AN APPROACH FOR OPTIMISING THE FLOW RATE CONDITIONS OF A DIVERGENT NOZZLE UNDER DIFFERENT ANGULAR CONDITIONS

Lam Ratna Raju¹, Ch. Pavan Satyanarayana², Neelamsetty Vijaya Kavya³

¹,³Department of Mechanical Engineering, Vignan’s Foundation for Science, Technology & Research (Deemed to be University), Vadlamudi, Andhra Pradesh, India.

²Institute M.Tech Student, Department of Mechanical Engineering, Acharya Nagarjuna University, NH16, Nagarjuna Nagar, Guntur, Andhra Pradesh

¹lamratnaraju@gmail.com, ²chpavansatyanarayana@gmail.com, ³kavyanilamsetty@gmail.com

Corresponding Author: Chelamalasetti Pavan Satyanarayana

https://doi.org/10.26782/jmcms.2020.07.00054

Abstract

A spout is a device which is used to offer the guidance to the gases leaving the burning chamber. Spout is a chamber which has a capability to change over the thermo-compound essentials created within the ignition chamber into lively vitality. The spout adjustments over the low speed, excessive weight, excessive temperature fuel in the consuming chamber into rapid gasoline of decrease weight and low temperature. An exciting spout is used if the spout weight volume is superior vehicles in supersonic airplane machines commonly combine a few sort of a distinctive spout. Our exam is surpassed on the use of programming like Ansys Workbench for arranging of the spout and Fluent 15.0 for separating the streams inside the spout. The events of staggers for the pipe formed spouts have been seen close by trade parameters for numerous considered one of a kind edges. The parameters underneath recognition are differentiated and that of shape spout for singular terrific edges by using keeping up the gulf, outlet and throat width and lengths of joined together and diverse quantities as same. The simultaneous component and throat expansiveness are kept regular over the cases. The surprise of stun became envisioned and the effects exhibited near closeness in direction of motion of Mach circle and its appearance plans as exposed in numerous preliminary considers on advancement in pipe molded particular spouts with assorted edges four°, seven°, ten°. Occurrence of stun is seen with higher special factors.

Keywords : Nozzle, Supersonic Rocket Engine, Divergent edges.
I. Introduction

Spout is applied to trade over the compound warmth imperativeness delivered inside the consuming chamber into dynamic vitality. The spout modifications over the low pace, high weight, inordinate temperature gas inside the consuming chamber into high speed gas of lessening weight and temperature. Swedish designer of French dive who, in endeavoring to amass a continuously talented steam engine, purposeful a turbine that become developed to become by means of planes of steam. The principal component – the one wherein warmness intensity of the hot high-strain steam from the pot transformed into changed over into engine control – turned into the spout from which the development blew onto the wheel. De Laval saw that the most talented change went off while the spout recently constrained, growing the speed of the fly to the speed of sound, and a short time later drawn out over once more. Over the speed of sound (yet not beneath it) this augmentation prompted a comparatively increase inside the pace of the fly and instigated a totally capable interchange of warmth vitality to movement. The speculation of air deterrent turned out to be initially proposed by means of Sir Isaac Newton in 1726. As in step with him, a streamlined power is predicated upon the thickness and speed of the fluid, and the shape and the size of the removing object. Newton’s speculation was sooner than extensive sought after by method for other theoretical game plan of simple movement issues.

The sum total of what these had been limited to move underneath mainstream circumstances, for instance air become foreseen to have reliable thickness and to move in mellow of weight and inaction. Nowadays steam turbines are the mainstream control wellspring of electric power stations and enormous pontoons, despite the way that they for the most part have a substitute structure to utilize the short steam stream, de Laval’s turbine expected to run at a strangely quick. Regardless, for rockets the de Laval gush turned out to be actually what gotten required. Computational Fluid Dynamics (CFD) is a planning instrument that permits experimentation. Its degree isn't constantly restricted to fluid components; CFD will be completed to any technique which joins shipping wonders with it. To manage a planning issue we can use excellent procedures simply like the clinical methodology, exploratory techniques utilizing designs. The clinical procedure is troubled and troublesome.

The investigate methodologies are over the top. In the event that any errors inside the arrangement were recognized all through the rendition giving it a shot, another model is to be committed clarifying all of the errors and again endeavored. This is a repetitive similarly as a worth eating up process. The introduction of Computational Fluid Dynamics has vanquished this problem essentially as modified the field of building. In CFD an issue is emulated in programming and the vehicle conditions related with the issue is numerically comprehended with PC help. Henceforth we would have the decision to anticipate the aftereffects of a trouble before experimentation. The present compositions objectives making sense of an incredible different plot for the spout which may convey the most extreme extraordinary outlet pace and meet the push conditions.
Stream uncertainties can be made in the spout because of the advancement if dazes which reduce the leave mach assortment similarly as push of the engine. This might be discarded by utilizing changing the unique factor. Here assessment has been coordinated on gushes with explicit edges five°, 10°, 15°. Experimentation utilizing the styles of each divergent part is an exorbitant similarly as a dreary technique. CFD exhibits to be a profitable instrument to vanquish these requirements. Here on this work the example of different dissemination parameters are in like manner separated.

Future Scope

The paintings is to be moreover proceeded via doing liquid movement investigation for both the fashions and bestowing rotational pace to the fan to quicken the air at lots higher fee. The present exam concentrates just on spherical geometry of the CD Nozzle and this work may be additionally proceeded with the aid of converting the geometry to square and oval at the channel, outlet and on the throat.

Limitations

The fundamental challenges associated with these setups emerge for the duration of the time spent satisfactorily assisting and cooling the focal attachment.

Objectives

1. Configuration, create and wreck down the exhibition of a C-D spout with a rotor and fan.
2. Rotor and fan to be established inside the CD spout on the throat territory.
3. Supplant the spherical and hole walled in location by using CD Nozzle.
4. Investigate the presentation of each by means of exam.

II. Literature Review

Varun, R Et.al, Srinivasan,k (2010) In the present work, the stagger instruct shape in a joined shifted gush is tried utilizing huge twirl reenactment (LES) framework subject to different subgrid models, including Smagorinsky-Lilly, Wall-Adapting Local Eddy-Viscosity (WALE) and Algebraic Wall-Modeled LES (WMLES). Thusly, the focal point of the overall test is to study unmistakable subgrid models for you to predict the region of ordinary and inclined staggers just as a tremendous numerical examination of a daze teach structure. In this one of a kind situation, dynamic network adaption procedures and the cream instatement inside the Fluent writing computer programs are done under the 3 dimensional assessment to diminish numerical mistakes and computational cost. The eventual outcomes of assorted subgrid models are differentiated and the available preliminary realities and it's miles demonstrated that the WMLES can give more noteworthy genuine outcomes than Smagorinsky–Lilly and WALE models. Quickly, the daze instruct conduct is obliged through utilizing the cooling divider temperature, delta Mach amount and channel Reynolds number inciting adjustments first and foremost phase of daze, daze...
decent, division among dazes, least weight and the most serious development Mach extend.

**Pandey, K.M Et.al (2010)** In this examination, Large Eddy Simulation (LES) has been applied to look at supersonic move and consuming in a model scramjet combustor. The LES model depends upon on an unstructured controlled amount discretization, utilizing total variational diminishing movement entertainment, of the isolated movement, weight, enthalpy, and inert/responsive scalar circumstances, used to painting the copying methodology. The design applied resembles the exploration office scramjet at the Institute for Chemical Propulsion of the German Aerospace Center (DLR) and involves an uneven outstanding channel with a wedge-formed flameholder at the base of which hydrogen is injected. Here, we investigate supersonic stream with hydrogen imbuement and supersonic flow with hydrogen implantation and start. With the quit reason for endorsement, the LES impacts are differentiated and preliminary information for speed and temperature at various pass-territories. Moreover, abstract assessments are similarly made among foreseen and anticipated shadowgraph photos. The LES figurings are fit as a fiddle for looking for each the non-reacting and reacting flowfields reasonably appropriately—especially we see that the LES rendition recognizes and isolates among attributes of the flowfields situated inside the assessments.

**Natta, Pardhasaradhi Et. al, (2012)** A Supersonic Swirling Separator (SSS) framework will be accommodatingly grasped in the combustible gas transmission pipelines so one can assemble and isolate the noteworthy hydrocarbons from the vaporous degree without basic weight incidents. In the present work, designs have been made Sto repeat the miracles drew in with the strategy: supersonic dissemination stimulating in a spout, stream spinning sway, dissimilar segment of liquid and vaporous ranges inside the end promptly pipe. The model allows a decent first try 1D plan of the whole separator.

**K.M. Pandey Et. al, (2010)** A customary Natural Gas mix become considered and, beginning from that, two stand-out revised mixes with decreased assortment of sections (3 and five) have been described so one can achieve a thought process pleasing and solid form with commendable computational time and effects for a significant shape instrument. The GERG EOS have been grasped inside the mixes appearing, completed in NIST REFPROP, which can be the most extreme explicit starting at now convenient. The thermodynamic one dimensional model allowed the estimation of the daze wave critical situation inside the different channel, the key thermodynamic habitations allocation close by the spout and the sythesis of the liquid and smoke degrees. An improved at this point altogether specific numerical methodology transformed into made to assess the two degree sound speed. A comprehensive affectability test changed into completed for the standard extent of the framework confine conditions. What's extra, a model for the amusement of the gas liquid outward segment was proposed. Joining the 2 models acknowledged the essential arrangement of the separator geometry and the trial of the conventional development and segment execution.
Shigeru Et. al, (2005) Investigation of the acoustic imprint conveyed through truncated best structure and push progressed illustrative spouts is driven all through each fixed and brief (startup) exercises. The truncated flawless structure gush experiences free-paralyze segment stream, while the surge prevalent illustrative spout experiences both free daze separation and limited stagger division stream states at some phase in startup. This exploration offers a prompt assessment of the acoustic imprint brought all through loosened daze parcel and limited daze office development states simultaneously as running under vague spout strain extents. During a concise scene, the endless wavelet change is applied to examine the acoustic imprints made with the guide of the spouts. The truncated flawless structure gush shows a moderate addition in broadband repeat essentialness with expanding gush strain share and with broadband shock upheaval showing up at better spout pressure extents. The push increasingly invaluable illustrative spout, nevertheless, recommends a bounty bigger affectability to the spout pressure extent. In particular, the free-shock segment to obliged daze division improvement, which occurs round spout strain rate 24.Four, is weakly uncovered inside the acoustic imprint nearby sideline elements to the spout. At gush pressure rate 13, the acoustic sign saw at shallow components to the spout lessens all of a sudden over a broad extent of frequencies. The end wonder is credited to the game plan of an open-completed subsonic center enveloped through a supersonic annular dissemination inside the push progressed informative spout eventually of free shock division exercises of the spout, which doesn't show in the truncated immaculate shape gush.

P. Padmanathan Et. al, (2012) Trial and numerical assessments are finished to separate the impact of coolant injector format on ordinarily film cooling execution in a different segment of a rocket gush. Two stand-out injector bearings are inspected: (1) shaped spaces with a difference edge of 15° (semi-unique injector) (2) totally exact beginning (completely dissimilar injector). A 2-dimensional, rotate symmetric, multispecies computational model utilizing compelled degree definition has been made and endorsed contrary to the exploratory records. The investigations gave an anticipated relationship of estimations for cooling feasibility for different blowing extents stretching out from 3.7 to 6. Results show that the semi unique arrangement initiates higher sufficiency stood out from totally divergent starting in any regard blowing extents. The spatially landed on the midpoint of reasonability results show that the examination among the two arrangements is significant at better blowing extents. The broadening in sufficiency became cycle 2 % at BR = three.7 in spite of the fact that it became cycle 12 % as a result of BR = 6. Numerical results show the nearness of elective flow distribution zones close to the fly go out for each the injectors. An greater distribution region present because of absolutely specific injector brought about a diffusion in blending of the coolant and popular, and a lower in film cooling execution.

Adamson, T.C Et. al, (1959) Secondary transverse infusion into the unique segment of an axisymmetric focalized unique propulsive spout is tested for the fluidic push vectoring influences. Coupled trial and numerical bloodless-circulation exam on the quantity of cases and angles became led in the shape of a French microsatellite.
launcher software. Five exploratory test models had been deliberate, built, and equipped with demonstrative gadgets. All exploratory test models had been reinforced with the aid of complete 3-dimensional numerical reenactments and in addition tested utilizing the greater spout models, instances, and investigation parameters. Relevant side energy and pitch vector factor of 5–9 deg have been performed in the 5–eight% scope of the optionally available to crucial mass-circulate rate proportion. Examination perspectives, labeled because the spout vectoring framework geometrical attributes, essential and elective circulation situations, and gasoline natural properties were discovered to predominantly impact the frenzy vectoring capabilities. Some in addition upgrades are proposed and completed in the streamlining of selected parameters.

Anderson Et. al, (2012) A spout for quickening compacted gas, preferably air, to supersonic paces which includes meeting, improvement, and fixing parcels characterised by way of a simple combo of circular segments and a line phase for the broadly useful of turning in a supersonic circulate to unearth or take away soil or different like cloth.

Versteeg Et. al, (2009) The preliminary presentation of a fluidic, multiaxis push vectoring (MATV) plot is displayed for an essentially fixed, afterburning gush implied in light of the fact that the conformal fluidic gush (CFN). This thought for stream oversee features symmetric imbuenment over the spout throat to give throttling to fly quarter control, and hilter kilter implantation to subsonically incline the sonic plane for fly vectoring. The determined headway of the CFN transformed into acquired a buddy paper (Miller et al. [1]). In that survey, basic structure factors had been demonstrated to be the overlap term and expansion district portion of the spout, and the area, aspect, and flow of mixed stream. Extents of authenticity had been vectoring limit, web push coefficient, and dispatch coefficient. A show of MATV was driven on a 20 percent scale CFN look at article over an extent of spout pressure extents (NPR), injector stream.

III. Methodology

Computational Fluid Dynamics (CFD) is a structuring gadget that allows experimentation. Its diploma isn't always obliged to fluid components; CFD can be associated with any method which incorporates transport wonders with it. To cope with a structure problem we can make utilization of numerous systems just like the interpretive approach, initial methodologies the use of fashions. The deliberate approach is surprisingly tangled and intricate. The exploratory strategies are extreme. In case any goofs inside the shape had been recognized within the midst of the version trying out, another version is to be made clearing up all of the mistakes and again tested. This is a boring and furthermore a value-ingesting up methodology. The advent of Computational Fluid Dynamics has vanquished this difficulty and changed the sphere of structuring. In CFD an problem is copied in programming and the vehicle conditions related with the difficulty is numerically settled with PC help. Consequently we'd have the ability to check the outcomes of an problem before experimentation. The present work is going for choosing an ideal multiple plot for the
spout which could provide the most preposterous outlet pace and meet the pushed necessities. Stream dangers may be made in the spout due to the method if incapacitates which reduce the leave Mach variety and also push of the motor. Thus we would have the ability to check the results of an trouble before experimentation. The present paintings is going for selecting an excellent extraordinary plot for the spout which could supply the satisfactory outlet speed and meet the pushed necessities. Stream dangers might be made inside the spout due to the development if stuns which decline the depart Mach wide variety and moreover push of the engine. This might be disposed of by way of differing the specific aspect. Here exam has been pushed on spout with various edges 4°, 7°, 10°. Experimentation the use of the models of each extraordinary part is an excessive and additionally a repetitive technique. CFD demonstrates up being a feasible device to triumph over those repressions. Here on this work the example of various stream parameters are likewise investigated.

Table 1: Design Parameters

| Parameter               | Value  |
|-------------------------|--------|
| Inlet diameter          | 25mm   |
| Throat diameter         | 10mm   |
| Exit diameter           | 35mm   |
| Total pressure(bar)     | 44.10  |
| Convergent angle(°)     | 30     |
| Divergent angle(°)      | 15     |
| Total temperature(K)    | 3400   |
| Mass flow rate(Kg/s)    | 826.0  |

Meshing In ANSYS Workbench

After the demonstrating is completed the pass section is to be completed. The module used to carry out coinciding is Fluid Flow (Fluent). The go section technique applied right here is Automatic Method and the paintings kind is selected as All Quad. The governing conditions utilized in work are as in step with the following: It is expected that there may be a one in all a kind, unmarried esteemed connection among the summed up co-ordinates and the physical co-ordinates which can be communicated as

$$\eta = \eta(x,y), \xi = \xi(x,y)$$

this also implies that,

$$x = x(\xi, \eta), y = y(\xi, \eta)$$
The converse change can be composed as pursues:

\[
\frac{\partial T}{\partial x} = \frac{\partial T}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial T}{\partial \eta} \frac{\partial \eta}{\partial x} \quad (eq. 1)
\]

\[
\frac{\partial T}{\partial y} = \frac{\partial T}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial T}{\partial \eta} \frac{\partial \eta}{\partial y} \quad (eq. 2)
\]

The Poisson Equation that is comprehended is of the structure as in the accompanying condition:

\[
\frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = P(\xi, \eta)
\]

\[
\frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = Q(\xi, \eta)
\]

Where P and Q are predefined uction's which can be utilized to control sort out gathering here on this undertaking Meshing take conveyance of a significant business, because of the reality we're picking up impacts by means of fluctuating the Number of divisions in work .The amount of divisions are changed at the vertical surfaces (channel and go out) and the willing surfaces (dividers). The going with adventure went into to work the geometry made. The made geometry was moved in to the cross territory workbench. The work utilized was tetrahedral work. The face-mapped concurring determination was utilized to the geometry with the end objective to maintain a strategic distance from the objectives blunders. The artistic creations got unobtrusive to a radical pondering the utilization of the refinement decision of the workbench. Resulting to cross stage, the straight, the center and as some separation as attainable have been named. This concurred geometry is starting at now outside made to the FLUENT workbench. In the FLUENT workbench the settings made are as grouped:
The work procured from the outset might be unstructured paintings (discern 1) and this will not be used to get specific effects. Since the rims are multicolored the paintings may be modified over into taken care of out go area by using the use of Mapped Face Meshing. The assessment is improved the circumstance five styles of move areas which can be gotten by means of changing the quantity of divisions in work. The assortment of the amount of divisions is performed at the inlet, go out and at the dividers of the spout. Coming up subsequent is the wording that is popular to make connection with the Number of divisions.

**Limit Conditions**

1. Mass stream delta
2. Outlet
3. Dividers

Assurance of the breaking factor zones should be performed in Workbench simplest, as there's no probability to illustrate the farthest factor zones in Fluent. Thusly fitting idea have to be taken at the same time as portraying the point of confinement situations in Workbench. With all of the zones portrayed virtually the work is conveyed to the solver. The solver utilized in this trouble is Ansys Fluent. The sent out paintings record is perused in Fluent for looking after the issue.

**Table 2: Problem setup**

| General                  | Solver type : Density-based |
|                         | 2D Space: Ax-symmetric      |
| Models                  | Energy equation : On        |
|                         | Viscous model : standard k-ε model |
| Materials               | Air-Ideal gas               |
Boundary Conditions

| Description                                      | Value/Condition |
|-------------------------------------------------|-----------------|
| Inlet-mass-flow boundary Enter mass flow rate   | = 826 kg/s      |
| Temperature                                      | = 3400 K        |
| Axis-axis boundary Outlet-pressure outlet boundary condition |                |

Divergent angle = 4°

**Fig. 2:** Divergent Angle 4° at Mach contour

The mach shape of the spout with the special fringe of 4° suggests the development of slanted stun in the multiple vicinity. Over the main stun, the Mach wide variety all of a sudden drops from 2.00 Mach to one.10 Mach. After this the rate of flow once more increments before the following stun occurs. This stun wave delivered is meditated from the dividers of the spout and it shapes any other stun in the disparate area itself. Now, the speed drops from 2.10 Mach to at least one.45 Mach. It very well may be visible that the rate once more increments and it arrives at 2.20 Mach at the exit of the spout. The positions where the stun takes place can be resolved from the Mach Vs role plot as regarded in determine.

**Fig. 3:** Divergent Angle 4° for Mach Vs position

It is determined that stun happens at the location 1m from the gulf vicinity and the stun fashioned due to the impact of the wave is framed at 2m from the delta phase. The velocity quantity is discovered to increment as we move from channel to go out.

*Copyright reserved © J. Mech. Cont.& Math. Sci.*
*Lam Ratna Raju et al*
The speed on the bay is 0.0853Mach (sub-sonic). At the throat section the velocity differs from zero.931Mach to one.04Mach. The velocity at the go out is visible as 2.2Mach (extraordinary-sonic).

**Fig. 4:** Divergent Angle 4° for fluid at Static pressure

In particular, it's miles the weight envisioned when the liquid remains, or very nonetheless. The above determine uncovers the way that the gasoline gets prolonged in the spout go out. The static weight within the delta is seen to be 2.39 e+06 Pa and as we circulate towards the throat there is a discount and the motivation at the throat is located to be 1.67e+06 Pa. After the throat, there's an abrupt increment within the static stress at the hub which suggests the occasion of the stun. After the stun there may be a mild decline inside the weight however it again ascends at the following stun. At that point it diminishes to an estimation of 1.15e+05Pa at the depart segment due to the extension of the liquid towards the go out of the spout.

**Fig. 5:** Divergent Angle 4° with Turbulent Intensity

The gulf phase has low disturbance of the worth 5.12e-02% and it increments closer to the spout. Similarly because the disparate vicinity begins the shape show exceedingly excessive estimation of violent power which is due to the abrupt extension of the circulation into the multiple phase. Here for this situation the stream inside the disparate place is enormously fierce as a result of the association of two stuns within the segment. From the shape the locale of first stun has a choppiness force of four.61e+00% and it thoroughly may be visible that the disturbance wins drastically after the subsequent stun. At that factor it drops to 3.59e+00% at the go away segment.
Divergent Angle = 7°

The variety in the mach form with increment within the disparate part from 4° to 7° may be seen from figure. Here it is perceptible that simply one stun has took place in the diverse vicinity. The gulf vicinity has a velocity of five.74e-02 Mach. At the throat the velocity shifts from 9.77e-01 and 1.11Mach. Over the stun the rate drops from 2.5Mach to 1.5Mach. The speed once more increments towards the exit of the spout. The depart velocity is visible as 2.55Mach alongside the hub of the spout. The varieties in velocity along the dividers of the spout are because of the consistency influences.

Fig. 6: Divergent Angle 7° with Mach contour

Fig. 7: Mach Vs position plot at Divergent Angle 7°

The scenario of stun can be found the mach plot as in fig.3.6 and it's far visible that the main stun occurs at 1.25m from the channel. It is sooner or later observed that the stunt has uprooted with the aid of approximately zero.3m as the particular factor accelerated from four° to 7°. The next stunt is observed to have moved out of the spout.

Fig. 8: fluid at Divergent Angle 7° with Static pressure
The static weight is visible as three.46e+06Pa at the gulf area. The weight dropped to approximately 2.56e+06Pa at the throat section and continues on diminishing to an estimation of 4.84e+04Pa. At the scenario of stun it has increased to at least one.13e+06Pa. At that point the static weight once more drops and it arrives at a low estimation of four.84e+04Pa on the leave region. Contrasted with the past case the estimation of static weight has dropped.

![Fig. 9: Divergent Angle 7° for Turbulent Intensity Contour](image)

Fierce Intensity The tempestuous power shape of the spout is as regarded in the parent At the gulf section, it's miles observed that the violent pressure is extraordinarily less, 5.36e-02%. The tempestuous energy increments because it offers to the throat section. At the throat vicinity it has come to an estimation of 3.63e+00 %. An abrupt increment in the fierce energy is regarded as the stun introduces. At the instatement of stun it has an estimation of about 6.61e+00% and there takes place an abrupt increment in the disturbance at the stunt function. It arrives at the most extreme estimation of one.14e+01% on the stun. At that point the disturbance diminishes, yet at the same time the impact of stun influences the choppiness till the depart location. At the leave, the fierce power is ready 7.21e+00% on the pivot

![Divergent Angle 10°](image)

The mach shape of the spout while the exclusive facet is made 10° is seemed formerly. Here likewise a solitary slanted stunt happens within the specific location. Over the stunt, the Mach variety is discovered to drop from 2.50Mach to approximately 1.75Mach. From the Mach variety Vs position plot (fig3.10), it's miles seen that the stunt occurs at approximately 1.25m from the delta phase. The stunt has
very little uprooted contrasted with the scenario while the disparate point became 7°. Be that as it is able to, the decrease in Mach range is less contrasted with the above case. The leave Mach range is 2.98Mach. Another belief is the lower in flow inversion contrasted with the 7° case. This can be seen from the mach shape near the dividers of the spout.

Fig. 11: Divergent Angle 10° with Mach Vs position

Fig. 12: Divergent Angle 10° Static pressure

Static stress At the bay region the static pressure is 3.50e+06 Pa and it has dropped to an estimation of two.39e+06Pa at the throat segment. After the throat vicinity, the static stress continues reducing and it drops to a totally low worth - 7.06e+02 Pa on the go away segment. An growth in the estimation of static stress is visible which speaks to the stun. The static weight has dropped extensively contrasted with the above cases.

Fig. 13: Nozzle at Divergent Angle 10° with Turbulent Intensity Contour
The violent power at the gulf section is 5.31e-02% that is a very little worth. This worth increments toward the throat region and arrives at 2.18e+00%. The sudden change in go sectional location has prompted a variety in choppiness energy close to the dividers. This disturbance receives subtle because the move sectional quarter increments similarly. Above all, the disturbance esteem suggests an abrupt increment on the scenario of stun. At the stun it has arrived at an estimation of four.91e+00%. A very close perception shows excessive violent power near the go away dividers. This is probably a direct result of the consistency affects

### Table 3: Conditions at exit section

| Case | Angle (deg) | Mach number | Static pressure (pa) | Turbulent intensity (%) |
|------|-------------|-------------|----------------------|------------------------|
| 1    | 4           | 2.20e+00    | 1.15e+05             | 3.59e+00               |
| 2    | 7           | 2.67e+00    | 4.84e+04             | 6.61e+00               |
| 3    | 10          | 2.98e+00    | -7.06e+02            | 4.30e+00               |

### Graph 1: Conditions at exit section different variations

### Table 4: Conditions at throat section

| Case | Angle (deg) | Mach number | Static pressure(pa) | Turbulent intensity(%) |
|------|-------------|-------------|---------------------|------------------------|
| 1    | 4           | 8.26e-01    | 1.91e+06            | 3.09e+00               |
| 2    | 7           | 9.71e-01    | 2.74e+06            | 4.23e+00               |
| 3    | 10          | 9.34e-01    | 2.76e+06            | 3.09e+00               |

Copyright reserved © J. Mech. Cont.& Math. Sci.
Lam Ratna Raju et al
IV. Conclusion

At the leave area, the Mach number is found to increase with increase in interesting side. It is 2.20e+00Mach for four° and it additions to the most extreme important estimation of 4.82e+00Mach for eleven°. So additionally at the throat area in like manner, the Mach wide assortment keeps expanding with increase in disparate point. It has increase from 8.26e+01Mach at four° to 1.25e+00Mach at eleven°. The static weight lessens with improved unique aspect. At four° it got 1.91e+06Pa and it dwindled to an estimation of - nine.32e+004Pa. It was obvious that sideways shocks are framed at some phase in course through the spout. At the factor while the particular viewpoint changed into four°, the dominating shock occurred at 1m from the channel and this wave pondered from the dividers of the spout has expedited each other paralyze at 2m. It become found that the development in disparate factor removes the daze toward the exit of the spout. At the factor when the particular edge changed into made 7° the paralyze become surrounded at 1.25m from cove and the accompanying stagger is executed from the spout As an affirmation of the outcomes got, a variant of the 11° spout can be made and circle representation structures, for instance, schlieren photographic technique might be used for trial of the flow plans.

References

I. Varun, R.; Sundararajan,T.; Usha,R.; Srinivasan,ok.; Interaction among particle-laden under increased twin supersonic jets, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 2010 224: 1005.
II. Pandey, K.M.; Singh, A.P.; CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, ISSN: 2010-0221.

III. Natta, Pardhasaradhi.; Kumar, V.Ranjith.; Rao, Dr. Y.V. Hanumantha.; Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD), International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 2, Issue five, September-October 2012, pp. 1226-1235.

IV. K.M. Pandey, Member IACSIT and A.P. Singh. K.M.Pandey, Member, IACSIT and S.K.YadavK.M.Pandey and S.K.Yadav, —CFD Analysis of a Rocket Nozzle with Two Inlets at Mach2.1, Journal of Environmental Research and Development, Vol 5, No 2, 2010, pp- 308-321.

V. Shigeru Aso, ArifNur Hakim, Shingo Miyamoto, Kei Inoue and Yasuhiro Tani “Fundamental examine of supersonic combustion in natural air waft with use of surprise tunnel” Department of Aeronautics and Astronautics, Kyushu University, Japan , Acta Astronautica 57 (2005) 384 – 389.

VI. P. Padmanathan, Dr. S. Vaidyanathan, Computational Analysis of Shockwave in Convergent Divergent Nozzle, International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 2, Issue 2, Mar-Apr 2012, pp. 1597-1605.

VII. Adamson, T.C., Jr., and Nicholls., J.A., “On the shape of jets from Highly below improved Nozzles into Still Air,” Journal of the Aerospace Sciences, Vol.26, No.1, Jan 1959, pp. Sixteen-24.

VIII. Lewis, C. H., Jr., and Carlson, D. J., “Normal Shock Location in underneath increased Gas and Gas particle Jets,” AIAA Journal, Vol 2, No.4, April 1964, pp. 776-777. Books

IX. Anderson, John D.Jr.; Modern Compressible Flow with Historical Perspective, Third edition, 2012

X. Versteeg. H.; Malalasekra.W.; An Introduction to Computational Fluid Dynamics The Finite Volume Method, Second Edition, 2009.

XI. H.K.Versteeg and W.Malala Sekhara, “An introduction to Computational fluid Dynamics”, British Library cataloguing pub, 4th version, 1996.

XII. Lars Davidson, “An introduction to turbulenceModels”, Department of thermo and fluid dynamics, Chalmers college of era, Goteborg, Sweden, November, 2003.

XIII. Karna s. Patel, "CFD analysis of an aerofoil", International Journal of engineering studies, 2009.
XIV. K.M. Pandey, Member IACSIT and A.P. Singh "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, 2017.

XV. P. Parthiban, M. Robert Sagayadoss, T. Ambikapathi, Design And Analysis Of Rocket Engine Nozzle by way of the usage of CFD and Optimization of Nozzle parameters, International Journal of Engineering Research, Vol. Three., Issue.5., 2015 (Sept.-Oct.).