Document details

‹ Back to results | 1 of 3 Next ›

International Journal of Mechanical and Production Engineering Research and Development
Volume 8, Issue 6, 2018, Pages 373-382

Numerical analysis of convergent-divergent nozzle using finite element method (Article)

Fharukh Ahmed, G.M.[a,b], Alrobaian, A.A.[c], Aabid, A.[d], Khan, S.A.[d]

[a]Department of Mechanical Engineering, Bearys institute of Technology, Mangalore, Karnataka, India
[b]Department of Mechanical Engineering, Government Engineering College, Huvinahadagali, Karnataka, India
[c]Department of Mechanical Engineering, Qassim University, Buraidah, Saudi Arabia

View additional affiliations ⌄

Abstract

⌄ View references (26)

In this paper, Finite element method (FEM) were used to simulate the different flow configuration. Convergent divergent (CD) nozzle was considered with extended divergent duct. 1 mm of microjets orifice diameter were arranged at ninety degrees at PCD 13 mm to control base pressure in a suddenly expanded flow. The designed Mach number of CD nozzleis 1.87 and area ratio 3.24 was considered. The different L/D of the duct was used from 2 to 10. The nozzle pressure simulated for 3, 5, 7, 9 and 11. In this case. Two-dimensional planar model was designed using ANSYS fluent analysis . The total wall pressure distribution and Mach number from inlet to the outlet was observed. From the results, it is detected that the microjets control the loss of pressure and decreases the drag at the suddenly

simulated for 3, 5, 7, 9 and 11. In this case. Two-dimensional planar model was designed using ANSYS fluent analysis . The total wall pressure distribution and Mach number from inlet to the outlet was observed. From the results, it is detected that the microjets control the loss of pressure and decreases the drag at the suddenly expanded region. The results also show, we can fix the flow parameter which will result in the maximum gain in the base pressure and velocity. In present study, the CD nozzle designed and modelled using available ANSYS fluent database: K-ε standard wall function turbulence model has been used and validated with the commercial computational fluid dynamics (CFD). © TJPRC Pvt. Ltd.

SciVal Topic Prominence

Topic: Pressure | Nozzles | micro jets

Prominence percentile: 32.230

Author keywords

ANSYS | C-D nozzle | CFD | Pressure & Mach Number

ISSN: 22496890
Source Type: Journal
Original language: English

DOI: 10.24247/ijmperddec201842
Document Type: Article
Publisher: Transstellar Journal Publications and Research Consultancy Private Limited (TJPRC)

References (26)

View in search results format >

All | Export | Print | E-mail | Save to PDF | Create bibliography