ANALYSIS OF THE FLOW IN RADIAL COMPRESSOR OF THE SMALL JET ENGINE MPM-20

Lukáš LAZÍK
Department of Aviation Engineering, Faculty of Aeronautics,Technical university of Košice, Rampová 7, 041 21 Košice, Slovak republic
E-mail: laziklukas@gmail.com

Marián HOCKO
Department of Aviation Engineering, Faculty of Aeronautics,Technical university of Košice, Rampová 7, 041 21 Košice, Slovak republic
E-mail: marian.hocko@tuke.sk

Summary. This thesis focuses on CFD analysis of airflow by centrifugal compressor of a small jet engine MPM-20, located in the laboratory of small jet engines in the faculty of aeronautics. In the introduction is developed thermodynamic calculation of the centrifugal compressor having aims to create conditions for comparison with CFD analysis. Another part of the work is divided chronologically as they respect the gradual development of the project and are the following parts: creating appropriate geometric model, the subsequent creation of computer mesh for geometric models and the creation of a computational model, further analysis using CFD methods, evaluation of the results and conclusions of the examination and comparison of CFD analysis with numerical calculation.

Keywords: CFD; centrifugal compressor; MPM-20; airflow; analysis; jet engine

1. INTRODUCTION

Nowadays, the calculations become increasingly widespread computer simulation programs, which include for example. STAR-CD, ANSYS, FLUENT. We know two basic methods of modeling. The first is the finite element method (FEA or FEM) and finite volume methods (FVM or CFD). The principle of these methods is discrediting solving partial differential equations. To simulate fluid flow is used the second one, and therefore the method of CFD (Computational Fluid Dynamics). Modeling this method plays an important role in facing the challenges of optimization calculations. CFD analysis of centrifugal compressor refers mainly to changes in basic parameters of the air stream flowing centrifugal compressors. Also it points out the characteristics of the flow and the point of origin undesirable turbulent flow.

2. SMALL JET ENGINE MPM-20

A small jet engine MPM-20 turbine trigger an adjustment TS-20 Fig.1, which start single jet air turbosupercharged engine AL-7F-1 of aircraft Su-7 for the needs of the experimental measurements in the school rehearsal small aircraft engines Department of Aerospace Engineering Faculty of Aeronautics Technical University in Košice.[1]

Small jet engine MPM-20 has a tangential input device, a single-stage centrifugal compressor with single impeller, an associated combustion chamber, single-stage axial non-chilled gas turbine reaction type. The adjustment of the turbine of the actuator to the TS-20 experimental small jet engine MPM-20 was removed by the output system consisting of a planetary reducer and a gas extraction system, which was replaced by a classical subsonic discharge nozzle now controllable design of the nozzle.[1]

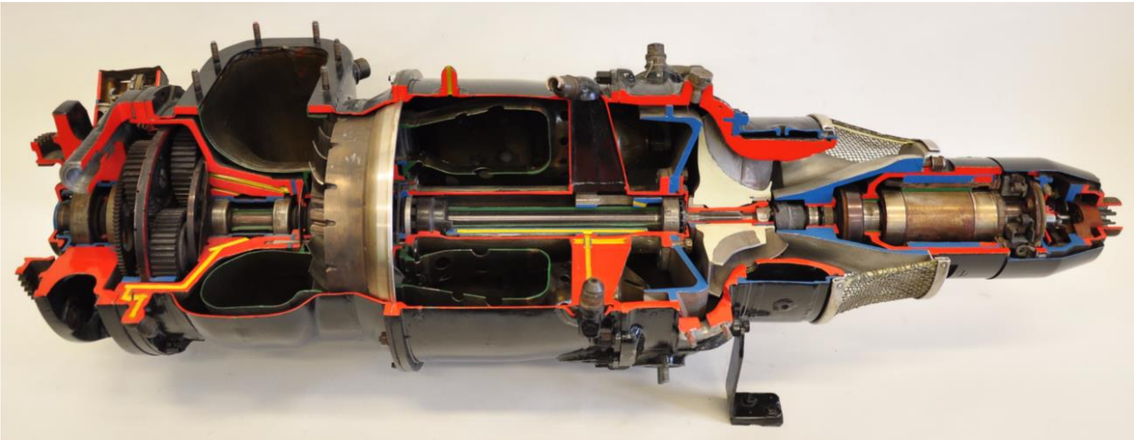

Figure 1 TS-21

3. STRUCTURE OF A CENTRIFUGAL COMPRESSOR OF MPM-20

Single-stage centrifugal compressor provides compression engine inlet air and its delivery to the specified pressure and quantity into the combustion chamber of the engine. Air into the rotating impeller centrifugal compressors Fig.2 supplied from the input system of the engine. Smooth entry of air into the impeller vanes. Vanes of the impeller, the particles of air forced to rotate. The effect of centrifugal forces of rotation and the air increases its absolute velocity, total pressure and total temperature at the outlet of the impeller of which projects into the vane diffuser and the bladeless. The bladeless a paddle diffuser occurs in expanding interscapular channels to reduce absolute air velocity and increase static temperature and static pressure. The centrifugal compressor from the compressed air enters through the annular output mechanisms of the engine combustion chamber.[1]

Vane impeller with 20 blades creates a moving inlet guide means for the air that enters the impeller. The curved blades 20, which rest on the impeller blades. Front overhang of the impeller blade is from 0.04 to 0.1 mm. Transmission of torque from the shaft to the impeller centrifugal compressor ensures grooved connection between the shaft and impeller latch.[1]

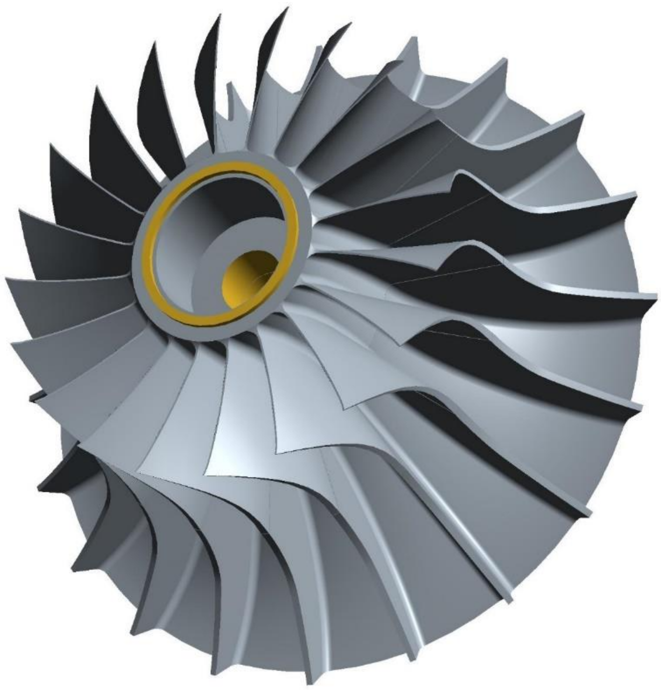

Figure 2 3D model of the rottating impeller in MPM-20

Radial-axial diffuser Fig.3 centrifugal compressors comprises forging alloy. It creates a back wall of a centrifugal compressor motor. It consists of a bladeless radial and axial diffuser vane diffuser.Bladed diffuser disk formed in the crown portion of which is 16 long and 16 short blades. The blades of the diffuser with its peripheral part bear against the inner surface of the outer housing of the input device of the engine. Four diffuser vanes have openings for the location of the fuel nozzles in the combustion chamber of the engine compartment. The exact location of fuel nozzles and fixing them is a paddle diffuser secured by means of a flange on the outer housing of the input Guidance system. The disc has a bladed diffuser centers the flange of the carrier tube, to which is fixed by screws.[1]

Figure 3 3D model of the diffuser in MPM-20

4. CFD ANALYSIS

Today there is CFD modeling becomes an integral part of the solution of the various projects, with specialization in analyzing flow centrifugal compressors are among the most complex and most difficult types of problems. It may be that a number of factors, such as: complex geometry of the compressor and the blade, the flow losses which occur, fluid dynamics, equipment dimensions, which influence the size and complexity of the network. As the challenge of this type is necessary to use a simplified mathematical models that describe us as closely as possible the phenomenon.

All CFD calculations are based on three fundamental conservation laws, which have general application. These include the law of conservation of mass, momentum and energy. The equations describing the flow are formally the same as it always is a conventional diffusion equation. If such an equation is applied to the three components of momentum (as a vector quantity), we get after small adjustments Navier-Stokes equations that describe the law of conservation of momentum.

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}=-\frac{1}{\rho}*\frac{\partial p}{\partial x}+\upsilon*\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+f_x$$

$$\frac{\partial}{\partial t}+\frac{\partial(vu)}{\partial x}+\frac{\partial(vv)}{\partial y}+\frac{\partial(vw)}{\partial z}=-\frac{1}{\rho}*\frac{\partial p}{\partial y}+\upsilon*\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+f_y \qquad (5.1)$$

$$\frac{\partial u}{\partial t}+\frac{\partial(wu)}{\partial x}+\frac{\partial(wv)}{\partial y}+\frac{\partial(ww)}{\partial z}=-\frac{1}{\rho}*\frac{\partial p}{\partial z}+\upsilon*\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+f_z$$

CFD mathematical modeling process begins with the formation of the 3D model Fig.4, which is usually applied in calculating the adjustment for implementation. If the work was created in the program model Creo 2.0. Model geometry based on the technical documentation turbine starter TS-20 centrifugal compressor and a real small turbojet engine MPM-20. When creating a model of the compressor was a major asset availability drawings together with real impeller, allowing to create a model of high dimensional and geometric accuracy. Using drawings and Interpolation trace was created by the base body of the compressor, and the exact shape of one blade.

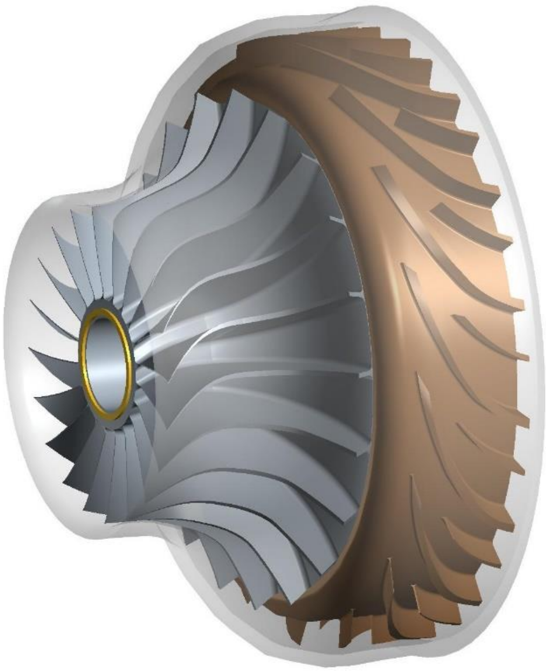

Figure 4 3D model of the centrifugal compresor of MPM-20

Consequently, it was necessary to create computing mesh. Creation of the mesh is made in ANSYS ICEM. To simplify the creation of the network it was decided that for all geometry will be used unstructured hybrid grid Fig.5, 'the NHS. NHS is specific in that the surface consists exclusively of a network of triangular elements (the. Triangles), and to it-binding volume of the tetrahedron formed by the network (so-called. Tetrahedrals). As said, the NHS is significantly less difficult to produce, since it is not necessary to divide the compounded calculation domain into blocks, which requires the formation of a structured network, but in the light of subsequent calculations may sometimes return to more longer calculation times, depending on the complexity and size of the responsibilities. Calculations in the not too quality NHS can also easily lead to unstable (divergent) resulting solutions. Regarding the difficulty of making the NHS, but this is not to say that her work is simple and does not add any pitfalls, quite the contrary. A problem in the development of networks NH suitable for solving the problems in the programs CFD ANSYS CFX and Fluent as the fact that the formation päťstenov prismatic layer (so-called. Prisms) at the surface, the program ICEM through the setting mode to the CFD, the objective of the points in complicated geometries these prismatic elements associated with the network around using pyramid elements (ie. pyramids). Unfortunately, ANSYS CFX is not able to count with these elements and the need to modify the properties of the network as long as the generator stops networks they comprise. An inability to preferred controls in ICEM is a big disadvantage compared to some competing programs, for example. ANSA, where this option is easily and thus can avoid making these pyramid elements. In this area development program ICEM hopefully a step forward and this possibility is here in the near future also appear.[5]

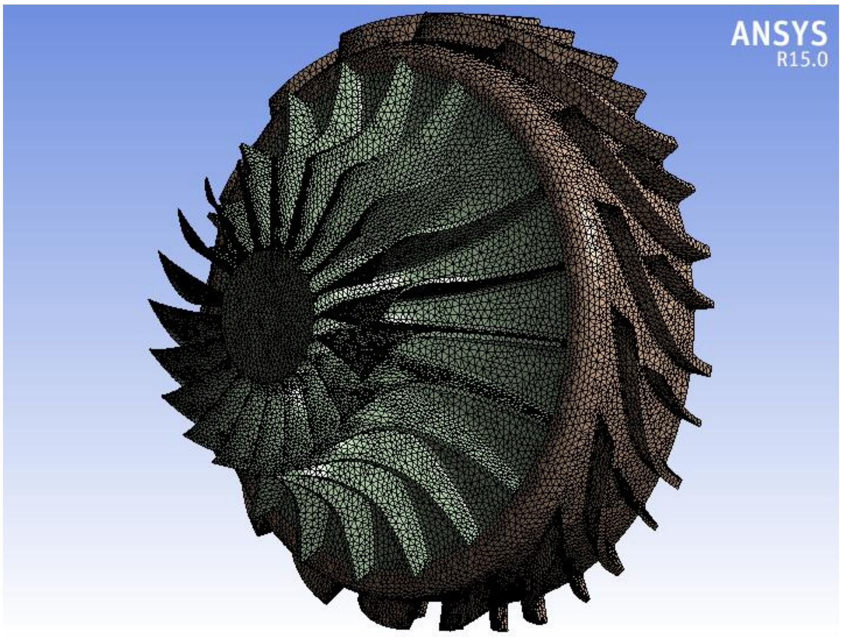

Figure 5 The resulting computational mesh of the centrifugal compressor MPM-20

The solver of all tasks was elected DENSITY-BASED type, which is based on solving the system of equations determining the continuity, momentum, energy (where necessary) and possibly ancillary (additional) equations simultaneously. In this type of solver are discrete, nonlinear governing equations linearized in order to obtain a system of equations for the determination of the dependent variables in each calculation cell. The resulting linearized system of equations is then calculated to obtain the updated flow field solution. Determining the equation for the additional angel fish are handled and then separated from the solution determining the equation of continuity, momentum and possibly of energy. The flow in all cases was considered as a compressible and non-stationary that is time sensitive.

This task required the use of a turbulent model, because it is impossible to assume that the flow of the impeller and diffuser was laminar. This is, however, the turbulent model was chosen, and it is assumed the presence of a turbulent boundary layer around the surface of the impeller and the diffuser. This setting, however, distorts information about the total resistance, but for convergence solutions is essential. Furthermore, since the aim of such work has not been established. total resistance, but the evaluation of flow centrifugal compressors and the use of turbulent model is completely in place and very convenient. As turbulence model was selected type SST (Shear Stress Transport). Model turbulence belongs to the category two equations models. Most of the two equations models can be classified into two groups between the models k-ε and k-ω. These models are used to calculate the viscosity viral with two equations and the equation of the turbulent kinetic energy and turbulent dissipation rate ε, or frequency ω dominant eddies.

5. RESULTS

The results were evaluated in a post-secondary program of CFX - Post to configure the stable operation of the compressor in computing speed 45 567.1 r / min. The average time of each calculation was in use 4 cores of approximately 13 hours.

Visualizing the flow field Fig.6 to obtain qualitative assessment of the results of calculations of flow. The calculation and the visualization focused on various parameters such as temperature, pressure, velocity of the air etc.

The air flow is sharply entrained impeller. It is obvious speed increase air flow in the impeller and the subsequent deceleration speed in space and bladeless diffuser vane. The rate of air flow was evaluated in the calculation sections of a centrifugal compressor air flow speed behaviour are different in at least, to confirm the accuracy of the setting that a particular computing environment. Also, these small deviations haughty importance CFD analyses in the field of aviation.

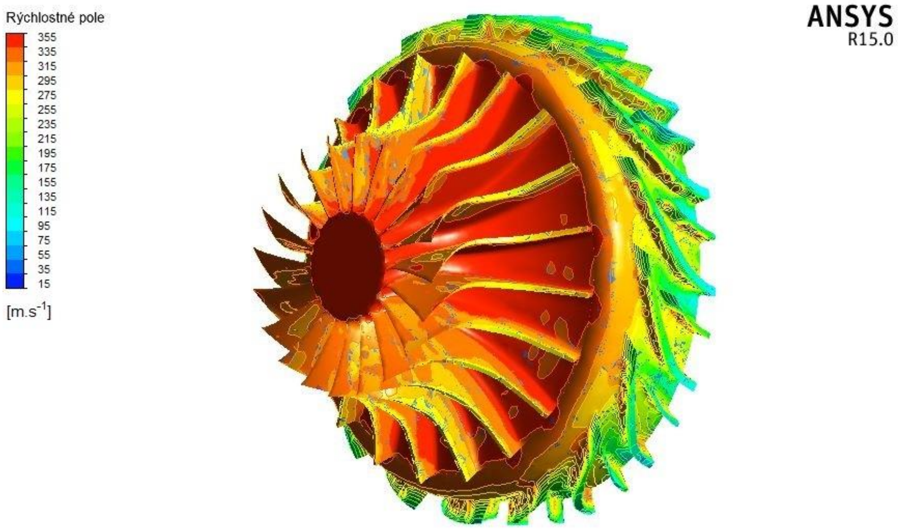

Figure 6 The course of the absolute velocity of the air flow by a compressor MPM-20

In Fig.7 shows a visualization of the actual air flow flowing centrifugal compressor motor. The picture can be seen in the current characteristics of the rotating impeller and the subsequent transition of turbulent flow in almost laminar flow in compressor diffusers. At the outlet of the vane diffuser direction vectors are most stable and facing the same direction. This was confirmed by visualizing a diffuser.

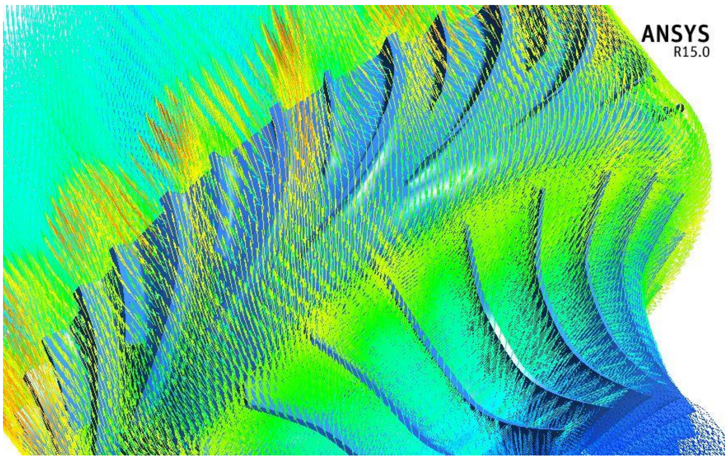

Figure 7 Visualization of flow direction of characteristics of the compressor MPM-20

Modified turbulent viscosity based on the specific procedural variables, so-called. turbulent viscosity model by which Spalart-Allmaras (and other models) reflects the impact of the turbulence. Turbulent viscosity is in the model described by one balance equations and constitutive relations. The aforementioned balance sheet balance equation, the chosen modified turbulent viscosity. Size turbulence can best determine the ratio of molecular and turbulent viscosity. Fig.8 shows the critical points to form turbulent flow, which is undesirable for better efficiency, and is an effort to eliminate the minimum.

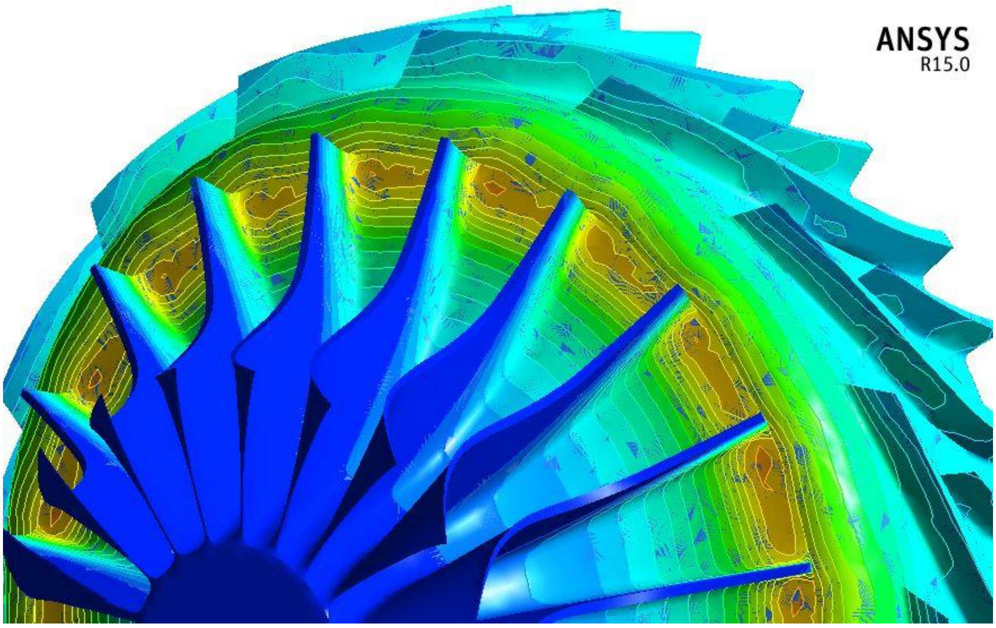

Figure 8 Places of turbulent flow in centrifugal compressor engine MPM-20

6. DEDUCTION

In this work it was analyzed airflow only for a stable mode of operation of the centrifugal compressor. Addressing the unstable operation of the compressor would be beyond the scope of this work. The solution flow in centrifugal compressors and compressor effects of unstable work on the engine and its component parts addresses in detail Ing. Marián Hock PhD. In an article for a scientific conference in Barcelona in 2014 titled "Options experimental verification of unstable compressor works in the laboratory of small jet engines." The article is based on practical experiments analyzing the process of unstable work centrifugal compressor (Surge) as a result of pinching the flow of air in the inlet tract. The aim of the article is to provide information on training, methodology and results of the experiments that were carried out in research of unstable phenomena in centrifugal compressor MPM-20 at the Laboratory of small jet engines Department of Aerospace Engineering Faculty of Aeronautics Technical University in Košice. Lessons learned from the above experiments allows for improved understanding of the process of developing unstable work in centrifugal compressor, thus preventing such phenomena in terms of the real operation of air turbosupercharged engine (ATSE), and combustion turbines (ST). The consequences of unstable work were analyzed for damaging a gas turbine.

Own contribution to the theory and practice I see mainly in reference to the possibility of usability CFD CFD is not just the individual parts for aircraft engines as well as various objects not only in the aviation industry. Further work presents various options and types of calculation as well as their advantages and disadvantages. Finally, this work could serve as a teaching aid in air flow centrifugal compressors as well as in the field of CFD analysis on soil Faculty of Aeronautics.

References

[1] HOCKO, M.: Malý prúdový motor MPM 20. Košice: VLA-MRŠ, 2002

[2] HOCKO, M.: Hodnotenie stavu LTKM na základe zmeny termodynamických parametrov. Košice, 2003 STN ISO 690: 1998, Dokumentácia - Bibliografické odkazy - Obsah, forma a štruktúra.

[3] HOCKO, M:VÝPOČTOVÉ CVIČENIE Z TEÓRIE LETECKÝCH MOTOROV - Približný výpočet odstredivého kompresora. Košice, 2009

[4] HOCKO, M.: Možnosti experimentálneho overenia vzniku nestabilnej práce kompresorov v laboratóriu malých prúdových motorov. Barcelona, 2014

[5] CHUDÝ, P.; FIĽAKOVSKÝ, K; FRIEDL, J.: Aerodynamic Analysis of Turboprop Engine Air Intake. Acta Polytechnica : Journal of Advanced Engineering. 2004, Vol. 44, No. 3,