ISBN: 978-3-95450-184-7 doi:10.18429/JACoW-IPAC2018-WEPMF022

COUPLED MULTI-PHYSICS SIMULATION FOR THE WATER COOLING LAYOUT OF A RHODOTRON CAVITY

L. Yang[†], X. Z. He, H. Li, S. Q. Liao, Institute or Fluid Physics, CAEP, Mianyang, Sichuan, China

Abstract

A Rhodotron-based electron accelerator served as micro-focused X-ray source is under development at IFP, CAEP. The RF-cavity, running in long pulse/ CW mode, will deliver 9 MeV energy to electron beams after multiple accelerations within the same field at a frequency of 107.5MHz. A substantial amount of average power loss with tens of kW will be dissipated on the RF surface of the cavity to maintain the operational field level. Efficient water cooling is critical to prevent large scale temperature rise for stable operation sake. Reasonable prediction of temperature rise becomes essential to assess a certain cooling layout in the design phase. The frequency drift and thermal stress on account of temperature variation and gradient on cavity wall respectively, could be computed accordingly. This paper presents a comprehensive coupled simulation involving electromagnetic, thermal and structural for the RF-cavity of Rhodotron.

INTRODUCTION

Rhodotron has been widely employed to generate X-ray for industrial irradiation since the concept of multiple accelerations in the same field supplied by an RF cavity (half wave resonator, HWR) was raised [1]. A micro-focused X-ray source driven by a 107.5MHz Rhodotron in long-pulse /continuous wave (CW) mode is undergoing R&D at IFP, CAEP. Aiming at higher resolution, the beam spot formed at the target location is required to be 0.2 mm other than a typical size of 2 mm for normal industrial CT machines [2]. After 10 passes, the cavity will deliver 9 MeV of energy to electron beams. The cavity will be made of stainless steel with copper coated on the inner surface to preserve cost. Pulsed RF power dissipation in the order of 100kW is yet required to generate an operational field level of 0.9 MV in the cavity. At high duty factor, the average RF power dissipation reaches tens of kW. For stable operation, water cooling layout on the cavity wall should be carefully concerned to prevent high temperature rise on the cavity wall. Due to the poor thermal conduction of stainless steel, the areas with high surface loss density in the order of 10^6 W/m^2 is dressed with a jacket instead of a cooling circuit attached. It allows these areas being fully covered by forced water so that efficient heat convection is benefitted.

With the known heat convection status on the wall/water interface, one can perform steady-state thermal analysis by using those common Finite Element Analyses codes to compute temperature rise at a certain power consumption. The computed temperature pattern over the cavity wall is necessary as a body load for the subsequent structural analysis and prediction for thermal induced frequency drift. Regarding to the water cooling problems with uniform meanders and heat flux, the heat convection coefficient could be well derived from theory, which is usually adopted by the thermal simulations for normal conducting RF guns, cavities [3-4]. In our case, however, the heat convection highly depends on the local heat flux and fluid field due to the non-regular cooling layout, therefore theoretical approximation is impractical to approach the distribution of heat convection coefficient. Alternatively, we take the advantage of the heat transfer enabled fluent package in ANSYS [5] that the heat convection over the wall/water interface could be internally computed and coupled with fluid dynamics simulation in the water volumes. The data mapping technology between non-shared mesh and node patterns in Workbench allows load import in multi-physics coupling. This paper will focus on the coupled simulation involving electromagnetic (EM), thermal, fluid and structural. The rest content is organized as follows. Description of the calculation procedures is detailed in section 2. Section 3 presents the achieved results for the Rhodotron cavity design at IFP.

PROCEDURE OF SIMULATION

The frame of coupled analysis engages 3 physics packages: EM, heat transfer enabled fluid and static structural. There are data links upon a shared geometrical model between each other to transfer load into downstream calculation. The steady temperature rise in cavity body at a certain RF power level and flow rate of water, accordingly, the frequency shift due to thermal expansion, and the maximum thermal stress are primarily overseen. All these packages are assembled in ANSYS Workbench. A work flow is plotted in Figure 1 which also illustrates the data transfer network.

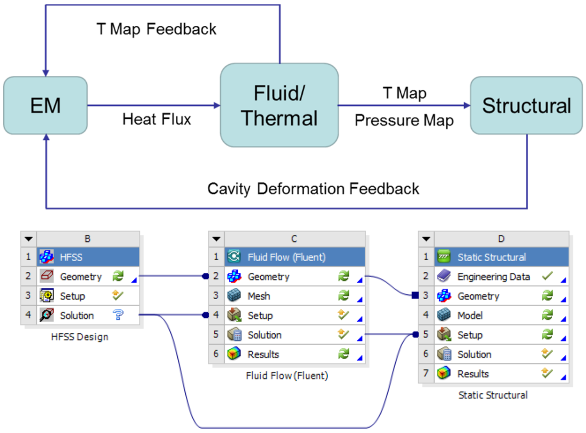

Figure 1: The work flow of the coupled analysis and a snapshot in ANSYS user interface.

ISBN: 978-3-95450-184-7 doi:10.18429/JACoW-IPAC2018-WEPMF022

The surface loss density calculated in HFSS is imported into Fluent as an energy source which will be conducted through the cavity wall in the solid zone and taken away by water flow in the fluid zone. The temperature distribution over the solid zone will be feedback to HFSS from Fluent. Since the resistance of the coated copper on the cavity inner surface is temperature dependent, an updated heat flux on account of material property variation resulted from another run in HFSS will be reloaded into Fluent, and accordingly, the temperature distribution could be renewed. One can perform this iterative process couple times to meet the convergence threshold. In our case, one iteration has already shown a reasonable convergence. The final temperature distribution as well as the pressure load applied by the water flow go forward into the static structural package. The deformation induced by thermal expansion and pressurized water and also the vacuum load in RF volume could be independently calculated or linearly superposed. The overall maximum stress is predicted by the latter one. The deformation will be captured by HFSS and mapped onto the mesh pattern for the new eigen mode calculation. The frequency shift is taking the difference to the initial eigen mode result.

Model Preparation

The final RF model was translated into a shell model as the cavity wall with thickness of 15 mm, see Figure 2. The cooling layout over cavity wall was designed in a way that cooling efficiency and mechanical stability were both considered. The inner conductor and the short ends are fully covered by a jacket with a 5 mm gap, while the outer conductor is cooled by numbers of regular meanders evenly distributed along the cavity circumference. Pressurized water will be injected towards the blind plates in the medium region of the inner conductor. All the water flows drained out of these meanders are collected by a belt like buffer and circulated back to a chiller. These meanders play another import role of stiffening rings to keep the large cavity body rigid against water pressure, vacuum in RF volume, and thermal expansion. Symmetrical boundary conditions allow using a wedge model (1/40 of the full model) with water volume filled in all the gaps and meanders for the coupled simulation.

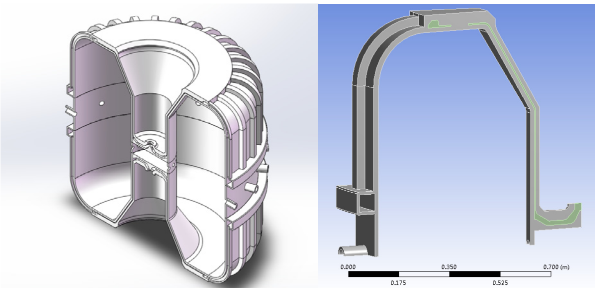

Figure 2: Left: Full cavity mechanical model; right: reduced model for coupled simulation, the water volume is coloured in green.

Simulation Setup

The field intensity in HFSS was adjusted to an overshot operational level (CW) with 15% margin to 68kW given by electromagnetic optimization as possible Q_0 degradation might take place during cavity fabrication (Figure 3). Temperature dependence of copper electric conductivity was enabled for the afterwards calculation with temperature feedback. The water flow rate was set to 55.7L/min which is equivalent to an average temperature rise of 10K at the outlet with respect to 77.5kW energy absorption by water. The water flow was injected at a typical temperature 300K. A rough estimation of Reynolds number in the areas with high flow velocities gives a range of 470~967. Hence, the fluid problem mostly falls in Laminar flow category (Re<2000). Large number of mesh ~ 1 million is generated even though a significant reduced model was used. The meshes in the boundary layers where heat convection is processed were refined as shown in Figure 4.

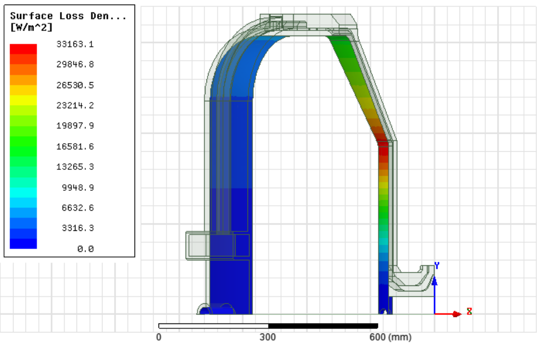

Figure 3: Surface loss density. The cavity wall dressed with water cooling jacket appears as the transparent portion.

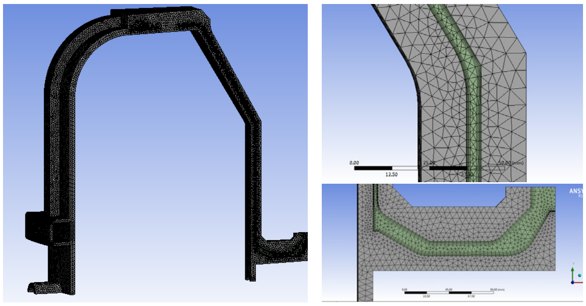

Figure 4: Mesh view in Fluent and zooming at locals.

RESULTS

After 40 iterations, the maximum temperature monitored in the entire processing reaches steady. The ramped up trace could be seen in Figure 5. The water temperature distribution in transverse plane on the outlet yields a mass-weighted average rise 9.94 K, and well consistent with the one predicted by a given flow rate. Figure 6 shows the temperature distribution with the initial heat flux load and the convection coefficient calculated from postprocessing. The coefficient varies over the whole interface with 3 orders of span, which is far beyond the capability of theoretical estimation. The negative value means heat transfer back to some areas at lower temperature than water flow. There are

ISBN: 978-3-95450-184-7 doi:10.18429/JACoW-IPAC2018-WEPMF022

two regions with appreciable temperature rise, the one subjected to high loss density and the other one located in the medium of two adjacent meanders where water cooling doesn't reach and the heat has to be conducted away by bulk stainless steel wall with poor thermal conductivity.

A 5% of Q_0 drop after temperature feedback was seen, and in turn caused additional 5.6K (Figure 7) rise to the initial maximum temperature, 353.9K. The heated cavity due to thermal expansion induces deformation with a maximum displacement of 0.5mm, and shifts the resonating frequency by a mount of -30.3 kHz. The maximum Von Mises stresses under various load were also calculated, listed in Table 1. The stresses arise under the loads that would appear in the real operation are yet significantly below the typical tensile yield strength of stainless steel, 214MPa.

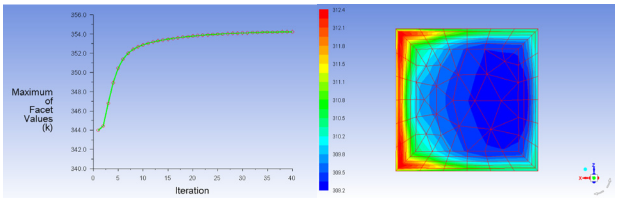

Figure 5: Left: maximum temperature trace over 40 iterations; right: water temperature distribution in transverse plane on the outlet.

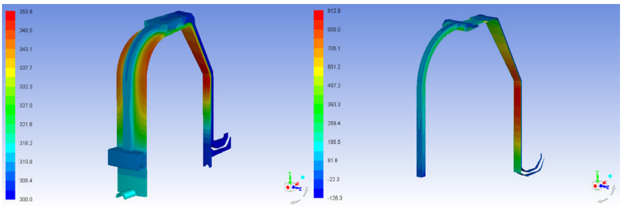

Figure 6: Temperature rise under initial heat flux load and the calculated convection coefficient.

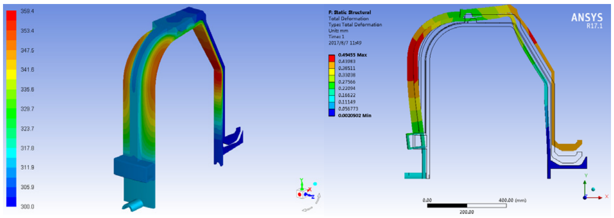

Figure 7: Updated temperature rise and induced deformation; the frame in solid line represents the undeformed.

Table 1: Frequency Shift and Von Mises Stresses Under Various Load

Load	Freq. Shift/kHz	Von Mises Stress/MPa
Vacuum	2.9	6.4
Vacuum + 4 bars water pressure	0.2	115.2
Vacuum + 4 bars water pressure +thermal at 80kW	-30.1	117.2

Sweeping Flow Rate and RF Power

Further simulations have been done by sweeping the flow rate and RF power. The flow rate was adjusted from 27.9 to 139.3 L/min, corresponding to the mean temperature rise in fluid at the outlet, 4~20K. The maximum temperature and the induced frequency shift are plotted in Figure 8. When the flow rate is being increased over 60 L/min, the temperature drop starts to be slow down, where the high temperature spots on the outer conductor mainly cooled by thermal conducting become dominant. The optimized flow rate to this particular cooling layout, therefore could be defined at somewhere close to 60L/min. The RF power was swept out at a fixed flow rate, 55.7L/min. In reality, the variable should be the duty factor in pulse mode while the field intensity within a pulse is maintained. Linear variation either for the temperature rise or the frequency shift along with increasing duty factor up to CW is shown in Figure 8 as well.

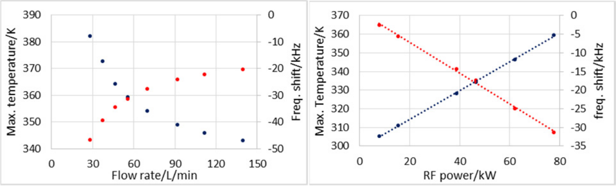

Figure 8: Left: the maximum temperature (blue) varies with the flow rate and the corresponding frequency shift (red); right: the maximum temperature (blue) varies with RF power at a fixed flow rate, 55.7 L/min, and the corresponding frequency shift (red).

CONCLUSION

Calculation techniques for comprehensive EM, thermal, fluid and structural coupled simulation of RF cavities have been described extensively. Reasonable prediction on RF induced temperature rise is applicable owing to the heat transfer enabled fluid simulation with internally computation on the convection coefficient, which must be accurately given in advance in thermal analysis. Cooling layout was severely designed for this specific RF cavity of a Rhodotron with a moderate temperature rise in simulation at a maximum power level. Mechanical stability under a variety of load conditions was checked. The optimized flow rate to the cooling design has been investigated by parameter sweeping.

REFERENCES

[1] J. Pottier *et al.*, "A new type of RF electron accelerator: the Rhodotron", *Nucl. Instrum. Methods-Phys.Res.B*, 40/41(1989)943.

[2] X. Z. He *et al.*, "A proposal of using improved Rhodotron as a high dose rate micro-focused X-ray source", in *Proc. SAP2017*, Jishou, China, 2017, TUPH08.

[3] F. Marhauser, "Finite Element Analyses for RF Photoinjector Gun Cavities", BESSY GmbH, Germany, TESLA FEL Report 2006 - 02.

[4] N. Hartman *et al.*, "Electromagnetic, Thermal, and Structural Analysis of RF Cavities Using ANSYS", in *Proc. PAC2001*, Chicago, USA, 2001, MPPH060.

[5] `http://www.ansys.com`