INTEGRATED FINITE ELEMENT ANALYSIS (FEA) IN THREE-DIMENSIONAL COMPUTER AIDED DESIGN PROGRAMS (CAD) - OVERVIEW AND COMPARISON

Bernd Roith, Alexander Troll and Frank Rieg

Department of Engineering Design and CAD, University of Bayreuth (Germany)

ABSTRACT

Nowadays faster product development is expected by customers but at the same time the costs of production and development have to be minimized. To meet these requirements more and more computer-assisted simulations are being used. This applies to many production procedures and a new standard of engineering by intelligent cross-linked simulations (ICROS) has been defined for these. The ICROS method describes the whole proceeding of development and production of a new product, including the material selection (material parameters), the design of the construction unit, the finite element analysis (FEA), the production simulation and the manufacturing of the prototypes. The constantly growing impact of such simulation programs led to the integration of tools covering the functions of the finite element analysis into all larger three-dimensional computer aided design programs.

This study shows the general differences between the FEA application tools, which are implemented in the CAD systems Pro/ENGINEER Wildfire, CatiaV5 and NX4, and investigates if these tools are generally suitable for the ICROS method. These tools consist of the import interface of geometry units, the pre-processor for the independent generation of a finite element mesh and the actual solver. The comparison is based on linear static analyses of individual engineering units. Three other standalone finite element programs were added to complete the comparison. These programs are Patran as pre-processor combined with Marc as solver, ABAQUS 6.6-1 and Z88 Version 12.0. The comparison is realised on the basis of several examples.

Keywords: linear finite element analysis, ICROS, software comparison

1   INTRODUCTION

1.1 ICROS

The life cycle of modern products is decreasing quickly because consumers demand innovative, state of the art products. Thus the product life cycle has to adapt in line with this, meaning that product development needs to be accelerated. This demand can be met by using modern computer-aided engineering. The engineering division not only has to make shape-giving decisions for the subsequent product but also has to check if the product satisfies all requirements. These requirements include for example mechanical sustainability and the parameters for manufacturing. The first aspect is in most cases determined by finite element analysis.

Within a complete product study a static analysis (strength determination) is executed, but in addition to this a simulation of production processes such as welding, milling or injection moulding should be performed. It is essential to connect the intersection between the different simulation results. An approach for linking these simulations is represented by the intelligent cross-linked simulations (ICROS) method [1, 2]. In the following paragraph a short example is described to outline the main principles shown in Figure 1.

The steel body of a car mainly consist of stamped panels which are joined together by a spot welding robot. Although the panels themselves are quite simple in design, the requirements that the auto body has to meet are very high. Before the production can begin a fundamental static analysis of the panel has to be performed. If the results of this analysis are satisfactory, a simulation of the manufacturing

process should be carried out either afterwards or even simultaneously. While the basic results of the linear finite element analysis are mainly displacements and van Mises stresses resulting from applied loads and boundary conditions, the prozess-simulation can determine influences resulting from heat treatment such as welding distortion. The most important question is to determine what can be done if the strength determination shows that the developed product does not satisfy the safety limit. The same applies to the manufacturing simulation [1, 3].
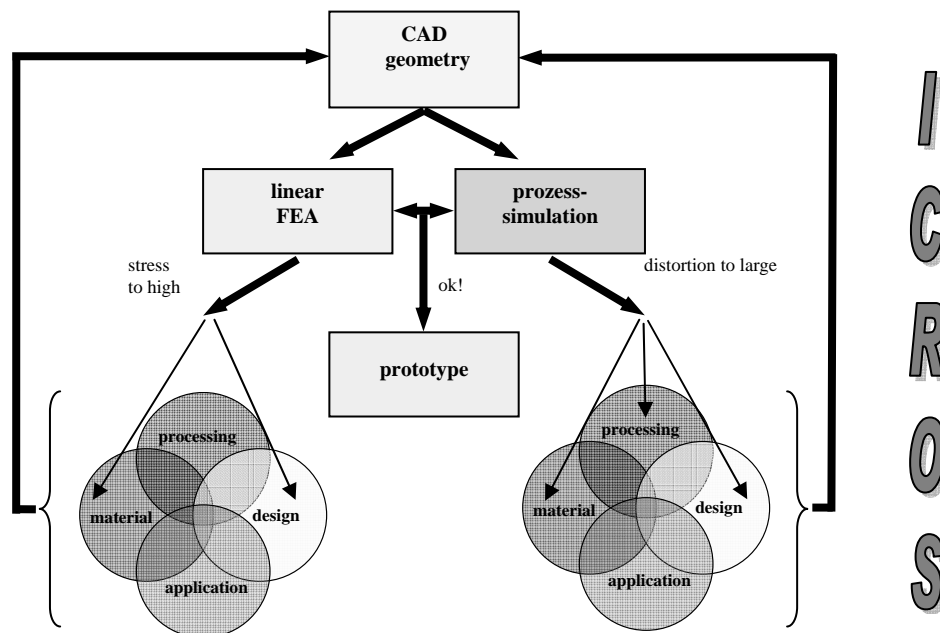

figure 1: cross linking of simulation programmes during the ICROS process [3]

To prevent possible mistakes when using ICROS, Alber has outlined the following proceeding [3]: After designing the geometrical model a strength determination and manufacturing simulation has to be undertaken. If all results are satisfactory, a prototype can be produced for further studies. Otherwise the whole proceeding must be redone with other parameters. For example, if the linear analysis shows that the part geometry is not suitable for the applied loads, the geometry has to be altered in order to receive a sufficient result. Altering the geometry will consequently also affect the manufacturing of the part. Therefore both the static analysis and the prozess-simulation have to be repeated to verify that the geometric changes do not have a negative effect on the welding distortion.
In order to speed up the product development when using the ICROS method it is advisable to begin the sequence of simulations with the linear static analysis, since this requires the least amount of time compared with other simulations.

1.2   CAD-integrated finite element analysis
There are many possibilities of performing a linear finite element analysis. One proceeding is to import the finished geometric model into corresponding software. Such a proceeding can lead to increased complexity if it is necessary to change the geometry, because the import must be redone after each modification. Integrated finite element modules of CAD-systems allow better time management because the import step is not required. Another advantage is that the design giving engineer can carry out the analysis on his or her own without the need to call on an intern or even third party specialist. Moreover there is a financial factor since the license fees for integrated FEA-tools are usually lower than those of standalone simulation software that often cover a wide range of applications, many of use for specialised simulation departments only.
Another useful advantage of integrated solutions is that the user only has to deal with one graphical interface within the complete software-suite, meaning that he does not have to become acquainted

with several different user interfaces of varying programs. Furthermore the know-how within the company increases because the constructing engineer multi-tasks the construction requirements and the simulation which will lead to faster product development in the future. Other developments in the field of finite elements deal with the direct usage of geometric representation for structural analysis. This applies to the usage of geometrical representation instead of generating a finite element mesh as described by Kagan [4]. In this case it is therefore necessary to use the original geometry as created in the CAD-system.

A lot of CAD-systems include integrated finite element analysis modules. Arabshahi [5] describes the difficulties software developers are facing when trying to implement such tools. A reason for such problems is the communication between the different modules of each CAD-system. This study investigates some of these integrated finite element analysis modules to show what can be achieved with the standard settings implemented in the different modules by simulating the work of a construction engineer when doing a strength determination. Typically the design giving engineer does not have extensive knowledge about the theories behind the simulation but is able to use the program and has a basic understanding of the proceeding of a simulation. The basic question is how significant the results are and whether or not they can be trusted. The following computations were accomplished with all integrated modules using the automatically suggested settings. This refers to e.g. geometric reduction, mesh controls, solver controls and pre-processor settings. Another comparison is based on the different element types that can be used within those systems. Finite elements build the finite element mesh. The finite element mesh can be understood as a discrete representation of the geometric model [6]. On this and other related topics several papers have been published. Kurowski [7] describes the effects chosen element types have for the results of a static analysis focusing on the differences between 4 node and 10 node tetrahedrons, whereas Ramos [8] outlines the differences between tetrahedrons and hexahedrons. Another contribution in the field of element comparison was made by Koch [9] showing the close relationship between choice of element type, geometrical model and mesh generation based on several simulation studies. Maus [10] discusses the influence of different element types with focus on the different input parameters and how they should be documented. A recent study about different FEA-programs shows the differences in the field of usability without comparing results of an analysis [11].

1.3   Software overview

As already mentioned, both CAD-integrated modules and standalone FEA-packages were used in this study to measure how good the performance of the integrated solutions is compared to the high-end simulation suites. Because of the great variety of CAD-software on the market, three of the most commonly used CAD-suites with integrated FEA-modules were used, and the simulations were done without changing the settings proposed by the systems. The systems chosen for the comparison are the current releases of the CAD-packages by PTC, Dassault and UGS.

Pro/ENGINEER Wildfire 2.0 by PTC has the integrated module Pro/Mechanica which can be used for structural and thermal analysis. The parts were created in the Pro/E standard module and the simulations were done in Pro/Mechanica using the internal solver [12]. The second CAE-package analysed was CATIA V5 R16. The product developed by Dassault Systèmes provides engineers with the module "Generative Structural Analysis" to carry out static FE-analysis of parts and assemblies [13]. NX 4 by UGS relies on the integrated module "Enhanced Simulation" that makes use of the solver NX-Nastran, an adaptation of MSC´s Nastran [14]. All of these CAD-suites are capable of meshing complex parts. For the comparison tetrahedron meshes were generated if possible both as 4-node and as 10-node tetrahedrons.

Three Packages from the wide range of standalone simulation-packages were also used. ABAQUS Version 6.6-1 from ABAQUS Inc., a member of Dassault Systèmes, allows the simulation of a large number of physical and mechanical, linear and nonlinear problems. For pre- and post-processing ABAQUS CAE was used, and for solving the simulation examples the iterative ABAQUS Standard was adducted. From the MSC Software Corporation MSC.Patran for pre- and post-processing and MSC.Marc`s linear solver were used. In contrast to most integrated finite element modules that can only solve linear problems, the standalone finite element software from MSC and ABAQUS Inc. are able to solve more complex cases using a multitude of different element types, material models and analysis types. In this study these software packets were used for comparison with integrated finite

element modules of CAD-systems. Therefore the settings within the standalone packets were adapted to approach those proposed by the integrated modules. The main aspect is the choice of the element type [15, 16].

In addition to the commercial software, the freeware program Z88 Version 12.0 developed by Rieg [17] was included in the comparison. This software offers the possibility to import structures meshed by Pro/MECHNICA or ABAQUS.

In the following the software packages were made anonymous and were numbered 1 to 6.

2.   EXPERIMENTAL

2.1   Presentation of the simulation examples

5 examples were selected to consider different aspects such as boundary conditions, loads and materials within the simulation. The first module was a beam. On this beam a constant line load was applied on one edge while the other edge was restrained. The material selected was standard steel with Young's Modulus of 206000 MPa and Poisson's ratio of 0.3. The advantage of this example was the possibility to get results (stress and displacement) by using engineering mechanics. Otherwise the beam example is very simple and therefore it was expected that the results of the different software packages would be similar.

The next example was a holding bush. A surface load in axial direction and a pressure on the outer surfaces were applied. An axial bearing on the surface at the end of the holding bush and radial bearing on the surfaces in the middle borehole were added as constraints. The same steel as in number one was used. Based on this example the differences in the interpretation of the surface loads of the different software could be shown.

The third example was a crankshaft. There is an axial bearing on one side of the crankshaft and a radial bearing on every side of the crankshaft. A surface force orthogonal to the rotation axis where the con-rod is attacking is given (Figure 2, Area A). The material of the crankshaft has the same properties as example number one. The main aim of this example was to confirm the results of example number two.
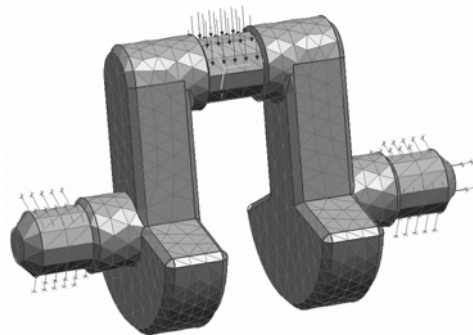

figure 2: boundary conditions and load for the crankshaft assigned in UGS NX 4.0

Another simulation example was a piston of a commercial diesel engine. A pressure was applied on the whole surfaces of the piston bowl and partially on the piston head. The thrust bearing was applied on the surfaces of the borehole where the piston pin is normally located. As material the above steel was used. This example was chosen because the load was a pressure.

As final example a fan was simulated, and boundary conditions and loads were applied. In the middle borehole a thrust bearing was fixed. A pressure was applied as load on every rotor blade (figure 3). The plastic material used in the simulation had a Young's Modulus of 17000 MPa and a Poisson's ratio of 0.335. The main reasons for simulating this example were that the geometry is the most complex in these tests and because it is the second example with pressure as load [18].
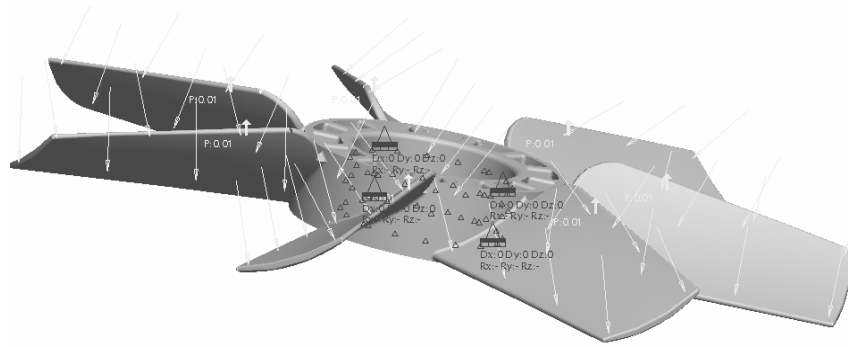
figure 3: boundary conditions and load for the fan assigned in Pro/ENGINEER Wildfire 2.0

2.2 Proceeding

The first step was the generation of the geometry within each CAD-system, because every system uses different internal formats to represent the geometry. Possible mistakes during this process would be brought forward into the integrated finite element module. Another option is the modelling in one CAD-system followed by a data exchange of the geometry with the other programs. During the export out of a CAD-system and the following import in an other CAD-system, mistakes can be made [19]. The complete proceeding of one simulation has to be done in one program, including the geometric modelling. There were exceptions for the standalone systems and Z88V12 because their primary function is not the design application. The geometry of the examples was designed in Pro/ENGINEER Wildfire 2.0 and imported using the parasolid format. The next step was the integration of the geometry into the simulation modules. Afterwards the material was assigned and the model was meshed using the implemented standard settings for the mesh generation. The standard settings included the method of meshing. One method is the h-method and the other is the p-method. The h-method uses more elements for the discretisation of the same geometry while the p-method uses element types with a higher order of degree of the polynomial (figure 4).
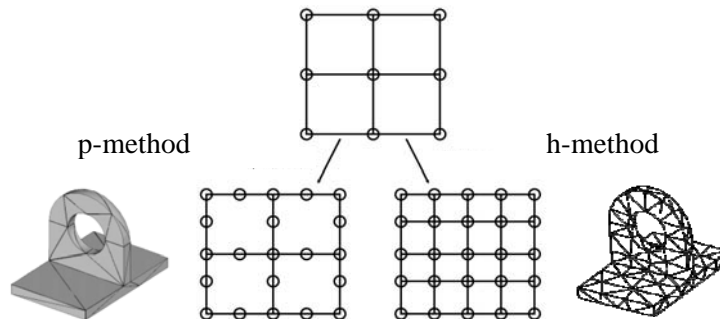

figure 4: difference between p- and h-method [9]

The element type (linear element (tet4) or quadratic element (tet10)) was only chosen if the software simulated with the aid of the h-method. No influence was exerted on the degree of the polynomial if the software simulated with the p-method. The last step for solving the problem was to apply the boundary conditions and the loads as described above. The whole proceeding is shown in figure 5.
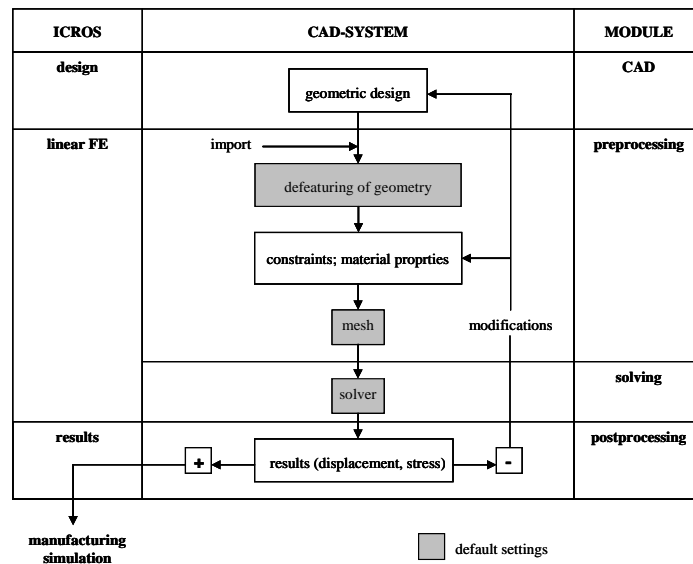
figure 5: proceeding

3. RESULTS AND DISCUSSION

The following tables (table 1 and table 2) show the results. Each software has two columns, one for every element type next to the software simulated with the aid of the p-method.

table 1: results for the beam and the holding bush

program	1[1]	2		3		4		5		6	
	beam										
method	p	h	h	h	h	h	h	h	h	h	h
e-type	tetra	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10
elements	211	664	664	646	646	2422	2422	34864	34864	31737	31737
nodes		272	1477	260	1423	858	4979	8442	56827	7846	52340
disp.	1,00/1,00	0,82	1,00	0,83	0,99	0,90	1,00	0,50	1,00	0,98	1,00
vMises	1,00/0,48	0,65	0,40	0,60	0,88	0,69	0,36	0,71	0,85	0,81	1,00
	holding bush										
method	p	h	h	h	h	h	h	h	h	h	h
e-type	tetra	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10
elements	1482	29051	28978	5403	5403	33220	33220	45617	45617	13787	14014
nodes		7679	49191	2167	11792	8727	56309	11100	11100	4017	25124
disp.	0,33/0,20	0,26	0,20	0,16	0,15	0,27	0,23	0,12	1,00	1,00	0,29
vMises	0,59/0,22	1,00	1,00	0,14	0,11	0,37	0,18	0,19	0,82	0,84	0,26

Legend:
listing of meshing method (method), element type(e-type), number of elements (elements), number of nodes (nodes), normalized displacement for each element type (disp.) and normalized stress for each element type (vMises) for the beam and the holding bush

---

[1] the first program uses the p-method, therefore an element variation was not possible. The first number by disp. and by vMises is normalized with the linear element and the second number with the quadratic element.

<p align="center">*table 2: results for the crankshaft, piston and fan*</p>

program	1[2]	2		3		4		5		6	
	crankshaft										
method	p	h	h	h	h	h	h	h	h	h	h
e-type	tetra	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10
elements	1140	11052	11052	3897	3897	47736	47736	90309	89940	47088	123296
nodes		2829	18423	1218	7319	11127	75744	18732	134302	10019	177520
disp.	1,00/0,98	0,85	0,96	0,77	0,96	0,95	1,00	0,59	0,60	0,60	0,63
vMises	1,00/0,95	0,31	0,46	0,20	0,35	0,45	0,51	0,40	1,00	0,23	0,34
	piston										
method	p	h	h	h	h	h	h	h	h	h	h
e-type	tetra	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10
elements	3888	12261	12261	5359	5359	63272	316675	63272	63272	15070	214778
nodes		3185	20818	1601	9950	13533	215679	13533	96341	3610	311614
disp.	1,00/1,00	0,67	0,77	0,57	0,73	0,77	0,70	0,77	0,70	0,596	0,65
vMises	1,00/1,00	0,53	0,73	0,46	0,85	0,58	0,84	0,58	0,96	0,47	0,88
	fan										
method	p	h	h	h	h	h	h	h	h	h	h
e-type	tetra	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10	tet4	tet10
elements	10970	17567	17567	25240	29624	130541	222783	35205	35205	49073	229613
nodes		5890	34868	8842	59968	34495	130541	10849	65840	14608	375055
disp.	1,00/0,89	0,27	0,89	0,38	0,97	0,53	0,90	0,35	1,00	0,42	0,90
vMises	1,00/1,00	0,26	0,85	0,20	0,77	0,77	0,91	0,38	0,94	0,43	0,94

Legend:
listing of meshing method (method), element type(e-type), number of elements (elements), number of nodes (nodes), normalized displacement for each element type (disp.) and normalized stress for each element type (vMises) for the crankshaft, the piston and the fan

Within these tables the method of meshing, element type, number of elements, number of nodes, normalized displacement for each element type and normalized stress for each element type are shown. The basis for normalization was the maximum value of all compared software.

---

[2] the first program uses the p-method, therefore an element variation was not possible. The first number by disp. and by vMises is normalized with the linear element and the second number with the quadratic element.

diagram 1: normalized displacement simulated with linear element type
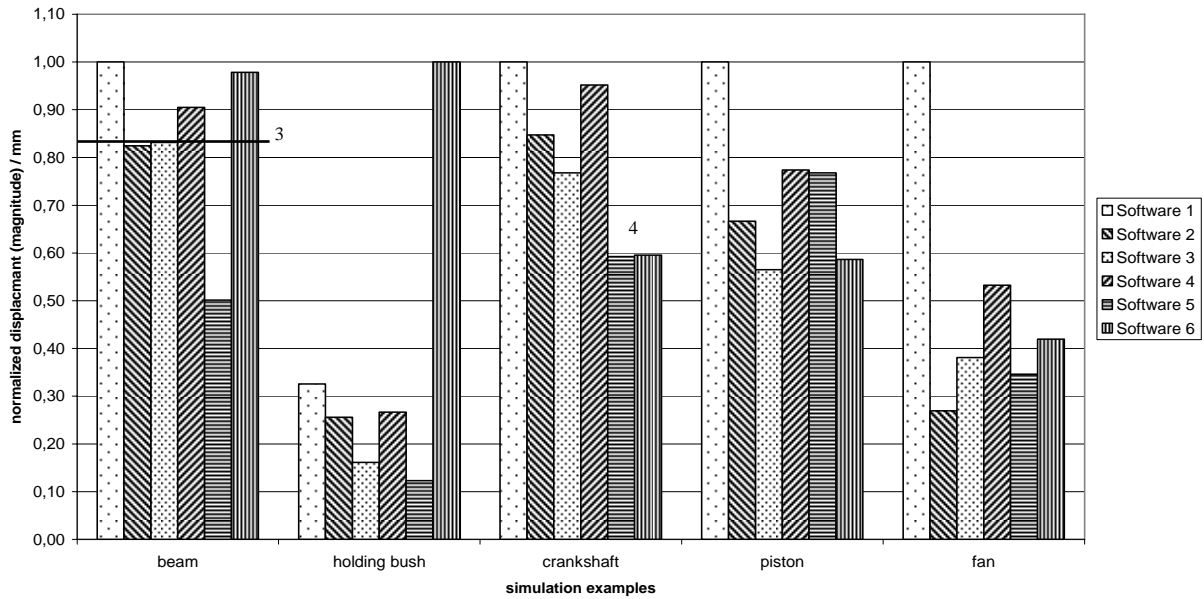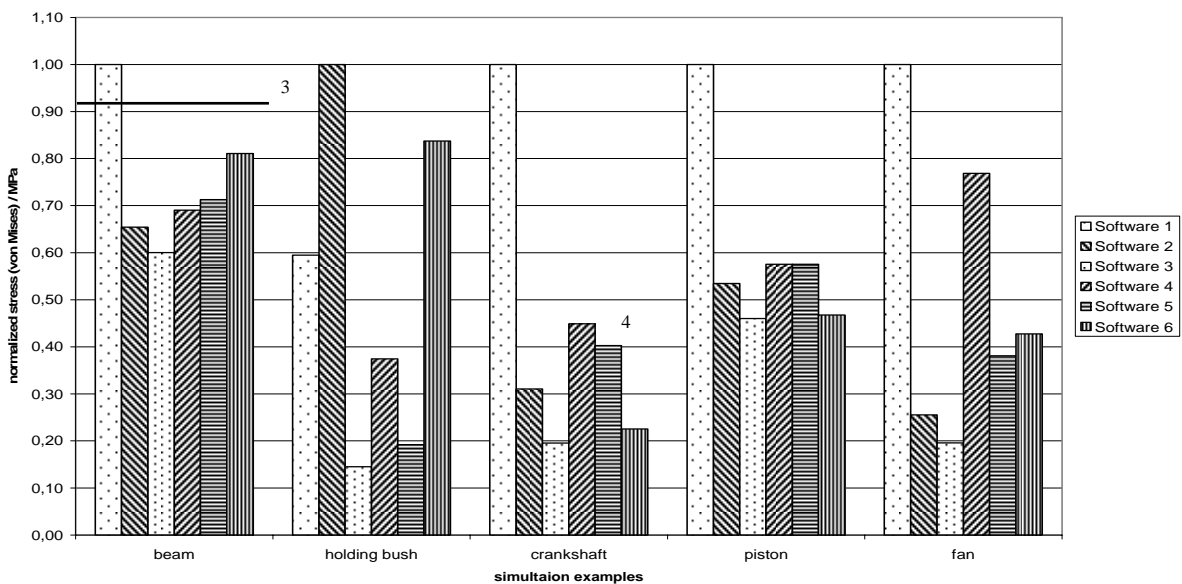

diagram 2: normalized von Mises stress simulated with linear element type

The normalized result for the beam example verified by using engineering mechanics was 0.82 for the displacement and 0.92 for the stress. At the corners of the rigid linear elements where the line load was applied maxima could occur, which led to a higher displacement. The difference between the results for the displacement obtained by using several software types was 50 percent. But most of the results showed a smaller tolerance. Within the simulation of the displacement only software 5 gives a much smaller value than the result obtained by using engineering mechanics. The average value for the displacement for example 1 is 0.86. This is a good value compared to the analytical result. Regarding the stress the result verified by using engineering mechanics is undershot five times. This is more dangerous than an exceeding. The average value of the stress is 0.79.

---

[3]: result obtained by using engineering mechanics
[4]: at software 5 and 6 the load was not a surface force but a pressure

With regards to the holding bush the highest displacement value of software 6 was due to the fact that the surface force could not be applied. The same effect was observed with software 5. The surface force was applied directly on the mesh by using nodal forces. The obverse rules were used as Bathe [20] and Zienkiewicz [6] describe. Those rules were applied for software 5 and 6, but the results were different. The high value of the displacement of software 6 was basically due to singularities on some nodes at the corners of the element where loads were applied. The other results show a difference of 20 percent. The displacement averages 35 percent (including software 6). The highest differences between the results of the software within the linear elements occur to the normalized stress the average value of which is 0.52.
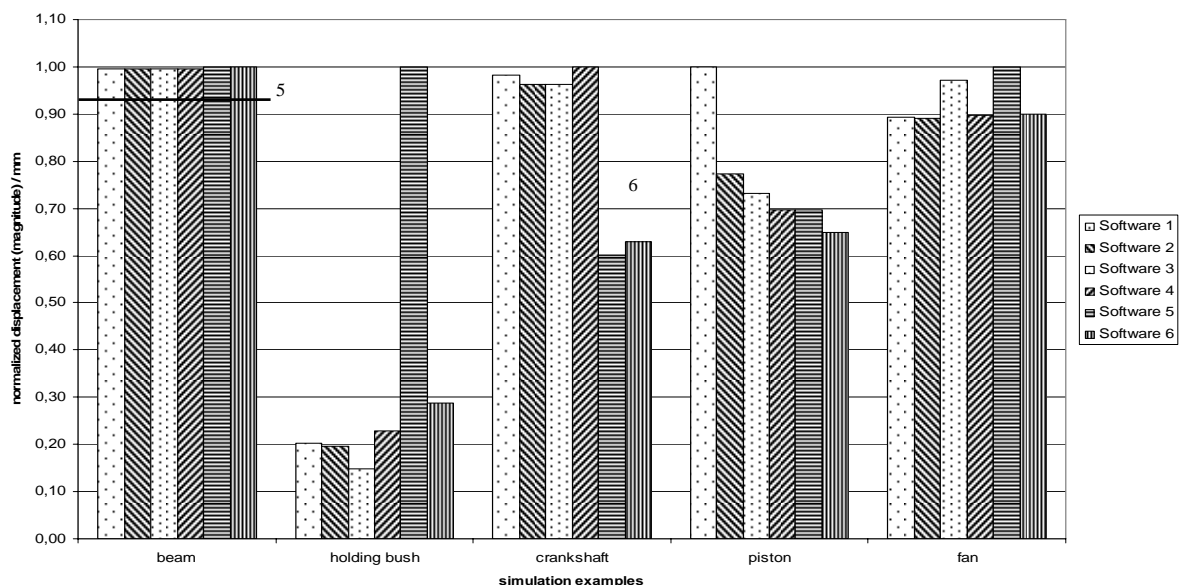
At the crankshaft the load was not a surface force but a pressure. This is essential for software 5 and 6. The consequence was that the results of these software were not considered within the comparison. The main difference between a surface force and a pressure is the direction. A pressure always knits vertical on a surface, whereas the force has a direction. In this example the direction of the force is vertical on the rotation axis of the crankshaft. The simulation results of the displacement when pressure was applied were almost equal. The difference of the displacement between other software was 25 percent. The average value is 89 percent (software 5 and 6 were not included). For the stress software 1 simulated a much higher value than the other three programs. Noticeably the difference between the results of the stress computed by software 5 and 6 was 50 percent, although the displacement was almost equal.

The piston was the first example every program simulated with pressure as load. Only software 1 shows problematic values for both results. The average value of the displacement is 0.72. The same is observed for the results for the stress. But the results of the stress simulated with the other programs are more similar in size to the displacement results.

The same results as in example 4 are shown in the last simulation study, the fan. Again the attribute pressure as load was used. The main feature of this example was the complex geometry, and it was designed to see if every program was able to mesh the geometry. The result with regards to this was that every program could mesh it without any problems. The average value of the displacement is 0.49. Regarding stress the results of example 4 could not be repeated in the fan example. Again Software 1 shows the highest value, but the difference amongst the results of the other programs is smaller.

The next two diagrams show the normalized displacement (diagram 3) and normalized stress (diagram 4) simulated with quadratic element type (tet10).
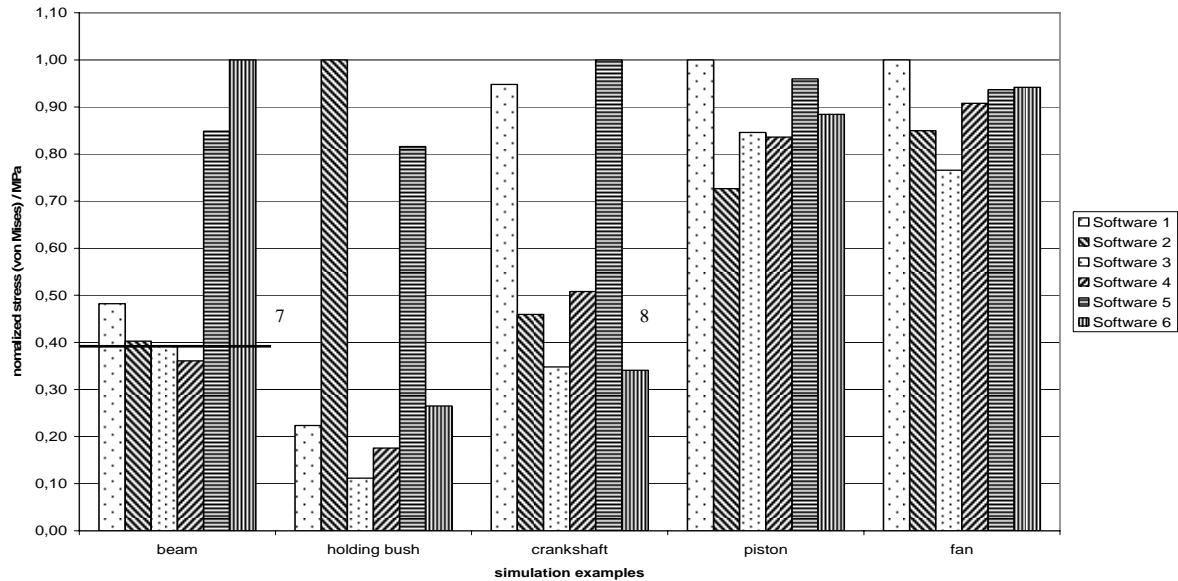
diagram 3: normalized displacement simulated with quadratic element type

---

[5]: result obtained by using engineering mechanics
[6]: at software 5 and 6 the load was not a surface force but a pressure

diagram 4: normalized stress (von Mises) simulated with quadratic element type

Regarding displacement the best result within the study was achieved in the simulation of the beam. All programs computed almost the same displacement value, although this does not apply for the stress. One reason for the higher stress value simulated with software 5 and 6 is that these programs could not apply linear line loads. Hence the line load was applied by using nodal forces. At the middle nodes of an element with a quadratic form function the nodal force is two thirds of the whole line force for this element resulting in higher stress values computed for these nodes. The results of the other four programs only had a difference of 0.1.

In the example of the holding bush the highest differences were simulated between the results for both the displacement and the stress. The surface force was manually applied by software 5 and 6 through a nodal force directly onto the mesh. The high value for the displacement simulated with software 5 is extraordinary considering that the displacement of software 6 is similar to the rest. Software 2 computed the highest stress. This was unexpected because software 5 has the highest displacement. The other programs showed a difference of 0.15.

The displacement calculated at the crankshaft example confirms the differences between a surface force load and a pressure load. Software 5 and 6 simulated with pressure as load and calculated almost the same result for the displacement. The same result was shown by software 1 to 4. For the stress one software out of every group always simulated a higher value as the rest of the group.

Software 1 shows the highest displacement value in the piston example. All other programs calculated a displacement 0.2 smaller than software 1. The average value of the stress is 0.88.

The best results were produced by example 5, the fan. The normalized displacements vary between 0.9 and 1.0. The normalized stresses show more differences but these are still only about 20 percent.

4. CONCLUSION

In this study several structural analyses with typical part geometries were made using six different software solutions. While three of these software systems were CAD-integrated finite element tools the others were standalone simulation-software-programs. This was done in order to determine whether or not the integrated modules were suitable for a linear strength determination within the ICROS method. All simulations in the integrated modules were done without modifying the standard settings proposed by the finite element analyses tools to exemplify an analysis in which the designing engineer only has basic knowledge of finite element procedures. The structural analyses were done with finite element meshes consisting of linear 4 node tetrahedrons and, if possible, of quadratic 10 node tetrahedrons.

---

[7]: result obtained by using engineering mechanics

[8]: at software 5 and 6 the load was not a surface force but a pressure

The actual results of these simulations lead to the conclusion that the examined FEA-modules are only suitable to a limited extent for appliance in ICROS. When an elementary structure (I-beam) that could also be verified using engineering mechanics was used the results for the maximum displacement showed a very good correlation. When 10 node elements were used, however, the stresses calculated by the programs showed significant differences. The examples with complex part geometry show a remarkably wide range of results both for the displacement as well as for von Mises stresses. The relative values showed a discrepancy of over 50 percent at times resulting in a stress that was twice as high.

The main statement for education is, that it is not enough to teach how to use a finite element software but the understanding of the finite element method and the mathematic background. It is important that new users of such softwarepackets have good knowledge about the FEM, so they can interpret the results.

Possible reasons for these results may be found in differences in the integration order the programs use or in the internal routines utilised to compute stresses. These internals of the integrated modules cannot be analysed by the user nor can they be altered. As a result it is assumed that the predicted maximum stress can only be a useful indication for strength determination and that further comparative studies have to be done in order to verify the results. From the viewpoint of a typical design engineer, employee of a big company, who only has access to one program and mainly has to rely on the computed von Mises stresses when dimensioning the part, it would most definitively be necessary to scrutinise the plausibility of the computed results further. Another way to validated the simulated results is to use prototypes or zero-series for excessive testing.

REFERENCES

[1] **Alber B., Hackenschmidt R., Rieg F.** Bauteilentwicklung mit Hochleistungskunststoffen - Intelligent vernetzte Simulationen mit anschließender Versuchsverifikation. *33. Tagung DVM-Arbeitskreis Betriebsfestigkeit*, Steyr, 2006, pp.

[2] **Hackenschmidt R., Alber B., Rieg F.** Intelligente Verknüpfung von Simulationsprogrammen. *CAD-CAM Report*, 2007, **4**.

[3] **Alber B., Hackenschmidt R., Rieg F.** ICROS-The selective approch to high-tech polymer product design - modelling and experimental verification. *International Design Conference - Design 2006, Vol. 1*, Dubrovnik, 2006, pp. 417-424 (The Design Society, Glasgow).

[4] **Kagan P., Fischer A.** Integrated mechanically based CAE system using B-Spline finite elements. *Computer-Aided Design*, 2000, **32**, 539-552.

[5] **Arabshahi S., Barton D. C., Shaw N. K.** Steps towards CAD-FEA Integration. *Engineering with Computers*, 1993, **9**, 17-26.

[6] **Zienkiewicz O. C., Taylor R. L.** . *The Finite Element Methode*. Butterworth Heinemann, Oxford, 2000.

[7] **Kurowski, P. M.** A closer look at model meshing. *Machine Design*, 2004, **82**, 82-88.

[8] **Ramos A., Simoes J. A.** Tetrahedral versus hexahedral finite elements in numerical modelling of the proximal femur. *Medical Engineering & Physics*, 2006, **28**, 916-924.

[9] **Koch, F.** *Elementauswahl bei Finiten Elemente Analysen*. Shaker Verlag, aachen, 2003.

[10] **Maus R., Weber C.** Dokumentation und Vergleich von FEM-Simulation. *CAD-CAM Report*, 2005, **10**, 50-53.

[11] **Gebhardt, C.** Virtueller Umlaufbiegeversuch. *KEM*, 2007, **3**.

[12] **Parametric, Technology Corporation**. *Simulation and Analyse*. Needham, 2005.

[13] **Dassault, Systèmes**. *Catia V5 Documentation*. Suresnes, 2005.

[14] **UGS, Corp.** *NX 4 Hilfebibliothek*. Plano, 2005.

[15] **Abaqus, Inc.** *Abaqus/CAE user`s manual*. Providence, 2006.

[16] **MSC.Software, Cooperation**. *Guideline MSC.Patran 2005 r2*. Santa Ana, 2005.

[17] **Rieg, F.** *Handbuch für Z88 Version 12.0*. Lehrstuhl für Konstruktionslehre und CAD, Bayreuth, 2006.

[18] **Rieg F., Hackenschmidt R.** *Finite Elemente Analyse für Ingenieure*. Hanser Verlag, München, 2003.

[19] **USA, Inc. Kubotek**. The 2006 CAD Interoperatbility Survey Results. *A Kubotek USA Study*

of the Design and Manufacturing Marketplace, october 2006, pp.

[20] **Bathe, K.-J.** *Finite-Elemente-Methoden*. Springer-Verlag, Berlin Heidelberg, 2001.

Contact: Dipl.-Ing. Bernd Roith
       Scientific Assistant
       Department of Engineering Design and CAD
       University of Bayreuth
       Universitaetsstr. 30
       95447 Bayreuth
       Germany
       Tel.: +49 921 55 7226
       Fax.: +49 921 55 7195
       E-Mail: bernd.roith@uni-bayreuth.de
       URL: http://www.uni-bayreuth.de/deparments/konstruktionslehre