Real world study with

# Code Saturne

Claus Andersen

# Introduction

For a long time I've wanted to try and use CFD for a practical purpose and compare it to real-world results, and now during my internship, I have the chance to try it out. For all intent and purposes, I'm far from an expert on CFD nor fluid dynamics, but I have done a few experiments and have a good general idea of how things come together in fluid dynamics/CFD. What this sums up to, is 'nothing is written in stone here' – There may be errors and the setup is probably far from optimal, there is always room for improvement.

Here is a hands on case with Code Saturne without getting too technical.

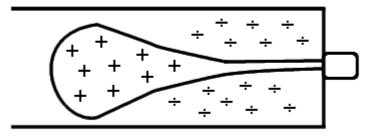
/Claus Andersen, April 20th '09

# Goal

To create a computer model that corresponds to real world – with the goal of further development of burner tube. This is an initial simulation to make sure I can make Code\_Saturne emulate real world conditions. The goal is however, not to be 100% physically correct, nor does time permit me to set up the experiment to a laboratory grade.

# Scenario

In a gas fire a burner tube is used to mix the fuel (in this case natural gas) with oxygen using the Venturi effect. The pressurized fuel is sent from the high pressure bottle, through a regulator and from the regulator through a nozzle into the burner tube. The gas jet from the nozzle creates a low pressure zone around the base of the jet which sucks in air through the aeration port and mixes it in the burner tube before it is sent into the burner1.



# Scenario

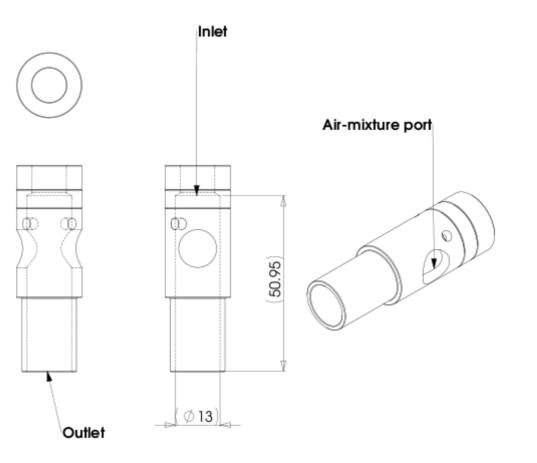
Among other things, the geometry of the nozzle and the velocity of the fluid determines the shape of the jet and the magnitude of the low pressure zone. In this case I've guestimated a discharge coefficient of 0.5, based on the geometry of the nozzle and calculated the exit velocity accordingly.

The exit orifice has a diameter of ~1.9mm



In this study I wont simulate what happens inside the nozzle, nor will I concern myself too much whether the shape of the gas jet is physically correct at the exit – That will be discussed a bit more later on.

# Scenario



# Mathematical approach

#### **Collecting data**

For my simulation I need to input some data into the model, this was done in the following manner:

- Measure volumetric flow rate of the gas going into the burner tube using a flowmeter on the actual gas fire and compare it to the technical documents specified by the test lab
- Guestimate a discharge coefficient for the nozzle using empirical numbers from a table
- Acquiring physical properties for natural gas (using pure methane)
- Measure pressure differential on the regulator
- Calculate exit velocity of gas jet using above values

#### **Result**

Calculating the exit velocity at the nozzle has been done in two ways – using the flow rate and using a theoretical approach. Both results equals  $\sim$ 38m/s but since I know that that is a bit high considering the setup and the values used, I back it down a bit to  $\sim$ 33m/s for the simulation.

See Media: <u>Mathcad-dyse.xmcd.pdf</u> MathCAD PDF for calculation.

# Simulation approach

#### • Software

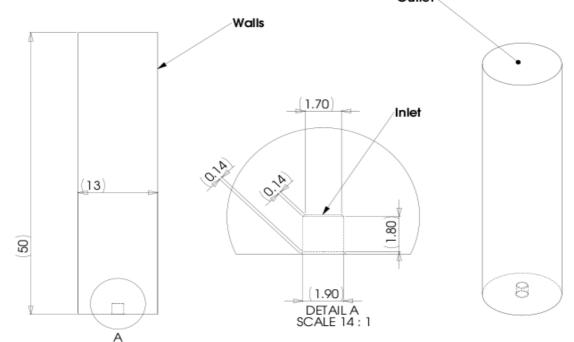
- SolidWorks for modeling the flow domain
- Salomé 4.1.4 for preparing geometry and meshing flow domain
- Code Saturne 1.4b for solving the simulation
- Paraview 3.4.0 for Post processing

Everything besides SolidWorks is run on Ubuntu 8.10 Linux – SolidWorks could easily be replaced by Salomé but laziness dictated I did it in SW at work.

The computer used is my trusty Zepto Znote 6014W, 1.7ghz Celeron, 1GB RAM

# Model description and preparation

The simplified flow domain consists of a cylinder (Ø13mm, h 50mm) with a simplified inlet (remember, I don't care too much about what happens near and around the inlet).



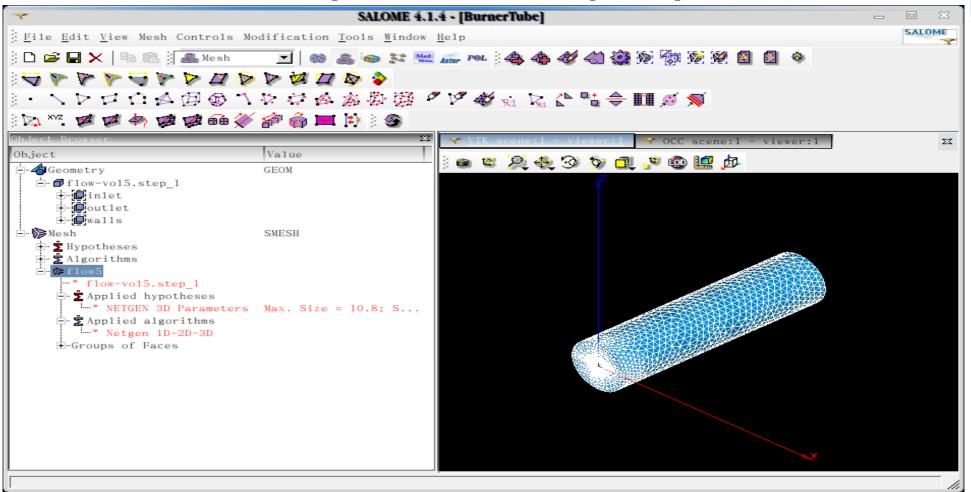
This is exported from SolidWorks in STEP file format and assigned groups in Salomé geometry module, before it is meshed in the meshing module using the following parameters

# Model description and preparation

NetGen 1D-2D-3D using the setting 'very fine' and 'quadratic' (2nd order) – the NetGen algorithm and geometry makes sure that the mesh is refined around the inlet and coarsens further up the flow domain. This yields the following mesh:

| ~              | Mesh cor | nputation succee | d              | S |
|----------------|----------|------------------|----------------|---|
| Compute mesh - |          |                  |                |   |
| e 💱            |          |                  |                |   |
| Name           |          |                  |                | _ |
| flow5          |          |                  |                |   |
| Mesh Infos —   |          |                  |                | _ |
|                | Total    | Linear           | Quadratic      |   |
| Nodes :        | 155940   |                  |                |   |
| Edges :        | 405      | 0                | 405            |   |
| Faces :        | 5692     | 0                | 5692           |   |
| Triangles :    | 5692     | 0                | 5692           |   |
| Quadrangles :  | 0        | 0                | 0              |   |
| Polygons :     | 0        |                  |                |   |
| Volumes :      | 111893   | 0                | 111893         |   |
| Tetrahedrons : | 111893   | 0                | 111893         |   |
| Hexahedrons :  | 0        | 0                | 0              |   |
| Pyramids :     | 0        | 0                | 0              |   |
| Prisms :       | 0        | 0                | 0              |   |
| Polyhedrons :  | 0        |                  |                |   |
|                |          |                  | C <u>l</u> ose |   |

## Model description and preparation



Again, without getting too technical, a way to make sure your mesh is optimized for the simulation, you can calculate the Courant number and adjust your mesh accordingly. ((u\*deltat)/deltax)<C

# Preparing and running simulation

The mesh is exported in MED file format to the folder ~/Study/MESH/ and the Saturne GUI is started from ~/Study/CASE1/DATA/ using ./SaturneGUI

Once the GUI has loaded, it is time to enter the values (the options/tabs not mentioned is left at default if nothing is stated).

Under Solution domain load the mesh file exported from Salomé :

|  | Code_Saturne user interface   | - O X        |
|--|---|--------------|
|  | <u>F</u> ile <u>T</u> ools Options                                  | <u>H</u> elp |
|  | 🗹 🐜 🔚 39 🛠 🗗 😫 🕺  |              |
| A Street   | Study name: BURNER  |              |
| MO-  | Case name: CASE1<br>XML file name: burnertube.xml                   |              |
|  |   |              |
| Calculation environm   | ent MESHES PERIODIC SYRTHES STAND-ALONE BOUNDARIES COUPLING RUNNING |              |
| <b>Solution Domain</b>   |   |              |
| Thermophysical mode  |   |              |
| Physical properties Additional scalars   | flow5.med   |              |
| 🖻 📂 Boundary conditions  |   |              |
| Definition of bounda   |   | <b>&amp;</b> |
| Calculation control  | nditions  |              |
| <ul> <li>Calculation control</li> <li>Calculation control</li> <li>Numerical parameters</li> </ul> | 5   |              |
| Calculation managem  |   |              |
| 🗋 Start/Restart  | Wern faces cutting: 🔶 on 🔺 off                                      |              |
| Length Prepare batch calcul  | ation   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |
|  |   |              |

Next go into Thermophysical models and set the turbulence model to k-epsilon – This is a high Renolds number model since that's what I have here. Initialization is set as seen here:

|  | <u>H</u> elp |
|--|--------------|
|  |              |
|  |              |
| Study name: BURNER   |              |
| Case name: CASE1   |              |
| XML file name: burnertube.xml  |              |
| Calculation environment  |              |
| Solution Domain Zone selection all cells   |              |
| Lein Definition of the volume zones  |              |
| Calculation features   |              |
| Velocity initialization  |              |
| Thermal model VelocitU 0.0 m/s   |              |
| 💼 Physical properties  |              |
| Additional scalars VelocitV 1.0 m/s     Boundary conditions                        |              |
| Definition of boundary regions     Dynamic variables bound, cond, VelocitW 0.0 m/s |              |
| Scalars boundary conditions  |              |
| Calculation control  |              |
| Calculation management   |              |
| Global initialization by reference velocity  |              |
| Prepare batch calculation  | _            |
| Reference velocity 1.0 m/s   |              |
|  |              |
|  |              |
|  |              |
|  |              |
|  |              |
|  |              |

#### Next the physical properties of the fluid needs to be set:

| Code_Saturne user interface       □ ○ ○ ○         Elle Tools Options       Help         Image: Study name:       Elle Tools Options         Image: Study name:       Case name:         Case name:       CASET         XML file name:       Durner tube xml         Image: Study name:       Case name:         Calculation environment       Case name:         Image: Study name:       Case name:         Calculation for the volume zones       Constant         Image: Thermosphysical models       Constant         Image: Toremosphysical models       Constant         Image: Toremosphysicalmodels       Constant   | Elle ⊥ools Ogtions       Help         Image: Second Se |  | Cada Saturaa usar iatarfasa  | X      |
|---|---|--|--|--------|
| Find properties     Beference values     Specific Heal     Constant     Casual value     Constant     Casual value     Constant     Constant     Constant     Casual value     Constant     Con | Find properties     Reference value     Substant     Sectific Heal     Constant     Reference Cp 2220.0 J/kg/K  |  | -  | 1      |
| <ul> <li>Identity and paths</li> <li>Definition of the volume zones</li> <li>Thermophysical models</li> <li>Calculation features</li> <li>Mobile mesh</li> <li>Turbulence models</li> <li>Thermal model</li> <li>Initialization</li> <li>Physical properties</li> <li>Reference values</li> <li>Fluid properties</li> <li>Gravity, hydrostatic pressure</li> <li>Additional scalars</li> <li>Definition of boundary regions</li> <li>Definition of boundary conditions</li> <li>Calculation control</li> <li>Numerical parameters</li> <li>Calculation management</li> <li>Start/Restart</li> </ul>   | Identity and paths       Density         Solution Domain       Definition of the volume zones         Thermophysical models       constant □         Calculation features       Mobile mesh         Turbulence models       Viscossity         Thermal model       Initialization         Initialization       constant □         Physical properties       constant □         Reference values       constant □         Thuid properties       specific Heat         Additional scalars       specific Heat         Optimic variables bound, cond.       Scalars boundary conditions         Calculation control       constant □         Numerical parameters       constant □         Calculation management       Start/Restart   |  | Study name: BURNER<br>Case name: CASE1                                   | Help   |
| Additional scalars<br>Boundary conditions<br>Definition of boundary regions<br>Dynamic variables bound, cond.<br>Scalars boundary conditions<br>Calculation control<br>Numerical parameters<br>Calculation management<br>Memory Management<br>Start/Restart   | Additional scalars<br>Boundary conditions<br>Definition of boundary regions<br>Dynamic variables bound. cond.<br>Scalars boundary conditions<br>Calculation control<br>Numerical parameters<br>Calculation management<br>Memory Management<br>Start/Restart   | <ul> <li>Identity and paths</li> <li>Solution Domain</li> <li>Definition of the volume zones</li> <li>Thermophysical models</li> <li>Calculation features</li> <li>Mobile mesh</li> <li>Turbulence models</li> <li>Thermal model</li> <li>Initialization</li> <li>Physical properties</li> <li>Reference values</li> </ul> | constant Reference p 0.656 <i>Viscosity</i> Constant Reference u 1.1e-05 |        |
|   | a da anti-anti-anti-anti-anti-anti-anti-anti-   | Gravity, hydrostatic pressure     Additional scalars     Boundary conditions     Definition of boundary regions     Dynamic variables bound, cond.     Scalars boundary conditions     Calculation control     Numerical parameters     Calculation management     Memory Management     Start/Restart                     | constant Reference Cn 2220.0   | J/kg/K |

Definition of boundary regions are set next – the names of the boundary regions corresponds to the group names assigned in Salomé.

|                                       | Code_Saturne user  | interface                    | _ 0 X              |
|---------------------------------------|--------------------|------------------------------|--------------------|
| <u>File</u> Tools                     | O <u>p</u> tions   |                              | <u>H</u> elp       |
|                                       | -                  | P. 57                        |                    |
|                                       | 3 🛠 🗗 🖇            |                              |                    |
|                                       |                    | Study name: BURNER           |                    |
|                                       |                    | Case name: CASE1             |                    |
| Mut.                                  |                    | XML file name: burnertube.xi | ml                 |
| Calculation environment               | Definition of boun | dary regions                 | <u> </u>           |
| Solution Domain                       | Label              | Zone Nature                  | Localisation       |
| Thermophysical models                 | walls              | 3 wall                       | walls              |
| Calculation features                  | inlet              | 1 inlet                      | inlet              |
| Mobile mesh     Turbulence models     | outlet             | 2 outlet                     | outlet             |
| Thermal model                         |                    |                              |                    |
| Initialization                        |                    |                              |                    |
| Physical properties                   |                    |                              |                    |
| Reference values     Fluid properties |                    |                              |                    |
| Gravity, hydrostatic pressure         |                    |                              | ₫                  |
| 产 Additional scalars                  | 81                 |                              | 2                  |
| Definition and initialization         |                    |                              |                    |
| L Physicals properties                | Label walls        |                              |                    |
| Definition of boundary regions        | 7                  |                              |                    |
| Dynamic variables bound. cond.        | Zone 3             |                              |                    |
| Calculation control                   | Nature             | wall 🗖                       |                    |
| Numerical parameters                  |                    |                              |                    |
| 😤 Calculation management              | Localisation walls |                              |                    |
| Memory Management                     | ,                  |                              |                    |
| Start/Restart                         |                    | Create Modify                | Delete             |
|                                       |                    |                              |                    |
|                                       |                    |                              | - 61               |
|                                       | Import groups and  | l references from Prepro     | cessor listing 🥨 🚽 |
|                                       |                    |                              |                    |
|                                       |                    |                              |                    |
|                                       |                    |                              |                    |

Entering the calculated velocity in the positive Y-direction in Dynamic variables bound. cond.

|   | 0                          | ode_Saturne u    | ser interface | e             |          | -            |              |
|---|----------------------------|------------------|---------------|---------------|----------|--------------|--------------|
|   | <u>F</u> ile <u>T</u> ools | O <u>p</u> tions |               |               |          |              | <u>H</u> elp |
|   | 📓 🐜 📄                      | 3 🛠 🎜            | 😵 🗐           |               |          |              |              |
|   |                            |                  | Study name:   | BURNER        | 1        |              |              |
|   |                            |                  | Case name:    | CASE1         |          |              |              |
| Miller.   |                            |                  | XML file nan  | ne: burnertuk | be.xml   |              |              |
| Calculation environme   | ent                        | Dynamic variab   | les bound. co | ond.          |          |              |              |
| Solution Domain   | ime zones                  | Label            | Zone          | Nature        |          | Localisation | <b>4</b>     |
| 👝 Thermophysical mode   |                            | walls            | 3             | wall          |          | walls        |              |
| Calculation features  |                            | inlet            | 1             | inlet         |          | inlet        |              |
| Turbulence models   |                            |                  |               |               |          |              |              |
| - Thermal model   |                            |                  |               |               |          |              |              |
| Physical properties   |                            |                  |               |               |          |              |              |
| Reference values  |                            |                  |               |               |          |              |              |
| Fluid properties  | pressure                   |                  |               |               |          |              | ∀            |
| 📂 Additional scalars  |                            | 5                |               |               |          |              |              |
| Definition and initiali   |                            |                  |               |               |          |              | -            |
| Boundary conditions   |                            |                  | Veloci        | ty compone    | ents u   | -1           |              |
| Definition of boundar   |                            |                  |               |               |          |              |              |
| Dynamic variables bo  |                            |                  | U             | 0.0           | m/s      |              |              |
| Calculation control   |                            |                  | v <b>F</b>    | 33.0          | m/s      |              |              |
| Numerical parameters           P         Calculation management |                            |                  | Ľ             | 00.0          | 11/2 3   |              |              |
| Memory Managemen  |                            |                  | W             | 0.0           | m/s      |              |              |
| Start/Restart   | ation                      |                  |               |               |          |              |              |
|   |                            | Turbulence       |               |               |          |              |              |
|   |                            |                  | Calculation b | y hydraulic   | diameter | -            |              |
|   |                            | _                |               |               |          |              |              |
|   |                            |                  | Hydraulic dia | umeter 0.0    | 1019     | m            | $\nabla$     |
|   |                            |                  |               |               |          |              |              |
|   |                            |                  |               |               |          |              |              |

Since this is a steady state condition, this is what I left it at in Steady Management.

Under Output control I set it to Post-processing every 'n' time step to 1 and select Ensight Gold as the output format (This is so ParaView can post-process the result):

|   | Code_Saturne user interface              | _ O X        |
|---|--|--------------|
| Ei  | ile <u>T</u> ools O <u>p</u> tions       | <u>H</u> elp |
|   | í ଲ 📙 30 🛠 🗗 🔗 🚀                         |              |
|   | Study name: BURNER                       |              |
| ACT -   | Case name: CASE1                         |              |
| (III)   | XML file name: burnertube.xml            |              |
| 👝 Calculation environment                         |  |              |
| Identity and paths                                |  |              |
| Solution Domain                                   | Output listing at each time step         |              |
| Definition of the volume<br>Thermophysical models | zones                                    |              |
| Calculation features                              | Post-processing                          |              |
| Mobile mesh                                       |  |              |
| Turbulence models     Thermal model               | Post-processing every 'n' time steps 🥏 🔤 | 1            |
| Initialization                                    |  |              |
| Physical properties                               | Fluid domain post processing             |              |
| Reference values     Fluid properties             | Domain boundary post processing          |              |
| Gravity, hydrostatic pres                         |  |              |
| Additional scalars                                | Type of post processing for mesh fixed   | =1           |
| Physicals properties                              |  |              |
| 📂 Boundary conditions                             | Execution Execution and                  |              |
| Definition of boundary re                         |  |              |
| Scalars boundary conditi                          |  |              |
| Calculation control                               | format binary 🔤                          |              |
| Steady management     Output control              |  |              |
| C Solution control                                | polygons to display 💴                    |              |
| Numerical parameters                              |  |              |
| Calculation management                            | polyhedra to display 💴                   |              |
| 🗋 Start/Restart                                   | his cardier                              |              |
| Prepare batch calculation                         | n big_endian 📕                           |              |
|   |  |              |
|   |  |              |
|   |  |              |

Once everything is set up, theres only one thing left: Load the 'runcase' file under Prepare batch calculation and press Code Saturne batch running:

|                                    |                            | Code_Saturn      | ie user in  | terface                  |          | - O X        |
|------------------------------------|----------------------------|------------------|-------------|--------------------------|----------|--------------|
| E                                  | <u>-</u> ile <u>T</u> ools | O <u>p</u> tions |             |                          |          | <u>H</u> elp |
|                                    | 🛛 ଲ 🛛                      | - 3 🛠            | <b>a</b> 🔗  | <b>~</b>                 |          |              |
|                                    |                            |                  |             | dy name: BURNER          |          |              |
|                                    |                            |                  |             | e name: CASE1            |          |              |
|                                    |                            |                  |             | L file name: burnertube. | ×ml      |              |
| 11. ···                            | [                          |                  |             |                          |          |              |
| Calculation environment            | <b>_</b>                   | Computer S       | Selection   |                          |          |              |
| Identity and paths Solution Domain |                            | ·                |             |                          |          |              |
| Definition of the volume           | zones                      |                  |             | Workstation              | -        |              |
| 👝 Thermophysical models            |                            |                  |             |                          |          |              |
| Calculation features               |                            |                  |             |                          |          |              |
| · Mobile mesh                      |                            |                  | Select t    | he batch script file     | runcase  |              |
| Thermal model                      |                            |                  |             |                          |          |              |
| Initialization                     |                            |                  |             |                          |          |              |
| 产 Physical properties              |                            | Prepare bat      | tch calcula | ation                    |          |              |
| - Reference values                 |                            |                  |             |                          |          |              |
| Fluid properties                   |                            | I                | Number of   | processors               | 1        |              |
| Additional scalars                 |                            |                  |             |                          |          |              |
| Definition and initializat         | tion                       |                  | User files  |                          | <b>a</b> |              |
| Physicals properties               |                            |                  |             |                          |          |              |
| Boundary conditions                | agiono                     |                  |             |                          | - 1      |              |
| Definition of boundary r           | -                          |                  | Advance     | d options                | 2        |              |
| Scalars boundary condit            |                            |                  |             | L                        | 1•       |              |
| Calculation control                |                            |                  |             |                          |          |              |
| Steady management                  |                            |                  |             |                          |          |              |
| Output control                     |                            |                  |             | Code_Satur<br>batch      | ne       |              |
| Numerical parameters               |                            |                  |             | running                  |          |              |
| Equation parameters                |                            |                  |             |                          |          |              |
| Global parameters                  |                            |                  |             |                          |          |              |
| Calculation management             |                            |                  |             |                          |          |              |
| Start/Restart                      |                            |                  |             |                          |          |              |
| Prepare batch calculatio           | m                          |                  |             |                          |          |              |
|                                    |                            |                  |             |                          |          |              |
|                                    |                            |                  |             |                          |          |              |
|                                    |                            |                  |             |                          |          |              |

This launches the calculation and outputs the temporary results in ~/tmp\_saturne and the finished results in ~/Study/CASE1/RESU As the calculation runs, issueing the terminal command tail -qf ~/tmp\_saturne/Study.CASE1.date/listing will give info on the progress and convergence of the calculation:

If everything went smoothly, Code\_Saturne rewards you with a "**Normal** Simulation Finished" and one can move on to...

Fire up ParaView and select

 $\textit{File} \rightarrow \textit{open} \rightarrow \textit{Study/CASE1/RESU/CHR}. \textit{ENSIGHT}. \textit{date/chr}. \textit{case}$ 

Click 'apply'

and select

Filters  $\rightarrow$  Alphabetical  $\rightarrow$  Cell data to point data

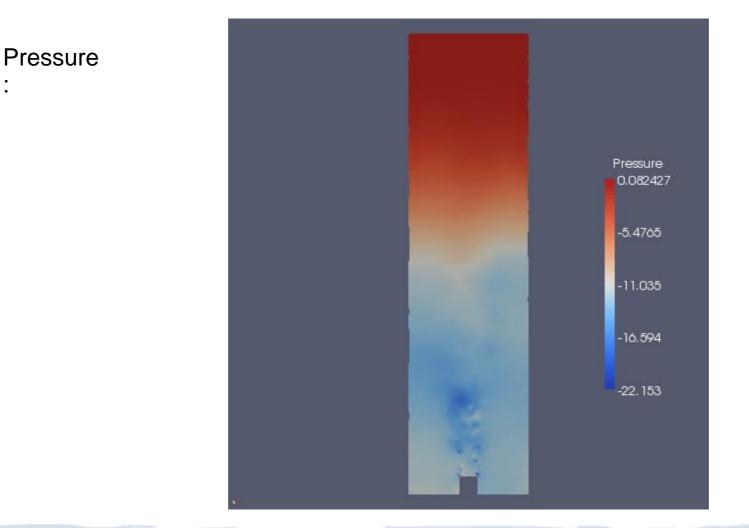
and

Filters  $\rightarrow$  Common  $\rightarrow$  Slice and de-select show plane select Z Normal –

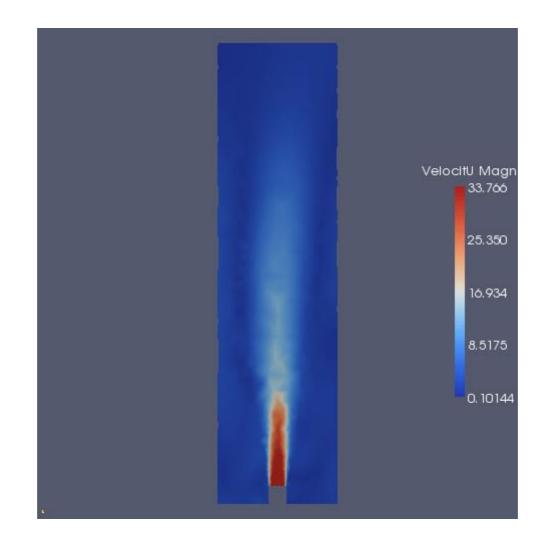
click apply

Cell data to point data will give a much more smooth representation of the fields and allow showing the velocity field as streamlines.

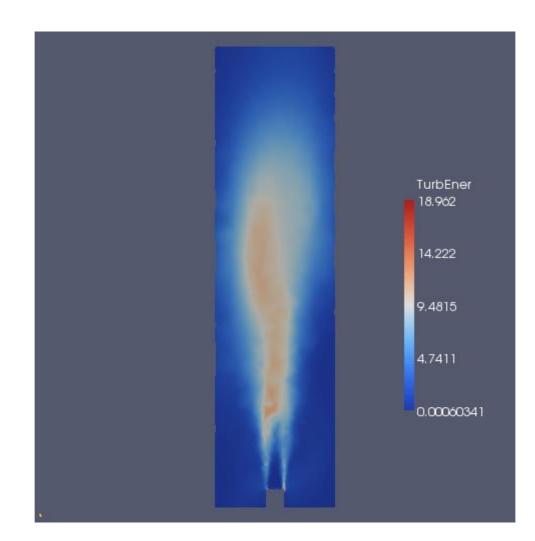
Images are captured at timestep 200 where we have a reasonably steady state:



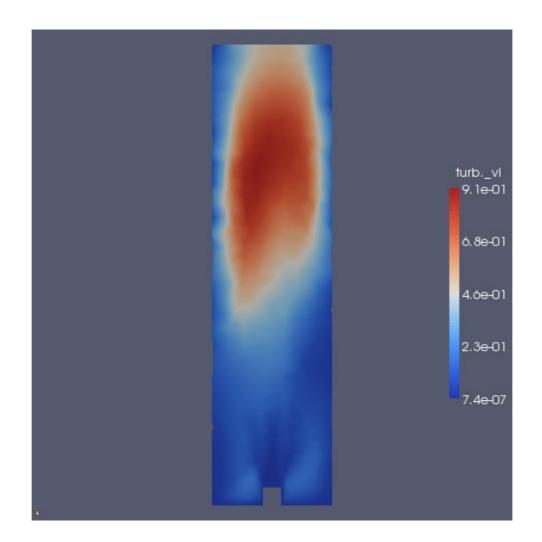
#### Velocity



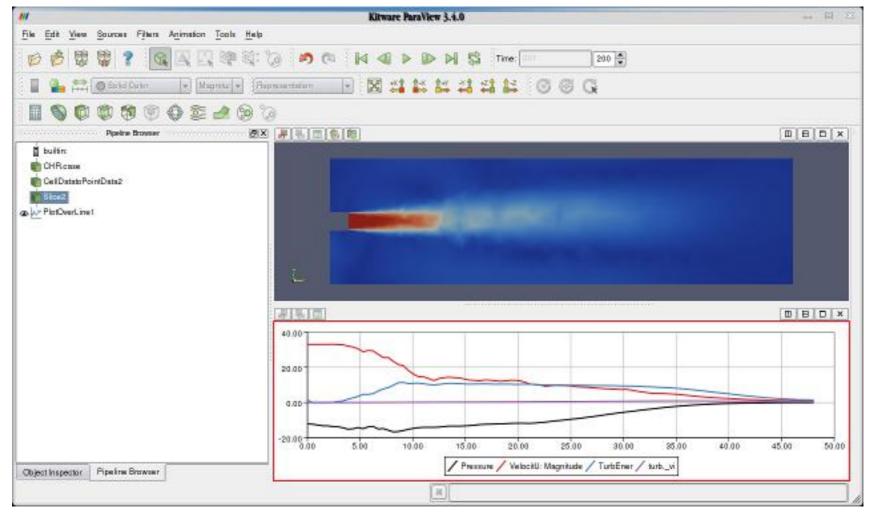
Turbulent energy



Turbulence – magnitude



ParaView set up with Plot over line to show values along the X axis:



So after post processing the simulation, what conclusions can be drawn? Velocity is indeed 33m/s as was the initial condition. The low pressure zone around the base of the jet is ~-22Pa, this is what I'm interested in, since this is what I plan to measure later on in the practical approach. The zones for turbulence and turbulent energy isn't that surprising at all – I'll keep that in mind for when I have to modify the burner tube later on.

So now I've gotten the simulation to finish and have reasonable derivatives(?), I proceed to do a practical test on the burner tube. First I want to measure the gas flowing into the burner tube; this is done with a flow meter connected between the gas bottle and the regulator. The results of the measurement is used in the MathCAD document both to hold up against the technical document on the gas burner and to control my theoretical calculation.

In this image the middle flowmeter is used and if you squint, you can see the ball hovering at around 6[l/min].



During the run of the flow measurement, I attach a manometer to the low pressure side of the regulator to make sure it has the proper inlet pressure: 1[kPa]



Next up is the actual measurement of the burner tube. I have attached a small fitting to the aeration port and sealed the ports with electrical tape. This should hopefully emulate the simulation setup.

Turning on the gas and letting it flow free through the burner tube with a micro-manometer attached, gave the following reading:



# Conclusion

Goal archived, I have a model on which I can build further on. Code\_Saturne can be a harsh mistress. I've have spent many many hours tweaking the shape of the mesh for a seemingly simple numerical analysis. Especially the inlet takes much care to model – the initial 'backlash' of the fluid as it enters the tube (as seen in the animations) has given me much grievance. The trick in this case since I was only interested in low pressures zone in the steady state, was to let the inlet protrude into the chamber to allow for the 'backlashing' – otherwise I got wildly inaccurate if any, results. CFD requires you to know exactly what you want to retrieve from the simulation and under what conditions. It simply does not converge if you don't have some idea of what mesh size, inlet velocity etc. etc. to apply. This takes much experimentation, at least for me since I don't have a formal background in fluid dynamics.

Don't do CFD on a 1.7ghz celeron if you have the choice – it will wear you out.