S966/006/2004

Closed valve flow field investigation using computational fluid dynamics

G DYSON
David Brown Union Pumps, Penistone, UK
J A TEIXEIRA and **P C IVEY**
School of Engineering, Cranfield University, UK

SYNOPSIS

The flow field of a centrifugal pump operating at closed valve was investigated using Computational Fluid Dynamics (CFD). The rotor stator interactions were shown to directly link to the developed flow regime. The impeller flow regime was divided into two distinct areas. The upper proportion at the discharge vanes contained a vortex, the size and motion of which was influenced by the vane number ratio of stator to rotor vanes and the position of the rotor vanes with respect to the stator vanes

The lower proportion of the impeller was filled with a strong flow reversal, which developed from the impeller hub and traversed to the shroud side spiralling out of the impeller eye into the suction duct. The suction duct flow regime contained a helically spiralling outflow near to the duct wall and a spiralling inflow core driven by the viscous forces imposed by the outflow regime. The pump casing showed a non-linear pressure development around the volute influenced by the stationary collector. The collector passages, which connect to the pump discharge, experienced pressure pulsations at the vane passing frequency.

1 INTRODUCTION

Flow field investigation within a centrifugal pump, operating at closed valve, has been the subject of little dedicated investigation. A study of the flow physics is difficult; the nature of the flow is unsteady with strong impulsive flow reversals within the impeller and pressure pulsations within the casing. These effects are aligned to the rotor stator interactions. Early investigations by Simpson and Cinnamond 1964(1) identified strong outlet eddies at the impeller periphery imposed above an inlet eddy encompassing two thirds of the impeller passage length. Pfleiderer 1961(2), Frost and Neilsen 1991(3) and Stirling 1982(4) used these observations as the basis for their closed valve pressure prediction methods.

These prediction methods assume that a proportion of the closed valve pressure is analogous to a solid rotating disc. Although the methods differ on their definitions of the internal diameter of this disc, they all use the impeller outside diameter as a major influence on the developed pressure. This is not surprising at it broadly concurs with the Euler analysis proposed in Stepanoff 1957(5). All available closed valve pressure prediction methods ignore the unsteady rotor-stator interaction effects. Despite this they are reasonably successful at prediction as evaluated by Dyson 2002(6).

Levin and Poliokovsky 1965(7) identified strong inlet re-circulation flows into the pump suction duct in a centrifugal pump impeller, running without a volute, at closed valve. These inlet re-circulation structures are reinforced by the Palgrave 1985(8) work on inlet re-circulation at partial flow. Frost and Neilsen and Yedidah 1993(9) proposed volute flow mechanisms, which assume a steady flow regime, Newton 1988(10) in his work on a centrifugal fan at closed valve points out the effect of the rotor-stator interactions at closed valve. Newton 1988 and Kaupert 1999(11) show in their work that a non-linear pressure rise around the volute exists.

Computational Fluid Dynamics has been previously applied to the closed valve problem by Newton with limited success. The qualitative prediction of the flow regime shows reasonable synergy with experimental data, but quantitatively the code over predicted the volute pressure rise by 165%. More recent work by Sun 2001(13) has proved successful in accurately predicting the pumps performance envelope.

CFD and computational capacity have now improved to such an extent that a commercial CFD code, CFX Tascflow can be used to accurately predict the flow regimes. This paper proposes a more complete picture of the flow field within a centrifugal volute pump operating at closed valve. The unsteady Rotor-Stator interaction effects are mapped out for impeller, collectors and suction channels.

2 CLOSED VALVE HEAD

When a pump runs at closed valve, the flow across the discharge valve is zero. The flow passing through a pump is not zero. A finite amount of liquid is passed through the wear ring landings and back to the pump's low-pressure inlet (Fig 1). This leakage is typically between 1-5% of the total best efficiency point flow, depending on the specific speed of the machine and the wear ring geometry.

Designers quantify the amount of leakage flow and consider it when using their streamline methods. When considering closed valve problems the difference between closed valve pressure with leakage and without leakage is considered negligible for the example provided.

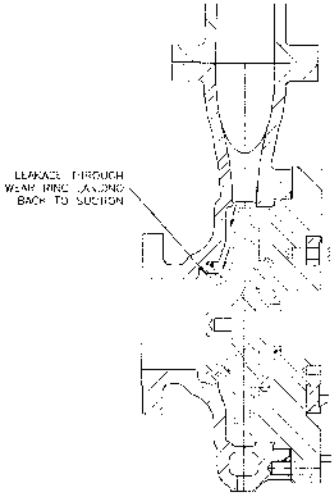

Fig 1 Pump leakage back to suction

The CFD methodology employed within this paper exploits the small differences in flow between the casing and impeller. The difference is used to ease the computational solution of the closed valve problem by allowing a finite flow, equivalent to the leakage flow, to pass through the outlet boundary. The impeller flow regime represents the actual flow field experiences when the machine is in operation. The casing flow is equivalent to extreme low-flow. The commercial software package CFX 5.6 requires an outlet boundary condition. This requirement is satisfied by the allocation of the leakage flow to the boundary.

The inlet boundary condition selection for the solution is important. Inlet boundary conditions are set as a pressure level. It is important that this boundary condition is at such a distance, away from the impeller eye, so the liquid expelled from the inlet as observed by Palgrave has dissipated and mixed out with the inflow over a distance of up to 20 multiples of the eye diameter. Little literature is available on the validity of boundary conditions which accept both inlet and outlet mass flux at regions prescribed purely by the solver. To achieve the satisfactory boundary conditions it is important to model not only the pump but also a proportion of the inlet and discharge piping up to the discharge valve.

The work by Sun included a modifier to the boundary conditions based on mass flow and the interaction effects of the flow with the system. This in part explains the importance of the system and the pumps reaction to it when investigating off-design performance as demonstrated by Kaupert 1999. In the case of zero flow such a modifier is replaced, in this instance, with modelled elements of the system piping.

3 CFD PREDICTIONS

The analysis and description of the pump flow physics is split into three elements. The descriptions of suction piping (Fig 10-11), collector (Fig 7-9) and impeller flows (Fig 3-5) are considered separately within these sections. The interactions between the components are covered with the effect of different collector vane numbers.

3.1 Impeller Flow

The mental model proposed by Simpson and Cinnamond 1964 for flow within the impeller defines the inner proportion of the impeller as filled with a standing vortex of a common radius of ½ of the impeller diameter. The proportion of liquid trapped above it drives this vortex, forcing fluid to re-circulate through the impeller eye and into the suction duct. The flow phenomenon from the CFD analysis generally conforms to the observations but superimposed on this steady state picture must be the unsteady effects.

3.2 CFD Impeller Flow Description For a Volute Pump (see Figs 2 and 3)

Position A (Fig 2 & 3) – The upper proportion of the impeller passage, close to the discharge area of the impeller, is filled with a standing eddy. The viscous forces applied by the stalled collector passages drive this eddy as the fluid around the impeller periphery is squeezed between the volute lip and the approaching impeller blade. It is further drives the intensity of the inlet vortex also.

Position B (Fig 2 & 3) – This represents the interface between the discharge rotating eddy and the inlet backflow eddy. This suction eddy is characterised by a local separation from the pressure side of the blade, close to the hub. This local development is manifest as the volume flux is reduced, becoming more stable as the flux approaches zero. Unsteadiness of the flow develops coupled to transient operational loading. At closed valve this complex inlet discontinuity gives rise to a span wise flux from pressure to suction side. The flow re-circulation zone viewed from the relative frame is characterised by this standing eddy rotating counter to the machine rotation.

Position C (Fig 2 & 3) – At this position the flux stream has passed from the hub across the passage span to the blade suction side in the meridional plane. The region is associated with strong secondary flow phenomenon and three-dimensionality, as the inlet eddy is driven out from the impeller and into the suction channel as described in section 5.

Position D (Fig 2 & 3) – The largest proportion of the streamline passage is characterised by strong flow re-circulation at mid position on the blade pressure side. The vortex, driven by the viscous interaction with the fluid trapped within the volute, experiences stochastic fluctuations that are characterised by the size of the discharge vortex with respect to the volute lip position. When a blade approaches the volute lip the relative volume flux is squeezed between the lip and the rotor-stator interface gap[1].

This suppresses the vortex development by providing a firm viscous boundary. As the impeller blade moves past the volute lip the diffusive contribution of the volute decays the intensity of the trapped casing fluid, allowing the impeller discharge vortex to move forwards along the pressure side of the impeller blade. A snapshot of this phenomenon can be seen in Fig 2 by comparison of the flow regime in individual impeller passages. The arrows in Fig 2 represent the position of the volute lips.

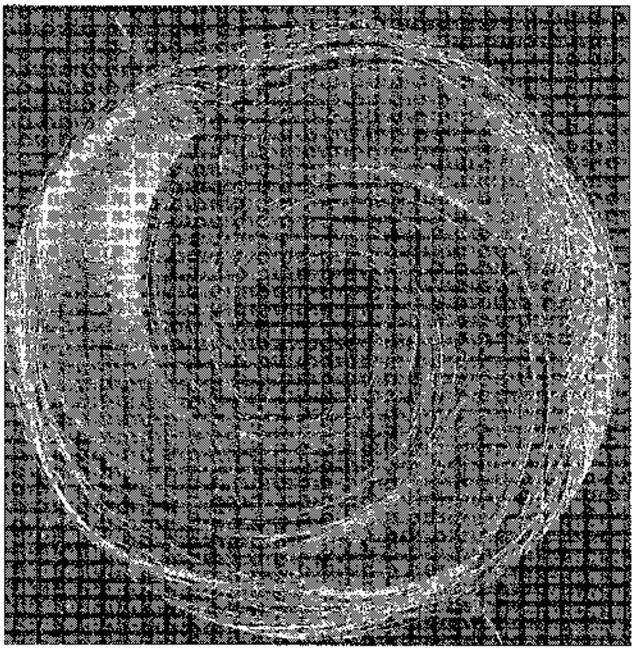

Fig 2 Impeller Streamlines at Closed Valve Low Solidity Impeller

[1] ~This can be as small as 2% of the impeller maximum diameter. For centrifugal pumps reducing this gap has a beneficial effect. The wake shed by the upstream rotor is stretched out into the stator passage generating a smaller mixing loss than if the wake were completely mixed out before entering the volute.

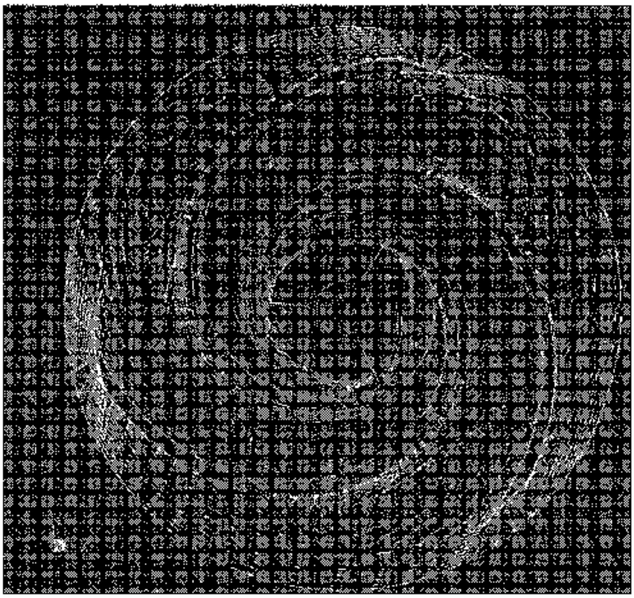

Fig 3 Velocity Vectors Within 3 Vane Pump Impeller at Closed Valve for a Low Solidity Impeller

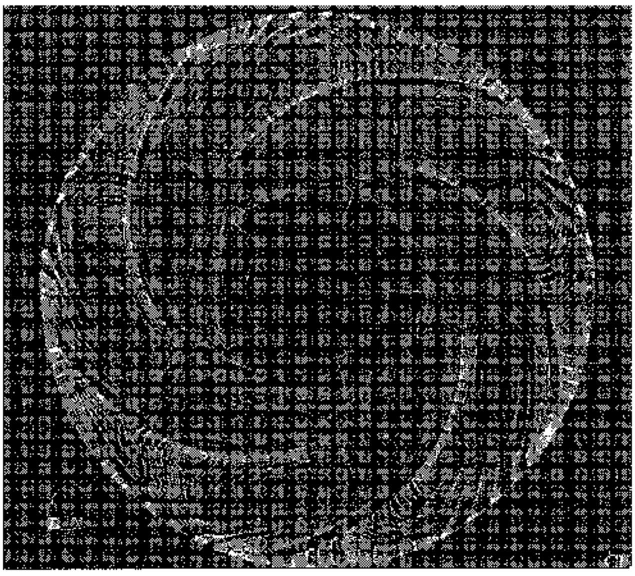

Fig 4 Velocity Vectors Within 5 Vane Pump Impeller at Closed Valve for a Medium Solidity Impeller

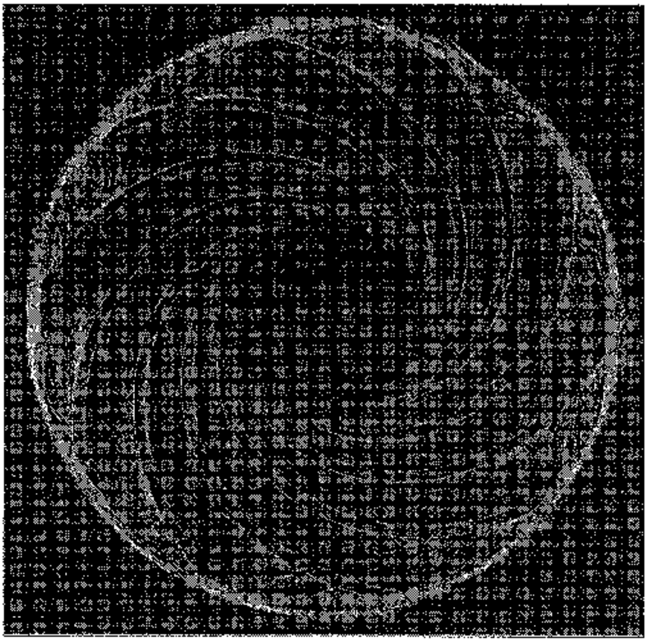

Fig 5 Velocity Vectors Within 10 Vane Pump Impeller at Closed Valve for a High Solidity Impeller

3.4 Blade Geometry Effects on Discharge Eddy Development

The vortex driven by the interaction with the stalled volute flow is a phenomenon, which is not consistently accounted for in the literature. It has been previously suggested by Newton that it is difficult to see and may have been missed by researchers. CFD studies consistently indicate its presence.

The vortex is linked to the position and number of blades in both the rotor and stator. The physical size and its development within the impeller passage is controlled by the number of impeller vanes. Its rotational direction and speed is firmly linked to the number of stator vanes.

For a low solidity impeller design with a low blade number, the vortex develops in the radial direction down into the impeller passage until it fills 50% of the channel (Fig.3) The intensity of this vortex increases as the impeller blade approaches the casing cut water as the driving viscous force of the liquid trapped between the lip and tip becomes dominant.

The converse is true when the impeller blade solidity is increased. The vortex at impeller exit extends in the radial direction only 20% into the impeller passage (Fig 5). Comparisons of Fig 3-5 indicate the extent of steady state vortex development within the passage with respect to increasing solidity. Experimental observations, which refer to this vortex, such as Acosta and Bowerman 1957 (14) define its motion as counter to the machine motion i.e. it rotates at a sub-synchronous speed counter to the pump rotation.

Whilst in the limited experimental studies available, this observation is true, it neglects the importance of the rotor-stator interaction between the varying rotor and stator blade systems which have a direct influence on the this direction of vortex motion.

3.5 Impeller blade number less than stator blades

Considering a case where the impeller blade number is less than the number of stator blades the vortex will appear to move counter to the machine rotation.

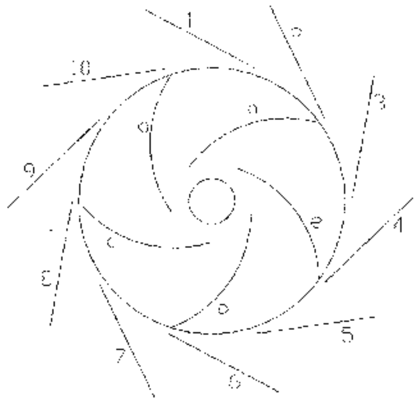

Fig 6 Representations of Impeller and Diffuser Vanes

Fig 6 represents a machine with a 5-vane impeller and a 10-vane diffuser. Passages 2,4,6,8 and 10 are congruent with impeller blades a-e. The standing vortex within the impeller exit, influenced by the stator vanes, will be identical within each of these impeller passages When the impeller has rotated 0.5 of the impeller blade peripheral pitch, this situation will be repeated as stator blades 3,5,7,9 and 1 become congruent with the impeller blades. In this manner the stalled vortex at impeller exit appears to rotate backwards, counter to the machine rotation.

The overall impression when the blade rows are viewed in entirety is one of 5 identical flow regimes occurring simultaneously. As the impeller rotates the subsequent blade encounters the identical influence of a subsequent stator blade after 0.5 impeller pitch rotations. This identical flow regime experienced after this 0.5 pitch rotation gives the impression that the vortex has passed counter to machine rotation at twice the synchronous speed when viewed from the relative frame.

If the impeller blade number and the stator blade number[2] are equal the vortex will appear not to move, as the pitch change influence of the stator is equal to the rotor at any instance. Each impeller passage experiences identical influence from the stator at the same instance.

[2] Pump Designers try to avoid this situation to limit the vibration potential of the machine

When the impeller vanes number is greater than the stator number the vortex will appear to move in the same direction as the machine rotation.

To put this into the context of a simple single volute pump, as analysed by Newton, the influence of the stator would occur once per blade pitch and no stator influence is passed onto subsequent vanes. It is understandable why some researchers did not register the phenomenon of the counter-rotating eddy, as it is entirely dependent upon the influence of the rotor and stator vanes.

4 VOLUTE CONTRIBUTIONS

When an impeller and casing reach a balance of angular momentum exchange, the centrifugal pump operates at its most efficient flow point. At extreme part load the miss-match of angular momentum causes the flow to decelerate within the volute.

At best efficiency point the pressure distribution around the spiral volute is linear, as the diffusive nature of the volute areas are designed with best efficiency operation in mind. The converse is true when a pump operates at zero flow. The reported non-linear pressure rise, Kaupert 1999, within the volute can be attributed to the stalled nature of the pump discharge passage.

Both discharge passages within the double volute pump contain low energy liquid filling the discharge volutes. This discharge blockage represents a barrier to the outlet flux of the impeller. As the impeller vane passes the casing throat the impeller outflow is forced between impeller vane tip and the volute lip, as it cannot escape into the outlet ducts.

This accounts for the non-linear nature of the time averaged pressure field as the pressure energy recovered within the diffusive volute around the impeller is transformed into low pressure, high velocity liquid Although this model is representative of the time-averaged situation it does not take into account the unsteady nature of the pressure pulsations within the volute throat and discharge ducts. Superimposed on the average pressure distribution are the pressure fluctuations due to the blade phase position with respect to the volute lip (Fig 7)

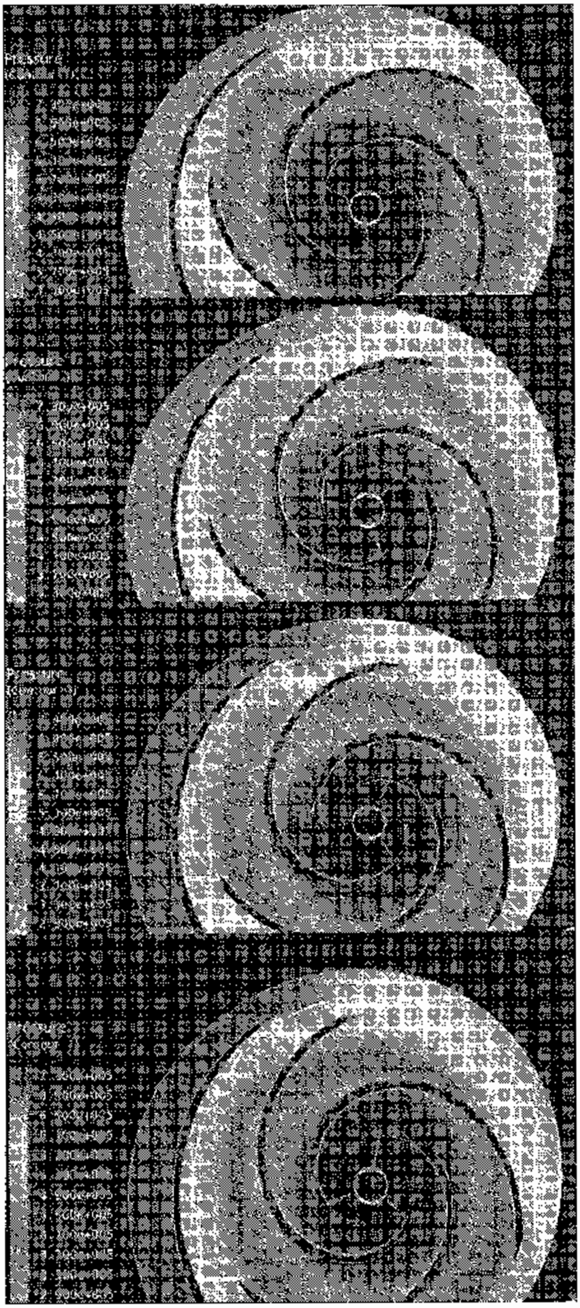

Fig 7 Unsteady Non Linear Pressure increase around the volute

This is manifest as an increasing pressure within the discharge passages, which reaches a maximum as the impeller blade passes the volute lip. The pressure pulsations are evident within these (Fig7) and are dominated by the blade passing frequency. The volute lips influence the impeller flow regime, limiting the development of the discharge impeller vortex.

The discharge passages of the pump are filled with low velocity fluid. The pressure fluctuations within this passage are dominated by the blade frequency. As the impeller blade tip approaches the volute lip, the pressure within the discharge duct reaches a maximum (Fig 7). As the blade tip passes the lip the pressure within the discharge duct decreases, but a pressure rise within the impeller is manifest.

The discharge duct described in the Newton single volute example, contained a rotating eddy in the discharge passage driven by the viscous forces imposed by the impeller discharge flow. This situation is not representative of the flow regime within a double volute pump is different.

The double volute design of volute spiral casing is popular with centrifugal pump designers as it is used balances the radial loads imposed on the casing. The design consists of two discharge ducts, used to transport 50% of the flow. These two ducts then join together at the pump discharge nozzle. These two ducts experience pressure pulsations at independent periods based on the impeller blade position. Although the fluid within the duct is moving at low speed, the situation of a driven vortex does not occur. The vortex described by Newton is broken by the addition of a second volute lip. The flow within the volute pulses between these divided discharge ducts as each duct experiences the pressure pulsation imposed upon it by the approaching vane (Figs 8 and 9).

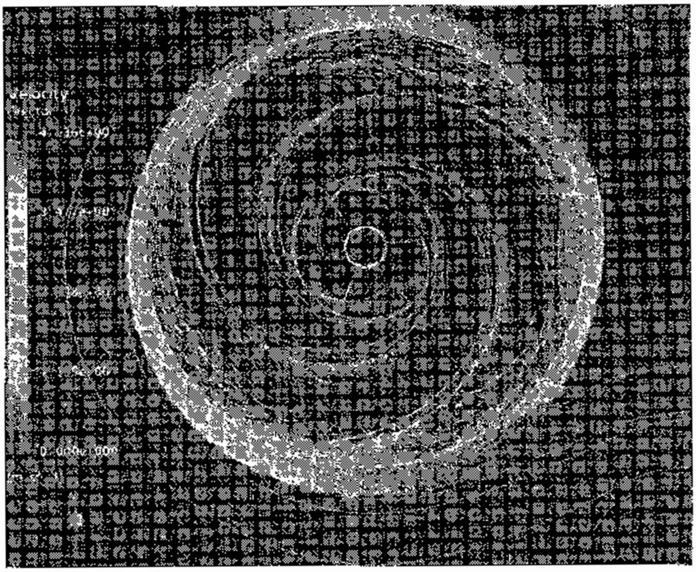

Fig 8 Pump Velocity Vectors

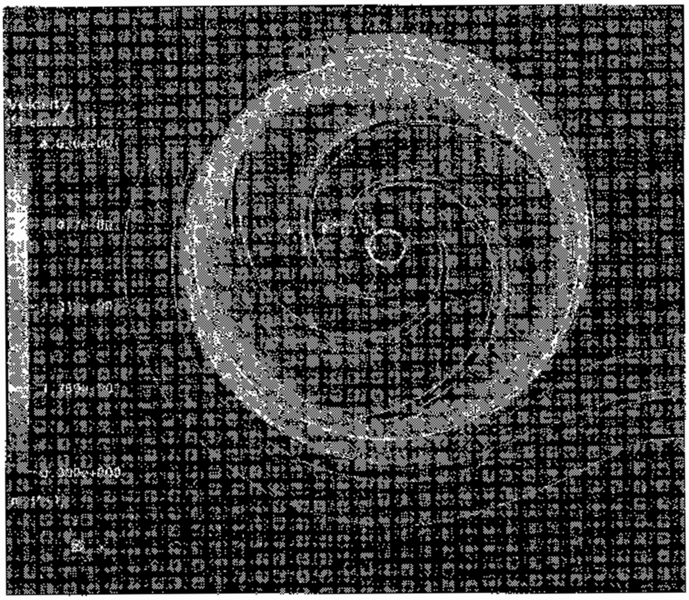

Fig 9 Pump Velocity Streamlines

5 Inlet Pipe flow

Levin and Poliokovsky carried out work to ascertain the suction re-circulation at shut-off conditions. Investigations took place on a radial un-shrouded machine. The investigation uncovered two distinct zones of fluid, which interact together within the suction channel. The mental model they proposed is somewhat simplified as it does not cover the three-dimensional interactions. Investigations using CFD yield the following observations for the two zones.

High-energy liquid is expelled from the impeller eye. This expelled liquid dominates the suction passage, occupying the 2/3 of the flow area from the pipe outer diameter downwards towards the channel centreline (Fig 10). The extent that the expelled liquid fills the suction channel remains unchanged with distance from the impeller. This flow spirals helically down the periphery of the suction pipe in a direction counter to the impeller rotation at a constant helical angle.

Each impeller blade generates individual streams. The tangential velocity imposed on the regime by the impeller dominates the flow causing the helical spiral angle to be approximately equal to the blade inlet angle. This angle does not diminish with distance from the impeller implying that both the tangential and axial components of the velocity decay proportionally to maintain this angle.

The inner 1/3 of the suction channel area contains a spiralling core of slower moving fluid. Viscous effects transmit tangential energy from the high-energy peripheral flow and drive this core in a helical spiral counter to the direction of pump rotation. Flow is dominated by the axial component of the velocity and tangential forces exerted by the peripheral flow cause the inner helical flow angle to be approximately double the outer angle. Again the helical spiral angle remains constant.

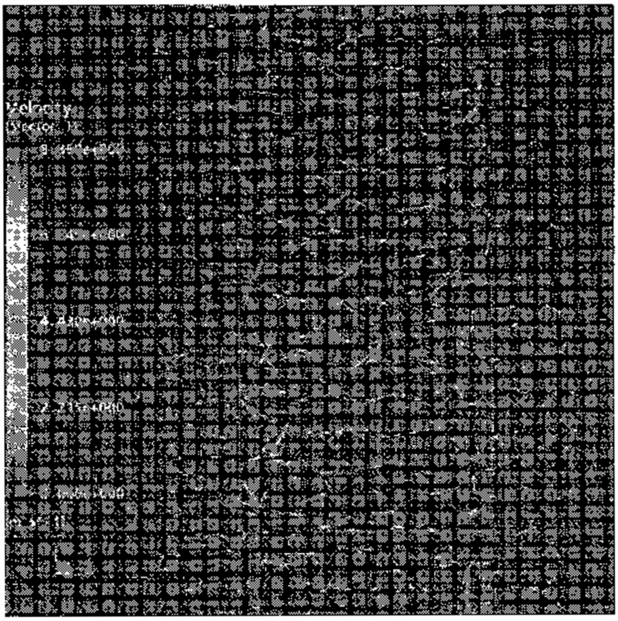

Fig 10 Internal Core of Spiralling Fluid in Suction Channel

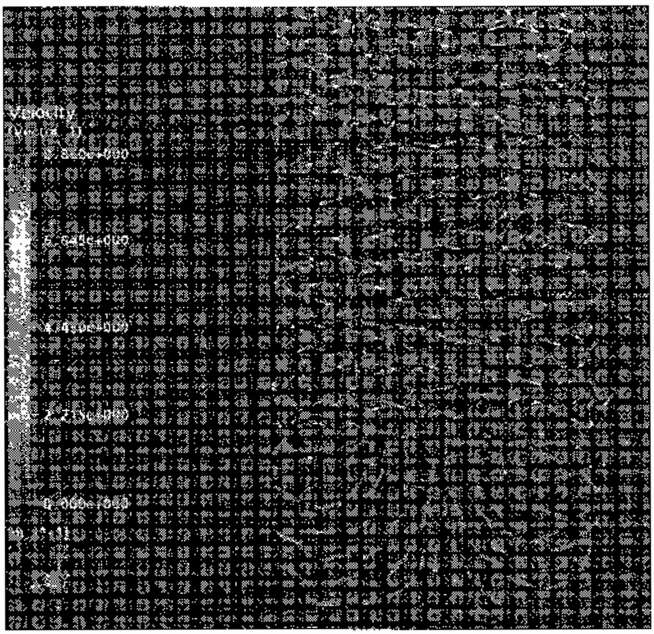

Fig 11 Helical Spiralling Inlet Backflow at Suction Duct Periphery

The interface between the re-circulating flow is predicted, by the CFD model, to be 14 multiples of the eye diameter. The position of this interface occurs when the viscous and friction effects have slowed the net velocity within the channel to zero. This "plugs" the suction pipe, encouraging the outer flow to reverse and form the inner core (Fig11). The extent that the re-circulating flow pushes into the suction duct is linked to the suction peripheral speed of the impeller. Increasing rotational speed of the impeller or the eye diameter pushes the extent of the re-circulation further down the suction passage.

6 CONCLUSION AND SUMMARY

The nature of the flow fields when the volume flux of a centrifugal pump approaches zero is undoubtedly unsteady. The Volute is filled with a slow moving low energy liquid which experiences pressure pulsations dominated by the vane pass frequency. Double volute pumps have discharge ducts that experience these pulsations independently of each other as the vanes approach the volute lips.

The outer proportion of the impeller is filled with a discharge vortex imposed on the impeller by the stationary vanes. The numbers of vanes within the stator influence the development and direction of this discharge vortex. The first 1/3 of the impeller is filled with a vortex generated by the boundary layer breakdown along the pressure side of the impeller vane close to the hub.

The liquid passes span-wise across the meridional plane and is expelled from the impeller eye at a constant helical angle approximately equal to the impeller inlet tip angle. This spiralling liquid has an influence on the inner core of the suction flow causing a rotation in this core, which reduces the incidence to the hub side of the vane. These effects all work together. The exchanges in energy from suction to impeller and on to the volute all influence the closed valve head value. The success of the model in predicting the closed valve head lies with its ability to capture all these flow effects.

The analogy of solid body rotation when applied to the closed valve head situation whilst appearing statistically significant does not truly represent the sophisticated energy exchanges required to fully describe the nature of the flow field. All of the prediction methods available rely on some form of empirical and statistical analysis Dyson 2002. Whilst these methods are of value the advances in computational modelling and processor speed will ultimately see them replaced by CFD analysis as a routine function of the design process.

References:

1.Simpson H.C. and Cinnamond (1964) "Studies of flow through centrifugal pump impellers." IMechE Proc. 1963-64 Vol. 178, Pt. 3 1(ii) Paper 8

2. Pfleiderer C. 1961 "Die Kreiselpumpen fur Flussigkeiten und Gase." Fifth Edition Springer-Verlag, Berlin.

3. Frost TH and Nilsen E 1991. Shut-off head of centrifugal pumps and fans. Proc. Inst. Mech. Engs Vol205 pp217-223

4. Stirling T.E. (1982) "Analysis of the design of two pumps using NEL methods." C183/82 Proc. ImechE Conf. "Centrifugal Pumps-Hydraulic Design."

5. Stepanoff A.J. (1957) "Centrifugal and axial flow pumps." John Wiley, New York Chapman and Hall, London

6. Dyson. G. (2002) A Review of Closed Valve Head Prediction Methods for Centrifugal Pumps.
Proc. Inst. Mech. Engs Vol216 p329 (9 pages)

7. Levin A.A. and Poliokovsky (1965) "To calculate the pressure characteristic of centrifugal pumps and fans at zero discharge." Isvestya AN SSSR Energetika 1, Transport 2 pp 129-133.

8. Palgrave. R(1985) –"Operating centrifugal pumps at partial capacity", 9th BPMA Technical Conference, Paper 6, Warwick University, Coventry (April 1985)

9. Yedidiah (1993). "The drooping curve of a centrifugal pump." Proc. ROCON 93 Rotating Machinery Conf. And Exposition, Nov. 12-13 New England, USA

10. Newton T M (1988). "Rotor –Stator interaction in radial flow pumps and fans at shut-off conditions". Pdh Thesis. Newcastle University.

11. Kaupert (1999) "The Unsteady Pressure Field in a High Specific Speed Centrifugal Pump Impeller Part1 – Transient Hysterisis in the characteristic". Vol 121 pp627-632

12. Acosta A J and Bowerman RD(1957)."An experimental study of centrifugal pump impellers"
Transactions of ASME Pp1821-1839

13. Sun.J. Tsukamota. H (2001) Off-design Performance Prediction for Diffuser Pumps Proc. Inst. Mech. Engs Part A. Journal of Power and Energy Vol215 IssA2 p191

Acknowledgement:

I would like to thank Ron Palgrave, David Brown Union Pumps and the Textron HPP for their ongoing support and encouragement with this research.

