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

The 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 here 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/are 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 of air 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 virtually 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,

> "I have been known to dangerously pontificate about intake bellmouths"

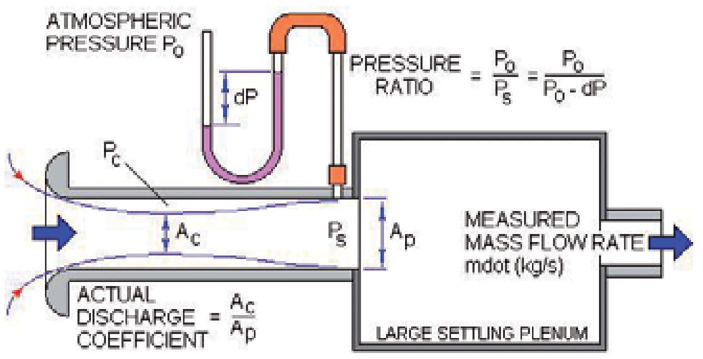

Fig.1 Measurement of Discharge Coefficients

tmdot (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/tmdot.

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' of 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 the sharp-edged plain pipe from the atmosphere (at 1.0 atm and 25 degC or 298 K). The Fig.2 shows the computed contours of pressure for the flow process. That there is indeed a 'vena contracta' is evident by the pressure drop from the entry at Po to Pc followed by pressure recovery to Ps at the pipe exit. It is worth mentioning that the pressure ratio PR is indeed Po/Ps and that Ps value is determined as an average

> "This is neither a simple nor a straightforward computation process"

of the pressure in all of the CFD computation cells across the duct at the 'Ps measuring station' and is not simply the atmospheric pressure divided by the plenum pressure. This average computation pressure Ps corresponds precisely with that obtained in an actual experiment by the 'manometer pressure gauge' shown in Fig.1.

Further evidence is given in Fig.3 by the computed temperature contours for the same process with a drop of 8 degC to the 'vena contracta' and a recovery of 3 degC by the end of the pipe. The most significant evidence is provided by Fig.4, showing nozzle-like flow from zero velocity at the entry to a high particle velocity (as Mach number) of 0.3 at the vena contracta, a still region surrounding it, and a reduction of velocity with diffusion to the pipe exit. That being the case, then that is how it must be theoretically analysed in order to derive a realistic discharge coefficient.

The theoretical situation is sketched in Figs.1 and 2. The theory, see the temperature-entropy diagram as an inset to Fig.2, must prescribe isentropic nozzle flow from the opening pressure Po through a vena contracta of area Ac, to be followed by a non-isentropic diffusing flow with entropy gain and pressure recovery to a pressure Ps at the full pipe area Ap, the upshot of which is a computed value of mass flow rate tmdot (g/s) which must correspond precisely with the measured value of mass flow rate mdot (g/s). The 'actual' [2,3] discharge coefficient CD is then calculated as Ac/Ap.

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 theoretician 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- ▶

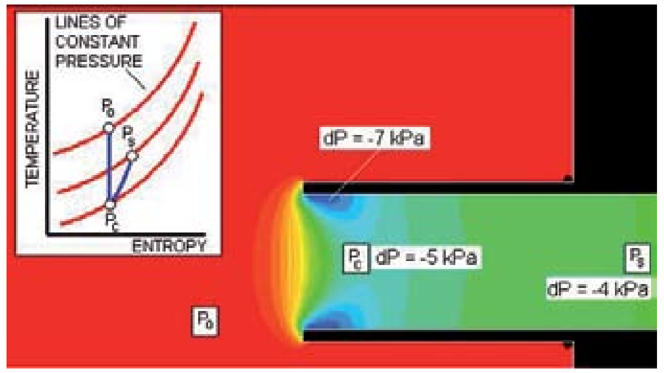

Fig.2 Thermodynamics of flow into a plain pipe end

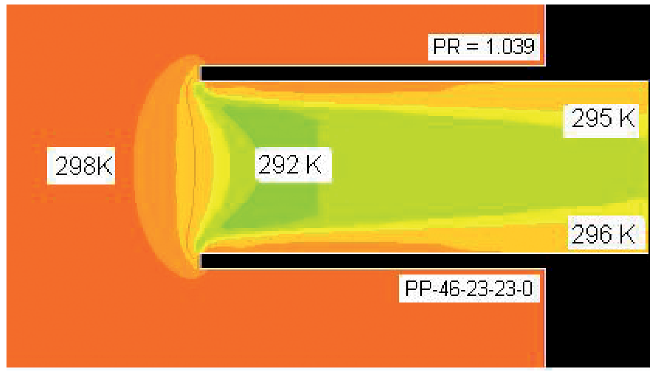

Fig.3 Temperature flow profiles into a plain pipe end

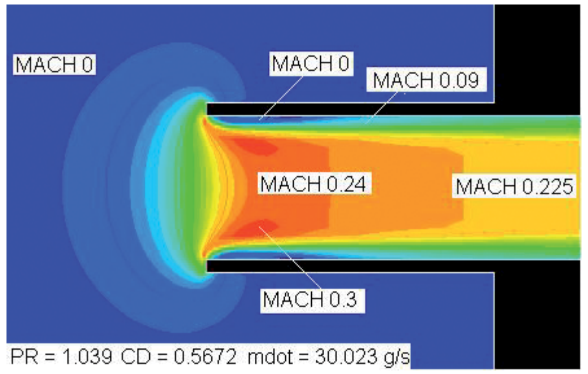

Fig.4 Velocity flow profiles into a plain pipe end

hearted and it is little wonder that many a major engine simulation package supplier has shied away from this, the only approach which will produce accuracy of engine simulation when the attained CD values are re-employed to help compute pipe end boundary conditions [2].

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-ball wrap-round radius installed at the end of the pipe. The second is a bellmouth with an aerofoil 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 aerofoil 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 show the computed Mach number (particle velocity) plots for the simple radius (as RAD-46-23-35-6 of Fig.5) and the ellipse profile (as 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 area Ac, is also given on those diagrams. In Fig.4, the sharp edged pipe case, the CD is 0.5672 and the measured mass flow rate is 30.023 g/s. In Fig.6, where there is a simple radius as the bellmouth, the CD is now 0.719 and the measured mass flow rate is 34.83 g/s. In Fig.7, the

"There is a considerable benefit in mass flow rate by the addition of even a simple radius"

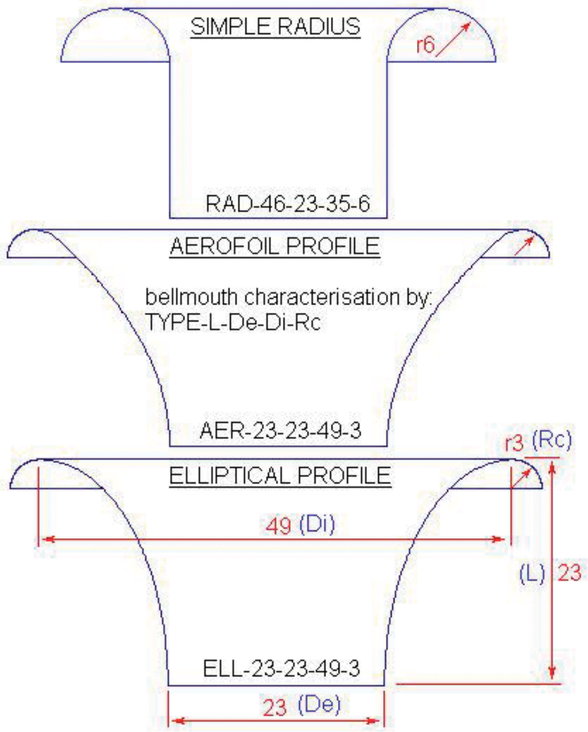

Fig.5 Nomenclature and shape of various bellmouths

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.

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-46-23-23-0 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

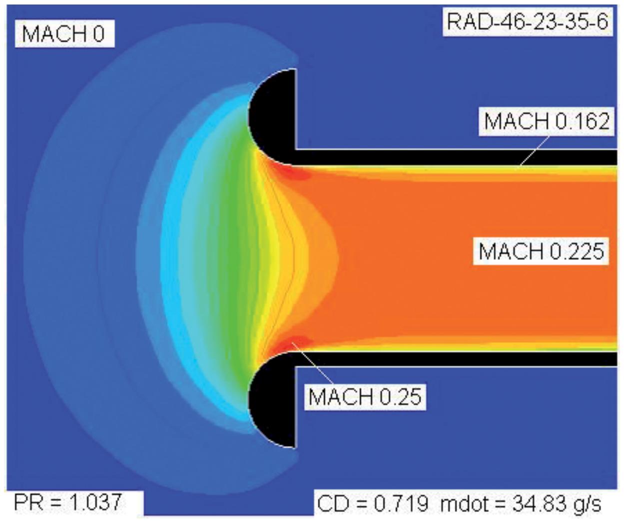

Fig.6 Velocity flow profiles into a radiused pipe end

"The elliptical profile comes out as the winner over the aerofoil profile"

that is not really a major issue as the instantaneous pressure ratio at an actual engine bellmouth during the peak of reflection of the intake pulse will rarely exceed 1.1. However, even at low pressure ratios the CD values will vary by 20% over a PR range from 1.04 to 1.1 and therefore cannot be considered as a constant. On this evidence, the FLUENT code can be trusted to provide us with an accurate prediction of air flow rates into intake bellmouths.

In Fig.9 are shown the (computed by FLUENT) mass flow rates mdot for the same plain pipe and the same simple radius bellmouth. Both are also a function of pressure ratio and here the difference between them remains at a near constant 16% at any given PR value. Lau [3] also quotes measured mass flow rates for the plain pipe and the simple 6 mm radius and those calculated here by CFD agree very closely with those measured, as seen in Fig.9.

DISCHARGE COEFFICIENTS FOR BELLMOUTHS

We will now discuss the results of the FLUENT CFD analysis of the fluid mechanics of the 3D flow and the analysis of its output data to acquire the 'actual' discharge coefficients. In Fig.10 is shown the CD values for a range of elliptical and aerofoil profile bellmouths and also for the simple radius bellmouth RAD-46-23-35-6. All profiled bellmouths have the same exit diameter De of 23 mm, some have lengths L of 23 or 46 mm, some have entry diameters Di of 40, 46 or 49 mm, and all have a corner radiis Rc of 3 mm. It can be seen that all profiled bellmouths exhibit a step increase in CD over the simple radius bellmouth but all lie rather closely together so that it is visually

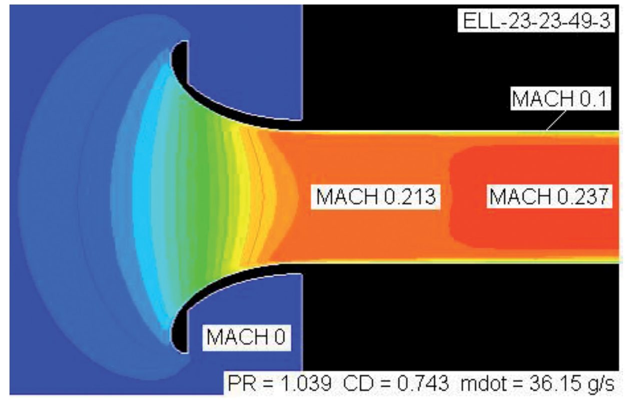

Fig.7 Velocity flow profiles into a bellmouth pipe end

difficult to separate them.

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 L 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 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

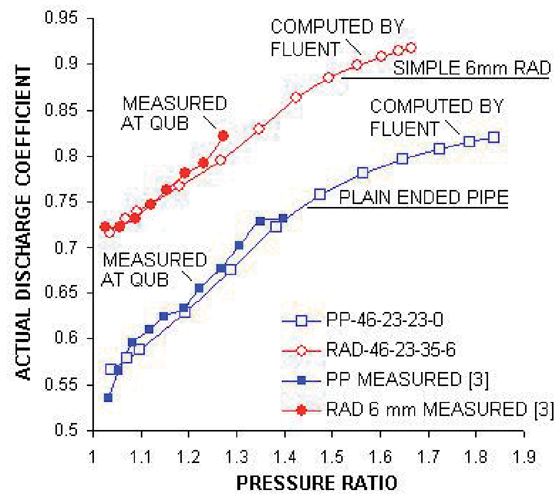

Fig.8 Measured and computed CD data at pipe ends

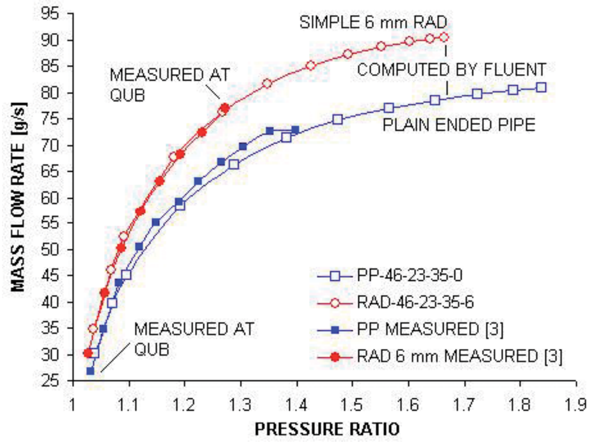

Fig.9 Measured and computed airflow rates at pipe ends

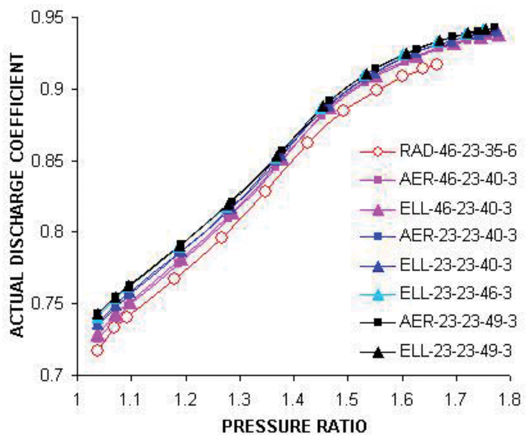

Fig.10 CD data for intake pipe bellmouths

shown the change of mass flow rate for all of the profiled bellmouths over that of the simple radius bellmouth. The change is expressed as a percentage. In the relevant pressure ratio PR band up to 1.1, the best bellmouths are those that are "short and fat" with the elliptical profiled bellmouth ELL-23-23-49-3 hailed as the winner. But the winning margin is very much a "short head" as its advantage over simplicity is a mere 1.5%. However, as that might just be an extra 1.5 hp per 100 hp, we would sooner have it as not!

CD DATA APPLIED WITH AN ENGINE SIMULATION

While the graphs and discussion above may be useful as an aid to understanding of the air flow behaviour at intake bellmouths, it is numbers that engineers need to be able to generally employ in design or, perhaps more specifically, within an engine simulation during the computation of pipe end boundary conditions. For the specific case in question, at the intake bellmouth, the reader will naturally think of it as an "inflow" process. Actually, this is an "outflow case" as the air is "outflowing" from a 'plenum', i.e., the atmosphere, through a 'restriction', i.e., the bellmouth, to a pipe, i.e., the intake pipe leading to the engine. For more theoretical information on the reverse flow case of "inflow", i.e., spitback, you should read a textbook [2] but this common phenomenon you have seen elsewhere in the engine as the exit of particles of exhaust gas at the end of an exhaust pipe!

In Fig.13 is shown some of the previous CD graph data replotted up to a pressure ratio just below 1.4. This range of pressure ratio covers the pressure wave reflection behaviour at the end of an intake pipe for naturally aspirated engines where that end either meets the atmosphere itself or the conditions of an intake airbox. The individual graphs of discharge coefficient for the plain ended pipe, the simple

"The best bellmouths are those that are short and fat with the elliptical the winner"

radius bellmouth and the elliptical profile bellmouth are curve-fitted with a third order polynomial and the equations are printed at the top of Fig.13. The "y" value is the CD and the "x" value is the pressure ratio PR. The quality of the curve fit is visibly good.

However, if the CD line is required within an engine simulation for a bellmouth fitted at an intake pipe where there is located a "restrictor" diameter, to be followed by a pipe, or more usually a diffuser pipe, as described in our previous articles [6,7], then a CD-PR equation only fitted over a pressure ratio up to 1.4 is not very helpful. Hence, the CD line for the best elliptical profile bellmouth, ELL-23-23-49-3, as graphed in Fig.10, is curve-fitted over the full pressure ratio range up to 1.8 and that trend line is expressed below as,

$$CD = 1.7869 - 2.9326PR + 2.5275PR^2 - 0.6446PR^3$$

This information is presented on the assumption that the designer would wish to use the best bellmouth design possible at the entry to the restrictor/diffuser and then simulate it accordingly. This same CD trend line is also applicable to another higher pressure ratio situation where a bellmouth is used and that is at the entry to the compressor of a turbocharger or a supercharger, particularly if that entry is geometrically restricted by the mandates of some racing engine classification.

This information is also presented on the assumption that these CD-PR equations will be applied into an accurate engine simulation where the theory used therein for its mathematical, gas dynamic and thermodynamic replay is that described briefly above and thoroughly in a textbook [2]. If that is not the case then the trend-line data in Fig.13, or the equation above, is meaningless and should not be used, i.e., "garbage in is garbage out".

DYNAMIC FLOW AT THE BELLMOUTH

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 and so the pressure ratio across the bellmouth varies with crankangle and the air particles will not only enter that intake pipe from the atmosphere but will also reverse (spitback) during various periods

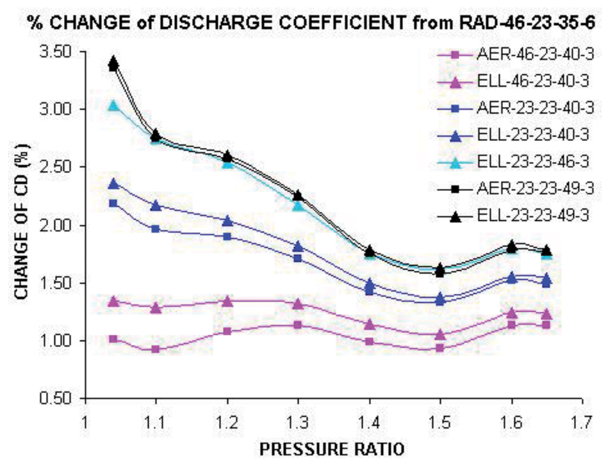

Fig.11 CD variations at intake bellmouths

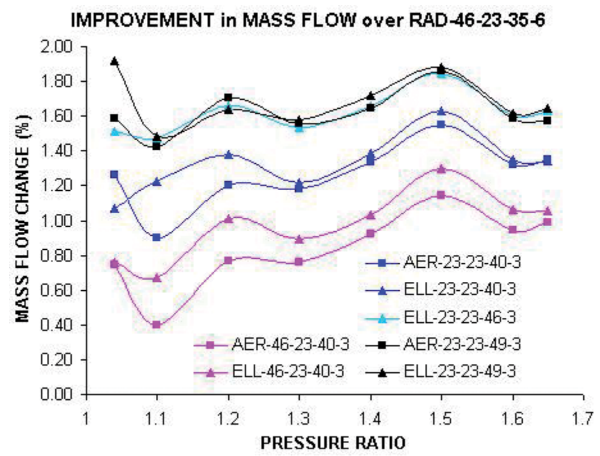

Fig.12 Airflow rate variations at intake bellmouths

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

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.

In recent times, and with the advent of ever 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 of the entire bellmouth including a short segment of the intake pipe beyond which the 1D engine simulation takes over. The engine used within the 1D simulation is the Seeley-Matchless G50 racing motorcycle engine of yesteryear, the geometry and performance characteristics of which are fully described elsewhere [2]. As the G50 engine had a 38 mm diameter intake duct, then new bellmouths for both the simple radius type and the elliptical profile type were designed [5] and their dimensions are shown labelled on Fig.14. In Fig.14 is presented snapshots of their particular particle velocity characteristics (as Mach number contours) at an engine speed of 7000 rpm. A plain-ended intake pipe is also included in this co-simulation computation series for its curiosity value.

In Fig.14a for each of the pipe end conditions tested is a snapshot at a particular crankangle of the "outflow" process from the atmosphere (to normal folk it is obviously an inflow process and only thermodynamic pedants such as your authors would consider it otherwise) at the point where maximum particle velocity is taking place at the bellmouth. The similarity of velocity contour (as Mach number) with the steady flow pictures of Figs.4, 6 and 7 is very apparent. These similarities lend some credence to the oft-used phrase of "quasi-steady flow" as used to describe the conventional theoretical approach in unsteady gas dynamics, which approach is founded in ▶

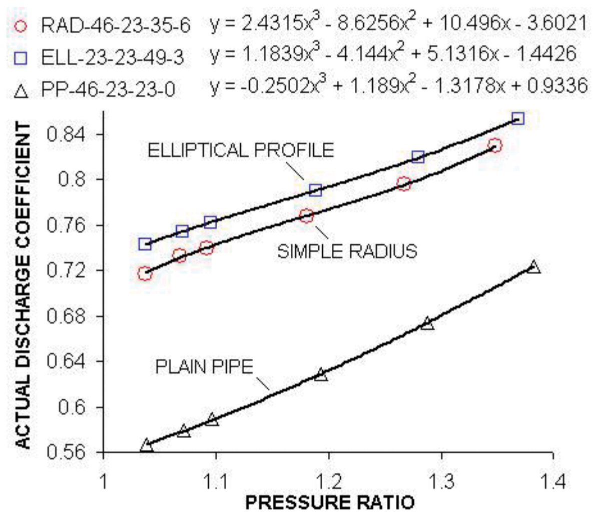

Fig.13 Curve fitted CD data for intake bellmouths

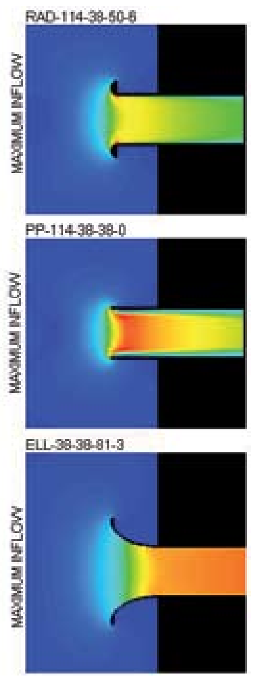

Fig.14a Dynamic particle inflow during co-simulation

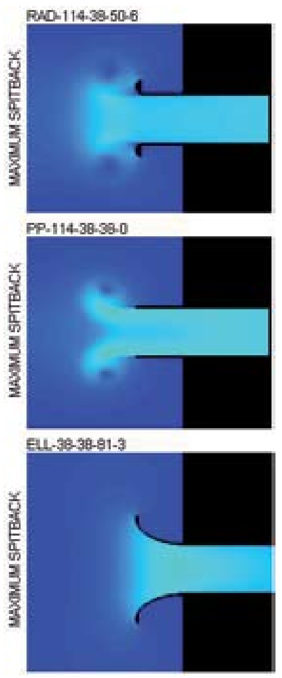

Fig.14b Dynamic particle 'spitback' during co-simulation

the notion that unsteady flow is but a sequence of differing steady flow processes conducted over very short time intervals.

In Fig.14b are the movie snapshots at a particular crankangle at the peak of the reverse flow process, commonly called "spitback", and again your pedantic authors will tell you that this is an "inflow" process to a large plenum called 'the atmosphere'. The particle flow entering the atmosphere is more pronouncedly strong for the weakest bellmouth, i.e., the plain-ended pipe, and vice-versa for the elliptical profile bellmouth. Indeed, it is so strong at the plain pipe end that it has formed a toroidal vortex (smoke 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 spitback of fuel for it is the spitback 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 questions may well relate to the "wrap-round" radius Rc. Is it necessary? How big should it be? Is it OK as a 'half-radius' as we show it here, or should it be a complete 'ball'? The answers are contained in Fig.15.

The steady flow CFD analysis is conducted with three similar elliptical profile bellmouths but one has our common 'half-radius' as seen in Fig.5 (Rc is 3 mm), another has a full 'ball' 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 all three) 46 mm entry diameter Di. The Fig.15 shows the variation from the worst case, which is the 'zero radius case' to the other two cases, as a change of CD expressed as a percentage (%). The results

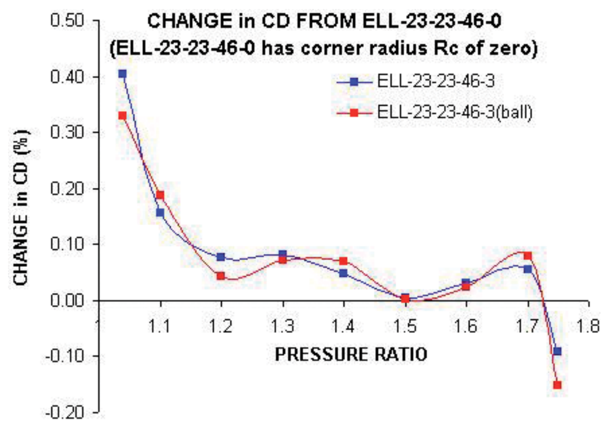

Fig.15 CD variations for bellmouth edge geometries

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

are somewhat inconclusive, although up to the very relevant pressure ratio of 1.1 it can be stated that a sharp-edged bellmouth is marginally inferior to a bellmouth with a Rc corner radius and that the use of a full 'ball' radius is unnecessary.

The second obvious set of questions will doubtless relate to the oft-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 23 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.

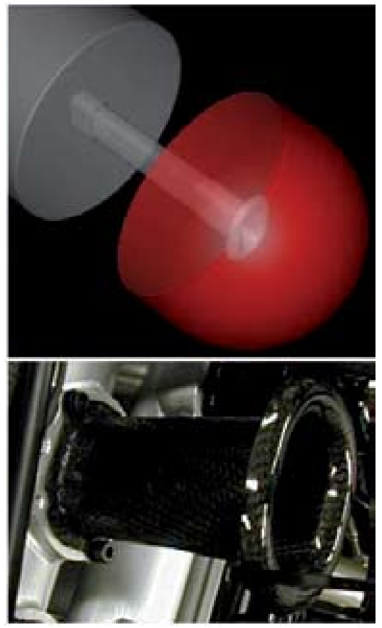

Fig.16 Rectangular bellmouth design

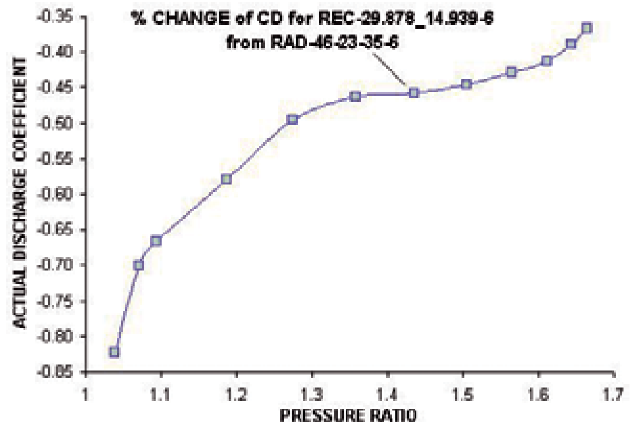

Fig.17 Loss of CD by a rectangular bellmouth

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 texts 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 Schlicting [9], with their inherent rotational turbulence characteristics, always provides computational difficulties for any CFD code.

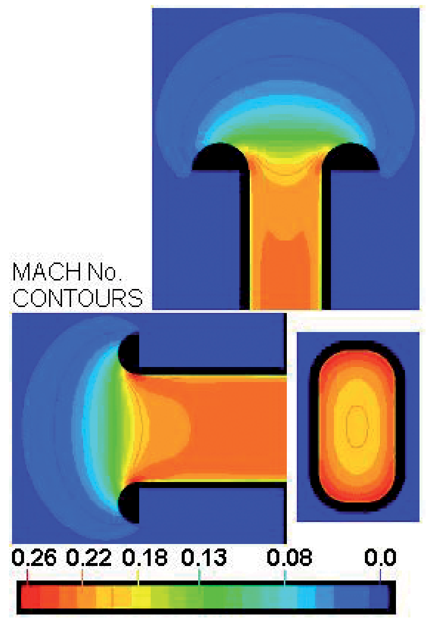

Fig.18 Velocity flow profiles at a rectangular bellmouth

"The computed CD results for the simple rectangular bellmouth are worse"

Quite irrespective of the above caveats, the computed CD results for the simple rectangular bellmouth are worse than 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. That 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.

CONCLUSIONS

While the specific conclusions have already been highlighted at each stage of the discussion above, the general conclusion might be glibly stated that the design of an intake bellmouth is not as difficult nor as vital to good engine breathing as might have been imagined. On the other hand, in racing, where the last few hp per 100 hp is the difference between winning and losing, the design exemplars discussed above are not to be lightly ignored. ■

REFERENCES

[1] FLUENT Inc., Lebanon, New Hampshire, www.fluent.com
[2] G.P. Blair, "Design and Simulation of Four-Stroke Engines", Society of Automotive Engineers, 1998, SAE reference R-186, p813.
[3] G.P. Blair, H-B. Lau, A. Cartwright, B.D. Ragunathan, D.O. Mackey, "Coefficients of Discharge at the Apertures of Engines", SAE International Off-Highway Meeting, Milwaukee, September 1995, SAE paper no. 952138, pp 71-85.
[4] G.P. Blair, F.M. Drouin, "The Relationship between Discharge Coefficients and the Accuracy of Engine Simulation", SAE Motorsports Engineering Conference and Exposition, Dearborn Michigan, December 8-10, 1996, SAE paper no. 962527
[5] 4stHEAD design software, Prof. Blair and Associates, Belfast, Northern Ireland, www.profblairandassociates.com.
[6] G.P. Blair, W.M. Cahoon, "Airbox Design Part 1", Race Engine Technology, Vol.1, No.2, 2003.
[7] G.P. Blair, W.M. Cahoon, "Airbox Design Part 2", Race Engine Technology, Vol.1, No.3, 2003.
[8] V.L. Streeter, "Fluid Mechanics", McGraw Hill, New York, 1961.
[9] H. Schlichting, "Boundary Layer Theory", McGraw Hill, New York, 1961.