Key steps for performing numerical analysis utilizing modern software tools

I A Magomedov¹,², Z A Dzhabrailov² and A A Elmurzaev³

¹ Faculty of information technology, Chechen State University, 32 Sheripov Street, Grozny, 364024, Russia
² Faculty of information technology, Chechen State University, 32 Sheripov Street, Grozny, 364024, Russia
³ Department of Technological Machines and Equipment. Grozny State Oil Technical University named after academician M. D. Millionshchikov, 100 H A Isaeva pr, Grozny, 364051, Russia

⁴ E-mail: ismwork@mail.ru

Abstract. The work was written to outline important steps of numerical analysis. This work presented the guidance to perform any analysis in the engineering design tools. At present, software tools are used in every field, and hence, it is essential to understand some techniques to utilize them. The following work gives guidance from the modeling of a given structure to the applying boundary conditions to it for further analysis. 3D modeling, the appliance of material contacts, constraints, loads and mesh are outlined in this work. Also, the work provides postprocessing guidance. For instance, the use of Sensitive Analysis tool to shorten the time during numerical analysis.

1. Introduction

We live in a time where new technology is dramatically influencing our life. It permeates almost every aspect of our life and, thus, making us dependent on them. It is true that they, innovations, make the lives of millions comfortable by performing some of the activities. These innovations can be met in our daily routine life whether it is a TV, refrigerator, electronic devices, vehicle, etc. Therefore, innovations can come in all different shapes and sizes, but the core of every technology lies in the software that drives it to perform its purpose.

Usage of software depends on the purpose of a given task. Different versions of software can be utilized to fulfill the required needs. As it was mentioned application of software packages are vast in the modern time. However, this work will focus on software packages, which are specifically made for numerical analysis and the steps that are required to analysis the given structure (task).

The use of software for research (analysis) has become commonplace. Numerical analysis right now is one of the largest branches of mathematics and with the dozen journals where scientists can publish their work [1,2]. Due to the computational power numerical analysis is applicable in many fields. The application of it can be found in all fields of engendering and furthermore: physical sciences, social sciences, medicine, business and some more. Therefore, today’s software makes possible nonlinearity or linearity of the structure to be calculated. Nevertheless, to validate such an analysis skilled researcher is required as results could be inaccurate.
There are a lot of tools in the market to utilize in performing numerical analysis. The following tools are used to perform both linear and non-linear numerical analysis: Solidworks, Inventor, Onshape, Abaqus, Fusion 360, Ansys, Rhinoceros, Solid Edge and etc. All of the mentioned software can handle most of the given tasks and should all of them give similar results for one given task. Therefore, it comes to the procedure of how analysis is set up. The procedure consists of pre-processing, simulation run and postprocessing.

2. Pre-processing
The following paragraph is devoted to pre-processing of engineering analysis (numerical analysis). The main part of any analysis is a structure. The structure is a representation of a real-world object, which is created using engineering design tools, such as Solidworks, Inventor, Ansys, and etc. In the analysis, the structure should be given some material properties, which will define the way the structure will behave. To analyze the structure external loads must be added. Once the loads are added the constraints must be concluded. At the last mesh must be generated and the structure will be ready to be analyzed. Hence, the followings are common procedures in pre-processing of any analysis: 3D model, materials, contacts, constraints, external loads and mesh.

2.1. 3D modelling
There are several variations that are used when creating a 3D model. The first way uses baselines, extrudes, etc. Another way is using coordinate data that can be entered into the application. Let's dwell on the first method since it is most simple. During the creation process and upon completion, one needs to make sure that the application shows the inscription "Fully defined". This inscription indicates that the model has no errors. "Fully undefined", on the other hand, indicates the presence of errors or the fact that the length is in some place is not specified, which in turn will lead to errors in the execution of the simulation and to inability to complete the analysis. If there is a need to add boundary conditions in a certain location, the dividing line is used. If the model is fully defined it can be then opened in the simulation section [3].

2.2. Materials and contacts
Materials can be applied once the structure is ready in the simulation. Adding material to the 3D objects is mandatory. Software packages, used for analysis have a library with different types of materials. If the required material is missing, then one can add or change the properties of the existing material. Material properties are important during the numerical solution, but sometimes due to the specific of the analysis, some of the properties can be ignored as they are not utilized in the analysis.

Contacts set up is optional and can be ignored, but sometimes they will effect the results depending on contact parameters. Therefore, the correct settings will give more relatable results. By default, if there are any contacts in the structure the set up for the contacts will be bounded. One can understand that to have contacts in the structure two or more objects must be assembled together for the contacts to be set. Hence, depending on the parameters of the contacts different results will occur.

2.3. Constrains and forces
Constrains and loads go together as they counterpart each other. Constraints are used to fix an object in space, otherwise, the object will move in the direction of the applied forces, while the forces will not act on the object itself (deform). There are different constrains, for example: regular fixation, coordinate fixation, cylinder fixation, etc. as constraints are set the forces can be applied to the object. Once the forces are added the magnitude and the direction can be specified. Both mentioned constraints and forces can be applied to vertexes, edges or polygons (segments), which are parts of an element of the structure (meshed structure).
2.4. Mesh
Mesh is used to separate objects into a set of elements as illustrated in the second structure on the right (figure 1). Depending on the study analysis the different types of polygonal mesh can be used: square, triangular, hybrid, and etc. The quality and accuracy of numerical analysis will depend on the size polygonal mesh: the smaller the mesh, the better results of the analysis. However, it is important to consider that an infinite decrease in the polygonal mesh will not lead to improved results, but on the contrary, will increase the time and complexity of the analysis. Therefore, the called “Sensitive analysis” can be utilized to shorten the time for the simulation run [4].

3. Simulation run
Simulation run or solver are used when a given structure is fully defined by adding materials, constraints, forces, and mesh. These steps are the minimal requirements for the solver to be run when performing numerical analysis in the above-mentioned software packages. Once Solver is fed by above mentioned information the following formula 1 is used by software packages. The formula can get dense and complex by giving the solver more complex geometry with many boundary conditions, but the fundamentals will stay the same. In simple words “F” is applied force or load, “k” is the stiffness of the given structure and “d” is displacement or results of the analysis.

\[
\{F\} = [k]\{d\}
\]  

(1)

4. Postprocessing

4.1. Results
As it was mentioned above the following formula 1 is utilized by the software to solve the numerical problems. But to get the results the following formula needs to rearranged as formula 2, which is represented below. Hence, the results can be gathered. By observing formula 2 displacement results can be calculated. Other results are derived from the displacement results.

\[
\{F\} \frac{1}{[k]} = \{d\}
\]  

(2)

4.2. Sensitive analysis
Sensitivity analysis (SA) is utilized to lessen the time required to analysis a given task by running the study over and over to find out the optimal solution. Many software packages are able to calculate (simulate) an enormous amount of data. It can be a complex analysis of fracture mechanics, simulations fluids flow, or composite analysis etc. Software packages mentioned in this work for numerical analysis are well used in industry as they are standards in those fields. In spite of this, the amount of time required to calculate complex analysis is huge. Therefore, can be introduced sensitivity analysis to deal with excessive time usage, where time and quality go together [5,6]. SA chart can be built by using data collected after simulation, for instance, deflection vs a number of elements in the structure (mesh elements) (figure 2). By observing figure 2 its clear the optimal number of elements of the mesh is

![Figure 1. Structure without and with applied mesh.](image)
Around 150. Below the value of 150 the results will vary. Hence, the pattern of correct/stable results will occur after the number of elements passes the value of 150.

**Figure. 2** Sensitive Analysis.

5. **Conclusion**

To conclude, this work provides all necessary to perform simple numerical analysis. However, the complex analysis also follows these steps but might go deep into some of the tools. The work explained all three stages: pre-processing, simulation run and post-process. The given steps are enough to perform any simple task and hence might give the reader some understanding of numerical analysis.

**References**

[1] Bahremand, A and Smedt F 2008 *Distributed Hydrological Modeling and Sensitivity Analysis in Torysa Watershed* (Slovakia: Water Resources Management) 22(3) p 293-408

[2] Zolkin A L, Tyshkov A S and Kalyakulin A N 2010 (Samara: SamGUPS) p 63-6

[3] Magomedov I A, Edaev R R and Rahimov A A 2019 The correct way of constructing 3d model for simulation 4(16) 164-8

[4] Steven J O 1998 A Survey of Unstructured Mesh Generation Technology 239-67

[5] Agus S and Xiaoping Du 2005 Probabilistic Sensitivity Analysis in Engineering Design using Uniform Sampling and Saddlepoint Approximation 1-3

[6] Saltelli A, Chan K and Scott E M 2000 *Sensitivity Analysis* (New York: John Wiley & Sons, Inc.) p 1-3