SAFETY ANALYSIS OF TITANIUM FOIL WINDOW AFTER SCAN FAILURE IN 10 MEV RF LINAC

R.Barnwal, S. Acharya, S.R. Ghodke, M.Kumar, K.C.Mittal, D.P.Chakravarthy, L.M. Gantayet
Accelerator and Pulse power Division, BARC, Mumbai-85, India

Abstract

A 10 MeV RF linac for development of industrial applications is operational at the Electron Beam Centre, Kharghar. It employs a 25 µm thick Titanium foil of 100 cm x 7 cm to extract the beam to atmosphere for irradiation purpose. The 10 µS, 250 mA peak current pulses with a repetition rate of 400 Hz are scanned along the foil with scanning frequency in the range 1-10Hz. There exists the possibility of the failure of the scanning system and the incidence of the beam at a particular location of the foil thus causing damage to it. A pertinent question is how much time can elapse without causing damage to the foil so that one can actuate the beam trip in such an eventuality. A thermal analysis of such a scenario has been carried out by using the finite element tool ANSYS. A Gaussian beam profile with 20 mm FWHM has been used with 1/16 J of energy absorption in each pulse. The absorption area is the projection of a circle on the curved surface of the foil. The heat loss mechanisms are the radiation and forced convection. As expected, the foil temperature rises with every pulse but the rate of rise reduces with temperature as the radiation loss becomes significant. In about one second duration, the temperature at the central node saturates at 1343K which is below the melting point of Titanium. The thermal stress approaches 75 % of the allowable stress at this point. However, electronic circuits incorporated in the system can activate a trip of the beam in less than 250 ms, thus ensuring safety of the foil.

INTRODUCTION

This paper aims for the prediction of temperature time profile of the titanium foil in case of the scan failure. The steep rise in temperature in a microsecond high intensity beam pulse duration is computed by using finite element method. The thermal deformation is also computed from the temperature rise in the foil and checked its mechanical strength and possible damage. Finite element package ANSYS is used for this analysis.

FE MODELLING OF FOIL USING ANSYS

The actual 3D geometry of the foil is modelled by using ANSYS preprocessor modelling tool. The Titanium material model is also done along with the variation in temperature. Model is meshed by using thermal SHELL57 element. It is a 4 noded rectangular element having features of conduction, edge and surface convection as well as radiation. Meshing of the model is done using smart element size selection. Refined the mesh near the centre where beam heats the foil directly over it.

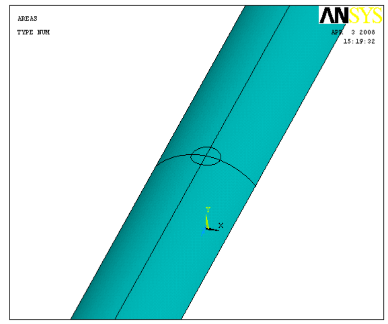

Figure 1: Layout of model.

Boundary conditions

- Convective heat transfer coefficients over foil concave surface. As blower directs the flow of air to the shape of foil, it is nearly a flow over a flat plate. The formula for heat transfer coefficients is given by

$$\overline{Nu} = 0.664 * \mathrm{Re}^{1/2} * \mathrm{Pr}^{1/3}$$

$$\bar{h} = 48 W / m^2 K \quad \text{at } 300K.$$

- Both surface of foil is make as a radiative surface, radiating heat to the outside at a temperature of 300K.
- Initial temperature of foil was taken 300K uniformly over the surface.

Thermal heat load

- Heat flux is defined at the centre of foil by beam heating. The nature of the beam is a Gaussian distribution defined by the following relationship

$$\mathrm{HF} = Ke^{Cr^2} Watt / m^2$$

where K=18.32X10^{4}Watt/m^2, C= -2.30X10^4 /m^2, and r is radial distance in meter. Total heat input, which is the integral over the cirular area shown in figure 1, comes to 25Watt. Therefore functional Gaussian input is defined in the ANSYS.

The first solution of the model is achieved in the steady state where temperature of each node of the model reaches to its saturation temperature. Fig 2 shown below is the steady state temperature distribution of the model. The maximum temperature is at the centre of foil, which is at 1343K. As we go away from the center, temperature gradually decreases and makes an almost circular profile of equi temperature zone. We can also observe the effect of Gaussian heat load is locally to some distance from centre after that the remaining foil is maintained at 300K only which is the outside temperature.

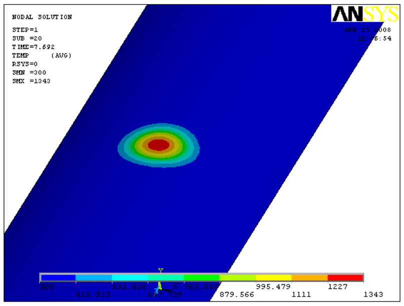

Figure 2: Steady state temperature distribution of foil.

In the second solution of the model transient analysis was performed where temperature time profile is generated. In this stage of analysis time step input is also considered. There is a 400 number of pulses of beam is coming in 1 second. Each pulse duration is 10 microsecond. To input this condition one separate load step file is written in APDL (Ansys parametric design language). The functional Gaussian heat flux is applied over 10 microseconds in every pulse gap of 2.5 milliseconds. In order to reduce the computing time of the system, feature of symmetry is used. Only the quarter of model is used for analysis. Fig 3 shown below the temperature distribution of the foil after 1 second in the quarter model. The maximum temperature is at the centre node, which is at 1310K near to its saturation value of 1343K from the steady state solution. It implies that the foil reaches to its saturation temperature nearly in 1 second.

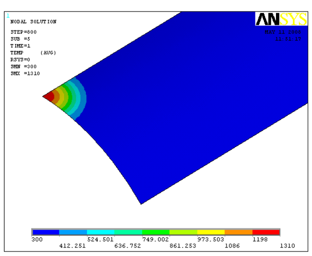

Figure 3: Temperature distribution after 1 second.

The graph shown below in Fig 4, is the temperature time profile is shown below. There are 4 nos. of line-plotted showing the temperature rise with time at different nodes of the foil.

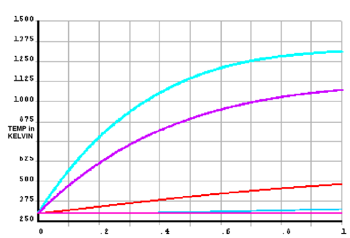

Figure 4: Time Vs Temperature profile

THERMAL DEFORMATION ANALYSIS

As the model is subjected to the above computed temperature distribution under the steady state condition (when saturation temperature is reached), it will undergoes to some thermal expansion. Because of this, some thermal stresses will also induced in the foil. This analysis can be performed by coupled field method. A coupled-field analysis is an analysis that takes into account the interaction (coupling) between two or more disciplines (fields) of engineering. In a sequentially coupled physics analysis, you can couple the two fields by applying results from one analysis as loads in another analysis. The load transfer occurs external to the analysis, and you must explicitly transfer loads using the physics environment. An example of this type of analysis is a sequential thermal-stress analysis where nodal temperatures from the thermal analysis are applied as "body force" loads in the subsequent stress analysis. The physics analysis is based on a single finite element mesh across physics.

The linear structural material is modelled. Structural SHELL63 element is selected which is compatible with thermal SHELL57 element Nodal temperatures from the thermal analysis are applied as "body force" loads in this subsequent stress analysis. A physics file is read into the database, coupled-field loads are transferred. The reference temperature input is given as 300K. The foil is also subjected to a differential atmospheric pressure; therefore 10^5 Pa (equivalent to 1 atmosphere) is given as the concave side pressure. All the boundary edges of the foil are fixed. The nodal displacement solution of the model is obtained as shown in the fig 5. It is clear that the foil at the centre is bulge inside out(because inside is in vacuum) almost in the Gaussian shape. At the centre maximum deflection is 3 mm.

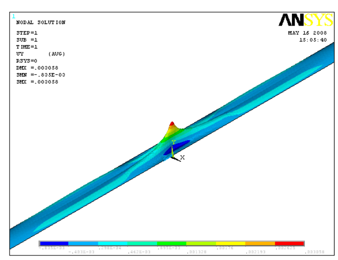

Figure5: Nodal displacement of the model

Though this thermal deformation reveals the picture of stress and deformation reasonably but accuracy of result can be improved by taking into accounts of non-linear material modelling with proper hardening parameter and stress stiffening effects.

REFERENCES

[1] ANSYS V10
[2] ANSYS, Inc. Theory Reference
[3] Heat Transfer by J. P. Holman, McGraw Hill Publication