عنوان مقاله:

(Numerical Investigation of Supersonic Flows on Innovative Nozzles (Dual Bell Nozzle

محل انتشار:

دوماهنامه مکانیک سیالات کاربردی, دوره 16, شماره 4 (سال: 1402)

تعداد صفحات اصل مقاله: 11

نویسندگان:

H. Kbab - *University of Blida۱, Aeronautics and Space Studies Institute, Aeronautical Sciences Laboratory, BP۲۷۰ Soumaa street, Blida, Algeria*

O. Abada - *University of Blida۱, Aeronautics and Space Studies Institute, Aeronautical Sciences Laboratory, BP۲۷۰ Soumaa street, Blida, Algeria*

S. Haif - *University of Blida۱, Aeronautics and Space Studies Institute, Aeronautical Sciences Laboratory, BP۲۷۰ Soumaa street, Blida, Algeria*

خلاصه مقاله:

The Dual Bell Nozzle is the most ambitious of several supersonic, altitude-compensating nozzle concepts for rocket engines. This design's objective is to enhance performance in two different evolving regimes (Sea-Level and High Altitude Mode) by self-adaptation with no mechanical control. The concept is simple in theory, but the structural efforts involved are significant. The study carried out in this paper is a simulation of the flows in this type of nozzle. Computational fluid dynamics (CFD) is increasingly used as an analytical tool in research and industry. Simulation is not a substitute for experimentation but a complement to it; it allows the analysis of the problem in real conditions (reproduce tests that are done in experimentation to better understand them and at lower cost) or, on the contrary, in extreme or marginal test conditions (extreme climates, installation defects, etc.). Through simulation, the studied system becomes more flexible. We can easily make parametric studies. Simulation almost always takes the form of a program or computer tool. These are commonly called simulation environments. Developments and progress over the past two decades have led to the emergence of a methodology that has become standard. As for any complex system, the control of a phenomenon is based on the identification and modularization of the tasks. Currently, the standard methodology divides the simulation process into four distinct tasks: geometric modeling, meshing, solving, and finally analysis and visualization. In this study, we will present a test case used to validate our computational models that will be used to optimize the profile of a dual bell nozzle. We will use the Ansys-ICEM environment to generate the meshes and the Ansys-Fluent environment to solve the equations of our models. Our results will then be .compared with experimental and numerical data from our literature review

کلمات کلیدی:

(CFD, Prandtl-Meyer expansion fan, ANSYS-Fluent, Supersonic flow, Method of characteristics (MOC