APPLICATIONS OF NUMERICAL ANALYSIS IN SIMULATION ENGINEERING AND MANUFACTURING TECHNOLOGIES

Haidar Amer [1]
[1]*Stefan Cel Mare University of Suceava, amerhaidar85@gmail.com*

Abstract: *This article summarizes the Numerical Analysis techniques and the basic uses of Finite Element Method in manufacturing and industry, furthermore discuss the Simulation and Numerical calculation power in design optimization and improving quality and reliability. In order to discuss the abilities and power of Finite Element method the article presents practical models which are commonly needed in engineering and manufacturing.*

Keywords: *Numerical Analysis, FEM, CFD, Simulation, Optimization*

1. Introduction

Numerical Analysis is the method to use numerical approximation for equations in a mathematical analysis. Numerical calculation create a great chance to use simulation engineering for all kinds of engineering projects using the high power of computers in our days. This analysis concentrates on obtaining approximate solutions for our problem with an acceptable margin of errors.

1.2. Historical background

Human has started using numerical analysis from a very long time; ancient Egyptian scientists described how to solve simple equations using approximate method (about 1600 BC). The major development of numerical analysis started with Newton and Leibnitz who invented mathematical procedures to solve complex equations and developed the concepts of numerical calculations.

Later important advanced steps for numerical analysis started with algorithms which created by Euler (1707-1783), Lagrange (1736-1813) and Gauss (1777-1855); [See 1, P 1-2].

2. An Overview of Numerical Methods

The main idea of numerical analysis is to study the physical model of our problem and obtain a suitable mathematical model that describes the problem and can control the variables and push forward for solution. Mathematical model means use approximate functions to change differential equations to integral algebraic ones, so we can use the power of computers to solve the new matrix of equations; [See 2, P100].
Based on computational discretization methods to divide geometry and enter boundary conditions we can discuss three basic methods for numerical analysis:
1- Finite Element Method (FEM)
2- Finite Volume Method (FVM)
3- Finite Difference Method (FDM)
The main differences between them as following:

2.1. FEM

FEM is a computational method to divide the geometry (CAD model) into small finite-sized elements in simplest possible shape, the name of this network of elements is Mesh and we call the contacts points between elements Nodes. In this method dependant values are stored at the elements and nodes; [See 3].

2.2. FVM

FVM subdivides the geometry to very small finite-sized volumes which we call them cells. Dependant values are stored in the center of the finite volume, and this method based on that many physical laws are conservation laws (Mass conservation, Momentum, Energy…), so from this idea we must define formulation which consist flux equations of conservation in an average sense over the cells; [See 4].

2.3. FDM

Most direct approach to discretize partial differential equations is FDM, so we can take a point in space where the continuum is representing and replace it with discrete equation. This method is suitable for simple and regular mesh and geometry; [see 5].

2.4. FEM basics

This method is the most difficult to apply however it has the best accuracy especially in complex CAD models, because of that we will concentrate on this method and its basics and applications.

For applying FEM for engineering model we need first to specific the physical governing equations which control the model, then to obtain mathematical model we need to change PDE to integral equations using approximate functions (linear, nonlinear, quadratic polynomial….); [See 3]. Using this functions and creating the Mesh which contains elements and nodes with specific number of Degree of Freedom (DOF^{s}) lead us to end up with a large sparse matrix equations system that can be solved by power calculation of computers.

We can right programming code for defining elements type, materials, boundary conditions …etc or using some international commercial codes and software like (ANSYS, ABAQUS, NASTRAN…).

3. FEM Industrial Applications

Nowadays engineers can use FEM to simulate almost all engineering problems and analysis (structural analysis, heat transfer, flux and fluid dynamics, electromagnetic….etc).
In machine manufacturing field we can mainly distinguish three basics applications:

1- Design optimization
2- Test quality and reliability for mechanical parts.
3- Improve manufacturing Process for mechanical assembly. [See 6, P 13].

We will discuss a practical example for each application which we may need it in manufacturing.

3.1. Design Optimization

Design optimization is to choose the best possible design using the available tools based on three principles which are: objective function, constraints and design variables.
In Figure 1 illustrates simple optimization procedure; [See 7].

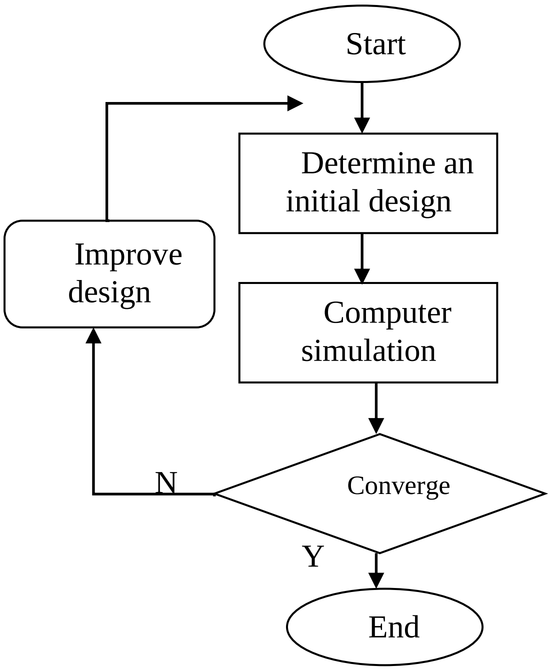


Figure1: *Simple optimization procedure using simulation engineering*

Simulation engineering allows designers to save time and money by replacing physical models with virtual ones and analyze these models under all kind of environments and boundary conditions so they can specify the efficient model to build or produce; [See 7].

We use FEM and CFD to design the Bulbous Bow section in marine vessels. Bulbous bow is an integrated part of the hull of ship; we add this part to the vessel's body to achieve two goals: first to change the flow distribution around the bow which means reducing the drag forces against ship movement in water and secondly to improve the hydrodynamic behavior of the ship; [See. 8]. Using bulbous bow has started since 1940s just for test without theoretical background, and till now design and enroll this part to ship's hull is still based on practical view and experience for the designer not on specific laws. In this article we create three types of bows for marine vessels and analyze them under similar conditions to know exactly which one is the best design to build later. Using simulation engineering helps us to optimize the design in effective time and cost because build this physical model is so much expensive; [See 10]. Fig.2, 3 and 4 show the models of bows which we simulate using FEM and CFD methods.

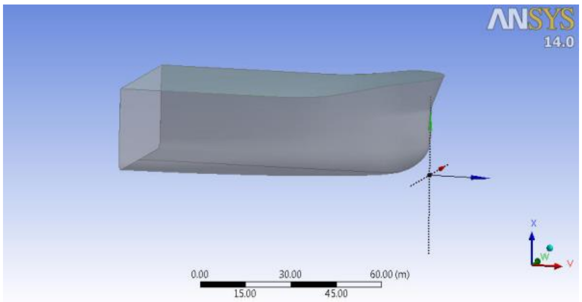


Figure 2: *Flat bow for ship using ANSYS-DM software designed by author 2016*

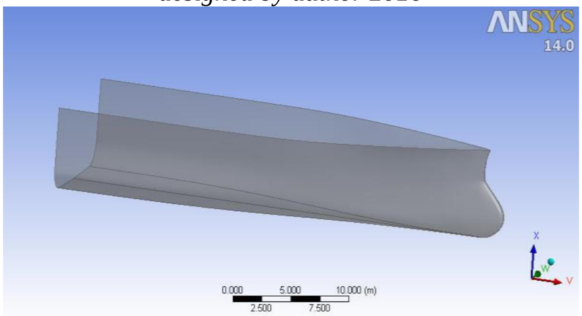


Figure 3: *Delta bow model for ship using ANSYS-DM software designed by author 2016*

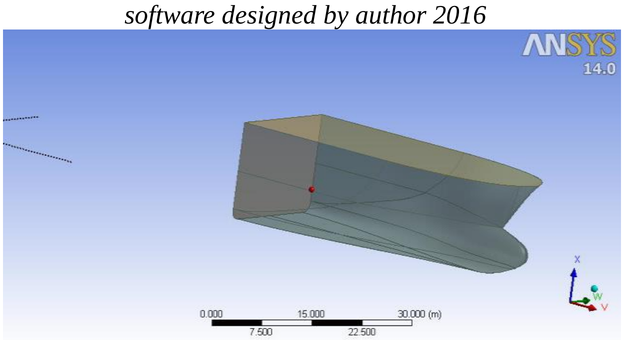


Figure 4: *Oval bow model for ship using ANSYS-DM software designed by author 2016*

Creating suitable mesh for the models and adapt the governing equations which control the water flow around the bodies (Mass equation, Momentum equation, Turbulent equation and free surface equation) are the basic steps to prepare the analysis and get acceptable accuracy; [See 8]. For accurate comparing between models we must to provide the same boundary conditions, flow properties and engineering similarity; [See 9].

To establish the optimization between models we will compare three basic results which represent the effects of bulbous bow on ship's behavior, which are:

1- Hydrodynamic pressure distribution on hull.
2- Drag force applied from water on hull.
3- Water wave created by hull during the move.

In Fig. 5 we can notice the hydrodynamic distribution on the hull with three different models of bows which we simulated. The contour surfaces show the pressure distribution from minimum values (blue color) to maximum values (red color) in Pascal.

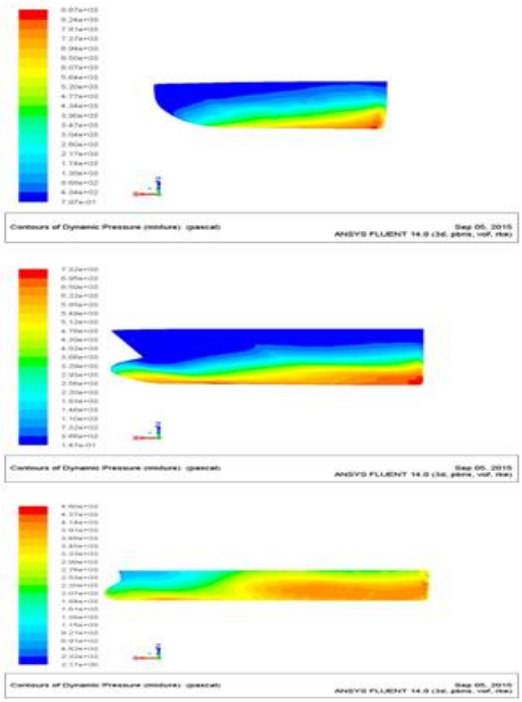


Figure 5: *Hydrodynamic pressure distribution on hull for Flat-Oval and Delta bows using ANSYS-FLUENT software, designed by author 2016*

Because of shape of bow, we have a different distribution for pressure and water flow. We can estimate drag resistance force which applied by water flow on ship's body using ANSYS-FLUENT calculations, estimated this force based on integral the forces which affect on hull surfaces in specific direction (against movement direction of ship). In spite of engineering similarity between models (materials, flow speed, volume, surface areas, smoothing…etc) we got the following results:

Table 1: Drag force entered on Ship's hull

Type of bow	Drag force (N)
Flat bow	682358
Oval bow	482010
Delta bow	140110

We can notice a big effect of bulbous bow on the drag force so on friction resistance which face the ship during the navigation, also the Delta-bow reduced the drag forces about six time smaller than Flat-bow. These results are compatible with results in ref.8 where the researchers used the experimental models to study bulbous bow effects; [see 8 & 10].

Moreover we estimate the wave which the ship will create during the movement in specific speed and by writing programming code on C language, in other meaning the free surface reaction against the bow. This technique called Volume of Fluids (VOF) which calculates the free surface particles in a period of time; [See 10].

Record the movement for a point on water surface gave us an assumption how the ship will create waves, so how we can issue wave making resistance for marine vessel using FEM.

Fig. 6 displays the wave creating by bulbous bow of the ship. From last diagram we notice that the Delta-bow creates a wave with maximum high 0.54 m for ship's speed 3m/s, however the Oval-bow makes wave with 0.76 m in same and 0.72 m height for Flat-bow model at same speed. From engineering view the wave creating by ships means wasting energy by ship so efficient design for bow is the design which creates less waves and less drag force on hull surfaces; [See 10].

Based on numerical results, we conclude that Delta-bow model achieves the objective functions (reducing drag resistance and wave making resistance) and respects the constraints and variables in design, so the bow-design is optimized using numerical calculation.

The second level of design optimization related to Delta-bow dimensions which are also related to ship's type, capacity, cost, and dimensions…etc. This level is not in our interests in this paper.

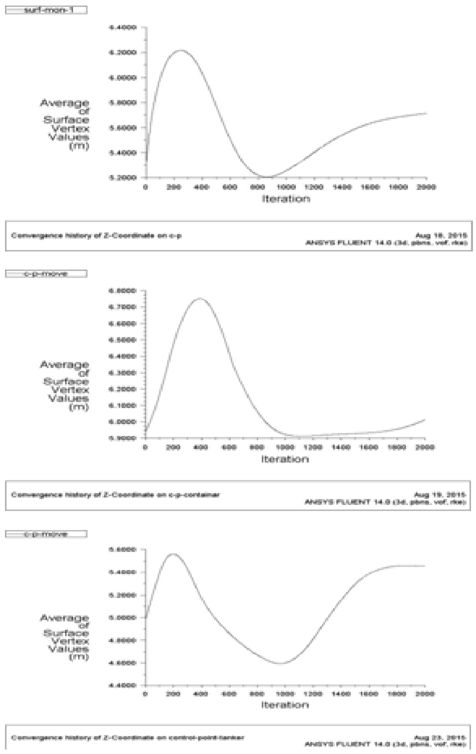


Figure 6: *Free surface wave created by Flat-bow, Oval-bow and Delta-bow respectively using ANSYS-FLUENT software*

3.2. Test Quality and Reliability

After design optimization using FEM, the designer can test the model in virtual environment which is very similar to real one and apply various scenarios of work on mechanical part or model, so he is able to issue the behavior of this part under different possible conditions like overloading, overheating, failure analysis, fracture, fatigue etc; [See 11].

Assurance the object's reliability and quality, under the critical work conditions which may occur during life-cycle time, is one of the most important duties for designer.

For discussing this application we design a mechanical Lever part and test it under various conditions. Lever is a mechanical part which affects the effort or force, needed to apply a certain amount of work and it's used by engineers in many manufacturing machines; [See 9 & 10]. Fig. 7 shows the CAD model for this lever and the elements network (Mesh) created for numerical analysis.

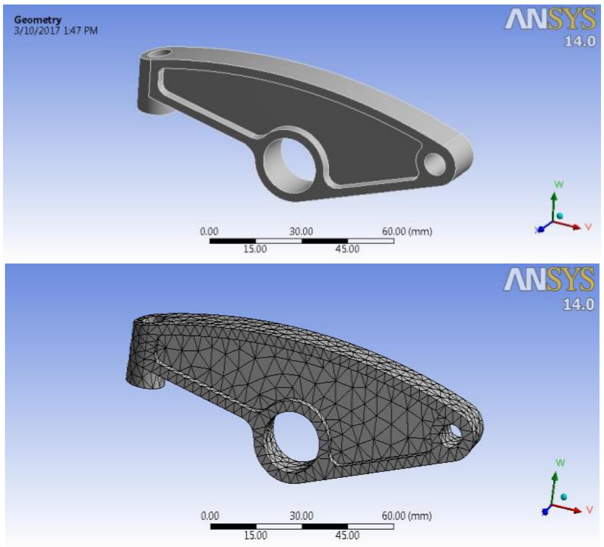


Figure 7: *CAD model for Lever part and the mesh of elements using ANSYS-DM software-2017*

For testing this lever we apply different values of bearing load and external moment on its model and estimate the maximum values of shear stress, equivalent stress and total deformation, so we can simulate the behavior of the lever and assure that has an acceptable quality and properties.

Table 2: Properties of materials used for simulation the Lever model

Material type	Density	Young module	yield strength
Steel S270	7850 Kg/m^3	2 E+5 MPa	270 MPa
Aluminum 2024	2770 Kg/m^3	71e+3 MPa	280 MPa

Properties of materials used for simulation the lever part is showed in Table 2. Choosing the right material related to lever's function and estimated life-cycle. We can experience much kind of materials (metals, composites, alloys…), in order to do the analysis, we perform the scenarios which applied on the level model and parameters specified to analyze (Table 3). We add an external moment and external pressure which may occur in the real life as an accident or because of overloading case.

Table 3: Lever simulation case studies

Case study	Bearing load	Bending moment	Pressure
1	2000 N	0	0
2	4000 N	10 N.m	0
3	6000 N	0	2 MPa
4	6000 N	20 N.m	4 MPa

The constraints and boundary conditions are displayed in fig. 8. These conditions contain the constraints on model, external forces and moments and other parameters which we selected to simulate; [See 12]

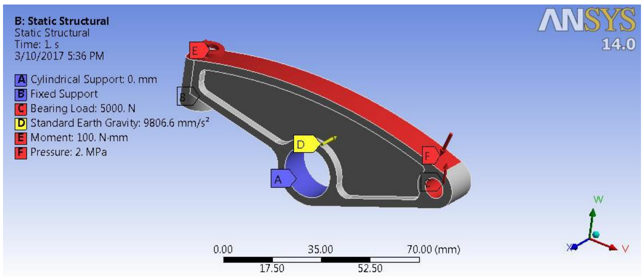


Figure 8: *Boundary conditions and forces applied on lever model using ANSYS-Structural Analysis*

Table 4: Output parameters results for test lever reaction for the case studies- Steel 270

Case study/S 270	Total Deformation mm	Max Equiv. stress MPa	Max shear stress MPa	Max Equiv. Strain mm^{-1}
1	0.0419	81.693	43.073	0.0004
2	0.0894	253.39	146.29	0.0013
3	0.1314	249.46	131.53	0.0012
4	0.1203	506.65	292.5	0.0027

Table 5: Output parameters results for test lever reaction for the case studies- material AL 2024

Case study/ S.S 316	Total Deformation mm	Max Equiv. stress MPa	Max shear stress MPa	Max Equiv. Strain mm^{-1}
1	0.1185	77.804	41.749	0.0011
2	0.2529	251.67	145.29	0.0037
3	0.3715	237.67	128.1	0.0034
4	0.3398	503.21	290.51	0.0057

Evaluation and validation the results need a high experience in numerical analysis and functions of the lever, so the engineer can clear up the weaknesses for mechanical part (stress concentration, cracks, failure, deformation…) and decide if the part has enough quality and reliability to use or needs more optimization and modifications. Fig. 9 illustrates stress distribution on lever's model in case No.2 and steel material, also fig. 10 displays deformation and max strain distribution.

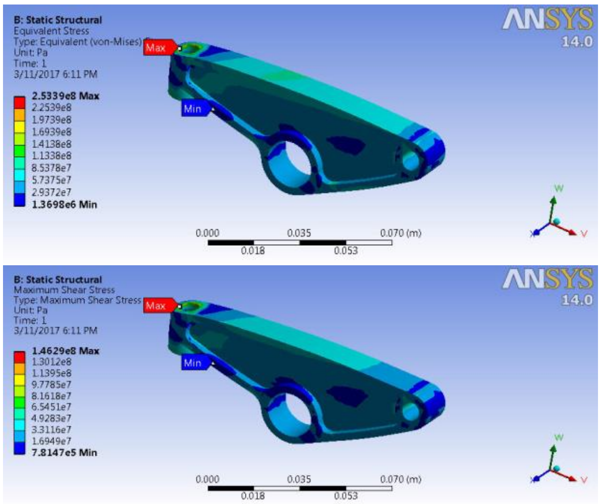
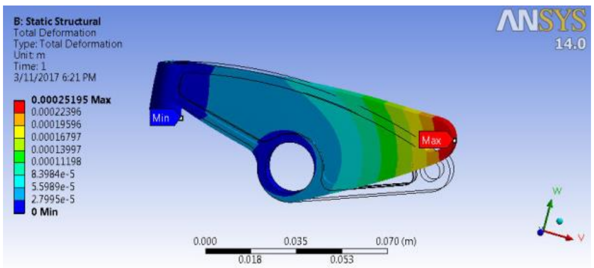


Figure 9: *Equivalent stress and Shear stress distribution (respectively) on Lever in case study No.2 with steel- using ANSYS-Structural software*



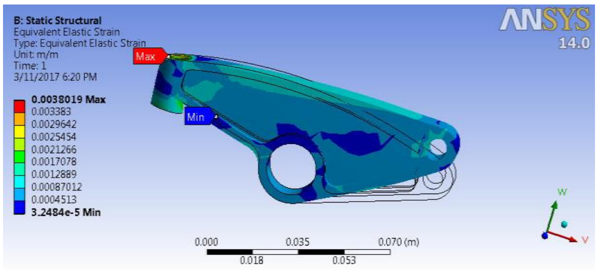


Figure 10: *Equiv. strain and total deformation distribution (respectively) on lever in case No.2 with steel mat.-using ANSYS-Structural software*

There are many types of results we can report, so we are able to study every parameter separately or the effects between designed parameters. Test quality and reliability for the mechanical part can be considered as a second level also from design optimization [see 7], which needs experimental experience from engineer to get effective results.

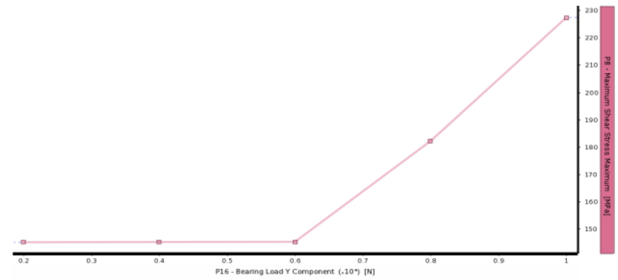


Figure 11: *Diagram max shear stress related to bearing load applying on Lever in various cases with AL material- using ANSYS-Post*

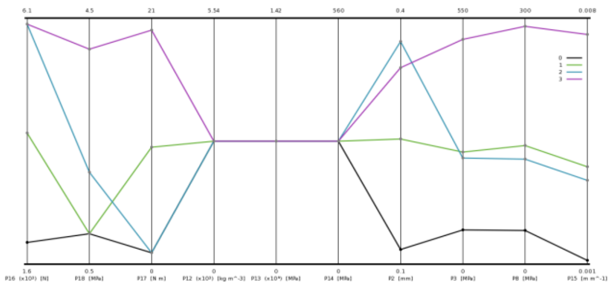


Figure 12: *Integrated diagram consists of all variable parameters in input and the output results for all cases with AL material- using ANSYS-Post*

We can isolate one parameter and analyze the changes in its value in all cases like what we showed in Fig. 11, where the relation between bearing load and maximum shear stress appears. Based on material properties the designer has to define the critical points which maybe will carry critical values for shear, normal or equivalent stress, these points are nominated to have failures or cracks during the life-cycle of mechanical part. Fig. 12 is combination of all results in all cases together, so analyzing relations between variables and output results lead the designer to the part's reliability in all possible cases.

3.3. Improve manufacturing Process for mechanical assembly

Mechanical machines have many mechanical and electrical parts, which they assembled together to achieve a special function. Every part has own material and properties due to this fact, the designer has to predict how the assembly part will perform its work in best possible way. Simulation engineering gives the designer a great chance to combine many mechanical parts in one mechanism and experience this assembly in similar environment to reality; [see 13].

Next level after design optimization and test the quality of this design is to put the mechanical part in its functional assembly and analysis total behavior during manufacturing process. In order to discuss this level we design pump model which contains Impeller, house case and pulley system. Three main parts with three types of materials, we need to assembly them as showed in fig. 13 and estimate the deflection of impeller and stresses in plastic pump housing under specific bearing loads, in other meaning assembly reliability.

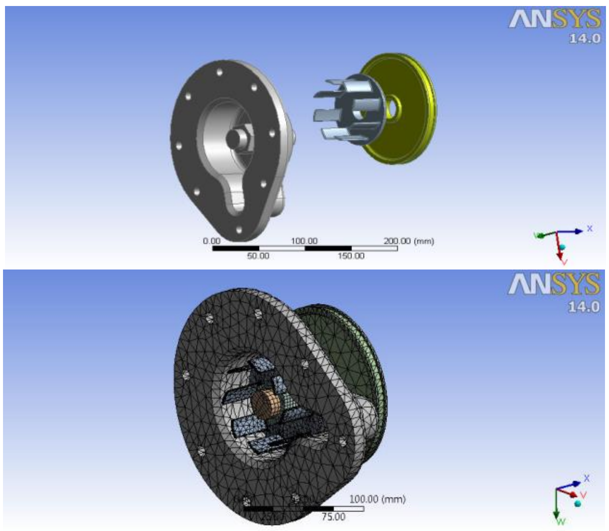


Figure 13: *Pump's part for assembly and mesh using ANSYS-Mechanical library-2017*

Implementation accurate boundary conditions and define type of contacts between mechanical parts are very important steps in order to obtain useful simulation, so by using ANSYS-Mechanical software we can define all contacts regions[See 14] similar as much as possible to real position (Fig. 14) and materials properties as shown in table 6.

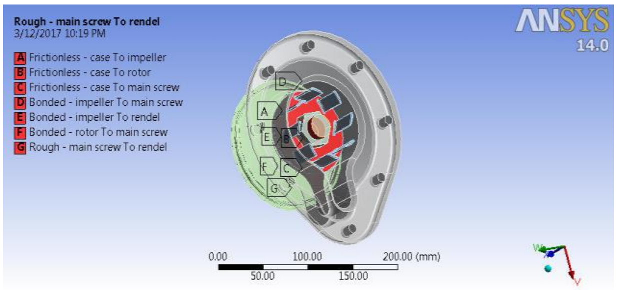


Figure 14: *Pump assembly contacts definition-using ANSYS-Mechanical*

Table 6: *Materials properties used in lever analysis*

Material type	Density	Young module	yield strength
Steel S270 (Pulley)	7850 Kg/m^3	2 E+5 MPa	270 MPa
Polyethylene (Housing)	950 Kg/m^3	1.1E+3 MPa	25 MPa
Bronze (Impeller)	7500 Kg/m^3	1.1E+5 MPa	280 MPa

By applying different bearing loads on pulley part, stress and deformation distribution can be estimated in all the assembly and separately in each part. Fig. 15 illustrates equivalent stress created on impeller under load 500N, and fig. 16 shows deformation in impeller due to this load.

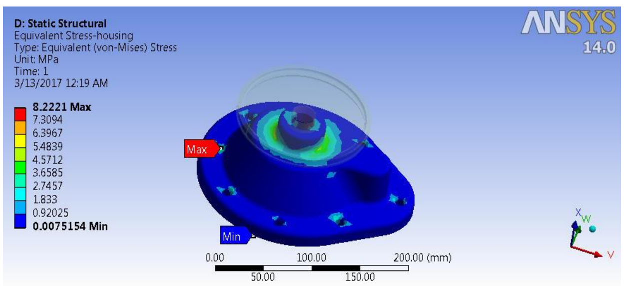


Figure 15: *Equivalent stress distribution on pump housing under 1000 N load- using ANSYS-Mechanical software*

Like last step we can create design parameters then change them to various values so the designer can define the critical load which under it, the housing of the pump will start to be defective.

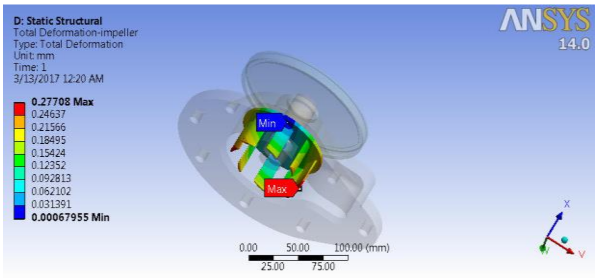


Figure 16: *Total deformation for impeller under 1000 N load- using ANSYS-Mechanical software*

Display the results on specific scopes helps designer to analyze each part's behavior under analysis conditions and in same time study the assembly group response to these conditions; [See 15]. Fig. 17 is shear stress distribution on impeller and axis which applied by pulley.

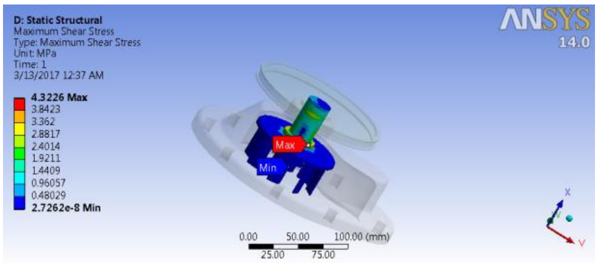


Figure 17: *Shear stress distribution under 1000 N loads- using ANSYS-Mechanical software*

4. Conclusion

- The article demonstrates the basic application of numerical analysis in manufacturing technology and Simulation engineering.
- The present study shows using numerical analysis in all practical steps of designing from design optimization to assembly testing before last order to build the physical model.
- Numerical analysis produces approximate results and it can't replace the experimental models tests, however it is a major step to achieve the best possible design in cost effective time and money.
- Validation a numerical result needs a special high experience in numerical codes and in manufacturing process to assure that these results can be used practically in industrial process, furthermore numerical results validation and evaluation deserve a special study to cover all errors criteria which help the designer to obtain the best accurate result.

References

[1] Atkinson, A.K, *Numerical Analysis*, University of Iowa press, 2011.
[2] Levy, D., *Introduction of Numerical Analysis*, CSCAMM center, University of Maryland press, 2011.
[3] FEM: https://www.quora.com/How-does-finite-element-analysis-help-manufacturers-to-improve-product-performance
[4] Pinho, F.T, *The finite Volume Method Applied to Computational Rheology*, **1** (2001) 63-100, Portugal, 2001.
[5] George, E.F., Wolfgang, R.W, *Finite Difference Methods for Partial Differential Equations*, Literary Licensing press, 2013.
[6] Quiza, R., *Hybrid Modeling and Optimization of Manufacturing*, Springer press, 10.1007/978-3-642-28085-6_2, 2012.
[7] James, T.A., *Introduction to Design Optimization*, University of Michigan, 2005.
[8] Shahid, M., Debo, H., *Computational Fluid Dynamics Based Bulbous Bow Optimization*, J. Marine Sci. Appl. (2012) 11: 286-294, China. [9] Kunes, J., *Similarity and Modeling in Science and Engineering*, Springer press, 2012.
[10] Nordas, E.D., *Optimization of Bow Shape for Large Slow Ships*, Norwegian University of Science and Technology, 2012.
[11] Boca, D.G., Gokkaya, H., *A Model to Improve the Quality Products*, ISSN 1584-0409, Turkey.
[12] Lever properties and loads can be seen in following link: http://www2.phy.ilstu.edu/pte/489.01content/simple_machines/simple_machines.html
[13] McLean, C., Kibira, D., *Virtual reality simulation of a mechanical assembly production line*, IEEE press, 7644346, 2003.
[14] Contacts types in ANSYS-software are available in this link: http://www.ansys-blog.com/what-are-the-differences-between-the-contact-formulations/
[15] George, D., *An Approach for Simulation design of Mechanical Assembled units*, ISBN 978-1-84626-xxx-x, Romania, 2011.