A picture containing text

Description automatically generated

Thermal simulations STL and Paraview

Computational Fluid Dynamic (CFD) simulations in general divide geometry up into smaller blocks and calculate energy and mass balance at the borders. Increasing the cell count, together with increasing the order of applied calculation schemes increases accuracy. Both measures increase the computational requirements too.

## Available simulation cases:

|  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- |
| name | experiment | No. people | heated glass | floorheating | Air Exchange Rate |
| n | n | T [C] | T [C] | n |
| STL\_IGU\_55 | 1 | 0 | 55 | 20 | 1 |
| STL\_IGU\_28 | 2 | 0 | 28 | 20 | 1 |
| STL\_IGU\_25\_FH\_25 | 3 | 0 | 25 | 25 | 1 |
| STL\_IGU\_55\_N1\_AER.5 | 4 | 1 | 55 | 20 | 0.5 |
| STL\_IGU\_28\_N2\_AER1 | 5 | 2 | 28 | 20 | 1 |
| STL\_IGU\_25\_N4\_AER2 | 6 | 4 | 25 | 25 | 2 |
| STL\_IGU\_28\_N6\_AER4 | 7 | 6 | 28 | 20 | 4 |
| STL\_IGU\_55\_N6\_AER4 | 8 | 6 | 55 | 20 | 4 |

## Boundary conditions and simulation setup:

|  |  |  |
| --- | --- | --- |
| Temperature | [K] | [C] |
| Tambient | 278.15 | 5 |
| T Technical space | 293.15 | 20 |
| T other testroom | 293.15 | 20 |
| T start | 293.15 | 20 |
| T inlet | 291.15 | 18 |
| skin temperature | 306.95 | 33.8 |
| Heat output person | 130W |  |

* k-epsilon turbulence model. Ideal gas, compressible.
* externalWallHeatFluxTemperature heat transfer
  + coefficient mode for wall temperature BC
  + power mode for mannequin temperature BC,
* simplefoam transient state
* P1 radiation model
* 1e-4 residuals on: P\_rgh, U, h, k, epsilon
* 1e-5 on G
* dt = 1s.

See the title page for geometry indication, the same data can be visualized in Paraview.

## Why Paraview

Because analyzing such vast 3D-datasets is common in multiple fields, there is a good use for a program like Paraview <https://www.paraview.org/> . This program allows you to analyze fields (U, P, T, Q, G) or any arbitrary data-format with a suitable reader.

In this document I will walk you through my workflow for Paraview, this comprises of the following:

1. Starting up an analysis by opening a suitable file
   1. I do this while the simulation is running, this allows me to refresh files and see the latest results as they are written
2. Analyze the geometry
3. Make suitable cuts in the 3D-geometry to visualize the results in 2D
4. Analyze the different available fields and check for anomalies
5. Create plots and export tabular data/screenshots

## Installation

Installation of ParaView is straightforward, go to the following site: <https://www.paraview.org/download/> Download the Paraview-x.xx.x-MPI executable version and install this.

# Tutorial

Some steps done below can be quickly done by loading the .pvsm files with the **Load state** button, the files are slightly different.

The files for this tutorial should be downloaded (at least one case from the table above)

## Opening paraview

After installation, open Paraview, and start with going to **File -> open (red)** and open the **STL\_IGU.foam** (red) file (this activates the openfoam reader built-in into Paraview)

Graphical user interface

Description automatically generated You can see the numbered folders in green, these represent simulation steps, if a transient solver was used, these would be timesteps, in this case they show different states of flow and temperature development. In general it is better to study the later simulation steps for transient simulations.

On the left in the red box you see the pipeline browser, here you see all operations represented as a tree structure. This allows you to setup complex data processing pipelines parallel from eachother.

Below that you see the properties box, you’ll spend a lot of time here setting up visualizations. Here it shows what mesh regions to show, starting with internalMesh is the best Idea, so press **apply** to start loading the simulation.

At first this doesn’t tell a lot, and immediately brings the problem of 3D simulations to light, data for each point in space.

Graphical user interface

Description automatically generated

In the green box above you see the field indicator, here you can select the speed field (**u**), pressure (**p**), radiation (**G**, W/m2)**,** temperature (**T**) and more. You could calculate new fields with postprocessing steps in ParaView (for example thermal comfort, PMV, etc). Select the Temperature field **T** for now.

In the yellow box you see the time indicator, you can go to the latest timestep available for now.

Now select **STL\_IGU.foam (blue)** in the **pipeline browser** and press **<ctrl+space>**, a pop-up **search-window** opens to tie new functions to your current selection. Now type **slice** and click the symbol (blue) and press apply in the **properties**. Now you get some insight into the simulation.

Graphical user interface

Description automatically generated

Graphical user interface

Description automatically generated with medium confidenceNow Make a similar cut for the Y-plane by opening the **search-window** and creating a new slice (select STL\_IGU.foam first). In the properties browser you get access to more settings by pressing the cogwheel, and you can quickly select the Y-plane as normal for cutting.

Now select **Slice1** and open the **search-window**, look for **Contour** and go to properties.

Now select **contour by: T** and clear the **value range** with the cross. Now make a new **range** with the range button from for example 290 – 305 in 25 steps.

**A screenshot of a computer

Description automatically generated with medium confidence**Select the file again and look for the **clip** function. In properties select **clip type** **scalar** and select **Scalars** **T.** Now untick the **invert** tickbox, and you can visualize all temperatures above the temperature set with the **value** slider. Graphical user interface

Description automatically generated

Now you can set a better color scheme to for example determine zoning by playing with the scale, see below screenshots.

A screenshot of a computer

Description automatically generated with medium confidence

Graphical user interface

Description automatically generatedAnother valuable function for characterization is the **plot over line** and **plot over time** function. Here I’ll show the first. Select the file and search for **plot over line** and **apply** in **properties.**

A plot shows, and as shown above 6 coordinates for 2 points can be set, the start and end-point of your line. Values can be retrieved along this line. If you scroll down, you can filter out fields to plot.

Another important trick is the extensive export functions paraview offers, select anything and press the screenshot buttons in the windows with **CTRL/SHIFT** modifier keys and you can save screenshots or tabular data immediately, or through the file menu after selection.

Next to that you can split windows and open for example the **spreadsheet view**, this allows you to view any compatible data as a table and export it as such. If you can’t export it conveniently yet, see if you can transform it with Paraview, it’s fast and efficient and easy to work with after you get used to it.

Graphical user interface

Description automatically generated

## Final exercise – compare different simulations

A screenshot of a computer

Description automatically generated with medium confidenceThe next goal will be to load multiple simulations in ParaView and compare them in the same view.

Start with adding a Z-normal slice to the original file, hide all other slices and operations by clicking the eyeball symbol, and center to the X-Y-plane with the button in the toolbar. Also close the other views and resize the color-legend.

Now rename the slice to represent your simulation, and open an additional **\*.foam** file in the pipeline browser (through **file** menu).

A screenshot of a video game

Description automatically generated with medium confidence

In the **properties** you can see which simulation folder is loaded, make sure they are different.

First we need to slice the newly loaded file the same. Select the new **STL\_IGU.foam** and make a **Z-normal slice.** Now we need to translate the slice up or left to not block the current visualization. In the properties browser select the latest slice, look for **tran** and enter a **-3.5** **y** translation. This will place it just below the previous cut.

A screenshot of a computer

Description automatically generated with low confidence

Application

Description automatically generated with medium confidenceThese steps can be repeated for all simulations and different slice directions, comparisons between different situations can be made. By playing with the scale and representation valuable insights can be made. This shows a scale from 290 to 301 K, which indicates the difference between a 28C pane and 55C pane well at this specific slice location.

## Load state

Load state is a handy feature to load a pre-set arrangement of windows and analysis, it also restores any tabs you created. Press **file** -> **load state** and find a **STL\_Tut.pvsm** or **STL\_ThermalAnalysis.pvsm**  file and press ok. A screenshot of a computer

Description automatically generated with medium confidence Then select **search under specified folder** in the pop-up and redirect to the **\*.foam** file of the case you want to load.

Graphical user interface, text, application

Description automatically generated