Journal of Physics: Conference Series

PAPER • OPEN ACCESS

Comparative study of finite element analysis software packages

Journal of Physics: Conference Series **1515** (2020) 032073 doi:10.1088/1742-6596/1515/3/032073

Comparative study of finite element analysis software packages

I A Magomedov[1] and Z S Sebaeva[2]

[1]Faculty of information technology, Chechen State University, 32 Sheripov Street, Grozny, 364024, Russia
[2] Faculty of secondary vocational education, Grozny State Oil Technical University named after academician M. D. Millionshchikov, 100 HA. Isaeva pr, Grozny, 364051, Russia

E-mail: ismwork@mail.ru

Abstract: research of the work is dedicated on comparing the abilities of Finite Element Analysis of software packages Abaqus (Computer-aided engineering), Ansys (Mechanical), SOLIDWORKS and Inventor Nastran. The next software packages were selected as they are the popular products in the market today. In the Introduction the meaning of Finite Element Analysis tool is explained. Abaqus (Computer-aided engineering), Ansys (Mechanical), SOLIDWORKS, Inventor Nastran are described in the part of Literature review. From point of view their capabilities and specific features by compared to each other. Using the available data form the software packages the comparison took place. The outcome of the research illustrated that all four software packages capabilities are almost the same. However, the most diverse from the list was SOLIDWORKS. Abaqus (Computer-aided engineering), Ansys (Mechanical), Inventor Nastran has similar abilities. The difference was that Inventor Nastran is not capable of performing acoustic analysis. SOLIDWORKS differ from others in terms of it does not performs acoustic, Electric/magnetic Fluid flow and Fluid structure interaction. it also was point out that limited versions, that come free has some restriction when dealing with mesh. It was said that limitation constrains the quantity of nodes in the structure, which limits the analysis. By last the results, it was observed from the given data that SOLIDWORKS is aimed mainly on modeling a part or assembling the whole structure and doing available analysis in it. Hence SOLIDWORKS is marketed and aimed on modelling and assembly the other three software packages Abaqus (Computer-aided engineering), Ansys (Mechanical) and Inventor Nastran are analysis tools, which are utilized by the investigators and industry.

1. Introduction
In the last three decade huge step forward for the finite element analysis. The tool become handy for the researchers to analyse almost every task that can be analysed practically. Progress of Finite Element Analysis (FEA) tool directly coincides with the accessibility and affordability of powerful calculating architectures (computers) [1]. In simple words, finite element analysis (FEA) described as a numerical approach, that is aimed to evaluate some difficulties related to engineering and physics (mathematical). It is used when dealing with complex constructions (body), material properties, applied loadings and constrains, where real approach results cannot be performed.

As every great inventions, finite element analysis tool similarly took its beginning around mid of 20 century. Its occurrence can be related to the necessity of lightweight structure, exact stress analysis and so on. At that, time researchers have no concept that their new discoveries will be used in such a manner. With the new inventions and concepts finite element analysis tool grew into a powerful tool with abilities to analyse complex problems.

There are many common areas where FEA utilized. Into these areas (groups) goes all kind of engineering fields: mechanical, aerospace, civil and automotive. The approach is designed to carry out dynamic, static and also linear and non-linear analysis etc.

Finite elements can be described as elements of one whole structure. For example, a 3D object can be divided into smaller elements (which are finite elements) and each element has its own number of nodes or nodal points depending on its complexity. Finite element analysis is not restrained to analyze only 3D problems, but able to analyses two-dimensional problems (the problem can be simplified in order to minimize the time recurrence for the analysis.

The general procedure when performing FEA. The procedure can be separated into the three main group of preprocessing, solution and postprocessing. Preprocessing covers all the boundary conditions, material selection and appliance and mesh generation and modification, also additional parameters might be applied such as surface smoothness, interaction and frequency etc. Second one is solution of the task and can be described as the solver of unidentified numbers of the primary field of variables. Last one is post processing of the problem. Postprocessor comprises sophisticated routines utilized for further plotting graph and illustrating results [2].

There are dozen of computer softs to perform FEA. The followings are used as FEA tool: ANSYS, Abaqus, SOLIDWORKS, Invertor Nastran etc. The list of commonly used FEA tool in the market can be extensive, but they all simply performs one task, which is to solve the problem (partial differential equations).

2. Literature review

Ansys is one from many of the commonly utilized product in the market for FEA. Ansys, inc develops a wide-ranging of computer-aided engineering CAD products, however best recognised for Ansys Mechanical/Multiphysics. FEA tool Ansys is self-contained analysing tool, which includes pre-processing, solver and post processing divisions. It can be pointed out that Ansys is capable of user-programming. The tool Command Language contains ten hundred command, which can be used to program or modify mesh, geometry, boundary conditions and many other futures. Therefore, Interactive and batch modes are used when dealing with FEA in Ansys. The batch mode needs commands to be inputted for the analysis to run and used mainly by those who are familiar with the Ansys Command Language. Consequently, the interactive mode refers to visualisation, where graphical interface is used to input data, select options etc [3].

Abaqus is powerful tool with user friendly interface for creating two dimensional sketches and three dimensional objects and then after application of boundary conditions can be transferred to simulation section (and numerous functions). The soft is manly separated into different stages. For instance, different stages consist of geometry creation, properties of material and also generator of mesh etc. Each modulus has its own tasks. CAE generates input file. The other two are dedicated to evaluation of the problem. Then the last-mentioned versions of Abaqus does the study, which is then sent to CAE to let researchers to visualize the progress of the problem. The product generates results (output data).

To visualise the outcome (results) the separate software Abaqus (here referring to Viewer) can be utilized. CAE used for pre-processing and post processing. CAE offers range of capabilities to study the following fields: acoustics, structure's damage, fracture and failure. Standard and Explicit (referring to software) are implicit and explicit solvers respectively. Standard is a finite element FE solver, which customs an implicit approach. The product is best suits with static investigation and low-speed dynamic events where accurate stress results are very significant. Whereas second one, which is Explicit software uses an explicit scheme, which ideal to evaluate nonlinear systems with very high number of complex interactions with applications of transient loads [4].

Software SOLIDWORKS mostly utilized for modeling separate parts and then assembling them for the future evaluations. SOLIDWORKS is most popular software for modelling complex structures and assembling different parts together in the marked. Almost 3.5 million of license sold worldwide. SOLIDWORKS has easies to work environment when modelling, assembling or analyzing. The product is capable of metal fatigue, pressure vessel, thermal structural etc. lately the product SOLIDWORKS is able to execute study of non-linear problem. As any other similar instrument for performing nonlinearity of a problem under deformations (load applications and conditions of material) SOLIDWORKS lets researchers to confirm (confirm/validate) product (structure) quality of the product's performance and safety throughout the creation [5].

Inventor is one of the Autodesk platform. The product is similar to SOLIDWORKS. The soft is self-contained and has built in sections or modulus. The sections include parts, assembly, drawing and presentation. The product has all stages from creating a structure with further assembly and its analysis. The software has simulation environment where FEA can be done [6].

3. Comparison

In this paragraph the above-mentioned software packages Abaqus (Computer-aided engineering), Ansys (refering to Mechanical), Inventor Nastran and Inventor (referring to Nastran) will be compared. The table 1 represents some available data of capabilities and futures of given softwares. However, over the time some of the listed parameters or functions might change and loose their usability. The comparison was done by using only data , which is given in the table 1, but there are other features that are not listed in this work (which might be the same or different for the products).

Table 1. comparison of competencies of the following products: Abaqus, Ansys, OLIDWORKS and Inventor

	Abaqus (CAE)	Ansys (Mechanical)	SOLIDWORKS	Inventor (+Inventor Nastran)
Self-contained	no	yes	yes	no
Graphical geometry modeler	Includes	Includes	Includes	Includes
Graphical manual meshing	Includes	Includes	No data	Includes
CAD import	Capable	Capable	Capable	Capable
Units aware	no	yes	No data	No data
Linear static	Performs	Performs	Performs	Performs
Nonlinear - large displacements	Performs	Performs	Performs	Performs
Nonlinear - contact	Performs	Performs	Performs	Performs
Transient linear	Capable	Capable	Capable	Capable
Transient onlinear	Capable	Capable	Capable	Capable
Natural frequency	Capable	Capable	Capable	Capable
Linear buckling	Capable	Capable	Capable	Capable
Acoustic	Capable	Capable	Not Capable	Not Capable
Heat transfer	Capable	Capable	Capable	Capable
Electric/magnetic	Capable	Capable	Not Capable	Capable
Fluid flow	Capable	Capable	Not Capable	Capable
Fluid structure interaction	Capable	Capable	Not specified	No data

Solid elements	Capable	Capable	Capable	Capable
Shell elements	Capable	Capable	Capable	Capable
Price	Limited free	Limited free	Not free	both

4. Discussion

Four popular software packages were compared to each other on behalf their ability to perform some features. By observing the table 1 one can agree on the similarity of the selected software. However, there is some small difference. The first two software packages Abaqus and Ansys (CAE and Mechanical respectively) has identical capabilities. The only difference they have is self-contained section and Unit aware part.

In Additional, these two software packages come as in two options full version and limited free. Limited version has constrained amount of nodes allowance. It means complex analysis where larger mesh generation is undoable. Hence limited version is more suited for simple analysis with lesser mesh. Inventor Nastran is also similar with the Abaqus and Ansys (CAE and Mechanical respectively). It has student version for three year of free license and the only difference of software is luck of capability of doing Acoustic analysis. SOLIDWORKS the only software, which is marked as version where no free license is is avalable from the list and has few limitation if to be compared with the other products. The limitation is not being capable to perform acoustic, Electric/magnetic Fluid flows. However, SOLIDWORKS is more powerful in modelling and assembly if compared to other three. Usually SOLIDWORKS is used to make a model or assembly of separate parts and then transfer the structure to Abaqus (CAE), Ansys (Mechanical) or Inventor Nastran.

5. Conclusion

To conclude the comparative study of capability of software packages was done. Four popular software packages were compared, which are Abaqus (CAE), Ansys (Mechanical), Inventor Nastran and Inventor Nastran. The brief introduction of each software packages were given. In the comparison paragraph the abilities of each software were compared. The results illustrated that three of the presented software were similar and SOLIDWORKS has been different from the three. Yet it was stated that SOLIDWORKS is more powerful in modelling and assembling, while others show the abilities to analyse complex problems by judging the table 1.

References

[1] Vijay K, Vinay K 2014 *Finite Element Analysis* (Pearson Education Press) p 4
[2] Indrajeet P 2008 Introduction to Finite Element Analysis. *ITTI Update* pp 1-5
[3] Kim H, Sankar V, Kumar A 2008 Introduction to Finite Element Analysis and Design, 2nd Edition p 363
[4] Izabela C W 2019 Basics of Abaqus CAE. *Finite Element Modeling of Textiles in Abaqus* pp. 13-57
[5] Matt L 2018 Mastering SolidWorks. (John Wiley & Sons) pp 13-57
[6] Kishore T 2017 Getting Started with Autodesk Inventor 2018 (Learn Autodesk Inventor 2018 Basics) pp 1-14
[7] Magomadov V 2019 Deep learning and its role in smart agriculture. *Journal of Physics: Conference Series*
[8] Lluís O A 2014 Comparison between ansys and abaqus using ultrasonic Guided waves *Department of Mechanical Engineering École de Téchnologie Supérieure* pp 75-76.
[9] Lloyd T 2006 Numerical Analysis, Oxford University pp 3-4
[10] Schäfer M 2006 Computational Engineering - Introduction to Numerical Methods Edition Number 1