A SIMULATION BASED THERMAL ANALYSIS OF A NEW CURRENT MONITOR AT PSI PROTON ACCELERATOR

Y. Lee*, P.-A.Duperrex, D. Kiselev and U. Müller, PSI, Villigen, Switzerland

Abstract

The Paul Scherrer Institute (PSI) operates a high power proton accelerator for a number of research projects in physics and medical sciences using protons, mesons and neutrons. Currently, a proton beam current of 2 mA with a beam power of 1.2 MW is routinely used. In the future, the upgraded injector and ring cyclotron will make a proton beam current of 3 mA possible. The enhanced beam power will generate higher thermal and mechanical loads to different components of the accelerator, such as collimators, targets and diagnostic monitors. In this paper, we present a simulation based thermal design of a new current monitor "MHC5" designed to sustain the 3 mA beam operation.

INTRODUCTION

During the 2009 shut down, a new current monitor "MHC5" has been installed for the high power proton accelerator at PSI, replacing an old prototype. The new MHC5 consists of a coaxial resonator, as shown in Fig. 1. It is located on the beam axis, approximately 8 m downstream of the 4 cm thick graphite target for muon and pion production. Thus, the current monitor is exposed to scattered particles and their secondaries from the graphite target. As main improvements, an active water cooling system and surface blackening have been newly applied, in order to sustain the severer thermal load from the enhanced beam power of 1.8 MW.

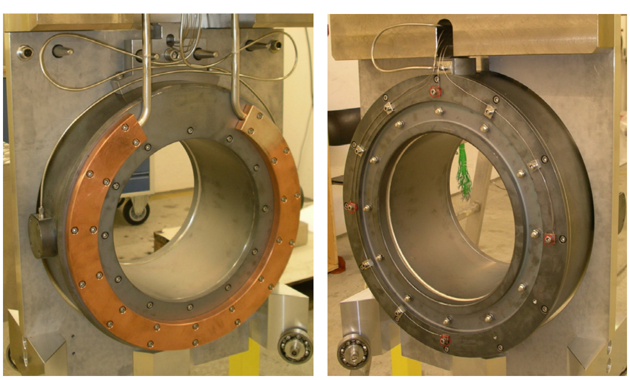

Figure 1: The new current monitor. The water channel is shown at the beam entry side (left). The four thermocouples are installed on the beam exit side (right), which are marked by four red circles.

In this paper, we present a simulation based study of the thermal characteristics of the new current monitor MHC5. The applied simulation methods include the coupled flow and heat simulations based on a second order accurate finite volume scheme and the Monte Carlo simulations for particle transport. The coupled flow and heat simulations have been performed by the multiphysics commercial tool CFD-ACE+ [1]. In order to validate the flow and heat simulation setting, a laboratory test has been performed to measure the water cooling efficiency. In particular, the transient behavior of the surface temperature at the chosen thermocouple locations due to water cooling has been measured, for different water speeds. The onset of fully developed turbulence flow for optimal heat exchange has been measured and then compared with the simulation results.

In order to predict the thermal energy deposition rate in MHC5, the Monte Carlo particle transport codes, MCNPX [2] and MARS [3] have been used. The Monte Carlo simulations have provided the volumetric heat source for the coupled flow and heat simulations, for the prediction of the operating temperature of MHC5.

In addition, the effect of surface blackening on the transient temperature rise delay has been investigated, in the absence of water cooling. The blackening of the monitor provides an additional safety margin for the critical operating temperature in case of a water cooling system failure.

SIMULATION METHOD VALIDATION FOR WATER COOLING EFFICIENCY

Experimental Setup

In order to exclusively investigate the effect of water cooling, the new MHC5 has been thermally isolated from the external thermal influences (convective and conductive cooling by air), using a case filled with fiber glass wool. The inlet water has been first heated up to the temperature near 80 ^{o}C by a 10 kW heater. After the current monitor reaches the equilibrium temperature near 80 ^{o}C, the water heating is stopped and the 30 ^{o}C cooling water is switched on. This process has been repeated for different inlet water velocities.

Eight K-Type thermocouples have been used for transient temperature measurements. Two have been used for the measurement of inlet and outlet water temperatures. Two have been placed on the copper plate providing the thermal contact between the cooling tube and the MHC5 body. The last four thermocouples have been placed at the exact thermocouple positions foreseen for the temperature monitoring during beam operation; see Fig. 1. The absolute accuracy of the used thermocouples is better than ± 2.2 ^{o}C.

After the cooling water inflow and subsequent initial transient phase, the temperature at the four monitoring

* yong-joong.lee@psi.ch

points at MHC5 has been observed to decrease exponentially in time, with a time constant τ. This is due to the fact that the wall heat exchange coefficient of the water flow at the cooling pipe surface is virtually a function of the water velocity or the Reynolds number only. The time constant could be estimated by fitting the experimentally obtained transient temperature data with the analytic formula given by

$$T(t) = T_{\rm w} + (T(t = t_0) - T_{\rm w}) \exp\left[-\frac{(t - t_0)}{\tau}\right]. \quad (1)$$

Here, $T_{\rm w}$ is the ambient water temperature and the time constant τ is a function of the thermocouple position, the heat conductivity of the current monitor and the wall heat transfer coefficient at the cooling pipe inner surface.

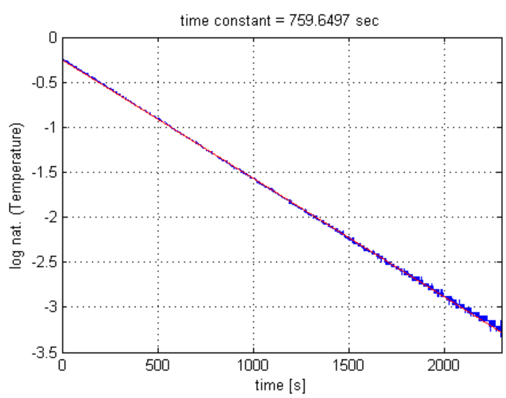

Figure 2: Semilog plot of the cooling down (blue) and exponential regression curve (red) for the time constant estimation, for the water speed of 4m/s.

Figure 2 is an exponential fitting of the measured transient temperature for the water speed 4 m/s, for example. The fit is excellent, and the time constant obtained from the graph is approximately 760 sec. Estimation of the time constants by fitting the experimental data has been done for each tested water speed and for the four temperature sensors.

Simulation Setup

The experimental setting has been built into the CFD (Computational Fluid Dynamics) simulation model which is based on a second order accurate finite volume flux scheme. The used simulation tool is CFD-ACE+[1]. For accurate calculation of the heat exchange rate between MHC5 and water, the shear stress transport (SST) $k-\omega$ turbulence model [4] has been used. The mesh in the boundary layer region has been refined until the desired value of the wall function is achieved.

Evaluation of Simulation with Test Data

Figure 3 compares the simulated and measured time constant τ, which shows the quality of the prediction power of

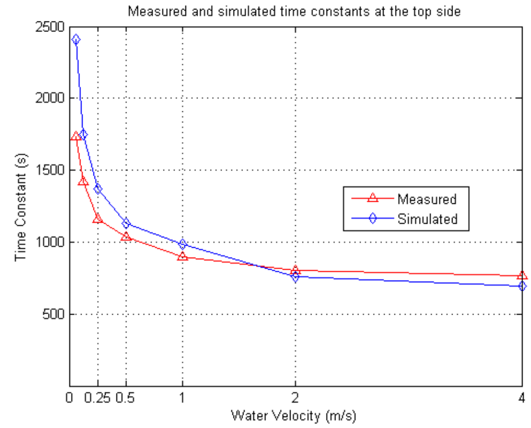

Figure 3: Comparison of measured and simulated time constants at the thermocouple located on the top side of MHC5.

the numerical simulation. The simulated and the measured time constants agree within 10 %, for the water speed larger than 0.5 m/s. There is relatively large disagreement in values, for the water velocities below 0.25 m/s. One reason is that the simulation setting has used the turbulence module which might describe the laminar flow region with low water speed inaccurately. Another reason is that there have been more uncertainties in accurately controlling the water velocity in the lab tests, for small mass flux. Nevertheless, the CFD simulation has captured the observed feature that the cooling efficiency increases with the inlet water speed.

For reference, the transition from the laminar to the turbulent flow begins to occur at around 0.25 m/s, inside the water pipe with diameter 10 mm. Then, the fully developed turbulent internal flow is formed at a water speed of approximately 0.4 m/s or a Reynolds number of $4.0 \cdot 10^3$; see Ref. [5]. Once the turbulence is fully developed, the improvement of the cooling efficiency with increasing water inlet speed is marginal. This phenomenon has been well reproduced by the simulation and the measurement. Above the water speed of 2 m/s no significant improvement of the cooling is to be expected.

MONTE CARLO SIMULATIONS FOR ENERGY DEPOSITION

In order to predict the thermal load, the proton energy deposition rate in the current monitor MHC5 should be calculated. For this purpose, Monte Carlo (MC) simulations have been performed, using MCNPX [2] and MARS [3]. The geometry model used for the MC simulations are shown in Fig. 4. In the figure, TgE is the 4 cm thick graphite meson target, C0, C1, C2 and C3 are collimators, and QHG 21 & 22 are the two magnetic quadrupoles for the focusing of the proton beam.

In MCNPX, the effect of the magnetic quadrupoles QHG 21 & 22 could not be taken into account, due to the lack of magnetic modules. In MARS, the quadrupole induced

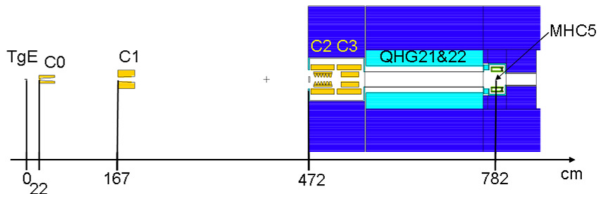

Figure 4: The geometry model for the MC simulations. The proton beam enters from the left.

beam deflection has been taken into account. The MCNPX and the MARS calculations respectively predicted 562 W and 345 W energy deposition rate in MHC5, for 3 mA proton beam current. The estimated value of MARS is smaller, which can be explained as follows. In MCNPX, the Coulomb scattering at the meson target has been observed to be stronger, which caused more widely scattered beam thereafter. In MARS, the beam was more focused due to the presence of quadrupole effect, and the current monitor was less exposed to scattered particle shower.

PREDICTION OF THERMAL CHARACTERISTICS

Operating Temperature with Water Cooling

From the tests and simulations for the efficiency of the water cooling, the water inlet flow of 2 m/s has been determined. For the prediction of the operating temperature of MHC5, we have performed the coupled flow and thermal simulations, for the energy deposition rate given by MCNPX. Figure 5 shows the simulated temperature distribution, for the future proton beam current 3 mA. The hot spot temperature is shown to be kept below 90 ^{o}C. This is smaller than the empirically set thermal failure temperature criterion of maximum 180 ^{o}C. For the energy deposition input from MARS, the peak temperature has been linearly scaled to be below 70 ^{o}C.

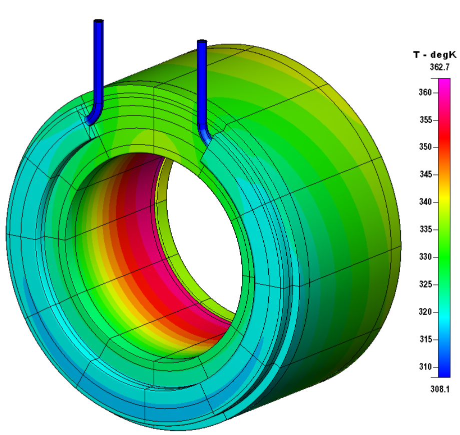

Figure 5: The simulated temperature profile of MHC5, for the proton beam current 3 mA.

Effect of Emissivity in Case of Water Cooling Failure

If the cooling water, for some reason, stops flowing, the temperature of the current monitor MHC5 will rise to the equilibrium temperature where the radiative heat transfer balances the energy deposition rate. For this case, the surface emissivity plays an important role. Therefore, the surface of the new MHC5 has been treated with silver and sulfur, raising the spectral emissivity from less than 0.05 of the untreated shiny surface up to 0.2.

The transient simulations for different emissivity have been performed for different surface emissivities, in case of water cooling failure. With the new surface treatment, the peak equilibrium temperature of MHC5 is expected to drop approximately by 132 ^{o}C, from 454 ^{o}C to 322 ^{o}C. If the water is assumed to be perfectly incompressible, the water temperature will rise to a boiling point within approximately 10 minutes after water blocking, which can cause serious damage to the accelerator system. Indeed, water is slightly compressive with a non-zero temperature dependent thermal expansion coefficient $\mathcal{O}(10^{-4})$ K^{-1}. This buoyancy effect has been also investigated by a CFD simulation. The heated water is driven upward by the buoyancy force, causing water flow in the channel with the speed $\mathcal{O}(10^{-1})$ m/s. This should significantly delay the time until the water reaches the boiling point.

CONCLUSIONS

The simulation based study of the thermal characteristics of the new PSI current monitor "MHC5" has been presented. The used CFD simulation method has been well validated by a lab test on cooling efficiency. The simulation and the validation test have shown that the operating cooling water speed 2 m/s for the pipe diameter 10 mm is optimal. Using CFD and Monte Carlo simulations, the operating temperature of the new MHC5 has been predicted to be below 90 ^{o}C for 3 mA proton beam operation, satisfying the engineering requirement of staying below 180 ^{o}C. In order to realize an additional safety margin in case of water cooling failure, blackening of the MHC5 surface with a silver and sulfur treatment has been done and its effect analyzed. The validation for the Monte Carlo simulations for the energy deposition rate in the current monitor is to be done once it is in stable operation.

REFERENCES

[1] CFD-ACE+, `http://www.esi-group.com`.

[2] D. Pelowitz, ed., "MCNPX User's Manual," Version 2.5.0, LA-CP-05-0369

[3] N. V. Mokhov, C. C. James, "The MARS Code System User's Guide," Version 15, 2007.

[4] F. R. Menter, "Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows", AIAA Paper 93-2906.

[5] J. P. Holman, "Heat transfer," McGraw-Hill, 2002.