Modeling and numerical investigation of thermo mechanical behavior for impeller design of centrifugal turbine

Sadie BELBACHIR1, Abedelkader LOUSDAD2, Ahmed ALLALI3, Ahmed ALAMI4, Belhadj BOUCHAM5

1LMPM, Mechanical engineering department, Faculty of technology, University of Sidi Bel Abbes, Algeria.
2Laboratory Mechanics of Structures and Solids (LMSS), Mechanical engineering department, Faculty of Technology, University of Sidi Bel Abbes, Algeria.
3Laboratory of applied mechanics, faculty of Mechanical Engineering, University of sciences and Technology of Oran (USTO), Algeria.
4Faculty of Technology -Department of Mechanical Engineering, University of Sidi Bel Abbes Algeria.
5Laboratory of Mechanics of Structures and Solids (LMSS), Faculty of Technology -Department of Mechanical Engineering, University of Sidi Bel Abbes, Algeria.

ARTICLE INFO
Received September 20th, 2020
Received in revised form December 30th, 2020
Accepted December 30th, 2020

ABSTRACT
The main goal of the present investigation is a contribution to the design of the rotor which is among the main turbine elements that require more attention in preliminary conceptual design for safety, reliability and high performance. The modeling and simulation are carried out using a fast tool computational ABAQUS software code. It allows a verification of the thermo mechanical characteristics at the different sections of the blades during the design of a centrifugal turbine. It will then be possible to use the design of the blades, and improvement by definition and by changing their shape. The results obtained put into evidence and highlights the effects of the coupling of thermal and mechanical loading conditions which may be considered in rotor design and improving turbine performance.

© 2020 EST-Khenifra, University of Sultan Moulay Slimane. All rights reserved.

1. Introduction:
Turbo-machines operate today under mechanical conditions and different temperatures. These temperature values often exceed the thermal limit allowed by the materials [1]. And therefore, an intense alteration can lead to the stoppage of work of the machine. These adverse thermal effects with a mechanical loading can continuously push the manufacturers of turbo machines to improve the techniques of their design. This task is conditioned by a good and deep understanding of the phenomenon of heat transfer coupled with a mechanical load in this type of machine [2-3]. The focus of this study is three-dimensional numerical simulation of the thermo mechanical behavior of a rotor of a centrifugal turbine made of steel. This is done using a finite element approach to evaluate the stresses in a 3D model of the rotor using the ABAQUS calculation code. It tries to make a design investigation while making changes in her geometry and the number of blades. The problem of mechanical and temperature resistance arises in this case. In fact, for the builders, this represents one of the major concerns occupying strongly their research spirit in order to find acceptable and satisfactory solutions. From the literature, it has been found that most research aims to calculate the flow coupled with heat transfer to predict the...
performance of turbo machines [4-6]. They are essentially based on the finite volume method using the Ansys software. Significantly, for example, in 2006, Moreno Salas [7] modeled the thermal exchanges in the volute of a radial turbine of a small gas turbine (75 kW) in 3D with the commercial code CFX. A 3D simulation of the flow and heat exchange in an internal cooling channel of a gas turbine blade has been the subject of research by A. Harizi et al. [8] in 2008. Y. Ribaud [9] in 2003 modeled thermal transfers in an ultra micro-gas turbine. Thermal analysis by calculation of flow coupled with heat transfer has been reported by Tom Heuer et al. [10] in 2005 For this purpose, the originality of this work lies in the fact that this study was carried out using the finite element method using the Abaqus software thus allowing the study of the thermo mechanical response of a rotor of a radial turbine. We must also note the complexity of the geometry studied which required us to use other codes in order to realize it as a 3D configuration and to inject it into the Abaqus software. For the interpretation and the validation of our results one was inspired by the works which touched the sizing of a rotor. Among these works are those of Benson (1977) [11] describing a method for determining the overall dimensions of a rotor of a radial turbine under operating conditions. An optimization procedure based on one-dimensional analysis of turbine operation was employed. The analysis includes forecasts of different losses in the turbine. Ebaid and Al-Hamdan (2004) [12] present in their paper a nonlinear optimization technique for the design of a radial turbine rotor. A radial turbine has been designed to operate at a high rotational speed. In addition, an average current line approach was employed to determine the optimal axial length of the inter-blade passage. On the other hand, among the design work done by Assuage in 2003 [13], he developed a comprehensive design and optimization approach in the design and performance analysis of centrifugal turbo machines and helical-centrifuge with incompressible fluid. Finally, to contribute to studies related to this field and to the application of the laws that govern the propagation of heat, we chose to numerically analyze the heat transfer properties by conduction under mechanical load in a steel rotor while making vary the following parameters: number of blades, diameter and width of the rotor. This is to deduce the fields of distribution of the stresses in the blades of a rotor in 3D of a radial turbine.

2. Material and methods:

2.1. Geometric configuration, Meshing and Loading:

In this paper the heat transfer by conduction under the action of pressure in a rotor of a centrifugal turbine is studied. The geometrical configuration is illustrated in figure 1a. The meshing of the rotor must be conforming to the geometrical and physical criteria and must take into consideration the overall constraints related to the study such as the direction of flow, the boundary conditions, etc. The meshing of the rotor is a structured tetrahedral mesh type as shown in figure 1b. The loading is modeled as shown in figure 1c. In order to see the effects of heat transfer on the mechanical behavior of the steel rotor.

![a) Physical configuration with and without cover. b) Physical configuration of rotor and meshing. c) Loading illustration of the steel rotor.](image)

Figure 1. Geometric configuration.

2.2. Simulation:

The simulation conditions introduced in the code are summarized in table1.

<table>
<thead>
<tr>
<th>Inlet conditions</th>
<th>Fixed end</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller internal pressure</td>
<td>0.3 MPa</td>
</tr>
<tr>
<td>Input Temperature</td>
<td>T_i = 300 °K</td>
</tr>
<tr>
<td>Output Temperature</td>
<td>T_o = 310°K</td>
</tr>
<tr>
<td>Conductivity</td>
<td>46 W/ m² °C</td>
</tr>
<tr>
<td>Expansion coefficient</td>
<td>0.12x10⁻⁴</td>
</tr>
<tr>
<td>Density</td>
<td>7.8 Kg/ m³</td>
</tr>
<tr>
<td>Specific heat</td>
<td>473 J/ Kg K</td>
</tr>
<tr>
<td>Impeller meshing</td>
<td>Structured</td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>127.66 MPa</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.33</td>
</tr>
</tbody>
</table>

Table 1. Simulation conditions.
3. Results and discussion:
The illustrations of the evolution of heat flux by varying the number of blades of the impeller, the width of the blade at the inlet and the external diameter of the impeller are shown in figure 2 to figure 4 in order to obtain a comparison.

3.1. Heat flux contour:
The modeling of heat transfer through the internal structure of the impeller involves both, the heat transfer equations and the mechanics of the structures [14-16]. The heat flux by conduction under the effect of a high temperature gradient at the outlet of the impeller \( T = 310 \text{ K} \) towards the low temperatures at the inlet \( T = 300 \text{ K} \). Figure 2 shows the heat flux contours for the case of impeller with five and eleven blades respectively. The results of the numerical simulation obtained in this case show the distribution of heat transmission by conduction through the entire wall of the impeller. In all the contours, a maximum of heat flux is observed at the blades toes in the lower of the impeller. The maximum heat flux is observed for the case of 11 blades impeller and is more or less high at the blades than at other parts of the impeller.

It is also observed a decrease in the heat flux due to the increase of the blade width at the inlet as shown in figure 3. An increase in the values of the heat flux is observed with respect to the variation of the blades number. A drop of heat flux from a maximum value of 220.4 W to 58.85W is observed due to the variation of the diameter of the impeller 160mm and 180mm respectively as shown in figure 4.

![Figure 2. Heat flux contour: a - Impeller 5 blades; b - Impeller 11 blades.](image)

![Figure 3. Heat flux contour: a - width \( b_1 = 10 \text{ mm} \); b - width \( b_2 = 15 \text{ mm} \).](image)

![Figure 4. Heat flux contour: a - diameter \( D_1 = 160 \text{ mm} \); b - diameter \( D_2 = 180 \text{ mm} \).](image)

3.2. Variation of the Von Mises' stresses
It is commonly known that the blades of the impeller condition the efficiency of the turbines. Thus, the prediction of their lifespan constitutes a primary objective of research. The blades are subjected to damaging stresses due to many factors such as temperature (often higher than the material melting temperature), the high temperature gradients and higher stresses induced by the complex geometry [17-18]. According to studies conducted on the performance of centrifugal turbines, it was observed an increase in pressure head, power and efficiency by increasing the number of blades.
However, in practice the use of a high number of blades is avoided for several reasons which may result in an excessive blockage of the flow at the rotor outlet, a great surface area causing high friction losses as well as due to the inertia and mass of the rotor which become relatively high. The numerical results of this work show that the maximum Von Mises stresses increase with increasing number of blades as illustrated in figure 5.

**Figure 5.** Plot of the maximum Von Mises’ stresses versus the number of blades.

Therefore, there is a stress concentration at the junction of the root and suction blade surface due the reduction in cross-section at the root while increasing the number of blades and also, at the extremity of the pressure blade surface. The maximum stresses can reach values much higher than the allowable stress of the material of the impeller. These high stress levels reduce considerably the life of the impeller and may lead to fracture resulting in a catastrophic damage of the turbine. Observation made on plot representing the evolution of the maximum Von Mises stresses as a function of the width of the impeller fig 6 shows that these stresses increase with increasing blade width at the inlet beginning at 15mm width. Similarly, the combination of effect of temperature and high stresses can cause a nucleation zone from which the defect propagates until the destruction of the impeller. Thus, the dimensioning of the impeller of a turbine requires considering the limits of the material capacity in terms of (tension, shear, creep, fatigue, ...) [19-20].

**Figure 6.** Plot of the maximum Von Mises’ stresses versus the width of the impeller.

4. **Conclusion:**

This study constitutes an important contribution to finite element rotor modeling for the analysis of the influence of the thermo mechanical effect. The goal was to calculate the stress fields coupled to the thermal transfer. In such problems, the consideration of thermo mechanical coupling is essential to understand the failure mechanisms of the wheel of a turbine. Since in general the wheels work under very severe conditions of use air overheated gas, it is necessary to know the way on which we base ourselves to make a most appropriate choice of the material with which the wheel is designed. From this work, it was deduced that all geometric changes and the number of dawns have an impact on the thermal and mechanical resistance of the wheel. It was possible to locate the stress concentration zones according to the change in the number of blades and also the width of the wheel. Also, there has been a maximum of heat flow at the blade roots at the lower ceiling of the wheel and it is observed in the case where the number of blades of the wheel is high. It can be concluded from this perspective that this type of study can contribute to decision making regarding the life of the turbine wheel.
References:


