Why Didn't The Supersonic Car Fly?*

KENNETH MORGAN C.Math.FIMA, OUBAY HASSAN AND NIGEL WEATHERILL C.Math.FIMA
University of Wales Swansea

The supersonic car, *ThrustSSC*, took the World Land Speed Record beyond the speed of sound on the Black Rock Desert in Nevada in October 1997. To achieve this feat, many challenging technological problems had to be addressed. One such problem was the aerodynamic design of the vehicle to ensure that it could be safely operated and, in particular, that it remained in contact with the ground at all speeds. Here we outline the role that was played by computational fluid dynamics in assisting the process of aerodynamic design.

INTRODUCTION

The first World Land Speed Record was set by Count Gaston de Chasseloup-Laubat in Achères, France on December 12 1898. Driving an electric vehicle, he set the Record at 39 mph. Since that initial event, the Record has been broken around sixty times and some of the major milestones achieved before 1997 are noted in Table 1. Initially, electric cars dominated and it was not until 1902 that a car powered by an internal combustion engine captured the Record. The first jet powered Record breaker was Donald Campbell's *Bluebird* in 1964. In the 1920s, Pendine Sands in South Wales was an attractive location for Record breaking. In fact, the Record was broken 5 times on Pendine Sands during this period by Malcolm Campbell and Parry Thomas. They took the Record from 146 to 175 mph. Interestingly, seventy years later, Pendine was also to play a role in assisting in the successful development of the supersonic car *ThrustSSC*. The exact nature of this role will be described later. The full list of Record breakers is dominated by British and American drivers while France, for example, has not held the Record since 1924. In a Record attempt, the recorded speed is the average speed achieved, over a measured mile, in two runs which must be made in opposite directions within a time interval of less than one hour. The vehicle must possess some basic characteristics eg it must have four wheels and a driver!

Table 1. The World Land Speed Record: major milestones achieved before 1997

Year	Driver	Nationality	Speed Attained (mph)
1898	Gaston de Ch-Laubat	France	39
1904	Louis Rigolly	France	103
1927	Henry Segrave	UK	203
1935	Malcolm Campbell	UK	301
1964	Donald Campbell	UK	403
1964	Craig Breedlove	USA	526
1965	Craig Breedlove	USA	600
1983	Richard Noble	UK	633

THE ThrustSSC PROJECT

In the early 1990s, Richard Noble, who held the Record at that time with the speed of 633 mph attained by *Thrust2*, began to think about breaking the Record again. An obvious initial target was 700 mph, but as this was not too far distant from the speed of sound at ground level, which is around 760 mph, he decided that he would assemble a team to attempt to take the Record to supersonic speed, ie faster than the speed of sound. Experience had shown, and we'll touch upon this again shortly, that this was not just going to be a matter of making minor modifications to *Thrust2*. This attempt was going to require a completely new design. The major challenge was not just to design a vehicle that could attain very high speeds, but to ensure that it could do this safely. This posed certain major technical difficulties. For example, to maintain the rigidity of the structure and the integrity of the wheels at high speeds required the identification of suitable materials; to ensure the basic stability of the vehicle required adequate control mechanisms; an appropriate propulsion system had to be selected and a decision was required concerning the number, and the type, of engines to be used. However, to answer the question posed by the title of this article, an understanding of the aerodynamics of the vehicle, ie the interaction between the moving vehicle and the air, was required. Therefore, we will concentrate here upon certain aspects of the aerodynamic design, as it was in this area that our work on computational fluid dynamics made an impact in the *ThrustSSC* project. Readers interested in discovering more about this, and other aspects of the supersonic car project, should consult the excellent text by Richard Noble[1] or the book produced by members of the *Thrust* Team[2].

*Based upon a public lecture presented at the University of Wales Aberystwyth, 29 October 1998

BASIC AERODYNAMIC DESIGN

The overall aerodynamic design of *ThrustSSC* was the responsibility of Ron Ayers. Important features of his basic original design are apparent in Figure 1, which is an early artist's impression of *ThrustSSC* at high speed on the Black Rock Desert in Nevada. The moving vehicle is subjected to the aerodynamic forces of lift and drag, and the magnitude of these

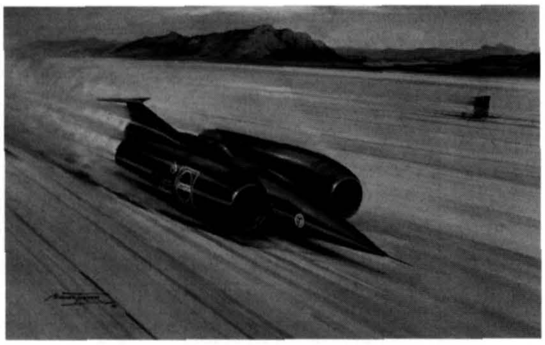


Fig. 1. An early artist's impression of ThrustSSC at speed on the Black Rock Desert in Nevada (reproduced with permission)

forces is governed by the vehicle's shape and speed. The design shows a long slender shape, to produce a low drag and also a small drag variation with increasing speed. The long wheelbase provides for stable steering and the longitudinal distribution of cross-sectional area of the vehicle is smooth. The power is provided by two Rolls Royce Spey jet engines. With the use of two engines, positioned as shown in the Figure, the centre of gravity can be located towards the front of the vehicle, for enhanced stability, and the front wheels can be widely spaced, to improve resistance to roll. In addition, the driver can be positioned, in the strongest part of the vehicle, near the centre of gravity, enabling rapid feedback response. Yaw stability is provided by a conventional highly swept tail-fin, with the horizontal fin mounted high to avoid the jet efflux from the engines. The attitude of the vehicle is controlled by an active suspension system. The technical soundness of these initial considerations helped ensure that this early artist's impression looked remarkably similar to the final design.

Assuming that the engines will produce enough thrust to overcome the drag at all speeds, the key problem for the designer is to optimise the shape to ensure that the lift force remains within bounds; too large a positive lift will invalidate the assumption that gravity will keep the vehicle on the ground, while a significant negative lift would destroy the vehicle's suspension. In the aerospace and related industries, the aerodynamic performance of new designs has traditionally been investigated by using wind tunnel experiments. In such experiments, a scale model of the vehicle, made to a high degree of accuracy, is held in the working section of the tunnel; air is passed over the model and the forces and moments on the model are measured. When performing the experiments, appropriate scaling factors have to be employed, to ensure that the main aerodynamic parameters are close to those encountered in the real, full scale, flow. Although wind tunnel testing has been a key ingredient in the design of most aircraft in use today, the approach is lengthy and expensive, with a single modern design often utilising thousands of hours of tunnel testing time. The building of models is costly, and minor changes in geometrical shape often require the construction of a new model. The tunnels themselves are expensive to build and operate and they have limited applicability for a full range of flight conditions.

The high speed wind tunnel testing of aircraft in cruise conditions normally involves air being passed over models which are held well away from the tunnel walls. For *ThrustSSC*, the correct experimental procedure should involve moving a model at high speed relative to a stationary simulated ground or moving the simulated ground at high speed with respect to a stationary model. Tunnel facilities capable of creating either of these scenarios were not available. This meant that, if tunnel testing was to be employed, the best that could be envisaged would be tests in which a model was held at rest close to a simulated stationary ground in a high speed stream. In fact, this approach had already been employed, with a limited degree of success, in the aerodynamic design of *Thrust2*. It is now known that this vehicle was operating at the limits of its capability, and it has been estimated that it would have lost contact with the ground if its peak speed had been only seven miles per hour faster[2]. Thus, it was felt that an alternative approach was necessary if the aerodynamic performance of *ThrustSSC* was to be confidently predicted over a range of speeds up to supersonic.

Computational Fluid Dynamics

Over the past thirty years, the aerospace industry has been making significant developments in an alternative testing procedure, based upon the use of computer simulation methods for the analysis of the aerodynamic performance of vehicle designs[3]. During this period, as wind tunnel costs have increased, the cost of high performance computers has decreased, and computers capable of performing certain complex flow simulations are now widely available. The process of using computers in this way to simulate realistic flows is termed computational fluid dynamics. In 1992, we were asked to consider if the computational fluid dynamics techniques that we had developed could be applied to assist in the design of *ThrustSSC*.

In computer form, the geometry of vehicle designs can be readily defined and modified and, hence, computational fluid dynamics offers the aerodynamicist a means of exploring a

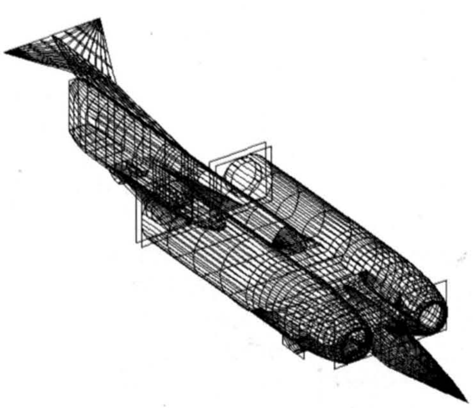


Fig. 2. The geometrical description of the vehicle surface

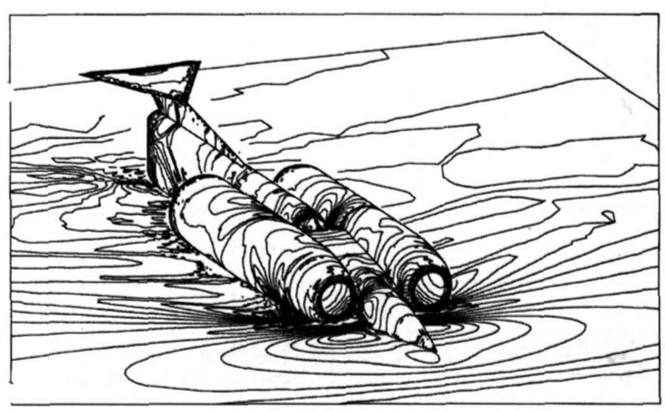


Fig. 3. Distribution of computed pressure contours for the vehicle at supersonic speed

wider range of vehicle shapes than can usually be accomplished, in available time scales, with wind tunnel testing alone. However, computational fluid dynamics has its own associated shortcomings. These are generally related to difficulties in modelling mathematically, and computing, flows involving the complex phenomena associated with extremes of aerodynamic design, such as the prediction of flow separation and turbulence. Lower order mathematical representations of fluid flow, involving simpler flow physics, can avoid some of these difficulties, while still providing useful information for many practical aerodynamic flows[4]. In the context of *ThrustSSC*, it was decided that a lower order computational model, based upon the assumption that the fluid was inviscid, would be appropriate. As no large regions of separated flow were likely to occur within the projected speed envelope, it was reasonable to expect that such a model would be capable of producing a good approximation to the distribution of pressure over the vehicle. This information would enable the lift force on the vehicle to be estimated. By adopting this choice of model, it was also possible to ensure that many different geometrical shapes could be analysed within the time and financial constraints that were being imposed by the project. Following the initial design phase, and before the decision was taken to proceed with the construction of *ThrustSSC*, the validity of employing this form of computational model would be investigated by performing a limited series of experimental rocket sled tests on a scale model of the vehicle. The results of this validation exercise will be presented below.

Flow Modelling for ThrustSSC

Modelling the air flow over *ThrustSSC* was accomplished by using the FLITE3D computer system at the University of Wales Swansea. The input to this system is the definition of the vehicle geometry, in the form of an assembly of mathematically defined surfaces and their intersection curves. The computational domain was defined to be the region surrounding the vehicle, and extending a prescribed distance from it in all directions. For computational efficiency, it was assumed that the flow was symmetric about the vertical plane through the central axis of the vehicle from the nose to the tail, so that only the flow over one-half of the vehicle was simulated.

The FLITE3D system requires that the computational domain be divided into an unstructured assembly of tetrahedral cells. To accomplish this, the boundary of the domain is first discretised into an assembly of triangular planar facets and the discretisisation of the domain volume then follows. These discretisation processes are fully automatic and generate points, at cell vertices, according to a user-specified point spacing distribution function[5, 6].

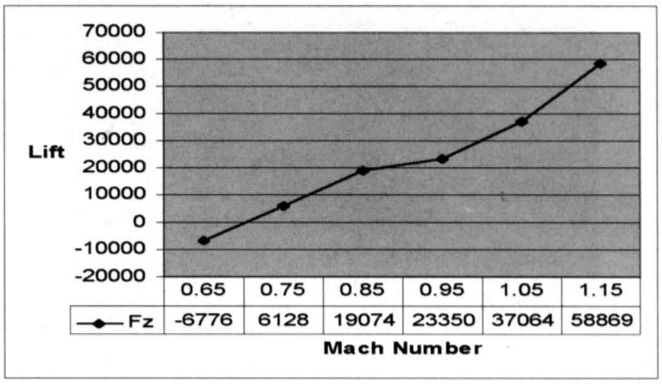


Fig. 5. Computed variation of lift with Mach number for the full vehicle

The equations governing inviscid rotational flow are the compressible Euler equations. These equations, expressing the conservation of mass, momentum and energy in the fluid, are considered in the conservative form

$$\frac{\partial \mathbf{U}}{\partial t} + \sum_{j=1}^{3} \frac{\partial \mathbf{F}^j}{\partial x_j} = 0$$

where the unknown $\mathbf{U}$ and the flux vectors $\mathbf{F}^j$ are defined by

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho\varepsilon \end{bmatrix} \qquad \mathbf{F}^j = \begin{bmatrix} \rho u_j \\ \rho u_1 u_j + p\delta_{1j} \\ \rho u_2 u_j + p\delta_{2j} \\ \rho u_3 u_j + p\delta_{3j} \\ (\rho\varepsilon + p) u_j \end{bmatrix}$$

Here $Ox_1x_2x_3$ is a cartesian coordinate system, t denotes the time, u_j is the fluid velocity in direction x_j, δ_{ij} is the Kronecker delta and p, ρ and ε denote the fluid pressure, density and total specific energy respectively. For the air flow simulations that are of interest here, the equation set is completed by the addition of the perfect gas equation of state. Before the solution of these equations can be attempted, appropriate boundary conditions must be prescribed on each surface bounding the computational domain.

The solution algorithm of FLITE3D is based upon an integral Galerkin approximate variational formulation of this classical problem statement[7]. To produce a practical algorithm for the simulation of high speed flows, consistent stabilisation and discontinuity capturing terms have to be added to this basic formulation. In the algorithm, the solution vector, $\mathbf{U}$, is assumed to vary linearly over each tetrahe-

	M = 0.65				M = 0.75				M = 0.85				M = 0.95			
	Fx	Fy	Fz	My	Fx	Fy	Fz	My	Fx	Fy	Fz	My	Fx	Fy	Fz	My
Nose	1903	-11466	-2531	-4658	2880	-18042	-2647	-4650	4280	-27546	-2966	-5097	6541	-42682	-2983	-4939
Centre Body (top)	-4524	34360	51485	-93018	-7017	43148	71337	-106185	-8442	58085	94762	-109792	-6376	88725	141641	-133698
Centre Body (bottom)	1986	-537	-56189	102523	3425	-311	-62214	80245	5818	785	-72716	74487	8570	3107	-115864	147292
After Body and Fin	686	10916	3088	-21442	1078	16693	3754	-26504	1416	23342	5240	-38878	2285	26540	7978	-50980
Tailplane	409	-108	-3265	31028	701	-172	-5152	50220	967	-232	-6828	67438	1286	-275	-8738	86728
Rear Wheel	413	-0.5	58	-851	602	-0.45	220	-1938	874	-0.3	434	-3797	914	-0.05	-155	1355
Inside Engines	920	601	578	1310	1466	864	830	1900	2208	1155	1148	2649	3180	1441	1471	3482
Total	**1793**	**33766**	**-6776**	**14892**	**3134**	**42180**	**6128**	**-6912**	**7121**	**55589**	**19074**	**-12990**	**16400**	**76856**	**23350**	**49240**

Fig. 4. Computed forces and moments on the vehicle components at different Mach Numbers

dral cell. Finite difference procedures are employed to discretise the time dimension and the solution is advanced by using a standard multi-stage explicit time stepping procedure. The computational implementation was designed to maximise efficiency on CRAY supercomputers with vector architecture and multi-tasking facilities.

The results of any computational simulation may be presented in both qualitative and quantitative form. An overall impression of the flow is obtained by using black and white, or colour-shaded, contours of selected flow variables. From such plots, flow features such as shock waves are readily detected. A more detailed analysis of the predicted aerodynamic performance can be determined from quantitative data, such as the contribution made to the lift and pitching moment by the individual geometrical components.

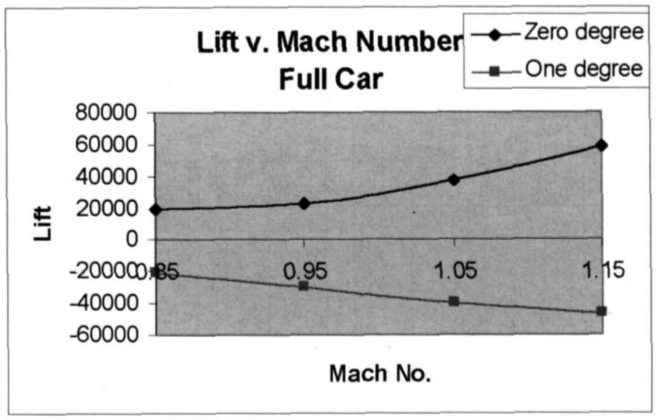


Fig. 6. The variation of lift with Mach number for the vehicle in normal attitude (Zero degree) and with the attitude changed to one degree nose down

The Simulations

Initially, computational simulations were employed to assist in the design of the nose cone and of the engine intakes. For these simulations, only the flow over the front section of the vehicle was analysed. This phase was followed by full vehicle simulations, which included the effects of the powered engines. For this stage, the geometry of the vehicle was described by an assembly of 56 surfaces, as illustrated in Figure 2. A rectangular box was employed to define the outer surface of the computational domain. The point spacing distribution function, which controls the domain discretisation process, was constructed so as to ensure that an adequate density of points was achieved in perceived critical areas of the domain. Based upon these considerations, a typical discretisation of the boundaries of the domain consisted of around 50,000 triangles, while a typical volume discretisation involved about 1 million tetrahedra. The computations were performed on a CRAY C90 computer and each steady state simulation required about 1 hour of cpu time.

Typical output from the computer simulations is displayed in Figure 3, which shows the distribution of contours of pressure on the ground and over the vehicle surface at supersonic speed. The quantitative data extracted from these computations provided the necessary information to drive the optimisation of the aerodynamic design. The facility to extract data particular to individual geometrical components of the vehicle, as shown in Figure 4, was particularly important in the evolution of the shape. In Figure 5, the computed lift is displayed for vehicle Mach numbers in the range 0.65 to 1.15. The vehicle Mach number is the ratio of the vehicle speed to the local speed of sound in air. It is clear that the lift increases rapidly as the vehicle approaches and exceeds the speed of sound. In this Figure, the unit on the vertical axis is pounds. Since the weight of the fully loaded vehicle is around 20 000 pounds, it is apparent that the lift needs to be reduced if the vehicle is to remain in contact with the ground at high speed. The lift could be altered by changing the vehicle shape. An alternative would be to change the attitude of the vehicle. In this case, the lift is altered as a different effective shape is presented to the oncoming air stream. Figure 6 shows the computed effect of a change of this type. It compares the variation of lift with Mach number for the vehicle in its normal attitude with that for the vehicle with its attitude changed to one degree nose down. The result of this change is seen to be significant, with the predicted lift decreasing with Mach number in the nose down case. Although these results were encouraging, and indicated that the aerodynamics of a supersonic vehicle could be controlled, independent validation of the modelling approach was necessary before a vehicle, designed on the basis of these predictions, could be operated confidently at high speeds.

Validation of the Simulation Procedure

The FLITE3D computer system had been extensively validated on aerospace geometries for a wide range of vehicle Mach numbers. However, it had never been applied previously to the simulation of a vehicle travelling at transonic and supersonic speeds near the ground. The ground effects, including reflection of shock waves between the ground and the underside of the vehicle, were unknown but clearly important in the design process. To check the validity of the mathematical and computational modelling that was being adopted, the *ThrustSSC* Team decided to undertake rocket sled tests at the Defence Test and Evaluation Organisation (DTEO) at Pendine Sands in South Wales. At the testing ground, 13 rocket powered runs were performed, using a 1:25 scale model of the vehicle. The model was fitted with nine pressure sensing gauges on its upper and lower surfaces and vehicle Mach numbers of 0.71, 0.96, 1.05 and 1.08 were attained. Computational simulations were performed at Swansea without access to the test data. A detailed comparison between the computational and test data was then undertaken by Ron Ayers. His original plot of corresponding pressure values is shown in Figure 7. Perfect agreement between the computational results and the test data would have resulted in a straight line at 45 degrees to the horizontal axis. The plot, therefore, shows a remarkable correlation between the two data sets. In addition, if conventional correction techniques for inviscid flow are applied[8], even the data points which do not appear to lie on the straight line are also brought into agreement.

This comparison, which was undertaken at the end of the initial design phase, validated the use of the computational fluid dynamics procedure for simulations of the flow over *ThrustSSC*. This exercise was critical to the success of the aerodynamic design process, as the excellent agreement which had been achieved provided the designer with the confidence nec-

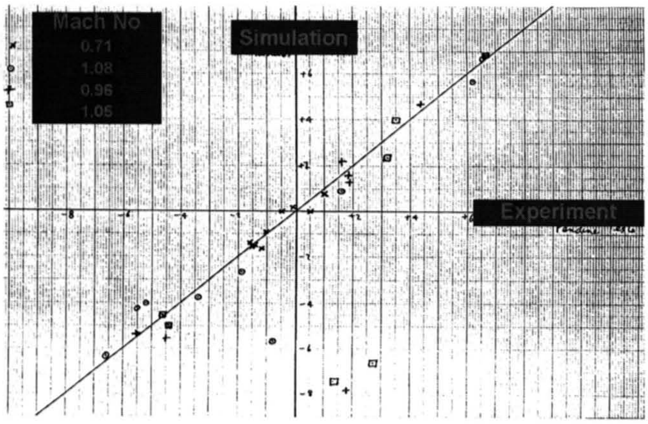


Fig. 7. The comparison between the pressure values observed in the Pendine experiments and the values predicted in the computer simulations

essary to enable him to use computational fluid dynamics predictions to guide and support design modifications throughout the full speed range. In particular, based upon results such as those shown in Figure 6, *ThrustSSC* was operated with a variable attitude of between zero and one degree nose down during the record breaking attempts.

CONCLUSIONS

Computational fluid dynamics technology, originally designed and developed to support the aerospace industry, was successfully used, over a period of five years, to assist in the design of the supersonic car *ThrustSSC*. The accuracy of the approach was validated by comparison with independent results produced by employing rocket powered models at the Pendine Testing Range in South Wales. At about 1000 Nevada time on October 15th 1997, *ThrustSSC* broke the World Land Speed Record and reached a supersonic speed of Mach 1.02 (763.035 mph).

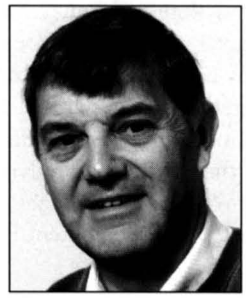
ACKNOWLEDGEMENTS

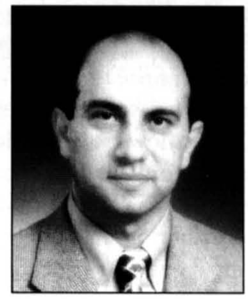
The authors would like to acknowledge the support and encouragement that they received throughout this project from Ron Ayers, who provided the key aerodynamic design input and the geometrical definition of *Thrust SSC*, and Richard Noble.

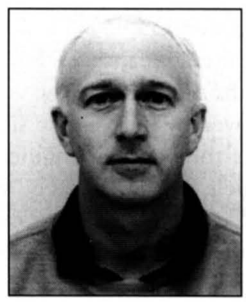
Certain computations were performed using computer time made available by Cray Research (UK) Ltd.

References

1. Noble, R. (1998). *Thrust,* Transworld Publishers, London.
2. The *Thrust* Team, (1998). *The Story of ThrustSSC,* Corgi Books, Transworld Publishers, London.
3. Raj, P. (1998). CFD at a crossroads: an industry perspective, in D A Caughey and M M Hafez (editors), *Frontiers of Computational Fluid Dynamics 1998,* World Scientific, Singapore, 429–448.
4. Jameson, A. (1989). Computational aerodynamics for aircraft design, *Science* **245**, 361–371.
5. Peiró, J, Peraire, J and Morgan, K. (1999). Advancing front grid generation, in J F Thompson, B K Soni and N P Weatherill (editors), *Handbook on Grid Generation,* CRC Press, Boca Raton, 17.1–17.22.
6. Weatherill, N P and Hassan, O. (1994). Efficient three-dimensional Delaunay triangulation with automatic point creation and imposed boundary constraints, *International Journal for Numerical methods in Engineering 37,* 2005–2039.
7. Morgan, K and Peraire, J. (1998) Unstructured grid finite element methods for fluid mechanics, *Reports on Progress in Physics* **61**, 569–638.
8. Lock, R C and Firmin, MCP. (1982). Survey of techniques for estimating viscous effects in external aerodynamics, in K W Morton and M J Baines (editors), *Numerical Methods in Aeronautical Fluid Dynamics,* Academic Press, London, 337–430.







Kenneth Morgan is a Professor in the Department of Civil Engineering at the University of Wales Swansea. He is also currently Dean of the Faculty of Engineering.

Oubay Hassan is a Senior Lecturer in the Department of Civil Engineering at the University of Wales Swansea.

Nigel Weatherill is a Professor in the Department of Civil Engineering at the University of Wales Swansea. He is also currently Chairman of Engineering.

Answer to Enigmaths 63 – Chessboard 6

[1] 8	[2] 1	[3] 2	[4] 5
[5] 1	4	9	6
8	[6] 9	8	[7] 3
[8] 3	8	9	4