

Diesel Engine Intake Port - Flow Stability and application of CFD using a Hybrid-Low-Reynolds near wall method.

J.W. Campbell¹, G. Hardy², F. Personeni²

¹ Imperial College London. Exhibition Rd., London SW7 2BX, United Kingdom

E-mail: j.w.campbell@ic.ac.uk

Telephone: +(44)(0)207 59 47109

² Iveco Motorenforschung AG, Arbon, CH-9320 Switzerland

E-mail: Gilles.HARDY@iveco.com

Telephone: +(41)(0)71 4477 281

Abstract. Traditionally, the design of intake port geometry for diesel engines has been guided by experimental testing using flow rigs measuring swirl characteristics under stationary conditions. These measurements, which are typically time averaged mean global characteristic values, are interpreted using a great deal of individual experience to balance the conflicting requirements of swirl number, flow permeability, packaging constraints, durability and cost of manufacture. The application of Computational Fluid Dynamics to aid understanding in-cylinder air motion is today a common tool. Nevertheless, convincing validation of CFD methods in predicting the intake port swirl flow have been elusive, hampered by the difficulties of accurate prediction of boundary layer separation and associated flow instabilities. In recent years, the difficulty of validation with steady flow swirl rigs has been somewhat circumvented by the conviction among the engine CFD community that it is the transient intake air motion which is of more relevance than the air motion obtained under stationary conditions. This conviction lends itself well to the acceptance of CFD simulation applied to the induction stroke which might be argued as of more relevance to combustion development than steady state flow measurements.

This paper presents a further determined effort, employing latest state-of-the-art industrial CFD methodology, to validate simulations of stationary port-valve flow rig swirl measurements. High definition geometry description, multi-million cell models, high order discretisation scheme, hybrid wall function combined with low Reynolds modelling and transient methodology have been employed. Significant improvements in accuracy of predicted port massflow are found as a result of the better accuracy of modelling boundary layer flow separation brought by low Reynolds near wall modelling. A further novel aspect is the potential of transient CFD studies to help understanding port flow instabilities which occur in typical compromised port-valve geometry.

Notation & Acronyms

RANS	<i>Reynolds Averaged Navier Stokes</i>
URANS/TRANS	<i>Unsteady/Transient Reynolds Averaged Navier Stokes</i>
EVM	<i>Eddy Viscosity turbulence Model</i>
RSTM	<i>Reynolds Stress Turbulence Model</i>
LES	<i>Large Eddy Simulation</i>
UD	<i>Upwind Differencing 1st order accurate Numerical discretisation scheme</i>
MARS	<i>Monotone Advection and Reconstruction Scheme, a robust 2nd order accurate discretisation scheme.</i>
HRWF	<i>High Reynolds Wall Function</i>
HYLR	<i>Hybrid Low Reynolds, combined approach of local low Reynolds near wall and wall function.</i>

1 Introduction

Numerical simulation of the aerodynamic flow in IC engines has been applied in engine research and development since the mid 1980s and in the two decades since there have been numerous studies published and methods established that were suggested to be robust and sufficiently accurate to assist the design of ports in an industrial context. The present studies began with the application of the conventional industrial CFD method, which will hereafter be referred to as High-Reynolds Wall Function (HRWF) method, for simulating the air motion through stationary engine valve-intake ports assembly. This work was intended as a relatively straightforward (as it was then believed) pre-cursor to making transient moving mesh simulations of the intake stroke and the closed cycle compression/combustion phase with spray and combustion. At the beginning of this work in year 2001, the conventional method of CFD being applied in industry in support of engine port design was and largely remains today that of High-Reynolds turbulence modelling combined with law-of-the-wall. Further, in the context of stationary valve-port simulations, a steady-state solver and a stable 1st order accurate discretisation scheme are also common industrial practice. It is widely acknowledged that these conventional HRWF methods have severe limitations with regard to accuracy and predictive capability of the local port-valve flow. Despite these shortcomings CFD simulation using HRWF appears to have found widespread acceptance as a tool to aid design of engine intake ports.

In typical studies of engine aerodynamic flow using CFD, for example the recent studies by Gil et al.[15] and by van den Heuvel et al.[8], the focus is the closed in-cylinder flow after the induction event and during compression. Experimental access during this closed phase is relatively straightforward compared with the more difficult port-valve-manifold region. Correspondingly, comparisons made with numerical simulations, for purposes of verification, are also looking at the bulk ensemble averaged flow patterns in the cylinder, downstream and some time after the flow through the valve-port itself. The aspect which is unfortunately absent, is the convincing validation of local accuracy of the CFD method in the upstream port-valve-seat region using local measurements.

Among the recent numerical studies which strive to improve the accuracy of the port-valve CFD simulation, Basara et al. [5, 4] applied a second moment closure (RSTM) method in combination with wall functions to the case of dual tangential intake port of a diesel engine and found global results to be in better agreement with experiment than those provided by a linear (EVM) eddy viscosity model. However, significant disagreement remained in the flowfield predicted by the RSTM compared to experiment; this was attributed to the use of wall functions. Basara had previously presented [3] results using RSTM combined with Low-Re $k - \epsilon$ model for a two-dimensional test case. He concluded in that study, the approach could be of great enhancement to complex three-dimensional industrial flows and supported abandoning wall function practice for certain industrial calculations. It appears as yet, that second moment closure method (RSTM), integrated through the near wall region, creates much numerical difficulty and a large increase in computational expense.

Bianchi et al.[7] and separately Bella et al [12] have applied the conventional CFD methodology to high performance gasoline intake ports and both found the accuracy of prediction for discharge coefficient to fall in the range of 4 to 12% depending upon lift case, turbulence model and grid resolution. In particular the studies of Bianchi et al [7] compared eddy viscosity turbulence models, with emphasis on performance of the near wall separation prediction; Concluding that none of the High-Reynolds form of EVM models with wall function, could predict the correct trend in discharge coefficient shown by three port geometry.

In other recent studies [17, 6], non-linear EVM with low-Reynolds near wall treatment has been applied and is verified to provide substantially better accuracy for predicting engine intake port discharge coefficient. In [6] Bianchi et al. concluded that accurate near wall treatment is fundamental in determining correct prediction of port discharge coefficient. Most importantly, [6] highlights the failure of conventional HRWF to correctly rank the performance of three alternative port designs, whereas the low-Reynolds approach gave the correct trend.

The present studies seek to extend the application of low-Reynolds near wall method, using EVM, to the case of diesel engine intake port design.

Intake Port Stability - an industrial perspective The term 'Stability' of an engine port may be interpreted slightly differently depending upon context. A 'stable' port from the point of view of engine product development is one whose stationary flow bench performance is consistent and insensitive to variations in tolerances which occur during manufacturing, for example to core movements during casting or precision of machining and location of the valve seat. It is desirable that the measured stationary flow characteristics of cylinder heads sampled from the production line are consistently in agreement with the characteristics measured on the prototype cylinder head. This interpretation of 'stability' would be better termed robust design, since the reasons for changes in flow behaviour can be attributed to approximations and tolerances during the sequence of events from translation of CAD through core prototyping, casting and CNC machining.

The ultimate quality assurance test of a production port, is how consistently it reproduces prototype combustion performance in the running engine. During experimental testing the acoustic noise can be a qualitative indication of the 'stability' of air flow in the port. Unfortunately the rather abstract characteristic of some port designs giving quiet/stable flow but others noisy/unstable flow does not correlate easily with engine performance on testbed. Nevertheless, it is known that two port designs of the same flow and nominally the same swirl levels as measured from the stationary flow rig, can exhibit different combustion behaviour on the engine. The question is how might we define and measure this additional characteristic port stability, and also can we simulate it using CFD for design purposes.

A recent example of extending the experimental method for characterising Intake Port flow is that of [13] where PIV method is used to obtain instantaneous swirl measurements from a stationary port flow rig and the data processed to yield a 'Swirl Stability Number' which characterises the '*...vortex stability of a given port flow*'. This particular test rig apparatus appears to be one which can be used on a regular basis in an industrial environment for characterising production cylinder heads.

An further recent example of a more sophisticated experimental method for characterising port flow performance is shown in [2] where LDA was applied to the case of a moving (dynamic) valve under otherwise 'steady' flow conditions. This raises the issue of the *dynamic* flow performance of the port during valve motion. The data presented in [2] has been processed to describe the ensemble mean flow behaviour. Although there could also be value in examining the statistics of the individual measurements.

Intake Port Stability - aerodynamic research perspective The geometry in even the most direct flowing gasoline engine port contains a high degree of curvature and bluff body features which create separated wake flows and associated unsteady vortex shedding structures. The valve stem is the most obvious obstruction whose presence may be compared to a cylinder in cross-flow and will thus generate vortex shedding. Then there is the valve head itself with bluff surface beneath the head of the valve around which the high speed annular free jet emerges. In a diesel engine port the design goal of swirl generation combined with constraints of cylinder head layout, size, packaging and cost of manufacture are dominant over achieving optimal port discharge coefficient. This leads to the port shape being compromised and features which give rise to separated flow in the upper port. From an aerodynamic simulation perspective, a diesel intake manifold/port is an extremely major challenge because it combines, inside a wall bounded geometry, many of those difficult flow features such as high streamline curvature, pressure gradients, stagnation regions, separation and re-circulation.

The presence of unstable flow features occurring within the port-valve assembly during nominally steady flow testing has been confirmed experimentally in research by [27, 21, 20] and investigated most recently in an industrial context by Bensler et al.[13]. The instability of a helical diesel engine intake port under steady flow test rig conditions was investigated in [27] using LDA measurement. This study evidenced that instantaneous velocity of the valve jet changes widely with time even in a steady flow condition, and the unsteadiness gives rise to instability of the swirl motion such that cyclic variation of swirl could occur in the operating condition of a running engine.

Experimental evidence of unstable flow for the case of an axisymmetric poppet valve is presented in [21, 20] where LDA and laser sheet visualisations revealed evidence of a 'jet flapping' instability below the head of the valve. This was attributed to a vortex shedding mechanism beneath the valve head similar to a Von Karman vortex street. Also, in the region around the valve stem, visualisation studies showed flow separation from the valve stem and a recirculation region which extended along the valve head to the valve gap. These and further related studies of dual intake port under steady state conditions [23], made statistical analysis of velocity measurements, confirming intermittent velocities consistent with jet instability/flapping of the mean flow. The presence of coherent, deterministic (non-turbulent) phenomena such as jet instability is indicated as making a large contribution to the r.m.s signal typically used to quantify engine in-cylinder turbulence. The result is an increase in the apparent level of turbulence intensity.

In-Cylinder Swirl Stability and Engine Cyclic Variation The influence and importance of in-cylinder turbulence and cyclic variation on gasoline engines is well documented, but is considered of less relevance to diesel engine design. Nevertheless, as the trend for diesel engine combustion moves towards a more premixed nature and in the extreme an HCCI mode, the subject of instabilities which occur during the intake phase and their contribution to turbulence levels or cyclic variation [9], does become more relevant. Conventional understanding of swirl in a diesel engine is that it promotes fuel-air mixing and macro mixing during combustion. It now also appears that swirl can assist in establishing an organised structured flow field which inhibits undesirable fluctuations and improves cyclic stability. The recent study in [22] shows '*The PIV flow measurements clearly show the cycle to cycle variation found in the flow field in a motored heavy duty diesel engine. The flow structure is more stable at higher swirl ratios.*' and emphasises the complex role of swirl in a diesel engine.

2 Experiment

Two different style of four valve HSDI engine port geometry have been used to provide experimental validation from a Stationary Flow test rig. The design denoted 'case A' in Fig.2 is a Siamese style intake port from a heavy-duty truck engine. The second design shown in Fig.1, 'case B' comes from a dual separated port design of a light duty truck engine. A summary of each engine operating conditions are shown in Table 1.

Table 1: Engine Operating conditions

Engine Case	Case A	Case B
Bore	125mm	96mm
Rated Speed	2100rpm	3900rpm
Valve inner Seat Dia.	37.6mm	29.2mm
Max. Valve Lift	11.5mm	10.5mm
Valve Open period[CA] at rated speed[sec]	230 0.0183	240 0.0103

Fig. 1: Port Case B

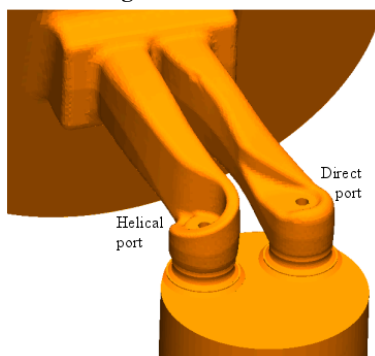
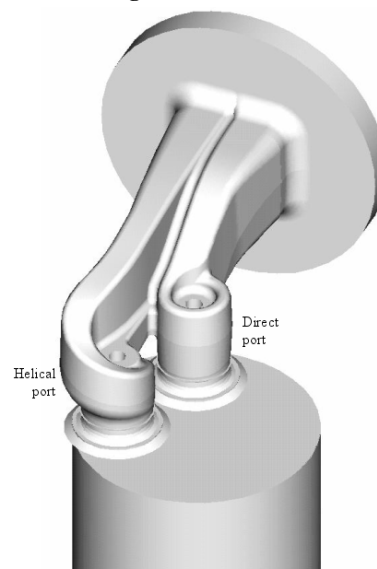


Fig. 2: Port Case A



The experimental stationary flow test rig is shown in Fig.3. The size of the rig is dominated by the large air tank below the cylinder head assembly. The rig uses an Impulse type torquemeter comprised of a honeycomb structure. A prototype or a production engine cylinder head is located at the top of the rig. A dummy cylinder is positioned below the cylinder head so that the impulse torquemeter is always located at a distance of 1.75 engine bore diameters below the port deck. Flow testing procedure is conducted at 1mm increments of valve lift using a constant **50mBar** pressure difference. In the case of a dual port configuration, testing may be done with either valve closed, as for case B in this work.

The relevant characteristic conditions for the experiment are shown in Table 2. The pressure ratio (1.05) and isentropic velocity indicate that a mild compressibility can be expected. Representative Reynolds number based on bulk flow velocity and valve diameter is circa 100,000. Also of interest is a Reynolds number $Re_{D_{st}}$ characteristic of the cross flow on the valve stem. Using the analogy of a cylinder in a cross flow, the value $Re_{D_{st}} \simeq 25,000$ indicates the subcritical regime giving rise to vortex shedding. An estimate of frequency of vortex shedding in the range 1100 to 1400Hz is made using the Strouhal number relation. This corresponds to a timescale more than one order of magnitude smaller than the opening duration of the intake valve.

Valve Seat angle study: For the Case B port, in addition to the individual (direct and helical) port measurements, a further set of measurements was available for an alternative 45 degree valve seat profile and a slightly modified port geometry, see Fig.4. This test case was chosen as the basis for performing a 2D simulation study (see section 4.2) to provide an economic pilot study comparing the near wall CFD methods.

3 Numerical Methodology

3.1 General

Numerical Discretisation scheme

The first order accurate Upwind (UD) scheme is computationally robust and efficient but, as shown in 3.4 can lead to substantial inaccuracy due to numerical diffusion especially in the application of swirling flow. The second order accurate Monotone Advection and Reconstruction (MARS) scheme is a proprietary STAR-CD scheme [1] based

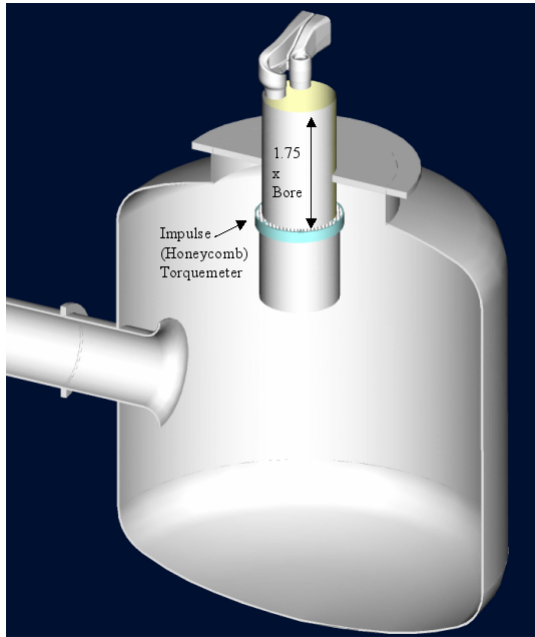


Fig. 3: Experimental Flow Rig

Table 2: Experimental Data

Engine	Valve Lift		Massflow [g/sec]	Torque [Nmm]
Case A	7mm	Dual	104	68.2
	11mm		119.1	106.6
Case B 30deg.	6mm	Dual	63.6	16.7
		Direct	34.6	12.7
		Helical	30.7	4.8
	10mm	Dual	69.7	38.3
		Direct	37.4	26.2
		Helical	33.8	3.8
Case B 45 deg.	6mm	Dual	63.4	22.1
	10mm		70.9	45.8

Table 3: Characteristic Conditions

$P_o=96.0\text{kPa}$ $T_o=297\text{K}$ $\rho_o=1.13\text{kg.m}^{-3}$ $\Delta P=5.0\text{kPa}$ Pressure Ratio: $\frac{P_o}{P_{cyl}}=1.055$ $V_{ideal} = \sqrt{\frac{\gamma}{\gamma-1} \frac{P_o}{\rho_o} \left[1 - \frac{P_{cyl}}{P_o} \right]} = 95.1 \text{ m/sec}$		
	Case A	Case B
Sample massflow (2 valves) $\dot{m}$	119 g/sec	70g/sec
Valve Seat Dia. D_v	38mm	29mm
Characteristic Velocity $V_{char} = \frac{\dot{m}}{2} / \left[\rho_o \left(\frac{\pi D_v^2}{4} \right) \right]$	47.6 m/sec	46.2 m/sec
$Re_{D_v} = \rho_o V_{char} D_v / \mu_o$	118 000	90,400
Valve Stem Dia. D_{st}	9.0 mm	7.0 mm
$Re_{D_{st}} = \rho_o V_{char} D_{st} / \mu_o$	28 200	21,600
Vortex shedding* Freq.[Hz]	1100 Hz	1400 Hz
(*Strouhal=0.21) Period [sec]	0.0009 sec	0.0007 sec

on a Total Variation Diminishing (TVD) approach which provides 2nd-order accurate numerical discretisation combined with robust numerical stability.

Transient RANS

Transient or also referred to as Unsteady RANS (URANS) is a time accurate solution of the ensemble averaged form of the Reynolds Averaged Navier-Stokes equations, which retain a time derivative term unlike time-averaged equations. Therefore this method is justified to capture the large scale organised (coherent) unsteady flow structures. In many cases of flows past bluff bodies, with stationary boundary conditions, the URANS method has been shown to capture the wake region and instabilities such as vortex shedding. In external aerodynamics of vehicles, URANS has been shown to quantitatively improve predictions of drag coefficient [16]. In the present context, a very relevant example of the URANS approach is that of [28] where URANS was used to investigate swirl stability in a gas turbine swirl burner. Some scepticism regarding the applicability of URANS to unsteady flows driven by turbulence mechanisms may be argued on the basis that the averaging assumes a separation of scales between the resolved motions and the turbulent fluctuations. In these studies the scales of expected coherent motions being captured in the mean flow are considered to be far removed from the scale of turbulent eddies.

Turbulence Modelling

Closure of the Reynolds stress terms in the RANS equations is obtained in the present studies by employing various two-equation eddy-viscosity type turbulence models (EVM). The choice of models used has been limited to those presently available in conjunction with the Hybrid near wall method as outlined below. The standard(compressible) $k-\epsilon$ [11], the standard non-linear cubic $k-\epsilon$ of Lien, Chen and Leschziner [18] and the SST $k-\omega$ model of Menter [19], have been used in both their high-Reynolds and low-Reynolds forms. There are numerous alternative EVM models, each with their own supposed strengths but which are presently not available in hybrid application. The most favoured alternative models will gain mention in the conclusions of this paper.

It is acknowledged that application of higher moment closure RANS turbulence modelling such as Reynolds Stress Transport Models (RSTM) or Large Eddy Simulation (LES) are in theory better candidates to solve the complex physics of non-isotropic turbulent flow occurring in engine ports. However, aside from their greater computational

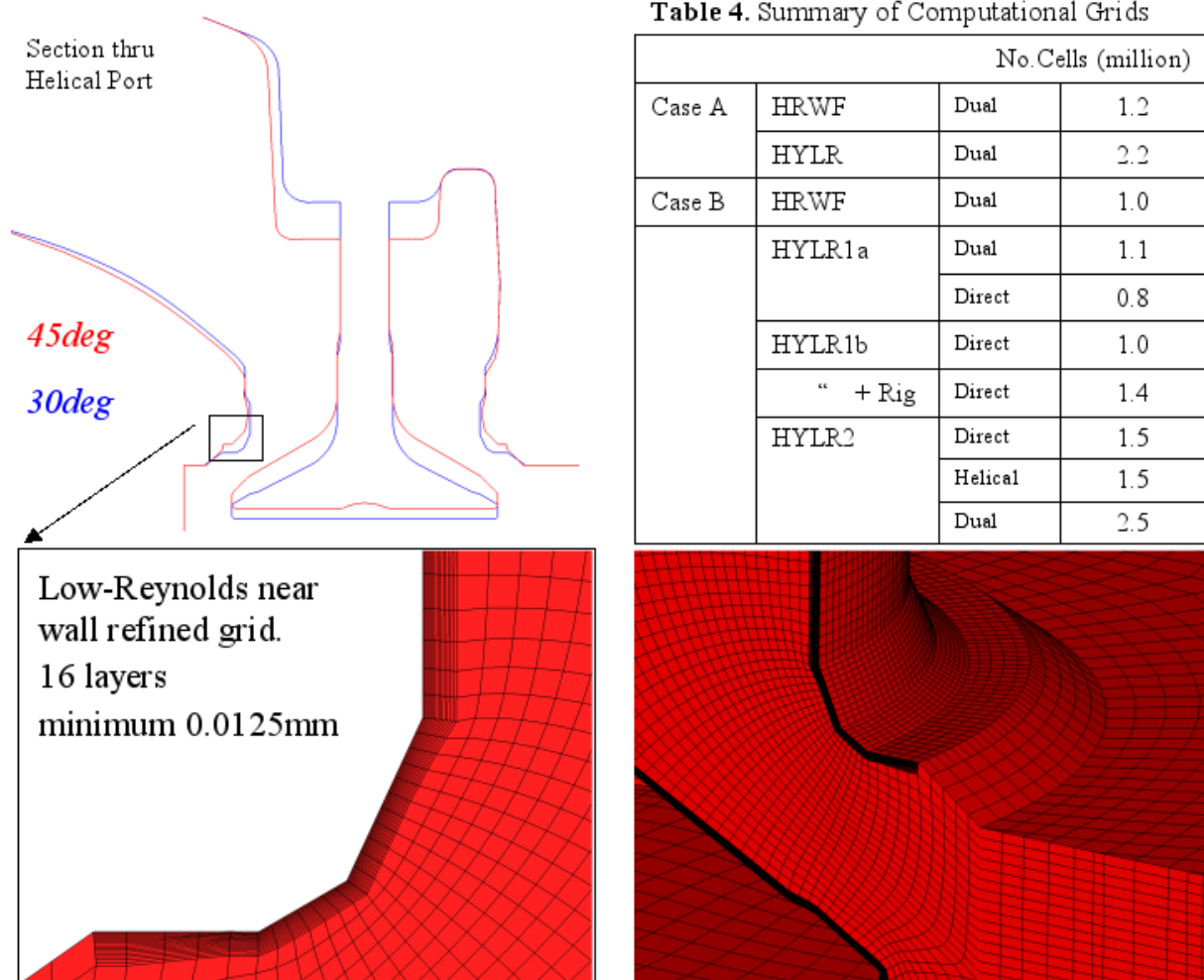


Fig. 4: Valve seat angle and Low Reynolds Mesh detail

expense, neither of RSTM nor LES are currently available in a robust industrial form in combination with suitably advanced near wall turbulence treatment.

3.2 Near Wall Modelling

Conventional High-Reynolds/Wall Function Methodology

In conventional High-Reynolds Wall Function (HRWF) method a turbulence model is used which is valid away from the wall boundaries, namely outside the viscous sub-layer of the wall boundary layer. The influence of the viscous dominated near wall regions are not resolved by the grid but are instead accounted using an empirical logarithmic law-of-the-wall analytic function - the 'Wall-Function'. This approach provides the wall boundary condition to the core flow, eliminating the need for a fine mesh to resolve the velocity gradients approaching the wall. Unfortunately the wall function approach is only accurate where the law-of-the-wall equilibrium conditions are met, which is rarely the case in real flows. Improvements to the standard wall function approach are available which provide a degree of sensitivity to pressure gradient effects. STAR-CD code [1] provides such a non-equilibrium wall function based on [24] and this has been applied in the high-Reynolds wall function calculations presented here. Aside from the inaccuracy of the wall function under attached boundary layer conditions, it's major limitation is a complete absence of capability to deal with separated flows where the wall shear stress τ_w vanishes and the associated friction velocity u_τ and y^+ have no meaning.

Low-Reynolds Turbulence Near Wall Method

In Low-Reynolds number near wall treatment, the turbulence equations are integrated up to the wall. The difference, compared to High-Reynolds form, is that account must be taken for the more dominant role of viscous effects

on the flow approaching the wall. This is usually accomplished by using a blending function. Physically, what happens is that the turbulent fluctuations which are free to move in the free stream, become strongly damped as their lengthscales must diminish while approaching the wall. In the turbulence modelling, damping functions are introduced to the dissipation rate (ϵ) transport equation and to the eddy viscosity term (μ_{tau}) [1]. The damping functions provide a mechanism of blending solution of the same equations throughout the whole domain. The disadvantage with the low-Reynolds method is the large increase in computational resource associated with the necessary fine grid in the wall region. Typically at least 16 near wall cells are needed where a single layer would exist for a wall function treatment.

Two-Layer Near Wall Method

The Two-Layer (TL) method is similar to low-Reynolds in respect of integrating turbulence up to the wall, and necessitating a fine grid to resolve the velocity gradient. However in the TL, the flow domain with regard to turbulence modelling is considered as two separate, coupled zones. In the near wall zone/layer the turbulence is modeled using a one-equation or a mixing length model, while in the outer freestream zone a two equation high Reynolds EVM is used. The TL method can be used in a 'hybrid' way by specifying a-priori which local wall regions to which it applies, whereas elsewhere the high Reynolds equations and wall functions will be calculated.

Hybrid Low-Reynolds Near Wall Method

The conventional two-layer approaches in STAR-CD have been effectively superseded by what is known as the 'Hybrid' near wall method [1, 26].

The STAR-CD Hybrid near wall treatment enables two-equation low-Reynolds turbulence modelling to be solved locally where the mesh refinement is sufficiently fine ($y+ \simeq 1$) while simultaneously retaining elsewhere the high computational efficiency provided by the wall function treatment (large cell size $y+ > 11$). The Hybrid implementation will automatically determine in which regions to apply the low-Re and where else to apply wall function. Further, the method can blend between low-Re and wall function in regions where the $y+$ values are not strictly valid for either single approach. In this way, the results are less sensitive to changes in near wall resolution. The method can provide a robust seamless blend of low-Re and wall function with accuracy determined by the users choice of local near wall resolution.

The Hybrid approach is extremely advantageous in the application to the engine port-valve assembly, because the benefits of a more rigorous boundary layer solution can now be applied locally to critical wall surfaces without necessitating the large computational expense of a globally refined near wall grid.

Presently, the Hybrid near wall treatment in STAR-CD is available with a choice of two-equation low-Re EVM models: standard $k-\epsilon$ [11], standard non-linear cubic $k-\epsilon$ [18] and the $k-\omega$ SST model of Menter [19].

3.3 Numerical Model Domain and Boundary Conditions Set-up

Consistent with the physical test rig, the computational model uses specified pressure boundary conditions giving a pressure drop of *50mBar* from inlet to exit of the model. The application of pressure boundaries to a computational model does sometimes produce numerical difficulties in initialising the calculation compared with a prescribed massflow condition. Nevertheless it is desirable to operate the computational model as realistic as possible to the physical test rig.

As indicated in Fig.5, two alternate model domains have been used in the calculations. For all model cases, the domain extends upstream of the port arms entrance into a substantial hemispherical volume. On the surface of the hemisphere, a pressure boundary is applied equal to the ambient pressure of the room containing the test rig. The downstream extent of the domain and associated boundary conditions is a topic of an on-going investigation. Following standard industrial practice, the model domain shown in Fig.5(a) has been used in the majority of calculations and all of those presented here have used this 'standard' domain. The model domain extends only to the base of the test rig dummy cylinder, a distance of 3.5 cylinder diameters, at which point a static pressure boundary condition is applied. However, using this condition can cause difficulty with calculation start-up and also exhibits recirculating flow across the 'exit' pressure boundary which is an undesirable condition at a model exit boundary. The cause of the recirculation can be understood due to two factors: one is the extent of the wake produced beneath the bluff body of the valve head, and second is the radial pressure gradient which must exist in the field of a swirling flow. The resulting effect is a suction effect at the centre of rotation. In the test rig, the swirling flow is being straightened by the presence of the torque-meter. Therefore it was decided to include simulation of the torque-meter in the model domain.

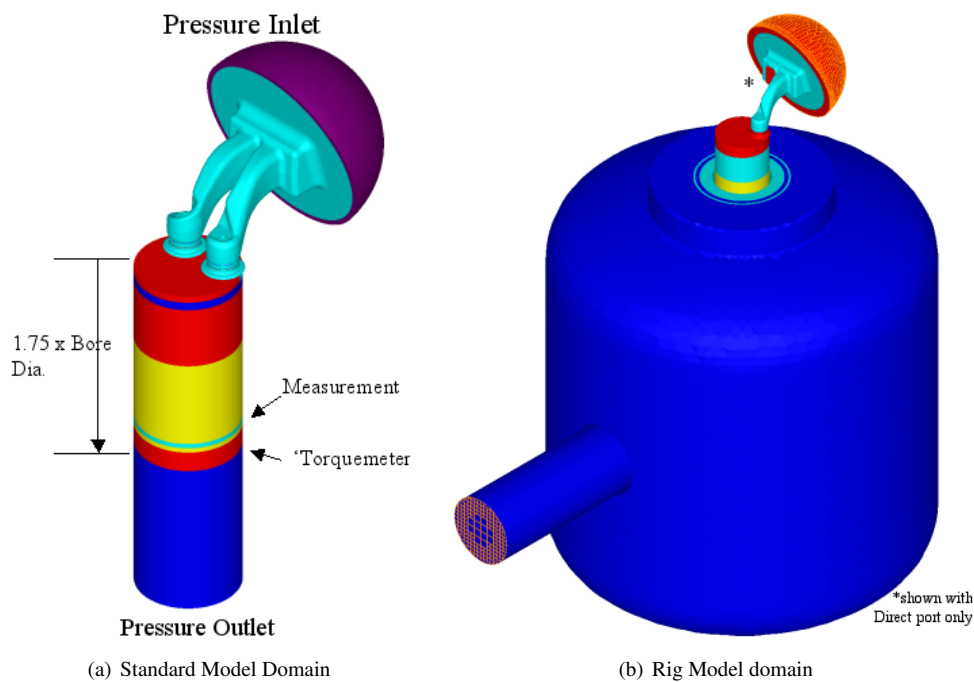


Fig. 5: Computational Model Domains

3.4 Numerical Torquemeter and Dummy Cylinder Test case

To simulate the torquemeter, several strategies have been considered and each one tested using the simplified case of a dummy cylinder without any engine ports. A forced vortex flowfield was specified at the cylinder inlet together with an axial velocity corresponding to measured massflowrates. An additional simplification is to run the test model with slip (friction-less) wall conditions so that in theory there should be no loss of angular momentum through the flow i.e. the numerical torquemeter should measure the same flux of angular momentum leaving the domain as the one prescribed at inlet.

A momentum sink strategy was selected as offering the easiest method of representing the presence of the torquemeter. A cell region at the same location and size as the torquemeter's honeycomb is designated to contain negative momentum source terms which are proportional to the in-plane (horizontal) components of momentum. In this way, all of the flow angular momentum is removed without affecting the axial momentum.

The Importance of Higher-Order Discretisation

A noteworthy finding from the dummy cylinder test case is the large loss of angular momentum (25%) when the UD discretisation scheme is used. Several variations of the dummy cylinder model have been constructed to confirm the strong discretisation influence. One example is shown in Fig.6(a) using a quarter cylinder sector model.

To re-cap, the dummy cylinder model uses a prescribed inlet swirl with slip wall conditions which should provide the same flux of angular momentum at the outlet as it is prescribed at inlet. Consequently, any loss of angular momentum must be due to numerical error arising from the discretisation scheme. With the UD scheme, it is found that when using a grid resolution typical of many industrial CFD models (approx. 4mm hex cell size), there is a very substantial 25% reduction in swirl level (angular momentum) from cylinder entrance to exit due to numerical discretisation error. When the same calculation is re-started with the higher order accurate MARS scheme the reduction of angular momentum becomes almost negligible as expected, confirming the better numerical accuracy afforded by the higher order scheme.

Hence it can be concluded that the UD scheme introduces substantial discretisation error and frankly should not be considered for calculations other than to provide a plausible initialised flowfield as a starting point.

3.5 Transient calculation procedure

In the transient simulations, the PISO algorithm is used with first order fully implicit time discretisation[1]. Initialisation of the transient solution field is made for all cases by first making a steady-state solver calculation using the UD scheme to obtain a converged solution field. This steady-UD solution field is then used as the basis for re-starting in transient mode and employing the MARS scheme.

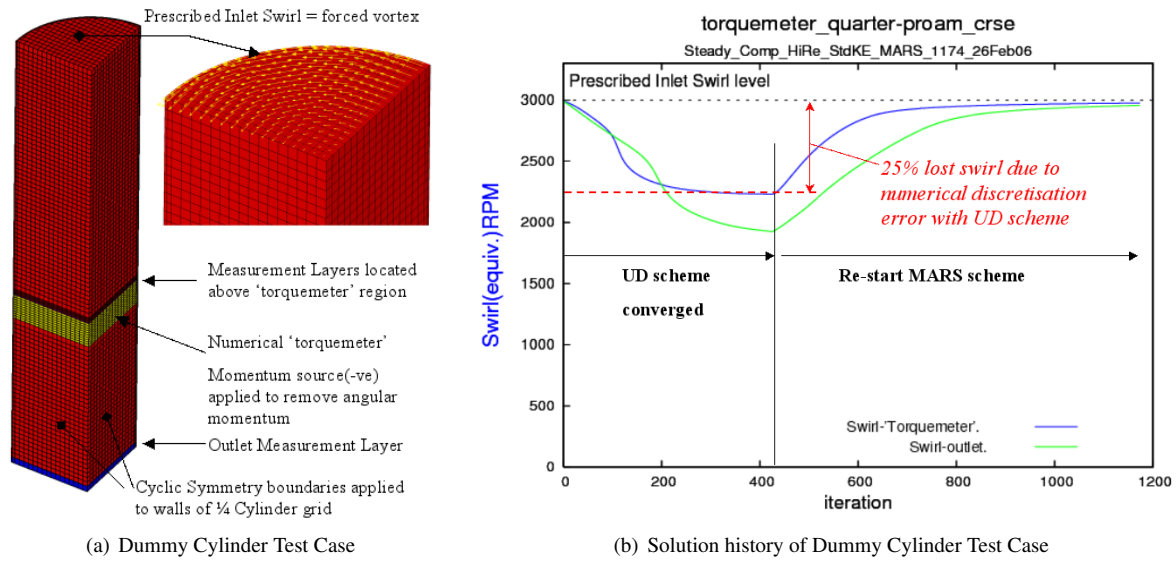


Fig. 6: Dummy Cylinder Test Case highlighting importance of Discretisation for Swirl accuracy.

Specification of transient timestep requires a compromise between meeting courant number restrictions and providing a realistic computational expense. A timestep of $1\text{E-}05$ seconds is applied in most cases and is found to provide reasonable limits on average and maximum local courant numbers. Based on an estimated throughflow time of 0.003sec and allowing for maybe ten cycles to start up, we may consider a calculation time requiring at least $0.03 / 1\text{E-}05$ equals 3000 timesteps.

4 Results and Discussion

4.1 Conventional High Reynolds-Wall Function Port Simulations

Prior to the application of the hybrid low Reynolds near wall method, an extensive range of calculation cases had been made using the conventional high Reynolds wall function (HRWF) approach. Several alternate meshing strategies and various high Reynolds eddy-viscosity turbulence models (EVM) were tested. A sample of the typical results are presented below.

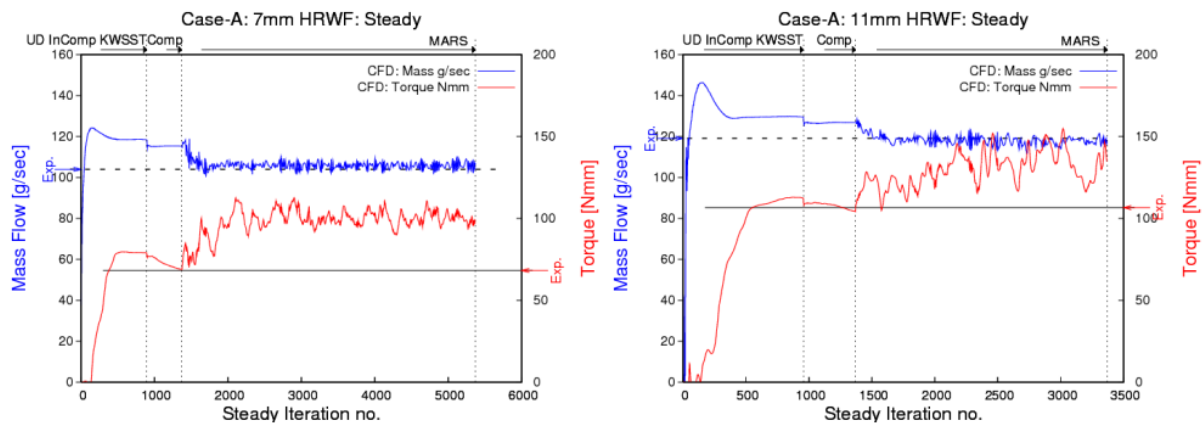


Fig. 7: Case-A 7mm and 11mm Lift HRWF Steady Global Results iteration history

Figure 7 shows typical steady calculation history for mid(7mm) and high(11mm) lift for case-A. The calculation begins as Incompressible UD, it is then re-started and converged with Compressible flow again with UD. Then, it is re-started a second time and run with MARS scheme. In these particular cases, a satisfactory prediction is found for the massflow at both lift cases. The torque level is not so accurate and shows large fluctuations, although in an industrial context it might also be interpreted as indicating an approximate 'converged' level. Figure 8 shows the global residuals corresponding to the calculations used for results in Fig.7. Satisfactory residual convergence levels of circa $1\text{E-}04$ are obtained for the UD calculations, consistent with the stable converged levels of the global

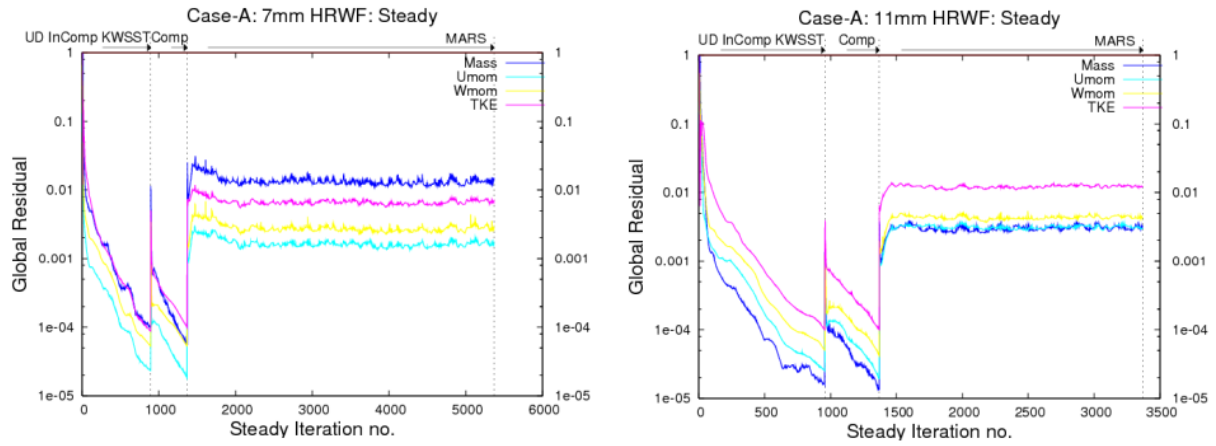


Fig. 8: Case-A 7mm and 11mm Lift HRWF Steady Residuals iteration history

values. However, the residuals levels rise by at least one-order of magnitude when the MARS solver is applied. Nevertheless, in an industrial CFD context given that the global engineering quantities shown in Fig.7 appear to converge at reasonable stable levels, these calculations may be considered acceptable.

When we move our attention to the second engine port case-B, we now see the result shown in Fig.9 for the 6mm lift dual case which shows a severe (+12%) over-prediction of the massflowrate. Also apparent is the very large fluctuation in torque level, which cannot with any stretch of imagination be considered 'converged'. Here we see the breakdown in our weak assumption of having achieved a 'converged' steady solution for the previous case-A simulations. This 6mm lift case-B model was re-calculated many times using alternative grids and turbulence models, but always yielded similar results as Fig.9 with large over prediction in massflow. The set-up was exhaustively checked and the same model built independently by two other CFD users, always yielding the same substantial over-prediction in the level of massflow.

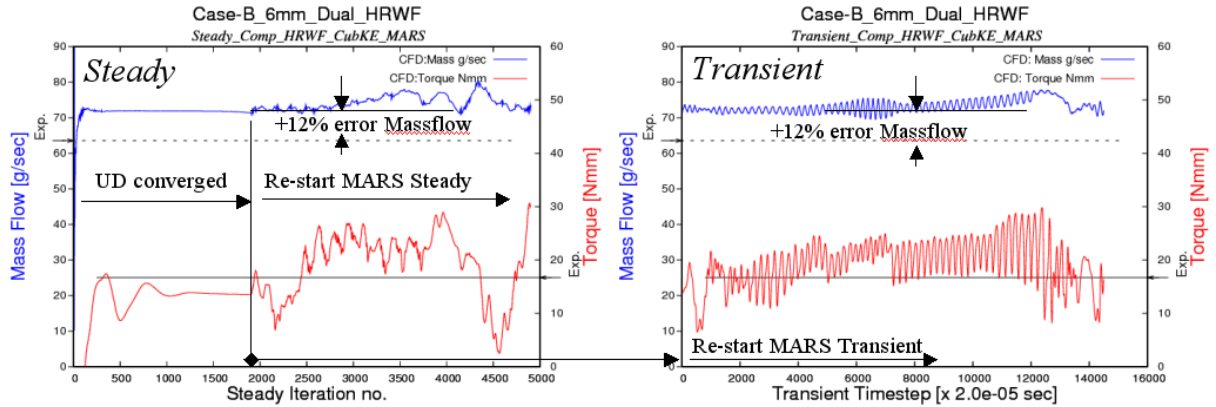


Fig. 9: Case-B 6mm lift dual port HRWF, Steady solver not converging, Transient confirms fluctuations

Transient solution using High Reynolds Wall Function method

It is well known that, in the majority of 3D bluff body flows, the wake of the real flow is not steady despite the stationary boundary conditions. In the field of external flow aerodynamics, it is often the case that with high resolution meshes and increasingly accurate turbulence modelling, a steady solver does not always give a stable solution. In reality the flow is not stationary, therefore attempting to solve the problem by assuming it to be stationary will lead to difficulties in terms of solution convergence and stability. The next logical step is to solve the equations in a time accurate way using a transient methodology. To help confirm whether the difficulty of steady convergence is related to some numerical difficulties or is inherent in the physics being captured by our modelling, the same case is re-started in transient solver mode. Figure 9 shows the history from transient solution re-starting from the steady converged UD field and calculating with MARS scheme. This particular calculation history was made over a long period with several re-starts and adjustments in timestep from $1e-05$ sec to $2e-05$ sec. The large 12% over-prediction in massflowrate remains. The solution shows a high degree of instability, particularly for the torque.

Conclusions from conventional High-Reynolds simulations

- Wall function approach cannot be relied upon for predictive accuracy in Massflowrate better than 12%
- Higher order accurate scheme in combination with high grid resolution begins to reveal large scale unsteady structures and will therefore not permit convergence with a steady-state flow solver, indicating that a transient solution method must be applied.

4.2 Two-Dimensional Hybrid Low-Re Valve Seat Test Cases

As a preliminary testcase of the low Reynolds method and it's hybrid application, 2-D axisymmetric models were made with two valve seat profiles, 30 degree and 45 degree, for case-B shown in Fig.4. The cases were built at the same 6mm lift height which were problematic in the HRWF simulations. In Figures 10 and 11 it is very obvious that the HYLRL method is able to predict the flow separation resulting in a reduced massflowrate. In actual fact, the reduced massflowrate compared with the HRWF is almost precisely -12%, the same magnitude as the discrepancy seen in the 3D HRWF calculations.

For the case of the 45 degree valve seat, the influence of the HYLRL is even more apparent, see Fig.12. Note, there have been here no corresponding 3D port simulations made for the 45 degree seat profile.

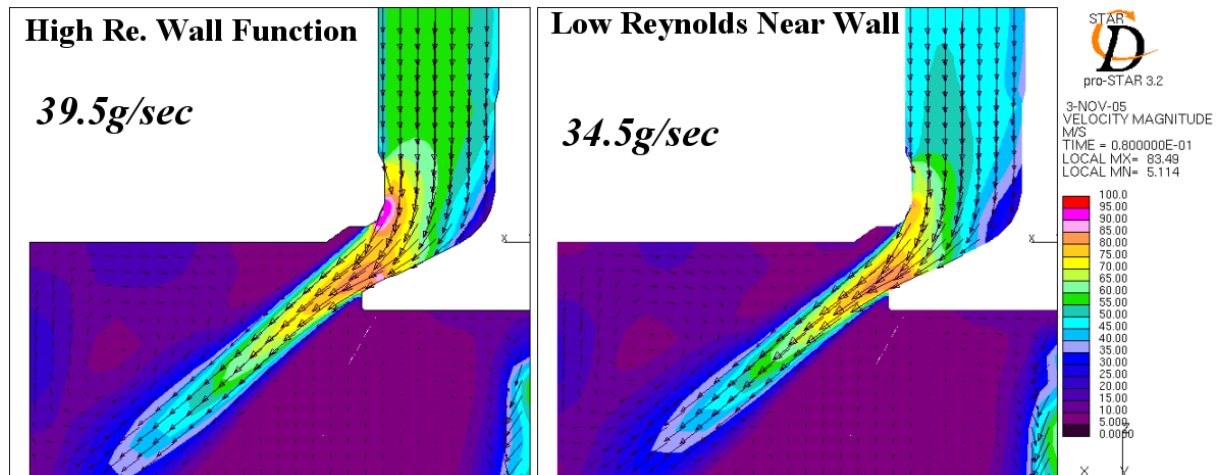


Fig. 10: 2D valve seat 30deg case

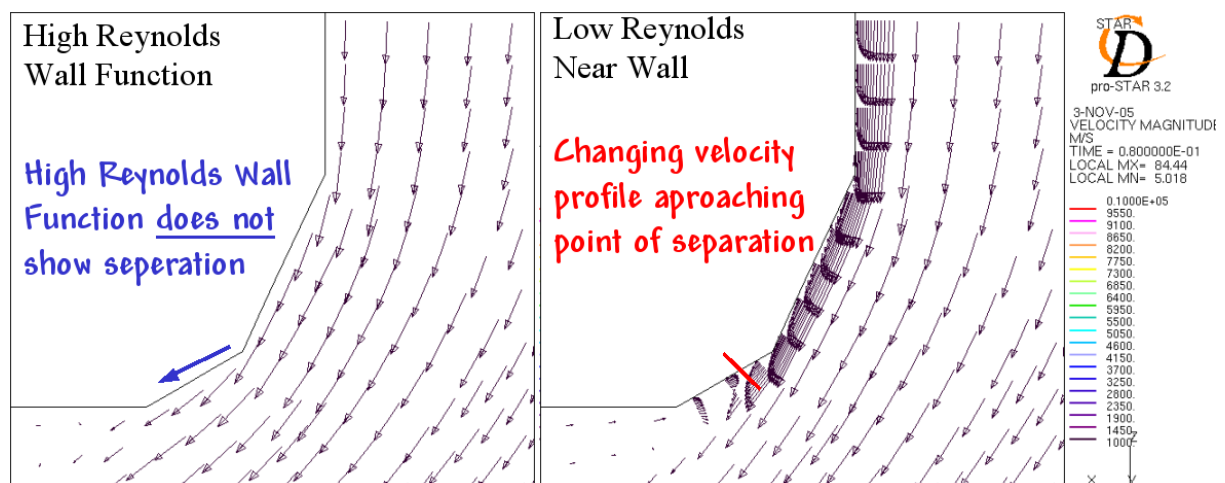


Fig. 11: 2D valve seat 30deg. case - Local separation

Figure 13 compares the field results from low-Reynolds simulation using three alternative turbulence models. It shows that performance of the low-Reynolds near wall method is strongly affected by the choice of turbulence

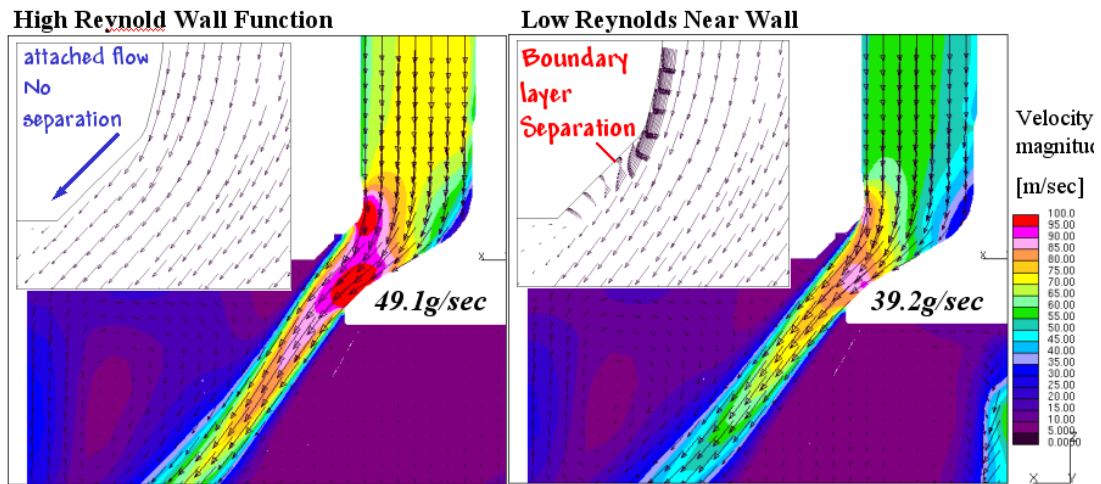


Fig. 12: 2D valve seat 45deg. case

model. In this example only the $k-\omega$ SST model of Menter [19] captures early separation. This result is not surprising since, as described by Hellsten [14], "...this model has been especially designed for predicting adverse pressure gradient flows where most of the two-equation models, and even higher order models, are known to fail severely."

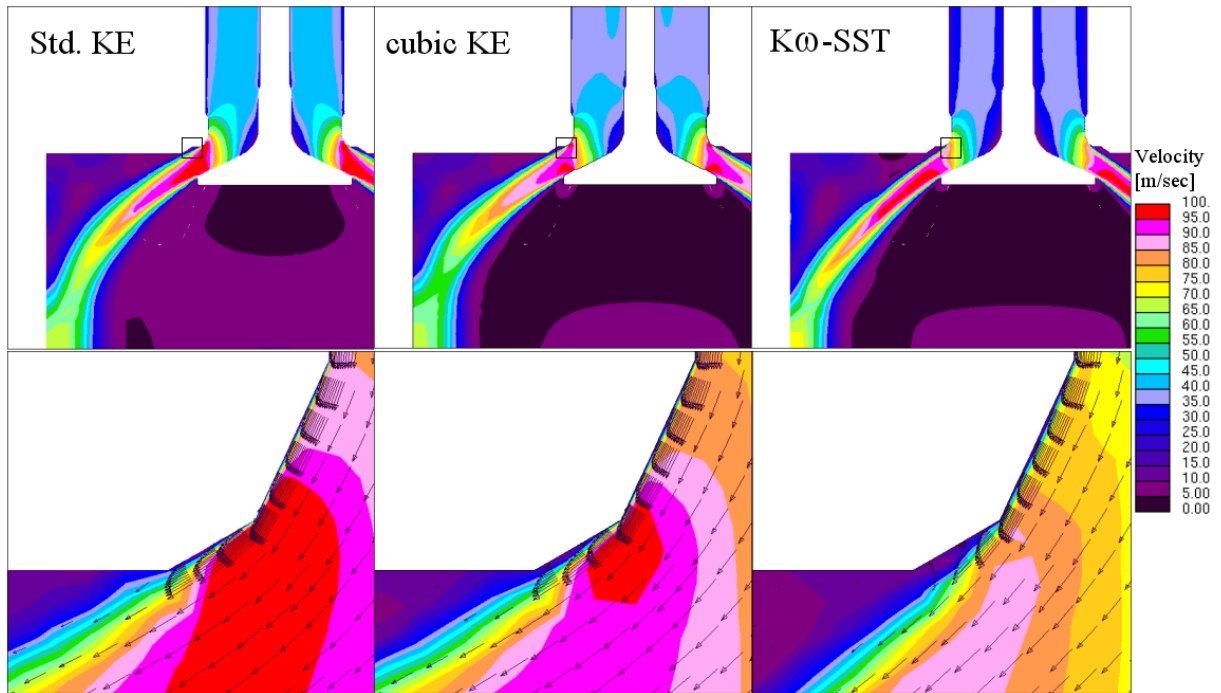


Fig. 13: Comparison of Low-Re EVM models for Separation at Valve Seat

Among other turbulence models which have not been tested in this work, Durbin's $k-\epsilon-v^2$ model (V2F)[10] is also recognised as being especially developed to better capture boundary layer separation. The V2F model is available in the STAR-CD code, although not in combination with the Hybrid near wall treatment. In all present 3D port studies shown in the following sections, the $k-\omega$ SST model has been applied.

4.3 Hybrid/Low-Reynolds (HYLR) Port simulations

Local Low-Reynolds applied to valve seat only (HYLR1)

In an attempt to realise the most economic benefit from improved near wall method, i.e. to limit the increase in number of computational cells, the first investigation applies mesh refinement only locally to the valve seat which may be considered to be the most critical region. The first results are presented under the name 'HYLR1a' in Fig.14 and show an immediate improvement in the level of predicted massflow.

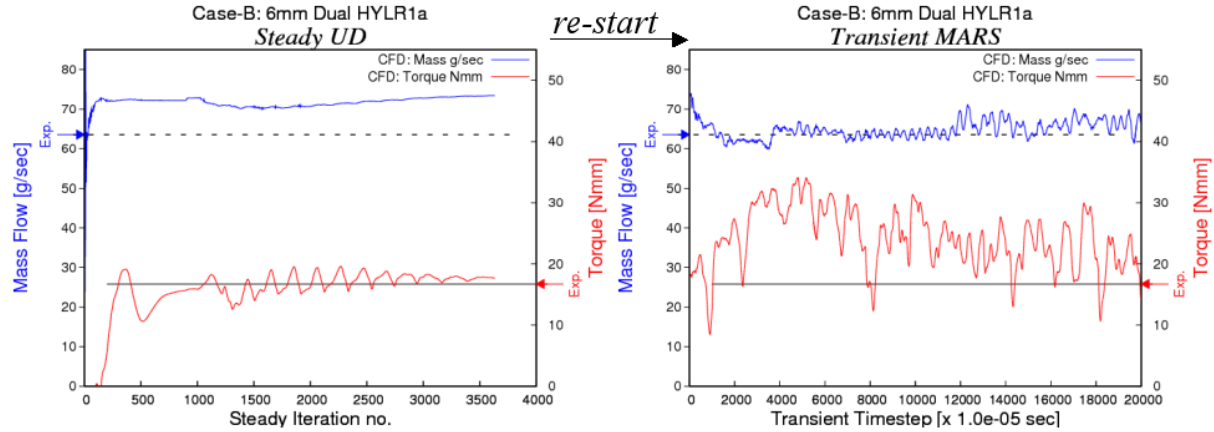


Fig. 14: Solution history for case-B 6mm lift Dual HYLR1a

Simulation of the dual port case may be considered more difficult because of the interaction of the jet flows from each of the two ports. Therefore it was considered to focus efforts first on the single port cases. In Fig.15 the result of calculating the Direct port only case-B at 6mm lift shows now larger and more periodic fluctuations in the massflow compared with the dual port case. This might be expected since we have removed the interaction of the two jet flows. The time averaged massflow looks in good agreement with measurement. The torque shows a large over-prediction compared to measured value and exhibits large irregular fluctuation.

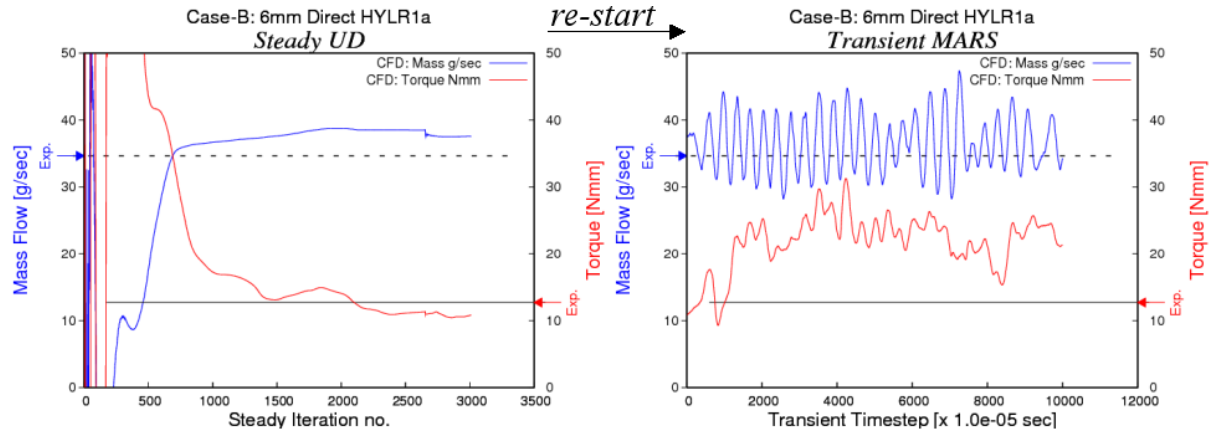


Fig. 15: Solution history for case-B 6mm lift Direct HYLR1a

At this point in the work, a new grid was constructed with some refinement and based on a custom polar template strategy. The new grid provides a better quality mesh for the purposes of local near wall refinement and more economic (in terms of less cells) extension of the near wall refinement for the studies which follow. This new grid is referred here as 'HYLR1b' and forms the basis from which the extended wall refined case of 'HYLR2' is built. A summary of the cell count for all the grids used is shown in table 4. The results of the HYLR1b case are shown in Fig.16 and are consistent with those of the HYLR1a case described previously.

Extended wall regions local Low-Reynolds (HYLR2)

Examination of the field results, in the case of the local valve seat low-Reynolds case (HYLR1), suggested that in addition to the valve seat there are several other regions where the occurrence of separated flow may play a critical role. These additional regions include the valve head in the vicinity of the throat, the valve stem and the port walls

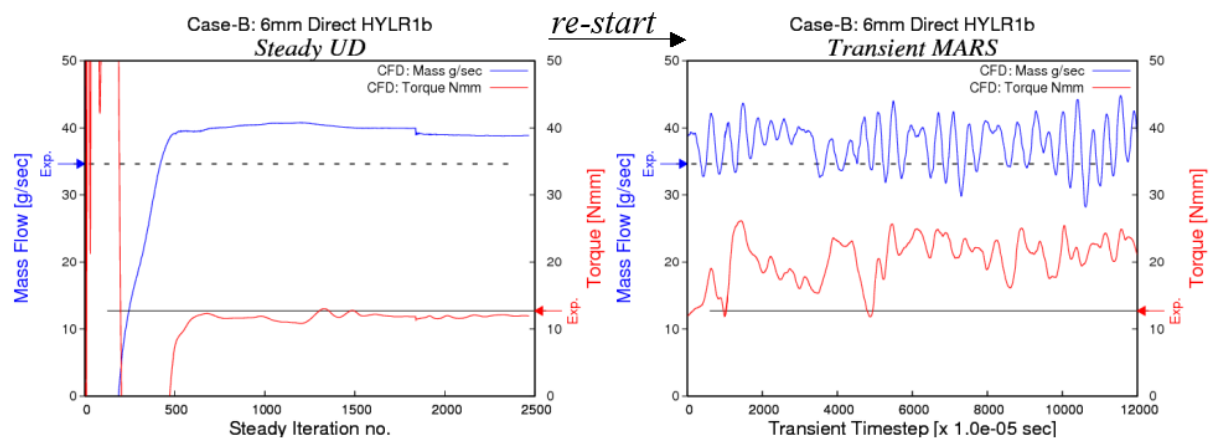


Fig. 16: Solution history for case-B 6mm lift Direct HYL1b

immediately upstream of the valve throat. A revised mesh (HYLR2) was created with these extended regions of low Reynolds near wall refinements.

Figure 17 shows a sample of the field result taken from each of the converged incompressible steady UD solutions for 6mm direct port HYL1b case and HYL2. The corresponding global history charts are shown in Fig.18. In the case of HYL2 model there is a substantial change in how the flow in the upper port approaches the stem/throat annulus, brought about by the separation of the flow in the roof of the port. Also, on the base of the port at the upstream edge of the valve seat, in case HYL2 the upstream extended low-Reynolds wall treatment appears to capture earlier separation of the flow which again modifies how the flow enters the port annulus surrounding the valve stem. This snapshot from the solution history, is example of the strong influence of extending the near wall treatment.

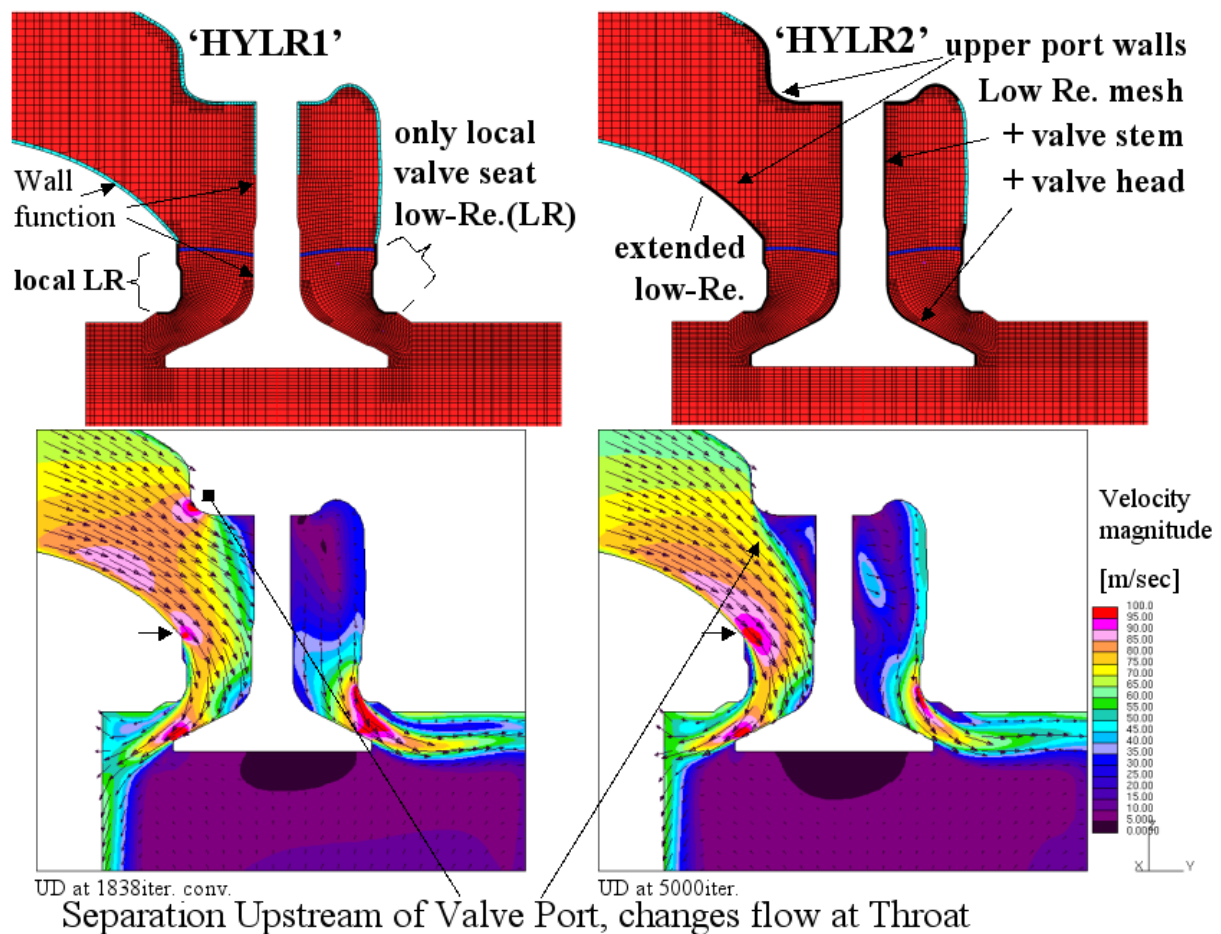


Fig. 17: Velocity Field change due to Extended Local Low-Reynolds

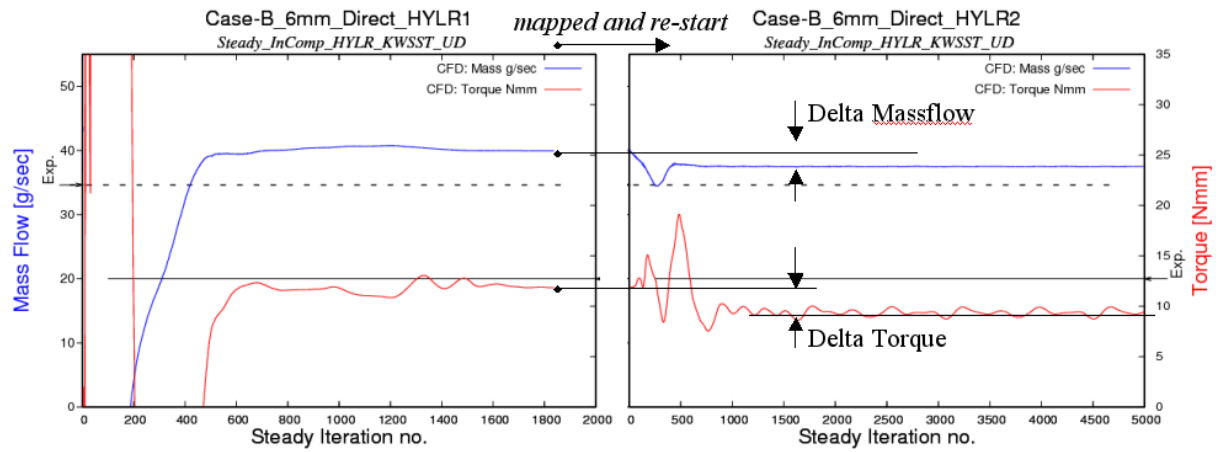


Fig. 18: Change from HYLR1b to extended HYLR2 Local Low-Reynolds on global Massflow and Swirl

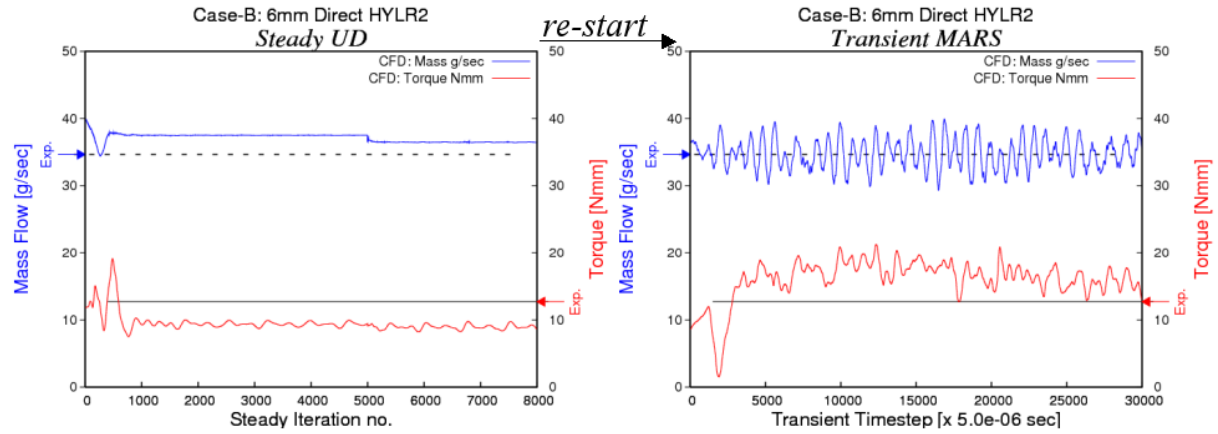


Fig. 19: Solution history for case-B 6mm lift Direct HYLR2

Figure 19 shows the solution history for the HYLR2 6mm lift Direct only port calculation. The massflow now exhibits large fluctuations but they appear periodic and the time average level is in close agreement with the measurement. Regarding the torque, it is noticeable how large an increase in level occurs when the MARS scheme is applied following the re-start from UD solution. This increase is associated with the discretisation error discussed in section 3.4. The

Figure 20 is the corresponding calculation history at 10mm lift for the HYLR2 case-B direct port. Here we see also strong periodic fluctuations in the massflow, similar to the 6mm case. The frequency spectra of this history is shown in Figure 27.

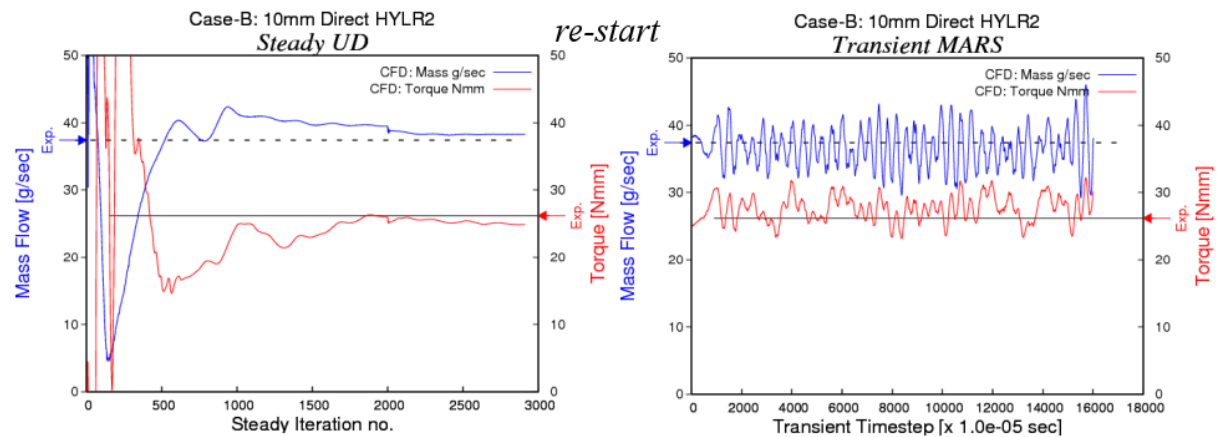


Fig. 20: Solution history for case-B 10mm lift Direct HYLR2

Figures 21 - 22 shows the global results history for the helical only case-B calculations.

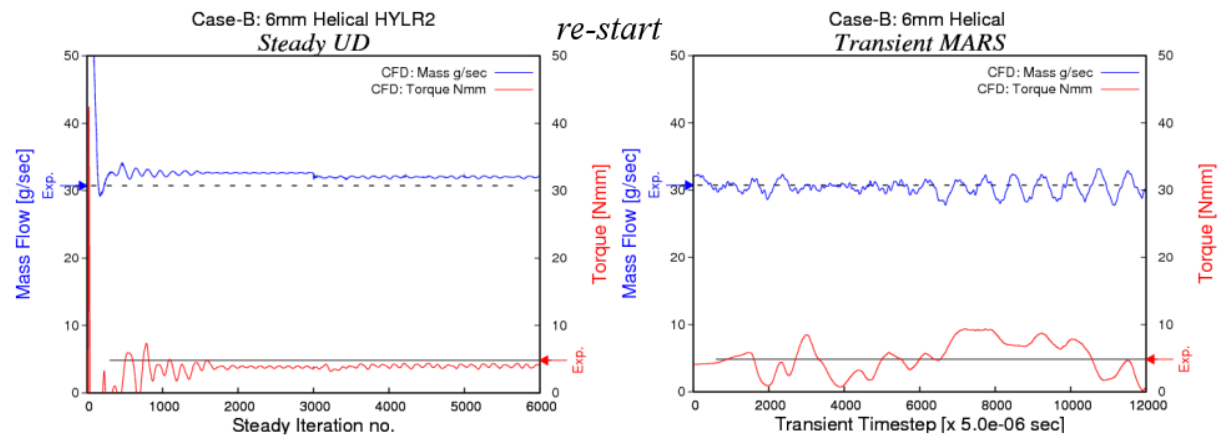


Fig. 21: Solution history for case-B 6mm lift Helical HYLR2

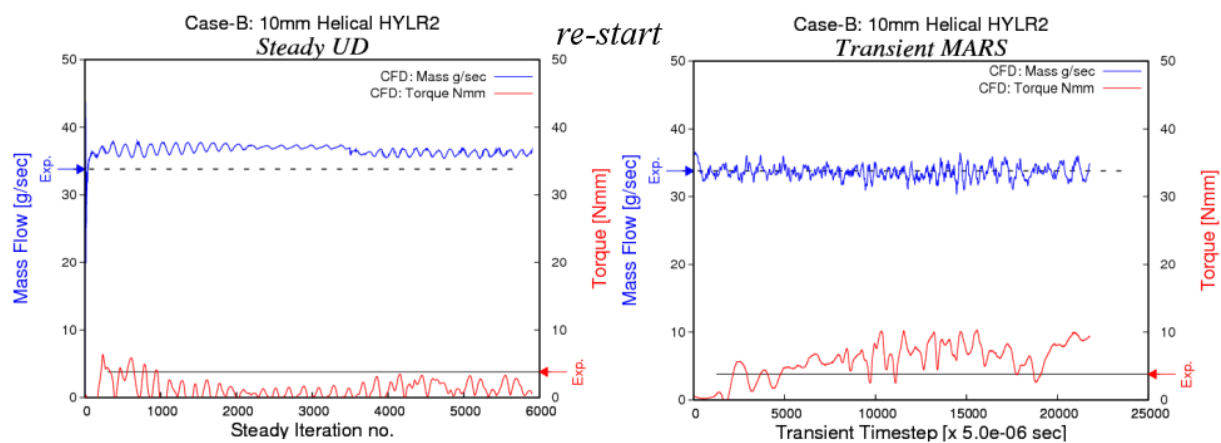


Fig. 22: Solution history for case-B 10mm lift Helical HYLR2

Finally, for the case-B, the results for the dual port case with HYLR2 model at 6mm and 10mm lifts are shown in Fig.23 and Fig.24. In both cases, the predicted levels of time-average massflow are in excellent agreement with the measured values. However, the torque prediction can be seen to fluctuate significantly and is substantially over-predicted for the 6mm lift case. The torque in the 10mm lift case, although fluctuating, does appear to have a time average level within 5% of the measurement.

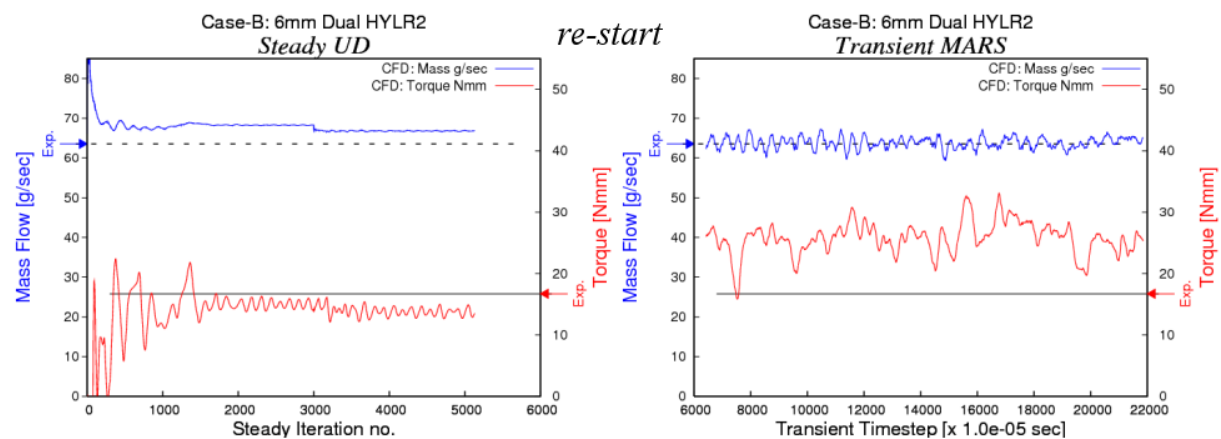


Fig. 23: Solution history for case-B 6mm lift Dual HYLR2

Applying the hybrid low-Reynolds approach to case-A, with extended wall regions as for case-B (HYLR2) gives the results shown in Fig.25 and Fig.26. The charts in Fig.25 show the steady calculation history with the

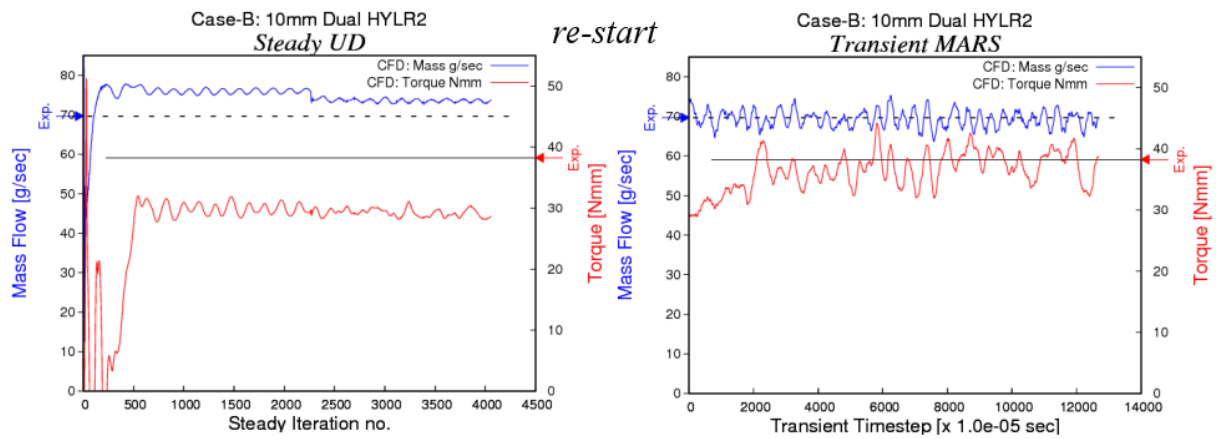


Fig. 24: Solution history for case-B 10mm lift Dual HYLR2

HYLR and may be compared directly with the previous HRWF results shown in Fig.7. In the 7mm lift case the HYLR results show much closer agreement with the torque level while retaining good massflow prediction. However, for the 11mm case, the massflow is now slightly under-predicted. Figure 26 shows the transient results corresponding to the case-A 7mm and 11mm HYLR calculations. The massflow level does appear to be stable in these case-A 7mm, however the torque shows strong irregular fluctuation.

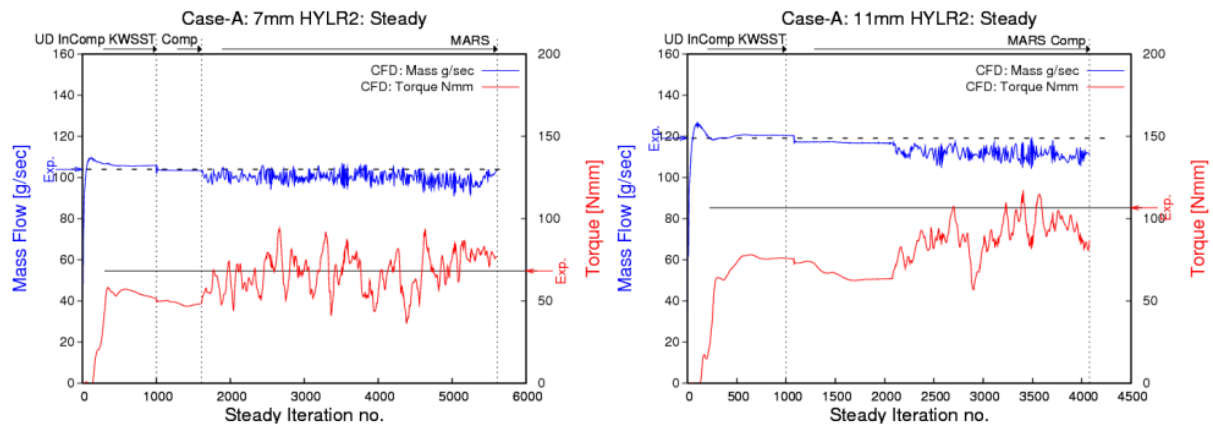


Fig. 25: Case-A 7mm and 11mm Lift HYLR2 Steady Iteration History, comparable with HRWF result in Fig.7

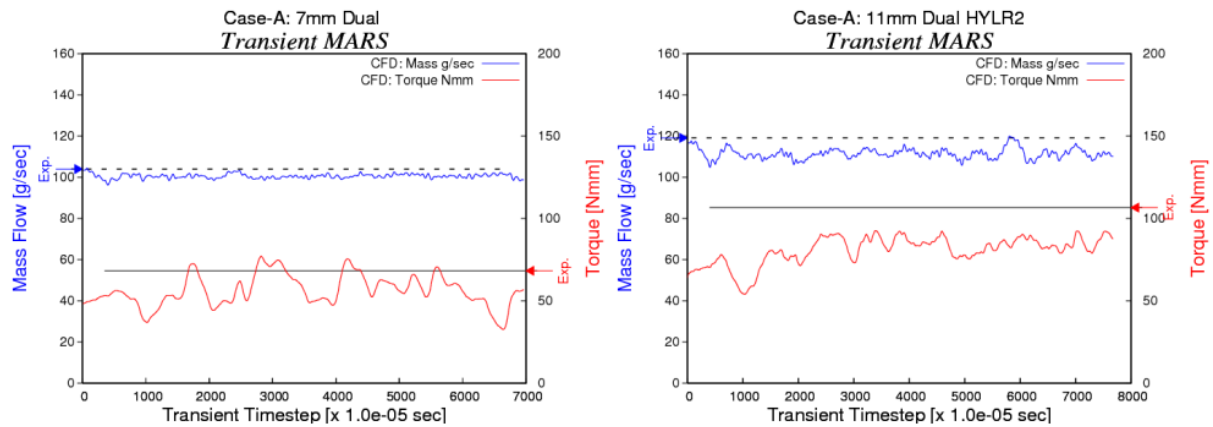


Fig. 26: Case-A 7mm and 11mm lift HYLR2 Transient Solution history

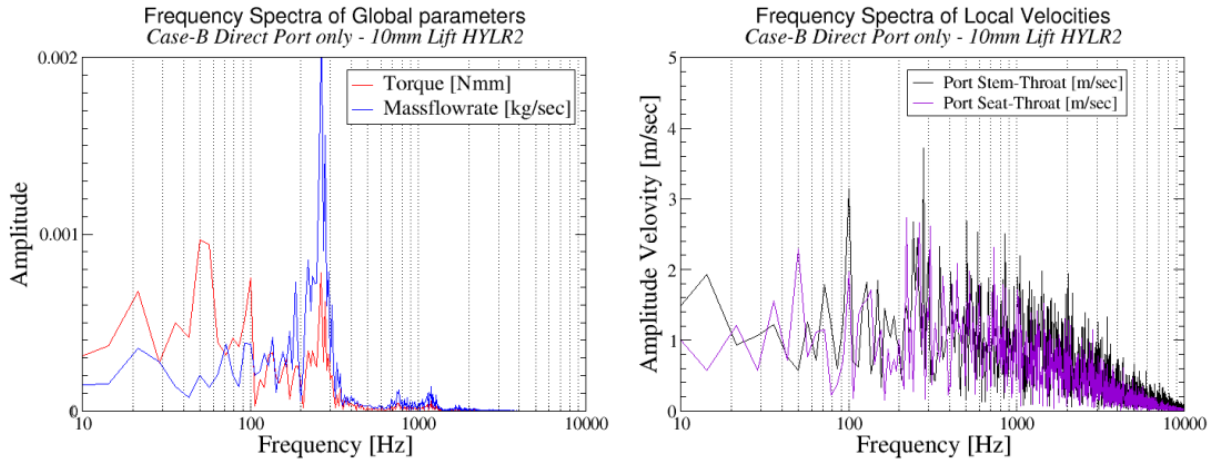


Fig. 27: Frequency analysis of Case-B Direct Port transient results

Stability analysis, Flow Visualisation and Animations

The emphasis in the present paper has been to compare the simulated global results for massflow and torque with the steady state experimental measurements. Unfortunately, in respect of assessing port stability, no experimental measurements are available. The design of the Impulse torquemeter contains oil damping which excludes possibility to detect fluctuations in the swirl field at frequencies indicated by the simulations. Measurements using acoustic techniques have not yet been attempted but are suggested for future work.

Frequency analysis of the CFD global data for the case-B Direct port at 10mm lift (Fig.20) is shown in Fig.27. The spectra shows a peak amplitude of fluctuation occurring at approximately 300Hz. This value is lower than the frequency of 1400Hz estimated for vortex shedding at the valve stem (see Table 2). This might suggest that vortex shedding at the stem is not occurring and another larger structure such as the base of the valve head and its wake, is the main source of unsteadiness. However, analysis of the the transient history of local velocities at the valve stem and throat, see right Fig.27 shows similar but broader spectra of frequencies including those at circa 1000Hz. This brings us to the analysis of the CFD field results. As with any multi-dimensional CFD, especially transient simulations, there is an enormous quantity of field information available and it is time consuming to process it. So far, only limited visualisation of the calculated flow fields has been made. The most revealing have been animations generated from the case-B direct port which show clearly vortex shedding occurring from the valve stem and interfering with the downstream jet leaving the port. The strong effect on the global parameters is evident in figures 15,16,19,20 which have distinctly larger and more regular periodic fluctuations than in the other dual or helical port cases. As indicated by the spectra shown in Fig.27, the frequency of fluctuations are in the same order of magnitude as, although rather lower than, the frequency from a simple Strouhal estimate. This is plausible for the complex three dimensional flow inside the port.

5 Conclusions

This work has investigated two main aspects of diesel engine port flow under stationary conditions.

1. Flow separation accuracy and prediction of global characteristic values of massflow and swirl.
2. Flow stability characteristic.

On the first aspect, the application of a new hybrid near wall approach is demonstrated to yield significant improvements to the accuracy of massflow prediction. On the second aspect, the application of transient URANS method is shown to have the potential to capture the large scale flow instabilities and is therefore a useful tool for investigating engine intake port instabilities.

- High resolution grid (cell count ≥ 1 million) combined with 2nd-order accurate numerical discretisation scheme begins to capture the inherent unsteady flow structures.
- Transient RANS method confirms that flow instability may be detected by the simulation and therefore has potential to provide a further characteristic for comparing performance of valve-port design.

- A hybrid near wall method permitting local low-Re near wall method in the critical valve seat region has been demonstrated to provide improved predictive capability for mass-flow rate by virtue of more accurate prediction of the flow separation location around the valve seat.
- Extending the low-Re near wall treatment to upstream wall surfaces within the port demonstrates further large changes in the port flow due to flow separation upstream of the valve seat region.
- Despite significant improvement in mass flow-rate prediction, the predictive accuracy of the swirl torque is unsatisfactory. The unsteady nature of the simulation also makes it difficult to interpret a mean torque level.

There are numerous possible reasons to explain why the torque prediction remains disappointing. First, it is most likely that turbulence modelling, particularly within the framework of EVM, remains still a long way from being adequate in the flow regime experienced by an engine port. Second, the transient nature of the flow may involve a longer starting-up/initial period before a representative condition is reached. Such behaviour could be dependent upon the boundary conditions, so that extended calculation using the full rig model domain may provide some answers. It is interesting to note that, on the experimental rig, a period of at least a few minutes is allowed prior to measurements being made.

Regarding turbulence modelling, the k - ω -SST model, despite the good performance regarding separation prediction, has been noted from recent testing [29] to perform rather badly in confined channel swirling flow. This is perhaps not surprising given that k - ω -SST model uses the Boussinesq linear constitutive relation. Correction to k - ω -SST model has been proposed by Hellsten [14] to make the model sensitive to flow curvature; this could be worth testing. As noted previously, the V2F model [19] is also recognised as being specially developed to better capture boundary layer separation and would seem an excellent candidate for future calculations. The non-linear k - ϵ - A_2 model is also a favourable candidate, as already demonstrated in [17] for a generic intake port test case. An outside candidate is the model of Spalart-Allmaras (S-A) [25] which is applicable in both near wall refined grid and freestream regions, it is tolerant of a wide range of y^+ , is robust and may provide an economic solution particularly given the expense of necessary transient calculation. Consideration of the S-A model leads to the suggestion of Detached Eddy Simulation (DES) approach in which the S-A model is best known. This brings us finally to any of the new generation of LES/URANS hybrid solutions currently under development.

A final conclusion, in respect of the necessity of transient solution, which now makes the study of steady-state engine port experiments using CFD an expensive activity. As noted in the abstract to this paper, given the significant computational expense, one might as well concentrate CFD resource on the more realistic case of the induction stroke calculation. The pitfall of this strategy, arises in the present limitations of near wall methods being used in moving mesh engine piston-valve simulation, namely HRWF. As demonstrated in this study, reliance on the HRWF method is a source of considerable error and does not provide a predictive capability when examining novel geometry features, exemplified by the port test cases shown here.

Potential Role of URANS method to understand engine Cyclic variations As suggested by some studies [9, 13] the large scale instabilities which occur during the intake phase can be a source of engine cyclic variation. Contemporary opinion seems to be that only LES can provide the necessary information to explain engine cyclic variations. However pure LES involving resolution of turbulence through to the wall boundary will remain prohibitively expensive in the near term. Progress is being made with development of hybrid LES/URANS methods. DES is a first possibility, but there are numerous others. A common requirement in all these new methods is to resolve in an accurate way the boundary layer. The adoption of URANS method for the near wall seems to become an established practice in most new CFD methods under development. Consequently, the present work makes a contribution by testing the latest URANS low-Reynolds method on a realistic geometry and emphasises the importance of the near wall behaviour for diesel engine port simulations. The present studies also show that an Unsteady RANS simulation, even when using a humble EVM turbulence modelling, has potential to capture the large scale unsteady structures which are sources of cyclic swirl variability.

Acknowledgements

The authors would like to express their gratitude to Iveco Motorenforschung AG for contributing experimental results to support this work. Also thanks go to Prof. A.D.Gosman (Imperial College, London) for valuable consultation.

References

- [1] *STAR-CD v3.2 Methodology*. CD-Adapco, London, 2005.
- [2] M. Auriemma, F. E. Corcione, U. Di Martino, and G. Valentino. Analysis of the intake flow in a diesel engine head using dynamic steady flow conditions. *SAE technical journal*, (2001-01-1307), 2001.
- [3] B. Basara. Calculation of steady flow through tangential dual intake port. In *Proceedings of FEDSM'98*, number FEDSM98-4896. ASME Fluids Engineering Division, June 1998.
- [4] B. Basara. Employment of the second-moment turbulence closure on arbitrary unstructured grids. *International Journal for Numerical Methods in Fluids*, 44:377–407, 2004.
- [5] B. Basara, V. Przulj, and A. Alajbegovic. Calculation of steady flow through tangential dual intake port. In *Proceedings of FEDSM'00*, number FEDSM2000-11330. ASME Fluids Engineering Division, June 2000.
- [6] G.M. Bianchi and S. Fontanesi. On the application of low-reynolds cubic k-epsilon turbulence models in 3d simulations of ice intake flows. *SAE Technical Paper*, (2003-01-0003), 2003.
- [7] G.M. Bianchi, G. Cantore, and S. Fontanesi. Turbulence modelling in cfd simulation of ice intake flows: The discharge coefficient preview. *SAE Technical Paper*, (2002-01-1118), 2002.
- [8] B. van den Heuvel, W. Willems, E. Karvounis, and H. Schulte. In-cylinder flow characterization of modern hsd diesel engines-numerical and experimental evaluation of current practices. In *Thiesel 2004*, 2004.
- [9] D.L. Reuss. Cyclic variability of large-scale turbulent structures in directed and undirected ic engine flows. *SAE technical journal*, (2000-01-0246), 2000.
- [10] P.A. Durbin. Separated flow computations with the k-epsilon-v2 model. *AIAA*, 33:659–664, 1995.
- [11] S.H. El Tahry. k-epsilon equation for compressible reciprocating engine flows. *AIAA J. Energy*, 1983.
- [12] G. Bella, A. DeMaio, and C.N. Grimaldi. Flow characterization of a high performance s.i. engine intake system - part 2: Numerical analysis. *SAE Technical Paper*, (2003-01-0632), 2003.
- [13] H. Bensler, L. Kapitza, J. Raposa, and U. Reisch. A new experimental method for determining port generated swirl flow. *SAE technical journal*, (2002-01-2846), 2002.
- [14] Antti Hellsten. Some improvements in menter's k-omega sst turbulence model. *AIAA*, 1997.
- [15] J.V. Pastor, X. Margot, A. Gil, and J.C. Donayre. A methodology to estimate the swirl number at tdc in di diesel engines: Through the combination of cfd and steady flow rig results. *SAE Technical paper*, (2004-01-1876), 2004.
- [16] Bahram Khalighi, S. Zhang, C. Koromilas, S.R. Balkanyi, Luis P. Bernal, G. Iaccarino, and P. Moin. Experimental and computational study of unsteady wake flow behind a bluff body with a drag reduction device. *SAE Technical Paper Series*, (2001-01B-207), 2001.
- [17] K. Suga, M. Nagaoka, N. Horinouchi, K. Abe, and Y. Kondo. Application of a three-equation cubic eddy viscosity model to 3-d turbulent flows by the unstructured grid method. *International Journal of Heat and Fluid Flow*, 2001.
- [18] F.S. Lien, W.L. Chen, and M.A. Leschziner. Low-reynolds-number eddy-viscosity modelling based on non-linear stress-strain/vorticity relations. In *Proc. 3rd Symp. on Engineering Turbulence Modelling and Measurements, Crete, Greece*.
- [19] F. Menter. Zonal two equation k-omega turbulence models for aerodynamic flows. *AIAA*, (paper-93-2906), July 1993.
- [20] S. Nadarajah, S. Balabani, M.J. Tindal, and M. Yianneskis. The effect of swirl on the annular flow past an axisymmetric poppet valve. *Proc. IMechE, Part C, Journal of Mechanical Engineering Science*, 1998.
- [21] S. Nadarajah, S. Balabani, M.J. Tindal, and M. Yianneskis. The turbulence structure of the annular non-swirling flow past an axisymmetric poppet valve. *Proc. IMechE, Part C, Journal of Mechanical Engineering Science*, 1998.
- [22] Henrik Nordgren, Leif Hildingsson, Bengt Johansson, Lars Dahlen, and Dennis Konstanzer. Comparison between in-cylinder piv measurements, cfd simulations and steady-flow impulse torque swirl meter measurements. *SAE technical journal*, (2003-01-3147), 2003.
- [23] N. St. Hill, P. Asadamongkon, and K.C. Lee. A study of turbulence and cyclic variation levels in internal combustion engine cylinders. In *Proc. 10th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal*, volume 22.4, July.
- [24] S.E. Kim and D. Doudhury. A near wall treatment using wall functions sensitized to pressure gradient. *ASME, FED*, 217, 1995.
- [25] P. Spalart and S.R. Allmaras. A one-equation turbulence model for aerodynamic flows. *AIAA*, 1992.
- [26] Rung T. Formulierung universeller wandrandbedingungen fuer transport-fleichungsturbulenzmodelle'. Technical Report 02/99, Technische Universitaet Berlin, 1999.
- [27] S. Tananbe, H. Iwata, and y. Kashiwada. Instability of steady valve flow in a swirl port. In *International Symposia COMODIA 90*, page 419:424, 1990.
- [28] B. Wegner, A. Maltsev, C. Schneider, A. Sadiki, A. Dreizler, and J. Janika. Assessment of unsteady rans in predicting swirl flow instability based on les and experiments. *International Journal of Heat and Fluid Flow*, 25, 2004.
- [29] M.I. Yaras and A.D. Grosvenor. Evaluation of one- and two-equation low-re turbulence models. parti-axisymmetric separating and swirling flows. *International Journal for Numerical Methods in Fluids*, 42:1292–1319, 2003.