Open Access Journal
Journal of Power Technologies 97 (3) (2017) 252–258
journal homepage:papers.itc.pw.edu.pl

Instability issues in the server room cooling system

Tomasz Kwiatkowski[a,*], Piotr Prusiński[a], Dariusz Zgorzelski[a], Sylwester Kozioł[a]

[a]*National Centre for Nuclear Research, A. Sołtana 7, 05-400 Otwock-Świerk, Poland*

Abstract

Since 2009, the National Center for Nuclear Research in Poland has been constructing its own High Performance Computing (HPC) Centre under the name Świerk Computing Centre (CIŚ). Now, it is ready reaching its target—1 PFLOPS—in December 2015. However at the early operation stage, one of its major problems was the unstable work of the HPC cluster cooling system, resulting in increased maintenance costs. The main aim of this work is to thoroughly investigate the origin of the problem and to find the best solution for it based on results from a Computational Fluid Dynamics (CFD) analysis. The constructors suspected that the oscillations in the flow domain are caused by thermal flow stratification, but something else was proved. In this paper, a wide range of cases will be analyzed, covering different work regimes of the installation as well as various geometry modifications. Finally, certain improvement to the current design will be suggested by the CFD Analysis Group.

Keywords: HPC, computer cluster, efficient cooling, CFD

---

1. Introduction

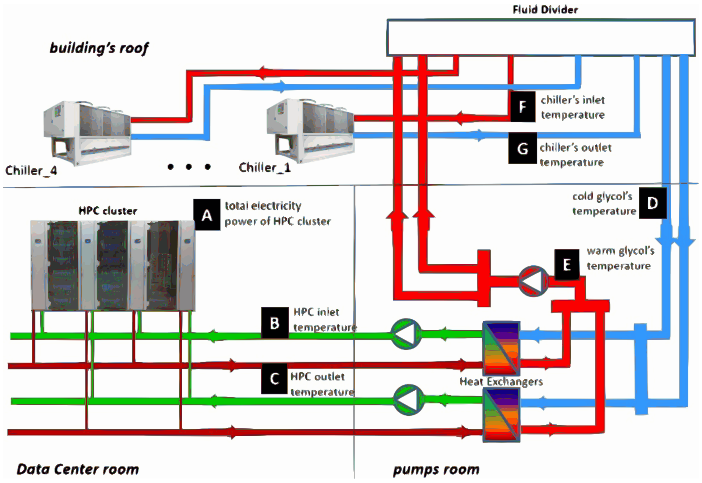

Figure 1: Scheme of the current cooling installation

Since the beginning of 2016, the National Center for Nuclear Research (NCBJ) in Poland can offer access to a brand new High Performance Computing facility called the Świerk Computing Center (CIŚ). However at an early stage in its operation, an unstable cooling system was blamed for higher than planned operating costs. These instabilities manifested themselves first by alternate operation of compressors attached to chillers, i.e. compressors changing their mode of operation in a binary way (on-off) in a short period of time. Until this study, the real origin of the problem was unclear, though it was believed that it related to high-frequency temperature oscillations of the cooling medium measured at a few points in the system (see Fig. 1 white letters B-G).

The main aim of this work is to investigate this problem thoroughly and propose a solution that fixes it through Computational Fluid Dynamics (CFD). As the observed system instabilities are caused by fluid temperature oscillations occurring in the interior of the fluid divider, only this part of the system will be of interest here.

The following sections contain a detailed problem description as well as the assumptions and resulting from them—the CFD model. Finally, the results, comments and design suggestions will be presented.

2. Need for analysis

The investigated cooling system was designed to provide up to 1 600 kW of cooling power.

A schematic representation of the entire analyzed installation is shown in Fig. 1. The installation consists of two main

---

[*] Corresponding author

Email addresses: `tomasz.kwiatkowski@ncbj.gov.pl` (Tomasz Kwiatkowski), `piotr.prusinski@ncbj.gov.pl` (Piotr Prusiński), `dariusz.zgorzelski@ncbj.gov.pl` (Dariusz Zgorzelski), `sylwester.koziol@ncbj.gov.pl` (Sylwester Kozioł)

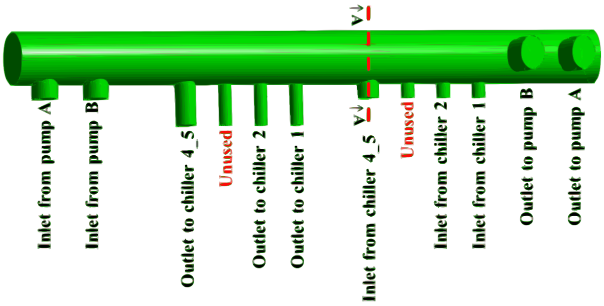

Figure 2: Outline of the 3D fluid divider

fluid circuits. The first, which uses water as a coolant, removes heat directly from the computer cluster (green/bronze pipelines). The HPC cluster can generate up to 1 600 kW of heat divided into 4 rows of computer cluster rack boxes. The second circuit uses glycol to transport the heat outside through the hydraulic/fluid divider (red/blue pipelines) and then, if needed, to the chillers. Between the circuits, there are two heat exchangers, each of 800 kW. The original blueprints assumed the existence of 5 chillers of about 400 kW each, two of which should be attached to one of the 4 legs of the fluid divider. One chiller was intended to serve only as back-up, so the total cooling capacity should always achieve 1 600 kW with a certain degree of confidence. This chiller, however, has never been delivered, so one leg is unused (see Fig. 2).

Under constant cooling conditions, the system is supposed to work in a stable way, but it does not. The issue is that the cold glycol temperature starts to oscillate regardless of the constant or near-constant electric power consumption by the HPC cluster which corresponds to constant heat generation. The evidence of that can be found in the report [1].

Consequently, chiller controllers start to turn on and off selected compressors in chillers. That causes an increase in power consumption compared to similar, but constant work of chillers. The water circuit and pumps did not show any anomalies and their work remained stable. Moreover, the chillers were examined thoroughly by authorized specialists, who proved they were in full working order. Thus, the fluid-divider is the only one element that can be the source of glycol temperature oscillations. Unfortunately, the fluid divider is in essence a kind of "black-box", with no viewfinder or measurement instrumentation inside and the only information available is gathered through a few point thermocouples distributed over the whole cooling system with low frequency as presented in Fig. 1. To prove these claims and to examine the fluid-divider, CFD simulation seems to be the best fit technique for use in this case.

3. Simulation of the current fluid divider

The fluid divider is an external component of a computer cluster cooling system that joins the glycol circuit with chiller fan circuits. Assuming the hydraulic divider has to be treated as a black-box, a numerical simulation became the only way to examine fluid flow inside the divider. That way the investigation may show possible anomalies in the work of the system. In order to perform this type of analysis in CFD code, the geometry, mesh and appropriate CFD model should be created and discussed.

The whole process of modelling was conducted using ANSYS Workbench 15.0. Geometry was modelled in Design Modeler, mesh was created in the Meshing module and simulations were performed in Fluent.

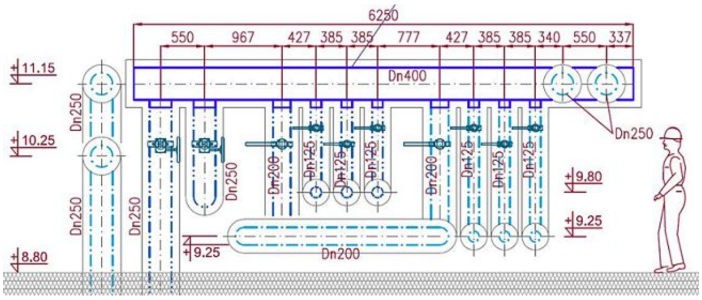

Figure 3: The dimensions of the fluid divider. A navy blue line indicates the investigated flow domain

The examined geometry consists of a cylindrical collector with a set of inlets and outlets. In figures 3 and 2 a schematic diagram of this device is presented. Starting from the left hand side of the diagram, there are two inlets from pumps A and B or to be more precise from pump sections A and B, subsequently there are four outlets to chillers (chillers 4_5, 3, 2 and 1—chiller 4_5 is a double chiller), next four chiller inlets and finally, two outlets to the pumps (rotated by 90 degrees).

The cut cell approach was chosen to create the mesh. This method [2] uses a uniform stationary background Cartesian mesh, as described in the report [1]. At this juncture, it is good to mention at least some advantages of the cut cell mesh [3]:

- It is always body-fitted;
- It provides simultaneously high fidelity geometric representation and a very coarse mesh;
- It has better mesh quality parameters than other approaches, i.e. no skew;
- It assumes cell refinement at the wall.

Subsequently, the divider geometry was cut by the mesh grid (Fig. 4). In consequence, the total divider volume (equals to 0.85 m^3) was split into 728 957 elements. Despite the simplicity of the geometry, the domain could not to be represented by the 2D model and 3D analysis had to be applied because of the lack of symmetry. The report [1] contains more detailed mesh data.

Knowing that the cooling power is coupled with the server electric power loads, it is rational to assume that the fluid divider will always be in a transient state. The boundary conditions were defined for the chosen cooling load variants. Thus, several different work regimes with different fluid flow

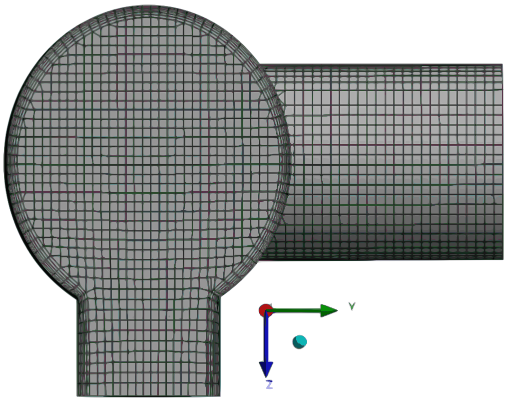

Figure 4: A y-z mesh cross-section with extra boundary layer (A-A corss-section marked as the dashed, red line in Fig 3.

loads were investigated. Limiting the number of variants to the lowest feasible amount, 3 load variants were investigated (more variants can be found in the report [1]):

Variant I: 40 l/s load with 2 pump and 2 chillers working (pump_A, pump_B, chiller_1, chiller_4_5)

Variant II: 20 l/s load with 1 pump and 2 chillers working (pump_A, chiller_1, chiller_4_5)

Variant III: 80 l/s load with 1 pump and 3 chillers working (pump_A, chiller_1, chiller_2, chiller_4_5)

The investigated domain contains only one fluid—namely glycol solution. Table 1 presents the initial and the boundary conditions.

Table 1: Initial and boundary conditions.

Condition		Definition
Inlets	Velocity inlet	Velocity profiles determined by periodic channels
Outlets	Outflow	
Operating conditions	Gravity (z-axis):	9.81 m/s^2
	Operating pressure:	101325 Pa
	Operating (Boussinesq) temperature:	288.15 K

When referring to turbulence, the $k-kl-\omega$ model was used in all presented simulations (transient and steady state). This model was chosen from a number of Reynolds Averaged Navier Stokes (RANS) formulations based on a sensitivity study, which is not a matter of discussion here. The $k-kl-\omega$ model was used mostly due to its ability to predict laminar-to-turbulent flow developments [4] [5].

Applying all the assumptions mentioned above and the conditions for transient and steady states, various data were obtained. In order to find the reasons for instabilities inside the divider, the two most important parameters were selected for final comparison, namely:

- temperature distribution,
- glycol solution concentration (mass fraction) at domain outlets with respect to the flow origin.

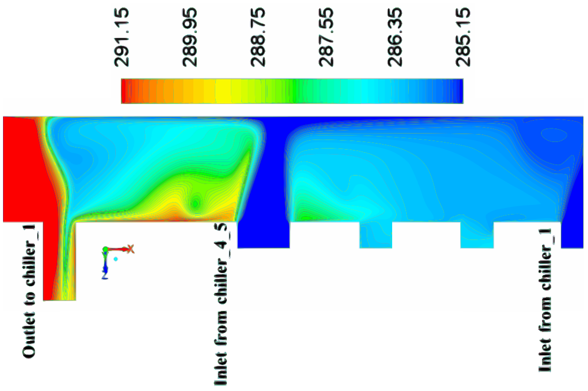

Figure 5: Temperatures in Variant I (40 l/s)

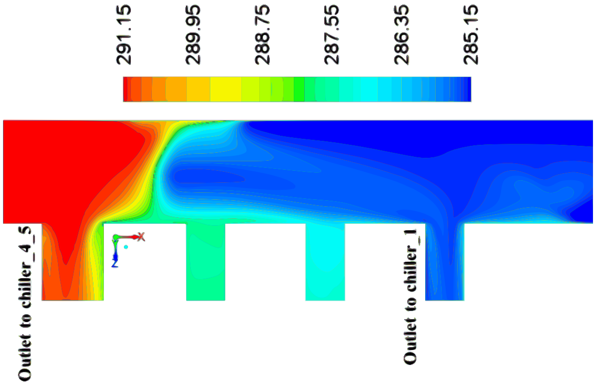

Figure 6: Temperatures in Variant II (20 l/s)

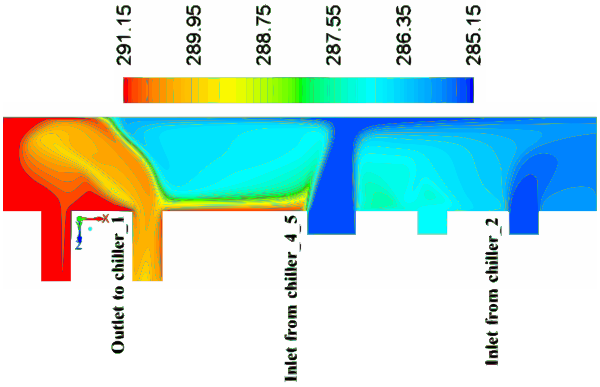

Figure 7: Temperatures in Variant III (80 l/s)

The temperature distribution from the steady state simulation, presented in figures 5 to 7, did not indicate any clear temperature front, instead they show a stationary mixing zone in the fluid.

However, transient results proved that the temperature oscillations in the fluid mixing zone (central part of the container) are significant. Mass fractions and temperatures in all vari-

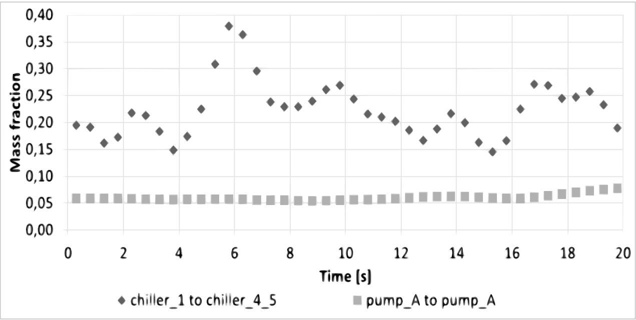

Figure 8: Mass faction in Variant III

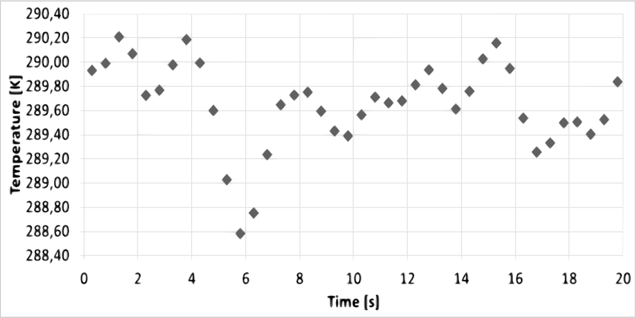

Figure 9: Temperature, K, on chiller_1 in Variant III

ants were analyzed. The results for Variant III are presented in figures 8 and 9.

In Fig. 8 the fluctuating mass fraction which varies between 15% and 35% at the inlet to chiller_4_5 is in fact a contribution from the chiller_1 outlet. The problem is not that the same fluid passes through a few chillers (because stage cooling is favorable), but that the process is unpredictable and features high frequency oscillations. Moreover, 5% of the flow appearing at the fluid divider outlet to pump section A actually bypasses the chillers totally.

The temperature oscillations (Fig. 9) in the chosen variant have an amplitude of 1.5 K, which means 25% of the maximum temperature gradient (285-291 K) in the container. It might have a negative influence on the cooling control system. This impacts the chillers, which may switch on and off alternately even if the cooling demand remains constant, causing inefficient use of them and higher energy consumption.

In each case, there was no element that could stabilize the temperature front. Thus, in all simulations of the current fluid divider transient, mixing zones were constantly changing their positions in the fluid divider.

4. Prospective solutions

The CFD results presented above and more deeply in the report [1] led the Management Board of CIŚ to the conclusion that a solution is required. Two teams were asked to produce solutions in parallel in two different areas:

- Control system of the cooling circuits analysis and reconfiguration;
- Fluid divider geometry modifications (baffles considerations).

The latter was assigned as a task for the CFD Analysis Group. This task is discussed in the following sections.

5. New geometry application

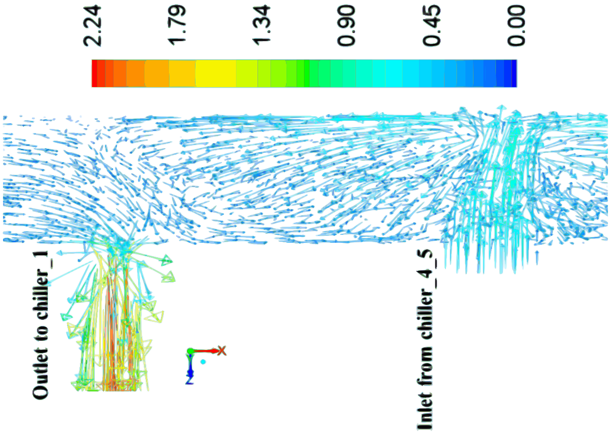

Figure 10: Velocity magnitude, m/s, in the mixing zone

Since the current geometry of the fluid divider was suspected to be the cause of instabilities in the fluid flow, some modifications of geometry were considered. The most likely space for oscillations to occur is in the domain between the inlets and outlets to the chillers. This is the place where cold and hot streams mix during the transient (presented in Fig. 10).

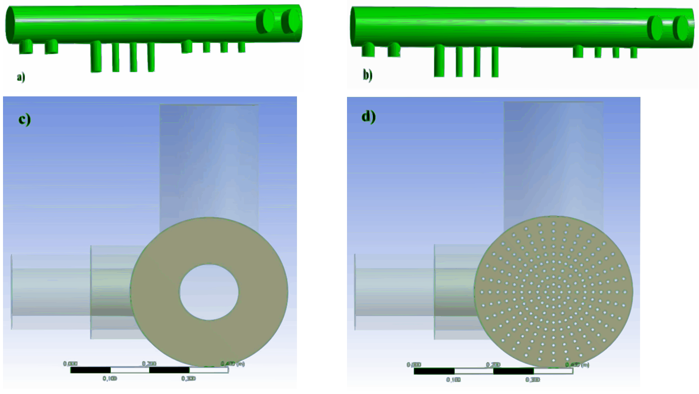

Figure 11: a) Current divider; b) Extended divider; c) Narrowing baffle; d) Sieve baffle

It is shown that the stream from a cold chiller leg (inlet from chiller_4_5) strikes directly into the upper wall of the collector and part of it is directed to a hot chiller leg (outlet from fluid divider to chiller_1). From the cooling point of view, that would be favorable (pre-cooling improves Coefficient Of Performance in cooling—COP), but in the case of unstable mixing, strong oscillations were observed. Moreover, part of the flow omits a set of chillers and goes directly from the pump outlet to the pump inlet and that also has to be fixed. Hence, several geometry modifications of the fluid divider central part were examined and only 3 of them were selected (Fig. 11) for the final comparison in this paper. For more information, the reader should look into the report [1].

Early on in the project, an idea was mooted of extending the central part of the divider. That might lead to retaining the mixing zone in the extended volume between the chiller inlets and outlets to reduce any sort of instabilities across a wider range. However, the extension could not be longer than 1 m due to the necessity of placing it on limited roof area.

Another idea was to make a wall with a bypass in the middle of the collector. This was based on the division of the fluid divider into the cold and warm sections, Instead, a simpler solution has been suggested, using a certain type of baffle to help steer the flow. Moreover, stage cooling is a favorable phenomenon, so the total division of the container into two parts is unacceptable. Additionally, in the case of low cooling loads the separated-sections-variant would create unnecessary pressure loss and cause additional power consumption by pumps.

To aid comparability, the baffles presented in this paper (Fig. 11) are of the same area, giving the same flow cross-sections. The equivalent flow area for both the circle in the narrowing baffle and for the sieve is 36 292 mm^2, so the diameter of the holes in a sieve baffle could be normalized to 10 mm in order to simplify the manufacturing process.

6. Comparison of results

In this case-to-case comparison, a few key factors are analyzed:

- mass fraction of the coolant from pump A, B and from the chiller_4_5 on outlets to pump A, B and chiller_1;
- temperature on outlets to pump A, B and chiller_1.

Mass fraction plots on outlet to pump A, B show the percentage of the flow that is bypassing the chillers and omitting the cooling process, which is undesirable from the thermodynamic point of view. Moreover, the plots show oscillation in these values (harmful for the control system).

Simulations of transient state are conducted for 20 seconds (limited computing resources and time for delivery of results). Plotted data are extracted with a 0.5 s time step, which suffices to clearly present changes in parameters over time. Each plot contains a comparison of the data for all geometries mentioned above.

5.1. Variant I (40l/s)

In this hydraulic load system, both pumps and two chillers are put into operation.

Starting with the reference case (with no modifications), it is worthwhile highlighting how much coolant is bypassing the chiller system. Summing the mass fractions from pump A to B and A to A gives approximately 5.5% of the stream from pump A omitting the chillers. Similar results are obtained with the stream from pump B, where 7.5% of the fluid is not cooled by the chillers. That means 13% in total! Changing the geometry of the divider cuts this value to 4% (extended divider) or even 1.65% (using a narrowing baffle), which will cause a reduction in temperature on the outputs to pumps. Thus, server room cooling will improve. The next challenging issue is the oscillation appearance observed both in the mass fractions and the temperatures on the inlet to chiller 1. The reference scenario (without modifications) illustrates significant instabilities in the fluid flow in the mixing part of the divider. The amplitude of the mass fraction of the coolant flowing from chiller_4_5 outlet to chiller 1 inlet is 4% (ranging from 8 to 12%) which is $\frac{1}{3}$ of the coolant transported in this way. That is crucial when considering automatic systems that control the cooling process. Based on the plotted values, one can conclude that the use of a baffle (with either a sieve or narrowing) can limit this effect significantly. Even the case with extension shows that some precipitation of balance occurs after the 10th second. On the other hand, the main disadvantage of using the baffles is a limitation of stage cooling corresponding to the coolant flow between the chiller_4_5 outlet and the chiller 1 inlet. Fig. 12 illustrates all these conclusions.

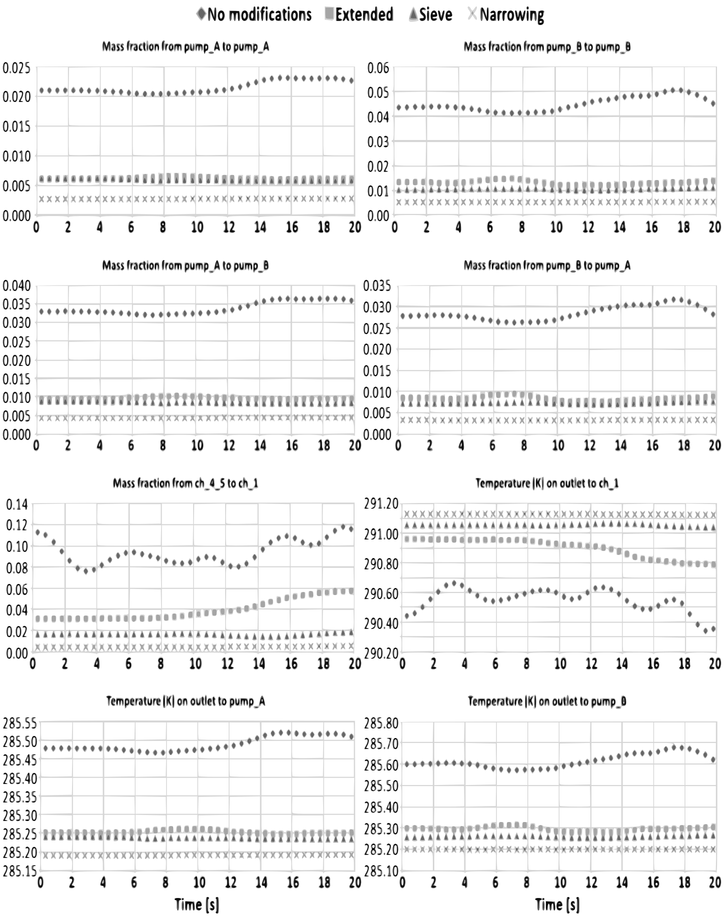

Figure 12: Changes of the mass faction and temperature in Variant I (40l/s).

5.2. Variant II (20l/s)

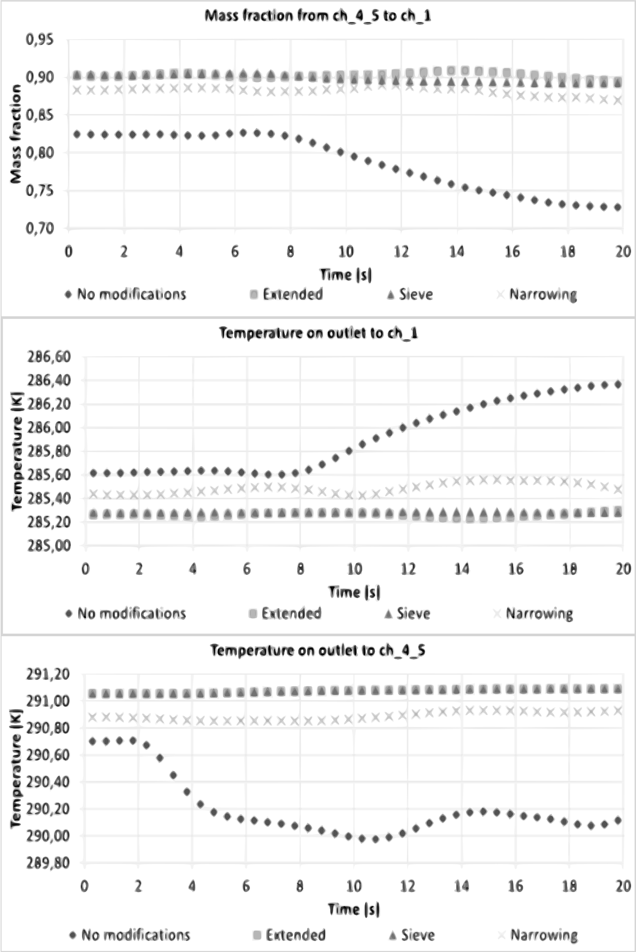

Figure 13: Changes of the mass fraction and temperature in Variant II (20l/s).

In this hydraulic load system, one pump and two chillers are put into operation. All remarks are in relation to plots in Fig. 13.

The system load here is the lowest of all the considered variants. The plots show that all 3 geometry modifications may be applied in order to cope with instabilities of the coolant flow. The plot of the temperature on the outlet to pump A is omitted here because of the constant value (291.15 K) of the temperature in all modifications due to the low coolant load. The case with a narrowing baffle is slightly less stable than other modifications, but on the other hand this type of baffle improves the efficiency of the cooling process in chillers by decreasing the temperature on the chiller_4_5 inlet (causing an increase of COP) and raising the coolant temperature on the outlet to chiller 1 caused by better coolant mixing. The temperature oscillations in the divider with narrowing are insignificant for chiller controllers due to their small magnitude (less than 0.15 K). It is worth mentioning that for this less loaded variant the reference case shows changes of temperature of about 0.8 K, which can be substantial considering the total change of the coolant temperature on the chillers is 6 K.

5.3 Variant III (80 l/s) In this hydraulic load system, one pump and three chillers are put into operation.

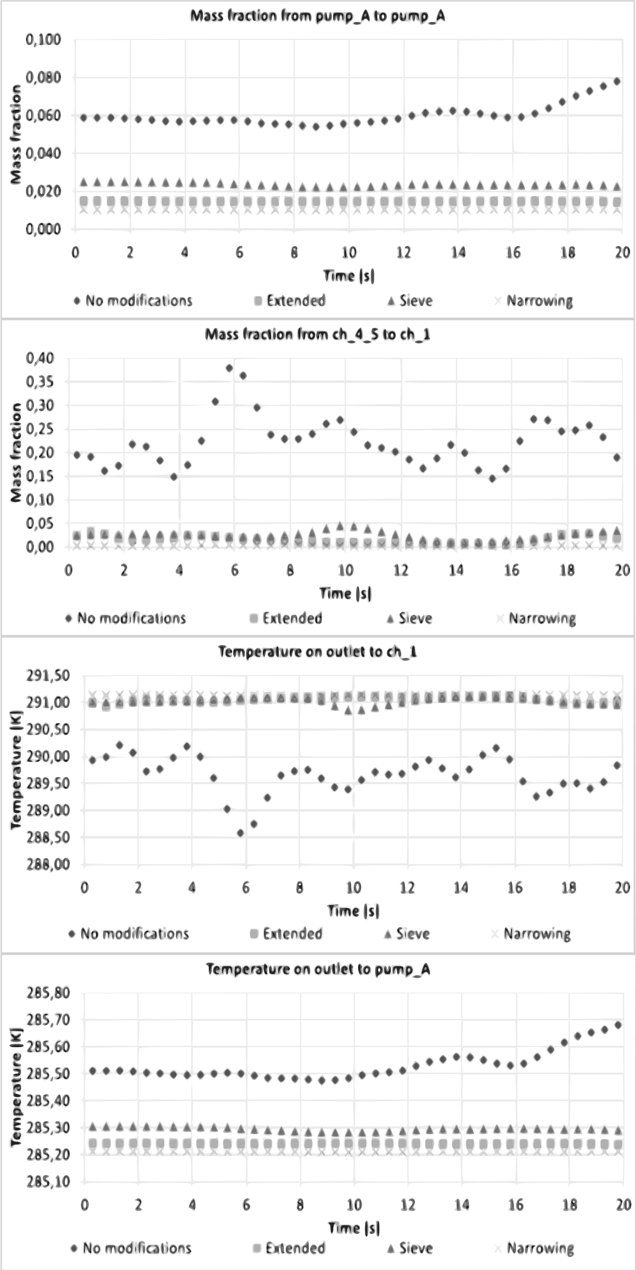

Figure 14: Changes of the mass fraction and temperature in Variant III (80 l/s)

Cooling load is 80 l/s and hence this is the most dynamic case. Simulation results are plotted in Fig. 14. The results of the simulation done on the existing divider design show that this regime of divider operation is unacceptable, because of the significant temperature gradient (1.5 K in 2 s) and fluctuations of the mass fraction on the outlet to chiller 1 which varies from 15% to 40%. The plots show that all geometry modifications may be treated as a solution, but the case with the narrowing provides the lowest temperature on outlets to pumps and the most stable coolant flow.

7. Conclusions

CFD analysis allows us to simulate coolant flow in a domain where it was impossible to install any measurement devices. This brought to light the reasons for certain anomalies. Summarizing the results of the simulations and considering various cases, the following conclusions can be outlined:

- The domain between chiller inlets and outlets is the space where mixing occurs and temperature gradient is the highest.
- The current fluid divider simulation indicated two problems occurring in its work: fluctuation in mass fractions at chiller inlets and coolant partially bypassing the chiller system. Both of them impact negatively on chiller efficiency.
- Higher fluid load causes stronger fluctuations, especially in the current existing geometry.
- In order to deal with the temperature front and to make it more stationary, a baffle in the central domain can be considered a practical solution. In this case, narrowing and sieve baffles cope very well with instabilities in the fluid. Extended geometry sometimes does not provide an acceptable solution.
- For the considered scope of cases, it was absolutely necessary to perform simulations of transients, as steady state simulation was not enough to select the optimal geometry.
- The narrowing baffle case provides the lowest temperature on outlets to pumps.
- The differences between the narrowing and sieve baffle were relatively small, but the geometry of narrowing is much simpler to manufacture so narrowing is the recommended modification.

As presented above, the problem is geometry-related and boundary-condition specific. In order to solve it, a geometry optimization procedure was necessary. CFD appearede to be the best tool for this purpose, as it can track flow behaviour at great resolution with no need to terminate operation of the cooling system. One needs to take into account that fluid divider termination means the computer cluster is expensive to shut down for hydraulic system maintenance and that the divider could also be difficult to handle as it needs to be maneuvered on the top of a building. Moreover, testing all the suggested designs in real life could take a long time, which is undesirable. In contrast, CFD tools can readily contribute to the process of designing the optimal shape. CFD analysis of a set of cases produces an optimal choice of design without shutting down of interfering with the system and is cheaper than building a comparable test stand. This recommendation enables the Management Board of CIŚ to make an informed decision as to the future course of action.

Acknowledgments

All the simulations that formed the basis for this paper were conducted thanks to EU and MSHE grant no. POIG.02.03.00-00-013/09—Świerk Computing Center (www.cis.gov.pl).

References

[1] D. Zgorzelski, P. Prusinski, T. Kwiatkowski, S. Koziol, Hydraulic divider geometry optimization with the use of cfd analysis tools, Tech. rep., National Centre for Nuclear Research, report NCBJ CIŚ Nr B –2/2015 (2015).

[2] D. Causon, D. Ingram, C. Mingham, A cartesian cut cell method for shallow water flows with moving boundaries, Advances in Water Resources 24 (8) (2001) 899–911. doi:http://dx.doi.org/10.1016/S0309-1708(01)00010-0.

[3] R. Kaczmarek, It's all about the mesh – a cartesian cut-cell approach (2013).
URL `http://bit.ly/2fUOFPB`

[4] W. Rahimi, H. nad Medjroubi, P. J., 2d and 3d numerical investigation of the laminar and turbulent flow over different airfoils using openfoam (2013).
URL `www.forwind.de/sowe/Site/Programfiles/SOWE2013Rahimi.pdf`

[5] D. K. Walters, D. Cokljat, A three-equation eddy-viscosity model for reynolds-averaged navier–stokes simulations of transitional flow, Journal of Fluids Engineering 130(20) (2008) 14. doi:10.1115/1.2979230.