



IHCantabria

UNIVERSIDAD DE CANTABRIA

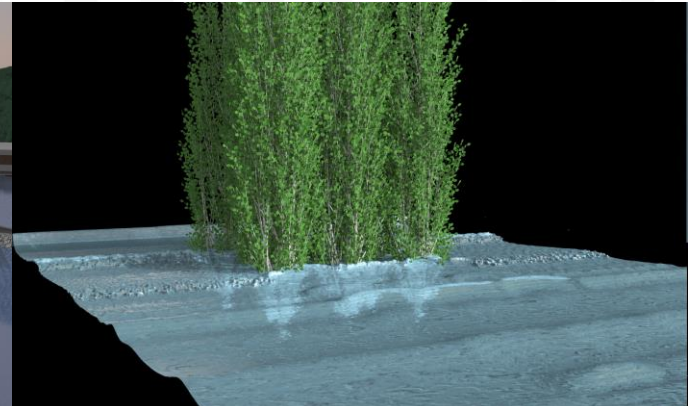
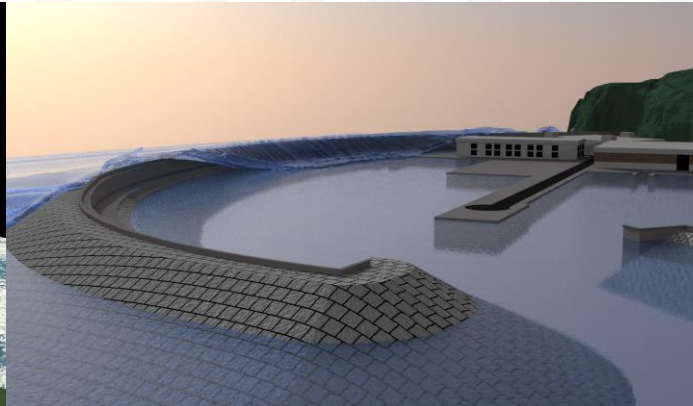
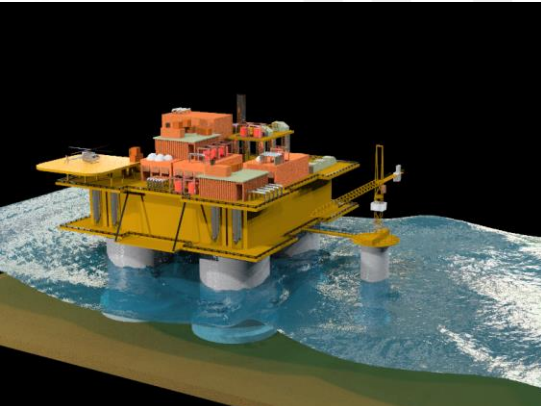
R+D+i for a Sustainable Development

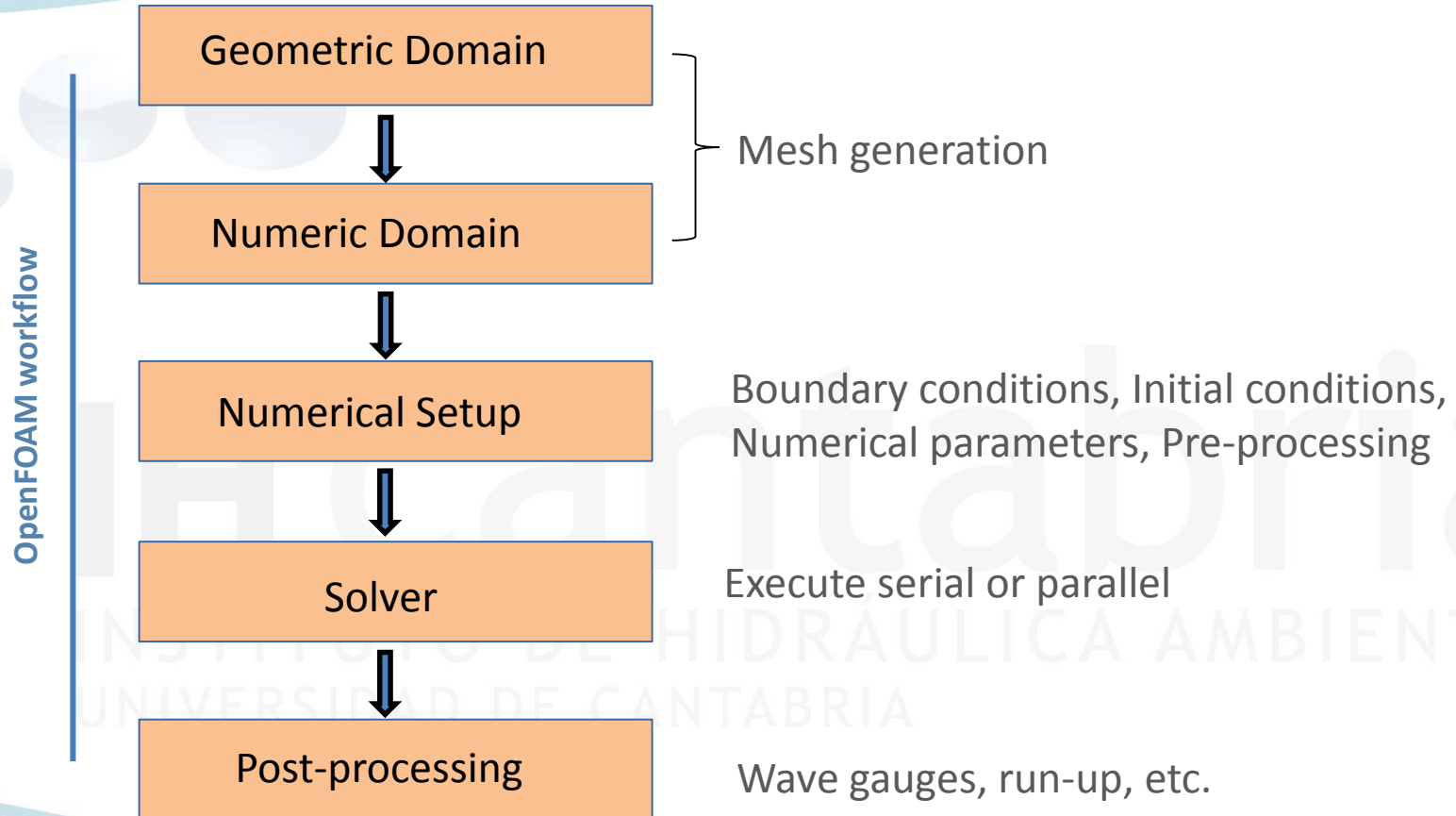
(IHFOAM GUI)

IHFOAM applied to Coastal Engineering

Regular waves in empty channel (2D)

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





OpenFOAM case

0

- alpha.water
- p_rgh
- U

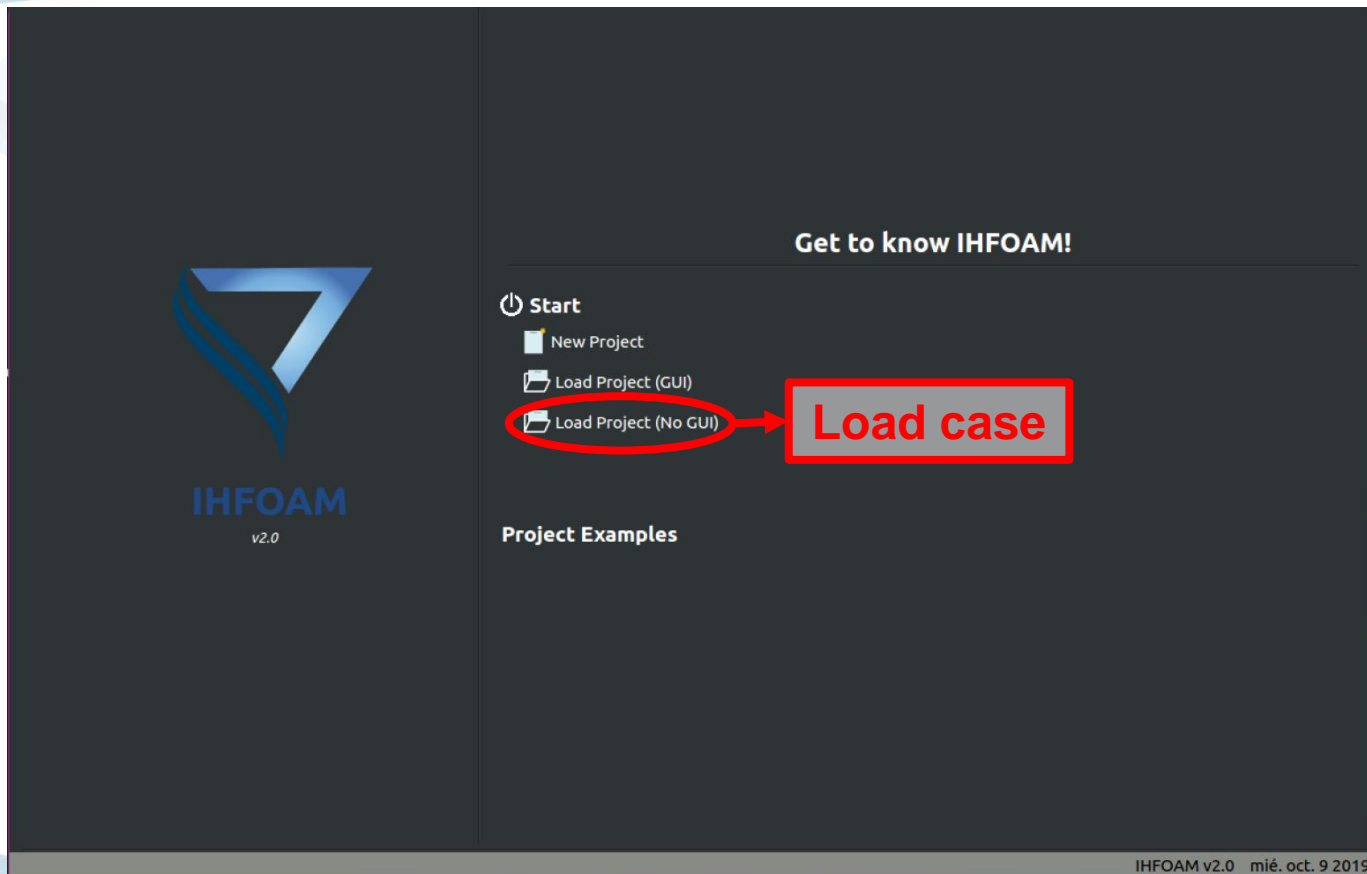
constant

- g
- transportProperties
- turbulenceProperties
- waveProperties

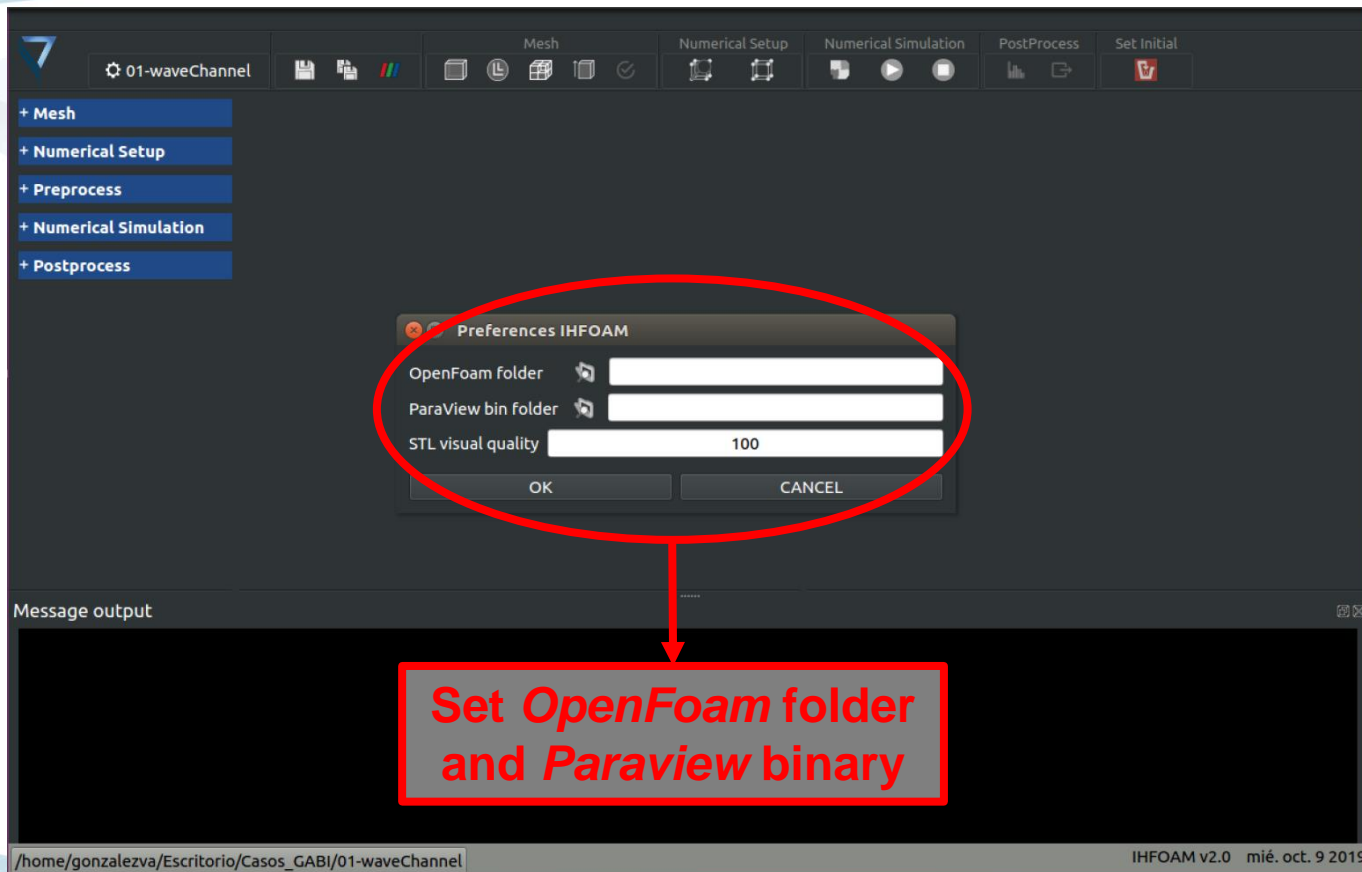
system

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

IHFOAM GUI

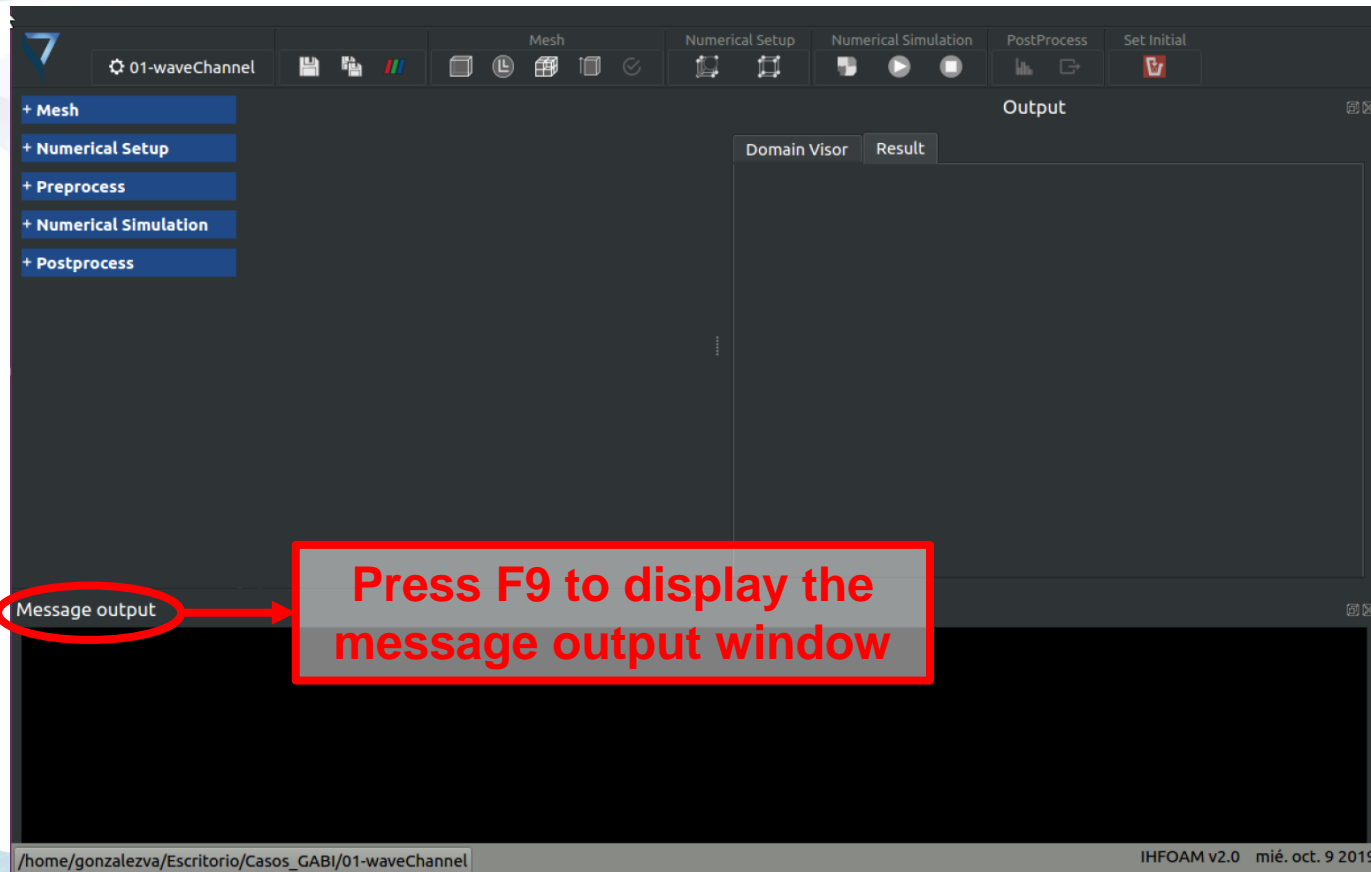


ia
ENTAL



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

ia
ENTAL



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

**Regular waves in empty
channel (2D)**

BlockMesh Attributes

Name	Units	Value
Space Mesh		2D
Type Mesh		static
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	0
Xmax	m	30
Ymin	m	0
Ymax	m	0.01
Zmin	m	0
Zmax	m	0.7
Nx		750
Ny		1
Nz		70

**Define geometric domain
and mesh discretization**

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL

blockMesh button

The screenshot shows the IHFOAM v2.0 software interface. The top toolbar contains several icons, with the 'BlockMesh' icon (a cube) highlighted by a red circle and an arrow pointing to the 'blockMesh button' label. The left sidebar shows a tree view with 'Mesh' expanded, containing 'BlockMesh' and 'SnappyHexMesh'. The 'BlockMesh' sub-menu is open, showing options like 'Import Geometry', 'Refinement', 'Castellated Controls', 'ExtrudeMesh', 'Patches', 'Numerical Setup', 'Preprocess', 'Numerical Simulation', and 'Postprocess'. The main window displays the 'BlockMesh Attributes' table, which includes fields for 'Name', 'Units', and 'Value'. The 'Space Mesh' field is set to '2D', and the 'Type Mesh' field is set to 'static'. Below the table, there is a red circle highlighting an 'IHFOAM' dialog box with an information icon and the text 'OK!! rm -rf constant/polyMesh/ && blockMesh End'. The bottom of the interface shows a 'Message output' window with text indicating the meshing process: 'patch 0 (start: 104180 size: 70) name: inlet', 'patch 1 (start: 104250 size: 70) name: outlet', 'patch 2 (start: 104320 size: 750) name: ground', 'patch 3 (start: 105070 size: 750) name: top', 'patch 4 (start: 105820 size: 105000) name: sides', and 'End'. The status bar at the bottom shows the file path '/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel' and the version 'IHFOAM v2.0' with the date 'mié. oct. 9 2019'.

Name	Units	Value
Space Mesh		2D
Type Mesh		static
convertToMeters		
Mesh dynamic		
Mesh main		
Xmin	m	0
Xmax	m	30
Ymin	m	0
Ymax	m	10
Zmin		
Zmax		
Nx		
Ny		
Nz		

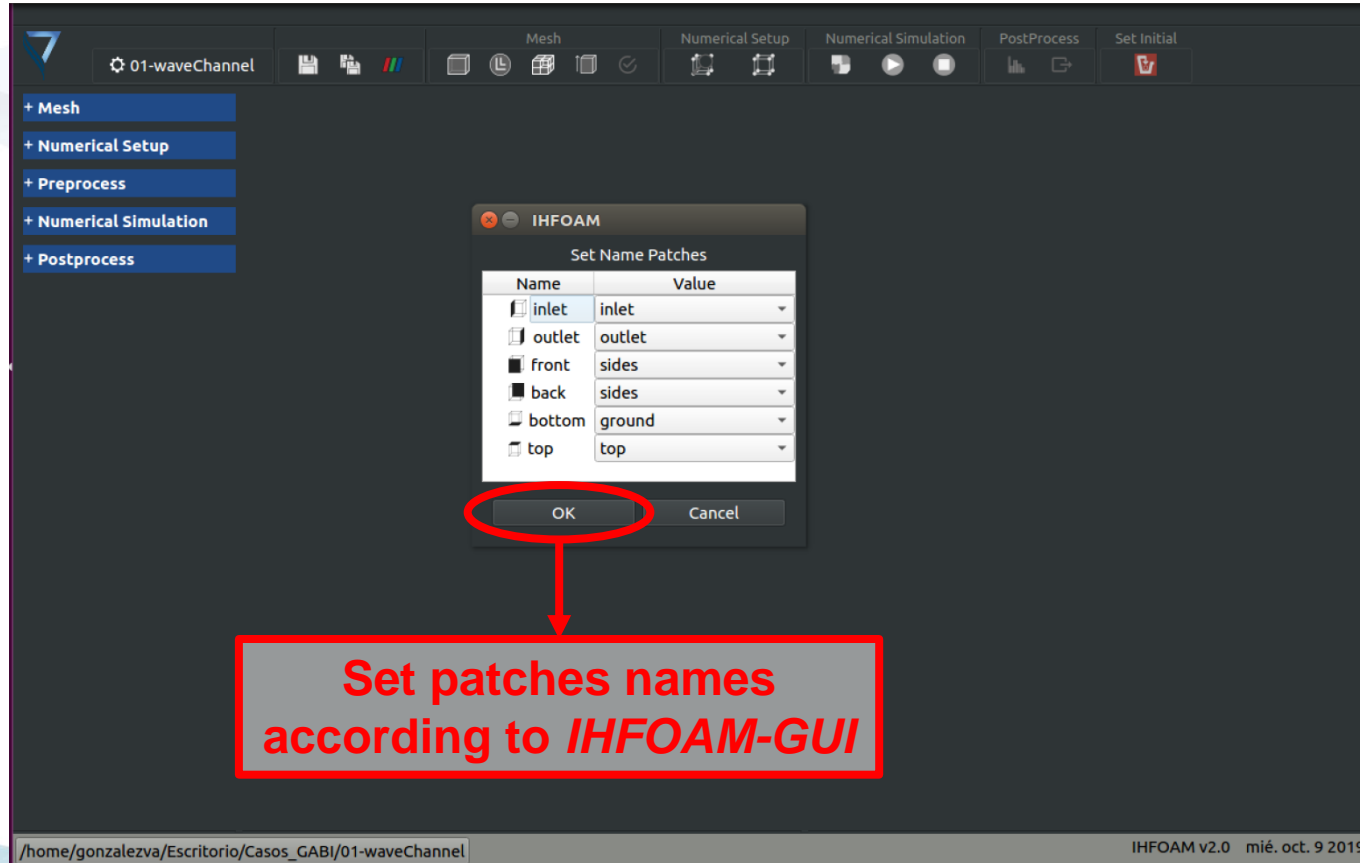
OK!! rm -rf constant/polyMesh/ && blockMesh End

Message output

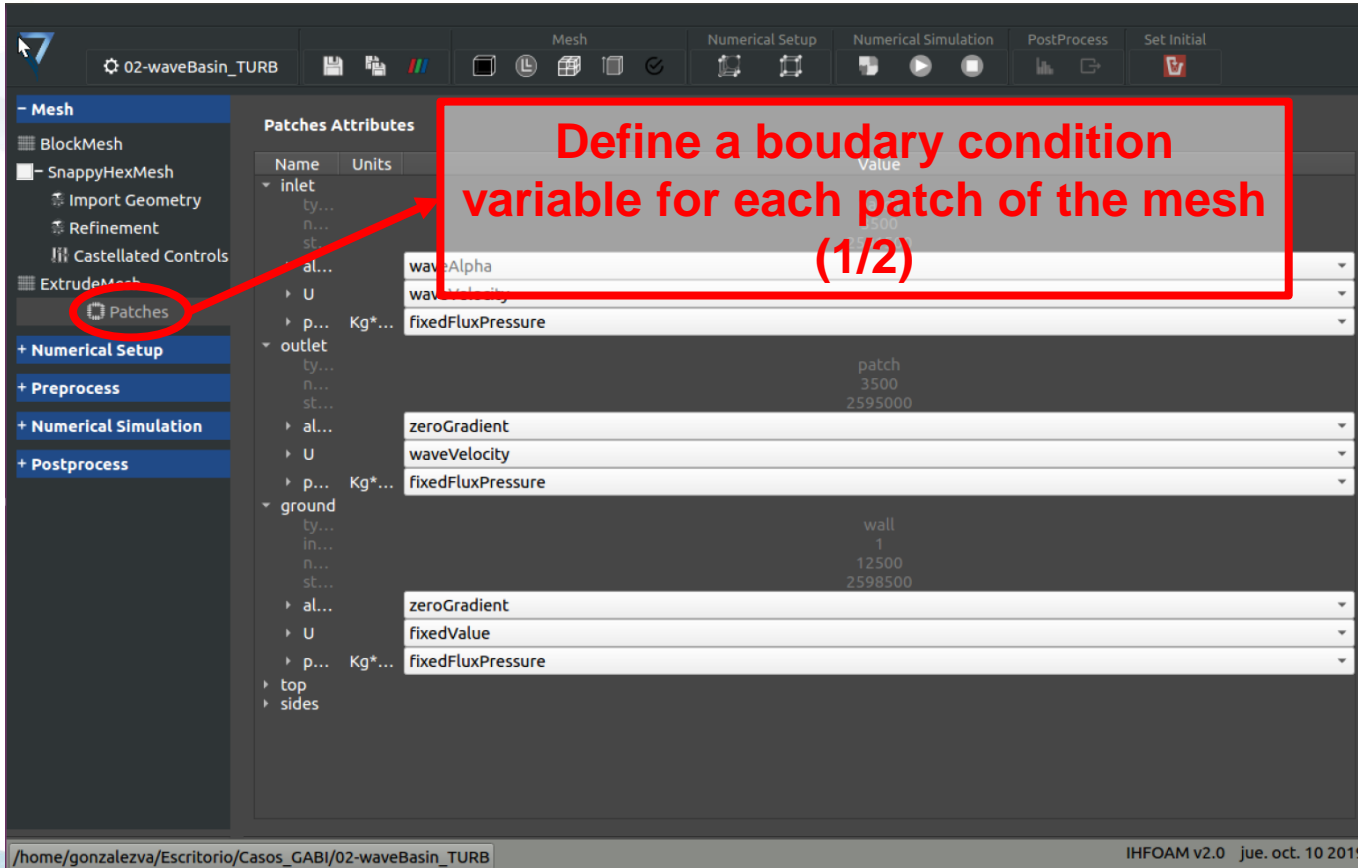
```
patch 0 (start: 104180 size: 70) name: inlet
patch 1 (start: 104250 size: 70) name: outlet
patch 2 (start: 104320 size: 750) name: ground
patch 3 (start: 105070 size: 750) name: top
patch 4 (start: 105820 size: 105000) name: sides
End
```

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel

IHFOAM v2.0 mié. oct. 9 2019



ia
ENTAL



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

Name	Units	Value
inlet		
al...		waveAlpha
U		waveVelocity
p...	Kg...	fixedFluxPressure
outlet		
al...		zeroGradient
U		waveVelocity
p...	Kg...	fixedFluxPressure
ground		
al...		zeroGradient
U		fixedValue
p...	Kg...	fixedFluxPressure
top		
sides		

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

Version: IHFOAM v2.0

Date: jue. oct. 10 2019

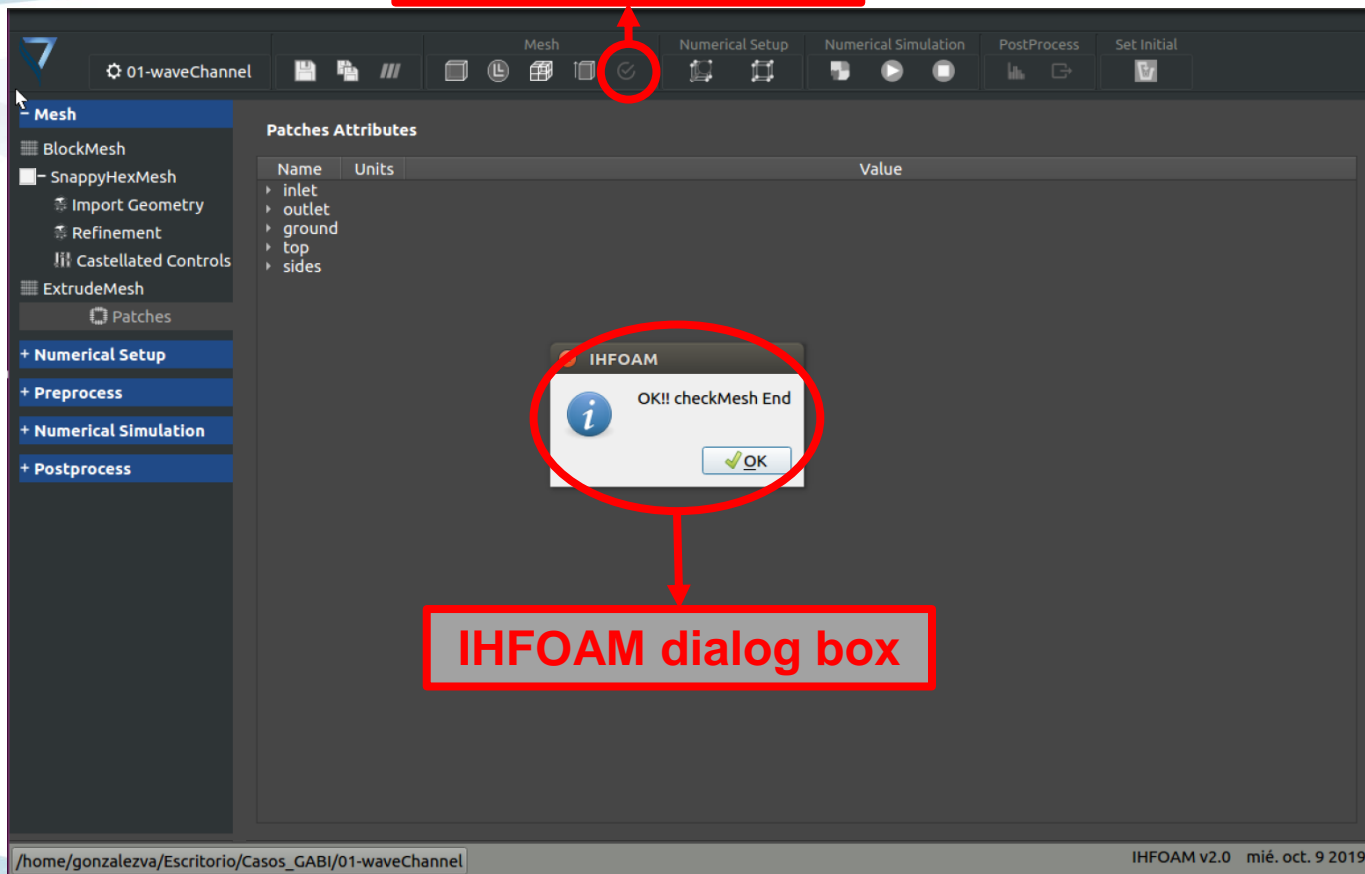
Define a boudary condition variable for each patch of the mesh (2/2)

Name	Units	Value
inlet		
outlet		
ground		
top		
al...		1250
U		20.11000
p...	Kg...	
sides		
ty...		patch
n...		35000
st...		2623500
al...		zeroGradient
U		slip
p...	Kg...	fixedFluxPressure

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

IHFOAM v2.0 jue. oct. 10 2019

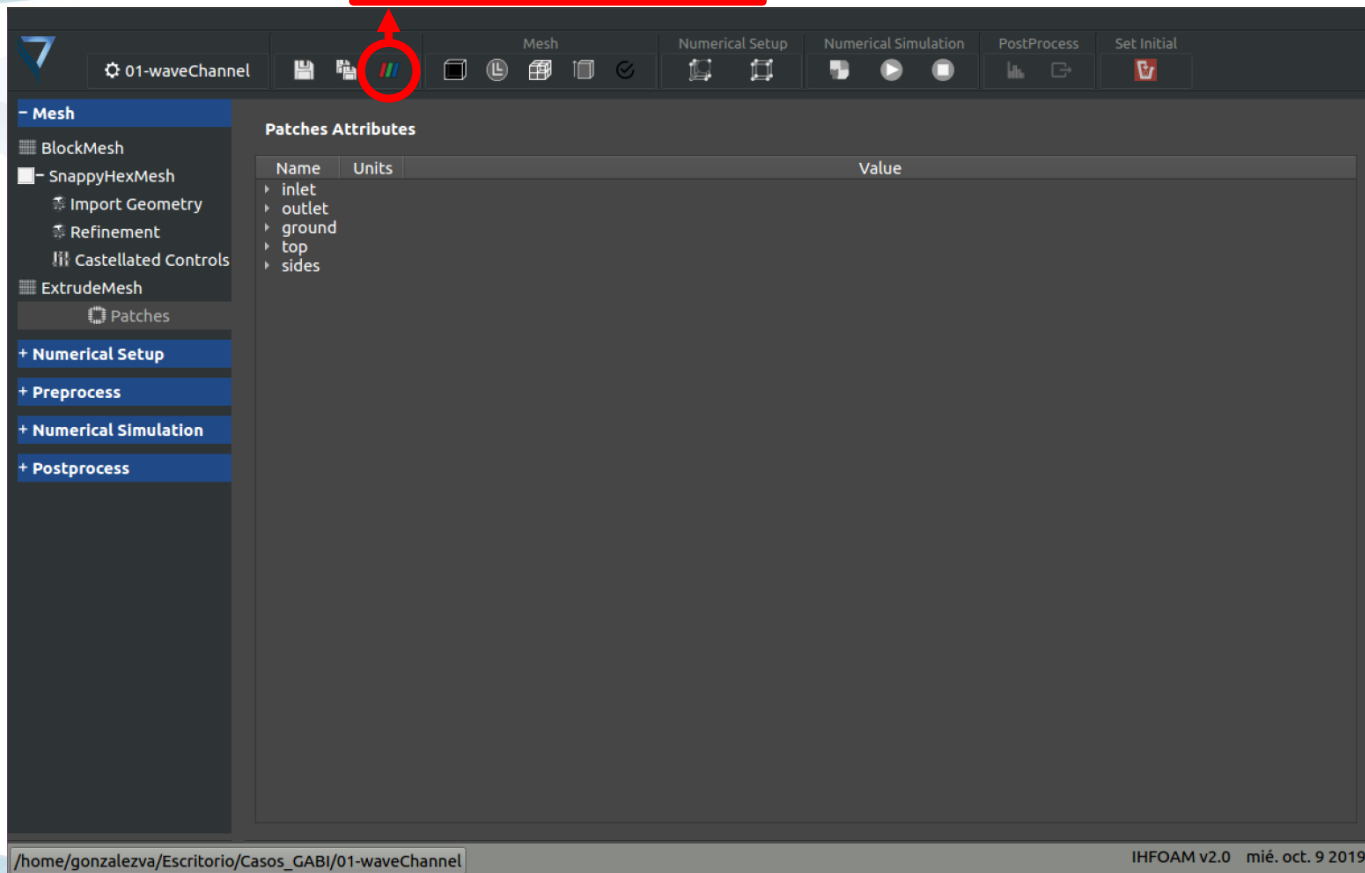
checkMesh button



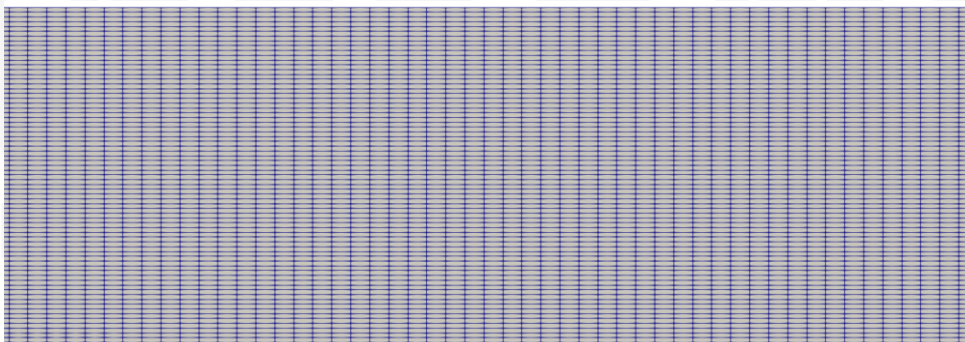
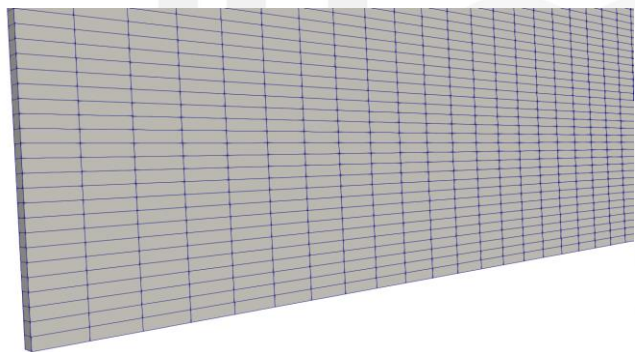
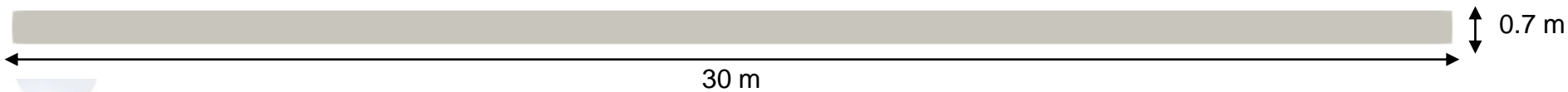
IHFOAM dialog box

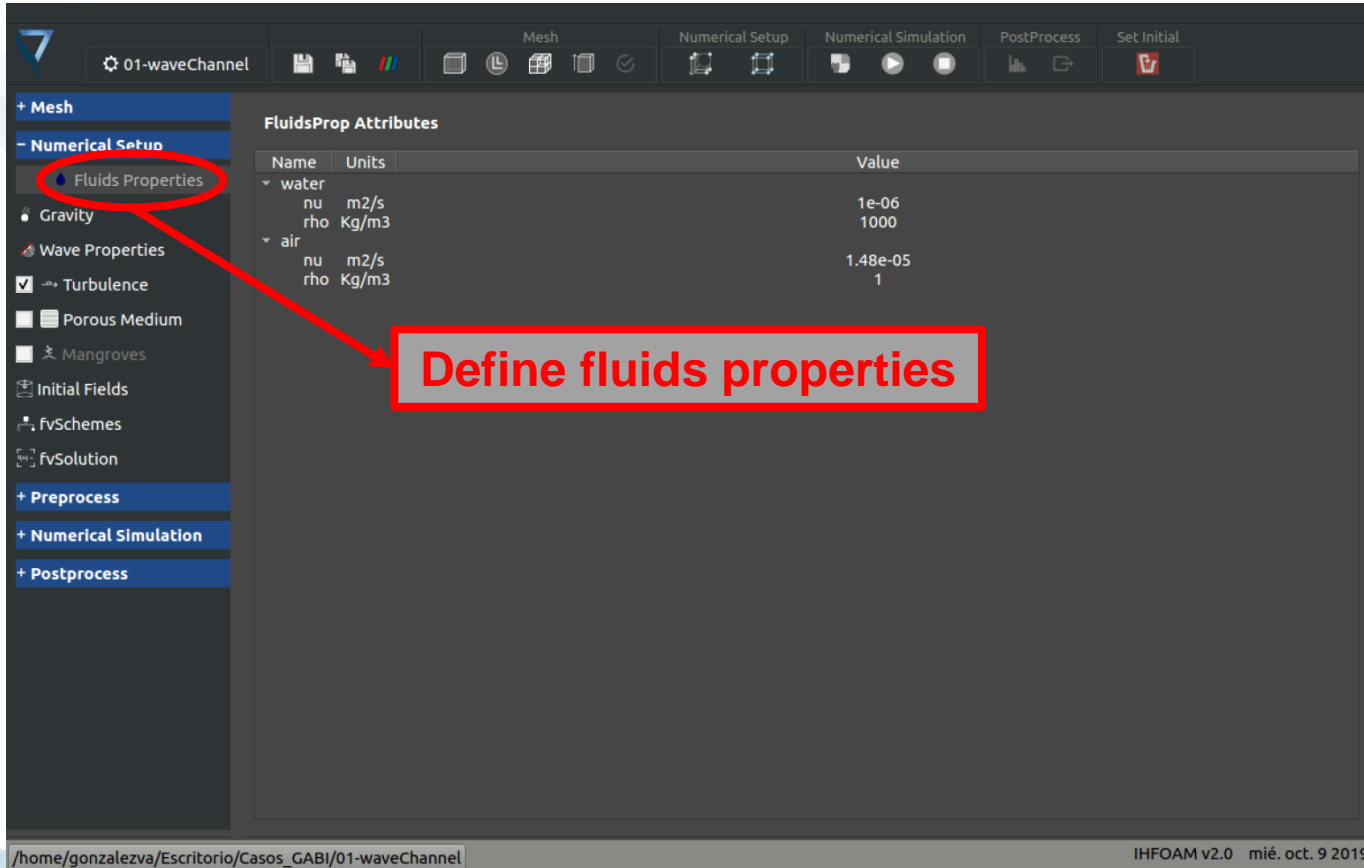
ia
ENTAL

Paraview button



ia
ENTAL





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

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

At the bottom of the window, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel 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 categories: Mesh, Numerical Setup, Wave Properties, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. Under Numerical Setup, 'Gravity' is selected and circled in red. A red arrow points from this selection to a red-bordered button labeled '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

At the bottom of the window, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel and the version/date as IHFOAM v2.0 mié. oct. 9 2019.

WaveProp Attributes

Name	Units	Value
WaveTheoryRange		
WaveTheoryRange		<input checked="" type="checkbox"/>
waveGeneration		inlet
waveModel		cnoidal
wavePeriod	s	2
waveHeight	m	0.1
waveAngle	degree	0
nPaddle		1
activeAbsorption		<input checked="" type="checkbox"/>
rampTime	s	2
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)

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel

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, Fluids Properties, Gravity, Wave Properties, ☒ Turbulence, Porous Medium, Mangroves, Initial Fields (highlighted with a red circle), 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	30
box yMin	m	0
box yMax	m	1
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' button, with a red arrow pointing from the button to the box.

At the bottom of the interface, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel, and the version and date are IHFOAM v2.0 mié. oct. 9 2019.

ia
ENTAL

setFields button

The screenshot displays the IHFOAM software interface. The top toolbar includes buttons for Mesh, Numerical Setup (highlighted with a red circle and an arrow pointing to the 'setFields' button), Numerical Simulation, PostProcess, and Set Initial. The left sidebar shows a tree view with categories like Mesh, Numerical Setup, Fluids Properties, Wave Properties, Turbulence, Porous Medium, Mangroves, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The main panel shows the 'InitFields Attributes' table.

Name	Units	Value
water		
box xMin	m	0
box xMax	m	30
box yMin	m	0
box yMax	m	1
box zMin	m	0
box zMax	m	0.4
alpha.water		1

A red circle highlights a terminal window titled 'IHFOAM' with the message: 'OK!! rm -rf 0 && cp -r 0.orig 0 && setFields End'. An arrow points from this window to the 'IHFOAM dialog box' label.

IHFOAM dialog box

The bottom section shows the 'Message output' terminal with the following text:

```
Setting field default values
Setting internal values of volScalarField alpha.water

Setting field region values
Adding cells with centre within boxes 1((0 0 0) (30 1 0.4))
Setting internal values of volScalarField alpha.water

End
```

The status bar at the bottom indicates the file path: /home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel and the version: IHFOAM v2.0 mié. oct. 9 2019.

The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view with categories: Mesh, Numerical Setup, Initial Fields, fvSchemes, and fvSolution. Under Numerical Setup, 'Turbulence' is checked. The 'InitFields Attributes' panel is open, showing a table of attributes for the 'water' domain.

Name	Units	Value
box xMin	m	0
box xMax	m	30
box yMin	m	0
box yMax	m	1
box zMin	m	0
box zMax	m	0.4
alpha.water		1

The 'Domain Visor' button is highlighted with a red circle. A red box with the text 'Press F10 to display the domain visor' points to it. The main window displays a 3D visualization of the wave channel, showing the water surface and the channel boundaries.

Output

Domain Visor

Press F10 to display the domain visor

Z (m)

Y (m)

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel

IHFOAM v2.0 mié. oct. 9 2019

Paraview button

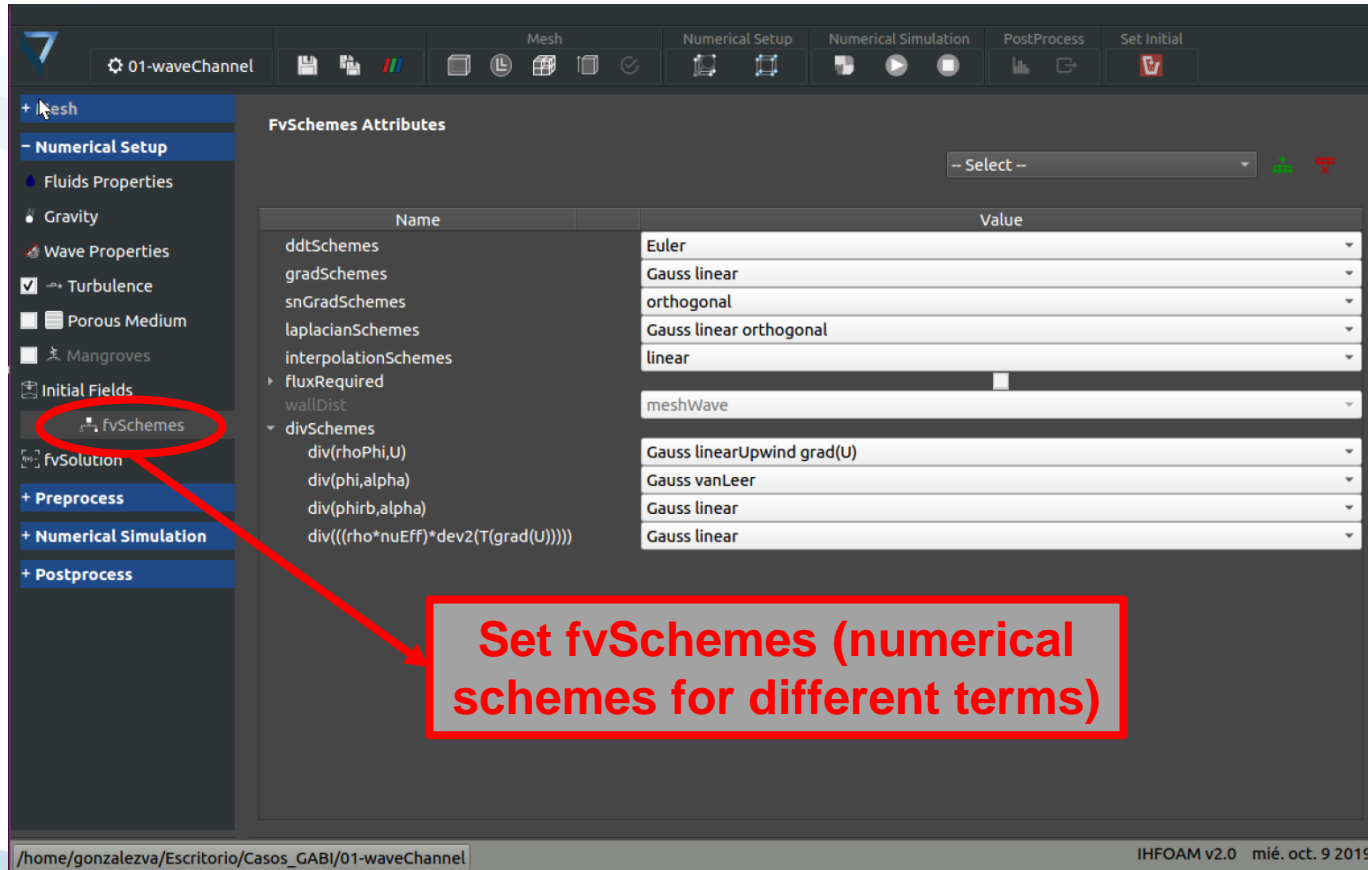
The screenshot shows the OpenFOAM v2.0 graphical user interface. A red circle and arrow highlight the 'Paraview' button in the top toolbar. The left sidebar contains a tree view with categories like 'Numerical Setup', 'Preprocess', 'Numerical Simulation', and 'Postprocess'. The 'Initial Fields' section is expanded, showing a table of attributes for the 'water' region.

Name	Units	Value
water		
box xMin	m	0
box xMax	m	30
box yMin	m	0
box yMax	m	1
box zMin	m	0
box zMax	m	0.4
alpha.water		1

The right panel shows a 3D visualization of the channel flow, with axes labeled 'Z (m)' and 'Y (m)'. The channel is represented by a blue volume, and the flow field is visualized with a color gradient. The status bar at the bottom indicates the file path: '/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel' and the version: 'IHFOAM v2.0'.



INSTITUTO DE HIDRÁULICA AMBIENTAL
UNIVERSIDAD DE CANTABRIA

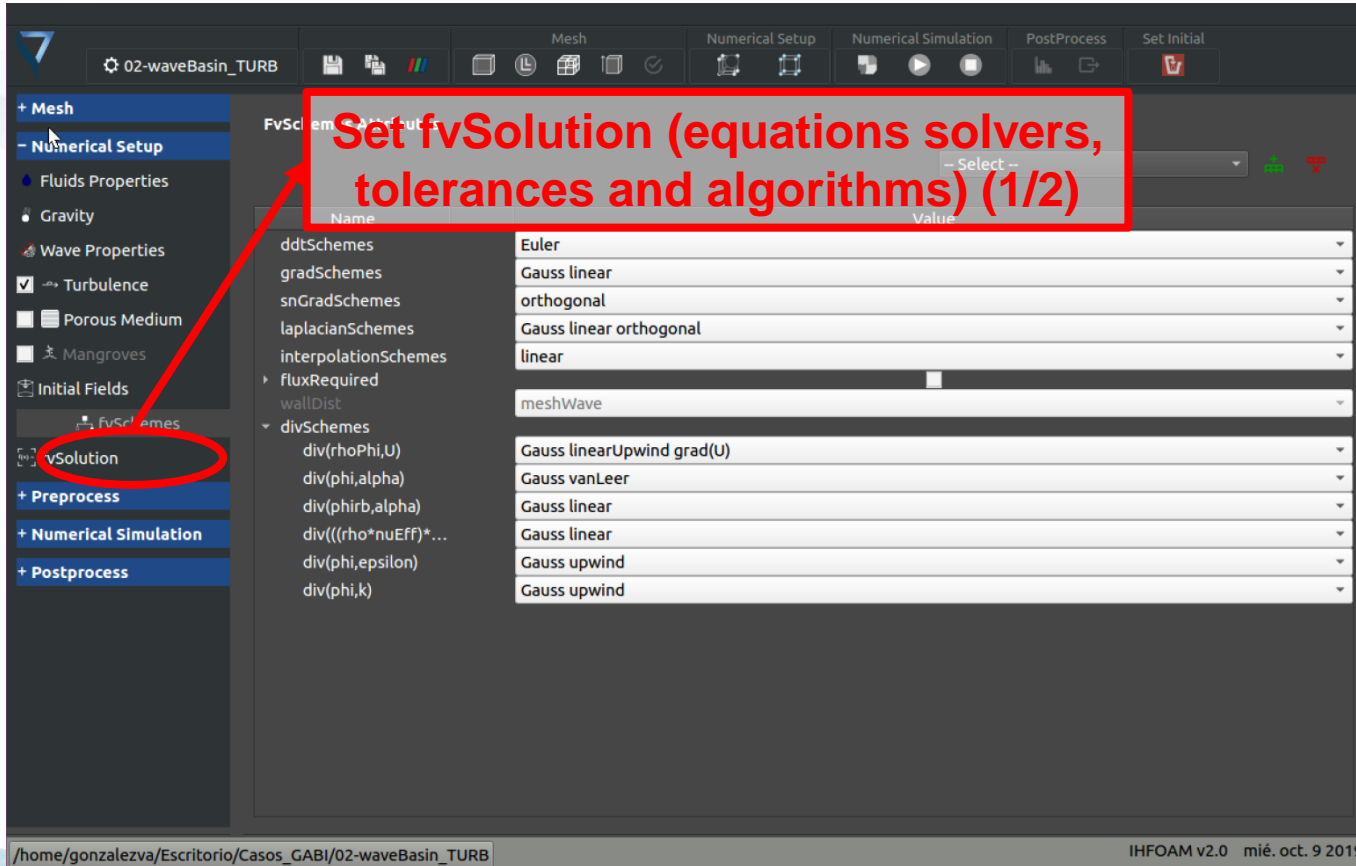


The screenshot shows the OpenFOAM GUI with the '01-waveChannel' case loaded. The 'FvSchemes Attributes' panel is active, displaying a table of numerical schemes for various terms. The 'fvSchemes' option is highlighted in the left sidebar, and a red box with an arrow points to it with the text 'Set fvSchemes (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(phiRb,alpha)	Gauss linear
div(((rho*nuEff)*dev2(T(grad(U))))	Gauss linear

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel

IHFOAM v2.0 mié. oct. 9 2019

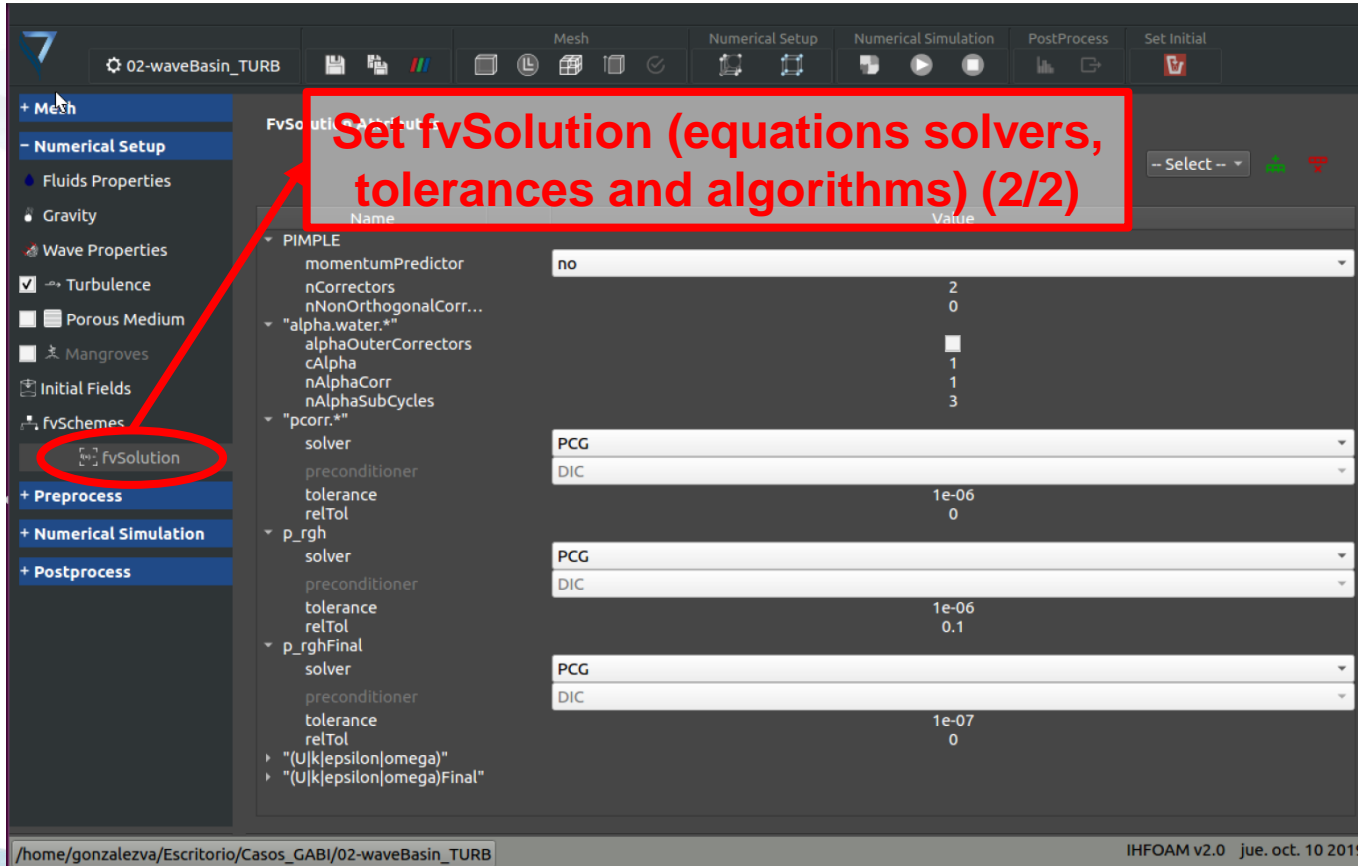


Set fvSolution (equations solvers, tolerances and algorithms) (1/2)

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

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

IHFOAM v2.0 mié. oct. 9 2019



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

Name	Value
PIMPLE	
momentumPredictor	no
nCorrectors	2
nNonOrthogonalCorr...	0
"alpha.water.*"	
alphaOuterCorrectors	
cAlpha	1
nAlphaCorr	1
nAlphaSubCycles	3
"pcorr.*"	
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

01-waveChannel

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	0.005	0	0.7	1001
2	line2	2	0.005	0	0.7	1001
3	line3	3	0.005	0	0.7	1001
4	line4	5	0.005	0	0.7	1001
5	line5	7.5	0.005	0	0.7	1001
6	line6	9	0.005	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

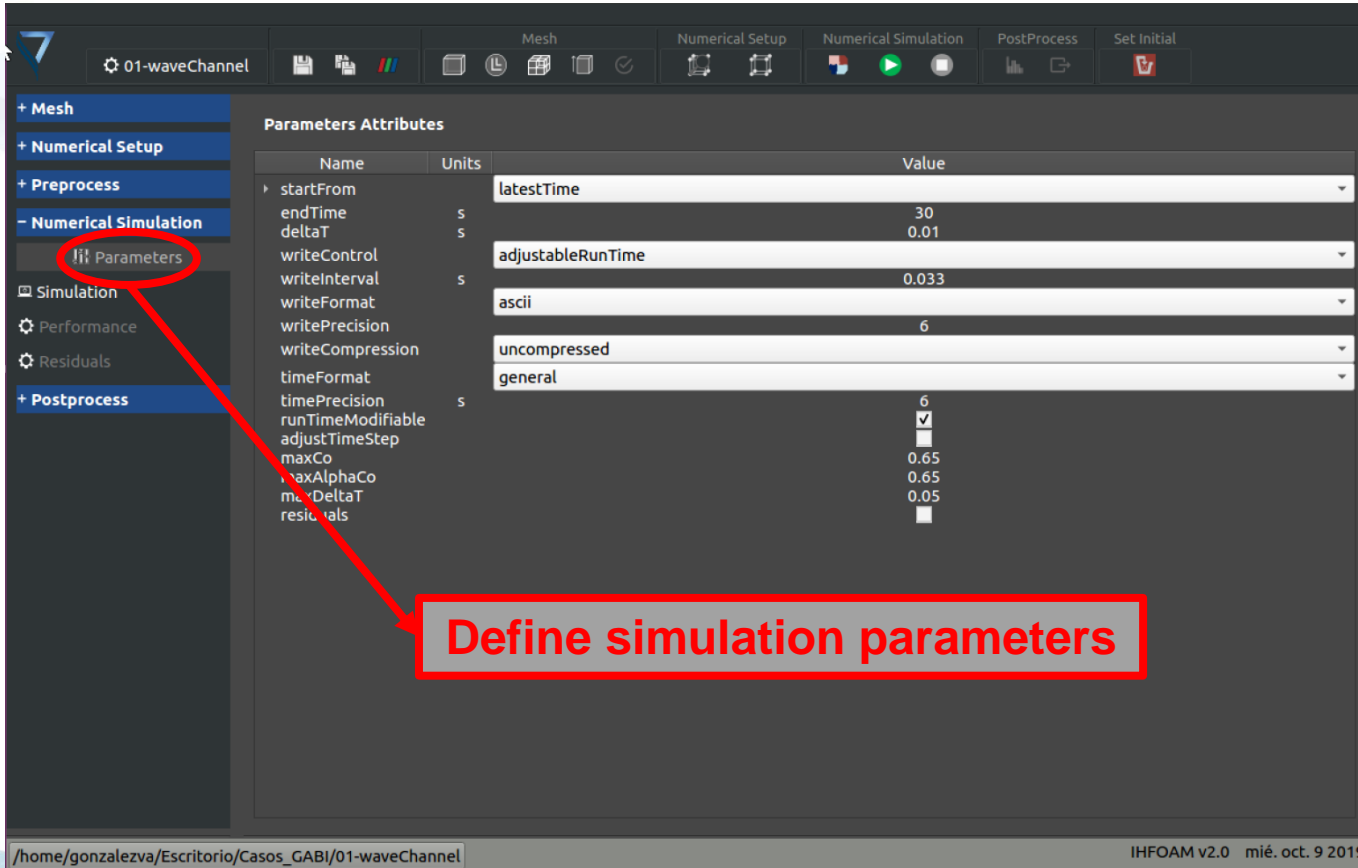
Domain Visor **Result**

Set free-surface gauges

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL



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 window displays the 'Parameters Attributes' table.

Name	Units	Value
startFrom		latestTime
endTime	s	30
deltaT	s	0.01
writeControl		adjustableRunTime
writeInterval	s	0.033
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' button 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/01-waveChannel and the version is IHFOAM v2.0, dated miércoles, 9 oct. 2019.

ia
ENTAL

Run/Execute button

The screenshot displays the IHFOAM v2.0 software interface. The top toolbar contains several icons, with the 'Run/Execute' button (a green play icon) highlighted by a red circle and an arrow pointing to a red box labeled 'Run/Execute button'. The left sidebar shows a tree view with the following items: '+ Mesh', '+ Numerical Setup', '+ Preprocess', '- Numerical Simulation', 'Parameters', 'Simulation' (highlighted with a red circle and an arrow pointing to a red box labeled 'Run OpenFOAM case'), 'Performance', 'Residuals', and '+ Postprocess'. The main panel, titled 'Simulation Attributes', contains a table with the following data:

Name	Units	Value
Numerical Solver		interFoam
Execution Type		Serial

The bottom status bar shows the file path '/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel' and the version 'IHFOAM v2.0' with the date 'mié. oct. 9 2019'.

Run OpenFOAM case

ia
ENTAL

01-waveChannel

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

+ Mesh
+ Numerical Setup
+ Preprocess
- Numerical Simulation
Parameters
Simulation
Performance
Residuals
+ Postprocess

Simulation Attributes

Name	Units	Value
Numerical Solver		interFoam
Execution Type		Serial

Output

Domain Visor Result

OK!! interFoam End

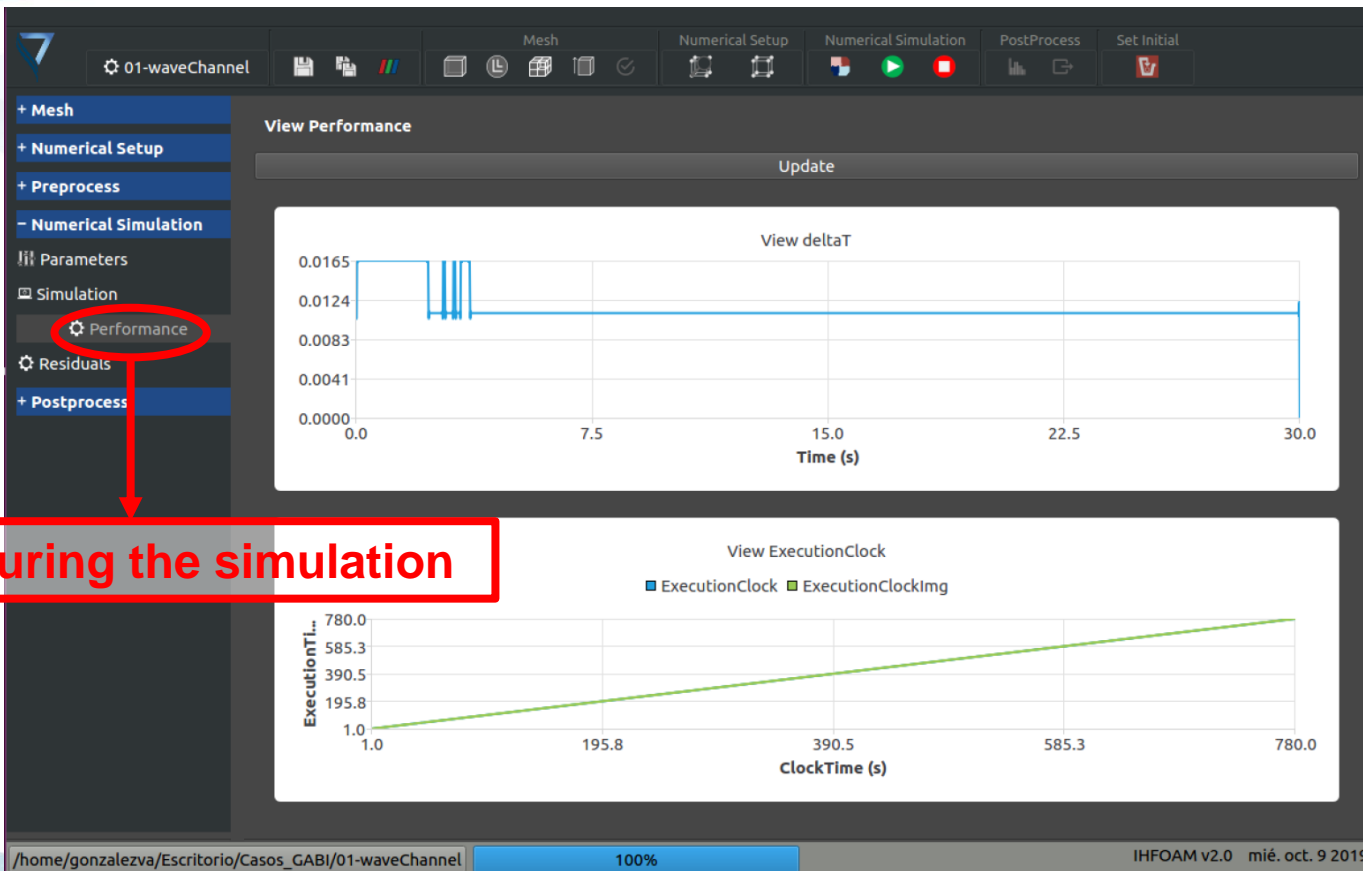
Case correctly executed!!

Message output

Updating shallowWaterAbsorption wave model for patch outlet
DICPCG: Solving for p_rgh, Initial residual = 0.0124302, Final residual = 7.14474e-05, No Iterations 1
time step continuity errors : sum local = 0.816721, global = 1.15409e-05, cumulative = 0.0503279
GAMG: Solving for p_rgh, Initial residual = 7.80906e-05, Final residual = 6.83676e-08, No Iterations 5
time step continuity errors : sum local = 1.11369e-05, global = 8.40828e-06, cumulative = 0.0503363
ExecutionTime = 779.36 s ClockTime = 780 s

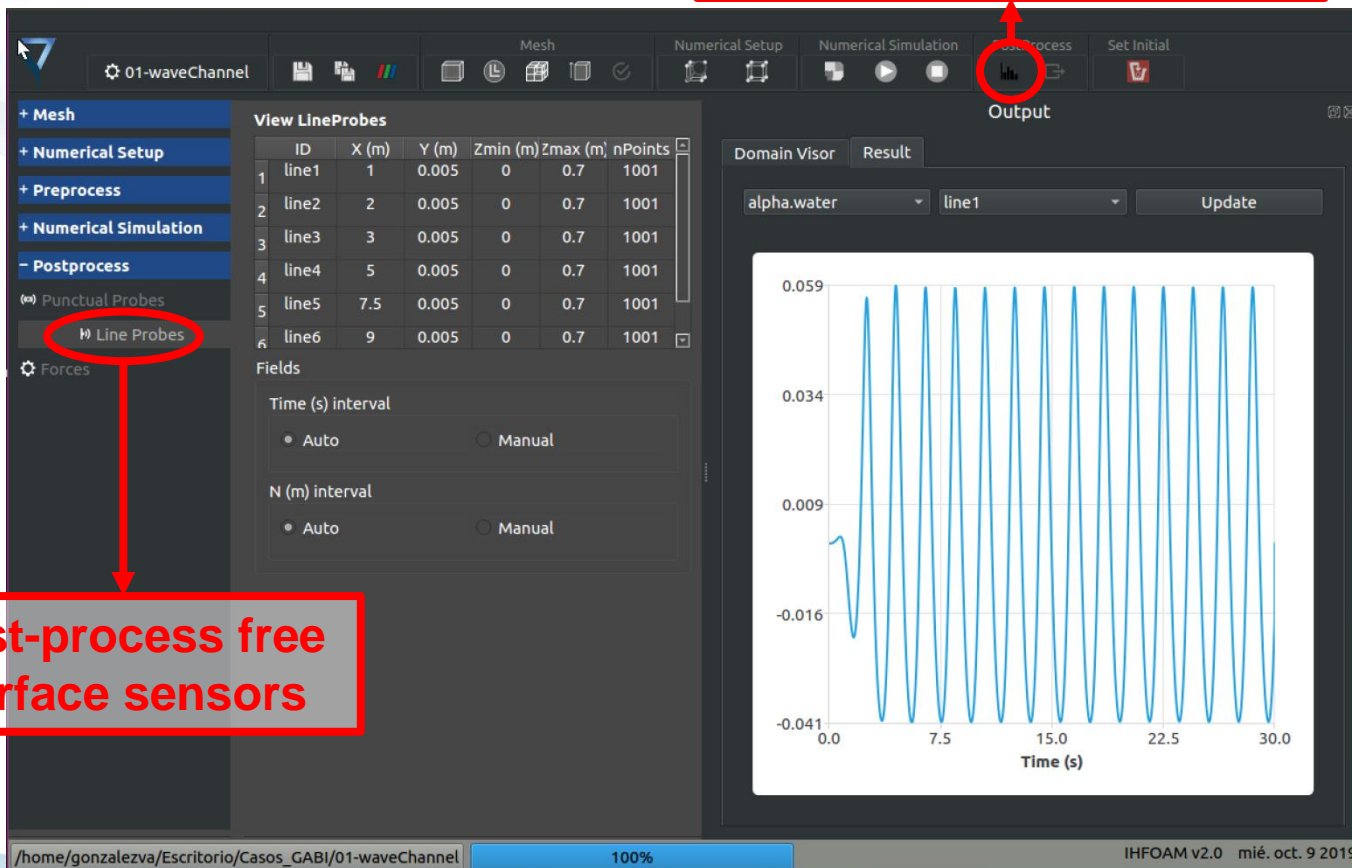
End

/home/gonzalezva/Escritorio/Casos_GABI/01-waveChannel 100% IHFOAM v2.0 mié. oct. 9 2019



plot dt during the simulation

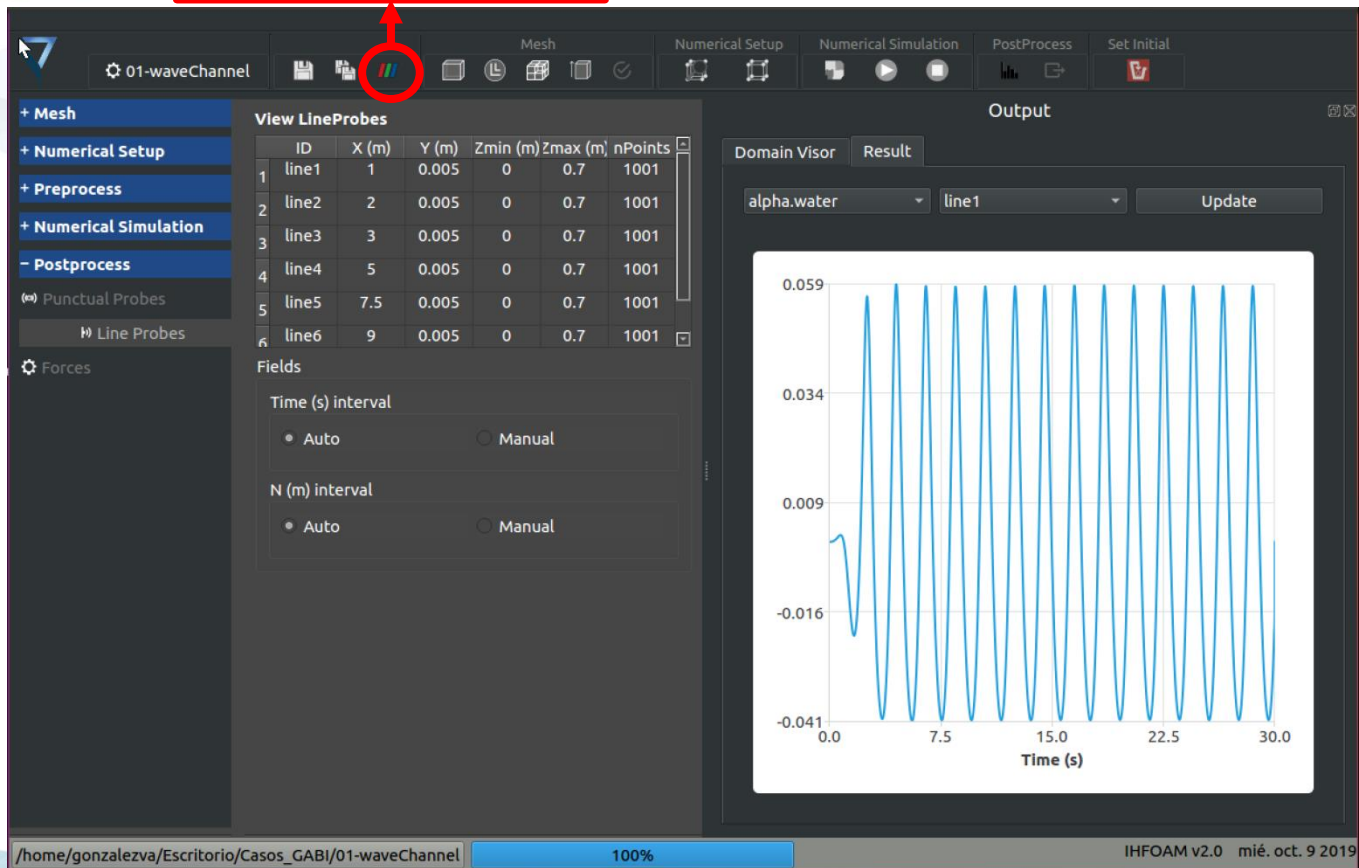
plot free surface sensors



Post-process free
surface sensors

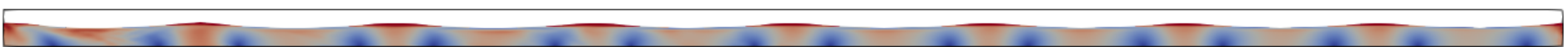
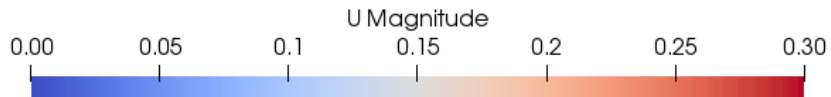
ia
ENTAL

Paraview button





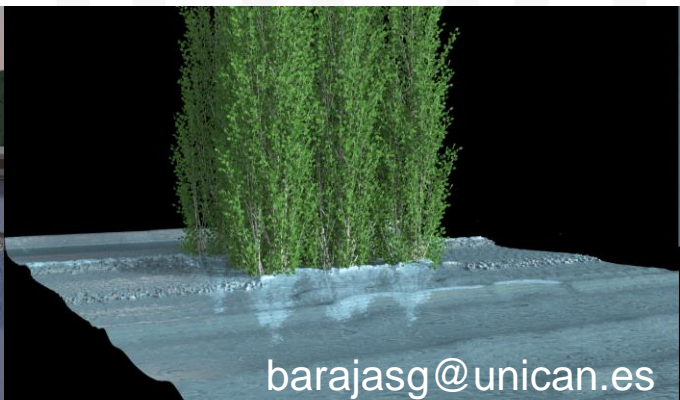
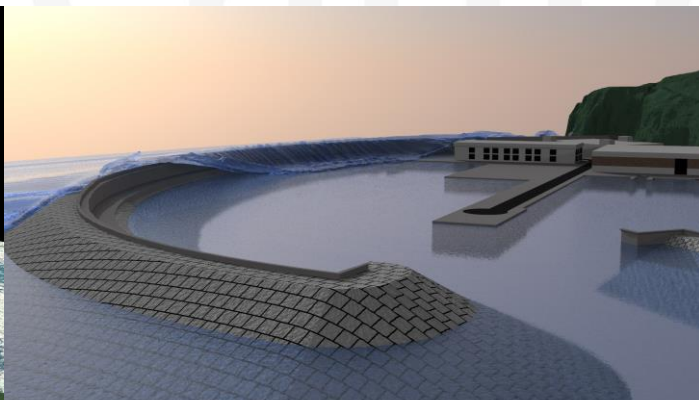
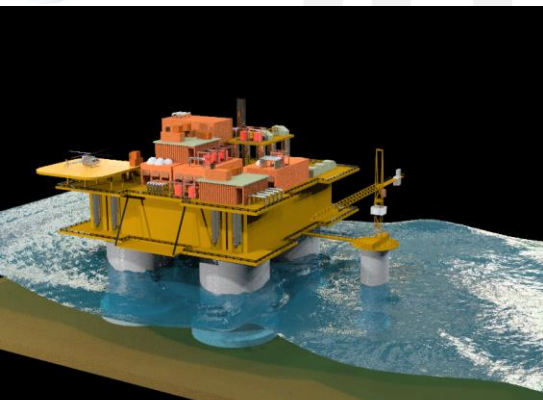
Time: 30.00 s.



Time: 30.00 s.



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



barajasg@unican.es