Energy Simulation of Climatic Wind Tunnel Plant

E.C.Roberts
Department of Civil and Building Engineering
Loughborough University

A Doctoral Thesis submitted in partial fulfilment of the requirements for the award of Doctor of Philosophy of Loughborough University

May 2000

Abstract

The Climatic Wind Tunnel (CWT) is a facility used by the motor industry to test vehicles under climatic extremes without the need for expensive overseas test programs.

This work focuses on the application of computer simulation to the Heating Ventilation and Air Conditioning (HVAC) plant that makes up a CWT facility. The objective being to reduce its operational costs through the identification of energy saving operational strategies.

When in operation the CWT has a peak power consumption of 3MW. The implementation of any measures that would reduce this peak load would give rise to considerable savings in the operating costs of the facility.

Computer simulation is an accepted technique for the study of systems operating under varying load conditions. Simulation allows rapid analysis of different strategies for operating plant and the effectiveness of achieving the desired effect without compromising the buildings performance.

Models for the components of the CWT have been developed and coded in Neutral Model Format. These models have then been linked together in a modular simulation environment to give a model of the complete plant. The CWT plant naturally decomposes into four major subsystems these being the test chamber, the soakroom, air make-up and refrigeration system

Models of all the primary and secondary HVAC plant are described as is how they constitute the systems that make up the CWT. Validation tests for individual components as well as for the systems have been carried out.

To illustrate the potential of the application of computer simulation into finding improved modes of operation that would reduce the energy consumption of the facility, four studies have been carried out. The studies involve the possibility of scheduling the operation of condenser fans as a function of refrigeration load and outside ambient temperature, methods for the pre-test conditioning of a vehicle, a reduction in the secondary refrigerant flow temperature and an increase in the thickness of the insulated panels from which the facility is constructed. The studies carried out showed that there was potential for moderate energy savings to be made in the operation of the facility and that extended simulation runs would allow for the in-depth assessment of a large range of possible modes of plant operation in order to identify the areas where the greatest savings are possible.

Keywords: Modelling, simulation, energy, Neutral Model Format, Climatic Wind Tunnel, chiller modelling, vehicle modelling.

Acknowledgements

"Evermore thanks, the exchequer of the poor", wrote Shakespeare. And yet that is all I can provide to the countless kindred spirits who readily strove to accommodate my pleas for help. But I am rich through knowing them, my poverty cast aside with a wealth of kindness. So many are such good friends, yet others just passing acquaintances, drifting out of reach by that surreptitious traitor, time.

There is argument that suggests I refrain from specifying individuals, for surely this list would end up looking like a telephone directory, yet at the risk of omitting someone I shall endeavour to name the most prominent.

Great thanks go to my supervisor Professor Vic Hanby, without whose seemingly infinite patience and guidance this work would be no more than blank paper.

To MIRA, whose financial support gave me the opportunity to be harassed on a monthly basis by Prof. Geoff Callow, David Fletcher and Neil Jones.

To Craig Wheatley, Adrian Horrell, Jayne Arnold, Jason Williams, Alistair May, Stella Artois and chicken balti I give a massive shout of thanks.

Special thanks to Wendy Lowery who always seemed to know the right thing to say.

The greatest debt I owe is to my family – Syd, Marion and Andrew – whom have supported and put up with me over the last few years and it is to them that I would like to dedicate this thesis.

The completion of this work leaves me with three questions that I suppose we must all ask ourselves at some time: Where am I going to? Why am I going there and most importantly will the pubs be open when I arrive?

TABLE OF CONTENTS

Chapter 1

Introduction

The expectations of today's motorist are higher than ever. Not only do vehicles need to be safe and efficient but also they need to be comfortable and rewarding to drive. In order for manufacturers to keep pace with reduced product time cycles [1], ever tightening legislation concerning emissions and safety and yet still deliver a vehicle that the motorist will regard as value for money, they must have an ongoing program of research and development. The Climatic Wind Tunnel (CWT) is a key facility in this program. It allows vehicles to be tested under the influence of a range of extreme climatic conditions, therefore reducing the need to carry out expensive and time consuming overseas testing.

The CWT is a significant user of energy. The systems needed to work the tunnel have a total capacity of 3MW, which leads to a annual running costs in the region of £100,000, a cost of approximately £500 per 16 hour shift. With the scarcity of earth's natural resources increasing the cost of their extraction and supply increases and inturn the cost of operating machinery that depends upon their usage grows. With the reluctance of the motorist to pay ever-increasing car prices it is up to the manufacturers to keep costs low. With cost reduction in mind the manufacturers will not allow for price increases from their suppliers. In order for these suppliers to remain competitive and still maintain the profit margin they require in order to survive, they must reduce costs through increased efficiency be that through the workforce or the machinery on which they rely. If improvements in the operating efficiency of the CWT plant could be found large savings in energy costs could be made.

This thesis describes an investigation into the modelling and simulation of the thermal systems that make up a Climatic Wind Tunnel and the subsequent application of the model to identify improved operational strategies that would yield savings in energy.

1.1 Modular simulation environments

The only technique available that allows the analysis of systems under varying load conditions is computer simulation. It has been shown in previous studies [2, 3, 4, 5, 6], that simulation has been used to model the performance of Heating, Ventilation and Air Conditioning (HVAC) systems and reduce their operating costs through improved control strategies and plant configurations.

Many building simulation environments have been developed all of which contain some degree of HVAC system simulation which is essential if they are to accurately estimate the buildings overall energy usage. As these building analysis tools give the user a list of pre-defined HVAC systems that may be included in the building under consideration they are inflexible when the need arises for analysing different or innovative designs.

To overcome this limitation a number of design and research tools have been developed that employ a modular approach to system simulation and are commonly referred to as Modular Simulation Environments.

The Modular Simulation Environment allows the system to be resolved into its constituent parts, each of which is represented by an individual model. The system is then assembled in much the same way as an engineer would draw a schematic representation of the system. The advantages of this approach in terms of increased flexibility are obvious, as different plant configurations can be assessed.

Other advantages of this approach include:

- The solution method is separated from the model and is effectively in the hands of experts in numerical techniques.
- Allows the modeller to concentrate on the modelling issues and not expend time on the solution method.
- The model code is more transparent, more portable and reusable by others.
- If problems with the solution of the equation set exist an alternative solver may be available.
- Allows for extension into the areas of part load performance analysis, plant sizing and optimisation.

A number of Modular Simulation Environments have emerged a list of which has been compiled by Sahlin [7]. Most of these tools are aimed for use by quite sophisticated users who have good grounding in mathematical modelling. Yet if these tools are to successfully cross over into mainstream design use, more user-friendly interfaces and comprehensive model libraries need to be developed.

1.2 The Climatic Wind Tunnel

The Climatic Wind Tunnel provides the motor industry with the opportunity of conducting research, environmental tests and product development of vehicles and components, under a whole range of climatic conditions. Typical tests that are carried out are:

- Engine cooling performance
- Air conditioning system development
- Hot fuel-handling tests
- Analysis of city driving
- Heater system development
- Cold start and drive away test
- Demist / defrost test

The vehicle testing is carried out in the facility's two temperature controlled areas, the soakroom and the test chamber. The soakroom is used to pre-condition a vehicle before it is moved into the test chamber where its performance at the test conditions can be analysed.

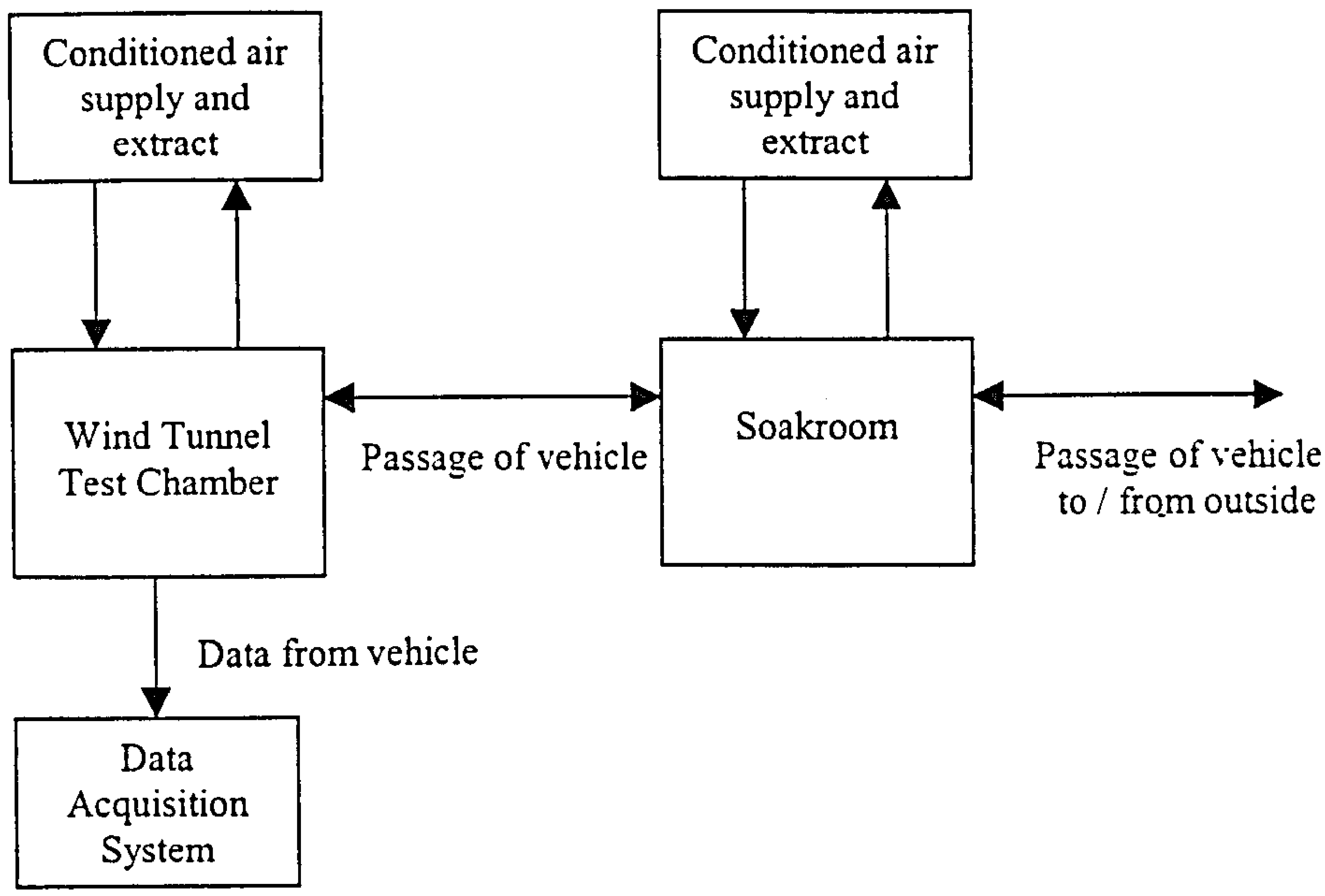


Figure 1.1 Vehicle test process diagram

Figure 1.1 illustrates how the vehicle moves between the temperature controlled zones of the CWT. Each area shown has its own independently controlled conditioned air supply and extract system and test data is only recorded from the vehicle in the wind tunnel test chamber.

1.3 Thesis objectives

The objectives of this research are to:

- Model the components that make up a Climatic Wind Tunnel facility.
- Using an existing Modular Simulation Environment assemble the models in a configuration that represents the Climatic Wind Tunnel thermal systems.

- Validate the model performance against empirical data recorded from the existing plant.
- Use the models of the CWT plant to carry out simulations configurations of different operating strategies in order to identify areas where it is possible to reduce energy consumption without compromising the performance of the facility.

1.3.1 Research methodology

The approach to the research is to characterise and document the existing CWT plant. Using the information gained of what the major CWT components are, compile of a library of plant models used in the CWT thermal systems. Review existing Modular Simulation Environments to enable selection of an appropriate platform in which to model the CWT. Search for existing available HVAC plant models that are applicable to re-use in the modelling work. The component models are to be linked together in an appropriate Modular Simulation Environment to form a representation of the CWT systems. Simulations using the plant models are to be carried out over a range of operating conditions and validated against empirical data recorded from the CWT whilst it is operating at the same conditions as are being simulated. The final analysis is to implement the models in varying plant operating strategies in order to gain insight into areas where energy can be saved without effecting the operational performance of the facility

1.3.2 Thesis structure

The major components of this thesis are:

- A review of published literature on Heating, Ventilation and Air conditioning (HVAC) simulation. HVAC component modelling, chiller and compressor modelling, the thermodynamic analysis and modelling of motor vehicles and the effect of frost formation upon heat exchangers with extended surfaces. A Review of existing Modular Simulation Environments and the reasons leading to the selection of one that lends itself to the problem of simulating the CWT thermal systems (chapter 2).
- Development of models that are particular to the Climatic Wind Tunnel project (chapter 3).
- A description of the systems that make up the Climatic Wind Tunnel and their modes of operation. The description of the development of a control strategy for the effective control of the heater and modulating valve(chapter 4).
- Validation of the simulation against empirical data recorded from the existing plant (chapter 5).
- Conduct an investigation into different plant operating strategies. Compare the effect of these alternative operational strategies to the strategies that are currently employed. Identify from comparison where a change in the mode of operation would lead to cost savings through reduced energy consumption (chapter 6).
- Draw conclusions from the research and suggest areas for continuation of the work (chapter 7).

1.4 References

[1] Fletcher D.W (1993). The design and construction of MIRA's Climatic Wind Tunnel. Society of Automotive engineers (SAE) Paper 931957. High Temperature Engineering and Testing.

[2] Braun J.E (1988). Methodologies for the design and control of central cooling plants. PhD Thesis. University of Wisconsin, Madison, USA.

[3] Bourdouxhe J-P (1997). Use of the simulation tools in the design and operating phases of the air conditioning equipment. Laboratory of Thermodynamics, University of Liège, Belgium.

[4] Hackner R.J, Mitchell J.W, Beckman W.A (1984). HVAC system dynamics and energy use in buildings – part 1. ASHRAE Transactions Vol. 90 pt.2b pg. 523 – 535.

[5] Hackner R.J, Mitchell J.W, Beckman W.A (1985). HVAC system dynamics and energy use in buildings – part 2. ASHRAE Transactions Vol. 91 pt.1b pg. 781 – 795.

[6] Hackner R.J, Mitchell J.W, Beckman W.A (1985). System dynamics and energy use. ASHRAE Journal, June 1985. Pg. 29 – 37.

[7] Sahlin P (1996). NMF Handbook: an introduction to the neutral model format. HTTP//:www.brisdata.se/

Chapter 2

Literature Review

The computer-based simulation of buildings and the systems that service them has long been established within the research community. Whilst the value of simulation in the research field has long been proven it is only recently that modelling and simulation techniques have begun to be made use of in industry. Computer modelling and simulation are the only tools that allow engineers the opportunity to rapidly analyze different system designs operating under non-design conditions. A review of previous work in the field of HVAC simulation has been carried out.

A simulation tool is of no use unless it has a library of models to allow simulation to take place. A modelling language that allows models to be expressed as equations and used in a number of different simulation tools is reviewed and its salient points highlighted.

The Climatic Wind Tunnel is used exclusively for the testing of motor vehicles. For simulation work to have realism a vehicle model needs to be included. Previous work into the development and modelling of vehicles is reviewed. The review pays particular interest to the heat transfer from the engine to its surroundings. This is because it is the engine that is the largest heat source / sink used within the Climatic Wind Tunnel and an effective model of the heat transfer of an engine needs to be developed.

At the heart of the Climatic Wind Tunnel is the refrigeration system. An investigation into previous chiller modelling work has been carried out.

A feature of heat exchangers operating at low temperatures is the build up of frost and ice upon their surfaces. As the heat exchangers in the Climatic Wind Tunnel can be operated at very low temperatures a review into the formation of frost and its modelling is presented.

The simulation work is to be carried out using an existing Modular Simulation Environment. A review of a number of existing environments has been conducted and comparison against a list of required criteria that the final selected platform will ideally posses has been made.

2.1 HVAC System Modelling and Simulation

With the evolution of computer programs since the 1960's the design and analysis of building performance has been revolutionised. Yet their widespread application in the design of building and Heating Ventilation and Air Conditioning (HVAC) design is still to be realised.

Building HVAC systems generally consist of primary plant i.e. boilers and refrigeration plant and secondary plant, which includes air handling units and pumps. An ASHRAE Task Group [1] defined system simulation applicable to "Energy Requirements for Heating and Cooling of Buildings" as:

> *"...predicting the operating quantities within a system (pressures, temperatures, energy and fluid flow rates) at the condition where all energy and material balances, all equations of state of working substances and all performance characteristics of individual components are satisfied."*

The same Task group also reports:

> *"It is essential that the dynamic characteristics of the building be considered in the calculation of the thermal loads, but the dynamic response of most systems is much more rapid than that of the building. For this reason a steady-state simulation is adequate for most energy calculations."*

The above acknowledges that plant components react much faster than the building fabric; typical response times are in the order of seconds and minutes as opposed to hours. The effect of the HVAC models on the total system performance is limited and therefore steady-state simulation is adequate for energy calculations. Yet if the performance of a system that is subject to rapid changes in load or the analysis of a control system is required then dynamic plant modelling is essential.

The original building energy analysis programs only allowed the user to model HVAC system configurations chosen from a set menu, under steady-state conditions. DOE-2 and BLAST [2, 3] are examples of this approach. This approach is inflexible and does not allow the user to analyse innovative one off designs more suited to the building under consideration.

One of the first dynamic modelling and simulation environments to emerge was TRNSYS in the mid 1970's [4]. TRNSYS broke away from the constraints imposed on HVAC modelling by the building energy analysis programs by allowing the user to build thermal systems component by component, in much the same way as a schematic diagram of a system is constructed. This type of simulation is often referred to as *modular,* which means that components and sub-systems are modelled as objects that can be interconnected to specify the model of the entire system. TRNSYS was originally developed for the simulation of solar energy systems but has since been successfully used on other studies in the HVAC field. Hackener et al [5, 6, 7] used it for the ASHRAE research project "HVAC System Dynamics and energy use in existing buildings". The aims of the project were to investigate the potential of energy saving HVAC operating strategies and the availability of "reliable" equipment models. The project modelled a number of water chillers and the air-handling units that they served. The results from the simulation showed that in the building in question a revised operational strategy would yield energy savings in the region of 8%. The project highlighted that the use of dynamic simulation could further reduce energy consumption by allowing a full exploration of control strategies available. Braun [8] used TRNSYS to model and simulate a central chilled water cooling plant and develop subsequent methodologies for its optimal control. Bourdouxhe [9] also developed large central chilled water cooling plant models that were used to analyse the energy consumption for a real chiller plant.

Silverman et al [10], acknowledged that in order for better systems to be developed a program that allowed the system designer to have the ability to define the system's components and the way in which they are interconnected was needed. From this a computer program similar to TRNSYS was developed that allowed the user to select a "node" representing an HVAC component and link it to another "node" (component). The resulting system could then be solved to give the resulting state-variables and allowed partial energy optimization.

Clark et al [11], in an effort to understand the dynamic interactions between a building shell, an HVAC system and control system developed a building system simulation program called HVACSIM+. HVACSIM+, stands for HVAC SIMulation PLUS other systems. The program employs the same methodology of a modular approach as used by TRNSYS, from which many of the ideas on which HVACSIM+ is based. It allowed the user to simulate the dynamic performance of the whole building / HVAC / control systems with control dynamics being modelled second – by – second. The HVAC system and building zone dynamics are calculated minute - by - minute and the heating / cooling loads being calculated on a 15 minute to 1 hour basis. This method of simulation avoids the error prone process of trying to simulate the entire building at a time step dictated by the fastest dynamic response in the system, usually the control system. As HVACSIM+ employed a better equation solver that TRNSYS it proved to be far better at simulating control systems.

A simulation environment is of no use unless it has access to well-developed component models. Component models may be regarded as mathematical statements describing the region under consideration [12]. The form in which the component model equations are written can be classified as *algorithmic*, i.e. equations which will give a solution representing the component behaviour for a given set of inputs. In which case the model produces coefficients which are then passed onto external differential equation schemes; a simulation environment that uses this approach is ESP [86].

Component models can be broken down into categories of description:

Fundamental models: These are models that are adequately described by established theoretical principles; numerical data requirements are limited to such quantities as thermo physical constants, which are usually reliable. Fundamental relationships feature strongly in internal component relationships such as discrete nodal schemes, but when applied globally to components they do not give a complete description. For example, mass and enthalpy balances describe the main functions of a mixing tee but some semi-empirical treatment is necessary if heat loss and pressure drops are to be modelled.

Semi-empirical models: These are widely available, very useful and generally incorporate reliable experimental data. Component behaviour is modelled as far as is possible from first principles, but empiricism is resorted to where theoretical treatments are unavailable or would be inappropriate. An example of where this approach may by used, is in the modelling of an air-water heat exchanger where empirical correlation's are used to determine the heat transfer coefficients and air and water pressure drops.

Empirical models: These are often referred to as "*black box*" models. This means that they are able to predict the response of a component to changes in operating conditions but lack any internal description of the component. Empirical models are used in many system simulations to model complex mechanical plant such as chillers, generally by curve fitting of manufacturers data. The level of output from this type of model may be considered adequate for most simulation purposes, but particular care is required in their formulation particularly when they are formulated through curve fitting. The accuracy of data on which the model is based is often in doubt, as the data on which they are based may not have been obtained under realistic operating conditions. An example of this is performance curves for refrigeration compressors. These are often obtained under standard sub-cooling and superheat conditions that are unlikely to be encountered in real life operation and hence the accuracy of the model is compromised.

Black box models can also be derived from direct measurements from an actual installation, this is often done in fault detection and diagnosis (FDD) work.

Algorithmic models: This type of model can be further subdivided:

Steady state: Steady state models are widely used due to their simplicity and have been proven adequate for a large number of applications. Many system simulations are based upon steady state models even when the output is used in the dynamic context, for example, where the input is based on a sequence of hourly weather periods. The justification for this technique is that if control process dynamics are not of interest then the response of the plant is usually much more rapid than the changes in the forcing functions (weather) so that a quasi-steady state analysis is sufficient.

Dynamic models: Originally this type of model was only used for studies into control system dynamics rather than predictions of component performance. Dynamic models address the transient behaviour of a component. Several typical situations that call for dynamic analyses are the investigations of:

i. Effects of disturbances to the system.

ii. System start-up transients.

iii. Control system stability.

An example of an algorithmic model is one for TRNSYS. The model shown in Figure 2.1 is for a steady-state evaporator (type 210).

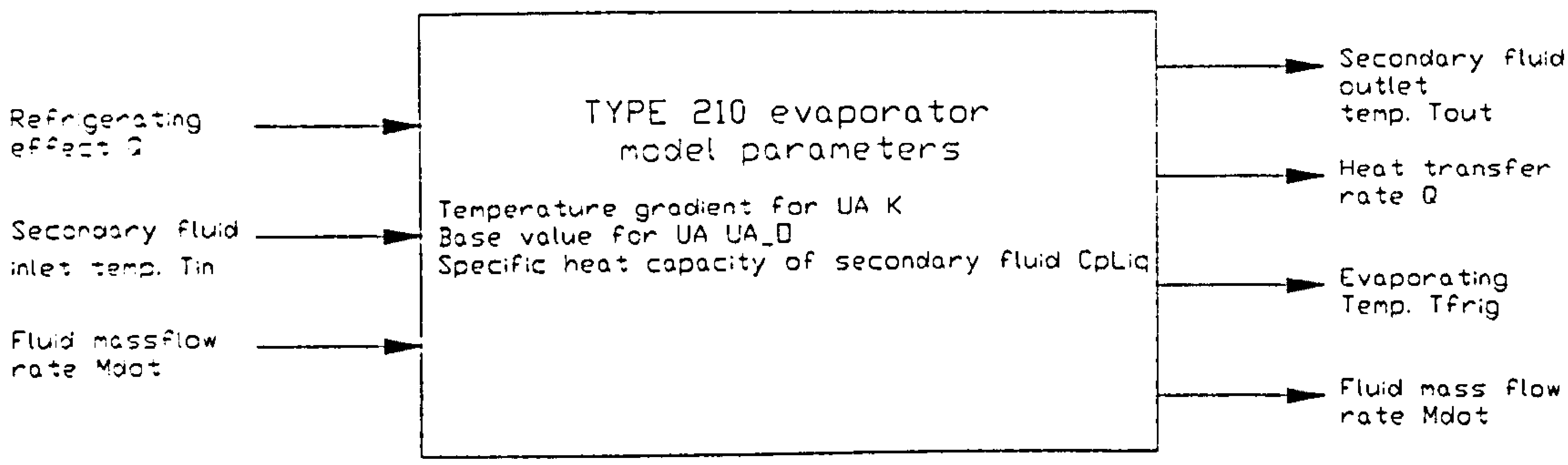


Figure 2.1 Information flow diagram for TRNSYS type 210

A fully annotated model description can be found in Appendix C.

The model is a specific algorithmic formulation to calculate evaporating temperature (Tfrig) from the refrigerating effect (Q). Some of the variables are passed through the model for interconnection with other models or for including in the output

Numerical models: This type of component models are quite different from the more familiar algorithmic type in that they cannot yield a solution representing component performance in their own right. They are better described as generators of coefficients that are passed onto a remote formalised solution process.

Clark et al [14] outline the development of dynamic plant models for use in the HVACSIM+ simulation tool. Dynamic models for pipes, ducts and heating / cooling coils are outlined and their output was shown to give good agreement with experimental data taken from actual plant.

Wright [15] suggests that if simulation software is to be of real use to the engineer, then component models must reproduce the performance of the components as measured by the manufacturer and give the designer the "look and feel" that they are used to. As each manufacturer produces performance data in their own format this influences model development, as the data produced must be convertiblē into a format for use in models. A steady-state model of a fan and its development using manufacturers data is then described.

Hanby [16] describes a technique for using component models to produce performance maps of the components under differing load conditions. This data is typically unavailable from manufacturers but would be of great benefit to the design process.

The design of HVAC systems relies upon the analysis of the performance requirements of the system at agreed design parameters, usually peak loads [17]. As the design process is still largely a manual task it would be impractical to use manual analysis to consider differing solutions to the problem and a "standard" system solution based upon the load criteria is employed. With advances in the simulation of the dynamic thermal performance of buildings and of HVAC plant, there is now a opportunity to alter the basis of HVAC plant sizing from the load calculation to an assessment of the ability of the plant to meet an installed performance specification [18].

2.2 Chiller modelling

Many studies have been carried out into the modelling and simulation of vapour compression refrigeration systems. The vast majority of these studies have concentrated upon the use of reciprocating compressors [19, 20, 21, 22].

The models vary greatly in complexity depending on whether the developers required the model to be steady state or dynamic. The steady-state models allow the investigator to see the state variables within the system for given evaporating and condensing temperatures. Whereas the dynamic model allows the investigator to follow the changes occurring as the system works to achieve a set final condition.

Following its origin at the Swedish Royal Institute in 1934 when A.J.R. Lysholm built the first prototype of the rotary screw compressor [23], it took until the late 1950's until the screw compressor was in widespread use within the refrigeration industry. Between 1946 and 1956 a great number of designs for screw compressors were produced but it was with the advent of rotor cutting machines in 1956 that gave the essential repeatable accuracy required in rotor production that a new program of development began [24].

Trulsson [25] developed an improved rotor profile that gave lower leakage between the rotors and was able to operate in an oil-free environment, which was a great leap forward in respect of efficiency, reliability, noise and maintenance. The main advantage to be gained from producing a compressor that ran in an oil-free environment is the simplification of the cooling arrangements and the elimination of the oil reclamation equipment, therefore reducing the size of space required for the compressor installation [26]. The capacity control system of a screw compressor and how the built-in volume ratio varies under part-load conditions was described by

Lundberg [27]. The operation of a twin-screw compressor under full and part-load conditions was described by Pillis [28]. He showed from testing, that with varying condensing temperatures a variable volume compressor offers significant savings in energy when compared to a fixed volume machine operating at the same conditions.

A model to investigate the performance of rotary screw compressors was developed by Firnhaber and Szarkowicz [29]. It gave a prediction of the compressors performance and evaluated the effects on efficiency of clearances and geometrical changes over a range of operating conditions.

Bráblik [30] developed an analytical model of an oil-free screw compressor. The model was built with the aim of observing the process of compression in conjunction with the dynamic processes in the discharge piping. The model was used to simulate an existing compressor design with the sole purpose of seeking the optimum design of the compressor and its operating conditions.

Similarly Sångfors [31] also produced an analytical model of a helical screw compressor for performance prediction. The simulation was developed to reduce the costs of experimental work in the design and development of new compressor models. It allowed better analysis and greater understanding of the compressors probable performance before a prototype was constructed.

Singh and Patel [32] produced a computer program that predicted the performance of oil-flooded twin-screw compressors. The program takes into account all leakage, viscous shear losses, oil cooling and inlet and discharge losses. Some empirical coefficients were used but through testing were shown to have good applicability. The program follows one compressor cavity from the inlet (0°) through to discharge (360°). The pressure in the cavity is computed from mass and energy conservation equations. The flow through the inlet and discharge ports is calculated as well as the various internal leakage's.

Jonsson [33] looked not only at the simulation of a twin-screw compressor but also at how the addition of an economiser (intercooler) enhances the performance of the refrigeration system. The addition of an economiser in a system is shown to improve its capacity and Coefficient of Performance (CoP). It is shown that a system utilising an economiser arrangement that the pressure ratio increases so does the increase in CoP. The effect of the inclusion of a second economiser in a system is also investigated. It is shown that not only would there be problems in the manufacture of such a device but there are numerical difficulties in its simulation that have to be overcome. The results from the simulation of the two-stage economiser show that the improvement in CoP is not as great as might be expected.

Bourdouxhe [9] detailed the development of a twin-screw chiller model for use in the ASHRAE Toolkit for Primary Energy Calculation. The chiller is represented by four components:

i. Twin-screw compressor
ii. Condenser
iii. Evaporator
iv. Expansion valve

The expansion of the refrigerant is assumed to be perfect, no heat exchange between the system and its environment is taken into account and no oil circulation is considered. The compressor was validated against data provided by manufacturers to which it gave very satisfactory agreement. The whole chiller model was validated against manufacturers' data and gave results of a maximum deviation of 2.2% from the real data.

Yik and Lam [34], produced numerical models of chiller plant that were developed from a second order curve fit to relevant manufacturers' data. The same method was used by Shelton and Weber [35] in their studies into the design flow rates for chiller plant. The models developed by Yik and Lam were formed from the relationship between power consumption (kW) to the cooling capacity (kW). The model development is based upon two assumptions these being:

i. That the flow rate through the chiller remains constant at all times.
ii. The chilled water temperature is kept at its respective design values.

These assumptions are normally the case in chiller plant, although the chilled water temperature is often reset to a higher level during times where part load conditions prevail, in order to save energy. The models developed were verified by comparison with data logged from a number of buildings in which that type of chiller modelled was operating. The results comparison showed that the model tended to overestimate the energy consumption of the chiller under consideration. The models were subsequently used to optimise the sequencing of a buildings chiller control.

2.2.1 Conclusion

A number of twin-screw compressor models have been developed all with differing degrees of detail. For the CWT project it has been decided to adopt the approach used by Yik and Lam as it has been proven to be accurate, easy to implement and is appropriate to the type of data that is available for the project.

2.3 Thermodynamic analysis and modelling of vehicles

The dawn of the motor vehicle occurred around 1769 when French military engineer, Nicholas Joseph Cugnot, built a steam-driven vehicle for the sole purpose of pulling artillery pieces [36]. Over the following decades engineers such as James Watt and Richard Trevithick improved upon the design. Trevithick developed a steam coach that operated on a route from Cornwall to London. The age of the steam coach was abruptly ended in 1865 by competition from the railways and strict new anti speed laws were passed.

It was during the 1860's that the internal combustion engine first became a practical reality [37]. Early developments of engines of this type used a mixture of coal gas and air at atmospheric pressure – there was no compression before combustion. Frenchman Etienne Lenoir was the first to develop a marketable engine of this type. The charge was drawn into the cylinder on the first half of the stroke, ignited with a spark, the pressure increased and the burned gases delivered the power to the piston for the second half of the cycle, Lenoir built about 5000 of these engines. He subsequently built a horseless carriage for use on the road, but eventually lost interest in the venture and nothing further came of it.

A method of carrying out the operations of a four-stroke internal combustion engine were described in an 1862 patent taken out by French civil servant, Alphonse Beau de Rochas. He lacked the means to develop the patent and offered it to Lenoir, who failing to realize its importance and potential turned it down.

A far more successful development was that made by Nicolaus Otto and Eugen Langen in 1867. They used the pressure rise resulting from combustion of the air-fuel charge in the outward stroke to accelerate a free piston and rack assembly so its momentum would generate a vacuum in the cylinder. Atmospheric pressure then pushed the piston inward, with the rack engaged through a clutch to the output shaft. The thermal efficiency of this engine was up to 11%, a big improvement on Lenoir's engine that had an efficiency of at best 5%. In order to overcome the engine's low thermal efficiency and its weight Otto proposed an engine cycle with four piston strokes:

- Intake
- Compression
- Power
- Exhaust

The prototype for his four-stroke engine first ran in 1876. The engine gave a great reduction in weight and volume and an increase in efficiency; this was the breakthrough that founded the internal combustion engine.

At this time Lenoir, realising his mistake, began to manufacture engines working on the same principle. Otto attempted to sue over infringement of his patent rights, but Lenoir had no difficulty in proving his engines were made under the now lapsed patent of Beau de Rochas. Even though Rochas' writing predates Otto's developments by some fifteen years, he never brought these ideas in to practice. Thus Otto, in a much broader sense, was the inventor of the internal combustion engine that we know today; the cycle was for many years (and sometimes still is) referred to as the Otto cycle.

After a century of development it might be thought that the internal combustion engine (also referred to as spark ignition engine) has reached its peak and there was little potential for further improvement. This is not the case. Internal combustion engines have continued to show improvements in power output, efficiency and reduced emissions; the modelling and simulation is now a major contributing factor in engine development.

Engines operate in one of two states, *transient* or *steady state.* It is the performance of components under transient operation that is of greatest interest to engineers, as it is at this time that they are subject to the greatest changes in temperature and load. Transient operation can be further sub-divided into two different modes of operation, *short-term transients* and *long-term transients* [38].

Short-term transients: These are imposed by the operator or control system in response to speed or load changes. They occur continuously during operation and may range from 1 to 20 seconds and may be considered to be made up of four basic modes:

- Increase in engine speed
- Decrease in engine speed
- Increase in load (torque)
- Decrease in load (torque)

Generally, these will occur in some combination, that is, either one of the speed changes occur with either of the load changes or vice versa. A typical example of the combination is a car accelerating at full throttle with the speed increasing. The torque then varies according to the engine characteristics, the torque first increasing with speed and then decreasing slightly toward the next gear change.

Long-term transients: These occur when a major change in temperature takes place in the engine and are of particular concern during the warm-up period. It will obviously vary from engine to engine and on the prevailing ambient and driving conditions but a typical value is in the region of eight minutes. They are of particular interest in the study of volatile fuel components and cooling systems. The cooling system must be of sufficient capacity that it prevents overheating under hot conditions but allows the engine to come up to operating temperature as quickly as possible.

During combustion the temperature reached by the burning gases is in the region of 1500 to 2000°C.

The heat of combustion is transferred in all directions to the metal of the combustion chamber, cylinder walls and piston by direct radiation and by convection currents of the gas scrubbing against the practically stationary gas film that forms on metal surfaces [39]. A thin film of oil exists between the stagnant gas layer and the cylinder wall and a thin layer of carbon separates the stationary gas from the piston crown and the combustion chamber. Heat then flows through the metal walls with minimum resistance. It should be noted that a material with perfect heat conducting properties would have no temperature gradient i.e. the temperature on both sides of the wall would be equal. A film of corrosion products, scale and contamination from the coolant forms on the opposite cylinder wall. Next to this is a stationary contact film, which separates the bulk of the liquid coolant from the layer of scale surrounding the cylinder barrel. The liquid coolant is circulated around the engine and is passed through the radiator where the heat transferred to the fluid from the combustion process is expelled to the surrounding atmosphere.

The flow of coolant around the engine block is regulated by the use of a thermostat. The thermostat blocks the coolant circulation to and from the radiator when the engine is cold or warming up, so that the trapped coolant in the cylinder block and cylinder head passageways absorbs and accumulates the rejected heat of combustion. The trapping of the coolant in the engine block causes the engine to reach its normal working temperature more rapidly. When the coolant in the thermostat housing reaches a pre-set level the thermostat begins to open. It does not open fully until the thermostat housing has risen to its designed operating temperature.

The effect of different engine coolant mixtures on the heat transfer performance of an engine was investigated by Bhowmick, Branchi and McAssey [40]. An experimental program was conducted to investigate what the effect of ethylene glycol / water and propylene glycol / water had on the overall heat exchange rate.

The effect of coolant heating in conjunction with lubricating oil heating and fuel vaporisation on the overall engine warm-up was investigated by Andrews, Harris and Ounzain [41]. Their results showed that the lubricating oil was the slowest component in the warm-up and may be the limiting factor in engine warm-up. They indicate that for an engine to achieve an optimum working temperature that in the region of 15 minutes from start-up must have elapsed. When the engine is not fully warmed up they found an increase in fuel consumption, gas emissions, lubricant degradation and engine wear.

An analysis of cyclic variability in combustion energy release was carried out by Daw et al [42]. They produced a model that allowed the rapid simulation of thousands of engine cycles that permitted the analysis of the cyclic variations. The model was concerned with the differences occurring in each phase of the four-stroke engine cycle and how it affected power output.

Patton, Nitschke and Heywood [43], detail a model that predicts friction mean effective pressure for a spark-ignition engine. The model is based upon a combination of fundamental scaling laws and empirical results. The friction losses in an engine can be divided into three categories:

i. Rubbing friction, losses from bearings, piston rings and valvetrain.
ii. Pumping losses, losses resulting from flow through the intake and exhaust valves.
iii. Auxiliary component losses, includes losses from oil and water pumps and alternator.

The model showed that the use of scaling laws made the results applicable over a wide range of operating conditions. At low speeds the friction of the pistons accounted for 40 – 60% of total engine friction where as at high speed pumping losses accounted for up to 50% of total loss.

In order to gain a greater understanding of the thermal processes occurring during the warm-up period Kaplan and Heywood [44] developed an engine model whose major components were based upon lumped thermal capacitance methods. The model took into account the exhaust system, coolant and oil flows and their respective heat transfer rates as well as friction heat generation relations.

Heat transfer in the model is as follows:

The oil reservoir receives thermal energy from the underside of the piston; oil is pumped to the cylinder head, where it undergoes a heat transfer (positive or negative depending up on the temperature) and returns to the reservoir. Heat is also transferred to ambient air at the bottom of the sump.

The block and cylinder head are separated by a gasket, which is assumed to prevent any significant heat transfer between the two components. They are linked by the coolant flowing through the block and then up through the head. The coolant circulates in the cylinder head until it reaches a predetermined temperature. At this temperature the thermostat opens allowing the coolant to flow through the radiator.

For the cylinder block, along with heat transferred from the combustion gasses, thermal energy is added due to friction at the piston / liner interface.

For the cylinder head, additional sources of heat transfer come from the oil flow and the exhaust port, where heat transfer from the hot exhaust gas to the head occurs.

The exhaust model is able to predict the gas and metal temperatures along its length. Since the exhaust pipe is long, gas and metal temperatures change continuously along its length, so a simple lumped capacitance model is not sufficient. The exhaust pipe is divided into 10 sections each small enough to use the lumped capacitance method.

The results from the model developed were compared to data taken from similar engine and the model's prediction for component temperatures during warm-up were found to be accurate. From the predictions given by the model, it can be observed that the engine warm-up time is governed by the thermal capacitance of its components. The pistons' heat up most rapidly, the thermal time constant depending upon the speed and load. The temperatures of the head block and coolant all increase at a similar but slower rate during the warm-up period, whilst the temperature of the oil reservoir lags behind.

Jarrier and Gentile [45] also developed a simulation to examine the thermal transients during the warm-up period. They developed nodal networks to represent the heat transfer processes (conduction, convection and radiation) occurring within the engine. Numerical nodes corresponding to the heat exchange surfaces are used to represent the physical boundaries within the engine. The simulation allows the study of the heat flux throughout the entire engine.

2.3.1 Conclusion

The engine models developed by Kaplan and Heywood, Jarrier and Gentile have been shown to give excellent predictions of the various heat fluxes occurring within an engine. Whilst either of these models would be applicable for use within the CWT simulation work the level of detail each of these models provide in terms of the heat flows between each of the individual components is far too great and neither model is focussed on the external heat transfer that is of interest in this project. For the CWT simulation work it is only necessary to consider the engine as a single mass that acts as a heat source or sink, what is occurring within the engine is of no interest. For this reason neither of the two previously discussed models are to be used. Secondly obtaining the detailed information required for the parameters used in the models would prove very difficult.

2.4 Frost formation on finned heat exchangers

When a surface is below the dewpoint of the air surrounding it condensation will form upon that surface. If the surface is cold enough the condense will form a layer of frost. In low temperature cooling applications heat exchangers are often operating far below the dewpoint of the air passing over them and are prone to frost build up on their surfaces.

Stoecker [46], Hosoda and Uzuhasi [47], were the first to highlight and carry out investigations into the effects of frost build. Both parties found that the frost layer caused deterioration in thermal conduction. As a result of this a decline in the overall heat transfer coefficient occurred and the cooling capacity of the heat exchanger deteriorates. It was found that the growth of frost caused a closing of the air passages which gives an increased air flow resistance. As a result, it is not possible to maintain the required air flow rate and the cooling capacity will further decline. The spacing of the fins is of great importance for as the frost builds, a close-finned coil will be subject to a greater reduction in airflow and therefore its capacity will be rapidly reduced.

Gatchilov and Ivanova [48] found that when frost begins to form it does so randomly on the tubes of the heat exchanger. When the frost layer begins to grow it then spreads along the fins of the heat exchanger. They observed that after 20 minutes the frost layer was of uniform thickness.

A number of computer programs have been developed in an attempt to simulate frosting process [49, 50, 51, 52, 53, 54]. The level of sophistication of each model differs greatly. The model developed by Sami and Duong [49] takes into account water migrating into the frost layer, altering its density as well as its thickness this inturn alters the heat transfer coefficient of the coil. Senshu et al [51, 52], developed a heat pump model that had a frost formation simulator integrated into it. The frost formation simulator was configured so that it was only operable when operating below 0°C. the frost formation simulator in the model relies upon two major simplifying assumptions:

- Frost is deposited evenly on the fin surface
- The frost surface layer is as smooth as the fin surface.

The acknowledged problem with trying to simulate the formation of frost, is that due to the nature in which it forms and the uncertainty of its structural nature, which inturn affects its thermal properties, an accurate simulation of its formation and build up is very difficult to achieve.

2.4.1 Conclusion

From the previous work carried into the modelling of frost formation on finned heat exchangers, it has been shown that due to the unpredictable nature of frost formation and the different way it forms at different temperatures it is very difficult to model.

As frost build up is only a feature of the low temperature heat exchanger within the air make-up plant and the complexities in its modelling it has been decided that there is little to be gained from incorporating it in a heat exchanger model.

2.5 Modular Simulation

2.5.1 Component modelling: Equation based models. The Neutral Model Format

Neutral Model Format (NMF) is an equation-based language for expressing models for use in existing and emerging modular simulation environments [55, 56, 57].

NMF was first proposed in 1989 by Sahlin and Sowell [55] as a format for the expression of component models that could be interfaced with a variety of simulation environments.

Sahlin [56] expressed the basic motivation for the development of NMF as:

> *"Without a comprehensive, validated library of ready made component models in a relevant application area most simulation environments*

are rather useless. To develop all necessary models from scratch is, in many projects, quite unrealistic. And since the cost of developing a substantial library easily exceeds the development cost of the simulation tool itself, it is important to be able to reuse what other people have already done."

NMF has two main objectives:

i. Models can be automatically translated into the local representation of several simulation environments i.e. the format is neutral with regard to the target solver.
ii. Models should be easy to understand and express for non-experts

The first objective allows the development of common libraries. A global NMF component library, SIMONE (SImulation MOdel NEtwork) exists in which developers may deposit models they have written. SIMONE can be found at: HTTP//:www.brisdata.se/nmf/simone.htm.

An NMF Translator parses NMF models into the appropriate model environment language. Translators have been developed for a number of simulation environments including IDA, TRNSYS and HVACSIM+ [56].

2.5.2 NMF – Basic Constructs

The following simple example is taken from the NMF Handbook [56] and illustrates some of the more important aspects of NMF. It is a model of a very simple thermal conductance Figure 2.2 with a linear relationship between heatflux, Q, and temperature difference T1 – T2. The NMF model description is shown in Figure 2.3.

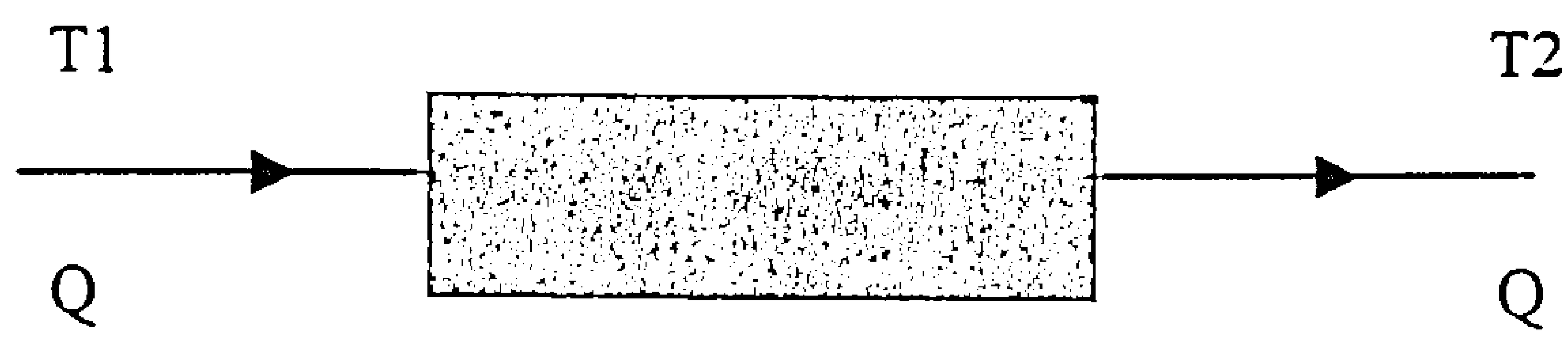


Figure 2.2 Simple thermal conductance

```
CONTINUOUS_MODEL                    tq_conductance

ABSTRACT          "Linear thermal conductance"

EQUATIONS

/* heat balance*/

0 = a_u * (T1 -T2) - Q:

LINKS

/*        type            name                   variables*/
          TQ              terminal_1 T1, POS_IN Q;
          TQ              terminal_2 T2, POS_OUT Q;

VARIABLES

/*        type            name      role      description*/
          Temp            T1        IN        "1st temp"
          Temp            T2        IN        "2nd temp"
          HeatFlux        Q         OUT       "flow from 1 to 2"

PARAMETERS

/*        type            name      role      description*/
          Area            a         S_P       "cross section area"
          HeatCondA       u         S_P       "heat transfer coeff"
          HeatCond a_u    a_u       C_P       "a*u"

PARAMETER_PROCESSING

a_u := a * u:

END_MODEL
```

Figure 2.3 NMF model of simple thermal conductance

2.5.3 NMF comments and reserved words

Comments used within an NMF model are enclosed within /* and */. There is no limit to the number of lines a comment may consist of and they may occur anywhere within the model.

Variable names, parameters and links are known as identifiers and must not exceed 31 characters in length. The identifier must start with a letter but may contain digits, underscores and dollar signs.

NMF is not case sensitive, but as a matter of convention and to aid readability, reserved words and variables are written in uppercase, whilst parameters are in lower case.

2.5.4 Global Declarations

Global declarations define units and variables. This definition assists in the exchange of models between simulation environments. The global declarations are held in a file within the translator called *global.nmf.* A developer my carry out changes to this list as is required.

Global declarations include quantity types, link types and global constants.

2.5.5 Continuous model

A continuous model operates with continuity though time i.e. time is a continuous variable. This is currently the only type of model available in NMF.

2.5.6 Abstract

The abstract section should contain a brief description of what the model is and who developed it.

2.5.7 Equations

The equation section is for expressing the relationship between model variables and parameters. The equations are expressed in their residual form i.e. $f(x_1, x_2, x_3....) = 0$. This means that all variables in the equation have equal status and they are merely stated in a relationship that is valid at all times. The number of equations within the model and their order are of no consequence to the generated algorithm.

User defined functions and subroutines may be called in the equation section from outside the model. The user-defined functions can be of one of four categories:

i. Functions in the NMF notation
ii. Functions in Fortran
iii. Functions in C
iv. Functions that are available in the target system as binaries i.e. no source code.

2.5.8 Links

In the links section of the model, the lines of communication are detailed. The thermal conductance model given has two links *terminal_1* and *terminal_2*. It is important to note that only variables appearing in the links section can interact with other models.

Not all target environments make use of links but they must always be specified within an NMF model. In environments supporting links models may be joined collectively or at variable level.

2.5.9 Variable and parameter declarations

Variables and parameters must be clearly detailed in an NMF model. Parameters are quantities that remain constant throughout every simulation. Each variable or parameter must be declared in four respects:

Type: As for links, variables and parameters are globally typed. As an alternative they may be declared GENERIC, which means they are compatible with any other type.

Name: The name given to the quantity.

Role: Parameters can be either supplied values, S_P, or they can be computed C_P. The user specifies the supplied parameters whilst the computed parameter is calculated in the parameter processing section of the model.

Variables may have one of four different roles:

i. IN
ii. OUT
iii. LOC
iv. A_S

The user must specify either a given (IN) or calculated (OUT) variable. The number of OUT variables must exactly match the number of equations given within the model. Locally assigned (LOC) variables receive their value by assignment. Assigned state variables (A_S) are variables that retain their value from one time step to the next.

Description: A description of each variable or parameter must be given and must not exceed 80 characters in length.

Parameter processing: The parameter processing section is where computed parameters (C_P) are calculated from supplied parameters (S_P). The code is executed only once at the start of the simulation. Standard or user-defined functions may be referred to, as in the equation section.

Much development work on NMF has now ceased and efforts are now concentrated on a "next generation modelling language" Modelica [58, 59, 60].

2.5.10 Modelica

Modelica is an effort to produce a standardised language for the modelling of such applications as electrical circuits, drive trains, thermodynamical systems and chemical processes [58]. It possesses the same aim as NMF in that it will assist in the interchange and reuse of models but is aimed at the entire modelling and simulation field rather than certain aspects of it.

2.5.11 Modelica fundamentals

In order to give an introduction to Modelica, consider the simple electrical circuit shown in Figure 2.4 [59].

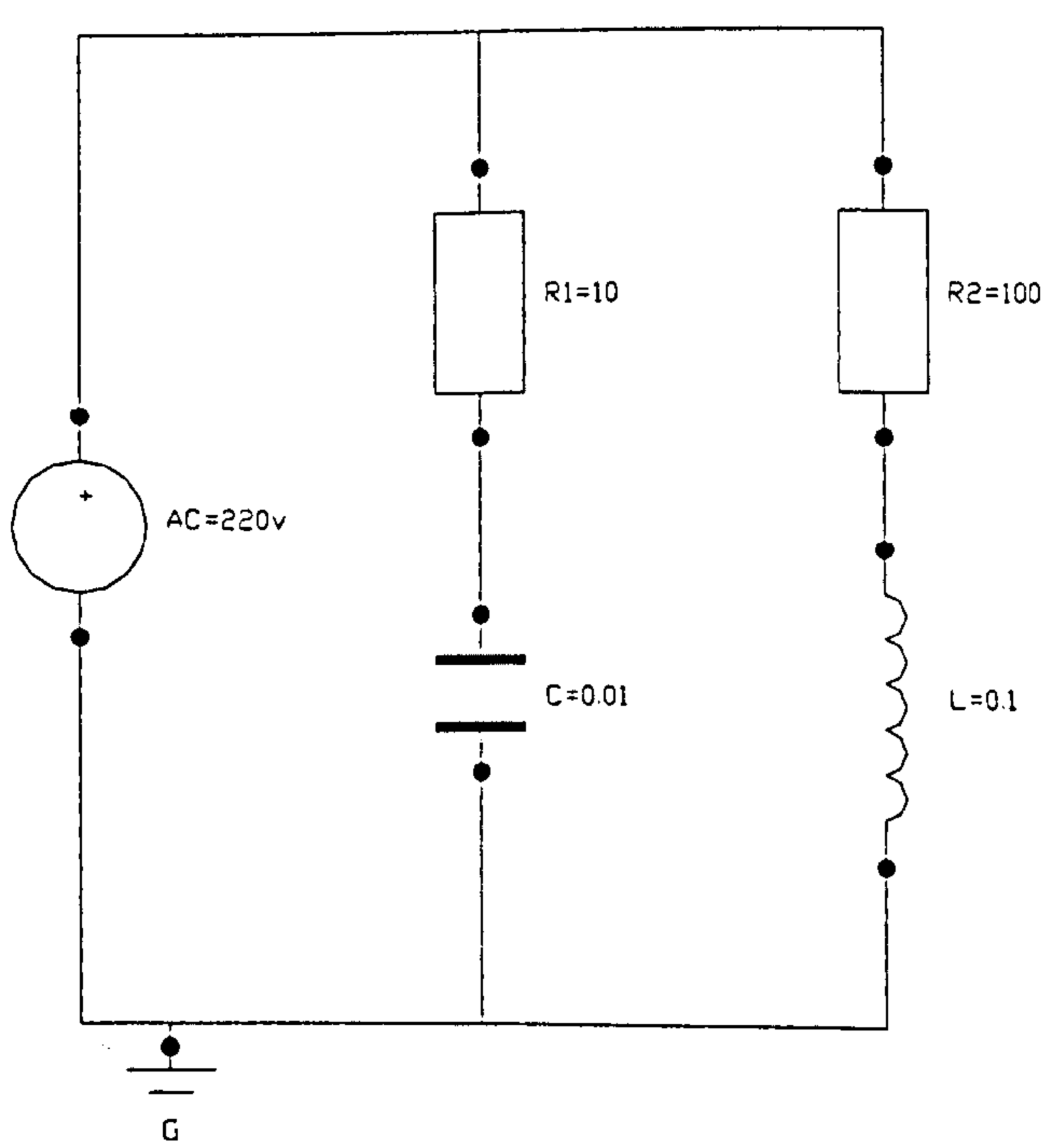


Figure 2.4 simple electrical circuit

```
Model Circuit
    Resistor   R1 (R=10),  R2 (R=100);
    Capacitor  C  (C=0.01);
    Inductor   L  (L=0.1);
    VsourceAC    AC;
    Ground         G;

Equation
    Connect (AC.p,  R1.p);  //capacitor circuit
    Connect (R1.p,  C.p);
    Connect (C.p,  AC.n);
    Connect (R1.p,  R2.p);
    Connect (R2.n,  L.p);
    Connect (L.p,  C.p);
    Connect (AC.n,  G.p);
End Circuit;
```

Figure 2.5 Modelica model of circuit in Figure 2.4

The circuit in Figure 2.4 can be broken up into a set of connected standard electrical components. There is a voltage source, two resistors, an inductor, a capacitor and a ground point. All these components are standard in all electrical model libraries.

A Modelica model of the circuit is given in Figure 2.5. The model specifies the circuit in terms of components and connections between them. The statement "Resistor R1 (R=10);" declares a component R1 of class Resistor and sets the default value of the resistance R to 10. The connections specify the interactions between the components. In other modelling languages connectors are referred to as ports or terminals. The language element **connect** is a special operator that generates equations taking into account what kind of quantities are involved.

A connector must contain all quantities needed to describe the interaction. For electrical components the quantities voltage and current are needed to define interaction via a wire. The types to represent them are declared as:

```
Type Voltage = Real (unit="V");
Type Current = Real (unit="A");
```

Where *Real* is the name of a predefinable variable type. A real variable has a set of seven restricted classes with specific names, such as **model, type** (a class which is an extension of built-in classes, such as **Real**), **connector** (a class which does not have equations and can be used in connections).

There are two possibilities in defining a class. The standard way is shown above in the definition of the electrical circuit. A connector class is defined as:

```
Connector Pin
   Voltage        V;
   Flow Current   i;
End  Pin;
```

The keyword **parameter** specifies that the quantity is constant during the simulation run, but the value can be changed between runs.

The above is constructed from an early version of the language (V.1) [58]. There is still much development work to be done before the modelling community begins to make any use of Modelica.

Modelica is being developed as a language of description that can be used in any simulation area and not slanted towards any particular one, unlike NMF that is directed more towards the building and HVAC simulation fields. It contains elements of successful existing languages that have been developed.

2.6 Simulation Environments

Stoecker [13] defines system simulation as the calculation of operating variables (such as pressures, temperatures and flow rates of energy and fluids) in a thermal system in a steady or transient state. A system is a collection of components whose performance parameters are interrelated.

Modelling and simulation are indispensable when dealing with complex problems. It allows analysis of physical systems before they are built and alleviates the need for expensive experiments. As well as the analysis of systems at their peak design loads, modelling and simulation allow assessment of systems operating at a variety of non-design conditions. Modelling is the development of a model that represents a real life component, whereas simulation is the process of using the model to analyse and predict the behaviour of the real component / system [61].

Modelling and simulation has been in use since the 1920's, but was the preserve of a handful of University groups [62]. It was not until the 1950's that advances in digital computers came about and their possible use for simulation explored. Selfridge [63], in his paper "Coding a general purpose digital computer to operate a differential analyser" showed the possibilities for computers in the simulation field.

Over the following decade intense activity in the simulation field saw many simulation programs become available. It was not until 1967 when the Continuous System Simulation Language (CSSL) [64] report was published that the concepts and languages of the simulation programs were unified.

A number of software products for different branches of engineering have been developed from the CCSL definition. It is some of these in particular the ones developed for HVAC simulation that are examined for their applicability to the project.

2.6.1 SIMULINK

Originally called SIMULAB [65], SIMULINK was integrated with the general purpose scientific program MATLAB in 1991.

SIMULINK is a software package for modelling, simulating and analysing dynamic systems [66]. SIMULINK provides a graphical user interface for building models as block diagrams, using "drag-and-drop" mouse operations. The interface allows the user to construct models in the same way as if producing a schematic diagram with pencil and paper

A model is defined by placing blocks selected from menus onto the work screen and then linking them together in order to give a meaningful representation of the system that is to be simulated. The simulation can be carried out either from within SIMULINK or from the MATLAB command window

Simulation consists of two distinct phases, initialisation and simulation.

Firstly, the block parameters are passed to MATLAB for evaluation. The resulting numerical values are used as the actual block parameters. The model hierarchy is flattened and each sub-system is replaced by the block it contains. Next, blocks are sorted into the order in which they need to be updated. The sorting algorithm constructs a list such that any block with direct feed through is not updated until the blocks driving its inputs are updated. Finally, the connections between the blocks are checked to ensure that the output of each block is the same as the input expected by the block it drives. When this is completed the simulation is ready to run. This process is known as problem reduction and is a feature of many programs.

A model is simulated using numerical integration. Each of the supplied integers depend upon the ability of the model to provide the derivatives of its continuous states. Calculating these derivatives is a two-step process. Firstly, each blocks output is calculated in the order determined by the sorting algorithm. Then in a second pass each blocks output is calculated based upon current time, it's inputs and its states. The resulting derivative vector is returned to the integrator, which uses it to calculate a new state. The resulting derivative vector is returned to the integrator, which uses it to calculate a new state. Once a new state is calculated, the sampled data blocks and scope blocks are updated.

Through the use of scope and other display blocks, the user is able to view the simulation results as it is running.

SIMULINK contains a comprehensive library of blocks that are used for constructing component models. Example block libraries are sources, which contain signal generators and sinks that contain output devices such as scopes. It is possible to create new blocks for inclusion within the component menus. The main drawback with SIMULINK is that it is a general-purpose simulation environment and is not particular to the HVAC field. Work is currently in progress [67], to produce a toolbox of HVAC components to be used in the design ands testing of control systems.

2.6.2 SPATS

SPATS – Simulation of the Performance of Air-conditioning and Thermal Systems – is a text-input steady state HVAC simulation tool developed at Loughborough University [68].

The user selects the required components to build up the desired network from the program's components library. A number identifies the links between the components; these links represent the flow of information from one component to the next.

Once the network has been defined to the users satisfaction, the simulation can be run. A number of boundary variables need to be specified by the user, these usually take the form of temperatures or massflow rates. The number of boundary variables that need to be specified within the network is equal to the total number of variables less the total number of equations; this ensures that the system is represented by 'n' equations in exactly 'n' unknowns, the program checks this to ensure that a well posed problem has been defined. The user prior to the beginning of each simulation may alter the boundary variables.

The equations are solved by expressing them in their residual form (i.e. $f(x) = 0$) and then using a reduced gradient optimised search (GRG2) [82]. GRG2 adjusts the values of all the unknowns so as to force the vector of residuals close to zero.

SPATS allows the user to exercise a limited degree of optimisation to the network; this finds the best operating point for one criterion i.e. energy consumption. An option exists for the user to carry out simulation of up to twenty-five time periods; this is called a load profile. The load profile allows the user to define differing sets of variables and see how the system reacts to them.

Problems that are apparent with SPATS are that if a wrong entry is made at the system definition stage, there is no way of editing this and the user has to start again which is time consuming especially with large networks. The same problem arises when the user is specifying boundary variables, there is no way of correcting a wrong input and the simulation has to be run with a wrong set of variables. A drawback with the output from SPATS is that it cannot be put directly into a spreadsheet or other analysis program

The major drawback is that the program is steady state, which is adequate when coupled to building energy simulation programs, which calculate the building energy loads hourly. The HVAC systems that make up the CWT have very short time constants. In order to effectively analyse the HVAC systems dynamic simulation needs to be used.

2.6.3 TRNSYS

TRNSYS – Transient System Simulation program – was developed at the Solar Energy Laboratory, University of Wisconsin during the 1970's [69]. It was one of the first modular simulation solvers and was primarily developed for the modelling of solar energy systems. Its modular approach makes it extremely flexible in the modelling of thermal systems of differing levels of complexity. TRNSYS comes with a library of pre-compiled component models, which allows the user to construct systems without the need for a compiler. Should the user require to develop their own models for specific applications, these maybe compiled and linked into the library.

TRNSYS: Incorporates a set of stand-alone utility programs, which allow the user to develop and test models, model systems and simulate those systems. These programs are:

TRNSHELL: A text based program that captures all the activities that are required to use TRNSYS. These activities incorporate file handling, editing plotting, the creation of TRNSED files and the running of TRNSYS.

TRNSED: This is a program that allows a user to run TRNSYS. It provides an interface for the user to view/alter information required for the system simulation.

DEBUG: A utility program that allows a user to test component models before incorporation into the component library.

PRESIM: Allows the user to create TRNSYS input files using a graphical interface. To form a system, the user connects icons representing the system components together graphically.

ONLINE: A component routine that plots variables to screen as the simulation progresses.

IISiBat: The latest tool to be added to TRNSYS was released with version 14.2 in 1997. IISiBat roughly translated from French stands for Intelligent Interface for the Simulation of Buildings. It is a general simulation environment program, adapted at the CSTB in France to house the TRNSYS simulation software [70]. The IISiBat package handles the necessary activities associated with TRNSYS, similar to the role of TRNSHELL. In addition to the normal TRNSYS functions, IISiBat allows the user to graphically create TRNSYS input files by connecting inputs and output of icons representing system components. The connections between the icons represent that represent the pipes, ducts or wires that connect the physical components the make up the system.

To plot the results of a simulation run the user uses IISiBat's spreadsheet tool, this automatically loads the output file generated by TRNSYS into the spreadsheet, where the results can be graphically viewed and manipulated.
In solving the ordinary differential equations associated with transient simulations, TRNSYS has the following solution methods available:

- Modified Euler Method
- Non-self-starting Heun method
- 4^{th} order Adams Method

The default solver is the Modified Euler Method, but the user may change the numerical method.

With the introduction of TRNSYS14.1 the user gained the ability to "back solve" problems [71]. This means that it does not matter whether input or outputs are defined as long as a valid set of simultaneous equations is defined, as the corresponding inputs can be calculated from the defined output. The solution method allows the use of variable time step, although TRNSYS employs a fixed time step throughout a simulation.

2.6.4 SPARK

SPARK [72] – Simulation Problem Analysis and Research Kernel – is a modular simulation environment based on a general differential/algebraic solver. This means it can be used to solve any kind of mathematical problem that is described in terms of differential and algebraic equations. As any physical system can be described in these terms SPARK can be used in many scientific and engineering fields, HVAC simulation being just one of these.

The initial prototype of SPARK [72] SPANK (Simulation Problem ANalysis Kernel), was developed at Lawrence Berkeley National Laboratory in 1986 [73]. The basic ideas were based upon previous work at the IBM Los Angeles Scientific Center [74] and later extended to allow the solution of differential equations [75]. In 1992 Nataf and Winklemann [76] developed the MACSYMA and Maple interfaces as well as carrying out many other improvements. During 1995/96 in preparation for its public release, SPARK was completely rewritten. In this rewrite a new class and problem description language was implemented to improve modelling flexibility, and the solver was redesigned to improve solution speed. As well as being rewritten, a graphical user interface was developed.

To describe a problem in SPARK solution begins by breaking it down into its component parts. A model must be then developed for each component not available in the SPARK library. As there may be several components of the same kind, SPARK object models are defined in a generic manner called classes.

SPARK object class models are described in an equation-based non-procedural text language.

To solve the differential/algebraic equations associated with dynamic simulation, SPARK has five different solution methods ranging from simple explicit formulae to complex explicit formulae used in predictor – corrector methods. The available methods being:

- Euler
- Backward – forward difference
- 4th order backward – difference object
- Adams – Bashforth – Moulton
- Milne 4th order

The user may specify different integration methods in differing parts of the same problem.
SPARK differs from other simulation engines in the way it solves problems. Rather than solving the problem directly, SPARK first builds a program that carries out the solution, an approach that maximises solution efficiency. To generate this program, graph – theory methods are used to decompose the problem into a series of smaller problems, called "components" that can be solved independently.

Recent work has been carried out [77] to demonstrate SPARK's problem reduction techniques and how this reduces execution time

SPARK comes with a HVAC toolkit based upon the ASHRAE Secondary Toolkit [85] for energy calculation and is supplemented with plant models from DOE-2 [2].

When SPARK is running there is output to the screen primarily to let the user know that the problem is being processed. The results from the simulation are written to an ASCII output file that can be opened by conventional spreadsheet programs.

Development work on SPARK is still in progress.

2.6.5 IDA

IDA is a modular simulation environment for the simulation of continuous systems. It was developed at the Swedish Institute of Applied Mathematics in co-operation with a group at The Department of Building Services Engineering KTH Stockholm [78]. Development began in 1987 original versions of IDA being called MODSIM [79].

IDA has very strong ties with Neutral Model Format (NMF), due to the involvement of key personnel being involved with both projects.

At the heart of IDA is a powerful differential / algebraic equation solver known as IDA Solver, (originally called MODSOL [79]). IDA Solver relies on pre-compiled models of the system components. The component models are held in Dynamic Link Libraries (DLL), which is generated automatically from the relevant component models. The interconnection of these components into working systems is carried out in a system description file.

In it's most basic form the IDA environment consists of an NMF translator, a Fortran compiler and IDA solver. IDA Solver is it's self-contained in a DLL, which if a hand coded system description file is being used, is called from a DOS prompt.

IDA Solver integrates the equations generated from a dynamic system using a variable time step, a method that is regarded as somewhat exotic in building simulation [80]. The use of a variable time step gives greater accuracy and consistency.

IDA allows the user to specify a range of alternative methods for the calculation of initial values. Several of these methods can be tried in sequence, the available methods being:

- Newton-Raphson iteration
- Newton homotopy with circular constant arc length
- Newton homotopy with tangent constant arc length
- Line seek with Newton direction
- Damped Newton iteration
- Steepest descent
- Hybrid Powell method (mix Newton and gradient)

IDA Modeller [81] is the graphical user interface of IDA solver. It allows the user to construct systems graphically from NMF based models. It allows hierarchical modelling, where any system may be used as a subsystem of another system.

IDA can be linked directly to the Microsoft EXCEL [83] spreadsheet tool and the output files from simulations can be opened directly from there.

IDA allows the dynamic modelling and simulation of systems. Modelling and simulation can be carried out by either using it in its basic form of NMF translator, Fortran compiler and IDA Solver or by means of the IDA Modeller interface.

IDA has been in use primarily for the simulation of building and energy systems since the early 1990's. During this time it has been used on a number of projects; these include natural ventilation, ventilation of road tunnels, refrigeration systems and district heating systems.

IDA Indoor Climate and Energy (ICE) [84] is a new tool based on IDA Modeller, for simulation of thermal comfort, indoor air quality and energy consumption. The principal requirement of ICE has been for its usability by non-experts. The user interface has been designed to support an infrequent user as well as the simulation expert. Wizards provide easy access to key input fields for common simulation tasks such as plant sizing.

IDA ICE may be used for most building types for calculation of:

- The full zone heat balance, including specific contributions from: sun, occupants, equipment, lights, ventilation, heating and cooling devices, surface transmissions, air leakage, cold bridges and furniture.
- Solar gain through windows with full account for local shading devices as well as surrounding buildings and other objects.
- Air and surface temperatures.
- Operating temperatures in multiple occupant zones.
- Directed operating temperature for estimation of asymmetric comfort conditions
- Comfort indices, PPD and PMV, at multiple occupant locations.
- Air CO_2 and moisture levels.
- Daylight level at and arbitrary room location.
- Air temperature stratification in displacement ventilation systems.
- Wind and buoyancy driven airflows through leaks and openings via a fully integrated airflow network model. This enables the study of temporarily open windows or doors between rooms.
- Power levels for primary and secondary system components.
- Total energy cost.

As with IDA the results from simulations can be exported directly into Microsoft EXCEL [83].

2.6.6 Conclusion

Five equation-based simulation tools, termed Modular Simulation Environments have been evaluated for their applicability to the Climatic Wind Tunnel project. In order to assess the applicability of each for the environments to the CWT project a number of criteria needed to be defined. The criteria around which the final decision of which platform would be used were as follows:

- Availability of existing component library
- Availability of appropriate NMF translator
- Source code availability
- Level of previous use in HVAC modelling
- Type of equation solver
- Output and portability to proprietary spreadsheet tools
- Commercial availability
- User interface
- Dynamic or steady-state simulation

All of the environments with the exception of SIMULINK have, to differing degrees, a proven level of capability in the modelling and simulation of HVAC systems.

All the simulation environments at their most basic level are accessed via text input. To gain widespread acceptance in the commercial field with non-expert users, graphical user interfaces (GUI's) are a highly desirable addition to the package.

A steady-state simulation tool is not appropriate for the analysis of the Climatic Wind Tunnel systems. This is due to the CWT'S thermal systems being made up of a great number of components that have a similar time response. A greater insight into possible directions for the reduction of energy consumption is possible by looking at the CWT'S systems whilst they are operating in a transient state.

Three Modular Simulation Environments strongly focused towards HVAC work and capable of dynamic simulation were considered for the undertaking of this task, these being: TRNSYS, SPARK and IDA.

TRNSYS developed during the mid 1970's at the University of Wisconsin Solar Energy Laboratory primarily for the simulation of solar energy systems. It has been used with success on a number of projects that have involved the modelling and simulation of HVAC systems and components [8,9].

SPARK is a simulation environment that is still currently under development at Lawrence Berkley Laboratory. It has been undergoing development work since its original conception in 1986. It has at present been used exclusively as a research tool and is unproven in a wider commercial setting. The first commercial release (WinSPARK 1.0) is due for release in the summer of 1999 – beyond the timescale of this project

IDA Simulation Environment is a product of collaboration between the Swedish Institute of Applied Mathematics and the Department of Building Services Engineering, Swedish Royal Institute of Technology. Development began in the late 1980's and IDA has been in use in the HVAC simulation field since 1990, during which time it has been used and proved successful on a number of projects. A new commercial package called IDA Indoor Climate and Energy aimed at the non-expert user is available. ICE is intended to be used in the simulation of thermal comfort, indoor air quality and building energy consumption.

TRNSYS, SPARK and IDA are all able to make use of the Neutral Model Format (NMF) description language [55] and translators are available for all environments. It has been decided that the IDA simulation environment will be used for the simulation work. The main reasons for this being that NMF is to be used for the description of the Climatic Wind Tunnels component models and there are very close ties between IDA and NMF. IDA allows the user the choice of a number of different numerical methods for finding the initial system operating conditions; this is likely to be a great advantage due to the large number of recirculating fluid loops present within the thermal systems.

2.7 References

[1] ASHRAE (1975). Procedures for simulating the performance of components and systems for energy calculations. American Society of Heating, Refrigerating and Air-Conditioning Engineers, New York. USA.

[2] DOE-2 users manual. Lawrence Berkley National Laborartory, California. USA.

[3] BLAST users manual. University of Illinois, Illinois, USA.

[4] Klein S.A, Beckman W.A, Duffie J.A. (1974). TRNSYS – A transient simulation program. ASHRAE Transactions Vol. 82 pt.2 pg. 623 – 633.

[5] Hackner R.J, Mitchell J.W, Beckman W.A (1984). HVAC system dynamics and energy use in buildings – part 1. ASHRAE Transactions Vol. 90 pt.2b pg. 523 – 535.

[6] Hackner R.J, Mitchell J.W, Beckman W.A (1985). HVAC system dynamics and energy use in buildings – part 2. ASHRAE Transactions Vol. 91 pt.1b pg. 781 – 795.

[7] Hackner R.J, Mitchell J.W, Beckman W.A (1985). System dynamics and energy use. ASHRAE Journal, June 1985. Pg. 29 – 37.

[8] Braun J.E (1988). Methodologies for the design and control of central cooling plants. PhD Thesis. University of Wisconsin, Madison, USA.

[9] Bourdouxhe J-P (1997). Use of the simulation tools in the design and operating phases of the air conditioning equipment. Laboratory of Thermodynamics, University of Liège, Belgium.

[10] Silverman.G.J, Jurovics S.A, Low D.W, Sowell E.F (1981). Modelling and optimisation of HVAC systems using network analysis. ASHRAE Transactions Vol. 87 pt 2 pg. 585 – 597.

[11] Park C, Clark D.R, Kelly G.E (1985). An overview of HVACSIM+, a dynamic building/ HVAC /control systems simulation program. Proceedings of Building Energy Simulation conference, Seattle. Pg. 175 – 185.

[12] Hanby V.I (1987). Simulation of HVAC components and systems. Building Services Engineering Research and Technology. Vol.8 n°.1 pg.5 – 8.

[13] Stoecker W.F (1989). Design of thermal systems, third edition.Mc-Graw-Hill, New York. USA. ISBN 0-07-061620-5.

[14] Clark D.R, Hurley C.W, Hill C.R (1985). Dynamic models for HVAC system components. ASHRAE Transactions Vol.91 pt.1 pg. 737 – 751.

[15] Wright J.A (1991). HVAC optimisation studies: Steady-state fan model. Building Services Engineering Research and Technology. Vol.12 n°.4 pg.129 – 135.

[16] Hanby V.I (1987). Performance modelling for HVAC plant. Journal of the Institute of Energy. December 1987 pg. 168 – 170.

[17] Murphy B, Murray M.A.P, Hanby V.I (1991). Simulation: a tool for improving the design process. IMechE paper C413/061.

[18] Hanby V.I (1998). Performance-Based sizing of HVAC systems. Proceedings System Simulation in Buildings. Liège, Belgium.

[19] Yasuda H, Touber S, Machielsen C.H.M (1983). Simulation model of a vapour compression refrigeration system. ASHRAE Transactions Vol. 89 pg. 408 – 425.

[20] Greyvenstein G.P (1988). A computer simulation model for the design and optimization of heat pumps and refrigeration systems. South African Journal of Science Vol. 84 pg. 494 – 502.

[21] Cecchini C, Marchal D (1991). A simulation model of refrigeration and air-conditioning equipment based on experimental data. ASHRAE Transactions Vol. 97 pg. 388 – 393.

[22] Herbas T.B, Berlinck E.C, Uriu C.A.T, Marques R.P, Praise J.A.R (1993). Steady-state simulation of vapour compression heat pumps. International Journal of Energy Research Vol. 17 pg. 801 – 816.

[23] Lysholm A.J.R (1943). New rotary compressor Proceedings of the Institution of Mechanical Engineers, 1943, London.

[24] O'Neill P, Beatts W (1970). The oil free screw compressor. Conference proceedings – Industrial Reciprocating and Rotary Compressors. Institution of Mechanical Engineers, London.

[25] Trulsson I (1970). A new development in rotary screw compressor design. Conference proceedings – Industrial Reciprocating and Rotary Compressors. Institution of Mechanical Engineers, London.

[26] Hodge J (1970). Some aspects of screw compressors. Conference proceedings – Industrial Reciprocating and Rotary Compressors. Institution of Mechanical Engineers, London.

[27] Lundberg A (1980). Capacity control for partial load operation of screw compressor units. ASHRAE Transactions Vol. 86 pt.1 pg. 485 – 492.

[28] Pillis J.W (1984). Design and application of variable volume ration screw compressor. ASHRAE Transactions paper no. 2876S.

[29] Firnhaber M.A, Szarkowicz D.S (1980). Modelling and simulation of rotary screw compressors. Proceedings Purdue Compressor Technology Conference, West Lafyette, Texas, USA.

[30] Bràblik J (1982). Analytical model of an oil-free screw compressor. Proceedings Purdue Compressor Technology Conference, West Lafyette, Texas, USA.

[31] Sångfors B (1982). Analytical modelling of helical screw machine for analysis and performance prediction. Analytical model of an oil-free screw compressor. Proceedings Purdue Compressor Technology Conference, West Lafyette, Texas, USA.

[32] Singh P.J, Patel G.C (1986). A generalized performance computer program for oil flooded twin screw compressors. Proceedings Purdue Compressor Technology Conference, West Lafyette, Texas, USA.

[33] Jonsson S (1991). Performance simulations of twin screw compressors with economizer. International Journal of Refrigeration Vol.14 pt 6 pg. 345 – 350.

[34] Yik F.W.H, Lam V.K.C (1998). Chiller models for plant design studies. Building Services Engineering Research and Technology. Vol. 19 N°. 4 pg. 233 – 241.

[35] Shelton S.V, Weber E.D. (1991). Modelling and optimisation of commercial building chiller / cooling tower systems. ASHRAE Transactions, Vol. 97, part 2 pg. 1209 – 1216.

[36] Gillespie T.D (1992). Fundamentals of vehicle dynamics, third edition. Society of Automotive Engineers. ISBN 1-56091-119-9.

[37] Heywood J.B (1988). Internal Combustion Engine Fundamentals. McGraw-Hill, New , New York. ISBN 0-07-100499-8.

[38] Sher E. (ed) (1998). Handbook of Air Pollution from Internal Combustion Engines: Pollutant Formation and Control. Academic Press, London. ISBN 0-12-639855-0.

[39] Heisler H (1995). Advanced Engine Technology. Arnold, London. ISBN 0-340-56822-4.

[40] Bhowmick S, Branchi C, McAssey E.V (1996). Heat transfer performance of engine coolants under sub-cooled boiling conditions. American Society of Mechanical Engineers Vol 26 pt.2 pg. 53 – 59.

[41] Andrews G.E, Harris J.R, Ounzain A (1988). Transient heating and emissions of an SI engine during the warm-up period. Society of Automotive Engineers, Technical paper 880264.

[42] Daw C.S, Kennel M.B, Finney C.E.A, Connolly F.T (1998). Observing and modelling non-linear dynamics in an internal combustion engine. Physical Review Vol.57 N°.3 pg. 2811 – 2819.

[43] Patton K.J, Nitschke R.G, Heywood J.B (1989) Development and evaluation of a friction model for spark ignition engines. Society of Automotive Engineers, Technical paper 890836.

[44] Kaplan.J.A, Heywood J.B (1991). Modelling the spark ignition engine warm-up process to predict component temperatures and hydrocarbon emissions. Society of Automotive Engineers, Technical paper 910302.

[45] Jarrier L, Gentile D (1997). Transient thermal behaviour simulation of a spark ignition engine. General review of Thermodynamics 1997 Vol.36 pg. 520 – 533.

[46] Stoecker W.F (1957). How frost formation on coils affects refrigeration systems. Refrigeration Engineering Vol. 65 pt2 pg. 42 – 46.

[47] Hosoda T, Uzuhasi H (1967). Effect of frost on the heat transfer coefficient. Hitachi Review Vol. 16 pt6 pg. 254 – 259.

[48] Gathilov T.S, Ivanova V.S (1979). Characteristics of frost formed on the surface of finned air coolers. Proceedings International Congress of Refrigeration Vol. 2 pg. 997 – 201.

[49] Sami S.M, Duong T (1989). Mass and heat transfer during frost growth. ASHRAE Transactions Vol 87 pt 1 pg. 158 – 165.

[50] Senshu T, Yasuda H, Oguni K, Ishibane K (1990). Heatpump performance under frosting conditions: Part 1 – Heat and mass transfer on cross finned tube heat exchangers under frosting conditions. ASHRAE Transactions Vol 96 pt 1 pg.324 – 329.

[51] Senshu T, Yasuda H, Oguni K, Ishibane K (1990). Heatpump performance under frosting conditions: Part 2 – Simulation of heatpump cycle characteristics under frosting conditions. ASHRAE Transactions Vol 96 pt 1 pg.330 – 336.

[52] Padki M.M, Sherif S.A, Nelson R.M (1989). A simple method for modelling the frost formation phenomenon in different geometries. ASHRAE Transactions Vol 95 pt 2 pg. 1127 – 1137.

[53] Oskarsson S.P, Krakow K.I, Lin S (1990). Evaporator models for operation with dry, wet and frosted fin surfaces. Part 1: Heat transfer and fluid flow theory. ASHRAE Transactions Vol 96 pt 1 pg. 373 – 380.

[54] Oskarsson S.P, Krakow K.I, Lin S (1990). Evaporator models for operation with dry, wet and frosted fin surfaces. Part 2: Evaporator models and verification. ASHRAE Transactions Vol 96 pt 1 pg. 373 – 380.

[55] Sahlin P, Sowell E.F (1989). A neutral format for building simulation models. Proceedings of Building Simulation '89. Vancouver, Canada.

[56] Sahlin P (1996). NMF Handbook: an introduction to the neutral model format. HTTP//:www.brisdata.se/. BrisData AB, KTH, Stockholm, Sweden.

[57] Sahlin P, Bring A, Sowell.E.F (1996). The neutral model format for building simulation. HTTP//:www.brisdata.se/. BrisData AB, KTH, Stockholm, Sweden

[58] Modelica design group (1998). Modelica – a unified object-orientated language for physical systems modelling, version 1e. Dynasim AB, Lund, Sweden.

[59] Mattsson S.E, Elmquist H (1997). Modelica – an international effort to design the next generation modelling language. Proceedings Computer Aided Control Systems Design, Gent, Belgium.

[60] Mattsson S.E, Elmquist H (1998). An overview of the modelling language Modelica. Proceedings Eurosim '98, Helsinki, Finland.

[61] Hensen J.L.M (1991). On the thermal interaction of building structure and heating and ventilation system. PhD Thesis

[62] Åström K.J, Elmqvist H, Mattsson (1998).Evolution of continuous-time modelling and simulation. 12th European Simulation Multiconference, ESM '98, Manchester, England.

[63] Selfridge R.G (1955). Coding a general-purpose digital computer to operate as a digital analyzer. Proceedings 1955 Western Joint Computer Conference.

[64] Strauss J.C (1967). The SCi continuous system simulation language (CSSL). Simulation, 9, pg. 281 – 303.

[65] Grace A.C.W (1991). SIMULAB, An integrated environment for simulation and control. Proceedings of the 1991 American Control Conference, pg. 1015 – 1020.

[66] Mathworks Inc (1996). Student edition of SIMULINK – users guide. Prentice Hall.

[67] Husaunndee A. et al (1997). SIMBAD: A simulation toolbox for the design and test of HVAC control systems. Building Simulation '97, 5th IBPSA conference, Prague, Checz Republic. Vol 2 pg. 269 – 276.

[68] Murray M.A.P (1984). Component based performance simulation of HVAC systems. PhD Thesis.

[69] Klein S.A, Beckman W.A, Duffie J.A (1976). TRNSYS – a transient simulation program. ASHRAE Transactions vol. 82 pt. 2 pg. 623 – 633.

[70] IISiBat - TRNSYS – A transient simulation program manual. Solar Energy Laboratory, University of Wisconsin. (1997).

[71] Fiskel A, Thornton J.W, Klein S.A, Beckman W.A (1994). Developments to the TRNSYS simulation program. Joint Solar Conference, American Society of Mechanical Engineers pg. 379 – 384.

[72] Simulation Problem Analysis Research Kernel (SPARK) – Users Manual (1997). Simulation Research Group, Lawrence Berkeley National Laboratory, California, USA.

[73] Sowell E.F, Buhl W.F, Erdem A.E, Winklemann F.C (1986). A prototype object-based system for HVAC simulation. Proceedings System Simulation in Buildings, Lièige, Dec. 1986.

[74] Sowell E.F, et al (1984). Generation of building energy system models. ASHRAE Transactions, vol. 90 pt. 2 pg. 573 – 586.

[75] Sowell E.F, Buhl W.F (1988). Dynamic Extension of the Simulation Problem Analysis Kernel (SPANK). Proceedings of the USER-1 Building Simulation Conference, Ostend, Belgium.

[76] Nataf J-M, Winkleman F.C (1992). Automatic code generation in SPARK: Appications of computer algebra and computer compilers. Simulation Research Group, Lawrence Berkeley Laboratory, California, USA.

[77] Haves P, Sowell E.F (1998). Component-based and equation based solvers for HVAC simulation: a comparison of HVACSIM+ and SPARK. Proceedings System Simulation in Buildings, Lièige, Dec. 1998.

[78] Sahlin P, Bring A (1991). IDA Solver: A tool for building and energy systems simulation. Proceedings IBPSA Building Simulation 1991, Nice, France.

[79] Sahlin P (1988). MODSIM: A program for dynamical modelling and simulation of continuous systems. Proceedings SIMS88 conference. Helsinki, Finland.

[80] Sahlin P (1996). Modelling and simulation methods for modular continuous systems in buildings. PhD Thesis. Royal Institute of Technology, Stockholm, Sweden.

[81] Sahlin P (1993). IDA Modeller: A man made interface for building simulation. Proceedings IBPSA Building Simulation 1993, Adelaide, Australia.

[82] Lasdon L.S, Et al (1978). Design and testing of a generalized reduced gradient code for nonlinear programming. ACM Transactions on mathematical software, Vol.4, N°. 1 pg. 34 – 50.

[83] Microsoft EXCEL spreadsheet: Microsoft Corp. Seattle, Wahington. USA

[84] IDA Indoor Climate and Energy (ICE): Bris Data AB, Stockholm, Sweden.

[85] ASHRAE Secondary Toolkit for Energy Calculation. American Society of Heating Refrigeration and Air Conditioning Engineers, Atlanta, Georgia. USA.

[86] Clarke J.A, McLean D.J (1986). ESP – A building and plant energy simulation system. ESP Manual. University of Strathclyde, ABACUS, Glasgow. Scotland.

Chapter 3

Component model development

Without a comprehensive model library any Modular Simulation Environment (MSE) is of no use [1].

A great number of component models have been developed for use in specific MSE's. These models are generally written in languages and formats that are specific to the simulation environment in question; example languages are C and FORTRAN.

With the increased use of simulation in the design of building thermal systems and the time constraints placed on projects, it is not possible to develop models for each component required in a particular system. It is because of this now paramount to reuse existing models that have been developed and proved to replicate a components real life operation. It was to facilitate this interchange of models across the simulation community that the Neutral Model Format (NMF) was proposed [2].

Since the original proposal NMF models have been developed for use in fire simulation and simulation of airflow through building zones. As well as these models the ASHRAE secondary toolkit for energy consumption [3] has been translated into NMF and along with the other models is freely available [4].

The models used in the Climatic Wind Tunnel (CWT) simulation work are listed in Table 3.1.

Airzone	Fan – single speed
Capacitance	Fan – two speed
Car	Heat exchanger - Drycoil
Compressor	Heat exchanger – CCSIM
Condenser	Heat flux node
Conductance	Humidifier
Converging wye	Humidistat
Diverging wye	Mixing tee
Diverging tee	Mixing valve
Diverting valve	Proportional controller – heating
Duct	Proportional controller – cooling
Evaporator	Two port modulating valve

Table 3.1 Models used in Climatic Wind Tunnel simulations

Some of the models used are taken from the NMF ASHRAE Secondary Toolkit [3], others are translations of existing models [5] (written for other MSE's) into NMF. A complete list of all models used can be found in Appendix A.

All models used in the make up of the air circuit use UNIAIR links. This type of link contains five terms:

- Pressure (Pair)
- Pollutant fraction(Xair)
- Temperature (Tair)
- Mass flow (Mair)
- Humidity content (Wair)

A number of models have had to be specifically developed for the project. It is the development of these models that is discussed within this chapter.

3.1 Airzone and thermal network

The two test areas (airzones) found within the CWT are composed from lightweight steel skinned Polyurethane foam elements and heavyweight concrete floor . The airzones formed by the panels are completely internal and are neither subject to external climate variations nor solar gain.

Instead of a specific model to represent an airzones, the approach used was to assemble the walls, roof and floor of the zone from a series of conductance and capacitance models. This method gives great flexibility in the construction and configuration of the fabric of the zone.

The model has been developed from basic principles of heat transfer as the construction of the entire fabric with the exception of the floors is of a thermally lightweight "sandwich" with dynamic characteristics far removed from normal building constructions.

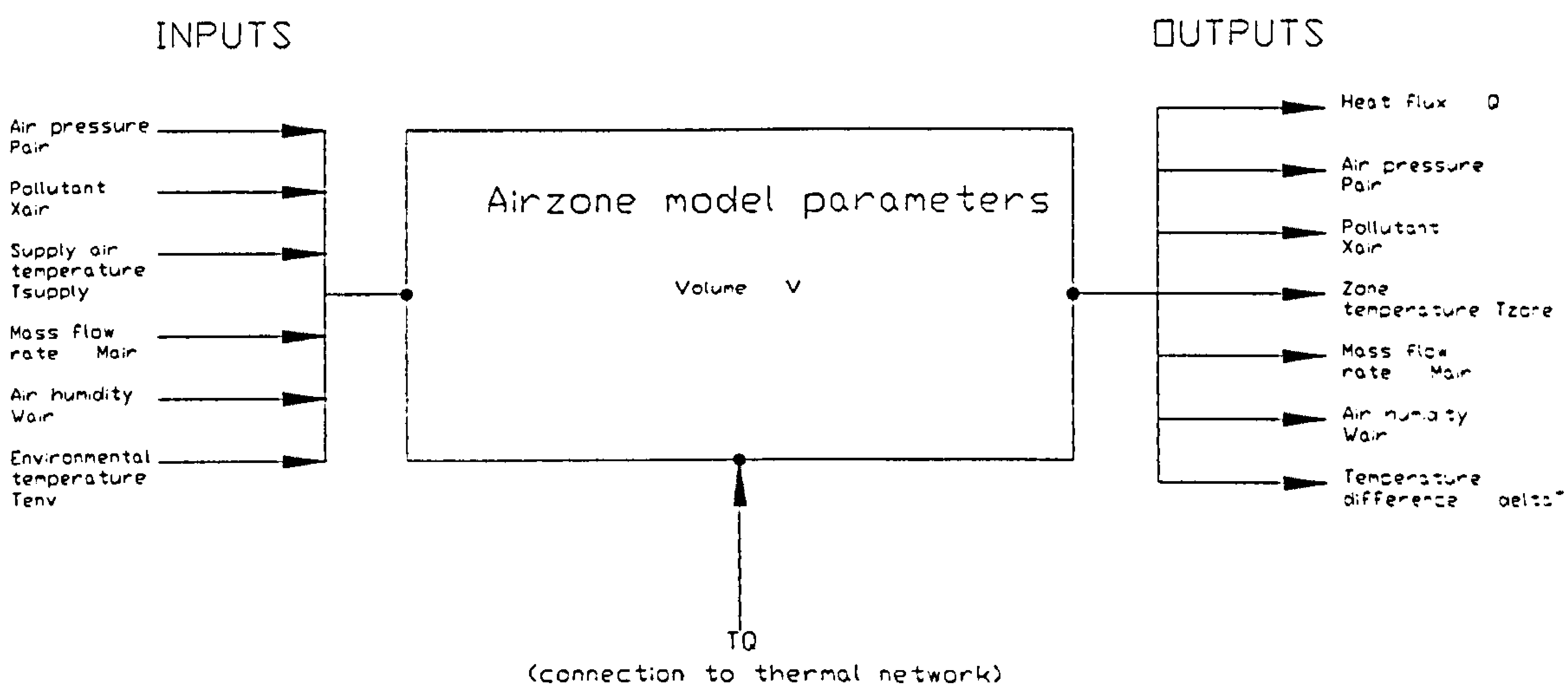


Figure 3.1 Information flow diagram for Airzone model

The user definable parameters for this component are:

Zone volume	(V)	m^3

The in/out variables for the model are:

Mass flow air	(M_{air})	kg/s
Air humidity ratio	(W_{air})	kg/kg
Heat flux from temperature node	(Q)	W
Temperature difference	(deltaT)	°C
Air supply temperature	(T_{supply})	°C
Zone temperature	(T_{zone})	°C
Environmental temperature	(T_{env})	°C
Air pressure	(P_{air})	Pa
Pollutant fraction	(X_{air})	Dimensionless

The equation for this model is:

$$CP_{air}V\rho\frac{dT_z}{dt} = M_a CP_{air}(T_s - T_z) + Q \tag{3.1}$$

where:

$\mathbf{CP}_{air}$ = Specific heat of air (J/(kg-K)

V = Volume (m^3)

ρ = Air density (kg/m^3)

$\mathbf{T_z}$ = Zone temperature (°C)

$\mathbf{M_a}$ = Mass flow rate of air (kg/s)

$\mathbf{T_s}$ = Supply temperature (°C)

Q = Heat flux (W)

Equation 3.1 is a dynamic energy balance about the mass of air in the room. It is assumed that the air can be described by a single temperature.

3.1.1 Thermal response of composite wall construction

The walls of the airzone are of a composite construction consisting of an inner and outer face of 1mm steel (coated in plastisol) enclosing a layer of 100mm thick blown polyurethane foam. The thermal response of these composite walls is a significant factor in the dynamic thermal response of the overall system.

The composite construction requires a numerical solution to the heat transfer equations and an accurate but computationally economical method of modelling the construction needs to be found.

3.1.1.1 Criterion for method evaluation

In order to accurately assess each method of modelling, a set of criteria need to be laid down in order that they may all be assessed having been subjected to the same test conditions. The type of test that they are to be subjected to is also of great importance.

If simplified thermal models are used the optimum configuration of the model will vary according to the forcing function which is applied to the test. The common test methods used are sinusoidal inputs and step inputs. Sinusoidal inputs are for simulations in which diurnal cycles are important, such as the response of buildings that are subject to external environmental temperature swings. Due to the nature of the CWT it is not exposed to these diurnal temperature swings and a step input test is deemed more appropriate. For the carrying out of the test the following test conditions were used in all cases:

- A step input temperature rise of 1°C applied to the inner face of the wall at time 0.
- Constant temperature of 0°C maintained in the space in contact with the outer surface of the wall.
- An initial temperature of 0°C was established throughout the construction.
- The heat flux into $1m^2$ of the inner surface of the wall from the air was used to measure the thermal response of the wall, i.e. the dynamics of the inside air temperature were regarded as being of primary concern.

A benchmark model was used to establish a datum by which the performance of the candidate models could be assessed. The assessment was carried out by examining visually the time history of the heat flux into the inner wall and secondly by summing the square of the deviations of the model from the benchmark at each of the computed points [6], which was in this case every 100 seconds.

As the model is that of a low capacitance construction (being essentially the same as a refrigerator construction) the energy balance technique used for its analysis is a text book technique [7] for analysing such systems.

3.1.1.2 Thermal properties of materials

The properties of the materials (wall and floor) used in the CWT construction were obtained from Incropera [7] and the wall surface resistances from the CIBSE Guide A [8]. Values used are summarised in Table 3.2.

Material	Conductivity ($W\ m^{-1}K^{-1}$)	Specific heat ($J\ kg^{-1}K^{-1}$)	Density ($Kg\ m^{-3}$)
Steel	60.5	434.0	7854.0
PU foam	0.026	1045.0	70.0
Concrete	1.4	880.0	2300.0

Table 3.2 Thermal properties of materials used in CWT construction

The wall and floor surface resistances were taken as 0.12 $K\ m^2\ W^{-1}$ giving a film coefficient of 8.33 $W\ m^{-2}\ K^{-1}$.

3.1.1.3 Model #1 - Benchmark model

A spatially discretized model consisting of linked thermal resistance and capacitance elements was selected as the benchmark [9, 10]. The first step in evaluating this model was to assess the dynamic thermal behaviour of the steel outer layers. The Biot number (equation 3.2) is the ratio of the internal thermal resistance of a solid to the boundary layer thermal resistance [15] and gives an assessment of whether a lumped parameter modelling approach is applicable. For lumped parameter modelling to be considered adequate a Biot number less than 1 must be achieved. The Biot number is given by:

$$\frac{(h.x)}{k} \qquad (3.2)$$

Where:

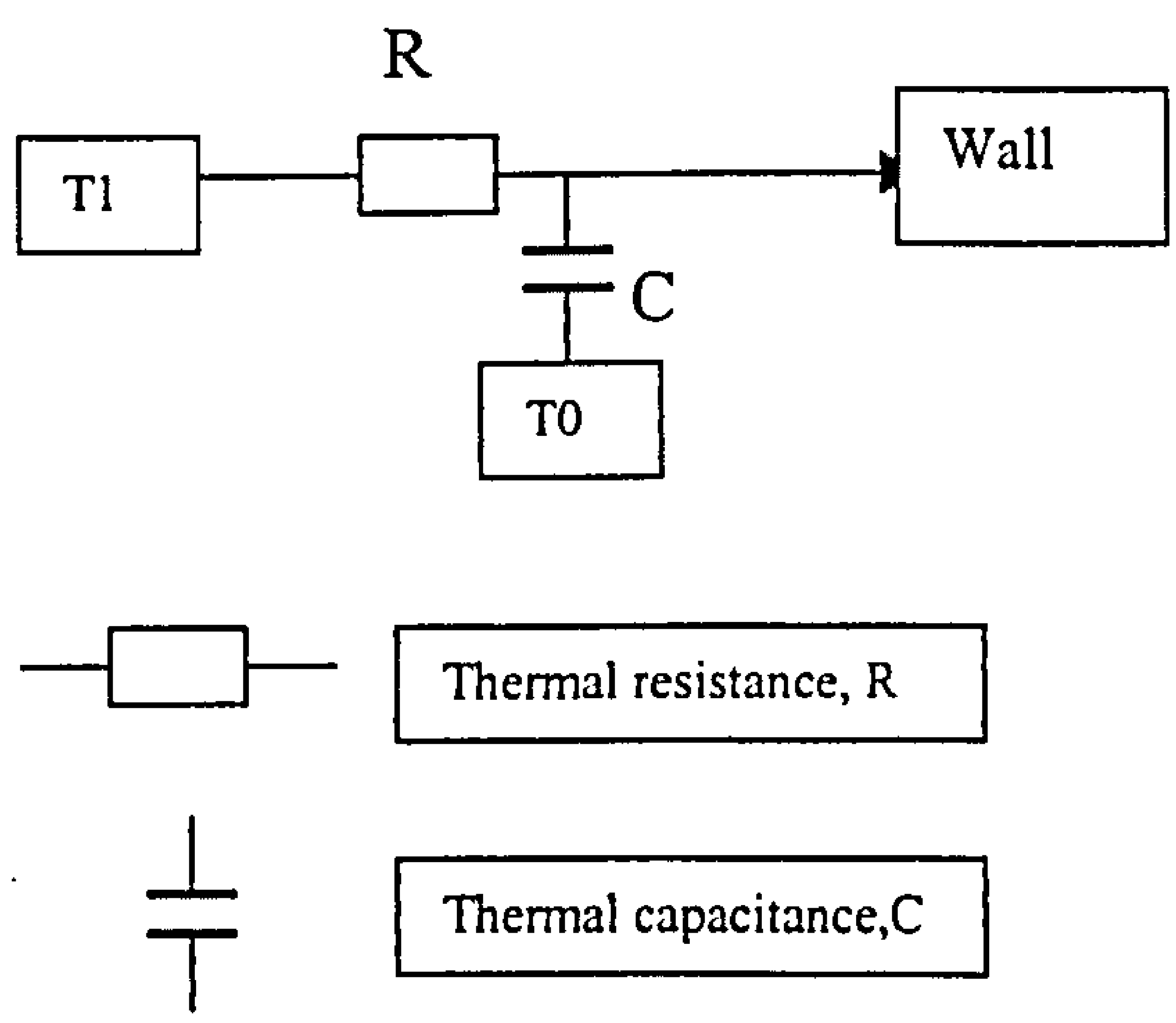
h = film coefficient of heat transfer

x = thickness of metal layer

k = thermal conductivity of layer

The Biot number has been calculated for the steel layer only and is low at 1.38e-4. This is much less than the value of 1 required and indicates that a single lumped capacitance will give a satisfactory model of its response. Fig 3.2 shows a schematic of the thermal network used to represent the steel layer.

The Biot number has only been calculated for the inner steel layer due to its dominant effect in the dynamic thermal response of the lightweight walls. The purpose of the following study is to assess to what level the polyurethane layer needs to be modelled.



Note: T1 and T0 are surface temperatures

Figure 3.2 R-C model of steel layer

In this case the R – element is the inside thermal resistance of the wall and the C - element the product of mass and specific heat for $1m^2$ of the steel.

The polyurethane (PU) layer was represented by three 'T' resistance-capacitance elements, this figure being more than adequate than adequately rigorous for this lightweight layer. In this case each resistance element represents one sixth of the total thermal resistance of the layer and each capacitance element one third of the total PU thermal capacitance.

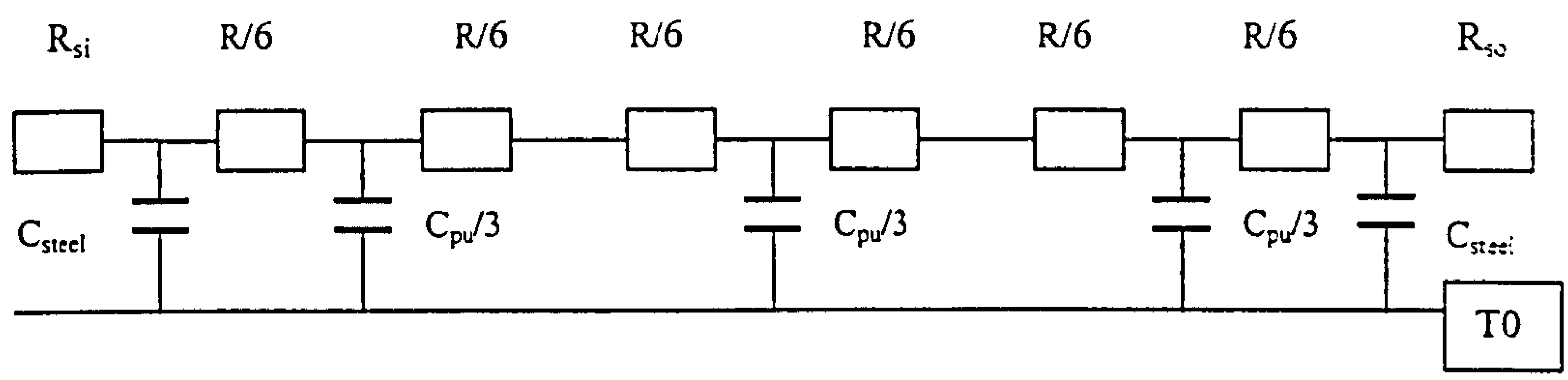


Figure 3.3 Model #1 - benchmark model
R-C network of the inner steel, PU layer, outer steel

3.1.1.4 Model #2 - Lumped parameter R-C wall

This model consists of a single classical R-C configuration [9, 10] with the total thermal capacity of the wall lumped into one element that is located centrally in the total resistance of the PU:

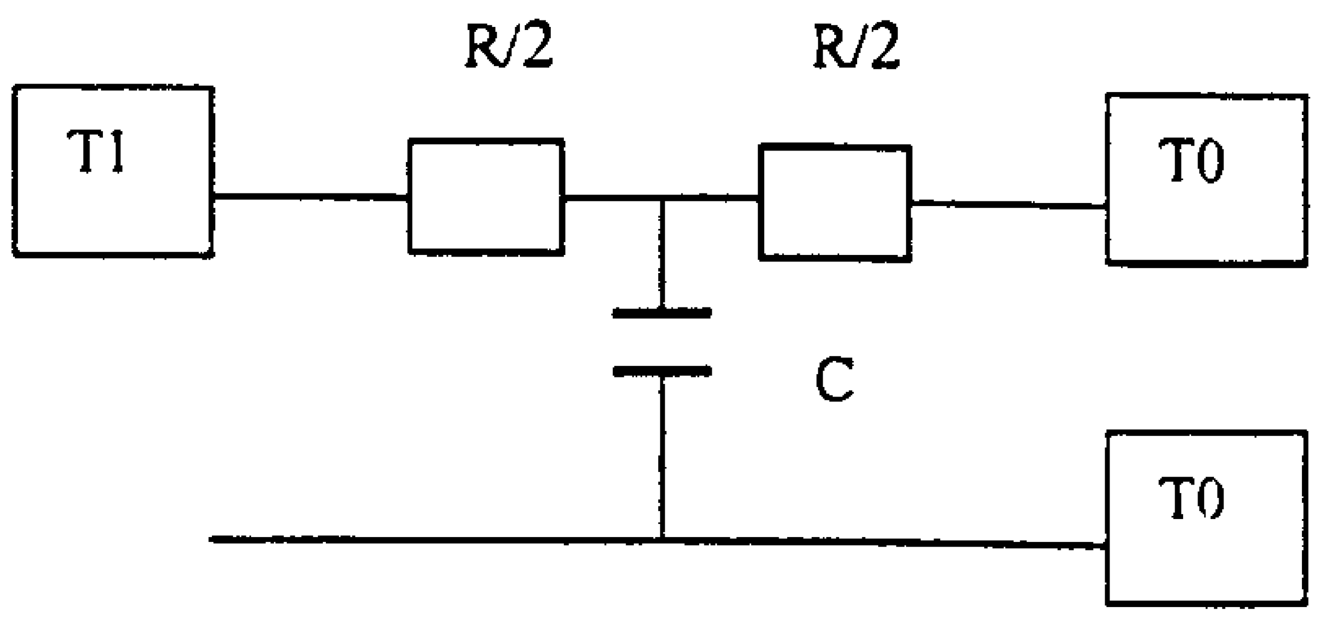


Figure 3.4 Model #2 - Lumped R-C model

3.1.1.5 Model #3 - Inner capacitance

In this configuration only the capacitance of the inner steel layer was accounted for, the capacitance of the PU layer and the outer steel cladding were ignored. The resistance of the PU layer and the outside surface resistance connect this capacitance to the inside by the inner surface resistance only, and to the outside. The thermal resistance of the 1mm steel is negligible.

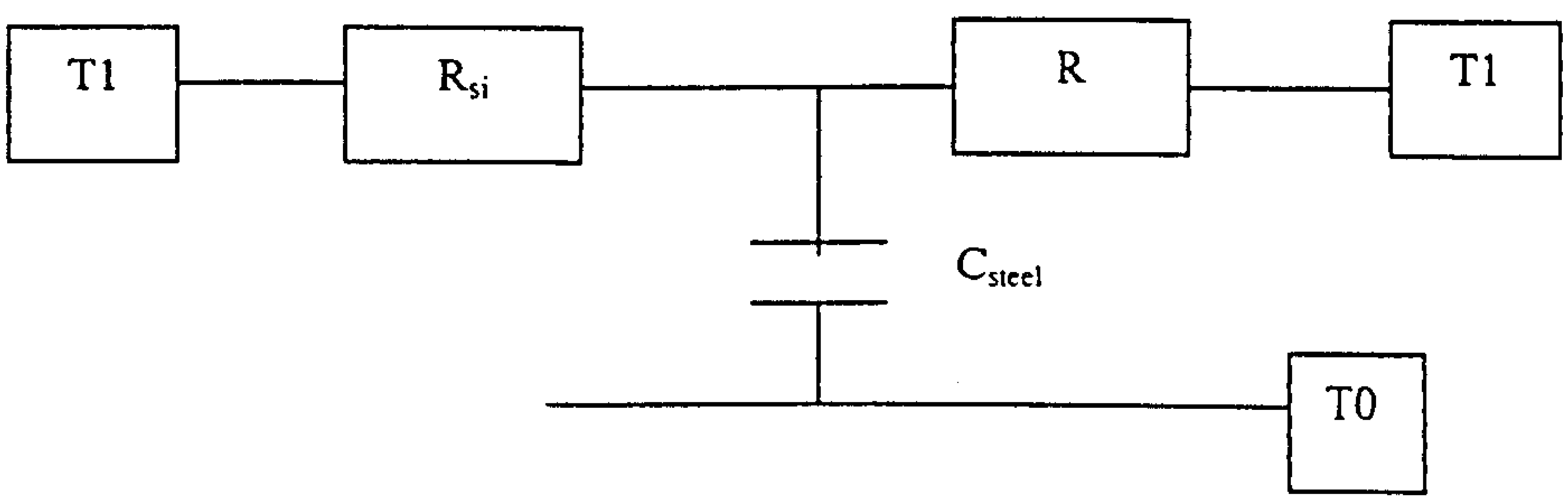


Figure 3.5 Model #3 – Inner capacitance

3.1.1.6 Model #4 - Pi configuration

This model consists of, from the inside, the inner surface resistance, capacitance of the inner steel layer, the resistance of the PU layer, the capacitance of the outer steel layer and finally the outer surface resistance [9].

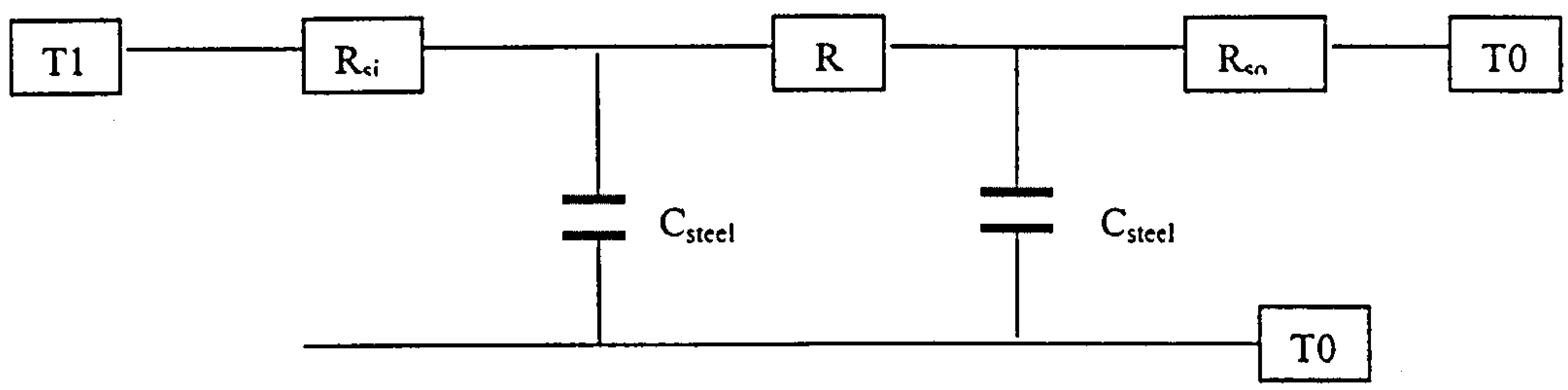


Figure 3.6 Model #4 - Pi configuration

3.1.1.7 Model #5 - Three capacitance

The three capacitance model is similar to model #4 (Figure 3.6), excepting that a single node representing the capacitance of the PU layer is placed at the mid-point of the PU resistance. This also related to the benchmark model, excepting that the PU is modelled by one capacitance element instead of three.

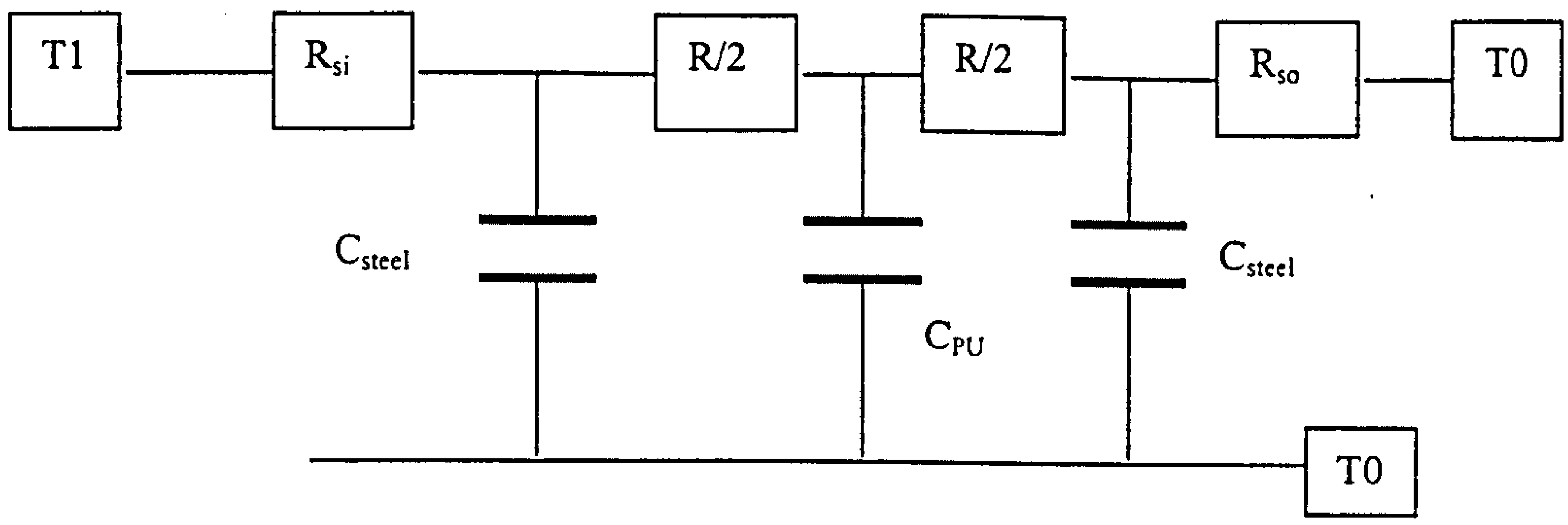


Figure 3.7 Model #5 - Three capacitance

3.1.2 Results and model comparison

The results of the benchmark simulation when this wall is subjected to a 1°C step temperature rise are shown in Figure 3.8.

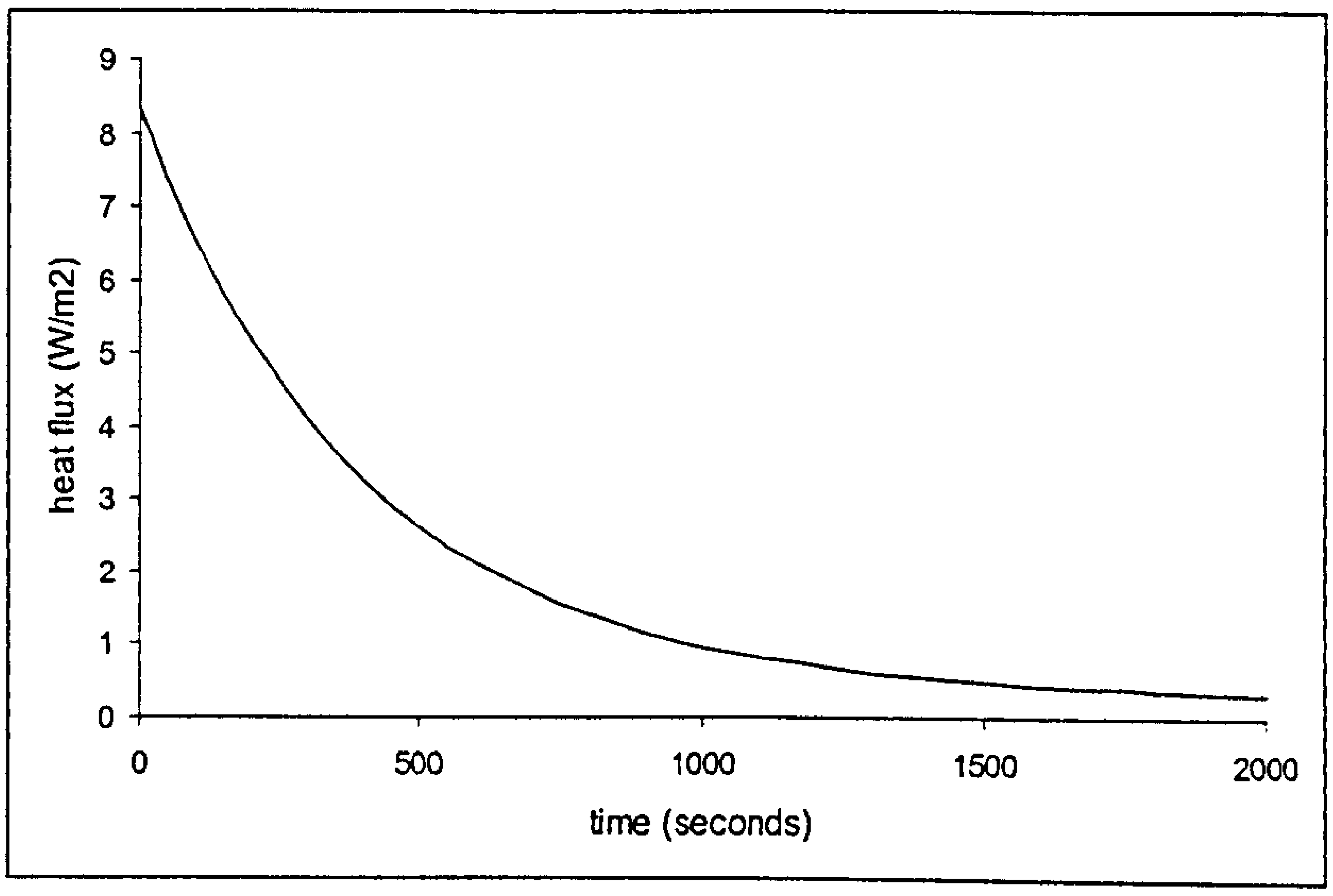


Figure 3.8 Thermal response of benchmark model

The results show that initially there is a heat flux into the wall of about 8 Wm^{-2}, with a slow decline towards the steady-state value of 0.26 Wm^{-2} after about 2000 seconds.

The thermal response of model #2 is shown in Figure 3.9 below. It can be seen that the distribution of capacitance leads to a severe underestimate of heat transfer into the wall, with a slight overestimate in the later stages of the test (when the heat flux is approaching the steady-state value).

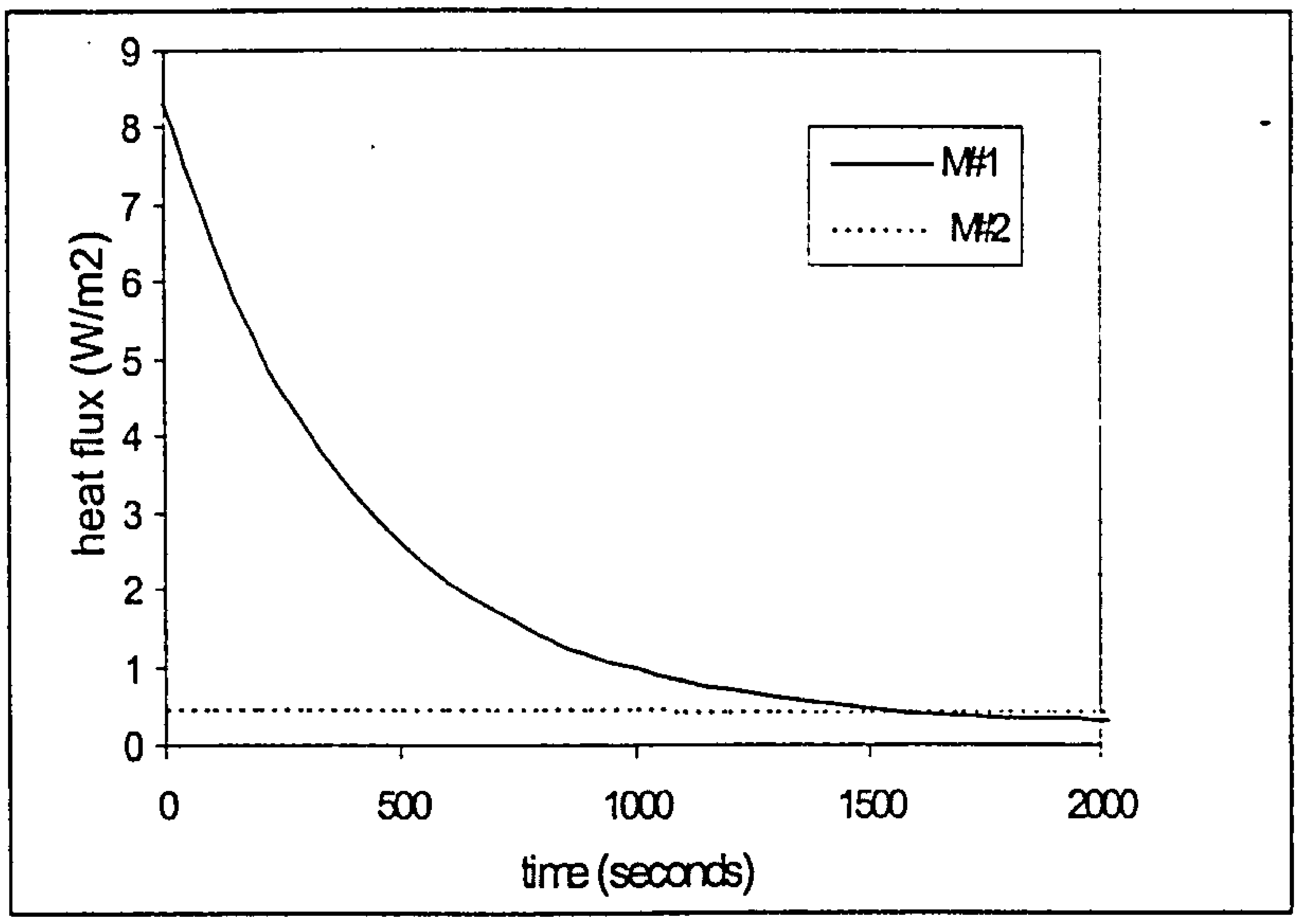


Figure 3.9 Response of lumped parameter model

The response of the remaining models M#3 – M#5 is shown on Figure 3.10. The benchmark model M#1 has also been included this plot.

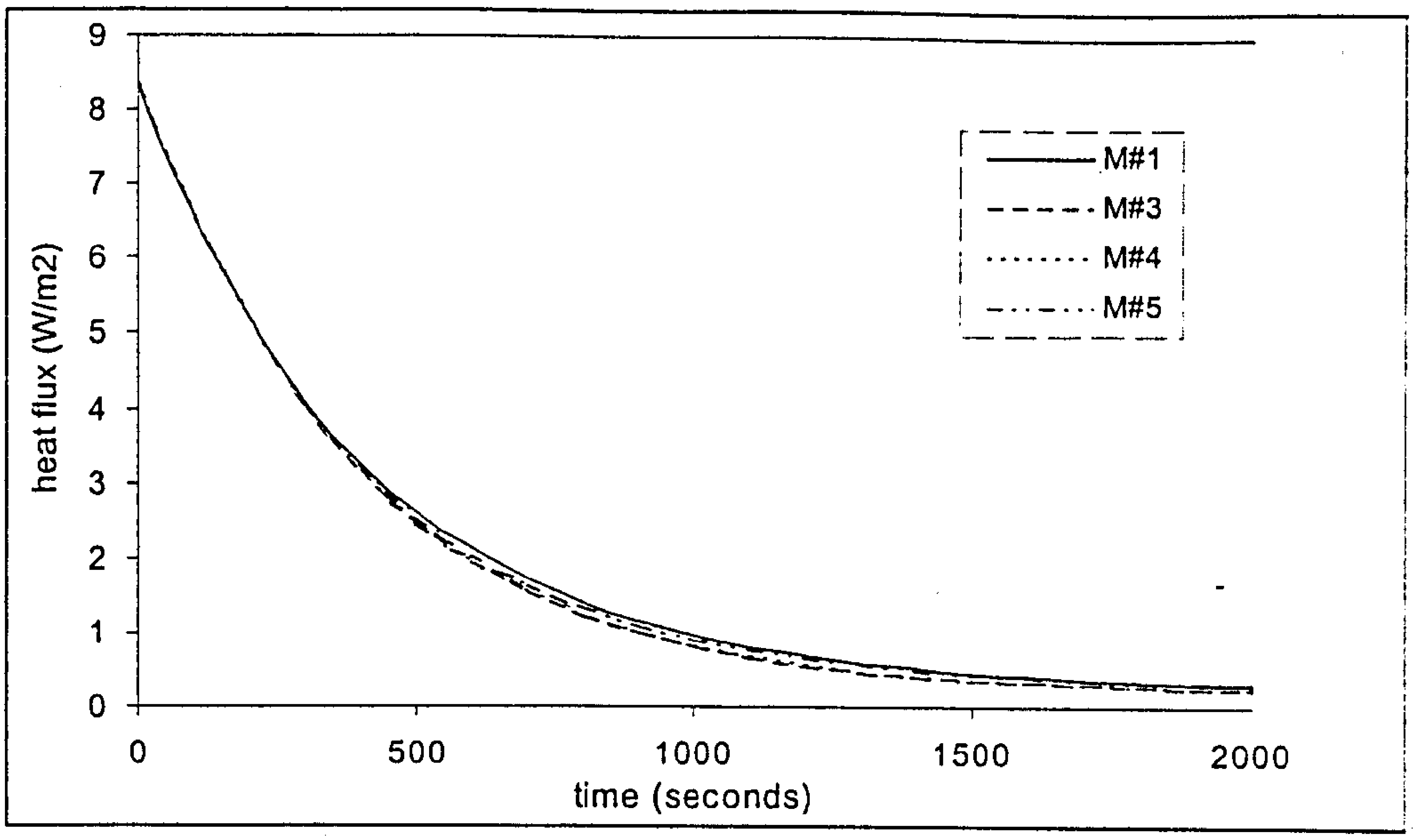


Figure 3.10 Response of models M#1 and M#3 - #5

It can be seen that a visual comparison indicates that the symmetrical lumped parameter model M#2 gives a very poor representation of the dynamic thermal response of this wall construction but that all the other models give results close to the benchmark

The results of analysing the sum of the squares of the deviation of the predicted heat flux (from the datum of the benchmark model) are shown in the Table 3.3 below.

Model	Squared deviation from benchmark
Lumped parameter (M#2)	153.12
Inner capacitance (M#3)	0.2504
Pi configuration (M#4)	0.2432
Three capacitance (M#5)	0.0380

Table 3.3 Results of squared deviation from benchmark model

These results show firstly that model M#2 has a significantly worse performance than the others. Secondly, that there is little to be gained by modelling the thermal capacitance of the outer steel layer (comparison of M#3 and M#4). There is a definite gain to be had by modelling the thermal capacitance of the PU insulation (M#5). Yet it is considered that in this case the computational simplicity of using the classical 'T' configuration of model M#3 makes this the most suitable form of model for dynamic heat transfer in the Airzone models used in CWT simulations.

The network of capacitance and conductance models representing the airzone walls are joined together using temperature/heatflux (TQ) links. Each link has two variables, a temperature and a heatflux. This encapsulation of component model behaviour is an important model structuring principle. The heatflux is positive into each model and positive out of it.

Note: from this point in the text the term “node” refers to a “Kirchoff Junction” which performs balances on heat or mass flow as appropriate.

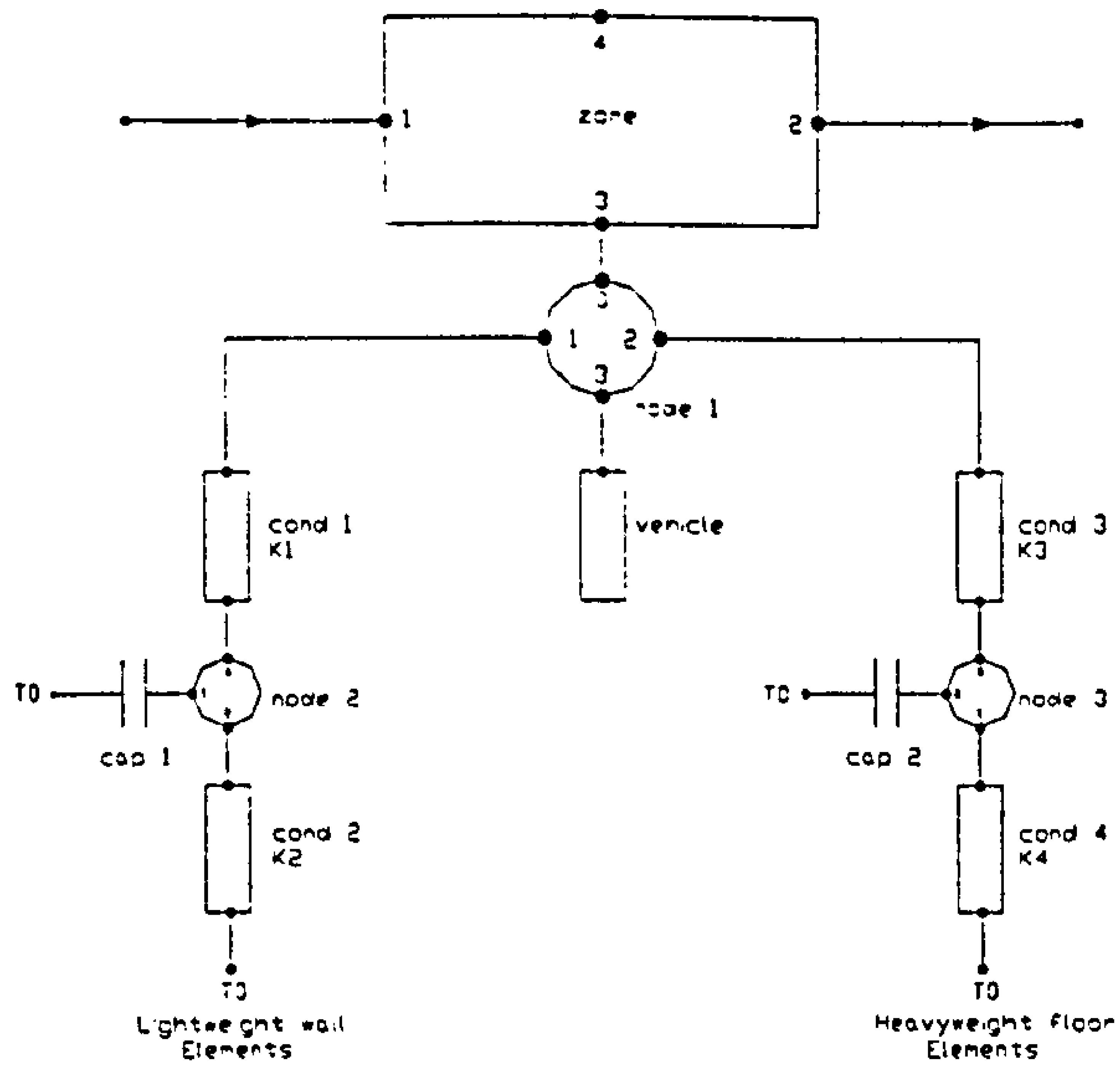


Figure 3.11 Thermal network representing airzone building fabric

3.1.3 Heavyweight floor construction

The heavyweight floor element is a homogenous layer of 300mm dense concrete. This layer due to its construction will be represented by the classical "T" conductance – capacitance network as illustrated in Figure 3.4. A distributed parameter model was initially considered for the representation of this element but through comparison was found to offer no advantage over the single node model employed in final network.

3.1.4 Test chamber dynamics

The airzone used to represent the test chamber has much in common with a duct in the way that the air flows through it. The test chamber has a maximum airflow of 600,000m^3/h the equivalent of 3600 ac/h. An investigation into the dynamic modelling of a duct using the nodal approach used for the airzone model was carried out [20]. The study involved constructing a duct model by linking together airzone models and their associated resistance-capacitance networks that represent the duct walls. The study found that to achieve the optimal representation of a duct 46 nodes are required. The study also showed that there was little difference in the time response of the duct when using 2 nodes compared to 46.With the simulation work being conducted using a beta test version of IDA Solver [21] the size of input file was limited. To include a representation of the test chamber made up of 46 nodes as well as all the associated HVAC plant models would have led to the maximum input file size being approached or exceeded. A it was shown that little was to be gained from such a rigorous modelling approach the simulation work was carried out using a single airzone to represent the test chamber.

3.2 Vehicle model development

As the purpose of the Climatic Wind Tunnel is to test vehicles under differing climatic conditions, it is imperative that a model of a vehicle is included to allow for its effect on the HVAC plant dynamics.

The test chamber is used to test vehicles under the following ways:

- Pull hot vehicle down to test conditions from operating temperature (vehicle not running).
- Vehicle start and running from cold in cold airstream.
- Vehicle start and running from hot in hot airstream.
- Vehicle start and running in hot and humid airstream.
- Change in airstream temperature at any point during test.

Only a simple model is required, as it is not important to look at what is occurring either inside the engine block or within the driver's compartment. The areas of interest are the heat flux to and from the surroundings and the resulting temperature changes within the engine. Hence the model is based upon assuming the engine to be one homogenous mass i.e. all the components are lumped together. The model is based upon a steady-state energy balance with first-order dynamics superimposed to simulate the heat-up and cool-down.

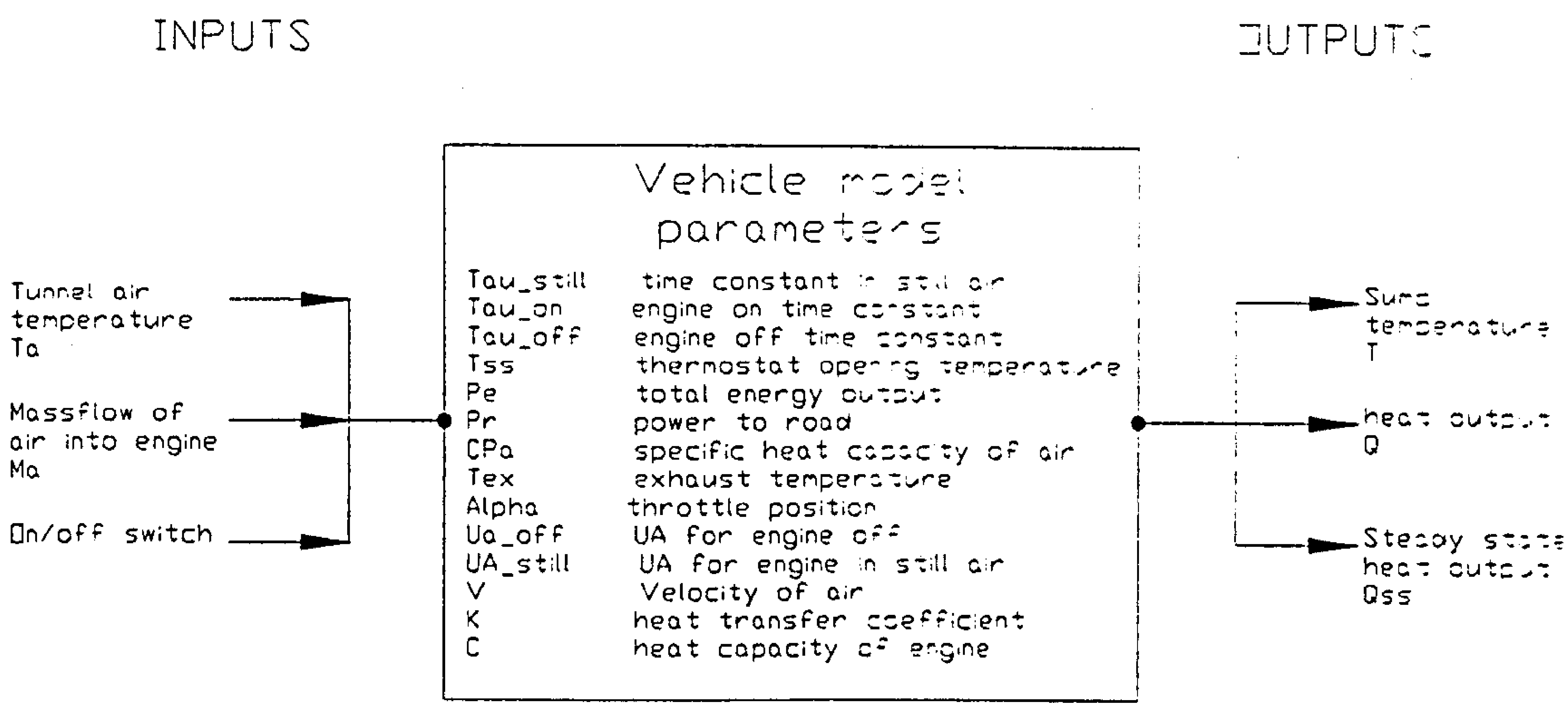


Figure 3.12 Information flow diagram for vehicle model

The user definable parameters for this model are:

Time constant in still air	(τ_{still})	Dimensionless
Engine on time constant	(τ_{on})	Dimensionless
Engine off time constant	(τ_{off})	Dimensionless
Thermostat opening temperature	(Tss)	Dimensionless
Total energy output of engine	(Pe)	W
Power to rolling road	(Pr)	W
Specific heat capacity of air	(Cpa)	J/kg K
Exhaust temperature	(Tex)	°C
Throttle position	(α)	Dimensionless
UA for engine off	(UA_{off})	W/K
UA for engine in still air	(UA_{still})	W/K
Tunnel air velocity	(V)	m/s
Heat transfer coefficient	(K)	W/K
Heat capacity of engine	(C)	J/K

The in / out variables for the model are:

Tunnel air temperature	(T_a)	°C
Mass flow of air into engine	(M_a)	kg/s
On / off switch	(Switch)	Dimensionless
Oil sump temperature	(T)	°C
Engine warm up heat output	(Q)	W
Engine steady state heat output	(Q_{SS})	W

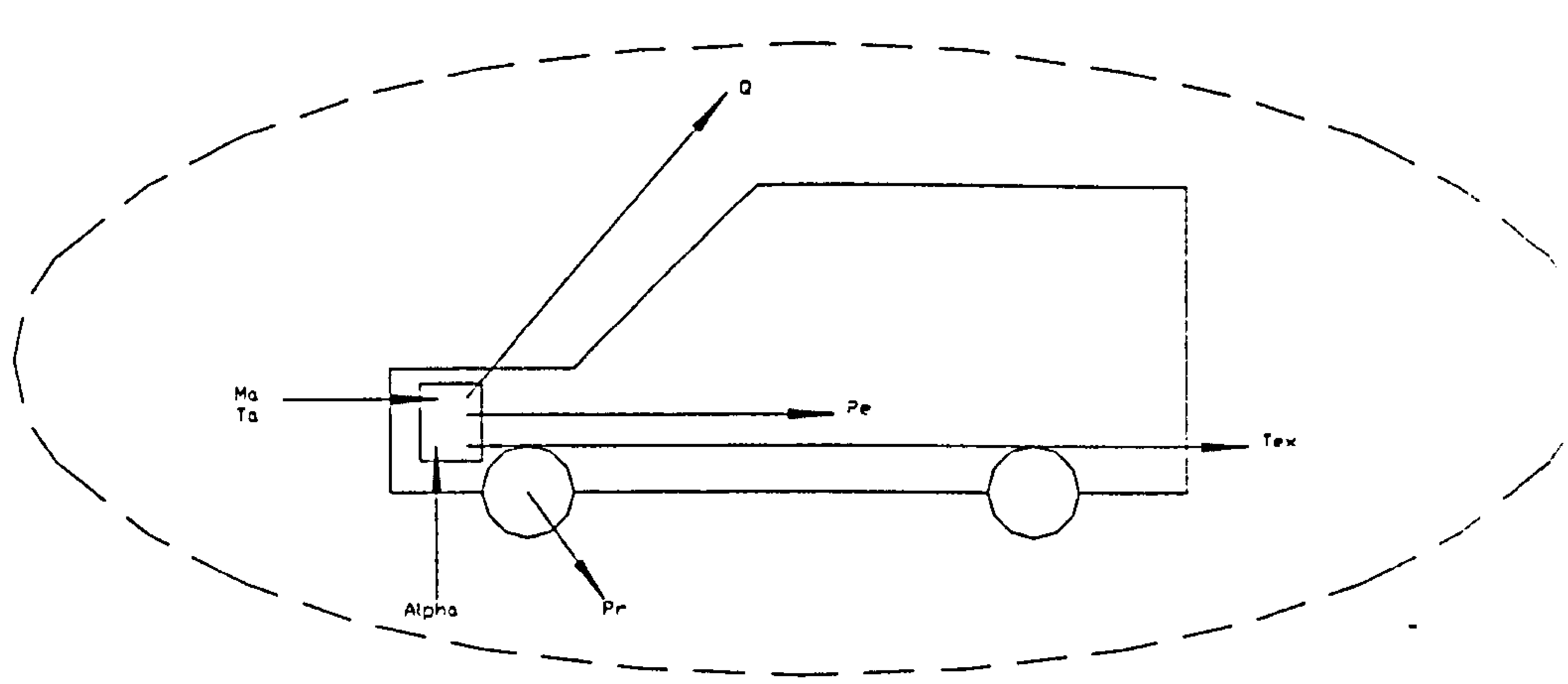


Figure 3.13 Vehicle model system diagram

The equations for this model are:

$$Q_{ss} = (\alpha.P_e) - (P_r + (\alpha.M_a.CP_{air})(T_{ex} - Ta)) \tag{3.3}$$

Equation (3.3) is an energy balance that calculates the steady state heat output of the engine when it is running. Its output is regulated by the factor α which represents the throttle position (0 – 1). The throttle position is multiplied by the total energy output of the engine to give the engine power output at the required driving condition. It is assumed that the exhaust temperature is constant.

Equation (3.4) calculates the temperature change occurring within the engine whilst it is running and is derived from equating the temperature rise of the engine to a first order lag curve. The use of a first order lag curve is justified by the recorded engine warm-up data shown in Figure 3.13 and 3.14.

$$T_t = T_{ss}\left(1 - e^{\frac{-t}{\tau_{on}}}\right)$$

$$T_t - T_{ss} = \left(-T_{ss} e^{\frac{-t}{\tau_{on}}} \right)$$

$$T' = \frac{(T_{ss} e^{\frac{-t}{\tau_{on}}})}{\tau_{on}}$$

$$T' = \frac{(T_{ss} - T_t)}{\tau_{on}} \quad (3.4)$$

Where:

T_{ss} = Steady state temperature (°C)

T_t = Engine temperature (°C)

T' = First derivative temperature (°C)

τ_{on} = Engine on time constant

t = time

In order to calculate the temperature of the engine as it cools down from its operating temperature equation (3.5) is used. The equation is derived from an exponential first order lag curve. Examination of the recorded cool down data for the engine presented in Figures 3.15 and 3.16 show that an exponential decay curve is well fitted to the data. It should be noted that T_{ss} is the starting temperature for the cooldown and the final temperature will be equal to the surrounding air temperature.

$$T_t = T_{ss} e^{\frac{-t}{\tau_{off}}} + Ta$$

$$T_t - T_a = -T_{ss} e^{\frac{-t}{\tau_{off}}}$$

$$T' = \frac{-(T_t - Ta)}{\tau_{off}} \tag{3.5}$$

Where:

T_{ss} = Steady state temperature (°C)

T_t = Engine temperature (°C)

T' = First derivative temperature (°C)

τ_{off} = Engine on time constant

t = time

T_a = Air temperature (°C)

$$Q = \frac{Q_{ss}(T_t - T_a)}{(T_{ss} - T_a)} \tag{3.6}$$

$$Q = UA_{off}(T_t - T_a) \tag{3.7}$$

Equation (3.6) calculates the heat flux for the engine when it is running and equation (3.7) calculates it when the engine is switched off.

$$UA_{still} = \frac{C}{\tau_{still}} \tag{3.8}$$

$$UA_{off} = UA_{still}(h.V) \tag{3.9}$$

$$\tau_{off} = \frac{C}{UA_{off}} \tag{3.10}$$

Equation (3.8) calculates the overall heat transfer coefficient of the engine in still air and equation (3.9) the heat transfer coefficient in moving air. It is necessary to evaluate values for UA in this way as an actual value is unavailable, but a value of the capacitance can be estimated from knowing the engine's mass and specific heat capacities and being able to measure its time constant. The time constant for the engine when it is not running is calculated by equation (3.10). The value for τ_{off} is required for tests where the vehicle is not running but is stationed within a moving air stream. The measured value of τ_{still} is the time constant for the vehicle cooling down in still air.

The parameter h is the heat transfer coefficient for the vehicle in moving air. Its value is calculated as laid down in [16].

The vehicle is taken to be a non-circular object in cross flow of a gas. This geometry is valid for Reynolds numbers (Re) from 5000 to 100,000. Assuming the maximum value for Re of 100,000 a check on the corresponding flow velocity can be made using equation (3.11).

$$Re = \frac{U\infty.De}{\nu} \tag{3.11}$$

Where:

Re = Reynolds number

U∞ = Free stream velocity

D*e* = Characteristic dimension

ν = Kinematic viscosity

For this geometry the flow maximum velocity in which it is valid is 2.68 m/s. It should be noted that the maximum air velocity achieved in the CWT is 55.5 m/s and far exceeds this value. It is assumed that the air velocity over the engine is reduced to a level within the range of validity due to the air having to pass through the cars front grille, radiator and cooling fan cowling before it reaches the engine block.

The heat transfer coefficient can be found from equations (3.12 and 3.13):

$$Nu = C.\mathrm{Re}^{n} \tag{3.12}$$

$$Nu = \frac{h.De}{k} \tag{3.13}$$

Where:

Nu = Nusselt number

C = Constant

n = Constant

Re = Reynolds number

h = Heat transfer coefficient

D*e* = Characteristic dimension

k = Thermal conductivity

3.2.1 Model time constants (τ)

When any quantity varies exponentially with time, the time required for a fractional change of amplitude is equal to:

$$100\left(1-\frac{1}{e}\right)=63.2\% \tag{3.14}$$

Where e is the exponential constant (the base of natural logarithms) [11].

The vehicle model requires two time constants (τ) one for when it is running (τ_{on}) and one for when the engine is switched off (τ_{off}).

In order to establish these two time constants, measurements of oil temperature from and engine of 1272cc were made. The temperature of the oil was recorded as opposed to the water, as it recognised within the vehicle testing industry that the oil temperature gives a far truer reflection of the overall engine temperature [12].

To carry out the required temperature measurements a 4 channel portable data logger was designed and constructed. The logger allowed the interval between measurements to be altered in one second intervals and was capable of recording upto 8640 separate events, one channel logging for 24 hours at 10 second intervals. The oil temperature was recorded via a K-type thermocouple fitted to a dipstick. This allowed the bulk temperature of the oil in the sump to be recorded. The data was recorded by driving the vehicle on the road and therefore only limited control over engine load and speed could be exercised

Figure 3.13 shows the resulting temperature profile for the engine oil temperature during warm up and running.

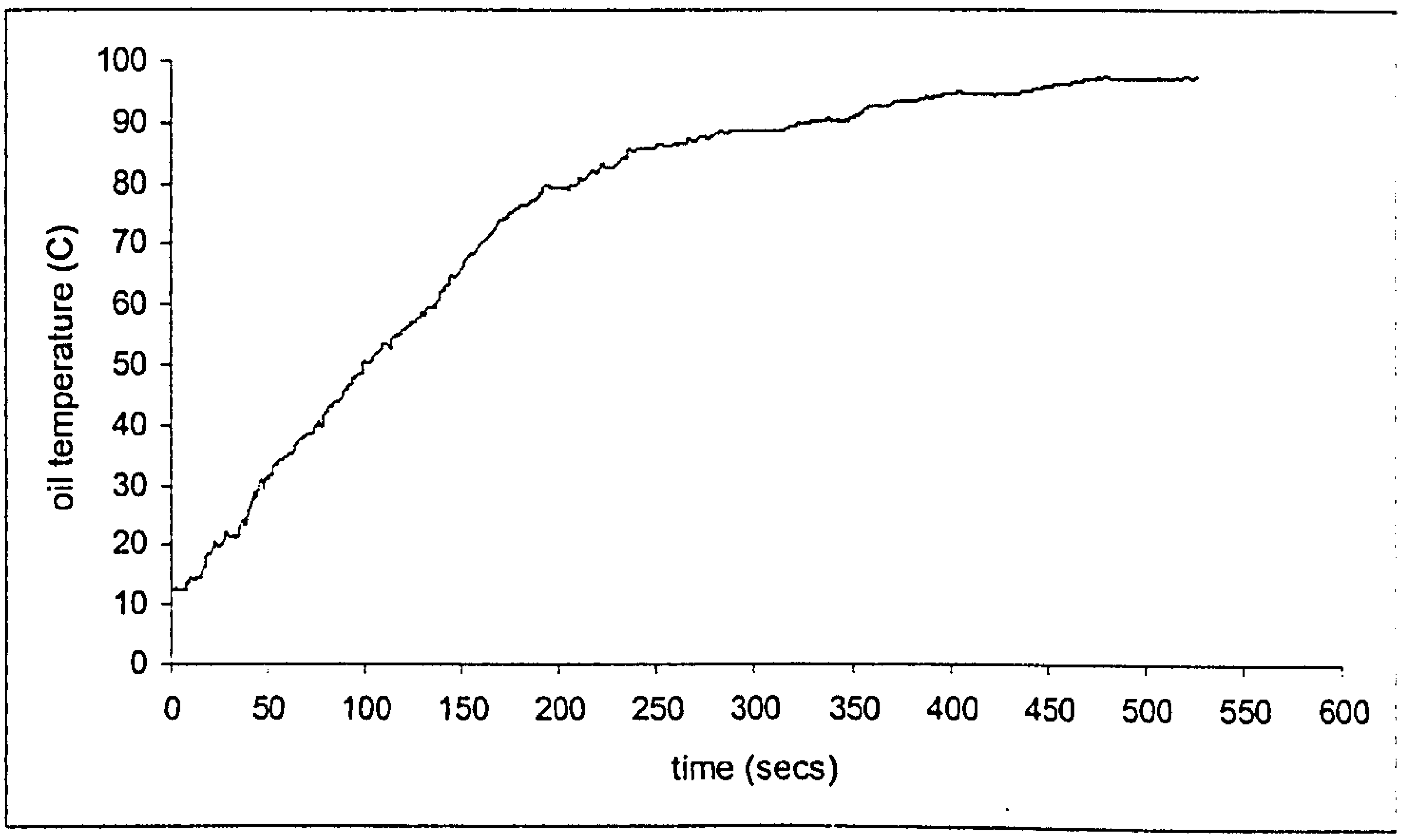


Figure 3.13 Engine oil temperature during warm up

The values recorded were normalised by means of:

$$Y' = \frac{(Y_{min} - Y)}{(Y_{min} - Y_{max})} \tag{3.15}$$

Where:

Y' = Normalised value

Ymin = Minimum temperature value

Ymax = Maximum temperature value

Y = Value to be normalised

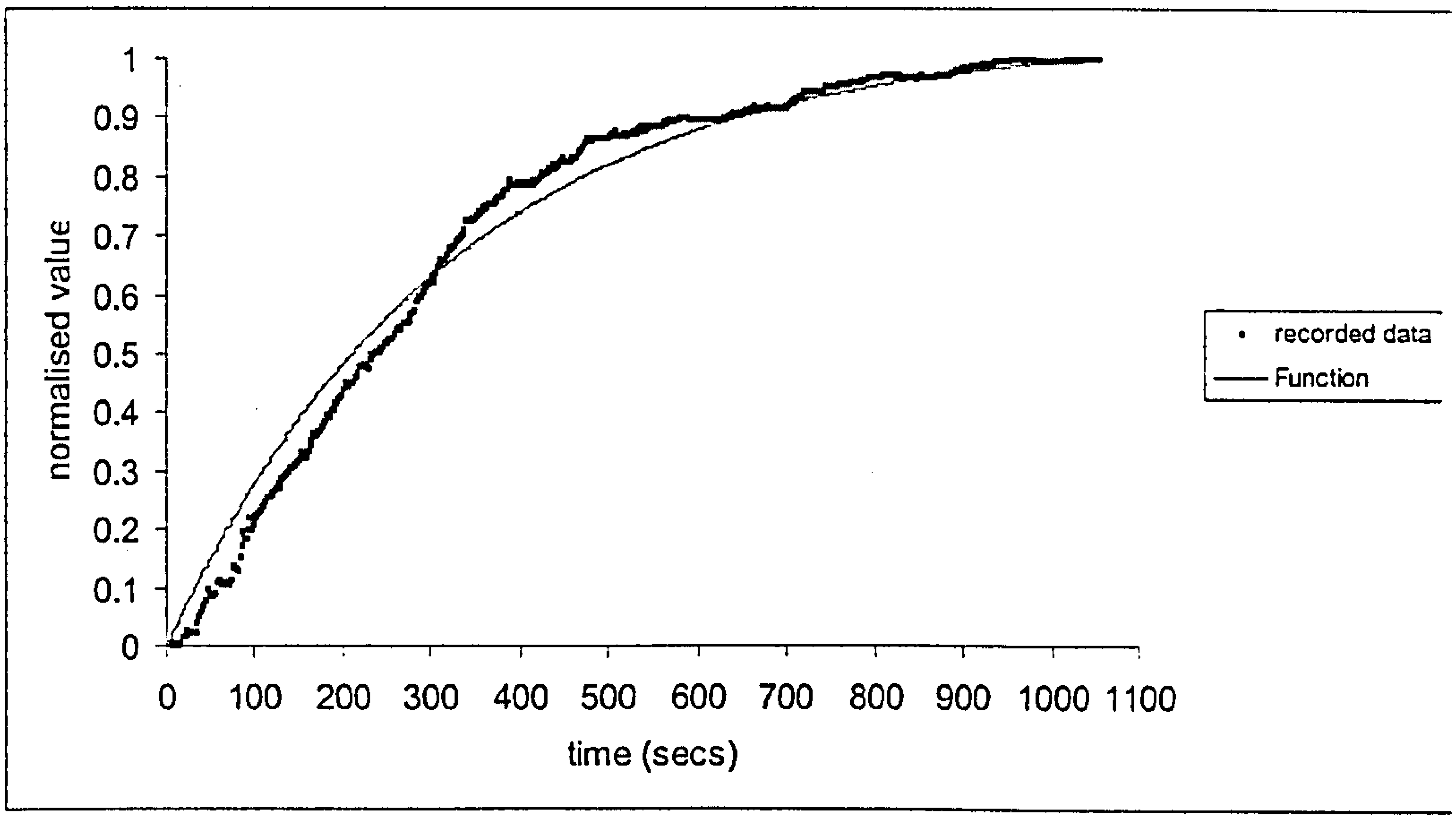


Figure 3.14 Curve fit to oil temperature data for running engine

A first order exponential curve was fitted to the data, this curve having the equation:

$$Y = \frac{1 - e^{\left(-3.29\left[\frac{x}{1050}\right]\right)}}{1 - e^{-3.29}} \tag{3.16}$$

Figure 3.14 shows the curve fit for the data and the resultant value for the time constant τ. The value for τ_{on} is 303 seconds.

Whilst it is appreciated that the time constant during warm-up will vary depending upon driving conditions imposed upon the engine and hence ideally the model should have time constants to reflect these conditions. Unfortunately due to the only method available for recording the engine oil temperature was under normal stop-start driving conditions. Under such circumstances it was impossible to maintain a constant engine load and hence only one engine time constant has resulted.

A similar process was carried out to establish the time constant for the engine when turned off (τ_{off}). Figure 3.15 shows the temperature data recorded for the vehicle during cool down.

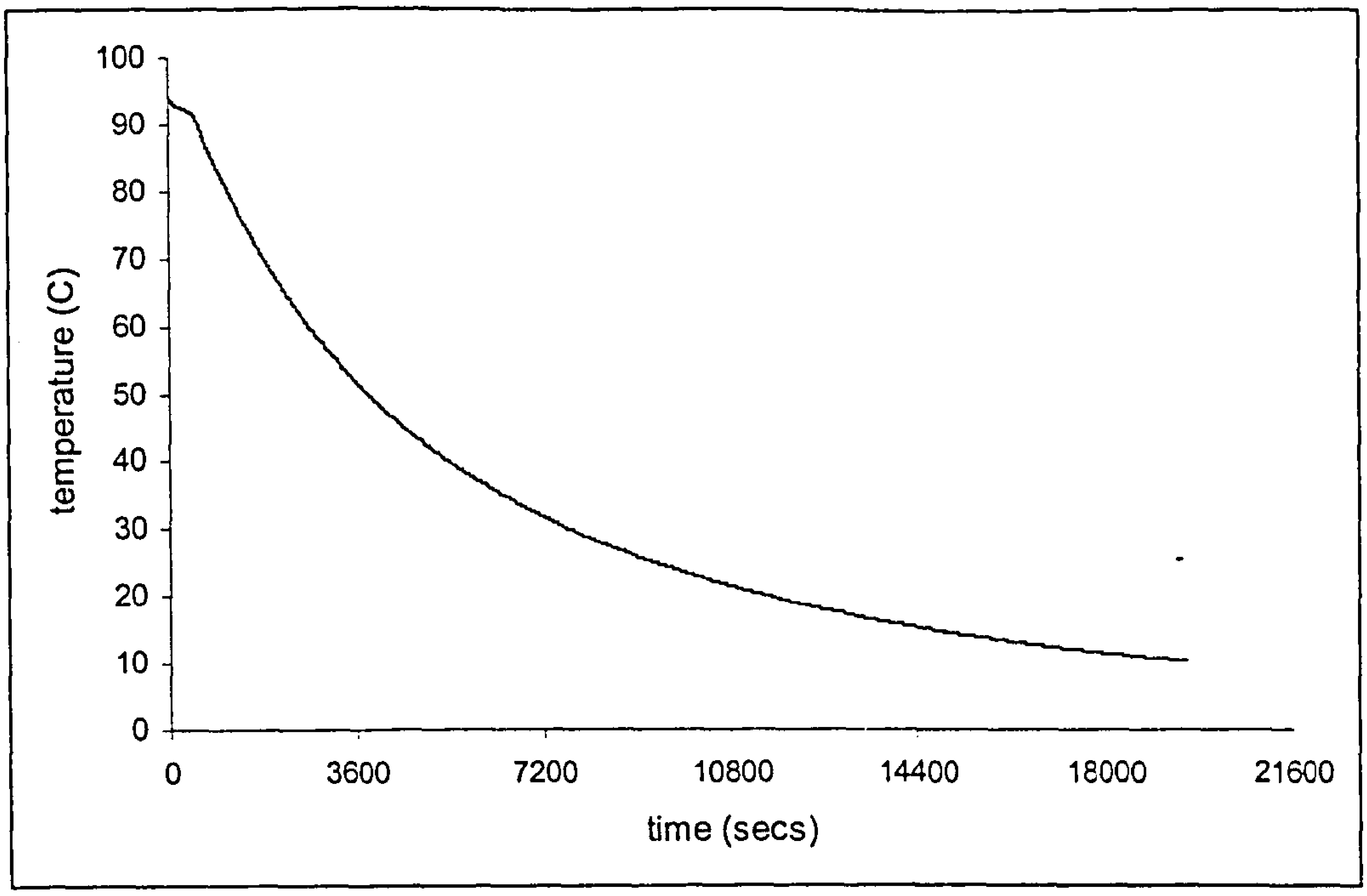


Figure 3.15 Oil temperature data for engine cool down

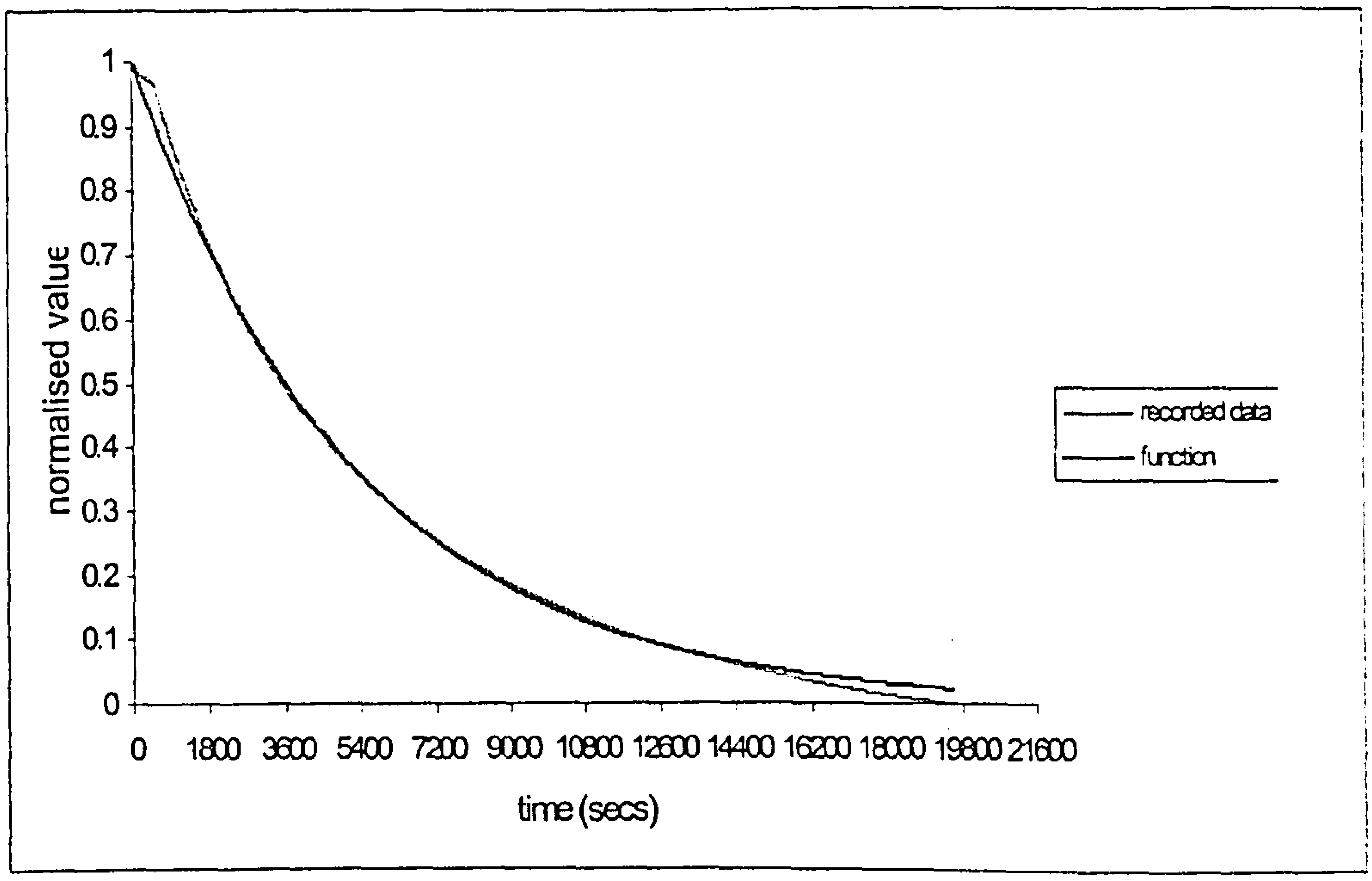


Figure 3.16 Curve fit to oil temperature data for engine cool down

A first order exponential decay curve was fitted to the data, this curve having the equation:

$$Y = \frac{1}{e^{(x.0.000192)}} \qquad (3.17)$$

Figure 3.16 shows the curve fit for the data and the resultant value for the time constant τ. The value for τ_{off} is 6805 seconds.

3.2.2 Exhaust temperature

The model requires an exhaust temperature. In reality this temperature will fluctuate continuously with engine load and along the length of the exhaust pipe.

Figure 3.17 shows the temperature fluctuations recorded at the exhaust tail pipe for a short journey lasting 15 minutes (860 seconds).

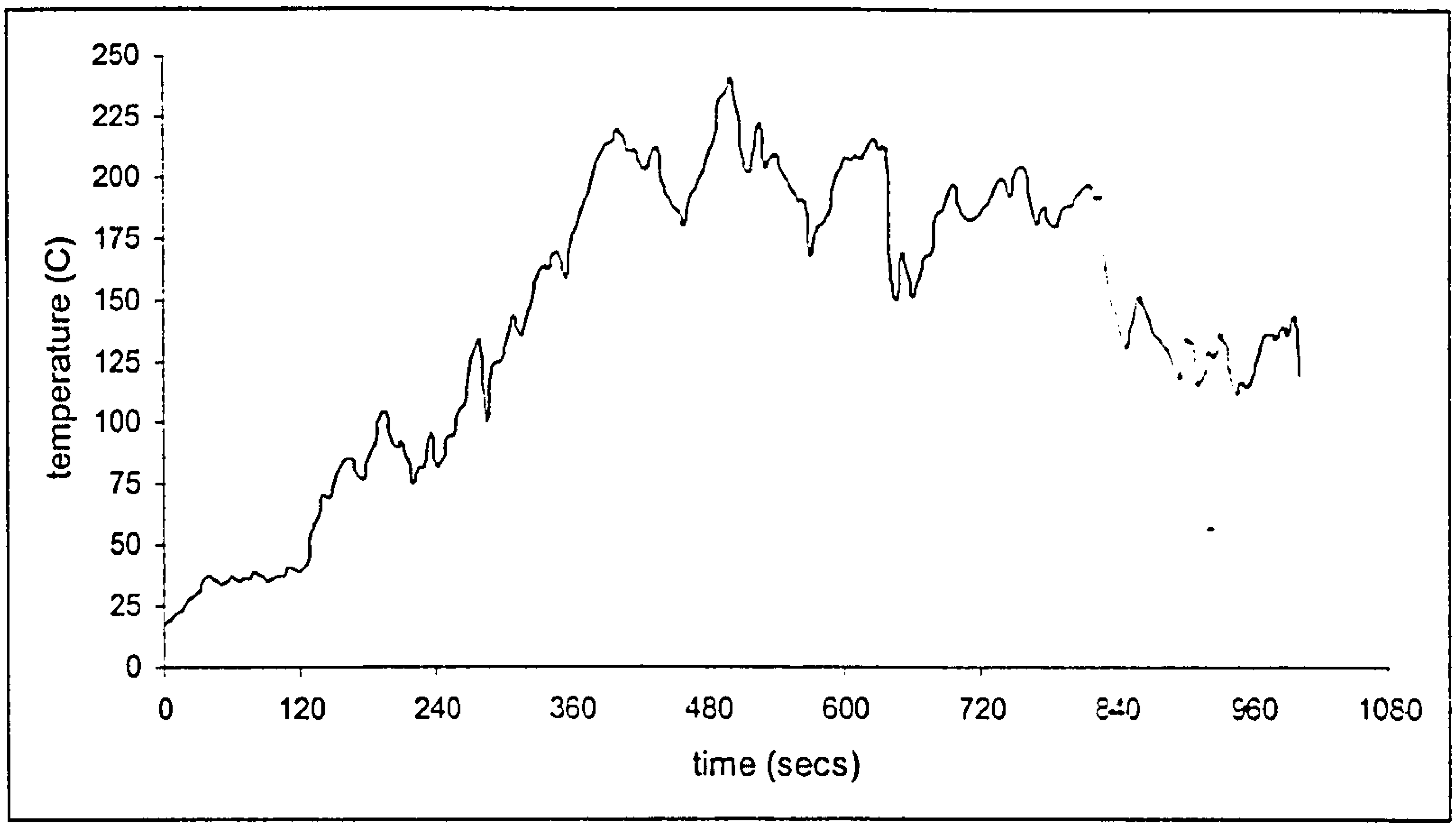


Figure 3.17 Exhaust temperature

As it would be near impossible to establish and representative temperature for differing driving loads and conditions an average temperature of 175°C was taken from the data shown in Figure 3.17. This temperature was arrived at from averaging only the temperatures between when the engine had reached its maximum operating temperature and the time the journey ceased.

3.3 Refrigeration system model development

The very large cooling loads (1.5 MW) occurring within the Climatic Wind Tunnel are catered for by a central refrigeration system.

The refrigeration system consists of two stage twin screw compressors with interstage cooling a shell and tube evaporator and air-cooled condenser.

3.3.1 Compressor model development

To develop a fundamental model of a compressor based upon established theoretical principals was not feasible in the time constraints of the project. An empirical model has been developed based upon curve fitting to manufacturers' data.

It has been shown in previous work [17, 18] that a second order polynomial curve fit. (equation 3.18) to manufacturers data gives a good representation of the compressors performance.

Manufacturers' data is often in doubt due to unrealistic operating conditions used in obtaining the data and the removal of any spurious data points prior to publication [13].

The data used to produce the curve fit was obtained from a computer based selection program [14]. The program allows the selection of multi-stage refrigeration plant with a correctly sized intercooler allowed for in the calculations. The exact refrigeration compressors and intercooler were selected from the database as well as the appropriate intercooling device. The program calculates the refrigerant properties and the resulting cooling capacities and power consumption from first principles.

The data for the compressors over a range of operating conditions was taken. These conditions were evaporating temperatures of –60°C to +10°C and condensing temperatures of 0°C to + 50°C. To this data second order curve fits of the form shown in equation 3.18 were made.

$$Y = (a_0) + (a_1 T_e) + (a_2 T_e^2) + (a_3 T_c) + (a_4 T_c T_e) + (a_5 T_c T_e^2) + (a_6 T_c^2) + (a_7 T_c^2 T_e) + (a_8 T_c^2 T_e^2) \qquad (3.18)$$

Where a_o to a_n are constants and the degree of the equation is the highest power of x. The curve fit to the data resulted in the following coefficients:

a_0	0.1166e03
a_1	0.2399e01
a_2	0.2917e-01
a_3	-0.2709e01
a_4	-0.2993
a_5	-0.4148e-02
a_6	0.1064
a_7	0.5015e-02
a_8	0.6396e-04

Table 3.4 Second order curve fit coefficients for high stage compressor

a_0	0.1667e03
a_1	0.2656e01
a_2	0.6666e-02
a_3	0.3e01
a_4	0.6366e-01
a_5	0.2313-03
a_6	-0.1641e-01
a_7	-0.4052e-03
a_8	-0.6595e-06

Table 3.5 Second order curve fit coefficients for low stage compressor

a_0	0.2957e04
a_1	0.5884e02
a_2	0.242
a_3	0.2805e02
a_4	0.1505e01
a_5	0.1728e-01
a_6	-0.4517
a_7	-0.1839e-01
a_8	-0.1692e-03

Table 3.6 Second order curve fit coefficients for combined cooling capacity

The condenser heat rejection is given by:

$$Q_{cond} = Q_{cool} + Q_h + Q_l \tag{3.19}$$

The model parameters are:

Curve fit coefficients	Coeff	Dimensionless

The model in/ out variables are:

Condensing temperature	Tc	°C
Evaporating temperature	Te	°C
Control link	X	Dimensionless
Cooling capacity	Q_{cool}	W
Condenser heat rejection	Q_{cond}	W
High stage power consumption	Q_h	W
Low stage power consumption	Q_l	W

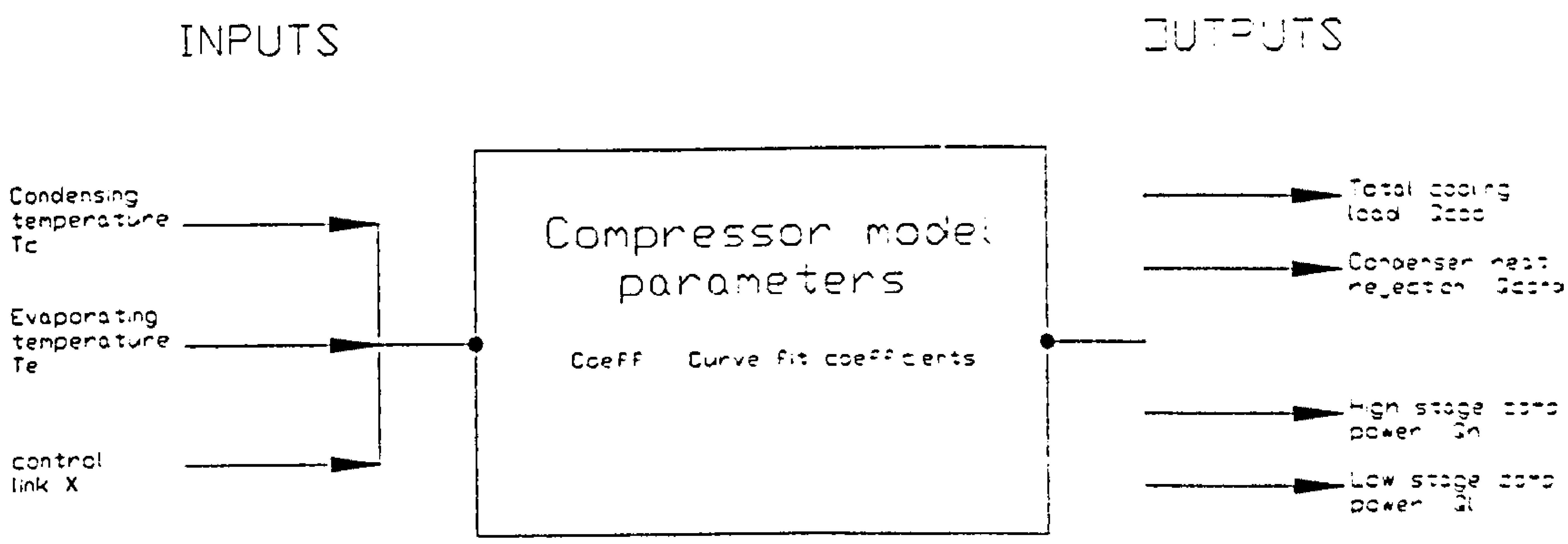


Figure 3.18 Information flow diagram for compressor model

The polynomial equations formed from the curve fit coefficients are multiplied by a factor X. This factor has a value of between 0 and 1 and serves the same purpose as a slide valve in a screw compressor by regulating its capacity. The value for X comes from a signal generated by a proportional controller sensing the fluid outlet temperature from the chiller. As the fluid leaving the chiller gets closer to the controller setpoint the output signal reduces and inturn off loads the compressors.

3.3.2 Condenser and evaporator model development

Both the condenser and evaporator are represented as classical heat exchangers. In both cases pressure drops have been considered and the refrigerant is considered to be isothermal i.e. there is neither superheating nor subcooling of the refrigerant. The secondary heat transfer medium in the evaporator in this case is Trichloroethylene but the model allows for other fluids to be used. In the condenser air is the secondary medium.

3.3.2.1 Condenser model development

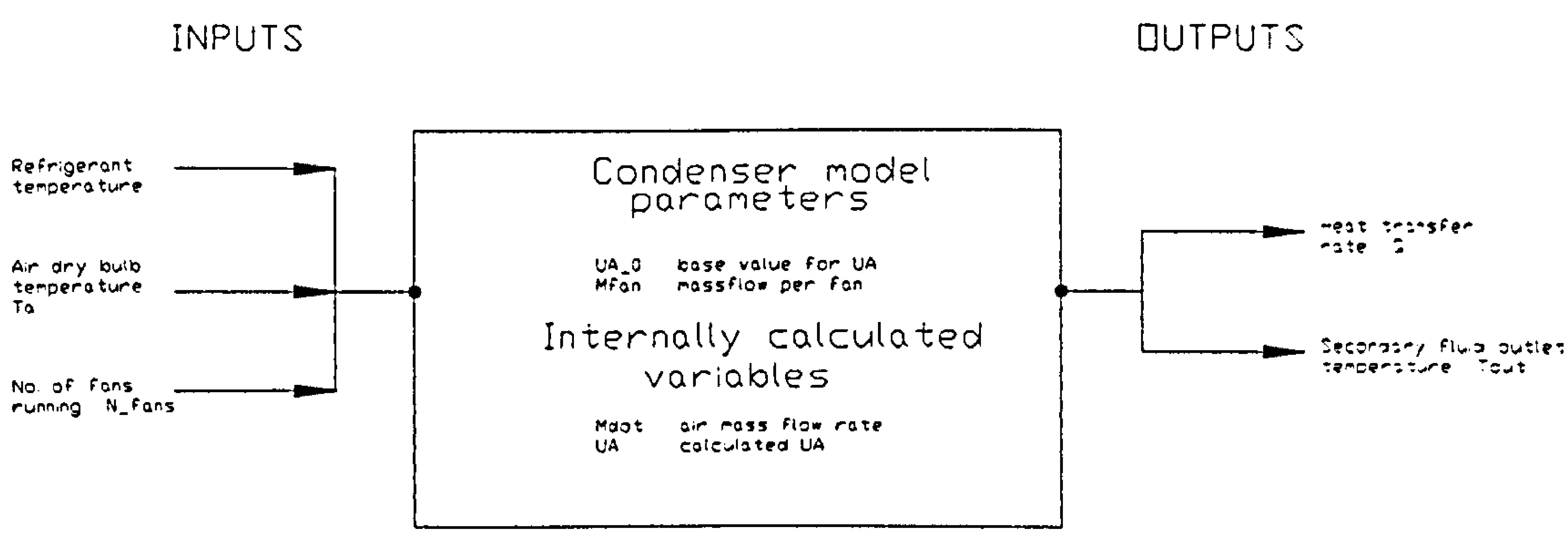


Figure 3.19 Information flow diagram for condenser model

The user definable parameters for the condenser are:

Base value for UA	UA_0	W/K
Mass flow rate per fan	M_{fan}	kg/s

The in/out variables for the model are:

Air mass flow rate	M_{dot}	kg/s
Refrigerant temperature	T_{frig}	°C
Air dry bulb temperature	T_o	°C
UA value	UA	W/K
No. of fans running	N_{fans}	Dimensionless
Heat transfer rate	Q	W
Air leaving temperature	T_{out}	°C

The condenser scales the value of UA in accordance to the number of fans running. Whilst it is appreciated that natural convection will exist when fans are not running. In order to reduce the model development time, in a first instance this has not been taken into consideration.

The equations for the condenser are:

$$M_{dot} = M_{fan} . N_{fans} \tag{3.20}$$

Equation (3.20) calculates the mass flow rate of air through the condenser. The model takes no account of natural convection occurring when no fans are running.

$$UA = UA_o . N_{fans} \tag{3.21}$$

Equation (3.21) adjusts the UA value according to the number of fans running.

$$Q = M_{dot}.CP_{air}.\Delta T \tag{3.22}$$

$$T_{out} = (T_{frig} - T_o).(1 - EXP.\left(\frac{-UA}{M_{dot}.CP_{air}}\right) + T_o \tag{3.23}$$

Equation (3.22) is an energy balance for the air and equation (3.23) is a rate equation based upon the effectiveness expression.

The condenser model requires two initial parameters to be identified, these being the base value for UA and the air mass flow rate.

The rating point for the condenser was obtained from available design data for the CWT [19] and is shown in Table 3.7. Using the relationships of equations (3.24 and 3.25) a value of UA can be found [15].

Load (kW)	1139	522
Outside wet bulb temp (°C)	19	19
Outside dry bulb temp (°C)	26	26
Condensing temp (°C)	38.5	38.5

Table 3.7 Design data of air-cooled condenser

$$\varepsilon = 1 - e^{-ntu} \tag{3.24}$$

$$ntu = \frac{UA}{C_{\min}} \tag{3.25}$$

Where:

Ntu = number transfer units

ε = effectiveness

Cmin = minimum heat capacity of fluids in heat exchanger

The volumetric flow rate ($\dot{Q}$) of air across the condenser is identified by equation (3.26):

$$\dot{Q} = \left(\frac{\Pi . D^2}{4}\right) V \tag{3.26}$$

Where:

D = Diameter of fan (m)

V = Face velocity of fan (m/s)

$\dot{Q}$ = Volumetric flow rate (m^3/s)

From the volumetric flow rate the mass flow rate is calculated by equation (3.27) for the parameter M_{fan}.

$$M_{fan} = \dot{Q} / v \qquad (3.27)$$

Where:

M_{fan} = Mass flow through fan (kg/s)

$\dot{Q}$ = Volumetric flow rate (m^3/s)

v = Specific volume (m^3/kg)

3.3.2.2 Evaporator model development

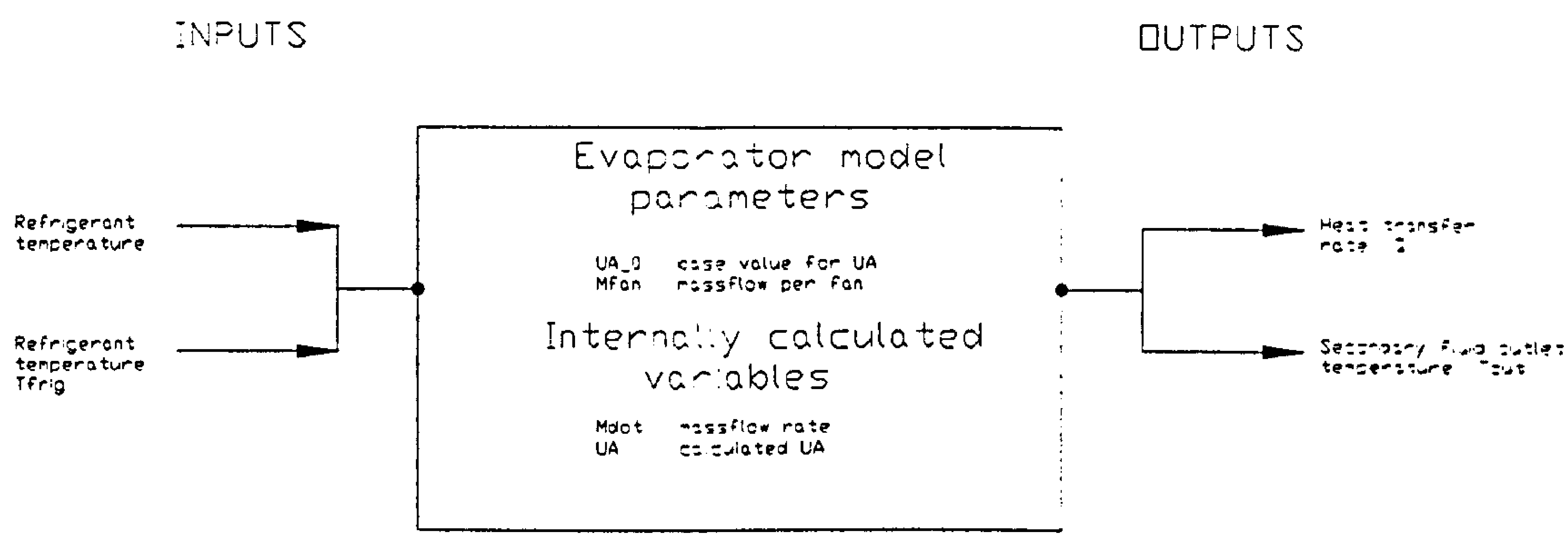


Figure 3.20 Information flow diagram for evaporator

The user definable parameters for the evaporator are:

Base value for UA	UA_0	W/K
Temperature gradient of UA	K	°C
Specific heat capacity	Cp_{liq}	J/kgK

The in/out variables for the model are:

Secondary fluid mass flow rate	M_{dot}	kg/s
Refrigerant temperature	T_{frig}	°C
Secondary fluid inlet temperature	T_{in}	°C
UA value	UA	W/K
Secondary outlet temperature	T_{out}	°C
Heat transfer rate	Q	W

Equations for the evaporator:

$$UA = UA_o + (K.T_{frig}) \tag{3.28}$$

Equation (3.28) calculates the overall heat transfer coefficient for the evaporator with the change in refrigerant temperature.

$$Q = M_{dot}.CP_{liq}.\Delta T \tag{3.29}$$

$$T_{out} = (T_{in} - T_{frig}).(1 - EXP.\left(\frac{-UA}{(M_{dot}.CP_{liq})}\right) - T_{in} \tag{3.30}$$

Equation (3.29) is a heat balance for the secondary fluid and equation (3.30) is a rate equation based on the effectiveness expression.

Equation (3.28) adjusts the UA value as the refrigerant temperature changes; this is by means of a parameter K.

K is calculated from two known design conditions for the evaporator [19], this data is shown in Table 3.8. The UA for each of these conditions can be calculated using equations (3.24) and (3.25). As only the two points were known a straight line evaluation of these points was carried out in order to assess a value for the parameter K, Figure 3.21 shows the evaluation.

Load (kW)	791	283
Flow rate (l/s)	160	160
Inlet temp (°C)	-12.2	-44.3
Outlet temp (°C)	-15.8	-45.5
Velocity (m/s)	77	81
Evaporator Temp (°C)	-20	-48.4

Table 3.8 Design data for evaporator

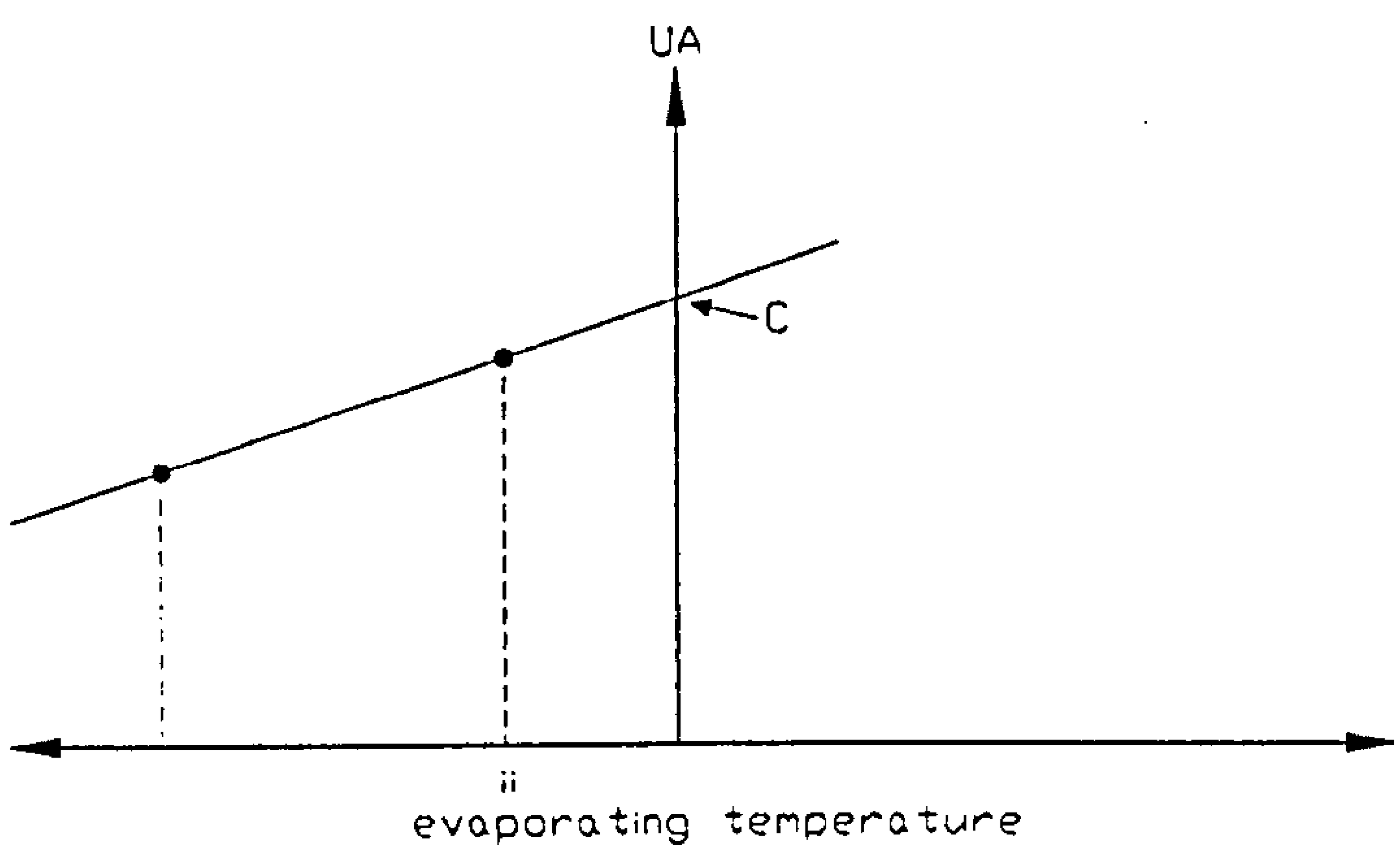


Figure 3.21: Straight line evaluation for parameter K

3.4 Air-liquid coil models

The two air-liquid heating/cooling coil models used for the CWT systems simulation are DRYCOIL and CCSIM from the ASHRAE NMF toolkit [3].

3.4.1 DRYCOIL

The DRYCOIL is a model of a sensible air-liquid heat exchanger. The model can be specified for use in a number of parallel, counter and crossflow configurations. The model requires the specification of a value for an overall heat transfer coefficient (UA). The model does not model any dehumidification of the air-stream nor any accumulation of frost and ice on the heat exchanger surface. The DRYCOIL model is to be used in the soakroom and test chamber sub-system models as neither systems require any dehumidification of their respective air-streams.

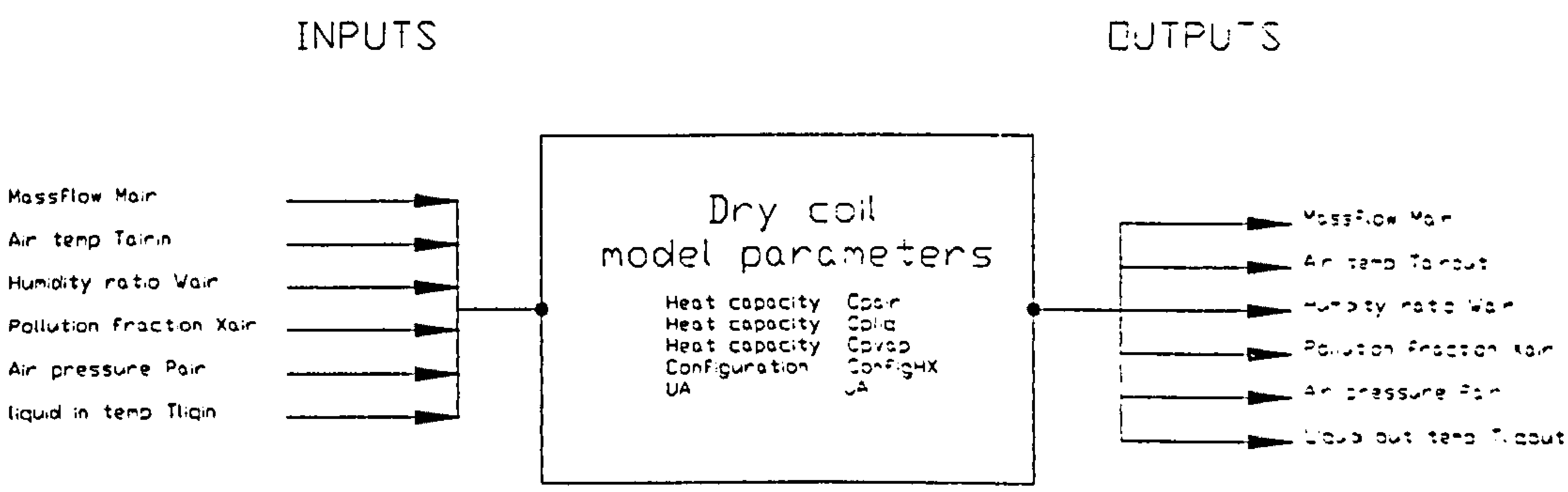


Figure 3.22 Information flow diagram for DRYCOIL model

Load (kW)	15	15
Airflow rate (m^3/h)	17000	17000
Air on temp (°C)	0	-40
Air off temp (°C)	-2.45	-42.1
Liquid flow rate (l/s)	14	14
Liquid in temp (°C)	-7.6	-45.3
Liquid out temp (°C)	-6.8	-44.5

Table 3.9 Design data for soakroom heat exchanger

Load (kW)	650	111
Airflow rate (m^3/h)	600,000	80,000
Air on temp (°C)	3.01	-36.7
Air off temp (°C)	0	-40
Liquid flow rate (l/s)	100	100
Liquid in temp (°C)	-10	-41.8
Liquid out temp (°C)	-5.2	-41

Table 3.10 Design data for test chamber heat exchanger

The drycoil model requires a calculated value of UA to be input as a parameter. This can be obtained from the above design data using the following:

$$LMTD = \frac{\Delta T_2 - \Delta T_1}{\ln\left(\frac{\Delta T_2}{\Delta T_1}\right)} \tag{3.31}$$

$$Q = UA\Delta T \tag{3.32}$$

Where:

LMTD = Log Mean Temperature Difference

ΔT_1 = temp hot fluid in – temp cold fluid out

ΔT_2 = temp hot fluid out – temp cold fluid in

3.4.2 CCSIM

CCSIM is a model that calculates the performance of a coil model when its extended fin surface is operating at one of three conditions: all wet, partially wet or completely dry. The coil can be used in a number of counter, parallel and crossflow configurations but its general application is for counter flow applications. The model requires the input of the rating data at which the coil design is based upon, from this the models initial conditions are

calculated. CCSIM model is used in the air make-up sub-system model as this system needs to dehumidify the air brought in from outside before it is supplied to the test chamber to make up for air used by the vehicle under test.

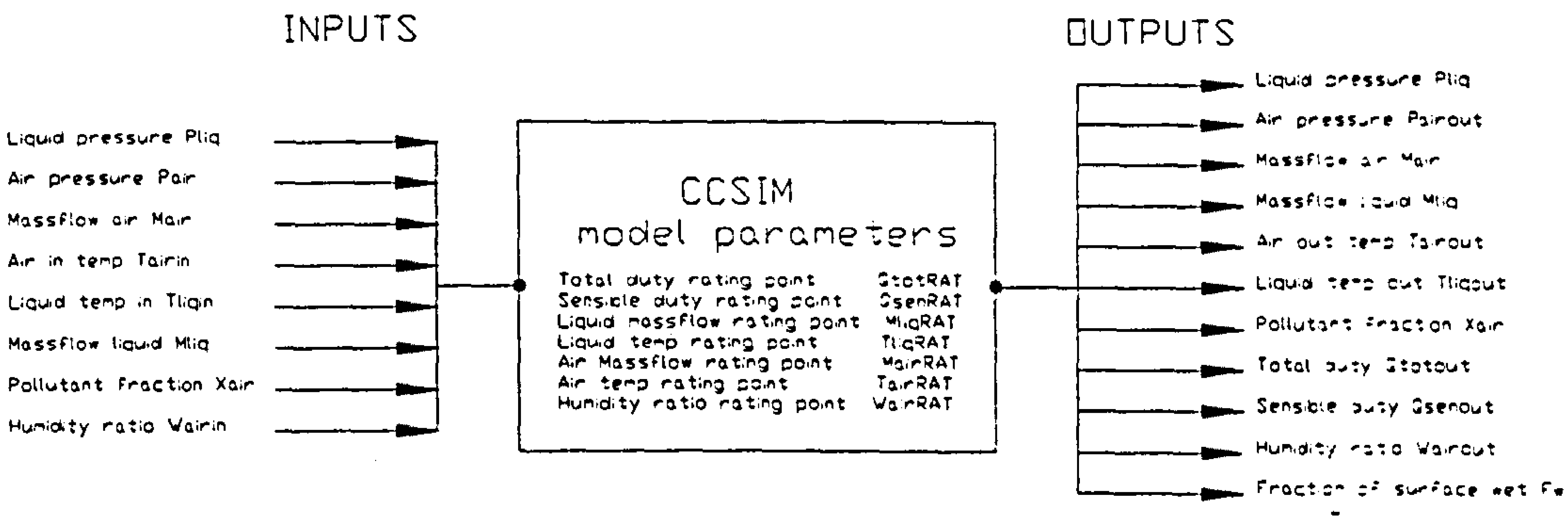


Figure 3.23 Information flow diagram for CCSIM

	High temperature heat exchanger	Low temperature heat exchanger
Load (kW)	23.3	31
Air flow rate (m^3/h)	1700	1700
Air on dry bulb temperature (°C)	26	2
Air on wet bulb temperature (°C)	19	2
Air off dry bulb temperature (°C)	2	-40
Air off wet bulb temperature (°C)	2	-40
Liquid flow rate (l/s)	20.95	20.95
Liquid in temp (°C)	0	-44.5
Liquid out temp (°C)	0.8	-43.1

Table 3.10 Design data for air make-up heat exchangers

3.5 Other models used in simulation

A number of models representing Tees, fans and other associated plant have been modelled. Most of these are existing equation-based models that have been converted into an NMF format. As these models have been used many times and in general are relatively simple in their description apart from testing that the models behave in a manner that is expected from them no detailed validation has been conducted.

One notable exception is the humidifier model that has been taken from the ASHRAE Secondary NMF toolkit [22]. The behaviour of this model was tested but as no validation tests made any use of the CWT's humidification capacity no data was available for empirical validation (Chapter 5). As no test required humidification the humidifier was not used in any of the scenarios modelled.

3.6 References

[1] Sahlin P (1996). NMF Handbook. An Introduction to the NMF. HTTP//: www.brisdata.se/. BrisData AB, KTH, Stockholm, Sweden.

[2] Sahlin P, Sowell E.F, (1989). A neutral format for building simulation models. Proceedings IBPSA Building Simulation 1989, Vancouver, Canada.

[3] ASHRAE Toolkit. ASHRAE NMF Toolkit. HTTP//: www.brisdata.se/. BrisData AB, KTH, Stockholm, Sweden.

[4] SIMONE: HTTP//: www.brisdata.se/. BrisData AB, KTH, Stockholm, Sweden.

[5] Murray M.A.P. (1984). Component based performance simulation of HVAC systems. PhD thesis. Loughborough University.

[6] Letherman K.M, Butler P (1980). A criterion for the accuracy of modelling of transient heat conduction in plane slabs. Building and Environment vol. 15 pg. 143 – 149.

[7] Incorpera F.P, DeWitt D.P (1990). Introduction to heat transfer. Wiley, New York.

[8] CIBSE (1988). CIBSE Guide Book A. Cartered Institution of Building Services Engineers, London.

[9] Davies M.G (1983). Optimum design of resistance and capacitance elements in modelling a sinusoidally excited building wall. Building and Environment vol. 18 pg. 19 – 37.

[10] Davies M.G (1983). A design procedure to model a transiently excited wall. Applied mathematical modelling vol. 7 pg. 182 – 188.

[11] Chambers Science and Technology Dictionary (1991). W & R Cambers, Edinburgh.

[12] Personal Communication D.W.Fletcher, Thermofluids Group, Motor Industry Research Association, Nuneaton, England.

[13] Wright J.A (1986). The optimised design of HVAC systems. PhD thesis, Loughborough University.

[14] Sabroe Matchmaster compressor program. Sabroe UK, Kings Norton, Birmingham. England.

[15] Incorpera F.P, Dewitt D.P (1996). Fundamentals of heat and mass transfer. 4th edition. Wiley, New York.

[16] Bayazitogiu Y, Ozisik M (1988) Elements of heat transfer. M^{c}Graw-Hill. ISBN: 0-07-100132-8.

[17] Yik F.W.H, Lam V.K.C (1998). Chiller models for plant design studies. Building Services Engineering Research and Technology. Vol. 19 N^{o}. 4 pg. 233 – 241.

[18] Shelton S.V, Weber E.D. (1991). Modelling and optimisation of commercial building chiller / cooling tower systems. ASHRAE Transactions, Vol. 97, part 2 pg. 1209 – 1216.

[19] DSMA Ltd.(1993). Engineering design specification for MIRA Climatic Wind Tunnel. DSMA Ltd. Vancouver, Canada.

[20] Hanby V.I, Roberts E.C, Fletcher D.W (1999). Modelling the dynamic thermal response of insulated ducts. Proceedings ISHVAC'99, Shenzhen. China.

[21] IDA Solver. BrisData AB, KTH, Stockholm, Sweden.

[22] ASHRAE NMF Secondary Toolkit. BrisData AB, KTH, Stockholm, Sweden.

Chapter 4

Plant Simulation

To carry out experimentation into the improvement of a system's performance is often impractical. This impracticality stems from the fact that any alteration of the systems operating conditions is likely to result in causing discomfort to the occupants or could seriously affect any process that the system was operating. If a model of the system was available, the model could be used to calculate how the system would have reacted to a certain change in its operating parameters. Simulation is thus an inexpensive and safe way in which to experiment with a system.

The development of a library of component models that make up the Climatic Wind Tunnels (CWT) thermal systems was detailed in chapter 3 and a number of existing Modular Simulation Environments (MSE's) suitable for the task of simulating these systems were reviewed in chapter 4. From these potential environments the IDA MSE was chosen to be the platform on which the simulation work was to be carried out.

This chapter describes the operation of the CWT and how the simulation is configured for the IDA environment.

4.1 Description of Climatic Wind Tunnel systems

The CWT consists of two temperature-controlled areas:

- Soakroom
- Test chamber

The soakroom is a thermally insulated space in which the vehicle can be pre-conditioned to any temperature within the range of -40°C / +55°C.

The soakroom is served by a fully recirculating closed loop air system. No extract or make-up air is required as the vehicle is not operational during its period in the soakroom. A typical vehicle pre-conditioning test is to soak the vehicle overnight (10 hours) to –30°C. This simulates the same conditions that the vehicle would be exposed to if it was left overnight in one of earth's arctic regions. From a test such as this a vehicle's cold starting ability can be assessed.

The Climatic Wind Tunnel (CWT) test chamber is where the active tests of the vehicle in motion are carried out. The CWT test chamber comprises of an air circuit, which produces a uniform, controlled flow of conditioned air in which the vehicles can be tested.

Movement of the vehicle is simulated by accelerating it on a dynamometer to the speed required for the test. The acceleration of the vehicle is carried out remotely from within the test chamber control room. Air is then passed over the vehicle at the corresponding speed; the combined effect simulates forward motion. The vehicle can be tested at any speed from 0 to 200 km/h. The 0 km/h test simulates the vehicle standing stationary in a city centre environment. During this test air is passed at high level across the chamber, thus removing heat as it rises from the vehicle.

In order to simulate the effect of the sun a solar simulation grid with a target area of 1.8m x 5.2m is installed above the dynamometer. This grid allows between 0.6 and 1.2 kW/m^2 of solar radiation to be simulated.

To simulate the vehicle travelling over hot roads an underbody heater is incorporated that allow the underside of the car to have its temperature raised to 30°C above the prevailing test conditions.

As with the soakroom the test chamber has an operating temperature range of between – 40 and +55°C. In addition to this it has humidity control that allows humidities upto 95% saturation to be achieved when the test air temperature is in the range of +5°C to +55°C.

The air make-up system is used to supply replacement air for that which is consumed by the vehicle, the exhaust gases are extracted and expelled to atmosphere.
The make-up air system brings the air in from outside filters it and then passes it through two heat exchangers. The temperature that this incoming air is cooled to is dependent upon the lowest temperature at which either the test chamber or soakroom is operating. The air is supplied at a constant mass flow rate of 5.2 kg/s into the test chamber air circuit. Atmospheric air pressure within the test chamber is maintained by means of pressure relief flaps within the structure. These flaps open when atmospheric pressure is exceeded.

A ring main of trichloroethylene (C_2Cl_3H) is used as the secondary heat transfer medium that couples the test chamber, soakroom and air make-up systems. Trichloroethylene is used as it has a freezing point of -80°C well below the CWT's lowest achievable flow temperature of -47°C.

The trichloroethylene is cooled by a two-stage refrigeration plant using twin-screw compressors and using R22 (monochlorodifluoromethane, $CHClF_2$)as the primary refrigerant. The refrigeration control system keeps the trichloroethylene at 15K below the lowest temperature required by the test chamber or soakroom. The 15K temperature difference is maintained until temperatures below -32°C are required. At this point the trichloroethylene will be supplied at its lowest possible temperature and the temperature difference between the secondary refrigerant and the air in the systems will be reduced, until at -40°C only a 7K difference exists.

Should either the test chamber or the soakroom be required to operate at higher temperatures, a 3-port valve is modulated so that flow from the main into the branch section becomes regulated. The trichloroethylene in this section is recirculated and passed through an electric heater, which heats it up to give the desired air conditions. Even when a high temperature test is in progress some cooling is required. This is due to the heat rejected by the vehicle under test being in excess of that required to maintain the desired tunnel condition.

The configuration of the soakroom and CWT test chamber allow them to operate independently of one another at different temperatures.

4.2 Definition of sub-systems

The CWT facility naturally divides up into two sub-systems, these being the test chamber and the soakroom. In addition to these the plant can be broken down into two further sub-systems, these being the air make-up and refrigeration systems.

A simplified schematic diagram of the CWT plant is shown in Figure 4.1 and detailed schematics of the test chamber, soakroom and air make-up systems are illustrated in Figures 4.2, 4.3 and 4.4 respectively. It should be noted that the schematics only contain plant items that are to be modelled. No pumps are included are pressure has not been modelled. Where fan models have been included, they are for heat gain to the air stream and contain no pressure terms.

The system schematics (Figures 4.1 to 4.4) contain the component models and the names of the variables that link them. Table 4.1 details the link types and the variables they contain.

Link name	Variable
UNIAIR	Massflow Temperature Humidity Pollutant fraction Pressure
M	Massflow
T	Temperature
X	Control signal
Sig_out	Control signal

Table 4.1 Link types and variable types

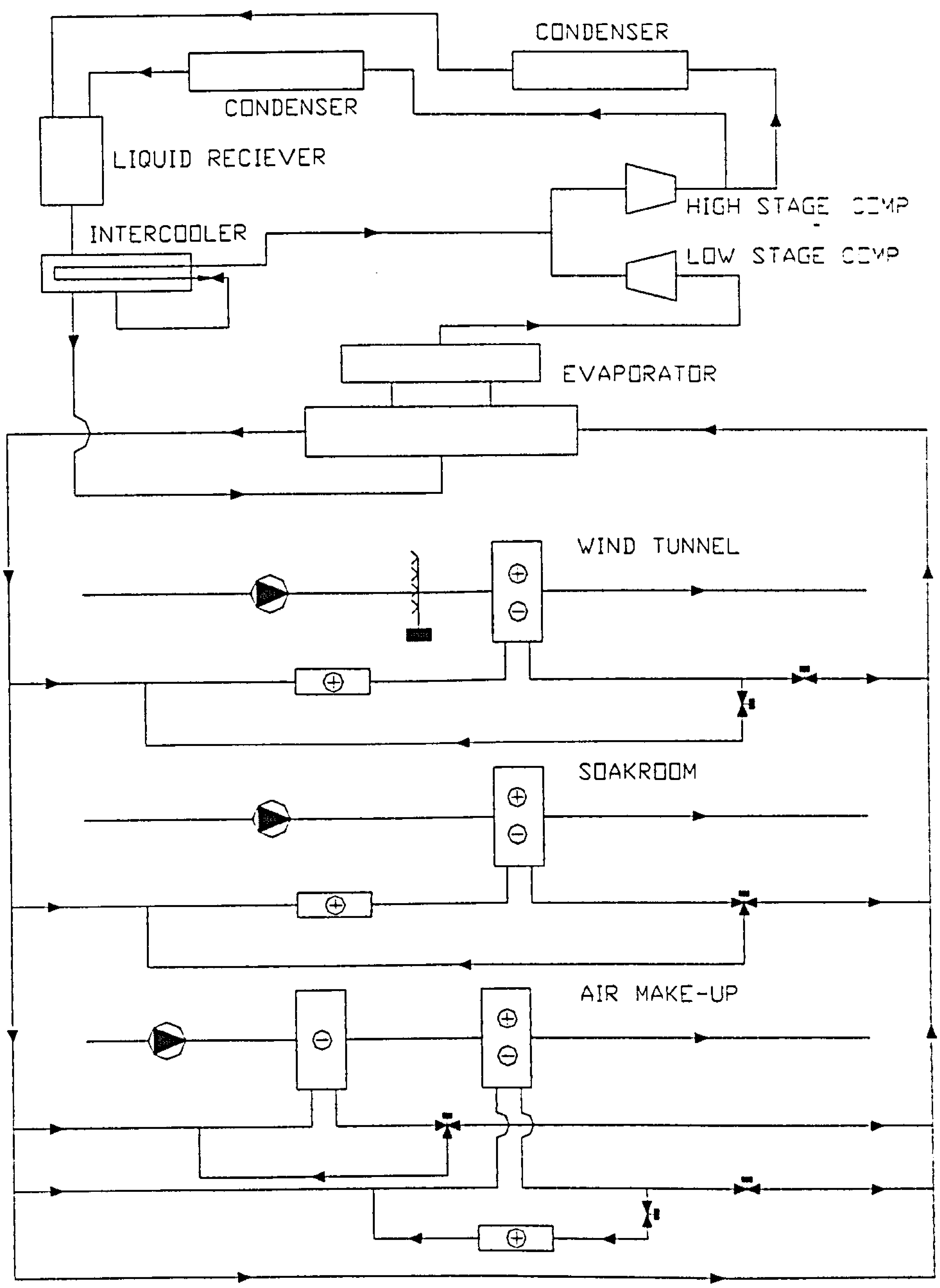


Figure 4.1 CWT plant schematic

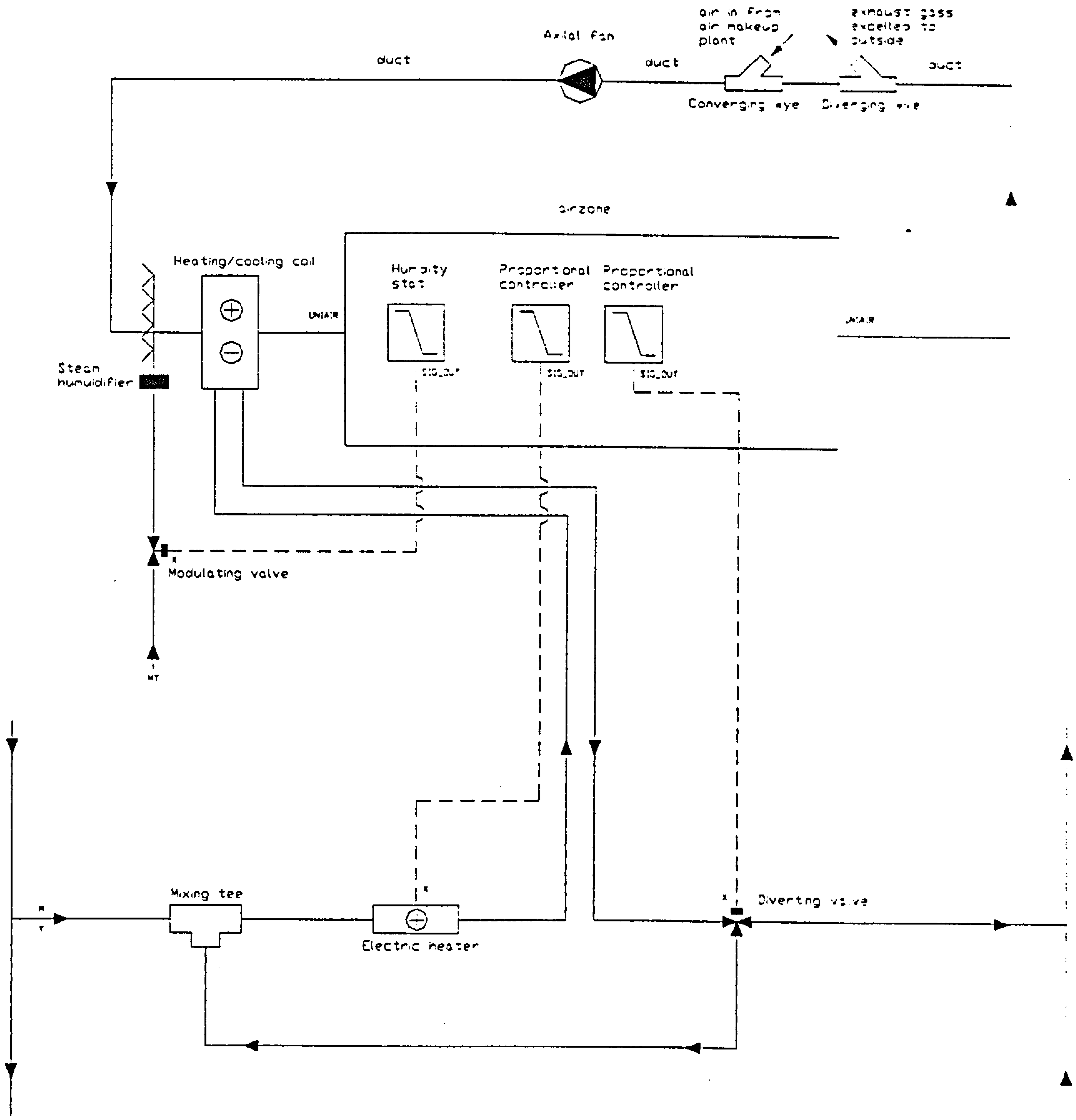


Figure 4.2 Test chamber plant schematic

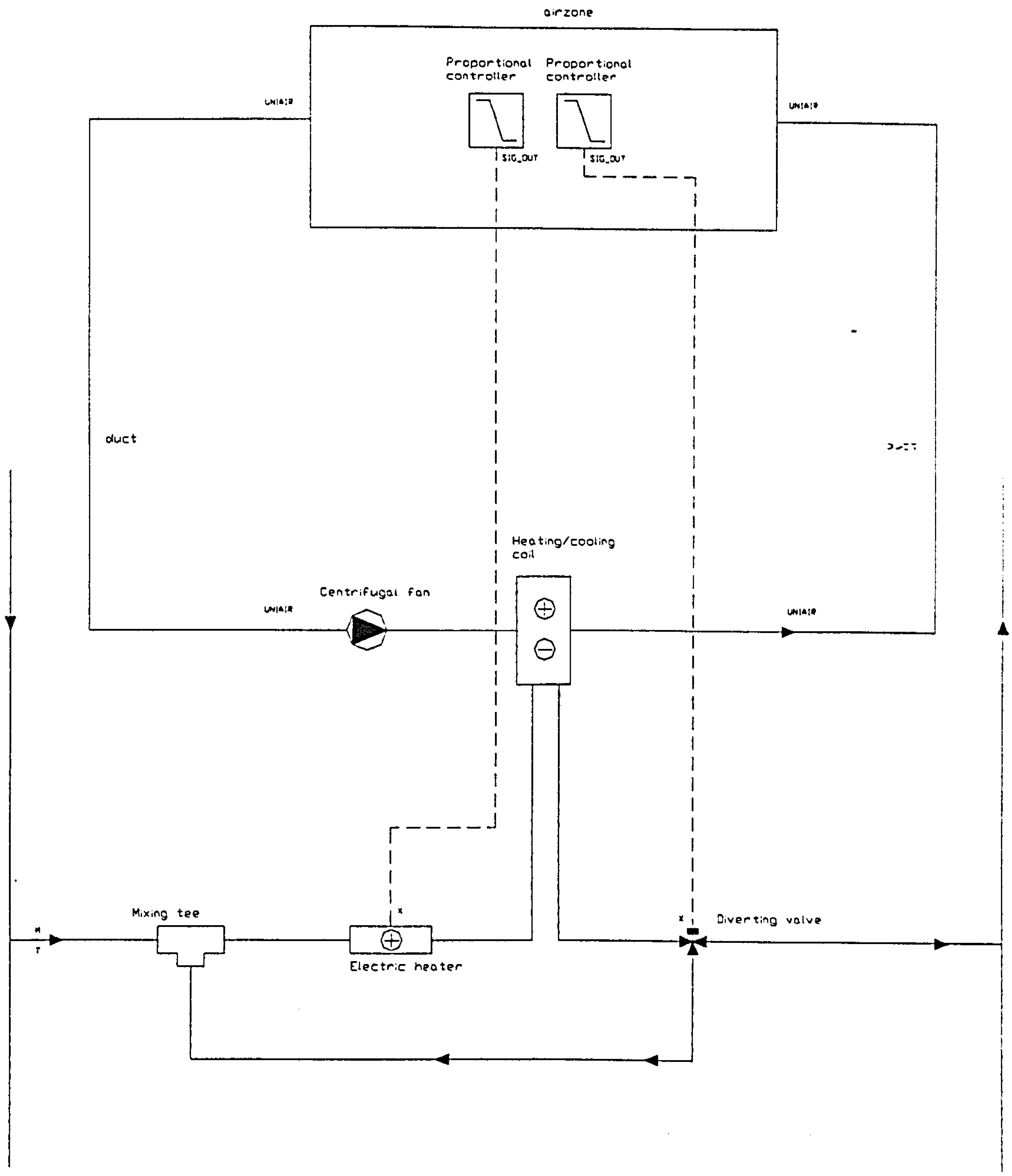


Figure 4.3 Soakroom plant schematic

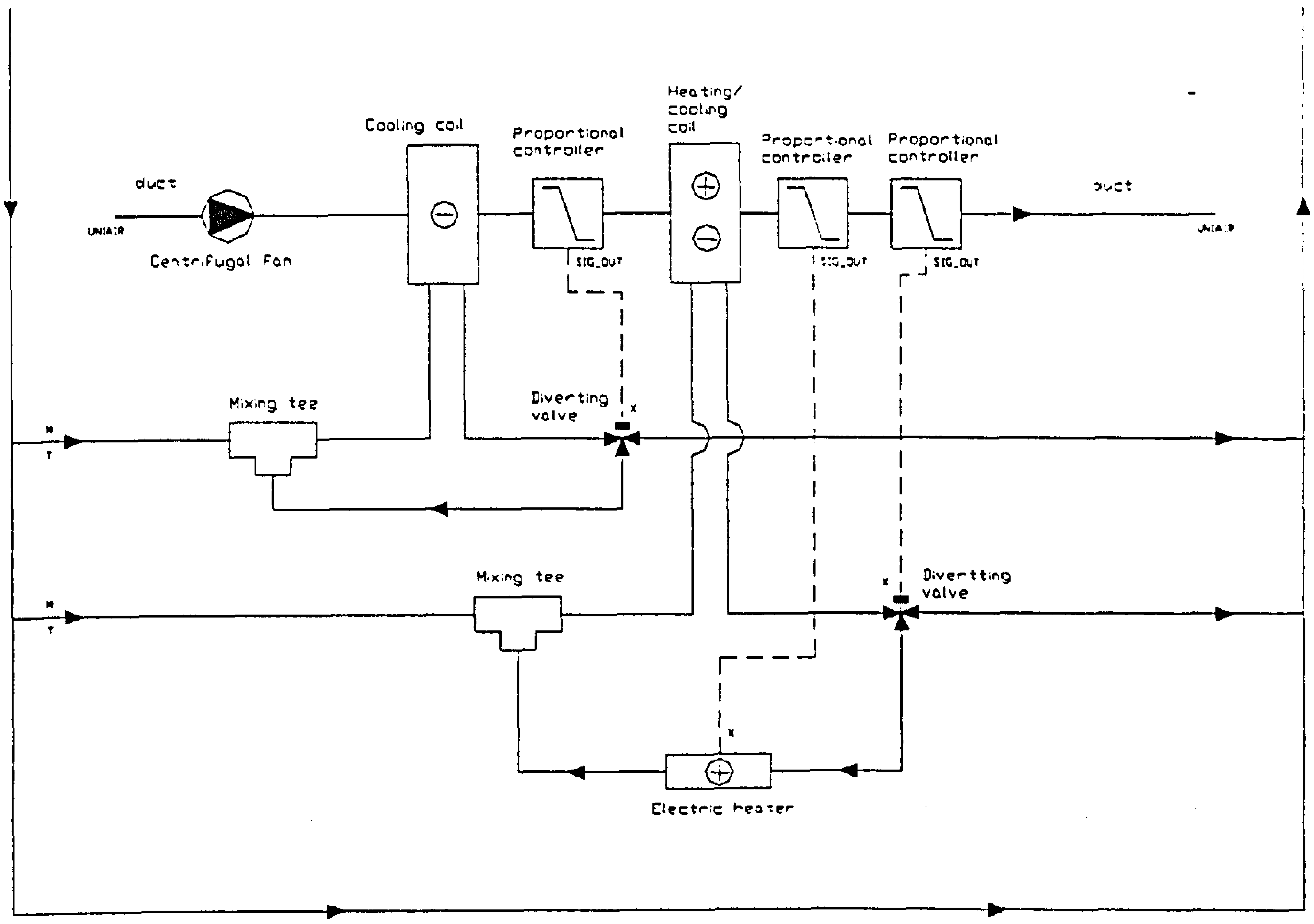


Figure 4.4 Air make-up plant schematic

4.3 Valve and heater control

The type of control used to modulate the electric heaters and three port valves within the CWT sub-systems is unknown. In the absence of such information idealised proportional control was used. Figure 4.5 shows the heater and valve configuration used in all CWT systems, schematics of the entire systems are shown in Figures 4.1 to 4.4.

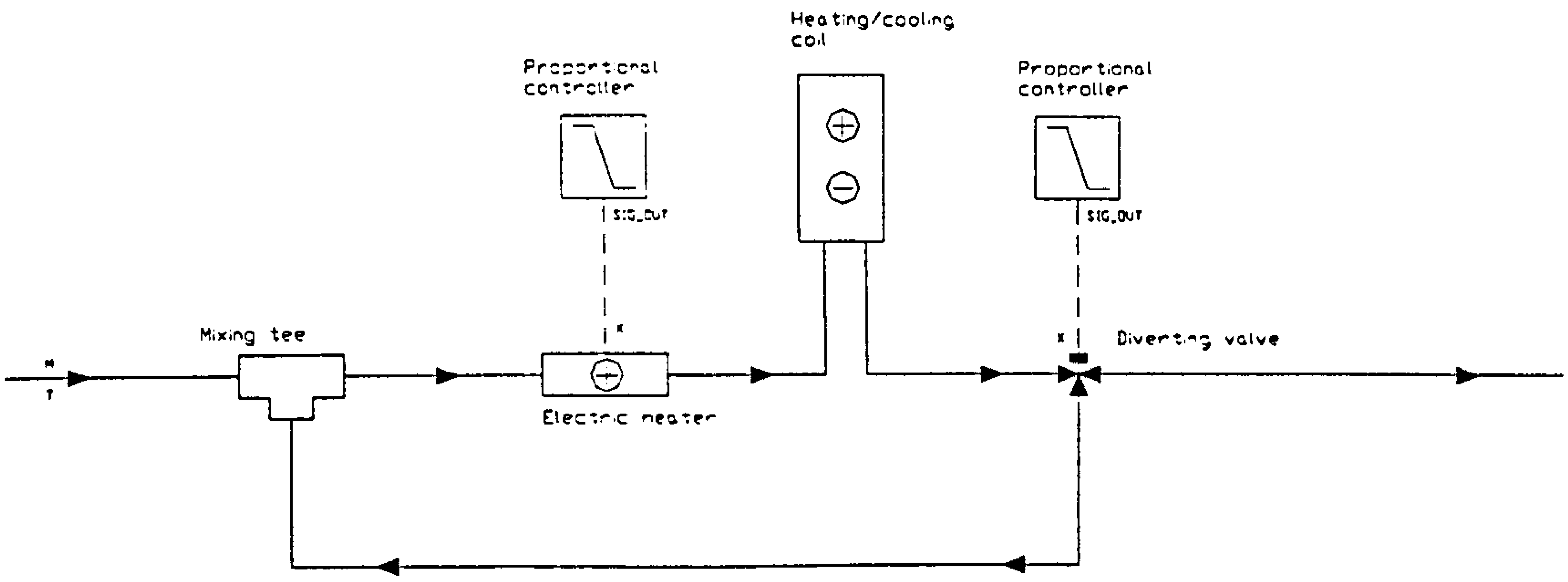


Figure 4.5 CWT three port valve and electric heater configuration

The valve and the heater need to be sequenced so that the heater would not come into operation until the three port valve was in 100% divert mode and conversely the valve would not modulate from full divert until the heater was inoperative. This relationship between the valve and the heater is so the heater is only used to heat the fluid in the re-circulating loop and does not heat the fluid in the main trichloroethylene loop. The sequencing was achieved by using two proportional controllers with separate set points and no overlap in the proportional band, an example schedule for a zone setpoint of 20°C is illustrated in Figure 4.6.

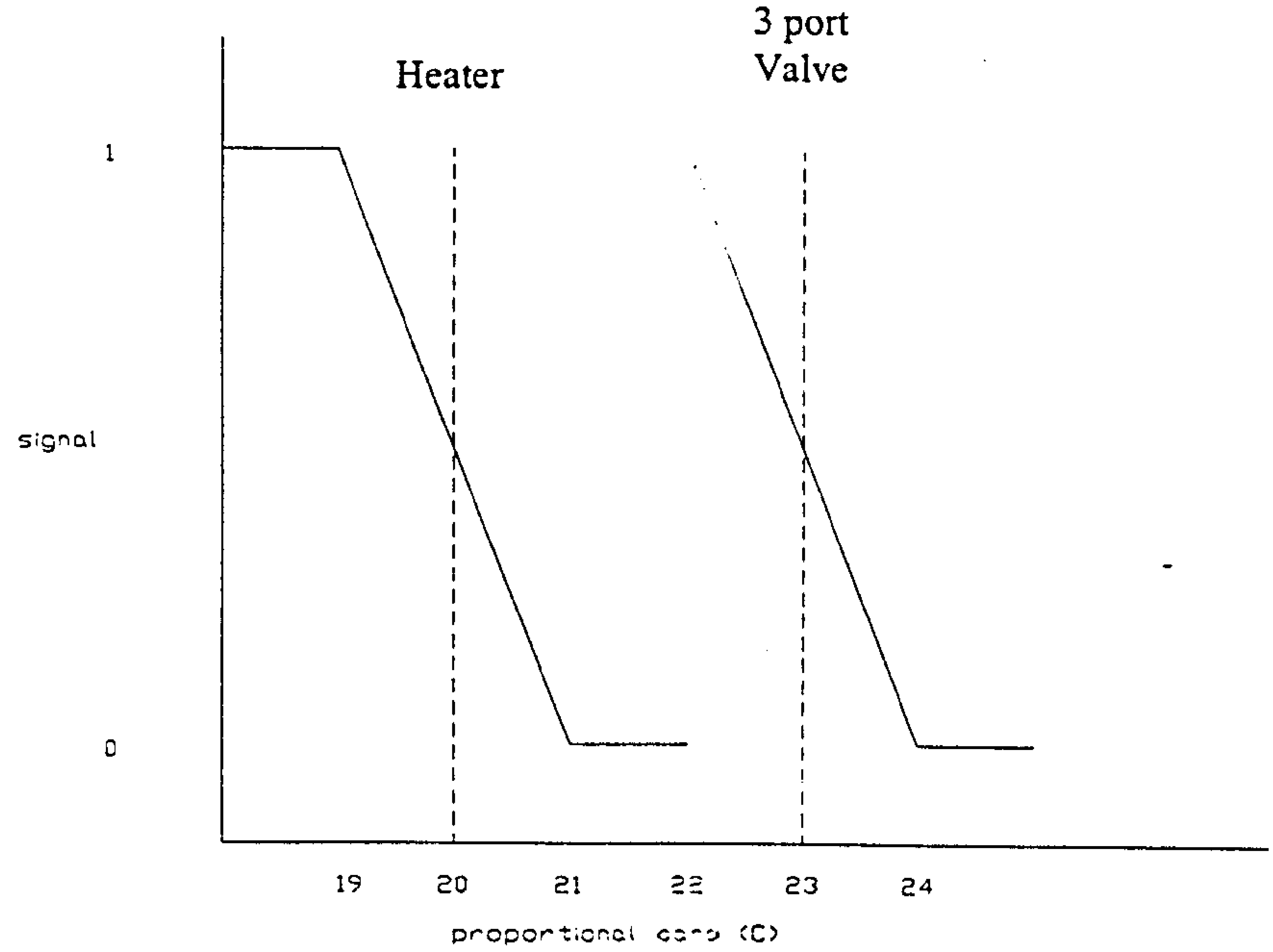


Figure 4.6 Example heater and valve schedule for zone setpoint of 20°C

The example shown in Figure 4.6 shows the zone temperature controlled by the heater that shuts down as the setpoint is reached. The heater control signal is 1 for full power and 0 for off and the valve control signal is 1 for the valve to be on full re-circulation and 0 for full flow through the valve. If the zone temperature continues to increase the three port valve begins to modulate from full re-circulation to allow cold fluid in from the main to cool the space.

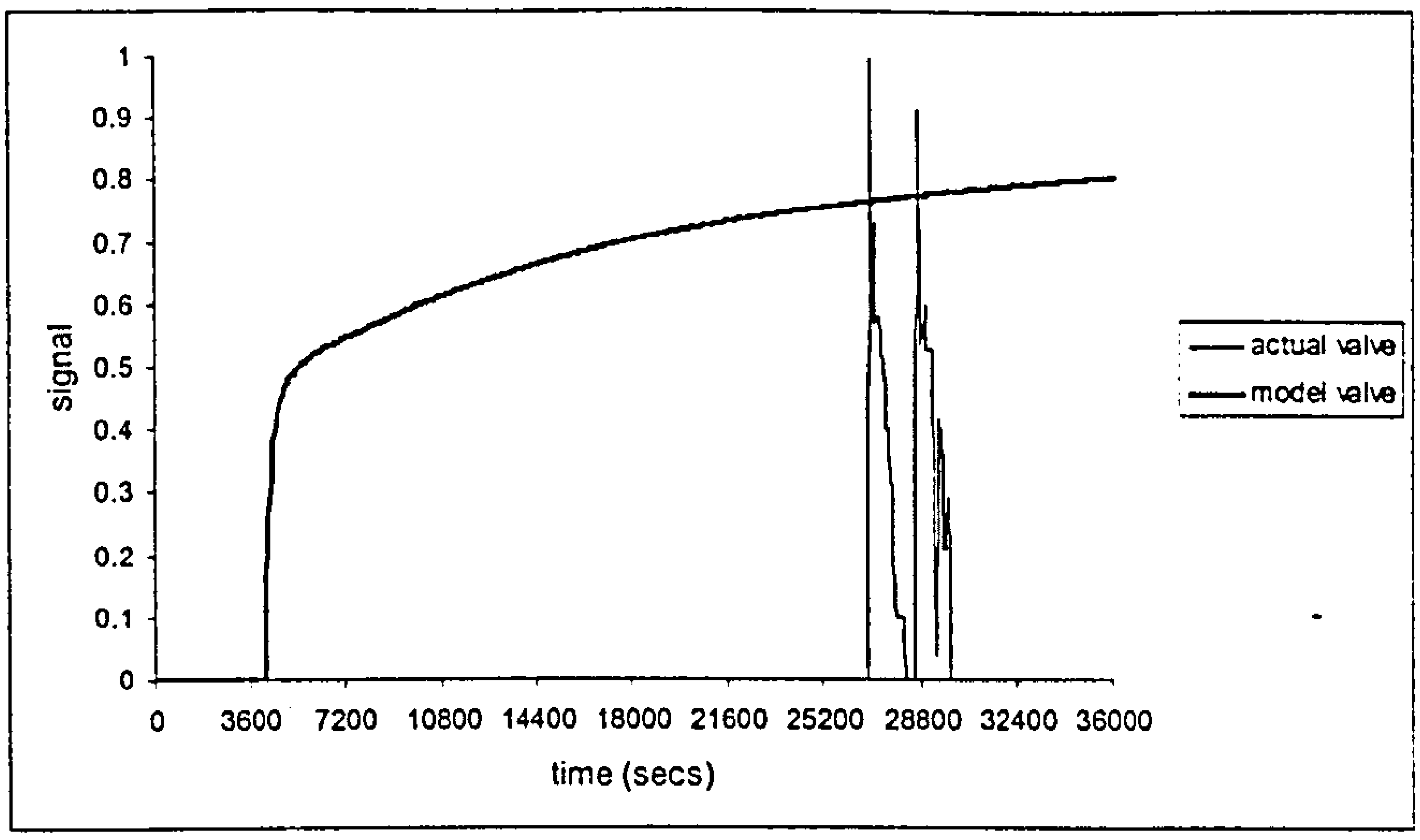


Figure 4.7 Comparison of actual signal to valve and that to the valve in the simulation

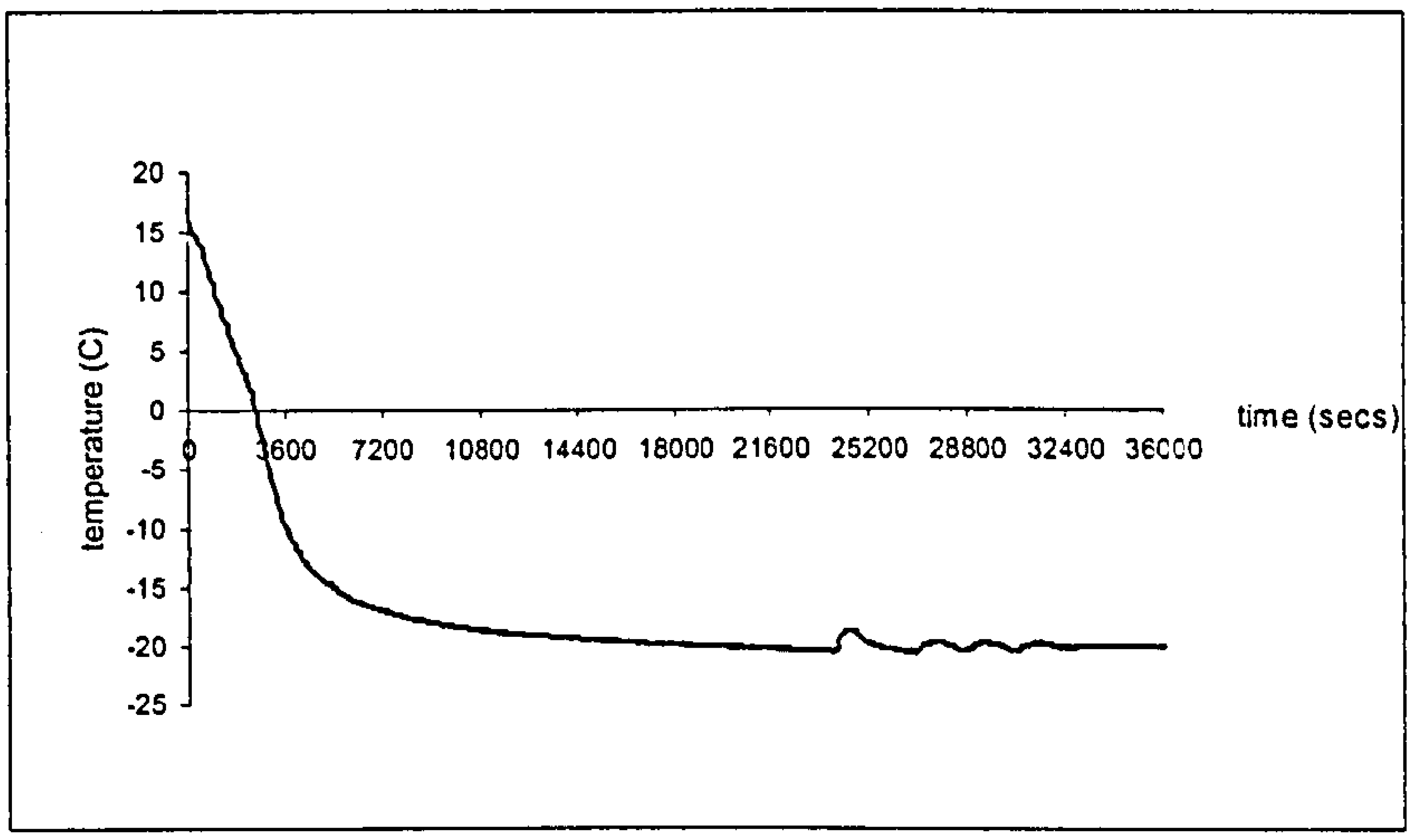


Figure 4.8 Actual soakroom temperature profile resulting from control valve action in 4.7

Figure 4.7 shows a comparison of the measured signal to the soakroom three port valve with a setpoint of –20°C and the signal to the simulated valve operating under the same conditions. Figure 4.8 shows the actual soakroom temperature response to the control valve action. It can be seen that the signal from the idealised controller used in the simulation progressively modulates the three port valve towards re-circulation as the set point is reached. Whereas the actual control signal recorded from the plant does not respond to the nearing of the set point and then modulates wildly although this has no apparent effect on the soakroom temperature. The oscillations in air temperature are caused by the supply fan shutting down and not the controller action, it can be seen that the controller action starts after the first temperature oscillation has occurred. Figure 4.7 clearly shows that no assumptions could be made as to what control algorithm is used. Whilst it is accepted that proportional control suffers from the problems of instability and offset (difference between setpoint and measured variable) [2] it benefits greatly from simplicity. It was decided that the probability of a slight offset occurring would not greatly affect the operation of the CWT simulation hence idealised proportional control was adopted as the control strategy employed.

4.4 IDA simulation development methodology

The methodology for developing a system definition input file for use by IDA Solver is defined in Appendix D.

Chapter 5

Verification and validation

Verification and validation is essential when developing any computer simulation program in order to give the user confidence in the results produced. Inaccurate prognosis is not necessarily down to errors within the program. All modelling involves assumptions and simplifications that are likely to introduce errors into the ensuing results.

Hensen [1], identified two groups of error sources, the first of these is known as external sources and are external to the internal workings of the program and are not under the control of the user. The types of errors expected within this group are:

- Differences between the actual and assumed boundary conditions.
- Differences between the actual use of the facility and that assumed by the user.
- Errors in deriving the simulation input files.
- Differences between the actual thermal and physical properties and those assumed by the user.

The second group of error sources are called internal errors. This type of error is directly linked to the internal workings of the program and is contained within the coding of the program. Types of errors expected within this group are:

- Differences between the actual heat transfer mechanisms operative in individual components and the algorithmic representation of those mechanisms in the program.
- Differences between the actual interactions of heat transfer mechanisms and their representation in the program.
- Coding errors.

Validation can fall into one of three categories [2]:

- Analytical
- Comparative
- Empirical

The first stage of any component model validation should start with the individual components as they are developed. Each component should be run in isolation and the results viewed in a context of "expected model behaviour" i.e. a heater heats and not cools, a humidifier puts moisture into an air stream and not takes it out.

Analytical testing involves the derivation of exact solutions by analytical means. These solutions can then be compared to the equivalent program predictions. Because of the wide range of building types and applications, analytical testing procedures are very difficult to devise.

Comparative validation is the comparison of the model against other models that have been subjected to a greater degree of previous testing. This inter-model comparison only shows how good one model is compared to another and not whether the model can replicate actual component behaviour.

Empirical testing involves the comparison of simulation output to data recorded from plant in operation. Empirical validation gives confidence that the results from the model are the same or are very similar to those recorded from the real life system. The problem is that data of this sort is often difficult and expensive to obtain and any measurement taken involves some degree of uncertainty usually stemming from the calibration of the sensing devices. It is difficult to provide enough suitable information for the validation of a dynamic simulation.

The validation of the Climatic Wind Tunnel model has been carried out using the above techniques. Greatest emphasis for the validation has been placed upon the empirical approach due to the availability of data through the facilities data acquisition system.

Details of all the model parameters and boundaries used in the validation work is given in Appendix F of this thesis.

5.1 Vehicle model validation

The vehicle model developed in chapter 3 is a semi-empirical model constructed using recorded data. As no temperature data is available for a vehicle operating at various wind speeds validation takes the form of "expected model behaviour".

The vehicle was simulated to be running in an air stream of 0°C and speeds of 0, 50 and 100 km/h. The resultant engine temperatures and heat flux from the simulation were logged and the results are shown in Figures 5.1 and 5.2.

The vehicle used in the soakroom tests was a Rover 200 2.0 litre diesel. The engine has the following details:

Dry engine weight:	187kg
Coolant capacity:	7.5litres of water
Oil sump capacity:	4.9 litres

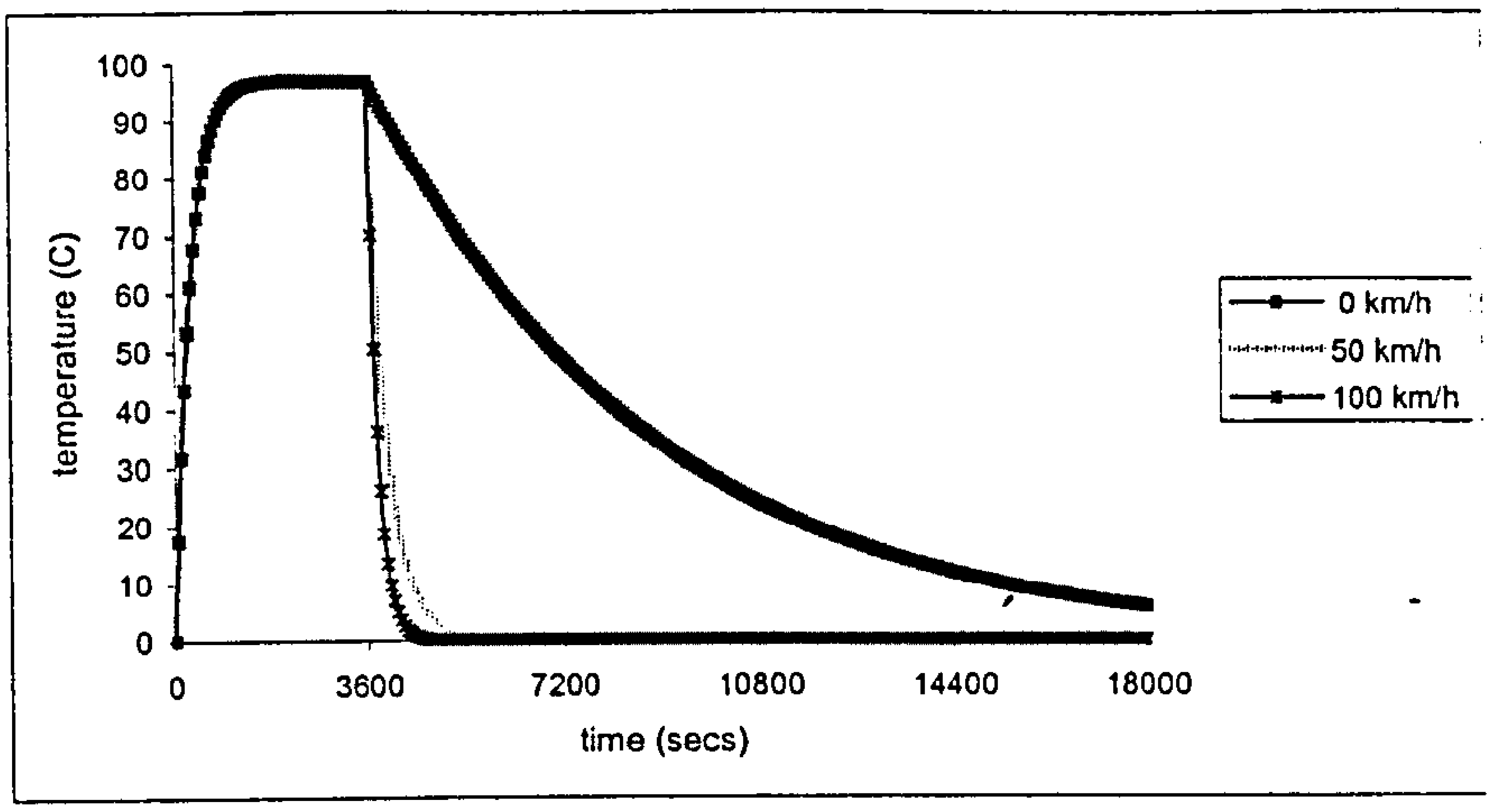


Figure 5.1 Engine temperatures for vehicle operating in different wind speeds

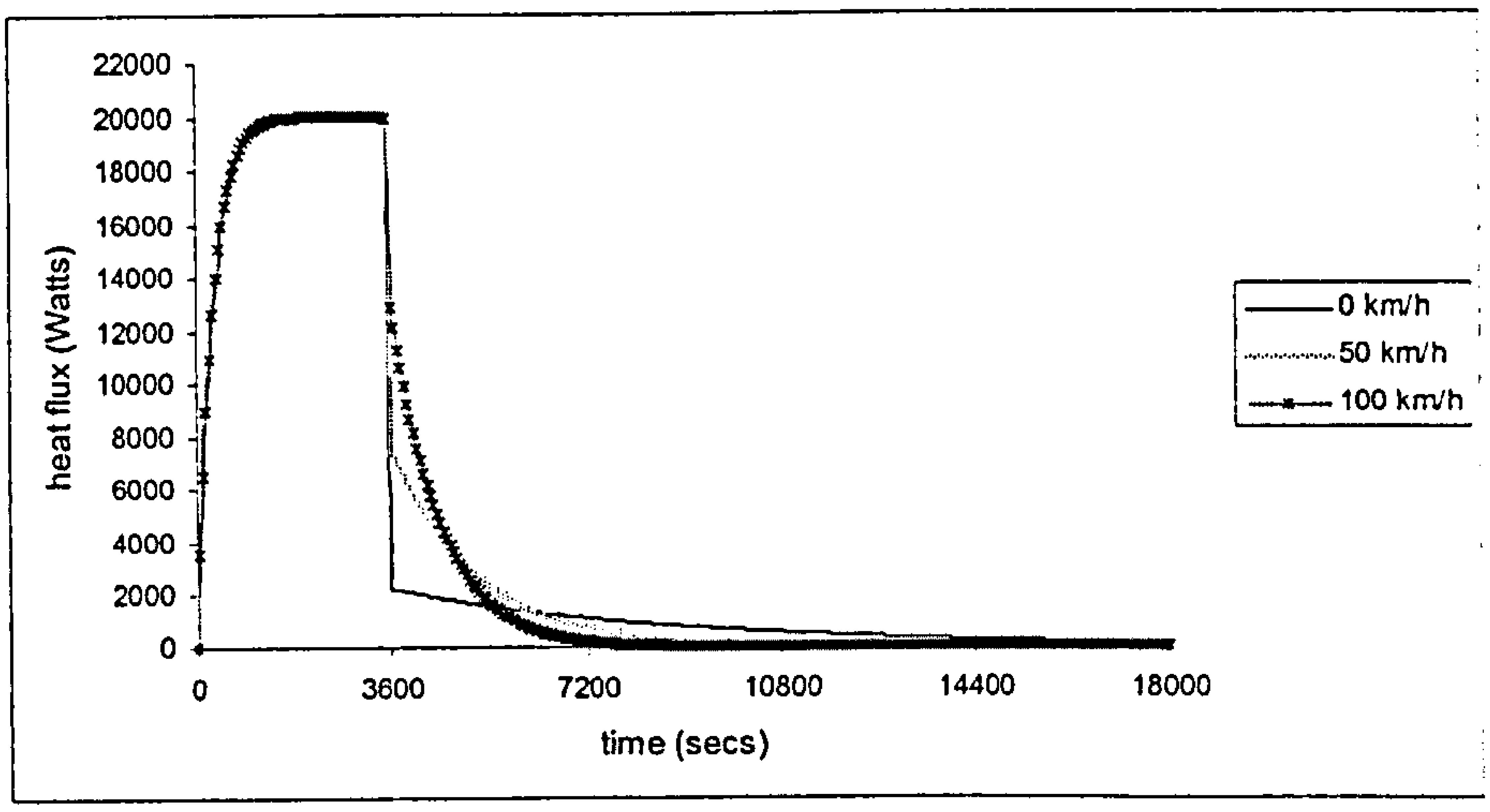


Figure 5.2 Heat flux from vehicle operating in different wind speeds

From Figure 5.1 it can be seen that as the wind speed increases the engine cools at a greater rate. This is corroborated by the heat flux from the engine shown in Figure 5.2. The heat transfer coefficient increases with wind speed so just after switch off the vehicle at 0 km/h has the lowest heat flux and the one at 100 km/h has the highest. As the engine stores only a finite quantity of heat it is depleted fastest by the 100 km/h air stream, hence the engine will cool quicker and have a lower heat flux than the others. Figure 5.1 shows that the engine is cooled to the surrounding conditions far quicker than the slower wind speeds and its heat flux is reduced to a very low level far quicker.

From this it can be said that the model operates as would be expected in that when switched on it heats upto a steady state level and then cools at varying rates depending on the speed of the air stream surrounding it.

5.2 Compressor model validation

The compressor model was developed from a curve fit to data provided by the manufacturers compressor selection model; the development process was detailed in chapter 3.

The model was tested with the coefficients listed in chapter 3 Tables 3.4 to 3.6and the results compared to data that was obtained from the manufacturers selection model. The comparison between the model and the manufacturers data included the high and low stage power consumption and the overall cooling capacity. A wide range of operating conditions were considered in the full-load regime. Evaporating temperatures were fixed at values of 10°C increments from –60 to 10°C and the condensing temperature varied from 0 to 50°C.
From the manufacturers data and that produced by the simulation the standard deviation and correlation coefficients for power and cooling capacity were calculated, these are shown in Table 5.1.

Compressor power consumption	Compressor cooling capacity
Standard deviation = 116629.1 Watts	**Standard deviation** = 1226195.5 Watts
Correlation coefficient (R^2) = 0.98	**Correlation coefficient** (R^2) = 0.99

Table 5.1 Standard deviation and correlation coefficients for compressor power consumption and cooling capacity

The results show a good overall positive correlation between the model and the available data upon which it is based.

From analysis of the comparison of the model prediction and the design data two potential areas where serious errors are likely; these being at either end of the performance envelope –60°C and +10°C.

At an evaporating temperature of -60°C the power consumption behaved as expected increasing as the condensing temperature increases. As condensing temperature increases the cooling capacity should fall yet the model predicts that it rises, this would result in an over estimation of the cooling for a given energy consumption.

When evaporating at +10°C the model behaved as would be expected with the power consumption increasing and the cooling capacity decreasing as the condensing temperature increases. But it should be noted that there are some large fluctuations in the design data at this temperature. How the Matchmaster program [4] calculates the energy consumption and cooling capacities is unknown and casts doubt as to its validity and the data that it produces.

5.3 Climatic Wind Tunnel model prediction and comparison with measured data

In order to validate the accuracy of the system model of the Climatic Wind Tunnel (CWT), its output has been compared to that measured from the actual facility. The data from the test facility was recorded using the data acquisition system used for the test vehicles. As use was made of this existing data recording facility it was not possible to log data from the plant when it was being used for vehicles under test. Fortunately some manufacturers use their own data acquisition systems which freed the resident one to record data from the plant when vehicles were under test. As the opportunities to obtain plant data were somewhat limited only a small number of tests could be recorded for both the soakroom and the test chamber.

5.3.1 Soakroom model prediction and comparison with measured data

The soakroom is used for the pre-conditioning of vehicles before they are tested and is predominantly used for low temperature test preparation. Vehicles are generally left in the soakroom for periods of 8 – 10 hours. This time span is equivalent to the vehicle being left unattended in the prevailing conditions and is long enough for it to reach the desired test condition.

The plant schematic used for the simulation is shown in Figure 4.3. Data was recorded from the plant both with and without a vehicle installed within the soakroom. Data without a vehicle was logged for conditions of –35°C, –20°C, +40°C, the actual and predicted air temperatures within the zone are shown on Figures 5.19, 5.20 and 5.21 respectively.

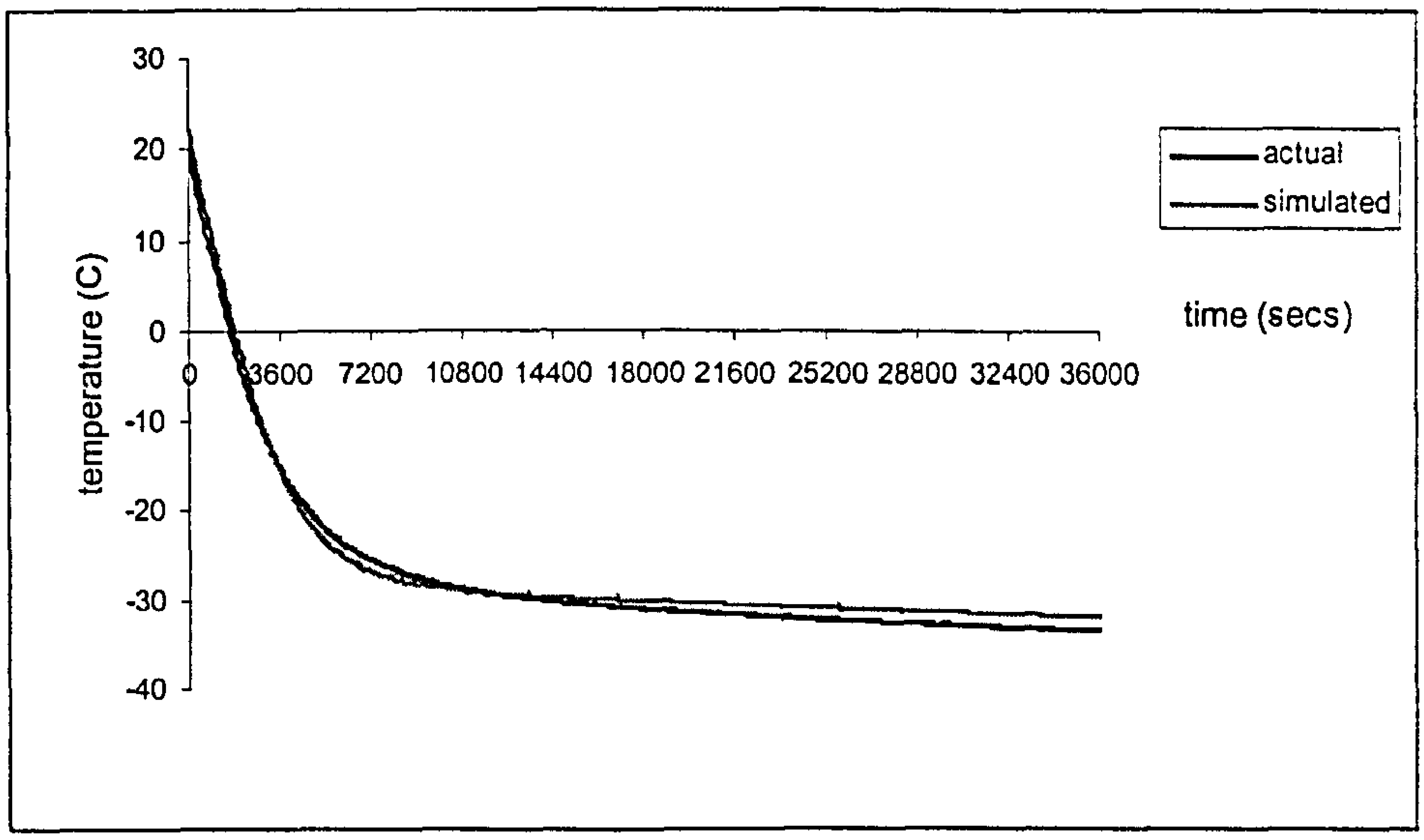


Figure 5.19 Comparison of actual and predicted soakroom temperatures, -35°C setpoint, no vehicle

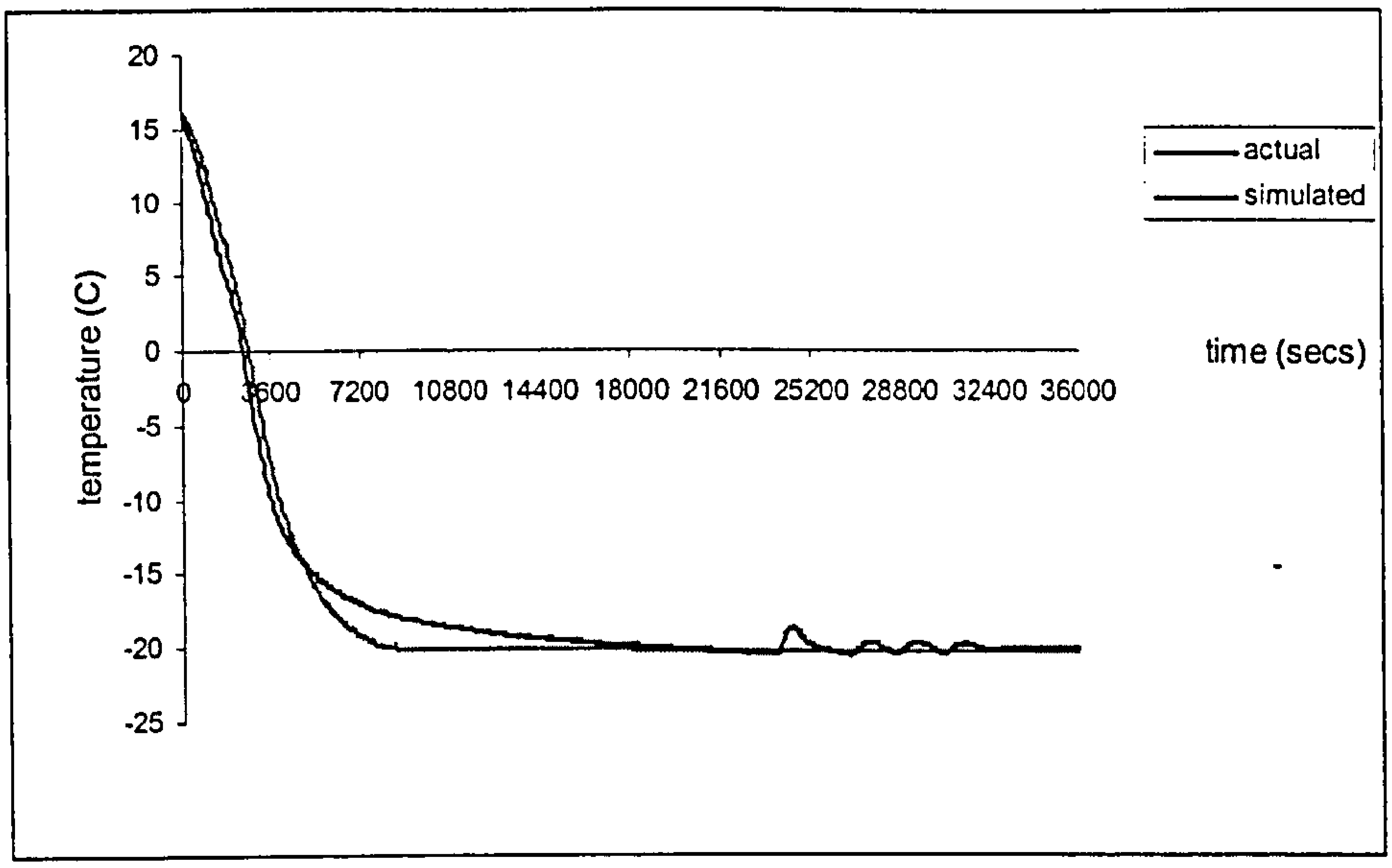


Figure 5.20 Comparison of actual and predicted soakroom temperatures, -20°C setpoint, no vehicle

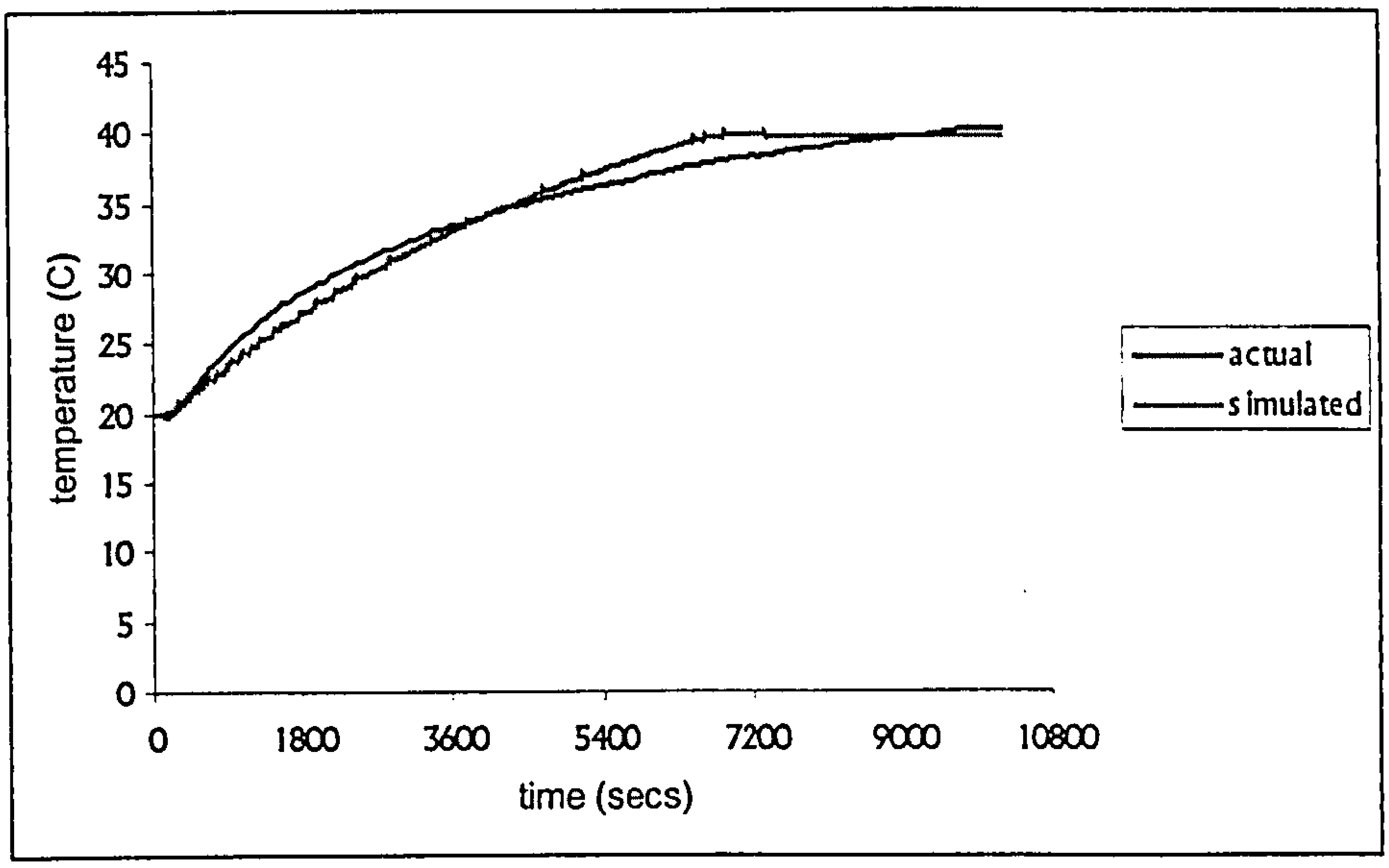


Figure 5.21 Comparison of actual and predicted soakroom temperatures, +40°C setpoint, no vehicle

Figure 5.19 shows excellent agreement between the data and the model prediction. It does indicate that the soakroom plant is potentially undersized, as it does not reach the set point during the 10 hours of the test.

From Figure 5.20 it can be seen that the model gives good agreement with the data down to –15°C. At which point it begins to over estimate the heat transfer occurring and as a consequence reaches the –20°C setpoint after 2 hours, some three hours before the actual soakroom achieved this condition. No comparison between the fluid flow temperatures out of the heat exchanger could be made due to a faulty sensor on the fluid exit port. The oscillations in the data recorded from the plant that start 6½ hours into the test are due to the air handling unit fan switching off. The fan in the soakroom has been found to be often shut down by the control system; as there is no data for the control system available this occurrence cannot be allowed for in the simulation.

Figure 5.21 again shows that the model has slightly overestimated the heat transfer occurring and achieved the setpoint an hour before the actual zone temperature reached this level.

It was not possible to obtain data for the soakroom with a vehicle installed for such a range of operating conditions. It was only possible to obtain data for the plant when conditioning a vehicle to +40°C; this is shown in Figure 5.22.

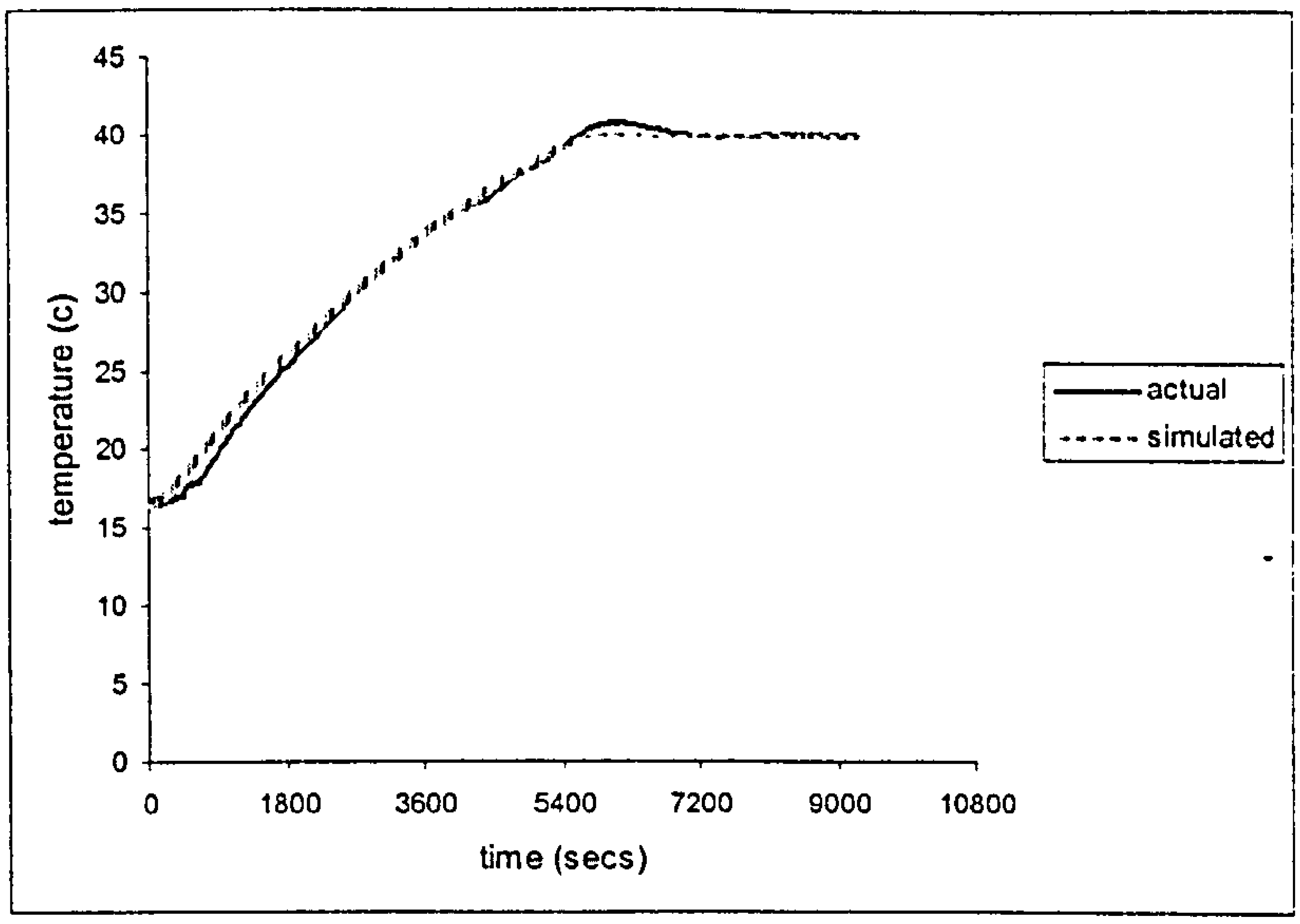


Figure 5.22 Comparison of actual and predicted soakroom temperatures, +40°C setpoint, with vehicle

Figure 5.22 shows good agreement with the data achieving the set point at the same time as the data. The achieving of the set point in a shorter time than when no vehicle was installed (Figure 5.21) can be attributed to the extra heat being input into the space by the vehicle that was driven into the space hot.

5.3.2 Test chamber model prediction and comparison to measured data

The test chamber is used to test vehicle engine cooling and environmental control systems under simulated driving conditions. The conditions within the test chamber are often not consistent as the vehicles are subjected to different test conditions throughout their test period.

It has been possible to obtain data for these changes in operating conditions for both heating up and cooling down of the air temperature. As the test chamber is never operated without a vehicle being present all the comparisons of actual and simulated data have a vehicle present. A schematic diagram of the test chamber can be found in Figure 5.2.

The vehicle under test at the time when the data upon which the validation is based was a Jaguar XJR 3 litre and has the following engine details:

Oil sump capacity	6.5 litres
Dry engine weight:	242kg
Coolant capacity:	10.2 litres of water

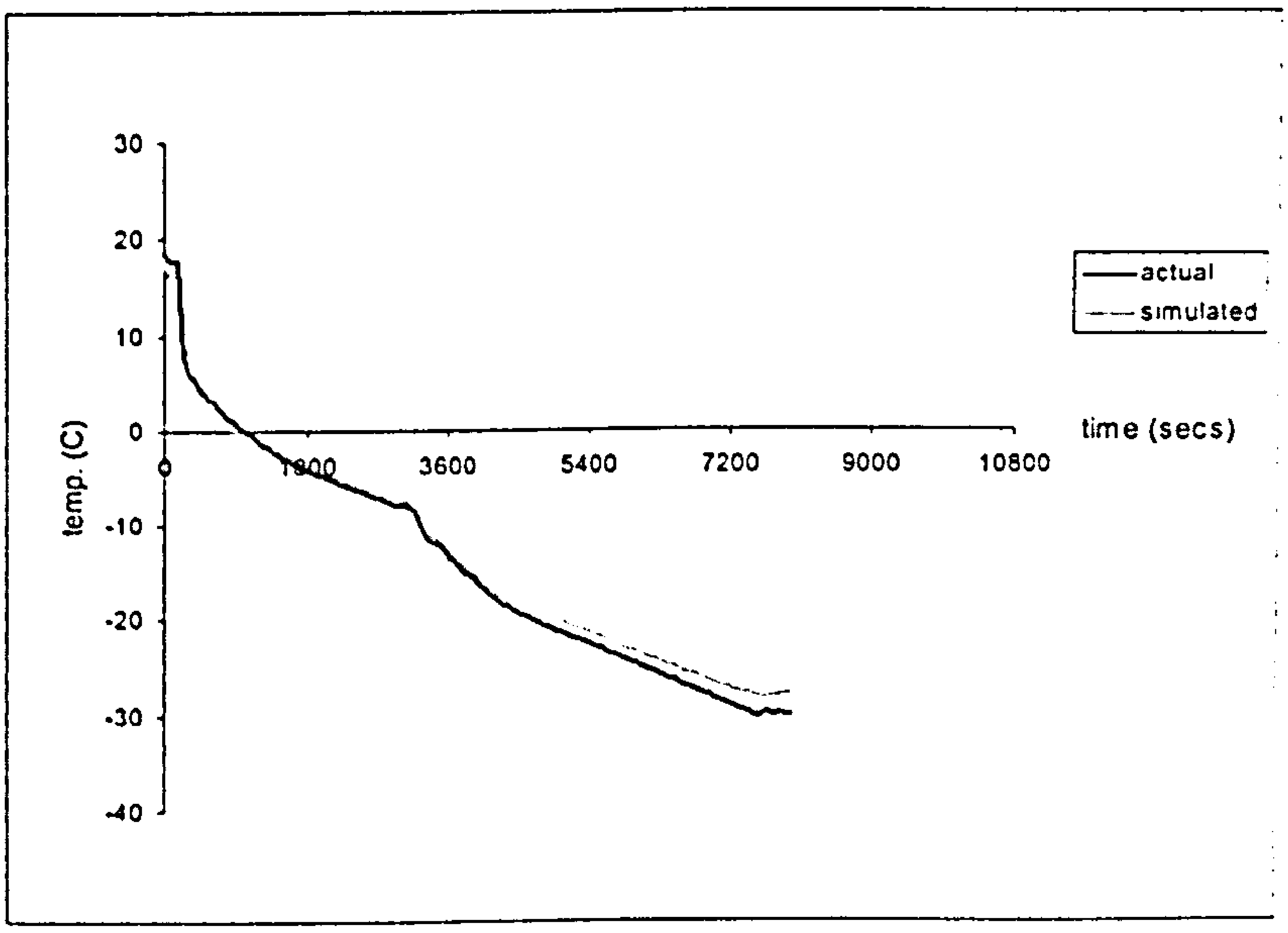


Figure 5.23 Comparison of actual and predicted test chamber temperatures, -30°C setpoint, with vehicle

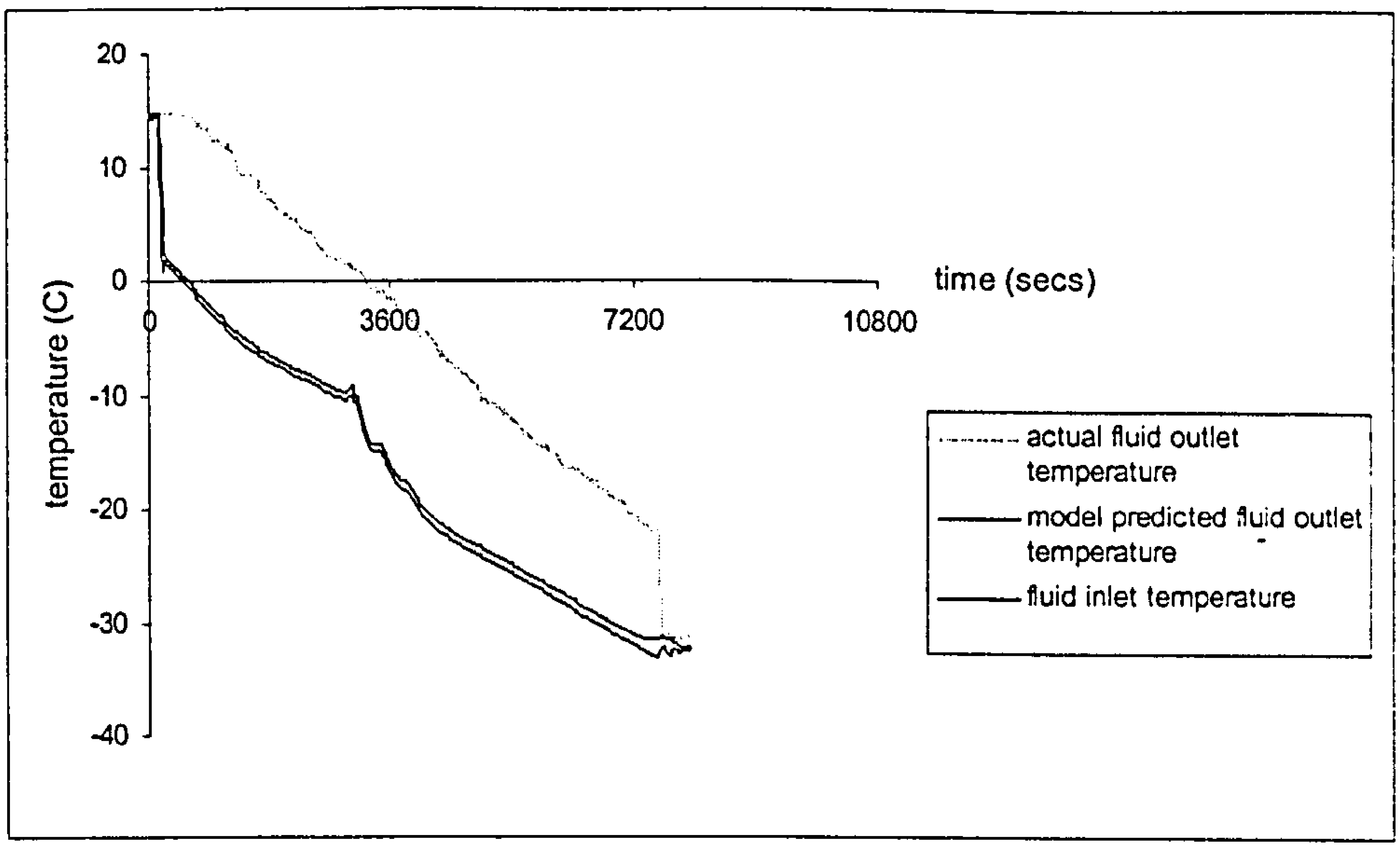


Figure 5.24 comparison of actual and predicted fluid inlet and outlet temperatures for test chamber heat exchanger, -30°C setpoint, with vehicle

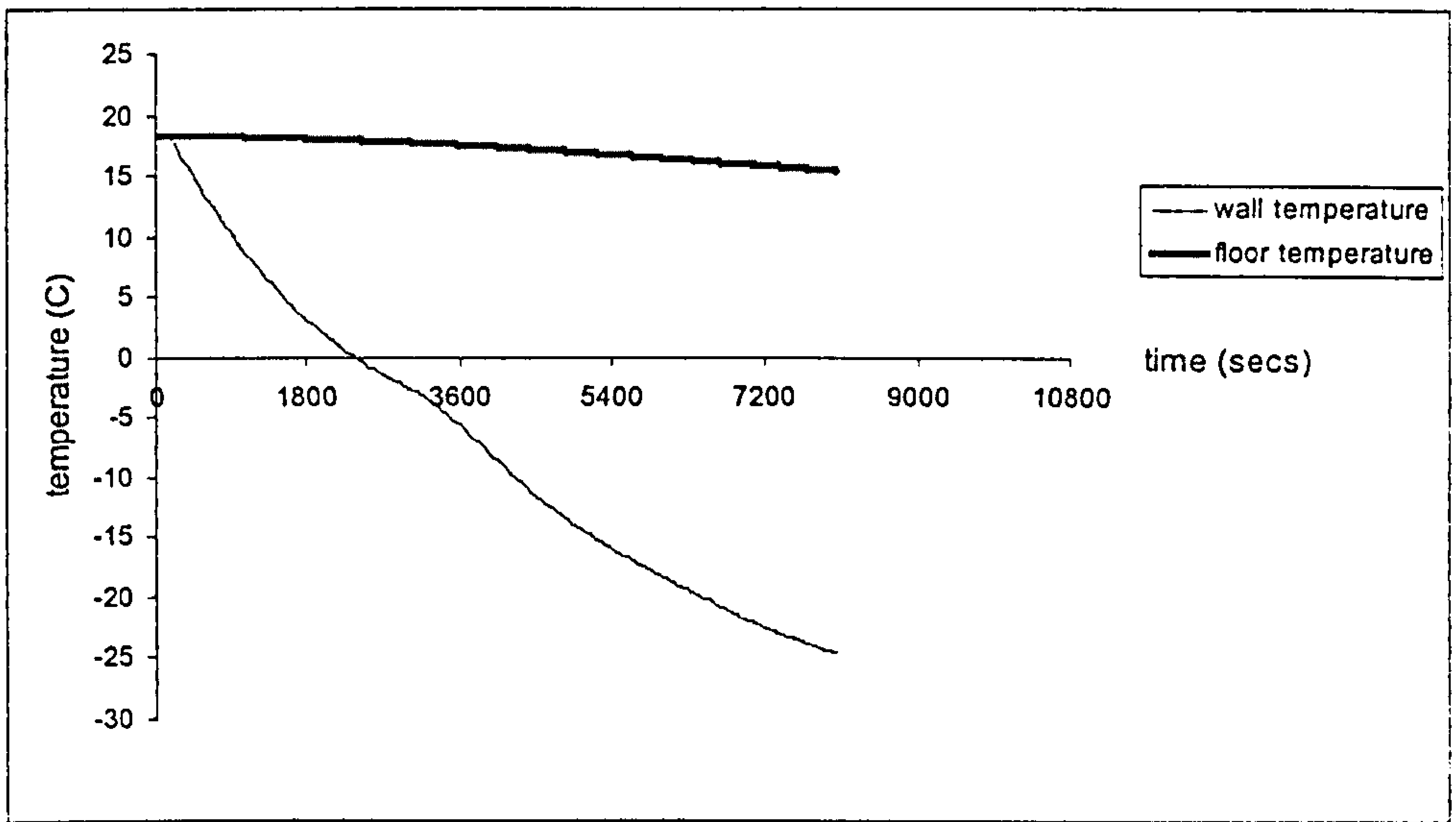


Figure 5.25 predicted test chamber wall and floor temperatures, -30°C setpoint, with vehicle

The comparison between actual and simulated air temperatures for the test chamber with a –30°C setpoint is shown in Figure 5.23. It can be seen that good agreement exists between the two data sets, with the model the model underestimating the final temperature by 2°C.

The predicted and actual fluid exit temperatures from the heat exchanger are shown in Figure 5.24. The simulation used the same input temperatures as were recorded from the plant during the test. It is shown that between the predicted outlet temperature and the flow a temperature difference of 1°C exists, where as the logged data shows a temperature difference of 10°C generally exists between the inlet and outlet temperatures. The disparity between the predicted and actual temperatures may be put down to the data used for the heat exchanger model. The model was developed using the design data [3] for the Climatic Wind Tunnel installation. This data gave the temperature difference between inlet and outlet temperatures of the heat exchanger to be 1°C. The model gives results that are true to this design criterion, but when they are compared to what is actually happening within the plant it is clear that the design criteria are not being met. It has not been possible to compare the entry and exit temperatures for the other heat exchangers in the installation due to faulty sensors. It is not possible to say whether this temperature difference is specific to this one coil or occurs in all cases. It is feasible that the heat exchangers were designed to have a 10K flow and return differential as this is the most common design temperature difference.

The predicted wall and floor temperatures are shown in Figure 5.25. It can be seen that the lightweight wall elements of the structure cool down quickly due to their low thermal capacitance. In contrast the heavyweight concrete floor cools down at a much slower rate due to its large thermal capacitance. In a facility such as the test chamber where sudden and quite large transient conditions can be imposed, the capacity of its structural elements to react quickly to these changes is important. In this case the large amount of heat stored within the floor slab may have a detrimental effect upon the HVAC systems in their ability to achieve specified conditions.

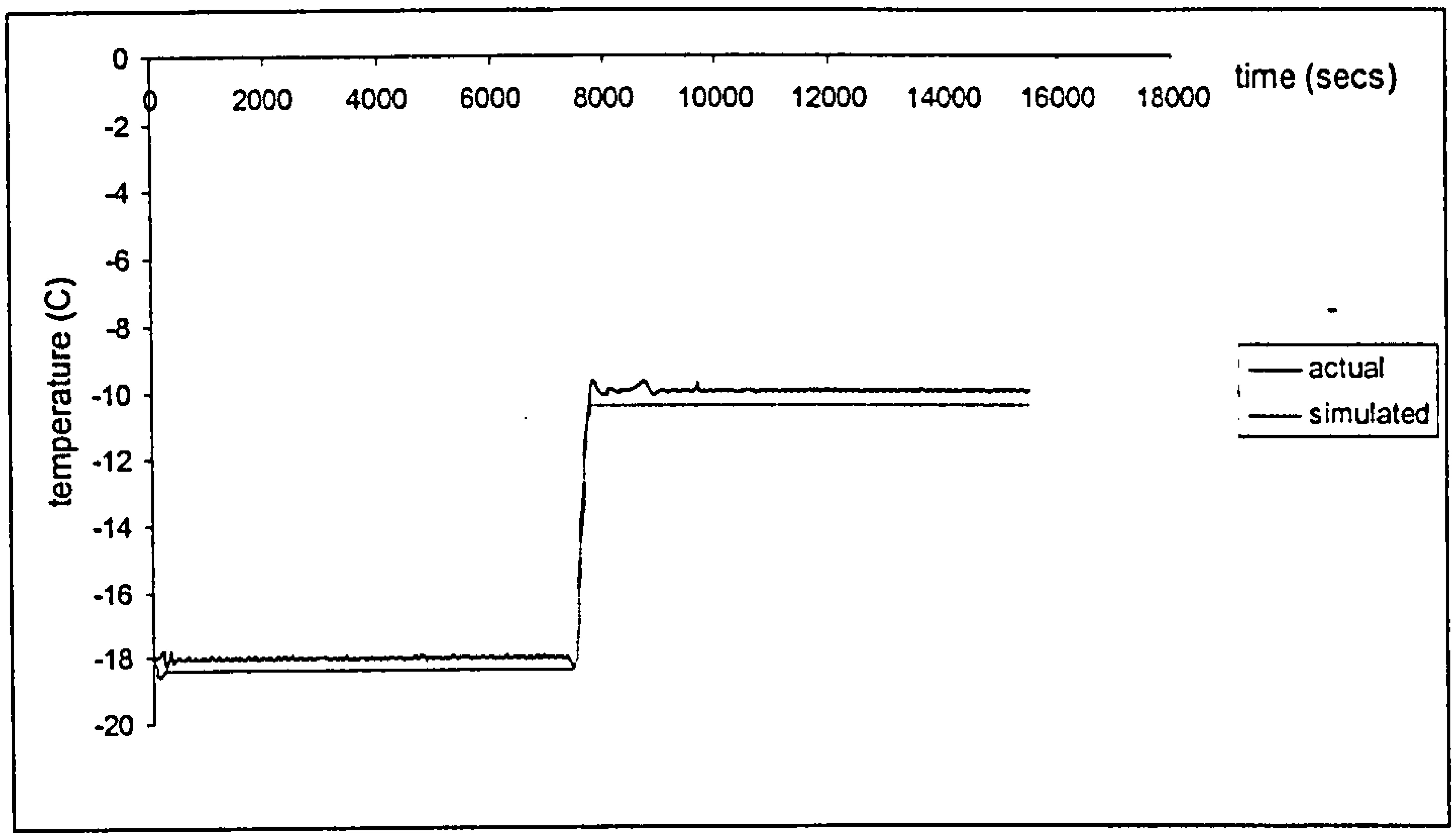


Figure 5.26 Comparison of actual and predicted test chamber temperatures, setpoint changing from –18°C to –10°C, with vehicle

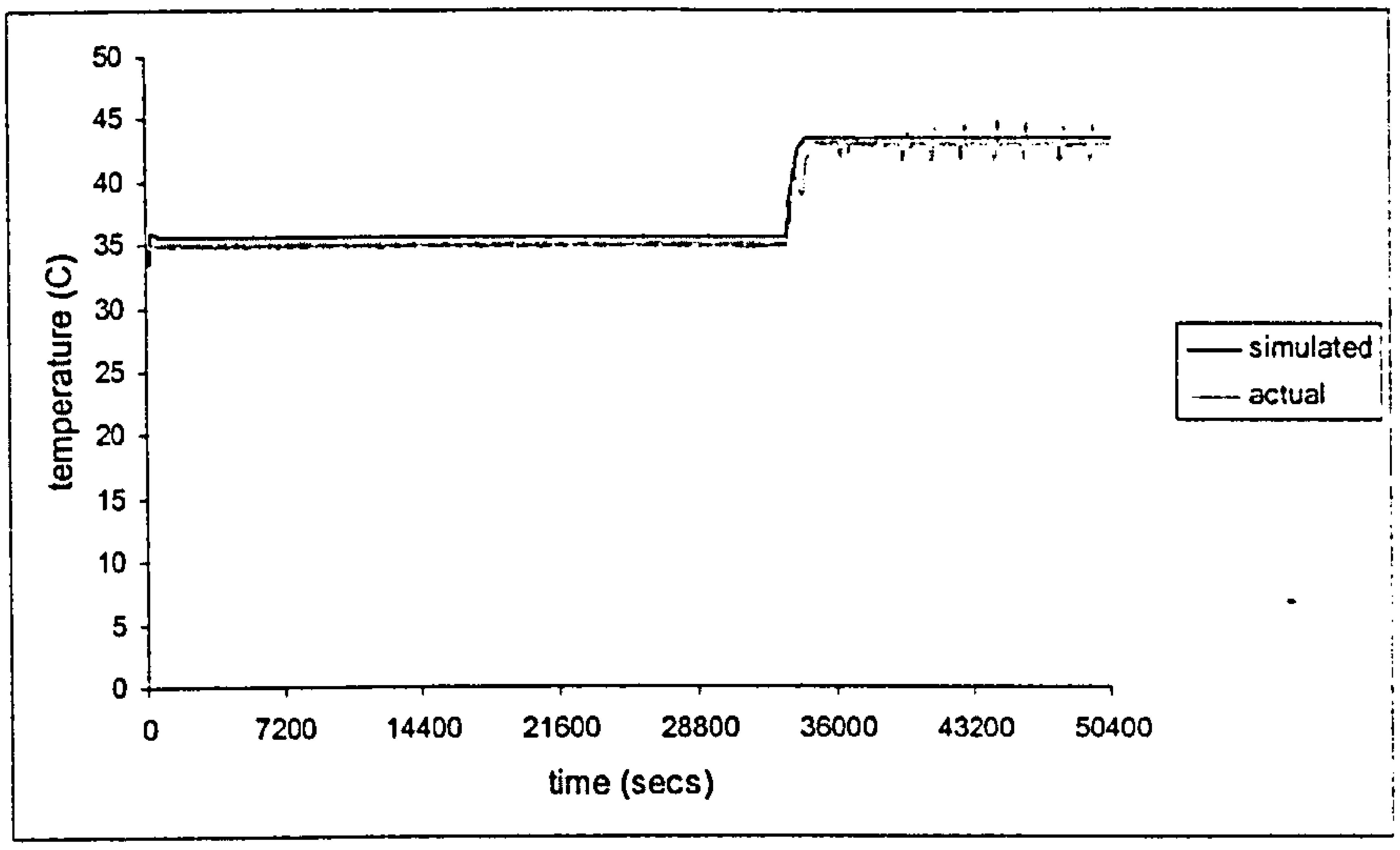


Figure 5.27 Comparison of actual and predicted test chamber temperatures, setpoint changing from +35°C to +43°C, with vehicle

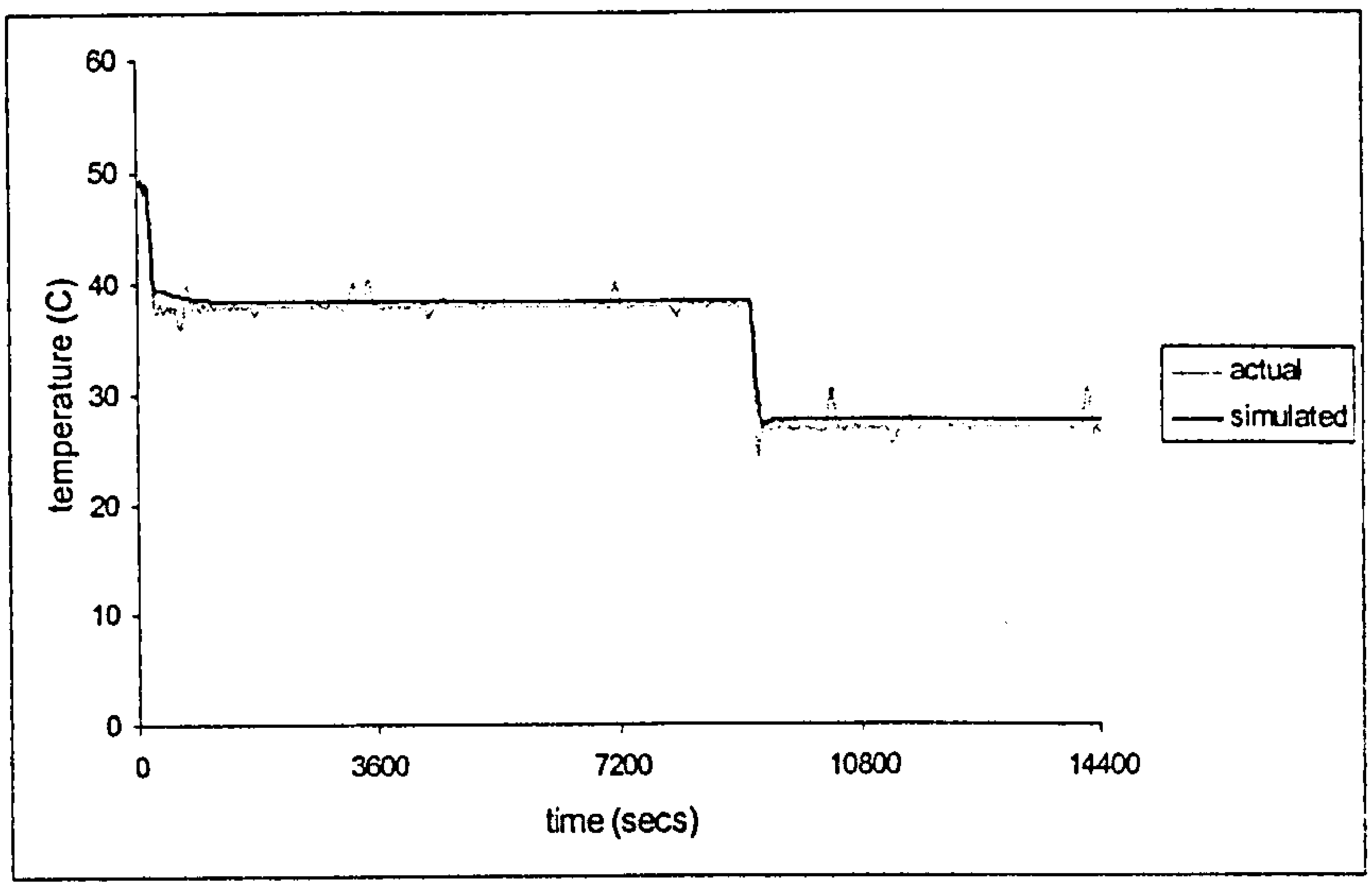


Figure 5.28 Comparison of actual and predicted test chamber temperatures, setpoint changing from +49°C to +38°C to +28°C, with vehicle

Figures 5.26, 5.27 and 5.28, show the comparison of actual and predicted air temperatures within the test chamber for a range of varying conditions. It can be seen that the model predicts the changes in setpoint well and gives good agreement with the measured data. This indicates that the model will give good prediction over a whole range of operating conditions that would be expected within the Climatic Wind Tunnel test chamber.

5.4 Conclusion

The purpose of validation and verification is to give the user confidence that the results produced by the simulation are representative of what would be expected from the real life plant.

Validation falls into one of three categories:

- Analytical
- Comparative
- Empirical

Analytical testing involves the derivation of exact solutions by analytical means that can then be compared to the equivalent program predictions. This type of testing is generally difficult to apply in building applications due the wide range of building types and their application.

Comparative testing is the inter comparison of results predicted by one program to a given set of inputs to the predicted results given by another program that has been subjected to the same inputs. This test only tells the user if his program gives similar output to another and gives little indication as to whether it can accurately predict the output of the system that it is supposed to be replicating.

Empirical testing is the comparison of simulation output to actual data recorded from an installation. Empirical data is often difficult and time consuming to obtain, but gives the best indication by far as to whether a simulation is giving results that are comparable to those expected in real life.

The output of the Climatic Wind Tunnel sub-system models have been compared to data that was gathered for the plant whilst operating under different conditions. It is shown that the prediction of the air temperatures within the test chamber and the soakroom are very closely matched to the data recorded from the plant.

The application of a simplified proportional control strategy to the systems has not affected the overall temperature control within the test chamber and soakroom. To refine the simulation further it would be necessary to establish the exact type of control used for the plant – information that was not available for this project.

The compressor model is a curve fit of data generated by a compressor selection program and in essence the validation took the form of an inter-model comparison. The model was shown to correlate very well to the data it was based upon. Problem areas are shown to exist at opposite ends of the performance envelope (evaporating temperatures of -60°C and +10°C). At –60°C the model gives a rise in power consumption when it should be giving a fall and a +10°C the data the model is based upon has large swings within it. Fortunately the extremes of the performance envelope are rarely visited in the CWT systems and the majority of operation falls within the area where very good agreement between the model and the data exists.

The vehicle is a model based upon recorded data. No data of exactly how the model should respond under different environmental conditions exists. It is shown that the model reacts as it would be expected to; it heats up to a steady state level when switched on and cools down at different rates depending on the velocity of the air surrounding it.

5.5 References

[1] Hensen J.L.M (1991). On the thermal interaction of building structure and heating and ventilation system. PhD Thesis, University of Eindhoven.

[2] Bloomfield D.P (1989). Evaluation procedures for building thermal simulation programs. Proceedings Building Simulation '89, Vancouver. Pg. 217 – 222.

[3] DSMA Ltd. MIRA Climatic Wind Tunnel design specification. DSMA Ltd. Vancouver, Canada.

[4] Matchmaster compressor selection program, Sabroe (UK) Ltd. Kings Norton, Birmingham. England.

Chapter 6

Application to Operational strategies

Use needs to be made of the models developed in improving the operational efficiency of the Climatic Wind Tunnel facility. Judgement upon how successful the application of the models developed to the improvement of efficiency will be based upon the reduction of the facility's energy consumption and in turn lowering its operational cost.

The model of the Climatic Wind Tunnel systems allows analysis of different control and plant configuration strategies. This chapter will look at some of the different strategies possible and highlight the practicability of their incorporation into an overall energy reduction scheme for the facility.

The decision of which operational strategies the model should be tested upon was carried out under the direction of the MIRA Thermofluids group. The areas investigated for potential savings are those of plant operation / construction that are of particular interest to them.

Due to the unknown nature of the electricity billing system in operation at the facility, no comparative cost savings have been able to be made. The CWT facility purchases its electricity on an annual basis from the MIRA Estates department and it has not been possible to ascertain details of how the costs are formulated.

This study has only concentrated upon the possible reduction of the energy consumption of the CWT. The implications of the revised operational strategies on the staffing of the CWT have not been investigated.

Details of all the model parameters and boundaries used in the operational strategies work is given in Appendix F of this thesis.

6.1 Refrigeration system control strategies

The refrigeration system is the largest consumer of power within the Climatic Wind Tunnel. The plant runs for a large proportion of the test day in order that it maintains the trichloroethylene at 15K below the lowest setpoint of either the test chamber or soakroom. Scope for the reduction in its power consumption is limited but potential exists for the investigation into the sequencing of the number of condenser fans running as a function of outside ambient temperature and refrigeration load.

The total power used by the refrigeration system is made up of two components the compressor power and the condenser fan power shown in Figure 6.1. As the refrigeration load begins to decrease i.e. as the setpoint in approached, the compressors will begin to off load through movement in the slide valve position reducing their power consumption. As the load falls the number of fans required to condense the refrigerant also reduces. The object of this study has been to analyse the effect of reducing the number of condenser fans for a given load condition in order to find the optimum point at which the compressor energy consumption begins to rise due to the increasing discharge pressure.

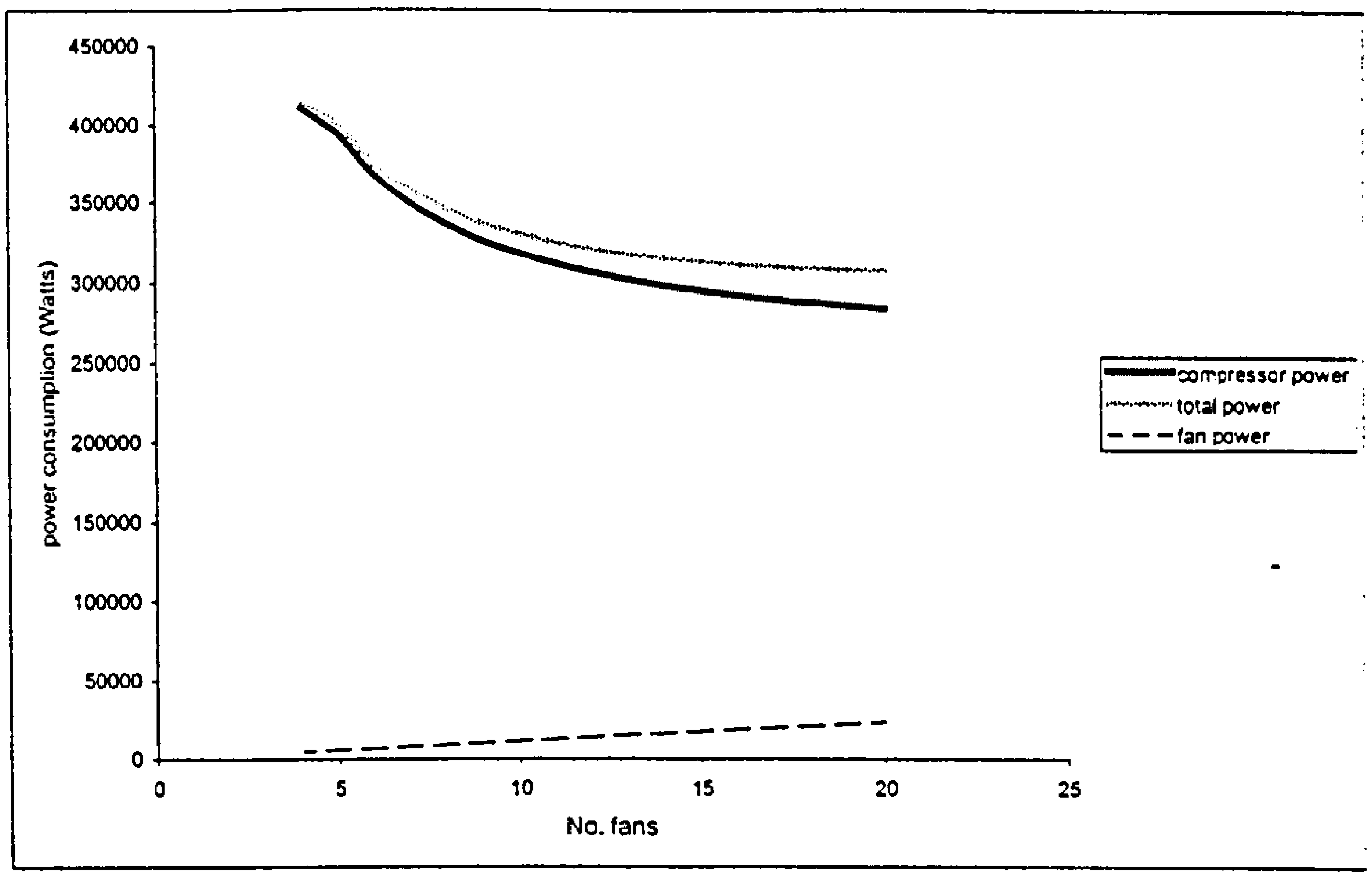


Figure 6.1 Components of refrigeration system power consumption. Full load operation, 25°C ambient temperature

Figure 6.1 shows the refrigeration system operating at full load in a typical outside summer air temperature of 25°C. It can be seen that as the number of condenser fans is reduced the power consumed by the compressors increases substantially. This shows that at full load conditions the lowest energy consumption of the plant will be when all the condenser fans are in operation. The simulation has been run for three different external conditions (summer, winter and mid-season) over the full capacity range of the compressors, reducing the number of fans in each case; the results of these simulations are shown in Figures 6.2, 6.3 and 6.4.

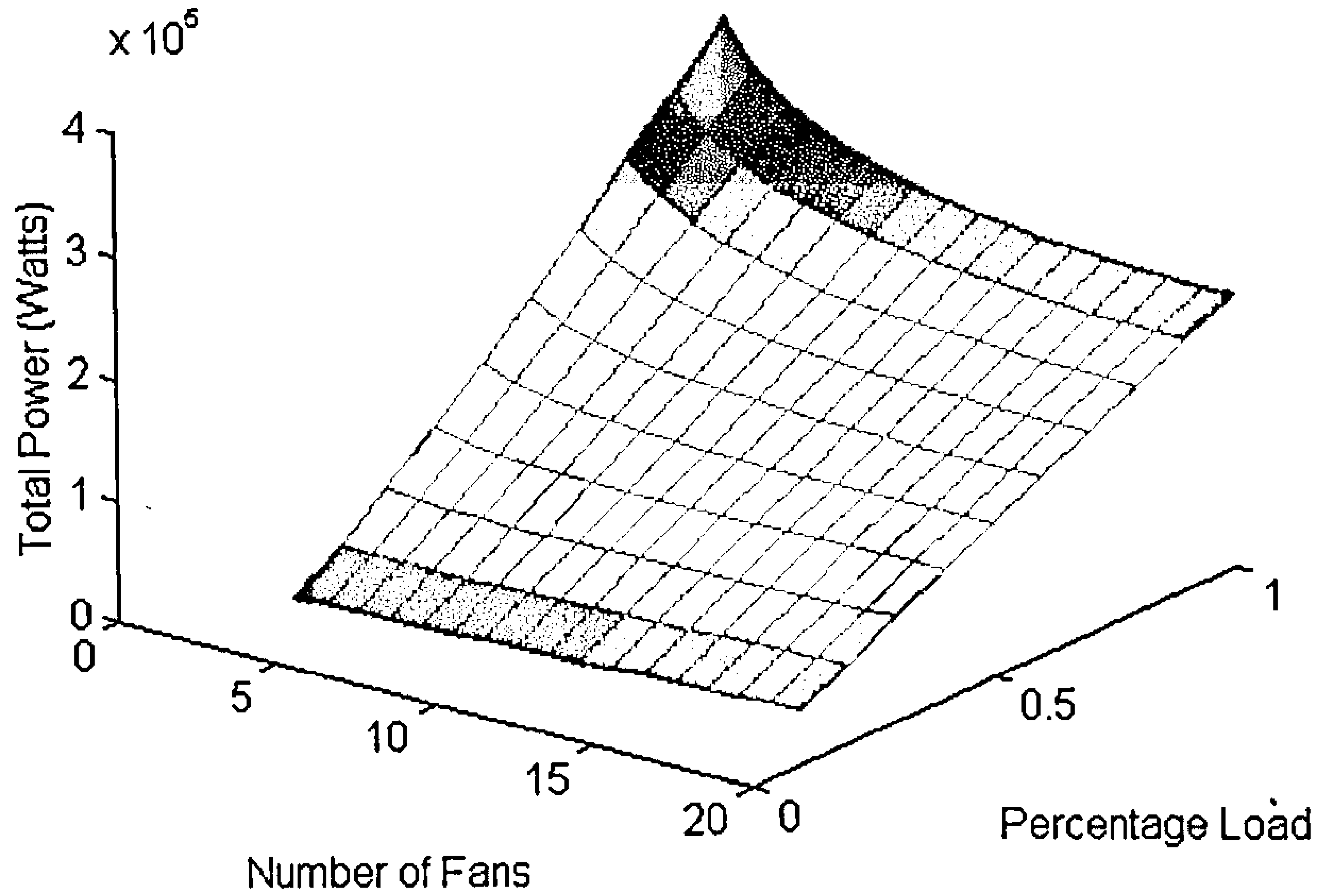


Figure 6.2 **Power consumption of refrigeration system over full range of operation with reducing condenser fans, 5°C ambient**

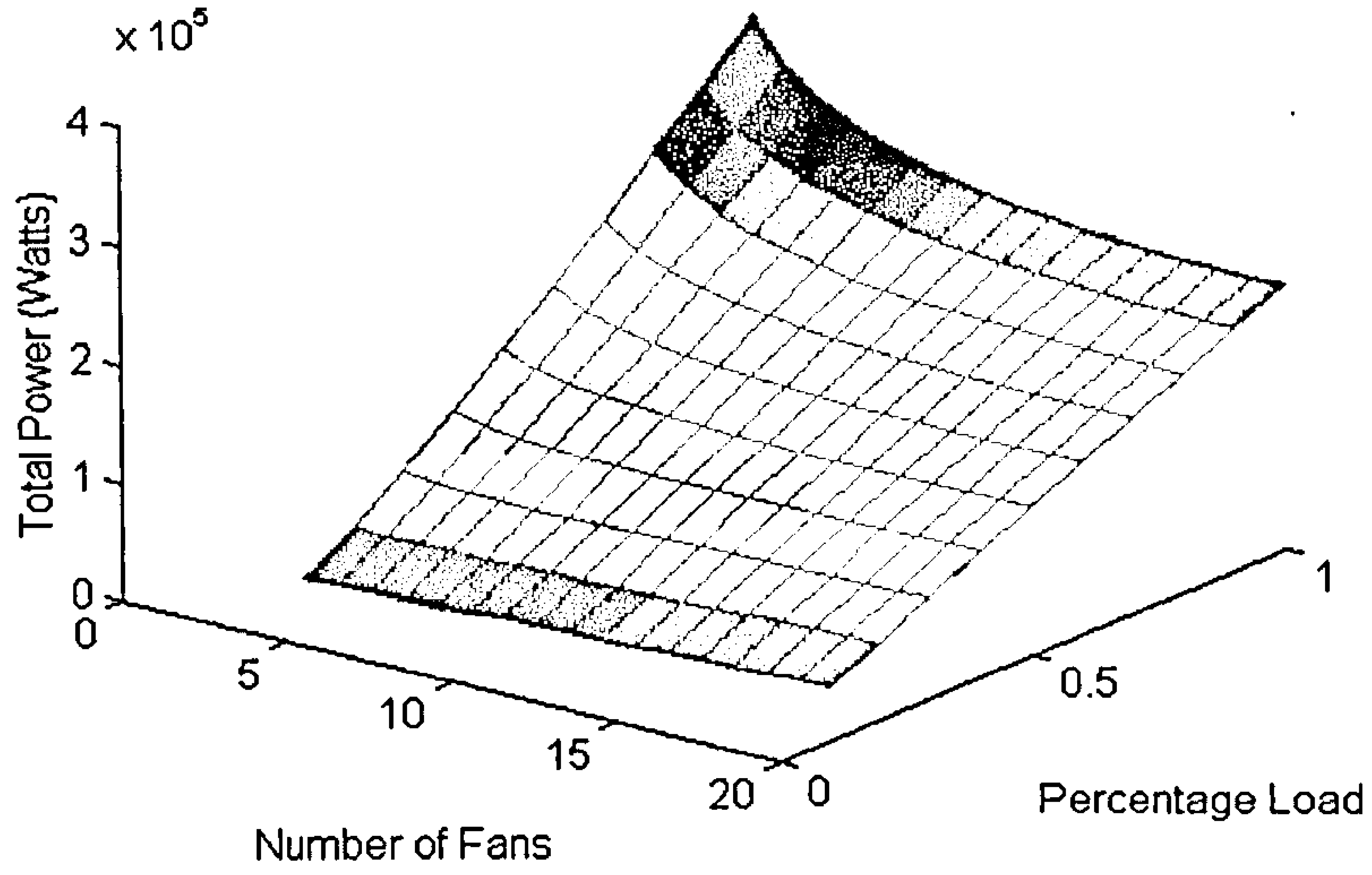


Figure 6.3 Power consumption of refrigeration system over full range of operation with reducing condenser fans, 15°C ambient

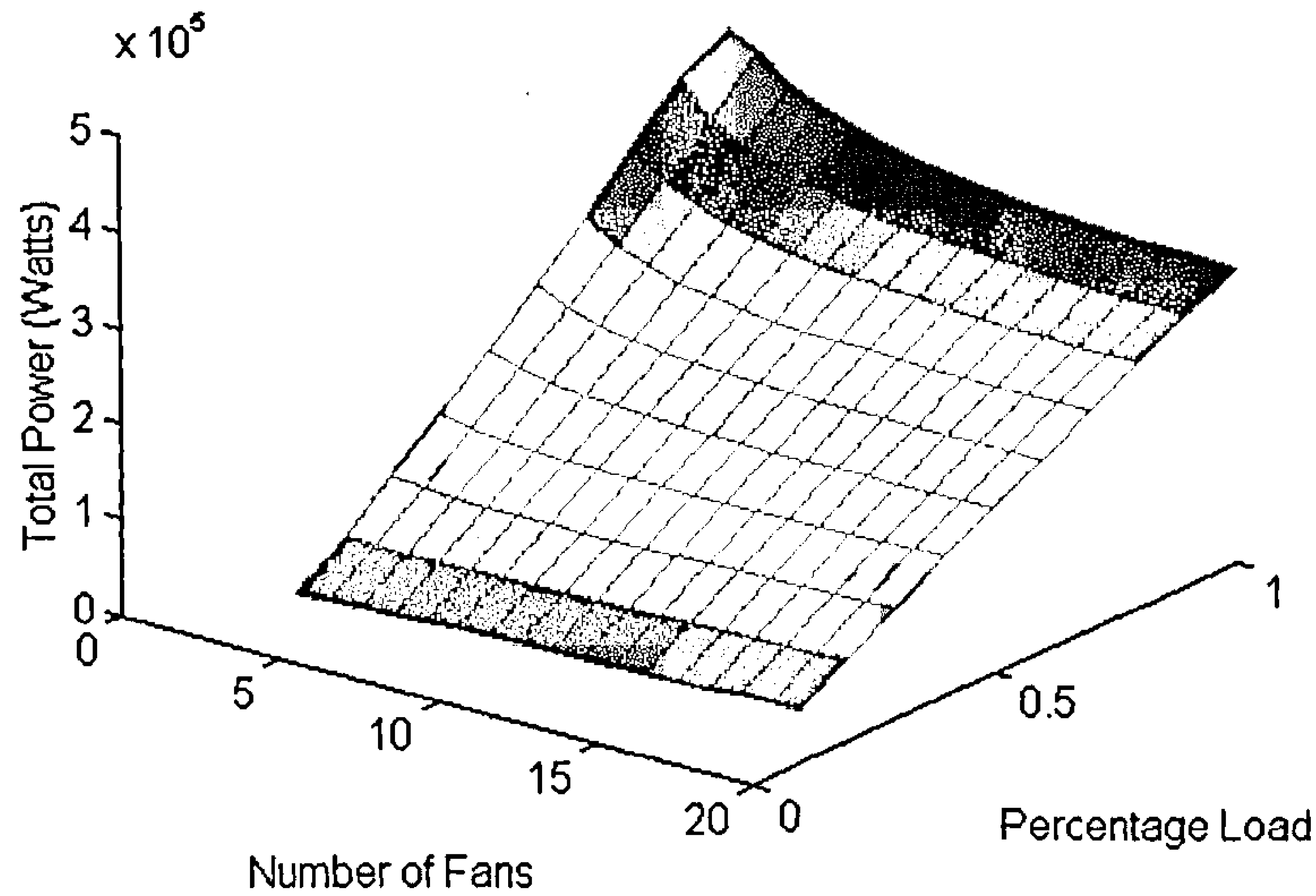


Figure 6.4 Power consumption of refrigeration system over full range of operation with reducing condenser fans, 25°C ambient

Figures 6.2 to 6.4 clearly show that for each condition there are a number of condenser fans that can be switched off before the compressors require an increase in power to maintain the refrigeration load.

The most substantial savings in energy are under part load conditions of 30% capacity. The results for this condition are shown in Figures 6.5 through 6.7 and are plotted at an appropriate scale. The results clearly show a distinct turning point after which any further reduction in the number of fans brings about a rise in the compressors power consumption.

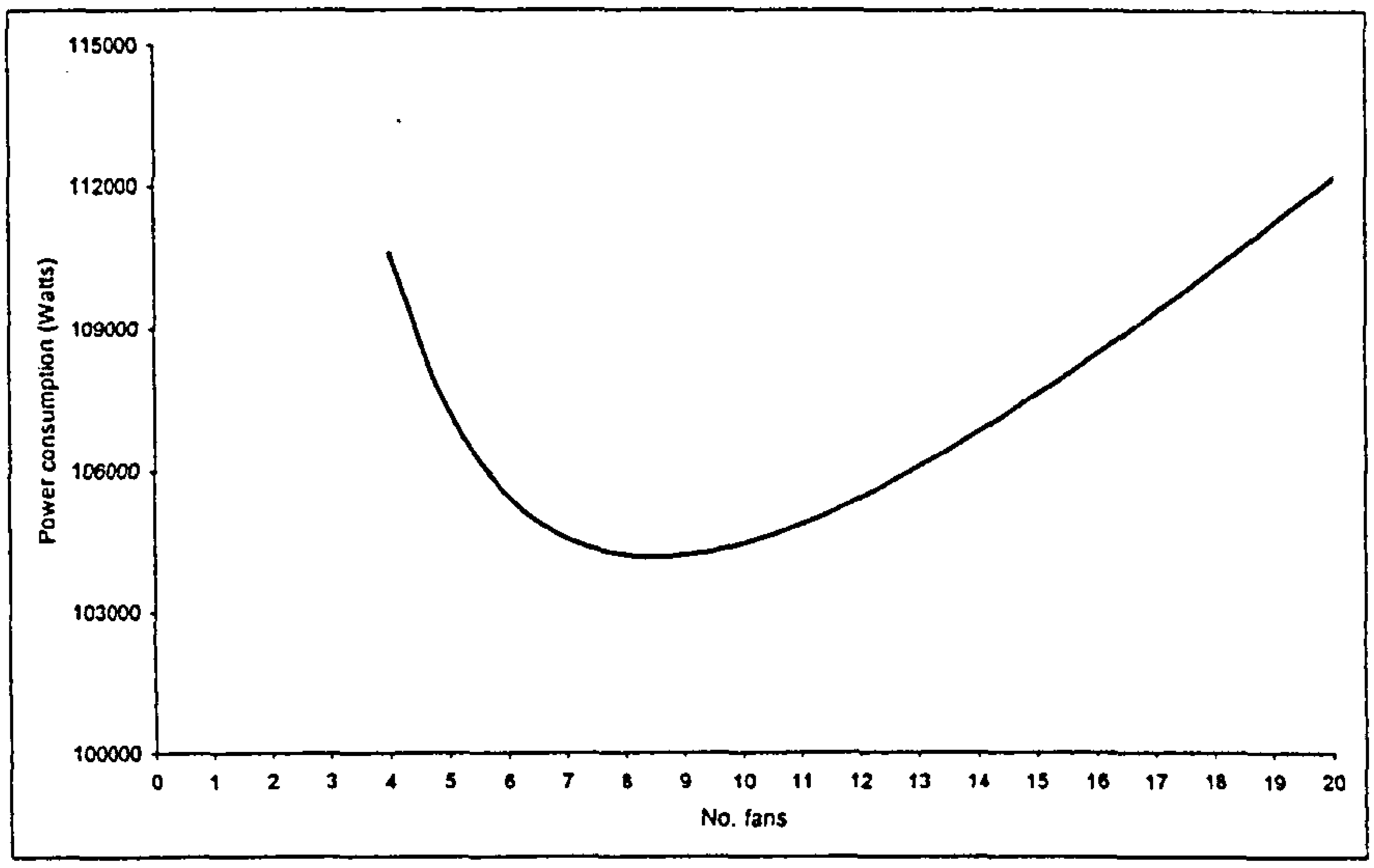


Figure 6.5 Total refrigeration power consumption at 30% load 25°C ambient

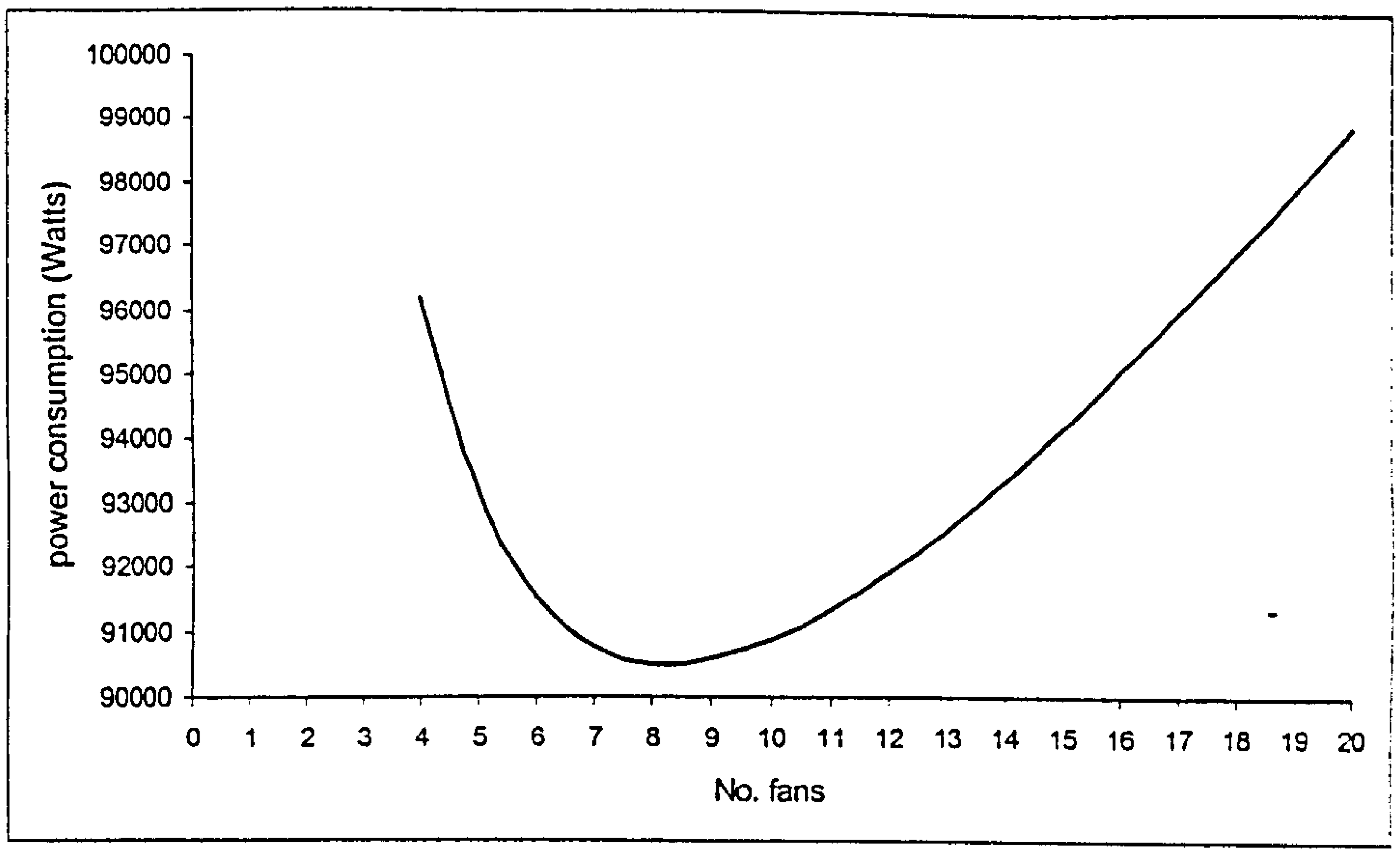


Figure 6.6 **Total refrigeration power consumption at 30% load 15°C ambient**

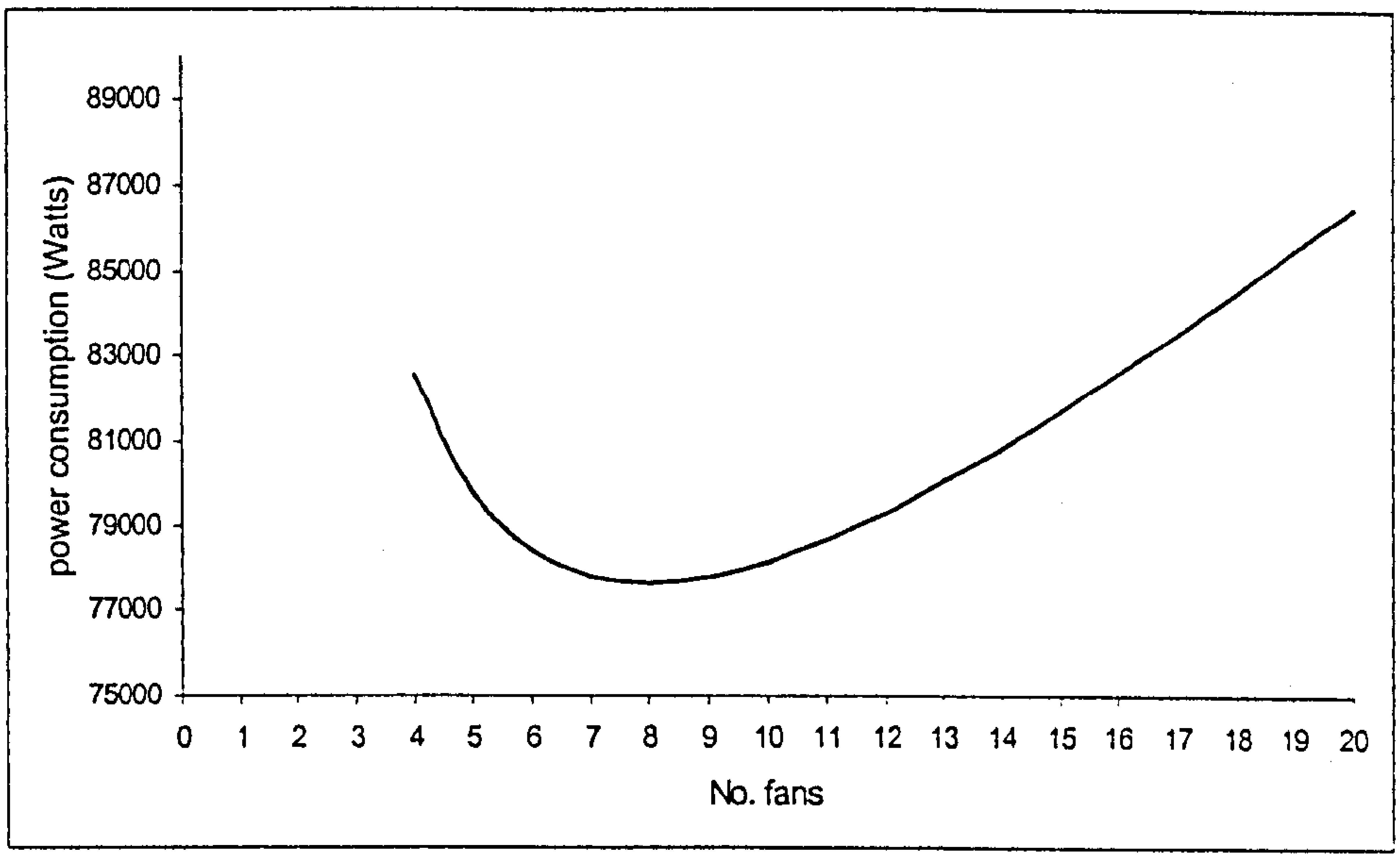


Figure 6.7 **Total refrigeration power consumption at 30% load 5°C ambient**

Figures 6.5, 6.6 and 6.7 show that at 30% load the lowest energy consumption of the refrigeration system occurs when there are eight condenser fans running. As the number of fans is reduced below this level there is a marked increase in power consumed by the system.

It is shown that there are possible savings available, if the number of condenser fans could be scheduled as a function of the refrigeration system load (a value that is available from the slide valve position) and the outside ambient air temperature. A simple rule-based algorithm to sequence the number of condenser fans running could be included in the control system software to accomplish this. The inclusion of such an algorithm to continuously monitor the refrigeration system load and the outside temperature, the energy consumption of the refrigeration plant would be maintained at an optimum at all points of operation. An example of the possible savings for a run of 8 hours duration where the plant is operating at 30% of full load capacity are shown in Table 6.1. The comparative power consumption for the optimised number of fans compared to the present control strategy that is employed shows a saving of 63.678 kWh a 7% power saving.

Present control strategy	**Optimised control strategy**	**Saving**
896.64 kWh	833.872 kWh	63.678 kWh

Table 6.1 Comparison of present fan control strategy to optimised strategy, for refrigeration system operating at 30% load, 25°C outside ambient for 8 hours.

6.2 Pre-conditioning of vehicle

The normal procedure for the conditioning of a vehicle for testing is to allow it to "soak" for 10 hours at the same conditions at which the test is to be performed.

In order to soak the vehicle for this time involves the refrigeration system and circulating pumps operating continuously over this period. It is possible to condition a vehicle within the test chamber itself taking advantage of the larger cooling capacity and the increased heat transfer to / from the vehicle afforded by the forced convection of the air stream passing over the vehicle.

To analyse any possible savings the data used for the –30°C pull down test (chapter 5, section 5.3.3.) is used for both the test chamber and the soakroom. The soakroom is allowed to operate for its normal 10 hour period, whilst the test chamber conditioning is run for 2½ hours.

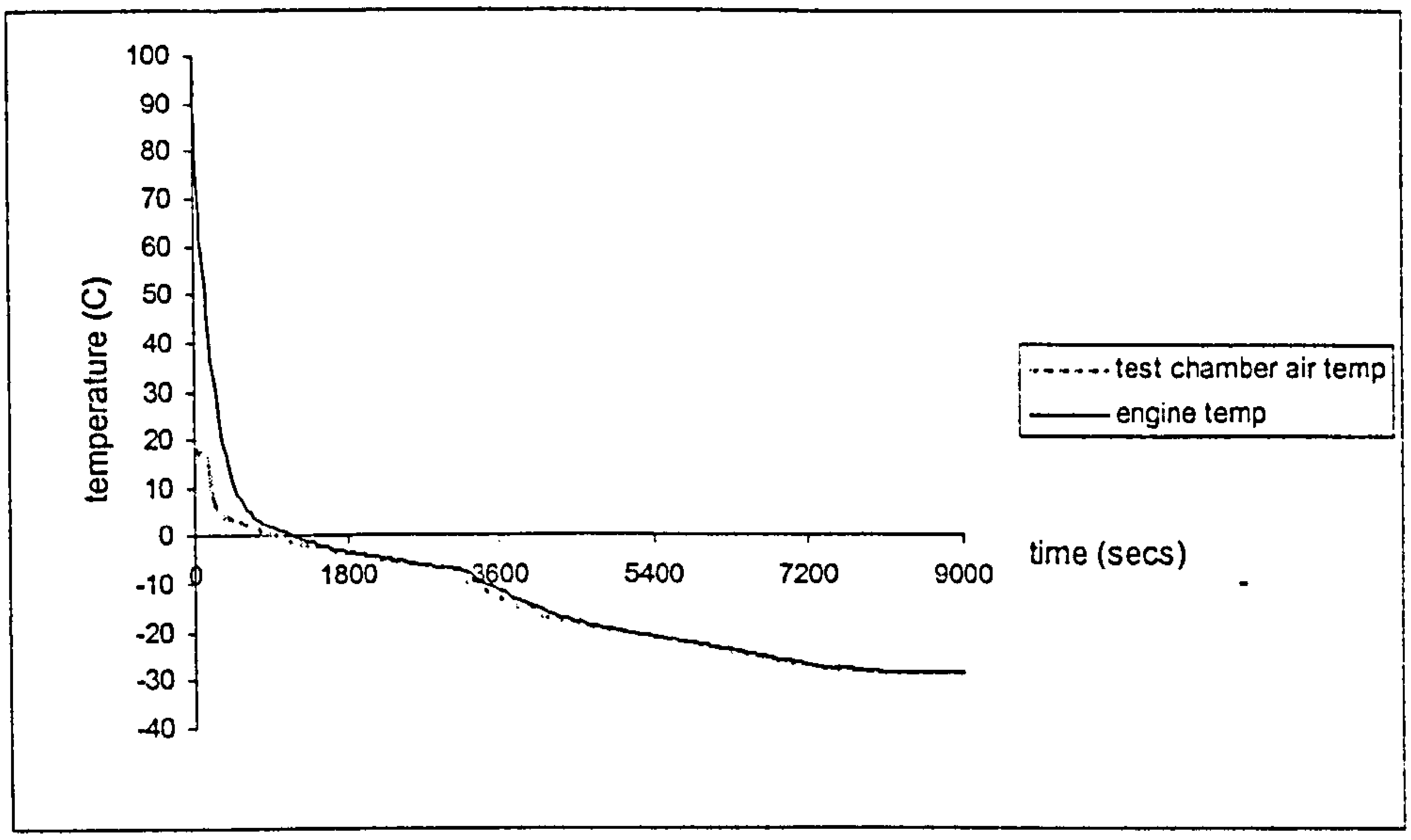


Figure 6.8 Predicted air and vehicle temperatures for conditioning of vehicle to –30°C in test chamber with wind speed of 120 km/h

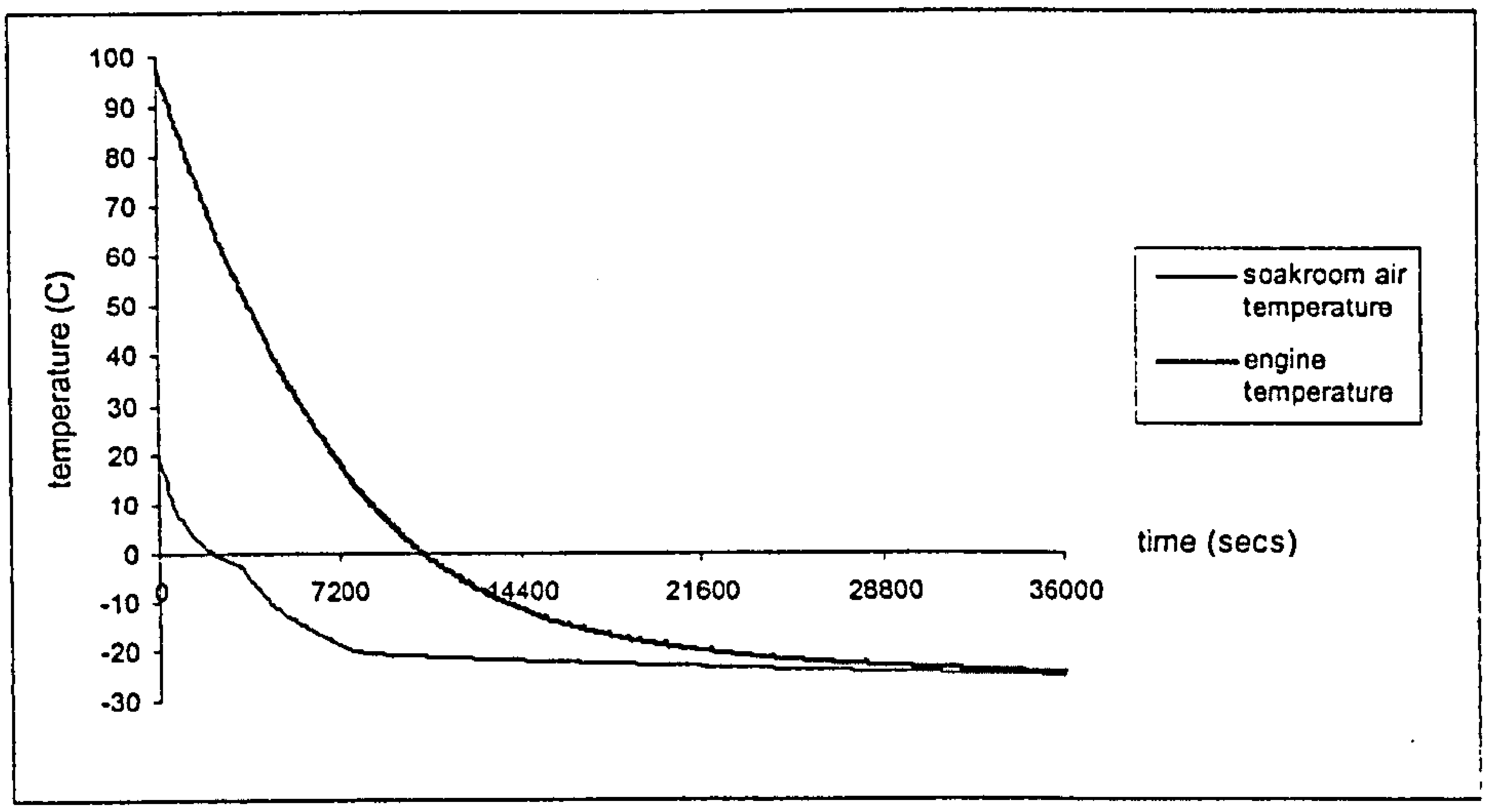


Figure 6.9 Predicted air and vehicle temperatures for conditioning of vehicle to –30°C in soakroom

From Figure 6.8 it can be seen that the vehicle temperature closely tracks the air temperature once the air stream has removed the majority of the heat from the engine. With the input data available the vehicle and air stream reach a temperature of –28°C at the end of the 2½ hour conditioning period. In comparison the air temperature in the soakroom Figure 6.9, is unable to reach a temperature lower than –25°C. Taking this into account it is clear from the air temperature profile that the air reaches a stable temperature of –25°C after 2 hours but the vehicle takes a further 8 hours until it has similar condition.

Initial analysis of the results suggest that conditioning the vehicle in the test chamber would not only be time saving but also considerably reduce the energy required to precondition a vehicle. Whilst the soakroom is run for a much longer period the test chamber uses a 385 kW fan to produce an air stream of the 120 km/h used for the simulation this alone affects the energy consumption picture drastically.

Figures 6.10 and 6.11 show a break down of the calculated energy consumption of both the soakroom and test chamber in the pre conditioning simulation. The total energy consumption shown is made up of two components. The first is the compressor power consumption used in the cooling and maintaining the temperature of the trichloroethylene, this power reduces as the setpoint is approached to a steady level where the trichloroethylene is maintained at its set condition. The second component is the power consumed by the ancillary components. The ancillary components include the pumps and fans used in each system, their power consumption is steady throughout the simulation period. Figure 6.12 shows a full breakdown of the proportion of energy used by all the system components in the test chamber simulation and Figure 6.13 shows the breakdown of energy for the soakroom simulation.

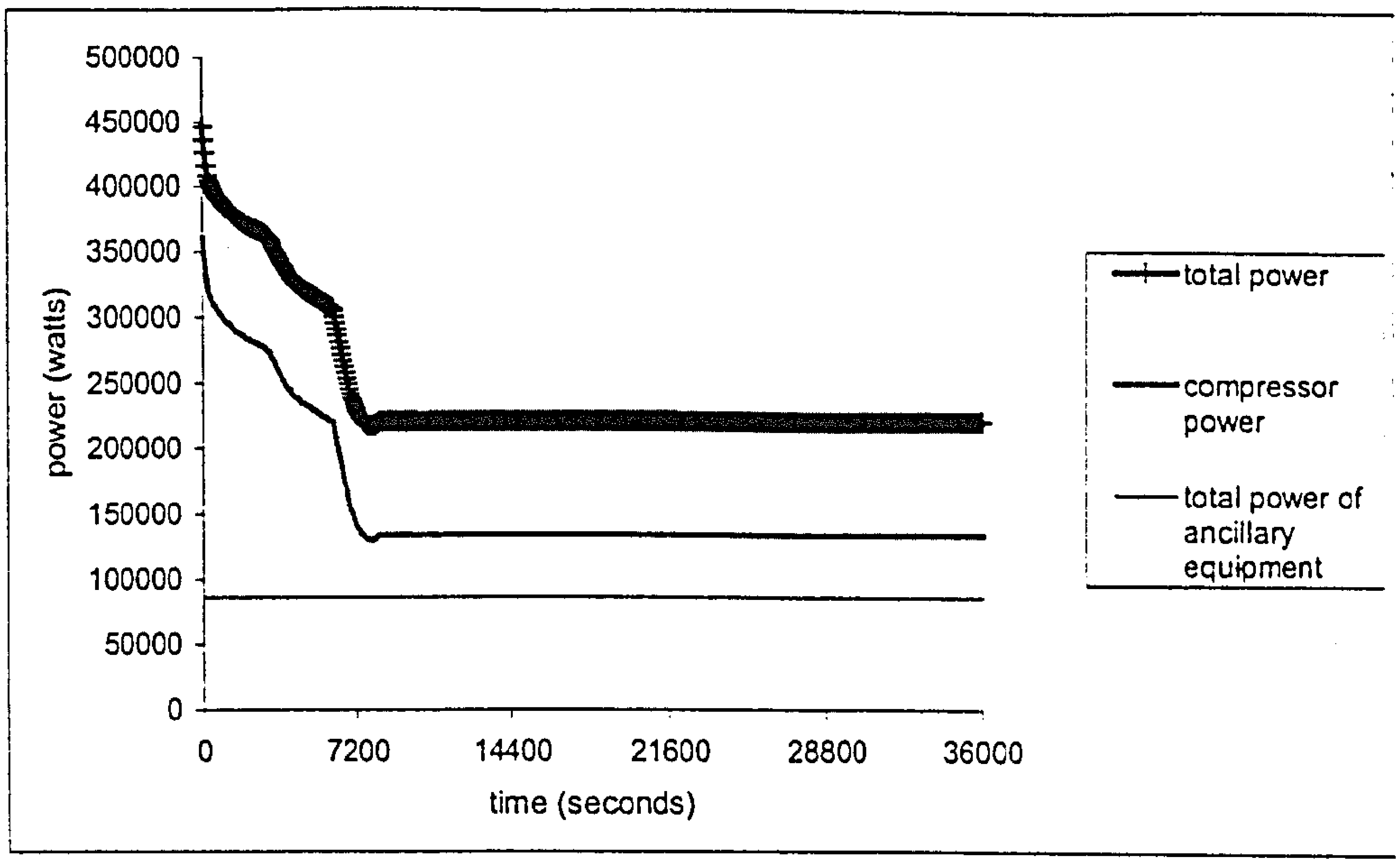


Figure 6.10 Predicted power consumption for conditioning of vehicle to –30°C in soakroom

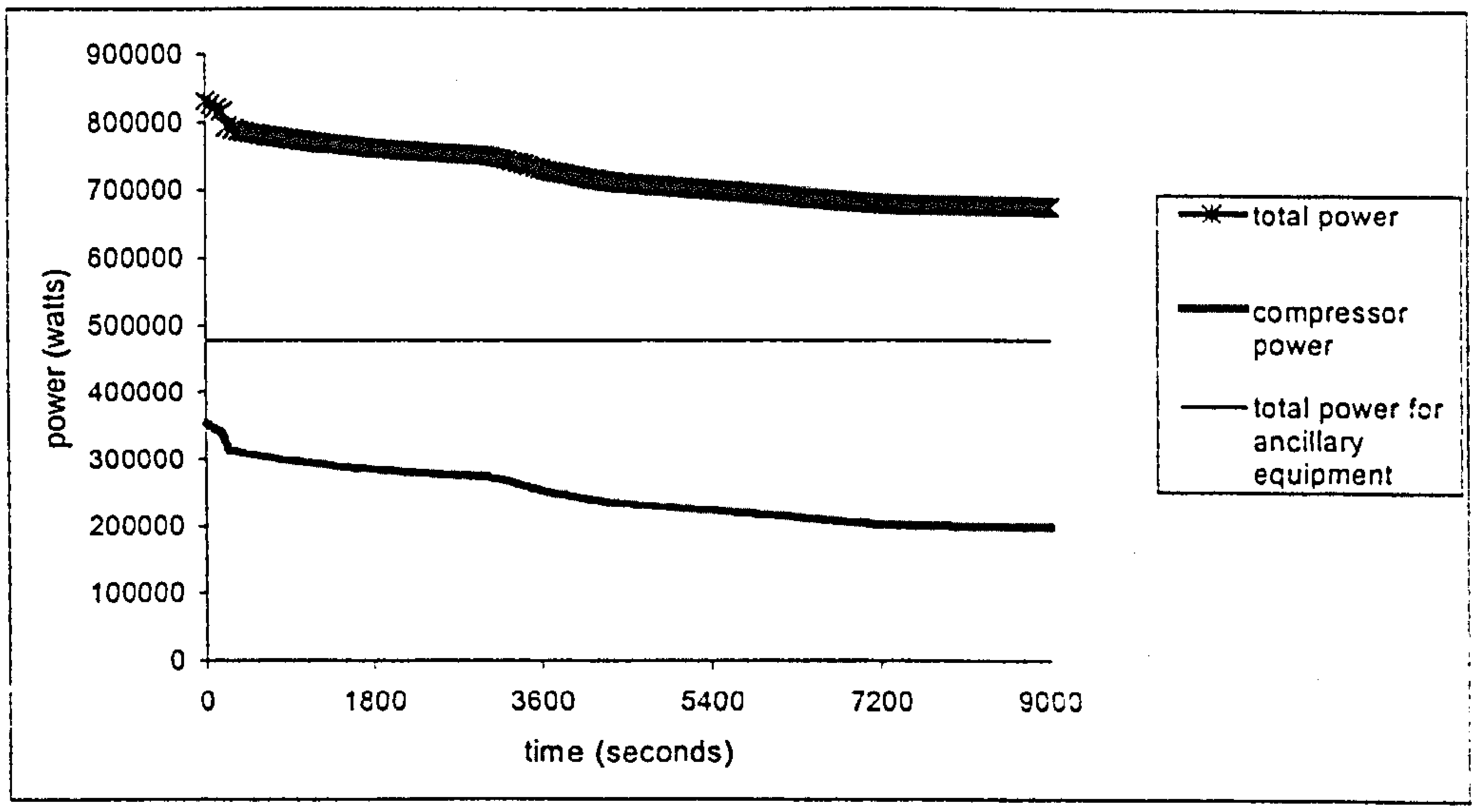


Figure 6.11 Predicted power consumption for conditioning vehicle to –30°C in test chamber

Test chamber	Soakroom	Saving
1580 kWh	2450 kWh	870 kWh

Table 6.2 Predicted power consumption of test chamber and soakroom for the conditioning of vehicle to –30°C

Table 6.2 shows the calculated energy consumption of both the test chamber and the soakroom whilst conditioning a vehicle to –30°C. It can be seen that making use of the extra heat transfer afforded by the air stream in the test chamber reduces the amount of energy required to pre condition the vehicle by 870 kWh or 35%.

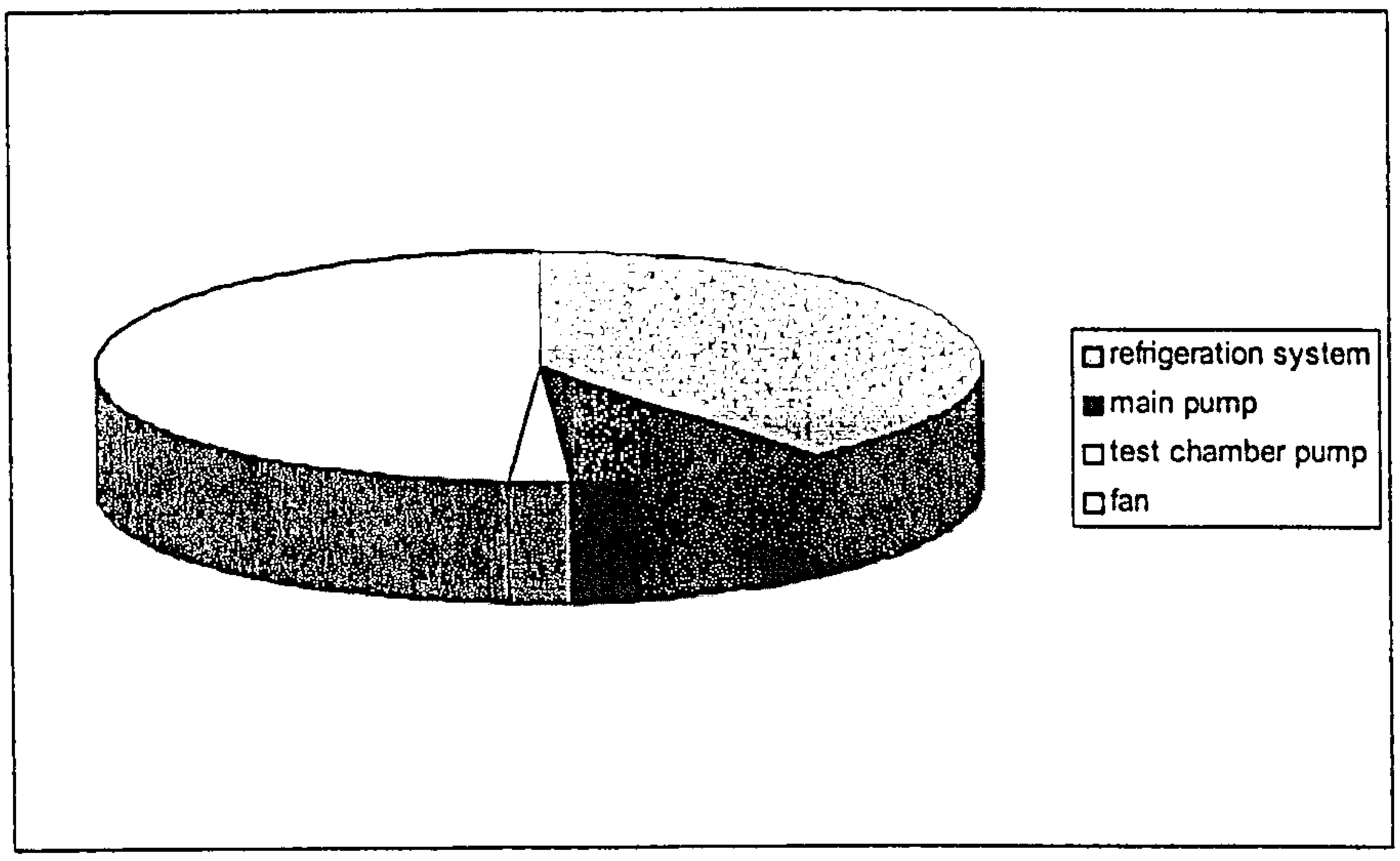


Figure 6.12 Breakdown of energy in the test chamber simulation

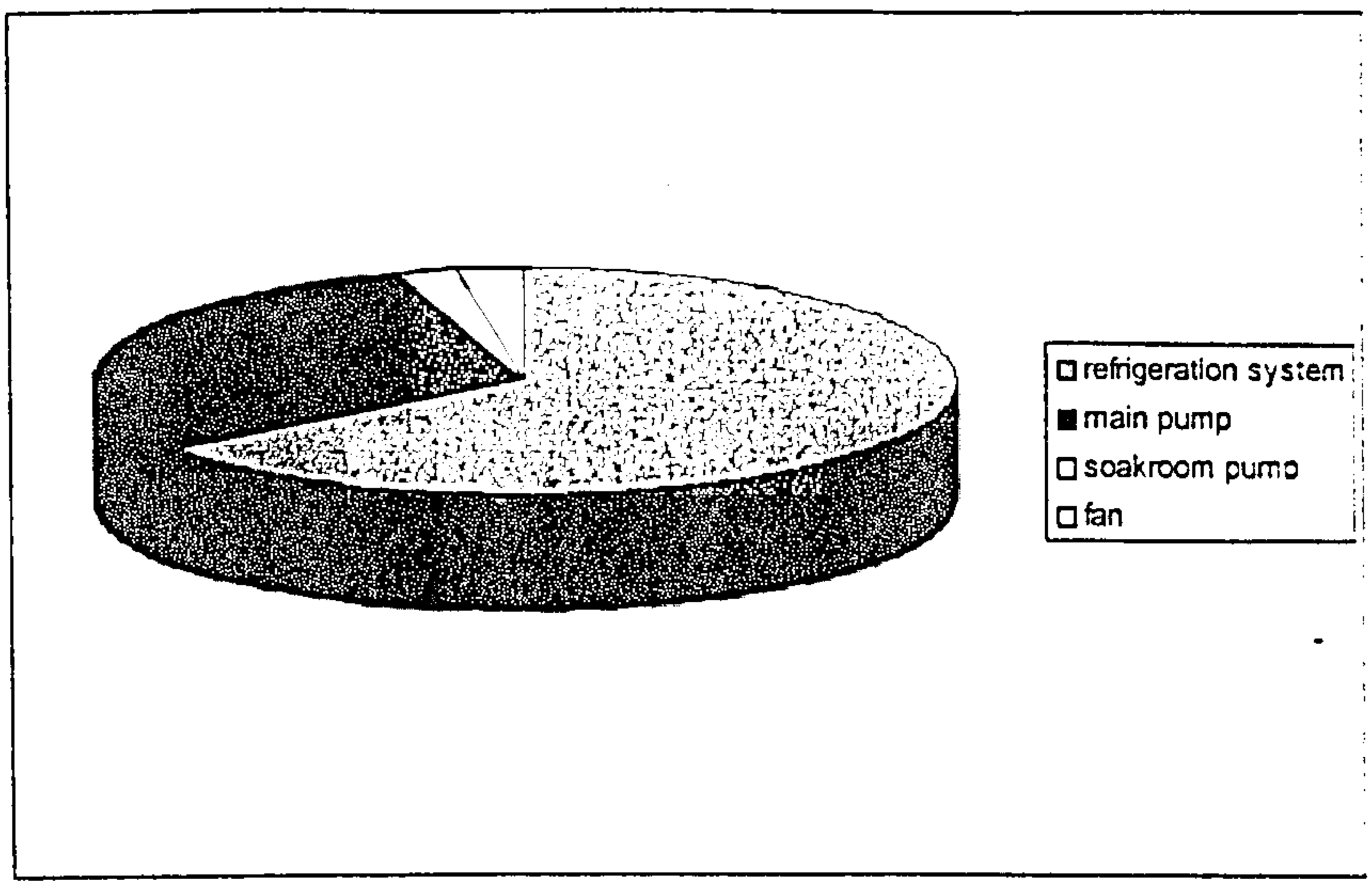


Figure 6.13 Breakdown of energy in the soakroom simulation

6.3 Reduction in flow temperature differential set point

The flow temperature for the trichloroethylene used within the Climatic Wind Tunnel systems is set at 15K below the required test condition. It is possible that an energy saving will be offered by reducing this temperature differential to 10K through set point on the flow temperature controller. It must be ensured that the test chamber and soakroom performance is not affected by this alteration to flow temperatures.

The simulations again concentrate upon a pull down test to –30°C. The flow temperatures out of the refrigeration plant and into the heat exchangers are set at –40 and –45°C. From these two conditions the resultant temperatures and energy consumption for each case can be calculated.

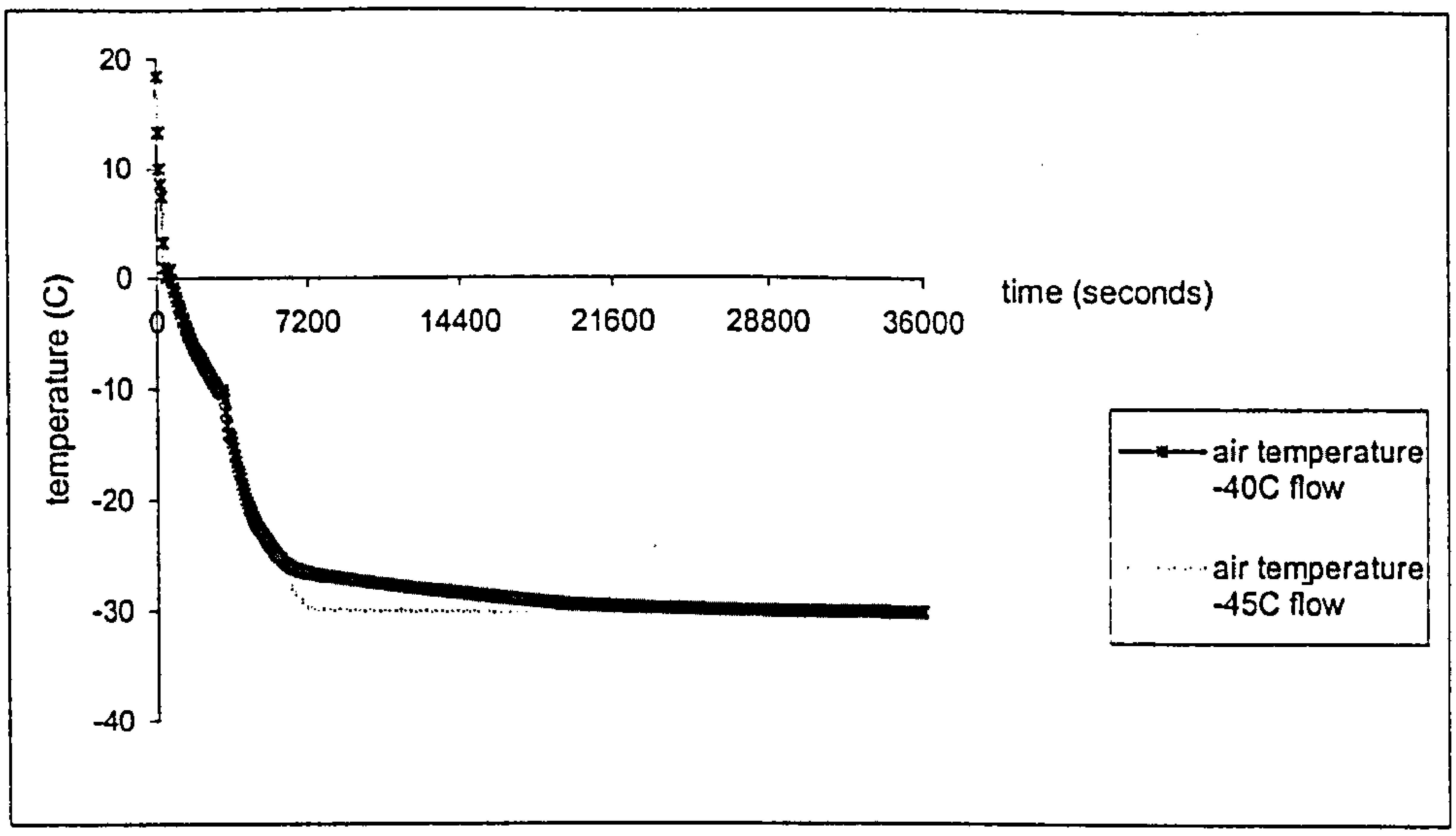


Figure 6.14 comparison of predicted soakroom air temperatures resulting from –40 and –45°C flow temperature into heat exchanger

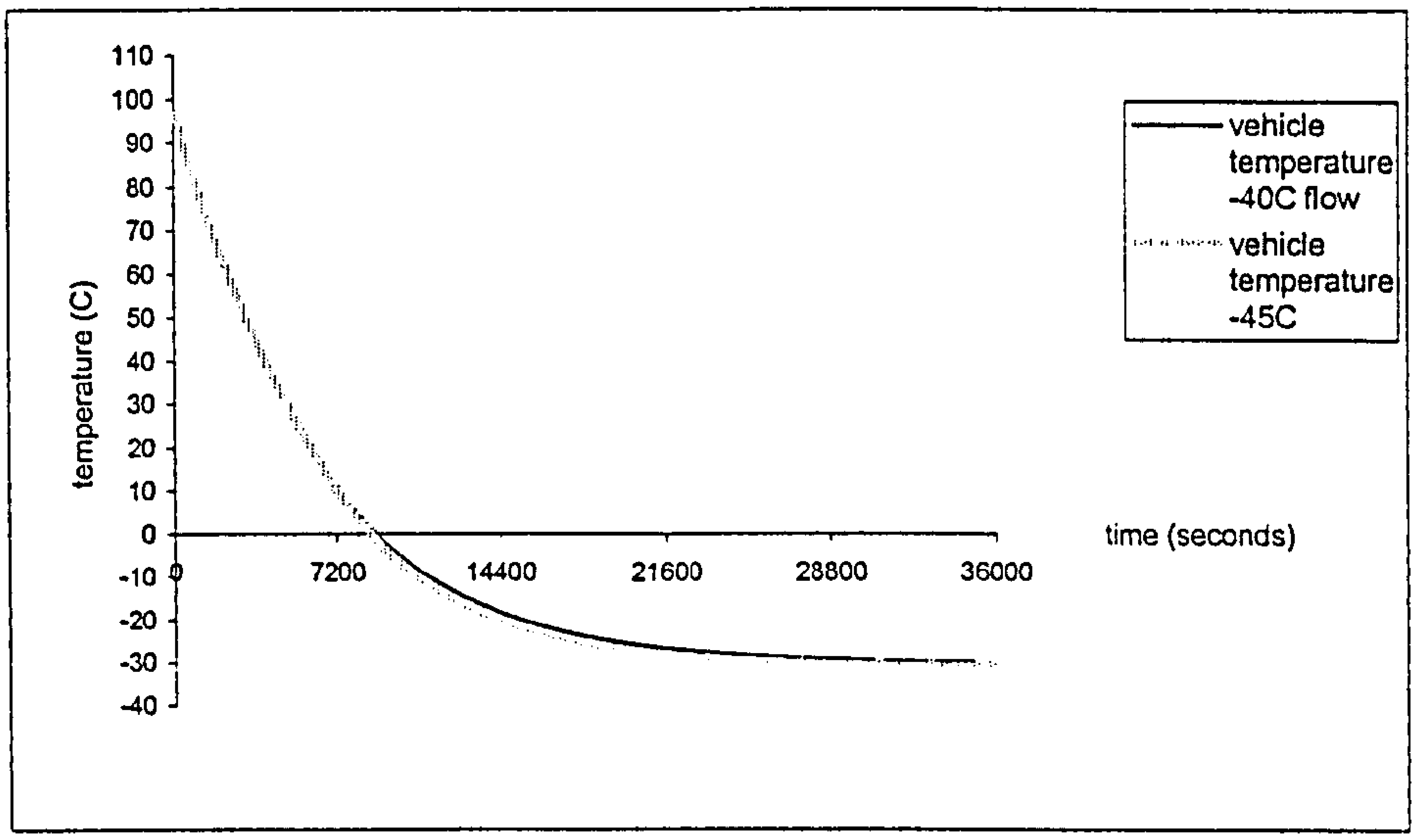


Figure 6.15 Comparison of predicted vehicle temperature in soakroom resulting from –40 and –45°C flow temperature into heat exchanger

Figures 6.14 and 6.15 show that there is no significant difference between operating the soakroom with a flow to air temperature differential of 10K rather than the 15K used. It can be seen from Figure 6.12 that the air temperature when using a 15K temperature difference reaches the set point quicker than when using a 10K differential, but the vehicle temperatures illustrated in Figure 6.13 show that there is little difference in the cooling rate of the vehicle.

-45°C flow	-40°C flow	Saving
2600 kWh	1850 kWh	750 kWh

Table 6.3 Predicted soakroom energy consumption for setpoint of –30°C and flow temperature differentials of 15 and 10K

Table 6.3 shows the comparison of predicted energy consumption for the soakroom during a pull down test to –30°C. It is shown that by reducing the temperature differential between the trichloroethylene flow and the air that a potential to save 750 kWh on this test exists, this equates at a 30% saving in the energy required to condition a vehicle to –30°C.

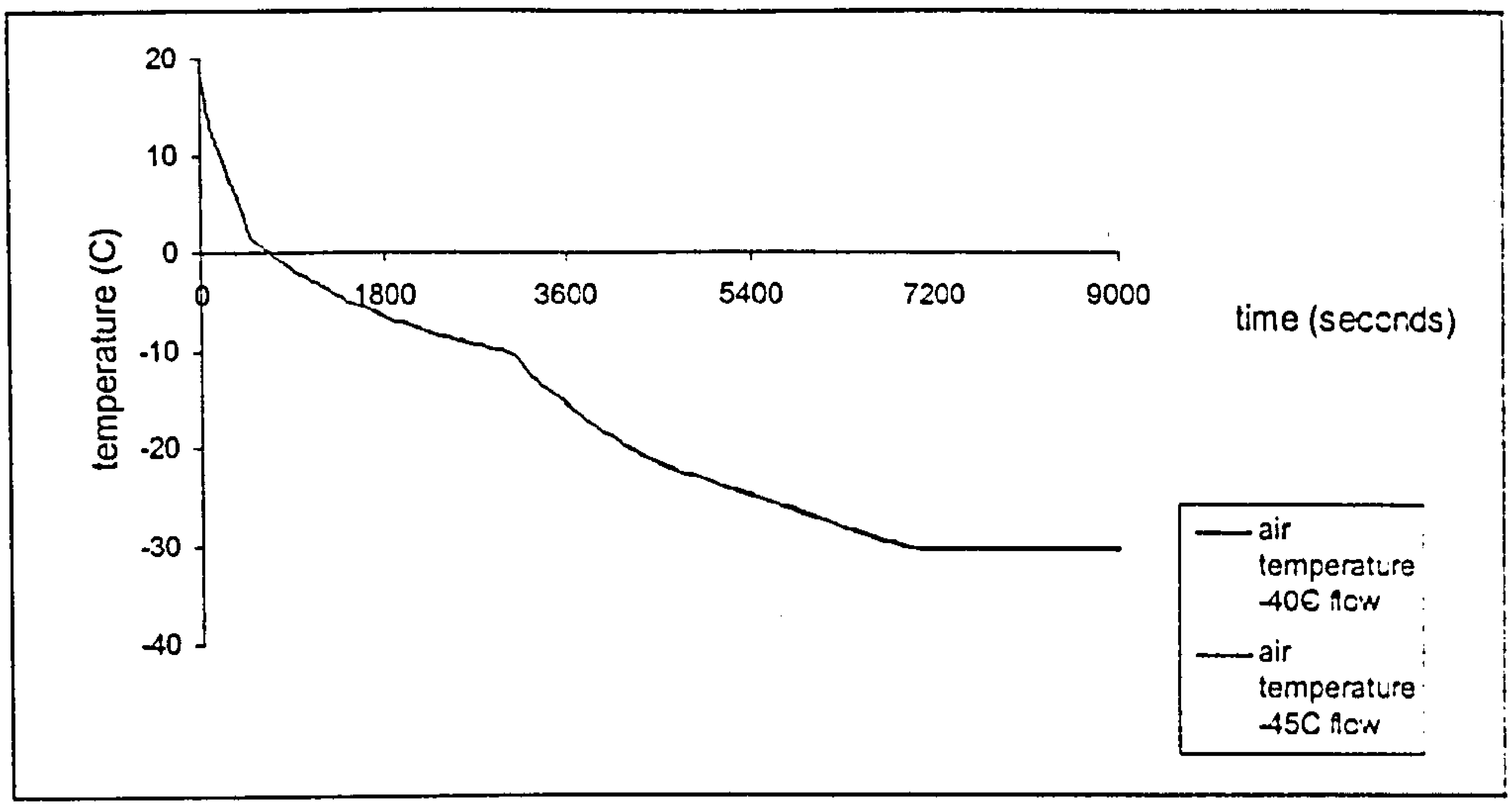


Figure 6.16 Comparison of predicted test chamber air temperatures resulting from –40 and –45°C flow temperature into heat exchanger

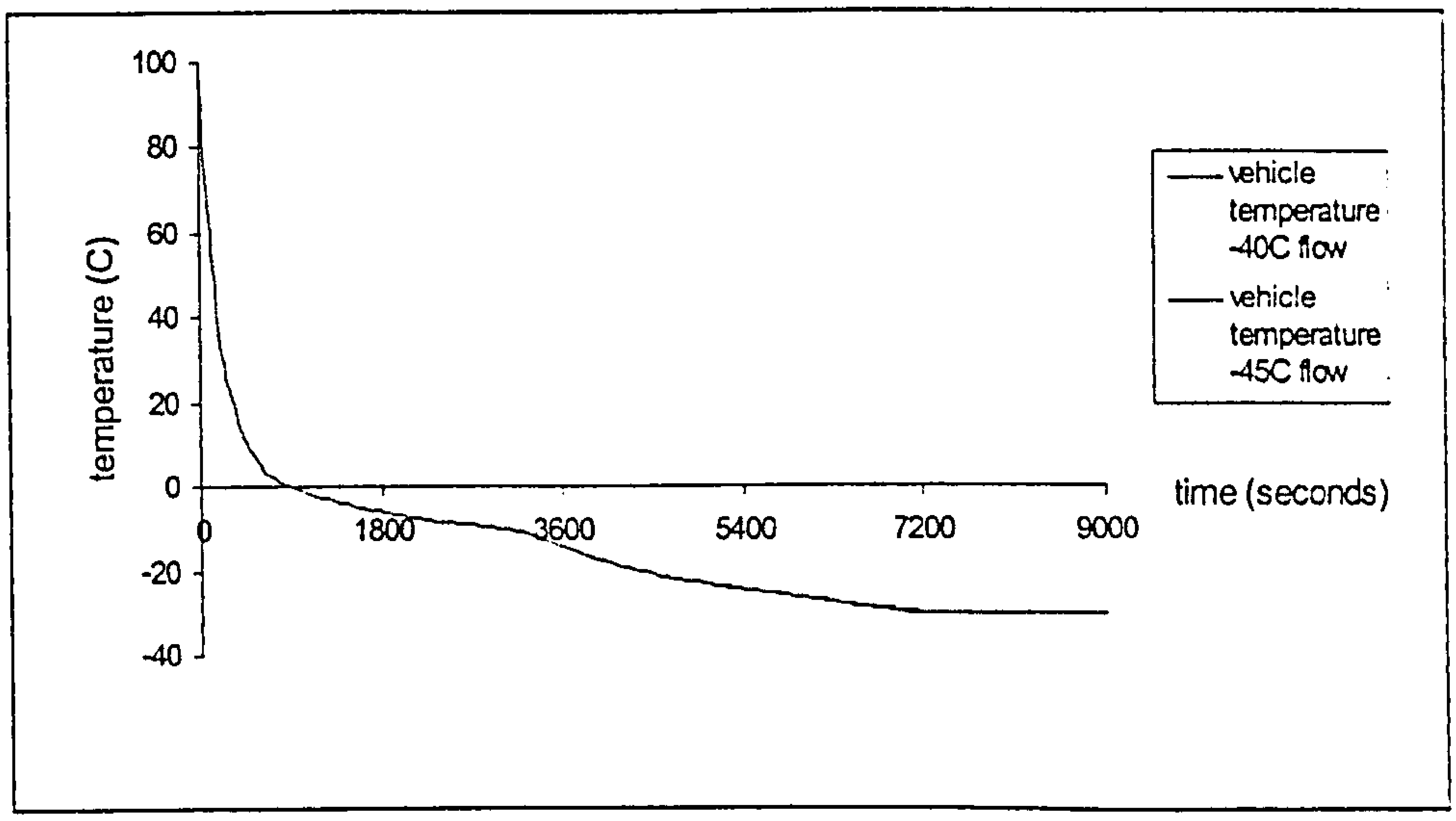


Figure 6.17 Comparison of predicted vehicle temperature in test chamber resulting from –40 and –45°C flow temperature into heat exchanger

-45°C flow	-40°C flow	Saving
1750 kWh	1640 kWh	110 kWh

Table 6.4 Predicted test chamber energy consumption for setpoint of –30°C and flow temperature differentials of 15 and 10K

Table 6.4 shows the comparison of predicted energy consumption for the test chamber during a pull down test to –30°C. It is shown that by reducing the temperature differential between the trichloroethylene flow and the air saves 110 kWh, an 8% reduction in the energy required to maintain a 15K temperature difference. With the test chamber being used more frequently and for longer periods than the soakroom the overall potential energy saving is far greater.

From Figures 6.16 and 6.17 it can be seen that a reduction in the temperature differential from 15 to 10K has no discernible effect on the performance of the air or vehicle temperature pull down rate for this set of operational conditions.

6.4 Increase in insulation thickness for the soakroom and test chamber

The soakroom and test chamber walls are constructed from insulated polyurethane panels of 100 mm thickness. This simulation is concerned with increasing the insulation thickness from 100 to 200 mm and its effect on the pull down time of the air and on the energy consumption of the Climatic Wind Tunnel systems. The 100 mm thick panel construction has a U value from 0.25 W/m^2K, doubling the insulation thickness to 200 mm will reduce this value to 0.125 W/m^2K.

The simulation is for a –30°C pull down test using a flow temperature into the heat exchanger of –40°C, the soakroom is run for 10 hours and the test chamber for 2½ hours.

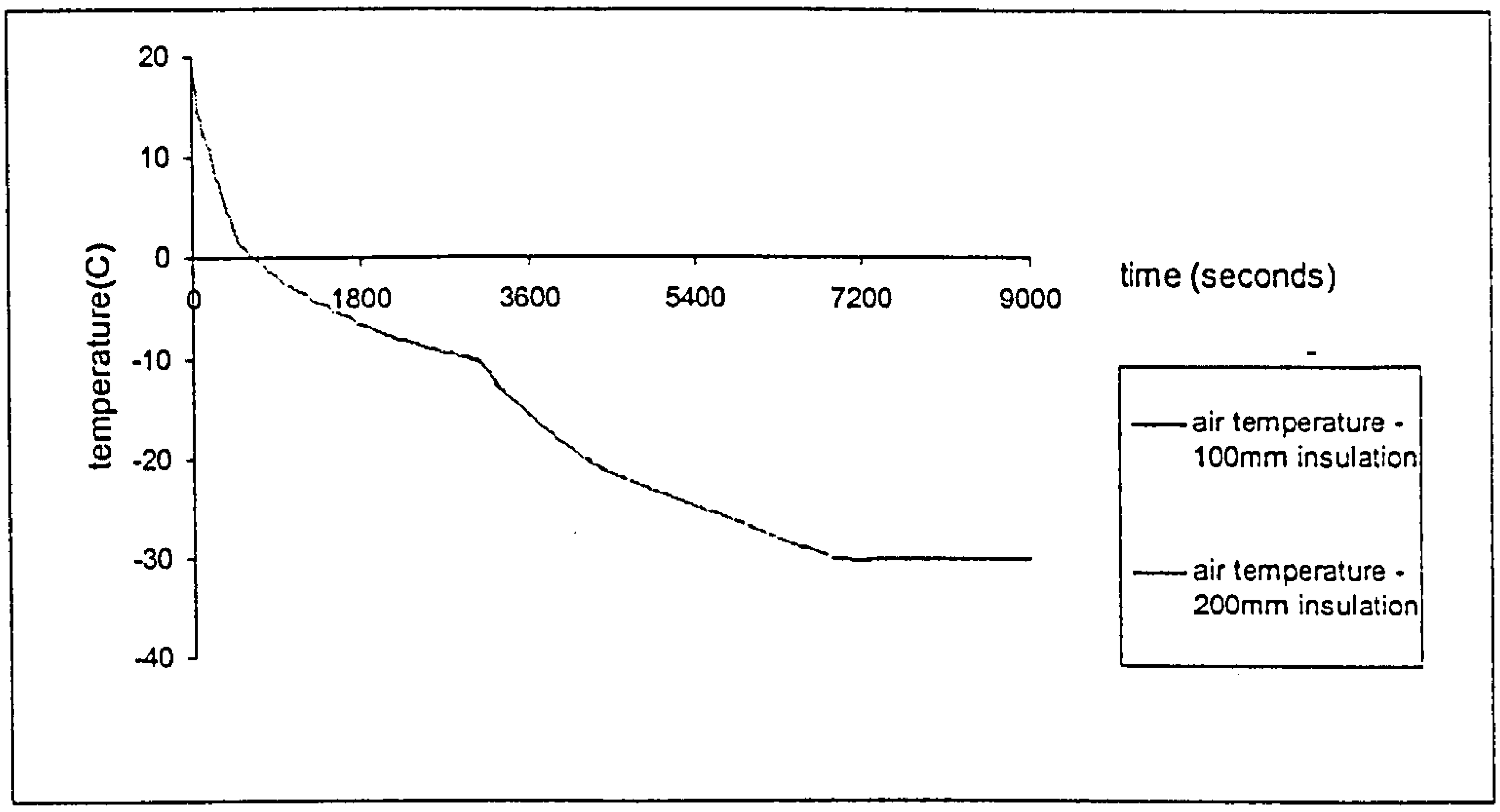


Figure 6.18 comparison of predicted soakroom air temperatures resulting from increasing the insulation thickness from 100 mm to 200 mm

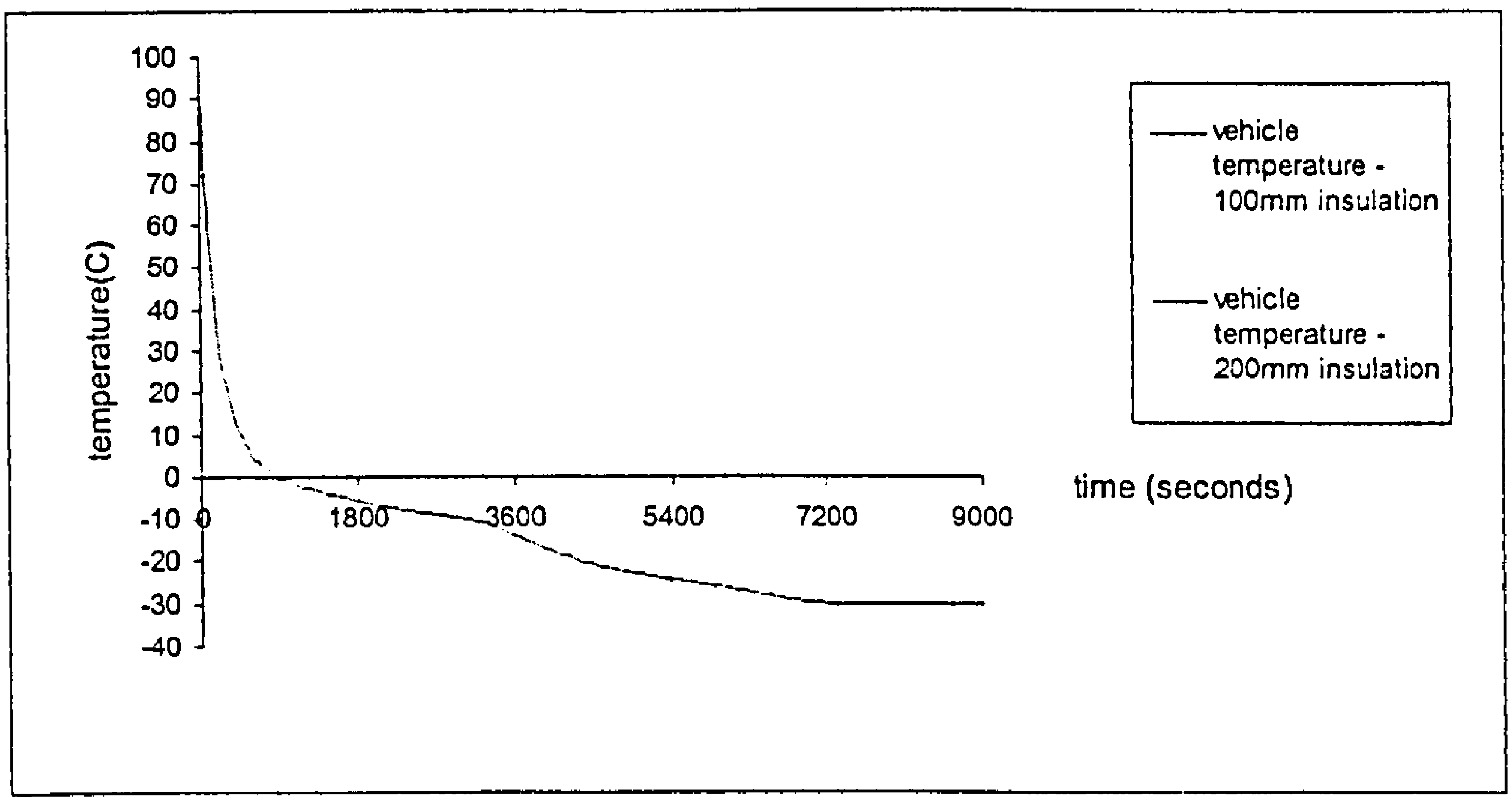


Figure 6.19 Comparison of predicted vehicle temperature in soak room resulting from increasing the insulation thickness from 100 mm to 200 mm

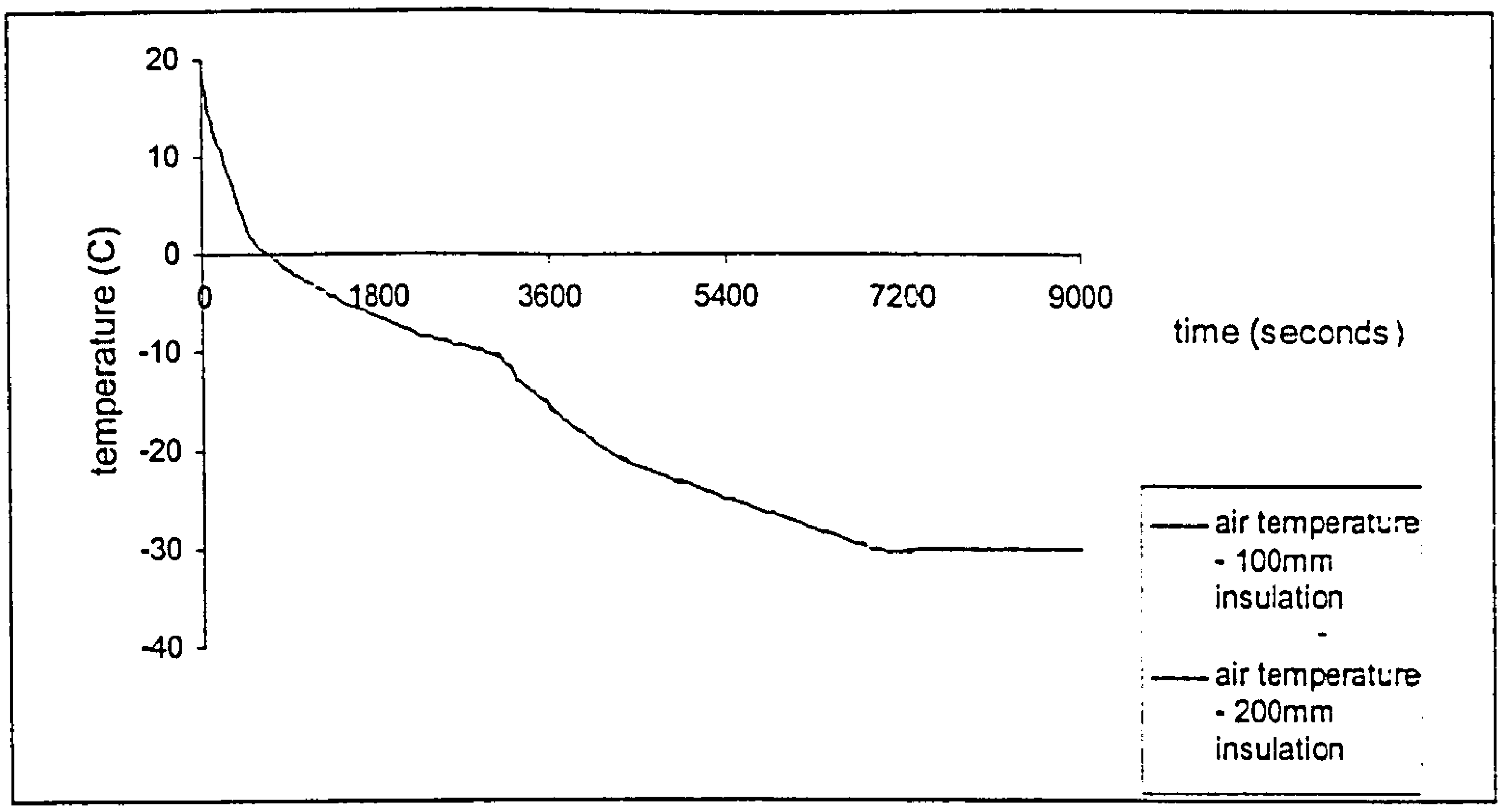


Figure 6.20 comparison of predicted test chamber air temperatures resulting from increasing the insulation thickness from 100 mm to 200 mm

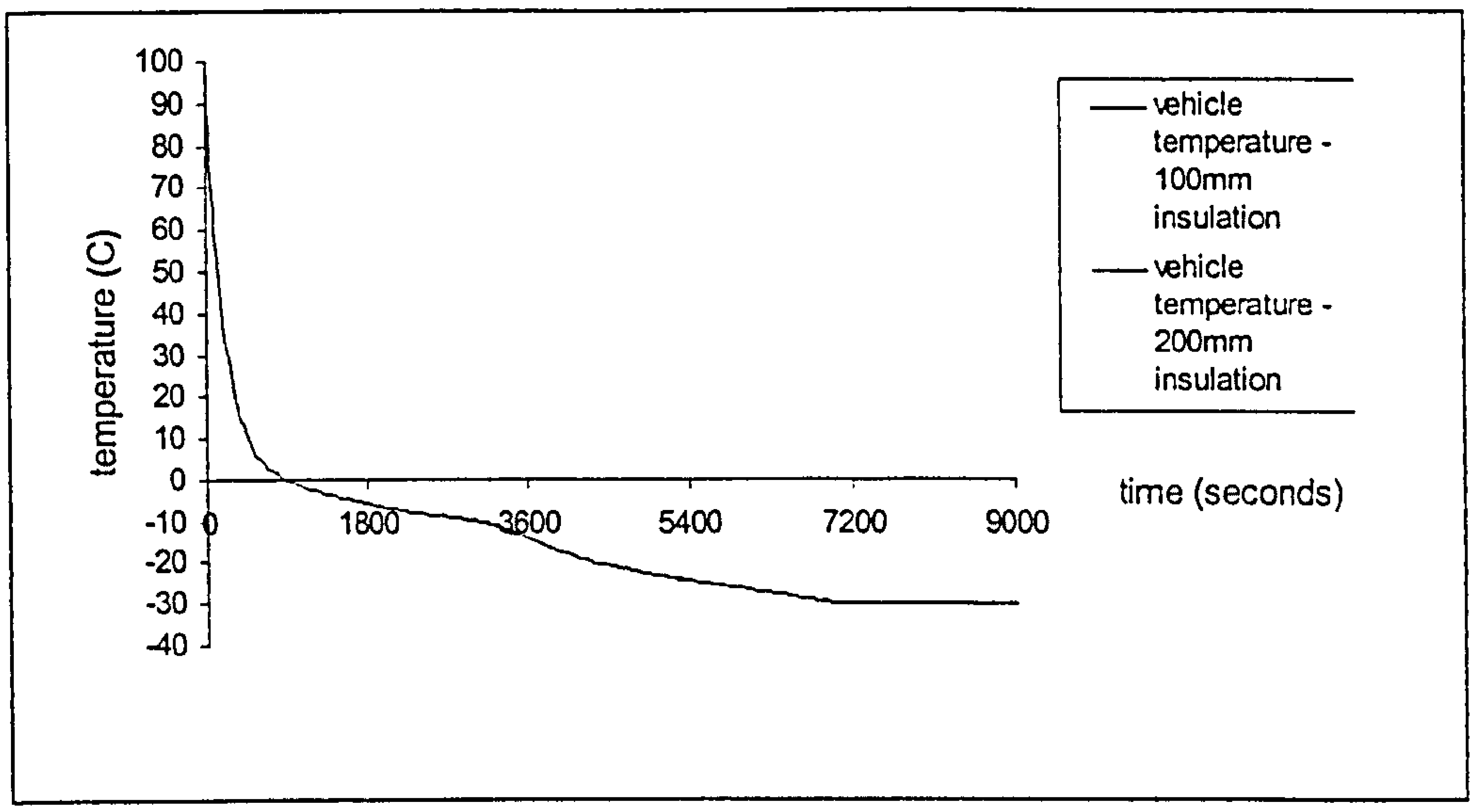


Figure 6.21 Comparison of predicted vehicle temperature in test chamber resulting from increasing the insulation thickness from 100 mm to 200 mm

100 mm insulation	200 mm insulation	Saving
1850 kWh	1755 kWh	95 kWh

Table 6.5 Predicted soakroom energy consumption for setpoint of –30°C and increase in insulation from 100 to 200 mm

100 mm insulation	200 mm insulation	Saving
1640 kWh	1590 kWh	50 kWh

Table 6.6 Predicted test chamber energy consumption for setpoint of –30°C and increase in insulation from 100 to 200 mm

Tables 6.5 and 6.6 show the predicted power consumption for the soakroom and the test chamber using 100 mm and 200 mm insulated panels. It can be seen that despite there being very little difference in the resulting air and vehicle temperatures (Figures 6.18, 6.19, 6.20 and 6.21) that for this set of operational conditions there is a saving of 95 kWh for the soakroom and 50 kWh for the test chamber. If the case of this saving is taken for an entire years operation for the test chamber (350 days) the minimum saving in power consumption would be 120,000 kWh. The increase in the insulation thickness would add to the construction cost of a CWT facility but the savings in the energy cost would offer a relatively short payback period for the investment.

6.5 Validity of results

It should be appreciated that the results from the studies have been achieved using models that involve simplifications of the actual physical processes occurring. Whilst it has been shown from the validation studies conducted that the systems constructed from these models give good agreement with the actual results recorded during tests, some deviations were highlighted.

The models of the systems used in these studies gives good indication as to areas in which substantial savings in running costs could be made, for more accurate prediction more detailed models would be required, one particular area where this is true is the refrigeration compressor.

6.6 Conclusions

Four different possible scenarios have been illustrated to show the possibility of applying the Climatic Wind Tunnel model to devising energy saving strategies.

The possibility of sequencing the number of condenser fans operating as a function of refrigeration load and outside ambient temperature was investigated. Three typical daytime conditions were chosen for winter (5°C), mid-season (15°C) and mid-summer (25°C). At each of these conditions the model was run at refrigeration loads from full capacity down to 10% of full capacity and the number of condenser fans running reduced from 20 to 4 at each condition. It was found that as the refrigeration load reduced the number of condenser fans could be reduced, reducing compressor power consumption whilst still achieving the required refrigeration capacity. For each load condition it was found that the number of fans required had an optimum level below which the compressor power consumption began to rise. An example test was shown with the refrigeration plant running at 30% load for an 8 hour period. The test compared the present condenser fan control strategy to the optimising of the number of fans operating with the refrigeration load and outside ambient conditions. This test showed a saving of 63 kWh, which is a 7% saving in the energy consumed to produce the same refrigeration effect.

The investigation shows that there is a potential for reducing energy consumption of the refrigeration load, if an algorithm could be included into the control system software that sequenced the condenser fans as a function of refrigeration load and outside ambient temperature.

The possibility of conditioning a vehicle ready for testing in the test chamber as opposed to the soakroom was investigated. A vehicle was simulated to be conditioned from hot to –30°C for a typical 10 hour period in the soakroom. The same vehicle was simulated being conditioned for a 2½ hour period in the test chamber. A shorter period was used for conditioning the vehicle in the test chamber than the soakroom due to the added heat transfer made available due to the vehicle being in a moving air stream and not still air as in the soakroom. It was shown that the vehicle in the soakroom took the full 10 hour period to reach the desired test condition whilst the vehicle in the test chamber cooled from its hot condition quickly and followed the air temperature very closely reaching the desired condition in 2¼ hours. From the comparison of the predicted energy consumption figures it is shown that 870 kWh were saved in pre-conditioning in the test chamber rather than the soakroom. Based on this predicted energy saving, if it is assumed that one vehicle per day must be pre-conditioned before testing then the reduction in energy consumption would be in the region of 216,000 kWh.

The possible savings associated with reducing the trichloroethylene flow temperature differential from 15 to 10K were investigated. The simulation was run for the same conditions used for the vehicle conditioning investigation (-30°C). A trichloroethylene flow temperature of –45°C was used to simulate the 15K temperature differential and the same simulation was run for a flow temperature of –40°C. It was shown that the reduced flow temperature did not effect the systems in achieving their desired condition nor did it affect the pull down of the vehicle in either the test chamber or the soakroom. This reduction in flow temperature differential for the operational conditions of performing a –30°C pull down test gave a predicted energy saving of 750 kWh. It would be very difficult to put a figure to the annual saving offered by the reduction in flow temperature differential due to the continuously varying test conditions required by the users of the facility. But based on the figure indicated from the –30°C pull down test that figure would be in the region of 200,000 kWh.

The final investigation into possible application to operational strategies involved the increasing of the thickness of the insulated panel from which the test chamber and soakroom are constructed from 100 mm to 200 mm. Again the simulation was for cooling the air down from ambient and the vehicle down from hot. It was shown that the increase in insulation had very little effect on the air temperatures or vehicle pull down rates. The increase in insulation showed a reduction in consumed power of 95 kWh for the soakroom and 50 kWh for the test chamber. If the case of this saving is taken for an entire years operation for the test chamber (350 days) the minimum saving in power consumption would be 120,000 kWh. The increase in the insulation thickness would add to the construction cost of a CWT facility but the savings in the energy cost would offer a relatively short payback period in the investment.

The four investigations carried out show how the model could potentially be employed as a tool to improve the operational efficiency of the Climatic Wind

Tunnel plant. This improved efficiency could be made through the analysis of different plant control strategies or the configuration of the plant itself, but to do so some more detailed models would be required.

Chapter 7

Conclusions of research and future work

7.1 Conclusions of the research

The research conducted has shown that a dynamic model of the thermal systems that constitute a Climatic Wind Tunnel can be applied to the task of improving its operational efficiency.

Neutral Model Format (NMF) [1, 2, 3], an equation-based language for expressing models for use in an number of existing and emerging Modular Simulation Environments was reviewed and its salient points were discussed.

Five equation-based simulation tools, termed Modular Simulation Environments were evaluated for their applicability to the Climatic Wind Tunnel project.

All of the environments with the exception of SIMULINK have, to differing degrees, a proven level of capability in the modelling and simulation of HVAC systems.

All the simulation environments at their most basic level are accessed via text input. To gain widespread acceptance in the commercial field with non-expert users, graphical user interfaces (GUI's) are a highly desirable addition to the package.

A steady-state simulation tool is not appropriate for the analysis of the Climatic Wind Tunnel systems. This is due to the CWT'S thermal systems being made up of a great number of components that have a similar time response. A greater insight into possible directions for the reduction of energy consumption is possible by looking at the CWT'S systems whilst they are operating in a transient state. Analysis of the plant whilst it is operating in its transient state is important if the operation of the control system is to be investigated and the effect of changes to its mode of operation carried out.

Three Modular Simulation Environments strongly focused towards HVAC work and capable of dynamic simulation were considered for the undertaking of this task. these being: TRNSYS, SPARK and IDA.

TRNSYS was developed during the mid 1970's at the University of Wisconsin Solar Energy Laboratory primarily for the simulation of solar energy systems. It has been used with success on a number of projects that have involved the modelling and simulation of HVAC systems and components [4, 5].

SPARK is a simulation environment that is still currently under development at Lawrence Berkley Laboratory. It has been undergoing development work since its original conception in 1986. It has at present been used exclusively as a research tool and is unproven in a wider commercial setting. The first commercial release (WinSPARK 1.0) is due for release in the summer of 1999 – beyond the timescale of this project

IDA Simulation Environment is the product of a collaboration between the Swedish Institute of Applied Mathematics and the Department of Building Services Engineering, Swedish Royal Institute of Technology. Development began in the late 1980's and IDA has been used in the HVAC simulation field since 1990, during which time it has been used and proved successful on a number of projects. A new commercial package called IDA Indoor Climate and Energy aimed at the non-expert user is also now available. ICE is intended to be used in the simulation of thermal comfort, indoor air quality and building energy consumption.

TRNSYS, SPARK and IDA are all able to make use of the Neutral Model Format (NMF) description language [6] and translators are available for all environments. It has been decided that the IDA simulation environment will be used for the simulation work. The main reasons for this being that NMF is to be used for the description of the Climatic Wind Tunnels component models and there are very close ties between IDA and NMF. IDA allows the user the choice of a number of different numerical methods for finding the initial system operating conditions; this is likely to be a great advantage due to the large number of recirculating fluid loops present within the thermal systems.
The simulation work for the project was carried out using IDA Solver β-test version 6.09. Use of the solver suffered considerably from a lack of user documentation that caused problems in identifying fundamental errors in the system description file. This lack of documentation has been corrected in the final release version.

A number of models that were particular to the Climatic Wind Tunnel application had to be developed, these included: a vehicle, an airzone to represent the test chamber and soakroom, a two stage screw compressor model and condenser and evaporator models. These models were linked together along with other models representing the valves, heaters, heat exchangers and fans used to give representative sub-systems of the Climatic Wind Tunnel thermal systems. The sub-system models were then validated against empirical data obtained from the actual plant under various operating conditions.

The purpose of validation and verification is to give the user confidence that the results produced by the simulation are representative of what would be expected from the real life plant.

Validation falls into one of three categories [7]:

- Analytical
- Comparative
- Empirical

Analytical testing involves the derivation of exact solutions by analytical means that can then be compared to the equivalent program predictions. This type of testing is generally difficult to apply in building applications due the wide range of building types and their application.

Comparative testing is the inter comparison of results predicted by one program to a given set of inputs to the predicted results given by another program that has been subjected to the same inputs. This test only tells the user if his program gives similar output to another and gives little indication as to whether it can accurately predict the output of the system that it is supposed to be replicating.

Empirical testing is the comparison of simulation output to actual data recorded from an installation. Empirical data is often difficult and time consuming to obtain, but gives the best indication by far as to whether a simulation is giving results that are comparable to those expected in real life.

The output of the Climatic Wind Tunnel sub-system models were compared to data gathered from the plant whilst it was operating under differing conditions. It was shown that the prediction of the air temperatures within the test chamber and the soakroom were very closely matched to the data recorded from the plant.

The control strategy used in the simulation work was that of idealised proportional control. This type of control was adopted due to the lack of information available on the actual control system employed in the control of the CWT's HVAC plant.

The compressor model is a curve fit of data generated by a compressor selection program and in essence the validation took the form of an inter-model comparison with the computer selection program. The model was shown to correlate very well to the data it was based upon. Problem areas were shown to exist at opposite ends of the performance envelope (evaporating temperatures of -60°C and +10°C). At -60°C the model gives a rise in power consumption when it should be falling and a +10°C the data the model is based upon has large swings within it. Fortunately the extremes of the performance envelope are rarely visited in the CWT systems and the majority of operation falls within the area where very good agreement between the model and the data exists.

The vehicle is a model based upon a first order fit to recorded data. No data of exactly how the model should respond under different environmental conditions exists. It was shown that the model reacts as it would be expected to; it heats up to a steady state level when switched on and cools down at different rates depending on the velocity of the air surrounding it. The size of the vehicle the model is representing can be changed by increasing or decreasing its capacitance value as can its power output, which governs how much heat it releases.

Four different possible scenarios were illustrated to show the possibility of applying the Climatic Wind Tunnel model to devising energy saving strategies. No cost comparisons for the predicted savings could be made due to uncertainties in the actual billing system used by the Climatic Wind tunnel facility.

The possibility of sequencing the number of condenser fans operating as a function of refrigeration load and outside ambient temperature was investigated. Three typical daytime conditions were chosen for winter (5°C), mid-season (15°C) and mid-summer (25°C). At each of these conditions the model was run at refrigeration loads from full capacity down to 10% of full capacity and the number of condenser fans running reduced from 20 to 4 at each condition. It was found that as the refrigeration load reduced the number of condenser fans could be reduced, reducing compressor power consumption whilst still achieving the required refrigeration capacity. For each load condition it was found that the number of fans required had an optimum level below which the compressor power consumption began to rise. An example study was carried out with the refrigeration plant operating at 30% load for an 8 hour period. From this it was shown that the optimised fan control strategy saved 63 kWh, which equated to a 7% saving in the energy consumed by the control strategy currently employed

The investigation shows that there is a possibility for reducing energy consumption if intelligent supervisory control sequencing the condenser fans as a function of refrigeration load and outside ambient temperature was included in the control system.

The possibility of conditioning a vehicle ready for testing in the test chamber as opposed to the soakroom was investigated. A vehicle was simulated to be conditioned from hot to –30°C for a typical 10 hour period in the soakroom. The same vehicle was simulated being conditioned for a 2½ hour period in the test chamber. A shorter period was used for conditioning the vehicle in the test chamber than the soakroom due to the added heat transfer made available due to the vehicle being in a moving air stream as opposed to still air as in the soakroom. It was shown that the vehicle in the soakroom took the full 10 hour period to reach the desired test condition whilst the vehicle in the test chamber cooled from its hot condition quickly and followed the air temperature very closely reaching the desired condition in 2¼ hours. From the comparison of the predicted energy consumption figures for this operation it was shown that 870 kWh of electricity was saved by pre-conditioning the vehicle in the test chamber rather than the soakroom. Basing an assumption that one vehicle per day needs to be conditioned on the figure of power saved in the –30°C pull down test the total annual reduction in electricity consumption would be in the region of 216,000 kWh.

The possible savings associated with reducing the temperature differential between the trichloroethylene and the air from 15 to 10K were investigated. The simulations were run for the same conditions used for the vehicle conditioning investigation (-30°C). A trichloroethylene flow temperature of –45°C was used to simulate the 15K temperature differential and the same simulation was run for a flow temperature of –40°C. It was shown that the reduced flow temperature did not effect the systems in achieving their desired condition nor did it affect the pull down of the vehicle in either the test chamber or the soakroom. This reduction in temperature differential gave a predicted energy saving of 750 kWh. Using this figure as a basis of the saving available from the employment of this strategy annual savings would be in the region of 200,000 kWh.

The final investigation into possible application to operational strategies involved the increase in the thickness of the insulated panels from which the test chamber and soakroom are constructed from 100 mm to 200 mm. Again the simulation was for cooling the air down from ambient and the vehicle down from hot. It was shown that the increase in insulation had very little effect on the air temperatures or the temperature pull down rates of the vehicle. A –30°C pull down test simulated that the increase in insulation gave a reduction in power consumption of 95 kWh for the test chamber and 50 kWh for the soakroom. If the saving predicted for the test chamber is taken over its entire annual usage (350 days) then an annual saving (based upon this figure) of 120,000 kWh is possible. The saving in energy costs offered by increasing the insulation thickness needs to be assessed against the cost of upgrading the CWT construction and its supporting structure, before its true worth could be assessed.

The four investigations that were carried out illustrate how the model of the sub-systems may be employed as a tool to improve the operational efficiency of the Climatic Wind Tunnel plant. The model can be used to look at many different aspects of the facility's operation and identify areas where further investigation may yield savings either through revised control strategy or plant configuration.

7.2 Future work

Further investigation into the control system for the CWT should be made as this would lead to development of a more realistic control regime for the systems rather than the idealised proportional control that was adopted in the absence of any control system information.

Further data should be gathered from the plant to allow validation studies to be carried out over a large range of operating conditions. The validation work highlighted areas where faulty sensors exist. These should be replaced to allow more detailed validation studies to be carried out.

A compressor model that allows the effects of different refrigerants to be analysed should be developed. This is particularly important as existing CFC refrigerants are phased out in accordance with the Montreal Protocol. A model with this capability would allow the analysis of how the application of HCFC refrigerants or ammonia would affect the operational capacity of the Climatic Wind Tunnel.

Cost analysis to fully appreciate the benefits offered by the predicted savings against the capital expenditure required for their implementation.

7.3 References

[1] Sahlin P, Sowell E.F (1989). A neutral format for building simulation models. Proceedings of Building Simulation '89. Vancouver, Canada.

[2] Sahlin P (1996). NMF Handbook: an introduction to the neutral model format. HTTP//:www.brisdata.se/

[3] Sahlin P, Bring A, Sowell.E.F (1996). The neutral model format for building simulation. HTTP//:www.brisdata.se/

[4] Braun J.E (1988). Methodologies for the design and control of central cooling plants. PhD Thesis. University of Wisconsin, Madison, USA.

[5] Bourdouxhe J-P (1997). Use of the simulation tools in the design and operating phases of the air conditioning equipment. Laboratory of Thermodynamics, University of Liège, Belgium.

[6] Sahlin P, Sowell E.F (1989). A neutral format for building simulation models. Proceedings IBPSA Building Simulation 1989, Vancouver, Canada.

[7] Bloomfield D.P (1989). Evaluation procedures for building thermal simulation programs. Proceedings Building Simulation '89, Vancouver. Pg. 217 – 222.

Appendix A

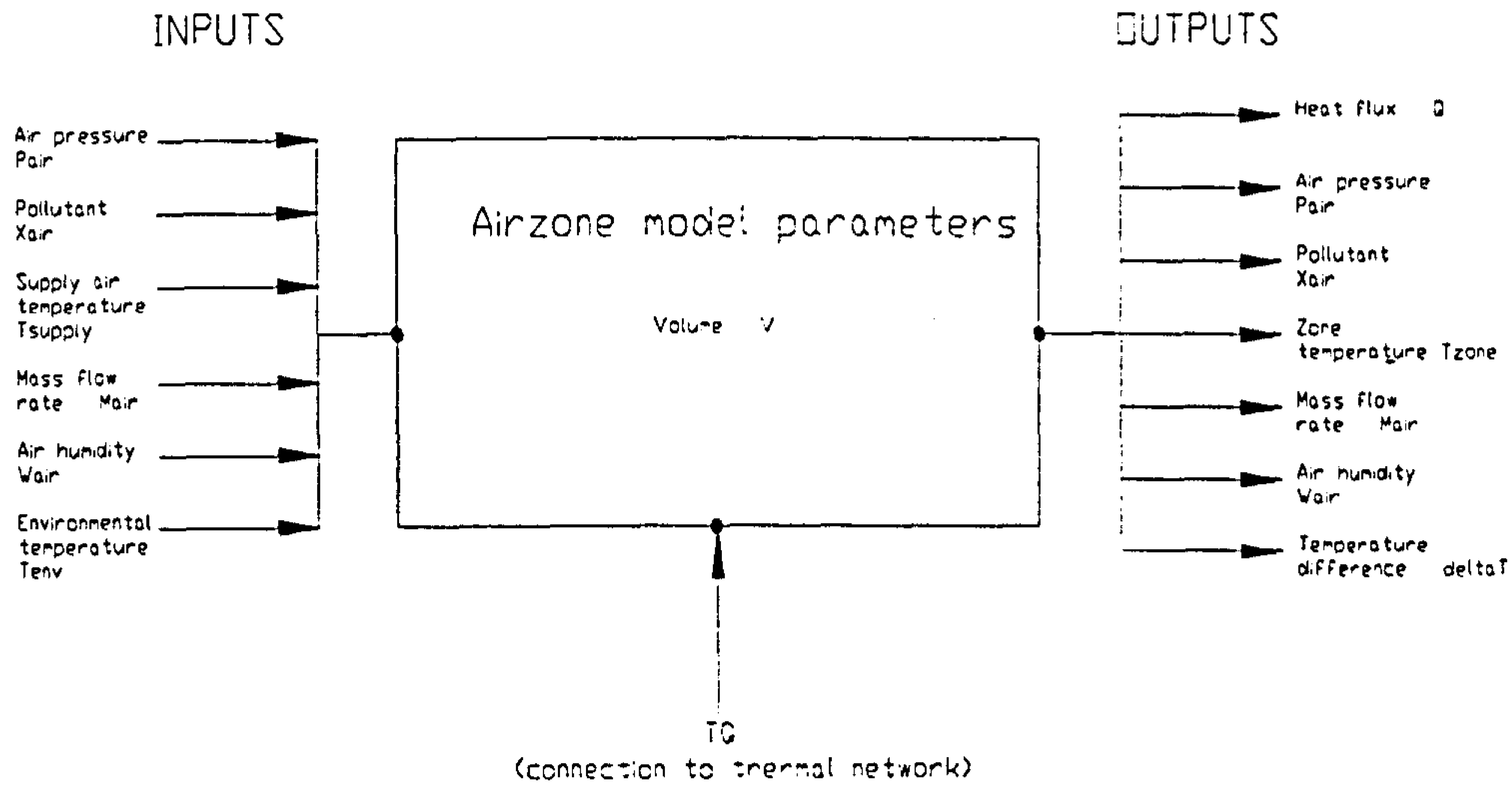
Models used in CWT simulations

General note:

X' notation used in models, refers to the first derivative $\frac{dx}{dt}$.

Airzone

Author: V.I.Hanby, E.C.Roberts



The user definable parameters for this component are:

Zone volume	(V)	m^3

The in/out variables for the model are:

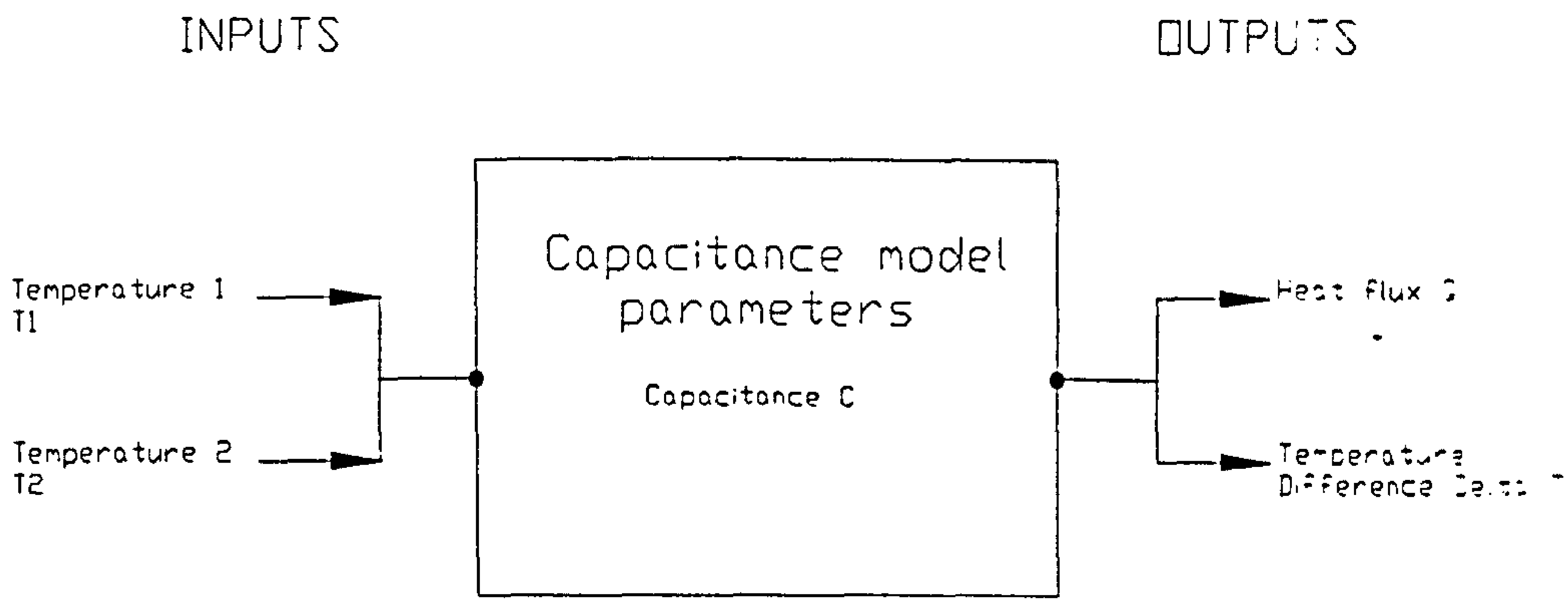
Mass flow air	(Mair)	kg/s
Air humidity ratio	(Wair)	kg/kg
Heat flux from temperature node	(Q)	W
Temperature difference	(deltaT)	°C
Air supply temperature	(Tsupply)	°C
Zone temperature	(Tzone)	°C
Environmental temperature	(Tenv)	°C
Air pressure	(Pair)	Pa
Pollutant fraction	(Xair)	Dimensionless

Model equations:

```
Cp_air*v*airdensity(Tzone)*deltaT' = Mair*Cp_air*(Tsupply
- Tzone) + Q


deltaT = Tzone - Tenv


Airdensity = (atmospheric pressure / (gas constant *
(airtemp +273))
```

Capacitance

Author: V.I.Hanby



The user definable parameters for this component are:

Capacitance	(C)	J/K

The in/out variables for the model are:

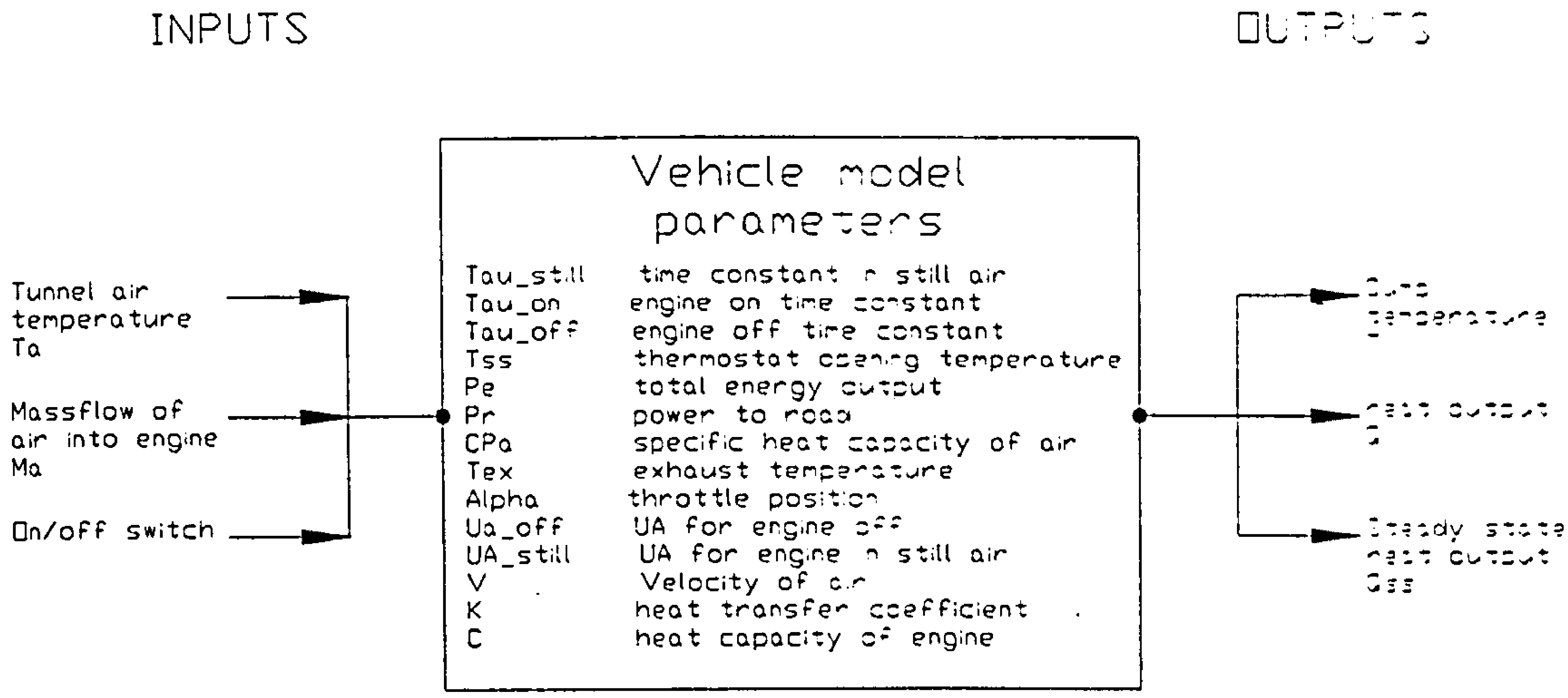
Heat flux	(Q)	W
Temperature difference	(DeltaT)	°C
Temperature at 1	(T1)	°C
Temperature at 2	(T2)	°C

Model equations:

```
C * DeltaT' = Q

DeltaT = T1 - T2
```

Car

Author: E.C Roberts, V.I.Hanby



The user definable parameters for this model are:

Time constant in still air	(Tau_still)	Dimensionless
Engine on time constant	(Tau_on)	Dimensionless
Engine off time constant	(Tau_off)	Dimensionless
Thermostat opening temperature	(Tss)	Dimensionless
Total energy output of engine	(Pe)	W
Power to rolling road	(Pr)	W
Specific heat capacity of air	(Cpa)	J/kg K
Exhaust temperature	(Tex)	°C
Throttle position	(Alpha)	Dimensionless
UA for engine off	(UA_off)	W/K
UA for engine in still air	(UA_still)	W/K
Tunnel air velocity	(V)	m/s
Heat transfer coefficient	(K)	W/K
Heat capacity of engine	(C)	J/K

The in / out variables for the model are:

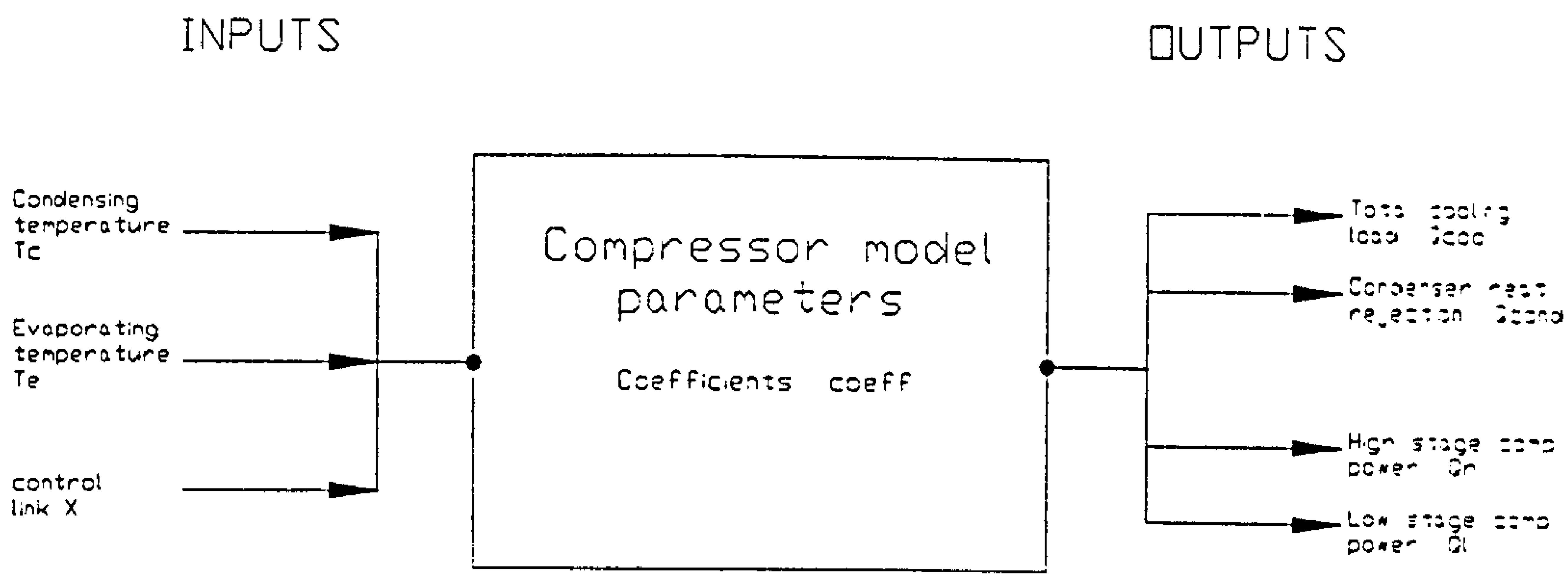
Tunnel air temperature	(Ta)	°C
Mass flow of air into engine	(Ma)	kg/s
On / off switch	(Switch)	Dimensionless
Oil sump temperature	(T)	°C
Engine warm up heat output	(Q)	W
Engine steady state heat output	(QSS)	W

Model equations:

```
QSS =(Alpha * Pe)-(Pr+(Alpha*Ma*Cpa)*(Tex-Ta))

T' = (TSS - T) / Tau_on

T'=  -(T - Ta) / Tau_off

Q = QSS*(T-Ta) / (TSS - Ta)

Q = UA_off*(T - Ta)

UA_still = C / Tau_still

UA_off   = UA_still(K * V)

Tau_off  = C / UA_off
```

Compressor

Author: E.C.Roberts



The model has no user definable parameters and its in/ out variables are:

Condensing temperature	Tc	°C
Evaporating temperature	Te	°C
Control link	X	Dimensionless
Cooling capacity	Qcool	W
Condenser heat rejection	Qcond	W
High stage power consumption	Qh	W
Low stage power consumption	Ql	W

The model uses a 2^{nd} order curve fit and uses the following coefficients:

a_0	0.1166e03
a_1	0.2399e01
a_2	0.2917e-01
a_3	-0.2709e01
a_4	-0.2993
a_5	-0.4148e-02
a_6	0.1064
a_7	0.5015e-02
a_8	0.6396e-04

Table A.1 Second order curve fit coefficients for high stage compressor

a_0	0.1667e03
a_1	0.2656e01
a_2	0.6666e-02
a_3	0.3e01
a_4	0.6366e-01
a_5	0.2313w-03
a_6	-0.1641e-01
a_7	-0.4052e-03
a_8	-0.6595e-06

Table A.2 Second order curve fit coefficients for low stage compressor

a_0	0.2957e04
a_1	0.5884e02
a_2	0.242
a_3	0.2805e02
a_4	0.1505e01
a_5	0.1728e-01
a_6	-0.4517
a_7	-0.1839e-01
a_8	-0.1692e-03

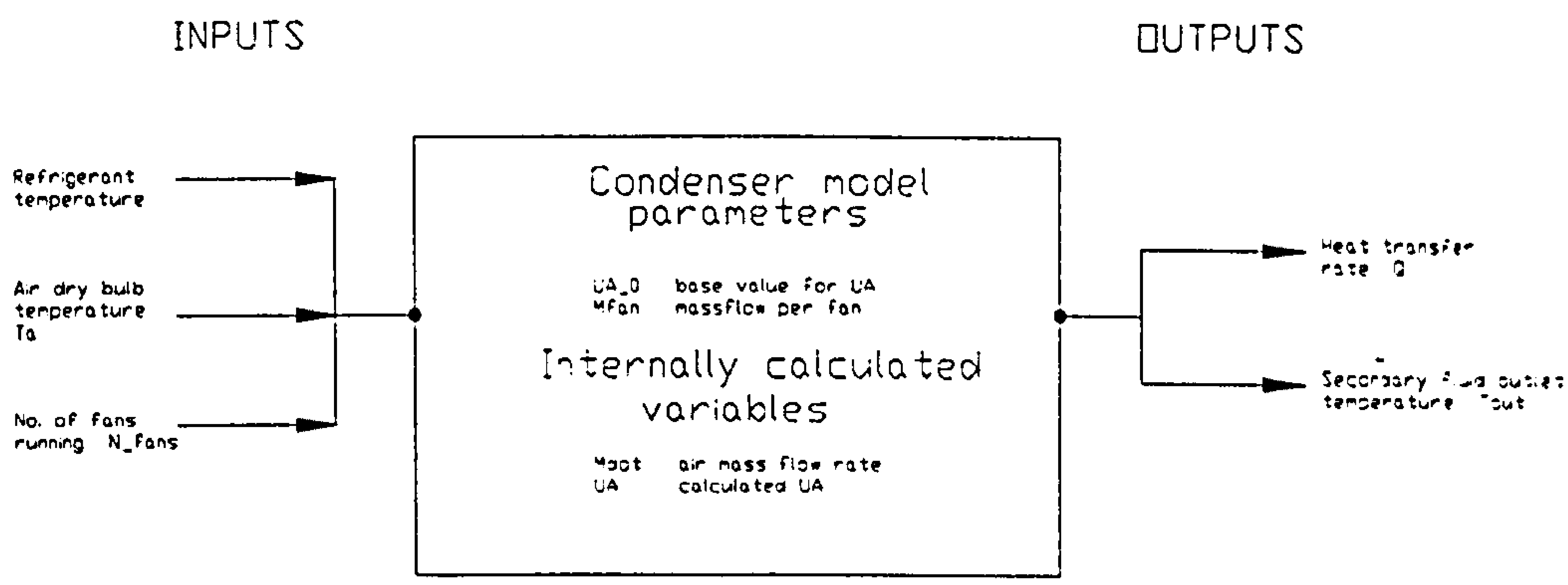
Table A.3 Second order curve fit coefficients for combined cooling capacity

The condenser heat rejection is given by:

```
0 = Qcool + Qh + Ql - Qcond
```

Condenser

Author: V.I.Hanby



The user definable parameters for the condenser are:

Base value for UA	UA_0	W/K
Mass flow rate per fan	Mfan	kg/s

The in/out variables for the model are:

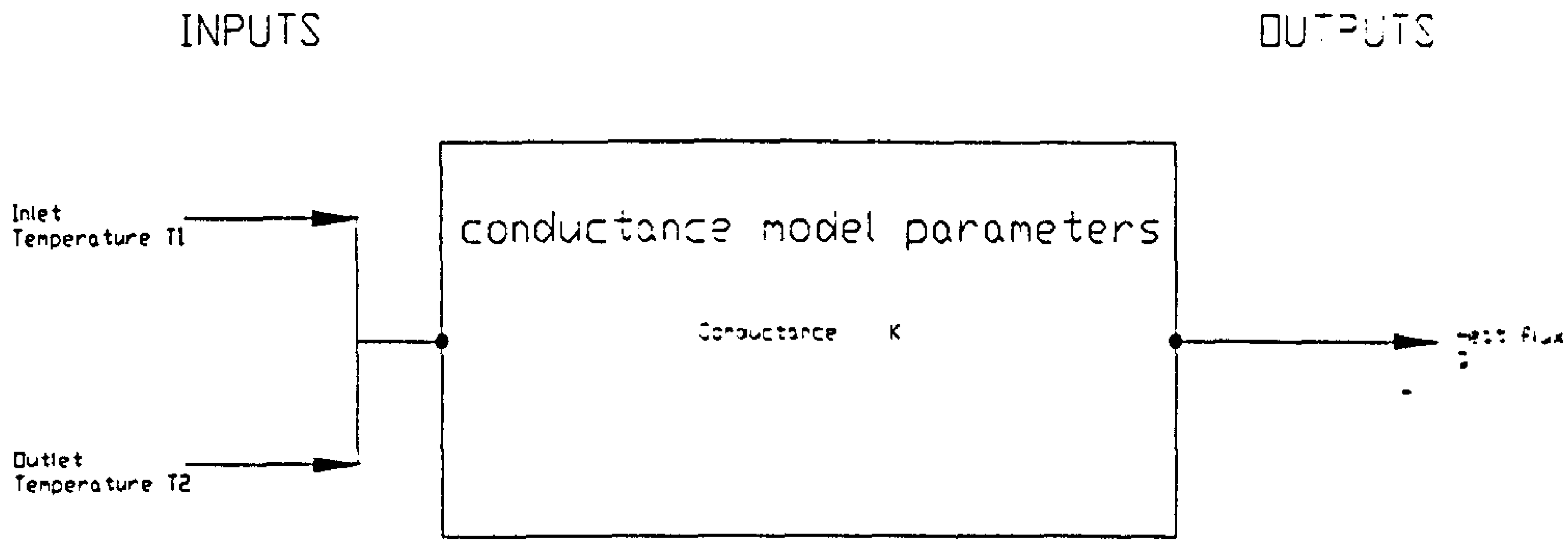
Air mass flow rate	Mdot	kg/s
Refrigerant temperature	Tfrig	°C
Air dry bulb temperature	To	°C
UA value	UA	W/K
No. of fans running	N_fans	Dimensionless
Heat transfer rate	Q	W
Air leaving temperature	Tout	°C

Model equations:

```
Mdot = MFAN*n_fans


UA = UA_0*n_fans


0 = Mdot*CP_AIR*(Tout - Ta) - Q


0 = (Tfrig - Ta)*(1-EXP (-UA / (Mdot*CP_AIR))) + Ta -
Tout
```

Conductance

Author: V.I.Hanby



The user definable parameters for this component are:

Conductance	(K)	W/K

The in/out variables for the model are:

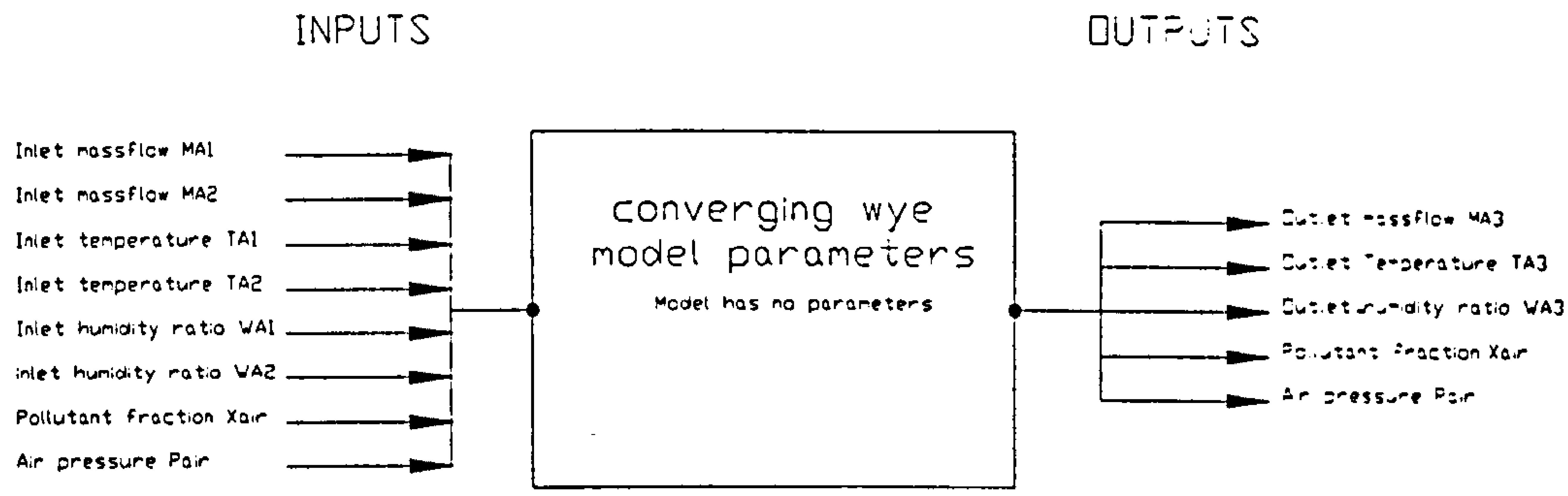
Heat flux	(Q)	W
Inlet temperature	(T1)	°C
Outlet temperature	(T2)	°C

Model equation:

```
Q = K * (T1 - T2)
```

Converging wye

Author: V.I.Hanby



The model has no user definable parameters and its in/ out variables are:

Inlet 1 air mass flow rate	(Ma1)	kg/s
Inlet 2 air mass flow rate	(Ma2)	kg/s
Outlet 3 air mass flow rate	(Ma3)	kg/s
Inlet 1 air temperature	(Ta1)	°C
Inlet 2 air temperature	(Ta2)	°C
Outlet 3 air temperature	(Ta3)	°C
Inlet 1 humidity ratio	(Wa1)	kg/kg
Inlet 2 humidity ratio	(Wa2)	kg/kg
Outlet 3 humidity ratio	(Wa3)	kg/kg
Pollutant fraction	(Xair)	Dimensionless
Air pressure	(Pair)	Pa

Model equations:

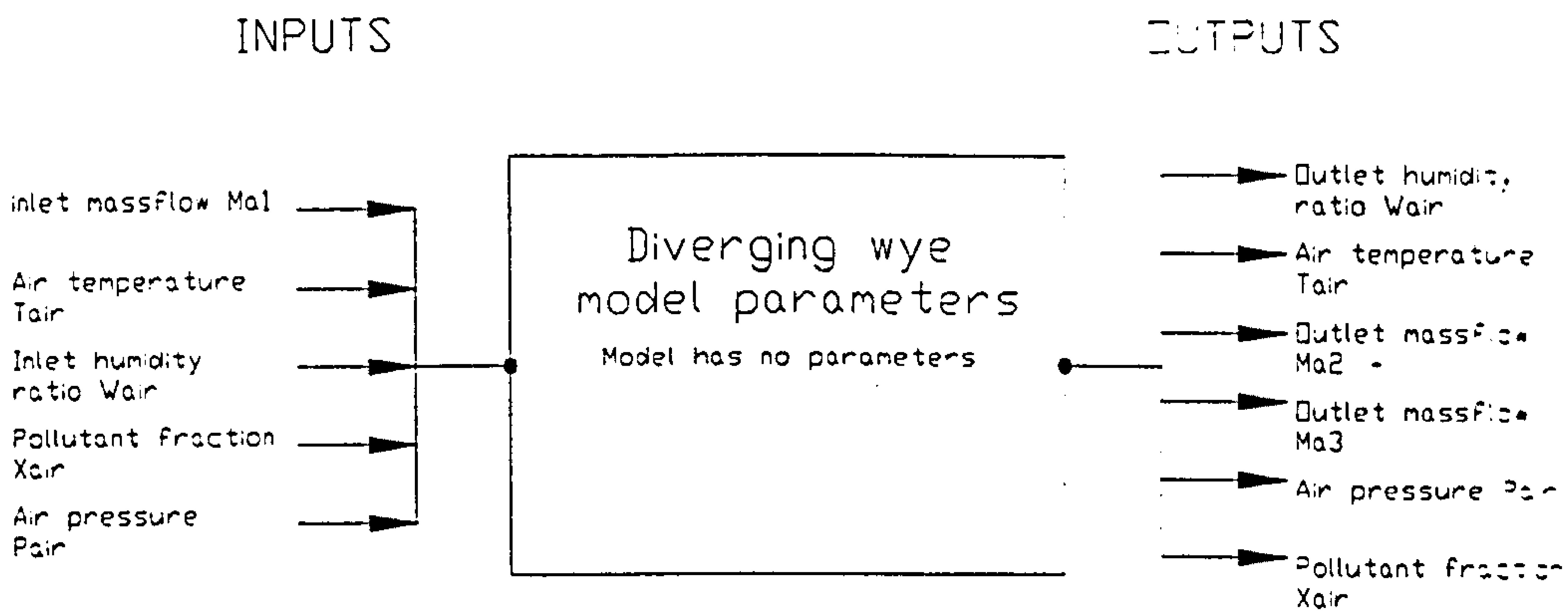
$$Ma3 = Ma1 + Ma2$$

$$Ma3*Ta3 = Ma1*Ta1 + Ma2*Ta2$$

$$Ma3*Wa3 = Ma1*Wa1 + Ma2*Wa2$$

Diverging wye

Author: V.I.Hanby



The model has no user definable parameters and its in/ out variables are:

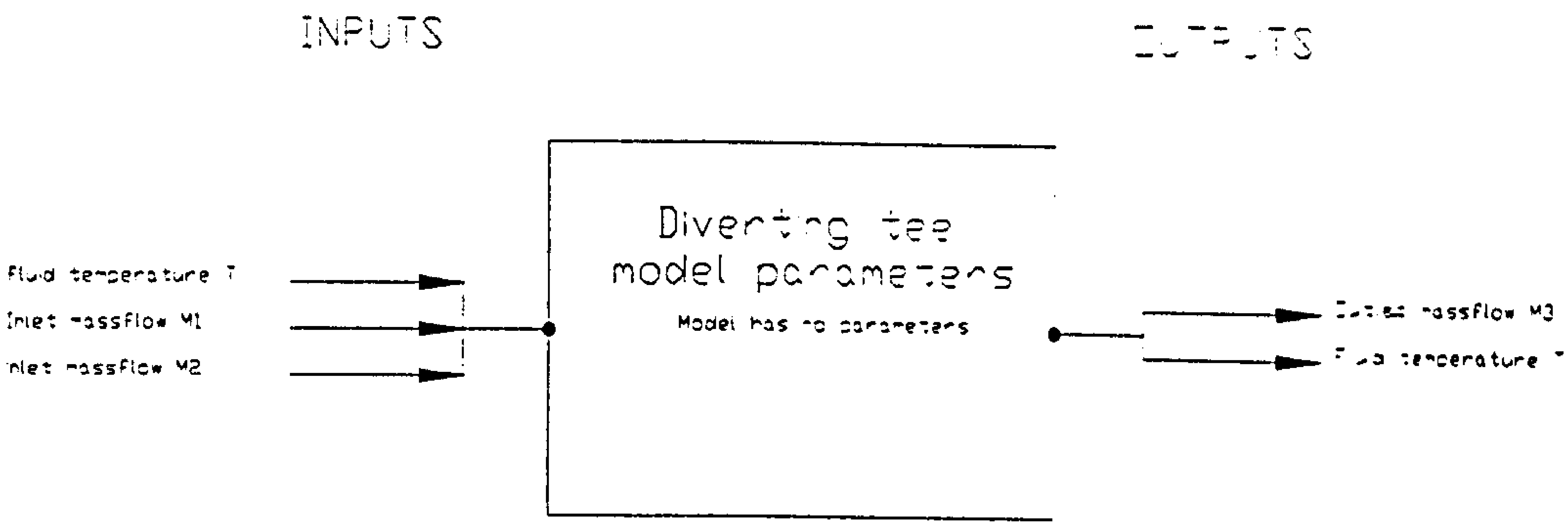
Inlet air mass flow rate	(Ma1)	kg/s
Outlet 1 air mass flow rate	(Ma2)	kg/s
Outlet 2 air mass flow rate	(Ma3)	kg/s
Air humidity ratio	(Wair)	kg/kg
Air temperature	(Tair)	°C
Pollutant fraction	(Xair)	Dimensionless
Air pressure	(Pair)	Pa

Model equation:

```
Ma1 = Ma2 + Ma3
```

Diverging tee

Author: E.C.Roberts



The model has no user definable parameters and its in/ out variables are:

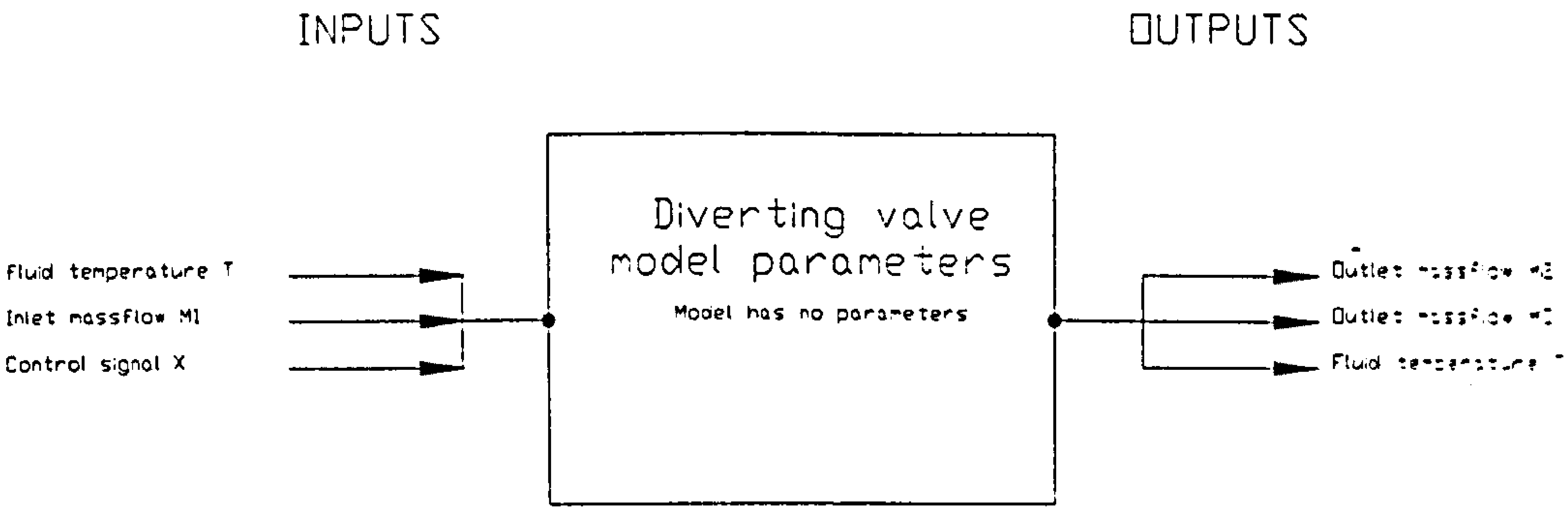
Fluid temperature	(T)	°C
Fluid inlet massflow	(M1)	kg/s
Fluid inlet massflow	(M2)	kg/s
Fluid outlet massflow	(M3)	kg/s

Model equation:

```
Ma1 = Ma2 + Ma3
```

Diverting valve

Author: E.C.Roberts



The model has no user definable parameters and its in/ out variables are:

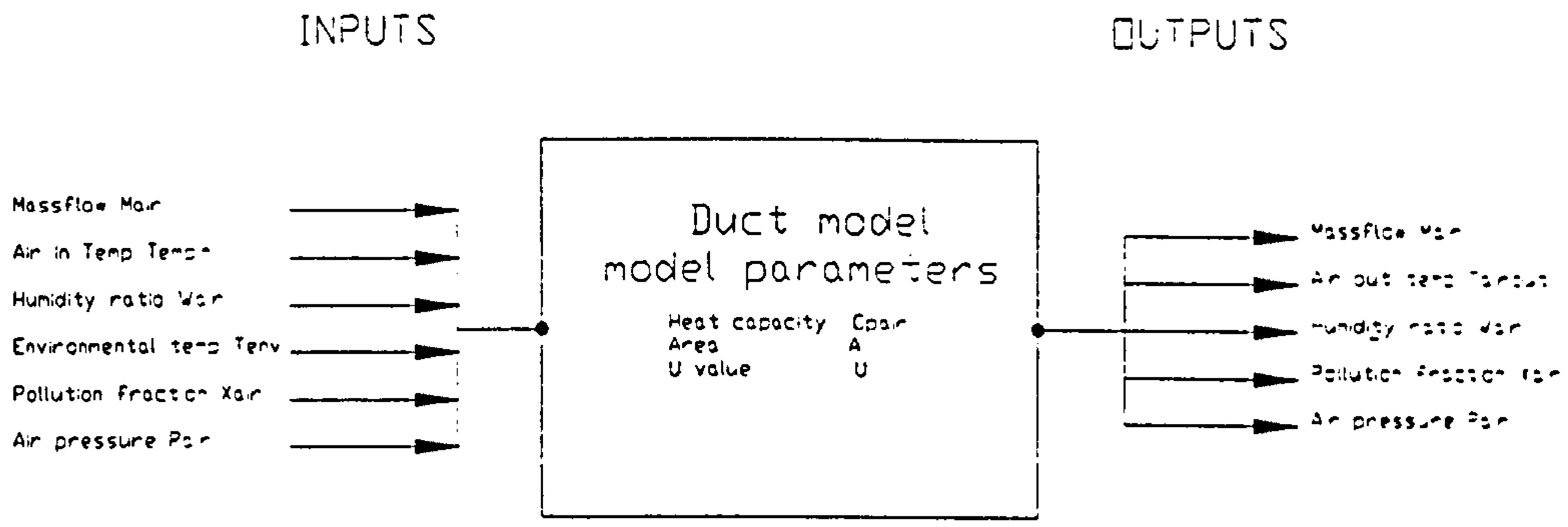
Fluid temperature	(T)	°C
Fluid inlet massflow	(M1)	kg/s
Fluid inlet massflow	(M2)	kg/s
Fluid outlet massflow	(M3)	kg/s
Control signal	(X)	dimensionless

Model equations:

```
M3  = (M1 * x)

M2  = M1 * (1-x)
```

Duct

Author: E.C.Roberts



The user definable parameters for this component are:

Duct U value	(U)	W/K
Area of duct	(A)	m^2
Heat capacity of air	(Cpair)	J/K

The in/out variables for the model are:

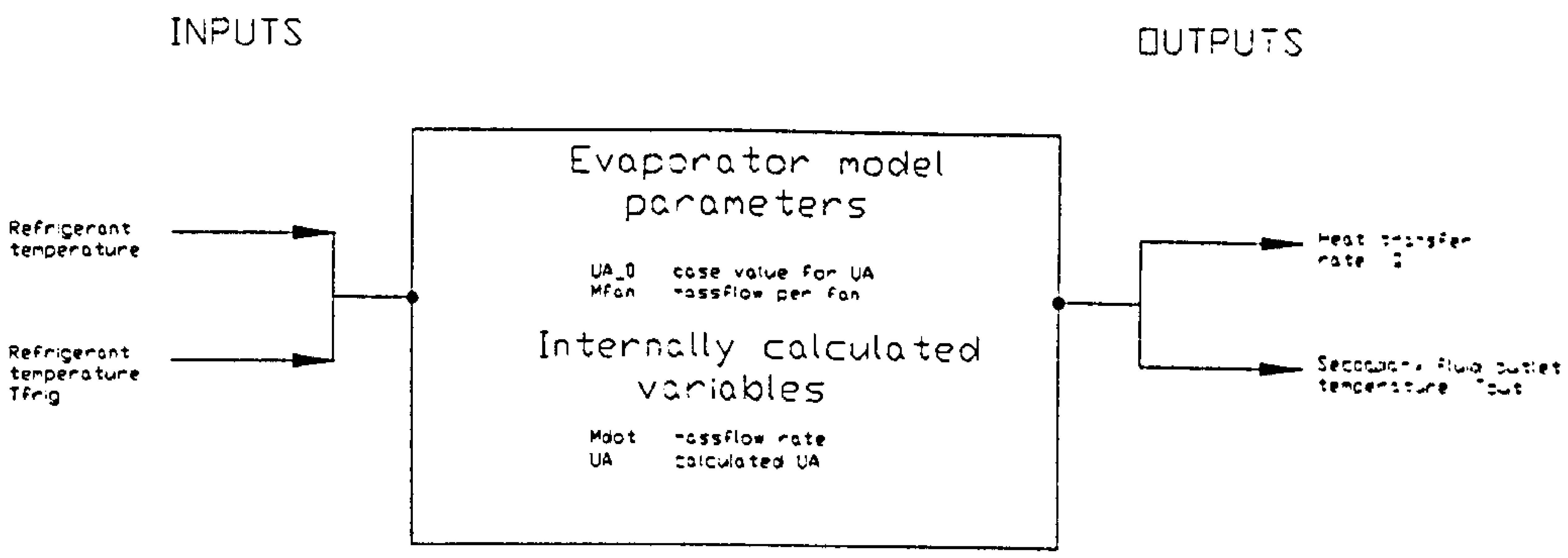
Inlet temperature	(Tempin)	°C
Outlet temperature	(Tempout)	°C
Massflow	(Mair)	kg/s
Humidity ratio	(Wair)	kg/kg
Environmental temperature	(Tenv)	°C
Pollution fraction	(Xair)	dimensionless
Air pressure	(Pair)	Pa

Model equation:

```
Tairout = Tenv + ((Tairin - Tenv) * Exp(-1*((U*A)/(Mair *
Cpair)))
```

Evaporator

Author: V.I.Hanby



The user definable parameters for the evaporator are:

Base value for UA	UA_0	W/K
Temperature gradient of UA	K	°C
Specific heat capacity	Cpliq	J/kgK

The in/out variables for the model are:

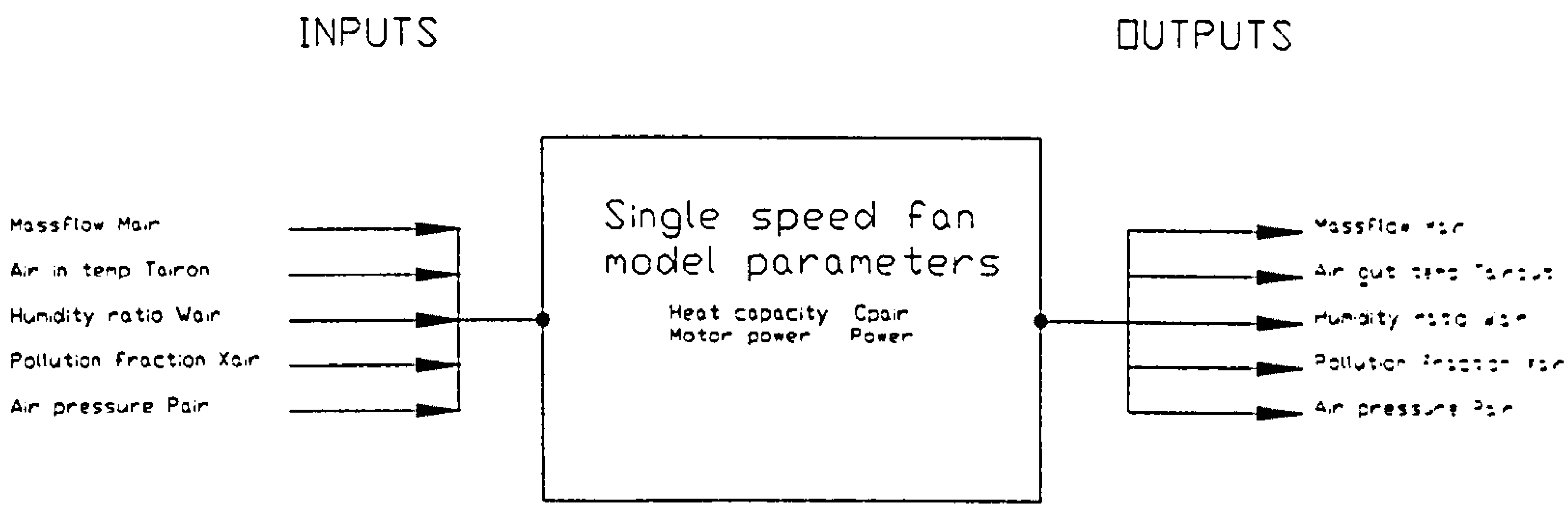
Secondary fluid mass flow rate	mdot	kg/s
Refrigerant temperature	Tfrig	°C
Secondary fluid inlet temperature	Tin	°C
UA value	UA	W/K
Secondary outlet temperature	Tout	°C
Heat transfer rate	Q	W

Model equations:

```
UA = UA_0 + K*Tfrig

Q = mdot*Cpliq*(Tin - Tout)

Tin + Tout = (Tin - Tfrig)*(1-EXP(-UA/(mdot*Cpliq)))
```

Single speed fan

Author: E.C.Roberts



The user definable parameters for this component are:

Heat capacity of air	(Cpair)	J/K
Fan motor power	(Power)	W

The in/out variables for the model are:

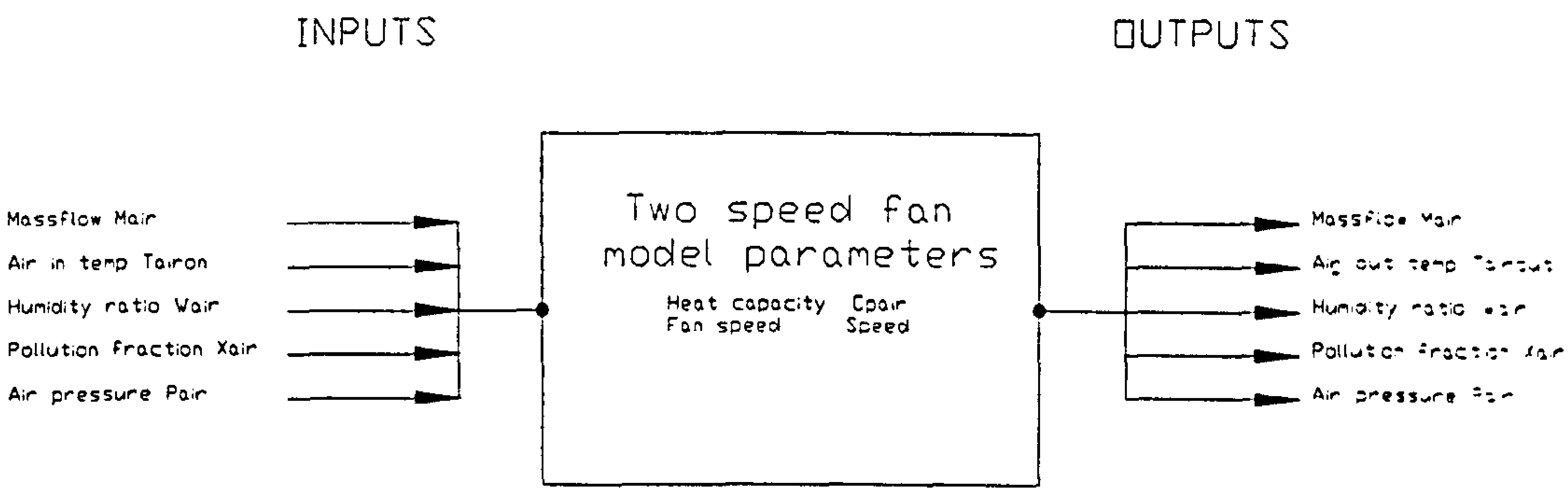
Inlet temperature	(Tairon)	°C
Outlet temperature	(Tairoff)	°C
Massflow	(Mair)	kg/s
Humidity ratio	(Wair)	kg/kg
Pollution fraction	(Xair)	dimensionless
Air pressure	(Pair)	Pa

Model equation:

```
Power = Mair * Cpair * (Tairoff - Tairon)
```

Two speed fan

Author: E.C.Roberts



The user definable parameters for this component are:

Heat capacity of air	(Cpair)	J/K
Fan motor speed	(Speed)	W

The in/out variables for the model are:

Inlet temperature	(Tairon)	°C
Outlet temperature	(Tairoff)	°C
Massflow	(Mair)	kg/s
Humidity ratio	(Wair)	kg/kg
Pollution fraction	(Xair)	dimensionless
Air pressure	(Pair)	Pa

Model equations:

If fan is running at full speed:

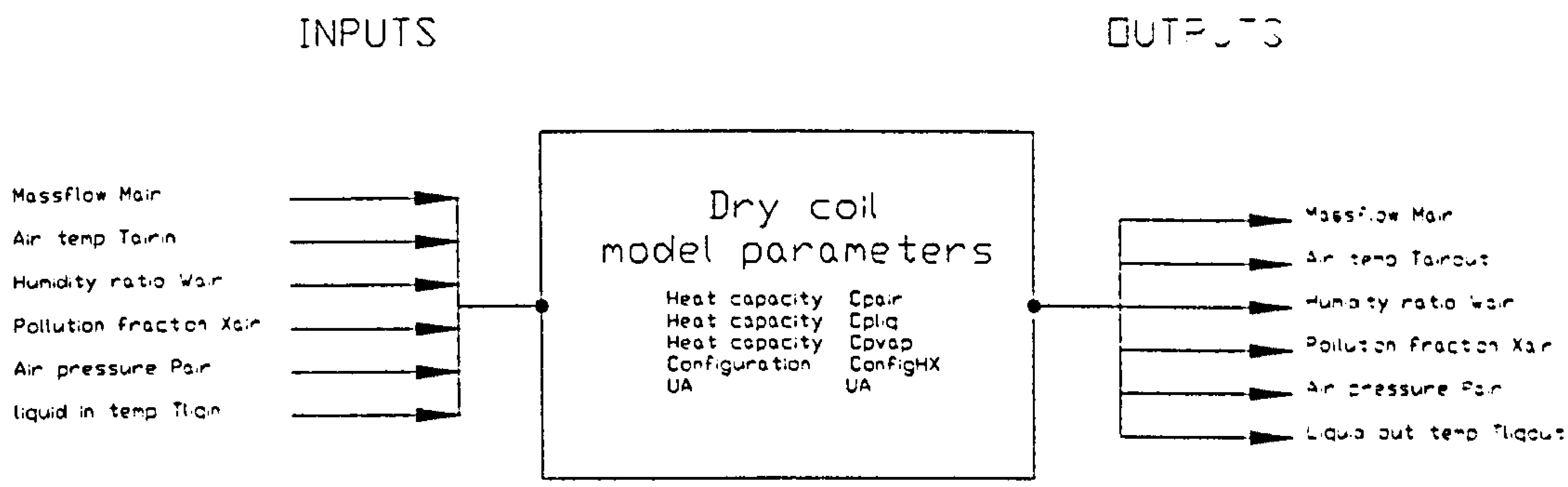
```
6000 = Mair * Cpair * (Tairon - Tairoff)
```

If fan is running at half speed:

```
4150 = Mair * Cpair * (Tairon - Tairoff)
```

Dry coil heat exchanger

Author NMF wrapper: E.C.Roberts
Author F77 sub-routine: J. Hyttinen



The user definable parameters for this component are:

Heat capacity of air	(Cpair)	J/K
Heat capacity of liquid	(Cpliq)	J/K
Heat capacity of water vapour	(Cpvap)	J/K
Heat exchanger configuration	(ConfigHX)	Dimensionless
Overall heat transfer coefficient	(UA)	W/K

The in/out variables for the model are:

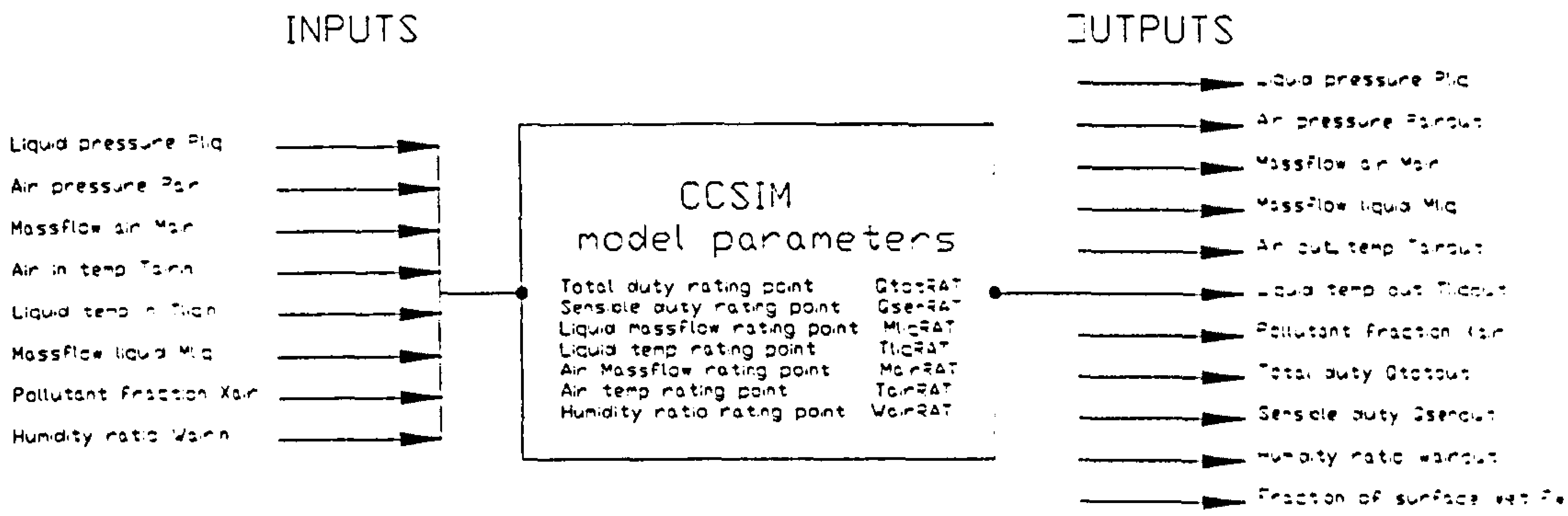
Inlet temperature	(Tairon)	°C
Outlet temperature	(Tairoff)	°C
Massflow	(Mair)	kg/s
Humidity ratio	(Wair)	kg/kg
Pollution fraction	(Xair)	dimensionless
Air pressure	(Pair)	Pa
Liquid inlet temperature	(Tliqin)	°C
Liquid outlet temperature	(Tliqout)	°C

Model equations:

NMF wrapper calls F77 DRYCOIL sub-routine from ASHRAE NMF Secondary Toolkit. Model does not simulate dehumidification of air stream or frost build-up on finned surface.

CCSIM heat exchanger

Author NMF wrapper: J.Hyttinen



The user definable parameters for this component are:

Coil total duty rating point	(Qtotrat)	W
Coil sensible duty rating point	(Qsenrat)	W
Liquid flow rate rating point	(Mliqrat)	kg/s
Liquid temp at rating point	(Tliqrat)	°C
Mass flow air at rating point	(Mairrat)	kg/s
Air temp at rating point	(Tairrat)	°C
Humidity ratio of air at rating point	(Wairrat)	kg/kg

The in/out variables for the model are:

Inlet air temperature	(Tairin)	°C
Outlet air temperature	(Tairout)	°C
Massflow of air	(Mair)	kg/s
Inlet humidity ratio	(Wairin)	kg/kg
Outlet humidity ratio	(Wairout)	kg/kg
Pollution fraction	(Xair)	dimensionless
Inlet air pressure	(Pairin)	Pa
Outlet air pressure	(Pairout)	Pa
Liquid inlet temperature	(Tliqin)	°C
Liquid outlet temperature	(Tliqout)	°C
Total duty	(Qtotout)	W
Sensible duty	(Qsenout)	W
Fraction of coil surface wet	(Fwetout)	dimensionless

Model equations:

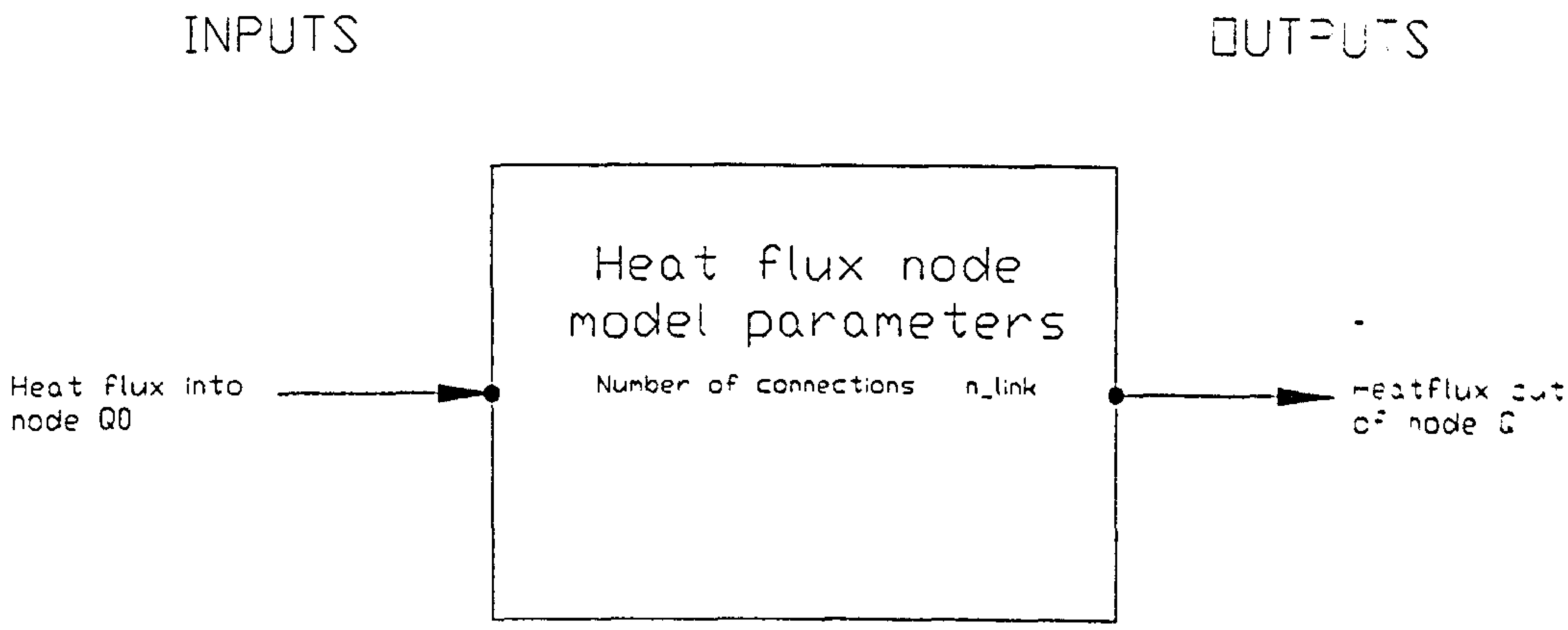
NMF wrapper calls F77 DRYCOIL, WETCOIL and psychormetric sub-routines from ASHRAE NMF Secondary Toolkit.

Model requires the input of rating point at which the coil being model was designed at.

Model does not simulate frost build-up on finned surface.

Heat flux node

Author: V.I.Hanby



The user definable parameters for this component are:

Number of links less one	(n_link)	dimensionless

The in/out variables for the model are:

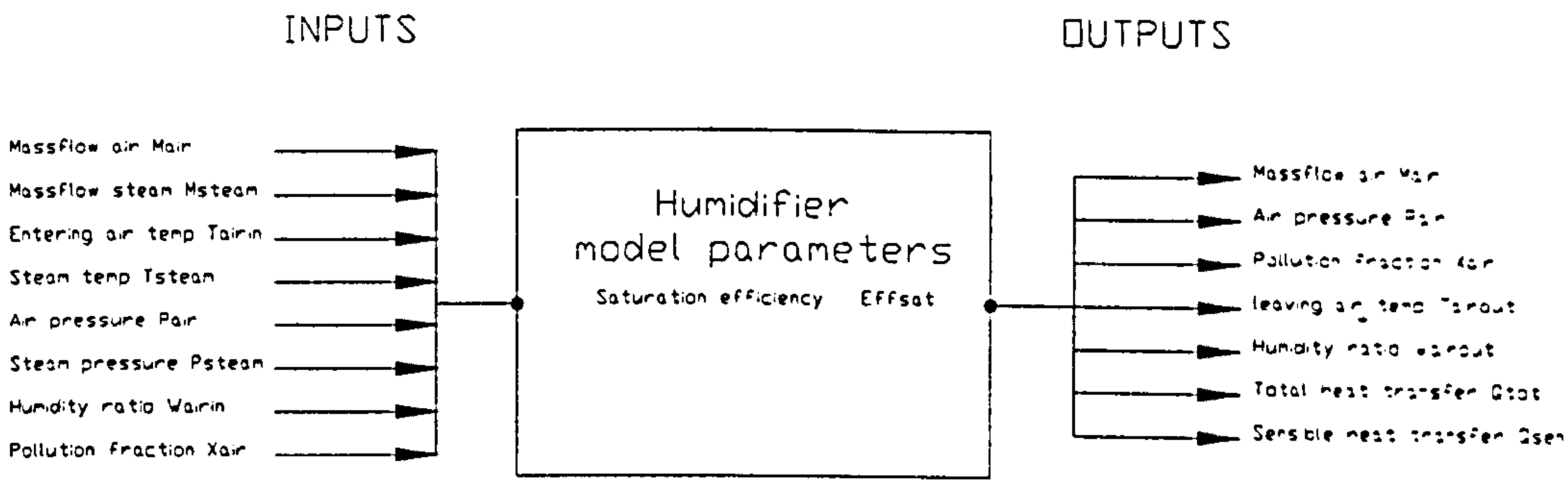
Node temperature	(T)	°C
Heat flux out of node	(Qo)	W
Heat flux into node	(Q)	W

Model equation:

```
0 = Q0 + SUM j = 1,n_link Q[j]
```

Humidifier

Author: J.Hyttinen



The user definable parameters for this component are:

Saturation efficiency	(Effsat)	dimensionless

The in/out variables for the model are:

Temperature of air in	(Tairin)	°C
Temperature of Steam	(Tsteam)	°C
Temperature of air leaving	(Tairout)	°C
Massflow of air	(Mair)	kg/s
Massflow of steam	(Msteam)	kg/s
Air pressure	(Pair)	Pa
Steam pressure	(Psteam)	Pa
Humidity ratio in	(Wairin)	kg/kg
Humidity ratio out	(Wairout)	kg/kg
Pollution fraction	(Xair)	dimensionless
Total heat transfer	(Qtot)	W
Sensible heat transfer	(Qsen)	W

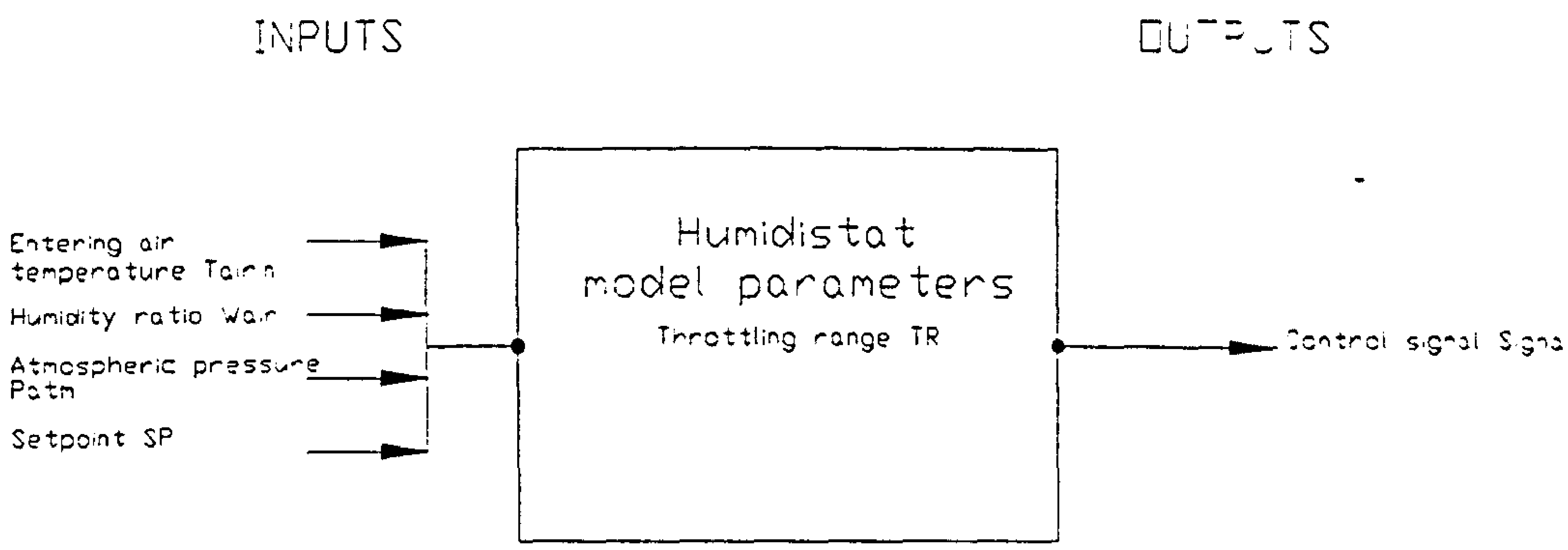
Model equations:

Calls psychrometric sub-routines from NMF toolkit.

```
cpMoist   = cp_air + WAirIn*cp_vap

TAirOut
=(mSteam*cp_vap*Tsteam+mAir*cpMoist*TAirIn)/(mSteam*cp_va
p+ mAir*cpMoist)

WMinOut   = WAirIn + mSteam/mAir

PSat     = SATPRES(TAirOut)

WSatOut   = 0.62198*PSat/(pAir-Psat)

WairOut =  (WSatOut*EffSat)

EntAirIn = cp_air*TAirIn + WAirIn*(hf_vap +
cp_vap*TAirIn)

EntAirOut  = cp_air*TAirOut + WAirOut*(hf_vap +
cp_vap*TAirOut)

Qtot = mAir*(EntAirIn-EntAirOut)

Qsen = mAir*cpMoist*(TAirIn-TAirOut)
```

Humidistat

Author: V.I.Hanby, E.C.Roberts



The user definable parameters for this component are:

Throttling range	(TR)	dimensionless

The in/out variables for the model are:

Temperature of air in	(Tairin)	°C
Humidity ratio	(Wair)	kg/kg
Atmospheric pressure	(Patm)	Pa
Setpoint	(SP)	dimensionless
Signal	(X)	dimensionless

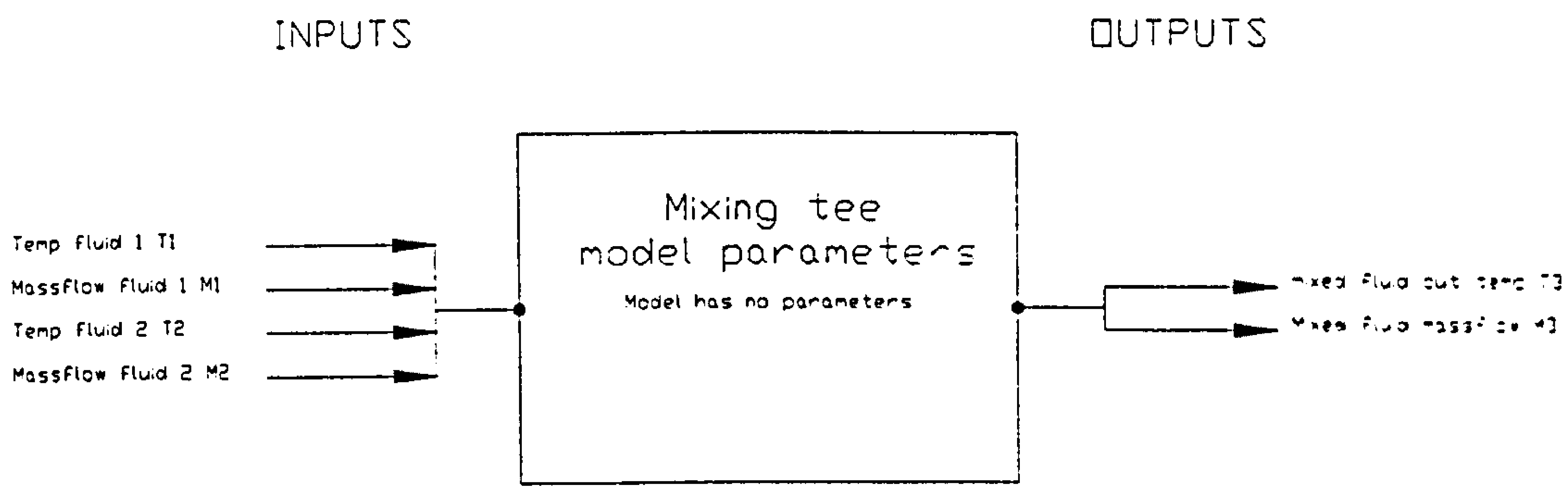
Model equations:

Model calls psychrometric sub-routines from NMF toolkit.

```
If relative humidity <= (sp-TR/2) then signal out = 1

If relative humidity >= (sp+TR/2) then signal out = 0

Otherwise 1-(rh - (sp-TR/2))/TR
```

Mixing tee

Author E.C.Roberts



The model has no user definable parameters and its in/ out variables are:

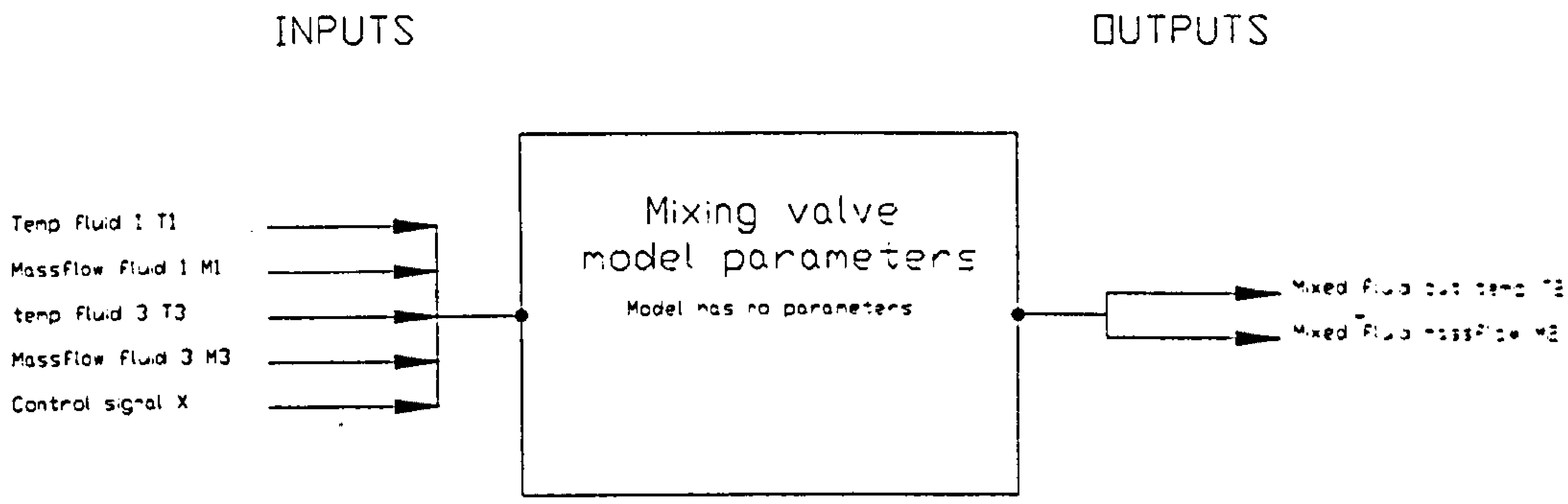
Fluid 1 temperature	(T)	°C
Fluid 1 inlet massflow	(M1)	kg/s
Fluid 2 temperature	(T)	°C
Fluid 2 inlet massflow	(M2)	kg/s
Fluid 3 temperature	(T)	°C
Fluid 3 outlet massflow	(M3)	kg/s

Model equations:

```
M3  =  M2  +  M1
(M3*T3)  =  (M2*T2)  +  (M1*T1)
```

Mixing valve

Author: E.C.Roberts



The model has no user definable parameters and its in/ out variables are:

Fluid 1 temperature	(T)	°C
Fluid 1 inlet massflow	(M1)	kg/s
Fluid 2 temperature	(T)	°C
Fluid 2 outlet massflow	(M2)	kg/s
Fluid 3 temperature	(T)	°C
Fluid 3 inlet massflow	(M3)	kg/s
Control signal	(X)	dimensionless

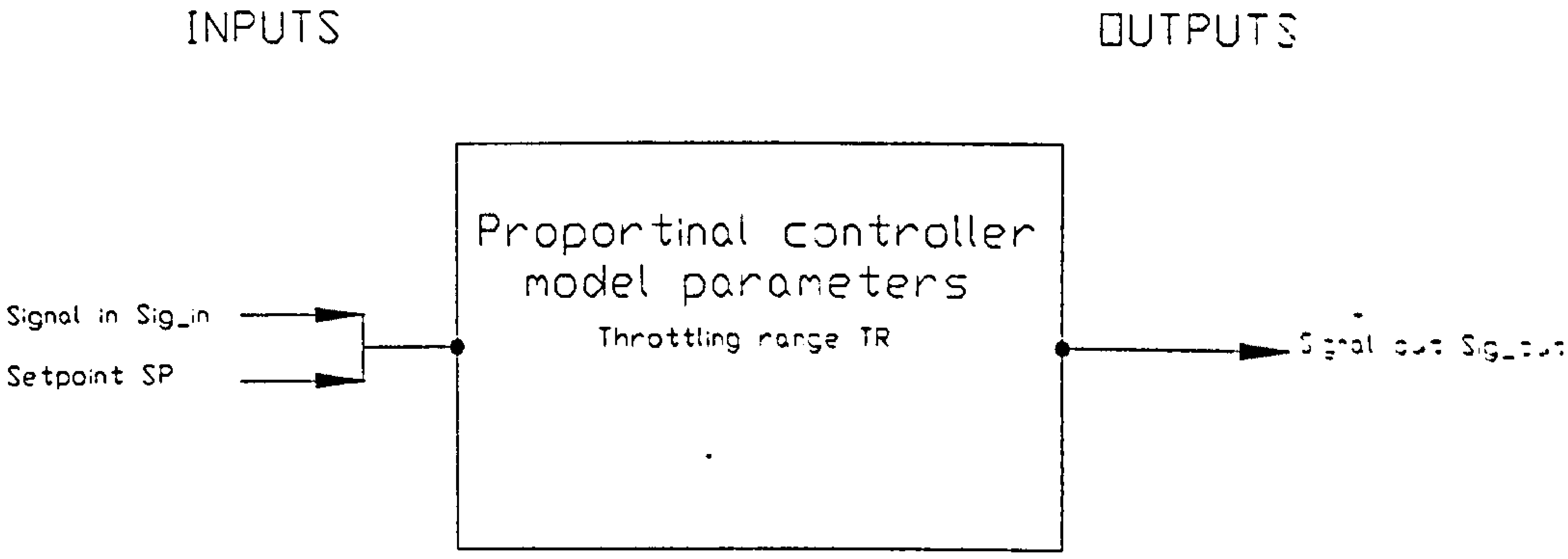
Model equations:

$$M2 = (x * M3) + ((1 - x) * M1)$$

$$(M2 * T2) = ((x * M3) * T3) + ((1 - x) * (M1 * T1))$$

Proportional controller – heating

Author: E.C.Roberts



The user definable parameters for this component are:

Throttling range	(TR)	dimensionless

The in/ out variables for the model are:

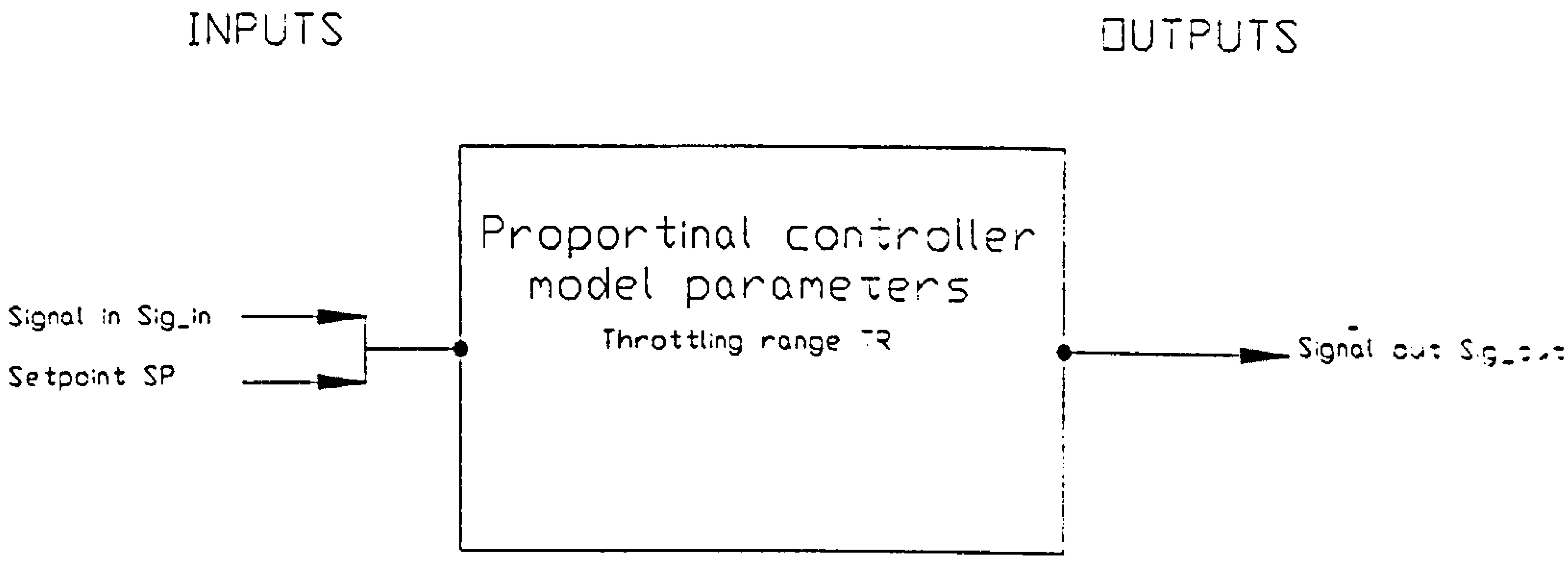
Control signal	(Sig_out)	dimensionless
Sensed variable	(Sig_in)	dimensionless
Setpoint	(Setpoint)	dimensionless

Model equations:

```
If sig_in < (setpoint-TR/2) then sig_out = 1

If sig_in > (setpoint+TR/2) then sig_out = 0

Otherwise sig_out = 1-(sig_in- (setpoint-TR/2))/TR
```

Proportional controller – cooling

Author: E.C.Roberts



The user definable parameters for this component are:

Throttling range	(TR)	dimensionless

The in/ out variables for the model are:

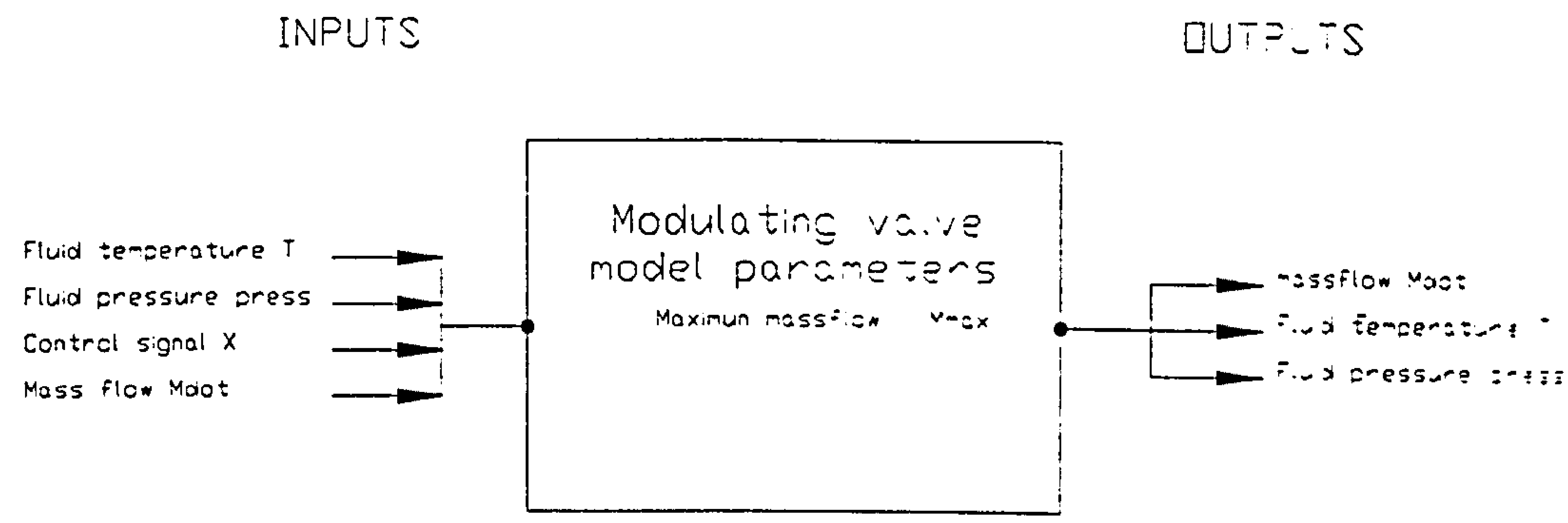
Control signal	(Sig_out)	dimensionless
Sensed variable	(Sig_in)	dimensionless
Setpoint	(Setpoint)	dimensionless

Model equations:

```
If sig_in < (setpoint-TR/2) then sig_out = 0

If sig_in > (setpoint+TR/2) then sig_out = 1

Otherwise sig_out = (sig_in- (setpoint-TR/2))/TR
```

Two port modulating valve

Author: E.C.Roberts



The user definable parameters for this component are:

Maximum flowrate	(Mmax)	kg/s

The in/ out variables for the model are:

Fluid temperature	(T)	°C
Fluid massflow	(Mdot)	kg/s
Control signal	(X)	dimensionless
Fluid pressure	(Press)	Pa

Model equation:

```
Mdot = (Mmax * X)
```

Appendix B

IDA system description file

The following example IDA system description file is for the refrigeration system used in the climatic Wind Tunnel simulations.

The sub-system consists of a two-stage compressor, condenser and evaporator. Control of the chiller is by proportional controller sensing the fluid outlet temperature from the evaporator.

```
ABSTRACT
" Refrigeration system by ECR on 19-4-99
  Proportional control loop added ECR 20-7-99
"

OPTIONS           !optional section used for configuration of IDA Solver

END_OPTIONS


FILES             !specifies paths for input and output data files.
OUTPUT refrig     !Output data file contents is specified in
   PATH *         !INTEGRATION section
END_FILES


CONSTANTS         !optional section used to define constant values for
                  !component parameters
END_CONSTANTS
```

```
TABLES                !allows tables of data to read as time dependent
END_TABLES            !boundaries

MODULES               !instantiates all the component models to be used in the
 MODULE comp1         !simulation
  TYPE compressor
 MODULE con1          !module name in simulation
  TYPE condenser      !NMF model name
   UA_0 11400.0       !parameter
   MFAN 25.0          !parameter
 MODULE evaporator
  TYPE evap
   UA_0 197260.0
   K 2143.0
   cpliq 960.0
 MODULE cont
  TYPE prop_C
   Tr 2.0
END_MODULES

CONNECTIONS           !lists the interconnection between component models
 compressor.terminal_2 = con1.comp_link
 evaporator.comp_link = compressor.terminal_1     !connection at link level
 cont.terminal_2 = compressor.terminal_5
 evaporator.tout = cont.sig_in                    !connection at variable level
END_CONNECTIONS
```

```
BOUNDARIES          !assigns the necessary boundary conditions to some model
 con1.n_fans 3.0    !variables
 con1.Ta 5.0
 evaporator.mdot 150.0
 evaporator.Tin -20.0
 cont.setpoint -30.0
END_BOUNDARIES

START_VALUES        !gives the solver state variable values to use at the start of
 DEFAULT 1.0        !simulation
 compressor.Qcool 1000.0
 compressor.Qcond 1000.0
 con1.Tout 20.0
 con1.Q 20000.0
 evaporator.Tout -20.0
 evaporator.Q 20000.0
END_START_VALUES

INTEGRATION         !details of integration time
 FROM 0             !start time
 TO 1               !initial time step
 STEP 1.0           !relative numerical tolerance
 TOL 0.01           !absolute numerical tolerance
 TOL_LIM 1.0

 LIST               !sends output to results file
  OUT_ALL
 END
```

```
LOG                     !logs output variable values to file at the specified time
                        interval
 OUT_TIMES
  0 1 1
 END_TIMES

  FILE refrigs          !contents for output data file (.PRN)
    compressor.Qh Qh
    compressor.Ql Ql
    compressor.Qcool cool     !module name...variable name...column heading in
    compressor.Qcond cond     !output file
    con1.Tout T2
    con1.Q heat_flux
    evaporator.Tout EvT2
    evaporator.Q Evapheat_flux
    cont.sig_out X
 END
END_INTEGRATION
```

APPENDIX C

TRNSYS type 210

Steady-state evaporator model

```
SUBROUTINE TYPE210(TIME,XIN,OUT,T,DTDT,PAR,INFO,ICNTRL,*)
```

! XIN = Input variable vector

! OUT = Output variable vector

! PAR = Parameters for model

```
C***********************************************************************
C  Steady-state evaporator model
C  Version 1.2 by VIH on 19/6/1999
C  accepts inputs in W, J etc as per IDA
C  reconfigured to calculate evaporating temperature from refrigerating
C  effect
C***********************************************************************
    INCLUDE '\TRNWIN\KERNAL\PARAM.INC'
    DOUBLE PRECISION XIN,OUT
    INTEGER*4 INFO
    DIMENSION XIN(3),PAR(16),OUT(20),INFO(15)

    CHARACTER*1
    TRNEDT,PERCOM,HEADER,PRTLAB,LNKCHK,PRUNIT,IOCHEK,+PRWARN
    CHARACTER*3 YCHECK(3),OCHECK(4)
C
C        local variables
C
         real CpLiq,eff,K,Mdot,NTU,Q,Tfrig,Tin,Tout,UA,UA_0
C
         COMMON /LUNITS/ LUR,LUW,IFORM,LUK
         COMMON /SIM/ TIME0,TIMEF,DELT,IWARN
```

```
      COMMON /STORE/ NSTORE,IAV,S(NUMSTR)
      COMMON /CONFIG/
      TRNEDT,PERCOM,HEADER,PRTLAB,LNKCHK,PRUNIT,IOCHEK,+PRWAN
C
C     PARAMETERS
C
C     K      PAR(1) temperature gradient of UA             (-)
C     UA_0   PAR(2) base value of UA                       (W/m^2K)
C     CpLiq  PAR(3) specific heat of secondary fluid       (J/kg-K)
C
C     INPUT VARIABLES
C
C     Mdot   XIN(1) fluid mass flow rate                   (kg/s)
C     Tin    XIN(2) secondary fluid inlet temp             (C)
C     Q      XIN(3) refrigerating effect                   (W)
C
C     OUTPUT VARIABLES
C
C     Tfrig  OUT(1)evaporating temperature                 (C)
C     Tout   OUT(2)secondary fluid outlet temperature      (C)
C     Q      OUT(3)heat transfer rate                      (W)
C     Mdot   OUT(4)fluid mass flow rate                    (kg/s)
C
      K=PAR(1)              ! Mapping centrally stored
      UA_0=PAR(2)           ! variables onto local
      CpLiq=PAR(3)          ! variable names
      Mdot=XIN(1)           ! for greater
      Tin=XIN(2)            ! transparency
      Q=XIN(3)              !
C
C     FIRST CALL                          !
```

```
      IF (INFO(7).EQ.-1) THEN                        ! Validity check to
            NP=3                                     ! ensure model has
            NI=3                                     ! been inserted
            ND=0                                     ! correctly into simulation
            CALL TYPECK(1,INFO,NI,NP,ND)             !
            DATA YCHECK/'MF2','TE1','PW2'/           !
            DATA OCHECK/'TE1','TE1','MF2','PW2'/     !
            CALL RCHECK(INFO,YCHECK,OCHECK)          !
      END IF
C
      UA=UA_0+K*Tfrig                    !
      NTU=UA/(Mdot*CpLiq)                !
      eff=1-EXP(-NTU)                    ! The algorithm
      Tfrig=Tin-(Q/(Mdot*CpLiq*eff))     !
      Tout=Tin-(Q/(Mdot*CpLiq))          !
C
      OUT(1)=Tfrig                       !
      OUT(2)=Tout                        ! Mapping output from calculation
      OUT(3)=Q                           ! onto TRNSYS output vector
      OUT(4)=Mdot                        !
C
      RETURN 1
      END
```

APPENDIX D

IDA Simulation Development Methodology

IDA simulation development methodology

In order to run the Dynamic Link Library (DLL) of NMF components, created by the NMF translator and FORTRAN compiler, a problem for that system needs to be defined. The problem is described in a file given the extension .IDA. The file is of a specific format required by IDA Solver [1] and contains suitable boundary conditions in order for the simulation to be performed. The file contains a number of sections which must be included in the system description file, three of these sections are optional and need not be included in the file in order for it to work. A sample IDA input file of a simple system is included in Appendix B and should be referred to in conjunction with the following.

D.1 System description

The first section heading is ABSTRACT. In this section, contained within quotation marks (“”), a brief description of the system under consideration is made. There is no limit on the length of comment but the first two lines appear at the beginning of the output file, so should be relevant.

D.2 IDA Solver configuration

This section in the system description file is optional and need not be included. In this section the user can modify the configuration of IDA Solver. A large number of configuration options are available to the user all of which are detailed in Appendix 6 of the IDA Programming guide [1]. The solver can be modified in two other ways;

via the command line if the solver is being used in DOS mode or via the IDA.CFG file found in the project directory.

D.3 Files

Files external to the system description file may be used for input data to time dependent boundary variables, or for storing output data for later processing. These files and the paths to their locations may be found in the FILES section. External files used to input data to boundary variables must be in a format generated by saving the required data with a .PRN file extension.

The output file contains a table of values of variables selected in the INTEGRATION section. Each line of the table corresponds to one time step, but the user may specify the time period at which the output is logged.

D.4 Constants

If the values of several parameters are the same they can be specified in the CONSTANTS section. This is convenient if the values of these parameters are to be changed during the course of the work, they may be changed en-masse rather than having to change each parameter in each model instance. This section is optional.

D.5 Tables for time dependent input

The TABLES section allows values for time dependent boundary variables to be read in. A table consists of two or more columns; the first always being reserved for time and each subsequent column corresponds to one variable. The time steps in the table do not have to be equal and step changes in values are accomplished by giving two lines of the same time step have the same value. IDA Solver uses linear interpolation when a value in-between two of the given time steps are required.

There is no limit on the number of tables used in a system description file, but large files of time dependent input should be read in from external files to keep the input file at a manageable size. This section is optional.

D.6 Instances of NMF components

Each component used in the system formation must be declared in the MODULES section of the system description file.

Each module should be given a name that is relevant to it in the context of the system. As many modules of the same type are likely to exist in one system the names for the modules should be carefully chosen and should give an idea of the system structure.

The component's parameters and corresponding values for that module are listed. The parameters must be listed in the order in which they appear in the parameters section of the NMF model.

D.7 Interconnection of modules

There are two methods of connecting modules available to the user; these are connections of variables or connection of links and may be freely mixed. A link is a communication port for information flowing to and from a model. A link encapsulates all of the models connecting variables and can only be used to connect to another model with the same link type.

Only component variables of the same type may be linked together. If these variables refer to physical flows (i.e. mass flow or heat flux) their positive direction is indicated in the NMF model by POS_IN or POS_OUT. If two variables with the same direction of flow (i.e. POS_IN to POS_IN) are to be connected a minus sign must proceed them. When connecting at the level of links this is automatically taken care of.

Most Modular Simulation Environments require that connections between components be carried out at the variable level. IDA allows the user to connect the components at the level of links, which greatly cuts down the number of links appearing in the system description file. Only components having the same link types can be connected as the variable sets associated with the links are automatically connected.

D.8 Boundary conditions

In order to perform simulations a number of boundary conditions have to be specified for a number of variables appearing in the links. Boundaries are external conditions that are either unchanged with time or are read in from a table of time dependent input (5.3.5).The number of boundary condition required by the simulation is calculated by equation 5.1

$$Boundaries = Variables - Equations \tag{4.1}$$

At the beginning of a simulation IDA Solver checks for a balance between boundaries, connections and variables, this check takes the form of equation 5.2.

$$Balance = INVariables - Boundaries - \left(\frac{Connections}{2}\right) \tag{4.2}$$

The number produced from this check is compared to the number of IN/OUT variables, if there is no match then the simulation is aborted.

Boundaries may be specified in one of a number of ways.

i. The variable is assigned a value that remains the same throughout the simulation.
ii. The boundary variable can be defined as a constant in the CONSTANT section.
iii. A table of time dependent variables given in the TABLES section may be used for the variable input.
iv. Data may be read in from an external file, specified in the FILES section.

D.9 Initial values

Initial values for the simulation are given in the START_VALUES section of the file. A number of variables may be given values that might be expected to be found at those points of the system at the commencement of the simulation. If a variable is not given an initial value it is assigned its default value. This default value may be wholly inappropriate to the simulation in question. It is therefore essential that all variables not given boundary conditions are give appropriate initial valuẽs.

If the simulation is the continuation of work on a system from that which has been previously carried out, it is possible to use the final values given by the earlier simulation as a starting point for the new one. Changing the file extension of the output file from .END to .BEG and including it in the FILES and START_VALUES sections will accomplish this.

D.10 Integration

IDA simulates the behaviour of a system between two time points, *time 1* and *time 2*. If the problem is steady state *time 1* is set equal to *time 2*, and the calculation process begins. The time for the simulation to run as well as the initial time step and tolerances are given in the INTEGRATION section of the system description file. The unit of time used is seconds and multiples of to represent hours (0, 3600, 7200 etc.).

The accuracy of the integration is controlled in each time step by comparing the calculated solution with the predicted solution. The deviations are checked at variable level as prescribed by two values tolerance (TOL) and tolerance limit

(TOL_LIM). If the deviations are larger than the tolerance limit the step is retraced and a shorter step is tried.

The remaining sections under the INTEGRATION heading are concerned with the output from the solver. The LIST section is where the user specifies what is to appear in the results file. Either the output for every variable at every time step is written to the file or certain variables at certain time steps can be written to the file.

D.11 Simulation output

IDA Solver outputs three types of file at the end of a simulation, these are:

i. *.RES
ii. *.END
iii. *.PRN

D.12 Output file *.RES

This file contains a reproduction of the system input file as well as information about the IDA Solver version used and the time and date. The file lists all of the components used in the simulated system and their variable values at each time step. Some general information about the simulation run is presented in a section towards the end of the file, this information includes time for integration and number of steps.

D.13 Output file *.END

The *.END file contains a simple list of all the variables in the simulation and their values at the last time step. This file may be used as a starting point for the continuation of the simulation, by including it in the system description file in the START_VALUES section and changing it's file extension to *.BEG.

D.14 Output file *.PRN

Output to this file is specified in the INTEGRATION section of the system description file. The file will contain a table of variables and their values at the time step specified. The first column is reserved for the time co-ordinate and the headings for the remaining columns are as those chosen in the system description file. IDA Solver can be linked to Microsoft EXCEL, which allow the output data to be opened directly.

D.15 IDA Solver Beta test version 6.09

The simulation work for the project has been carried out using IDA Solver β-test version 6.09. The use of the solver has suffered from a lack of user documentation that has caused problems in identifying fundamental errors in the system description file. In the final release version there is still a lack of documentation but there is improved support and a greatly improved solver.

D.16 References

[1] IDA Programming Guide. Version 1.1β. Bris Data AB, Stockholm, Sweden. HTTP://www.Brisdata.Se/

[2] Levermore G.J. (1994). Building Energy Management Systems 1st Edition. E & F Spon, London. ISBN. 0-419-15290-3

Appendix E

Publications to date

Modelling the dynamic thermal response of insulated ducts

V.I.Hanby, E.C.Roberts, D.W.Fletcher

ISHVAC 99, November 1999, Shenzhen, China.

MODELLING THE DYNAMIC THERMAL RESPONSE OF INSULATED DUCTS

V I Hanby[1], E C Roberts[1] and D W Fletcher[2]

[1]Department of Civil and Building Engineering
Loughborough University, UK.
[2]Motor Industry Research Association
Nuneaton, UK.

Abstract

A method for the dynamic thermal modelling of a duct is proposed, based on discretization of the duct into a sequence of well-mixed flow nodes. This enables the time delay produced by the fluid flow in the duct to be simply modelled in any time domain simulation. An optimal level of discretization, based on fully developed turbulent flow, is presented.

The thermal response is based on a second order model for each discretized node, based on the thermal capacitance of the fluid and of the duct inner wall. It is demonstrated that the model predicts a time delay followed by a rapid initial response due to flow effects, followed by slower dynamics controlled by the thermal inertia of the duct walls. A comparison is made with two published dynamic models.

Introduction

Dynamic plant simulation

Dynamic models of HVAC plant components have been developed much less intensively than have building fabric elements. Generally, plant components react faster than the building fabric; typical response times are seconds or minutes as opposed to hours. In combined building/HVAC plant models, this results in the plant dynamics having limited overall effect on the total system performance and also the resulting stiff equation set can cause numerical difficulties. Dynamic plant modelling is essential if a simulation of the performance of its control system is required, and highly desirable if the system being modelled is subject to rapid changes in load.

A number of specialised simulation tools are used in dynamic plant simulation. An example of a general simulation environment that is being increasingly used in HVAC simulation work is MATLAB/SIMULINK [1], for which plant component models are becoming increasingly available [2]. This program represents dynamics using the classical frequency domain formulation, an approach that makes integration with building models very difficult. Models based on time domain formulation are much more flexible in this respect, with component models being widely available for programs such as ESP-r [3], HVACSIM+ [4], IDA [5], SPARK [6] and TRNSYS [7].

Plant components within these simulation environments are frequently modelled in the steady-state, or represented in a simple lumped parameter, first-order form. The additional complexity implicit in a more rigorous approach is often regarded as unjustifiable; also practical dynamic performance data, required for the calibration of more advanced models, is generally not available. This paper describes a fundamental approach to the modelling of components characterised by dynamics resulting from thermal capacitance, together with transport delays caused by internal fluid flow. Such components include heat exchangers, coils, pipes and ducts. The method described in the paper has been developed to allow the dynamic modelling of long, insulated ducts hence is described within this context.

Duct modelling

The dynamic response of a fluid conduit results from two basic mechanisms: the time required for a fluid element to flow through the component (residence time) and dynamic heat flow within the fabric of the component. This paper describes an approach, which combines both these elements in a form that enables a straightforward implementation of the model to be made in modular simulation environments.

The contribution of fluid flow to duct dynamics is generally based on the concept of a delay time, given by

$$t_d = \frac{V}{\dot{v}} \tag{1}$$

This has two limitations.

1. The concept is based on an idealised piston or plug flow model, which assumes a flat velocity profile across the duct.
2. In a time domain simulation, the time step is either fixed (e.g. TRNSYS) or automatically set to an efficient value by the solver (e.g. IDA). Access to the system time is either not possible, or can cause numerical instability.

A method of representing time delay has been described by Clark et al [8], in which the temperature distribution as a function of time and distance is given as a fifth order polynomial in distance, with time-dependent coefficients. The coefficients are determined by Gaussian elimination at each timestep in the simulation. The duct is discretized into five sections in this model.

$$T(x,t) = \sum_{i=0}^{5} a_i(t)x^i$$

The method described in this paper is based on modelling the duct as a discrete number of well-mixed nodes, each characterised by a single fluid temperature. The thermal response of each node is described by a second-order model and the number of nodes can be selected to give a representation of the time delay based on the residence time distribution in the duct. This approach does not require explicit access to the internal representation of time in the simulation program. If the time step is user-specified, then clearly this must be set to a value determined by the response of the duct.

Residence time distribution

The F diagram

A convenient way of considering residence time distribution is given by considering the following: at time $t=0$ a passive fluid property in a steady-flow system undergoes a step change (for example, the colour could change from clear to red). The variable $F(t)$ represents the fraction of red material in the outgoing fluid at time t. The analysis is further simplified if we use dimensionless time, given by $\tau = \frac{\dot{v}t}{V}$.

The actual shape of the F-diagram in a duct or pipe depends primarily on the velocity profile. Clearly the plug flow assumption is predicated on a flat velocity profile across the duct, hence all the red fluid elements arrive at the exit at the same time. If a velocity profile exists, then the faster moving elements near the centerline will arrive more quickly than the average. In the case of laminar flow, the maximum velocity is double that of the mean fluid velocity, hence red material would first appear in the exit flow at $\tau = 0.5$. The F-diagram for the limiting conditions of plug flow, well-mixed flow (in which an entering fluid element is instantaneously dispersed throughout the volume) and laminar flow are shown in Fig 1.

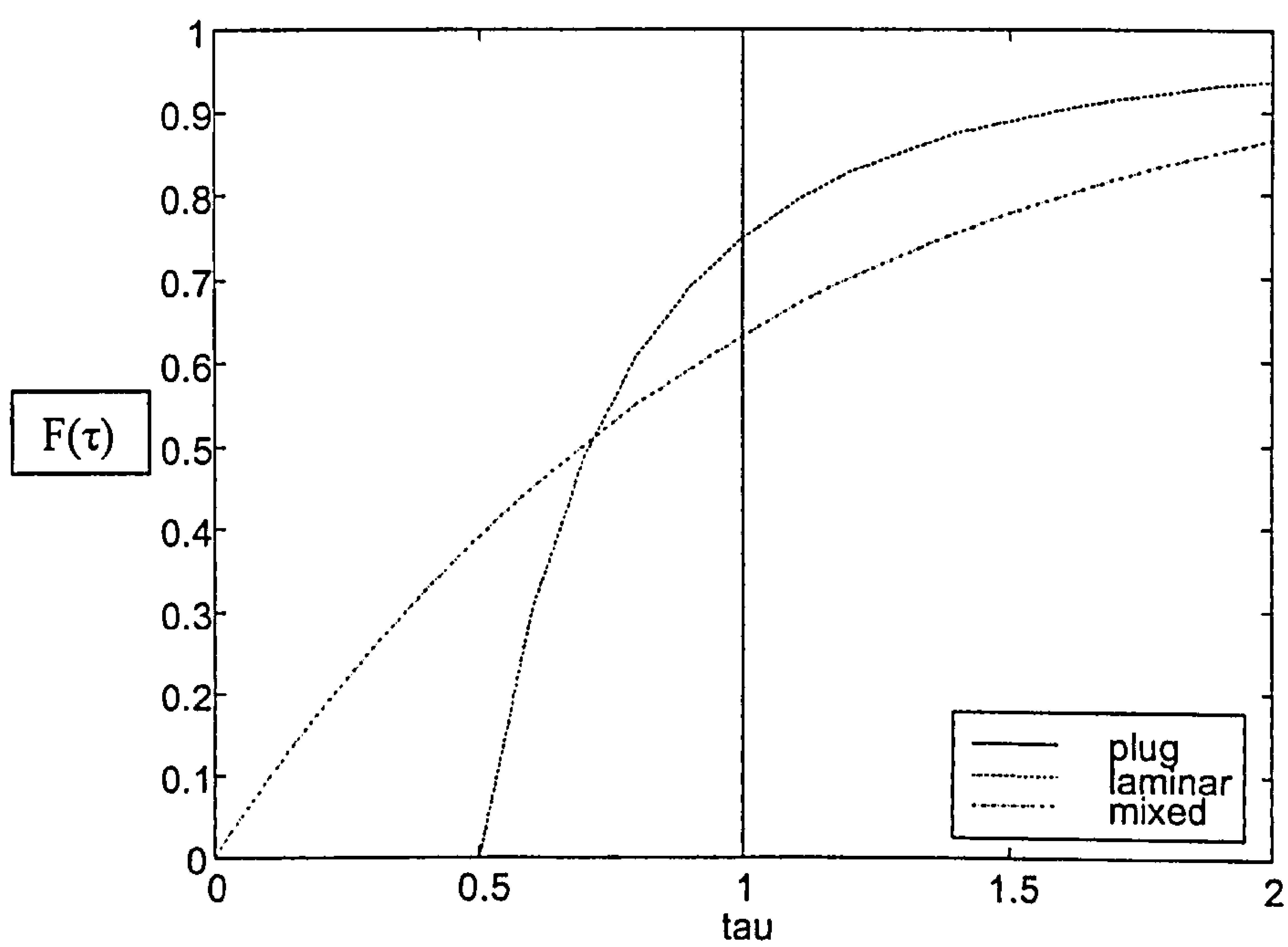


Fig 1. F-diagrams for plug, laminar and well-mixed flow.

Neglecting mixing, an F-diagram can be obtained by integration of the velocity profile. For laminar flow this easily shown to give

$$F(\tau) = 1 - \frac{1}{4\tau^2} \qquad \tau \geq 0.5$$

For any given situation, the F diagram can be obtained by three means.

1. Direct experiment measurement (using a tracer fluid, as in ventilation measurements).
2. Numerical modelling using CFD techniques.
3. A closed-form equation, taking into account velocity profile and diffusivity, to incorporate mixing.

The last of these approaches is considered in this paper.

Turbulent velocity profile-based F diagram for a circular duct

A power law velocity profile is assumed:

$$u(r) = u_{\max}(1 - r)^{\frac{1}{n}}$$

The maximum velocity $u_{\max}$ is related to the mean $\bar{u}$ by

$$\bar{u} = 2\frac{u_{\max}}{\left(\frac{1}{n}+1\right)\left(\frac{1}{n}+2\right)}$$

$n = 7, \quad 2000 < N_{Re} < 100\,000$

$n = 8 \quad N_{Re} > 100\,000$

A correction factor allows for the effects of eddy diffusivity as a function of distance from the duct wall [9]:

$$\beta = \frac{(n-1)^2 R}{0.32nL}$$

The combined effects of velocity profile and diffusion give the following expression for F

$$F(\tau) = 1 + \frac{n+1-\tau^n(2n-1)}{n(\tau^n)^2\tau} + \frac{\beta(n+1)(2n+1)^2(2\tau^n-1)(\tau^n-1)^2}{4n^2(\tau^n)^4} \tag{2}$$

where the third term (containing β) is the diffusive correction factor. This F diagram is shown in Figure 2 for $n = 7$ and $\beta = 0.01$. A similar expression was derived by Bosworth [10] and compared with experimental results which showed that equation (2) tended to over-predict the extent of longitudinal mixing. It should be noted that the length: diameter ratio of the duct only affects the diffusivity correction term. A fully developed, power-law velocity profile is assumed throughout the length of the duct.

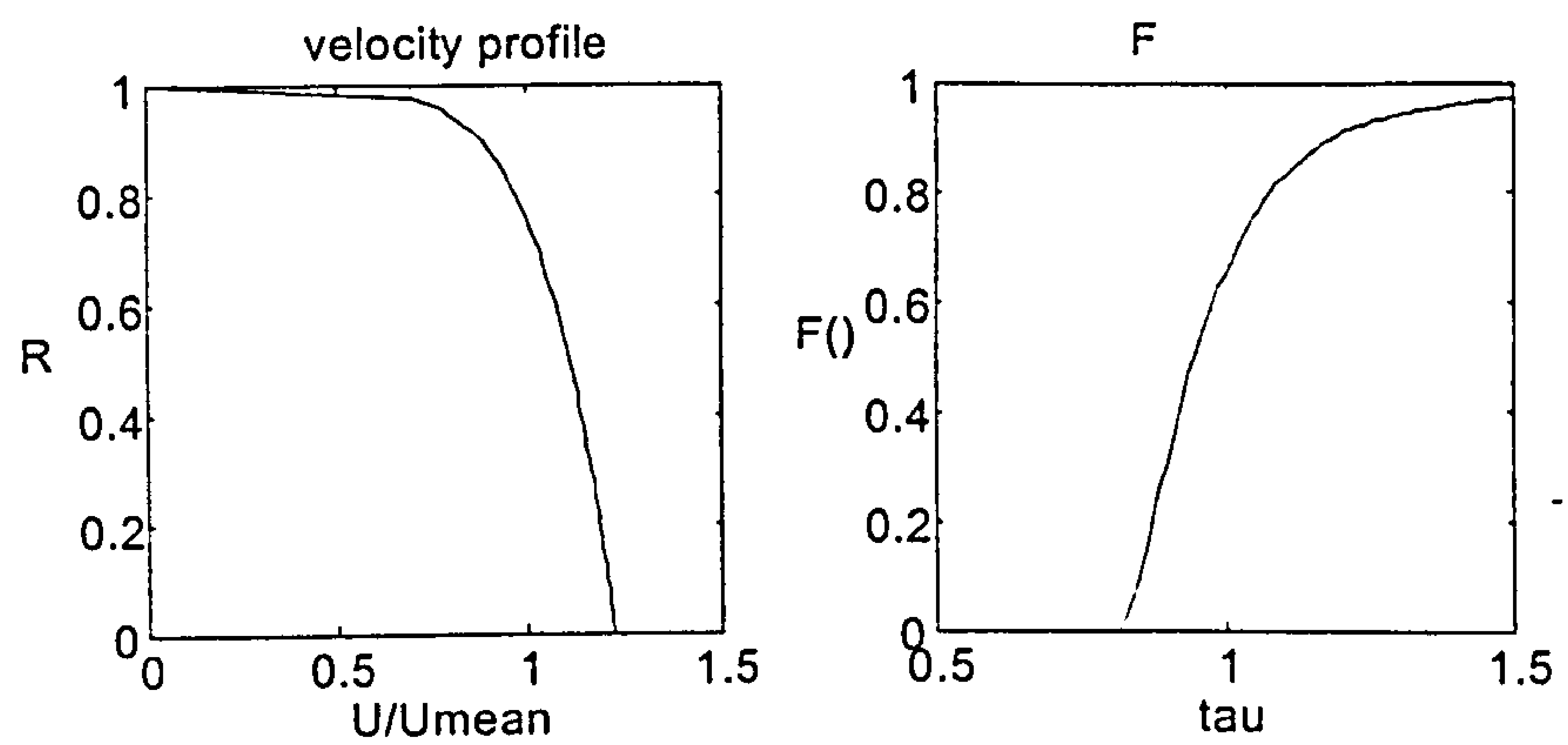


Fig 2 Velocity profile and F-diagram for turbulent flow in duct

Well-mixed node model

The approach adopted in this paper is to obtain an approximation to a benchmark F diagram (here taken as that given by equation (2)) by defining a model consisting of a number of well-mixed nodes in series. This gives a simple modular structure to the model, with computational simplicity and stability, as each well-mixed node is first order with respect to the fluid flow.

The single, well-mixed zone is a familiar model in building studies, as it is the basis for most room ventilation analyses. For a step change in input the response is a simple exponential rise.

$$F(\tau) = 1 - e^{-\tau}$$

If we consider a number of such nodes in series (i) such that the volume of each node is V/i, then it can be shown that the resulting F diagram is given by

$$F(\tau) = 1 - e^{-i\tau}\left\{1 + i\tau + \frac{1}{2!}(i\tau)^2 + \ldots \frac{1}{(i-1)!} i\tau^{(i-1)}\right\} \qquad (3)$$

As the number of nodes is increased, the order of the response rises: as the number of nodes approaches infinity the response approaches that of plug flow. Figure 3 shows the F-diagram for 2, 20, 40 and 80 nodes in series, together with the analytical result given by equation (2).

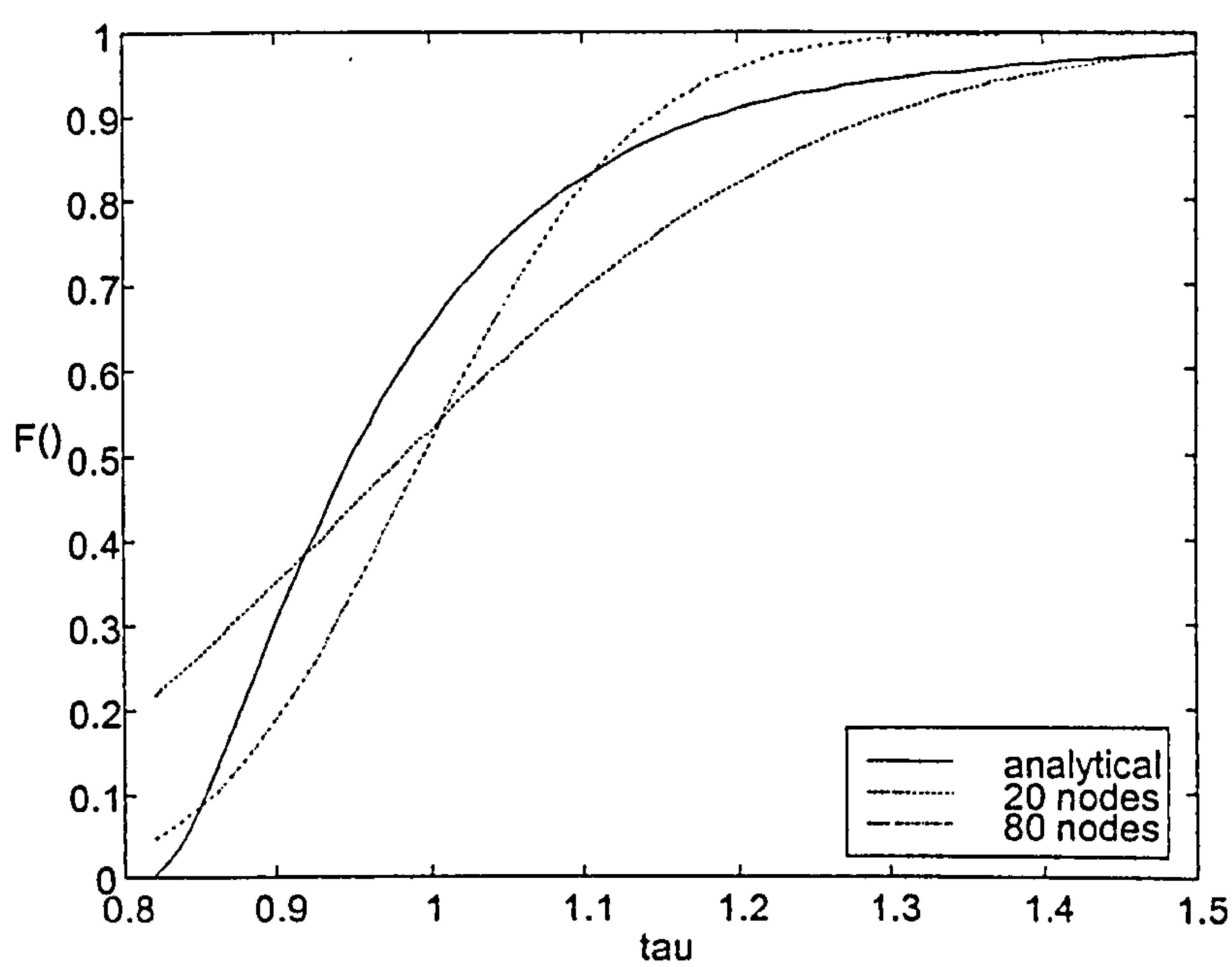


Fig 3. F-diagrams for well-mixed nodes in series.

The optimum number of nodes was established by evaluating the area enclosed between the relevant curve and that of the benchmark, over the range $0.8 < \tau < 1.5$

The results are shown in Figure 4, which shows that the optimum number of nodes is 46, but that 20 gives a reasonable approximation to the optimum.

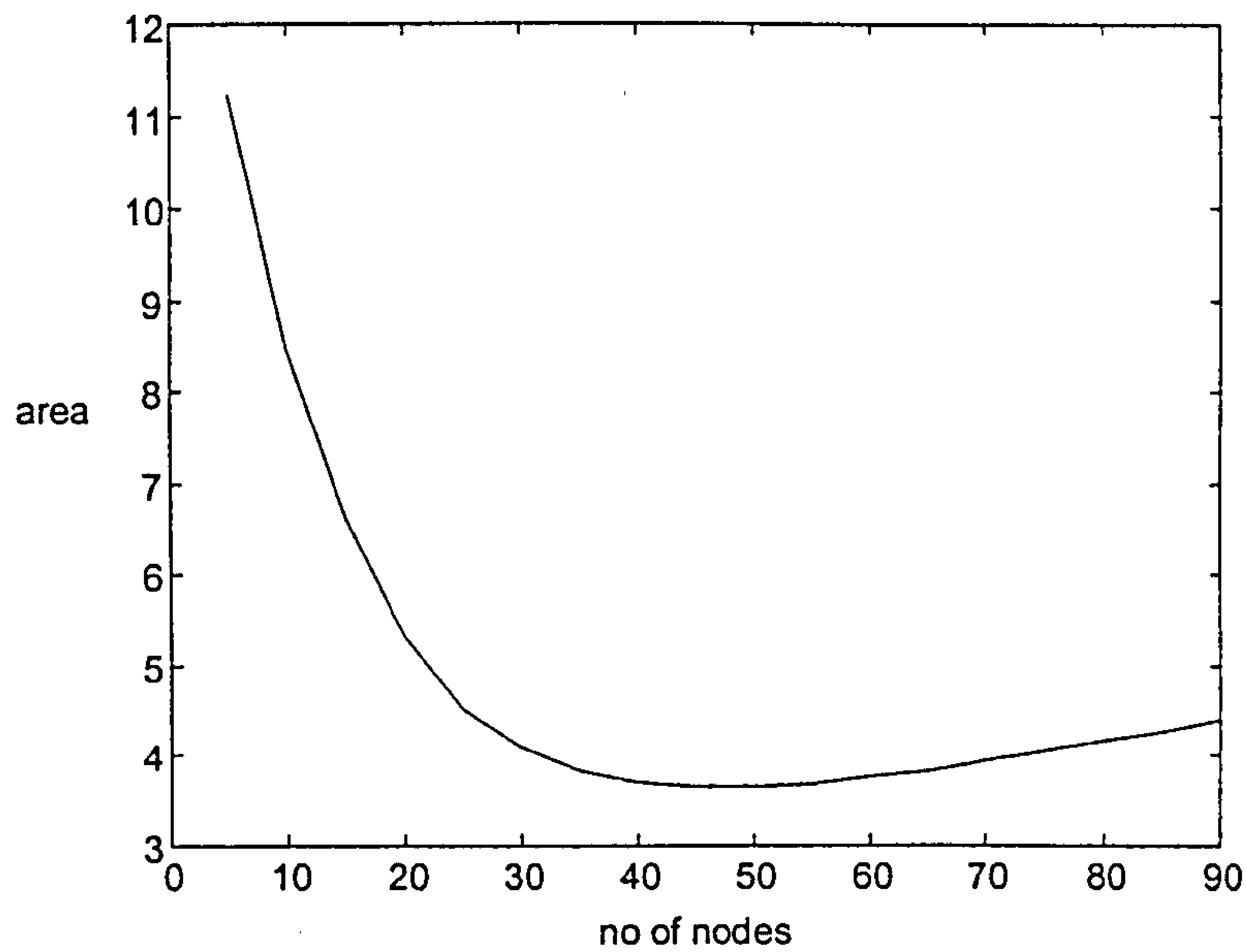


Fig 4. Optimal representation of duct.

Thermal dynamics

The prototype duct which formed the basis of this study is of standard refrigeration configuration, 1mm steel inner lining, 50 mm insulation, 1mm steel outer lining. The thermal properties are summarised in Table 1.

material	k (W m^{-1}K^{-1})	ρ (kg m^{-3})	c_p (J kg^{-1}K^{-1})
steel	60.5	7854.0	434.0
PU insulation	0.026	70.0	1045.0

Table 1 Thermal properties of duct materials (after [11]).

The inside heat transfer film coefficient was calculated from a standard correlation for forced convection [11]

$$N_{Nu} = 0.023(N_{\mathrm{Re}})^{0.8}(N_{\mathrm{Pr}})^{0.4}$$

and the outside film coefficient from empirical data for convective and radiative heat transfer for pipes in still air [12].

Dynamic radial heat transfer was modelled by a discrete nodal scheme. Assuming an internal film coefficient of 8.5 Wm^{-2} K^{-1}, the Biot number for the inside steel liner was 1.4 x 10^{-4}, hence justifying a lumped capacitance for this layer. The duct is 21.2 m long, has a cross sectional area of 0.64 m^2 and under normal operation has a mean residence time of 2.8 seconds.

In order to obtain the most compact representation, a comparison was made between the following modelling schemes.

1. One capacitance node in each steel layer and three in the insulation layer.
2. One capacitance node in each of the three layers.
3. A capacitance node in the inner steel layer only.

These alternatives were evaluated by comparing the heat flux transferred into the duct wall resulting from a unit step change in inside fluid temperature. The comparison of methods (1) and (3) is shown in Figure 5: it is apparent that the differences are small, hence option (3) was chosen as the most compact thermal network representation.

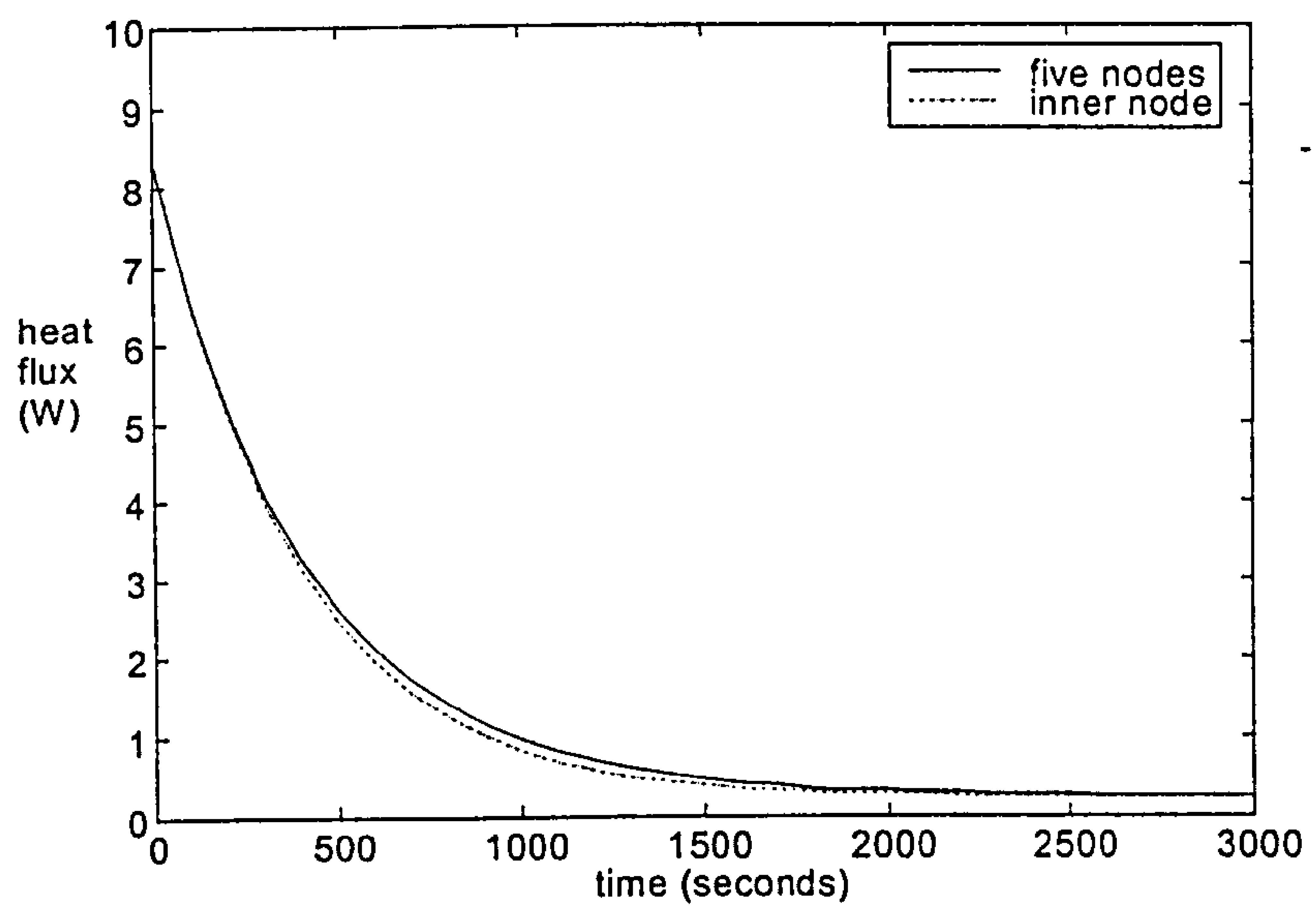


Fig 5 Comparison of duct element thermal models.

Simulation of duct response

Thermal network

A diagram of the thermal network representation of the duct is shown in Fig 6. Each element consists of a well-mixed node, a conductance between the fluid and the inner layer (the thermal resistance of the steel layer is neglected) the thermal capacitance of the inner layer and a conductance (insulation and external film coefficient) between the capacitance and the outside. An energy balance on each duct node yields

$$C_a \frac{dT_i}{dt} = \dot{m}c_p(T_{i-1} - T_i) - h_c A_i (T_i - T_{w,i})$$

To implement the model, four component models (duct node, liner thermal capacitance, conductance and Kirchhoff node) were written in Neutral Model Format [13] and the model equations solved using IDA solver.

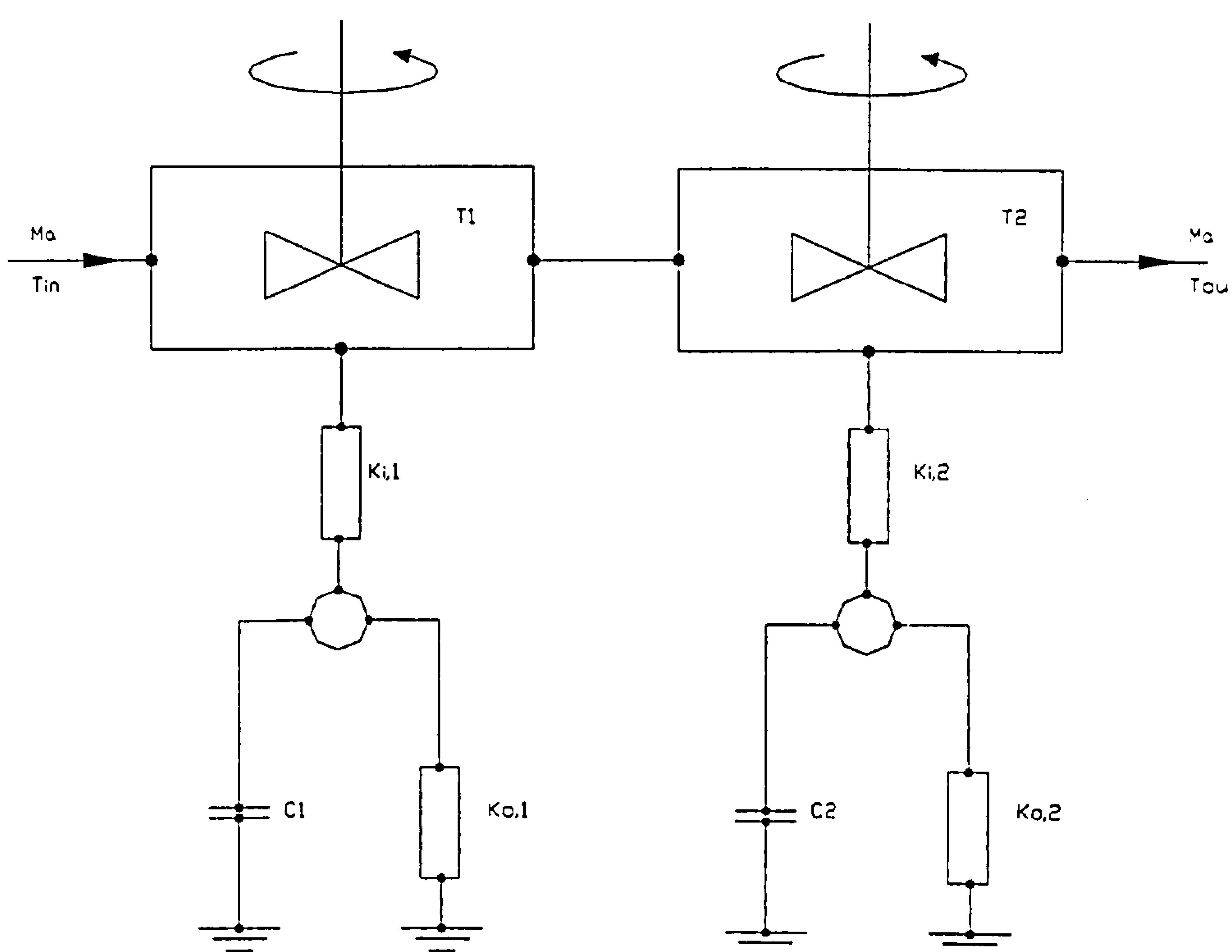


Fig 6. Schematic of discretized duct thermal model.

Model output

The model was used to predict the outlet temperature of air in the duct in response to a unit step increase in air temperature at the inlet, for different levels of duct discretization. The results are shown in Fig 7, for varying numbers of nodes (i). It can be seen that the method predicts a time delay followed by a rapid rise in outlet temperature (caused by the fluid transport), then a much slower rate of temperature increase due to the thermal inertia of the duct walls. At least 10 nodes are needed to give a reasonable modelling of a time delay. The duct walls have a time constant of the order of 600 seconds, which is significant in the context of overall plant dynamics.

Inter-model comparison

It has not proved possible to measure the dynamic response of the prototype duct in order to obtain a comparison with experimental data. The model output has therefore been compared to that of three published dynamic duct models. Tobias [13] derived a transfer function for the response of fluids flowing through ducts coils or pipes. The governing partial differential equations expressed the fluid temperature as a function of time and distance, but the wall temperature as a function of time only, hence the thermal capacitance of the wall was lumped and not longitudinally distributed. Tobias introduced a simplification to transform the partial differential equations into ordinary differential equations.

A simplified model was described by Clark [8]. This superimposed a first order dynamic response onto a time delay given by

$$t_d = \frac{V}{\dot{v}}$$

the dynamics were first order with respect to the fluid temperature.

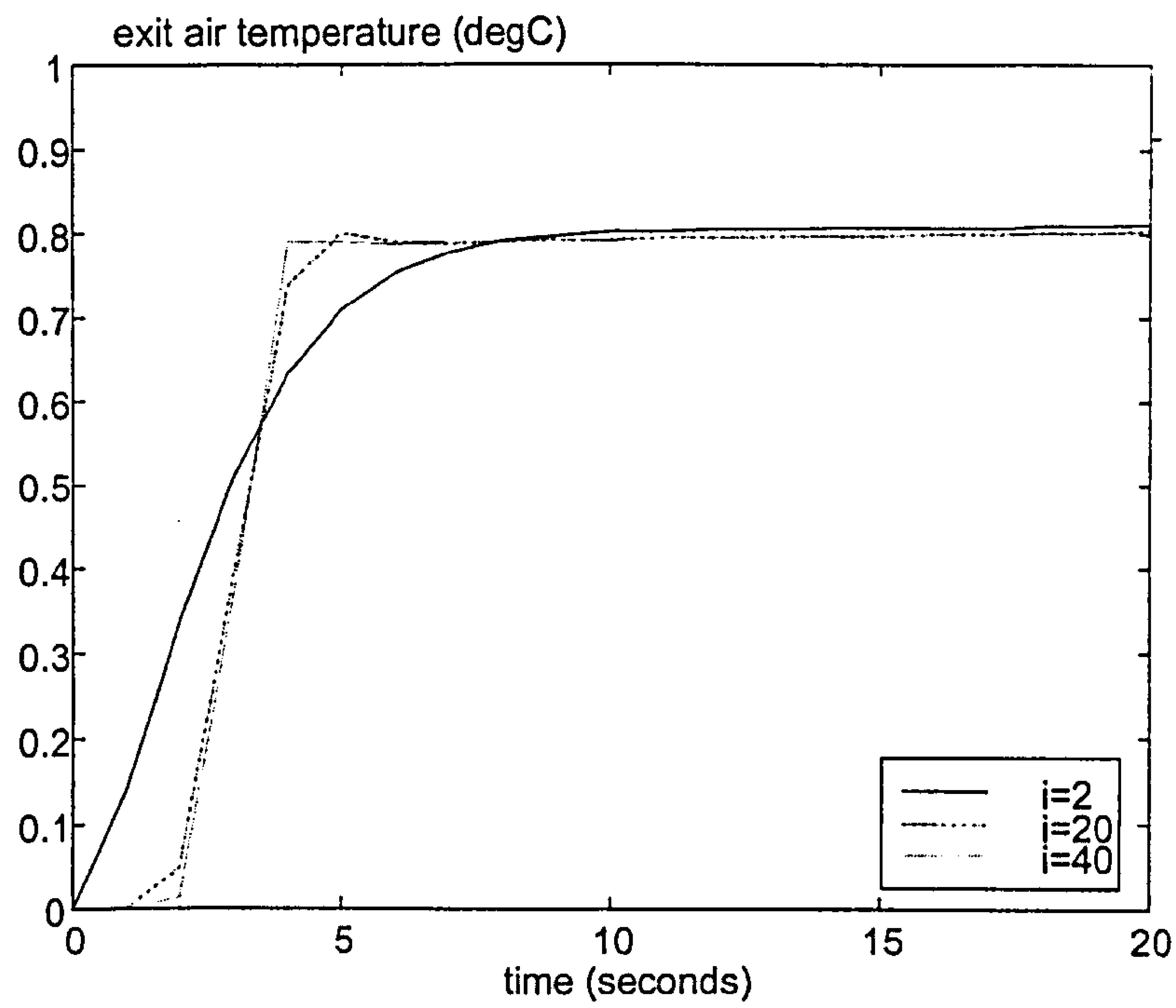


Fig 7 Response of the model to step temperature input.

$$\frac{dT_{out}}{dt} = \frac{T_{ss} - T_{out}}{t_c} \tag{4}$$

where the time constant t_c is given by $\left[\frac{h_i}{h_i + h_o}\right]\frac{C_m}{\dot{m}c_p}$

Clark [15] derived a time-domain transformation of Tobias' approximate transfer function. This model has three components: a time delay as per equation (4), a step rise in temperature followed by a first order temperature rise. For a unit step input:

$$T_{out} = \lambda + (1-\lambda)(1-e^{-\frac{t}{t'_c}}) \qquad \text{(for } t > t_d\text{, } T_{out} = 0 \text{ otherwise)} \tag{5}$$

where $\lambda = e^{-\frac{h_i A}{\dot{m} c_p}}$

$t'_c = t_c e^{-\alpha}$ and

$$\alpha = \frac{UAh_i}{2\dot{m}c_p h_o}$$

A comparison of the response of the discretized model with equations (4) and (5) is shown in Fig 8. It can be seen that the discretized model has a higher initial increase in temperature than that predicted by equation (5), followed by a slower rise in exit air temperature. The simplified model of equation (4) does not produce a rapid initial temperature rise, but this is somewhat compensated for by the fact that it has a smaller time constant than the more detailed model (5).

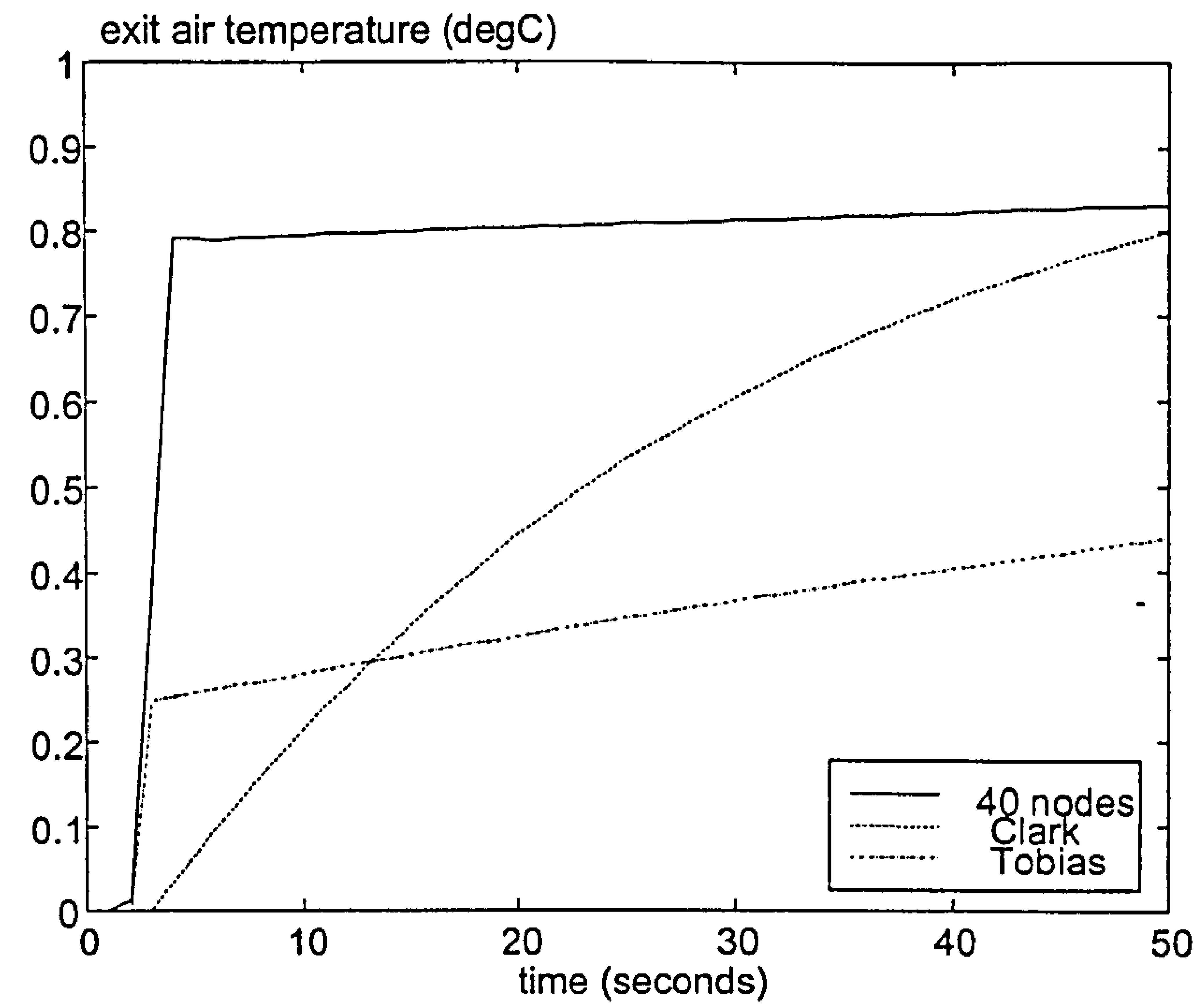


Fig 8. Inter-model comparison of response to a unit step input.

It is difficult to comment on these comparisons without experimental data to use as a benchmark. The short-term dynamics (0 – 10 s) are defined by the fluid flow in the duct, the longer-term response by heat conduction and capacitance effects. The model described differs from the published closed-form solutions in having its thermal capacitance lumped in each discretized segment, hence distributed longitudinally. The published models assume a single duct material temperature.

A longitudinal conduction link was originally incorporated into the model to account for conduction along the inner lining of the duct. This was abandoned as longitudinal heat fluxes were found never to exceed 10^{-4} of the radial values for any duct segment.

Conclusion

A dynamic model for the thermal response of duct/pipe systems is described, based on discretization of the duct into well-mixed nodes. Consideration of the residence time distribution in a duct, calculated from the radial velocity profile and eddy diffusivity of the flow, indicates an optimal level of discretization of 46 nodes, although satisfactory performance should be obtained with around 20. This result is independent of the length: diameter ratio of the duct, but assumes fully-developed flow throughout.

The method generates a response that includes a characteristic time delay and does not need explicit access to the system time in a dynamic simulation environment. If the simulation program internally generates the time step, these dynamics should appear automatically. If the time step is user-specified, then it must be set to a lower value than the mean residence time of the fluid in the duct.

A comparison has been made with other published dynamic duct models: the discretized model shows a steeper initial response (caused by the fluid flow characteristics), followed by a more gradual temperature rise due to the thermal dynamics of the duct material. Immediate future development is aimed at producing experimental data of duct and pipe thermal response for model validation.

List of symbols

a	polynomial coefficient	-
A	surface area	m^2
c_p	specific heat (constant pressure)	$kJ\ kg^{-1}K^{-1}$
C_a	fluid thermal capacitance	$kJ\ kg^{-1}K^{-1}$
C_m	duct wall thermal capacitance	$kJ\ kg^{-1}K^{-1}$
F	fraction of passive fluid component	-
h_c	convective heat transfer coefficient	$W\ m^{-2}K^{-1}$
h_i	inner surface convective heat transfer coefficient	$W\ m^{-2}K^{-1}$
h_o	outer surface convective heat transfer coefficient	$W\ m^{-2}K^{-1}$
i	integer	-
k	thermal conductivity	$W\ m^{-1}K^{-1}$
L	duct length	m
$\dot{m}$	fluid mass flow rate	$kg\ s^{-1}$
n	integer	-
N_{Nu}	Nusselt number	-
N_{Re}	Reynolds number	-
N_{Pr}	Prandtl number	-
r	relative radial position	-
R	duct radius	m
t	time	s
t_c	time constant	s
t_d	time delay	s
T	temperature	C
T_{out}	fluid outlet temperature	C
T_{ss}	steady-state fluid temperature	C
T_w	wall temperature	C

u	longitudinal velocity	$m\ s^{-1}$
U	overall coefficient of heat transfer	$W\ m^{-2}K^{-1}$
$\dot{v}$	volume flow rate	m^3s^{-1}
V	system volume	m^3
x	longitudinal distance	m
β	diffusivity correction	-
ρ	density	$kg\ m^{-3}$
τ	dimensionless time	-

References

1. Technical literature on MATLAB 5.3, The Mathworks, Boston, USA.

2. Husaunddee, A, Riederer, P and Visier, J-C. "Coil modelling in the SIMBAD toolbox – numerical and experimental validation of the cooling coil model". Paper #32, *Proceedings of SSB '98*, University of Liege, Belgium (1998).

3. ESRU, Energy Simulation Research Unit, Faculty of Engineering, University of Strathclyde, UK.

4. Park, C. D R Clarke and G E Kelly. "An overview of HVACSIM+, a dynamic building/HVAC/control simulation program" *Proceedings of the First Annual Building Energy Simulation Conference*, Seattle, WA (1985).

5. Technical literature on IDA Solver, Bris Data AB, Vasterlanggata 27, 111 29 Stockholm, Sweden.

6. Ayres et al. "SPARK Installation and User Guide", Ayres Sowell Associates Inc. USA (1997).

7. Klein et al. "TRNSYS - a transient simulation program", Solar Energy Laboratory, University of Wisconsin-Madison, Madison, WI (1996).

8. Clark, D R, Hill C R and Hurley C W. "Dynamic models for HVAC system components". *ASHRAE Transactions*, 91, pp737 – 751, (1985).

9. Danckwerts, P V. "Continuous flow systems", *Chemical Engineering Science*, 2. No 1, pp 1 – 13, (1953).

10. Bosworth, R C L. *Phil. Mag.* 39, pp 847 – 850, (1949).

11. Incropera, F P and D de Witt. "Introduction to heat transfer" Wiley, New York, (1990).

12. McAdams, W H. "Heat transmission" McGraw-Hill, p. 127, (1954)

13. Sahlin, P and E F Sowell. "A neutral model format for building simulation models" Proceedings of Building Simulation '89, IBPSA, Vancouver (1989).

14. Tobias, J R. "Simplified transfer function for temperature response of fluids flowing through coils, pipes or ducts" ASHRAE Transactions, 79, pp 19 – 22, (1973).

15. Clark, D R. "HVACSIM+ building systems and equipment simulation program reference manual" US Department of Commerce, National Bureau of Standards, Gaithersburg, MD, USA (1985).

Acknowledgement

The authors would like to acknowledge the Motor Industry Research Association for financial support of this work and Professor R A Smith of Loughborough University for helpful and stimulating discussions.

APPENDIX F

Model parameters and boundaries used in validation studies and applications to operational strategies

Vehicle model:

Vehicle at 0 Km/h:

Model: car

Parameter	Parameter value
Tau_on	303
Tau_off	6805
TSS	97.0
Pe	38000.0
Pr	18000.0
Cpa	1005.0
Tex	175.0
Alpha	1.0
V	0.0
K	1.1
C	121370.0

Soakroom system:

The following are the parameters used in all studies involving the soakroom. The only addition would be a vehicle at 0 Km/h.

Trichloroethylene flow temperatures and valve set points are read in from input tables and contain real data, due to their size they are not reproduced here.

Module: duct return

Parameter	Parameter value
Tenv	20
U	3.84
Cpair	1005
A	51.34

Module: heater and valve controllers

Parameter	Parameter value
TR	1

Module: airzone

Parameter	Parameter value
V	1800
Tenv	15

Module: Heatflux node to connect to airzone

Parameter	Parameter value
N_link	2

Module: heater

Parameter	Parameter value
UA	150
Qmax	50000
W	10000
M	14
Tenv	15
Cpliq	960

Module: dry coil

Parameter	Parameter value
UA	9500
Cpair	1005
Cpvap	1860
Cpliq	960
Ma	8.36

Module: duct supply

Parameter	Parameter value
Tenv	15
U	3.84
Cpair	1005
A	67.25

Module: Heatflux node for lightweight elements

Parameter	Parameter value
N_link	2

Module: Heatflux node For heavyweight elements

Parameter	Parameter value
N_link	2

Module: inner conductance for lightweight elements

Parameter	Parameter value
K	1799.9

Module: outer conductance for lightweight elements

Parameter	Parameter value
K	54.4
T2	15

Module: inner conductance for heavyweight elements

Parameter	Parameter value
K	900

Module: outer conductance for heavyweight elements

Parameter	Parameter value
K	50.0
T2	15

Module: thermal capacitance of lightweight elements

Parameter	Parameter value
C	7.36e5
T2	15

Module: thermal capacitance for heavyweight elements

Parameter	Parameter value
C	35.0e6
T2	15

Module: two speed fan

Parameter	Parameter value
Speed	1
Cpair	1005

Test chamber validation:

The following are the parameters used in all studies involving the soakroom. The only addition would be a vehicle at a relevant velocity.

Trichloroethylene flow temperatures and set points are read in from input tables and contain real data, due to their size they are not reproduced here.

Module: heater

Parameter	Parameter value
UA	150
Qmax	20000
W	10000
M	100
Tenv	15
Cpliq	960

Module: dry coil

Parameter	Parameter value
UA	50000
Cpair	1005
Cpvap	1860
Cpliq	960

Module: duct supply

Parameter	Parameter value
Tenv	15
U	3.84
Cpair	1005
A	425.88

Module: duct return

Parameter	Parameter value
Tenv	20
U	3.84
Cpair	1005
A	327.6

Module: heater and valve controllers

Parameter	Parameter value
TR	1

Module: airzone

Parameter	Parameter value
V	231.4
Tenv	15

Module: Heatflux node to connect to airzone

Parameter	Parameter value
N_link	2

Module: Heatflux node for lightweight elements

Parameter	Parameter value
N_link	2

Module: Heatflux node For heavyweight elements

Parameter	Parameter value
N_link	2

Module: inner conductance for lightweight elements

Parameter	Parameter value
K	1799.9

Module: outer conductance for lightweight elements

Parameter	Parameter value
K	54.4
T2	15

Module: inner conductance for heavyweight elements

Parameter	Parameter value
K	900

Module: outer conductance for heavyweight elements

Parameter	Parameter value
K	50.0
T2	15

Module: thermal capacitance of lightweight elements

Parameter	Parameter value
C	7.36e5
T2	15

Module: thermal capacitance for heavyweight elements

Parameter	Parameter value
C	35.0e6
T2	15

Module: Single speed fan

Parameter	Parameter value
Power	385000
Cpair	1005

Module: Stem humidifier

Parameter	Parameter value
Effsat	0.95
Patm	101325

Refrigeration system:

The following are the parameters used in all studies involving the refrigeration system.

Trichloroethylene flow temperatures and set points are read in from input tables and contain real data, due to their size they are not reproduced here.

Module: condenser

Parameter	Parameter value
UA_0	11400
Mfan	25
Fanpower	1200
n-fans	20
Tair_on	15

Module: evaporator

Parameter	Parameter value
UA_0	197260
K	2143
Cpliq	960
M	100

Module: evaporator

Parameter	Parameter value
TR	2

Air make-up system:

The following are the parameters used in all studies involving the air make-up plant.

Trichloroethylene flow temperatures and set points are read in from input tables and contain real data, due to their size they are not reproduced here.

Module: heater

Parameter	Parameter value
UA	150
Qmax	50000
W	10000
M	100
Tenv	20.95
Cpliq	960

Module: CCSIM

Parameter	Parameter value
ConfigHX	1
Qtotrat	21260
Qsenrat	13860
Cpliq	960
Mliqrat	28.28
Tliqrat	0
Mairrat	1
Tairrat	26
Wairrat	0.011
Dprate	100
Dp	0

Module: duct in

Parameter	Parameter value
Tenv	15
U	3.84
Cpair	1005
A	4.62

Module: duct return

Parameter	Parameter value
Tenv	20
U	3.84
Cpair	1005
A	52.36

Module: heater and valve controllers

Parameter	Parameter value
TR	1

Module: Single speed fan

Parameter	Parameter value
Power	2000
Cpair	1005