

## عنوان مقاله:

Fluid flow analysis of the impeller of a backward centrifugal fan

## محل انتشار:

چهارمین کنفرانس ملی مهندسی مکانیک و هوافضا (سال: 1398)

تعداد صفحات اصل مقاله: 13

## نویسندگان:

Saeid Ghouli - *Fatigue and Fracture Laboratory, School of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran*

Majid Reza Ayatollahi - *Fatigue and Fracture Laboratory, School of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran*

## خلاصه مقاله:

Centrifugal fans, as a category of industrial fans, are frequently applied in different engineering practices. This is mostly owing to their main duty, which is changing the air flow direction by 90 degrees, from inlet to outlet. To reach this aim, fan impeller must revolve around its central axis, absorbing air from the inlet and discharging it towards the outlet. Being the key component in this fluidic system, the impeller consists of a back plate, a number of blades and a shroud, all of which are necessary to be investigated from the computational fluid dynamics (CFD) point of view. To this end, a 3-D model of an impeller of a selected centrifugal fan is considered for the analysis. This model is transferred into ANSYS Workbench to perform a CFD analysis using ANSYS CFX. A well-shaped structured mesh is also produced by using ANSYS TurboGrid. The flow is assumed to be steady state and a double precision calculation method is acquired. Area circumferential average (ACA) pressure is obtained from inlet to outlet versus streamwise location, which complies well with the boundary conditions. The pressure contour is displayed graphically on the 50% span of the blade, demonstrating its pressure and suction sides; this is also presented through a graph which plots the corresponding pressure loads on the blade. Moreover, the velocity streamlines along with velocity values are schematically illustrated in local and absolute coordinates of the impeller. Resultant pressure contours exerted on the impeller components, ensued from the fluid interaction with solid parts, are also obtained in the present study. To verify our results, the residuals of the CFD simulation are shown to gradually converge to appropriate final values, which manifests the good accuracy of the analysis.

## کلمات کلیدی:

Centrifugal fan, impeller, CFD analysis, ANSYS CFX, air pressure

## لینک ثابت مقاله در پایگاه سیویلیکا:

<https://civilica.com/doc/934237>

