Aerodynamic design optimization of a bellmouth shaped air intake for jet engine testing purposes and its experiment based validation

Koray Sevinç
TUSAŞ Engine Industries Inc., Eskişehir, Turkey
E-mail: mail@koraysevinc.com

Abstract. Jet engine ground tests are carried out with bellmouth shaped air intakes in order to measure the mass flow rate and the inlet flow quantities while supplying the engine an undisturbed flow. Although there are a variety of design standards and academic guidelines for the design of bellmouths, they can be designed to reach a lower total pressure loss coefficient, a less disturbed radial velocity profiles for the engine and a less thickened boundary layer at the engine inlet. This enhanced design can be achieved with the help of optimization algorithms and computational fluid dynamics (CFD) computations thanks to the increasing average computational power and simplicity of the flow problem. In this study, a genetic algorithm based optimization method and a RANS solver under a commercial software platform are applied to find a bellmouth design with less loss and less disturbed flow. An elliptical bellmouth lip profile with a linear extension part is defined with parametrization and a wide range of parameter space is investigated during the design work. Optimum design candidates are presented comparatively regarding the design purpose. The selected design candidate is produced and tested for CFD validation.

1. Introduction
During jet engine ground tests, or during early component rig tests, a critical need appears to be satisfied which is the measurement of the flow rate, accurately. Accurate flow measurement is essential for characterization of the test article. Intake nozzles in form of bellmouths or Venturi nozzles are used in general for this purpose.

Although the main aim of the bellmouth shaped test intakes is to measure air mass flow rate accurately, their design should be taken with care. One of the performance affecting factors on a jet engine comes from the aerodynamic performance of the engine air intake which have an interface with the compressor unit. The aerodynamic design of the intake is so important that when it is not well handled it may affect the compressor surge margin [1]. Mass flow rate measurements with accuracy of up to 0.5% can be found in the literature [2,3]. In some cases, Venturi nozzles can be preferred over bellmouths for more accurate flow measurements, although they can be less practical. In some cases, bellmouth use can be still preferred although there is a Venturi nozzle installed in the system [4]. This is to supply good quality flow to the system.

When the literature and the industrial standards are investigated, guidelines for bellmouth design can be found. However, this critical part, can be specifically designed and even optimized for the specific jet engine. Considering the literature, bellmouth CFD computations can be found both for 2D and 3D while the literature is short on optimization work of a jet bellmouth intake [1,5-6].
2. Design problem and numerical approach
For a given axial inflow aero engine, a bellmouth shaped intake is required. A base geometry is
dimensioned based on the ANSI/AMCA 210 standard for a given engine diameter. The design problem
is defined by 5 input variables (lip height, lip length, section length, arc radius and arc angle) for a given
range and 3 output variables (loss coefficient, $\zeta$, maximum velocity magnitude, $V_{max}$ and maximum axial
exit velocity, $u_{exit,max}$) to be minimized.

The design problem given, is solved by a commercial CFD solver and its genetic algorithm based
optimizer. The numerical model is kept small assuming the flow to be mostly axisymmetric and less
three-dimensional.

2.1. The base bellmouth shape
In order to decide on the base shape concept of the bellmouth, a literature survey is performed. Based
on the outcome of the study the concept of the shape is fixed. Bellmouth design approaches found in the
literature are discussed under this section.

Idel’chik describes the design of a bellmouth shape based on a combination of circular and linear
sections in the Handbook of Hydraulic Resistance (Figure 1a). Loss coefficient data for a wide range of
$r/D$ (0-0.22) is given in the book [7].

The ANSI/AMCA 210 standard describes a bellmouth design based on a combination of elliptical
and linear sections (Figure 1b). A very narrow design space is given to the designer. The standard
recommends the use of a linear section of length with a variation of only 0.5% of the diameter [8].

A flexible design concept is defined for the base design. The dimensions of each geometry are
defined independently considering the strong optimization algorithm in hand. The initial dimensioning
is made based on the ANSI/AMCA 210. The schematic and convention for the base design is shown on
Figure 2.

![Figure 1. Schematic of bellmouth shapes defined by different sources.](image)

![Figure 2. Schematic of the base design.](image)
2.2. Numerical approach
The design problem is solved with the help of a CFD solver and an optimizer. The details are given in this section.

The 2-dimensional axisymmetric numerical domain is defined by keeping the outer boundaries far from the problem geometry. Similar axisymmetric CFD approaches for intake problems can be found in the literature [5,6]. A typical combination of total quantities at the inlet – static pressure at the outlet is used. Additionally, the outlet static pressure is controlled by the solver to match desired mass flow rate. While the bellmouth boundaries are defined as smooth walls, the extension of the bellmouth towards the large domain’s outlet is defined as free-slip wall. The top boundary of the domain is defined as free-slip wall as well.

Total amount of 130k cells used to discretize the numerical domain by aiming $y^+ \approx 1$ at the no-slip walls (Figure 3). Minimum orthogonal quality is found to be 0.2 while the maximum aspect ratio is around 110. Majority of the cells are quad cells while very few are triangles.

![Image](image1.png)

(a) whole domain (grid-off), (b) close-up view (grid-on),

**Figure 3.** The numerical domain.

As the CFD solver, an unstructured finite volume solver ANSYS FLUENT is used for this study. The axisymmetric flow solver with high order schemes (QUICK) for the advection terms’ discretization is employed. The coupled algorithm is employed to make the pressure based solver solve momentum and continuity equations together [9].

Considering the interest on the boundary layer on this study a k-ω based turbulence model, k-ω SST, is used. The turbulence intensity of 5% and turbulent viscosity ratio of 10 are defined for the incoming flow.

The optimization is handled by ANSYS DesignXplorer. The optimization technique with a Gaussian process regression (Kriging) and multi-objective genetic algorithm (MOGA) is applied for this study (Table 1) [10].

| #Samples/Iteration | 43 | Max. #iteration | 7 |
|--------------------|----|-----------------|---|
| Max. allowable Pareto % | 70 | Mutation probability | 0.01 |
| Converg. Stability % | 2 | Crossover probability | 0.98 |

**Table 1.** Adaptive multiple-objective optimization criteria

The parametrized bellmouth geometry has five input parameters for a given engine diameter, as discussed under section 2.1. Considering the base design parameters are set to the standard, the design space was not needed to be extended too much. Approximately, ±15% of the base design parameter
values are used to create design space. The design space boundaries and the objectives of the optimizer are listed in the Table 2.

Table 2. Design space boundaries and optimizer’s objectives.

| Input Parameters | Min  | Max  | Output Parameters | Objectives |
|------------------|------|------|-------------------|------------|
| Height / D       | 0.6  | 0.9  | ζ                 | minimize   |
| L_{ellipse} / D  | 0.9  | 1.2  | u_{exit,max}      | minimize   |
| L_{line} / D     | 0.9  | 1.2  | V_{max}           | minimize   |
| R_{arc} / D      | 0.1  | 0.2  |                   |            |
| α_{arc} [rad]    | 2.1  | 2.6  |                   |            |

3. Experimental approach

The experimental campaign is carried out to validate the design study. The air is sucked through the bellmouth by using an aspirator where a turbine meter is located in between. The temperature and pressure measurements are carried out at the test chamber and inside of the turbine meter. Once the air flow rate is kept constant, wall static pressure values are recorded through differential pressure transducers. The experimental setup schematic is shown on Figure 4.

A special attention is paid to the production of the wall static taps. The taps are manufactured by drilling and counter bore operations, allowing the wet surface to keep its natural form in the flow path as this approach is guarantees the best measurement in case the it is machined burr-free [11].

4. Results and discussion

An extensive numerical design work with experimental validation is carried out. The optimization outputs are investigated and compared under this section. The optimizer is guided with three objectives but the decision on the desired design is made also based on some field data and flow behavior.

4.1. Mesh independency study

In order to proceed further with the numerical computations, a mesh independency study is carried out. By doubling the mesh count, three different grid resolutions were investigated through four essential performance parameters including three design objective function parameters. Considering errors ranging between 0.004-0.476%, medium dense grid is selected for the optimization study (Table 3).
Table 3. Mesh independency study.

| Mesh# | C_d [-]   | Err. [%] | ζ [-]   | Err. [%] | u_exit,max [-] | Err. [%] | V_max [-] | Error [%] |
|-------|-----------|----------|---------|----------|--------------|----------|----------|----------|
| 68k   | 0.978856  | -0.021%  | 0.0213447 | 2.099%   | 1.00575      | 0.189%   | 1.0388   | 0.055%   |
| 133k  | 0.979021  | -0.004%  | 0.0210053 | 0.476%   | 1.0036       | -0.025%  | 1.03843  | 0.019%   |
| 283k  | 0.979064  | -        | 0.0209058 | -        | 1.00385      | -        | 1.03823  | -        |

4.2. Optimization outputs
The optimizer is asked to list 8 optimum candidates out of all design points (DPs) so that the final decision can be made in detail by the designer not only by comparing the three output parameters. The optimization run data is shown on Table 4. Although the number of maximum iterations were set to 7, the optimization has converged in 4 iterations for the given target.

Table 4. Optimization run data.

|                | Pareto % | #DP | Stability % | # Failed DP | # Iterations | # Candidates |
|----------------|----------|-----|-------------|-------------|--------------|--------------|
|                | 70       | 202 | 1.87        | 3           | 4            | 8            |

After the candidate design points are reported by the optimizer, each of the designs investigated in more details. A general performance parameters chart is prepared including three main objectives and additionally; discharge coefficient, C_d, non-dimensionalized displacement thickness, δ*, non-dimensionalized momentum thickness, θ and shape factor, H (Table 5). A similar approach on duct performance characterization can be found in the literature [12].

The parameters ζ and C_d give idea about overall performance of the intake while the rest of the parameters talk more about the boundary layer behavior and local velocity peaks.

Most of the parameters are quite close to each other. The most varied parameters can be listed as u_exit,max and V_max. These parameters show that DP110, DP122, DP141, DP148 and DP162 are relatively behind their competitors. In other words, DP6, DP124 and DP107 appear to be more desirable.

Table 5. General performance parameters for candidate design points.

| Parameters | DP110 | DP162 | DP122 | DP6 | DP124 | DP141 | DP107 | DP148 |
|------------|-------|-------|-------|-----|-------|-------|-------|-------|
| ζ          | 0.0204| 0.0209| 0.0205| 0.0209 | 0.021 | 0.0211| 0.0212| 0.0206 |
| u_exit,max  | 1.0037| 1.0043| 1.0043| 1.0034| 1.0043| 1.0048| 1.0035| 1.004  |
| V_max       | 1.0464| 1.038 | 1.0424| 1.0382| 1.0354| 1.0532| 1.0364| 1.0423 |
| C_d         | 0.9792| 0.9788| 0.9791| 0.9788 | 0.9788 | 0.9788| 0.9787| 0.979  |
| δ*          | 3.80E-03| 3.90E-03| 3.82E-03| 3.86E-03| 3.90E-03| 3.89E-03| 3.90E-03| 3.84E-03 |
| θ           | 2.53E-03| 2.61E-03| 2.55E-03| 2.58E-03| 2.61E-03| 2.61E-03| 2.62E-03| 2.56E-03 |
| H           | 1.5    | 1.495 | 1.5    | 1.495 | 1.495 | 1.489 | 1.492 | 1.498 |

The Pareto front for the optimization study is shown on Figure 5. The candidate designs are concentrated at a point as it can be seen from the graph. Due to the quite close output parameters variation of the candidates, the finalist candidate is selected by not only considering the general performance parameters but also by going through the field data. The finalist design point, DP6, is marked in red colour and the selection process is explained in the following part.
In addition to the general parameters, wall static pressure distribution on the intake is quite interesting. Because this graph shows how the flow sees the bellmouth and how it reacts near surface (Figure 6). The flow will accelerate up to the throat of the bellmouth and then with a sudden short deceleration it will be stabilized with a growing boundary layer up to the exit station of the bellmouth.

When the plots are compared, DP141 and DP110 appear to be the most varied designs. DP110 starts to accelerate a bit late and that causes a lower minimum pressure than most of the other designs. On the other hand, DP141 immediately starts to accelerate also with a relatively steep slope which causes to reach minimum pressure. Both two designs seem to be risky due to abrupt pressure rise towards the exit.
due to reaching a lower minimum than the other designs. While the rest of the designs are very similar to each other.

Another parameter to consider is the axial velocity distribution along the centerline. The velocity distribution along the centerline two characteristics with the concave and the convex part (Figure 7). DP141 again appears to be the most aggressive design point while the rest are quite similar. This distribution motivates to check the behaviour of its derivative for a better comparison.

![Figure 7. Centerline axial velocity distribution.](image)

Figure 8 shows the centerline axial velocity gradient distribution for the design points. DP141, DP110, DP122 and DP148 shows the highest velocity gradients. These design points can be avoided due to abrupt change in behaviour.

![Figure 8. Centerline axial velocity gradient-x distribution.](image)

Figure 9 and Figure 10 show the bellmouth exit axial velocity profiles; whole profiles and the close-up views of the profiles. Figure 9 shows that the boundary layer growth is up to 5% of the radius while most of the profiles seem very similar to each other. A closer look at Figure 10 shows the difference. It
can be seen that the weak disturbances in the flow go up to 50% of the radius for most of the designs. It should be noted that the error on the maximum axial velocity at the exit was 0.025% with regard to the fine grid, for this reason on this scale level comparison of the designs is still valid.

Based on the close-up view DP6, DP107 and DP110 has the least overshoot which makes them more desirable in this perspective considering that a less disturbed flow at the engine inlet is beneficial. Considering the previous comparisons, DP110 appears to be an undesirable design when compared to the others. Removing DP110 from the final candidates leads us to a decision between the remaining design candidates; DP6 and DP107. When Figure 10 is considered for the remaining designs, it can be observed that DP6 reaches unity before DP107 does. It means that DP6 has more developed and stable boundary layer when compared to DP107. For these reasons DP6 is chosen over the rest, succeeding to the experimentation.

Figure 9. Intake exit velocity profile.

Figure 10. Intake exit velocity profile (Close-up).
4.3. Experiment based validation results

The experimental campaign was carried out for the maximum aspirator power, although it was not capable of increasing the air flow rate up to the CFD flow rate. For that reason, an additional computation is carried out for the maximum flow that was reached during the experiments in order to achieve validation study.

Wall static pressure measurements are carried out at ten axial locations multiplied by two circumferential locations making twenty static pressure measurements.

![Figure 11. Intake wall static pressure distribution.](image)

The pressure distribution along bellmouth wall is plotted against CFD results on Figure 11. The error bars with 0.5% total experimentation uncertainty are plotted, showing CFD results staying well within the experimental data.

There is one clear behavioural discrepancy between the experimental data behaviour and the computational data behaviour at the station just before the bellmouth exit, at 0.85 normalized length location. This slight increase of static pressure in the experiments might be sourced by the stagnation field of six probes, which are located near 0.1 normalized length location. Besides this behaviour, the computational data and experimental data are in good agreement.

5. Conclusion

Bellmouth shaped test intakes are used for jet engine testing purposes in order to measure the air flow rate while supplying uniform air to the engine. There are design guidelines for bellmouth shapes both in the literature and in the standards. However, a more tailored intake design can be made by using today’s computational capacity, current CFD and optimization algorithms. Considering the problem to be an axisymmetric flow problem, it is just a brief work for today’s engineer.

In this study an elliptical concept bellmouth geometry is parametrized and modeled as an axisymmetric flow CFD problem within a multi-objective genetic algorithm based optimizer. The designs are examined through the essential parameters: $\zeta$, $u_{exit,max}$, and $V_{max}$. So, the optimizer’s task was to minimize total pressure loss coefficient, maximum axial velocity at the bellmouth exit and maximum velocity magnitude in the computational domain.

Based on the optimization run, 8 candidate design alternatives are obtained out of 200 design points. The designs are investigated in detail both with overall parameters and flow quantity distribution data.
In order to achieve comprehensive investigation and comparison of the final design candidates, additional global parameters such as boundary layer parameters ($\delta^*$, $\theta$ and $H$) are calculated and field data profiles plotted against each other. Design point 6 was chosen at the end of the investigation and succeeded to experimental work.

Experimental study was carried out for a lower mass flow rate than the target in the optimization study due to aspirator capacity limitation. For that reason, the numerical computation of the DP6 is repeated for the corresponding flow rate and validation is done through axial static pressure profile comparison. CFD data stayed well in the uncertainty range of the experimental data.

This study covers a way of approaching to design of a jet engine test intake in terms of geometrical parametrization, specification of design criteria for an optimization algorithm and for a detailed investigation technique including validation of the computational study. Considering the potential of the applied technique, a very similar approach can be applied to jet engine platform ducting and intake design problems as well where an optimization work cannot be avoidable.

Acknowledgements
The author wish to express thanks to TUSAŞ Engine Industries (TEI) for their support.

References
[1] Zierer T and Matyschok B 1997 Design, development and verification of gas turbine gt24 air intake ASME 1997 Turbo Asia Conference
[2] Fite E B 2006 Fan performance from duct rake instrumentation on a 1.294 pressure ratio, 725 ft/sec tip speed turbofan simulator using vaned passage casing treatment NASA TM-2006-214241
[3] Smith S C 1985 Airflow calibration of a bellmouth inlet for measurement of compressor airflow in turbine-powered propulsion simulators NASA-TM-84399
[4] Brandstetter C, Pages V, Duquesne P, Paoletti B, Aubert S and Ottavy X 2019 Project PHARE-2 – a High-Speed UHBR Fan Test Facility for a new Open-Test-Case ASME. J. Turbomach. 141(10)
[5] Beale D K, Hand T L and Sebourn C L 2001 Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities 37th Joint Propulsion Conference and Exhibit
[6] Sebourn C L 2002 Computational Investigation of the Discharge Coefficient of Bellmouth Flow Meters in Engine Test Facilities PhD thesis The University of Tennessee
[7] Idel’chik I E 1966 Handbook of hydraulic resistance
[8] AMCA 1999 Laboratory methods of testing fans for aerodynamic performance rating ANSI/AMCA 210-99 ANSI/ASHRAE 51-1999
[9] ANSYS Inc 2016 ANSYS FLUENT Version 17.2 Theory Guide
[10] ANSYS Inc 2016 ANSYS DesignXplorer Version 17.2 User’s Guide
[11] AGARD 1990 Recommended practices for measurement of gas path pressures and temperatures for performance assessment of aircraft turbine engines and components
[12] Ho S S H 1990 Subsonic inlet duct flows PhD thesis University of Salford