ELSEVIER

Finite Elements in Analysis and Design 35 (2000) 199–212

FINITE ELEMENTS IN ANALYSIS AND DESIGN

www.elsevier.com/locate/finel

# Finite element simulation of a twin-cam 16-valve cylinder structure

Shiang-Woei Chyuan[a,b,*]

[a]Department of Mechanical Engineering, National Taiwan University, Taipei, Taiwan, R.O.C.
[b]Chung Shan Institute of Science and Technology, P.O. Box 90008-15-3, Lung-Tan, Tao-Yuan 325, Taiwan, R.O.C.

**Abstract**

In the present paper, a new twin-cam 16-valve cylinder head and block structure accompanied with several important sub-components under firing load and assembly loads has been investigated using the finite element method. The finite element analysis was executed using a commercial finite element analysis software package, MSC/NASTRAN (trademark of MacNeal-Schwendler Corporation) and the finite element modeling and pre–post processing was carried out using a popular computer-aided engineering tool, I-DEAS (trademark of Structural Dynamics Corporation). The physical behavior of the upstand design of gasket bead and liner, the stiffness distribution of cylinder head, the preload of cylinder head bolts, the residual insertion loads of valve guides and valve seats, and firing pressure have been thoroughly discussed. Meanwhile, the sealing and structural response analyses under assembly and firing load cases highlighted several areas of interest. Suggestions obtained from this project have been forwarded to designer successfully, for incorporation of adjustment, and to other proper areas for design evaluation. 

*Keywords:* FEA; Upstand; Sealed fluid pressure; Cold assembly; Residual stress

## 1. Introduction

Recent developments in numerical techniques and computer simulation methods have resulted in a very prestigious progress in engineering analysis. With the increasing developments of digital

* Correspondence address: Chung Shan Institute of Science and Technology, P.O. Box 90008-15-3, Lung-Tan, Tao-Yuan 325, Taiwan, R.O.C. Fax: 886-347-13318.
*E-mail address:* yeaing@iris.seed.net.tw (S.-W. Chyuan)

0168-874X/00/$ - see front matter 
PII: S0168-874X(00)00002-0

computer power, finite element method [1,2] was considered to be one of the most powerful computer-aided engineering (CAE) design tools for engineers. Complex structural configurations can be modeled using finite elements and the response at any desired point of the structure can easily be determined. In the last, the finite element method has been evolving to be a widely accepted tool for the solution of pragmatic engineering problems.

In engine system design, the cylinder head structure is among the most complicated parts, and, numerical techniques are necessary to simulate the physical behavior and to evaluate the structural integrity of the different designs, and, to minimize the cost of product development. In the automobile system, the cylinder structure is the most important component affecting the performance of a car and it can be divided into two components: cylinder head and cylinder block accompanied by other important sub-components such as: combustion chamber, spark plug, cams, oil rail, cold-water passage, cylinder head bolts, inlet valves, exhaust valves, valve seats, valve guides, gasket, liner … etc. A gasket bead and body and liner are fastened by cylinder head bolts between the cylinder head and block in order to ensure that the prevailing gasket pressure should exceed the sealed fluid pressure of about 1.0 $Kgf/mm^2$ ( = 1421 psi) by at least 10%. The gasket–liner interaction is completed by the wet liner design where different upstand values (The height of gasket bead is higher than the height of gasket body and the height of liner is also higher than the height of cylinder block in order to get a higher internal normal stress in the area of gasket bead under cylinder bolt preload) are chosen for both components. The use of critical cases for all components would be conservative for the head and unconservative for the gasket sealing performance. A compromise was, therefore, chosen which used a harder gasket with a moderate, combined liner and gasket upstand. For designers, lower cylinder head bolt preload will result in inadequate gasket normal pressure response and affect the performance of cylinder system. Higher assembly bolt preload may result in over gasket pressure and damage the fragile gasket bead and body and liner. Therefore, the selection of an appropriate preload value of cylinder head bolts becomes a design task for the engineer. In addition, in the cylinder head structure, there are valve guides and valve seat inserts fasten by cold assembly. (Before the valve guides and valve seat inserts were put into the parent bore in the cylinder head, high temperature was added in the areas of parent bore to enlarge the inside diameter for assembly.) For the structural cases, the nominal design interference was modeled by applying proper thermal loads to the inserts and the valve guides. This makes them expand from the nominal parent bore diameters to the required nominal insert diameters and nominal guide diameters. Therefore, the valve inserts and guides could be tied to the parent bore in the cylinder head; radially over the whole circumferential contact area and axially. It should be noted that valves are also the inevitable sub-components of cylinder head. In the load path design of cylinder head, inlet valves apply gas load through the valve head, to the valve seat, and the inlet and exhaust valves were tied to the corresponding valve seats, normal to the seat, over the whole circumferential contact area. From the above statements concerning cylinder structure, one can see that it is a very difficult and laborious task to predict the physical response of cylinder system during the design phase. Therefore, the use of computer simulation technique to analyze the sealing and structural behavior of cylinder structure in the preliminary design stage is very important and necessary. In this paper, a successful and feasible numerical model was used and discussed thoroughly.

In engineering analysis, a theoretical model was the first choice for researchers and scientists because of totally correct and unique solution. But in the pragmatic design problem, the theoretical

model was scarcely utilized to predict physical response because of the very complex geometrical design and load path. Therefore, a powerful numerical method was introduced to analyst to face the difficulty. Among different numerical approaches, finite element method (FEM) and boundary element method (BEM); and, the increasing developments of digital computer power have moved from being research tools for select groups to become powerful design tools for engineers. For the boundary element method [3], the easy data preparation due to one dimension reduction makes it attractive for special practical use. For problems with singularities (for example: seepage flow problems, crack … etc.), it is well known that BEM accompanied with dual integral formulation [4] became a very effective analytical model. However, the main drawback of BEM is that it is very difficult to apply in the field of extremely complicated design. In the sealing and structural analyses of complex geometrical cylinder structure, BEM is difficult to apply. Therefore, FEM has become the most widely used numerical technique for analysis of cylinder systems because the complex structural configurations can be modeled using finite elements (for example: solid rocket motor [5]) and the response at any desired point of the structure can easily be determined [6].

## 2. Finite element modeling and loading analysis

### *2.1. Fundamentals of finite elements and constraints*

In linear elastic static analysis by the displacement method, stiffness and mass properties (used in the generation of gravity and inertia loads) are input either as properties of elements or as properties of grid points. In finite elements, constraint elements are very useful and convenient for specifying the special boundary condition. In the finite element model of cylinder structure, there are several contact areas (for example: valve seat inserts, valve guides, gasket, liner, cylinder head bolt, … etc.) concerning multi-point constraints. Therefore, constraints are employed in this paper for the following purposes: (1) to specify the prescribed enforced displacements (2) to simulate the continuous behavior of displacement in the interface area (3) to enforce rest condition in the specified directions at grid points of reaction.

### *2.2. Solid modeling and meshing of cylinder structure*

Because of the complexity of geometrical design and load path of cylinder head, it is not easy to model the complicated stiffness distribution of cylinder structure using simply analytical model. Therefore, a 3-D solid model (Fig. 1) was chosen for the twin-cam 16-valve cylinder system in order to predict the stress and strain response in detail. Since the engine is symmetrical, a quarter ($\frac{1}{4}$; 4-valve) model of the cylinder structure with specified boundary condition on the cut faces was used. The density on some critical areas (for example, the contact areas between valve seat inserts and cylinder head) is much higher than on other sub-critical parts in order to reduce the number of degrees of freedom. Finally, a quarter finite element model of cylinder structure with 10 608 solid elements (CHEXA; CPENTA) accompanied with some plate and beam elements and 16 947 grid point nodes was finished as shown in Fig. 2a (The sub-model of cylinder head is shown in Fig. 2b, the sub-model of cylinder block and liner is shown in Fig. 2c, the sub-model of valves, valve seats and valve guides is shown in Fig. 2d, and the sub-model of gasket bead, gasket body and cylinder

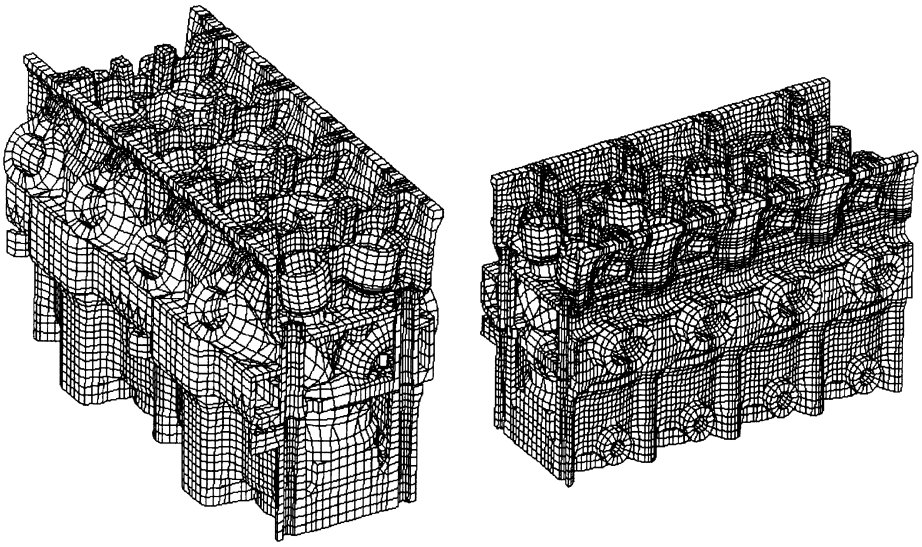

Fig. 1. A thorough model with 3-D solid element mesh for the new twin-cam 16-valve cylinder structure.

head bolts is shown in Fig. 2e.) The number of degrees of freedom of this quarter model is 51 720. The finite element analysis was executed on an IBM590/RS6000 computer system, and the CPU time for a static run under a single load case was 336.62 s. When a personal computer system with Pentium Pro 200 processor was used, the CPU time required for a static run under a single load case was 369.642 s. However, when a personal computer system with Pentium II 233 processor was used, the CPU time required for a static run under a single load case was 243.016 s.

### 2.3. Boundary condition of cylinder structure

Because the load path is different for the different loading configurations, the prescribed boundary condition types under diverse loads are also changed case by case. Constraint equations were used to ensure that loads associated with the bolts, valve seat inserts, valve guides, valves, gasket and block were applied. Some of the uses of single- and multi-point constraints in this paper are given in the following sections.

#### 2.3.1. Cylinder head and block under firing load case

Nodes on the cut face of the single bore model were restrained by single-point constraints (SPCs) in the basic $Y$-direction. This constraint equation type gives the effect of a repeated symmetrical model, where the loads produced at one cut face are applied to the opposite face. The firing load case, using symmetry conditions, would simulate the effect of all cylinders firing. This analytical condition is not realistic and leads to an overestimation of the stresses and strains near the cut faces of both sides. To correct this, an anti-symmetric load case is introduced to give opposing forces on adjacent cylinders by using SPCs in the basic $X$- and $Z$-direction. A scaled combination of the two half-firing-load cases gives the effect of only one cylinder firing with loads canceling on adjacent cylinders.

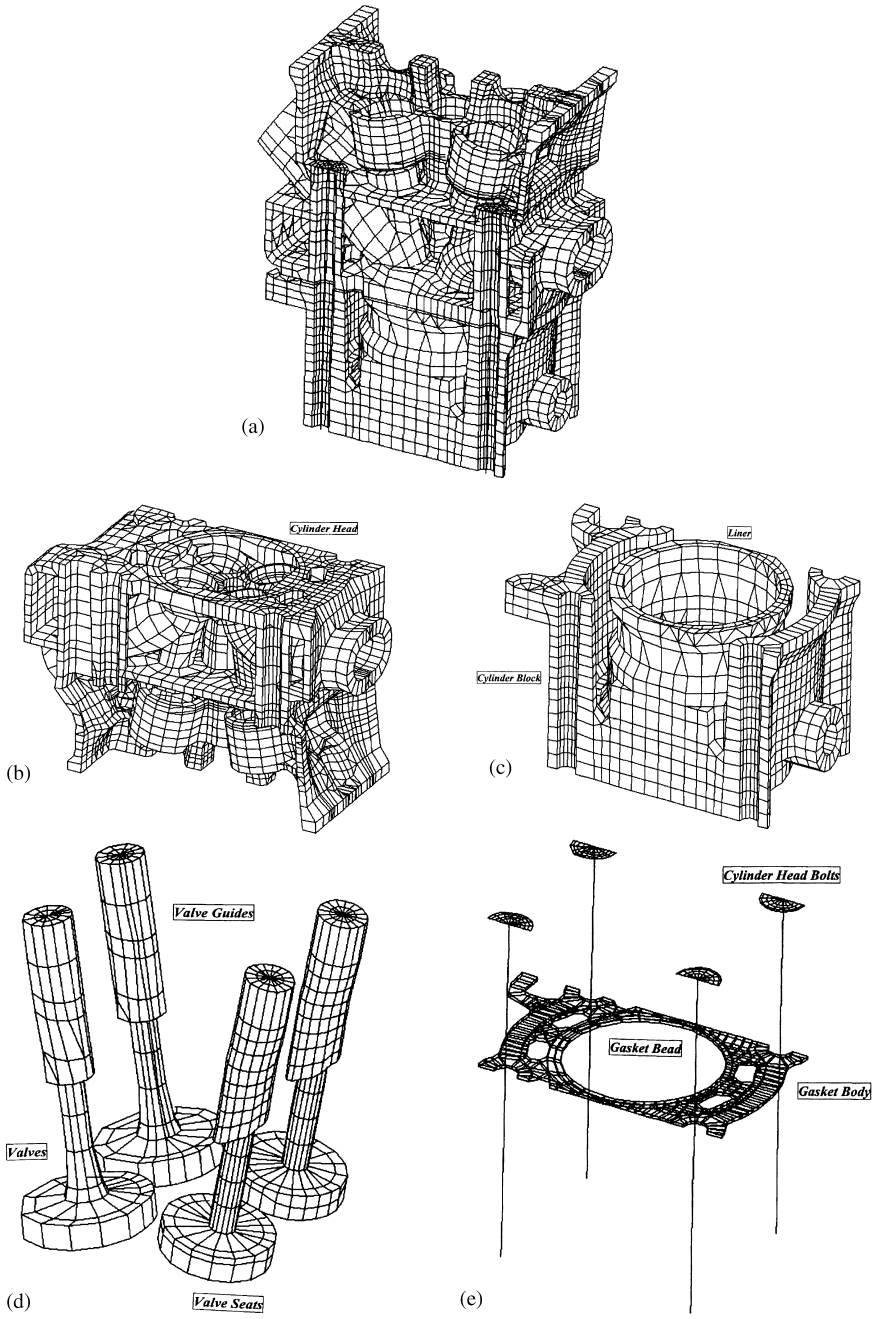


Fig. 2. (a) A quarter ($\frac{1}{4}$) finite element model of cylinder structure; (b) A quarter ($\frac{1}{4}$) finite element model of cylinder head; (c) A quarter ($\frac{1}{4}$) finite element model of cylinder block and liner; (d) The finite element model of valves, valve seats and valve guides; (e) The finite element model of gasket bead, gasket body and cylinder head bolts.

#### 2.3.2. Cylinder head and block under cylinder bolt preload case

From the viewpoint of machine design, the bolts were constrained to the head, at their top end, and to the block, at their bottom end. At the bolt head, the loads are applied from bolt to the load washer and from washer to cylinder head in the axis of bolts (vertical direction; axis of bolts) by MPCs accompanied with some proper artificial factors (1.0, − 1.0). The bolts are restrained to the washers, and hence to the head, in local $Y$ using two corresponding node pairs on the cut faces but on opposite sides of the component. This ensures no relative rotation of the components in the local $Z$ plane. At the bottom end, the thread region of the bolt is also tied in the axis of bolts by MPCs accompanied with some proper artificial factors (1.0, − 1.0). As with the bolt heads, the threads are also tied in local $Y$ along vertical corresponding node pairs. The nodes on the cut faces of the bolts, head and block were constrained to the symmetrical boundary condition using SPCs like the firing load case. This condition satisfies the repeated symmetry requirements on the cut faces under cylinder bolt preload case. As for the gasket, bolt preload is transmitted between the bolt and head, by means of washers, bolt and block, mainly in the vertical direction. Therefore, the liner and gasket model was tied to the cylinder head with some upstand in the normal direction using MPCs accompanied with some proper artificial factors (1.0, − 1.0) to simulate the displacement continuous response of corresponding node pairs.

#### 2.3.3. Valve seat inserts and valve guides under cold assembly loads case

Displacements at nodes on the interface contact areas between the valve seat insert or valve guide and the parent bore, in the head, were defined in the local coordinate systems centered on the relevant valve. The valve seat inserts and valve guides were then restrained to the parent bore with MPCs accompanied with some appropriate artificial factors (1.0, − 1.0) between corresponding nodes, radially over the whole circumferential contact areas for simulating the displacement continuous response of corresponding node pairs. Therefore, the analytical assumption can simulate correctly the physical behavior of cylinder structure under cold assembly loads case.

#### 2.3.4. Valves under firing load case

Structurally, the valve was used to apply firing gas loads to the valve seat. The valve seat interface nodes were constrained with MPCs accompanied by some appropriate artificial factors (1.0, − 1.0), normal to the contact face between seat and valve, over the whole circumferential contact area, for simulating the displacement continuous response of corresponding node pairs. This was achieved using the local spherical coordinate systems, one for each level of nodes in the seat.

### 2.4. Loading analysis of cylinder structure

#### 2.4.1. Firing load case

A firing gas load was applied to the cylinder head to represent the maximum cylinder pressure found during the power curve. This consisted of a scaled combination of the symmetric and anti-symmetric load cases. PLOAD4 which simulate distributed pressure was used in the valves areas to simulate the effect of firing load case.

*2.4.2. Assembly loads*

The assembly loads comprised the valve seat insert and valve guide residual insertion loads due to cold assembly, the cylinder head bolt preload and upstand of gasket bead and liner.

*2.4.2.1. Cylinder head bolt preload case.* The bolt preload which was specified by the designed engineer, was taken as the largest predicted load, at the time, to apply the maximum loads to the cylinder head structure. The temperature distribution was specified by TEMPs for temperature values relative to a reference temperature together with appropriate prescribed boundary condition were used in the model of bolts to generate the required axially compressive loads. This temperature value has to be iterated several times during initial verification runs since it is dependent on the stiffness distribution of cylinder head and block structure.

*2.4.2.2. Valve seat insert and valve guide residual insertion loads case.* Interference on the inserts and guides was taken to be some design values. TEMP input was used to define the proper temperature values relative to a reference temperature with proper prescribed boundary condition which was used in the inlet valve seat inserts, exhaust valve seat inserts and valve guides to produce the required local radially tensile loads.

*2.4.3. Upstand of liner and gasket bead*

The structural analysis used in the upper half of the block, complete with liner, to react the cylinder head preloads. The liner upstand was taken to be incorporated in the nominal gasket bead upstand, which compensated the stiff gasket used to obtain higher internal head stresses and to generate consistent gasket sealing conditions. The upstand of liner and gasket bead was taken to be some value provided by the design engineer, to get sufficient gasket normal pressure for lower bolt preload in order to ensure that the prevailing gasket pressure should exceed the sealed fluid pressure of about 1.0 $Kgf/mm^2$ by at least 10%. TEMPs were used to specify the temperature field in the liner and gasket bead areas to get the required normal compressive loads.

## 3. Finite element analysis

### *3.1. Stress and strain analysis of cylinder structure*

*3.1.1. Firing load case*

Under the firing load on the valve areas, the displacement distribution of cylinder structure illustrated in Fig. 3, the maximum value is 0.0356 mm. Meanwhile the stress distribution of cylinder head can be found in Fig. 4, the maximum principal stress value is 6.25 $Kgf/mm^2$ located at the local area between inlet valve seat inserts and exhaust valve seat inserts due to local bending moment and the minimum principal stress value is $-8.26$ $Kgf/mm^2$ located at the interface areas between valve seat inserts and cylinder head because of local compressive loads. The application of firing load to the combustion chamber forces the cylinder head away from the gasket, liner and cylinder block, effectively reversing the cylinder bolt preload case. The intermediate deck, around the valve guides, is pushed up in both firing load and assembly loads cases. Engineering experience suggests that this region is prone to structural failure because of firing load if detailed design or

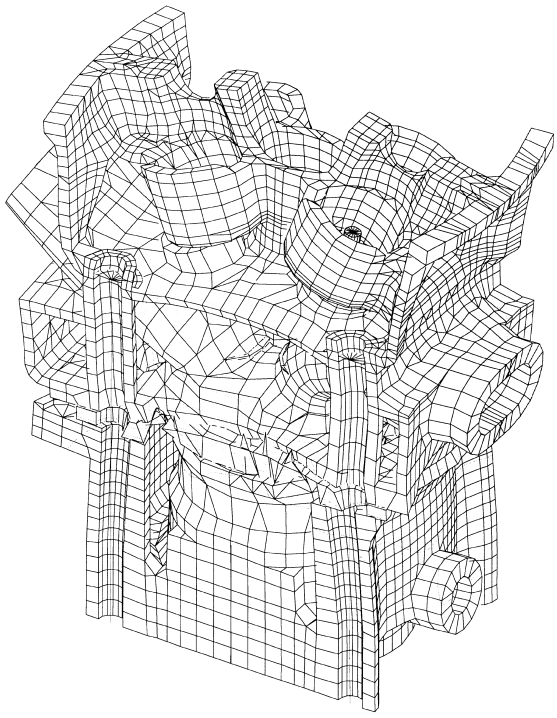

Fig. 3. The displacement distribution of cylinder structure under the firing load on the valve areas, the maximum value is 0.0356 mm.

material specification is not carefully specified. The narrow bridge of deck between the valve guide boss and the oil rail should be free of stress raisers on both sides by using generous fillets. In addition, the surface finish in this area should be carefully investigated during quality assurance procedure to ensure that porosity or sand inclusions are avoided. The cold water passage in this area may be reduced to give a local thickening of the deck or the upper surface may be corrugated to give the same effect.

### *3.1.2. Assembly loads case*

Under the assembly loads on the cylinder head bolts, valve seat inserts and valve guides, the maximum displacement value is 0.178 mm. Meanwhile the stress distribution of cylinder head can be found in Fig. 5, the maximum principal stress value is 25.00 Kgf/mm$^2$ located at the interface areas between valve guides and cylinder head because of local bending moment and the minimum principal stress value is $-28.80$ Kgf/mm$^2$ located at the interface areas between cylinder head bolts and cylinder head due to local compressive loads. The bolt bosses show the largest deformation under assembly loads and also the highest stresses and strains. The high stresses are predominantly compressive and originate from the cylinder head bolt preload. It can be found that the local stresses, at the bolt contact points, are near the allowable strength of cylinder head design material. These high stresses are, however, very local. The loads are applied through models of the bolt heads and washers which offer a realistic stiffness and the results may be considered correct.

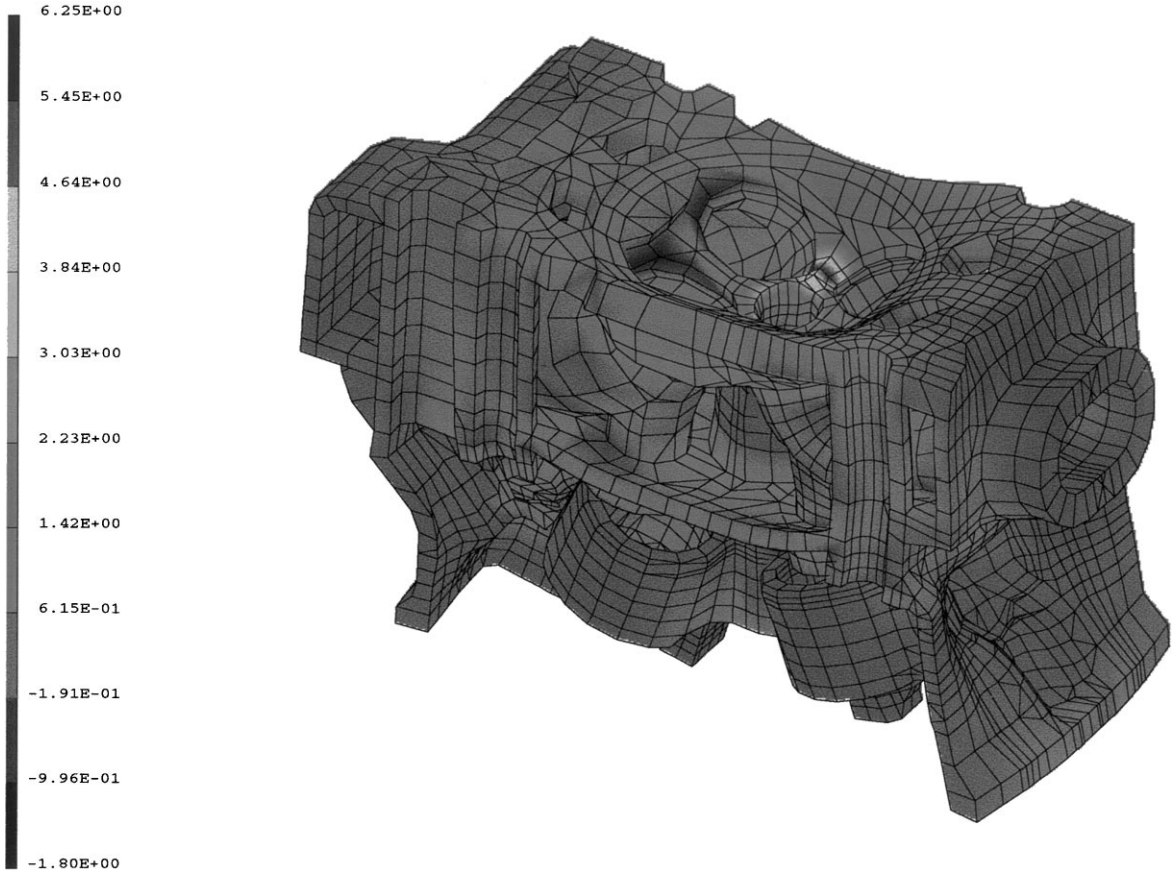


Fig. 4. The maximum principal stress distribution of cylinder head under the firing load on the valve areas, the maximum principal stress value is 6.25 Kgf/mm$^2$ located at the local area between inlet valve seat inserts and exhaust valve seat inserts due to local bending moment.

Hence, the bosses may be expected to yield locally under much higher bolt preload during assembly procedure between cylinder head and block. This is not a problem since the load will be more effectively redistributed across the boss cross section as a result and very local yielding on the interface areas between cylinder head and bolts is permitted in the viewpoint of pragmatic engineering for appropriate gasket normal pressure design goal.

### *3.1.3. Assembly/gas-combined loads case*

Under the assembly/gas-combined loads on the valves, cylinder head bolts, valve seat inserts and valve guides, the maximum displacement value is 0.200 mm. Meanwhile the stress distribution of cylinder head can be found in Fig. 6, the maximum principal stress value is 25.20 Kgf/mm$^2$ located at the interface areas between valve guides and cylinder head because of local bending moment and the minimum principal stress value is $-30.40$ Kgf/mm$^2$ located at the interface areas between cylinder head bolts and cylinder head due to local compressive loads. Therefore, the basic design

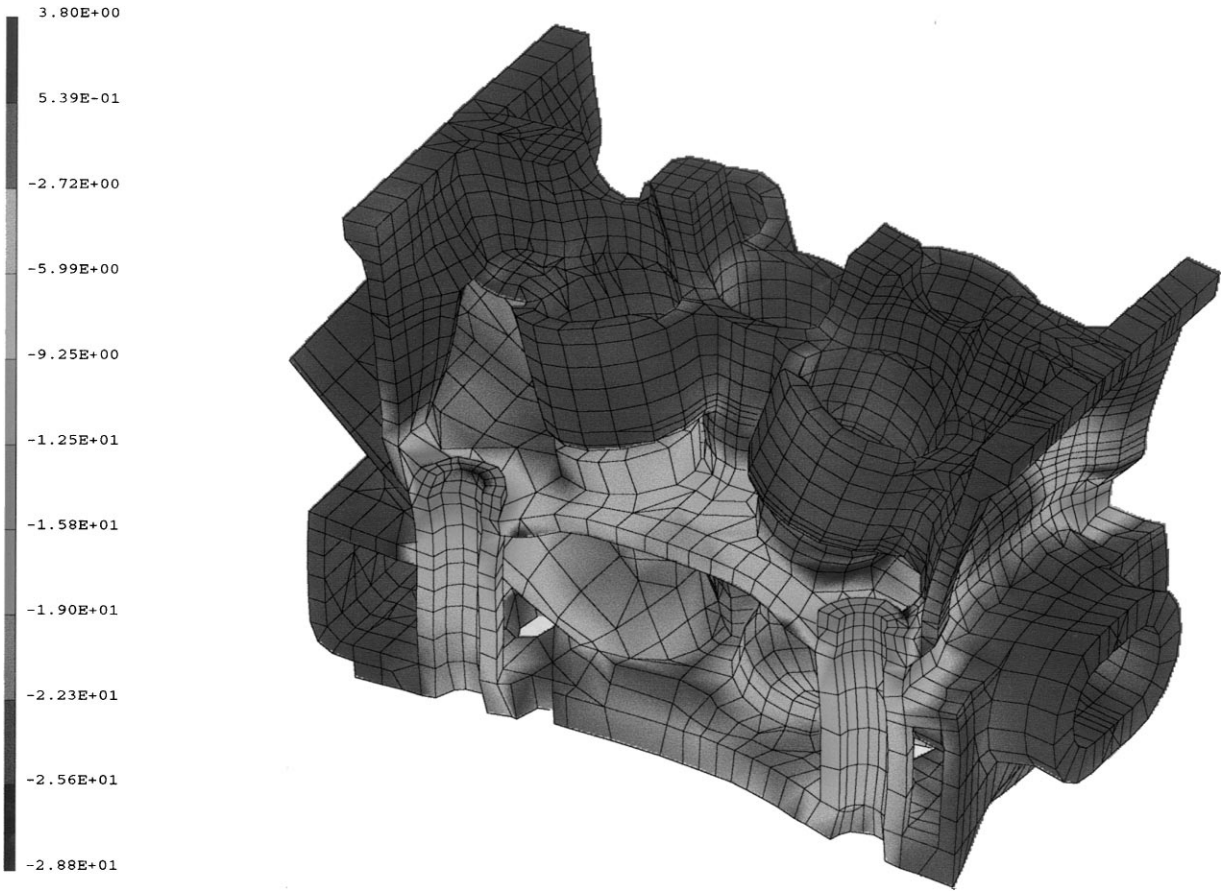


Fig. 5. The minimum principal stress distribution of cylinder head under the assembly loads on the cylinder head bolts, the minimum principal stress value is $-28.80$ Kgf/mm$^2$ located at the interface areas between cylinder head bolts and cylinder head due to local compressive loads.

goal that the maximum principal stress value of cylinder structure should not exceed the allowable strength of cylinder head design material about 32.5 Kgf/mm$^2$ can be satisfied.

### 3.2. Sealing analysis of cylinder structure

#### 3.2.1. Assembly loads case

Under the assembly loads case, the normal stress distribution of gasket can be found in Fig. 7, the maximum normal pressure value is 2.33 Kgf/mm$^2$ located at the internal gasket bead area because of the upstand of liner and gasket bead.

#### 3.2.2. Assembly/gas-combined loads case

Under the assembly/gas-combined loads case, the normal stress distribution of gasket can be found in Fig. 8, the maximum normal pressure value is 1.67 Kgf/mm$^2$ located at the internal gasket

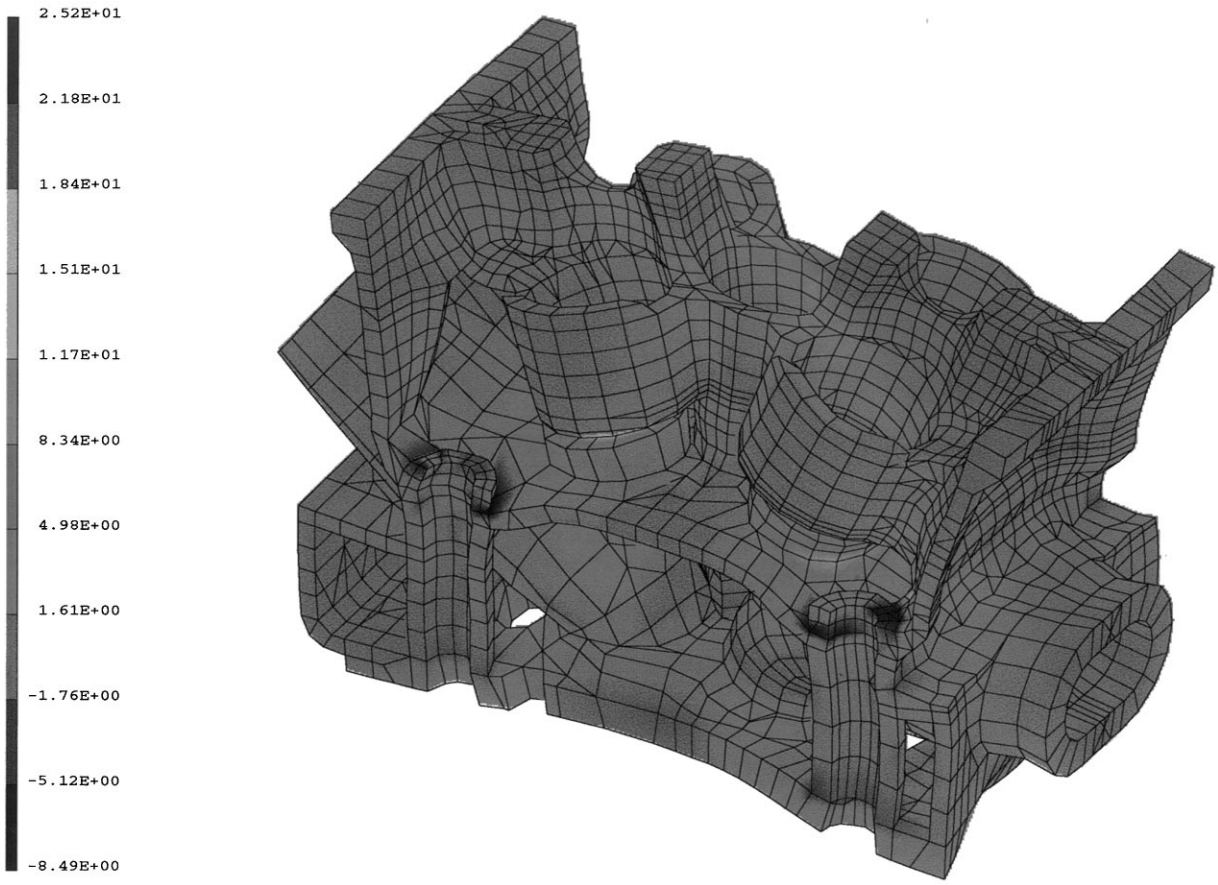


Fig. 6. The maximum principal stress distribution of cylinder head under the assembly/gas-combined loads, the maximum principal stress value is 25.20 Kgf/mm$^2$ located at the interface areas between valve guides and cylinder head because of local bending moment.

bead area due to the upstand of liner and gasket bead. In this case, the maximum normal pressure value is lower than that of the assembly loads case because the load path of firing gas pressure is opposite with cylinder head bolt preload. Though the maximum normal pressure value of this case is lower, the design goal that the prevailing gasket pressure should exceed the sealed fluid pressure about 1.0 Kgf/mm$^2$ by at least 10% can be achieved.

## 4. Conclusions and recommendations

(1) An elaborate and extensive structural and sealing analysis of cylinder system under firing gas load and assembly loads case was carried out using a commercial analysis FEA software package [7,8] with a CAE pre–post processor [9]. A 3-D solid model was adopted to obtain detailed analysis results. The structural and sealing analyses highlighted several areas of interest.

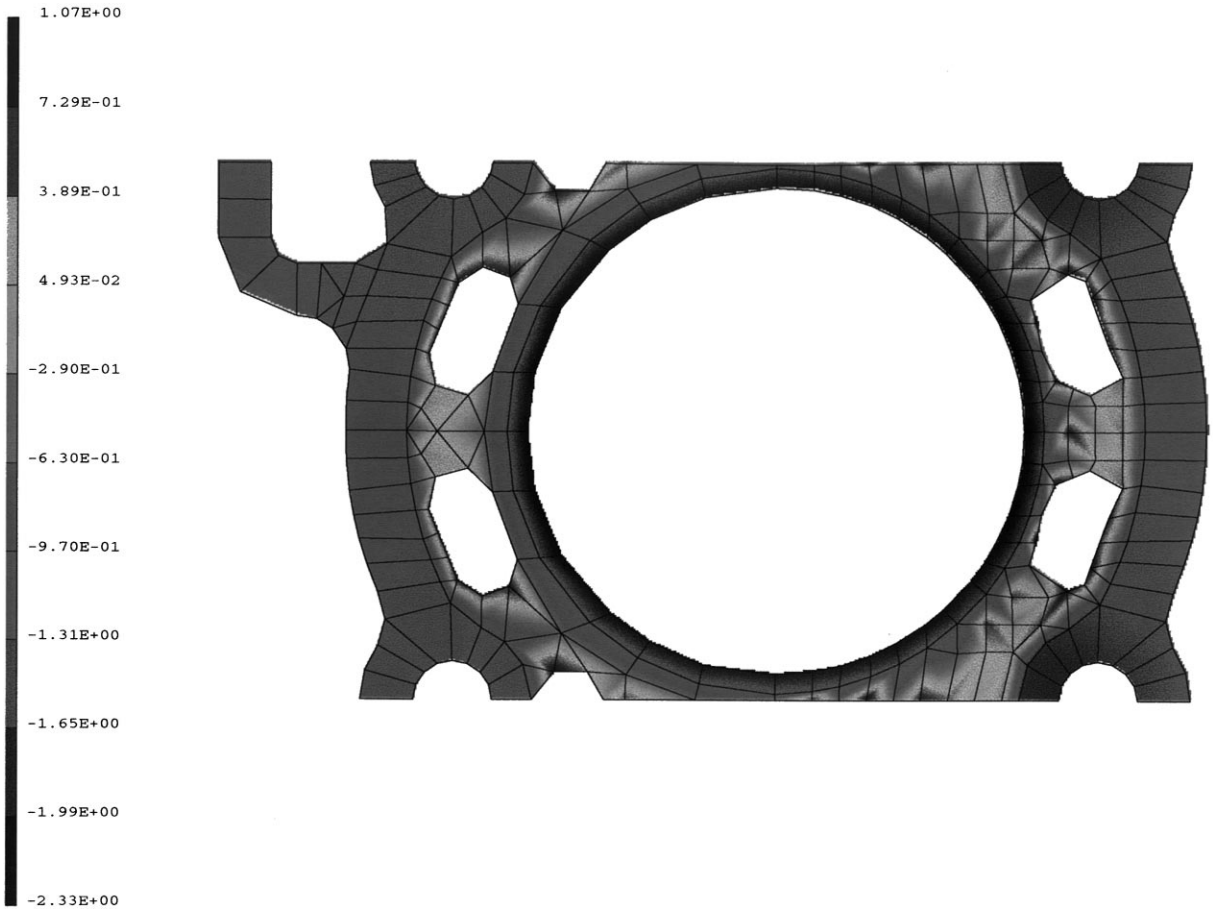


Fig. 7. The normal stress distribution of gasket under the assembly loads case, the maximum normal pressure value is 2.33 Kgf/mm$^2$ located at the internal gasket bead area because of the upstand of liner and gasket bead.

Recommendations resulting from this work have been forwarded to designer, for incorporation of adjustment and modification, and to other appropriate areas for design evaluation.

(2) The small bridge of metal between the tappet housing and the inlet port was found to be more highly loaded in the assembly/gas-combined loads case. The assembly loads case generates stresses which are largely static; however, the firing load case, is dynamic and has a very high rate of repetition. Therefore, firing load-induced stresses, while lower in magnitude than assembly loads case, have a higher damage potential. Therefore, surface finish should be investigated and monitored using the nondestructive test (NDT) during quality checks and development in detail because a small initial crack will result in horror behavior of rapid crack growth and fatigue failure under the cyclic dynamic loads.

(3) Because the configuration of cylinder head is too complex to predict the critical area and failure mode in the design phase, the finite element simulation becomes the best method to obtain the stress distribution under different loading cases. Without the analysis on the project, designers

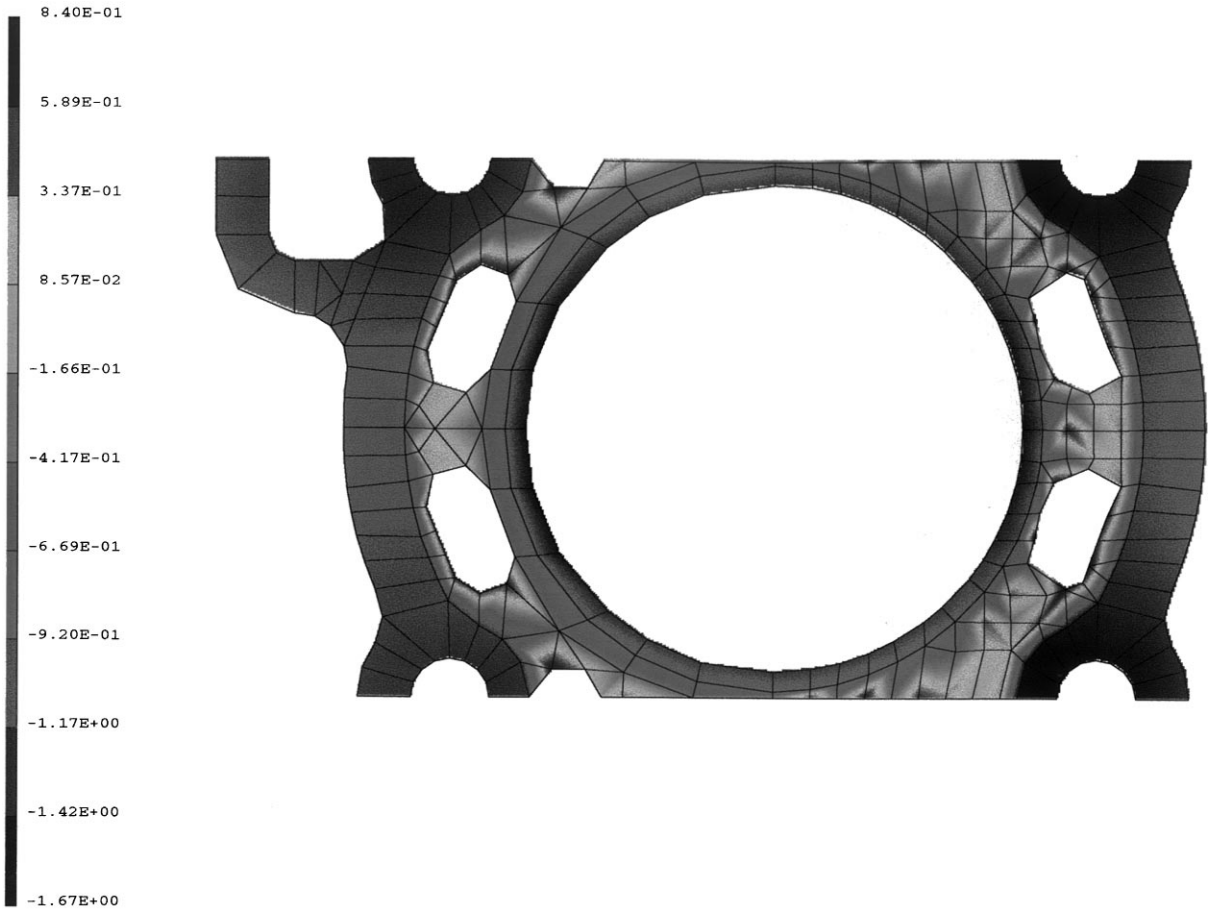


Fig. 8. The normal stress distribution of gasket under the assembly/gas-combined loads case, the maximum normal pressure value is 1.67 Kgf/mm$^2$ located at the internal gasket bead area due to the upstand of liner and gasket bead.

need much more time for trial and error to get a feasible design type accompanied by many test data. Therefore, the analysis of this work can reduce project span time and save the total project cost. In the initial design, the original cold assembly of valve seat inserts and valve guides was not proper in the design phase, and the maximum stress value located at the interface areas between valve guides and cylinder head because of local bending moment was very high. Recommendation obtained from the analyst has been forwarded to the designer, for incorporation of modification, and the basic design goal that the maximum stress value of cylinder structure should not exceed the allowable strength of cylinder head design material about 32.5 Kgf/mm$^2$ can be satisfied. In addition, the designer has also modified the bolt preload value and upstand design because of the finite element simulation results from this project.

(4) About the dynamic response of cylinder head under firing load case, there are two ways to do this in engineering analysis process. One is to perform a real transient analysis and to compare the dynamic stress response with static stress response. The other is to add a dynamic magnified factor

(for example, 2.0) on the static analysis result. Because the maximum principal stress of static analysis under firing load case is 6.25 Kgf/mm$^2$, and the maximum equivalent stress is 12.50 Kgf/mm$^2$. Because the maximum equivalent stress does not exceed the allowable strength of cylinder head design material about 32.5 Kgf/mm$^2$, the basic design goal (cylinder head with no yielding and structural failure under firing load case) can be satisfied. In addition, we can use an $S$–$N$ or $\varepsilon$–$N$ curve of design material to avoid fatigue failure under the cyclic dynamic firing loads.

## Acknowledgements

The author gratefully acknowledges Dr. J.Y. Hwang, Dr. D.C. Liu, Mr. N.C. Shieh and Dr. W.M. Chang of Chung-Shan Institute of Science and Technology for helping the part work of finite element meshing and solid modeling.

## References

[1] O.C. Zienkiewicz, K. Morgan, Finite Elements and Approximation, Pineridge Press, Swansea, 1983.
[2] K.J. Bathe, Finite Element Procedures in Engineering Analysis, Prentice-Hall, Englewood Cliffs, NJ, 1982.
[3] C.A. Brebbia, J.C.F. Telles, L.C. Wrobel, Boundary Element Techniques, Springer, Berlin, 1984.
[4] J.T. Chen, H.-K. Hong, S.W. Chyuan, Boundary element analysis and design in seepage problems using dual integral formulation, Finite Elements Anal. Des. 17 (1994) 1–20.
[5] Shiang-Woei Chyuan, A study of loading history effect for thermoviscoelastic solid propellant grains, Comput. Struct., in press.
[6] S.W. Chyuan, C.Y. Chiou, Application and modeling of finite element method on structural mechanics and thermal analysis, Abstracts of Invited Lectures and Short Communications Delivered at the Sixth International Colloquium on Numerical Analysis and Computer Science with Applications, August 13–17, 1997, Plovdiv, Bulgaria, pp. 31–33.
[7] Harry G. Schaeffer, MSC/NASTRAN primer: static and normal modes, Schaeffer Analysis, Inc., 1982.
[8] J.T. Chen, S.L. Lin, C.Y. Chiou, S.W. Chyuan, J.Y. Hwang, W.R. Harn, W.T. Chin, Finite element analysis and engineering applications using MSC/NASTRAN, Northern Gate Publication, Taipei, Taiwan, 1996 (in Chinese).
[9] I-DEAS User's Guide, Finite Element Modeling, SDRC, 1990.