

Back to Elements - Tetrahedra vs. Hexahedra

Erke Wang, Thomas Nelson, Rainer Rauch

CAD-FEM GmbH, Munich, Germany

Abstract

This paper presents some analytical results and some test results for different mechanical problems, which are then simulated using finite element analysis with tetrahedral and hexahedral shaped elements. The comparison is done for linear static problems, modal analyses and nonlinear analyses involving large deflections, contact and plasticity.

The advantages and disadvantages are shown using tetrahedral and hexahedral elements. We also present the limitations in operating and hardware systems when solving large finite element models by using quadratic tetrahedral elements.

Some recommendations and general rules are given for finite element users in choosing the element shape.

Introduction

Today, the some finite element method is not only applied to mechanical problems by some specialists anymore who know every single finite element and its function. The finite element method has become a standard numerical method for the virtual product development and is also applied by designers who are not permanent users and have less detailed understanding of the element functionality.

With the rapid development in hardware performance and easy-to-use finite element software, the finite element method is not used only for simple problems any more. Today finite element models are often so complex that a mapped mesh with hexahedral shaped elements is often not economically feasible. Experience shows that the most efficient and common way is to perform the analysis using quadratic tetrahedral elements. As a consequence of that, the total number of the degrees of freedom for a complex model increases dramatically. Finite element models containing several millions degrees of freedom are regularly solved. Typically iterative equation solvers are used for solving the linear equations. Figure 1 shows typical models meshed with tetrahedra and hexahedra elements.

With modern finite element tools it is not difficult to represent results as color pictures. However, the correctness of the results are actually the cornerstone of the simulation. The correctness of the numerical results crucially depends on the element quality itself. There are no general rules which can be applied just to decided which element shape should be preferred but there do exist some basic principles and also certain experiences from applications which can be very helpful in avoiding simulation errors and in judging the validity of the results.

In this paper we compare some analytic solutions and experimental results with finite element results coming from a mesh of tetrahedra and hexahedra. We also compare the solutions on tetrahedra and hexahedra for complex models, performing linear and nonlinear static and dynamic analyses.

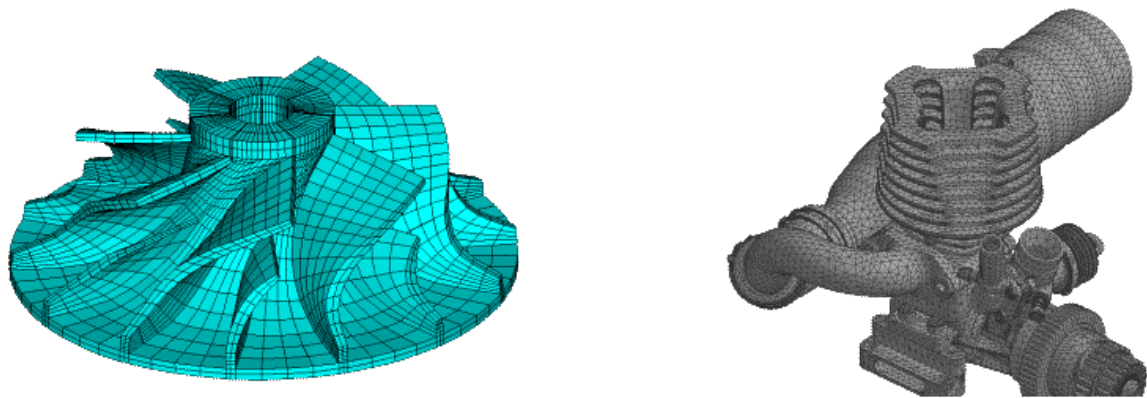


Figure 1. Typical tetrahedra and hexahedra model

Analytical results vs. tetrahedra and hexahedra element solution

Let us consider a pure bending problem which can be calculated analytically using beam theory. We compare the analytic results (displacement and stresses) with the finite element solution using linear hexahedral elements.

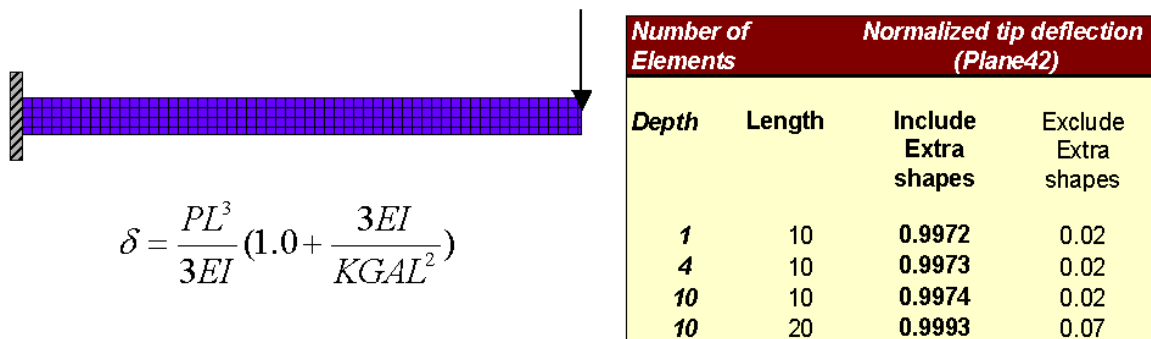


Figure 2. Beam bending problem: Analytical and numerical results of the tip deflection

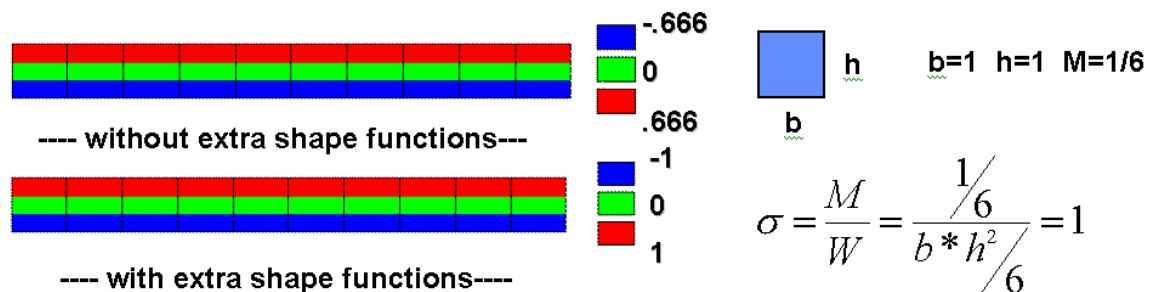


Figure 3. Beam bending problem: Analytical and numerical results of the bending stress

The finite element model with linear hexahedral elements, not including the extra shape functions or the enhanced strain formulation, shows an incorrect result in the stress distribution. Note, that also the error in the displacements cannot be eliminated just by increasing the number of the elements in depth. This is shown in the table from Figure 2 above. This phenomena is known as shear locking:

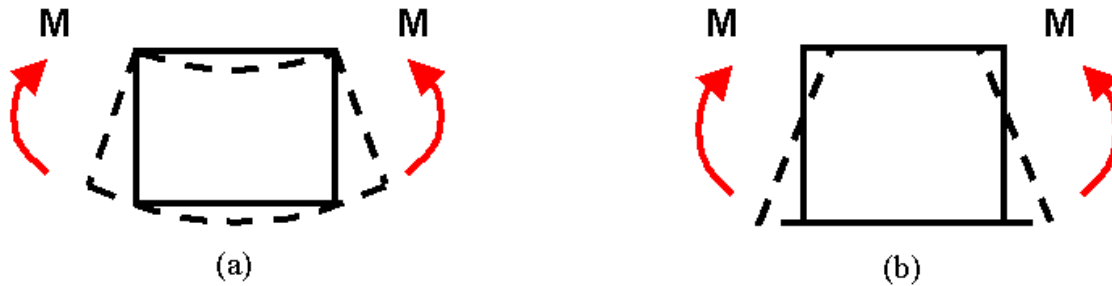


Figure 4. Bending figure using elements with and without extra shape functions or enhanced strain formulation

Figure 4(a) above shows the correct and expected deformed configuration figure in case of a pure bending load. This is just obtained if elements with extra shape functions or the enhances strain method are used. If these technologies are not chosen the wrong result in the figure 4(b) is obtained.

- It is important to know that for bending dominated problems only linear hexahedra elements lead to good results if extra shape functions or enhanced strain formulations are used.

Now we solve the same problem using tetrahedral elements with and without mid-side nodes:

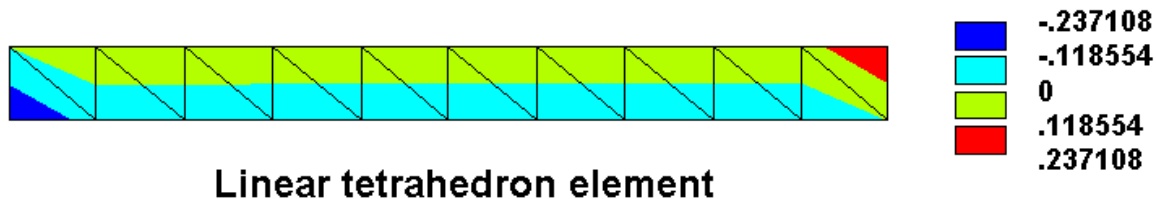


Figure 5. Beam bending problem: Wrong results using linear tetrahedra and a coarse mesh

- Linear tetrahedrons tend to be too stiff in bending problems.

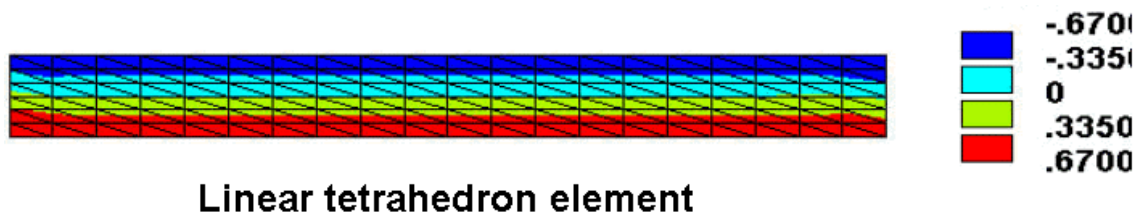


Figure 6. Beam bending problem: Wrong results using linear tetrahedra and a fine mesh

- By increasing the number of the elements in depth the structure is still too stiff

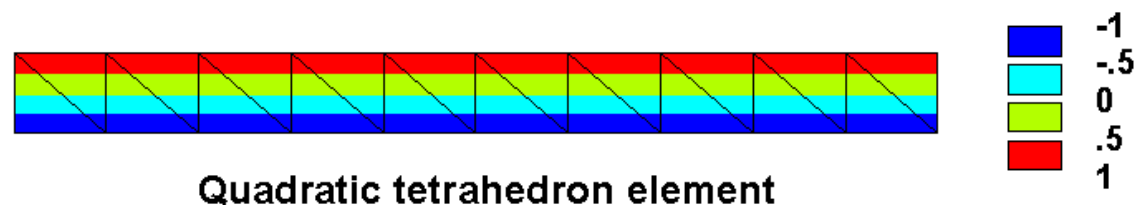


Figure 7. Beam bending problem: Correct results using quadratic tetrahedra with a coarse mesh

- The quadratic mid-side node tetrahedron element shows the exact analytic solution for pure bending dominated problems even with a coarse mesh with only one element in depth.

Figure 8 shows the summary of the beam bending problem solved analytically and numerically using linear hexahedra with extra shape functions, quadratic tetrahedra and linear tetrahedra.

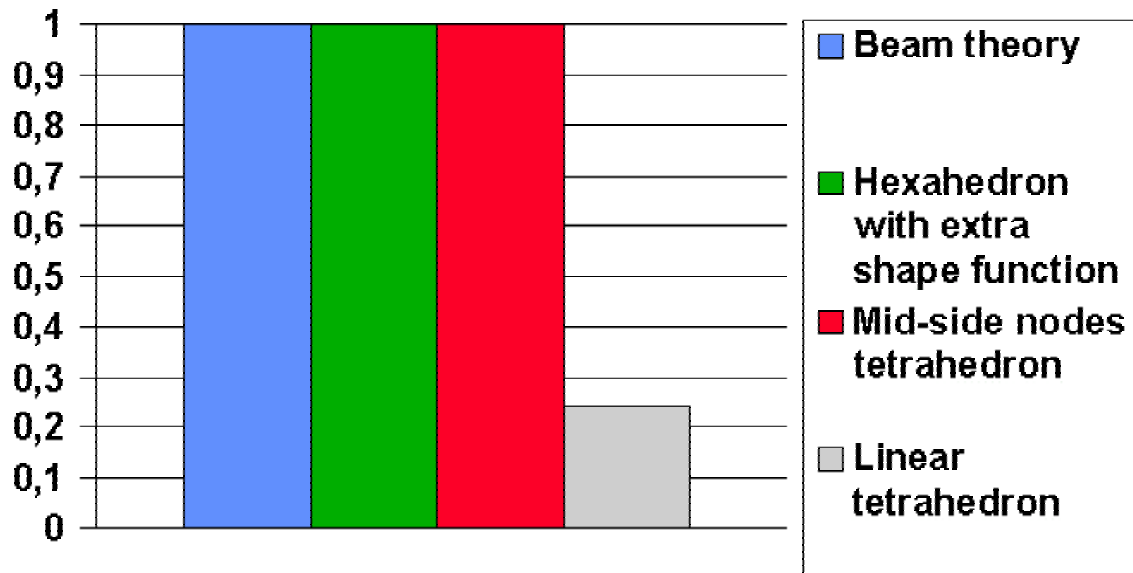


Figure 8. Beam bending problem: Comparing the results

- It is obvious that using a linear tetrahedron element yields unacceptable approximations. The user should not use it for bending dominated problems. On the other hand quadratic mid-side node tetrahedra elements are good for bending dominated problems.

To illustrate the difference in mapping the stiffness of a structure in a correct manner using different element types we perform a modal analysis of a cantilever beam. The first two frequencies and mode shapes are computed. We take the solution of quadratic hexahedra elements as a reference solution and compare the results with a mesh of quadratic tetrahedra and linear tetrahedra with a coarse and a fine mesh, respectively. The results are shown in Figure 9:

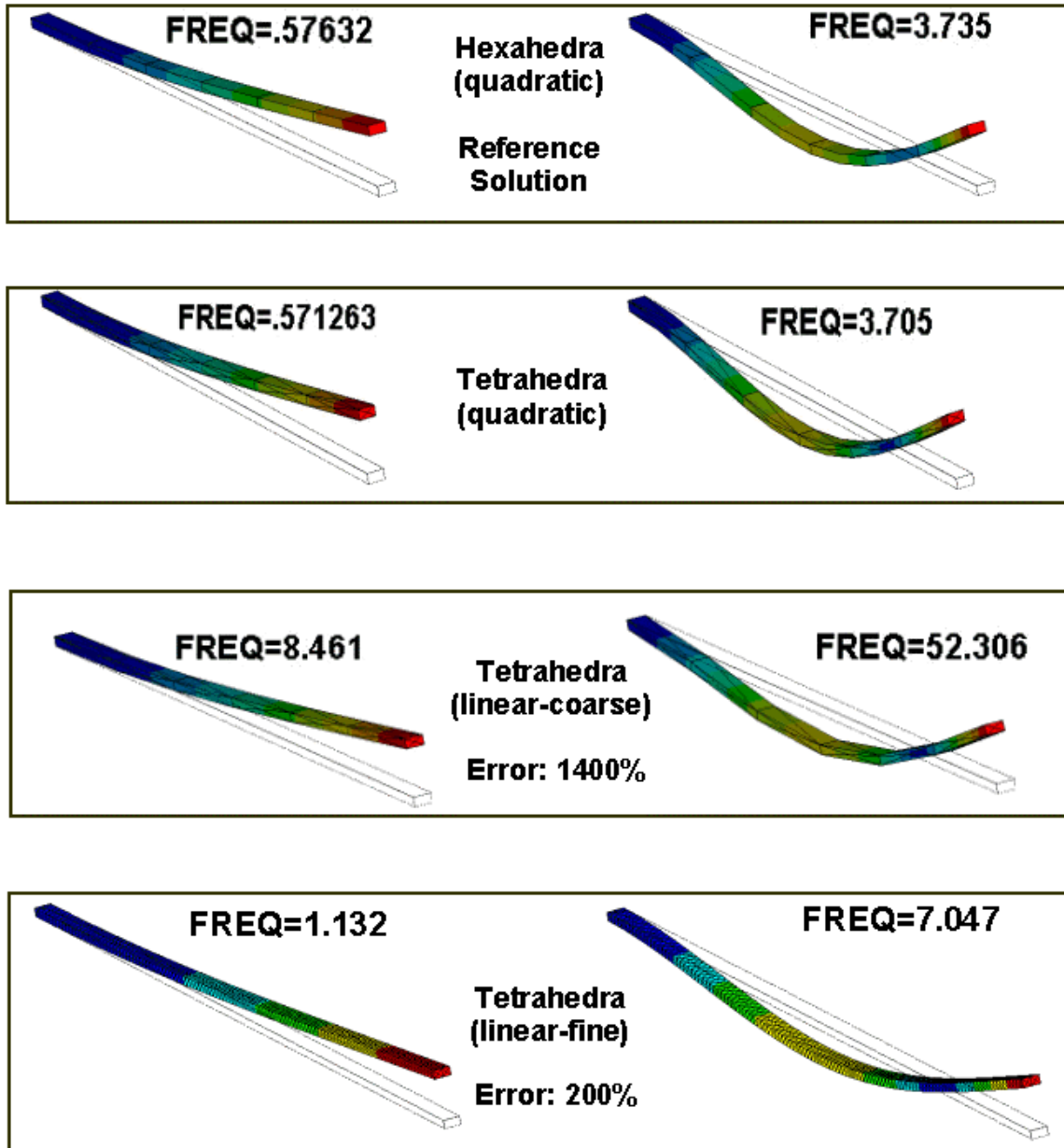


Figure 9. Element quality and stiffness check in a modal analysis

- A good agreement in modeling the stiffness of the structure correctly is just obtained if quadratic tetrahedra elements are taken. Even the fine mesh of linear tetrahedra elements does not result in a good approximation of the solution.

In the next test we investigate the stress concentration in a specimen under different states of loading. We compare the result coming from a mesh of quadratic tetrahedral elements with a mesh of quadratic hexahedral elements with the analytic solution. Figure 10 shows the tensional stress distribution for hexahedral elements, tetrahedral and the closed solution.

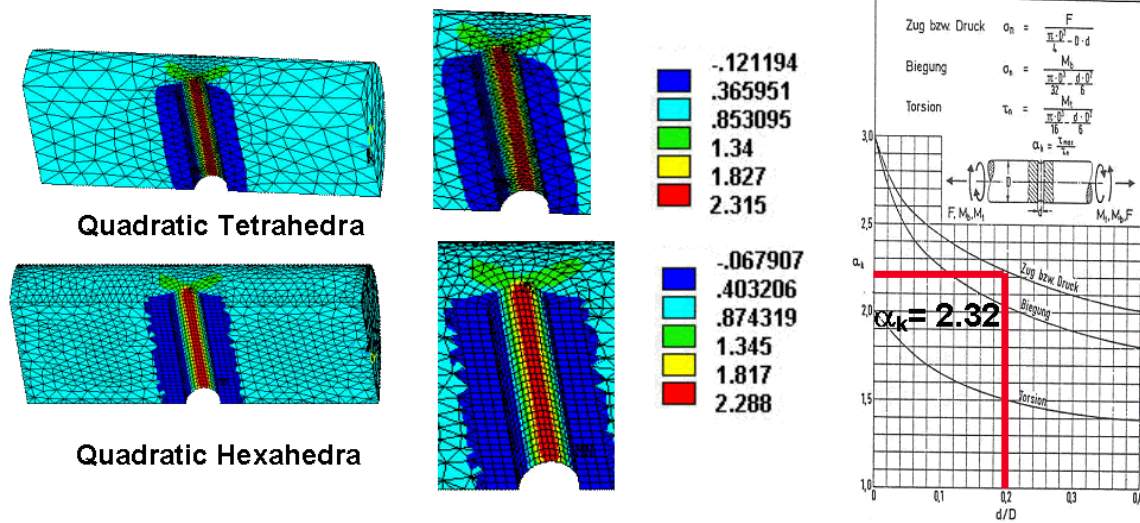


Figure 10. Tension test: Analytical stress concentration and numerical results

Figure 11 shows the bending stress distribution for hexahedral elements, tetrahedral and the closed solution.

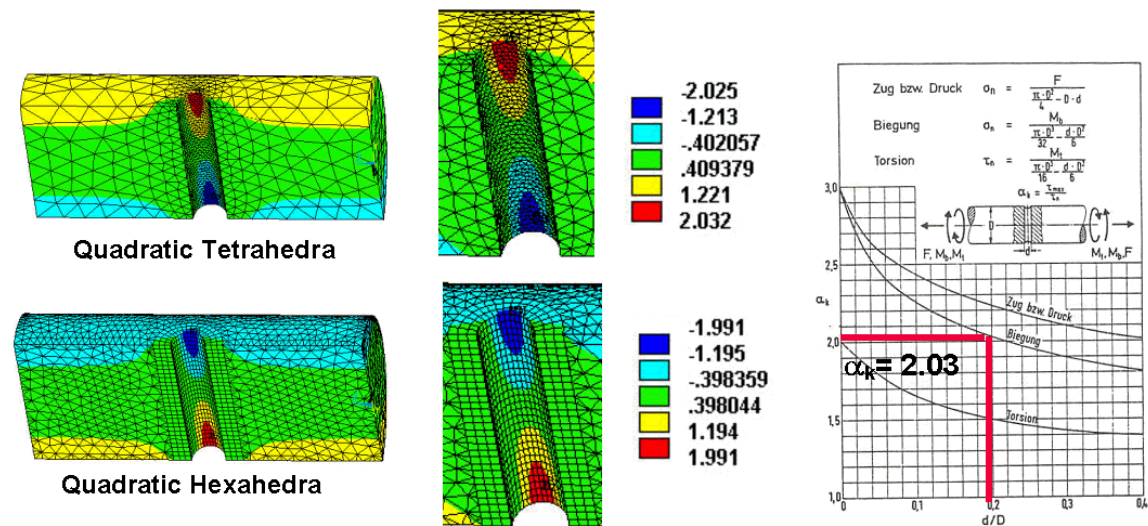


Figure 11. Bending test: Analytical stress concentration numerical results

Figure 12 shows the torsional stress distribution for hexahedral elements, tetrahedral and the closed solution.

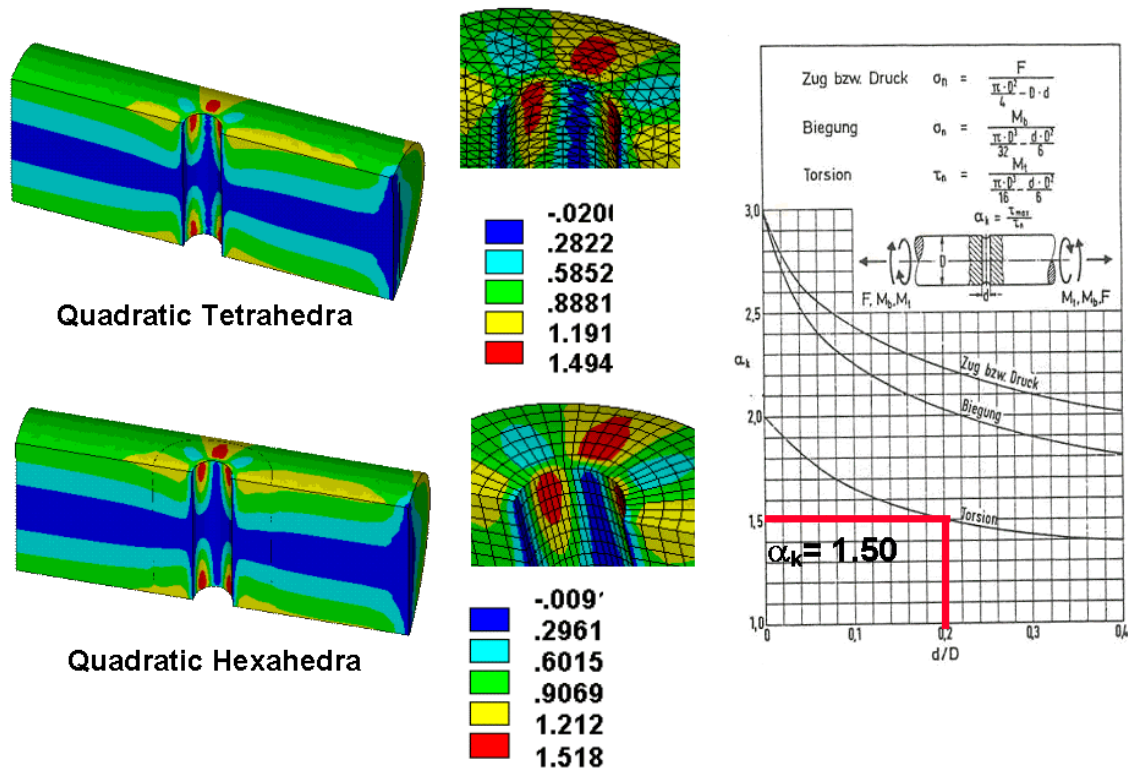


Figure 12. Torsion test: Analytical stress concentration numerical results

The results of all three stress concentration tests are shown in Figure 13. It turns out that the quality of quadratic tetrahedra is really good in comparison with the results of quadratic hexahedra and also with the analytical solution.

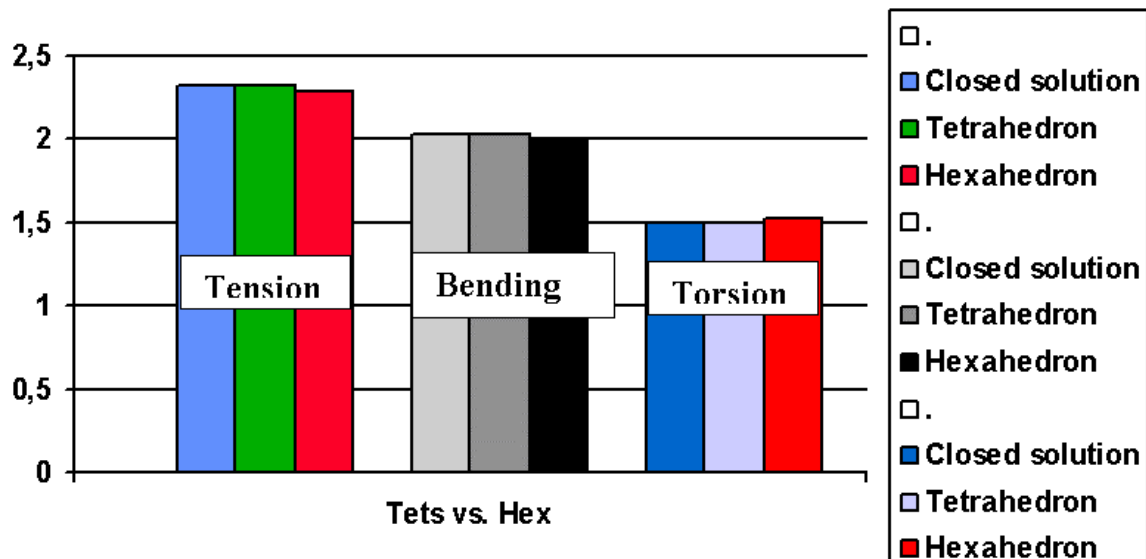


Figure 13. Stress concentration test: Comparing the results

Considering contact problems high stress gradients occur at the contact region. Traditionally, for the nodal stiffness based contact formulation only the hexahedral elements without mid-side nodes are used to achieve a reasonable contact result.

However, ANSYS developed a contact algorithm in which the contact stiffness is based on the results at the integration points. As a consequence of that, structural elements with quadratic shape function can also be used. As we mentioned above quadratic mid-side node tetrahedral elements can predict the local stress concentration very well so they can also be used for contact problems to achieve accurate results. The normal stress distribution for hexahedral elements is shown in Figure 14(a) and for tetrahedral elements is shown in Figure 14(b).

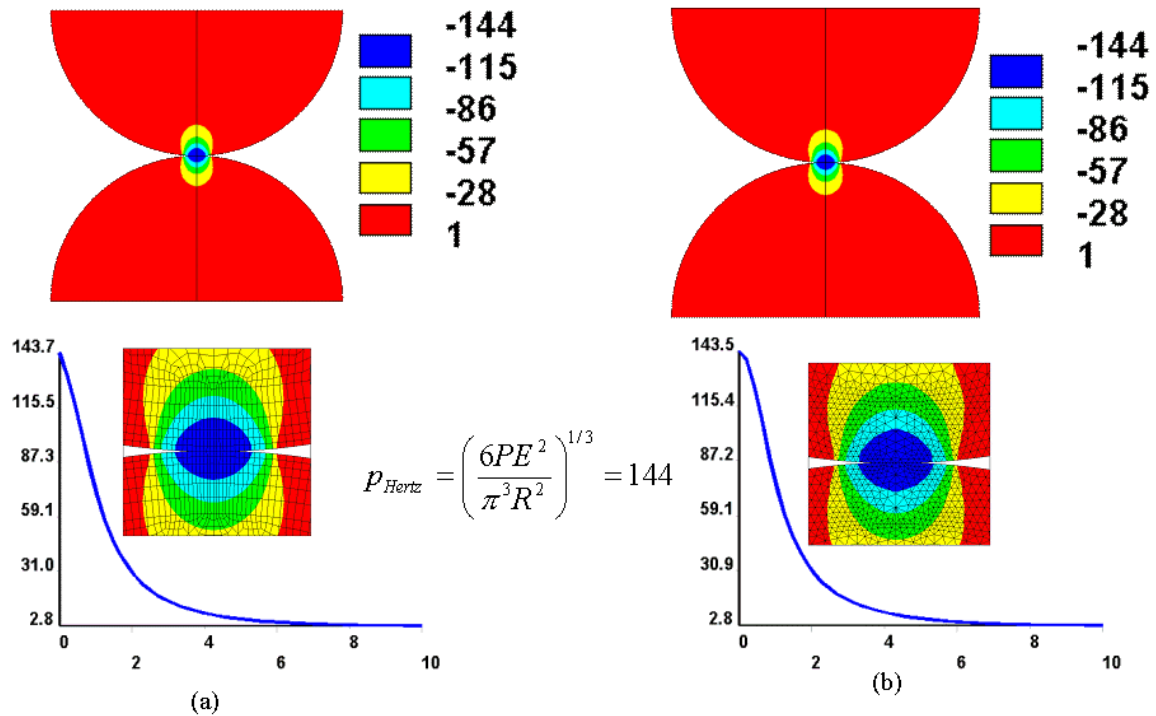


Figure 14. Hertz contact tests with tetrahedra and hexahedra elements and analytical result

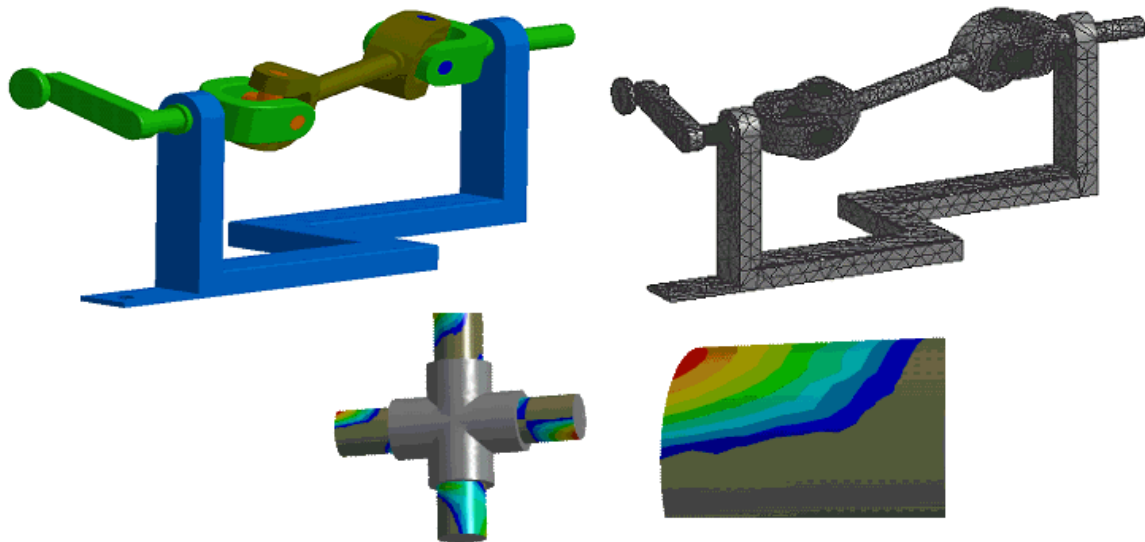


Figure 15. Contact pressure distribution using quadratic mid-side tetrahedra elements

Experimental results vs. tetrahedral element solutions

In this section we will compare the numerical results of a finite element analysis using quadratic mid-side node tetrahedral elements with experimental results. Three examples of linear structural mechanics will be shown.

Example 1

An exhaust guidance support of a diesel engine was investigated under different static loads. Experiments have been performed to validate the accuracy of the finite element simulation with quadratic mid-side node tetrahedral elements. Bonded contact is used to glue different parts of the assembly. The difference between the numerical and the experimental results were less than 5%.

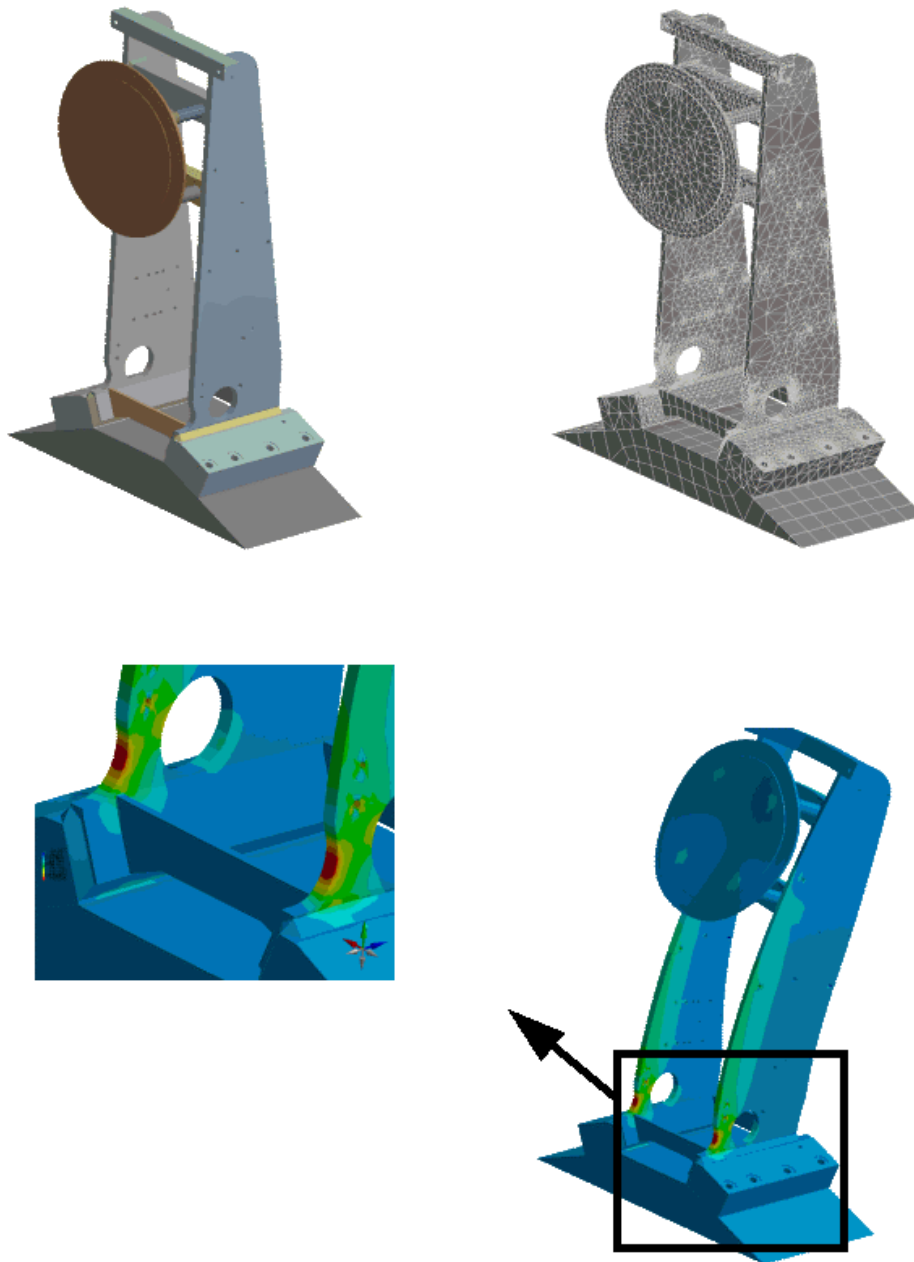


Figure 16. Assembly with quadratic mid-side tetrahedra elements and stress distribution

Example 2

In this example the stiffness of a brake housing under static loading is investigated:

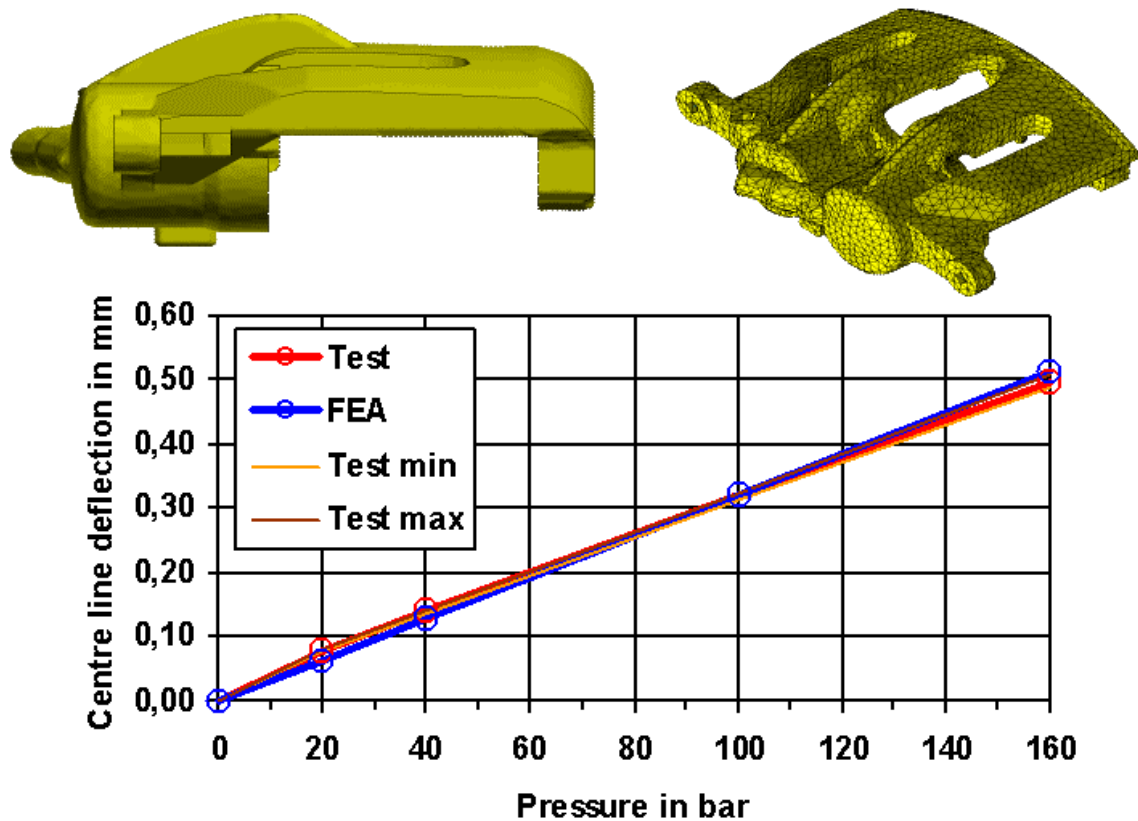


Figure 17. Experimental results vs. tetrahedra finite element results of a brake housing

As one can see, the FE results with tetrahedral elements matches the tests results quite well.

Example 3

In this example we compare the first ten frequencies of a breaker carrier with experimental results:

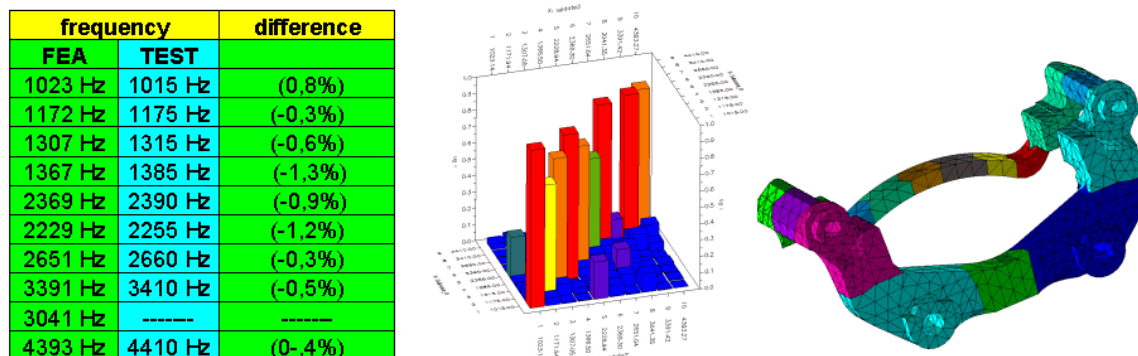


Figure 18. Experimental results vs. tetrahedra finite element results of a breaker carrier

First 10 frequencies show, that the FE results with tetrahedral elements matches the tests results with less than 1% error.

Tetrahedral and hexahedral element solution in nonlinearities

Now we will compare the tetrahedra elements with the hexahedral elements in nonlinear applications. The aim of this section is just to show that the calculated numerical results coming from a quadratic tetrahedra discretization are reasonable compared with an equivalent hexahedra discretization.

Example 1

A nonlinear contact simulation has been performed first to compare the local stress coming from a quadratic tetrahedra discretization with the results from a quadratic hexahedra discretization. The material behavior is linear. Geometric nonlinearities have been ignored.

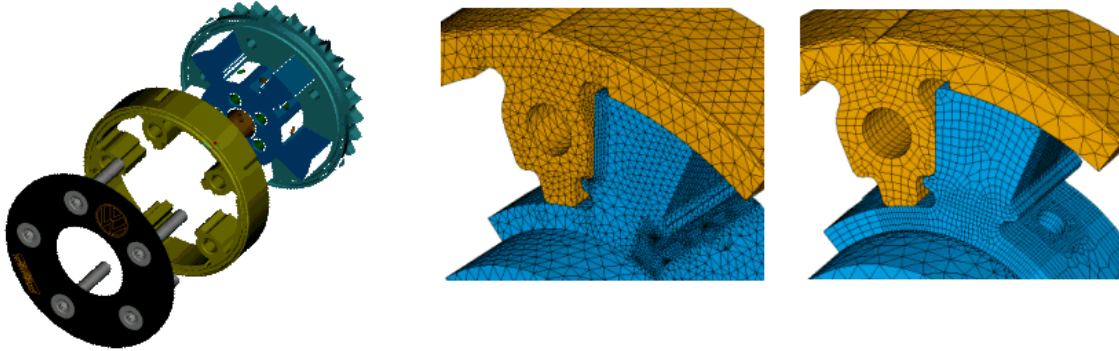


Figure 19. System to be simulated and two discretizations with quadratic hexahedra and tetrahedra elements

Figure 20 shows equivalent stress distributions for tetrahedra and hexahedra are shown here from the nonlinear contact analysis under complicated loading condition. As one can see, both models show the similar stress distribution and the amplitude.

The advantage for hexahedra is, one can achieve the good stress result, without having very fine mesh.

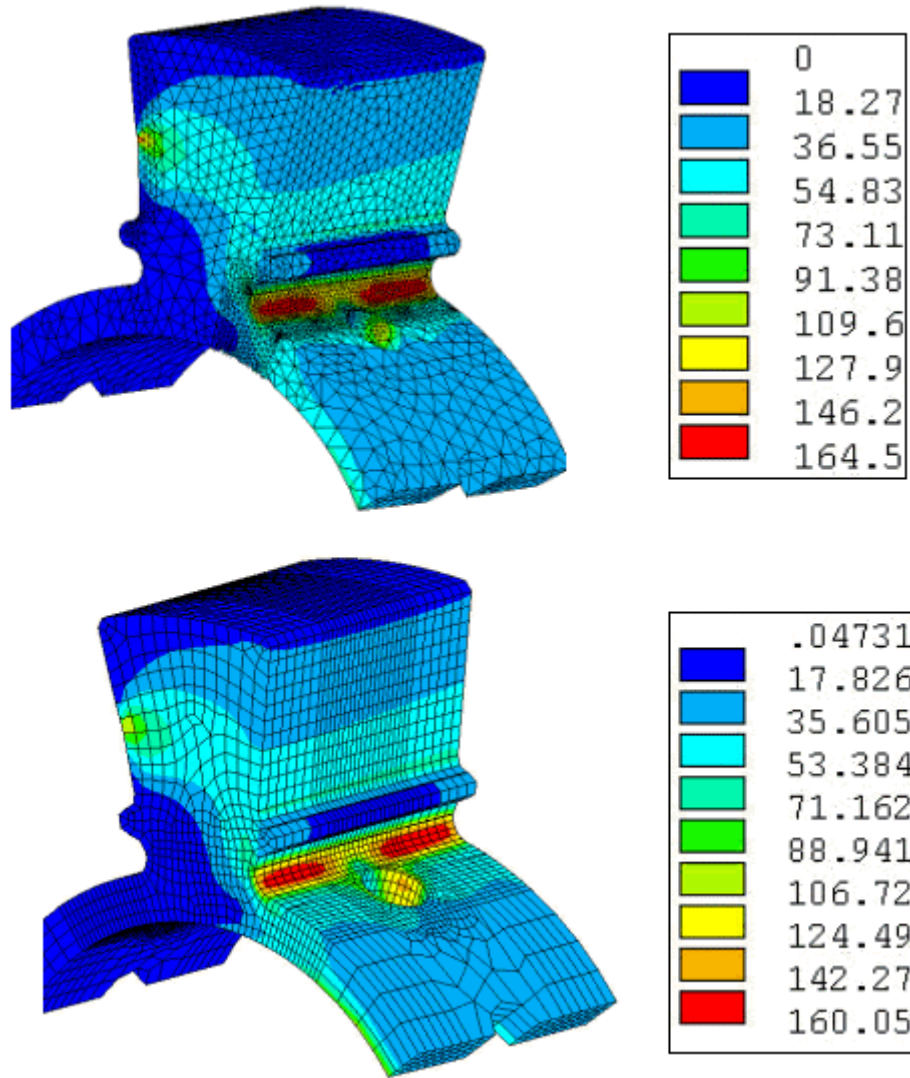


Figure 20. Numerical results for the equivalent stress of the quadratic tetrahedra and hexahedra discretization

Example 2

In this nonlinear contact simulation the effect of geometric nonlinearities has been included together with nonlinear material behavior. Again, we compare the results of quadratic tetrahedra and hexahedra.

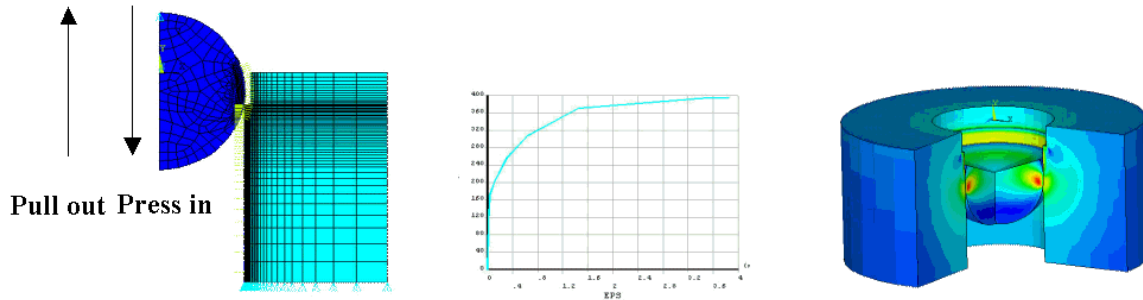


Figure 21. Finite element model (hexahedra elements), material behavior and its equivalent stress distribution

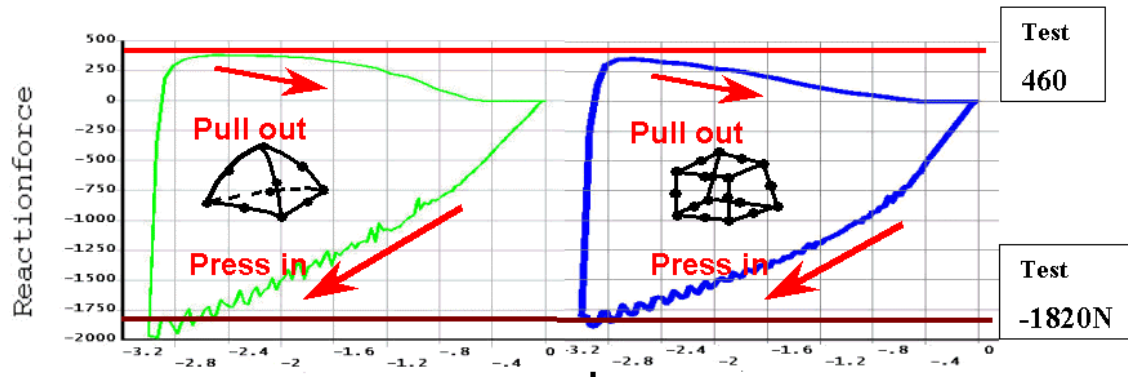


Figure 22.

This figure shows the force needed, with tetrahedral and hexahedra, to press in and pull out the ball. The up and low limit force from the tests are shown with the straight lines.

This figure shows the interference, with tetrahedral and hexahedra, during the deformation process.

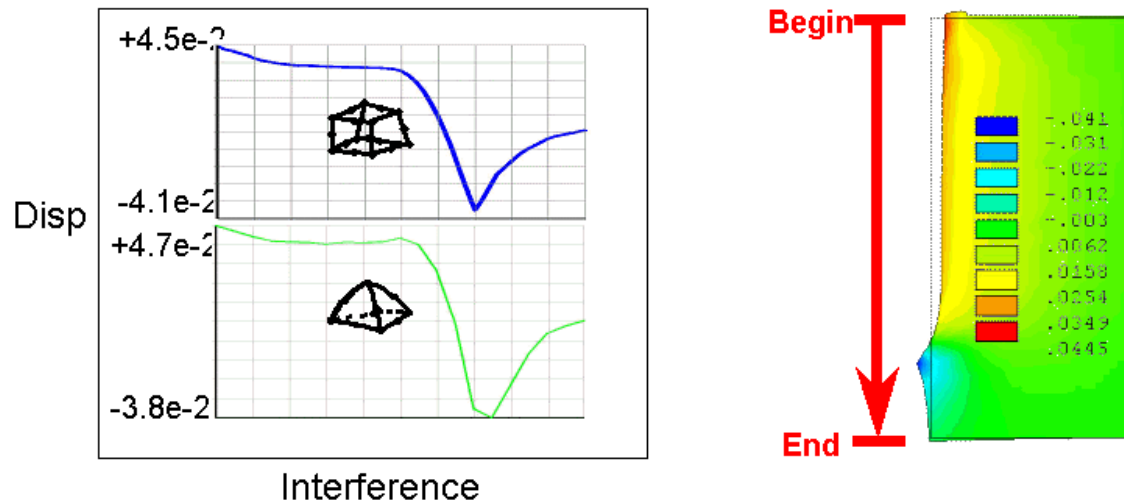


Figure 23. Numerical results of the quadratic tetrahedra and hexahedra discretization

The quality of tetrahedra elements in thin-walled structures

Now we will investigate the quality of quadratic tetrahedral elements when used for simulating the mechanical behavior of thin-walled structures. We investigate the stiffness of the plate by performing a modal analysis and compare the numerical results with the analytic solution for the first frequencies.

Because of the nature of thin-walled structure (no stiffness normal to plane) usually Kirchhof-Love or Reissner-Mindlin based shell elements are used for the finite element simulation instead of classical displacement based solid elements.

The geometric modeling effort to be able to use finite shell elements might be expensive nowadays since for shell applications the user typically needs a mid-surface model. However, most of the CAD models are 3D solid models and the user must work on the solid model to obtain a mid-surface model which is usually not an easy task. For very complicated 3D solid models it is very difficult and maybe even impossible to get the mid-surface in an efficient way. It follows that more and more thin-walled 3D solid models are meshed and calculated using quadratic tetrahedral elements.

Caution must be taken in using tetrahedral elements for thin-walled structure since the structural behavior could be much too stiff in bending, if the element size comparing to the thickness is not properly chosen. This also might result in numerically ill-conditioned stiffness matrices.

In this paper we investigate the stiffness of a simply supported rectangular plate performing a modal analysis using quadratic tetrahedral elements. The first four frequencies are compared with the analytical solution of this problem. Different length/thickness ratios of the plate have been investigated. The aim is to find an in-plane element size so that the error is below 2%. For all calculations an element over the thickness has been used.

As a result of our study we publish a factor which gives us when multiplied by the thickness of the plate a proper in-plane element size to map the stiffness of the plate in a correct manner:

$$\text{IN-PLANE ELEMENT SIZE} = \text{FACTOR} * \text{THICKNESS OF THE PLATE}$$

For length/thickness ratios of 50,100,500,1000,2000,3000,4000 and 5000 the studies have been performed. The results are shown in Figure 24 and Table 1:

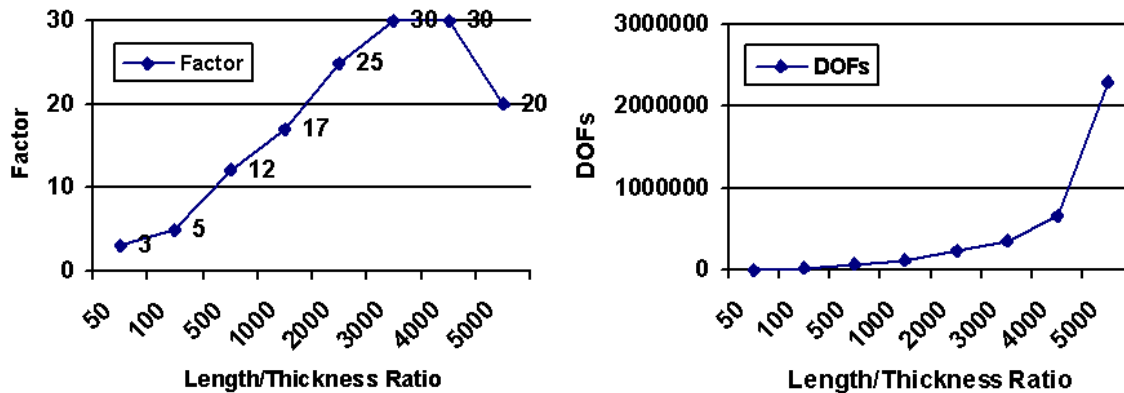


Figure 24.

Table 1. Factors to calculate reasonable in-plane element sizes for tetrahedral elements of the plate problem.

Length/Thickness Ratio	50	100	500	1000	2000	3000	4000	5000
Factor	3	5	12	17	25	30	30	20
DOFs	11000	15000	65000	127000	230000	360000	650000	2300000

Conclusion

Ten years ago because of the hardware and the software limitation only relatively small finite element models of approximately 100,000 degrees of freedom could be solved efficiently.

Usually the engineer had to simplify the mechanical system in order to minimize the number of unknown degrees of freedom. There were several problems in this process:

- 1) The simplification process was usually expensive. It took days, weeks, even months.
- 2) If the design changed after the simulation the whole simplification process must have been done again because the finite element simulation was not really integrated in the virtual product development.
- 3) Usually the simplified finite element model could not predict local stress distributions. Additional models were necessary to investigate local characteristics.
- 4) Only experienced finite element engineers were able to do the simplification.

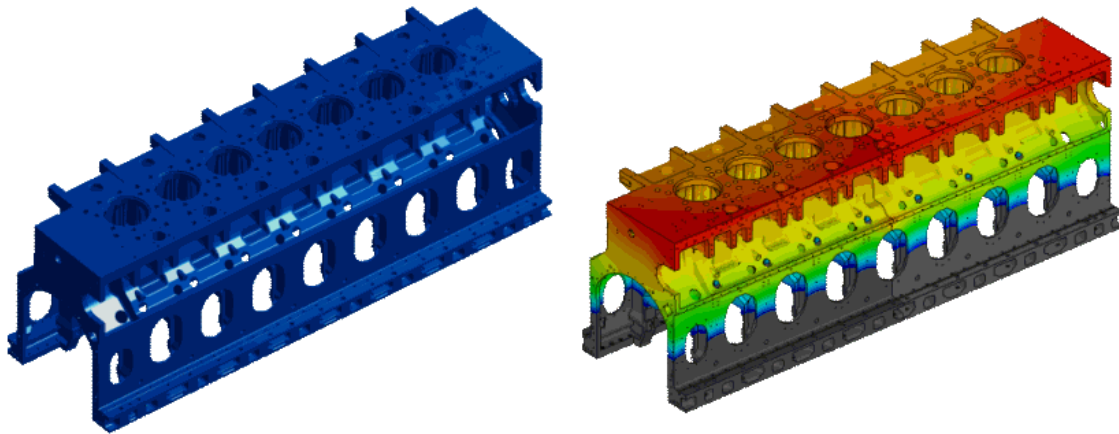


Figure 25. Engine block with complex geometry: system and resulting first mode shape of a modal analysis

For the kind of geometry shown in Figure 25 one engineer needed at least weeks to get a simplified finite element model if shell elements or hexahedra elements were to be used.

Today, 3D CAD solid models are typically meshed with quadratic tetrahedral elements. Very often this yields large models of about 2 to 3 million degrees of freedom. Nowadays on modern machines such models can be handled mostly without any problems. For the above engine block the first six natural frequencies have been calculated successfully on a window XP machine with 3 GB RAM within approximately 3 hours (elapsed time).

In this paper we showed that the quality of quadratic tetrahedra elements is good if reasonable element edge sizes are used. To sum up we give the following recommendations in choosing a solid element type:

- 1) Never use linear tetrahedron elements. They are much too stiff.
- 2) Quadratic tetrahedral elements are very good and can always be used.
- 3) Linear hexahedral elements are sensible with respect to the corner angle (see ANSYS benchmark in ANSYS verification manual). The users should be careful to avoid large angles in stress concentration regions. Extra shape functions or the enhanced strain formulation should be activated for bending dominated problems.
- 4) Quadratic hexahedral elements are very robust, but computationally expensive.
- 5) For thin-walled structure the limit element edge/thickness ratio to use tetrahedra is about 2000.

Our results and recommendations are also outlined schematically in the following Figure 26. It holds for the two dimensional and also for the three dimensional case:

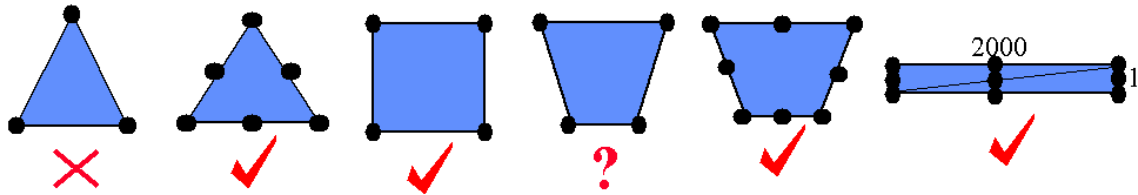


Figure 26. Solid element recommendations

On a windows 32 bit computer the largest finite element model we solved had about 2.5 million degrees of freedom. On a 64 bit windows XP machine we were even able to solve problems with 14 million degrees of freedom and on a 64 bit unix or linux machine we solved even problems with about 20 million degrees of freedom. In the next two or three years we expect to be able to solve problems with 50 million up to 100 million degrees of freedom successfully.

References

- [1] ANSYS Theory Reference, Release 6.1, Swanson Analysis Systems, Inc., 2001
- [2] Johnson, K.L., Contact mechanics, Cambridge university press
- [3] A comparison of all-Hexahedra and all Tetrahedral Finite Element Meshes for elastic & elastoplastic analysis. Proceedings 4th International Meshing Round table Sandia National Labs, pp 179-181, Oct. 1995