Gordon P. Blair, Senior Associate, Prof. Blair & Associates and W. Melvin Cahoon, Senior Engineer-Specialist, Volvo Penta of the Americas, discuss how to optimise the design of an engine air intake bellmouth and the use of advanced software as an aid to the process.

Best bell

T
he design of a bellmouth at the end of the intake tract of a reciprocating internal combustion engine is not a topic that has ever occupied much space within the pages of the technical literature. One could come to the not unnatural conclusion that it cannot be a topic of any real significance. That viewpoint, right or wrong, is very much at odds with the efforts made by the designers of the nacelles for aircraft gas-turbine engines who put much experimental and theoretical effort into the design shape of the leading edge of their engine pods.

Speaking personally (writes Blair), I have always been curious about the proper design method for intake bellmouths, indeed I have been known to dangerously pontificate about it; “dangerously” in the sense that my real experimental or theoretical knowledge of that design process is “dangerously” inadequate. Behind the writing of this paper, with the modern availability of computational fluid dynamics (CFD) and with the expert efforts of my co-author using the FLUENT code [1], not only can real design information be provided on the topic but also our mutual curiosity has been satisfied. We present this work both for your interest and for numerical assimilation into your design systems.

DERIVATION OF DISCHARGE COEFFICIENTS

The effectiveness of the flow regime at any boundary at the end of a pipe in an engine is expressed numerically as a ‘discharge coefficient’, i.e., a Coefficient of Discharge or CD. In history, and even today, they were measured experimentally using a steady flow rig, much as shown in Fig.1.

The pipe-end boundary under examination, in this case an intake bellmouth, is placed before a settling tank/plenum and a steady flow is sucked through it into the tank by a vacuum pump. Typically, most production/commercial rigs like this will induce a tank pressure some 28 inches of water below atmospheric pressure, which is a pressure ratio, PR, of some 1.07. As you will find, the numerical value of the discharge coefficient is a considerable function of pressure ratio and, as many pipe end boundaries are exposed to pressure ratios up to the sonic flow condition (where PR is virtual 2), a rig which can generate a maximum pressure ratio of just 1.07 is just not adequate. However, as an intake bellmouth is normally exposed to pressure ratios of 1.1 or less, this type of commercially-available experimental rig would suffice for that purpose. At The Queen’s University of Belfast, at an earlier point in history, we had quite superb experimental facilities and could measure most pipe end boundary conditions up to the sonic threshold PR values approaching 2.0 [2,3].

Traditionally, in the literature, one measured the mass flow rate, as seen in Fig.1, and noted that as mdot (g/s). Some rather carelessly computed it merely as a volume flow rate. In the particular case of the bellmouth, the researcher then computed a theoretical mass flow, t

mdot (g/s), through the pipe area Ap, at the experimental pressure ratio PR, i.e., Po/Ps, using some such theory as the “St Venant” equation, or other subsonic nozzle theory. That researcher then conventionally quoted the discharge coefficient CD as the ratio of the measured to the theoretical mass flow rate, i.e., mdot/mdot.

A useful number, maybe, but one that is well-nigh useless for application into an unsteady flow simulation of the flow in a real engine, assuming that one expects accuracy from that computation. Some readers may well be alarmed to read that you can still pay megabucks for a theoretical engine simulation that precisely uses that particular approach. We will not belabour you with a full description of why there is a correct/incorrect way to derive CD values, which are accurately applicable within an engine simulation for it has been thoroughly covered already [2,3].

The “correct” way to derive CD values is exemplified in Figs.2-4, in conjunction with Fig.1. In Fig.1, the illustrated theoretical contention is that the flow will form a ‘vena contracta’ area Ac inside the pipe somewhat less than the full pipe area Ap. The ‘actual’ discharge coefficient CD is then defined as Ac/Ap and a theoretical analysis must be created to compute that Ac value at any given pressure ratio [2,3].

To illustrate the reality of this flow regime, Fig.2 shows the outcome of a computation by the FLUENT CFD code for the case of inflow into a plain pipe end, with the experimental or theoretical knowledge of that design process is “dangerously” inadequate. Behind the writing of this paper, with the modern availability of computational fluid dynamics (CFD) and with the expert efforts of my co-author using the FLUENT code [1], not only can real design information be provided on the topic but also our mutual curiosity has been satisfied. We present this work both for your interest and for numerical assimilation into your design systems.

Fig.2 Thermodynamics of flow into a plain pipe end

Fig.3 Temperature flow profiles into a plain pipe end

This is neither a simple nor a straightforward computation process. Firstly, the theoretical equations are all non-linear polynomial functions of pressure, temperature, density, and particle velocity. No single solution is a direct solution, but the mathematician must continuously vary the value of Ac in the computation until a unique value of Ac produces precisely the measured values of Ps and mdot. The iterative process to get there is not for the mathematically faint-hearted.

“I have been known to dangerously pontificate about intake bellmouths”

Fig.1 Measurement of Discharge Coefficients

Fig.2 Thermodynamics of flow into a plain pipe end
SPECIAL INVESTIGATION: DESIGN OF AN INTAKE BELLMOUTH

THE BELLMOUTH DESIGNS TO BE ANALYSED

In Fig.5 is sketched, to scale, the bellmouths which will be analysed by the FLUENT CFD software. The first is a simple semi-circle wrap-round radius installed at the end of the pipe. The second is a bellmouth with an aerodynamic profile (NACA type) and the third is a bellmouth with an elliptical profile [5].

All bellmouths are characterised by their basic data for "Type", length L, exit diameter De, entry diameter Di, and entry corner radius Rc. The "Type" can be a sharp edged plain pipe (PP), a simple radius (RAD), an aerodynamic profile bellmouth (AER), or an elliptical profile bellmouth (ELL). A wide range of dimensions for all such bellmouths were tested and most of the more significant ones are reported upon below. Before that point in the discussion, look at Figs.6 and 7, which were tested and most of the more significant ones are reported upon below. Before that point in the discussion, look at Figs.6 and 7, which show the computed Mach number (particle velocity) plots for the simple radius (RAD-46-23-35-6 of Fig.5) and the elliptical profile (ELL-23-23-49-3 of Fig.5). The simple radius in Fig.6 shows less of the pronounced vena contracta so evident in Fig.4 for the plain pipe, but the elliptical profile in Fig.7 has almost no vena contracta so smooth is the flow entry. A more fundamental message, reflecting the increasing interest in fluid mechanics of the 3D flow and the analysis of its output data to produce accuracy of engine simulation when the attained CD values are re-employed to help compute pipe end boundary conditions [2].

THE MEASURED MASS FLOW RATE

In the discussion thus far, there is reference to a measured mass flow rate (mdot) into the bellmouth when actually it is really referring to a mass flow rate as computed by the computational code FLUENT modelling the bellmouth attached to a settling tank as shown in Fig.1. In short, the CFD code is modelling the intake bellmouth and the entire apparatus as a replica for an actual experiment with real hardware instead. Is this justified?

At The Queen's University of Belfast (QUB), much experimental work was conducted in this area (2,3) and one series of experiments did measure the inflow of air into a plain ended pipe and a simple radiused bellmouth, the physical dimensions of which were identical to those described here as PP-46-23-30-0 and RAD-46-23-35-6. It was conducted as a final-year project by a most capable student, H.B. Lau [3]. In short, we can now directly compare the CD values as measured by Mr Lau and as computed by FLUENT. They are shown in Fig.8.

The correspondence between the measured and CFD-computed CD values are very close both numerically and as a trend with pressure ratio. You will observe that the CD values are indeed a considerable function of pressure ratio. You will also note that at QUB we could not exceed an experimental pressure ratio of about 1.3 even though the apparatus at QUB had a flow capacity at least 5 times more than most commercially available flow rigs. However, for an intake bellmouth more sophisticated bellmouth case with an elliptical profile, the CD is 0.743 and the measured mass flow rate is 36.15 g/s. The fundamental message is that there is a considerable benefit in either CD (27%) or mass flow rate (16%) by the addition of even a simple radius at a pipe end to make a bellmouth, but the gain in CD above that simplicity to an optimum may be only 4% more.

To illustrate the potential effect on engine performance, as air mass flow breathed is potentially engine torque produced, in Fig.12 is presented here, the corner radius Rc can be usefully designed as 0.08 mm. The visualisation problem is rectified in Fig.11, where the change in CD for all profiled bellmouths is expressed as a percentage over that for the simple radius bellmouth. It becomes apparent that the improvement in CD is very much a function of the entry diameter Di and is less of a function of either the profile or the length L, which rationale is echoed by the colour and symbol coding of the several graphs. This is but a small selection of all of the bellmouths studied but showing them all would merely provide further confusion, not enhanced clarity.

While there is not much in it, the elliptical profile comes out as the winner over the aerofoil profile. In the all-important pressure ratio PR range up to 1.1 one can conclude that the best bellmouth has an advantage in CD terms of some 3.5% over the simplest bellmouth. In design terms, one can usefully conclude that "short and fat" is best with an optimum length criterion of one diameter De, and an optimum entry diameter Di of some 2.13 times the exit diameter De, and with an elliptical profile. Although the investigations are not presented here, the corner radius Rc can be usefully designed as 0.08 mm times the entry diameter Di.

To illustrate the potential effect on engine performance, as air mass flow breathed is potentially engine torque produced, in Fig.12 is presented here, the corner radius Rc can be usefully designed as 0.08 mm. The visualisation problem is rectified in Fig.11, where the change in CD for all profiled bellmouths is expressed as a percentage over that for the simple radius bellmouth. It becomes apparent that the improvement in CD is very much a function of the entry diameter Di and is less of a function of either the profile or the length L, which rationale is echoed by the colour and symbol coding of the several graphs. This is but a small selection of all of the bellmouths studied but showing them all would merely provide further confusion, not enhanced clarity.

While there is not much in it, the elliptical profile comes out as the winner over the aerofoil profile. In the all-important pressure ratio PR range up to 1.1 one can conclude that the best bellmouth has an advantage in CD terms of some 3.5% over the simplest bellmouth. In design terms, one can usefully conclude that "short and fat" is best with an optimum length criterion of one diameter De, and an optimum entry diameter Di of some 2.13 times the exit diameter De, and with an elliptical profile. Although the investigations are not presented here, the corner radius Rc can be usefully designed as 0.08 mm times the entry diameter Di.

To illustrate the potential effect on engine performance, as air mass flow breathed is potentially engine torque produced, in Fig.12 is presented here, the corner radius Rc can be usefully designed as 0.08 mm. The visualisation problem is rectified in Fig.11, where the change in CD for all profiled bellmouths is expressed as a percentage over that for the simple radius bellmouth. It becomes apparent that the improvement in CD is very much a function of the entry diameter Di and is less of a function of either the profile or the length L, which rationale is echoed by the colour and symbol coding of the several graphs. This is but a small selection of all of the bellmouths studied but showing them all would merely provide further confusion, not enhanced clarity.

While there is not much in it, the elliptical profile comes out as the winner over the aerofoil profile. In the all-important pressure ratio PR range up to 1.1 one can conclude that the best bellmouth has an advantage in CD terms of some 3.5% over the simplest bellmouth. In design terms, one can usefully conclude that "short and fat" is best with an optimum length criterion of one diameter De, and an optimum entry diameter Di of some 2.13 times the exit diameter De, and with an elliptical profile. Although the investigations are not presented here, the corner radius Rc can be usefully designed as 0.08 mm times the entry diameter Di.
The best bellmouths are those that are short and fat with the elliptical profile. The special investigation: design of an intake bellmouth.

“ Whereas a 1D simulation will take but minutes to complete, a 1D-3D co-simulation can take days”

“Whereas a 1D simulation computes crankangle by crankangle, but normally using what is theoretically described as a 1D (one-dimensional) procedure. In recent times, and with the advent of more sophisticated 3D (three-dimensional or CFD) codes, it is possible to co-simulate where elements of an engine ducting are segregated and computed by FLUENT (say) and the remainder of the engine ducting and cylinders are computed by the 1D engine simulation. The 1D engine simulation then feeds the instantaneous thermodynamic state and gas dynamic conditions to the CFD computation at either end of the segregated region and similarly receives updated instantaneous data in return with which to continue its 1D calculations. This is referred to as “co-simulation” and is a most powerful tool to examine regions of an engine ducting where the 1D simulation is theoretically weak, such as at branches in pipes or at an exhaust collector junction where the flow is decidedly three-dimensional.

These computations are best conducted on high performance computer workstations. Whereas a 1D engine simulation will take but minutes to complete on a modern fast PC, a 1D-3D co-simulation can take many hours, even days, to conclude.

In this case, we have prepared a co-simulation by FLUENT for the entire bellmouths including a short segment of the intake pipe beyond which is theoretically described as a 1D (one-dimensional) procedure. During the cycle [2]. That, of course, is what an accurate engine simulation computes, crankangle by crankangle, but normally using what is theoretically described as a 1D (one-dimensional) procedure.

The CFD analysis of the flow by FLUENT [1] and the subsequent analysis of that flow computation to determine the CD coefficient [2,3] is conducted under steady flow conditions, just as if it was experimentally executed on a flow bench. However, the actual bellmouth is placed on an engine which breathes most unsteadily.

The CD PR equations will be applied into an accurate engine simulation, the CFD analysis of the flow by FLUENT [1] and the subsequent analysis of that flow computation to determine the CD coefficient [2,3] is conducted under steady flow conditions, just as if it was experimentally executed on a flow bench. However, the actual bellmouth is placed on an engine which breathes most unsteadily.

In this case, we have prepared a co-simulation by FLUENT for the entire bellmouths including a short segment of the intake pipe beyond which is theoretically described as a 1D (one-dimensional) procedure. During the cycle [2]. That, of course, is what an accurate engine simulation computes, crankangle by crankangle, but normally using what is theoretically described as a 1D (one-dimensional) procedure.
SPECIAL INVESTIGATION: DESIGN OF AN INTAKE BELLMOUTH

Fig.14a Dynamic particle inflow during co-simulation

Fig.14b Dynamic particle ‘spillback’ during co-simulation

Fig.15 CD variations for bellmouth edge geometries

Fig.16 Rectangular bellmouth design

Fig.17 Loss of CD by a rectangular bellmouth

“There is a message here for those who install fuel injectors pointing into intake bellmouths”

The steady flow CFD analysis is conducted with three similar elliptical profile bellmouths but one has our common ‘full-radius’ as seen in Fig.5 (Rc is 3 mm), another has a full ‘half-radius’ of 3 mm that folds right back to the outside of the bellmouth. Yet another has no radius at all but has a sharp-edged pipe end at the (common to that folds right back to the outside of the bellmouth. Yet another has a full ‘half-radius’ of 3 mm that forms a toroidal vortex (implied ring) at the pipe end. Such a phenomenon has been seen and photographed before in high speed Schlieren image experiments conducted at QUB more than thirty years ago [2, pp 154-157]. There is a message here for those who install fuel injectors pointing into intake bellmouths; use a ‘short and fat’ bellmouth to reduce the spillback of fuel for it is the spillback of air which propels it.

ANSWERING THE OBVIOUS QUESTIONS

At this point many a reader will be saying, “… is that it? …” and forming a question beginning with “… what if… ? …”. To forestall many an e-mail, we have examined a couple of such cases. The first obvious question may well relate to the “wrap-round” radius Rc. Is it necessary? The second obvious set of questions will doubtless relate to the oil-used rectangular intake duct shape as seen in Fig.16. In a four-valve head design it is somewhat difficult to smoothly connect the twin intake passages at each of the intake valves into a single round intake duct and often a rectangular intake duct is considered the effective compromise. But is it, especially if it leads to a rectangular bellmouth? In Fig.16 is seen a photo of just such a bellmouth and above it the CFD geometric model to assess its CD characteristics by FLUENT [1].

The rectangular duct geometry used has an aspect ratio of 2:1 with four corner radii each of 6 mm and the width and height are 29.878 and 14.939 mm, respectively, giving the same area as the round 2.5 mm pipe used for all previous CFD flow experiments’. The bellmouth is created by a simple 6 mm radius around the perimeter, making it the rectangular equivalent of the round pipe RAD-46-23-35-6. This rectangular bellmouth is labelled as REC-29.878_14.939-6.

The FLUENT CFD computations are run at varying pressure ratios up to 1.7 and the CD values are analysed at each PR from the mass flow data determined by CFD [2,3]. The results of these calculations are seen in Fig.17, but they might have been anticipated by reading almost any text on fluid mechanics [8] or studying the experimental observations on the loss-creating vortices at the entry corners to rectangular pipes [9]. Any such tests will show that the hydraulic radius, conventionally calculated as ‘area/wetted perimeter’, for the 23 mm round pipe is D/4 or 5.75 mm but that for our selected rectangular pipe is 5.24, a loss of some 9%. However, in Fig.17, the loss of CD for the rectangular pipe is computed through FLUENT as no worse than 0.83% at the lowest pressure ratio. The reality of an actual CD steady flow measurement might show greater losses in CD for the rectangular bellmouth as the corner vortices seen by Schlichting [9], with their inherent rotational turbulence characteristics, always provides computational difficulties for any CFD code.

Quite irrespective of the above caveats, the computed CD results for the simple rectangular bellmouth are worse than for that for the simple radius RAD-46-23-35-6 and, as a glance at Fig.10 will confirm, this simple radius bellmouth was a numerical step below all of the profiled bellmouths. As it is rather difficult to design a rectangular profiled bellmouth, it will inevitably have a rectangular entry, the general conclusion must be that rectangular intake ducts and rectangular intake bellmouths should be avoided by design if at all possible.

The flow regime has the asymmetric fluid mechanic difficulties described in the literature [8,9] is confirmed in Fig.18, where the computed (particle velocity) Mach number profiles across each axis and at a section 6 mm inside the entry are illustrated.

REFERENCES


“The computed CD results for the simple rectangular bellmouth are worse”