The 9th TSME International Conference on Mechanical Engineering
11-14 December 2018
Phuket, Thailand

CAE 3D – A Computer-Aided Engineering Software Package for Engineering Education

S. Otarawanna[1,*], K. Ngiamsoongnirn[1], A. Malatip[1], and P. Eiamaram[2]

[1] National Metal and Materials Technology Center (MTEC), National Science and Technology Development Agency (NSTDA), 114 Thailand Science Park, Paholyothin Rd., Klong 1, Klong Luang, Pathumthani 12120, Thailand
[2] Design&Engineering Consulting Service Center (DECC), National Science and Technology Development Agency (NSTDA), 131 Thailand Science Park, Paholyothin Rd., Klong 1, Klong Luang, Pathumthani 12120, Thailand
[*] Corresponding Author: somboono@mtec.or.th, Tel. +66 2 564 6500 ext.4320

Abstract

This article presents a Computer-Aided Engineering (CAE) software application, CAE 3D, developed by research teams in Thailand under the funding of MTEC (National Metal and Materials Technology Center). The software has the capability to analyze 3D solid mechanics, heat transfer and mechanical vibration problems by the finite element method, and 3D fluid dynamics problems by the finite volume method. In contrast to most CAE freeware packages, the user interface of CAE 3D is purely graphical user interface (GUI) instead of containing some text-based user interfaces. Additionally, CAE 3D is designed to work on a personal computer with the Microsoft Windows operating system. As the software is user-friendly and capable of analyzing various types of engineering problems, it is suitable for being used as a learning tool for engineering education. For example, learners can analyze some engineering problems by themselves in CAE 3D to study the CAE technology.

Keywords*:* Computer-Aided Engineering software; Finite element; Finite volume; Graphical user interface; Engineering education

1. Introduction

Nowadays, Computer-Aided Engineering (CAE) software is an indispensable tool in design and engineering. CAE software is commonly employed by engineers to assess the performance of the design, estimate the safety margin and identify weaknesses of the design [1-2]. CAE engineers, or so-called CAE analysts, are then required to report the CAE results to their colleagues, supervisors and/or customers. Therefore, the thorough understanding of CAE is not limited to just CAE analysts, but also the relevant people mentioned.

To fully understand the CAE technology, ones need to have some experience in using CAE software themselves. CAE freeware applications offer the advantage that users can install and use the software freely in their own personal computers. However, most CAE freeware packages contain some text-based user interfaces causing some inconvenience for users. This paper presents the 3D CAE freeware “CAE 3D” possessing only graphical user interface (GUI). CAE 3D was developed by research teams in Thailand under the funding of MTEC (National Metal and Materials Technology Center). In this article, the programming concepts and key features of CAE 3D are described. Two examples of using CAE 3D for analyzing engineering problems are also demonstrated to show the capability of CAE for engineering education.

2. Software Structures and Features

CAE 3D is a Windows-based application and its user interface is entirely GUI. CAE 3D consists of four modules for solving different kinds of engineering problems. The modules SOLID, HEAT, VIBRA and CFD are for solid mechanics, heat transfer, vibration and fluid dynamics problems, respectively. After installing the software, there is a desktop icon shown in figure 1(a) where the user can double-click to open the window in figure 1(b). Each module of CAE 3D can be accessed by clicking the respective region in figure 1(b).

After entering one of the four modules (SOLID, HEAT, VIBRA and CFD), the screen appears like figure 2. Figure 3 magnifies the tree-view diagram marked by the dash box in figure 2. Items under *Model* in figure 3, i.e. *Sketch*, *Geometry*, *Constraints*, *Mesh*, *Calculation* and *Result*, are the steps of CAE software usage in sequence which are summarized as follows:

- *Sketch* / *Geometry*: Model creation by sketching or importing an STL file
- *Geometry*: Applying material properties
- *Constraints*: Applying boundary conditions
- *Mesh*: Meshing the geometry
- *Calculation*: Running the analysis
- *Result*: Visualizing results

These steps can be classified into the three main components of the CAE software: the pre-processor, solver and post-processor (figure 3). All the four modules contain similar pre-processor and post-processor while the solvers are different.

The pre- and post-processors of CAE 3D were developed by Microsoft Visual C# (the C# language tool in Microsoft Visual Studio [3]). Microsoft Visual Studio is an integrated development environment (IDE) for developing computer programs, web applications and mobile applications. Its features include Microsoft .NET Framework [4]. The .NET Framework contains a library of classes, interfaces, and value types allowing access to system functionality. Furthermore, it provides language interoperability (each language can use codes programmed by other languages) across several programming languages. Therefore, Microsoft .NET Framework allows the programmer to benefit from good attributes of other languages. For example, Microsoft Visual C# can employ the concept of the Object-oriented programming (OOP) like Microsoft Visual C++ while still benefiting from the ease of creating GUI like Microsoft Visual Basic.

The 2D and 3D vector graphics of the pre- and post-processors of CAE 3D were developed by OpenTK [5]. OpenTK is a C# graphics library providing access to graphics tools for rendering 2D and 3D vector graphics.

For the solvers of CAE 3D, the SOLID, HEAT, VIBRA and CFD modules contain solvers for solid mechanics, heat transfer, vibration, fluid dynamics problems, respectively. All the solvers were developed by using Intel Fortran Compiler, which is part of Intel Parallel Studio XE [6].

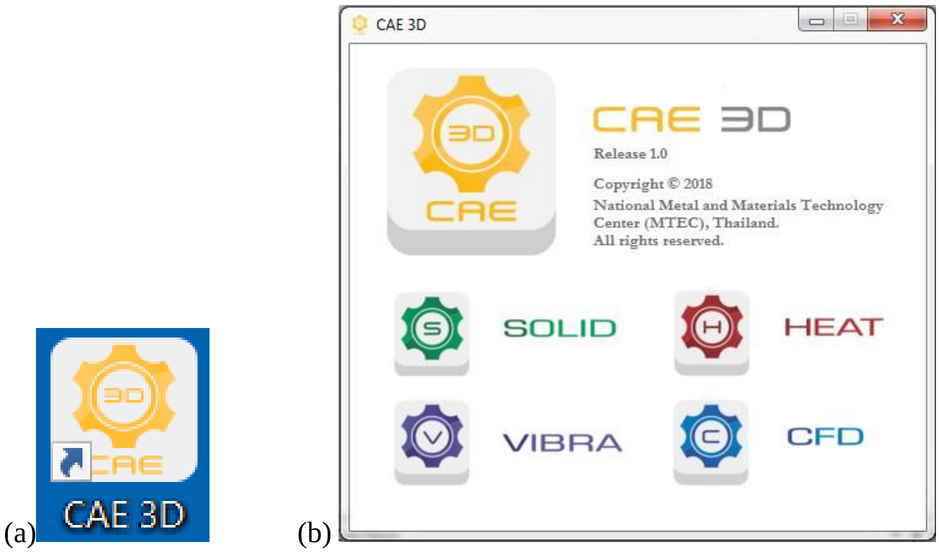

Figure 1 (a) The desktop icon and (b) the pop-up menu of CAE 3D.

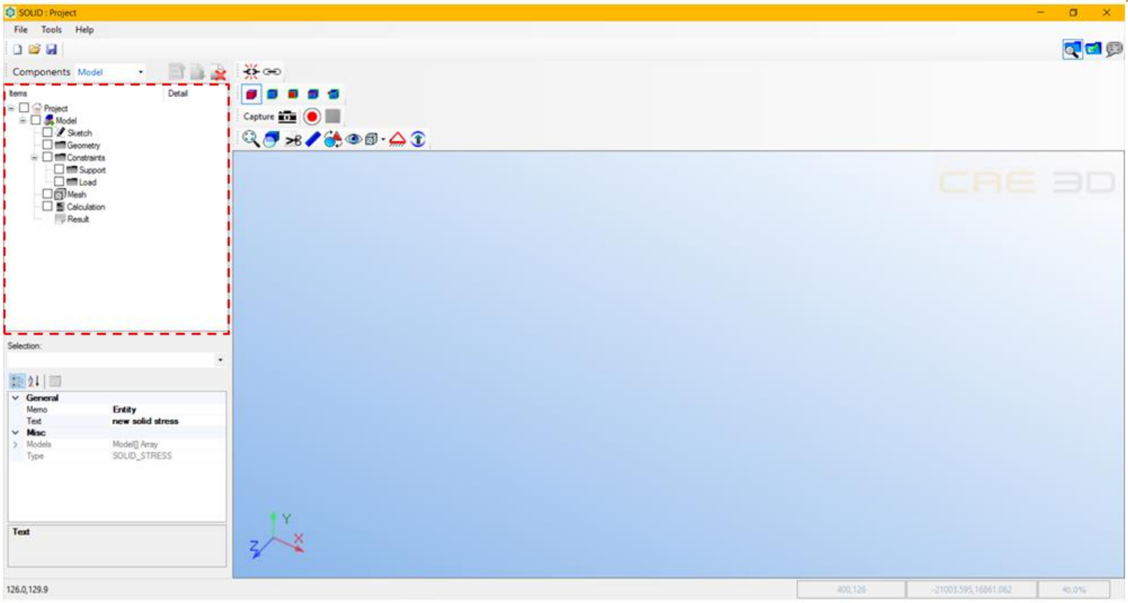

Figure 2 Typical appearance of the screen after entering one of the modules of CAE 3D.

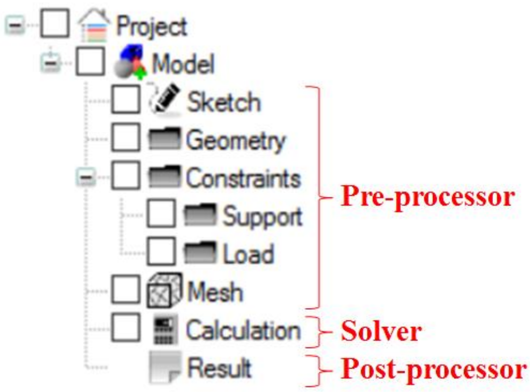

Figure 3 Magnified view of the tree-view diagram marked by the dash box in Fig. 2.

3. Examples

Two examples are chosen to demonstrate the capability of the CAE 3D software for analyzing 3D engineering problems. Although the SOLID module is used in both examples, analysis concepts and procedures are similar for the other modules (HEAT, VIBRA and CFD). It is worth mentioning that the validity of computational results obtained from CAE 3D has been verified by comparing them with those computed by ANSYS [7], which is one of the most popular commercial CAE software packages.

The first example is a structural beam with a T-shaped cross-section subjected to a force of 100,000 N at one end and fixed at the other end (figure 4). As the beam's geometry is relatively simple, it is convenient to use CAE 3D for creating the beam model. The beam geometry can be built by drawing the T-shaped cross-section and then extruding the drawn 2D cross-section into a 3D beam. Material properties and boundary conditions can be applied as shown in figures 5-6, respectively. Figure 7 shows how the user can generate the finite element mesh and control the average element size. The analysis result can be illustrated in the forms of deformed shape and fringe plot of different values (figures 8-9).

For the second example, a chair is loaded with the force of 200,000 N as shown in figure 10. As the chair geometry is relative complicated, it is more convenient to create the geometry in a CAD (Computer-Aided Design) software package, e.g. SolidWorks [8], Unigraphics NX [9] and AutoCAD [10]. The geometry built in CAD software can be saved in the STL (STereoLithography) format and subsequently imported into CAE 3D (figure 11). The subsequent steps, i.e. applying material properties, applying boundary conditions, meshing and visualizing results, are similar to those performed in the first problem. For the result, figure 12 shows the fringe plot of the total deformation on the deformed chair.

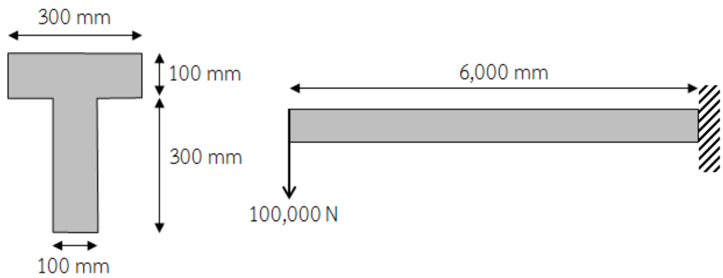

Figure 4 Problem statement of the beam problem.

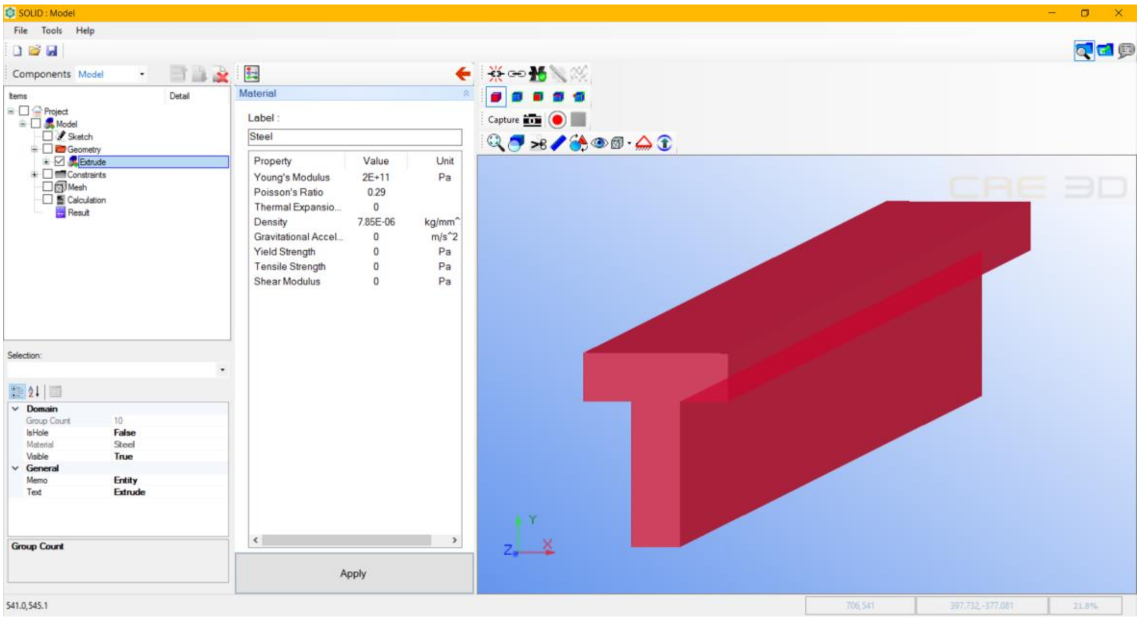

Figure 5 Applying material properties to the beam.

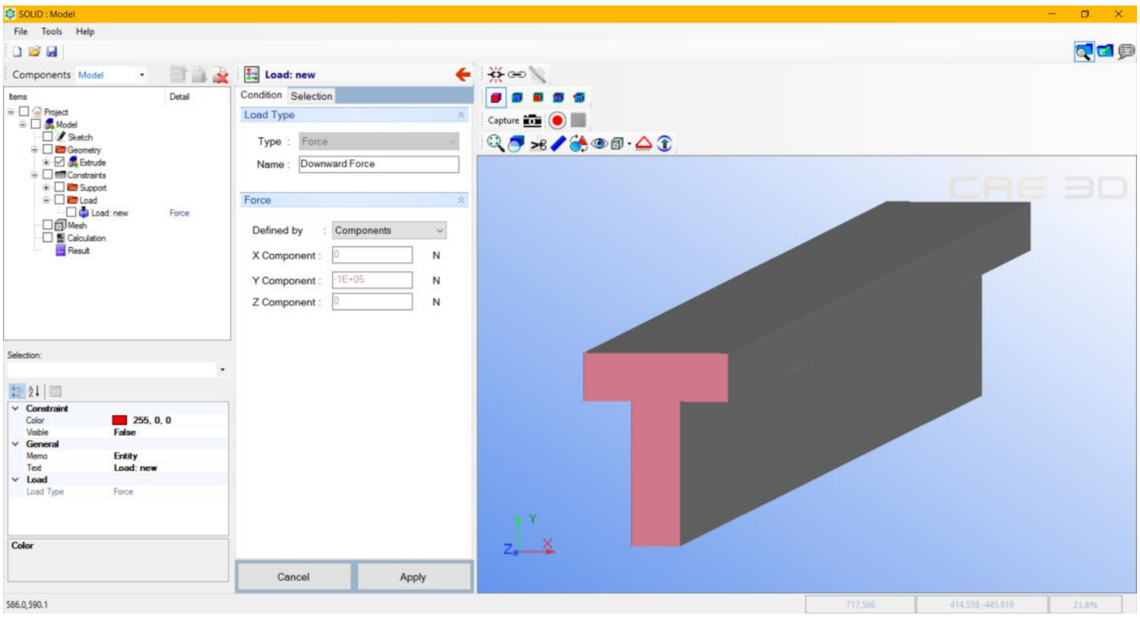

Figure 6 Applying boundary conditions to the beam.

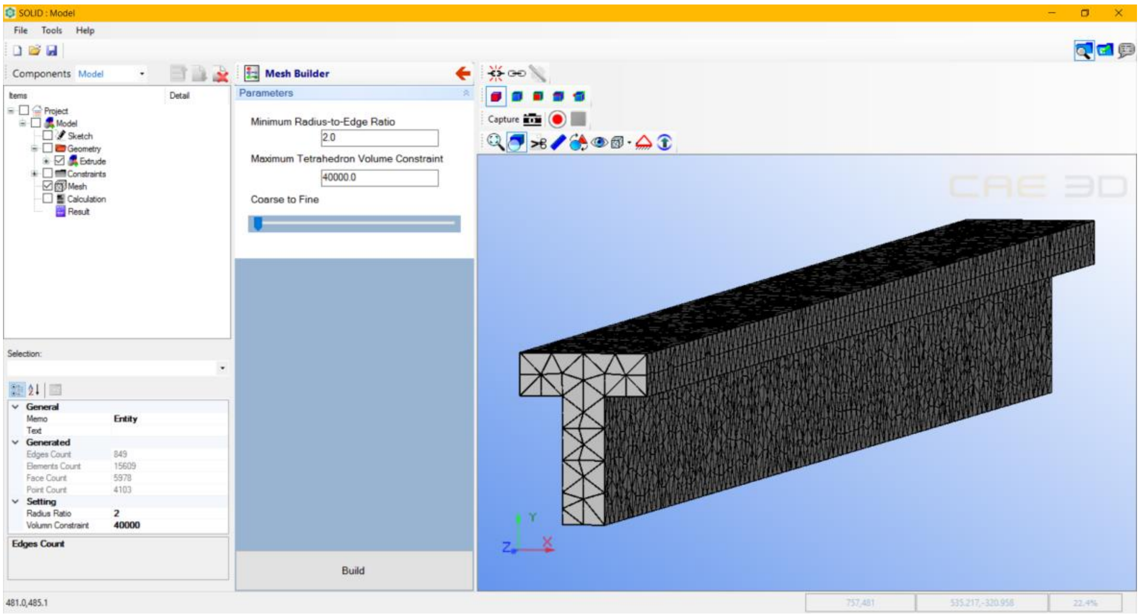

Figure 7 Generating mesh for the beam.

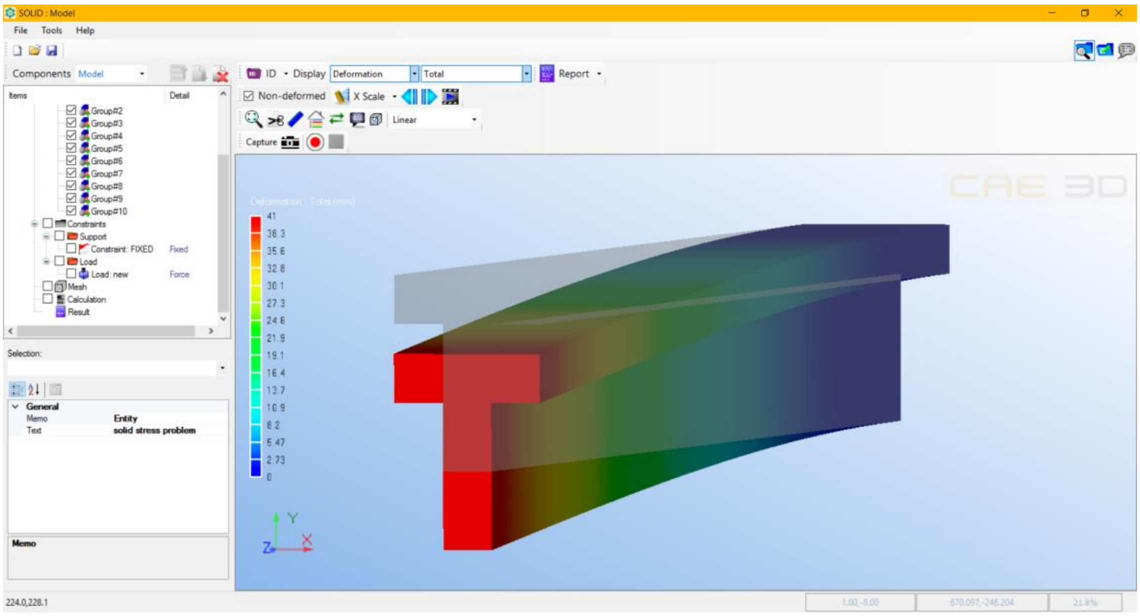

Figure 8 Fringe plot of the total deformation on the deformed beam.

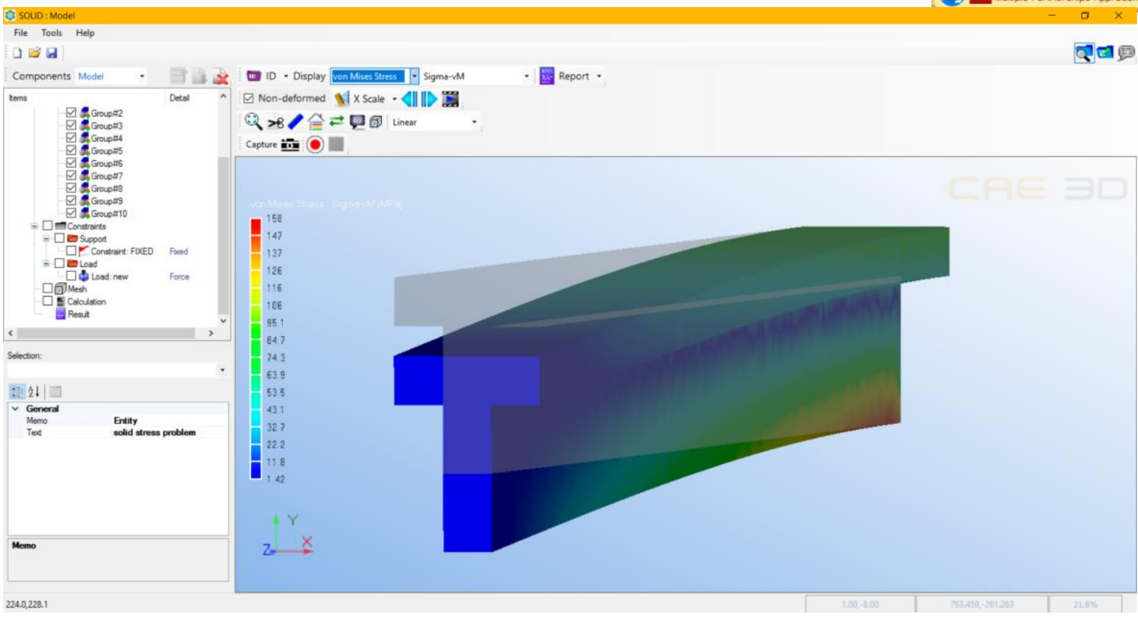

Figure 9 Fringe plot of the von Mises stress on the deformed beam.

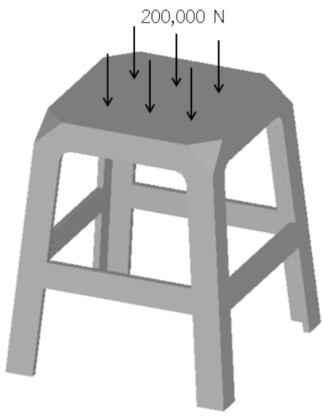

Figure 10 Problem statement of the chair problem.

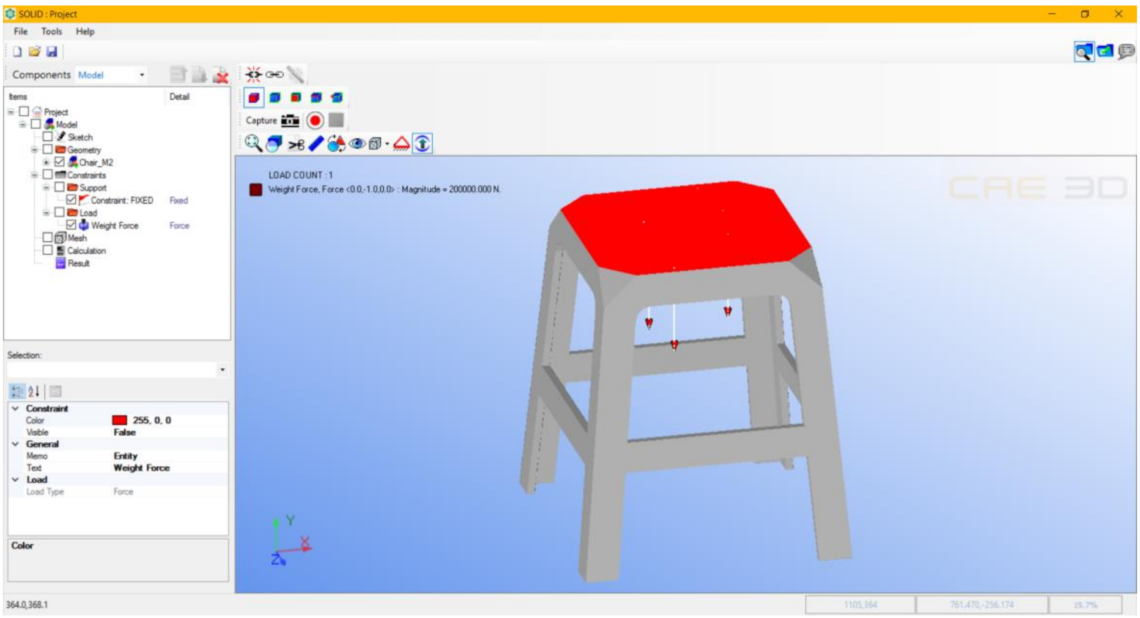

Figure 11 3D model of the chair and the boundary conditions applied.

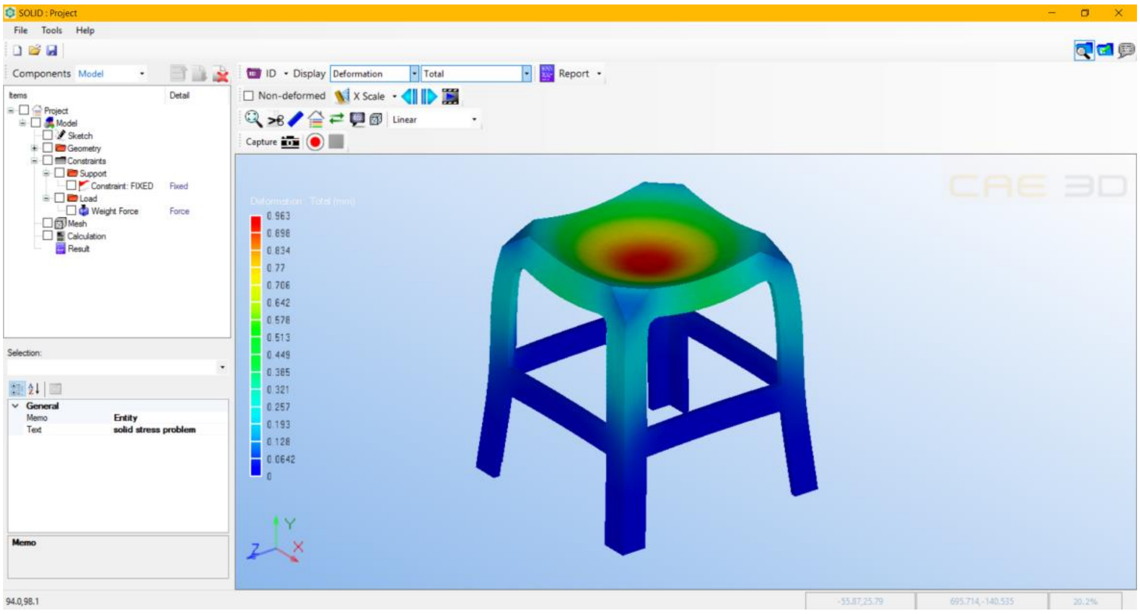

Figure 12 Fringe plot of the total deformation on the deformed chair.

4. Conclusion

The CAE 3D software has been presented in terms of programming concepts and software capability. From the two examples demonstrated, it can be seen that CAE 3D is beneficial for CAE learners to use as a learning tool. The learners can experience the CAE usage environment in this software. Furthermore, they themselves can adjust parameters, such as geometry, material properties and boundary conditions, to explore the effects of a given setting on the results.

Acknowledgements

The authors would like to acknowledge MTEC for supporting this work. All contributors for the solver parts of the CAE 3D are also acknowledged.

References

[1] N. S. Gokhale, S. S. Deshpande, S. V. Bedekar and A. N. Thite, "Practical Finite Element Analysis", Finite To Infinite, 2008.
[2] P. Dechaumphai, "Finite Element Method: Fundamentals and Applications", Alpha Science, 2010.
[3] https://visualstudio.microsoft.com/
[4] https://www.microsoft.com/net/learn/what-is-dotnet
[5] https://opentk.github.io/
[6] https://software.intel.com/en-us/fortran-compilers
[7] https://www.ansys.com/
[8] https://www.solidworks.com/
[9] https://www.plm.automation.siemens.com/global/en/products/nx/
[10] https://www.autodesk.eu/products/autocad/overview