Citation for published version:

Chew, J 2004, 'CFD Studies of Dynamic Gauging', *Chemical Engineering Science*, vol. 59, no. 16, pp. 3381-3398. <https://doi.org/10.1016/j.ces.2004.03.042>

DOI:

[10.1016/j.ces.2004.03.042](https://doi.org/10.1016/j.ces.2004.03.042)

Publication date:

2004

Document Version

Peer reviewed version

[Link to publication](#)

University of Bath

Alternative formats

If you require this document in an alternative format, please contact:
openaccess@bath.ac.uk

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy

If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

CFD Studies of Dynamic Gauging

J.Y.M Chew, S.S.S. Cardoso, W.R. Paterson and D.I. Wilson

*Department of Chemical Engineering
University of Cambridge
New Museums Site
Pembroke Street
Cambridge
CB2 3RA
UK*

Revised Manuscript

submitted to

Chemical Engineering Science

January 2004

© JYMC, SSSC, WRP and DIW

CFD studies of dynamic gauging

J.Y.M Chew, S.S.S. Cardoso, W.R. Paterson and D.I. Wilson

Department of Chemical Engineering, University of Cambridge, New Museums Site, Pembroke Street, Cambridge, CB2 3RA, UK

Abstract

Dynamic gauging is a non-contact technique for measuring the thickness of soft deposit layers on solid surfaces immersed in liquid environments, *in situ* and in real time. The technique works by inducing a flow into a nozzle located close to, and normal to, the deposit surface; the relationship between pressure drop and mass flow rate yields a measure of the distance between the nozzle and the deposit, whence the thickness of the deposit can be deduced.

Computational Fluid Dynamics (CFD) studies were performed to illuminate the fluid dynamics of this technique, with particular focus on the flow patterns and on the stresses imposed on the surface. The governing Navier-Stokes equations were solved using the Augmented Lagrangian Method implemented by the commercial partial differential equation solver, *FastfloTM*. The code was first tested successfully against previous studies in the literature featuring confined, slow Homann flows, where fluid flowed out of a nozzle. Then, simulations of gauging flows, where fluid enters a nozzle from a confined entry region, were compared with experimental data; good agreement was observed. Laminar Newtonian flows have been investigated, with Reynolds number at the nozzle throat in the range $0 < Re_t < 2200$. The shear and normal stresses on the gauged surface were predicted using the output from the CFD simulations. An initial comparison of experimental results for power-law fluids (aqueous CMC solutions) demonstrated the versatility of the technique and implied its

applicability to more complex fluids, which would be useful for industrial application. The success of this study will enable (i) use of the gauge to measure the strength of deposits, (ii) optimization of the shape of the nozzle for different tasks and (iii) extension of the technique to power-law fluids.

Keywords: Dynamic gauging; CFD; Fouling; Cleaning; Nozzle; Soft deposits.

1. Introduction

The attachment and growth of fouling layers on fluid-handling and heat transfer equipment can cause significant deterioration in hydraulic or thermal performance. Deposit growth frequently results in processes being shut down for cleaning, incurring both production losses and cleaning costs. Fouling not only increases pressure drops and reduces heat transfer rates but can also threaten process or product quality, *e.g.* by violating hygiene requirements in food manufacture. Selection of effective fouling mitigation methods, and enhancement of cleaning methods, requires knowledge of the underlying deposition and cleaning mechanisms.

Of central importance in fouling and cleaning studies is establishing the thickness of fouling layers, preferably *in situ* and in real time. Existing techniques include indirect estimation, based on changes in pressure drop or heat transfer coefficient, use of direct contact by a stylus, and measurement of ultrasound transmittance. Tuladhar *et al.* (2000) reviewed such techniques and identified their main limitations as: distortion of deposit due to contact, the need for prior knowledge of deposit properties, the limitations of measurements providing only averaged values, and the need for expensive and sophisticated equipment. A further complication is that many fouling layers generated in liquid environments are 'soft' and 'gel-like', being heterogeneous liquid-saturated structures that collapse on drainage or yield readily to a stiff contact probe.

Dynamic gauging is a novel technique which has been shown to be a successful non-contact technique for measuring the thickness of soft deposits *in situ* and in real time (Tuladhar *et al.*, 2000). Those authors used the technique to study the swelling and removal of layers of denatured whey protein from steel surfaces exposed to solutions of aqueous sodium hydroxide, simulating a common method for cleaning-in-place in the dairy industry. Invention of the technique was inspired by the pre-existing non-contact method of pneumatic gauging (Macleod *et al.* (1962), Macleod & Todd (1973), Jackson (1978)). Figure 1 shows a schematic of a typical gauging nozzle. A fixed hydrostatic head induces a flow into a nozzle located normal to and close to the surface of interest. The pressure drop/flow rate characteristic is sensitive to the nozzle-surface separation, h ; steady flow measurements can be made very quickly.

Knowledge of the nozzle location in space allows one to calculate the location of the surface, and thereby any change resulting from deposition or cleaning. The apparatus is simple and cheap to construct and operate; data can be generated rapidly. The prime assumptions are that the surface being studied is effectively stiff and impermeable.

Experimental studies using pneumatic gauging (Bridge *et al.*, 2001) and dynamic gauging (Tuladhar *et al.*, 2002) indicated that the forces imposed by gauging flows on weak deposit layers could cause significant deformation of the surface. This is undesirable for gauging, but the onset of deformation – which could be recorded by gauging – is related to film strength. Knowledge of the stresses imposed by the gauging flows on the surface would therefore afford a method for measuring film strength *in situ*. The aim of this work was to further our understanding of the fluid dynamics of gauging flows, in particular the flow patterns and the stresses acting on the surface. Moreover, the predicted shear and normal stress distributions can be used as a guide for optimizing the shape of the nozzle for different applications. The main part of this study is concerned with Newtonian liquids in the laminar regime *i.e.* for Reynolds number at the nozzle throat between 0 and 2200.

The flow pattern in dynamic gauging shows similarities with those of Homann and impinging jet flows. In Homann flow, fluid approaches a plane (a stagnation boundary) symmetrically about the axis normal to the plane (Middleman, 1995). Impinging jets and confined Homann flows have been studied extensively as a result of their wide range of engineering applications. Axisymmetric impinging jets were first studied numerically by Glauert (1956), the velocity components and shear stresses at the surface being calculated using the boundary layer equations for the laminar regime. For flows confined between parallel plates, van Heiningen *et al.* (1976) and Saad *et al.* (1977) obtained numerical solutions for the flow field and heat transfer characteristics for two-dimensional and axisymmetric laminar impinging jets, respectively. For a submerged laminar jet impinging on a plane, Deshpande & Vaishnav (1982) used a finite difference technique to solve the steady state Navier-Stokes (N-S) and continuity equations. They reported the velocity fields and the shear stress and pressure distributions on the impingement plate. Their work was coordinated with a study of canine endothelium tissue by Vaishnav *et al.* (1983), where shear stress predictions were linked to evidence of deformation from microscopy

images. Numerical modelling of impinging jet flow and heat transfer was reviewed by Polat *et al.* (1989), including cases of unconfined jets and jets confined by a plane wall, for both the laminar and turbulent regimes. Miranda & Campos (1999) reported a study of a laminar impinging jet confined by a conical wall; the N-S equations were solved using finite differences and the results compared with measurements made by laser Doppler anemometry. Their measurements are simulated here as part of code verification.

Most of the literature involving flows through nozzles is concerned with fluid directed outwards from confined and unconfined nozzles, impinging on a plate. There appears to be little work reported on flows directed inwards, *i.e.* into a nozzle, such as arise in dynamic gauging. This paper presents a numerical investigation of such flows, with all flows being treated as axisymmetric, steady, laminar, incompressible and either Newtonian or quasi-Newtonian. The N-S and continuity equations are solved numerically, using a finite element method, to predict the velocity and pressure fields, the streamlines and the stresses acting on the gauged surface. Numerical results are compared with results from the literature and with experimental data from our laboratory. This study also considers briefly the applicability of this technique to a non-Newtonian (power-law) liquid. CFD has been widely used for decades to test designs before they are built; here, we introduce this approach to design of nozzles.

2. CFD studies on the gauge

2.1 Governing equations

The N-S and continuity equations, written in dimensional form, are:

$$\text{Navier – Stokes : } \rho \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla^* \mathbf{v} \right) = -\nabla^* p + \mu \nabla^{*2} \mathbf{v} + \rho \mathbf{g} \quad (1)$$

$$\text{Continuity : } \quad \text{div } \mathbf{v} = \nabla^* \cdot \mathbf{v} = 0 \quad (2)$$

where $\mathbf{v}$ is the velocity vector, p the pressure, t time and the superscript ‘*’ on ∇ and ∇^2 implies that the operators are dimensional.

In the physical situation, the pressure difference driving the flow is generated by a gravitational head. In the simulation, however, it proves convenient to neglect gravity within the flow field and instead simply to impose a chosen pressure difference to drive the flow (Tritton, 1988). Thus for steady flow, the non-dimensional N-S and continuity equations are equations (3) and (4) respectively, where $\mathbf{V}$ is the dimensionless velocity vector and P is the dimensionless pressure.

$$\text{Navier – Stokes : } \mathbf{V} \cdot \nabla \mathbf{V} = -\nabla P + \frac{1}{Re} \nabla^2 \mathbf{V} \quad (3)$$

$$\text{Continuity : } \quad \text{div } \mathbf{V} = \nabla \cdot \mathbf{V} = 0 \quad (4)$$

where

$$\mathbf{V} = \frac{1}{2v_c} \mathbf{v} \quad (5)$$

$$\nabla = l_c \nabla^* \quad (6)$$

$$P = \frac{p}{4\rho v_c^2} \quad (7)$$

$$Re = \frac{2\rho v_c l_c}{\mu} \quad (8)$$

and the subscript c implies a characteristic value. The values of the characteristic length, l_c , and characteristic velocity, v_c , are defined later.

In two-dimensional or axisymmetric problems, it is usually useful to determine the streamlines of the flow. For the gauging problem, we employ the cylindrical coordinate system (r, θ, z) , the flow being assumed axisymmetric, with no swirl component of velocity; all the terms involving θ vanish. The stream functions and vorticity are then represented by equations (9) to (12) (Middleman, 1995).

$$V_z = -\frac{1}{R} \frac{\partial \psi}{\partial R} \quad (9)$$

$$V_R = \frac{1}{R} \frac{\partial \psi}{\partial Z} \quad (10)$$

$$\omega^* = \frac{\partial V_R}{\partial Z} - \frac{\partial V_z}{\partial R} \quad (11)$$

$$\omega^* R = \frac{\partial^2 \psi}{\partial Z^2} - \frac{1}{R} \frac{\partial \psi}{\partial R} + \frac{\partial^2 \psi}{\partial R^2} \quad (12)$$

where Z and R are, respectively, the dimensionless z and r coordinate, ω^* the dimensionless vorticity, V_z and V_R are the dimensionless velocity components in the z and r directions, and ψ the stream function. The vorticity is made dimensionless by normalizing against v_c/l_c . The values of the characteristic length, l_c , and characteristic velocity, v_c , are defined later.

The dimensionless wall shear stress, τ^* , is calculated from equation (13), which replicates the notation of Deshpande & Vaishnav (1982)

$$\tau^* \equiv \frac{\tau_{wall}}{4\rho v_c^2} = \frac{\omega^*_{wall}}{Re} \quad (13)$$

where τ_{wall} is the shear stress at the surface being studied and ω^*_{wall} is the dimensionless vorticity at the surface.

The CFD code used to solve the above N-S and continuity equations was written using *Fastflo*TM, a commercial partial differential equation solver. Spatial discretization was performed using the Finite Element Method (FEM) and solutions were generated using the Augmented Lagrangian Method (ALM). In this work, the grids used in the simulation were generated using the internal mesh generator in *Fastflo*TM. The domain was represented using unstructured triangular elements and

the governing equations were solved using a quadratic approximation. Further information concerning ALM and FEM is given in *FastfloTM* (2002), Zienkiewicz & Taylor (1991), Connor & Brebbia (1976), Gallagher *et. al* (1975) and the references therein.

2.2 Validation of the CFD code

The code was first used to solve two test cases involving flows directed outwards through different nozzles, these cases having previously been solved by finite difference methods.

The first case is the impingement of a laminar jet confined by a conical wall, described by Miranda & Campos (1999). Figure 2 shows the essentials of the conical cell. A fully developed, laminar, pipe flow impinges on a flat, round plate positioned normal to the axis of the nozzle, and is confined by a conical upper wall. The flow leaves the conical cell radially, through the gap at the periphery. The flow was treated as axisymmetric and steady. Figure 3 shows a comparison of the predicted streamlines (for $Re = 685$ defined at the point **Jet** shown in Figure 2) obtained using the *FastfloTM* code, with those predicted by Miranda & Campos, who discretized the flow equations using a finite difference technique and represented the non-rectangular domain (area under the conical wall) with non-uniform grids. The predicted streamlines show very good agreement and indicated that the code was suitable for simulating confined nozzle flows. Note that ‘ R ’ (dimensionless r -coordinate) in our simulation is equivalent to ‘ r ’ in their study. As further comparison, the dimensionless radial velocities at one location are plotted alongside Miranda and Campos' experimental data and numerical predictions in Figure 4. The figure indicates that the predictions of the current CFD code are in excellent agreement with the experimental data and, in some cases, provide better predictions than the finite difference method used by the original workers. Convergence criteria for this case were set by requiring $|\delta V_i|$ and $|\delta P|$ (dimensionless) from successive iterations to be less than 5×10^{-4} and 5×10^{-3} , respectively. The CPU-time on a 1.2 GHz PC and the number of iterations required for convergence varied from 0.5 to 1 hour and 50 to 200, respectively, depending on the value of Re and the size of the grid.

The second test case considered was the impingement of a vertical submerged laminar jet on a horizontal plane, described by Deshpande & Vaishnav (1982). The predicted flow patterns (Figure 5 – Reynolds number defined at the jet exit *i.e.* $Z = 4$) and the maximum shear stresses on the impingement plane (Figure 6) agree well with the previously reported numerical predictions. Both case studies indicate that the ALM/FEM approach is a reliable method for simulating laminar flows through nozzles.

2.3 Computational models and boundary conditions

Three computational models were considered to describe the fluid dynamics of the gauging flow. The purpose of the different models was to test the sensitivity of the solutions to the approximations made in describing the flow distant from the nozzle. Figures 7(a)-(c) show the configuration of the models, which differ in the specification of the input boundary. In Model 1, a purely axial feed is assumed, *i.e.* it flows normally across the inlet surface, positioned parallel to and remote from the plane wall. The pseudo-surface is a cylindrical surface concentric with, and far from, the axis of symmetry: flows beyond this surface are assumed to be negligible. In Model 2, it is instead the inlet surface that is concentric with, and far from, the axis of symmetry; the inlet flow, being purely radial, crosses this surface normally. The top-surface is now the surface above which all flows are treated as negligible. Model 3 assumes that the feed crosses both of the feed surfaces defined above *i.e.* it consists of a combination of radial and axial flows.

Figure 8 shows Model 1, with coordinates non-dimensionalised against the tube radius, *i.e.* $R_{tube} = 1$.

The dimensionless dependent variables follow immediately from substitution into equations (5)-(8) and (13), where l_c is the radius of the tube section 1, v_c is the mixing-cup mean fluid velocity in that section and Re_{tube} represents the Reynolds number of the flow in that section. The Reynolds number at the nozzle throat (Re_t) and Re_{tube} differ by a constant factor, which depends only on the ratio of the tube and nozzle diameters.

Boundary conditions (BCs) for this problem are specified by assigning boundary tags (**I** to **VI**) to the different boundaries in the computation domain, as shown in Figure 7(a). The physical justifications of the boundaries are as follows:

Boundary I - the axis of symmetry:

I.i No radial flow across the cell axis *i.e.* $V_R = 0$

Boundary II - the plane wall/gauged surface:

II.i No-slip condition *i.e.* $V_R = 0$

II.ii Impermeability condition *i.e.* $V_Z = 0$

Boundary III - the pseudo-surface:

Boundary **III** is the cylindrical pseudo-surface concentric with, and remote from, the axis of symmetry. Beyond boundary **III**, the flows are assumed not to contribute to the overall flow through the gauge. The distance of the boundary from the axis of symmetry (R_{plane}) is taken to be four times the radius of the tube. It is shown later that this distance is sufficiently large. At the boundary, the flow is purely axial.

III.i Axial flow only, *i.e.* $V_R = 0$

Boundary IV - the inlet:

The inlet flow enters far from the plane. This boundary's distance from the plane wall (L_2) is taken to be five times the radius of the tube. It will be shown later that this is a sufficiently large distance to ensure that the streamlines at boundary **IV** are parallel and normal to the inlet surface *i.e.* $\partial\psi/\partial Z = 0$.

Boundary V - the tube and nozzle outer wall:

Here we impose non-slip and impermeability along the solid boundaries. Both are imposed throughout the length of boundary **V** by requiring

V.i $V_R = 0$

V.ii $V_Z = 0$

Boundary VI - the outlet:

Boundary **VI** is an approximation of the discharge outlet of the physical apparatus. The outlet position in the simulation, however, is not the entire

length of the siphon tube, but instead just the length (L_I) required for the flow to become fully developed, so that essentially

$$V_z = 1 - R^2 \quad (14)$$

L_I varies with the values of Re_t and h/d_t , as summarized in Table 1.

The values of stream function at the axis of symmetry, the plane wall and the pseudo-surface are set to zero. The value of stream function at the tube inner wall and the nozzle inner and outer walls is determined by integrating equation (9) from points C to D (Figure 8), yielding $\psi = -0.25$.

2.4 Modelling procedure and convergence tests

Simulations were performed by specifying Re_{tube} (thus determining Re_t) and h/d_t : the streamlines, shear and normal stresses were then calculated from the predicted velocity and pressure fields.

The parameter range investigated includes $0 < Re_t < 2200$ and $0 < h/d_t < 0.65$; experimental studies indicated that the flow was relatively insensitive to h/d_t at larger values of h/d_t . Physical properties used are those of water at 20°C. The solutions presented were obtained using Model 1 unless otherwise stated. The standard nozzle geometry was:

- (i) $d = 4.0$ mm
- (ii) $d_t = 1.0$ mm
- (iii) $w = 0.5$ mm
- (iv) $\lambda = 0.1$ mm
- (v) $\alpha = 45^\circ$

This geometry yields the relationship $Re_t = 4 \times Re_{tube}$ for the simulations described here.

FastfloTM allows single and double precision modes for calculations. Initial studies indicated that the solutions obtained using these two modes were indistinguishable, so simulations were performed using the single precision mode to reduce computation

time and memory requirements. Convergence was tested by comparing the values of V_R , V_Z and P (dimensionless) from successive iterations: dimensionless tolerances were set at $|\delta V_i| < 5 \times 10^{-4}$ and $|\delta P| < 5 \times 10^{-3}$, respectively. The CPU-time on a 1.2 GHz PC and the number of iterations required for convergence varied from 12 to 24 hours and 4000 to 8500, respectively, depending on Re and the size of the grid.

Several diagnostic tests were performed to ensure the accuracy of the solutions. In one, the number of iterations for one of the simulation runs was increased by 50%: the solution did not drift away from the converged result. In a second, the number of grid points used in the simulation was varied using different degrees of mesh refinement to determine when grid dependency had been eliminated. In general, more grid points were required for the higher end of the Reynolds number range, with associated increases in computing time. It is worth noting that more grid points (smaller elements) were concentrated around the nozzle to ensure accuracy there, since flow separations and larger velocity gradients are expected to occur in this region. Figure 9 shows a typical example of the grid refinement.

Simulation tests were also performed to check on the boundary conditions at boundaries **III** and **IV** (Figure 7(a)). Sensitivity tests on the choice of L_2 (Figure 8) were performed by repeating the calculations, using larger values of L_2 while keeping all other parameters constant. The flow fields predicted for $L_2 = 5$ and 10 were numerically identical. Further simulations showed, moreover, that the stream function gradients (in the z -direction) were negligibly different from zero at $Z = 2$ (BC for boundary **IV**). Therefore, $L_2 = 5$ was used for all subsequent simulations. A similar sensitivity test on the value $R_{plane} = 4$ (Figure 8) indicated that the flow fields were insensitive to R_{plane} . The flow patterns obtained with $R_{plane} = 4$ and 8 were numerically identical, so the former value was employed in these simulations.

3. Results and Discussion

3.1 Streamlines

The streamlines calculated using models 1-3 for the case $Re_t = 260$, $h/d_t = 0.125$ are shown in Figures 10(a)-(d). The figure suggests that the flow field can be divided into three distinct regions, namely (i) the undisturbed-flow region, far from the nozzle; (ii) the suction region, around the nozzle mouth; and (iii) the recirculation region, within the nozzle. On inspection, regions (ii) and (iii) prove to be essentially identical for the three models, being insensitive to differences in region (i). This result indicates the unimportance of the flow approximations made far from the nozzle. The shear and normal stress distributions on the plane wall for all three models (reported later) also showed negligible differences in the positions of interest.

Figure 11 shows the streamlines obtained using Model 1, where the flows remote from the nozzle are approximated by two hypothetical surfaces, namely the inlet surface and the pseudo-surface (boundaries **III** and **IV**). The flows are predominantly axial in the undisturbed-flow region due to the BC imposed at boundary **IV** *i.e.* $\partial\psi/\partial Z = 0$. Also, at the inlet surface *i.e.* boundary **IV** ($Z = 5$), the z -wise gradients of the stream functions, ψ , approach zero. The flow changes from an axial to a radially-convergent flow as it approaches the suction region, in which the flow is predominantly radial and is sensitive to Re_t , h and w . Downstream from the suction region, the flow diverges in the nozzle throat, generating recirculation. At larger values of Re_t , the flow separates downstream of the nozzle entry and then reattaches to the inner wall of the tube further downstream. Comparison of the two halves of the gauge represented in Figure 11(a) shows that at lower Re_t , the recirculation region is smaller and flow reattachment occurs earlier.

Figure 11(b) indicates the existence of a flow transition at low Re_t ; here, for $h/d_t = 0.20$, with Re_t varied in the range 8 – 20. The recirculation pattern has vanished at $Re_t = 8$, suggesting that the flow has entered the creeping flow regime. (This transition may explain the difference in the dependence of C_d on Re_t evident in Figure 13. We further discuss this point beneath. The range of critical Re_t numbers when flow transitions occur was found by simulation to be insensitive to h/d_t .)

3.2 Comparison with experimental data

Experimental data for dynamic gauging in a quasi-static liquid were available from Tuladhar (2001), or generated by the present authors, using water and aqueous sucrose solutions in Tuladhar's apparatus. In the experimental system, the hydrostatic head driving the flow, s , is set, the clearance, h , fixed, and the mass flow rate, m , measured to give Re_t . The experimental values of Re_t supply the abscissae of the symbols plotted in Figure 12; the lines show the experimental values of s . In the simulations, the clearance and mass flow are fixed equal to the experimental values. The values of the hydrostatic head, s_s , are calculated from the computational output, via the pressure differences, and supply the ordinates of the symbols plotted in Figure 12: the comparison of s and s_s thereby effected corresponds to the range $0.07 < h/d_t < 0.26$ likely to be used in practice. The simulations and measurements show excellent agreement and suggest that the assumptions made in the model, particularly that of laminar flow throughout the entire field, are satisfactory.

The flow through the nozzle is described by

$$m = C_d \frac{\pi}{4} d_t^2 \sqrt{2\rho \Delta P_{13}} \quad (15)$$

where

$$\Delta P_{13} = \Delta P_{14} - \Delta P_{34} = \rho g s - \frac{128 \mu m l_{eff}}{\pi d^4 \rho}, \quad (16)$$

m = mass flow rate measured experimentally,

l_{eff} = equivalent length of the siphon tube,

and the stations are those marked in Figure 1.

Calculation of ΔP_{13} requires an estimate of l_{eff} , which was estimated thus: consider the *real* length of the siphon tube, measured from station 3 of Figure 1 to the discharge end of the tube. We define its equivalent length, l_{eff} , as the length of the *hypothetical* straight tube that would support the same resistance to flow as does this *real* tube (the two lengths differ because of the pressure drops in the bends of the real tube). The value of l_{eff} was determined by separate experiments, for which the nozzle had been unscrewed from the tube, performed at high values of h (typically $h > 20$

mm). The total pressure drop due to the hydrostatic head, s , is then equal to the pressure drop over the real tube, including losses due to the bends. Hence, for fully developed laminar flow,

$$\rho g s = \frac{128 \mu m l_{eff}}{\pi d^4 \rho} \quad (17)$$

$$l_{eff} = \frac{\rho^2 g s \pi d^4}{128 \mu m} \quad (18)$$

In practice, there are irreversible energy losses due to the complexity of the flow near the constrictions (nozzle tip and clearance region); it is these that are accounted for by the discharge coefficient, C_d , as shown in equation (15). Experimental work by Tuladhar *et al.* (2000), using water as the gauging fluid, established that C_d for the gauging nozzle was a weak function of Re_t (equations (19) and (20)) over a useful range.

$$1000 C_d = \left(0.3571 \exp \left[-5.0613 \sqrt{\frac{h}{d_t}} \right] \right) Re_t + \left(-70.3 + 3721.2 \frac{h}{d_t} - 2238.3 \left(\frac{h}{d_t} \right)^2 \right) \quad (19)$$

$$1000 < Re_t < 4000 \quad (20)$$

Figure 13(a) shows a plot of the variation of C_d with Re_t for a wider range of Re_t . The solid and dotted lines show respectively the current CFD predictions and Tuladhar's empirical model (equation (19) – $d_t = 1$ mm, $d = 4$ mm, $w = 0.5$ mm, $\lambda = 0.1$ mm and $\alpha = 45^\circ$), extrapolated whenever necessary, while the dark and light symbols are respectively the current and Tuladhar's experimental measurements.

To confirm our CFD predictions at low Re_t ($Re_t < 500$), experiments were performed using aqueous sucrose solutions of 15%, 25% and 35% w/w. Sucrose solution was used instead of water to achieve the desired lower values of Re_t . The fluid properties of the sucrose solutions, namely the density and viscosity, are well documented. For cross-checking, an Ostwald viscometer (1619N02/ Type B) was used to measure the viscosity of the sucrose solutions; it was found that the measured values for all three concentrations agreed with the literature (Mathlouthi & Genotelle, 1995) to within 4%, as in Table 2.

Figure 13(a) shows reasonable agreement between the CFD predictions and the experimental C_d values. At lower Re_t , the extrapolations of equation (19) show a significant mismatch with the simulations and measurements, revealing the inadvisability of extrapolating Tuladhar's high- Re_t model to low Re_t . It is also noteworthy that the value of Re_t at which the simulation and the extrapolation of Tuladhar's empirical model start to differ appreciably increases with increasing h/d_t . For instance, at $h/d_t = 0.10$ and 0.20 , the critical values are approximately $Re_t = 320$ and 600 respectively (Figure 13(b)). In that figure, C_d^* is the asymptotic discharge coefficient *i.e.* it corresponds to $h/d_t \rightarrow \infty$.

It is interesting to note that C_d decreases with a decrease in Re_t . This is expected, as equation (15) assumes there is a balance between pressure forces and the inertia of the fluid $\Delta P_{13} \sim \rho U^2$. Here U is a scale for the velocity between stations 1 and 3 (see Figure 1). Such a balance is realistic for high Re_t flows, and hence C_d is close to 1 in such cases. At smaller Re_t , however, viscous effects balance the pressure forces and, as a result, we expect $C_d^2 \sim \{\mu(U/d_t)\} Re_t / \Delta P_{13}$, *i.e.* $C_d^2 \sim Re_t$. We also note that, for a given Reynolds number, C_d decreases as h/d_t decreases. This implies that the total pressure loss in the complex flow near the constriction is significant. Therefore it will be fruitful to divide the total pressure drop across the gauge (*i.e.* $\Delta P_{Total} = \rho g H$) into components. Figure 1 shows the different sections analyzed (ΔP_{PQ} , ΔP_{QR} , ΔP_{RS} , ΔP_{ST}) and Figures 14(a)-(c) show the breakdowns of the total pressure drop for $h/d_t = 0.10$, 0.20 and 0.65 at $Re_t = 4$, 20 and 400 respectively. Note that the pressure drop values are the average values over the surface concerned. Figure 14(a) shows that the pressure loss under the rim of the nozzle is a significant portion of ΔP_{Total} ($\Delta P_{QR} > 70\%$) for $4 \leq Re_t \leq 400$ when $h/d_t = 0.10$. This is because the flow into the nozzle is substantially affected by the friction with the gauged surface and the rim. Also from Figure 14(a) we can infer that the pressure drop contributions from the tube section ($\Delta P_{ST} < 15\%$) are small. Therefore, we can conclude that the low C_d values in Figure 13(a) are mainly due to the large pressure drop under the rim of the nozzle. When h/d_t is increased to 0.20 , the decreased contributions of ΔP_{QR} to ΔP_{Total} for all the values of Re_t are compensated by ΔP_{RS} , as seen in Figure 14(b). This is because the effect of the gauged surface started to moderate: thus $h/d_t = 0.20$ is the recommended

upper working limit for the gauge, advice consistent with Tuladhar's (2001). When the nozzle is far from the plane surface, at $h/d_t > 0.25$, the presence of the surface has little effect on C_d . The dominating pressure drops are ΔP_{RS} and ΔP_{ST} , as shown in Figure 14(c). The discharge coefficient then tends towards an asymptotic value of ~ 0.9 (Figure 13(b)). The experimental data and simulation results in Figure 13(b) show excellent agreement; both data sets indicate that C_d is insensitive to Re_t in the range considered. It can also be seen from Figures 14(a)-(c) that ΔP_{RS} increases with Re_t . This is because the scales of the recirculations in the divergent section of the nozzle for higher Re_t are larger, corresponding to greater pressure losses. Finally, we should note that at low Re_t (< 40), there is a significant discrepancy between the numerical predictions and the experimental data for C_d . We believe this is a result of the way in which ΔP_{I3} was calculated using equation (16), where an average l_{eff} value was used at both high and low Re_t . The average value of l_{eff} may not adequately represent the low Re_t case because the effect of the bends in the tube is dependent on Re_t .

3.3 Extension to power-law fluids

Several of the fluid environments in potential applications of the dynamic gauging technique involve non-Newtonian rheologies. The power-law model (defined by equation (21)) is one of the simplest constitutive equations used to describe the behaviour of non-Newtonian fluids (Pnueli & Gutfinger, 1997).

$$shear\ stress = k \times (shear\ rate)^n \quad (21)$$

where n and k are two rheological parameters.

Colombo and Steynor (2002) performed a feasibility study of dynamic gauging using aqueous carboxy-methyl-cellulose (CMC) solutions, which exhibited power-law behaviour. The CMC powders available were in two forms *i.e.* 'high viscosity' and 'low viscosity' types (BDH Laboratory Supplies). They observed m/h profiles similar to those observed for Newtonian liquids, and found that they could be mapped to Newtonian behaviour by characterizing the flow in terms of discharge coefficient, C_d , and the Metzner-Reed Reynolds number (Chhabra & Richardson, 1999), defined as

$$Re_{t,MR} = \frac{\rho v_t^{2-n'} d_t^{n'}}{8^{n'-1} k'} = \frac{\rho v_t d_t}{\mu_{eff}} \quad (22)$$

where the rheological parameters,

$$n' = n \quad (23)$$

$$k' = k \left(\frac{3n+1}{4n} \right)^n \quad (24)$$

and the effective viscosity,

$$\mu_{eff} = k' \left(\frac{8v_t}{d_t} \right)^{n'-1}. \quad (25)$$

Table 3 summarizes the rheological parameters measured for the CMC solutions using parallel plate and concentric cylinder rheometers. The high viscosity CMC had longer mean chain length than the low viscosity material. In general, at low CMC concentration, the solution is less viscous (higher $Re_{t,MR}$ for given H) and n approaches 1 (Newtonian behaviour). Colombo and Steynor used a large nozzle (with $d_t = 2$ mm, $d = 4$ mm, $w = 0.2$ mm, $\lambda = 0.1$ mm and $\alpha = 30^\circ$) because the CMC solutions were significantly more viscous than water. The modified Reynolds numbers obtained were in the lower range, *i.e.* $Re_{t,MR} < 100$.

The corresponding simulations were performed for gauging with a quasi-Newtonian fluid in Colombo and Steynor's nozzle configuration over the range of $Re_{t,MR}$ observed in the experiments. The aim of this work was to verify the C_d versus $Re_{t,MR}$ mapping observed by Colombo and Steynor at these low $Re_{t,MR}$ values, and thereby test the applicability of dynamic gauging to power-law fluids. Calculation of the stress distributions on the gauged surface, requiring implementation of fully non-Newtonian viscous terms, was not performed.

Figure 15 shows the experimental C_d values plotted against $Re_{t,MR}$. The error bars indicate the considerable uncertainty in the determination of the rheological properties of the CMC solutions. The uncertainties in the experiments are also evident from the C_d calculations because Colombo and Steynor's experimentally-derived equivalent length of tube (determined by substituting equation (25) into equation (18)) was found to be less than the actual length of the tube (average $l_{eff} \approx 0.665$ m *cf.* actual length =

0.695 m), which is aberrant. So the experimental C_d values in Figure 15 were recalculated using the actual length of the tube instead of the average l_{eff} . Although the CFD predictions (using the quasi-Newtonian fluid formulation) do not agree perfectly with the experimental values, the trends are notably similar. This encourages further investigation, using different power-law fluids and/or improved rheological characterization. It is interesting to note the significant mismatch between the extrapolation of Tuladhar's empirical model for this nozzle and the simulations and experimental data, as was also observed for the Newtonian fluid case. Equation (26) is Tuladhar's empirical model portraying the C_d versus Re_t relationship for a nozzle with $d_t = 2$ mm, $d = 4$ mm, $w = 0.2$ mm, $\lambda = 0.1$ mm and $\alpha = 30^\circ$.

$$1000C_d = \left(0.18 - 2.49 \frac{h}{d_t} + 12.10 \left(\frac{h}{d_t} \right)^2 - 20.43 \left(\frac{h}{d_t} \right)^3 \right) Re_t + \left(-180.5 + 6424.9 \frac{h}{d_t} - 10076 \left(\frac{h}{d_t} \right)^2 \right) \quad (26)$$

It can also be inferred from Figure 15 that the asymptotic discharge coefficients (at large $h/d_t \geq 0.34$) are strongly dependent on $Re_{t,MR}$. This is partly due to the high viscosity of the liquids, as the values of s used ($s = 300$ mm) are not large enough to generate large flow rates. The trend is, however, similar to the trends for Newtonian fluids, as shown in Figure 13(a) for $Re_t < 100$, albeit for a different nozzle configuration. The discharge coefficient is strongly dependent on Re_t .

3.4 Shear stress distributions on the plane surface

The stresses acting on the gauged surface/plane wall can be readily calculated from the numerical solutions. Figures 16(a) and (b) shows the wall stress distributions calculated for the case $Re_t = 260$ and $h/d_t = 0.125$ using Model 1. Also plotted are the (dimensionless) residuals for Models 2 and 3, being defined as

$$Residuals \text{ for Model 2} = |Prediction \text{ of Model 1} - Prediction \text{ of Model 2}| \quad (27)$$

$$Residuals \text{ for Model 3} = |Prediction \text{ of Model 1} - Prediction \text{ of Model 3}| \quad (28)$$

The magnitudes of the residuals in the region of interest (*i.e.* $0.0 < R < 0.65$) are small ($< 5\%$) compared to the actual stresses. For example, the largest residual, of *ca.* 0.36 (dimensionless) at $R \sim 0.2$ for Model 2 compares favourably with the stress value of 7.2. This shows that the predictions from the three models were again practically indistinguishable, indicating that the undisturbed-flow region is unimportant to the working of the gauge and that the choice of the model is trivial. The normal stress decreases towards the axis ($R = 0$), peaking at the radial position of the inner rim of the nozzle. The shear stress imposed on the surface is zero at the centreline, *i.e.* $R = 0$, and approaches zero asymptotically for large R . A maximum is observed at $R \sim 0.25$ (inner radius of nozzle) and there is a shoulder located outside the nozzle outer radius ($R \sim 0.62$). Figure 17(a) shows the shear stress distributions at $Re_t = 4$ and 904, both at $h/d_t = 0.10$. There is a pronounced shoulder near the radial position of the outer rim of the nozzle for the $Re_t = 904$ case. This could be due to inertial effects at higher Re_t because when Re_t is decreased to 4, the shoulder vanishes. For all cases considered, the peak shear stress values occur at $R \sim 0.25$, *i.e.* close to the inner radius of the rim of the nozzle. This is consistent with Tuladhar's (2001) experimental observation; he recorded that deposit distortions were sometimes seen, most often located under the rim of the nozzle. The peak value of the wall shear stress is plotted as a function of Re_t in Figure 18. The figure is potentially deceptive; at constant s , increasing Re_t would be expected to increase the peak wall shear stress. On the plot, the peak values, however, are actually decreasing as Re_t is increased. The decrease is caused by the increase in the value of h/d_t , *i.e.* as the gauge is located further from the surface. The decrease is moderate when $h/d_t < c. 0.20$ (depending on s); at higher values, the maximum shear stress decreases rapidly (most evident for $s = 340$ mm). This is consistent with experimental observations, which indicate that the presence of the gauged surface is unimportant when $h/d_t > 0.25$.

In Tuladhar's (2001) experiments on the cleaning of whey protein, the dynamic gauge was operated in a duct flow; visual inspection revealed that the whey protein deposit was sometimes distorted during gauging. The deposit distortions were probably due to the wall shear stress exerted by the gauge suction or by the bulk flow of cleaning solution, or by both. Tuladhar used mean bulk velocities of 0.03 – 0.30 m/s, corresponding to wall shear stresses due to bulk flow of *c.* 0.016 Pa and 0.44 Pa respectively. The shear stresses exerted by a dynamic gauge in quasi-stagnant liquid

under similar conditions would be two or three orders of magnitude larger (*e.g.* 20 Pa $< \tau_{wall} < 70$ Pa in the working range of the gauge, $0 < h/d_t < 0.20$), suggesting that the gauging flow was the chief cause of disruption. More detailed calculation of stresses imposed by dynamic gauging in duct flow will require solution of the 3-dimensional N-S equations.

Figure 17(b) shows the calculated normal stress distribution on the gauged surface at $Re_t = 4$ and 904. The largest values of the suction pressure again occur within the inside radius of the nozzle, *i.e.* for $R < 0.25$. The difference in pressure distributions is very evident. At low Re_t , the high pressure region is flat, whereas at high Re_t , there is a peak at the inner radius location, accompanied by a compression zone between $0.45 < R < 0.55$, generated by the inertial effect of the fluid in the convergent area in the suction region. The existence of such a peak is a characteristic of high Re_t flows. Figure 16(b) shows the shape of the pressure profile for an intermediate value of $Re_t = 260$.

3.5 Design of nozzles

CFD was used to predict the effects of different nozzle shapes on the gauging system. This is very useful in avoiding time-consuming fabrication of apparatus and possibly costly mistakes, while still permitting testing of a variety of alternative designs. Three design aspects are covered in this paper, being the nozzle angle, α , the width of the nozzle rim, w , and tube diameter, d . Each aspect was varied individually while keeping all the other design parameters constant. The following dimensions were unaltered throughout this design study (refer to labels in Figure 1)

- (i) $d_t = 1.0$ mm
- (ii) $h/d_t = 0.20$
- (iii) $\lambda = 0.1$ mm.

Figures 19(a) and (b) show the shear and normal stress distributions on the gauged surface for different nozzle angles, namely $\alpha = 30^\circ$, 45° and 60° at $Re_t = 20$ and 400 respectively. Figure 19 shows, once again, pronounced shoulders near the outer rim of the nozzle for higher Re_t , but not for lower Re_t . Also, it is clearly shown that the effect of the nozzle angle on the stresses acting on the surface is insignificant for both

high- and low- Re_t flows. This implies an advantage for fabrication of the nozzle because a slight uncertainty in the machining of the angle of the nozzle can be tolerated since the effect on the stresses acting on the gauged surface is negligible. Pressure drop analysis indicates that, at low Re_t , the fractional pressure drops are also very similar for all three angles. On the other hand, for the high Re_t case, the pressure drop in the divergent section of the nozzle increases slightly with the nozzle angle. This observation is supported by the flow patterns which show that the size of the recirculation region increases with nozzle angle thus increasing the pressure drop in the divergent section. This increase in pressure drop becomes more prominent as Re_t is further increased. Consequently, it is concluded that for high Re_t flows, the value of C_d decreases as the nozzle angle is increased whereas for low Re_t the effect of the nozzle angle is negligible.

The rim widths investigated in this study were $w = 0.25$ mm, 0.5 mm and 1.0 mm. Figures 20(a)-(d) show the shear and normal stress distributions at different rim widths at $Re_t = 20$ and 400 . These distributions are dissimilar at different rim widths for both high and low Re_t flows. Nevertheless, Figures 20(a) and (b) show that the peak shear stresses are negligibly different at the same Re_t . The shear stresses start to decrease rapidly at the outer rim of the nozzle, as expected. Therefore, among the three different widths, it is the largest, $w = 1.0$ mm, that subjects the total area of the gauged surface to the greatest shear force. In other words, the total shear forces acting on the gauged surface increase with rim width. The normal stress distributions in Figures 20(c) and (d) show that the magnitudes of the peak normal stresses differ significantly, being greatest for $w = 1.0$ mm. The total normal force acting on the gauged surface is also greatest for $w = 1.0$ mm. The flow patterns, however, for both high and low Re_t , are insensitive to rim width. It can readily be proved by pressure drop analysis that the total pressure drop due to the constriction, especially under the rim of the nozzle, increases with increasing rim width. Therefore, C_d decreases with increasing rim width, at constant h/d_t .

The effect of tube diameter on the stresses is shown in Figures 21(a) and (b). Larger diameter would be expected to promote a larger recirculation region. The pressure loss due to the nozzle expansion (divergent section) would therefore increase accordingly. This would cause the value of C_d to decrease as the tube diameter is

increased. Figures 21(a) and (b) show that the peak shear stresses decrease slightly and shift towards the centreline of the gauge when the tube diameter is increased from 4 mm to 8 mm. In contrast, the peak normal stresses increase as the tube diameter is increased.

The success of this study lets us conclude that designing for both larger rim width and larger tube diameter will be preferable for low Re_t flows, because such designs will increase the pressure drop around the nozzle at the expense of the pressure drop in the tube. In other words, the gauge will be more sensitive to the location of the nozzle relative to the gauged surface.

4. Conclusions

This work demonstrates the application of CFD as a quantitative tool to describe the fluid mechanics of dynamic gauging. The potential power of combining CFD with gauging experiments to measure the strength of deposits has been revealed. CFD has been used for optimization of nozzle shape. CFD calculations have supported the practicality of using some simple non-Newtonian fluids as gauging fluids.

To verify the CFD code, two test cases involving flow outwards from confined and unconfined nozzles were modelled. These validations showed that the code was able successfully to model flows through nozzles. For the dynamic gauge, simulations were compared with experimental results in both dimensional and dimensionless forms. Hydrostatic heads were compared for the parameter range $0 < Re_t < 2200$ and $0.07 < h/d_t < 0.65$: agreement was good, as it was for discharge coefficients, thus increasing confidence both in the simulations and the experimental results. From the pressure drop analysis, it was shown that the pressure drop under the rim of the nozzle dominates when h/d_t is low (< 0.10) whereas when h/d_t is high (> 0.20), the pressure drop in the divergent section and the tube section dominate.

Simulations suggest that the flow field of the gauge can be divided into three distinct flow regions, namely the undisturbed-flow region, the suction region, and the recirculation region. Comparisons of models showed that the choice between different approximate representations of the flows far from the nozzle is unimportant and suggests that only the suction region and the recirculation region are important to the working of the gauge. The three-region flow structure disappeared for $Re_t < 8$,

indicating that flow transitions occur between $Re_t = 8$ and 20; this range is found by simulation to be insensitive to the value of h/d_t being studied.

CFD simulations showed that the maximum wall shear stresses were located under the rim of the nozzle, close to its inner radius, *i.e.* at $R \approx 0.25$. The peak suction pressures, however, occurred within the inside radius of the rim of the nozzle, *i.e.* in $0 < R < 0.25$.

This work also shows the convenience of CFD for studying various designs of the nozzle without their being built. The design studies show the effects of different nozzle angles, rim widths and tube diameters on the shear and normal stresses acting on the gauged surface. It is concluded that larger rim width and/or tube diameter can increase the sensitivity of the gauge to the clearance between the nozzle and the gauged surface.

Based on the C_d versus $Re_{t,MR}$ profiles for viscous power-law fluids, the applicability of this gauging technique in simple non-Newtonian environments was demonstrated.

Notation

Roman

C_d	Discharge coefficient accounting for flow complexity and energy loss	-
d	Inside diameter of tube	m
d_t	Inside diameter of nozzle throat	m
g	Acceleration due to gravity	m/s ²
h	Clearance between the nozzle tip and gauging surface	m
H	Dimensionless clearance between nozzle and surface	-
k	Rheological power-law consistency	-
k'	Modified rheological power-law consistency	-
l_c	Characteristic length	m
$l_{eff.}$	Equivalent length of the siphon tube	m
L_i	Dimensionless length, $i = 1$ or 2	-
m	Mass flow rate	kg/s
n	Rheological power-law index	-
n'	Modified rheological power-law index	-
P	Dimensionless pressure	-
p	Pressure	Pa
p_c	Characteristic pressure	Pa
R	Dimensionless r -coordinate	-
Re	Reynolds number	-
Re_t	Reynolds number in nozzle throat	-
$Re_{t, MR}$	Metzner-Reed Reynolds number in the throat	-
Re_{tube}	Reynolds number in siphon tube	-
R_{nozzle}	Dimensionless inside diameter of nozzle	-
R_{plane}	Dimensionless radius of plane wall	m
r_{tube}	Inside radius of siphon tube	m
R_{tube}	Dimensionless inside diameter of tube	-
s	Hydrostatic head	m
s_s	Simulated hydrostatic head	m
v_c	Characteristic velocity	m/s
V_R	Dimensionless velocity component in r -direction	-
v_t	Velocity in the throat	m/s
V_Z	Dimensionless velocity component in z -direction	-
w	Width of nozzle rim	m
W	Dimensionless width of nozzle rim	-
Z	Dimensionless z -coordinate	-

Greek

ΔP_{xy}	Pressure drop between point x and y	Pa
α	Angle of the nozzle	°
δP	Pressure change from successive iteration	-
δV_i	Velocity change from successive iteration	-
λ	Length of nozzle exit	m
μ	Viscosity of Newtonian process fluid	Pa.s
μ_{eff}	Effective viscosity of non-Newtonian process fluid	Pa.s
ρ	Density of process fluid	kg/m ³
τ_c	Characteristic shear stress	Pa
τ^*	Dimensionless shear stress	-
τ_{wall}	Wall shear stress	-
ω^*	Dimensionless vorticity	-
ω^*_{wall}	Dimensionless vorticity at the plane wall	-
ψ	Stream function	-

Abbreviations

ALM	Augmented Lagrangian Method
BC	Boundary condition
CFD	Computational Fluid Dynamics
FEM	Finite Element Method
N-S	Navier-Stokes

Acknowledgements

Financial support from the Cambridge Commonwealth Trust for JCYM, and experimental data from A.Q. Colombo, E.J. Steynor and T.R. Tuladhar are all gratefully acknowledged.

References

- Bridge, S.P., Robbins, P.T., Paterson, W.R., & Wilson, D.I. (2001). A pneumatic gauging sensor for measuring the thickness of soft films, *TransIMechE Part E : Process Mechanical Engineering*, 215, 19-27.
- Chhabra, R.P., & Richardson, J.F. (1999). *Non-Newtonian Flow in the Process Industries*, Chapters 1-3, Butterworth-Heinemann, Oxford, U.K.
- Colombo, A.Q., & Steynor, E.J. (2002). *Developing Dynamic Gauging*. Part IIB Research Project Reports, Department of Chemical Engineering, University of Cambridge, U.K.
- Connor, J.J., & Brebbia, A. (1976). *Finite Element Techniques for Fluid Flow*, Chapters 1 and 2, Newnes-Butterworths, London, U.K.
- Deshpande, M.D., & Vaishnav, R.N. (1982). Submerged laminar jet impingement on a plane. *Journal of Fluid Mechanics*, 114, 216-236.
- Gallagher, R.H., Oden, J.T., Taylor, C., & Zienkiewicz, O.C. (1975). *Finite Elements in Fluids Volume 1*, John Wiley & Sons, London, U.K.
- Glauert, M.B. (1956). The wall jet. *Journal of Fluid Mechanics*, 1, 625-643.
- Jackson, G.F. (1978). An assessment of the application of pneumatic gauging to the measurement of mass transfer coefficients. *Journal of Physics – Engineering Scientific Instruments*, 11, 569-575.
- Macleod, N., & Todd, R.B. (1973). The experimental determination of wall-fluid mass transfer coefficient using plasticized polymer surface coatings. *International Journal of Heat and Mass Transfer*, 16, 485-504.
- Macleod, N., Cox, M.D., & Todd, R.B. (1962). A profilometric technique for determining local mass transfer rates – Application to the estimation of local heat-transfer coefficients in a nuclear reactor. *Chemical Engineering Science*, 17, 923-935.
- Mathlouthi, M., & Genotelle, J. (1995). *Rheological Properties of Sucrose Solutions and Suspensions: Sucrose Properties and Applications: edited by Mathlouthi, M. & Reiser, P.* pp. 126-154, Blackie Academic & Professional, New York, U.S.A.
- Middleman, S., (1995). *Modeling of Axisymmetric Flows: Dynamics of Films, Jets and Drops*, Chapters 1 and 5, Academic Press, New York, U.S.A.
- Miranda, M., & Campos, B.L.M. (1999). Impinging jets confined by a conical wall: laminar flow predictions. *AIChE Journal*, 45(11), 2273-2285.
- “FastfloTM Tutorial Guide Version 3.0,” Chapters 1-15, 19-21. Nag Ltd., Oxford, U.K.
- Pnueli D., & Gutfinger C., (1997). *Fluid Mechanics*, Chapter 14, Cambridge University Press, Cambridge, U.K.
- Polat, S., Huang, B., Mujumbar, A.S., & Douglas, W.J.M. (1989). Numerical flow and heat transfer under impinging jets: A Review. *Annual Review of Numerical Fluid Mechanics and Heat Transfer*, 2, 157-197.
- Saad, N.R., Douglas, W.J.M., & Mujumbar, A.S. (1977). Prediction of heat transfer under an axisymmetric laminar impinging jet. *Industrial and Engineering Chemistry Fundamentals*, 16, 148-154.
- Tritton, D.J., (1988). *Physical Fluid Dynamics*, 2nd Edition, pp. 58-59. Oxford University

- Press, Oxford, U.K.
- Tuladhar, T.R., Paterson, W.R., MacLeod, N., & Wilson, D.I., (2000). Development of a novel non-contact proximity gauge for thickness measurement of soft deposits and its application in fouling studies. *The Canadian Journal of Chemical Engineering*, 78, 925-947.
- Tuladhar, T.R., (2001). *Development of a Novel Sensor for Cleaning Studies*. Ph.D. Dissertation, University of Cambridge, U.K.
- Tuladhar, T.R., Paterson, W.R., & Wilson, D.I., (2002). Investigation of alkaline cleaning-in-place of whey protein deposits using dynamic gauging, *Trans. I. Chem. E.*, 80, Part C, 199-213.
- van Heiningen, A.R.P., Mujumbar, A.S., & Douglas, W.J.M., (1976). Numerical prediction of the flow field and impingement heat transfer caused by a laminar slot jet. *Journal of Heat Transfer*, 98, 654-658.
- Vaishnav, R.N, Patel, D.J, Atabek, H.B, Deshpande, M.D, Plowman, F., & Vossoughi, J. (1983). Determination of the local erosion stress of the canine endothelium using a jet impingement method. *Journal Biomechanical Engineering*, 105, 77-83.
- Zienkiewicz, O.C., & Taylor, R.L. (1991). *The Finite Element Method, 4th Edition, Volume 1 and 2*. McGraw-Hill, New York, U.S.A.