THERMAL SIMULATION OF LINAC CAVITY OF 10 MEV RF LINAC

Rajesh Barnwal, S.R. Ghodke, Mahendra Kumar, V.T. Nimje, K.C. Mittal, D.P. Chakravarthy, L.M. Gantayet

Accelerator and Pulse power Division, BARC, Mumbai, India

Abstract

The 10 MeV RF linac at Electron Beam Centre Kharghar, has 33 cells assembly fabricated in three parts. Of these, the top and the bottom parts consist of 14 cells each while the middle part has 5 cells, including the power feed cavity. Water cooling is provided to the cavity by brazing Cu tube spirally over the cavity surface. Detailed conjugate heat transfer modelling has been carried out using the FLUENT flow modelling software. The temperature distribution profile is obtained which identify the hot spot zones. The further thermal optimization has been carried out based on the results and more uniform thermal profile is obtained by modification in the cooling tubes design and water flow rate.

INTRODUCTION

Computational Fluid Dynamics (CFD) is a computer based tool for simulating the behaviour of systems involving fluid flow, heat transfer and other physical processes. This is accomplished by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region. Computational Fluid Dynamics is now an established industrial and research tool, helping to reduce design time and improve process efficiency. The CFD software used in this study is FLUENT

DESIGN REQUIREMENTS

Due to high RF power loss in the linac tube during operation, it requires an active cooling system designed to satisfy the following requirements:

- Uniform temperature profile along the axis of the linac tube must be there which significantly lower the detuning range of the cavity. Uniform and non-uniform heat flux distributions are used to assess the design compliance with this criterion.
- The cavity cooling design has to minimize the pumping power associated with the fluid flow. This concern is addressed by creating a geometry that provides small hydraulic resistance as well as by a judicious selection of the coolant inlet velocity.
- Heat transfer at the solid/liquid interface is by single phase forced convection. Since the coolant is water, operating limits have to be identified and established.

FLOW MODELING

The actual 3D geometry of the cavity is modelled by using GAMBIT preprocessor modelling tool. In the operating LINAC the cooling tube is spirally brazed over the cavity having water input from one end and output is from another end. Meshing of the model is also created in the GAMBIT. Fine the meshes near the zone of fluid solid interaction surface. The turbulence model selected for this simulation is the k-epsilon model. It is an industry standard model which has proven to be stable and numerically robust while at the same time offering good solution accuracy. Material property is defined for solid cavity side is OFHC copper and fluid side is process water.

Boundary conditions:

- For the flow inlet boundary condition, the normal speed in option has been selected because it provides better simulation stability and convergence. The direction of the velocity is normal to the boundary and constraints are imposed such that the flow direction is parallel to the boundary surface normal, for each element face at the boundary. As far as heat transfer is concerned, a constant inlet temperature is specified for this type of boundary condition. The water inlet condition specified for the operating LINAC is 1.6m/sec at 22^0C.
- The flow outlet boundary condition utilizes an outlet relative static pressure which is constrained such that its average value equals the value specified as input.
- Wall – This type of boundary condition is used define the physical limits for domain of interest that is investigated. When the domain under scrutiny is a fluid domain, the velocity of the fluid at the wall boundary is set to zero. For the solid cavity side the wall boundary is set to natural convective loss heat transfer coefficient of 4W/m^{2}K at 25^0C of bulk outside temperature.

Thermal heat load:

- Average RF power loss in the cavity is 15KW for the 6MW peak power, 10microsecond pulse RF input power. The profile of RF power loss from RF analysis is used to define the heat flux load associated with the cavity.

.

Case is solved in the FLUENT. The iterative solver is converged all the flow equations like continuity, momentum and energy up to the order of 10^{-5}.

Figure 1: Temperature profile of operating LINAC.

The temperature distribution profile obtained from the simulation is shown in Fig. 1 above (only top part of cavity). The temperature in the solid cavity side reaches to 349K near the top flange. Also the cavity temperature is gradually rises from bottom to top. Though the simulation has predicted the fairly reasonable temperature profile but in reality temperature may be in the higher side because of the poor contact of brazed cooling coil. The new cooling design of cavity is proposed with some modification in the previous one. All the cooling channels are made parallel inlet and outlet from the single inlet and outlet header. The shape of the channel is also optimized to C type with one face is in direct contact with the cavity surface. This feature eliminates the poor contact of cooling channel to the cavity. The middle flanges are also provided with cooling slots. The flow velocity in the channel was optimized at 3m/sec (correspond to 210 lpm) for 1^0C rise of water temperature.

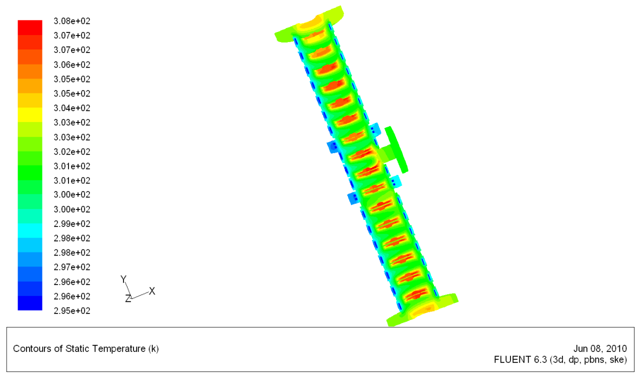

Figure 2: Temperature profile of new LINAC.

The figure shown above the temperature distribution profile of the new cooling design of cavity. Temperature profile obtained is uniform alog the axis of cavity also the temperature range of the cavity is narrowed down to within 12 °C.

REFERENCES

[1] FLUENT 6.3 User's Guide
[2] GAMBIT 2.2 User's Guide
[3] Heat Transfer by J. P. Holman, McGraw Hill Publication