External Incompressible 3D flow CFD Analysis
OpenFOAM Tutorial
Eng. Sebastian Rodriguez
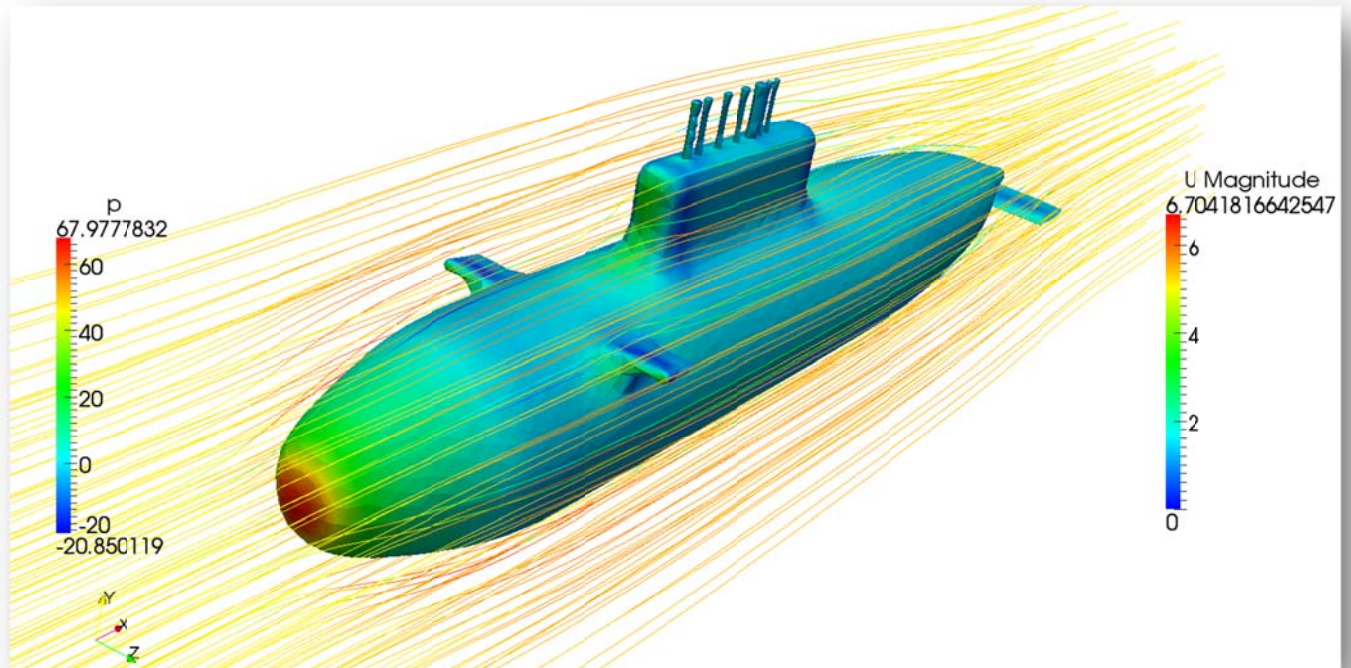
www.libremechanics.com

External Incompressible 3D Analysis

OpenFOAM (Open Source Field Operation and Manipulation) it's an Open Source Software project claim to be one of the best CFD tools currently available, the constant development and its highly technical structure, the fine implementation of common solvers and the possibility to edit and create equations and mathematical cases make OpenFOAM useful tool on researching. OpenFOAM is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, including computational fluid dynamics (CFD). The code is released as free and open source software under the GNU General Public License

"*In common plastic analysis, deformations are easily inferred, but de fluid computational dynamics its governed by a wide group of less known physics laws and variables that make the results almost unpredictable by simple observation"*

As the user is probably aware by now, this document make a number of simplifying assumptions as the tutorial progressed, this is done in the interest of gaining a clearer understanding of the fundamental without getting bogged down in special details and exceptions. By no means it hast the complete history of external 3D fluid analysis handling on OpenFoam, it is much broader in scope that can be presented in a single document such as this, but it is sincerely hoped that this tutorial will enable one to do a better job on the definition, solution and study of this kind of analysis.

Background

The external flow CFD analysis are commonly used to study the behavior of objects submerged in continuous non static fluids, it aims to determine how efficiently a body can move throw the medium and how they affect each other in the process.

Here the fluid it's no confined between wall type conditions such as pipes or confined flow cases, the fluid its free to move around the object and interact only with its external "wet" layer, this means that there is the flow involving the body shape what must be analyzed not the core.

Evaluating the pressure distribution on the structure generated by the resistances of the fluid the engineers can optimize the design to decrease the amount of energy needed to move the body or the disturb impact on the fluid patch that leaves the pass of the structure.

This tutorial its based over the at
www.LibreMechanics.com Simple Openfoam 3D tutorial at http://www.libremechanics.com/?q=node/25

Case introduction.

This basic tutorial its design to be a guide for the creation of simple external incompressible 3D flow cases on OpenFOAM, it must be complemented with further reading and understanding of FEA and CFD theory and by no means this tutorial most replace the although complex yet useful documentation from OpenFOAM itself and related sources.

In this case, the dummy Kilo-Class submarine model its completely submerged under water and its advancing completely straight throw the water medium. It is no common to have an small restricted medium-- boundary like in this example, the user must be careful in make the right assumptions in this type of procedure, the dimensions of the medium affects the turbulence model and may lead to unreal results.

According to merely informative sources this type of submarine has a maximum submerged velocity of 25 knots (12.86 m/s) , but for this first reference simulation the speed will be limited to 10 knots (5.14 m/s). No additional forces, velocity's, or pressure points will be configures over the model, the only goal of the case it's to determine the flow path over the submarine surface.

In real design process not just the hydrodynamic behavior dictates the submarine shape, it involves some deep and complex study's which naval engineers master. *This analysis its completely representative and educational* due the lack of real data such as dimensions, velocity and materials properties the model has been created by merely observation and assimilation of some real cases.

Image 1 Kilo-Class submarine, no attribution.

3D Modeling.

The model was create using merely observation of a Kilo-Class submarine completely submerged on a continuous media, the model stands completely separate form any external surface in horizontal position and align to the X axis.

The whole external surface is made by spline revolutions and curved ends, the meshing of this kind of surfaces will be explained at the Meshing section.

A single body model was created for the submarine, geometry as hatches, propeller, periscope and others were ignored on purpose to keep the model as simple as possible keeping its characteristic shape. Some basic rules as non-negative or cero volume sections on the model are also respected for this 3D model in order to perform an optimal and simple discretization.

The dimensions of the fluid domain respond to an acceptable volume where the fluid can develop its total turbulence , The distance between the domain external layers and the model can affect the pressure or velocity's on the model, The bigger the fluid domain is it will be a greater load to the meshing process and it will require much more memory to perform the analysis and even to compute the result files.

After the model was created a mold surrounding volume counterpart was created to match the external surface in a hollow cavity, this is done by most common CAD tools, the final hollow model has the next characteristics:
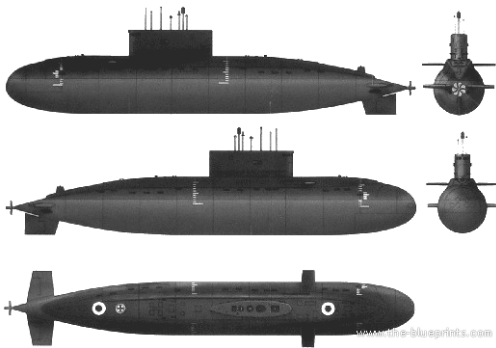
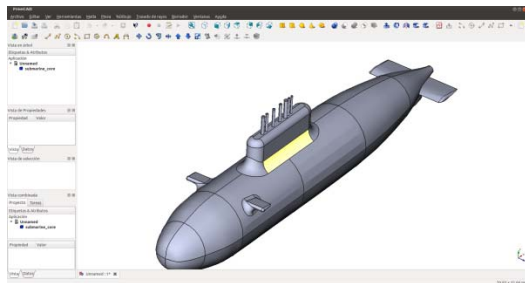

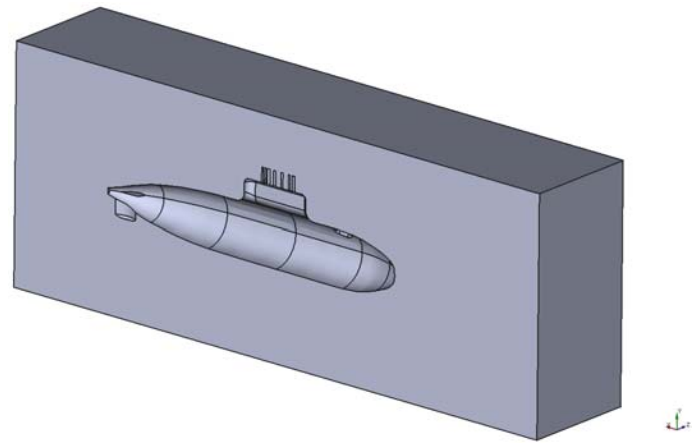**Image 2 Kilo-Class representative sketch, no attribution**

Image 3 CAD core model.

Image 4 3D water volume model, cut.

Water External Domainl:
- Lenght : 130m
- Width: 50m
- hight: 50m
- Total water volumen: 319000 m³

Submarine Model (hole):
- Lenght : 70m
- Width: 20m
- hight: 20m

Units

http://www.openfoam.org/docs/user/basic-file-format.php

Units on OpenFoam need to be set in order to be accepted by the solver, the program makes an early check for unit's congruency and stops if anything unusual is detected. In this case there is no need for unit combinational scheme as other FEA tool which makes easiest the reading of results.

No.	Property	SI unit	USCS unit
1	Mass	kilogram (kg)	pound-mass (lbm)
2	Length	metre (m)	foot (ft)
3	Time	second (s)	
4	Temperature	Kelvin (K)	degree Rankine (R)
5	Quantity	kilogram-mole (kgmol)	pound-mole (lbmol)
6	Current	ampere (A)	
7	Luminous intensity	candela (cd)	

Example: [0 2 -1 0 0 0 0]

Where each of the values corresponds to the power of each of the base units of measurement listed in Table. The table gives the base units for the International (SI) and the United States Customary System (USCS) but OpenFOAM can be used with any system of units. All that is required is that the input data is correct for the chosen set of units. It is particularly important to recognize that OpenFOAM requires some dimensioned physical constants, e.g. the Universal Gas Constant.

Physical parameters

The fluid of interest in this case its common sweet water, incompressible fluid.

It is assumed that the pressure drops are not enough to fall below the vapor pressure of the water and make any phase change during the analysis, so the solver chosen is the SimpleFoam for incompressible fluids:

- Density: 1000 kg/m^3
- Kinematic Viscosity: 1,0 e-6m^2/s
- Inlet velocity: 5,14 m/s

$$k = \frac{2}{3}\left(U'^2_x + U'^2_y + U'^2_z\right)$$

$$\varepsilon = \frac{0{,}09 \cdot k^{0.75}}{B \cdot l}$$

The turbulence model used k-Epsilon in external flows applications has special considerations depending of the application, it may be even not recommended for special turbulent models

The E model needs an special configuration where the medium dimensions are considered, since the is not an external 0 velocity layer (wall) limiting the vortex energy transfer there is not a standard Specific Length magnitude to compute the energy dissipation equation (E), therefor the external combined dimensions of the inlet face and a multiplier factor B can be manipulated to reach a substantial and stable solution to the case.

$$k = \frac{2}{3}\left(U'^2_x + U'^2_y + U'^2_z\right)$$

$$U'^2_x = U'^2_y = U'^2_z$$

$$k = \frac{2}{3}(0.257^2)$$
$$k = 0{,}04403 m^2 s^{-2}$$

$$\varepsilon = \frac{0{,}09 \cdot k^{0.75}}{B \cdot l}$$

$$\varepsilon = \frac{0.09 \cdot 0{,}04403^2 m^2 s^{-2}}{0{,}01 \cdot 2500}$$

$$\varepsilon = 6{,}9791^{-6} \frac{m^2}{s^3}$$

Meshing with Gmsh

Even though OpenFOAM have a build in tool called **blockMesh** for multi-block simple geometry meshing, there is a need in CFD for multiple meshing tools that cover a range of complexity meshing task.

Gmsh is a 3D finite element grid generator with a build-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities. Gmsh is built around four modules: geometry, mesh, solver and post-processing. The specification of any input to these modules is done either interactively using the graphical user interface or in ASCII text files using Gmsh's own scripting language.

- The Gmsh meshing process begins at the import of the CAD geometry, many formats are supported by the tool, depending de CAD used to create the model the user may choose the correct format, in this case the model is saved as a .XX file, opening the file the main shape can be checked for any overlap geometry.
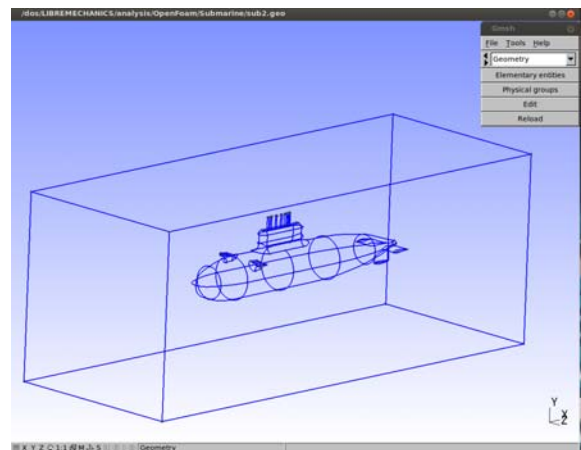

Image 5 CAD inported on Gmsh.

- The boundary are created before the meshing process, using the inside tool of Gmsh of physical parameters we can choose the faces from the model and selecting them together to create the boundary groups, note that the Graphical View of Gmsh won't ask the set name or any other identification, it just create a consecutive set (1,2,3 …) with the group of faces are selected.

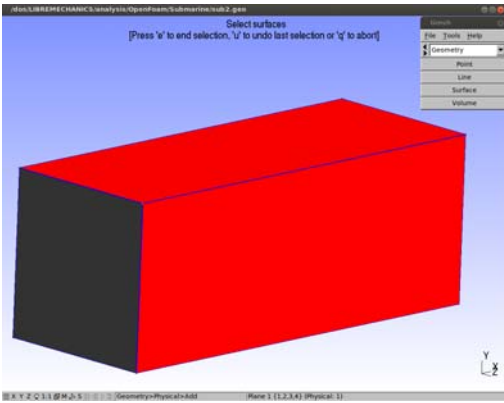*Eng. Sebastian Rodriguez* **www.libremechanics.com**

Image 6 Boundary sets in Gmsh.

To finish the set creations the file case.geo must be changed to give the sets the real names that will be used in the processing of the analysis, changing the name between the parenthesis the user can give costume names to the sets, for this tutorial the case.geo file must look something like this:

It is possible and even easier to create the case.geo file from scratch without the use of the Gmsh GUI, but the user must know for sure the identification of the faces; wich one are walls, inlets, outlets and such.

Just meshing the volume won't be enough to create a good preprocessing model, in this type of analysis the density of the mesh are quit important, to maintain the result consistency; in contrast, other FEA elastic material analysis may use special density concentration on some model places like angles, holes or irregular shapes; but this type of analysis require a uniform density to precisely predict the flow velocity on the medium.

- • The element size can be selected before the mesh process on the Options menu and advanced tab, experimentation and experience may lead to a good element size, it is

recommended to star in a moderate big element size (coarse mesh) and compare results as the mesh becomes thinner, a good pint to stop will be where the resulting data change over the change on the mesh does not exceed the 1% of magnitude.
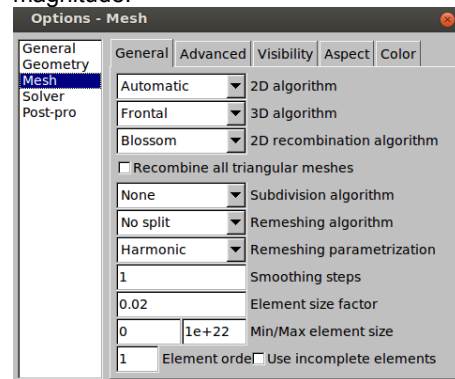

Image 8 Element size configuration Gmsh.

After the process, we check the quality of the mesh, quantity and size of elements and nodes, and finally save it as a Gmsh file "case.msh"

- o **Nodes : 193.637**
- o **Elements: 1,080,753**

- • The export process is t done by a tool named gmshToFoam, a common OpenFoam installation already have this tool available and it doesn't require any configuration to used, just the command:

gmshToFoam case.msh

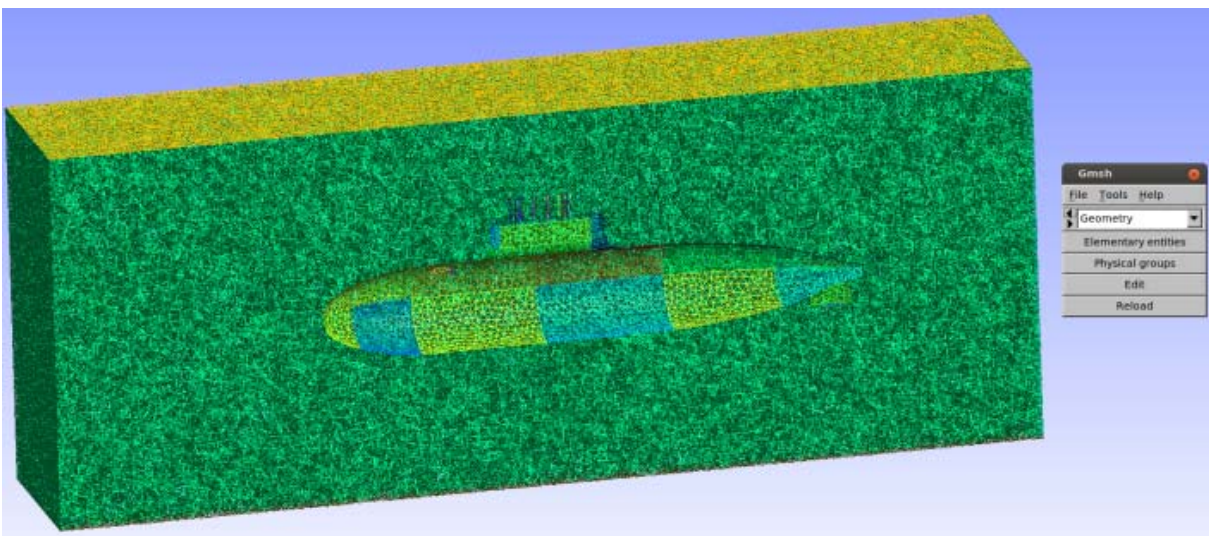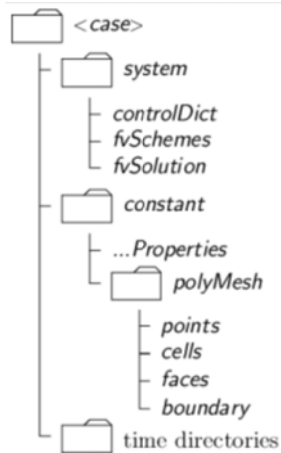This will create the mesh files on the file Constant on the case folder.

Image 7 Mesh from cavity model.

Structuring the case folders

The case definition is not a simple file containing all the information, several files and folder constitute the case structure which order is mandatory for any solver to run. Many pre-processing tool form OpenFoam create some folders by default for the user just to complete and change parameters.

This gives any case great flexibility and scalability where changes over the mesh, boundary conditions, solver parameters can be easily changed without affect any other part of the case. Time directions are the folder for the time steps data which constrain the analysis and those who are solved by the process, normally a "0" folder denotes the first time step where some boundary conditions are save it.

The initial case folder for this tutorial can be downloaded from Libre Mechanics web page or the Source Forge portal

Setting up the case

Even if the mesh files where already created form the import process from Gmsh, the boundary conditions characteristics most be changed depending on the case definition:

http://www.openfoam.org/docs/user/boundaries.php

```
2
(
   IN
   {
      TYPE        PATCH;
      NFACES      102442;
      STARTFACE   1884376;
   }
   WALLS
   {
      TYPE        WALL;
      NFACES      8410;
      STARTFACE   1986818;
   }
)
```

The "0" folder files respond to the initial conditions described in the previous chapters, the files data are edited as bellow:

```
U--------------------------

DIMENSIONS      [0 1 -1 0 0 0 0];

INTERNALFIELD   UNIFORM (0 0 0);

BOUNDARYFIELD
{
IN
{
    TYPE        FIXEDVALUE;
    VALUE       UNIFORM (5.140000
0.000000 0.0000);
}
WALLS
{
    TYPE        FIXEDVALUE;
    VALUE       UNIFORM (0. 0. 0.);
}
}
```

```
E---------------------------
DIMENSIONS      [0 2 -3 0 0 0 0];

INTERNALFIELD   UNIFORM 0.0000069791;

BOUNDARYFIELD
{
IN
{
 TYPE        FIXEDVALUE;
    VALUE       UNIFORM 0.0000069791;
}
WALLS
{
    TYPE
EPSILONWALLFUNCTION;
    VALUE       UNIFORM 1.000000;
}
}
```

```
K--------------------------------------
DIMENSIONS      [0 2 -2 0 0 0 0];

INTERNALFIELD   UNIFORM 0.04403;

BOUNDARYFIELD
{
IN
{
    TYPE        FIXEDVALUE;
    VALUE       UNIFORM 0.1;
}
WALLS
{
    TYPE        KQRWALLFUNCTION;
    VALUE       UNIFORM 0.010000;
```

Eng. Sebastian Rodriguez   **www.libremechanics.com**

```
}
}
```

```
TRANSPORT PROPERTIES --------------------
---

TRANSPORTMODEL  NEWTONIAN;

NU          NU [ 0 2 -1 0 0 0 0 ] 0.000001;

CROSSPOWERLAWCOEFFS
{
   NU0          NU0 [ 0 2 -1 0 0 0 0 ] 1E-06;
   NUINF        NUINF [ 0 2 -1 0 0 0 0 ] 1E-
06;
   M          M [ 0 0 1 0 0 0 0 ] 1;
   N          N [ 0 0 0 0 0 0 0 ] 1;
}

BIRDCARREAUCOEFFS
{
   NU0          NU0 [ 0 2 -1 0 0 0 0 ] 1E-06;
   NUINF        NUINF [ 0 2 -1 0 0 0 0 ] 1E-
06;
   K          K [ 0 0 1 0 0 0 0 ] 0;
   N          N [ 0 0 0 0 0 0 0 ] 1;
}
```

The maximum steps for the solver may be limited to save resources on the solving process but even this number is high the process stop at the step where the convergence criteria is reached.

The step "time" can be "anything" comfortable value which only controls the format of the steps used by the solver to reach the stability solution, due the solver characteristic (Steady-state solver for incompressible, turbulent flow) the only interesting step for results interpretation will be the last one were the convergence is reached, this result state is time independent; any other step is an approximation, a footprint of the iterative process of the solver.

```
ControlDict ------------------------
----

application    simpleFoam;
startFrom      latestTime;
startTime      0;
stopAt         endTime;
endTime        800;
deltaT         1;
writeControl   timeStep;
writeInterval  40;
purgeWrite     0;
writeFormat    ascii;
writePrecision 6;
writeCompression off;
timeFormat     general;
```

```
timePrecision  6;
runTimeModifiable true;
```

```
Relaxation factors ----------------
---------------------

relaxationFactors
{
   p          0.3;
   U          0.7;
   k          0.7;
   epsilon    0.7;
   R          0.7;
   nuTilda    0.7;
}
```

Solving the case

The size of the mesh and the steps the solver must run, require the use of multiple cores for the OpenFOAM solver to decrease the solving time.

- **./cleanAll.sh**
    - *This command cleans the working folder from previous results.*
- **mpirun -np 4 simpleFoam -parallel**

The solver creates the steps folders on the main work case direction, each step contains the velocity and pressure values, this structure allows to exchange result data from case to case.

Its recommendable to pay attention to the console output to capture any warning or error from the solver messages.

Post processing

ParaView is an open source mutiple-platform application for interactive, scientific visualization. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities. ParaView allows to import the model to visualize the result and also add another mesh, images and cad models to references, for example the model of the submarine created at firt to generate the hollow water body can be combined with the result files to better understanding.
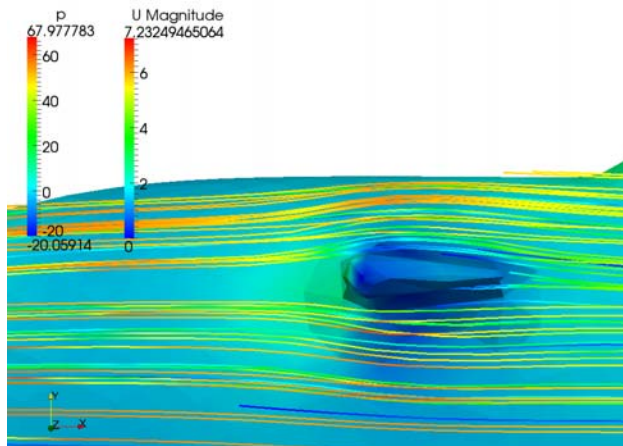
Image 9 Flow around front rudder.

The interest data for this tutorial will be the flow path over the submarine shape and how the water interacts, the pressure points on the structure and the turbulence generated.

Interpreting the pressure concentration around the cavity the submarine design can be later loaded in another analysis to check the material resistance.
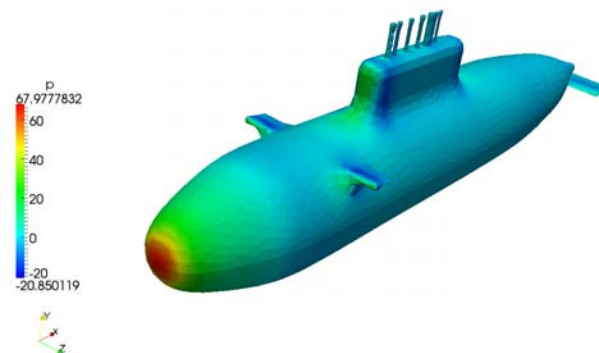

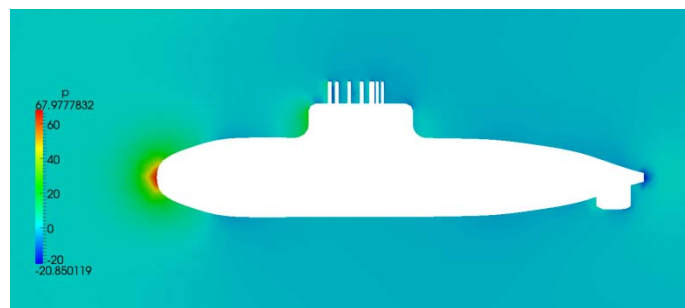
Image 11 Pressure concentration on core geometry.

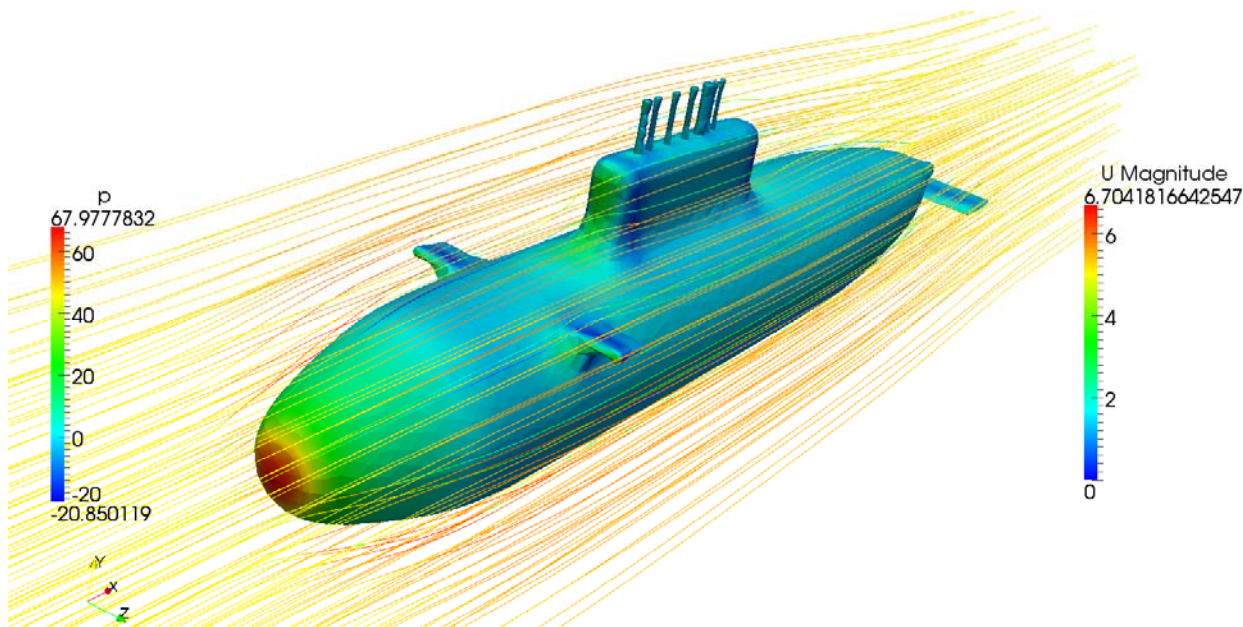**Image 12 Pressure concentration on fluid.**
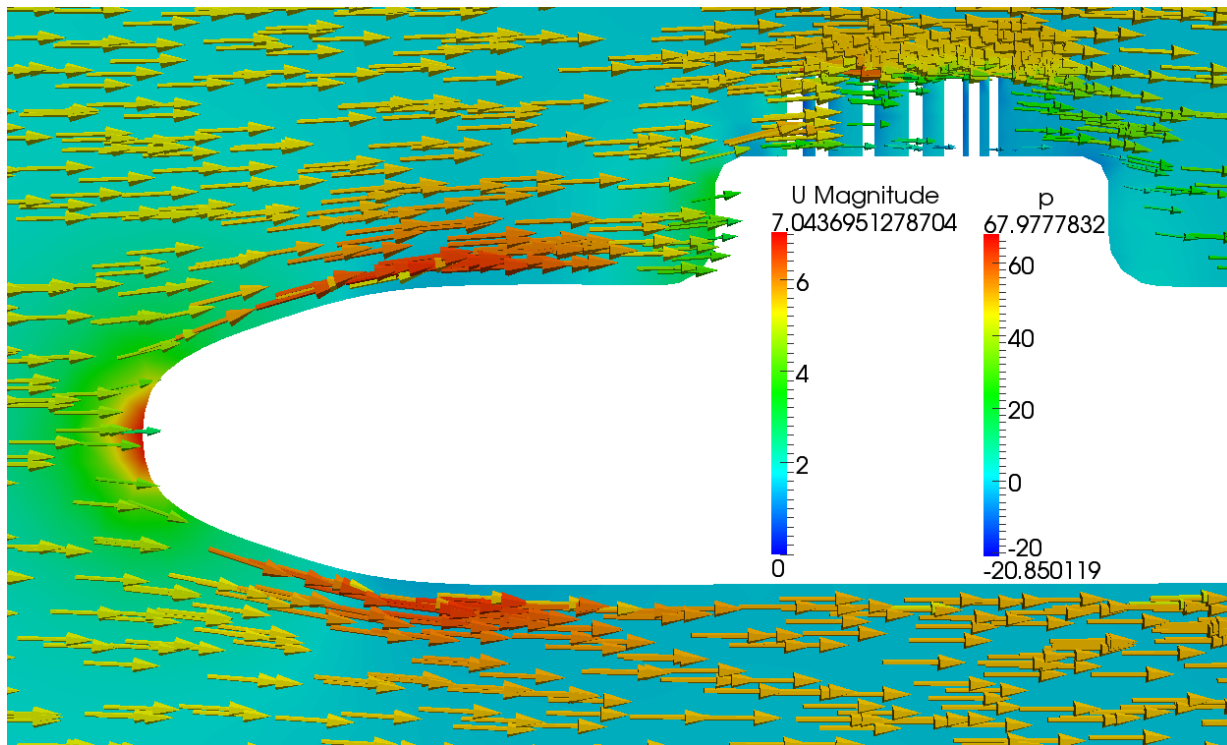

Image 10 Velocity and pressure results, global.

Eng. Sebastian Rodriguez **www.libremechanics.com**

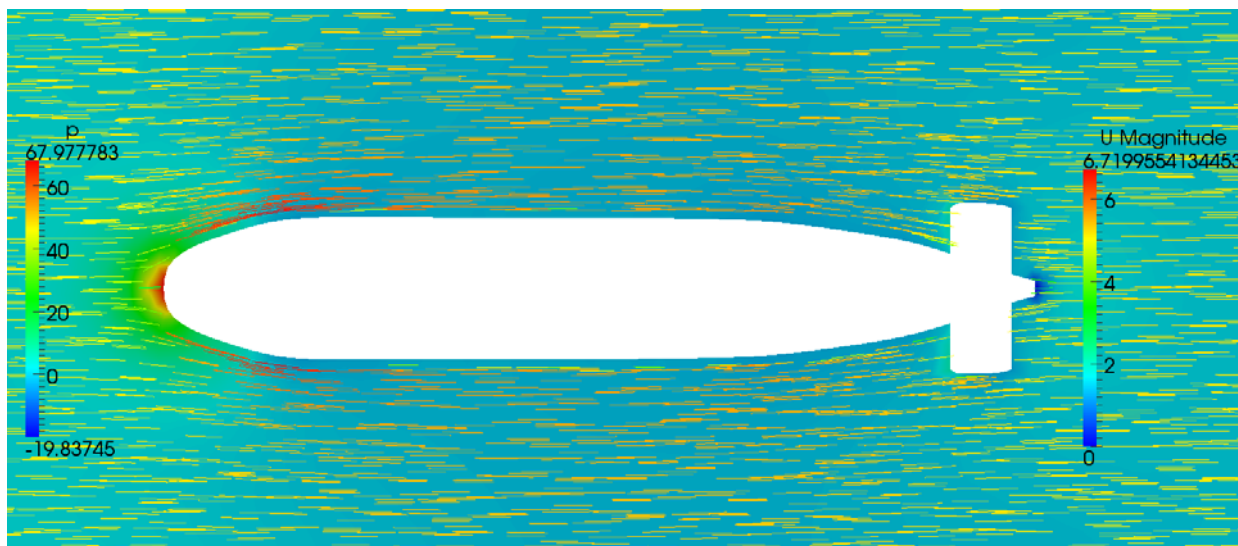**Image 13 Velocity and pressure over a cut view.**

Image 14 Velocity and pressure over a another view.

General Sources:

- Book: Hirsch, Numerical Computation of Internal&External Flows, 1994

- Book: McDonough, Lectures in Computational Fluid Dynamics of Incompressible Flow, 2007

- Web: OpenFoam tutorials, December 2013, http://www.openfoam.org/docs/user/tutorials.php

- Web: The Durham OpenFOAM Tutorial, December 2013, http://www.dur.ac.uk/g.l.ingram/download/openfoamtutorial2012.pdf

- Web: Simple beginning 3D OpenFOAM Tutorial, December 2013 http://www.libremechanics.com/?q=node/25

- Web: Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities, December 2013, http://geuz.org/gmsh/

- Web: OpenFoam solver documentation, December 2013, http://www.openfoam.com/features/standard-solvers.php

- Web: Choosing Turbulence Models for Your Applications, December 2013, http://www.innovative-cfd.com/turbulence-models.html

- Web: Guidelines for Specification of Turbulence at Inflow Boundaries, December 2013, http://www.esi-cfd.com/esi-users/turb_parameters/

Acquiring the case files

For the ease follow of this tutorial the different used and generated files named are available for download, allowing the user to skip or compare any step of the tutorial by its own. Please keep in mind that any file may vary from user to user by the meshing and computational conditions, but it does not meaning this difference will represent an error of processing.

- Geometry
- Mesh
- Case files
- Result Files

Most of the documents recourses as images and this tutorial its available at http://www.libremechanics.com/ and the sourceforge page

More Information

There are multiple ways to acquire more information about OpenFoam and CFD analysis in general, useful for further work:

- OpenFOAM official documentation http://www.openfoam.org/docs/
- OpenFOAM wiki http://openfoamwiki.net/index.php/Main_Page

- Libre Mechanics web page:

  http://www.libremechanics.com/

Please feel free to redistribute comment, suggest and contribute to this or any documentation found on **Libre Mechanics** web by contacting the author at contribute section.