Hydraulic study and optimisation of water treatment processes using numerical simulation

K. Craig*, C. De Traversay, B. Bowen*, K. Essemiani**, C. Levecq** and R. Naylor***

* Vivendi Water Australia, Level 37, Gateway, 1 Macquarie Place, Sydney 2000, NSW, Australia (E-mail: *kcraig@vivendiwater.com.au*)

** Vivendi Water Anjou Recherche, Chemin de la Digue B.P 76, 78603 Maisons Laffitte, France (E-mail: *christelle.de-traversay@generale-des-eaux.net*)

Water Science and Technology: Water Supply **Vol 2 No 5–6 pp 135–142** © IWA Publishing 2002

Abstract Until recently, water treatment plants were frequently designed solely on the basis of the biological, chemical and physical constraints of processes. Nowadays, the use of Computational Fluid Dynamics (CFD) software enables the dimensioning of water treatment processes by taking into account the real hydraulic behaviour of processes. That has be done for the Coliban Water Aqua 2000 project, which consists of the construction of three water treatment plants. The disinfection performance of three ozone contactors were compared using the CFD software, Fluent. Moreover, the CFD application has been extended to a large range of water treatment processes in recent years. The paper presents several of these: flocculation tanks, UV reactors and secondary settling tanks.

Keywords Clarifiers; computational fluid dynamics (CFD); flocculation; oxidation contactors; UV reactor

Introduction

The main objectives in drinking water and wastewater processes are to meet quality requirements, and to guarantee the process efficiency at the optimal cost. Computational Fluid Dynamics (CFD) is a powerful modelling tool that enables improvement and optimisation of the hydraulic and process efficiency in water industry applications. With the drive toward system optimisation and compliance business drivers, the conventional thinking has been repeatedly challenged, and it has been demonstrated that CFD solutions do have a value when applied to perceived “low-technical” problems. In conjunction with this, advances in hardware and software capabilities have helped to make CFD more accessible as well as practical, allowing direct commercial benefits to be gained from its use. CFD is a computer-based methodology to solve the fundamental equations of fluid flow: the Navier–Stokes equations coupled with species, turbulence and multiphase transport equations and closure models using a finite volume technique.

Vivendi Water has been using the CFD software, Fluent, since 1992 to investigate the performance and possible designs of various treatment processes.

Results of this study

This paper outlines some of the CFD works carried out in Anjou Recherche Vivendi Water. They include ozonation contactors, flocculation tanks, UV reactors and secondary settling tanks.

The ozonation contactors

The Coliban Water Aqua 2000 Project is a BOOT project awarded to Vivendi Water for the design, construction and 25 year operation of three water treatment plants ranging in size from 7.7 ML/day to 126 ML/day, with reservoirs in the Bendigo region of Victoria, Australia. The contract has very stringent water quality targets that have been adopted from the United States and European Union drinking water standards. The CFD tool has been used to ascertain the disinfection performance of the ozone contactors of the three water treatment plants: Sandhurst, Kyneton and McCay. These ozone contactors have different designs in their inlet and their water height/compartment length ratio. The residence time distribution (RTD) enables the quantification of the oxidation contactors by calculating the hydraulic efficiency (ratio of the time for which 10% of the fluid went out of the contactor, t_{10}) and the hydraulic residence time, τ (HRT, ratio of volume by flow rate). The disinfection efficiency is good or excellent when the hydraulic efficiency is more than 0.5. The RTD (and consequently the HRT) can be experimentally measured or determined using CFD (de Traversay *et al.*, 1998). Previous studies have validated the CFD model used to simulate the ozonation contactors by comparing experimental and numerical results (de Traversay, 2000; Levecq *et al.*, 2001). Figure 1 and Table 1 present the isometric view and the main characteristics of the Kyneton, McCay and Sandhurst ozone contactors, respectively.

The RTD were calculated at the outlet of each compartment for the three ozone contactors and the global hydraulic efficiency was determined (see Figure 2).

The disinfection efficiencies are excellent for the Kyneton and McCay ozone contactors but not for the Sandhurst contactor. The performance of the contactors is linked to their water height/compartment length ratio (7.1 for Kyneton, 8.3 for McCay and 1.1 for Sandhurst). With increasing ratio of water height/compartment length the hydraulic behaviour tends more to a plug flow. Although the ratios of the Kyneton and McCay contactors are very close, the hydraulic efficacy is much more significant for McCay. This is explained by the use of a centered inlet instead of a lateral one as for the Kyneton contactor.

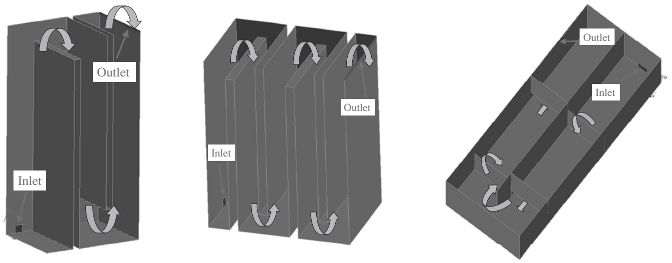
To improve the disinfection efficiency of the Sandhurst contactor, another configuration was tested and consists of the addition of three more baffles (see Figure 3).

The addition of baffles increases the water height/compartment length ratio from 1.1 to 2.1 and the hydraulic efficiency from 0.47 to 0.64 (see Figure 4). The additional baffles enable breakage of the main recirculation loops in the first compartment. The amplitude of the RTD related to the first compartment is less important.

The numerical simulation of the three ozone contactors of the Coliban project has enabled the quantification of the hydraulic performance before construction. According to the CFD study, the Sandhurst ozone contactor design was improved by adding three baffles.

The flocculation tanks

The coagulation–flocculation steps aim to agglomerate the suspended particles to form

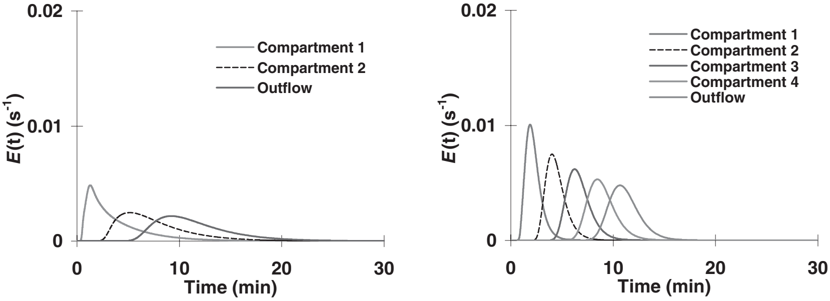


Kyneton McCay Sandhurst

Figure 1 Isometric view of the Kyneton, McCay and Sandhurst ozone contactors

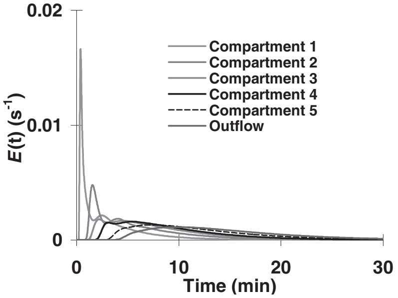
Table 1 Main characteristics of the Kyneton, McCay and Sandhurst ozone contactors

Contactor	Number of compartments	Ratio of water height/compartment length
Kyneton	3	7.1
McCay	5	8.3
Sandhurst	6	1.1



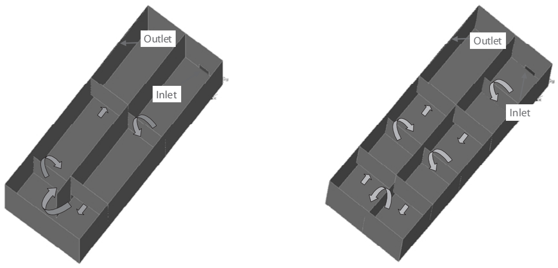
Hydraulic efficiency : 0.61
Kyneton

Hydraulic efficiency : 0.86
McCay



Hydraulic efficiency : 0.47
Sandhurst

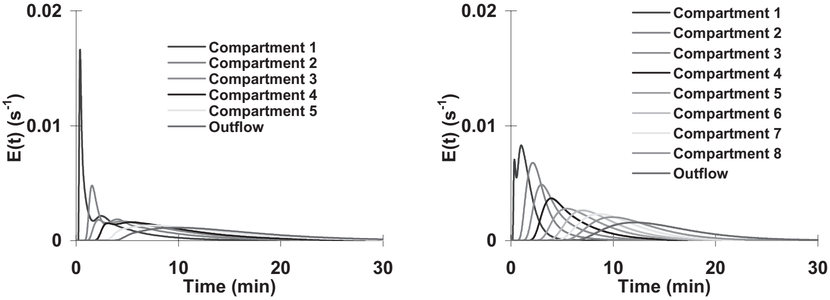
Figure 2 RTD at the outlet of each compartment of the ozone contactors



First configuration

Second configuration

Figure 3 Isometric view of the two configurations of the Sandhurst contactor



Hydraulic efficiency : 0.47
First configuration

Hydraulic efficiency : 0.64
Second configuration

Figure 4 RTD at the outlet of each compartment for the two configurations of the Sandhurst contactor larger flocs that are easier to settle. These flocs are formed in a continuously stirred vessel, and the process efficiency depends on many hydraulic parameters such as the mixing rate, the impeller type, the reactor geometry, the HRT, etc.

CFD has been used to evaluate the mixing performance of a mechanically stirred vessel equipped with various impellers (Essemiani, 2000). In the flocculation process, CFD can be very helpful to evaluate the effect of the hydraulic parameters on the flocculation process, and thus optimise it by testing different scenarios.

Figure 5 presents the geometry of a flocculation tank equipped with an axial impeller in a turbulent regime flow. The velocity field induced by this impeller is thus presented in Figure 6. The highest values are located in the impeller region.

The CFD model used to simulate the mixing has been validated on one hand by a comparison with published measurements of the power dissipated and the impeller pumping flow rate as presented in Table 2. On the other hand, RTD experiments on a pilot were performed for different inlet flow rates and impeller velocities. Figure 7 shows a validation of the numerical model by the RTD for one test. All the tests show a good agreement between the experimental and the numerical RTD.

Figure 8 represents the local gradient *G* distribution in a vertical plane; it highlights the region where maximum and minimum *G* values are located. The rate of floc formation is directly proportional to the velocity gradients in the stirred tank. Until now the flocculation process was described by a global hydraulic gradient *G*. It is equal to the square root of the power dissipated per volume/kinematic viscosity. It does not tell us anything about the

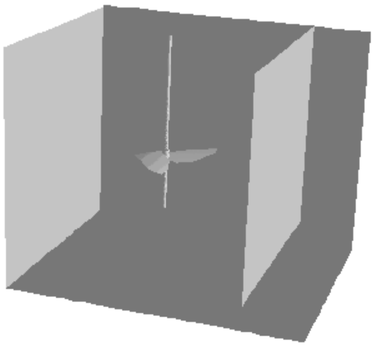


Figure 5 Geometry of the flocculation tank

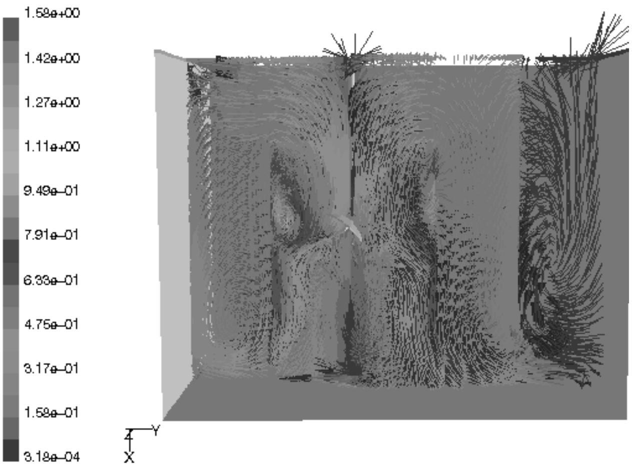


Figure 6 Velocity flow field in a vertical plane

shear rate distribution in the tank. Therefore, we can expect different performances from different mixing rates and impeller types at the same global parameter G. Conversely, the local G distribution obtained by CFD will determine where the break up and the coalescence will occur in the tank.

The UV reactors

The disinfection performance of the UV reactors is directly linked to the lamp irradiation, the exposure time in the reactor and the water UV transmittance. By describing these physical phenomena and properties, CFD enables the determination of the average fluence (AF) and/or the reduction equivalent fluence (REF). Both are calculated from the fluence distribution of the UV reactor for given operating conditions. The AF is just the arithmetic average fluence of the fluence distribution, while the determination of the REF needs the knowledge of the fluence-response curve of the biodosimeter used (see Figure 9). The fluence-response curve corresponds to static manipulations using a collimated beam apparatus. The REF is determined by multiplying each percentage of volume of the fluence distribution by the corresponding biodosimeter residual (Féliers *et al.*, 2001) (taken from the fluence-response curve), giving a total residual. The REF is the fluence corresponding to this total residual.

Bioessays were carried out with *Bacillus subtillis* spores as biodosimeter in the Uvaster® pilot which consists of a tubular stainless steel reactor equipped with an axial medium pressure lamp (see Figure 10). Three setpoints were established by calculation to provide a 400 J m^{-2} average fluence at three UV transmittance values. The fluence distributions were determined using CFD in the same operating conditions (see Figure 11). The CFD methodology consists in coupling the irradiance field calculated by the multiple point source summation model (MPSS) provided by the United States Environment Protection Agency (Bolton, 2000) and the hydrodynamics.

The comparison between experimental (experimental REF) and numerical (REF and AF for two reflection coefficients at the stainless steel wall) results shows that the AF does not represent reality in terms of microorganism reduction (Figure 12). It is necessary to couple the fluence-response curve to CFD to determine the REF. For the lowest transmittance the

Table 2 Comparison of experimental and numerical values of the global parameters

Parameter	Experiment	CFD	% Error
Power dissipated (W)	30	25	16
Pumping flow rate (m^3 h^{-1})	360	285	20

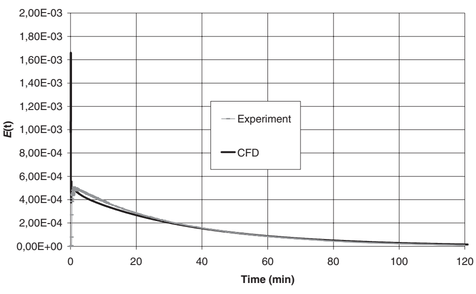


Figure 7 Experimental and numerical residence time distribution

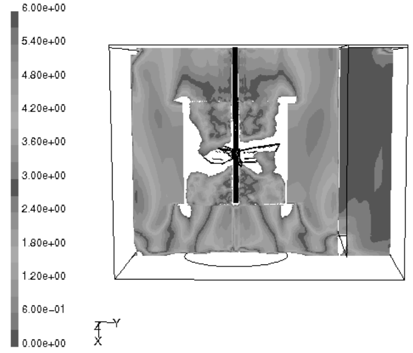


Figure 8 Local velocity gradient G distribution

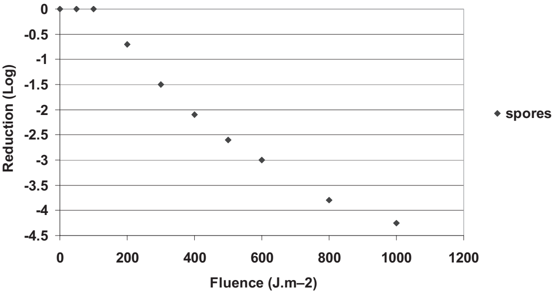


Figure 9 Fluence-response curve of *Bacillus subtillis* spores

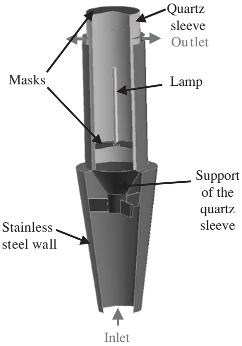


Figure 10 Uvaster® pilot

calculated REF, with reflection or not at the stainless steel wall, is very close to the experimental one. But the reflection must not be neglected for transmittance values of up to 20%.

Secondary settling tanks

The treated wastewater/biological sludge mass separation is achieved by gravity sedimentation in secondary settling tanks (SST). The optimisation of the SST is then controlled by the optimisation of their hydraulic functioning to avoid the re-suspension of the settled sludge (Dahl, 1993; Ekama *et al.*, 1997). This optimisation will allow the surface flow rate to increase to higher values and ensure that the SST sustains the high flow rate due to storm events.

The French public water agency *Agence de l'Eau Seine Normandie* and Anjou Recherche have launched a project to optimise the hydraulic behaviour of the SST using

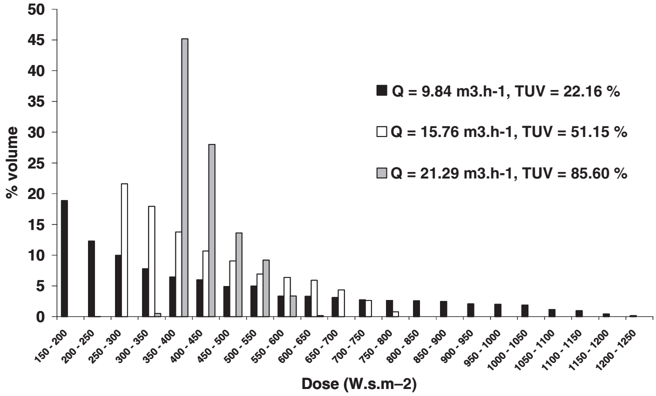


Figure 11 Fluence distributions for different flow rates and UV transmission

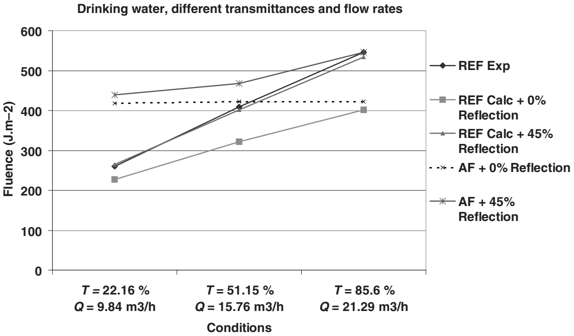


Figure 12 Comparison of the experimental REF and the calculated AF and REF

CFD. The validation of the numerical results is made with comparison with experimental measurements of the sludge blanket height and the total suspended solids (TSS) at different points of the SST of Morlaix WWTP, France. This comparison is made for wet weather operating conditions and after a storm event. The operating conditions of the dry and the storm events are presented in Table 3.

The SST of Morlaix is presented in Figure 13. It is 27 m in diameter equipped with a clifford (0.7 m long and 5 m in diameter) and a horizontal deflector of 4 m diameter situated at 1.5 m below the free surface. The slope is 20%. The tank volume is 2,335 m^3 and the mean depth is 3.9 m. For these operating conditions TSS profiles were measured at different positions, Figure 14 and Figure 15 represent the numerical and experimental profiles of TSS at different positions for dry weather and after a storm, respectively.

Globally the experimental and numerical TSS profiles are close, except close to the bottom of the clarifier. In this region the difference is important. Two reasons may explain this discrepancy. First, there are not enough experimental points in this region, the TSS is increased by a factor 2 in 35 cm, and there is only one measurement point. Second, in our simulation we considered the sludge viscosity as constant, but in reality the sludge viscosity is modified due to the settling in this region.

The increase of the hydraulic rate due to the storm is taken into account correctly by the simulation. Table 4 presents the simulated and measured height of sludge zone, the difference is low in comparison with the measurement error (15%).

Figures 16 and 17 represent the sludge volume fraction for the dry and storm weathers. In the normal conditions the sludge blanket is located at the tank bottom. As soon as the hydraulic rate is increased due to the storm, the sludge blanket increases and the higher flow rate induces vortices in the clifford zone that mixes the sludge.

The CFD model has been validated for experimental measurements. The model correctly predicted the increase of the hydraulic rate due to a storm event and its effects on the height of the sludge zone and the TSS profiles in the tank. It is now possible to optimise the SST for the permissible hydraulic flow rate, optimal dimensions of the inlet, recirculation and outlet, and the testing of the existing configurations.

Table 3 Operating conditions

Parameter	Dry weather	Storm
Sludge volume index (mL.g^{-1})	260	250
Influent flow rate (m^3.h^{-1})	238	1000
Recirculation rate (%)	156	133
Mixed liquor SS (g.L^{-1})	2.30	2.50
Return activated sludge SS (g.L^{-1})	3.75	2
Outlet TSS (mg.L^{-1})	3	10
Comparative sludge volume (mL.L^{-1})	598	625
Solid loading rate (kg.m^{-2}h^{-1})	0.96	4.36
Surface overflow rate SOR (m.h^{-1})	0.16	0.75
Height of sludge zone (m)	–1.75	–3.25[a]

[a] The reference plane is the water surface level, the sludge blanket height is thus negative.

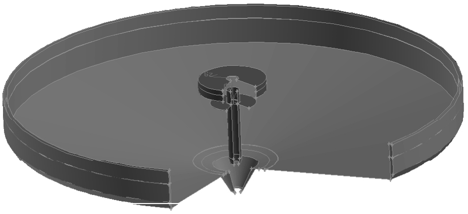


Figure 13 Morlaix's SST

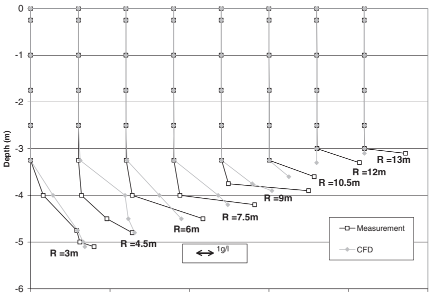


Figure 14 TSS profile: dry weather

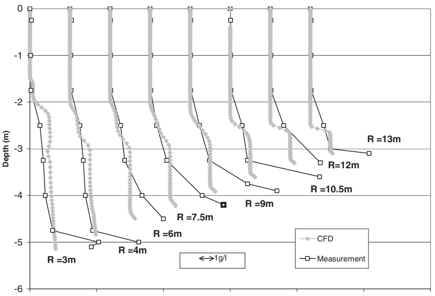


Figure 15 TSS profile: storm

Table 4 Comparison of simulated and measured height of sludge zone

Case	Height of sludge zone (m) – measurements	Height of sludge zone (m) – CFD	Difference (%)
Dry weather	–3.25	–3.20	2
Storm	–1.75	–2.10	17

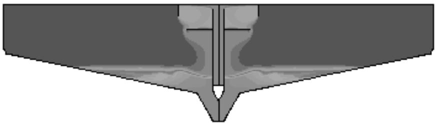


Figure 16 Sludge volume fraction in dry weather

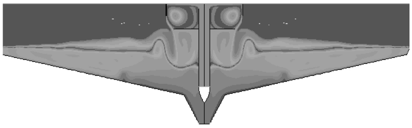


Figure 17 Sludge volume fraction in a storm

Conclusion

The very wide variety of applications proves that Computational Fluid Dynamics (CFD) answers a large proportion of hydraulic questions in the water industry. CFD is a helping-hand tool for retrofitting, optimisation or design of processes in both drinking water and wastewater treatment. One of the next goals is to develop CFD abilities to describe the coupling between the hydrodynamics and the already known, as well as the new biological–chemical models.

References

Bolton, J.R. (2000). Calculation of ultraviolet fluence rate distributions in an anular reactor: signifance of refraction and reflexion. *Wat. Res.*, 34(13), 3315–3324.

Dahl, C. (1993). *Numerical Modelling of Flow and Settling in Secondary Settling Tanks*. ATV project EF-359.

de Traversay, C. (2000). *De la Mécanique des Fluides à la Modélisation Systémique: Application aux Réacteurs D'oxydation en Potabilisation*. PhD Thesis, University of Rennes, France.

de Traversay, C., Luck, F., Wolbert, D. and Laplanche, A. (1998). CFD: a useful tool for designing the ozonation contactor. *Proceedings IOA Conference*, Poitiers, 23–25 September 1998, 5(25), 1–10.

Ekama, G.A., Barnard, J.L., Günthert, F.W., Krebs, P., McCorquodale, J.A., Parker, D.S. and Wahlberg, E.J. (1997). *Secondary Settling Tanks: Theory, Modelling, Design and Operation*. IAWQ, Scientific and Technical Report No. 6.

Essemiani, K. (2000). *Potentialities of Computational Fluid Dynamics for the Mixing Tanks*. PhD thesis, University of Toulouse, France.

Féliers, C., Laplace, C., Vidal, A., Levecq, C., de Traversay, C. and Cervantes, P. (2001). *Evaluation and monitoring of the fluence in a vertical reactor using biodosimetry, actinometry and CFD approaches*. IWA 2001, Berlin, 15–19 October 2001 (Preprint).

Levecq, C., Essemiani, K. and de Traversay, C. (2001). Computational Fluid Dynamics Application in Water Treatment. *CCWI 2001 Conference*, Leicester, 3–5 September.