



Effective Modeling and Simulation of Engineering Problems with COMSOL Multiphysics

¹Oladejo K. A., ²Abu R., ³M. D. Adewale

¹Department of Mechanical Engineering, Obafemi Awolowo University, Ile-Ife, Nigeria.

²Department of Mechanical Engineering, University of Ibadan, Nigeria.

³School of Science and Technology, National Open University of Nigeria, Nigeria

ABSTRACT

Mastery of finite element theory is necessary to write finite element program, but it is not necessary to use COMSOL Multiphysics package. It is only necessary to understand the physical problem at hand, how to use the finite element program, and how to interpret the results. This paper gives an introduction to modeling in COMSOL Multiphysics, as being employed in the course heat transfer, by discussing the basic steps of modeling process, from generating the geometry to post-processing. This was demonstrated by applying the package to steady-state heat conduction in a domain. Three different temperature profiles, of different orientation, were obtained from the domain, and COMSOL Multiphysics automatically generated the graphical illustration of the temperature profile. The output of the package was compared with the solutions generated by analytical application of finite difference method to the same case. A spreadsheet program was used to graphically illustrate the profile for the analytical solution. The results were in good agreement. The evaluation illustrates the powerful and flexible nature of COMSOL for solving scientific and engineering problems; and as teaching and research tool in engineering education.

Keywords: *COMSOL Multiphysics, FEM, Gauss – Seidel method*

1. INTRODUCTION

Modeling and simulation are becoming indispensable tools in design, optimization, and research. An important part of these tasks is the estimation of the influence of different parameters that describe a model or simulation [1]. COMSOL Multiphysics is a finite element modeling software that is applicable to all fields of engineering and science. It comprises tailored modeling solutions for Structural Mechanics, Chemical Engineering, and Electromagnetic. It facilitates modeling of all physical phenomena based on Partial Differential Equations, PDEs, such as heat transfer, stress and strain analysis, and fluid flow [2]. The model equations, or physics, can either be applied in a ready-to-use form, or specified freely to suit any type of physical phenomenon (linear, non-linear or time dependent). It is used mainly when hand calculations cannot provide accurate results. This is often the case when the geometry or process in question is very complex. When solving the PDEs that describe a model, COMSOL applies the finite element method (FEM). It runs that method in conjunction with adaptive meshing and error control as well as with a variety of numerical solver [3].

Finite element analysis (FEA) was first termed by R.W. Clough in a paper published in 1960, but the roots of the theory relates back to the Ritz method of numerical analysis, first introduced in 1909. Using the calculus of variations, R. Courant applied the Ritz method to obtain “piecewise approximations” of solutions for problem of equilibrium and vibrations in 1943 [4]. Further development of these ideas continued through the 1940’s and 50’s. By 1953, engineers began to use computers to solve structural problems. The paper by Turner, Clough, Martin, and Topp, published in 1956 is considered a major turning point in the development of FEA. The paper dealt with the “stiffness and deflection of

complex structures” and contributed to increased interest in the method. In 1960 the term “finite element” was first used, and around this time numerical methods began to be widely used in the aerospace industry [5]. In 1963, FEA was recognized as a specific technique, and a serious academic discipline. This led to much greater research, leading to its application for heat transfer and fluid mechanics problems in addition to structural. By the early 1970’s, FEA had become established as a general numerical technique for the solution of any system of differential equations. The method was still only used at this time on a limited basis, however, because of the availability of powerful computers in industry [6]. The use was primarily in the aerospace, automotive, defense, and nuclear industries.

With the advent of micro-computers in the 1980’s, however, the methods have become more widely used. It is now possible for engineers in virtually every industry to take advantage of this powerful tool. COMSOL Multiphysics is used in research, product development and teaching, in such fields as: Acoustics, Antennas, Bioscience, Bioengineering, Chemical reactions, Diffusion, Ecology, Electromagnetic, Environmental science, Fluid dynamics, Fuel cells, Geophysics, Heat transfer, Math/Applied PDEs, MEMS, Microwave engineering, Nanotechnology, Optics and photonics, Physics, Porous media flow, Quantum mechanics, Radio frequency components, Semiconductor devices, Structural mechanics, Transport phenomena, Wave propagation, Any combination of the above etc [7]. Oladejo et al. [8] presented an in-house built model to solve two-dimensional conduction heat transfer in a square domain using the concept of finite difference method.

The main aim of this study is to demonstrate the standard process of modeling using the package for heat transfer problems. The package is used to simulate steady-state



temperature distribution for a 2-D domain. This is because the analytical solutions can be easily obtained for proper comparison. The implementation of the package will reduce frustration for students and provide better platform to understand the basic principles of thermal analysis.

2. PACKAGE IMPLEMENTATION

Problem definition

A flat metal plate of dimensions 1.2 m x 1.2 m is subjected to a temperature of 100° C on one side, 25° C on two sides and 0° C on the fourth side as shown in Figure 1.

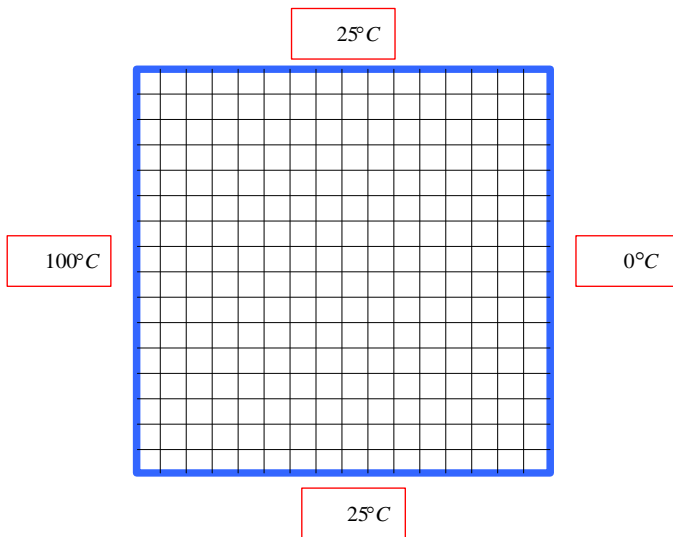


Figure 1: Graphical Illustration of the Problem

3. ALGORITHM FOR THE COMSOL SOLUTION

- (a) Start COMSOL, a Model Navigator screen appears.
- (b) Select an application mode from the Model Navigator.
 - (i) Set the **Space dimension to 2D**.
 - (ii) Click on + of **Heat Transfer**.
 - (iii) Click on + **Conduction**.
 - (iv) Click on **Steady-state Analysis**.
 - (v) Select Element: **Lagrange – Linear** to use linear elements. (Note that the default element is always quadratic).
 - (vi) Click **OK**. This is illustrated in Figure 2.

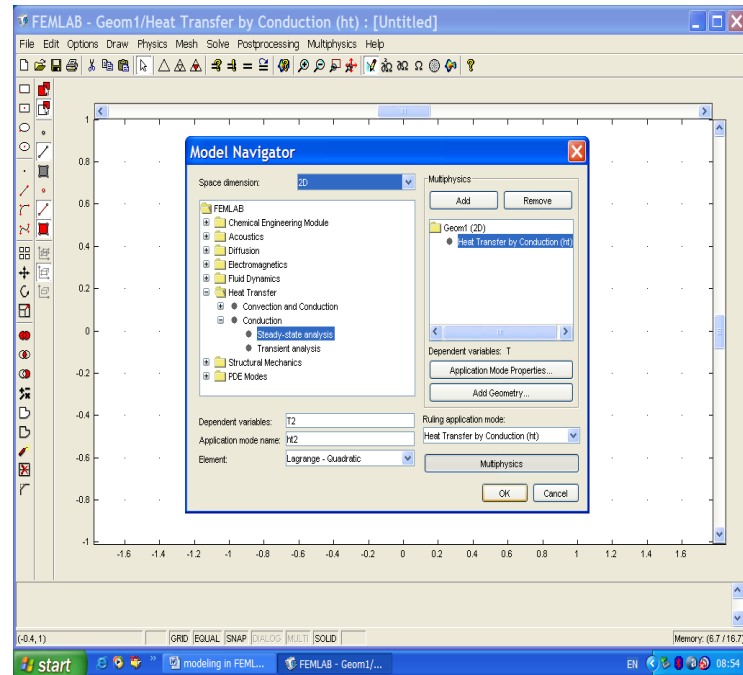


Figure 2: Model Navigator Interface

- (c) **Draw the Geometry in Draw Mode**
 - (i) Click on **Rectangle/Square** on the left-hand toolbar.
 - (ii) Click at **(0, 0)** and, whilst holding down the mouse button, drag the cursor to **(1.2, 1.2)** and then release the mouse button.
 - (iii) Click on **Zoom Extents** on the top horizontal toolbar to get a better view.
- (d) **Set the Sub domain Settings and Boundary Conditions in Physics mode.**
 - (i) Select **Physics -> Boundary Settings . . .**
 - (ii) In Boundary Selection, click on boundary **1** to highlight it. Look on the drawing to see which boundary this is.
 - (iii) Select Boundary Condition: **Temperature**.
 - (iv) Change the temperature T_0 to 100.
 - (v) In Boundary Selection, click on boundary **2** to highlight it. Look on the drawing to see which boundary this is.
 - (vi) Select Boundary Condition: **Temperature**.
 - (vii) Change the temperature T_0 to 25.
 - (viii) In Boundary Selection, click on boundary **3** to highlight it. Look on the drawing to see which boundary this is.
 - (ix) Select Boundary Condition: **Temperature**.
 - (x) Change the temperature T_0 to 25.
 - (xi) In Boundary Selection, click on boundary **4** to highlight it. Look on the drawing to see which boundary this is.
 - (xii) Select Boundary Condition: **Temperature**.



- (xiii) Change the temperature T_0 to 0.
- (xiv) Press **OK**.

- (e) **Initialize a finite element mesh in Mesh mode.**
A mesh is partition of the geometric model into small units of simple shapes. Select Mesh -> Initialize Mesh.
- (f) **Solve the problem in Solve mode.**
Select Solve -> Solve Problem.
- (g) **Visualize the Solution in Post-processing mode.**
The package automatically goes from solve mode into post-processing mode. The temperature distribution in the plate (as automatically generated by the package) is shown in Figure 3.

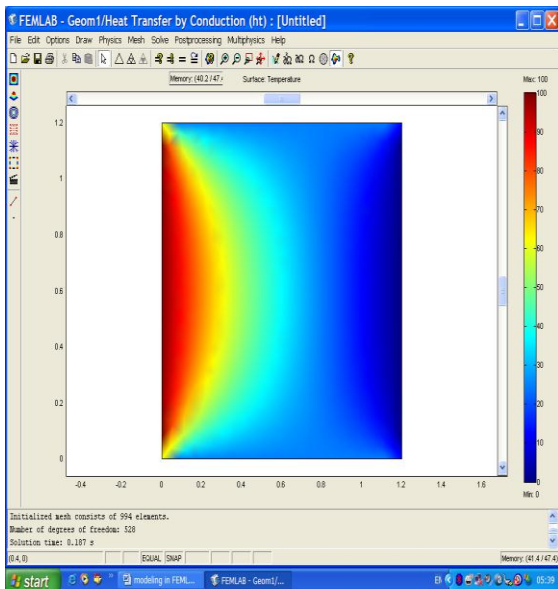


Figure 3: Temperature distribution resulting from the boundary conditions.

- (i) Click on **Draw Line for Cross-Section Line Plot** on left-hand toolbar. Press the mouse button down when the cursor is at **(0.4, 0)**, drag the cursor to **(0.4, 1.2)** and release the mouse button. A cross-section temperature plot along this line should be displayed. (Temperature profile on vertical line).
- (ii) Click on **Draw Line for Cross-Section Line Plot** on left-hand toolbar. Press the mouse button down when the cursor is at **(0, 0.8)**, drag the cursor to **(1.2, 0.8)** and release the mouse button. A cross-section temperature plot along this line should be displayed. (Temperature profile on horizontal line).
- (iii) Click on **Draw Line for Cross-Section Line Plot** on left-hand toolbar. Press the mouse button down when the cursor is at **(0, 0)**, drag the cursor to **(1.2, 1.2)** and release the mouse button. A cross-section temperature

plot along this line should be displayed. (Temperature profile on diagonal line).

4. ANALYTICAL SOLUTION OF THE CASE STUDY

The method of solution is to iterate through all the grid points, calculating a better approximation to the temperature at each point (i, j) in turn from the equation (1) as diagrammatically shown in Figure 4 [9].

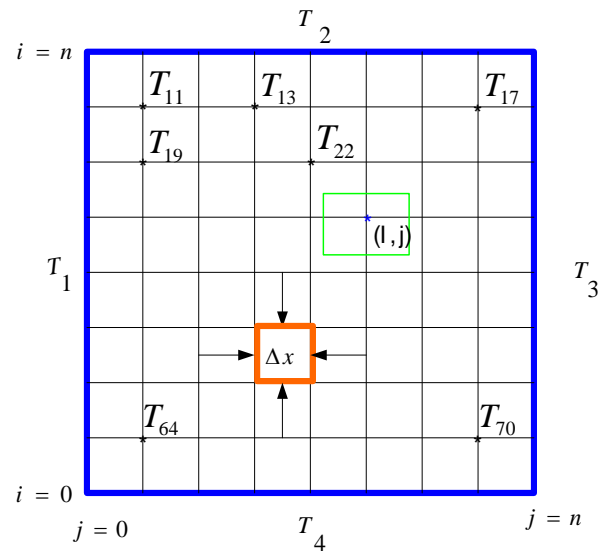


Figure 4: Two-dimensional finite difference element configuration

$$T_{i,j} = \frac{T_{i-1,j} + T_{i+1,j} + T_{i,j-1} + T_{i,j+1}}{4} \tag{1}$$

Where,

$$i = 1, 2, 3, \dots, n-1, \text{ and} \\ j = 1, 2, 3, \dots, n-1.$$

Equation (1) simplifies the Gauss – Seidel method of iteration. To start, the minimum temperature of the boundary condition is assumed everywhere within the plate. The process of iteration through all grid points is repeated until further iterations would produce, it is hoped, very little change in the computed temperatures. The iteration stops if the sum ϵ , over all grid points, of the absolute values of the deviations of the temperatures from their previously computed values, falls below a small quantity ϵ_{max} . Computations will also be discontinued if the number of complete iterations, k, exceeds an upper limit, k_{max} [10]. For the problem, diagrammatically shown in Figure 5, there are four internal nodes and three elements on a row so there will be four equations to evaluate four unknown variables.

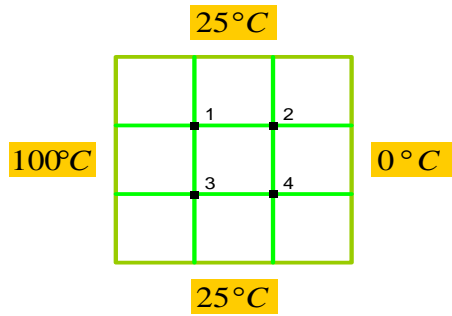


Figure 5: Graphical illustration of nodes for analytical solution

Node 1,

$$T_2 + 100 + 25 + T_3 - 4T_1 = 0$$

$$T_1 = (T_2 + T_3 + 125) / 4 \tag{2}$$

Node 2,

$$0 + T_1 + 25 + T_4 - 4T_2 = 0$$

$$T_2 = (T_1 + T_4 + 25) / 4 \tag{3}$$

Node 3,

$$T_4 + 100 + T_1 + 25 - 4T_3 = 0$$

$$T_3 = (T_1 + T_4 + 125) / 4 \tag{4}$$

Node 4,

$$0 + T_3 + T_2 + 25 - 4T_4 = 0$$

$$T_4 = (T_2 + T_3 + 25) / 4 \tag{5}$$

Gauss-Seidel method of iteration was used to solve the linear equations (2), (3), (4) and (5) as shown in Table 1. The values of temperatures at some grid points on the horizontal, vertical and diagonal lines defined in Figure 6 are shown in Table 2.

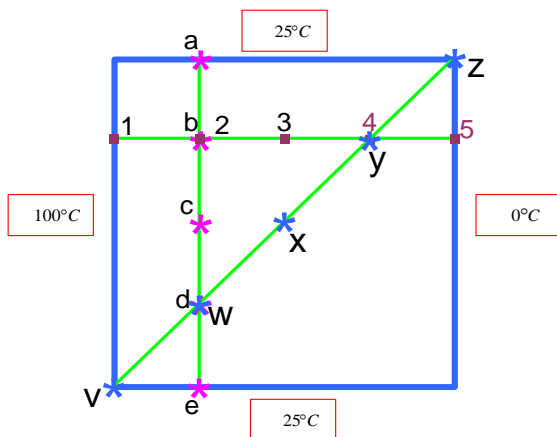


Figure 6: Illustration of problem definition with critical points in the domain.

Table 1: Analytical Solution by Gauss-Seidel Iteration

N	T_1	T_2	T_3	T_4
0	0	0	0	0
1	31.250	14.060	39.060	19.530
2	44.530	22.270	47.270	23.640
3	48.640	24.320	49.320	24.660
4	49.660	24.830	49.830	24.920
5	49.920	24.960	49.960	24.980
6	49.980	24.990	49.990	24.990
7	49.995	24.996	49.996	24.998
8	49.998	24.999	49.999	25.000

Table 2: Temperatures at Some Points in the Domain

Vertical		Horizontal		Diagonal	
Poin t	Temperatur e	Poin t	Temperatur e	Poin t	Temperature
a	25.0000	1	100.0000	v	65.5000
b	49.9990	2	49.9980	w	49.9990
d	49.9980	4	24.9990	y	24.9990
e	25.0000	5	0.0000	z	12.5000

5. DISCUSSION OF RESULTS

From the analytical solutions obtained from the 8th iteration of the FDA, stated in Table 1, one would be able to get the steady-state temperatures for points (a,b,d,e), (1,2,4,5), and (v, w, y, z) as depicted in the domain shown in Figure 6. These are clearly shown in Table 2.

A spreadsheet package [11] is used to graphically illustrate the temperatures for easy comparison with that of COMSOL Multiphysics package. Figure 7 and Figure 8 show COMSOL Multiphysics generated temperature profile for the vertical line and analytically generated temperature profile for the vertical line respectively. Figure 9 and Figure 10 show COMSOL Multiphysics generated temperature profile for the horizontal line and analytically generated temperature profile for the horizontal line respectively. Figure 11 and Figure 12 show COMSOL Multiphysics generated temperature profile



for the diagonal line and analytically generated temperature profile for the diagonal line respectively.

The minor deviation of the temperature profile in the analytical solution from that of COMSOL Multiphysics is primarily due to poor modeling, discretization (such as insufficient short time steps); and round off and truncation (depending on numerical representation, or the solution algorithm).

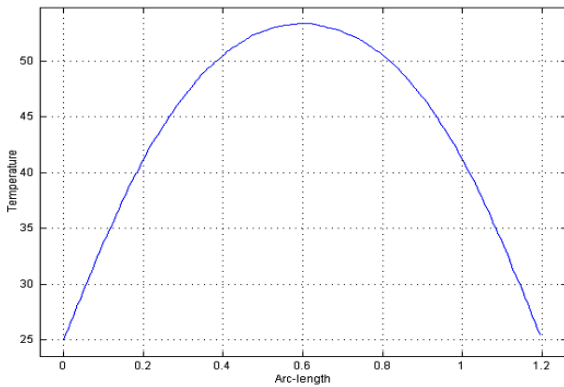


Figure 7: COMSOL Multiphysics generated temperature profile for the vertical line.

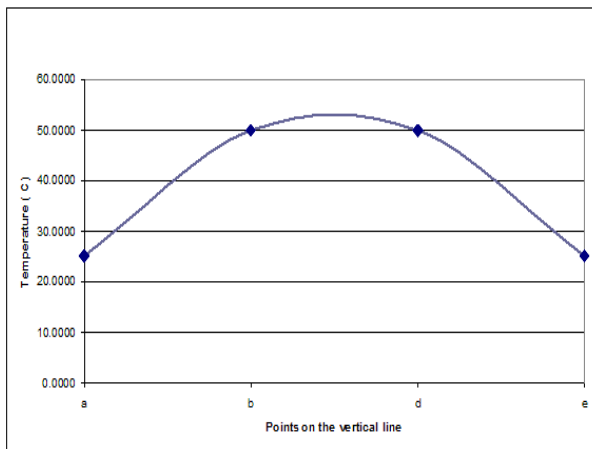


Figure 8: Analytically generated temperature profile for the vertical line.

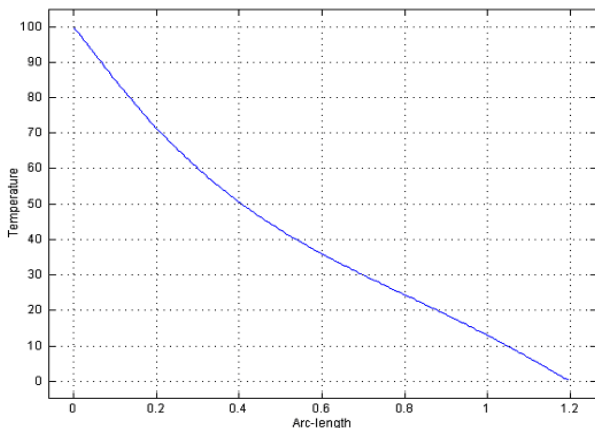


Figure 9: COMSOL Multiphysics generated temperature profile for the horizontal line.

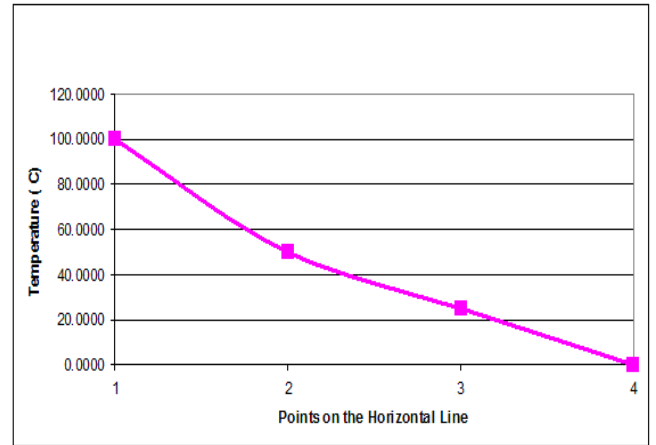


Figure 10: Analytically generated temperature profile for the horizontal line.

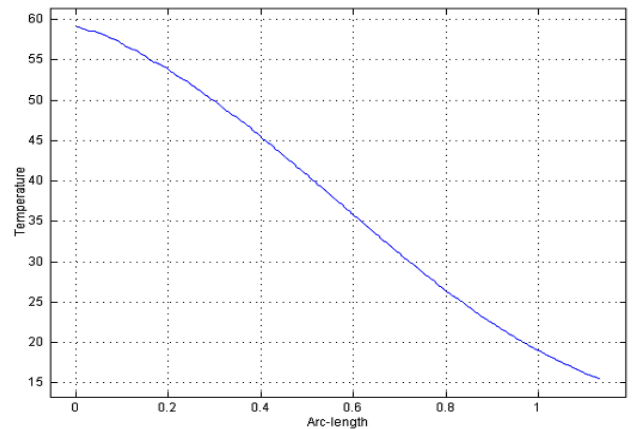


Figure 11: COMSOL Multiphysics generated temperature profile for the diagonal line.

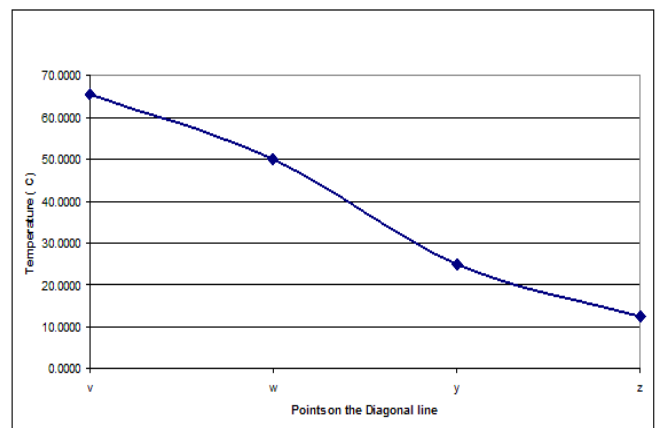


Figure 12: Analytically generated temperature profile for the diagonal line.

6. CONCLUSIONS

This paper has illustrated the usefulness of COMSOL Multiphysics 3.0 as a teaching and research aid in engineering education. The case-study presented shows the simplicity, effectiveness, efficiency, reliability, and accuracy of the



package. The motivation for using COMSOL Multiphysics to model steady-state temperature distribution is simple as:

- (i) It avoids any presumption that the students or engineers can read, write and edit source code.
- (ii) All of the calculation methodology is open and clear to see.
- (iii) Graphs of results can easily be displayed.
- (iv) A parametric study is well facilitated with little structured input required, i.e. a range of “what if” studies can be undertaken to look for key trends or relationships together with the potential successes and failures of the numerical method.

COMSOL Multiphysics is a FEM software that is not commonly used as other FEM packages like ABAQUS, FLUENT and ALGOR that are commonly used. This being an illustration of how COMSOL Multiphysics can be used to solve a simple two dimensional problem, it is considered appropriate to compare it with other FEM packages. Several factors are considered when comparing COMSOL Multiphysics with other FEM packages. The factors used are as follows: Listing Price, speed of processing a simple geometry model with simple boundary conditions, Ease of Use with User Interface, Customer Support times available and online tutorials. Findings from the questionnaire conducted among the users of these FEM packages and the prices we got from the vendors and the dealers of these software show that if you compare the price, COMSOL Multiphysics is significantly more expensive than other FEM Packages. However, if the analysts are able to put the software to use for product development, the cost will be recovered in a very short time in terms of price because COMSOL Multiphysics program is a flexible, robust design analysis and optimization package. COMSOL Multiphysics features file compatibility throughout the family of products and across all platforms. The Multiphysics nature of COMSOL allows the same model to be used for a variety of coupled-field applications, such as thermal-structural, magneto-structural, and electrical-magnetic-flow-thermal. The summary of the comparison is shown in Table 3.

Using COMSOL Multiphysics, the analysts can readily explore the effects of changing the physical problem. i.e. looking at the influence of different thermo-physical properties and the application of different boundary conditions. For engineering undergraduates such conduction investigations are useful in their own right. The COMSOL implementations of the numerical method also allow an insight into the numerical methodology and see factors which affect its performance. Numerical stability, rate of convergence, the compromise between computational time and accuracy together with the influence of time and space step lengths can all be explored. This study, synthesized with the conduction studies, truly enables a deeper understanding of the problem and moves the study from a skilling exercise to an understanding one.

Table 3: Temperatures at Some Points in the Domain

FEM Packages	Listing Price	speed of processing a simple geometry model with simple boundary conditions	Ease of Use with User Interface	Customer Support times available and online tutorials
Comsol	\$7995	Very Fast	Very Easy and flexible to use	Not Enough
Ansys Classic	\$6800	Very Fast	Very Easy and flexible to use	Enough
Workbench	\$5600	Very Fast	Very Easy and flexible to use	Enough
Abaqus	\$7000	Very Fast	Easy and Flexible to use	Enough
Fluient	\$6640	Very Fast	Easy and Flexible to use	Not Enough
Algor	\$29.95	Very Fast	Easy and Flexible to use	Not Enough

ACKNOWLEDGMENTS

The authors gratefully acknowledged the provision of the trial version of COMSOL Multiphysics package by Mr. Tolani Owoosina of Simulo Technologies, a representative of COMSOL Inc. and their associated companies in Nigeria, and opportunity to be trained on the application of the package by National Agency for Science and Engineering Infrastructure (NASeni), Abuja, Nigeria.

REFERENCES

- [1] Nicholas M. B. (1987). “Finite Element Analysis of Microcomputers”, McGraw-Hill New York.
- [2] COMSOL AB (2004), “FEMLAB Quick Start”; Tnergatan 23 SE-111 40 Stockholm, Sweden.
- [3] Wilensky, U. (1998), “NetLogo Heat Diffusion model,” <http://ccl.northwestern.edu/netlogo/models/HeatDiffusion>, Center for Connected Learning and Computer-Based Modeling, Northwestern University, Evanston, IL.
- [4] Schijndel A. W. M. (2002), “Modeling and solving building physics problems with



- FemLab”http://www.sciencedirect.com/science?_ob=ArticleListURL&method=list&ArticleListID=465219388&sort=d&view=c&acct=C000050221&version=1&urlVersion=0&userid=10&md5=806cf382860b32cb0a7b74ca47fcc530
- [5] Spyrakos C. (1994), “Finite Element Analysis in Engineering Practice”, West Virginia University Press, USA.
- [6] Hutchens G. J. and Gupta N. K. (2000), “A Spreadsheet Program for Steady-State Temperature Distributions”;<http://sti.srs.gov/fulltext/tr2000132r1/tr2000132r1.html> This document was prepared in conjunction with work accomplished under Contract No. DE-AC09-96SR18500 with the U.S. Department of Energy.
- [7] James G. (2003), “Advanced Modern Engineering Mathematics”, Pearson Prentice Hall, Harlow, England.
- [8] Oladejo K. A., Adetan D. A. and Bamiro O. A. (2007); “A software tool for determining steady-state temperature distributions in a square domain by the finite-difference method” The International Journal of Mechanical Engineering Education; Vol. 35, No. 4, pp 305 -315.
- [9] Donald R. P. and Leighton E. S., (1998), “Theory and Problems of Heat Transfer”, Schaum’s Outline Series in Engineering, McGraw-Hill Book Company, New York, pp 48.
- [10] Carnahan B., Luther H. A., and Wilkes J. O., (1976), “Applied Numerical Methods”, John Wiley & Sons, New York, pp 486.
- [11] Kharab A. (1995), “Use a spreadsheet program in a two-dimensional heat conduction problem,” Computer Methods in Applied Mechanics and Engineering 12(2), 173-181.

BIOGRAPHIES

Oladejo K.A., is a PhD holder in Mechanical Engineering from University of Ibadan, Nigeria. He is an Academic Staff of Obafemi Awolowo University, Ile-Ife, Nigeria. He is a registered Engineer with the Council for Regulations of Engineering in Nigeria (COREN). His current research interests include CAD, Modeling, Simulation and Production Engineering.

Abu R., is currently a PhD student and academic staff of Department of Mechanical Engineering at the University of Ibadan in Nigeria. He obtained master’s degree in Mechanical Engineering from University of Ibadan in Nigeria. He is a registered Engineer with the Council for Regulations of Engineering in Nigeria (COREN). He is interested in solid mechanics and computer applications.

Adewale M.D., is a postgraduate student in Information Technology in the School of Science and Technology at the National Open University of Nigeria in Nigeria. He obtained both ordinary diploma and bachelor’s degree in Computer Technology and Food Engineering respectively from Obafemi Awolowo University, Ile-Ife in Nigeria. He is a member of National Society of Black Engineers USA and a graduate member of Nigerian Society of Engineers. He is currently the Computer Network Consultant for National Open University of Nigeria (Ibadan Study Centre). His interests involve research into modeling and simulation, as well as automation.