BEAM IMPACT OF THE ILC COLLIMATORS

George Ellwood, Justin Greenhalgh, CCLRC/RAL, Chilton, Didcot, Oxon

Abstract

Spoilers in the ILC Beam Delivery System are required to survive a minimum of 1-2 direct impacts from each energetic electron or positron bunch of charged particles without failure, in addition to maintaining low geometric and resistive wall wakefields. The transient shock wave resulting from rapid localised beam heating and its implications for spoiler design are studied using ANSYS. The realistic patterns of energy deposition are taken from FLUKA.The results presented quantify uncertainties in the predictions and consider possible options for spoiler jaws for the ILC.

INTRODUCTION

The ILC (International Linear Collider) aims to accelerate almost 3000 bunches of electrons and positions to a centre of mass of 500 GeV in the first stage and, in an anticipated upgrade to 1 TeV or more. After accelerating the bunches it is necessary to collimate them to remove any excess particles that travel with them in the form of a halo of charged particles. These collimators are required to survive at least 2 bunches impinging on their surface. It is clear that the highly energetic beams may melt or at the very least, cause considerable stress and fracture of the material of which the collimators are composed. This paper has concentrated on the stress problem and has assumed that the temperature increase is insufficient to melt the material. This leaves the problems of dynamic loads, the shock wave issues and small heated zone which require small element sizes and the rapid heating which necessitates short time steps.

All our simulations have been undertaken with a 250GeV pulsed beam composed of 2820 bunches each of which contains $2x10^{10}$ particles of length 1ps with an inter-bunch spacing of 337ns. The beam monitoring and protection system should be able to detect an errant beam and direct it towards the beam dump within 2 bunches. The collimator will more than likely be of the spoiler and absorber type, with the geometry of the collimators will be decided primarily on the wakefield studies. The effects of the beam striking the spoiler jaw have been considered in this paper.

TYPES OF ANALYSIS

In order to look at the material's response against time transient analysis needed to be performed. The heat deposition figures were calculated using FLUKA and given in units of $GeVcm^{-3}$, these were then converted into Wm^{-3} for use in ANSYS. Thermal-structural coupled field analysis needs to be performed in order to calculate the dynamic stress from a given heat generation load. A coupled-field analysis is an analysis that takes into account the interaction between two or more fields of engineering. There are several ways of doing this.

The direct coupled field analysis requires the use of a coupled field element which contains all necessary degrees of freedom and solves for each at every time step.

Indirect coupling uses a sequential method whereby the thermal solution is obtained and then used as a body load for the structural analysis; both solutions have to be performed for each time step. The loads must be transferred between thermal and structural environments externally from the solver. [1]

A third method can be used when studying events that occur over so short a time period that heat transfer within the material is negligible and can be ignored. This method is very similar to the indirect method in that both thermal and structural analyses are performed. The difference is that only one thermal analysis is performed and this is taken to be constant for all the subsequent structural analyses. Although less accurate this method is preferred for some solutions because the thermal solution does not need to be performed for every time step and the structural elements require less CPU time to solve than coupled field elements. During the time scales of the order of 400ns, heat transfer by conduction travels ~5µm whereas the shock wave travels ~2mm, so conduction has little affect and can be ignored.

FLUKA, ANSYS AND THE USE OF TABLES

FLUKA was used to produce simulations of heat deposition caused by a beam strike of energy 250GeV hitting a section of spoiler jaw made from Ti-6Al-4V alloy measuring 1mm x 0.1mm x 3mm. A bin size of 10µm x 10µm x 500µm was found to be the minimum required so as to capture the peak data. This data was then converted into Jm^{-3} and then divided by 1ps to give heat deposition values in Wm^{-3}. The data was input into ANSYS by use of the table function. The heat deposition due to particle cascades within the material is from several different mechanisms with different time profiles. For simplicity a conservative assumption that all the energy is deposited within 1ps was made.

Previous work has looked at using FLUKA and ANSYS. [2] Since then ANSYS has been developed and incorporates a table function. Tables in ANSYS give the value of heat deposition in 3D co-ordinates within the geometry. ANSYS then interpolates these values and applies them as a load to the nearest node. The table size doesn't have to match the mesh of the finite element model, the use of tables doesn't add to the solution time of the model.

Thermal analysis was performed in order to find out the rise in temperature expected from 2 beam strikes. The heat load found using FLUKA was applied, then after 337ns the load was applied again. This leads to a temperature increase of 233°C per beam strike, reaching 466°C in total. Due to the small time gap between strikes heat from the first strike couldn't dissipate through conduction.

Coupled field analysis was performed but this underestimated the maximum stress, this was due to the relatively coarse mesh used. A limit of 6mm has been found to be the minimum mesh size that can be solved in a reasonable amount of time in 3D, although a mesh in the micron range is required to capture the peak heat generation. This has led to some theoretical 2D studies of shockwaves in the micron range.

SHOCKWAVE STUDIES

Initial Studies

A simple model was created this was essentially 1D but was constructed using a strip of 2D structural elements. This model had an overall length of $5x10^{-8}$m in x, the first $1x10^{-8}$m formed the heated zone. Square elements of $1x10^{-11}$m were used to mesh the model. It was assumed to be in plane stress, meaning it was taken to be stress free in the z direction. Work has also been done using plane strain to capture Poisson effects but this is not reported here. The load was applied as a temperature to the structural model so that a thermal solution did not need to be performed for each time step. ANSYS was used in µMKSV mode to enable it to work with very small distances. Unless otherwise specified the material used was Aluminium with the material properties shown in table 1. The material was assumed to be perfectly linearly elastic with no yield or failure stress and to be anisotropic, with only thermal expansion in x allowed.

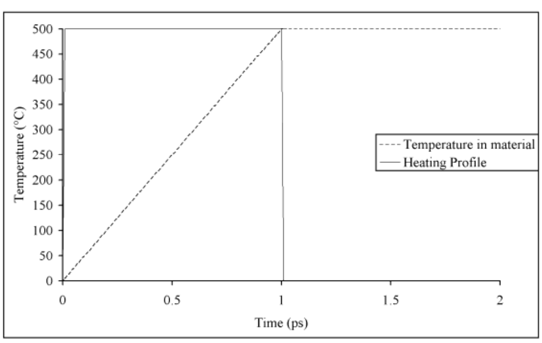

Figure 1: Heating and temperature profile.

Table 1: Material properties.

Young's Modulus	70 GPa
Poisson's ratio	0.27
Density	2700 kgm^{-3}
CTE	23 $E^{-6}.°C^{-1}$
Speed of Sound in Material	5091 ms^{-1}

A temperature rise of 500°C in the material was applied that would correspond to sudden heating of the collimator. This is shown in figure 1.

The stress in the x direction was plotted as a function of x at several times, this is shown in figure 2. Unconstrained thermal expansion of the heated zone would be $1.15x10^{-10}$m; in this case the inertia of the unheated region doesn't allow the heated zone to expand. This gives rise to a strain of -0.0115 which leads to a stress of -805MPa. This agrees with the results from ANSYS. After 1ps, at the end of the heating period, the stress in the heated zone has risen to -805MPa.

The shockwave propagates in the x direction, travelling at the speed of sound in Aluminium. At 5ps it had left he heated zone completely. As the shockwave left the heated zone it halved in magnitude but doubled in length.

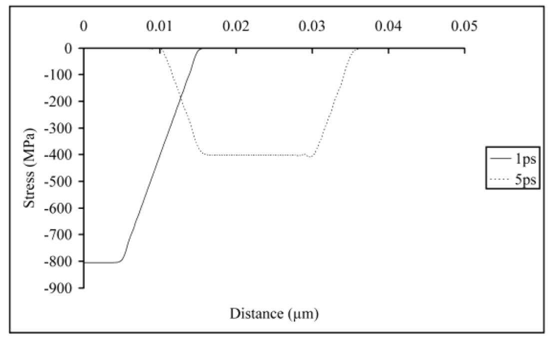

Figure 2: Stress in x in as shockwave leaves heated zone.

Size And Shape Of Heated Zone And Effects Of Ramp Rate

Different lengths of heating zone were tried. Increasing the length significantly there was found to have little effect on the maximum stress. When the heated zone was reduced it was found that the stress did not reach the maximum theoretical value of -805MPa. The critical size was found to be the distance travelled by the speed of sound in the material during the heating period. A heating period of 1ps was a conservative underestimate; in reality the heating period will be longer which will increase the critical size of the heated zone. This agreed with earlier work on the subject. [2]

Different ramp rates were tried but found to have little affected the maximum stress level. The maximum stress was only affected by the temperature reached.

A different model was created with two heated zones, to test the affect of having a non-uniform heated region. Again this model had an overall length of $5x10^{-8}$m in x, the first $1x10^{-8}$m was heated to 500°C, $1x10^{-8}m < x < 2x10^{-8}m$ was heated to 250°C. At 1ps the maximum stress is the same as that of the uniform heated zone, but after 5ps the shape of the shockwave has changed significantly. There is a pronounced lip due to the two heated zones. See figure 3.

Minimum Element and Time Steps

The model was scaled up to have an overall length of 50µm in x, the first 10µm formed the heated zone. This was the minimum bin size required by FLUKA to capture the heating events. The effect of element size on the results was studied. This looked at the maximum stress in the material after 5ns when the shockwave had left the heated region. See figure 4. As the element size increased the accuracy decreased. When the element size was the same length as the heated zone it gave an error of ~25%. The time steps were studied using an element size of 5μm. The results of which are shown in figure 5. A time step of 200ps was found to be the limit for capturing shock effects in the material. Sub 1ps time steps would be required for the heating phase.

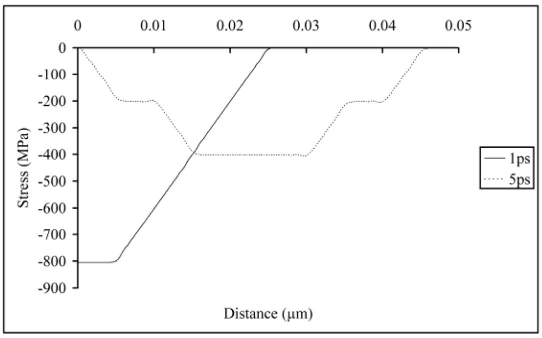

Figure 3: Shock wave caused by graduated heated zone.

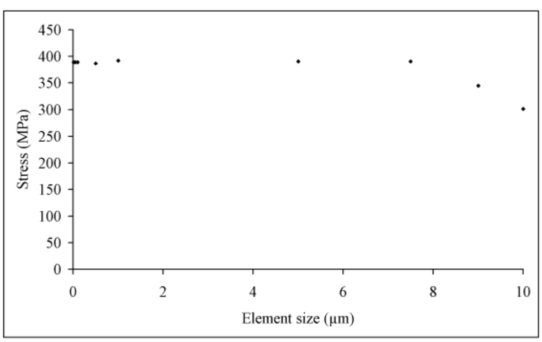

Figure 4: Element size.

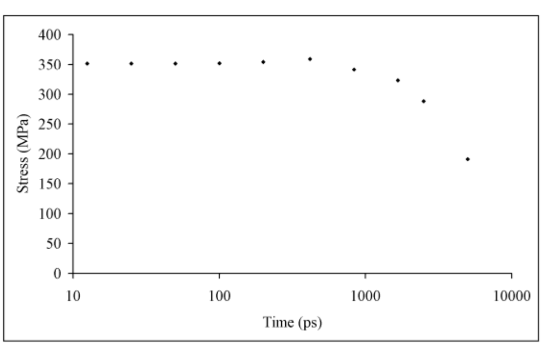

Figure 5: Time steps.

CONCLUSIONS

With a combination of FLUKA and ANSYS temperature rises and stresses within materials due to sudden heating has been calculated. The direct impact of two bunches on the collimators has been shown raise the temperature by no more than 466°C. This is significantly below the melting point of the Ti-6Al-4V alloy which the collimator is composed. To improve the accuracy of the results presented would require a mesh spacing better than 10μm and time steps of less than 200ps.

Neither the heating rate nor the shape of the heated zone has an affect on the maximum stress seen within the material.

Acknowledgements

The authors would like to thank all of the Spoiler Wakefield and Mechanical Design team especially Nigel Watson and Juan Luis Fernandez-Hernando.

REFERENCES

[1] ANSYS Manual Release 9.

[2] J. Zazula, "From Particle Cascade Simulations (FLUKA) to Finite Element Heat Transfer and Structural Deformation Analyses (ANSYS)." CERN-SL/BT(TA) (1995).

[3] P. Sievers, "Elastic Stress Waves in Matter due to Rapid Heating by an Intense High-Energy Particle Beam." CERN LAB.II/BT/74-2 (1974).