ISBN: 978-3-95450-227-1 ISSN: 2673-5490 doi:10.18429/JACoW-IPAC2022-TUPOMS022

COOLING CHALLENGES IN A NEG-COATED VACUUM CHAMBER OF A LIGHT SOURCE

S. Talebi Motlagh[†,1,2], A. Danaeifard[1], J. Rahighi[1], F. Saeidi[1], F. Zamani[2]
[1]Iranian Light Source Facility Institute for Research in Fundamental Sciences (IPM), Tehran, Iran
[2]University of Kashan, Kashan, Iran

Abstract

In a Light Source, unused synchrotron radiation is being distributed along the walls of the chambers. Due to the small conductance of the chambers, vacuum pumping will be based on the distributed concept, and then non-evaporable getter (NEG) coating is extensively used. The vacuum chambers are made of copper alloys tube, and cooling circuits are welded to the chamber to remove the heat load from the radiation generated. Filler metal creates a brazed joint between the water-cooling pipe and the vacuum chamber body. The thermal conductivity of the fillers is less than the vacuum chamber body. On the other hand, the velocity of the water in the cooling pipe is a critical parameter in thermal calculations that must be taken into account. So, in this paper, we study and investigate the effects of the filler metal and the cooling water velocity on cooling the NEG-coated chambers.

INTRODUCTION

The ILSF storage ring lattice is based on 20 five-bent achromats; Each achromat contains three unit cells and two matching cells. The unit cells have a 3.9° bending magnet, while the matching cells deflect the beam 3.15° [1]. The radiation of these five dipole magnets is uniform in the horizontal direction (plane of the storage ring), while in the vertical direction, it follows a narrow Gaussian profile.

The irradiate power and the radiation power density on the vacuum surface are simulated by Synrad+ software [2]. It is assumed that the chamber body will absorb all the radiation, and no reflection would happen. Hence, the facets' sticking factor has been considered equal to 1. The cooling of this chamber and its challenge will be studied in the present work.

The commonly used thermal absorber design criteria are [3]:

1. The maximum cooling wall temperature T_{Max}^{CW} should be lower than water boiling temperature $T_{boiling}$ at the pressure of the water in the cooling tubes
2. The maximum temperature of the chamber T_{Max}^{Ch} must be significantly lower than the melting point of the copper and the brazing temperature.
3. The maximum temperature rise in the chamber should be less than 300°C for Glidcop and 150°C for oxygen-free high thermal conductivity (OFHC) copper, which are also used at APS [4].

THEORY

The total radiative power of all the bending magnets in a ring is determined by the electron's energy $E\,[GeV]$, the bending magnet's field $B\,[T]$, and the electron beam current $I_b\,[mA]$, as the following equation [5]:

$$P_{total}[kW] = 26.6E^3[GeV]I_b[A]B[T] \quad (1)$$

Correspondingly, the power density on the beam axis can be obtained as follows:

$$\frac{dP}{d\Omega}\left[\frac{W}{mrad^2}\right]|_{\psi=0} = 5.44E^4[GeV]I_b[A]B[T] \quad (2)$$

Since the magnitude of the magnetic field in the dipoles of the ILSF storage ring is about $B = 0.567\,T$ according to Eq. (2), at a current of 400 mA and an energy of 3 GeV, the total output power due to synchrotron radiation will be 162.95 kW. It means that for 100 dipole magnets in the ring, each bend chamber wall receives 1.05 kW radiation power.

A maximum value of 8.35 W/mm^2 for the power density has been obtained analytically.

Most of the power will be distributed along the water-cooled vacuum chambers. So, a cooling circuit channel is welded to each chamber to remove the heat load from the radiation generated as seen in Fig. 1. In this research, a simple model of a curved vacuum chamber in a dipole without photon extraction is studied to investigate the thermal effects, as seen in Fig. 2. Although the electrons travel parallel with the chamber in a curved path but the radiant photons go in a straight line. So, significant radiation collisions will be happen along the second half of the chamber. Therefore, the initial part of the chamber will have an ambient temperature, while the final part will be warm due to the radiation, as discussed here.

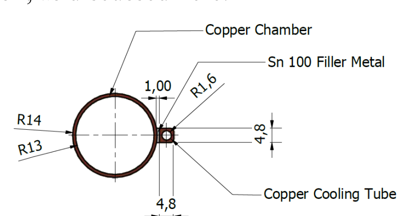

Figure 1: The ILSF chamber cross-section view.

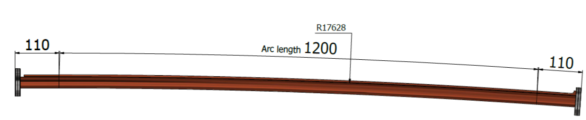

Figure 2: Simple model of the vacuum chamber in a dipole without photon extraction.

† s.talebimotlagh@ipm.ir

ISBN: 978-3-95450-227-1 ISSN: 2673-5490 doi:10.18429/JACoW-IPAC2022-TUPOMS022

The vacuum chambers of the ILSF storage ring will be fabricated using an oxygen-free copper alloy. The small amount of silver in this alloy helps the chamber increase its resistance to softening.

The velocity of water in the cooling pipes should be less than 3 m/s to keep the flow-induced vibrations within acceptable levels and larger than 1 m/s to avoid the accumulation of air in the circuits.

A filler metal (such as SN100C) will be used to create a low-temperature brazing joint between the water-cooling pipe and the vacuum chamber body. The thermal conductivity of this alloy is less than the conductivity of copper. The thermal conductivity coefficient of OFS copper is 388 $W/m°K$ while the thermal conductivity coefficient of SN100C is 64 $W/m°K$. A thickness of 1 mm against brazed filler metal has been considered in the thermal simulations, which is a strict assumption. Nevertheless, it will be sure that it will not encounter any problems in practice. The effect of this filler conductance has been studied in this work.

SIMULATIONS

Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) model is based on the fundamental equations governing fluid dynamics: mass, momentum, and energy conservation. CFD helps anticipate fluid flow behavior based on mathematical models using software tools. This method is widely used and accepted as a proper engineering tool in the industry. The CFD simulation process involves several different steps involved in fluid flow analysis.

Finite Element Analysis (FEA)

FEA is a famous method to solve numerically differential equations moving up in engineering and mathematical modeling. The FE*A* subdivides an extensive system into smaller, simpler parts called finite elements to solve a problem. The thermal analysis of the presented chamber has been performed based on FEA. The ANSYS software [6] is used to simulate the thermal distribution of the chamber body. The result of the thermal analysis will be used for stress analysis in the next step.

The heat generated on the vacuum chamber's inner surfaces is transferred to the water that passes through the cooling pipe by the convection process. Several quasi-experimental equations are available to calculate the convective heat transfer coefficient from the pipe wall to the water. Since the heat is transferred from the hotter body to the water pipe walls, the heat absorbed is approximately equal to the radiation heat.

In FEA, cooling is entirely defined by the convective heat transfer coefficient and water temperature.

CFD modeling has not been commonly used in the past for absorber cooling calculations in the synchrotrons. In Ref [7] authors refer to a heat transfer film coefficient value that varies from 10 to 20 $W/m^2°K$ with no or little precisions concerning the cooling channel dimensions and the flow rate.

For our case, it is decided to study a simple model of a cooling channel with a water flow rate (1-3m/s), to extract the maximum and average heat transfer coefficient in the cooling tube wall. Also, the maximum and average temperatures of the chamber body are interesting to compare with FEA results. All CFD simulations were made using the k-ε turbulence model [8], and then it is compared with the results of FEA.

Heat transfer coefficient h will be calculated by ANSYS from the equation $q'' = h(T^{C\,W} - T^{\infty})$ where q'' is the heat transfer rate, $T^{C\,W}$ is the cooling wall temperature and T^{∞} is the temperature of the reference. The cooling calculations using CFD have been done with three different flows. In this method, it is assumed that the inlet water temperature is 25 degrees. Heat flux due to synchrotron radiation colliding with the inner wall of the vacuum chamber was calculated using Synrad+ software. The cooling tube was hypothesized to be brazed into the vacuum chamber with a filler material with about 1 mm thickness. The maximum and average temperature of the vacuum chamber was simulated using Fluent software [6]. The maximum wall temperature of the cooling pipe is too critical since it should be less than the boiling temperature of the cooling water.

The heat transfer coefficient between the wall of the cooling tube and water was calculated by the software. Calculated convection coefficients at different locations of the cooling tube wall have different values because the wall temperature of the cooling pipe can be different in dissimilar locations, Figs. 3 and 4. The maximum value of the convection coefficient of the cooling pipe wall and the average of this quantity for the whole wall are obtained that all listed in Table 1.

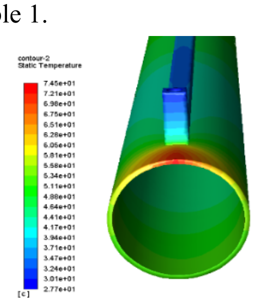

Figure 3: CFD simulations of temperature distribution with 3 m/s flow rate.

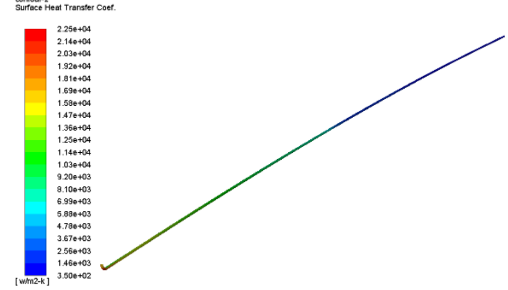

Figure 4: CFD simulations of heat transfer coefficient between the wall of the cooling tube and water with 3 m/s flow rate.

ISBN: 978-3-95450-227-1 ISSN: 2673-5490 doi:10.18429/JACoW-IPAC2022-TUPOMS022

Table 1: CFD Results for Different Water Flow Rates

Flow rate (m/s)	h_{ave} ($W/m^2°C$)	h_{Max} ($W/m^2°C$)	T^{Ch}_{Ave} (°C)	T^{Ch}_{Max} (°C)	$T^{C\,W}_{Max}$ (°C)
3	5800	22500	44	74.5	50
2	4080	17800	51	78.5	60
1	2257	11800	71	98	87

In the next step, with the heat transfer coefficient obtained with CFD, the cooling calculations of the chamber with the FEA modeling were performed, Figs. 5 and 6. Cooling calculations were done with the maximum value of the heat transfer coefficient and its mean value, which are shown in Table 2. In this method, a single convection coefficient should be considered for the entire wall of the cooling pipe chamber. As with the CFD method, the maximum mean temperature of the chamber is simulated.

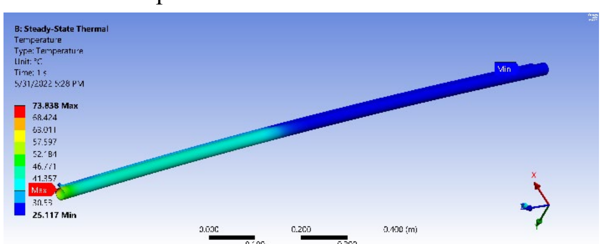

Figure 5: FEA simulations of temperature distribution with $h = 22500\,W/m^2°C$.

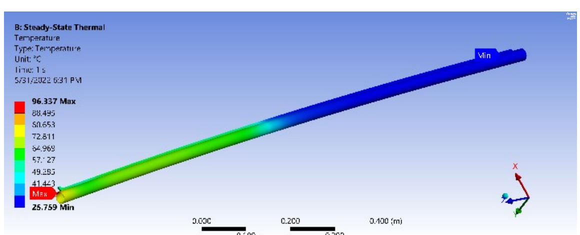

Figure 6: FEA simulations of temperature distribution with $h = 5800\,W/m^2°C$.

Table 2: FEA Simulation With the Heat Transfer Coefficient Results Obtained From CFD

Flow rate (m/s)	h ($W/m^2°C$)	T^{Ch}_{Ave} (°C)	T^{Ch}_{Max} (°C)	$T^{C\,W}_{Max}$ (°C)
3	22500	34	74	37
3	5800	46	96	61
2	17800	35	76	39
2	4080	54	109	75
1	11800	38	81	44
1	2257	73	143	113

Assuming that the material used as a filler in welding has the same heat transfer coefficient as the copper, the chamber cooling simulations have been done again. Comparing the results with the situation in which the actual conductivity of the filler was considered can help us understand this effect. For our simple case, considering the filler thermal conductivity, FEA thermal analysis shows that the maximum temperature of this chamber is about 8 degrees higher than a similar chamber in which the filler's thermal conduction is assumed to be equal to the copper. For the average chamber temperature, this difference is about 2°C; the results are shown in Table 3.

Table 3: Simulation Results, Assuming the Thermal Conductivity of Filler Material as the Same as the Thermal Conductivity of Copper

Flow rate (m/s)	h ($W/m^2°C$)	T^{Ch}_{Ave} (°C)	T^{Ch}_{Max} (°C)
3	22500	32	66
3	5800	44	89
2	17800	33	68
2	4080	52	101
1	11800	36	73
1	2257	71	136

CONCLUSION

To estimate the amount of heat transfer coefficient in vacuum chamber cooling simulations, cooling simulations with different flow rates were done by FEA. After obtaining the heat convection coefficient in these simulations, it was seen that the maximum amount of heat convection coefficient of the cooling pipe and water could be used in finite element calculations. In the process of welding the cooling pipe to the vacuum chamber, in practice in some places the distance between them may be more than usual. Such a gap will be filled by the filler. The filler at the welding place increases the temperature of the chamber locally. Considering the maximum temperature of 150°C for the chamber and a filler with a thickness of 1 *mm* increases the maximum temperature of the chamber by 8°C, which is not negligible in the calculations and may lead to further problems. So, it is recommended to consider the maximum temperature of 140°C on the design level for the enclosures.

REFERENCES

[1] E. Ahmadi, M. Jafarzadeh, J. Rahighi, H. Ghasem, and S. M. Jazayeri, "Designing an Ultra Low Emittance Lattices for Iranian Light Source Facility Storage Ring", in *Proc. IPAC'16*, Busan, Korea, May 2016, pp. 2858-2860. doi:10.18429/JACoW-IPAC2016-WEPOW016

[2] R. Kersevan and M. Ady, "Recent Developments of Monte-Carlo Codes Molflow+ and Synrad+", in *Proc. IPAC'19*, Melbourne, Australia, May 2019, pp. 1327-1330. doi:10.18429/JACoW-IPAC2019-TUPMP037

[3] L. Zhang, J. Biasci, and B. Plan, "ESRF thermal absorbers: temperature, stress and material criteria", *2nd International Workshop on Mechanical Engineering Design of Synchrotron Radiation Equipment and Instrumentation*, 2002.

[4] S. Sharma, E. Rotela, and A. Barcikowski, High heat-load absorbers for the APS storage ring, no. ANL/ASD/CP-102380, Argonne National Lab., IL, US, 2000.

[5] J. A. Clarke, The science and technology of undulators and wigglers, vol. 4, OUP Oxford, 2004.

[6] https://www.ansys.com

[7] S, Sharma, "A novel design of high power masks and slits", in *Proc. Mechanical Engineering Design of Synchrotron Radiation Equipment and Instrumentation Int. Conf., (MEDSI'14)*, Melbourne, Australia, 2014.

[8] Ansys Fluent Tutorial Guide", Ansys INC, release 15, 2013.