ICAS-94-5.5.2

COMPUTER ANALYSIS AND SIMULATION OF AIRCRAFT FUEL SYSTEMS

BY

I. PLASTOW, BRITISH AEROSPACE AIRBUS LIMITED, FILTON, UK.
AND
D.A. COWLING*, UNIVERSITY OF BRISTOL, UK.

Abstract

Currently the design of aircraft fuel systems is accomplished primarily from experience gained over many earlier programmes and has relied on expensive test results for flight clearances rather than comprehensive computer-based simulations. With the advent of high powered inexpensive computer systems the possibility exists for the application of computer based methods to reduce the cost of testing and to refine designs by conducting analyses which are too complex by conventional methods. This paper reviews the results of a programme to investigate the effectiveness of computational methods in the fuel system design process. The application of network models is investigated, with new forms of analysis which have hitherto been impractical using conventional methods. A novel approach to the analysis of component performance is introduced, with the application of computational fluid dynamic (CFD) analysis to determine the operational characteristics of complex pipe components. It is concluded that the application of computational methods to the design of aircraft fuel systems enables a greatly increased number of analyses to be undertaken in a reduced time. Ultimately the application of computational methods will result in a better quality, safer design, at a reduced cost.

Introduction

Aircraft fuel systems have advanced significantly from early aircraft when they simply supplied fuel to the engines and there was little restriction on space and positioning of tanks. With the advancement in aerodynamic design and the introduction of the gas turbine engine there became a requirement for today's aircraft to travel further and faster. In line with this development has been the refinement of airworthiness safety requirements. As a result, fuel systems have developed considerably from their predecessors with an increased emphasis on safety and fuel utilisation. Today's aircraft fuel systems can be considered as complex fluid networks that not only supply fuel to the engine, but can also control the position of the centre of gravity of the aircraft, provide wing bending relief and act as a heat sink to other systems. Typically a modern large transport aircraft's fuel system will contain in excess of thirty control valves, fifteen pumps and numerous specialised components, all of which are required to perform in both the heat of a tropical day (54°C) and the cold of the troposphere (-50°C).

Whilst fuel systems themselves have advanced considerably, the design and analysis methods have not developed at the same rate. Indeed the methods commonly used today are similar to those used on earlier aircraft. Current analyses to verify the operation of the design are based around steady state graphical methods. These methods are adequate for the steady state analysis of simple systems. When the analysis requirements, size and complexity of today's aircraft fuel systems are considered the capability of these graphical methods is severely limited. Furthermore there is an increasing requirement for dynamic analysis of these systems for which

---

* Also Stirling Dynamics Ltd, Bristol, UK.

graphical methods are impractical.

Fuel system components are rarely available 'off the shelf', due to their non-standard nature, stringent safety requirements and extremes of operation. Often no performance data are available particularly for complex shaped components such as junctions. In these cases a means of determining their performance is required.

This paper reviews the results of a British Aerospace Airbus Ltd. and University of Bristol collaborative programme to investigate the application of computational methods to the fuel system design process to overcome the shortcomings outlined above. The main aims of this programme have been to investigate and introduce:

- Computational network analysis methods.
- Computational fluid dynamics (CFD) to the modelling of component characteristics.

The major results of this programme are discussed in the context of network analysis and CFD component modelling and are illustrated with practical examples. The paper closes with a case study which highlights the development of a CFD model of the operational characteristics of a complex component. The results of which are then applied in a network analysis.

Network Analysis

The aim of network analysis is to determine the pressure and flow rate at various points within a network. A network is a system of components linked to form a circuit. For the purpose of network analysis, a component can be described by the equations and data governing its inlet and outlet pressure and flow rate. In addition to the pressure and flow rate, data such as pump speed, valve position etc., may be calculated. These variables may be used for information purposes, or may be required to define component performance and thus have an important effect on system behaviour.

The pressure and flow rate of a component can be defined in terms of a pressure loss and compressibility effects. Pressure loss is defined[1,2] by the 'general loss equation':

$$\Delta p_i = k_i \rho \frac{u_i^2}{2} \tag{1}$$

where Δp_i is the pressure loss across the ith component, u_i the velocity of the fluid flow, ρ the density of the fluid and, k_i is an empirical value relating the pressure loss to u_i^2 and is referred to as the loss coefficient. Expanding for volumetric flow rate and rearranging gives:

$$q_i = \sqrt{\frac{2A_i^2 \Delta p_i}{k_i \rho}} \tag{2}$$

where A_i is the cross-sectional area of the ith component.

The rate of change of pressure due to compressibility effects is defined as:

$$\frac{dP_i}{dt} = \frac{\beta}{V_i} \sum q_i \tag{3}$$

where β is the bulk modulus of the fluid and V_i is a compliant volume in which the rate of change of pressure dP_i/dt can be considered to be constant. From equation (3) flow rate is defined as:

$$\sum q_i = \frac{V_i}{\beta} \dot{P}_i \tag{4}$$

where for steady state conditions.

$$\sum q_i = 0 \tag{5}$$

Two approaches have been adopted for network analysis through the use of two commercial software packages. The first uses numerical integration of the dynamic equations (4) with pressure loss accounted for by the general loss equation (2) using Bath*fp*[3]. The second is the iterative solution of the set of general loss equations (2) for the network using Flowmaster[4,5].

Steady State Analysis

Both the numerical integration and iterative solution methods have been successfully used for the analysis of steady state conditions within a network. The iterative solution method solves directly for steady state conditions using only equation (2). Whereas the numerical integration solution method solves equations (2) and (4) for steady state conditions via a dynamic analysis with steady state inputs ie. constant inlet flow rate and pressure.

All of the sub-systems which make up an aircraft fuel system require steady state analysis to determine their design pressure and flow rates. Traditionally these analyses have been carried out using graphical methods. With the increased complexity of aircraft fuel systems and the refining of airworthiness requirements it is desirable to replace traditional methods with computational methods.

The application of computational methods to steady state analysis in place of traditional methods both increases the number and complexity of analyses that can be carried out, and reduces the time required for analysis. This has provided an increased understanding of the operation and performance of the systems.

An example of a network that has been analysed to determine its performance under steady state conditions is shown in Figure 1.

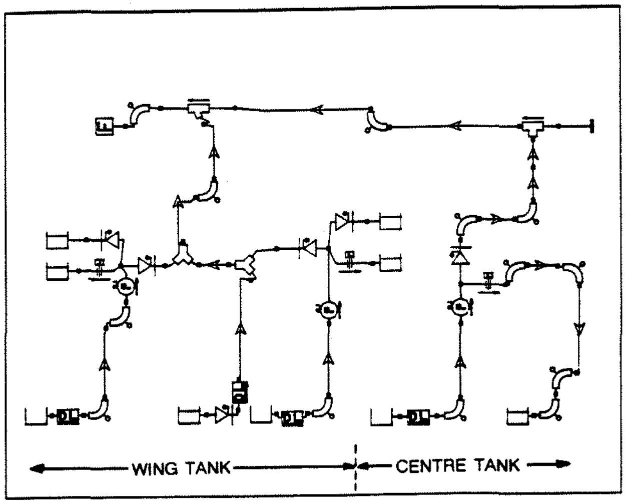

Figure 1 Typical Steady State Network

This network is interesting because it highlights problems with the application of the iterative solution method when multiple solutions are possible. For this method the solution obtained is dependent on where the first iterations predict an initial estimate of solution. This can be illustrated if a simplified system characteristic for a tri-stable system with an appropriate pump characteristic as shown in Figure 2 is considered.

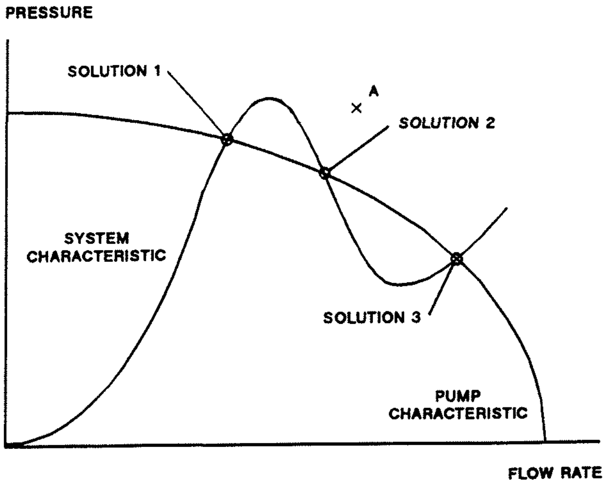

Figure 2 Simplified Tri-Stable System and Pump Characteristic

If the initial iterations predict a solution at *A* then the iterative solution will solve at solution *2*. Although this is a mathematically correct solution, it may not be achievable in the actual system. The solution achieved by the actual system will be dependent on which direction it is 'approached' along the system curve, ie. if the flow rate is increasing or decreasing. If we consider the flow rate to be increasing from zero flow for the characteristic depicted in Figure 2, then solution *1* will be achieved. If the solutions were in close proximity on the system curve the iterative solution can oscillate between solutions. The dynamic solution method does not suffer from these problems because the solution is always 'approached' along the system characteristic, therefore the solution achieved is dependent on the initial conditions set as for the actual system.

Dynamic Analysis

Both the numerical integration and iterative solution methods have been successfully used for the analysis of dynamic conditions within networks. The iterative solution method solves for a dynamic case through a series of steady state analyses carried out at a fixed user selected time step, with any time dependent properties such as valve position, pump speed etc. adjusted as appropriate at each time step. Inertias, rates of change of pressure etc. are taken into account with approximations of the differential equations.

Dynamic analysis is applicable to many of the sub-systems which make up an aircraft fuel system, and its application has greatly expanded the range and complexity of network analyses that can be undertaken. This has resulted in a greatly increased understanding of the operation of systems.

Without the computational methods many of the analyses requiring a dynamic analysis would have been carried out by steady state analysis of the critical conditions with allowances made for any dynamics. In cases were analysis is not possible, reliance has been placed on test rig, ground and flight test results for confirmation of design parameters. These approaches have by necessity resulted in 'over-designed' solutions. The increased understanding of the system provided by dynamic network analysis results in the 'over-design' factors being removed and systems being designed with reduced weight and yet the same or even increased safety.

The following sections give examples of the extended capability provided by dynamic network analysis through the application of computational methods.

Refuel System. Refuel system analysis has been greatly extended by the analysis of pressure surge due to tank inlet valve closure. Results of this type of analysis are given in Figure 3.

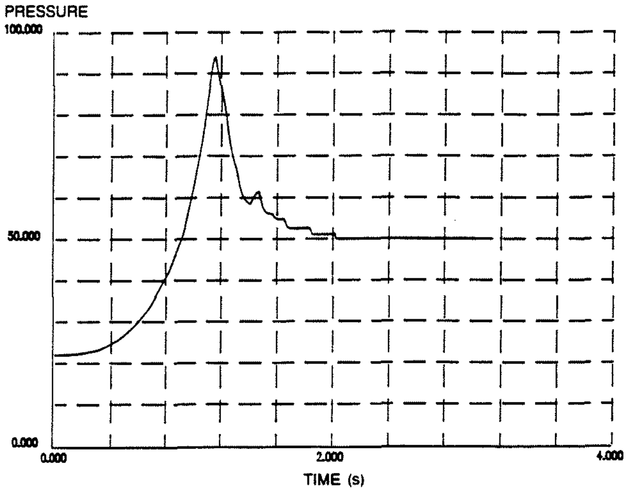

Figure 3 Typical Refuel Pressure Surge

However it is difficult to make reliable predictions of surge pressures using any method in other than straight pipes. Bends, junctions and other components produce reflection waves which add to the fundamental wave. The interaction of these reflection waves with the fundamental wave is not fully understood and therefore not precisely modelled. Nevertheless comparison of aircraft test results with simulation results indicates that simulated pressure surges, though not giving an exact replication of those measured on the aircraft, do give a very good indication of the surge pressures that may be encountered.

Without computational methods, determination of pressure surge has relied solely on aircraft test results. With computational methods, refuel pressure surge can be analysed early in the design, and any changes, if required, introduced before the aircraft is tested.

Fuel Transfer. Fuel transfer systems are dynamic systems which require models of the fuel tanks with accurate height, volume descriptions for a complete analysis. These have been developed as part of this work programme(1). This allows not only the transfer rates to be analysed, but enables the volume of fuel within the tanks to be monitored throughout the transfer.

Without computational methods, analyses are restricted to steady state at the critical condition of a transfer with the flow rates assumed constant throughout. The fuel tank models enable the effects on transfer rates of depletion of fuel levels in the tanks due to transfer and engine feed to be considered. This is particularly important for cases of gravity transfer and for transfers with degraded pump performance, for example when pumping with hot JETB fuel at altitude. In these cases the head of fuel within the tanks has a significant effects on the transfer rates.

The application of computational methods is illustrated with an example of an analysis of transfer between centre and inner tanks (shown in Figure 4) with the tank inlet valve to the inner tank failed open. The transfer is intended to be from centre to inner tank and is achieved by pump transfer. The case being considered is for JETB at a temperature of 54°C and an altitude of 41,000ft. Although this combination of fuel type, temperature and altitude are highly remote (JETB is usually only used in arctic climates, and the atmospheric temperature at 41,000ft is -50°C) it is a possibility and so represents an extreme design condition. With JETB at this altitude and temperature the performance of the transfer pump is degraded to the point where transfer initially takes place from the inner tank to centre tank through the failed valve, because of the greater head of fuel in the inner tank. Simulation of this case with the dynamic analysis method determined that as the fuel volume, and hence head, in the inner tank was reduced through transfer and engine feed, the transfer pump was able to overcome the head of the inner tank and commence transfer from the centre to inner tank. This is shown in Figure 5 a plot of the fuel volumes in the inner tanks and centre tank against time.

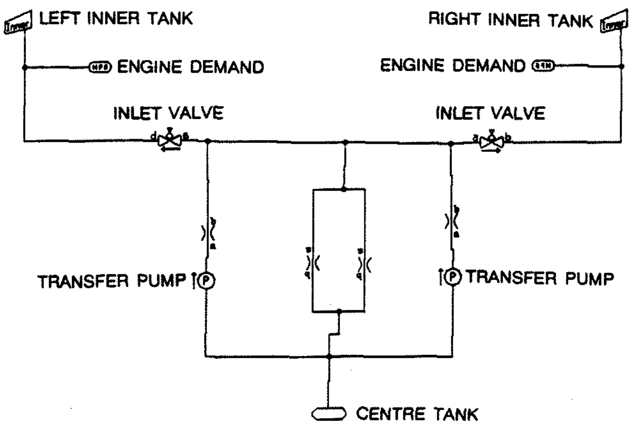

Figure 4 Transfer Analysis Network.

A natural extension of fuel transfer analysis has been to extend it to flight profile modelling. This models the operation of the transfer system for a total flight and takes into account the burn rate of the engines and all transfers. Implementation of this type of analysis has required limited aspects of the fuel system control computer to be incorporated by controlling the position of tank inlet valves as a function of the volume within the tank. This has been implemented through an extension to the standard fuel tank model which sets the position of the tank inlet valve dependent on the volume of fuel within the tank.

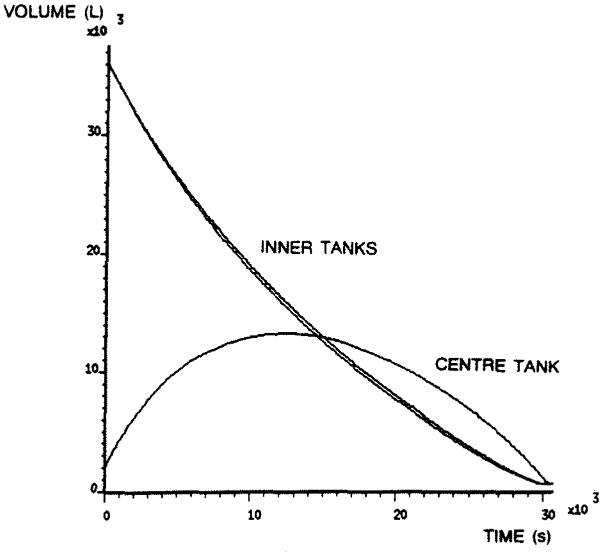

Figure 5 Plot of Tank Volumes.

Vent System. Vent system analysis has been significantly extended with the analysis of the operation of the vent system during emergency descent.

Emergency descent is defined as a descent from 40,000ft to sea level in 110 seconds. This is based on the scenario of a cabin depressurisation at this altitude.

The effect of this descent on the fuel system, and in particular the vent system, is that air passes into the fuel tanks through the vent system. With the high rate of descent, the air reaching the tanks will be at a lower pressure than the external atmosphere, causing a pressure differential across the tank wall. This analysis is conducted to ensure the pressure differential does not exceed structural limits and cause a tank to implode.

In simulations of this type the fuel tanks are modelled with accumulators with the tanks height, volume characteristics incorporated. This allows the compression of the air within the tank to be modelled. The accumulators are valid as models of fuel tanks until the accumulator fills whereupon the compression of the air is no longer modelled correctly. Typical results of an emergency descent analysis are given in Figure 6.

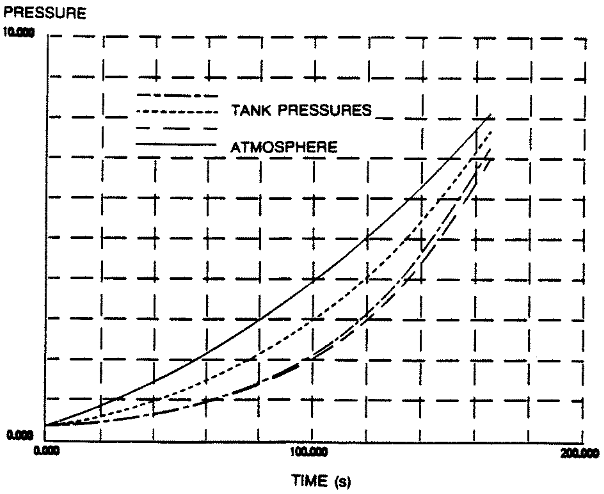

Figure 6 Typical Emergency Descent Pressures

Prior to this type of analysis the diameter of the vent system would be increased by an appropriate allowance to ensure the differential pressure across the tank wall did not exceed structural limits. This has generally resulted in vent systems being oversized.

Benefits of Network Analysis

The introduction of computational network analysis to the simulation of aircraft fuel systems outlined above, enables a competent engineer who is not necessarily an expert user of any of the software, to carry out all the design analyses in a reduced time scale. In the past the complex nature of the systems has required a specialist to carry out the majority of the analyses.

The application of computational network analysis to the design and analysis of aircraft fuel systems has greatly extended the range and complexity of analyses available, and enables many more analyses to be conducted in a reduced time scale and results in a greater confidence in the design and reduced costs.

CFD Component Modelling

With the increased number and complexity of network analyses available through the application of the computational methods, there is a requirement to provide the operational characteristics of a greater number of components to be used in the network analyses. Traditionally test rigs have been the only means of obtaining this data. In practice this required the data being collected to be critical or be in a large enough quantity to justify the cost of testing. CFD provides data similar to that obtained from the test rigs, but with no restriction on the number of measuring points. The cost is also significantly less. CFD introduces a new dimension to component modelling with the fluid flow through the component being modelled rather than the actions of the component on the fluid flow. This enables the pressure, flow characteristics of a component to be determined based solely on its inlet and outlet conditions, and geometry. Although the CFD models themselves are not appropriate for direct inclusion, results from their analyses describing the operational characteristics of components can be included in component models for network analysis.

CFD has been applied in two ways during this work programme:

- Determination of loss coefficients for components.
- Verification of proposed component designs through the modelling of flow paths within the components.

Examples of both of these applications are given in the case study which concludes this paper.

Junctions have required their loss coefficients and flow paths to be modelled more than other components. This is attributed to the lack of data on standard junction loss coefficients. Additionally, the junctions within aircraft fuel systems tend to be of a complex shape which leads to complex flows.

To address this lack of loss coefficient data a test programme has been undertaken to determine loss coefficients for combining symmetrical 'Y' junctions down to included angles of 30°. In addition to the testing of the junctions, CFD models of all the junctions tested were developed with the intention of validating the application of CFD to the modelling of junctions, in particular combining symmetrical 'Y' junctions. Typical results are given in Figure 7 which shows the loss coefficients for the 60° 'Y' junction for both the test rig and CFD analysis.

From the results of the test programme and CFD analysis it was concluded that CFD can adequately model combining symmetrical 'Y' junctions(1). Care has to exercised for junctions with large included angles (in excess of 120°), and to a lesser extent for flow ratios above 0.6 if a $k-\varepsilon$ turbulence model is used. It was found that for these conditions the turbulence intensity due to the combining inlet flows was too great for the $k-\varepsilon$ turbulence model. To accurately model turbulence of this intensity requires a turbulence model specially developed for the modelling of the high turbulence found in combining flows.

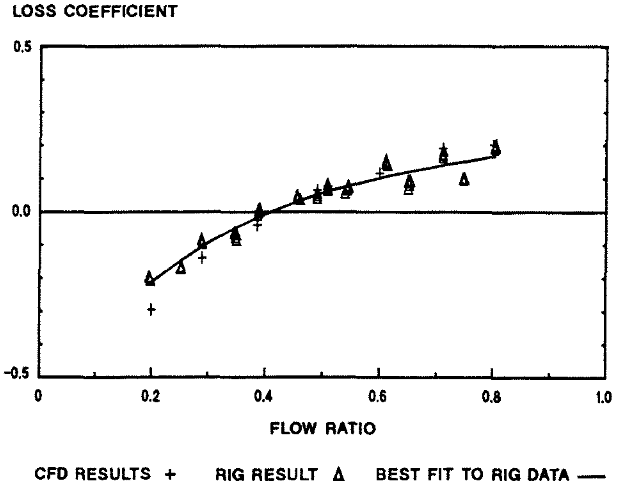

Figure 7 CFD and Test Rig Results For 60° 'Y' Junction.

Benefits of CFD Component Modelling

The introduction of CFD to the analysis and assessment of a components performance makes a large range of data for development of component models now available. The range and number of applications of CFD to the modelling of component characteristics is almost unlimited, and as such ensures that almost any new component can be modelled and then incorporated in a network analysis model.

Case Study:
Refuel in a Partitioned Tank

The case study considers the sizing of refuel restrictors in a tank partitioned into two cells, as shown in Figure 8. Normally the main requirement when sizing refuel restrictors is to achieve a known flow rate. However in this case the main requirement was to maintain equal levels in the two cells. Furthermore the flow rate through each restrictor was unknown due to the arrangement of the system.

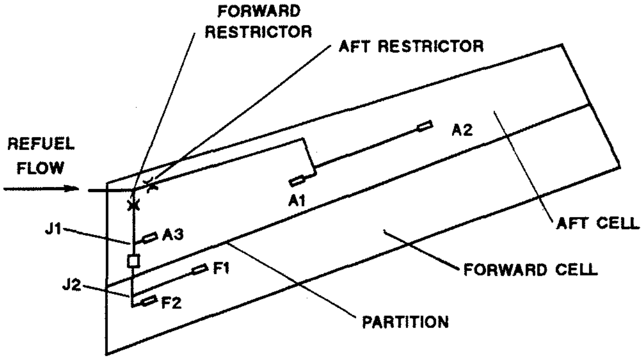

Figure 8 Case Study Refuel System.

During initial simulations and aircraft tests the refuel system was unable to maintain equal fuel levels in the two cells during refuel. The cause of this was investigated and a modification proposed and incorporated. This case study highlights the role of the computational methods in influencing the design of the refuel system, including:

- How CFD and network analysis were used to determine why the refuel system was unable to achieve equal fuel levels.
- The use of CFD to confirm the proposed modification.
- The use of network analysis to successfully size the refuel restrictors with the modification incorporated.

Throughout this design study the difference in height of the surface of fuel in the forward and aft cells is referred to as the 'fuel step across the partition', and is illustrated in Figure 9.

Refuel System

The tank involved in this study is split into two cells, forward and aft, linked by the refuel line, and is refuelled via two restrictors as shown in Figure 8. The aft restrictor feeds the two outboard aft cell diffusers *A1* and *A2*, and the forward restrictor feeds the two forward cell diffusers *F1* and *F2* and the inboard aft cell diffuser *A3*. During normal operation the refuel line linking the aft and forward cells maintains equal levels within the two cells. However it was anticipated that during refuel the refuel line would be unable to maintain equal fuel levels in the two cells, and therefore the refuel restrictors were sized to achieve this.

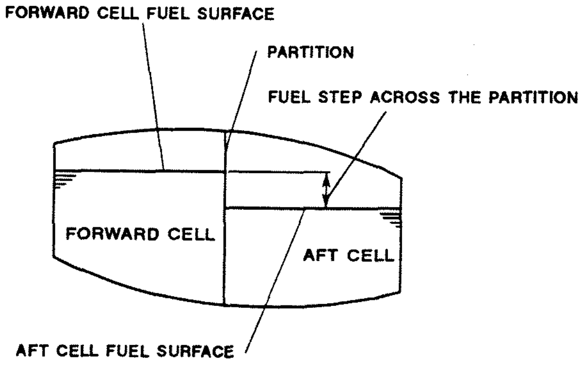

Figure 9 'Fuel Step Across The Partition'

The refuel system was simulated with a dynamic network analysis (Figure 10). This simulation, in addition to providing the pressures and flow rates within the refuel system, also has accurate models of the forward and aft fuel cells volume height descriptions, enabling the fuel levels and volumes in each to be monitored throughout refuel. The losses of all the components within the simulation were modelled with standard loss coefficients.

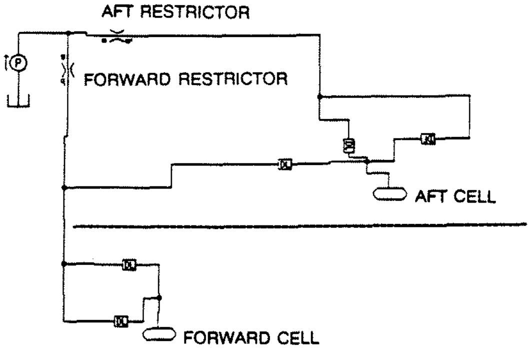

Figure 10 Simulation Network.

Initially this simulation was used to model the behaviour of the test aircraft with a 35mm forward and 22mm aft restrictor. To provide more data further tests and simulations were conducted with the forward restrictor removed.

<u>Initial Results</u>

For both cases tested the simulation was found to predict a fuel step across the partition 26cm forward high of those measured on the aircraft. Additionally for the case with the forward restrictor removed an aft high fuel step across the partition was measured on the aircraft. This indicated that a component downstream of junction *J1* (Figure 8) had a loss considerably higher than those attributed to standard components. Junction *J2* (Figure 11) was found to be the most probable cause, its complex shape suggesting the possibility of a high loss.

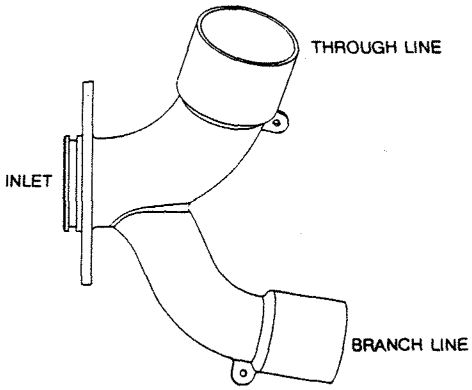

Figure 11 Junction J2.

To assess the loss characteristics of junction *J2* a two-dimensional CFD model was developed. This is shown in Figure 12. A two-dimensional model was developed in preference to a three-dimensional model because at this stage the time scale did not permit the development of a three-dimensional model. The two-dimensional model represented the junction to a sufficient accuracy(1) because although the junction was not truly co-planar it did not lay far out of plane.

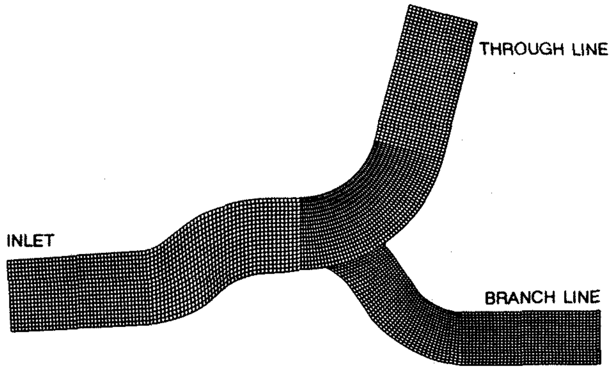

Figure 12 Two-Dimensional CFD Model.

Velocity vector plots from the CFD analyses are given in Figure 13. These show a large recirculation zone in the branch line which supplies diffuser *F2* and effectively reduces its diameter by half.

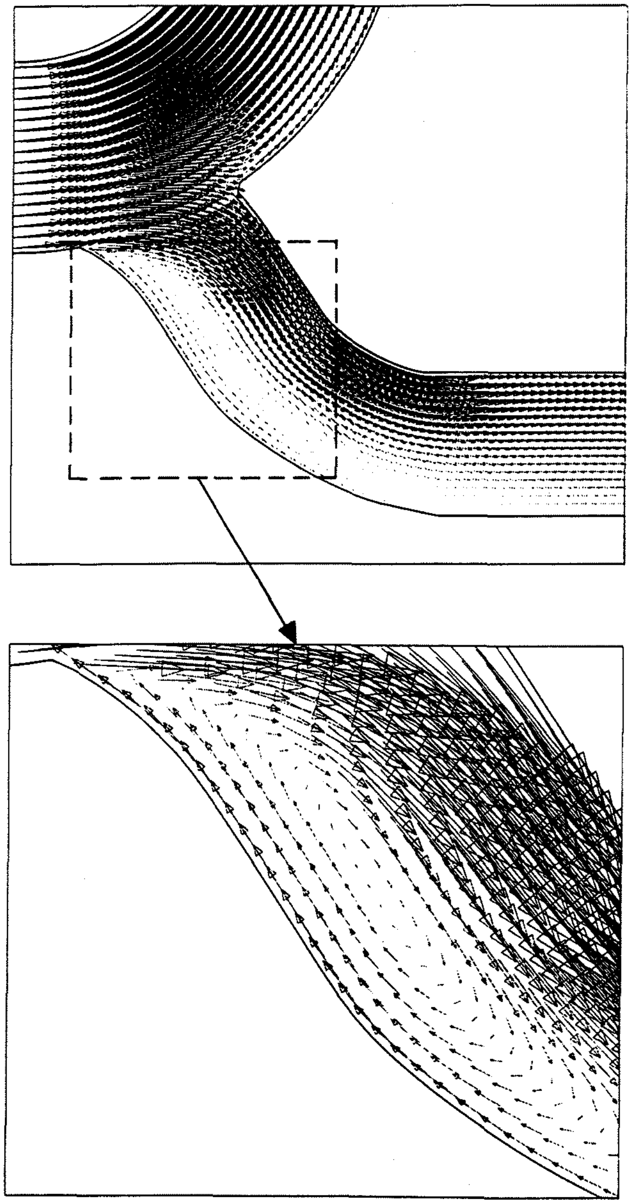

Figure 13 Recirculation Zone in Junction Branch.

The effects of this recirculation were modelled in the network simulation with the inclusion of a restrictor representing the reduction in diameter. With the previously tested restrictors included the simulation was found to be in close agreement with the aircraft results. It was therefore concluded that the aircraft was unable to produce a level surface across the partition due to the restriction in flow to the forward cell caused by the recirculation zone in junction *J2*. This also accounted for the difference in simulation and test results. Based on these analyses junction *J2* was re-designed to reduce the size of the recirculation zone and hence its loss.

If the computational methods, and particularly CFD, had not been available there would have been no means of assessing the actual impact of junction *J2* on the operation of the refuel system. The decision to re-design this junction, if made, would have been based solely on assumptions and engineering judgement.

Junction Design and Modelling

The modified design of the junction is shown in Figure 14.

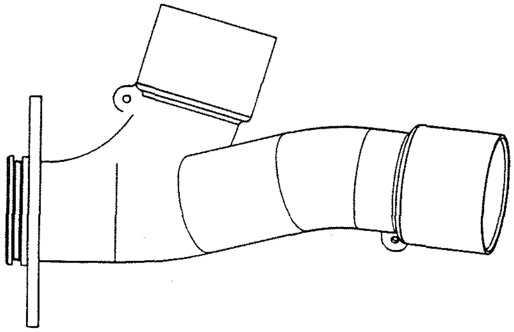

Figure 14 Modified Junction.

Based on the experience of the initial tests a CFD model of the new junction was developed to assess its performance before the junction was manufactured and installed on the aircraft. In this case time was available to develop a three-dimensional CFD model (Figure 15). The advantage of three-dimensional CFD models over two-dimensional CFD models is that they will supply a more accurate prediction of the performance of the junction, particularly for three-dimensional effects such as recirculation out of the two-dimensional plane. The major problem with three-dimensional CFD models is the development of the geometry itself. This is particularly true for intersecting circular cross sections.

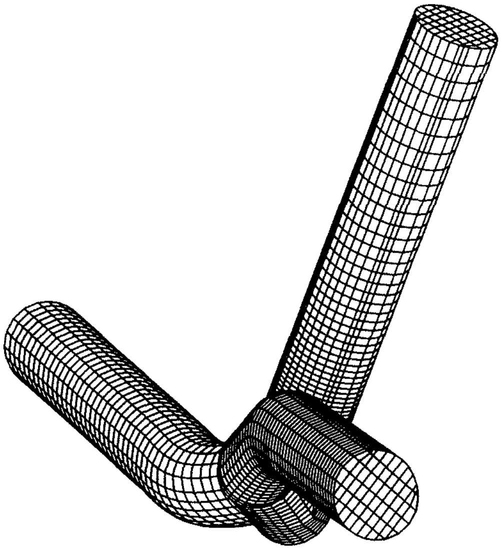

Figure 15 Three-Dimensional CFD Model.

The CFD model of the junction was run for various flow ratios to determine the location of any recirculation zones. Only small recirculation zones typical of those found within junctions of this type(2), were located.

The CFD model was also used to determine the loss coefficient of the junction at the various flow ratios tested. These loss coefficients were used in the simulation of the system to provide an accurate model of the junction.

Incorporation of Refuel System Modifications

With the new junction included further simulations and aircraft tests were carried out to determine the appropriate restrictor sizes. From these it was found a near zero fuel step across the partition could be achieved throughout refuel.

A set of typical results showing the fuel step across the partition, both predicted by the simulation and measured on the aircraft, plotted against time are given in Figure 16. These results show the good agreement obtained between the simulation and aircraft tests.

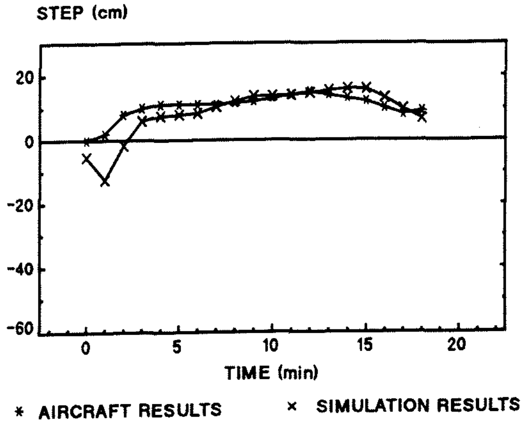

Figure 16 Typical Simulation and Aircraft Results.

Conclusions of Case Study

The application of the computational methods to this case study have enabled the modification to the refuel system to be incorporated based on analytical results rather than assumptions.

The computational methods provided the only means of analysis for this particular case. If the methods were not available traditional methods would not have been able to carry out the analyses in the time available.

- The sizing of the refuel restrictors would have been carried out through a series of estimations of restrictor sizes and confirmation through aircraft tests. This is an inappropriate method of sizing the restrictors due to the cost of aircraft testing.
- The determination of the flow paths through the junctions could not have been carried out with traditional methods. Any modifications would have been based on assumptions. The CFD results gave confidence to undertake the re-design and manufacture.

Closure

Application of computational methods to the design and analysis of aircraft fuel systems greatly increases the number, range and complexity of simulations. This is most evident with the inclusion of dynamic analysis and CFD component modelling. With traditional methods analyses of these types are rarely undertaken due to the excessive time required.

The computational methods outlined in this paper, give the aircraft fuel system design engineer a set of tools which enable detailed design and analysis prior to testing and flight. The application of these tools and associated methods achieves a greater understanding of the system and a reduced time from concept to first flight together with a safer design and reduced costs.

References

1. Plastow, I., "The Development and Application of Computational Methods for the Design of Aircraft Fuel Systems", PhD Thesis, University of Bristol, Submitted October 1993.
2. Miller, D. S., Internal Flow Systems, 1st Edition, BHRA Fluid Engineering, 1978.
3. Richards, C. W., Fung, K. C. J., "The Bath*fp* User Guide", The Fluid Power Centre, University of Bath, 1993.
4. Flowmaster User Manual, Flowmaster International Ltd., UM-3.4, 1993.
5. Plastow, I., "Simulation of Fuel Systems, Evaluation of Software Packages", University of Bristol, 1988.

Acknowledgements

The authors wish to acknowledge Mr M.A. Laceby formerly of British Aerospace and Dr R. Stirling formerly of the University of Bristol for initiating this work, colleagues at British Aerospace who have provided assistance and advice in the course of this work and Mr C.J. Slack for his aid in the preparation of this paper.

This work has been carried out with the support of the Department of Trade and Industry.

Disclaimer

The views expressed in this paper are those of the authors, and do not necessarily represent those of British Aerospace Airbus Ltd. or the University of Bristol.