

The background image shows a modern architectural complex at night. On the left, a curved building with a grid-like facade is brightly lit from within. In the center, a multi-story building with a colorful, modular facade (red, orange, yellow, green) is also brightly lit. To the right, another building with a similar grid-like facade is partially visible, its lights reflecting off the glass. The sky is dark, and streetlights are visible in the distance.

# IHCantabria

UNIVERSIDAD DE CANTABRIA

---

R+D+i for a Sustainable Development

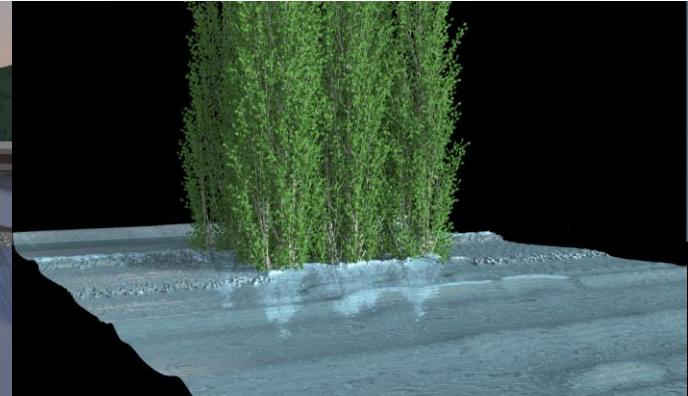
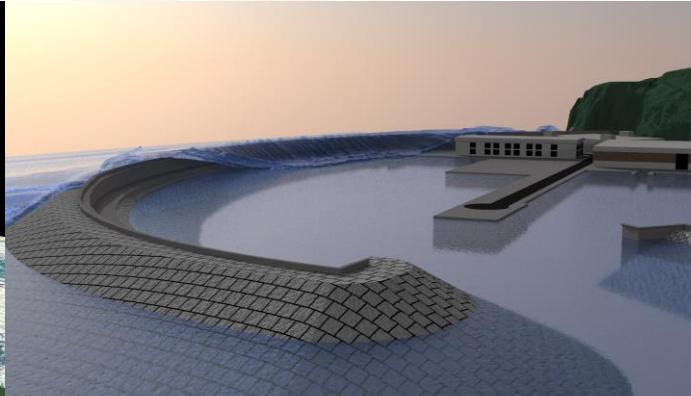
(IHFOAM GUI)

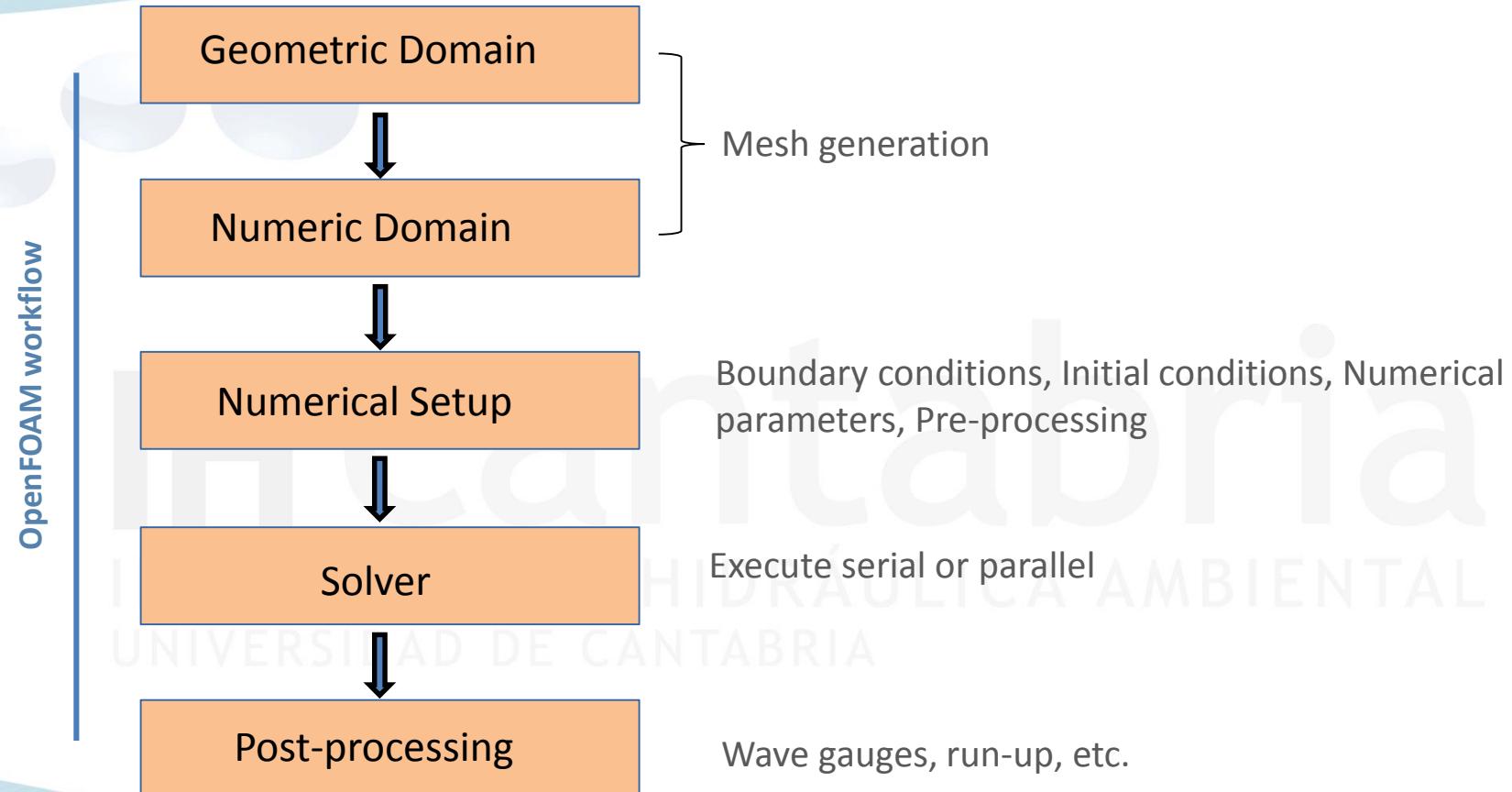


# IHFOAM applied to Coastal Engineering

Regular waves interaction with a rubble-mound breakwater

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





OpenFOAM case

0

- alpha.water
  - p\_rgh
  - U
  - porosityIndex
- k
  - epsilon
  - nut

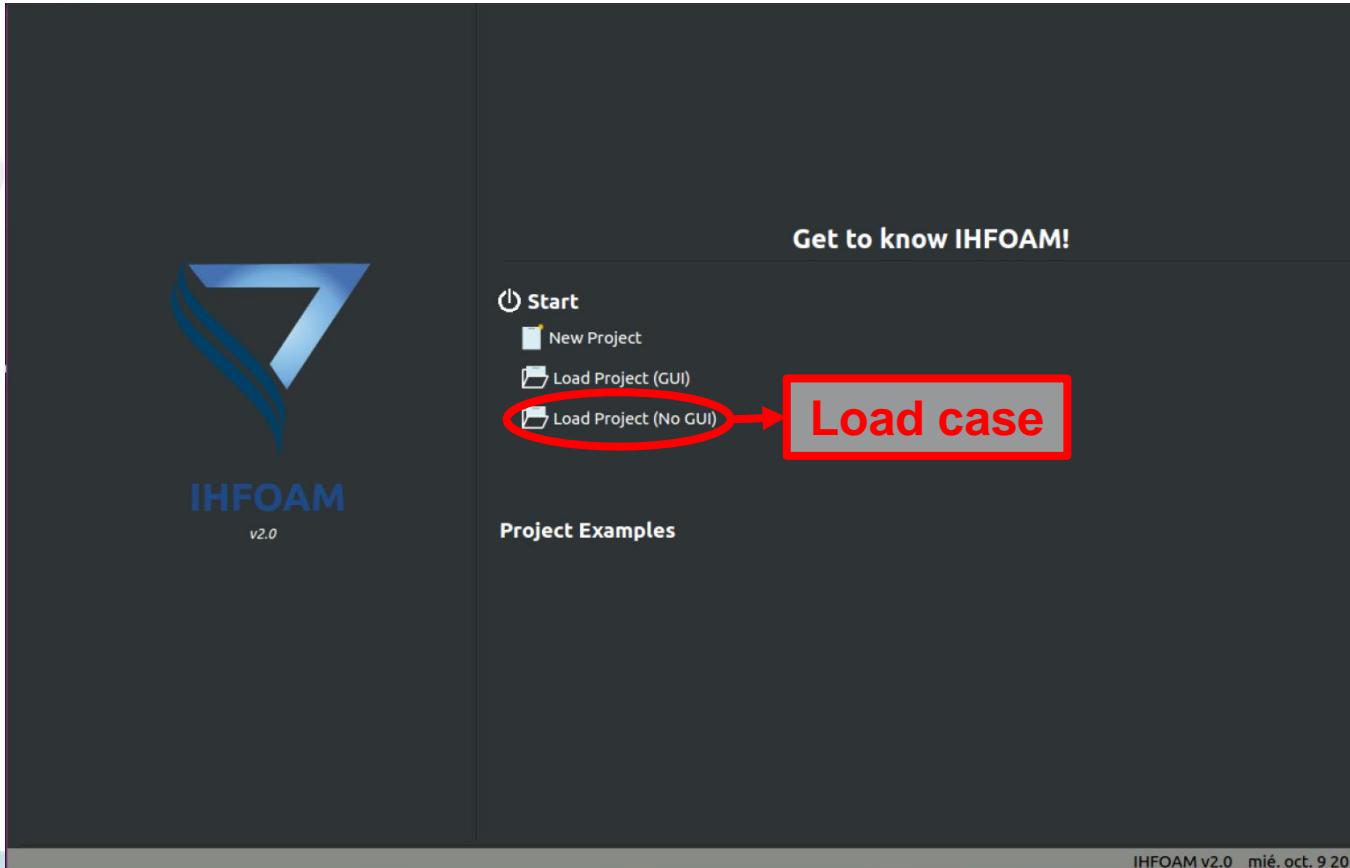
constant

- g
  - transportProperties
  - turbulenceProperties
  - porosityDict
- waveProperties

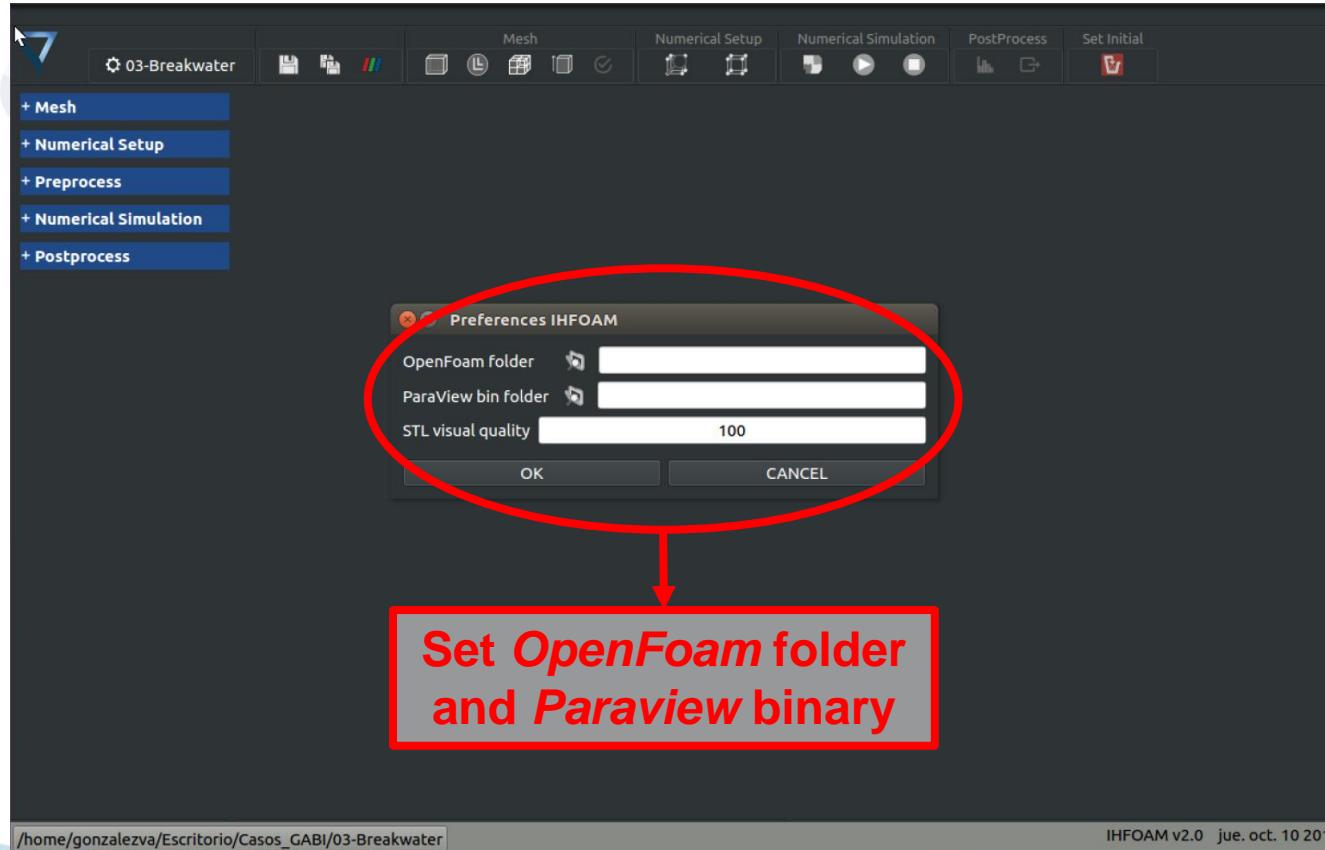
system

- blockMeshDict
  - setFieldsDict
  - snappyHexMeshDict
  - extrudeMeshDict
- fvSchemes
  - fvSolution
  - decomposeParDict
  - controlDict

IHFOAM GUI



IHFOAM v2.0 mié. oct. 9 2019

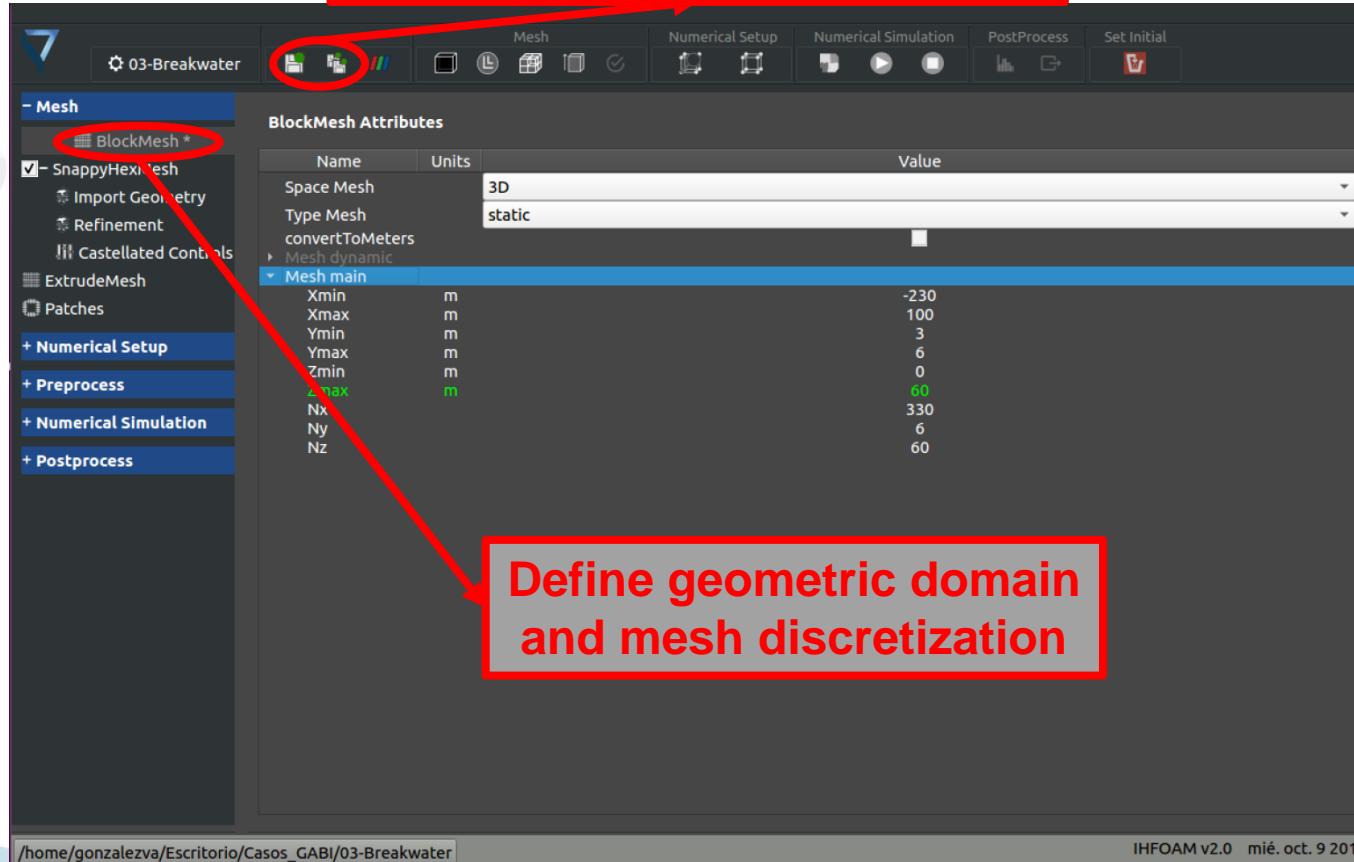


Set *OpenFoam* folder  
and *Paraview* binary



Save single pannel modified  
or save all pannels modified.

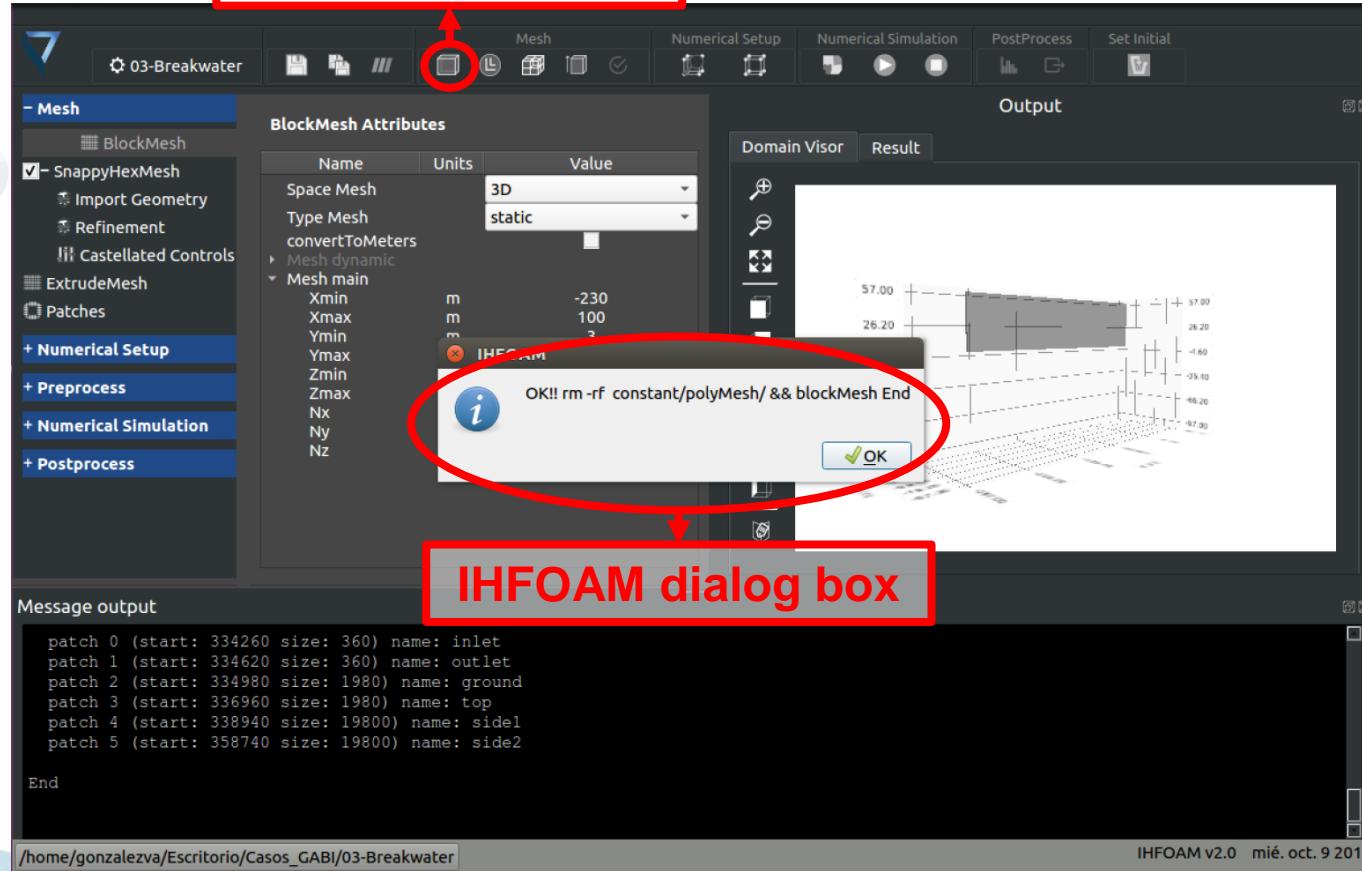
Regular waves interaction with a rubble-mound breakwater

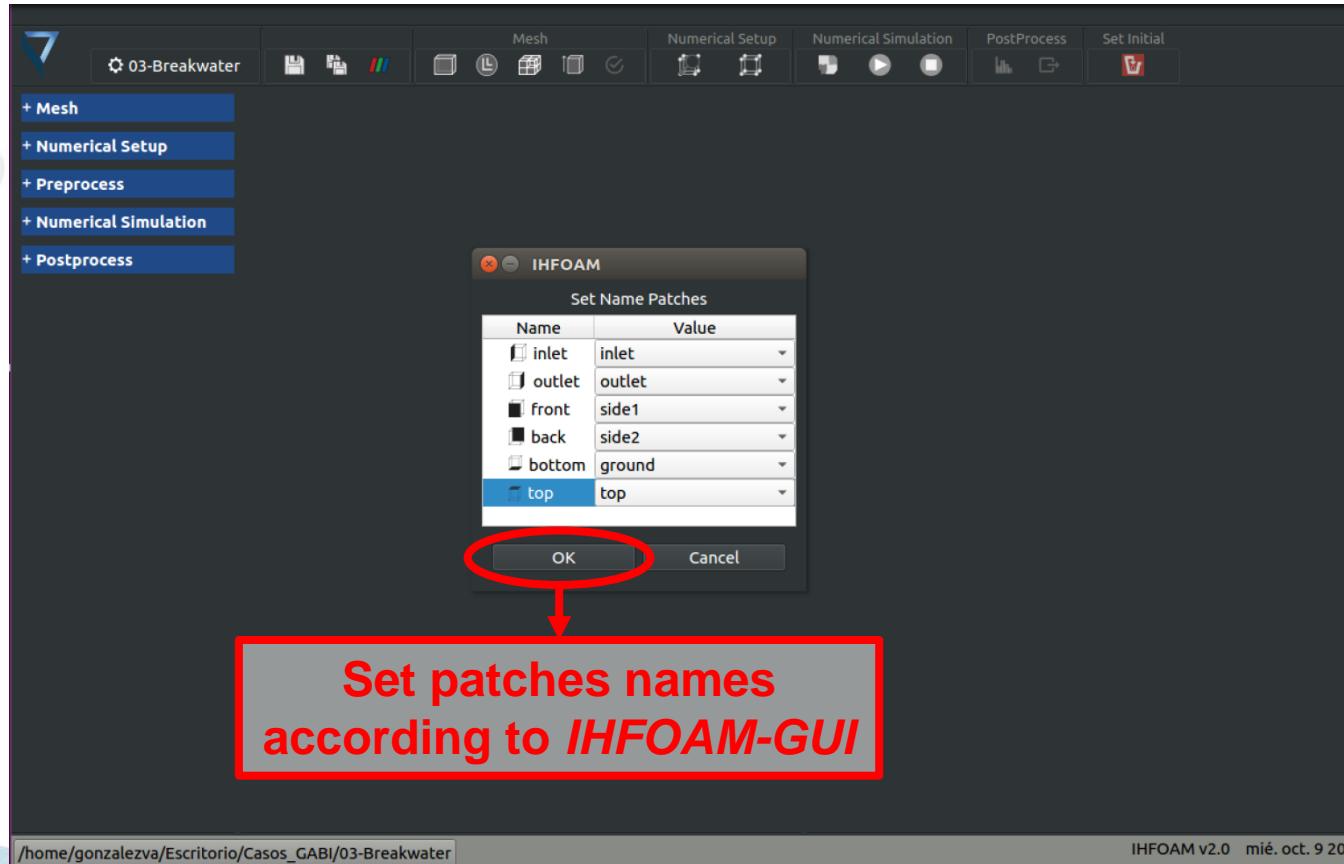


The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains navigation tabs: Mesh, Numerical Setup, Preprocess, Numerical Simulation, and Postprocess. The Mesh tab is active, showing a tree structure with 'BlockMesh' selected, which has 'SnappyHexMesh' as its child. Under SnappyHexMesh, there are options for Import Geometry, Refinement, and Castellated Controls. Other tabs include ExtrudeMesh, Patches, and a plus sign for more options. The main panel displays 'BlockMesh Attributes' with a table:

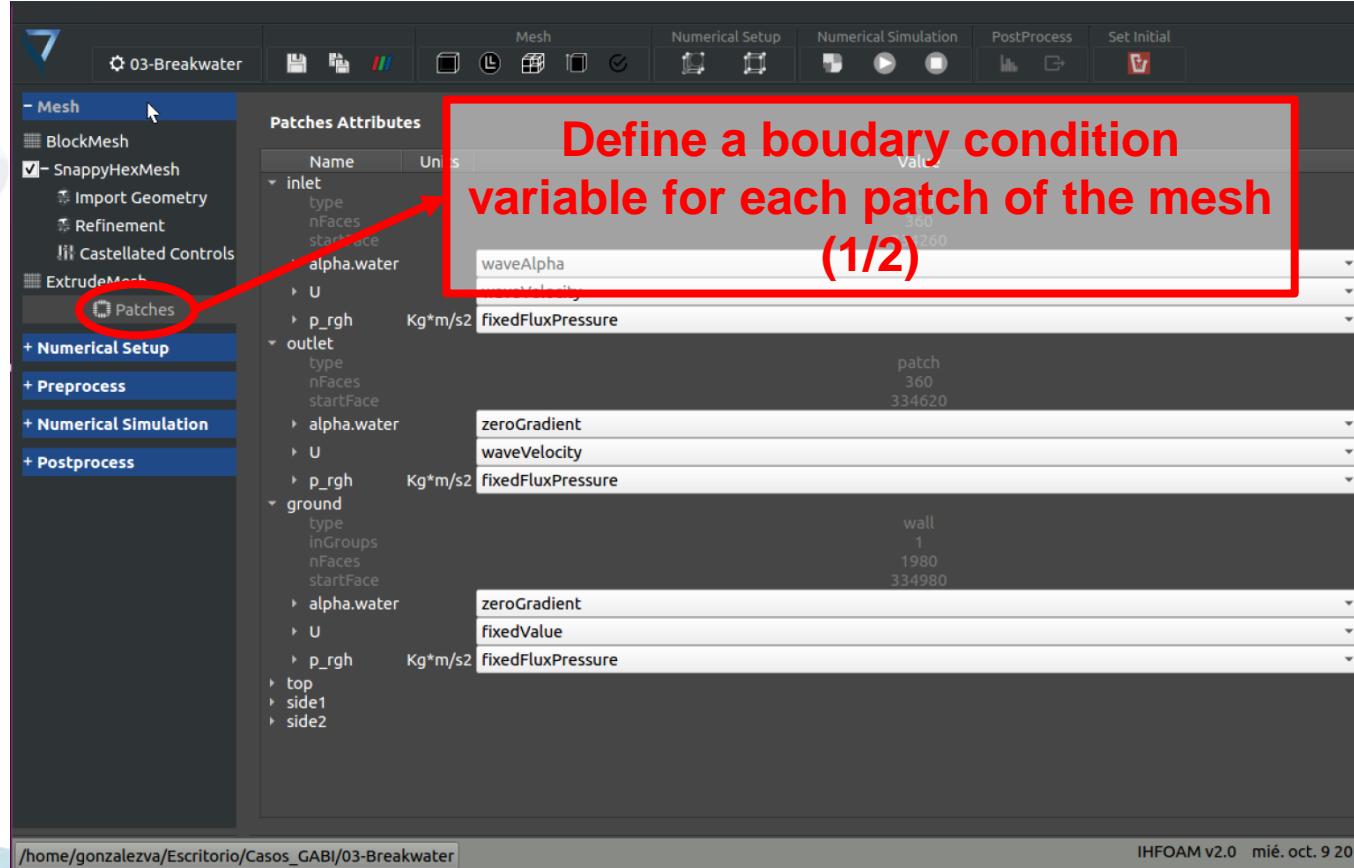
Name	Units	Value
Space Mesh	3D	
Type Mesh	static	
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	-230
Xmax	m	100
Ymin	m	3
Ymax	m	6
Zmin	m	0
Nx	m	60
Ny		330
Nz		60

A red box highlights the 'BlockMesh' node in the tree, and another red box highlights the 'Save' icon in the top toolbar. A red arrow points from the 'Save' icon to the text 'Save single pannel modified or save all pannels modified.' A second red arrow points from the 'Nx' value in the table to the text 'Define geometric domain and mesh discretization.'



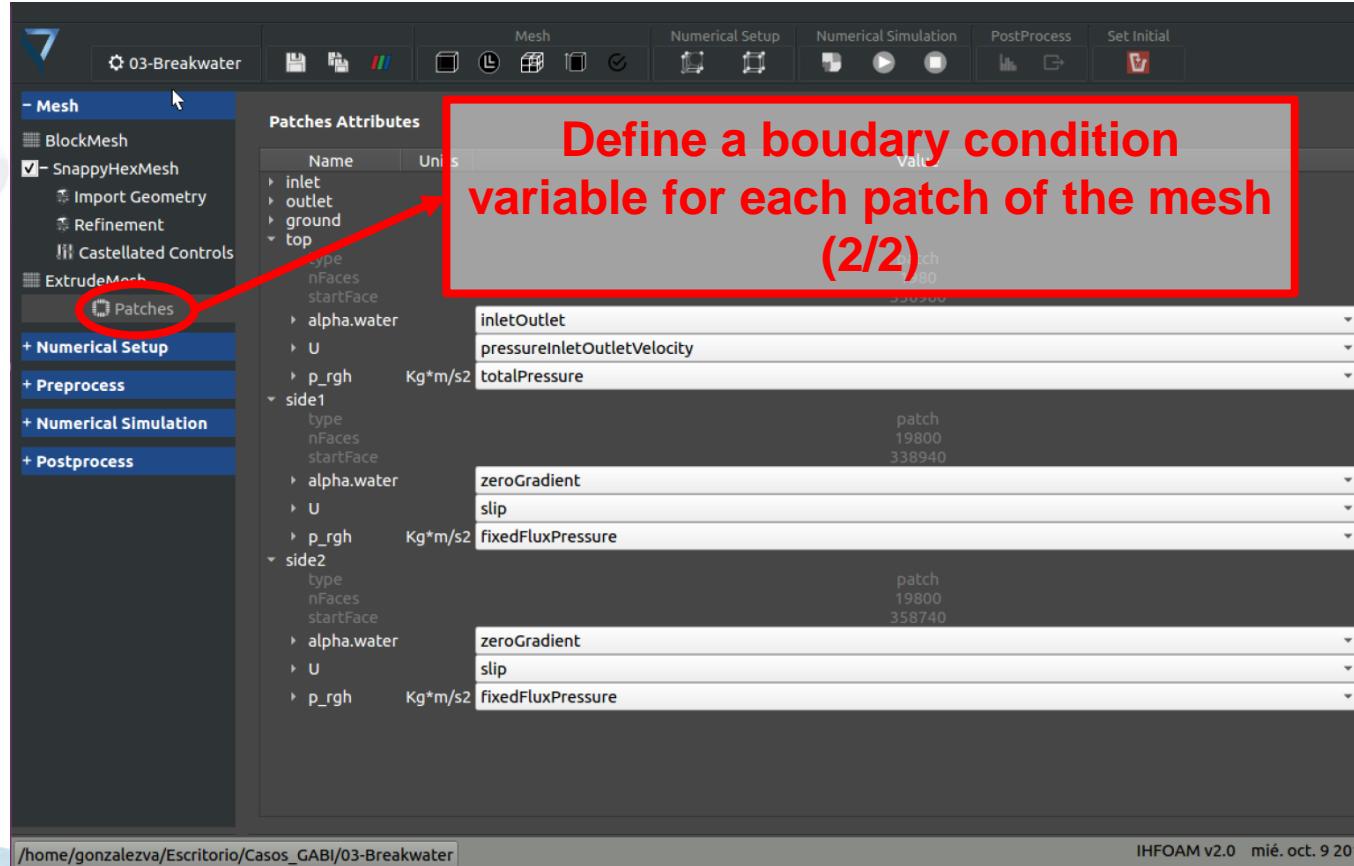


Define a boundary condition variable for each patch of the mesh (1/2)

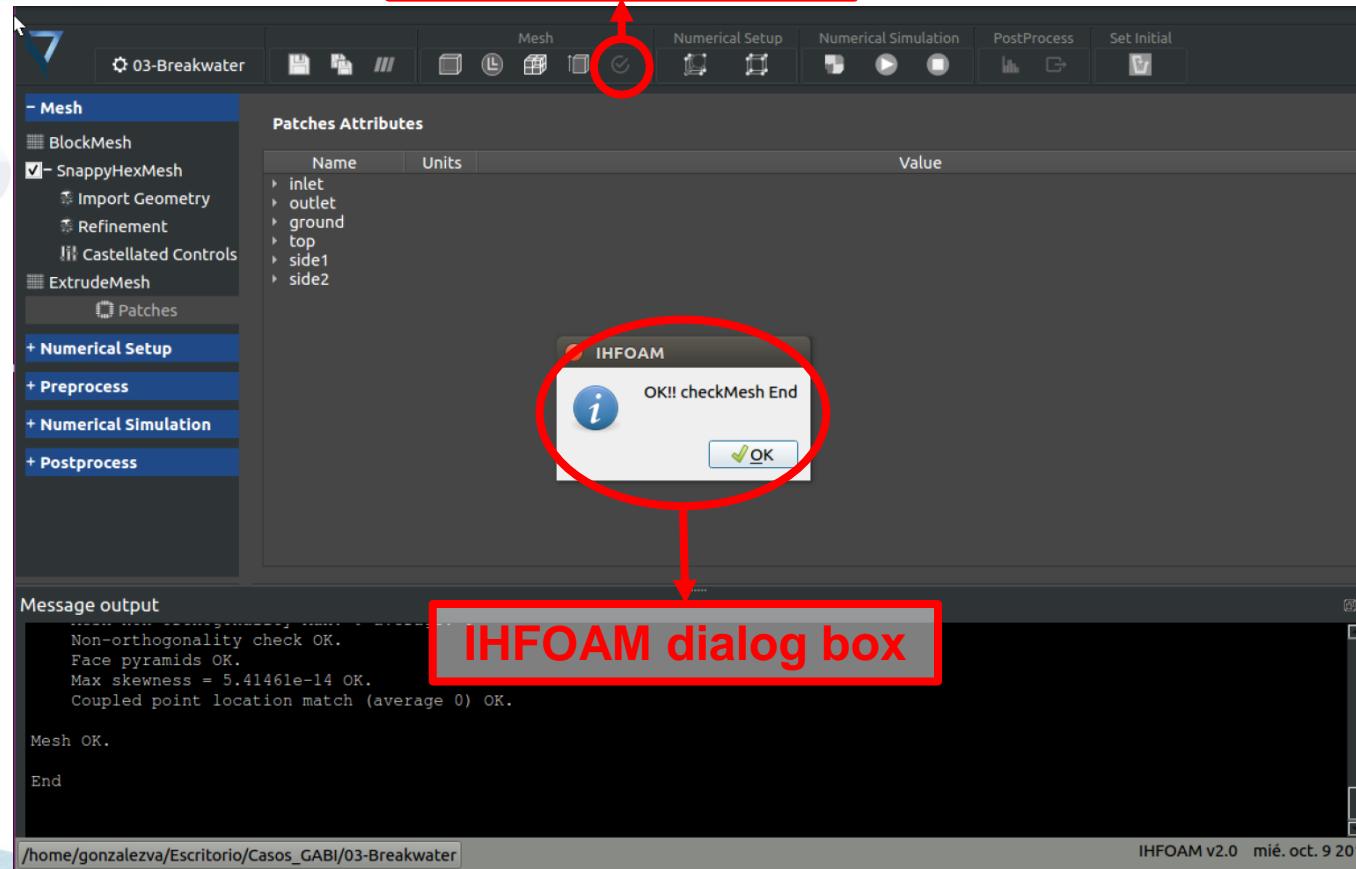


Name	Units	Value
inlet		waveAlpha
outlet	Kg*m/s <sup>2</sup>	fixedFluxPressure
ground		patch
top		wall
side1		
side2		

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

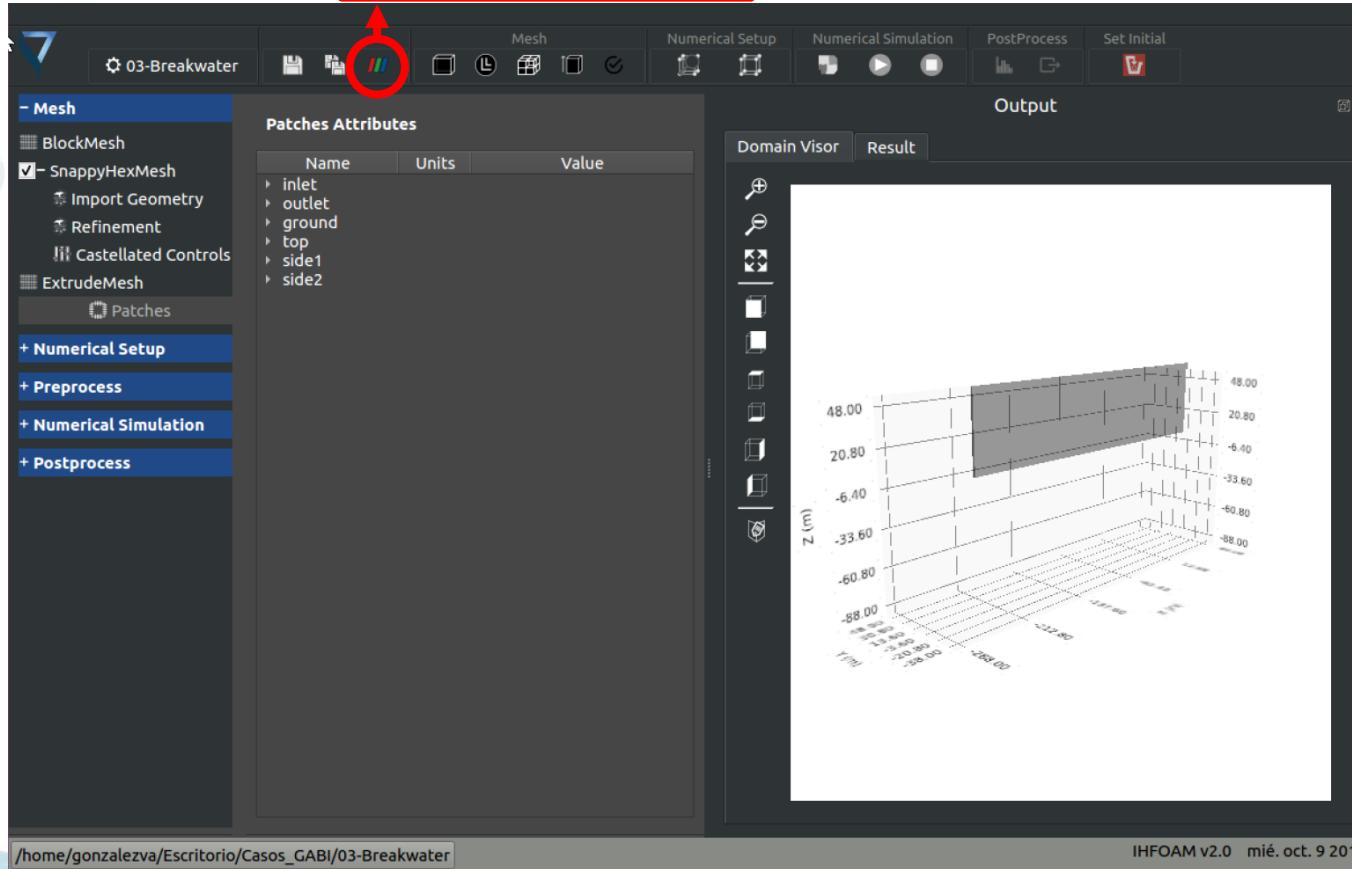


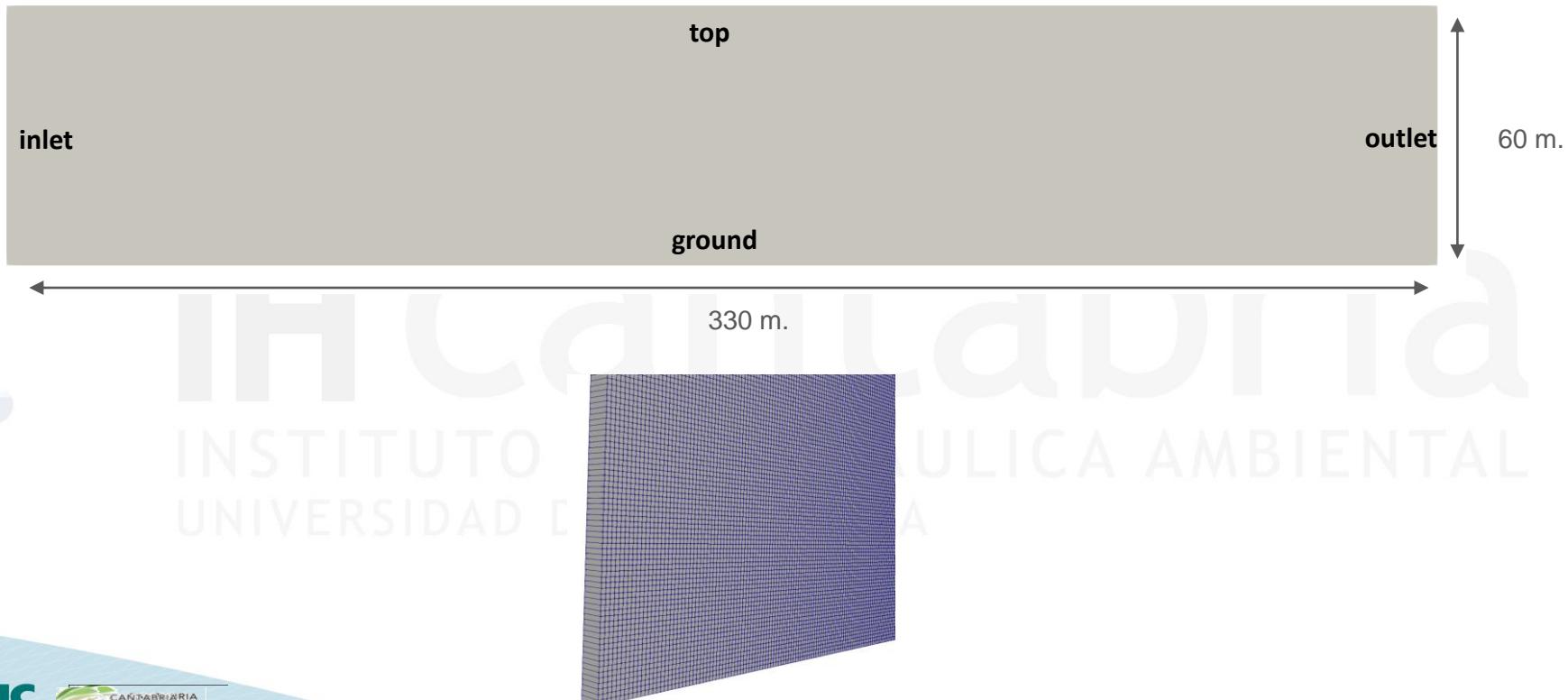
Name	Units	Value
inlet		
outlet		
ground		
top		
alpha.water		
U		
p_rgh	Kg*m/s <sup>2</sup>	totalPressure
side1		
alpha.water	zeroGradient	
U	slip	
p_rgh	Kg*m/s <sup>2</sup>	fixedFluxPressure
side2		
alpha.water	zeroGradient	
U	slip	
p_rgh	Kg*m/s <sup>2</sup>	fixedFluxPressure

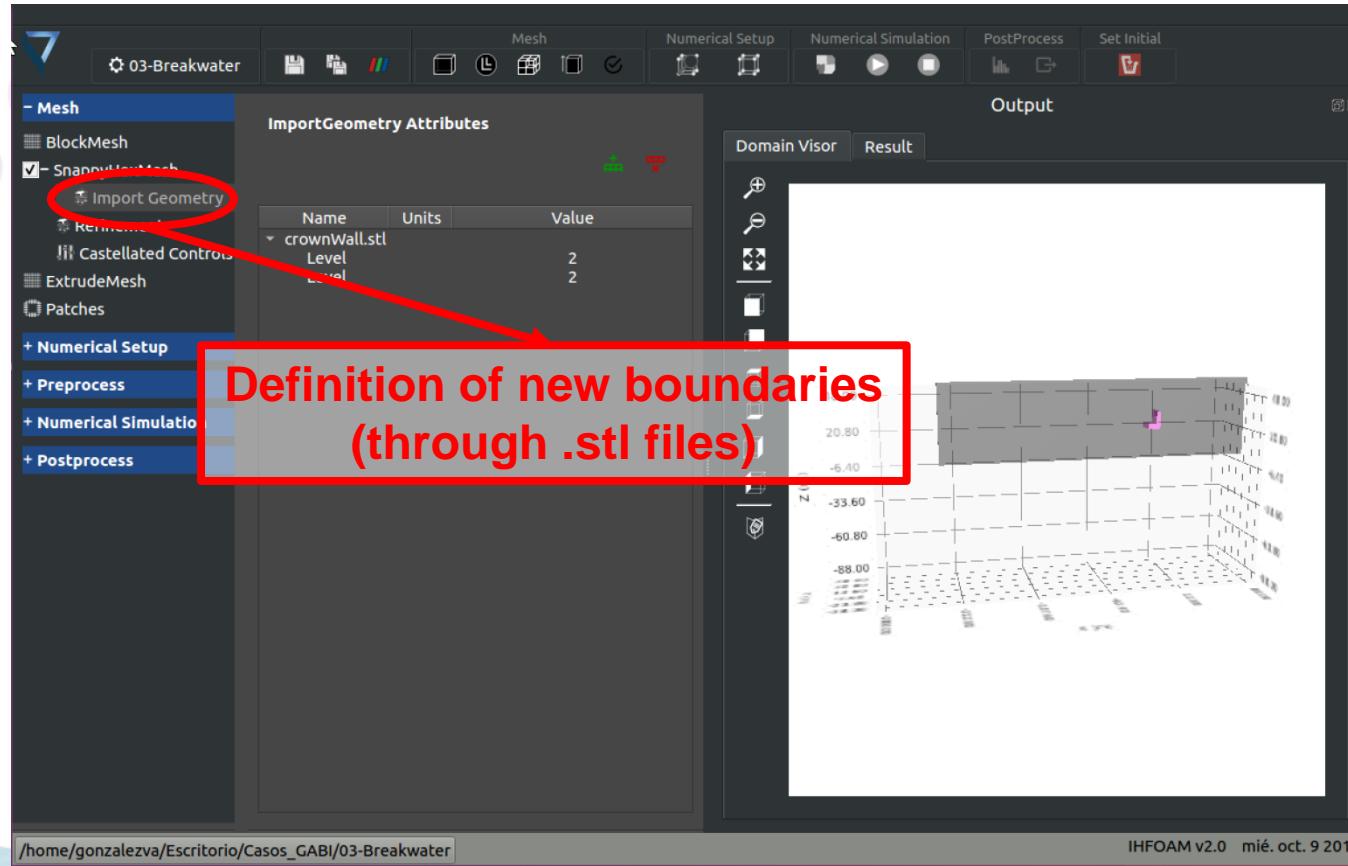


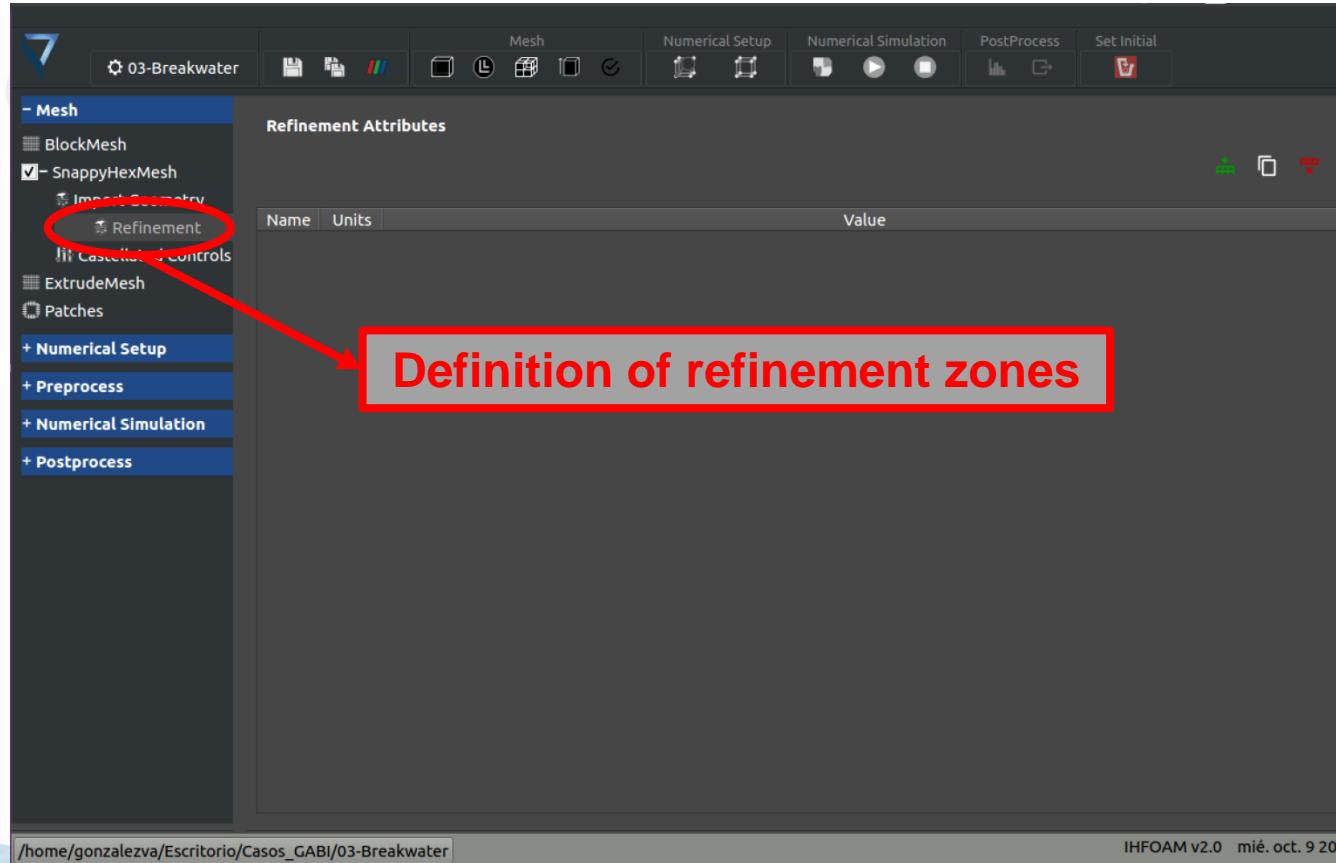
IHFOAM dialog box

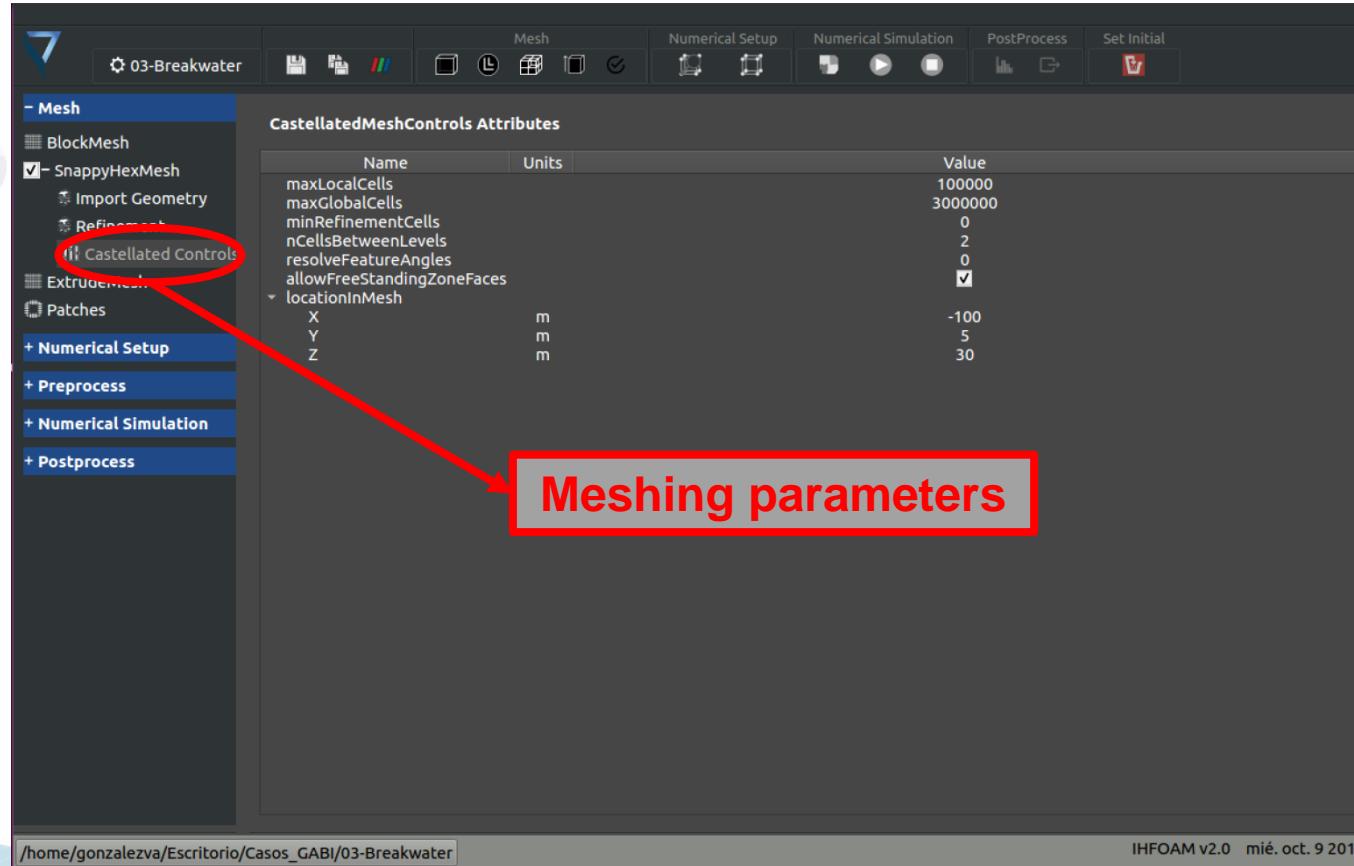
## Paraview button











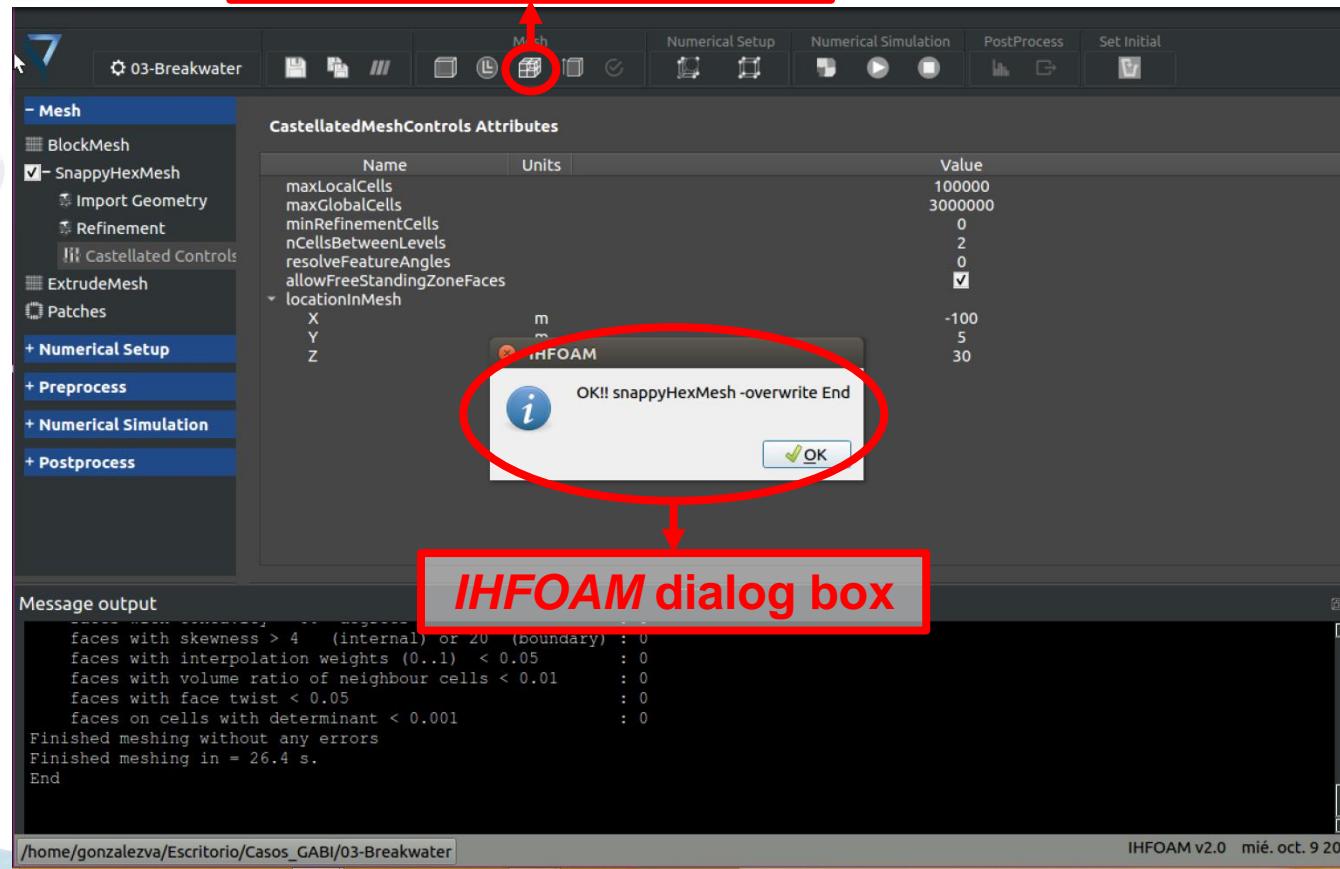
The screenshot shows the IHFOAM v2.0 software interface. The left sidebar has a 'Mesh' section expanded, showing options like BlockMesh, SnappyHexMesh (with Import Geometry and Refinement), Castellated Controls (which is circled in red), Extrudement, Patches, and several collapsed sections. The main area displays 'CastellatedMeshControls Attributes' with the following table:

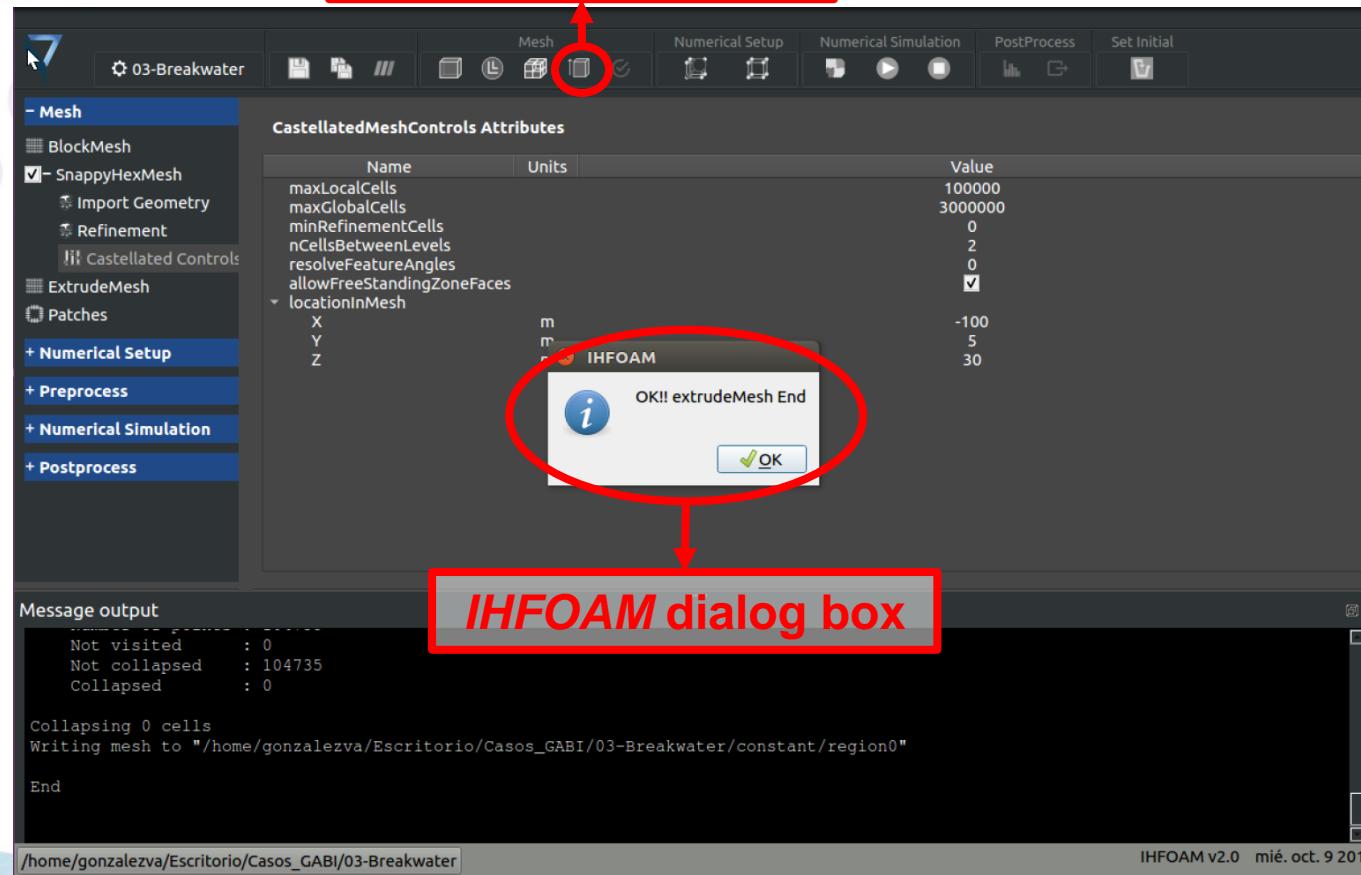
Name	Units	Value
maxLocalCells		100000
maxGlobalCells		3000000
minRefinementCells		0
nCellsBetweenLevels		2
resolveFeatureAngles		0
allowFreeStandingZoneFaces		<input checked="" type="checkbox"/>
locationInMesh		
X	m	-100
Y	m	5
Z	m	30

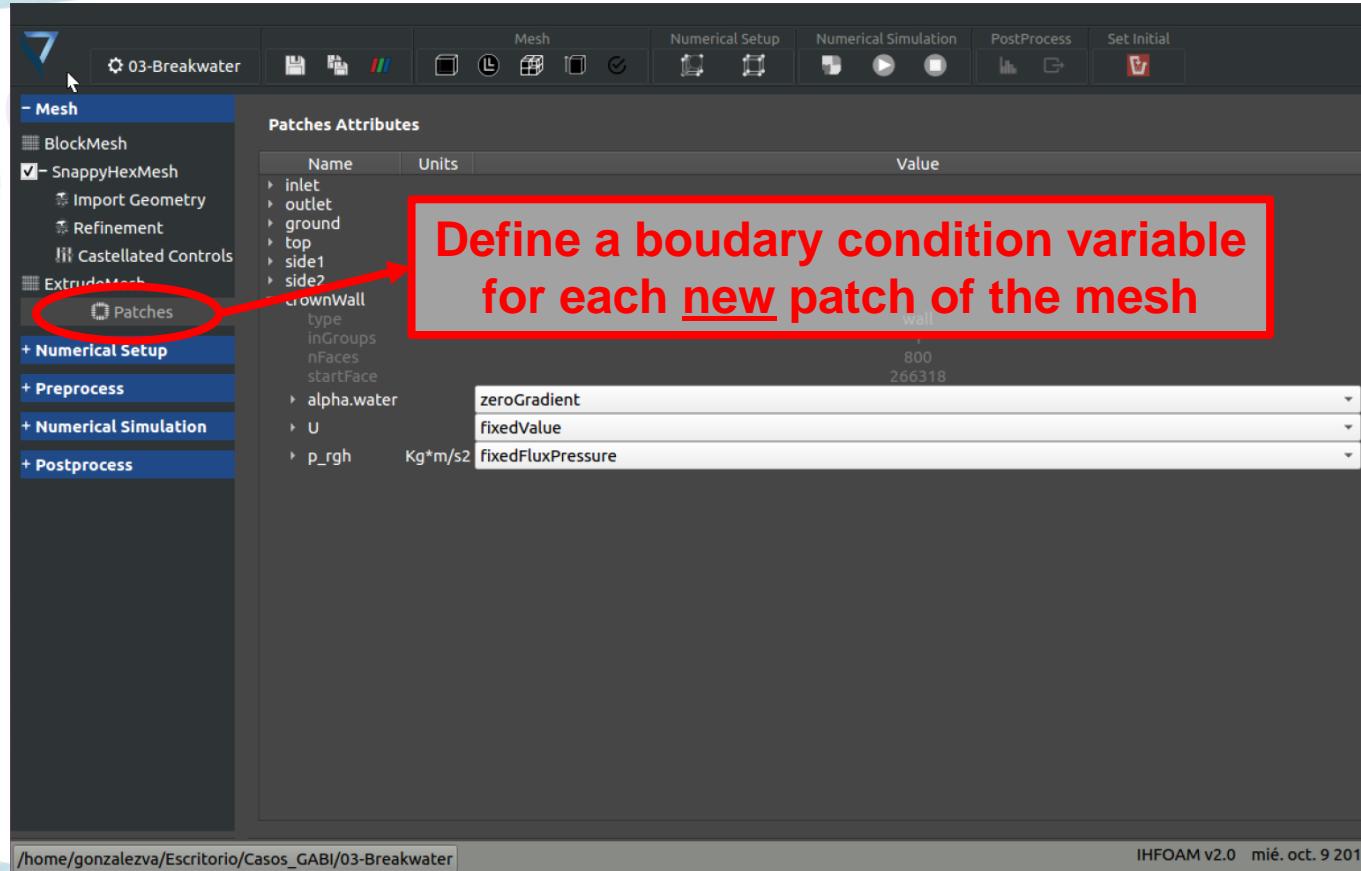
A large red arrow points from the 'Castellated Controls' section in the sidebar to the 'Meshing parameters' text in the bottom right of the main window.

**Meshing parameters**

/home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater      IHFOAM v2.0      mié. oct. 9 2019

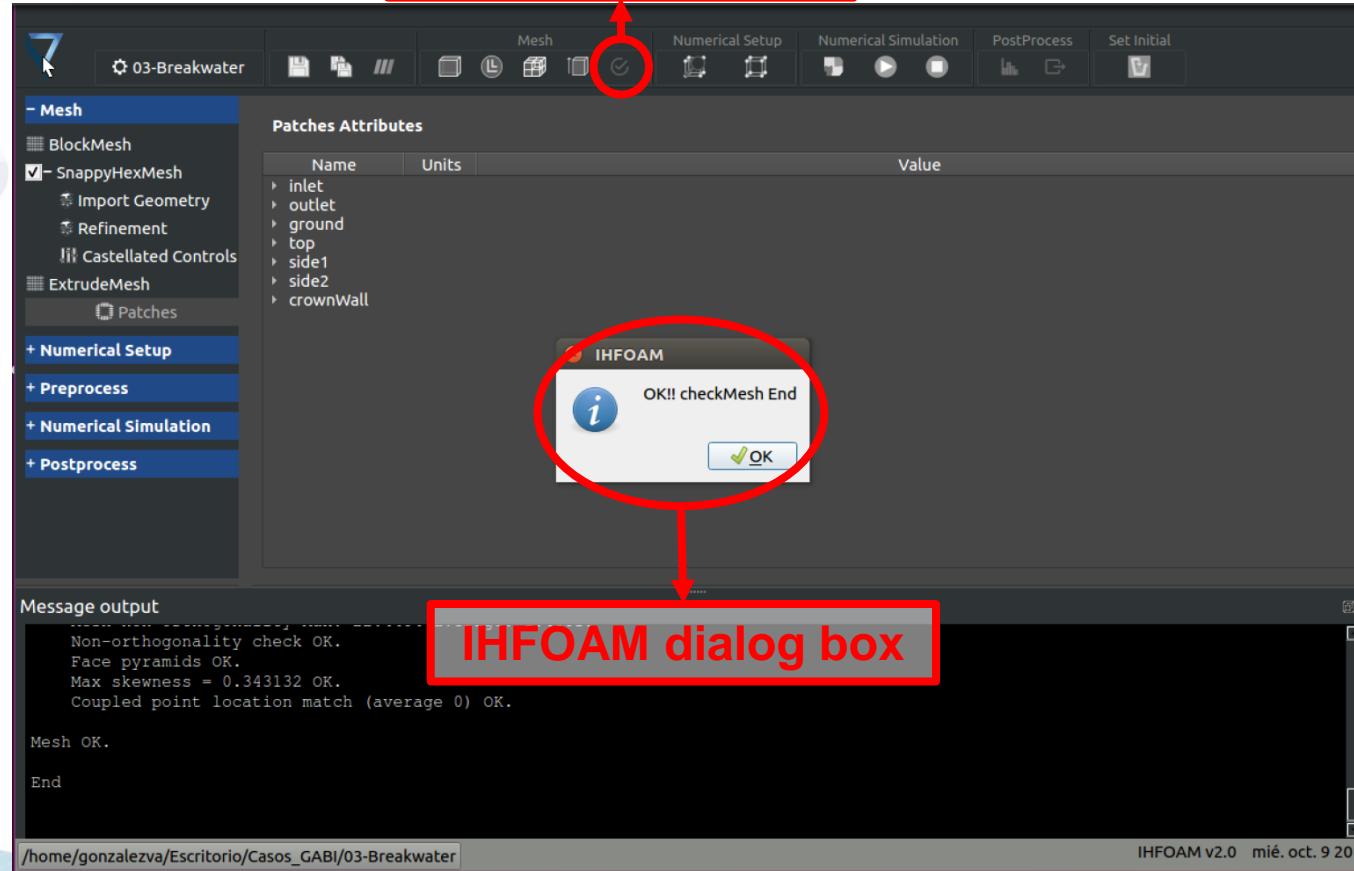






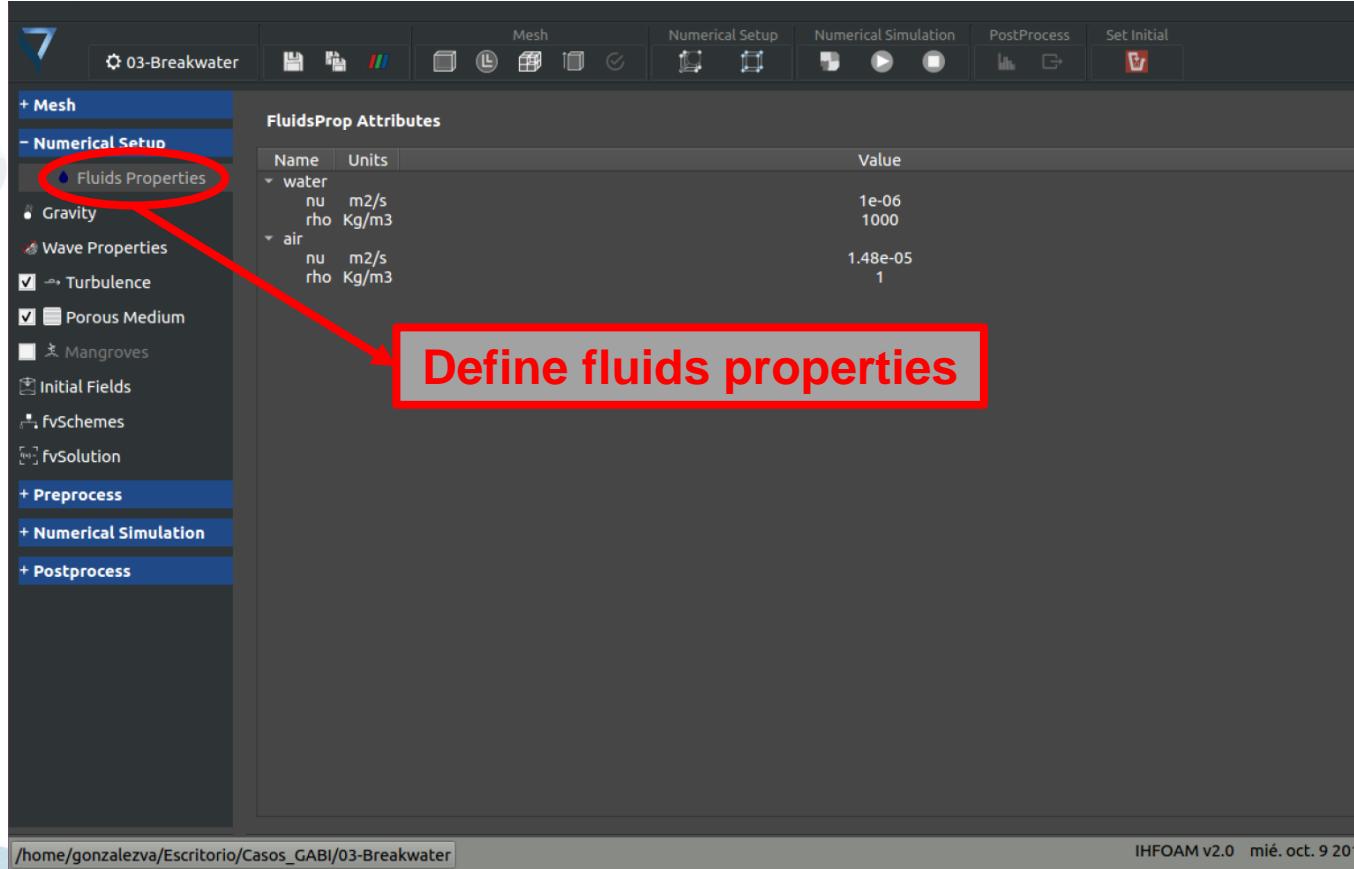
Define a boundary condition variable for each new patch of the mesh

Name	Units	Value
inlet		
outlet		
ground		
top		
side1		
side2		
crownWall		wall
type		
inGroups		
nFaces		800
startFace		266318
alpha.water		zeroGradient
U		fixedValue
p_rgh	Kg*m/s <sup>2</sup>	fixedFluxPressure



/home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019



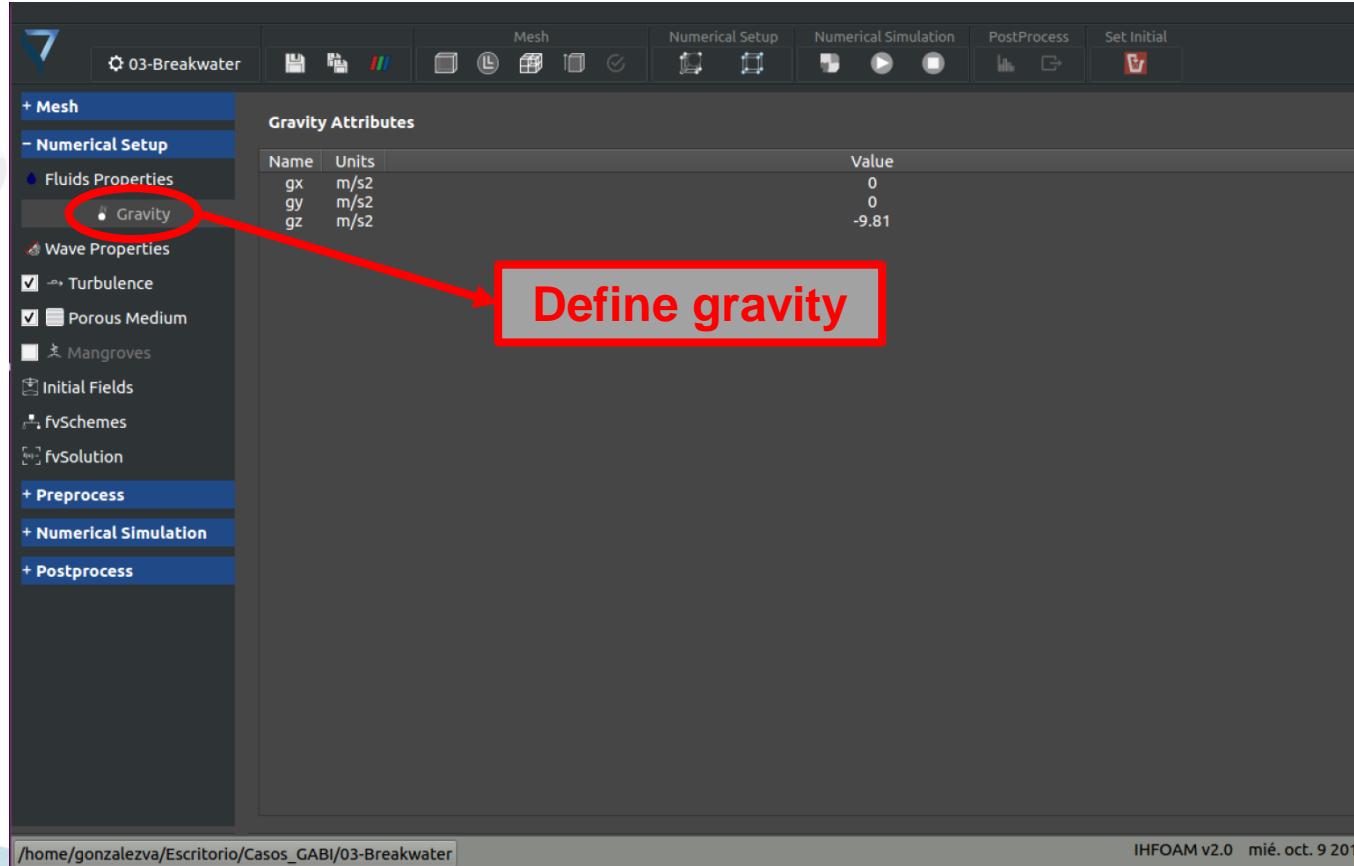
The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view of project components: Mesh, Numerical Setup (selected), Fluids Properties (highlighted with a red circle), Gravity, Wave Properties, Turbulence, Porous Medium, Mangroves, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The main panel displays the "FluidsProp Attributes" table with the following data:

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

A red arrow points from the "Fluids Properties" item in the sidebar to the text "Define fluids properties" which is enclosed in a red box.

Path: /home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater

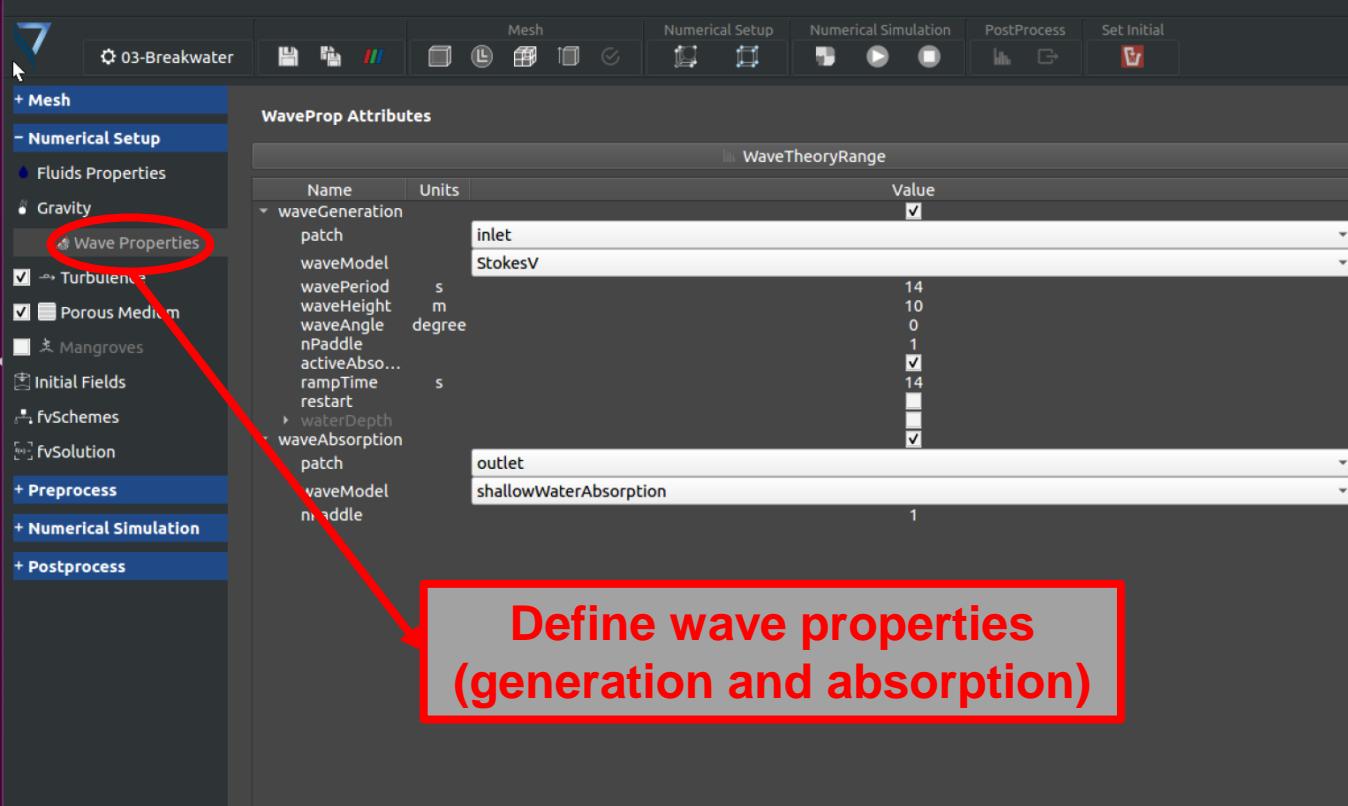
IHFOAM v2.0 mié. oct. 9 2019



The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view of project components: Mesh, Numerical Setup (selected), Fluids Properties, Wave Properties, Turbulence, Porous Medium, Mangroves, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The 'Gravity' item under Numerical Setup is highlighted with a red circle. A red arrow points from this circle to a red-bordered box containing the text "Define gravity". The main panel displays the "Gravity Attributes" table:

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

At the bottom of the interface, the path is shown as /home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater and the version is IHFOAM v2.0 mié. oct. 9 2019.



The screenshot shows the IFOAM v2.0 software interface. The left sidebar contains a tree view of project components: Mesh, Numerical Setup (selected), Fluids Properties, Gravity, Wave Properties (circled in red), Turbulence, Porous Medium, Mangroves, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The main panel displays 'WaveProp Attributes' with a table titled 'WaveTheoryRange'. The table has columns for Name, Units, and Value. The 'waveGeneration' section includes rows for patch (inlet), waveModel (StokesV), wavePeriod (14), waveHeight (10), waveAngle (0), nPaddle (1), activeAbo... (checked), rampTime (14), restart, waterDepth, waveAbsorption, patch (outlet), waveModel (shallowWaterAbsorption), and handle (1). A red arrow points from the 'Wave Properties' item in the sidebar to the 'waveGeneration' section in the table. A red box highlights the text 'Define wave properties (generation and absorption)'.

Define wave properties  
(generation and absorption)

03-Breakwater

+ Mesh

- Numerical Setup

- Fluids Properties
- Gravity
- Wave Properties
- Turbulence
- Porous Medium
- Mangroves
- Initial Fields
- fvSchemes
- FvSolution

+ Preprocess

+ Numerical Simulation

+ Postprocess

Mesh

Numerical Setup

Numerical Simulation

PostProcess

Set Initial

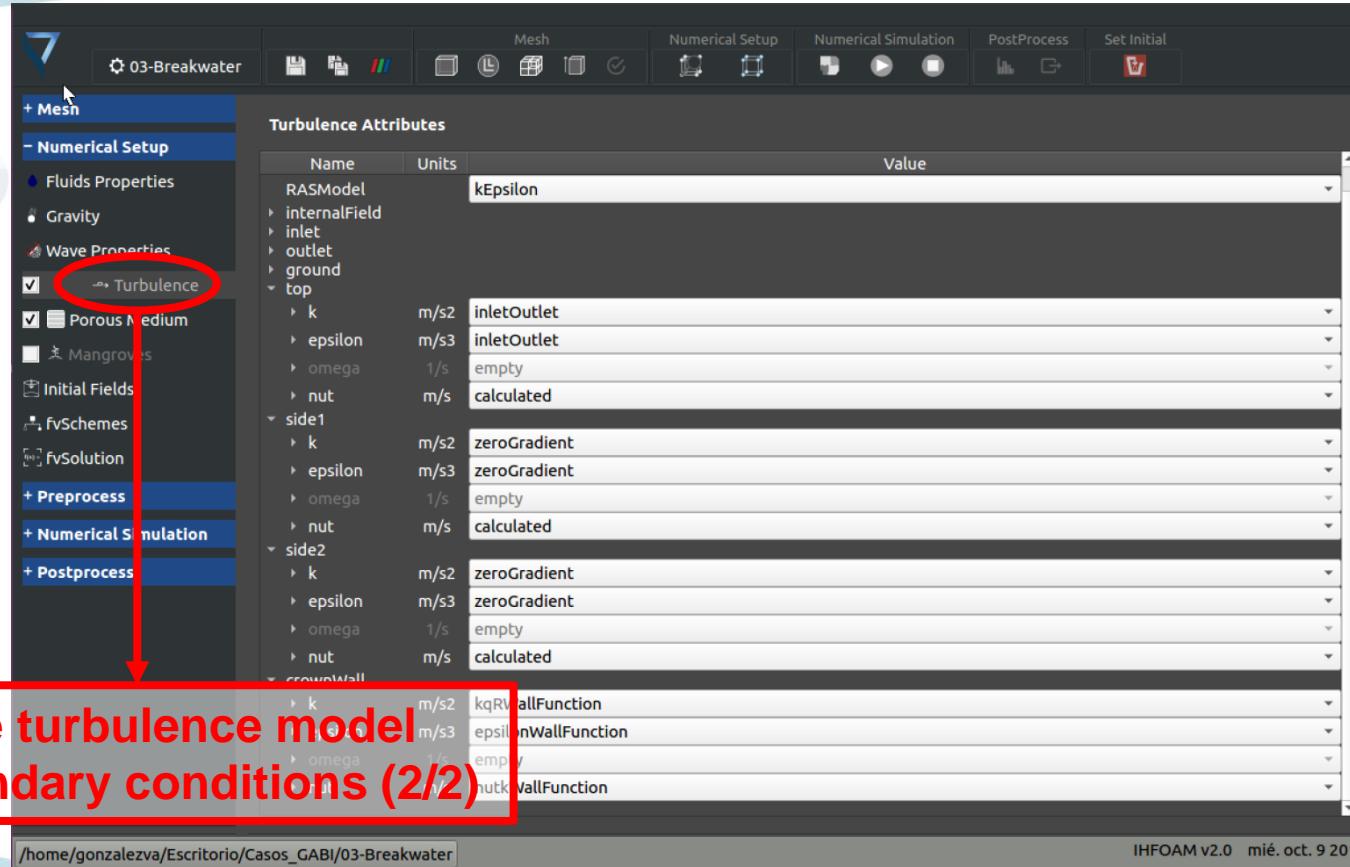
Turbulence Attributes

Name	Units	Value
simulationType		RAS
RASModel		kEpsilon
internalField		
k	m <sup>2</sup> /s <sup>2</sup>	0.96
epsilon	m <sup>2</sup> /s <sup>3</sup>	0.0038479
omega	1/s	0
nut	m <sup>2</sup> /s	0
inlet		
k	m/s <sup>2</sup>	zeroGradient
epsilon	m/s <sup>3</sup>	zeroGradient
omega	1/s	empty
nut	m/s	calculated
outlet		
k	m/s <sup>2</sup>	zeroGradient
epsilon	m/s <sup>3</sup>	zeroGradient
omega	1/s	empty
nut	m/s	calculated
ground		
k	m/s <sup>2</sup>	kqRWallFunction
epsilon	m/s <sup>3</sup>	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

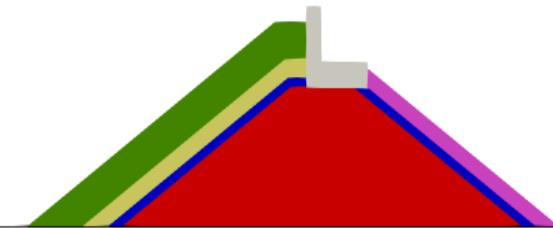
IINFOAM v2.0 mié. oct. 9 2019



Define turbulence model and boundary conditions (2/2)

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

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



UNIVERSIDAD DE CANTABRIA

The screenshot shows the IHFOAM v2.0 software interface. On the left, a sidebar lists various numerical setup options. A red circle highlights the 'Porous Medium' option under the 'Turbulence' section. A large red arrow points from this highlighted area down to a red-bordered box at the bottom left containing the text 'Define porous layers (1/2)'. The main central area displays a 'Porous Attributes' table with data for different STL files. To the right, a 3D plot shows a cross-section of a rubble-mound breakwater structure.

Name	Units	Value
core.stl		
D <sub>50</sub>		0.3
a	m	200
b	m	0.8
c	m	0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
Filter.stl		
D <sub>50</sub>		0.4
a	m	200
b	m	1
c	m	0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
cubes_6tn.stl		
D <sub>50</sub>		1.37
a	m	200
b	m	1.1
c	m	0.34
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10
cubes_65pt.stl		
outsidePoints		
outsidePoints X	m	-75
outsidePoints Y	m	2
outsidePoints Z	m	10

Output

Domain Visor Result

Z (m)

Define porous layers (1/2)

/home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

The screenshot shows the GABI software interface with the following details:

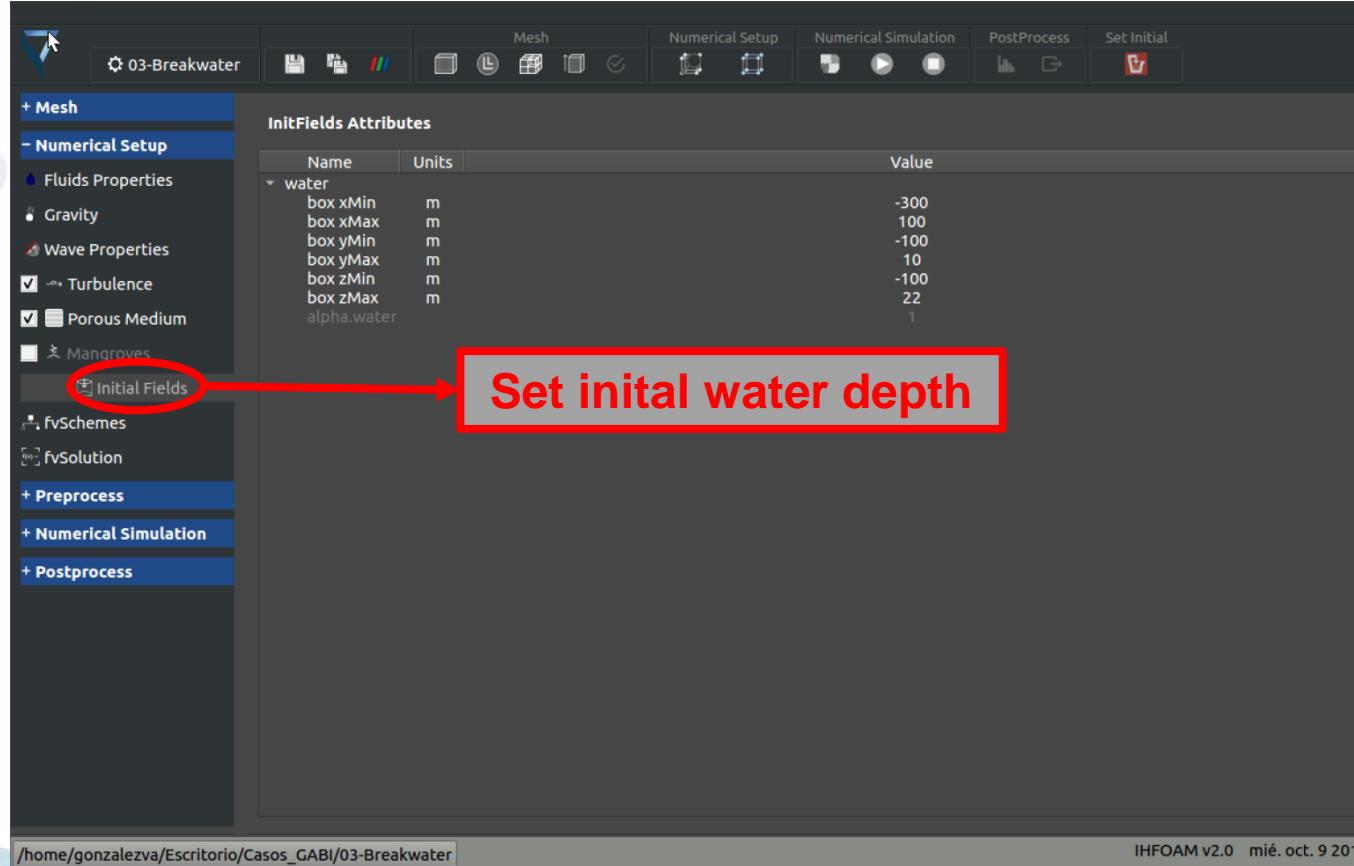
- Left Panel (Navigation):**
  - + Mesh
  - Numerical Setup
    - Fluids Properties
    - Gravity
    - Wave Properties
    - Turbulence
    - Porous Medium** (highlighted with a red circle)
  - Mangroves
  - Initial Fields
  - fvschemes
  - FvSolution
  - + Preprocess
  - + Numerical Simulation
  - + Postprocess
- Porous Attributes Panel:**

**Porous Attributes**

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

A 3D plot showing a cross-section of a rubble-mound breakwater. The vertical axis is labeled Z (m) with values from -100.00 to 60.00. The plot shows a grey rectangular area representing the domain, with a pink triangular shape representing the rubble mound. A blue line indicates the water level at approximately Z = 28 m.

Define porous layers (2/2)



The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view of project components: Mesh, Numerical Setup (selected), 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:

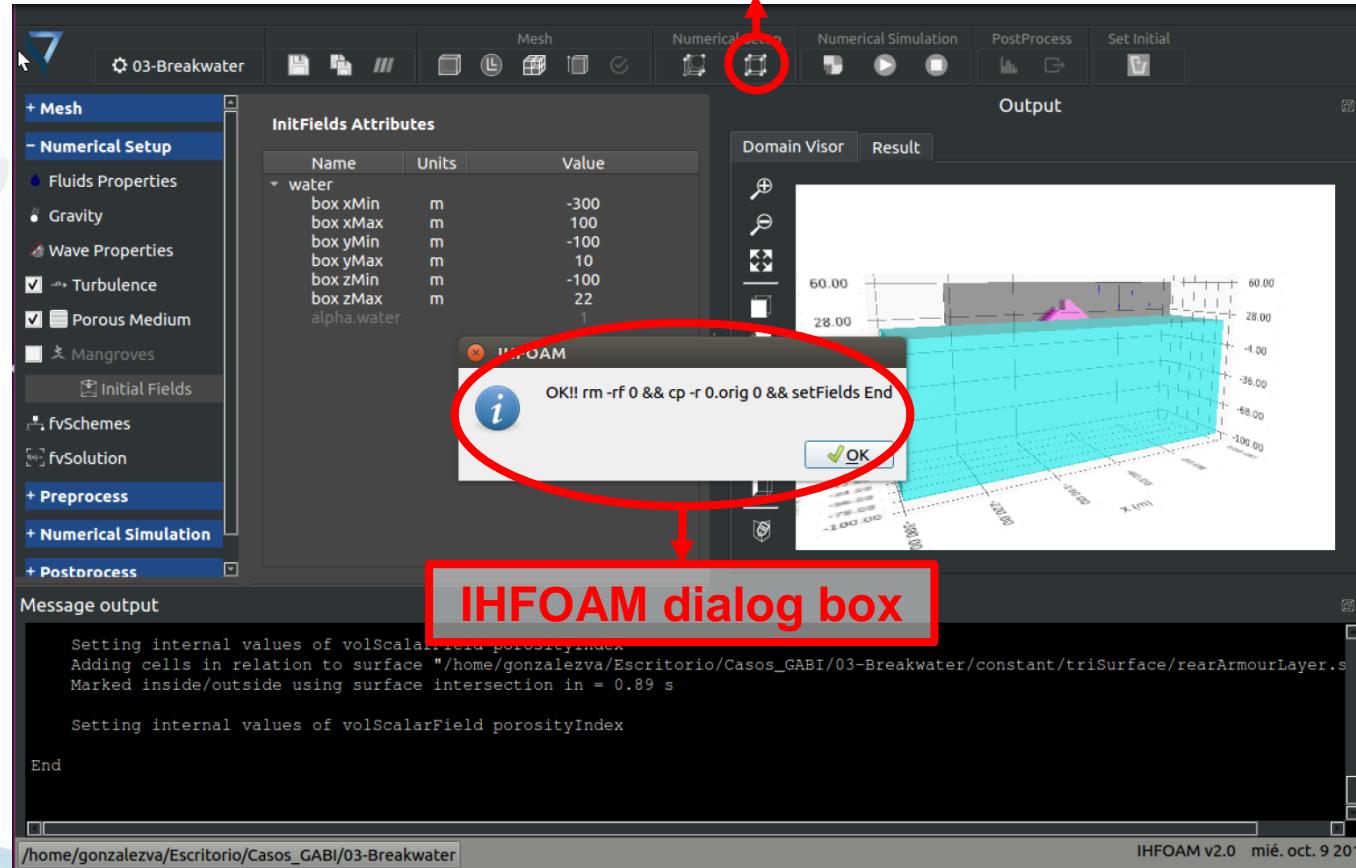
Name	Units	Value
water		
box xMin	m	-300
box xMax	m	100
box yMin	m	-100
box yMax	m	10
box zMin	m	-100
box zMax	m	22
alpha.water		1

A red arrow points from the 'Initial Fields' entry in the sidebar to the table, with the text 'Set initial water depth' overlaid in a red box.

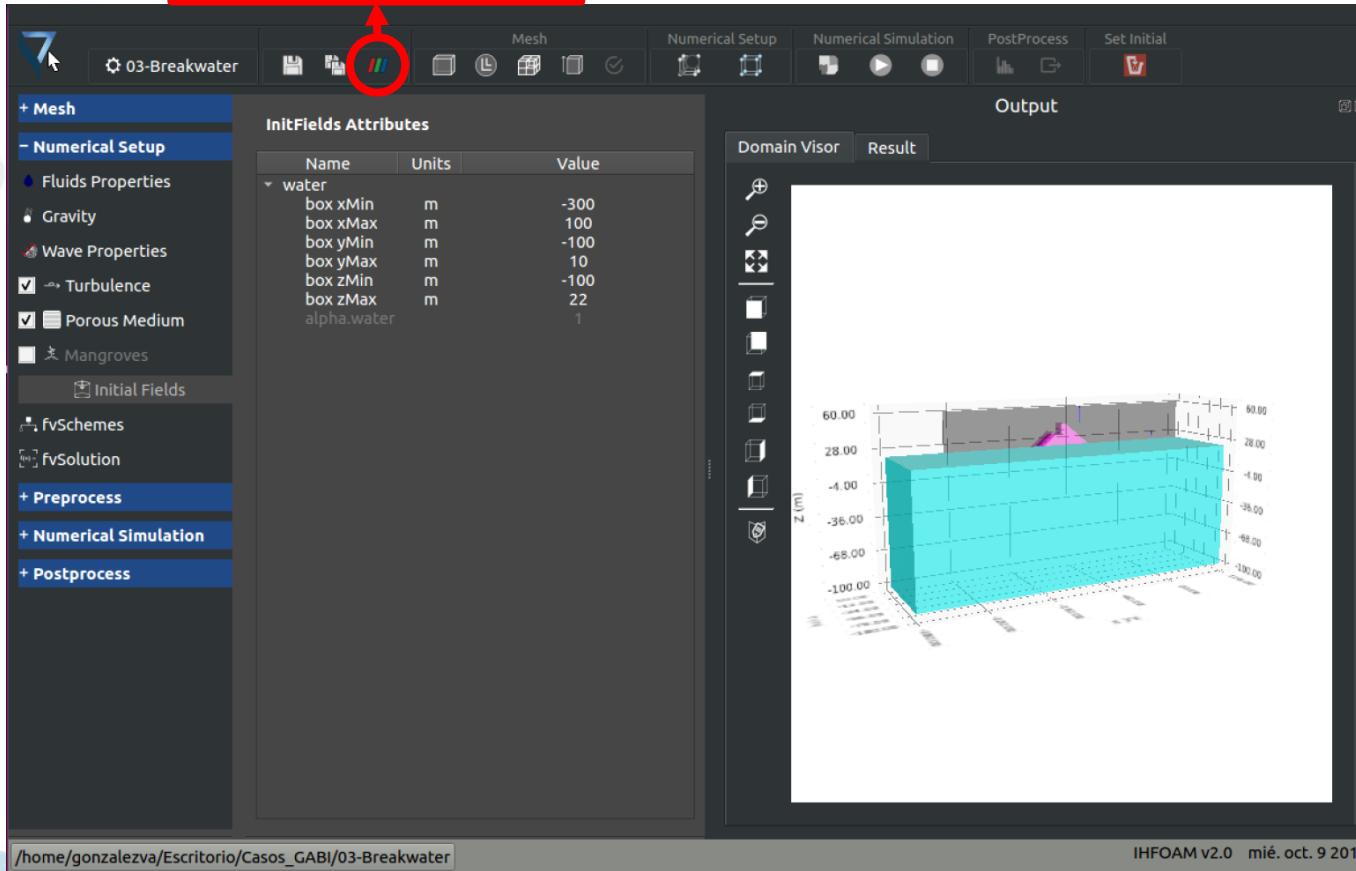
/home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019

**setFields button**

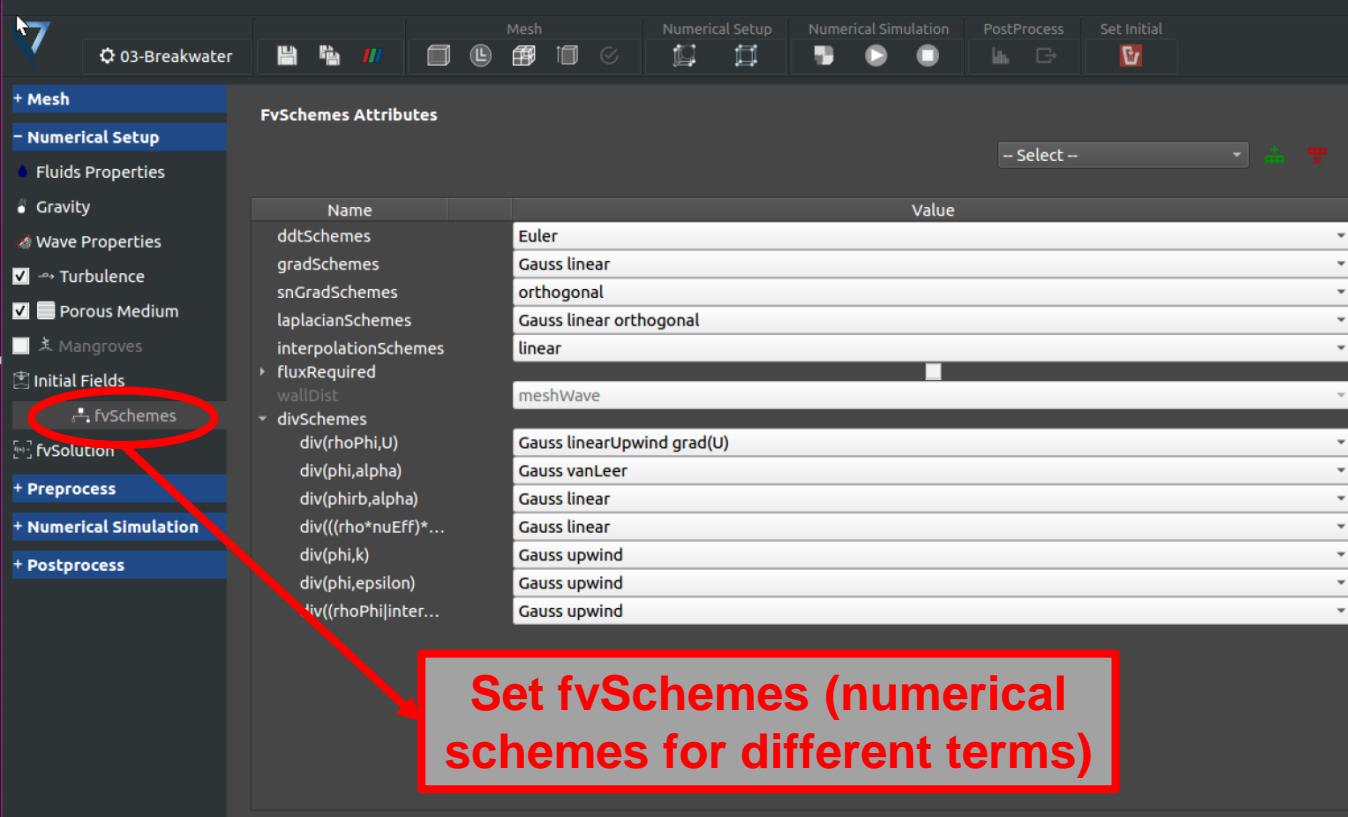


## Paraview button





INSTITUTO DE HIDRAULICA AMBIENTAL  
UNIVERSIDAD DE CANTABRIA



**fvSchemes Attributes**

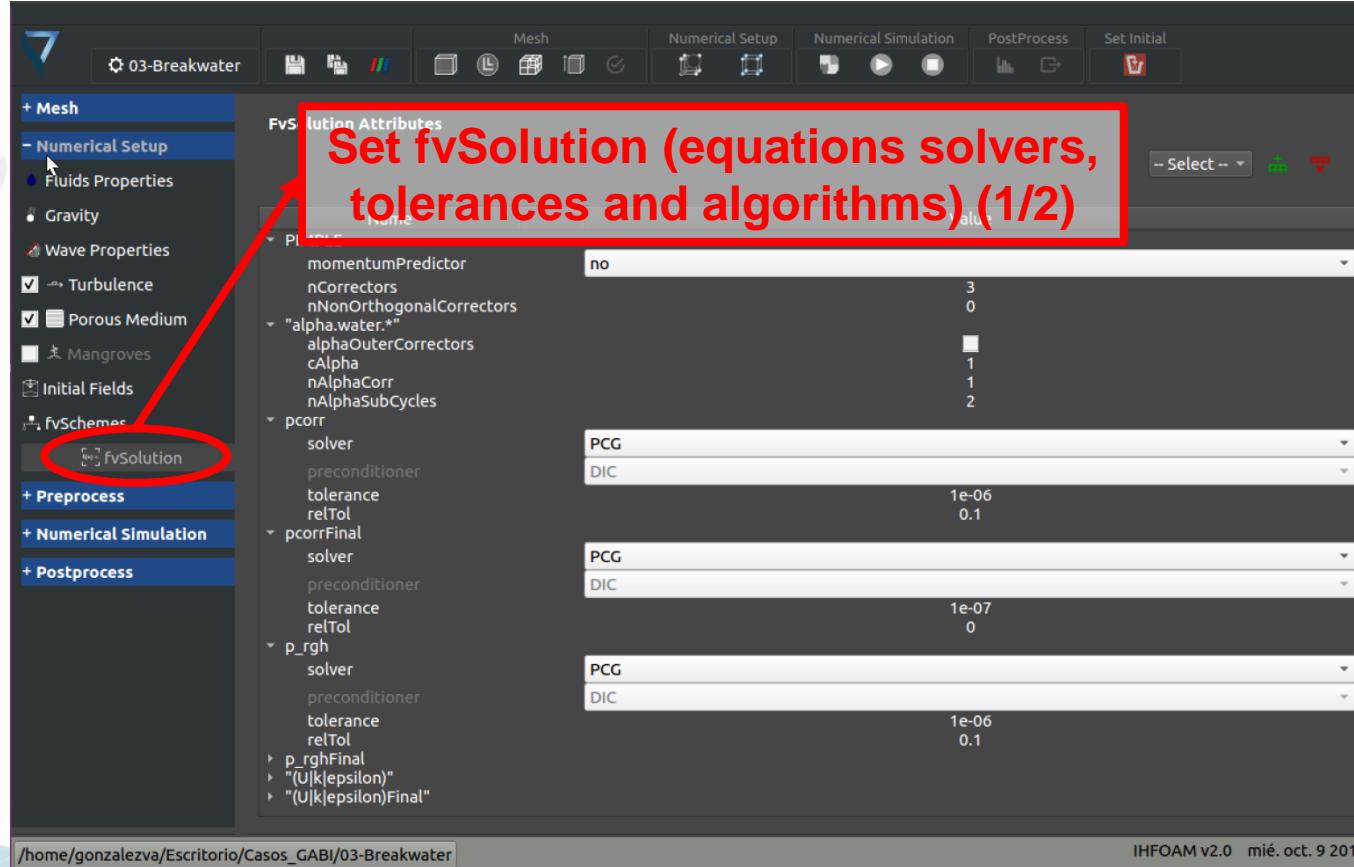
-- Select --

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

**Set fvSchemes (numerical schemes for different terms)**

/home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater

IHFOAM v2.0 mié. oct. 9 2019



The screenshot shows the IHFOAM v2.0 software interface. On the left, there is a navigation tree with the following items:

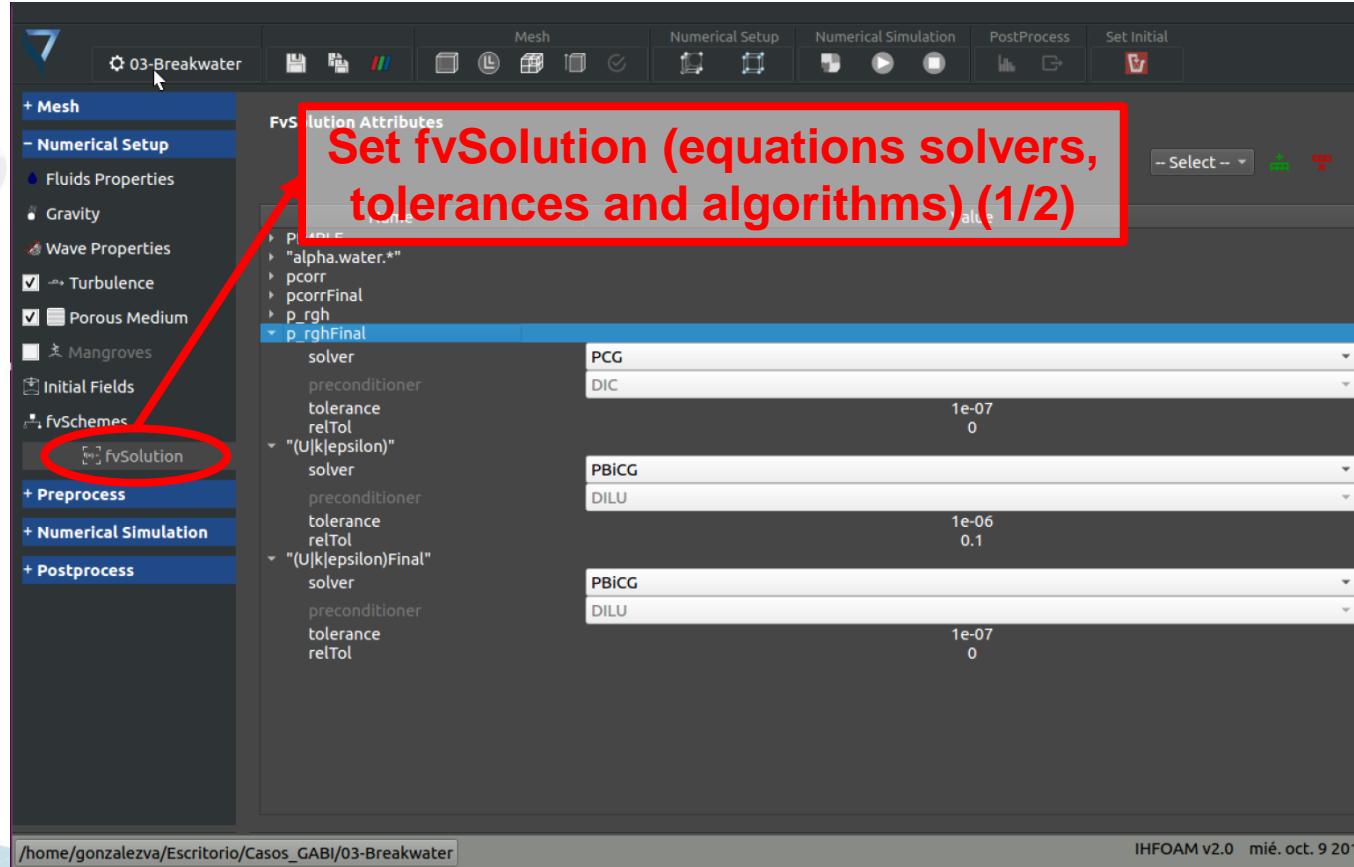
- + Mesh
- Numerical Setup
  - Fluids Properties
  - Gravity
  - Wave Properties
  - Turbulence
  - Porous Medium
  - Mangroves
- Initial Fields
- fvSchemes
- + Preprocess
- + Numerical Simulation
- + Postprocess

A red circle highlights the "fvSolution" item under the "fvSchemes" section.

The main panel displays the "fvSolution Attributes" configuration. A large red box surrounds the top portion of this panel with the text: "Set fvSolution (equations solvers, tolerances and algorithms) (1/2)".

The configuration table shows the following settings:

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



The screenshot shows the IHOAM v2.0 software interface with a red box highlighting the 'fvSolution' section in the 'Numerical Setup' tree view. A red arrow points from the 'fvSolution' node in the tree to the detailed configuration window.

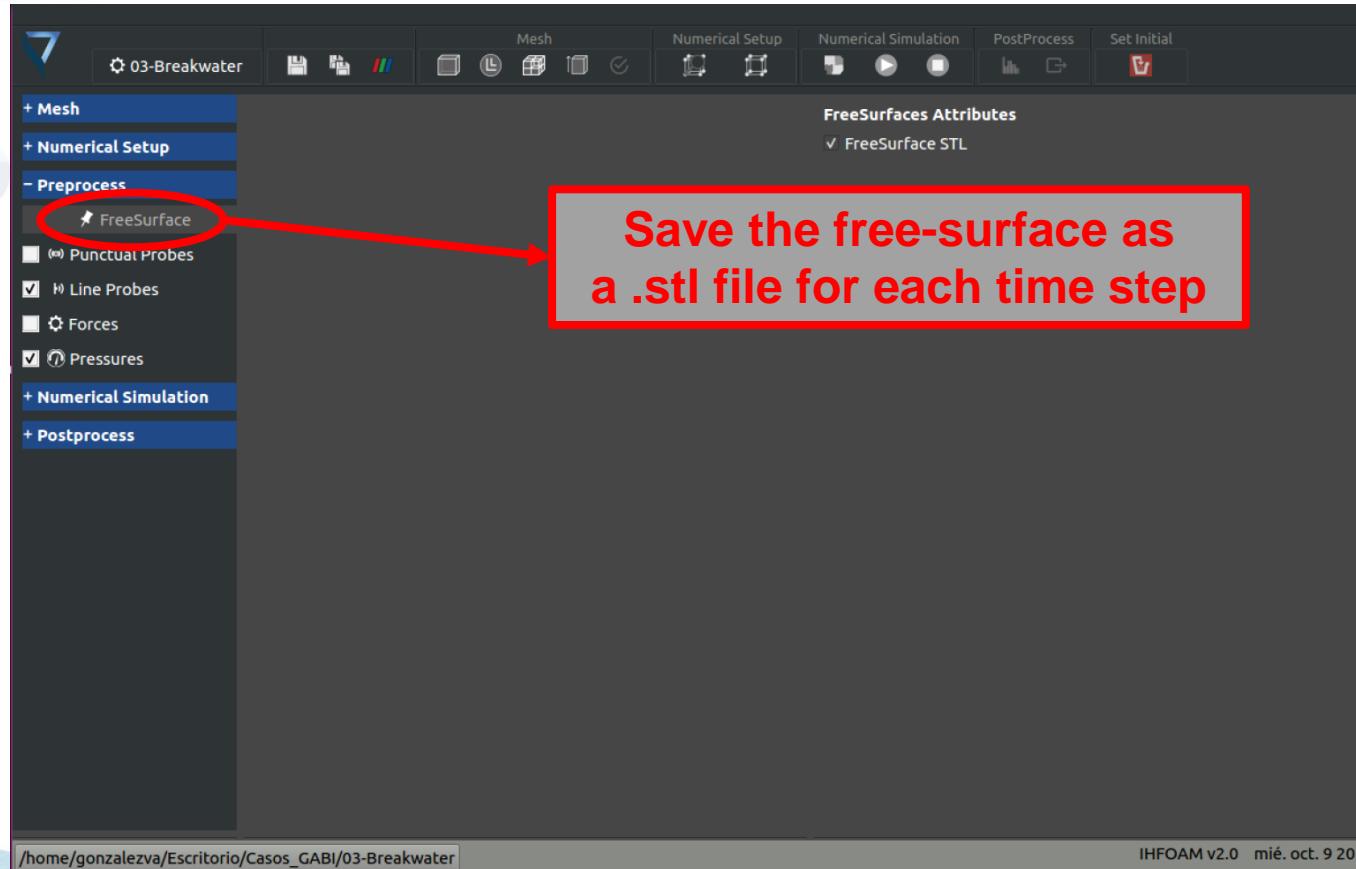
**fvSolution Attributes**

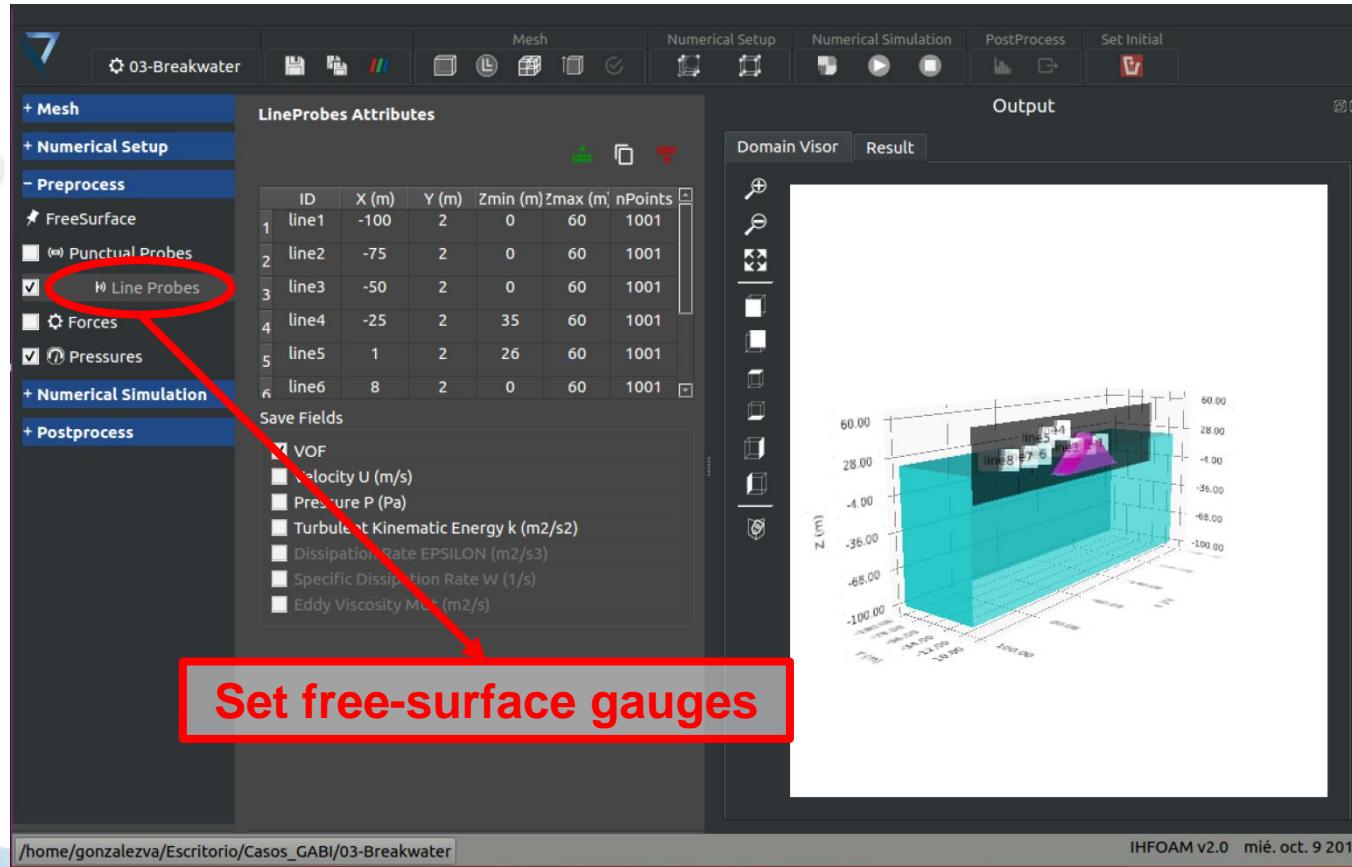
**Set fvSolution (equations solvers, tolerances and algorithms) (1/2)**

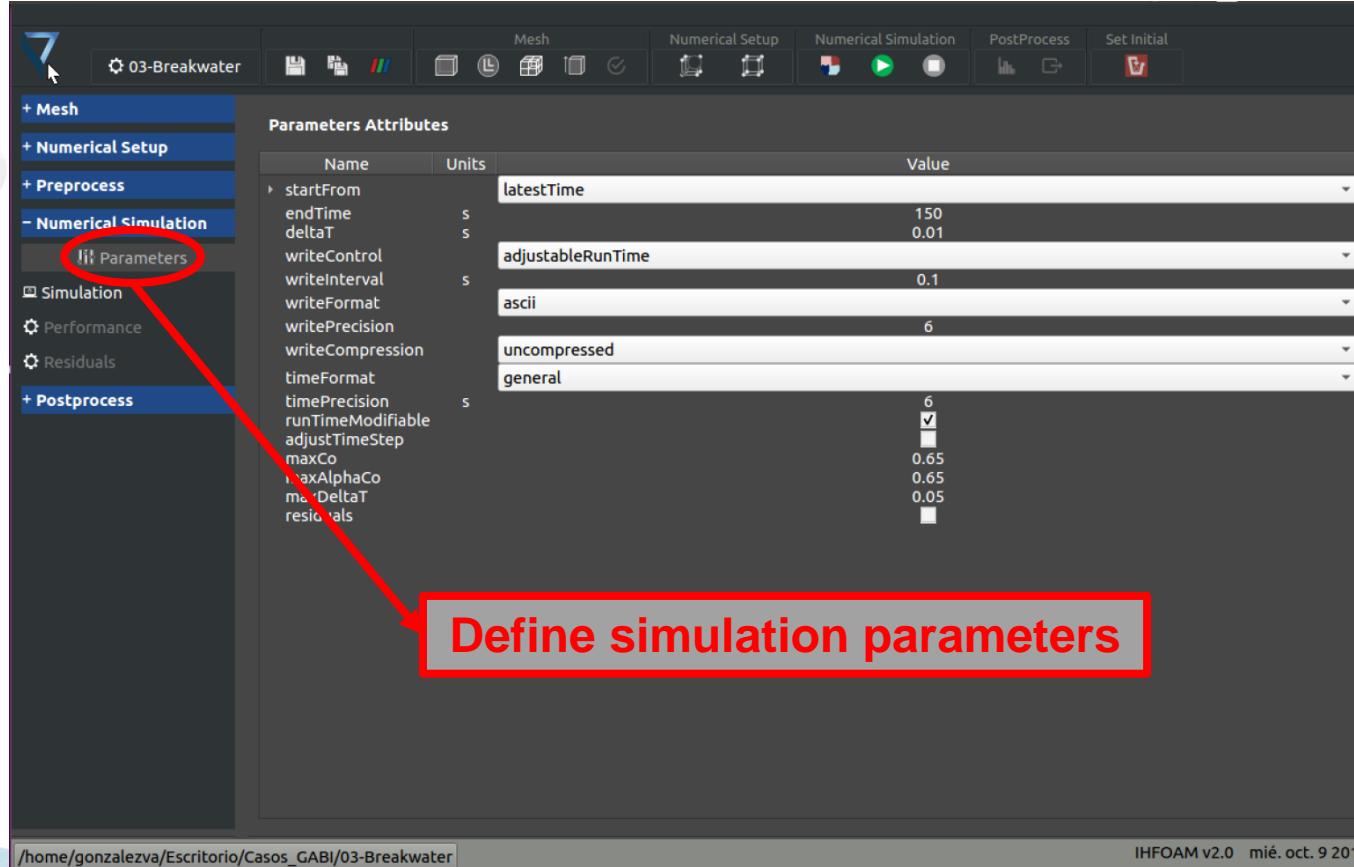
Equation	Solver	Preconditioner	Tolerance	RelTol
"alpha.water:*	PCG	DIC	1e-07	0
"(U k epsilon)"	PBiCG	DILU	1e-06	0.1
"(U k epsilon)Final"	PBiCG	DILU	1e-07	0

/home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater

IHOAM v2.0 mié. oct. 9 2019



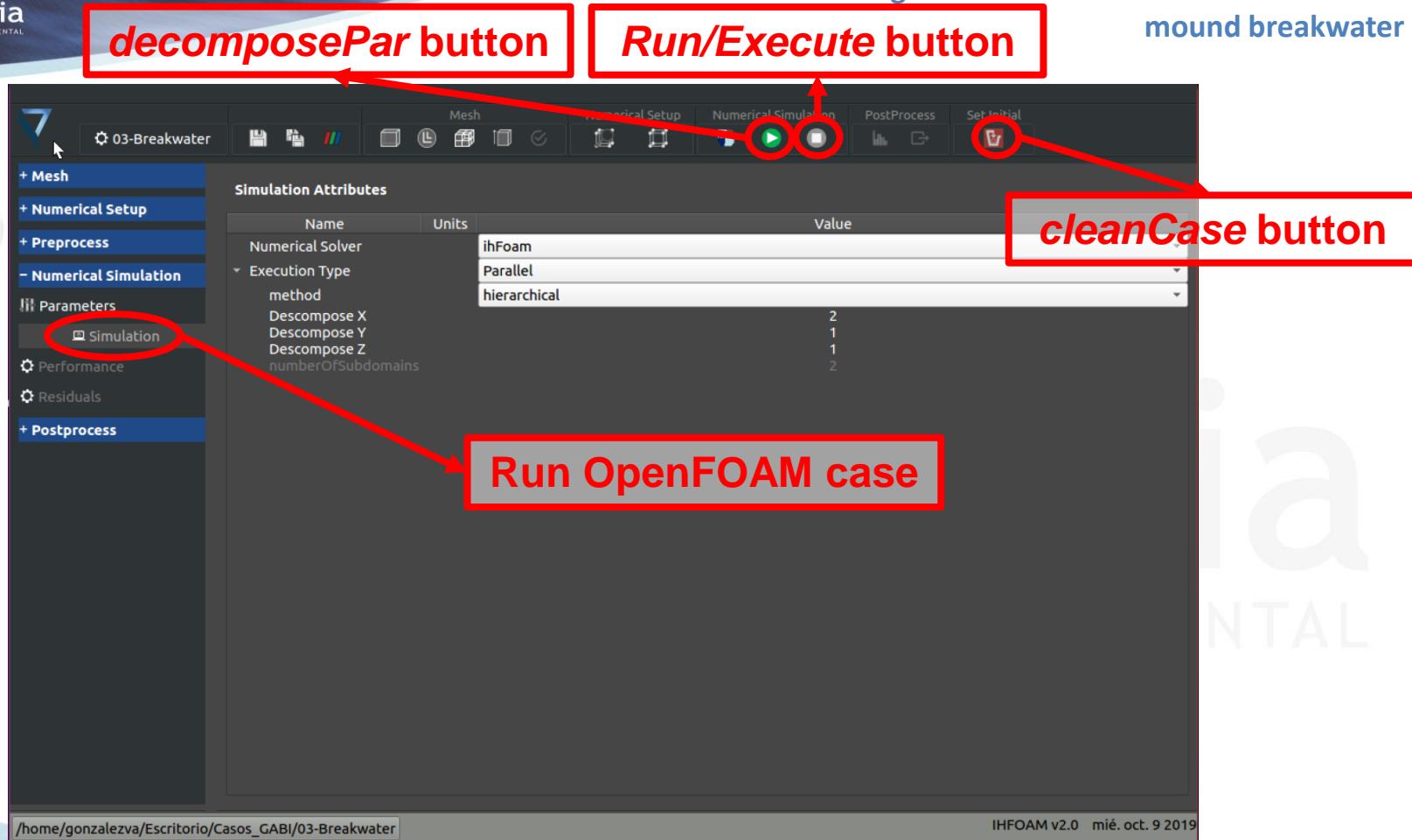




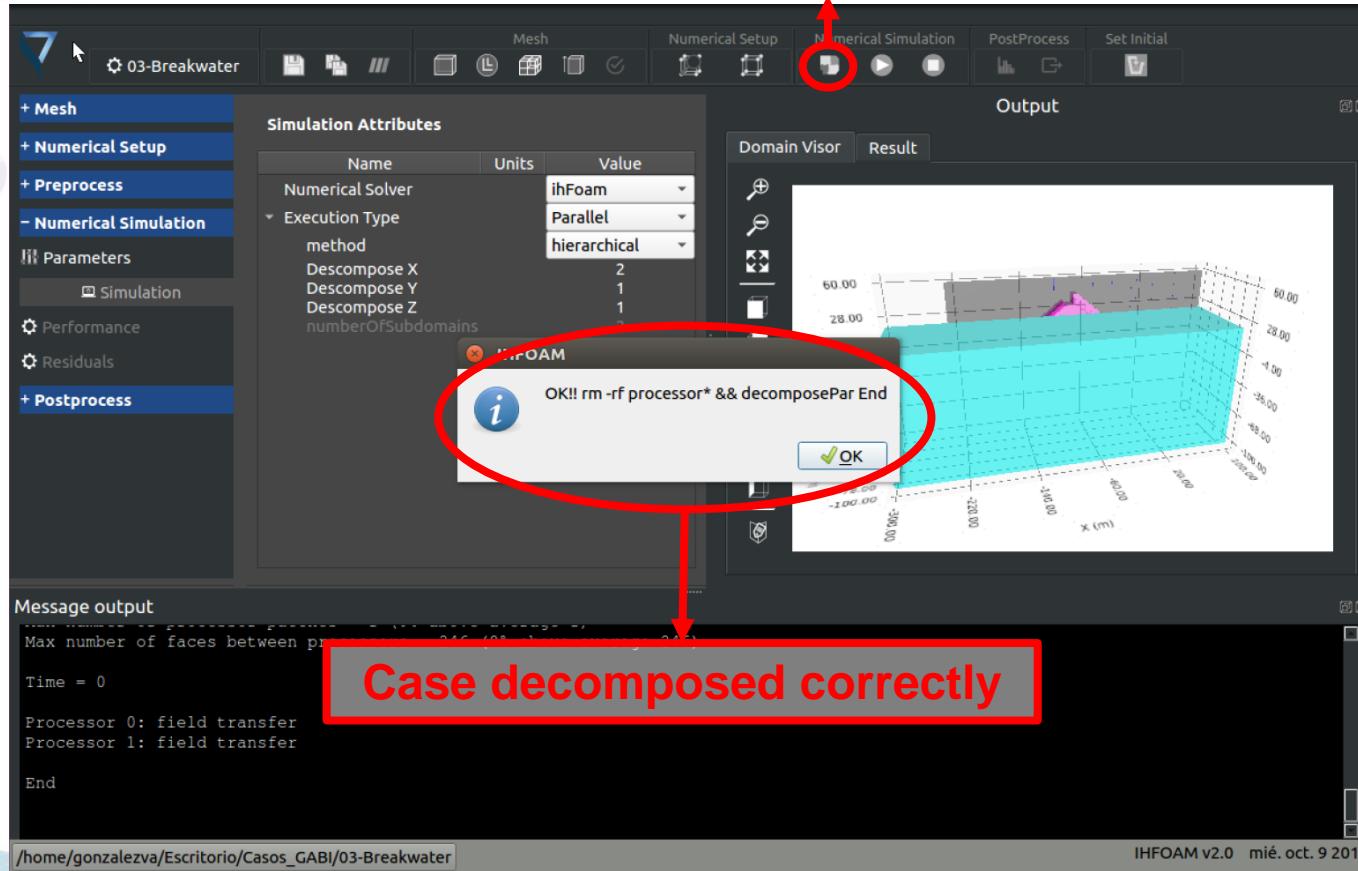
The screenshot shows the IHFOAM v2.0 software interface. The left sidebar has a tree view with categories like Mesh, Numerical Setup, Preprocess, Numerical Simulation, and Postprocess. The 'Numerical Simulation' category is expanded, and its 'Parameters' sub-item is highlighted with a red circle and a red arrow pointing to a red box containing the text 'Define simulation parameters'. The main area displays a table of numerical 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	s	general
timePrecision		6
runTimeModifiable		<input checked="" type="checkbox"/>
adjustTimeStep		<input type="checkbox"/>
maxCo		0.65
maxAlphaCo		0.65
minDeltaT		0.05
residuals		<input type="checkbox"/>

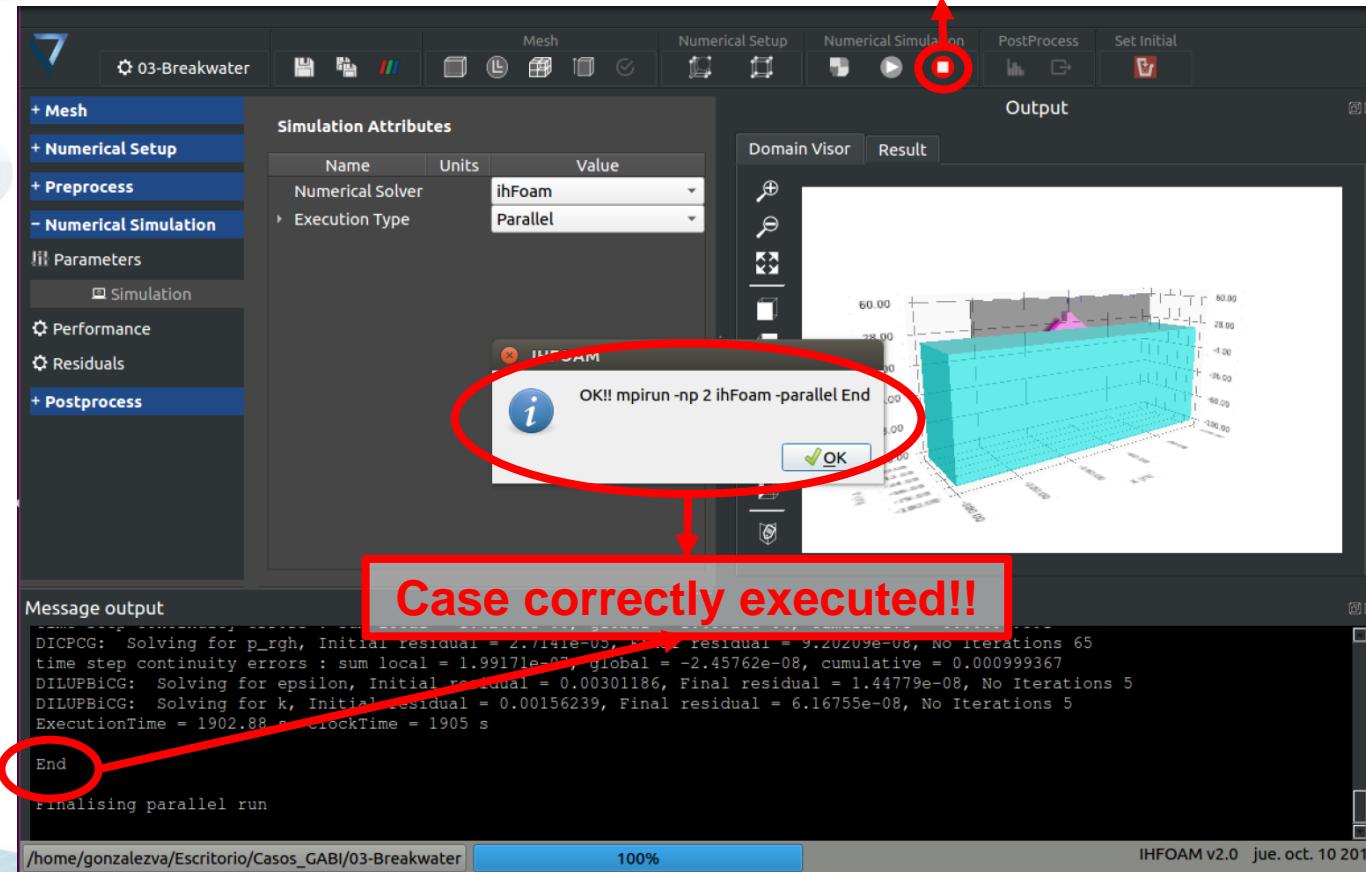
At the bottom left, the file path is shown as /home/gonzalezva/Escritorio/Casos\_GABI/03-Breakwater. At the bottom right, the text IHFOAM v2.0 mié, oct. 9 2019 is displayed.

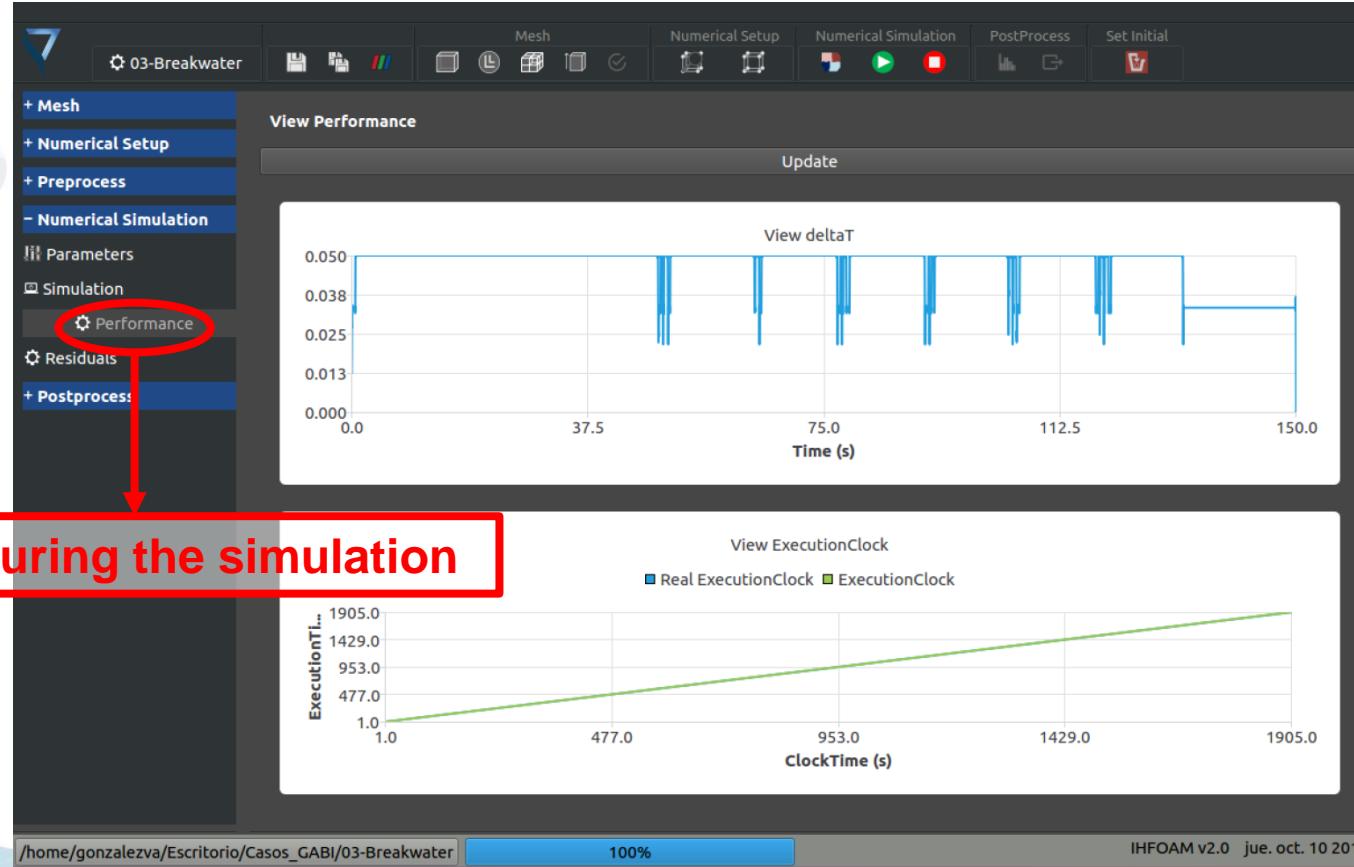


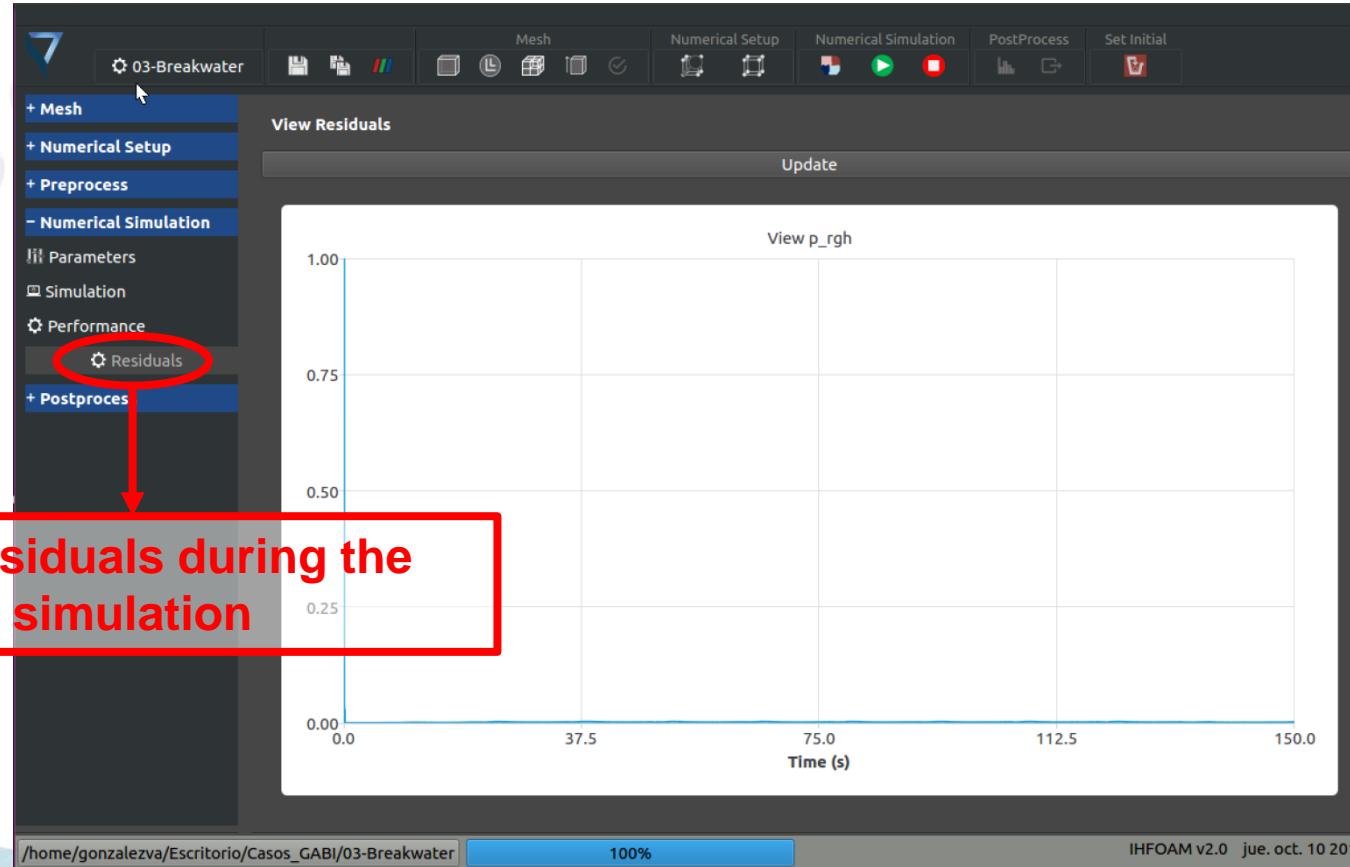
## decomposePar button



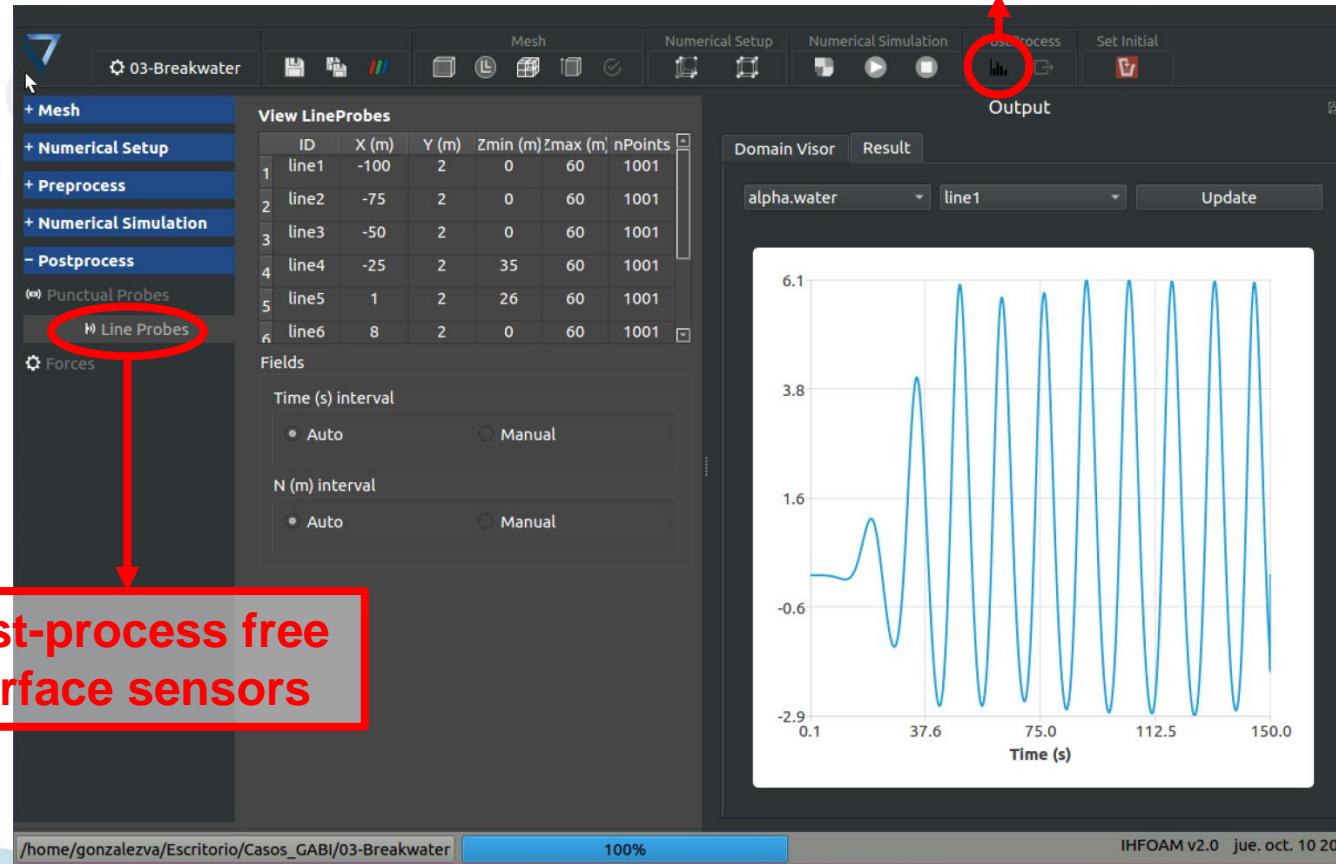
Run/Execute button



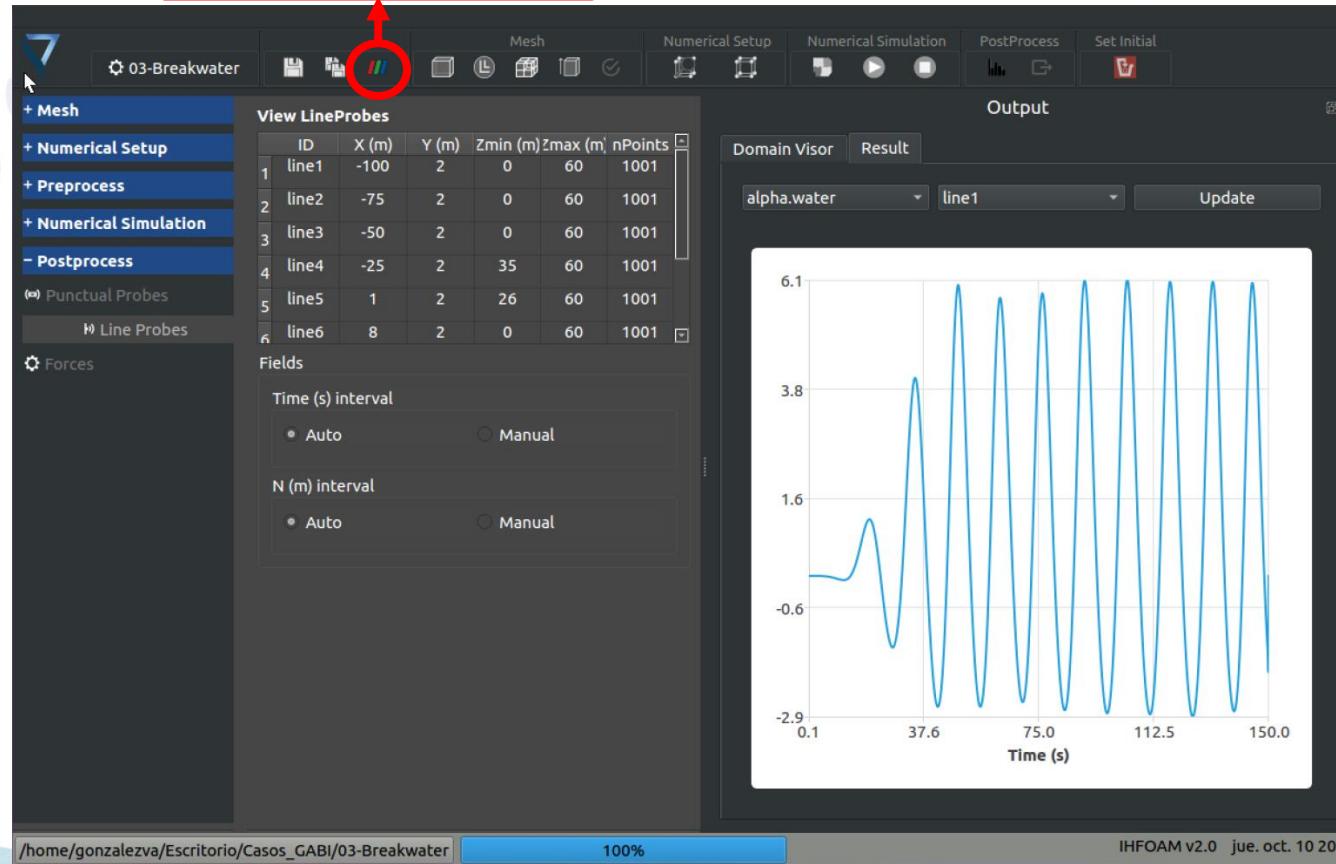


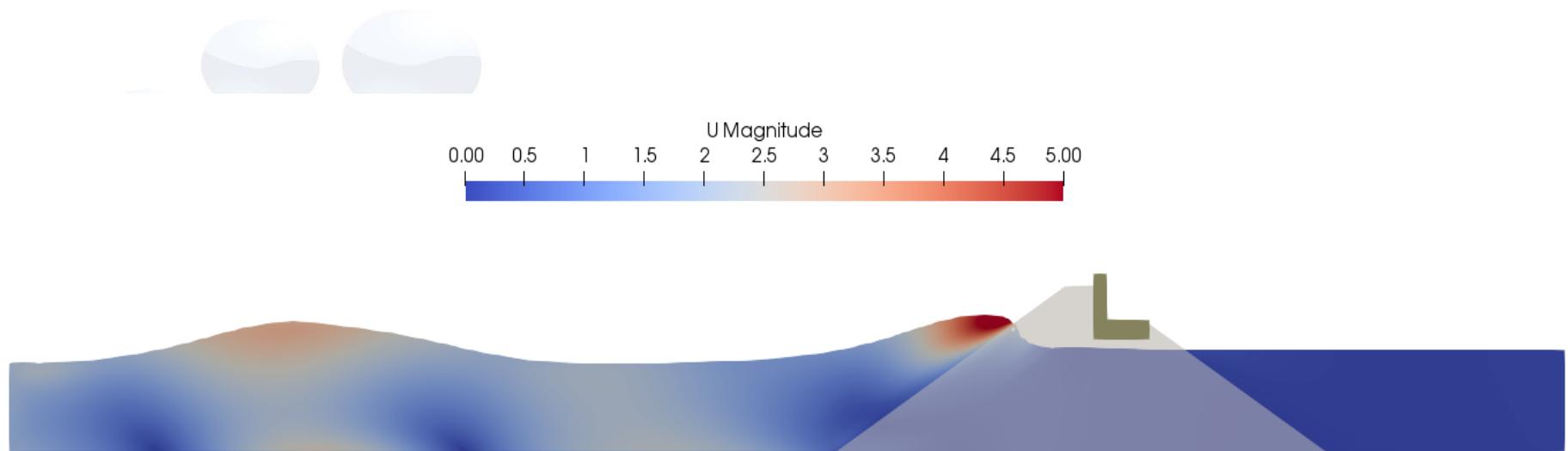


## plot free surface sensors



## Paraview button



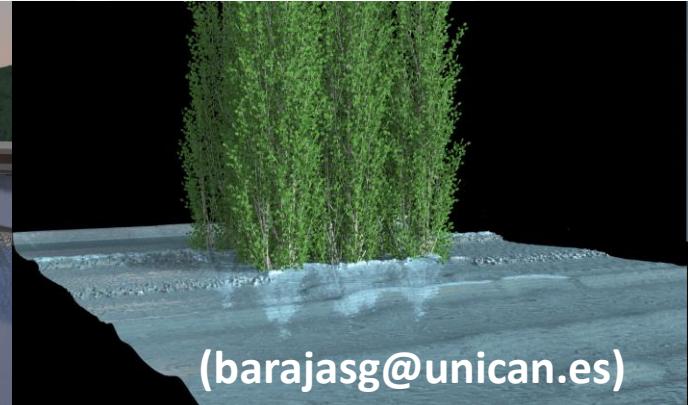
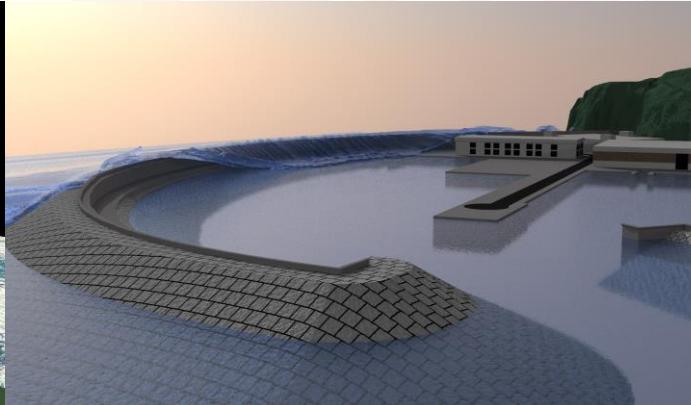


Time: 124.00 s.

UNIVERSIDAD DE CANTABRIA



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



**(barajasg@unican.es)**