

Utilização do código OpenFoam na resolução de problemas de hidrodinâmica

José M. P. Conde

UNIDEMI - Unidade de Investigação e Desenvolvimento
em Engenharia Mecânica e Industrial

Faculdade de Ciências e Tecnologia

Universidade Nova de Lisboa

Portugal

Resumo

- Significado de OpenFOAM
- Máquina virtual – VirtualBox
- OpenFoam.com
- OpenFoam.net
- IHFoam
- OlaFoam
- Verificação e validação
- Exemplos

Significado de OpenFOAM

- Open – aberto (em inglês)
- FOAM – Field Operation And Manipulation.
- é uma caixa de ferramentas (TOOLBOX) C++ para o desenvolvimento de solucionadores (SOLVERS) numéricos customizados e utilitários de pré/pós-processamento para a solução de problemas de mecânica contínua, incluindo mais proeminentemente dinâmica de fluidos computacional (CFD).

Máquina virtual

- Windows, existem versões Windows mas utilizam emuladores
- LINUX, as diferentes versões funcionam em linux
- Mas o computador que utilizo tem Windows, o que posso fazer?
- Instalar o VirtualBox, que é emulador de Linux (e outros sistemas).

VirtualBOX

<https://www.virtualbox.org/wiki/Downloads>

The screenshot shows the VirtualBox website's download page. At the top, there is a navigation bar with links for Home, Download, Documentation, and Community, along with a search box. The main heading is "Download VirtualBox". Below this, a paragraph explains the PUEL license. The page is divided into two main columns. The left column, titled "VirtualBox Platform Packages", lists various operating systems: Windows hosts, macOS / Intel hosts, macOS / Apple Silicon hosts, Linux distributions, Solaris hosts, and Solaris 11 IPS hosts. A red arrow points to the "Windows hosts" link. The right column, titled "VirtualBox Extension Pack", contains a license agreement text, a link to the FAQ, and three buttons: "PUEL License FAQ", "PUEL License Text", and "Accept and download". A red arrow points to the "Accept and download" button. At the bottom of the page, there are three boxes: "Change Log" (with a link to "List of changes"), "File Checksums" (with links to "SHA256 checksums" and "MD5 checksums"), and "User Guide" (with links to "User Guide (PDF)" and "User Guide (HTML)").

VirtualBox

Home Download Documentation Community

Download VirtualBox

The VirtualBox Extension Pack is available for personal and educational use on this page under the PUEL license. The VirtualBox Extension Pack is also available under commercial or enterprise terms. By downloading, you agree to the terms and conditions of the respective license.

VirtualBox Platform Packages

VirtualBox 7.1.4 platform packages

- Windows hosts
- macOS / Intel hosts
- macOS / Apple Silicon hosts
- Linux distributions
- Solaris hosts
- Solaris 11 IPS hosts

Platform packages are released under the terms of the [GPL version 3](#)

VirtualBox Extension Pack

VirtualBox 7.1.4 Extension Pack

This VirtualBox Extension Pack Personal Use and Educational License governs your access to and use of the VirtualBox Extension Pack. It does not apply to the VirtualBox base package and/or its source code, which are licensed under version 3 of the GNU General Public License "GPL".

See our [FAQ](#) for answers to common questions.

VirtualBox Extension Pack Personal Use and Educational License

[PUEL License FAQ](#) [PUEL License Text](#) [Accept and download](#)

Change Log

[List of changes](#)

File Checksums

[SHA256 checksums](#) [MD5 checksums](#)

User Guide

[User Guide \(PDF\)](#) [User Guide \(HTML\)](#)



Ferramentas



Nova



Adicionar



Definições



Ignorar



Iniciar



ubuntu18.04

Desligada



ubuntu1604

Desligada



win xp32bit

Desligada



Windows XP 32bits

Desligada



ubuntu

Desligada

Geral

Nome: ubuntu18.04
Sistema Operativo: Ubuntu (64-bit)

Sistema

Memória Base: 4096 MB
Processadores: 2
Ordem de Arranque: Disquete, Ótico, Disco Rígido
Aceleração: Nested Paging, KVM Paravirtualization

Display

Memória de Vídeo: 16 MB
Controlador de Gráficos: VBoxVGA
Servidor do Ecrã Remoto: Desativado
Gravação: Desativada

Armazenamento

Controlador: IDE
IDE Secondary Device 0: [Unidade Ótica] Vazio
Controlador: SATA
Porta SATA 0: ubuntu18.04.vdi (Normal, 50.00 GB)

Áudio

Controlador Anfitrião: Windows DirectSound
Controlador: ICH AC97

Rede

Adaptador 1: Intel PRO/1000 MT Desktop (NAT)

USB

Controlador USB: OHCI, EHCI
Filtros de Dispositivo: 0 (0 ativos)

Pastas partilhadas

Nenhuma

Descrição

Nenhuma

Pré-visualizar

ubuntu18.04

Basic Expert

Search settings 🔍

- Geral**
- Sistema
- Display
- Armazenamento
- Áudio
- Rede
- Portas Série
- USB
- Pastas Partilhadas
- Interface do Utilizador

Geral

Basic Avançado Descrição **Encriptação de Disco**

Ativar Encriptação de Disco

Disk Encryption Cipher: *Leave Unchanged*

Inserir nova palavra-passe: _____

Confirmar nova palavra-passe: _____

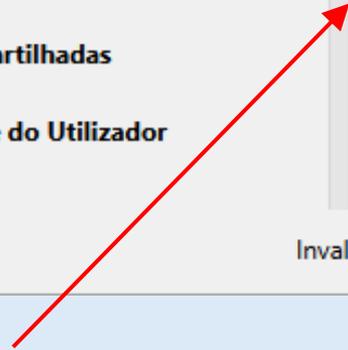
Sistema

Placa Mãe Processador Aceleração

Memória base: _____ 4096 MB

4 MB 16384 MB

Ordem de arranque: Disquete Ótico



Invalid settings detected 🚨

OK Cancelar Ajuda

Basic Expert

Search settings 🔍

Geral

Sistema

Display

Armazenamento

Áudio

Rede

Portas Série

USB

Pastas Partilhadas

Interface do Utilizador

Geral

Basic Avançado Descrição **Encriptação de Disco**

Ativar Encriptação de Disco

Disk Encryption Cipher: *Leave Unchanged*

Inserir nova palavra-passe:

Confirmar nova palavra-passe:

Sistema

Placa Mãe **Processador** Aceleração

Processors:  4

1 CPU 8 CPUs

Execution Cap:  100%

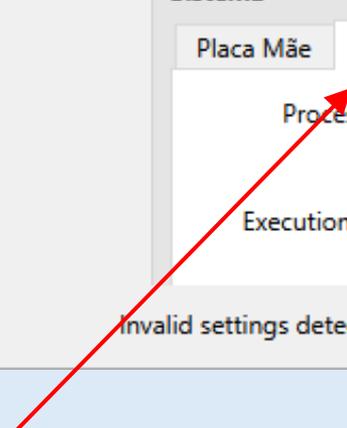
1% 100%

Invalid settings detected ⚠

OK

Cancelar

Ajuda



Basic Expert

Search settings

- Geral
- Sistema
- Display
- Armazenamento**
- Áudio
- Rede
- Portas Série
- USB
- Pastas Partilhadas
- Interface do Utilizador

Armazenamento

Dispositivos

- Controlador: IDE
 - Vazio
- Controlador: SATA
 - ubuntu18.04.vdi

Atributos

Disco rígido: Porta SATA 0

Unidade de Estado Sólido

Hot-pluggable

Informação

Tipo (Formato): Normal (VDI)

Virtual size: 50.00 GB

Actual size: 5.55 GB

Detalhes de armazenamento: Dynamically allocated s...

Localização: C:\Users\jpc\VirtualBox...

Ligado a: ubuntu18.04

Encryption key: --

Áudio

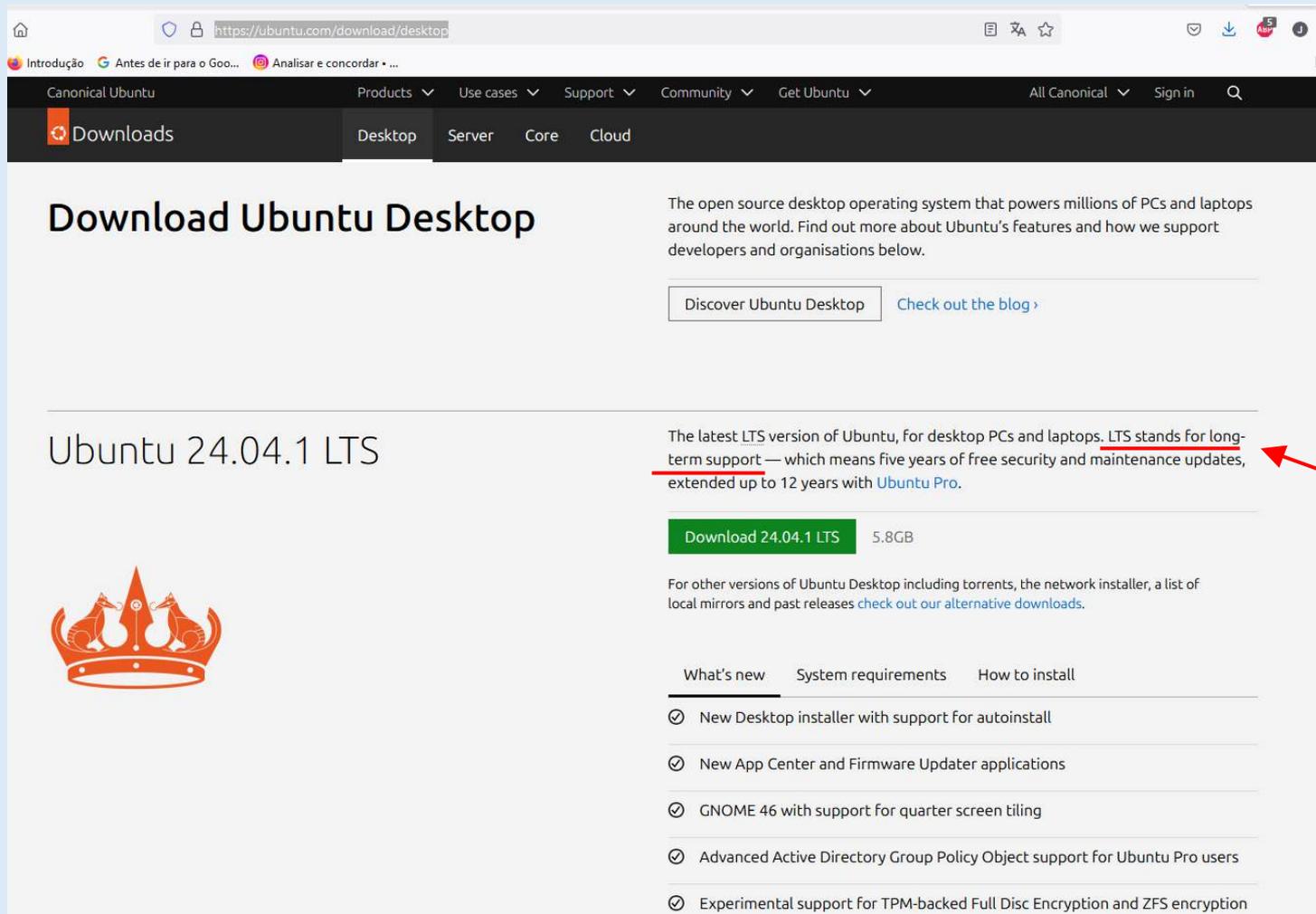
Ativar Áudio

Invalid settings detected

OK Cancelar Ajuda

Instalar ubuntu

- Descarregar Ubuntu
(<https://ubuntu.com/download/desktop>)



The screenshot shows the Ubuntu website's download page for the desktop version. The browser address bar shows the URL <https://ubuntu.com/download/desktop>. The navigation menu includes 'Downloads', 'Desktop', 'Server', 'Core', and 'Cloud'. The main heading is 'Download Ubuntu Desktop'. Below it, there is a description of Ubuntu as an open-source desktop operating system. A green button labeled 'Download 24.04.1 LTS' is visible, with '5.8GB' next to it. A red arrow points to the text 'LTS stands for long-term support' in the description. Below the download button, there are links for 'What's new', 'System requirements', and 'How to install'. A list of features is provided, including a new desktop installer, App Center, GNOME 46, and advanced security features.

Canonical Ubuntu Products Use cases Support Community Get Ubuntu All Canonical Sign in

Downloads Desktop Server Core Cloud

Download Ubuntu Desktop

The open source desktop operating system that powers millions of PCs and laptops around the world. Find out more about Ubuntu's features and how we support developers and organisations below.

Discover Ubuntu Desktop [Check out the blog](#)

Ubuntu 24.04.1 LTS

The latest LTS version of Ubuntu, for desktop PCs and laptops. LTS stands for long-term support — which means five years of free security and maintenance updates, extended up to 12 years with [Ubuntu Pro](#).

[Download 24.04.1 LTS](#) 5.8GB

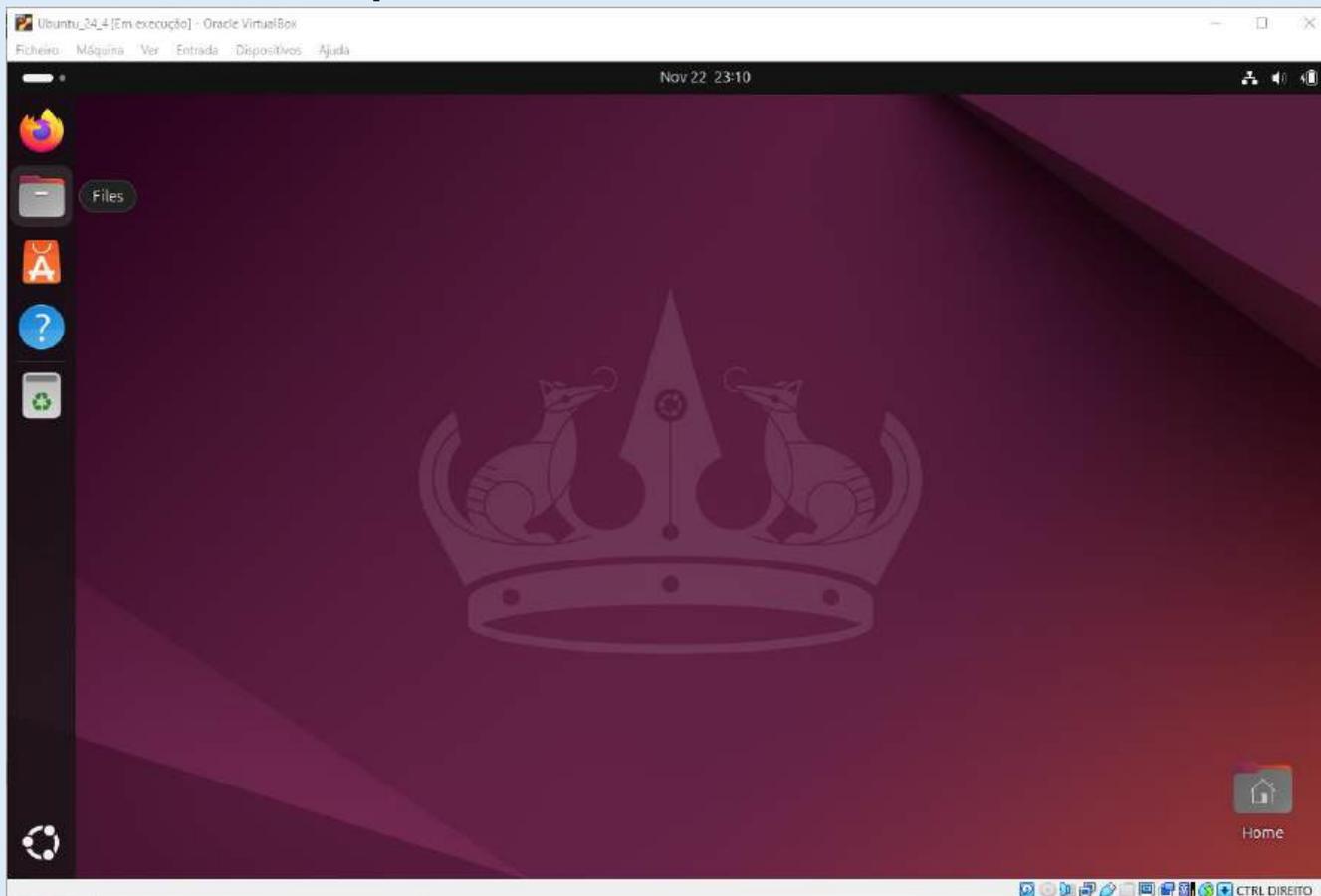
For other versions of Ubuntu Desktop including torrents, the network installer, a list of local mirrors and past releases [check out our alternative downloads](#).

[What's new](#) [System requirements](#) [How to install](#)

- ✓ New Desktop installer with support for autoinstall
- ✓ New App Center and Firmware Updater applications
- ✓ GNOME 46 with support for quarter screen tiling
- ✓ Advanced Active Directory Group Policy Object support for Ubuntu Pro users
- ✓ Experimental support for TPM-backed Full Disc Encryption and ZFS encryption

Instalar ubuntu

- Na virtualbox
- Ou numa máquina física.



OpenFOAM

- <https://www.openfoam.com/>
- <https://openfoam.org/>
- Ihfoam (<https://ihfoam.ihcantabria.com/>)
- Olaflow (<https://olaflow.github.io/>)

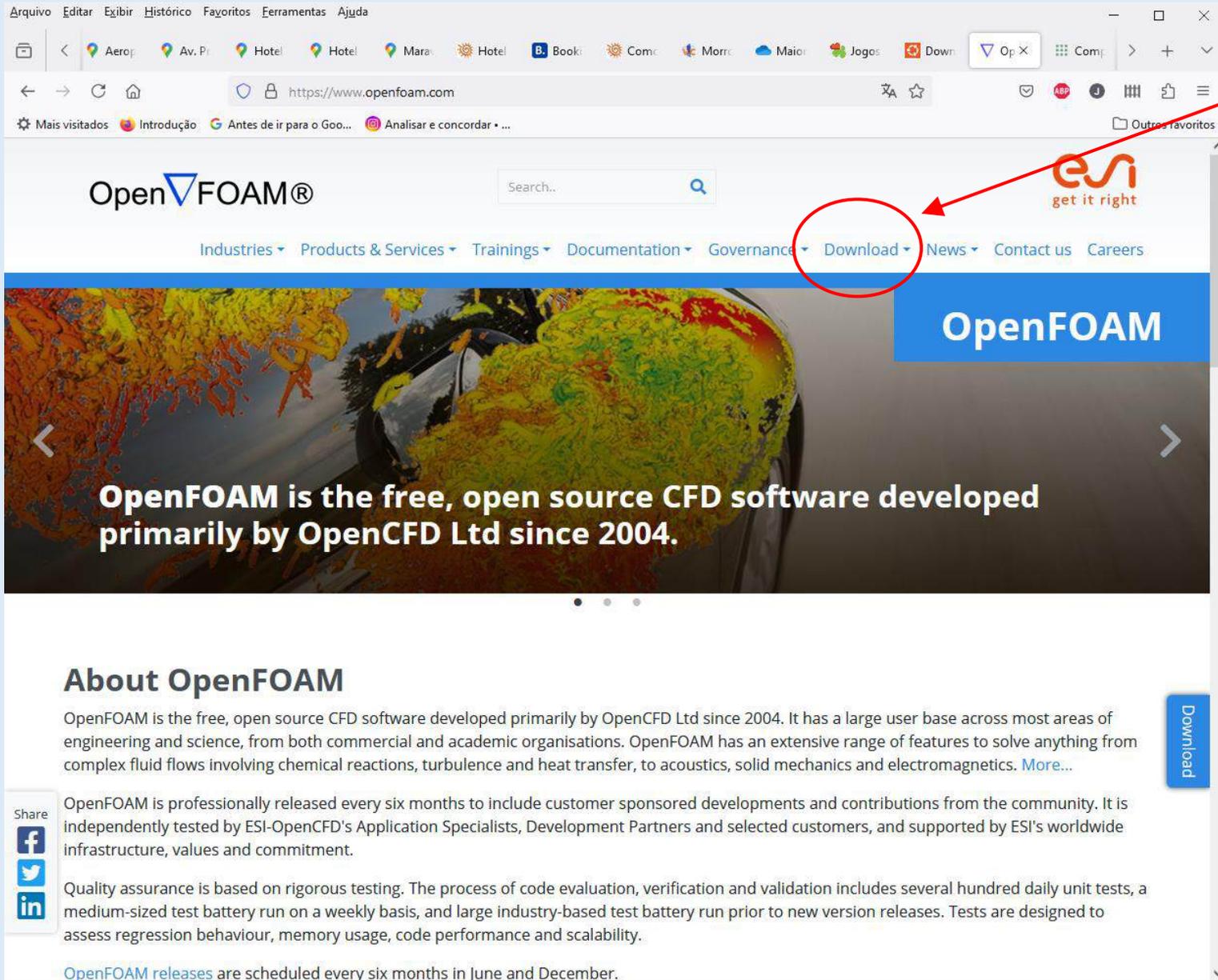
História:

- OpenFoam foi desenvolvido por
- **Hrvoje Jasak e Henry Weller.**



- <https://www.cfd-online.com/Forums/openfoam/152605-original-openfoam-author-s.html>
- <https://en.wikipedia.org/wiki/OpenFOAM>
- <https://www.ams.org/journals/notices/201404/rnoti-p354.pdf>

OpenFOAM.com



Arquivo Editar Exibir Histórico Favoritos Ferramentas Ajuda

Aero... Av. P... Hotel Hotel Mara... Hotel B. Book... Comc... Morre... Maior... Jogos... Down... Op X... Comp... > + v

https://www.openfoam.com

Mais visitados Introdução Antes de ir para o Goo... Analisar e concordar...

Outros favoritos

OpenFOAM® Search..

Industries Products & Services Trainings Documentation Governance **Download** News Contact us Careers

esi get it right

OpenFOAM

OpenFOAM is the free, open source CFD software developed primarily by OpenCFD Ltd since 2004.

About OpenFOAM

OpenFOAM is the free, open source CFD software developed primarily by OpenCFD Ltd since 2004. It has a large user base across most areas of engineering and science, from both commercial and academic organisations. OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to acoustics, solid mechanics and electromagnetics. [More...](#)

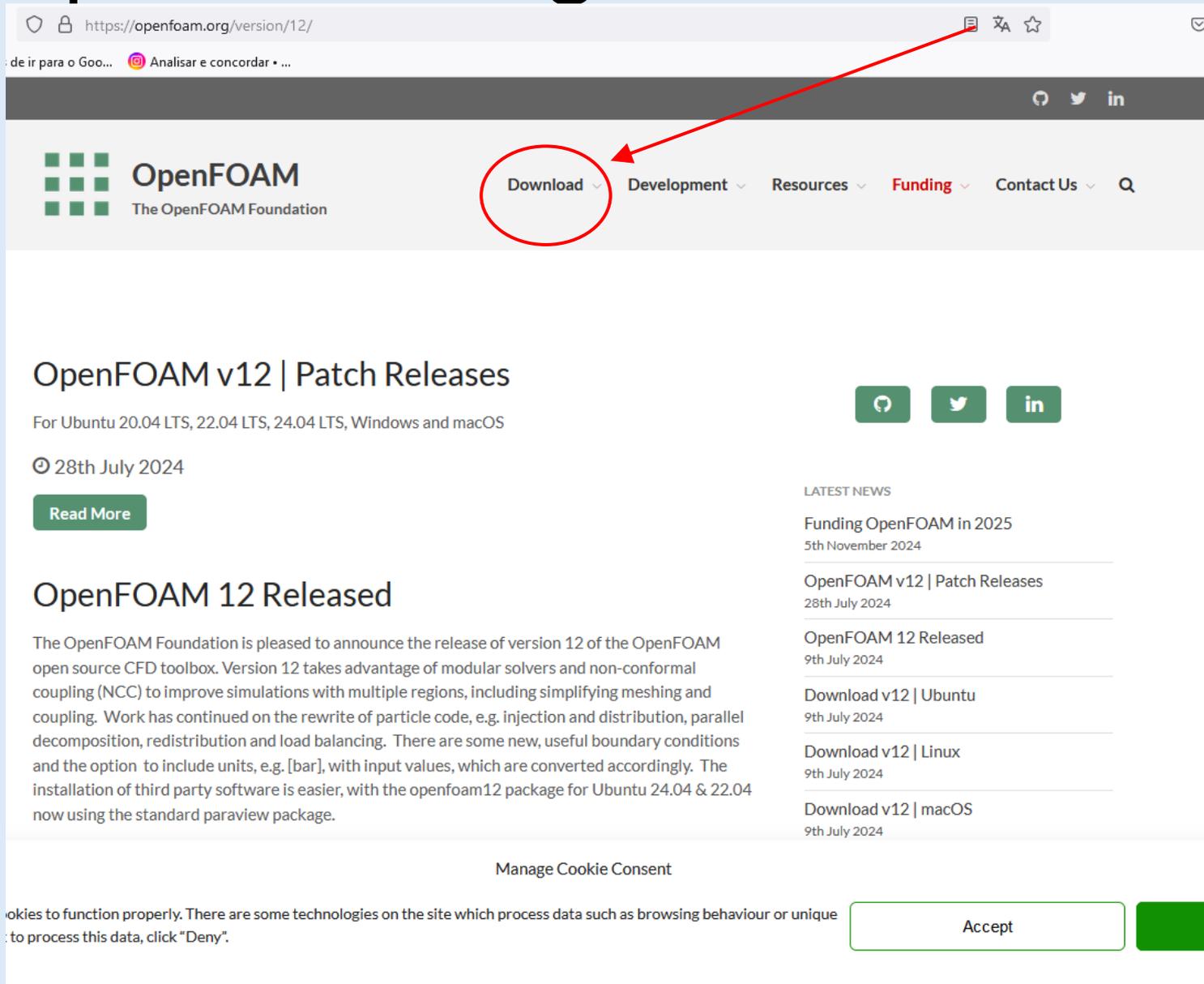
OpenFOAM is professionally released every six months to include customer sponsored developments and contributions from the community. It is independently tested by ESI-OpenCFD's Application Specialists, Development Partners and selected customers, and supported by ESI's worldwide infrastructure, values and commitment.

Quality assurance is based on rigorous testing. The process of code evaluation, verification and validation includes several hundred daily unit tests, a medium-sized test battery run on a weekly basis, and large industry-based test battery run prior to new version releases. Tests are designed to assess regression behaviour, memory usage, code performance and scalability.

[OpenFOAM releases](#) are scheduled every six months in June and December.

Download

OpenFOAM.org



de ir para o Goo...  Analisar e concordar • ...

 **OpenFOAM**
The OpenFOAM Foundation

[Download](#) ▾ [Development](#) ▾ [Resources](#) ▾ [Funding](#) ▾ [Contact Us](#) ▾ 

OpenFOAM v12 | Patch Releases

For Ubuntu 20.04 LTS, 22.04 LTS, 24.04 LTS, Windows and macOS

 28th July 2024

[Read More](#)

OpenFOAM 12 Released

The OpenFOAM Foundation is pleased to announce the release of version 12 of the OpenFOAM open source CFD toolbox. Version 12 takes advantage of modular solvers and non-conformal coupling (NCC) to improve simulations with multiple regions, including simplifying meshing and coupling. Work has continued on the rewrite of particle code, e.g. injection and distribution, parallel decomposition, redistribution and load balancing. There are some new, useful boundary conditions and the option to include units, e.g. [bar], with input values, which are converted accordingly. The installation of third party software is easier, with the openfoam12 package for Ubuntu 24.04 & 22.04 now using the standard paraview package.

LATEST NEWS

- [Funding OpenFOAM in 2025](#)
5th November 2024
- [OpenFOAM v12 | Patch Releases](#)
28th July 2024
- [OpenFOAM 12 Released](#)
9th July 2024
- [Download v12 | Ubuntu](#)
9th July 2024
- [Download v12 | Linux](#)
9th July 2024
- [Download v12 | macOS](#)
9th July 2024

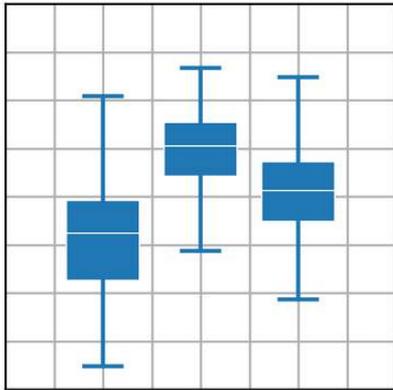
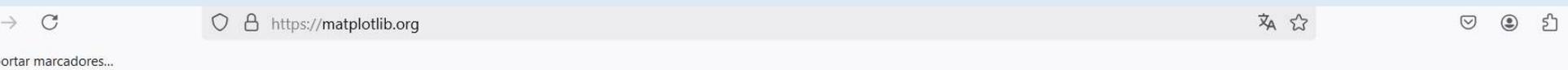
Manage Cookie Consent

okies to function properly. There are some technologies on the site which process data such as browsing behaviour or unique
to process this data, click "Deny".

[Accept](#) 

matplotlib

O iinfoam e o olafoam tambem necessitam do modulo pylab para fazer graficos. Matplotlib é uma livreria Python 2D.



boxplot(X)

Matplotlib: Visualization with Python

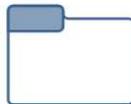
Matplotlib is a comprehensive library for creating static, animated, and interactive visualizations in Python. Matplotlib makes easy things easy and hard things possible.

- Create [publication quality plots](#).
- Make [interactive figures](#) that can zoom, pan, update.
- Customize [visual style](#) and [layout](#).
- Export to [many file formats](#).
- Embed in [JupyterLab](#) and [Graphical User Interfaces](#).
- Use a rich array of [third-party packages](#) built on Matplotlib.

[Try Matplotlib \(on Binder\)](#)



[Getting Started](#)



[Examples](#)



[Reference](#)



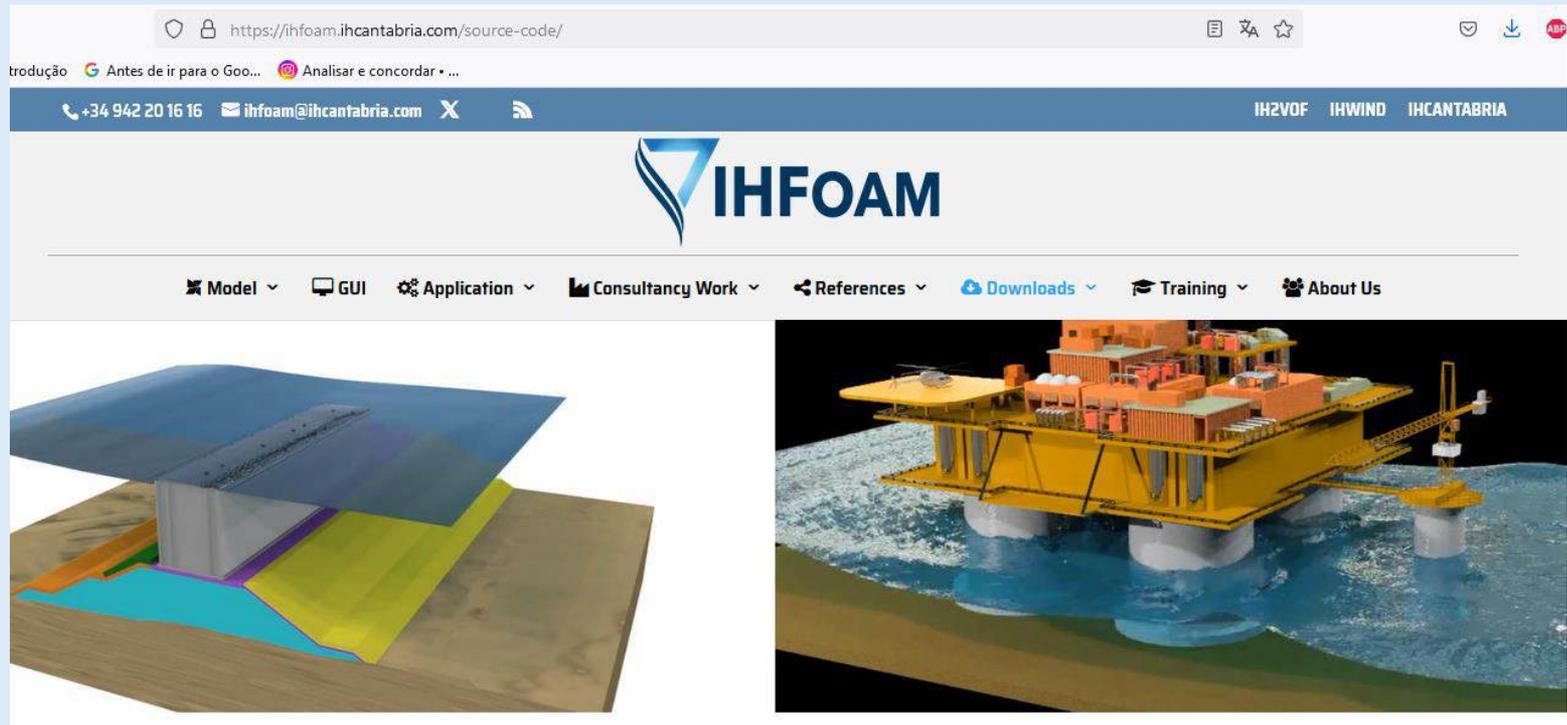
[Cheat Sheets](#)



[Documentation](#)

IHFOAM

(<https://ihfoam.ihcantabria.com/>)



The screenshot shows the IHFOAM website interface. At the top, there is a browser address bar with the URL <https://ihfoam.ihcantabria.com/source-code/>. Below the address bar is a navigation bar with contact information: phone number +34 942 20 16 16, email ihfoam@ihcantabria.com, and social media icons for X and RSS. The main header features the IHFOAM logo, which consists of a stylized blue 'I' and the text 'IHFOAM'. Below the logo is a navigation menu with the following items: Model, GUI, Application, Consultancy Work, References, Downloads, Training, and About Us. Two 3D simulation images are displayed below the menu. The left image shows a cross-section of a structure with a blue top layer, a yellow middle layer, and a cyan bottom layer, all on a brown base. The right image shows a yellow offshore platform with orange structures on top, situated on a blue sea with white waves, and supported by white cylindrical piles on a brown seabed.

As The Environmental Hydraulics Institute [IHCantabria](http://www.ihcantabria.com/) is an official contributions to [OpenFOAM](http://www.openfoam.com/), **IHFOAM** has become a part of the official release by OpenCFD (<http://www.openfoam.com/>). Therefore, once you download and install OpenFOAM in your computer, you will find the boundaries conditions and some tutorials.

You can upgrade OpenFOAM-v1612+, OpenFOAM-v1706, OpenFOAM-v1712, OpenFOAM-v1806, OpenFOAM-2006, OpenFOAM-2012, OpenFOAM-2106, OpenFOAM-2112, OpenFOAM-2206 and OpenFOAM-2212 by adding:

- The solver for two-phase flow within porous media by means of the VARANS equations (version 1.0) ([Download](#) )
- The solver for two-phase flow within porous media by means of the VARANS equations (version 2.0) (soon to be released)

IHFOAM was initially developed to simulate numerically all the processes related to coastal engineering. In a constant development, IHFOAM has now become a more friendly and flexible tool by adding all the known capabilities to a graphical user interface. Written in Python, this new capability decreases considerably the learning curve and provides engineers a powerful tool to deal with projects in a visual way.

IHFOAM

(<https://ihfoam.ihcantabria.com/>)



- Javier L. Lara contribuiu para o desenvolvimento do Cobras (Cornell Breaking Wave and Structures) (depois designado IH2VOF) e o IH3VOF, mas teve problemas de convergência por isso passou para o OpenFOAM.

OlaFlow (<https://olaflow.github.io/>)

- Pablo Higuera Caubilla desenvolveu o IHFoam e depois o OlaFLOW.
- Sendo o código muito parecido com a primeira versão de IHFOAM.



Extrutura do OpenFOAM

- O OpenFOAM é uma biblioteca C++, utilizada principalmente para criar executáveis, conhecidos como aplicações.
- As aplicações se enquadram em duas categorias:
 - solucionadores, cada um concebido para resolver um problema específico em mecânica contínua;
 - utilitários concebidos para executar tarefas que envolvem a manipulação de dados.

OpenFoam.com versão v2406

e

OpenFoam.org versão v12

Extrutura do OpenFOAM

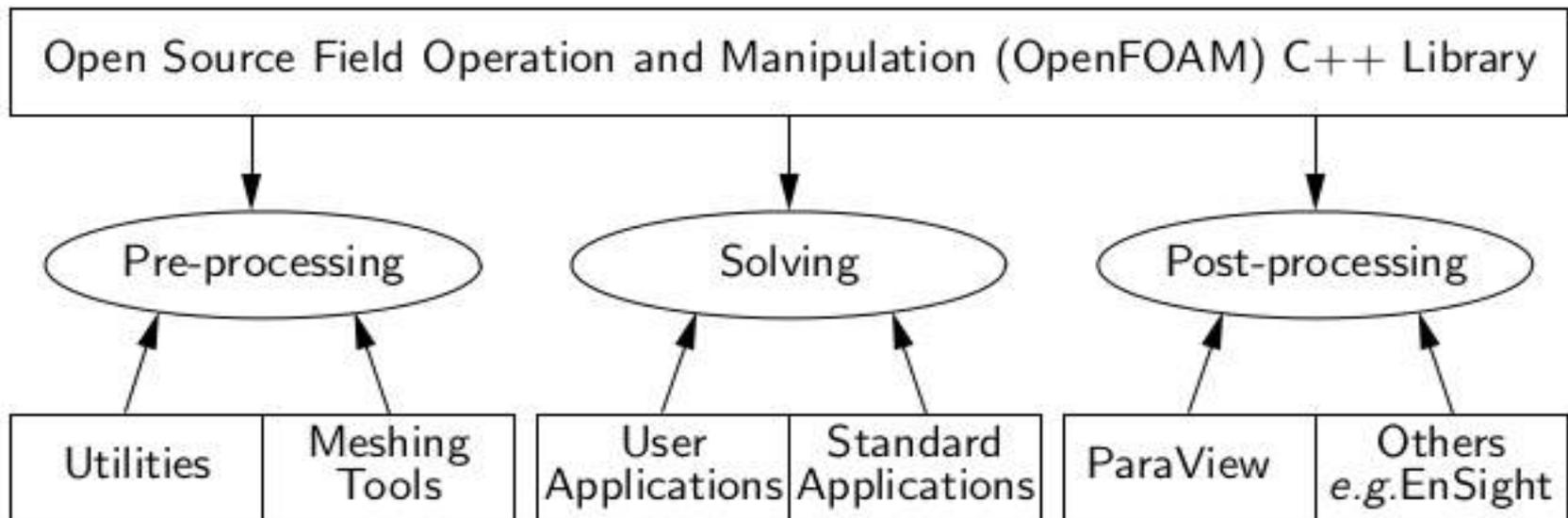
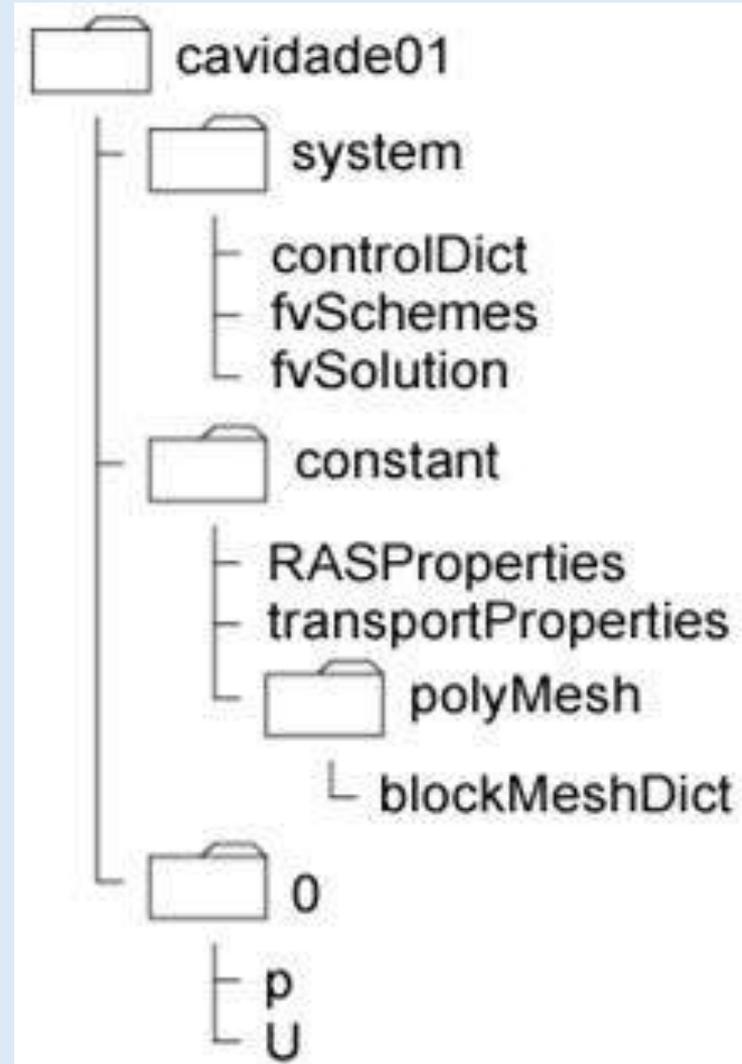
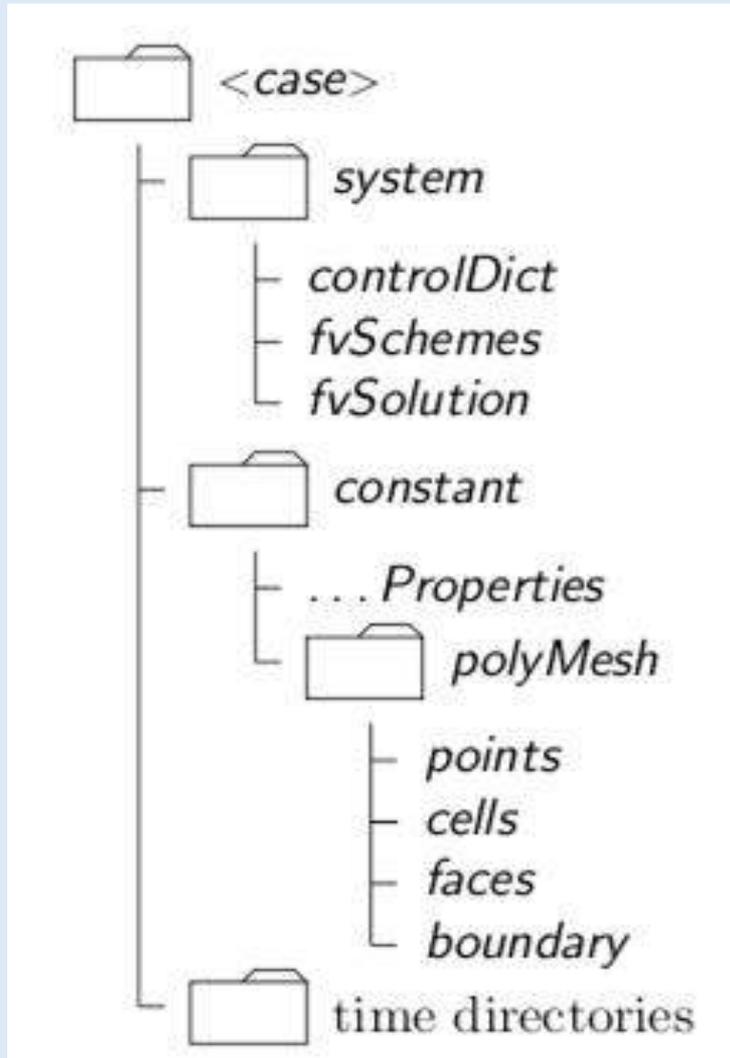


Figure 1.1: Overview of OpenFOAM structure.

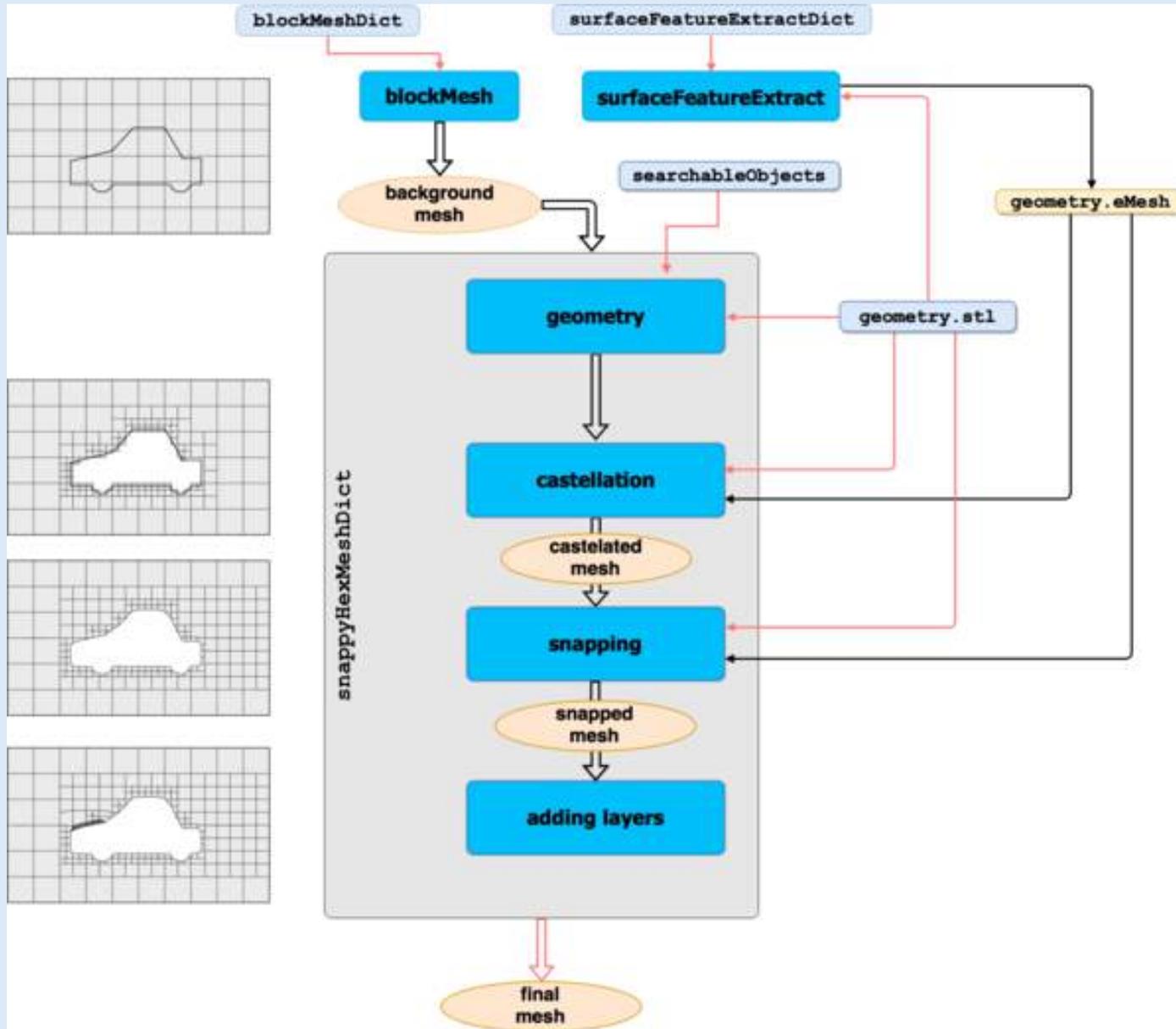
Extrutura do OpenFOAM

- A distribuição OpenFOAM contém numerosos solucionadores e utilitários que cobrem uma vasta gama de problemas.
- O OpenFOAM é fornecido com ambientes de pré e pós-processamento.
- A interface para o pré e pós-processamento são utilitários OpenFOAM, garantindo assim um tratamento consistente dos dados em todos os ambientes.

- A estrutura básica de directórios para um caso OpenFOAM



Utilitário: snappyHexMesh



- Outro método: conversão de malha

Malha feita
no gambit.

Mesh conversion	
ansysToFoam	Converts an ANSYS input mesh file, exported from I-DEAS, to OpenFOAM format
cfx4ToFoam	Converts a CFX 4 mesh to OpenFOAM format
datToFoam	Reads in a datToFoam (.dat) mesh file and outputs a points file. Used in conjunction with blockMesh
fluent3DMeshToFoam	Converts a Fluent mesh to OpenFOAM format
<u>fluentMeshToFoam</u>	Converts a Fluent mesh to OpenFOAM format including multiple region and region boundary handling
foamMeshToFluent	Writes out the OpenFOAM mesh in Fluent mesh format
foamToStarMesh	Reads an OpenFOAM mesh and writes a PROSTAR (v4) bnd/cel/vrt format
foamToSurface	Reads an OpenFOAM mesh and writes the boundaries in a surface format
gambitToFoam	Converts a GAMBIT mesh to OpenFOAM format
gmshToFoam	Reads .msh file as written by Gmsh
ideasUnvToFoam	I-Deas unv format mesh conversion
kivaToFoam	Converts a KIVA grid to OpenFOAM format
mshToFoam	Converts .msh file generated by the Adventure system
netgenNeutralToFoam	Converts neutral file format as written by Netgen v4.4
plot3dToFoam	Plot3d mesh (ascii/formatted format) converter
sammToFoam	Converts a STAR-CD (v3) SAMM mesh to OpenFOAM format
star3ToFoam	Converts a STAR-CD (v3) PROSTAR mesh into OpenFOAM format
star4ToFoam	Converts a STAR-CD (v4) PROSTAR mesh into OpenFOAM format
tetgenToFoam	Converts .ele and .node and .face files, written by tetgen
writeMeshObj	For mesh debugging: writes mesh as three separate OBJ files which can be viewed with e.g. javaview

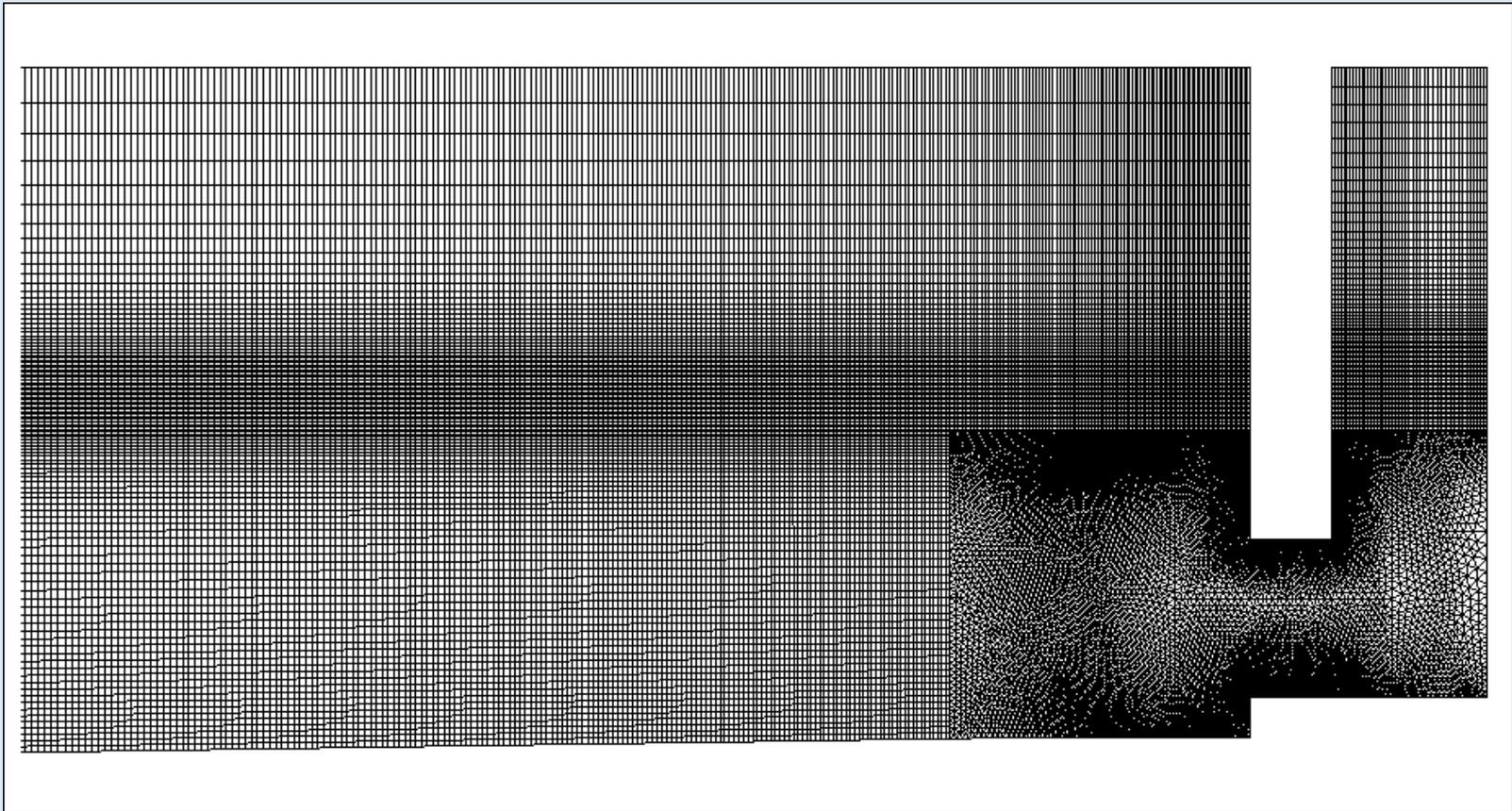
exemplo

Escrever o comando no terminal:

```
>fluentMeshToFoam canal10.msh
```

Fazer a malha noutro programa. Ex: gambit .

```
>rotateMesh -constant -noZero "(0 1 0)" "(0 0 1)"
```



IHFOAM (ou OLAFLOW)

- Solver IHFoam é baseado no solver InterFoam que permite resolver problemas bifásicos, permite um grande número de teorias de geração de ondas e absorção ativa de ondas.
- Identifica a superfície livre pelo método VoF. Este método utiliza uma variável, a fração de volume, que varia entre 1 e 0 consoante a quantidade existente de água na célula, 1 se for somente água, 0 se for o ar e entre 1 e 0 se for a interface entre ar e água.

IHFOAM

- Para obter uma interface bem definida, utiliza o método limitador multidimensional universal para solução explícita (Multidimensional Universal Limiter for Explicit Solution, MULES).
- Na resolução do sistema de equações RANS é utilizado o algoritmo PIMPLE, que é uma combinação dos algoritmos SIMPLE e PISO.

➤ Sistema de equações

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot [\rho \mathbf{u} \mathbf{u}^T] = \quad (1)$$

$$-\nabla p^* - \mathbf{g} \cdot \mathbf{x} \nabla \rho + \nabla \cdot [\mu \nabla \mathbf{u} + \rho \boldsymbol{\tau}] + \sigma_T \kappa_\alpha \nabla \alpha$$

$$\nabla \cdot \mathbf{u} = 0 \quad (2)$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot [\mathbf{u} \alpha] + \nabla \cdot [\mathbf{u}_r \alpha (1 - \alpha)] = 0 \quad (3)$$

$$\boldsymbol{\tau} = \frac{2}{\rho} \mu_t \left[\frac{1}{2} (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \right] - \frac{2}{3} k \mathbf{I} \quad (4)$$

➤ Modelo de turbulência k - ε standard ou outros.

Estrutura do caso

Os casos OpenFOAM são configurados utilizando vários ficheiros de entrada de texto simples localizados nos três diretórios:

- *system*

controlDict

fvSchemes

fvSolution

<system dictionaries>

- *constant*

polyMesh

<constant dictionaries>

- *<initial time directory>*

<field files>

<result time directories>: field predictions as a function of iteration count or time

postProcessing: data typically generated by **function objects**

data conversion, e.g. VTK

Solvers

Modules

[Basic solvers](#)

[Combustion solvers](#)

[Compressible flow solvers](#)

[Direct Numerical Simulation solvers](#)

[Discrete method solvers](#)

[Electro-magnetics solvers](#)

[Financial solvers](#)

[Heat transfer solvers](#)

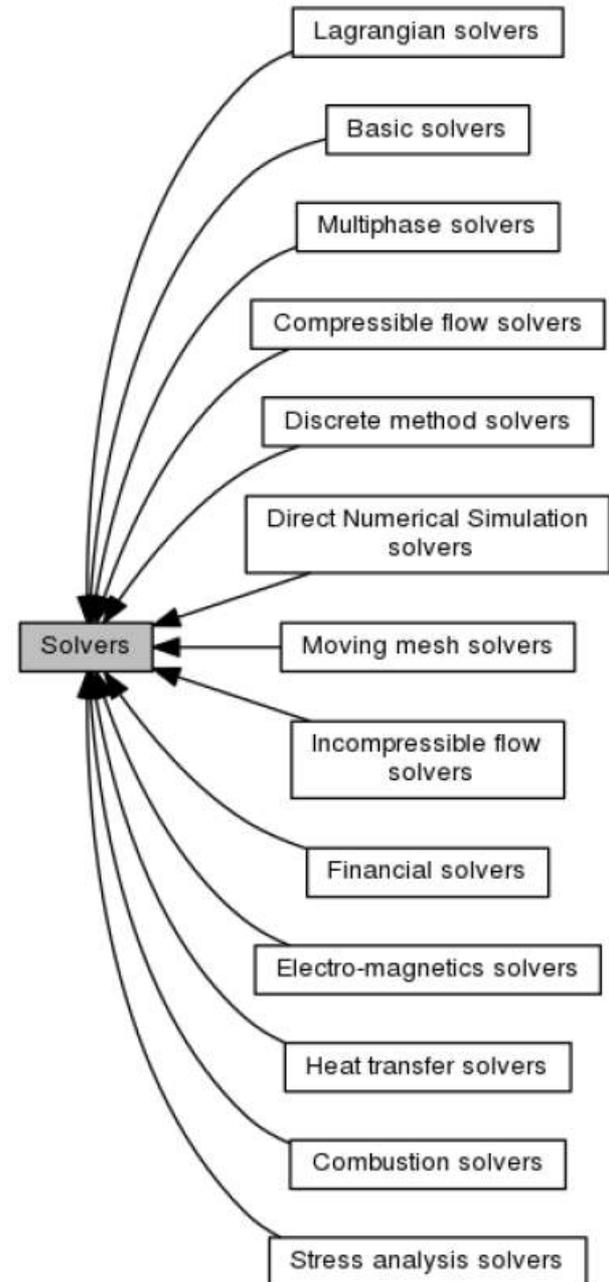
[Incompressible flow solvers](#)

[Lagrangian solvers](#)

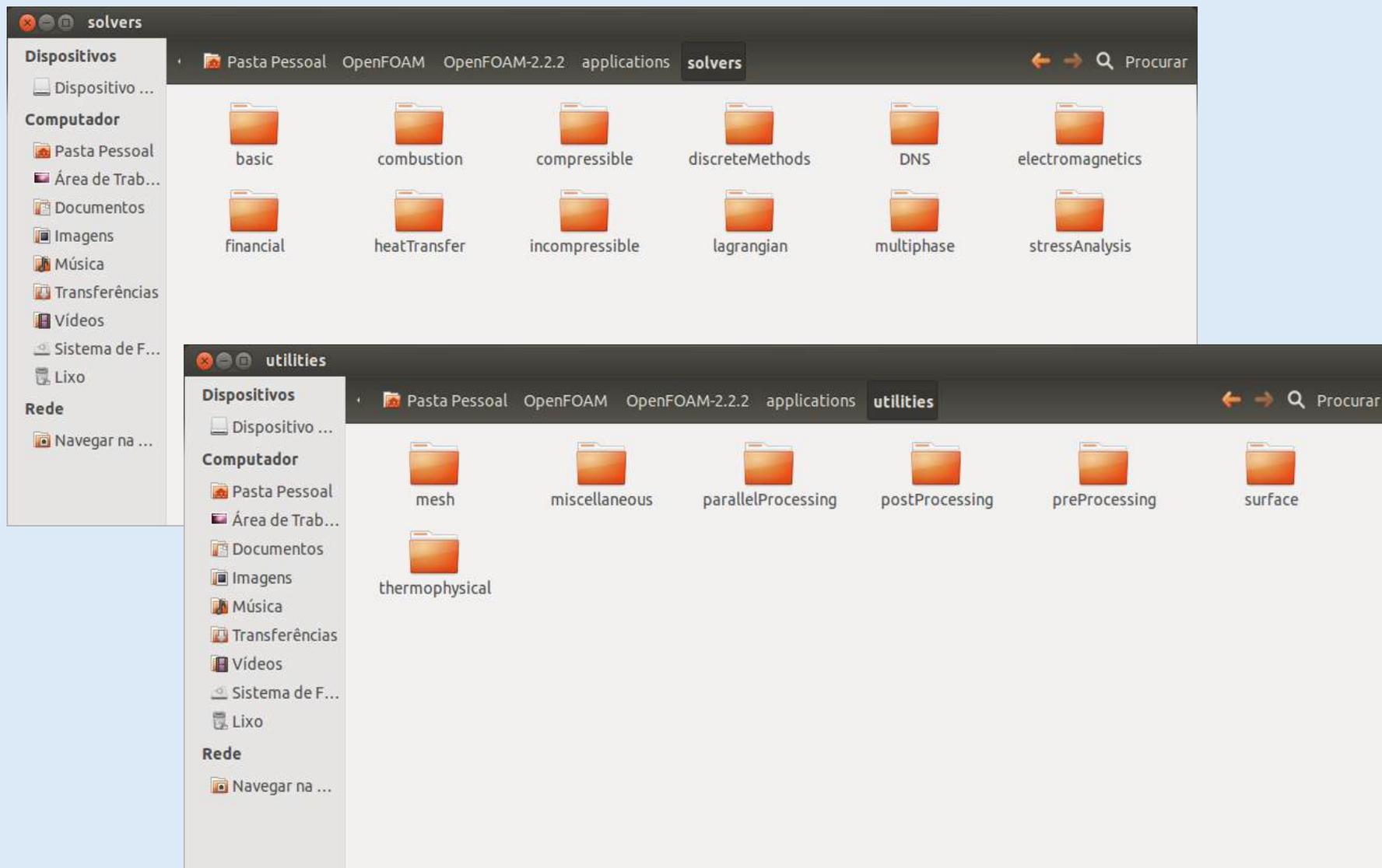
[Moving mesh solvers](#)

[Multiphase solvers](#)

[Stress analysis solvers](#)



Solvers e utilities



Exemplo:

Utilizados recentemente

Início

Ambiente de trabalho

Documentos

Imagens

Música

Transferências

Vídeos

Lixo

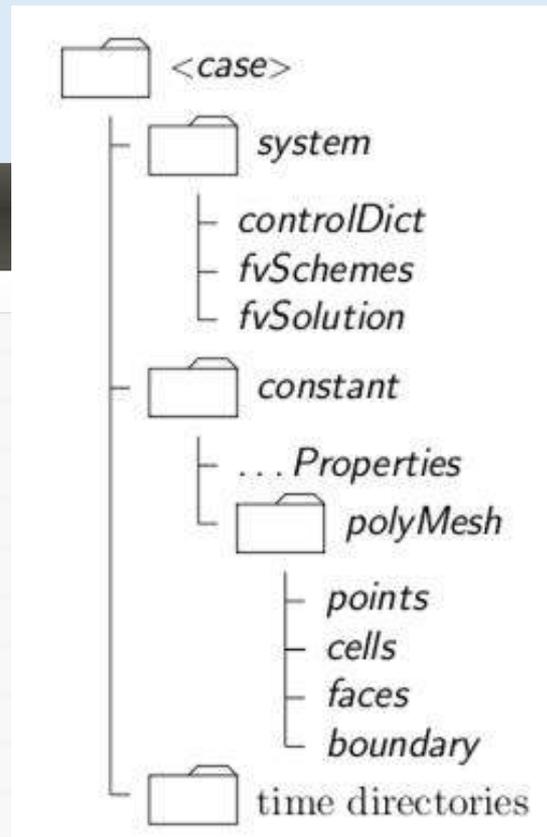
Rede

Computador

Volume de 1,0 TB

Ligar ao servidor

Nome
0.15
0.1
0.05
system
gaugesVOF
0
constant
postProcessed.tar.gz
log.reconstructPar
waterfall-comb-IH.avi
video_wf_comb_gnu_v1
log.olaFlow
log.decomposePar
log.setFields
runCase
mutriku2d2_v03.msh
runCase_mariana
multigauges_gnuplot_mariana
cleanCase_mariana
isoSurface_v7.py
postSensVOF_v3.py



Nome	Tamanho	Tipo	Modificado
0.15	10 itens	Pasta	Out 12
0.1	10 itens	Pasta	Out 12
0.05	10 itens	Pasta	Out 12
system	8 itens	Pasta	Out 11
gaugesVOF	22 itens	Pasta	Out 11
0	7 itens	Pasta	Out 4
constant	6 itens	Pasta	Set 4
postProcessed.tar.gz	28,2 MB	Arquivo	Out 12
log.reconstructPar	715,8 kB	Texto	Out 12
waterfall-comb-IH.avi	8,7 MB	Vídeo	Out 12
video_wf_comb_gnu_v1	4,7 kB	Texto	Out 12
log.olaFlow	33,3 MB	Texto	Out 12
log.decomposePar	3,8 kB	Texto	Out 11
log.setFields	1,4 kB	Texto	Out 11
runCase	957 bytes	Aplicação	Out 11
mutriku2d2_v03.msh	10,8 MB	Texto	Out 2
runCase_mariana	245 bytes	Aplicação	Ago 26
multigauges_gnuplot_mariana	906 bytes	Texto	Ago 26
cleanCase_mariana	298 bytes	Aplicação	Ago 26
isoSurface_v7.py	3,5 kB	Texto	Ago 21
postSensVOF_v3.py	2,8 kB	Texto	Ago 6

Pasta constant/polyMesh



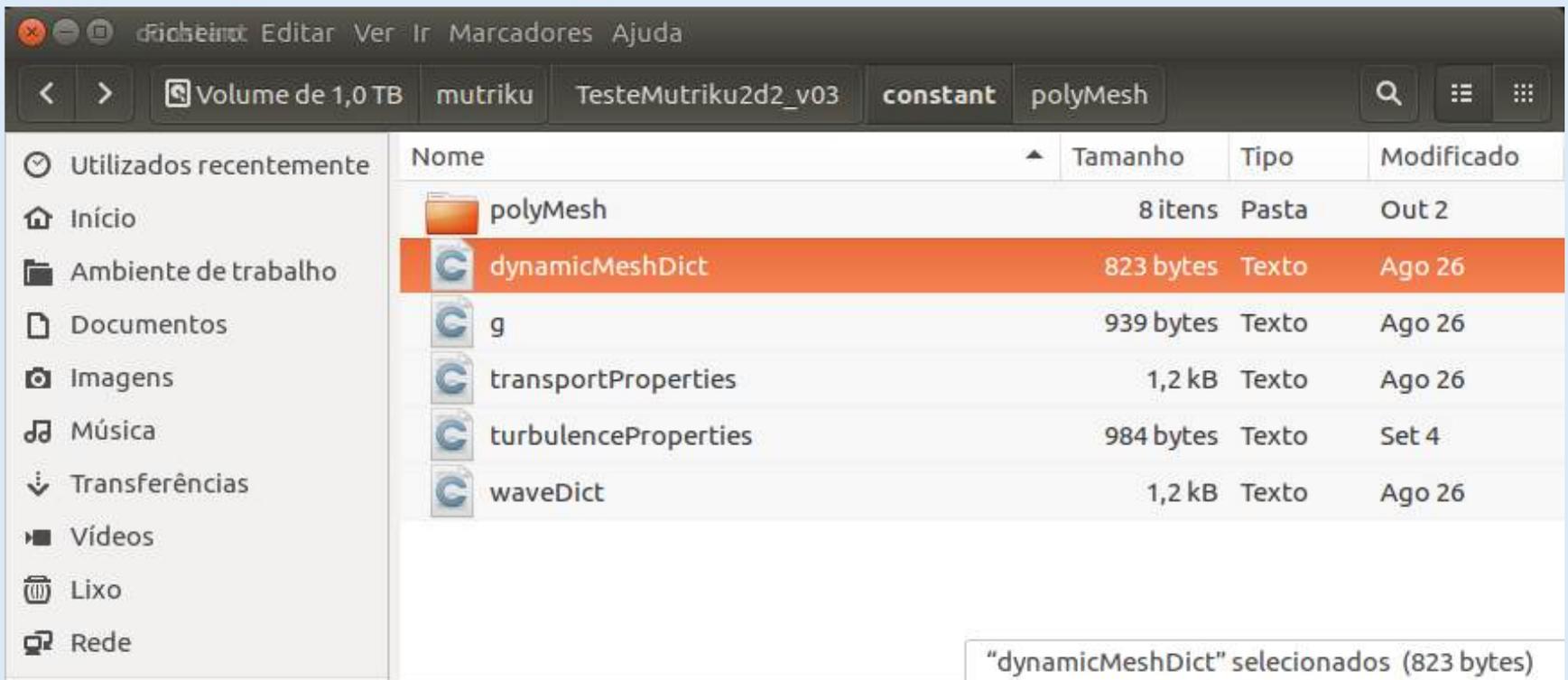
The screenshot shows the polyMeshEditor application window. The breadcrumb path is "Volume de 1,0 TB > mutriku > TesteMutriku2d2_v03 > constant > polyMesh". The file explorer displays a list of files in the "constant/polyMesh" directory. The files are listed in a table with columns for Name, Size, Type, and Modified. The "boundary" file is highlighted.

Nome	Tamanho	Tipo	Modificado
boundary	1,9 kB	Texto	Out 2
cellZones	877 bytes	Texto	Out 2
faces	13,4 MB	Texto	Out 2
faceZones	877 bytes	Texto	Out 2
neighbour	1,4 MB	Texto	Out 2
owner	2,9 MB	Texto	Out 2
points	7,9 MB	Texto	Out 2
pointZones	878 bytes	Texto	Out 2

Aqui é importante ver o file boundary que tem o nome e tipos de fronteiras.

Pasta constant

Os arquivos colocados na pasta constante estão relacionados às propriedades físicas da simulação.



The screenshot shows a file explorer window with the following details:

- Window title: Ficheiro Editar Ver Ir Marcadores Ajuda
- Address bar: Volume de 1,0 TB mutriku TesteMutriku2d2_v03 constant polyMesh
- Left sidebar: Utilizados recentemente, Início, Ambiente de trabalho, Documentos, Imagens, Música, Transferências, Vídeos, Lixo, Rede
- Main view: A table listing files in the 'constant' folder.

Nome	Tamanho	Tipo	Modificado
polyMesh	8 itens	Pasta	Out 2
dynamicMeshDict	823 bytes	Texto	Ago 26
g	939 bytes	Texto	Ago 26
transportProperties	1,2 kB	Texto	Ago 26
turbulenceProperties	984 bytes	Texto	Set 4
waveDict	1,2 kB	Texto	Ago 26

“dynamicMeshDict” selecionados (823 bytes)

- Malha estática

```
/*-----*- C++ -*-----*/
|=====|
|  \ \  /  | F i e l d      | OpenFOAM: The Open Source CFD Toolbox
|  \ \  /  | O p e r a t i o n | Version: dev
|   \ \  /  | A n d           | Web:      http://www.OpenFOAM.org
|    \ \  /  | M a n i p u l a t i o n |
|-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       dynamicMeshDict;
}
// *****

dynamicFvMesh      staticFvMesh;
```

Mass	kilogram (kg)
Length	metre (m)
Time	second (s)
Temperature	Kelvin (K)
Quantity	kilogram-mole (kgmol)
Current	ampere (A)
Luminous intensity	candela (cd)

```

/*-----*- C++ -*-----
|=====
|  \ \ / /   F i e l d   |   O p e n F O A M :   T h e
|  \ \ / /   O p e r a t i o n   |   V e r s i o n :   1 . 6
|  \ \ / /   A n d   |   W e b :   w w w .
|  \ \ / /   M a n i p u l a t i o n   |
|-----
/*-----

```

```

FoamFile
{
    version      2.0;
    format       ascii;
    class        uniformDimensionedVectorField;
    location     "constant";
    object       g;
}
// *****

dimensions      [0 1 -2 0 0 0 0];
value           ( 0 0 -9.81 );

// *****

```

```
/*-----*- C++ -*-----*/
|=====|
|  \ \ /  | F i e l d      | OpenFOAM: The Open Source CFD Toolbox
|  \ \ /  | O p e r a t i o n | Version: 1.6
|   \ \ /  | A n d          | Web:      www.OpenFOAM.org
|    \ \ /  | M a n i p u l a t i o n |
\*-----*/
```

```
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       turbulenceProperties;
}
// ***** //
```

```
simulationType  RAS;

RAS
{
    RASModel      kEpsilon;           Modelo de turbulência utilizado
    turbulence    on;
    printCoeffs   on;
}

// ***** //
```

```

/*-----*- C++ -*-----*/
|=====|
|  \ \  /  | F i e l d           | OpenFOAM: The Open Source CFD Toolbox
|  \ \  /  | O p e r a t i o n   | Version: 3.0.0
|  \ \  /  | A n d                | Web:      www.OpenFOAM.org
|  \ \  /  | M a n i p u l a t i o n |
/*-----*- C++ -*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       transportProperties;
}
// *****

phases (water air);

water
{
    transportModel Newtonian;
    nu              [0 2 -1 0 0 0 0] 1e-06;
    rho            [1 -3 0 0 0 0 0] 1000;
}

air
{
    transportModel Newtonian;
    nu              [0 2 -1 0 0 0 0] 1.48e-05;
    rho            [1 -3 0 0 0 0 0] 1;
}

sigma            [1 0 -2 0 0 0 0] 0.07;

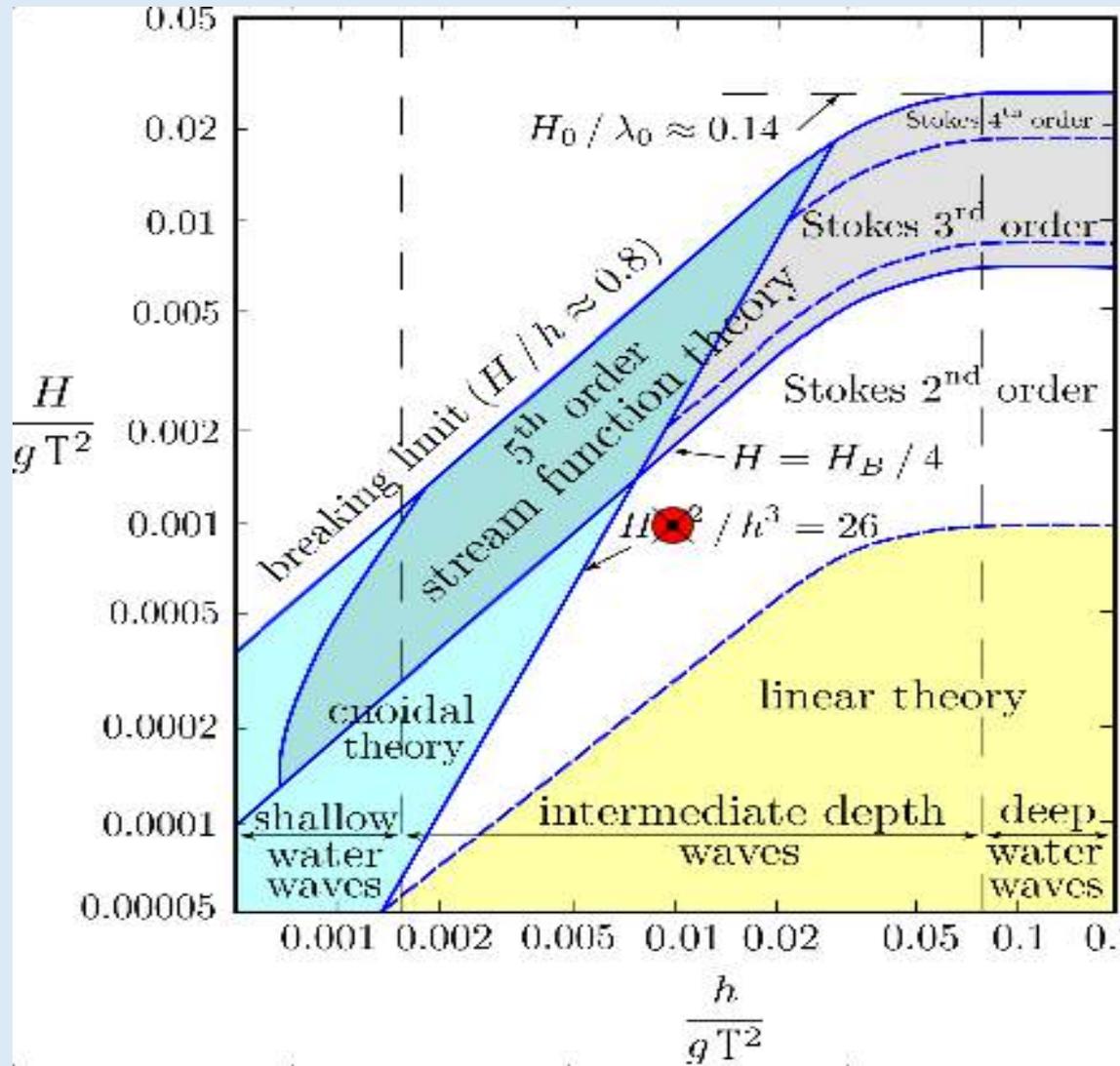
// *****

```

Aqui **considere** **tensao superficial**, talvez possa fazer uma simulação com este valor a zero.



Condições de simulação:



```
/*-----*\
|=====|
|  \ \ /  | F i e l d      | OpenFOAM: The Open Source CFD Toolbox
|  \ \ /  | O p e r a t i o n | Version: 1.3
|  \ \ /  | A n d           | Web:      http://www.openfoam.org
|  \ \ /  | M a n i p u l a t i o n |
|-----*\
/*-----*\
```

```
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       waveDict;
}
// ***** //
```

```
waveType      regular;
waveTheory     StokesII;
genAbs         0;
absDir         0.0;
nPaddles      1;
waveHeight     1;
wavePeriod    10;
waveDir        0.0;
wavePhase     5.76;

// Change both entries to true to re-read this dictionary upon restart.
rereadAlpha   false;
rereadU       false;

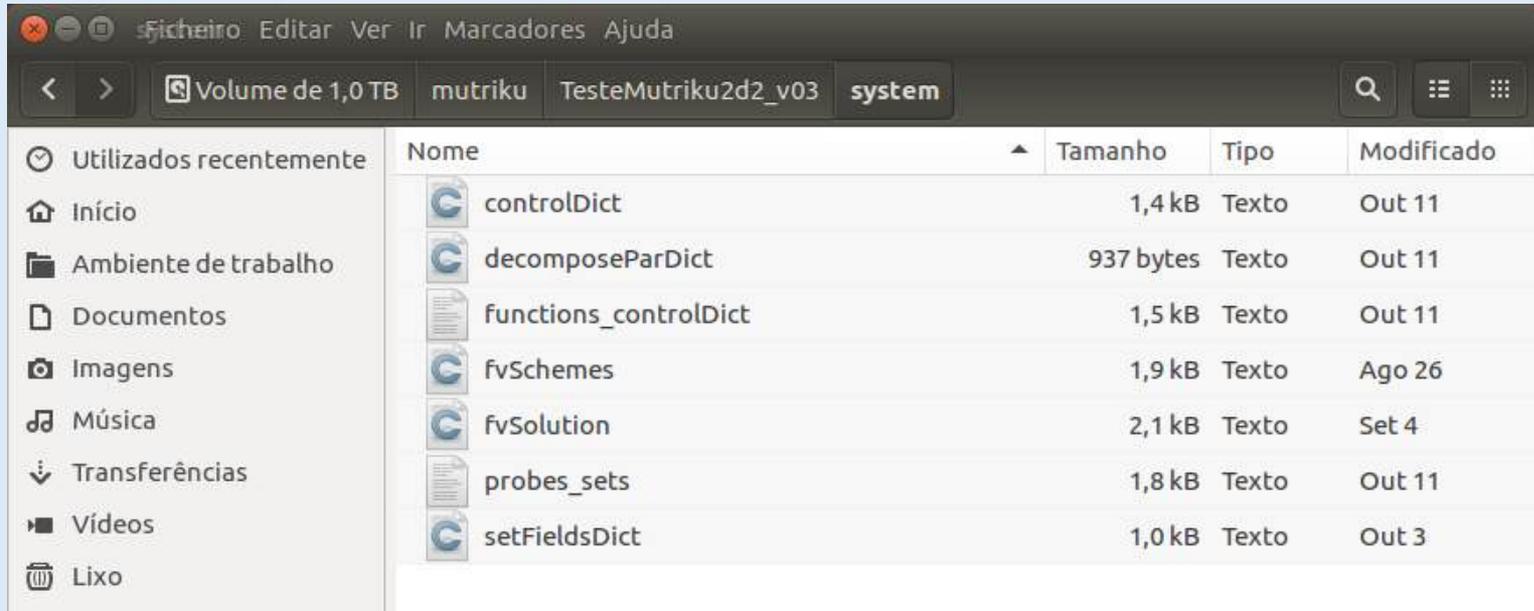
// ***** //
```

Teoria das ondas

Absorção desligada



Pasta system



Nome	Tamanho	Tipo	Modificado
controlDict	1,4 kB	Texto	Out 11
decomposeParDict	937 bytes	Texto	Out 11
functions_controlDict	1,5 kB	Texto	Out 11
fvSchemes	1,9 kB	Texto	Ago 26
fvSolution	2,1 kB	Texto	Set 4
probes_sets	1,8 kB	Texto	Out 11
setFieldsDict	1,0 kB	Texto	Out 3

decomposeParDict:

Só alterei para 8 processadores com scotch.

```
/*-----*\
|=====|
|  \ \ /  F i e l d      | OpenFOAM: The Open Source CFD Toolbox
|  \ \ /  O p e r a t i o n | Version: 1.3
|   \ \ /  A n d         | Web:      http://www.openfoam.org
|    \ \ /  M a n i p u l a t i o n |
\*-----*/

FoamFile
{
    version            2.0;
    format             ascii;
    location           "system";
    class              dictionary;
    object             decomposeParDict;
}
// *****

numberOfSubdomains 8;

method              scotch;

// *****
```

Set fields

Para definir onde é água.

```
    location    "system";
    object      setFieldsDict;
}
// ***** //

defaultFieldValues
(
    volScalarFieldValue alpha.water 0
);

regions
(
    boxToCell
    {
        box (0 -5 -15) (415.0 5 0);

        fieldValues
        (
            volScalarFieldValue alpha.water 1
        );
    }
);
```



```
writeFormat      ascii;

writePrecision   6;

writeCompression off;

timeFormat       general;

timePrecision    6;

runTimeModifiable yes;

adjustTimeStep  yes;

maxCo            0.25;
maxAlphaCo      0.5;

maxDeltaT       0.025;

functions

{
  #includeIfPresent "functions_controlDict";
}

// ***** //
```

functions_controlDict:

```
sets
{
    type                sets;
    functionObjectLibs ("libsampling.so");
    enabled              true;

    writeControl        timeStep;    // Alternative: outputTime
    writeInterval       1;

    interpolationScheme  cellPoint;
    setFormat           raw;

    fixedLocations      false;

    fields
    (
        U
        alpha.water
    );
    #includeIfPresent "probes_sets";
}

freeSurface
{
    type                surfaces;
    functionObjectLibs ("libsampling.so");
    //outputControl     outputTime;
    writeControl        timeStep;
    writeInterval       1;

    surfaceFormat       raw;
}
```

```

interpolationScheme cellPoint;

fields
(
    alpha.water
);
surfaces
(
    // interpolatedIso
    topFreeSurface
    {
        // Iso surface for interpolated values only
        type          isoSurface;    // always triangulated
        isoField      alpha.water;
        isoValue      0.5;
        interpolate   true;

        //zone          ABC;          // Optional: zone only
        //exposedPatchName fixedWalls; // Optional: zone only

        // regularise   false;        // Optional: do not simplify
        // mergeTol     1e-10;        // Optional: fraction of mesh
        bounding box
        // to merge points (default=1e-6)
    }
);
}

```

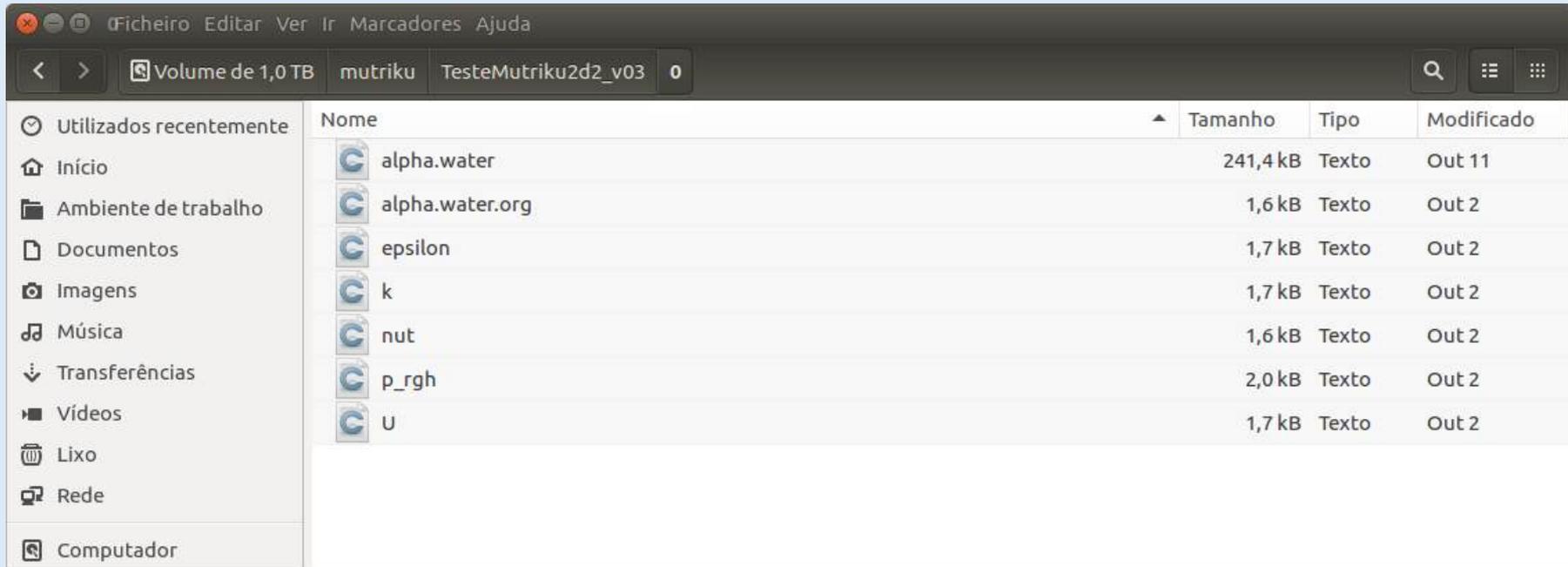
probes_sets:

Vou colocar sondas de nível em:

X= 20, 50, 100, 150, 200, 250, 300, 302, 305.7, 307.2, 308.7

```
sets
(
  gauge00
  {
    type      midPointAndFace;
    axis      xyz;
    start     (20 0.00 -10.);
    end       (20 0.00  6.6);
  }
  gauge01
  {
    type      midPointAndFace;
    axis      xyz;
    start     (50 0.00 -10.);
    end       (50 0.00  6.6);
  }
  gauge02
  {
    type      midPointAndFace;
    axis      xyz;
    start     (100 0.00 -10.);
    end       (100 0.00  6.6);
  }
  gauge03
  {
    type      midPointAndFace;
    axis      xyz;
    start     (150 0.00 -10.);
```

Condições de fronteira e iniciais (pasta 0)



The image shows a file explorer window with the following details:

- Window title: Ficheiro Editar Ver Ir Marcadores Ajuda
- Address bar: Volume de 1,0 TB mutriku TesteMutriku2d2_v03 0
- Left sidebar (Navigation pane): Utilizados recentemente, Início, Ambiente de trabalho, Documentos, Imagens, Música, Transferências, Vídeos, Lixo, Rede, Computador
- Main pane (Table view):

Nome	Tamanho	Tipo	Modificado
alpha.water	241,4 kB	Texto	Out 11
alpha.water.org	1,6 kB	Texto	Out 2
epsilon	1,7 kB	Texto	Out 2
k	1,7 kB	Texto	Out 2
nut	1,6 kB	Texto	Out 2
p_rgh	2,0 kB	Texto	Out 2
U	1,7 kB	Texto	Out 2

Correr o caso

Para correr abrir terminal

```
jpc@deep03:~$ cd '/media/jpc/94228DEB228DD322/mutriku/TesteMutriku2d2_v03'
```

```
jpc@deep03:/media/jpc/94228DEB228DD322/mutriku/TesteMutriku2d2_v03$ of1712  
./runCase
```

Posso fazer o comando e correr o script todo, mas prefiro fazer passo a passo. Estando os passos neste script.

```
./runCase
```

file runCase

```
#!/bin/bash

# Source run functions
echo Source run functions...
. $WM_PROJECT_DIR/bin/tools/RunFunctions

# Preparing 0 folder
#echo
#echo Preparing 0 folder...
#cp alpha.water.org alpha.water

# Create the computational mesh
#echo
#echo Create the computational mesh
#fluentMeshToFoam mutriku2d2_v03.msh
#rotateMesh -constant -noZero "(0 1 0)" "(0 0 1)"

# Check the computational mesh
echo
runApplication checkMesh

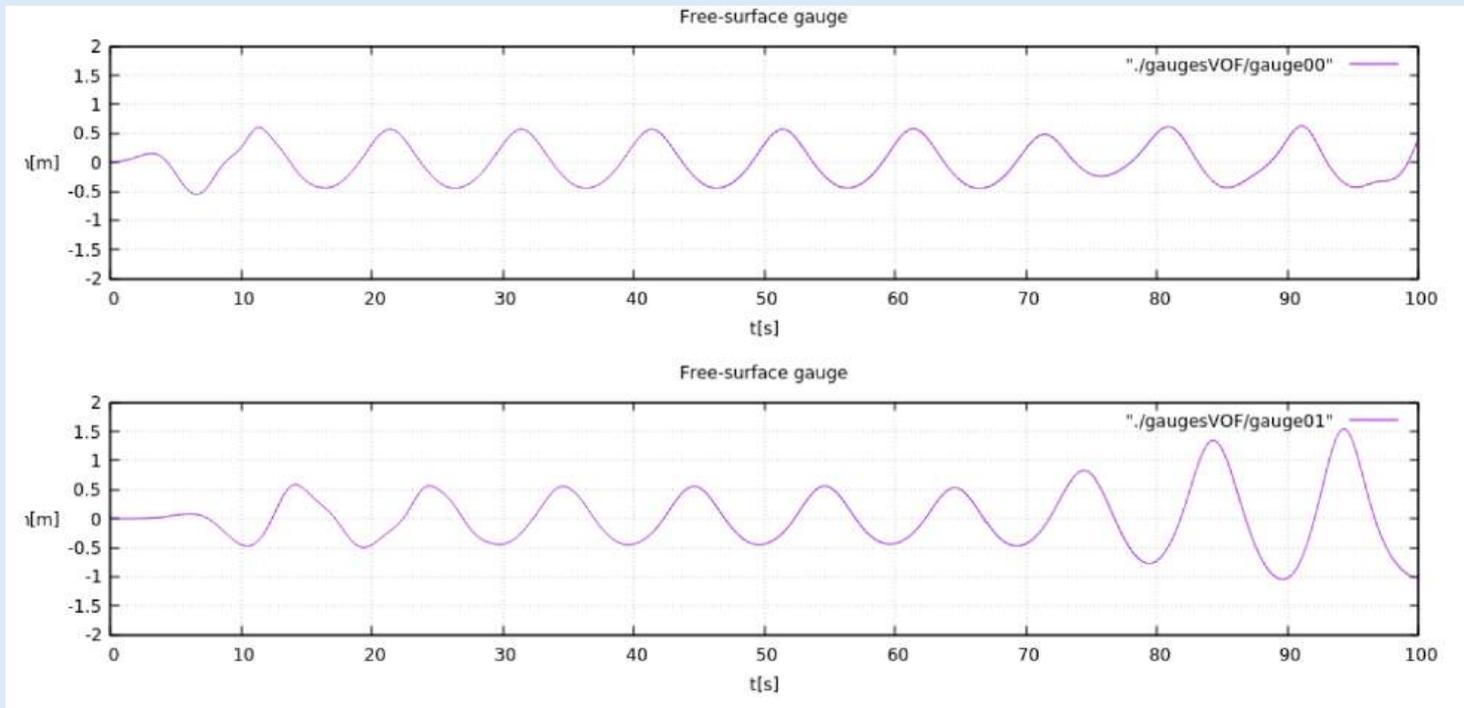
# Setting the fields
echo
runApplication setFields

##### Run Case Serie #####
# Run the application serie
#echo
#runApplication $application
```

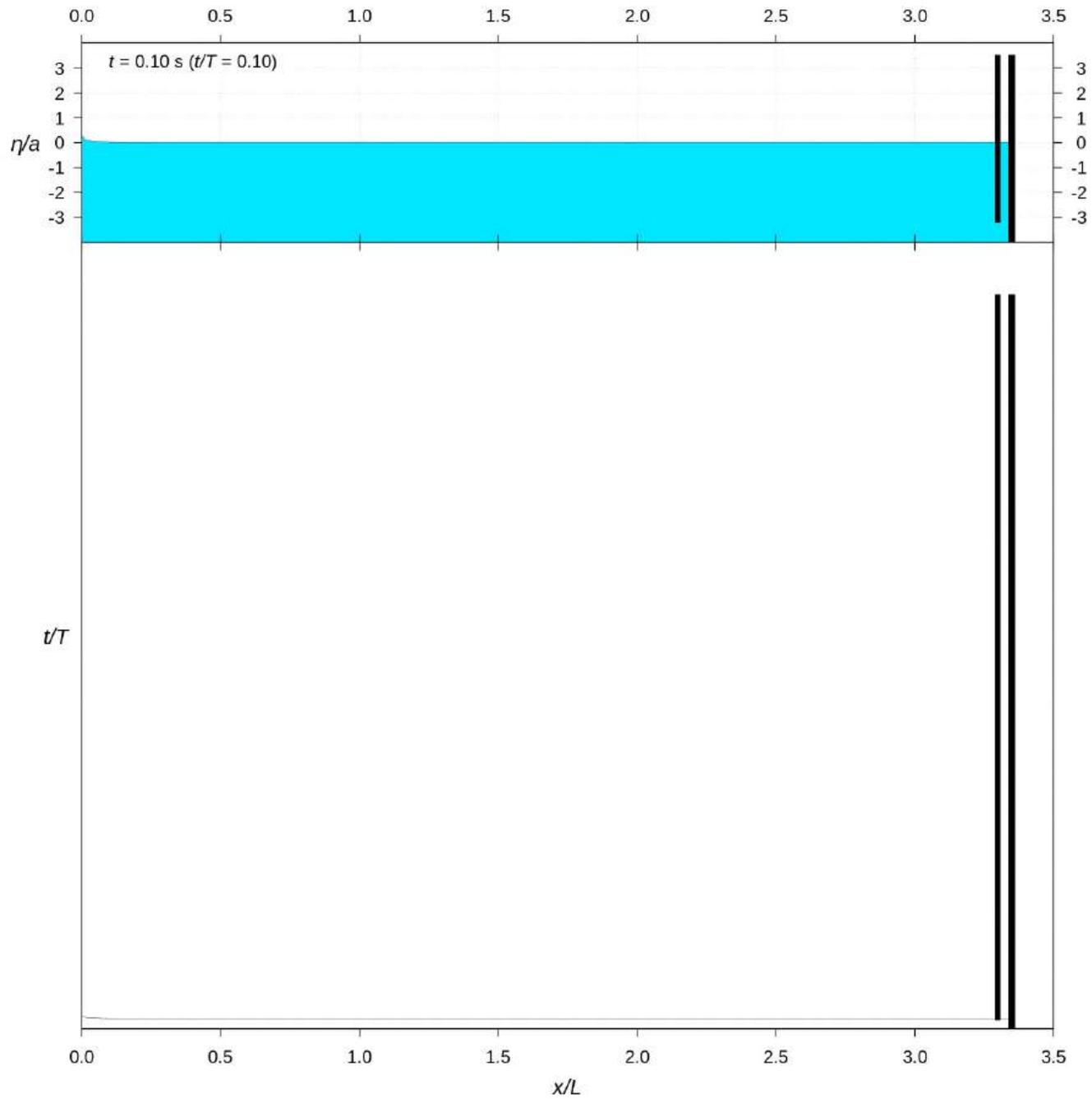
```
##### Run Case Parallel #####  
# Run application parallele  
echo  
runApplication decomposePar  
  
echo  
runParallel $(getApplication)  
  
echo  
runApplication reconstructPar  
  
##### Post-processing #####  
# Post-processing analysis  
  
echo  
echo Calculating probes...  
python postSensVOF_v3.py  
  
echo  
echo Writing isoSurfaces Files...  
python isoSurface_v7.py
```

Gráfico gnuplot para sondas

Para fazer um gráfico rápido das sondas de superfície livre pode-se fazer o script gnuplot

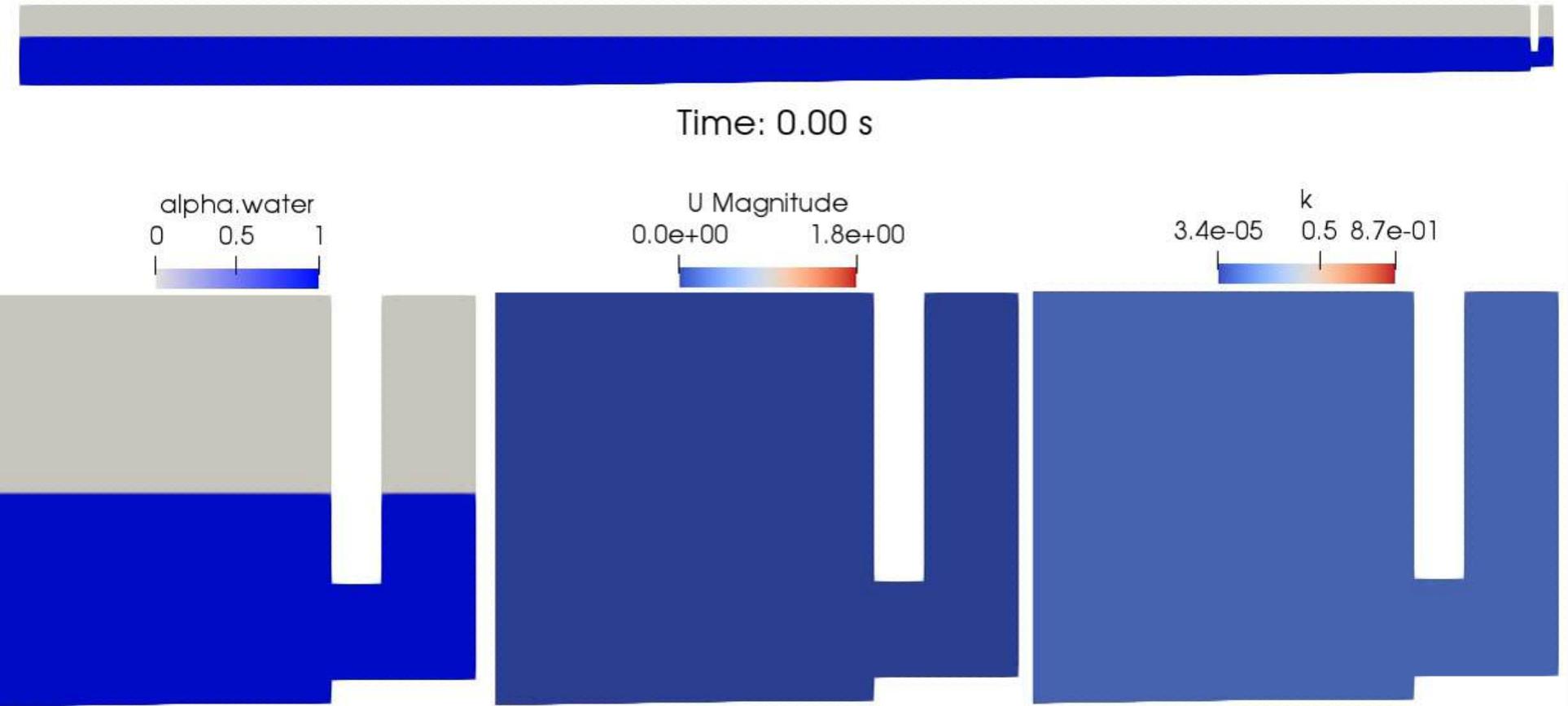


IHFoam/OlaFlow

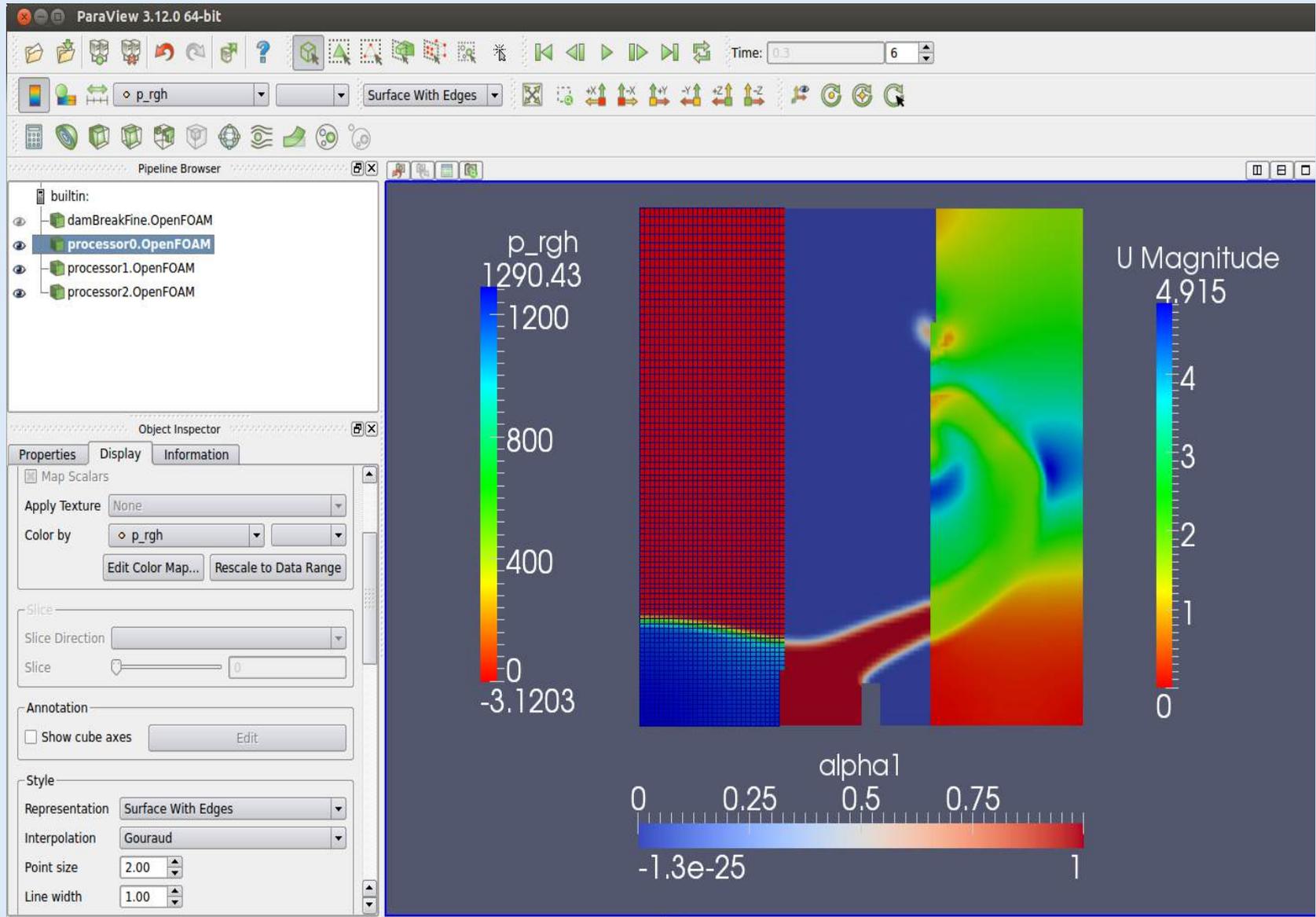


Exemplos: -paraview

Mutriku IHFoam/OlaFlow



Parafoam



Verificação e validação

Verificação

Resolver corretamente as equações.

‘Solve the equations right’.

Verificação do código - verificar se um programa de cálculo resolve correctamente as equações através da avaliação de erros de uma solução conhecida. Técnica das soluções fabricadas (“Method of Manufactured Solutions”)

Verificação dos cálculos - determinação de uma estimativa do erro de uma solução numérica para a qual a solução exacta é habitualmente desconhecida.

Validação

Resolver as equações corretas.

‘Solve the right equations’.

Ex. Modelo de turbulência correto.

Validação

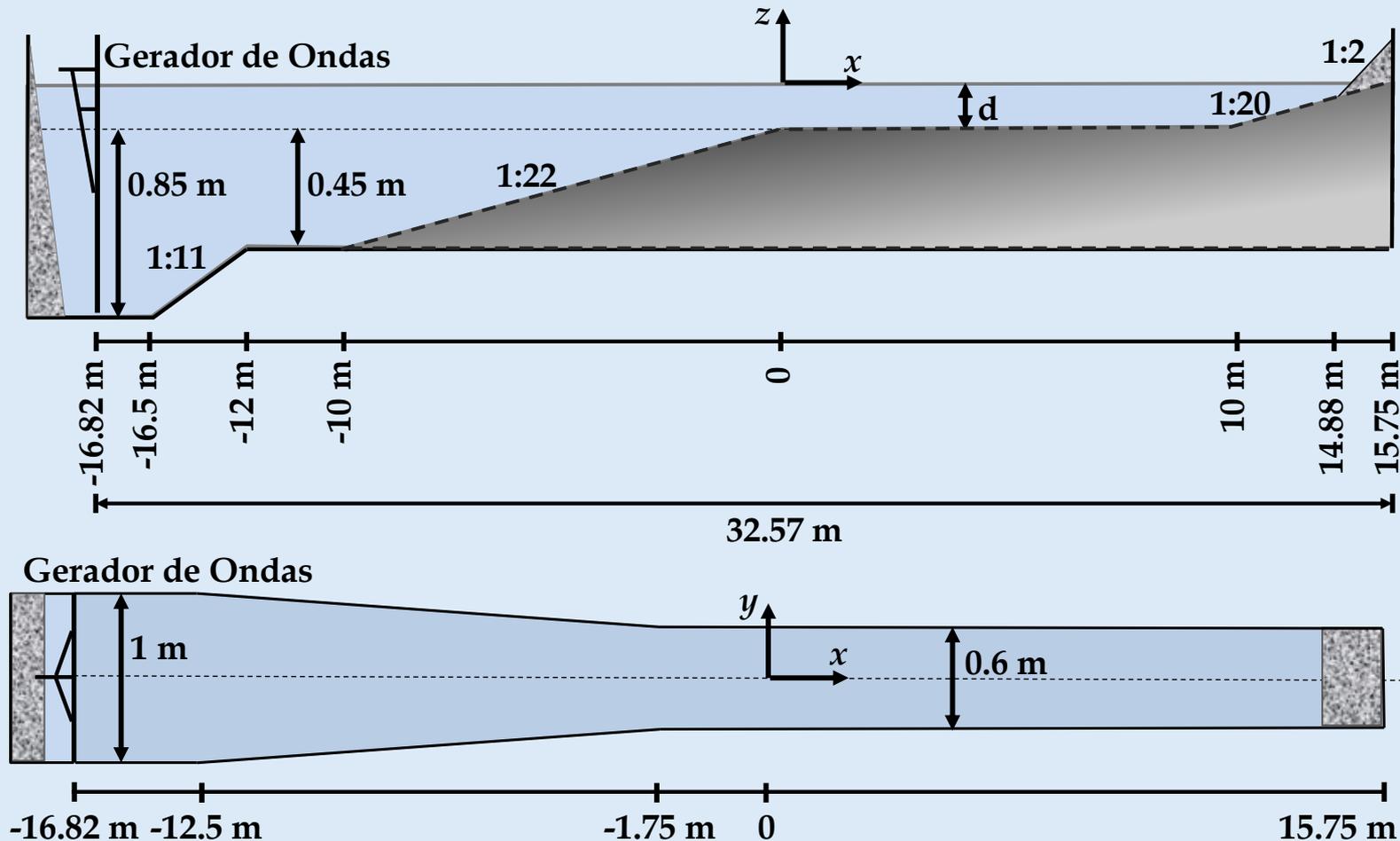
Ensaios em modelo físico



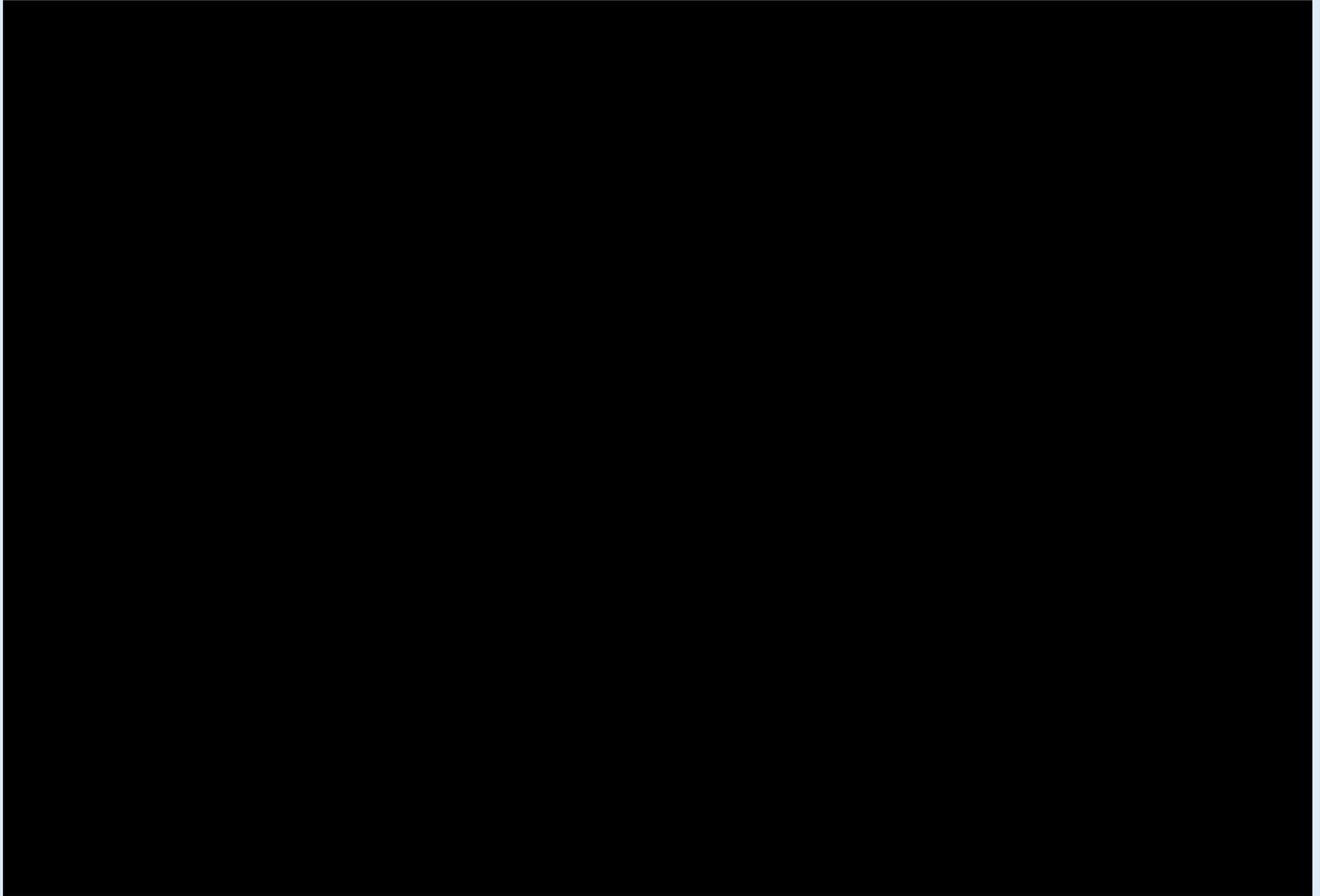
Vista lateral do canal (esquerda), gerador de ondas (centro) e vista de cima do canal (direita).

Ensaio em modelo físico

Canal de ondas: perfil longitudinal (cima) e planta (baixo).



Verificação e validação



Modificac o OpenFOAM

