



38TH
INTERNATIONAL
CONFERENCE
ON COASTAL
ENGINEERING

ROME
2024

8-14
SEPTEMBER



UC

ih
FUNDACION
INSTITUTO DE HIDRÁULICA
AMBIENTAL DE CANTABRIA



IH cantabria
INSTITUTO DE HIDRÁULICA AMBIENTAL
UNIVERSIDAD DE CANTABRIA

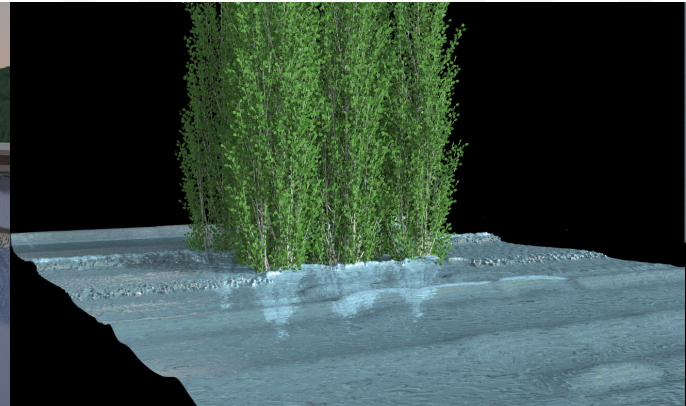
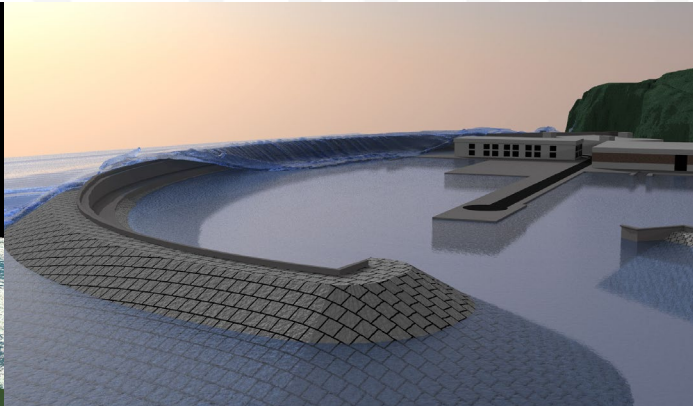
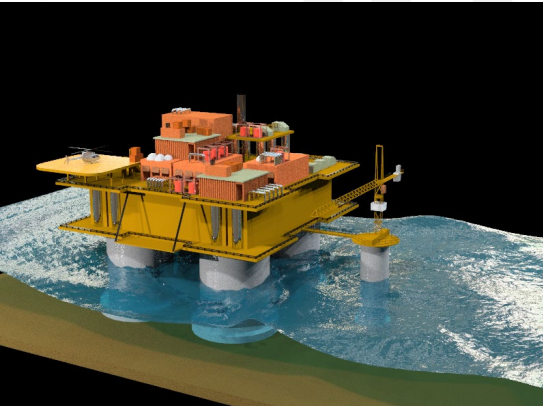
ROMA
TRE
UNIVERSITÀ DEGLI STUDI

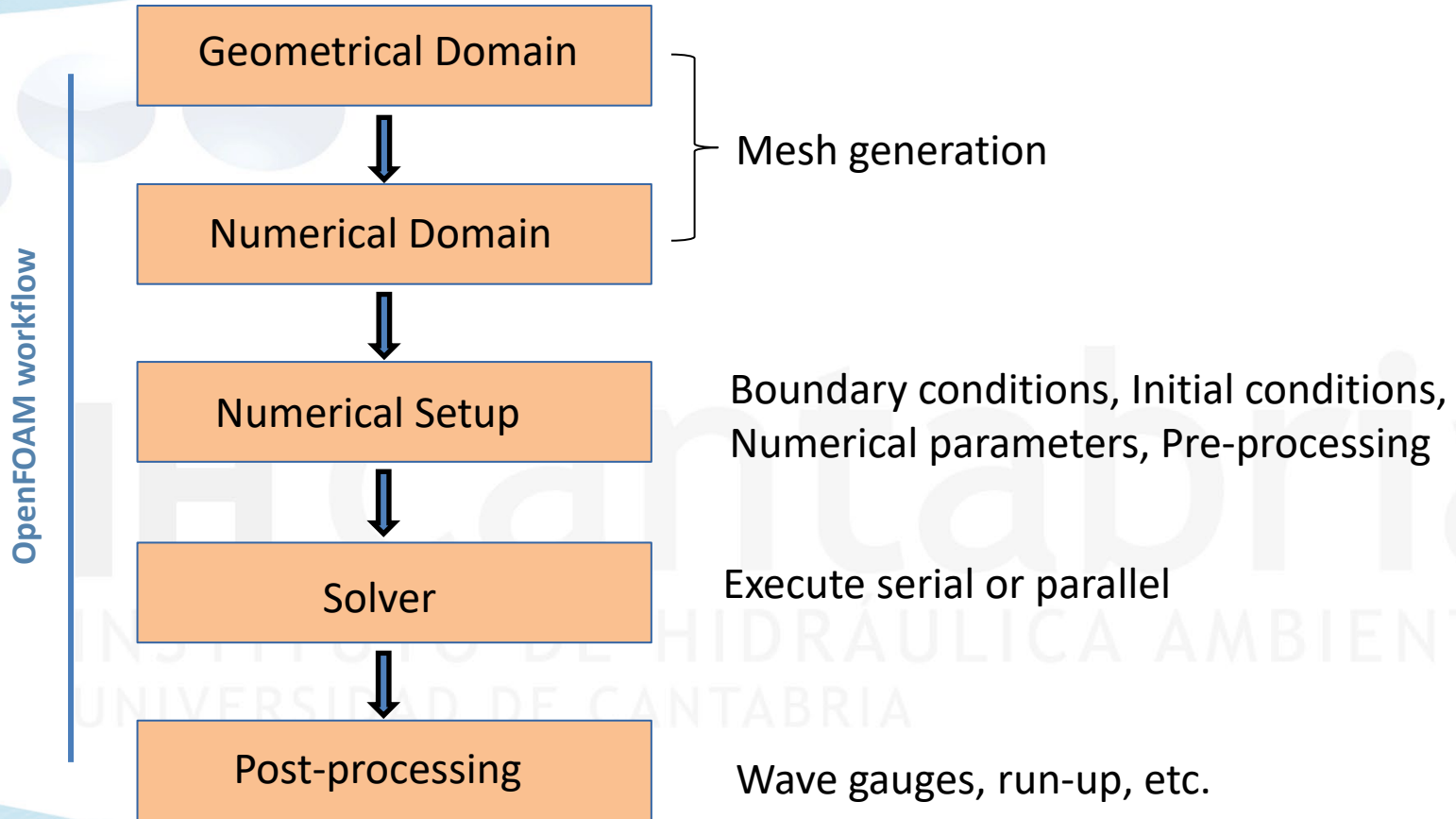


IHFOAM applied to Coastal Engineering

Regular waves in an empty wave flume

Javier L. Lara, Gabriel Barajas, María Maza, Alessandro Romano.





OpenFOAM case

0

- alpha.water
- p_rgh
- U

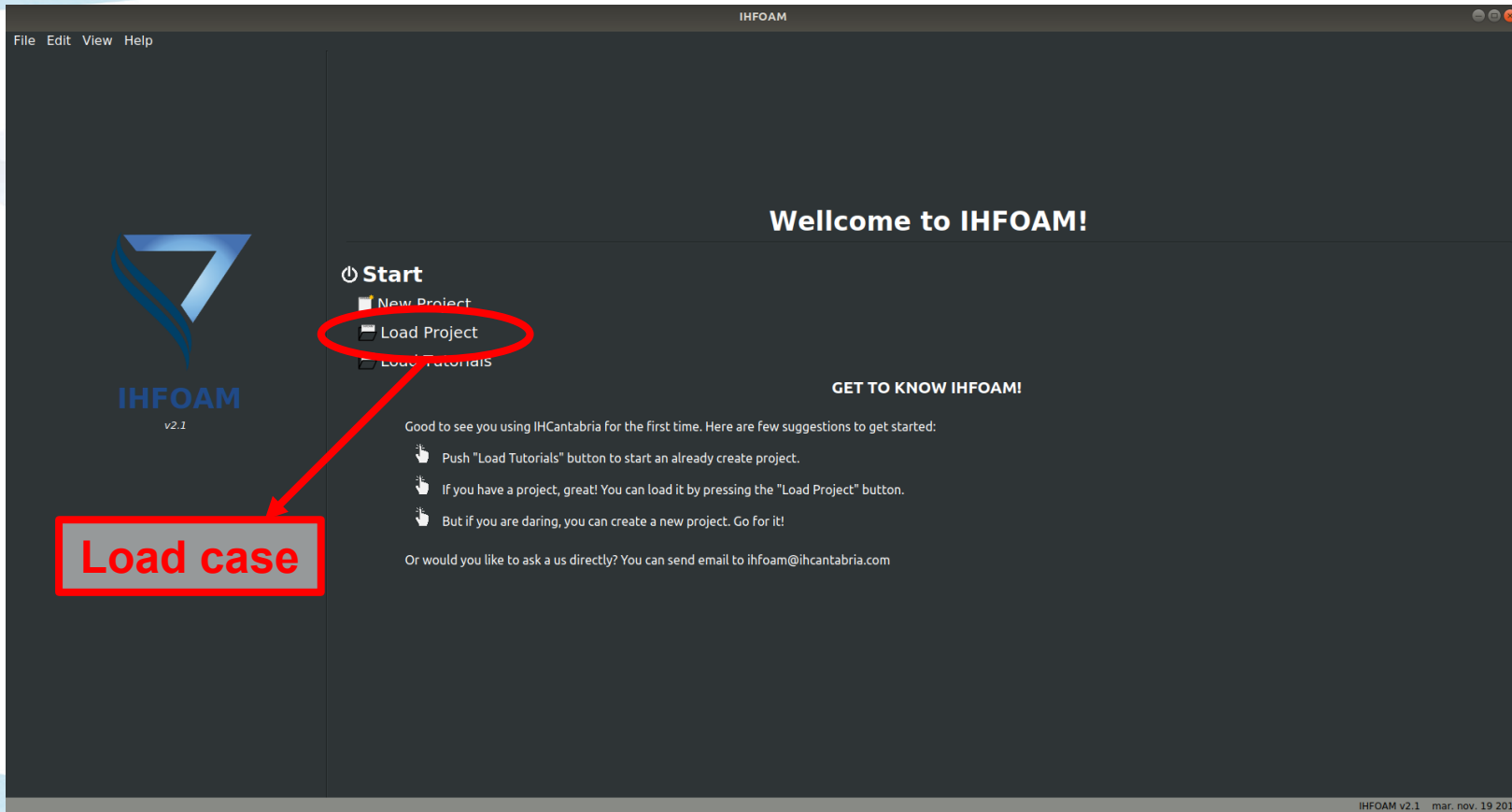
constant

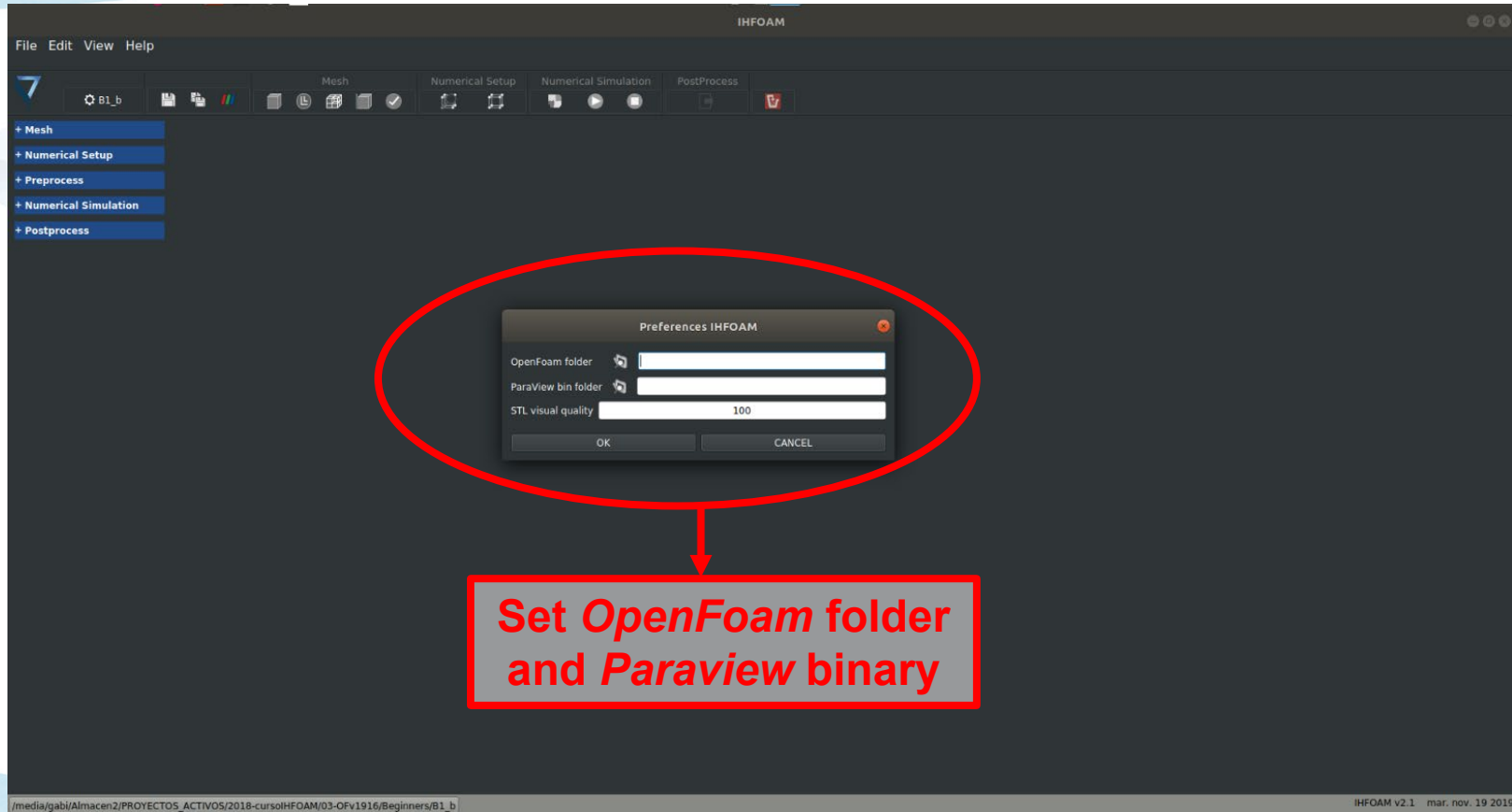
- g
- transportProperties
- turbulenceProperties
- waveProperties

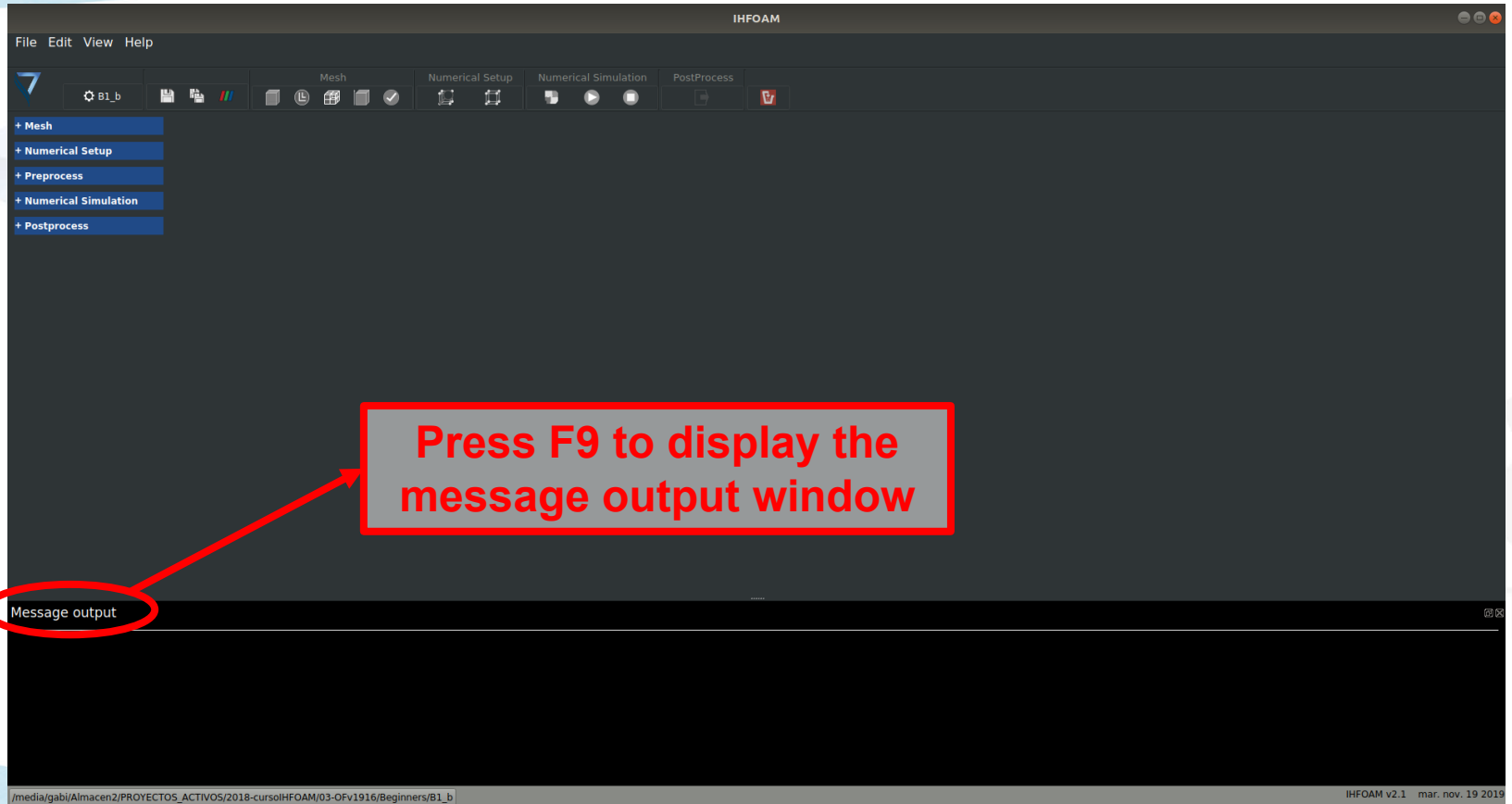
system

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

IHFOAM GUI







IHFOAM

File Edit View Help

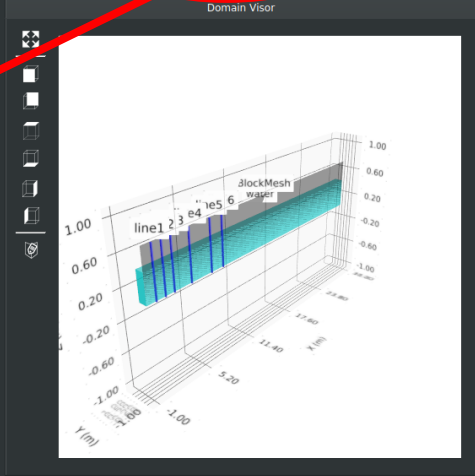
B1_b

Mesh Numerical Setup Numerical Simulation PostProcess

+ Mesh
+ Numerical Setup
+ Preprocess
+ Numerical Simulation
+ Postprocess

Press F10 to display the Domain visor window

Output
Domain Visor



Message output

line1 2 3 e4 e5 6
blockMesh
water

Y (m)

X (m)

Z (m)

Save single pannel modified
or save all pannels modified.

The screenshot displays the IHFOAM software interface. The top menu bar includes File, Edit, View, and Help. Below it is a toolbar with icons for various functions. The left sidebar shows a tree view with categories like Mesh, Numerical Setup, Numerical Simulation, and Postprocess. The 'Mesh' category is expanded, showing options like BlockMesh, Snap, Import Geometry, Refinement, and Castellated Controls. The 'BlockMesh' option is selected, and its attributes are displayed in the main panel.

Name	Units	Value
Space Mesh		2D
Type Mesh		static
convertToMeters		
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

On the right side, the 'Output' panel shows a 'Domain Visor' with a 3D visualization of the wave flume domain. The domain is a rectangular box with a blue mesh. The axes are labeled X (m), Y (m), and Z (m). The Z-axis ranges from -1.00 to 1.00. The X-axis ranges from -1.00 to 1.00. The Y-axis ranges from -1.00 to 1.00. The domain is divided into several regions, with labels like 'line1', 'line2', 'line3', 'line4', 'line5', 'line6', and 'Block-Mesh' visible.

At the bottom, there is a 'Message output' panel.

Define geometric domain
and mesh discretization

blockMesh button

The screenshot shows the IHFOAM software interface. The top menu bar includes File, Edit, View, and Help. Below it is a toolbar with various icons. A red circle highlights the 'blockMesh' button in the toolbar, with an arrow pointing to a red box containing the text 'blockMesh button'. The main window is divided into several panels. On the left is a sidebar with a tree view showing the project structure. The central panel displays the 'BlockMesh Attributes' table. On the right is a 'Domain Visor' showing a 3D visualization of the mesh. At the bottom is a 'Message output' panel showing the results of the blockMesh command.

BlockMesh Attributes

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

IHFOAM dialog box

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

Domain Visor

3D visualization of the mesh showing the domain boundaries and the mesh structure.

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
  
```

Set patches names according to IHFOAM-GUI

Name	Value
inlet	inlet
outlet	outlet
front	sides
back	sides
bottom	ground
top	top

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
  
```

ih cantabria

a
TAL

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

The screenshot shows the IHFOAM software interface. On the left, a sidebar contains a tree view with categories: Mesh, Numerical Setup, Preprocess, Numerical Simulation, and Postprocess. Under the 'Mesh' category, 'Patches' is highlighted with a red circle. A red arrow points from the text box above to this 'Patches' option. The main window displays the 'Patches Attributes' table, which lists boundary conditions for different patches in the mesh.

Name	Units	Value
inlet		
type		patch
nFaces		70
startFace		104180
alpha.water		waveAlpha
value		0
inletValue		0
U		waveVelocity
value		0
p_rgh	Kg*m/s2	fixedFluxPressure
value		0
outlet		
type		patch
nFaces		70
startFace		104250
alpha.water		zeroGradient
value		0
inletValue		0
U		waveVelocity
value		0
p_rgh	Kg*m/s2	fixedFluxPressure
value		0
ground		
type		wall
nGroups		1
nFaces		750
startFace		104320
alpha.water		zeroGradient
value		0
inletValue		0
U		fixedValue
value		0
p_rgh	Kg*m/s2	fixedFluxPressure
value		0
top		
sides		

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

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

Name	Units	Value
inlet		
outlet		
ground		
top		
type		patch
nFaces		750
startFace		105070
alpha.water		inletOutlet
value		0
inletValue		0
U		pressureInletOutletVelocity
value		0
p_rgh		totalPressure
value		0
sides		
type		empty
nFaces		105000
startFace		105820
alpha.water		empty
value		0
inletValue		0
U		empty
value		0
p_rgh		empty
value		0

checkMesh button

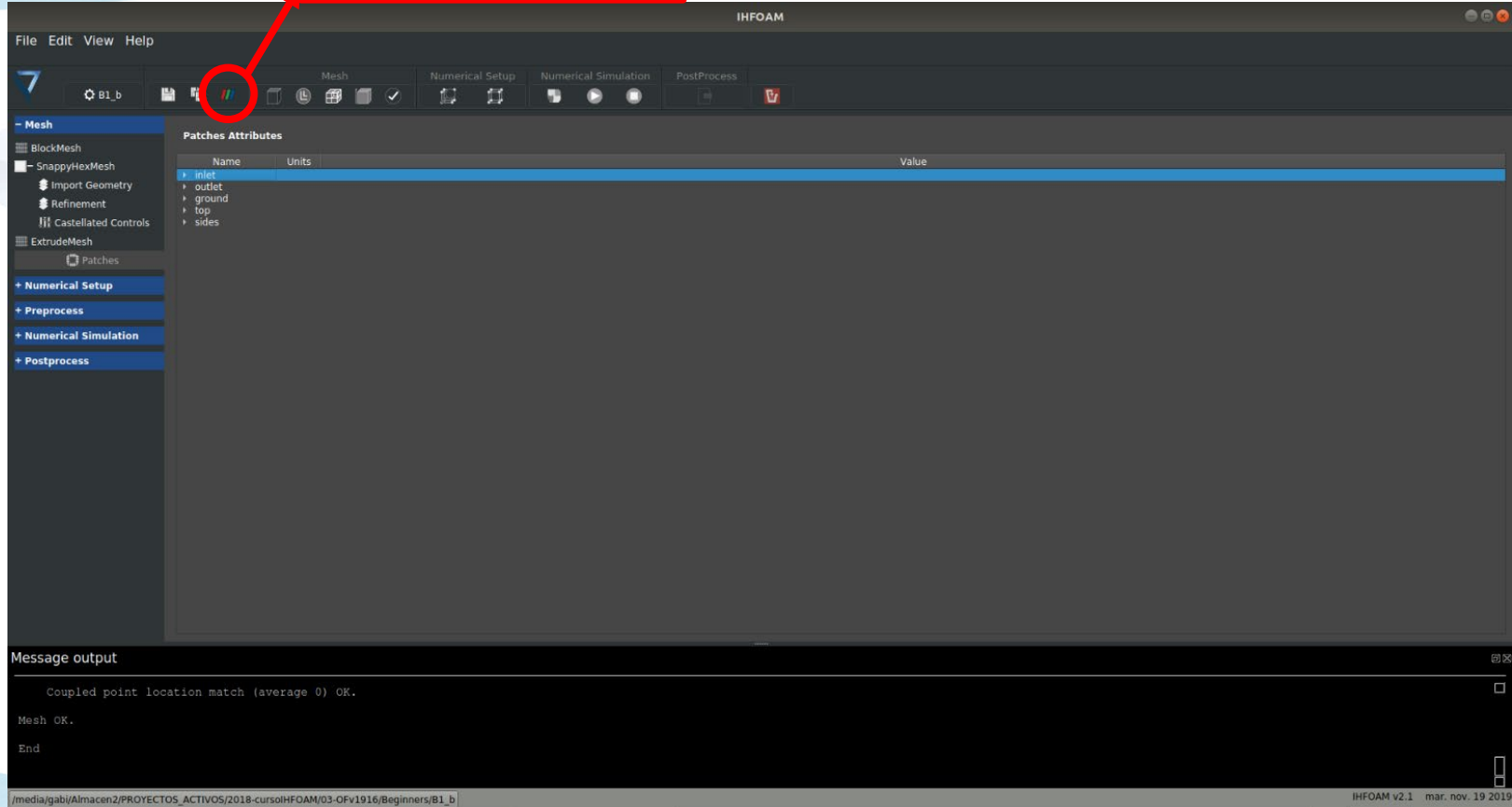
The screenshot displays the IHFOAM software interface. The top menu bar includes File, Edit, View, and Help. Below it is a toolbar with various icons. The 'Numerical Setup' tab is active, and the 'checkMesh' button, represented by a checkmark icon, is circled in red. A red arrow points from the 'checkMesh' button to a red-bordered box containing the text 'IHFOAM dialog box'. This dialog box is a small window titled 'IHFOAM' with an information icon and the text 'OK!! checkMesh End', with an 'OK' button. The background shows the 'Patches Attributes' table with columns for Name, Units, and Value. The table lists various patches like inlet, outlet, ground, top, alpha.water, U, p_rgh, and sides, with their respective units and values. The 'Message output' panel at the bottom shows the text: 'Coupled point location match (average 0) OK.', 'Mesh OK.', and 'End'.

Name	Units	Value
inlet		
outlet		
ground		
top		
type		patch
nFaces		750
startFace		105070
alpha.water		inletOutlet
U		pressureinletOutletVelocity
p_rgh	Kg*m/s2	totalPressure
sides		
type		empty
nFaces		105000
startFace		105820
alpha.water		empty
U		empty
p_rgh	Kg*m/s2	empty

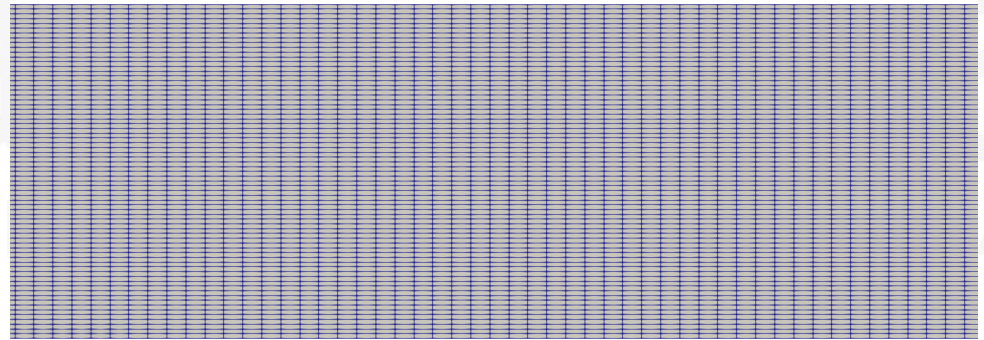
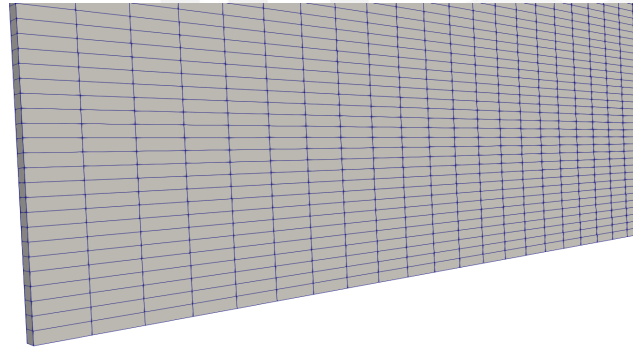
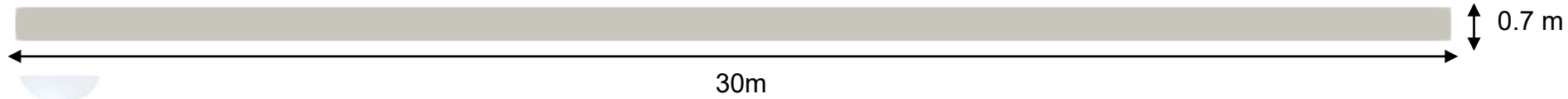
Message output

```
Coupled point location match (average 0) OK.  
Mesh OK.  
End
```

Paraview button



a
ITAL



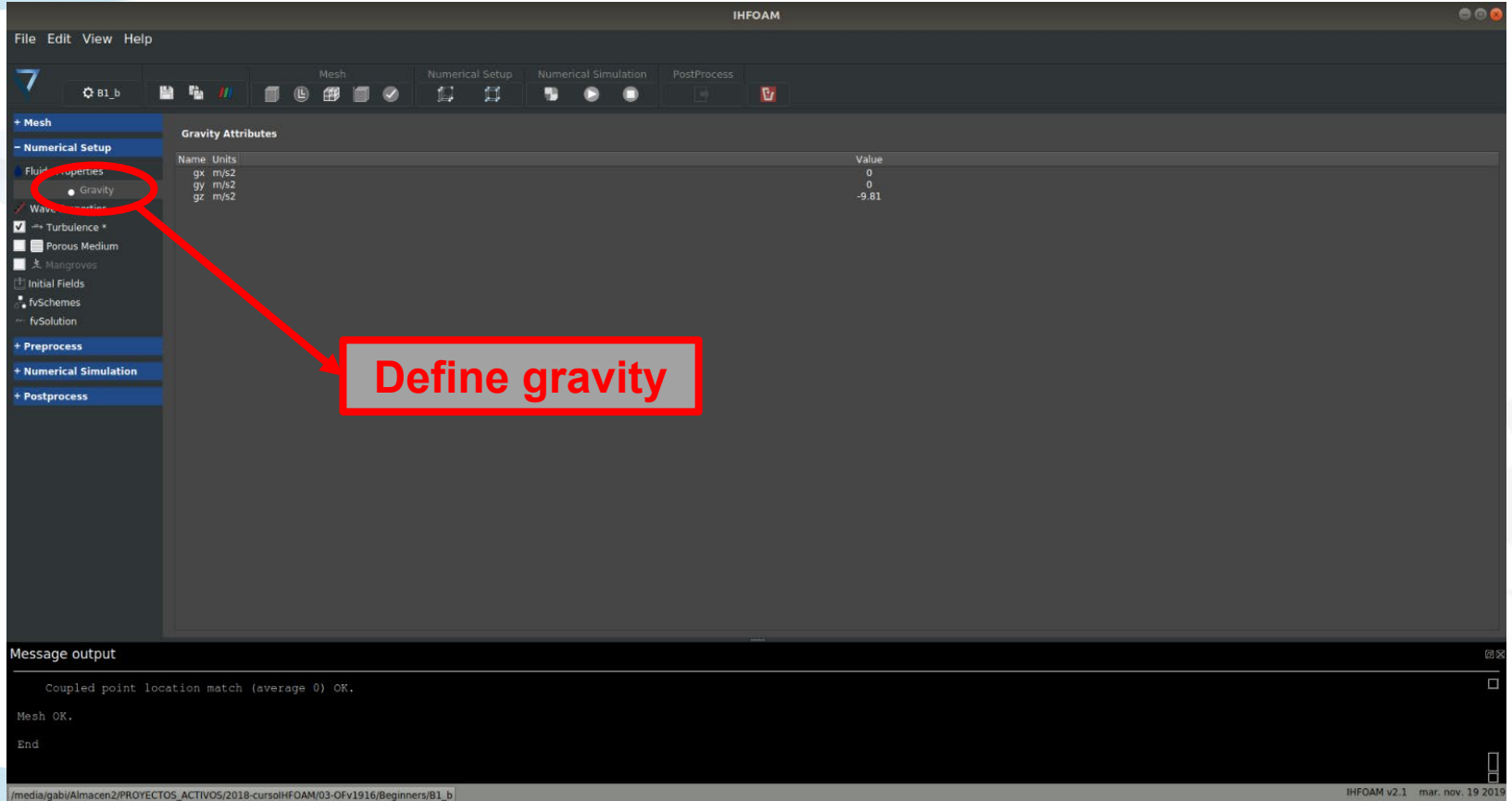
The screenshot shows the IHFOAM software interface. The left sidebar has a tree view with the following items: Mesh, Numerical Setup, Fluids Properties (selected), Wave Properties, Porous Medium, Mangroves, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The 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

Message output

```
Coupled point location match (average 0) OK.  
Mesh OK.  
End
```

Define fluids properties



The screenshot shows the IHFOAM software interface. On the left sidebar, under 'Numerical Setup', the 'Wave Properties' option is highlighted with a red circle. A red arrow points from this circle to a text box that reads 'Define wave properties (generation and absorption)'. The main panel displays the 'WaveProp Attributes' settings. It includes a table for 'WaveTheoryRange' and two expandable sections: 'waveGeneration' and 'waveAbsorption'.

Name	Units	Value
patch		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	m	
patch		outlet
waveModel		shallowWaterAbsorption
nPaddle		1

Message output:

```
Coupled point location match (average 0) OK.
Mesh OK.
End
```

Path: /media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolIHFOAM/03-Ofv1916/Beginners/B1_b

IHFOAM v2.1 mar. nov. 19 2019

The screenshot shows the IHFOAM software interface. The sidebar on the left contains a tree view with the following categories: Mesh, Numerical Setup, Preprocess, Numerical Simulation, and Postprocess. Under Numerical Setup, there are sub-items: Fluids Properties, Gravity, Wave Properties, Turbulence (checked), Porous Medium, and Initial Fields (highlighted with a red circle). A red arrow points from this circle to a red-bordered box containing the text "Set initial water depth".

The main window 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

At the bottom, the "Message output" panel shows the following text:

```
Coupled point location match (average 0) OK.  
Mesh OK.  
End
```

a
TAL

setFields button

The screenshot shows the IHFOAM software interface. The 'Numerical Setup' tab is active, displaying the 'InitFields Attributes' table. A red circle highlights the 'setFields' button in the toolbar. A red arrow points from this button to a dialog box titled 'IHFOAM' that appears in the center of the screen. The dialog box contains the message: 'OK!! rm -rf 0 && cp -r 0.orig 0 && setFields End' and an 'OK' button. Below the dialog box, a red arrow points to a red-bordered box containing the text 'IHFOAM dialog box'. At the bottom of the interface, the 'Message output' window shows the following text:

```
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: `/media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolIHFOAM/03-OFv1916/Beginners/B1_b` and the version: `IHFOAM v2.1 mar. nov. 19 2019`.

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

Press F10 to display the
domain visor

The screenshot displays the IHFOAM software interface. The main window is titled 'IHFOAM' and contains a menu bar (File, Edit, View, Help) and a toolbar. The left sidebar shows a tree view with categories: Mesh, Numerical Setup, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The 'Numerical Setup' window is active, showing 'InitFields Attributes' with a table of parameters.

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 'Domain Visor' window on the right shows a 3D visualization of the domain. It includes a coordinate system with axes X (m), Y (m), and Z (m). The domain is a rectangular box with dimensions 30m in X, 1m in Y, and 0.4m in Z. The domain is divided into regions labeled 'line1', 'line2', 'line3', 'line4', 'line5', 'line6', 'line7', 'line8', 'line9', 'line10', 'line11', 'line12', 'line13', 'line14', 'line15', 'line16', 'line17', 'line18', 'line19', 'line20', 'line21', 'line22', 'line23', 'line24', 'line25', 'line26', 'line27', 'line28', 'line29', 'line30', 'line31', 'line32', 'line33', 'line34', 'line35', 'line36', 'line37', 'line38', 'line39', 'line40', 'line41', 'line42', 'line43', 'line44', 'line45', 'line46', 'line47', 'line48', 'line49', 'line50', 'line51', 'line52', 'line53', 'line54', 'line55', 'line56', 'line57', 'line58', 'line59', 'line60', 'line61', 'line62', 'line63', 'line64', 'line65', 'line66', 'line67', 'line68', 'line69', 'line70', 'line71', 'line72', 'line73', 'line74', 'line75', 'line76', 'line77', 'line78', 'line79', 'line80', 'line81', 'line82', 'line83', 'line84', 'line85', 'line86', 'line87', 'line88', 'line89', 'line90', 'line91', 'line92', 'line93', 'line94', 'line95', 'line96', 'line97', 'line98', 'line99', 'line100'. The domain is colored blue and green. A red circle highlights the 'Output' button in the top right corner of the Domain Visor window.

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

```
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
```


Paraview button

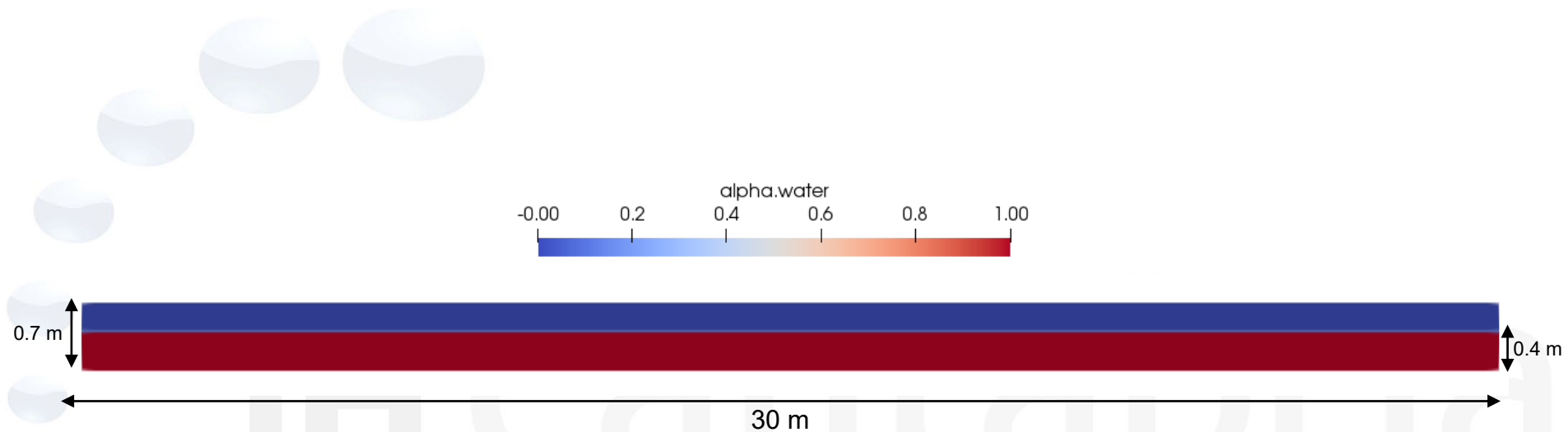
The screenshot shows the IHFOAM software interface. The top menu bar includes File, Edit, View, and Help. Below it is a toolbar with various icons. The 'Paraview' icon, which is a red cube, is circled in red. A red arrow points from the text 'Paraview button' to this icon. The main window is divided into several panels. On the left is a sidebar with a tree view showing the project structure: Mesh, Numerical Setup, Numerical Simulation, and PostProcess. The 'Numerical Setup' panel is active, showing a list of 'InitFields Attributes' with columns for Name, Units, and Value. The 'Output' panel on the right shows a 3D visualization of the domain, with a grid and a blue rectangular region. The 'Message output' panel at the bottom shows the following text:

```
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

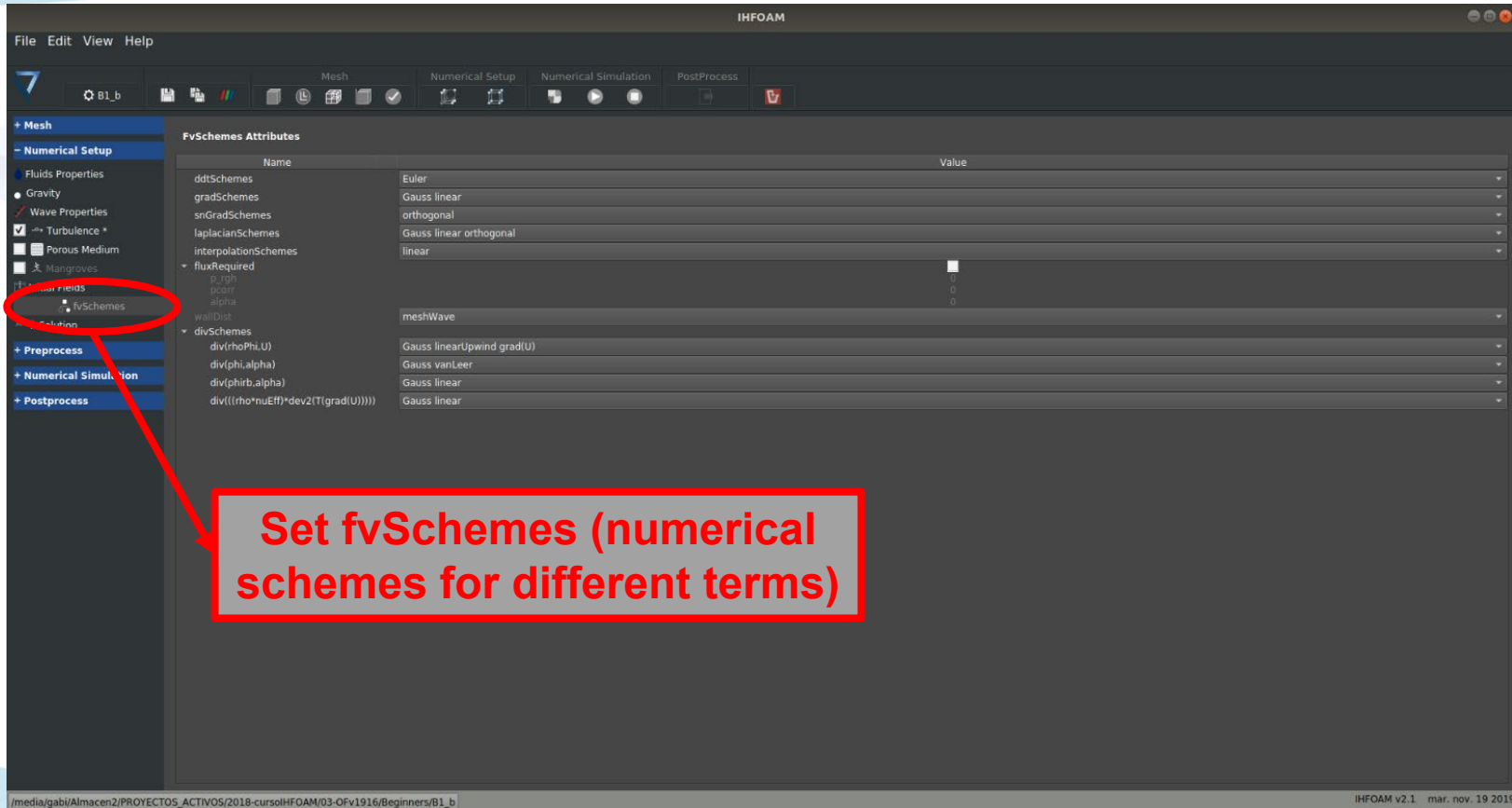
End
```

/media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolIHFOAM/03-Ofv1916/Beginners/B1_b

IHFOAM v2.1 mar. nov. 19 2019



INSTITUTO DE HIDRÁULICA AMBIENTAL
UNIVERSIDAD DE CANTABRIA



The screenshot shows the IHFOAM software interface. The left sidebar contains a tree view with the following categories: Mesh, Numerical Setup, Preprocess, Numerical Simulation, and Postprocess. Under Numerical Setup, there are sub-items: Fluids Properties, Gravity, Wave Properties, Turbulence (checked), Porous Medium, Mangroves, and fvSchemes (highlighted with a red circle). A red arrow points from the 'fvSchemes' item to a text box.

Set fvSchemes (numerical schemes for different terms)

The main window displays the 'fvSchemes Attributes' table:

Name	Value
ddtSchemes	Euler
gradSchemes	Gauss linear
snGradSchemes	orthogonal
laplacianSchemes	Gauss linear orthogonal
interpolationSchemes	linear
fluxRequired	<input type="checkbox"/>
p_rgh	0
pcorr	0
alpha	0
wallDist	meshWave
divSchemes	
div(rhoPhi,U)	Gauss linearUpwind grad(U)
div(phi.alpha)	Gauss vanLeer
div(phi.b.alpha)	Gauss linear
div(((rho*nuEff)*dev2(T(grad(U))))	Gauss linear

Set fvSolution (equations solvers, tolerances and algorithms)

Regular waves in an empty wave flume

The screenshot shows the 'fvSolution' settings in the IHFOAM v2.1 interface. The left sidebar has 'fvSolution' selected. The main panel displays a tree view of solver settings. A red arrow points from the title to the 'fvSolution' option in the sidebar.

Name	Value
PIMPLE	
momentumPredictor	no
nCorrectors	2
nNonOrthogonalCorrectors	0
alpha.water	
alphaOuterCorrectors	
cAlpha	1
nAlphaCorr	1
nAlphaSubCycles	3
p_rgh	
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	GAMG
smoother	DiC
tolerance	1e-07
relTol	0
U	
solver	PBICG
preconditioner	DILU
tolerance	1e-06
relTol	0.1
Ufinal	
solver	PBICG
preconditioner	DILU
tolerance	1e-06
relTol	0

/media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolIHFOAM/03-OFV1916/Beginners/B1_b

IHFOAM v2.1 mar. nov. 19 2019

File Edit View Help

Mesh Numerical Setup Numerical Simulation PostProcess

+ Mesh

+ Numerical Setup

- Preprocess

FreeSurface

Virtual Probes

Line Probes

Cornea

Pressures

+ Numerical Simulation

+ Postprocess

LineProbes Attributes

	X (m)	Y (m)	Zmin (m)	Zmax (m)	nPoints
line1	1	0.005	0	0.7	1001
line2	2	0.005	0	0.7	1001
line3	3	0.005	0	0.7	1001
line4	5	0.005	0	0.7	1001
line5	7.5	0.005	0	0.7	1001
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 Mut (m2/s)

Output

Domain Visor

Set line gauges

/media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolIHFOAM/03-OFv1916/Beginners/B1_b

IHFOAM v2.1 mar. nov. 19 2019

Select domain visor

Output

Show labels in window

File Edit View Help

Mesh Numerical Setup Numerical Simulation PostProcess

+ Mesh

+ Numerical Setup

- Preprocess

FreeSurface

Punctual Probes

Line Probes

Forces

Pressures

+ Numerical Simulation

+ Postprocess

LineProbes Attributes

	X (m)	Y (m)	Zmin (m)	Zmax (m)	nPoints
line1	1	0.005	0	0.7	1001
line2	2	0.005	0	0.7	1001
line3	3	0.005	0	0.7	1001
line4	5	0.005	0	0.7	1001
line5	7.5	0.005	0	0.7	1001
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 Mut (m2/s)

Domain visor

File Edit View Help

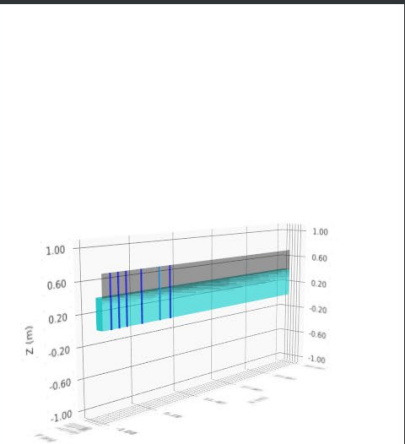
Mesh Numerical Setup Numerical Simulation PostProcess

+ Mesh
+ Numerical Setup
+ Preprocess
- Numerical Simulation
+ Simulation
+ Residuals
+ PostProcess

Parameters Attributes

Name	Units	Value
startFrom		latestTime
startTime	s	0
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 checked="" type="checkbox"/>
maxCo		0.65
maxAlphaCo		0.65
maxDeltaT		0.05
residuals		<input type="checkbox"/>

Output
Domain Visor



Define simulation parameters

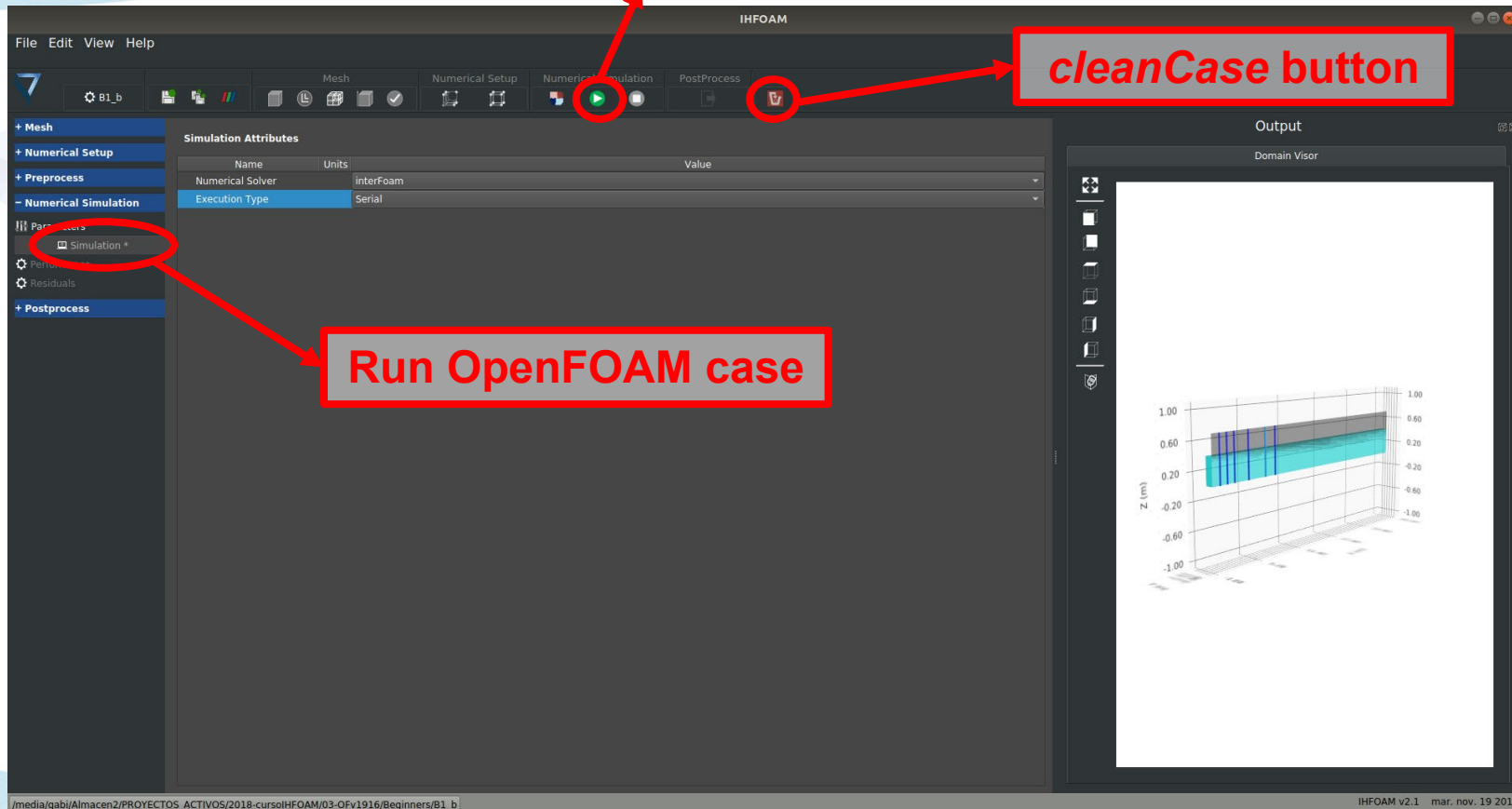
/media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolIHFOAM/03-Ofv1916/Beginners/B1_b

IHFOAM v2.1 mar. nov. 19 2019

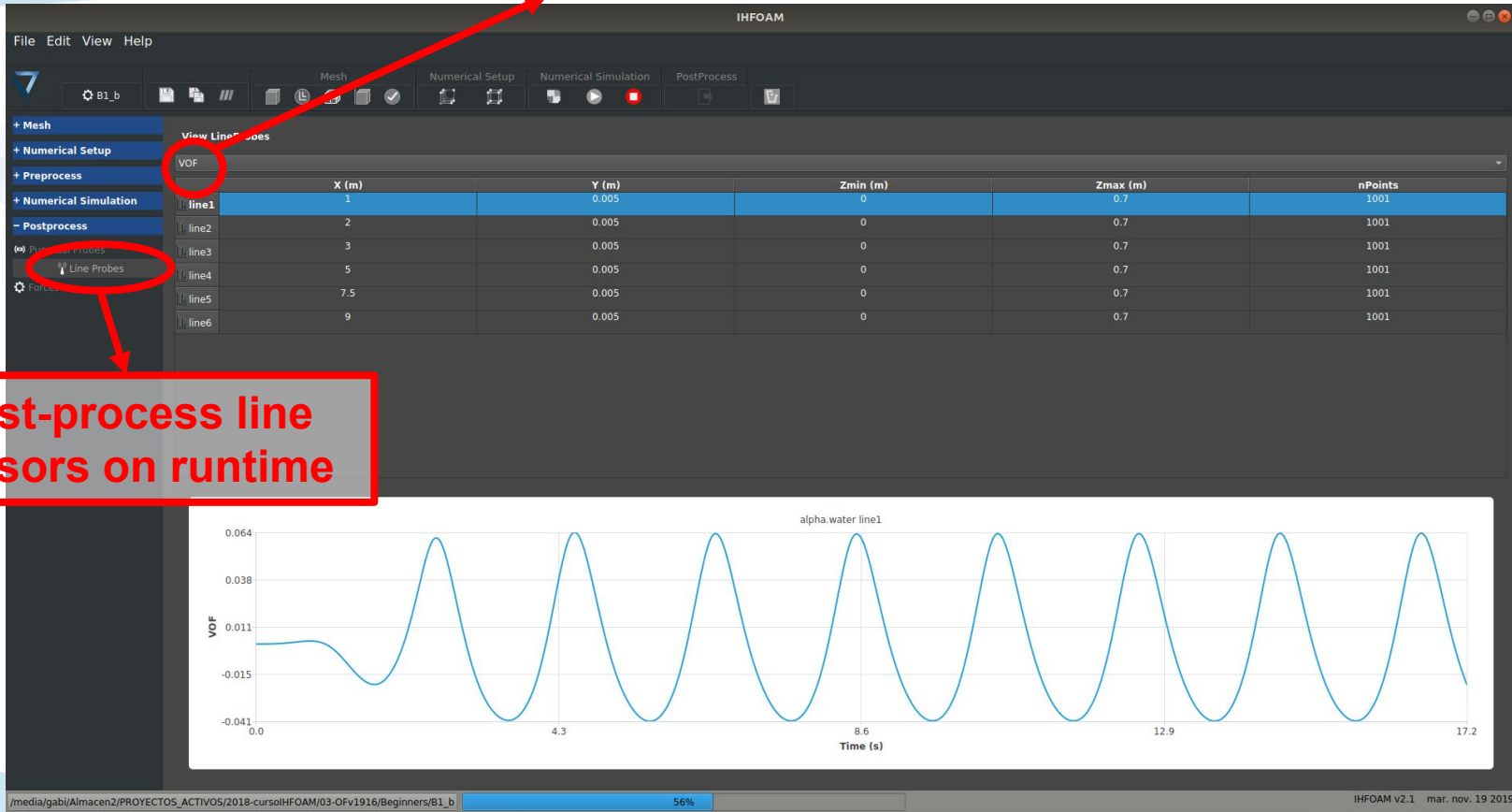
Run/Execute button

cleanCase button

Run OpenFOAM case



plot free surface sensors



at
ITAL

ihFOAM

File Edit View Help

Mesh Numerical Setup Numerical Simulation PostProcess

Mesh

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

Refinement Attributes

Name	Units	Value
------	-------	-------

Output

Domain Visor

ihFOAM

OK!! InterFoam End

OK

Case correctly executed!!

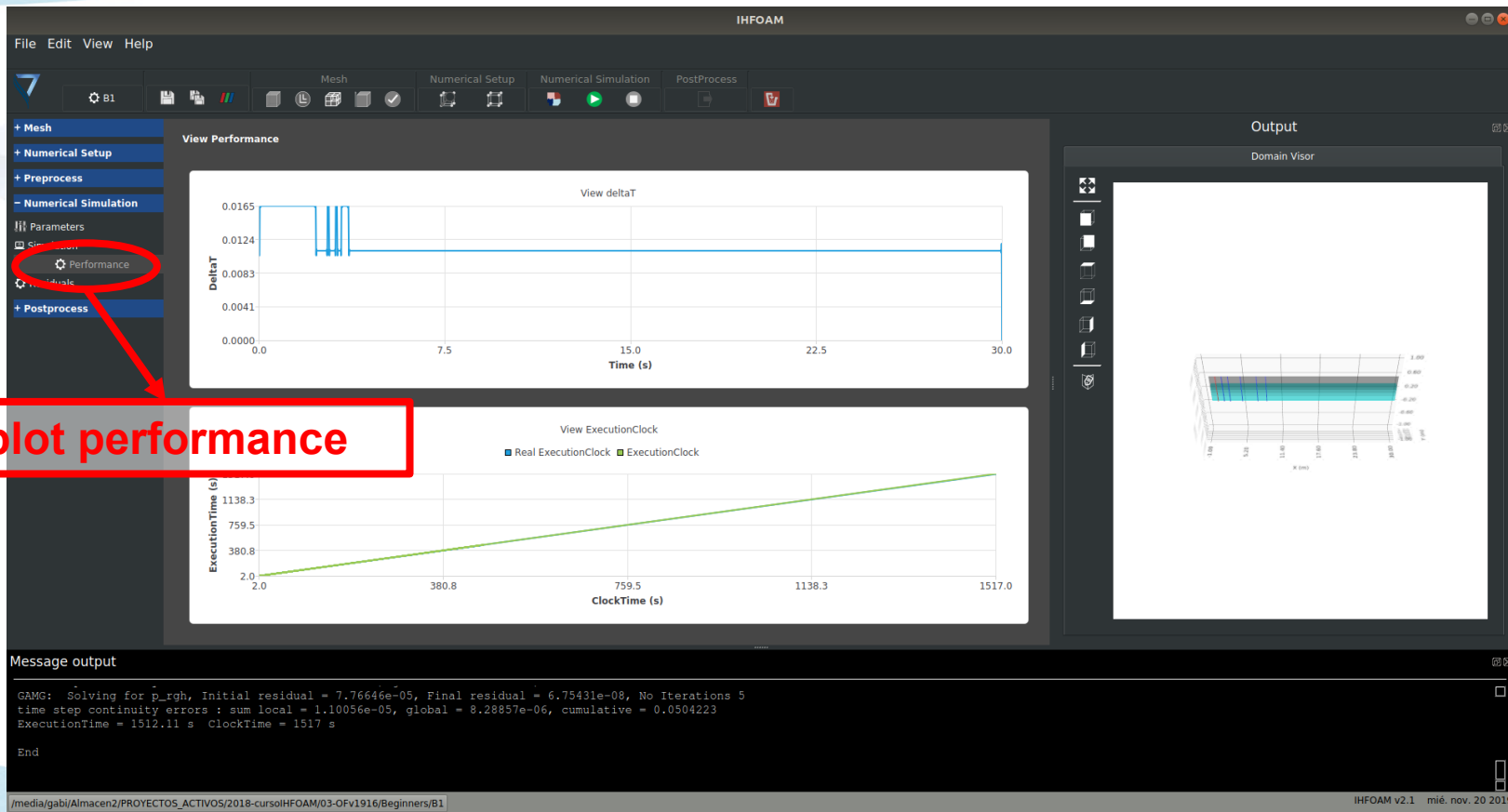
Message output

```
Updating shallowWaterAbsorption wave model for patch outlet
DICPCG: Solving for p_rgh, Initial residual = 0.0173975, Final residual = 7.10205e-05, No Iterations 1
time step continuity errors : sum local = 0.0116409, global = 1.14021e-05, cumulative = 0.050414
GAMG: Solving for p_rgh, Initial residual = 7.76646e-05, Final residual = 6.75431e-08, No Iterations 5
time step continuity errors : sum local = 1.10056e-05, global = 8.28857e-06, cumulative = 0.0504223
ExecutionTime = 1512.11s, ClockTime = 1517 s
```

End

/media/gabi/Almacen2/PROYECTOS_ACTIVOS/2018-cursolihFOAM/03-Ofv1916/Beginners/B1_b 100%

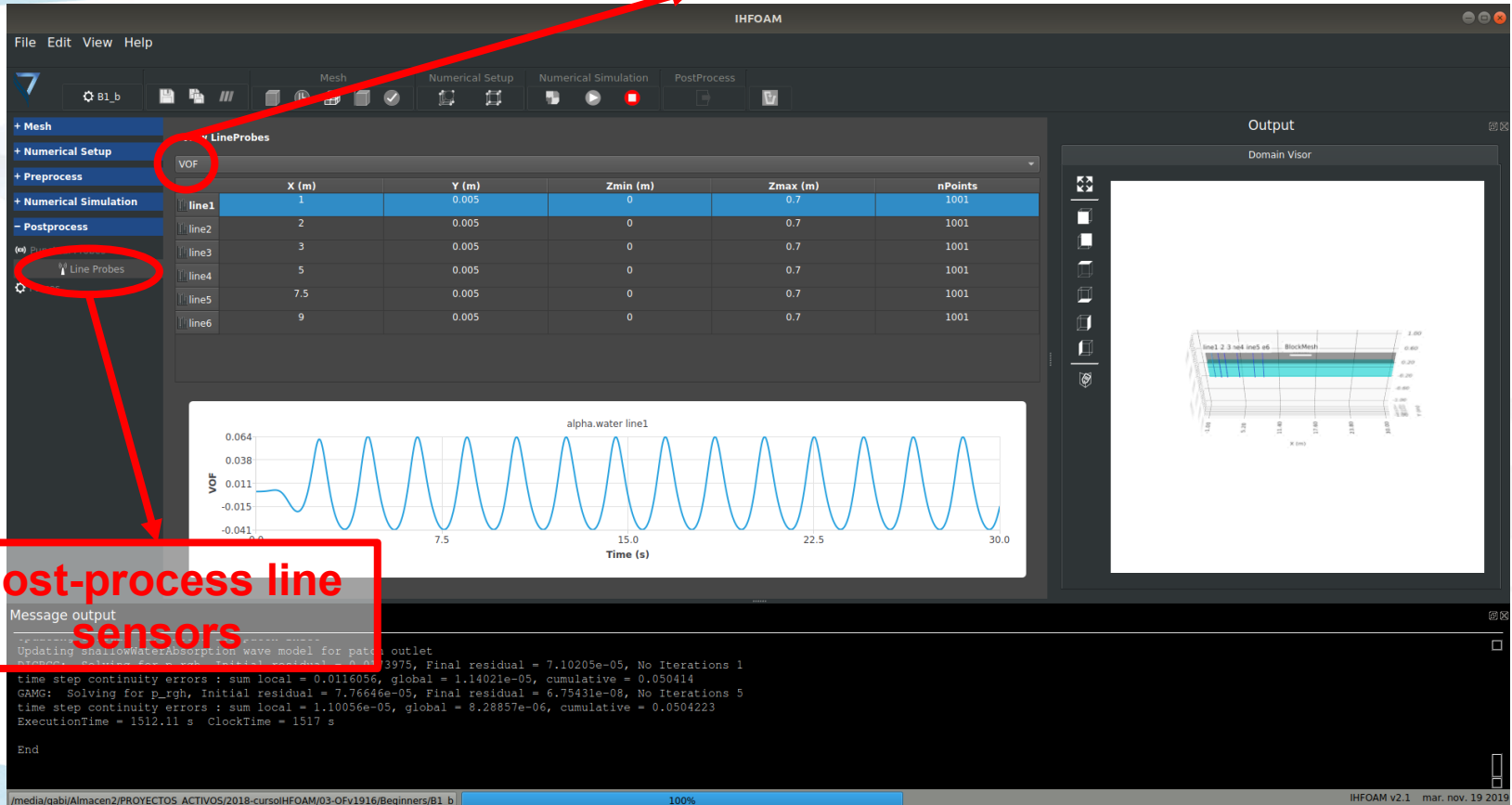
ihFOAM v2.1 mar. nov. 19 2019



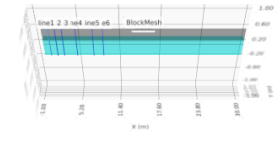
plot performance

ATAL

plot free surface sensors

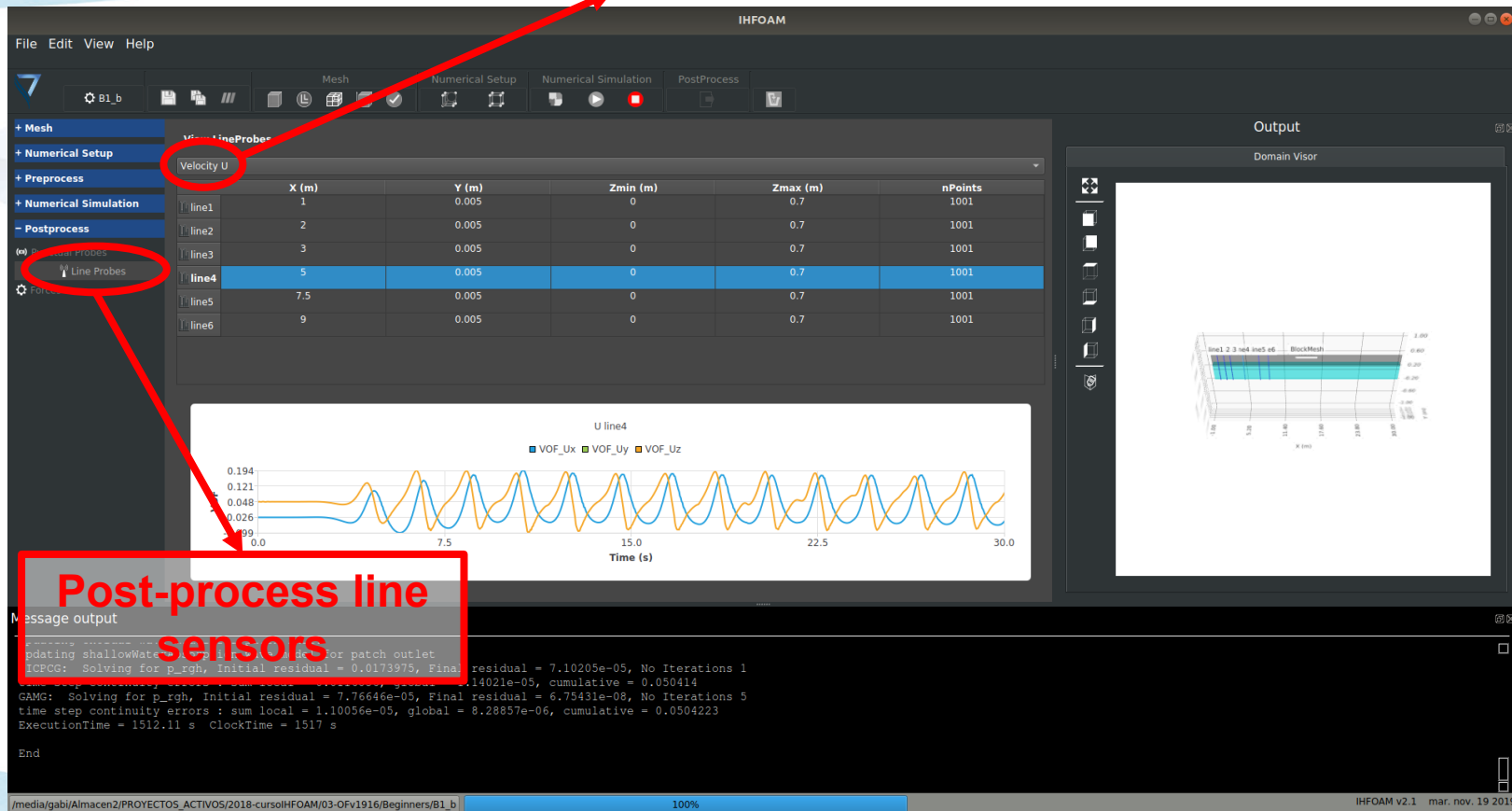


Post-process line
sensors



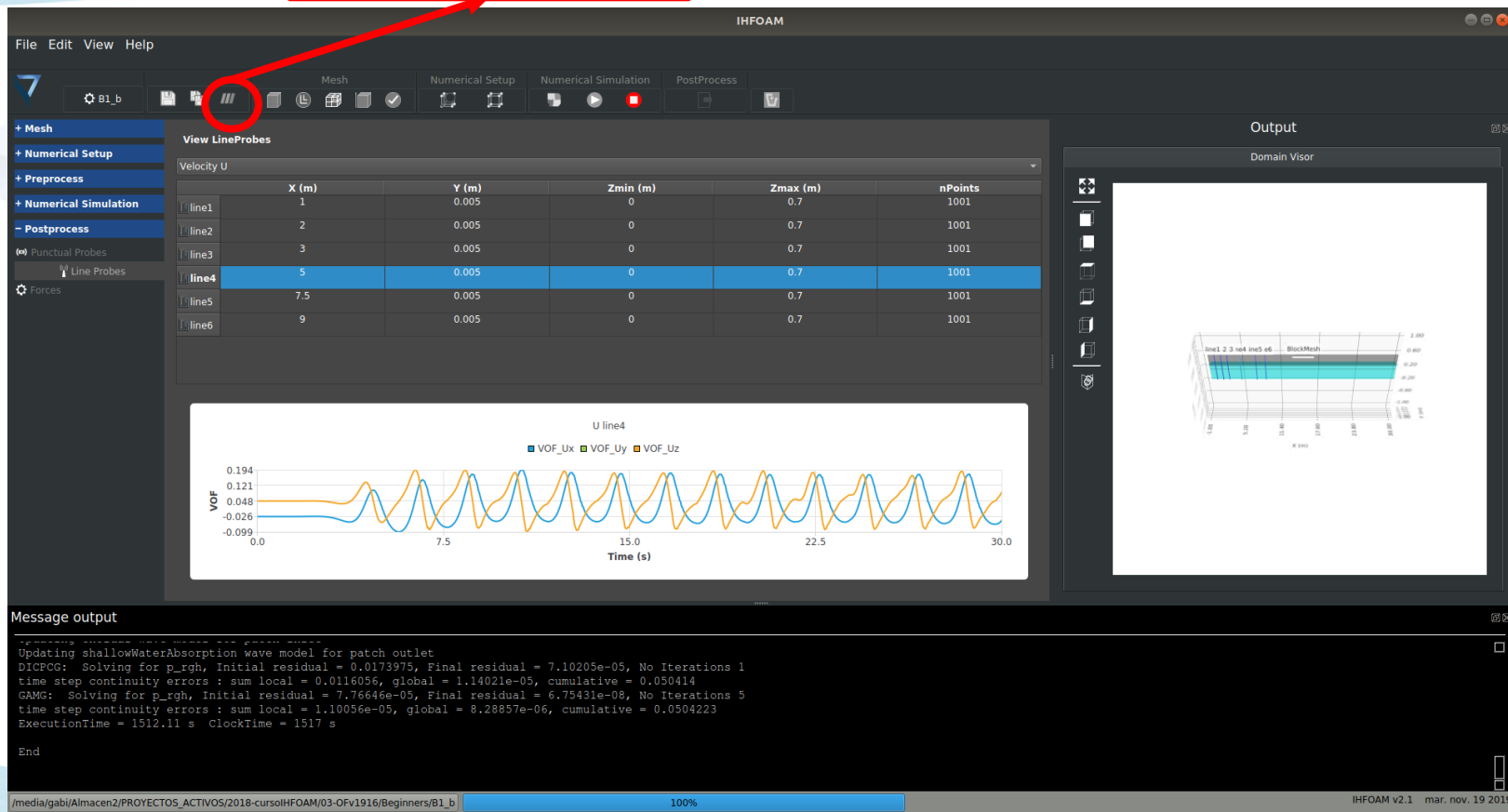
@
TAL

plot velocity sensors



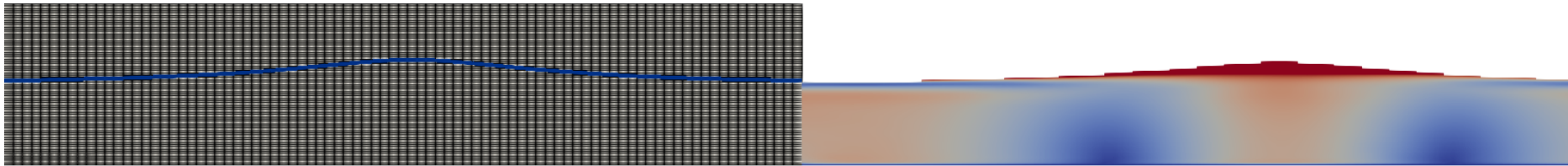
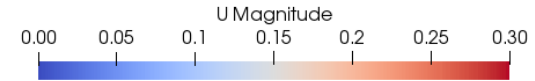
Post-process line
sensors

Paraview button



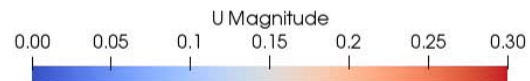


Time: 29.01 s.





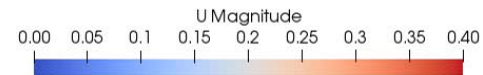
Time: 0.00 s.



- EXERCISE: Change wave parameters.



Time: 0.00 s.





Javier L. Lara, Gabriel Barajas, María Maza, Alessandro Romano.

