EUROPEAN ORGANIZATION FOR NUCLEAR RESEARCH
ORGANISATION EUROPÉENNE POUR LA RECHERCHE NUCLÉAIRE

CERN - ST Division

CERN-ST-2001-014
30 January 2001

TARGET COOLING FOR THE TOF EXPERIMENT

D. Gasser

Abstract

In the TOF experiment, a high-energy proton beam is sent on a lead target, which releases an important amount of heat inside. In order to avoid that the target temperature rises over the lead melting point, a cooling system was designed. Starting from the heat distribution generated inside the target by the proton beam, steady and unsteady simulations of the target thermal behavior were carried out. These simulations were performed with different ways of cooling and in the various beam configurations foreseen for the experiment. It will be explained how these simulations helped to design a suitable water cooling system for the target. The hydraulic circuit providing the cool water will be briefly described as well.

Presented at the 4th ST Workshop
Chamonix, France, 30 January - 2 February 2001

1 INTRODUCTION

The design of innovative Accelerator-Driven Systems for incineration of nuclear waste, energy production and radio-isotope activation for medical applications as well as many other subjects in Nuclear Physics, requires an experimental determination of neutrons cross sections, elastic and inelastic across the whole Mendeleiev table.

The TOF facility aims at study systematically and with excellent resolution, neutron cross sections of almost any element.

In the TOF experiment, a proton beam is sent on one side of a lead target. The spallation mechanism, taking place inside the lead, produces a neutron source on the other side of the target. The neutrons are then canalized through a vacuum pipe to an experimental area located 200 m downstream of the target.

Because of its high energy, the proton beam releases an important amount of heat inside the lead target. In order to avoid that the target temperature rises over the lead melting point, a cooling system for the target was designed.

After a short description of the experiment (target, proton beam, heat power released), it is explained how the target cooling system was developed. The hydraulic circuit providing the cooled water for the target is briefly described at the end.

2 THE LEAD TARGET IN TOF EXPERIMENT

2.1 Description of the target and the proton beam

The target is a cube with an alcove on one of its face (see

Figure **1**). The proton beam goes along the Z-axis and hits the lead inside the alcove. A neutron source is then produced on the opposite side.

Looked from the top, the proton beam has a 10° angle relative to the Z-axis, in the YZ plane. The energy of the beam is 20 GeV/c.

The proton beam consists of a sequence of bunches sent onto the target with a certain period of time, called a supercycle. The time length of a supercycle is 14.4 seconds. Each bunch contains 7.10^{12} protons and is emitted during a very short time of 6 nanoseconds. Two beam configurations are foreseen:

- 1 bunch per supercycle (i.e. 1 bunch every 14.4 s),
- 4 bunches per supercycle (i.e. 4 bunches every 14.4 s at the time 0, 1.2, 2.4 and 3.6 second).

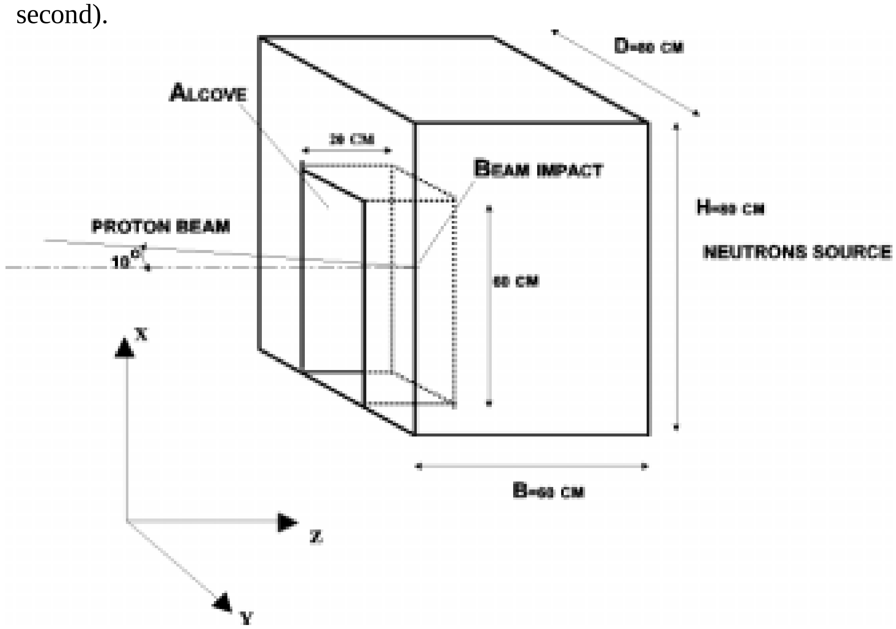

Figure 1: target geometry

2.2 Heat released inside the target

A neutronic code allows to calculate the energy released inside the target by the proton beam. This energy Q is given in Gev/cm^3/protons. Knowing the number of proton per bunch (np = 7.10^{12}), the corresponding volumetric heat power P (in W/m^3) released inside the target can be calculated as:

$$P = \frac{Q * np * 1.6 \cdot 10^{-19} * 10^{9} * 10^{6}}{t} \qquad \textbf{(1)}$$

Where $1.6.10^{-19}$ is the energy in Joules of 1 eV, the factor 10^9 converts the Gev in eV and the factor 10^6 converts the cm^3 in m^3.

P can be averaged, in first approximation, over one super-cycle (t = 14.4 s) or given as an instantaneous figure for each bunch (t = 6.10^{-9}s).

It can be seen in Eq. (1) that the ratio $\frac{np * 1.6 \cdot 10^{-19}}{t}$ corresponds to the beam intensity Ib, which yields :

$$P = Q \cdot Ib \cdot 10^{15} \qquad \textbf{(2)}$$

3 DEVELOPMENT OF THE TARGET COOLING

3.1 First solutions foreseen

The very first solution foreseen to cool down the target was to use ambient air free convection. Therefore, a thermal simulation of the target was performed with StarCD[1] considering an ambient temperature of 20^o C at the lead surface and a heat transfer coefficient with the air of 5 W/m^2/K which is a standard value for free convection. The volumetric heat power P applied inside the target is averaged over one super-cycle (Eq. (1) with t=14.4s). This means that the simulation is done with a steady state approach, which is enough to get a first estimation of the temperature figures. The calculation was performed in the configuration of four bunches per super-cycle (np=$4*7.10^{12}$); this corresponds to a total power released inside the target of about 3200W.

The temperature distribution arising from this simulation is shown on Figure 2 on a YZ plane cutting the center of the target. The beam impact is located on the left side in the center of the alcove. The temperature iso-contours show clearly the 10 degrees inclination of the beam. Moreover, the iso-contours are almost perpendicular to the lead surface because of the small heat exchange with air. The hottest spot is located inside the target a few centimeters after the impact. The temperature inside the target remains in the range between 200 and 315^o C which is too much knowing that the lead melting point is around 320^o C. Indeed, the lead may get soft even around 200^o C, which is not compatible with the experiment. The free convection cooling is therefore not enough.

[1] StarCD is a commercial Computational Fluid Dynamic code, developed by Computational Dynamics Ltd.

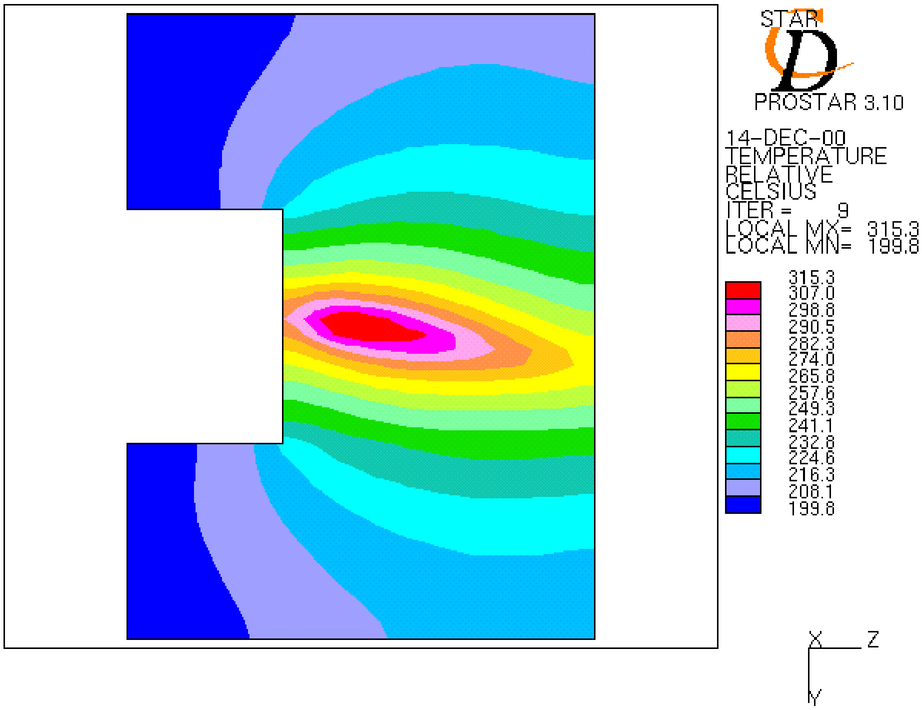

Figure 2: temperature distribution inside the target (air free convection cooling)

Another simulation was done with an air forced convection around the target (i.e. heat transfer coefficient of 20 $W/m^2/K$). The temperature figures arising from this simulation were still too high.

3.2 Dimensioning of the water cooling

The coolant finally chosen for the target is demineralised water which, thanks to its important heat capacity, is able to remove the rather big amount of heat generated inside the lead (3200 W). The practical solution is to immerse the target in a tank full of water and to provide a circulation inside, with two main streams, one along the side with the alcove (proton side) and one along the opposite side (neutron side). The reason for this is that on the first simulation (Figure 2), it can be noticed that these two sides are the hottest ones and has to be cooled down with the best efficiency in comparison with the other faces of the target. As it can be seen on Figure 3, five holes are dig on the target tank, on each side of the two walls perpendicular to the Yaxis. Pipes are connected to the holes in order to bring the water into the tank through the inlet holes and bring the water out through the outlet holes, and this with a constant flow rate.

In order to provide a good heat transfer coefficient along the two main streams, the water flow rate has to be high enough in order to ensure a turbulent flow. This flow is assumed as an internal one inside a pipe of rectangular section L x w, where L is the length equal to the target height (H=80 cm see

Figure **1**) and w is the width equal to the water thickness between the target wall and the tank. The influence of the alcove, on the proton side is neglected in first approximation. The width w is 3 cm on the proton side and 5 cm on the neutron side. These thickness figures were chosen mainly for physical reasons, the water acting as a moderator for the neutrons as well.

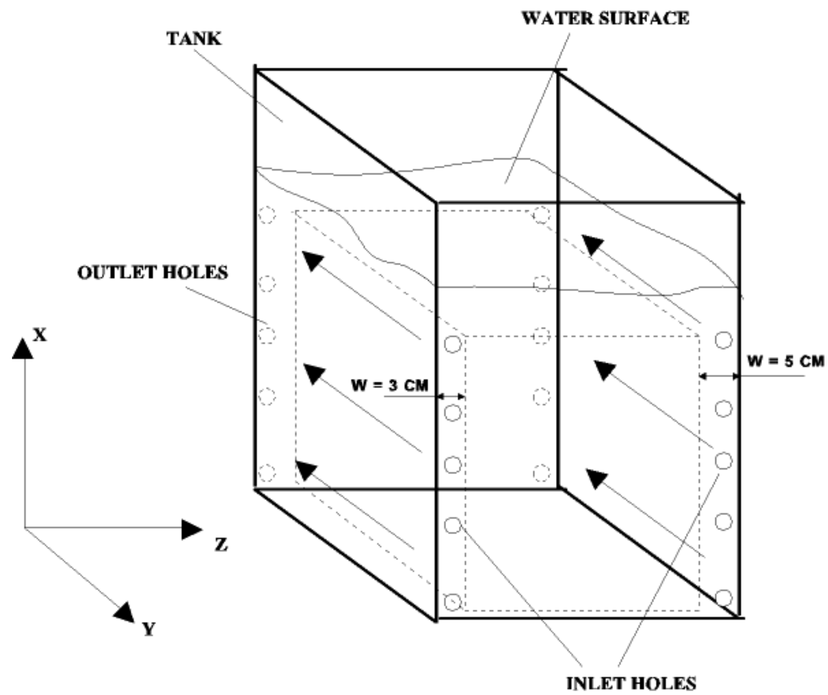

Figure 3: water cooling of the target

The Reynolds number of the flow along the two main streams is given by :

$$\frac{V \cdot Dh}{\nu} \qquad \textbf{(3)}$$

where ν is the water kinematic viscosity (1.1 10^{-6} m^2/s^2) , V the velocity and Dh the hydraulic diameter given by $\frac{4 \cdot L \cdot w}{2 \cdot (L + w)}$. In order to have a well established turbulent regime for both streams, the Reynolds number has to be over 6000, which give V=0.125 m/s on the proton side (w = 3 cm) and V=0.075 m/s on the neutron side (w=5 cm). This makes a total flow rate q on the two sides of 6 l/s.

The heat transfer coefficient h between lead and water can then be estimated from nusselt number, i.e.: $h = \frac{Nu \cdot \lambda}{Dh}$ where λ is the water heat conduction coefficient (0.598 W/m/K). The Nusselt number is given by the following formula [1]:

$$Nu = \frac{\frac{f}{8} \cdot (\mathrm{Re} - 1000) \cdot \mathrm{Pr}}{1 + 1.27 \cdot \left(\frac{f}{8}\right)^{0.5} \cdot \left(\mathrm{Pr}^{\frac{2}{3}} - 1\right)} \qquad \textbf{(4)}$$

Where Pr is the Prandtl number and f the friction factor at the lead water interface estimated from the following formula [2]:

$$f = \frac{0.25}{\log\left(\frac{\Delta}{Dh \cdot 3.7} + \frac{5.74}{Re^{0.9}}\right)^2} \quad \textbf{(5)}$$

where Δ is an average roughness equal to 0.046 mm.

h is estimated to about 1000 W/m^2/K for both streams, which is a good figure and should ensure an efficient heat transfer at the lead/water interface.

3.3 Simulation of the water cooling

In order to check that the water cooling solution is working, StarCD simulations of the target cooled down by two water streams were performed. The model (mesh and boundary conditions) used for these simulations is shown on Figure 4. The model takes into account the fluid (water) and the solid (lead) with the heat transfer between both media.

The black cells correspond to the two water streams cooling down the proton and neutron sides, with the inlet velocity in yellow ; the inlet temperature is set to 30^{o}C which is the figure than can be achieved in summer ; in green are represented the two water streams outlets. The two water streams are longer than the target for simulation purpose, in order to get a permanent flow regime all along the sides of the target. Moreover, the alcove is taken into account in this model, it generates in the simulation a flow vortex taking place inside the alcove.

In blue are represented the four remaining sides of the target cooled down by air free convection with a heat transfer coefficient of 5 W/m^2/K and an ambient temperature identical to inlet water temperature (30^{o}C). This model does not fit to the exact reality because the four sides are actually cooled by water since the whole target is immersed in water. But it has to be considered that the water flow velocity is very low along these four sides with a small heat transfer coefficient. This situation is approximated as an air free convection cooling. So, the simulation is done in the worst possible situation in order to have a kind of safety margin.

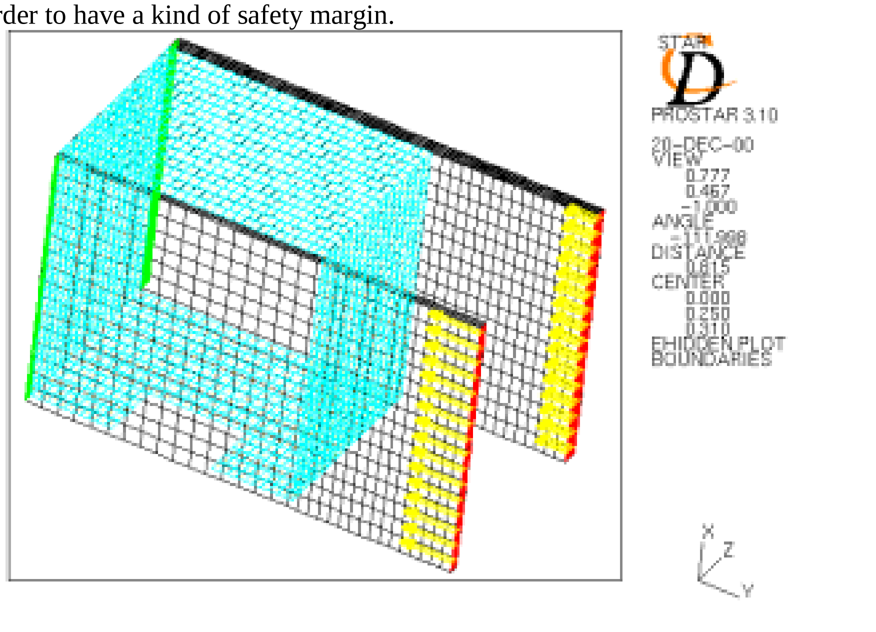

Figure 4: mesh and boundary conditions

The two first simulations were done on a steady case (volumetric power P averaged over one super-cycle), for the two beam configurations: one bunch and four bunches per super-cycle. The temperature distributions inside the target, arising from these two simulations are shown on Figure 5 and Figure 6 on a YZ plane. The maximum temperature reached inside the target is 56^{o}C for one bunch per super-cycle and 135°C for four bunches per super-cycles. These two figures remain far above the melting point. At 135°C, the target might get softer in the center but this phenomenon is limited to the center of the cube and should not affect its stability. On the lead surface, the temperature do not rise over 70°C thanks to the important heat transfer coefficient (1000 W/m^2/K) provided by the water flow. The lead surface should not get soft at such a temperature.

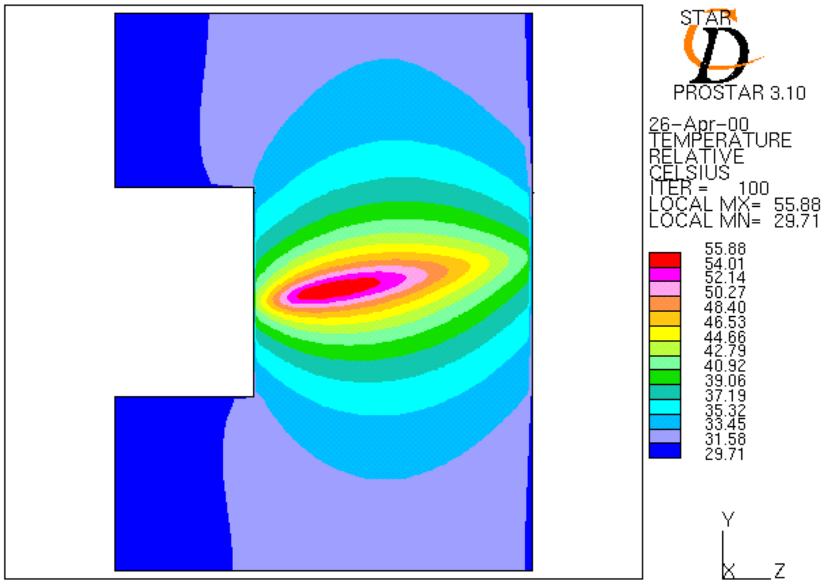

Figure 5: temperature distribution inside the target with water cooling (steady case, 1 bunch per super-cycle).

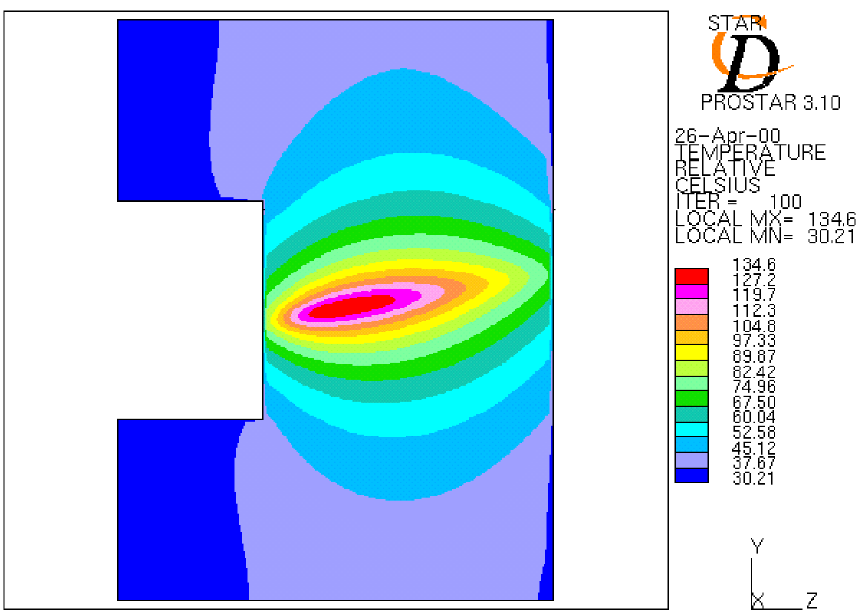

Figure 6: temperature distribution inside the target with water cooling (steady case, 4 bunches per super-cycle).

The two previous simulations, performed in a steady case, gave a good evaluation of the temperature figures.

But the heat power is actually released instantaneously at each bunch (during 6 nanoseconds), which means that the target heating is unsteady. This unsteady effect should have a minor influence on the temperature distribution in comparison with the steady case. Nevertheless, two unsteady simulations were carried out for the two beam configurations. The simulation were performed over one super-cycle (14.4 s), the heating power being applied during 6 nanoseconds (eq. (1) with $t=6.10^{-9}$ second):

- at time 0s for the first beam configuration,
- at times 0, 1.2, 2.4 and 3.6 s for the second beam configuration.

In fact, the simulations were done over several super-cycles. Indeed, the target first gets hotter through a transient regime and then reaches a permanent unsteady regime. In order to reach faster this regime, the initial conditions applied were those arising from the steady simulation. The permanent unsteady regime was reached when, for one super-cycle, the temperature distribution inside the target at time 0 and 14,4 s were identical. The results of these two simulations show that the temperature iso-contour have the same shape as in the steady case and that the lead surface do not rise over 70°C, except on a very small area, around the beam impact point, where the temperature reached 100°C only the time of a bunch (6 nanoseconds).

Figure 7 shows the time evolution of the maximum temperature reached inside the target during one super cycle, in the case of four bunches (the time axis scans two super-cycle). It can be seen that the temperature reaches 175°C at the fourth bunch, but the average value over one super-cycle corresponds to the maximum temperature got in the steady case (see Figure 6).

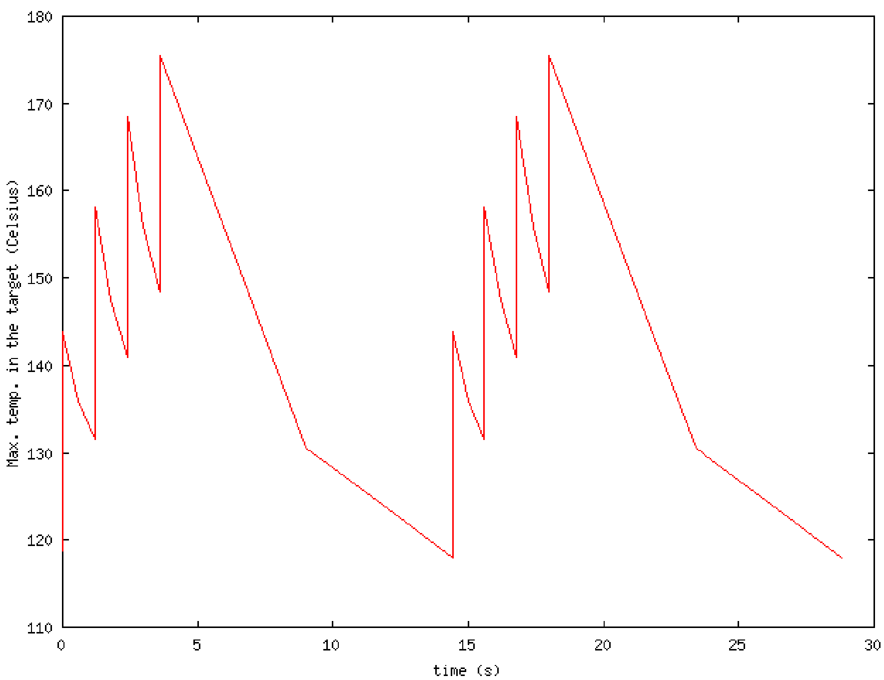

Figure 7: time evolution of the maximum temperature in the target (4 bunches)

The TOF experiment has been commissioned in the configuration of one bunch per super cycle. Some thermocouples were installed inside the lead, and especially one in the center of the target. Figure 8 compares the time evolution of the temperature measured at this point, with the estimation done by StarCD. What is worth comparing between the two curves are the temperature amplitude at each bunch and the average temperature level over one super-cycle. The temperature amplitude measured by the thermocouple is about 6°C and the one estimated by the calculation is 4°C. The averaged temperature level estimated by StarCD is higher of 15°C in comparison with the measurement, which makes sense since the simulation is done in an unfavorable case (see section 3.3) with an overestimated thermal charge on the target. Nevertheless, it must be mentioned that the measurement where done in winter when the water temperature provided by the cooling system is about 20°C, in summer, this temperature is about 30°C, which should reduce of 10°C the shift of 15°C observed between calculation and measurement.

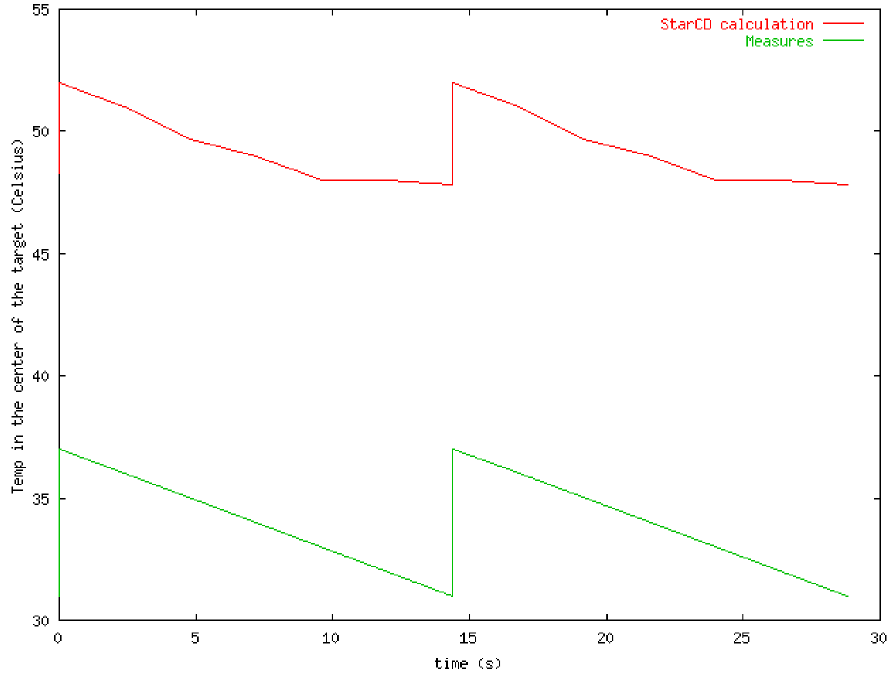

Figure 8 : Time evolution of the temperature in the center of the target (1 bunch) – Comparison with measures.

4 OUTSIDE HYDRAULIC CIRCUIT FOR THE TARGET COOLING

A hydraulic circuit, feeding the target tank with cooled demineralised water, was built in the TOF experimental zone. The principle of this circuit is described on Figure 9. The secondary loop cools down the target, and the heat is removed by a heat exchanger, which is connected to the primary water network. This network is itself connected to the cooling towers. In the secondary loop, the variable velocity pump ensures a constant flow rate (6 l/s) through a PID loop controlled by a set point measured by the flow transmitter (FT). On the primary circuit, the regulating valve (RV) ensures a constant temperature at target tank inlet through a PID loop controlled by a set point measured by the temperature transmitter (TT).

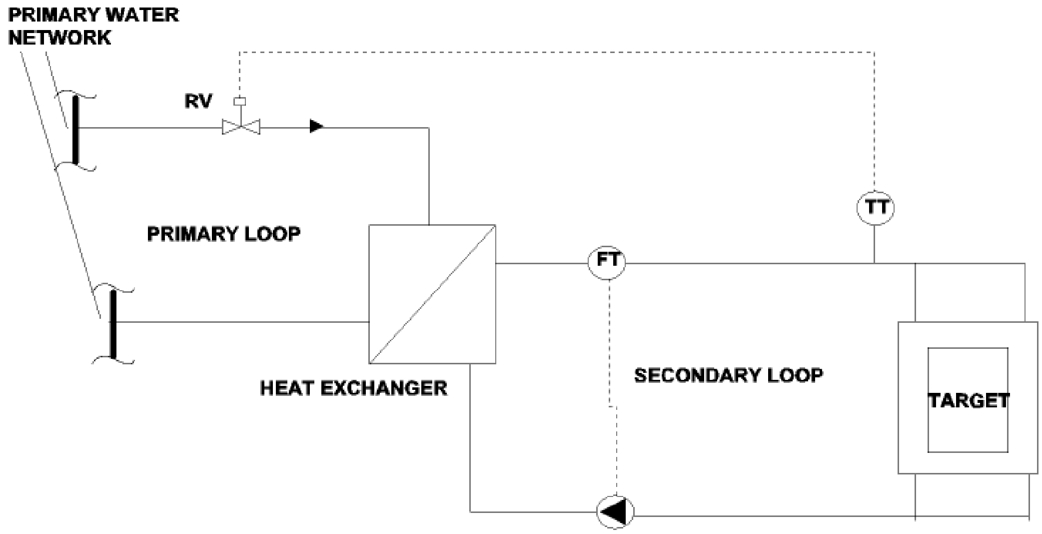

Figure 9: cooling circuit principle

5 CONCLUSION

A cooling system was designed for the target. It consists of providing a flow circulation inside a tank, where the whole target is immersed. It was shown how the simulations of the target thermal behavior helped to define the system, mainly the flow rate figure and the way of ensuring the flow circulation around the target.

The measurements of the temperature reached inside the target during the experiment confirmed that the simulation did not give an underestimation of the temperature loads, which would have led to wrong a design.

The target and the tank were installed in the TOF experimental zone; an outside hydraulic circuit, with a heat exchanger, feeding the tank with cool water was built as well.

REFERENCES

[1] Fundamentals of Heat and Mass Transfer, Franck P. Incropera, David P. DeWitt, Wiley edition, fourth edition

[2] Mechanics of Fluids, Irving H. Shames, Mc Graw-Hill international edition, third edition