Benchmark Tests on ANSYS Parallel Processing Technology

Kentaro Suzuki
ANSYS JAPAN LTD.

Abstract

It is extremely important for manufacturing industries to reduce their design process period in order to keep a competitive position in worldwide markets. CAE has been and will have an even more important role in shortening the design process. To effectively use CAE as a part of the design process, the ability of doing analysis in an affordable way, using large-scale real life problems is commonly required.

One of the requirements is the development of a finite element method suitable for parallel computational processing using a number of CPUs. A reduction of computational time is achieved by using parallel machines such as multiprocessing servers or network clusters. Consequently the burden of model simplification, a major part of total engineering time, can be reduced enormously.

ANSYS Inc. has released parallel processing solvers, DDS (Distributed Domain Solver) and AMG (Algebraic Multigrid Solver) in ANSYS version 5.7 in order to meet the requirements from the market.

In this paper, simple tests are first carried out to confirm the features and capabilities of DDS and AMG for different types of analyses. Furthermore, real-life problems such as thermal stress of an engine block (7.3 million DOF) and elastic-plastic-creep behavior of an IC packaging (about 84,000 DOF) are analyzed. The effectiveness of ANSYS parallel processing solvers is also studied.

Introduction

In manufacturing industries, it is important to keep oneself competitive, in order to survive in the highly competitive international market. To be competitive means: high product quality, short product development cycle and the right product release time. Recently, CAE has been playing an ever more important role in the product development process. It has become a common practice to introduce CAE into the design process. At the same time, the reduction of engineering time for CAE is required to stay competitive. There are two issues concerning engineering time reduction in CAE:

1) Ability to solve very large models. This eliminates the need for an experienced engineers' time consuming model simplifications.
2) Ability of solving a real-life model in a realistic time range, that is to say, the requirement for high computational speed.

In addition to these requirements, control software necessary for parallel processing such as PVM (Parallel Virtual Machine) and MPI (Message Passing Interface) has been developed in recent years. Parallel computer equipped with multiple CPUs and Cluster machines (inexpensive Personal Computers interconnected by networks) have also been commercialized. With these dramatic advances in software and hardware 1), large-scale application examples with parallel processing have been published

The solution to these requirements has not been satisfactorily provided so far. However, with the ever-growing FEM technology and computational science, it is becoming possible. ANSYS Inc. has released parallel processing solvers, DDS (Distributed Domain Solver) and AMG (Algebraic Multigrid Solver) in ANSYS version 5.7 in order to meet the requirements from the market. Parallel processing provides a solution to the above issues.

Able to solve very large models

The inability of solving large models means a limitation in analysis mesh size. Model simplification is often necessary to fit the model and get results in a reasonable time. For instance, for a real 3D problem, it is often converted to a 2D problem; such as, plane strain, plane stress or asymmetric according to the geometry shape of the object or load conditions. Components having a bar shape can be modeled with truss elements (axial tension), or modeled with beam elements (bending effect) as required. The analyst needs to select the appropriate element type and provide different section properties for truss or beam element. For a thin plate with a thickness, it can be simplified with thin shell elements, assuming that the effect of stress in the thickness direction is negligible. Take the analysis of a structural component in an assembly as an example, usually the component is modeled individually and loads/constraints from the surrounding components/structures are applied onto the component. These techniques are often used in FEM modeling and can also be considered as a kind of simplification. It simplifies the analysis model by removing the factors which are considered to be unimportant. However, these techniques must be used with careful consideration and judgment and require a lot of engineering experience and know-how. Solving the simplified model saves computational time, however the simplification itself could be very time consuming. In addition the results can be meaningless if the assumptions made are erroneous. It will be extremely difficult, if not impossible, for a designer to simplify a real problem, reduce the size of the model to fit the computer resource, get meaningful results and finally improve his design.

However, with the possibility of large model analysis using parallel processing, the burden for model simplification is greatly reduced (if not totally removed). In other words, with a powerful solid mesh generator one can analyze the model as the read-in CAD geometry. This saves huge amounts of modeling time for the analyst and it allows designer to do difficult analyses without CAE background.

Up to now, analyses has been performed by highly qualified analysts. Designers have to wait for design evaluation from CAE analysts. With the large-scale parallel processing, it becomes possible for designers to do analysis modeling, assess the CAE analysis results by themselves and improve their designs.

Able to solve a real-life model in a high speed

Another merit of parallel processing for FEM is the high-speed computation. If the large-scale problem mentioned above is solved with an unsatisfactory computational speed, for a multi-step non-linear analysis, it could take days or weeks until a result can be obtained. Moreover, parameter studies are usual practice in the design process. One or a few parameters (including the geometric sizes) are adjusted to satisfy the design requirements or to find the best performance of the component. These kinds of parameter studies can take months to be completed for large problems. Not only is the computer being used for long periods of time, but also the design process may have to be stopped for days or weeks waiting for analysis evaluation results. If multiple design modifications are necessary and each needs to be verified by the analysis, the design process period will be unacceptable. One of the criterions for a competitive design process, e.g. time to market, will not be satisfied.

With much improved computational speed by parallel process technology, even for a problem with millions of DOF, it has become possible to run the job over night. The unused hardware resources during nights can be fully used for the analysis. In case of parameter studies, if the response of structure due to one set of parameters can be obtained in a short time, the designer maybe able to know quickly how to alter the design (parameters). As a result, the design process period can be shortened.

Industry specific requirements

Automotive industry

The automotive industry has the fiercest competition in terms of reduction of product development cycle and manufacturing cost while keeping the best performance as possible. Therefore CAE is widely applied to all of the aspects of automobile design process and the role of CAE is becoming even more important. In the automobile industry, for instance, to study the vibration characteristics or stress analysis for fatigue life of a white body, the FEM model could easily exceed the capacity of a single CPU. For the thermal stress analysis of an engine block including its auxiliaries with solid element, due to the complicated geometry of the parts such as the internal water jackets, the total number DOF can be on the order of 10 million. Naturally the ability of dealing with large models becomes a MUST in this industry.

Another typical example is the fatigue simulation of a train arm. A suspension of a car is an assembly of train arm, coil, absorber, sub-frame, etc. Ideally the fatigue simulation model should consist of all the related components in the assembly. If possible, the whole car body should be modeled so that the load acting on the train arm can be accurately captured. However, considering the reality of computational time, the train arm is often modeled individually. The influences from surrounding components are converted to springs elements, constrain conditions. This kind of modeling technique needs a full understanding of mechanics and long-time experience. The validity of the each simplification needs to be tested which is usually very time-consuming. Otherwise meaningful conclusions will not be obtained. If large modeling is available, it is possible to model the assembly as it is without all (or most of) the simplifications.

Even if the solver can solve large models, there is still a consequent need for the pre/post processor to cope with the large models. Pre/post processing is an operation that one has to do it facing the terminal screen. If the pre/post processor is not large model oriented, one is most likely to feel frustrations in mesh generation or result evaluation. In other words, the requirement for large modeling is not only for a parallel processing oriented solver but also a parallel processing oriented pre/post processor.

Electricity / electronic industry

The electricity/electronic industry is another major field in which CAE is widely applied. In addition to electric/magnetic applications, thermal conductivity and thermal creep analysis of semi-conductor package is also a typical application of using FEM. The dramatic increase in the number of devices and functionality of the latest ultra large-scale integration (ULSI) designs have resulted in increased chip size. Concurrently, package dimensions have been shrinking. These two competing trends are leading to even more rigorous requirement on the thermal, electrical and mechanical characteristics of the package technology. One of the dominant issues is the solder fatigue in solder joints under cyclic thermal loads. Since the experiment for thermal fatigue of a solder joint is not only time consuming but also expensive, CAE simulation shoulders an important role. This simulation would typically entail a finite element method employing creep and/or viscoplastic constitutive models. After the viscoplastic strain energy density accumulated per cycle during thermal or power cycling by FEM analysis, the strain energy density is then used with crack growth data to calculate the number of cycles to initiate cracks, and the numbers of cycles to propagate cracks through a joint. An example of solder joints, BGA (Ball Grid Array), of an IC package is shown in Fig. 1. As shown in the figure, IC package consists of numbers of components and has to be modeled 3 dimensionally. Consequently the solder joint analysis has been a heavy job with a large number of nodes and elements 2).

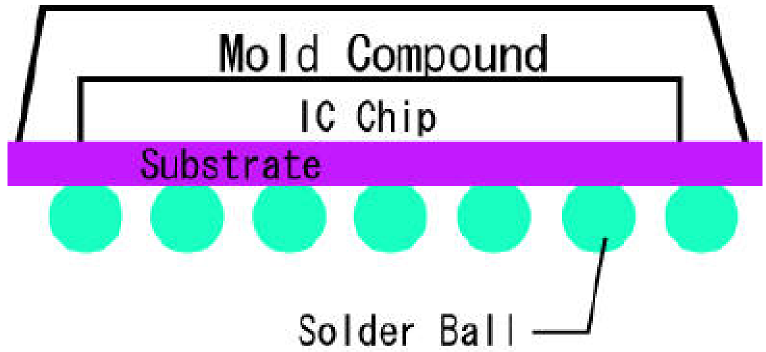

Figure 1 - Ball Grid Array (BGA) package

The current status is such that, with sophisticated mesh distribution and careful simplification of the analysis, the model can be compressed to approximately 20 thousand elements. Still, it could take days to complete the analysis. It does not imply that this is the best practice. It is only because this is the size of a model that can be analyzed within a few days with the current CAE environment (the machine and software capability). It would be highly efficient and accurate to model with a sufficient mesh density everywhere, so that time for considerations of sophisticated mesh distributions can be saved, provided the analysis can be done in few hours or within one day.

The development of lead free solders is required due to increasing environmental and health concerns on the toxicity of lead. Tin-silver lead free solders seem to be an attractive candidate alloy to meet the requirements. It is reported 3) that the Sn-3.5Ag-0.75Cu lead free material is superior to the 63Sn37Pb in creep behavior, i.e. the former has a longer solder joint life. Since Sn-Pb solder has been a main steam in IC packages, enormous knowledge and experience on Sn-Pb has been accumulated. It is expected that for the new lead free solder joint, CAE will again play an important role. Huge amounts of analysis for developing the new solder joint will be necessary and parallel processing for large-scale problem will be rigorously required from IC package makers.

Parallel Processing technology provided by ANSYS Inc.

In ANSYS, two parallel processing solutions, released in ANSYS Version 5.7, are provided to deal with the large-scale problems:

DDS (Distributed Domain Solver)

AMG (Algebraic Multigrid Solver)

The following is a brief introduction of the two methods and a report of the benchmark tests on the effectiveness of the methods

Distributed Domain Solver

The Distributed Domain Solver (DDS) is based on domain partition method 4). The concept of the domain partition method is that the model is partitioned into numbers of small domains and each domain is analyzed individually using a direct solver. The unbalance on the boundary of the domains is taken care of by the iterative solver. Since each domain is analyzed individually and the solution for each domain is independent to the others, this method is considered suitable for parallel processing.

DDS in ANSYS is suitable for solving large models on SMP and DMP machines and also on the cluster systems with the fast growing Linux operating system. However, since the equilibriums on the boundaries are solved with an iterative method, it is not effective for ill-conditioned models.

The scalability of ANSYS DDS was studied. The analysis model is shown in Fig. 2. The linear elastic model has a DOF of 1.5 million. The speedup (scalabilities) with different number of CPUs is shown.

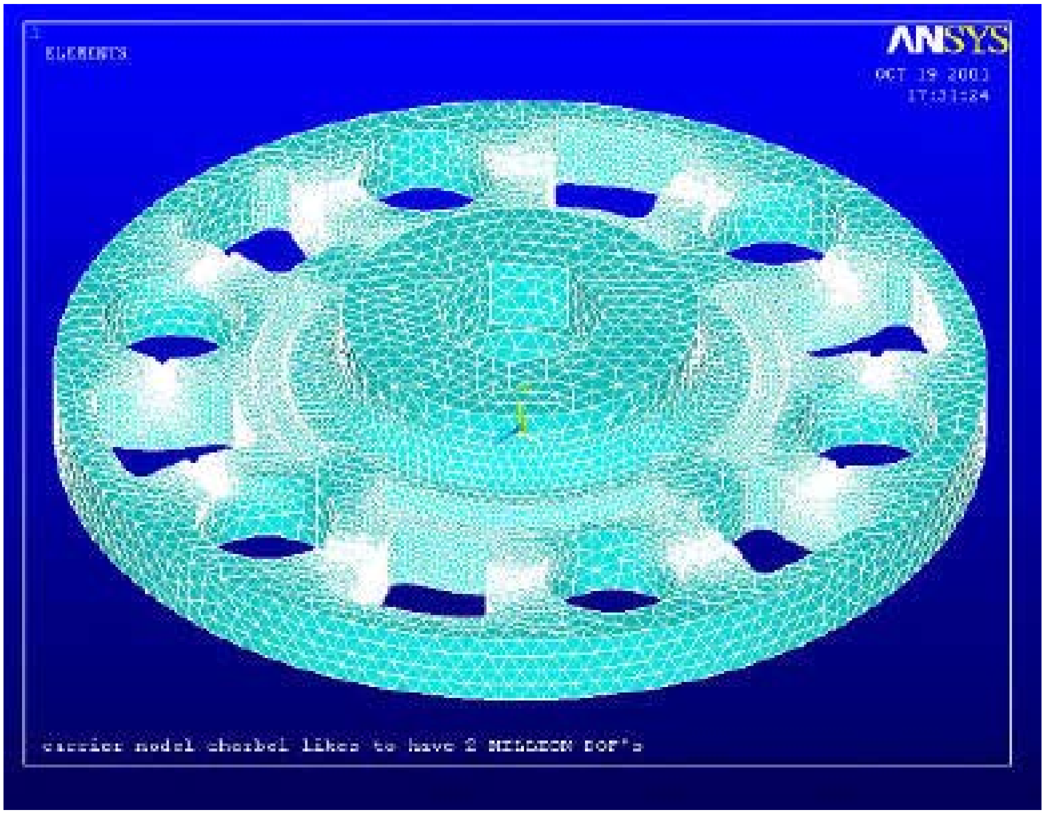

Figure 2 - FE model for DDS benchmark test

The machine used for this analysis was a HP V2500. One analysis result, the stress contour, is shown in Fig. 3. Figure 4 shows the relation between CPU number and computation time. It is shown that with 16 CPUs, the solution time is 11.6 times faster than a single CPU. Table 1 shows a computational time comparison of 3 models with 3 different DOF numbers analyzed from 1 CPU up to 16 CPUs. Speedups of up to 12 times were achieved.

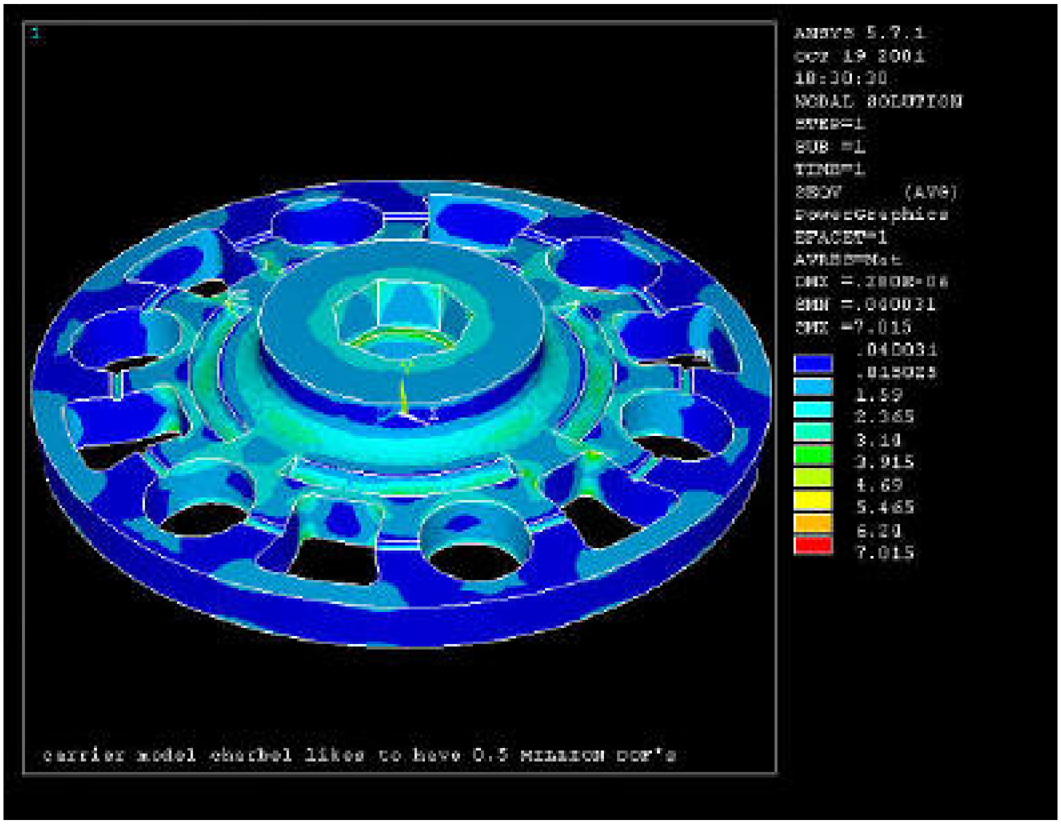

Figure 3 - Distribution of von mises stress

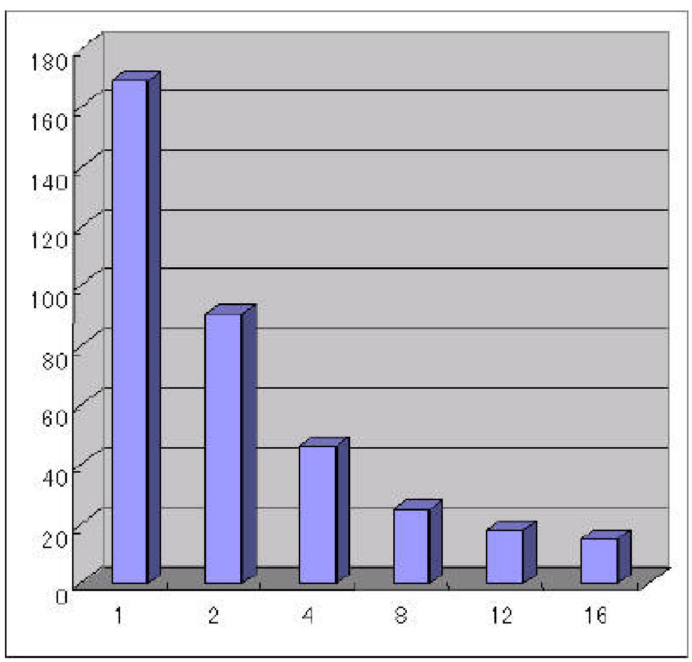

Figure 4 - Relation between CPU number and computation time

Table 1. Computation time using 1CPU/16CPU

D.O.F [million]	1CPU [min]	16CPU [min]
0.5	41	3.7
1.5	170	14.7
2.0	250	23.6

A 7.3 million DOF linear elastic problem is done in order to check to what extent of large-scale problem that DDS can deal with. The analysis model is same as above but with a denser mesh. The machine for analysis is a HP SuperDome. Table 2 gives the information of the analysis and the computational times.

Table 2. Specification of the 7.3 million D.O.F. linear elastic model

Number of elements	1,736,728
Number of nodes	2,441,863
Degree of freedom	7,325,589
Number of subdomains	6,117
Number of CPUs	64
Computation time	42 [min]

From the investigation discussed above, it is concluded that for large-scale linear elastic models, a very high scalability can be achieved with ANSYS DDS.

Thermal Stress Analysis for an Engine Block

A thermal stress analysis for an engine block is carried out using ANSYS 5.7.1 with HP SCA (8CPUs). The CAD geometry of an engine block with 4 cylinders is meshed with 3-D high order tetrahedron using ICEM CFD. Figure 5 and 6 show geometry and FE model of engine block. The parallel processing solver, DDS was selected for the analysis. Number of nodes and elements in the model are listed in Table 3.

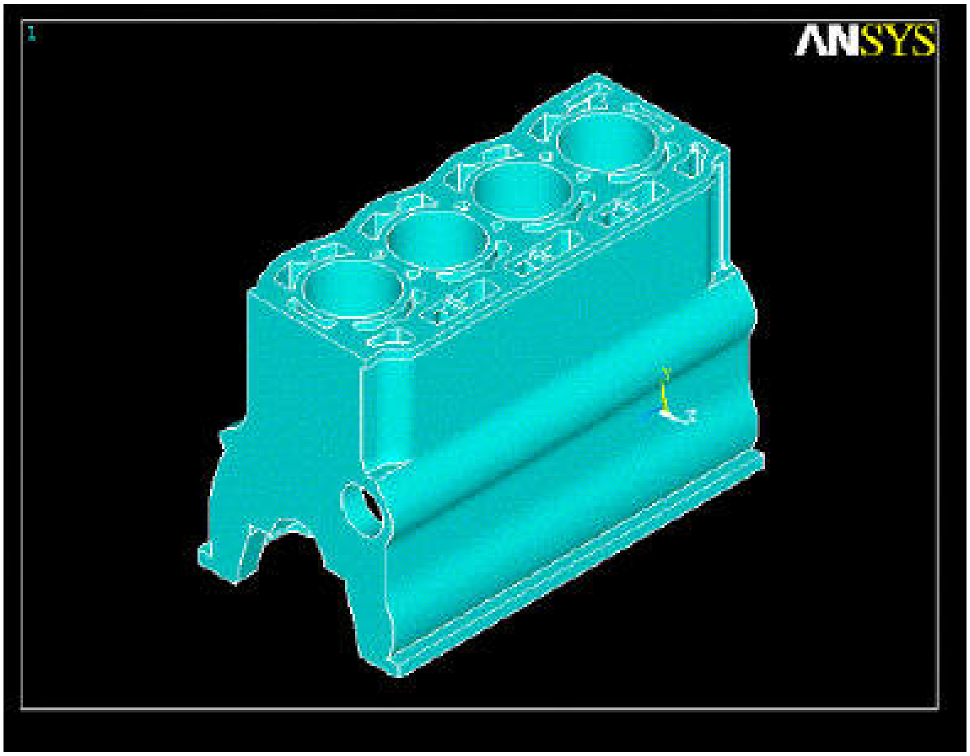

Figure 5 - Geometry of Engine block

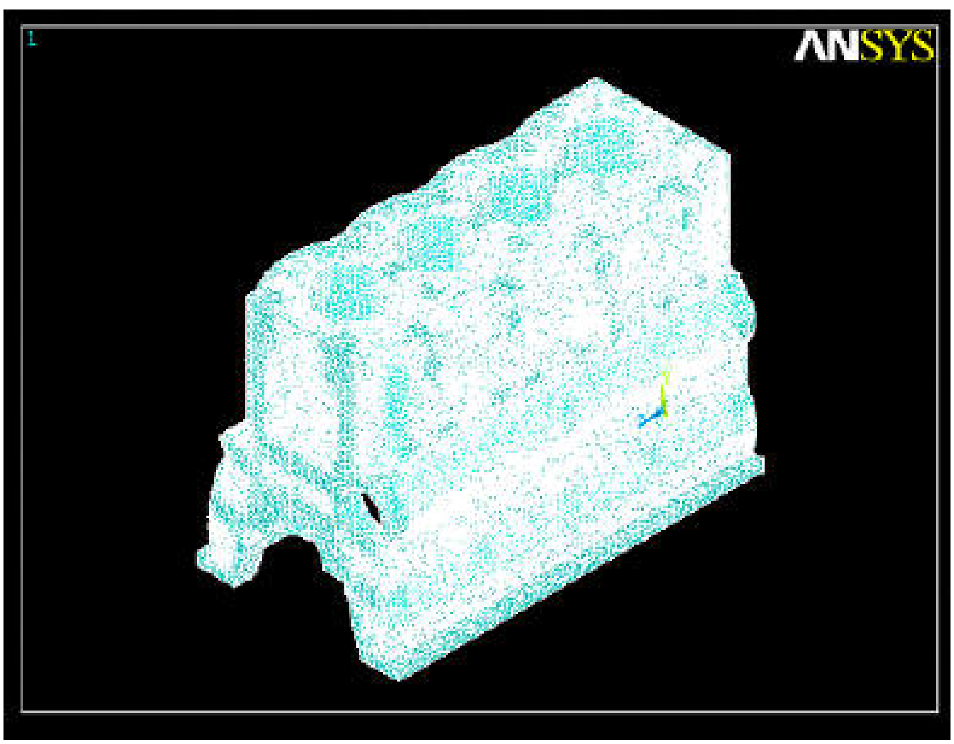

Figure 6 - FE model for Engine block

Table 3. Specification of Engine block FE model

Elements	1,010,642
Nodes	1,485,675
Degree of freedoms	4,457,025

Figure 7 shows the distribution of Von Mises stress in engine block. The computational time for the analysis is 98 minutes. ANSYS Fast Graphics is used for the post processing. It is confirmed that even for such a large-scale model, the post processing with the Fast Graphics is still quick.

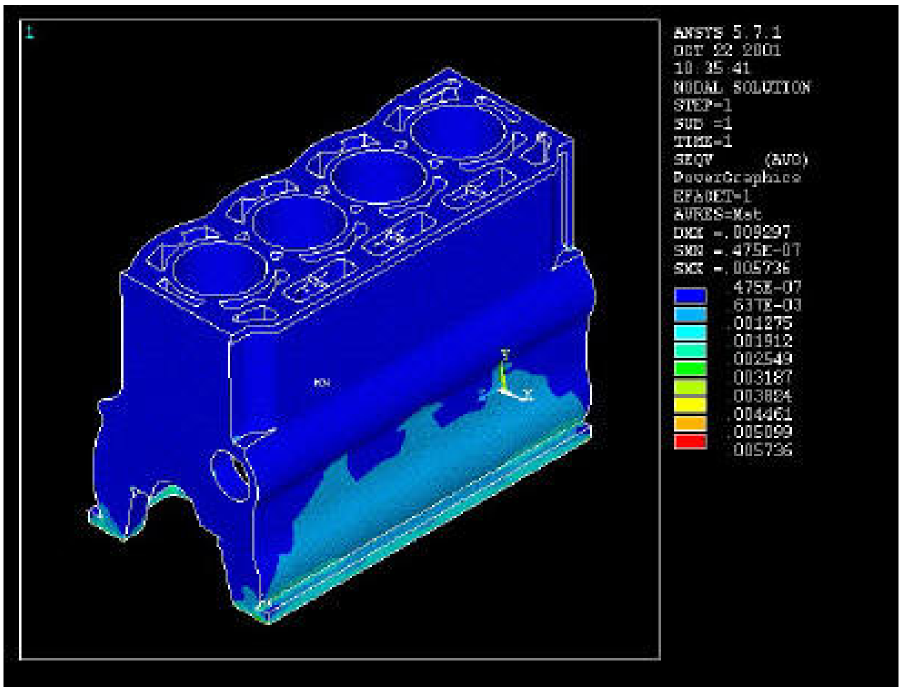

Figure 7 - Distribution of von mises stress in the engine block

Algebraic Multigrid Solver

Another parallel processing method introduced in ANSYS 5.7 is the Algebraic Multigrid Solver (AMG) method. AMG is an iterative solver. Direct solvers often run into difficulty with large nonlinear models (Disk/memory requirements can be high). Iterative solvers are very efficient in solving large models in terms of low disk requirements and relatively low memory requirements.

The convergence rate of an iterative solver is determined from a “Condition Number”. Condition number is the ratio of the maximum and minimum eigenvalues. The higher the condition number is, the more iterations needed to converge. Condition number of 1 is ideal (converge in 1 iteration). A pre-conditioner, C, is used to decrease the condition number of [K] so that convergence is faster. In an iterative process, $C^{-1}Ku = C^{-1}f$ is solved, where C is somewhat similar to K. If C = K, then perfect preconditioning, i.e. exact solution u is obtained in 1 iteration. The more C is similar to K the better the condition number. The ANSYS AMG pre-conditioner is based on Algebraic Multigrid method using a coarsened assembled K. The ANSYS AMG parallel implementation is based on commercially available POSIX thread library.

Compared to other iterative solvers, ANSYS AMG works better for ill-conditioned models and handles constraint equations well. Ill-conditioning can be caused by poorly constrained models such as those with constrained by weak springs, large aspect ratios and other poor element shapes, mixture of element types (Beam/Shell/Solids), very high and low stiffness present in the same model, rapid contact status change, or large numbers of constraint equations/couplings. In case of a model using shell and/or beam element which causes the in-plane stiffness is much higher than the bending stiffness, the stiffness matrix will be ill-conditioned and the convergence rate will be very slow if iterative solver such as PCG is used. There are many structural models for which shell element is appropriate. To date, there have been no satisfactory solutions for large-scale Shell/Beam models. ANSYS AMG parallel implementation may provide an effective solution to the large-scale shell/beam models. However the ANSYS AMG parallel implementation does not support DMP systems. Additionally, ANSYS AMG, like all iterative solvers, has difficulty solving models containing Lagrange multiplier elements.

ANSYS AMG solver was tested using a 3-dimensional contact model. As shown in figures 8 and 9, contact occurs when one of two cantilevers is pressed by a predefined displacement at the free end. Contact is modeled using surface-to-surface contact elements. The model has 1.05 million DOF. Figure 10 shows the comparison of the analysis on HP Superdome with 1 to 12 CPUs. For this problem, the best performance obtained was using 8 CPUs. The computational speedup was not as effective when the number of CPUs exceeds 8. Comparing to 1 CPU, the scalability is about 4.0 if 8 CPUs are used for the problem.

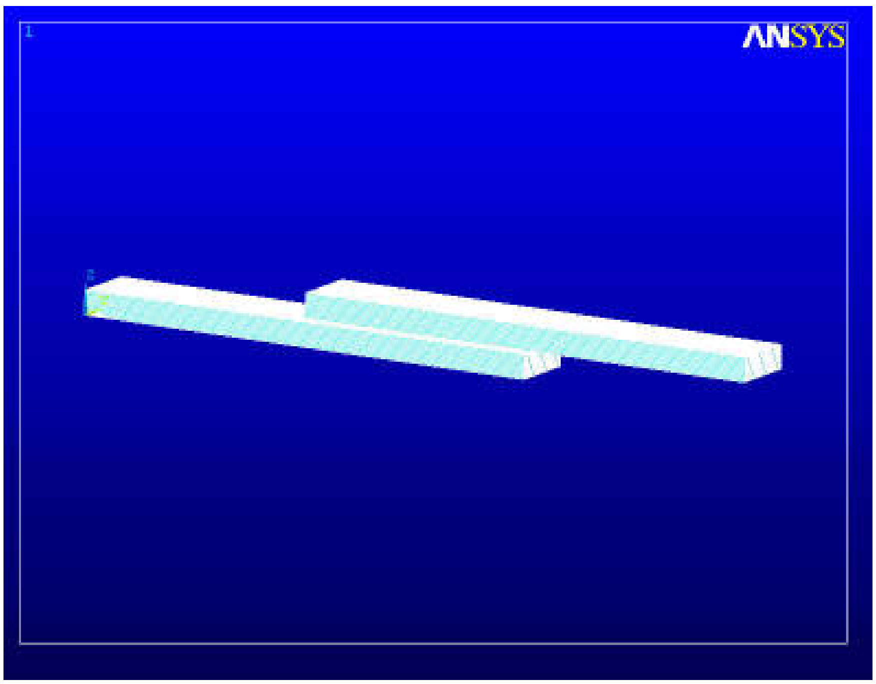

Figure 8 - FE model for AMG benchmark test

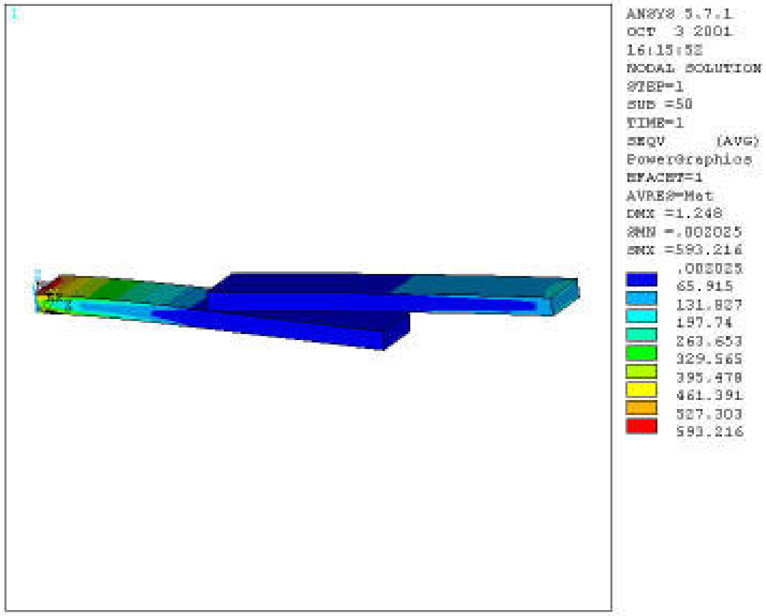

Figure 9 - Deformation shape and distribution of von mises stress

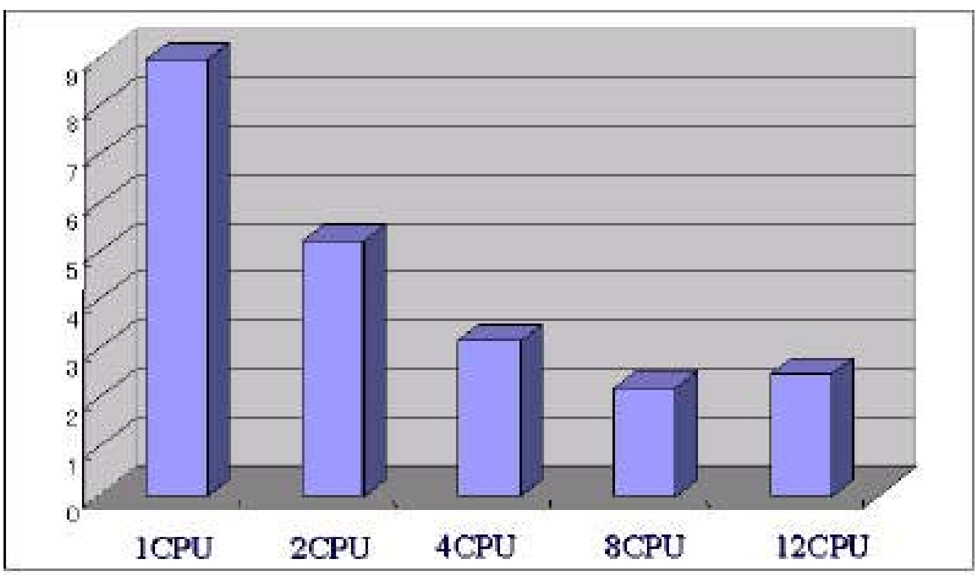

Figure 10 - Relation between CPU number and computation time

Elastic-Plastic Creep analysis for IC package

The IC package is modeled with 3-dimensional solid elements focused on the solder joints. Elastic, plastic and creep constitutive models are employed at the same time. From the geometry symmetry, 1/4 of IC package is modeled and only thermal load is applied. Figure 11 shows the FE model for IC package. Temperature dependent Norton Law and the implicit algorithm are used to capture the creep strain. Since the problem is highly non-linear, it is believed that ANSYS AMG is the appropriate solver to use. Element Number and node number of the model is listed in Table 4. The analyses are carried out by HP V2500 with 1/2/4/8 CPUs.

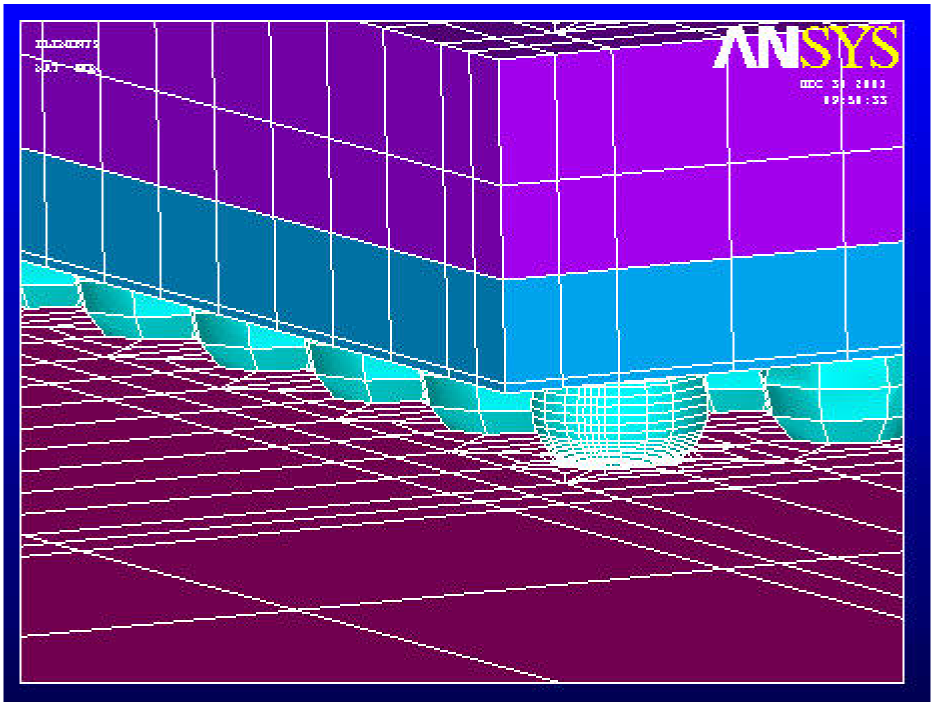

Figure 11 - FE model for IC package (focus on Solder joint)

Table 4. Specification of IC package FE model

Elements	26,224
Nodes	27,992
Degree of Freedom	83,976
Materials	5

The analysis results are shown in Fig. 12. The relation between CPU numbers to computational time is shown in Figure 13. For this model with 8 CPUs, the computational time is 33 minutes, and the speedup is 3.6 times.

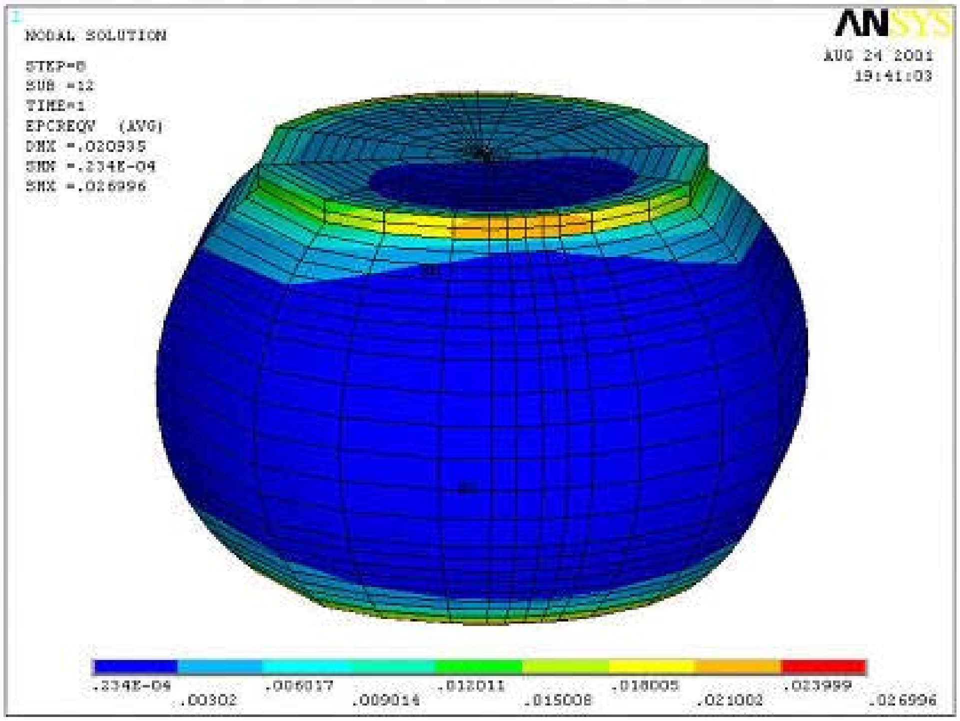

Figure 12 - Distribution of equivalent creep strain

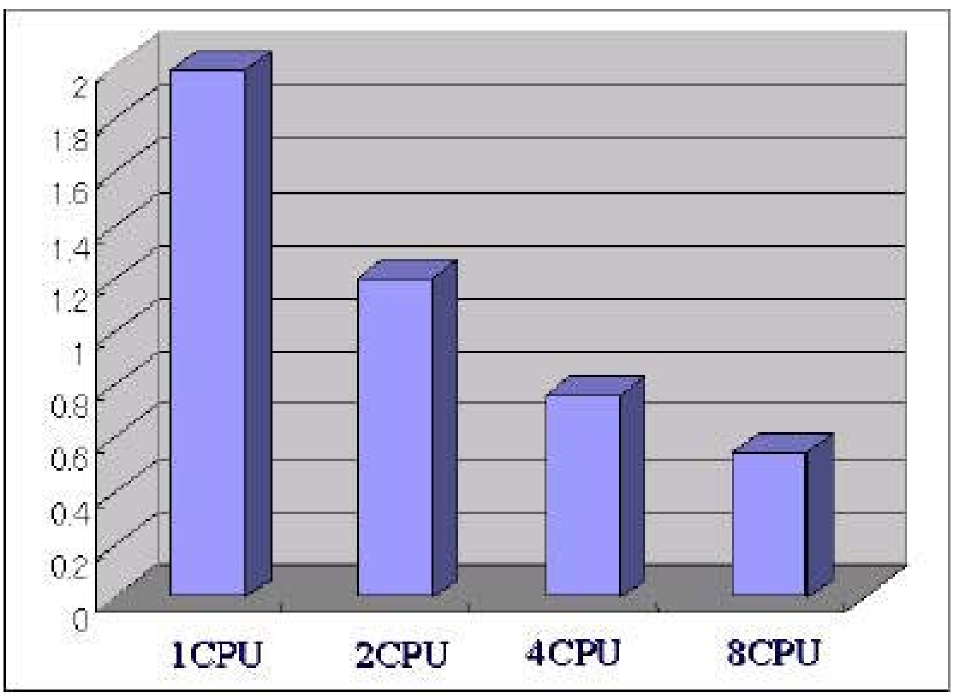

Figure 13 - Relation between CPU number and computation time

From the benchmark tests discussed above, it is believed that the current ANSYS AMG parallel implementation is an appropriate tool for nonlinear problems. The best performance can be expected when CPU number is 4 to8. However as noted before, one of the limitations of ANSYS AMG solver is that the machine has to be of SMP type.

Conclusion

Based on the requirement from the manufacturing industry for solving large-scale FEM models, the effectiveness of the solutions (DDS, AMG) provided by ANSYS Inc. were studied and tested with numbers of simple and real life benchmark tests. The conclusion drawn from our study is that for large-scale linear or low nonlinear models the current ANSYS DDS is an appropriate tool and high scalabilities can be expected even up to 32 CPUs. Whereas ANSYS AMG is a good tool for highly nonlinear problem and speedup can be expected up to 8 CPUs.

References

1) M.Mukai, T.Kawakami, K.Takahashi and N.Iwase, Thermal Fatigue Life Prediction of Solder Joints using Stress Analysis, Proceeding of 1997 IEMT/IMC, (1997),204

2) Nikkei Electronics, 2000.12.4, No.784, 146-160

3) G.S.Almasi and A.Gottlieb, Highly parallel computing, The Benjamin/Cummings Publishing Company (1994)0

4) R.Glowinski, O.V.Dinh and J.Periaux, Domain decomposition methods for non-linear problems in fluid dynamics, Computer Methods in Applied Mechanics and Engineering, 40(1983),27-109