



**AECENAR**

Association for Economical and Technological Cooperation  
in the Euro-Asian and North-African Region

[www.aecenar.com](http://www.aecenar.com)



MEAE – *A member of AECENAR Applied Research Center*

[www.aecenar.com/institutes/meae](http://www.aecenar.com/institutes/meae)



[www.temo-ek.de](http://www.temo-ek.de)

# **Abstract**

Computational Fluid Dynamics (CFD) is the study of fluid flow dynamic status using computer software. It is undergone for a prototype of an incineration power plant, present in Ras Nhache/Batroun, North Lebanon. There were some problems concerning the design which should be solved by a CFD study. In our study of computational fluid dynamic (CFD) we are interested to find the pressure and velocity values for this incineration power plant, in order to strengthen the used computer tools. We discretize the design in order to calculate the values for each point; the discretization makes the calculation possible - using a solver. Then the results are visualized.

# **Résumé**

Le dynamique des fluides computationnelle est l'étude de l'état dynamique de l'écoulement de fluide en utilisant un logiciel informatique; pour un prototype de centrales d'incinération, présents dans RasNhach, nous avons quelques problèmes devraient être résolus en faisant étude CFD pour la conception. Dans notre étude de la dynamique des fluides computationnelle (CFD), nous avons intérêt pour trouver les valeurs de pression et de vitesse pour une centrale d'incinération, afin de renforcer l'équipement utilisé. Et nous discrétisons la conception afin de calculer les valeurs pour chaque point, la discrétisation faciliter le calcul, en utilisant un solveur qui peut nous donner les résultats avec visualisation.

# Table of contents

<b><i>Abstract</i></b>	<b><i>II</i></b>
<b><i>Table of contents</i></b>	<b><i>III</i></b>
<b><i>List of figures</i></b>	<b><i>IV</i></b>
<b><i>List of symbols</i></b>	<b><i>VI</i></b>
<b><i>Introduction</i></b>	<b><i>1</i></b>
<b><i>1. Chapter I: Basics</i></b>	<b><i>3</i></b>
1.1.Incineration power plants	3
1.2.Study of literature	4
1.3.Arguments supporting and against incinerations	6
1.4.History	8
1.5.Pump utility	10
1.6.Software Tools used	11
1.6.1. Discretization methods	13
1.7.What are the Navier-Stokes Equations?	13
1.7.1. Incompressible fluid flow	14
1.7.2. Newtonian and Non-Newtonian fluid flow	14
<b><i>2. Chapter II: Contribution</i></b>	<b><i>16</i></b>
2.1.Solve incineration design	16
2.1.1. OpenFOAM solver	16
2.1.2. Using Gmsh	16
2.1.3. Elmer solver	18
2.1.4. Using Elmer	20
<b><i>3. Chapter III: Results and discussion</i></b>	<b><i>31</i></b>
<b><i>4. Conclusion and future work</i></b>	<b><i>39</i></b>
<b><i>5. References</i></b>	<b><i>40</i></b>
<b><i>6. Annex</i></b>	<b><i>42</i></b>
6.1.Annex 1: Planning in March 2015	42
6.2.Annex 2: Updated Planning July 2015	42
6.3.Annex 3: Tool chains used for CFD analysis	43
6.4.Annex 4: Offers for 30 MW Steam Turbine and Pump	44

# List of figures

Figure 1-1: steps in a incineration an power plant.....	3
Figure 1-2: illustrating of the incineration power plant zone .....	4
Figure 1-3: Distribution of incineration power plant in Europe .....	5
Figure 1-4: An incineration power plant in (A) Torsvikverket in Jönköping (B) Roskilde, Denmark (C) Esbjerg Denmark (D) near Bergen (Oslo).....	5
Figure 1-5: CFD plot for assessment of residence time in an afterburning chamber in Esberg, Denmark incineration power plant.....	6
Figure 1-6: The real flow (A) and the computation flow form (B) .....	10
Figure 1-7: Fluid path through the centrifugal pump. ....	10
Figure 1-8: FreeCAD design example .....	11
Figure 1-9: OpenFOAM window when calculation velocity values.....	11
Figure 1-10: Elmer windows for a quarter pipe .....	12
Figure 1-11: Paraview window for the velocity values .....	12
Figure 1-12: Shear stress vs Rate of shear strain for several categories .....	15
Figure 2-1: Chain of tools 1 using OpenFOAM .....	16
Figure 2-2: FreeCAD incineration power plant design.....	17
Figure 2-3: Meshing incineration power plant design .....	17
Figure 2-4: Chain of tools 2 using Elmer.....	18
Figure 2-5: Design of a quarter pipe introduce in Elmer .....	18
Figure 2-6: Noted design.....	19
Figure 2-7: Setup for each part (the same) .....	20
Figure 2-8: Choice of Navier-Stokes equation .....	21
Figure 2-9: iteration conditions in Elmer .....	22
Figure 2-10: Material used for study .....	23
Figure 2-11: Initial conditions introduce in Elmer .....	24
Figure 2-12: Inlet boudary conditions given by the turbine .....	25
Figure 2-13: Meshed pipe viewing in Elmer.....	26
Figure 2-14: Pipe outlet boundary condition .....	26
Figure 2-15: Meshed corner viewing in Elmer.....	27
Figure 2-16: Corner outlet boundary condition.....	27
Figure 2-17: Water path meshing in Gmsh .....	28
Figure 2-18: Water path illustrated in Elmer .....	29
Figure 2-19: Water path outlet boundary condition .....	30
Figure 3-1: Elmer velocity color results (in m/s) .....	31
Figure 3-2: Elmer velocity vector results in m/s .....	31
Figure 3-3: Elmer pressure color results in Pa .....	32
Figure 3-4: The studied design.....	33
Figure 3-5: Water path .....	33
Figure 3-6: Result files in Elmer .....	34
Figure 3-7: Meshed pipe viewing in Elmer .....	35
Figure 3-8: Velocity variation x-component in m/s.....	35
Figure 3-9: Velocity variation y-component in m/s.....	36
Figure 3-10: Velocity variation z-component in m/s .....	36

Figure 3-11: Velocity vector into the pipe in m/s .....	37
Figure 3-12: Pressure variation in Pascal.....	37
Figure 3-13: Input water pressure into the pipe in Pascal.....	38
Figure 3-14: Output water pressure in Pascal .....	38

# List of symbols

$\rho$  is density of the fluid (assumed to be a known constant);

$u \equiv (u_1, u_2, u_3)$

$p$  is fluid pressure;  $\mu$  is viscosity, and  $FB$  is a body force.

$D/Dt$  is the substantial derivative expressing the Lagrangian, or total, acceleration of a fluid particle in terms of a convenient laboratory-fixed Eulerian reference frame;

$\nabla$  is the gradient operator;

$\Delta$  is the Laplacian;

$\nabla \cdot$  is the divergence operator;

$V(t)$  is the velocity;

$\mu$  is the constant shear stress and is linearly dependent on the velocity gradient;

$Q$  is the volume flow.

# Introduction

Fluid dynamics is the study of fluid flow; in our case we study this flow for an incineration power plant; studying the flow of steam and liquid water into the pipes (The main critical component is the pressure vessel with about 100 bar pressure differences).

But this study should be a computational study; our goal is to obtain codes that take the fluid conditions and make us sure that we can design an upscaled version of the vaporizer of an incineration power plant (about 40 MW) in Tripoli.

These codes take the equations of fluid dynamics and make discretization to solve them with computer programs.

But we have to know that the continuous equations of fluid dynamics are hardly possible to solve and computational fluid dynamics makes it easier to solve them after discretization.

In our study; we will show the velocity and pressure in the path of water from pump into the water tank by simulation.

To apply this goal we have to use codes of an existing program like OpenFOAM or Elmer and use a designer program (as FreeCAD) a visualization tool as Paraview.

In the report three chapters contain the study; in the first chapter we introduce the basics that we use in our study, in the second chapter the contribution is done, in the third chapter the results are published.

At the beginning we had the following **working packages**:

- CAD Modeling: Up-scaled CAD Model with "ProE" or "FreeCAD" (was to be done by another student –see above).
- Mesh Generation: A mesh generation "C++" code shall be taken from the open source code "OpenFoam" and migrated to TEMO\_IPP-CFD tool - in our research we use Gmsh for meshing.

- Solver: A finite difference and a finite volume "C++" code shall be taken from the open source code "OpenFoam" and migrated to TEMO\_IPP-CFD tool or using another tool for solving (Elmer).
- Visualization: Shall be done with the tool Paraview.



# 1. Chapter I: Basics

In this chapter we introduce the fundamentals of our study, and cite some other precedent studies in this field, with a history concerning the development of computational fluid dynamics.

## 1.1. Incineration power plants

INCINERATION is a waste treatment technology that involves burning commercial, residential and hazardous waste. Incineration converts discarded materials, including paper, plastics, metals and food scraps into bottom ash, fly ash, combustion gases, air pollutants, wastewater, wastewater treatment sludge and heat.

It follows several steps illustrated in figure 1-1.

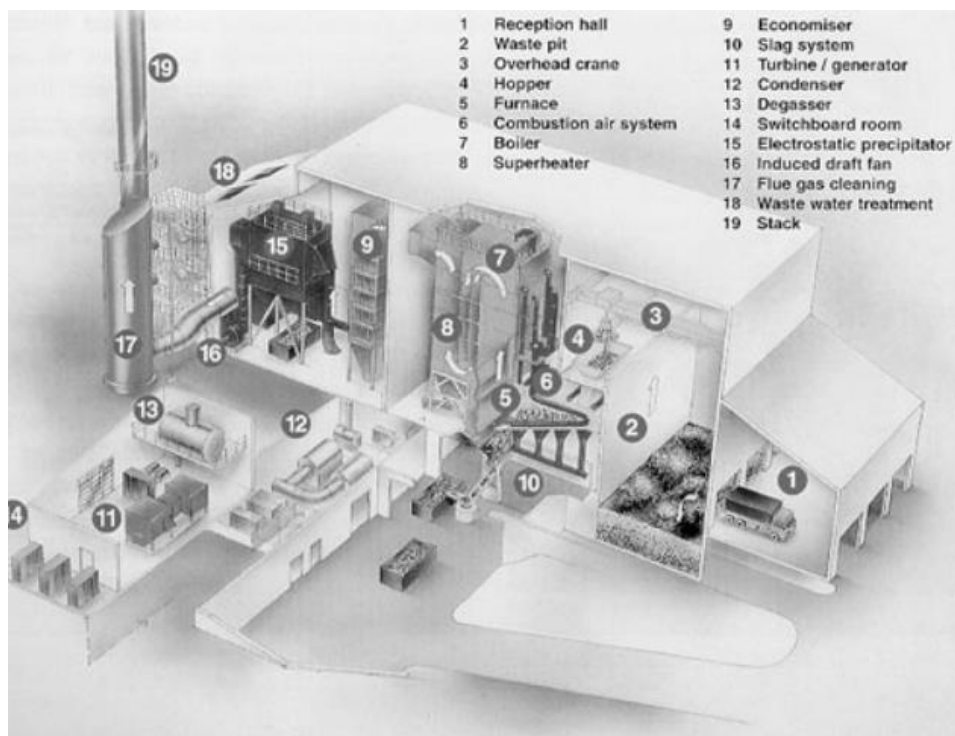


Figure 1-1: steps in a incineration an power plant

The ENERGIZE multistage grate is the core and the bottom surface of the furnace. It carries and transports the burning solid matter from the feeding section to the ash extractor.

The grate consists of identical elements grouped in 3 zones:

Drying zone, combustion zone, burnout zone.

1. In the drying zone, moisture in the waste is evaporated by the heat in the furnace and the radiation from the first empty pass which is positioned just above this zone.
2. In the combustion zone, the actual combustion takes place.
3. The burnout zone is a buffer to guarantee the burnout quality and to cool the ashes.

A step between each zone causes the waste to drop from one zone to the next, creating the necessary mixing of the burning matter<sup>[1]</sup>.

The 3 zones illustrate in the figure 1-2.

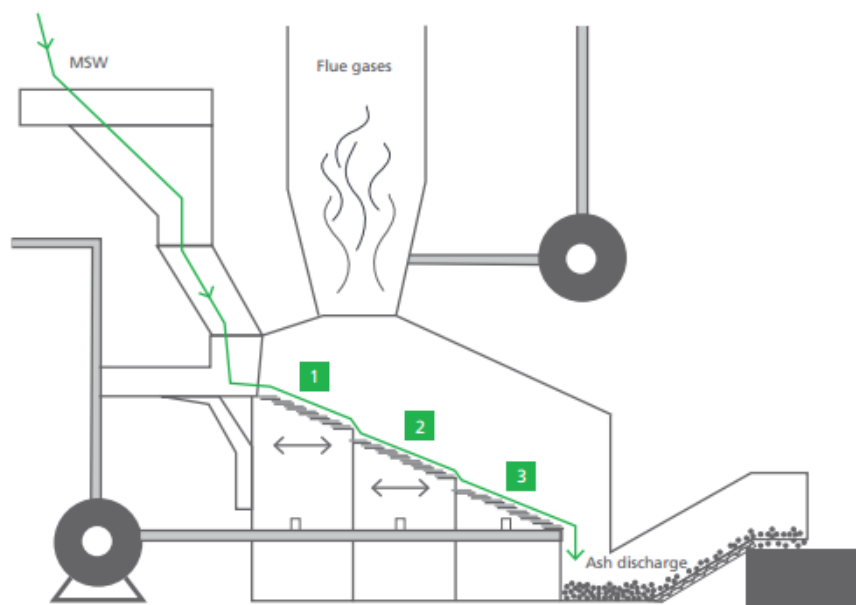


Figure 1-2: illustrating of the incineration power plant zone

## 1.2. Study of literature

There are many applications of waste incineration power plants, especially in Europe, see Figure 1-3, and it is necessary to study them. But it is impossible to view published studies in internet; we can only see the incineration power plant's design with global studies.

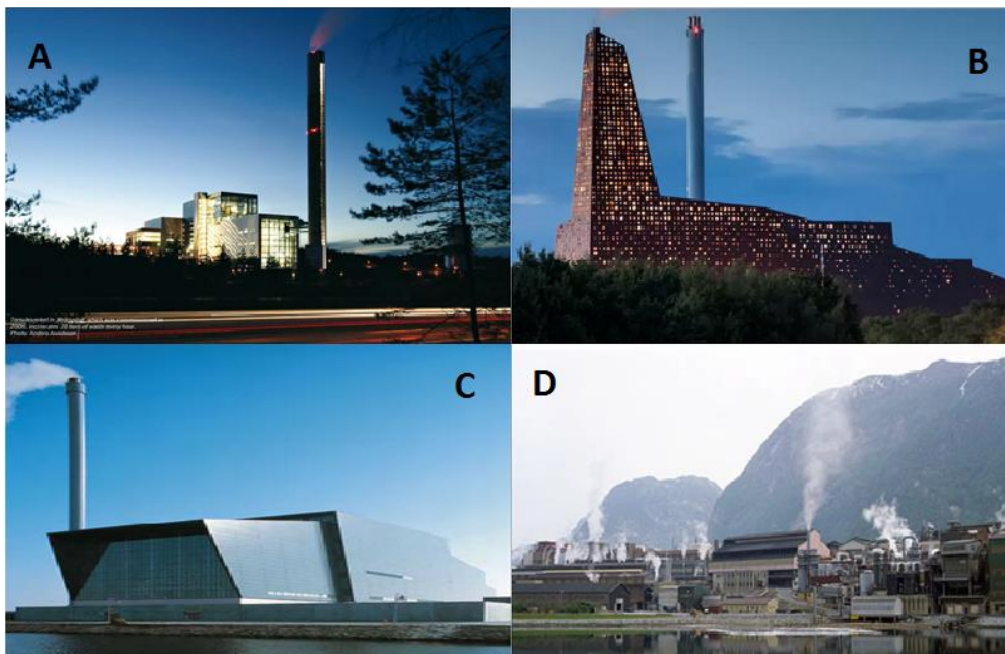


Figure 1-4: An incineration power plant in (A) Torsvikverket in Jönköping (B) Roskilde, Denmark (C) Esbjerg Denmark (D) near Bergen (Oslo).

An incineration plant in Sweden Figure 1-4 (A) represents a similar amount of waste in the industry as well. Waste incineration provides heat corresponding to the needs of 810,000 homes, around 20 per cent of all the district-heating produced. It also provides electricity corresponding to the needs of almost 250,000 homes.

International comparisons show that Sweden is the global leader in recovering the energy in waste<sup>[2]</sup>.

In Roskilde, Denmark figure 1-4 (B) when waste from nine surrounding municipalities and from places abroad will be incinerated at the Roskilde plant. According to its developer KARA/NOVEREN, this will be enough to produce electricity for around 65,000 homes and heat for around 40,000 homes<sup>[3]</sup>.

We can say that the incineration power plant of in Esbjerg, Denmark figure 1-4 (C), attached with some historic study concerning development of the incineration power plant design and CFD information<sup>[4]</sup> in figure 1-5:

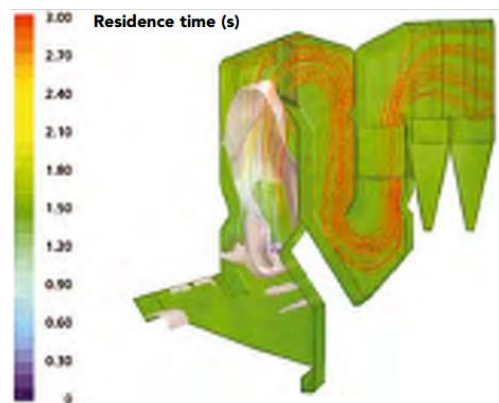


Figure 1-5: CFD plot for assessment of residence time in an afterburning chamber in Esbjerg, Denmark incineration power plant.

Christoffer Back Vestli says, communications adviser for the Oslo in Bergen figure 1-4 (D): municipality. "At the moment, the city of Oslo can take 410,000 tonnes of waste a year and we import 45,000 tonnes from the UK. Europe as a whole currently dumps 150m tonnes of waste in landfills every year, so there is clearly great potential in using waste for energy."<sup>[5]</sup>

### 1.3. Arguments supporting and against incinerations

Usage of incineration for waste management is divisive. The debate for incinerators generally involves business interests, regulations of government, activists of environment and citizens.

### *Arguments supporting incinerations*

The first concern for incineration stands against its injurious effects over health due to production of furans and dioxin emission. However, the emission is controlled to greater extent by developing of modern plants and governmental regulations.

Incineration plants are capable for producing energy and can substitute power generation plants of other sort.

The bottom ash after the process is completed is considered non-injurious that still is capable for being land filled and recycled.

Fine particles are removable by processing through filters and scrubbers.

Treating and processing medical and sewage waste produces non-injurious ash as product.

### *Arguments against incinerations*

Extremely injurious matter needs adequate disposing off. This requires additional miles and need special locations for land filling this material.

Although after a lot of regulations and restrictions and developments concerns are still alive about emission of furans and dioxins.

Incinerating plants are producers of heavy metals, which are injurious even in minor amounts.

IBA (**Incinerator Bottom Ash**) is consistent over a considerably high level of heavy metals and can prove fatal if they are not disposed off or reused properly.

Initial investment costs are only recovered through long periods of contract for incinerating plants.

Local communities always have opposed the presence of incinerating plant in the locality.

The upheld view is to recycle, reuse and waste reduction instead of incineration<sup>[6]</sup>.

## 1.4. History

In the past, incineration was conducted without separating materials thus causing harm to environment. This unseparated waste was not free from bulky and recyclable materials, even. This resulted in risk for plant workers health and environment. Most of such plants and incinerations never generated electricity.

Incineration reduces the mass of the waste from 70 to 80 percent. This reduction depends upon the recovery degree and composition of materials. This means that incineration however, does not replace the need for landfilling but it reduced the amount to be thrown in it.

Incineration comes with a number of benefits in specific areas like medical wastes and other life risking waste. In this process, toxins are destroyed when waste is treated with high temperature.

Incineration or thermal treatment of waste is much popular in countries like Japan where there is scarcity of land. The energy generated by incineration is highly demanded in countries like Denmark and Sweden. In the year 2005 it was estimated that 4.8 percent of the electricity consumed by Danish nation was produced by incineration and the amount of heat was some 13.7 percent out of total. Other than Denmark and Sweden many European countries are recovering heat and electricity from waste<sup>[6]</sup>.

During the last three or four decades, computer simulations of physical processes have been used in scientific research and in the analysis and design of engineered systems. The systems of interest have been existing or proposed systems that operate at design conditions, off-design conditions, failure-mode conditions, or accident scenarios. The systems of interest have also been natural systems. For example, computer simulations are used for environmental predictions, as in the analysis of surface-water quality and the risk assessment of underground nuclear-waste repositories. These kinds of predictions are beneficial in the development of public policy, in the preparation of safety procedures, and in the determination of legal liability. Thus, because of the impact that modeling and simulation predictions

can have, the credibility of the computational results is of great concern to engineering designers and managers, public officials, and those who are affected by the decisions that are based on these predictions<sup>[7]</sup>.

The development of computer fluid dynamics has been closely associated with the evolution of large high-speed computers. At first the principal incentive was to produce numerical techniques for solving problems related to national defense. Soon, however, it was recognized that numerous additional scientific and engineering applications could be accomplished by means of modified techniques that extended considerably the capabilities of the early procedures. This paper describes some of this work at The Los Alamos National Laboratory<sup>1</sup>, where many types of problems were solved for the first time with the newly emerging sequence of numerical capabilities. The discussions focus principally on those with which the author has been directly involved<sup>[8]</sup>.

CFD **Computational fluid dynamics** is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions<sup>[1]</sup>.

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- Mathematical modeling (partial differential equations)
- Numerical methods (discretization and solution techniques)
- Software tools (solvers, pre- and post-processing utilities)

CFD enables scientists and engineers to perform "numerical experiments" (i.e. computer simulations) in a "virtual flow laboratory"<sup>[9]</sup>.

In the figure 1-6 (A and B) we see the comparison of the real fluid flow which is continuous and the computational fluid flow which is discretize.

---

<sup>1</sup> at Los Alamos Laboratories the first nuclear bomb was developed during World War II.

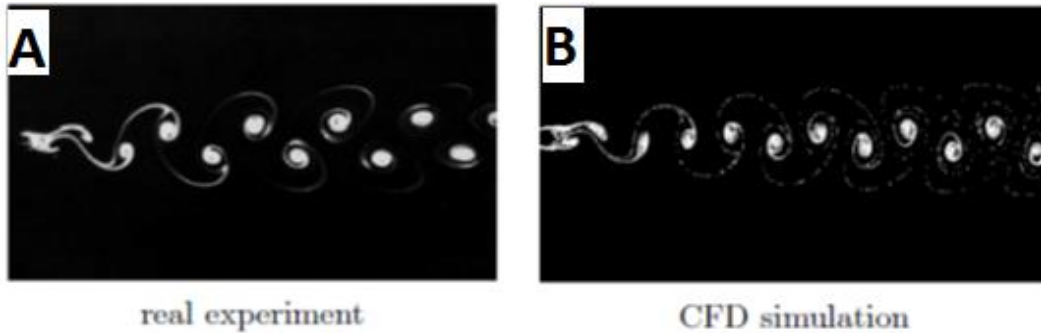


Figure 1-6: The real flow (A) and the computation flow form (B)

## 1.5. Pump utility

An increase in the fluid pressure from the pump inlet to its outlet is created when the pump is in operation. This pressure difference drives the fluid through the system or plant. The centrifugal pump creates an increase in pressure by transferring mechanical energy from the motor to the fluid through the rotating impeller. The fluid flows from the inlet to the impeller center and out along its blades. The centrifugal force hereby increases the fluid velocity and consequently also the kinetic energy is transformed to pressure. Figure 1-7 shows an example of the fluid path through the centrifugal pump<sup>[17]</sup>.

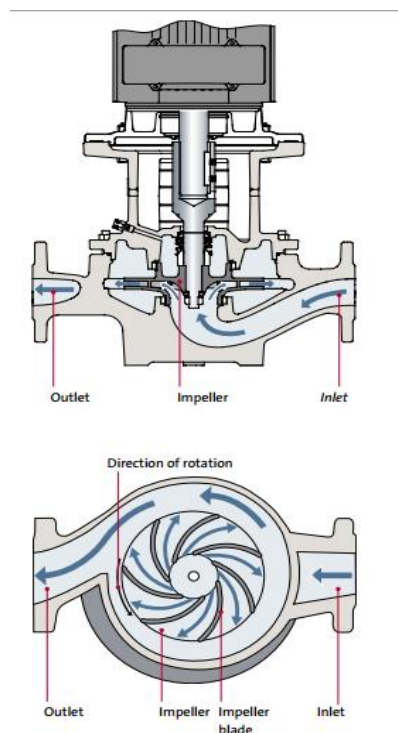


Figure 1-7: Fluid path through the centrifugal pump.





**Elmer** is an open source solver for 3D design; he is giving a numerical solver and pre/post-processing utilities for the solution of mechanics problems, including computational fluid dynamics (CFD).

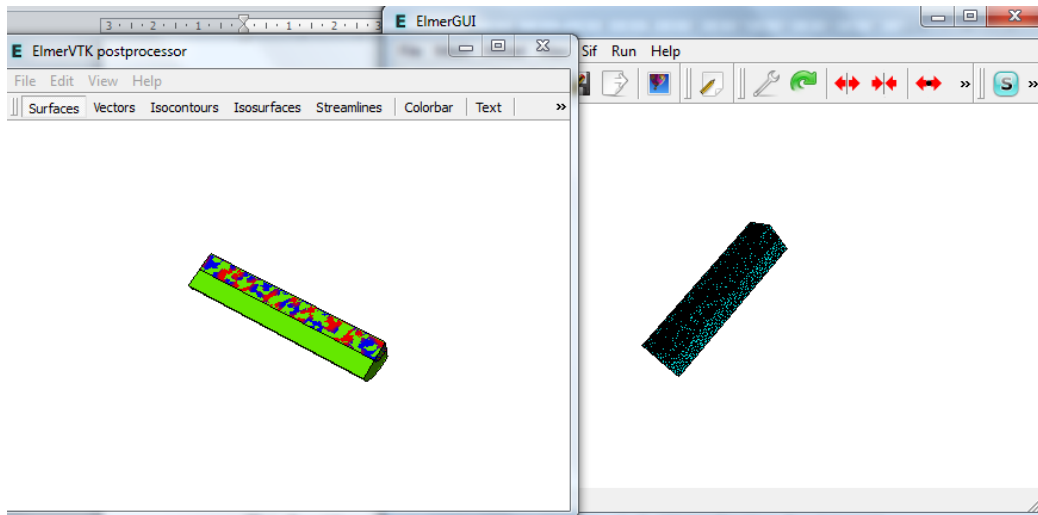


Figure 1-10: Elmer windows for a quarter pipe

**The ParaView** is an open source multiple-platform application for interactive, scientific visualization. It has a client-server architecture to facilitate remote visualization of datasets, and generates level of detail models to maintain interactive frame rates for large datasets.<sup>[12]</sup>

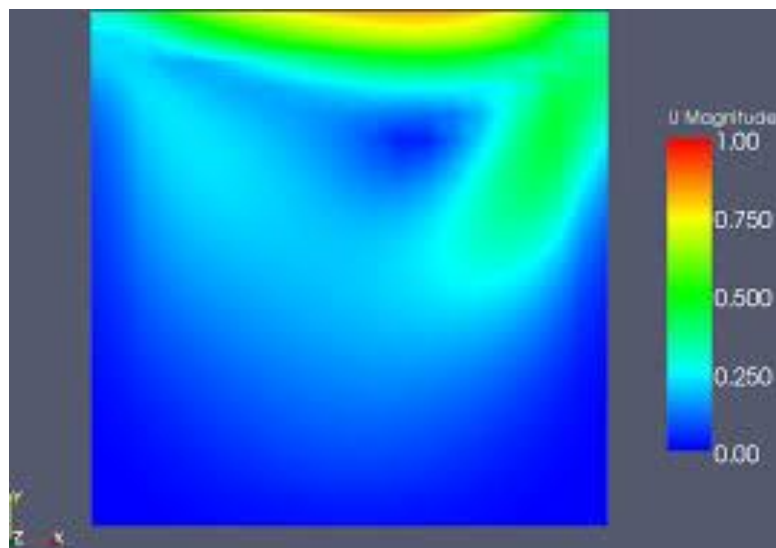


Figure 1-11: Paraview window for the velocity values

We don't compile codes but just use solver software that has already compiled codes. These programs give us results. We use only free software and if possible

open source; all we need is introduce the design after we made it, and the boundary conditions with many treatments concerning the iterative.

The solvers and meshing software used discretization methods, which are described at the following - with finite volume for OpenFOAM and finite element for Elmer and Gmsh.

### 1.6.1. Discretization methods

The solvers in use can have three methods for solving which: finite element method, finite different method, and finite volume method.

Finite element method is the discretization in order to obtain matrix which change the equation of physical quantity to a linear equations when solve is more easy.

Finite difference method is the discretization in order to transform the differential equation to different linear equations then a matrix.

Finite volume method is the discretization in order to discretization of the integral forms of the conservation equations by discretization of each term except<sup>[13]</sup>.

## 1.7. What are the Navier-Stokes Equations?

The equations of viscous, incompressible fluid flow, known as the Navier–Stokes (N.–S.) equations after the Frenchman (Claude Louis Marie Henri Navier) and Englishman (George Gabriel Stokes) who proposed them in the early to mid-19th Century, can be expressed as

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \mu \Delta \mathbf{u} + \mathbf{F}_B \quad (1.1a)$$

$$\nabla \cdot \mathbf{u} = 0, \quad (1.1b)$$

We remind the reader that the first of these equations (which is a threecomponent vector equation) is just Newton’s second law of motion applied to a fluid parcel—the left-hand side is mass (per unit volume) times acceleration, while the right-hand side is the sum of forces acting on the fluid element. Equation (1.1b) is simply

conservation of mass in the context of constant-density flow. In the sequel we will provide alternative forms of these basic equations<sup>[14]</sup>.

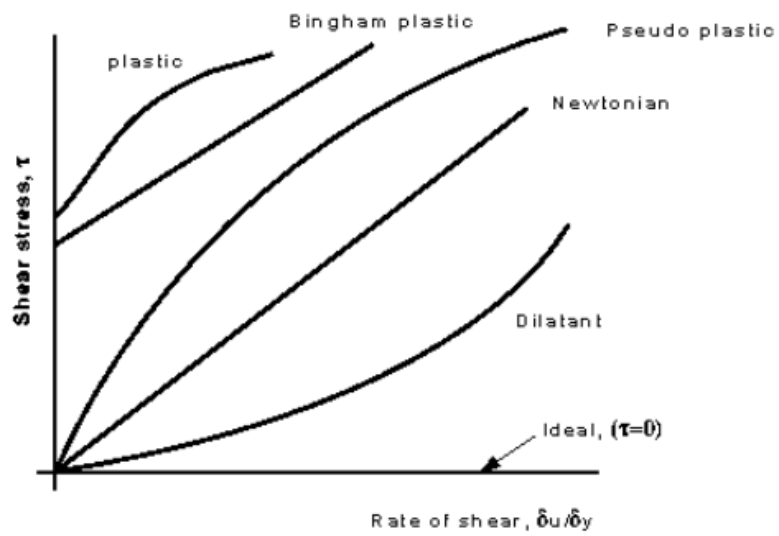
### **1.7.1. Incompressible fluid flow**

A fluid flow is said to be incompressible if volumes of blobs of fluid do not change. So we have the general law: A fluid flow is incompressible if and only if  $\text{div } \mathbf{u} \equiv 0$ .

In fact, we have that incompressibility holds if and only if  $\int_{V(t)} \text{div } \mathbf{u} = 0$  for all  $V(t)$ . Since  $\mathbf{u}$  is a smooth function and  $V$  can be any blob, this implies the assertion<sup>[15]</sup>.

### **1.7.2. Newtonian and Non-Newtonian fluid flow**

Even among fluids which are accepted as fluids there can be wide differences in behaviour under stress. Fluids obeying Newton's law where the value of  $\mu$  is constant are known as Newtonian fluids. If  $\mu$  is constant the shear stress is linearly dependent on velocity gradient. This is true for most common fluids. Fluids in which the value of  $\mu$  is not constant are known as non-Newtonian fluids. There are several categories of these, and they are outlined briefly below. These categories are based on the relationship between shear stress and the velocity gradient (rate of shear strain) in the fluid. These relationships can be seen in the graph below figure 1-12 for several categories<sup>[16]</sup>.



Shear stress vs. Rate of shear strain  $\delta u/\delta y$

Figure 1-12: Shear stress vs Rate of shear strain for several categories

In our report we study the incompressible Newtonian fluid flow.

## 2. Chapter II: Contribution

### 2.1. Solve incineration design

Our study is to find out the pressure and the velocity values for path of water from the pump into the water tank of an incineration power plant, so we should insert the design of this part of the power plant, which is a freeCAD design, into a solver in order to run program and obtain results.

#### 2.1.1. OpenFOAM solver

First we used OpenFOAM (finite volume method solver) as solver. We tried many methods (use command, change format files...) to input the FreeCAD design to the OpenFOAM solver but we didn't manage it. So we changed the solver. But one of these methods is the use of Gmsh which can mesh the design; we use Gmsh despite we changed the solver because it can mesh a big design (water tank 6 meters length and a pipe with 12 meters length) like ours.

#### 2.1.2. Using Gmsh

Using OpenFoam didn't resolve our study, so we used new meshing software - Gmsh (discretization is implemented by finite element method) which we used afterwards in the study. The first way to solve:

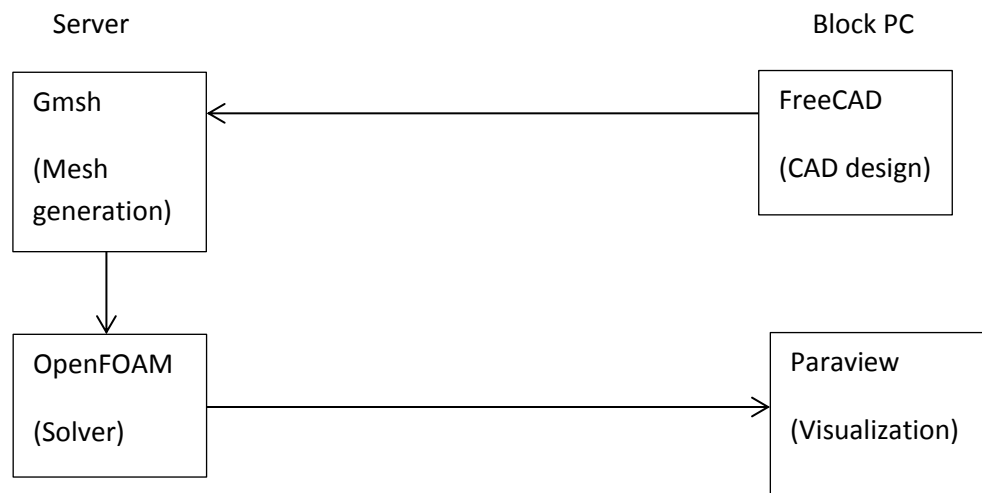


Figure 2-1: Chain of tools 1 using OpenFOAM

The design of incineration power plant in FreeCAD, from<sup>18</sup>:

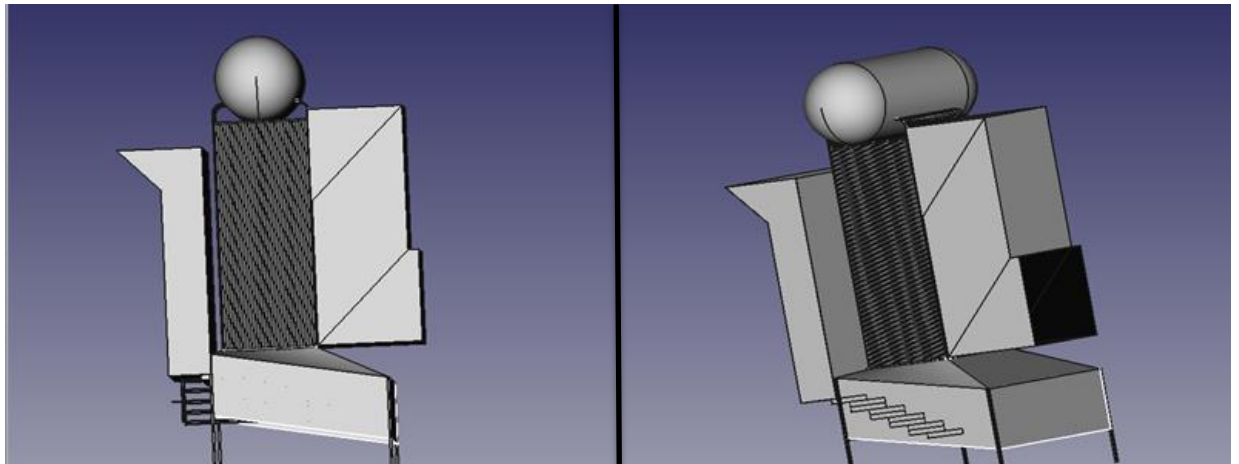


Figure 2-2: FreeCAD incineration power plant design

We obtain meshing design using Gmsh figure 2-3:

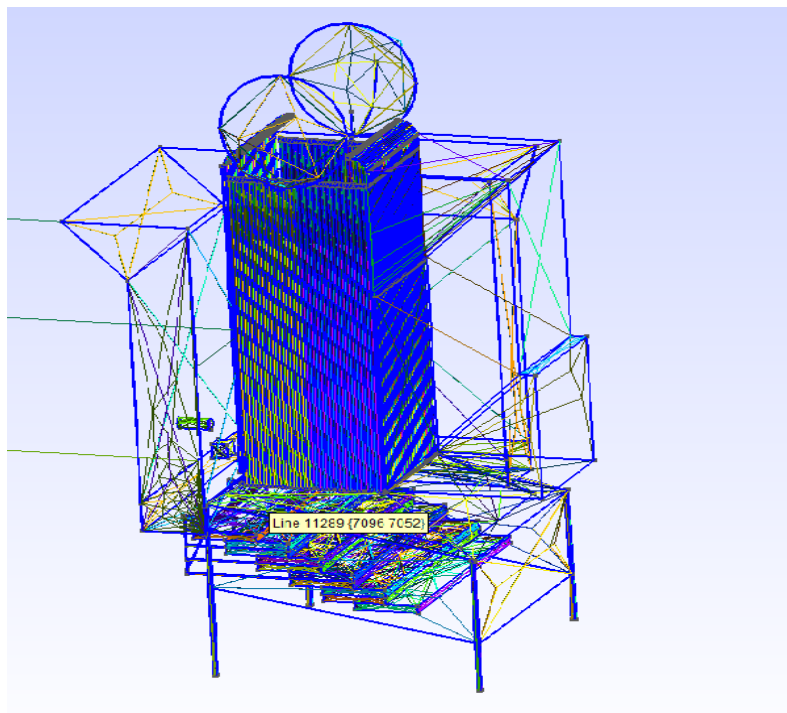


Figure 2-3: Meshing incineration power plant design

To input the CAD design into Gmsh we have to save FreeCAD design in STEP exchange format (e.g. file.stp).

We have to say that we tried to input the meshing design into OpenFOAM in many ways (using command, change format...), in windows and Linux, but we didn't succeed. For this reason we changed the solver and tried another chain of tools.

### 2.1.3. Elmer solver

The Elmer solver implements the finite element method.

The new way to solve "all in server":

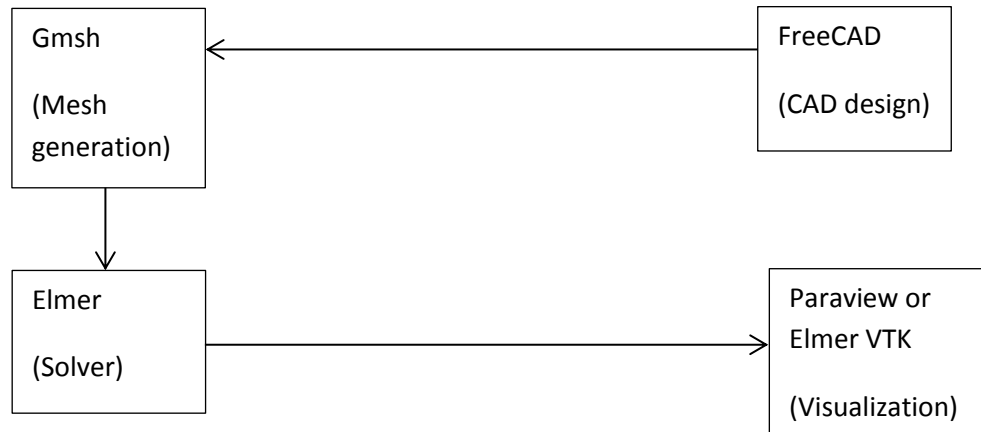


Figure 2-4: Chain of tools 2 using Elmer

Elmer can read the design of pipe (example: quarter pipe in figure 2-5). Then the following is done:



Figure 2-5: Design of a quarter pipe introduce in Elmer

- Design of pipe with meshing on gmsh and saving in file.msh format
- Specify the Initial conditions
- Choose the **Navier-Stokes equation**
- Specify the material used (water for the internal face, and steel (stainless) for the external face)
- Define the boundaries in the design before introducing the condition of each boundary



- Select **run start solver**
- Then select start **ElmerPost** (Paraview) or **ElmerVTK**.

Before we move to the results, we should know how we obtain the initial conditions:

- We need a pump which generates 40 bars ( $40 \cdot 10^5$  Pa), and we can find conditions in Annex 4.
- So the pressure is  $40 \cdot 10^5$  Pa, the flow rate is  $450 \text{ m}^3/\text{h} = 0.125 \text{ m}^3/\text{s}$ .
- We use water, so density is  $1000 \text{ kg}/\text{m}^3 = \text{mass}/\text{volume}$ ; so  $\frac{\text{mass}/\text{s}}{\text{volume}/\text{s}}$ ; we can deduct that the volume flow Q equal  $0.125 \text{ m}^3/\text{s}$ .
- $Q = \frac{\text{volume}}{\text{time}} = \frac{\text{section} \cdot \text{displacement}}{\text{time}} = \text{section} \cdot \text{velocity}$ ; with  $\text{section} = \pi \cdot \text{radius}^2 = 0.003 \text{ m}^2$ .
- So velocity is 41.67 m/s.
- We observed that the flow is in 2 dimensions in each path, and we assume that the values in both velocity directions are equal:
  - ◆ When we have  $v_x$  and  $v_y$  for example,  $v_x^2 + v_y^2 = v^2$  so  $v_x = v_y = 29.47 \text{ m/s}$ , and  $v_z = 0$ .
  - ◆ And so on.

And before we show results we have to know some notes to illustrate in the figure bellow:

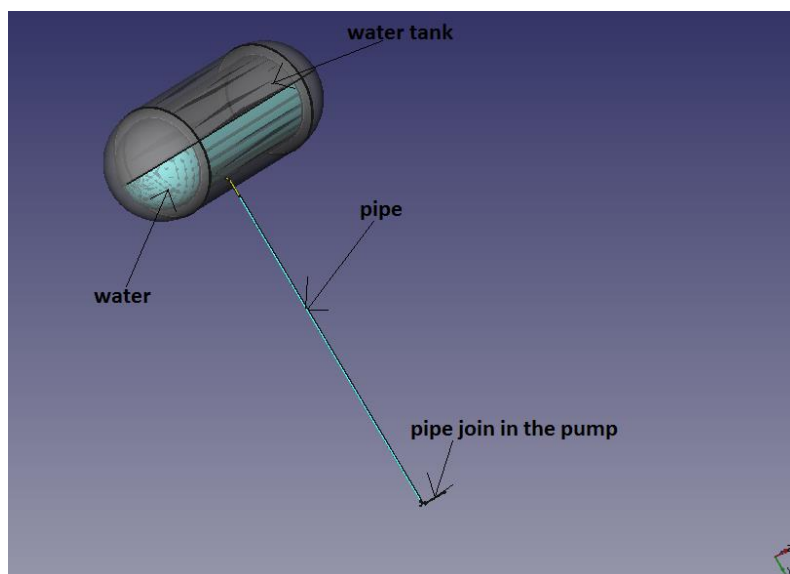


Figure 2-6: Noted design

Where in the figure 2-6 the pipe (height=12m, radius=0.38m) is connected to a Water tank (height=6m, radius=2m) to supply it of water; than the pipe connect with another pipe which connect it to the pump that pumps water into the water tank.

### 2.1.4. Using Elmer

First we define setup (see Figure 2-7) for each part which we study:

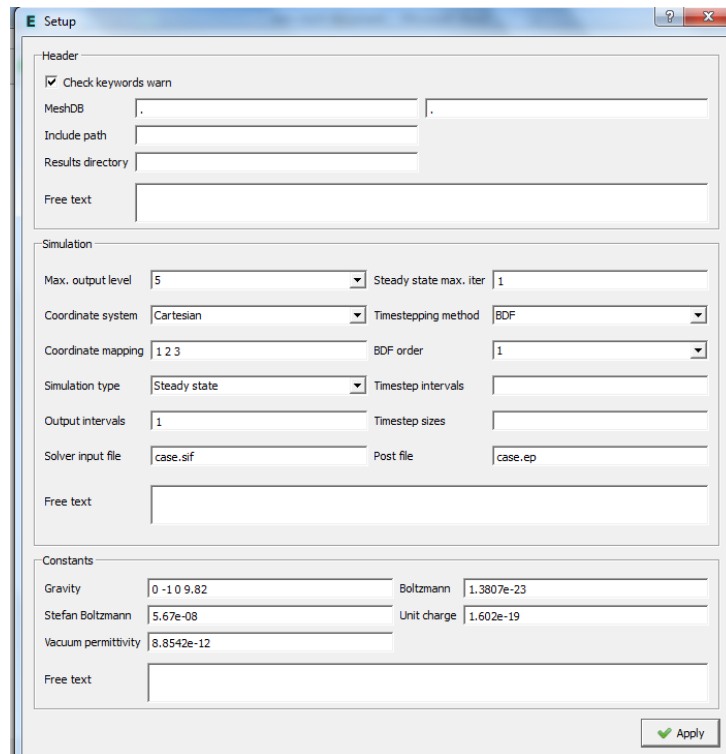


Figure 2-7: Setup for each part (the same)

Then we define the equation as seen in Figure 2-8 (Navier-Stokes equation in our case):

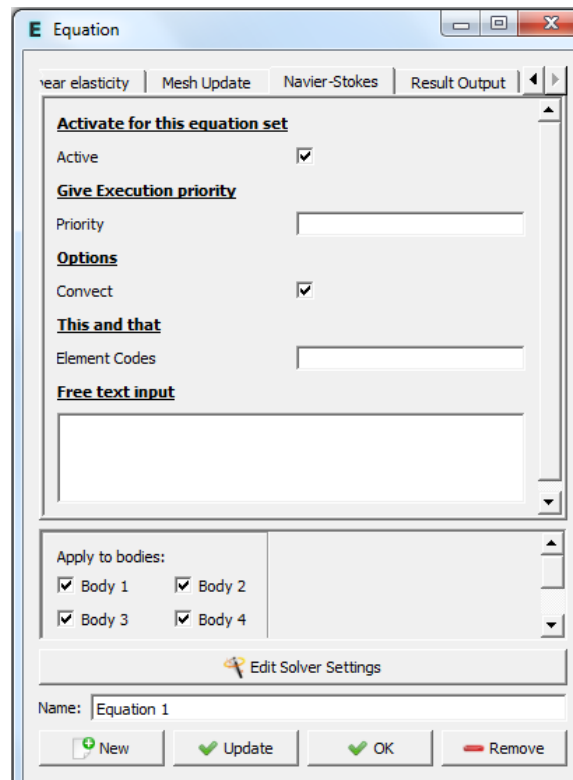


Figure 2-8: Choice of Navier-Stokes equation

We have to say that in Edit Solver Settings we define the solve properties; e.g. number maximal iterations (see Figure 2-9).

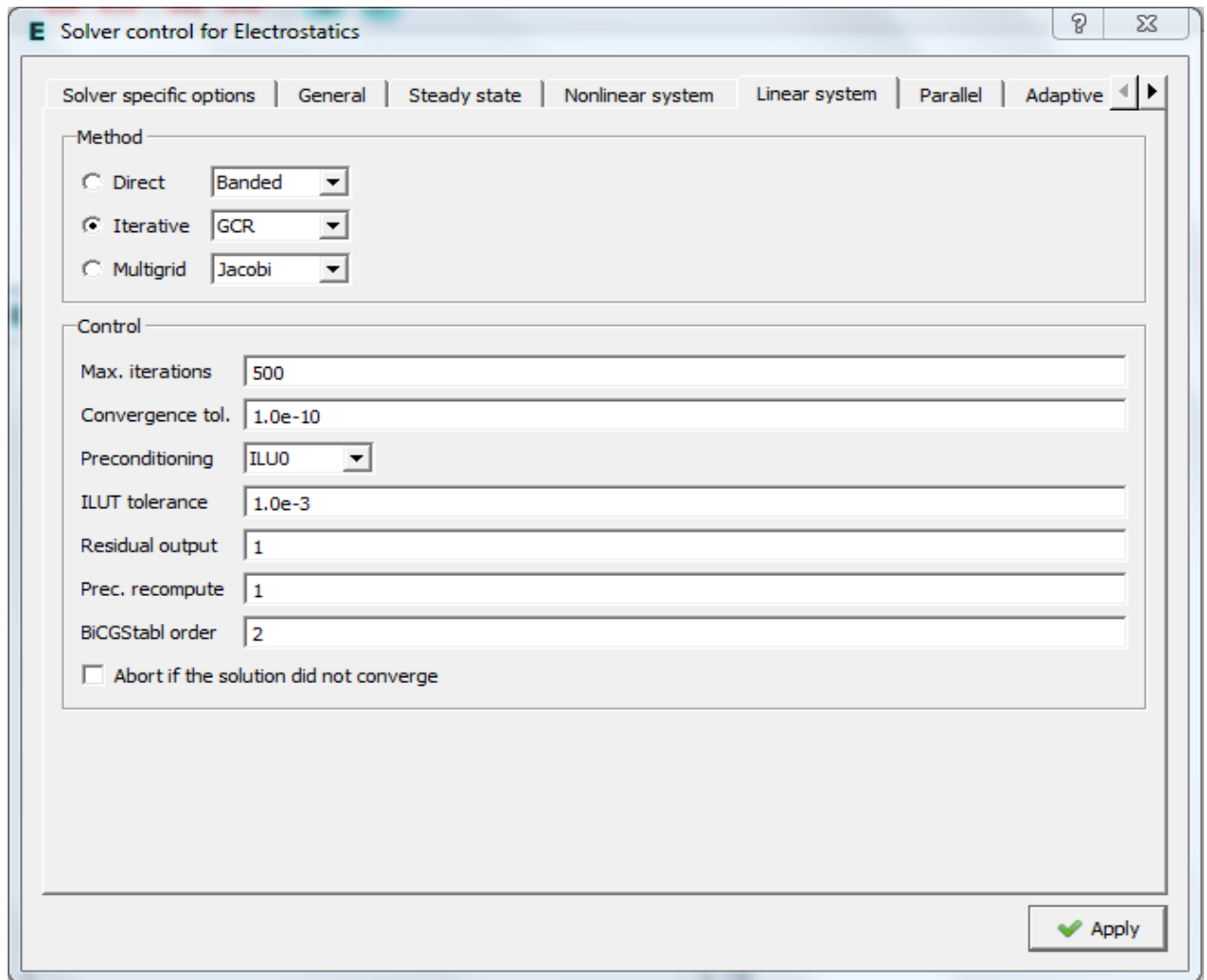


Figure 2-9: iteration conditions in Elmer

Thirdly we introduce the properties of the used material (Figure 2-10):

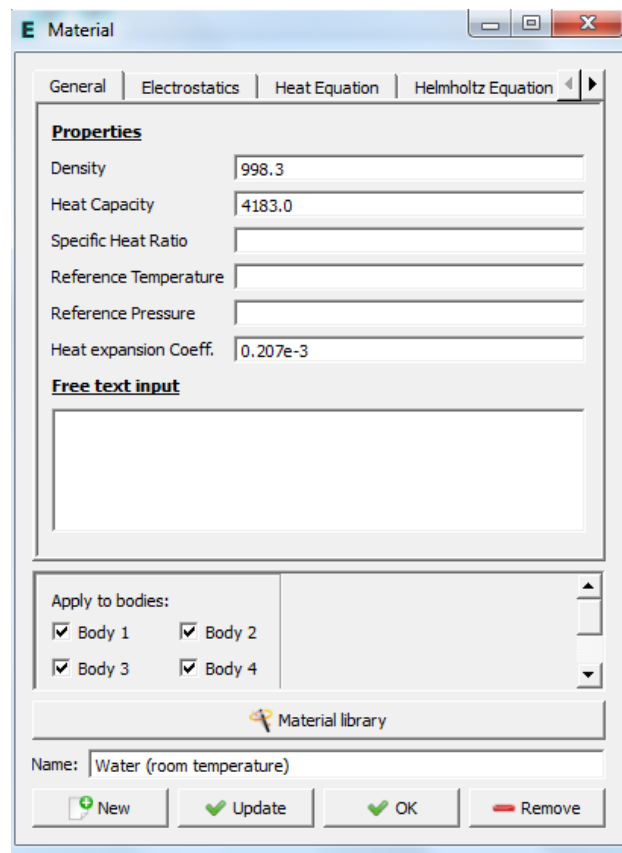


Figure 2-10: Material used for study

But we can choose an existing material like water, as in our case, when the properties are already defined.

Fourthly we introduce the initial conditions (see Figure 2-11) - velocity in m/s and pressure in Pa:

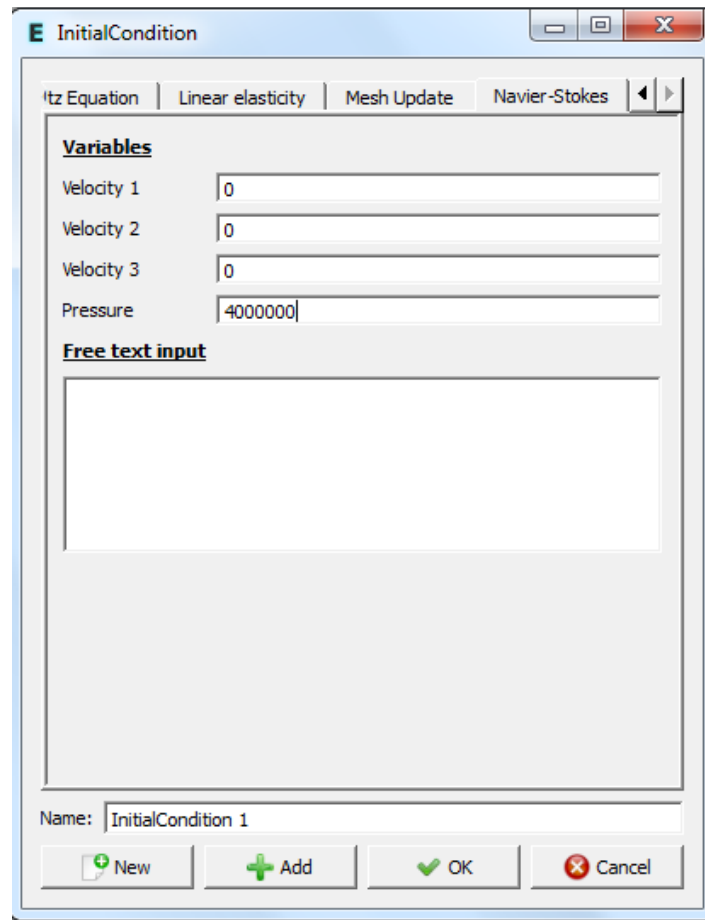
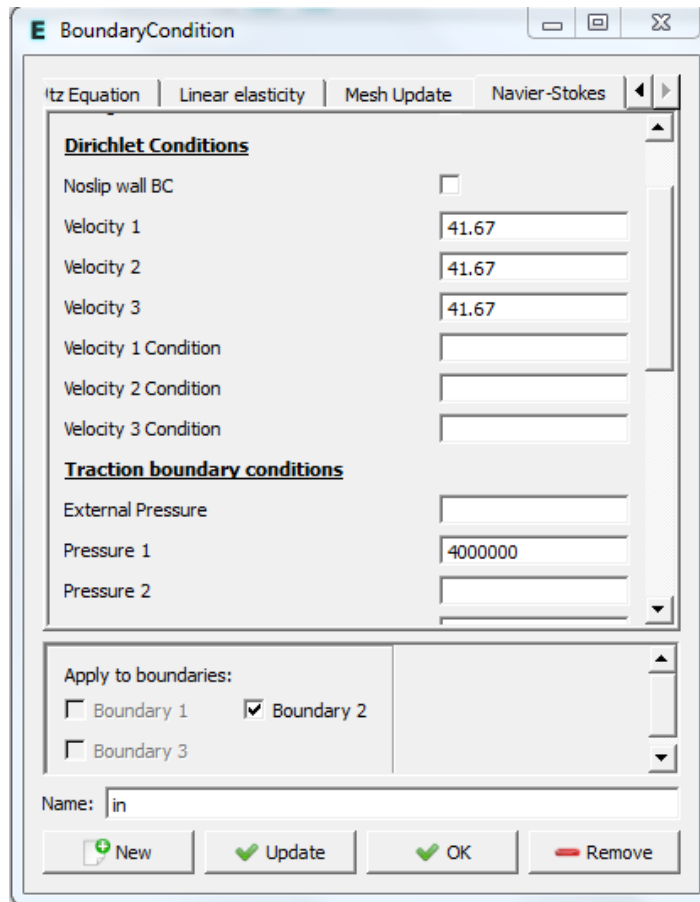


Figure 2-11: Initial conditions introduce in Elmer

We propose that the velocity is beginning by 0 and the pressure  $4 \cdot 10^5$  Pa.

Finally we introduce the boundary conditions that based in the conditions of turbine used.

Boundary conditions divide in two part; inlet boundary conditions where we introduce the turbine conditions figure 2-12:



**Figure 2-12: Inlet boudary conditions given by the turbine**

And outlet boundary conditions where the axes which not affect the flow must not be considered; so it differs from one part to another; we considered each part separately.

First we take a 1 direction fluid flow, fluid flow in the pipe figure 2-13:

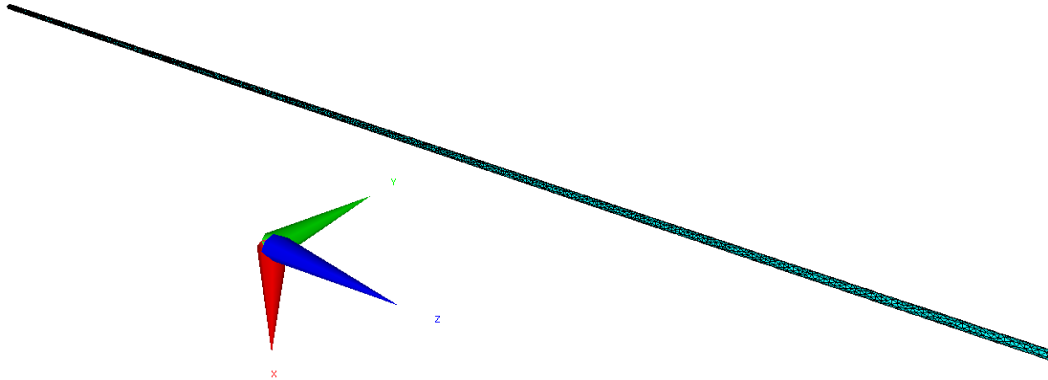


Figure 2-13: Meshed pipe viewing in Elmer

The outlet condition figure 2-14:

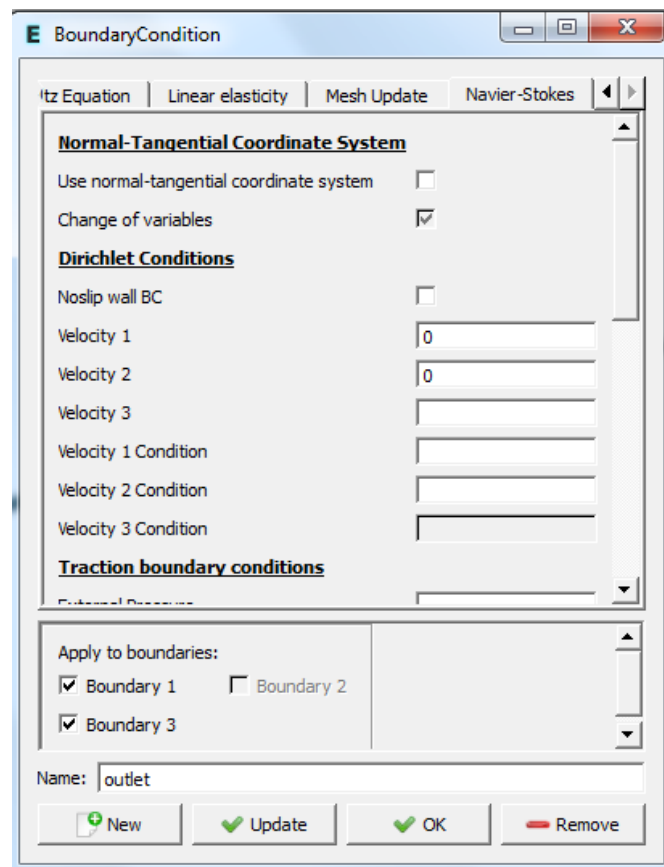


Figure 2-14: Pipe outlet boundary condition

We have to enter the x and y components, and leave the z component empty where the solver gives us the solution.



Taking now a 2 directions fluid flow, fluid flow in the corner figure 2-15:

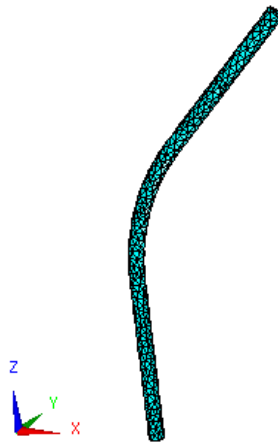


Figure 2-15: Meshed corner viewing in Elmer

We see that the water is flowing following x and z axes. The outlet condition of corner figure 2-16:

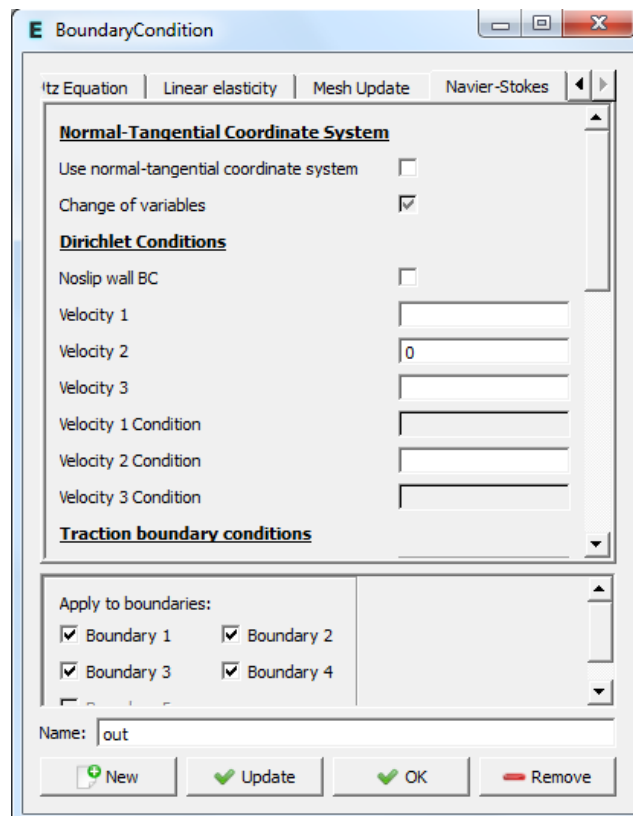
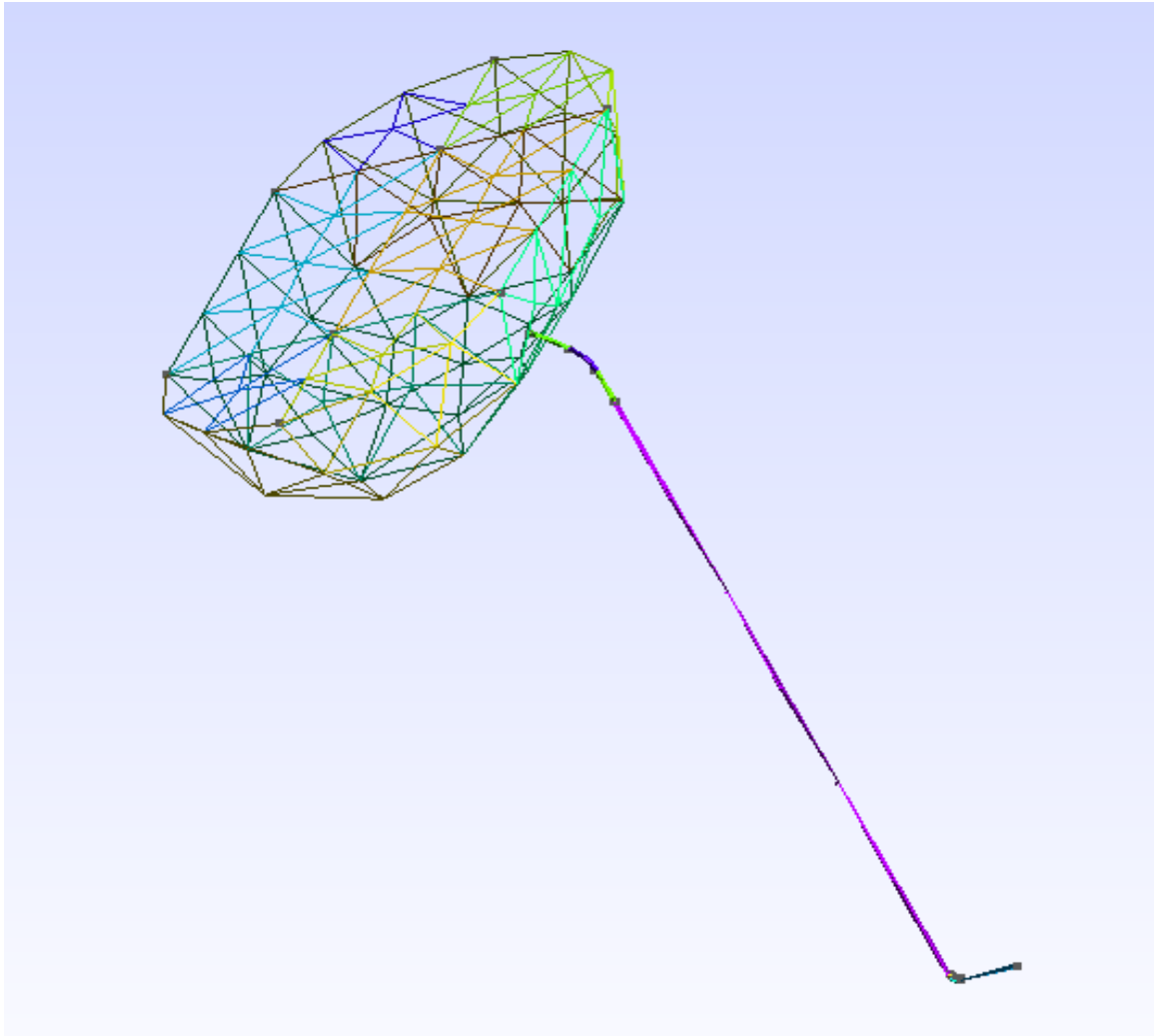


Figure 2-16: Corner outlet boundary condition

The water path from pump to the water tank figure 2-17: Now we have to discretize the water in the pressure tank using gmsh or Elmer, but Elmer is unable to discretize a big design so we use gmsh:



**Figure 2-17: Water path meshing in Gmsh**

We introduce the water design to the Elmer software with the initial conditions, velocity equation, and boundary conditions that we make in the Elmer model:

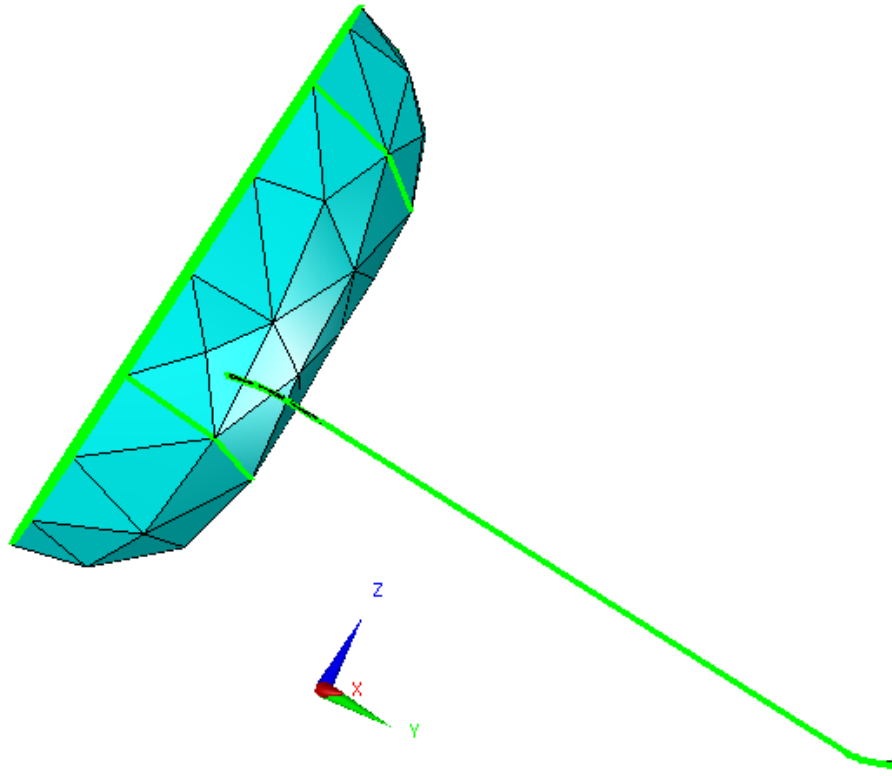


Figure 2-18: Water path illustrated in Elmer

We see that the water is flowing following y and z axes, so we have to introduce the initial conditions following yz plane:

The outlet conditions of the water path are shown in Figure 2-19:

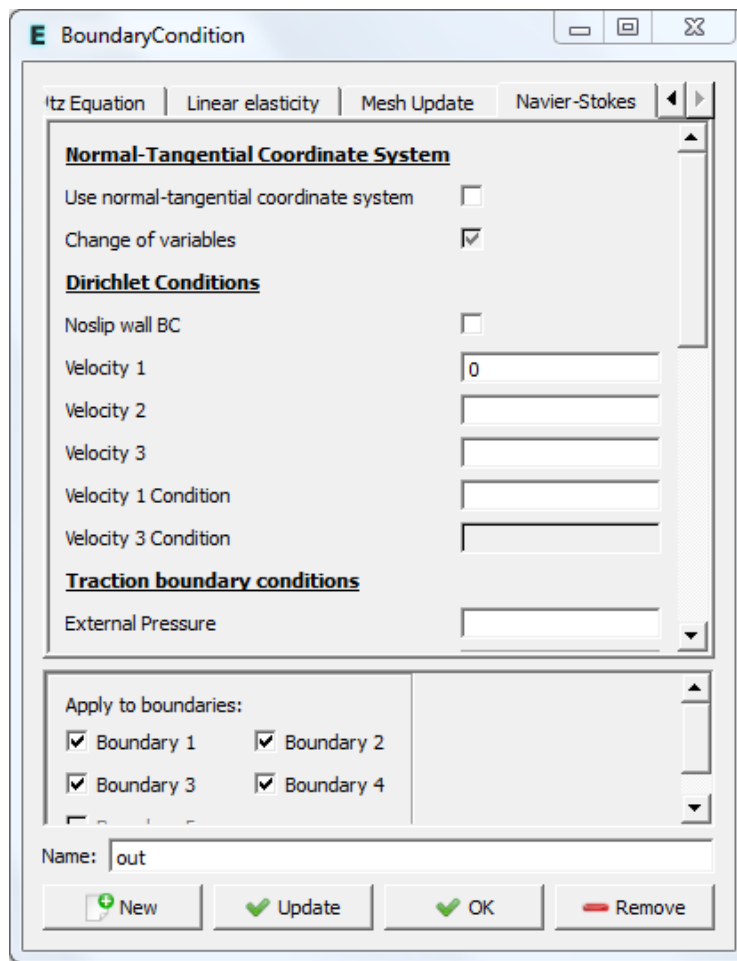


Figure 2-19: Water path outlet boundary condition

But we cannot achieve results on the corner and the complete passage of the water pump to the reservoir because of the problems found in software that is able to study a large design; for this reason we make our study only into the pipe.

### 3. Chapter III: Results and discussion

We obtain the velocity values (start solver) and visualization (ElmerPost) of water in the pipe example that we made:

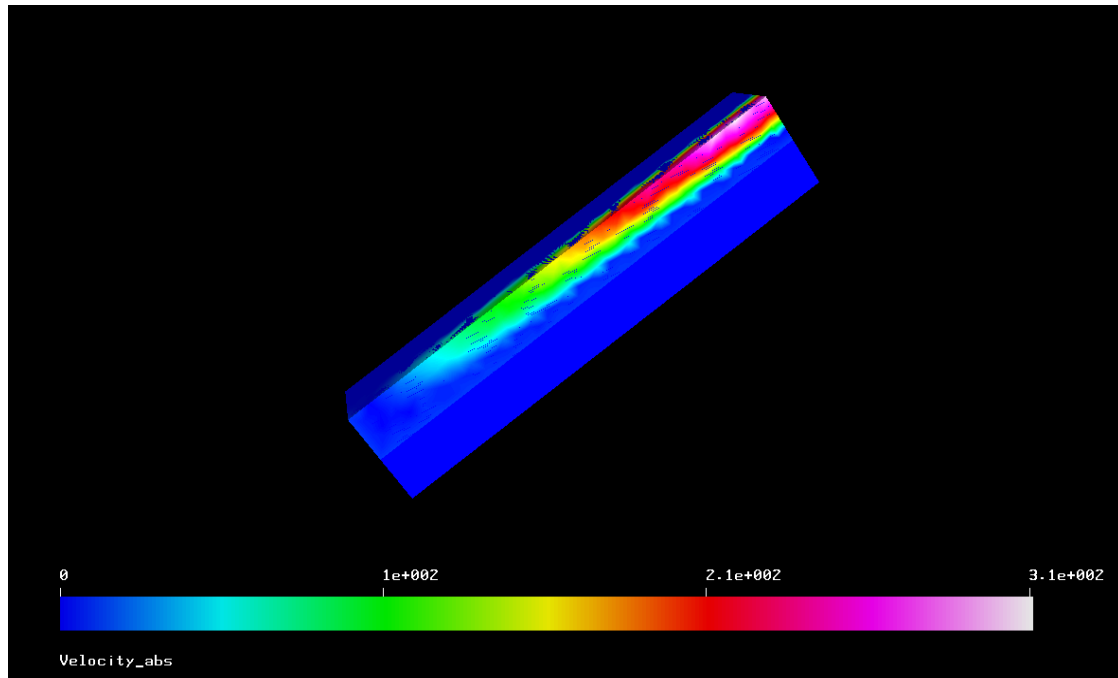


Figure 3-1: Elmer velocity color results (in m/s)

The velocity is changed as follows:

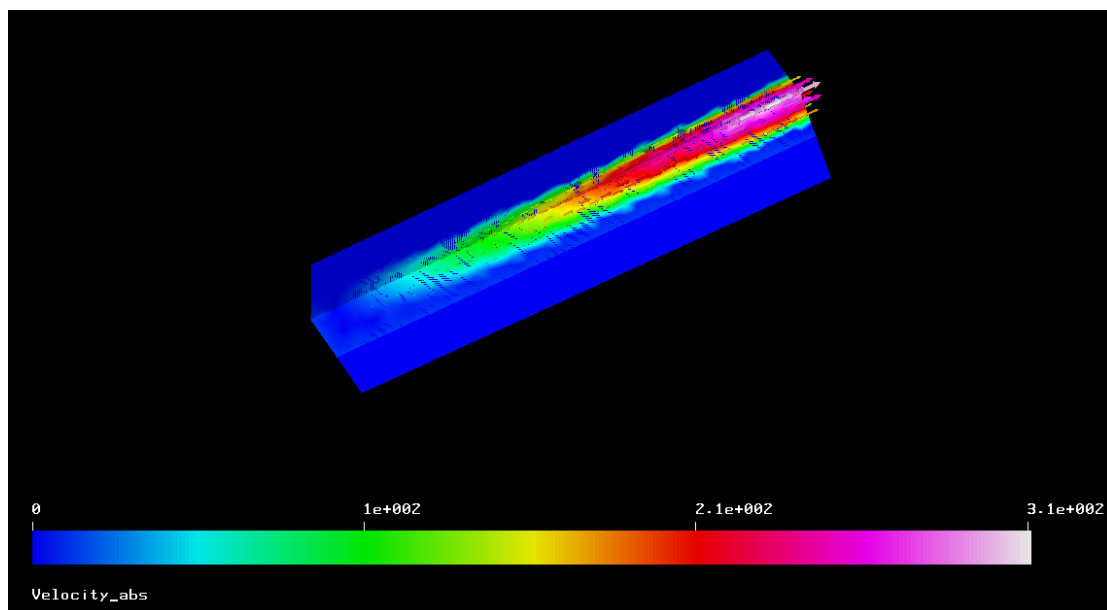


Figure 3-2: Elmer velocity vector results in m/s

And the pressure is following the figure 3-3:

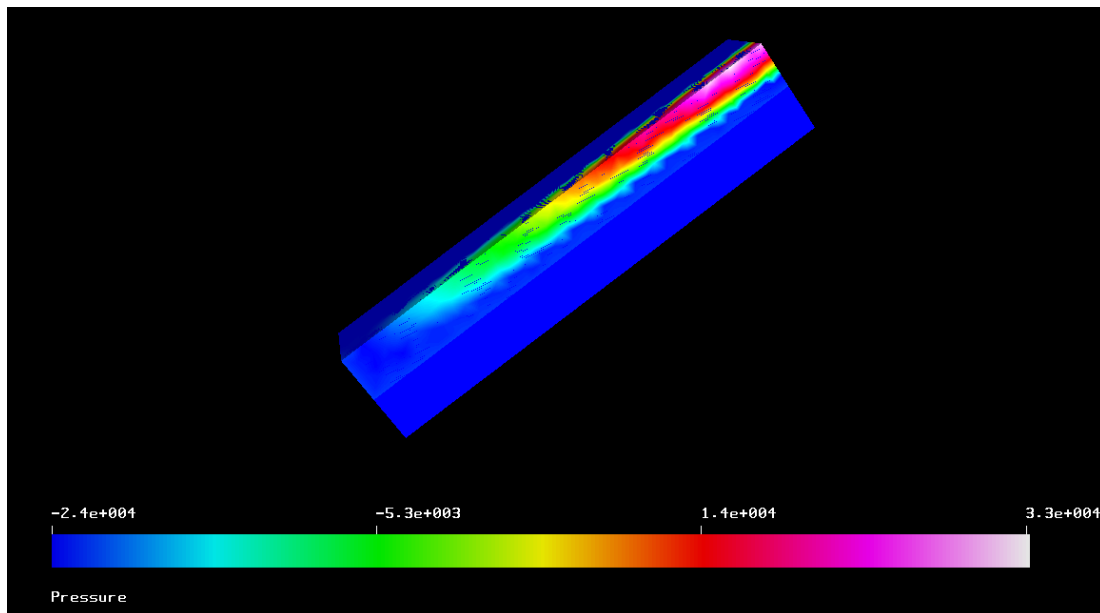
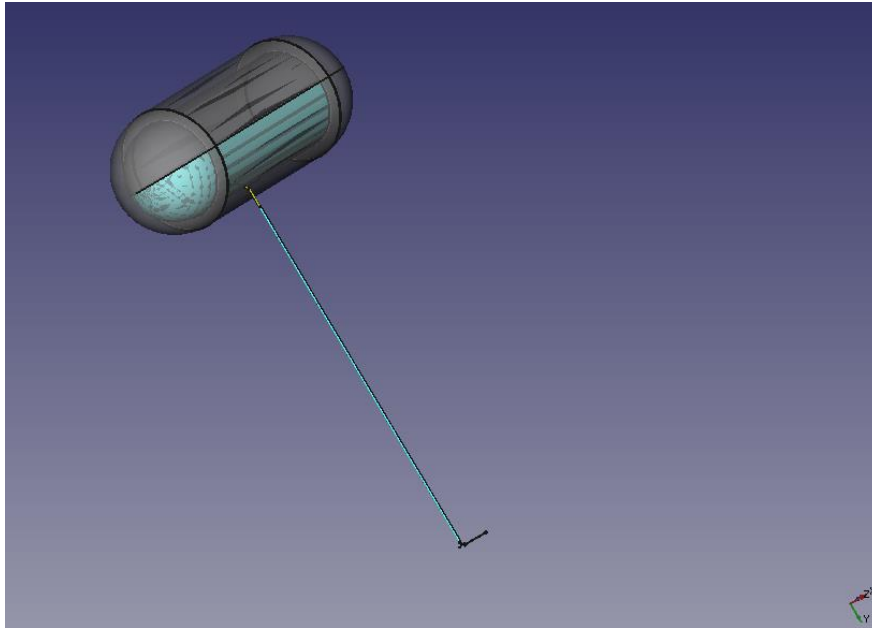


Figure 3-3: Elmer pressure color results in Pa

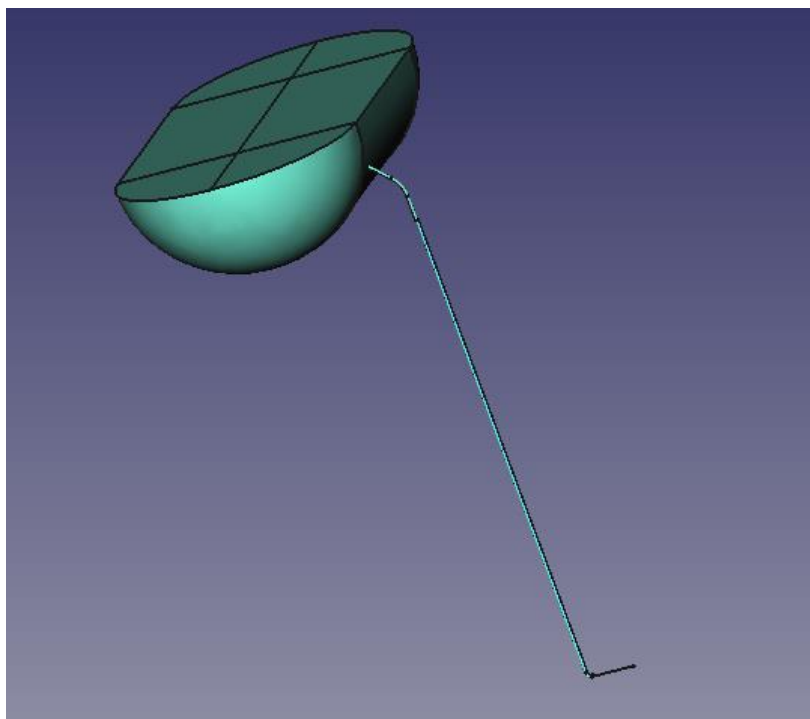
The velocity and pressure values are maximum in the middle than it decreases going towards the ends of the pipe and that because of friction of pipe in the fluid which illustrated in figures 3-1, 3-2, and 3-3.

Now we introduce the incineration power plant design (pump-water tank) to Elmer software in the following figure 3-4:



**Figure 3-4: The studied design**

The materialization of water makes us introduce the following design into Elmer:



**Figure 3-5: Water path**

Our study is difficult in a dual core personal computer; so we do the studies in a quadcore server networked with the personal computer. We make study in the server and move the result (files and figures) to the personal computer. After we run the program with the finite element method, we obtain some files shown below:

Name	Date modified	Type	Size
case.ep	18/8/2015 11:46 AM	EP File	1,564 KB
case.sif	20/8/2015 11:51 PM	SIF File	3 KB
egproject	17/8/2015 11:02 AM	XML File	96 KB
ELMERSOLVER_STARTINFO	20/8/2015 11:51 PM	File	1 KB
mesh.boundary	18/8/2015 11:44 AM	BOUNDARY File	322 KB
mesh.elements	18/8/2015 11:44 AM	ELEMENTS File	431 KB
mesh.header	18/8/2015 11:44 AM	HEADER File	1 KB
mesh.nodes	18/8/2015 11:44 AM	NODES File	161 KB
netgen.prof	20/8/2015 11:57 PM	PROF File	1 KB
water.FCStd	17/8/2015 11:02 AM	FCSTD File	11 KB
water.msh	17/8/2015 11:02 AM	MSH File	1,028 KB
water.stp	17/8/2015 11:02 AM	STP File	76 KB

**Figure 3-6: Result files in Elmer**

Case.ep is the file that contains the velocity and pressure values.

Case.sif is the file that contains the conditions introduced.

Mesh.boundary is the file that contains number of boundary elements, number of elements belongs to the boundaries, the elements surround the boundary, type of codes of the elements, and the nodes of elements.

Mesh.elements is the file that contains identification of the elements, body's material of this element, type of code, nodes of element.

Mesh.header is the file that contains number of nodes, number of elements, and number of boundary elements.

Mesh.node is the file that contains number of nodes, index of parallel execution nodes, and the node coordinates.

Water.FCStd is the FreeCAD design file.

Water.stp is the gmsh design file.

And water.msh is the Elmer meshing file.<sup>[10]</sup>



We remember the coordinate system x, y and z:

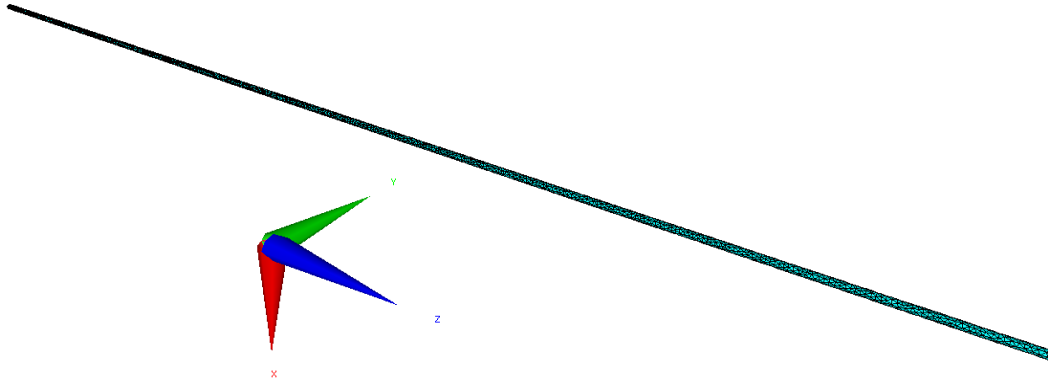


Figure 3-7: Meshed pipe viewing in Elmer

The color of variable value of velocity and pressure illustration:

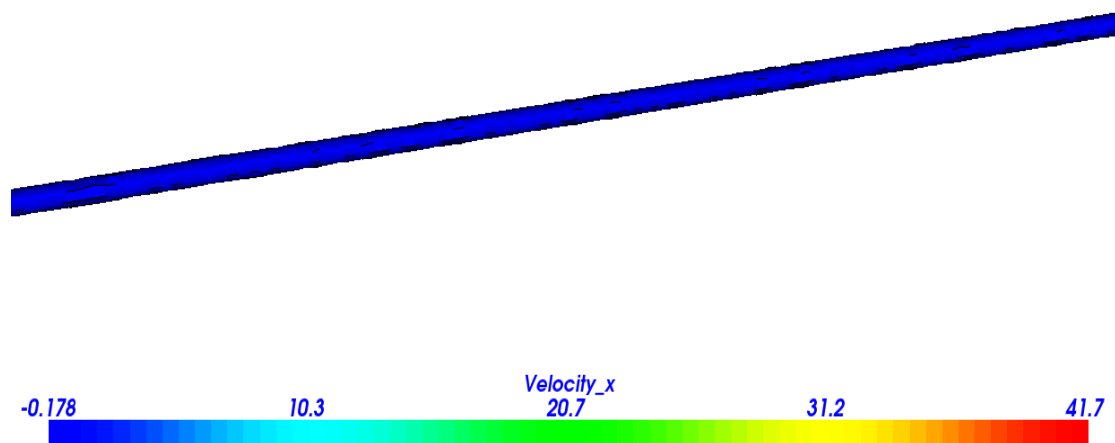


Figure 3-8: Velocity variation x-component in m/s

This figure (figure 3-8) of velocity values x component shows that the blue color identify the minimum value (or the value which the velocity is in opposite direction) of velocity; then the value increases to reach the maximum in the red color.

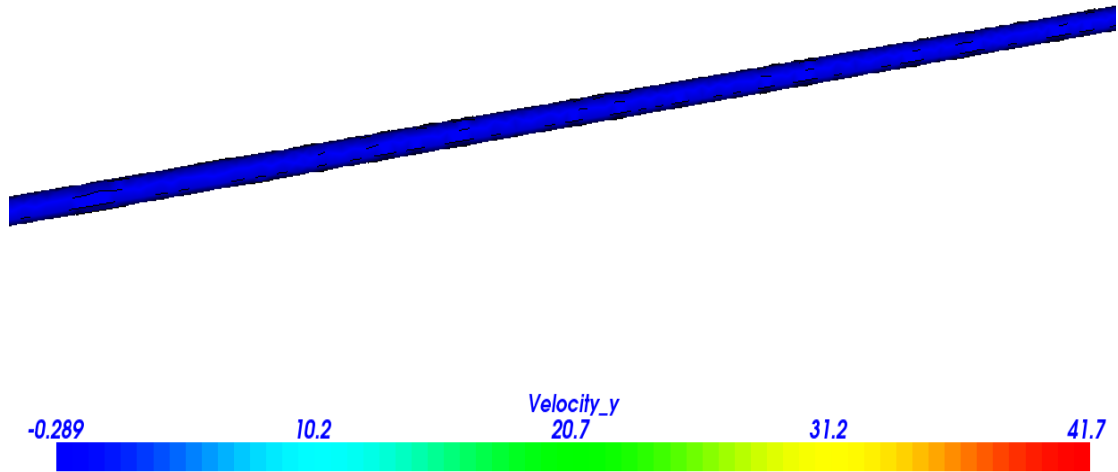


Figure 3-9: Velocity variation y-component in m/s

In the figure 3-9 we see the same result concerning the arrangement of color but for the y-component of velocity.

And the same for the y-component in the figure 3-10 but we deduct that the velocity<sub>z</sub> is constant because the flow is following 2-directions x and z.

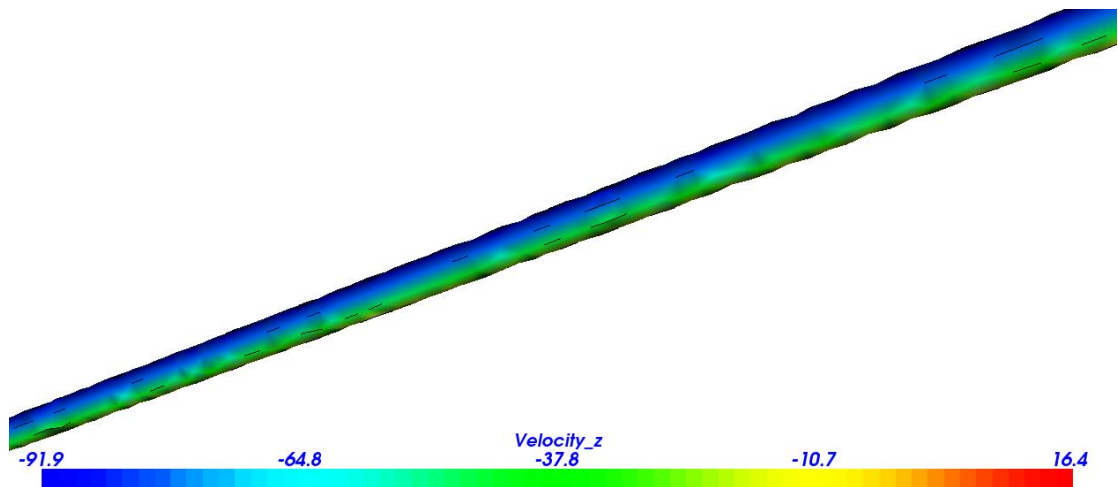


Figure 3-10: Velocity variation z-component in m/s

So we should interest to the position of green, yellow, and red color for velocity and pressure to know where study is should be fixed.

Water flowing into the pipe following the z direction so the velocity in the x and y direction are null and it varies along the axis z.

Now the vector of velocity into the pipe is showing in figure 3-11:

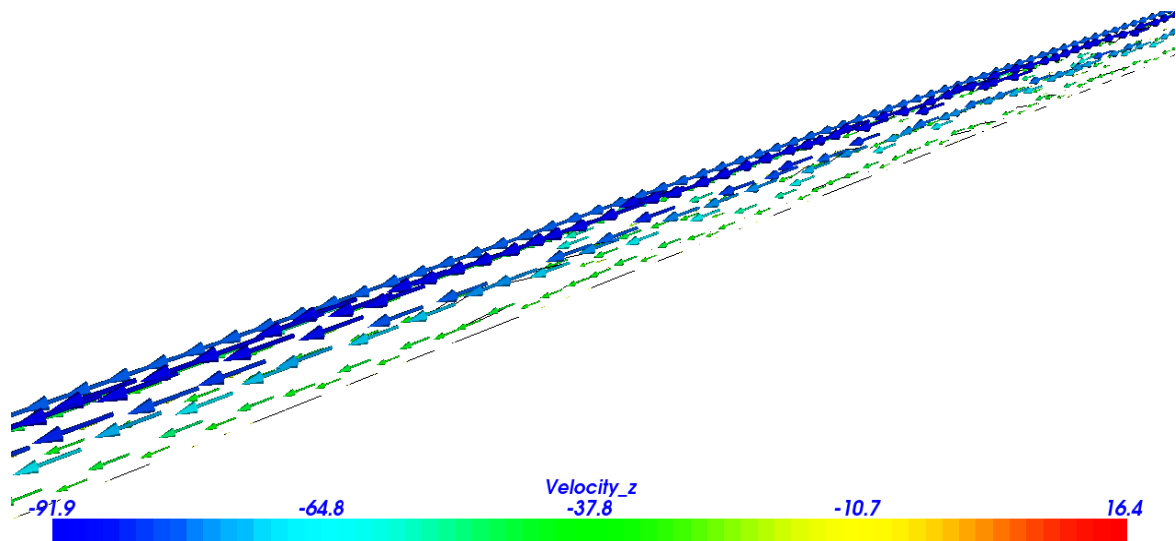


Figure 3-11: Velocity vector into the pipe in m/s

Here in figure 3-12, we consider the variation of the pressure:

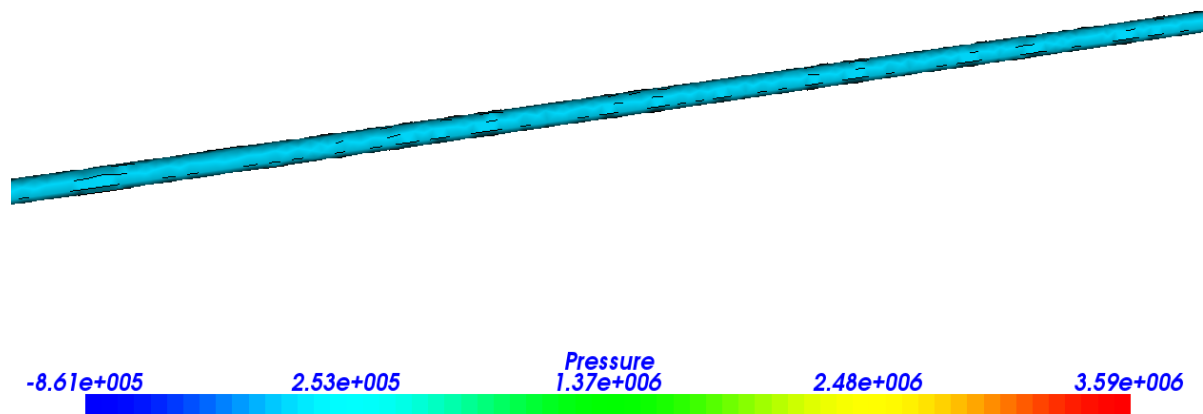


Figure 3-12: Pressure variation in Pascal

This figure (figure 3-12) of pressure values shows that the blue color identify the minimum value of pressure; then the value increases to reach the maximum in the red color.

We deduct that the blue color indicated that the pressure is varied between  $2.53 \cdot 10^5$  and  $1.37 \cdot 10^6$  Pascal.

In the figures 3-13 and 3-14 we show the input and the output of water pressure in the pipe respectively:

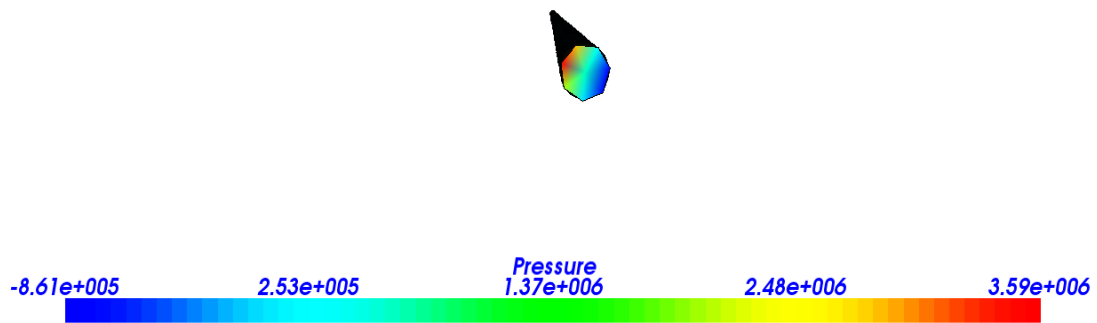


Figure 3-13: Input water pressure into the pipe in Pascal

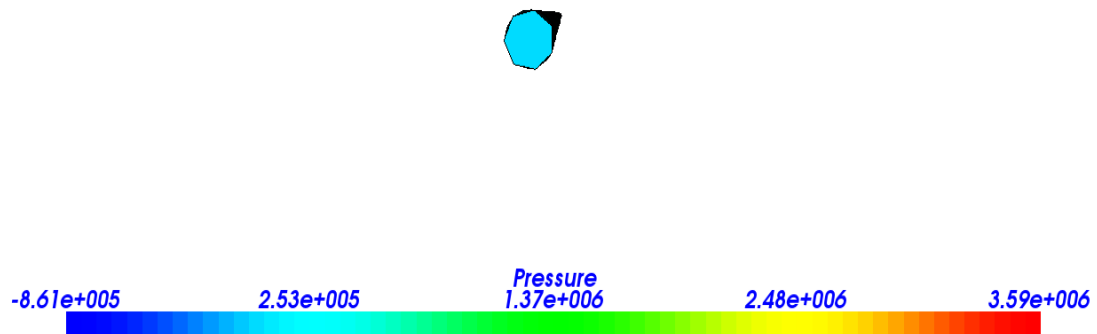


Figure 3-14: Output water pressure in Pascal

We can deduct that the pressure is reduced and this is the cause of the passage of the water pump to the reservoir.

## **4. Conclusion and future work**

Our study was to know the fluid flow pressure and velocity of a part of an incineration power plant using finite element method. We investigated the path of water and evaluated the velocity and pressure for the pipe which feeds water from the pump to the pressure tank under conditions which we introduced. We used FreeCAD to draw the design, then Gmsh for meshing and at the end Elmer for solving (the fluid dynamic equations) and visualizing (the resulting flow).

The velocity and pressure values that we obtain are necessary for knowing the conditions of material used in the design, and for improvement of the incineration power plant by adding a valve or vary the thickness.

It shall also be remarked that OpenFOAM can solve the CFD problem but our study was limited in time, so the OpenFOAM study should be continued and the correct command should be found so that we can insert the FreeCAD design (or Gmsh meshing design) into OpenFOAM and solve it to find velocity and pressure using the finite volume method, and visualize using Paraview.

And we have to say that our study concerned the incompressible fluid flow, so it is probably to change the condition of study and add the necessary condition to study the compressible fluid flow for example. So our study is open to use by another students to continue it.

And we have to know that our study is not just useful for the steam path to incineration power plant, it is useful for any domain of fluid mechanic when it is important to discretize the domain and obtain results which can use in industrial applications.

## 5. References

[1] WASTE to ENERGY INTEGRATED SOLUTIONS FOR THE THERMAL TREATMENT OF TOXIC AND NON-TOXIC; WATERLEAU protecting the 4 elements: WASTE Radioweg 18, 3020 Herent (Leuven).

[2] AVFALL SEVERAGE; Swedish Waste Management. © Avfall Sverige AB Literature references.

[3] Roskilde's glowing incinerator powers 65,000 households

[STU ROBARTS](#)

SEPTEMBER 16, 2014

[4] By Heron Kleis, Babcock & Wilcox Vølund and Søren Dalager, Rambøll, 100 YEARS OF WASTE INCINERATION IN DENMARK, From Refuse Destruction Plants to High-technology Energy Works

[5] Trash to cash: Norway leads the way in turning waste into energy, UK cities pay to send rubbish to Norwegian incinerators, but green campaigners warn of dangers

[6] <http://www.wrfound.org.uk/articles/incineration.html>

[7] SAND2002 - 0529 Unlimited Release Printed March 2002 Verification and Validation in Computational Fluid Dynamics1 William L. Oberkampf Validation and Uncertainty Estimation Department Timothy G. Trucano Optimization and Uncertainty Estimation Department Sandia National Laboratories P. O. Box 5800 Albuquerque, New Mexico 87185

[8] Science Direct: Journal of Computational Physics 195 (2004) 414-433, review Fluid dynamics in Group T-3 Los Alamos National Laboratoru (LA-UR-03-3852) Francis H. Harlow Los Alamos National Laboratoru. MS-8216. Los Alamos NM 87544. USA Receiver 12 June, received in revised from 25 August 2003; accepted 3 September 2003.

[9] Introduction to Computational Fluid Dynamics. Instructor: Dmitri Kuzmin Institute of Applied Mathematics University of Dortmund kuzmin@math.uni-dortmund.de <http://www.featflow.de>

[10] <http://www.freecadweb.org>

**[11] 2011-2015 OpenFOAM Foundation**

OPENFOAM is a registered trademark of OpenCFD Ltd.

**[12] Sandia National Labs Achieves Breakthrough Performance Using NVIDIA(R) Technology for Scientific Visualization**

[13] Introduction to Computational Fluid Dynamics (CFD), 3rd edition, John F. Wendt (Editor), A von Karman Institute Book Authors of used part: J. Anderson, R. Grundmann

[14] LECTURES in COMPUTATIONAL FLUID DYNAMICS of INCOMPRESSIBLE FLOW: Mathematics, Algorithms and Implementations J. M. McDonough Departments of Mechanical Engineering and Mathematics University of Kentucky

[15] A Gentle Introduction to the Physics and Mathematics of Incompressible Flow Course Notes, Fall 2000 Paul Fife

[16] <http://www.efm.leeds.ac.uk/CIVE/CIVE1400/PDF/Notes/section1.pdf>

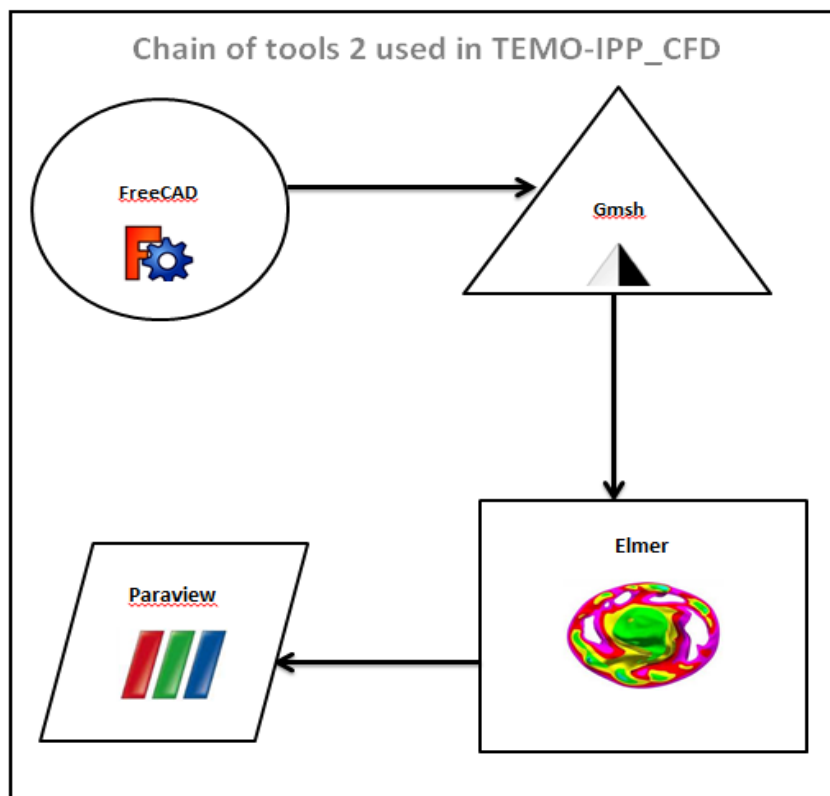
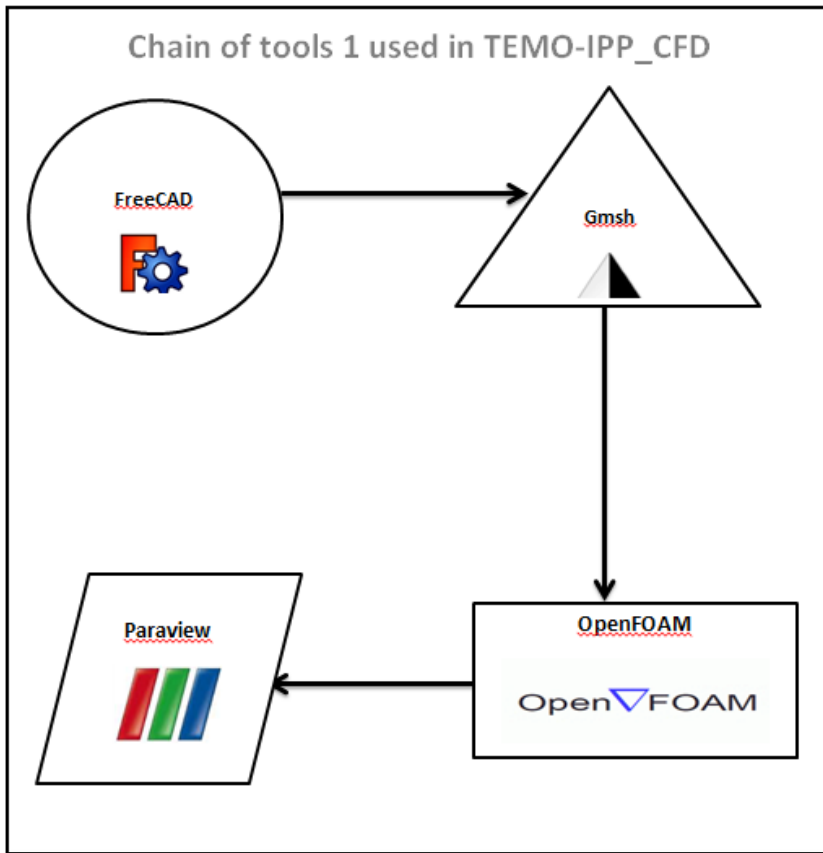
[17] GRUNDFOS RESEARCH AND TECHNOLOGY. The Centrifugal Pump; GRUNDFOS Management A/S

[18] Banan EL Kerdi, Mechanical Analysis of an upscaled version of the Vaporizer (pressure vessel and circulation tubes) of the incineration pilot power plant TEMO-IPP Master Thesis, AECENAR, 2015





### 6.3. Annex 3: Tool chains used for CFD analysis



## 6.4. Annex 4: Offers for 30 MW Steam Turbine and Pump

POWER PLANTS  
OIL REFINERIES  
SALE & RELOCATION

**LOHRMANN**  
INTERNATIONAL

Taunusstr. 5a      Tel. +49 (0) 611-50402-0      www.lohrmann.com  
65183 Wiesbaden/Germany      Fax +49 (0) 611-50402-50      info@lohrmann.com

<b>For Sale:</b>	<b>Pre-owned 30.2 MW Steam Turbine Generator Extraction – Condensing Type</b>
<b>Ref.-No:</b>	<b>STG-29.33</b>

### Brief plant history:

The steam turbine generator was part of a combined cycle power plant commissioned in 1996 and shut down 2012 after approx. 86,562 operating hours.

### Description of major plant components:

#### Turbine

Manufacturer, Type	ABB Turbinen Nürnberg GmbH, VEE 40 Double extraction condensing turbine, Type 4 Stages HP, 9 stages MP and 9 stages LP part
Max. terminal power	30.2 MW
Number of extractions/ bleeders	1 pcs./ 2 pcs.
Live steam parameters	120 bar, 520 °C, 210 t/h
Bleeding 1 normal/max.	36 / 41 bar, 365 / 415 °C, max. 80 t/h
Bleeding 2 normal/max.	6 / 8.5 bar, 170 / 333 °C, max. 75 t/h
Extraction normal/max.	16 / 20.5 bar, 275 / 300 °C, max. 20 t/h
Exhaust normal/max.	0.047 / 0.08 bar, max. 18 °C
Rate max.	10.2 / 55.6 t/h

#### Gear

Parameters      Type spur gear, single helical  
30.2 MW, 1500 rpm (from 5859 rpm)

#### Generator

Parameter      Three-phases  
37500 kVA, 10500 V, 50 Hz  
Rotating speed      1500 rpm

Cooling      TEWAC

#### Condenser

Pressure inside condenser      0.06 / 0.073 bar  
Cool. water parameters      3000 m³/h, 27 °C  
Amount of steam      41 / 56 t/h

### Major auxiliary equipment / Accessories

Start system, lubrication system, hydraulic system, gears, and acoustic enclosure



Further information on request

#### Important Disclaimer:

Although the statements and technical information contained herein are believed to be materially accurate as of the date hereof, no representation or warranty is given as to the accuracy of any of the information provided. This document contains confidential information intended only for the use of the addressee. If you are not the intended recipient of this information then you are hereby notified that any use, dissemination, distribution or reproduction of this message is prohibited.

STG-29.33 Short

Page 1 of 1

Budget Price for STG 29.33: FOB 3.150.000 EUR (Offer from 12 Oct 2015)



**Series SIHImulti MSL, MSM**

horizontal, multi-stage centrifugal pumps according to ISO 5199 / EN 25199 for pressures up to 40/63 bar.

Flow rate: up to 450 m<sup>3</sup>/h

Head: up to 630 m ▶

<http://www.sterlingsihi.com/cms/home/products-services/liquid-pumps/high-pressure-pumps.html>

