عنوان مقاله:

Numerical simulation of fluid flow inside compressor of turbocharger

محل انتشار:

نهمین همایش بین المللی موتور های درونسوز (سال: 1394)

تعداد صفحات اصل مقاله: 6

نویسندگان:

Seyed Shahabeddin Alaviyoun - *PhD Candidate of Mechanical Engineering, K.N. Toosi University of Technology and Design Department of IPCO, Tehran, Iran*

Masoud Ziabasharhagh - *Associate Professor, Faculty of Mechanical Engineering, K.N. Toosi University of Technology, Tehran, Iran*

خلاصه مقاله:

The turbocharger has an irreplaceable role in improving combustion engine power, reducing fuel consumption and decreasing emissions. Through looking up the literature of different turbocharger, the compressor has been the subject of numerous numerical and experimental studies. In this research, internal flow field inside compressor of turbocharger is simulated using CFD software. Results of grid independence study in centrifugal compressor of turbocharger indicated that while the grid numbers of single-passage is above 300000, the increasing of the grid number has little influences on the numerical results of the compressor performance. Numerical simulation is carried out with CFX to investigate steady state flow and k –ε turbulence model is used for the simulation. Boundary condition at the inlet was set as the total pressure and total temperature and at the outlet, static pressure for higher mass flows and mass flow for lower mass flows were set. The boundary conditions were obtained from result of one dimensional simulation of turbocharged engine in GT-POWER software. In order to validate these numerical results, experimental results of turbocharger is used (Compressor characteristics map) and comparison of numerical and experimental results shows that maximum difference of numerical and experimental results at low rotational speed is less than 7% .which is acceptable

کلمات کلیدی:

turbocharger, compressor, internal flow, steady state simulation, characteristics map