

PAPER • OPEN ACCESS

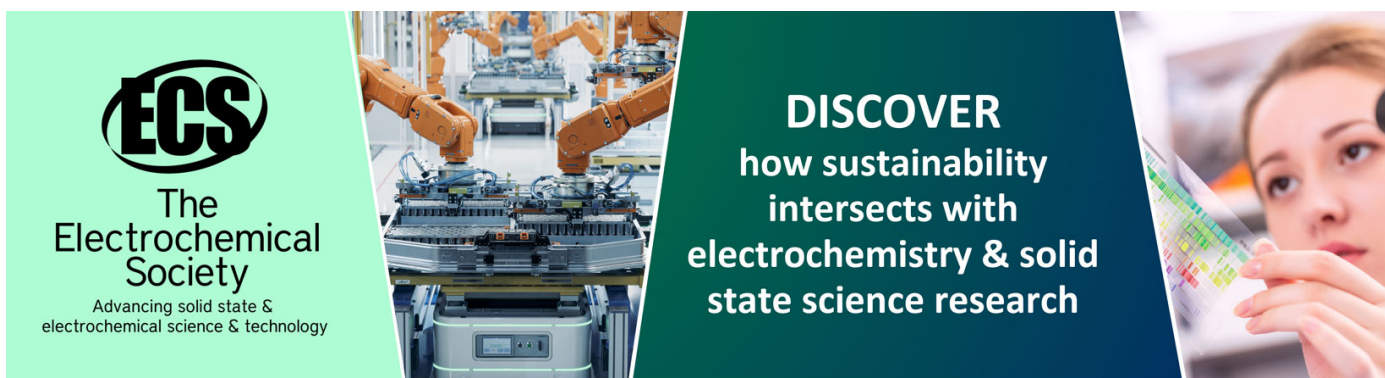
Numerical Investigation of the Influence of the Input Air Irregularity on the Performance of Turbofan Jet Engine

To cite this article: Y Novikova and V Zubanov 2018 *IOP Conf. Ser.: Mater. Sci. Eng.* **302** 012028

View the [article online](#) for updates and enhancements.

You may also like

- [Leading edge topography of blades—a critical review](#)
Robert J K Wood and Ping Lu
- [A novel bootstrap ensemble learning convolutional simple recurrent unit method for remaining useful life interval prediction of turbofan engines](#)
Chengying Zhao, Xianzhen Huang, Huizhen Liu et al.
- [Two Degree-of-freedom Synthesis Control for Turbofan Engine with Slow Actuator Dynamics and Uncertainties](#)
Meiyin Zhu, Xi Wang, Keqiang Miao et al.



ECS
The
Electrochemical
Society
Advancing solid state &
electrochemical science & technology

DISCOVER
how sustainability
intersects with
electrochemistry & solid
state science research

Numerical Investigation of the Influence of the Input Air Irregularity on the Performance of Turbofan Jet Engine

Y Novikova¹ and V Zubanov¹

¹Samara National Research University, 34, Moskovskoe shosse st., Samara, Russia

E-mail: zubanov.vm@ssau.ru

Abstract. The article describes the numerical investigation of the input air irregularity influence of turbofan engine on its characteristics. The investigated fan has a wide-blade, an inlet diameter about 2 meters, a pressure ratio about 1.6 and the bypass ratio about 4.8. The flow irregularity was simulated by the flap input in the fan inlet channel. Input of flap was carried out by an amount of 10 to 22,5% of the input channel diameter with increments of 2,5%. A nonlinear harmonic analysis (NLH-analysis) of NUMECA Fine/Turbo software was used to study the flow irregularity. The behavior of the calculated LPC characteristics repeats the experiment behavior, but there is a quantitative difference: the calculated efficiency and pressure ratio of booster consistent with the experimental data within 3% and 2% respectively, the calculated efficiency and pressure ratio of fan duct – within 4% and 2.5% respectively. An increasing the level of air irregularity in the input stage of the fan reduces the calculated mass flow, maximum pressure ratio and efficiency. With the value of flap input 12.5%, reducing the maximum air flow is 1.44%, lowering the maximum pressure ratio is 2.6%, efficiency decreasing is 3.1%.

1. Introduction

During operation, on the input to turbofan engine receives air having a non-uniform field of velocity and pressure [1, 2]. This is due to many factors: wind, scythe blowing at work on the ground and at the evolution of the aircraft in air, design and technological projections in the pathflow of the input device. The stable operation of low pressure compressor (LPC) is only possible to a certain degree of irregularity of air in the input device [3]. For this reason, it is necessary to predict the air irregularity influence on the engine workflow. One of the numerical methods that can make such a forecast is modern CFD software package such as NUMECA Fine/Turbo. In such software systems, an approach of using a nonlinear harmonic analysis (NLH-analysis) was implemented to calculate the unsteady effects in turbomachines. NLH techniques allow to significantly reducing the calculation time in comparison with traditional methods of non-stationary 3D modeling [4]. This is achieved by presenting the flow in the form of quasi-three-dimensional flows and thus as a set of two-dimensional tasks, and it makes possible to calculate multistage turbomachinery within a reasonable time.

In the present work, the numerical modeling results of the air flow irregularity impact at inlet turbofan gas turbine engine on the characteristics of its fan were shown.

2. CFD-model description

The test fan (Figure 1) has a wide-blade, inlet diameter of about 2 m and a pressure ratio of about 1.6, the bypass ratio of about 4.8.



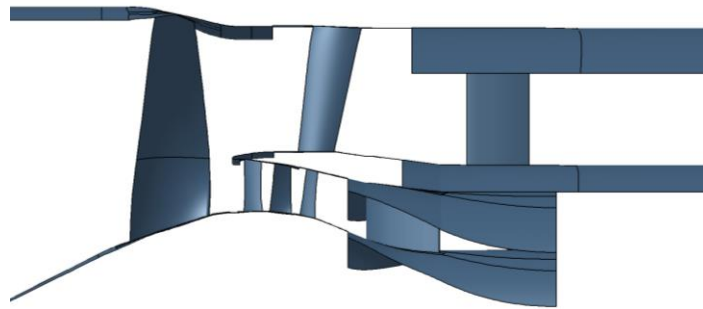


Figure 1. Investigated fan

Numerical study was performed in two stages. In the first stage, the LPC numerical model was created in NUMECA Fine/Turbo CFD software and a comparison of the experimental data and the calculated characteristics in a stationary mode was carried out. In the second stage, the numerical investigation of the air irregularity on fan performance was conducted using NLH method in NUMECA Fine/Turbo [5]. The air irregularity was simulated by input of the flap in the inlet channel of fan. The flap input was carried by an amount of 10 to 22.5% of the diameter of input channel with increments of 2.5%. For each flap position the fan performance was calculated using NLH method in NUMECA Fine/Turbo.

Numerical model of the LPC was created in the NUMECA AutoGrid 5 software. The initial geometric model of the computational domain was constructed based on technical documentation. The geometric model contained the following elements: fan blade, booster and bearing racks (Figure 2). Radial clearance under rotor blades was considered [6].

The LPC numerical model was created in the stationary axisymmetric statement. The blades deformation under the applied forces influence was considered. The shape of the deformed blades was found as a result of the strength calculation, it was loaded into a 3D model of the fan and booster blades.

The created model was divided into finite elements of the structural mesh using program Numeca Autogrid 5 (Figure 3). The total number of elements in a model of about 7 million. The fan domain contains about 2 million elements, each of the other blade domains contains about 0.5 million elements [7].

The regions around the rotor wheels (RW) and the guide vanes (GV) were allocated in the computational domain. The region around GV was calculated in the stationary coordinate system. The region around RW was calculated in the rotating frame, its speed coincides with the rotor speed.

Calculations were performed in a stationary mode using the Mixing Plane interface for data transfer between rotating and stationary domains. The Mixing Plane interface averages the flow parameters in the circumferential direction in the upstream and transmits as a boundary condition in the region located downstream. The turbulence model Spalart-Allmaras was used.

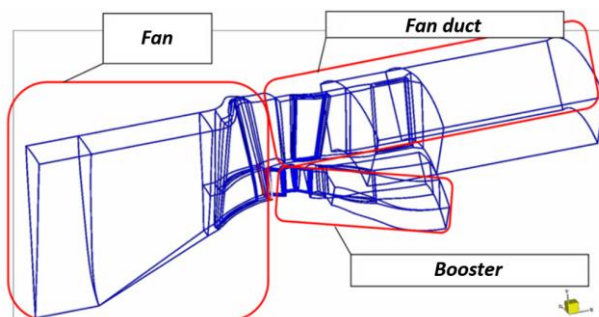


Figure 2. Domains of numerical model of investigated compressor

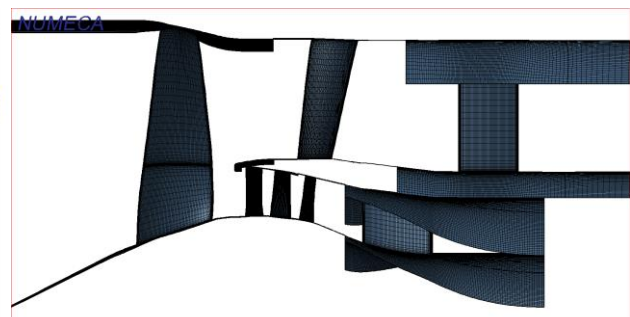


Figure 3. Finite element model of investigated LPC

The values of total pressure $p^*=101,325 \text{ kPa}$ and total temperature $T^*=288,15 \text{ K}$ were set as boundary conditions at the LPC inlet. The flow direction at the inlet region was set as axial. The mass flow values at the outlet of fan duct and booster were set at the outlet boundary conditions. Mass flow ratio determines the value of bypass ratio.

3. Result Analyses

The pressure and efficiency characteristics calculations of booster channel and fan duct was carried out using created numerical model for three modes equal to 0.95, 1, 1.05 of relative rotor speed (Figure 4).

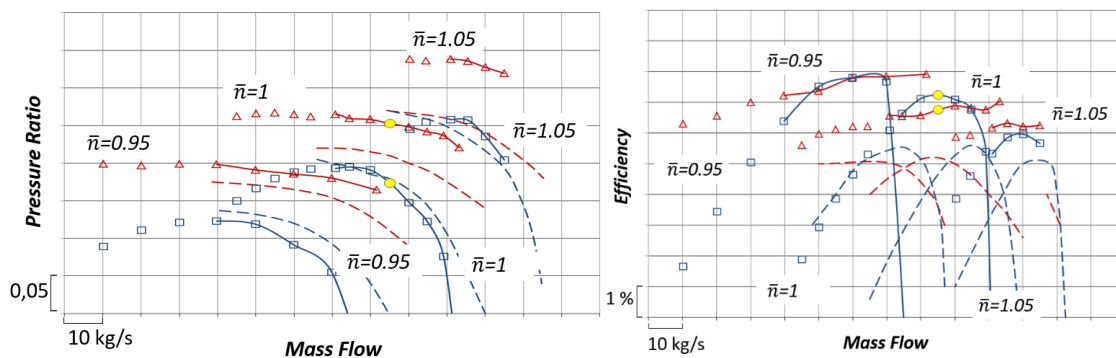


Figure 4. Comparison of experimental and calculated relative characteristics of the LPC of NK-36ST obtained using different computational models

Figure 4 shows the comparison of the calculated LPC characteristics (markers in the form of triangles and squares) and LPC testing experimental data (dashed lines). Markers that are not connected by a line - the calculation at this mode remains stable, but the calculated efficiency and pressure ratio values continuously decrease with the iteration passage, i.e., these modes are between sustainable and unsustainable solution.

From the analysis of dependencies presented in Figure 4 shows that the behavior of the characteristics of the calculated LPC repeats the behavior of the experiment, but there is a quantitative difference. Thus, the calculated values of the efficiency of booster different with experimental by from 1 to 3% and efficiency of fan duct different with experimental from 1.5 to 4%. The calculated values of the degree of pressure ratio in the booster different with the experimental values by up to 2%, the fan duct differ with experimental values by up to 2.5%. Figure 5 shows the calculated field of Mach number in LPC in the meridional plane as averaging in the circumferential direction.

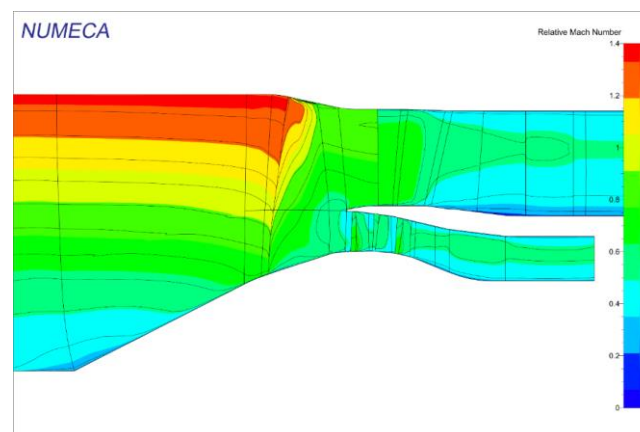


Figure 5. Calculated field of Mach number in LPC

Calculation of the fan characteristics with simulated input air irregularity was performed using the method of nonlinear harmonic analysis. Since the method limitation does not allow the study of the irregularity effect within the double-duct statement, it was decided to carry out research in the formulation of a single-stage fan. Investigated fan stage (Figure 6) consisted of a fan blade and the guide vane of booster which was "elongated" on the distance from the hub of the inner loop to the peripheral contour of the fan duct. It is assumed that this model will allow qualitative assessment of the air irregularity impact on the fan performance.

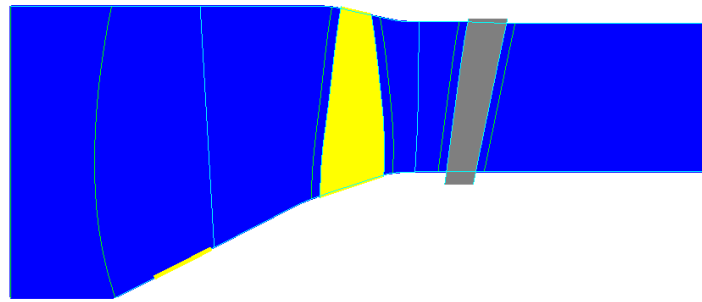


Figure 6. Investigated fan stage

A family of discrete models with the input device and simulator of air irregularity was created to perform the study (Figure 7). The models differ from each other only by the size of the flap extension into the flow of the input device. Input flap carried by an amount of 10 to 22.5% of the diameter of input channel with increments of 2.5%. The input device was modeled entirely (full model), the fan blades and the guide vanes were modeled by sectors corresponding to one inter-blade channel (Figure 8).

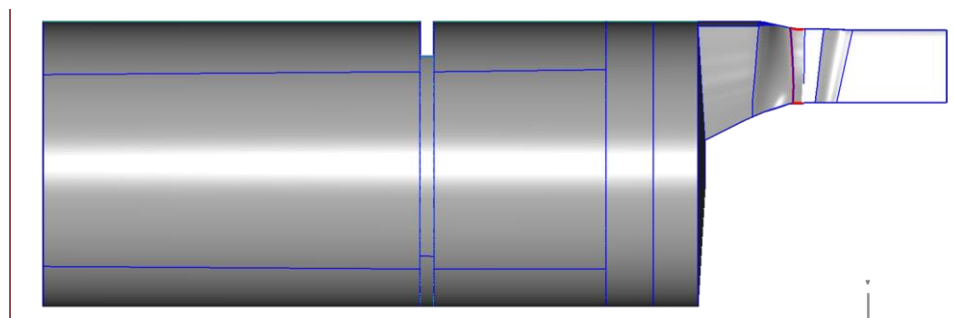


Figure 7. Numerical model of the fan with the simulator of air irregularity

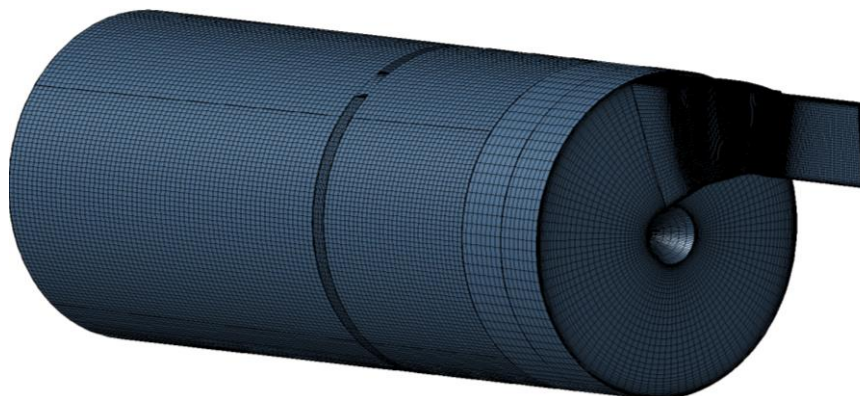


Figure 8. Finite element model of input device and the fan stage

The adiabatic efficiency and pressure ratio characteristics of the fan stage with the air irregularity simulator were calculated with the created numerical model (Figure 9). The calculations were carried out in a non-stationary formulation using the method of nonlinear harmonic analysis. The calculations were performed in the software package NUMECA Fine/Turbo using the following settings of numerical model: *Number of frequencies 3, Max number of perturbations 2*.

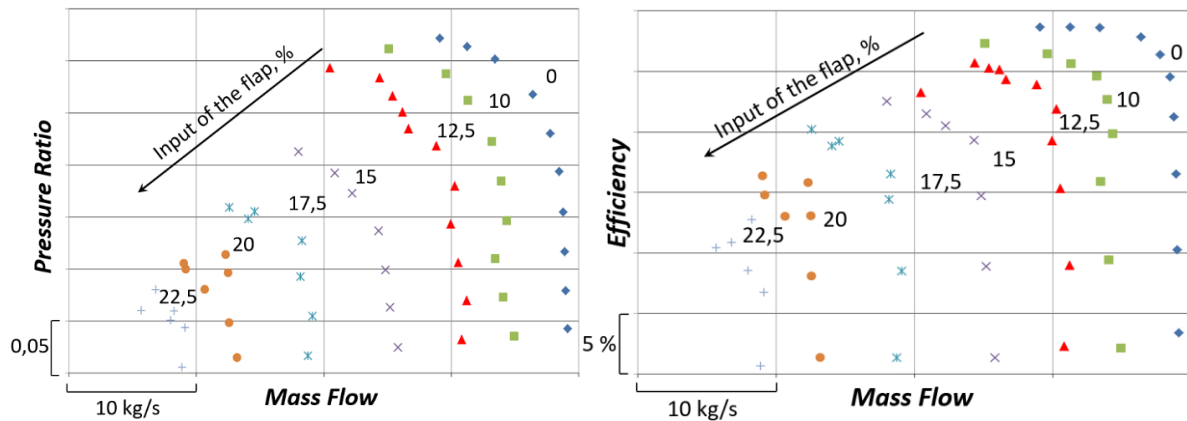


Figure 9. Calculated performance map of fan stage considering the simulator of air irregularity

Analysis of the stage fan characteristics shows that increasing the air irregularity level in the input stage of the fan reduces the calculated value of mass flow, the calculated value of maximum pressure ratio and calculated value of efficiency. With the value of flap input 12.5%, reducing the maximum air flow (absolute, not the given) is 1.44%, lowering the maximum pressure ratio (from the entrance to the inlet device to exit) is 2.6%, decrease in efficiency (including losses in the input device with the flap) is 3.1%.

Figure 10, 11 shows the total pressure field and Mach number field in front of the fan section corresponding to the flap input in 12.5% of diameter of input channel.

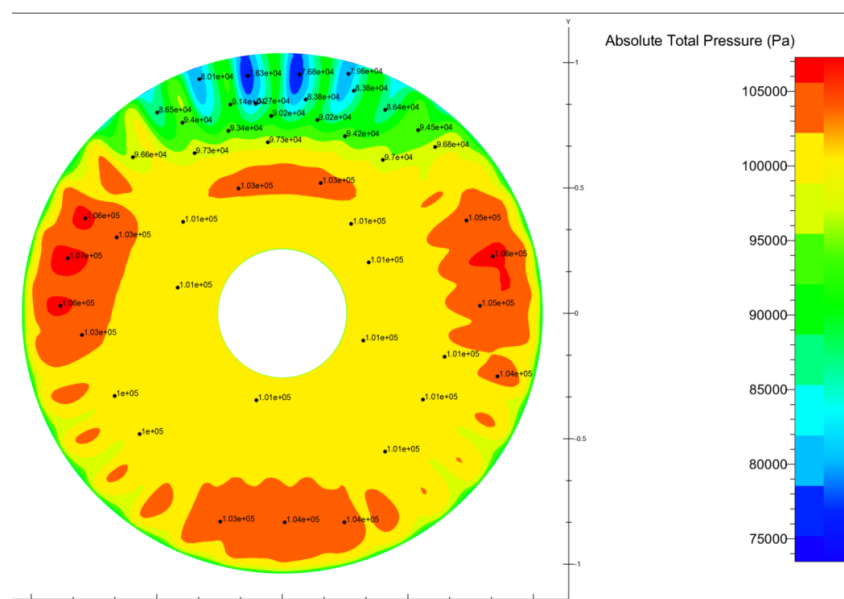


Figure 10. Total pressure field in front of the fan section corresponding to the input of the flap in 12.5%

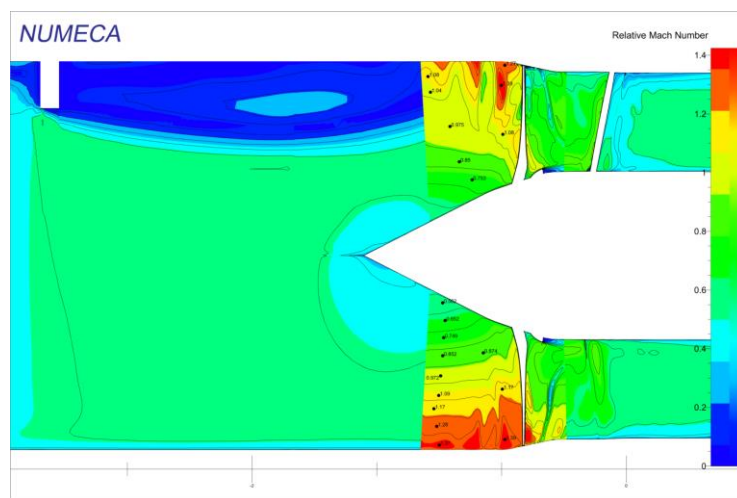


Figure 11. Field of the relative Mach number of the fan channel

4. Conclusion

Calculation model of LPC was created, which allows calculating the characteristics of LPC to a sufficient degree of accuracy. The model consists of fan stages, the fan duct and booster.

The numerical investigation of the air irregularity influence on the fan stage performance was performed. It is shown that the air irregularity increase leads to reduction of the maximum air flow and maximum pressure ratio, to decrease efficiency. It can cause LPC and engine failure.

Acknowledgments

This work was supported by the Ministry of education and science of the Russian Federation in the framework of the implementation of the Program of increasing the competitiveness of Samara National Research University among the world's leading scientific and educational centres for 2013-2020 years.

References

- [1] Kulagin, V. V., 2002, "Teoria, raschet i proektirovanie aviacionnyh dvigateley i energeticheskikh ustanovok. Uchebnik. Osnovy teorii GTD. Rabochiy process I termodinamicheskii analiz", (The theory, calculation and design of aircraft engines and power plants: a Textbook. Fundamentals of the theory of the CCD. Workflow and thermodynamic analysis), 616 p.
- [2] Frohnäpfel, D.J., Ferrar, A.M., Bailey, J.M., O'Brien, W.F., Lowe, K.T. "Measurements of fan response to inlet total pressure and swirl distortions produced by boundary layer ingesting aircraft configurations", 54th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum, Paper No. AIAA 2016-0533.
- [3] Von Karman Institute for Fluid Dynamics, Aero-engine Design: From State of the Art Turbofans Towards Innovative Architectures, Lecture Series 2008-03.
- [4] Kolmakova, D., Popov, G. "Methods of improving the axial compressor flow passage to reduce the flow circumferential nonuniformity", Proc. of ASME 2015 Gas Turbine India Conference, Paper No. GTINDIA2015-1276.
- [5] NUMECA, Theoretical Manual, FINE/Turbo v8.9 Flow Integrated Environment, NUMECA.inc., Belgium, 2011.
- [6] Shablii L. S. and Dmitrieva I. B., 2014, "Blade geometry transformation in optimization problems from the point cloud to the parametric form," Russian Aeronautics, 57(3), 2014, pp. 276-282.
- [7] Kuz'michev, V.S., Tkachenko, A.Y., Krupenich, I.N. and Rybakov, V.N., 2014, "Composing a virtual model of gas turbine engine working process using the CAE system "ASTRA"," Research Journal of Applied Sciences, 9(10), pp. 635-643.