

CFD Driven Parametric Design of Air-Air Jet Pump for Automotive Carbon Canister Purging

Dhaval Vaishnav, Mohsen Ehteshami, Vylace Collins, Syed Ali, Alan Gregory,
and Matthew Werner
Ford Motor Company

ABSTRACT

A jet pump (also known as ejector) uses momentum of a high velocity jet (primary flow) as a driving mechanism. The jet is created by a nozzle that converts the pressure head of the primary flow to velocity head. The high velocity primary flow exiting the nozzle creates low pressure zone that entrains fluid from a secondary inlet and transfers the total flow to desired location. For a given pressure of primary inlet flow, it is desired to entrain maximum flow from secondary inlet. Jet pumps have been used in automobiles for a variety of applications such as: filling the Fuel Delivery Module (FDM) with liquid fuel from the fuel tank, transferring liquid fuel between two halves of the saddle type fuel tank and entraining fresh coolant in the cooling circuit. Recently, jet pumps have been introduced in evaporative emission control system for turbocharged engines to remove gaseous hydrocarbons stored in carbon canister and supply it to engine intake manifold (canister purging). Naturally aspirated engines use vacuum pressure inside the intake manifold for canister purging. However, turbocharged engines operate at or above atmospheric pressure. Hence, a jet pump is used in which the high pressure compressed air from the turbocharger flows through a nozzle and creates necessary vacuum to facilitate canister purging. This paper describes the CAE driven parametric design process of such a jet pump. Flow velocity through nozzle is often in high subsonic or supersonic regime. Hence, a CAE method needs to consider coupled flow along with local mesh refinements and additional boundary layer cells. For subsonic regime, results from the coupled flow solver were nominally same as less resource intensive segregated flow solver. However, for supersonic regime, the difference in performance was found to be up to 10% due to air compressibility effects. The paper also covers details about the CAE-test correlation. The validated CAE method is employed to understand the effect of numerous geometrical parameters such as: nozzle diameter, area ratio of throat and nozzle, diffuser length, nozzle shape and purge port diameter. Performance curves (purge flow vs. turbocharger pressure) are developed for each of these parameters. Purge flow was specifically found to have a strong dependency on the area ratio and purge port diameter. Optimum area ratio of 0.04 and larger purge port provided significant performance improvement. In conclusion, this paper aims to develop performance curves and explore effect of several geometrical parameters on the performance of the jet pump to add to the existing knowledge base especially the automotive literature.

CITATION: Vaishnav, D., Ehteshami, M., Collins, V., Ali, S. et al., "CFD Driven Parametric Design of Air-Air Jet Pump for Automotive Carbon Canister Purging," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1316.

1. INTRODUCTION

Jet pump is a device with no moving parts which works by transferring momentum from the high velocity primary flow to the secondary flow. The operating principle of jet pumps is the entrainment of surrounding fluid by shear effects. A typical jet pump mainly comprises of a primary (boost) port, a nozzle, a mixing chamber, a suction (secondary) port, a throat and a diffuser.

Jet pumps can be broadly categorized into two types: 1) cross flow jet pump 2) annular flow jet up. As shown in figure 1b and figure 1c, the purge flow entrance is perpendicular and parallel to boost flow in the cross flow jet pump and annular jet pump, respectively. They have been widely used in various industries such as aerospace, petrochemical, refrigeration, etc. [1, 2, 3, 4], due to their simple structure (no rotating part and excellent sealing ability), low packaging footprint and superior cavitation performance. In automotive industry, liquid-liquid jet pumps are commonly used to

facilitate fuel and coolant transfer. Recently, ejectors have found their use in evaporative emission control system to pull hydrocarbons (HC) stored in carbon canister and supply it to engine (canister purging).

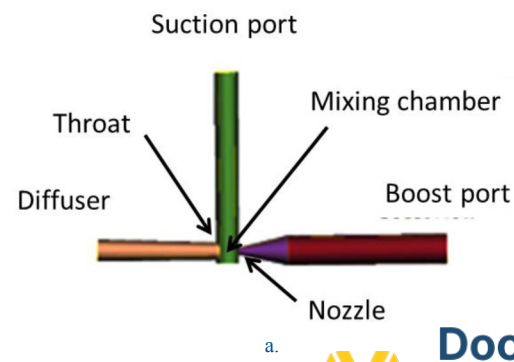


Figure 1. Schematic and configurations of jet pumps: a) typical parts, b) cross-flow jet pump and c), annular jet pump

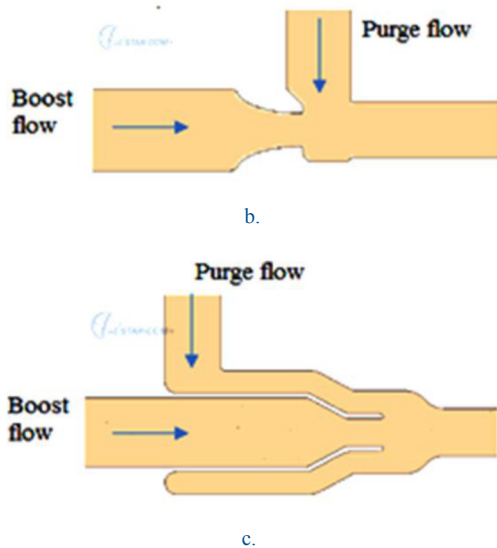


Figure 1. (cont.) Schematic and configurations of jet pumps. a) Typical parts, b) cross-flow jet pump and c), annular jet pump

Figure 2 presents the evaporative emission (EVAP) system including the vent lines/valve, carbon canister, and canister purge valve. EVAP system limits the fuel vapors from escaping into the atmosphere. It transfers the fuel vapors from the fuel tank to a hydrocarbon storage device called the carbon canister. The carbon canister stores the fuel vapors until the engine is able to use it. To regenerate the carbon canister, the intake fresh air flow purges the fuel vapor from the canister to the engine.

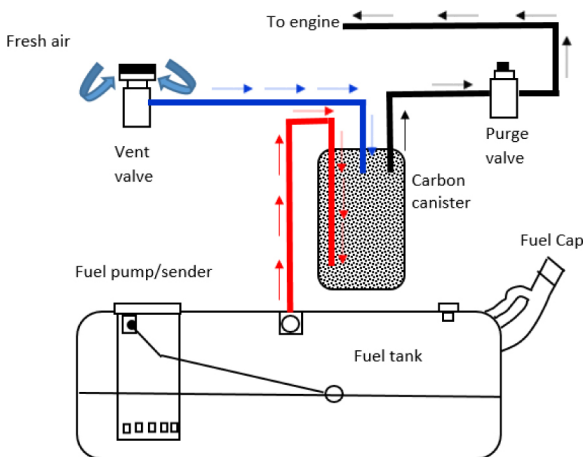


Figure 2. Schematic of EVAP system

For naturally aspirated engines, the vacuum pressure inside the intake manifold is employed to generate flow of HC. However, turbo-charged engines operate at or above atmospheric pressure. Hence, many times the engine manifold is under positive pressure. For such cases, an ejector is used in which the high pressure air from turbo charger flows through a nozzle and creates necessary suction to facilitate canister purging. Figure 3 shows the system diagram for boosted engine EVAP system. The ejector is located after check valve 2. A recirculation line supplies high pressure boost flow from compressor (primary flow) to the jet pump (ejector). The purge port

(secondary outlet) of the jet pump is connected to carbon canister via check valve and CPV (canister purge valve). The discharge from the jet pump is released in the line leading to the compressor.

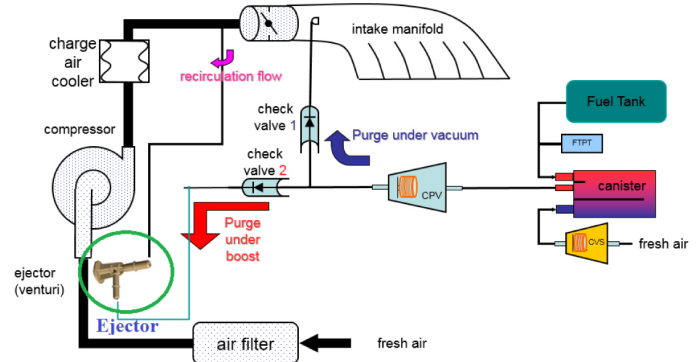


Figure 3. Schematic of a boosted engine purge flow path

For the canister purging application, objective of the design optimization is to maximize the purge flow for the given boost pressure. By maximizing the purge flow, canister purging time can be reduced. A design challenge is the packaging of the jet pump. In order to avoid structural damages, the jet pump is mounted inside the air induction system (AIS) duct which restricts available packaging space. In addition, jet pumps operate at wide range of flow velocities ranging from low subsonic to low supersonic. These considerations should also be taken into account for designing of efficient, robust and reliable jet pumps.

2. LITERATURE REVIEW

Several studies have focused on understanding the hydrodynamics of jet pumps and their design using computational fluid dynamics (CFD). Zhu et al. studied the effects of nozzle exit position, and the mixing section converging angle on the performance of the ejector [5]. They considered a compressible flow and a steady state k-ε turbulence model for their simulation. Their results show that the optimum nozzle exit position is proportional to the mixing section throat diameter and increases with the boost flow pressure. They also found out that the mixing section converging angle influences the ejector performance characterized by entrainment ratio by 26.6%. As the boost flow pressure rises, higher converging angles will be required in order to obtain the highest performance.

Pianthong studied ejectors with constant-pressure mixing and constant-area mixing using CFD techniques [6]. Realizable k-ε turbulence model was used in that simulation and the results obtained were in good agreement with experimental data. Hemidi et al. investigated the operation of a supersonic ejector by applying CFD techniques [7], [8]. They found out there are discrepancies in the local flow features especially at high boost pressures although the predicted entrainment rates were in good agreement with those obtained in experiments over a wide operation range. Furthermore, they proposed a way to predict the choking cross-section based on sonic line location, Mach number field, and operation curve. Fan et al. investigated the effect of an ejector's geometry on its performance using CFD models considering compressible air. They studied the ejector [9]. Key design variables considered were the diffuser inlet and

outlet radii, diffuser and nozzle length, and the nozzle inlet and outlet radii. Their results suggested that inlet radius of the diffuser and nozzle are creating large impacts on the purge flow rate. However, it should be noted that decreasing the diffuser and nozzle too much leads to supersonic flow entering the diffuser resulting in a drop of the ejector performance. Besides, it was found out that the length of nozzle and diffuser does not affect the ejector's performance. The effect of the pressure-based algorithms for solving the flow, being coupled versus segregated approaches, has been controversial [10], [11]. It has been reported that the condition has been in favor of segregated solver [11]. However, the situation has been changing recently. Darwish et al. [10] solved a laminar incompressible flow applying a fully coupled solver on an unstructured grid. They compared the computational cost of the coupled solver against a segregated solver. Their results showed that the CPU time needed for the coupled solver to converge to the solution is significantly lower compared to the segregated solver and the reduction rate increases with the size of the grid. Hanby et al. [12] compared the performance of coupled and segregated solvers for an incompressible swirling flow and found out that the coupled solver provides better convergence for models with dense computational meshes.

To summarize, understanding the effect of geometrical parameters of an ejector on its performance and optimizing the geometry leading to the highest efficiency is of great interest to reduce the time required to purge the canister. This study is aimed at validating the CAE method and conducting parametric design studies for air jet pumps using computational fluid dynamics. Section 3 describes the experimental setup that was used to collect the test data and validate CAE method. In section 4, the CFD model of an air jet pump is described and the results obtained through segregated and coupled solvers are compared against the experimental data. Section 5 covers parametric design studies. The parameters used in this study are: nozzle area ratio, diffuser length ratio, nozzle length ratio and the diameter of the purge tube. Section 6 summarizes the findings of this study.

3. EXPERIMENTAL SETUP

Figure 4 presents the bench test set up used to characterize the performance of the jet pump. The test setup includes two flow meters to measure primary flow (i.e. high pressure boost flow) and secondary flow (i.e. flow through the purge port that connects with carbon canister). The setup also includes a pressure gauge to measure boost pressure. During the test, boost pressure is held constant and resulting primary (boost) flow and secondary (purge) flow rates are measured. The test is repeated for multiple values of boost pressure.

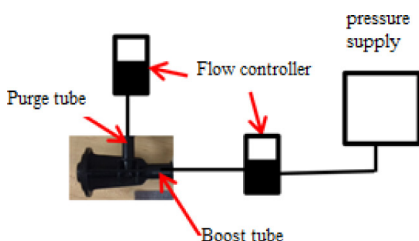


Figure 4. Bench test set up used to characterize the performance of jet pumps

4. CAE METHOD DEVELOPMENT AND VALIDATION

4.1 Governing Equations

The flow in the jet pump is governed by the compressible, implicit unsteady three dimensional form of the fluid flow conservation equations. The energy equation including viscous dissipation is coupled to the Reynolds-averaged Navier-Stokes equation for ideal gas. The governing equations can be presented as follows.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

$$\frac{\partial}{\partial t} (\rho E) + \frac{\partial}{\partial x_i} (u_i (\rho E + P)) = \vec{\nabla} \cdot \left(\alpha_{eff} \frac{\partial T}{\partial x_i} + u_j (\tau_{ij}) \right) \quad (3)$$

$$P = \rho RT \quad (4)$$

$$\tau_{ij} = \mu_{eff} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad (5)$$

Some of the simulations are done for steady state case. For those cases, the time-dependent terms would be equal to zero. For transonic flow regime, the problem forms a stiff system which is inherently difficult to invert. Running in transient mode produces an effective under-relaxation (UR) for the solution. Fundamentally, a steady state and a transient mode solver are the same but with different time-step sizes. The steady state solver considers a very large time-step size which is usually called a pseudo time-step. In the case of transonic flow where large gradients are present, the steady step size may be too large causing instability. This can be alleviated with controlled UR values through solving the problem by transient solver. The turbulence model used is realizable k-epsilon with all wall Y+ treatment models which is based on Boussinesq hypothesis in which an eddy viscosity is assumed. This makes the Reynolds stress tensor proportional to the mean rate of deformation.

4.2 CFD Model

Figure 5 presents three views of the CFD model; the first view shows the jet pump positioned inside a box surface to mimic far field conditions. A mass flow inlet boundary condition is specified at the primary flow (boost flow) port and a pressure outlet boundary condition is specified at the secondary flow (purge flow) port. Based upon the mass flow rate specified at the boost port, suction is created in the mixing chamber which entrains flow from the purge port. The resulting flow exits from diffuser into the AIS duct. The pressure at

the purge port, and box faces are assumed to be atmospheric. No slip condition is considered for all walls. The turbulence intensity at the inlet flow is considered to be 0.1.

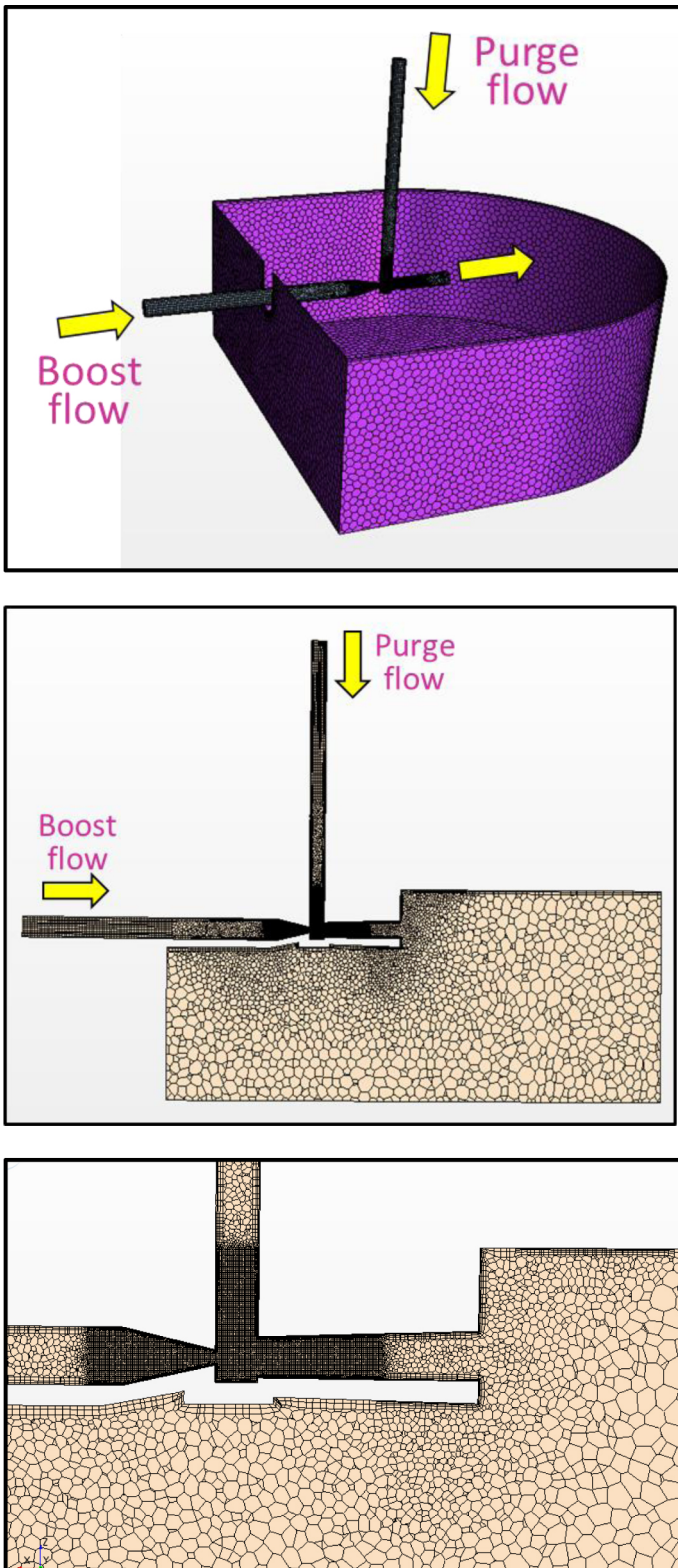


Figure 5. Schematic of the domain considered for the jet pump model

In this study, ANSA and STAR-CCM+ were used for grid generation and CFD simulation, respectively. The mesh size is about one million cells comprising of polyhedral cells (in the core of the jet pump model) and prism layer elements (close to walls). The size of the near wall prism layer is decided to keep the Wall Y^+ less than 10 in order to capture viscous sublayers. Number of prism layers is decided to capture complete boundary layer and achieve smooth transition to core mesh. Polyhedral mesh is preferred over hexahedral mesh for this application as it captures recirculation regions more accurately. Additionally, polyhedral mesh offers benefits of faster convergence with less iteration, robust convergence to lower residual values, and faster solution runtimes. Solution-based approach was also used to achieve efficient refinement by reducing the cell size in areas of interest like the near field of nozzle, mixing bubble, purge tube close to the mixing bubble, and the diffuser.

4.3 Model Validation

In order to substantiate the CFD model results, segregated steady flow and coupled transient flow solvers were applied. The results obtained from the two solvers are presented in figure 6. There is a reasonable agreement between the test results and the simulation results. Figure 6 indicates that the coupled flow solver is able to provide more accurate results compared to the segregated flow solver. The flow within the ejector's nozzle is often transonic or supersonic, where air compressibility changes the density and temperature significantly. In addition, there is a strong coupling between the pressure and temperature in the coupled flow solver; also the conservation of mass, momentum and energy equations are solved simultaneously for each single cell, thereby providing more accurate predictions [13]. However, the coupled solver takes more run time. In order to make it more efficient, solution of the segregated flow solver was used as initial condition for the coupled flow solver.

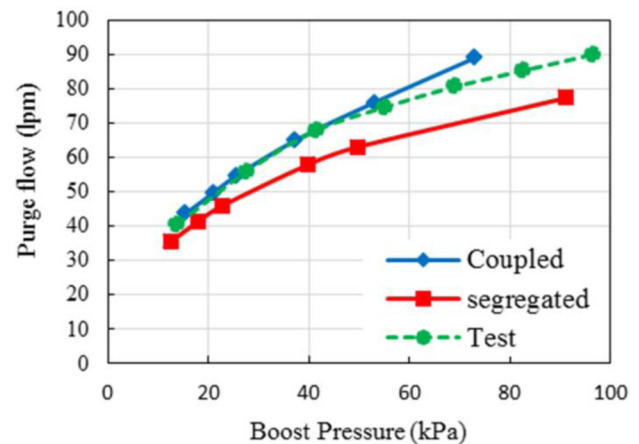


Figure 6. Comparison of test data and simulation data

Figure 7 provides a comparison of boost pressure versus boost flow rate obtained from test and numerical simulation through segregated and coupled solvers. The trend is in agreement with that of figure 6. It can be seen that for the segregated flow the boost flow rate flattens at lower boost pressure (~20kPa). That is explainable by the fact that the Mach number of the flow solved by segregated solver increases at a higher rate compared to that of the one solved by coupled solver. This is attributed to the isothermal nature of the segregated solver.