Silo Fires - Simulation of Gas Injection in a Porous Bed

Mickaël Freindorf

ENSIAME Université de Valenciennes, France

SP Technical Research Institute of Sweden

Silo Fires - Simulation of Gas Injection in a Porous Bed

Mickaël Freindorf

Abstract

Fires that occur in silo storages are difficult to extinguish and can create dangerous conditions. Fires in storages of wood pellets often occur because of the heat generated by oxidation reaction inside the silo. Indeed conventional extinguishment techniques are not suitable for this kind of problem. The most used and efficient technique is to fill the silo with inert gas like nitrogen, the process is very easy, the aim is to replace all the air containing oxygen by nitrogen and thus quench the fire, and stop the oxidation process. This technique has been used several times with success, the most recent case in Sweden is the extinguishment of a fire silo in Kristinehamn (November 2007).

A lot of empirical knowledge has been gathered about the subject but still some questions are not answered, like: What is the behavior of the gas spreading inside the silo? What is the impact of the porous media on the flow and the concentration distribution inside the silo? Is it possible to set a general law to have the number of inlets against the total area that has to be covered?

This report try to find answers to the previous questions by simulating the filling with nitrogen of a silo full of wood pellets using the COMSOL software. Ignition and pyrolysis is not included in the simulation. Here we are investigating the gas distribution in a cold silo.

The major observations resulting from the study show that the concentration distribution is in the shape of a parabolic front of 100% concentration that grows faster upwards than laterally, this means that for high and narrow silos only one inlet is enough to fill it completely. Inlet velocities or diameters have no major impact on the gas distribution, the main parameter is the injection rate.

Acknowledgements

I would like to thank my SP supervisors Mr. Per Blomqvist and Mr. Bijan Adl-Zarrabi for their help and their kindness during work and breaks.

Great thanks to all the SP Brandteknik team for their very nice welcoming and their support for me and my work.

I am also gratefulness to Mr. Patrick Van Hees, Mr. Cavel and Mlle Nashida Bourabaa who made this internship at SP in Sweden possible.

Table of Contents

1 Introduction

The price of oil is reaching peaks that have never been seen, and the use of biofuels like wood pellets is increasing more and more. Thus large storage areas grow up, sometimes without acknowledging the consequences of such huge amount of biofuels. One of the most dangerous consequences is spontaneous ignition. This means a fire that appears in those storages without external heating. Normally the first rise of temperature is due to microbial activity, then the temperature is high enough for exothermic oxidation reaction, and after a while pyrolysis can occur. For storages of wood pellets slow oxidation is suspected to be the first step in the self heating process as the fuel is too dry to have any significant biological activity. Most of the time when smoke is detected from a deep-seated fire it is too late, then the one thing that can be done is to try to quench the fire by injecting inert gas.

This study will try to give answers to questions such as: How to model the material inside a silo and the flow that goes through? What is the impact of the porous media on the flow and the concentration distribution inside the silo? What is the impact of the temperature difference between gas and the porous media? And finally, is it possible to set a general law for the number of inlets versus the total area.

Answers will be given as follows: firstly the theory used for the model is described, secondly the modeling of different problems with finite element software is discussed, and finally an analysis of the results is made.

COMSOL multiphysics (phemlab) with the addition of others parameters as the porous media had been used in order to build a model to study injection of gas in a cold (no fire) porous material. For more information about the settings of the model please refer to appendix 7.

This work was made by Mickaël Freindorf during his internship at SP as a part of the second year at ENSIAME (French engineering school specialized in mechanics). Mickael's supervisor at SP Fire Technology was Per Blomqvist.

2 Simulation of silo fire extinguishment

2.1 Theory about the model used for simulation

Simulation of injection through a packed bed of pellet where self ignition is taking place is a very complex problem in which several phenomenon are coupled. This is why at this stage of the study we assume that no ignition is in progress inside the silo. The core of the problem is a diffusion problem driven by an incompressible flow. First we describe exactly how to couple the system of equations, and which parameters that influence each other.
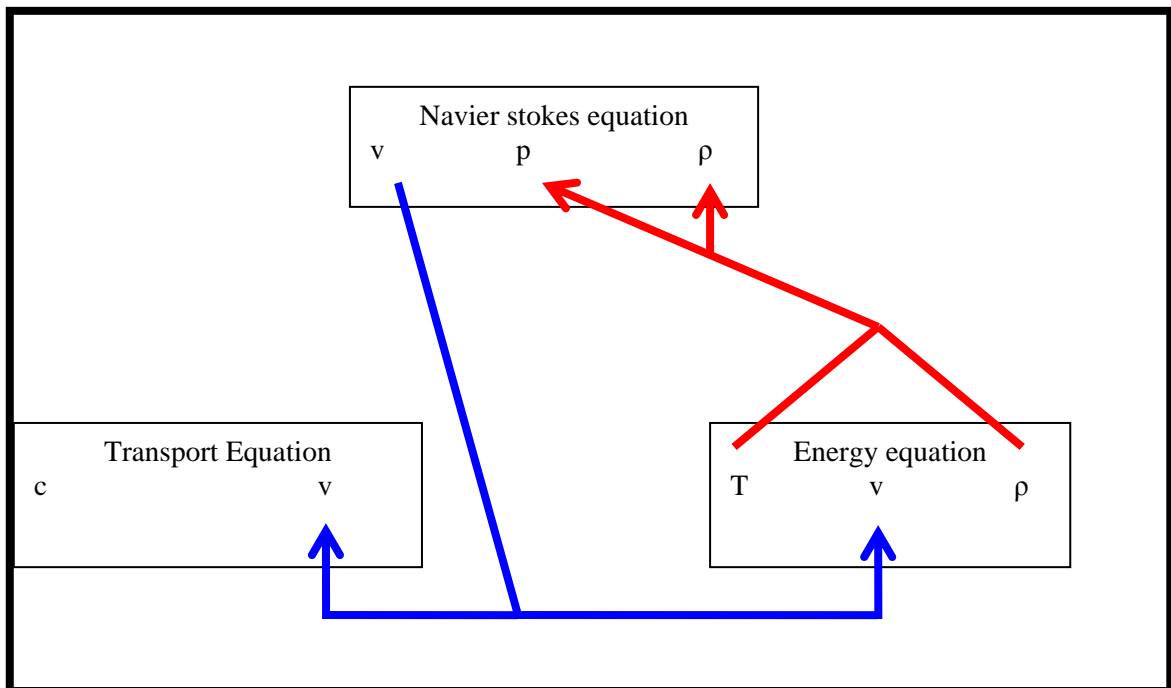

Figure 2-1   Equations and variables coupling.

As shown in Figure 2-1, three equations are needed to model the problem, with a fourth which is not explicitly shown. That is the ideal gas equation.

2.1.1 Modelling of the flow

The first step would be to model the flow with the transient Navier Stokes equation given below:

$$\frac{\partial \vec{v}}{\partial t} + \vec{v} \bullet \nabla \vec{v} = -\nabla p + \eta \Delta \vec{v} + \rho \vec{g}$$

Equation 1: Navier Stokes equation.

This equation is good to describe a flow in an empty silo, it means without any porous media. In a study of injection of nitrogen in a porous media we have to consider the weights of air and nitrogen as significant. One way to describe the flow through the porous media is to add source terms which will describe the pressure drop due in one hand to the permeability and viscosity and in other hand to the inertial loss that can be expected at high velocities. We have the following equation.

$$\frac{\partial \vec{v}}{\partial t} + \vec{v} \bullet \nabla \vec{v} = -\nabla p + \eta \Delta \vec{v} \ - \frac{\eta}{\alpha} \vec{v} - \frac{C \bullet \rho \bullet \vec{v} \bullet |\vec{v}|}{2} + \rho \vec{g}$$

Equation 2: Navier Stokes equation in porous media.

The two additional terms were taken from the Darcy's equation, where α is the permeability coefficient in square meter and C the inertial resistance factor. These factors can be determined by empirical calculations, in our case the permeability was already determined from experiments, but C can be found with the Ergum equation.

$$C = \frac{3,5 \bullet (1-\varepsilon)}{D_p \bullet \varepsilon^3}$$

$$\alpha = \frac{D_p^2 \bullet \varepsilon^3}{150 \bullet (1-\varepsilon)^2}$$

Equation 3: Empirical expressions for inertial coefficient and permeability.

ε is the porosity of the bulk, and Dp the average diameter of the pellets. Two different values were found for two different porosities:

For ε =0,5 and Dp=8mm: C=1750m^{-1}
For ε =0,25 and Dp=8mm: C=21000 m^{-1}

2.1.2 Modeling of the material transport

Once the flow is found, we still have to find the equation to describe the diffusion phenomenon, the best way is to directly use the diffusion equation with convection.

$$\varepsilon \bullet \frac{\partial c}{\partial t} + \nabla(-D_{eff}\nabla c) = R - \vec{v} \bullet \nabla c$$

Equation 4: Transport equation.

As can be seen that the influence of porosity can be found in all scalar time derivative terms. R is the source term which is equal to zero in our case, D_{eff} is the effective diffusivity of the bulk, it was determined by simulating free diffusion through a packed bed of pellets, different geometries were tried and an average was done. However, we do not need a great accuracy on that parameter because the convective term of the equation is going to be much larger. The free diffusive term was seen to represent less than 1% of the total flux. More explanations are available in appendix 1.

2.1.3 Modeling of the temperature influence

The temperature influence is the last thing that has been modelled, but at the moment it is still to premature to speak about fire in the silo.

Different choices were possible to model the problem. At first we can consider a porous media as a biphasic problem that induce the use of two equations as it is done in the FIRCOSIM program (FIRCOSIM is an advanced CFD program package that includes simulations of reactive porous media. The software is owned and distributed by Dr. Zhenghua Yan, Lund University, Sweden). One equation is used to describes the energy of the fluid through the porous media, the second equation is used to describes the energy of the solid part, the pellets.

$$\varepsilon \frac{\partial \rho_g Cp_g T}{\partial t} + \nabla \bullet (\rho_g Cp_g T.\vec{v}) = \varepsilon \nabla \bullet (\lambda_g \nabla T)$$

Equation 5: Fluid energy equation.

$$\frac{\partial \rho_s Cp_s T}{\partial t} = \nabla \bullet (\lambda_s \nabla T)$$

Equation 6: Solid energy equation.

That solution was finally rejected; in fact the coupling between the two energy equations was impossible with COMSOL, or would require deeper knowledge of the software. As an alternative an effective energy equation can be used as it is described in the Fluent manual. It means that we can use only one equation in addition to the others, which gives of course a faster computation.

$$\frac{\partial ((\rho_g Cp_g + (1-\varepsilon)\rho_s Cp_s)T}{\partial t} + \nabla \bullet (\rho_g Cp_g T.\vec{v} - \lambda_{eff} \nabla T) = 0$$

Equation 7: Effective energy equation.

With: $\lambda_{eff} = \varepsilon \lambda_g + (1-\varepsilon)\lambda_s$.

2.1.4 Assumptions and variables coupling

In Fluent and FIRCOSIM all the coefficients are considered as variables, it means they are time and space dependent, thus they have to be involved in the derivatives. COMSOL provide simplified equations as incompressible flow and energy, for which density is assumed constant; the main assumption in our case is the incompressible flow. The Mach number can be calculated for an ideal gas with the assumption that the temperature difference between the gas and the silo is low enough to assume adiabatic conditions.

$$M = \frac{v}{\sqrt{\gamma RT}}$$

With $\gamma=1,4$ for an ideal gas, R=287 J•kg-1•K-1 and v=1mm/s. It gives M = 2E-6.

This low value of the Mach number means that we can assume the gas is incompressible.

Another assumption that is dependent of the previous assumption appears in the ideal gas equation. This state equation permits to couple density to temperature, but in our case we will not take account of the pressure variations. In fact most of the time, the cold gas injected is at 273K and the silo has a temperature of 300K. The maximum pressure drop is from the bottom to the top of the silo and it reach only 500 Pa (in a large tower silo), with an initial values of pressure of one bar everywhere. So relatively it means the temperature variation is about 10% against a pressure variation which is about 1%. By that analysis we can see that the temperature is the major parameter which would have an impact on the density. Finally we will use a fixed, pressure operator in the ideal gas.

The temperature difference between the injected gas and the content in the silo is relatively low, and we can assume constant conductivities and specific heats.

2.2   Kristinehamn silo Modeling

2.2.1   The aim

In Kristinehamn a silo storage previously used for corn has been converted to store wood pellets. One of the eight silos had burnt because of a self ignition. SP was immediately called to find a solution, and then a hole was made in the bottom of the silo and an inlet probe was introduced through which nitrogen injection was possible. The first appearance of the injected nitrogen in the top of the forty meters silo was detected after approximately five hours. After one day of inerting, the emptying of the silo started, and it continued for two days. The day after, nearby silos were investigated for risks for ignition. The gas composition of the air in the top of the silos was measured to check the oxygen, carbon monoxide, carbon dioxide and hydrocarbons concentrations. Temperatures were also checked. At the end of the investigations there were no clear signs that a new ignition should occur.

Figure 2-2   Kristinehamn silo.   **Figure 2-3**   Gas inlet.

The aim of the filling simulations based on the Kristinehamn conditions was to have a first idea of the important things to model, and to know if there are some parameters that require any measurements and the accuracy that should be used for it.

Several simulations were made to study the influence of different parameters. First a study of the flow without taking into account the temperature differences with and without pellets. Then a study of the influence of viscosity, density and the two parameters at the same time with temperature were done, here we will focus on the results with density and viscosity variables.

The following sections have been placed in order of work progress, each section corresponds to a step in the work.

2.2.2 The geometry

A very simple geometry was chosen because the first goal of the study was to find a model for the material inside the silo. The size of the Kristinehamn silo was chosen. We have the case of a silo with only one inlet in the right bottom corner, so we can study its filling in the longitudinal plan which contains the generating axes of the two cylinders (see Figure 2-4). We are only interested in what happens in the silo, thus we can disregard the inlet pipe. At the end we have a rectangle with a diameter of 8 meter and a height of 40 meter with an inlet placed on the right side (20 cm in diameter) centred 1,1m from the bottom. This is the representation of the area of the silo which contains pellets.
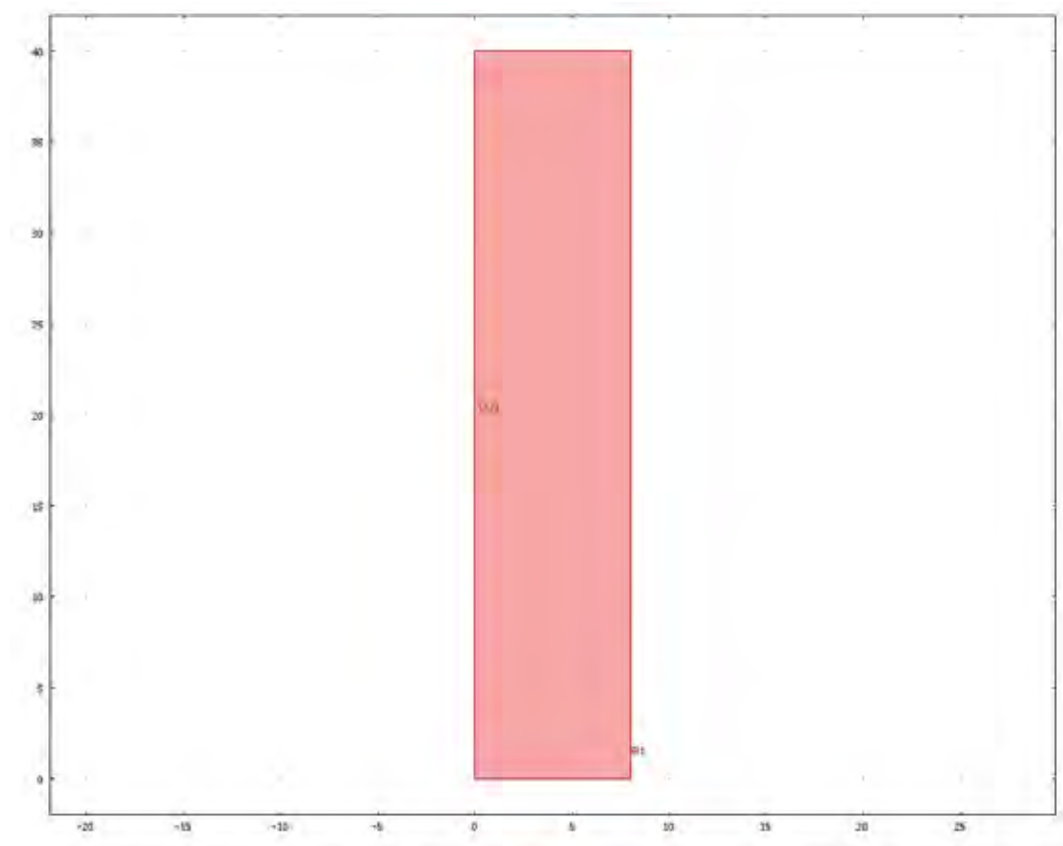
Figure 2-4 Silo geometry.

The mesh was made using triangle elements. A very fine mesh near the inlet can be seen, because this is where important variations of velocity and pressure are expected (see Figure 2-5). The mesh counts 4241 elements.
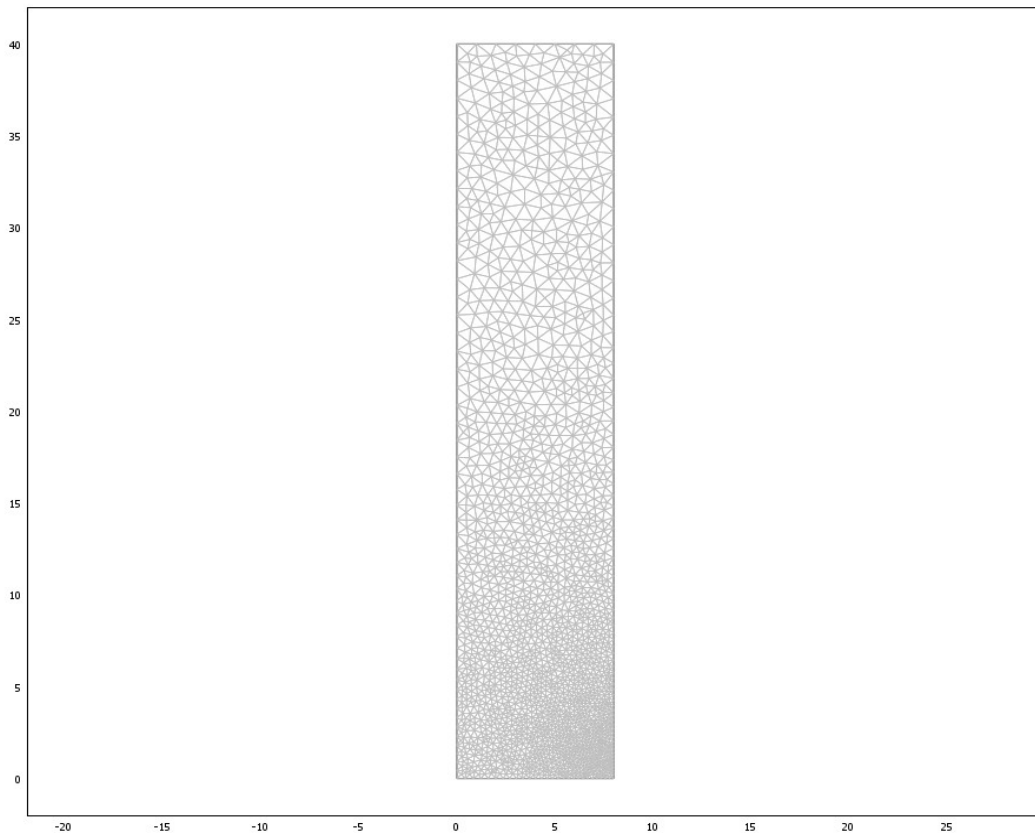
Figure 2-5 Mesh shape.

2.2.3   Modeling of the physic of the problem

In this part we will speak more about the physical aspect of the modeling.

Assumptions:
In our case the velocity at the inlet is 4,9E-2 m/s. By a conservation law we deduce a velocity inside the silo of 1,23E-3 m/s, this value induce a Reynolds number of 620 for a temperature of 300K and a dynamic viscosity of 1,78e-5 Pa.s. It means that we are in a laminar flow regime and that we can neglect the pressure drop due to the inertial term, in fact it represents less than 3% of the pressure drop due to the permeability and viscosity.

(Δp inertia = 0,0015 Pa/m for ε = 0,5 and Δp viscous = 0,73 Pa/m).

Here we will not introduce chemical species; fluids are defined only by their density and viscosity.

2.2.3.1   The incompressible flow

The sub-domain:
Here you describe the properties of the material from the point of view of the flow:
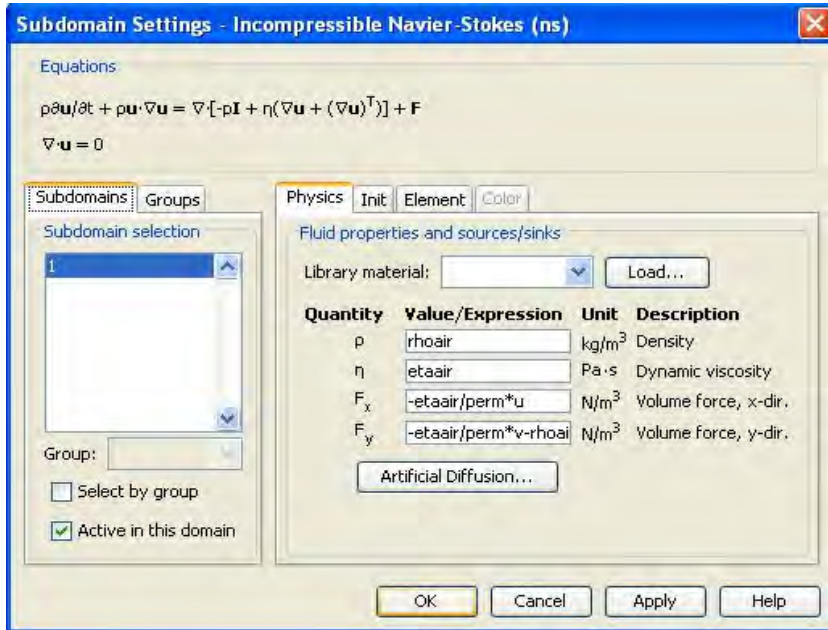
Figure 2-6 Navier Stokes Subdomain Setup window.

- Density ρ: defined as an expression by the ideal gas equation.
- Dynamic viscosity η: defined as an expression by the Sutherland equation.
- Fx: $-\dfrac{\eta}{\alpha}u$
- Fy: $-\dfrac{\eta}{\alpha}v - \rho g$
- u(t0):0
- v(t0):0
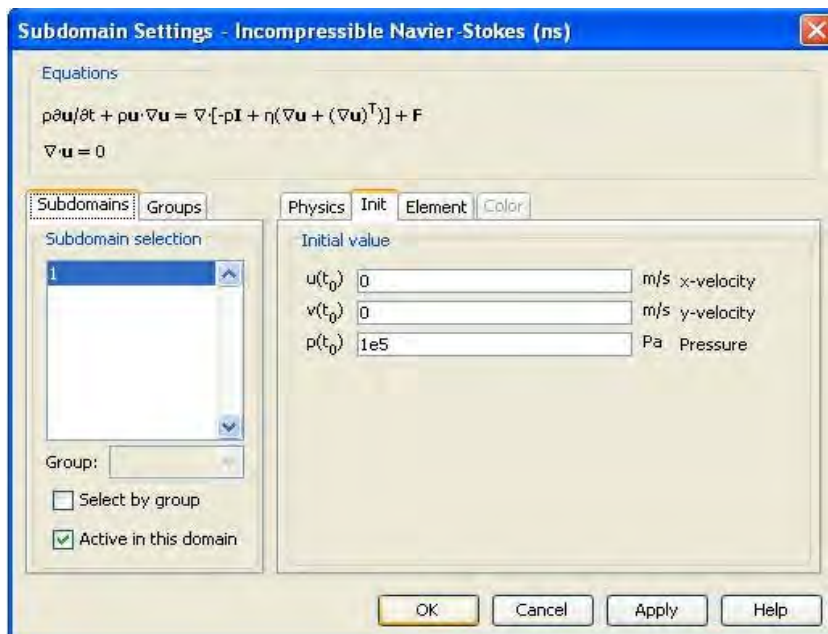
Figure 2-7 Navier Stokes Initial conditions setup window.

- p(t0):1E5

Boundaries:

Here you describe the flow. As you can see you have the choice between different types of boundaries. All the sides are defined as "No slip" it means that we consider the fluid has an adherence on the wall. The Top of the silo is defined as a Normal flow/ pressure, it means we have an outflow which is perpendicular to the outlet surface with a pressure of one bar.
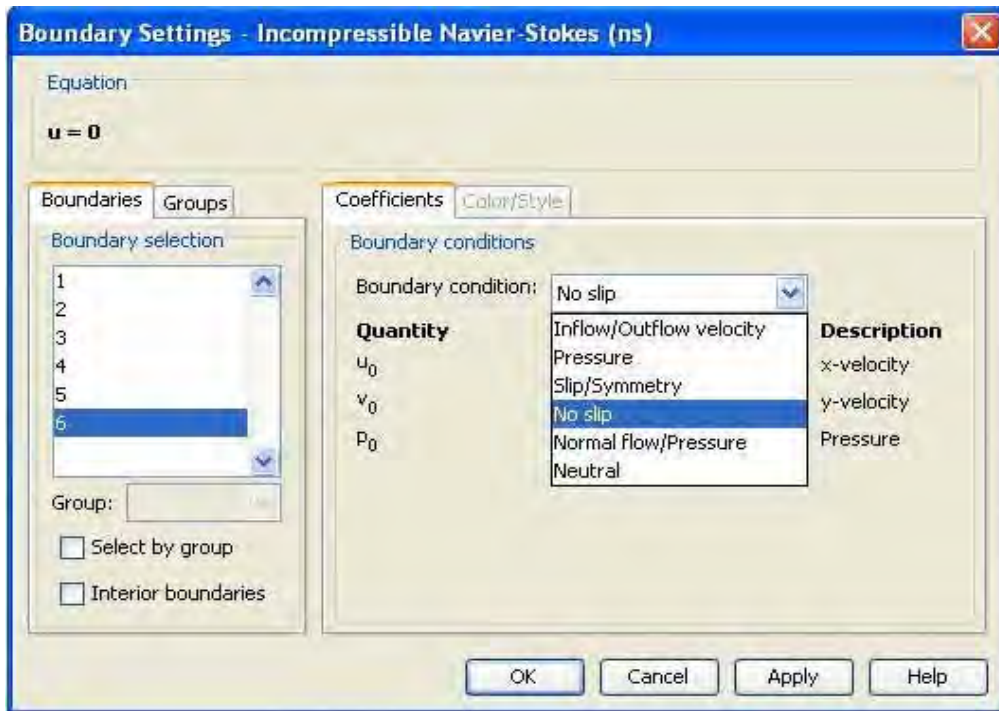
Figure 2-8   Navier Stokes Boundary conditions setup window.

- Inlet: Inflow/outflow velocity
    - u0=-0,049
    - v0=0
    - p0=1E5
- Outlet: Normal flow/pressure
    - p0=1E5

2.2.3.2　　The material transport

As for the fluid we enter the parameters with the GUI (Graphical User interface) for the material transport phenomenon.
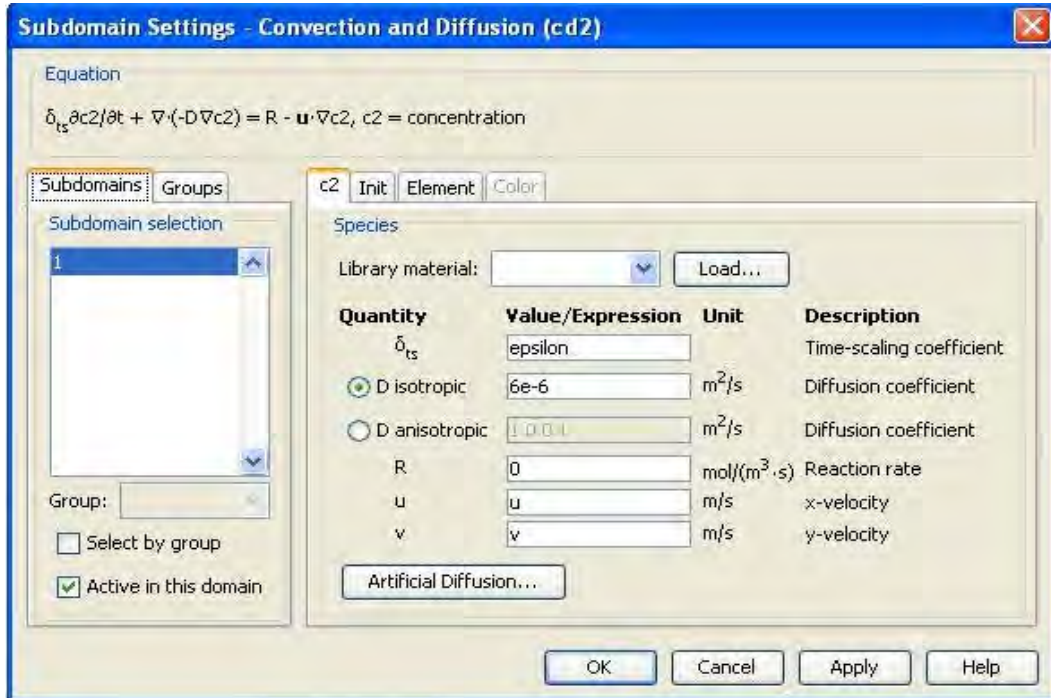
Figure 2-9　Transport subdomain setup window.

The subdomain: Here you describe the properties of the material from the point of view of the diffusion:
- δts =ε the porosity 0,5
- Disotropic=6e-6 m²/s
- u=u
- v=v by doing like that you couple the system of equations.
- Artificial diffusion isotropic: 0,11 This is a coefficient which permit to avoid oscillations of the computed data.
- c(t0)=32,8mol/m3 the nitrogen concentration in air.

Boundaries:
- Inlet: Concentration
  - C0=41mol/m3 Set to 100%
- Outlet: Convective flux
- Others: Insulation/symmetry

The idea is to have a constant concentration at the inlet, and then the velocities will drive the diffusion from the inlet to the outlet.

2.2.3.3　　The energy

Here we will use almost the same idea as above, but instead of concentration we would have a constant temperature.

Figure 2-10   Energy subdomain setup window.

<u>The subdomain</u>: Here you describe the properties of the material from the point of view of the energy:

- δts=ε the porosity 0,5 (set to 1 for an empty silo).
- k isotropic=effective conductivity.
- ρ= defined as an expression by the ideal gas equation.
- Cp=cpair=1003 J.kg -1.K -1
- Q= -(1-epsilon)*rhop*cpp*diff(T,t) in that way we define the weak terms in the equation.
- u=u
- v=v by doing like that you couple the system of equations.
- Artificial diffusion isotropic: 0,2 This is a coefficient which permit to avoid oscillations of the computed data.
- T(t0)=300K the initial temperature in the silo.

<u>Boundaries</u>:

- Inlet: Temperature
    - o   T0=273K
- Outlet: Convective flux
- Others: Insulation/symmetry

19

2.2.4 Results

The results presented in Figure 2-11- Figure 2-15 are from early simulations with the model. The shortcomings of the model at that stage are discussed in 2.2.5. Please see appendix 2 for the results from the final model. The difference in results are mainly in the concentration distribution.

2.2.4.1 Concentrations

50% porosity: The concentration in a silo with pellets that induce 50% porosity after 14h of injection (see Figure 2-11 and Figure 2-12).
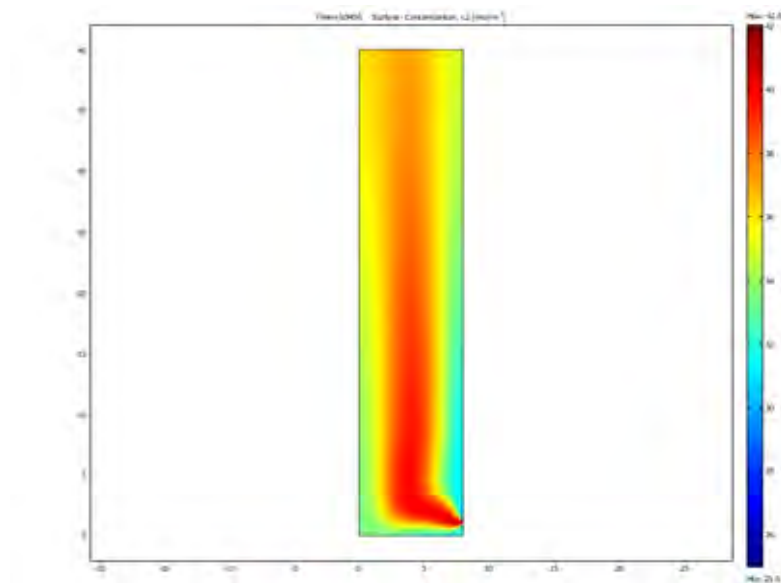
Figure 2-11 Concentration field with 50% porosity after 14 hours of injection.
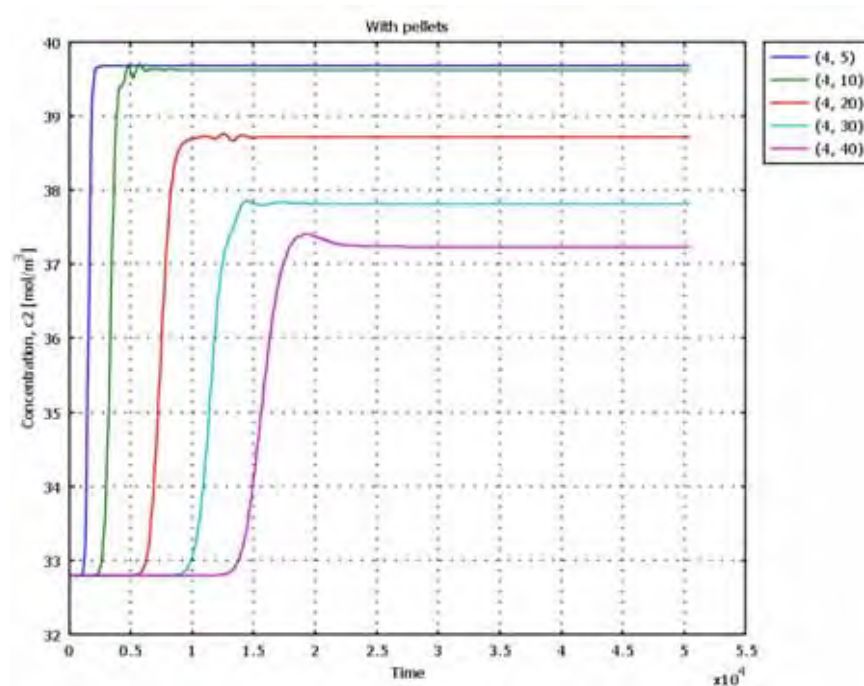
Figure 2-12 Concentration profile against time in the middle of the silo at 5,10,20,30 and 40m.

We can see the concentration field never really looks homogenized. Some kind of disturbance at 5 meters can be seen. We will see later on that it might be because of the temperature difference. The graph shows that concentration decrease slowly with height, we can also see first variations of concentration at the top after 14 000 seconds (4h). The steady state is reached after 2,5E4 seconds (7h). For such a little inlet velocity those values are quite interesting, in fact the filling is fast, this is partly due to the porosity. Indeed half of the silo is occupied by pellets which mean that there is only an half silo to fill.

2.2.4.2   Velocities and streamlines

Here we will check velocity profile and streamlines.
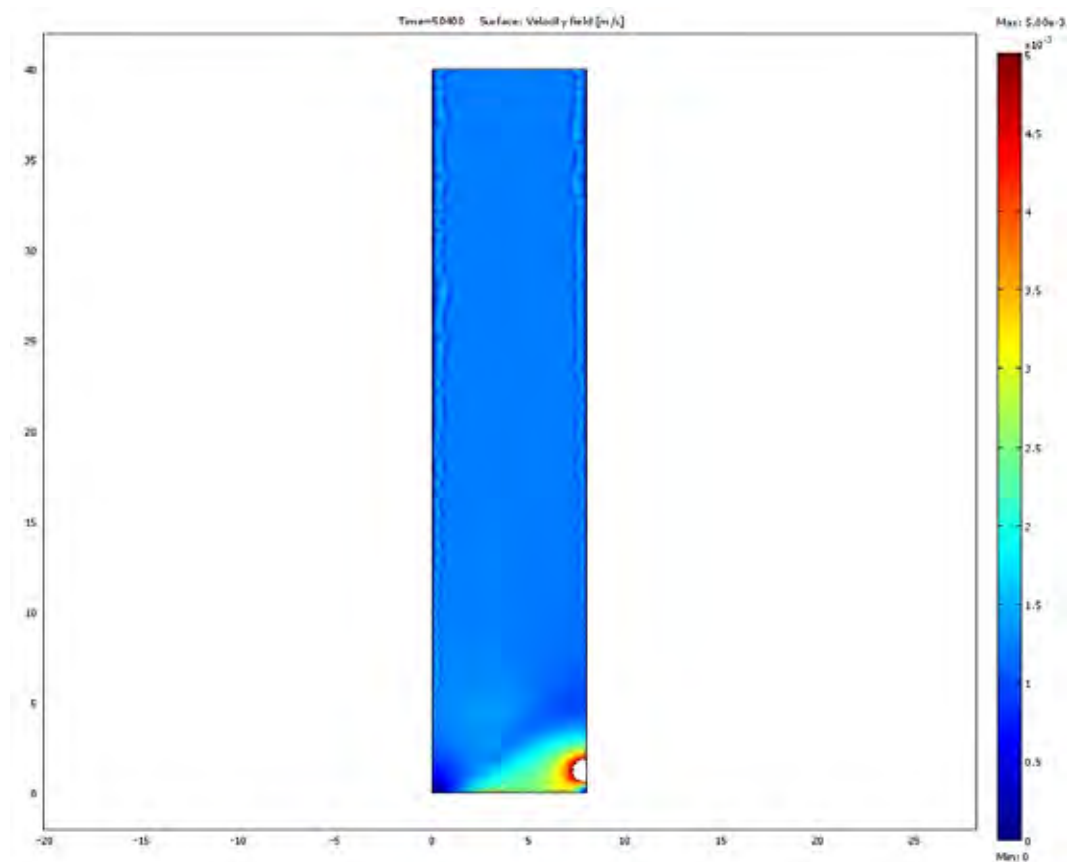
Figure 2-13   Velocity field with 50% porosity.

This kind of velocity profile could be surprising for an empty silo but in our case it is a characteristic shape. Just right to the inlet velocities are relatively high, and because of the gravity the flow goes down. What is really important to notice is the homogeneous velocity field in the upper part of the silo; in fact, in that stage the flow is mainly due to the pressure drop terms that were added earlier in the Navier Stokes equation and as the pressure gradient is uniform there we have a uniform velocity. (Note that the disturbance near the walls is due to numerical instabilities. A finer mesh close to the walls should have avoided this disturbance.)

The streamlines shows a disturbance at 5 meters.
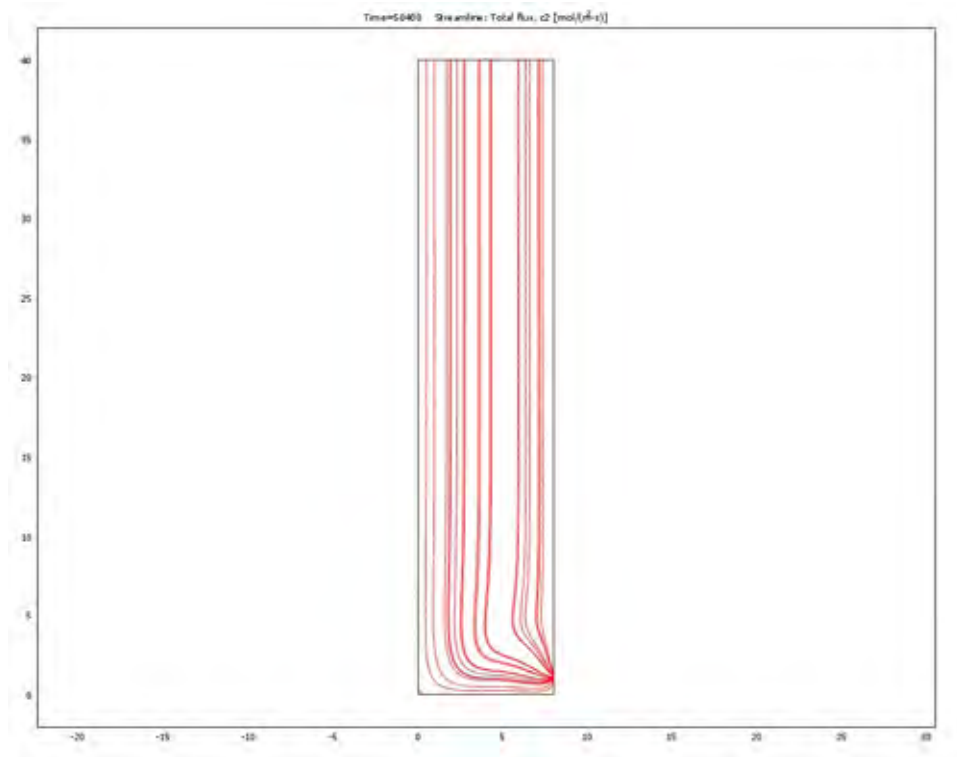
Figure 2-14   Streamlines with 50% porosity.

2.2.4.3   Temperature profile
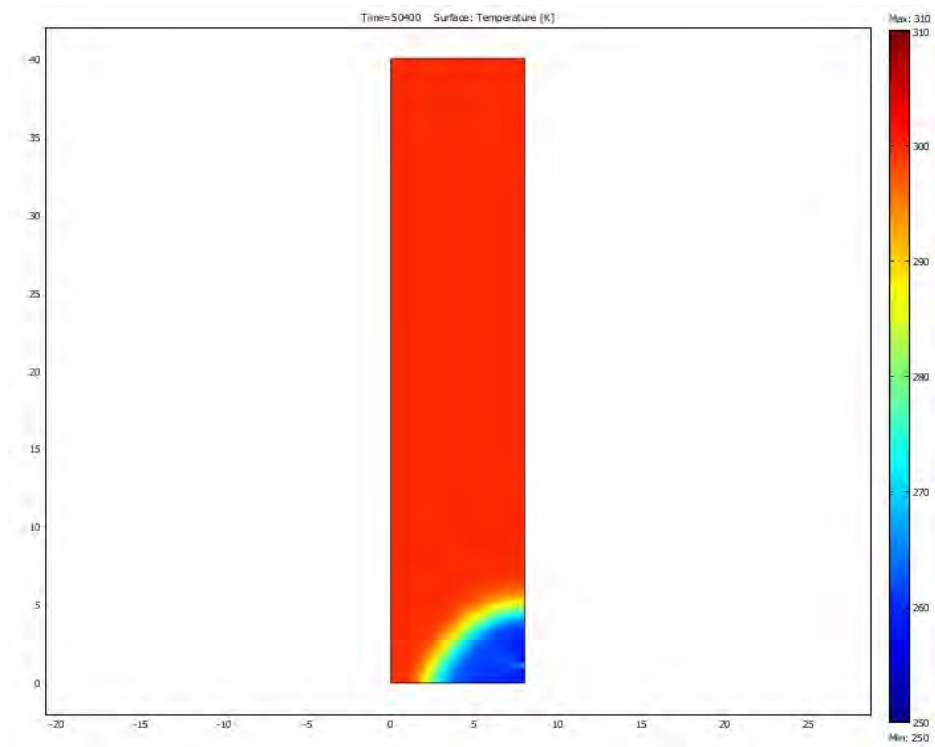
Figure 2-15   Temperature field with 50% porosity.

We can clearly see the temperature front at the bottom of the silo. As was said previously, perturbations in the streamlines can be observed, the temperature front and the anomalies are clearly similarly located. However by comparing it to the concentration profile, we can clearly see that temperature has not a great impact on the filling of the silo because of its very slow propagation. In fact when the concentration reaches its steady state, the temperature front is below 5 meters height.

2.2.5   Shortcomings of the model

The size of this silo and the velocity values have been used to mimic an extinguishment that took place in Kristinehamn. The first variations of gas concentration in the top of the silo were noticed after approximately five hours. Simulations give us a value of four hours which is very close to the reality and the one hour remaining can be explained by the top air phase which is not present.

As the problem is in two dimensions and NOT axisymmetric the inlet velocity is lower than it should be because the velocity in the silo has to be conserved. Thus if a 3D simulation has to be run, we should have a higher velocity at the inlet and the consequences would be that the inertial term could not be neglected.

The top of the silo is not represented, and chemical species are not specified. We also assume that the air molar mass is equal to the nitrogen molar mass.

In the first setting of the model the inlet velocity was constant along the boundary. At the edges of the inlet the high velocity induced numerical instabilities as the velocity nearby was zero. To remedy this problem the velocity profile for the inlet boundary was set by a parabolic function. Starting from zero at the boundary edges and reaching its maximum in the centre of the inlet.

Using the first settings for the inlet boundary resulted in that a concentration of 100 % never was reached. When improving the velocity profile for the inlet boundary 100% concentration was reached in the whole silo for corresponding simulations. The reason or the mechanism for this is not known, but one can assume that the numerical instability close to the inlet in the first case resulted in a propagation and amplification of the error. Please refer to the appendix 2 to check the real concentration shape.

2.2.6   Conclusions

This first approach of a model for silo extinguishing gives a core on which other parameters can be added. The concentration distribution is approximately known and also the behavior of velocity with pellets. The next step of this study would be to study the distribution of the gas in a very large silo to investigate the number of inlets that would be enough to cover the entire area of the silo.

2.3 Simulation of gas distribution

2.3.1 The aim

Now we are able to set a porous media with the major part of the constraints it involves. We would like to have a better knowledge about the behavior of the filling in a very large silo to avoid wall influence with only one inlet in the centre which is characterized by different diameters and different velocities. We have a maximum injection rate of 6280kg/h for one (the largest) inlet, and for simulating a corresponding number of smaller inlets we have to divide this injection rate by the number of inlets we want. A working plan was done with different flows, velocities and diameters (see appendix 3). These parameter were determined by the number of inlets wished in the silo for different extinguishment device configurations (1,5,10 and 25 inlets).

The aim is to check the behavior of the gas spread with one of the inlets that would be in the silo with high and low velocities (3 and 30m/s). For example if the extinguishment device includes only one inlet then we run a simulation with the whole device but if the device has more than one inlet then we run the simulation with only one of those and check its behavior at high an low velocity in order to have an idea of the whole device.

2.3.2 The geometry

As the silo is a cylinder and the inlet is circular and its centre and normal vector of the disc are in conformation with the axe of the silo, the problem is axisymmetric, thus it can be solved in two dimensions. Here we use an extreme case with a silo which is almost as tall as wide; it means 30 meters height for forty meters in diameter.

As we can see in Figure 2-16 the inlet is in the left bottom corner. Figure 2-17 shows us the mesh of the geometry. One of the problems was with high velocities for small diameters, in fact if a coarser grid worked with larger diameters at high velocities it was not the case with the smaller ones. As a comparison has to be done later on between all simulations a similar grid has to be used for all simulations. The square near the inlet is totally neutral, it has been set just to have a very fine mesh in that area because of the high pressure gradient in that location and the big changes in velocity that are induced. A mapped mesh (squares) was used there because of its higher quality but as it costs more degrees of freedom, the rest of the silo had to be meshed with a free mesh (triangles) with a better refinement near the square and boundaries. Finally we have almost 6500 elements which is quite heavy but necessary to avoid oscillations.
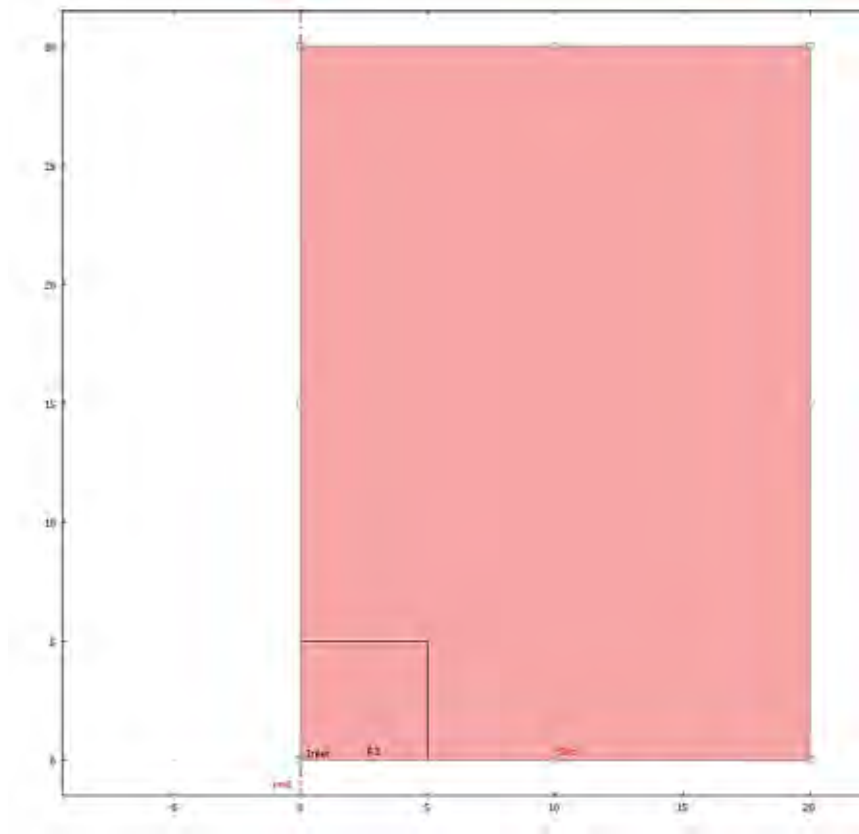
Figure 2-16   Large silo geometry.
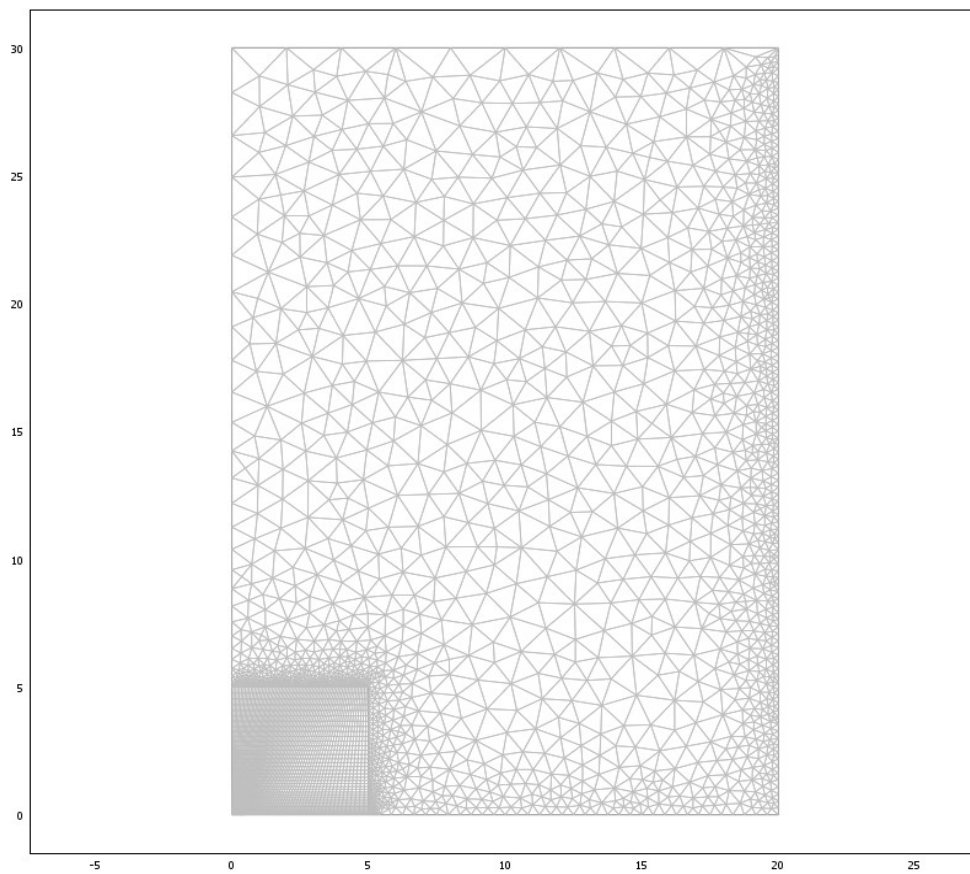
Figure 2-17   Large silo mesh.

2.3.3 Modeling of the physic of the problem

Exactly the same physical problem was set as before, the only difference comes from the inlet velocity. Instead of u0=-0,049m/s and v0=0m/s we have u0=0 and
v0= Uavg*1.5*(1-s²)*(1-exp(t/0,1)) where Uavg is the average velocity through the inlet, (1-s²) represents a parametric function with s the parameter of the boundary which goes from 0 (at the beginning of the boundary) to 1 (at the end of the boundary) in that way we obtain half of a parametric function, 1.5 comes from the integration of the velocity through the inlet as following:

$$\text{Uavg} = \int_0^1 U\max \cdot (1 - s^2)ds$$

After calculation we have: Umax=1.5 Uavg

This expression is very convenient to express velocity directly with the value of the desired average velocity.

The function of the last term (1-exp(-t/0.1) is to progressively increase the velocity. The velocity's has reached steady state in 0.5 seconds. This time is very small compared to the total filling time and its consequences on the simulation can be neglected.

2.3.4 Results

2.3.4.1 Concentration profile

To compare the results from all cases simulated, plots of concentration against time at fourteen hours, in the middle of the silo, at 5, 10, 20 and 30 meters can be found in the appendix 4.

The first injection rate value permits to totally fill the silo in twelve hours, for other values of the injection rate we have the same evolution with a delay. This is an important finding, it means that only one inlet is enough to inert an entire silo.

We can see that inlet velocity has no impact on the gas spread, in fact for each set of simulations with the same number of inlets there is no visible difference between high and low velocity.

Another important point can be seen by checking the gas spread for all simulations. We can observe the same evolution for all configurations with just a delay in time for lower injection rates. In fact we always observe some kind of parabolic front which spread faster upwards than laterally. As the top of the silo is inerted faster than the sides, there would be a big waste of gas for large silo. In contrary for high silos, the solution of one inlet is enough.

2.3.4.2 Pressure profile

Appendix 5 gives the shape of the pressure field in the silo and its evolution with the height. To be able to compare graphs, the maximum pressure value for high velocities had to be cut. The following table gives maximum pressure values in the silo for each case.

Table 1 Maximum pressure in the silo.

Simulation no	Number of inlets	Injection rate (kg/h)	Inlet velocity (m/s)	Inlet diameter (mm)	Maximum Pressure in Bar
1	1	6280	3	770	1,028
2	1	6280	30	243	1,64
3	5	1256	3	344	1,015
4	5	1256	30	109	1,3
5	10	628	3	243	1,012
6	10	628	30	77	1,21
7	25	251	3	154	1,009
8	25	251	30	49	1,13

The graphic below sum up the pressure evolution against height for all cases.
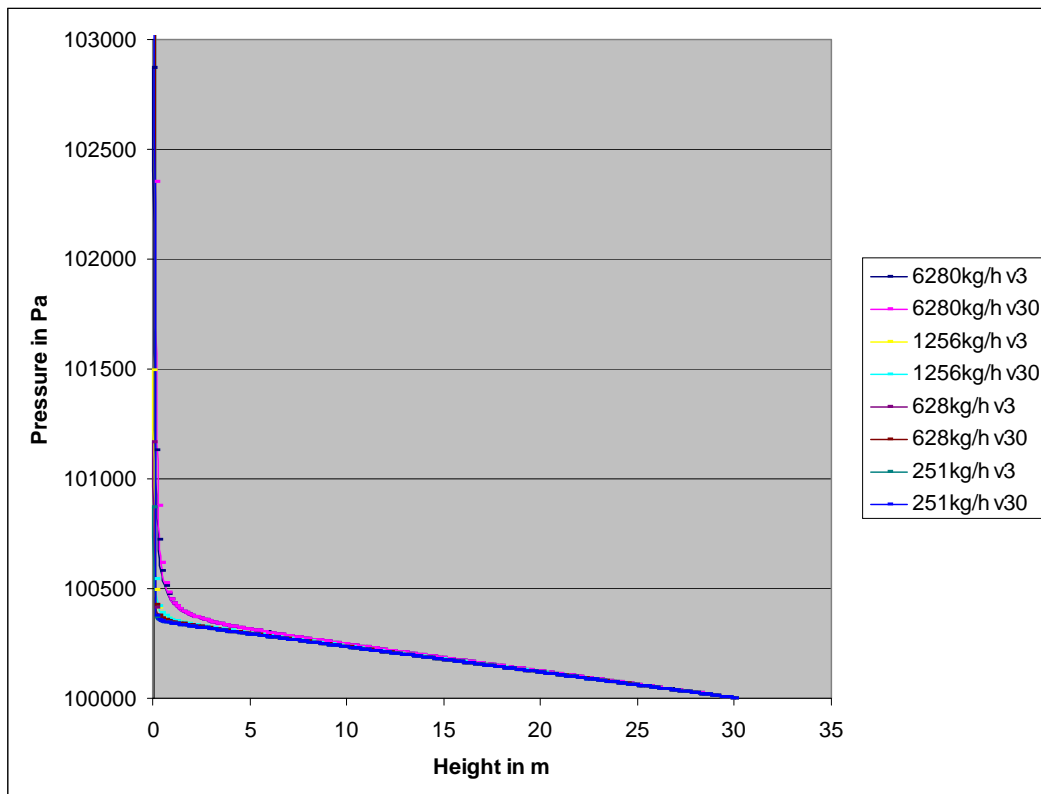
Figure 2-18 Pressure evolution against height for all cases.

Considering all cases, as expected the pressure and its gradient are very high close to the inlet. Then they decrease very quickly. The gradient is homogeneous up to five meters, and it should in this area induce a very uniform velocity as observed in the Kristinehamn silo simulation.

Differences between high and low velocities for each number of inlets cases are not so pronounced, in fact we just have an important difference of the pressure just next to the inlet. The difference between high and low velocities goes from 40% for the highest injection rate to 11% for the lowest, close to the inlet. At heights over 5 m there is no pressure difference between inlet velocities and injection rates.

Now let analyze differences between each injection rate cases. The first observation is that a constant value of the gradient of pressure is reached fastest for the lower injection rate. And that the value of the gradient is the same for all simulations, approximately 12 Pa/m.

2.3.4.3 Velocity profile and streamlines

In appendix 6 velocity fields can be compared between all different inlet cases, and also between high and low velocities. However to compare results easily between all cases Figure 2-19 shows velocities in the centre of the silo against height for all cases.
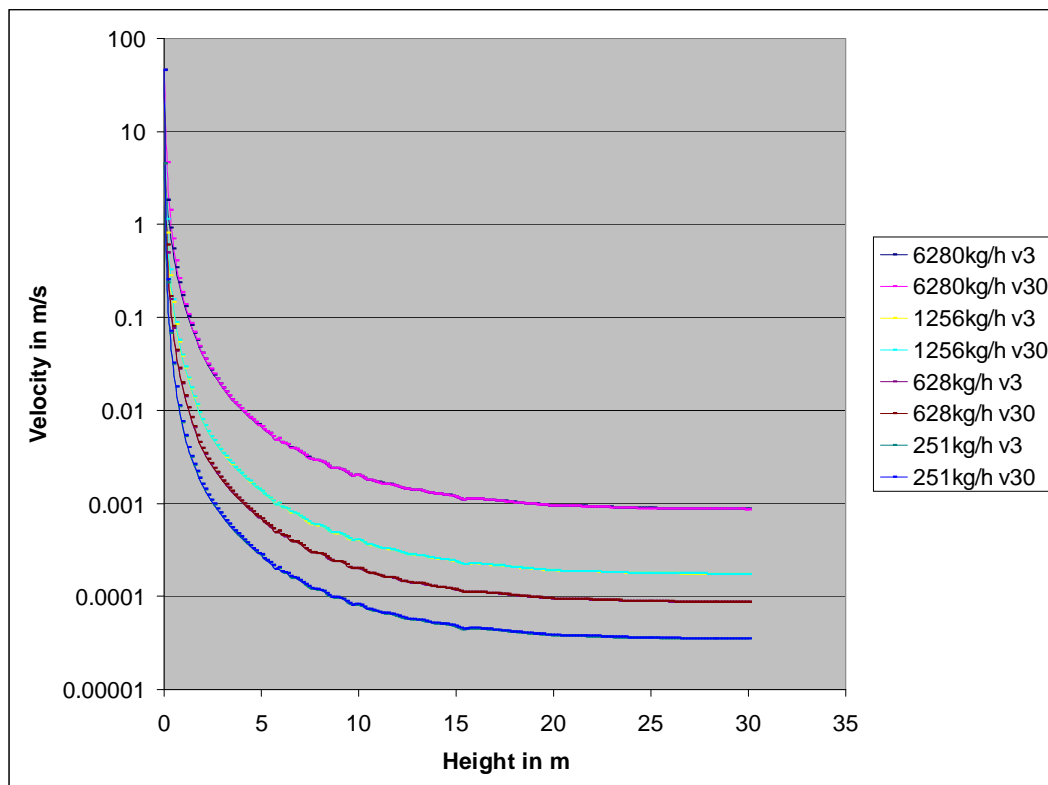
Figure 2-19 Velocity evolution against height for all cases.

Two conclusions can be made from these results. Firstly, we have a very similar velocity shape as observed for the Kristinehamn simulation case, with a fast velocity decreasing and a homogeneous distribution up to 5 meters height, which decreases faster with lower injection rates. The velocity in the uniform area is also lower with a higher number of inlets. This can explain why in those cases the concentration did not reach a steady state. We can still observe some kind of dead end in the right bottom corner.

Secondly there are clearly no differences in general velocity shape between cases with high and low inlet velocities, the only thing that differs is the initial velocity value which is 45m/s for high velocity and 4,5m/s for low velocity, but after less than half a meter we have exactly the same value which is 1m/s for 5024m^3/h. The same conclusions can be made for other injection rates, the only difference is that velocity fields conform faster with lower injection rate.

Concerning the streamlines, we have almost the same shape for all simulations. As the injection is from the bottom no disturbance because of the temperature difference can be seen.
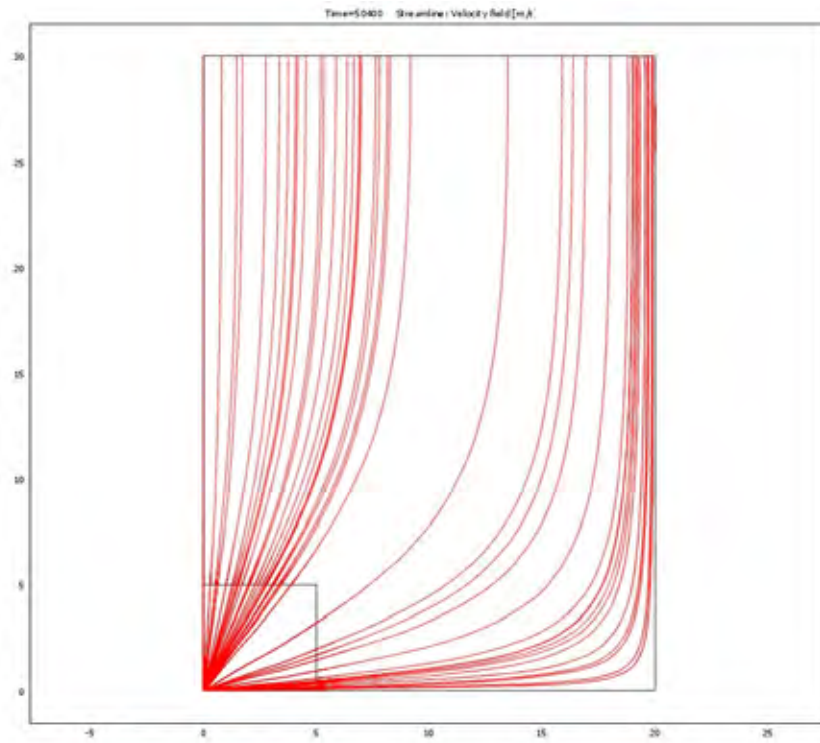
Figure 2-20   Streamlines for 770mm diameter inlet and 3m/s injection.

2.3.4.4   Temperature profile

We will make a very rough analysis of the temperature. In fact differences between simulations are not of real interest, it is always the same shape, with a front advancement which decreases when the injection rate goes down as it is shown on Figure 2-21.

The high or low velocity has no impact on the temperature front. It seems that the main factor is the injection rate. Of course one could say that the growing of the temperature front is very small and ask why. Actually the growing of the area at 0°C is not linear as it is dependent of convection; the temperature decrease relatively quickly around the inlet then when the velocity is lower the temperature propagation is governed by conduction thus it is slower.
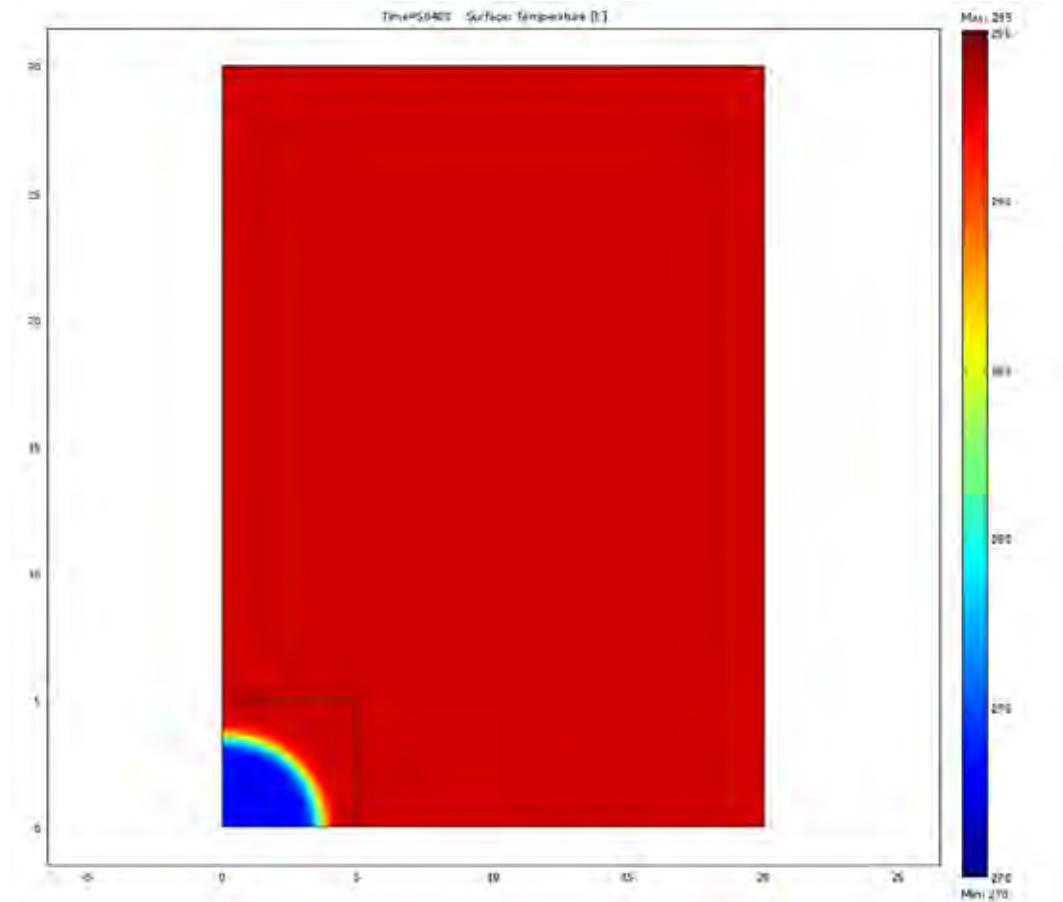
Figure 2-21   Temperature profile at 50400s for 770mm diameter inlet and 3m/s injection.

2.3.5   Shortcomings of the model

Here the problem is axisymmetric and we use true velocities, so there is no problem concerning the inertial loss.

We still make the following approximations, the top of the silo has not been represented, and chemical species are not specified. We still assume that the air molar mass is equal to the nitrogen molar mass and can be used for the two gases.

This time a parabolic inlet was used and we have a 100% concentration everywhere when the steady state is reached. Some simulations were performed with a plug inlet, and we still had maximum concentration everywhere, but within a shorter time. (A stabilization technique was not used here, but was used in the earlier runs with a plug flow where the grid at the inlet was less optimal.) It is important to keep in mind that difference is less than 10% and we do not have to take account of it in this specific study as long as we just study the general shape of the gas spread, but if later on more accurate simulations have to be done this point should be checked deeper.

2.3.6   Conclusions for the large simulations

We have clearly established the general shape of filling a silo with inert gas. Now we know that the inlet velocity of the gas or the diameter of the inlet has no major influence

on the spreading of the gas. The only parameter that has an influence is the injection rate, which permits a more or less fast filling.

Another valuable point is that the spread rate is faster upwards than laterally, it means that the top of the silo reach a 100% concentration faster than the area close to the walls. This means that for higher and narrow silos only one inlet is enough to fill it completely. On the other hand for very wide silos the choice of only one inlet with high flow can be a bad choice in regard of the cost of the process. Indeed a lot of gas is wasted unnecessary through the top of the silo. It would be preferable to have more inlets in order to cover a larger area in this case.

3 Conclusions

This study has resulted in a good model base for simulation of gas flows in a porous media with COMSOL software. It has also been possible to simulate different scenarios, and we are now aware of the behavior of the gas spreading in a silo. This result is very valuable for the silo builder and could also with additional research provide guidelines to fire brigades.

Further improvement of the model could be to add an air phase on the top of the silo and to include the difference between air and nitrogen, i.e. include transport equations for species.

4   References

The literature listed below have been valuable in the work with the model.

[1] Siew Mei Lim,Michael Yit Lin Chew, Buildig Department, School of Design and Environment, National University of singapore.(Interflam2007),Interaction between natural convection and chemical reaction in spontaneous ignition of solids

[2] Fluent 6 user's guide volumes 1 and 2

[3] Ulf Göransson, Department of Fire Safety Engineering Lund University Sweden,Determination of material properties for fire modelling

[4] Zhenghua Yan, Per Blomqvist, Ulf Göransson, Göran Holmstedt,Lars Wadsö, Patrick Van Hees, Validation of CFD model for simulation of spontaneous ignition in biomass fuel storage

[5] http://www.lmnoeng.com (Web site to determine constants)

[6] COMSOL manual

Appendix 1: Effective diffusivity calculation
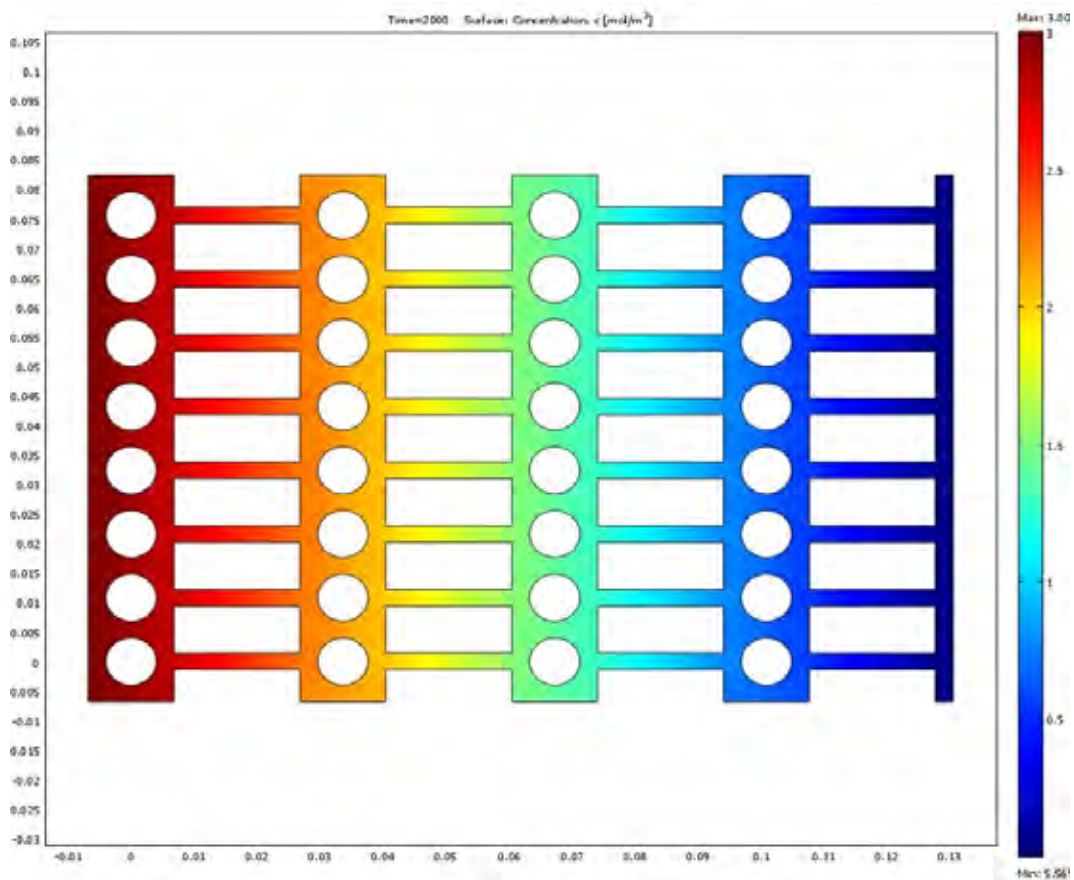
Figure A1-1   Concentration profile from the determination of an effective diffusivity.

The example in Figure A1-1 gives the concentration field through a geometry supposed to represent pellets. The method was taken from the COMSOL manual. Of course it is very far from the reality, this is why several configurations were used and an average of the effective diffusivity was calculated.

The principle is very simple. We set an initial concentration of zero in the entire domain, and then we apply a concentration on one side. When the steady state is reached we just have to use the Fick law to find the effective diffusivity, which is dependent of the tortuosity.

$$\vec{J} = -\text{Deff} \bullet \nabla c$$

Where J is the diffusive flux in mol.m-2.s-1 and Deff the effective diffusivity in m².s-1 and c the concentration in mol.m-3.

Finally we find a value of 6E-6 m².s-1 for Deff..

Reference:

COMSOL manual – Diffusion and convection

Appendix 2: Kristinehamn silo simulation result
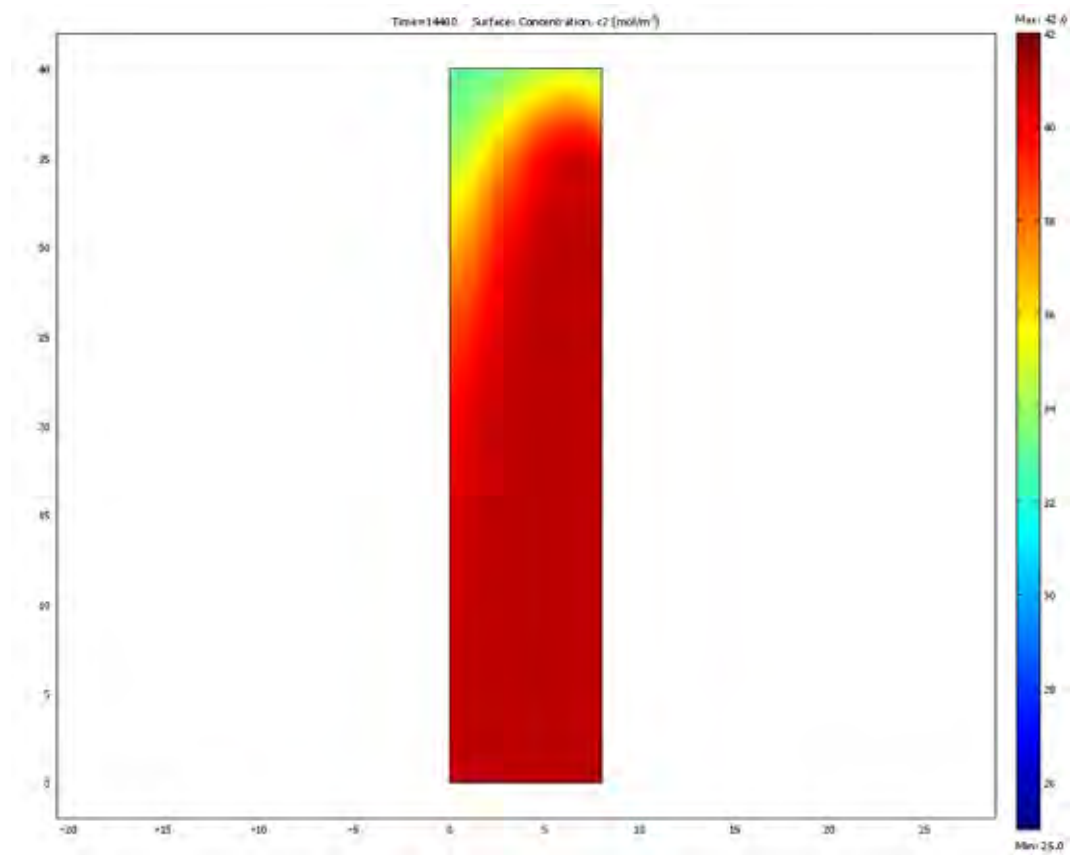
Figure A2-1 Kristinehamn silo concentration after 4 hours with a parabolic velocity shape.

Appendix 3: Simulation plan for gas distribution study

Instructions for initial simulations in the BRANDFORSK project:

"Based on the discussions during the telephone meeting today, below is an updated table specifying the simulation scenarios.

Please note that all simulations at this moment should be made on a 40 m diameter and 30 m high silo with just one gas inlet in the centre of the silo.

Instead of "high/low" inlet velocity, as shown in the previous document, I'm specifying the inlet diameter instead. After our telephone meeting we decided to use 3 m/s as "low" velocity and 30 m/s as "high" velocity which corresponds to a factor of 10. Based on this I have calculated the inlet diameters corresponding to each specified injection rate. In the transformation of the injection rate from kg/h to m3/h, I have assumed a gas temperature of 0 ˚C of the nitrogen which corresponds to 0,8 m3 /kg."

Table 2   Plan for silo simulations - revised after telephone meeting 2007-11-20.

Simulation no	Number of inlets	Injection rate (kg/h)	Injection rate (m^3 /h)	Inlet velocity (m/s)	Inlet diameter (mm)
1	1	6280	5024	3	770
2	1	6280	5024	30	243
3	5	1256	1005	3	344
4	5	1256	1005	30	109
5	10	628	502	3	243
6	10	628	502	30	77
7	25	251	201	3	154
8	25	251	201	30	49

Appendix 4: Simulation results - concentration

Simulation 1 - 5024 m3/h, 3 m/s, diam. 770 mm (single inlet situation)
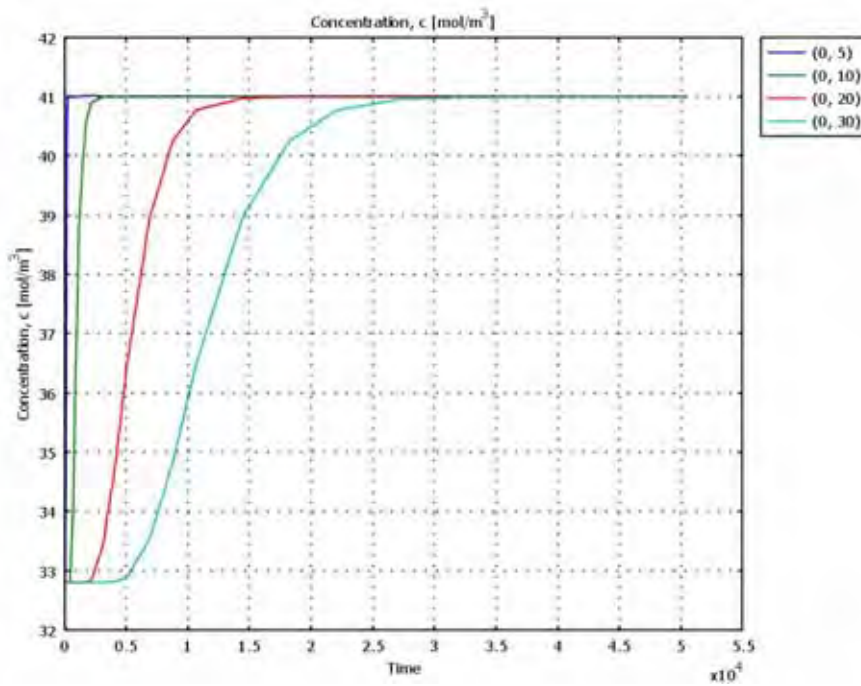
Figure A4-1   Simulation 1: concentration against time at 5,10,20 and 30m height in the centre. (Note: the x-scale is in seconds. 50400 sec = 14 h.)
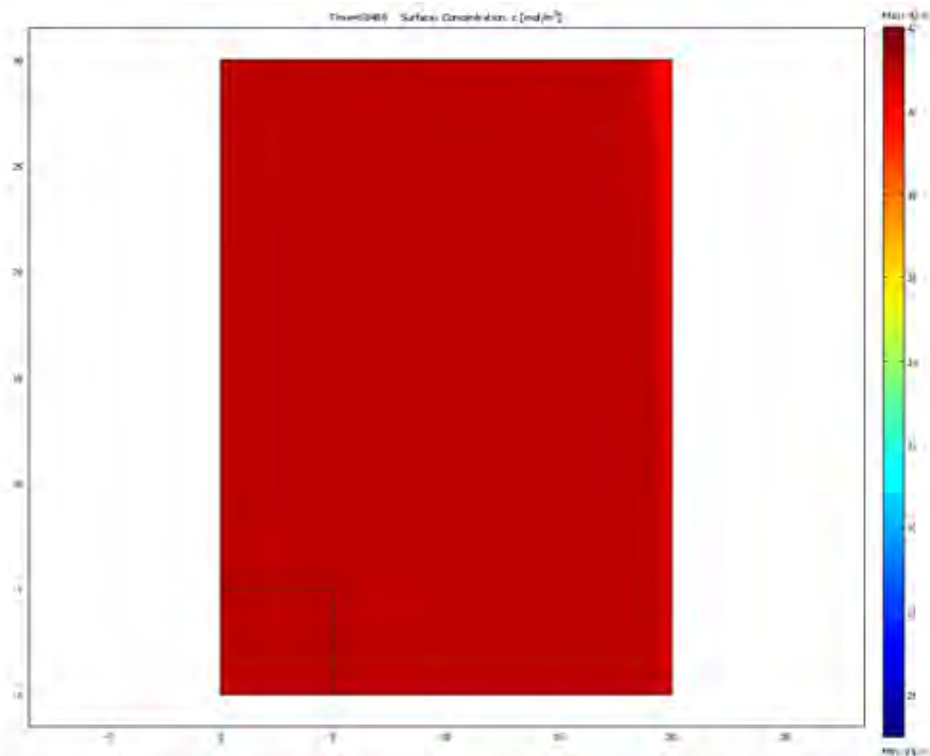
Figure A4-2   Simulation 1: concentration field.

Simulation 2 - 5024 m3/h, 30 m/s, diam. 243 mm (single inlet situation)
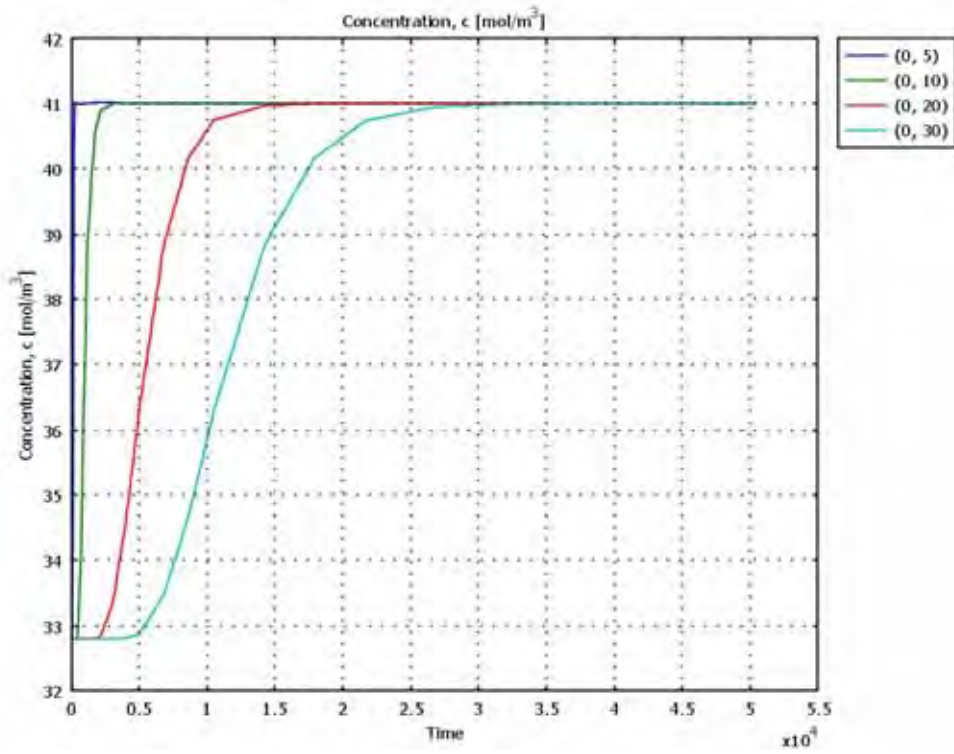
Figure A4-3   Simulation 2: concentration against time at 5,10,20 and 30m height in the centre.
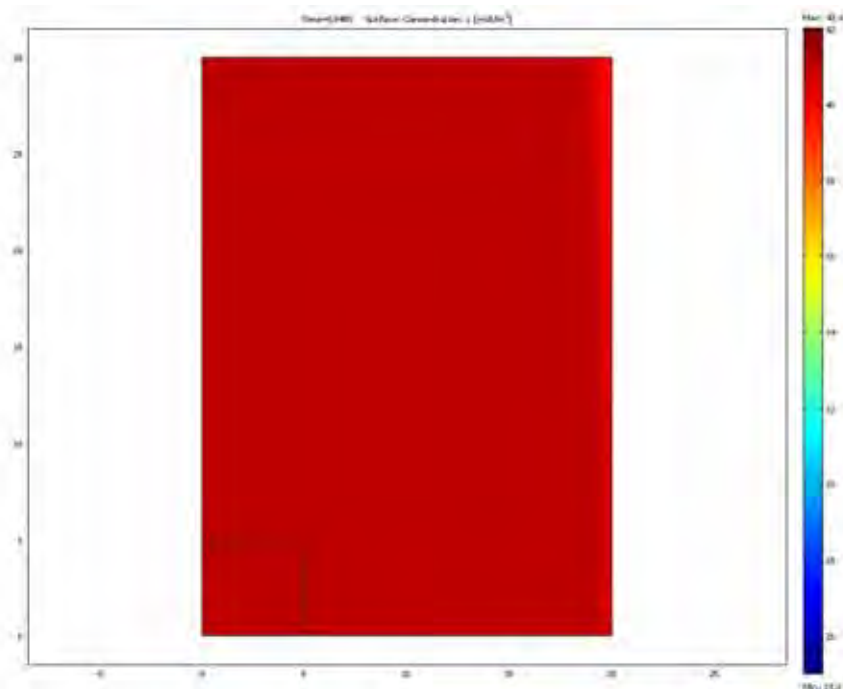
Figure A4-4   Simulation 2: concentration field.

Simulation 3 - 1005 m3/h, 3 m/s, diam. 344 mm (five inlet situation)
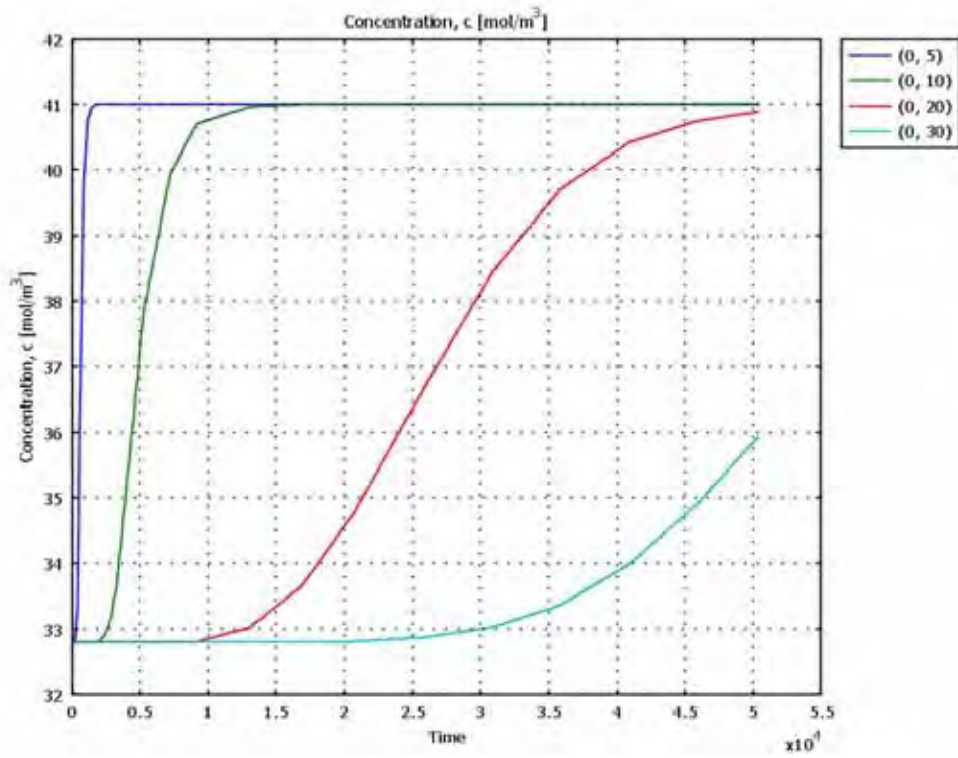
Figure A4-5   Simulation 3: concentration against time at 5,10,20 and 30m height in the centre.
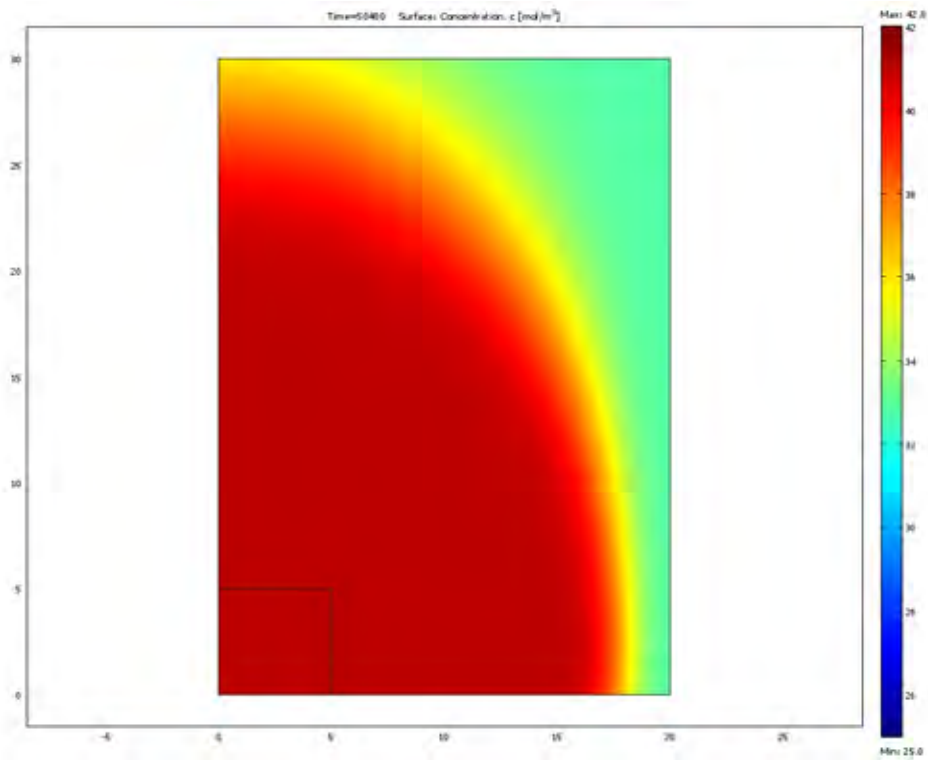
Figure A4-6   Simulation 3: concentration field.

Simulation 4 - **1005 m3/h, 30 m/s, diam. 109 mm (five inlet situation)**
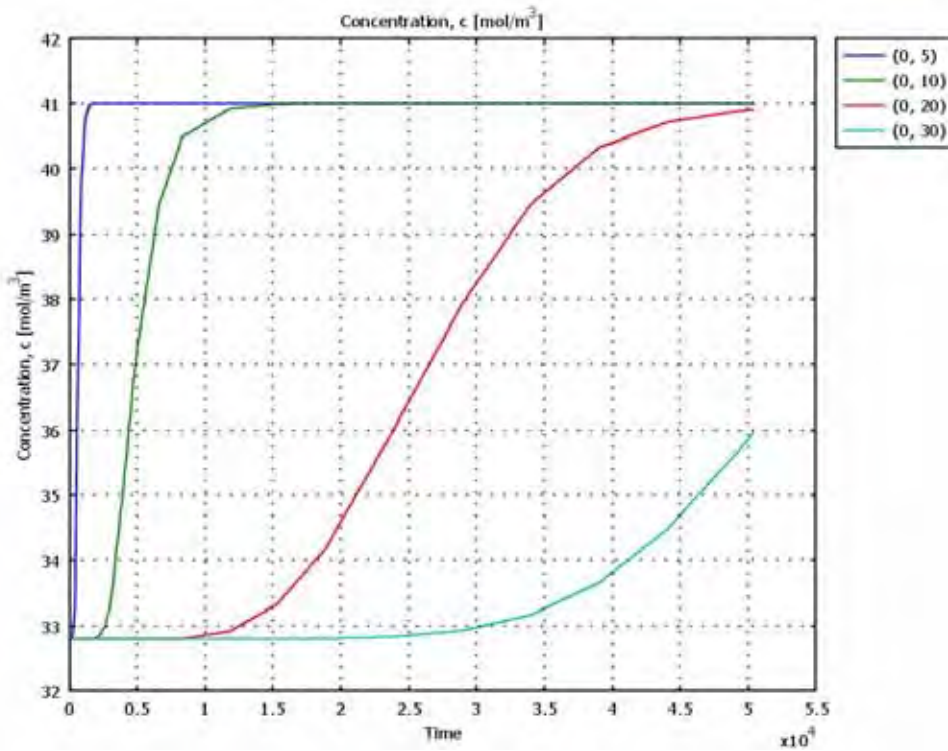
Figure A4-7 Simulation 4: concentration against time at 5,10,20 and 30m height in the centre.
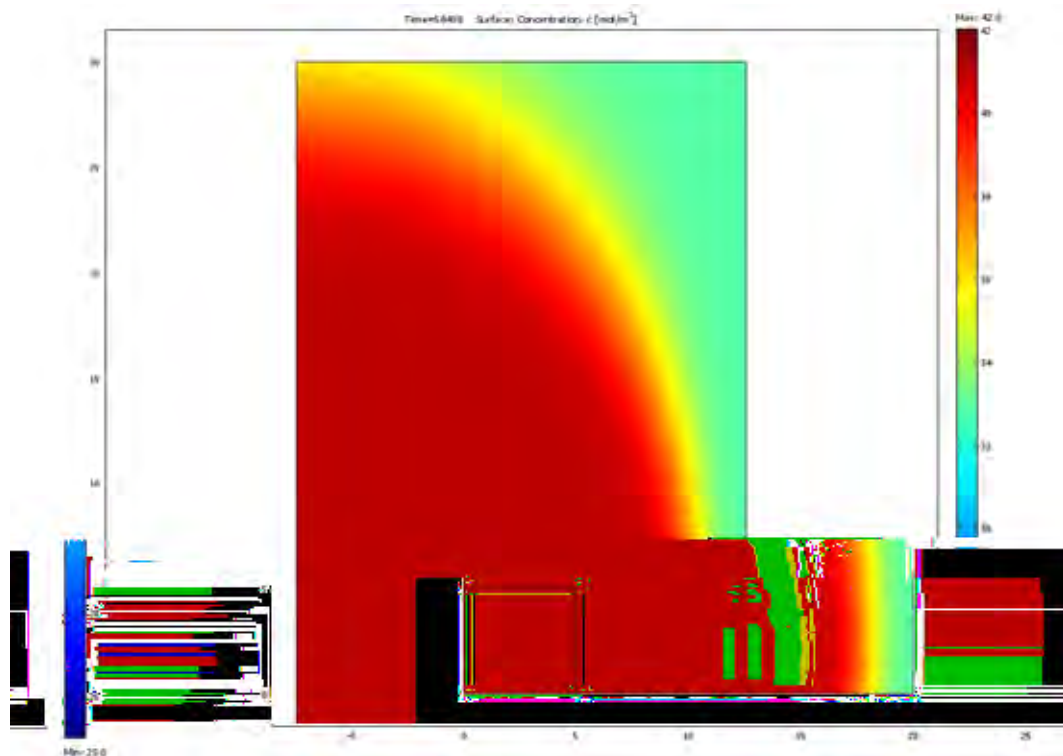
Figure A4-8 Simulation 4: concentration field.

Simulation 5 - 502 m3/h, 3 m/s, diam. 243 mm (10 inlet situation)
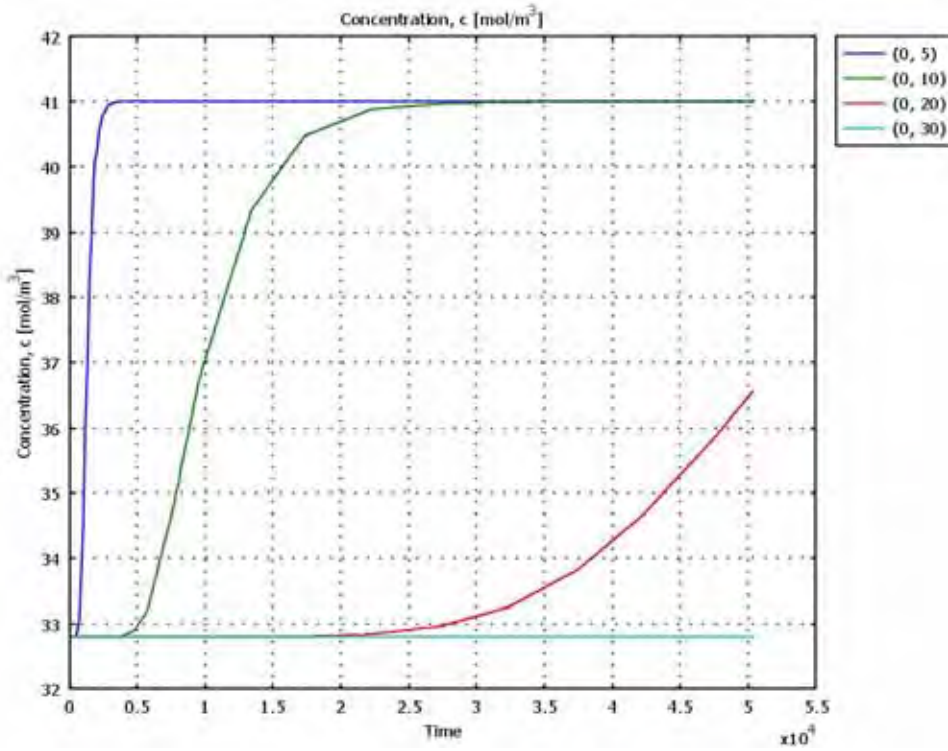
Figure A4-9   Simulation 5: concentration against time at 5,10,20 and 30m height in the centre.
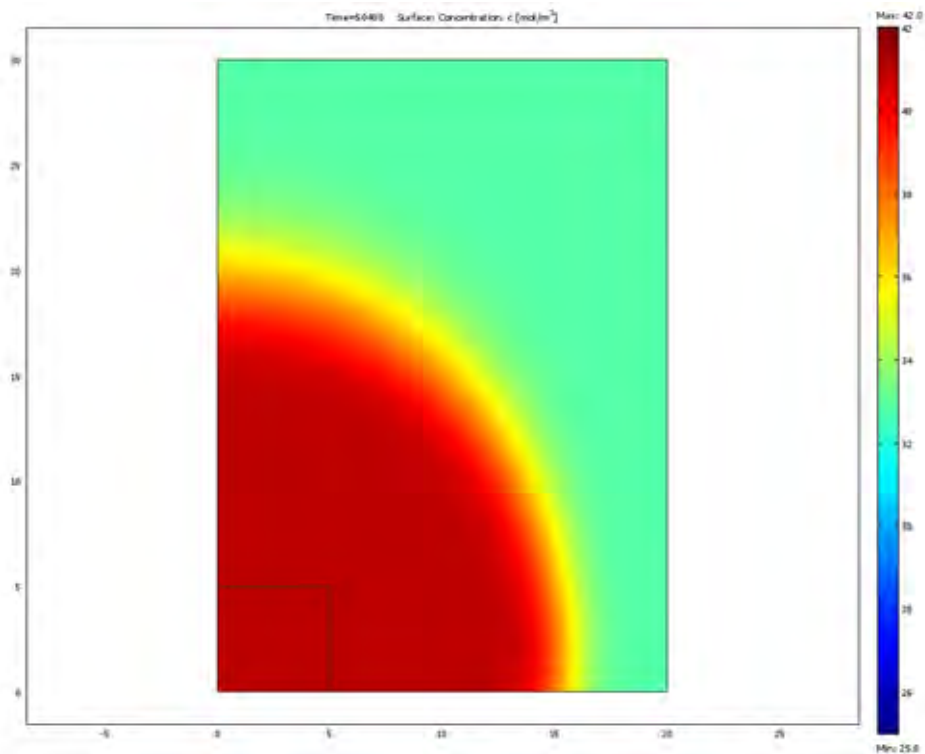
Figure A4-10   Simulation 5: concentration field.

Simulation 6 - 502 m3/h, 30 m/s, diam. 77 mm (10 inlet situation)
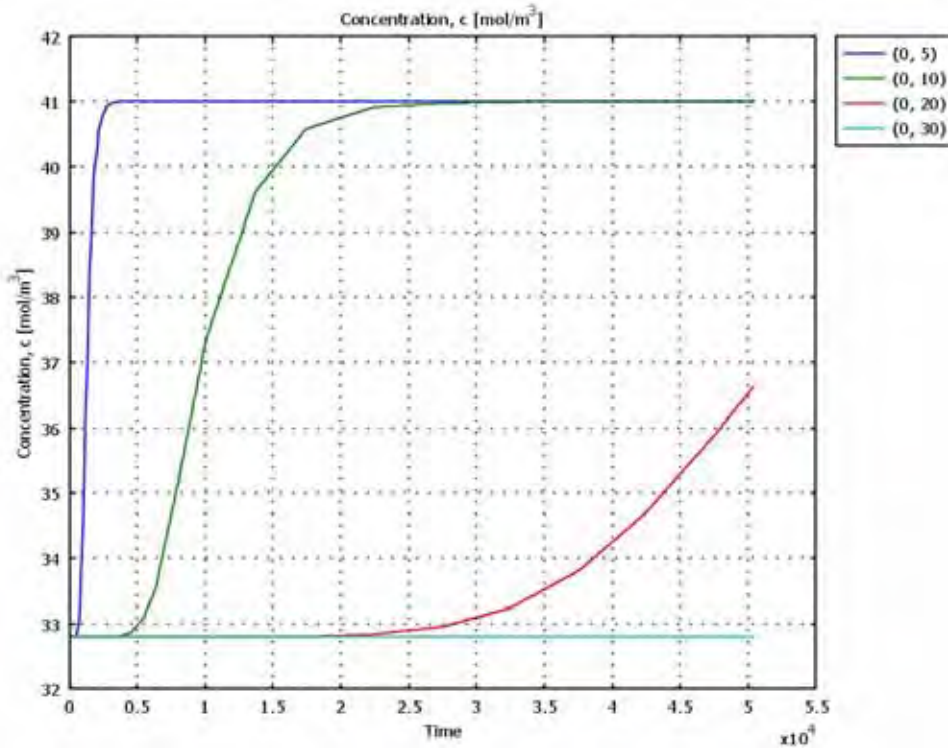
Figure A4-11 Simulation 6: concentration against time at 5,10,20 and 30m height in the centre.
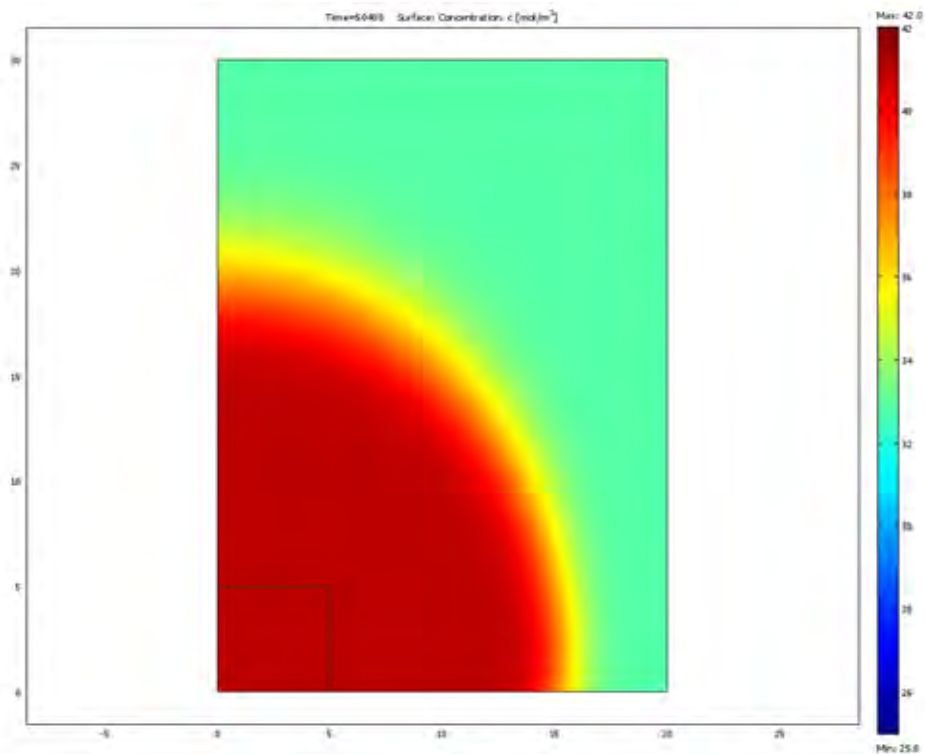
Figure A4-12 Simulation 6: concentration field.

Simulation 7 - 201 m3/h, 3 m/s, diam. 154 mm (25 inlet situation)
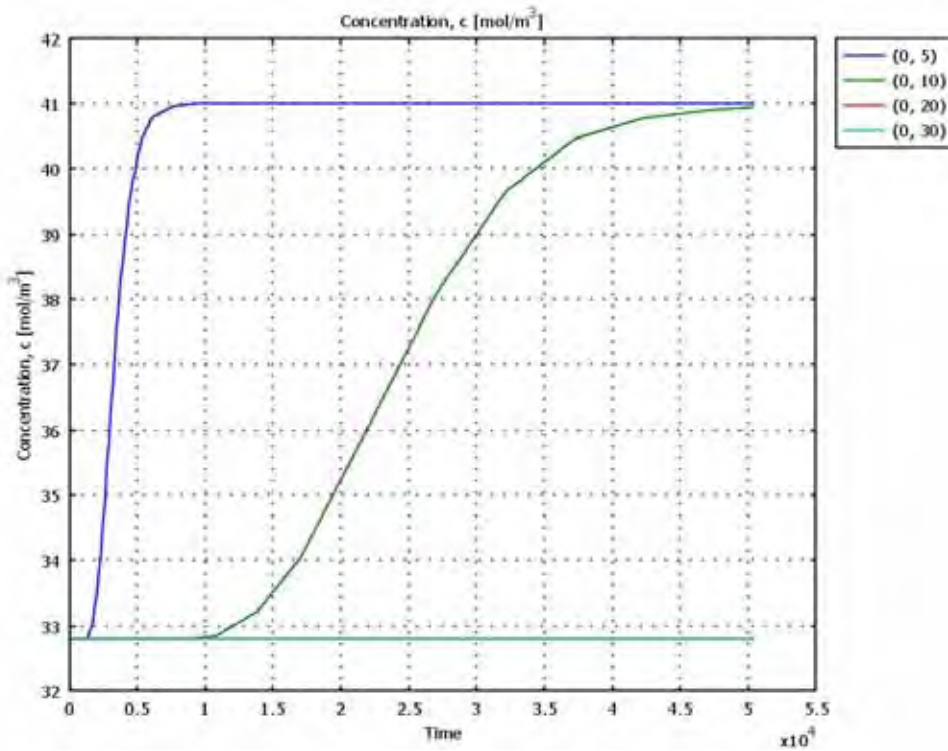
Figure A4-13 Simulation 7: concentration against time at 5,10,20 and 30m height in the centre.
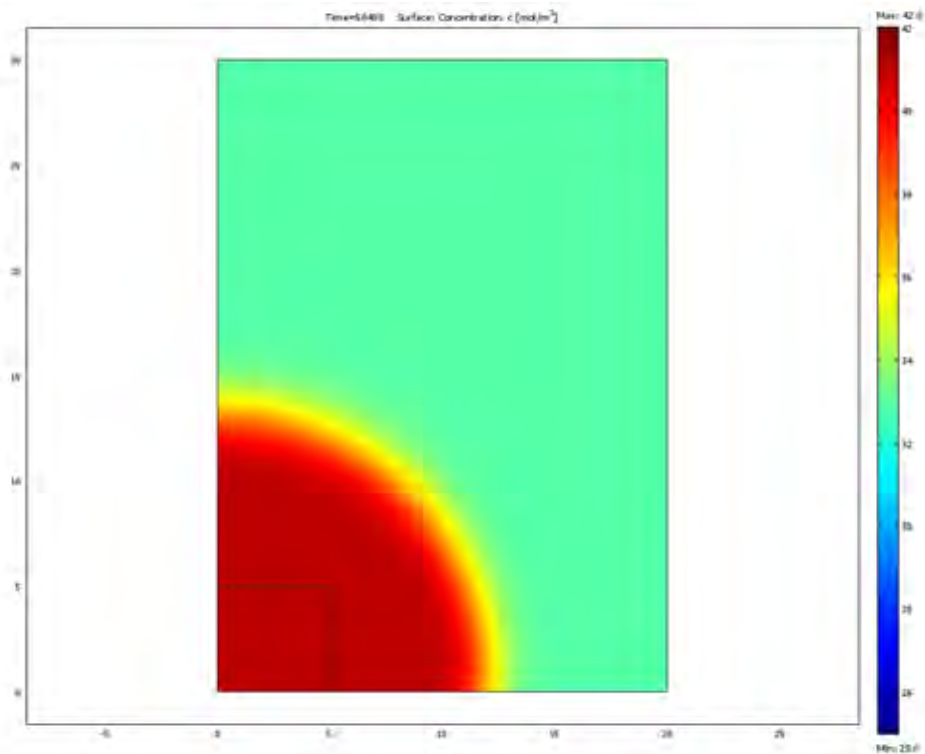
Figure A4-14 Simulation 7: concentration field.

Simulation 8 - 201 m3/h, 30 m/s, diam. 49 mm (25 inlet situation)
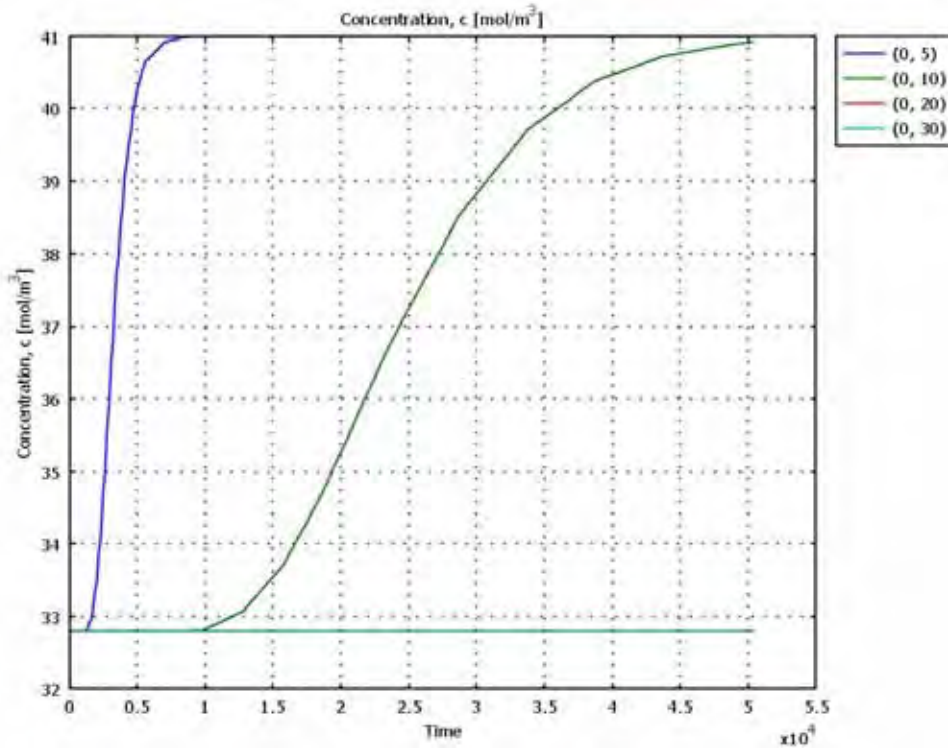
Figure A4-15   Simulation 8: concentration against time at 5,10,20 and 30m height in the centre.
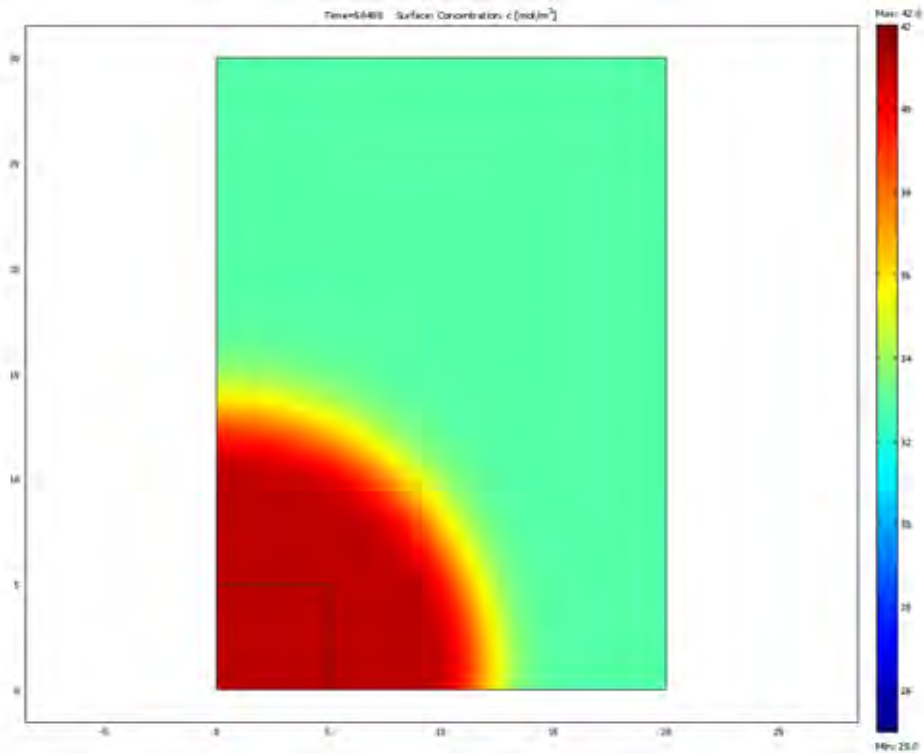
Figure A4-16   Simulation 8: concentration field.

Appendix 5: Simulations results- pressure

<u>**Simulation 1**</u> **- 5024 m3/h, 3 m/s, diam. 770 mm (single inlet situation)**
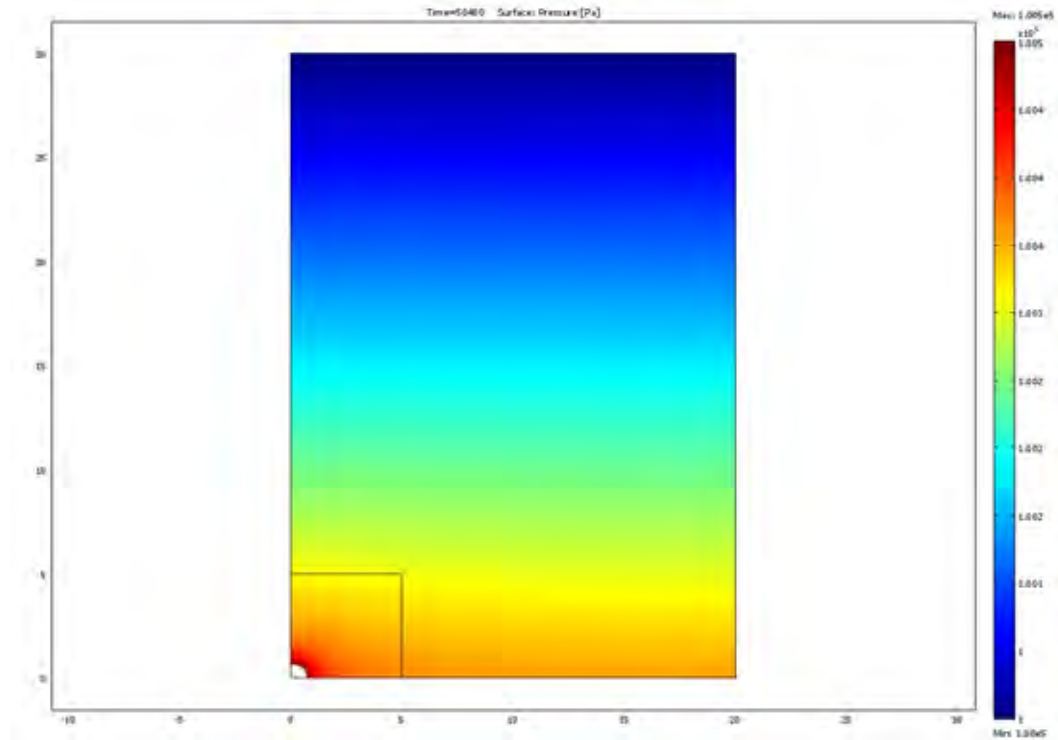
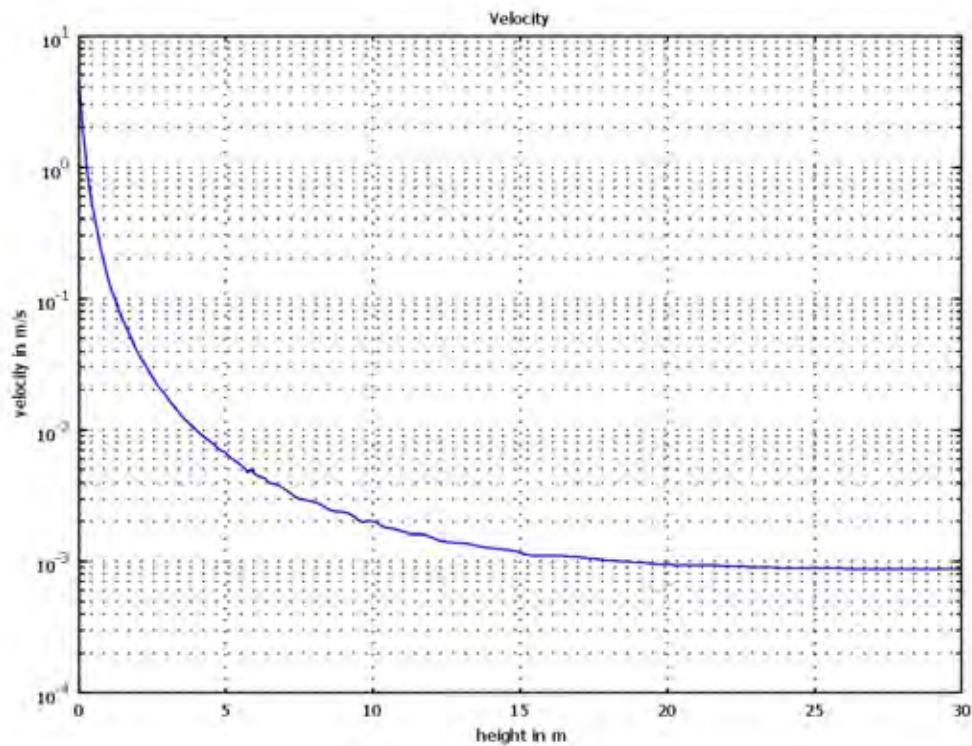**Figure A5-1**   Simulation 1: pressure field.
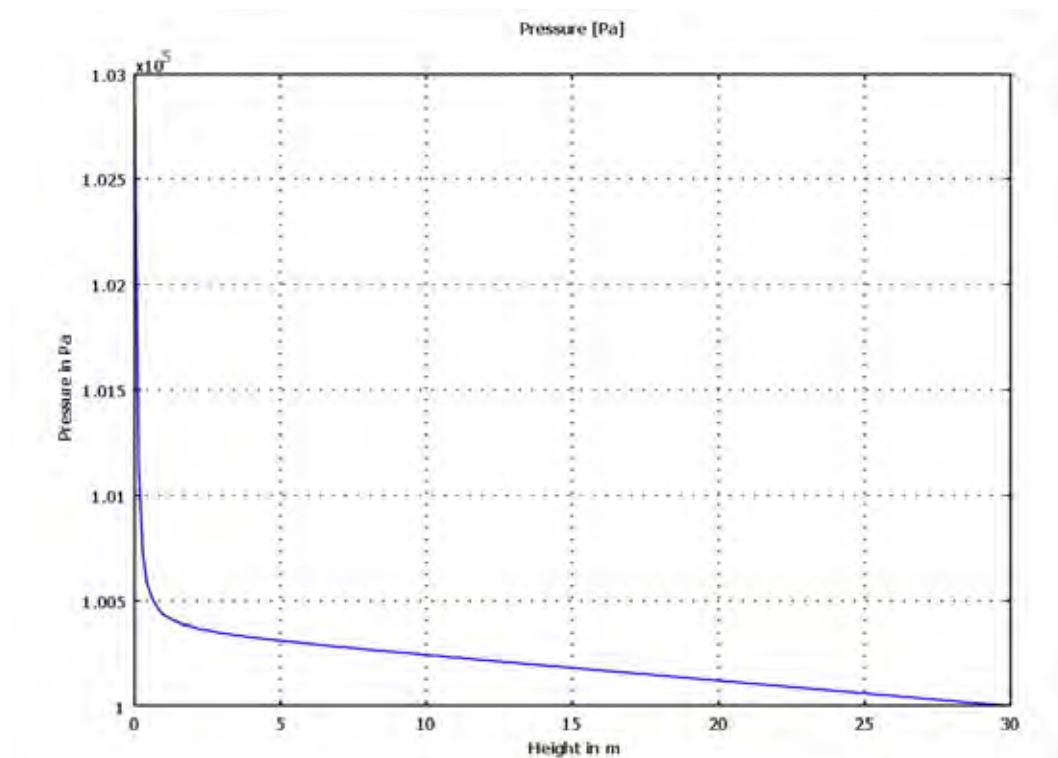
Figure A5-1   Simulation 1: cross section pressure plot against height in the centre.

**Simulation 2 - 5024 m3/h, 30 m/s, diam. 243 mm (single inlet situation)**
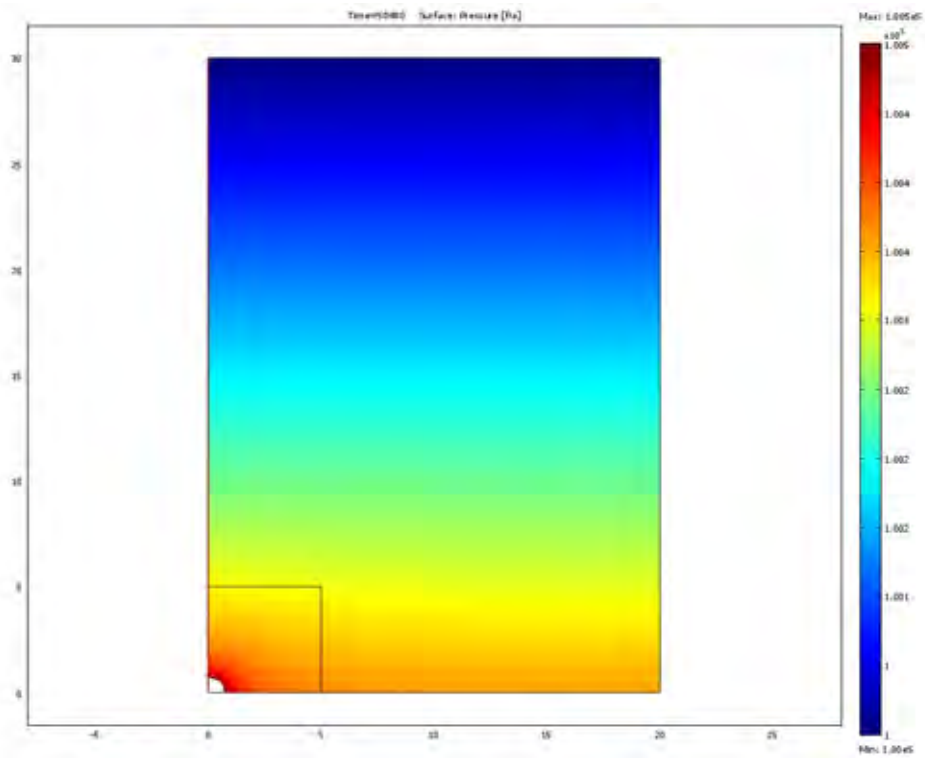
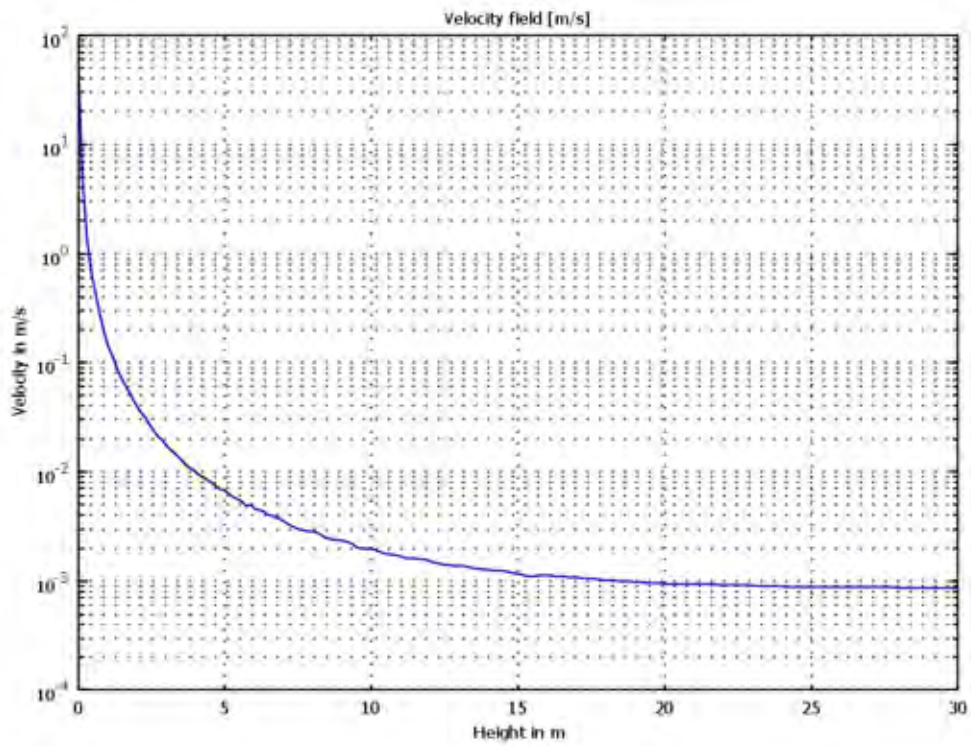**Figure A5-3**   Simulation 2: pressure field.
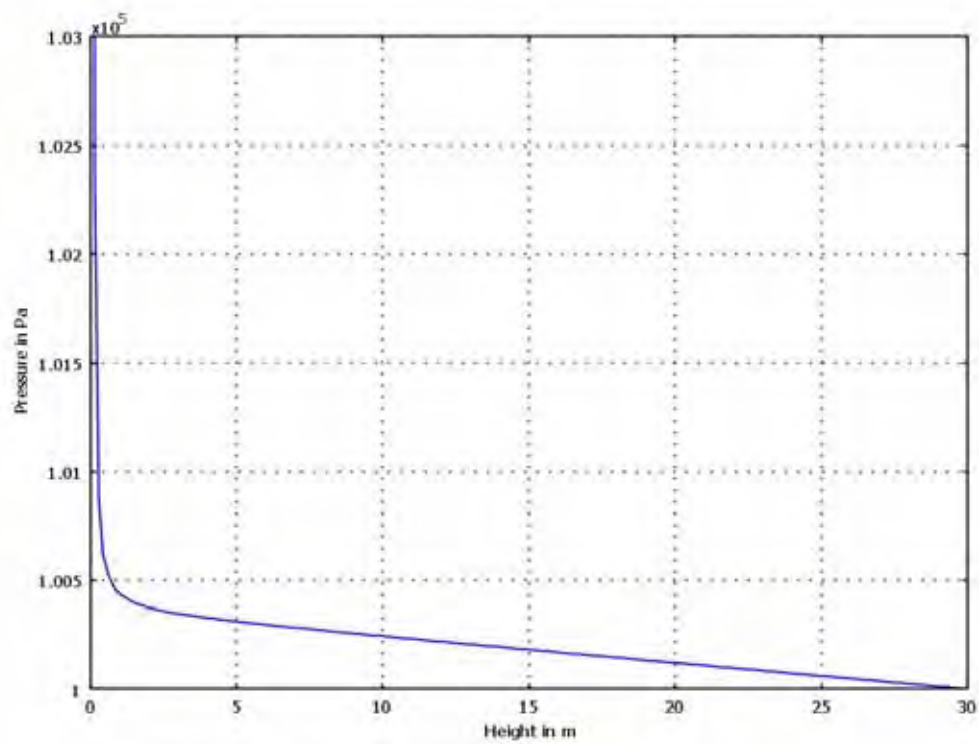
Figure A5-4   Simulation 2: cross section pressure plot against height in the centre.

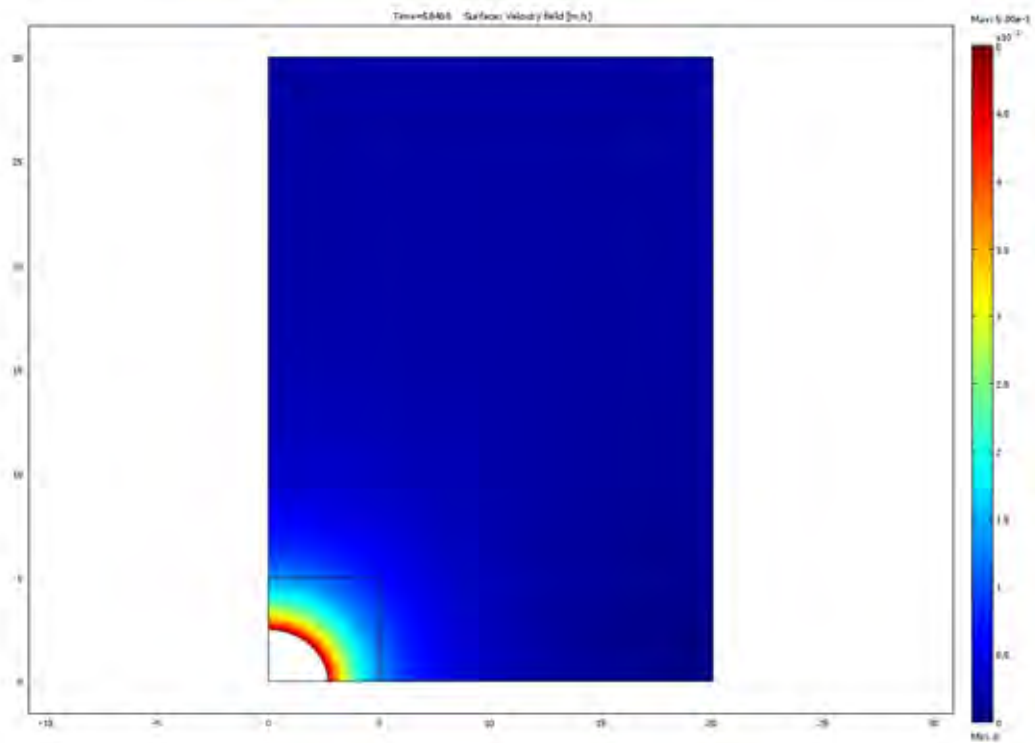**Simulation 3 - 1005 m3/h, 3 m/s, diam. 344 mm (five inlet situation)**
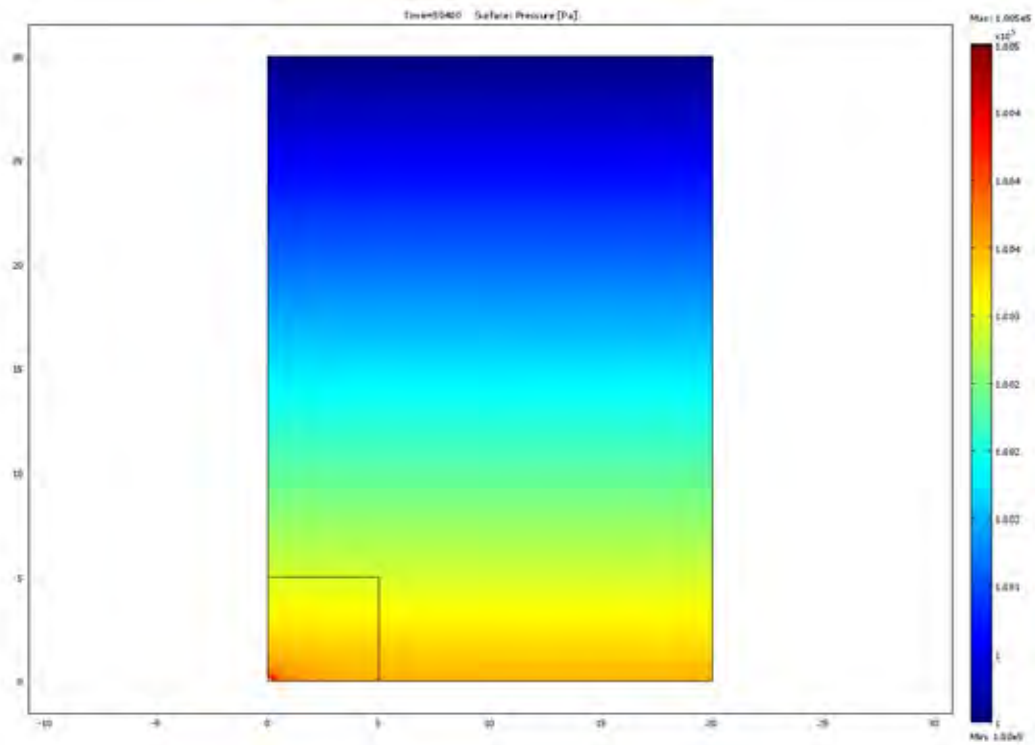
Figure A5-5   Simulation 3: pressure field.
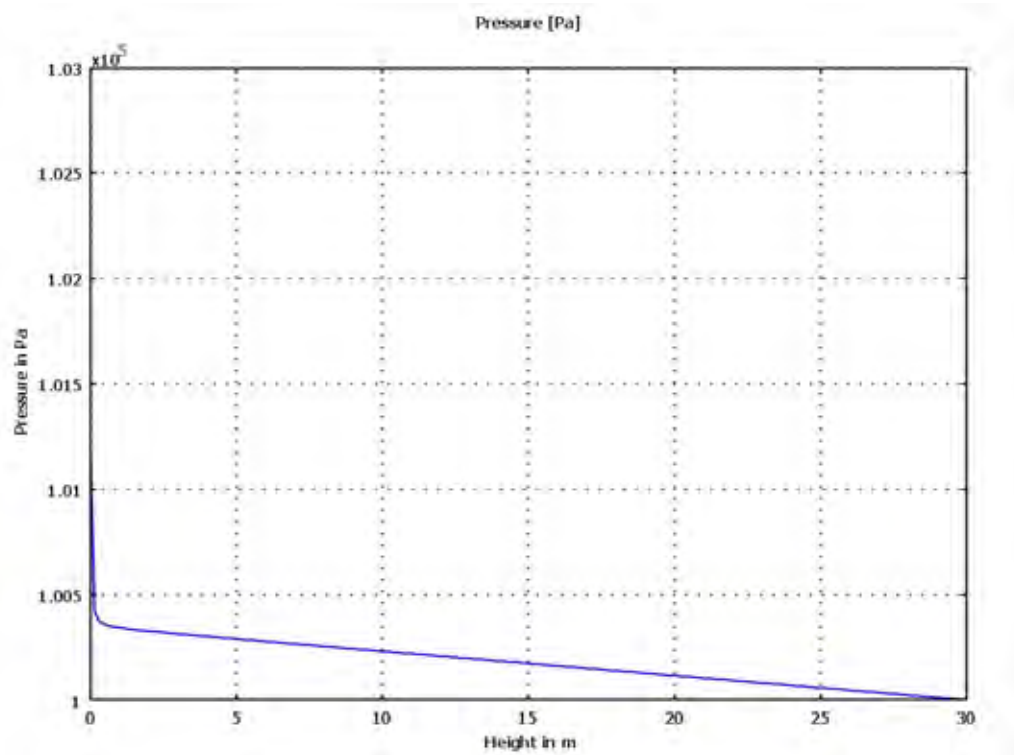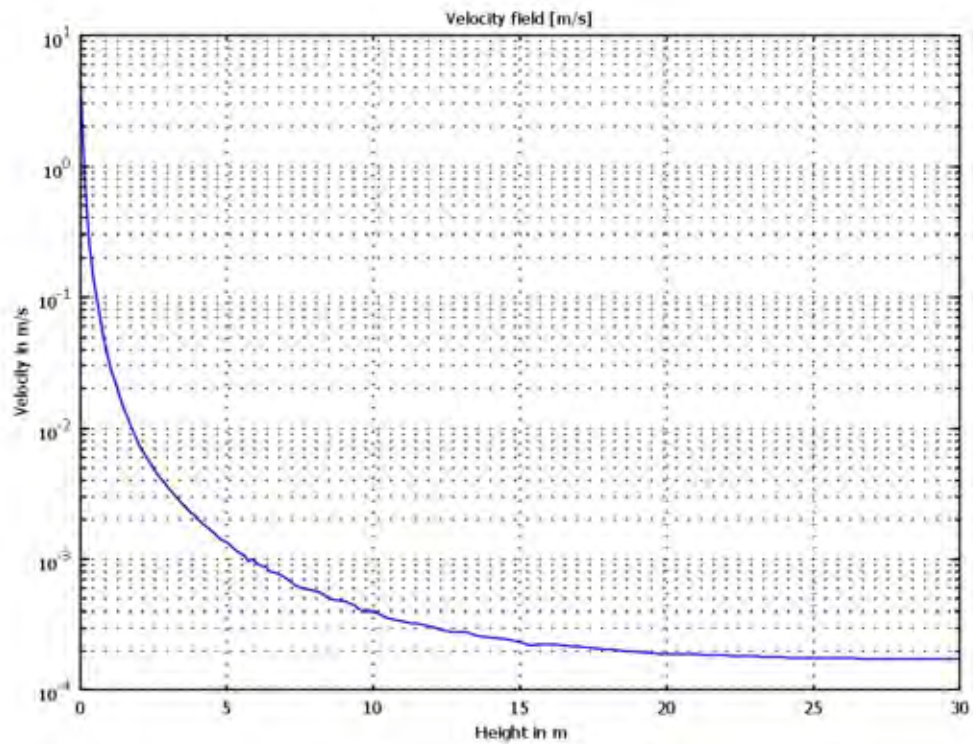

Figure A5-6   Simulation 3: cross section pressure plot against height in the centre.

Simulation 4 - 1005 m3/h, 30 m/s, diam. 109 mm (five inlet situation)
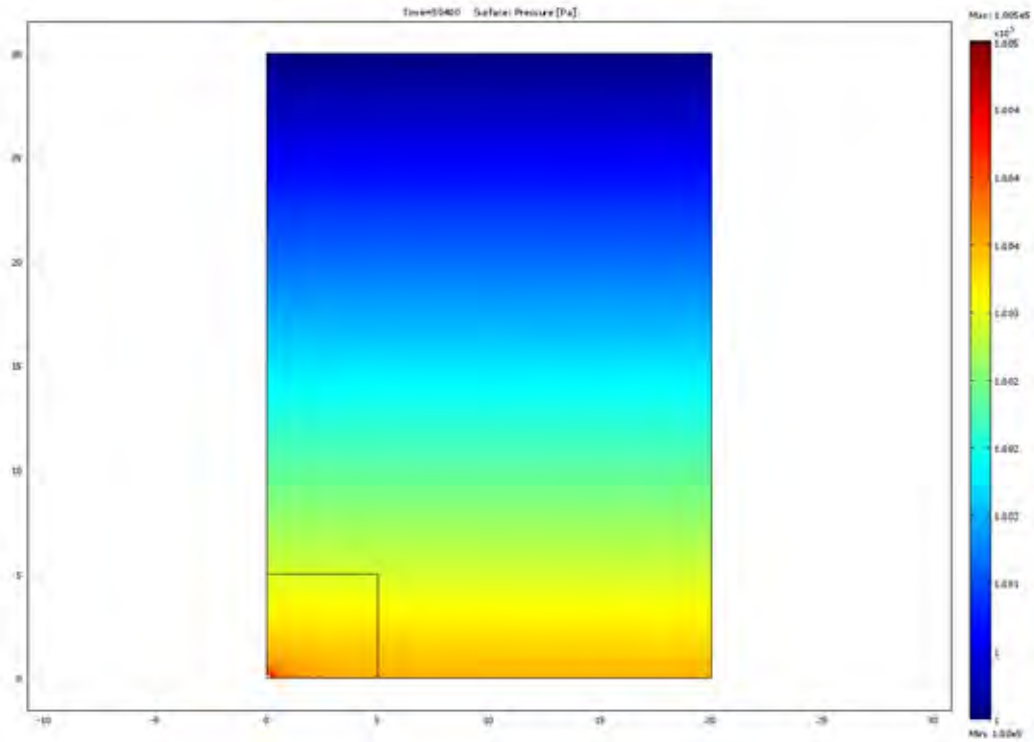
Figure A5-7   Simulation 4: pressure field.
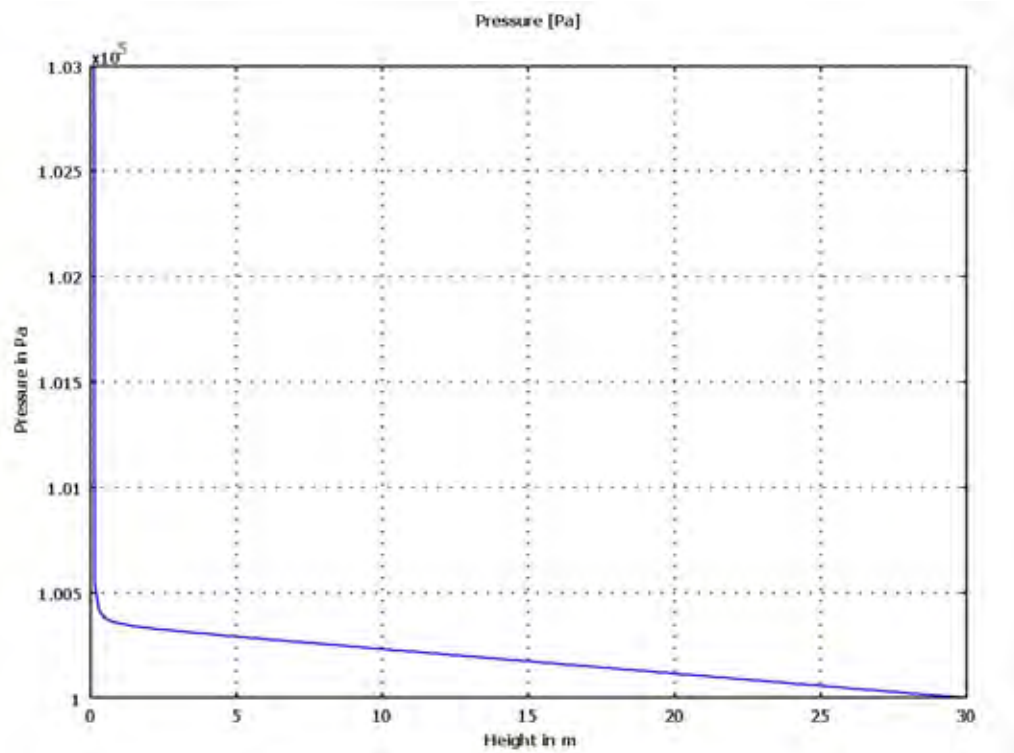
Figure A5-8   Simulation 4: cross section pressure plot against height in the centre.

Simulation 5 - 502 m3/h, 3 m/s, diam. 243 mm (10 inlet situation)
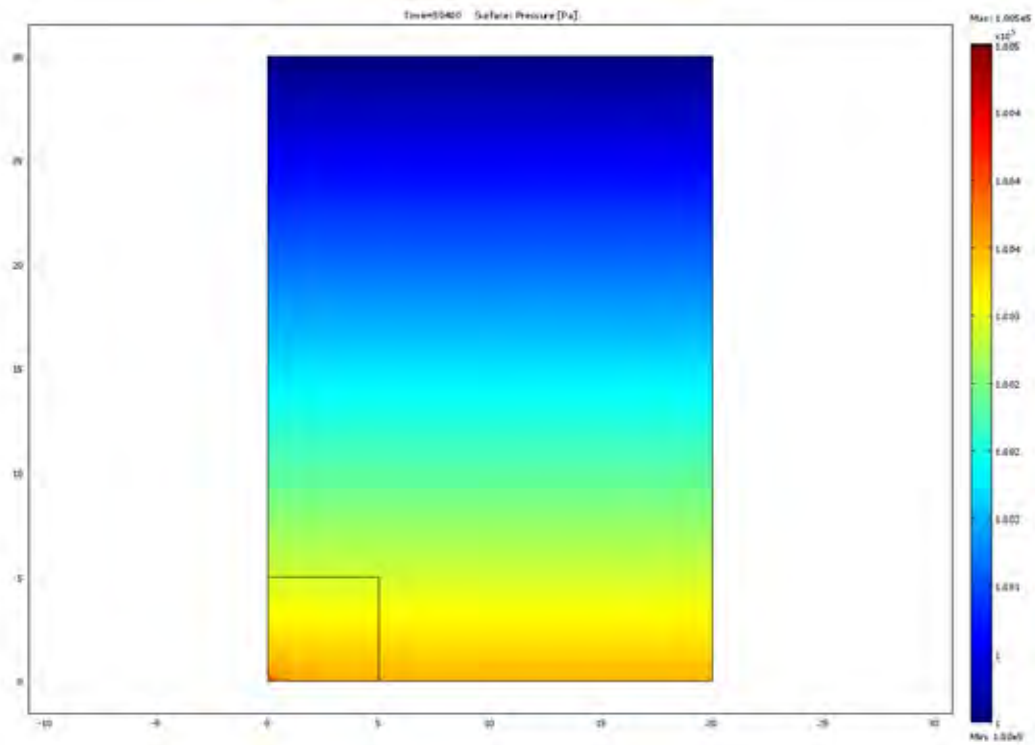
Figure A5-9   Simulation 5: pressure field.
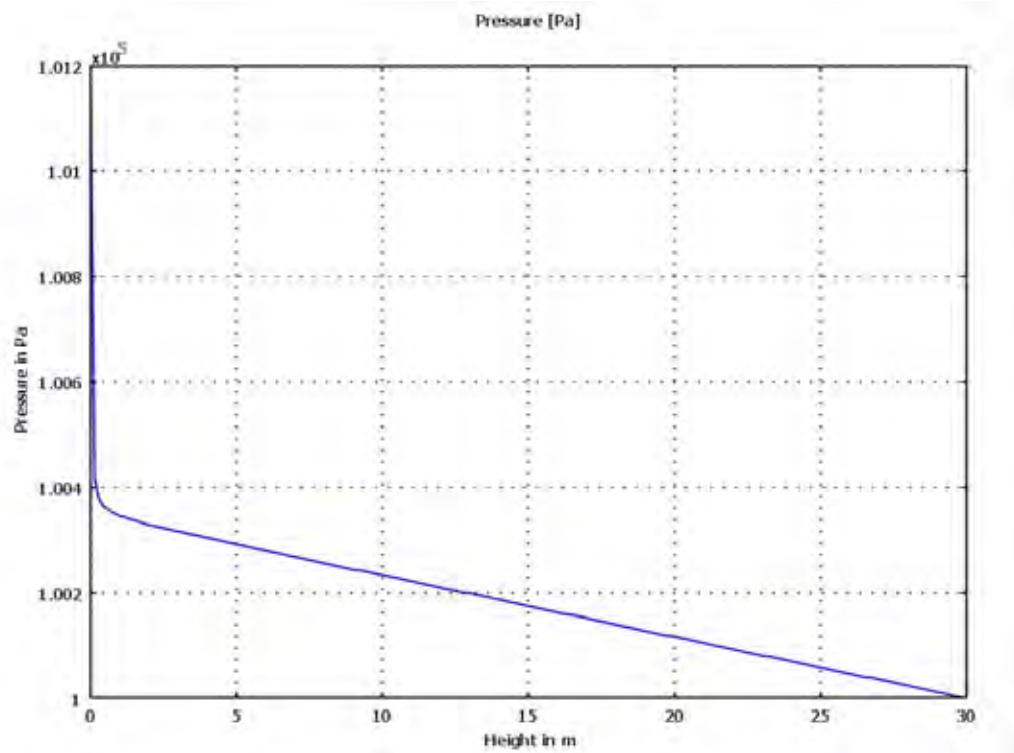
Figure A5-10   Simulation 5: cross section pressure plot against height in the centre.

Simulation 6 - 502 m3/h, 30 m/s, diam. 77 mm (10 inlet situation)
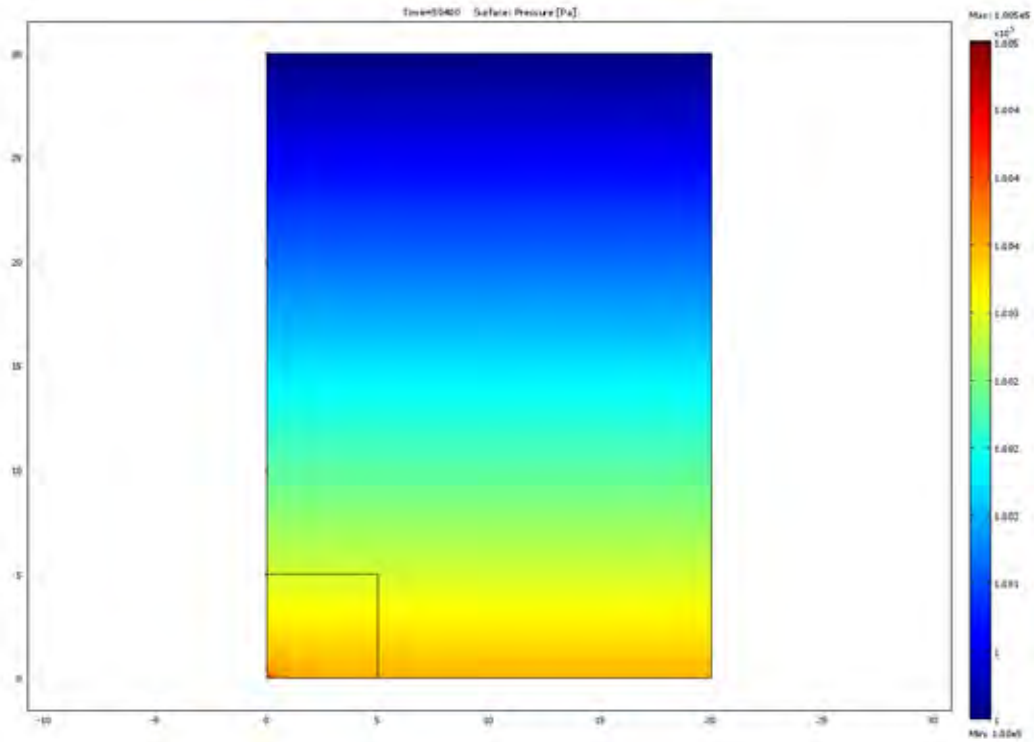
Figure A5-11   Simulation 6: pressure field.
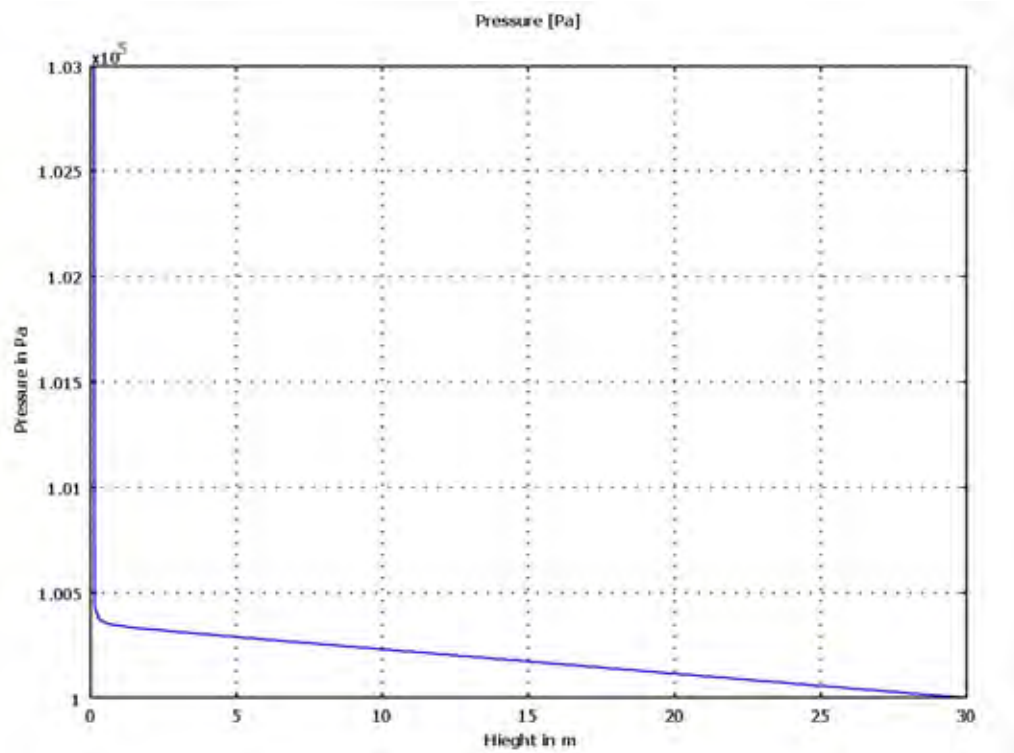
Figure A5-12   Simulation 6: cross section pressure plot against height in the centre.

Simulation 7 - 201 m3/h, 3 m/s, diam. 154 mm (25 inlet situation)
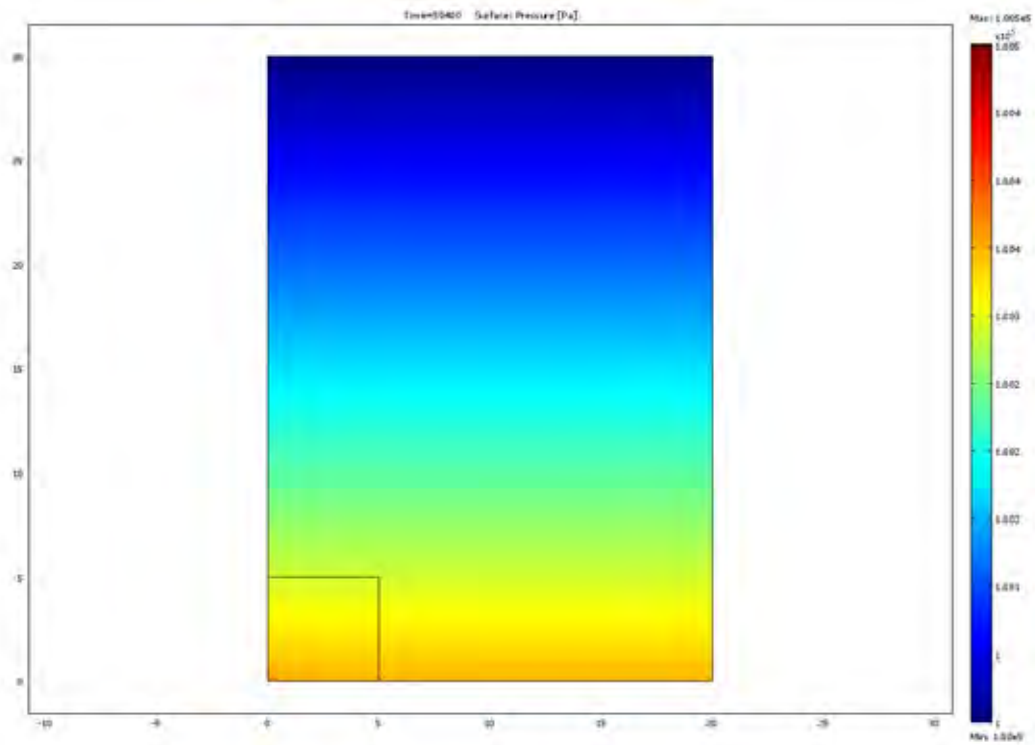
Figure A5-13   Simulation 7: pressure field.
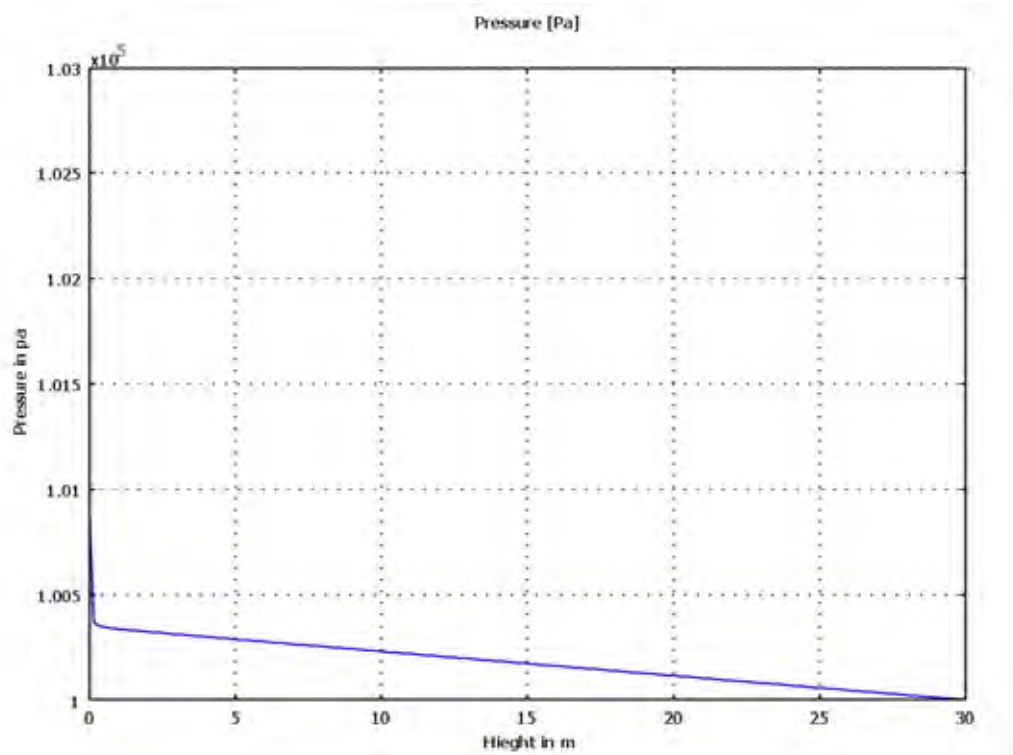
Figure A5-14   Simulation 7: cross section pressure plot against height in the centre.

Simulation 8 - 201 m3/h, 30 m/s, diam. 49 mm (25 inlet situation)
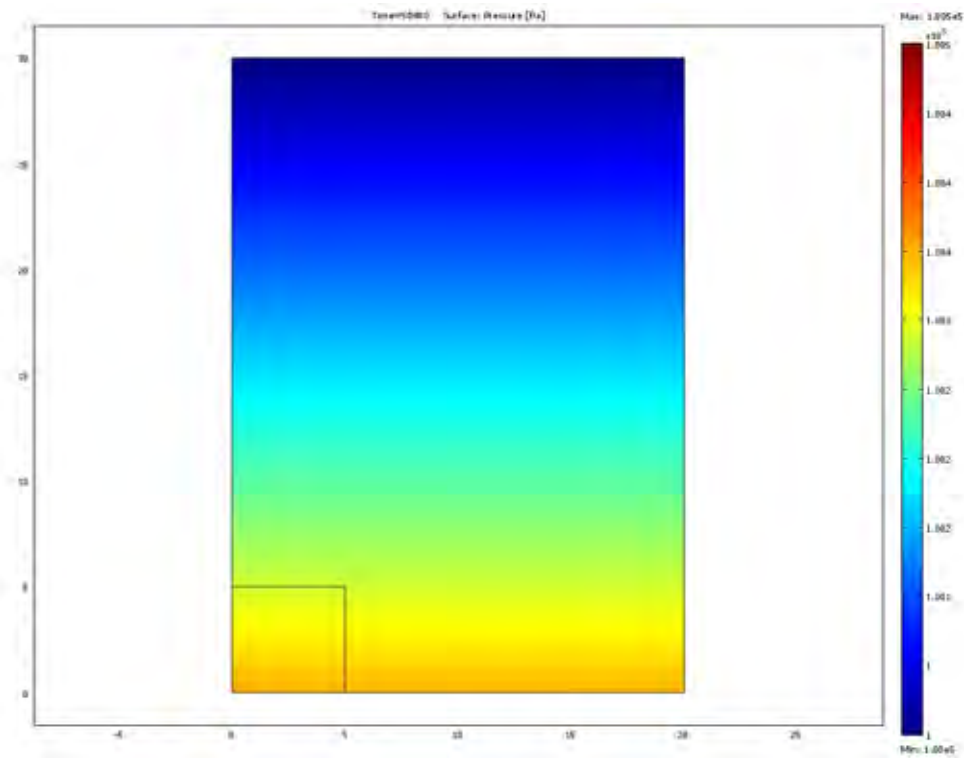
Figure A5-15   Simulation 8: pressure field.
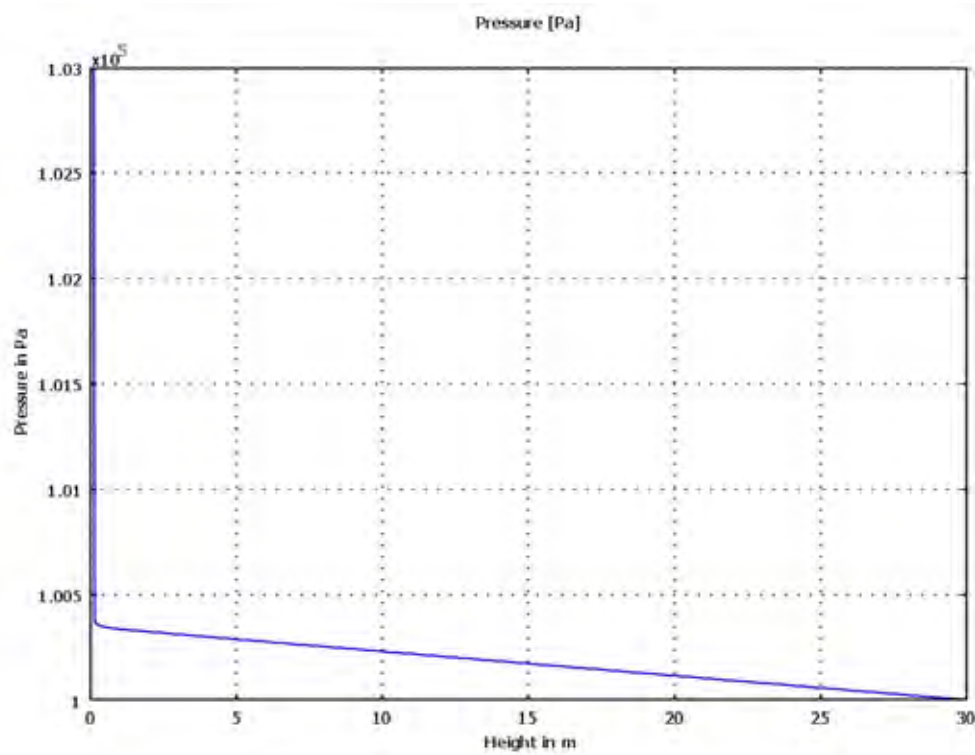
Figure A5-16   Simulation 8: cross section pressure plot against height in the centre.

Appendix 6: Simulations results- velocity

<u>Simulation 1</u> - **5024 m3/h, 3 m/s, diam. 770 mm (single inlet situation)**
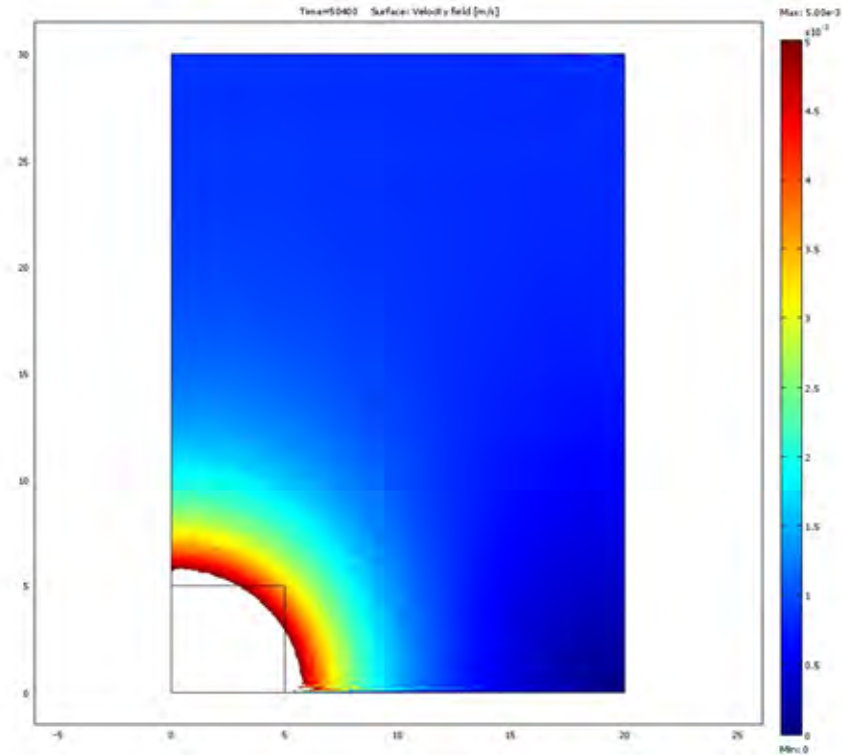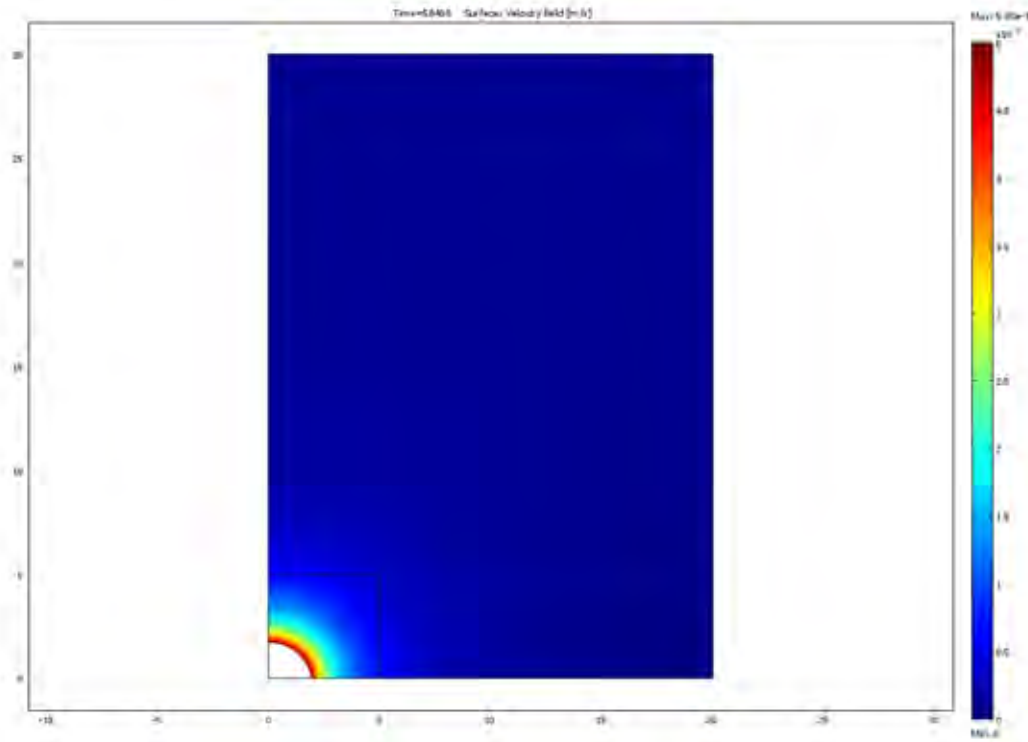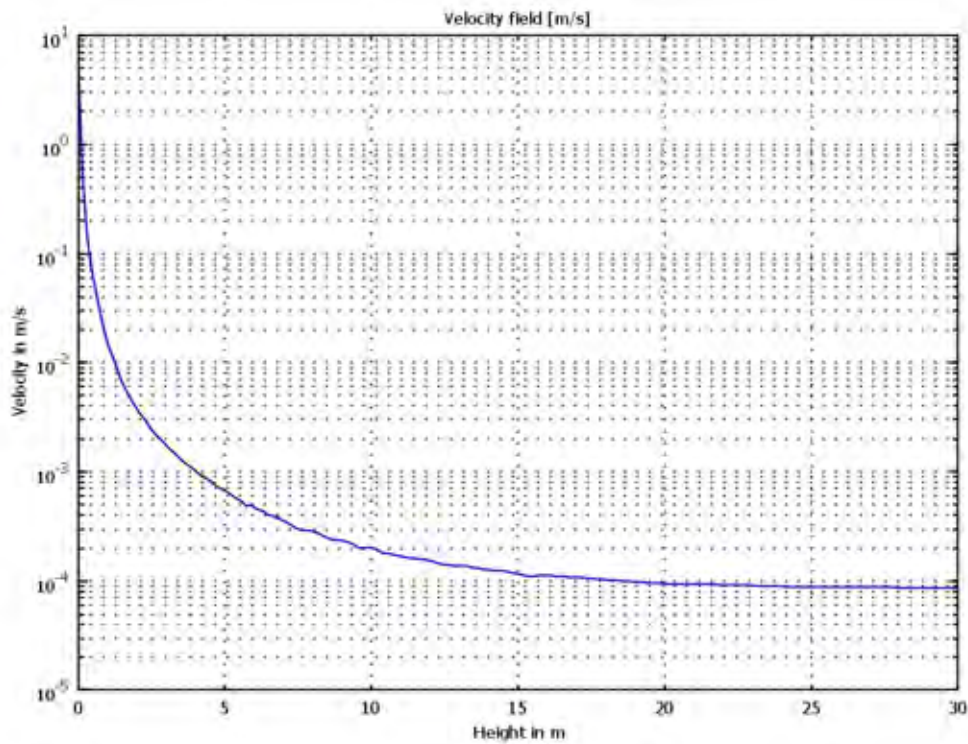

Figure A6-1   Simulation 1: velocity field.

Figure A6-2   Simulation 1: cross section velocity plot against height in the centre.

Simulation 2 - 5024 m3/h, 30 m/s, diam. 243 mm (single inlet situation)
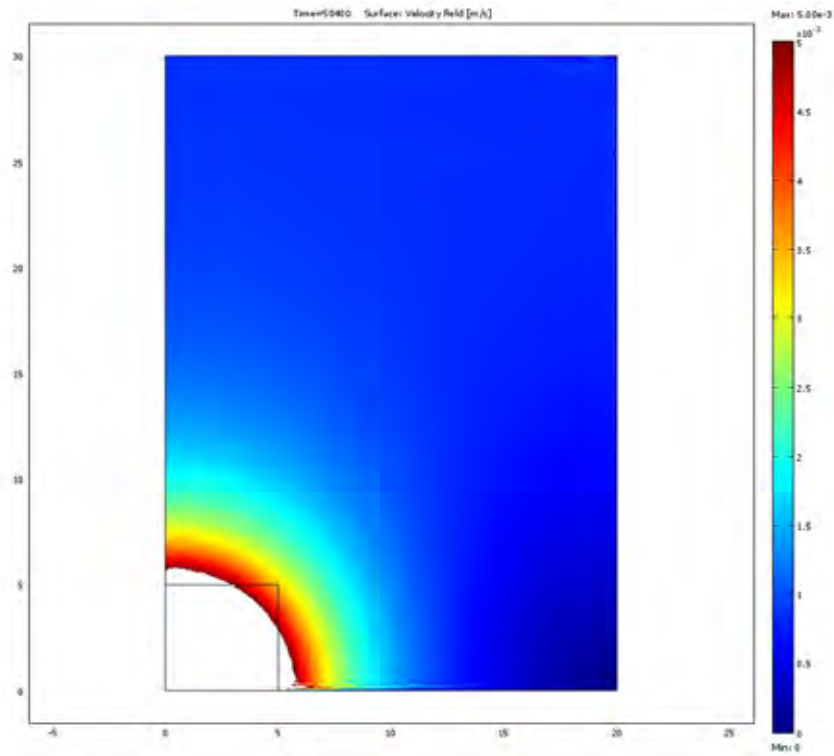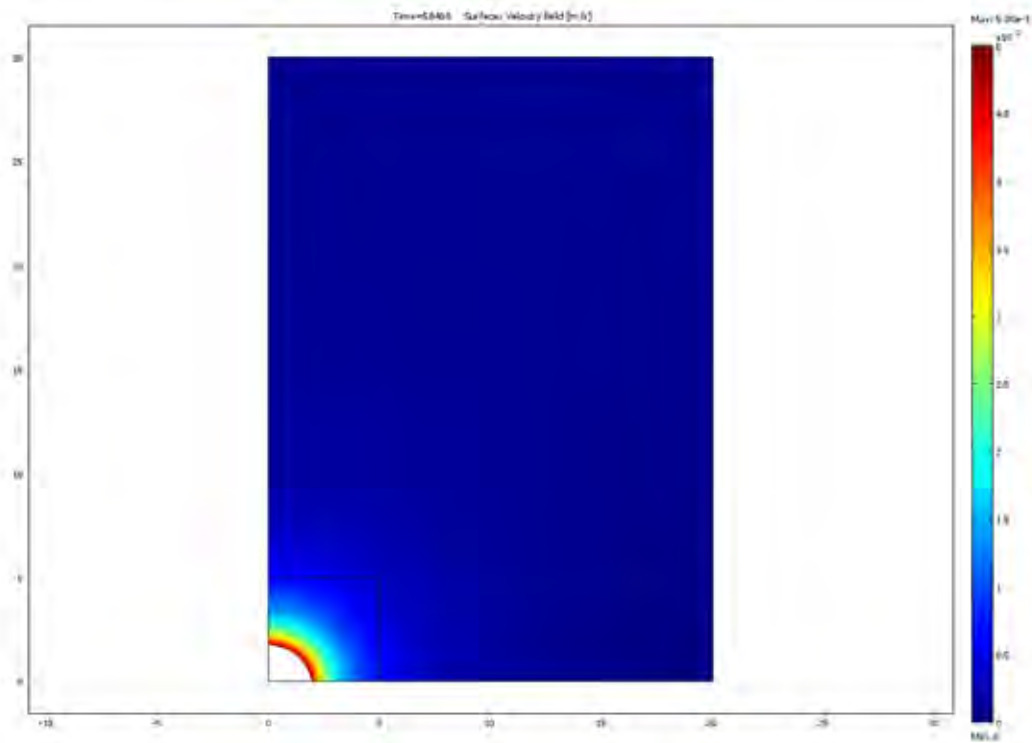

Figure A6-3   Simulation 2: velocity field.
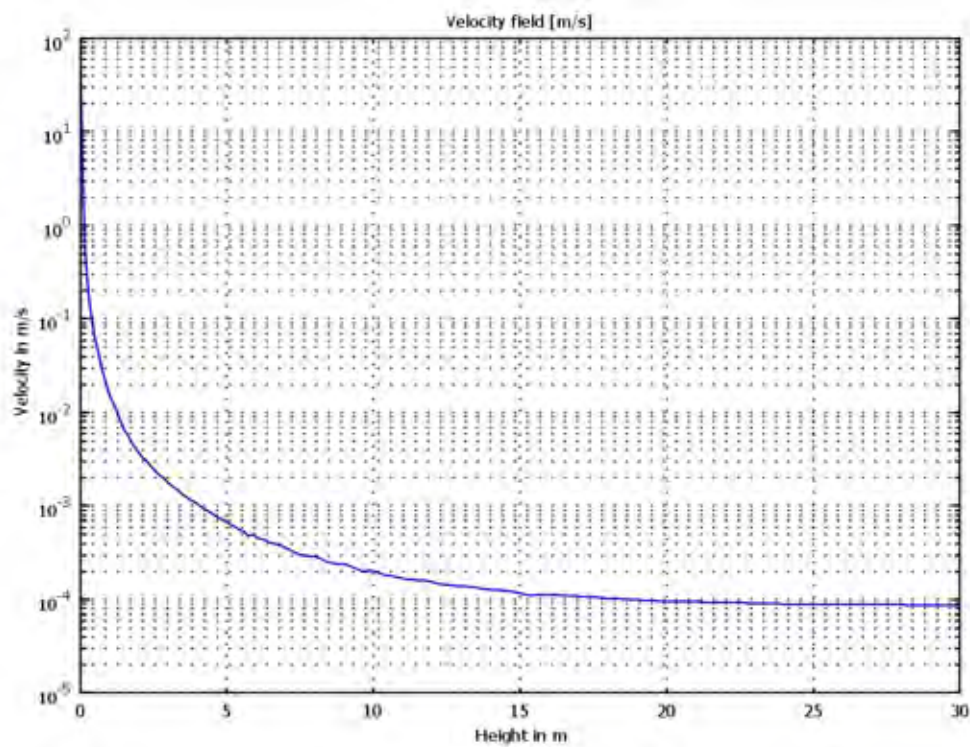

Figure A6-4   Simulation 2: cross section velocity plot against height in the centre.

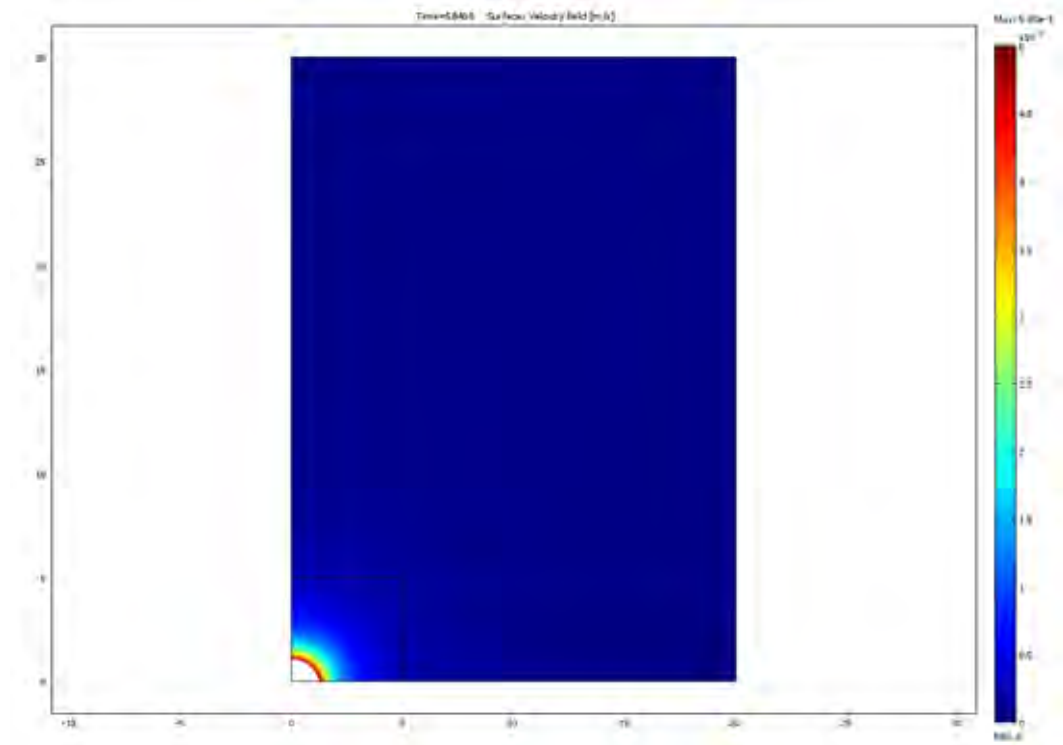**Simulation 3 - 1005 m3/h, 3 m/s, diam. 344 mm (five inlet situation)**

Figure A6-5   Simulation 3: velocity field.

Figure A6-6   Simulation 3: cross section velocity plot against height in the centre.

Simulation 4 - 1005 m3/h, 30 m/s, diam. 109 mm (five inlet situation)
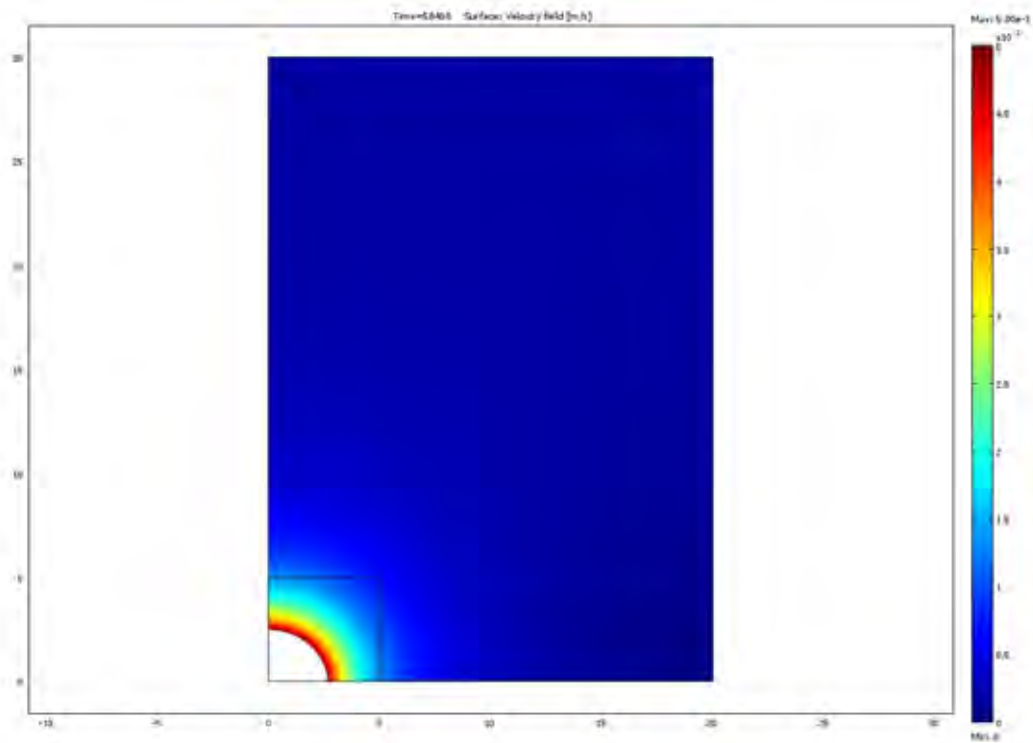
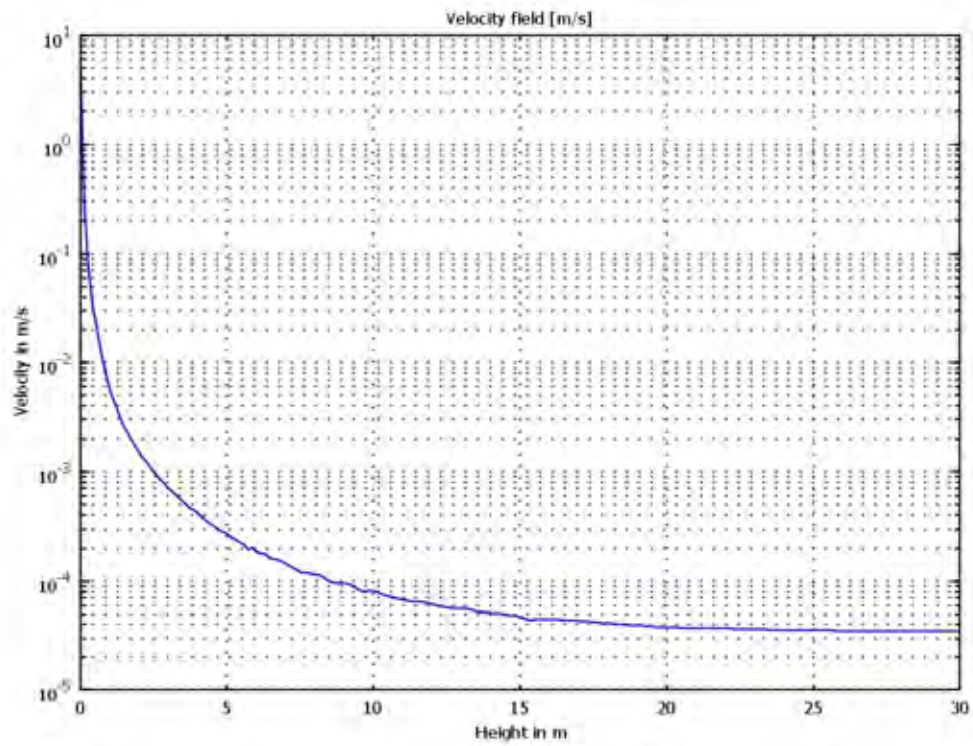**Figure A6-7**   Simulation 4: velocity field.
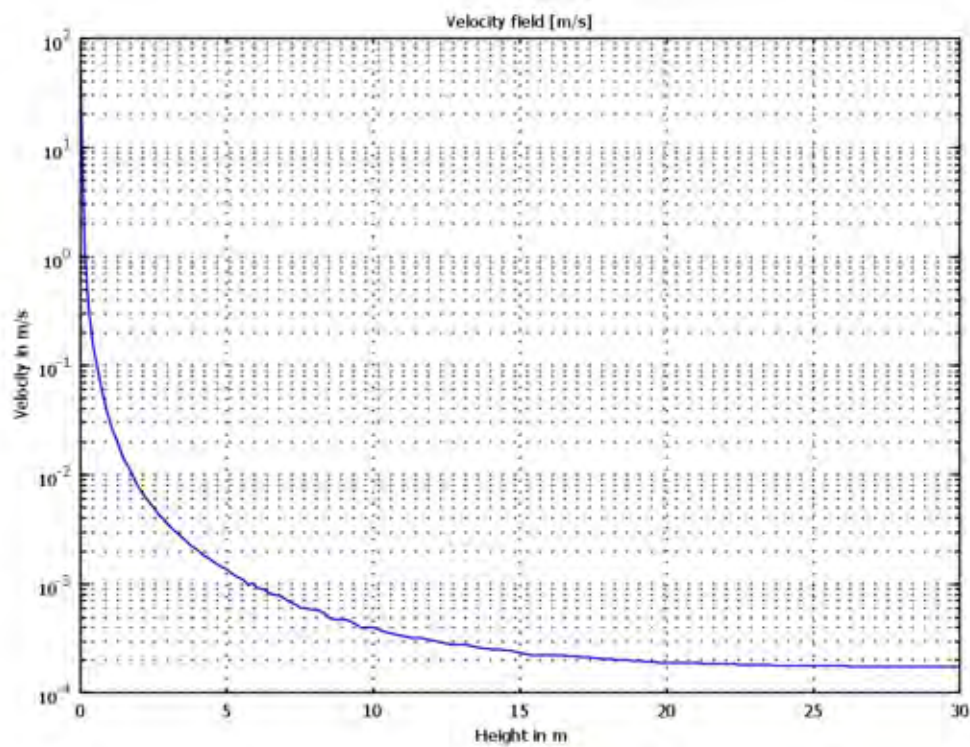
Figure A6-8   Simulation 4: cross section velocity plot against height in the centre.

Simulation 5 - 502 m3/h, 3 m/s, diam. 243 mm (10 inlet situation)

Figure A6-9   Simulation 5: velocity field.

Figure A6-10   Simulation 5: cross section velocity plot against height in the centre.

Simulation 6 - 502 m3/h, 30 m/s, diam. 77 mm (10 inlet situation)

Figure A6-11 Simulation 6: velocity field.

Figure A6-12 Simulation 6: cross section velocity plot against height in the centre.

63

Simulation 7 - 201 m3/h, 3 m/s, diam. 154 mm (25 inlet situation)

Figure A6-13   Simulation 7: velocity field.

Figure A6-14   Simulation 7: cross section velocity plot against height in the centre.

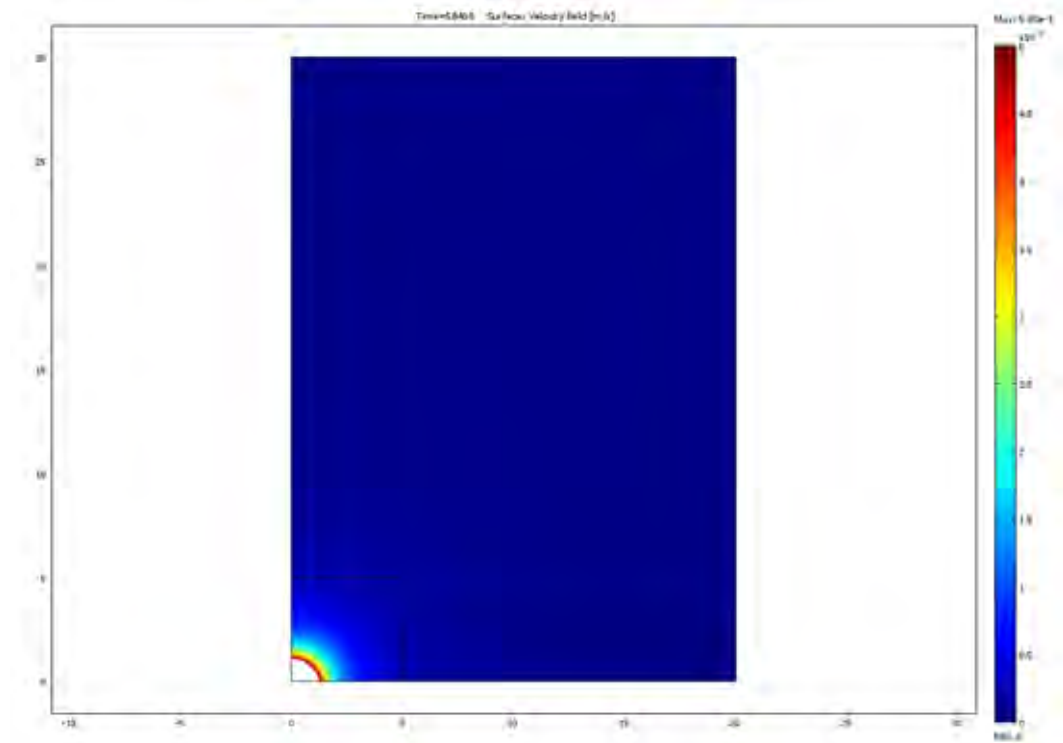**Simulation 8 - 201 m3/h, 30 m/s, diam. 49 mm (25 inlet situation)**
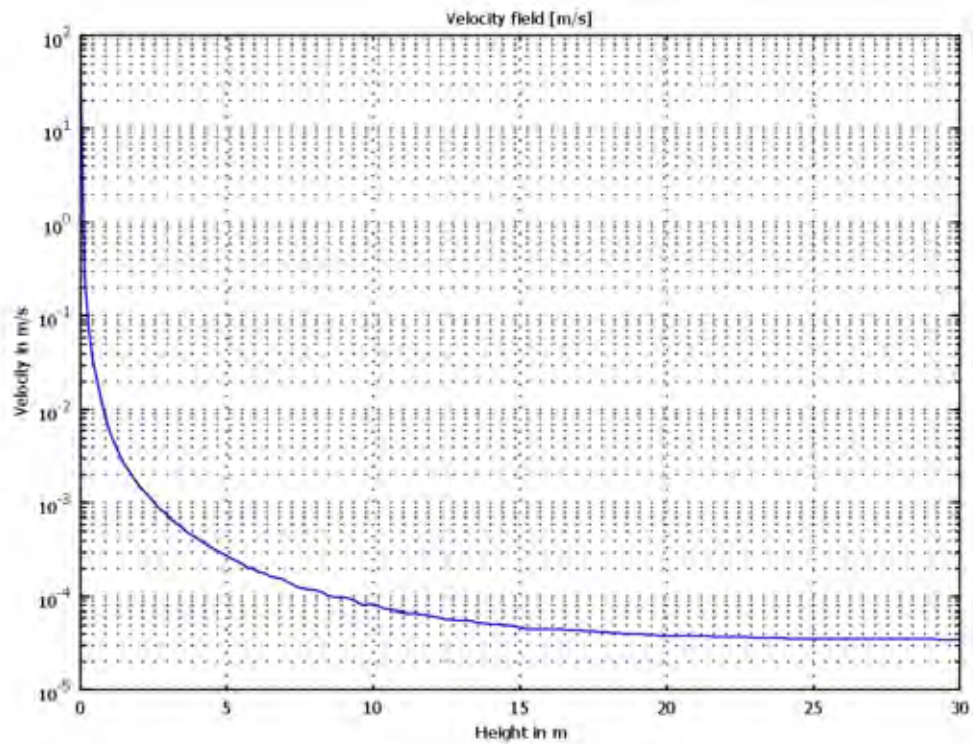

Figure A6-15   Simulation 8: velocity field.

Figure A6-16   Simulation 8: cross section velocity plot against height in the centre.

Appendix 7: Simulation of a porous media with COMSOL – A practical guide

COMSOL multiphysics is a very flexible finite element software. It includes many different equations that can be coupled together in a relatively easy way. Here we are going to discuss more in detail how to set a porous media with the available equations. In other words, how to modify the existing equations to solve our problem.

1   The problem

We would like to model a flow of nitrogen through a porous media and follow the evolution of the Nitrogen concentration with time. The porous media is stored in a silo of four meters in diameter and five meters height with an injection pipe at the bottom of 20 cm diameter. The generating axes of the two cylinders are in conformation. Let us set an injection velocity of 3m/s. The temperature in the silo is of 300K and the nitrogen injected is at 273K.

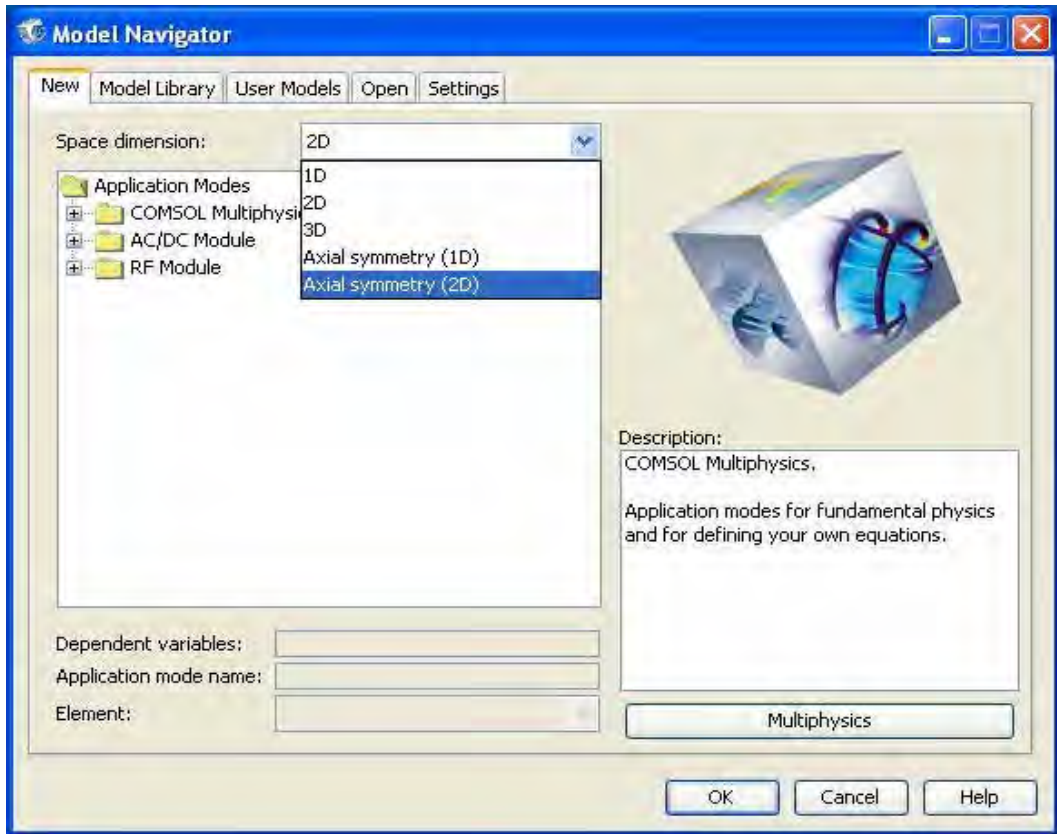# 2   The COMSOL multiphysics interface

COMSOL provide a GUI (Graphical User Interface) which is very intuitive and easy to understand. To make a simulation you proceed as with all software of the same type. First you work with the pre-processor to define your geometry, the physic of your problem and the mesh that permit to discretizise the domain of calculations. Secondly you use the solver that solves the different PDEs (Partial Differential Equations) you have chosen. Thirdly you use the post processing to monitor your results.

2.1   Which equation to choose?

When you launch COMSOL you arrive on the following interface:
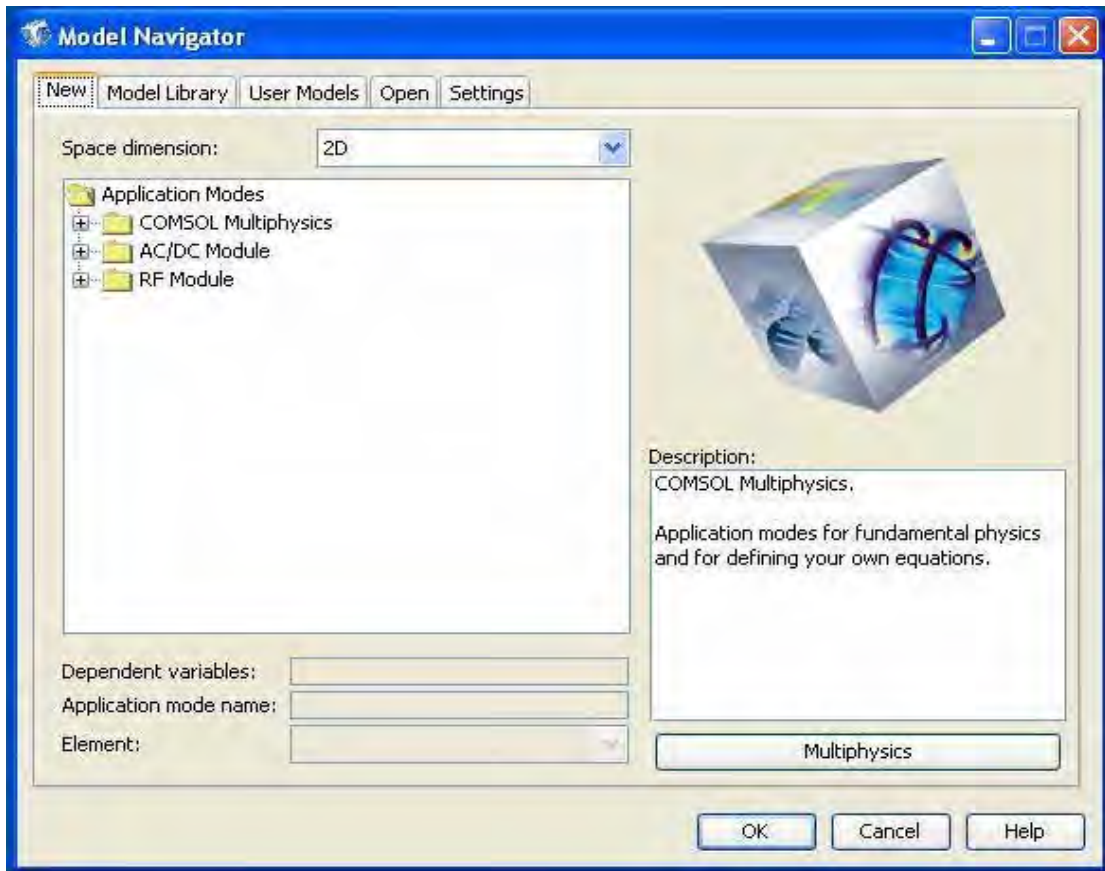
There are different menus on the top (new, model library …) if you want a new model, go to new, and COMSOL provides a series of examples that you can find in the model library. You can also save you own examples in the user models. In our case we want to do a new model so go to new. Here you have first to set the space dimension. In our case we would work with an 2D axisymmetric problem because the geometry is axisymmetric.

Once this is done you have to choose one of the three available equation libraries. The "COMSOL Multiphysics" library gather the general equations mostly use, this is what we need so let tick it.
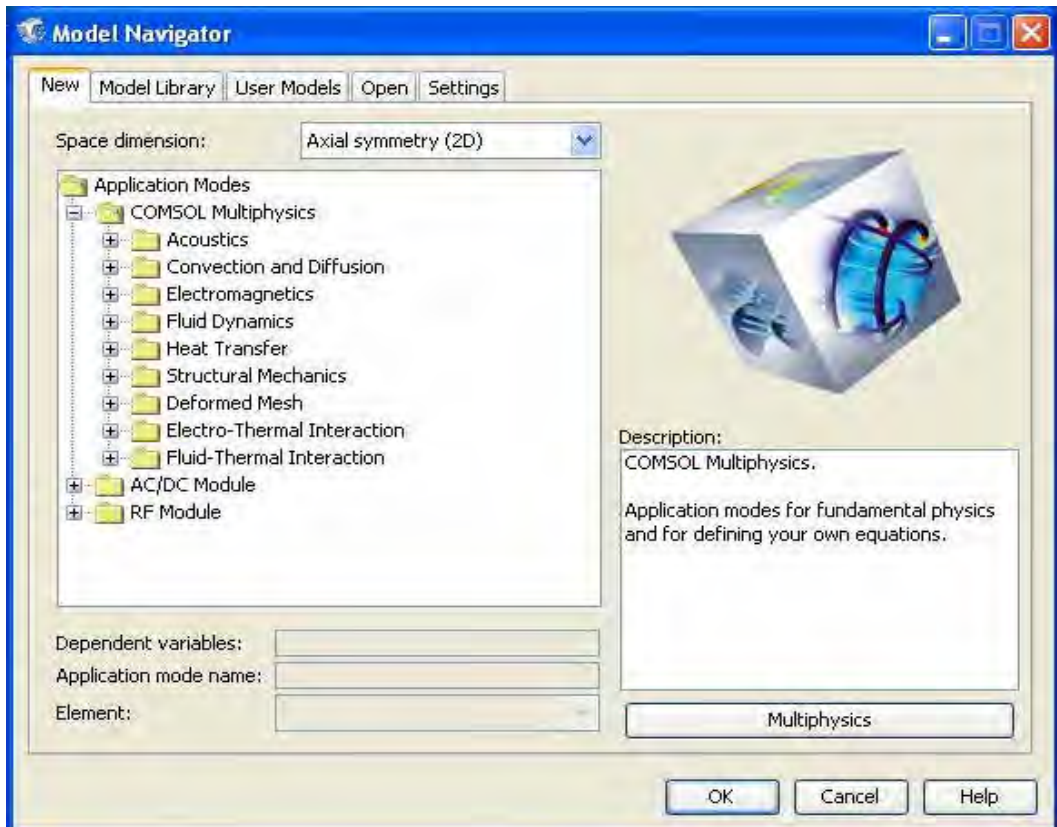
Here you have the choice between several equation. We would need a fluid dynamic description for the flow, a convection and diffusion description for the solute transport and a heat transfer equation to describe the temperature differences.

But let begin with only one equation to model the flow through the porous media. Click on the "Fluid Dynamics" folder.
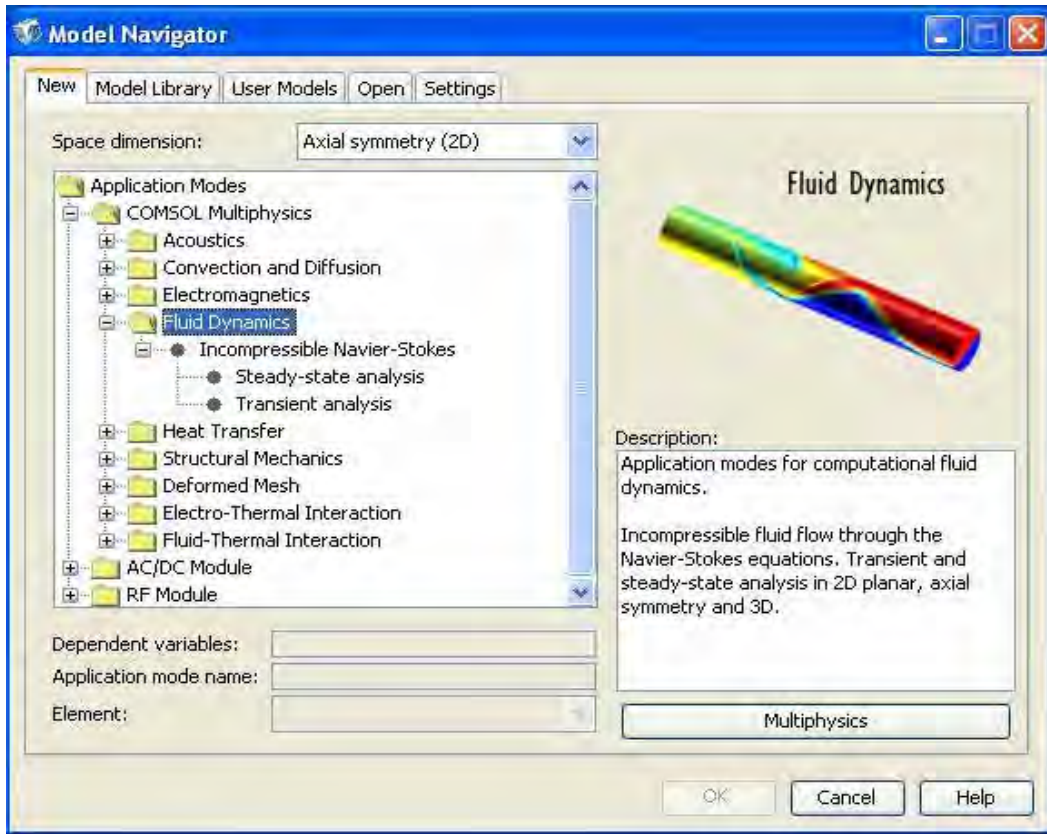
Now let tick the "Incompressible Navier Stokes", then you have the choice between a steady or a transient state analysis, as we are going check an evolution with time we need to solve a transient problem. So double click on "Transient analysis".

Now you can see the GUI. On the top of the main window all the menus correspond step by step to the definition of the model.

2.2   How to define a geometry?
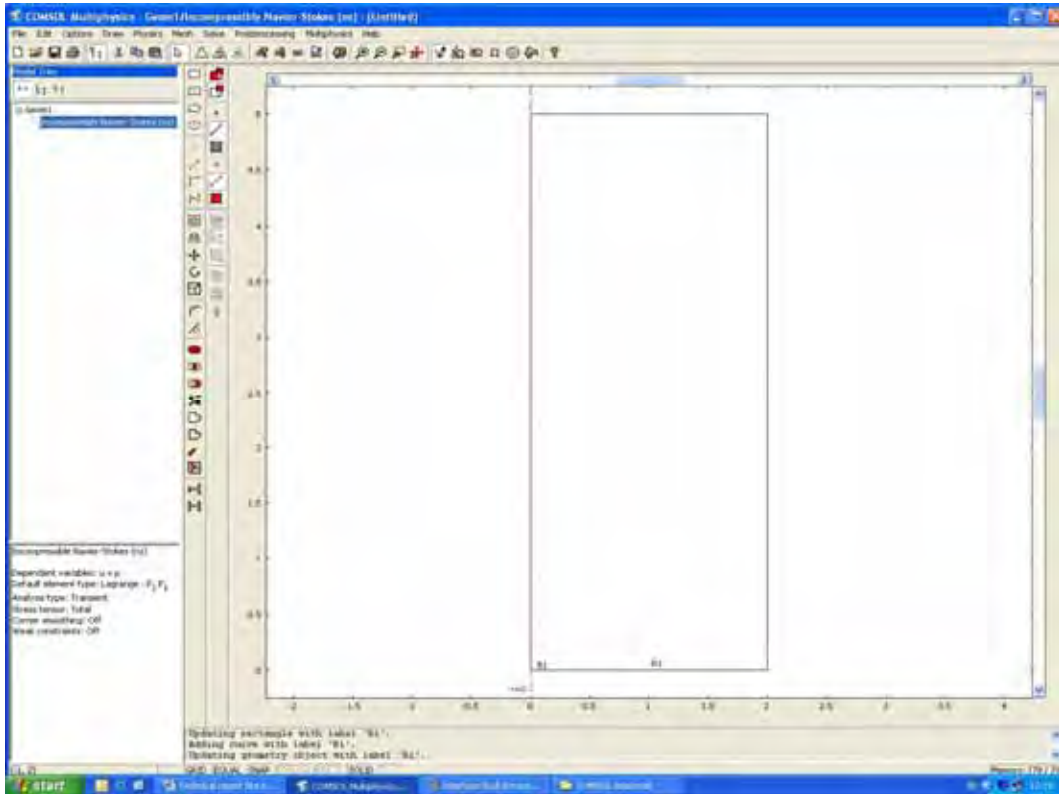
First you have to define your geometry. As you can see you have a doted line in the centre of the screen. This is the symmetry axis. We are going to draw our geometry by doing a square and a curve.

(1) Click on the square.

(2) Draw a square in the drawing area. It is called R1 and this is now the silo.

(3) Now we want to specify the height and the wide of the square. So select the square and click on the "Draw" menu and select "Object Properties" There enter the desired dimensions as follows:
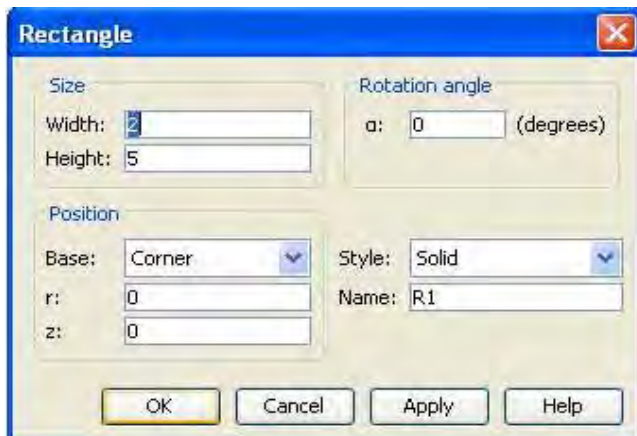
(3)

(4) Click on the segment element and draw a segment. Fix the dimensions you want by doing just as you have done for the square.
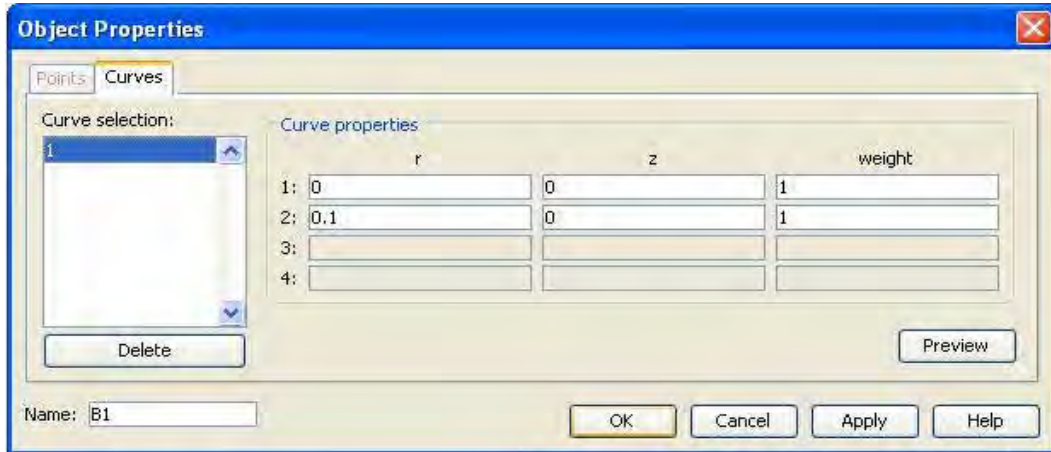
Now the geometry is ready.

2.3   How to set the physic of the problem?

Concerning the theory of the problem please refer to the report. Now we want to use the equations described in the theory. Let begin by the momentum equation (Navier stokes).

First of all we have to define all the coefficients we are going to use. To do that go to the "Option" menu and click on "Constants…", then enter all the data you need as follows:
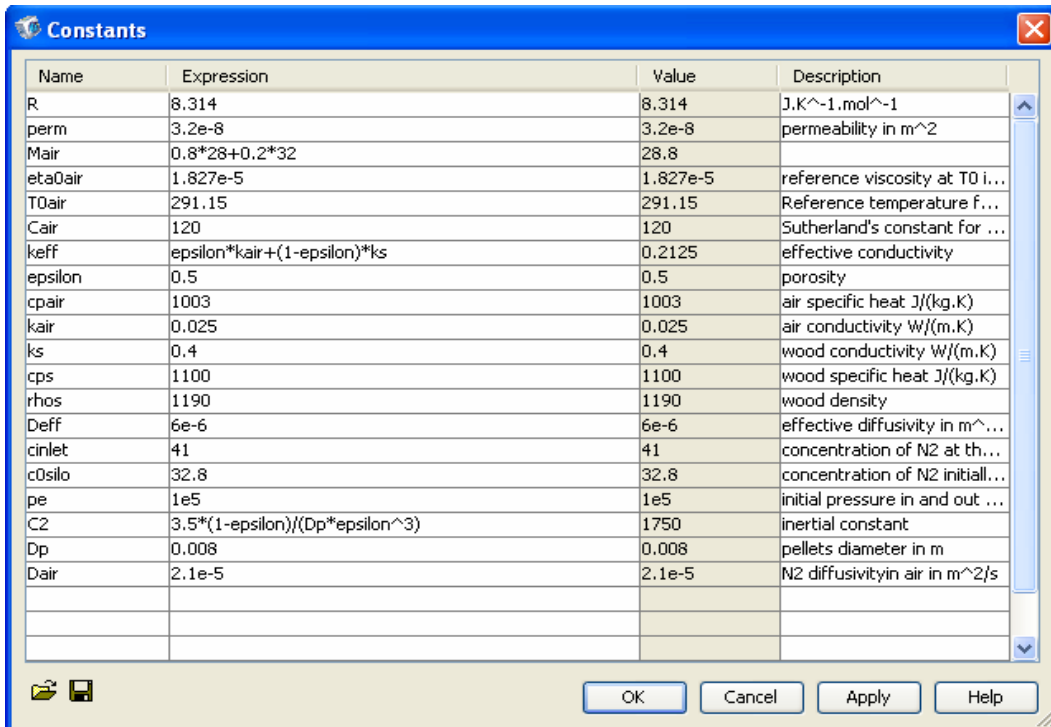

Now click again on "Option" and click on "Expression" and again on "Global Expression", here you would define expressions that are going to be valid and usable in all the domain, you could have done it also in only one subdomain. Then enter expressions as follows:
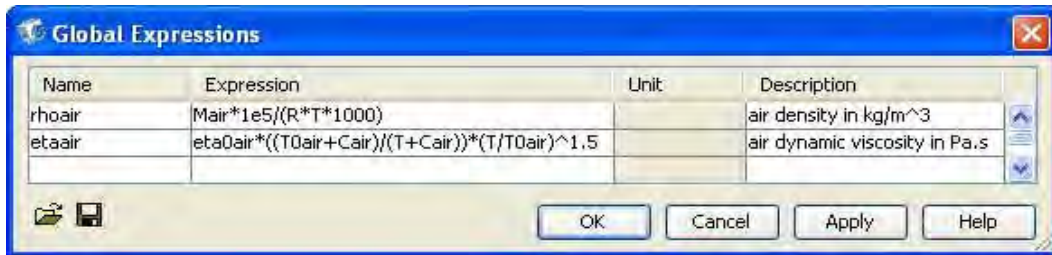
On the left side of the main window you have a model tree. Right click on the desired equation. Then you have the choice of setting the subdomain or the boundary. Let set the subdomain by left clicking on it.
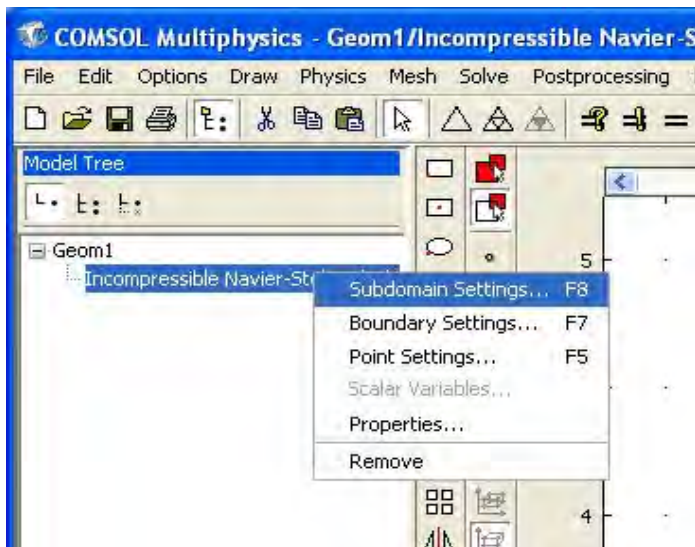
Now you have access to the subdomain window.
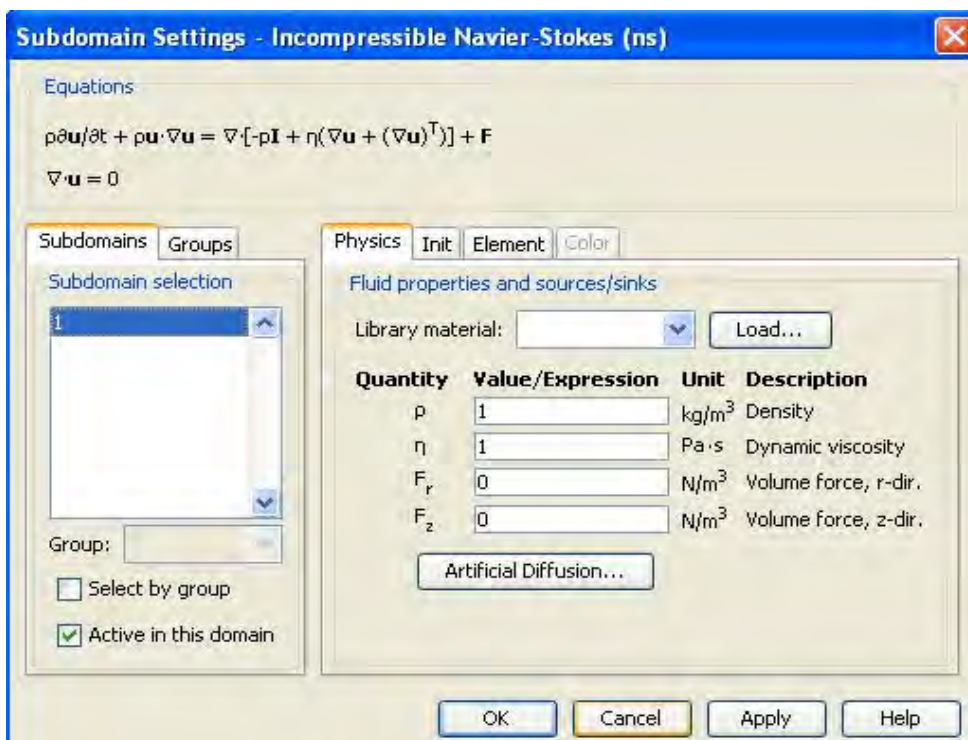
Now we are going to use the constants we have entered before.

(1) In density type "rhoair"
(2) In dynamic viscosity type "etaair"

In Fr and Fz we are going to add the terms due to the porous media.

(3) In Fr type -etaair/perm*u-0.5*C2*rhoair*u^2
(4) In Fz type -etaair/perm*v-0.5*C2*rhoair*v^2-9.81*rhoair

Now you have to define the initial conditions in the "Init." Menu. Just enter "pe" in "p(t0)"

The subdomain is now defined. Let us define Boundary conditions by using the model tree as mentioned above.¨
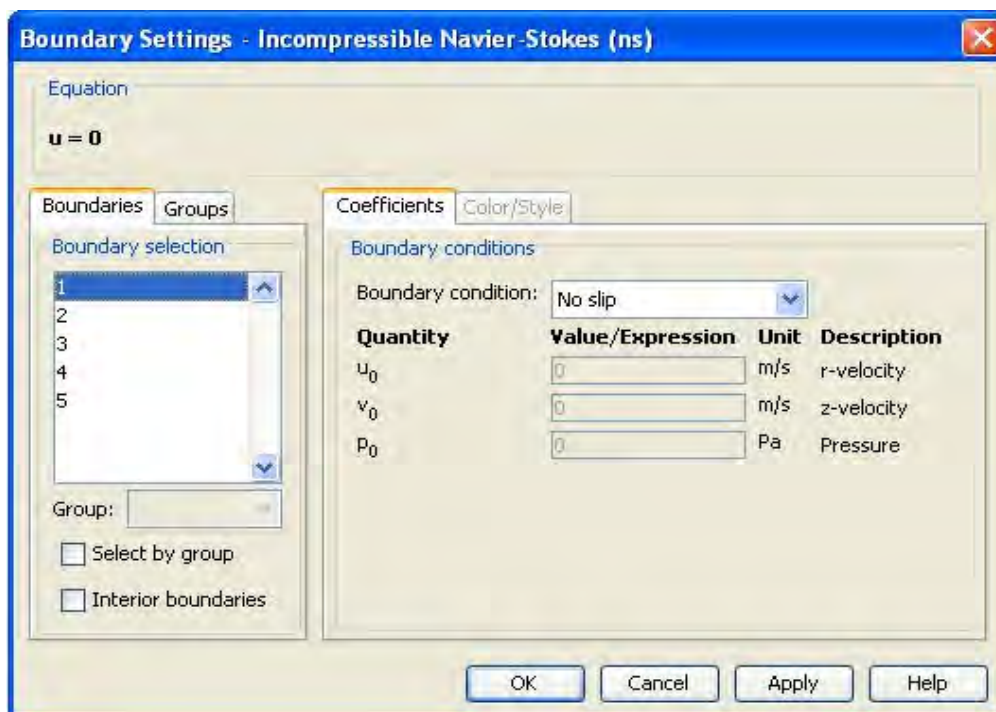
Set boundaries as below:

1: "Axial symmetry"
2: "Inflow outflow" u0=0 and v0=3*1.5*(1-s^2)*(1-exp (-t/0.1))
3: "Normal flow/pressure" p0=pe
4;5: "No slip"

Now all the physic of the problem is ready. Just add in the constants T=300; it would be used to calculate the density. T is in Kelvin.

2.4   How to set a grid system

In the main window click on the "Mesh" menu. Here you can see that you have the choice between "Free Mesh Parameters" and "Mapped Mesh Parameters". The free mesh would discretizise the geometry with triangles, it is a good way to start, but difficult to set

precisely a grid especially if you want a finer grid in certain areas like in our case close to the inlet. In that case it is sometimes better to choose the mapped mesh (squares elements) but they are also less efficient concerning the using of the computer memory. Each Time you would have to do compromises.

Let try the free mesh. In the Mesh menu click on "Free Mesh Parameters" There go to the subdomain menu. And set the "Maximum Element Size" to 1.5 and the "Element Growth Rate" to 1.05 and click remesh and Ok. Now you can see in the subdomain the grid system. As desired it is finer close to the inlet.

2.5 A word about the solver

In the menu "Solve" Click on "Solver parameters" Here you set up the solver. Most of the time you just would have to change the time stepping in the general menu. It is set per default to "0:0.1:1" It means that the data recording starts at 0 seconds (the simulation starts always at 0 seconds) then every 0.1 seconds the data are saved and stored but the calculation time step can be lower than 0.1 seconds. "1" means the end of simulation is at time 1 second and this is also the last time that data are saved. The solver chooses automatically the time step of calculations and changes it in function of the degree of variations. It is preferable to keep it automatic.

In our case we can keep the default values as the transient state of the velocity has duration of 0.5 seconds.

Now in the "solve" menu just click "Solve Problem".

## 2.6 How to check and manipulate the results

One of the most important parts of the work with simulations is to check the results. Sometimes it can be good to follow in real time what is calculated. For that click on the "Post Processing" Menu in the main window then on "Probe Plot Parameters."
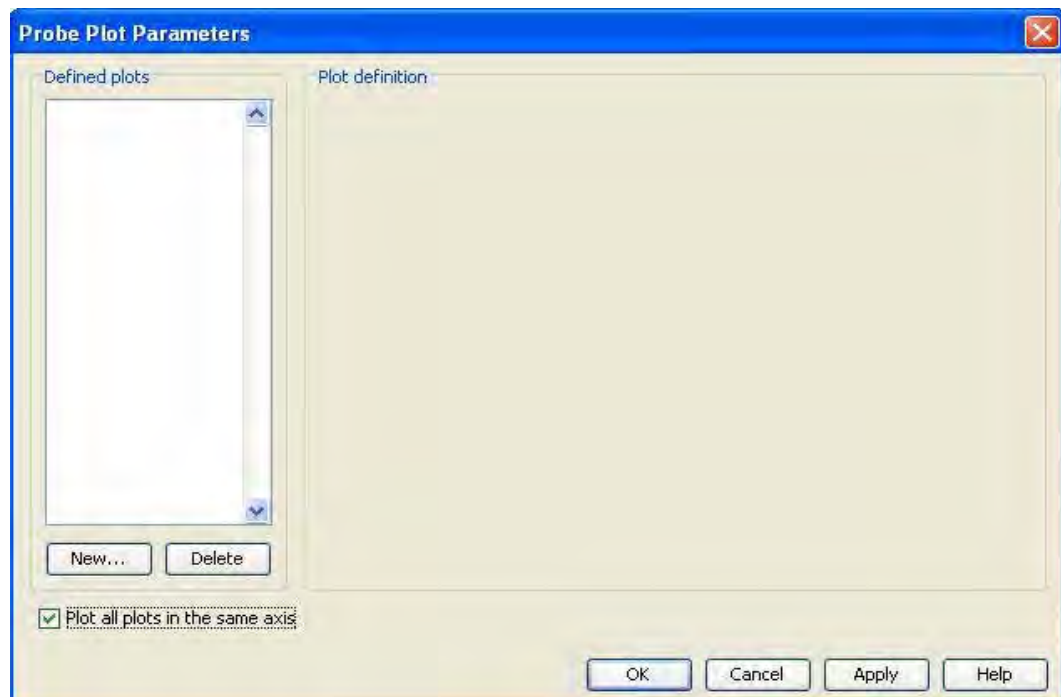
Here you define a function of time (e.g. temperature) that would be computed during the solving. Click new.
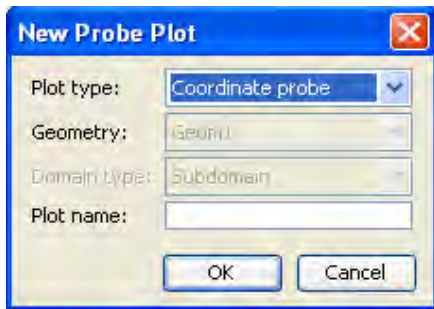
Now choose "Coordinate Probe" in the Plot Type and enter a name. Then select the quantity that you want to compute and enter the coordinates of the point you want to check. You can add several plots like that. Finish by clicking OK. Let check the result by solving again and checking the velocity for r=0m and z=0.5m.
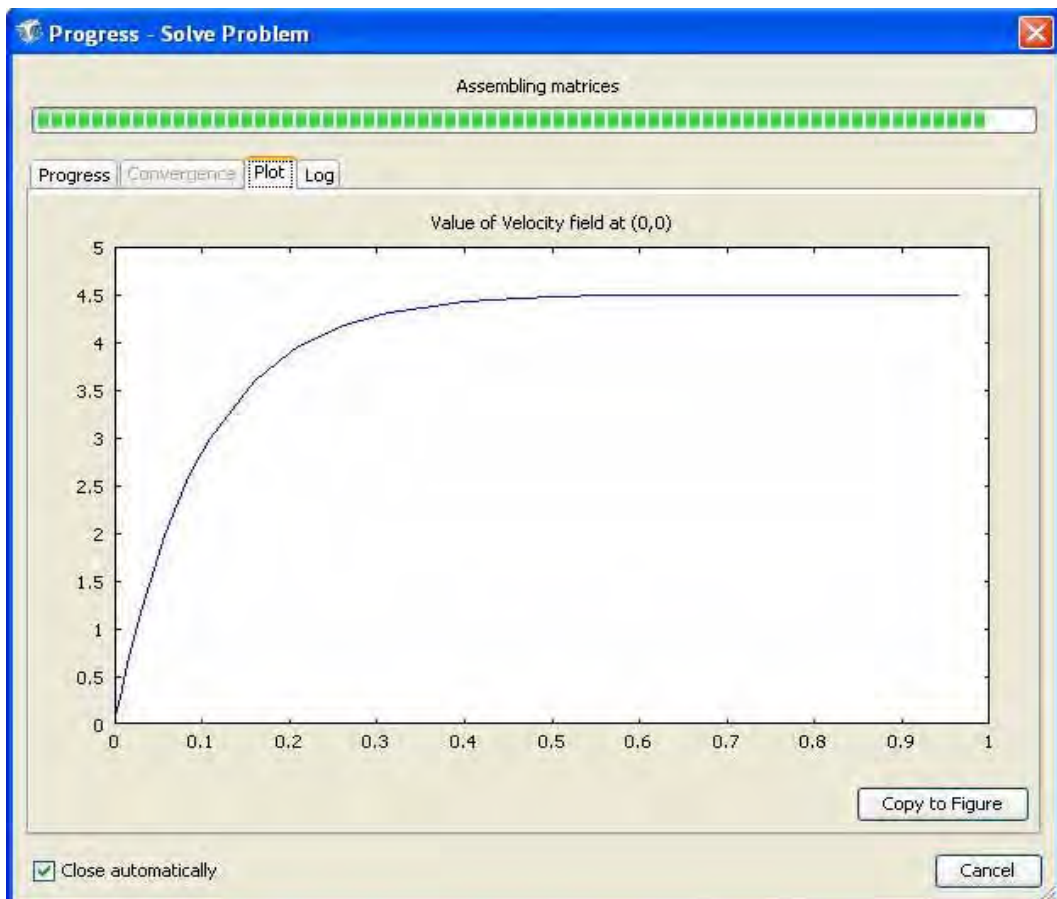
More about the results. You can check it by different plots; you could find all type of plots in the main window, "Post Processing" menu and "Plot Parameters".

If you want more quantitative results, still in the "Post Processing " choose "Cross Section Plot Parameters" Here you can set the quantity you want against time in different points that you define by their coordinates. You can also set a line and plot a quantity against the arc length of the line.

2.7 How to add equations

Now we have set the flow, we still need an equation for the spreading of the nitrogen (Transport equation) and an equation to compute the temperature difference (Energy equation).

In the main window click on the multiphysics menu and select "Model navigator". You are again in the equation library. As before select in the "COMSOL Multiphysics" folder, the "Heat Transfer" folder, then choose "convection and conduction" and select the transient case, then click on the add button.

Let do the same for the "Convection and Diffusion" equation. Once you have finished go back to the main window by clicking OK.

Now we are again going to do what we have done previously in section 2.3.

Energy equation:

First of all we have to remove "T" from "constants" in fact it becomes useless because we have an energy equation. So go to the option menu, then click on "Constants" and erase T, then click apply and leave.

Right click on the "convection and conduction" equation in the model tree. Let set the subdomain by left clicking on it.

Now you have access to the "convection and conduction" subdomain window. We assume all the coefficients constant here so we can bring them out from the derivatives.
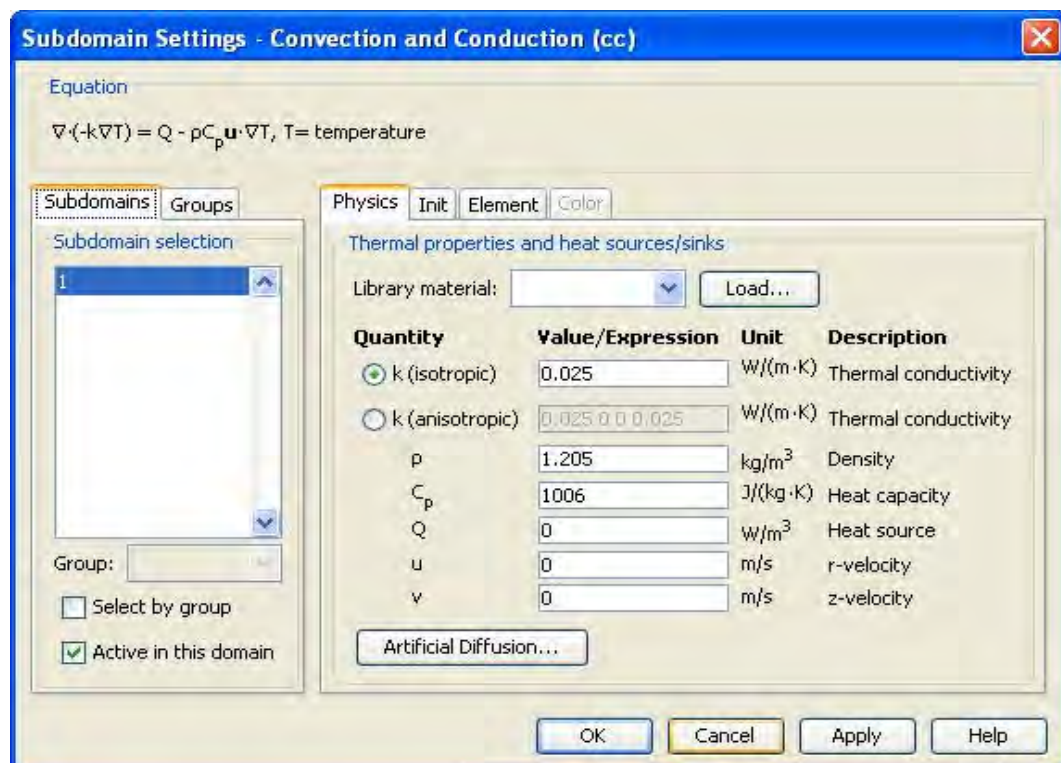
(1) In Thermal conductivity type "keff"
(2) In density type "rhoair"

(3) In Heat capacity type epsilon*cpair

(4) In Heat source type -(1-epsilon)*rhos*cps*diff(T,t)

(5) In r-velocity type u

(6) In z-velocity type v

Now you have to define the initial conditions in the "Init." menu. Just enter "300" in "T(t0)".

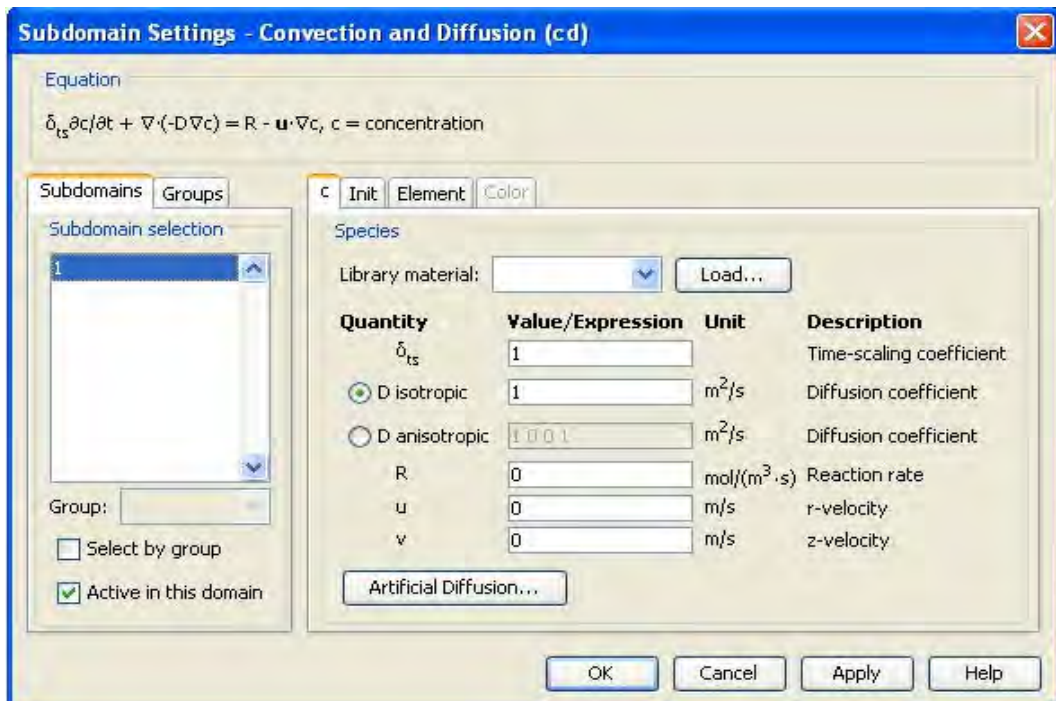The subdomain is define let define Boundary conditions by using the model tree as mentioned above.

Set boundaries as below:

1: "Axial symmetry"

2: "Temperature" T0=273K

3: "Convective flux"

4;5: "Thermal Insulation"

Transport Equation:

Right click on the "Convection and Diffusion" equation in the model tree. Let set the subdomain by left clicking on it.

Now you have access to the "Convection and Diffusion" subdomain window. We assume all the coefficients constant here so we can bring them out from the derivatives.

(1) In Time scaling coefficient type "epsilon"

(2) In Diffusion coefficient type "Deff"

(3) In Reaction Rate type 0

(4) In r-velocity type u

(5) In z-velocity type v

Now you have to define the initial conditions in the "Init." menu. Just enter "32.8" in "c(t0)".

The subdomain is define let define Boundary conditions by using the model tree as mentioned above.

Set boundaries as below:

1: "Axial symmetry"
2: "Concentration" c0=32.8K
3: "Convective flux"
4;5: "Insulation/Symmetry"

Now the whole system is set. You can now try to run the simulation. You can also modify parameters in the solver to make the simulation longer that 1 second (see section 2.5).

2.8   Instability problems

Sometimes with high velocities at the inlet you can have instabilities problem. Then you should have this error message.
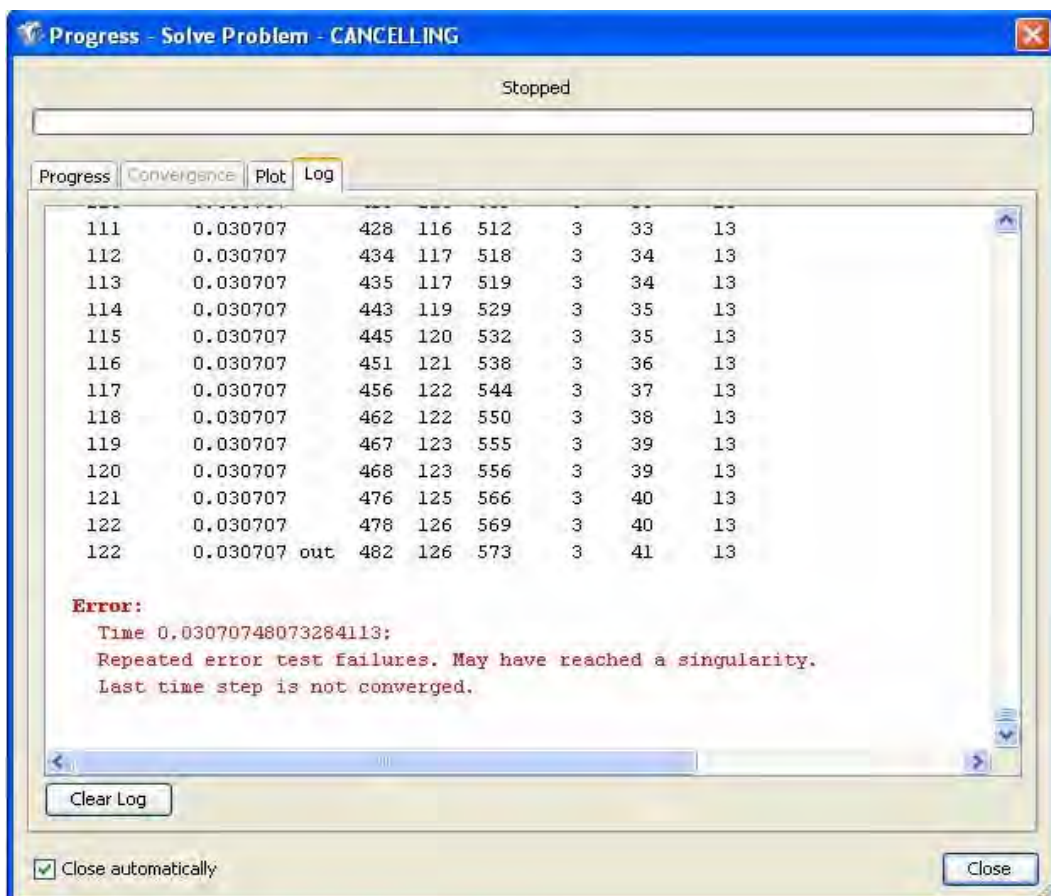
Then you have different parameters that can help you. If you run a large scale simulation and you cannot increase the number of elements by refining the mesh you have to improve the approximation function between each node by setting a higher degree

function as following. In the model tree right click on the desired equation, then click on " Subdomain". Then choose the Elements menu.

The default element is "Lagrange P2P1" for the "Navier Stokes" equation, set a higher degree like "Lagrange P4P3" for example. For the other equations set it to "quintic".

If it still does not work you can try to refine you mesh by adding element or by changing the elements type.

Finally if you still have problems you can try to use a stabilization technique. Then please refer to the COMSOL manual.

3   Conclusions on COMSOL

COMSOL provide many different PDEs that permit to model many problems. Even if the equation you need is not available, with a little bit of literature you can find an equation and set it by yourself.

SP Technical Research Institute of Sweden develops and transfers technology for improving competitiveness and quality in industry, and for safety, conservation of resources and good environment in society as a whole. With Sweden's widest and most sophisticated range of equipment and expertise for technical investigation, measurement, testing and certification, we perform research and development in close liaison with universities, institutes of technology and international partners.

SP is a EU-notified body and accredited test laboratory. Our headquarters are in Borås, in the west part of Sweden.
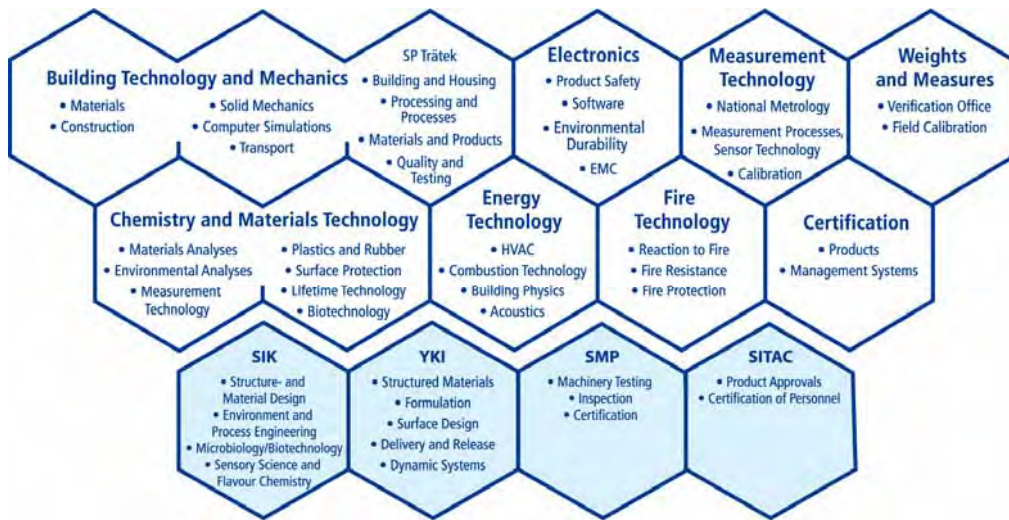
SP is organised into eight technology units and four subsidiaries