Thermax

1 2 3 Excel

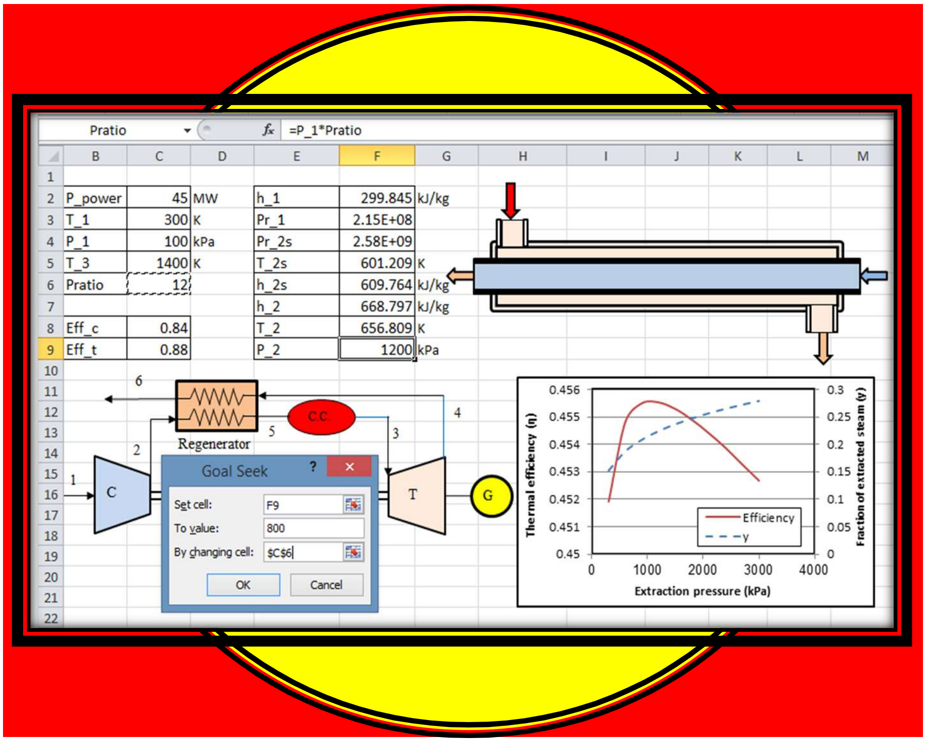
Research

Design

Fundamentals

1

Computer-Aided Analyses and Optimisation of Fluid-Thermal Systems using Excel



Mohamed M. El-Awad

September, 2025

This book is dedicated to the memory of my parents

El-Awad and Amna

May Allah bless their souls in Heaven

Preface

The three thermofluid subjects (thermodynamics, fluid mechanics, and heat-transfer), which are fundamental components of the curricula for most engineering specialisations, are traditionally taught by using data tables for fluid properties and charts for approximate analytical solutions of complex systems and design-based analyses. Although the repetitive use of tables and charts helps the students to acquire the three lower-order thinking skills (LOTS); *Remember, Understand,* and *Apply*, of tables and charts are not suitable for conducting sensitivity and design optimisation type of analyses that help them to acquire the higher-order thing skills (HOTS); *Analyse, Evaluate,* and *Create*. In this respect, computer-aided learning (CAL) methods can be more effective as shown by research conducted at a number of top-class universities. CAL methods are also more suitable than traditional methods for online teaching that has now become a necessity rather than a choice. Realising the benefits of CAL, most thermofluid textbooks now include computer-based exercises and mini-projects. Although a number of commercial software are available and can be used for such exercises and projects, these might not be available or affordable to many students and academic institutions particularly in developing countries. General-purpose spreadsheet applications, such as Microsoft Excel, can be the ideal solution to this problem.

With its wide availability on personal computers, ease of use, and powerful graphical tools, Excel has been used as a computational platform in various engineering courses. Numerous papers on the use of Excel as a teaching aid have also been published in relevant journals or presented at specialised conferences over the past two decades. However, most of the previous experiences and efforts, if not all, focus on a particular type of analyses or a particular feature of Excel and do not describe a general pedagogical approach. Moreover, the papers, which are written in the form of short research articles, cannot provide sufficient information for using Excel as an effective educational modelling platform for the various types and levels of thermofluid analyses. This book is the first volume in a set of books that complement the previous efforts by describing an Excel-based modelling platform which is adequate for computer-aided thermofluid analyses and showing how the platform can be used for conducting the various types of these analyses. The book adopts a learning-by-example approach and many of its examples have been adopted from standard thermofluid textbooks so that the students can verify their Excel solutions and look for any additional information if required.

The Excel-based modelling platform described in the book has four elements; (i) Excel's user-interface and built-in functions, (ii) the Solver add-in that comes with Excel, (iii) Microsoft Office Visual Basic for Applications (VBA) programming language, and (iv) an Excel add-in called Thermax that provides numerous VBA functions for fluid properties as well as a couple of useful numerical tools. While Excel's user-interface and Solver are adequate for most fluid-dynamics and heat-transfer analyses, the Thermax add-in is needed for thermodynamic analyses while VBA is needed for the development of custom functions when the analytical model cannot be executed by only using Excel's built-in functions and Thermax functions. Appropriately used, the Excel-based modelling platform minimises the effort of developing the analytical models for thermofluid analyses so that more attention can be paid to the application of the relevant principles.

Exercises are given at the end of most chapters of the book that help the students to sharpen their skills related to the particular topic and more challenging exercises and mini projects are provided as an appendix. Certain topics that require detailed treatments are also discussed in the appendices that supplement the main chapters. The book provides a sufficiently detailed and organised material for a stand-alone course on computer-aided analyses and optimisation of fluid-thermal systems, but it can also be used to supplement the existing courses of fluid mechanics and heat-transfer. Although it has been written mainly for educational purposes, it is hoped that the book can also be useful for practicing engineers and researchers in the field.

Acknowledgements
The development of Thermax and the writing of this set of books would not have been possible without benefitting from the efforts of many colleagues who have made their publications, data, and software available in the open literature or on their websites. A special gratitude goes to the Mechanical Engineering Department at the University of Alabama (USA) whose initiative "*Excel for Mechanical Engineering*" both inspired and helped me to develop Thermax. I am also indebted to Universiti Putra Malaysia and Universiti Tenaga Nasional (Malaysia), the University of Khartoum (Sudan), and the University of Technology and Applied Sciences (Oman) for their generous support at different periods of my academic career and hope that they find my efforts a worthy token of appreciation and gratitude. Last, but not least, I am grateful to my beloved family for the unfailing support I needed desperately to complete this work at a very challenging time.

CONTENTS

1

Introduction

The electrical energy required to operate the refrigerators and air-conditioners and many other essential appliances in our daily life mainly comes from power-generation plants that burn fossil fuels. Fossil fuels are also the main source of energy in the industrial and transport sectors. Apart from being non-renewable, large-scale combustion of fossil fuels is responsible for the increasingly devastating effects of global warming at different parts of the world. Therefore, proper design of energy transportation, conversion, and utilisation systems is crucial for curtailing these effects. The design of these systems is mainly based on the principles of thermofluids: *thermodynamics*, *fluid mechanics*, and *heat transfer*. While thermodynamic analyses usually assume steady-state and isotropic fluid properties, fluid mechanics and heat-transfer analyses take into consideration the temporal and spatial variations of the fluids properties. Therefore, fluid mechanics and heat-transfer analyses lead to multi-dimensional and/or non-linear equations which are difficult to solve without using computer-aided methods. Many dedicated applications have been developed for computer-aided thermofluid analyses, but there are also many advantages for using the general-purpose software Microsoft Excel for educational purposes. This chapter reviews the main principles of fluid mechanics and heat transfer, highlights the advantages of computer-aided thermofluid analyses, and describes the Excel-based modelling platform used throughout this book for these analyses.

1.1. A brief review of fluid-dynamics and heat-transfer

The two main principles of thermofluid analyses are the conservation of mass (the continuity equation) and the conservation of energy (the first-law of thermodynamics) that are used to obtain mathematical equations for analysing the relevant systems and processes. The analytical models developed for the systems by applying these two principles take different mathematical forms depending on whether the system under consideration is open or closed and whether the flow is steady or unsteady. Since the flow can be laminar or turbulent, compressible or incompressible, etc., and the heat-transfer can be by conduction, convection, or radiation, the analyses need numerous auxiliary relationships in order to quantify the various parameters involved in the main equations such as pressure-variations, friction losses, and rates of heat-transfer. Considering the wide scope of the topic, the main concepts of fluid dynamics and heat-transfer are reviewed here by considering typical applications of each subject.

1.1.1. Fluid-dynamics

In addition to pipes and ducts, fluid-transporting systems require various equipment such as pumps and compressors, control valves, flow-diversion devices and flow-measuring devices. The principles of *fluid dynamics* help us to estimate the power needed for overcoming friction in these equipment and to determine suitable types and sizes for them. To illustrate the application of these principles, consider the pump-pipe system shown in Figure 1.1 that conveys a liquid between two non-pressurised tanks A and B through a pipe of known length L, diameter D, and roughness ε. Suppose that we want to determine the needed pump power for transporting the liquid at a certain flow rate Q. The required pump power ($\dot{W}$) is determined from the following "power equation" [1]:

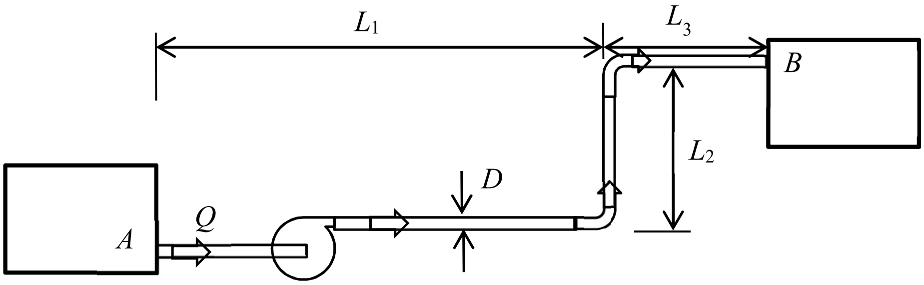


Figure 1.1. Schematic diagram of a simple pump-pipe system

$$\dot{W} = \frac{\gamma \times Q \times h_p}{\eta} \tag{1.1}$$

Where γ is the specific weight of the transported liquid, Q is the volume flow rate of the liquid, h_p is the pump head needed to circulate the fluid through the pipe from A to B, and η is the combined efficiency of the pump and the electric motor. For a steady flow of an incompressible fluid, h_p can be determined from the following "energy equation":

$$h_p = h_{f,total} + (Z_B - Z_A) + (V_B^2 - V_A^2)/2g \tag{1.2}$$

Where $h_{f,total}$ is the total head loss through the system due to friction, Z_A and Z_B are the elevations at points A and B, respectively, and V_A and V_B are the corresponding fluid velocities. If the two tanks are not open to the atmosphere, the energy equation should include another term for the pressure difference between the tanks. The total friction head loss $h_{f,total}$ consists of two parts: the *major friction loss* (h_f), which is the part lost in the pipe itself, and the *minor friction head loss* (h_c), which is the part lost in other components of the system like nozzles, elbows, valves, etc. The major friction loss can be determined from the following Darcy-Weisbach equation:

$$h_f = f\frac{L}{D}\frac{V^2}{2g} \tag{1.3}$$

Where f is the dimensionless Darcy friction factor, V the fluid velocity, L the total length of the pipe, and D the internal diameter of the pipe. The value of the friction factor, which depends on the roughness of the pipe surface and on whether the flow is laminar or turbulent, can be obtained from a Moody diagram or calculated from a relevant formula. For laminar flows, it can be calculated from:

$$f = 64/\mathrm{Re} \qquad \mathrm{Re} < 2300 \tag{1.4}$$

Where Re is the Reynolds number defined as:

$$\mathrm{Re} = VD/\nu \tag{1.5}$$

Where ν is the kinematic viscosity of the flowing fluid. For a turbulent flow in rough pipes, f can be obtained from the following Swamee-Jain formula:

$$f = 0.25 / \left[\log_{10}\left(\frac{\varepsilon}{3.7D} + \frac{5.74}{\mathrm{Re}^{0.9}} \right) \right]^2 \qquad \mathrm{Re} > 4000 \tag{1.6}$$

For more accuracy, the friction factor for a turbulent flow can be determined by using the following Colebrook-White formula (frequently referred to as the Colebrook equation):

$$\sqrt{\frac{1}{f}} = -2.0 \log_{10}\left(\frac{\varepsilon / D}{3.7} + \frac{2.51}{\mathrm{Re}\sqrt{f}} \right) \tag{1.7}$$

The Colebrook equation is an example of the implicit equations met in thermofluid analyses that need to be solved iteratively. For turbulent flows in smooth tubes, f can be determined from the first Petukhov formula:

$$f = \left(0.790\ln(\mathrm{Re}) - 1.64\right)^{-2} \qquad 10^4 < \mathrm{Re} < 10^6 \tag{1.8}$$

Chemical engineers usually determine the pipe friction by using the following Chezy-Manning equation instead of the Darcy-Weisbach equation:

$$h_f = 2f \frac{L}{D} \frac{V^2}{g} \tag{1.9}$$

Where f is the Fanning friction factor. Comparison with Equation (1.3) reveals that the value of the Fanning friction factor is four times the corresponding value of the Darcy friction factor. Civil engineers determine the friction head loss in water-transporting pipes by using the following Hazen-Williams equation:

$$h_f = \frac{10.67\, L Q^{1.852}}{C^{1.852} D^{4.8704}} \tag{1.10}$$

Where C is a coefficient that depends on the roughness of the pipe. Unlike Equations (1.3) and (1.09), Equation (1.10)) is applicable for both laminar and turbulent flows.

The minor friction losses, h_c, can be determined from the following equation:

$$h_c = \sum_{1}^{n} K \frac{V^2}{2g} \tag{1.11}$$

Where n is the total number of components in the fluid system and K is a coefficient the value of which can be found for each component in relevant tables.

Given the values of the length and diameter of the pipe and its material or roughness and the flow rate and fluid viscosity, the equations described above can be used to determine the required pump power. In principle, the equations can also be used to determine the maximum flow rate of the fluid to be delivered via a pipe of a certain diameter such that the friction loss in the system or the needed pump power does not exceed a specified limit. Moreover, by taking into consideration the initial cost of the pump-pipe system (which increases with D), and the cost of electrical energy needed by the pump (which decreases with D), the equations can also be used to determine the economic pipe diameter D_{opt} that gives the lowest total owning cost for the system over its entire life-time. The equations are also applicable for analysing and optimising pipe-networks.

The principles of fluid dynamics also enable us to select the appropriate type and size of the pump for a given pump-pipe system by matching the "pump curve" with the "system curve". This is achieved with the help of pump characteristic curves usually provided by the manufacturers. In many situations a single pump or compressor may not be adequate for the required flow rate or delivery pressure and more than one pump or compressor have to be used. In this situation, the principles of fluid dynamics help us to decide when to arrange the pumps/compressors in parallel or in series.

1.1.2. Heat transfer

The design practices of energy-conversion equipment that deal with the transfer of thermal energy such as boilers, condensers, and heat exchangers are mainly based on the principles of *heat transfer*. Three independent physical laws are used in heat-transfer analyses to quantify the *rate* of heat transfer between an object and its surroundings depending on whether the transfers is by conduction (Fourier's Law), convection (Newton's law of cooling), or radiation (Stefan-Boltzmann law). The physical properties that determine the rate of heat transfer by conduction, radiation, and convection are the thermal conductivity (k), the surface emissivity (ε) and absorptivity (α), and the heat-transfer coefficient (h), respectively. While k, ε, and α are material or surface-specific, h depends on both the fluid and the flow. Numerous analytically-obtained relationships and empirical formulae are used for determining h depending on whether the flow is forced or natural and whether the flow is internal or external to the system being considered. These formulae usually give the Nusselt number (Nu) which is related to h as follows:

$$h = \frac{k}{D} Nu \tag{1.12}$$

Where D is the pipe's diameter. Many analytical or empirical formulae are used for determining the Nusselt number for forced or natural flows over single tubes, bank of tubes, plates, etc. For example, the following Dittus-Boelter equation is used for determining Nu inside a fluid-transporting pipe due to forced convection [2]:

$$Nu = 0.023\,\mathrm{Re}^{0.8}\,\mathrm{Pr}^{n} \tag{1.13}$$

Where Re is the Reynolds number, Pr the Prandtl number, and n is a constant that takes a value of 0.4 when the pipe is being heated and 0.3 when it is being cooled.

The subject also describes the methods that can be used to minimise or maximise the rate of heat-transfer between the system's components or between the system and its surroundings by means of thermal insulation, fins, heat-pipes, etc. To illustrate the use of heat-transfer concepts in thermal-insulation analyses, consider the metal pipe shown in Figure 1.2 that has an internal radius r_1 and external radius r_2. The pipe carries a fluid at a temperature T_i, while the surrounding air is at a different temperature T_∞.

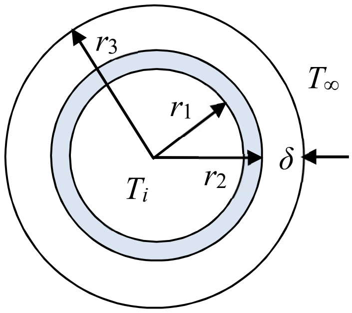


Figure 1.2. Schematic for an insulated metal pipe

The temperature difference between the pipe and the surroundings causes heat gain or heat loss to/from the pipe and, in order to reduce this undesired heat gain or heat loss, the pipe has to be covered by an insulating material. The principles of heat transfer help us to account for the effect of thermal insulation on the rate of heat-transfer ($\dot{Q}$) to/from the pipe which can be calculated from [2]:

$$\dot{Q} = \frac{T_i - T_\infty}{R_{th}} \tag{1.14}$$

Where R_{th} is the combined thermal resistance to heat-transfer by conduction, convection, and radiation, which is given by:

$$R_{th} = \frac{1}{h_i A_1} + \frac{\ln(r_2/r_1)}{2\pi L k_1} + \frac{\ln(r_3/r_2)}{2\pi L k_2} + \frac{1}{h_o A_3} \tag{1.15}$$

Where h_i and A_1 are the heat-transfer coefficient and surface area inside the pipe, respectively, h_o and A_3 are the heat-transfer coefficient and surface area outside the insulated pipe, respectively, L is the length of the pipe, and k_1 and k_2 are the thermal conductivities of the pipe and the insulation, respectively. To simplify the analysis, h_o in Equation (1.15) is allowed to take into account the heat-transfer by both convection and radiation to/from the insulation surface. The thickness of the metal pipe is usually small compared to its diameter, while its thermal conductivity is much higher than that of the insulation material. Therefore, the equation can be simplified further by neglecting the term that represents the thermal resistance due to conduction through the pipe.

For other design applications Equations (1.14) and (1.15) are used to determine the required thickness of insulation (δ) for reducing the rate of heat transfer to the required tolerance or for controlling the surface temperature within a range that is dictated by safety or other practical considerations. Although the thicker the insulation the lower will be the rate heat transfer, the cost of insulation increases with its thickness and, therefore, adding more insulation may not be economically profitable beyond a certain thickness. By extending the above heat-transfer model so that the cost of insulation and the value of the saved thermal energy can be calculated and compared, the above equations can also be used to determine the economically optimal thickness of insulation (δ_{opt}).

In many design applications the objective is to enhance the transfer of heat instead of minimising it. Figure 1.3 shows a metal pipe with circular fins attached to its surface so as to boost the rate of heat-transfer between the fluid being transported with the pipe and the surrounding medium, usually air. In this respect, the principles of heat transfer can be used to develop the required mathematical models that determine the the rate of heat transfer from the pipe so as to evaluate the effectiveness and efficiency of the fins.

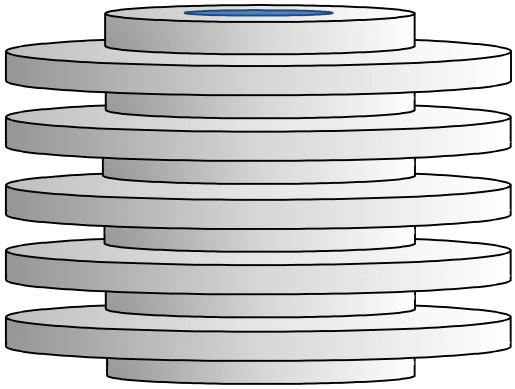


Figure 1.3. Circular fins attached to a metal pipe

Another important application of heat-transfer principles is that related to the design and selection heat-exchangers. A heat-exchanger is any device that allows the transfer of thermal energy between two fluids through a separating surface usually a pipe, a duct, a tube, or a plate. Figures 1.4 and 1.5 show two types of heat-exchangers commonly used in industries and power-plants.

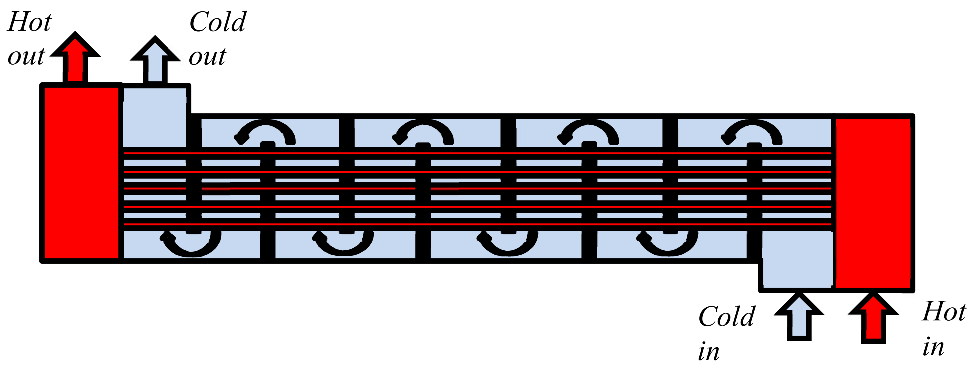


Figure 1.4. A parallel-flow shell-and-tube exchanger

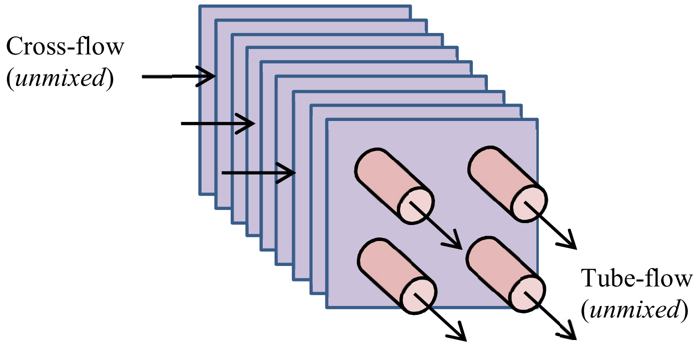


Figure 1.5. A cross-flow exchanger with both streams unmixed (adapted from [2])

Figure 1.4 shows a shell-and-tube heat-exchanger, while Figure 1.5 shows a cross-flow heat-exchanger. Heat-exchanger analyses either aim at determining the required size (i.e., surface area) for a specified heat-transfer duty or determining the exit temperatures of the two streams from a specified heat-exchanger type and size. Two methods are used for these two types of analyses which are the log-mean temperature difference (LMTD) method and the effectiveness-number of transfer units (ε-NTU) method. Complex thermal systems use heat-exchanger networks [HENs] and finding the configuration that minimises the annual cost of the network is also based on the principles of heat transfer.

1.2. Advantages of computer-aided thermofluid analyses

Apart from saving time and eliminating possible human errors, computer-aided thermofluid of analyses offer a number of advantages over traditional analytical methods that use property tables and charts. An important advantage of computer-aided methods is their ability to give more realistic results by avoiding unnecessary simplification of the models and by using more accurate formulae for fluid properties. Moreover, they offer reliable techniques for iterative solutions and optimisation analyses and for the analyses of complex fluid-thermal systems. In what follows, these advantages are illustrated by means of relevant examples.

A. Avoiding excessive simplification of the model

In many situations, traditional analytical methods adopt excessive simplifications of the analytical models which makes their results grossly deviate from the behaviour of real systems. A good example of this situation is given by the models of internal-combustion (IC) engines. Traditional air-standard models of IC engines, such as the Otto cycle and the Diesel cycle, neglect heat-transfer and friction losses, treat the combustion process as heat-addition from an external source, and use constant specific heats of the working fluids. These assumptions enable the engine processes to be represented by simple closed-form relations for calculating the amount of heat added to the engine and net work from it [3]. However, air-standard models usually overestimate the engine's output and thermal efficiency. By comparison, computer-aided models of IC engines closely mimic the behaviour of actual IC engines by taking into consideration the geometrical as well as the thermodynamic characteristics of the engines. Therefore, these models can be used to investigate the effect of important design and operation factors such the ignition or injection timing on the engine performance or the effect of engine' speed on the specific fuel consumption. However, the formulation of these models leads to a set of ordinary differential equations that need to be solved simultaneously by using a numerical method such as the Newton-Raphson method [4].

B. Accurate representation of fluid properties and processes

The ideal-gas law ($Pv=RT$) can be used with reasonable accuracy to determine the specific volume of a superheated vapour. However, when the temperature approaches the saturation line, the value of the specific volume thus determined departs significantly from the actual volume. More accurate estimates can be obtained by using more complex models such as the following Soave-Redlich-Kwong (SRK) equation of state [5]:

$$P = \frac{R_u T}{\tilde{v} - b} - \frac{a\alpha}{\tilde{v}(\tilde{v} + b)} \tag{1.16}$$

Where P is the absolute pressure of the gas, $\tilde{v}$ is the molar specific volume, R_u is the universal gas constant, T is the absolute temperature of the gas, and the constants a, b and α are fluid-dependent. Figure 1.6 shows the deviations from the tabulated values by those obtained from the ideal-gas law and the SRK equation of state for refrigerant R134a at 0.2 MPa. The figure shows that the error of the ideal-gas law is more that 2% even at high temperatures and increases as the temperature approaches the saturation value, but the accuracy of the SRK equation remained higher than 99% even close to the saturation line. However, the SRK equation is implicit in $\tilde{v}$ and, therefore, it cannot be used directly to determine the specific volume. A number of standard iterative procedures (e.g. Newton-Raphson method) can be used to solve the equation, but they are more suitable for computer-aided analyses than hand calculations. Another important implicit equation for thermofluid analyses is the Colebrook-White equation, Equation (1.07), that determines the friction factor (*f*) for turbulent pipe-flows. Since the equation involves *f* on both sides and needs to be solved iteratively, the explicit relationships such as the Swamee-Jain formula are preferred in conventional analytical models even though the Colebrook-White equation is more accurate. Many other nonlinear equations like the SRK equation and the Colebrook-White equation give advantage to computer-aided thermofluid analyses by enabling more realistic and accurate estimations.

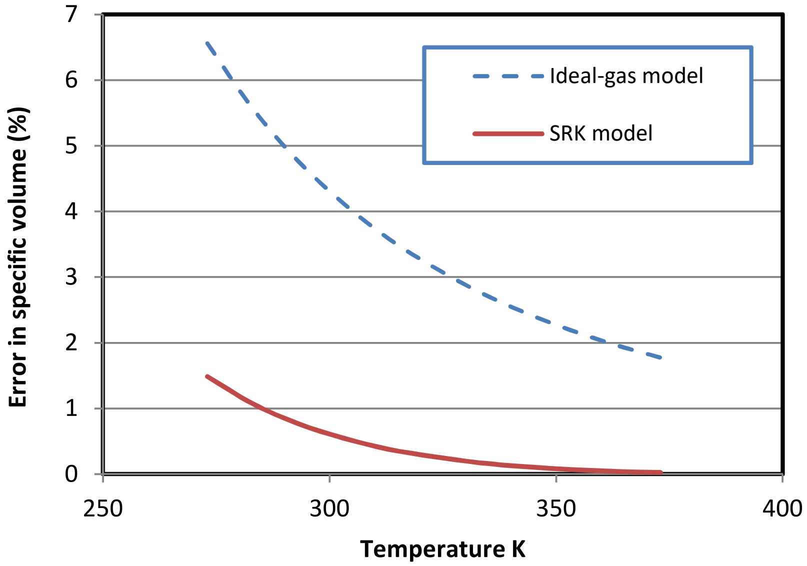


Figure 1.6. Errors in the specific volume of R134a by the ideal-gas law and the SRK equation of state

C. Dealing with iterative solutions and optimisation analyses

Thermofluid analyses that require iterative solutions are very common. A good example of these is given by the pump-pipe analyses discussed in Section 1.1.2. The problems that require the friction head loss to be determined when both the diameter and flow rate are known can be solved in a straightforward manner by using Equation (1.03). However, in design analyses of pump-pipe systems we may need to find the flow rate in a given pipe that gives a specified head loss or to find a suitable pipe diameter for specified head loss, flow rate, and pipe length. In these two cases, the friction factor *f* cannot be determined in advance because it depends on the Reynolds number. Therefore, these two types of pipe-flow problems, referred to as type-2 and type-3 problems, need to be solved by iteration. It is much easier to carry out the iterative process to the required level of accuracy by using a computer-aided method than by doing it manually. Two examples of the analyses that require iterative solutions in heat-transfer and thermodynamics are the rating analyses of heat exchanger and the determination of the adiabatic flame temperature by first-law analysis of the combustion process.

Optimisation analyses are needed for determining the best design for a fluid-thermal system such as the optimum intermediate pressure for the two-stage air-compression system and the optimum steam-extraction pressure for the regenerative Rankine cycle discussed in Section 1.1.1 and the economic insulation thickness for a pipe discussed in Section 1.1.3. While certain simple optimisation analyses that involve a single design parameter can be performed by means of calculus techniques and graphic tools, optimisation analyses of complex systems that involve multiple design variables and multi-objective optimisation analyses require the use of computer-aided techniques.

D. Analyses involving complex models

The complexity of modelling certain fluid-thermal systems makes their analyses only possible with the help of computer-aided methods. The model complexity can be either due to the complexity of the physical structure of the system itself or the complexity of its mathematical representation. An example of the physically complex systems is the pipe network shown in Figure 1.7 that consists of four pipe loops and four consumption points fed by two water tanks; tank A and tank B. Suppose that the flow rates from the two supply tanks are specified together with the pipe diameters and lengths and it is required to determine the discharges at the four consumption points. Although the solution is mainly based on the principles of fluid dynamics discussed in Section 1.1.2, it is difficult to solve the problem by using manual analytical methods especially when a minimum or a maximum pressure level is to be met at the discharge points. In this case, a computer-aided method, such as the Hardy-Cross method, has to be used [6, 7]. The optimisation analyses of heat-exchanger networks give another example of the models that deal with physically complex systems [8].

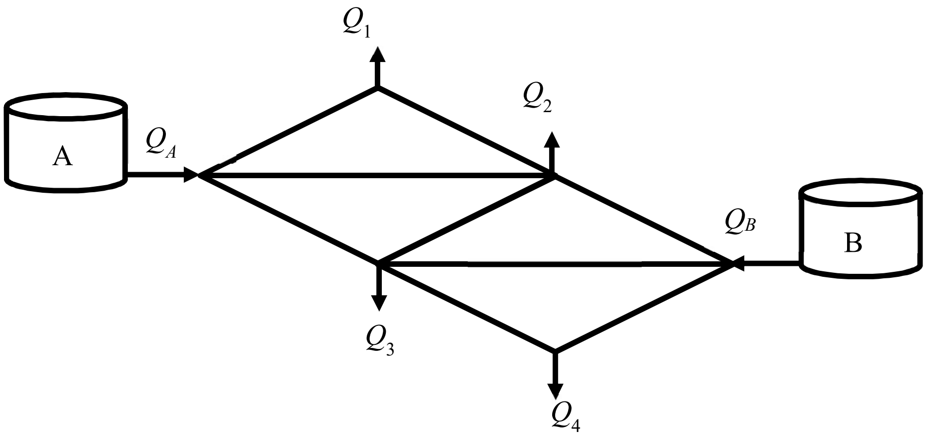


Figure 1.7. A looped pipe network supplied by two tanks

Examples of the mathematically complex thermofluid models that need computer-aided numerical methods are found in multi-dimensional fluid-flow and heat transfer analyses. This type of analyses involves coupled and nonlinear partial differential equations that have to be solved by using computational fluid dynamics (CFD) methods such as the finite-volume method or the finite-difference method. These types of analyses in particular require the use of dedicated software because the same software can be used for many fluids and system configurations. Only a dedicated software can give us the ability to choose the fluid type and system configuration and provide the required information regarding the flow whether it is compressible or incompressible, laminar or turbulent, with or without mixing or chemical reaction, etc., as well as the information required by the numerical method to construct its grid. Luckily, many commercial CFD applications are available nowadays that offer great flexibility and user-friendliness.

1.3. The Excel-based modelling platform for thermofluid analyses

Microsoft Excel is commonly used for data visulation and analyses and has also been used for dealing with simple computer-based operations like matrix inversion and matrix multiplications. However, Excel is equipped with numerous features and tools that make it a capable modelling platform for a wide range of engineering analyses including its Goal Seek command and the Solver add-in [9-11]. The "Developer" ribbon in Excel provides a programming language called Visual Basic for Applications (VBA) that can be used for developing customised user-defined functions (UDFs) not provided by Excel. The Developer ribbon also allows the use of macros to remove the tedium of parametric studies and repetitive calculations. The main limitation of Excel for thermofluid analyses, which is the lack of built-in functions for fluid properties, could be resolved by various academic and research institutions via the development of suitable add-ins [12-15].

This book uses an Excel-based modelling platform that includes, in addition to Excel, Solver, and VBA, an educational add-in called Thermax [16-19]. Thermax provides seven groups of property functions for ideal gases, saturated water and superheated steam, synthetic and natural refrigerants, atmospheric humid air for psychrometric analyses, two aqua solutions for vapour-absorption refrigeration, chemically-reacting substances, and air at standard atmospheric pressure for the usual fluid dynamics and heat-transfer analyses. Thermax also provides two interpolation functions and a Newton-Raphson solver for nonlinear equations that enhance the usefulness of the Excel-based modelling platform. Table 1.1 summarises the roles of the four components of the Excel-based modelling platform as used in this book.

1.4. Closure

The following three chapters describe the Excel-based modelling platform in more details and illustrate its use for a common type of computer-aided thermofluid analyses which is iterative solutions. Chapter 2 focuses on the features of Excel that are mostly needed for computer-aided thermofluid analyses, such as its matrix functions and its Goal Seek command. Chapter 3 introduces the other three components of the modelling platform, gives examples of using the three solution methods offered by Solver, and describes the development of user-defined functions with VBA. Chapter 3 also shows how the property functions provided by Thermax can be used in Excel formulae. Chapter 4 gives examples of using Excel's Goal Seek command and Solver for iterative solutions in the fields of fluid dynamics, heat-transfer, and thermodynamics.

Table 1.1. Roles of the four components of the Excel-based modelling platform

Component	Role
Excel	• Provides the basic functions needed for the development of mathematical models including the general mathematical functions and the matrix-operation functions • Provides the Goal Seek command that can be used to perform unconstrained iterative solutions involving a single parameter • Allows circular calculations which can be a convenient method for dealing with iterative solutions of a set of linear or non-linear equations in certain analyses • Provides graphical tools needed for data visualisation and analyses • Allows macros to be recorded for repetitive calculations
Solver	• Offers three solution options that suit different types of analyses • Allows constrained iterative solutions involving multiple parameters • Allows optimisation analyses with single and multiple design variables and can be used for multi-objective optimisation
Thermax	• Provides the physical properties of various fluids • Provides two interpolation functions for tabulated data and a Newton-Raphson solver for non-linear equations such as the Colebrook-White equation and the SRK equation
VBA	• Needed for developing additional fluid property functions or other functions not provided by Excel or Thermax, e.g.: - numerical solvers for large systems of linear equations - custom function for standard pipe dimensions - custom functions for global-warming potential (GWP) of the various refrigerants

The four chapters 5 to 8 deal with fundamental analyses in heat-transfer and fluid-dynamics that require the use of computer-aided methods. Both Chapter 5 and Chapter 6 deal with the numerical solution of the heat-conduction equation by using the finite-difference (FD) method. While Chapter 5 focuses on the steady conduction equation, Chapter 6 focuses on the transient equation. Similarly, both Chapter 7 and Chapter 8 deal with hydraulic analyses of fluid systems. Chapter 7 focuses on hydraulic analyses of multi-pipe and pump-pipe systems, while Chapter 8 describes a method for using Solver for hydraulic analyses of gravity-driven pipe-networks with looped, branched, and mixed configurations. The last four chapters of the book illustrate the use of the Excel-based platform for design analyses and optimisation of various type of fluid-thermal systems.

References

[1] C. T. Crowe, D. F. Elger, B. C. Wiliams, and J. A. Roberson, *Engineering Fluid Mechanics*, 9th edition, John Wiley & Sons, Inc., 2009.

[2] Y.A. Cengel and A.J. Ghajar, *Heat and Mass Transfer: Fundamentals and Applications*. 4th edition, McGraw Hill, 2011.
[3] W.W. Pulkrabek, *Engineering Fundamentals of the Internal Combustion Engine*, 2nd Edition, Pearson Education International, 2004.
[4] C.R. Ferguson, *Internal Combustion Engines*, John Wiley & Sons, 1986.
[5] Y.A. Cengel, and M.A. Boles, *Thermodynamics an Engineering Approach*, McGraw-Hill, 7th Edition, 2007
[6] A. Rivas, T. Gómez-Acebo, and J. C. Ramos. The application of spreadsheets to the analysis and optimization of systems and processes in the teaching of hydraulic and thermal engineering, *Computer Applications in Engineering Education*, Vol. 14, Issue 4, 2006, pp. 256-268.
[7] D. Brkic, Spreadsheet-Based Pipe Networks Analysis for Teaching and Learning Purpose, Spreadsheets in Education (eJSiE): Vol. 9: Iss. 2, Article 4, 2016. Available at: http://epublications.bond.edu.au/ejsie/vol9/iss2/4
[8] A. Bejan, G. Tsatsaronis, M. Moran, *Thermal Design & Optimisation*, John Wiley & Sons, 1996.
[9] M. Niazkar, S.H. Afzali, Application of Excel Spreadsheet in Engineering Education, First International & National Conference on Engineering Education, Shiraz University, 10-12 November 2015
[10] Z. Ahmadi-Brooghani, Using Spreadsheets as a Computational Tool in Teaching Mechanical Engineering, Proceedings of the 10th WSEAS International Conference on computers, Vouliagmeni, Athens, Greece, July 1315, 2006, 305-310
[11] S.A. Oke, Spreadsheet Applications in Engineering Education: A Review, Int. J. Engng Ed. Vol. 20, 2004, No. 6, 893-901
[12] A. Karimi, Using Excel for the thermodynamic analyses of air-standard cycles and combustion processes, ASME 2009 Lake Buena Vista, Florida, USA.
[13] The University of Alabama, Mechanical Engineering, Excel for Mechanical Engineering project, Internet: http://www.me.ua.edu/excelinme/index.htm (Last accessed July 11, 2019).
[14] J. Huguet, K. Woodbury, R. Taylor, Development of Excel add-in modules for use in thermodynamics curriculum: steam and ideal gas properties, American Society for Engineering Education, 2008, AC 2008-1751.
[15] K. Mahan, J. Huguet, K. Woodbury, R. Taylor, Excel in ME: Extending and refining ubiquitous software tools, American Society for Engineering Education, 2009, AC 2009-2295.
[16] M.M. El-Awad, M.S. Al Nabhani, K.S. Al Hinai, A. Younis, Development and Validation of an Excel Add-In for Determining the Properties of Various Refrigerants, Proceedings of First National Conference on Recent Trends in Applied Science, Engineering and Technology (CASET 2K19), Ipri College of Technology, June 11, 2019.
[17] M.M. El-Awad, A Multi-Subject Excel Add-In for Fluid Properties and its Use for analysing Cascade and Multi-Stage Compression Refrigeration Cycles, The Electronic Journal of Spreadsheets in Education (eJSiE) Vol. 12, Issue 1, April 18, 2019.

[18] M.M. El-Awad, Energy and exergy analyses of the evaporative-regenerative gas-turbine cycle using Excel-Thermax, International Journal of Innovative Research in Science, Engineering and Technology (IJIRSET), Volume 4, Issue 10, October 2015, 9855-9862.

[19] M.M. El-Awad, A multi-substance add-in for the analyses of thermo-fluid systems using Microsoft Excel, International Journal of Engineering and Applied Sciences (IJEAS), Volume-2, Issue-3, March 2015, 63-69.

2

Excel

With its graphical tools, iterative tools, and the "Developer" options, Excel forms the backbone of the modelling platform for the various analyses considered in this book. Excel allows the manipulation of stored data by providing a large set of built-in functions and its user-interface provides many tools for general data analyses, but this chapter focuses on those needed for building analytical models relevant to the book's methodology. In this respect, the chapter highlights the use of "cell-labelling" for writing Excel's formulae instead of the usual referencing by location and illustrates the use of Excel's matrix functions for the solution of linear systems of equations. The chapter also illustrates the use of the "Goal Seek" command and the solution of nonlinear equations by "circular calculations" and its final section on Excel's graphical tools illustrates the use of the "trendline" feature for curve-fitting of tabulated data. More detailed information about Excel can be found in more specialised books, e.g., Walkenbach [1,2].

2.1. Elements of Excel's user-interface

Excel's user-interface allows the user to adjust the appearance of the workspace and present his/her primary data and analysis results in various forms. To allow easy access to the large number of functions, tools and commands provided by the user-interface, it is divided into a number of elements with different purposes. For example, Figure 2.1 shows a screenshot of an Excel sheet that stores the scores obtained by a group of students in one semester.

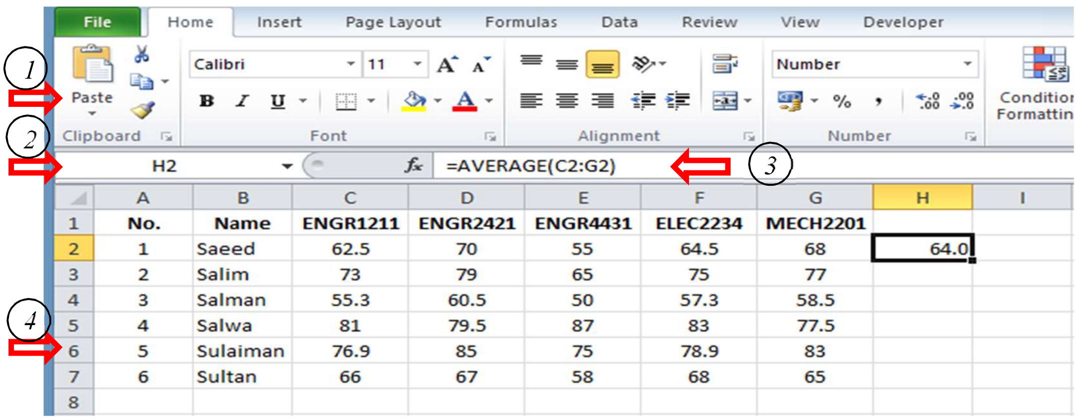


Figure 2.1. The main elements of Excel's user-interface

Figure 2.1 shows four elements of Excel's user-interface, which are:

1. The ribbon
2. The name box
3. The formula bar
4. The workspace

The **Ribbon**, which occupies the top part of the sheet, organises the numerous commands provided by Excel into nine "tabs", e.g., the **File**, **Home**, and **Insert** tabs. Each tab consists of a number of command-groups that have a common purpose. For example, the **Developer** tab shown in Figure 2.2 allows the user to write customised functions using the Visual Basic for Applications (VBA) language and to record **Macros** together with other useful development tools.

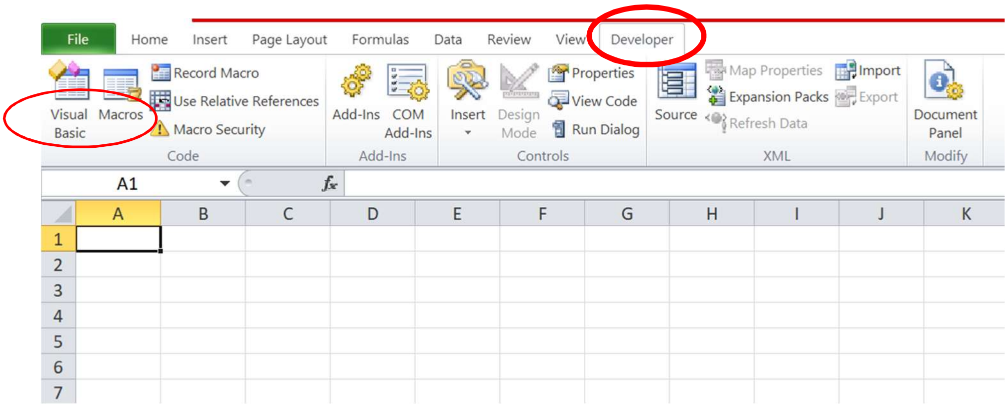


Figure 2.2. Elements of the Developer tab

The **Workspace**, which is the main part of the sheet, is divided into a grid of columns and rows that form separate "**cells**" at their intersections. A cell is referred to by a letter and a number, e.g., A1, B3, H2, etc. The letter represents the cell's column, while the number represents its row. The **Name box** shows the name of the cell where the pointer is positioned which is H2 in Figure 2.1. As Figure 2.1 shows, a cell can simply contain a character data, such as "Saeed" and "Salim", or a numerical data, such as 62.5 and 70. A cell can also contain a formula for data manipulation using the numerous built-in functions provided by Excel. The formula bar in Figure 2.1 reveals the formula typed in cell H2 that uses the built-in function "**AVERAGE**" to determine the average score of the first student in the list (Saeed) in the five subjects as 64.0. Note that, unlike the number or character cells, a cell that includes a formula must start with the equal sign "=". The writing of Excel's formulae will be explained in more details in the following section.

2.2. Excel's formulae

In general, Excel's formulae include mathematical or logical operators, built-in functions, and cell references. Excel provides two ways to refer to a particular cell in a formula; either by its location in the sheet, e.g., A2, C10, etc., or by giving it a relevant name, e.g., efficiency, diameter, etc. The two methods will be illustrated below.

2.2.1. Cell reference by location

To illustrate this method, let us write a formula to calculate the area of a circle that has a radius of 5 m. Open a new Excel sheet and type the number 5, which is the radius of the circle, in cell A1 as shown in Figure 2.3. Now, go to cell A2 and type the following formula:

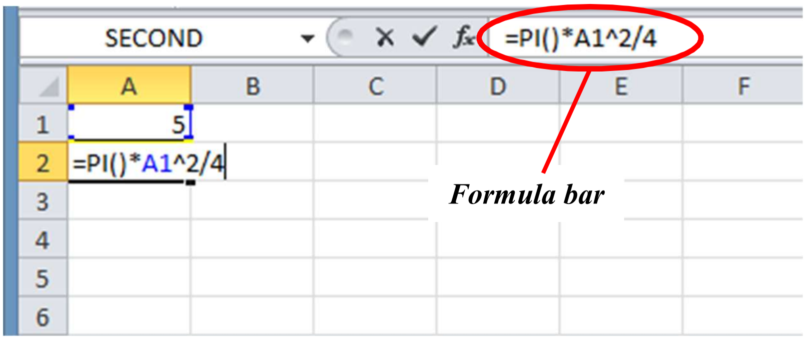


Figure 2.3. Writing an Excel formula to determine the area of a circle

=PI()*A1^2/4

The formula starts with the equal sign as mentioned above. The function "**PI()**" is a built-in function that returns the value of Archimedes' constant π. The formula also contains a reference to cell A1 that stores the value of the circle's radius, the multiplication operator *, the power operator ^, the division operator /, and the constants 2 and 4. Note that the formula is shown in the formula bar which can also be used to edit the formula. Pressing the **Enter** key after typing the formula, the result will be as shown in Figure 2.4; which is 19.63495 square metres. The following example shows how Excel's formulae and built-in functions can be used in a simple thermodynamic analysis.

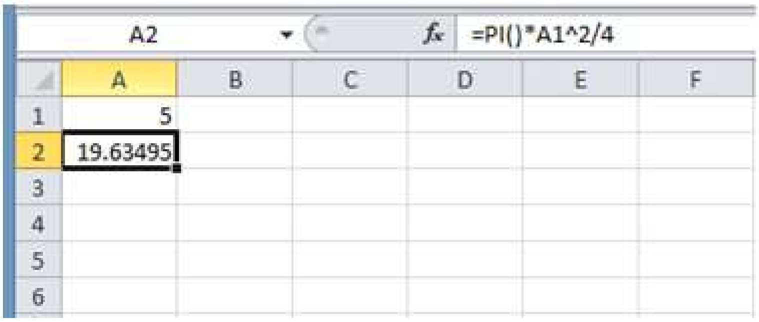


Figure 2.4. The Excel sheet with a formula that determines the area of a circle

Example 2-1. Determining the error in the specific volume of R134a calculated by using the ideal-gas law

It is required to develop an Excel sheet that calculates the error in the specific volume (v) of superheated refrigerant R134a that results from applying the ideal-gas law at a pressure of 200 kPa (T_{sat} = -10.09ºC) and temperatures in the range 0ºC to 100ºC (273 to 373 K).

Solution

Figure 2.5 shows the Excel sheet prepared for this example. The pressure (P), the gas constant (R), and the temperature (T) are stored in columns A, B, and C, respectively. Since the pressure (P) and the gas constant (R) do not change in this example, Excel allows for these parameters to be stored in single cells instead of being repeated as shown in Figure 2.5 (see Section 2.4.1).

E2 f_x =B2*C2/A2

	A	B	C	D	E	F	G
1	P	R	T	v_table	v_ideal		
2	200	0.08149	273	0.10438	0.1112339		
3	200	0.08149	283				
4	200	0.08149	293				
5	200	0.08149	303				
6	200	0.08149	313				
7	200	0.08149	323				
8	200	0.08149	333				
9	200	0.08149	343				
10	200	0.08149	353				
11	200	0.08149	363				
12	200	0.08149	373				
13							

Figure 2.5. The sheet developed for Example 2-1

Column D stores the values of v obtained from relevant property tables and column E stores the corresponding values obtained by using the ideal-gas law, i.e.:

$$v = RT / P \tag{2.1}$$

Where, P and T are the absolute pressure and temperature, respectively, and R is the gas constant (for R134a, $R = 0.08149$ kJ/kg.K). Note that the formula bar in Figure 2.5 reveals the formula in cell E2 that applies Equation (2.1).

Figure 2.5 shows the tabulated value of the specific volume (v_table) and that determined by Equation (2.1) (v_ideal) at 273K. The percentage error of the ideal-gas law in estimating the specific volume is given by:

$$Error = \frac{v_{Ideal} - v_{Table}}{v_{Table}} \times 100 \tag{2.2}$$

To determine the percentage error at 273K, go to cell F2 as shown in Figure 2.6 and type the following formula, which is equivalent to Equation (2.2):

=(E2 – D2)/D2*100

Note that the formula bar in Figure 2.6 reveals the above formula for 273K. When you press the **Enter** key, the number **6.566** will appear in cell F2 as shown in the figure.

F2 f_x =(E2-D2)/D2*100

	A	B	C	D	E	F	G
1	P	R	T	v_table	v_ideal	error_v_ideal	
2	200	0.08149	273	0.10438	0.1112339	6.5662483	
3	200	0.08149	283	0.10922	0.1153084	5.5743911	
4	200	0.08149	293	0.11394	0.1193829	4.776944	
5	200	0.08149	303	0.11856	0.1234574	4.1306933	
6	200	0.08149	313	0.12311	0.1275319	3.5917878	
7	200	0.08149	323	0.12758	0.1316064	3.1559414	
8	200	0.08149	333	0.13201	0.1356809	2.7807363	
9	200	0.08149	343	0.13639	0.1397554	2.4674463	
10	200	0.08149	353	0.14073	0.1438299	2.2026931	
11	200	0.08149	363	0.14504	0.1479044	1.974869	
12	200	0.08149	373	0.14932	0.1519789	1.7806389	
13							

Figure 2.6.The completed sheet developed for Example 2-1

To find the percentage errors at other temperatures you can simply copy the formula in cell F2 and paste it on cells F3 to F12. Figure 2.6 shows the completed Excel sheet for the required temperature range. The calculated values of the errors show that the maximum error occurs at the lowest temperature, which is 273K. Note that the error decreases gradually as the temperature increases (refer to Figure 1.10).

2.2.2. Use of cell labels

The usual reference to cells by their columns and rows suits perfectly statistical analyses in which the same formula is applied to a large body of data that is stored column-wise or row-wise. For example, we may want to determine the average value, maximum value, or minimum value of the tabulated data. However, thermofluid analyses usually involve the application of numerous formulae to a small set of data, e.g. the diameter or length of a pipe, the fluid-flow rate, the density or viscosity of the fluid, etc. For such analyses, it is more convenient to give the cell a meaningful name or "label" that matches its content. The cell can then be referred to by its label instead of its relative location. This method makes it easier to recognise the quantities involved in the Excel formulae.

For the purpose of illustration, let us develop an Excel sheet to compare the density of air before and after an isentropic compression process from an initial condition of P_1 = 100 kPa and T_1 =300K to a final pressure of P_2 = 800 kPa. The two air densities involved can be calculated from the ideal-gas law as follows:

$$\rho_1 = P_1 / RT_1 \qquad (2.3)$$

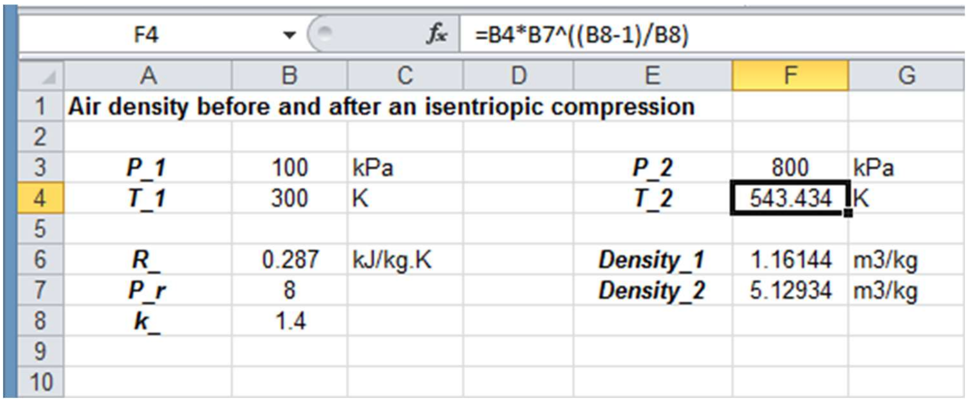
$$\rho_2 = P_2 / RT_2 \qquad (2.4)$$

Where R is the gas constant (for air, $R = 0.287$ kJ/kg.K). For an isentropic process, T_2 is related to T_1 according to the following approximate relationship:

$$T_2 = T_1 \times \left(P_2 / P_1\right)^{\frac{k-1}{k}} \tag{2.5}$$

Where k is the ratio of specific heat at constant pressure (c_p) and at constant volume (c_v). At the given temperature range, k for air can be taken as 1.4. Note that the symbol k is also used for the thermal conductivity in Appendix A, Table A.1.

Figure 2.7 shows the sheet prepared for this analysis in which the respective cell labels are typed in the column to the left of the different pressures and temperatures, while the corresponding units are written in the column to the right of each quantity. This is also done to the other quantities in the calculations. The sheet also shows the units of the different properties involved for more clarification.



F4 | fx =B4*B7^((B8-1)/B8)

	A	B	C	D	E	F	G
1	Air density before and after an isentriopic compression						
2							
3	P_1	100	kPa		P_2	800	kPa
4	T_1	300	K		T_2	543.434	K
5							
6	R_	0.287	kJ/kg.K		Density_1	1.16144	m3/kg
7	P_r	8			Density_2	5.12934	m3/kg
8	k_	1.4					
9							
10							

Figure 2.7. Excel sheet for calculating the air densities before and after compression

Placing the cursor on cell F4 makes the formula bar reveal the formula used for the calculation of the temperature T_2 according to Equation (2.5) which is:

=B4*B7^((B8-1)/B8)

The problem with the above formula is that it is not self-explanatory and one has to go to each cell in the formula in order to figure out what the formula represents; which is very difficult when there are many such formula in the model. The formula can be made easily understandable by using familiar labels to refer to the different cells involved. To do that, select the cells in columns A and B as shown in Figure 2.8, then go to **Formulas** and, at the **Name Manager**, select **Create** from **Selection**. When the form shown in Figure 2.8 appears to you, tick the "Left column" option. Pressing the "**OK**" button will make Excel create names for the different values in the selection box according to the labels written on the left column. The cell F3 that stores the value of P_2 can also be associated with its corresponding label in cell E3.

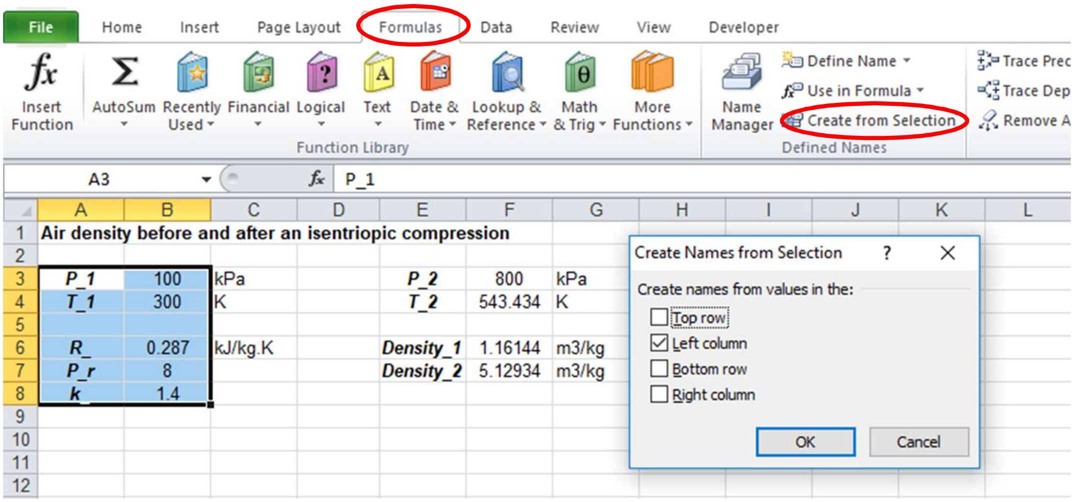


Figure 2.8. Creating names for a selected group of cells

Now, retype the formula in cell F4 that determines T_2 as follows:

=T_1*P_r^((k_-1)/k_)

The formula bar in the sheet shown in Figure 2.9 reveals the formula with the corresponding labels instead of the location references. The procedure can also be applied to the formulae that determine the two densities.

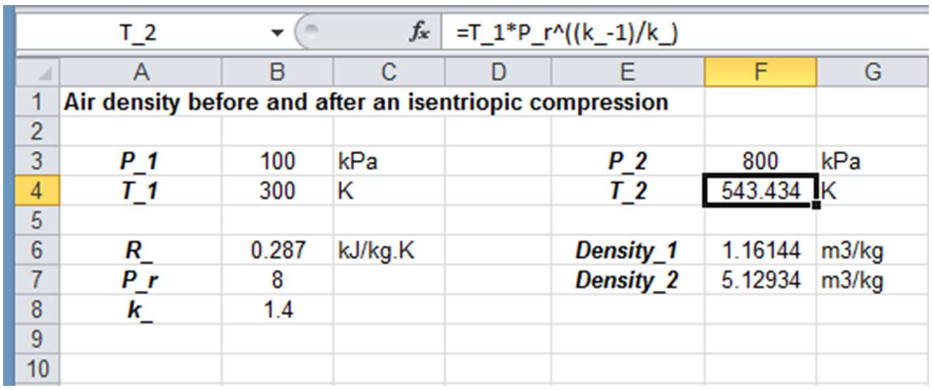


Figure 2.9. Formulae using cells labels instead of locations

Labelled formulae are easier to edit than those using location referencing particularly when intricate formulas are involved. Another advantage of cell-labelling is that if you copy a labelled formula and paste it in any other cell you will get the same formula and the same answer, which is not the case if you copy a formula that uses the usual referencing by location in another cell. When naming your cells, choose suitable representative names for the variables involved, e.g. P_1 and T_1 for P_1 and T_1. Note that Excel does not accept "P1" or "T1" as labels since these can be confused with usual cell references by locations. To avoid this, Excel automatically changes these labels to "P1_" and "T1_". To reveal all the formulae in the sheet, press the control key (ctrl) with the tilde key (~). More information about Excel's formulae can be found in Walkenbach [1].

2.3. Excel's built-in functions

Excel provides a large library of built-in functions for data manipulation, like the **AVERAGE** function, and other mathematical functions commonly used in engineering analyses like the **PI**, **SIN**, and **COS** functions. To view the full range of Excel's functions, type "=" in any Excel cell as shown in Figure 2.10 and then place the cursor on the **Insert Function** "*fx*" button in the formula bar and click it. The dialog box shown in Figure 2.11 will appear to you. You can list all the categories via the "**Select a category**" slot.

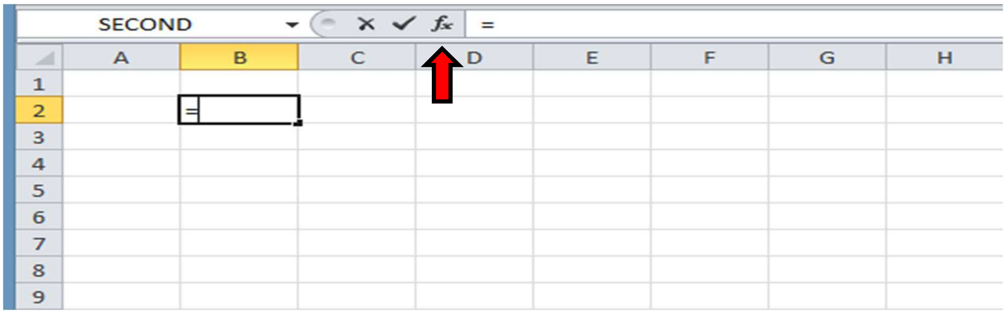


Figure 2.10. Exploring Excel's built-in functions

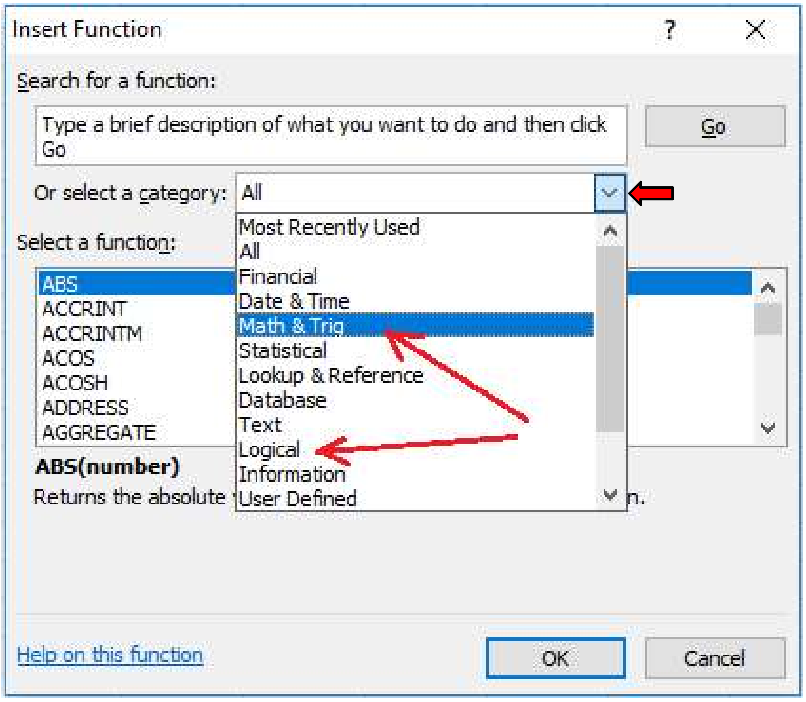


Figure 2.11. The various categories of Excel's built-in functions

The **Math & Trig** group includes the mathematical and trigonometric functions used in different types of engineering analyses. Figure 2.12 shows some of the numerous functions in this group. Note that a brief explanation of the function you select automatically appears below the selection window. For example, the explanation given to the **ABS** function is that it returns the absolute value of a number. The functions **ACOS**, **ASIN**, and **ATAN** apply the familiar inverse trigonometric functions: $\cos^{-1}$, $\sin^{-1}$, and $\tan^{-1}$, respectively. By scrolling down the list, you can find many other familiar functions. The following discussion focuses on two types of functions that are needed for the development of analytical models in subsequent chapters of the book, which are (a) the logical functions and (b) the functions for matrix operations.

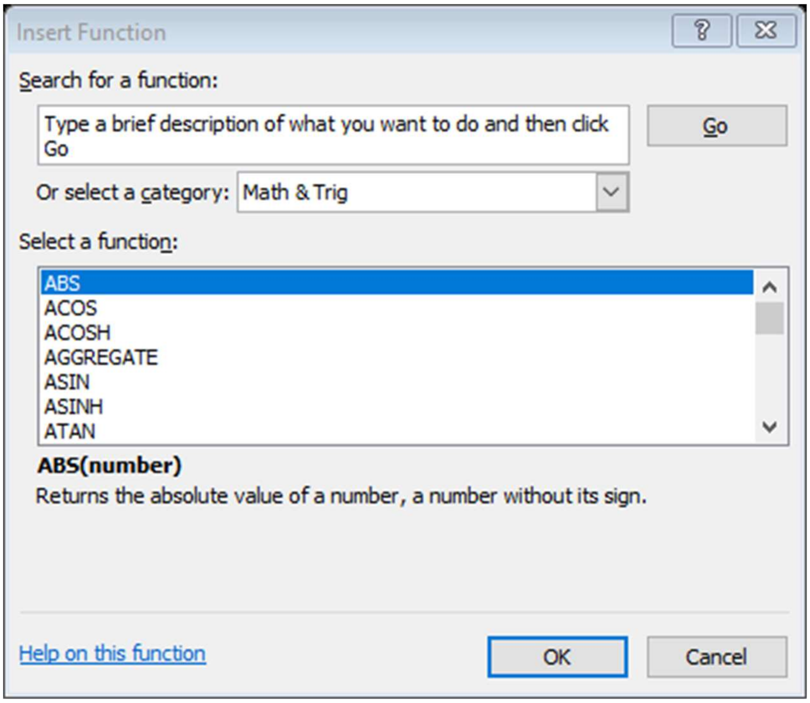


Figure 2.12. Common mathematical functions supported by Excel

2.3.1. Logical functions

To determine the major friction loss (h_f) in a pipe by using the Darcy-Weisbach equation we have to establish whether the flow is laminar or turbulent so as to select the relevant formula for the friction factor (f). The flow remains laminar before the Reynolds number (Re) reaches a certain value, which can be taken as 2,000, but the flow can only be considered fully turbulent beyond Re = 3000. There is a transitional region between laminar and turbulent flows when 2000 < Re < 3,000. Suppose that we want to write an Excel formula that determines the type of flow from the given value of the Reynolds number. Using a simple **IF** function, we can write the following formula:

=IF(Re<=2000, “Laminar”, “Turbulent or transitional”)

Note that the above **IF**-formula does not differentiate between turbulent flow and transitional flows. However, we can do that by using a second logical test inside the first logical test according to the following nested IF function:

=IF(Re<=2000, "Laminar", IF(Re>=3000, "Turbulent", "Transitional"))

Figure 2.13 shows an Excel sheet containing the above formula (shown in the formula bar) and the response of the formula when the value of Re stored in cell C2 is 500, which is "Laminar". Depending on the value of Re, the result of the If-formula can be "laminar", "Turbulent", or "Mixed". Excel supports six other logical functions; **AND**, **FALSE**, **IFERROR**, **NOT**, **OR** and **TRUE**. These functions can be combined in the same formula so as to handle more intricate formulae.

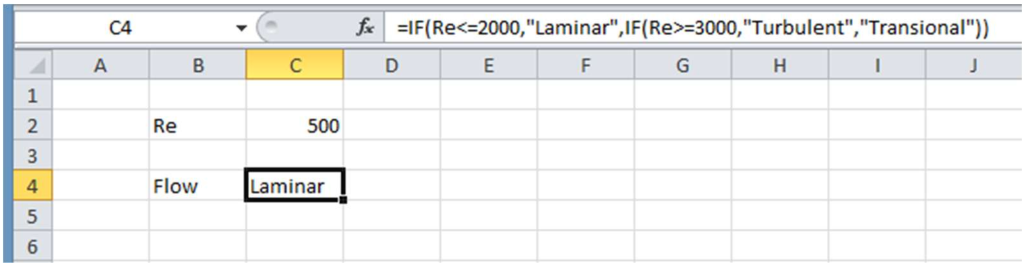


Figure 2.13. A formula using the nested IF function to determine the type of flow

2.3.2. Functions for matrix operations

A group of adjacent cells can be treated by Excel as a matrix or a vector and one of its function groups allow for the addition, subtraction, and multiplication of these matrices and vectors according to the established rules of matrix operations. Figure 2.14 shows a 3x3 matrix (A) stored in the cells B3:D5 and a vector (b) stored in cells F3:F5.

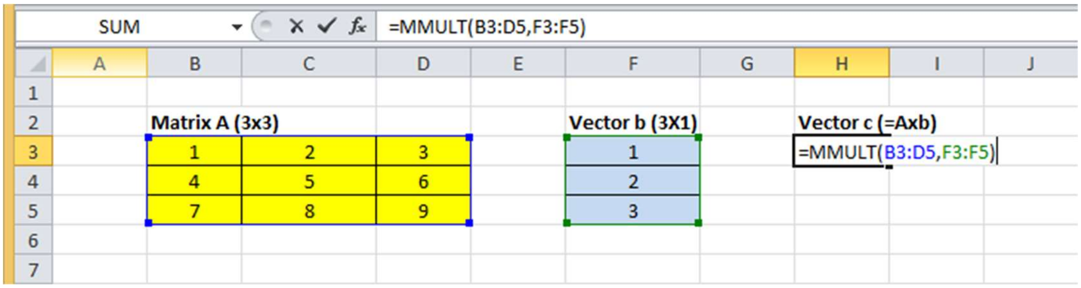


Figure 2.14. Step 1 of using the matrix multiplication function

Matrix (A) and vector (b) can be multiplied and the result stored in a third vector (c) by using the matrix function **MMULT**. The procedure is as follows:

1. After keying in the data of matrix (A) and vector (b) as shown on Figure 2.14, position the cursor at cell H3 and type the formula: =MMULT(B3:D5;F3:F5).

2. Now press ENTER key and cell H3 will take the value 14, which the result of multiplying the first row of the matrix with the vector (b) as Figure 2.15 shows.

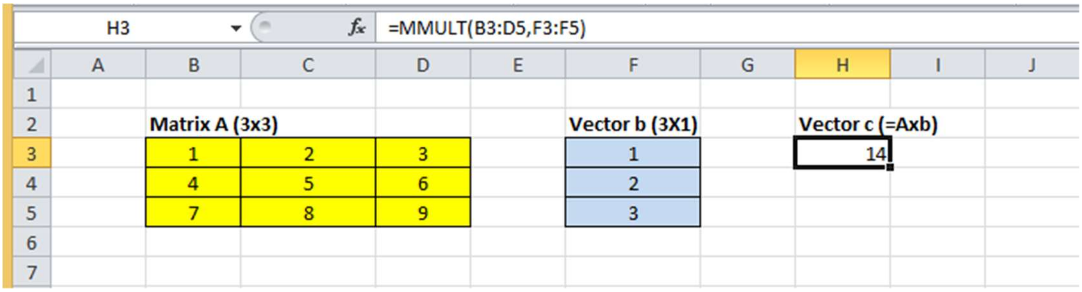


Figure 2.15. Step 2 of using the matrix multiplication function

The other two elements of the result vector will not appear automatically. To view the complete solution vector, do what follows:

1. Select the cells H3:H5 as shown on Figure 2.16,
2. Press the function key F2 once and then simultaneously hold the (**SHIFT** + **CONTROL**) keys together and press ENTER. The complete solution vector (c) will now appear as shown on Figure 2.17.

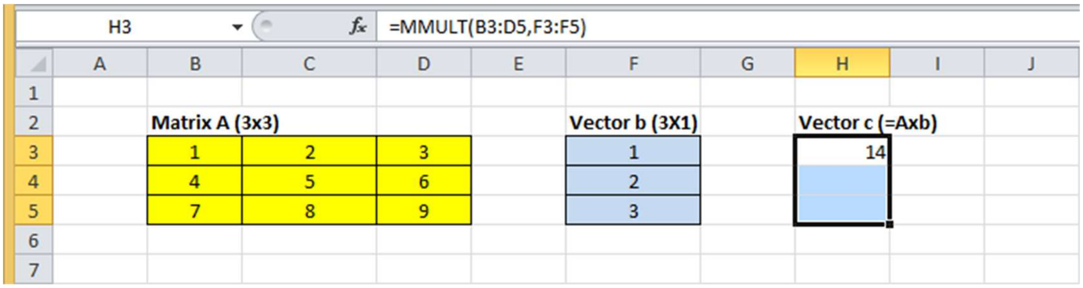


Figure 2.16. Step 3 of using the matrix multiplication function

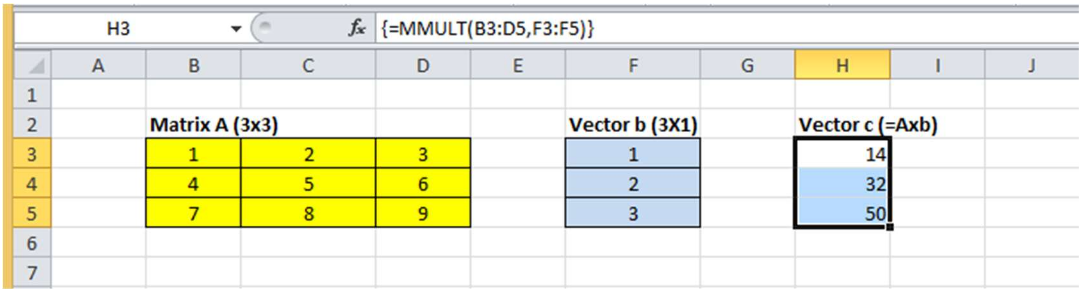


Figure 2.17. Step 4 of using the matrix multiplication function

Another matrix-operation function provided by Excel is the matrix-inversion function **MINVERSE** which is useful for the solution of linear systems of equations. The following example illustrates the use of this function.

Example 2-2. Using the matrix inversion function
By using the **MINVERSE** function, find the inverse of matrix [A] given by:

$$[A] = \begin{bmatrix} 1 & 0 & 3 \\ 0 & 5 & 6 \\ 7 & 0 & 5 \end{bmatrix}$$

Solution
The first step of the solution is to enter the elements of the matrix as shown on Figure 2.18. After entering the data, go to cell F2 and type the formula "=MINVERSE(B2:D4)". When you press **ENTER**, this cell will have the value -0.3125, which is the first element of the inverse matrix $[A]^{-1}$ shown on Figure 2.19.

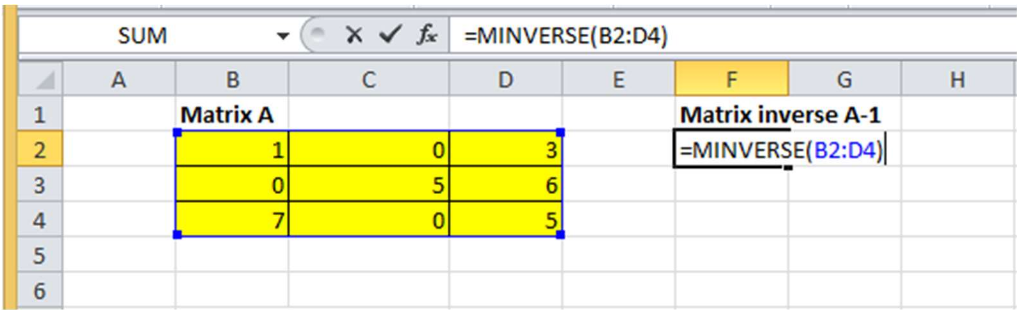


Figure 2.18. Step 1 of using the MINVERSE function

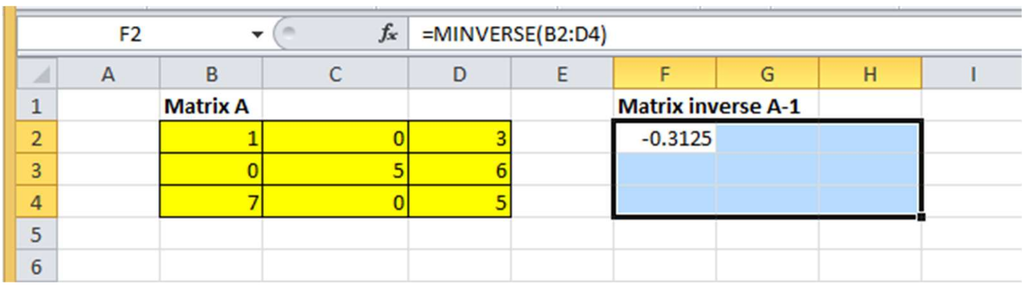


Figure 2.19. Step 2 of using the MINVERSE function

In this case, the result of the matrix operation is another matrix. Starting with the formula in cell F2, select the range F2 to H4 as shown on Figure 2.19. Press and release the function key F2 and then simultaneously hold the **CTRL+SHIFT** keys and press **ENTER**. The other elements of the inverse matrix $[A]^{-1}$ will then appear as shown on Figure 2.20. You can check the solution by multiplying matrix [A] with its inverse by using the **MMULT** functions as explained above. The steps of applying the procedure are shown on Figures 2.21 to 2.23. As should be expected, Figure 2.23 shows that the resultant matrix is the identity matrix.

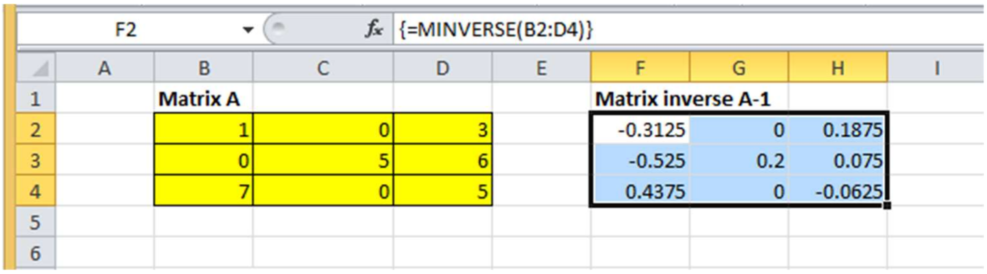


Figure 2.20. The complete inverse matrix $[A]^{-1}$

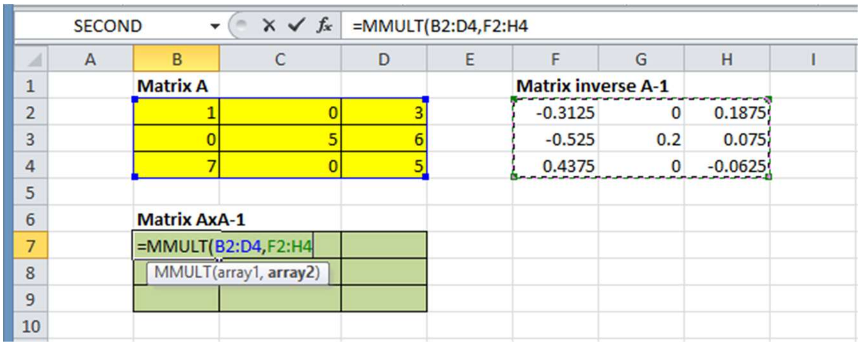


Figure 2.21. Multiplying matrix [A] by its inverse $[A]^{-1}$

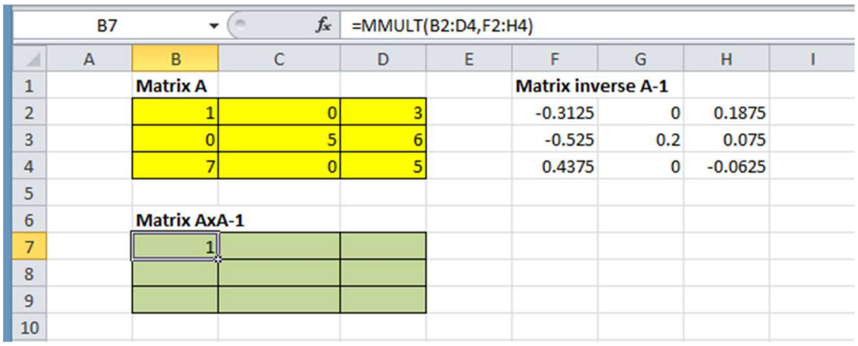


Figure 2.22. The first element of the product (identity) matrix

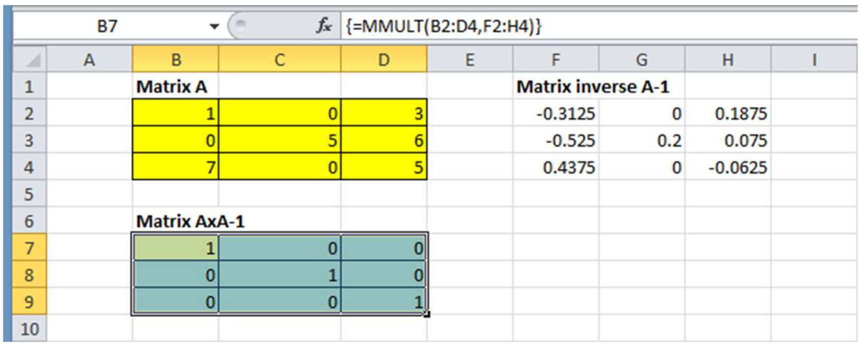


Figure 2.23. The complete solution which is the identity matrix

2.3.3. Solution of linear system of equations

The problems that involve systems of linear equations are very common in engineering analyses. In thermofluid analyses an example of such problems is the solution of the heat conduction equation by the finite-difference method. One of the methods offered by Excel for solving linear systems of equations is by applying the matrix-inversion method. For illustration, consider the following linear system written in matrix notation:

$$[A]\{x\}=\{y\} \tag{2.6}$$

Where [A] is the coefficient matrix, $\{x\}$ the vector of unknowns, and $\{y\}$ the right-side or "load" vector. By applying the matrix-inversion method, the solution vector $\{x\}$ can be obtained as follows:

$$\{x\}=[A]^{-1}\{y\} \tag{2.7}$$

Where $[A]^{-1}$ is the inverse of matrix [A]. The method is applied in two steps: (1) inversion of the coefficient matrix and (2) multiplication of the inversed matrix with the load vector. The resulting vector is the solution. The following example illustrates the procedure of applying the method by using Excel's matrix functions.

Example 2-3. Solution of a system of linear equations

Find the values of x_i in the following system of linear equations:

$$\begin{bmatrix} 14 & 14 & -9 & 3 & -5 \\ 14 & 52 & -15 & 2 & -32 \\ -9 & -15 & 36 & -5 & 16 \\ 3 & 2 & -5 & 47 & 49 \\ -5 & -32 & 16 & 49 & 79 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \\ x_4 \\ x_5 \end{Bmatrix} = \begin{Bmatrix} -15 \\ -100 \\ 106 \\ 329 \\ 463 \end{Bmatrix} \tag{2.8}$$

Solution

Note that the system is symmetric; which is typically the case with the linear systems that arise in the solution of heat-conduction problems by the finite-difference method. For larger systems of equations, the symmetry of the system can be utilised for reducing the required computer memory by storing only half of the coefficient matrix. However, this requires a complicated computer programming. For small systems like the one considered here, it is more convenient to use Excel's matrix inversion and multiplication functions. Figure 2.24 shows the Excel sheet that stores both the coefficient matrix [A] and the load vector $\{y\}$. The inverse matrix $[A]^{-1}$, which is obtained by following the procedure described in the previous section, is stored below the coefficient matrix as shown in the figure. The inverse matrix $[A]^{-1}$ is then multiplied with the load vector {y} and the result stored below the load vector as shown on Figure 2.25. The complete solution is shown on Figure 2.26. The first element is practically zero and, therefore, the solution vector is{x} = (0, 1, 2, 3, 4).

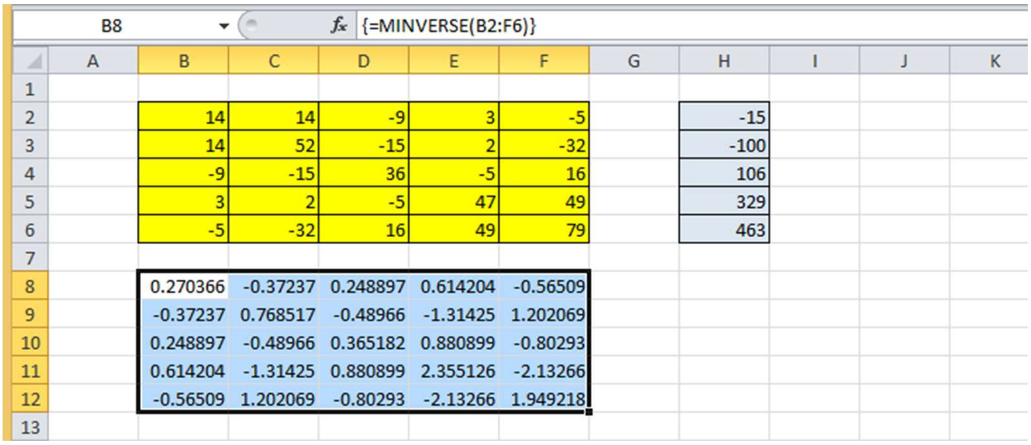


Figure 2.24. The coefficient matrix [A], the load vector {y} and the inverse matrix $[A]^{-1}$

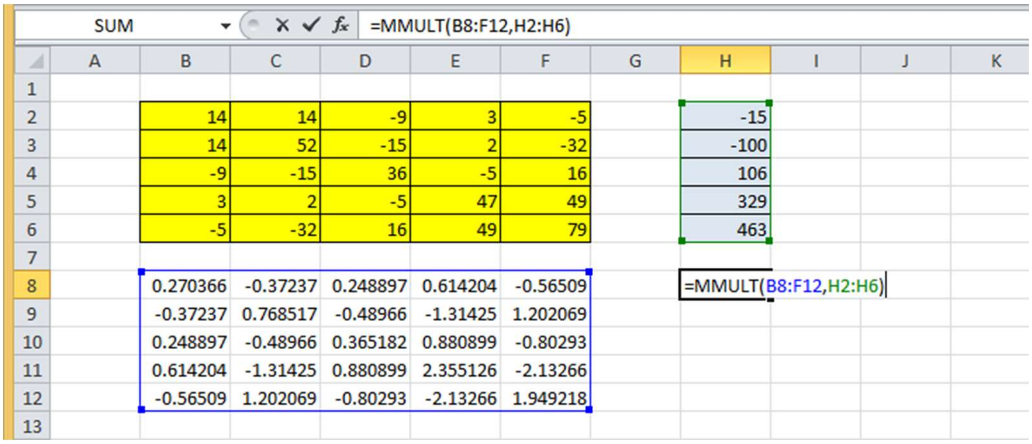


Figure 2.25. The first step of multiplying the inverse matrix $[A]^{-1}$ with the load vector {y}

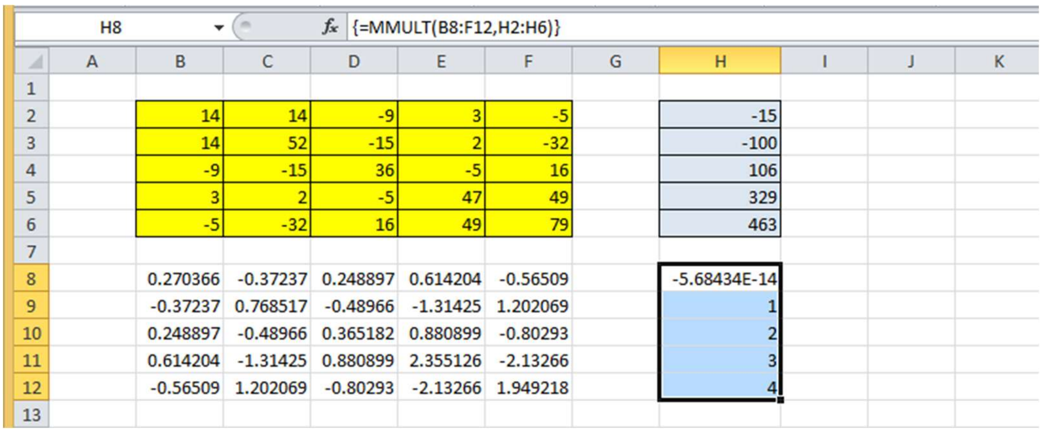


Figure 2.26. The complete solution vector {x}

It should be mentioned that the linear systems generated in multi-dimensional heat-transfer and fluid-dynamics analyses are usually too large to be solved efficiently by using the matrix-inversion method described above. As mentioned in Chapter 1, page 11, such analyses require dedicated software for other reasons any way. Small systems of linear equations can also be solved by using Solver as described in Chapter 3.

2.4. Itearative solutions with Excel

Iterative solutions are another type of analyses commonly encountered in engineering analyses. Two methods can be used to perform iterative solutions with Excel: (i) by using the **Goal Seek** command and (ii) by using **circular calculations**. In what follows the two methods are illustrated with the help of simple examples.

2.4.1. Itearative solutions with Goal Seek

The **Goal Seek** command is used for finding the value of an independent variable (x) that yields a specified value of a dependent variable (y). It is a simple, yet very useful tool for "What-if" analyses. The following example illustrates how this command can be used to solve a nonlinear equation.

Example 2-4. Solution of a nonlinear equation by Goal Seek

A centrifugal pump is used for lifting water from the utility network at the ground level to a tank at the top of a building that is 30-m high as shown on Figure 2.27. The pump's characteristic curve can be represented by the following formula:

$$h_p = h_0 - aQ - bQ^2 - cQ^3 \qquad (2.9)$$

Where h_p and Q are the pump's head (m) and discharge (m^3/s), respectively, and h_0, a, b, and c are constants the values of which are 47.22, 2.985×10^3, 1.549×10^5, and 2.348×10^8, respectively. Neglecting friction losses in the pipe, determine the water flow rate (m^3/s) that can be delivered by the pump.

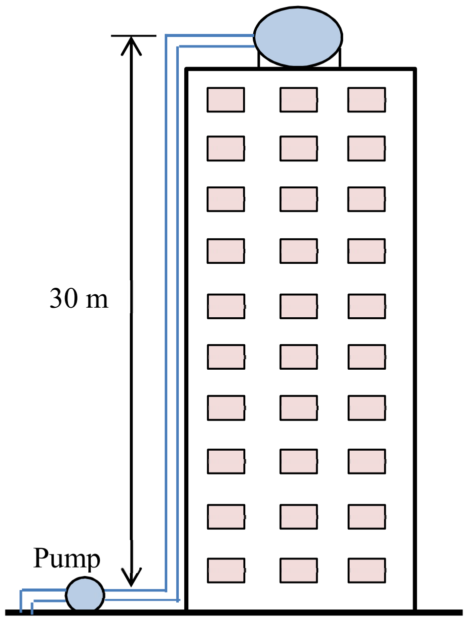


Figure 2.27. Schematic for Example 2-4

Solution

Figure 2.28 shows the Excel sheet prepared for this example in which the values of the four constants in Equation (2.9) are stored on the left side of the sheet. The pump's head is calculated at various values of the discharge and plotted as shown on the figure. We can see from the plot that the value of Q that yield h_p = 30 m is approximately 0.003 m^3/s.

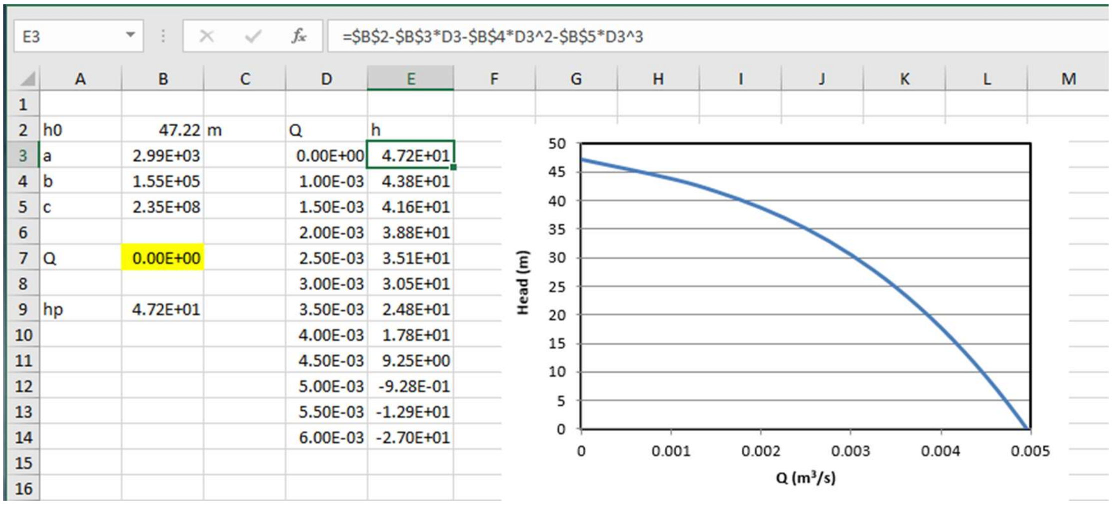


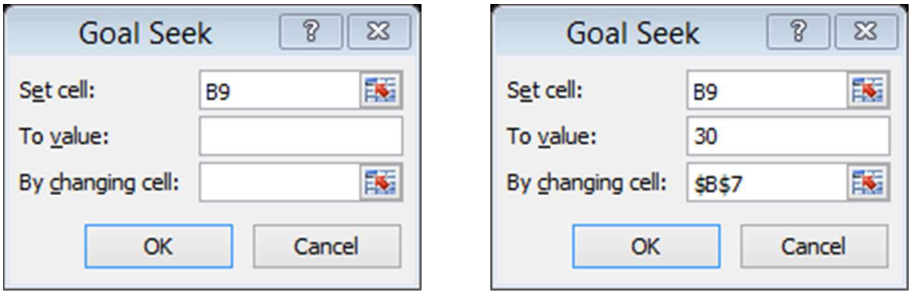
Figure 2.28. Excel sheet for Example 2-4

To solve the problem by using Goal Seek, enter an initial guess for Q in cell B7, say 0, and then enter the following formula that uses Equation (2.9) to calculate h_p in cell B9:

=B2-B3*B7-B4*B7^2-B5*B7^3

Note the dollar sign ($) that has been added to the references of the four constants, e.g. B2 has become B2. The formula bar in Figure 2.28 reveals a copy of the above formula in cell E3 that uses Equation 2.9 to calculate the head from the discharge. Copying and pasting this formula in the following cells keeps the references to the cells that store the four constants and only changes D3 to D4, D5, etc. To activate the Goal Seek command, go to the **Data** tab, select the **What-If-Analysis** option in the **Data Tools** group and then select **Goal Seek,** as shown on Figure 2.29. The Goal Seek dialog box shown on Figure 2.30.a will then appear to you and asks you to select the "**Set cell**", i.e. the cell that contains the dependent variable, which is B9 in this case. You also have to specify the value sought for this cell and the adjustable cell that stores the parameter to be changed. In this case, we seek the value in the cell B9 to be 30 by changing the value of the cell B7. The completed form is shown on Figure 2.30.b.

Figure 2.29. Activation of the Goal Seek command



(a) (b)

Figure 2.30. Goal Seek Set-up for Example 2-4: (a) before completion (b) the completed box

By pressing the "**OK**" button after completing the Goal Seek form, Excel will change the value in the adjustable cell (B7) until the Set cell (B9) acquires the required value. As shown on Figure 2.31, the answer obtained for Q is 0.003 m^3/s, which agrees with the estimated value from the plot on Figure 2.28.

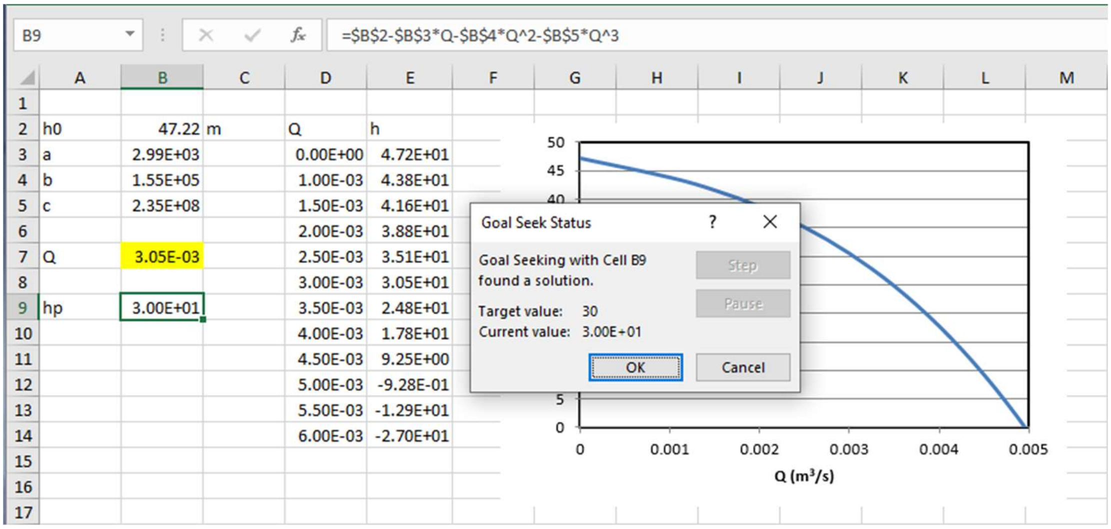


Figure 2.31. Goal Seek solution for Example 2-4

2.4.2. Iterative solution with circular calculations

A circular reference occurs when an Excel formula refers to its own cell in a direct or indirect manner. This occurs, for example, when solving a system of linear or non-linear equations because the various elements of the solution depend on each other. **Circular calculation** allows Excel to itearate until all the formulae involved are satisfied. The following example illustrates this special feature which is useful for thermofluid analyses.

Example 2-5. Determining the final temperature of heated air

Heat is added to a piston-cylinder device that contains one kg of air initially at 300K. If 100 kJ of heat is added to the air at constant pressure, determine the final temperature of air taking into consideration that its molar specific-heat ($\tilde{c}_p$) varies with temperature according to the following formula:

$$\tilde{c}_p = a + bT + cT^2 + dT^3 \quad \text{[kJ/kmol]} \tag{2.10}$$

Where a = 28.11, b =1.97x10^{-03}, c = 4.80x10^{-06}, and d = -1.97x10^{-09}.

Solution

From the defenition of specific heat, the final temperature (T_2) is given by:

$$T_2 = T_1 + Q/(\tilde{c}_p / M) \tag{2.11}$$

Where T_1 is the initial temperature, Q is the amount of heat added, and M is the molar mass (for air, M=29). If the variation of $\tilde{c}_p$ with temperature is ignored and its value at T_1 alone is used, Equation (2.11) determines T_2 as 399.73K. However, the result can be more accurate by using Equation (2.10) to determine $\tilde{c}_p$ at the average temperature, T_{avr} = $(T_1+T_2)/2$. Figure 2.32 shows the Excel sheet developed for this method which reveals the formulae inserted in cells F2, F4, and F6.

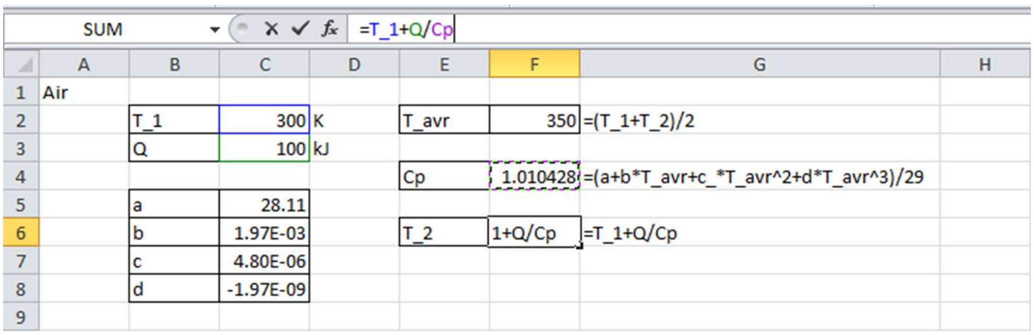


Figure 2.32. Excel sheet developed for Example 2-5

As soon as we type Equation (2.11) in cell F6, Excel will make the warning message that there is a circulare refernce as shown on Figure 2.33.

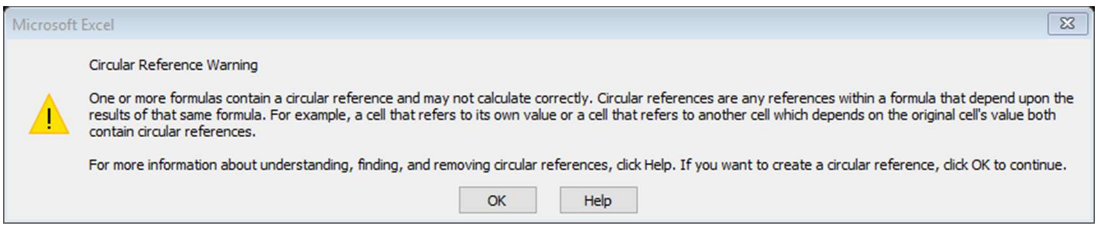


Figure 2.33. The circular-reference prompt

The circular reference occurs because T_2 depends on $\tilde{c}_p$ according to Equation (2.11) while $\tilde{c}_p$ itself depends on T_2 according to Equation (2.10). If we press the “OK” button shown on Figure 2.33, the cells involved in the circular reference whill be identified as shown on Figure 2.34. In this case, three cells are involved in the circular reference, which are F2, F4, and F6.

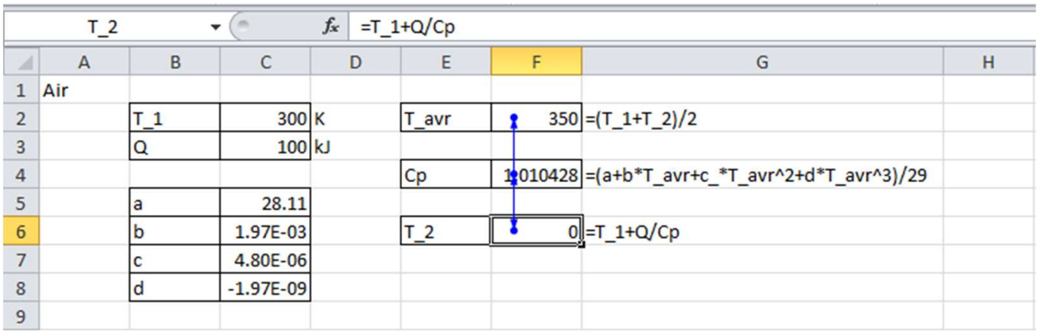


Figure 2.34. The cells involved in the circular reference

By allowing circular calculations, Excel will iterate to determine the values of both T_2 and $\tilde{c}_p$ that satisfy the relevant equations. However, the iterative-calculation option is not allowed by default. To allow it, go to **File** and select **Options**. The **Backstage View** form shown on Figure 2.35 will appear to you. Select **Formulas**, then the form will appear as shown on Figure 2.36. Enable iterative calculations by ticking (√) the box indicated in the figure and press the “OK” button. Excel can now iterate to find the values of T_2 and $\tilde{c}_p$ that simultaneously satisfy Equations (2.10) and (2.11). Figure 2.37 shows the solution found by this method, which is T_2 =398.976K. By using a constant value for $\tilde{c}_p$, the value obtained earlier was 399.73K. The difference between the two solutions will increase as more heat is added to the air.

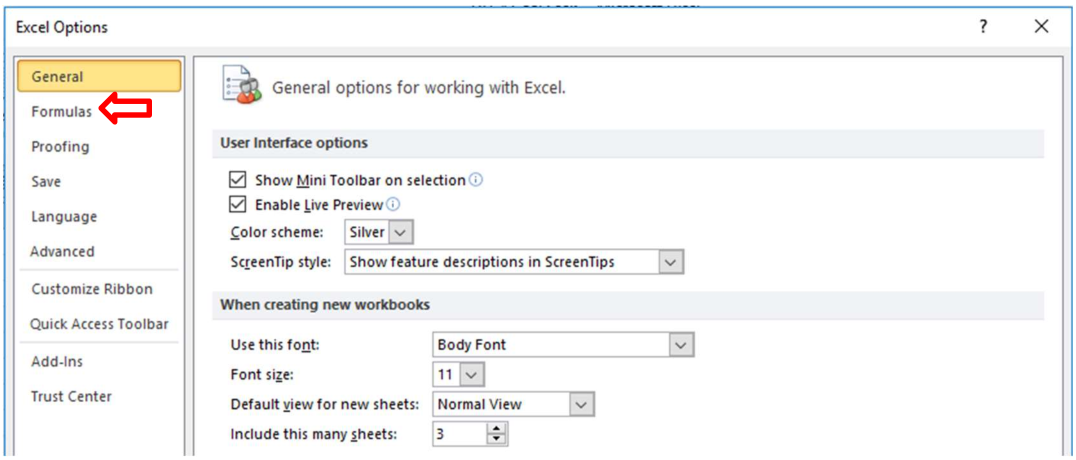


Figure 2.35. Selecting Excel's option (Formulas)

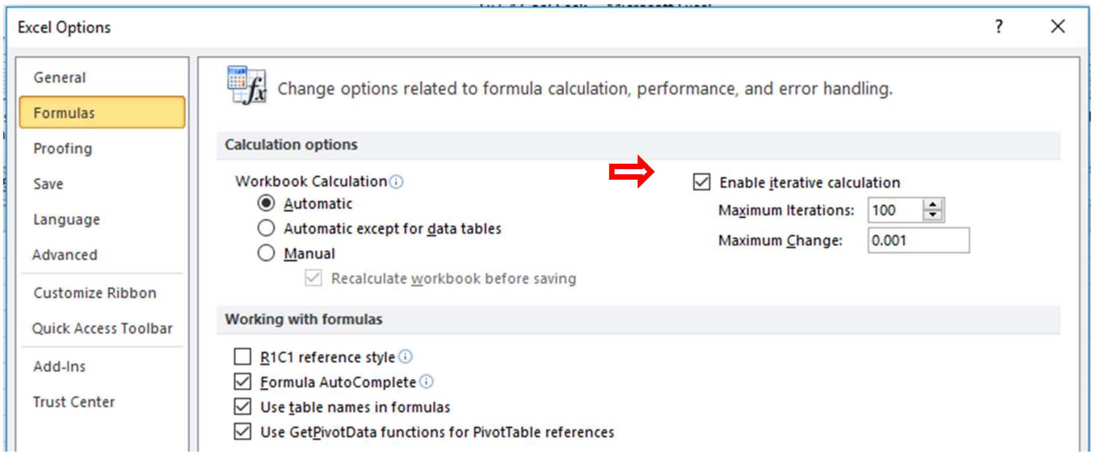


Figure 2.36. Enabling iterative calculations from Excel's Formulas option

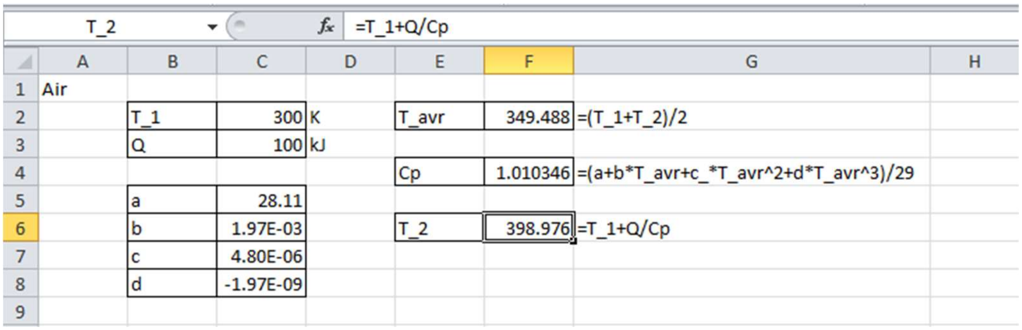


Figure 2.37. Solution of Example 2-5 by circular calculations

This example can also be solved by using the Goal Seek command. With this method, we start the iterative solution by providing Excel with a guessed value for T_2, call it T_{2o}, based on which a new value for T_2 is calculated, called it T_{2c}. Since the guessed value T_{2c} is unlikely to be correct, it will be different from T_{2o}. Goal Seek is then used to adjust the value of T_{2o} until the difference (Diff = T_{2c} - T_{2o}) vanishes.

Most iterative solutions in this book are obtained by using the Goal Seek command. However, the circular-calculation option is more useful than Goal Seek in certain situations as demonstrated in Chapters 5 and 6 that deal with the solution of the heat-conduction equation with the finite-difference method. The option can also be used to solve small systems of linear or non-linear equations. This is left as an exrcise (refer to Exercise 2.3).

2.5. Excel’s graphical tools for data presentation and analysis

Excel has numerous graphical tools that can be used to present the stored data in a variety of charts. Figure 2.38 shows one type of Excel charts that displays the annual variation of temperature and relative humidity at a hypothetical location on a certain day. The figure shows a line chart in which the temperature is scaled on the primary y-axis (on the left) while the humidity is scaled on the secondary y-axis (on the right). This arrangement is useful for displaying two or more types of data that differ significantly in magnitude, such as the net specific work and thermal efficiency of a power cycle, on the same chart.

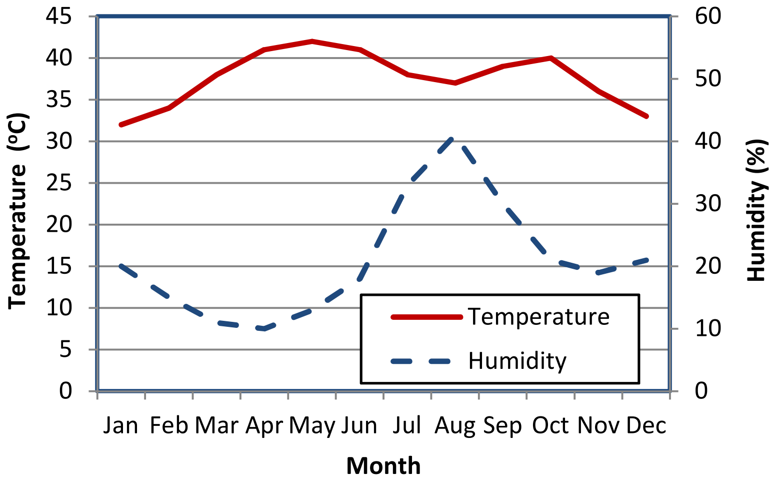


Figure 2.38. An example of line charts

Excel supports other types of charts that allow the user to select the most appropriate way to display his/her data in the form of bar, area, or scatter charts. For more information about the different types of Excel's charts, the reader can refer to specialised references such as Walkenbach [2]. A number of tutorials and videos that show how to create different types of charts can also be found in the internet.

A useful feature of Excel is that its charts provide a curve-fitting capability of numerical data by using the **Trendline** feature. This capability is particularly useful for computer-aided thermofluid analyses because it can be used to convert tabulated fluid-properties and other data into analytical equations that make the data more suitable for iterative solutions and optimisation analyses. To illustrate the use of this feature, consider Table 2.1 that shows properties of saturated water in the range 0.001ºC – 60ºC. These values of the saturation pressure (P_{sat}) and saturated liquid enthalpy (h_f) are used in psychrometric analyses of air-conditioning applications. For computer-aided analyses, it is useful to convert these data into relevant equations.

The trendline feature provides a number of options, which include exponential, linear, logarithmic, polynomial, and power equations as shown on Figure 2.39. To fit a trendline to the data, we have to create line charts for the two properties as shown on Figures 2.40.a and 2.40.b. Trendlines can then be added on the line charts. Figures 2.40.a and 2.40.b also show the corresponding trendline equations of the tabulated data as determined by using polynomial equations. As Figure 2.40.b shows, a linear equation is adequate for the h_f data since its variation over the given temperature range is mild. However, a third-order polynomial is required to represent the variation of P_{sat} with temperature as shown in Figure 2.40.a.

Table 2.1. Properties of saturated water at temperatures in the range 0ºC- 60ºC taken from Cengel and Boles [3]

TºC	P_{sat} [kPa]	v_f [m^3/kg]	v_g [m^3/kg]	u_f [kJ/kg]	u_g [kJ/kg]	h_f [kJ/kg]	h_g [kJ/kg]	s_f [kJ/kg.K]	s_g [kJ/kg.K]
0.01	0.6117	0.001000	206.00	0.000	2374.9	0.001	2500.9	0.0000	9.1556
5	0.8725	0.001000	147.03	21.019	2381.8	21.020	2510.1	0.0763	9.0249
10	1.2281	0.001000	106.32	42.020	2388.7	42.022	2519.2	0.1511	8.8999
15	1.7057	0.001001	77.885	62.980	2395.5	62.982	2528.3	0.2245	8.7803
20	2.3392	0.001002	57.762	83.913	2402.3	83.915	2537.4	0.2965	8.6661
25	3.1698	0.001003	43.340	104.83	2409.1	104.83	2546.5	0.3672	8.5567
30	4.2469	0.001004	32.879	125.73	2415.9	125.74	2555.6	0.4368	8.4520
35	5.6291	0.001006	25.205	146.63	2422.7	146.64	2564.6	0.5051	8.3517
40	7.3851	0.001008	19.515	167.53	2429.4	167.53	2573.5	0.5724	8.2556
45	9.5953	0.001010	15.251	188.43	2436.1	188.44	2582.4	0.6386	8.1633
50	12.352	0.001012	12.026	209.33	2442.7	209.34	2591.3	0.7038	8.0748
55	15.763	0.001015	9.5639	230.24	2449.3	230.26	2600.1	0.7680	7.9898
60	19.947	0.001017	7.6670	251.16	2455.9	251.18	2608.8	0.8313	7.9082

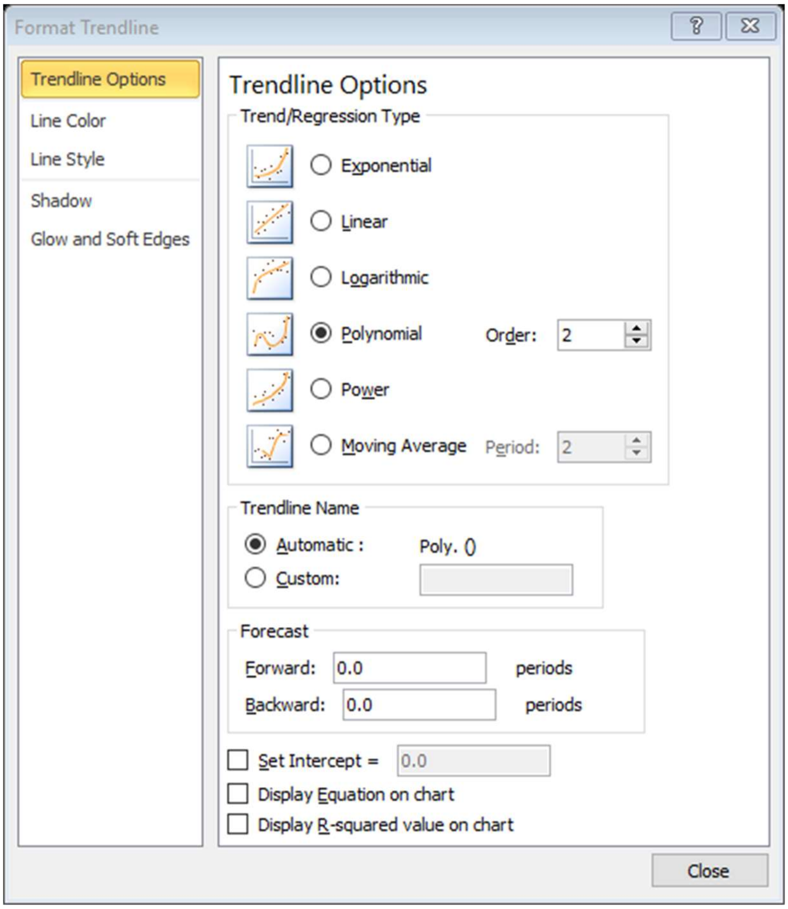
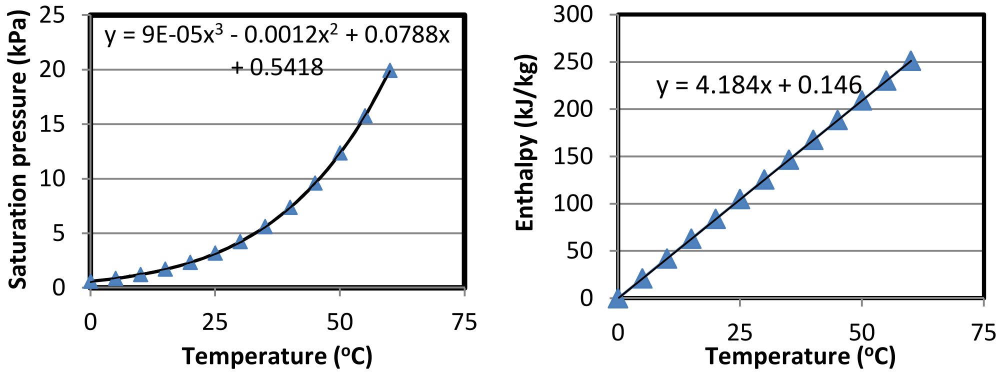


Figure 2.39. The Format Trendline window



(a) (b)

Figure 2.40. Fitting trendlines on tabulated data for water of (a) saturation pressure and (b) saturated liquid enthalpy

2.6. Closure

This chapter focusses on the basic features of Excel that are needed for developing Excel-aided models for thermofluid analyses. The chapter highlights the importance of using cell labelling with Excel's formulae and illustrates the use of Excel's general mathematical functions, logical functions, and the functions for matrix operations. The chapter also demonstrates the use of Excel's two iterative tools: Goal Seek and circular calculations. In spite of its simplicity, the Goal Seek command is very useful for iterative solutions as shown in later chapters of this book. Finally, the chapter illustrates the usefulness of Excel's charting tools for the presentation of tabulated data particularly the trendline feature.

It should be mentioned that the **Developer** tab of Excel's user-interface provides a number of useful features that enhance the effectiveness of Excel as modelling platform for thermofluid analyses, but these features have not discussed in the chapter. Two of these features are the ability to develop user-defined functions with VBA and the ability to record **macros** for conducting repetitive calculations and parametric analyses. VBA will be discussed in Chapter 3, but for more information about the use of macros the reader needs to refer to more specialised books on Excel or the internet.

References

[1] J. Walkenbach, *Excel 2010 Formulas*, Wiley Publishing Inc., 2010
[2] J. Walkenbach, *Excel 2007 Charts*, Wiley Publishing Inc., 2007
[3] Y. A. Cengel and M. A. Boles. *Thermodynamics an Engineering Approach*, McGraw-Hill, 7th Edition, 2007
[4] S. C. Chapra and R. P. Canale, *Numerical Methods for Engineers*, 6th Edition, McGraw Hill, 2010

Exercises

1. The following table shows measured values of the temperature by two different methods compared to the correct corresponding values. Find the average error for each method.

Correct T **(ºC)**	***Method*** **1**	***Method*** **2**
0	0.1044	0.1112
10	10.1092	10.1153
20	20.1139	20.1194
30	30.1186	30.1235
40	40.1231	40.1275
50	50.1276	50.1316
60	60.1320	60.1357
70	70.1364	70.1397
80	80.1407	80.1438
90	90.1450	90.1479
100	100.1493	100.1520

2. The following table shows the data for the saturation pressure of a certain fluid. Use a nested IF statement to develop an interpolation formula that determines the saturation pressure of the fluid at any temperature in the range 5ºC ≤ T ≤ 30ºC.

T(ºC)	P_{sat} (kPa)
5	0.872
10	1.228
15	1.705
20	2.339
25	3.169
30	4.246

3. A system of algebraic equations can be expressed in matrix form as follows:

$$\begin{bmatrix} 70 & 1 & 0 \\ 60 & -1 & 1 \\ 40 & 0 & -1 \end{bmatrix} \begin{Bmatrix} a \\ b \\ c \end{Bmatrix} = \begin{Bmatrix} 636 \\ 518 \\ 307 \end{Bmatrix}$$

Solve the system of equations to determine the values of the three unknowns a, b, and c by using:

(a) The matrix inversion method
(b) The iterative solution option.

This exercise is based on Example 9.11 in Chapra and Canale [4]. The answer is: a = 8.5941, b=34.4118, and c = 36.7647.

4. Figure 2.P4 shows a triangular fin attached to the surface of a wall. Solution of the conduction heat transfer equation with the finite-difference method resulted in the following system of linear equations the solution of which gives the temperatures in ºC at different distances from the fin base as shown in the figure:

$$-8.008\ T_1 + 3.5\ T_2 = -900.209$$

$$3.5\ T_1 - 6.008\ T_2 + 2.5\ T_3 = -0.209$$

$$2.5\ T_2 - 4.008\ T_3 + 1.5\ T_4 = -0.209$$

$$1.5\ T_3 - 2.008\ T_4 + 0.5\ T_5 = -0.209$$

$$T_4 - 1.008\ T_5 = -0.209$$

Use Excel functions to solve the above system of linear equations.

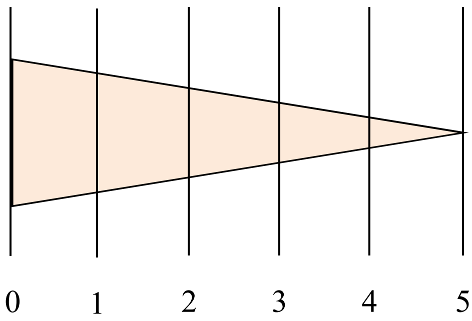


Figure 2.P4. Triangular fin

5. Adopting suitable names in your formulae, prepare an Excel sheet for calculating the frictional loss (h_f) in a circular pipe of diameter D, length L, and roughness k_s. Use your sheet to determine h_f in the following cases:

(a) D = 25 cm, L = 150 m, V = 2 m/s, k_s = 0.045 mm, carrying water at 20ºC.
(b) D = 25 cm, L = 150 m, V = 0.2 m/s, k_s = 0.045 mm, carrying oil at 20ºC.
(c) D = 25 cm, L = 150 m, V = 7 m/s, k_s = 0.045 mm, carrying air at 20ºC.

Use the Darcy-Weisbach equation and determine the values of the kinematic viscosity from relevant property tables.

6. Using a line chart, plot the variation of sine θ for $-180 \leq \theta \leq 180$ in steps of 10ºthen add cosine θ on the same chart.

7. Using the data shown in Table 2.1, make a line chart for v_f and v_g. Add polynomial trendlines for both properties and comment on the trendlines equations.
8. The table below shows some of the thermo-physical properties of air at atmospheric pressure and different temperatures. Use Excel charts to show the variation of the properties ρ, β, c_p, k, α, μ, v, and Pr with temperature and use trendline to obtain suitable equations for these properties.

T (K)	ρ (kg/m^3)	$\beta\times10^3$ (1/K)	c_p (J/kg.K)	k (W/m.K)	α (m^2/s)	$\mu\times10^6$ (N S/m^2)	$\nu\times10^6$ (m^2/s)	Pr
273	1.252	3.66	1011	0.0237	19.2	17.456	13.9	0.71
293	1.164	3.41	1012	0.0251	22.0	18.240	15.7	0.71
313	1.092	3.19	1014	0.0265	24.8	19.123	17.6	0.71
333	1.025	3.00	1017	0.0279	27.6	19.907	19.4	0.71
353	0.968	2.83	1019	0.0293	30.6	20.790	21.5	0.71
373	0.916	2.68	1022	0.0307	33.6	21.673	23.6	0.71
473	0.723	2.11	1035	0.0370	49.7	25.693	35.5	0.71
573	0.596	1.75	1047	0.0429	68.9	29.322	49.2	0.71
673	0.508	1.49	1059	0.0485	89.4	32.754	64.6	0.72
773	0.442	1.29	1076	0.0540	113.2	35.794	81.0	0.72

9. The volume V of liquid in a spherical tank of radius r is related to the depth h of the liquid by:

$$V = \pi h^2(3r - h)/3$$

Using the Goal Seek command, determine the value of h for the tank with r=1 m and V = 0.5 m^3.

This exercise is based on Problem 8.9 in Chapra and Canale [4]. Answer: h = 0.431 m.

3

Solver, VBA, and Thermax

Excel is the main component of the modelling platform used in this book for thermofluid analyses by providing the user-interface, numerous built-in functions, two iterative tools, and the graphic tools. However, the three auxiliary components; Solver, VBA, and Thermax make Excel an adequate and effective modelling platform for thermofluid analyses. This chapter focuses on these three components and illustrates their use by means of relevant examples. Initially, the chapter shows how to activate Solver and use its three solution methods for performing optimisation analyses and solving systems of linear and nonlinear equations. The chapter then shows how VBA can be used for developing custom functions and how Thermax functions can be used in Excel formulae.

3.1. Solver

Solver is a multi-purpose iterative tool developed by Frontline Systems [1] as an Excel add-in for "What-if" analyses. Compared to Excel's own iterative tool, which is the Goal Seek command described in Chapter 2, Solver offers the following advantages:

1. While Goal-Seek can be used for simple problems that involve only one decision variable, Solver can deal with more difficult problems in which the objective cell is affected by numerous decision variables.
2. Goal Seek allows only a required value of the objective cell to be achieved, but beside that Solver enables Excel to perform an optimisation analysis by finding the maximum or minimum value for the formula in the objective cell.
3. Solver allows constraints to be applied on the iterative solution.
4. Solver offers three solution methods that suit different types of problems and gives the user a number of numerical options for applying these methods. One of these methods is the Evolutionary method which is needed for multi-objective optimisation analyses.

3.1.1. Activation of Solver

Like the Goal Seek command, Solver is found in the **Data** tab as shown in Figure 3.1. If it doesn't appear in your **Data** tab, then go to **File**, click **Options**, and select **Add-Ins**. From the **Manage** option at the bottom of the menu select **Excel Add-ins** and then click "**Go**". The **Add-Ins** dialog box shown in Figure 3.2 will be shown. To add **Solver** to the add-ins menu, tick (√) on the "**Solver"** option and return to the **Data** tab. When you click the **Solver** button from the **Data** tab, the **Solver Parameters** dialog box shown in Figure 3.3 will be shown.

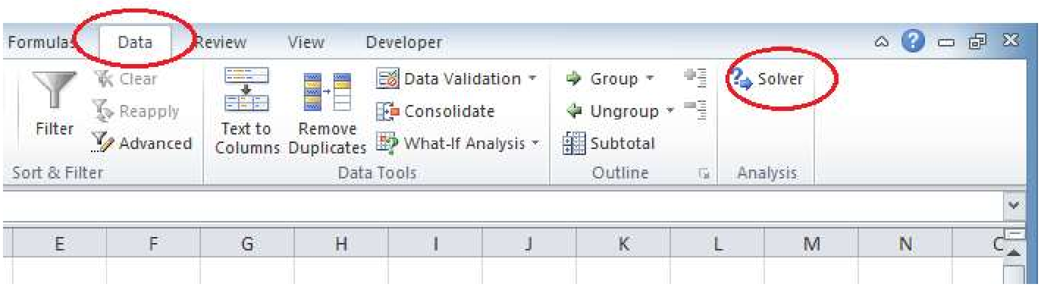


Figure 3.1. The Solver add-in in the Data tab

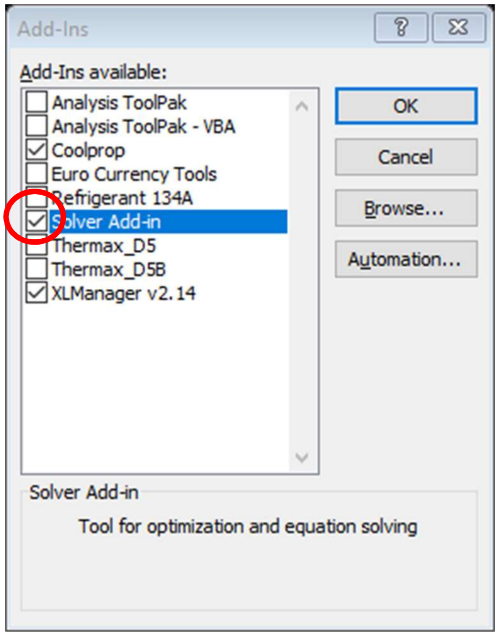


Figure 3.2. Activating Solver from the menu of Excel add-ins

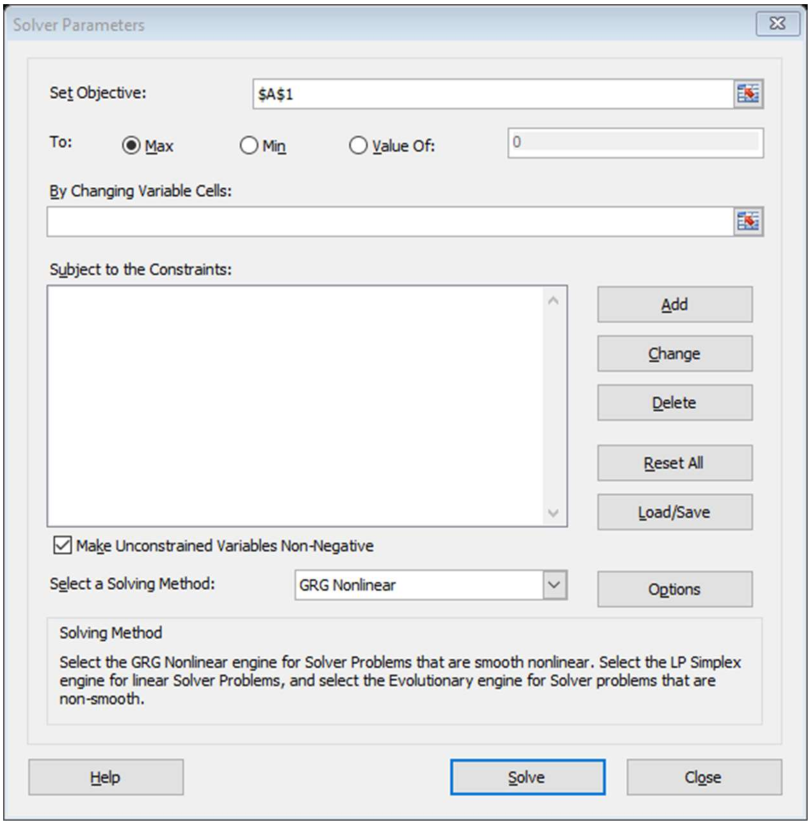


Figure 3.3. Solver Parameters dialog box

Solver Parameters dialog box helps the user to select a formula in one cell, called the **objective cell**, and a group of other cells, called **decision variables** or variable cells, that are directly or indirectly related to the formula in the objective cell. By adjusting the **decision variables**, the **objective cell** can be maximised, minimised, or made to acquire a certain value. As shown on Figure 3.3, **constraints** can be applied on the sought values of the decision variables. To suit different types of problems, Solver offers three solution methods which are:

1. The **GRG Nonlinear** method,
2. The **Evolutionary** method
3. The **Simplex LP** method.

The **GRG Nonlinear** method involves the determination of the function's gradient like the Steepest Descent method [2, 3]. The **Evolutionary** method adopts a variety of genetic algorithms and local search methods [4]. Both the **GRG Nonlinear** method and the **Evolutionary** method are suitable for non-linear problems. The **Simplex LP** method, which is a linear-programming method, is suitable for linear problems. As shown on Figure 3.3, Solver uses the **GRG Nonlinear** method by default. The following sections give examples of using the three solution methods.

3.1.2. The GRG Nonlinear method

In an optimisation analysis we may require the objective function to be maximised or minimised depending on the situation at hand. For example, the optimisation of a pipe insulation requires its total cost to be minimised, while the optimisation of a central thermal power plant requires its thermal efficiency to be maximised. The following example illustrates the use of the **GRG Nonlinear** method in optimisation analyses.

Example 3-1. Finding the minimum value of a quadratic function

Find the minimum value of the following quadratic function in the specified range.

$$f(x) = x^2 - 2x - 1; \qquad -2 \le x \le 3 \tag{3.1}$$

Solution

Figure 3.4 shows the Excel sheet developed for this example. The line chart inserted in the figure shows the variation of f with x from which we can estimate that the minimum value of f is -2 and occurs at x =1. An initial value for x is entered in cell **B3** based on which the function $f(x)$ is calculated in cell **B6** according to Equation (3.1). Note the curser is placed on cell **B6** to reveal the formula typed in the cell, which is:

= B3^2-2*B3−1

We can now use Solver to determine the minimum value of the function. To do so, select **Solver** from the **Data** tab and fill its parameters dialog-box as shown on Figure 3.5 that shows the top part of the completed box.

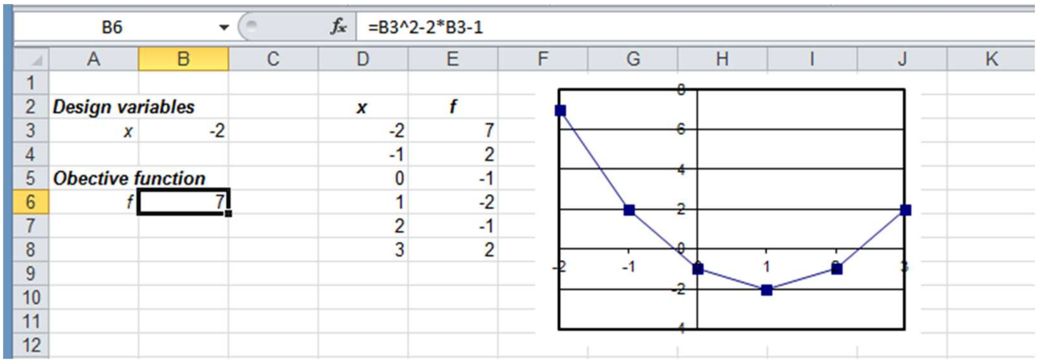


Figure 3.4. Excel sheet for determining the local minimum of the quadratic function

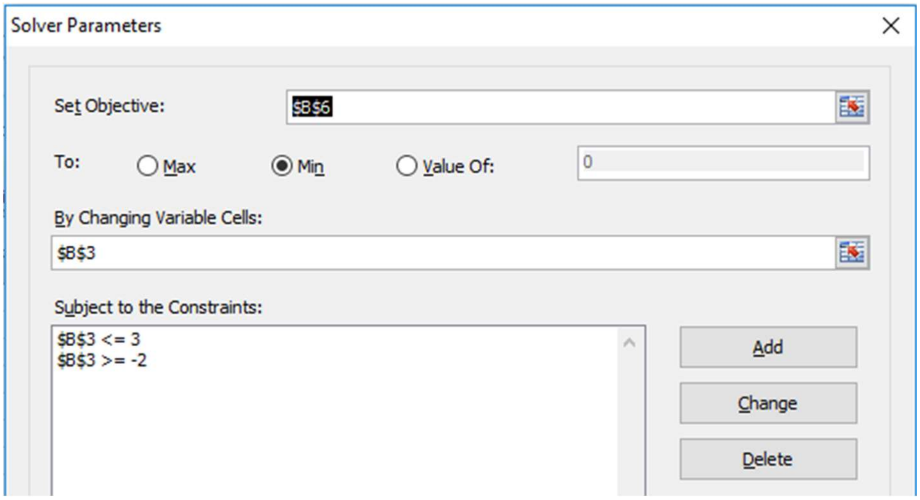


Figure 3.5. The completed Solver dialog box for Example 3-1

The dialog box in Figure 3.5 has been filled as follows:

Set Objective:	**B6** and **Min** have been selected for this option since want the value of the function in cell **B6** to be minimised.
By Changing Variable Cells:	**B3** which is the cell that stores the value of the independent variable x.
Subject to the Constraints:	Two constraints have been added that specify the minimum and maximum values of x as $x \geq -2$ and $x \leq 3$, respectively.
Select a Solving Method:	The **GRG Nonlinear** method (the default option).

Pressing the "**Solve**" button of the completed parameters box sparks Solver to find the required solution. As Figure 3.6, shows, the answer found by Solver is $x = 1, f = -2$, which agrees with the function's graph shown on Figure 3.4.

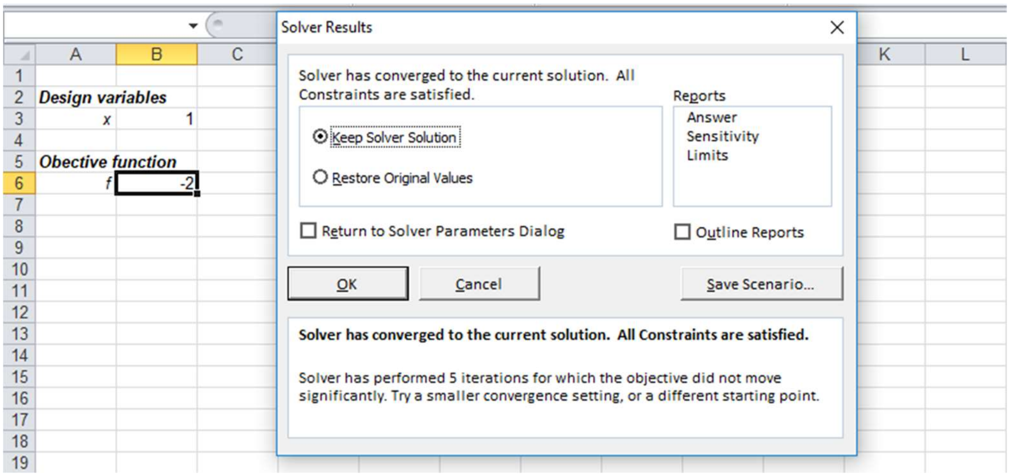


Figure 3.6. Solver solution for Example 3-1

3.1.3. The Evolutionary method

When the function to be optimised has more than one point of inflection, the solution found by the **GRG Nonlinear** method may be only a local minimum or maximum. The following example illustrates the capability of the **Evolutionary** method to find the global optimum solution in this situation for a simple function.

Example 3-2. Finding the global minimum of a function

Determine the global minimum value for the following function:

$$f(x) = x\sin(x) \qquad 3 \le x \le 14 \qquad (3.2)$$

Solution

Figure 3.7 shows the Excel sheet developed for solving this example. The insert shows that the function has two minima in the specified range of x; *viz*., at $x \approx 5$ and $x \approx 11$. Let us first solve the problem with the **GRG Nonlinear** method. An initial value for x has been entered in cell **B2** based on which the function f is calculated in cell **B4**. The formula bar in Figure 3.7 reveals the formula entered in the cell **B4**. Figure 3.8 shows the completed Solver parameters dialog-box with two constraints that specify the upper and lower limits for x. From Figure 3.9 that shows the solution found by the **GRG Nonlinear** method it is clear that it found the local minimum at $y = -4.81$ which is nearer to the initially specified value. In order to locate the global minimum by the method, the solution has to start with an initial guess that is closer to the global minimum. Therefore, the method offers a **Multi-start** option for such cases. Figure 3.10 shows the parameters-box with this option being selected and Figure 3.11 shows the new solution which is $y = -11.041$. With the **Evolutionary** method this option is not necessary. To use this method, the set-up shown in Figure 3.8 only needs Solver's solution method to be changed to "**Evolutionary**". Figure 3.12 that shows the solution obtained by this method confirms that the method produced the global minimum at $y = -11.041$.

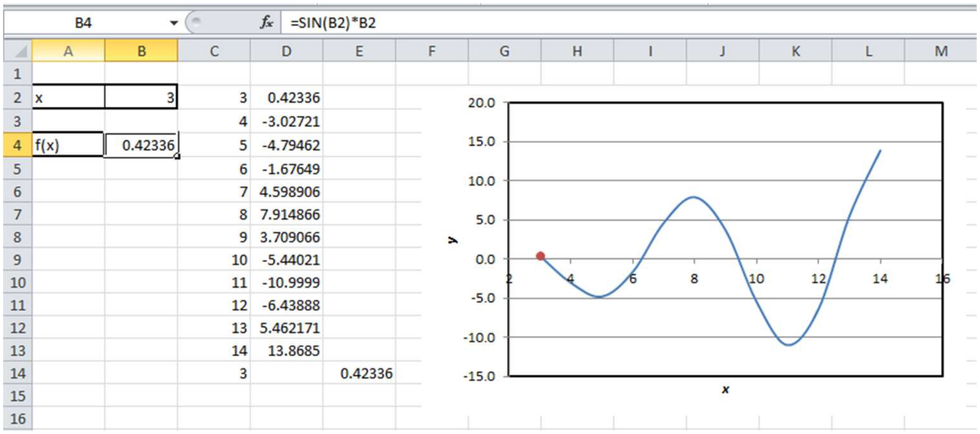


Figure 3.7. The Excel sheet for Example 3-2

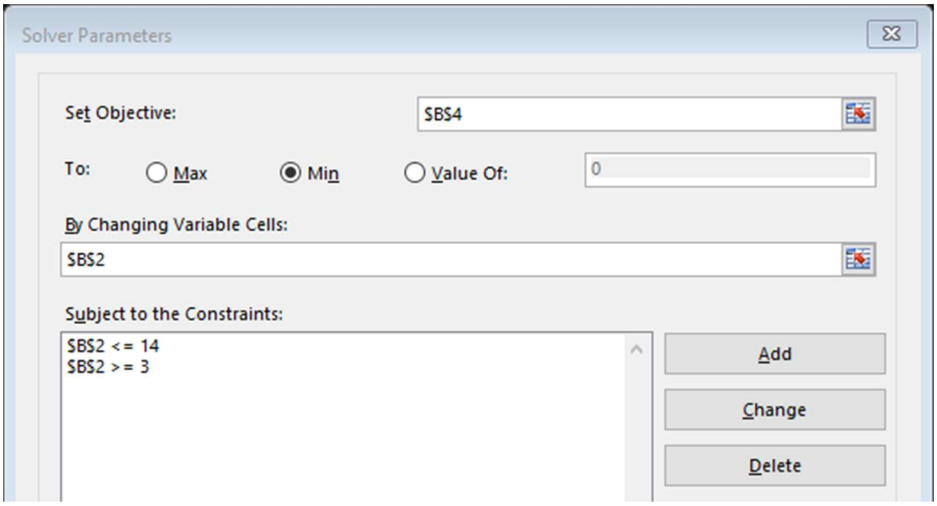


Figure 3.8. Solver set-up for Example 3-2 with the GRG Nonlinear method

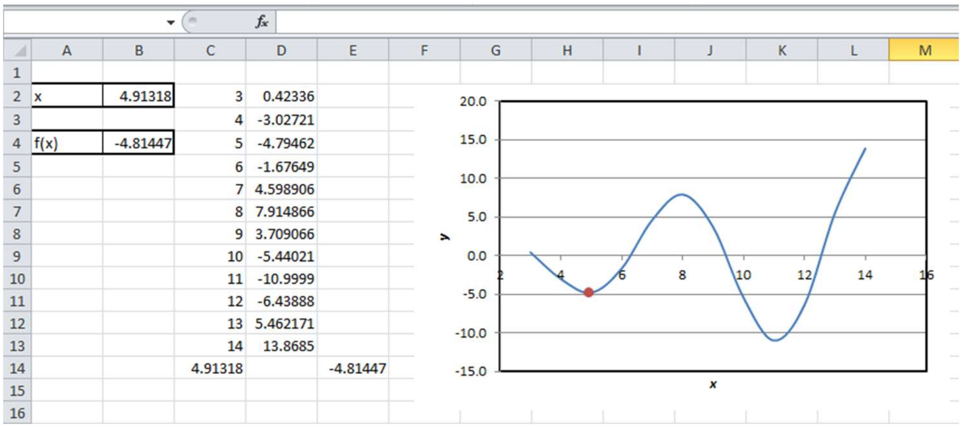


Figure 3.9. Solver solution for Example 3-2 with the GRG Nonlinear method

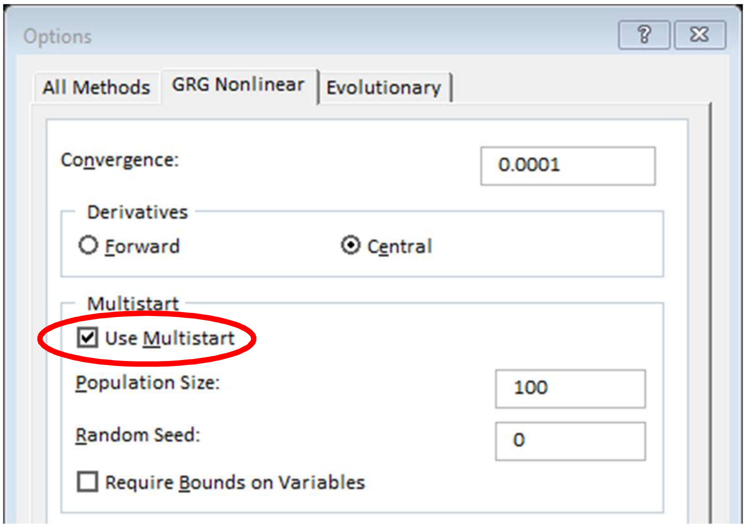


Figure 3.10. Activation of the multi-start option of the GRG Nonlinear method

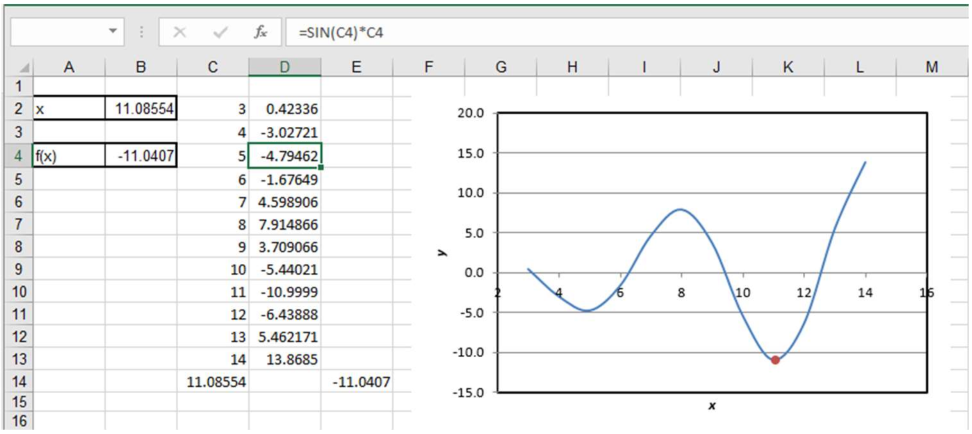


Figure 3.11. Solution of the GRG Nonlinear method with the multi-start option

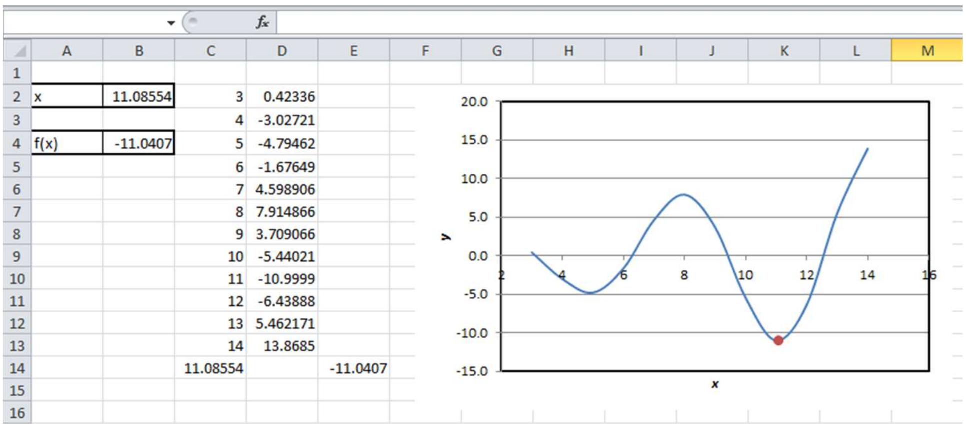
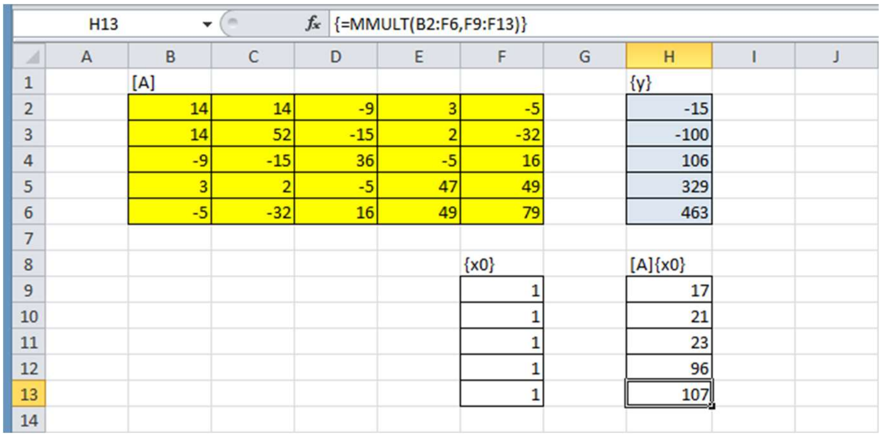


Figure 3.12. Solver solution for Example 3-2 with the Evolutionary method

While the **GRG Nonlinear** method took less than a second to solve the above problem, the **Evolutionary** method took more than a minute with the same computer. The **Evolutionary** method is useful for optimisation analyses that involve non-smooth and discontinuous functions, which are difficult to solve with the **GRG Nonlinear** method. It is also the method to use when dealing with multi-objective optimisation analyses.

3.1.4. The Simplex LP method

This method provides another option for solving small systems of linear equations that can be used in addition to the methods described in the Chapter 2 . The method will be illustrated by reconsidering the problem of Example 2-3. Figure 3.13 shows a new Excel sheet for solving the problem with the present method.



H13 {=MMULT(B2:F6,F9:F13)}

	A	B	C	D	E	F	G	H	I	J
1		[A]						{y}		
2		14	14	-9	3	-5		-15		
3		14	52	-15	2	-32		-100		
4		-9	-15	36	-5	16		106		
5		3	2	-5	47	49		329		
6		-5	-32	16	49	79		463		
7										
8						{x0}		[A]{x0}		
9						1		17		
10						1		21		
11						1		23		
12						1		96		
13						1		107		
14										

Figure 3.13. Excel sheet for solving Example 2-3 with Solver

The top part of the sheet stores the coefficient matrix [A] and the right-hand vector {y} of the system of linear equations to be solved. The procedure starts with a guessed solution which is stored as vector {x0} in cells **F9:F13**. All the elements of this vector are given a value of 1 as shown in Figure 3.13. The coefficient matric [A] is then multiplied by the guessed vector {x0} using the "**MMULT**" function of Excel and the result stored in cells **H9:H13**. If the initial guess is the correct answer, the multiplication [A]{x0} will be the same as the true right-hand side vector, i.e.,

$$[A]\{x0\} = \{y\} \tag{3.3}$$

However, Figure 3.13 shows that the vector [A]{x0} is different form the true right-hand side vector {y} stored in cells **H2:H6**. Solver can now be used to adjust the variable cells **D9:D13** so that all elements of the vector [A]{x0} become equal to their counterparts in vector {y}, i.e.:

H9 = H2
H10 = H3
H11 = H4

H12 = H5
H13 = H6

Solver set-up for this task is shown in Figure 3.14. Note that the objective cell is left blank and the **Simplex LP** method is selected as the solution option. In this case, Solver will iterate to find the values of the decision variables that satisfy all the imposed constraints. The solution found by Solver using the above set-up is shown on Figure 3.15.

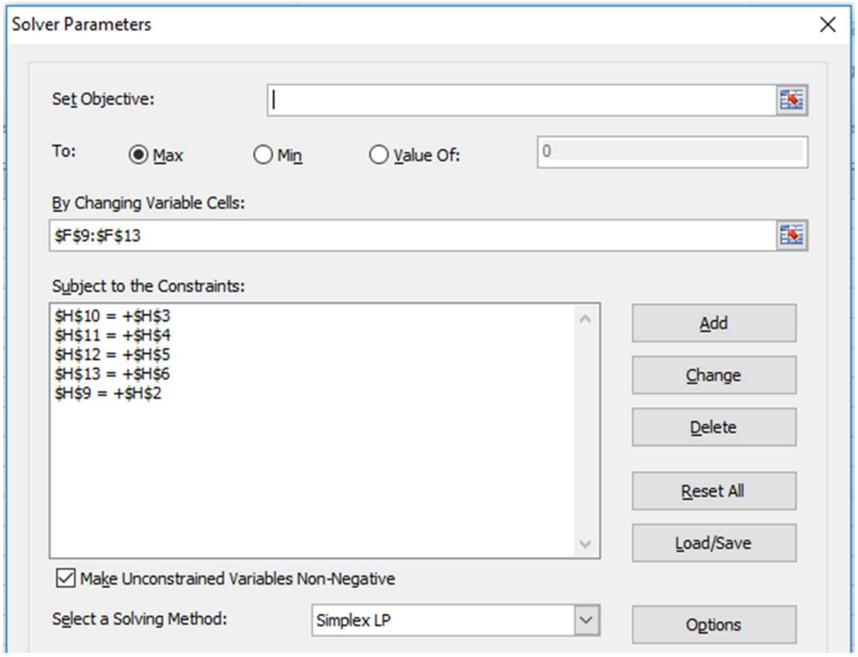
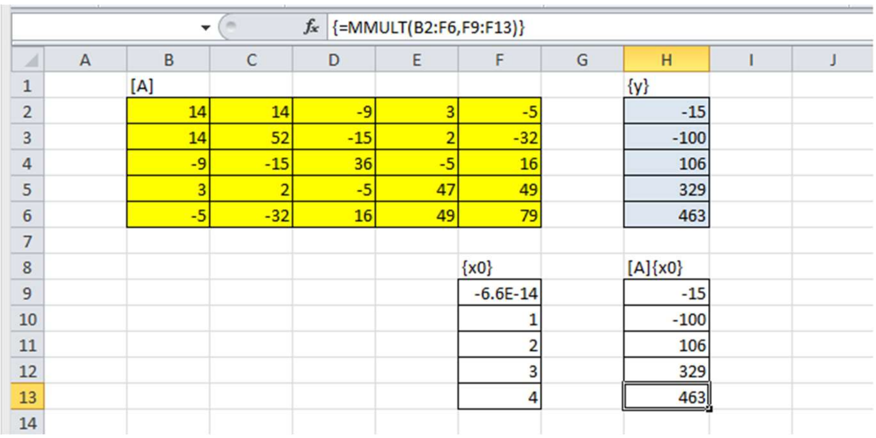


Figure 3.14. Solver set-up for Example 2-3 with the Simplex LP method


{=MMULT(B2:F6,F9:F13)}

	A	B	C	D	E	F	G	H	I	J
1		[A]						{y}		
2		14	14	-9	3	-5		-15		
3		14	52	-15	2	-32		-100		
4		-9	-15	36	-5	16		106		
5		3	2	-5	47	49		329		
6		-5	-32	16	49	79		463		
7										
8						{x0}		[A]{x0}		
9						-6.6E-14		-15		
10						1		-100		
11						2		106		
12						3		329		
13						4		463		
14										

Figure 3.15. Solution of Example 2-3 with the Simplex LP method

All the elements of the [A]{x0} are now equal to their corresponding elements in the vector {y}. The first element of the solution vector, which is -6.6×10^{-16}, is practically zero. Therefore, the solution is [0,1,2,3,4], which is the same as that obtained in Example 2-3 by using the matrix-inversion method. The advantage of Solver compared to the matrix-inversion method is that it can be used for solving systems of nonlinear equations by following the procedure (Refer to Problem 3.5 in the Exercises).

3.1.5. The default options of Solver solution methods

Solver gives it user additional flexibility by allowing alternative options for monitoring and adjusting the precision and computer time of its three solution methods. While some options are common to all three solution methods, others are particular to the **GRG Nonlinear** or the **Evolutionary** method. By clicking the "**Options**" button in Solver's parameters dialog box shown in Figure 3.16, the dialog box shown in Figure 3.17 will be shown. Figure 3.17 shows the default settings of the options that are common to all three solution methods. In certain situations, some of these default settings may have to be changed in order to reduce the computation time or increase the precision of the solution. Sometimes, Solver fails altogether to find the solution if the default options are used. As shown in Section 3.1.3, the multi-start option had to be used for the GRG Nonlinear method to solve the problem of Example 3-2. An option that is common to the three methods is use of automatic-scaling (AS) which is favourable in certain situations but not always. AS enables Solver to handle a poorly-scaled model, i.e., a model in which the values of the objective and constraint functions differ by several orders of magnitude. By using AS, the values of the objective and constraint functions are scaled internally in order to minimise the differences between them.

Figures 3.18.a and 3.18.b show the default settings which are particular to the **GRG Nonlinear** method and the **Evolutionary** method, respectively. The **GRG Nonlinear** method uses by default the forward difference (FD) approximation of derivatives. Most of the analyses presented in later chapters of the book used the **GRG Nonlinear** method with the default FD approximation. More information about Solver options for the **GRG Nonlinear** method can be obtained from the developer's website [5].

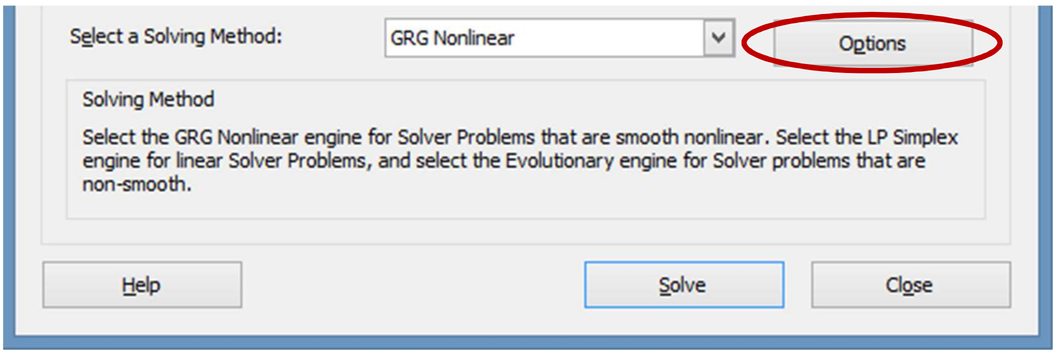


Figure 3.16. Solver options in the Properties dialog box

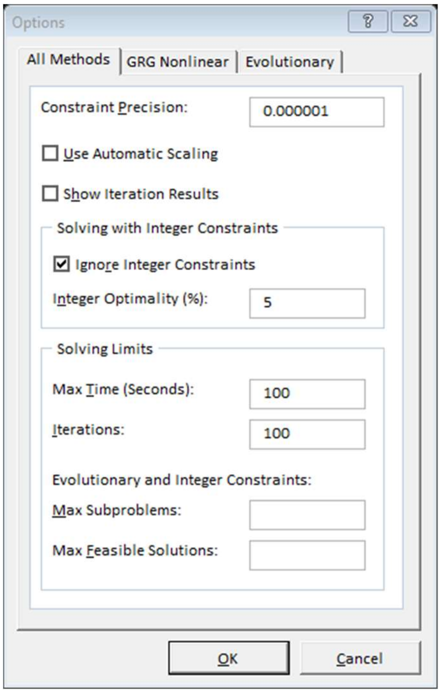
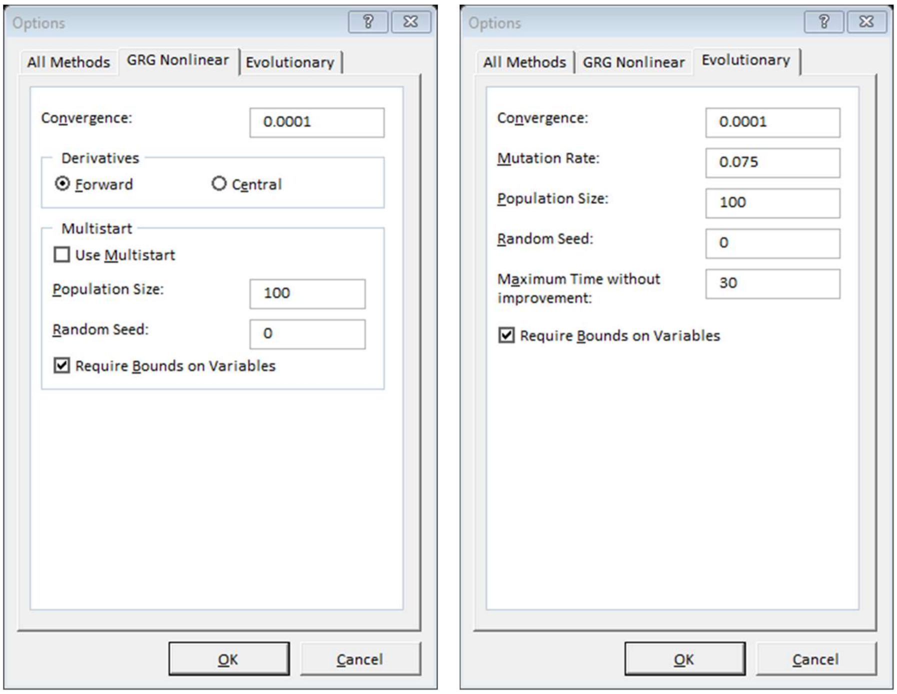


Figure 3.17. Default Solver options adopted in the analyses for all solution methods



(a) (b)

Figure 3.18. The default Solver options specific to: (a) the GRG Nonlinear method and (b) the Evolutionary method

Figure 3.18.b shows the default settings used by the **Evolutionary** method. Note that this method has more adjustable parameters than the **GRG Nonlinear** method. According to the default set-up, the population size is 100 and the maximum allowable time without improvement is 30 seconds. Compared to the **GRG Nonlinear** method, the **Evolutionary** method needs longer computer times. The cases solved with the **Evolutionary** method in later chapters of the book to do not show a clear advantage to this method over the **GRG Nonlinear** method for problems that involve a single-objective. However, the method is useful when the problem at hand involves multiple conflicting objectives and a suitable trade-off between them is to be found.

3.2. VBA and the development of user-defined functions

Although Excel's user-interface provides numerous built-in functions for data analyses in general, there are situations where the analytical model requires the development of a customised user-defined function (UDF) that is not provided by Excel. This arises, for example, in thermodynamic analyses that require functions that determine the properties of fluids at various pressures and/or temperatures. This section illustrates the process of activating VBA and using it to develop simple UDFs.

3.2.1. Activation of VBA

As shown in Figure 3.19, VBA is found on the left side of the **Developer** tab. If the **Developer** tab is not shown in the ribbon of your Excel sheet, then go to **File**, select **Options**, and then select **Customise Ribbon** from the **Backstage View** shown in Figure 3.20. From the **Main Tabs**, select the **Developer** check box and then click "**OK**". The **Developer tab** will now be shown in the ribbon of your Excel sheet.

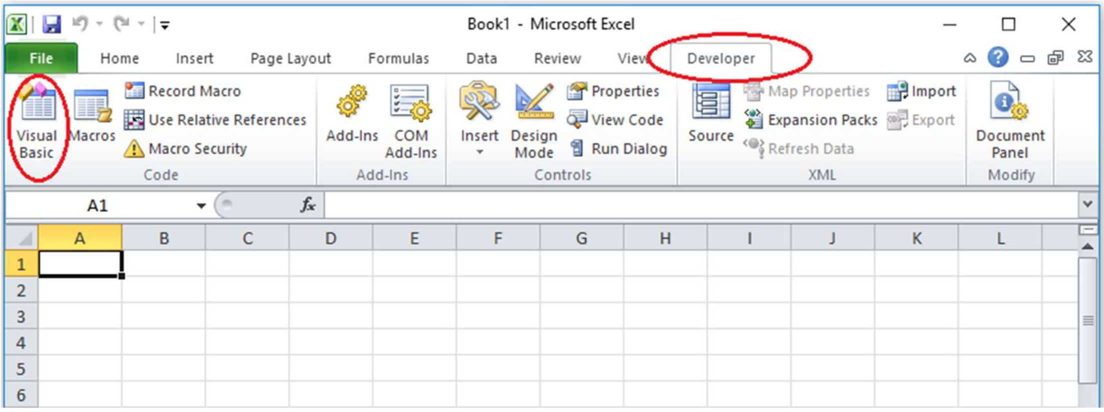


Figure 3.19. Selection of VBA from the Developer tab

3.2.2. Development of UDFs

To start writing a UDF, go to **Developer** tab menu and select **Visual Basic**. The Visual Basic editor will appear to you as shown on Figure 3.21. Select **Insert** → **Module** and the blank page shown on Figure 3.22 will be open for you to type the VBA code.

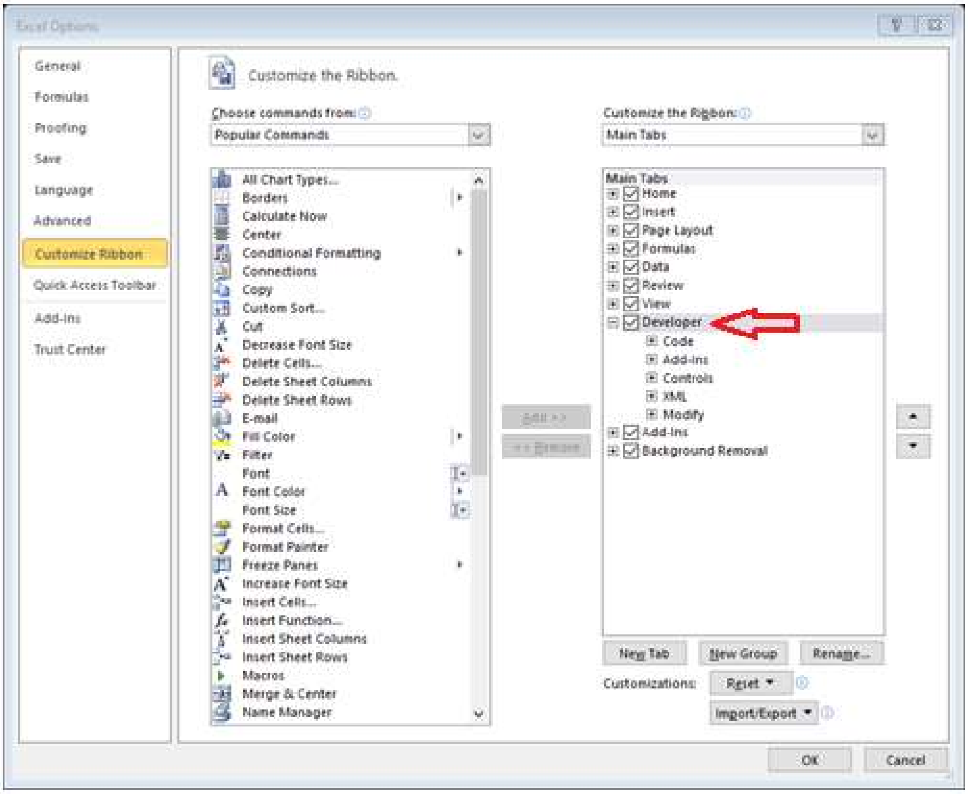


Figure 3.20. Adding VBA to the Developer tab

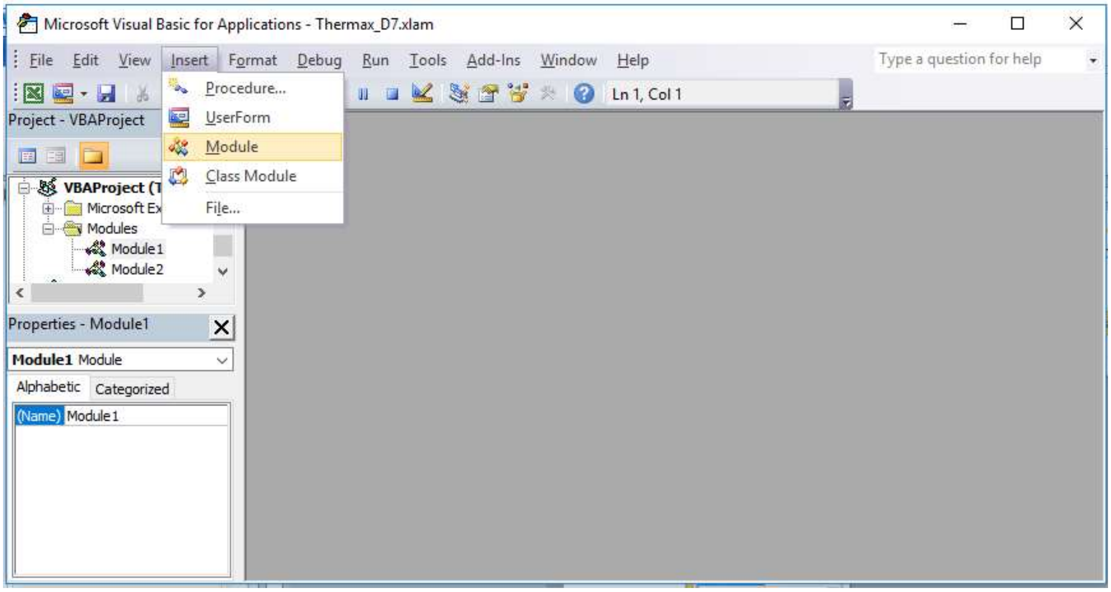


Figure 3.21. Inserting options for VBA

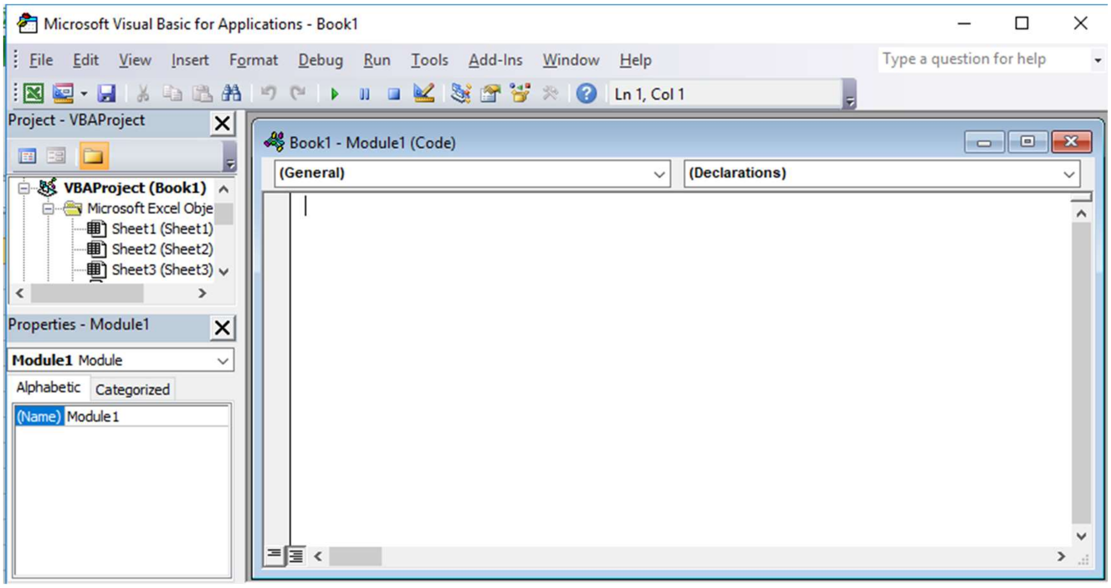


Figure 3.22. Inserting a new VBA module

As a first example, let us write a VBA function for determining the area (A) of a circle given its diameter (D) using the following mathematical equation:

$$A = \pi D^2 / 4 \quad (3.4)$$

The following UDF determines the circle's area according to Equation (3.4):

```
Function Circ_area(Dia)
Pi = 3.141593
Circ_area = Pi * Dia ^2 / 4
End Function
```

Note that the first line in the code starts with the word "**Function**" followed by the name given to the function, which is "**Circ_area**". The required input parameters are specified between two brackets after the function name. The present function has only one input parameter which is the diameter (Dia). As soon as you type the first line of the code and press the "**Enter"** key, the editor will automatically add the **End** line of the function. Now, type the rest of the code as shown in Figure 3.23. After typing the code correctly, the function can be used via Excel UI just like any built-in function as shown in Figure 3.24. Note that the formula bar in Figure 3.24 reveals the formula in cell B2 as:

= Circ_area(10)

Where the number 10 refers to the diameter of the circle, which is the only input to the UDF.

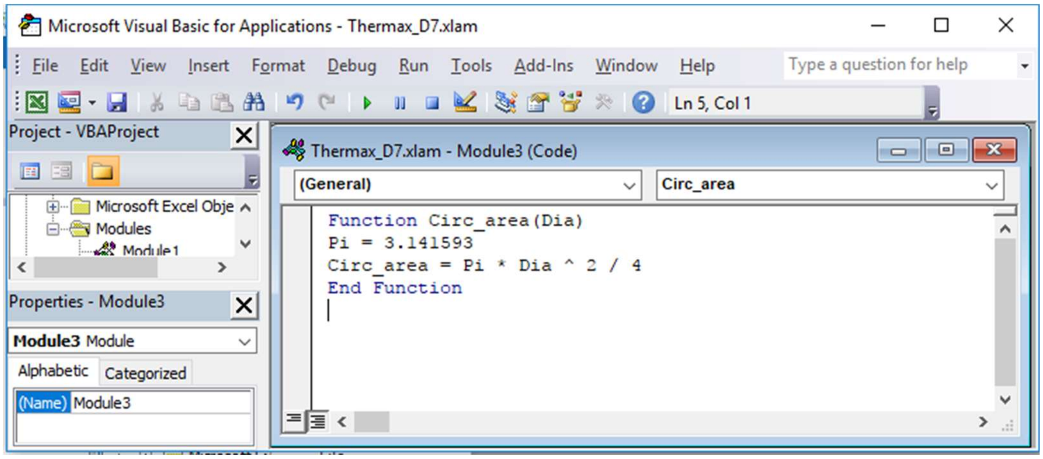


Figure 3.23. A UDF for caclcuating the area of a circle with a given diameter

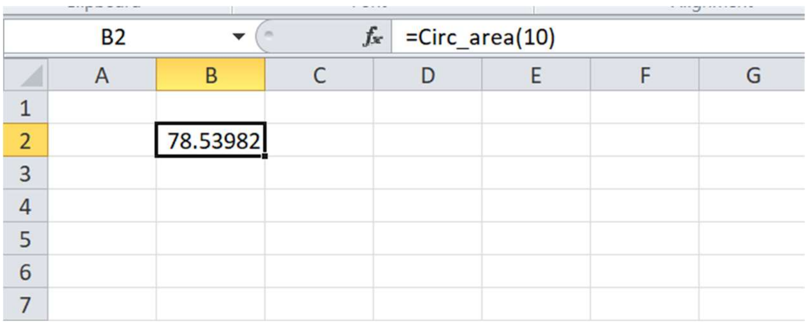


Figure 3.24. Using the "Circ_area" function in Excel

In writing the UDF shown on Figure 3.23, we assigned a value for the constant π because VBA does not provide a built-in function for it like Excel. However, it is possible to call the built-in functions **PI** provided by Excel within the VBA function as follows:

```
Function Circ_area(Dia)
Pi = Application.WorksheetFunction.Pi()
Circ_area = Pi * Dia ^ 2 / 4
End Function
```

This is very useful since built-in functions, like MIN and MAX, can be used to minimise the programming effort needed for developing the required UDF. More information about this and other features of the VBA language can be found in the references [7-9].

For thermodynamic analyses, VBA is useful for developing UDFs for fluid properties. As an example, let us develop a UDF that determines the specific-heat at constant pressure (c_p) for air. For an ideal gas, the molar specific-heat, $\tilde{c}_p$, is given by the following polynomial [6]:

$$\tilde{c}_p = a_0 + a_1 T + a_2 T^2 + a_3 T^3 \qquad \text{[kJ/kmol.K]} \qquad (3.5)$$

Where T is the absolute temperature and a_0, a_1, a_2, and a_3 are constants that have different values for different gases. For air, their values are 28.11, 0.1967x10^{-2}, 0.4802 x10^{-5}, and -1.966 x10^{-9} in this order. The following VBA function, called "cp_air", determines c_p ($c_p = \tilde{c}_p / M$), in kJ/kg.K, for air based on Equation (3.5):

```
Function cp_air(TempK)
a0 = 28.11
a1 = 0.00196
a2 = 0.000004802
a3 = -0.000000001966
M = 28.97
cpbar = a0 + a1 * TempK + a2 * TempK ^ 2 + a3 * TempK ^ 3
cp_air = cpbar / M
End Function
```

The only input for the function is the absolute temperature (TempK). Figure 3.25 shows the VBA function and the formula bar in Figure 3.26 shows how the function can be used in an Excel formula to determine c_p for air at 300K. The value returned by the function is 1.0037 kJ/kg.K. By making suitable extensions, the function can easily be used to determine the values of c_p for other ideal gases in addition to air (Refer to Exercise 3.8). The various functions provided by Thermax for the thermo-physical properties of water, refrigerants, etc., have been developed by writing similar functions with VBA.

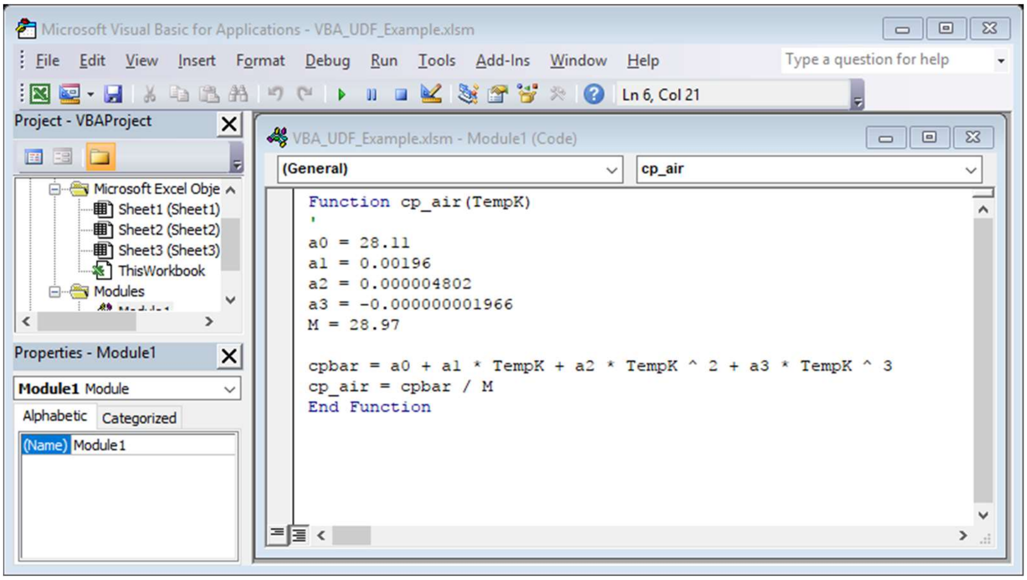


Figure 3.25. A UDF for caclcuating the molar specific-heat for air

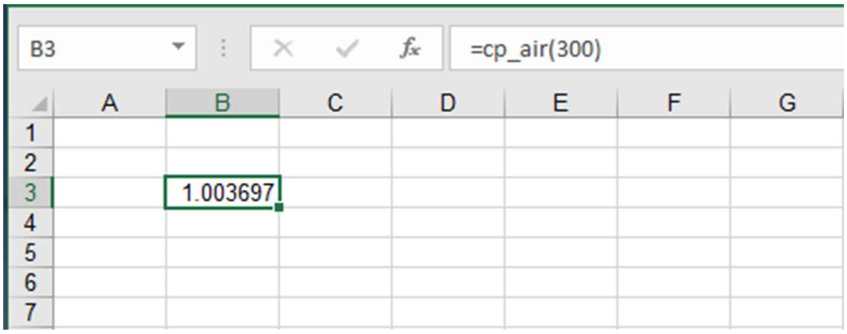


Figure 3.26. Using the cp_air function in an Excel formula

3.3. Thermax installation and use

Thermax enables Excel to be used as a modelling platform for a wide range of thermofluid analyses [10]. Before the add-in can be recognised by Excel you have to install it in your computer. To do that, open the **Thermax.xla** file and then save it as an "**Excel Add-in**". Recent Excel versions locate all add-ins in a certain folder in the computer and automatically direct you to that location when you want to save a new add-in. Save the Thermax add-in in the specified location and **restart** Excel in order to activate it. Open a new Excel sheet and then do the following:

1. Go to **File** and then click **Options**.
2. Select **Add-Ins**. From the **Manage** ribbon at the bottom of the menu select **Excel Add-ins** and then press **Go**. The pull-down menu shown in Figure 3.27 will appear to you.
3. To add **Thermax** to the add-ins menu, tick (√) the corresponding box.
4. If for any reason you saved the add-in in a location that is different from the default folder, then click on **Browse** and search for it in the destination folder and select it.

Once installed, Thermax functions can be used in Excel's formulae just like the built-in functions. For illustration, let us start a formula by entering the equal sign (=) in the cell B2. If you now press the *fx* button in the formula ribbon, the **Function Wizard** shown in Figure 3.28 will be shown. The Function Wizard first lists the various categories of built-in functions provided by Excel. Scroll down the list of function categories and select the **User-defined** functions. Then, all the functions provided by Thermax will be listed alphabetically as shown in Figure 3.29. The first function in the list, **Air_Data**, is the auxiliary function that stores the data for the thermo-physical properties of air at standard atmospheric pressure. This function is called by other functions in the same category to obtain the values of these properties at the required temperature. To start using the add-in functions, scroll down the list and select the function **Airk_T** that determines the thermal conductivity (*k*) of air at a given temperature. Upon pressing the **OK** button, the **Function Arguments** box shown in Figure 3.30 will be shown.

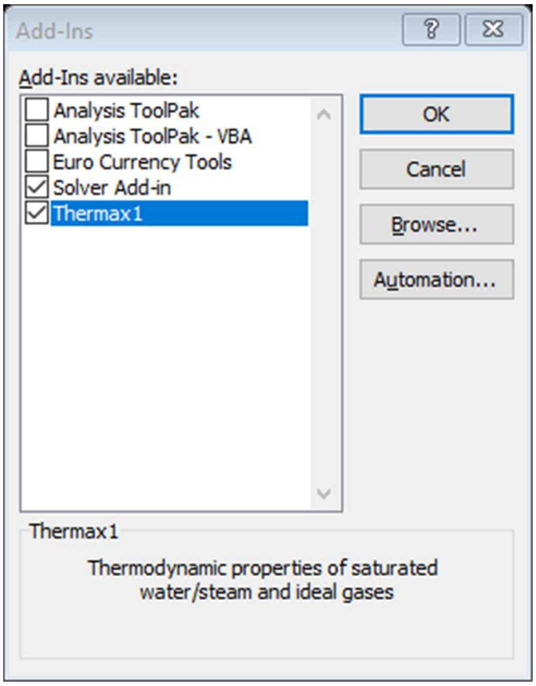


Figure 3.27. Adding Thermax to the menu of Excel add-ins

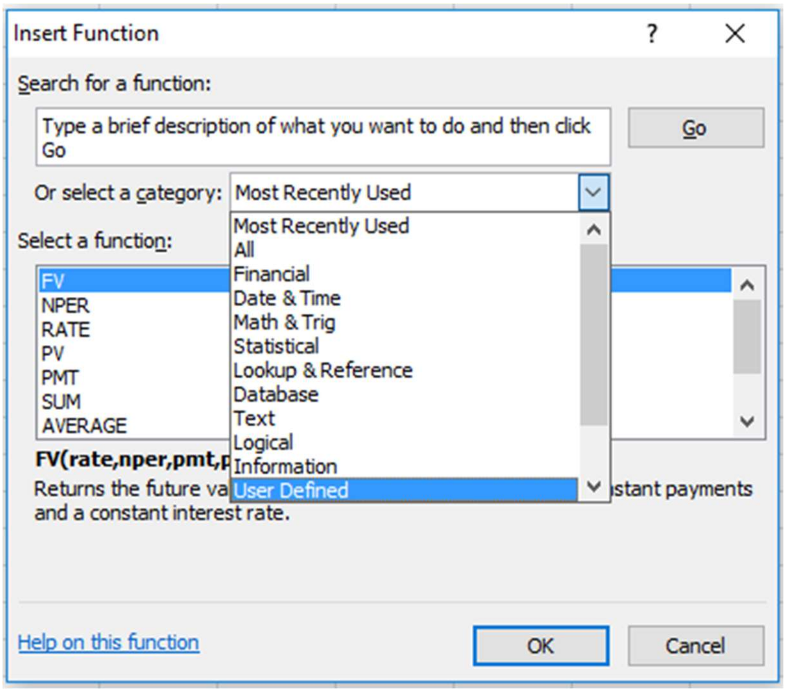


Figure 3.28. Finding the add-in user-defined functions in the Function Wizard

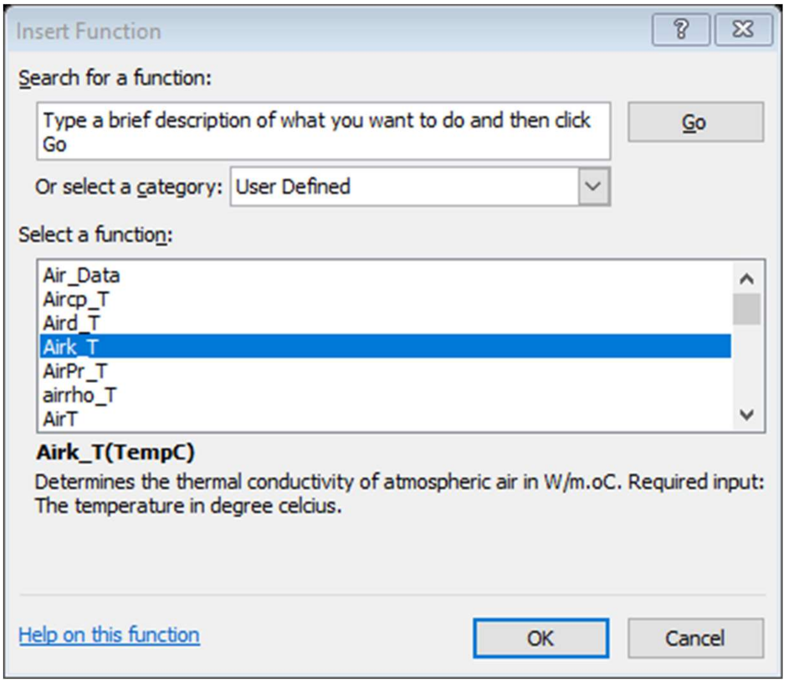


Figure 3.29. Thermax functions listed alphabetically in the User Defined category

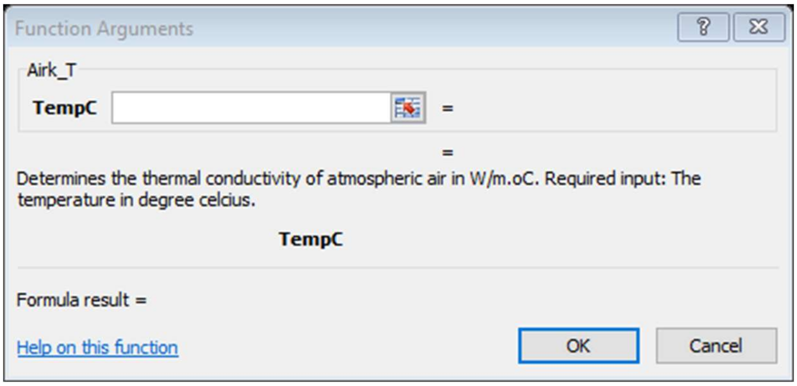


Figure 3.30. The Function Arguments box for the "Airk_T" function

Figure 3.30 shows that this function has one input parameter, which is the temperature in °C "TempC", and gives a brief description of its intended use. Let us use the function to determine the thermal conductivity for air at 25°C. Fill the form by entering the value of the temperature, 25, as shown in Figure 3.31. Note that the formula ribbon now shows the formula in cell **B2**, which is **"=Airk_T(25)"**, and the function form shows the calculated value of *k*, which is 0.02551 W/m.°C. When you press the "OK" button, this value will appear in the cell **B2**. You can check this value with that given in Table A.1 of Appendix A.

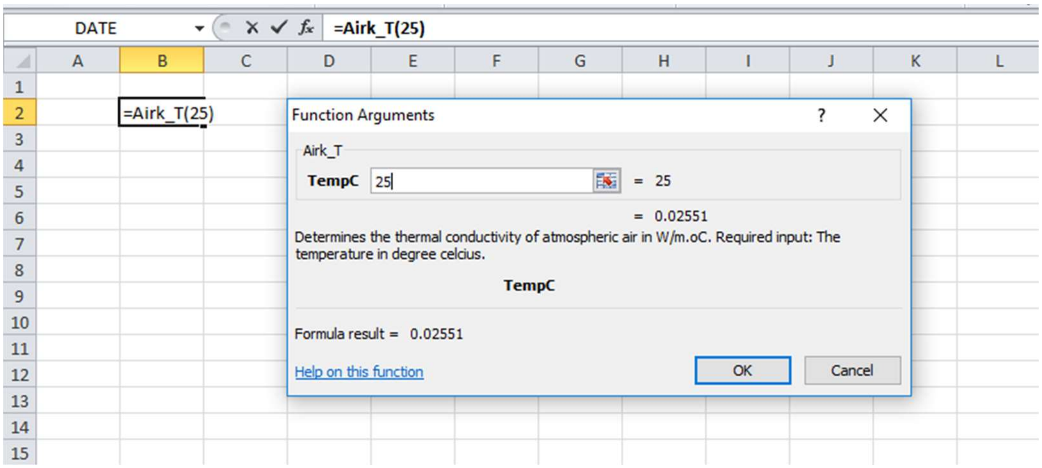


Figure 3.31. Using the function "Airk_T" to determine the thermal conductivity of atmospheric air at 25°C

In certain situations, the confinement of Excel's formula in one cell becomes too restrictive for developing the analytical model. This situation arises, for example, when an iterative process involves a non-linear equation such as the Colebrook-White equation or the Soave-Redlich-Kwong equation of state. In this case, a UDF is needed solve the nonlinear equation and return the result to Excel's formula like a built-in function. For such a case, Thermax provides a Newton-Raphson solver for nonlinear equations in addition to its property functions. Appendix B shows how to use this tool. Appendix B also describes two interpolation functions provided by Thermax for tabulated data which are useful for including additional fluid properties or other tabulated data needed in a thermofluid analysis.

3.4. Closure

This chapter introduces the three auxiliary components of the Excel-based modelling platform used in this book for thermofluid analyses; Solver, VBA, and Thermax. Starting with Solver, the chapter gives examples of using the three solution methods it provided to deal with different types of problems and highlights certain situations that require their default settings to be modified. The chapter also shows how VBA can be used for developing user-defined functions not provided by Excel and explains the procedure for installing Thermax and using its functions in Excel formulae. Appendix B demonstrates the use of the two interpolation functions for tabulated data provided by Thermax and its Newton-Raphson solver for solving nonlinear equations. Thermax also provides a couple of other useful functions.

References

[1] FrontlineSolvers, internet: http://www.Solver.com/ (Last accessed January 18, 2023).
[2] L.S. Lasdon, R.L. Fox, M.W. Ratner, Nonlinear optimization using the generalized reduced gradient method, Revue Française d'Automatique, Informatique et

Recherche Opérationnelle, tome 8, V3 (1974), p. 73-103. Available at: http://www.numdam.org/article/RO_1974__8_3_73_0.pdf
[3] Wikkipedia, https://en.wikipedia.org/wiki/Gradient_method
[4] Wikkipedia, https://en.wikipedia.org/wiki/Evolutionary_algorithm
[5] FrontlineSolvers, internet: https://www.solver.com/excel-solver-change-options-all-solving-methods
[6] Y. A. Cengel and M. A. Boles. *Thermodynamics an Engineering Approach*, McGraw-Hill, 7th Edition, 2007
[7] J. Walkenbach, *Excel 2013 Power Programming with VBA*, Wiley Publishing, Inc. 2013.
[8] J. L. Latham, *Programming in Microsoft Excel VBA, An Introduction*, 2008. Internet: http://ies.fsv.cuni.cz/default/file/download/id/21101 (Last accessed November 29, 2015).
[9] http://www.fontstuff.com/vba/vbatut01.htm (Last accessed November 26, 2017).
[10] M.M. El-Awad, *Thermax: Thermodynamic Analyses of Energy-Utilisation Systems using Excel* - https://www.academia.edu/24381392/Thermodynamic_Analyses_of_Energy_Utilisation_Systems_Using_Excel
[11] S.C. Chapra and R.P. Canale, *Numerical Methods for Engineers*, 6th Edition, McGraw Hill, 2010.

Exercises

1. A system of algebraic equations can be expressed as follows:

$$\begin{bmatrix} 70 & 1 & 0 \\ 60 & -1 & 1 \\ 40 & 0 & -1 \end{bmatrix} \begin{Bmatrix} a \\ b \\ c \end{Bmatrix} = \begin{Bmatrix} 636 \\ 518 \\ 307 \end{Bmatrix}$$

Solve the system of equations by using Solver to determine the values of the three unknowns a, b, and c. This exercise is based on Example 9.11 in Chapra and Canale [11]. The answer is: $a = 8.5941$, b=34.4118, and $c = 36.7647$.

2. Draw a line chart with Excel to show the variation of the following function in the range $0 \leq x \leq 4$:

$$f(x) = 2 \sin x - x^2/10$$

Use Solver to find the maximum of the function in the specified range. Based on Example 13.1 in Chapra and Canale [11]. The answer is: $f(x) = 1.7757$ at x =1.4276.

3. The curve shown in Figure 3.P3 is a plot of the function:

$$f = e^{\theta} \sin(2\theta)$$

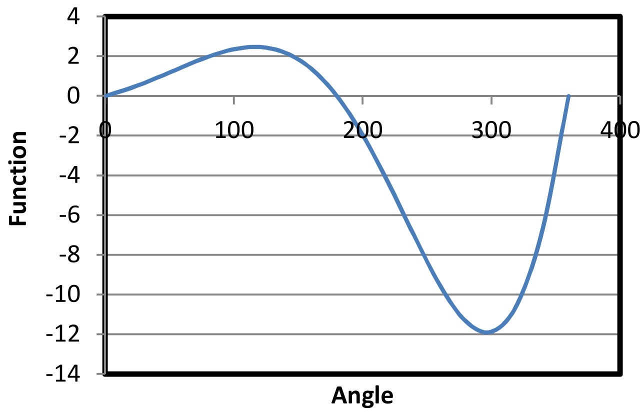


Figure 3.P3 A composite function

Use Solver to find:

a) The minimum value of the function and the corresponding angle
b) The maximum value of the function and the corresponding angle
c) The angle at which value of the function equals 4.0

4. Using the Excel sheet developed to solve Example 3-1 by the **GRG Nonlinear** method, study the effect of using central-difference approximation of derivatives instead of the default forward-difference approximation on the solution.
5. Consider the following set of simultaneous nonlinear equations:

$$x^2 + xy = 10 \qquad \text{(A)}$$

$$y + 3xy^2 = 57 \qquad \text{(B)}$$

To solve the system with Solver, rearrange the equations as follows:

$$u(x, y) = x^2 + xy - 10 = 0 \qquad \text{(C)}$$

$$v(x, y) = y + 3xy^2 - 57 = 0 \qquad \text{(D)}$$

Create two cells (**B1** and **B2**) to hold initial guesses for x and y. Enter the function values themselves, $u(x, y)$ and $v(x, y)$ into two other cells (**B3** and **B4**). The initial guesses may result in function values of u and v that are far from zero. Determine the sum of the function squares, i.e. $u^2 + v^2$, and store it in cell **B5**. Use Solver to find the values of x and y in cells **B1** and **B2** (the Changing cells) that make the value in cell **B5** (the objective cell) equal to zero. Using this procedure, find the roots of the above system starting with initial guesses of $x = 1$ and $y = 3.5$.

This exercise is based on Example 7.5 in Chapra and Canale [11]. The correct pair of roots are x=2 and y=3.

6. The volume V of liquid in a spherical tank of radius r is related to the depth h of the liquid by:

   $V = \pi h^2(3r - h)/3$

   Using VBA, develop a user-defined function that determines h at any given values of r [m] and V [m^3]. Check your function with r=1 m and V = 0.5 m^3. Answer: h = 0.431 m.
7. Extend the UDF developed in Section 3.2 for determining the specific heat for air so that it can be used for other ideal gases as well. Note that in this case, the function will have another input parameter which is the name of the gas.
8. Using suitable formulae for the thermodynamic properties of superheated steam, develop user-defined functions with VBA for determining the specific enthalpy and entropy of superheated steam from its pressure and temperature.
9. Using suitable formulae for the thermodynamic properties of superheated refrigerant R134a, develop user-defined functions with VBA for determining properties, e.g., enthalpy and entropy, of superheated R134a from its temperature and pressure.
10. Using the data for properties of air at atmospheric pressure given in Appendix A, Table A.1, develop an Excel sheet that can be used to determine the kinematic viscosity of air at any given temperature in the range 200 – 1000K by using:

    a. The trendline feature of Excel
    b. The linear-interpolation function (**Interpl**) provided by Thermax.

11. Develop a VBA function to determine the friction factor from the Colebrook-White equation and use it with the NRM solver provided by Thermax (refer to Appendix B) to determine the frictional losses (h_f) in a circular pipe that carries air at 20°C with the following data:

    D = 25 cm, L = 150 m, V = 7 m/s, k_s = 0.045 mm.

4

Iterative solutions

Iterative solutions are needed in thermofluid analyses for a number of reasons. This chapter gives examples of such analyses and shows how the iterative solution can be handled by using the two iterative tools provided by Excel; the Goal Seek command and Solver. While the Goal Seek command can deal with the simple type of iterative solutions that involve a single parameter, Solver is needed for those involving multiple variables and requiring certain constraints to be satisfied by the iterative solution. When the analytical model involves a nonlinear equation, such as the Colebrook-White equation, it becomes difficult to use only Goal Seek and Solver. For such problems, the chapter shows how a Newton-Raphson solver written with VBA can be used to deal with the nonlinear equation separately, leaving the main iteration loop to Goal Seek or Solver.

4.1. Iterative solutions by using Goal Seek

Despite of its simplicity, the Goal Seek command can be used to solve most thermofluid problems that require iterative solutions. This section presents three examples that demonstrate its use for typical analyses that require iterative solutions in fluid-dynamics, thermodynamics, and heat-transfer.

4.1.1. Type-2 and type-3 pipe-flow analyses

The frictional head loss (h_f) in a pipe depends on a number of factors that characterise the pipe itself as well as the fluid being transported. For a straight pipe with no fittings carrying a viscous Newtonian and incompressible fluid, the frictional head loss is determined by the following Darcy-Weisbach equation:

$$h_f = f \frac{L}{D} \frac{V^2}{2g} \tag{4.1}$$

Where f is the Darcy friction factor, L the length of the pipe, D its diameter, V the fluid velocity, and g the gravity acceleration constant. The friction factor f can be obtained from Equation (1.4) if the flow is laminar and from Equation (1.6) or (1.7) if it is turbulent. Practical pipe-flow problems described by Equation (4.1) can be divided into three types [1]:

1. Type-1 problems – require the determination of h_f when both the pipe's diameter and fluid velocity (or flow rate) are known.
2. Type-2 problems – require the flow rate to be determined for specified values of h_f and pipe diameter.
3. Type-3 problems – require the pipe diameter to be determined for given values of h_f and flow rate.

Type-1 problems can be solved in a straight-forward manner by using Equation (4.1) to determine the friction head loss. However, both type-2 and type-3 problems require iterative solutions because the Reynolds number and, therefore, the friction factor, f, cannot be determined without knowing D or V. For type-2 problems (i.e. unknown flow rate), the iterative procedure can be avoided by using extended Moody diagrams that require the determination of the following dimensionless parameter [2]:

$$\mathrm{Re}\, f^{0.5} = \frac{D^{1.5}}{\nu}\left(\frac{2gh_f}{L}\right)^{0.5}. \tag{4.2}$$

Apart from the inaccuracy of visual chart interpolation, the procedure is difficult to adopt in optimisation or parametric analyses. By using the Goal Seek command, both type-2 and type-3 problems can be solved more accurately. The following example shows how Goal Seek can be used to solve a type-3 problem. It is based on Example 8.4 in Cengel and Cimbala [1].

Example 4-1. Solution of type-3 pipe flow problems
Heated air at 1 atm and 35°C is to be transported in a 150-m-long circular plastic duct (ε=0.045 mm) as shown on Figure 4.1 at a rate, Q, of 0.35 m^3/s. If the head loss in the duct is not to exceed 20 m, determine the smallest required diameter for the duct.

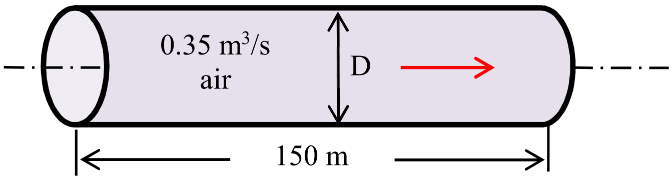


Figure 4.1. Schematic for Example 4-1 (adapted from Cengel and Cimbala [1])

Solution with Goal Seek
The problem can be solved by calculating the friction head loss at different diameters of the duct and then selecting the diameter that gives the required head loss which is 20 m. The iterative solution proceeds as follows:

1. Select a diameter for the inner pipe (D)
2. Calculate the velocity of the hot air (V), $V=Q/A$, $A=\pi D^2/4$
3. Calculate the Reynolds number in the pipe (Re), $\mathrm{Re}=VD/\nu$
4. Calculate the friction factor (f) using Equation (1.4) or (1.6)
5. Calculate the friction head loss (h_f) from Equation (4.1)
6. If $h_f \neq 20$ m, repeat steps 1 to 5

Figure 4.2 shows the Excel sheet developed for this example which is divided into three parts: (i) problem data (ii) calculations, and (iii) results. The data part shows the information given in the question. The value of the kinematic viscosity of air at 35°C (ν = 1.655x10^{-5} m^2/s) was obtained from Cengel and Cimbala [1] and fixed throughout the calculations. Cell-labelling is applied in the formulae and Figure 4.2 reveals the formulae used in each cell of the calculations part. Note the If-function in cell **F10**.

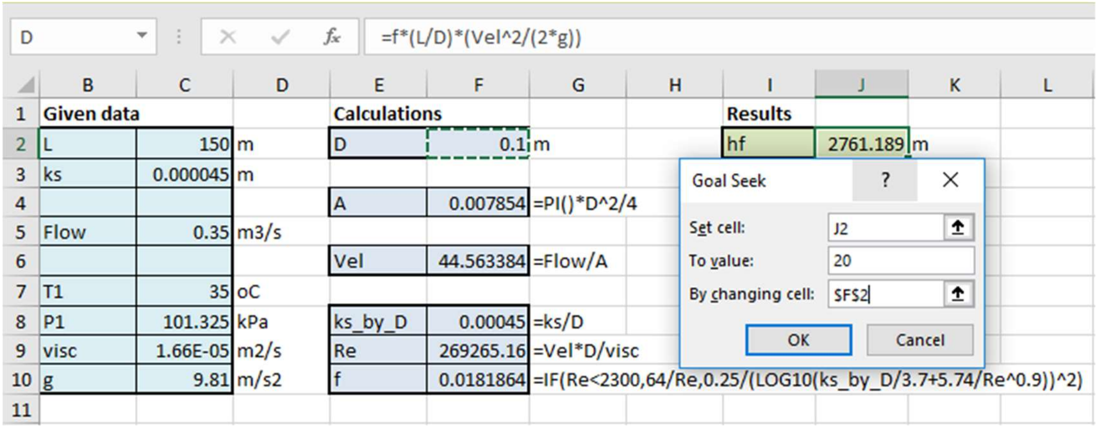


Figure 4.2. Excel sheet and Goal Seek set-up for Example 4-1

As Figure 4.2 shows, for an assumed duct diameter of 0.1 m the friction head loss exceeds 2761 m. Figure 4.2 also shows the completed Goal Seek dialog box that requires the command to change the diameter in cell **F2** and iterate until the friction head loss in cell **J2** attains the required value of 20 m. Figure 4.3 shows the answer found by Goal Seek, which is $D \geq 0.27$ m. This answer agrees with that given by Cengel and Cimbala [1]. A similar procedure can be used to solve type-2 flow problems by iterating over the flow rate instead of the diameter.

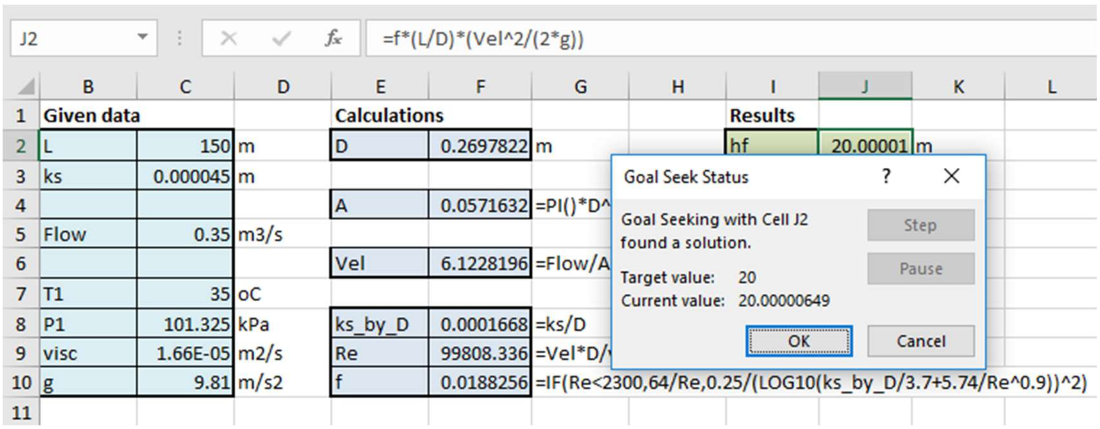


Figure 4.3. Goal Seek solution for Example 4-1

4.1.2. Thermodynamic analyses involving gas mixtures

Without the usual idealisations and simplifications applied in thermodynamic analyses, most of these analyses, if not all, would require iterative solutions. A commonly used thermodynamic approximation is treating air as a pure gas even though it is a mixture of nitrogen, oxygen, and water vapour with small traces of other gases. Computer-aided analyses with fluid property functions such as those provided by Thermax enable more realistic models to be used by treating air as a mixture of gases instead of a single gas. However, if the temperature of the gas mixture is not known and has to be determined, as in the analyses of combustion processes, an iterative solution will be required. The following example shows how the problem can be solved by using Goal Seek.

Example 4-2. Constant-pressure expansion of air

Figure 4.4 shows a piston-cylinder device that contains a mixture of oxygen and nitrogen. Initially at 100 kPa, 330K the gas mixture contains 21% oxygen and 79% nitrogen by volume and occupies 0.1 m^3. 50 kJ of heat is transferred to the gas causing it to expand at constant pressure. Neglecting heat losses to the surroundings and treating oxygen and nitrogen as ideal gases, determine the final temperature of the gas inside the cylinder.

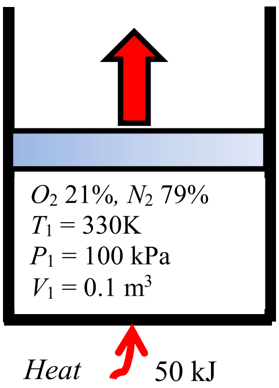


Figure 4.4. Schematic diagram for Example 4-2

The analytical model

By treating air as a single pure gas and using a constant specific heat, c_p, of 1.005 kJ/kg, the final temperature, T_2, can easily be determined from:

$$T_2 = T_1 + Q/(c_p m) \tag{4.3}$$

Where, T_1 is the initial temperature, Q is the amount of heat added, and m is the total mass. Substituting the values of T_1, Q, c_p and m in Equation (4.3) gives T_2 = 803.13K. To solve the problem by treating air as a mixture of two gases, the solution procedure applies the first-law of thermodynamics to the expansion process. Neglecting changes in kinetic and potential energies, for a constant-pressure expansion the first law reads:

$$Q = m_{O2}\left(h_{2\_O2} - h_{1\_O2}\right) + m_{N2}\left(h_{2\_N2} - h_{1\_N2}\right) \tag{4.4}$$

Where, m_{O2} and m_{N2} are the masses of oxygen and nitrogen in the device, h_{1_O2} and h_{2_O2} are enthalpies of oxygen at the initial and final temperatures, respectively, and h_{1_N2} and h_{2_N2} are the corresponding enthalpies for nitrogen. The correct value of the final temperature is that at which the amount of heat added as obtained from Equation (4.3) is equal to the given value, which is 50 kJ. The values of enthalpy for O_2 and N_2 in Equation (4.3) can be determined by using the relevant Thermax function for ideal gases, **Gash_TK**, and the masses m_{O2} and m_{N2} can be obtained from the ideal-gas law using the corresponding partial pressures as follows:

$$m_{O2} = \frac{(0.21P_1)V_1}{R_{O2}T_1} \tag{4.5}$$

$$m_{N2} = \frac{(0.79\,P_1)V_1}{R_{N2}T_1} \tag{4.6}$$

Where R_{O2} and R_{N2} are the gas constants for oxygen and nitrogen, which are 0.2598 kJ/kg.K and 0.2968 kJ/kg.K, respectively.

Solution with Goal Seek

Figure 4.5 shows the Excel sheet developed for this example. The data part includes the initial pressure, temperature, and volume of the gas mixture together with the mole fractions and gas constants of oxygen and nitrogen. The initial partial pressures of oxygen and nitrogen, P1_O2 and P1_N2, are calculated from the total initial pressure (P_1) and the respective volume fractions, y_O2 and y_N2, as shown in cells **E2** and **E3**, respectively. The masses of the two gases in the mixture (m_O2 and m_N2) are calculated in cells **E5** and **E6**, respectively, and the total mass (m_total) in cell **E8**.

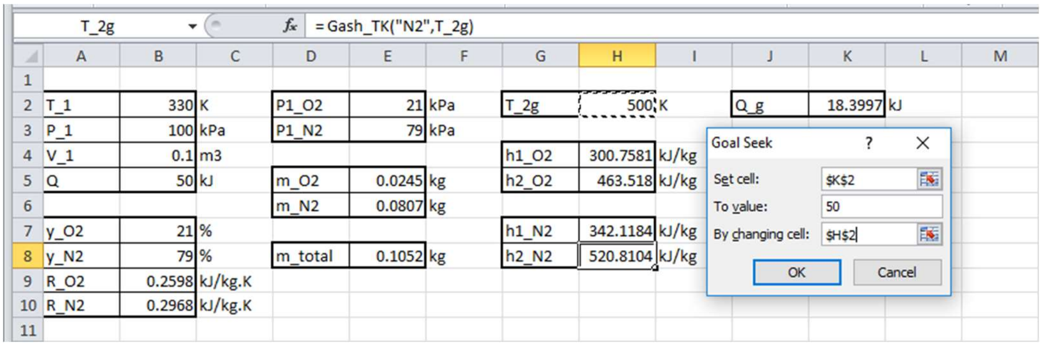
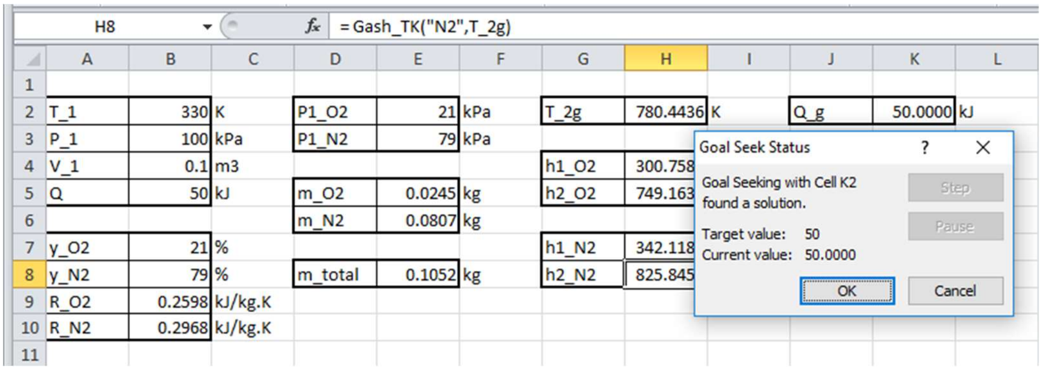


Figure 4.5. The Excel sheet developed for Example 4-2 by using Thermax functions

Starting with a guessed value for the final temperature, T_2g, which is 500K, the initial and final enthalpies of oxygen and nitrogen are determined by using Thermax function **Gash_TK** at the corresponding temperatures. Equation (4.3) is then used to determine the total amount heat added in the process (Q_g). With the guessed final temperature, Equation (4.3) determined the total amout of heat as 18.4 kJ, which is less than the actual values of 50 kJ. To find the appropriate final temperature, the guessed temperature T_2g has to be adjusted by Goal Seek so that the value of Q_2g equals 50 kJ. Figure 4.5 shows the required Goal Seek set-up and Figure 4.6 shows the solution obtained by Goal Seek, which is 780.444K. This value differs from that obtained by treating the gas mixture as a single gas by 22.69K and the deviation between the two values is expected to increase as the amount of heat added and, therefore, T_2 increases.



	A	B	C	D	E	F	G	H	I	J	K	L
1												
2	T_1	330	K	P1_O2	21	kPa	T_2g	780.4436	K	Q_g	50.0000	kJ
3	P_1	100	kPa	P1_N2	79	kPa						
4	V_1	0.1	m3				h1_O2	300.758				
5	Q	50	kJ	m_O2	0.0245	kg	h2_O2	749.163				
6				m_N2	0.0807	kg						
7	y_O2	21	%				h1_N2	342.118				
8	y_N2	79	%	m_total	0.1052	kg	h2_N2	825.845				
9	R_O2	0.2598	kJ/kg.K									
10	R_N2	0.2968	kJ/kg.K									
11												

Figure 4.6. Goal Seek solution for Exampe 4-2 by using Thermax functions

4.1.3. Convection heat-transfer analyses

Like the friction factor (f) in pipe-flow analyses, the convection heat-transfer coefficient (h) is not a physical property of the fluid itself but that of the flow and, as for the case of type-2 and type-3 flow problems, convection heat-transfer analyses frequently involve iterative solutions. The following example shows how Excel's Goal Seek command can be used for such analyses. The example is based on Example 10.1 in Holman [3].

Example 4-3. Overall heat-transfer coefficient for pipe in air

Hot water at 98ºC flows through a 2-in schedule 40 horizontal steel pipe (k =54 W/m·◦C) and is exposed to atmospheric air at 20ºC as shown in Figure 4.7.

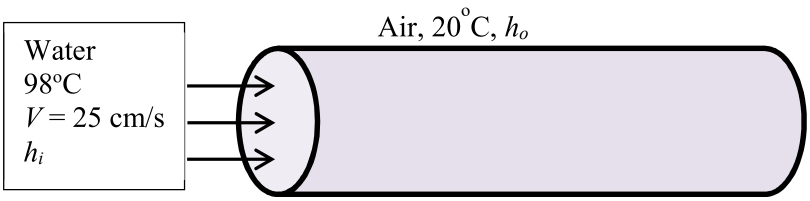


Figure 4.7. Schematic for Example 4-3 (adapted from Holman [3])

If the water velocity is 25 cm/s, calculate:

(a) the rate of heat-transfer through the pipe,
(b) the temperatures at the inside and outside surfaces of the pipe, and
(c) the overall heat-transfer coefficient based on the outer area of the pipe.

Properties of water at 98ºC are: ρ = 960 kg/m^3, μ = 2.82 x 10^{-4} kg/m.s, k = 0.68 W/m.ºC, Pr = 1.76. For a 2-in schedule 40 pipe, D_i = 5.25 cm and D_o = 6.033 cm.

The analytical model

Using the thermal-resistance concept, the rate of heat-transfer through the pipe, Q, is given by:

$$Q = \left(T_w - T_\alpha\right) / R_{th} \tag{4.7}$$

Where T_w and T_∞ are the water temperature and air-temperature, respectively, and R_{th} is the total thermal resistance that consists of the thermal resistances due to heat-transfer by convection inside the pipe (R_i), by conduction through the steel pipe (R_p), and by convection outside the pipe (R_o). The three resistances are given by [3]:

$$R_i = \frac{1}{A_l h_i} \tag{4.8}$$

$$R_p = \frac{\ln\left(D_i / D_o\right)}{2\pi k} \tag{4.9}$$

$$R_o = \frac{1}{A_o h_o} \tag{4.10}$$

Where A_i and A_o are the inside and outside areas of the pipe and h_i and h_o are the corresponding heat-transfer coefficiens. The internal heat-transfer coefficien h_i is determined from the corresponding Nusselt number (Nu) by:

$$h_i = Nu \frac{k_w}{D_i} \tag{4.11}$$

Where, k_w is the thermal conductivity of water. The Nusselt number itself is determined from emprirical equations depending on the type of the flow, i.e., natural or forced, laminar or turbulent. For the turbulent forced internal flow (to be confirmed later), Nu is obtained from the Dittus-Boelter equation, Equation (1.13):

$$Nu = 0.023\,\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.4} \tag{4.12}$$

Where, Re and Pr are the Reynolds number and Prandtl number, respectively. For the external flow, Holman [3] used the following simplified equation for free laminar convection from a horizontal pipe to air at atmospheric pressure:

$$h_o = 1.32\left(\frac{T_o - T_\infty}{D_o}\right)^{1/4} \tag{4.13}$$

Both R_i and R_p can be determined directly from the given data, but R_o depends on h_o which cannot be determined directly since T_o is not known. Therefore, the problem has to be solved by adopting an iterative approach by assuming a value for T_o based on which h_o is determined and, consequently, Q. The value of Q thus obtained can be used to calculate corresponding values for T_i and T_o from:

$$T_i = T_w - Q.R_i \tag{4.14}$$

$$T_o = T_i + Q.R_p \tag{4.15}$$

If the guessed value for T_o is correct, then it will be the same as that obtained from Equation (4.15). Otherwise, a new guess for T_o has to be made repeatedly until this condition is met. Once this is achieved, the overall heat-transfer coefficient (U_o) based on the outside area (A_o) can be obtained from:

$$U_o = \frac{1}{A_o\left(R_i + R_o + R_p\right)} \tag{4.16}$$

Solution with Goal Seek

The Excel sheet developed for this example is shown in Figure 4.8. The properties of the pipe, water, and air are entered in the data part on the left side of the sheet. The cells are labelled and the figure shows the formulae used in the calculations. The calculations part at the central part of the sheet starts with a guessesd value for the pipe's outside temperature (T_og) of 50ºC. Based on this value, the sheet determines the outside heat-transfer coefficient (h_o) from Equation (4.13) and the thermal resistance associated with it (R_o) from Equation (4.10). Following the analytical model described above, the sheet determines the three thermal resistances (R_i, R_p, and R_o), and then calculates the rate of heat-transfer (Q), inside temperature (T_i), outside temperature (T_o), and overall-heat transfer coefficient (U).

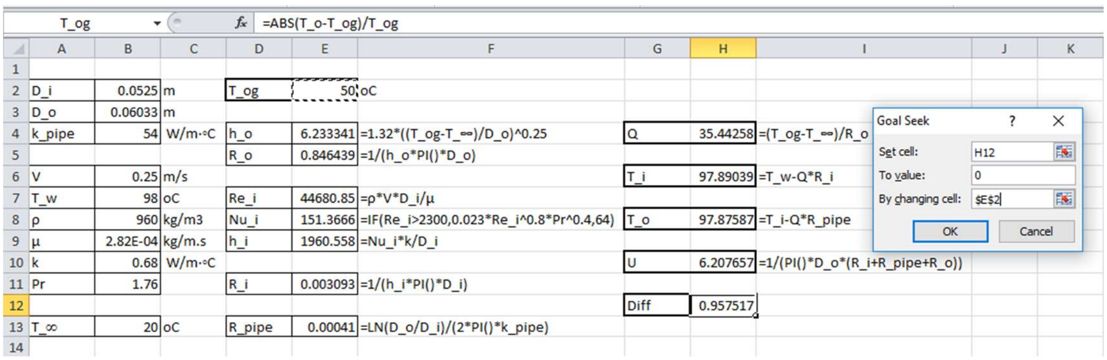


Figure 4.8. Excel sheet developed for Example 4-3

As Figure 4.8 shows, the value of T_o calculated from Equation (4.15) is 97.876°C, which is different from the initially guessed value (T_og = 50°C). The formula bar reveals the formula entered in cell **H12** that calculates the difference between the calculated exit temperature (T_o) and the guessd value (T_og) as a fraction of T_og. The exit temperature that makes the difference vanishes can be found by using the Goal Seek command and Figure 4.8 shows the required set-up. The solution found by Goal Seek is shown on Figure 4.9. Table 4.1 that comapres the present results with those given by Holman [3] confirms the accuracy of the iterative solution with Goal Seek.

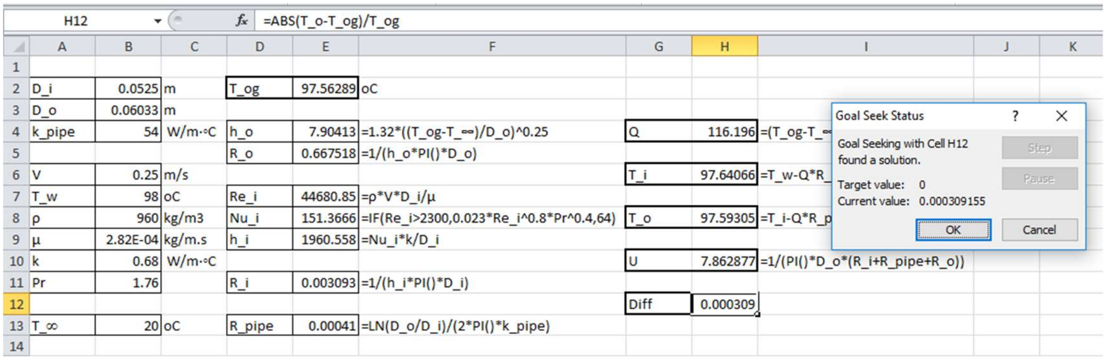


Figure 4.9. Solution obtained by Goal Seek for Example 4-3

Table 4.1. Comparison of the presnt Goal Seek solution with that given by Holman [3]

	Holman [3]	Goal Seek
T_i	97.65	97.64
T_o	97.6	97.59
h_i	1961.0	1960.56
h_o	7.91	7.90
U_o	7.87	7.86

4.2. Constrained iterative solutions with Solver

Compared to Goal Seek, Solver offers greater flexibility for dealing with iterative solutions by allowing for multiple changeable cells and constraints to be imposed on the solution. This section illustrates the need for these additional features in thermofluid analyses by two examples from the areas of fluid-dynamics and thermodynamics.

Example 4-4. Determining the maximum water flow rate to avoid cavitation

Water at 20°C (γ = 9810 N/m^3 and ν =1.006x10^{-6} m^2/s) is to be pumped from a large reservoir via a pump-pipe system as shown in Figure 4.10. The pump is positioned vertically at a level which is 9 m above the surface of the reservoir and horizontally at 1 m from the vertical section of the pipe. The pipe is made of commercial steel pipe (ε = 0.046 mm) and has a 2″ nominal diameter.

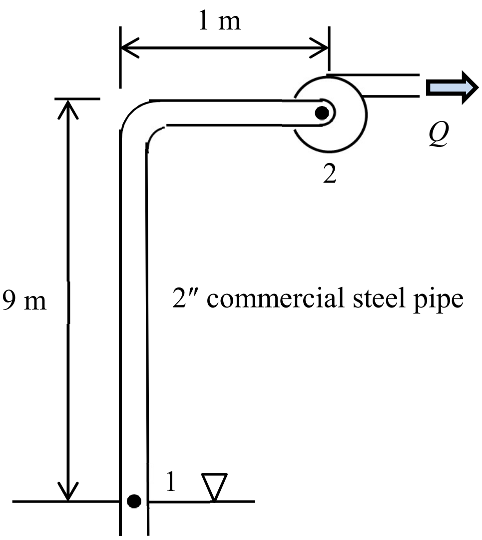


Figure 4.10. Schematic for the pump-pipe system in Example 4-4

Determine the maximum allowable water flow rate (Q) such that:

1. The water velocity (V) is to be in the range 1.4-2.8 m/s for economic considerations.
2. The pressure at the pump inlet must be greater than the saturation pressure of water at 20ºC, which is 2.338 kPa, to avoid cavitation.

These two conditions are important design considerations for pump-pipe systems. The example, which is basically a type-2 pipe flow problem, is based on a similar example given by Schumack [4].

The analytical model

The energy equation between the pipe inlet (point 1) and the pump inlet (point 2) is:

$$\frac{p_1}{\gamma} + Z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + Z_2 + \frac{V_2^2}{2g} + h_f \tag{4.17}$$

Where γ stands for the specific weight of water, z for the elevation, V for the water velocity, g for the gravitational acceleration, and h_f for the friction loss in the pipe. For a large reservoir $V_1 \approx 0$. Taking point 1 as a reference, i.e. $Z_1 = 0$, and noting that the water velocity in the pipe is uniform, i.e. $V_2 = V_1 = V$, the energy equation reduces to:

$$p_2 = \gamma\left(\frac{p_1}{\gamma} - Z_2 - \frac{V^2}{2g} - h_f\right) \tag{4.18}$$

The velocity V is related to the pipe diameter (D) and water flow rate (Q) as follows:

$$V = 4Q / \pi D^2 \tag{4.19}$$

Neglecting minor losses, the friction loss can be calculated from the Darcy-Weisbach equation, Equation (4.1), which needs an auxiliary formula to determine the friction factor (f) depending on whether the flow is laminar or turbulent.

Solution with Solver

Figure 4.11 shows the Excel sheet developed for this example. The data part on the left side stores the problem data such as the diameter, roughness, and length of the pipe, etc. The central part stores a guessed value for the water velocity (V=1.0 m/s) in cell **E2**. Based on the guessed water velocity, the sheet performs the necessary calculations according to the analytical model given above. Figure 4.11 reveals the formulae used in these calculations. Note that an **IF**-statement is used to calculate the friction factor (f) depending on the value of the Reynolds number (Re). Cell **E6** calculates the friction loss (hf). Based on the calculated value of friction loss, the pressure at point 2 (P_2) is calculated from Equation (4.18) and stored in cell **E7**. The right side of the sheet contains the single cell **I2** that determines the flow rate (Q) and the formula in this cell is shown in the formula bar.

Q | fx =0.25*PI()*D^2*V

	A	B	C	D	E	F	G	H	I	J	K
1											
2	D	0.05252	m	V	1.00	m/s		Q	0.0021664	m3/s	
3	ε	0.000046	m								
4	L	10	m	Re	52520	=V*D/v					
5	Z_2	9	m	f	0.023651	=IF(Re<2000,64/Re,0.25/(LOG10(ε/3.7/D+5.74/Re^0.9))^2)					
6	P_1	100	kPa	hf	0.229523	=f*L/D*V^2/(2*gc)					
7	v	1.00E-06	Pa.s	P_2	8.958382	=(P_1*1000/γ-Z_2-V^2/(2*gc)-hf)*γ/1000					
8	γ	9810	N/m3								
9	gc	9.81	m/s2								
10											

Figure 4.11. The Excel sheet developed for Example 4-4

Based on the assumed water velocity of 1.0 m/s, the calculated values of hf and P_2 are 0.2295 m and 8.958 kPa, respectively. Since the pressure at point 2 is higher than the minimum desired level of 2.338 kPa, while the water velocity (V) is less than the minimum economic value of 1.4 m, there is room to increase the flow rate. The task can be left to Solver and Figure 4.12 shows the set-up that requires Solver to maximise the value of the flow rate Q while satisfying the three constraints shown in the figure. The first constraint on the iterative solution requires the value of P_2 in cell **E7** to be higher than or equal to 2.338 kPa. The two other constraints are to satisfy the limits on the water velocity imposed by economic limits, i.e., 1.4 m $\leq V \leq$ 2.8 m.

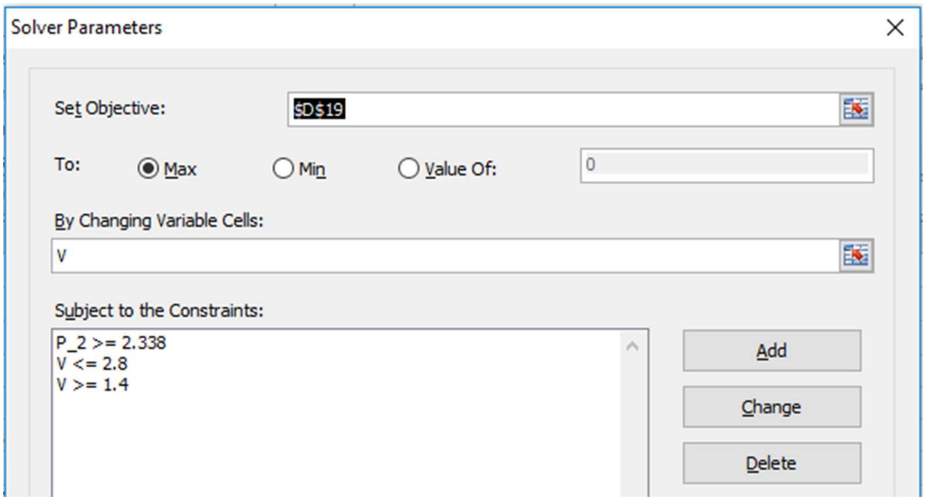


Figure 4.12. Solver parameters dialog box for Example 4-4

Pressing the "Solve" button will trigger Solver to search for the solution. The solution found by the **GRG Nonlinear** method is shown in Figure 4.13. The value determined for the water velocity is 1.90 m/s which lies within the limits imposed by the economic constraint. The pressure at point 2 is equal to 2.338 kPa which is the minimum pressure level required to prevent cavitation. Therefore, the corresponding flow rate of 0.00413 m^3/s is the maximum flow rate to be recommended.

Q =0.25*PI()*D^2*V

	A	B	C	D	E	F	G	H	I	J	K
1											
2	D	0.05252	m	V	1.90	m/s		Q	0.004125	m3/s	
3	ε	0.000046	m								
4	L	10	m	Re	100001.5	=V*D/ν					
5	Z_2	9	m	f	0.021901	=IF(Re<2000,64/Re,0.25/(LOG10(ε/3.7/D+5.74/Re^0.9))^2)					
6	P_1	100	kPa	hf	0.770567	=f*L/D*V^2/(2*gc)					
7	ν	1.00E-06	Pa.s	P_2	2.338005	=(P_1*1000/γ-Z_2-V^2/(2*gc)-hf)*γ/1000					
8	γ	9810	N/m3								
9	gc	9.81	m/s2								
10											

Figure 4.13. Solver's solution for Example 4-4

Example 4-5. Restrained expansion of air inside a piston-cylinder device

Figure 4.14 shows a piston-cylinder device that initially contains 0.05 m^3 of air at 200 kPa and 317K. At this state, a linear spring is touching the piston but exerting no force on it. 72.7 kJ of heat is then transferred to the air causing the piston to rise and compress the spring. The cross-sectional area of the piston is 0.25 m^2 and the spring's constant (k) is 150 kN/m, Air can be treated as an ideal gas with a specific heat at constant volume (c_v) that varies linearly with the temperature according to the formula:

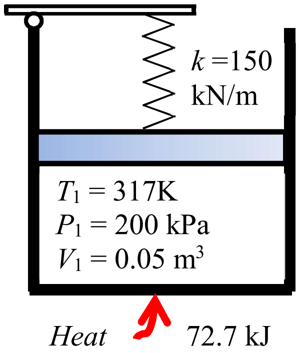


Figure 4.14. Schematic and pressure-volume diagrams for Example 4-5 (adapted from Cengel and Boles [5])

$$c_v = 0.645+0.0002T \tag{4.20}$$

Where T is the temperature in K and c_v is in kJ/kg.K. Determine the final volume, pressure, and temperature of the air inside the cylinder after the heat-addition process.

Comment

This example is based on Example 4-4 given by Cengel and Boles [5]. However, unlike the present example, Cengel and Boles [5] specified the final volume to be 0.1 m^3 instead of specifying the amount of heat added. When the final volume (or final pressure) is given, the problem can be solved in a straightforward manner without iteration. However, in the present example T_2, V_2, and P_2 at the final state depend on the amount of heat added. Another factor that makes the present example more difficult than the one given by Cengel and Boles [5] is that the specific heat c_v for air is not treated as constant but varies according to Equation (4.20). The specific value of 72.7 kJ given in this example has been chosen such that the final volume will be 0.1 m^3 as specified by Cengel and Boles [5] so that the present final pressure on the piston and total work will be comparable to their corresponding values even though the data of the two examples are different.

Like Example 4-2, the problem can be solved by using the first-law of thermodynamics together with the ideal-gas law, but the variation of the specific heat with temperature makes it necessary to adopt an iterative solution. Moreover, the addition of the linear spring in this example introduces a new factor, which is the variation of pressure with air expansion. Since the present iteration process involves both the temperature and the volume (or pressure), the Goal Seek command cannot be used. Therefore, this example requires Solver to start the iterative procedure with assumed values for both the final temperature and the final volume, referred to as T_2^* and V_2^* respectively.

The analytical model

The final pressure P_2 is given by:

$$P_2 = P_1 - \frac{k\Delta x}{A} \tag{4.21}$$

Where A is the base area of the piston and Δx is the reduction in the spring's length given by:

$$\Delta x = \frac{V_2^* - V_1}{A} \tag{4.22}$$

The total work (W), i.e., the summation of the air expansion work and the work done against the spring, can now be obtained from:

$$W = \frac{(P_1 + P_2)}{2}\left(V_2^* - V_1\right) \tag{4.23}$$

The final temperature can be determined by applying the first-law of thermodynamics to the piston-cylinder device as a closed system:

$$Q - W = m(u_2 - u_1) = m\bar{c}_v(T_2 - T_1) \tag{4.24}$$

Where Q is the amount of heat added, u is the internal energy, m is the mass of air inside the cylinder, and $\bar{c}_v$ is the average specific heat of air at constant volume. The mass and specific heat of air can be obtained from:

$$m = P_1 V_1 / RT_1 \tag{4.25}$$

$$\bar{c}_v = 0.645 + 0.0002\left(T_1 + T_2^*\right)/2 \tag{4.26}$$

Rearranging Equation (4.24), the final temperature T_2 is given by:

$$T_2 = T_1 + \frac{Q - W}{m\bar{c}_v} \tag{4.27}$$

Note that both W and $\bar{c}_v$ depend on the assumed values of T_2^* and V_2^*. Using the values obtained for T_2 and P_2, the final volume V_2 can be determined from the ideal-gas law:

$$V_2 = mRT_2 / P_2 \tag{4.28}$$

If the initially guessed volumes of T_2^* and V_2^* are correct, then they will be the same as T_2 and V_2 obtained from Equation (4.27) and Equation (4.28), respectively. Otherwise, new values for T_2^* and V_2^* have to be used until the differences between the calculated and guessed values become negligibly small. This multi-variable iterative process can be performed with Solver as shown below.

Solution with Solver

Figure 4.15 shows the Excel sheet developed for this example which reveals the formulae used in it. The left side of the sheet accommodates the problem data. The calculations part starts by assumed values for the final temperature (T_2g = 500K) and final volume V_2g = 0.15 m^3. Based on the assumed final volume, the sheet determines the compression of the spring (Δx), spring force (Fspring), final pressure (P_2), and total work involved (Work). The final temperature (T_2) is then calculated from the first-law according to Equation (4.27), and the final volume (V_2) from the ideal-gas law, Equation (4.28).

T_2 fx =T_1+(Q-Work)/m/Cv

	A	B	C	D	E	F	G	H	I	J
1										
2		P_1	200	kPa	T_2g	500		T_2	826.5425	K
3		T_1	317	kPa	V_2g	0.15		V_2	0.059259	m3
4		V_1	0.05	m3						
5					m	0.109915	=P_1*V_1/Rgas/T_1			
6		Q	72.7	kJ						
7		Area	0.25	m	Δx	0.4	=(V_2g-V_1)/Area			
8					Fspring	60	=kspring*Δx			
9		kspring	150	kN/m	P_2	440	=P_1+Fspring/Area			
10					Work	32	=0.5*(P_1+P_2)*(V_2g-V_1)			
11		Rgas	0.287	kJ/kg.K						
12					Cv	0.7267	=0.645+0.0002*(T_1+T_2g)/2			
13										

Figure 4.15. Excel sheet developed for Example 4-5

As Figure, 4.15 shows, the calculated values T_2 and V_2 are different from the initial values T_2g and V_2g. Solver can now be used to adjust the guessed value of T_2g and V_2g until they become the same as the calculated values. Figure 4.16 shows the set-up that requires Solver to change the values of T_2g and V_2g in cells F2 and F3, respectively, until the two specified constraints are satisfied: (i) T_2 = T_2g and (ii) V_2=V_2g. Note that the "**Set Objective**" option has been left blank. Figure 4.17 shows the solution obtained by the **GRG Nonlinear** method of Solver, which is T_2g = 1014.864K and V_2g = 0.1 m^3. At this state, the final pressure on the piston is 320.0 kPa and the total work is 13.0 kJ. These values agree with their corresponding values given by Cengel and Boles [5] whose analysis gave the same values.

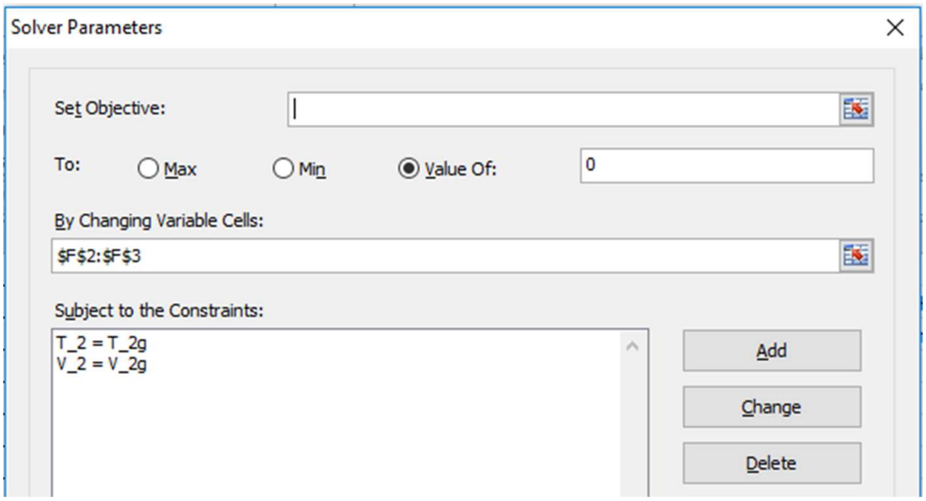
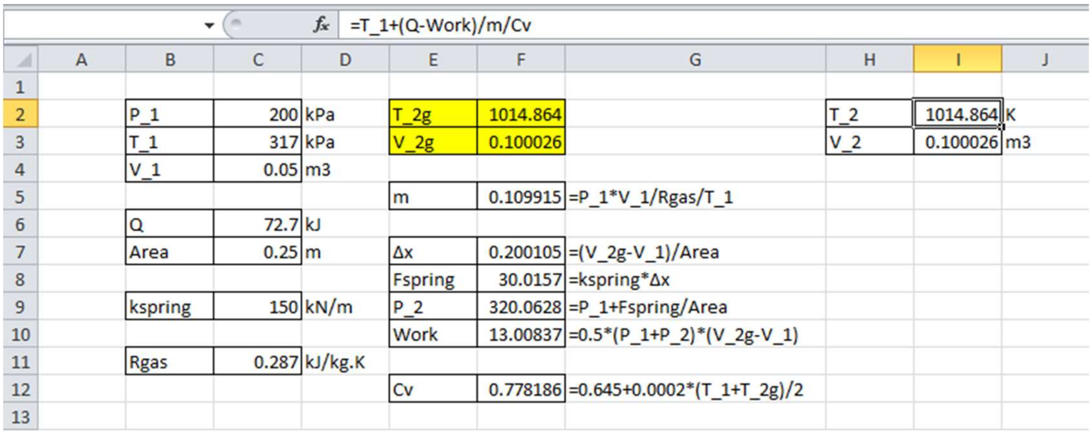


Figure 4.16. Solver's set-up for Example 4-5



=T_1+(Q-Work)/m/Cv

	A	B	C	D	E	F	G	H	I	J
1										
2		P_1	200	kPa	T_2g	1014.864		T_2	1014.864	K
3		T_1	317	kPa	V_2g	0.100026		V_2	0.100026	m3
4		V_1	0.05	m3						
5					m	0.109915	=P_1*V_1/Rgas/T_1			
6		Q	72.7	kJ						
7		Area	0.25	m	Δx	0.200105	=(V_2g-V_1)/Area			
8					Fspring	30.0157	=kspring*Δx			
9		kspring	150	kN/m	P_2	320.0628	=P_1+Fspring/Area			
10					Work	13.00837	=0.5*(P_1+P_2)*(V_2g-V_1)			
11		Rgas	0.287	kJ/kg.K						
12					Cv	0.778186	=0.645+0.0002*(T_1+T_2g)/2			
13										

Figure 4.17. Solver's solution for Example 4-5

4.3. Iterative solutions involving nonlinear equations

To determine the head loss due to friction (h_f) in Example 4-1, the friction factor (f) for the turbulent pipe flow was obtained from Equation (1.6) which is an explicit equation. However, for a turbulent pipe flow, f can be determined more accurately by using the following Colebrook-White equation [1]:

$$\sqrt{\frac{1}{f}} = -2.0\log\left(\frac{\varepsilon / D}{3.7} + \frac{2.51}{\mathrm{Re}\sqrt{f}}\right) \tag{4.29}$$

Since the equation involves the friction factor f in both sides, it needs to be solved iteratively in order to determine f. Therefore, for type-2 and type-3 flow problems using this equation involves two nested iterations; an inside iteration to determine f and an outside iteration to determine the pipe's diameter or flow rate. To allow the Colebrook-White equation to be used in iterative solutions and optimisation analyses, Thermax provides a Newton-Raphson solver (**NRM**) for this equation which can also be used for other nonlinear equations. Appendix B illustrates the use of the **NRM** solver by considering another nonlinear equation which is the Benedict-Webb-Rubin equation. In the present situation, the **NRM** solver will be used to solve the Colebrook-White equation leaving the main iteration for Solver or Goal Seek. For illustration, let us reconsider Example 4-1 and solve it by the using Equation (4.29) to determine *f* instead of Equation (1.6). The **NRM** solver requires the intended nonlinear equation to be provided as a separate user-defined function. The one needed for the Colebrook-White equation is listed below:

```
Function colebrook(x, e, Re)
' Colebrook equation for the friction factor
        colebrook = 1/Sqr(x) + (2/log(10))*Log(e /3.7 + 2.51/Re/Sqr(x))
End Function
```

Note that in VBA syntax the term "log" is used for the natural logarithm "ln", which is different from Excel. Figure 4.18 shows the Excel sheet developed for solving Example 4-1 with the Colebrook-White equation.

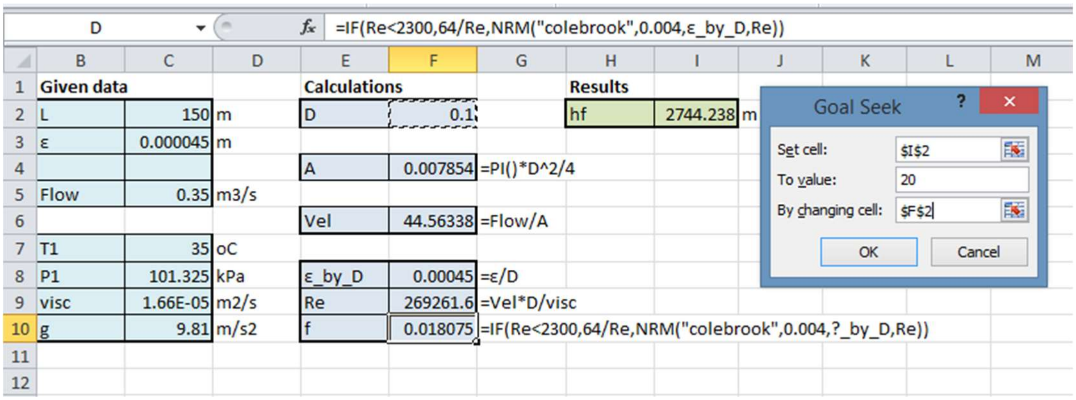


Figure 4.18. Excel sheet for Example 4-1 using the Colebrook-White equation

The only difference from the sheet shown in Figure 4.2 is the content of the cell **F10** that calculates friction factor. Figure 4.18 shows the formula typed in this cell as:

=IF(Re<2300,64/Re,NRM("colebrook",0.004,ε_by_D,Re))

The first input to the **NRM** solver, "**colebrook**", refers to the separate function that contains the Colebrook-White equation while the second input, **0.004**, is an initial guess for *f*. The last two arguments, **ε_by_D** and **Re**, respectively, are values of the two cells **F8** and **F9** that store the roughness-diameter ratio (ε/D) and the Reynolds number (Re) at which f is to be determined. For other nonlinear equations, these two arguments are replaced by relevant input parameters. For the Benedict-Webb-Rubin equation, refer to Appendix B that considers this equation.

Figure 4.18, which shows the calculations for a selected diameter of 0.1 m, shows that the value of the friction factor obtained by the Colebrook-White equation is 0.018075 and the corresponding friction loss is 2744.2 m. These values are slightly different from those obtained with Equation (1.6) as shown in Figure 4.2. The diameter that keeps the loss below 20 m can be determined by using Goal Seek and Figure 4.18 also shows Goal Seek set-up for finding the value of D that makes the friction head loss equal to 20 m. As Figure 4.19 shows, the answer found by Goal Seek with the Colebrook-White equation is $D \geq 0.27$ m, which is the same answer obtained earlier in Example 4-1.

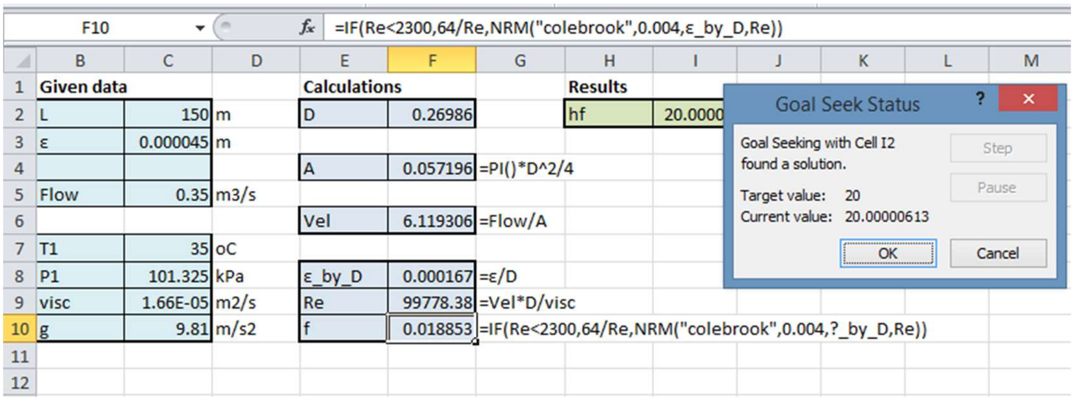


Figure 4.19. Goal Seek solution for Example 4-1 using the Colebrook-White equation

4.4. Closure

This chapter deals with thermofluid analyses that require iterative solutions and shows how Excel's Goal Seek command and Solver can be used for solving typical problems from the areas of fluid dynamics, thermodynamics, and heat-transfer. While the Goal Seek command can easily perform the simple type of iterative solutions that involve a single parameter, Solver can perform the more difficult iterative solutions that involve multiple changeable cells and require constraints to be applied to the iterative solution. The chapter also shows how the Newton-Raphson solver provided by Thermax can be used to deal with the iterative solutions that involve nonlinear equations such as the Colebrook-White equation and the Benedict-Webb-Rubin equation.

References

[1] Y. A. Cengel and J. M. Cimbala, *Fluid Mechanics Fundamentals and Applications*, 3rd edition, McGraw-Hill, 2006.
[2] C. T. Crowe, D. F. Elger, B. C. Williams, and J. A. Roberson, *Engineering Fluid Mechanics*, 9th edition, John Wiley, 2009.
[3] J. P. Holman, *Heat Transfer*, 10th edition, McGraw-Hill. 2010.
[4] M. Schumack, Solution of complex pipe flow problems using spreadsheets in an introductory fluid mechanics course, Proceedings of the 2004 American Society for

Engineering Education Annual Conference & Exposition, Session 3666, Pages 9.1108.1 - 9.1108.13. Available at https://peer.asae.org/solution-of-complex-pipe-flow-problems-using-spreadsheets-in-an-introductory-fluid-mechanics-course

[5] Y. A. Cengel, and M. A. Boles, *Thermodynamics. An Engineering Approach*. 5th edition, McGraw-Hill, New York, 2006.

[6] M.J. Moran and H.N. Shapiro, *Fundamentals of Engineering Thermodynamics*, 5th edition, John Wiley, & Sons. Inc. 2006.

[7] M.M. El-Awad, Use of Excel's 'Goal Seek' feature for Thermal-Fluid Calculations, the Electronic Journal of Spreadsheets in Education (eJSiE), Vol. 9, Iss. 1, Article 3, 2016.

Exercises

1. Consider the case in Example 4-1. Suppose that the only available pipe diameter is 20 cm and we want to maintain the same maximum limit on the friction head loss of 20 m by reducing the water flow rate. Using the Goal Seek command, determine the water flow rate that gives the required result. Answer: 0.157 m^3.
2. Using the Excel sheet developed for Example 4-2, determine the final temperature for air when the amount of heat added is 50, 100, 150, and 200 kJ. Also calculate the final temperature from Equation (4.3) by using a constant specific heat (c_p) of 1.043 kJ/kg.K. Plot the values obtained for the final temperature (T_2) with the amount of heat added by the two methods and comment on the result.
3. A gas mixture consisting of O_2 and CO_2 with mole fractions 0.2 and 0.8, respectively, expands isentropically and at steady state through a nozzle from 700 K, 500 kPa to an exit pressure of 100 kPa as shown in Figure 4.P3. Determine the temperature at the nozzle exit, in K.

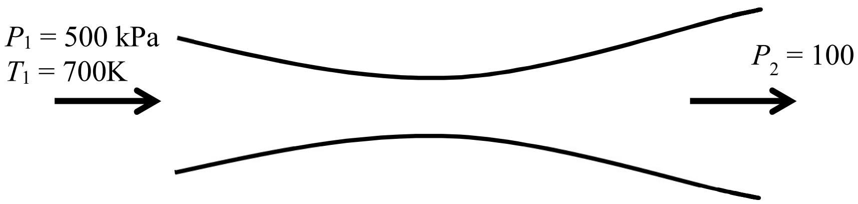


Figure 4.P3. Isentropic expansion in a nozzle

Using the approximate constant-specific heat method, the exit temperature (T_2) can be determined from:

$$T_2 = T_1 \times \left(P_2 / P_1\right)^{\frac{k-1}{k}} \tag{A}$$

Where k is the ratio of the specific heats for the mixture. Using k = 1.304, the resulting exit temperature is 480.9K. Using the exact variable specific heat method, T_2 is determined by requiring the total entropy change to be zero, i.e.:

$$y_{O2}\left[s_{O2}^{0}(T_2)-s_{O2}^{0}(T_1)-R_{O2}\ln\frac{P_2}{P_1}\right]$$
$$+\,y_{CO2}\left[s_{CO2}^{0}(T_2)-s_{CO2}^{0}(T_1)-R_{CO2}\ln\frac{P_2}{P_1}\right]=0 \qquad \text{(B)}$$

Where y_{O2} and y_{CO2} are the volume fractions of O_2 and Co_2, respectively, and R_{O2} and R_{CO2} are the molar masses for O_2 and Co_2, respectively. The values of s_{O2}^{0} and s_{CO2}^{0} can be determined by using the relevant function provided by Thermax. Equation (B), that requires an iterative solution, can be solved by using the Goal Seek command. This exercise is based on Example 12.4 in Moran and Shapiro [6]. Answer: T_2 = 514.05K.

4. Steam is be condensed at 30°C on the shell side of the multi-pass shell-and-tube heat exchanger shown in Figure 4.P4. The condenser has 8-tube-passes with 50 tubes in each pass. Its overall heat transfer coefficient is 1000 W/m^2·°C. Cooling water (C_p = 4180 J/kg·°C) enters the tubes at 15°C at a rate of 55,000 kg/h. The tubes are thin-walled, and have a diameter of 1.5 cm and length of 2 m per pass. (a) determine the outlet temperature of the cooling water by using the ε-NTU method and (b) develop an Excel sheet to determine the outlet temperature by using the Goal Seek command and the LMTD method [7].

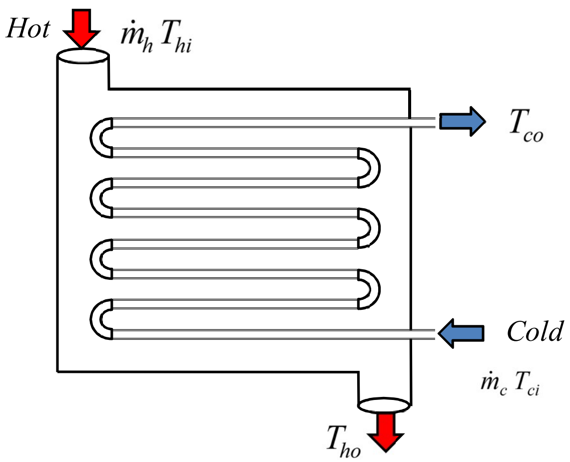


Figure 4.P4. A multi-pass shell-and-tube heat exchanger

5. Reconsider the case in Example 4-5. An alternative solution of this problem that also takes into consideration the variation of specific heat for air with temperature can be obtained by using the ideal-gas property functions provided by Thermax instead of Equation (4.20). Show that this solution can be obtained by using the Goal Seek

command instead of Solver and compare your solution with that given in Example 4-5.

6. Consider the semi-infinite slab shown in Figure 4.P6 that is suddenly exposed to convection environment at T_∞. The temperature (T) at a depth x from the surface at any time is given by [3]:

$$\frac{T - T_i}{T_\infty - T_i} = 1 - erf(X) - \left[\exp\left(\frac{hx}{k} + \frac{h^2 \alpha \tau}{k^2}\right)\right] \times \left[1 - erf\left(X + \frac{h\sqrt{\alpha\tau}}{k}\right)\right] \quad \text{(A)}$$

Where α and k are the diffusivity and thermal conductivity of the slap material, respectively, T_i is the initial temperature of the solid, T_∞ is the environmental temperature, τ is the elapsed time in seconds, and $X = \left(2\sqrt{\alpha\tau}\right)$. Equation (A) requires an iterative procedure because the time (τ) appears in both terms on the right-hand side of the equation.

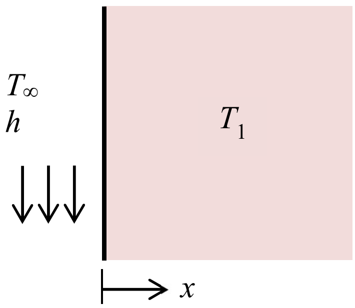


Figure 4.P6. Semi-infinite slab with convection heat-transfer

A large slab of aluminium (k = 215 W/m.°C, $\alpha = 8.4 \times 10^{-5}$ m²/s) at a uniform temperature of 200°C is suddenly exposed to a convection-surface environment of 70°C with a heat-transfer coefficient of 525 W/m²·°C. Calculate the time required for the temperature to reach 120°C at the depth of 4.0 cm for this circumstance.

This problem is based on Example 4-5 in Holman [3] for which the answer is approximately 3000 seconds.

Water at 60°C enters a tube of 3-cm diameter at a mean flow velocity of 1.2 cm/s. If the tube is 3.0 m long and the wall temperature is constant at 80°C, what will be the exit water temperature? Use Goal Seek to perform the iterative solution of this problem. To determine the viscosity of water at any temperature, develop a user-defined function based on the data shown in Table A.2 in Appendix A. This exercise is based on Example 4-2 in Holman [3]. Answer: 73.0°C.

5

Finite-difference solution of the steady heat-conduction equation

Differential equations occur frequently in thermofluid analyses, particularly in heat-transfer and fluid-dynamics analyses. In heat-transfer, the equations describe the spatial and/or temporal variation of temperature in the medium. In fluid-dynamics, they describe the variation of fluid velocity and scalar quantities such as temperature and contaminant concentration. By replacing the differential equation with a system of linear equations that can be solved using standard methods, numerical solution methods, such as the finite-difference (FD) method and the finite-element (FE) method, enable us to deal with multi-dimensional and complex geometries. The FD method is easier to apply than the FE method because it directly replaces the derivatives in the equation by finite differences. This chapter focuses on using the FD method for solving the steady heat-conduction equation. Two problems of steady one-dimensional heat conduction with simple boundary conditions and without heat generation are first solved with the analytical solution method and then with the FD method so as to highlight the differences between the numerical and the analytical method. The FD method is then applied to solve the one-dimensional heat-conduction equation with heat generation and the two-dimensional heat-conduction equation. The chapter presents two approaches for applying the FD method with Excel the first of which assembles the system of linear equations and uses Excel's matrix functions to solve it, while the second approach uses circular calculations and, therefore, does not require the system of linear equations to be explicitly assembled.

5.1. Analytical solution of the steady one-dimensional conduction equation

The unsteady heat transfer by conduction in a three-dimensional medium with heat generation but a constant thermal conductivity is governed by the following partial differential equation [1]:

$$\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}+\frac{\dot{e}_g}{k}=\frac{1}{\alpha}\frac{\partial T}{\partial t} \tag{5.1}$$

Where k is the thermal conductivity, $\dot{e}_g$ the volumetric rate of heat-generation, and α the thermal diffusivity of the medium ($\alpha = k/\rho c_p$). For steady one-dimensional (1D) heat-transfer by conduction without heat generation, the equation reduces to:

$$\frac{d^2 T}{dx^2}=0 \tag{5.2}$$

Equation (5.2) can be solved analytically by integrating it twice, i.e.:

$$\frac{dT}{dx}=C_1 \tag{5.3}$$

$$T(x)=C_1 x+C_2 \tag{5.4}$$

Values of the two unknown constants C_1 and C_2 can then be determined if given two independent boundary conditions. The following two examples illustrate the application of the analytical solution method by using simple boundary conditions.

Example 5-1. One-dimensional heat-conduction with prescribed-temperatures
The inner and outer sides of a large plane wall of thickness L = 0.3 m and thermal conductivity k = 0.8 W/m^2.°C are kept at constant temperatures of 20°C and 5°C, respectively. Determine the temperatures at x = 0.1 m and x = 0.2 m during a steady heat transfer by conduction.

Solution
In this case, the two constants in Equation (5.4) can be determined by applying the boundary conditions as follows:

$T = T_0$ at $x = 0$; Which leads to $C_2 = T_0$

$T = T_L$ at $x = L$; Which leads to $C_1 = (T_L - T_0)/L$

Substituting the values obtained for C_1 and C_2 in Equation (5.4) yields:

$$T(x) = T_0 - \frac{x}{L}\left(T_L - T_0\right) \tag{5.5}$$

The solution, which indicates that the temperature changes linearly between the two wall surfaces, should be expected since k is constant and there is no heat generation. Substituting for x/L = 1/3 and x/L = 2/3 in Equation (5.5), gives the two temperatures:

$T_{0.1}$ = 10°C

$T_{0.2}$ = 15°C

Example 5-2. One-dimensional heat-conduction with a specified heat-flux
Consider the steady heat conduction in a large plane wall of thickness L = 0.3 m and thermal conductivity k = 0.8 W/m^2.°C. Suppose that the inner side of the wall is subjected to a constant heat flux $\dot{q}$ = 50 W/m^2 while the outer side of the wall is kept constant at 40°C. Determine the temperature at the inner side of the wall (x=0) and at the centre of the wall (x=L/2).

Solution
Equation (5.4) also applies for this case and values of the constants C_1 and C_2 can be determined by applying the two boundary conditions:

$\dot{q}_0$ = 50 W/m^2

6

Finite-difference solution of the transient heat-conduction equation

This chapter extends the methodology presented in the previous chapter so as to solve the transient 1D and 2D heat-conduction equations with the finite-difference (FD) method. The chapter initially considers the transient 1D conduction-equation to present two schemes for applying the method: (i) the explicit forward-in-time central difference in space (FTCS) scheme and (ii) the implicit backward-in-time central difference in space (BTCS) scheme. For the implicit scheme that requires a system of linear equations to be solved, the two methods for using Excel are presented. The first method explicitly forms and solves the linear system by using the tri-diagonal matrix algorithm (TDMA), while the second method does not form the linear system but uses Excel's iterative-calculation option to solve the nodal equations. The transient 2D conduction equation is also solved with both the explicit and implicit schemes. Since solving the 2D conduction equation with the implicit scheme requires a large system of linear equations to be assembled and solved, only the method of applying Excel's iterative-calculation option is presented.

6.1. Transient 1D conduction: the explicit FTCS formulation

The transient one-dimensional heat-transfer by conduction is governed by:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \tag{6.1}$$

As shown in the previous chapter, the FD method replaces the spatial derivative ($\partial^2 T / \partial x^2$) by the central difference defined by Equation (5.13). For the temporal term ($\partial T / \partial t$) either forward or backward differences can be used, i.e.:

$$\left.\frac{\partial T}{\partial t}\right|_{FW} \approx \frac{T_m^{i+1} - T_m^i}{\Delta t} \tag{6.2}$$

$$\left.\frac{\partial T}{\partial t}\right|_{BW} \approx \frac{T_m^{i} - T_m^{i-1}}{\Delta t} \tag{6.3}$$

Where, i, i-1, and i+1 refer to three time levels. Depending on whether Equation (6.2) or (6.3) is selected, two FD formulations for Equation (6.1) are possible: (i) a forward-in-time central-difference in space (FTCS) scheme and (ii) a backward-in-time central-difference in space (BTCS) scheme. This section focuses on the first scheme.

By using Equation (6.2), the FD approximation of Equation (6.1) becomes:

$$\frac{T_m^{i+1} - T_m^i}{\Delta t} = \alpha \frac{T_{m+1}^i - 2T_m^i + T_{m-1}^i}{\Delta x^2} \tag{6.4}$$

Note that the temperatures at the three nodes on the r.h.s of Equation (6.4) are evaluated at the same time level i. Equation (6.4) can be rearranged as follows:

$$T_m^{i+1} = T_m^i + \tau\left(T_{m+1}^i - 2T_m^i + T_{m-1}^i\right) \tag{6.5}$$

Where, τ is the dimensionless mesh Fourier number defined as:

$$\tau = \frac{\alpha \Delta t}{\Delta x^2} \tag{6.6}$$

Equation (6.5) calculates the temperature at the new time step (i+1) from the known values at the old time step (i). Starting with a known temperature field, it can be used directly to compute the new temperature field because all the values of T_m^{i+1} can be updated independently. The FTCS scheme is easy to implement, but it is only conditionally stable by requiring the time step Δt to be chosen such that [1]:

$$\tau < \frac{1}{2} \tag{6.7}$$

The following example shows how the scheme is applied.

Example 6-1. Solving the transient 1D conduction equation with the FTCS scheme
Transient heat-conduction occurs in a plate with length L = 0.1 m and diffusivity α = 0.1. The boundary conditions are $T(0,t) = T(L,t) = 0$ and the initial temperature distribution is given by:

$$T(x) = \sin(\pi x/L) \tag{6.8}$$

Use the FD method with a grid of 21 equally-spaced nodes and a suitable time step to determine the temperature variation at t = 0.2 s.

The exact temperature variation is given by [1]:

$$T(x,t) = \sin\left(\frac{\pi x}{L}\right)\exp\left(-\frac{\alpha \pi^2 t}{L^2}\right) \tag{6.9}$$

Solution
By dividing the length of the plate into 21 equally-spaced nodes, Δx= 0.1/20 = 0.05 m. According to Equation (6.7), the maximum allowable time step Δt is given by:

$$\Delta t = 0.5 \times 0.05^2 / 0.1 = 0.0125 \text{ s}$$

7

Hydraulic analyses of multi-pipe and pump-pipe systems

Pump-pipe systems used in the various industrial and residential applications usually consist of multiple pipes and pumps connected in parallel or in series. The hydraulic analyses of multi-pipe and multi-pump systems are based on the basic principles of mass and energy conservation discussed in Chapter 1 by considering a simple pump-pipe system. However, simultaneously satisfying the continuity and energy equations in the analyses of multi-pipe systems usually requires an iterative solution. An iterative solution is also needed for determining the operating point for any pump-pipe system whether it involves a single pump or multiple pumps. This chapter shows how Excel with its iterative tools, Goal Seek and Solver, can be used for analysing multi-pipe systems and determining the operating point for a single pump with various pipe arrangements.

7.1. Analyses of multi-pipe systems

Figure 7.1 shows three multi-pipe arragements in which three pipes are connected (a) in series, (b) in parallel, and (c) at a junction. In general, the pipes may have different lengths, diameters, and roughness. They may also have different elevations at their entrance and exit points.

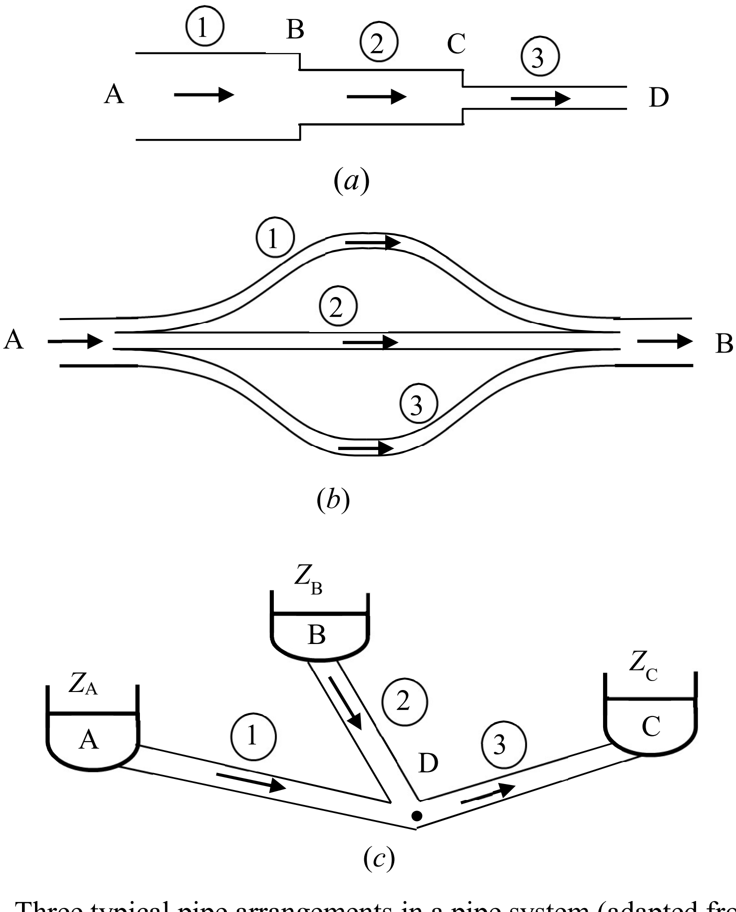


Figure 7.1. Three typical pipe arrangements in a pipe system (adapted from White [1])

As for the case of a single pipe, three types of flow problems can arise with multi-pipe analyses depending on whether the pipe diameters and flow rates are known in advance or not. If both the pipe diameters and flow rates are known a priori then the problem is classified as a type-1 flow problem, but if one of these is to be determined, then the problem is classified as a type-2 or a types-3 flow problem. Like the case of a single pipe, both type-2 and type-3 flow problems for multi-pipe system require iterative solutions. The following subsections shows how Solver can be used to perform the iterative solutions for type-2 and type-3 analyses.

7.1.1. Type-2 flow analyses

To illustrate the Excel-aided analysis procedure for this type of flow, consider the following example which is based on Example 8-7 in Cengel and Cimbala [2].

Example 7-1. Pumping water through two parallel pipes

Water at 20°C is to be pumped from reservoir *A* to reservoir *B* through two 36-m-long pipes connected in parallel as shown on Figure 7.2. The elevation of reservoir *A* is 5 m while that of reservoir *B* is 13 m. The two pipes are made of commercial steel and their diameters are 4 and 8 cm as shown in the figure. Water is to be pumped by a 70% efficient motor-pump combination that draws 8 kW of electric power during operation. The minor losses and the head loss in the pipes that connect the parallel pipes to the two reservoirs can be neglected. Determine the total flow rate of the pump and the flow rate through each of the two parallel pipes.

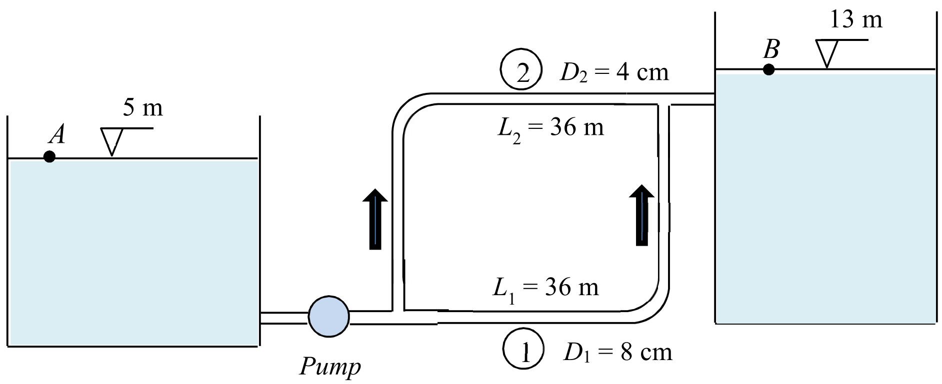


Figure 7.2. The piping system in Example 7-1 (adapted from Cengel and Cimbala [2])

The analytical model

Since the fluid at both points *A* and *B* are open to the atmosphere, $P_A = P_B = P_{atm}$. Also, the fluid velocities at both points are zero ($V_A = V_B = 0$) and, therefore, the energy equation, Equation (1.2), between these two points simplifies to:

$$h_p = h_f + (Z_B - Z_A) \qquad (7.1)$$

Where h_p is the pump head and h_f is the total friction loss between the two reservoirs. Since the pressures at the entrance and exit points of the two pipes are the same, the friction head losses in the two pipes must be the same, i.e.,

$$h_{f1} = h_{f2} = h_f \tag{7.2}$$

The friction head losses in the two pipe are given by the Darcy-Weisbach equation:

$$h_{f1} = f_1 \frac{L_1}{D_1} \frac{V_1^2}{2g} \tag{7.3}$$

$$h_{f2} = f_2 \frac{L_2}{D_2} \frac{V_2^2}{2g} \tag{7.4}$$

Given the two flow rates Q_1 and Q_2, the water velocities in two pipes, V_1 and V_2, can be calculated from:

$$V_1 = Q_1 / A_1 \tag{7.5}$$

$$V_2 = Q_2 / A_2 \tag{7.6}$$

For steady flow of an incompressible fluid, the total flow-rate given by the pump (Q) is the summation of the flow-rates in the two parallel pipes (Q_1 and Q_2), i.e.,

$$Q = Q_1 + Q_2 \tag{7.7}$$

Finally, the power of the pump ($\dot{W}_p$) is given by the power equation as follows:

$$\dot{W}_p = \gamma \times Q \times h_p / \eta \tag{7.8}$$

Where γ is the fluid's specific weight and η is the combined pump-motor efficiency. In the present case, the pump-power is known to be 8 kW. This can be used to determine the unknown flow rate (Q) and the pump head (h_p) by the following iterative procedure:

1. Assume the pipe flow rates Q_1 and Q_2
2. Calculate the corresponding velocities V_1 and V_2 from Equation (7.5) and Equation (7.6), respectively.
3. Based on the two velocities, calculate the friction factors and friction head losses in the two pipes from Equation (7.3) and Equation (7.4), respectively, and calculate the pump power from Equation (7.8)

4. Compare the resulting pipe friction losses to each other using Equation (7.2), and pump power to the specified value of 8 kW. If: (i) the pipe friction losses are different or (ii) the pump power is not equal to 8 kW, go back to step 1 and repeat the procedure until both conditions are reasonably satisfied.

Excel implementation

Figure 7.3 shows the Excel sheet developed for this example. The data part, positioned on the left-side of the sheet, shows the given information about the pump-pipe system such as the lengths and diameters of the two pipes, density and viscosity of water, pump efficiency, etc. The calculations part begins at column E with assumed values for the two pipe flow rates (Q_1 = Q_2 = 0.01 m^3/s). Based on these assumed flow rates, the sheet determines the corresponding velocities in the two pipes (V_1 and V_2), the Reynolds numbers (Re_1 and Re_2), and the friction factors (f_1 and f_2). For a turbulent flow, the friction factor is obtained from the Colebrook-White formula. The pipes friction losses, hf_1, and hf_2, are then calculated and the total pump power (P_pump) is determined from Equation (7.8) as shown in the formula bar.

Power | fx =ρ*g_*Q_total*(hf_1+(Z_B-Z_A))/η_p/1000

	A	B	C	D	E	F	G	H	I	J	K	L
1												
2	L_1	36	m	Q_1	0.01		Re_1	158519.6		Power	2.734141	kW
3	D_1	0.08	m	Q_2	0.01		Re_2	317039.2				
4	L_2	36	m									
5	D_2	0.04	m	Q_total	0.02		ε_by_D_1	0.000563				
6	Z_A	5	m				ε_by_D_2	0.001125				
7	Z_B	13	m	A_1	0.005027							
8	η_p	0.7		V_1	1.989437		f_1	0.019547				
9	ρ	998	kg/m3				f_2	0.02106				
10	μ	1.00E-03	kg/m.s	A_2	0.001257							
11	ε	0.000045	m	V_2	7.957747		hf_1	1.774383				
12	g_	9.81	m/s2				hf_2	61.1772				
13												

Figure 7.3. Excel sheet for Example 7-1

With the guessed pipe flow rates, Figure 7.3 shows that the pump power is only 2.734 kW, which is considerably less than the required value. Moreover, the values of h_{f1} and h_{f2} are different and, consequently, Equation (7.2) is not satisfied. Solver can now be used to find the correct values of Q_1 and Q_2 and Figure 7.4 shows the required set-up for its parameters dialog-box. Note that the pump power is specified as the **Set Objective** and required to have a value of 8 kW. In this case, Solver will iterate to satisfy the imposed constraint, which is requirement to satisfy Equation (7.2). According to the solution found by the **GRG Nonlinear** method of Solver with this set-up as shown on Figure 7.5, the flow rates are Q_1 = 0.0258 m^3/s and Q_2 = 0.00415 m^3/s, giving a total pump flow rate of 0.03 m^3/s. Table 7.1 compares this solution with that obtained by Cengel and Cimbala [2]. The figures on the table show only small differences between the two solutions even though Cengel and Cimbala [2] determined the friction factors f_1 and f_2 from the

Colebrook-White equation. Appendix B shows how this implicit equation can be handled within an iterative solution by using the Newton-Raphson solver provided by Thermax.

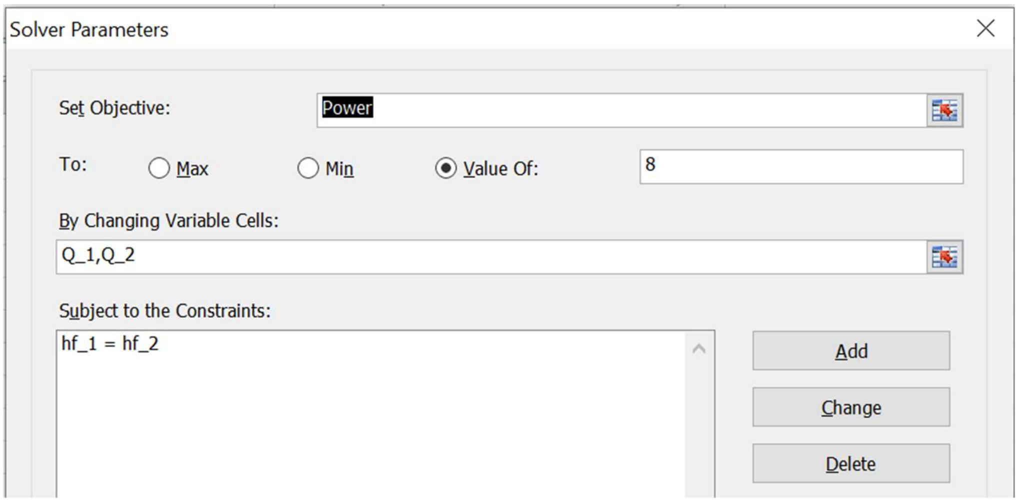
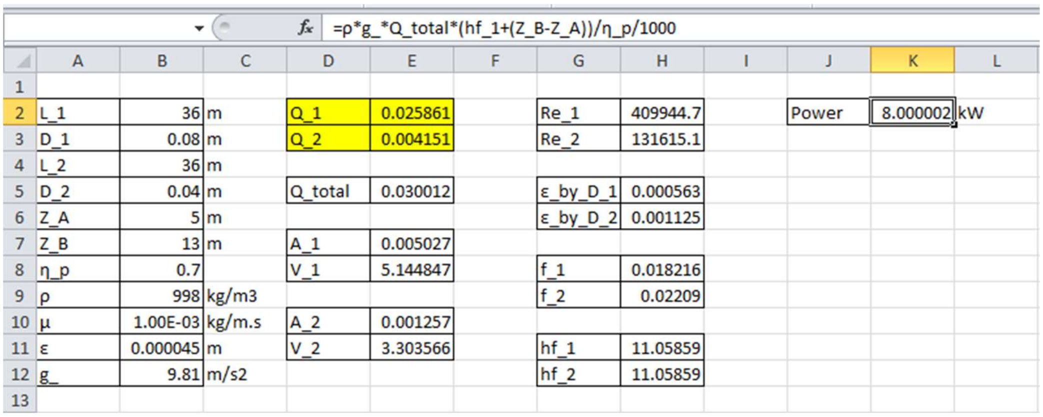


Figure 7.4. Solver's set-up for Example 7-1



=ρ*g_*Q_total*(hf_1+(Z_B-Z_A))/η_p/1000

	A	B	C	D	E	F	G	H	I	J	K	L
1												
2	L_1	36	m	Q_1	0.025861		Re_1	409944.7		Power	8.000002	kW
3	D_1	0.08	m	Q_2	0.004151		Re_2	131615.1				
4	L_2	36	m									
5	D_2	0.04	m	Q_total	0.030012		ε_by_D_1	0.000563				
6	Z_A	5	m				ε_by_D_2	0.001125				
7	Z_B	13	m	A_1	0.005027							
8	η_p	0.7		V_1	5.144847		f_1	0.018216				
9	ρ	998	kg/m3				f_2	0.02209				
10	μ	1.00E-03	kg/m.s	A_2	0.001257							
11	ε	0.000045	m	V_2	3.303566		hf_1	11.05859				
12	g_	9.81	m/s2				hf_2	11.05859				
13												

Figure 7.5. Solver's solution for Example 7-1

Table 7.1. Comparison of the present solution with that of Cengel and Cimbala [2]

Parameter	Cengel & Cimbala [2]	Present solution	Deviation (%)
Q_1	0.0259	0.025861	-0.151
Q_2	0.00415	0.004151	0.024
Re_1	410,000	409,944.5	-0.014
Re_2	131,600	131,615.3	0.012
f_1	0.0182	0.018216	0.088
f_2	0.0221	0.02209	-0.045
h_f	11.1	11.05861	-0.373

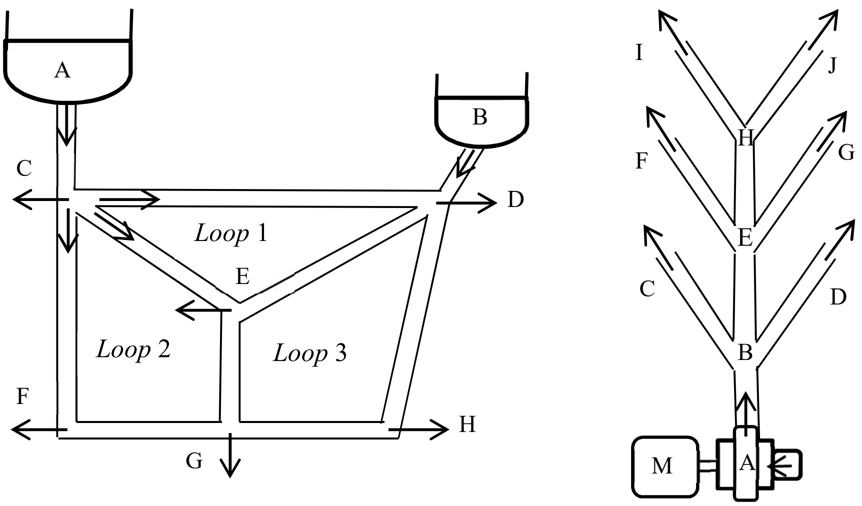
8

Hydraulic analyses of pipe networks

As for simple pump-pipe systems and pump-pipe systems with multiple pipes; hydraulic analyses of pipe-network are also based on the two principles of mass and energy conservation coupled with auxiliary formulae for determining the pipe friction. However, pipe-network analyses are more challenging because they involve more unknown flows and, therefore, more continuity equations and energy constraints to be simultaneously satisfied. The methodology of applying the two principles also depends on whether the pipe-network is looped, branched, or mixed. The chapter initially illustrates the limitation of mathematical methods for pipe-network analyses by considering a simple looped network with three pipes and then presents the Excel-Solver method that can be used to deal with the analyses of looped, branched, and mixed pipe networks.

8.1. The methodology of looped pipe-network analysis

Figure 8.1 shows two types of pipe network arrangements used in practice: (a) looped networks and (b) branched networks. The flow in the pipe network can be driven by the effect of gravity or by a motor-pump system. The connection points, called nodes, can be sources (supply points), sinks (demand or load points), or just junctions. The looped network on Figure 8.1.a has two sources, which are the reservoirs A and B, and six load points. The branched network on Figure 8.1.b has a single source, which is the pump A, and six load points. In a typical hydraulic analysis, the flow rates at the sources and outlets are given together with the lengths, diameters, and roughness of the pipes and the requirement is to determine the flow rates and pressure levels at the different nodes.



(a) (b)

Figure 8.1. Pipe network arrangements: (a) looped and (b) branches networks

The analytical model for a looped network

According to the notation scheme used on Figure 8.1, a single letter refers to a location or a pipe junction, e.g., A, B, etc., two letters refer to a pipe, e.g., AB, BC, etc., whereas numbers refer to pipe loops, e.g. loop 1, loop 2, etc. In order to analyse the network shown on Figure 8.1.a that has eight pipes, eight independent equations have to be formed from mass and energy balances.

The continuity equation can be applied at two levels; (i) the whole pipe network, and (ii) any junction (node) in the network. Applied to the whole network shown on Figure 8.1.a for a steady flow of an incompressible fluid, the continuity equation leads to:

$$Q_A + Q_B = Q_C + Q_D + Q_E + Q_F + Q_G + Q_H \tag{8.1}$$

Applied to any junction in the network, the continuity equation requires the algebraic sum of all the flow rates meeting at the junction to be zero. For example, at junction C the equation becomes:

$$Q_C = Q_{AC} - Q_{CD} - Q_{CE} - Q_{CF} \tag{8.2}$$

Application of the continuity equation at the six nodes in the network shown on Figure 8.1.a leads to five (number of nodes minus 1) independent linear equations. The remaining three equations have to be obtained from the energy equation.

The general form of the energy equation, Equation (1.2), does not only account for the friction losses in the pipes, but also for the differences in elevations between the end points. However, for simplicity we will assume here that all pipes of the network under consideration are on the same horizontal plane. A special form of the energy equation can be written for a closed loop in which point B in the equation coincides with point A. In this case, the algebraic sum of the head losses in all the pipes forming a closed loop is balanced by any heads generated by inline booster pumps in the loop, i.e.,

$$\sum_1^M h_f = \sum_1^N h_P \tag{8.3}$$

Where M is the number of pipes forming the closed loop, N is the number of booster pumps in the loop, h_f is the head loss in each pipe (including the minor losses), and h_P is the head produced by a booster pump. When there is no booster pump within the loop, Equation (8.3) reduces to:

$$\sum_1^M h_f = 0 \tag{8.4}$$

Applying Equation (8.4) to the three loops in the network provides the three (equal to the number of loops) additional equations needed to solve the problem. The solution of the eight equations should give the unknown values and directions of the flow rates in the

9

Optimisation analyses using Solver

The design of thermofluid systems involves many factors such as the physical, economic, environmental, and safety factors. Taking all these factors into consideration is a multi-objective optimisation problem which is difficult to solve by using the conventional analytical methods. However, usually the main concern for the designer of these systems is to maximise their energy efficiencies and minimise their initial and running costs. Therefore, the designer seeks to find a compromise between the lifetime saving in energy cost achieved by improving the efficiency of the system and the additional cost due to this improvement. With this supposition, the optimisation analysis aims to determine the most suitable design for the system by defining a single optimisation objective that takes both factors into consideration which is minimising the combined annualised cost. The environmental factor can also be included by adding a relevant penalty cost to the total cost. This chapter describes an Excel-aided method for such single-objective optimisation and applies it for optimisation analyses of various types of thermofluid systems. Unlike the traditional analytical method, this method does not requires many simplifying assumptions for the development of the objective function so that the problem can be solved with the conventional mathematical methods. By using Solver, the method can also take into consideration any relevant constraints on the optimisation process such as physical (spatial or temporal) constraints or safety constraints**.** The chapter also evaluates the use of the Evolutionary method of Solver instead of the GRG Nonlinear method for such single-objective optimisation analyses.

9.1. The Excel-aided optimisation method

Any optimisation analysis involves two main steps: (i) developing the objective functions that quantify the optimisation criteria in terms of the system's design and operating parameters and (ii) using an optimisation method to find the values of these parameters that optimise (maximise or minimise) the objective functions. In the case of thermofluid systems, the first step involves the development of a mathematical analytical model for the system being considered based on thermofluid concepts. For the second step, numerous methods are available that can be broadly classified as classical calculus-based deterministic methods or evolutionary methods which are inspired by natural evolution and use concepts like selection, reproduction, and mutation [1,2]. The GRG Nonlinear method and the Evolutionary method provided by Excel's Solver belong to the first type and the second type of these two classes, respectively. While Excel, Thermax. and VBA provide an adequate platform for the development of the analytical models, Solver can be used to search for optimal solutions of complex design problems. The following example illustrates the Excel-aided method by considering a two stage air-compression system for which we want to determine the intermediate pressure that minimises the total compression work. To highlight the limitations of the traditional method that uses calculus compared to the Excel-aided method, the optimisation problem is also solved analytically in Appendix G after making several simplifying assumptions.

Example 9-1. Excel-aided optimisation of a two-stage air compressor

The two-stage air compressor shown on Figure 9.1 takes atmospheric air at 300K and 100 kPa and delivers it at a final pressure of 900 kPa. The adiabatic efficiency (η_c) of both compressor stages is 85%. The effectiveness (ε) of the intercooler that cools the air after the first stage is 85%. The intercooler causes a pressure drop of 5 kPa. Determine the exit pressure of the first stage (P_x) that minimises the total compression work.

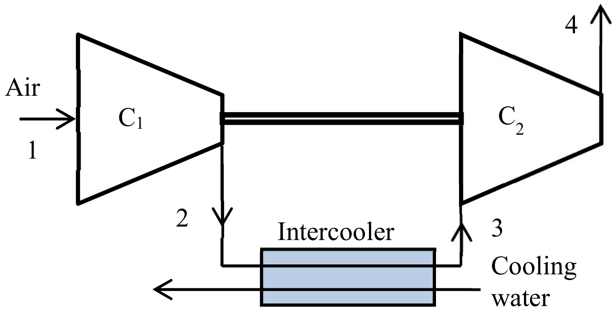


Figure 9.1. Schematic diagram of the two-stage, intercooled air compressor

The analytical model

Figure 9.2 shows the *T*-*s* diagram of the two-stage compression system that takes into consideration the friction losses in the two compression processes by the increased entropy. States 2s and 4s are the states of the air after ideal isentropic compression processes, while states 2 and 4 are the actual corresponding states. The analytical model also allows for the imperfection of the intercooler by taking into consideration its pressure losses (ΔP_{IC}) and heat-transfer effectiveness (ε).

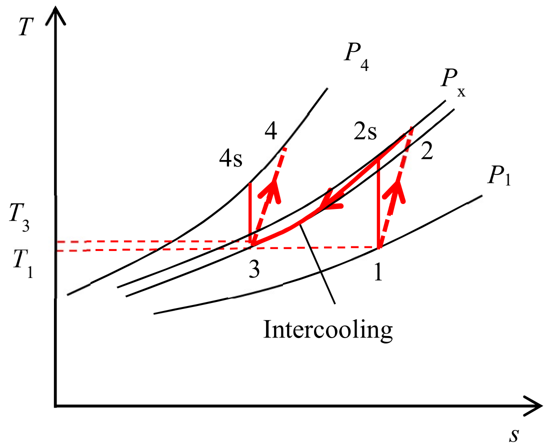


Figure 9.2. *T*-*s* diagram for the two-stage air compressor

By using the functions provided by Thermax for ideal gases, the analytical model applies the exact variable-specific-heat method. Given the inlet air temperature (T_1), the relative pressure at state 1 (P_{r1}) is determined by using the relevant function, which is **GasPr_TK**. Assuming an initial value for the intermediate pressure (P_x), the relative pressure at state 2s (P_{r2s}) is calculated from the following relationship [3]:

10

Design analyses of fluid-thermal systems

Design is inherently an iterative process for which computer-aided methods can help to overcome the tedium of repetitive analyses and avoid possible human errors. However, in the case of fluid-thermal systems, the use of computer-aided methods is unavoidable because they usually involve the solution of linear systems of equations or nonlinear equations and require optimisation analyses to meet the physical constraints while ensuring the safety and economic feasibility of the system. This chapter shows how Excel can be used for such analyses by considering five cases of fluid-thermal systems. The first case is that of thermal insulation for which the design analyses focus on selecting its thickness from safety and economic considerations. The second and third cases are those of a hot-water generation system with two sources of steam and a double-pipe heat exchanger. These two cases involve single-objective optimisation analyses to determine their optimum configurations by minimising their total annualised costs. The fourth case is that of a pump-pipe system which we want to select a suitable pump and a suitable standard-pipe size. The fifth case is that of a rectangular aluminium fin for which we seek to determine the thickness and length that minimise its weight as well as maximise the heat-transfer it extracts by conducting a dual-objective optimisation analysis.

10.1. Design analyses of thermal insulation

Figure 10.1.a shows a pipe that transports a fluid at a temperature ($T_{\infty 1}$) which is higher than that of the surroundings ($T_{\infty 2}$). The temperature difference causes the hot fluid being transported to lose part of its energy to the cooler ambient air by heat-transfer. The rate of heat-transfer from the surface of the pipe can be reduced by covering it with an insulation material as shown on Figure 10.1.b.

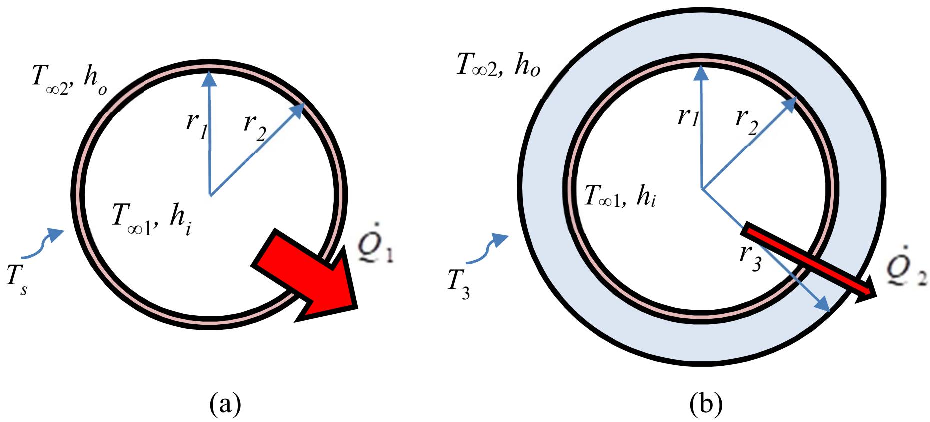


Figure 10.1. Fluid-transporting-pipe (a) without insulation and (b) with insulation

Without insulation, the rate of heat-transfer, $\dot{Q}_1$, is given by [1]:

$$\dot{Q}_1 = (T_{\infty 1} - T_{\infty 2}) / R_{th} \tag{10.1}$$

Where R_{th} is the thermal resistance to heat-transfer through the pipe as given by:

$$R_{th} = \frac{1}{h_i A_1} + \frac{\ln(r_2/r_1)}{2\pi L k_{pipe}} + \frac{1}{h_o A_2} \tag{10.2}$$

Where h_i and A_1 are inside heat-transfer coefficient and the corresponding surface area, respectively, h_o and A_2 are outside heat-transfer coefficient and the corresponding surface area, respectively, and L and k_{pipe} are the length and thermal conductivity of the pipe, respectively. After insulation, the reduced rate of heat-transfer, $\dot{Q}_2$, is given by:

$$\dot{Q}_2 = (T_{\infty 1} - T_{\infty 2}) / R_{th2} \tag{10.3}$$

Where R_{th2} is the increased thermal resistance after the insulation as given by:

$$R_{th2} = \frac{1}{h_i A_1} + \frac{\ln(r_2/r_1)}{2\pi L k_{pipe}} + \frac{\ln(r_3/r_2)}{2\pi L k_{ins}} + \frac{1}{h_o A_3} \tag{10.4}$$

Where k_{ins} is the thermal conductivity of the insulation and A_3 is its outside surface area of insulation.

At a steady state, the rate of heat transfer $\dot{Q}_2$ can also be determined by applying Newton's law of cooling:

$$\dot{Q}_2 = h_o A_3 (T_3 - T_{\infty 2}) \tag{10.5}$$

Where T_3 is the temperature of the outside insulation surface. Equation (10.5) can be used to determine the value of T_3 once Q_2 has been calculated as follows:

$$T_3 = T_{\infty 2} + \dot{Q}_2 / h_o A_3 \tag{10.6}$$

By taking into consideration the cost of energy and the cost of insulation, the above analytical model can be used to determine the value of the resulting energy saving and the economic insulation thickness. Apart from saving energy, the purpose of insulation can be to reduce the surface temperature of the pipe for safety considerations and there can be other reasons for imposing a lower or upper limit for the surface temperature of insulation. For example, the surface temperature of the insulation material may not be allowed to drop below a certain value in order to prevent condensation of the air-borne water vapour over the insulation surface [2]. In this case, the analytical model can be used to determine the required thickness of insulation such that the surface temperature doesn't fall below the wet-bulb temperature at the local condition.

10.1.1. Determining the insulation thickness from safety considerations

11

Determination of the economic pipeline diameter

The pipeline systems used in oil and gas industries are not only costly to purchase, but also to operate. While they require various types of auxiliary equipment for supporting and protecting the pipe system and instruments for flow measurement and control, the large pumps needed for operating these systems require a non-stop supply of electrical energy in order to overcome friction in the pipes, bends, and valves. Therefore, the design of pipelines must be a careful balance between their installation and running costs. In this respect, an important step in the design process is the selection of the pipeline diameter. Before computers and computer software became widely available as they are today, the engineers used an analytical method for selecting the optimum diameter that involves arduous calculus manipulations. Using the method, closed-form solutions can be obtained for water only by using the Hazen-Williams equation and for laminar flows only by using the Darcy-Weisbach equation. However, for turbulent flows in general the method requires the use of dimensionless charts in order to avoid iterative solutions. This chapter describes the traditional analytical method in details before presenting a much simpler Excel-aided method and demonstrating its generality for all fluids and all flows.

11.1. The objective function for pipeline optimisation

For economic optimisation of a pipeline, or any industrial pump-pipe system, the objective is to minimise the total cost of the system over its entire lifetime. Following the method described by Janna [1], the total cost for a pipeline system includes three components: (i) the installation cost (ii) the operation or running cost, and (iii) the maintenance cost as given by the following equation:

$$C_T = L\left(c_{PF} + c_m\right) + C_{OP} \tag{11.1}$$

Where, L is the pipe length, c_{PF} the annualised installation cost of the pipes, fittings, and pumps per metre-length of the pipe, c_m the annual maintenance cost fittings and pumps per metre-length of the pipe, and C_{OP} the annual cost of operation. How the three costs are evaluated is described below.

The installation cost of a pump-pipe system

Pipes come in standard sizes that are designated by two numbers which refer to the pipe's **nominal diameter** and **schedule**, e.g., a 6-nominal and schedule 40 pipe. The nominal size is associated with the outside diameter of the pipe, but not necessarily equal to it, while the pipe's schedule refers to its thickness. Table 11.1 lists some standard pipes with their outside and inside diameters. Schedule 40 is the standard for sizes up to nominal 12, while for larger sizes the standard schedule is 30 or 20. Pipe costs depend on their material and manufacturing process, nominal diameter, and schedule. Table 11.2 shows the costs per linear metre (c_P) in \$/m for commonly used pipes made of PVC [2], two grades of stainless-steel pipes [3], and two grades of copper tubes [4]. The cost of fittings and pumps (c_F) is taken as a fraction, F, of the pipe cost:

$$c_F = F \times c_P \tag{11.2}$$

Table 11.1. Selected standard-size pipes (adapted from Janna [1])

Nominal diameter	D_o(in)	D_o(cm)	D_i(cm)	**Nominal diameter**	D_o(in)	D_o(cm)	D_i(cm)
½	0.840	2.134	1.580	**12**	12.750	32.39	30.33
¾	1.050	2.667	2.093	**14**	14.000	35.57	33.65
1	1.315	3.340	2.664	**16**	16.000	40.64	37.73
1¼	1.660	4.216	3.504	**18**	18.000	45.72	43.81
1½	1.900	4.826	4.090	**20**	20.000	50.80	48.89
2	2.375	6.034	5.252	**22**	22.000	55.88	53.97
2½	2.875	7.303	6.271	**24**	24.000	60.96	59.05
3	3.500	8.890	7.792	**26**	26.000	66.04	64.13
4	4.500	11.43	10.23	**30**	30.000	76.20	74.29
6	6.625	16.83	15.41	**32**	32.000	81.28	79.34
8	8.625	21.91	20.27	**34**	34.000	86.36	84.45
10	10.750	27.31	25.46	**36**	36.000	91.44	89.53

Table 11.2. Costs of PVC and stainless-steel pipes and copper tubes per linear metre ($)

Nominal diameter	PVC [2]	Stainless steel 304 [3]	Stainless steel 316 [3]	Copper type L [4]	Copper type M [4]
¼				4.59	
⅜				6.69	5.22
½	1.94	5.64	8.33	8.10	5.94
¾	2.36	6.95	9.28	13.15	9.64
1	3.25	9.77	11.32	18.76	14.66
1¼	4.00	13.58	16.24	25.85	21.32
1½	4.66	15.48	20.37	33.26	29.39
2	6.26	21.88	24.99	53.46	46.31
2½	10.00	33.10	40.84	76.00	67.17
3	12.66	43.43	51.53	101.38	89.31
4	17.71	65.80	81.25	180.66	167.25
6	32.41	106.40	133.99	419.87	
8	43.39	184.30	218.78	791.27	
10	60.98	267.55	287.75		
12	80.23	321.05	340.86		
14	123.82				
16	156.06				
18	235.77				
20	275.95				
24	396.65				

Therefore, the total annualised cost of the pipe, fittings and pumps per metre-length of the pipe (c_{PF}) is then given by [1]:

$$c_{PF} = \alpha(c_p + c_F) = \alpha(1+F)c_P \tag{11.3}$$

Where a is the amortisation rate which depends on the annual rate of interest (i) and the number of payment years (N) according to the following relationship:

$$a = i \div \left[1 - \left(\frac{1}{1+i}\right)^N\right] \tag{11.4}$$

For sufficiently large values of N, the following approximate relationship can be used:

$$\alpha = 1/N \tag{11.5}$$

The maintenance cost

The annual maintenance cost (c_m) per metre-length of the installed pipe system is taken as a fraction, b, of the cost of the pipe, fittings and pumps [1], i.e.

$$c_m = b(c_P + c_F) = b(1+F)c_p \tag{11.6}$$

The operation cost

The operation cost is mainly the cost of electrical energy needed by the pumps to overcome friction losses in the pipes and fittings as well as the elevation differences in the system. The power needed to overcome friction losses is given by:

$$\dot{W}_f = \rho g Q h_f / \eta \qquad \text{[W]} \tag{11.7}$$

Where, ρ is the fluid density, g the gravitational acceleration, Q the volume flow rate through the pipe, h_f the head loss caused by friction in the pipe including minor losses, and η the combined pump-motor efficiency. In terms of mass flow rate ($\dot{m} = \rho Q$), the equation becomes:

$$\dot{W}_f = \dot{m} g h_f / \eta \qquad \text{[W]} \tag{11.8}$$

The power required for overcoming the difference in elevation between the inlet and exit of the pipe is given by:

$$\dot{W}_z = \dot{m} g (Z_2 - Z_1)/\eta \text{ [W]} \tag{11.9}$$

Where Z_1 and Z_2 are the elevations at the inlet and outlet points, respectively. The total annual operation cost is then given by:

$$C_{OP} = \left(\dot{W}_f + \dot{W}_z\right)c_2\tau / 1000 \quad (11.10)$$

Where c_2 is the cost of electricity per kilowatt-hour (\$/kW-h) and τ is the total number of operation hour per year. Substituting for $\dot{W}_f$ and $\dot{W}_z$ from Equation (11.8) and Equation (11.9), respectively, the total annual operation cost becomes:

$$C_{OP} = \frac{\left[h_f + \left(Z_2 - Z_1\right)\right]\dot{m}gc_2\tau}{1000 \times \eta} \quad (11.11)$$

The total annualised cost of the pipeline

Substitution of the installation, maintenance, and operation costs in Equation (11.1) yields the following equation for the total annualised cost of a pipeline:

$$C_T = \left(\alpha + b\right)\left(1 + F\right)Lc_p + \frac{\left[h_f + \left(Z_2 - Z_1\right)\right]\dot{m}gc_2\tau}{1000 \times \eta} \quad (11.12)$$

Equation (11.12) is the required objective function for least-cost pipeline optimisation. The equation can be used to determine the diameter at which the total pipeline cost is minimal by using either analytical or computer-aided methods. The analytical method described in the following section follows that given by Janna [1].

11.2. The analytical method for pipe-diameter optimisation

The optimum pipe diameter can be obtained analytically by differentiating Equation (11.12) with respect to the diameter (D) and equating the derivative to zero, i.e.:

$$\frac{\partial C_T}{\partial D} = \left(\alpha + b\right)\left(1 + F\right)L\frac{\partial c_p}{\partial D} + \frac{\dot{m}gc_2\tau}{1000 \times \eta}\frac{\partial h_f}{\partial D} = 0 \quad (11.13)$$

Note that the difference in elevation ($Z_2 - Z_1$) does not appear as a parameter in Equation (11.13) and, therefore, it does not affect the value of the optimum diameter (D_{opt}). The solution of Equation (11.13) yields the optimum diameter, but before the differentials of c_P and h_f, can be determined the analytical relationships for these two parameters in terms of the diameter D have to be provided. An easily differentiable relationship between the pipe cost c_P and its diameter D is given by the following power function:

$$c_p = c_1 D^n \quad (11.14)$$

Where c_1 and n are constants the values of which are determined by curve-fitting the pipe cost data. Figure 11.1 shows the power functions for the pipe materials given in Table 11.2 obtained by plotting the data on an Excel chart and using Excel's trendline feature to fit the data. Table 11.3 shows the corresponding values of c_1 and n for the five pipe types shown on Figure 11.1.

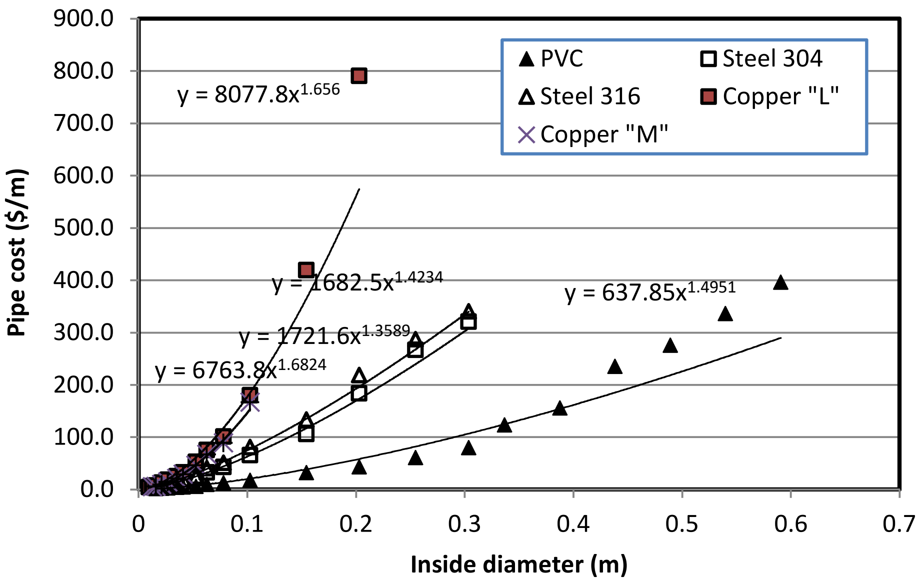


Figure 11.1. A plot of the pipe-cost data given in Table 11.2

Table 11.3. Values of the two constants c_1 and n in Equation (11.14) for five pipe materials

Pipe material	c_1	n
PVC	637.9	1.495
Stainless steel 304	1682.5	1.423
Stainless steel 316	1721.6	1.358
Copper (type L)	8077.8	1.656
Copper (type M)	6763.8	1.682

Substituting for c_p from Equation (11.14) and differentiating with respect to the pipe diameter, Equation (11.13) becomes:

$$n(\alpha + b)(1 + F)c_1 D^{n-1} L + \frac{\dot{m} g c_2 \tau}{1000 \times \eta} \frac{\partial h_f}{\partial D} = 0 \qquad (11.15)$$

Further progress with the analytical optimisation method depends on the formula used to evaluate the pipe friction h_f. The following sections consider two options: (i) the Hazen-Williams equation and (ii) the Darcy-Weisbach equation.

11.2.1. Optimisation with the Hazen-Williams equation

In the design of water pipe-systems, the pipe head loss is usually determined by using the following Hazen-Williams equation [5]:

$$h_f = \frac{10.67 L Q^{1.852}}{C^{1.852} D^{4.8704}} \qquad (11.16)$$

Where C is a roughness coefficient that depends on the pipe's material and surface condition. Typical C values for selected pipe materials are shown in Table 11.4. The equation is limited to water, but it applies for both laminar and turbulent flows.

Table 11.4. *C*-values in the Hazen-Williams equation for selected pipe materials [5]

Pipe	***C*-value**
Cast-iron (new)	130
Cast-iron (10 years)	107 – 113
Galvanised iron	120
Steel	90 – 110
Copper	130 – 140
Concrete	100-140
PVC	140

Equation (11.16) provides the necessary function for the friction loss in terms of the pipe diameter. Substituting for $\partial h_f / \partial D$ from Equation (11.16) in Equation (11.15) yields:

$$n(\alpha + b)(1 + F)c_1 D_{opt}^{n-1} L - 4.8704 \frac{10.67 \dot{m} g L Q^{1.852} c_2 \tau}{1000 \times C^{1.852} D_{opt}^{5.8704} \eta} = 0 \qquad (11.17)$$

Rearranging, Equation (11.17) yields:

$$D_{opt}^{n+4.8704} = \frac{51.9672 \dot{m} g Q^{1.852} c_2 \tau / 1000}{n(\alpha + b)(1 + F)c_1 C^{1.852} \eta} \qquad (11.18)$$

The absence of the pipe length, L, in Equation (11.18) means that it also does not affect the optimum diameter. By taking the density of water as 1000 kg/m^3, the equation can be simplified as follows:

$$D_{opt} = \left[\frac{509.7982 Q^{2.852} c_2 \tau}{n(\alpha + b)(1 + F)c_1 C^{1.852} \eta} \right]^{\frac{1}{n+4.8704}} \qquad (11.19)$$

Apart from avoiding an iterative solution, an important advantage of Equation (11.19) is that it can be used to determine the optimum diameter for both laminar and turbulent water flows. The following example illustrates the use of the equation.

Example 11-1. Optimum pipe diameter using the Hazen-Williams equation
A schedule 40 commercial steel pipe is used to convey water to a heat exchanger. The flow rate of water is 1 kg/s. Calculate the optimum economic diameter and the corresponding water velocity based on the following data:

c_1 = \$750/m$^{2.2}$
n = 1.2
b = 0.01
F = 7.0
N = 7 years
c_2 = \$0.04/kW.hr
τ = 7880 hr/year
η = 75%

Solution
For commercial steel $C = 90$
The volume flow rate $Q = \frac{\dot{m}}{\rho}$ =0.001 m^3/s.
The amortisation rate α = 1/7 = 0.143

Substituting in Equation (11.19) gives:

$$D_{opt} = \left[\frac{509.7982 \times 0.001^{2.852} \times 0.04 \times 7880}{1.2(0.143 + 0.01)(1+7) \times 750 \times 90^{1.852} \times 0.75}\right]^{\frac{1}{1.2+4.8704}}$$

= 0.022 m or 2.2 cm

The water velocity at the optimum diameter is calculated as follows:

$$V_{opt} = \frac{Q}{A_{opt}} = \frac{4Q}{\pi D_{opt}^2} = \frac{4 \times 0.001}{\pi \times 0.022^2} = 2.6 \text{ m/s}$$

Note that the water velocity is within the recommended range which is between 1.4 and 2.8 m/s [3]. The type of flow in the pipe can be determined from the velocity. Taking the absolute viscosity of water as 8.90x10^{-04} Pa.s, the corresponding Reynolds number is:

$$\text{Re} = \frac{0.022 \times 2.6}{8.9 \times 10^{-4}} = 64{,}589.91$$

The value of the Reynolds number indicates that the flow is turbulent. Referring to the data shown in Table 11.1, we realise that the obtained optimum diameter falls between two standard pipe diameters; which are ¾ inch and 1 inch. As a rule-of-thump, it is preferable to select the pipe with the larger nominal diameter, which is the 1-inch pipe.

11.2.2. Optimisation with the Darcy-Weisbach equation (laminar flow)

Using the Darcy-Weisbach equation, the pipe friction h_f in Equation (11.15) can be obtained from:

$$h_f = f\frac{L}{D}\frac{V^2}{2g} = f\frac{8L\dot{m}^2}{g\rho^2\pi^2 D^5} \tag{11.20}$$

The Darcy-Weisbach equation is advantageous over the Hazen-Williams equation because it can be used for various fluids including water. However, the friction factor depends on the Reynolds number, which in turn depends on the pipe diameter. For a laminar flow, the friction factor is given by the following linear relationship:

$$f = \frac{64}{Re} \tag{11.21}$$

Where Re is the Reynolds number ($\mathrm{Re} = 4\dot{m}/\mu\pi D$). The linear relationship between the friction factor and the Reynolds number for a laminar flow enables a closed-form solution to be obtained as shown below.

Substituting in Equation (11.20):

$$h_f = \left(\frac{64\mu\pi D}{4\dot{m}}\right)\left(\frac{8L\dot{m}^2}{g\rho^2\pi^2 D^5}\right) = \frac{128L\dot{m}^2}{g\rho^2\pi D^4} \tag{11.22}$$

Differentiating h_f with respect to the diameter, D:

$$\frac{\partial h_f}{\partial D} = \frac{\partial}{\partial D}\left(\frac{128\mu L\dot{m}}{g\rho^2\pi D^4}\right) = \frac{4\times 128\mu L\dot{m}}{g\rho^2\pi D^5} \tag{11.23}$$

Substituting from Equation (11.23) in Equation (11.15) we get:

$$n(\alpha + b)(1+F)c_1 D_{opt}^{n-1} L - \frac{\dot{m}gc_2\tau}{1000\times\eta}\left(\frac{4\times 128\mu L\dot{m}^2}{g\rho^2\pi D^5}\right) = 0 \tag{11.24}$$

Rearranging:

$$D_{opt}^{n+4} = \frac{0.512\mu\left(\dot{m}^2 c_2 \tau\right)}{n(\alpha + b)(1+F)c_1\left(\eta\pi\rho^2\right)} \tag{11.25}$$

Or:

$$D_{opt} = \left(\frac{0.512\,\mu\left(\dot{m}^2 c_2 \tau\right)}{n(\alpha + b)(1+F)c_1\left(\eta\pi\rho^2\right)}\right)^{\frac{1}{n+4}} \tag{11.26}$$

It should be noted that the optimum diameter determined by Equation (11.26) depends on the density as well as the viscosity of the fluid. Therefore, this equation can take into consideration the effect of temperature on the fluid's viscosity. The following example illustrates the procedure of using the equation.

Example 11-2. Optimum diameter for a laminar flow

Glycerine (ρ = 1252 kg/m^3, μ= 0.27 Pa.s) is to be pumped from a storage tank to a bottling machine at a rate of 0.34 l/s via a 10-m long schedule-40 pipe made of stainless steel 304. The pipe's connection to the bottling machine is 1 m higher than its inlet from the storage tank. Determine the optimum diameter for the pipe if the other rated data is as follows:

c_2 = \$0.12/kW.hr, F = 6.75, N = 10 years, η = 75%, τ = 7000 hr/yr, b = 0.01

Solution

From Table 11.3, for stainless steel 304, c_1 = \$1682/m$^{2.423}$ and n = 1.423.

$a = 1/N = 0.1$

$Q = 0.34/1000 = 0.00034$ m^3/s

$$\dot{m} = \rho Q = 1252 \times 0.00034 = 0.42568 \text{ kg/s}$$

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{4\dot{m}}{\pi D\mu} = \frac{4 \times 0.42568}{\pi \times 0.27 \times D} = \frac{2.00738}{D}$$

Since the numerator in the above equation (≈ 2.0) is small, we can assume that the flow is laminar and use Equation (11.26) to calculate the optimum diameter:

$$D_{opt} = \left[\frac{0.512 \times 0.27 \times (0.42568)^2 \times 0.12 \times 7000}{1.423(0.1+0.01)(1+6.75) \times 1682 \times 0.75 \times \pi \times 1252^2}\right]^{\frac{1}{1.423+4}}$$

= 0.02646 m or 2.65 cm.

We can now check the validity of our assumption that the flow is laminar by calculating the Reynolds number using the value obtained for the diameter:

$$\mathrm{Re} = \frac{2.00738}{0.02646} = 75.86$$

The small value of Re confirms that the flow is laminar.

11.2.3. Optimisation with the Darcy-Weisbach equation (turbulent flow)

Generally, the friction factor for turbulent flows depends on both the Reynolds number and roughness of the pipe and the relationship between the friction factor and the Reynolds number is nonlinear. This makes it difficult to use the procedure for the laminar flow described above in order to obtain an explicit formula for D_{opt} like Equation (11.26). Analytical solutions could be obtained for two limiting cases: (i) smooth pipes and (ii) pipes with complete turbulence. For smooth pipes the friction factor depends on the Reynolds number only and not on the pipe's roughness, while for pipes with complete turbulence the friction factor depends on the pipe roughness only and not on the Reynolds number. The earliest equation developed for pipe diameter optimisation with turbulent flows in smooth pips was the Genereaux equation [6], while Genic et al. [7] described an equation that can be used for pipes with complete turbulence.

Treating f as constant in Equation (11.20) and differentiating h_f with respect to D gives:

$$\frac{\partial h_f}{\partial D} = f\frac{-40L\dot{m}^2}{g\rho^2\pi^2 D^6} \tag{11.27}$$

Substituting in Equation (11.15) and rearranging gives:

$$D_{opt}^{n+5} = \frac{40f\,\dot{m}^3}{n(\alpha+b)(1+F)} \times \frac{c_2\tau/1000}{c_1\eta\pi^2\rho^2} \tag{11.28}$$

Or,

$$D_{opt} = \left[\frac{0.04f\,\dot{m}^3}{n(\alpha+b)(1+F)} \times \frac{c_2\tau}{c_1\eta\pi^2\rho^2}\right]^{\frac{1}{n+5}} \tag{11.29}$$

Since the friction factor depends on the pipe's diameter, Equation (11.29) is implicit in D, but can be used to determine D_{opt} by using a trial-and-error procedure. An initial guess for D_{opt} is made based on which the Reynolds number and friction factor are evaluated.

When substituted in the right-hand side of Equation (11.29), the result should be equal to D_{opt} itself. Otherwise, a new value for D_{opt} is tried until a reasonably close value is found.

To avoid the trial-and-error process, Janna [1] describes dimensionless graphs that directly determine the friction factor from two dimensionless numbers π_1 and π_2 defined as follows:

$$\pi_1 = \left(f.\mathrm{Re}^{n+5}\right)^{1/6} = \left[\frac{128}{5\pi^3}\frac{\dot{m}^2}{\mu^5}\left(\frac{4\dot{m}}{\pi\mu}\right)^n\left(\frac{n(\alpha+b)(1+F)c_1\eta\rho^2}{c_2\tau/1000}\right)\right]^{1/6} \tag{11.30}$$

$$\pi_2 = \frac{\varepsilon/D}{\mathrm{Re}} = \frac{\varepsilon}{D}\left(\frac{\pi D\mu}{4\dot{m}}\right) = \frac{\pi\varepsilon\mu}{4\dot{m}} \tag{11.31}$$

The second dimensionless number, π_2, is called the roughness number (Ro). Since π_1 depends on n, a graph is needed for each value of n. Figure 11.2 shows the graph which is applicable for the case n = 1.2. Janna [1] provides two more graphs for n = 1.0 and n=1.4. The two numbers π_1 and π_2 have to be calculated from the given problem data and used to read the corresponding friction factor off the respective graph. The value of f thus determined can then be substituted in Equation (11.29) to determine D_{opt}.

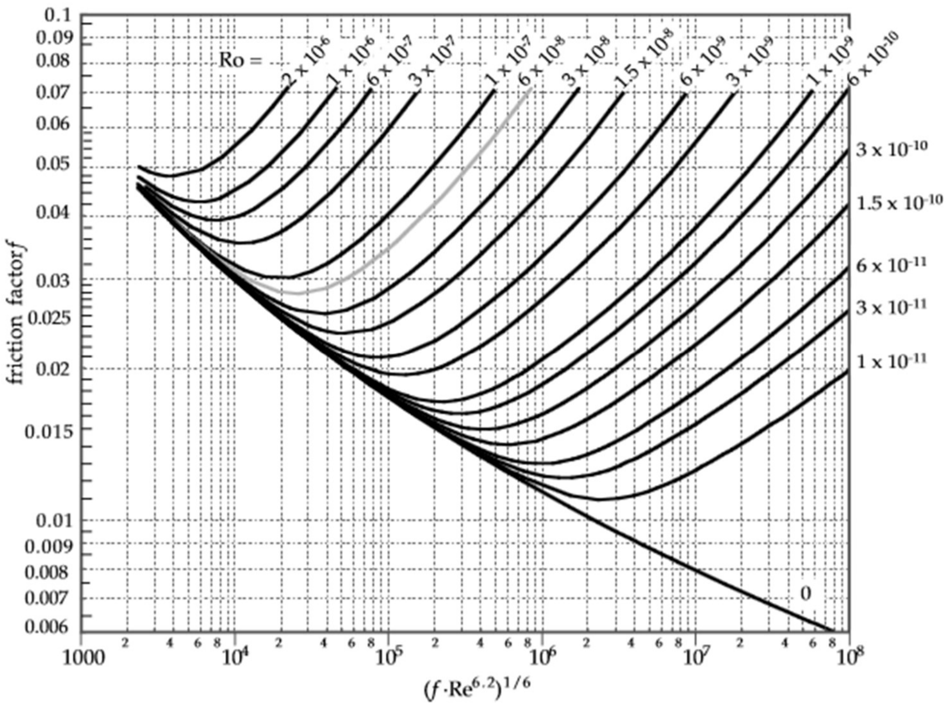


Figure 11.2. Friction factor graph for n = 1.2 (adopted from Janna [1])

The dimensionless graphs like the one shown on Figure 11.2 enable D_{opt} to be determined without a trial-and-error process, but each graph is only useful for a specific value of n. Since n differs with the pipe materials as shown in Table 11.3, a large number of such graphs would be required to cater for all possible values of n. Alternatively, Figure 11.2 can be used for other values of n by using the graph shown on Figure 11.3 that has been developed with Excel by following a similar procedure to that described by Alciatore and Janna [8] for preparing Figure 11.2. For a better accuracy, the following Cheng formula was also used for calculating the friction factor [1]:

$$f = \left[-2.0\log\left\{ \frac{\varepsilon}{3.7065D} - \frac{5.0452}{\mathrm{Re}} \log\left(\frac{1}{2.8257}\left[\frac{\varepsilon}{D}\right]^{1.1098} + \frac{5.8506}{\mathrm{Re}^{0.8981}} \right) \right\} \right]^{-2} \tag{11.32}$$

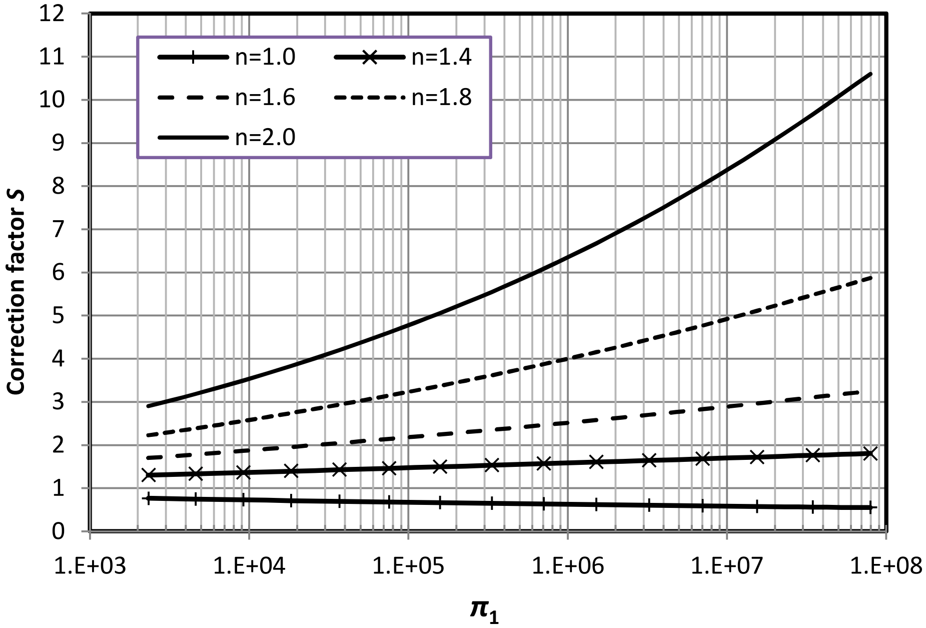


Figure 11.3. The adjustment factor S for reading the friction factor from Figure 11.2

With this new graph, the friction factor can be read from Figure 11.2 at the same values of π_2, but with an adjusted value of π_1, called π'_1, which is calculated from the following relationship:

$$\pi_1^{/} = \pi_1 / S \tag{11.33}$$

Where S is a correction factor that is read from Figure 11.3 given the value of n. With the friction factor thus determined, Equation (11.29) can be used to calculate D_{opt} following the same procedure described earlier. The following example illustrates the procedure.

Example 11-3. Optimum diameter for a turbulent flow
A type-M copper tube is to be sized for an air conditioner that uses Freon-22 (ρ= 1197 kg/m^3, μ=3.51x10^{-4} N.s/m^2). The flow rate is 1.0 l/s in the portion of the system that is to be selected. Determine the optimum economic diameter for the tubing where liquid refrigerant is being conveyed, given the following information:

c_2 = \$0.12/kW.hr
F = 7.0
Life time = 8 years
η = 62%
τ = 4500 hr/year
b = 0.01

Solution
From Table 11.1, for copper tube ε = 0.0015 cm, or 0.0000015 m. From Table 11.3, for type M copper tube c_1 = \$6763/m$^{2.682}$, n = 1.682.

α = 1/8 = 0.125

Q = 1.0/1000 = 0.001 m^3/s

$\dot{m} = \rho Q$ = 1197 x 0.001 = 1.197 kg/s

$$\mathrm{Re} = \frac{4\dot{m}}{\pi D\mu} = \frac{4\times 1.197}{\pi\times 3.51\times 10^{-4}\times D} = \frac{4.342\times 10^3}{D}$$

Note that the large value of the numerator in the Reynolds number indicates that the flow is turbulent. Substitution of the above data in Equation (11.29) gives:

$$D_{opt} = \left[\frac{0.04\times f\times 1.197^3}{1.682(0.125+0.01)(1+7)}\times\frac{0.12\times 4500}{6763\times 0.62\times\pi^2\times 1197^2}\right]^{\frac{1}{1.682+5}}$$

$$= 0.0383 f^{0.15}$$

The two dimensionless quantities π_1 and Ro are now calculated as follows:

$$\pi_1 = \frac{128}{5\pi^3}\frac{1.197^2}{\left(3.51\times 10^{-4}\right)^5}\left(\frac{4\times 1.197}{\pi\times 3.51\times 10^{-4}}\right)^{1.2}\left(\frac{1.682((1/8)+0.01)(1+7)\times 6763\times 0.62\times 1000^2}{(0.12/1000)\times 4500}\right)^{1/6}$$

= 4.25047x10^5

$$\text{Ro} = \frac{\pi \times 0.0000015 \times 3.51 \times 10^{-4}}{4 \times 1.197} = 3.455\text{x}10^{-10}$$

The value of the friction factor can be determined by using the graph on Figure 11.2 at the same value of Ro, but with an adjusted value of π_1 given by Equation (11.33). The value of S from Figure 11.4 at $n = 1.67$ and $\pi_1 = 4.25\text{x}10^5$ is approximately 3.0. Therefore, the adjusted value $\pi'_1 = 4.25047\text{x}10^5/3.0$, or $1.41683\ \text{x}10^5$.

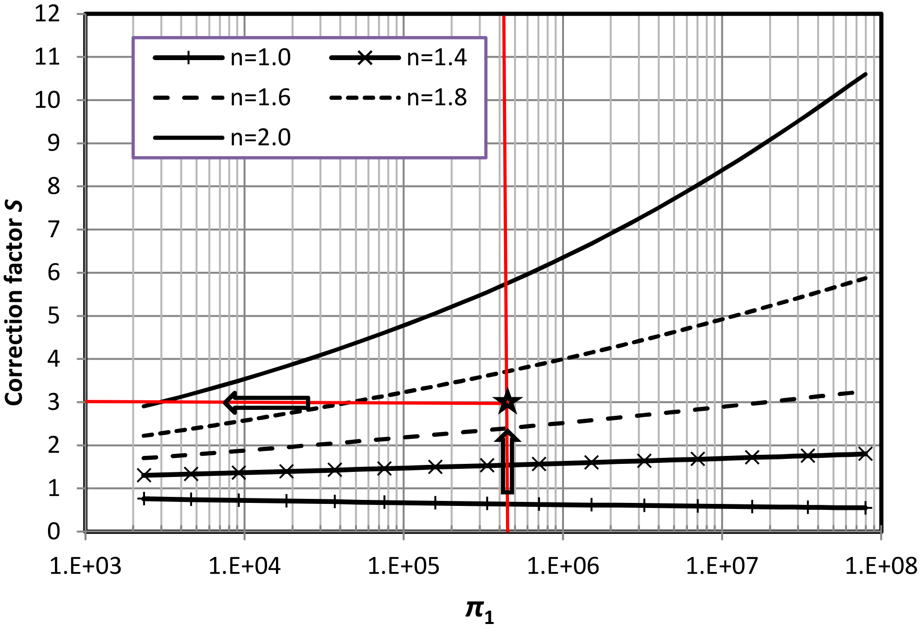


Figure 11.4. Determining the correction factor S for Example 11-3

At $\pi_1 = 1.41683\text{x}10^5$ and Ro = $3.455\text{x}10^{-10}$, Figure 11.2 gives $f \approx 0.016$. Therefore, the optimum diameter is:

$$D_{opt} = 0.0383 f^{0.15} = 0.0383\text{x}(0.016)^{0.15} = 0.02065 \text{ m or } 2.065 \text{ cm}$$

We can now determine the Reynolds number using the calculated value of the optimum diameter of D_{opt}.

$$\text{Re} = 4.34 \times 10^3 / 0.02065 = 210{,}248.77$$

The value of Re confirms that the flow is turbulent. The accuracy of the procedure can now be checked by calculating the friction factor based on the determined value of D_{opt} and comparing it with that obtained earlier from Figure 11.3. Using Equation (11.32), the friction factor is:

$$f = \left[-2.0\log\left\{ \frac{0.0000015}{3.7065 \times 0.02065} - \frac{5.0452}{210248.77}\log\left(\frac{1}{2.8257}\left[\frac{0.0000015}{0.02065}\right]^{1.1098} + \frac{5.8506}{210248.77^{0.8981}} \right) \right\} \right]^{-2}$$

$= 0.0161$

This value of f is the same as the value found earlier from Figure 11.2.

11.3. The Excel-Solver method for pipe-diameter optimisation

The analytical method for determining the economic pipe diameter suffers from two drawbacks the first of which is that it does not provide a general methodology for dealing with both laminar and turbulent flows. Although this can be avoided by using the Goal Seek command or Solver to perform the iterative solution required by the turbulent flow, the analytical method still suffers from the second drawback which is the need to use the pipe cost-diameter power relationship given by Equation (11.14), or another similar equation, which may not accurately represent the pipe-cost data. In general, determining the optimum pipe diameter by minimising the total cost does not require the use of the power relationship which is only needed for convenience of applying the analytical method. By using Excel, the total cost can easily be minimised with Solver. The Excel-Solver method described in this section enables the pipe-cost relationship to be represented by a more accurate function than the power function either by using Excel's "trendline" feature to obtain a suitable polynomial or by using the interpolation function "**Interpl1**" provided by the Thermax add-in and described in Appendix B. Unlike Equation (11.14), the interpolation function returns the exact values given in the table.

To illustrate the Excel-Solver method, let us go back to the pump-pipe system in Example 11-3 and determine its optimum diameter by using Equation (11.12) instead of Equation (11.29). The Excel sheet developed for this purpose is shown on Figure 11.5.

cpo | fx =Interpl1(D,Sheet2!D3:D12,Sheet2!E3:E12,9)

	B	C	D	E	F	G	H	I	J	K	L	M
4	Life	8	year	ε	1.5E-06	m	D	0.04	m	cpo	28.14956	$
5	a	0.125	per 10 yr							C_PT	30.40153	$
6	b	0.01		Qlitre	1	l/s	Q	0.001	m3/s	C_op	0.146883	$
7	FF	7					m	1.197	kg/s			
8	C_2	0.12	$/kW.hr	L	1	m	Re	108551.8		C_T	30.54841	$
9	t	4500	hr/yr	H_1	1	m	f	0.017799				
10	η	0.62		H_2	1	m	V	0.795775	m/s			
11	g	9.81	m/s2				hf	0.014362	m			
12				ρ	1197	kg/m3	Wf	0.168644	W			
13				μ	3.51E-04	Pa.s						
14												

Figure 11.5. The Excel sheet developed for Example 11-3

The given data are stored on the left part of the sheet. The Excel formula that calculates the friction factor enables the sheet to be used for both laminar and turbulent flows. For turbulent flows, the sheet uses the Swamee-Jain equation for the friction factor, but a more accurate equation, such as the Cheng equation, can also be used. In order to determine the pipe cost by the linear interpolation function, the pipe-cost data is stored in Sheet 2 of the same workbook as shown on Figure 11.6.

J24 fx

	A	B	C	D	E	F
1					Copper M	
2		Pipe	Di(cm)	Di(m)	$/m	
3		0.375	1.252	0.01252	5.2152	
4		0.5	1.58	0.0158	5.9368	
5		0.75	2.093	0.02093	9.6432	
6		1	2.664	0.02664	14.6616	
7		1.25	3.504	0.03504	21.32	
8		1.5	4.09	0.0409	29.3888	
9		2	5.252	0.05252	46.3136	
10		2.5	6.271	0.06271	67.1744	
11		3	7.792	0.07792	89.3144	
12		4	10.23	0.1023	167.2472	
13						

Figure 11.6. Cost data for the type-M copper tube in Example 11-3

The formula bar on Figure 11.5 shows the following formula which is used to determine the unit cost (cpo) from the piper diameter(D) with the interpolation function "**Interpl1**":

=Interpl1(D;Sheet2!D3:D12;Sheet2!E3:E12;9)

Based on the guessed pipe diameter of 0.04 m, the sheet calculates the Reynolds number (Re), friction factor (f), friction losses (hf), and the power needed to overcome friction losses (W). It then determines the pipe unit cost (cpo), annualised cost of the pipe, pump and fittings, and annual maintenance cost (C_PT), and cost of operation (C_op) which are stored in cells **L4 – L6**. The total annualised cost (C_T) in cell **L8** is the summation of C_PT and C_op. Figure 11.5 shows that the total cost for the guessed pipe diameter of 0.04 m is about $30.55, which is not the minimum cost. Solver's set-up for determining the diameter that minimises the total cost is shown on Figure 11.7. Two constraints have been imposed for the lower and upper values of the diameter. With this set-up, the GRG Nonlinear method of Solver failed when automatic-scaling was used. Figure 11.8 shows the solution obtained without automatic-scaling, which is 2.093 cm. Although this solution is more accurate than the 2.065-cm diameter obtained in Example 11-3 by using the analytical power function, the difference in not significant since both methods led to a nominal diameter of ¾ inch.

The developed Excel sheet can also be utilised to assess the sensitivity of the optimum diameter to possible changes in the costs involved. By adjusting the pipe diameter, the sheet shown on Figure 11.5 was used to calculate the total cost at various diameters as shown on Figure 11.9.

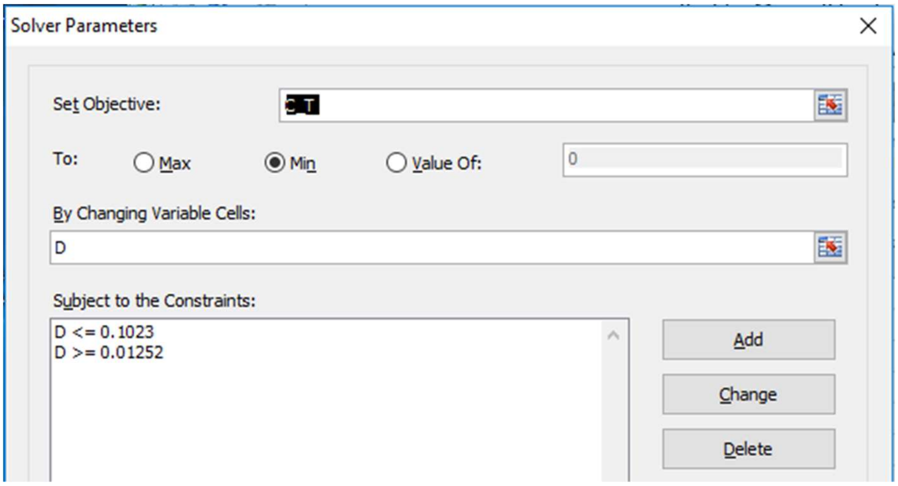
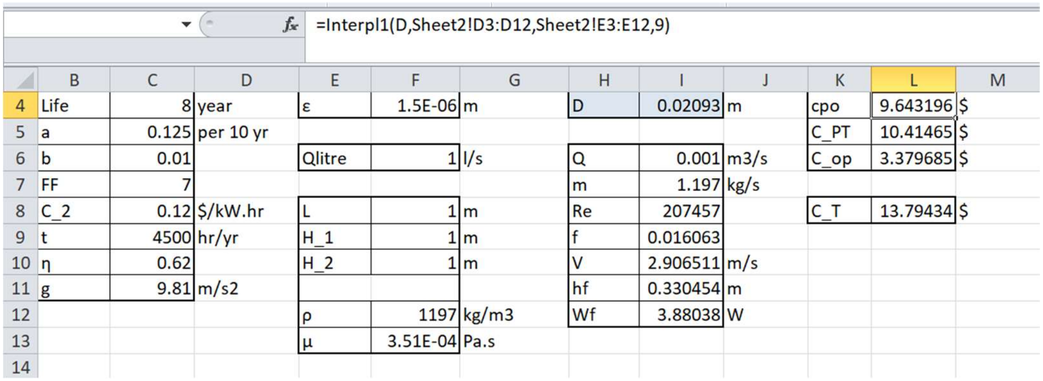


Figure 11.7. Solver's set-up for Example 11-3



=Interpl1(D,Sheet2!D3:D12,Sheet2!E3:E12,9)

	B	C	D	E	F	G	H	I	J	K	L	M
4	Life	8	year	ε	1.5E-06	m	D	0.02093	m	cpo	9.643196	$
5	a	0.125	per 10 yr							C_PT	10.41465	$
6	b	0.01		Qlitre	1	l/s	Q	0.001	m3/s	C_op	3.379685	$
7	FF	7					m	1.197	kg/s			
8	C_2	0.12	$/kW.hr	L	1	m	Re	207457		C_T	13.79434	$
9	t	4500	hr/yr	H_1	1	m	f	0.016063				
10	η	0.62		H_2	1	m	V	2.906511	m/s			
11	g	9.81	m/s2				hf	0.330454	m			
12				ρ	1197	kg/m3	Wf	3.88038	W			
13				μ	3.51E-04	Pa.s						
14												

Figure 11.8. Solver's solution of Example 11-3

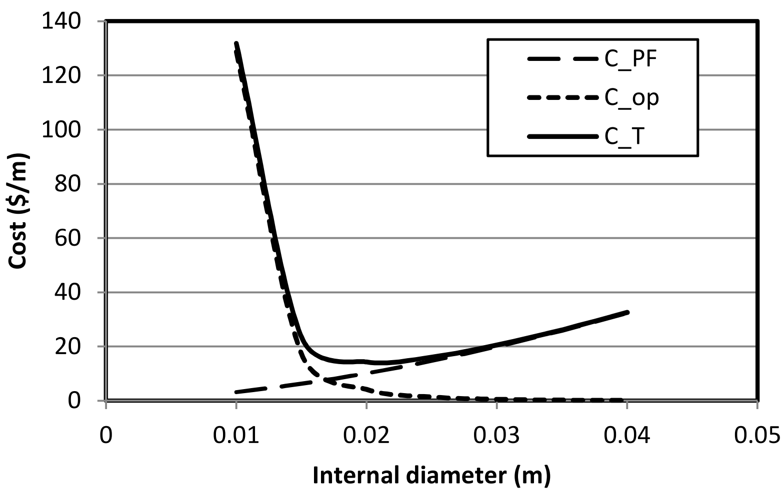


Figure 11.9. Variation of the costs of the pipe in Example 11-3 with its diameter

Figure 11.9 reveals that a pipe internal diameter of up to 2.7 cm can be used without substantially increasing the total cost of the system. Therefore, it is wise to select a 1-in nominal-diameter pipe so as to cater for future increases in the operating costs.

11.4. Closure

This chapter deals with the economic optimisation of pipeline diameters by using both analytical and computer-aided optimisation methods. The analytical method that obtains the optimum pipe diameter by differentiating the total annualised-cost function needs an analytic function for the price of standard pipe sizes in terms of the diameter. With this method an equation for determining the optimum pipe diameter could be obtained for both laminar and turbulent flows in water-transporting pipes by using the Hazen-Williams equation to calculate friction losses. However, by using the more general Darcy-Weisbach equation, an explicit equation for the optimum pipeline diameter could only be obtained for a laminar flow. For a turbulent flow, the analytical method determines the optimum diameter by using non-dimensional graphs in order to avoid the iterative solution involved. The chapter demonstrates the generality of the computer-aided method that uses Excel and Solver to solve the basic objective function for optimisation with the Darcy-Weisbach equation and replaces the power function used in the analytical method with a direct interpolation of the diameter-cost data.

References

[1] W.S. Janna, *Design of Fluid Thermal Systems*, third edition, CENGAGE Learning, 2011
[2] USP United States Plastic Corporation, http://www.usplastic.com/
[3] British Metals LLC. Brismet, http://brismet.com/
[4] Copper tube: www.camlee.com/wp-content/uploads/2013/06/CPRICE.pdf
[5] Wikipedia contributors, https://en.wikipedia.org/wiki/Hazen%E2%80%93Williamsequation
[6] R.P. Genereaux, Fluid-flow design methods, Industrial and Engineering Chemistry, 29 (1937) 385-388.
[7] S.B. Genic, B.M. Jacimovic, and V.B. Benic, Economic optimization of pipe diameter for complete turbulence. Ene7rgy and Buildings, 45 (2012), 335-338.
[8] D. Alciator and W.S. Janna, Modified pipe friction diagrams that eliminate trial-and-error solutions, Proc. Of the National Fluid Dynamics Congress, Part 2, July, 1988, available at: https://www.engr.colostate.edu/~dga/dga/papers/pipefriction.pdf

12

Pipe-network optimisation

Considering the high initial and running costs of large pipe-networks, their design process may not only seek to minimise the total owning cost of the network but to maximise its social benefits as well as minimise its negative effects on the environmental. Therefore, the network's layout itself can be one of the design considerations. This chapter extends the methodology presented in Chapter 8 for hydraulic analyses of pipe-networks with pre-determined layouts by adding an objective function for economic optimisation of the pipe-networks. The chapter demonstrates the usefulness of Excel and Solver for the optimisation of both gravity-fed and pump-fed pipe-networks. The effect of the formula used for calculating the major pipe friction losses on the analyses results is studied by considering three options which are; the Darcy-Weisbach equation with the Swamee-Jain formula, the Darcy-Weisbach equation with the Colebrook-White formula, and the Hazen-Williams equation.

12.1. The pipe-network optimisation problem

The least-cost pipe-network optimisation problem that seeks to determine the minimum total owning cost of the network is hard to solve because of three reasons: (i) the non-linearity of the equations involved (ii) the availability of pipes in discreet sizes and costs, and (iii) the requirement to satisfying certain discharge and pressure criteria. For illustration, consider the two-loop gravity-fed pipe network shown on Figure 12.1 which was first presented by Alperovits and Shamir [1] as a test case for their network optimisation method. The network carries water from an elevated tank to 7 consumption points via 8 pipes all of which are 1000-m long and have the roughness of cast-iron (ε = 0.26 mm, C = 130).

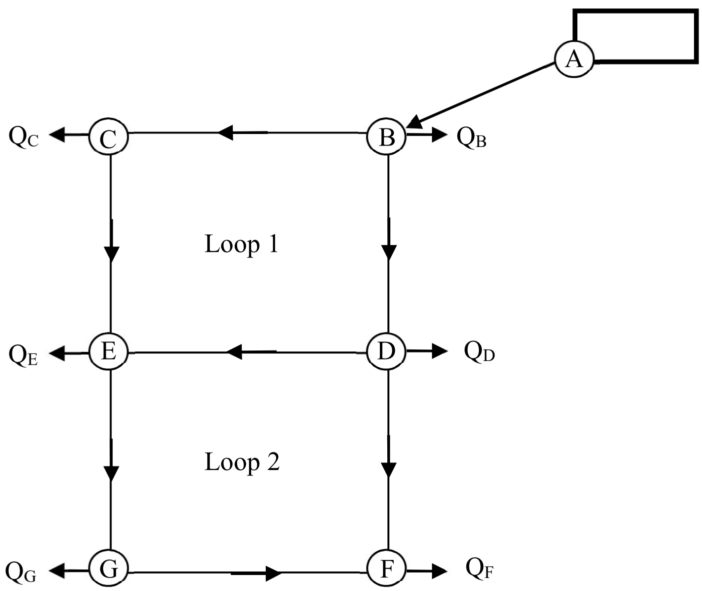


Figure 12.1. A two-loop gravity-fed pipe network

The minimum pressure level at any consumption point is required to be 30 metres column of water (m.c.w). Alperovits and Shamir [1] sought to minimise the network's total installation cost by selecting from 14 available pipe sizes with the costs shown in Table 12.2.

Table 12.1. Node data for the two-loop network

Node	Node demand (m^3/h)	Ground level (m)
1	(Reservoir) -1120	210
2	100	150
3	100	160
4	120	155
5	270	150
6	330	165
7	200	160

Table 12.2. Cost data for the two-loop network

Diameter (in)	Diameter (m)	Cost (units)	**Diameter (in)**	Diameter (m)	Cost (units)
1	0.0254	2	**12**	0.3048	50
2	0.0508	5	**14**	0.3556	60
3	0.0762	8	**16**	0.4064	90
4	0.1016	11	**18**	0.4572	130
6	0.1524	16	**20**	0.5080	170
8	0.2032	23	**22**	0.5588	300
10	0.2540	32	**24**	0.6096	550

Though much simpler than most pipe networks met in practice, the total number of possible combinations for the 8 pipes forming the network and the 14 available pipe sizes is 14^8. Therefore, trying to determine the pipe diameters that minimise the total cost by simple trial-and-error would be a futile effort. Engineers usually apply certain rule-of-thumps, like specifying a range for the economic velocity, so as to minimise the number of possible pipe arrangements, but the resulting pipe diameters determined this way are unlikely to ensure optimality. Numerous attempts have been made to develop computer models for pipe-network optimisation over the last four decades and, unlike hydraulic analyses of pipe-networks, the development of optimisation methods for pipe-network is still an active area of research. For detailed treatment of this topic, the reader may refer to the more specialised sources listed as references [2-4].

12.2. The analytical model for pipe-network optimisation

Pipe-network optimisation has three main aspects: (i) the hydraulic-analysis model, the objective function for optimisation, and (iii) the computer-based solution method. The three aspects of the Excel-Solver method are described below by referring to the two-loop pipe-network shown on Figure 12.1.