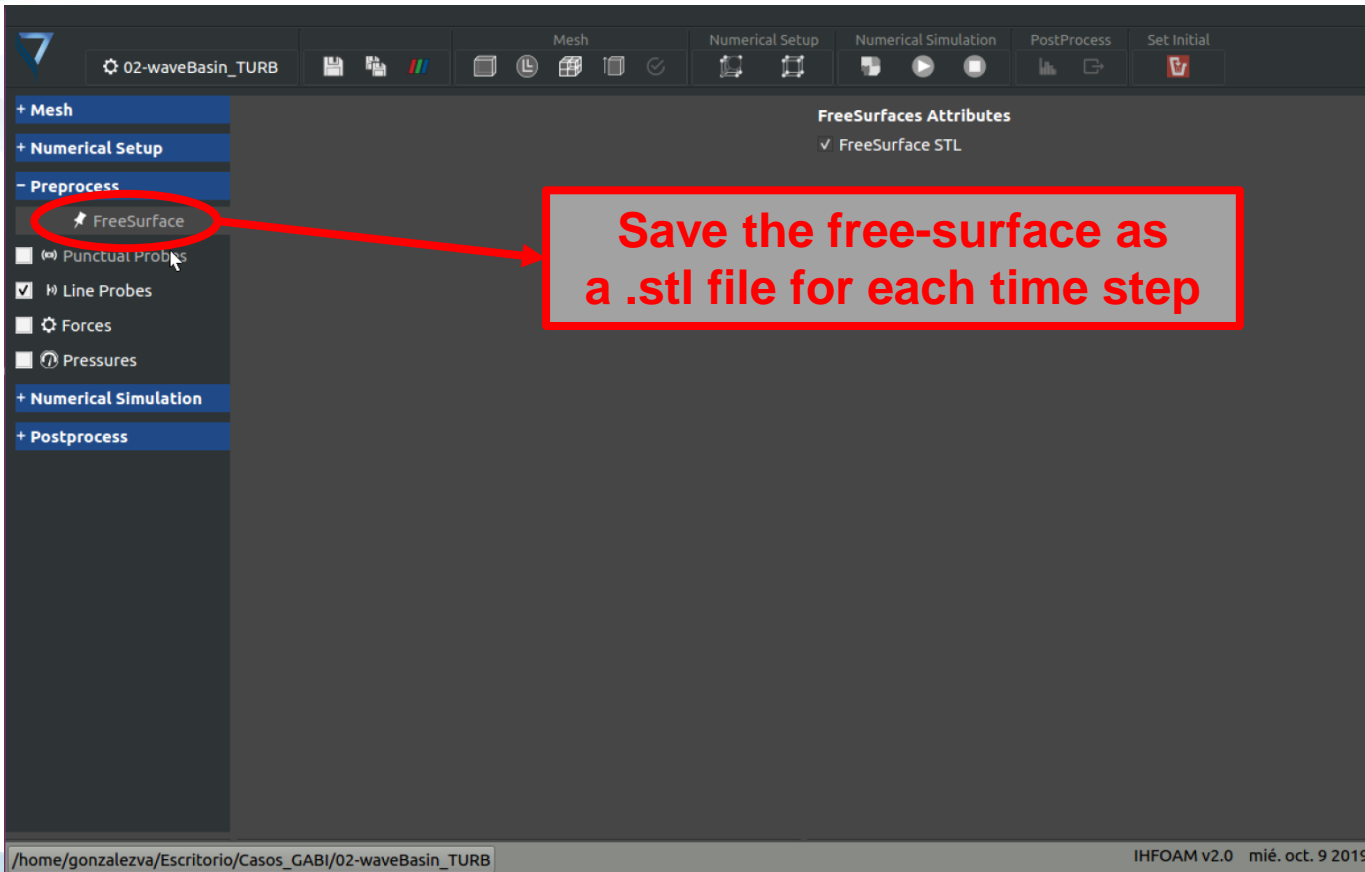
IHFOAM v2.0 jue. oct. 10 2019

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

Name	Value
PIMPLE	
"alpha.water.*"	
"pcorr.*"	
p_rgh	
p_rghFinal	
"(U k epsilon omega)"	
solver	PBiCG
preconditioner	DILU
tolerance	1e-06
relTol	0.1
"(U k epsilon omega)Final"	
solver	PBiCG
preconditioner	DILU
tolerance	1e-06
relTol	0

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 jue. oct. 10 2015



02-waveBasin_TURB

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	1	1	0	0.7	1001
2	line2	2	1	0	0.7	1001
3	line3	3	1	0	0.7	1001
4	line4	5	1	0	0.7	1001
5	line5	7.5	1	0	0.7	1001
6	line6	9	1	0	0.7	1001

Save Fields

☒ VOF

☒ Velocity U (m/s)

☐ Pressure P (Pa)

☐ Turbulent Kinematic Energy k (m2/s2)

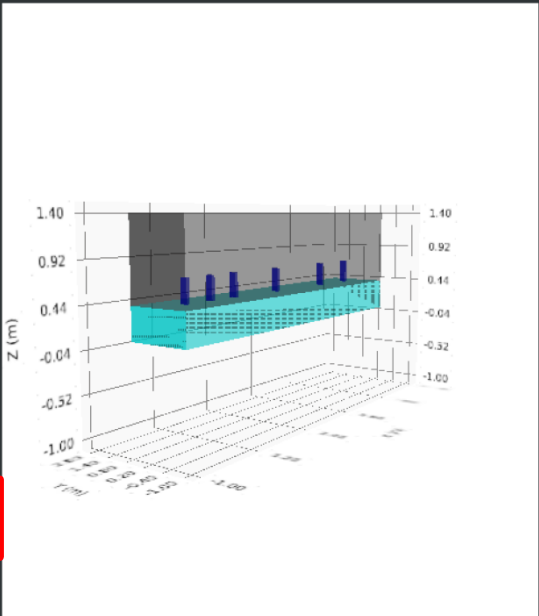
☐ Dissipation Rate EPSILON (m2/s3)

☐ Specific Dissipation Rate W (1/s)

☐ Eddy Viscosity Mu_t (m2/s)

Output

Domain Visor Result



Set free-surface gauges

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 mié. oct. 9 2019

The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains 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 displays the 'Parameters Attributes' table.

Name	Units	Value
startFrom		latestTime
endTime	s	10
deltaT	s	0.01
writeControl		adjustableRunTime
writeInterval	s	0.33
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/02-waveBasin_TURB and the version is IHFOAM v2.0, dated mié. oct. 9 2019.

ia
ENTAL

decomposePar button

Run/Execute button

cleanCase button

Simulation

Run OpenFOAM case

The screenshot shows the OpenFOAM GUI interface. The left sidebar contains a tree view with the following items: + Mesh, + Numerical Setup, + Preprocess, - Numerical Simulation, Parameters, Simulation (circled in red), Performance, Residuals, and + Postprocess. The main panel displays 'Simulation Attributes' with a table of simulation parameters. The top toolbar contains icons for Mesh, Numerical Setup, Numerical Simulation (with 'decomposePar' and 'Run/Execute' buttons circled in red), PostProcess, and Set Initial (with a 'cleanCase' button circled in red). Red arrows point from the external labels to these specific buttons and the 'Simulation' menu item.

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

/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB

IHFOAM v2.0 mié. oct. 9 2019

decomposePar button

The screenshot shows the OpenFOAM 2.2.0 GUI. The 'Numerical Simulation' tab is active, and the 'decomposePar' button is highlighted with a red circle and an arrow pointing to it from the text 'decomposePar button'. Below the button, a message box displays the command 'OK!! rm -rf processor* && decomposePar End'. A red arrow points from this message box to a red box containing the text 'Case decomposed correctly'. The 'Simulation Attributes' panel shows the 'Numerical Solver' set to 'interFoam', 'Execution Type' set to 'Parallel', and 'method' set to 'hierarchical'. The 'Output' panel shows a 3D visualization of the wave basin. The 'Message output' panel at the bottom shows the following text:

```
Time = 0

Processor 0: field transfer
Processor 1: field transfer
Processor 2: field transfer
Processor 3: field transfer

End
```

The status bar at the bottom indicates the path '/home/gonzalezva/Escritorio/Casos_GABI/02-waveBasin_TURB' and the version 'IHFOAM v2.0' dated 'mié. oct. 9 2019'.

Case decomposed correctly

Run/Execute button

The screenshot displays the IHFOAM v2.0 software interface. The 'Numerical Simulation' tab is selected, and the 'Run/Execute' button is highlighted. A message box indicates successful execution: 'OK!! mpirun -np 4 interFoam -parallel End'. The message output window shows the end of the simulation: 'End'.

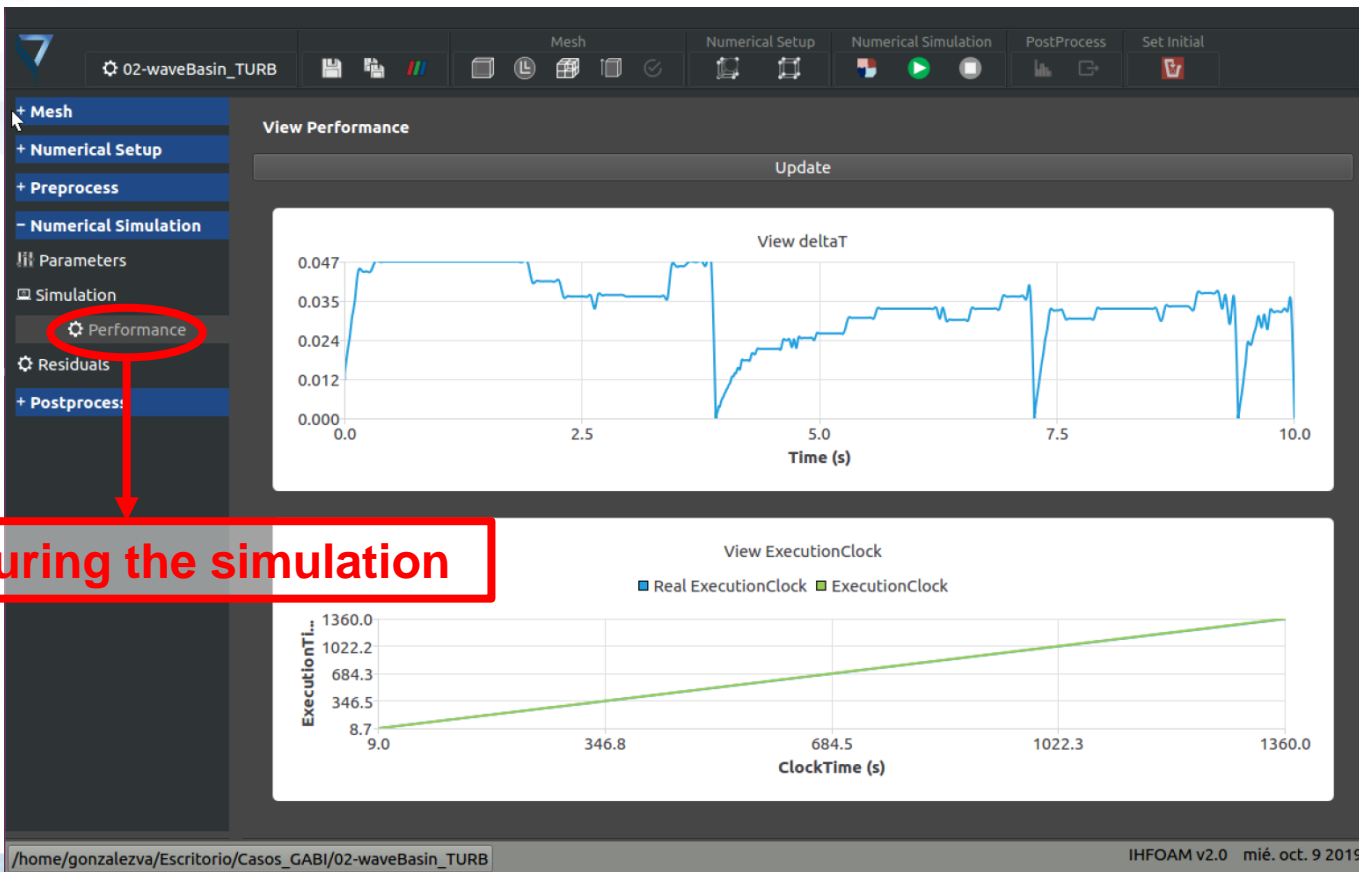
Simulation Attributes

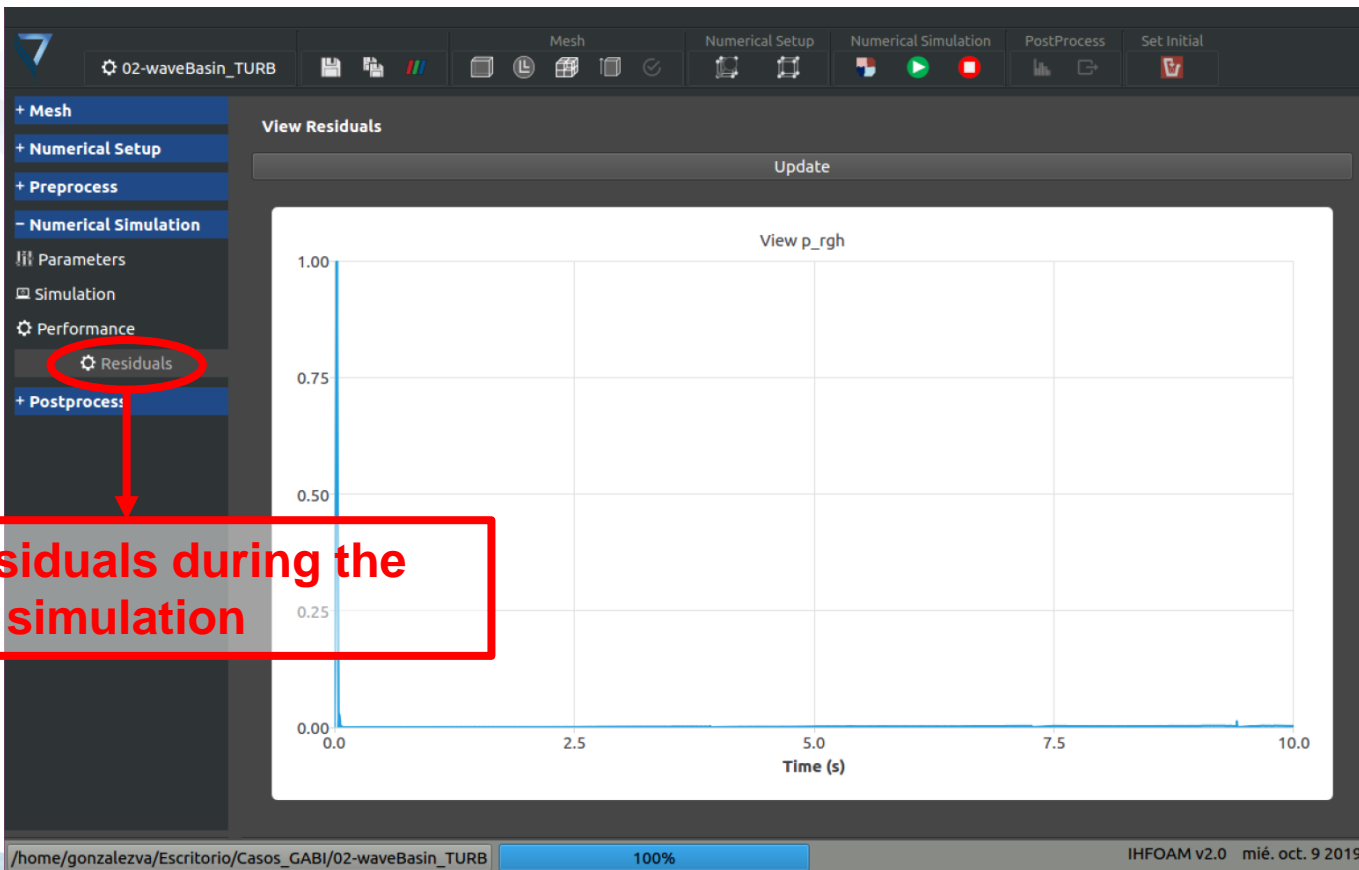
Name	Units	Value
Numerical Solver		interFoam
Execution Type		Parallel

Message output

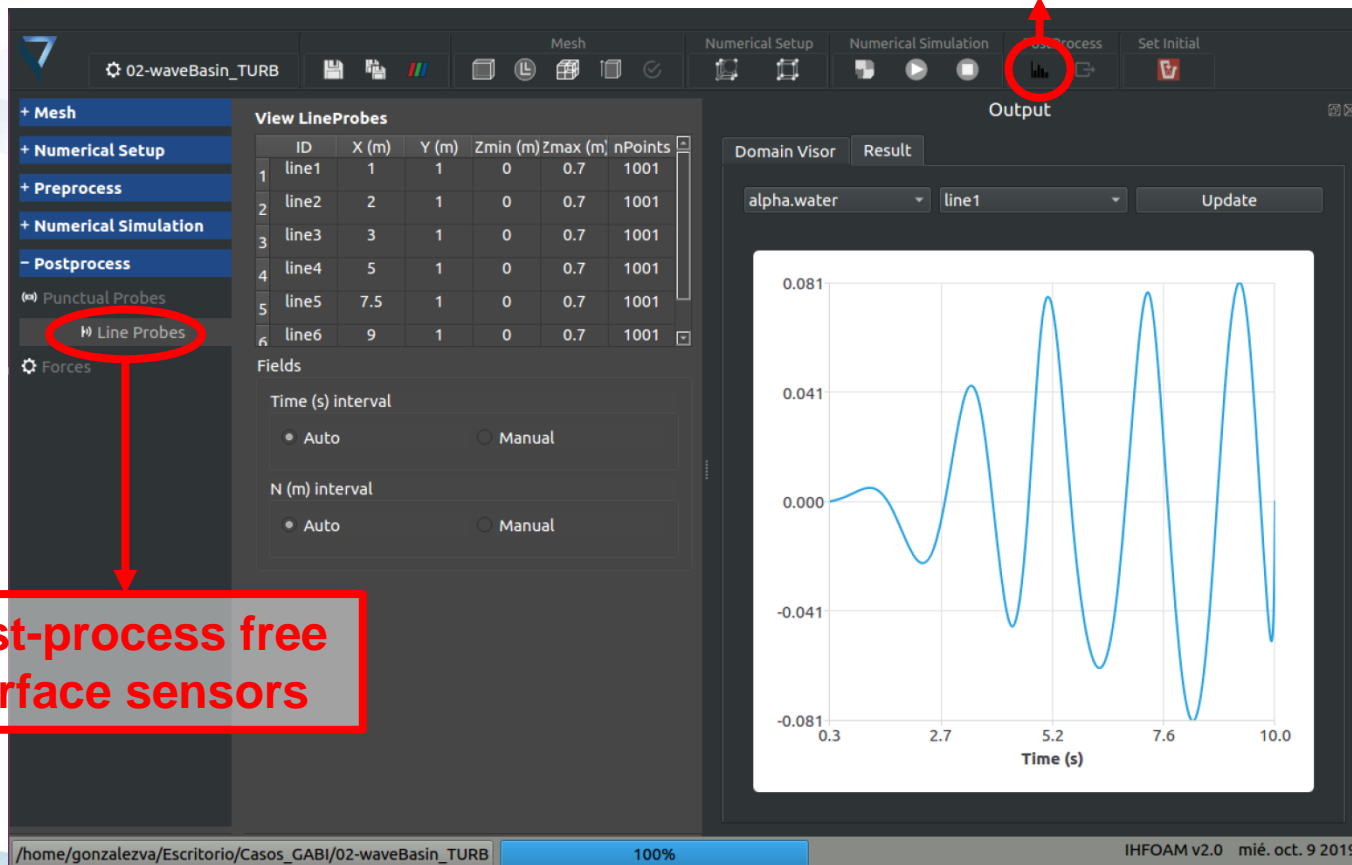
```
DICPCG: Solving for p_rgh, Initial Residual = 0.000175773, Final Residual = 9.19187e-08, No Iterations 44  
time step continuity errors : sum local = 4.39954e-09, global = 3.59172e-07, cumulative = 0.000127926  
DILUPBiCG: Solving for epsilon, Initial residual = 0.0306964, Final residual = 7.2401e-07, No Iterations 9  
DILUPBiCG: Solving for k, Initial residual = 0.0200523, Final residual = 8.22797e-07, No Iterations 10  
ExecutionTime = 1355.69 s, ClockTime = 1360 s  
End  
Finalising parallel run
```

Case correctly executed!!



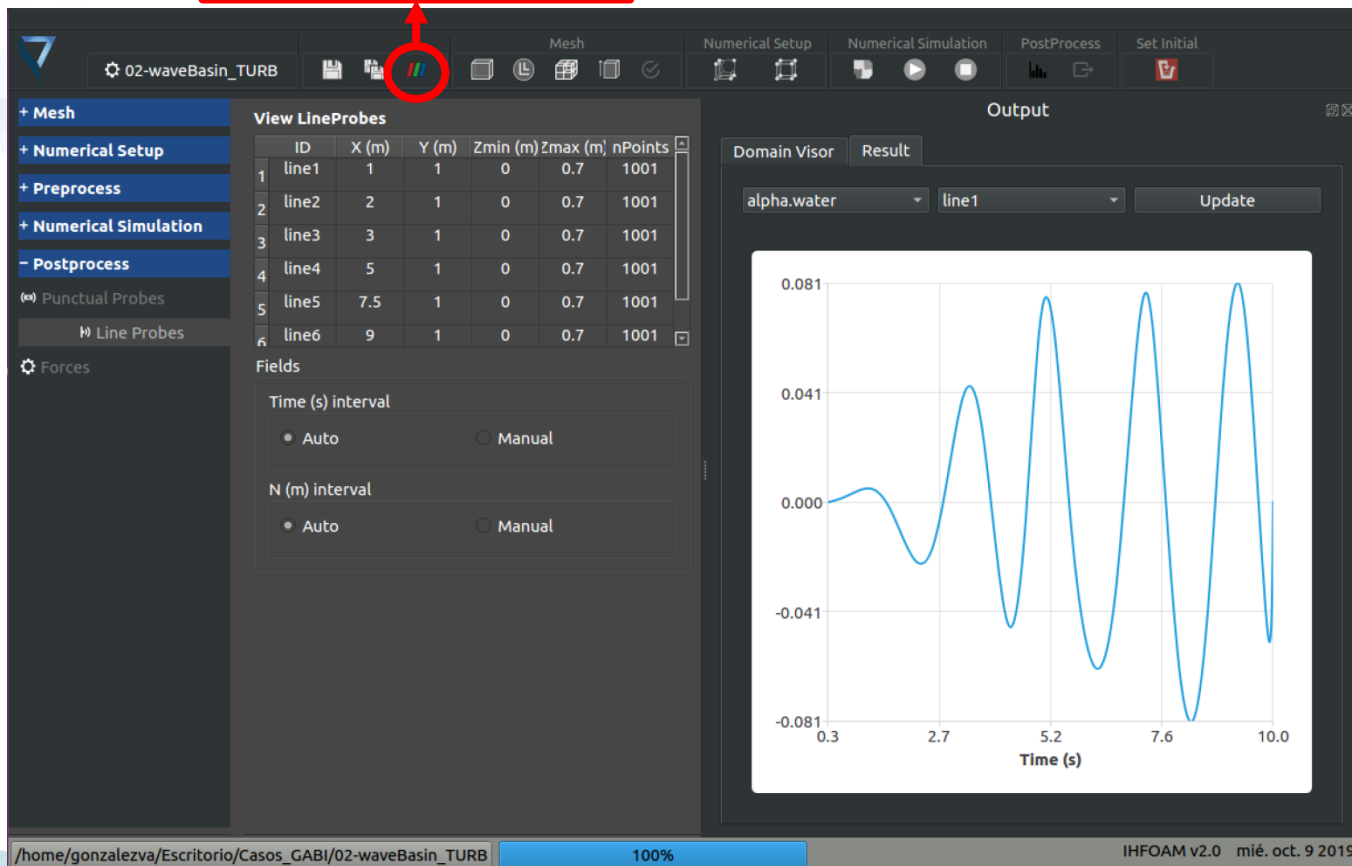


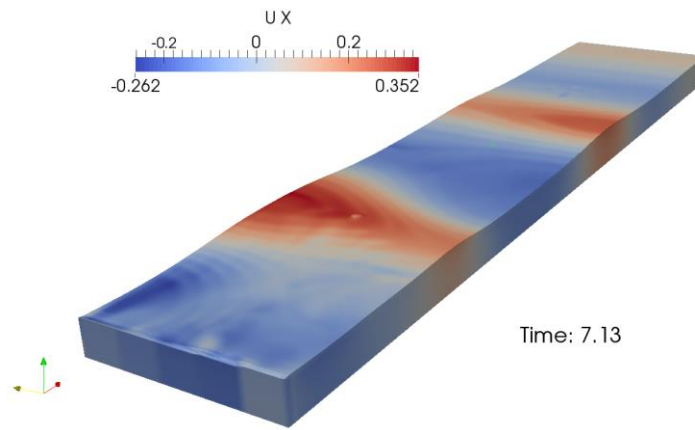
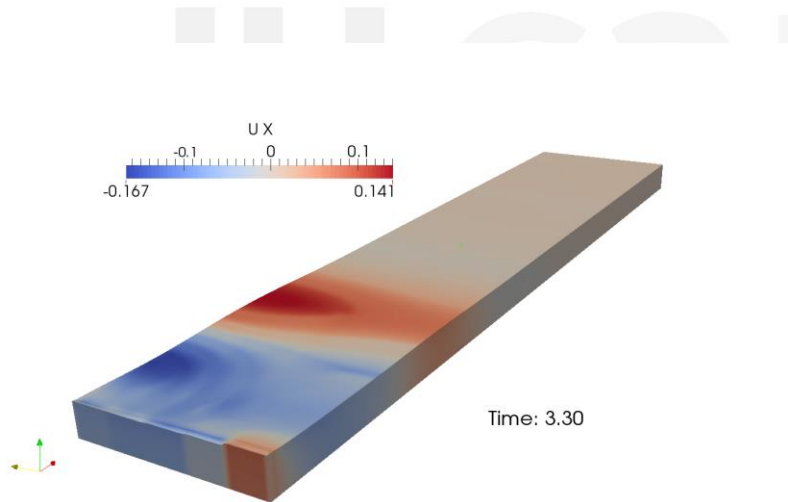
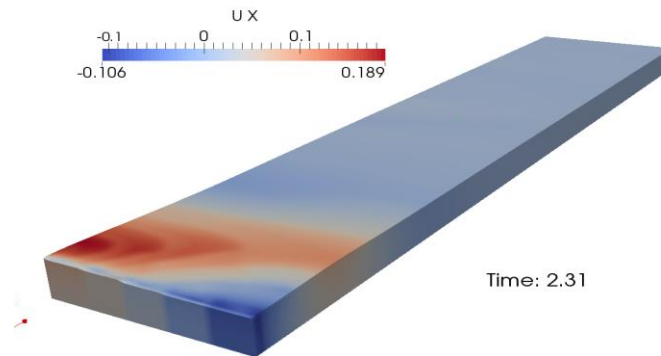
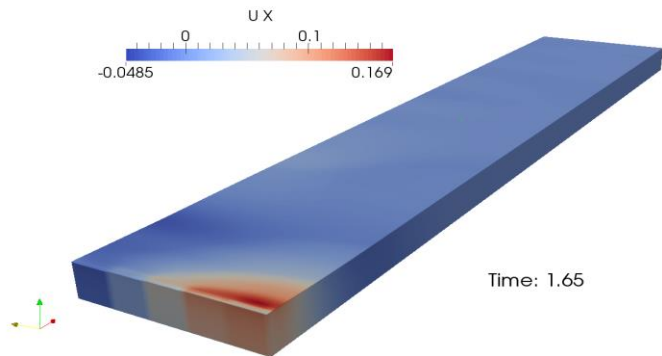
plot free surface sensors



Post-process free surface sensors

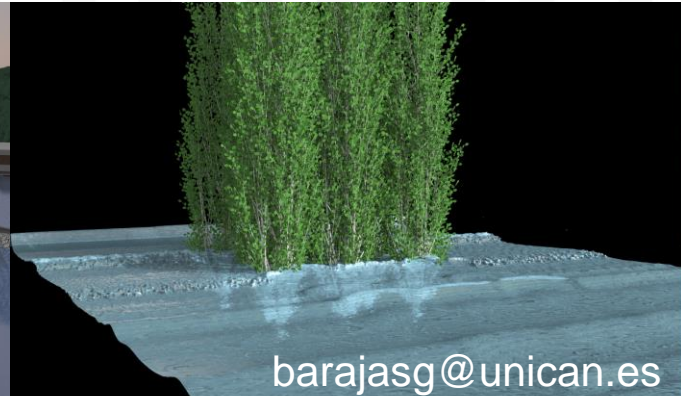
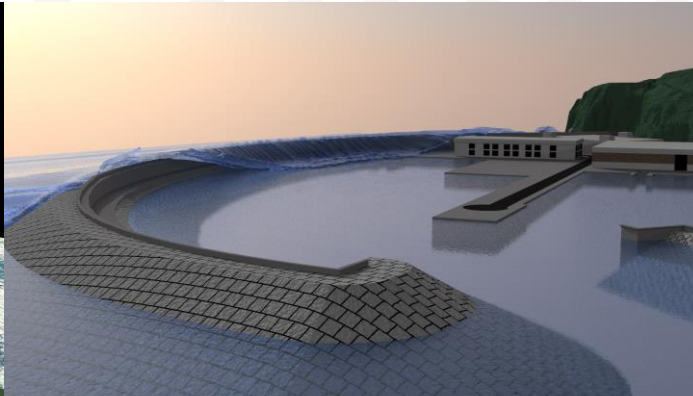
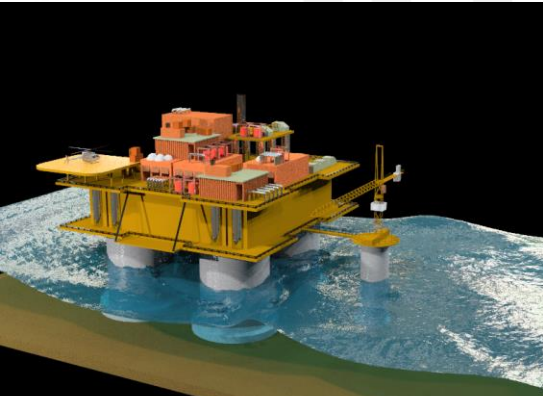
Paraview button







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



barajasg@unican.es