NONLINEAR IMPACT ANALYSIS OF PROTECTIVE PANEL STRUCTURE UNDER MISSILE BLAST LOADS

HANAMANT YARAGUDRI[1], CHETHAN H N[2],
[1]Assistant Professor, [2]PG Scholar,
Department of Mechanical Engineering, New Horizon College of Engineering,
Bengaluru, India-560103.

***Abstract*—** In the present work, a panel structure designed to take missile load need to be optimized for impact loads arising from pieces of blast. The blast loads triggers pieces of iron particles of size 2 kg with a velocity range of 22m/sec. So the panel structure design should consider all possibility of the blast loads for structural safety. The work is for an explosive testing center where missile explosions are tested. A separator panel structure is required to separate the various testing centers. So the design of panel should take the load of explosions.

***Keywords*—** *Impact,Panel Structure, Blast Load, software ANSYS.*

I.INTRODUCTION

Software increasingly includes design optimization tools that would not have been possible just a few years ago. Increased raw computing power has reduced “single” analysis run times so that multiple analysis runs can be performed in reasonable time. Improvements in CAD software that allow for geometry models that can be easily and automatically changed have also helped, allowing for schemes of multiple analysis to be run without human intervention. More over advances in convergence techniques have allowed for some reasonable degree of accuracy in these automatic schemes. But the optimization is not an easy process. It is like the design process itself - potentially infinitely complex and perfectly unique for each specific situation. Successful design optimization requires complete understanding of the software limitations, a willingness to simplify the problem as much as possible, and skill in deciding which factors are important in improving the design. It’s important to recognize that optimization problems increase in complexity exponentially with the number of variables used. While a two or three variable optimization scheme might seem like a simple problem, this is really close to the practical limit for any real FEA “pure” optimization today. Often only one variable is possible, a case where optimization is not even needed, since the same result can be obtained by performing a sensitivity study (as is often provided in FEA software) with the one variable or even by simply making a few different analysis runs individually. But the real work lies in working with multiple variables where problems are compounded.

II.PROBLEM STATEMENT

Analysis of the panel structure under blast loads for structural safety and design optimization is the main objective of the problem. The problem objectives include

- Computational fluid dynamic analysis to find the pressure acting on the panel structure
- Theoretical validation for the sections considered
- Finite element optimization of the panel design
- Assembly Analysis

III.REQUIREMENT

Many times, a blast in the assembly site creates lot of problem. This may results to damage of vehicles, building and workers. So proper design of the panel structures is very important for dynamic loads and also for static loads. Sufficient ribbing is required for strengthening of the members. Even sandwich panel design can also be considered for sufficient strength. Due to the advances in the finite element technology, design simulation helps the designer for proper selection of material, grade and thickness.

IV. METHODOLOGY

- Computational fluid simulation to find pressure acting on the surface of the panel.
- Structural design based on Shells and plate concept
- Finite element analysis for required thickness of the problem under blast and impact loads
- Geometrical built up and structural design
- Analysis and stress representation
- Report generation

V.DESIGN SPECIFICATIONS

- The panel structure should be of 6 meters width,6 meters length and 7 meters height.
- The material of the panel should be mild steel plates
- The blast pressure should be considered based on CFD evaluation.
- Maximum blast particle size is 2 kg in mass.
- Maximum velocity of the blast is 22 m/sec.

VI.CFD SIMULATION

Initially to find the blast pressure reaching to the panel structure, a computational fluid analysis is carried out using Ansys Flotran. Plane 142 element is used for analysis to find the velocity and pressure plots in a compressible air media.

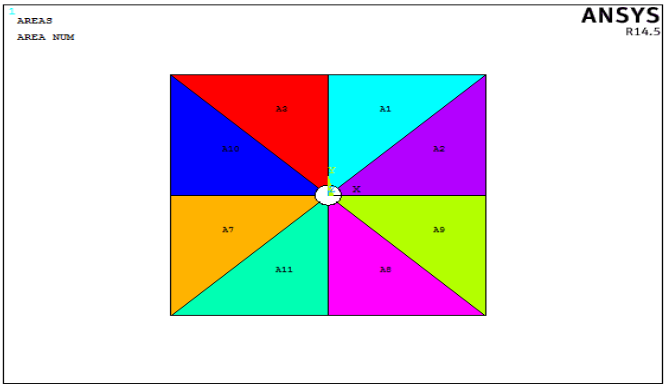

Figure 1: Split geometry for map meshing

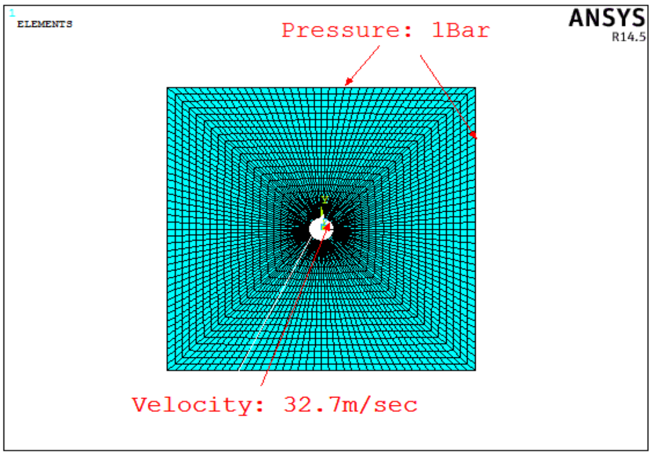

Figure 2: Map mesh of the geometry

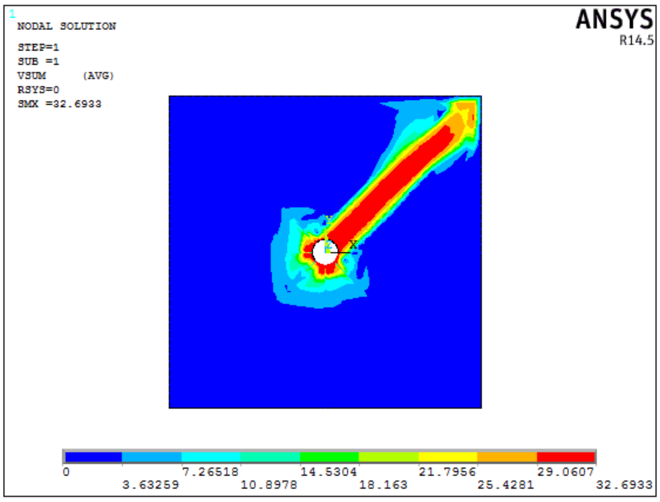

Figure 3: Vector plot

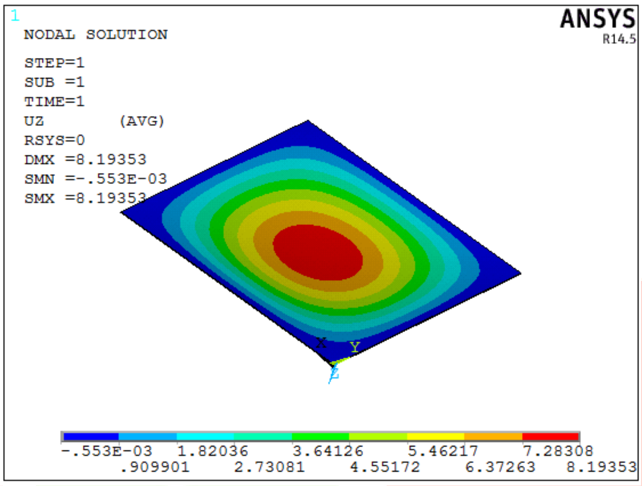

Figure 4: Deflection in the plate for 3mm thickness

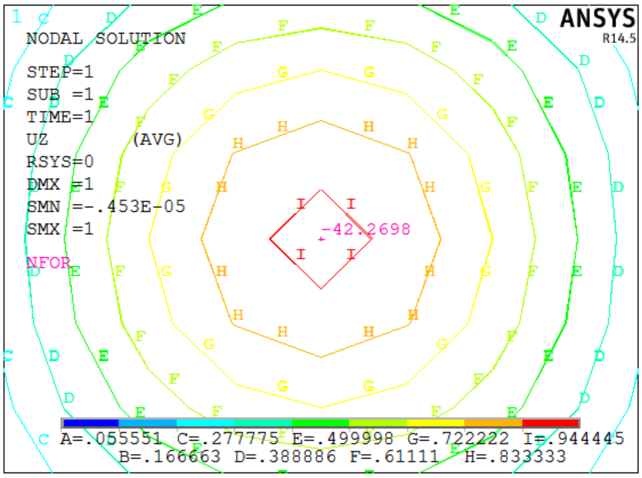

Figure 5: Reaction development at the center of the structure for unit displacement

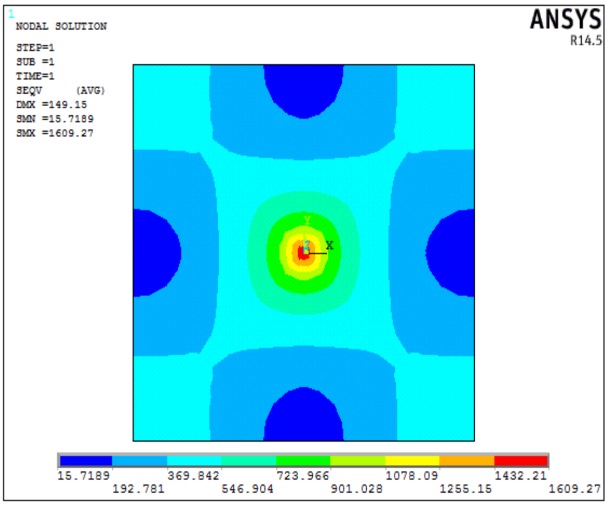

Figure 6: Von-Mises stress in the problem

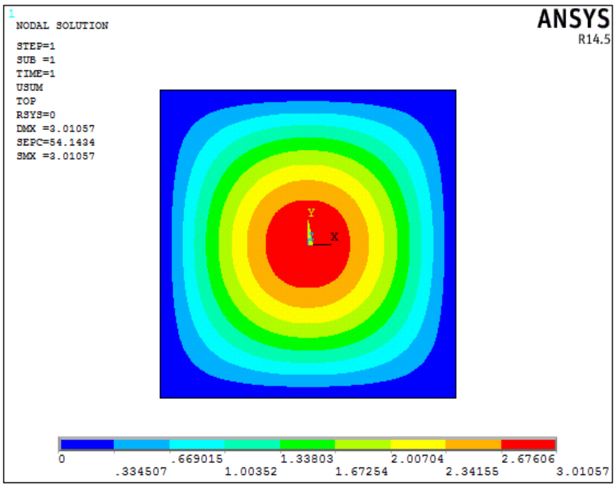

Figure 7: Displacement plot for the final iteration

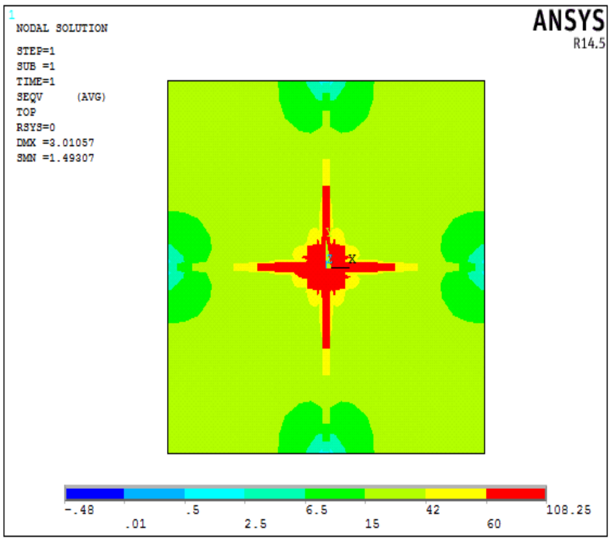

Figure 8: Von-Mises stress in the plate structure

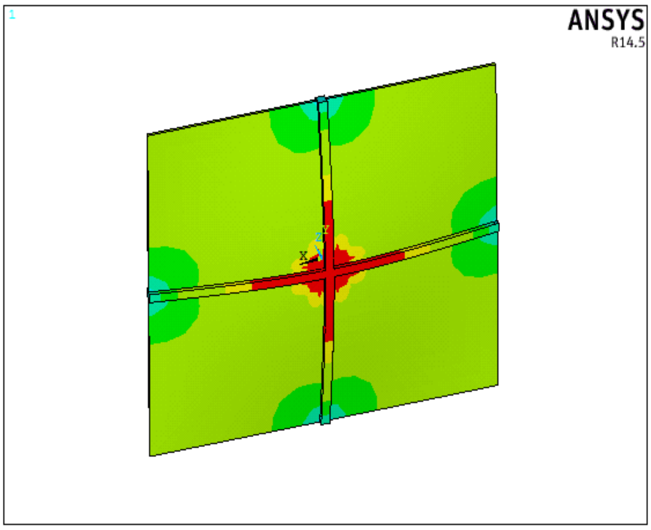

Figure 9: Stress in the rib structure

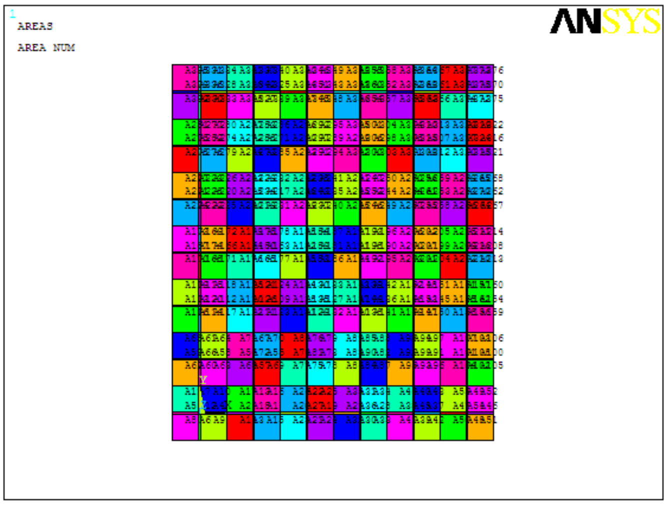

Figure 10: Full frame

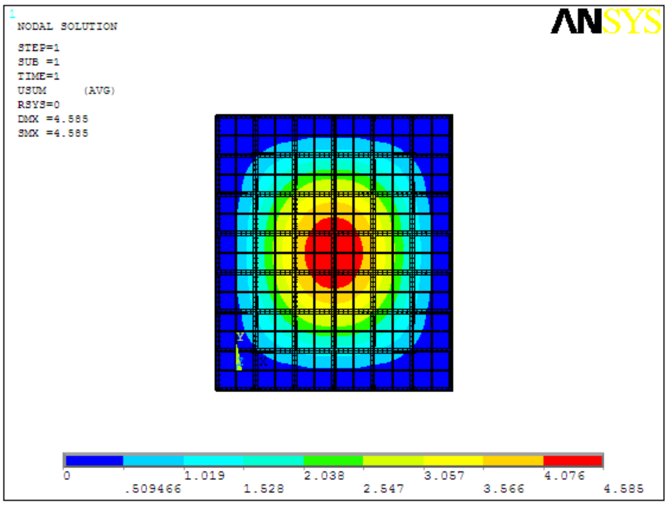

Figure 11: Overall displacement plot

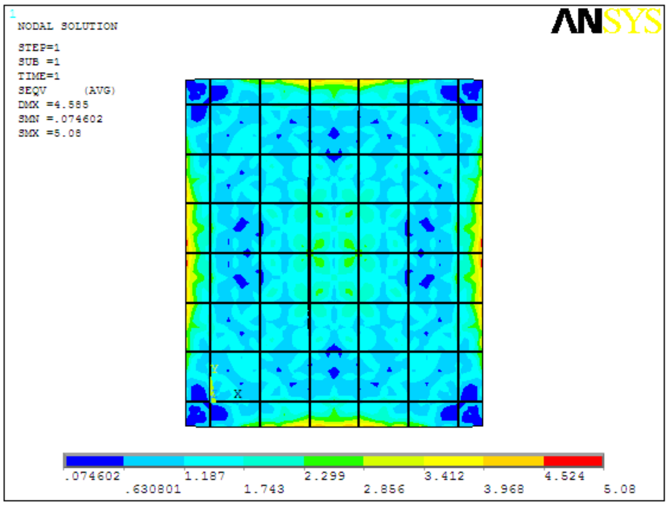

Figure 12: Stress in the plate material (maximum Stress: 5.08MPa)

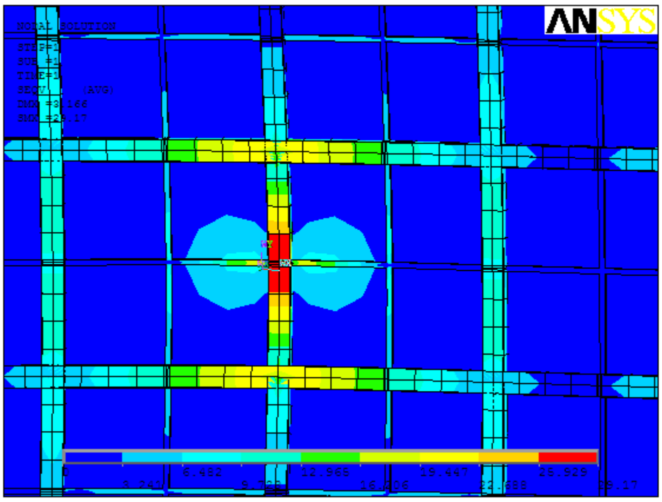

Figure 13: Stress corresponding to impact load

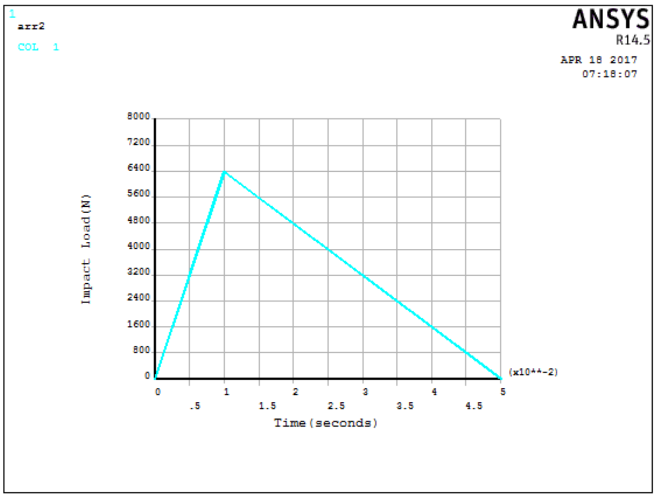

Figure 14: Load history curve

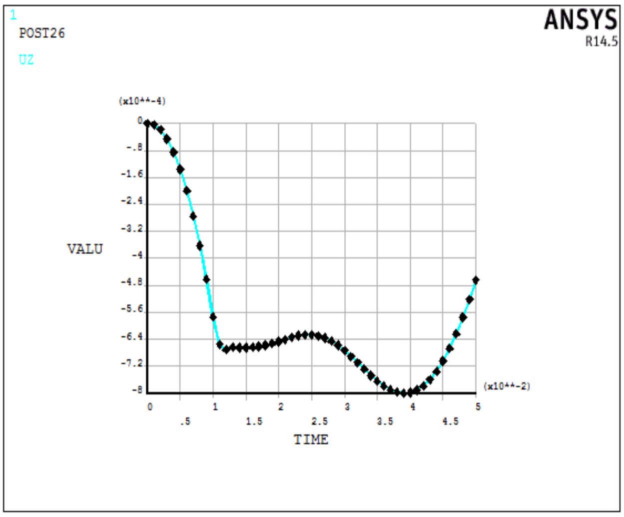

Figure 15: Displacement response with 2% damping

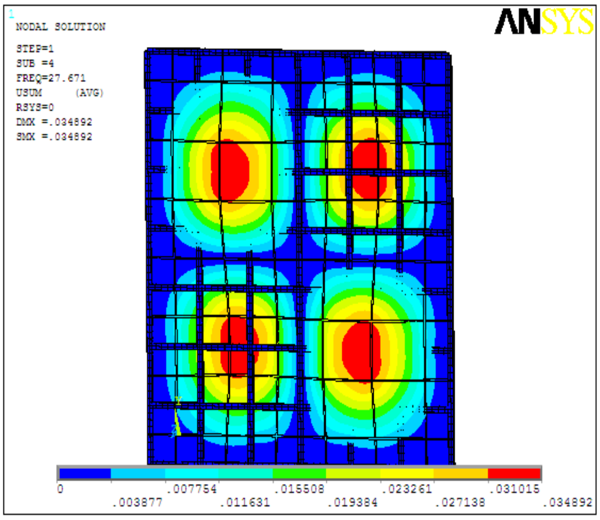

Figure 16: Shape corresponding to 4th natural frequency of 27.671Hz

VII.CALCULATIONS

Mass of the particle: 2 kg
Velocity of the component: 22m/sec
Assuming conversion of total KE to PE
$0.5\ Mv^2=0.5\ KX^2$
$2*22^2=42270X^2$
X=0.151m
F=KX=42270*0.151=6382.77N~6383N
Mass of 2 kg with a dynamic velocity of 22m/sec can create a load of 6382.77N in the localized region.
$1m^3$ volume of steel has 7800kg mass.
2 kg mass will occupy $0.256e\text{-}4m^3$
Assuming a spherical component of steel
$(4/3)\Pi r^3=0.256e\text{-}4m^3$
r=0.018m or 18mm radius
Diameter of Impact =36mm

VIII.RESULTS AND DISCUSSION

The panel structure need to be designed for safe working of employs inside blast testing center. So finite element analysis mixed with theoretical calculations are used to define model and analyses a panel structure safe working for blast loads. The overall summary is as follows.

Initially computational fluid analysis is carried out to find the blast pressure acting on the walls of the panel structure. A two dimensional model based on Ansys/Flotran is used to find the velocity and pressure acting on the panel structure. Plane142 element, a CFD element is used for analysis. From the CFD analysis, it is observed that 1000N load acting on every square meter area of the panel structure. Further structural analysis is carried out to find the thickness requirement of the panel structure. Initially for blast pressure load, a minimum 3mm thickness is required for the panel structure to prevent buckling of the plate. Further formulae's based on theory of plates and shells; the deflection value is validated from both finite element solution and theoretical solution. The results show excellent coherence between theoretical and finite element solutions. To find dynamic equivalent static load, unit deflection method is applied on the plate structure to find the stiffness of the plate. The result shows, stiffness of the structure equal to 42270 N/mm from finite element analysis. Based on the stiffness of the plate, dynamic equivalent static load is calculated and the value is around 6383 N for 2kg mass impacting with 22 m/sec velocity. Further impact area is calculated based on spherical component missile hitting the panel structure. The diameter of the impact object is found to be 36 mm diameter. Further panel structure is built with necessary loading area and analysis is carried out for the required thickness under impact loads. The geometry design is optimized considering rib width, thickness and plate thickness as design parameters. The design summary shows minimum requirements of the design parameters for safety. The rib width required is 25 mm, plate thickness is 8mm and center rib thickness 20.5 mm. The final results show complete safety of the problem for the given loads.

IX.CONCLUSION

Further bolt calculations are carried out to find the minimum size of the bolt. Total 8 bolts are considered for the problem and based on shear strength; the size of the bolt is fixed as 6mm. Even the finite element analysis shows safety of the assembly for the given loads with 8 bolts. The stresses and deformations are observed to be within the limits.

Finally complete assembly analysis is done for given loads using shell and beam elements. The results shows; assumed dimensions works for the given loading conditions as the deformation and stresses are well within the working range. Finally impact analysis is carried out in the transient domain along with nonlinear properties with large deformation effects and the results shows safety of the complete structure under maximum response conditions. Further modal analysis is carried out and obtained natural frequencies shows no possibility for resonance as the frequencies are much higher compared to the operational frequencies. Therefore no possibility for resonance in the system.

X. FUTURE SCOPE

- Harmonic response can be analysed.
- Possible thermal effects can be considered for structural safety.
- Possible spectrum conditions can be applied to find the dynamic response of the system.
- Composite usage can be checked.
- Buckling analysis can be carried out.

REFERENCES

1. R.J.Duffin, E.L.Peterson and C.Zener, Geometric programming: theory and applications, Wiley, New York.
2. G.B.Dantzig Linear Programming and Extensions, PrincetonUniversity Press.
3. R. A. Gettatly and R. H. Gallagher, “A procedure for automated minimum weight structural design”.
4. Part I - Theoretical basis, Part II - Applications,” Aero. Quart. Part I, Vol. 17, pp. 216-230 and pp.332-342.
5. M.M. Denn, Optimisation by variational methods, McGraw-Hill, New York .
6. G.S.G. Beveridge and R.S. Schechter, Optimisation : theory and practice, McGraw-Hill, New York.
7. J.L. Kuester and J.H. Mize, Optimisation techniques with FORTRAN, McGraw-Hill, New York.
8. M.J. Panik, Classical Optimisation: foundations and extensions, North-Holland Publishing Co., Amsterdam.
9. D. Koo, “Elements of Optimisation” Springer-Verlag, New York.