*19th International Symposium of Ballistics, 7–11 May 2001, Interlaken, Switzerland*

# STRUCTURAL ANALYSIS OF A KINETIC ENERGY PROJECTILE FOR MEDIUM CALIBER GUN

**KK Pathak and M Jha**

*Armament Research and Development Establishment (A.R.D.E),*
*Dr. Homi Bhabha Road, Pashan, Pune – 411 021, India*

A 2D axisymmetric finite element analysis of a kinetic energy projectile for medium caliber gun has been carried out to check its in-bore structural integrity. The objective of structural analysis has been to obtain accurate stress field in the penetrator and sabot by considering proper contact definition at the threads. A high stress concentration with steep spatial gradient has been predicted in the fillet region of rear thread of the penetrator. A high spatial rate of change of the stress has also been observed in the rear thread of the projectile. Although, stresses are very high at the fillet of the rear thread, they reduce to safe limit rapidly within 25 micron depth into the material of the penetrator.

## INTRODUCTION

Firing a kinetic energy ammunition from a medium caliber gun integrated with the infantry combat vehicles has distinct advantages in respect of fire power and target engagement capabilities of the vehicle. The design of medium caliber kinetic energy projectile is however not an easy task. Since, the projectile is launched from a medium caliber and small length barrel as compared to that of a normal tank barrel, a very high accoleration is required to develop necessary muzzle velocity to cause desired effects on the target. The kinetic energy projectile considered for study, experiences a pressure of 300 MPa and an extremely high acceleration (75,000 g) which is almost double than that experienced in the normal tank gun. Thus, ensuring in-bore structural integrity of the projectile under such severe loading conditions becomes a difficult design task.

Drysdale [1] used the mechanics of materials approach to obtain a closed form analytical solution of the simplified boundary value problem for kinetic energy projectile. The load transfer between the sabot and the penetrator was considered through the shear traction between them and a safe design of the projectile was evolved which ensured in-bore structural integrity. The effects of the load transfer through threads were not modelled in the above study. Pflegl, et. al. [2] and Holis [3] have performed 2D axisymmetric structural analyses of the projectile with the help of finite element method. The above finite element structural analyses of kinetic energy projectile were performed without contact definition at the threads. Such analysis approach may not predict accurate stresses at the stress

Home
Table of Contents

concentration regions due to the absence of contact related phenomena occuring at the threads.

In this work, a 2D axisymmetric finite element model with proper contact definition at the threads between sabot and penetrator has been considered for performing structural analysis. The objective of structural analysis has been first to obtain overall stress distribution in the full projectile and then to predict accurate stresses in the penetrator and sabot at the stress concentration locations by a submodel analysis. The finite element model has been prepared using 4-node quadrilateral solid and node-to-surface contact elements. The finite element model has been generated using exact geometric details of parts, which includes the thread geometric details on penetrator and sabot.

## STRUCTURAL ANALYSIS APPROACH

A 3D transient dynamic analysis with material non-linearity and appropriate contact interface at the threads between sabot and penetrator would be the most appropriate structural analysis for predicting realistic in-bore structural behavior of kinetic energy projectile. But, this analysis requires large computational resources and is very costly. A fairly reasonable estimate of stress field, however, can be obtained by a 2D axisymmetric analysis. In this work, a simplifled 2D axisymmetric finite element analysis has been performed to obtain stress distribution in the projectile. The linear elastic structural analysis with non-linear contact boundary has been employed for the projectile in the following two phases

| | |
|---|---|
| Phase-I: | Finite element analysis on a coarse model of full projectile |
| Phase-II: | Advanced submodel finite element analysis of high stress concentration region predicted by the Phase-I analysis. |

The objectives of these analyses have been to first obtain overall stress distribution in the projectile from a coarse model and then, to predict accurate stresses in the penetrator and sabot at high stress concentration region by using a submodel analysis. Both analyses have been carried out using commercial finite element analysis program ANSYS.

## FINITE ELEMENT MODELS

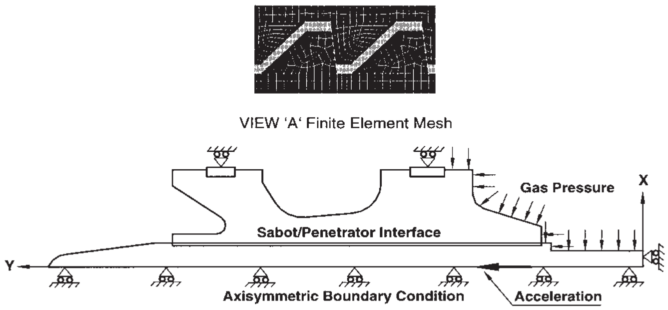

Figure 1.

Home
Table of Contents

## Finite Element Model for Phase-I Analysis

The axisymmetric model of the kinetic energy projectile with enforced boundary conditions is shown in Fig 1. This finite element model has been prepared using exact geometric features of parts, which include the thread details (excluding fillet radius) on the sabot and penetrator. The projectile is discretized with 4-node quadrilateral solid elements. The interface at the threads between sabot and penetrator is modelled using node-to-surface contact elements to allow for realistic load transfer from the sabot to the penetrator. The finite element model of the projectile consists of 35,603 nodes, 33,688 quadrilateral solid elements and 3,696 node-to-surface contact elements. A mapped meshing technique has been used to generate a good quality mesh. The quality of finite element mesh in the threads of the projectile is shown in the enlarged View ‘A’ in Fig. 1. Base pressure of 300 MPa and body force corresponding to translational acceleration of 75,000 g are applied on the projectile. The rollers along the central axis of the projectile display the axisymmetric boundary condition and the rollers at the bands show radial constraint imposed by the barrel. The projectile is constrained in y-direction at its rear end to prevent rigid body motion as well as to account for the inertial load due to the tail-fin. The material properties taken for structural analysis are given in Table 1.

Table 1

| Material | Density ($kg/mm^3$) | Modulus of Elasticity (MPa) | Poisson’s Ratio | Yield Strength (MPa) |
|---|---|---|---|---|
| Tungsten Alloy | 17.0 E-06 | 0.31E06 | 0.31 | 1050 |
| Aluminium Alloy | 2.8 E-06 | 0.6895 E05 | 0.33 | 520 |

## Finite Element Model for Phase-II Analysis

After careful study of Phase-I analysis results, it is found that very high von Mises stress has been induced in the vicinity of rear thread of the projectile. Thus, it would be appropriate to consider the two rearmost threads of the projectile for advanced submodel analysis.

Home
Table of Contents

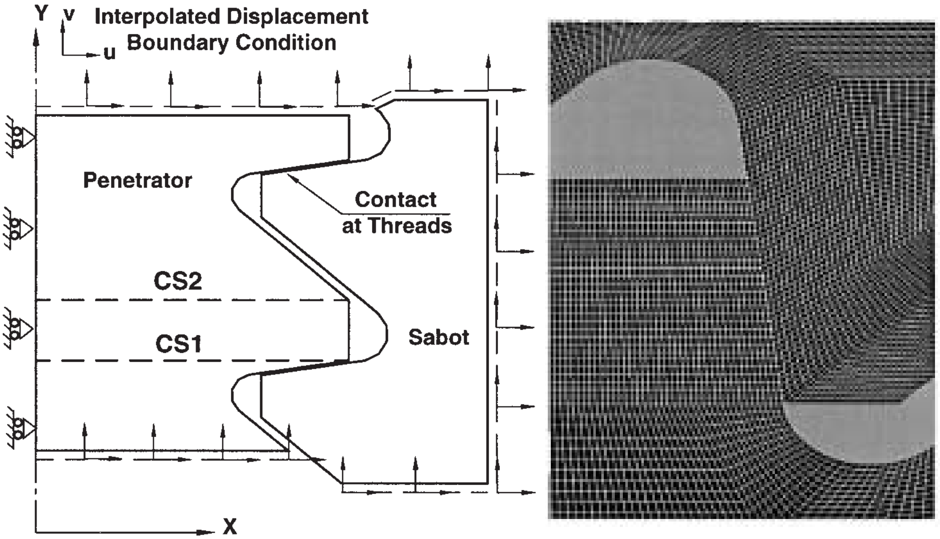

Figure 2. View 'B' Finite Element Mesh.

Fig. 2 shows the geometric shape and the enlarged view of finite element mesh for the thread contact portion of the submodel. The geometry considered for the submodel also includes the fillet radius of the thread. An utmost care has been taken to create finite element mesh of the submodel for obtaining accurate stress field in high stress concentration regions. The quality of finite element mesh developed for the submodel is depicted in the enlarged View 'B' in Fig. 2. The submodel is also discretized using 4-node quadrilateral solid and node-to-surface contact elements. The submodel consists of 35,669 4-node quadrilateral solid and 3,536 node-to-surface contact elements. The interpolated displacement boundary conditions obtained on the cut boundary of the coarse model from previous analysis are applied on the submodel. The interpolated boundary conditions are shown by arrows placed on dotted line in Fig. 2.

## RESULTS AND DISCUSSION

The overall distribution of various stresses in the full projectile has been obtained by performing Phase-I linear static structural analysis with non-linear contact boundary. The radial variation in axial, radial, hoop and von Mises stresses at the rear, middle and front threads of the penetrator are presented in Figs. 3, 4 and 5 respectively. The variation in axial and von Mises stresses along the centerline of the penetrator is shown in Fig. 6. The material used for the penetrator and sabot are ductile in behavior and follow von Mises failure criterion. Therefore, the magnitude of von Mises stress would speak about the possibility of material yielding under the applied loads. It is seen in Fig. 3 that the magnitudes of all stress components are predicted to be very low and constant in the core of the penetrator. But the axial and von Mises stresses at the rear thread are quite high due to very

Home
Table of Contents

high inertial load transfer and stress concentration. The radial, hoop and axial stresses are found to be tensile in the fillet region of the rear thread of the penetrator. The von Mises stress in the fillet region of the rear thread of the penetrator is predicted to be approximately 1600 MPa as shown in Fig. 3.

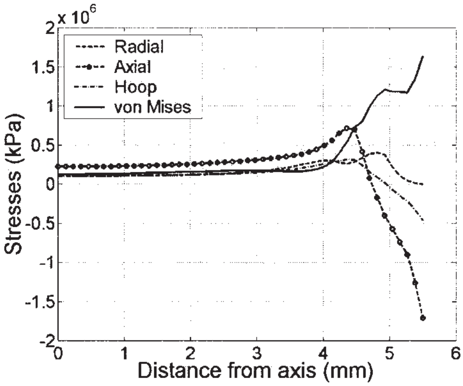

Figure 3.

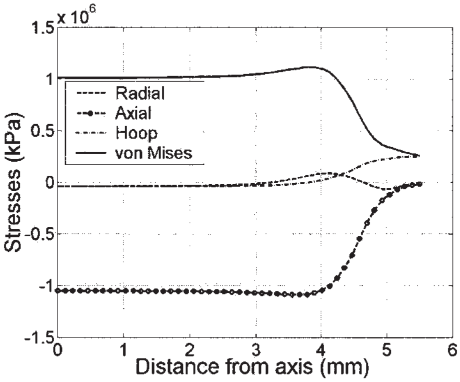

Figure 4.

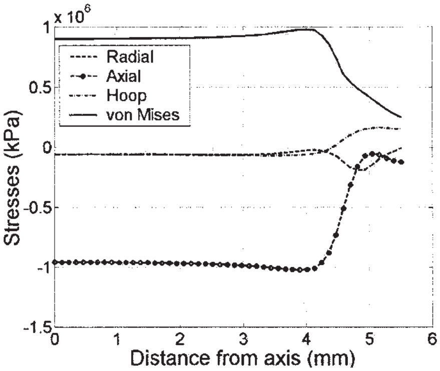

Figure 5.

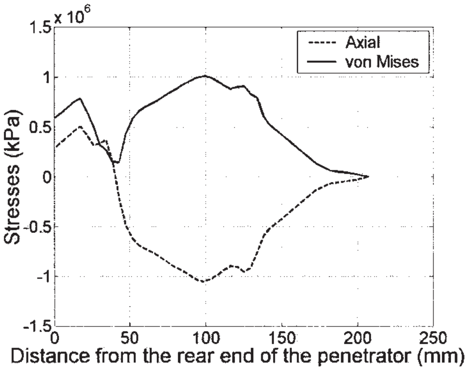

Figure 6.

Figs. 4 and 5 show the radial variation in axial and von Mises stresses in the middle and front threads of the penetrator. The distribution of stresses are predicted to be identical in both cases. The axial stress at the cross-section in the middle and front threads are found to be constant and compressive upto the thread root and then reduces to zero over the thread thickness. Fig. 6 shows that the rear portion of the penetrator core (up to rear thread) has been subjected to axial tensile stress due to pulling of the tail whereas the penetrator core ahead to rear thread has been subjected to overall axial compressive stress due to the pushing effect. It is also observed that the axial compressive stress in the core of the penetrator increases upto middle of the penetrator and then starts decreasing towards the tip.

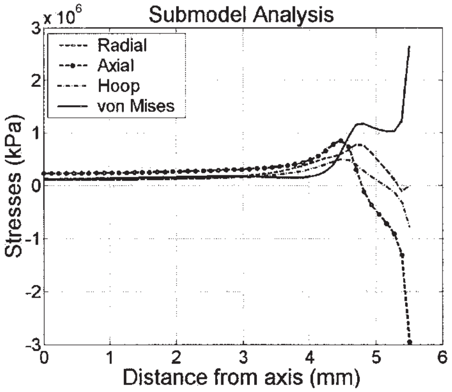

Figure 7.

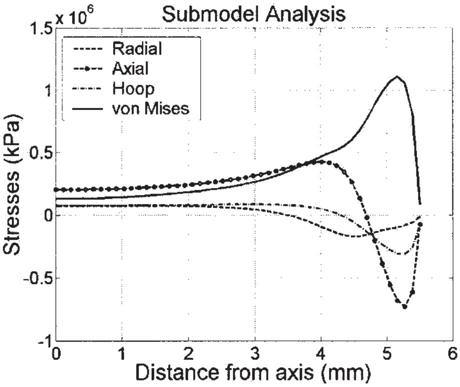

Figure 8.

Since, high stress concentration has been predicted in the fillet region of the rear thread of the projectile, an advanced submodel analysis for the rear thread portion of the projectile is performed in Phase-II analysis to obtain accurate stress distribution. Two cross-sections CS 1 and CS2 (see Fig. 2) located near contact and non-contact sides of the rear thread are considered for displaying radial variation of all stress components in the penetrator. The variations of radial, axial, hoop and von Mises stresses at two cross-sections in the rear thread of the penetrator are depicted in Figs. 7 and 8. In these figures, the most important observation is that there is very high spatial rate of change of the stress in the rear thread of the projectile. Although the stresses are very high at the fillet, they drop off rapidly with depth into the material of the penetrator and remains constant in the core. The maximum von Mises stress in the fillet region of the rear thread is predicted to be approximately 2600 MPa. The predicted value of the maximum von Mises stress is due to the assumed material linearity. Nevertheless, it indicates that some plastic deformation would take place in the fillet of the thread. Also, if some flaw is present at the rear thread fillet, a catastrophic failure may occur. Hence, to avoid permanent deformation or any catastrophic fallure, a change in the design of thread is envisaged.

## CONCLUSIONS

The finite element structural analyses on coarse model of full projectile and refined submodel of rear thread portion of the projectile have been performed. A 2D axisymmetric finite element model with proper contact definition at the threads between sabot and penetrator has been employed for performing structural analysis. The structural analysis has provided better understanding of overall distribution of radial, axial, hoop and von Mises stresses in the projectile. A very high concentration of stress at the fillet of rear thread of the penetrator has been predicted and the von Mises stress value is found to be more than double the value of stress developed in the core of the penetrator. The radial, hoop and axial stresses are found to be tensile in the fillet region of the rear thread of the penetrator. This is an alarming situation for opening out of micro cracks and giving birth to fracture initiation in the penetrator. High spatial gradient stress field has been predicted in the rear thread portion of the penetrator. Although, the stresses are very high in the fillet of the rear thread, they reduce rapidly to safe limit within 25 micron depth into the material of the penetrator and remain constant in the core. The maximum von Mises stress in the fillet region of the rear thread of the penetrator is predicted to be in the thread design.

## ACKNOWLEDGEMENT

Computational infrastructure of CADAS at ARDE was used for this work. Authors express their gratitude to the Director, ARDE for his valuable technical suggestions and granting permission to publish this paper.

## REFERENCES

1. W. H. Drysdale, “Design of Kinetic Energy Projectiles for Structural Integrity”, *Technical Report [BRL-TR-02365], Ballistic Research laboratory,* USA, 1981
2. G. A. Pflegl, J. H. Underwood, G. P. O’Hara, “Structural Analysis of a Kinetic Energy Projectile during Launch”, *Technical Report, US Army Armament Research and Development Command,* July 1981
3. Michael S. 1. Holis, “Use of Finite Element Stress Analysis in the Design of a Tank Cannon Launched Training Projectile”, *Technical Report, US Army Research Laboratory,* Sept. 1994

Home
Table of Contents