Summary

Some papers dealing with the modelisation of injection of cement fluids in cavities like vertebras for the vertebroplasty and kyphoplasty, have already been publicized. Previously, advances made in the matter were the basis of the Level Set. But those projects done on the software called COMSOL multiphysics, did not try to simulate real injections in cavities.

This paper goes further on that way. Even if the experiments could not be done with real alpha-TCP, a viscous liquid was used to simulate similar conditions to the ones experienced in vertebroplasty and kyphoplasty. These results have been compared to those given by the numerical problems computed on COMSOL and it has been shown that the computational power provided by actual personal computers, is nowadays enough to obtain results approaching reality. This paper explains also how to use the Level Set mode on numerical software, and explains also the signification of the results obtained.

Moreover, an opening is done by introducing the porous equations that govern the fluid flow in porous medias. To approach the real situation of vertebroplasty and kyphoplasty, a fluid melted with sand was injected to little pieces of sponge to see the distribution of sand inside the sponge porosity and the pressure constraints.

Please note that video supports and pictures accompany major part of the chapters. They are available on the annex A of the DVD.

Table of contents

1. Glossary

PFC: Proyecto final de carrera, Spanish expression for the term undergraduate project

IS14: Injection Station with fourteen cavities, this experimental tool has been fabricated during the PFC in order to obtain experimental data. (For more information see p.39)

CMC: Carboxymethyl Cellulose (CMC) is a cellulose derivative. Very expanded in the food industry with the intention of thicken the water containing food as puree vegetables. (For more information see p.43)

2. Preface

2.1. Origin of the project

This project is consecutive to the PhD thesis of Maria Daniela Vlad entitled "New development in calcium phosphate bone cements approaching spinal applications" **[1]** issued in December 2008 under the supervision of Prof. Dr. Enrique Fernandez Aguado for the department of Materials Science and metallurgical Engineering of the Universitat Politecnica de Catalunya (UPC). At the end of her PhD thesis, Daniela suggested that researches should be done on the fluid diffusion in the vertebras. Two students Solenn Laforgue in her PFC "Aproximación computacional al estudio de la inyectabilidad de cementos óseos" **[2]** in February 2008 (ETSEIB) and Amin Gargouri in his PFC "Aproximación computacional al estudio de la inyectabilidad de cementos óseos y medidas ultrasónicas" in February 2009 **[3]** (ETSEIB) have already worked on flow simulations with the software named COMSOL using the Level Set method which is the computational mode.

2.2. Motivation

The purpose of this project is at term, to model a flow of cement comporting micro particles for kyphoplasty and vertebroplasty which are occurring in a bone, a micro porous material. **[4] [5] [6] [7]**

2.3. Pre-requiresites

It is necessary to have a global knowledge about fluids mechanic and finite elements modeling. It is also necessary to have a global knowledge about vertebroplasty, kyphoplasty and biocements. **[4] [5] [6] [7] [8] [9] [10]**

3. Introduction

3.1. Aim of the project

Our goal was to characterize different kind of fluid flows in order to: in one hand determine the conditions of validity of a different kind of simulations with COMSOL and on the other hand to get experimental results about flows in porous materials. **[10] [11] [12] [13] [14] [15]**

3.2. Protocol of the project

The protocol was as follow:

a) The first step was starting to work with the software COMSOL to create computational models and improve the results by using with efficiency all the options offered by the Level Set mode of the software.

b) Second step, the computational model of a macroscopic cavity (empty or comporting one obstacle) filling with a single phase fluid has been compared to a similar experimentation. It has been determined in which measure the results were coherent and possibly improvable.

c) Third step, the experimentation of a macroscopic cavity filling (comporting two inputs and obstacles creating a multi path web) with a single phase fluid has been compared to a similar computational model. The purpose was to simulate a complex flow and it has been determined in which measure the results were coherent and possibly improvable.

d) Finally experimentations have been tried on cavities full of sponge. Filling has been provided with fluid and fluid comporting sand. Particular observations have been made on the pressure curves and on the dispersion of sand grains in the sponge.

4. Overview of COMSOL handling

The following chapter is an overview of the handling of the software COMSOL Multiphysics for beginners. **[12] [16] [17]** (see also Annex B: COMSOL video tutorial)

4.1. File opening

Space dimension: when opening the software, you may choose between linear, planar or volume work. In our case, when modeling a fluidic flow, we worked particularly in 2D, but also in Symmetrical 2D and 3D. The axial symmetry implicates the presence of a symmetrical axis on your draw sketch.

Model choice: you can open a new project, specifying the type of model you wish. In our case: MEMS Module, Microfluidics, Two-phase flow, Laminar, Level Set.

You may also open an existing model (Model library), a model you created before (Open) and choose some language and units options (Settings).

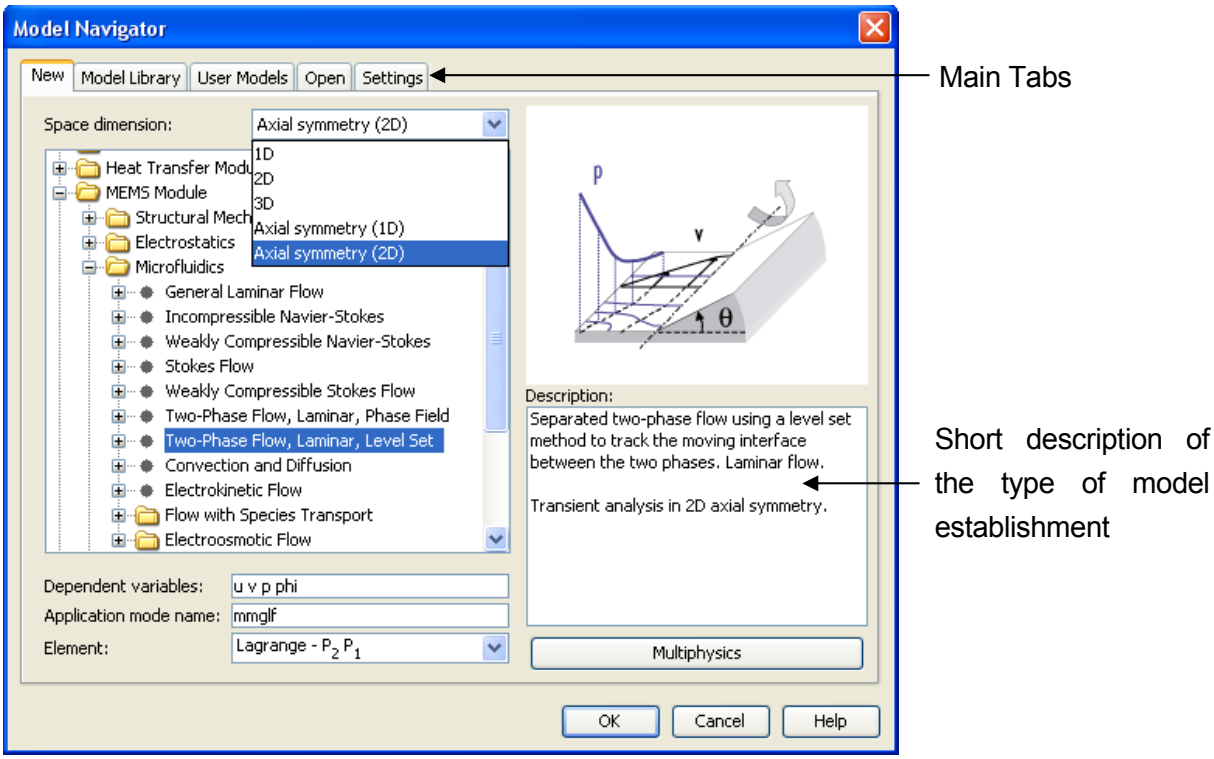


Fig.4.01. COMSOL opening window

Click OK and you will be directed to the main interface.

It opens directly on a typical draw interface with draw tools, zooms and a model tree on the left.

4.2. Drawing

You may draw your model; define size, volumes, etc. In our example a long path representing a needle arrives on an empty volume CO2 and a boundary B1 has been created to represent the initial fluids interface.

The DRAW tab on the top contains some other draw options and also the OPTIONS tab permits to modify the axes and grid settings.

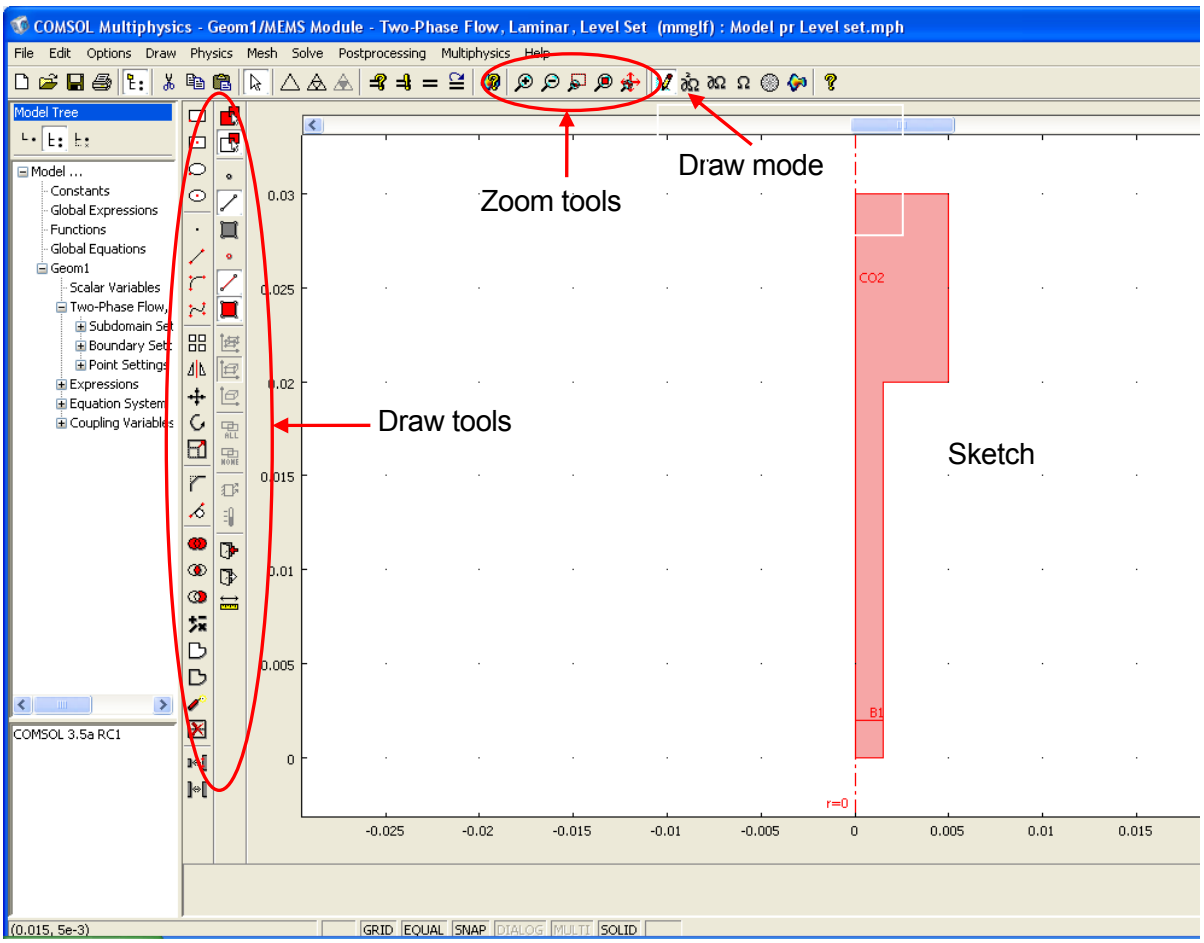


Fig.4.02. Draw interface on COMSOL

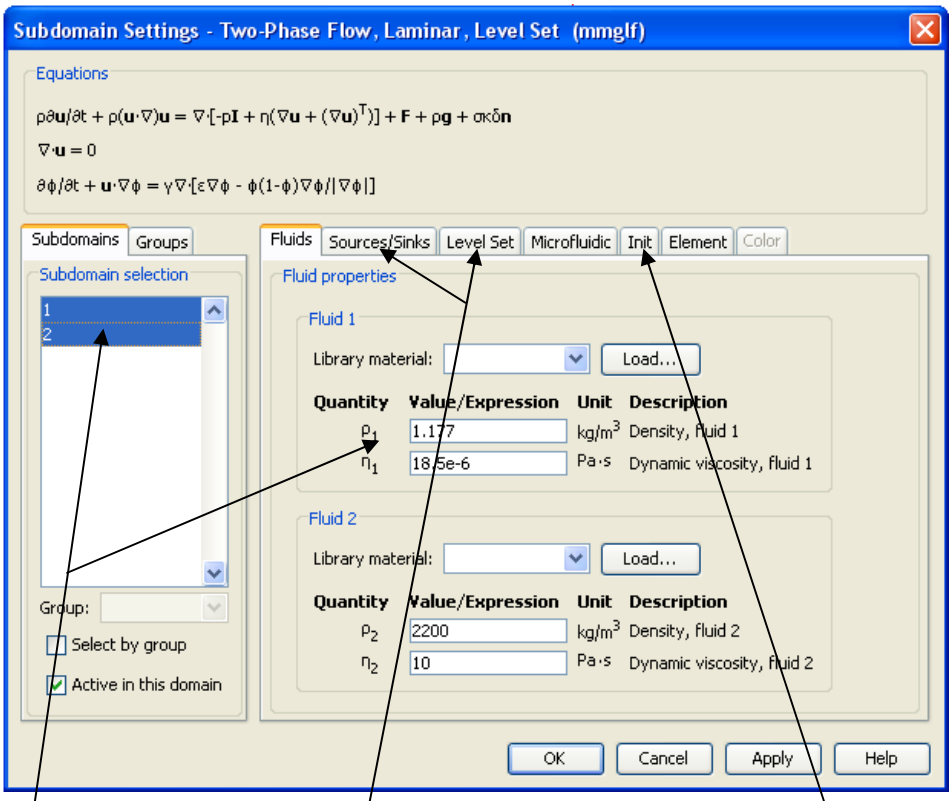
Note: this piece has been drowned by: drawing a first rectangle, adjusting the size, adding a second rectangle, adjusting the size again, selecting both pieces and creating a union automatically called CO2, suppressing interior boundaries, and at last adding a line automatically called B1.

The software will automatically recognize the existence of two domains.

After drawing your piece you can enter the properties of each area.

4.3. Physics settings (subdomain and boundary settings)

Click the tab PHYSICS, the two main parts you have to inform are: Subdomain settings and Boundary settings.

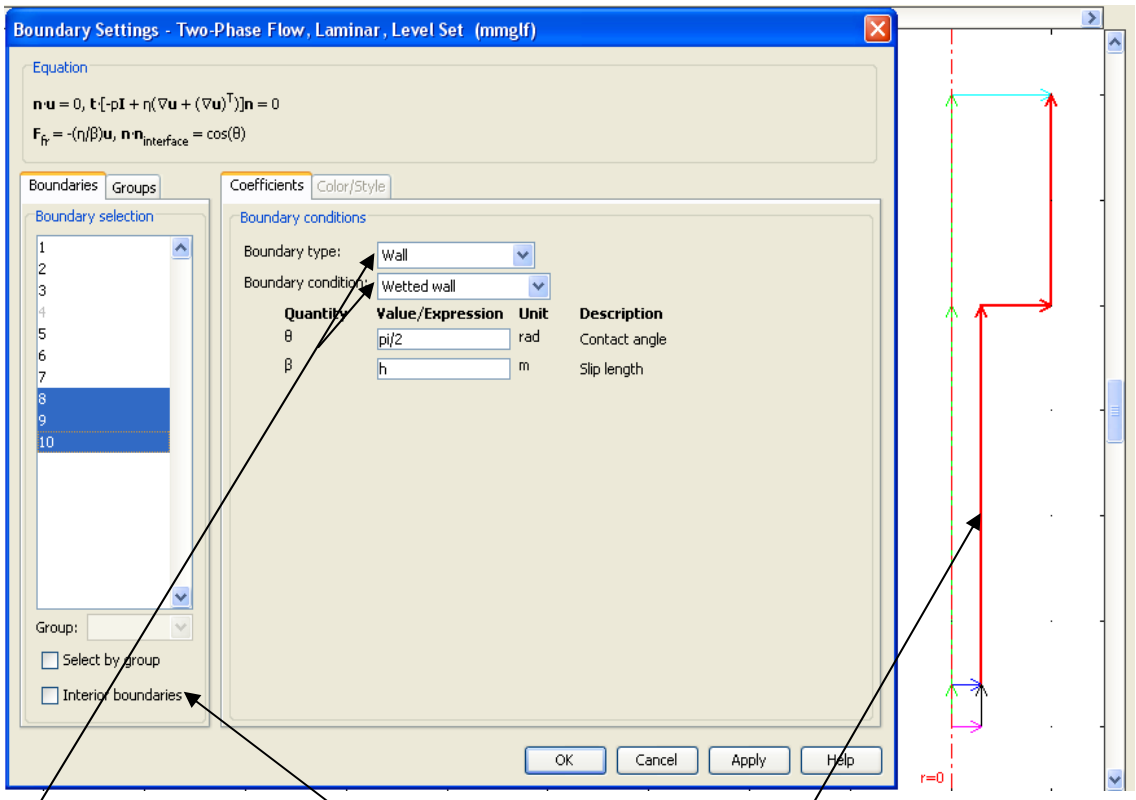


1/ Select for both subdomains the fluid properties

2/ The Sources/Sinks and Level Set tabs and their specific values are described in detail in a following part

3/ Select for each subdomain the initial present fluid

Fig.4.03. Subdomain settings window



1/ Describe for each boundary the type (wall, inlet, outlet, symmetry boundary, etc) and condition

2/ Click Interior boundaries and define the condition: initial fluid interface at the needed place

Note: you may control on your draw which line (or surface if you are working in 3D) is affected to which number. Moreover you may select many bounds in the same time

Fig.4.04. Boundaries setting window

4.4. Meshing

When subdomain and boundary settings are defined, you may mesh your piece. Meshing does not require any convergence study. However the form and quantity of elements will of course have an influence on the quality of your results but also on the rapidity of computation. You have to find a compromise.

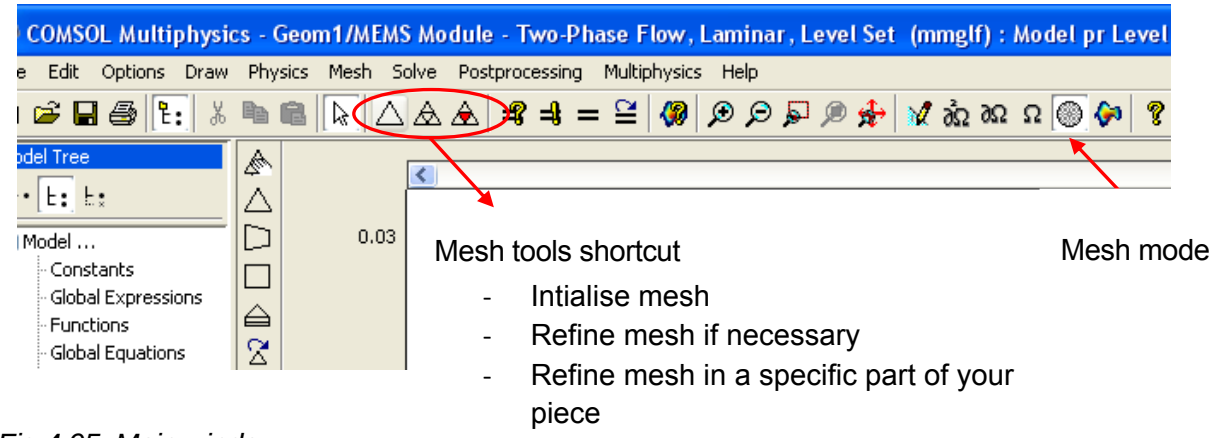
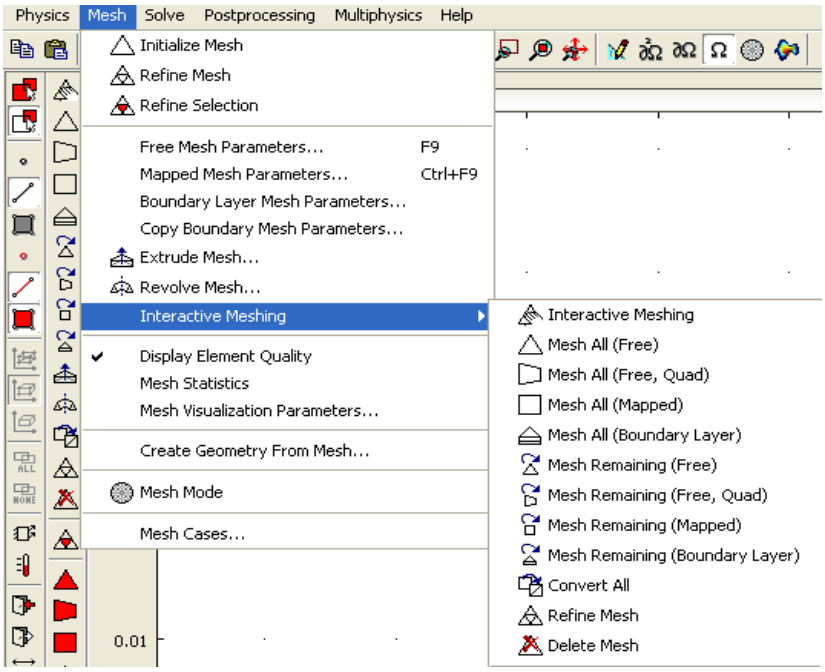


Fig.4.05. Main window

Fig.4.06. our 300 elements piece made by using the free mesh parameters, predefine mesh normal size: (See above)

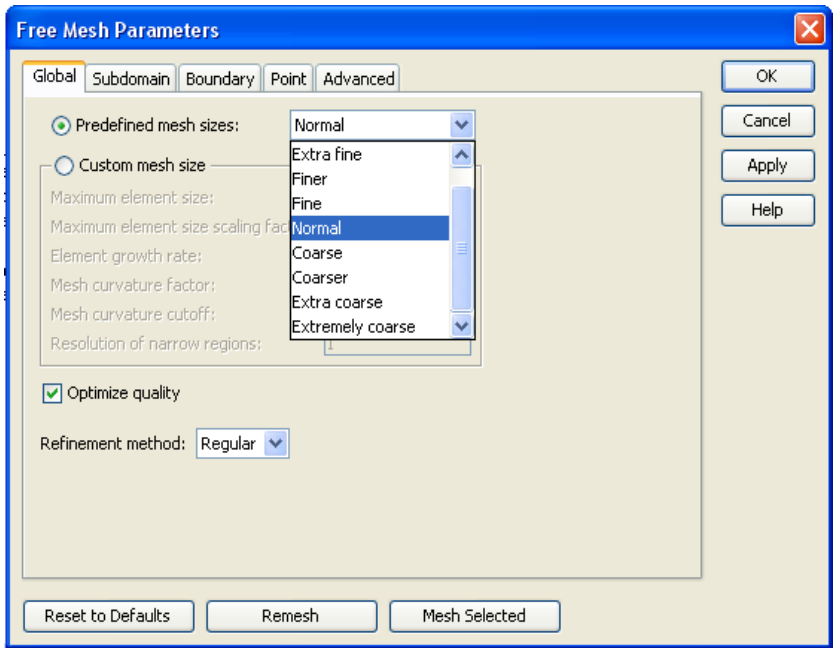
You may also use the MESH tab to get more options:



You will get options to define the geometry of your mesh, the possibility to mesh partially your piece, etc…

Fig.4.07. Detail of the Mesh tab

At last you can use the Free Mesh Parameters icon:



With this tab you may select a predefined mesh size and apply it on your draw by clicking Remesh.

You may also define the number of bounds that you wish to have on each boarder or interface.

Fig.4.08. Free Mesh Parameters window

4.5. Computation

You can now start the computation of your model. The type of computation will depend on your simulation.

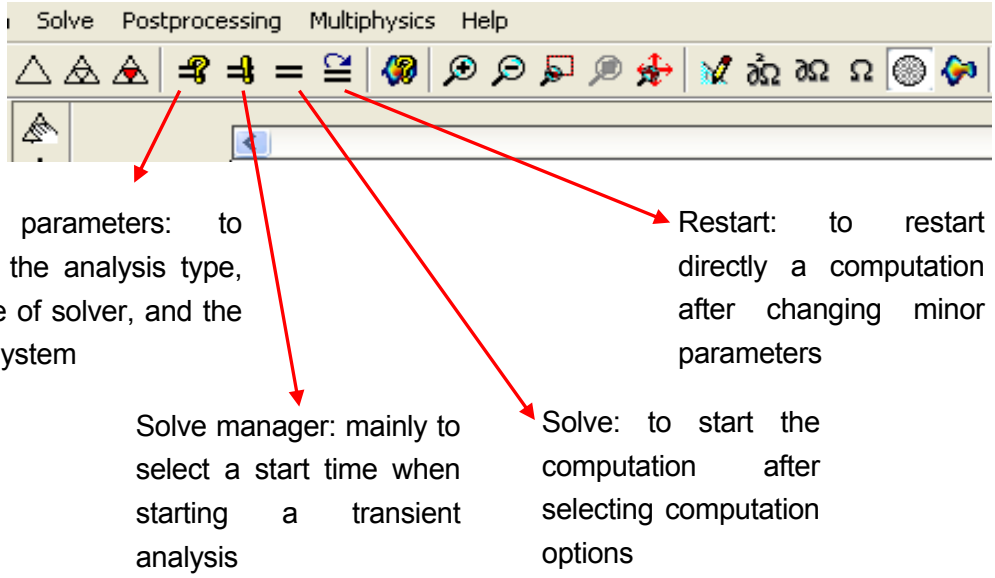
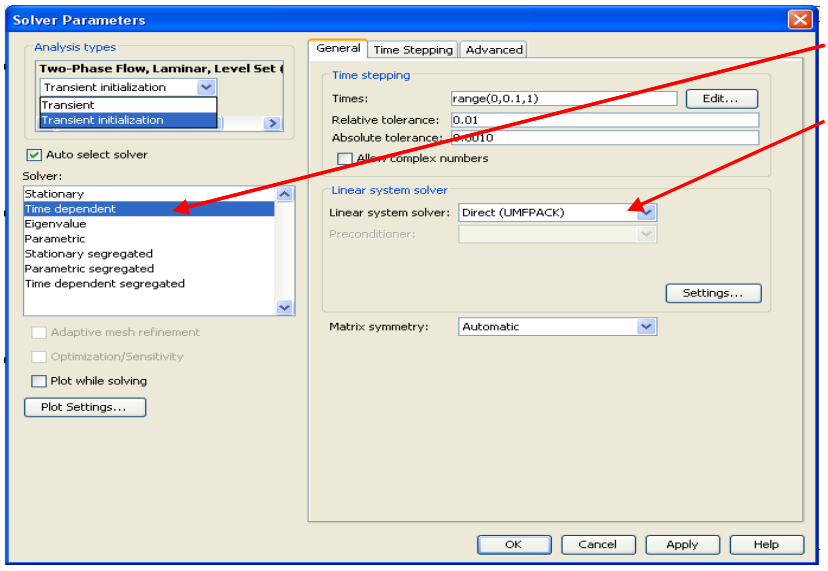


Fig.4.09. Detail of the solving tools

In the case of a laminar two phase flow, you want to solve a time dependant problem to visualize the advancing front.



- Select Time dependant solver
- The system solver will always be Direct (UMFPACK) (best

Fig.4.10. Solver parameters window

After that you will proceed in two steps:

1/ Initialization

- Click transient initialization and OK (fig JJ)
- Open the Solver Manager (see fig KK) and select the Initial value expression evaluated using current solution in first box and Zero in second box. Click OK
- Click Solve.

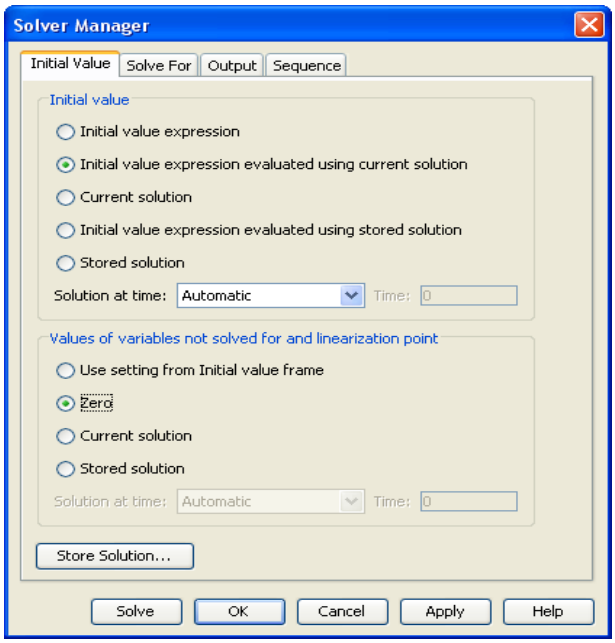


Fig.4.11. Solver Manager window

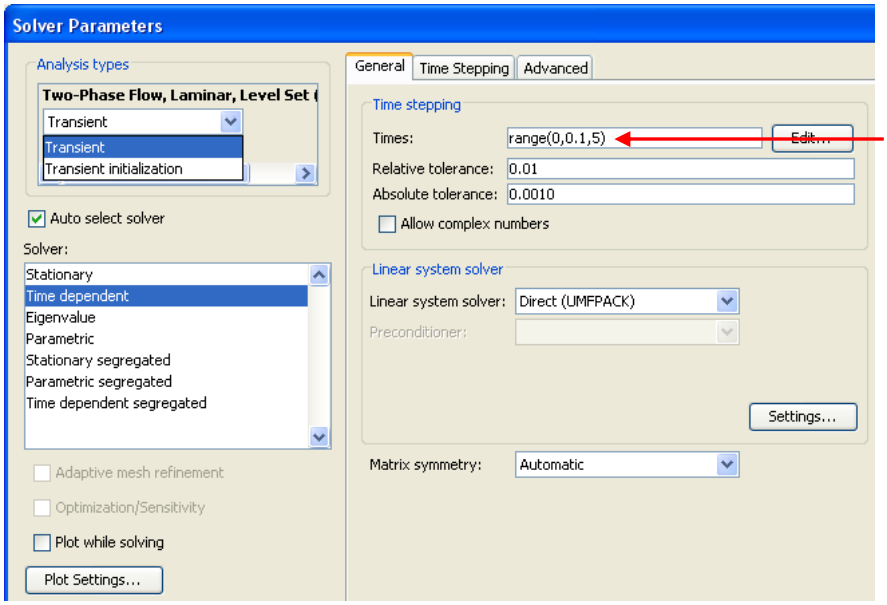
The transient initialization should not be very long (about 2 seconds for our 300 elements piece)

2/ Transient computation

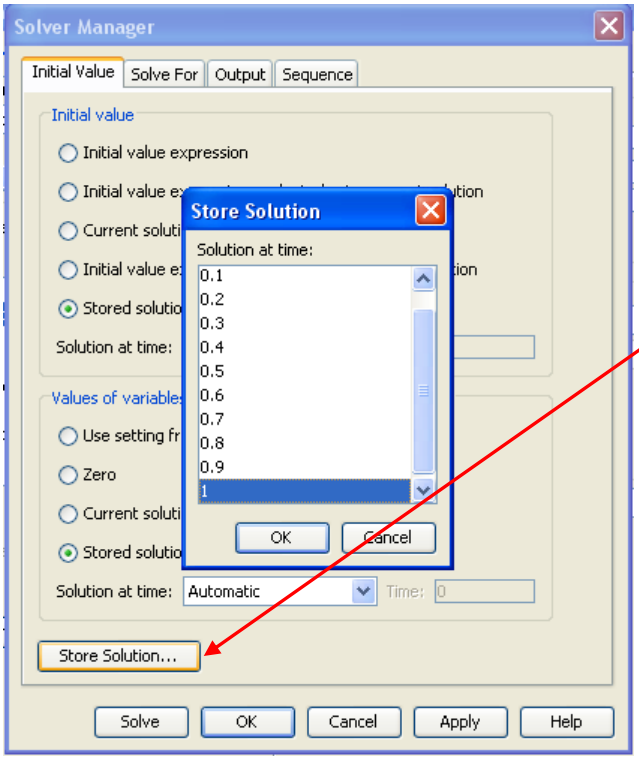
- Change the time stepping for the duration you want to simulate your flow (see fig LL)
- Select Transient analysis type
- Click Apply an OK
- In the Solver Manager (see fig MM), select stored solution in both box.
- Click Store Solution at the bottom of the window and select Solution at time: 1 (it means that the computation will start at the end of the initialization)
- Click Apply and OK
- Click Solve

You may check the duration and the convergence of your computation on the appearing window. You may stop the computation if the convergence becomes too high (10e9).

Convergence is the inverse of the time step for the solution calculation. As a consequence, the smaller is the time step, the higher is the convergence. Consequently, it is not good to have to high convergence, but to low is not good either.



Change the time stepping for the duration you want to simulate your flow (here 5 seconds)



Click Store Solution at the bottom of the window and select Solution at time: 1 (it means that the computation will start at the end of the initialization)

4.6. Step of post processing

4.6.1. Solver parameters

After the transient initialization that is used to define the initial fluid interface, a step depending on time, and were the fluid are going to move has to be done. For this click on the solver parameters button. In the window: Solver Parameters, in analysis types say it is a Transient step (depending on time). In the time edition enter the first value and the final value of the experimentation time, with the step size. Ex:

First value: 0s

Last value: 7s

Step size: 0.2s

Click on replace to change the previous value. And continue using Direct (UFPACK) as the linear system solver.

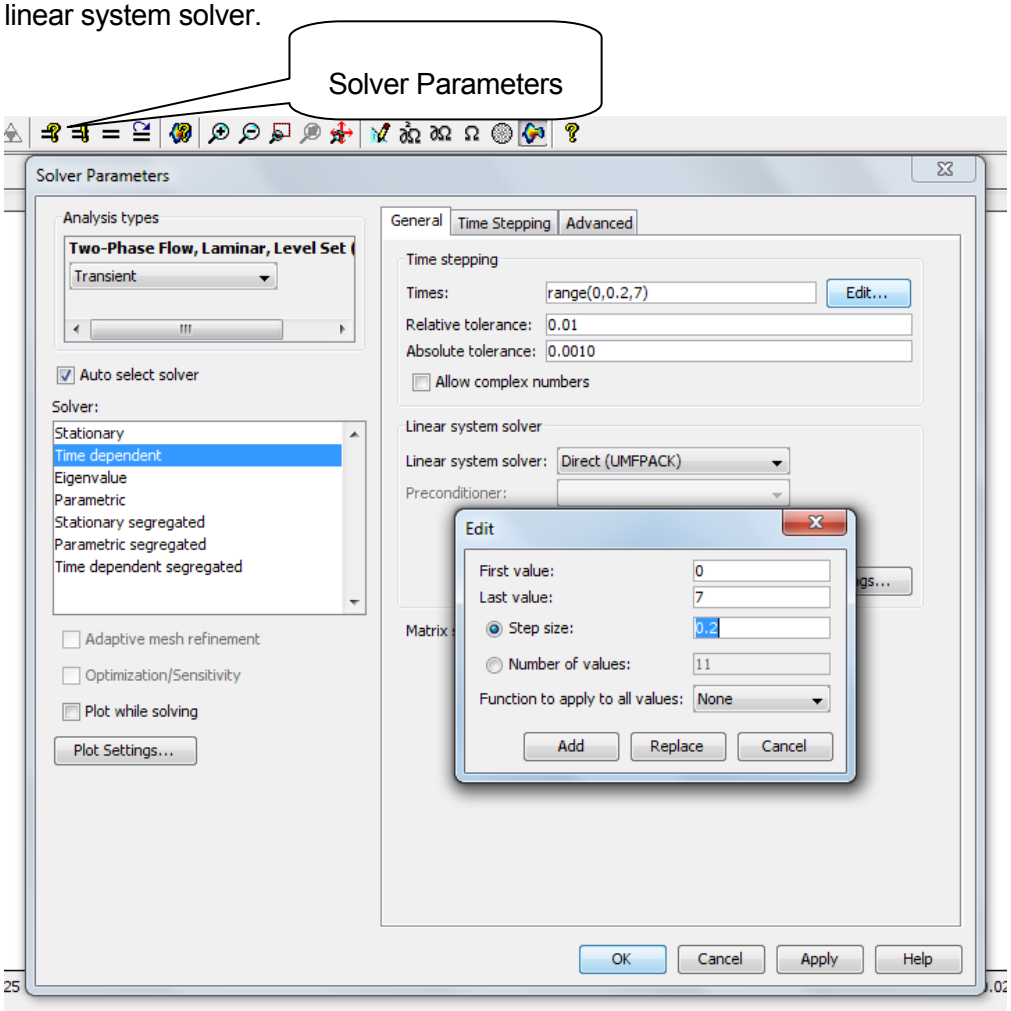


Fig.4.12. Solver parameters window

After this step, the program has to know how the initial interface between the two phases is. For this the better is to say to the software to take the last value of the transient initialization step. Click on the solver manager to enter on the following window.

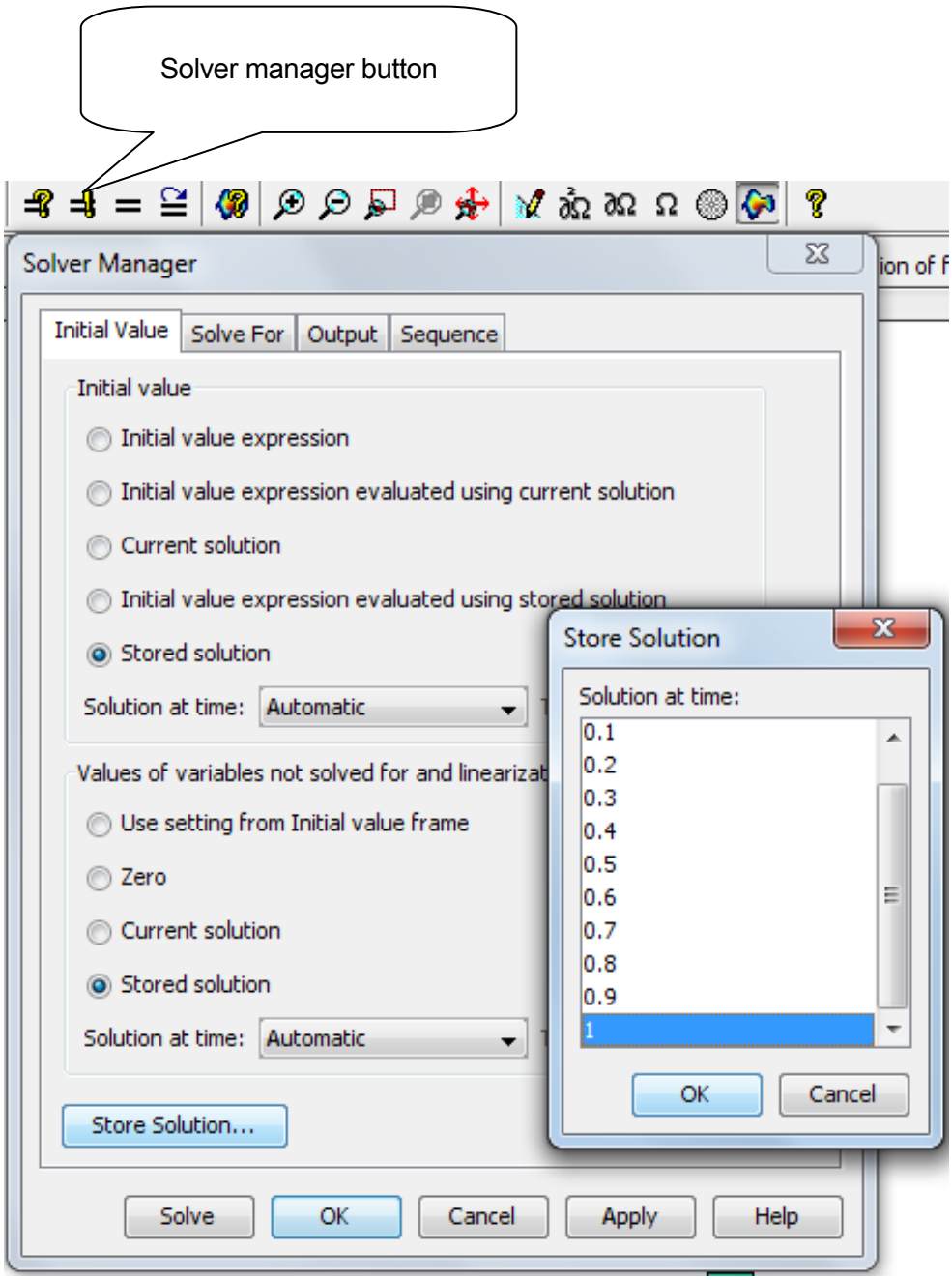


Fig.4.13. Solver manager window

Click on solve and wait while the program is processing all the information. In function of the mesh, the computer used and all the parameters on input, the processing time will be more or less long. Try then to optimize the system in function of the calculus power available and the precision of the problem that is required.

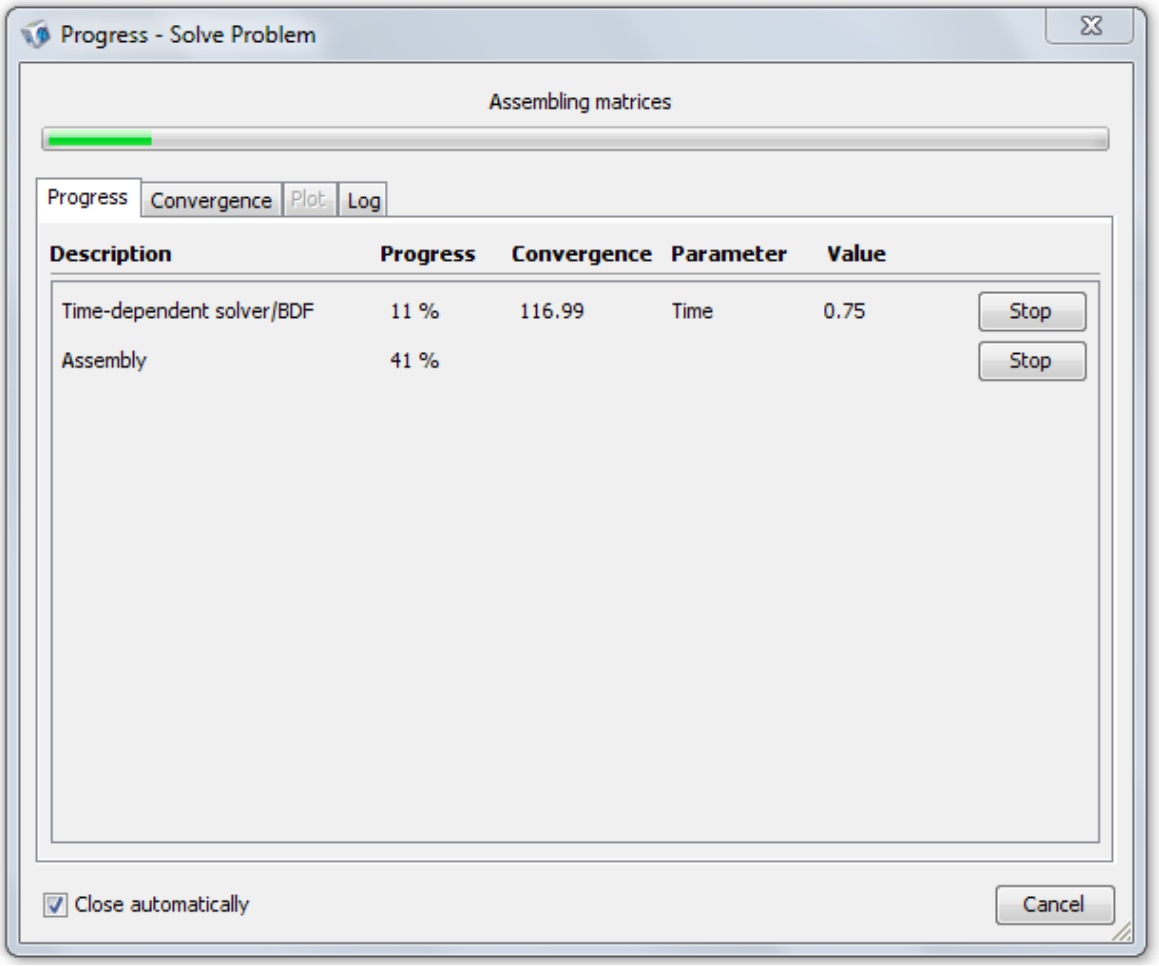


Fig.4.14. Window showing the progress of the computing

4.6.2. Solution and analysis

The more important part is the solution and the analysis of this solution. After this step, the solver will close itself; a result of the volume fraction of fluid 2 is given as shown in the picture below.

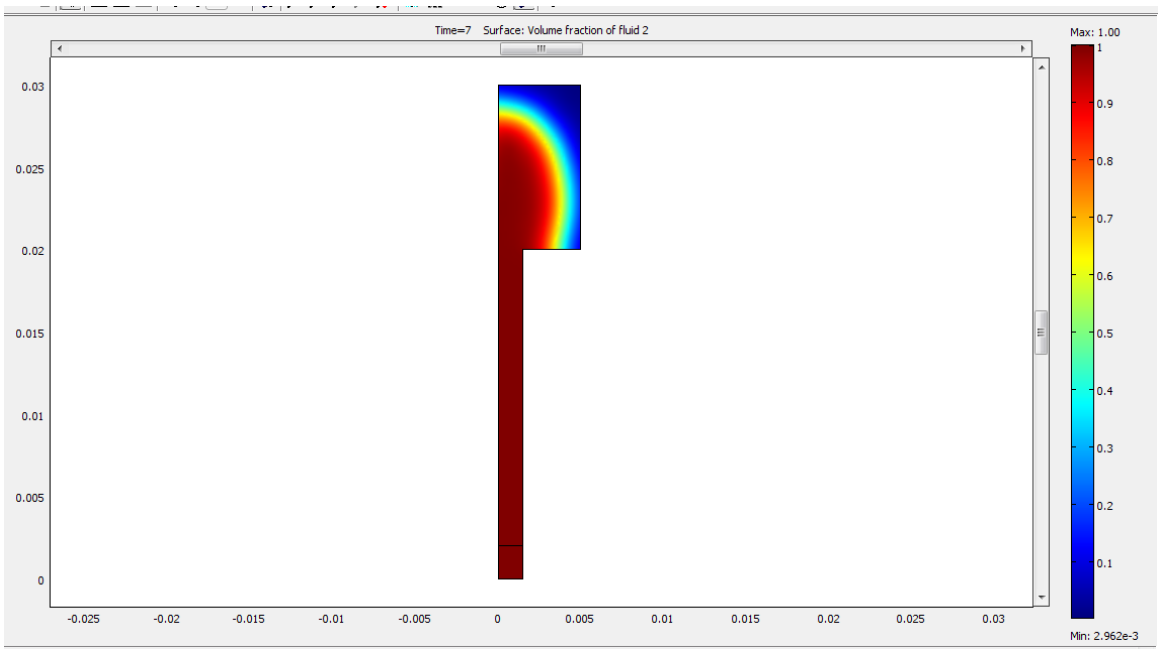


Fig.4.15. Picture showing the volume fraction of fluid 2 in fluid 1 using Level Set Method on COMSOL

All the results can be analyzed as the programmer want, by opening the plot parameters window.

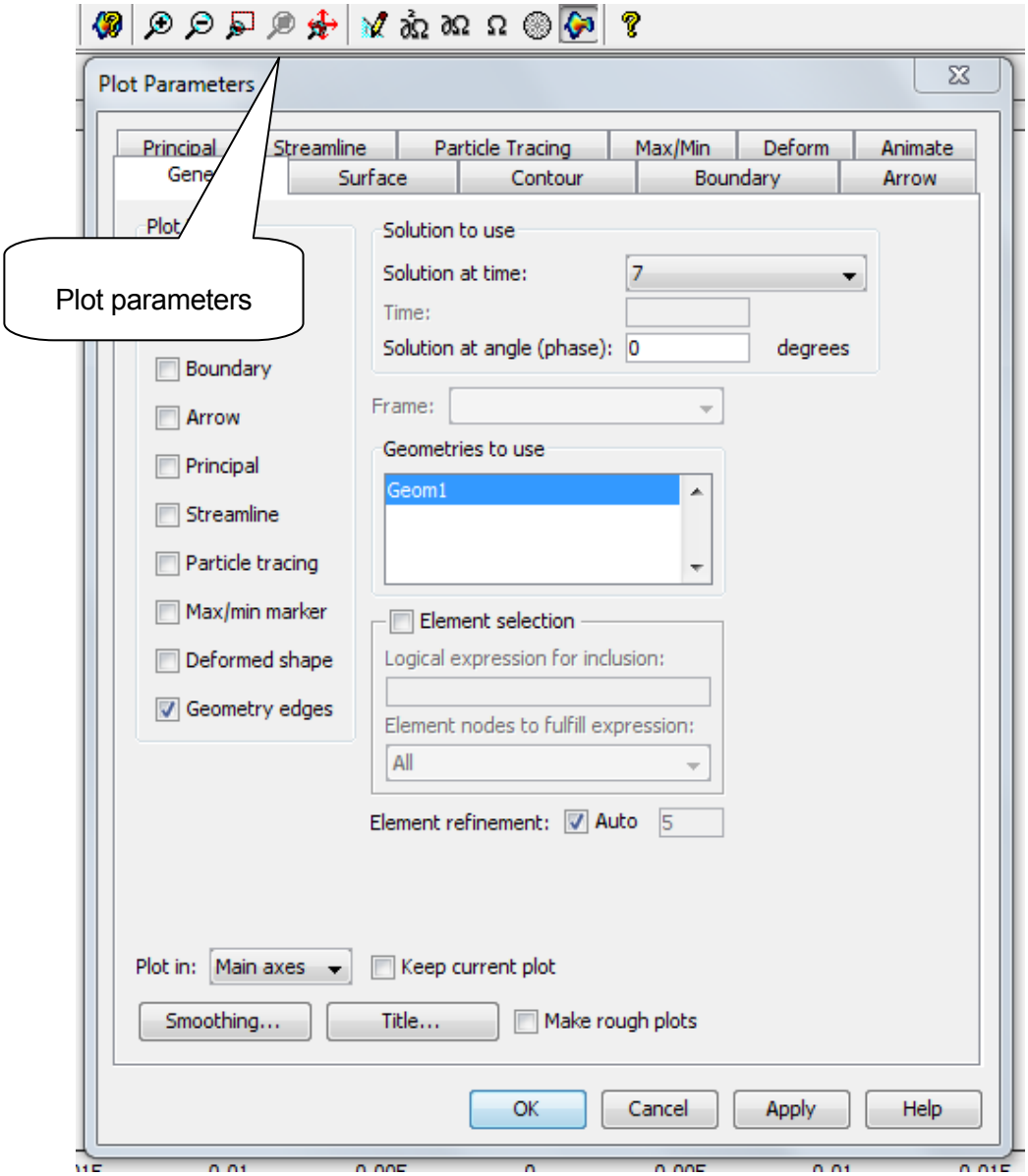


Fig.4.16. Plot Parameters Window allowing the user to change the visualization of the results

In this window, it can be choose the time for analysis, all the properties that the software needs to give. For example the person doing the problem can change if they want to work with the volume fraction of the flow 2 or the flow 1. It is possible also to see the pressure distribution o, the velocity field or the dynamic viscosity.

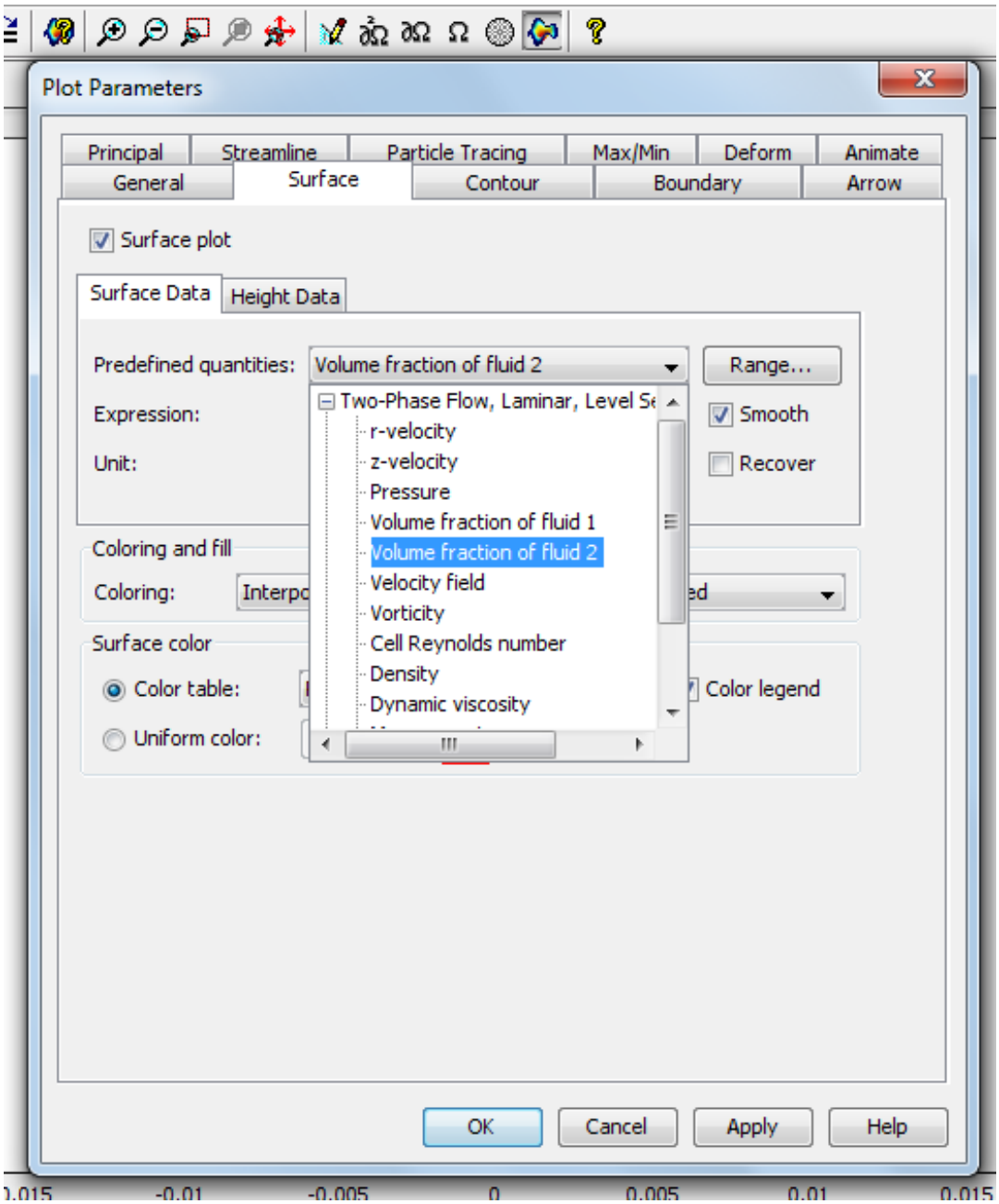


Fig.4.17. Plot Parameters Window allowing the user to change the visualization of the results in this case looking at the volume fraction of fluid 2 in fluid 1

An important thing that can be done to differentiate easily the interface between the two flows is to check in the general tab the contour and the boundary buttons as in the picture below. In the boundary tab just check uniform color and put a black color to distinguish the boundary surface. And then in the contour tab, check the vector with isolevels button and put for example 0.75 (this is the volume fraction of the follow 2). Don't forget to check also the field button.

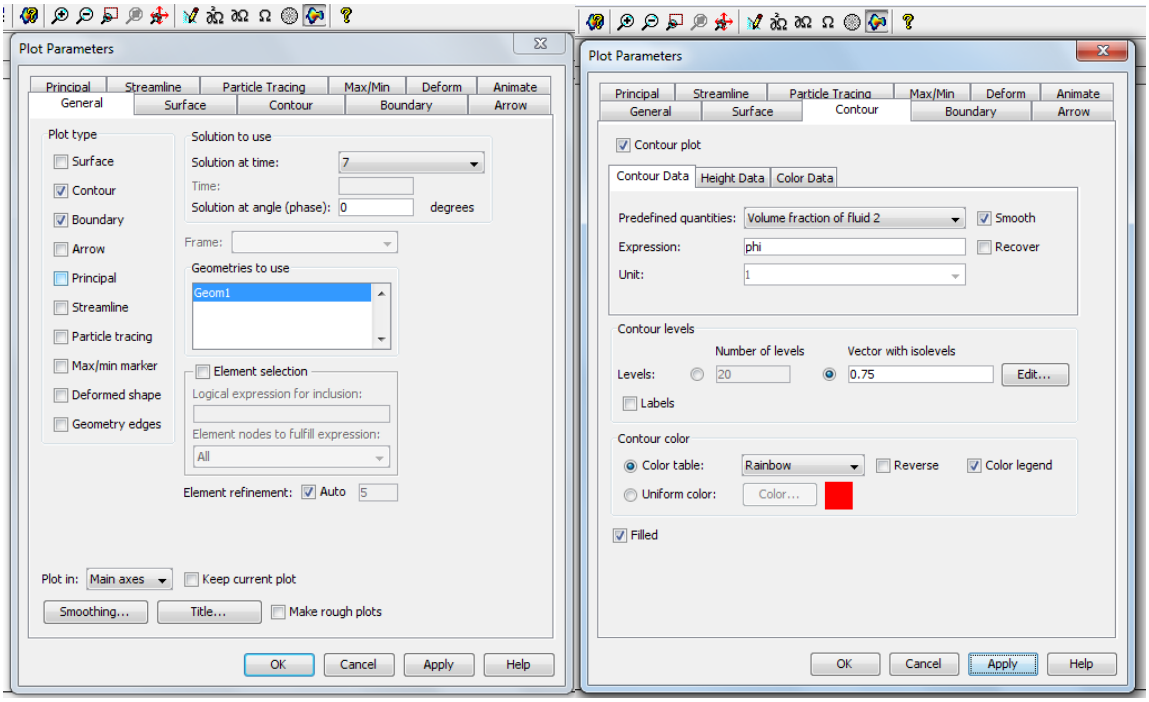


Fig.4.18. Plot Parameters Windows used to simplify the visualization of Level-set problems

At the end of this procedure we obtain this kind of interface:

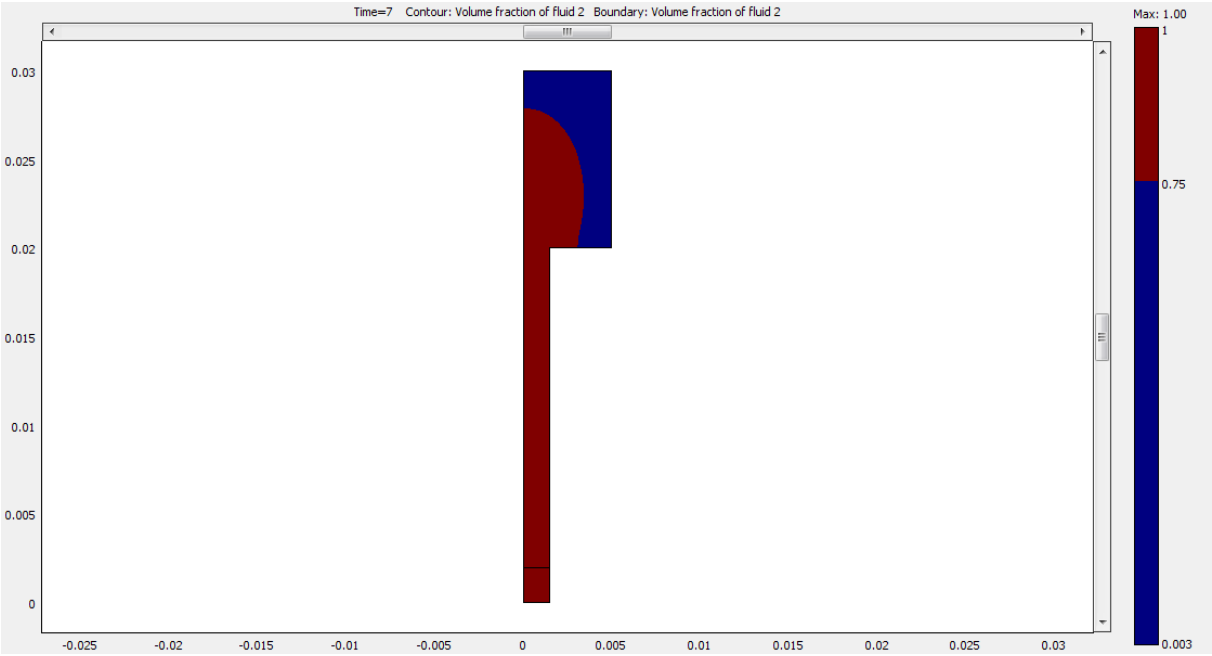


Fig.4.19. Final result of the simplification in the visualization

This is better to see the advancing front level, to analyze and compare with the experimental results. This time a very good separation between the two phases has been obtained.

4.7. Summary conclusion

This chapter explains how to make a model on COMSOL Multiphysics. This is relatively easy if the different steps described are followed. To make it even easier, a Tutorial has been done. This Tutorial can be open in the CD given with the paper and contains explanation of how to do some models on COMSOL in a more advanced way than this text. A few models have been done in 3D, and how to obtain the results has been described with accuracy.

The problem is that once the results are obtained, a layer which in reality doesn't exist is present in each model. The aim in the next chapter is to prevent this thick layer to appear, and reduce it the more that the material used for calculation could. All this is done by changing some physical and mathematical values.

5. What is Level Set?

Flow problems with moving interfaces or boundaries occur in a number of different applications, such as fluid-structure interaction, multiphase flows. In the case of Kyphoplastie, this method is very useful to describe the progression of the flow in a certain space. Then one of the possible ways to track moving interfaces is to use a Level Set method. A certain contour line of the globally defined function, the Level Set function, then represents the interface between phases.

The method of Level Set is defined by some parameters that can be modified in COMSOL Multiphysics. But two of these parameters are the more important: gamma and epsilon. Each of them is going to modify the properties between the phases interface. The aim of this work is to see how they work, what they do in the interfaces and at the end be able to choose parameters that describe well the real situation. **[12] [16] [18]**

Here is the principal equation that uses COMSOL Multiphysics for the two phases Level set method.

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = \gamma \nabla \cdot \left(\varepsilon \nabla \phi - \phi(1-\phi) \frac{\nabla \phi}{|\nabla \phi|} \right)$$

Where phy is the ratio of one of the two phases, gamma and epsilon the two parameters of the level set that will be explained later on, and the interface between the phases depend on the time.

5.1. Influence of the parameter gamma on the interface

This parameter gamma is called the reinitialization parameter and is expressed in m/s. To understand the influence of gamma on the distribution flow, we have modeled a very simple case. A thin tube can be considered as the syringe of the kyphoplasti, from where the cement is coming and a bigger space that the cement will invade. The object is axial symmetric, and has the following dimensions

	THIN TUBE	BIGGER TUBE
L (IN CM)	**2**	**1**
L (IN CM)	**0,15**	**0,5**

Tab.5.01. Dimensions of the model for the Level Set

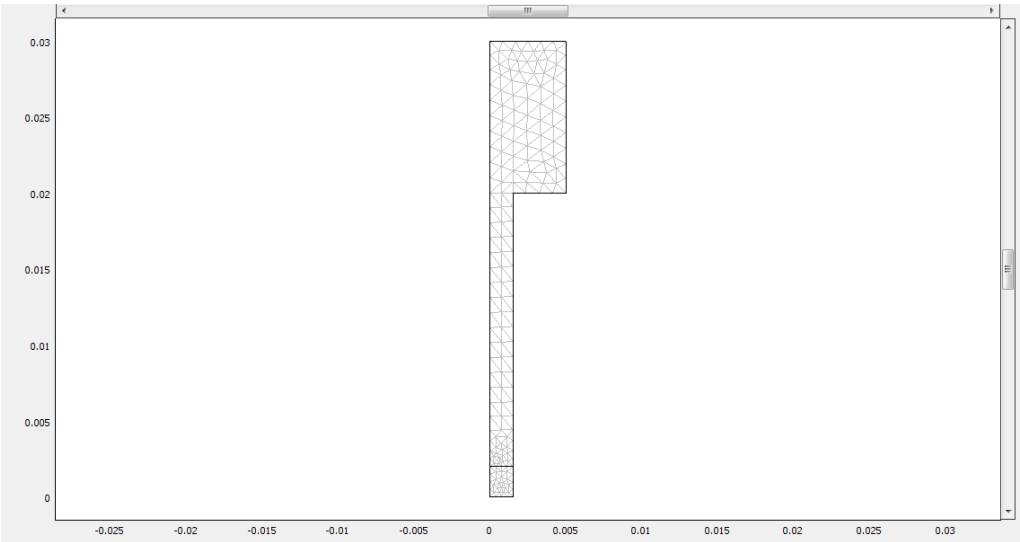
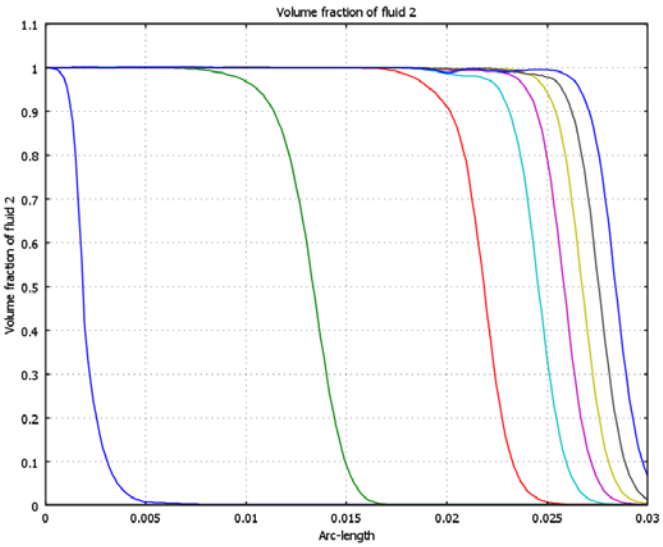
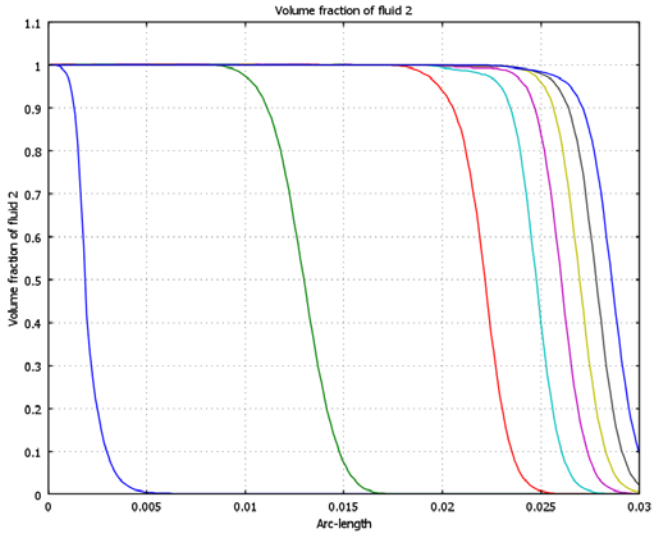


Fig.5.01. This is the model used for all the level set study with the dimension mentioned in the Tab.5.01.

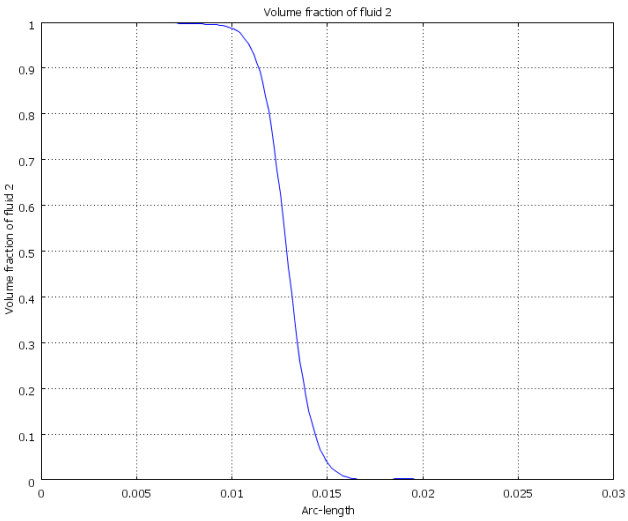
Different kinds of gamma have been used to describe this modeling. And the results are given by the plots that show the fraction of the phase 2 in the phase 1. Normally we should obtain plots with shapes very similar to the shapes of Heaviside function.



This curve was obtained with a gamma of 0.1 m/s. This curve undergoes some diffusion.

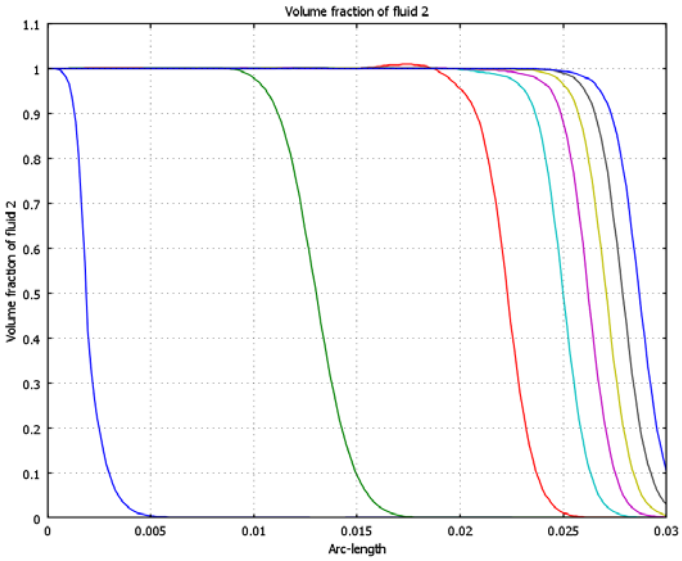


This curve was obtained with a gamma of 0.01 m/s. This curve undergoes less diffusion than with the parameter gamma 0.1 m/s.

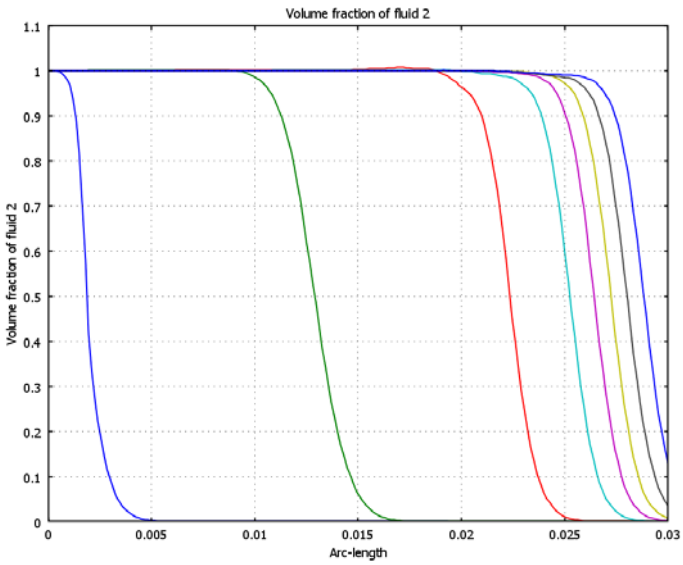


This curve was obtained for a value of gamma of 0.5 m/s. This curve undergoes a bit more of diffusion than for a lower gamma value.

The difference of the diffusion between these plots is not very clear. This parameter gamma modifies the diffusion in the inflexion points of the curves. The difference is more visible with the two examples coming down, but the problem is that, with so low values the level set introduce some little errors, like the volume fraction of the phase two in the one being higher than 1. This error can be fixed with the parameters of crosswind and streamline diffusion that will be described further. Moreover, the difference between a gamma of 0.01 or 0.001m/s is very small.



This curve was obtained with a gamma of 0.05 m/s. This curve undergoes less diffusion than with the parameters gamma 0.1 m/s or 0.01 m/s, but introduces some errors.

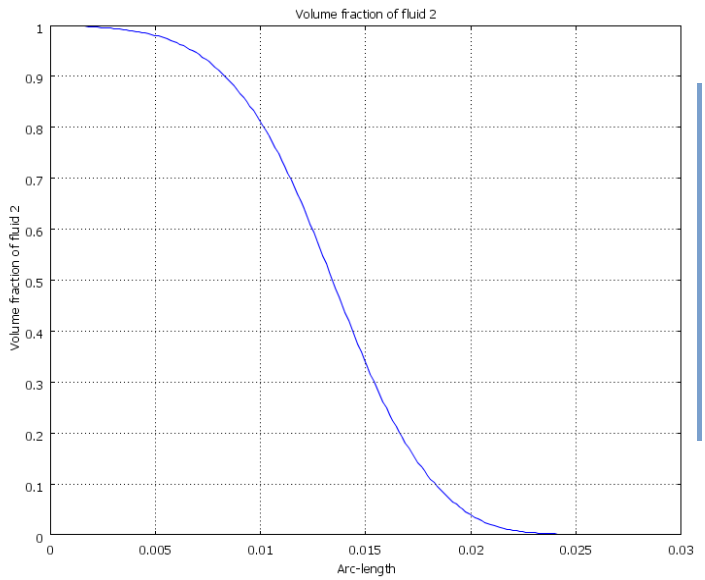


This curve was obtained with a gamma of 0.03 m/s. This curve undergoes less diffusion than with the parameters gamma 0.1 m/s; 0.01 m/s or 0.005m/s, but introduces some errors that can be fixed by the artificial diffusion.

In the case of this study, the best results were given by a gamma that was equal to the entrance velocity in the syringe. In our case the best value for gamma is 0.01 m/s. In the notice of COMSOL Multiphysics, the suitable value for this parameter is the maximum velocity flow in the model. So in our case it should be 0.02 m/s.

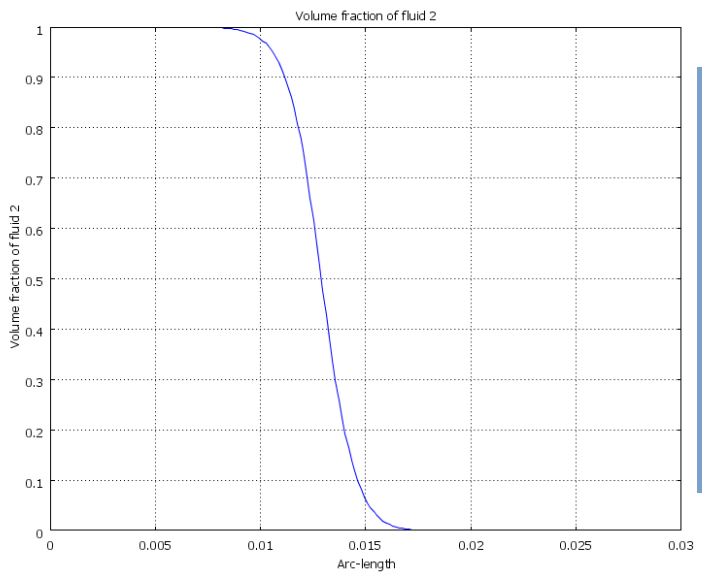
5.2. Influence of the parameter epsilon on the interface

This epsilon parameter controls the thickness of the phase. In our case, this thickness must be really thin because the two phases are not miscible. This study will help to control this parameter and explain the suitable value to use in each case. Another time the same model is going to be used. The initial value proposed by the software is ½* hmax_mmglf with hmax_mmglf representing the maximum length of an element mesh. The experience was done by varying the multiplier factor between 0.1 and 2. The next values are for the experience at the time 1s and show the quantity of diffusion for each value of epsilon.



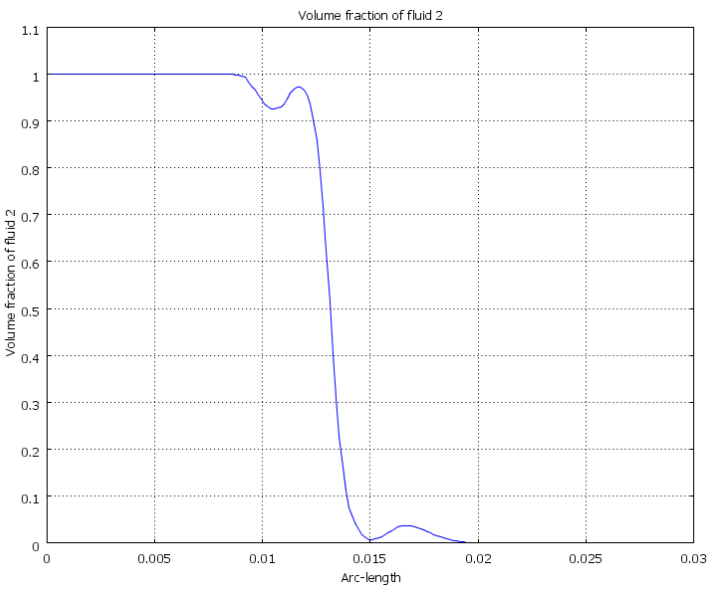
*This curve was obtained for a multiplier of 2 so: 2*hmax_mmglf=1.7e-11. This curve is really soft, and very far from the model of Heaviside.*

Decreasing the multiplier factor is the good thing to do to obtain better results.



*This curve was obtained for a multiplier of 0.5 so: 0.5*hmax_mmglf. This value of the multiplier is the default value proposed by the program.*

The results obtained with this value are much better. We have a thinner interface between the two phases. With lower value, the result could be improved, and then be less far from reality. With the multiplier being 0.4, we obtain better results, but the problem is that the compilation cannot be achieved. The program doesn't converge. The COMSOL Multiphysics help say that decreasing the mesh of the object improve the reduction of the two phase thickness. But for computational problems some tests have been done without results.



*This curve was obtained for a multiplier of 0.1 so: 0.1*hmax_mmglf. In this example the result is perfect. But some problems of convergence and numerical errors have appeared*

This last example is really interesting, not just because the result is quasi perfect, and very similar to a Heaviside equation, but also because of the numerical errors that appear here. First of all it is important to say that it was impossible to obtain so good results, because of computational resources, and so the model did not converge. The numerical errors that appear here are interesting, because we can see the limit of this model, it is not perfect. Looking to this curve, it has no physic sense. This kind of errors can be fixed by inducing some artificial diffusion. There are two types of artificial diffusion, the cross wind and the streamline diffusion. The artificial diffusion will be explained in the next chapter.

5.3. The Sources and Sink tab

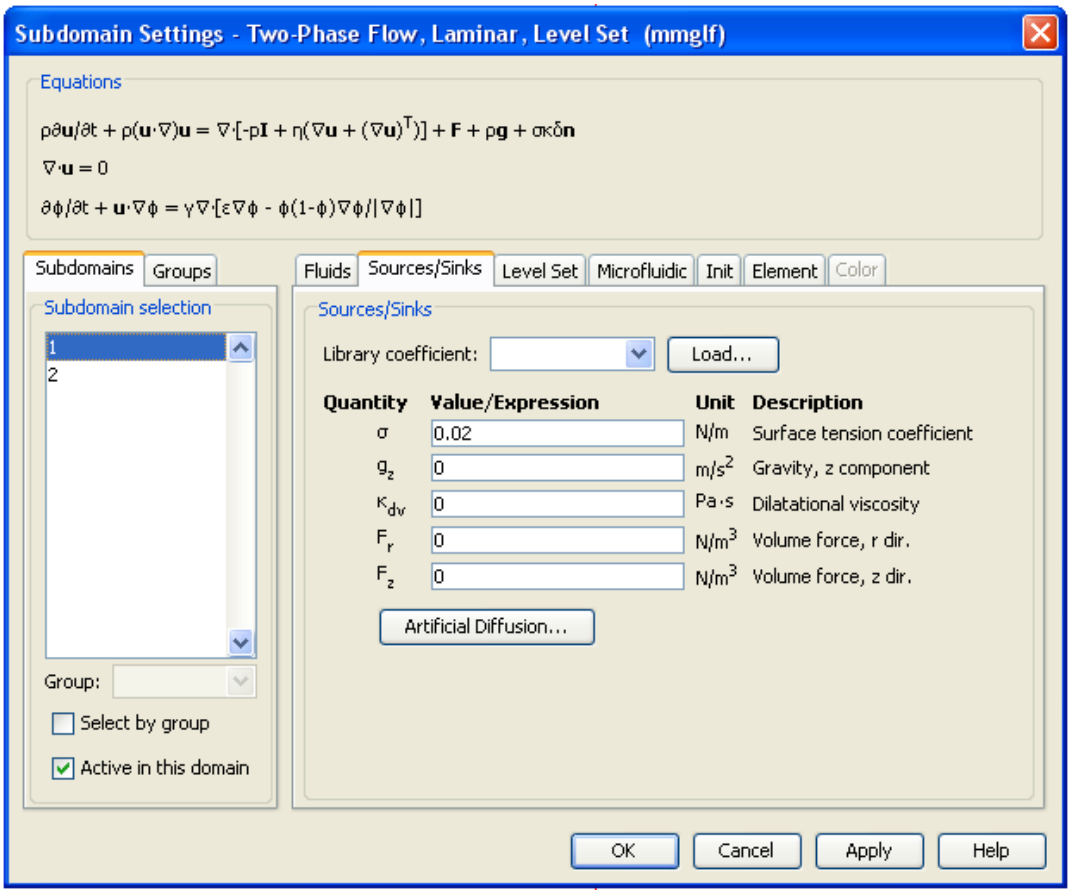
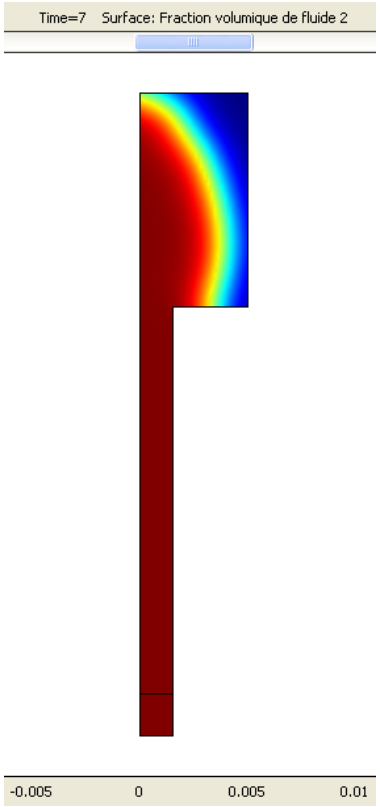


Fig.5.02. Screenshot of the Sources/Sink window appearing in Subdomain settings.

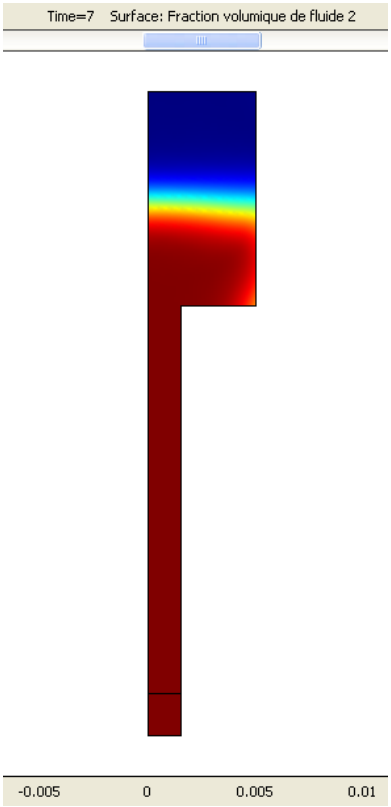
The Source/Sinks tab permits to add parameters such as surface tension coefficient, dilatational viscosity, gravity, and volume forces.

The surface tension parameter characterizes the property of a liquid to be attracted to another surface. It has been modified from 0,002 to 0,8 with all other coefficients from other tabs remained unchanged.

We observed the evolution of the curves of the volume fraction of the cement every second during 7 seconds which means 8 curves representing the advancement of the front of the cement in the tube.

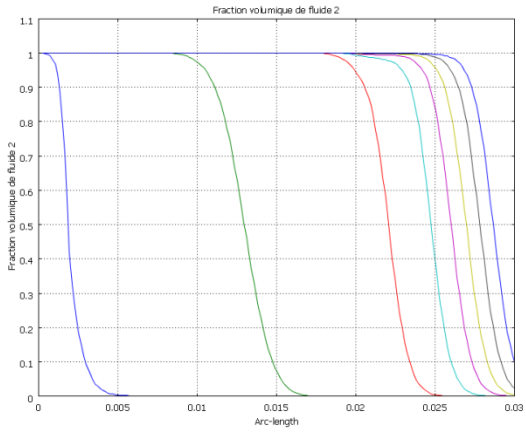


a) Paste behavior: fast growth in the propagation direction with a weak lateral expansion

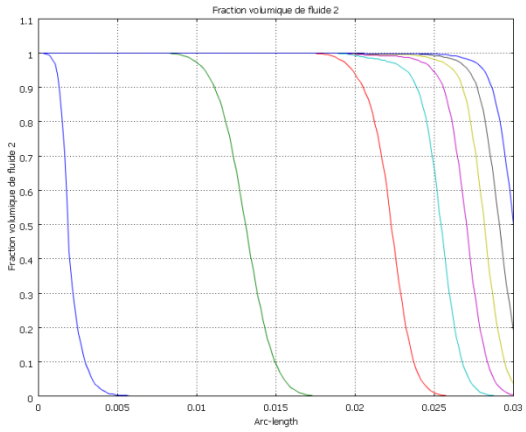


b) Liquid behavior: slowly growth in the propagation direction but with an important lateral expansion

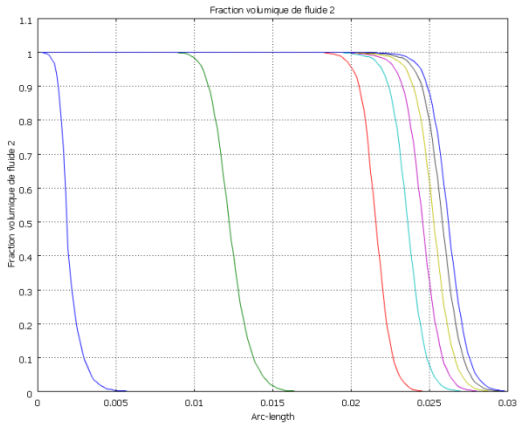
Fig.5.03. The higher the surface tension coefficient is the more the cement behaves like a liquid and fills the entire tank showing an attractive behavior to the lateral surface of the empty volume.



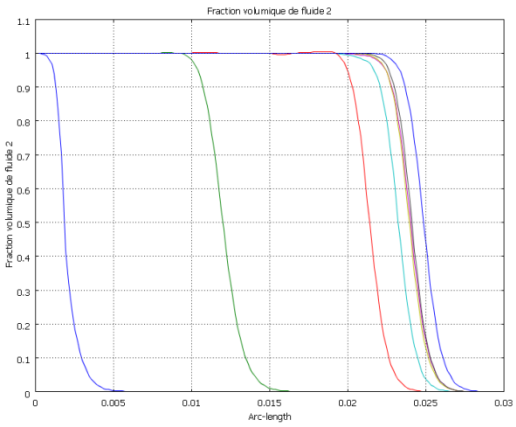
σ=0,002



σ=0,02



σ=0,2



σ=0,8

Fig.5.04. The higher the surface tension coefficient is the slowly apparently advancing the cement is. This is due to the fact that a part of the volume of the cement is necessary to full the lateral part of the tank.

However it is interesting to notice that the advancing profile of the cement is modified. The dispersion of the values of volume fraction is less important with a higher surface tension coefficient.

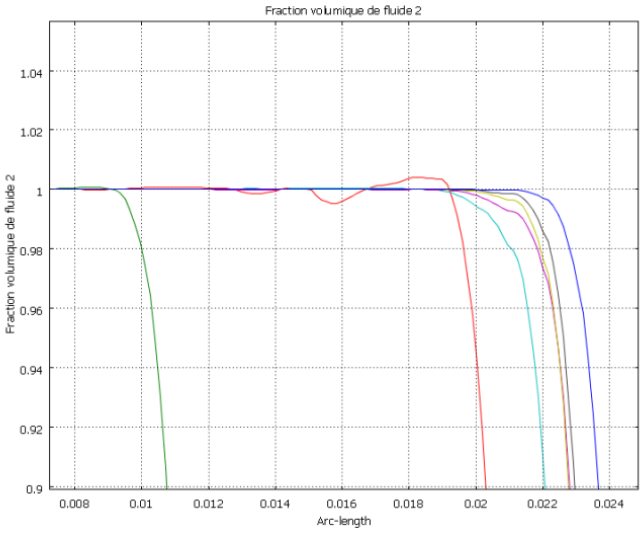


Fig.5.05. We may observe some noise when the surface tension coefficient is too high (0,8) characterized by waves and a volume fraction of cement higher than 1.

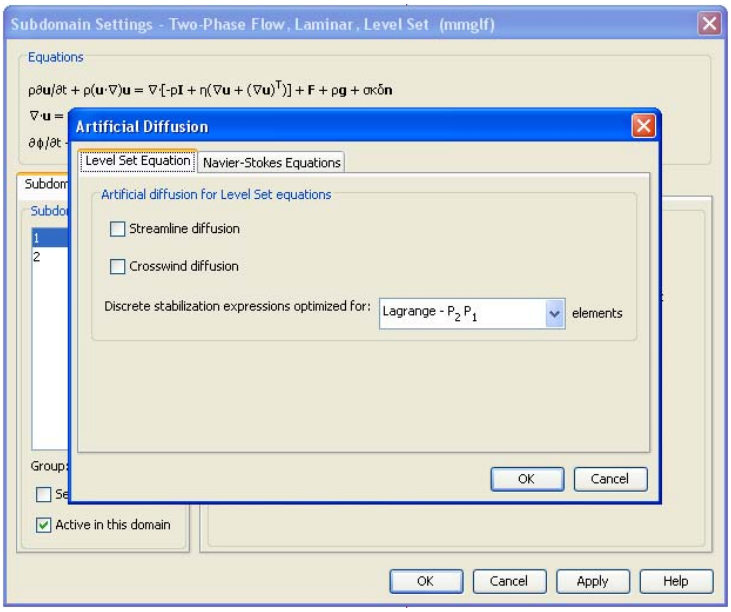
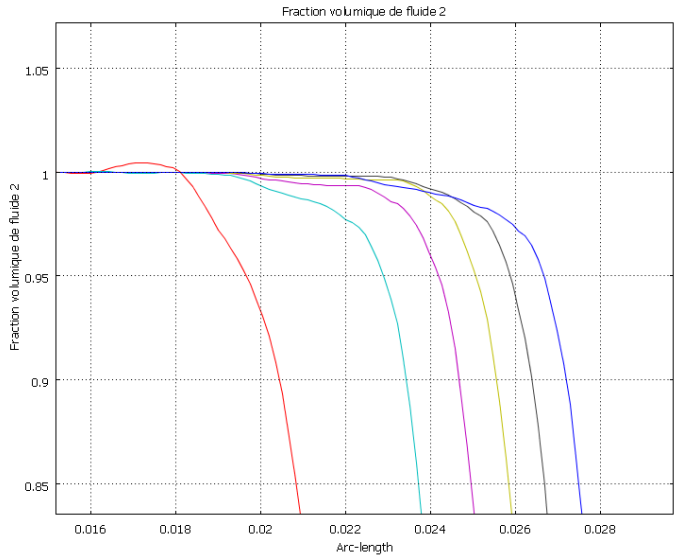


Fig.5.06. Screenshot of the Sources/Sink-Artificial Diffusion window.

The introduction of cement is purely a problem of convection. A characteristic problem occurring in this type of model establishment is the noise appearing near the inflexion points of the curve. This problem is relieved thanks artificial diffusion which permits to smooth the curve without the need for mesh refinement.

Streamline diffusion occurs in the direction of the flow.

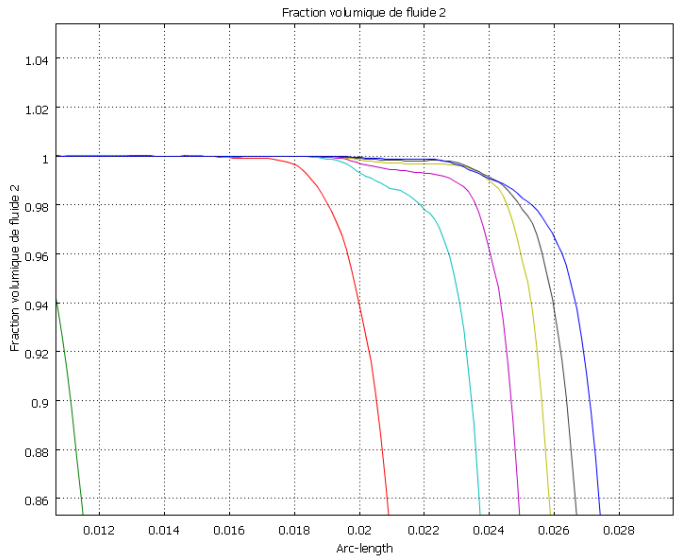
Crosswind diffusion occurs in the orthogonal direction of the flow.



Crosswind diffusion off

Streamline diffusion off

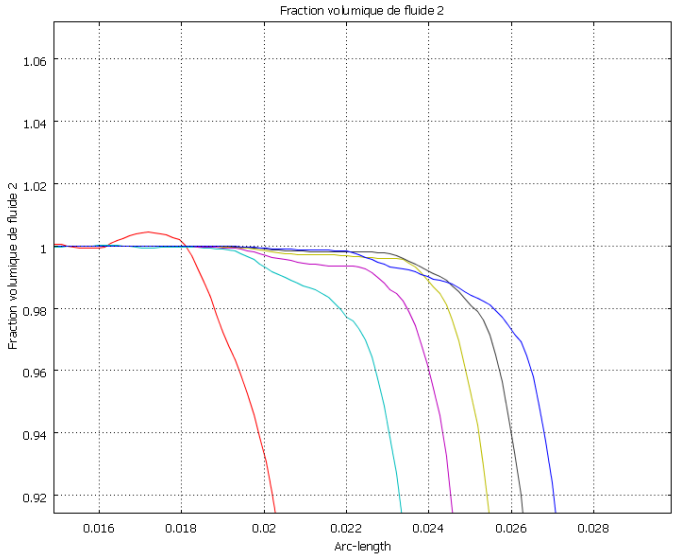
We observe a kick on the red curve characteristic of uncertainty of resolution



Crosswind diffusion off

Streamline diffusion on

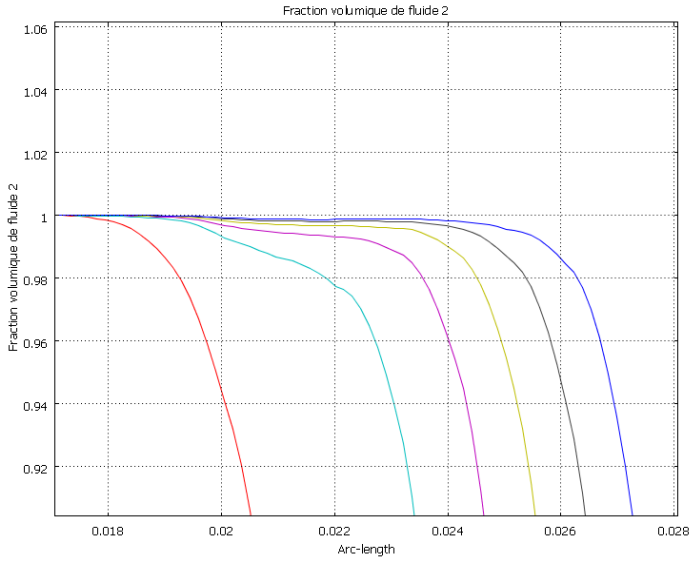
The kick on the red curve barely disappeared and the curve is smoother proving the interest of streamline diffusion



Crosswind diffusion on

Streamline diffusion off

The kick on the red curve is not completely disappeared which proves that crosswind diffusion is less efficient that streamline diffusion



Crosswind diffusion on

Streamline diffusion on

No kick is appearing

Fig.5.07. It appears that the ad of streamline diffusion is sufficient to reduce the noise. The crosswind diffusion may not be used in our model establishment.

5.4. Summary conclusion

In this chapter all the variables involved in the apparition of the characteristic layer of Level-set are defined. And some numerical experiments have been done to highlight the effect of these variables on a computation. Some values define the layer itself; others define some mathematical corrections that can be done to minimize the numerical errors.

An important thing to learn from this chapter is that with more numerical power, better results can be reached. The parameters can be the more accurate that the user need, and the mesh the finer as possible.

But to get the best of the computers that are at your disposal, these parameters have to be chosen very smartly. That's why before doing the real experiments some numerical results should be obtained to have the optimum results, with the help of this chapter.

To put in evidence the accuracy of COMSOL's computations, and the results that are obtained, some real experiments with fluids have been done. For this, different kinds of injections of liquid have been done inside cavities.

6. Building of an injection station

6.1. Interest of an injection experiment

The goal of this project as it has been said was to find a way to model an injection of cement like those used in kyphoplasty and vertebroplasty. The models on the computers with COMSOL Multiphysics were done. The aim was to obtain results very similar to experimental data, however without any experimental data it was very complicated to compare something. As a result, we decided to make a little model which is provided of fourteen cavities to dispose inside some obstacles and then compare the interface of the fluid with the corresponding COMSOL Multiphysics model. We choose to do six cavities of 40*60 mm and with just 1 cm of thickness. This thickness was chosen to consider that this model is like a two dimensional problem, and then simplify the problem. These cavities are the betters to do the experiments with the obstacles. As the human eye cannot see really well the changes in the interface fluid, it is better to do big models than really small ones. But in case of the project is following a good way, some little cavities were done to simulate the injection in real vertebras.

6.2. Drawing IS14 with Catia

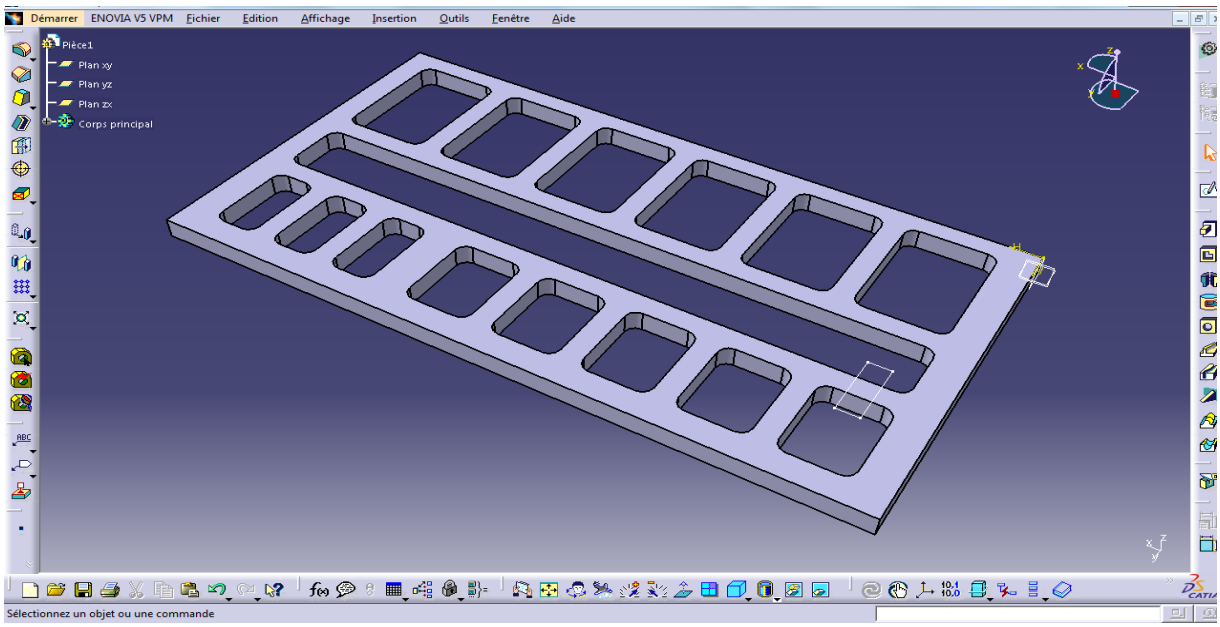


Fig.6.01. Perspective picture of the InjectionStation14 (IS14) perspective view

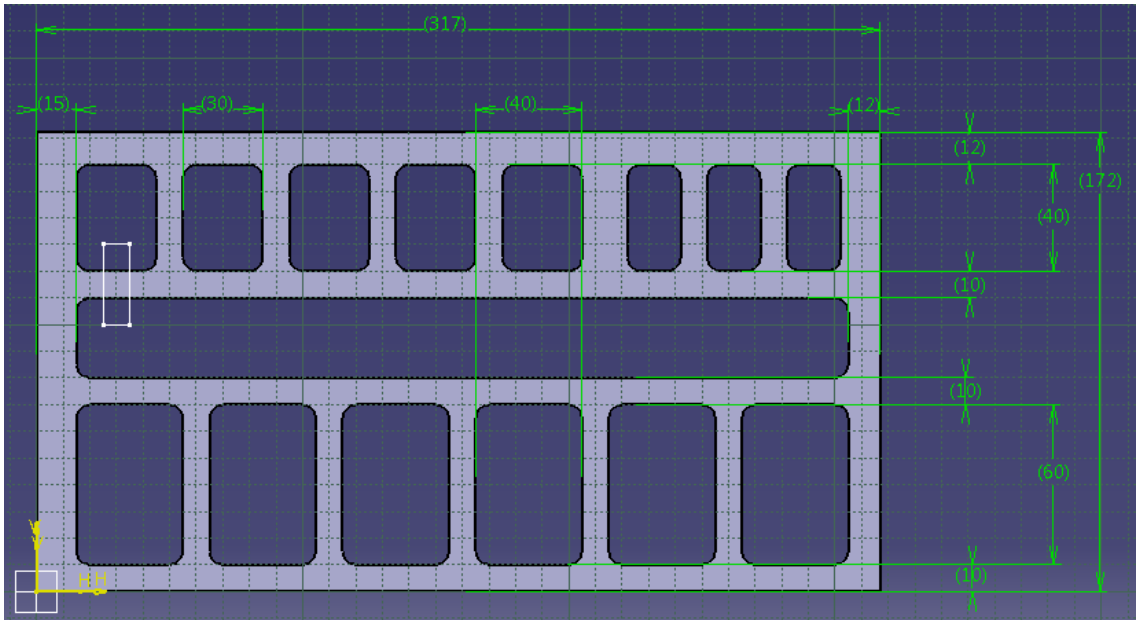


Fig.6.02. Dimensions (in mm) of the injection mini laboratory named InjectionStation14 (IS14). Top view

Five cavities of 30*40*10 mm3 have been done to test cavities resembling more to cavities in the vertebras. And three very small cavities to specify the fluid interface without obstacles. These cavities will give the properties that are needed by COMSOL. And by Comparing the models made on it and the experiments, verify the COMSOL results.

The IS14 is made with transparent PMMA above and another sheet of PMMA to see the interface between the air and the fluid. In each of these 40*60*10 mm boxes an experience of injection can be achieved and filmed to compare with the numerical results. Fourteen holes with a diameter of 3mm are done in the middle of each cavity. These holes are the place where the injection is going to be done.

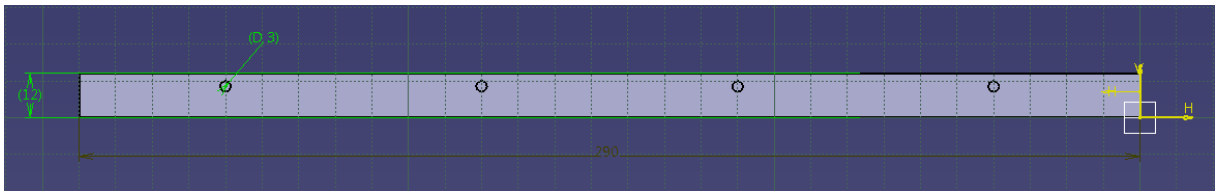
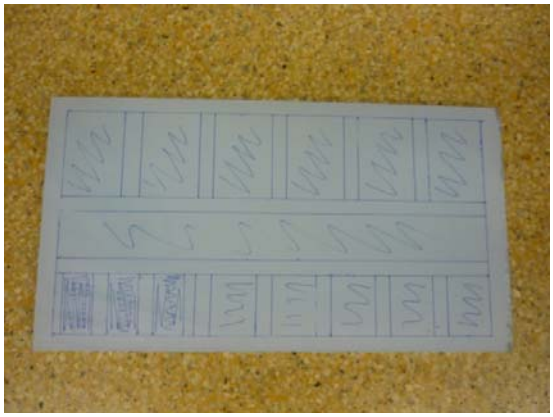


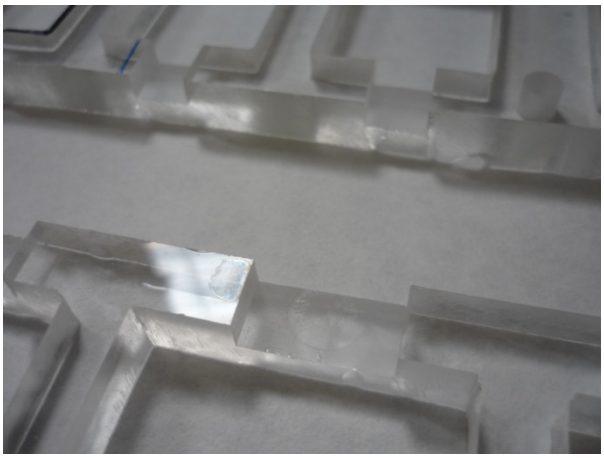
Fig.6.03. Dimensions (in mm) of the injection mini laboratory named InjectionStation14 (IS14), Side view

6.3. Construction of IS14



From the draws and perspective pictures made with Catia the volumes that had to be removed from the plate have been drawn.

Series of holes have been made in the volumes with a drill machine using a diameter 10mm drill, after what the off cuts have been removed and the surfaces have been successively leveled with a rough file and smoothed with a fine file.



It was necessary to connect each test volume to a central tank in order to avoid an air compression in the test volume. That is why 5mm depth notches have been made with a milling machine using a diameter 10mm milling-cutter and cutting systematically of 0,5mm.

PMMA is a brittle material so it is necessary to lock the plate carefully on the work table and to perform the holes very slowly, especially when drilling close to the corners and borders.

Next the top and bottom plates have been put on both sides of the holed plate and holes have been made to lock them together with diameter 6 screws and wing nuts.

Finally, little holes have been made in the transversal side in the middle of each cavity to link the needle with a diameter 3mm drill.

Digging holes in the plate has been made without any automation so it was a quite fastidious work. It required barely two weeks of work to get the product finished because of regular incompatibilities of schedule with the technicians to access to the machine room. If this kind of object has to be made, we recommend if possible to use automation or at last to use the milling machine even to dig the holes in order to gain a precious time.

6.4. Summary conclusion

The IS14 was done in approximately two weeks of work. A numerical version has been done to discuss with the technicians about the dimensions and to choose the better way to do it.

Since this IS14 is done, the injection tests and the comparison between the numerical and the experimental data could be made.

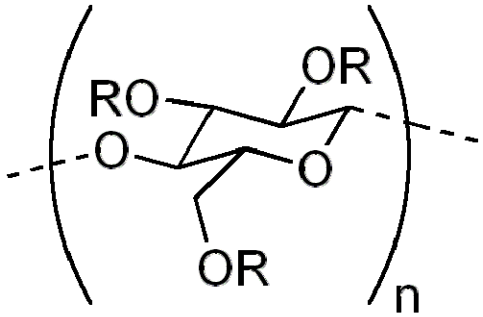
7. Intrinsic properties of the fluid

7.1. Preparation of the fluid

For this experiment, the fluid that is required is a high viscosity one, because with high viscosity, and low velocity, the flow is not going to be turbulent. Like this, the experiment could be analyzed with human eyes. And the second point why a high viscosity fluid is needed is because the initial cement fluid used in kyphoplastie and vertebroplasty is a high viscosity fluid.

Two fluids could be used for our experiment: Carboxymethyl cellulose (CMC) or Gum Arabic .

Carboxymethyl cellulose (CMC) is a cellulose derivative with carboxymethyl groups (-CH_2-COOH) bound to some of the hydroxyl groups of the glucopyranose monomers that make up the cellulose backbone. CMC is used in food science as a viscosity modifier or thickener, and to stabilize emulsions in various products including ice cream. As a food additive, it has E number E466.



$$R = H \text{ or } CH_2CO_2H$$

Fig. 7.01. Carboxymethyl cellulose (CMC)

The gum Arabic is a natural gum made of hardened sap taken from two species of the acacia tree; *Acacia senegal* and *Acacia seyal*.

At the end the Carboxymethyl cellulose was the fluid that was chosen because we had 1,6kg of powder to do it. The problem is that the first pot of CMC powder was full of CMC hydrophobic. So after ten minutes of rest two phases could be distinguished. For our

experiment this was not really interesting. And for making 100ml of product 100g had to be melt with water. This was not a good product to do our fluid of CMC.

CMC can be also hydrophilic and then become a hydrogel when melted with water. To obtain an hydrophilic powder a salt is needed. This salt is sodium salt. The polar carboxyl groups render the cellulose soluble and chemically reactive. The reaction of this powder with the water is a reaction of polymerization between the carboxyl groups. It is because of this that the structure at the end is a hydrogel.

Before making any real fluid, some tries were done to see which quantity should be added to 100ml of water to obtain a consistent fluid. A 2.5%, 5% and 7.5% in weight were done. This means, for example for the 2.5% in weight, that for 100ml so 100g of water, we add 2.5g of CMC. This quantity is of CMC is really small. So CMC hydrogel is a good fluid, really cheap, to start the experiments.

After choosing the fluid that was going to be used, some values are needed by COMSOL Multiphysics to compute the models. These intrinsic values are the different properties that characterize a fluid material, as the viscosity, the fluid density. And some properties of the surface in which the fluid is flowing, like the contact angle.

7.2. Determining the density and the viscosity of the fluid

The density of the fluid is very easy to calculate. The final product is inserted in a recipient of 200ml, and then weighted without the mass of the recipient. In our case the density of the final product is: 1037kg/m^3.

One of the really important values that COMSOL Multiphysics needs is the Viscosity of the fluid that is used. The viscosity is a measure of the resistance of a fluid which is being deformed by either shear stress or extensional stress.

The values of the viscosity were then very important. So an Ostwald viscometer was used in order to measure the value of this viscosity. The problem is that even the fluid with 2.5% in weight was too viscous for this kind of viscometer. It took all night to flow along the capillary. And the limit was missed because of this. At the end it was decided to calculate the viscosity using a small ball that is dropped into a container with the fluid that is studied, and applying the Stokes formula.

The idea is the following: the little ball is dropped inside the container containing the fluid.

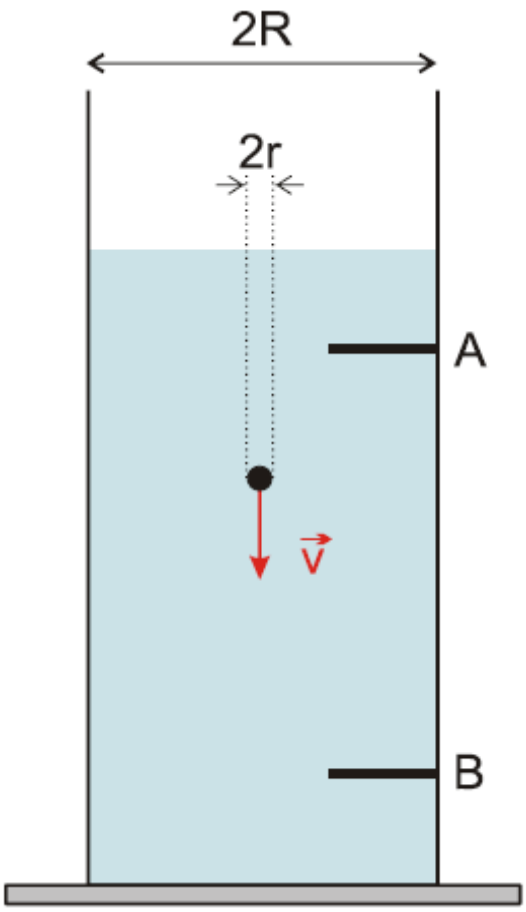


Fig. 7.02. Fall of a small ball in a recipient containing the fluid that has an unknown viscosity.

The forces acting in this case are:

- the weight of the small ball: $P = m * g = \rho_{ball} * V_{ball} * g$
    - m = mass of the ball
    - V = Volume of the ball
    - ρ = density of the ball
- The buoyancy : $F_A = \rho_{liquid} * V_{ball} * g$
- The viscous friction force given by Stokes: $F_f = 6\pi * r * \eta * v$
    - r = radius of the small ball
    - v = constant fall rate
    - η = dynamic viscosity of the fluid

These forces are the ones that act in the case that a small ball of radius r without initial velocity is released in a recipient of radius R with the fluid that is studied. (r << R)

At the beginning of the fall, the small ball accelerates. It is suppose that the velocity of this ball is constant when it reaches a mark named A (see fig. 7.02.). Then between A and B the speed is constant.

When the speed is constant, the formula that governs the system is:

$$\sum \vec{F} = \vec{0}$$

And then if everything is replaced in this formula:

$$P - F_A - F_f = 0$$

So:

$$6\pi * r * \eta * v = V_{ball} * g * (\rho_{ball} - \rho_{liquid})$$

And then the Stokes formula to calculate the dynamic viscosity is found:

$$\eta = \frac{V_{ball} * g * (\rho_{ball} - \rho_{liquid})}{6\pi * r * V}$$

The dynamic viscosity can be calculated in a very easy way, with the help of this formula. During our experience, we decided that after five centimeters, the small balls had already a constant speed, so it was assumed that this was our point A as in the fig. 7.02. The distance that the small ball had to travel is 24cm, and here is the table with all the time results.

time (in s)	**30,48**	**31,66**	**31,9**	**29,03**	**31,35**	**30,03**
speed (cm/s)	0,787402	0,758054	0,752351	0,826731	0,76555	0,799201

Tab.7.01. Speed and time of the freefall of a ball in the fluid

Now every component of the Stokes formula is known. Applying this values to this formula, we obtain the dynamic viscosity of 22,6 Pa*s.

Something that is a bit problematic is the fact that the recipient, in which the small ball is released, has not infinite dimensions. And this can have some interactions with the fall of the

ball. And then influence on the value of the dynamic viscosity. And this problem is really true; the Stokes equation is only good for containers of infinite size. A correction can be made to calculate the real value of the dynamic viscosity; this correction is a geometrical parameter lambda.

$$\lambda = 1 + 2{,}1 * \frac{r}{R}$$

The Stokes equation with the geometrical parameter becomes:

$$\eta = \frac{V_{ball} * g * (\rho_{ball} - \rho_{liquid})}{6\pi * r * V} * \frac{1}{\lambda}$$

Applying the Stoke equation with the parameter lambda, the result of the dynamic viscosity is much lower: 18,4 Pa*s.

7.3. Determining the contact angles between fluid and cavities surfaces

The important values characterizing the fluid have been obtained. But COMSOL Multiphysics needs the properties of the walls on which the fluid is going to flow. These walls are called wetted walls and the Contact angle of each surface must be given. The contact angle is the angle at which a liquid interface meets the solid surface. The contact angle is specific for any given system and is determined by the interactions across the three interfaces.

In our case, the method to calculate the contact angle was really simple. With a camera, pictures of little droplets were taken on different surfaces, and with geometrical software, the angles were found. In the pictures that follow, two droplets on different surfaces.

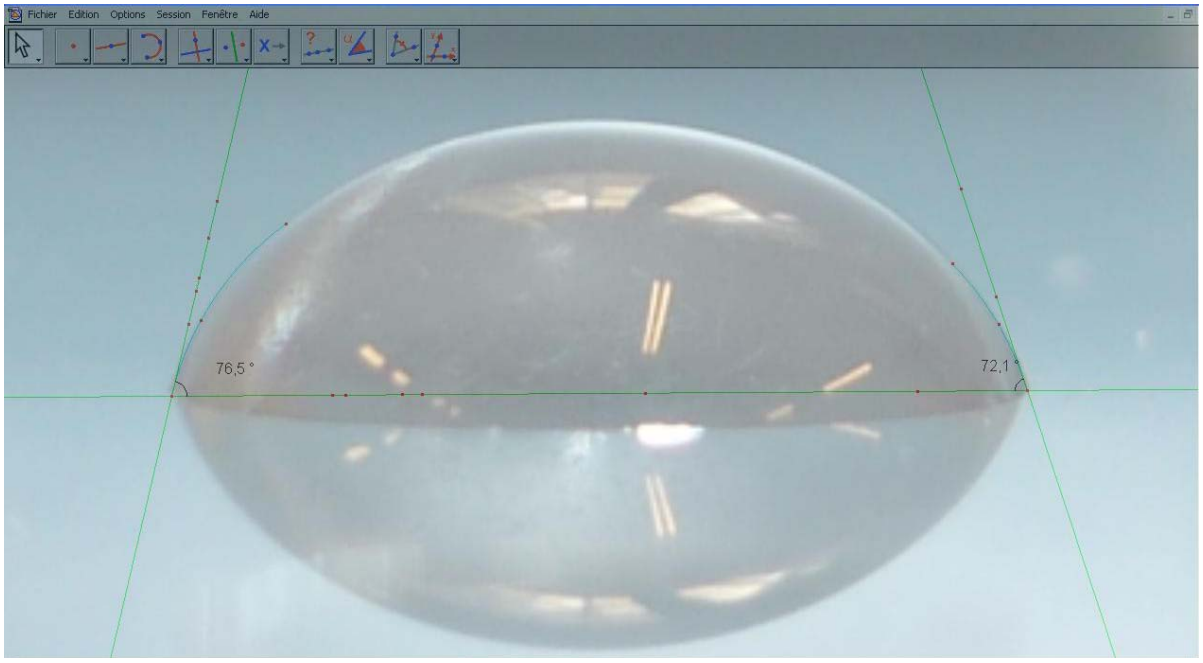


Fig.7.04. A droplet of CMC product on the upper surface.

On the upper and the lower surfaces, the contact angle measured was 74,3°. But the edges of the InjectionStation14 have another contact angle, because the edges were done with some machinery. The contact angle found for the edges is: 62,1°.

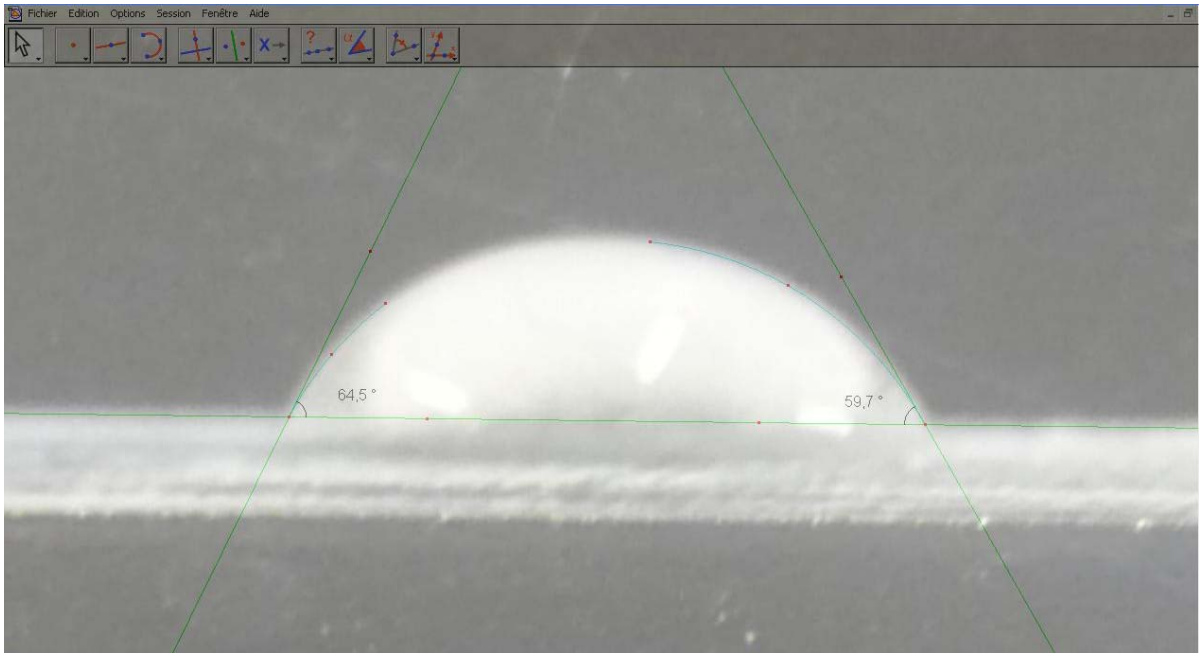


Fig.7.05. A droplet of CMC product on the edge surface done by cutting the PMMA.

The contact angle of the modeling clay was also deduced from this kind of pictures. The surface was plane, but a little rougher to simulate really fine the surface of the modeling clay inside the cavity.

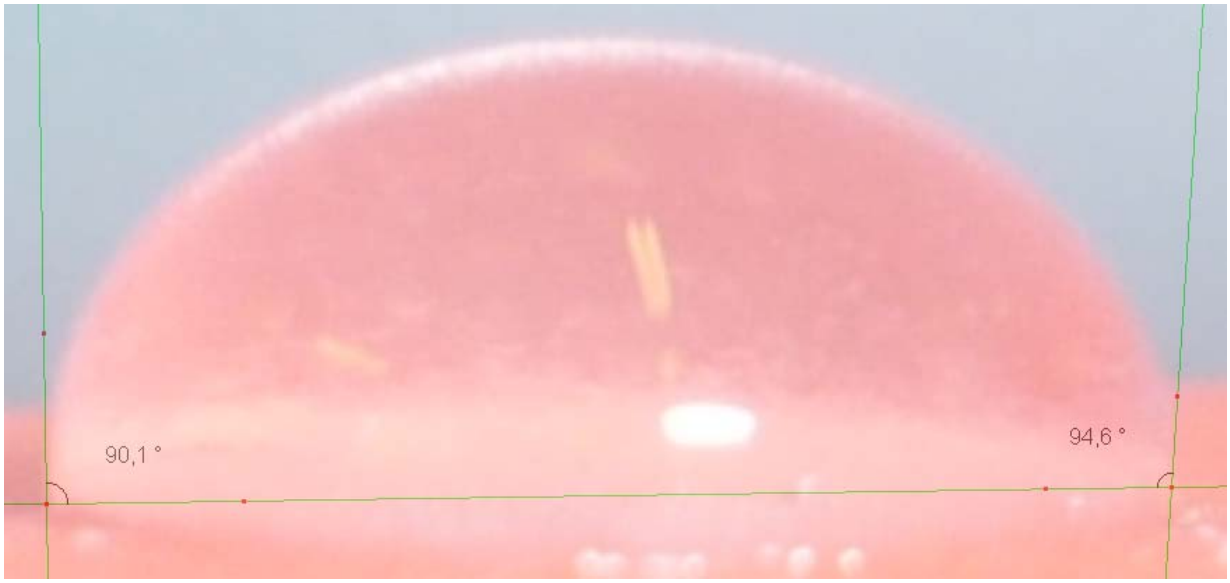


Fig.7.06. A droplet of CMC product on modeling clay surface.

The contact angle on this surface is assumed to be 90°, which is the default value that the program gives to the materials.

During the first model some problems appeared. One of these problems was that looking at the results, the interface between the materials and the CMC was the symmetry of what it is in the experience. That is because on COMSOL Multiphysics the value needed is 180° minus the value of the contact angle. Then the shape result and the interface are very good looking.

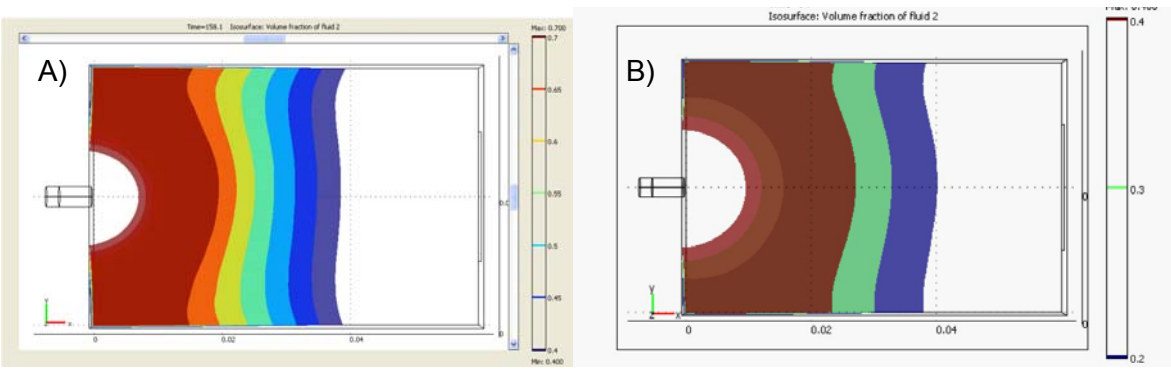


Fig.7.07. A) Model with the different front levels because with bad geometry B) Correction of the angle geometry on COMSOL

7.4. Summary conclusion

This chapter describe the properties of the fluid that COMSOL need to compute the models. These data are very important because all the fluid equations are based on these intrinsic properties. Then with all these initial conditions, the different computations could be run.

After some days of calculation, the results on COMSOL were found, and at first sight they were quite good. Let's now compare them to the experimental results.

8. Injection phase

8.1. Compression speed calculation

The purpose was to control the injection speed in order reproduce the real evolution of the flow. In the case of kyphoplastia the linear injection speed in the needle is of 1 cm/s.

We calculated the compression speed of the syringe to get the good linear speed in the output of the needle. Of course this calculation depends on the type of the syringe.

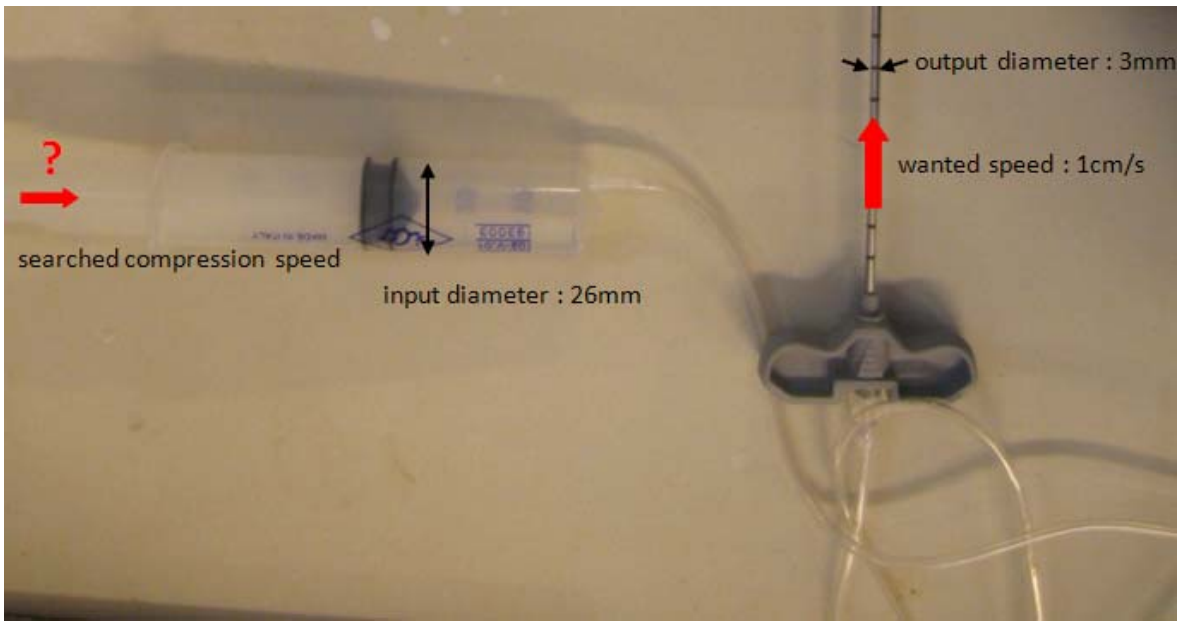


Fig. 8.01. Picture with various speeds and diameters in the syringe-needle assembly.

Output flow $Qo = Vo * So$ With $Vo = 1cm/s$ and $So = 0,07cm^2$

$Qo = 0,07cm^3/s$

Compression speed $Sc = Qi/Si$ With $Qi = Qo$ and $Si = 5,3cm^2$

$Sc = 0,0132m/s$

$Sc = 7,9mm/min$

The needed compression speed is 7,9 mm/min. To make it possible, a tensile machine has been used in a compressive mode with the syringe locked between the two grips.

8.2. Experimental assembly

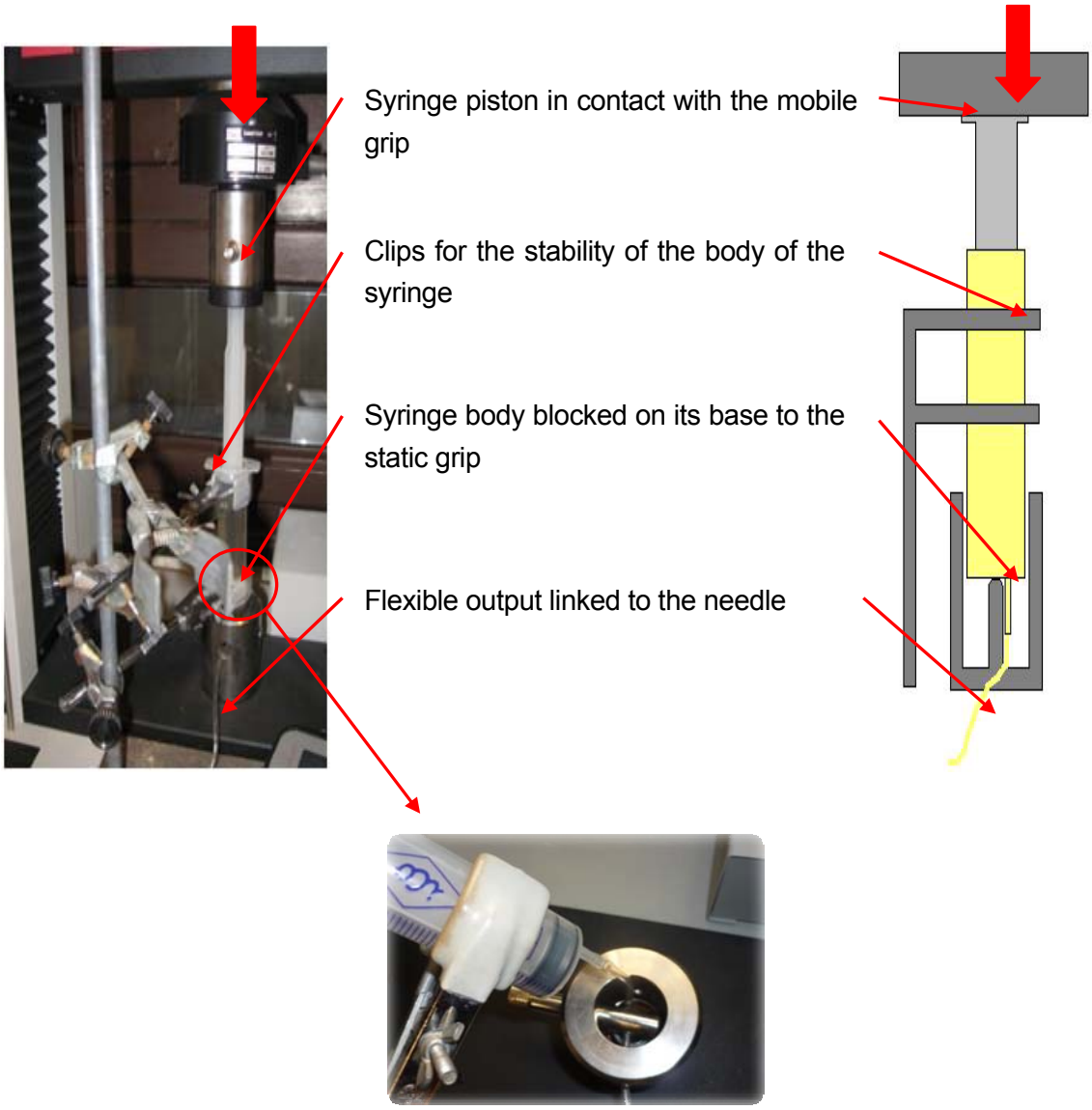


Fig. 8.02. Picture, outline and detail of the assembly

It is important on the experimental assembly to keep the needle and its piston very vertical during the compression phase in order to avoid transverse stresses which could break the piston because of torsion.

8.3. Tensile machine software: TestWorks

The tensile machine uses software which permits to create a compression program in order to automate a compression with constant and controlled speed on a definite distance. It also permits to see the evolution of the pressure in the syringe body.

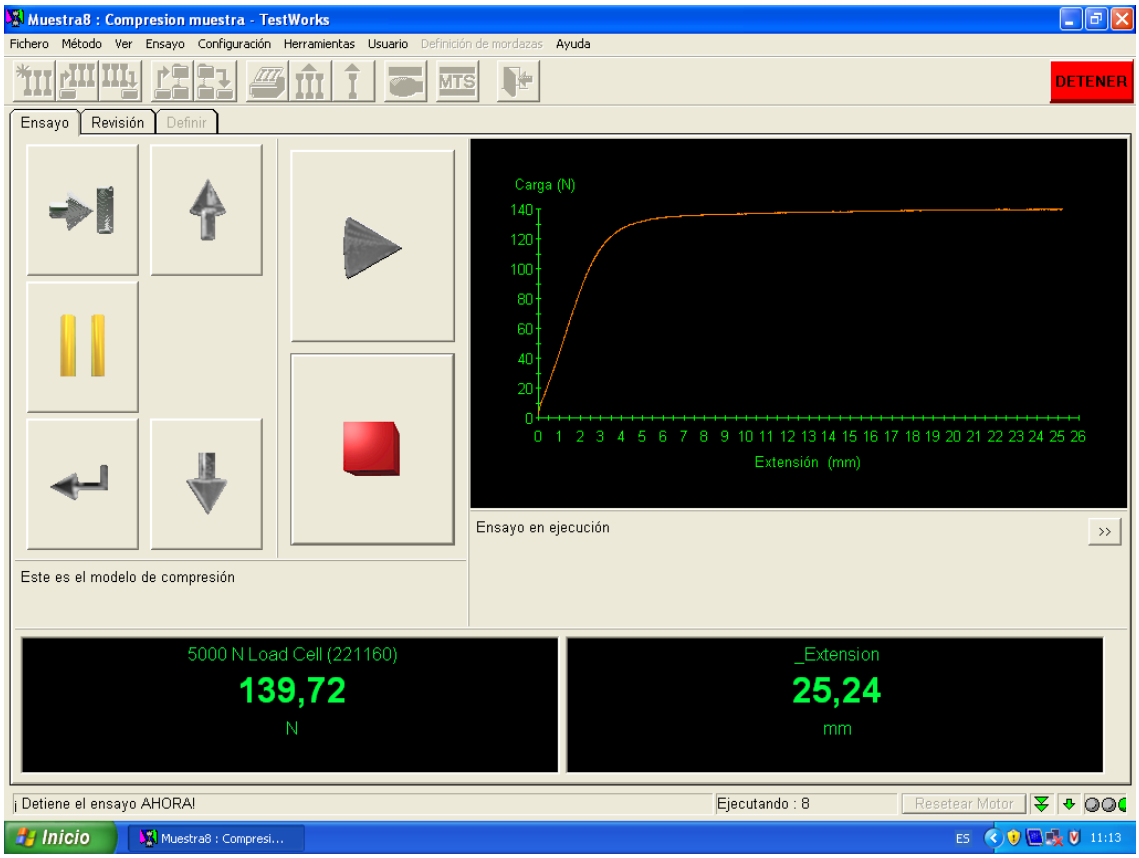


Fig 8.03. The interface of TestWorks permits to observe in live the value and the curve of the applied force. In our case, the applied force is, once stabilized after transitory phase, of 140 Newton.

Applied pressure on the piston joint $Pj = Fp * St$ With $Fp = 140N$ and $St = 5{,}3 * 10^{-4} m^2$

$$Pj = 2600hPa$$

$$Pj = 2{,}6bar$$

The applied pressure on the piston joint is evaluated at 2,6 bar which is a correct value in relation to the syringe mechanical capacity.

The pressure depends on the length of the cable. In this case its length is of 120 cm.

8.4. Summary conclusion

This chapter explains the method that is used to inject the liquid inside the cavities of the IS14. It describes also how the velocity values that were needed to have the same conditions on COMSOL and in the experimental part were calculated.

It also describes how to measure the pressure applied on the syringe. The calculation of pressure will be reused for the sponge filling.

9. Approximate the fluid velocity with COMSOL

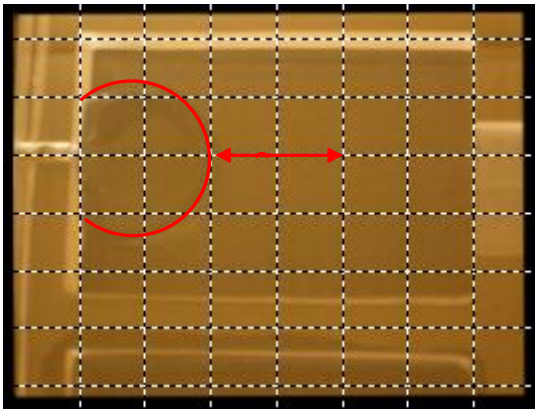
The phenomenon of diffusion which is added in a Level Set computation on COMSOL implicates that the velocity of the front of the fluid does not exist as a linear and well defined speed. The velocity depends on each isolevel. This property of the isolevels is used to compare the models as two well defined phase flow. As there is no possibility to determine directly the velocity of each isolevel, it is necessary to make some manipulations to associate the experimental velocity and the good computational velocity.

The idea was to:

a) Measure the experimental front velocity
b) Determine the computational velocities for several isolevels
c) Deduce the adequate isolevel which has the same speed as the experimental liquid front

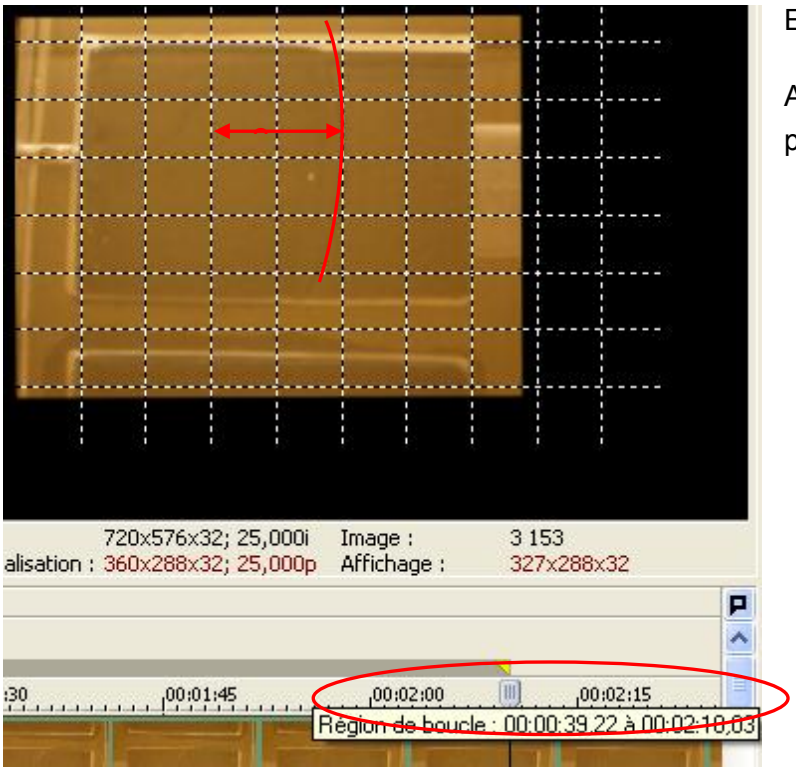
9.1. Experimental velocity measurement for the empty cavity

The velocity of the front of the fluid is **not** linear during the whole filling up of the cavity but it is supposed to be so on a short distance.



Start point:

Thanks the video software Sony Vegas Movie Studio a 1cm length grill is put on the cavity and a start point is defined.



End point:

At the end point, the time of progression can be read.

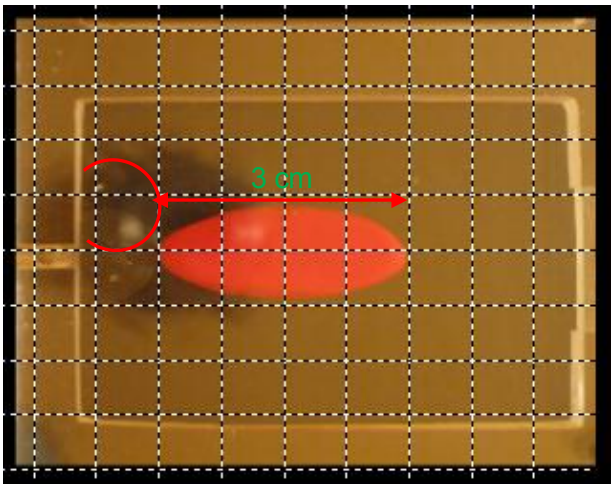
Fig.9.01. Screenshots of the video software to observe the progression of the fluid in the empty cavity.

Experiment	Distance (m)	Start time (s)	End time (s)	Run time (s)	Velocity (mm/min)
	0,02	39	130	91	13,2

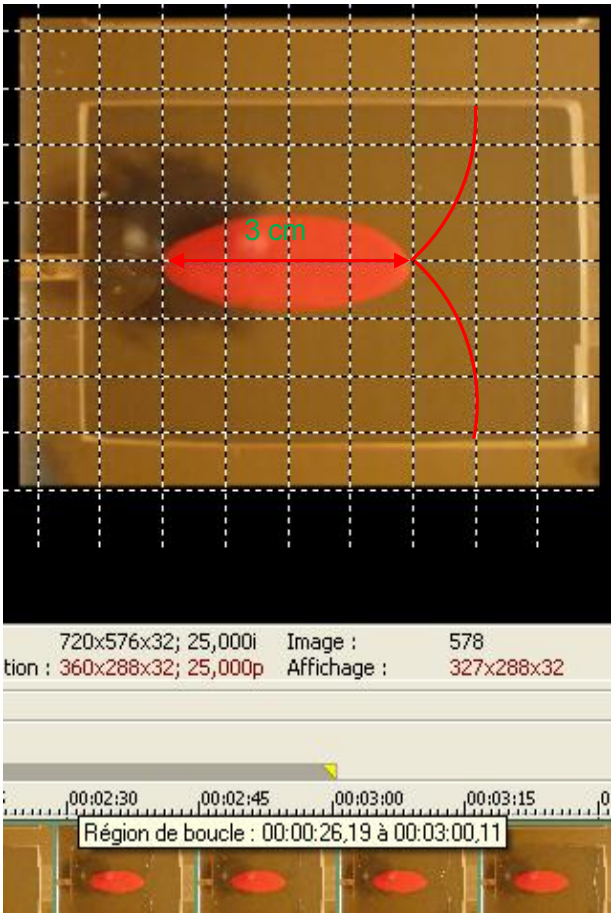
Tab.9.01. Parameters to determine the velocity of the fluid in the experimental case.

The tab.9.01. shows the start time and the end time. With this value and the distance stridden, the velocity could be calculated.

9.2. Experimental velocity measurement for the cavity with the obstacle



The start point is defined at the beginning of the modeling clay obstacle. The measurement is done on a 3cm length path.



The end point, is defined has the point when the left and the right flow in each part of the plasticine beginn to touch each other.

Fig.9.02. Screenshots of the video software to observe the progression of the fluid in the modeling clay cavity.

Experiment	Distance (m)	Start time (s)	End time (s)	Run time (s)	Velocity (mm/min)
	0,03	26,23	180,11	153,88	11,7

Tab.9.02. Parameters to determine the velocity of the fluid in the experimental case with plasticine.

This is the same case as before, but some plasticine has been putted in the cavity. This changes a little the value of the velocity. That is why; the value of the isolevel will not be the same as for the previous example.

9.3. Measurement of the computational velocities of several isolevels

The needed time to cover the same distance as the experiment is measured for several isolevels. For this, 5 isolevels were observed at the same time to determine the different velocities, and look which one will be more like-looking.

9.3.1. Deduction of the adequate isolevel for the empty cavity

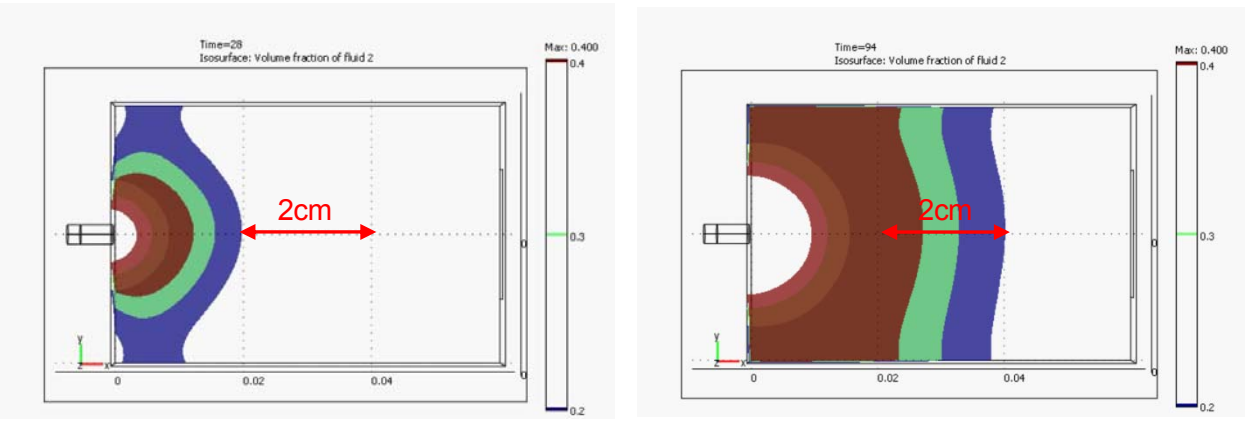


Fig.9.03. Cover times are read above the COMSOL video for each isolevel (above for the 0,2 isolevel in the empty cavity).

COMSOL	Distance (m)	Start time (s)	End time (s)	Run time (s)	Velocity (mm/min)
ISOSURFACE					
10%	0,02	17	61	44	27,3
20%	0,02	28	94	66	18,2
30%	0,02	42	126	84	14,3
40%	0,02	60	165	105	11,4
50%	0,02	80	208	128	9,4
60%	0,02	110	263	153	7,8
70%	0,02	155	338	183	6,6
80%	0,02	220	435	215	5,6
90%	0,02	330	570	240	5

Tab.9.03. Parameters to determine the velocity of the isosurface that could be compare to the experimental data.

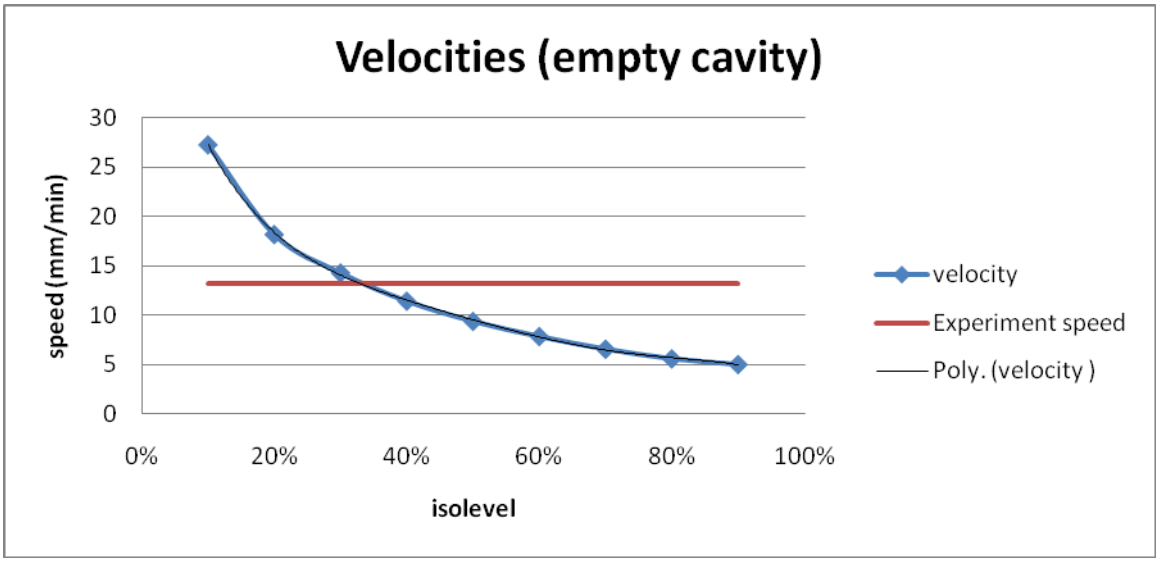


Fig 9.05. The experimental speed in the empty cavity is represented by the 0,33 isolevel. This value is confirmed by an order 5 quadratic approximating the computational velocity.

The tab.9.03. and the Fig.9.05.shows that the 33% isolevel is the best one to compare with the experiment done in a empty cavity.

9.3.2. Deduction of the adequate isolevel for the modeling clay cavity

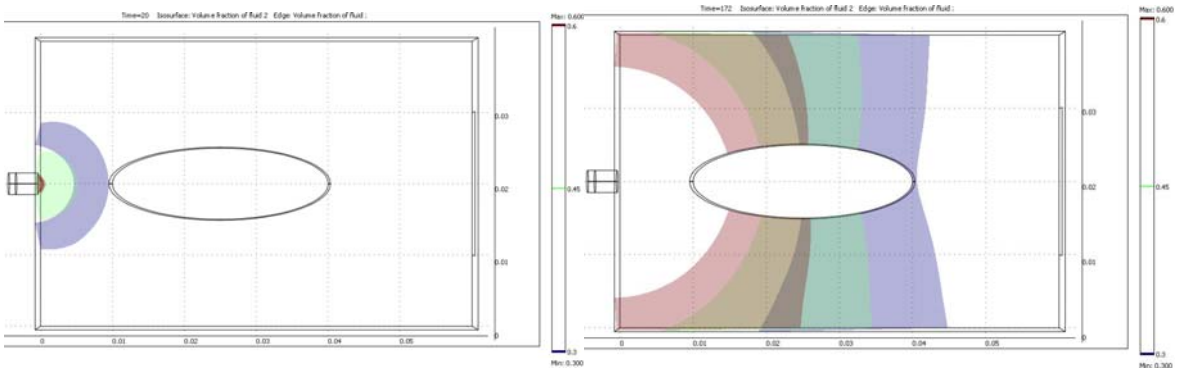


Fig.9.06. cover time read for the 0,3 isolevel in the cavity with modeling clay.

COMSOL	Distance (m)	Start time (s)	End time (s)	Run time (s)	Velocity (mm/min)
ISOSURFACE					
20%	0,03	14	130	116	15,51724
28%	0,03	19	164	145	12,41379
30%	0,03	20	172	152	11,84211
32%	0,03	22	180	158	11,39241
44%	0,03	33	232	199	9,045226

Tab.9.04. Parameters to determine the velocity of the isosurface that could be compare to the experimental data.

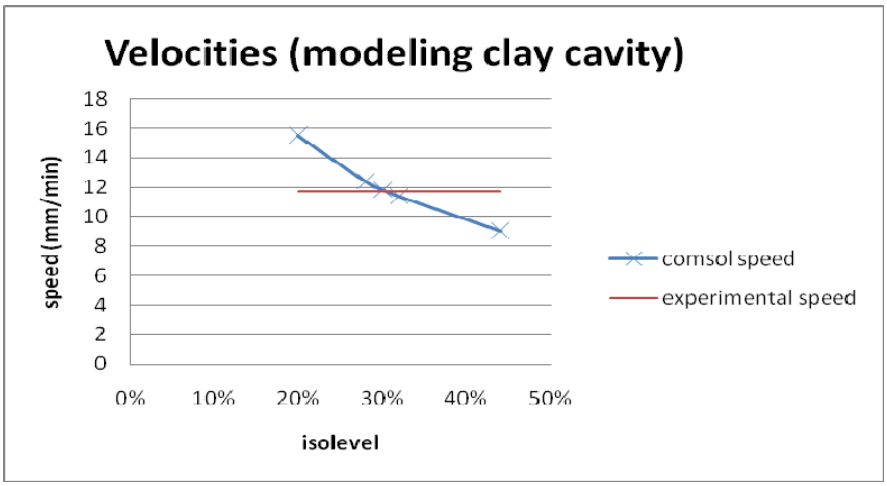


Fig.9.08. The experimental speed in the modeling clay cavity is represented by the 0,31 isolevel.

In the case of a cavity with an oval of plasticine inside, the isolevel found is 31%.

9.4. Summary conclusions

To compare with efficiency the results given by COMSOL and the experimental data, some isolevel values have been found for the two different shapes. Even if the isolevel which represents the best the real shape of the fluid is not the same in both cases, it can be said that the values are very similar one to each other. This value stands around the isolevel 0,3.

We invite you to visualize the final results on the videos “empty cavity” and “modeling clay cavity” given in annex A folder.

10. Porosity

One of the mains purposes of this paper is to show how the hydroxyapatite flows in a porous bone. This can be done by modeling a little piece of porous bone on COMSOL Multiphysics. And once some results have been obtained, find a way to model the flow of hydroxyapatite inside the porous vertebra, like running a model which involves Darcy's law or the Brinkman's equations. These equations will be explained below.

The first thing to do is to search all the values that will help to characterize the bone porosity, and will be needed by COMSOL to run the models.

Looking in the scientific literature, some very important characteristics values could be found.

The dimension of the porosity of a bone is very often between 100 and 2000μm. The model should then have a porosity of this kind of dimension.

Darcy's law and Brinkman's equation use both of them a concept called permeability. The permeability is a measurement of the ability of a porous material to transmit fluids. This value is very common in fluids and earth science as it means a porous medium. In the case of bone, this value of permeability is 1.35e-13m² **[19] [20] [21] [22]**.

Once these values are known a model similar to the bone structure can be modeled. But as the computational power is not sufficient to run this kind of problem in 3D, the model has to be in 2D and cannot be really big. An illustration of this: for a simulation of vessels in a structure of 6*4mm², COMSOL needed 16773 seconds to compute it. It is really long for a very small problem. So this simulation is just an approximation of the real thing. Because it is well known that bone is a 3D structure, and when osteoporosis appears, the bone structure gets more porous.

10.1. Level-set on a micro-porous material

The model drawn on COMSOL has the following dimensions: 6mm*4mm. Some channels where the liquid is going to flow are modeled inside.

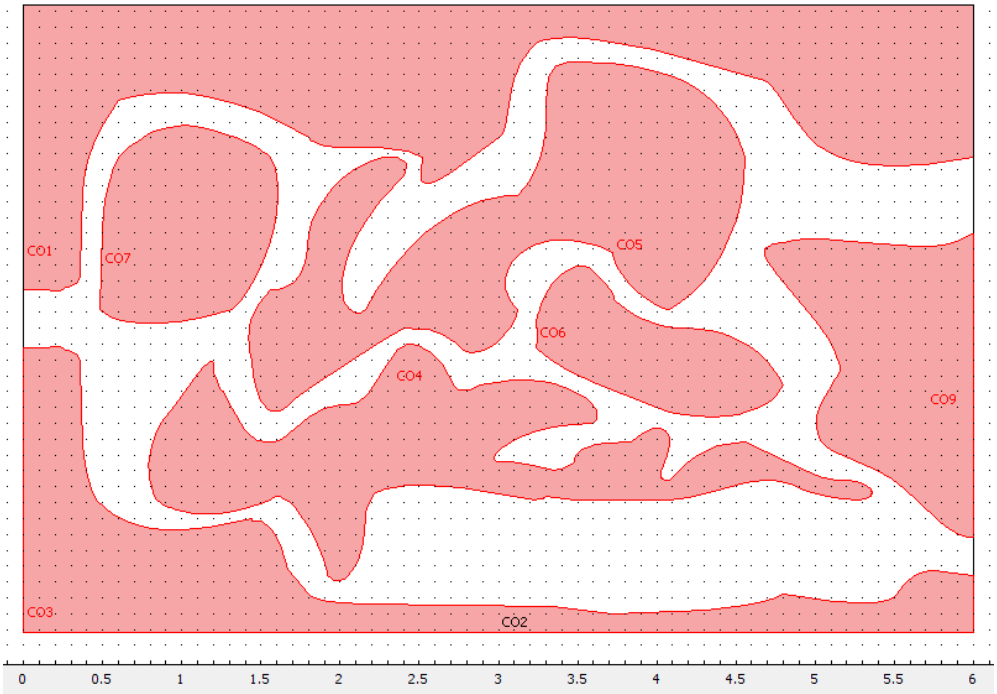


Fig.10.01. Modeling of a porous micro-bone

Looking to the dimensions of the channels, those are between the dimensions found on the scientific literature. This model was realized from the picture above, which is a micrograph of a section of human bone.

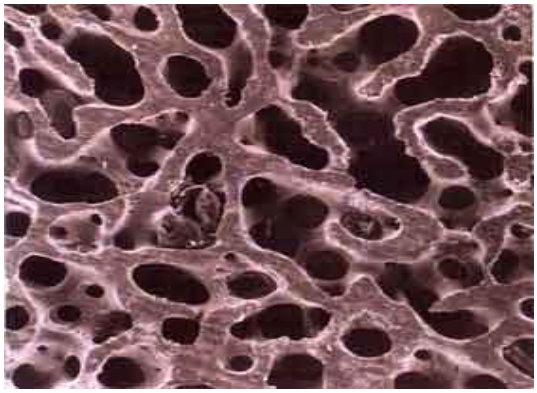


Fig.10.02. Section of human bone

The resolution of this model was run on the Level Set Method of COMSOL Multiphysics. But this resolution of the problem even if it is a small one, needs plenty of computational resources. It took several hours to finish solving the entire problem and needed around 700Mb of RAM.

The second area of research in the porous material is to see how the flow behaves in this kind of material. As it was said before, two laws can be applied: Darcy's law or Brinkman's equation **[23] [24] [25]**. Darcy's law is a phenomenological derived constitutive equation that describes the flow of a fluid through a porous medium. This law is used in a variety of applications like hydrogeology, to describe oil, gas and water flows in the petroleum tank. Darcy's law is a simple proportional relationship between the instantaneous discharge rate through a porous medium, the viscosity of the fluid and the pressure drop over a given distance.

$$Q = \frac{-\kappa A}{\mu}\frac{(P_b - P_a)}{L}$$

With Q representing the units of volume per time, K is the permeability of the porous material, A the cross sectional area, P_b-P_a is the pressure drop flow occurring from high pressure towards low pressure (opposite the direction of increasing gradient. Hence the negative sign in Darcy's law). μ is the viscosity of the fluid and L the length on which the pressure drop occurs.

On the other hand brinkman's equation is an extension of Darcy's law. This extended equation has an effective viscosity term. This correction term is used to see the flow through environments where the grains of the material are porous themselves.

Above the Level set of this section is done. All the cavities are filled, but some ones remain empty. The model is very interesting, but the problem is that hydroxyapatite is made from alpha TCP so small grains, and this can create some agglomerate. Those agglomerates are going to clog the pores, and our purpose is to model an obstruction by this agglomerates.

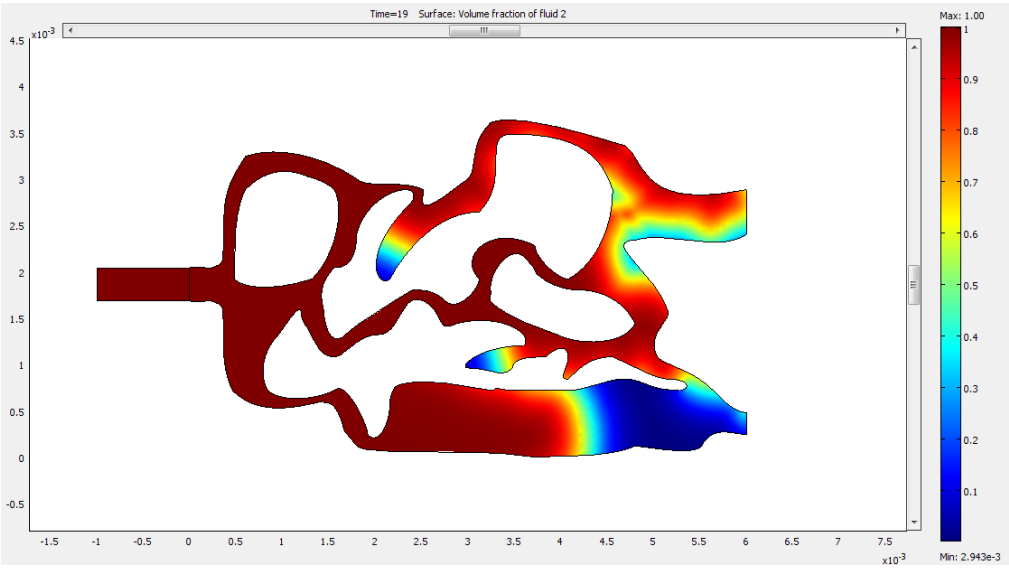


Fig.10.03. analysis of the results of flow in porous micro-bone

In the example above, the fluid was coming from the left and was going out by two places of release.

10.2. Comparison between COMSOL results and experimental data

To see the obstruction of the vessels by the agglomerates, a different porous model has been made by a 3D printing. A 3D model could not be done at the real bone scale. All the dimensions were multiplied by 10. The Porous model was previously designed on Catia. It comports a first layer of material with a thickness of 4mm and a second layer that has the same shape as a human bone. This layer had a thickness of 6mm. Two entries with a diameter of 3mm and one out were done in two opposite sides.

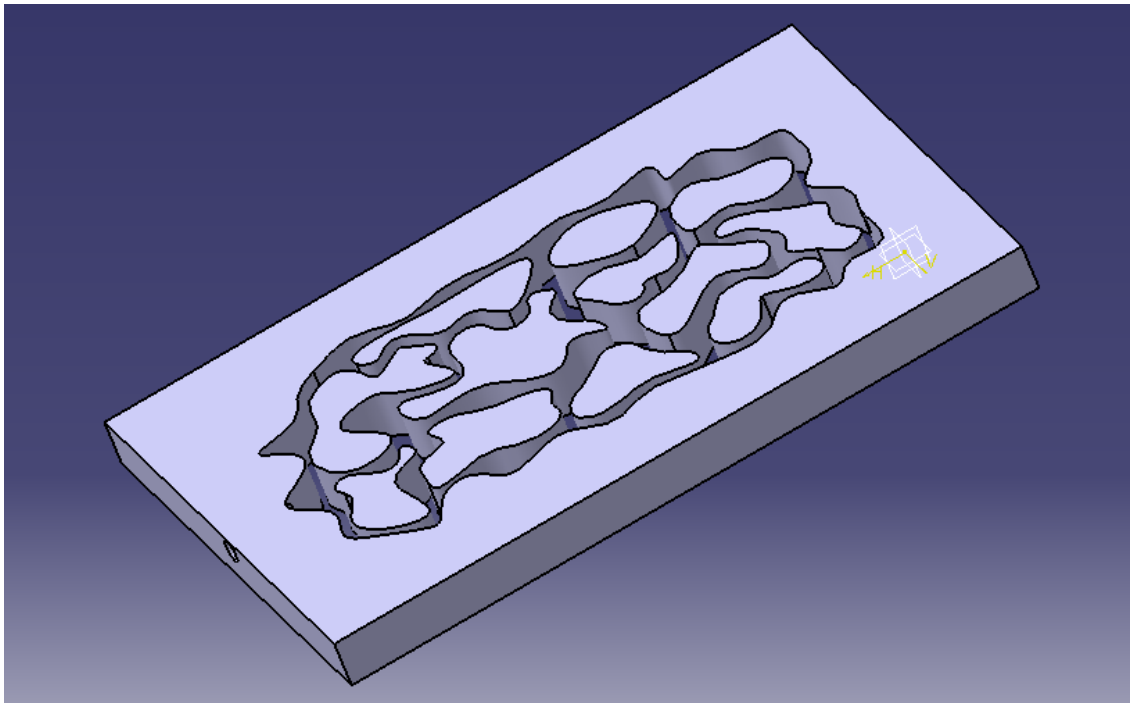


Fig.10.04. Modeling of a porous micro-bone for 3D printing

This model (fig.10.04) was recreated on COMSOL Multiphysics. As a Catia model is not compatible to run on COMSOL, a 3D model has been designed on Catia, and has been opened on COMSOL to extract a 2D shape. This was done with the help of the third degree Bezier curve, as shown in the fig.10.05. This work is not really pleasant, but the results at the end were really good.

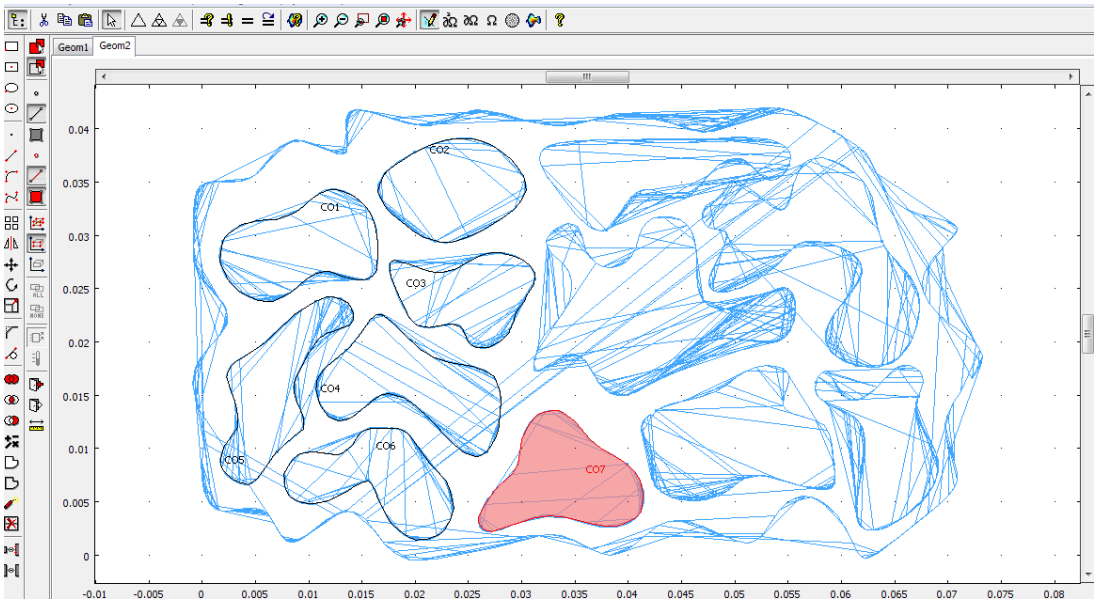


Fig.10.05. Reproduction of the Catia model on COMSOL

Once the model on COMSOL was done, the calculus could be made. And the 3D printed model could also be done thanks to the Catia file. Two 3D printed models were done in a 3D printing machine with the help of the software SD view. This 3D printing machine works sticking plastic layers with a thickness of 0,16mm on each other, and cut the shape on it.

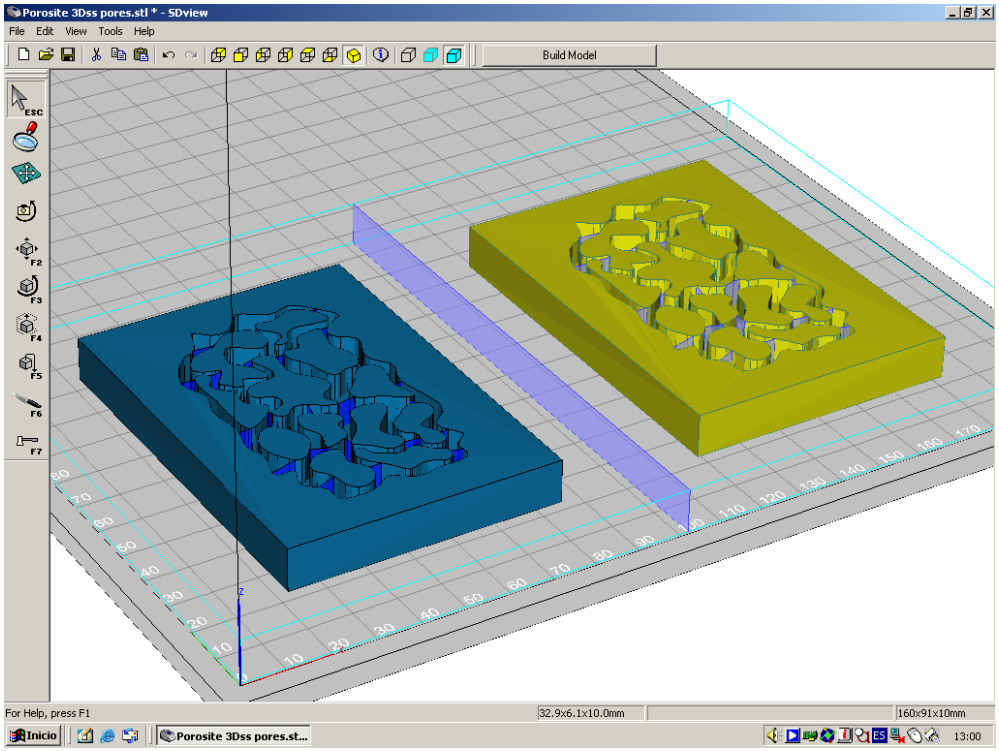


Fig.10.06. Software to run the 3D Printing machine

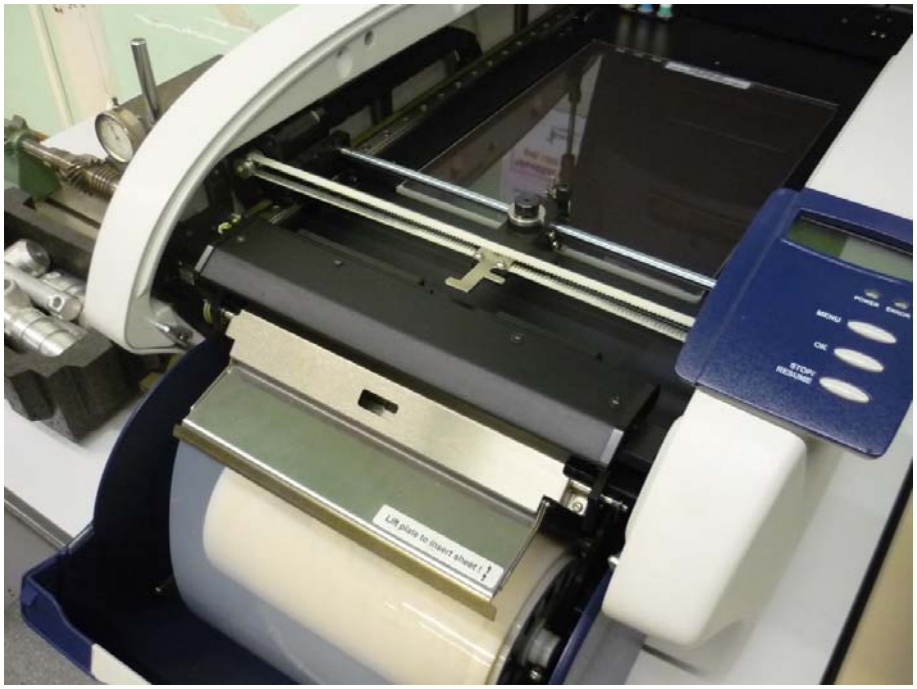


Fig.10.07. Picture of the 3D printing machine

In the places that have to model a compact material, the different layers are glued. But for the places where the fluid is going to take place, anti-glue is used. The printing was done in 6 hours.

But these models were not ready for the experiments. Once those two models were finished, the inside shapes had to be removed with tweezers. This work is very meticulous and long, and has to be done with patience.

Fig.10.08. One of the two 3D printed models after having removed all the unnecessary parts.

10.2.1. The model on COMSOL and the results

The fluid that is used in this example is going to have the following properties:

Viscosity: 0,3 Pa.s

Density: 1025 kg/m^3

On the software COMSOL Multiphysics, the aim is to insert some particle tracing. This will allow us to see the most integrated paths. First the model has to be executed, and the result is in the picture next page.

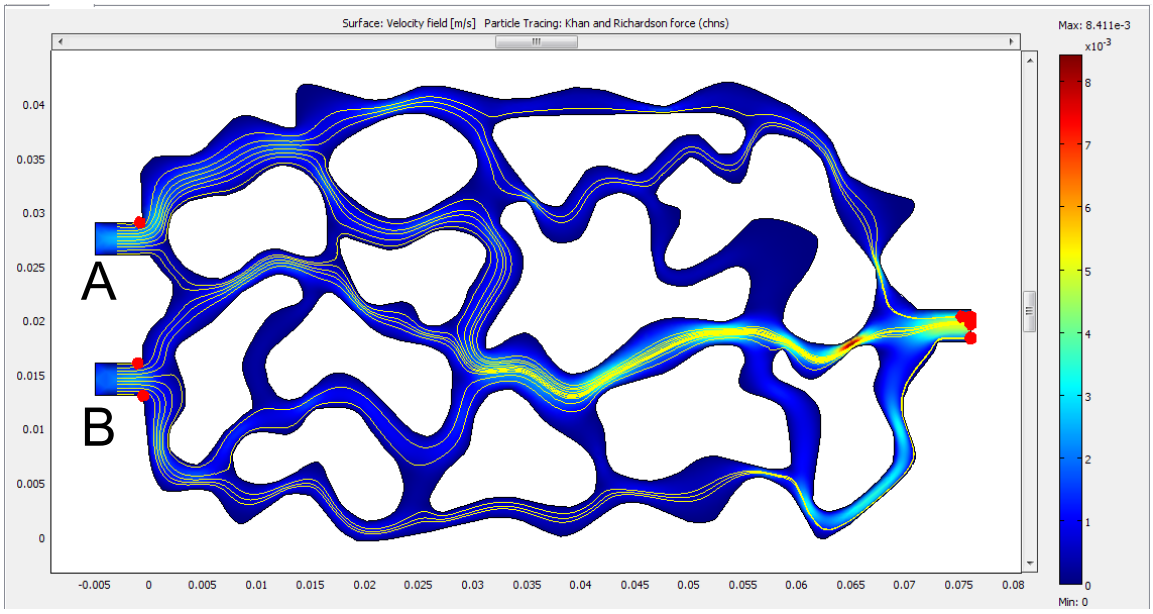


Fig.10.09. Representation of the particle tracing in the porous bone model

To compare the data given by COMSOL with the experimental data, we used on COMSOL a function that is called particle tracing. This function allows adding particles that will follow the path given by the energy of the fluid. This is not the more easy function to use on COMSOL, but just some values have to be input on the program for this operation.

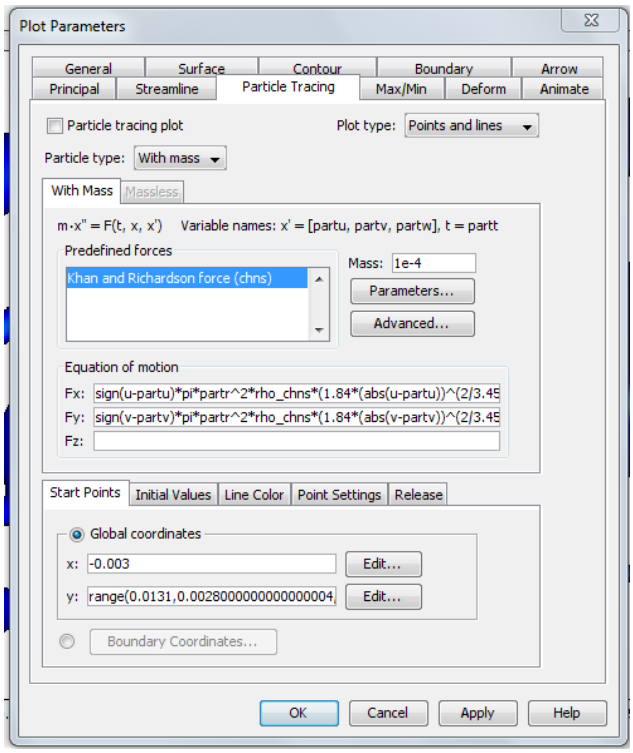
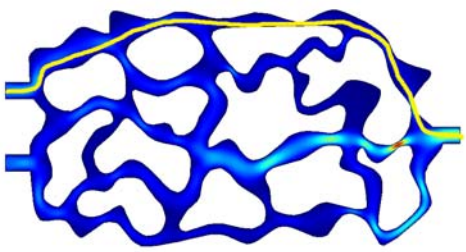
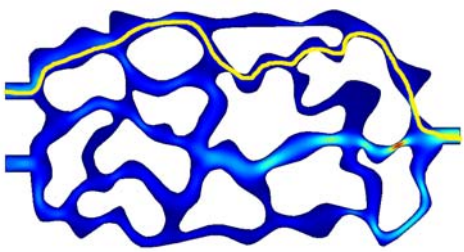
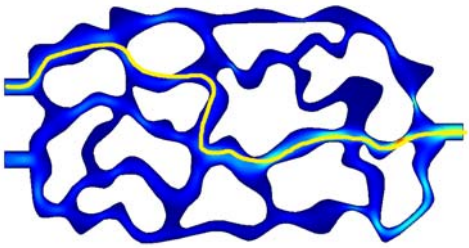
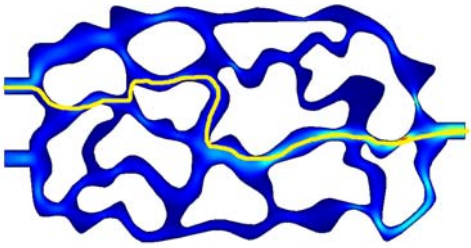
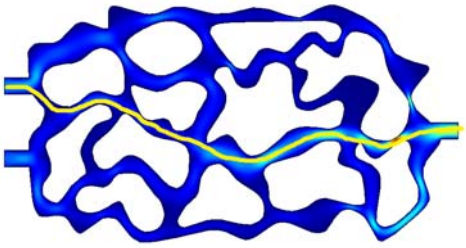
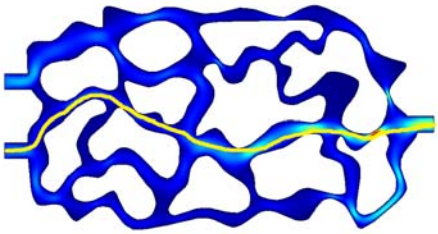
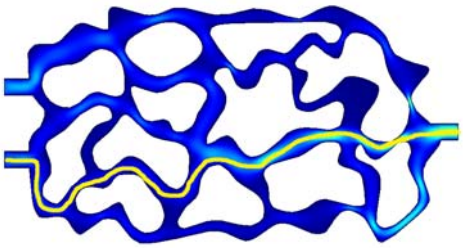


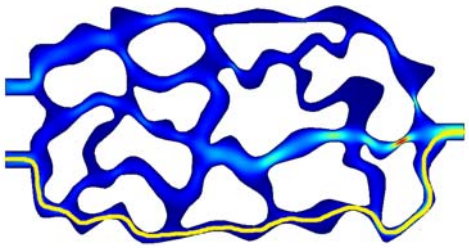
Fig.10.10. Properties of the particle tracing in the case of the porous bone model

These values are the properties of the particles, like the dimensions, the mass. At time 0 those particles should be in one specific place of the 2D space, or in other cases in the 3D space. These particles will follow the fluid direction, but the real direction of the fluid is given by some equations of motion. In this case the equations of motion that are used are the equations of Khan and Richardson forces. These equations define how the particles will be drag by the velocity of the fluid.

To compare the COMSOL model and the experimental data, a statistical model is built: ten particles with a mass of 0,1g are disposed in each outlet. The way that they are going to follow is recorded. In the COMSOL model, three particles crashed on the walls without any explanation. So the value of useable data particles is seventeen. We have two outlets in this model: the upper one named outlet A and the down one named B. Nine particles were effective in the outlet A, and eight in the outlet B.

The path taken by the particle	Number of particles on this way	Probability for the particle of following this path	Coming from the outlet A	Coming from the outlet B
	1	5,9%	11,1%	0%
	2	11,8%	22,2%	0%

	4	23,5%	44,4%	0%
	1	5,9%	11,1%	0%
	2	11,8%	22,2%	0%
	3	17,6%	0%	37,5%
	2	11,8%	0%	25%

	3	17,6%	0%	37,5%

Tab.10.01. Probability for the fluid of following different ways

In this table all the probabilities for the fluid of following one path or another are given. These probabilities are given by the data afforded by COMSOL. And the aim is to see if it is comparable to the experiment in the 3D model. For this an experience was realized. The velocity in each outlet was of 0,19mm/s.

10.2.2. The same experiment but using 3D models and a specific fluid

First of all a new fluid was elaborated. The purpose in this case was to obtain a fluid with just a little more viscosity than the water. As the precedent fluid for the injections in cavities, we used Carboxymethyl cellulose (CMC). At the end the fluid had 1,5% in mass of CMC. This multiplied by 30 the viscosity of the water. The porous model was filled with this liquid.

To differentiate the liquid inside the model and the liquid that was injected to see the paths, methylene blue was used. This liquid with blue methylene can be injected inside the injection tube with a little syringe of 10ml that has a very fine needle. The injection of the transparent liquid is done by a syringe of 60 ml, so 74 minutes of injection can be done without having to stop the injection. The fluid is injected at a velocity of 0,013ml/s. During the fluid injection, the injection of the blue methylene is followed by human eyes and recorded with a video camera.

Before starting, the needle punctured the tube and was not moved from there since the end of the experimentation. The fluid was then injected in the model and after 2 minutes, the methylene blue was injected in just one of the outlets. This is because it is easier to see the blue coming from just one of the outlets than coming from the two of them at the same time. It was necessary to wait that the model was completely full of liquid before to inject the methylene blue in order to observe a continuous and well established flow.

Once the injection is done, the paths that follow the blue methylene can be seen. The pictures below show the lines done by the blue methylene.

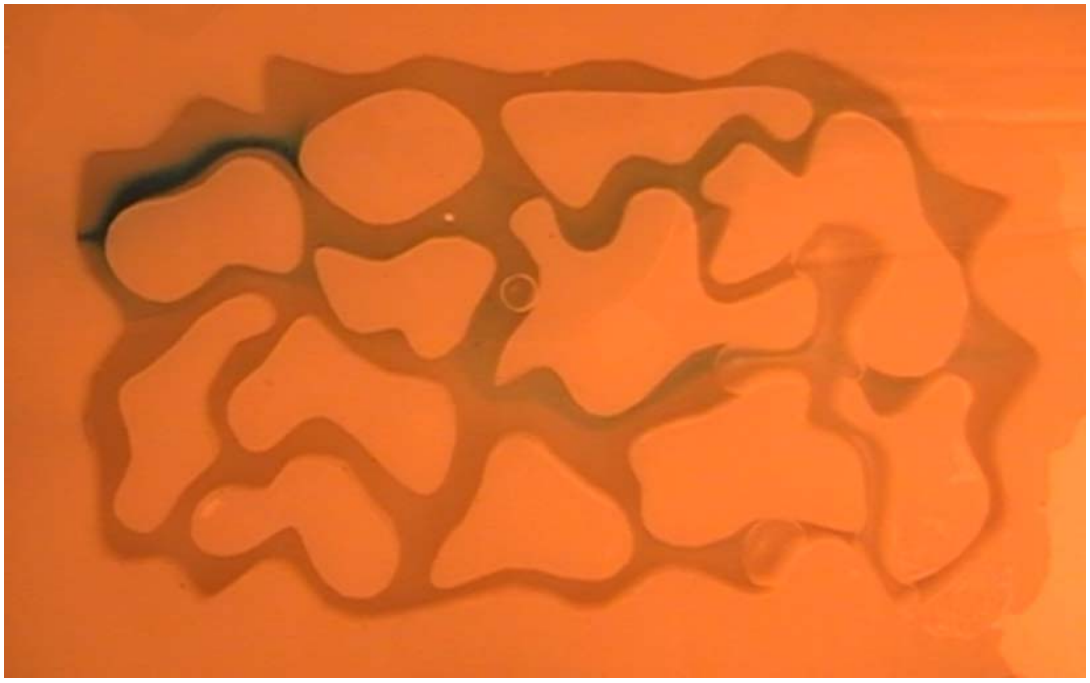


Fig.10.11. Methylene blue flowing inside the 3D printed model at time t=0s. Outlet A

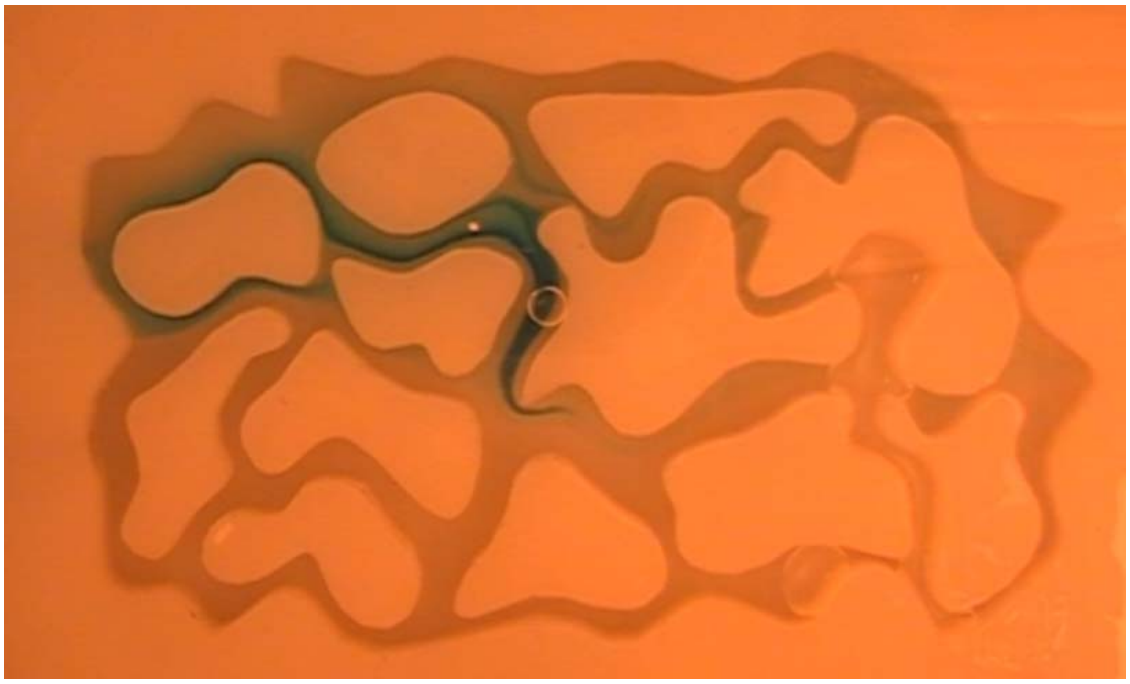


Fig.10.12. Methylene blue flowing inside the 3D printed model at time t=60s. Outlet A

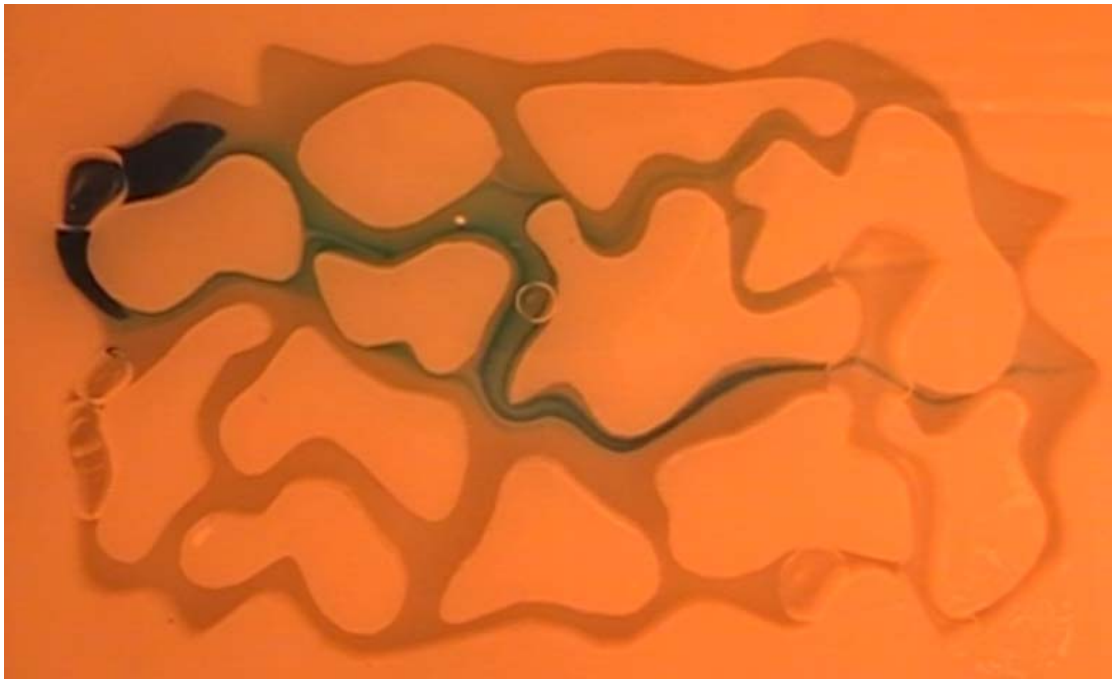


Fig.10.13. Methylene blue flowing inside the 3D printed model at time t=94s. Outlet A

Looking at the pictures above, and comparing these pictures to the statistical tab.10.01., it is possible to see that COMSOL predicted very well the way that the fluid was going to follow. Indeed in those pictures, the methylene blue coming from the outlet A is more visible in the path that has 44% of probability to be followed. And the other minority probabilistic ways are also represented in a clearer blue in those pictures. (To compare the COMSOL probabilities by yourself with the experimental results, please report to the video: “fluid flow tracing in a 3D printed model with methylene blue used as tracer in Outlet A”)

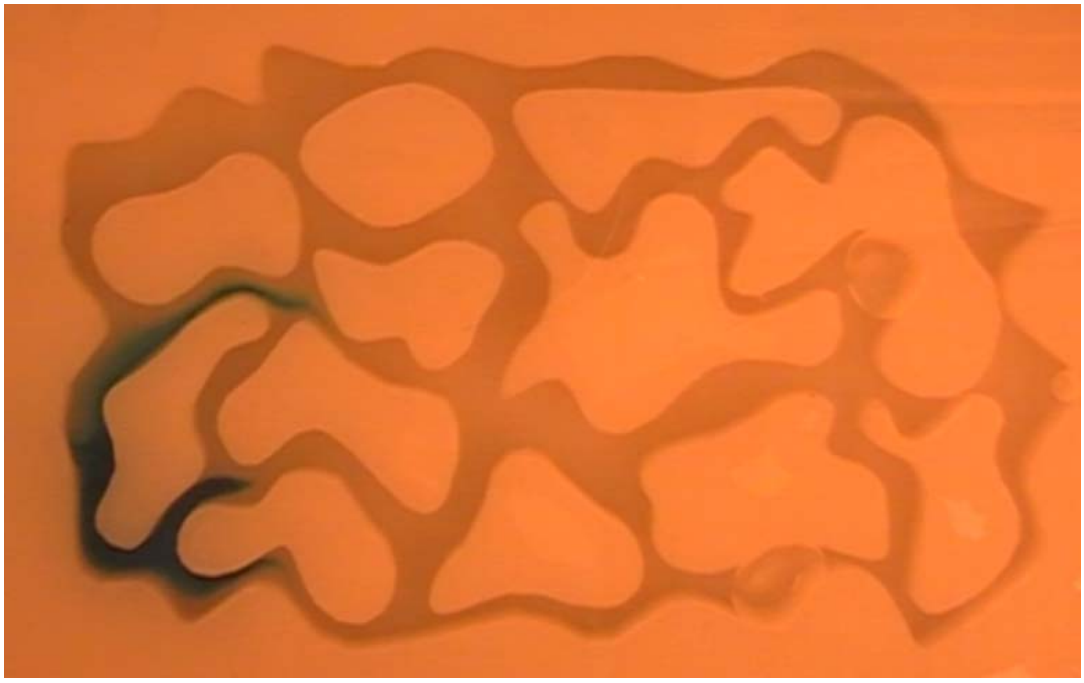


Fig.10.14. Methylene blue flowing inside the 3D printed model at time t=0s. Outlet B

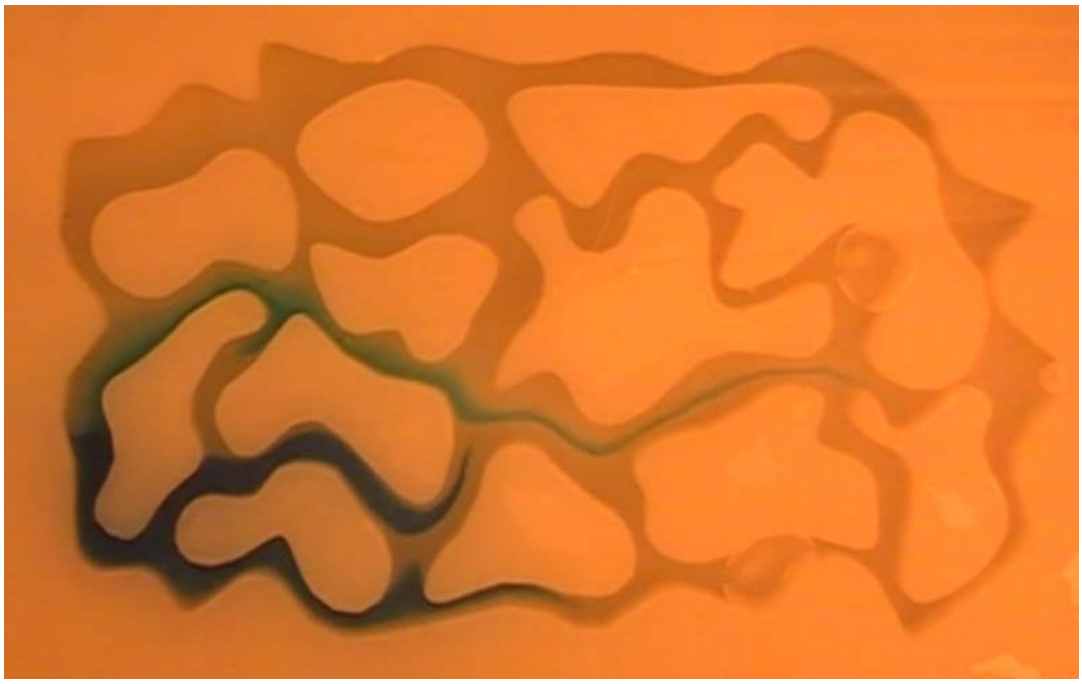


Fig.10.15. Methylene blue flowing inside the 3D printed model at time t=45s. Outlet B

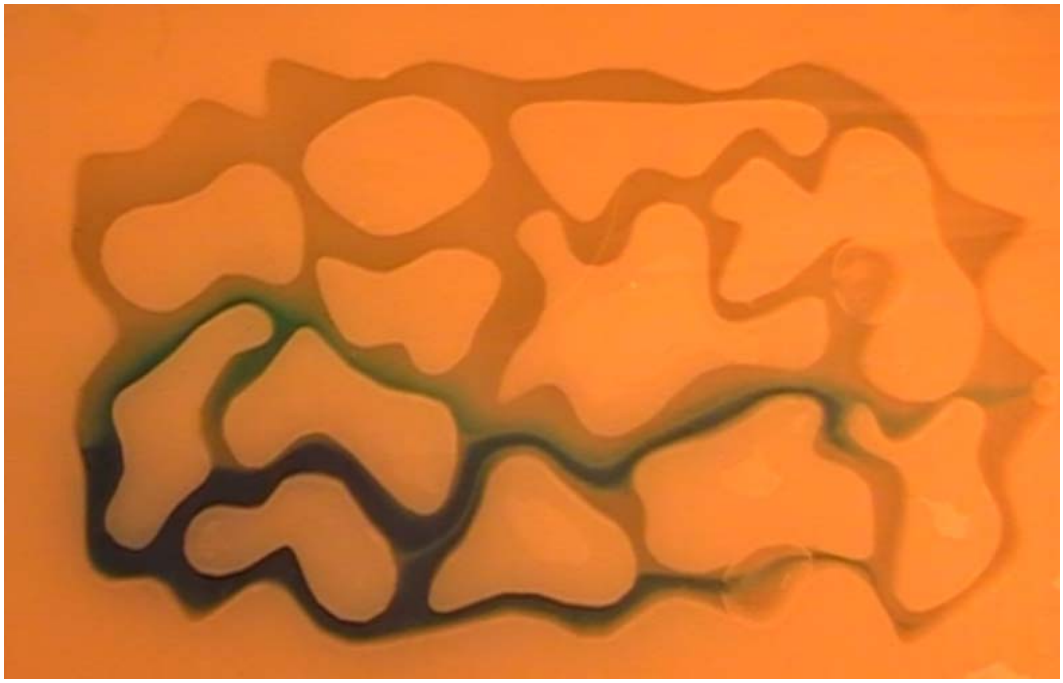


Fig.10.16. Methylene blue flowing inside the 3D printed model at time t=77s. Outlet B

Those three pictures above are from the movie taken of the injection in the outlet B. Once more COMSOL was right, the paths that were indicated by the probabilities as the more followed seemed to be taken by the methylene blue in the experiment. The path in the middle seems to be followed by much more fluid than the upper one, and it is the only difference with COMSOL. Even if a little of blue can be seen in some paths that were not indicated by COMSOL. But those paths can be neglected, as we will see after with the analysis of the pressure head. (To compare the COMSOL probabilities by yourself with the experimental results, please report to the video: “fluid flow tracing in a 3D printed model with methylene blue used as tracer in Outlet B”)

10.3. Other interesting results

Something interesting to observe on COMSOL is the stagnant zones. With a certain method the dynamic of the fluid can be seen, in other terms the intern energy of the fluid due to the pressure exerted on the container. The physic term used for this is the pressure head. Mathematically it is expressed as the following expression:

$$\Psi = \frac{p}{\gamma} = \frac{p}{\rho \cdot g}$$

Where Ψ is is pressure head (Length, typically in units of m);

p is fluid pressure (force per unit area, often as kPa units);
γ is the specific weight (force per unit volume, typically N/m^{-3} units);
ρ is the density of the fluid (mass per unit volume, typically kg/m^{-3});
g is acceleration due to gravity (rate of change of velocity, given in m/s^{-2});

In the picture below, this pressure head is modeled. The paths most integrated by the fluid have a greater slope than the stagnant surfaces. So it is easy to see where the fluid is going to flow, and also which path the particles will integrate.

The aim at this point is to see the behavior of some particles inside the model. And then compare the results with those given by COMSOL Multiphysics.

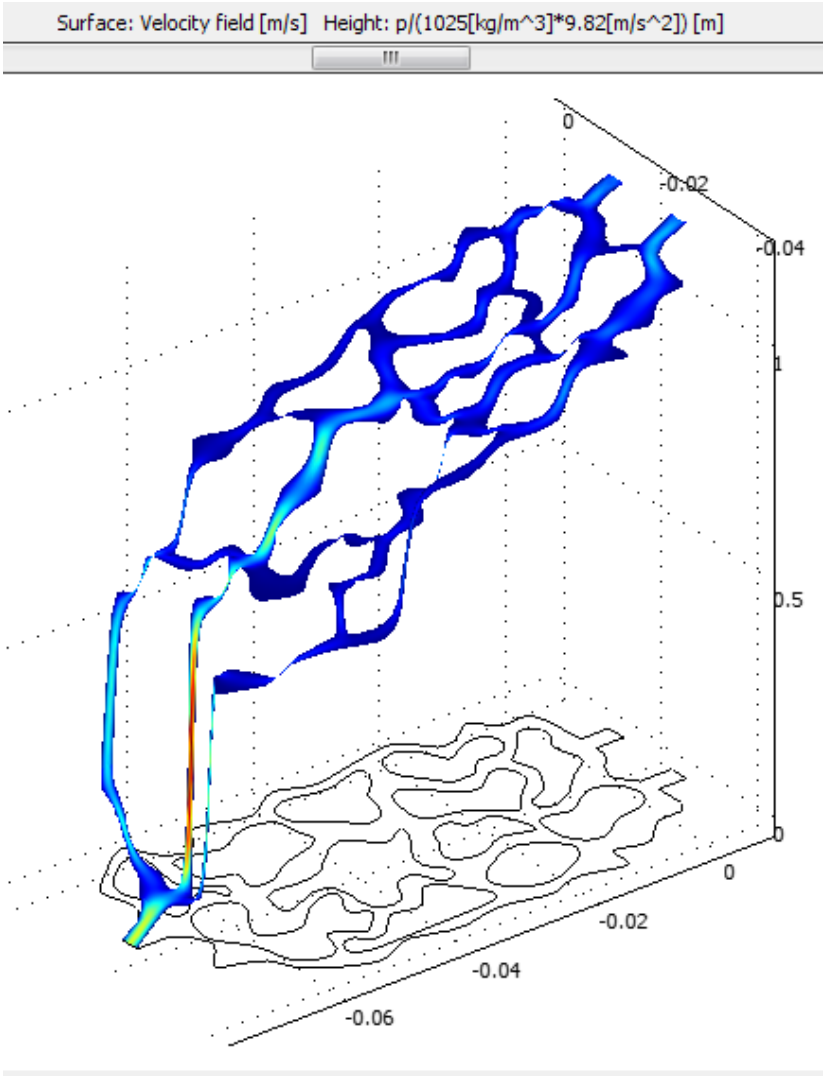


Fig.10.17. Representation of the pressure head in the porous bone model

10.4. Summary conclusion

This chapter makes an introduction on the phenomenon that affect the fluid flow inside a porous media as a bone. First of all the results given by COMSOL seem to fit with the experimental results. The accuracy of the program computation compared with the experimental data is pleasantly surprising.

It has been decided to go forward with porous media and to make some injections inside sponge. Injections of simple fluid and of fluid comporting solid particles have been made.

We invite you to visualize videos relative to particle tracing from the additional videos folder in annex A.

11. Experimental opening: filling of porous cavities

11.1. Purpose

The last part of the experimentations consisted in a filling of cavities comporting a sponge. The purpose was to recreate a porous environment. The filling has been realized with a 5,5% in weight CMC added fluid and a 5,5% in weight CMC and 5% in weight sand added fluid. The presence of sand has an interest in the visualization of particles progression when moved by the fluid convection but disturbed or stopped in their movement by the porosities of the sponge.

Experimental data and curves given by the tensile machine have been exploited in order to find relevant results about applied pressure.

Sponges have been cut after filling when using color-free fluid in order to observe the repartition of the sand in the volume. In some cases methylene blue has been added to facilitate the visualization of the fluid progression during video recording.

11.2. Encountered problems with the pressure

The 5% fluid used for the empty cavities filling has been used in a first time to make the experiments. Some results and values were uncertain.

On one hand the falling ball technique (see chap.7) gave a lower viscosity for the fluid containing sand 10,6 Pa.s, instead of 18,4 Pa.s for the fluid without sand which means that the presence of sand reduces the viscosity of the fluid. This may have a logic explanation with the fact that inert particles of sand are breaking three dimensional links between CMC particles, reducing the ability of the fluid to resist to shear stresses.

But on the other hand the pressure curves recorded when filling were either giving very similar results for both types of fluids or giving non reproducible results for the filling of the empty cavity with the fluid alone and also non reproducible results for the filling of a sponge with the fluid with sand. Moreover, by measuring the speed in the cavity with the video software Sony Vegas Pro, there was no visible change in the way the fluid progressed in the cavity.

However the falling ball measure had not been made in the same container and could be a source of mistake.

Moreover the pressure curves are very responsive to the cable torsion and the way the syringe and needle are linked.

So we decided to restart entirely the experimental protocol in order to get trusted results. 1,2 liters of a 5,5% in weight CMC added fluid has been made and respectively: viscosity using falling ball technique, empty cavity filling and sponge filling have been conducted. Then 5% in weight of sand has been added to the fluid and exactly the same protocol has been conducted with the same tools.

11.3. Experimental results

11.3.1. Viscosity

The falling ball technique used on a length of 27 cm gave a viscosity of 23,2 Pa.s for the basic fluid and 21,3 Pa.s for the fluid with sand. So the viscosity was a little bit lower, the previous explanation, of CMC links broken by the sand particles remain plausible. However the difference is much lower than what was calculated in a first time.

11.3.2. Empty cavity filling

11.3.2.1. Speed comparison

Still there was no visible change between the two fluids in the way they progressed in the cavity by measuring the speed in the cavity with the video software Sony Vegas Pro. This similarity looks more credible considering the difference of 1,9 Pa.s between the two viscosities instead of 7,8 Pa.s.

11.3.2.2. Pressure comparison

Pressure and flow are compared with the same experimental assembly.

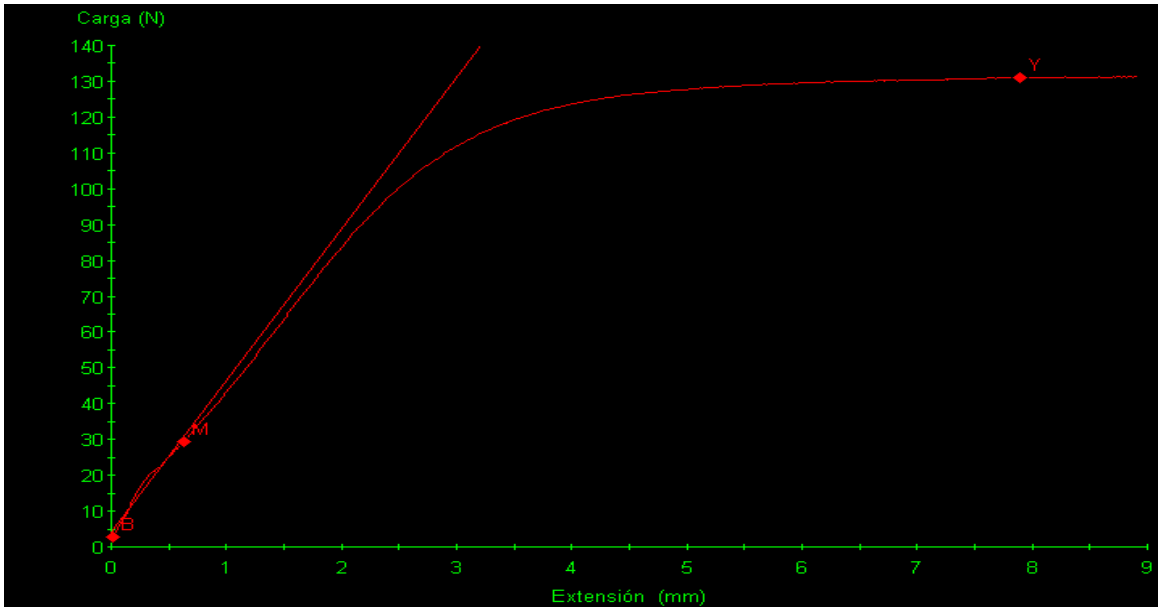


Fig.11.01. The applied force curve of an empty cavity filling (cable length 103 cm) with a 5,5% added CMC fluid. The resulting force is established around 130 N which represents a 2,45 bar pressure (calculus method chap.9).

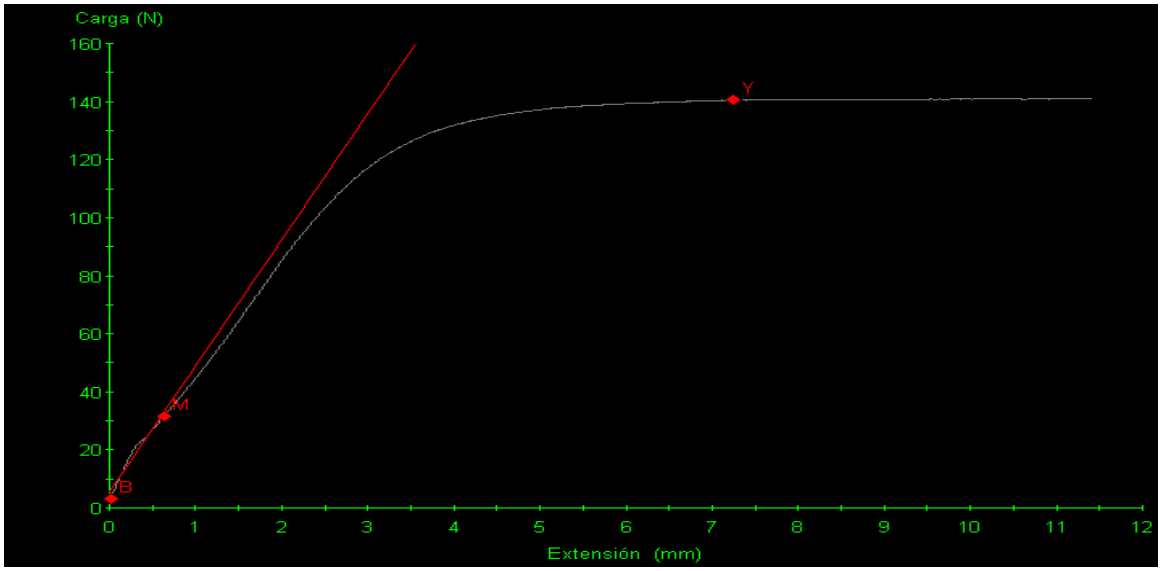


Fig.11.02. The applied force curve of an empty cavity filling with a 5,5% CMC and 5% added sand (cable length 103 cm). The resulting force is established around 140 N which represents a 2,64 bar pressure.

Interpretations: it appears that in spite of a lower viscosity, the fluid with added sand requires more pressure to be injected truth a cable and needle. It may be supposed that the sand particles in the cable and needle act as a brake by consuming energy when sliding against the inner surface of the cable and needle. The behavior is different in a 3 mm diameter pipe as in a container with a diameter considered as infinite.

It is noticed that in both cases the applied force is established and stabilized after a 6 mm compression of the tensile machine.

11.3.3. Sponge filling

11.3.3.1. Sponge models

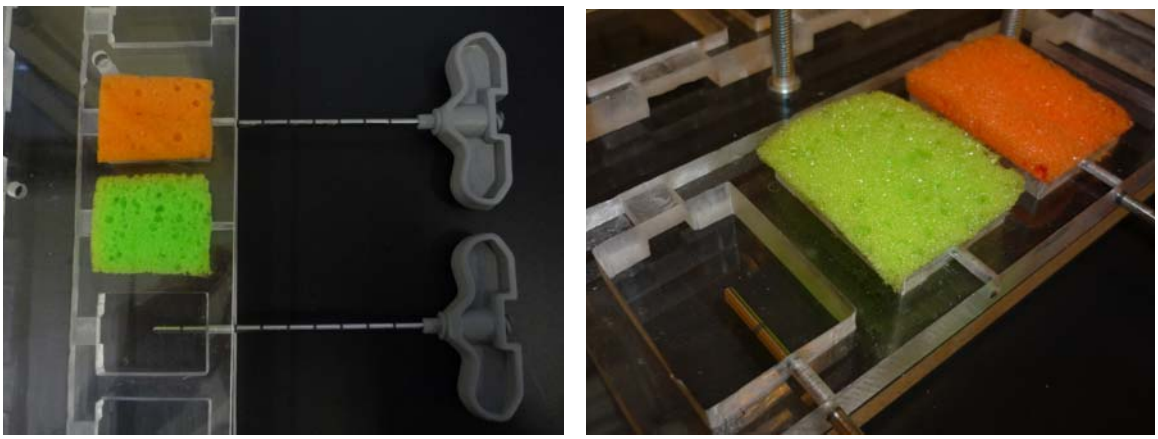


Fig.11.03. General view (on the left) and detail (on the right) of the sponge filling assembly showing the green sponge (type G in the following part) the orange sponge (type O in the following part) and the needle pushed in the orange sponge and in a empty cavity to figure out that the filling takes place in the middle of the sponge.

The type G sponge comports bigger porosities than the type O sponge.

Note: the PMMA top cover is not represented on the right picture but is used to keep the sponge locked in a finite volume.

11.3.3.2. Pressure curves

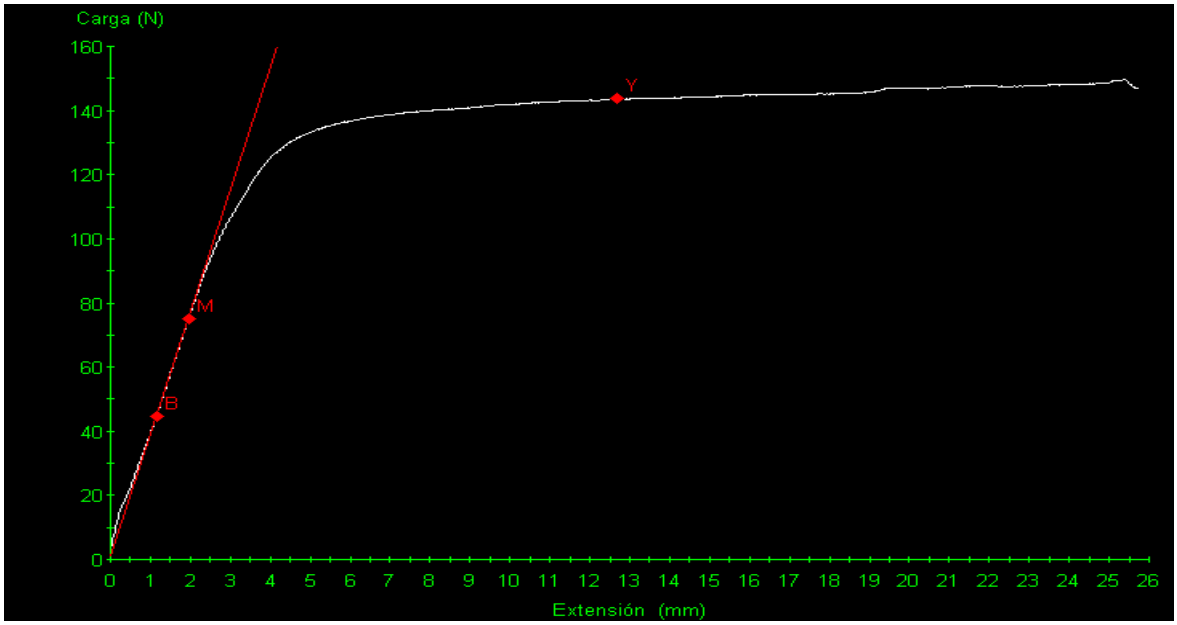


Fig.11.04. The applied force curve of a type O sponge filling with a 5,5% added CMC (cable length 103 cm). The resulting force is established around 149 N which represents a 2,8 bar pressure. The applied force is stabilized after a 6 mm compression of the tensile machine.

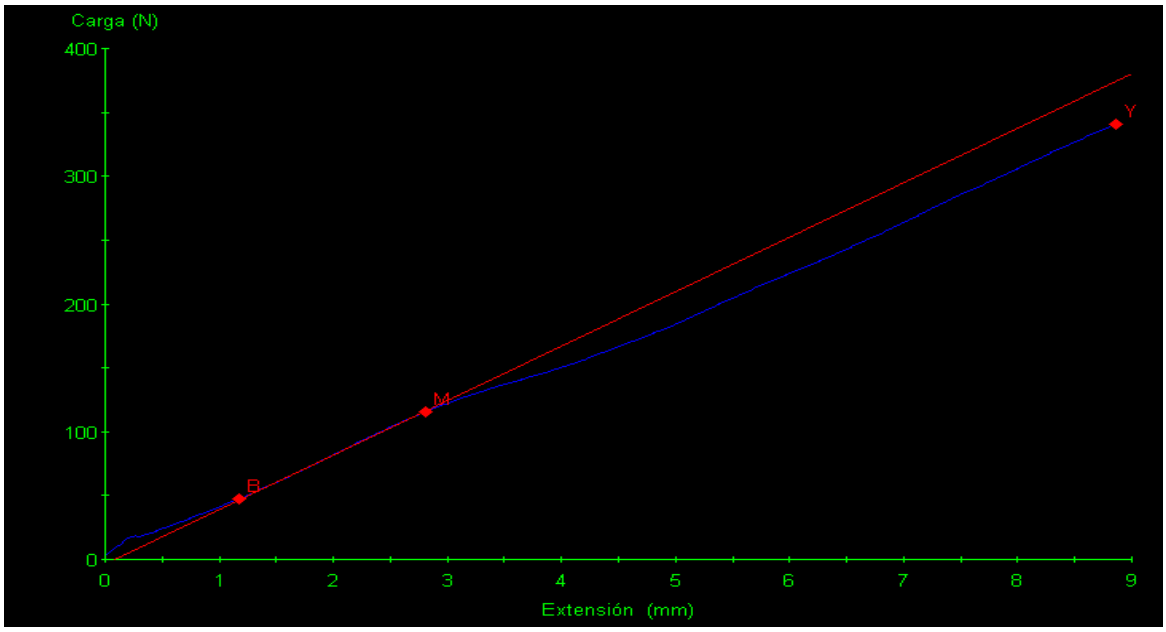


Fig.11.05. The applied force curve of a type O sponge filling with a 5,5% CMC and 5% added sand (cable length 103 cm). The applied force does not get stabilized and the cable-needle or the cable-syringe link breaks around 350 N for the applied force which represents a 9 mm compression of the tensile machine.

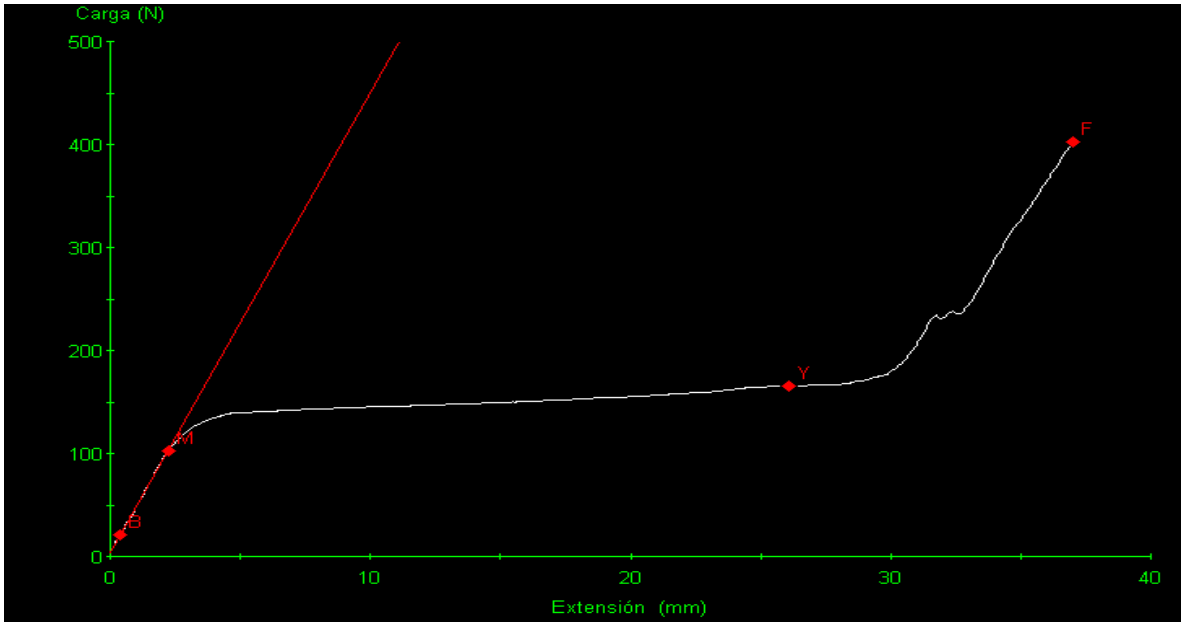


Fig.11.04. The applied force curve of a type G sponge filling with a 5,5% CMC and 5% added sand (cable length 103 cm). The applied force gets stabilized around 150 N after a 6 mm compression of the tensile machine and a slope of recompression appears after a 30 mm compression. The cable-needle or the cable-syringe link breaks around 400 N for the applied force which represents in this case a 37 mm compression of the tensile machine.

11.3.3.3. Calculus of the sand volume injected in the type G sponge

This calculus is made on the linear part of the injection on 24 mm from 6 to 30 mm of compression (see fig above).

Compression length / Compression speed = 24 mm / 8 mm/min = 3 min

The compression time is 180 s.

Filling speed: 1 cm/s * π * $(1,5\ mm)^2$ = 0,07 ml/s

Filling volume: filling speed * compression time = 12,7 ml

Filling mass: filling volume * ρ = 13,7 g

Mass of injected sand: 13,7 * 5% = 0,68 g

It is supposed that 0,68 g of sand are injected during the linear part of the injection.

11.3.3.4. Sand progression in the sponge

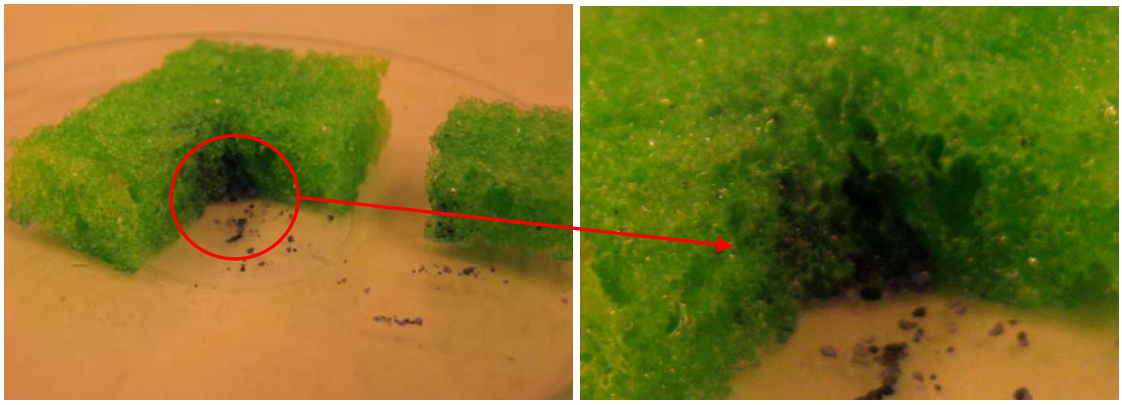


Fig.11.07. View of the type G sponge after it has been cut (left), and a zoom showing the dispersion of sand in the sponge (right).

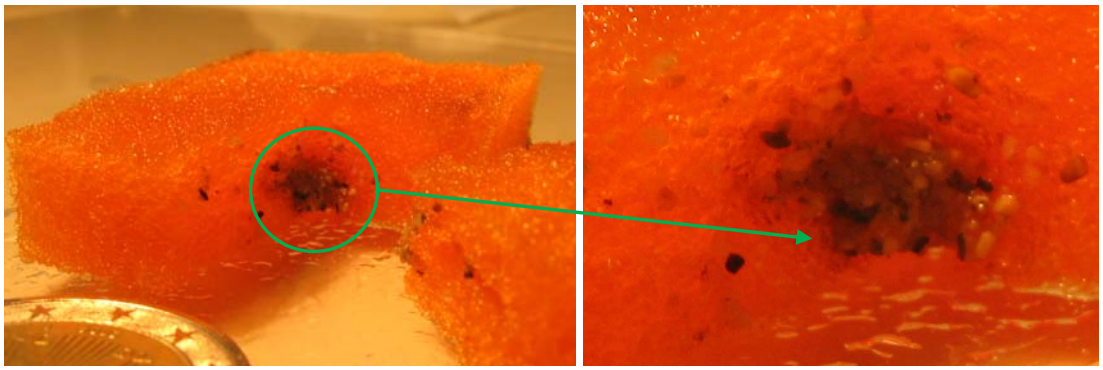


Fig.11.08. View of the type O sponge after it has been cut (left), and a zoom showing the agglomeration of sand in the sponge (right).

11.3.3.5. Interpretations

The type O sponge comports very small porosities and the sand remains locked in the needle. So the sponge behaves like a sand particles filter and the pressure increases constantly. The type G sponge permits to the sand to progress slightly in the porosities. It is supposed that for this reason the behavior remains unchanged compared to the fluid without sand until the filling area of the sponge is full of accumulated sand and then, the pressure starts to increase.

11.4. Summary conclusion

The purpose of this experimental opening was to obtain preliminary results concerning the pressure and particles convection during the fluid progression in porous materials. Particular pressure curves and particles agglomerate occurring during this kind of filling have been experimentally highlighted.

More characterizations and researches could be done on porous media filling in order to realize complete simulations on COMSOL with all the required parameters.

We invite you to visualize the video “sponge filling” given in annex A.

12. Our mistakes and error messages on COMSOL

Working with COMSOL may reach regularly to disappointment. The main reason is that a computation may take a quite long time and the result is not every time the one you expected or does not simply give any result. To avoid this kind of disappointment and hopefully to provide you a gain of time, we will explain in a first time two of our misfortune and the conclusions that we draw of and in a second time we will list the different kind of error message you may obtain with computation.

The first problem looks actually very easy to fix but… We regularly started the same 3D model of a 6*4*1cm cavity and a 3mm diameter pipe arriving in. It became an automatism to draw the model, include parameters and run it but more than once we ran models with errors concerning boundary settings (forgetting an outlet, wrong inlet speed, etc…) and we realized that we made a mistake only hours later, after the program ran uselessly.

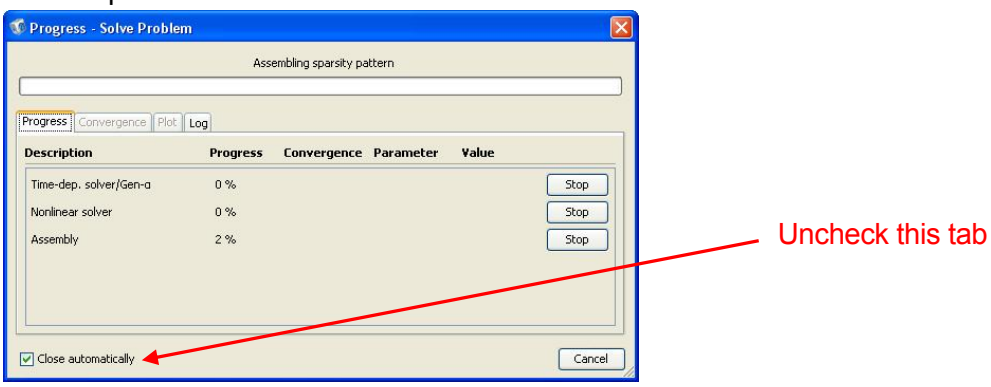
How did we figure it out?

- As working as partners, one was drawing the model and the other one systematically checked the model before to run the computation. It is simple but necessary. If you are working alone and nobody may check your parameters, do not hesitate to check twice.

Second problem was a regular “blackout”: we started a computation and when we came back a few hours later, the software had simply closed without keeping any data.

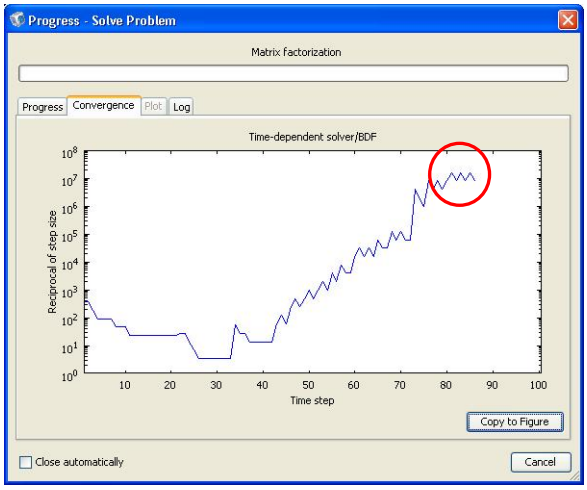
How did we figure it out?

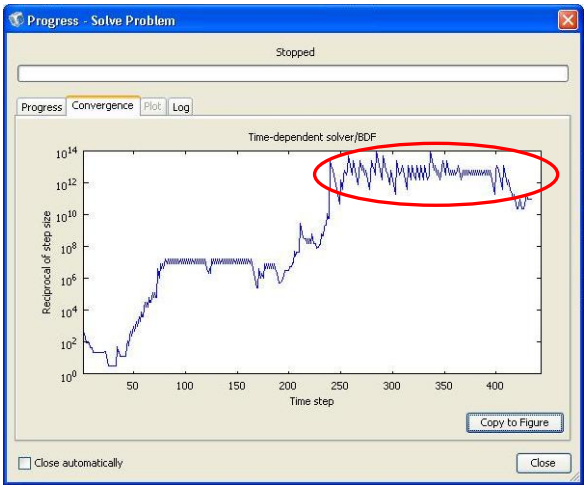
- First we discovered that it was useful to let the window open after the end of the computation: it permits to read the error message that occurred during the computation



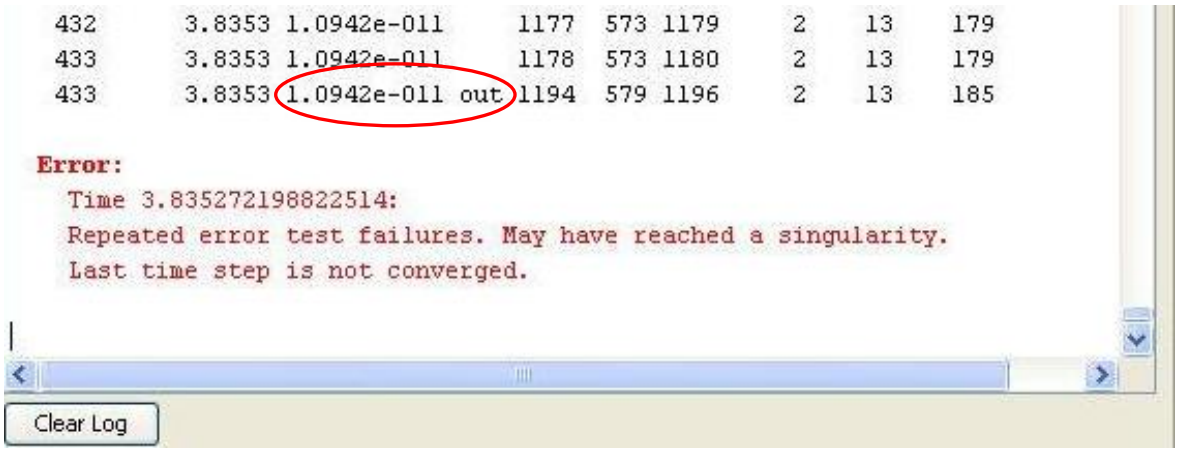
- Then it appeared that some computation are asking too much RAM resource and are bugging

12.1. Error messages





These two screenshots have been printed after respectively 90 and 433 steps and we can see that the program diverges respectively around 10^8 and 10^{14} before to stop.



After 433 steps the program stopped, step size was very small (around 10^{-11}) and the program could not “go forward” anymore because of a redundant computation problem.

How to figure it out?

There is no miracle solution to solve this problem: you may have to reset your model, reduce your mesh, and try again…

```
176      1.5017        0.01      821  144  821     2     -     47
176      1.5017 9.5367e-009 out  845  155  845     2     -     57

Error:
  Time 1.501718759536744:
  Nonlinear solver interrupted since convergence rate above limit.
  Last time step is not converged.
```

Clear Log

This message appears when the solver is not able to obtain correct values. Convergence curve goes too high and computation stops. As for a singularity problem, there is no miracle solution but try.

```
63     0.30322   0.0075349      185   64  186     1     0     19
64     0.31829     0.01507      187   65  188     1     0     19

Error:
  Time-dependent solver interrupted.
  Not all time steps returned.
```

Clear Log

This message appears when you voluntarily stop the computation. You may however continue the computation by running it from the last time value.

13. Point of view on COMSOL software

The first feeling when starting with COMSOL is that this software is easy to run, with a quite simple interface and a logic way to process from draw to results passing by subdomain and boundary settings and meshing tools.

Still, when it is necessary to modify a model, modifications are easy to do and do not require (in our case) to destroy the model tree.

However, when a model does not work, the reasons of the problem remain too often unexplained. Error messages appear for too much demanding computation but when the problem comes from a mesh irregularity or anything else that it is not possible to have knowledge about, there is no explanation to fix the problem and it takes sometimes several hours to find a semblance of solution.

The results are very easy to see and to study. Sometimes however they look quite imprecise with a rough numeric scale. We had several examples of meaningless pressure or speed in our results and still it is hard to understand where the mistake is coming from.

At last, it is very difficult to create a multiphysics model without a perfect theoretical knowledge of the involved physic and equations. Interactions between computational parts are complex and sources of mistakes. It remains however an interesting option of the software.

To conclude, it can be said that during the four month of use of COMSOL, the computation had been easy and fast made with useful results even if the veracity of some of them is difficult to evaluate. Starting with basic computation, models have been slowly more and more complex bringing with them always more and more complex problems. Some have been solved thanks to the acquit experiment and time but some remained unexplained because of a lack of purely computational knowledge. My feeling at the end is to have used a really small part of the enormous abilities of computation that this software is able to do in the fluids mechanic and that I do not even imagine.

14. Environmental impact

This project has been made mostly using computers for the simulations of the fluid flows. And another part of this project was using some machines to fabricate all the components used for the experimental data. And in the laboratory the products that were used were not harmful to nature, nor for men. The only environmental impact this project has caused is the electricity used by the machines.

However, the students doing this project have minimized this impact by stopping the computers and all the machines during the night when nothing had to be done. The water used for cleaning and washing all the glassware was the optimum quantity that could be used.

To conclude this project was nature friendly due to the very low impact on the environment that the project brought.

15. Project cost

This project started the seven of September 2009 and finished the 2 of February 2010, it is considered that with the extra work, the project was done in five months including twenty days of 7 hours a day.

Determining the cost of this project is done by dividing the costs in categories. Below, all the categories done to calculate the costs are written.

a) Costs associated with computer usage

b) Costs associated with software

c) Costs associated with machines usage

d) Costs associated with Laboratory glassware and products usage

e) Costs associated in Human Resources

15.1. Costs associated with computer usage

The informatics system is considered as necessary. A lot of work had to be done with powerful computers, so we decided that the best to do is to account that rent all the computers was more expensive than bought them.

Material	Costs associated with computer usage			
	Number of elements	Cost per unit (€/u)	Time of use (h)	Total Cost (€)
Personal computers	2	600,00 €	--	1 200,00 €
Workstation HP	1	1 000,00 €	--	1 000,00 €
PC HP	1	1 000,00 €	--	1 000,00 €
Electricity		181,00 €		181,00 €
TOTAL Cost associated to computer usage				3 381,00 €

15.2. Costs associated with software

Lot of different software was used for doing this project:

Material	Costs associated with Software			
	Number of elements	Cost per unit (€/u)	Time of use (h)	Total Cost (€)
COMSOL License	4	8 000,00 €	--	8 000,00 €
CATIA License	1	10 000,00 €	--	10 000,00 €
Sony Vegas platinum	1	100,00 €	--	100,00 €
CAMTASIA	1	261,00 €	--	261,00 €
Cabri	1	143,00 €	--	143,00 €
Maple	1	1 000,00 €	--	1 000,00 €
Pack Office 2007	2	475,00 €	--	950,00 €
SUPER (video converter)	1	0,00 €	--	0,00 €
TOTAL Cost associated to software usage				20 454,00 €

15.3. Costs associated with machines usage

This project needed plenty of machines to make the experimental montages. The cost were reduced by trying doing them the more quickly as possible, and with good results in doing them.

Material	Costs associated with machines usage			
	Number of hours	Cost per hours (€/h)	Time of use (h)	Total Cost (€)
Tensile testing machine	1	60,00 €	10	600,00 €
3D Printing machine	1	60,00 €	7	420,00 €
Drill	1	60,00 €	20	1 200,00 €
Milling machine	1	60,00 €	4	240,00 €
photo camera	2	125,00 €	--	250,00 €
Video camera	1	400,00 €	--	400,00 €
TOTAL Cost associated to machines usage				2 510,00 €

15.4. Costs associated with Laboratory glassware and products usage

This part is really cheap because no specific product and expensive one were used. Everything was thought to be less expensive as it could have been.

Material	Costs associated with Laboratory glassware and products usage			
	Number of elements	Cost per unit (€/u)	Time of use (h)	Total Cost (€)
CMC powder	400g	1€/Ton	--	neglected
Water	100L	0,02€/L	--	2 €
Methylene blue	5g	1€/g	--	5 €
Glassware	--	200 €	--	200 €
additional stuff (Plasticine, syringe)	--	8 €	--	8 €
PMMA and screw	--	53 €	--	53 €
TOTAL Cost associated to Laboratory usage				268,00 €

15.5. Costs associated in Human Resources

This section of the project was the more expensive. It was taken in account the fact that a company when employs someone had to pay the person, and also employer contributions. In this paper we assumed that the employer contributions were the same price that the wage of the person.

Furthermore we hypothesized that a technician is paid 10€/hour of work (+ 10€/hour of work for the employer contribution). And an engineer is paid 15€/hour of work (+ 15€/hour of work for the employer contribution).

Material	Costs associated in Human Resources			
	Number of elements	Cost per unit (€/u)	Time of use (h)	Total Cost (€)
Technicians	1	20,00 €	80	1 600,00 €
Engineer	2	30,00 €	700	42 000,00 €
TOTAL Cost associated in Human Resources				43 600,00 €

15.6. Costs associated to the Project

At the end the budget used for this project is:

Material		Costs associated to the Project	
	Cost per day (€/d)	Cost per month (€/m)	Total Cost (€)
Computer Usage	33,81 €	676,20 €	3 381,00 €
Software Usage	204,54 €	4 090,80 €	20 454,00 €
Machines Usage	25,10 €	502,00 €	2 510,00 €
Laboratory Usage	2,68 €	53,60 €	268,00 €
Human resources	436,00 €	8 720,00 €	43 600,00 €
Total costs associated to the project		14 042,60 €	70 213,00 €

This project could seem really expensive at the end. However, this is not too much. First of all the employer contributions have been added to the cost. This employer contribution cost 21k€. The Software used in this project, are the property of the EUTEIB, and could be used by more persons that just the persons using it for the project. Looking at this points, the project is really more cheaper than the data given in this paper.

Conclusions

This project provided coherent results concerning the capacity of COMSOL to produce relevant simulations of the progression of two phase flows (empty cavity and modeling clay obstacle) by using the Level Set method. It requires however to carry out a “hand made” post processing to obtain a like-looking simulation.

Good results have also been provided concerning the Incompressible Navier-Stockes simulation of a continuous single phase flow in a bone looking model.

These results are encouraging concerning a full parameters included simulation of a bone filling in a case of kyphoplasty or vertebroplasty. The experimental opening has been brought in this direction and may have a future in the next projects. It would be interesting to design a real 3D bone structure (for example by drawing a 3D model by using a combination of 2D curves on Catia) and to compute it on a COMSOL study by using the Multiphysics mode in order to combine all physical effects that can be found in this kind of filling (like capillarity, porosity, grain size, …).

To put this in, it will be necessary to improve the interface quality, manage the capilority and grain effects and provide a way to reproduce accuratly a piece of bone in 3D that will be easily runned on COMSOL.

Acknowledgements

Thanks to our tutor R. Torres for his perpetual good mood and constant supports. He was always saying that the results were incredible, and making us thinking that we did good work. Always there to listen at our problems and try to find out a solution.

Our tutor E. Fernandez for the briefings about vertebroplasty and kyphoplastie, his technical support and help in finding some other interesting applications that get us very far in that project.

J. Grau for his acute advices about COMSOL that made us wins through all our problems.

T. Travieso for his precious help on the conception of the 3D printed model.

D. Romanillos and S.Calles for their availability and help for the use of machines.

The EUETIB for providing us offices, computers, machine tools and the GRICCA laboratory for our work.

Bibliography

Bibliographical references

[1] M.D. VLAD. Proyecto de tesis doctoral: *Desarrollo y caracterización de cementos inyectables de fosfato de calcio para aplicaciones en cirugía vertebral*. Director tesis: Prof. Dr. Enrique FERNANDEZ. Convocatoria: 3 de julio de 2007.

[2] S. LAFORGUE, *Aproximación computacional al estudio de la inyectabilidad de cementos óseos,* Marzo 2008

[3] *A. GARGOURI, Aproximación computacional al estudio de la inyectabilidad de cementos óseos y medidas ultrasónicas*, Marzo 2009

[4] I.H. LIEBERMAN, D. TOGAWA, M.M. KAYANJA. *Vertebroplasty and kyphoplasty: filler materilas*. Spine J 2005; 5:305S-316S.

[5] G. BAROUD, M. BOHNER. *Biomechanical impact of vertebroplasty and Postoperative biomechanics of vertebroplasty*. Joint Bone Spine 2005; 73(2):144-150.

[6] G. BAROUD, T. STEFFEN. *A new cánula to ease cement injection during vertebroplasty*. Eur Spine J 2005, 14:474-479.

[7] I.H. LIEBERMAN, S. DUDENEY, M.K. REINHARDT, G. BELL. *Initial outcome and efficacity of kyphoplasty in the treatment of painful osteoporotic vertebral compression fractures*. Spine 2001; 26:1631-1638.

[8] G. BAROUD, M. BOHNER. *Injectability of calcium phosphate pastes*. Biomat. 2005; 26:1553-1563.

[9] S. SARDA, E. FERNANDEZ, J. LLORENS, S. MARTINEZ, M. NILSSON, J.A. PLANELL. *Rheological properties of an apatitic bone cement during inicial setting*. J. Mat. Sc. Mat. Med 2001; 12:905-909.

[10] G. LEWIS. *Injectable bone cements for use in vertebroplasty and kyphoplasty: State of the art review*. J. Bio. Mat. Re. PartB: Applied Biomat. 2005; 76B(2):456-468.

[11] C. HIRSCH. *Numerical Computation of Interna land External Flows. Fundamentals of Computational Fluid Dynamics.* Edition Elsevier, 2007.

[12] R. TORRES, J. GRAU. *Introduccion a la mecanica de fluidos y transferencia de calor*

con COMSOL multiphysics. Addlink Software Científico, 2007.

[13] J.N. REDDY, D.K. GRATLING. *The finite element method in heat transfer and fluid dynamics*. CRC Taylor-Francis, 2001.

[14] J. WILKES. *Fluid mechanics for chemical engineers with microfluidics and CFD.* Prentice Hall, 2005.

[15] J.H. FERZIGER, M. PERIC. Computational methods for fluid dynamics. Springer Verlag, 2002.

[16] COMSOL Multiphysics, User's guide. Version 3.5, COMSOL AB, 2006.

[17] COMSOL Multiphysics. *Chemical engineering module*, User's guide, Version 3.5, COMSOL AB, 2006.

[18] S. OSHER, R.FEDKIW, *Level Set methods and dynamics implicit surfaces*, Applied Mathematical Sciences 153, Springer 2003.

[19] ETIENNE MALACHANNE, DAVID DUREISSEIX, PATRICK CAÑADAS & FRANCK JOURDAN, *Indentification expérimentale et numérique de la perméabilité de l'os cortical pour la modélisation du remodelage*

[20] S. OSHER, R.FEDKIW, *Indentification expérimentale et numérique de la perméabilité de l'os*

[21] http://courses.washington.edu/bonephys/index.html

[22] http://www.osteoporosis-surgery.com/moyens_corail.htm

[23] EDWARD J. GARBOCZI, Solution of the Brinkman Equation: Multiple Scale Porous Media

[24] HAIDONG LIU, PRABHAMANI R. PATIL AND UICHIRO NARUSAWA, On Darcy-Brinkman Equation: Viscous Flow Between Two Parallel Plates Packed with Regular Square Arrays of Cylinders

[25] T ZEISER, M BASHOOR-ZADEH, A DARABI, AND G BAROUD, Pore-scale analysis of Newtonian flow in the explicit geometry of vertebral trabecular bones using lattice Boltzmann simulation

Complementary biography

[26] G. BAROUD, M. BOHNER, P. HEINI AND T. STEFFEN, Injection biomechanics of bone cements used in vertebroplasty, (2004) 487–504

[27] BUDDY D. RATNER, PH.D., ALLAN S. HOFFMAN, SCD., FREDERICK J. SCHOEN, M.D., PH.D., JACK E. LEMONS, PH.D., BIOMATERIALS SCIENCE An Introduction to Materials in Medicine, 2nd Edition

[28] C. JOHNSON, *Numerical solution of partial differential equations by the finite element method*, Studentliteratur, 1987

[29] M.P. GINEBRA, F.J. GIL, J.A. PLANELL, Cursos de biomaterieales en la ETSEIB, UPC de Barcelona (SPAIN)