TETRAENER

OPTIMAL BALANCING OF DEMAND AND SUPPLY THROUGH RES IN URBAN AREAS

REF. EC: TREN/05/FP6EN7S07.48739/513489

INTEGRATED PROJECT
FP6 Priority 6.1 -- "SUSTAINABLE ENERGY SYSTEMS"

Deliverable	GE WP 1.1 Del 2	METHODOLOGY FOR THE DESIGN OF LAKE AND RIVER WATER UTILISATION SYSTEMS

Due date of deliverable: M18
Actual submission date: 14/01/08
Start date of project:.......30/11/05
Duration:..........................5 years

Organisation name of lead contractor for this deliverable: EIG
Revision: 0

Project co-funded by the European Commission within the Sixth Framework Programme (2002-2006)		
Dissemination Level		
PU	Public	X
PP	Restricted to other programme participants (including the Commission Services)	
RE	Restricted to a group specified by the consortium (including the Commission Services)	
CO	Confidential, only for members of the consortium (including the Commission Services)	

A. Taruffi, P. Haas, J. Richard

1 Table of contents

2 Abstract

This report describes a methodology to design systems utilizing lake and river water for cooling. It corresponds to delivrable 2 of the work package 1.1.

This methodology is used to define the best location of the water intake and outlet. Lake and river models are developed and used to evaluate the potential utilization of lake and river water in district heating and cooling systems. Outlet devices for a fast mixing of the exhaust water with the environment are studied and a conceptual methodology for the design of intake and outlet in lakes and rivers are presented.

The methodology is applied to two cases: The Leman Lake and the Rhone River in the urban part of the Geneva region. These two cases are used to validate the approach based on in field measurements.

3 Introduction

The use of lake and river water for district heating and cooling shall be done with respect to environmental constraints. The direct use or the potential increase in efficiency of the systems shall not move the problem to others concerns. The deep and the location of the outlet shall be selected according to the courantology. An analysis of the exhaust water streams shall be done and the exhaust location selected in a way to insure the sensible regions of the lake or river from the ecological point of view are not affected.

During projects using lake or river water for cooling or heating, some parameters are fixed. These are

- The meteorology
- The courantology
- The bathymetry of the lake or river and geometrical aspects

The engineers can then work on other parameters such :

- The location and deep of the outlet
- The mixing efficiency of the outlet
- The flow volume and temperature of the exhaust (cooling or heating power)

During the TetraEner project we analyzed the possibility to use CFD techniques and field measurements to select the best outlet location. A special technique using a GPS system and a sonar is first used to generate a tridimensional model. This model is then used in a computational fluid dynamics (CFD) code to simulate the flow and the thermal behavior of the lake or the river. The streamlines of the heated effluents and the temperature fields are computed.

These results are then compared to measurements in a way to validate the proposed technique.

4 Objectives

4.1 Definition of the best location for water intake and outlet

If the best location for the inlet can be selected easily and is generally the coldest point in the acceptable vicinity, the location of the exhaust point is more complex. It is defined by the geographical coordinates and the deep.

An analysis of all environmental aspects related to a lake or river shall be done prior to define the location of the inlet and outlet. According to this analysis, a choice can then be made in regards of environmental objectives. The simulation of the exhaust water trajectory is an important tool in the process of selecting the best location. Meteorological scenarios can be simulated and the impact evaluated. This study will give a methodology to define the best geographical coordinates and deep for the inlet and outlet.

4.2 Definition of a conceptual methodology for the design of intakes and outlets

The study presents a conceptual methodology to define intakes and outlets for lakes and rivers. The differences between these two cases are explained and the methodology adapted. The outlet for a lake will not be the same as for a river; due to the fact for a river the receiving water has momentum. This can be used to mix efficiently the heated effluent.

Another difference between the two cases is the temperature. In the case of the river, the temperatures are higher. The water can generally not be used directly and a cooling machine is needed. In this case, the thermal power to be absorbed is increased.

4.3 Design of outlet devices for a fast mixing of the heated effluent with the river

A family of devices able to mix the heated effluent with the receiving environment is developed. They use the momentum of the receiving stream to generate vortices or favorable mixing zones.

Environmental laws mention generally a fast mixing of the waters is favorable. This can be done using such devices.

5 The situation

5.1 State of the art – Some major existing projects

The use of lakes and rivers for cooling as a hot source or directly without chiller exist since about 10 years. Some major projects are summarized in table 1.

Project	Place	Source, cooling power	Year	Reference
Stockholm Energi AB	Stockholm, Sweden	Baltic sea 25 MW	1995	FERMBAECK G. 1995
Purdy Wharf	Halifax, Canada	Pacific ocean 45 MW	2005	
LSC - Cornel University and Ithaca school district	New York, USA	Ithaca lake 15 MW	2000	STEARNS & WHELER LLC 1998
DLWC	Toronto, Canada	Ontario lake 200 MW	2005	
GLN - Genève Lac Nations	Geneva, Switzerland	Leman lake 15 MW	2006	EU FP6 - TetraEner Demonstration activity
Rhône	Geneva, Switzerland	Rhône river 30 MW	1980	BEAUCHAMP H. AND AL. 2007

Table 1 : Some major cooling projects using lake or river water

These are presented more in details in W2.4 of this project.

In this deliverable we will use two projects of this table, one based on the use of the water of lake, and the second based on the use of the water of a river. The problem is not the same from the flow behaviour point of view. These projects are

- Genève-Lac-Nations (GLN)
- Rhône, Geneva

For these projects we will show how to use standard engineer tools, including CFD analysis, to help engineer and environmental planners to design efficient systems.

Figure 1 : the Leman Lake in Geneva

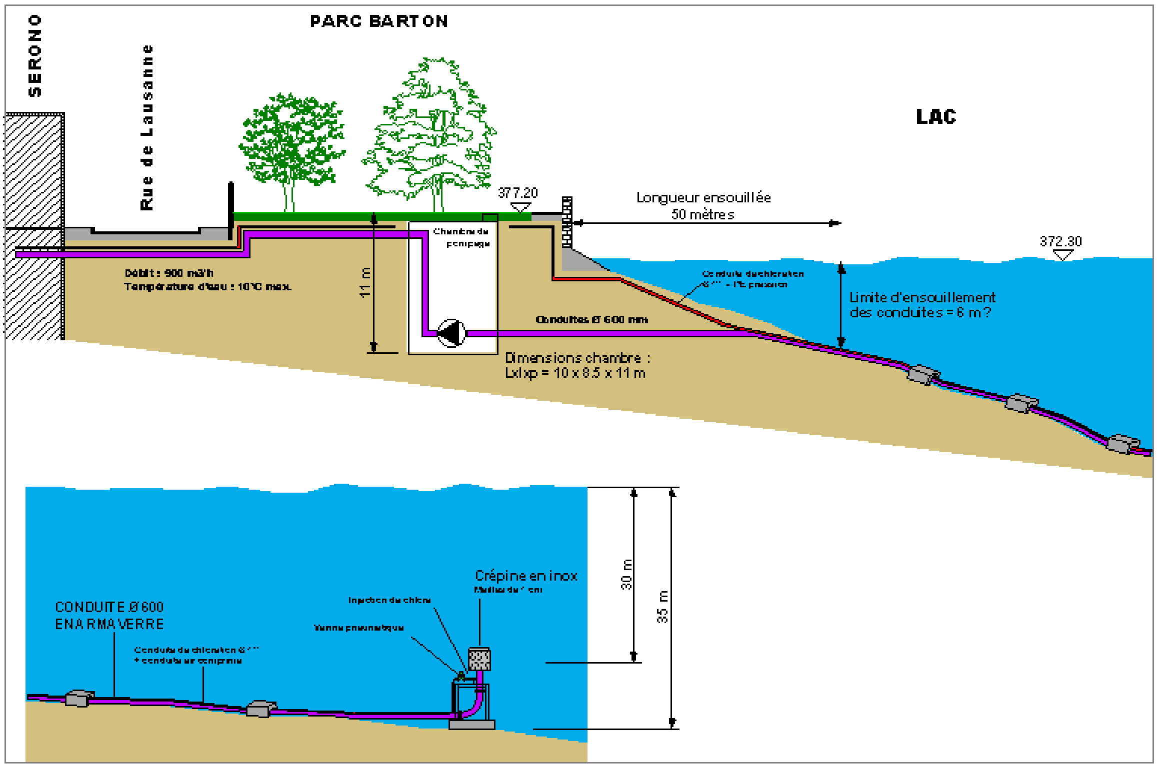


Figure 2 : GLN inlet and installation in Parc Barton

Figure 3 : GLN project inlet device

Figure 4 : GLN project outlet device

Figure 5 : GLN project distribution network

Figure 6 : GLN project pumps

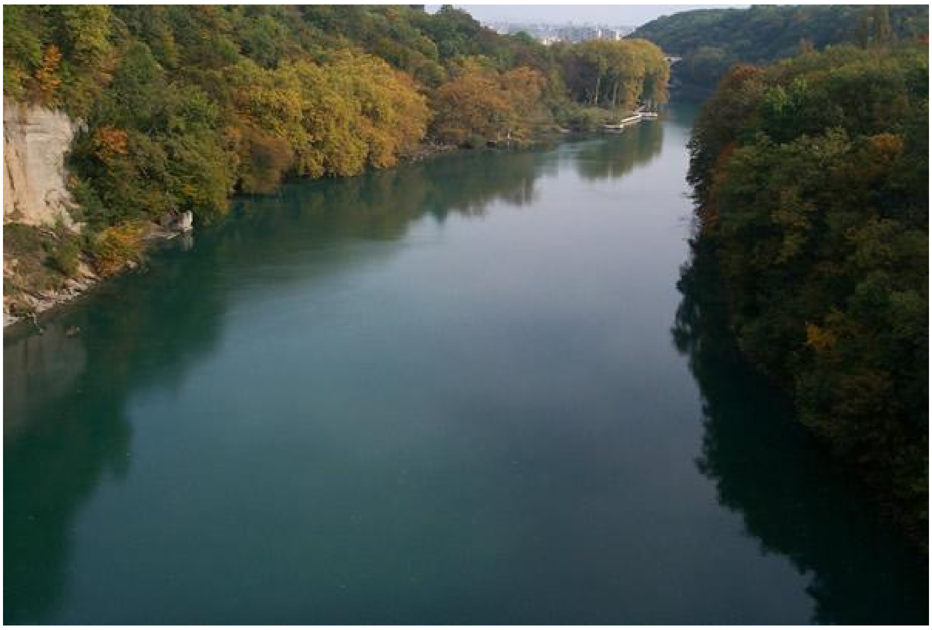


Figure 7 : Rhône river downstream Geneva

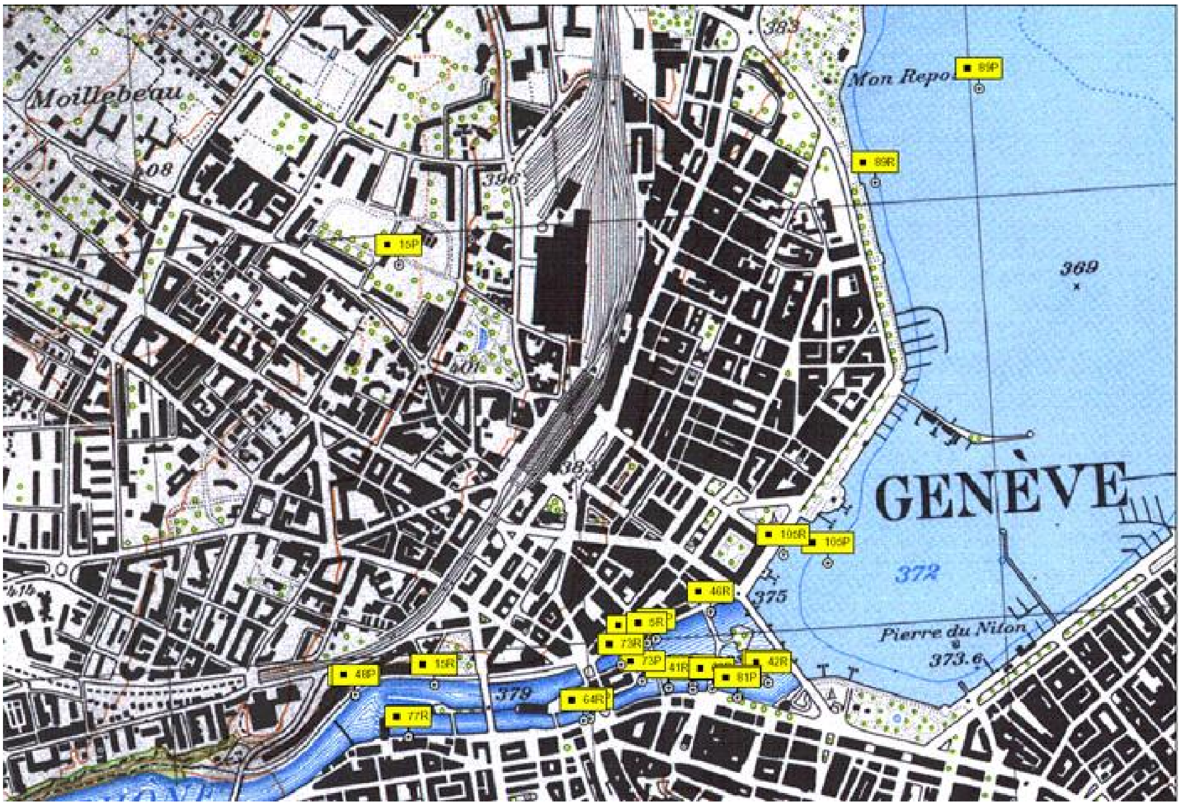


Figure 8 : Locations of authorized heated effluents in Geneva (yellow boxes)

N°	Designation ProE	Owner	Y [m]	X [m]	H [m]	Flow [l/min]
5	Manor	Grands magasins Manor	357	103	369.4	9700
15	FPSG	Fondation prof. et sociale de Genève	870	169	368.7	1320
28	BNP	BNP Paribas (Suisse) SA	398	106	370.2	250
31	COOP	Coop City, centre Rhône-Fusterie	256	221	369	3600
39	DRESDNER	Dresner Bank (Suisse) SA	255	221	368.6	4000
41	UBS	U.B.S.	312	216	368.5	11000
42	ZURICH	Zurich Assurance	74	216	370.15	2250
46	HOTEL_BERGUES	Hotel des Bergues	201	39		2383
48	CPCZ	Caisse de pention du canton de Zurich	1060	187	367.75	3833
62	GP	Au Grand Passage-Innovation SA	147	243		1500
64	CSFB	Credit Suisse First Boston	520	277	368.6	6500
73	BCG	Banque Cantonnale de Genève	422	150	370.1	2000
77	SLCI	Societe Le Capital Immobilier	940	289		2000
78	LLOYDS	Lloyds bank				300
81	CAI	Credit Agricole Indosuez (Suisse) SA	150	243		130
111	LATSIS	CGI Immobilier	1062	189	367.75	1333

Table 2 : Authorized heated effluents in Geneva (situation 2005)

5.2 Bathymetry measurement

For this study it is necessary to obtained bathymetry measurements. Several ways exist for that. The most common is based on the use of a sonar. This techniques coupled with the use of a GPS Global Positionning System) has been developed recently. A boat equipped with these tools is used to record along trajectories the depth of the lake or the river. A treatment of these measurements is made and a map with bathymetry lines is produced.

The bathymetry map used for case 1 of this study has been produced by the Forel Institute (Geneva University).

Figure 9 : The Forel Institute boat "La Licorne"

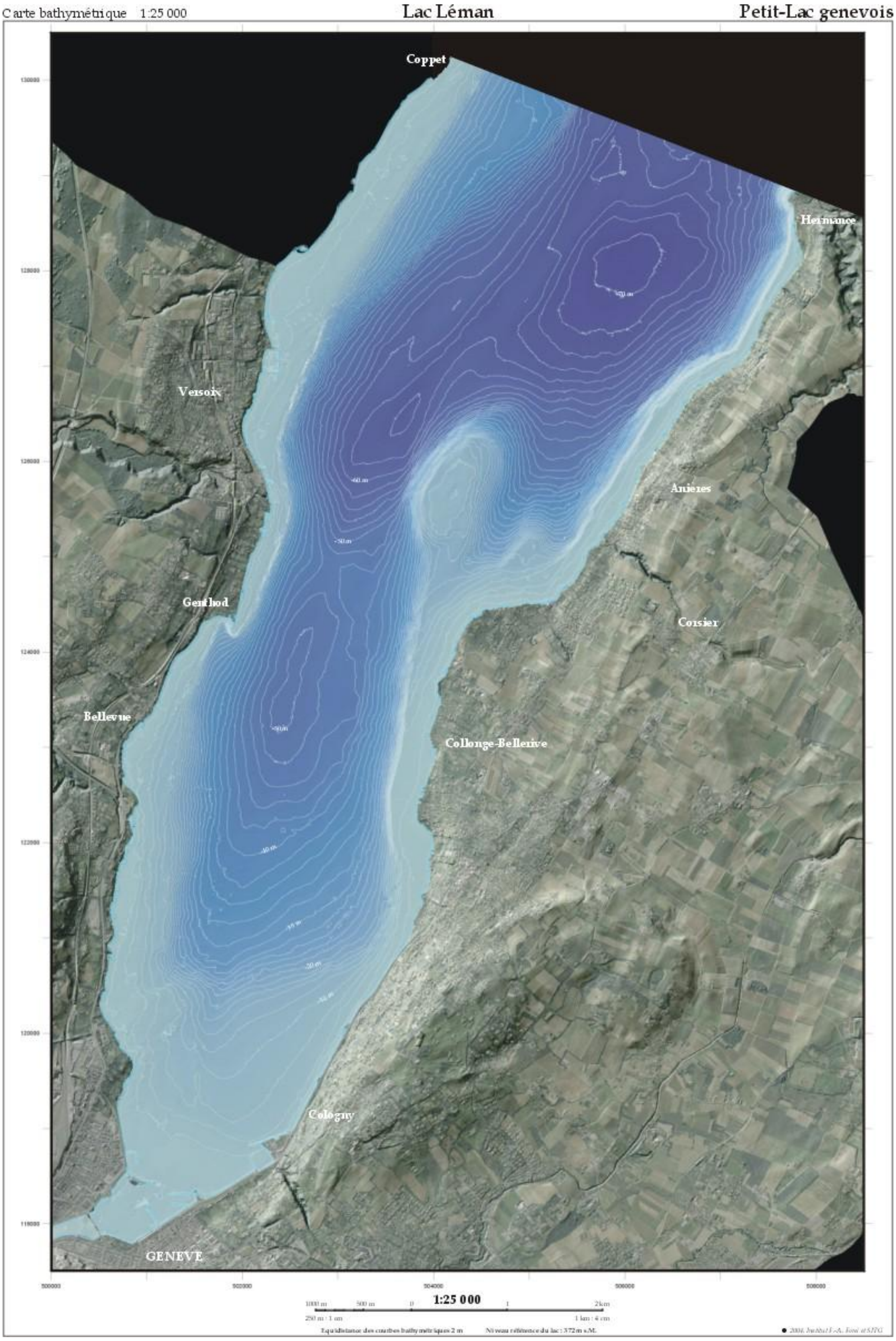


Figure 10 : Bathymetry of the Leman Lake in the Geneva region
(Forel Institute University of Geneva)

5.3 Interest in the use of lake or river water as an energy source

A thermodynamic system used for heating or cooling need two sources. The first is the building or the components to be heated or cooled, and the second is the environment. This last can be the air, water from lake or river, or the ground. In the case of cooling, the Carnot efficiency is much higher if the temperatures of the sources are smaller. That means the use of lake or river water increases considerably the efficiency when compared to air based systems (air cooled condensers).

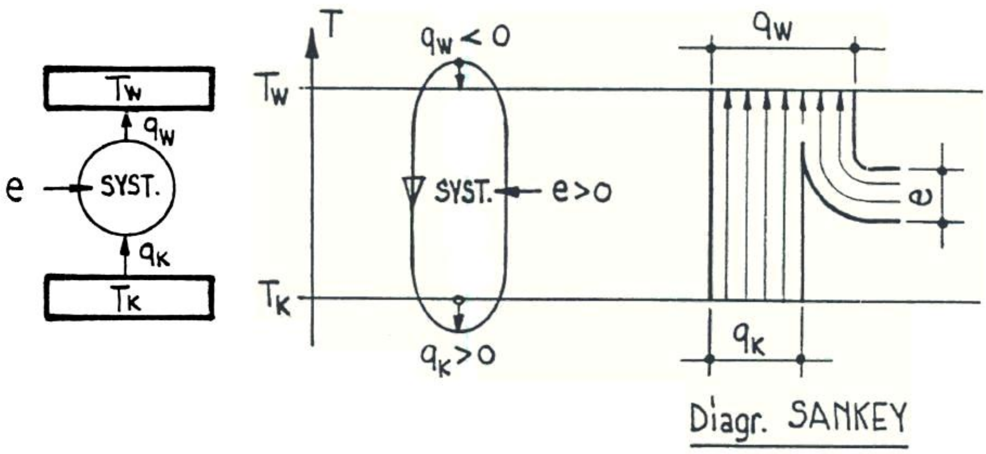


Figure 11 : Dualtherm thermodynamic system. q_w : heating, q_k : cooling, e : mechanical work.

5.4 Thermal effects of energy discharge on the fauna and flora - Legal policies

Rejecting hot water in a lake or a river has several impacts on the fauna and flora. The most important of them is the moving of fish to satisfy temperature needs. This is also the case for micro-organisms or invertebrates. Bigger organisms as fishes will follow them if they move.

Also the fact in some cases we take the water from lake depths it is possible to take some chemical inactive species and to return them a higher temperature where it should be more active.

Each country has its own legal policies to protect environment, including the effects of heated effluents. In Switzerland, and for the two examples mentioned previously, federal and provincial legislations describe the conditions to be respected in a document called "Ordonnance sur la protection des eaux" (Oeaux, october 28th, 1998).

For a river, this means (Switzerland) :

- The rejected water must not be higher than 30 °C and should not increase the river temperature more than 1.5 °C after an homogeneous mixing
- The river shall not be used for heating if its temperature is higher than 25 °C
- The rejected waters must be mixed quickly with the river water; the technical resources used to insure this mixing must be subscribed to the department in charge in order to validate it

For a lake (in Switzerland), this kind of policy doesn't exists. The federal environmental laws shall be satisfied. This includes the need to perform a complete evaluation of the environmental impact.

Applicable laws shall be studied in regards to the country where the project is made.

5.5 Main differences between lakes and rivers for using water as an energy source

The main differences between lakes and rivers for using water as an energy source is the behaviour of the flow in the vicinity of the exhaust device. In the case of the river, the kinetic energy of the flow can be used to favorize the mixing of the heated effluent in the receipting fluid. In the other case, the flow is governed much more by natural convection and diffusion.

For rivers, we have developed static devices able to shorten the mixing length. These are presented more in details in paragraph 8.8.

6 Methodology

6.1 Methodology presentation

The methodology presented in this study includes four steps :

1. Bathymetry measurements
2. Geometrical model construction
3. Definition of typical meteorological situations
4. Simulations
5. Project evaluation

Four techniques are used :

- Echo sounding
- Surfaces reconstruction (geometrical models)
- Computational Fluid Dynamics (CFD)
- Heat exchange models

A computational fluid dynamics (CFD) approach is used coupled with data obtained from previous studies regarding heat exchanges of lakes and swimming pools.

6.2 Methodology to define geometrical models of lakes and rivers convenient to CFD

This is a very nice application of Reverse Engineering as one aim of RE is to provide a geometrical virtual model that suits to reality. The model should have enough detailed features to allow the physical behaviour of the water flow in the lake and Rhone River thought CFD simulation but it also should be too heavily detailed for computation time reason and hardware limitation.
The basic information needed for the Reverse engineering process is about the ground of Rhone River and part of Lake of Geneva (Lac Léman). The format and the structure of those data depend of the measurements means: several sections map through Rhône River or dense cloud of point for the Lac Léman. In both cases Reverse Engineering process has to start with some data re-ordering and cleaning. According to the structure of the input data the method used to model the ground may be some different in the building surface function but with a common objective: generate a solid model – close volume known as CAD B-Rep model (Boundary-Representation).

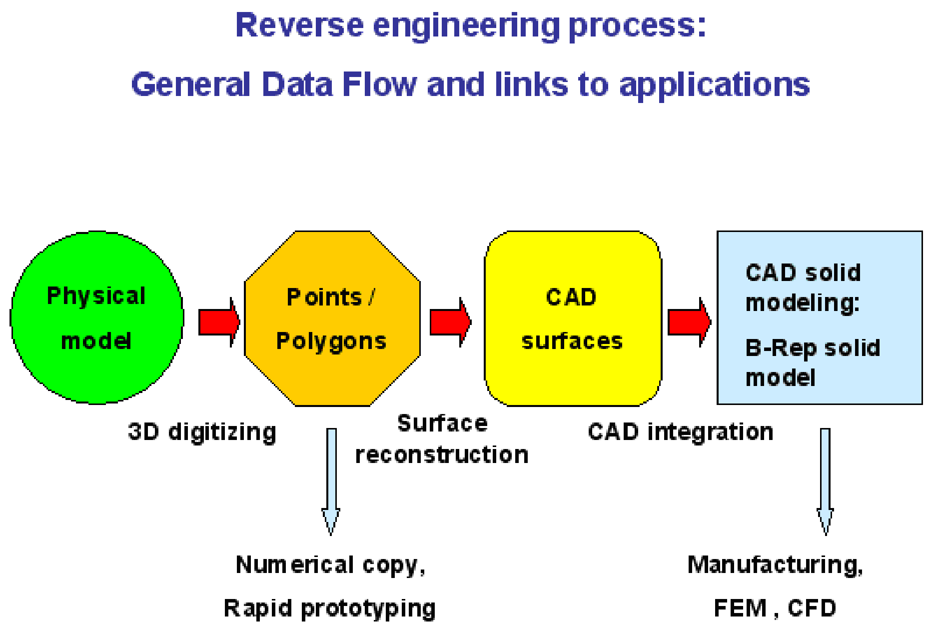


Figure 12 : Reverse engineering process

Model for CFD in FLUENT software: CAD solid model as B-Rep. The input geometry for CFD computation has to define a closed volume more usually known as solid model that satisfies the Euler-Poincaré formula:

$$v - e + f = 2 - 2g$$

v number of vertex (summits)
e number of edges
f number of faces
g number of holes

Most CAD system are using for such purpose the B-Rep model = Boundary-Representation; this B-Rep model may be exported in STEP format – a well known standard of data exchange format. Basically it means that the closed volume, the solid, is described by a set of (boundary) faces. In a B-Rep model, a face is a continuous surface delimited boundaries curves – a face can be flat but may also have some complex shape. It is that geometrical information that is used by le CFD geometrical pre-processor software (GAMBIT) to define the (tetrahedral) mesh needed by the CFD computation. It is interesting to notice that starting with a discrete data of the physical object (points) we first built a continuous model so to achieve with discrete model:

- the solid B-Rep model that is built with B-Splines and NURBS surfaces is a smooth and continuous model as points coordinates may be generated at any requested refinement
- the (tetrahedral) mesh is obviously a discrete model

In a simplified way the solid model of a river or lake extremity is a mapping of a cube as following: Face1 = In-flow section, Face2 = out-flow section, Face3 = lower surface - ground topology, Face4 = upper surface – water/air interface, Face5 and Face 6 = river or lake borders. The boundary conditions for CFD computation are then defined on those faces.

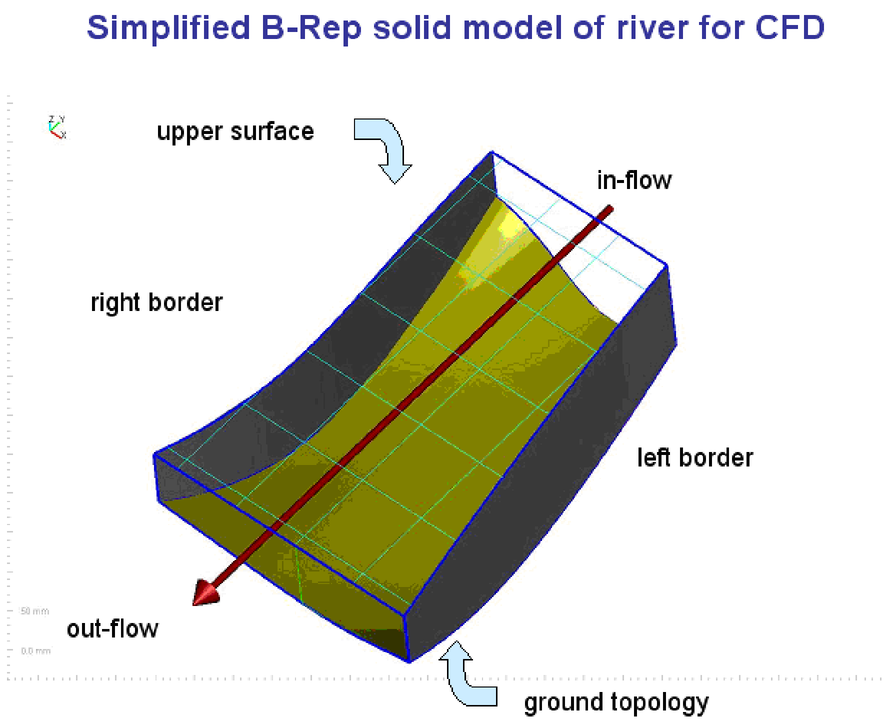


Figure 13 : Solid model representation

The presence of Face5 and Face 6 as vertical walls is important for CFD mesh in order to avoid degenerate tetrahedrons.

6.3 Heat and mass exchanges at the surface

The heat exchanges of swimming pools have been studied by the CUEPE (University of Geneva). The results are reported in the reference Molineaux and al. 1991 and 1994. These will be used as boundary conditions.

To determine the heat flux that must be imposed on the surface, we have to calculate the contribution of three physical phenomenon : the convection, the phase change (evaporation or condensation) and the radiation. We also impose the wind effect on the surface by applying a shear stress in the wind direction.

$$\Delta q = j_{convection} + j_{radiation} + j_{evap/condensation} \qquad (1.1)$$

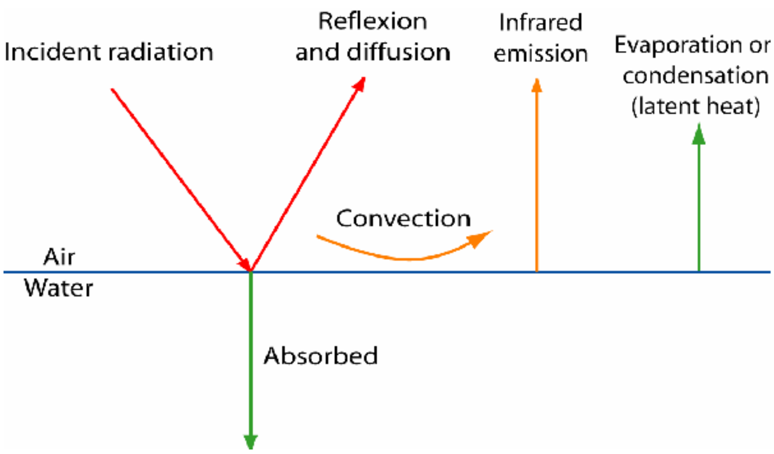


Figure 14 : Heat exchanges at the water surface

<u>Convection</u>

This contribution is determined by the following relations (Molineaux and al. 1991 and 1994):

$$j_{convection} = h_{convection} * (T_{air} - T_{water}) \left[\frac{W}{m^2} \right] \qquad (1.2)$$

with :

$$h_{convection} \approx 3 + 2 * U_{wind} \quad (1.3)$$

<u>Phase change at the surface</u> (evaporation / condensation)

$$j_{evaporation} = h_{convection} \times \frac{L_{water}}{Cp_{air}} \times \frac{M_{water}}{M_{air}} \left[\frac{Pvs(Twater)}{P_{air}} - U \frac{Pvs(Tair)}{P_{air}} \right] \left[\frac{W}{m^2} \right] \quad (1.4)$$

with:

- $L_{water} \left[\frac{J}{Kg} \right]$
- Cp_{air} the air specific heat $\left[\frac{J}{KgK} \right]$
- $M_{water} = 18 \left[\frac{g}{mole} \right]$ the molar mass of water
- $M_{air} = 28.8 \left[\frac{g}{mole} \right]$ the molar mass of air
- $Pvs(Twater)$ vaporization pressure of water (Pa)
- $Pvs(Tair)$ vaporization pressure of air (Pa)
- U the relative humidity (%)
- P_{air} the air ambient pressure (Pa)

Depending on if the air near the water surface is saturated with water or not, we obtain a gain or a loss of energy of the water volume.

<u>Radiation</u>

$$j_{radiation} = e_{water} \frac{G_{meteo}}{24[h]} + e_{water} A - e_{water} \sigma T_{water}^4 \quad (1.5)$$

where:

- e_{eau} the emissivity of water

- $G_{météo}$ the energy brought to the lake during 1 day (meteorological data) $\left[\frac{W}{m^2}h\right]$
- A the radiation the atmosphere on the water surface $\left[\frac{W}{m^2}\right]$
- s the Stefan Bolzmann constant
- T_{water} the temperature of the water $[K]$

A is obtained with an empiric relation (Bolz and Fritz, 1950, see Molineaux and al 1991):

$$A = s\,T_{air}{}^{4}\left(1+K_c B^{2.5}\right)\cdot\left(0.820-0.250\cdot 10^{-0.126 Pvs(Tair)}\right) \quad (1.6)$$

where Pvs is the vaporization pressure. B depends on the type of clouds and Kc of the area percentage of clouds. These 2 parameters are equal to zero when we don't have any clouds.

6.4 Wind shear stress at the surface

The wind shear stress t_p at the water surface is determined by the expressions

$$C_f \cong 0.045\left(\frac{n}{U_\infty d_\infty}\right)^{\frac{1}{4}} = \frac{t_p}{\frac{1}{2} r U_\infty^2} \quad (1.7)$$

with :

- C_f the friction coefficient
- n the cinematic viscosity of the air $\left[\frac{m^2}{S}\right]$
- U_∞ the velocity of the wind out of the boundary layer $\left[\frac{m}{S}\right]$
- d_∞ the height of the boundary layer created by the ground (250 m here), determined by empiric relations:
- $\frac{U}{U_\infty} = \left(\frac{h}{h_\infty}\right)^a$ with U the velocity at a height h, h_∞ the height of the boundary layer and a an empiric factor depending of the situation (for our situation, a = 0.16)
- t_p the shear stress $\left[\frac{N}{m^2}\right]$

- r the density of the air $\left[\frac{kg}{m^3}\right]$

6.5 The simulation program (CFD)

The modeling of both models that are presented in this document, the approach includes the following steps:

- We first have to import a CAD file with the 3D geometry in a meshing code. The purpose of this is the discretisation of the domain in several cells (tetraedrals or hexaedrals, depending on the situation)
- When the meshing is completed, it is imported in a CFD program (Fluent here). All the physical and numerical aspects of the problem are defined: the boundary conditions, the turbulence modeling, the solver, the discretisation type and every parameter relative to these variables.
- The simulation can then be done (the time calculation depends on the model meshing size and the complexity of the physics). When the solution is converged, we can visualize the solution on the geometry domain (for each variable we are interested in). We can then discuss about the results and compare it to measurements in order to validate the model.

6.6 Development of lake and river models

Ideally, to consider all the physical phenomenon that occur in the situation we want to model, we have to include the following effects:

- Heat exchange (activation of energy equation and heat flux on the lake surface)
- Wind (shear stress at the lake surface)
- Gravity
- Variation of water density in function of temperature (piecewise linear model)
- Turbulence (k – epsilon model has been selected)

7 Case 1 : GLN project in the Leman lake

7.1 Case description

The first case we have selected for developing the approach presented here is the Genève - Lac - Nations (GLN) project in the Leman Lake. This project is a demonstration activity of the TetraEner project and is fully described in WP 2.1 to 2.4 (see also chapter 5).

7.2 Geometrical model of the Leman Lake

In few words the reverse engineering process for generating a model of Lac Léman has been the following:

- Collecting data – filtering – grid 3D points model generation
- Definition of a convenient units and origin for CFD geometrical model
- Polygonization of the full data (grid 3D points model) enabling shaded display
- Scaling in Z only help to see the complexity of the topography
- This Z-scaled representation is also very useful for the following surface reconstruction in order to control the surface.
- NURBS surface fitting
- Solid B-rep model generation

The information we started with are measurements data from Institut Forel. A boat equipped with sonar and DGPS has provided a depth map of the considered region of Geneva Lake: that boat named “La Licorne” has scanned the area of the “Petit Lac” trough several trip during 2003 2004. The sonar gives the depth under the boat with a accuracy of less than 0.5 meters and the DGPS (Differential Global Positioning System - linked to SwissTopo) the position of the boat with à accuracy of less than 1 meter. Thus the collected data are huge sets of points witch repartition is mostly inhomogeneous: the 3D points are very dense along the boat path but the distance between two parallel boat paths is about 50 metres. Some filtering is needed as the sonar catch some echoes that are not relevant of the lake’s ground (fishes, water plants). Thus, at that stage of the lake’s ground topography reconstruction is a 3D point model witch is competed by the lake border set at a constant level. This mostly inhomogeneous points repartition is not convenient for topography representation. It is then converted in a grid 3D points model: 1 point each 5 meters is computed by linear interpolation of the original 3D points model (With this a complete map has been provided by Institut Forel).

For the considered area this grid 3D points model represent about 180 000 points – 1 point each 5 meters – that has to be converted in a model suited to CFD simulation. This is the starting point of the next process stage: the reverse engineering. The main stages of this process are:

- Working with points:
  - Importing and data (grid 3D points model as an ASCII file) – defining the origin convenient for the CFD
  - Defining a polygon on the grid 3D points model
  - Inspection, cleaning, adding some points to the imported data

- Working with NURBS surfaces, CAD model generation:
  - Building one NURBS surface fitting the lake's ground
  - Defining a closed volume with complementary surfaces
  - Building a CAD solid model
  - Defining with more detail the lake's border on the CAD solid model

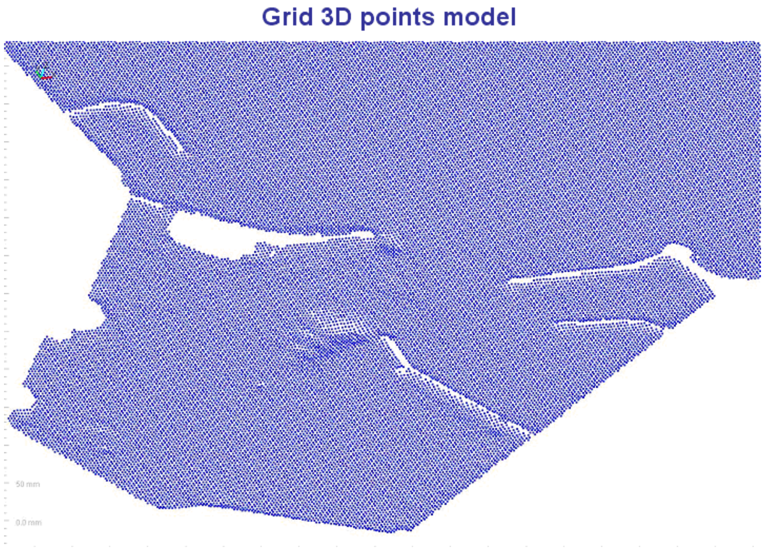


Figure 15 : Part of the lake ground near "Le jet d'eau de Genève": with only dots display as 3D points - it is very difficult to see the topography.

Figure 16 : Polygon built on the grid 3D model

With the polygon virtual shading is possible to see the topography. One observes that the lake's ground is very flat… The ground of the lake is very flat in fact – only 18 m at the deepest point where the lake is 2.8 km large!

The origin of the model is set at the place of the pipe outflow. Using a Z scale factor to show better the topography - part of the fine texture is due to sonar scanning artefacts.

Inspecting the lake's ground topography with the grid 3D point data. The Z scale factor shows also artefacts due to the scanning method.

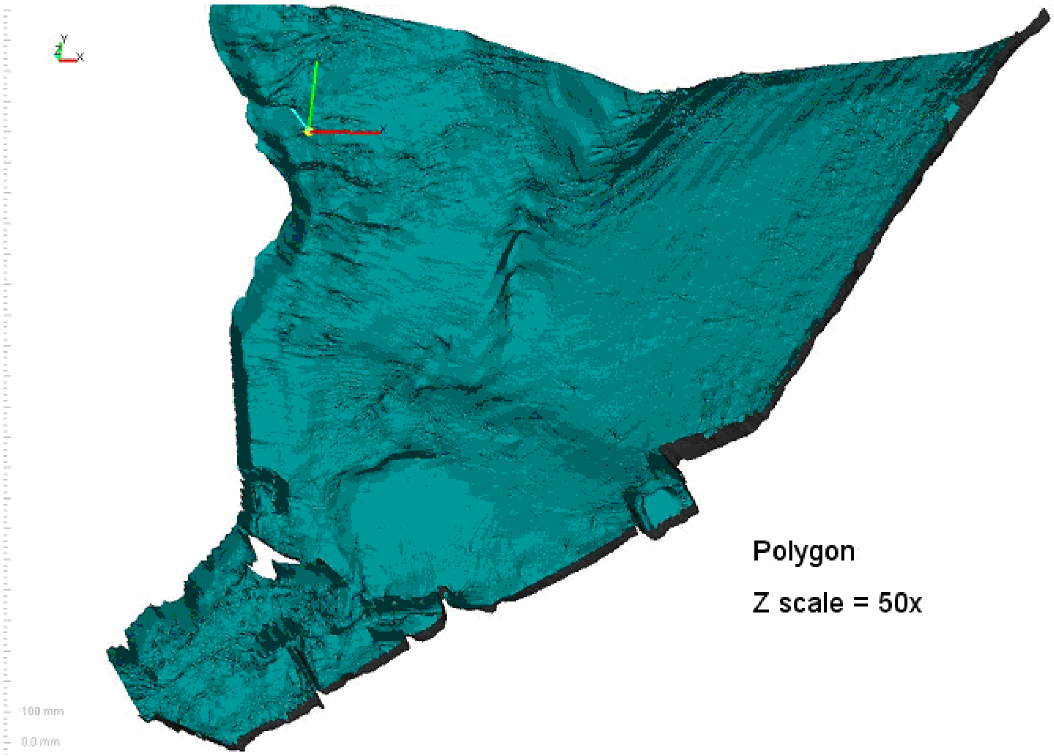


Figure 17 : Lake's ground topography

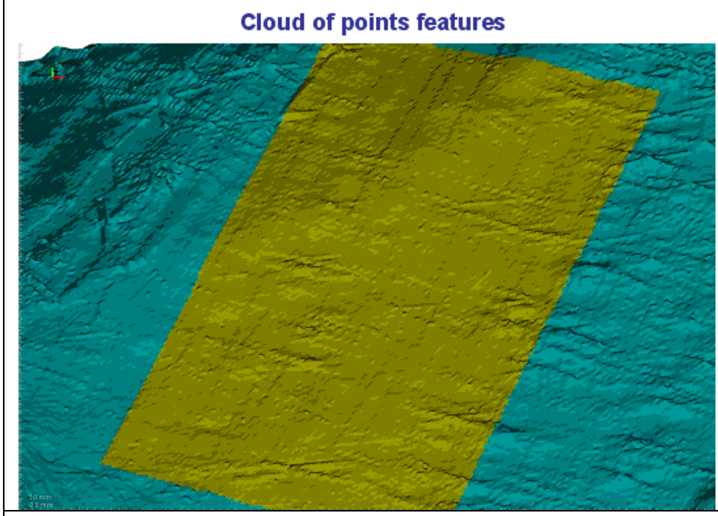
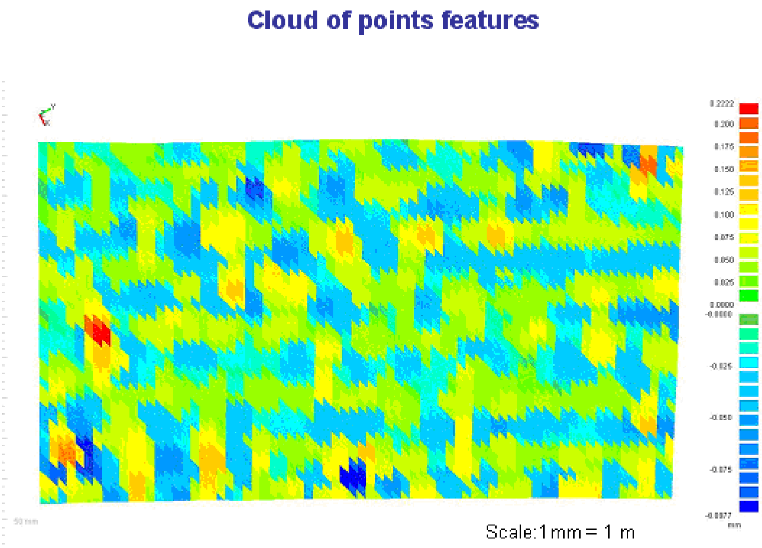
	Sonar scanning artefacts: Path of the boat are visible as kind of trail. Some extra echoes give kind of cross peaks.
	Sonar scanning artefacts: The resulting measurement noise has a standard deviation of 0.025 [m]

Figure 18 : Measurements of the cloud of points

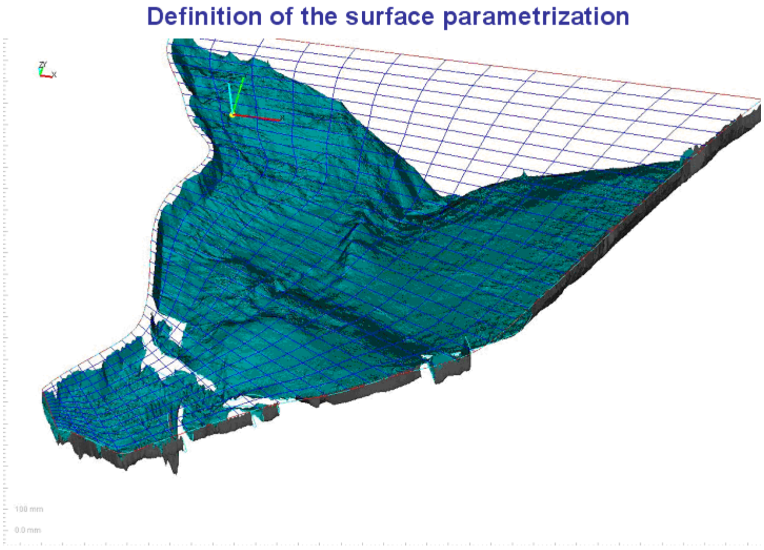


Figure 19 : Geometrical model of Lac Léman - Working with NURBS surfaces, CAD model generation

The parameterization of the surface has to be defined according to the main feature of the lake's ground topology. On the main two lake's sides borders are built curves that extrapolate slightly the lake border shape.

The flexibility of the NURBS surface is given by its number of knots and control points. On this figure the knots are given by the crossing point of the displayed curves network.
The NURBS surface is then defined by wrapping the parameterization on the grid 3D points.

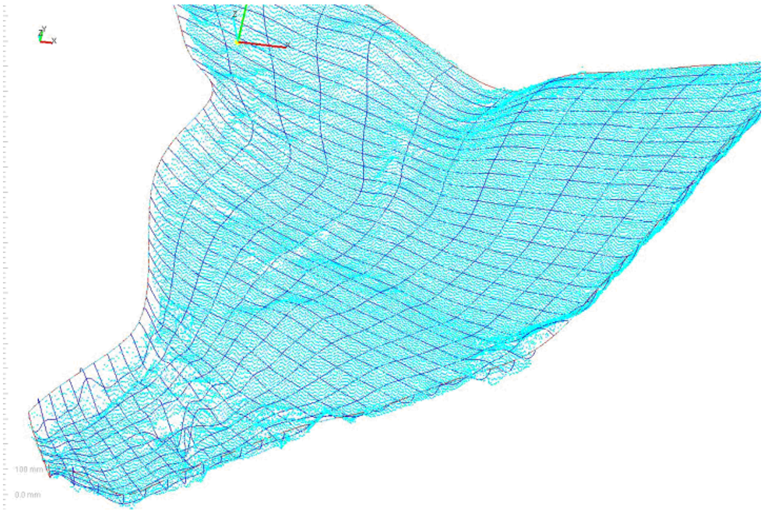


Figure 20 : NURBS surface fitting to the lake's ground

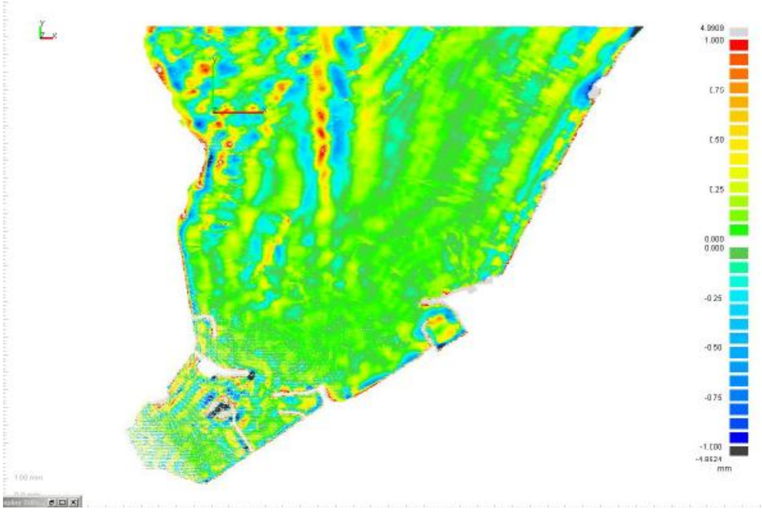


Figure 21 : NURBS surface to grid 3D points difference

The NURBS surface definition has been done with a scale factor 50 on the Z axis. This was temporary – then it is reduced to the right scale. It is important to control that the NURBS surface fits well to the input grid 3D points.
In case of too big differences, another parameterization should have been defined with more knots and control points.

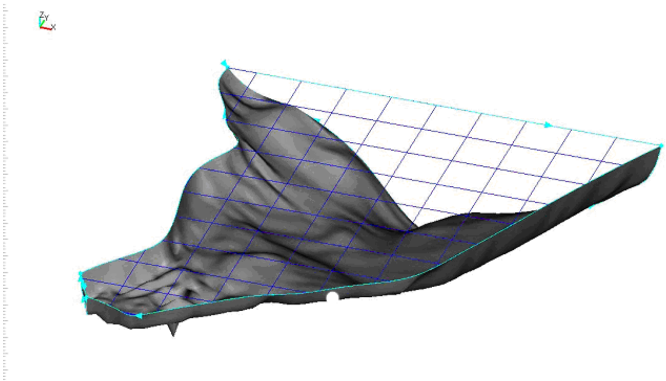


Figure 22 : Defining a closed volume with complementary surfaces

In order to define a solid (B-Rep Model) complementary surfaces have to be defined witch will be considered as faces of the solid.
Here 3 surfaces have to be added: the top surface, the in-flow surface and the out-flow surface.

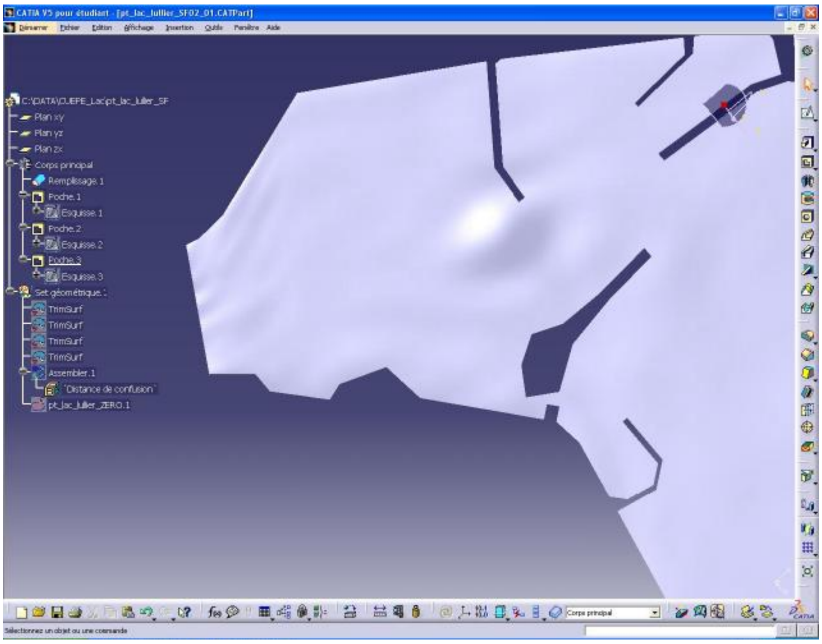


Figure 23 : Defining a CAD solid model - completing border lake's details

The final part of the lake is here shown by underneath in order to make perceptible the ground topography.
Once a B-Rep CAD solid model is defined, it is completed by trimming operation drawing with quite a lot of detail the lake's border.

This trimming has a benefit aspect as it defines vertical walls; even if this is very small relatively to the size of the lake, it avoids having degenerate mesh for CFD.

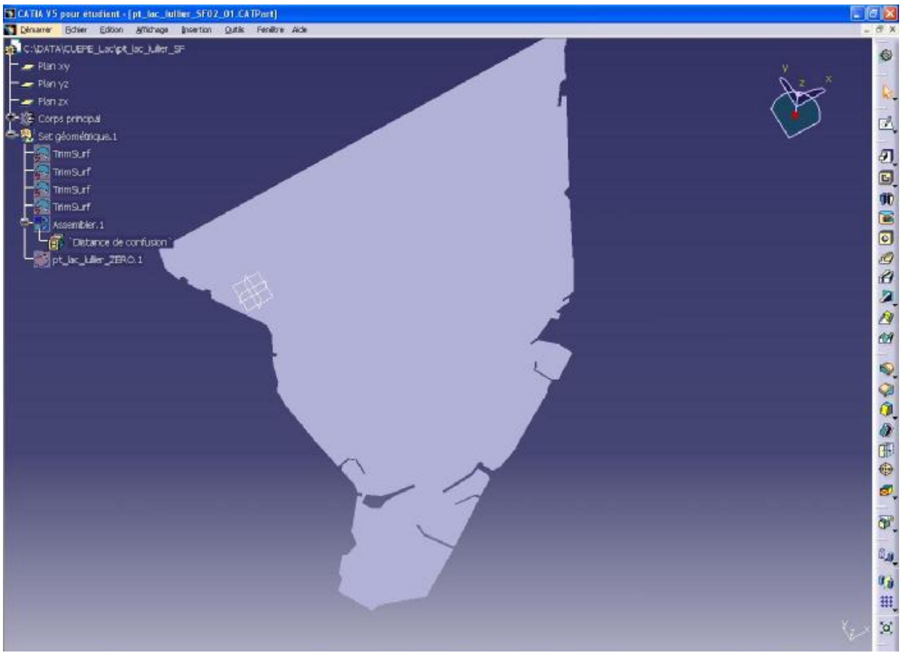


Figure 24 : Compete CAD solid B – rep model

7.3 Exhaust pipe modeling/dimensioning

In function of the mean mass flow (here about 1000 kg/s, 1 m^3/s of water) in the pipe, methods exist to determine an ideal straight conduct diameter. The head loss (for our case, viscous head loss) can be calculated with empiric relations, in function of the roughness, the flow field, the flow type (Reynolds number), the length of the conduct and the fluid density).

Relations between these variables can be found for example in “Memento des pertes de charge”, I.E. Idel’cik, Collection de la Direction des Etudes et Recherches d’Electricité de France, Chap. 2.

First, we have to choose a mean velocity in the conduct (knowing the mass flow). Generally, we choose 1 – 2 m/s. This gives us a conduct diameter.

After that, knowing the velocity and dimensions of the conduct, with the Idel’cik relations, we can determine the head loss in the conduct. It is then possible to choose the pump that corresponds to this regime.

In our case, at the end of the conduct, the 1 m^3/s water mass flow is extracted out of the pipe by 16 circular wholes. The mass flow and velocity is so decreasing along the conduct. Dimensions and plans of the GLN pipe are illustrated on the following pictures.

Length of the conduct (Station – diffuser) = 158 m.

The pipe can then be implemented in the lake geometry.

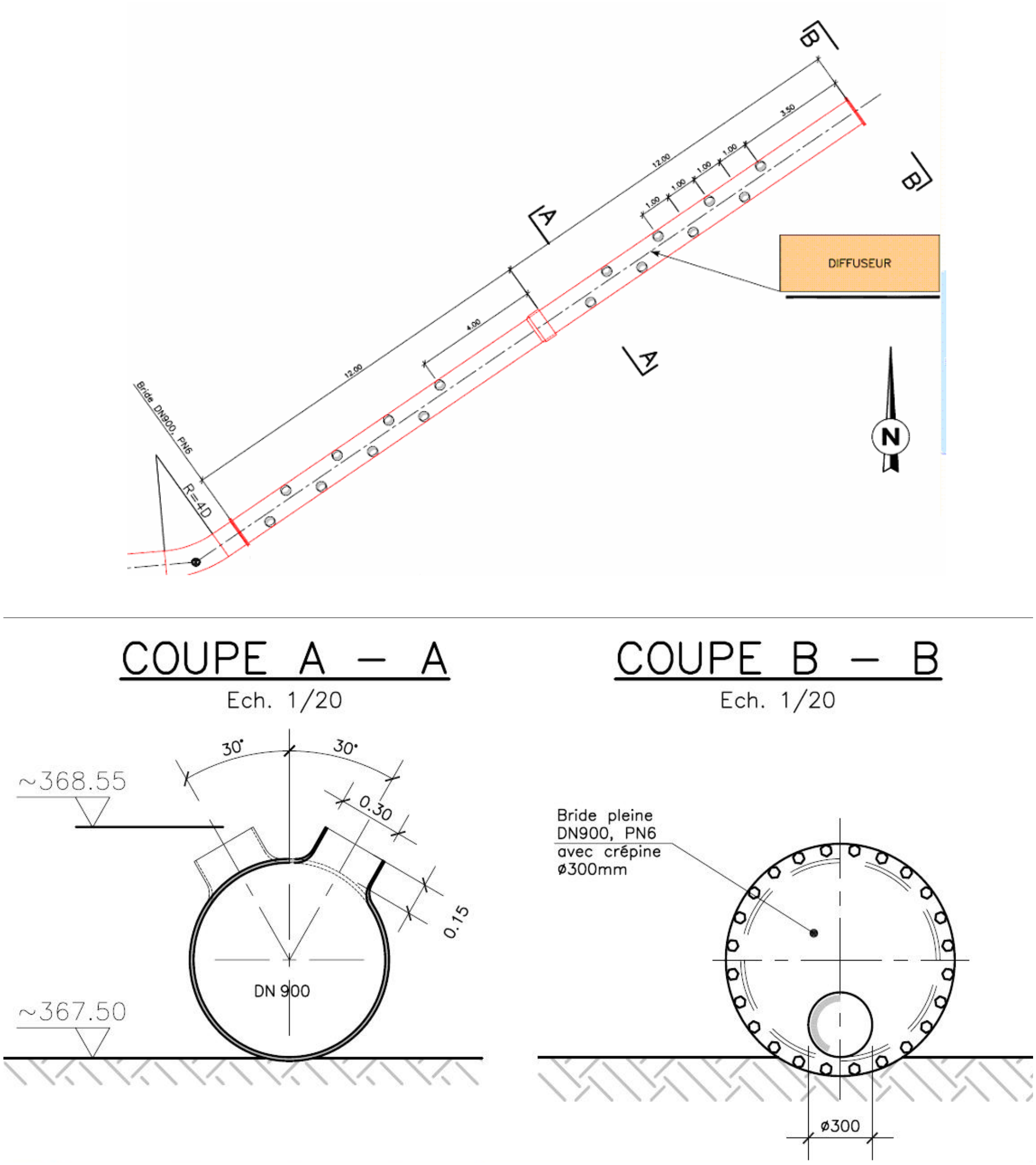


Figure 25 : dimensions of the diffuser

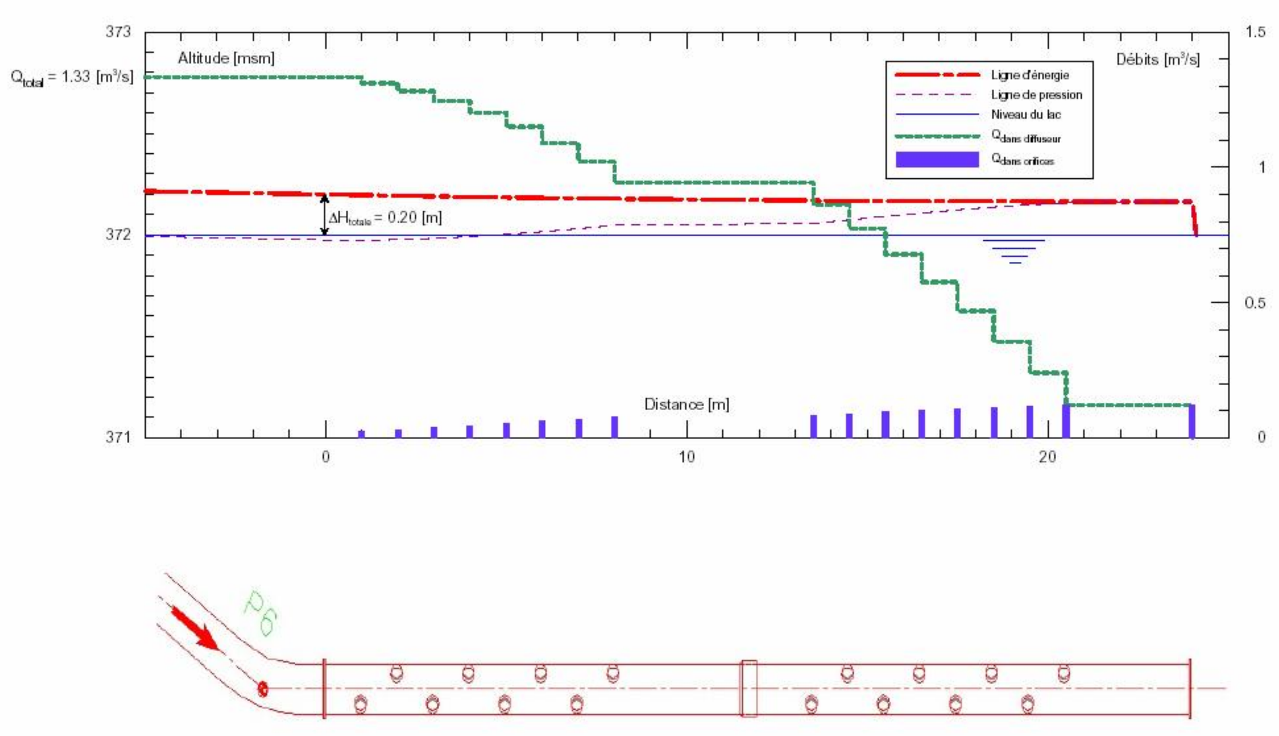


Figure 26 : Hydraulic characteristics of the diffuser (SIG)

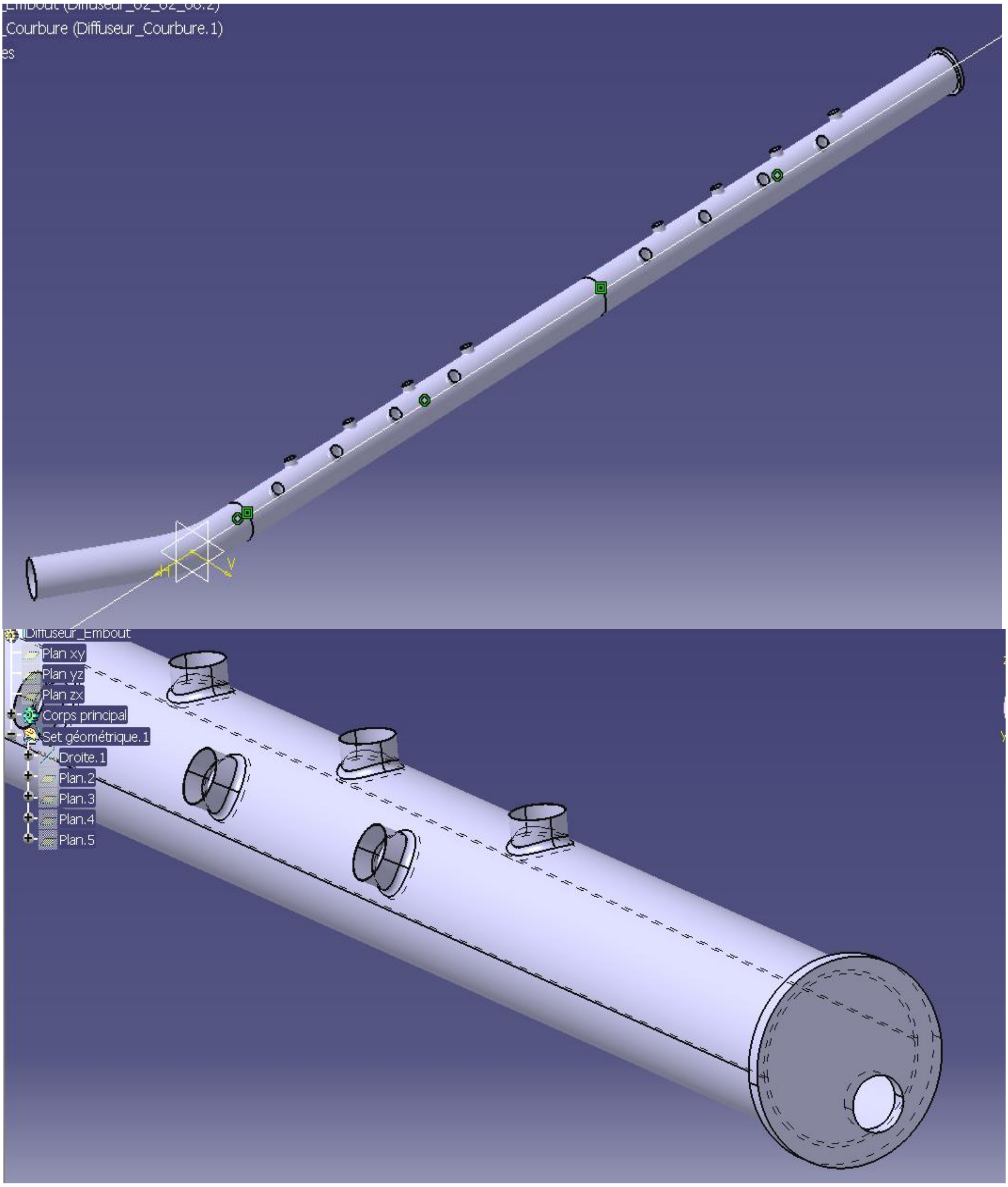


Figure 27 : CAD geometry of the diffuser

7.4 In situ tests performed

Measurements have been performed at several locations. They can be used to validate and determine boundary conditions for numerical simulation. These measurements are described on the WP 2.4.

7.5 Boundary conditions

We have chosen boundary condition in function of the information we had about the flow conditions of this part of the leman lake. Here is the list of physicals boundary conditions we have imposed:

- The borders and the bottom of the lake are considered to be adiabatic walls.
- The lake surface is considered to be a slipping wall, so that we can impose a specified shear stress condition and a heat flux, corresponding to the sum of convection, evaporation/condensation and radiation effects.
- The line where the TAM measurements have been done is considered as a velocity inlet. At this boundary, we impose a velocity and temperature profile. For the temperature, it corresponds to the measurement done by the CUEPE during WP 2.4 (the points along the domain inlet on the picture below), and for the velocity, we have used the data of the study from 1931.
- The boundary corresponding to the Mont Blanc's bridge is considered as a velocity outlet. We impose a constant velocity field so that the mass flow at this boundary corresponds to the mass flow of the Rhône.

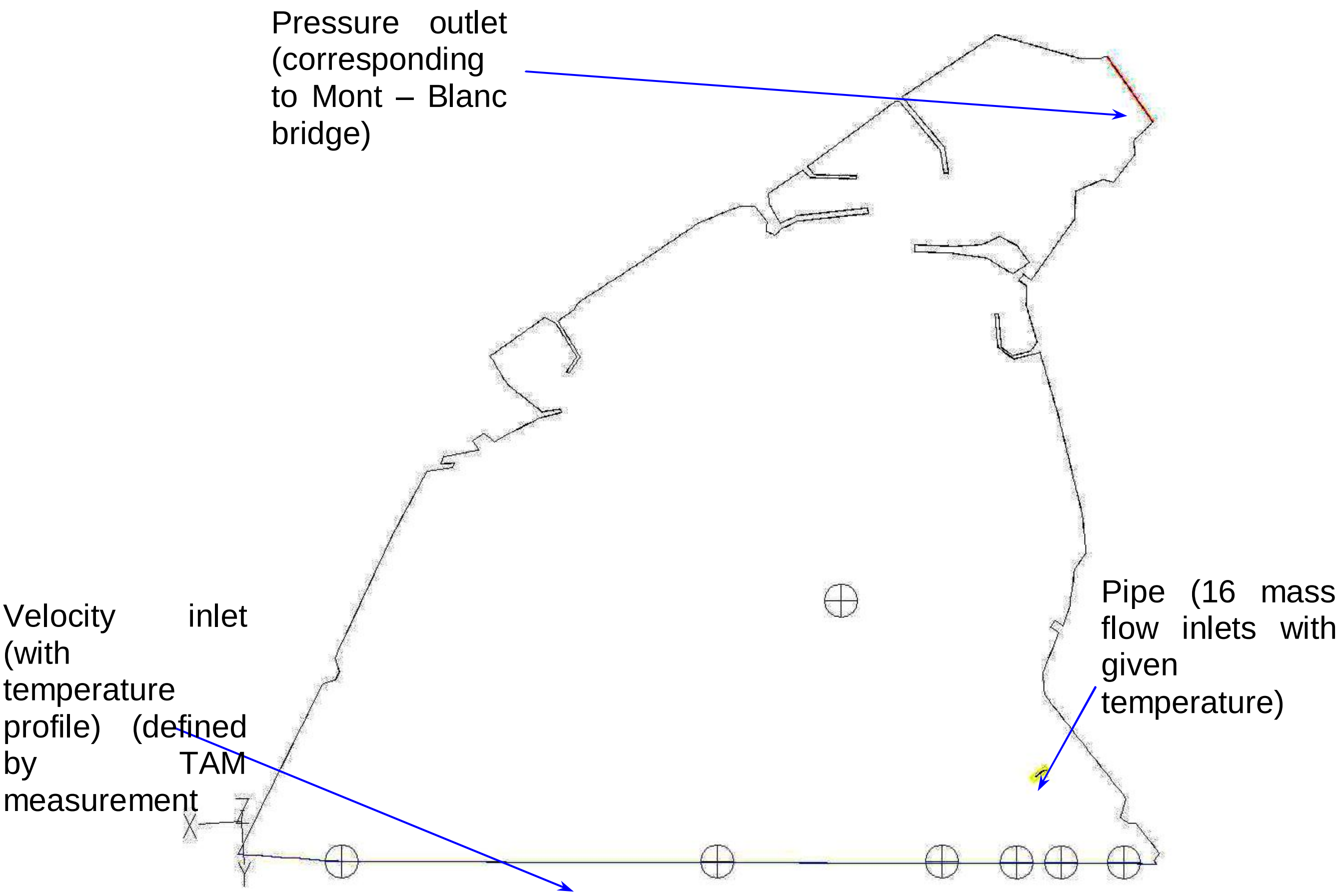


Figure 28 : Geometry and boundary conditions

7.6 The numerical model

For the meshing of the domain, according that the main component of the velocity field was horizontal, we decided to create a structural grid with hexaedral cells, oriented with the flow. The cooper scheme has been used in this case. The principle of this algorithm is that we reproduce the same meshing configuration of a face (here the surface or the bottom of the lake) on several upper layers (10 here). It gives also two advantages:

- Anywhere in the domain, the calculations will give us 10 values in the vertical direction. This is optimal for example to compare temperature profiles.
- This type of meshing involves a lesser number of cells, in comparison to a non – structured meshing (tetraedral) for example. This will generate a model that asks less calculation time.

We had to separate the model into 3 parts in order to be able to impose such a structural meshing. Without this modification, the meshing code was creating deformed cells and it would not have been possible to obtain any results with the CFD code.

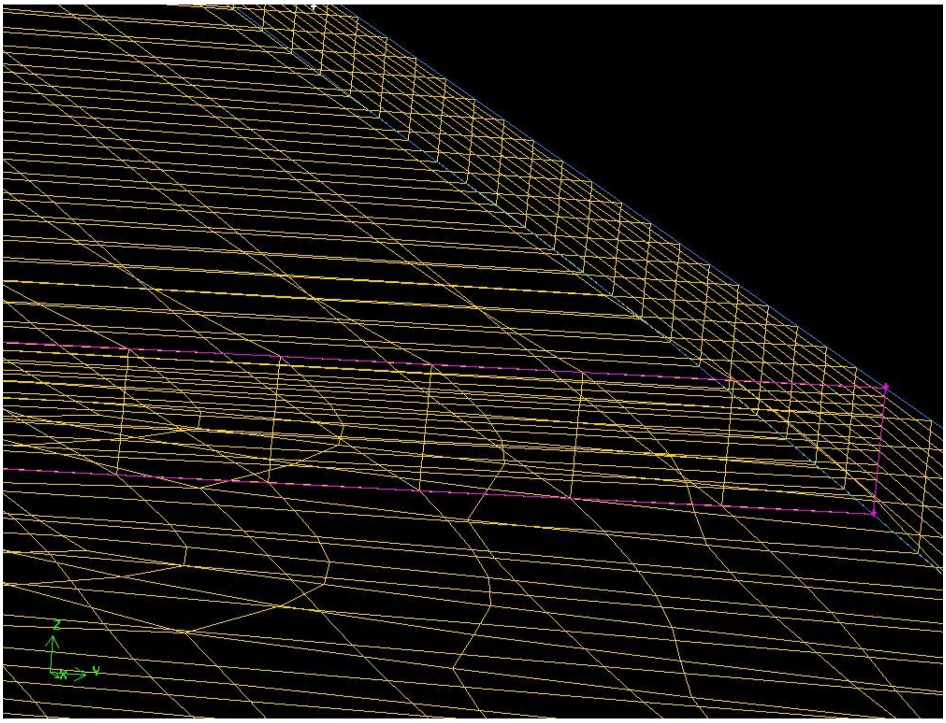


Figure 29 : Illustration of the hexaedral structural meshing of the domain
(with 10 layers in the vertical direction)

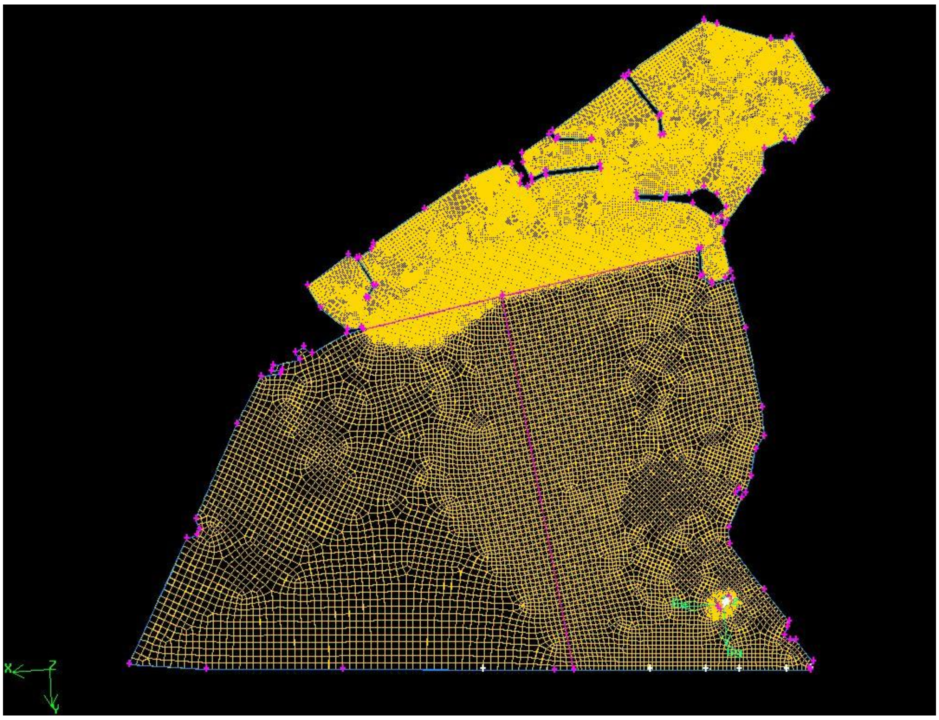


Figure 30 : Complete view of the meshed geometry

The zone where the pipe stands cannot accept a structural meshing, due to the geometry. We separated a zone around the pipe and imposed an interface to authorize the presence of both hexaedral and tetraedral meshes. The mesh is also refined in this region. This is made possible by imposing a size function.

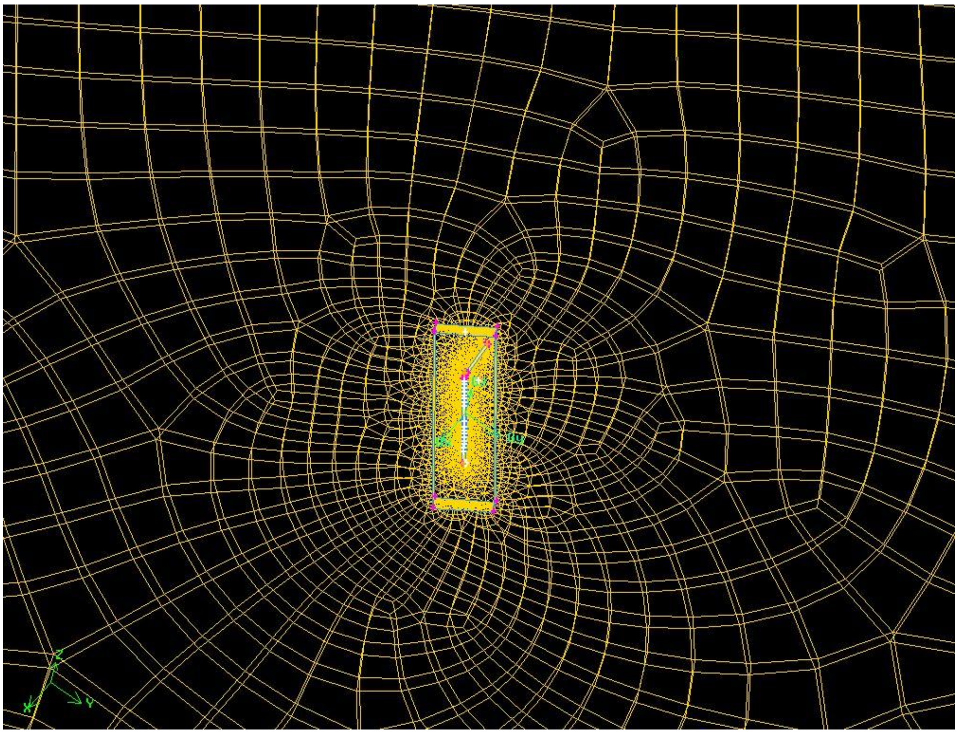


Figure 31 : Zoom on the pipe – zone mesh (interface wit non – structural mesh (tetraedral cells)

For the solver and discretisation parameters we used the following options:

- Flow considered to be stationary.
- Pressure based and segregated solver
- Pressure discretisation: PISO
- Other discretisation schemes : first order
- Low to very low (0.1 – 0.3) under – relaxation factors

We encounter convergence difficulties by imposing all these specifications at the same time during the calculation process certainly due to the gravity effects and the profile of temperature and velocity we have to impose on the inlet. To obtain a good result, we had to proceed like this:

- First impose a constant velocity profile to obtain a realist velocity field, without activating energy equation, gravity and variable density.
- Then converge a calculation by activating the energy equation and imposing the real velocity – temperature field at the inlet of the domain, corresponding to the measurements that have been done.
- Impose a constant heat flux and shear stress on the lake surface. The shear stress field will perturb the velocity field on the surface, and the temperature field.

- Then activate the gravity and variable density.

7.7 Results with North wind (Bise)

Numerical values :

Mean air temperature [K]	295.02
Mean water temperature [K]	296.3
Pipe temperature [K]	287
Relative humidity [%]	68.7
Wind speed [m/s]	1.52
X Shear stress [N/ m^2]*	-0.01146
Y Shear stress [N/ m^2]	-0.01637

* Reference of the coordinates axis from the CFD model

Convection [W/m^2]	-8
Evaporation/condensation [W/m^2]	85
Radiation [W/m^2]	224
Total [W/m^2]	301

In this case, the water from the pipe is 287 [K], which is cooler than the lake at this period (mean temperature about 291 [K]). We expect so to see effects of these mass flow inlets on the temperature field near the bottom of the lake (with effects of the gravity, cold water goes down).

In this case, the water from the pipe is 287 [K], which is cooler than the lake at this period (mean temperature about 291 [K]). We expect so to see effects of these mass flow inlets on the temperature field near the bottom of the lake (with effects of the gravity, cold water goes down).

Velocity field :

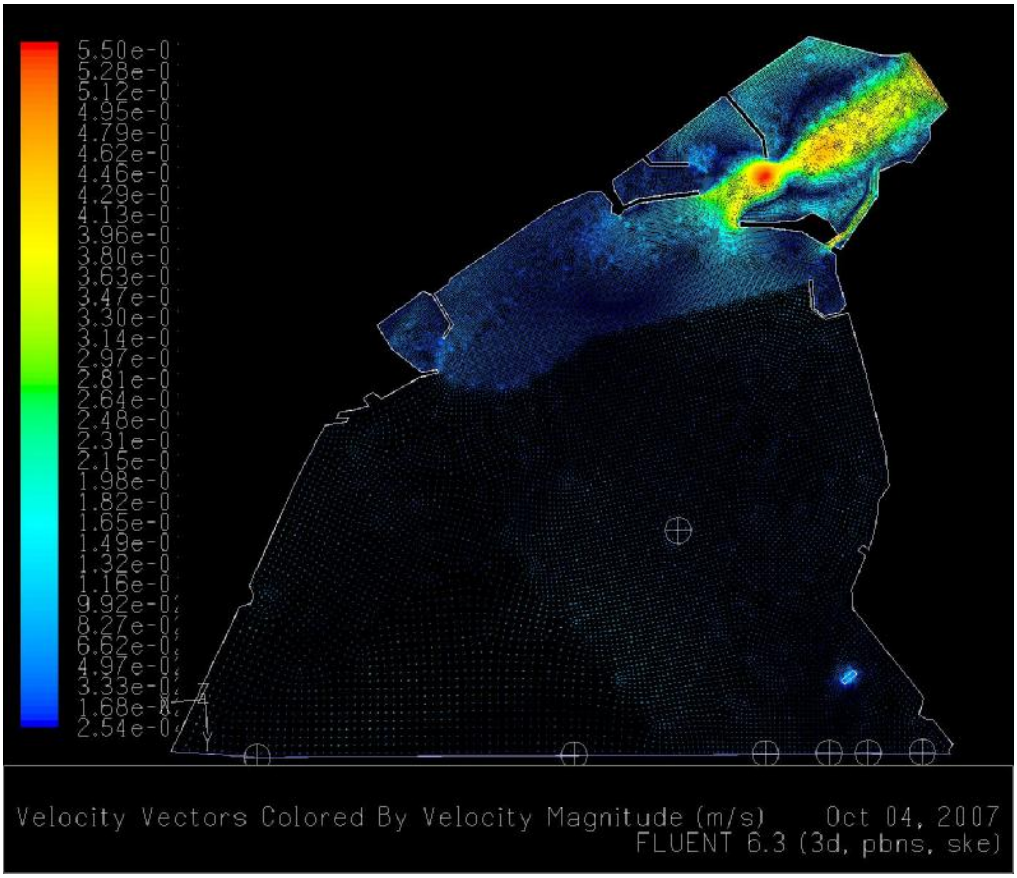


Figure 32 : Velocity field on the lake surface

The values are generally low (maximum value near the Mont Blanc bridge: 0.5 [m/s]). We can notice that the direction of the flow field is different between the surface and the near – bottom zone. This is the effect of the north wind.

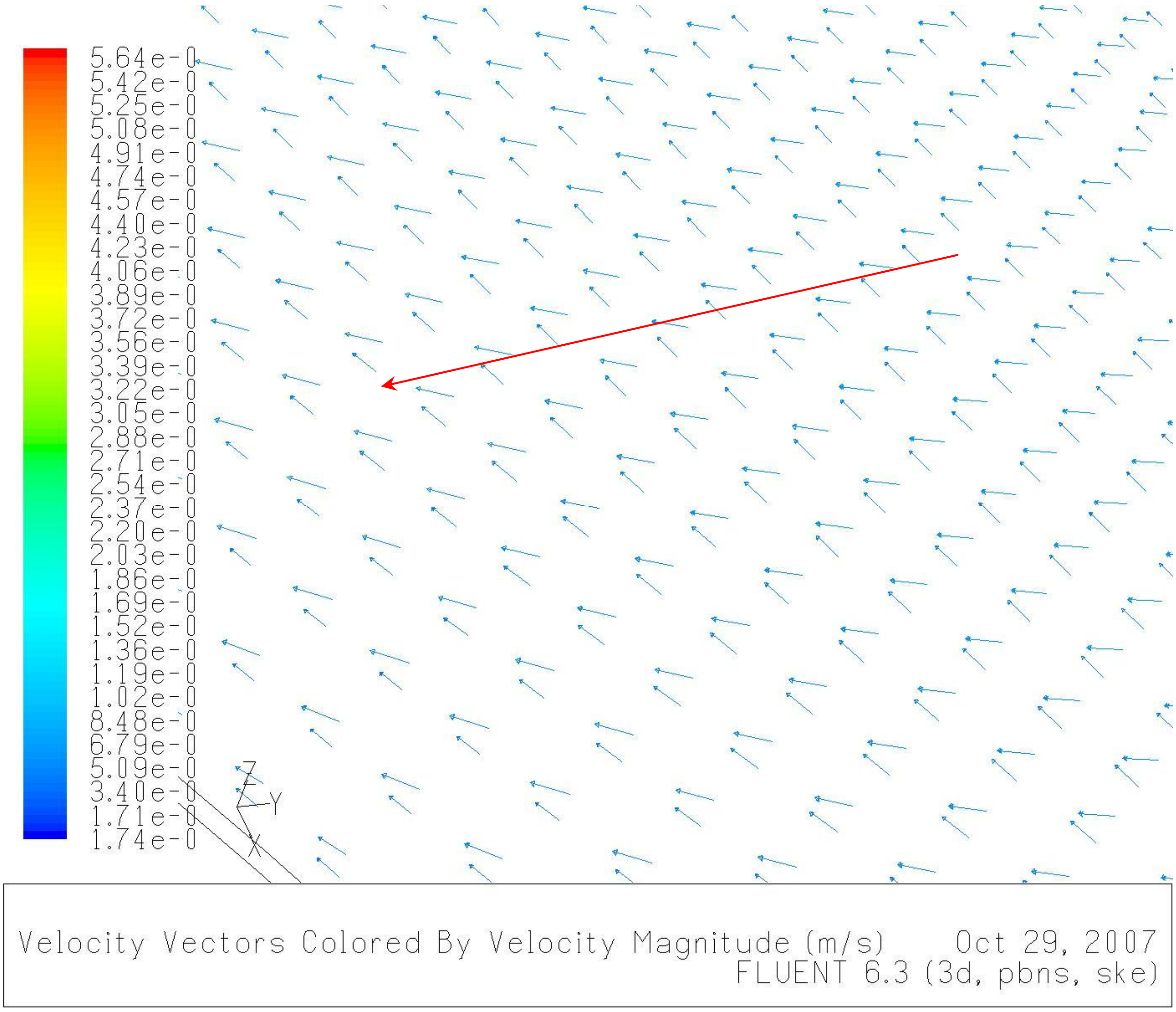


Figure 33 : Velocity field on the lake surface and bottom to illustrate the effect of the north wind. The red arrow shows the direction of the wind.

Temperature field:

Surface and bottom:

The temperature field on the surface is relatively uniform for the large zone of the domain. For the bottom of the lake although, the values are lower (especially for the zones where the lake is deep). We notice some warmer zones that correspond to stagnant water. When the lake section becomes smaller, in some zones, the temperature equalizes itself between the bottom and the surface. This is certainly due to the fact that the lake is lesser deep in this zone and there is more water mixing from different depths. The depths vary from 14 to about 2 meters (in the region of the Mont Blanc's bridge).

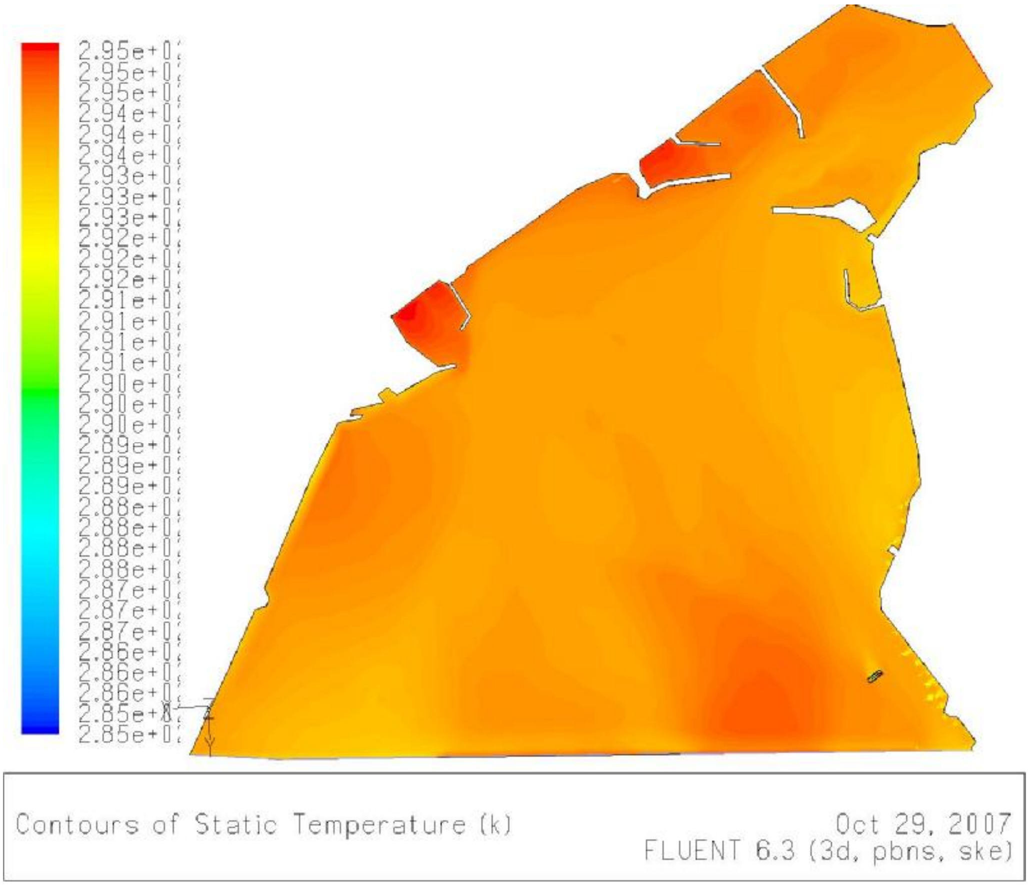


Figure 34 : Temperature contours at the lake surface

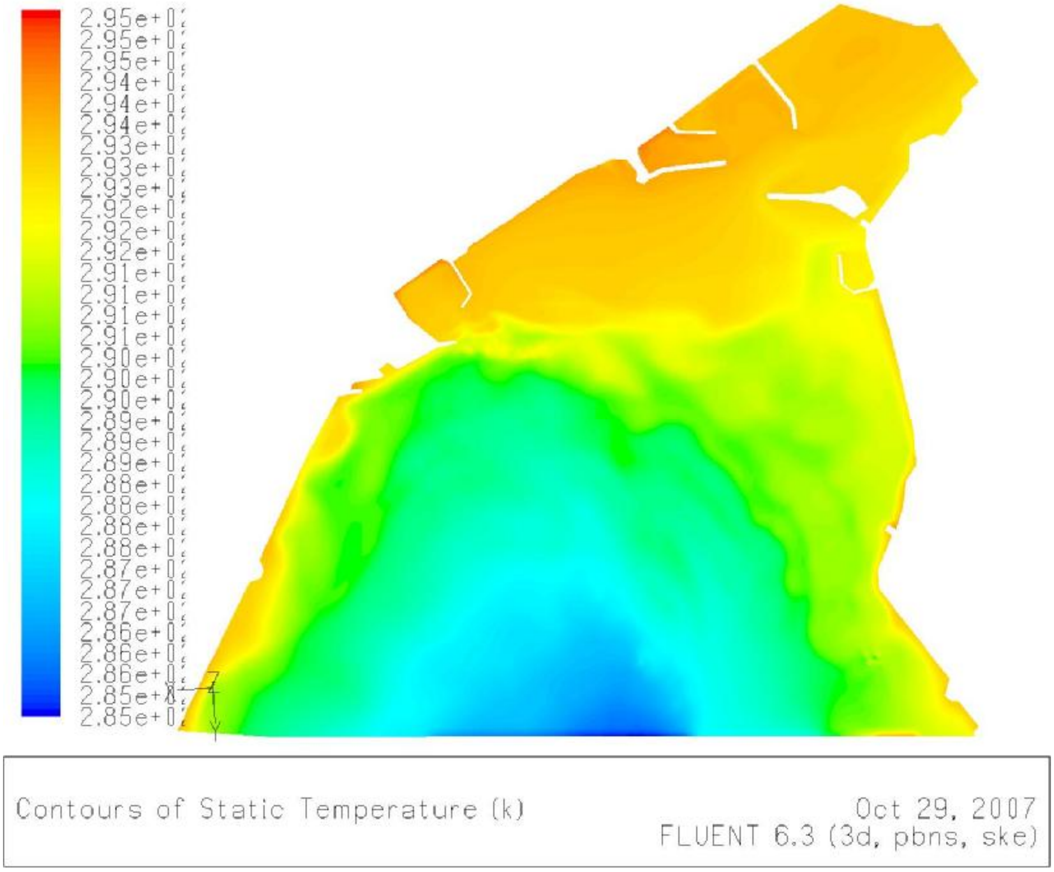
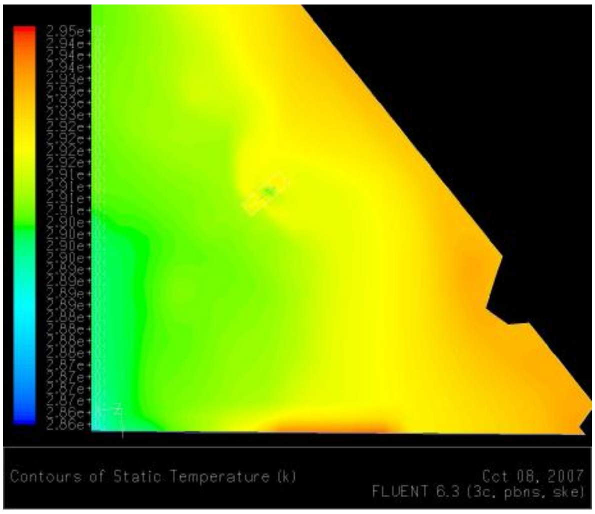


Figure 35 : Temperature contours at the lake bottom

Influence of the pipe:

The flow field has been calculated with and without activating the inlets from the pipe in order to be able to examine the influence of it. The two following print screens illustrate that. The water from the pipe has an influence on the temperature field, but his effect disappears fast completely after a few meters.



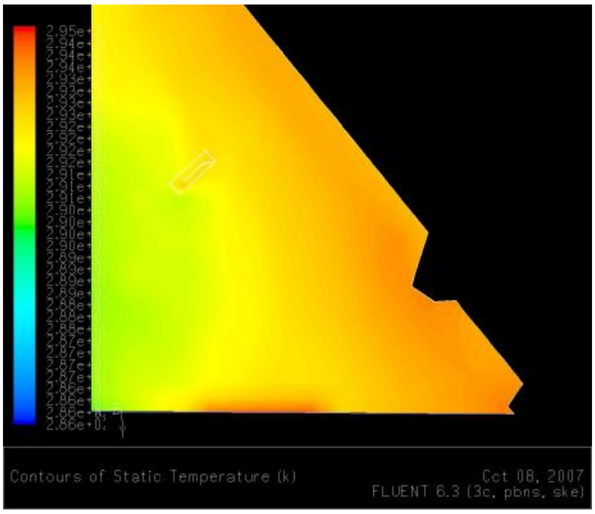


Figure 36 : Temperature field near the pipe region when the mass flow inlets of the pipe are settled to 0 (right) or experimental value

Zoom on the pipe for flow field simulation with mass flow inlets from the pipe (left) and without it (right).
There is fast no influence on the surface flow between these two simulations. As expected, the temperature dependant density has an important influence on the temperature field here. This can be observed with the two following screens: We have represented pathlines from the inlets of the pipe. The inlets are oriented in the direction of the lake surface, but most of the pathlines show that the water drawn down fastly.

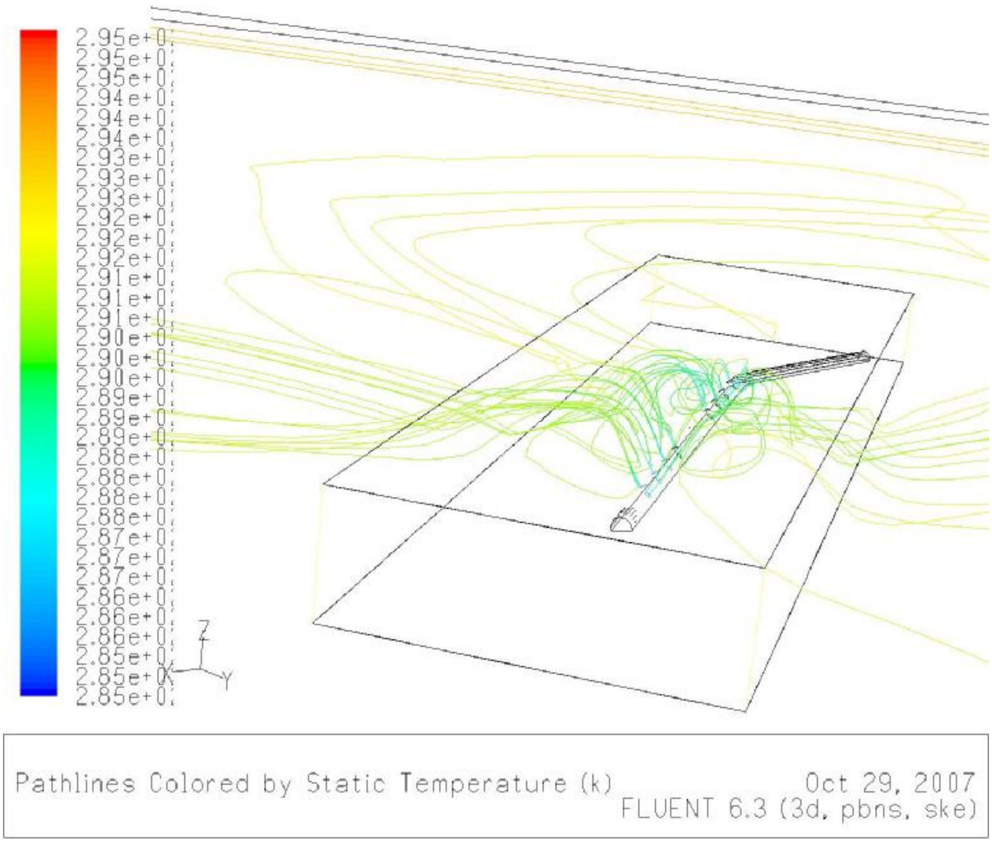


Figure 37 : Pathlines in the vicinity of the outlet pipe

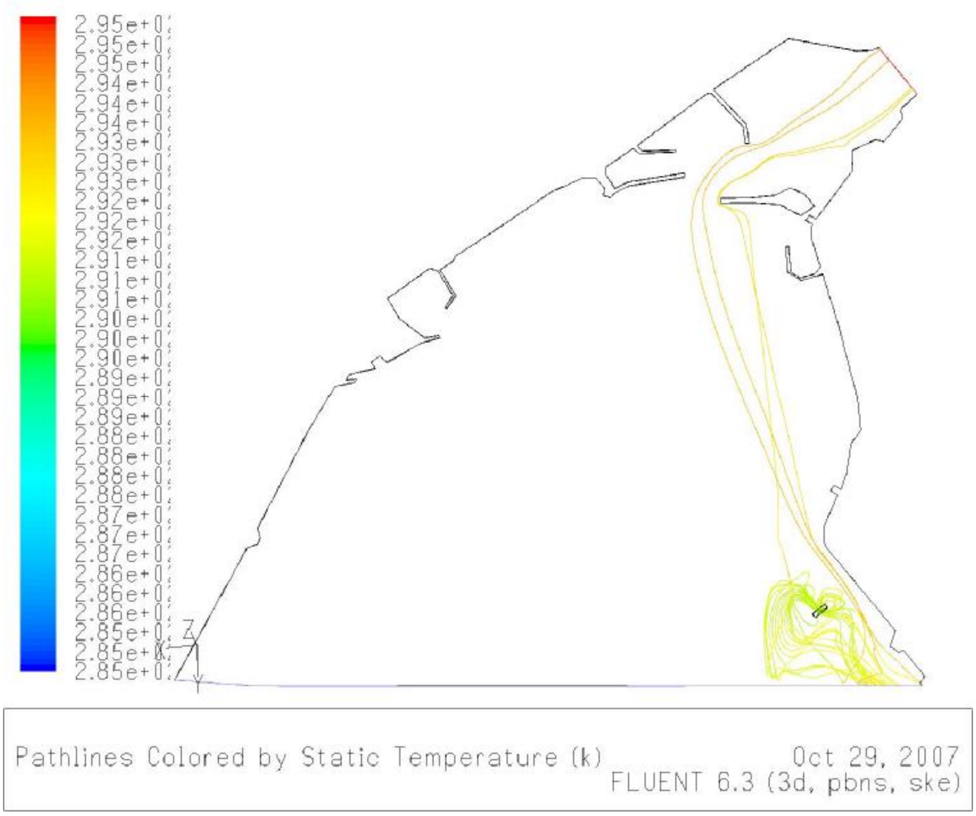


Figure 38 : Pathlines in the whole domain

The pathlines shows that most of the water from the pipe goes directly in the direction of the Mont- Blanc bridge (Rhône). Some take the opposite direction. In some cases, the particle goes up to the surface and takes the same direction as the north wind (we can check that on the print screen above).

7.8 Results with South-West wind

In this case, the difference between the temperature from the pipe (about 287.5 [K]) and the mean temperature of the lake (varies between 299 – 300 [°K]) is bigger than the last case. The other big difference in comparison to the conditions for the 26.06.2006 is the direction of the wind that has been inversed.

Velocity field:

The velocity profile for the inlet of the domain is the same than the one that was used in the other case. The velocity field near the Mont Blanc bridge does not change (due to the high values caused by the reduction of the passage section), but for the rest of the domain, the orientation of the water velocity is totally different for the surface (does not change a lot for values near the bottom of the lake).

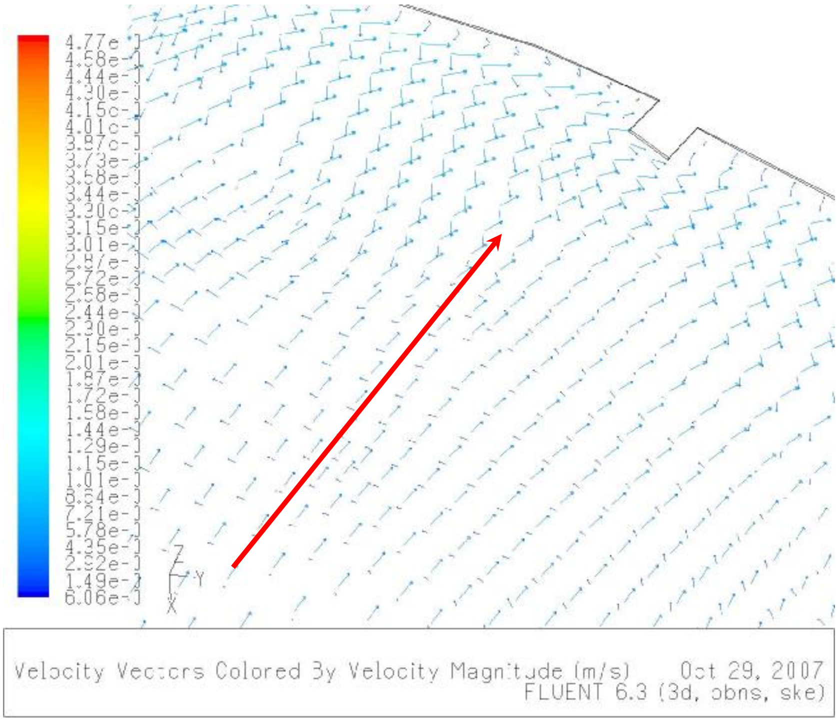


Figure 39 : Temperature field (red arrow: direction of the south – west wind)

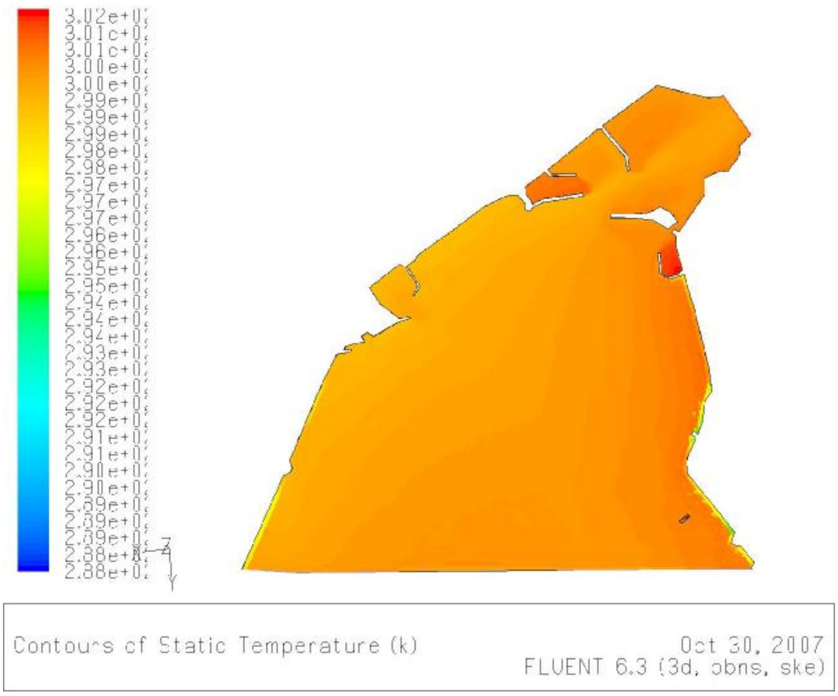


Figure 40 : Temperature field at the lake surface

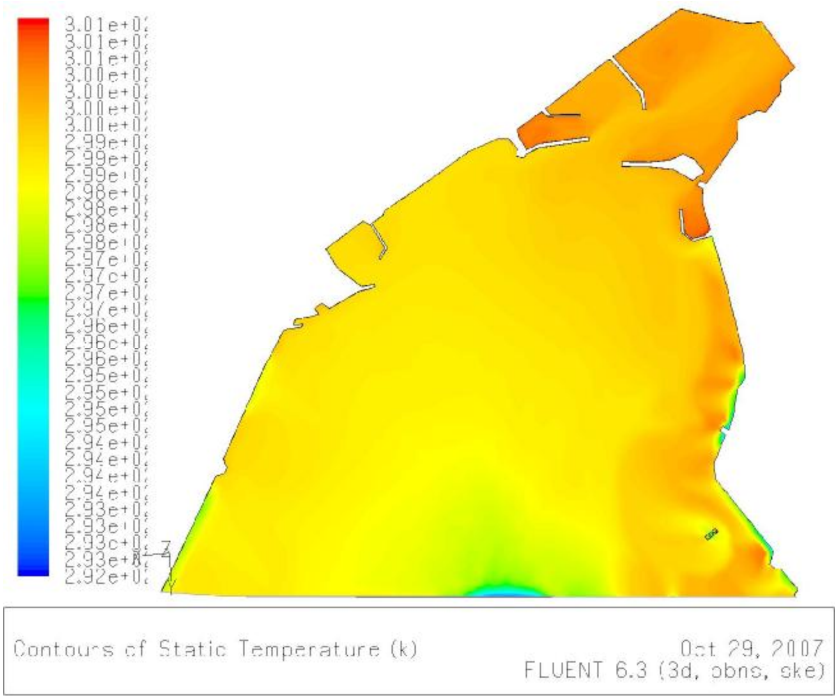


Figure 41 : Temperature field at the lake bottom

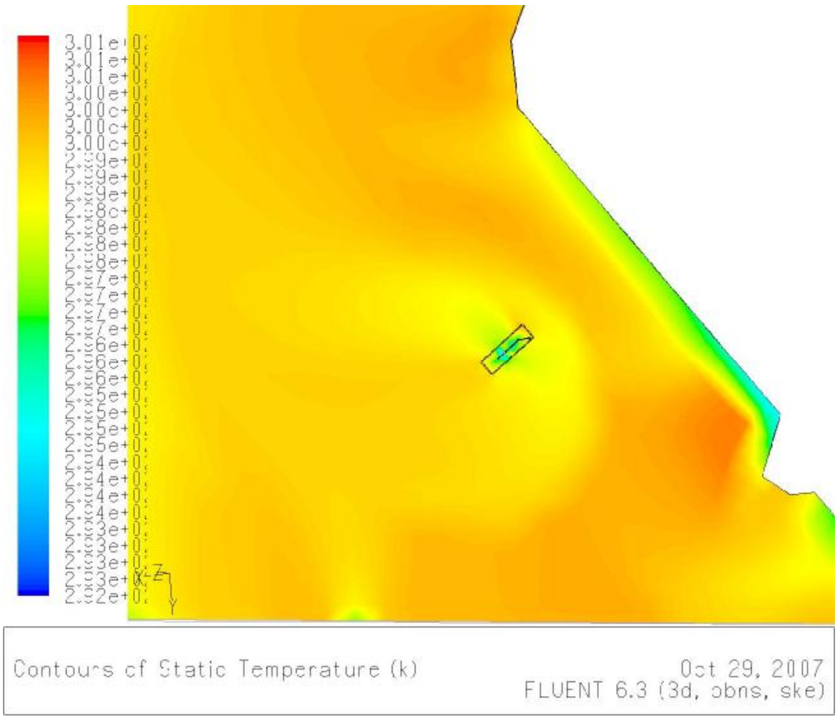


Figure 42 : Influence of the water from the pipe at the bottom lake level

The temperature field is very similar in comparison with the precedent case. The main difference is the mean temperature, which is higher than the situation in June 2006. The zoom of the pipe region on the next screenshot shows that its effect is more important, due to the temperature difference between the lake surface and the pipe inlets.

We can notice that the variations of temperature are diminishing as the lake bottom level is getting smaller. This is illustrated with the following screenshots (on planes perpendicular to the surface).

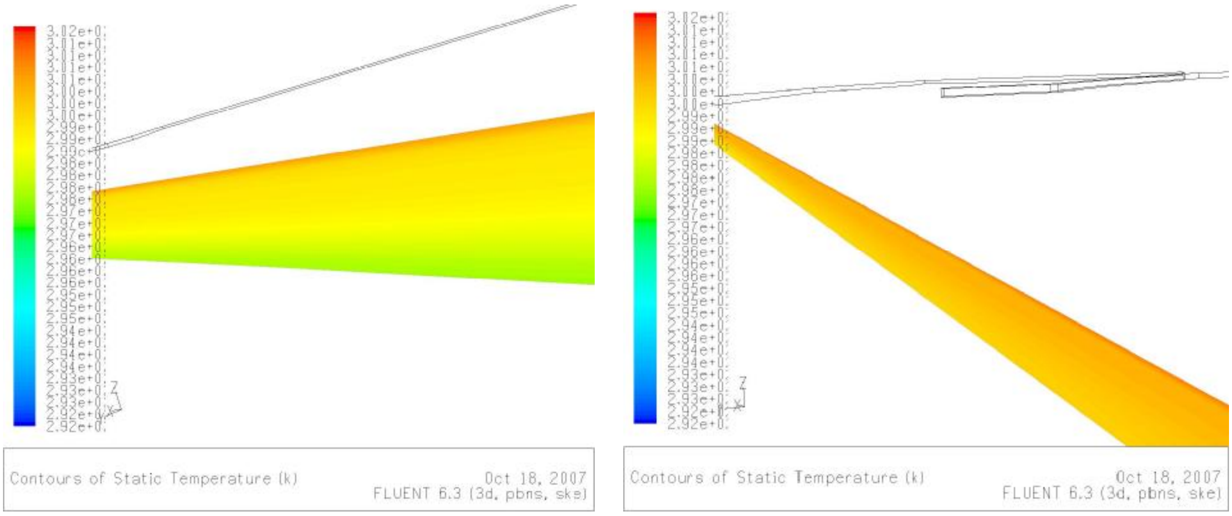


Figure 43 : Temperature field on cut views. Left: lake bottom level: -14 m; right: lake bottom level: -5 m the temperature field is more homogeneous (same scales for both pictures)

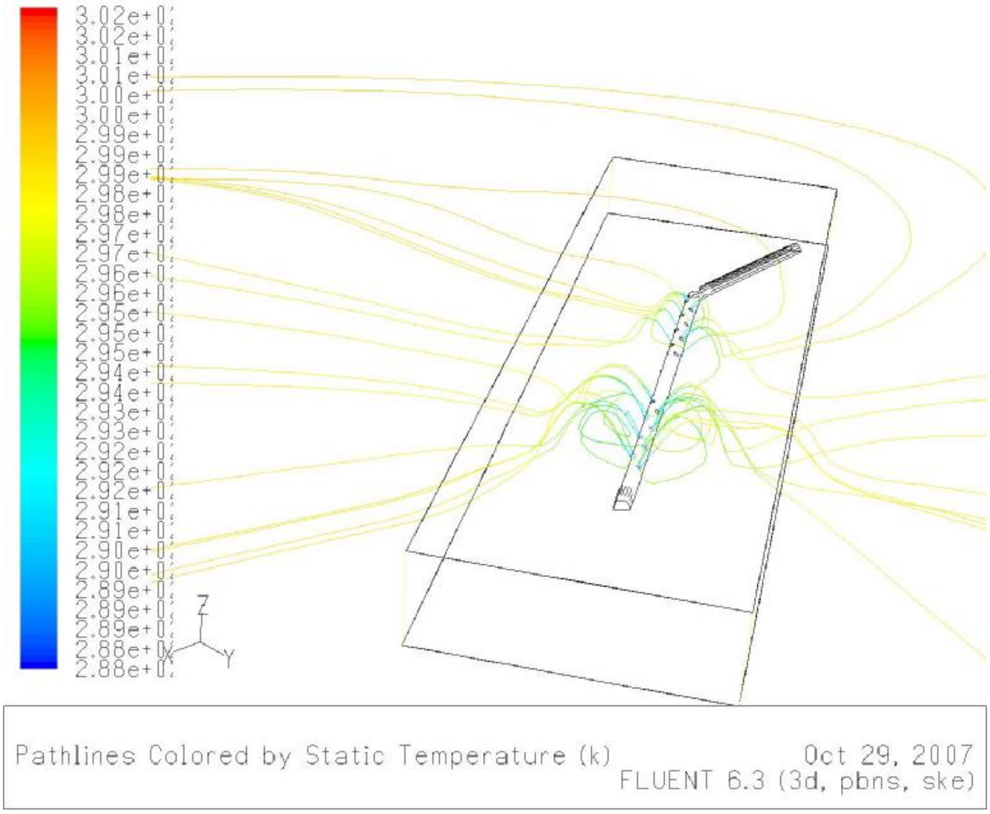


Figure 44 : Pathlines in the pipe region

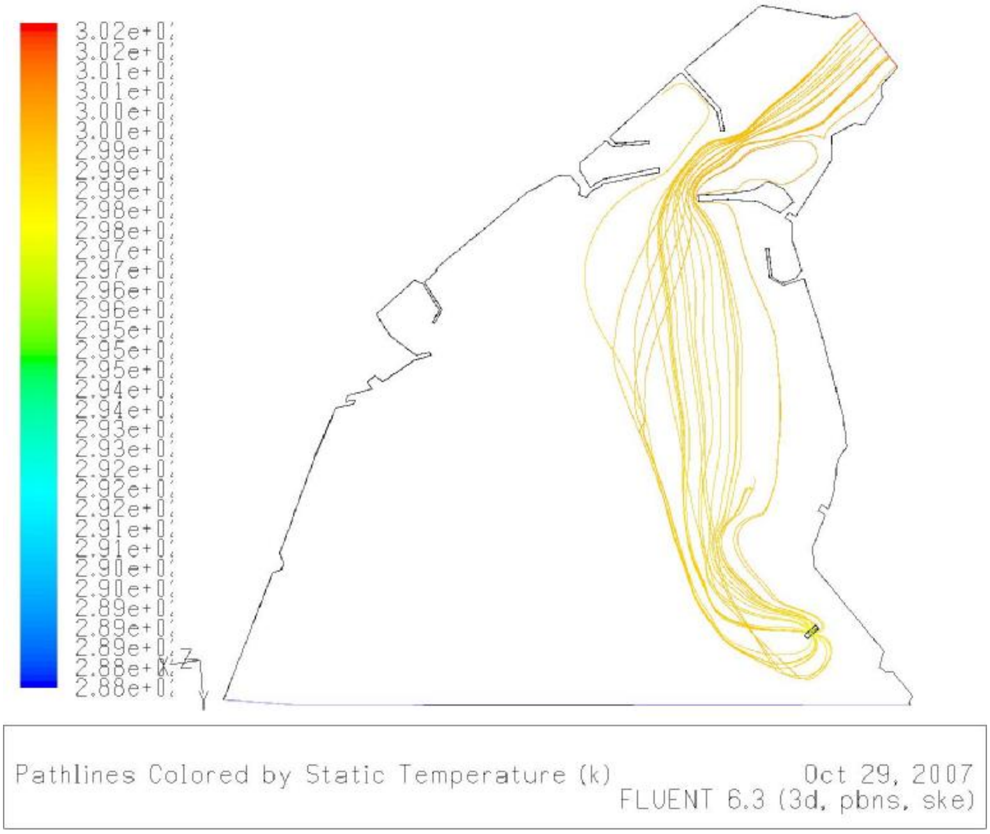


Figure 45 pathlines on the whole lake surface

7.9 Analysis and discussion

After analyzing the calculation results, it results the several following points:

- The temperature and velocity fields calculated seems are realistic (configuration of velocity vectors at the lake bottom and surface, temperature fields with gravity effects, mixing of the water from the pipe with the flow field of the lake,...).
- The effect of the velocity profile is very important for the calculation of the rest of the flow. The problem is that for this data, we only have approximate values (courantology study from 1931). There is also velocity measurements that were done at only one point (eastern direction from the TAM points) but it is not sufficient to determine a precise velocity profile from experimental data.
- The thermal effect of the water from the pipe is very local and is mixing very fast with the external flow and its influence disappears after a few meters.
- For the north wind case, the pathlines from the pipe shows that at the lake bottom level, a recirculation effects (most of the time they are going in the inlet domain direction). This corresponds to what has been observed. No similar effect is obtained with the south – west wind situation.

In conclusion, this complex model gives realist results, but we did not have sufficient data in order to validate it with measurements.

8 Case 2 : The Rhône river in Geneva

8.1 Case description

In down town Geneva the water of the Rhône is used to cool buildings, telecomunication and computer systems for a total rejected thermal power of about 30 MW. Even if this value seems really acceptable, a risk of concentration exists. The effluents are generally located near the borders of the river. We can observe an increase of the water temperature along the stream. The biggest part of the river doesn't participates and a rough estimation of the acceptable rejected power the river can accept can not be done using the flow volume of the river only. The localizations of the effluents shall be take into account.

There are now 16 cooling installations distributed on both sides of the Rhône River between the Mont–Blanc's bridge and the Seujet dam. The effect of these rejected water sources must be quantified in order that the hydro – thermal utilization of the Rhône River may be realized in acceptable conditions. For this, measurements of the water temperature upstream and downstream thermal exhausts has been used to validate a tridimensional model of the Rhône river.

In the Geneva region, all the rivers are considered being "trout" areas. The Rhône river is considered to be a river from the Mont – Blanc bridge to the Peney bridge, where begins the Verbois dam (considered to be a lake). This section is cut by the Seujet Dam, which represents an obstacle that can be considered as a new boundary condition (the mixing being very strong). Our study will be limited to the urban part of the Rhône river (between the Mont–Blanc's bridge and the Seujet Dam.

8.2 Heated effluents

The pumping and exhausts sites situations are represented on Figure 8. Data about the existing rejects in the Rhône river part are given wit the Table 1. The power which is currently injected by the thermal rejects in the Rhône River is worth about 30 MW.

8.3 Boundary conditions

The energetical flows applied on the Rhône River are the same as for the lake (case 1). The assessment of these fluxes gives a global heating or cooling effect. See paragraph 6.3 for more details.

8.4 The numerical model

The basic information from the ground of Rhône River comes from several sections map. The geometrical model is built on splines curves that fit to those sections. The ground surface is built by interpolation in-between those curves. Note that the final CAD solid B-rep model topology is quiet complex: there are a lot of holes due to the islands and the bridge pillars.

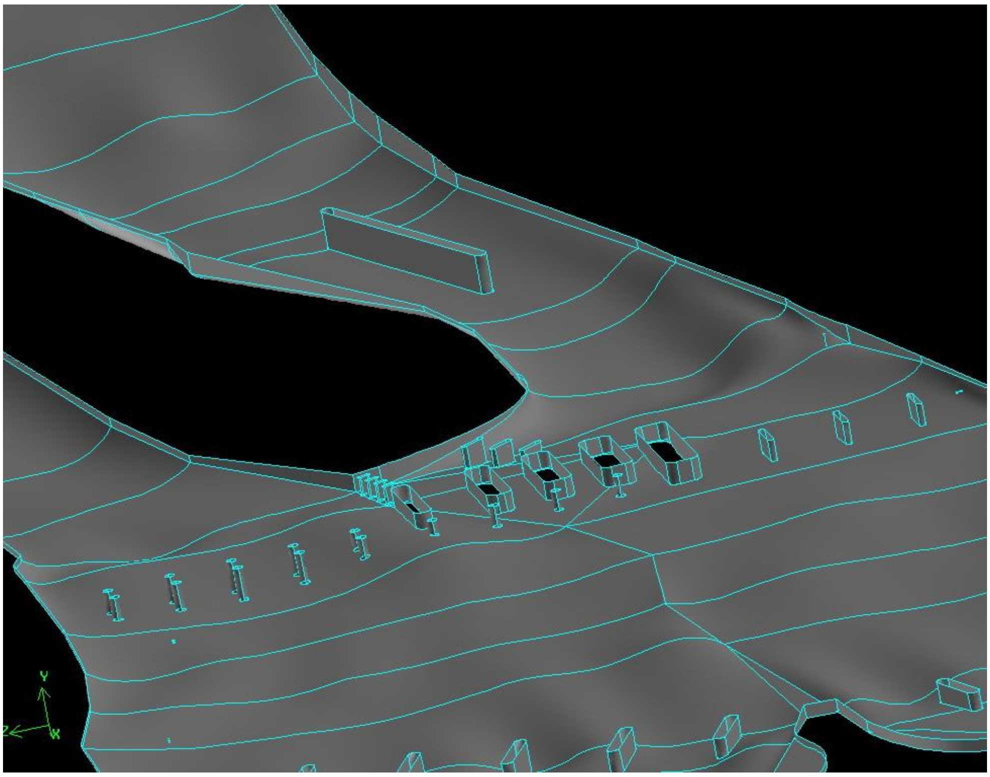


Figure 46: View of Rhône model bottom surface

A geometrical model has been built, based on bathymetric data realised by the F.A. Forel Institute. These measurements were obtained with a boat equipped with a GPS and a sonar. The cloud points are used to create cuts every ten meters. In the Seujet zone, bathymetry measurements have been realised during the dam construction in 1996. Smoothing surfaces have been created and a volume has been generated (figures 46 and 47).

The simulation model (which will be described in the next chapter) needs to discretize the domain. For this, we have generated two millions of tetraedral volumes, which sizes vary in function of the distance from the considered point to some surfaces (the bottom and borders of the river, the bridge pillars and the pipes. The dimension of the tetraedral edges varies between 0.1 and 1.5 meter. The growing ratio between two adjacent cells is not bigger than 1.2.

The calculation model is based on a three dimensions finite volumes scheme. It is described in the literature (Patankar S. 1980; Ferziger J H, Peric M. 2002). A κ–ε model with two equations and two variables is used to represent the turbulence for scales smaller than the cell dimensions. The two additional variables to the Navier – Stokes equations are the fluctuation kinetic energy κ and its dissipation rate ε. The model used has been described by Launder B. E. and Spalding D. B. and is called the standard κ–ε model. Simulations with the RNG model (which has been described by Choudhury in 1993) has also been tried.

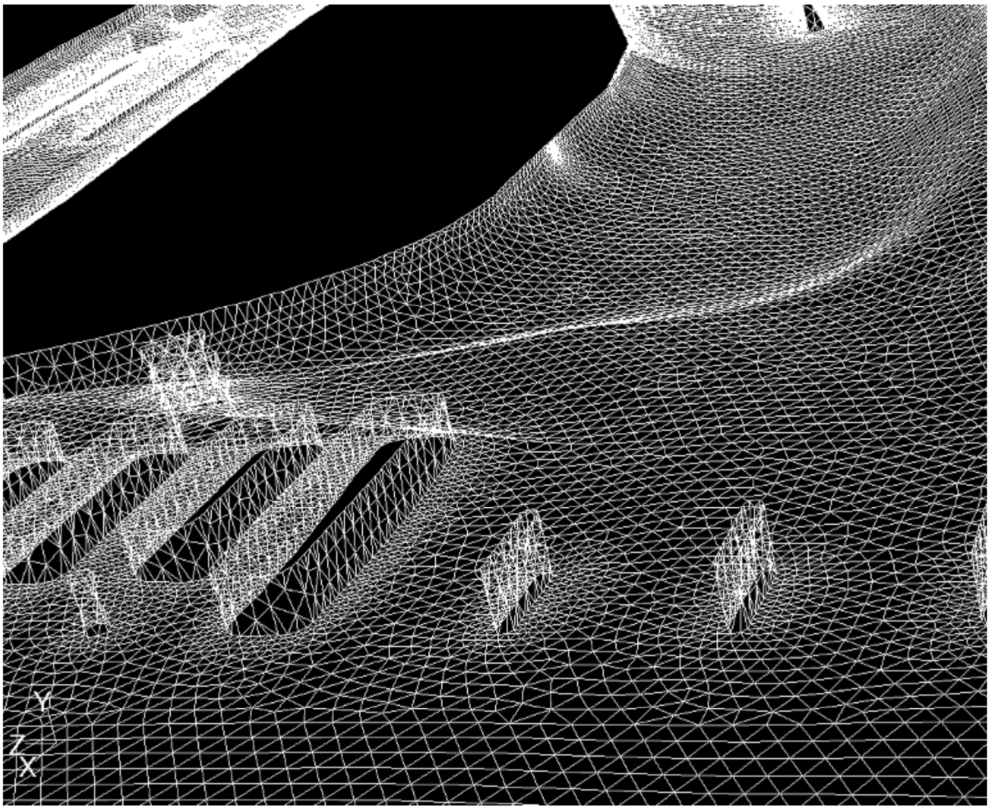


Figure 47: View of mesh near bridge pillars and the isle

The boundary conditions are assumed stationnary. A heat flux is applied to the river surface. It is calculated from the solar radiation, the convection and the evaporation (Molineaux, Lachal, Guisan 1994). The upstream (Mont-Blanc bridge) boundary is a mass flow inlet with temperature and velocity values given by the measurements obtained (Beauchamp, H. 2006).

Variable	Value	Units
Time	11h-16h	
Diffuse radiation	339.3	[W/m^2]
Incident radiation	228	[W/m^2]
Relative humidity	54.8	[%]
Air température	21.4	[°C]
Wind speed	4	[m/s]
Rhone temperature	22.9	[°C]
Heated effluent temperature	30	[°C]
Rhone turbulence level	1	[%]
Heated effluent turbulence level	5	[%]
Convection coefficient at surface	158.5	[W/m^2 K]
Rhone flow volume	126	[m^3/s]
Rhone flow speed at Mont-Blanc bridge	0.22	[m/s]
Transit time from Mont-Blanc bridge to Seujet	84.87	[min]

Table 3 : Boundary conditions for July 12th, 2005

The initial conditions are all uniform. A temperature and a velocity corresponding to the upstream are imposed values (Mont – Blanc bridge) are applied to the whole domain. Then, at the initial time, thermal loadings are imposed and the calculations begin.

8.5 Numerical model validation

Several situations have been simulated. They correspond to different days. The results presented after represent the situation of the 12 July 2005 at 10h00. This represents a typical situation that can be observed during the summer. These results have been validated using in situ measurements (Beauchamp, H., 2006).

The goal of these measurements is to validate the simulation model developed. The values measured at the Machine bridge level on the Besançon Hughe border (left side) will be used to validate the model. The table 3 summarizes the applied boundary conditions.

8.6 Results for the Rhône river

The model converges after about 12 hours calculation. Figure 48 shows the velocity field obtained at the surface level. There are zones with strong recirculations downstream the Rousseau Island, and local velocity increases near obstacles like bridge pillars.

Figure 51 shows the temperature field. It is easy to show the drags let by the effluents. Although the small temperature difference, there is fast no more thermal mixing between the rejects and the Rhône main flow.

The model has been validated on the base of experimental results. Figure 52 shows comparison between calculated and measured values for temperature profiles at the Machine bridge level. The thermal effects corresponds well to the modeling results, which gives two important information : the model gives a good representation of the Geneva urban Rhône river real thermal situation, and the effect of each heated effluent can be quantified, individually or for all the rejects.

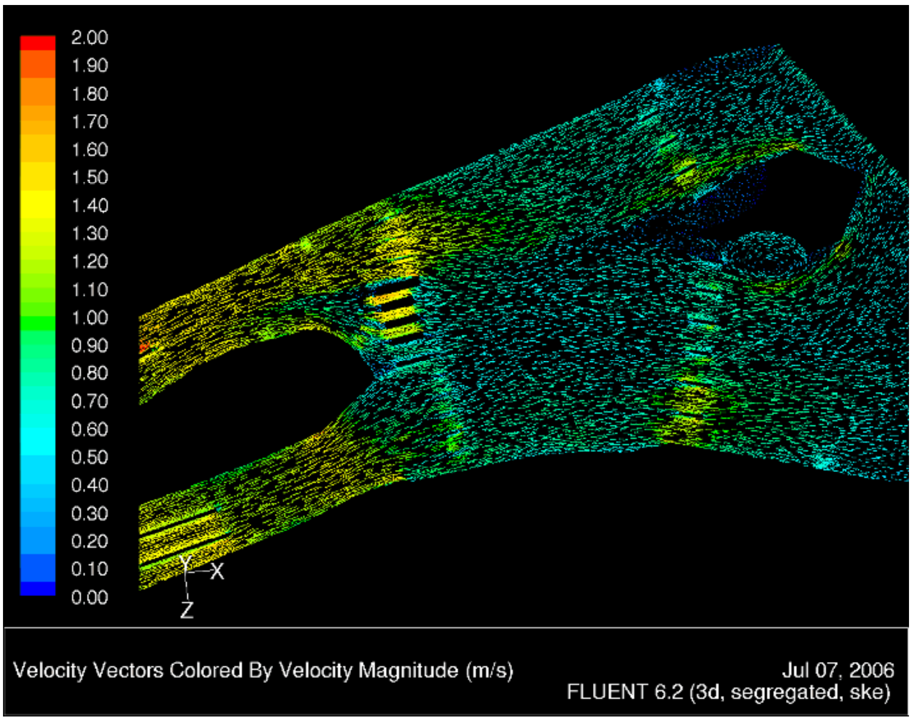


Figure 48 : Velocity vectors at the surface level (July 12th, 2005)

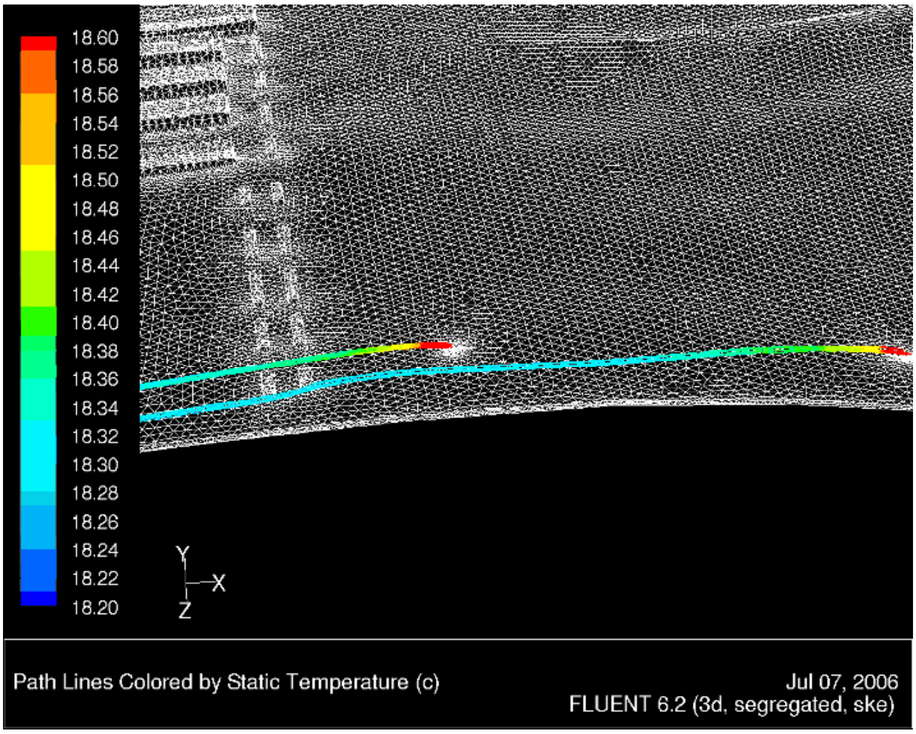


Figure 49: Rejects 39 and 41 : Reject water temperature evolution (12 July 2005)

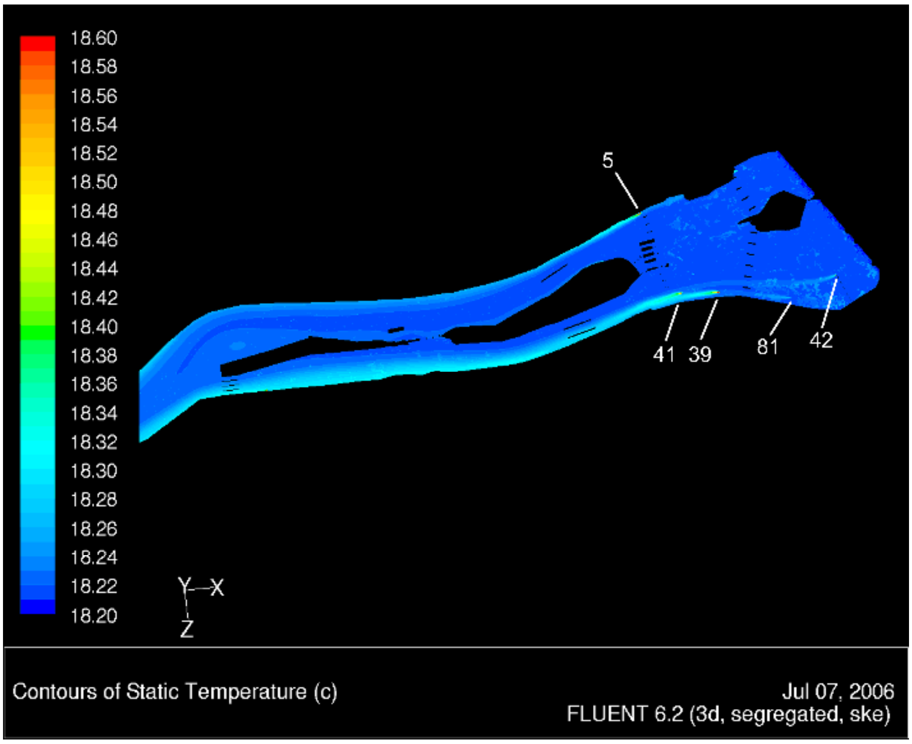


Figure 50 : Temperature field (depth 2 m, July 12th, 2005)

8.7 Analysis and discussion

In the vicinity of the exhaust point, the effluent flow and the river velocities are very different (direction and amplitude). In this region, a strong mixing exists. A few meters downstream, the situation is very different. The rejected water moves with a velocity equal to which of the river. This goes slightly up to the surface by natural convection mechanisms. Hot zones are forming and carried hundreds meters downstream.

A detailed study about the exchanges along the trajectories shows that 60 meters downstream the rejected point, the temperature reaches a value that is about 0.1 °C higher than the river (figure 54). This value is then constant for hundreds meters. The temperature gradient varies from 5 °C/m (near the outlet) to less than 0.01 °C/m 100 meters downstream. In figure 53 we can see the enthalpy evolution, we observe that 90 % of the energy exchange occur in the first 2 meters downstream the exhaust.

A study about the evolution of velocities and vorticity along the trajectories shows that the distributions of these variables are similar to which of the energy exchange. We conclude when the vorticity or an important velocity difference between the heated effluent and the river main flow exists, the mixing, and then the heat exchange is strong.

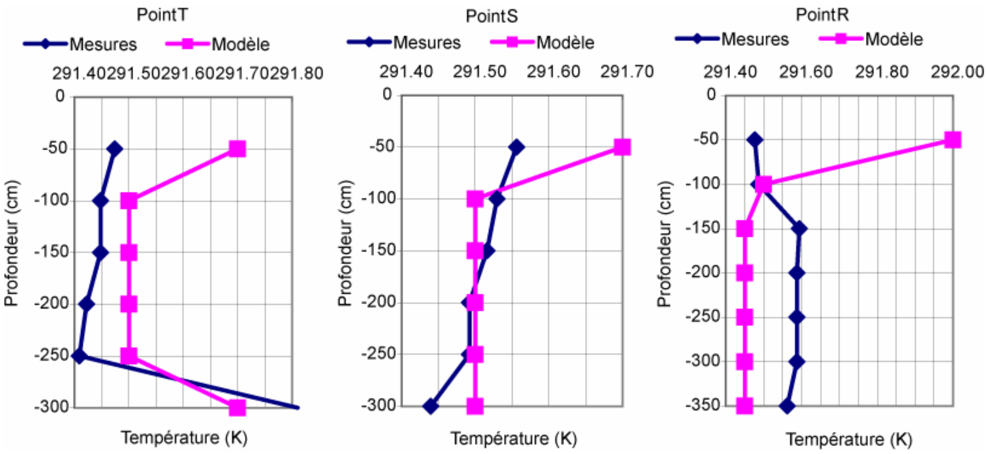


Figure 51 : Validation of the model with the temperature profiles measured at the Machine bridge level (12 July 2005) (measurements from Beauchamp, H., 2006)

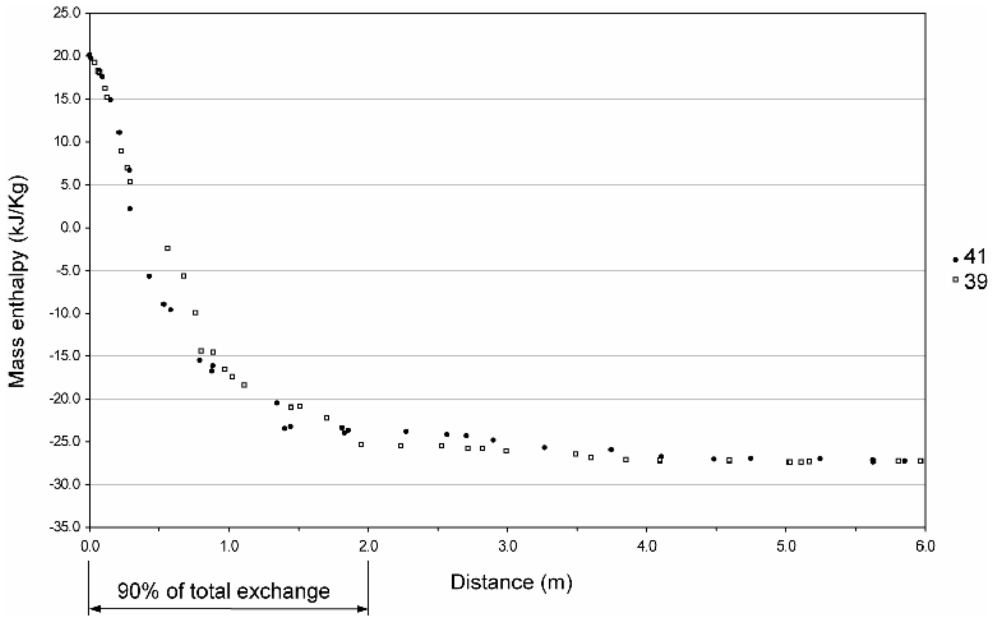


Figure 52 : Discharges 39 and 41 : Enthalpy evolution along effluent trajectories (July 12th, 2005)

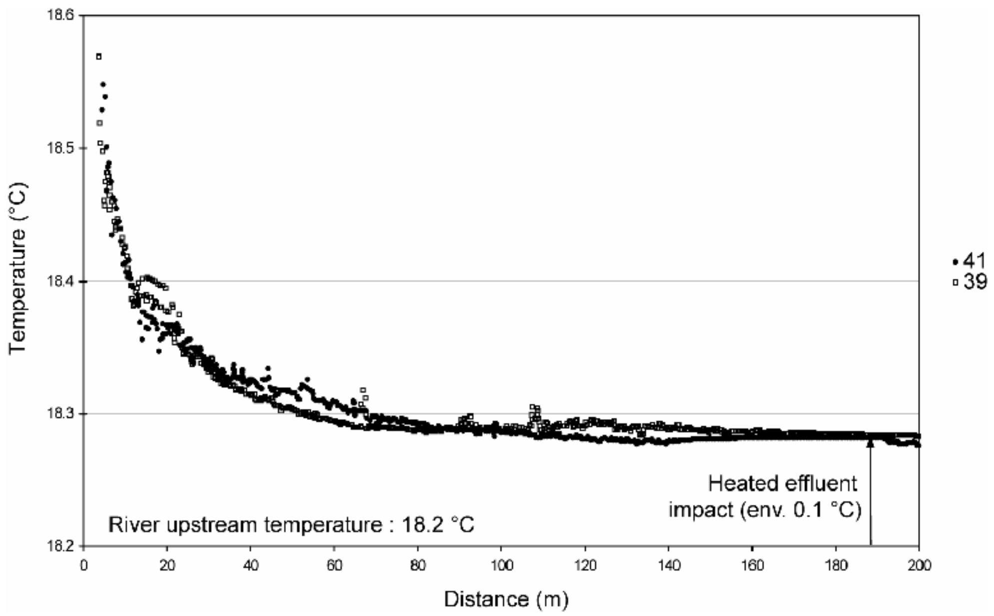


Figure 53 : Rejects 39 and 41 : Temperature evolution along effluent trajectories (July 12th, 2005)

The strong velocity difference near the exhaust involves a strong mixing. Then, the rejected water is leaded and goes up slowly to the surface. When the velocity difference between the effluent and the river water is getting smaller, we observe the volume of water being transported and exchanging quasi no energy. The temperature varies with very small gradients.

The study done by Beauchamp and al., 2006, shows that high temperature gradients are not incompatible with the biological processes existing in the river. By imposing a 0.2 °C limit for the after mixing temperature augmentation in the low flow periods (night and weekend), the nominal thermal power of the effluents could reach a value 3 times bigger than currently (about 100 MW).

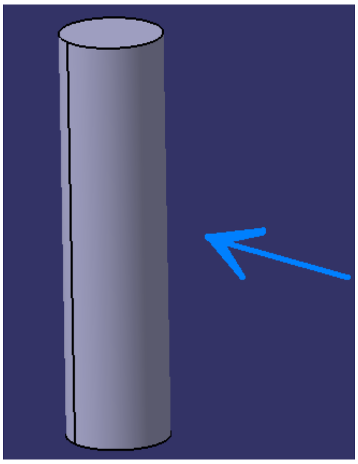
This study allowed us to show the effects of the existing heated effluents in the Rhône river. More precisely we have evaluated the mixing distances of them and we have identified the existing concentration zones.

In order to guaranty the aquatic habitation conservation, the development of the Rhône river utilization as a cold source must be integrated in a rational energy use optic. The existing authorizations are not creating pollution thermal situation in the legal meaning, although their effects are observable. Local heating could exist downstream the exhausts and accumulate along the river borders. These effects can be evaluated and minimalized with the help of the tools presented here.

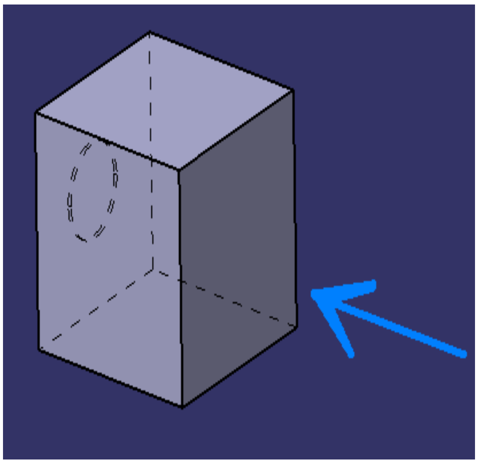
At last, an aspect to take in consideration is the temporal shift between the more ecological sensitive period (winter and spring) and the period of the exploitation of the resource for hydro – thermal purpose (summer principally). This aspect is note taken in consideration by the actual legislations.

8.8 Design of outlets for a fast mixing

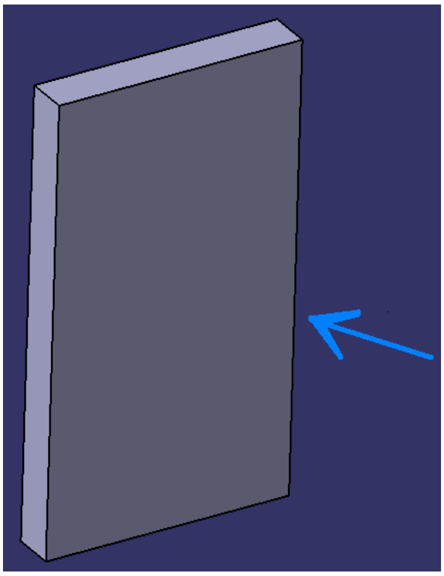
In a way to mix quickly the heated effluents with the receipting river several outlet designs have been evaluated using a CFD simulation approach similar as which described previously. Here are some outlet devices we have evaluated. These bodies have been placed upstream and downstream the pipe in order to determine which position is the most accurate.



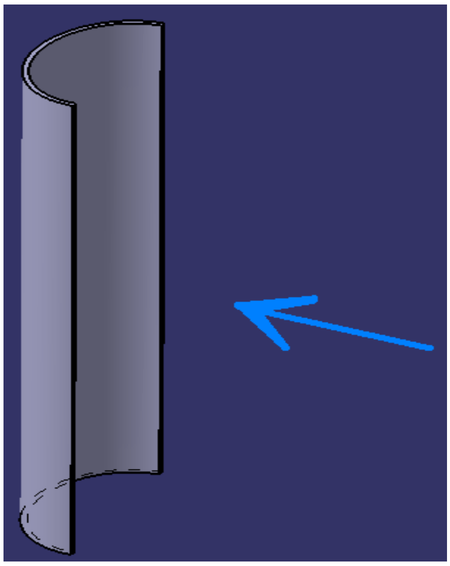
Cylinder



Cube

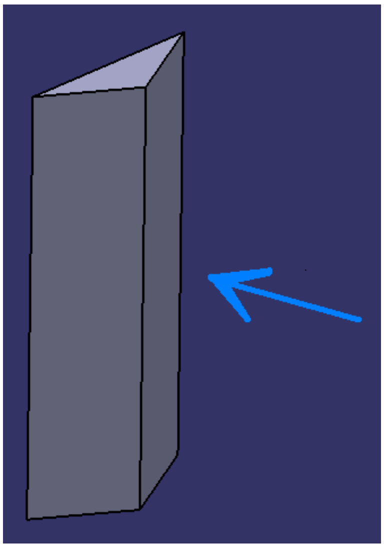


Flat plate



U

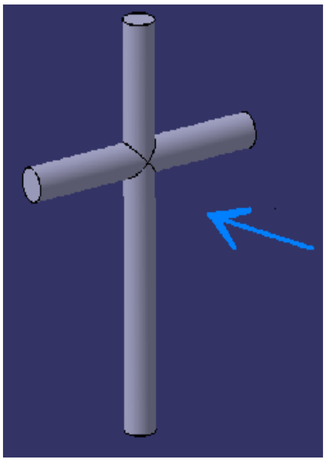
Figure 54 (a) : Several obstacles configuration used is to mix the flow in the reject area



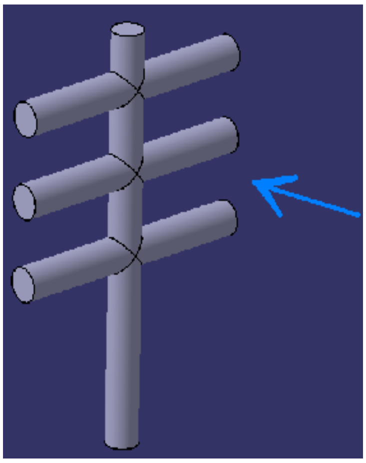
Prism

Figure 55 (b) : Several obstacles configuration used is to mix the flow in the reject area

With a geometry obtained by the use of several simple bodies, it is possible to obtain a higher or asymmetric flow perturbation. Here are several configurations that have been evaluated.

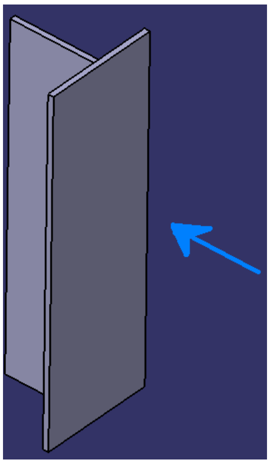


Cross

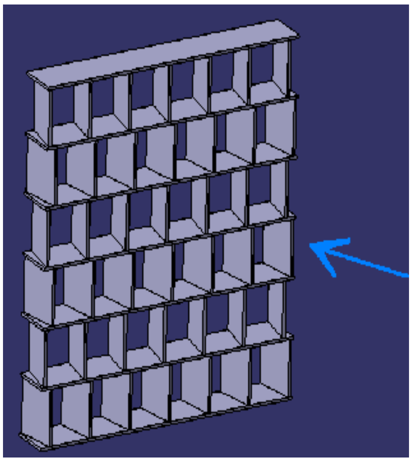


Multi cross

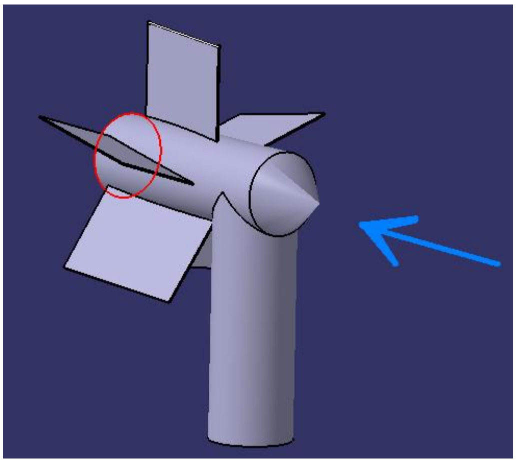
Figure 56 (a) : Other more complex geometries



2 Perpendicular planes



Grid



Propeller

Figure 57 (b) : Other more complex geometries

In this case, the rejected water comes from the rear of the propeller (red circle on the previous picture). It is possible to vary the number of blades and their incidence angle.

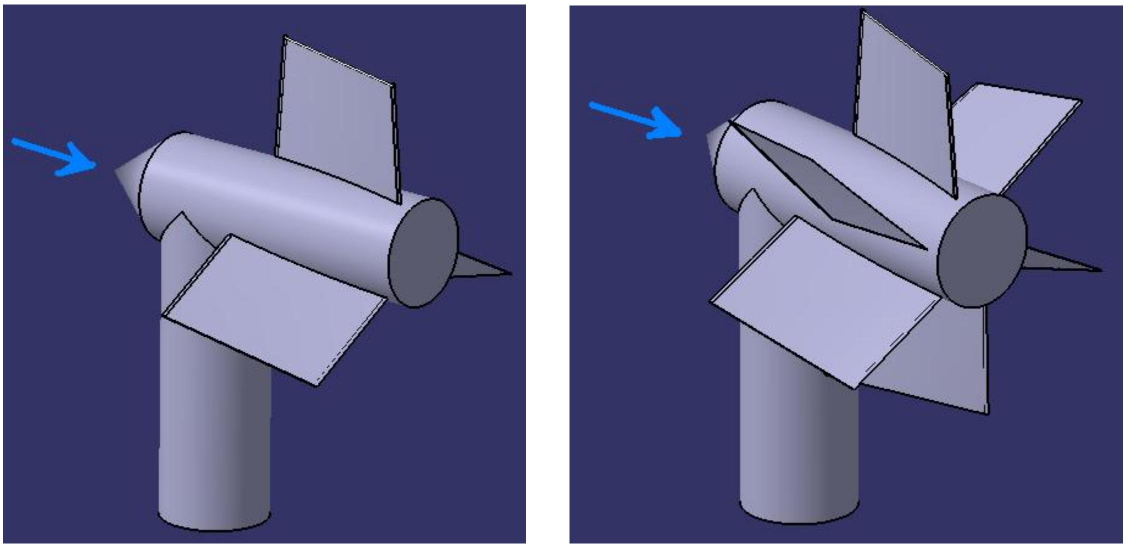
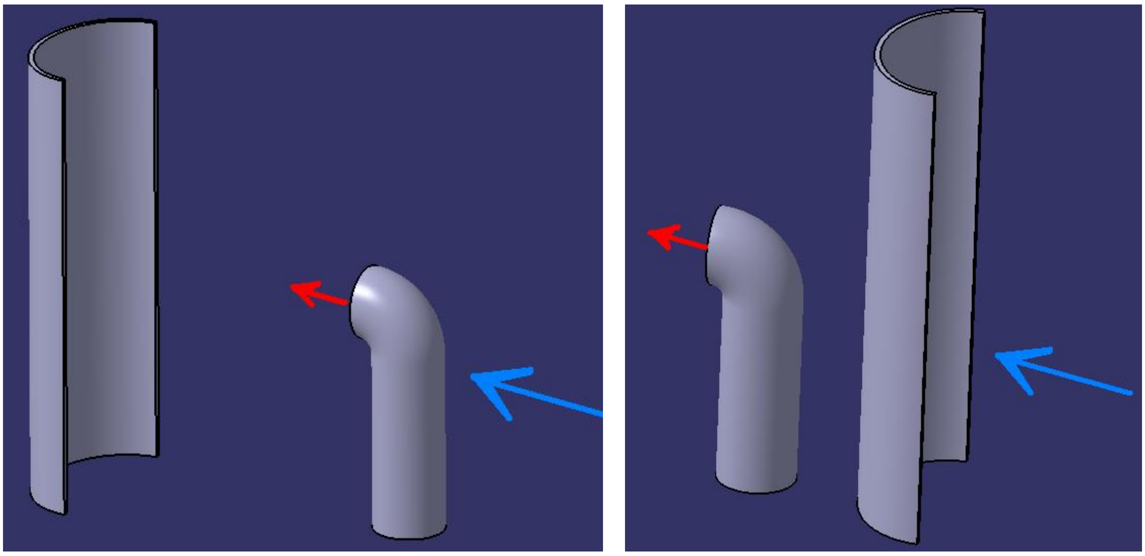


Figure 58 : Two types of propellers

In our case, we chose to orient the reject field in the same direction than the river flow. Wit this configuration, it is easier to observe the influence of the perturbation from the obstacle. Then, as said before, we had choice to put the obstacle downstream or upstream the pipe.



Obstacle placed downstream **Obstacle placed upstream**

Figure 59 : Mixing devices with obstacles placed upstream or downstream

After having modeled the simulation model, the pipe inlet is imposed to a different temperature than the river flow, so it is possible to check which configuration presents the better mixing accuracy.

Qualitative comparisons have been done to determine which types of devices were the most efficient. We have observed that the U plane and the 2 perpendicular planes configurations were the ones we tested who present the better flow mixing.

9 Conclusion

In this project we have defined a methodology based on

- Bathymetry measurement from echosounding
- Spline modeling
- CFD techniques

We have shown this methodology represents an efficiency tool in the prediction of temperature and velocity fields. These can then be used to analyze the impact of cooling system on the environment. Mixing lengths, temperature distribution and water speed can be analyzed and compared with the requirements of local legal policies.

Two cases have been studied as examples : The Leman lake GLN project and the Rhone River in Geneva region. For the Leman Lake, the temperature distribution and the particle trajectories have been calculated. The best location for the exhaust can then be choose based on simulation before the project start. In the case of the GLN project, the results obtained in WP 2.4 will be compared to which obtained in this WP.

For the Rhône River, the results have been validated by measurements taken by the CUEPE from Geneva University. The method shows an impact of the heated effluent along the river borders. The majority of them are located along the borders and their impacts are cumulated. We have shown also 90% of the energy exchange between the heated effluent and the main flow occur in the 2 meters downstream the exhaust. Then a temperature difference of 0.1 °C exists on distances higher than 1 km.

References

BEAUCHAMP H. 2006. Heated effluents and the Rhône River in Geneva : Thermal pollution or sustainable development ?, Master en sciences naturelles de l'environnement, Centre universitaire d'étude des problèmes de l'énergie CUEPE, Université de Genève, mémoire n° 140.

BEAUCHAMP H., BEETSCHEN C., HAAS P., LACHAL B., VIQUERAT P. - A., PAMPALONI E. 2007. Heated effluent and the urban Rhône in Geneva - Measurements and simulations, Archives des sciences, SPHN, Geneva.

BEETSCHEN C. 2005. CARMEN - Un modèle du Rhône pour la gestion des rejets thermiques, travail de diplôme, Groupe de compétences en mécanique des fluides et procédés énergétiques CMEFE, filière de génie mécanique, University of applied sciences of western Switzerland HES-SO, Genève.

BETANT A., PERRENOUD G. 1931. Etudes sur la partie occidentale du lac de Genève - Courants et températures, Mémoires de la société de physique et d'histoire naturelle de Genève, Vol. 41, facsicule 2.

BINYET E. 2006. TetraEner - Geneva sub-project "Genève-Lac-Nations", travail de diplôme, Groupe de compétences en mécanique des fluides et procédés énergétiques CMEFE, filière de génie mécanique, University of applied sciences of western Switzerland HES-SO, Genève.

BINYET E., SCHWEIZER C., WITSCHARD M. 2006. Projet Carmen et Genève-Lac-Nations, projet annuel, Groupe de compétences en mécanique des fluides et procédés énergétiques CMEFE, filière de génie mécanique, University of applied sciences of western Switzerland HES-SO, Genève.

CERTEJAN T. 2003. Suivi des apports thermiques dans le Rhône genevois, Rapport interne, Service cantonal de l'écologie de l'eau DomEau SECOE, Etat de Genève.

CH 1998. OEaux - Ordonnance sur la protection des eaux, RS 814.201. Swiss laws, Bern.

CIPEL - COMISSION INTERNATIONALE POUR LA PROTECTION DES EAUX DU LEMAN CONTRE LA POLLUTION. 2004. Rapports sur les études et recherches entreprises dans le bassin lémanique, programme quinquennal 2001 - 2005.

CHOUDHURY D. 1993. Introduction to the renormalization group method and turbulence modeling, Fluent Inc. technical memorandum TM-107.

FERZIGER J. H., PERIC M. 2002. Computational methods for fluid dynamics, 3rd ed. Springer, Berlin.

FERMBAECK G. 1995. District cooling in Stockholm using sea water, proceeding of the 10th annual cooling conference, International District Energy Association.

HUTCHINSON G. E. 1957. A treatise on limnology, John Wiley & Sons Inc., New York.

LAUNDER BE, SPALDING DB. 1972. Lectures in mathematical models of turbulence, Academic Press, London.

MOLINEAUX B, LACHAL B, GUISAN O. 1991. Analyse thermique de cinq piscines avec capteurs solaires, Rapport interne, Centre universitaire d'étude des problèmes de l'énergie CUEPE, Université de Genève.

MOLINEAUX B, LACHAL B, GUISAN O. 1994. Thermal analysis of five outdoor swimming pools heated by unglazed solar collectors, Solar Energy, Vol 53, pp 21-26.

PATANKAR S. 1980. Numerical heat transfer and fluid flow, 1rst ed. Hemisphere Publishing Corporation, New York.

SANTAMOURIS M, ADNOT J, ALVAREZ S, KLITSIKAS N ET AL. 2004. Cooling the cities / Rafraîchir les villes. Ecole des Mines de Paris, Paris.

SECOE - SERVICE CANTONAL DE L'ECOLOGIE DE L'EAU DOMEAU 2004. Rappel succints des conditions à respecter concernant le rejet des eaux utilisées à des fins hydrothermiques, Etat de Genève.

STEARNS & WHELER LLC 1998. Draft Environmental impact statement, lake source cooling LSC, Cornell University, New York.