Computational Fluid Dynamic Analysis of Abrasive Jet Nozzles

Dr D V Sreekanth
St martins Engineering college

Saurav Das
St.Martin's Engineering College

P.Santosh Kumar Patra
ST. MARTIN'S ENGINEERING COLLEGE

ISSN: 1533-7812
Vol-21-Issue-06-June-2020

Computational Fluid Dynamic Analysis of Abrasive Jet Nozzles

Dr.D.V. Sreekanth[1], Dr.Saurav Das[2*], Dr. P. Santosh Kumar Patra[3], Dr.M.Sreenivasa Rao[4]

[1]Professor and H.O.D, Mechanical Department, St. Martin's Engineering College, Secunderabad, Telangana, 500100, Email: hodmech@smec.ac.in

[2*]Professor and R&D in-charge, Mechanical Department, St. Martin's Engineering College, Secunderabad, Telangana, 500100, Email: drsauravme@smec.ac.in (Corresponding Author)

[3]Principal & Professor, CSE Department, St. Martin's Engineering College, Secunderabad, Telangana, 500100, Email: principal@smec.ac.in

[4]Professor, Mechanical Department, JNTUH, Hyderabad, Telangana, Email: raoms63@rediffmail.com

ABSTRACT

Abrasive Jet Machining (AJM) utilizes mechanical energy and is one of the advanced machining processes. Energy in its direct form is acting for metal removal. The erosion takes place due to high speed abrasive particles mixed with carrier gas. Micro abrasive particles are propelled by a pressurized air of different velocities. AJM includes drilling, grinding, honing, and deburring. The common nozzle shapes used for machining by AJM are rectangular or circular. In the present work a conical nozzle with convergent hole is introduced. This present work investigates the flow of abrasive particles in the inlet of the convergent nozzle. A 3D model of a nozzle design and size of abrasive particles was carried out for the erosion process is done in CREO software of Computer Aided Design (CAD). The different machining conditions for the abrasive flow at inlet and outlet of the nozzle is analyzed by Computational Fluid Dynamics (CFD) simulation. The simulation results are analyzed, and the differences are identified. For each nozzle diameter (1mm, 2mm, 3mm) the analysis is made at different pressures and velocities. The influence of inlet nozzle velocity, pressure, nozzle diameter and Stand Off Distance (SOD) on the abrasive flow is investigated and the results are presented.

Keywords: Abrasive Jet Machining, Computational Fluid Dynamics, nozzle, Stand Off Distance

1. INTRODUCTION

Computational fluid dynamics, commonly called CFD, is a fluid mechanics branch that uses numerical techniques and algorithms to solve and analyze fluid flux problems. Computers are used to measure the interaction of liquid and gas with boundary defined surfaces to simulate them. Better options can be found with high-speed supercomputers [1].

Navier – Stokes equations, which describe any single phase (gas, liquid, but not both) fluid flow, are the fundamental basis for nearly all CFD problems. These equations can be simplified by removing terms that define viscous steps to generate the Euler equations. Further simplification results in complete potential equations by eliminating terms defining vortices. Finally, these equations can be linearized to generate linearized potential equations for minor disturbances in subsonic [2].

ISSN: 1533-7812
Vol-21-Issue-06-June-2020

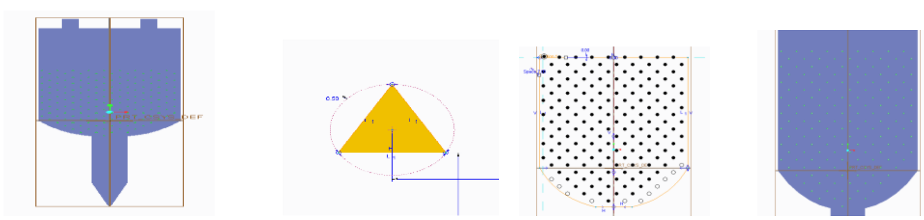
ANSYS Fluent also allows the flow solution to refine or coarsen the mesh. We can move the mesh to the fluent ANSYS or build y mesh for 3D geometries using meshing mode of ANSYS. All remaining operations are conducted in the Fluent solution mode; including boundary conditions, fluid characteristics description, and solution execution, meshes processing, post-processing processing and results viewing. The serial solver ANSYS Fluent handles input and output, data storage and flow field calculations with the aid of a single computer solver processes. ANSYS Fluent also uses a cortex tool that handles the user interface and the basic graphical functionalities of ANSYS Fluent. The parallel resolution of ANSYS Fluent allows a solution to be measured using several processes on the same system or on separate network computers [3].
In this article, the CFD study takes place via a simulation of the nozzle boundary, which limits the nozzle diameter from 1 mm to 8 mm. A modeling software called a CREO software is used for CFD analysis.

2. CREO

CREO Parametric is the 3D product standard, providing industry-leading usability tools that encourages best practices in design and ensure compliance. Integrated CREO CAD / CAM / CAE solutions enable the design of outstanding products faster than ever while optimizing creativity and quality.
Industrial requirements may change, and the time pressure might continue to increase, but product design needs remain the same-irrespective of the scope of the project, the solution offered by CREO is strong, simple to implement and affordable [4].

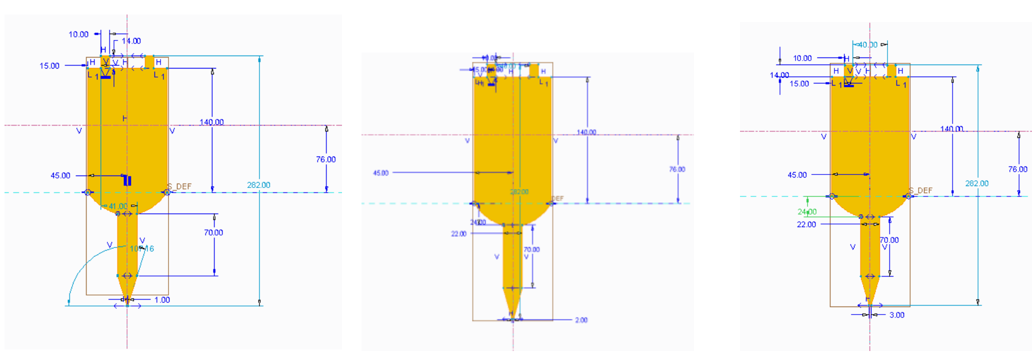
3. DESIGN

Based on different modules of CREO the part design, assembly, Drawing and Sheet metal manufacturing of mixing chamber modules are made.



(A) (B) (C) (D)

Figure 1 (A to D) : A) 2-D design of AJM Mixing chamber B) outer boundary in nozzle and C&D) Final distribution of abrasive particles



(A) Nozzle with 1mm dia (B) Nozzle with 2mm dia (C) Nozzle with 3mm dia

Figure 2 (A to C): Nozzles of different diameters designed with CREO

4. CFD ANALYSIS

Ansys is used along with Fluent for CFD analysis. Fluent uses client / server architecture. The process followed is standard ANSYS operating procedures. This allows it to run on client desktop workstations and powerful application servers as separate simultaneous processes. This architecture enables effective output, interactive control and full flexibility among various machine types or operating systems. ANSYS Fluent

ISSN: 1533-7812
Vol-21-Issue-06-June-2020

offers total mesh versatility, including the possibility of solving flux problems by means of unstructured meshing which can be generated with relative ease in complex geometries [3]. The nozzle ranges are 1mm to 3mm and the inlet pressures of 2 kg/cm^2, 4 kg/cm^2, 6 kg/cm^2, 8 kg/cm^2.

MODULE 1: Nozzle diameter: 1mm

CASE 1: At inlet pressure 2 Kg/cm^2

Once the pre-processing is over the model is set for analysis. The steps are boundaries for meshing, completed mesh, location of abrasive inlet, alternative abrasive inlet and location of abrasive outlet at an air pressure of 2Kg/cm^2 at velocity of 177 m/s are analyzed as shown in Figures 3 and 4. The output-based parameters selection is analyzed by considering 1mm nozzle diameter and velocity 177 m/sec as given in Figure 4. The inlet and outlet conditions are observed, wherein the velocity contours show the same effect, as Figure 3 shows an exploded view as compared to Figure 4. It clearly shows that the tip of nozzle is always the most happening point of mass flow velocity. The Mass Flow Rate Convergence and Residuals with number of iterations are shown in Graphs 1, 2. They depicts that the mass flow residuals as shown in Graph 1 and the iteration residuals in Graph 2, proves that the results are analyzed after the convergence of results.

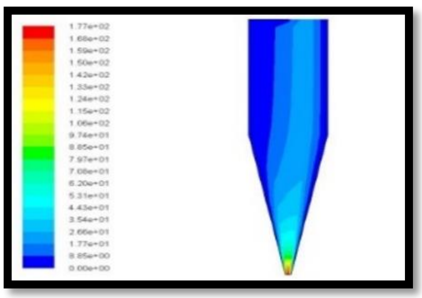


Figure 3: Nozzle output at 2 Kg/cm^2

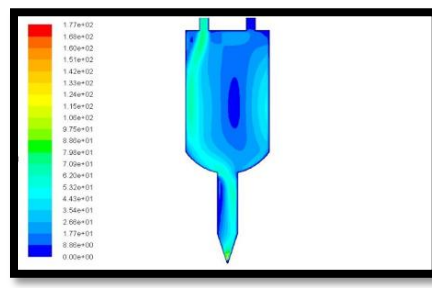
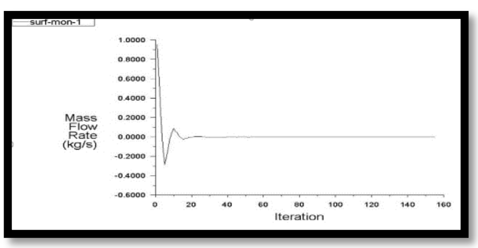
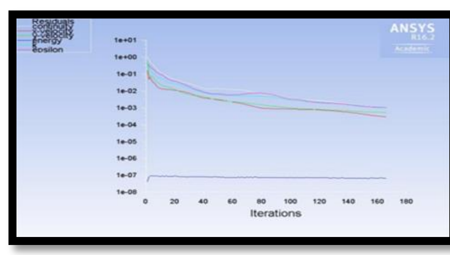


Figure 4: Velocity parameter at the nozzle



Graph 1: Mass Flow rate Convergence



Graph 2: Scaled Residuals

CASE 2: At inlet pressure 4 Kgs/ cm ^{2}

In Case 2 the nozzle inlet pressure is taken as 4 Kgs/ cm ^{2} at velocity 251 m/sec. The nozzle output is analysed in Figure 5 which shows that the most influencing area is nozzle of tip. The variation in velocities is discussed in Figures 5 and 6. The Mass Flow Rate Convergence and Residuals with number of iterations are discussed in Graphs 3 and 4. As obvious the tip of the nozzle is again subjected to more velocity fluctuations as compared to the other area, as observed from the Figures.5 and 6. Also, the Graphs 3 and 4 shows that the mass residuals and scaled residuals proves the convergence criteria.

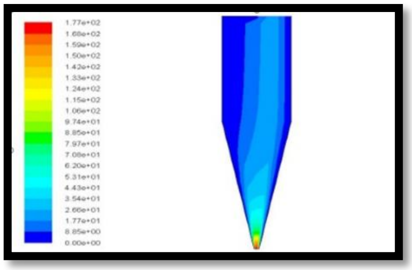


Figure 5: Nozzle output (4Kg/ cm ^{2})

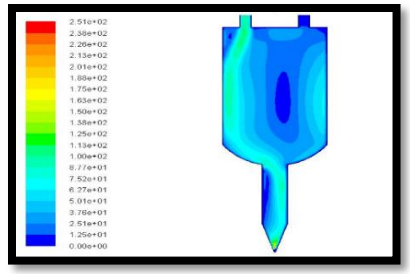
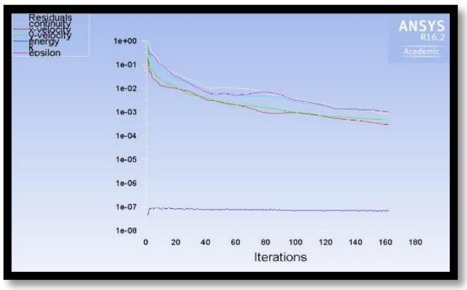
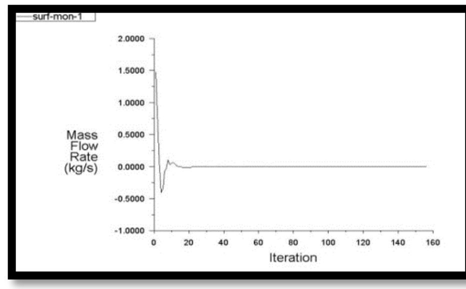


Figure 6: Velocity Variations



Graph 3: Mass flow rate convergence



Graph 4: Residuals Vs Iterations.

CASE 3: At inlet pressure 6 Kgs/ cm ^{2}

In Case 3 the nozzle inlet pressure is taken as 6 Kgs/ cm ^{2} at velocity 307 m/s. The nozzle output is analysed in Figure 7 which shows that the most influencing area is nozzle tip. The variation in velocities is shown in Figure 8. The Mass Flow Rate Convergence and Residuals with number of iterations are discussed in Graph 5 and 6. By Figure 7 it can be analysed that the effect of pressure at nozzle tip is higher as compared with Figure 8.

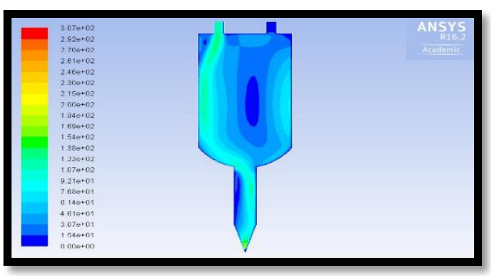


Figure 7: Nozzle output (6Kg/ cm ^{2})

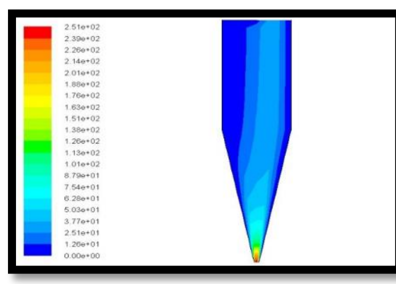
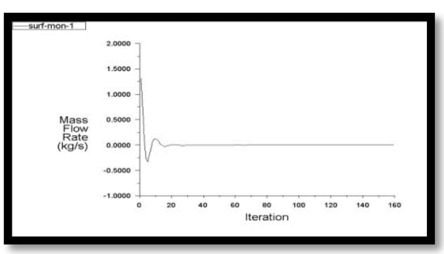
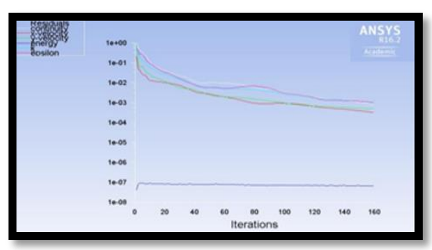


Figure 8: Velocity Variation



Graph 5: Mass flow rate convergence



Graph 6: Residuals Vs Iterations

CASE 4: At inlet pressure 8 Kgs/ cm ^{2}

In Case 4 the nozzle inlet pressure is taken as 8 Kgs/cm^2 at velocity of 355 m/s. The nozzle output is analysed in Figure 7 which shows that the most influencing area is nozzle tip. The variation in velocities is

ISSN: 1533-7812
Vol-21-Issue-06-June-2020

discussed in Figure 8. The Mass Flow Rate Convergence and Residuals with number of iterations are shown in Graphs 7 and 8. By Figure 9 it can be analysed that the effect of pressure at nozzle tip is high compared with 6 kg/cm^2. Figure 10 gives the Influence of air flow at the tip of nozzle.

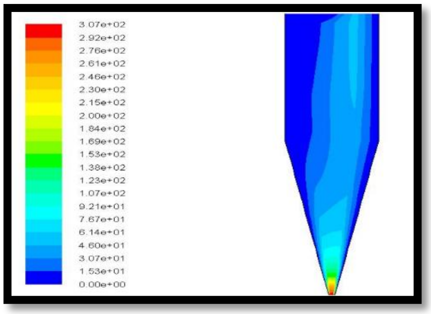


Figure 9: Nozzle output (8Kg/ cm ^{2})

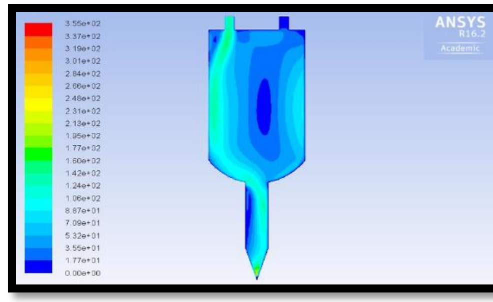
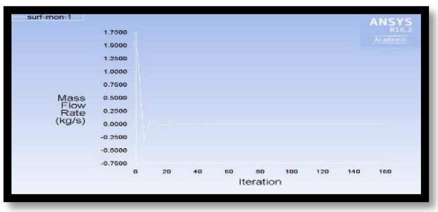
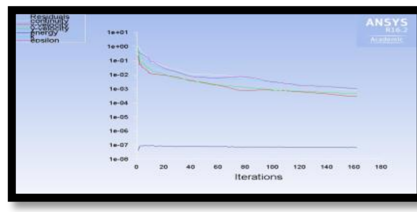


Figure 10 : Velocity variations for air flow



Graph 7: Mass Flow rate Convergence



Graph 8: Residual Vs Iterations.

MODULE 2 : Diameter Of Nozzle: - 2mm

CASE 1: At inlet Pressure 2 Kg/ cm ^{2}

The output based parameters selection is analysed by considering 2mm nozzle diameter and velocity 168 m/sec at pressure 2 kg/cm^2 as given in Figure 11 and 12.The Mass Flow rate Convergence and Residuals with number of iterations are discussed in Graphs 9 and 10.

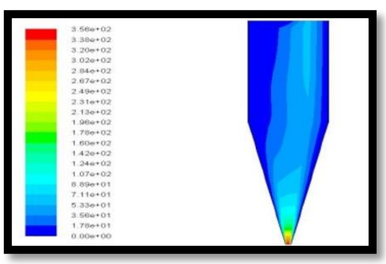


Figure 11: Nozzle output (2Kg/ cm ^{2})

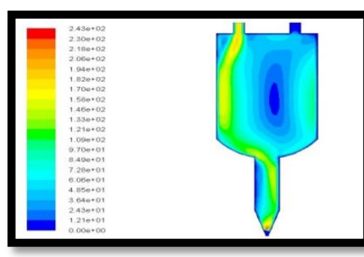
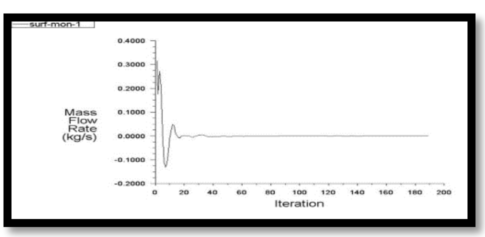
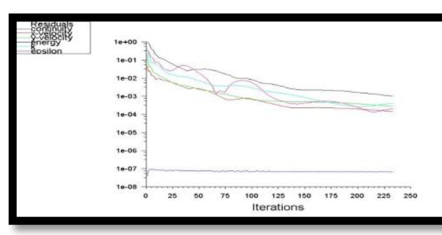


Figure 12 : Velocity variations



Graph 9: Mass Flow rate Convergence



Graph 10: Residual Vs Iterations.

ISSN: 1533-7812
Vol-21-Issue-06-June-2020

CASE 2: Nozzle with inlet pressure 4 Kgs/ cm ^{2}

In Case 3 the nozzle inlet pressure is taken as 4 Kg/ cm ^{2} at velocity of 298 m/s. The nozzle output is analysed in Figure 13 which shows that the most influencing area is nozzle tip. The variation in velocities is discussed in Figure 14. The Mass Flow rate Convergence and Residuals with number of iterations are discussed in Graph 11 and 12. By Figure 13 it can be analysed that the effect of pressure at nozzle tip is little high because of the reason that the pressure is medium.

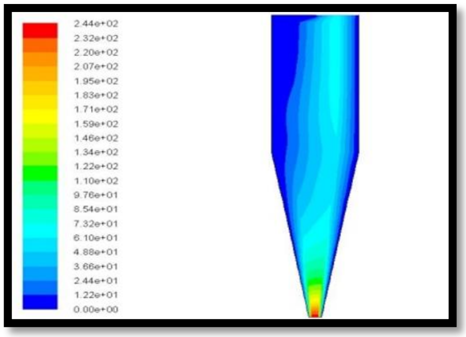


Figure 13: Nozzle output (4Kg/ cm ^{2})

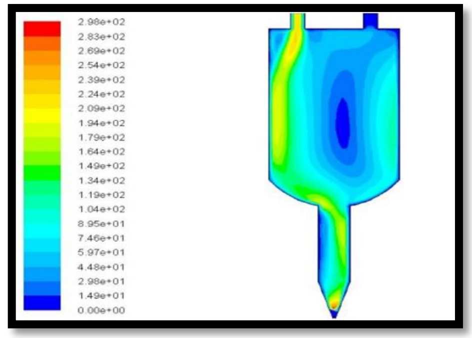
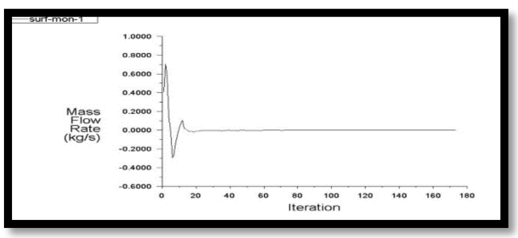
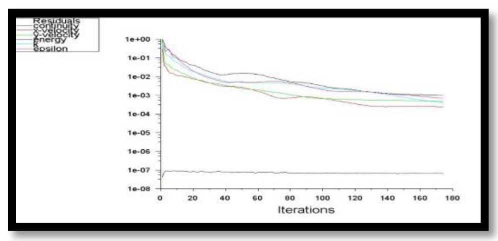


Figure 14: Velocity Variations



Graph 11: Mass Flow rate Convergence



Graph 12: Residual Vs Iterations.

CASE 3: At inlet pressure 6 Kgs/ cm ^{2}

In Case 3 the nozzle inlet pressure is taken as 6 Kg/ cm ^{2} at velocity of 345 m/s. The nozzle output is analysed in Figure 15 which shows that the most influencing area is nozzle tip. The variation in velocities is discussed in Figure 16. The Mass Flow Rate Convergence and Residuals with number of iterations are discussed in Graph 13 and 14. By Figure 15 it can be analysed that the effect of pressure at nozzle tip is high because of the reason that the pressure is medium.

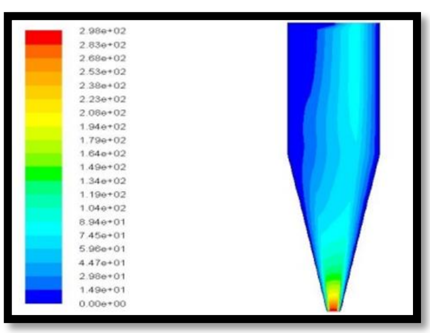


Figure 15: Nozzle output (6Kg/ cm ^{2})

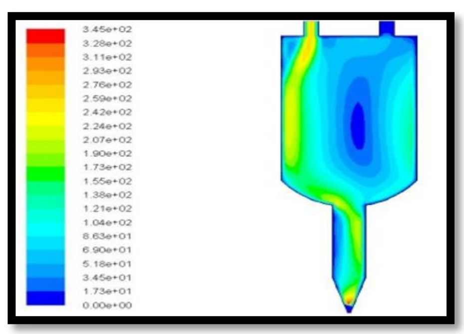
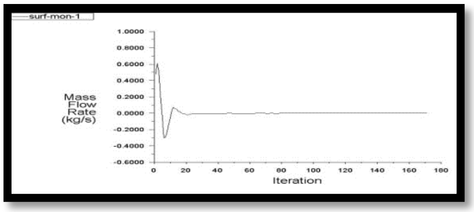
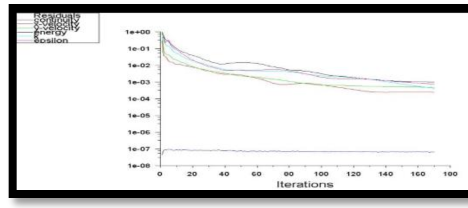


Figure 16: Velocity Variations

ISSN: 1533-7812
Vol-21-Issue-06-June-2020



Graph 13: Mass Flow rate Convergence



Graph 14: Residual Vs Iterations

CASE 4: At inlet pressure 8 Kgs/ cm ^{2}

In Case 4 the nozzle inlet pressure is taken as 8 Kg/ cm 2 .at a velocity of 445 m/s. The nozzle output is analysed in Figure 17 which shows that the most influencing area is nozzle tip. The variation in velocities is discussed in Figure 18. The Mass Flow rate Convergence and Residuals with number of iterations are discussed in Graphs 15 and 16. By Figure 17 it can be analysed that the effect of pressure at nozzle tip is very high because of the reason that the pressure is high.

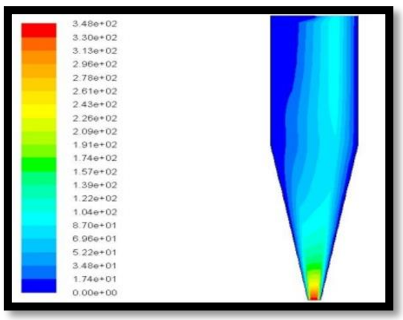


Figure 17: Nozzle output (8Kg/ cm ^{2})

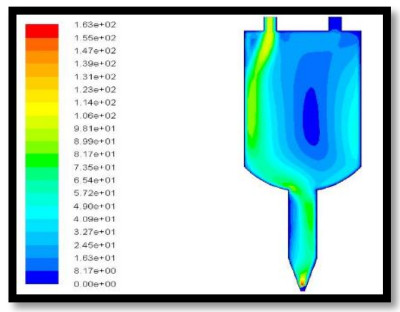
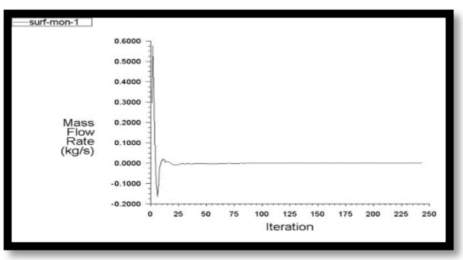
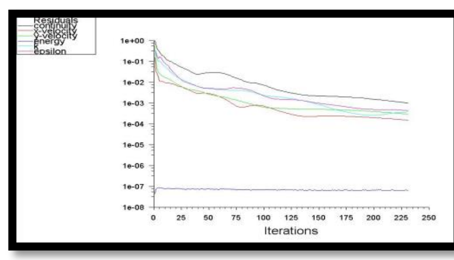


Figure 18: Velocity Variations



Graph 15: Mass Flow rate Convergence



Graph 16: Residual Vs Iterations

MODULE 3 : Diameter Of Nozzle: - 3 MM

CASE 1: Nozzle with inlet pressure 2 Kgs/ cm ^{2}

In Case 4 the nozzle inlet pressure is taken as 2 Kg/ cm^2 at velocity of 232 m/s. The nozzle output is analysed in Figure 19 which shows that the most influencing area is nozzle tip. The variation in velocities is discussed in Figure 20. The Mass Flow Rate Convergence and Residuals with number of iterations are discussed in Graph 17 and 18.

ISSN: 1533-7812
Vol-21-Issue-06-June-2020

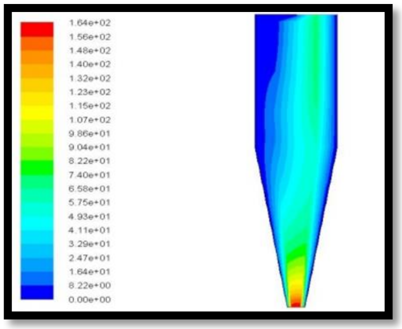


Figure 19: Nozzle output (2 Kg/ cm ^{2})

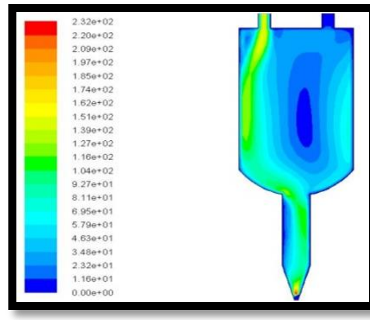
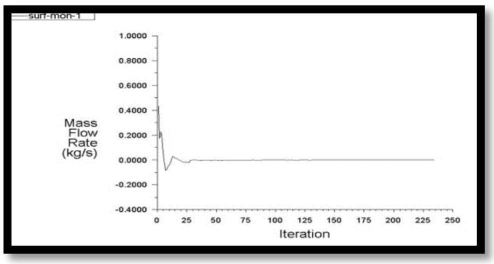
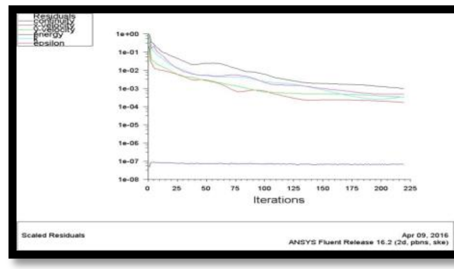


Figure 20: Velocity Variations



Graph 17: Mass Flow rate Convergence



Graph 18: Residual Vs Iterations

CASE 2: At inlet pressure 4 Kgs/ cm ^{2}

In Case 2 the nozzle inlet pressure is taken as 4 Kg/ cm 2 at velocity of 292 m/s. The nozzle output is analysed in Figure 21 which shows that the most influencing area is nozzle tip. The variation in velocities is discussed in Figure 22. The Mass Flow Rate Convergence and Residuals with number of iterations are discussed in Graphs 19 and 20.

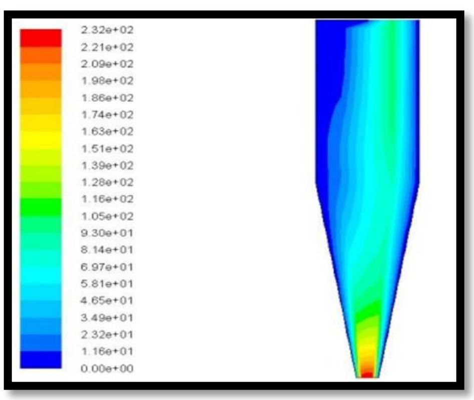


Figure 21: Nozzle output (4Kg/ cm ^{2})

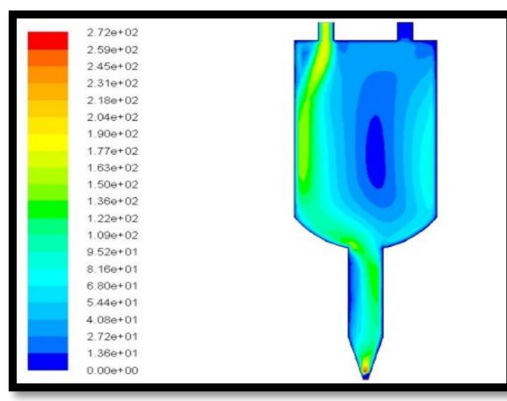
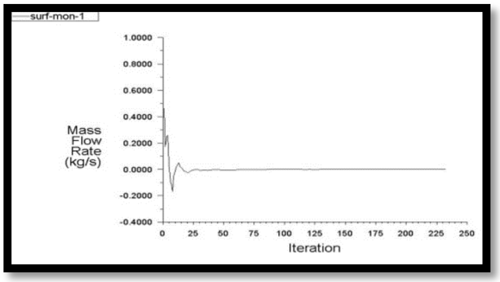
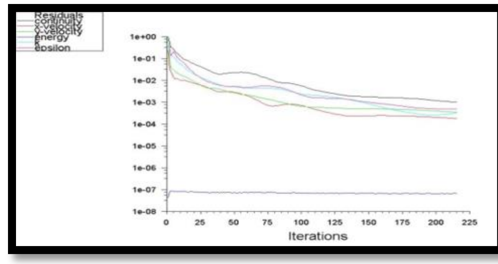


Figure 22: Velocity Variations



Graph 19: Mass Flow rate Convergence



Graph 20: Residual Vs Iterations

CASE 3: NOZZLE with inlet pressure 6 Kgs/ cm ^{2}

In Case 3 the nozzle inlet pressure is taken as 6 Kg/ cm 2 at velocity of 339 m/s. The nozzle output is analysed in Figure 23 which shows that the most influencing area is nozzle tip. The variation in velocities is discussed in Figure 24. The Mass Flow Rate Convergence and Residuals with number of iterations are discussed in Graph 21 and 22.

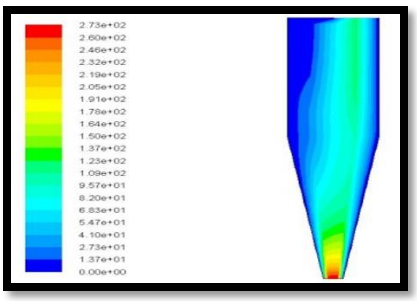


Figure 23: Nozzle output (6 Kg/ cm ^{2})

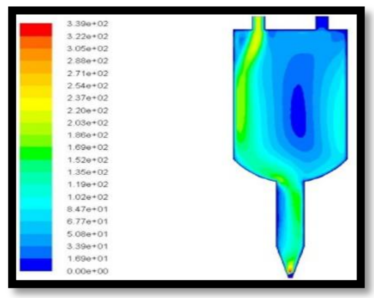
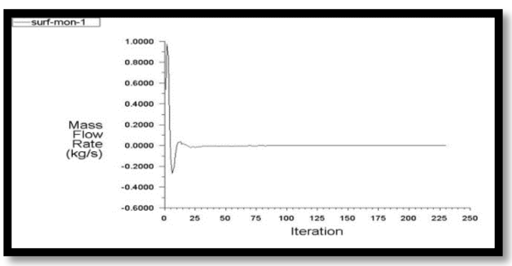
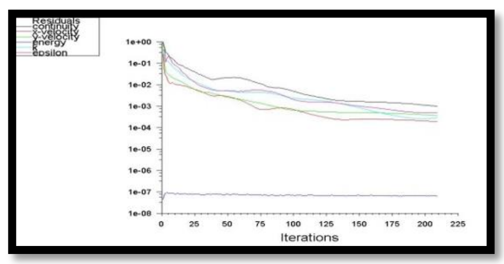


Figure 24: Velocity Variations



Graph 21: Mass Flow rate Convergence



Graph 22: Residual Vs Iterations

CASE 4: At inlet pressure 8 Kgs/ cm ^{2}

In Case 4 the nozzle inlet pressure is taken as 8 Kg/ cm 2. The nozzle output is analysed in Figure 25 and 26 which shows that the most influencing area is nozzle tip. The Mass Flow Rate Convergence and Residuals with number of iterations are shown in Graphs 23 and 24 which shown convergence.

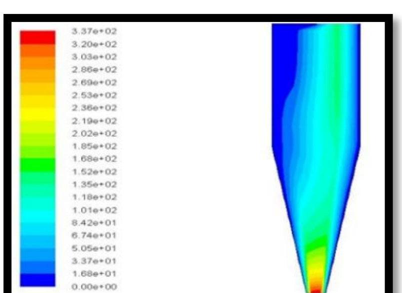


Figure 25: Nozzle output (8 Kg/ cm ^{2})

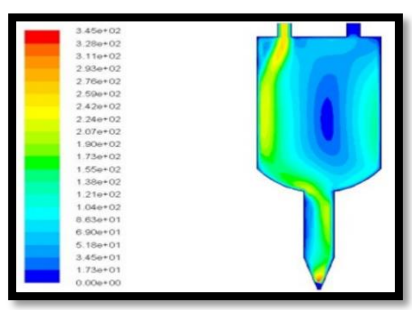
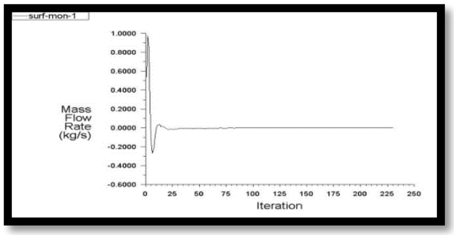
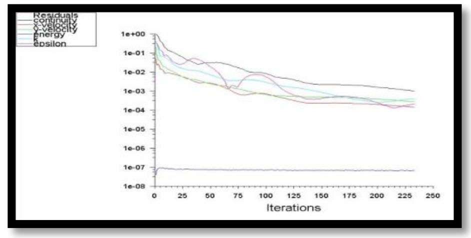


Figure 26: Velocity Variations in the whole domain

ISSN: 1533-7812
Vol-21-Issue-06-June-2020



Graph 21: Mass Flow rate Convergence



Graph 22: Residual Vs Iterations

5. CONCLUSIONS

The CATIA CREO architecture and meshing was done in the experimental study of Abrasive Jet nozzles using CFD study and is used in the smooth analysis. Importance and outcomes of CFD analyses.

- Figure 1A to 1D indicates the design of nozzles and distribution of abrasive particles in mixing chamber, followed by 2A to 2C represents inlet and meshing.
- The Figure 3,5,7,9,11,13,15,17,19,21,23,25 represents the velocity at the throat of nozzle.
- Figure 4,6,8,10,12,14,16,18,20,22,24 and 26 represents the flow of air in mixing chamber with variation in velocity.
- Graph 1,3,5,7,9,11,13,15,17,19,21 represents the mass flow rate of convergence at different nozzle diameters and pressures
- Graph 2,4,6,8,10,12,14,16,18,20,22 represents the Residual Vs number of iterations at various nozzle pressures in related with different modules on nozzle diameters.
- The pressure and the MF and Nozzle diameter changeswith varying pressures of 2, 4, 6 and 8 Kg / cm2 of 1 mm, 2 mm, and 3 mm nozzles, to compare the flow characteristics.
- The speed in a 1 mm nozzle at an eight kg / cm2 pressure has been found to be very high and the experimental results (by increasing MRR pressure) correlate to the actual working model.
- The 2 mm nozzle has a high velocity of 8 kg / cm2 and the experimental data (in addition to the MRR pressure) almost matches the real operating model.
- The velocity of a 3 mm pitcher, 6 kg / cm2, is found to be very high, with almost the actual working pattern being consistent with experimental results (by the pit diameter of the MRR at optimum pressures).

Reference

1. U. Prisco and D'Onofrio, M.C., Three-dimensional CFD simulation of two-phase flow inside the abrasive water jet cutting head, International Journal for Computational Methods in Engineering Science and Mechanics, 9 (2008), pp. 300-319.
2. H. Liu, J. Wang, R.J. Brown, N. Kelson, Computational fluid dynamics (CFD) simulation of ultrahigh velocity abrasive waterjet, Key Eng. Mater. 233–236 (2003) 477–482.
3. ANSYS, ANSYS Fluent Theory Guide, ANSYS, Inc. Canonsburg, PA.
4. Manolea Daniel, Petrianu Cristian, Rules of Defining Sketches in PRO/ENGINEER Wildfire / CREO / CREO Parametric, ANNALS of the ORADEA UNIVERSITY,
Fascicle of Management and Technological Engineering, Volume XI (XXI), 2012, NR2.