

Document details

< Back to results | 1 of 3 Next >

Export Download Print E-mail Save to PDF Add to List More... >

Full Text View at Publisher

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

^aDepartment of Mechanical Engineering, Bearys institute of Technology, Mangalore, Karnataka, India
^bDepartment of Mechanical Engineering, Government Engineering College, Huvinahadagali, Karnataka, India
^cDepartment of Mechanical Engineering, Qassim University, Buraidah, Saudi Arabia

View additional affiliations v

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 nozzles 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

Metrics

0 Citations in Scopus
0 Field-Weighted Citation Impact

PlumX Metrics
Usage, Captures, Mentions, Social Media and Citations beyond Scopus.

Cited by 0 documents

Inform me when this document is cited in Scopus:
Set citation alert > Set citation feed >

NEW! SciVal Topic Prominence is now available in Scopus.
Which Topic is this article related to? View the Topic.

68157 Planning ac...pdf ^

Show all x

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/ijmperddc201842
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

Set citation alert > Set citation feed >

Related documents

[CFD analysis of convergent-divergent nozzle flow and base pressure control using micro-JETS](#)

Khan, A. , Aabid, A. , Khan, S.A.
(2018) *International Journal of Engineering and Technology(UAE)*

[CFD analysis of cd nozzle and effect of nozzle pressure ratio on pressure and velocity for suddenly expanded flows](#)

Khan, S.A. , Aabid, A. , Baig, M.A.A.
(2018) *International Journal of Mechanical and Production Engineering Research and Development*

[An investigation to control base pressure in suddenly expanded flows](#)

Pathan, K.A. , Dabeer, P.S. , Khan, S.A.
(2018) *International Review of Aerospace Engineering*

[View all related documents based on references](#)

Find more related documents in Scopus based on:

NEW! SciVal Topic Prominence is now available in Scopus. ✕

Which Topic is this article related to? [View the Topic.](#)

