



IHCantabria

UNIVERSIDAD DE CANTABRIA

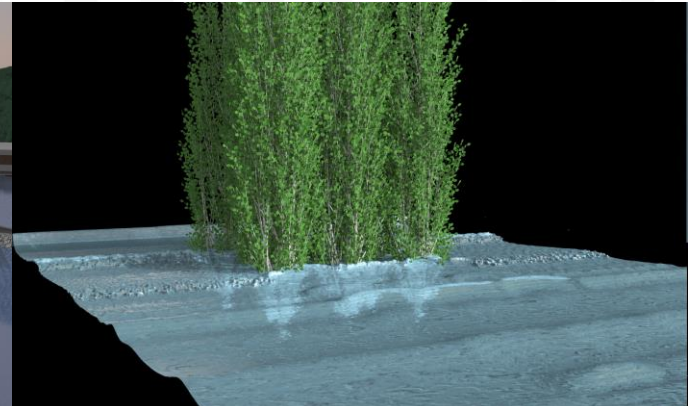
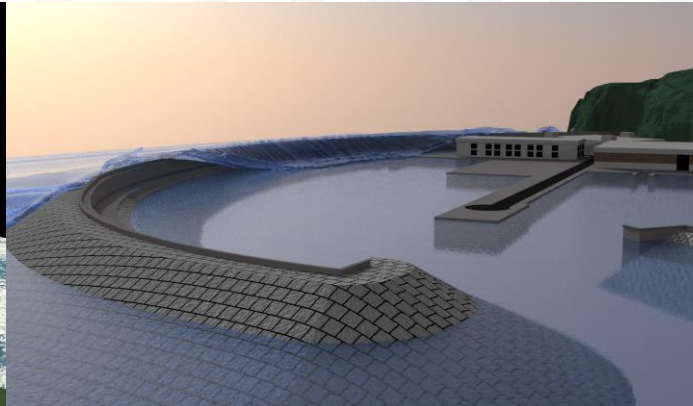
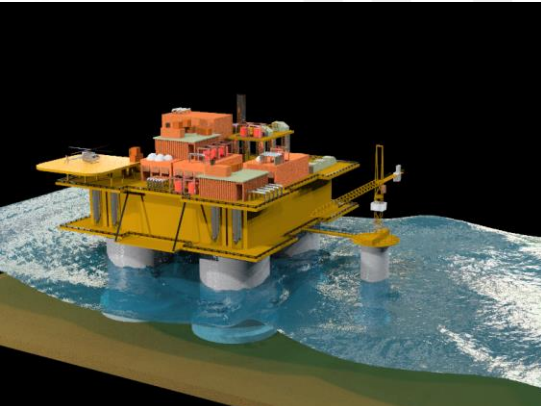
R+D+i for a Sustainable Development

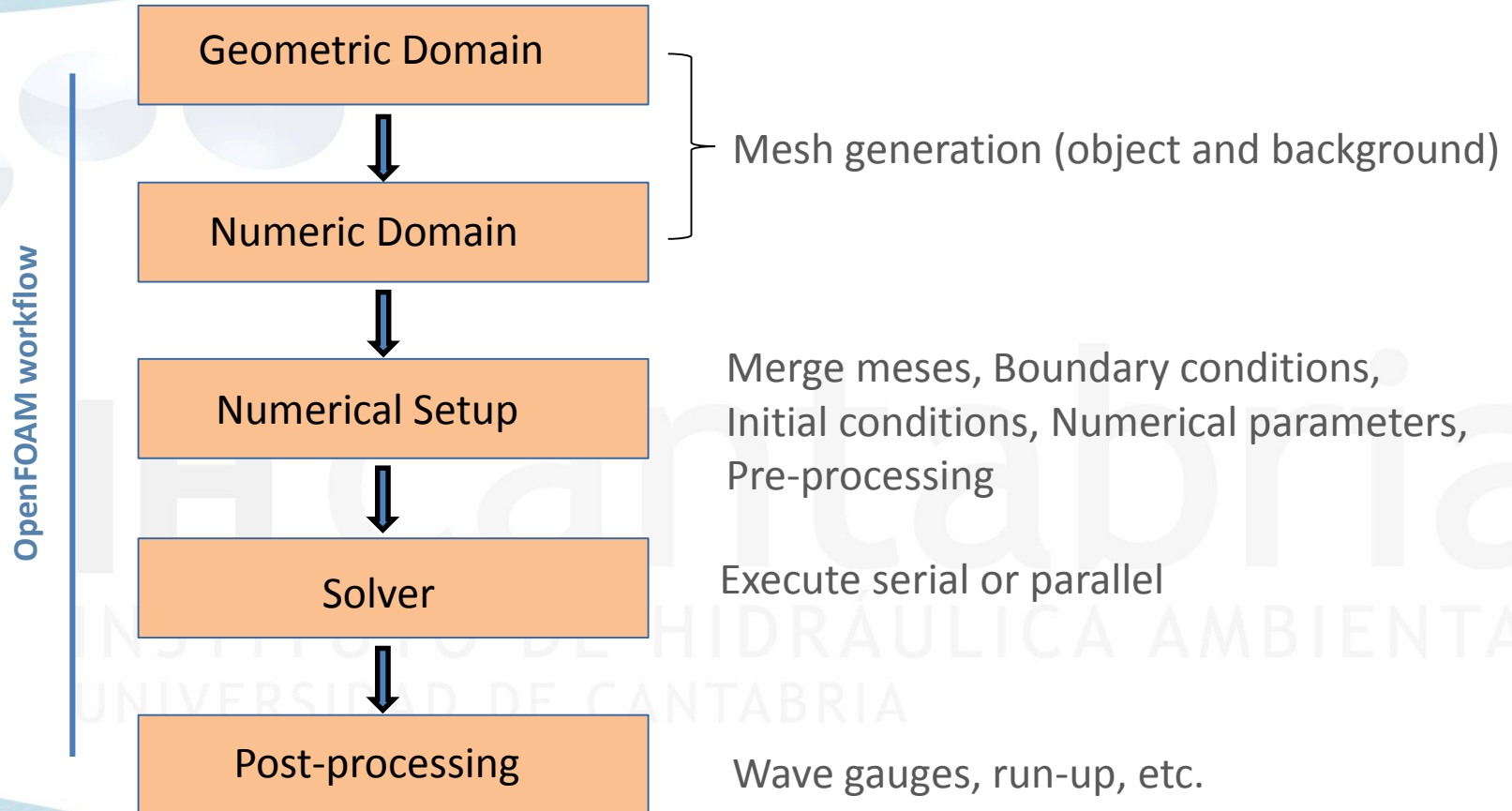
(IHFOAM GUI)

IHFOAM applied to Coastal Engineering

Regular waves interaction with cylinders (3D)

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





OpenFOAM case

0

- alpha.water
- p_rgh
- U
- k
- epsilon
- nut

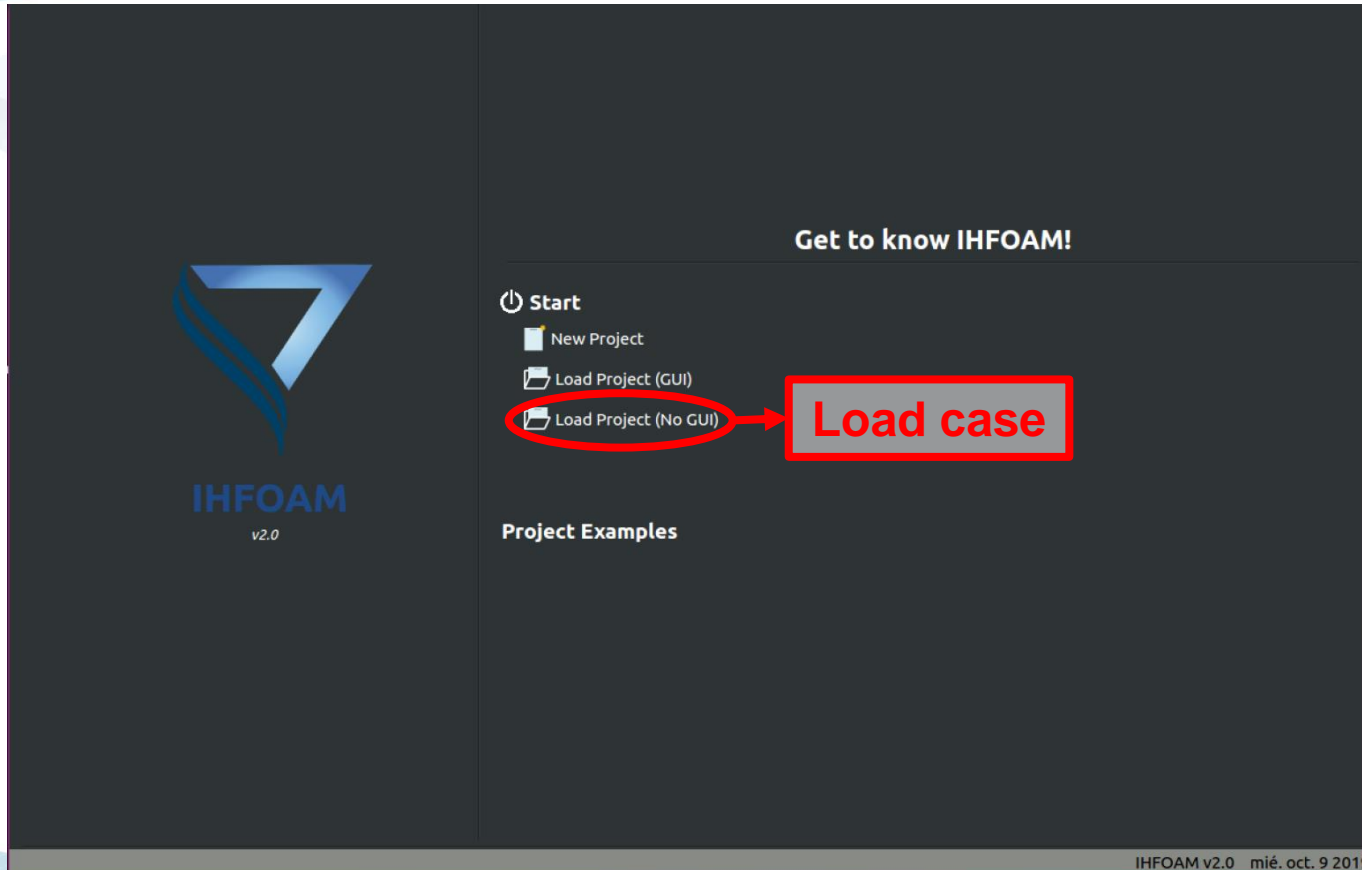
constant

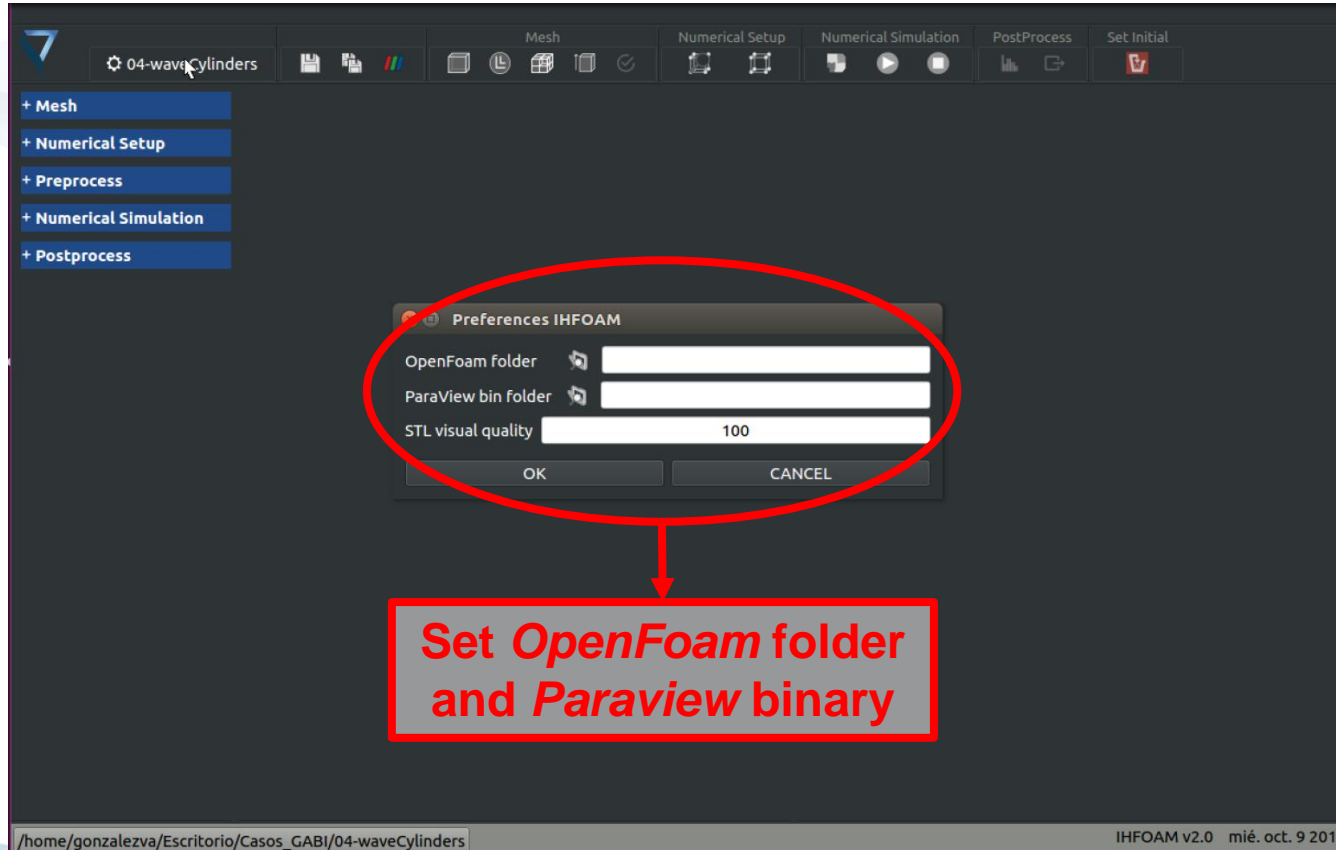
- g
- transportProperties
- turbulenceProperties
- waveProperties

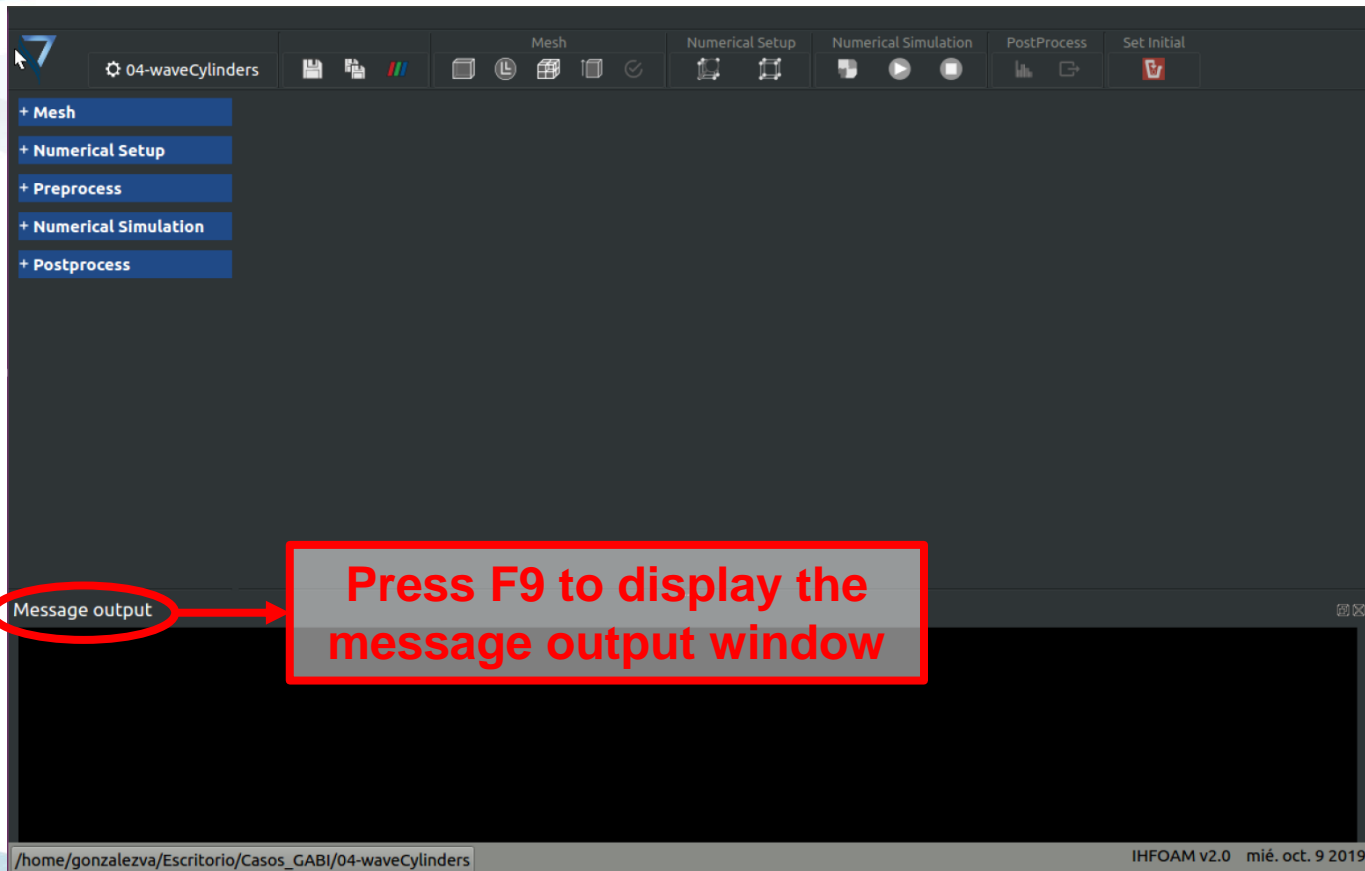
system

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

IHFOAM GUI







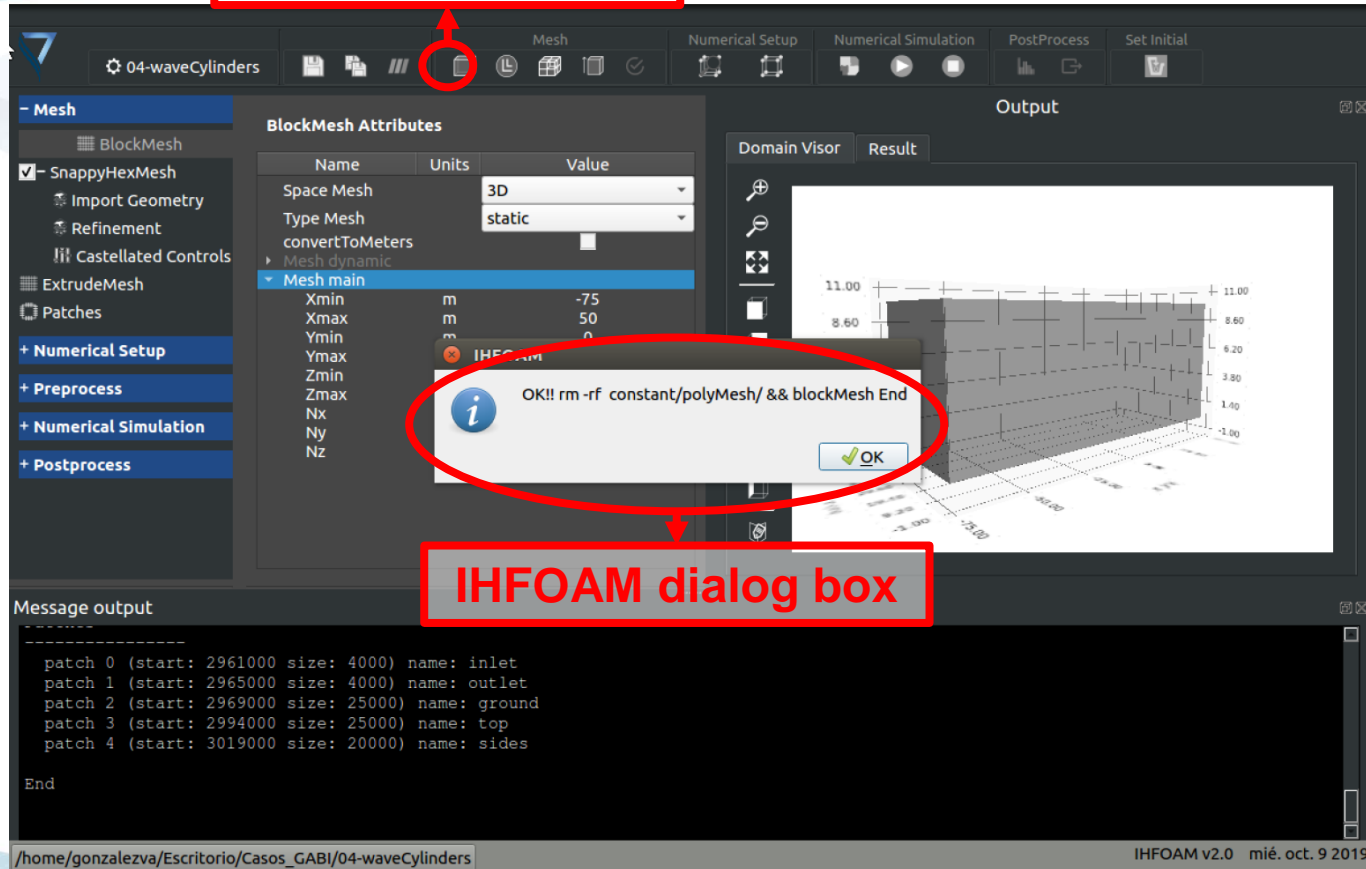
Save single pannel modified
or save all pannels modified.

BlockMesh Attributes

| Name | Units | Value |
|-----------------|-------|--------|
| Space Mesh | | 3D |
| Type Mesh | | static |
| convertToMeters | | |
| Mesh dynamic | | |
| Mesh main | | |
| Xmin | m | -75 |
| Xmax | m | 50 |
| Ymin | m | 0 |
| Ymax | m | 50 |
| Zmin | m | 0 |
| Zmax | m | 10 |
| Nx | | 250 |
| Ny | | 100 |
| Nz | | 40 |

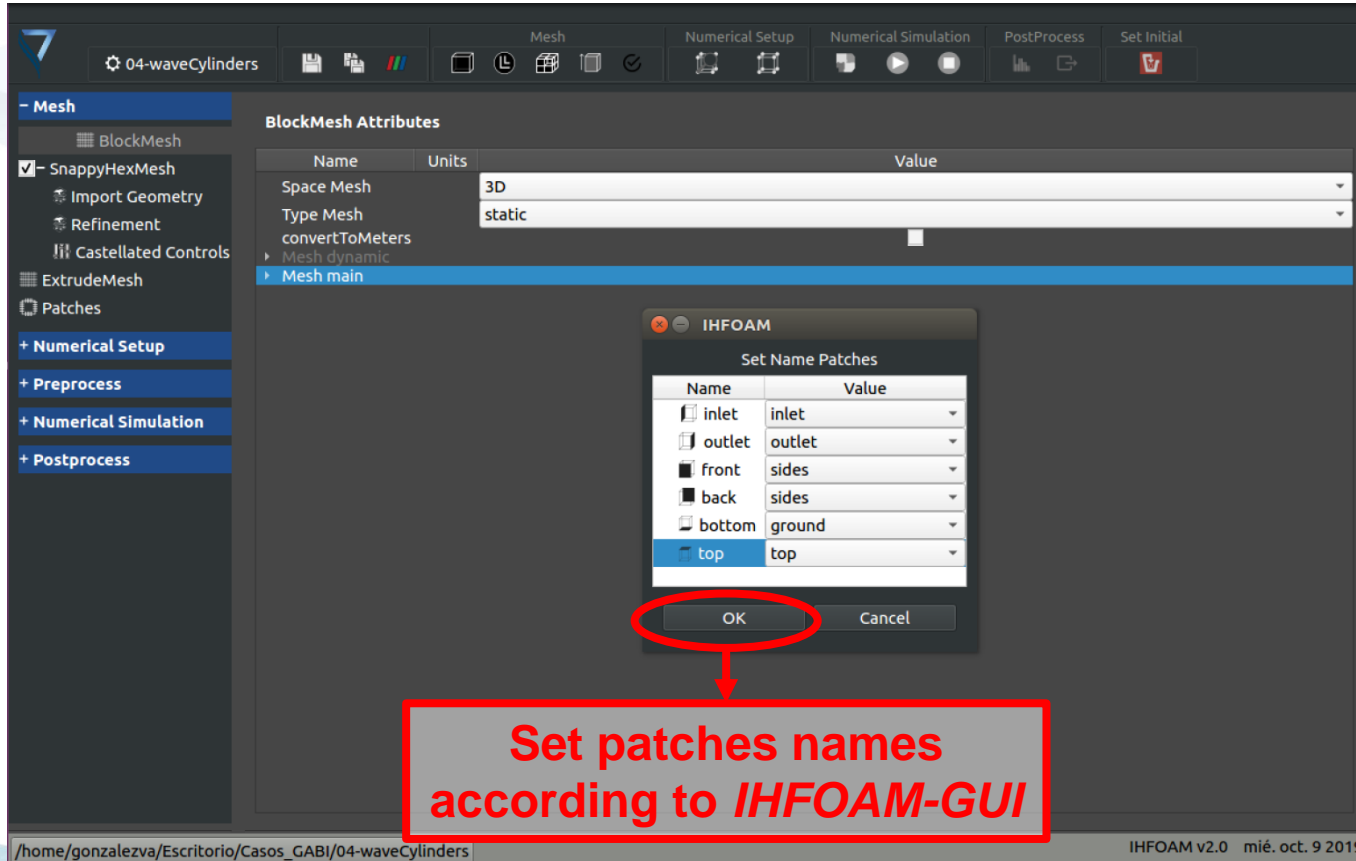
Define geometric domain
and mesh discretization

blockMesh button



IHFOAM dialog box

ia
ENTAL



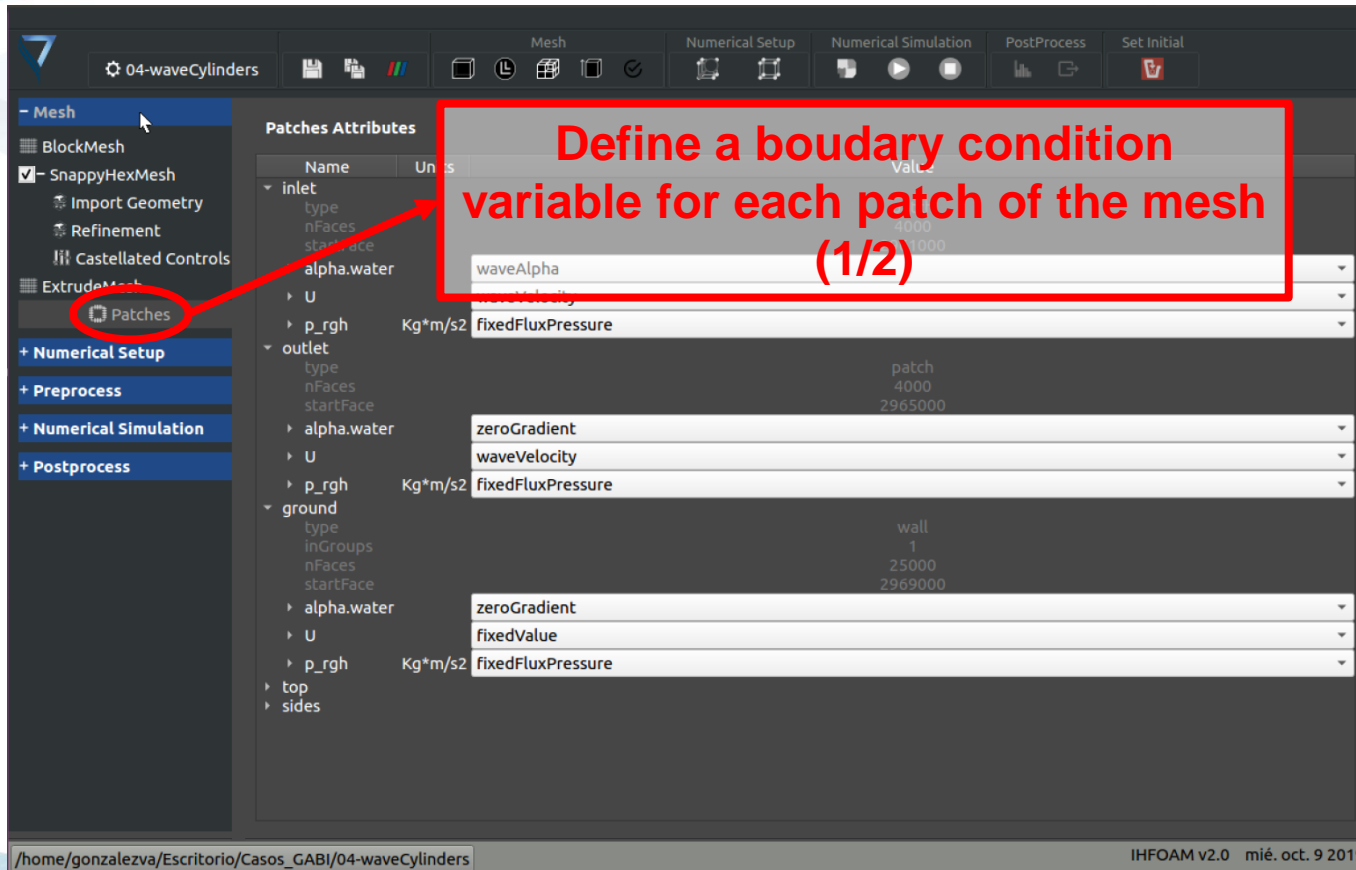
BlockMesh Attributes

| Name | Units | Value |
|-----------------|-------|--------|
| Space Mesh | | 3D |
| Type Mesh | | static |
| convertToMeters | | |
| Mesh dynamic | | |
| Mesh main | | |

Set Name Patches

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

Set patches names according to *IHFOAM-GUI*



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

| Name | Units | Value |
|-------------|---------|-------------------|
| inlet | | |
| type | | patch |
| nFaces | | 4000 |
| startFace | | 2961000 |
| alpha.water | | zeroGradient |
| U | | waveVelocity |
| p_rgh | Kg*m/s2 | fixedFluxPressure |
| outlet | | |
| type | | patch |
| nFaces | | 4000 |
| startFace | | 2965000 |
| alpha.water | | zeroGradient |
| U | | waveVelocity |
| p_rgh | Kg*m/s2 | fixedFluxPressure |
| ground | | |
| type | | wall |
| inGroups | | 1 |
| nFaces | | 25000 |
| startFace | | 2969000 |
| alpha.water | | zeroGradient |
| U | | fixedValue |
| p_rgh | Kg*m/s2 | fixedFluxPressure |
| top | | |
| sides | | |

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL

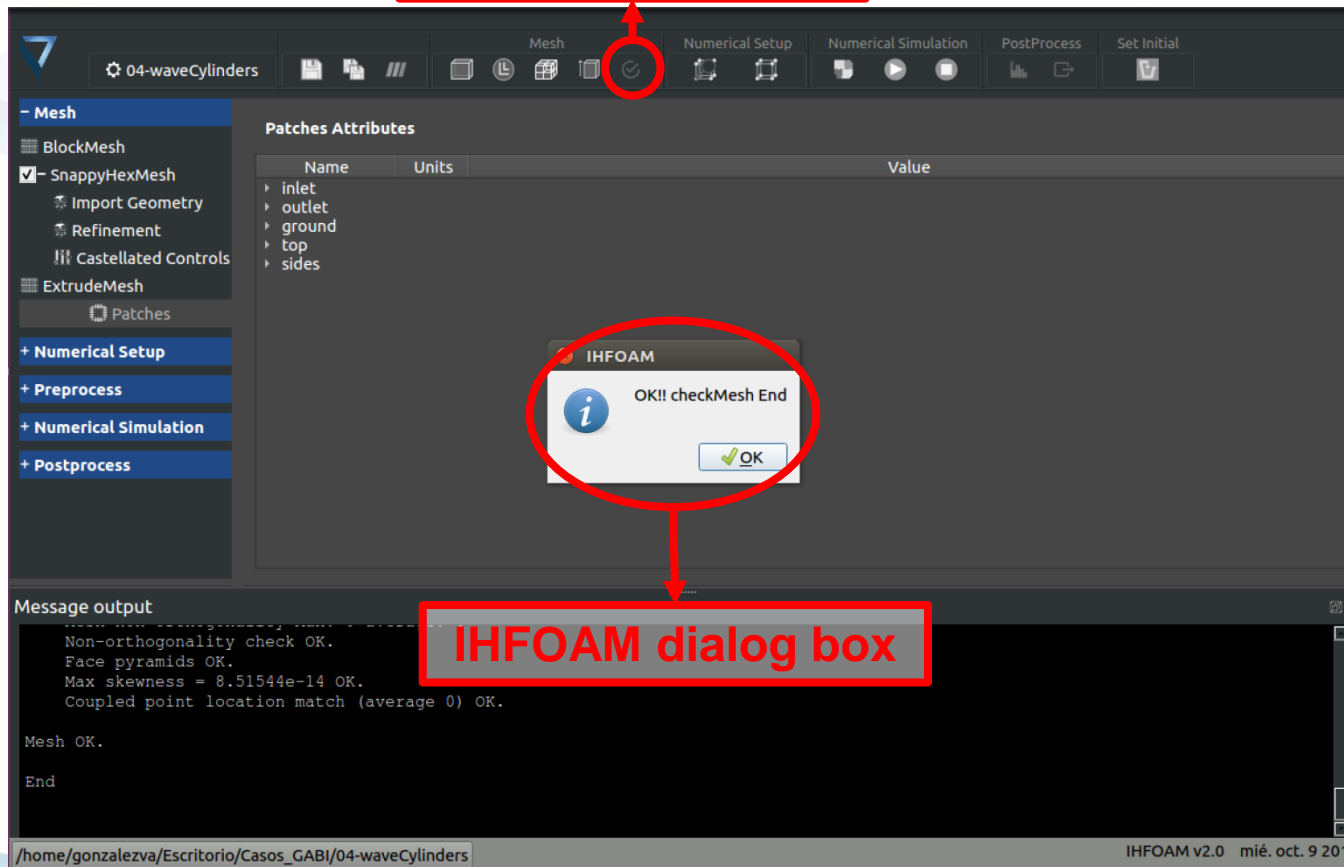
Define a boundary condition variable for each patch of the mesh (2/2)

| Name | Units | Value |
|-------------|---------|-----------------------------|
| inlet | | |
| outlet | | |
| ground | | |
| top | | |
| type | | patch |
| nFaces | | 20000 |
| startFace | | 3019000 |
| alpha.water | | inletOutlet |
| U | | pressureInletOutletVelocity |
| p_rgh | Kg*m/s2 | totalPressure |
| sides | | |
| type | | patch |
| nFaces | | 20000 |
| startFace | | 3019000 |
| alpha.water | | zeroGradient |
| U | | slip |
| p_rgh | Kg*m/s2 | fixedFluxPressure |

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

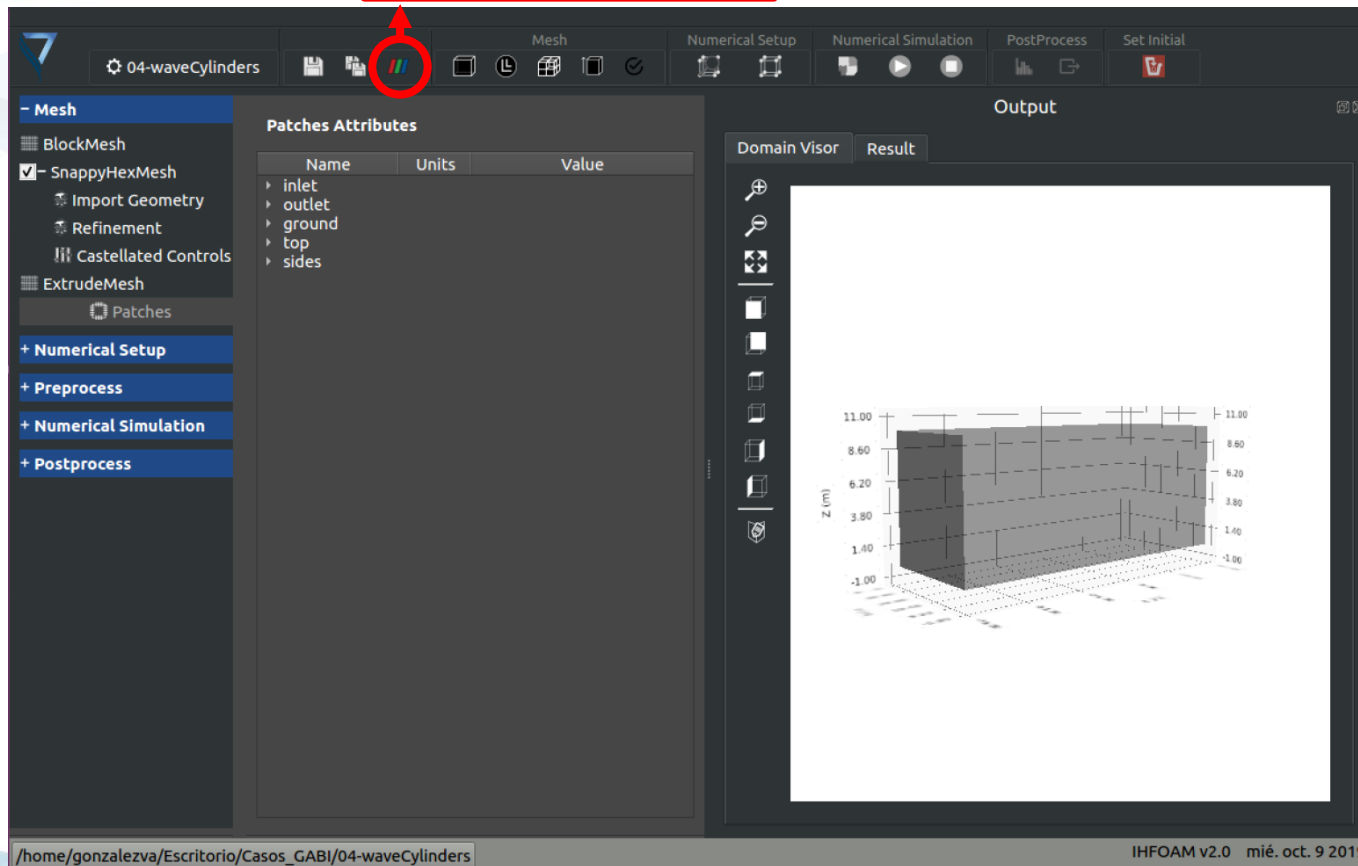
IHFOAM v2.0 mié. oct. 9 2019

checkMesh button

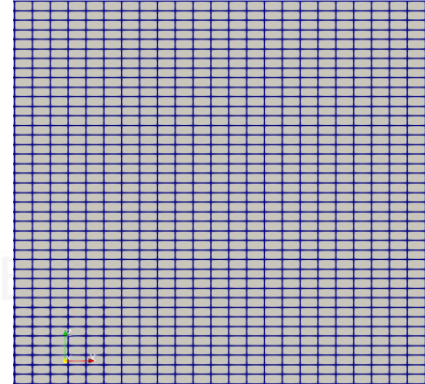
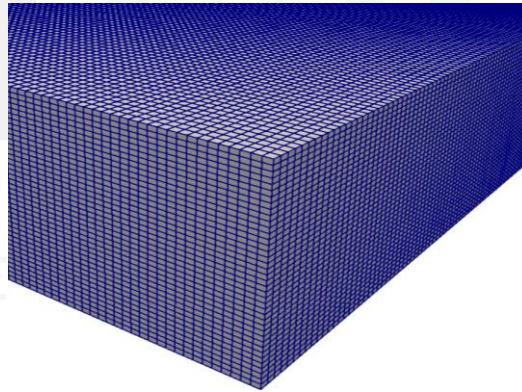
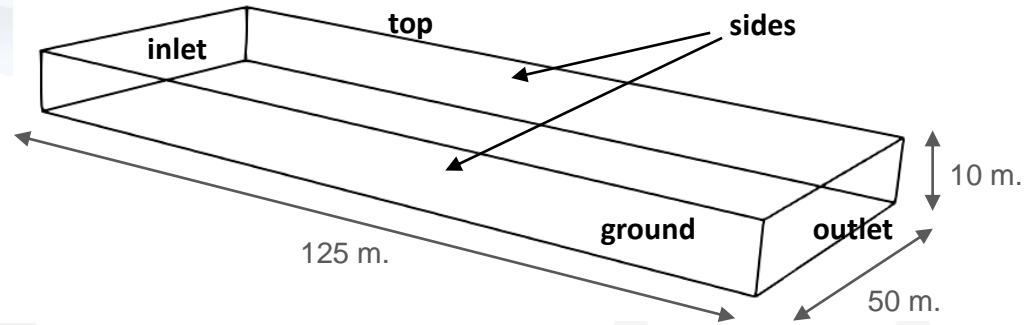


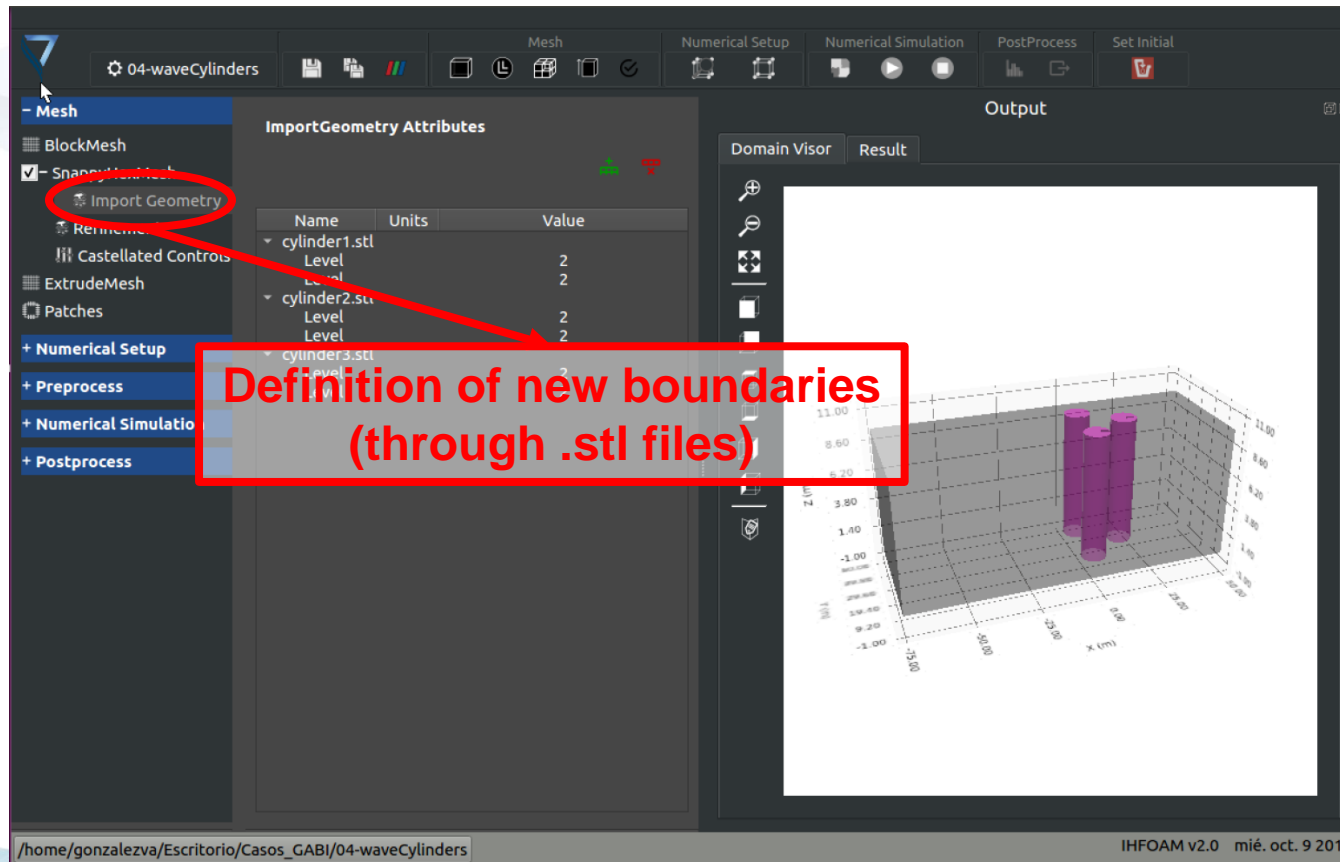
ia
ENTAL

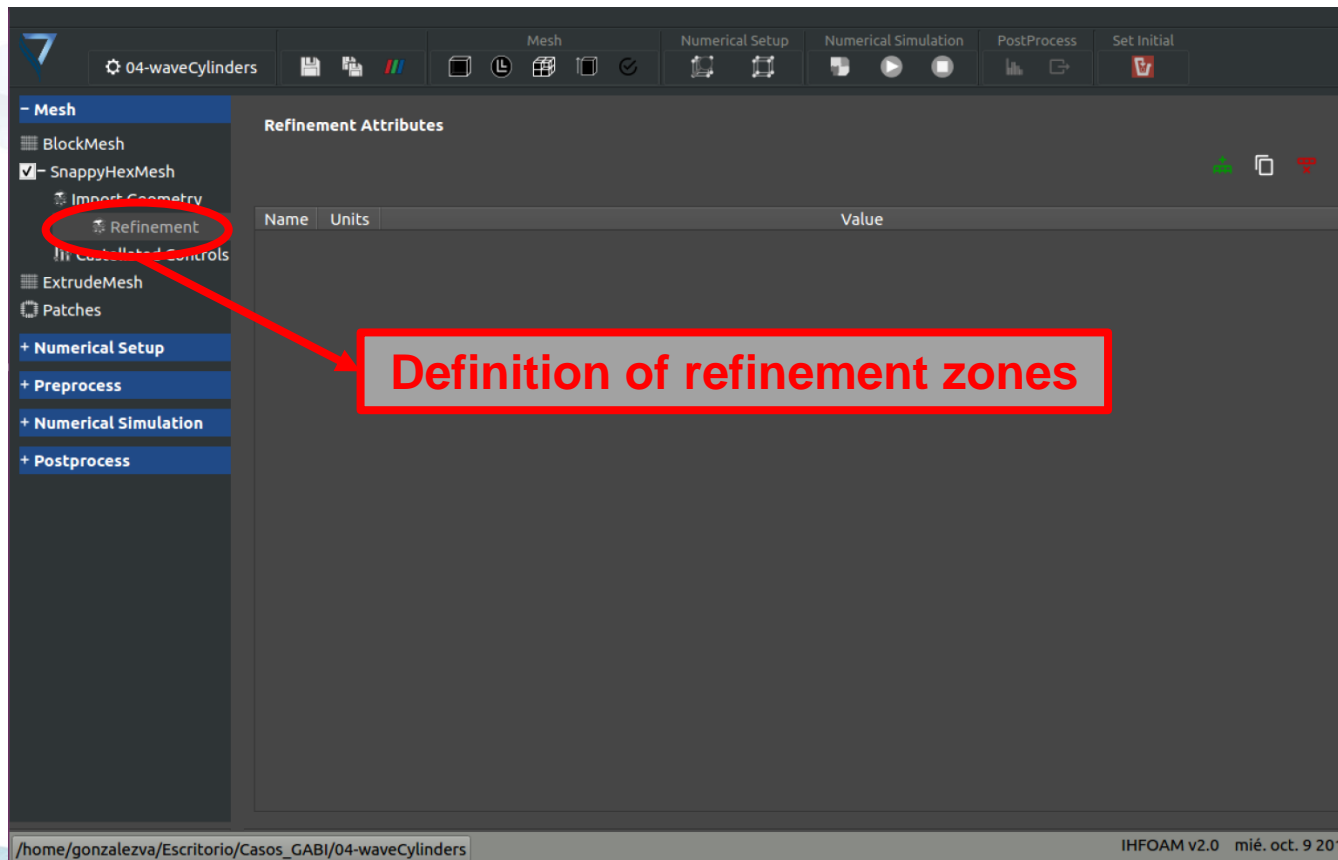
Paraview button



ia
ENTAL







Definition of refinement zones

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

| Name | Units | Value |
|----------------------------|-------|-------------------------------------|
| maxLocalCells | | 100000 |
| maxGlobalCells | | 3000000 |
| minRefinementCells | | 0 |
| nCellsBetweenLevels | | 3 |
| resolveFeatureAngles | | 0 |
| allowFreeStandingZoneFaces | | <input checked="" type="checkbox"/> |
| locationInMesh | | |
| X | m | -40 |
| Y | m | 25 |
| Z | m | 5 |

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

At the bottom of the window, the file path is shown as `/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders` and the version is `IHFOAM v2.0` dated `mié. oct. 9 2019`.

snappyHexMesh button

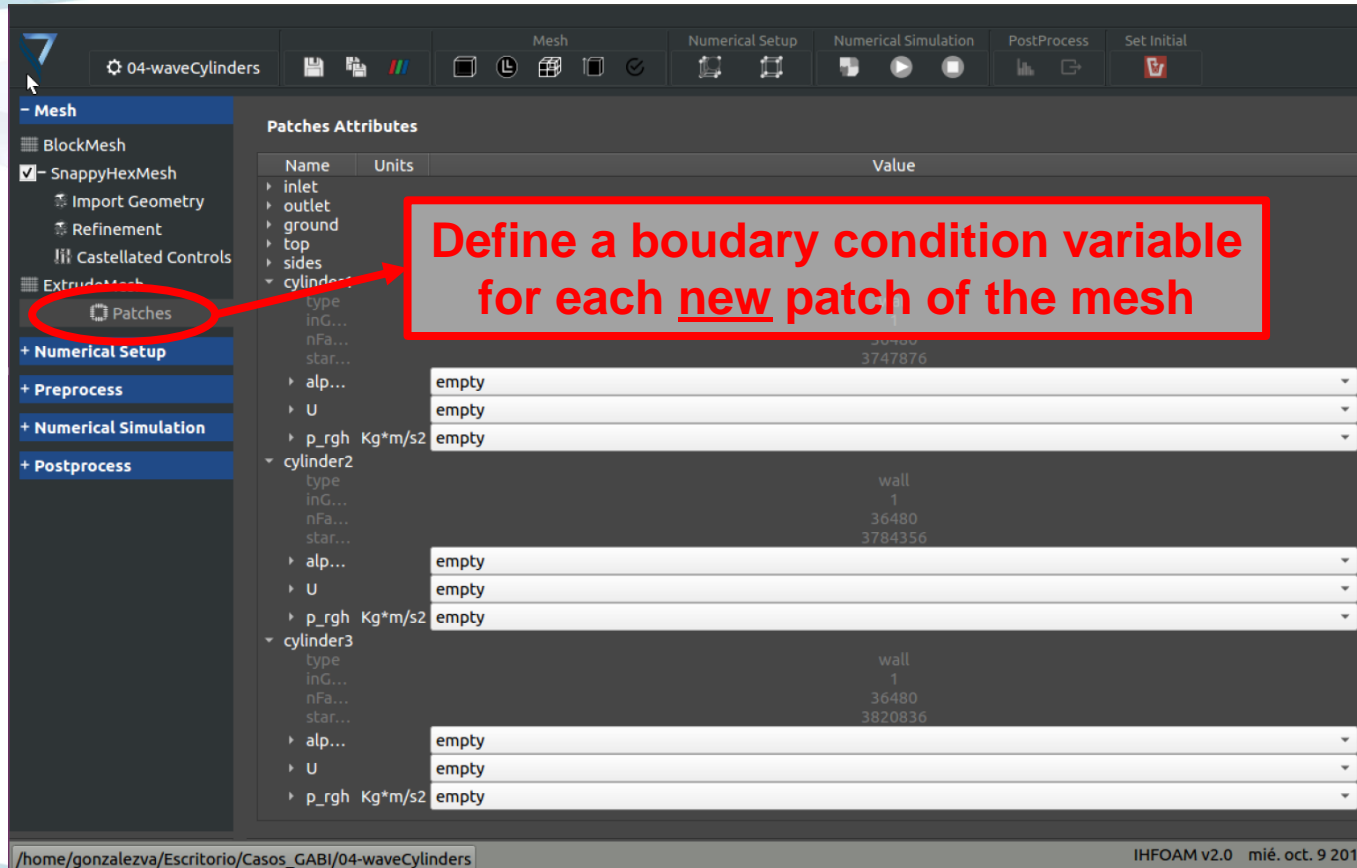
The screenshot shows the IHFOAM software interface. The top toolbar contains several icons, with the 'snappyHexMesh' icon (a cube with a grid) circled in red. Below the toolbar, the 'Mesh' tab is selected, and the 'CastellatedMeshControls' attributes are displayed. The 'snappyHexMesh' checkbox is checked. The 'IHFOAM' dialog box is open, showing the message 'OK!! snappyHexMesh -overwrite End' and an 'OK' button. The 'Message output' window at the bottom shows the following text:

```
faces with skewness > 4 (internal) or 20 (boundary) : 0
faces with interpolation weights (0.1) < 0.05 : 0
faces with volume ratio of neighbour cells < 0.01 : 0
faces with face twist < 0.05 : 0
faces on cells with determinant < 0.001 : 0
Finished meshing without any errors
Finished meshing in = 305.15 s.
End
```

The status bar at the bottom indicates the file path: `/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders` and the version: `IHFOAM v2.0` dated `mié. oct. 9 2019`.

IHFOAM dialog box

ia
ENTAL



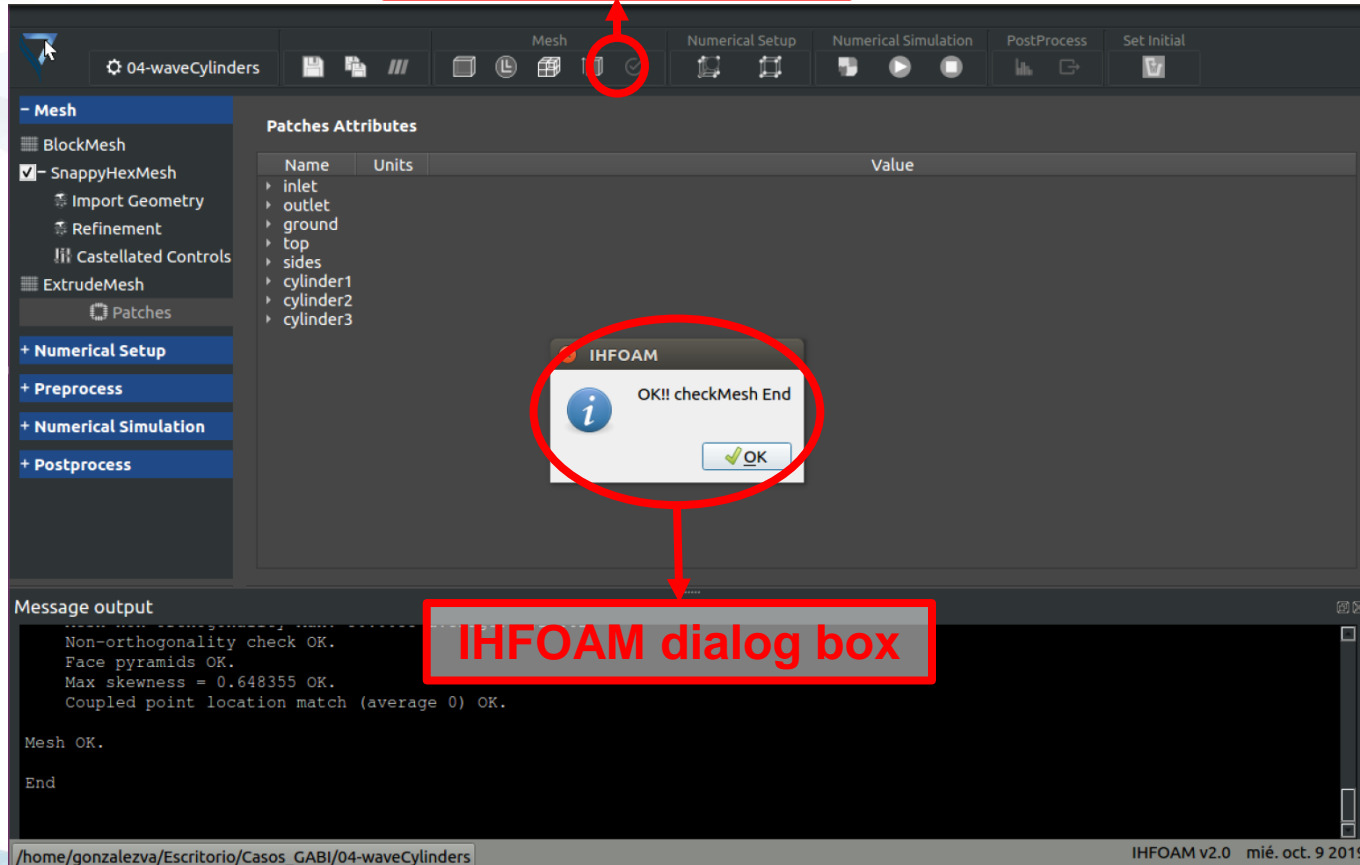
The screenshot shows the IHFOAM v2.0 software interface. The left sidebar contains a tree view with the following items: Mesh, BlockMesh, SnappyHexMesh (checked), Import Geometry, Refinement, Castellated Controls, ExtrudeMesh, and Patches (highlighted with a red circle). The main panel is titled 'Patches Attributes' and displays a table of boundary conditions for different patches.

| Name | Units | Value |
|-----------|---------|---------|
| inlet | | |
| outlet | | |
| ground | | |
| top | | |
| sides | | |
| cylinder1 | | |
| type | | wall |
| inG... | | 1 |
| nFa... | | 36480 |
| star... | | 3747876 |
| alp... | | empty |
| U | | empty |
| p_rgh | Kg*m/s2 | empty |
| cylinder2 | | |
| type | | wall |
| inG... | | 1 |
| nFa... | | 36480 |
| star... | | 3784356 |
| alp... | | empty |
| U | | empty |
| p_rgh | Kg*m/s2 | empty |
| cylinder3 | | |
| type | | wall |
| inG... | | 1 |
| nFa... | | 36480 |
| star... | | 3820836 |
| alp... | | empty |
| U | | empty |
| p_rgh | Kg*m/s2 | empty |

A red box with the text "Define a boudary condition variable for each new patch of the mesh" is overlaid on the interface, with a red arrow pointing to the 'Patches' option in the left sidebar.

The status bar at the bottom shows the file path: /home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders and the version: IHFOAM v2.0 mié. oct. 9 2019.

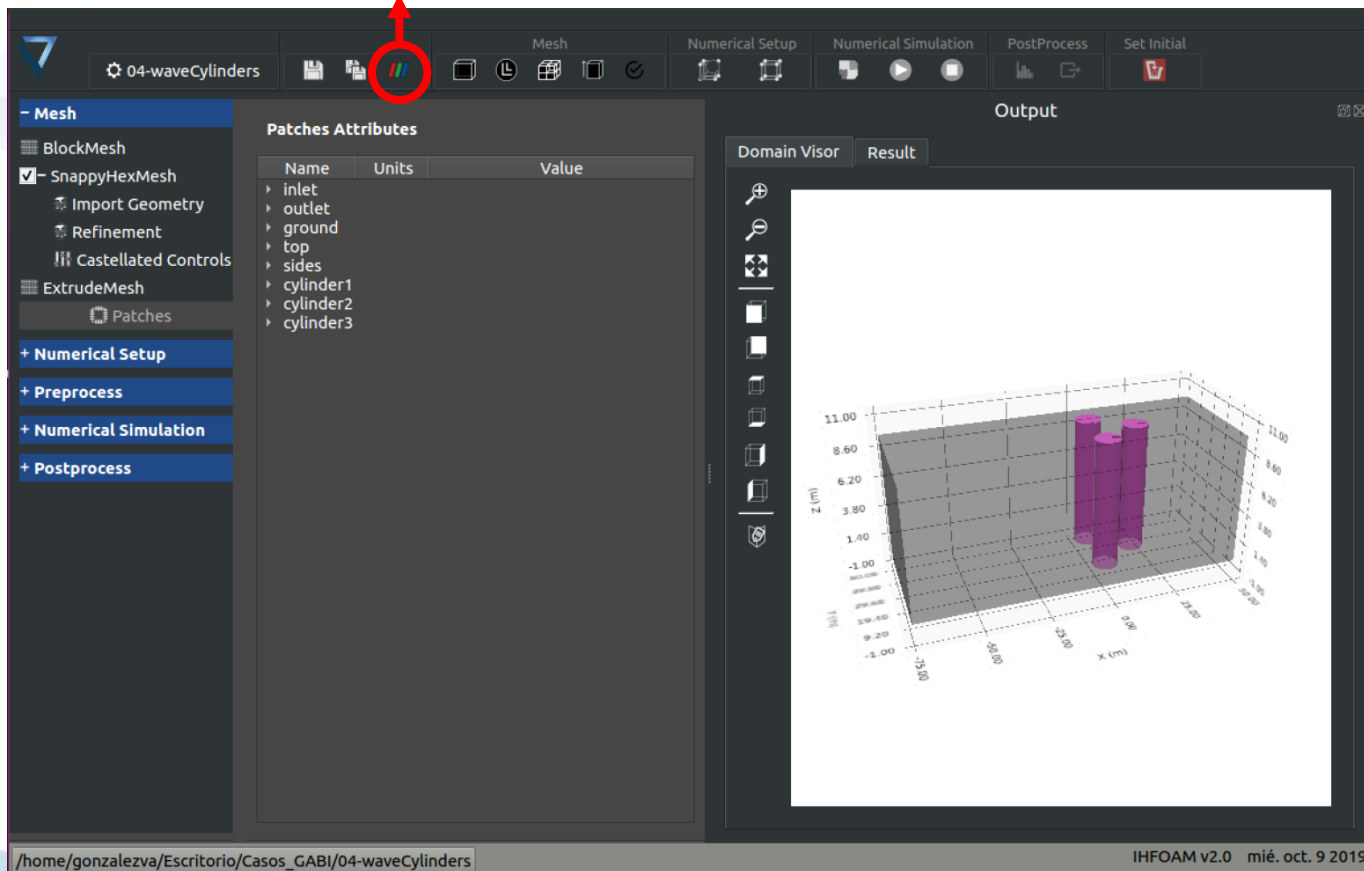
checkMesh button

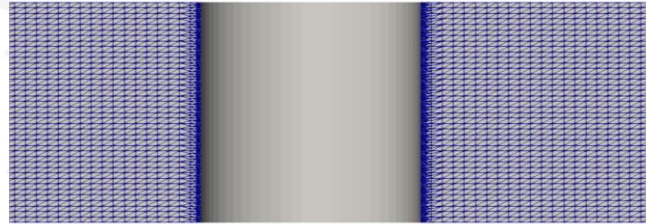
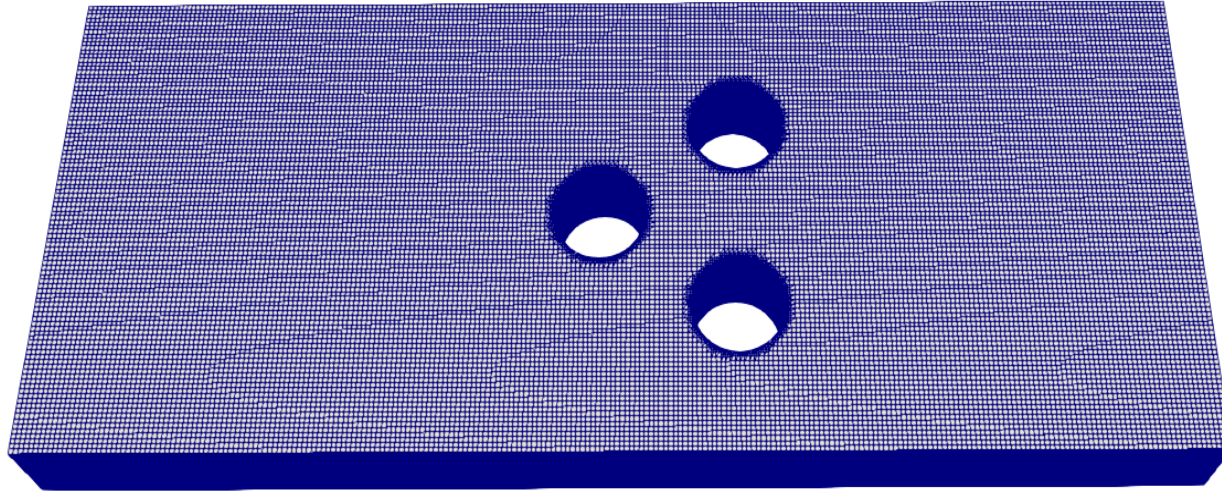


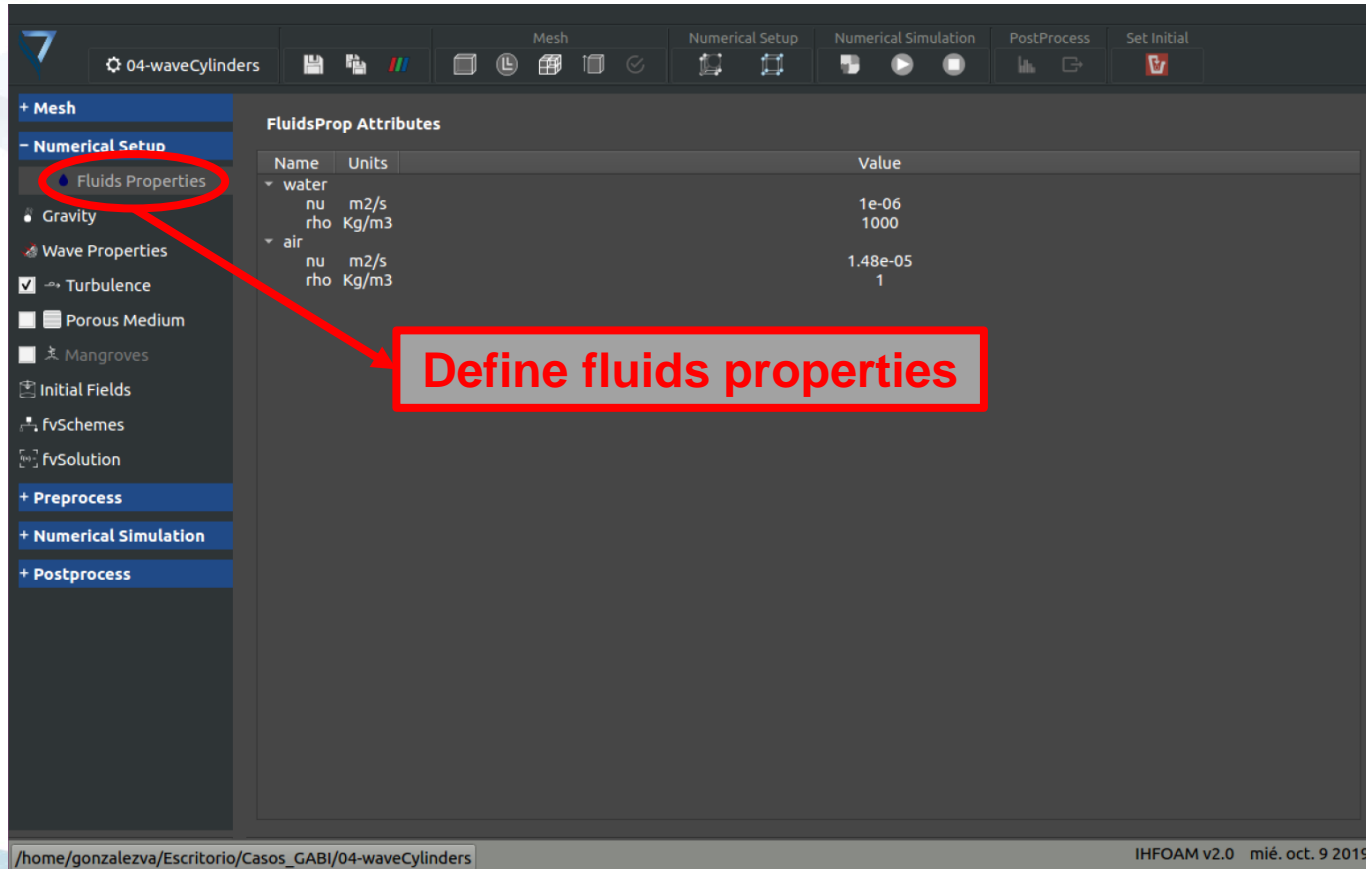
IHFOAM dialog box

ia
ENTAL

Paraview button







The screenshot displays the IHFOAM v2.0 software interface. The top menu bar includes 'Mesh', 'Numerical Setup', 'Numerical Simulation', 'PostProcess', and 'Set Initial'. The left sidebar shows a tree view with 'Numerical Setup' expanded, and 'Fluids Properties' highlighted. A red arrow points from this label to a red box with the text 'Define fluids properties'. The main panel, titled 'FluidsProp Attributes', contains a table with fluid properties.

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

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

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

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

A red arrow points from the 'Gravity' option in the sidebar to a red box containing the text 'Define gravity'.

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL

WaveProp Attributes

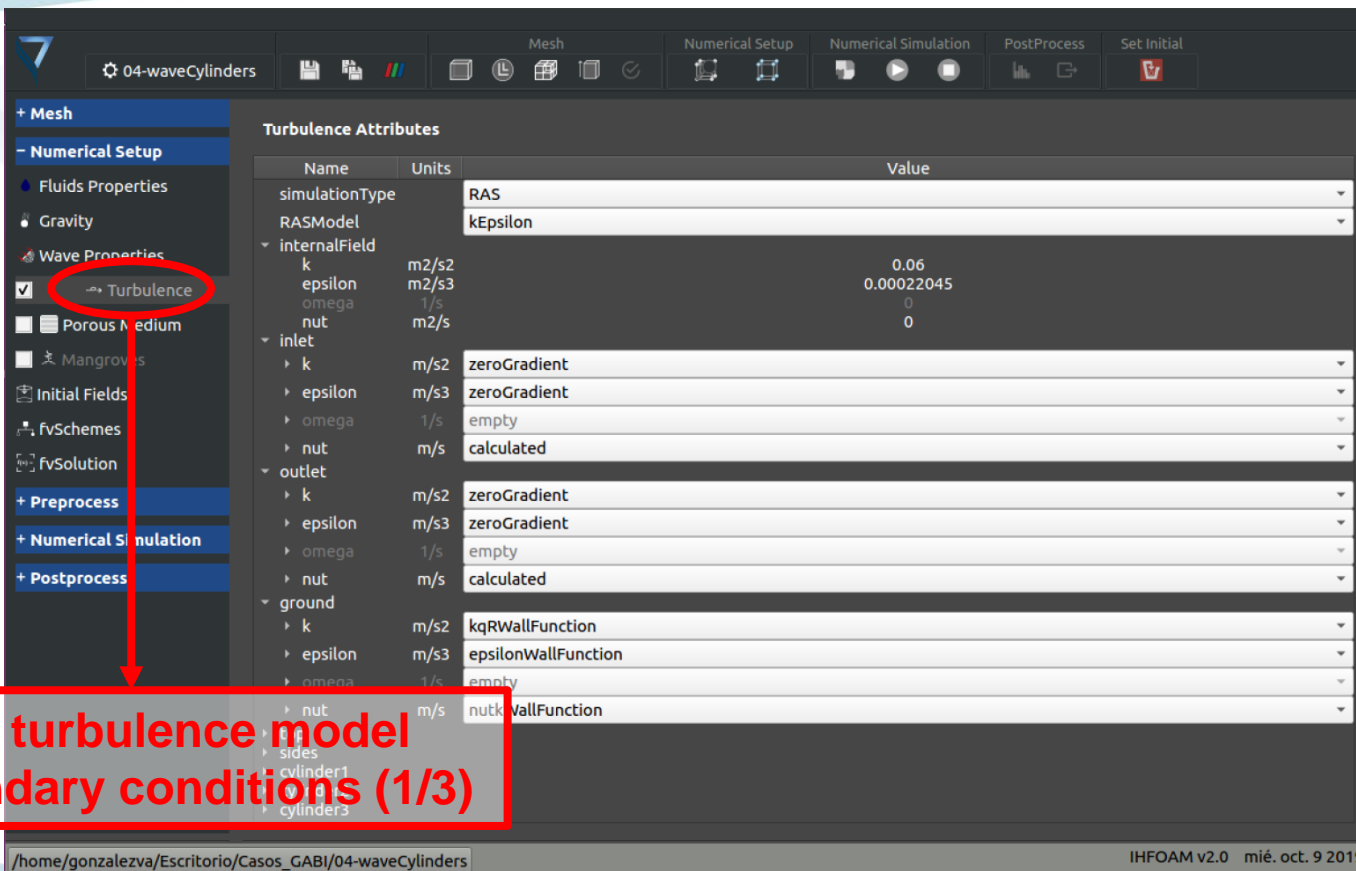
| Name | Units | Value |
|-----------------|--------|-------------------------------------|
| WaveTheoryRange | | <input checked="" type="checkbox"/> |
| waveGeneration | | <input checked="" type="checkbox"/> |
| patch | | inlet |
| waveModel | | StokesV |
| wavePeriod | s | 7 |
| waveHeight | m | 2.5 |
| waveAngle | degree | 0 |
| nPaddle | | 8 |
| activeAbsor... | | <input checked="" type="checkbox"/> |
| rampTime | s | 7 |
| restart | | <input type="checkbox"/> |
| waterDepth | | <input type="checkbox"/> |
| waveAbsorption | | <input checked="" type="checkbox"/> |
| patch | | outlet |
| waveModel | | shallowWaterAbsorption |
| nPaddle | | 4 |

**Define wave properties
(generation and absorption)**

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL



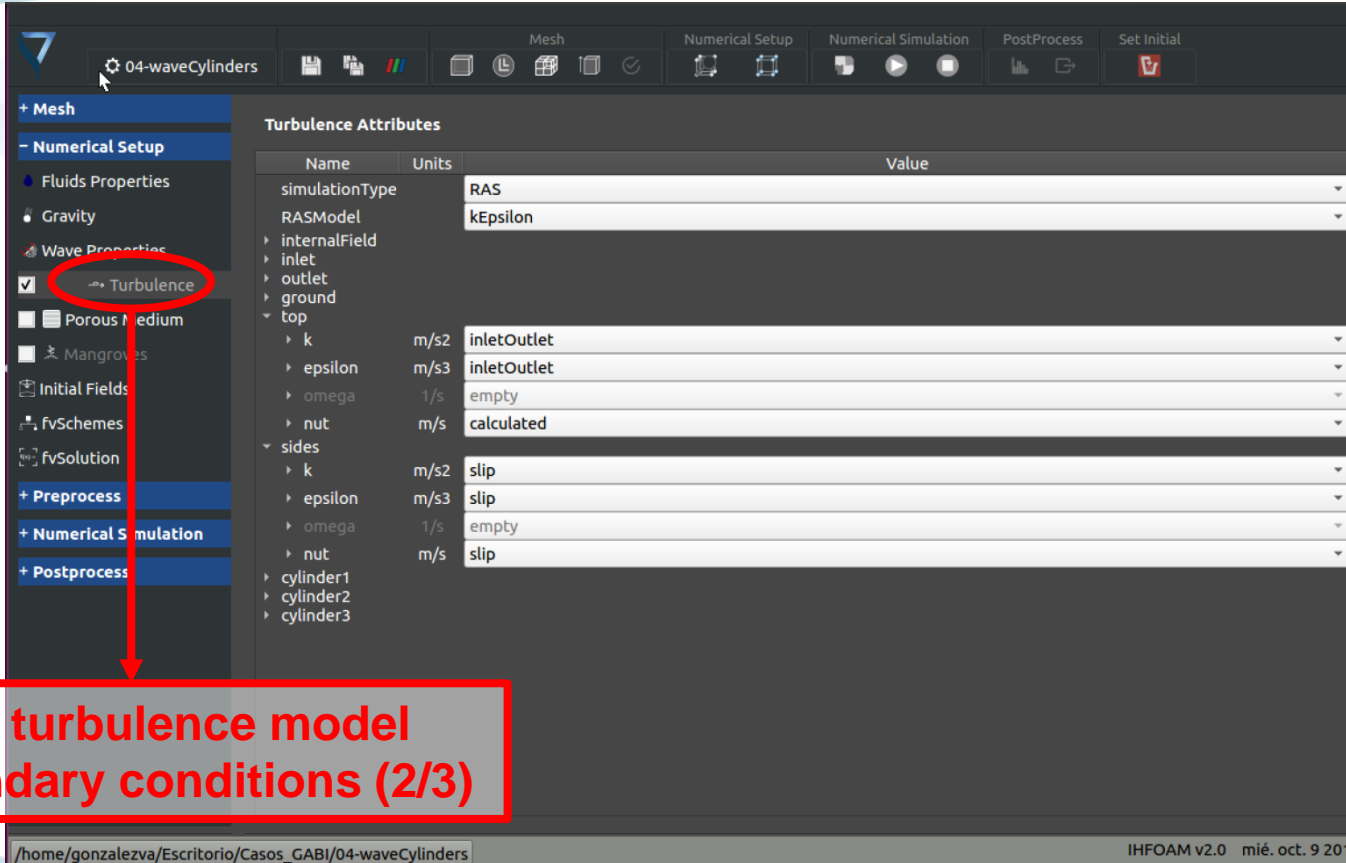
Turbulence Attributes

| Name | Units | Value |
|----------------|-------|---------------------|
| simulationType | | RAS |
| RASModel | | kEpsilon |
| internalField | | |
| k | m2/s2 | 0.06 |
| epsilon | m2/s3 | 0.00022045 |
| 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 |

Define turbulence model and boundary conditions (1/3)

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019



04-waveCylinders

Turbulence Attributes

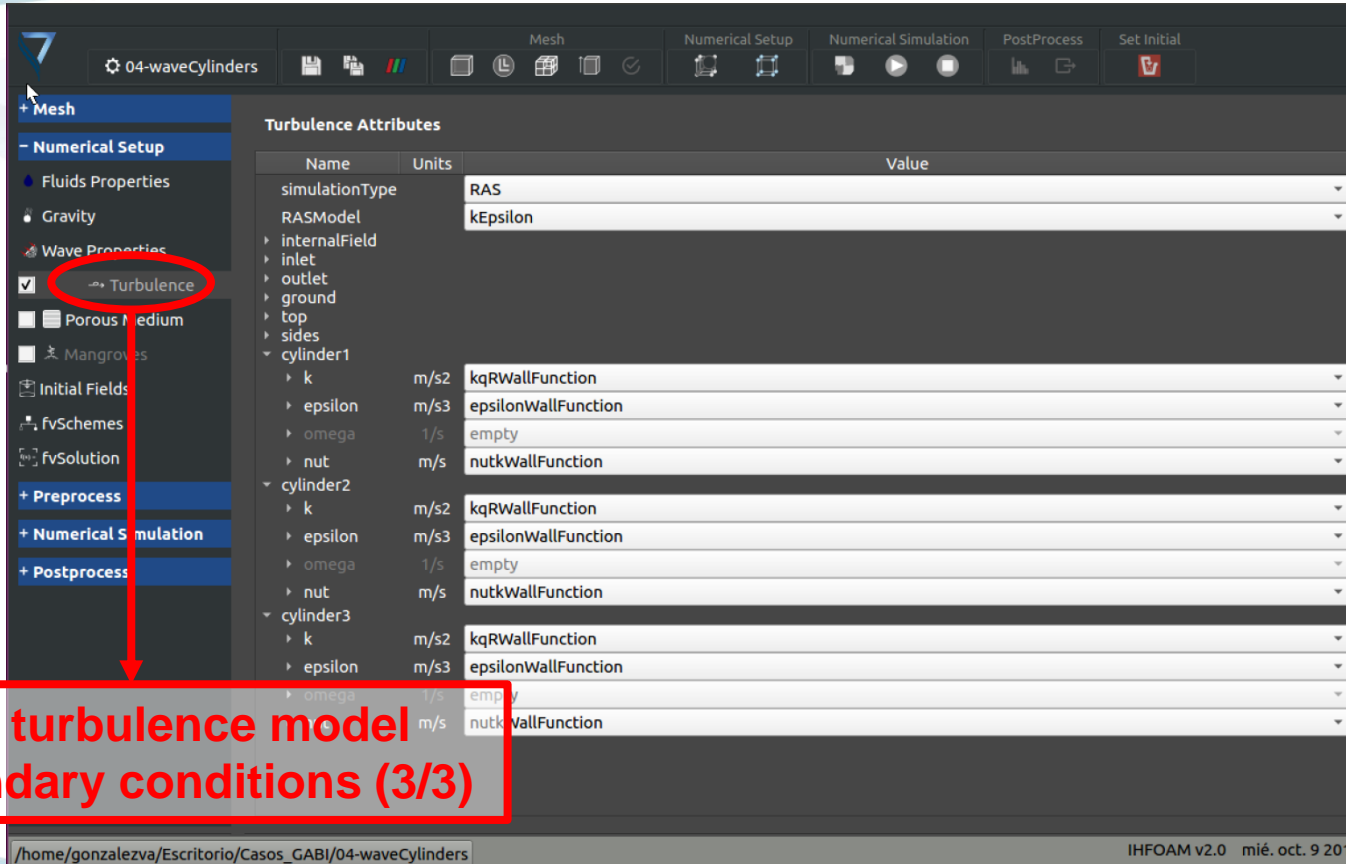
| Name | Units | Value |
|----------------|-------|-------------|
| simulationType | | RAS |
| RASModel | | kEpsilon |
| internalField | | |
| inlet | | |
| outlet | | |
| ground | | |
| top | | |
| k | m/s2 | inletOutlet |
| epsilon | m/s3 | inletOutlet |
| omega | 1/s | empty |
| nut | m/s | calculated |
| sides | | |
| k | m/s2 | slip |
| epsilon | m/s3 | slip |
| omega | 1/s | empty |
| nut | m/s | slip |
| cylinder1 | | |
| cylinder2 | | |
| cylinder3 | | |

Define turbulence model and boundary conditions (2/3)

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019

ia
ENTAL



04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

+ Mesh

- Numerical Setup

Fluids Properties

Gravity

Wave Properties

☒ Turbulence

☐ Porous Medium

☐ Mangroves

Initial Fields

FvSchemes

FvSolution

+ Preprocess

+ Numerical Simulation

+ Postprocess

Define turbulence model and boundary conditions (3/3)

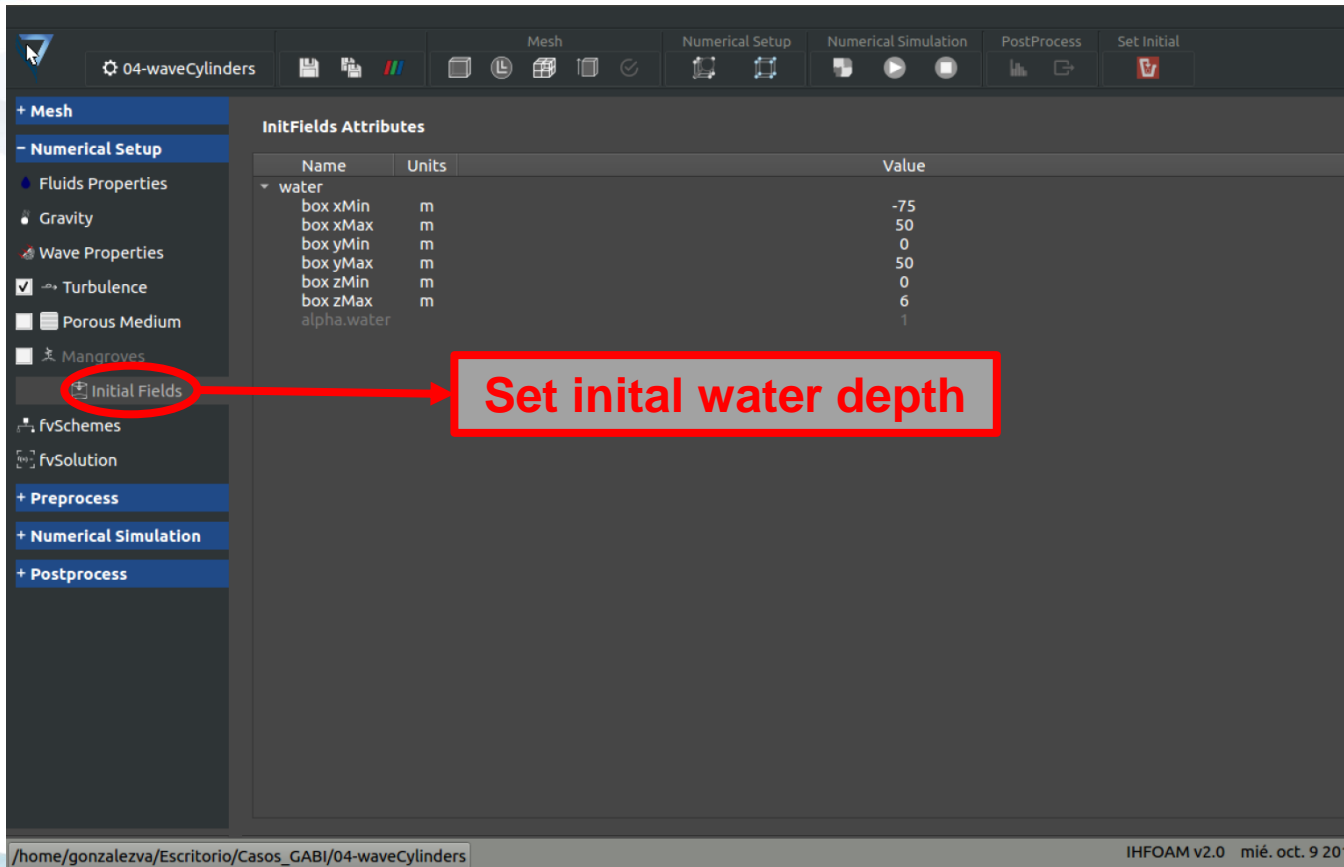
Turbulence Attributes

| Name | Units | Value |
|----------------|-------|---------------------|
| simulationType | | RAS |
| RASModel | | kEpsilon |
| internalField | | |
| inlet | | |
| outlet | | |
| ground | | |
| top | | |
| sides | | |
| cylinder1 | | |
| k | m/s2 | kqRWallFunction |
| epsilon | m/s3 | epsilonWallFunction |
| omega | 1/s | empty |
| nut | m/s | nutkWallFunction |
| cylinder2 | | |
| k | m/s2 | kqRWallFunction |
| epsilon | m/s3 | epsilonWallFunction |
| omega | 1/s | empty |
| nut | m/s | nutkWallFunction |
| cylinder3 | | |
| k | m/s2 | kqRWallFunction |
| epsilon | m/s3 | epsilonWallFunction |
| omega | 1/s | empty |
| nut | m/s | nutkWallFunction |

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

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, 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 | -75 |
| box xMax | m | 50 |
| box yMin | m | 0 |
| box yMax | m | 50 |
| box zMin | m | 0 |
| box zMax | m | 6 |
| 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/04-waveCylinders and the version is IHFOAM v2.0, dated miércoles, 9 oct. 2019.

setFields button

The screenshot displays the IHFOAM software interface. The top toolbar includes buttons for Mesh, Numerical Setup (highlighted with a red circle and arrow), Numerical Simulation, PostProcess, and Set Initial. The left sidebar shows a tree view with categories like Mesh, Numerical Setup, Fluids Properties, Gravity, Wave Properties, Turbulence, Porous Medium, Mangroves, Initial Fields, fvSchemes, fvSolution, Preprocess, Numerical Simulation, and Postprocess. The central panel is divided into 'InitFields Attributes' and 'Output'. The 'InitFields Attributes' table lists parameters for 'water' with units and values. The 'Output' panel shows a 3D domain visualization with three purple cylinders. A red circle highlights the 'setFields' button in the toolbar. A red oval highlights the 'IHFOAM dialog box' showing a message: 'OK!! rm -rf 0 && cp -r 0.orig 0 && setFields End'. The bottom panel shows the 'Message output' log.

| Name | Units | Value |
|-------------|-------|-------|
| water | | |
| box xMin | m | -75 |
| box xMax | m | 50 |
| box yMin | m | 0 |
| box yMax | m | 50 |
| box zMin | m | 0 |
| box zMax | m | 6 |
| alpha.water | | 1 |

Message output

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

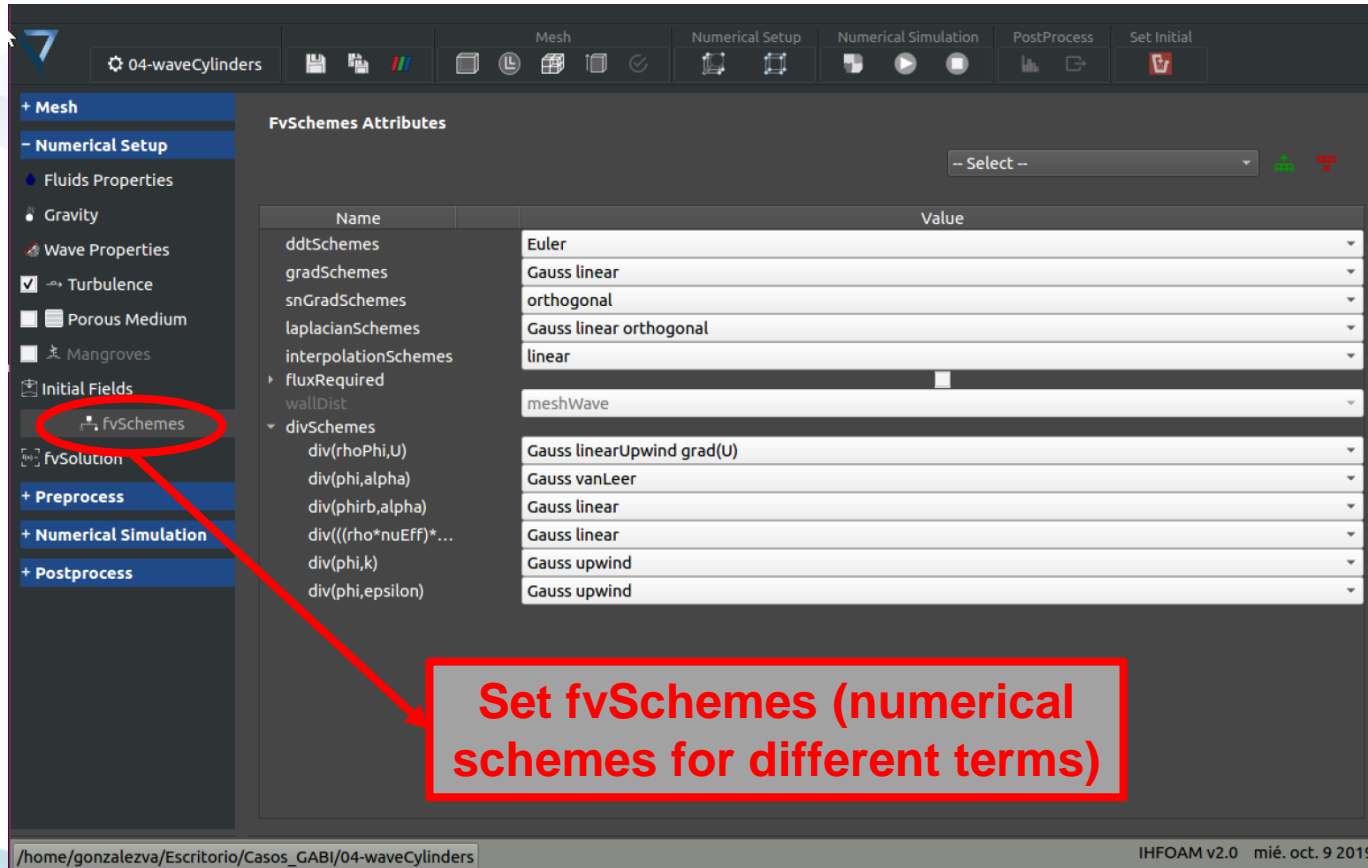
Setting field region values
Adding cells with centre within boxes 1((-75 0 0) (50 50 6))
Setting internal values of volScalarField alpha.water

End
```

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019

IHFOAM dialog box

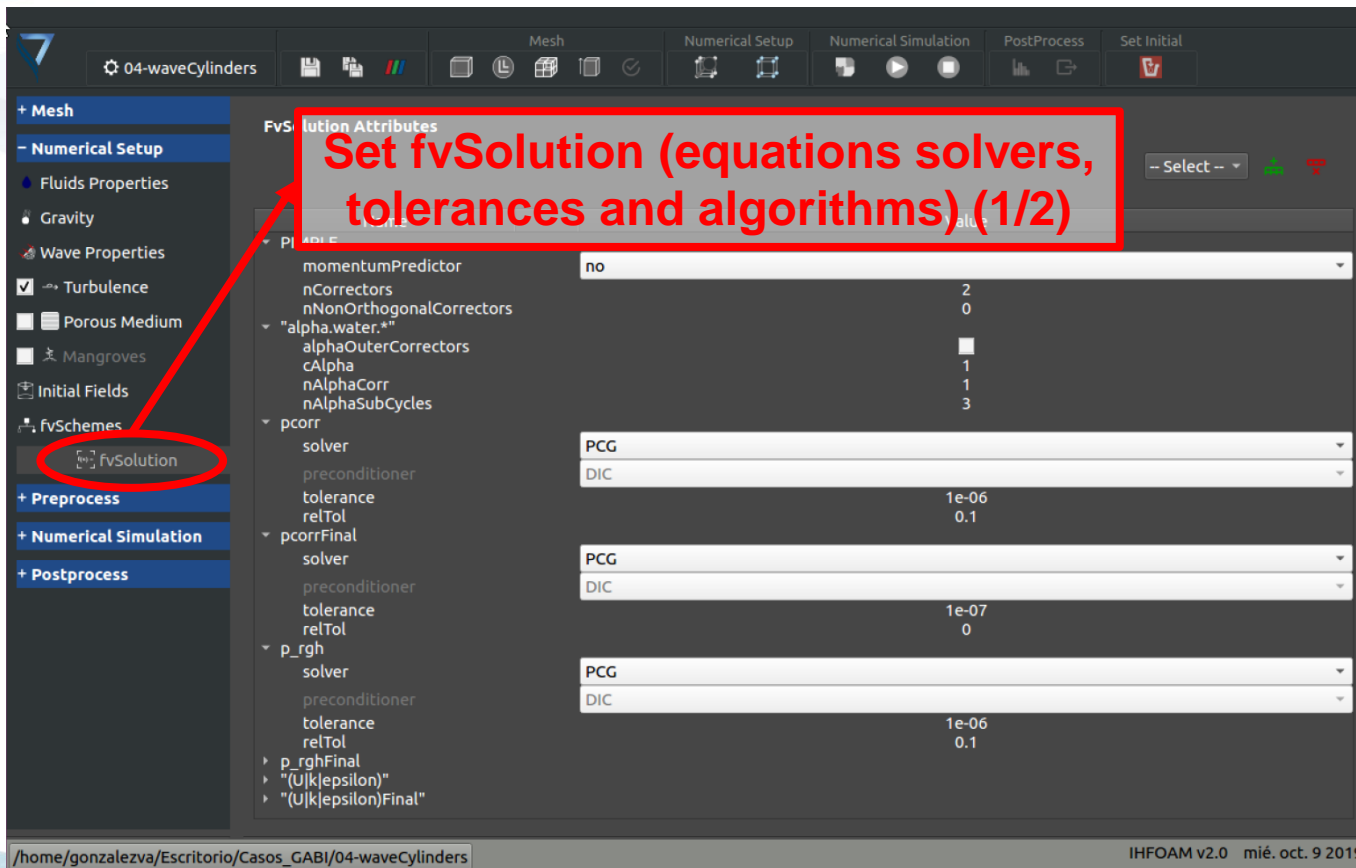


The screenshot shows the OpenFOAM GUI with the 'FvSchemes Attributes' panel. The left sidebar has a red circle around the 'fvSchemes' option. A red arrow points from this circle to a red-bordered box with the text: **Set fvSchemes (numerical schemes for different terms)**.

The 'FvSchemes Attributes' panel displays a table of numerical schemes for various terms:

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

The bottom status bar shows the path: /home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders and the version: IHFOAM v2.0 mié. oct. 9 2019.



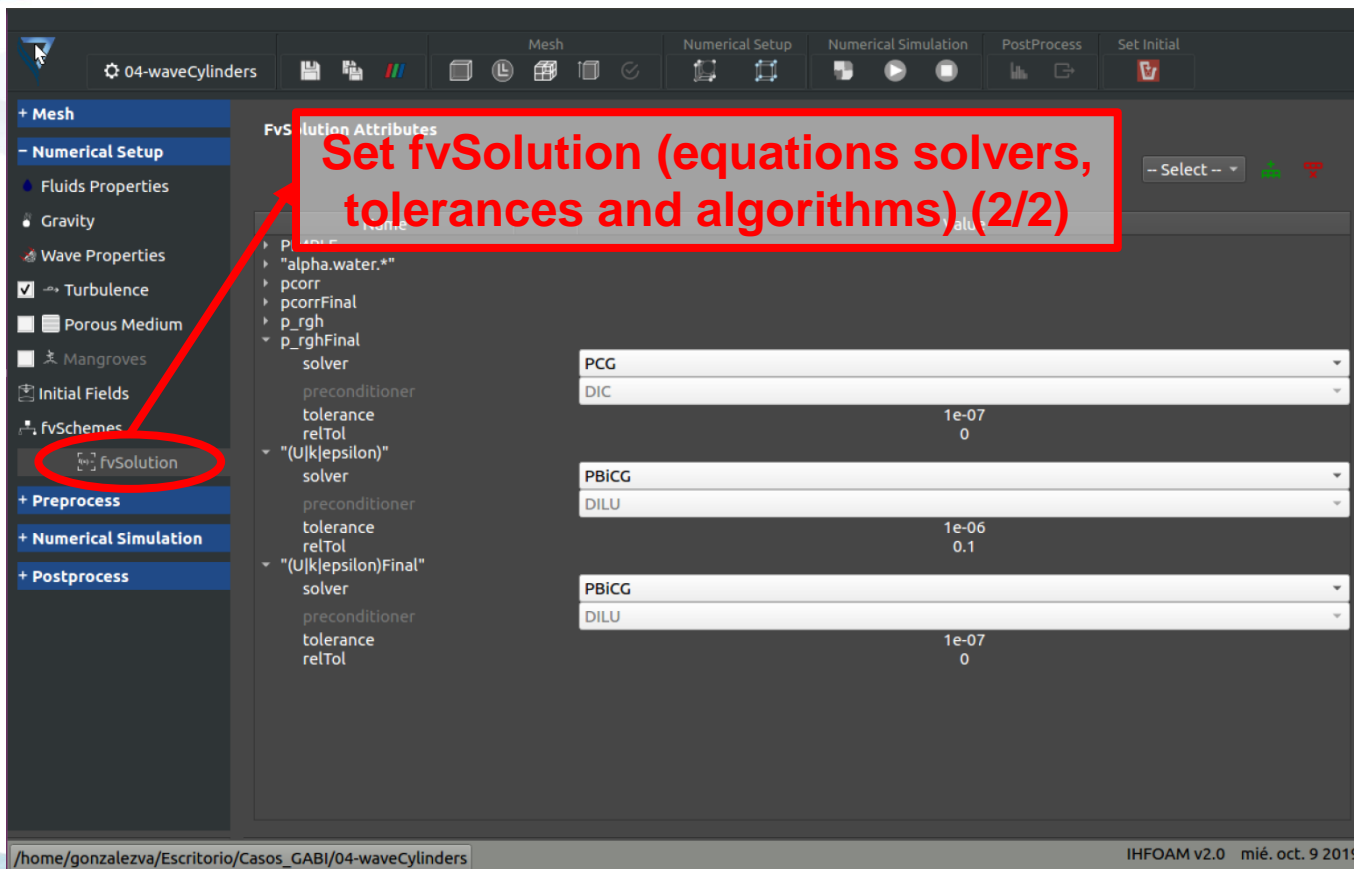
Set fvSolution (equations solvers, tolerances and algorithms) (1/2)

The screenshot shows the 'fvSolution' configuration panel in the IHFOAM v2.0 software. The left sidebar lists various setup categories, with 'fvSolution' highlighted. The main panel displays the configuration for the 'fvSolution' attributes, organized into a tree structure. The configuration includes settings for the 'momentumPredictor', 'alpha.water.*', 'pcorr', 'pcorrFinal', and 'p_rgh' solvers, along with their respective preconditioners, tolerances, and relative tolerances.

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

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019



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

FvSolution Attributes

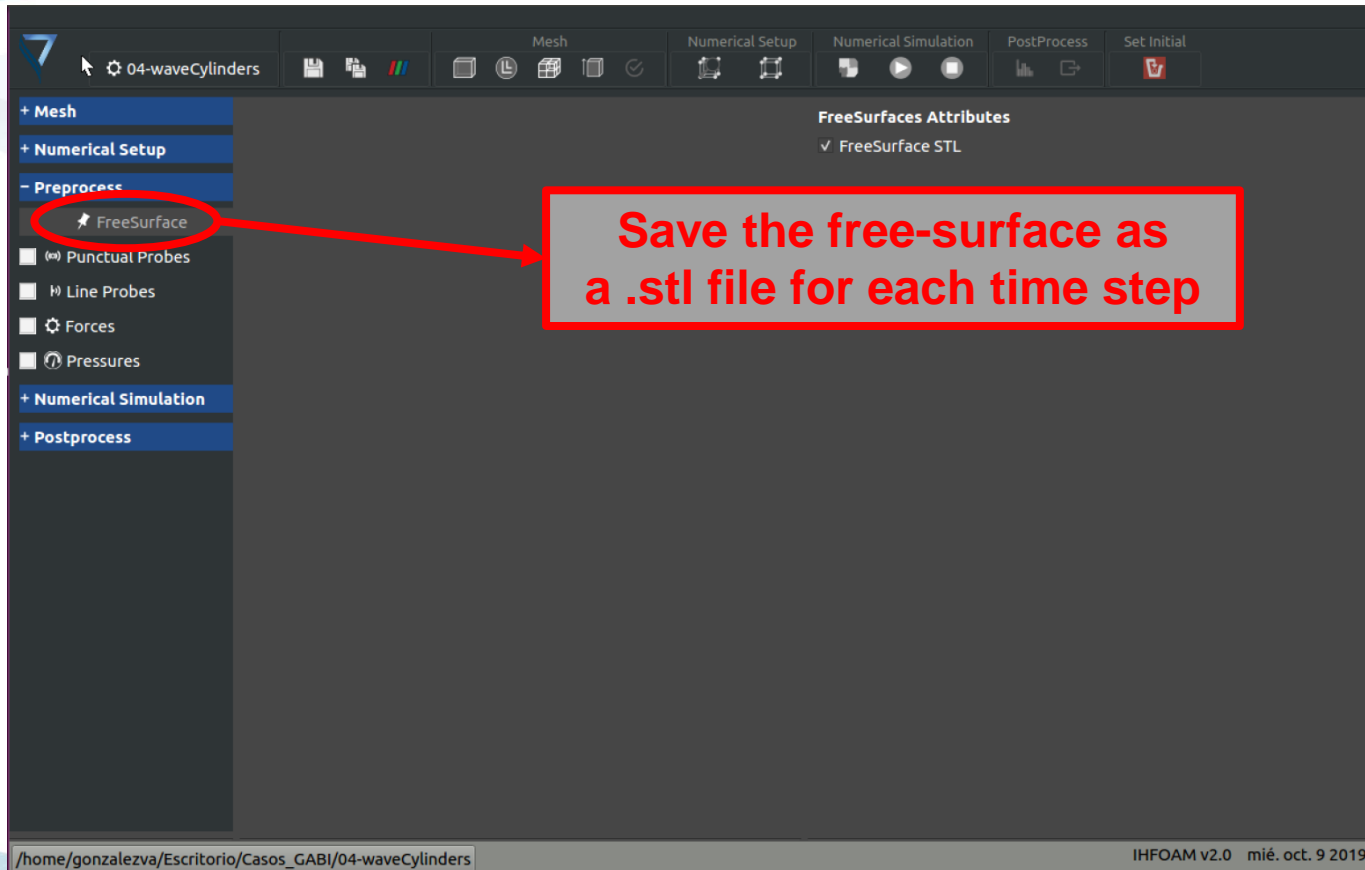
PCG
DIC
tolerance 1e-07
relTol 0

PBiCG
DILU
tolerance 1e-06
relTol 0.1

PBiCG
DILU
tolerance 1e-07
relTol 0

/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019



IHFOAM

File Edit View Help

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

- Mesh

- BlockMesh
- ✓ SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- Patches

- Numerical Setup

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

- Preprocess

- FreeSurface
- Punctual Probes
- Forces *
- Pressure

+ Numerical Simulation

+ Postprocess

Attributes

Add cylinder1

| Name | Units | Value |
|-------------------|----------|-------------------------------------|
| Forcescylinder1_1 | | |
| outputControl | timeStep | |
| outputInterval | | 0 |
| writeFields | | <input checked="" type="checkbox"/> |
| CoR | | |
| CoR X | m | 0 |
| CoR Y | m | 0 |
| CoR Z | m | 0 |

Output

Domain Visor Result

Set patch force sensors (1/3)

/home/gabi/Desktop/IHFOAM/04-waveCylinders

IHFOAM v1.0 mié. sept. 4 2019

ihFOAM

File Edit View Help

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

- Mesh

- BlockMesh
- ✓ SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- Patches

- Numerical Setup

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

- Preprocess

- FreeSurface
- Punctual Probes
- Forces *
- Pressure

+ Numerical Simulation

+ Postprocess

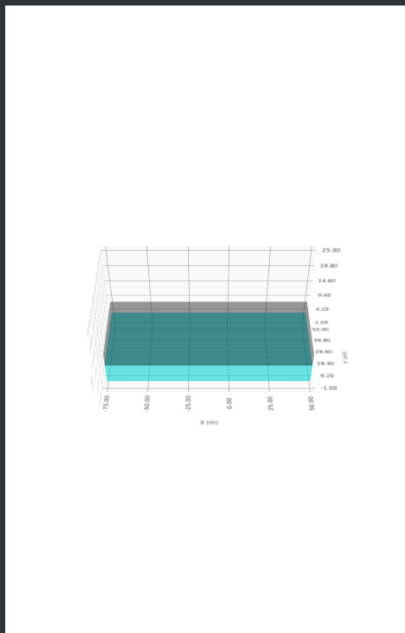
Attributes

Add cylinder2

| Name | Units | Value |
|-------------------|----------|-------------------------------------|
| Forcescylinder2_1 | | |
| outputControl | timeStep | |
| outputInterval | | 0 |
| writeFields | | <input checked="" type="checkbox"/> |
| CoR | | |
| CoR X | m | 0 |
| CoR Y | m | 0 |
| CoR Z | m | 0 |

Output

Domain Visor Result



Set patch force sensors (2/3)

/home/gabi/Desktop/IHFOAM/04-waveCylinders

ihFoam v1.0 mié. sept. 4 2019

ihFOAM

File Edit View Help

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

- Mesh

- BlockMesh
- SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- Patches

- Numerical Setup

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

- Preprocess

- FreeSurface
- Punctual Probes
- Forces *
- Pressure

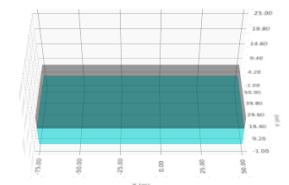
Attributes

Add cylinder3

| Name | Units | Value |
|-------------------|----------|-------------------------------------|
| Forcescylinder3_1 | | |
| outputControl | timeStep | |
| outputInterval | | 0 |
| writeFields | | <input checked="" type="checkbox"/> |
| CoR | | |
| CoR X | m | 0 |
| CoR Y | m | 0 |
| CoR Z | m | 0 |

Output

Domain Visor Result



Set patch force sensors (3/3)

ihFOAM

File Edit View Help

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

- Mesh

- BlockMesh
- SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- Patches

- Numerical Setup

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

- Preprocess

- FreeSurface
- Punctual Probes
- Line Probes
- Pressures *

+ Numerical Simulation

+ Postprocess

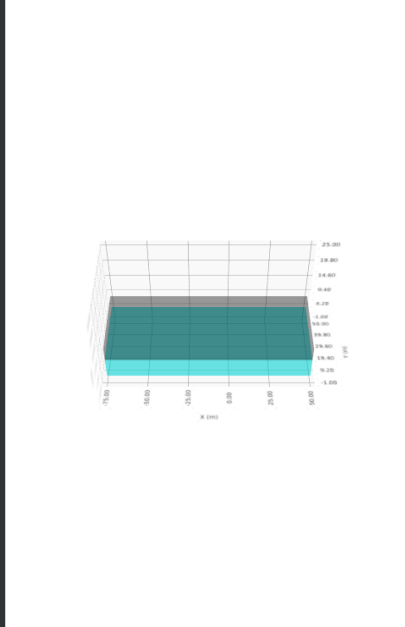
Attributes

Add cylinder1

| Name | Units | Value |
|---------------------|-----------|-------|
| Presscylinder1_1 | | |
| outputControl | timeStep | |
| outputInterval | | 0 |
| surfaceFormat | vtk | |
| interpolationScheme | cellPoint | |

Output

Domain Visor Result



Set patch pressure sensors (1/3)

/home/gabi/Desktop/IHFOAM/04-waveCylinders

IHFOAM v1.0 mié. sept. 4 2019

ihFOAM

File Edit View Help

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

- Mesh

- BlockMesh
- SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- Patches

- Numerical Setup

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

- Preprocess

- FreeSurface
- Punctual Probes
- Line Probes
- Pressures *

+ Numerical Simulation

+ Postprocess

Attributes

Add cylinder2

| Name | Units | Value |
|---------------------|-----------|-------|
| Presscylinder2_1 | | |
| outputControl | timeStep | |
| outputInterval | | 0 |
| surfaceFormat | vtk | |
| interpolationScheme | cellPoint | |

Output

Domain Visor Result

Set patch pressure sensors (2/3)

/home/gabi/Desktop/IHFOAM/04-waveCylinders

ihFoam v1.0 mié. sept. 4 2019

ihFoam

File Edit View Help

04-waveCylinders

Mesh Numerical Setup Numerical Simulation PostProcess Set Initial

- Mesh

- BlockMesh
- SnappyHexMesh
 - Import Geometry
 - Refinement
 - Castellated Controls
- Patches

- Numerical Setup

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

- Preprocess

- FreeSurface
- Punctual Probes
- Line Probes
- Pressures *

+ Numerical Simulation

+ Postprocess

Attributes

Add cylinder3

| Name | Units | Value |
|---------------------|-----------|-------|
| Presscylinder3_1 | | |
| outputControl | timeStep | |
| outputInterval | | 0 |
| surfaceFormat | vtk | |
| interpolationScheme | cellPoint | |

Output

Domain Visor Result

Set patch pressure sensors (3/3)

/home/gabi/Desktop/IHFOAM/04-waveCylinders

ihFoam v1.0 mié. sept. 4 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 | 50 |
| 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 interface, the file path is shown as /home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders and the version is IHFOAM v2.0, dated miércoles, 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 (expanded), Parameters, Simulation (circled in red), Performance, Residuals, and Postprocess. The main panel displays the 'Simulation Attributes' table. The table has three columns: Name, Units, and Value. The rows are: Numerical Solver (interFoam), Execution Type (Parallel), method (hierarchical), Decompose X (2), Decompose Y (2), Decompose Z (1), and numberOfSubdomains (4). The 'decomposePar' button is located in the top toolbar, and the 'Run/Execute' button is also in the top toolbar. The 'cleanCase' button is located in the top toolbar. The 'Simulation' button is located in the left sidebar. The 'Run OpenFOAM case' button is located in the bottom right corner of the main panel.

| 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/04-waveCylinders

IHFOAM v2.0 mié. oct. 9 2019

decomposePar button

Simulation Attributes

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

Message output

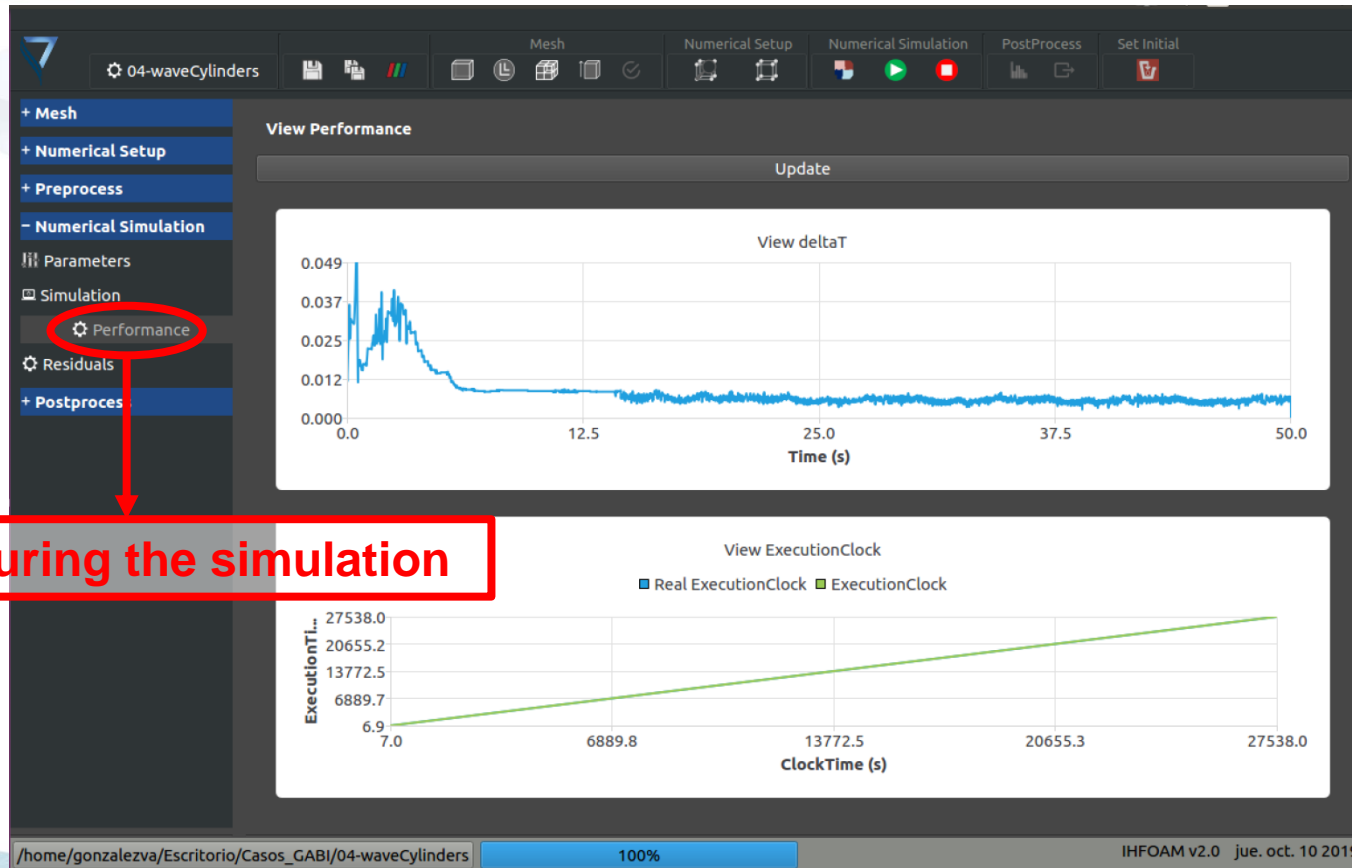
```
Time = 0  
  
Processor 0: field transfer  
Processor 1: field transfer  
Processor 2: field transfer  
Processor 3: field transfer  
  
End
```

Case decomposed correctly

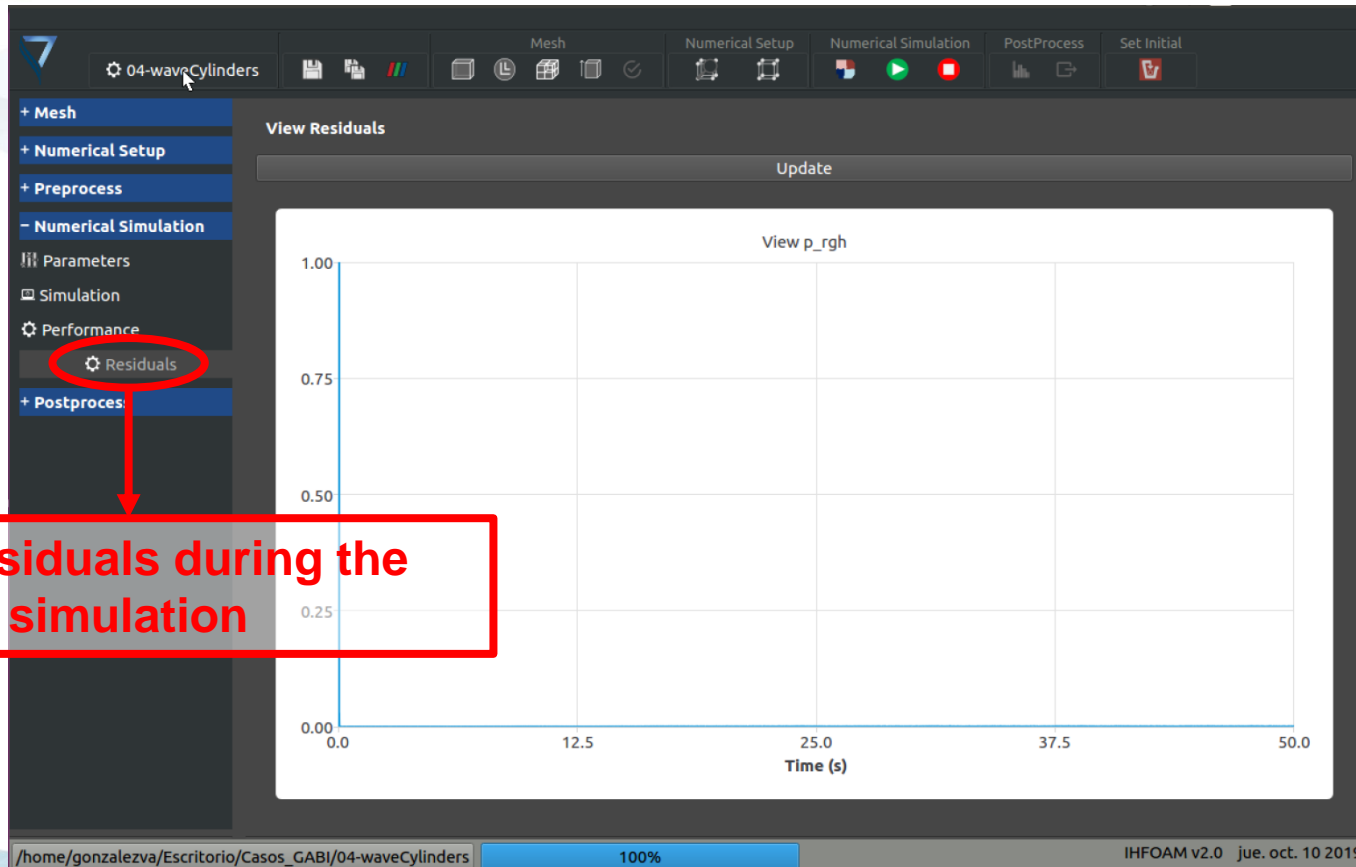
Run/Execute button

The screenshot displays the IHFOAM v2.0 software interface. The top toolbar includes tabs for Mesh, Numerical Setup, Numerical Simulation, PostProcess, and Set Initial. The 'Numerical Simulation' tab is selected, and the 'Run/Execute' button (a play icon) is highlighted with a red circle and an arrow pointing to the text 'Run/Execute button' above it. The 'Simulation Attributes' panel shows the 'Numerical Solver' set to 'interFoam' and the 'Execution Type' set to 'Parallel'. A message box in the center of the screen displays the text 'OK!! mpirun -np 4 interFoam -parallel End' with an 'OK' button. The 'Domain Visor' panel on the right shows a 3D visualization of the simulation domain with two cylinders. The 'Message output' window at the bottom shows the execution log, and the word 'End' is highlighted with a red circle. The status bar at the bottom indicates the file path '/home/gonzalezva/Escritorio/Casos_GABI/04-waveCylinders', the progress '100%', and the version 'IHFOAM v2.0' dated 'jue. oct. 10 2019'.

Case correctly executed!!



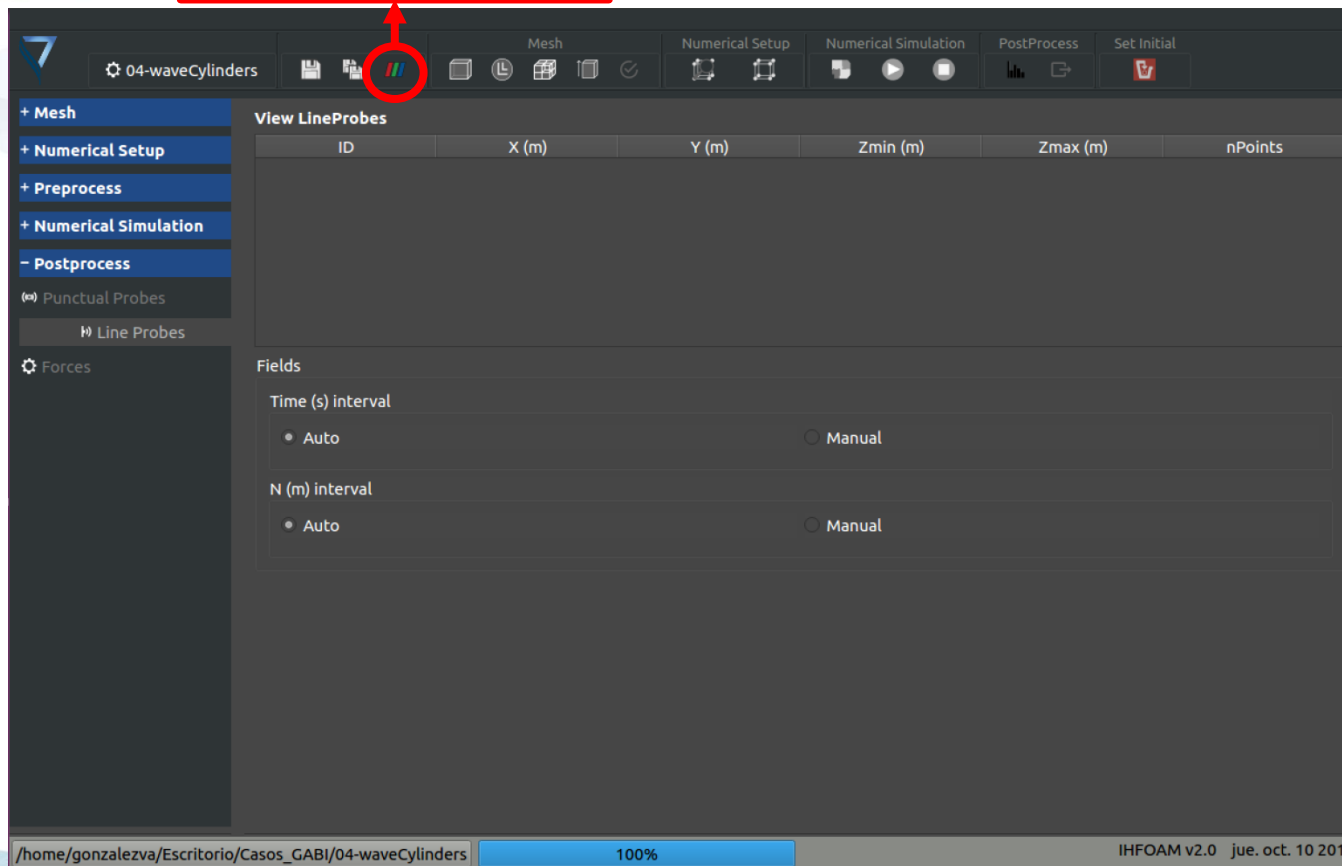
ia
ENTAL



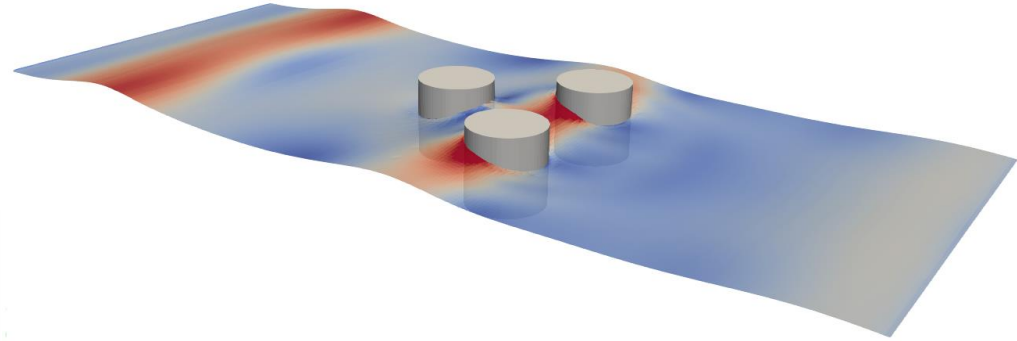
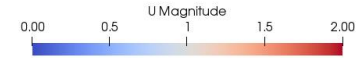
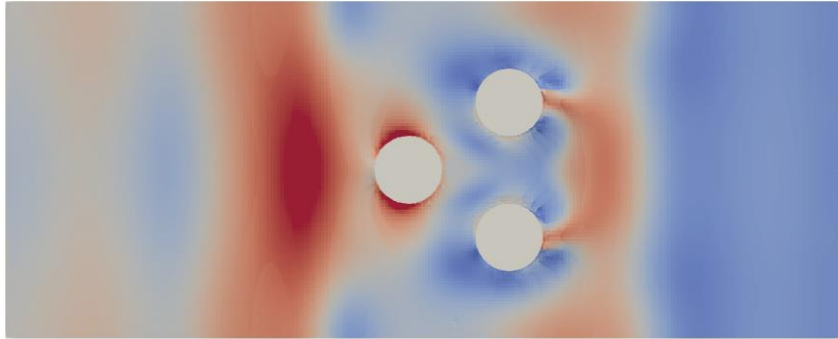
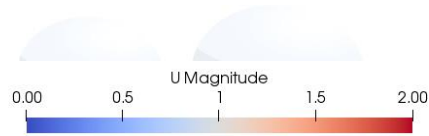
plot residuals during the
simulation

ia
ENTAL

Paraview button

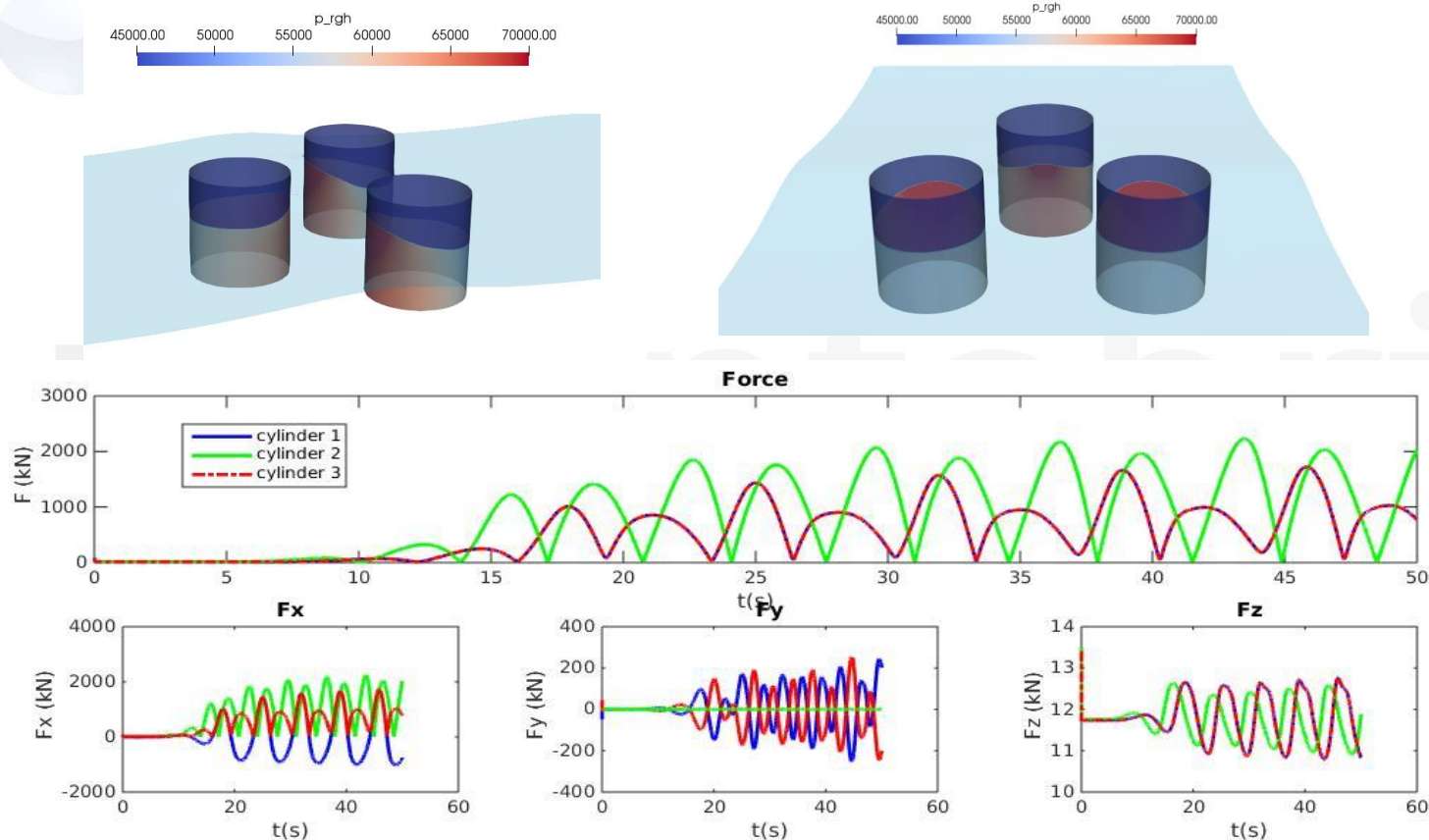


ia
ENTAL



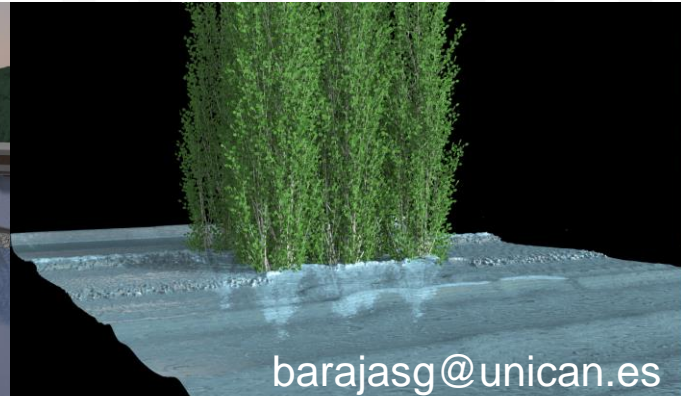
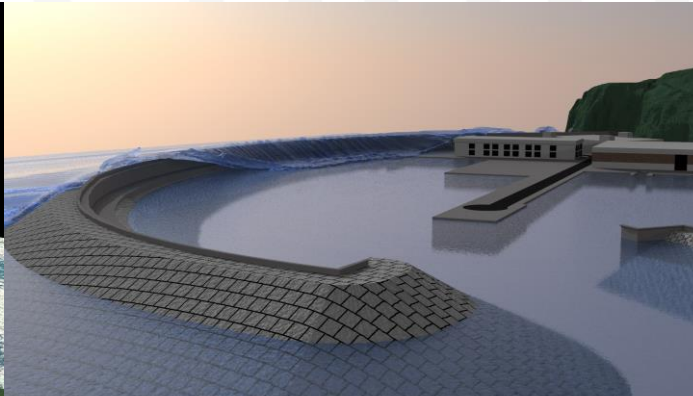
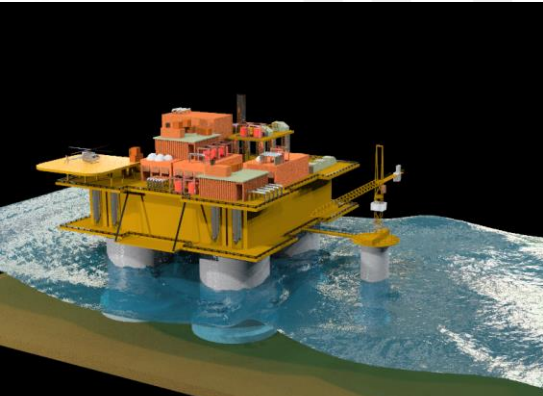
INSTITUTO DE HIDRÁULICA AMBIENTAL
UNIVERSIDAD DE CANTABRIA

- Postprocessing using Matlab (forces on each cylinder , taken from the *postProcessing* folder):





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



barajasg@unican.es