COMPARISON OF SOME FEM CODES IN STATIC ANALYSIS

Bachelor's thesis

Mechanical Engineering and Production Technology

Riihimäki 2014

Dilip Neupane

Clarification of signature

HAMK
UNIVERSITY OF APPLIED SCIENCES

ABSTRACT

Riihimäki
Degree Programme in Mechanical Engineering and Production Technology
Design of Mechanics

Author	Dilip Neupane	**Year** 2014
Subject of Bachelor's thesis	Comparison of some FEM codes in static analysis	

ABSTRACT

The aim of this thesis work was to compare some finite element method codes and their theories in a static analysis with few different examples. The comparison was made between open source and commercial FEM codes. This thesis was commissioned by HAMK University of Applied Sciences. This work was mainly based on the use of the finite element methods in a static analysis.

The primary goal of this thesis was to conduct a research project on open source FEM codes; among them to select a few codes and to compare them with commercial FEM codes to their features and result from different examples with the same boundary conditions, material properties and geometry. The examples were linear and nonlinear static problems.

At first, a research project was conducted on finite element methods and on a finite element method in a static analysis. Based on this, a general comparison was made on the basis of working fields, an internal module, the operating system, the base language and supported file types for each selected FEM code. Then five test examples were set up and theoretical solutions were obtained for each test example. Finally, theoretical solutions were compared to numerical solutions obtained from each FEM code wherever possible. Only the FEM results were compared to the test example if theoretical solutions were not possible.

The overall conclusion from this thesis project was that, open source finite element method codes can also give similar results compared to commercial FEM codes if used with a similar amount of care and knowledge. It was observed that most of the free or open source FEM codes could perform only a linear static analysis whereas some of them were capable of nonlinear analysis. It was also found that open source FEM code were difficult to learn compared to commercial FEM codes. Although it is difficult to learn these, the main advantage with them is that they are free and offer an alternative choice for those who cannot purchase a commercial FEM code license.

Keywords Finite element method, static analysis, linear/nonlinear, FEM codes

Pages 71 p. + appendices 10 p.

Contents

List of Figures

List of Tables

Notations

Symbols most often used in stress analysis appear in the following list. Matrices and vectors are denoted by boldface type.

$\mathbf{R}$	global loading matrix
$\mathbf{f}$	element loading matrix
$\mathbf{K}$	global stiffness matrix
$\mathbf{k}$	element stiffness matrix
$\mathbf{U}$	global displacement matrix
$\mathbf{u}$	element displacement matrix
U	strain energy stored in an element
E	Young modulus
$\mathbf{E}$	stress-strain relation matrix
$\boldsymbol{\varepsilon}$	strain matrix
ε	strain
$\boldsymbol{\sigma}$	stress matrix
$\mathbf{B}$	Element strain displacement matrix,
σ	stress
τ	shear stress
u,v,w	displacement at each nodes
$\mathrm{v(x)}$	deflection function
A	cross-sectional area of an element
I	second moment area of an element
$\mathrm{N_i(x)}$	shape function
$\mathrm{F,q}$	force, distributed load
M	moment
Γ	boundary conditions

Abbreviations

DOF	degree of freedom
FEM	finite element method
FEA	finite element analysis
CFD	computational fluid dynamics
GPL	General public license
GUI	Graphical user interface
1-D, 2-D, 3-D	one-dimensional, two-dimensional and three-dimensional

1 INTRODUCTION

A finite element method is a numerical solution technique for solving different types of field problems and it has been used for effective digital simulation. There are lots of finite element codes which are used to solve different field problems. These codes must be purchased or are available for free under a GPL license (open source).

The objective of this thesis work was to compare some FEM codes and their theories, interface, capabilities and results in a static analysis with different examples. A list of open source FEM codes was collected and three codes were selected for detailed study. Two additional commercial FEM codes were also chosen for detailed examination. Then the FEM codes were compared to each other based on their working field, solution procedure, type of elements, interface, learning curve, supported files, etc. After that, five test examples were selected and theoretical solutions were obtained, then a static analysis was performed with each selected FEM code. The selected codes for comparison were as follows:

- ANSYS Workbench 15.0
- Creo Simulate 2.0
- Calculix 2.7
- Z88 Aurora
- Gmsh

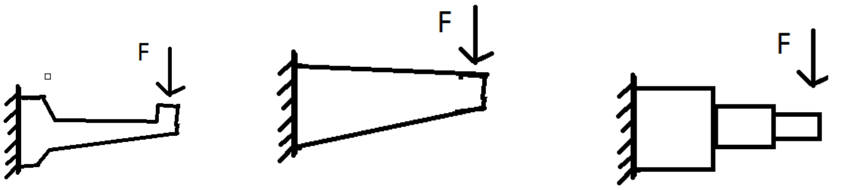
Only a general comparison was made with each FEM code and only some result quantities were compared. In most examples, the result quantities were maximum total displacement and equivalent von mises stress. The results obtained from each FEM code will change if improved in mesh quality. The program default convergence criteria were considered and mesh was refined by the author’s decision.

This thesis is divided into three parts. Chapters two and three form the first part which is based on theoretical research on the finite element method and finite element methods in a static analysis. Chapters four and five are the second part based on research on FEM codes and a detailed study of the selected FEM codes. A general comparison of the FEM codes was made in this part. The third part consists of Chapters six and seven where there are test example descriptions and the results obtained from each selected FEM code. There are five test examples described here: three of them are linear problems and the remaining two are nonlinear problems. The nonlinearity was caused by contact and materials. The theoretical and numerical solutions obtained from each selected FEM code compared here.

2 BASIC OF FINITE ELEMENT METHOD

2.1 What is Finite Element Method?

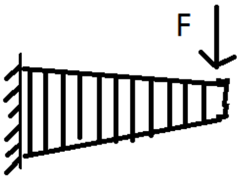
The finite Element Method (FEM) is a method of an analysing process where real life structures are divided into finite pieces to obtain solution for a large class of engineering analysis. Mathematically, The FEM is an approximate method for solving field problems. It is also called finite element analysis (FEA). FEM is a numerical or computational technique for solving different field variable like displacement, stress, strain, temperature, electric charge, etc when boundary conditions of field variables are given. An example of FEM is provided in Figure 1 where a physical (real) system is assumed with a mathematical model and FEA discretization has been applied on it. Upon good representation of a real physical system into mathematical model, and increasing discretization, FEM solution approaches exact solution to mathematical model which is called convergence in FEM analysis. Hence, when used effectively, FEM can enable innovation that would be impossible or tedious with any other methods.



Physical (Real) system

Mathematical Model

FEM discretization



Increase in discretization

Figure 1 An approximate nature of FEA

FEM can be applied into solving different static and dynamic engineering problems, from stress analysis of simple a beam structure (1D) or a large complicated machine (3D) to dynamic responses under different mechanical, electrical, magnetic or thermal loading. Today, FEM has been used in aerospace, aeronautical, defence, consumer product and industrial equipment industries. Also, with rapid development of different CAD software and advance computation systems, FEM are used in materials science, biomedical engineering, medicine, biology, physic, etc (Chen and Liu, 2014). Table 1 summarizes some application examples using FEM.

Table 1 Application of FEM (Chen and Liu, 2014)

Field	**Application examples**
Solid/Structural mechanics	Wind turbine blade design optimization, structure failure analysis, crash simulation, nuclear reactor component integrity analysis, beam and truss design and optimization, limit load analysis, etc
Heat conduction	Combustion engine, cooling and casting modelling, electronic cooling modelling, etc
Acoustic flow	Seepage analysis, aerodynamic analysis of cars and airplanes, air conditioning modelling of a building, etc.
Electronics/electrostatics/electromagnetics	Electromagnetic interference suppression analysis, sensor and actuator field calculations, antenna design performance predictions, etc

2.2 A brief history of FEM

The basic idea of FEM originated from advances in aircraft structural analysis. The foundation of the FEM was first developed by Courant in 1940s and the stiffness matrix for truss, beam and other elements were developed during 1956s by Courant and other people. The term *finite element* was first coined and used by Claugh in 1960s whereas the first book of FEM by Zienkiewicz and Chung was published in 1967s.Used of computer FEM codes emerged during 1970s and till today advanced FEM codes are available to solve different field problems. In recent years, several significant development has been emerged in FEM software which were introduction of p-element, integrations sensitivity, FEM codes on desktop computers and development of powerful CAD programs to model complex geometry. A brief history of FEM can be summarized as follow (Chen and Liu, 2014).

Year	**Major Milestone**
1943	Variation method which laid foundation of FEM (Courant)
1956	Stiffness method for beam, truss
1960	The term *finite element* coined
1967	First book of FEM by Zienkiewicz and Chung
1970	FEM applied to non-linear problems and large deformations
1970s	Computer implementation on solving FEM
1980s	Used of microcomputer and GUI
1990s	Large structural systems analysis, nonlinear and dynamic problems
2000s	Multiphysics and multiscale problems
2014s	Powerful FEA tools

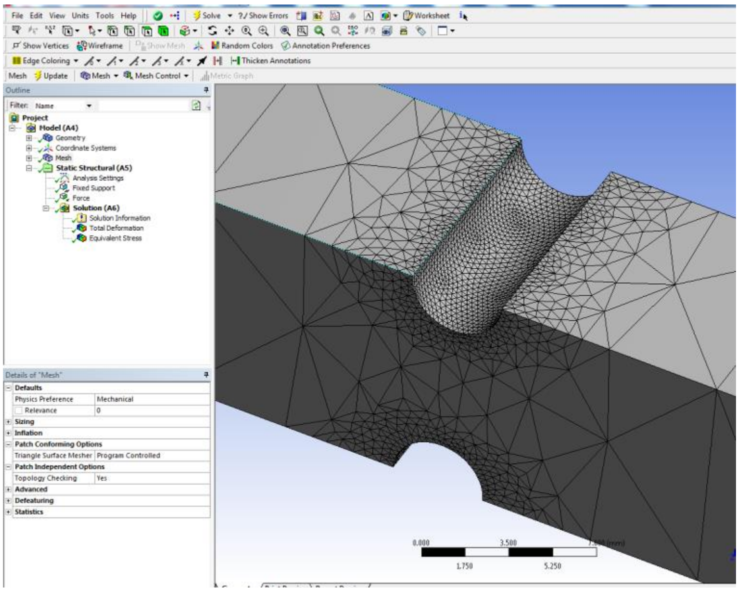
2.3 A General procedure of FEM

The general procedure of FEM can be summarize by following certain steps and these steps can be further classified into pre-processing, solution and post processing steps.

- Discretization of mathematical model into finite number of elements.
- Selection of interpolation functions to connect different nodes.
- Development of the element matrix for an element.
- Assembly of the element matrices to obtained global matrix for entire FEM model.
- Apply boundary conditions.
- Solution of equations.
- Additional computations and results

Pre-Processing:

This procedure include defining the geometry, material properties and boundary condition for the physical model. Usually, the structure is modelled using a CAD program that either comes with the FEM code or separate software. Then the element are selected with suitable interpolation functions and using these elements, the structure is discretize into finite pieces which is called meshing. The material properties and loading are defined in this procedure.



Defaults	
Physics Preference	Mechanical
☐ Relevance	0
Sizing	
Inflation	
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	Yes
Advanced	
Defeaturing	
Statistics	

Figure 2 Pre-processing using Ansys workbench 15.0

Solution:

In this step, the geometry, boundary condition, material properties and loading are applied to generate matrix equation for each element, which are then assembled to generate global matrix equation. The global equation is

$$R = KU \tag{1}$$

Where,

$\boldsymbol{R}$ = global loading matrix
$\boldsymbol{K}$= global stiffness matrix
$\boldsymbol{U}$= global displacement matrix

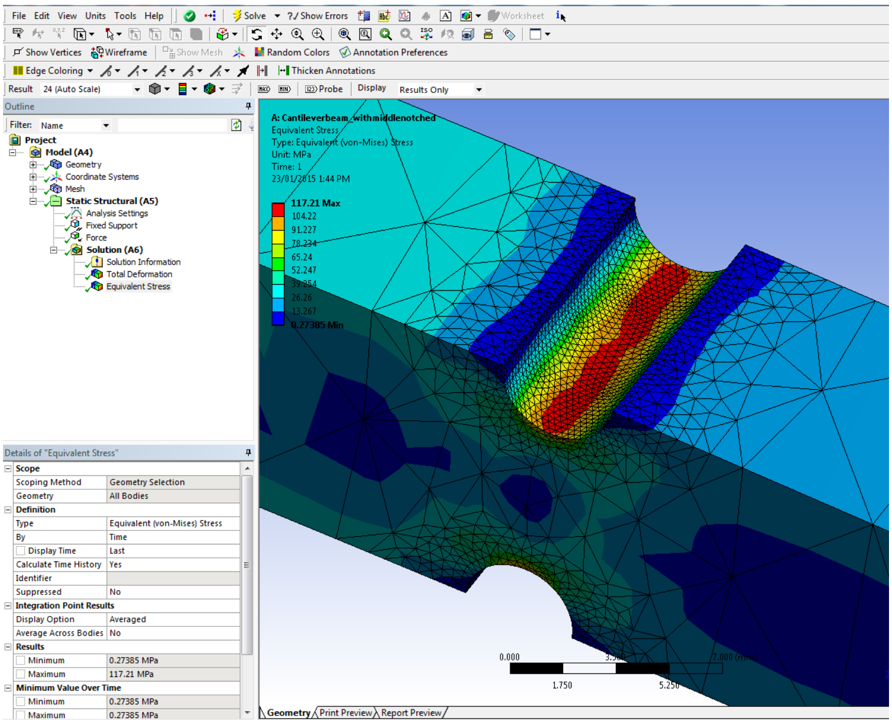
Also, for element equation, equation one can be applied where a global loading matrix becomes element loading, a global stiffness matric become element stiffness and a global displacement matrix becomes element displacement. Finally, the unknown form equation one is solved in this step.

Table 2 provides different FEM application disciplines with DOF and loading vectors. Here, in this table, DOF is the unknown parameters that should be solved using equation 1 which is global displacement matrix (Madenci and Guven, 2006).

Table 2 Degree of freedom and loading vector for different disciplines using FEM

Discipline	DOF (U)	Loading Vector (R)
Solid/Structure mechanic	Displacement	Force
Electrostatic	Electric potential	Charge density
Heat conduction	Temperature	Heat flux
CFD	Displacement potential, pressure, velocity	Particle velocity, fluxes
Magneto static	Magnetic potential	Magnetic intensity

Post-Processing:
After the solution step, post processing is the last step in a FEM analysis where the results obtained after solving global equations are manipulated and gathered to generate the result. The results may be graphical, contour plots, animation, etc. Post-processing is very useful to understand the raw data which are obtained from the solution step. Usually, the raw data are difficult to understand.



A: Cantileverbeam_withmiddlenotched
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
23/01/2015 1:44 PM

Details of "Equivalent Stress"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) Stress
By	Time
☐ Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
☐ Minimum	0.27385 MPa
☐ Maximum	117.21 MPa
Minimum Value Over Time	
☐ Minimum	0.27385 MPa
☐ Maximum	0.27385 MPa

Figure 3 Post process result using Ansys workbench 15.0

2.4 Element and shape functions

As mentioned earlier, real structures are discretized into small pieces and theses small pieces are called elements. An element can be one-, two or three-dimensional. These three types of element are also called line, surface or area, solid or volume element respectively. Each element is connected between nodes using shape functions. Shape functions also may be on a different "order" where that term refers to the order of the shape function that defines the distribution of displacement across the elements. Figure 4 shows some common finite elements ranging from line to volume elements with two type of element order (shape function) which are linear and quadratic.

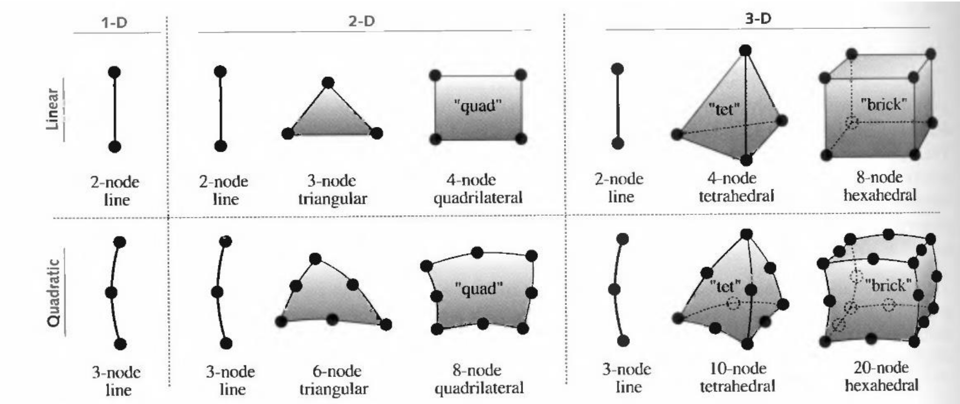


Figure 4 1-D, 2-D and 3-D elements with linear and quadratic shape functions (Norton,2006)

The element order or shape function can be linear, quadratic, and cubic and so on up to nth order of polynomial function. Today, commercial FEM code (Creo Simulate 2.0) can have up to 9th order of polynomial. Figure 5 illustrates linear, quadratic and cubic shape functions for line elements.

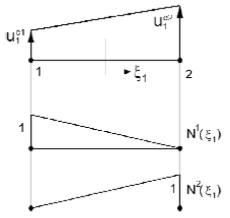
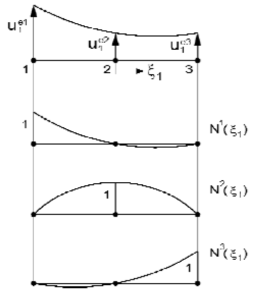
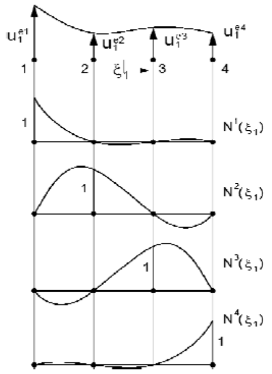
	Linear $p = 1$ 1 2	Quadratic $p = 2$ 1 2 3	Cubic $p = 3$ 1 2 3 4
$N^1(\xi_1)$ $N^2(\xi_1)$ $N^3(\xi_1)$ $N^4(\xi_1)$	$\frac{1}{2}(1-\xi_1)$ $\frac{1}{2}(1+\xi_1)$	$\frac{1}{2}(\xi_1-1)\xi_1$ $1-\xi_1^2$ $\frac{1}{2}(1+\xi_1)\xi_1$	$\frac{9}{16}(1-\xi_1)(\xi_1^2-\frac{1}{9})$ $\frac{27}{16}(\xi_1^2-1)(\xi_1-\frac{1}{3})$ $\frac{27}{16}(1-\xi_1^2)(\xi_1+\frac{1}{3})$ $\frac{9}{16}(\xi_1+1)(\xi_1^2-\frac{1}{9})$
			

Figure 5 Lagrange interpolation function for line elements (Enes, 2009).

There are also two type of element type's categories by various FEM codes which are called h-elements and p-elements respectively. The most common elements used by FEM codes are h-elements which are also called classical elements. These elements orders is typically limited to quadratic, therefore mesh refinement must be done to achieved convergence and it is done by increasing the number and size of h-element near regions of high stress gradient. P-elements are the element which allow higher order shape function to element edge. This type of element are popular nowadays in many commercial FEM codes.

These type of element and shape function have their own application and uses. Truss, beam, frame element are model using line element. For example, 1-D beam element have two nodes and each node have two degree of freedom. Surface element are used for plane stress and strain problems and solid element are used for those type of application when 1-D and 2-D analysis no longer valid or accurate. Upon good choice of element and shape function, errors can be reduce to get acceptable results. Also, good choice of element and shape function can reduce computation time and cost in FEM analysis.

2.5 Errors by FEM

As we already know FEM is a numerical method which discretises the structure into finite pieces. FEM is also a computational technique. The result obtained by FEM contains basically two type of errors which are as follow (Budynas, Nisbett and Shigley, 2008):

Computational errors:

These errors are due to round off errors from the computer floating point calculations and due to errors generated by numerical integration. These errors cannot be eliminated but can be reduce so that they do not really influence to final results. Most commercial FEM codes concentrate to reduce the theses error where free FEM codes have only few features to reduce these errors.

Discretization errors:
These errors are due to discretization of the structure into finite pieces. The geometry and the displacement distribution of a real structure is continuously vary. When using finite number of element to model the structure, the discretize structure cannot be fully matched with real model which causes errors. These errors can re reduce using smaller element and good interpolation functions.

3 FINITE ELEMENT METHODS IN STATIC ANALYSIS

FEM has been most extensively used in both linear and non-linear static analysis. The various types of static problems solved using FEM in this field include elastic, elastoplastic, and viscoplastic analysis of beam, frame, truss, plate, shells and solid structure. Usually, static analysis includes analysis of stress, strain and displacement under static loading for one-, two- or three-dimensional problems. In this chapter, general theory of elasticity has been discussed. Also, the general formulation of 1-D, 2-D and 3-D elements stiffness has been discussed with detailed formulation of beam element stiffness matrix. After that, general solution method of static analysis has been shown for linear and non-linear problems. The global equation for static analysis is same as in equation 1 and solving static problems is exactly the same as mention above in FEM general procedure. At the end of this chapter, the causes of nonlinearity in static analysis and a comparison between symbolic solution between beam theory and FEM theory has been shown.

3.1 Basic equations of solid mechanics

The primary aim any stress analysis is to find the distribution of displacement and stress under static loading and boundary conditions. The following equation are satisfied if there exist analytical solution for a given problems which are based on theory of elasticity. The table 3 shows type of equation in 1-D, 2-D and 3-D problems.

Table 3 Type of equation and number of equation in 1-D, 2-D and 3-D problems (Rao, 2005).

Types of equations	**Number of equations**		
	In 3-D problems	In 2-D problems	In 1-D problems
Equilibrium equation	3	2	1
Stress-displacement relation	6	3	1
Stress-strain relation	6	3	1
Total no. of equation	**15**	**8**	**3**

Similarly, the unknown quantities whose number is equal to the number of equation available, in various problems are shown in table 4.

Table 4 Unknown quantities in 1-D, 2-D and 3-D problems (Rao, 2005)

Unknowns	**In 3-D problems**	**In 2-D problems**	**In 1-D problems**
Displacements	u, v, w	u, v	u
Stresses	σ_x , $\sigma_y, \sigma_z,$ $\tau_{xy}, \tau_{yz}, \tau_{zx}$	$\sigma_x , \sigma_y , \tau_{xy}$	σ_x
Strains	$\varepsilon_x , \varepsilon_y , \varepsilon_z$ $\varepsilon_{xy} , \varepsilon_{yz} , \varepsilon_{zx}$	$\varepsilon_x , \varepsilon_y , \varepsilon_{xy}$	ε_x
Total no. of unknowns	**15**	**8**	**3**

Thus, we have number of equation equal to number of unknowns to find stress, strain and displacement. There will be some additional equations which must be consider in practise which are equilibrium equations and boundary conditions equations. Although all the analytical solution has to satisfy above equations, but numerical solution like FEM solution does not satisfy all the equations stated above. This is very important to understand in finite element relations and also estimating the order of error involved in the finite element solution by knowing the extent to which the FEM solution violates the basic equations.

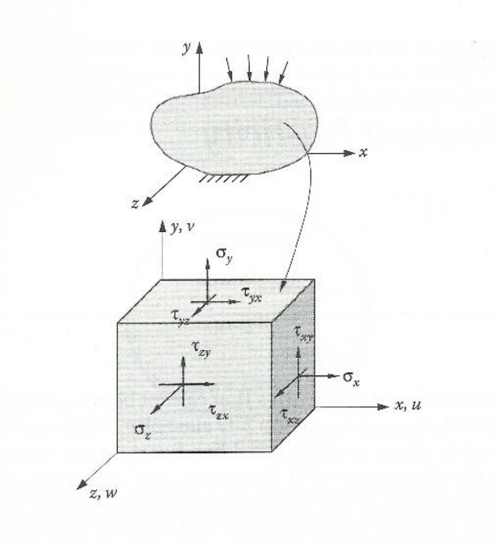


Figure 6 Stress and strain in a 3-D elastic body (Chen and Liu, 2011).

The stresses and strains at a point in a 3-D elastic body are

$$\sigma = (\sigma) = \begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{pmatrix} \text{ or } \left(\sigma_{ij}\right) \qquad \varepsilon = (\varepsilon) = \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} \text{ or } \left(\varepsilon_{ij}\right) \tag{2}$$

The stress and strain relation (for isotropic materials) in 3-D is given by

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{pmatrix} = \frac{E}{(1+\upsilon)\cdot(1-2\upsilon)} \cdot \begin{pmatrix} 1-\upsilon & 0 & 0 & 0 & 0 & 0 \\ 0 & 1+\upsilon & 0 & 0 & 0 & 0 \\ 0 & 0 & 1+\upsilon & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\upsilon}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\upsilon}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\upsilon}{2} \end{pmatrix} \cdot \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} \tag{3}$$

Which can be express in matrix form,

$$\sigma = E\cdot\varepsilon \tag{4}$$

The displacement field can be describe as

$$u = \begin{pmatrix} u(x,y,z) \\ v(x,y,z) \\ w(x,y,z) \end{pmatrix} = \begin{pmatrix} u_1 \\ u_2 \\ u_3 \end{pmatrix} \tag{5}$$

Similarly, the strain-displacement relation in 3-D

$$\begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{pmatrix} = \begin{pmatrix} \frac{d}{dx}u \\ \frac{d}{dy}v \\ \frac{d}{dz}w \\ \frac{d}{dx}v + \frac{d}{dy}u \\ \frac{d}{dy}w + \frac{d}{dz}v \\ \frac{d}{dz}u + \frac{d}{dx}w \end{pmatrix} \tag{6}$$

The stresses and body force vector *f* at each point satisfy the following three equilibrium equation for electrostatic problems.

$$\frac{d}{dx}\sigma_x + \frac{d}{dy}\tau_{xy} + \frac{d}{dz}\tau_{zx} + f_x = 0$$
$$\frac{d}{dx}\tau_{xy} + \frac{d}{dy}\sigma_y + \frac{d}{dz}\tau_{yz} + f_y = 0 \qquad (7)$$
$$\frac{d}{dx}\tau_{zx} + \frac{d}{dy}\tau_{yz} + \frac{d}{dz}\sigma_z + f_z = 0$$

And, boundary condition at each point on the boundary Γ and at each direction, either displacement or traction (stress on the boundary) should be given, that is

$$u = u_i \;\; on\, \Gamma_u \qquad (specific\ displacment)$$
$$t = t_i \;\; on\, \Gamma_\sigma \qquad (specific\ displacment) \qquad (8)$$

in which the barred quantities denote given values, and the traction is defined by $t_i = \sigma_{ij}\, n_j$ or in matrix form

$$\begin{pmatrix} t_x \\ t_y \\ t_z \end{pmatrix} = \begin{pmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{pmatrix} \cdot \begin{pmatrix} n_x \\ n_y \\ n_z \end{pmatrix} \qquad (9)$$

With n being the normal. The following figure shows boundary of a 3-D elastic domain.

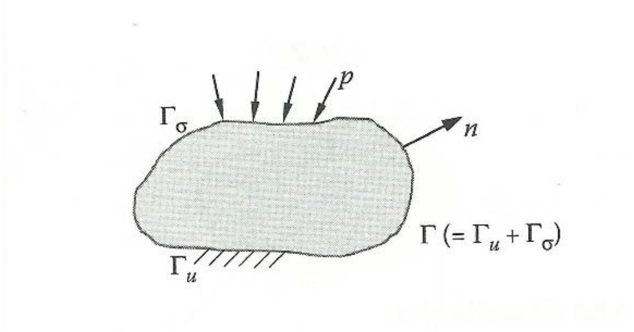


Figure 7 The boundary of a 3-D domain (Chen and Liu, 2011).

Similar equation as mention above will be used for 1-D and 2-D problems where the number of equation and quantities are same as mentions on table 3 and 4. For example, the stresses and strain for a 2-D elastic body is given by

$$\sigma = (\sigma) = \begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} \text{ or } (\sigma_{ij}) \qquad \varepsilon = (\varepsilon) = \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{pmatrix} \text{ or } (\varepsilon_{ij}) \qquad (10)$$

Finally, for 3-D analysis, above equation are solved in order to obtain stress, strains and displacement fields which is similar for 1-D and 2-D analysis.

Usually analytical solution of 1-D and 2-D are easy but it is very difficult for 3-D, therefore numerical methods or FEM are used for solving these type of field problems.

3.2 General formulation of elements

In this section, general formulation of solid elements stiffness for 3-D elasticity problems has been summarized by using the energy approach method. The same formulation method can be used in formulation of line and surface element stiffness matrix. A detailed derivation of beam element stiffness has been shown at the end of this section.

General formulation of Solid elements:
For formulation of solid element stiffness matrix, we first interpolate the displacement field within a 3-D element using shape function *N_{i,}* which is

$$u = \sum_{i=1}^{N} \left(N_i \cdot u_i\right) \qquad v = \sum_{i=1}^{N} \left(N_i \cdot v_i\right) \qquad w = \sum_{i=1}^{N} \left(N_i \cdot w_i\right) \tag{11}$$

where u_i,v_i and w_i are nodal values of displacement on the element and N is the number of nodes on that element.

We can write equation 11 in matrix form which is

$$\begin{pmatrix} u \\ v \\ w \end{pmatrix} = \begin{pmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & \dots \\ 0 & N_1 & 0 & 0 & N_2 & 0 & \dots \\ 0 & 0 & N_1 & 0 & 0 & N_2 & \dots \end{pmatrix} \cdot \begin{pmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ v_2 \\ w_2 \\ \dots \end{pmatrix} \tag{12}$$

or

$$u = Nd \tag{13}$$

Now, we can derived strain vector using relation given in equation 11 and equation 6, which is

$$\varepsilon = Bd \tag{14}$$

where, **B** is the matrix relating the nodal displacement vector **d** to strain vector **ε**. Considering the strain energy stored in an element which is given by

$$U = \frac{1}{2} \int_V \sigma^T \varepsilon dV \tag{15}$$

We can obtained general formula for *element stiffness matrix* which is

$$k = \int_V B^T EBdV \tag{16}$$

where, the dimension of **k** are 3N x 3N.

Similarly, for formulation of surface element, same procedure and equation are used except equation 12. The equation 12 can be replace with

$$\begin{pmatrix} u \\ v \end{pmatrix} = \begin{pmatrix} N_1 & 0 & N_2 & 0 & \dots \\ 0 & N_1 & 0 & N_2 & \dots \end{pmatrix} \cdot \begin{pmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \dots \\ \dots \end{pmatrix} \tag{17}$$

Formula of beam element stiffness:

The general element equation for a four degree freedom beam element is same as equation 1 which can be written in following form

$$\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix} = \begin{bmatrix} F_i \\ M_i \\ F_j \\ M_j \end{bmatrix} \tag{18}$$

$$or$$

$$ku = f$$

where ***k*** = element stiffness, ***u*** = nodal displacement and ***f*** is nodal force vector.

To derive the element stiffness given in equation 18 using energy approach, we can represent the deflection of a beam v(x) using shape function and corresponding nodal values **u.** The four shape function are as follow

$$\begin{aligned} N_1(\mathrm{x}) &= 1 - 3x^2 / L^2 + 2x^3 / L^3 \\ N_2(\mathrm{x}) &= x - 2x^2 / L + x^3 / L^2 \\ N_3(\mathrm{x}) &= 3x^2 / L^2 - 2x^3 / L^3 \\ N_4(\mathrm{x}) &= -x^2 / L + x^3 / L^2 \end{aligned} \tag{19}$$

And the deflection is given by

$$v(x) = Nu = \left[N_1(\mathrm{x}) \quad N_2(\mathrm{x}) \quad N_3(\mathrm{x}) \quad N_4(\mathrm{x}) \right] \begin{bmatrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{bmatrix} \tag{20}$$

which is a cubic function where $N_1 + N_3 = 1$ and $N_2 + N_3 L + N_4 = x$.

To derived the beam element stiffness matrix, we consider the curvature of the beam which is

$$\frac{d^2 v(x)}{dx^2} = \frac{d^2}{dx^2} Nu = Bu \tag{21}$$

where, B is the strain-displacement matrix given by

$$\begin{aligned} B &= \frac{d^2}{dx^2} N = \left[N_1''(\mathrm{x}) \quad N_2''(\mathrm{x}) \quad N_3''(\mathrm{x}) \quad N_4''(\mathrm{x}) \right] \\ &= \left[-\frac{6}{L^2} + \frac{12x}{L^3} \quad -\frac{4}{L} + \frac{6x}{L^2} \quad \frac{6}{L^2} - \frac{12x}{L^3} \quad -\frac{2}{L} + \frac{6x}{L^2} \right] \end{aligned} \tag{22}$$

Now, strain energy stored in the beam element is given by following equation which is same as equation 15 and applying the basic equation of simple beam theory we obtain element stiffness for a beam element which same as in equation 16 where E is not a matrix but it is material properties called young modulus.

$$\begin{aligned} U &= \frac{1}{2} \int_V \sigma^T \varepsilon dV \\ U &= \frac{1}{2} \int_0^L \int_A \left(-\frac{My}{I} \right)^T \frac{1}{E} \left(-\frac{My}{I} \right) dA dx = \frac{1}{2} \int_0^L M^T \frac{1}{EI} M dx \\ &= \frac{1}{2} \int_0^L \left(\frac{d^2 v(x)}{dx^2} \right)^T EI \left(\frac{d^2 v(x)}{dx^2} \right) dx = \frac{1}{2} \int_0^L (Bu)^T EI \, (Bu) \, dx \\ &= \frac{1}{2} u^T \left(\int_0^L B^T EIB dx \right) u \end{aligned} \tag{23}$$

Finally the beam element stiffness is given by

$$k = \int_0^L B^T EIBdx \tag{24}$$

Applying value of **B** from equation 22 and carrying out integration on element stiffness equation 24 we obtained following element stiffness matrix for a four degree of freedom beam element which is as follow.

$$k = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \tag{25}$$

3.3 Solution of Equilibrium equation in static analysis

The global equation as mention in equation 1, which is **R= KU** can be linear or non-linear base on linear or non-linear static problems. Therefore, there are separate solution method for linear and non-linear problems which are as follow (Bathe, 1996).

Solution of linear equation:
In linear analysis, both **R** and **U** are function of time t. That's way, there are two type of solvers used in FEM for solving linear system of algebraic equation which are direct method and iterative methods.

Direct method include solving equation using algorithm based on gauss elimination. This type of solution method is suitable for small to medium problems with less DOF (typically 1000000 range).The solution time for solver is depend upon dimension of the matrix and bandwidth of the FEM systems. This method handle multiple load cases easily.

Iterative method include solving equation using algorithm based on the Gauss-Seidel and Conjugate Gradient methods. This type of method is suitable for large problems or bulky structure with large DOF and bandwidth. The solution time is unknown beforehand but they converge faster. This method must be solved repeatedly for different load cases.

Solution of non-linear equations:
In non-linear analysis, both **R** and **U** are function of time t as well as **K** is function of **U** which makes the equilibrium equation non-linear. Numerical methods are unable to solve nonlinear equation explicitly for **U** as a function of **R.** Therefore, a nonlinear problem is solved by taking a sequence of linear steps. The general procedure for solving non-linear equation is to iterate in the solution. The popular solving technique are based on Newton-Raphson method; the BFGS method known as quasi-Newton method, which is an alternative form of Newton-Raphson iteration; and Load-Displacement-Constraint methods, which is frequently used for the calculation of the collapse load of a structure.

3.4 Nonlinearity

All most all physical structure exhibit nonlinear behaviour. The sources of nonlinearity in the physical system can be geometric nonlinearity, material nonlinearity and constraint nonlinearity. Constraint nonlinearity are caused by contact.

Geometric nonlinearity occurs due to change in the geometry of physical system. There are two main type of geometric nonlinearity which are large deflection and rotation, and stress stiffening. Stress stiffening occurs when the stress in one direction affect the stiffness in another direction. For example, a fishing rod with low lateral stiffness under a lateral load experience large deflection and rotation.

Material nonlinearity occurs due to nonlinear strain-stress curve of material. This is due to material property. A typical nonlinear stress-strain curve in given below. Typical material nonlinearity are plasticity, creep, nonlinear elastic, viscoelasticity and hyper elasticity.

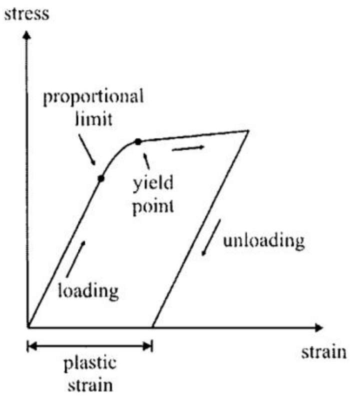


Figure 8 Nonlinear material response under loading and unloading (Madenci and Guven, 2006).

3.5 An example with analytical and FEM solutions

A uniformly loading cantilever beam deflection is computed using Euler-beam theory and FEM solution. A comparison between maximum deflection and graph of deflection curve has been shown in this section. The beam has length L, second moment of area I and young modulus E.

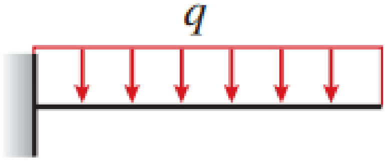


Figure 9 Cantilever beam with uniform loading q

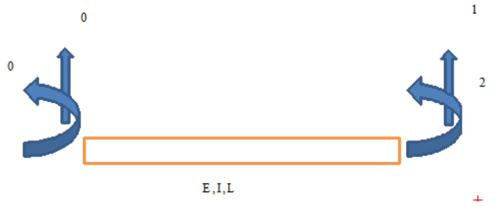
The deflection of beam using beam theory is given by

$$v(x) = \frac{-q\cdot x^2}{24\cdot E\cdot I}\cdot\left(6\cdot L^2 - 4\cdot L\cdot x + x^2\right) \qquad (26)$$

And, maximum deflection occur at L, which is

$$v(L) \rightarrow -\frac{L^4\cdot q}{8\cdot E\cdot I} \qquad (27)$$

Now, solution using FEM has been shown below. The above beam is modelled as follow



Total number of degree of freedom is 2, number of element is 1 and 2 nodes. The element stiffness using equation 25 and loading vector are given by

$$k_1 := \frac{E\cdot I}{L^3}\cdot\begin{array}{cc} \begin{array}{cccc} 0 & 0 & 1 & 2 \end{array} & \\ \begin{pmatrix} 12 & 6\cdot L & -12 & 6\cdot L \\ 6\cdot L & 4\cdot L^2 & -6\cdot L & 2\cdot L^2 \\ -12 & -6\cdot L & 12 & -6\cdot L \\ 6\cdot L & 2\cdot L^2 & -6\cdot L & 4\cdot L^2 \end{pmatrix} & \begin{array}{c} 0 \\ 0 \\ 1 \\ 2 \end{array} \end{array} \qquad r_1 := \begin{pmatrix} \frac{-q\cdot L}{2} \\ \frac{-q\cdot L^2}{12} \\ \frac{-q\cdot L}{2} \\ \frac{q\cdot L^2}{12} \end{pmatrix}\begin{array}{c} 0 \\ 0 \\ 1 \\ 2 \end{array} \qquad (28)$$

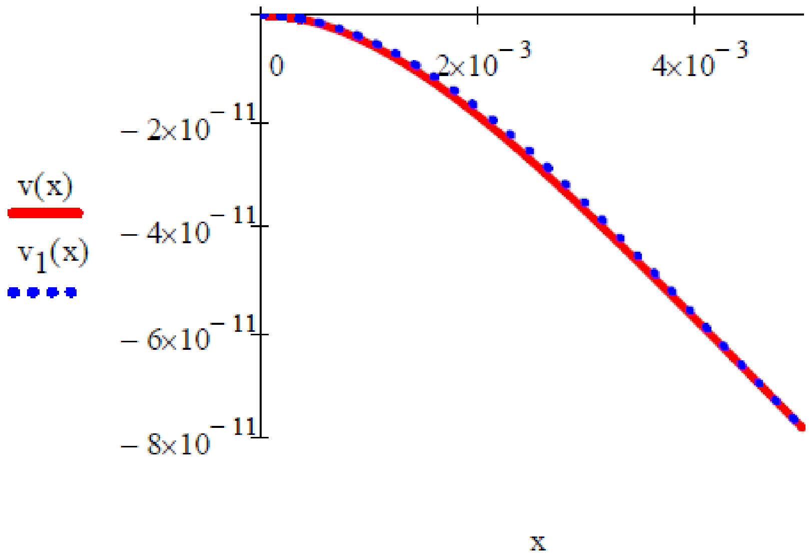
The global stiffness matrix **K** and loading vector **R** becomes

$$K := \frac{E\cdot I}{L^3}\cdot\begin{array}{cc} \begin{array}{cc} 1 & 2 \end{array} & \\ \begin{pmatrix} 12 & -6\cdot L \\ -6\cdot L & 4\cdot L^2 \end{pmatrix} & \begin{array}{c} 1 \\ 2 \end{array} \end{array} \qquad R := \begin{pmatrix} \frac{-q\cdot L}{2} \\ \frac{q\cdot L^2}{12} \end{pmatrix}\begin{array}{c} 1 \\ 2 \end{array} \qquad (29)$$

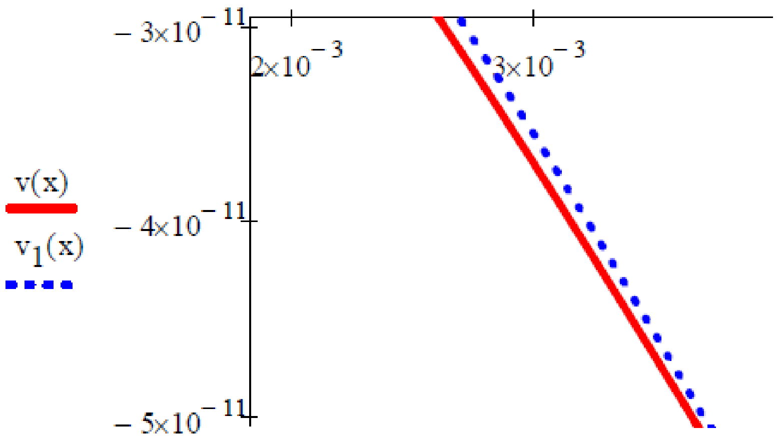
Using equation 1, global displacement as well as element displacement has been solved which is

$$U := K^{-1}\cdot R \rightarrow \begin{pmatrix} -\frac{L^4\cdot q}{8\cdot E\cdot I} \\ -\frac{L^3\cdot q}{6\cdot E\cdot I} \end{pmatrix}\begin{array}{c} 1 \\ 2 \end{array} \qquad (30)$$

Assuming, L= 0.005, E=1, q=1, I=1 with their SI units, deflection of the beam from beam theory and FEM are plotted as follow. This assumption values are imaginary for good graphical representation.



Deflection graph. v(x)= beam theory solution $v_1(x)$= FEM solution
X= length of beam



Zoom in the middle of first graph

Figure 10 Comparison of exact and FEM solution for cantilever beam

It is observed that the nodal value are same from exact solution and FEM solution which are given in equation 27 and equation 30. But the values at middle of the element is not same. This is due to error from FEM solution. Errors can be minimize by using more element or increasing order of element shape function which is given in equation 19.

4 FEM CODES

Computers have revolutionized the practise of engineering. When we talk about product design, classical method like tedious hand drawing are replace by computer-aided design using different CAD software. Similarly, analysis of product design are also replaced by computer aided analysis tools. One of the most popular and widely used CAE tools is FEM codes or finite element method software. FEM codes are modern calculators which can solve large engineering problems. FEM codes are computer codes or program written in different programing language which are based on algorithms developed for solving different fields problems using finite element method. Wide range of FEM software are available today for solving different engineering field problems. Some of them are open source code under GPL licencing whereas most of them are commercial.

This chapter summarize why computer codes are necessary for finite element methods and what are the available FEM software. At the end of this chapter, the reason for using free or open source FEM code has been mentioned.

4.1 Use of computer codes in FEM

FEM usually consist of calculation of linear or non-linear equation for global equation mentioned in equation 1. Solving this equation using hand calculation is limited to number of equation inside global equation. Also, iteration must be done for solving this equations which is impossible for large 3-D analysis which have higher number of elements and large number of equations. Therefore, computer codes are used for almost all engineering analysis using finite element methods.

We can categories FEM codes into two groups which are commercial FEM codes and free or open source FEM codes. The list of common FEM codes are mentions below which are suitable for static analysis.

4.1.1 Commercial FEM codes

Wide range of commercial FEM codes are available today for solving wide range of engineering problems. These commercial code are not only limited solving engineering problems but they are also used recently in the area of physic, chemistry, biomedical engineering, etc. The main advantage of these codes are user friendly interface and easy learning process. Most of the commercial codes are attached with a CAD software whereas few of them are only FEM codes.

Commercial codes provide continues support and training for their users. Most of them need to be renew for licencing over a certain time and few of them provide license forever once purchased. Most of them are based on h-element technology whereas few of them are based on p-element technology.

The following are the list of commercial FEM codes used for solving different field problems.

- ANSYS
- ADINA
- ABAQUS
- COMSOL Multiphasic
- Creo Simulate
- MSC/NASTRAN

A list of few commercial codes are in the Appendix.

4.1.2 Free or open source FEM codes

Apart from the commercial FEM codes, there are free or open sources FEM codes used these days by different academic institution, students and professionals. These FEM codes are provided for free under GNU General Public License with source codes. These source codes are mainly written by groups of academic or individual for certain purpose with certain limitation. Only few of them can solve wide range of engineering problems but most of them are designed and coded for specific purpose. Complicated geometry can't be modelled using these codes, therefore complicated geometry are modelled using separate CAD program and they are imported. Usually free FEM codes does not provide support and training for general users but user guide and tutorial for few examples are provided on their respective websites.

The following are the major list of free or open source FEM codes for different filed problems.

- Calculix
- Elmer
- Z88/Z88Aurora
- Gmsh
- FEbio
- GetFEM++
- Free FEM
- OpenFOAM

A list of few free or open source codes are in the Appendix.

4.2 Free or open source FEM codes?

The main reason to use open sources FEM codes is that, the codes are distributed freely with source codes and can be easily downloaded from internet. Huge amount of money must be spend for commercial software license which can be saved using open source codes. Free FEM codes can't give result as compare to commercial codes, but they can be alternative if you don't want purchase commercial codes. Some of the free FEM codes can give equivalent result as compared to commercial codes if use correctly. Commercial code do not provide an insight information into the formulation and solution method. With available source codes, it can enhance the research and learning process for development of new FEM codes.

5 DETAILED STUDY AND GENERAL COMPARISON OF FEM CODES

From the above FEM codes listed under commercial or open source FEM codes, the following codes are selected for detailed study and a comparison has been made between them. A comparison has been made on the basis of user interface, working field, solving procedure or internal module within, learning curve quality, operating system and base language, supported files formats, element types and element order and solving methods for global equation.

The following codes are studied in detailed.

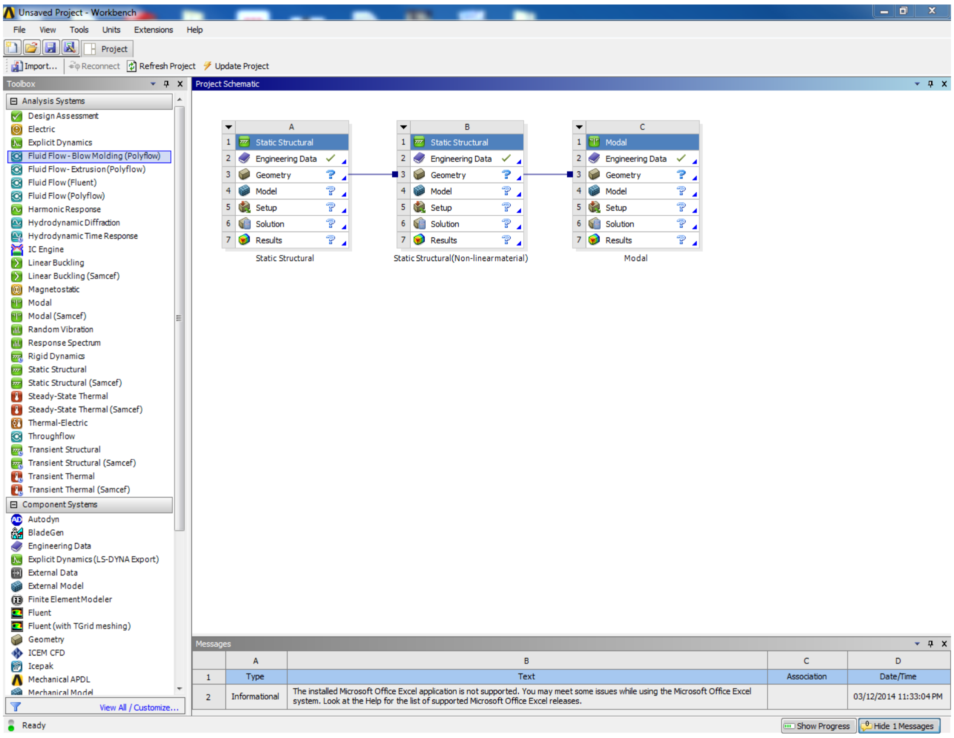
- ANSYS Workbench 15.0
- Creo Simulate 2.0
- Calculix
- Z88/Z88Aurora
- Gmsh

The reason why these FEM code has been chosen for study are as follow:

- Availability of the codes(only ANSYS and Creo available for student use during thesis process)
- h-element and p-element method
- Availability of FEM codes for windows and sufficient tutorial and users guides.
- less programing knowledge required to use the codes

5.1 ANSYS WORKBENCH 15.0

ANSYS workbench 15.0 is a commercial FEM code, which is part of ANSYS 15.0 and is developed by Ansys, Inc. It can perform structural, explicit, thermal, fluid, electromagnetic and coupled physic analysis. This is user friendly interface platform which can perform different type of analysis using same user data and geometry under a same workbench project. An analysis problems is called a project on workbench. This is widely used industry standard FEM codes based on h-element methods. ANSYS can be open in three modes based on the interaction between user and the ANSYS program. They are interactive, batch and combined mode. ANSYS workbench is interactive mode where platform is based on graphical user interface, which is composed of menus, dialog box, and different windows. This type of mode is suitable for beginners. Batch mode is the method to use ANSYS program without GUI which involves an input file written in ANSYS Parametric Design Language (APDL). Combined mode is a combination of both interaction and batch modes.



	A	B	C	D
1	Type	Text	Association	Date/Time
2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system. Look at the Help for the list of supported Microsoft Office Excel releases.		03/12/2014 11:33:04 PM

Figure 11 ANSYS Workbench 15.0 interface

ANSYS workbench is capable of simulating problems in wide range engineering disciplines but a short review of structural analysis has been discussed in this chapter. For example, static structural analysis includes analysis of deformation, stress, strain fields as well as reaction forces in a solid body. The types of analysis performed inside structural analysis are as follow.

- Static analysis
- Modal analysis
- Harmonic analysis
- Transient Dynamic
- Eigenvalue Buckling

Inside static analysis, it is capable of simulating linear or nonlinear problems. Nonlinearity includes geometric, materials and changing status nonlinearity. Nonlinear material behaviour in ANSYS workbench is characterised as plasticity, creep, nonlinear elastic, viscoelasticity and hyperleasticity whereas geometry nonlinearity is characterised by large deformation and rotation. Similarly, changing status nonlinearity is characterised by nonlinearity caused by contact between two bodies in their assembly. Symmetry conditions can be also applied to ANSYS workbench if the physical system exhibits symmetry in geometry, material properties and loading. Convergence is based on element size, therefore mesh should be refined in the areas where there is higher stress gradient.

A general procedure for solving static structural problem using ANSYS workbench is given below.

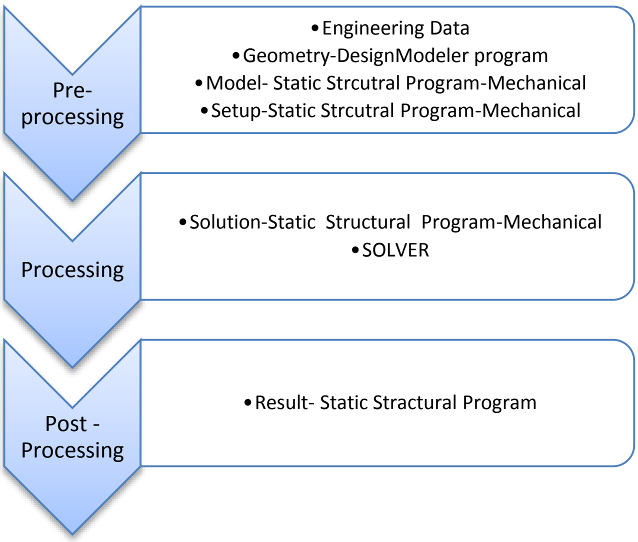


Figure 12 The basic static FEA process in ANSYS workbench

ANSYS workbench simulate physical system in three main phase as mention above in the figure 12. Basically, it consists of six step for solving a problems. Pre-processing includes *engineering data* where material properties like young modulus, poison ratio and yield strength are specified, *geometry* where physical CAD model is modelled using Design Modular program, *Model* where mathematical model of object or assembly is divided in mesh, *Setup* where boundary conditions are applied respectively using Static Structural Program. Static structural program is the solver and the post-processor for ANSYS workbench for static problems. Then the global equation is solved using the *solver* and the *result* are post process. Result can be obtained in graphical representation or tabular data. Different type of geometry model from different CAD system can be imported inside workbench. FEM data also can be imported and exported to another FEM solver or FEM codes.

There are two types of unit system, base units and common units. All common units are derived from base units. Base unit include chemical amount, current, luminance, mass, solid angle, time and temperature. Other units like electric charge, force, pressure, stress, etc are common units. It support predefined units system and also user can defined units system. Both unit system are based on base units. The following are few predefined unit system offered by ANSYS Workbench. These predefined unit system can be edit or delete.

- Metric (kg, m, s, °C, A, N, V) (default unit system)
- SI (kg, m, s, K, A, N, V)
- US engineering (lbm, in, s, R, A, lbf, V)

The following are few type of element that can be found on ANSYS element library. ANSYS library consists of wide variety of element types. Typical ANSYS elements used up to 3rd order polynomial for shape function. User can defined own element type also if needed.

Table 5 Element type and geometric entity of ANSYS Workbench 15.0

Element geometry types	3-D Element	2-D Element	1-D Element
Line	Beam, truss, frame	Beam, trust, frame	Beam, spring
Area or surface	Fluid, shell	Plane, shell, plate, axisymmetric	
Volume	prism, tetrahedral, brick, pipe, fluid		

5.2 CREO SIMULATE 2.0

Creo Simulate 2.0 is a commercial FEM code developed by PTC, Inc. It was known as Pro/Mechanica on previous version of Creo Simulate. This is completely different FEM code based on p-element method compared to other FEM codes which are usually based on h-element methods. It simulate in standalone mode or integrated mode with PTC Creo Parametric 2.0. It can simulate structural and thermal analysis. Structural analysis consists of linear static, static with small displacement contact, modal, linear buckling analysis whereas thermal analysis consist of linear steady state analysis. Creo simulate can be run in two modes, native modes and FEM mode. Native mode use own creo solver but FEM mode uses ANSYS or NASTRAN solver. Both structural and thermal analysis can be simulate in FEM mode using ANSYS and NASTRAN solver. It can also perform nonlinear structural analysis and nonlinearity are characterized as follow

- Large Deformations
- Contacts
- Hyperelasticity
- Plasticity
- Nonlinear Springs

It support wide range of materials. Typical materials library include metal and plastic materials. User defined materials are also possible. It support isotropic material property which is assigned to geometry, isotropic material failure limit properties and temperature dependent structural material properties. The strain-stress response can be linear, hyperelastic and elasto-plastic for these materials.

Convergence in Creo are based on the polynomial order of the element shape function. This is only the FEM codes that support polynomial order up to 9 degree. Based on polynomial order, there are two type of convergence method in FEM which are Single-pass adaptive and multi-pass adaptive. Single-pass adaptive convergence method use up to three degree of polynomial shape function whereas Multi-pass adaptive use higher order element shape functions.

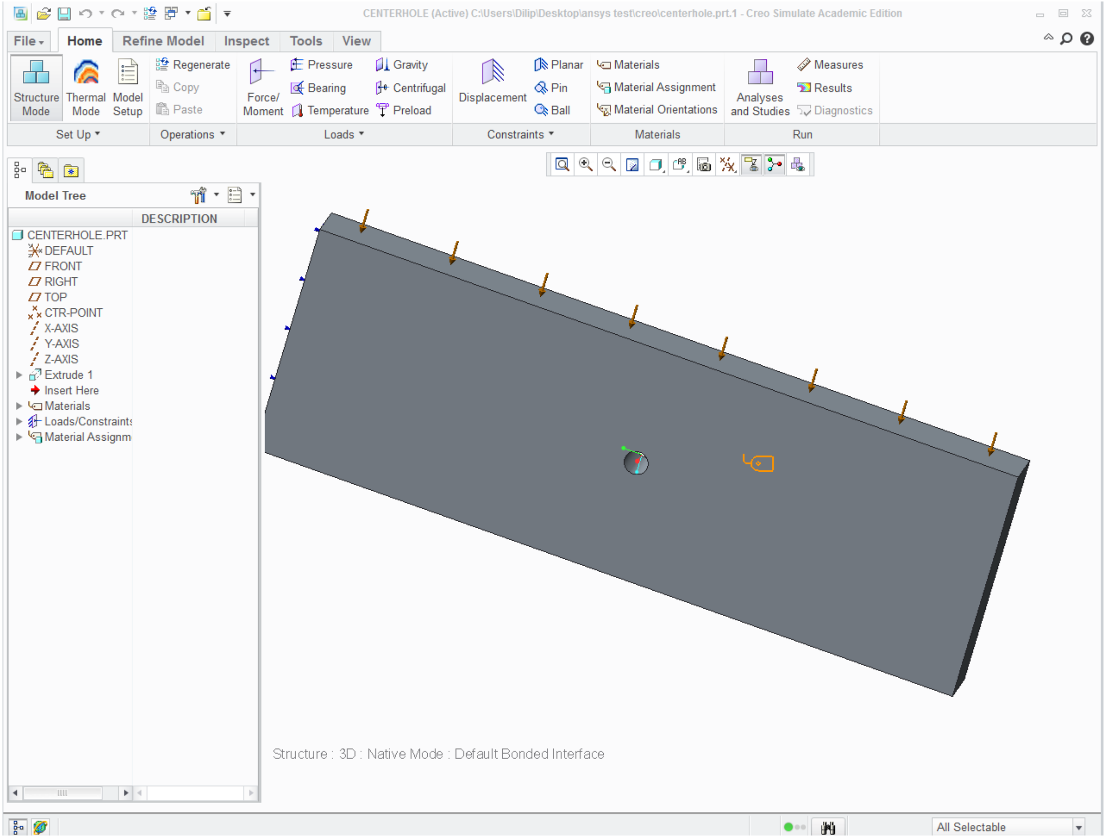


Figure 13 Creo Simulate 2.0 interface

A general procedure for solving static problems using Creo Simulate is given below.

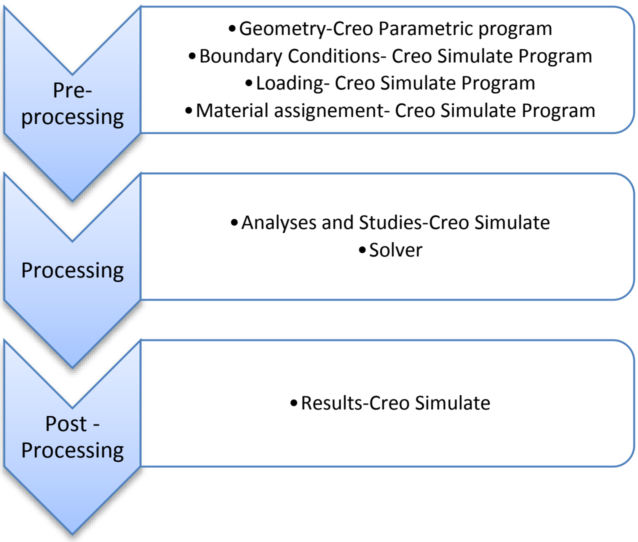


Figure 14 The basic static FEA process in Creo Simulate

Creo Simulate static problems as shown in above figure. It does not have CAD modelling software inside it. Usually geometry are modelled using Creo Parametric. If the geometry must be import from another CAD software, then also Creo Parametric is used to create geometry for Creo Simulate. After that, general procedure includes applying boundary conditions and loading to the geometry. The materials are assigned from material library. Mesh can be created automatically using AutGEM. AutoGEM is automatic mesh generator. Element shape and size can be change as well. After the solution is converged, the result can be obtained in fringe, contour plot, vector plot, animation, etc. Multiple result can be displayed in same window.

It supports SI units and other derived units as well. The input can be in any units but the final result are displayed in default SI units or user defined display unit which can be configure from configuration setting.

The following are some element type used in Creo Simulate.

Table 6 Element type and geometric entity of Creo Simulate 2.0 (Help.ptc,com, 2014)

Geometric Entity	**Solid Model Element types**	**2-D plane strain Element types**	**2D-plane stress Element types**
points	Beam Spring masses	Spring, masses	Spring, masses
curves	Beam	2-D shells	
surfaces	Shells (quadrilaterals, triangles)	2-D solids	2-D plates
Volumes	Solid (brick, wedge and tetrahedron)		

5.3 CALCULIX

Calculix is an open source or free FEM codes developed by Guido Dhondt and Klaus Wittig. The program consist of two parts, CalculiX GraphiX (CGX) and CaluliX CrunchiX (CCX). CGX is a program for pre- and post-processor developed by Kluas Witting and CCX is a solver program developed by Guido Dhondt. It is based on h-element method. It can simulate linear and nonlinear static, linear frequency, linear and nonlinear dynamic, buckling and thermal analysis. It also can simulate CFD problems as well as Laplace and Helmholtz problems by analogy.

CGX is designed to generate finite element model and display result generated by solver. It can generate and display beam, shell and brick element up to quadratic shape function. Other element like pentahedral and tetrahedral elements can be displayed but not generated. Therefore, another mesh program like Gmesh and Netgen are used to generate good quality mesh. It also can generate input data for other commercial FEM codes like Nastran, Abaqus and ANSYS.

CCX is designed to solved static, thermal, CFD, buckling and frequency analysis. In static analysis, it can perform linear and nonlinear static analysis and the nonlinearity may be caused by geometry, material or contact. Geometric nonlinearity includes large deformation. Plasticity also can be added to the model. This solver program is written in FORTAN and C language.

This FEM code does not defined units. The units are defined by the user, when input data is written for solve program. The user can choose any systems on units. Same system of units must be follow for all input, if different units is followed in same analysis, the numerical result will contain errors. Like ANSYS and Creo Simulate, it is not possible input units in different systems. For example, it is not possible to input force in "**N**" and pressure in "**psi".** The possible system of unit suggested by Dhondt user manual are as follow.

Table 7 Suggestion examples for different possible unit systems. (Dhondt, 2013)

Quantities	**System of units**		
	m, kg, s, K	**mm, g, s, K**	**mm, N, s, K**
Density	$1\frac{kg}{m^2}$	$10^{-6}\frac{g}{mm^2}$	$10^{-12}\frac{Ns^2}{mm^4}$
Mass	1kg	1g	$10^{-3}\frac{Ns^2}{mm}$
Young's Modulus	$1\frac{kg}{ms^2}$	$1\frac{g}{mms^2}$	$10^{-6}\frac{N}{mm^2}$
Force	$1\frac{kgm}{s^2}$	$10^{6}\frac{gmm}{s^2}$	1N

Calculix can support different material properties which can be by default inside material library or user defined materials. Material properties like linear elastic, isotropic hyperelastic, deformation plasticity, large deformation incremental isotropic, large deformation creep, fiber reinforced anisotropic hyperelastic, etc. A general procedure for solving static problems using Calculix is given below.

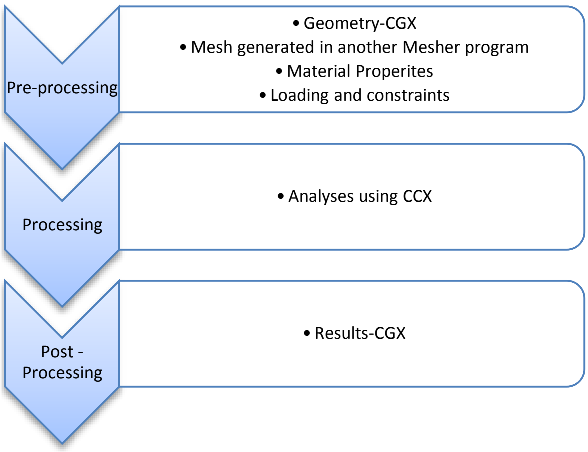
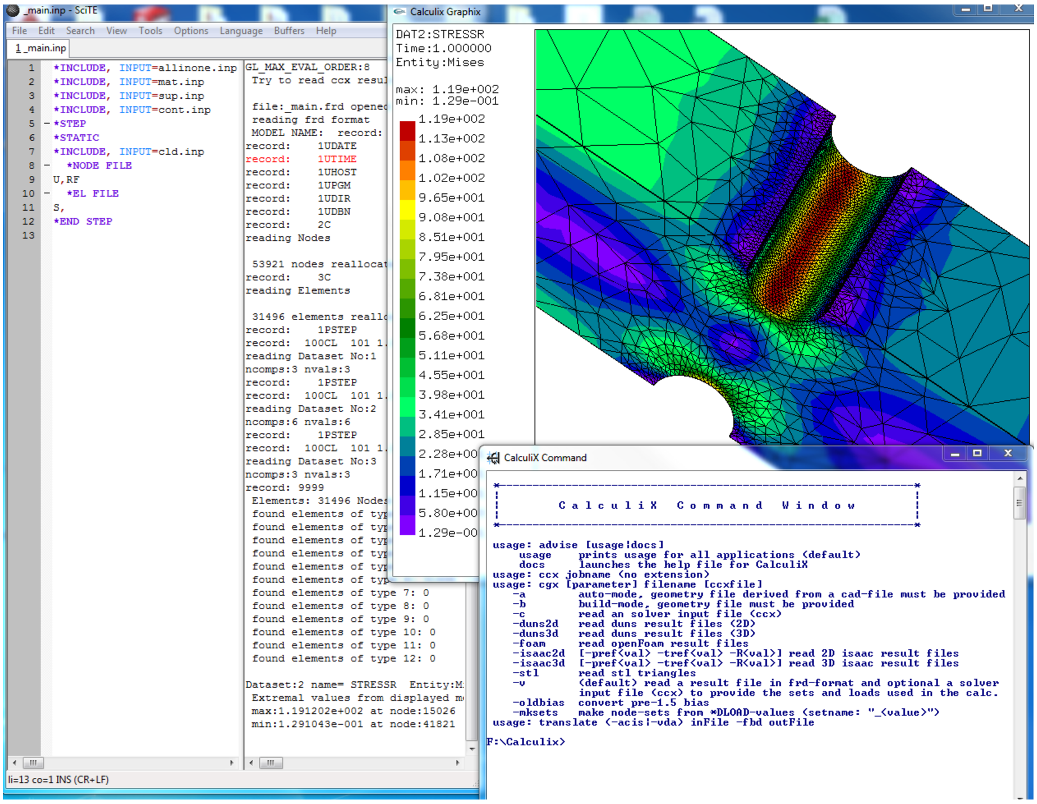


Figure 15 The basic static FEA process in Creo Simulate



```
*INCLUDE, INPUT=allinone.inp
*INCLUDE, INPUT=mat.inp
*INCLUDE, INPUT=sup.inp
*INCLUDE, INPUT=cont.inp
*STEP
*STATIC
*INCLUDE, INPUT=cld.inp
  *NODE FILE
U,RF
  *EL FILE
S,
*END STEP
```

```
GL_MAX_EVAL_ORDER:8
 Try to read ccx resul

 file:_main.frd opene
 reading frd format
 MODEL NAME:  record:
record:    1UDATE
record:    1UTIME
record:    1UHOST
record:    1UPGM
record:    1UDIR
record:    1UDBN
record:    2C
reading Nodes

 53921 nodes realloca
record:    3C
reading Elements

 31496 elements reall
record:    1PSTEP
record:  100CL  101 1.
reading Dataset No:1
ncomps:3 nvals:3
record:    1PSTEP
record:  100CL  101 1.
reading Dataset No:2
ncomps:6 nvals:6
record:    1PSTEP
record:  100CL  101 1.
reading Dataset No:3
ncomps:3 nvals:3
record: 9999
 Elements: 31496 Nodes
 found elements of typ
 found elements of typ
 found elements of typ
 found elements of typ
 found elements of typ
 found elements of typ
 found elements of type 7: 0
 found elements of type 8: 0
 found elements of type 9: 0
 found elements of type 10: 0
 found elements of type 11: 0
 found elements of type 12: 0

Dataset:2 name= STRESSR  Entity:M
 Extremal values from displayed m
 max:1.191202e+002 at node:15026
 min:1.291043e-001 at node:41821
```

```
*----------------------------------------------------------*
¦                                                          ¦
¦        C a l c u l i X   C o m m a n d   W i n d o w     ¦
¦                                                          ¦
*----------------------------------------------------------*

usage: advise [usage¦docs]
    usage    prints usage for all applications (default)
    docs     launches the help file for CalculiX
usage: ccx jobname (no extension)
usage: cgx [parameter] filename [ccxfile]
   -a        auto-mode, geometry file derived from a cad-file must be provided
   -b        build-mode, geometry file must be provided
   -c        read an solver input file (ccx)
   -duns2d   read duns result files (2D)
   -duns3d   read duns result files (3D)
   -foam     read openFoam result files
   -isaac2d  [-pref<val> -tref<val> -R<val>] read 2D isaac result files
   -isaac3d  [-pref<val> -tref<val> -R<val>] read 3D isaac result files
   -stl      read stl triangles
   -v        (default) read a result file in frd-format and optional a solver
             input file (ccx) to provide the sets and loads used in the calc.
   -oldbias  convert pre-1.5 bias
   -mksets   make node-sets from *DLOAD-values (setname: "_<value>")
usage: translate (-acis¦-vda) inFile -fbd outFile

F:\Calculix>
```

Figure 16 Calculix interface

The following element types are available inside CGX.

Table 8 Some element type and node number of Calculix

Solid Elements	Surface or area elements	Line Elements
Brick element (8-node) (20-node)	Plane stress and stain element (6-node) (8-node)	Beam element (3-node) (2-node)
Incompressible elements (20-nodes)	Axisymmetric element (6-node) (8-node)	Linear and Nonlinear springs (2-node)
Tetrahedral elements (4-node) (10 node)	Shell Element (6 node) (8-node)	Gap element (2-node)
Wedge Element (6-node) (15-node)		

5.4 Z88/Z88AURORA

Z88 is an open source or free, fast, powerful and compact FEA programs which can solve wide range of structural mechanical and static problems. The FEM code was originally created by Professor Frank Rieg in 1986 and currently being further developed by his team at University of Bayreuth under his supervision.

Z88Aurora is extended version of compact Z88 which was developed in 2009. **Auora** stand for **a**dvance **u**ser inte**r**face f**o**r **r**eliable fe**a**. It contained Z88 solvers but it offers a GUI and completely new pre- and post-processing software. It is more user friendly program which can be used with basic knowledge of FEM. It is based on h-element method. Z88Aurora is a software for static analysis. It can perform linear and nonlinear static analysis. The nonlinearity can be used for large displacement analysis only. Beside static analysis, it can simulate thermal and natural frequency analysis. This code is a powerful and complex computer program but it is still under development, therefore all the functions are not implemented. How Z88 deals with other programs and utilities hasn’t been tested yet. The units are managed by user. It can’t convert units from one system of unit to another. The material database integrated in Z88 Aurora uses the unit’s mm/t/N.

A general procedure for solving static problems using Z88Auora is given below.

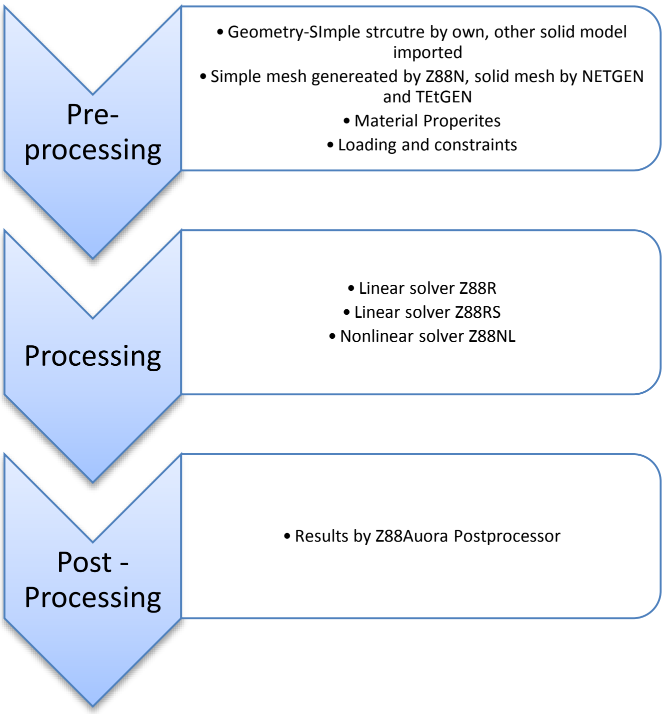
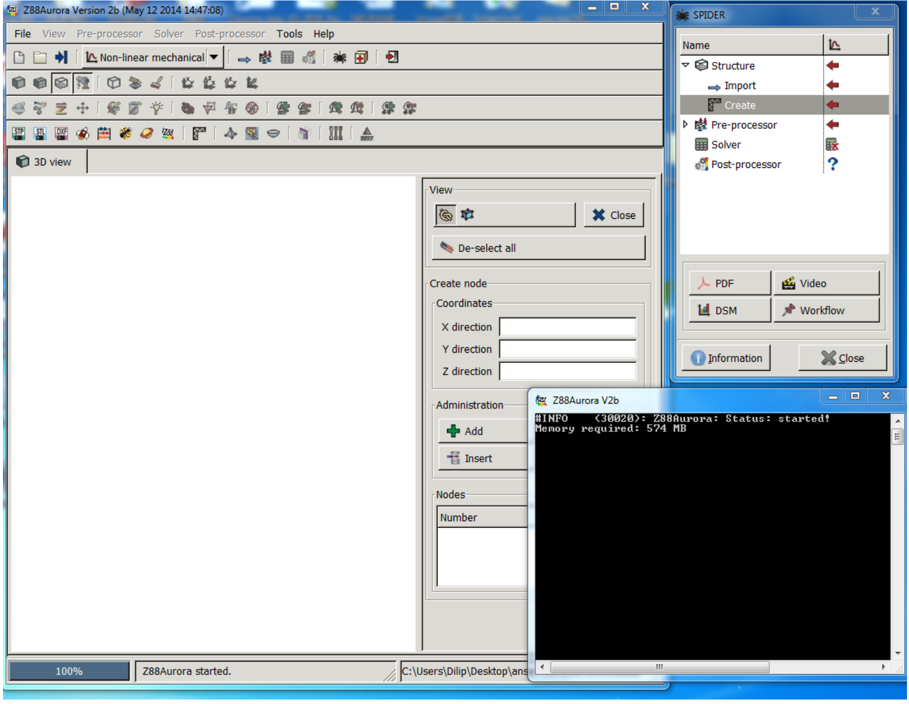


Figure 17 The basic static FEA process in Z88Auora

Simple structural shape using beams, truss and frame can be build inside Z88 pre-processor using different beam and truss element which is called super element inside Z88. For solid model, different CAD software needed to model and the geometry and for solid meshed, two freeware mesher program has been integrated inside Z88Aurora. Material library consists of few material library and user defined material is also possible. Currently it support only linear materials. There are three different solver based on following features inside Z88 solver program.

- A Cholesky solver
- A direct sparse matrix solver,
- A sparse matric iteration solver



Z88Aurora started.

```
#INFO    (30020): Z88Aurora: Status: started!
Memory required: 574 MB
```

s

Figure 18 Z88Aurora interface

The following element are supported by Z88Aurora.

Table 9 Some element type supported by Z88Aurora solver.

Solid Elements	Surface or area elements	Line Elements
Hexahedron (linear or quadratic shape function)	Plane stress element (quadratic, Cubic shape function)	Beam element
Tetrahedron (linear or quadratic shape function)	Plate element(quadratic, cubic)	Truss element
Tetrahedral elements (4-node) (10 node)	Shell element (quadratic)	Cam element
Wedge Element (6-node) (15-node)		

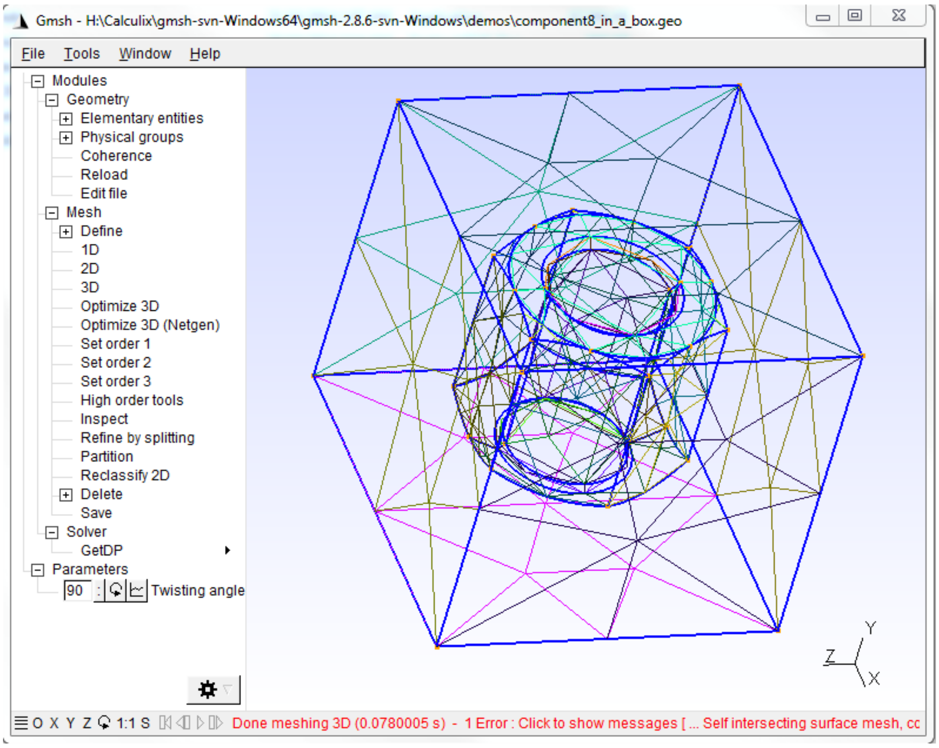
5.5 GMSH

Gmsh is an open source 3D finite element mesh generator with a build in CAD engine. It is also a post processor. It was created by Christophe Geuziane and Jean-Francois Remacle. The CAD engine enable to create 1-D, 2-D or 3-D solid model. Gmesh is divided into four modules which are geometry, mesh, solver and post-processing modules. All input to these modules are given to the program either by using graphical user interface or in the text files which is written in own Gmsh scripting language.

Gmesh uses a boundary representation to describe geometries. All model are created in a bottom-up flow by successively defining points, lines, surface and volume. For example, to model a solid model, first point are defined and the points are joined together lines. The line segment can be straight line, circles, splines, ellipses, etc. These line segments together form a surface and combining these surface will result a solid model.

The second module is mesh generator. It can generate three-dimensional solid mesh by different element shape and size. Line, surface and volume element are possible to create using Gmsh. Triangular, quadrilateral, tetrahedral, prism, hexahedral and pyramids type of element are created. The mesh generated by Gmsh can be import in different FEM codes.

This is not actually a FEM solver but it has own default solver which is called GetDP. It can solve only linear static and thermal problems. External solvers can be interfaced with Gmsh using Unix or TCP/IP sockets, which permits to modify solver parameters, launch external computations and post-process the results.


Gmsh - H:\Calculix\gmsh-svn-Windows64\gmsh-2.8.6-svn-Windows\demos\component8_in_a_box.geo

Twisting angle: 90

Done meshing 3D (0.0780005 s) - 1 Error : Click to show messages [... Self intersecting surface mesh, cc

Figure 19 Gmsh interface

5.6 General Comparison

In this chapter, general comparison among above studied FEM codes has been outline on the basis of working fields, analysis types, internal modules, software quality, operating systems, user interface and files types. Also, comparisons have been made on the basis of element types and convergences.

The following table shows the comparison between studied commercial and open source FEM codes on the basis of working fields. Most of the general commercial FEM codes are general purpose FEA tools. The basis modules integrated with their CAD software can simulate basic structural and thermal analysis. In case of Open source FEM codes, they are usually written for certain field problems. All the studied codes can simulate linear or nonlinear static problems. Here, on the Table 10, types of structural analysis are listed according to their respective number mention below under structural analysis types and a tick mark is given for specific codes if they can simulate listed working fields.

Table 10 Commercial and open source FEM codes classification by working fields

Working Fields	**Ansys**	**Creo Simulate**	**Calculix**	**Z88 Aurora**	**Gmsh**
Structural Analysis	1,2a 2b 2c 3 4	1,2a 2b 2c 3 4	1,2a 2b 2c 3 4	1 2a 4	1
Explicit analysis	√		√		
Thermal analysis	√	√	√	√	
Fluid Dynamic	√		√		
Electromagnetic	√				
Coupled Physic	√				

Structural analysis type

1. Static linear
2. Static nonlinear
    a. Geometric nonlinearity
    b. Contact Modelling
    c. Material Nonlinearity
3. Buckling
4. Frequency

Another comparison is made on the basis of internal module within. The basic FEA process includes pre-processor, processor and post-processor modules. Most of commercial FEM codes have all the modules but free FEM codes does not have all the modules. Some of them are only solver or processor which need another software to pre-process and post-process. The following table shows the comparison on the basis of internal modules.

Table 11 Commercial and open source FEM codes classification by internal module within

FEM codes	Pre-processor	Solver(proces-sor)	Post-Processor
Ansys	√	√	√
Creo Simulate	√	√	√
Calculix	√*	√	√
Z88/Z88 Aurora	√*	√	√
Gmsh	√		√

*another pre-processor is used for quality mesh.

Now, the comparison is made on the basis of user interface and learning curve quality. This comparison is made by the author during the interaction between author and the FEM codes while analysing the Test Examples. The learning curve quality are based on author own opinions. The following table shows comparison on the basis of user interface and learning curve quality.

Table 12 User interface and learning curve quality

FEM codes	Classification				
	1	2	3	4	5
Ansys					√
Cero Simulate					√
Calculix		√			
Z88/Z88 Aurora				√	
Gmsh			√		

The following table compare the codes on the basis of operating systems and base language. It was found that the normal operating system for commercial and open source FEM codes is Windows. Some of open source codes were found only for Linux. Here, the base language refers to the programing language on which the FEM code was written. Commercial FEM codes does not provide any source codes, therefore, they also does not mention what types of programing language is used on developing their software's. But, open source FEM codes provide the source codes and the codes can be change according to user needs. Most of the open source codes were written in C or FORTRAN. Nowadays, basic FEM codes can be found on MATLAB platform also.

Table 13 Operating system and base language

FEM codes	Linux	Windows	Apple OS	Base Language
Ansys	√	√		-
Creo Simulate		√		-
Calculix	√	√		C++
Z88/ Z88 Aurora	√	√	√	C
Gmsh	√	√	√	C++

In FEA process, it is very important to transfer data form one software to another software. All the analysts in the world do not use same FEM codes. So there must be a communication between FEM codes to transfer data from one company to another company or consultant. Geometry are modelled using CAD and they are imported to FEM codes. Some FEM codes can perform only processing whereas some can only perform pre-processing and post-processing only. Hence, it is very important to know the types of files supported by each FEM codes during analysis. The following table provide comparison on the basis of files types their native modes, files types which can be imported on and files types which can be exported to another FEM codes. A files type's descriptions has been provide below the table. The number inside the table represent files types provided under files type's descriptions below. ANSYS and Creo support wide range of files. More information can be obtained from their respective websites.

Table 14 File types supported (native, import and export)

FEM codes	**Native**		**Import**	**Export**
	Input	**Output**		
Ansys	ANSYS	ANSYS	1, 2, 3, 4, 5, 6, 7, 8, 9, 10	15, 16, 2
Creo Simulate	Creo	Creo	1, 2, 3, 4, 5, 6, 7, 8, 9, 10	13, 15
Calculix	14	*.FRD	5, *.FRD, 22 20	14, 15, 21, 22, 20, 5
Z88 Aurora	19	19	1, 2, 12, 13, 14, 15, 17, 18	5
Gmsh	4,11	4	4, 2,3,5,1,23, 13	4,2,3,5,1,23, 13

Files types descriptions:

1. Geometry STEP(*.STP, *.STEP)
2. Geometry IGES (*.IGS, *.IGES)
3. Geometry BRep (*.BREP)
4. Geometry Gmsh GEO (*.GEO)
5. STL files (*.STL)
6. Geometry ACIS
7. Geometry AutoCAD, Inventro
8. Geometry Catia
9. Geometry Creo
10. Geometry Unigraphic
11. Mesh- Gmsh MSH (*.MSH)
12. AutoCAD DXF files (*.DXF)
13. NASTRAN files (*.BDF, *.NAS)
14. ABAQUS files (*.INP)
15. ANSYS files (*.ANS)
16. ANSYS Design Modeler Database (*.agbd)
17. COSMOS files (*.COS)
18. Z88 files (*.TXT)
19. Z88Aurora project files (*.Z88)
20. OpenFOAM
21. CodeAster
22. Duns
23. IMAGE (*.BMP, *.JPG, *.JPEG, *.PBM, *.PNG, *.PPM)

6 DETAILED TEST EXAMPLES

There are five test examples in which three are linear static analysis and two are nonlinear static analysis problems. Test example 4 and test example 5 have contact and material nonlinearity respectively. The examples are very simple which can be analysed with basic theory. In most examples, 1-D or 2-D analysis will be sufficient, but 3-D analysis are performed. In each test examples, 3-D geometry is modelled with a CAD program (Creo Parametric 2.0) and exported to a STEP file format. Same STEP file is used in each FEM codes as a geometry sources.

In this chapter, each test example detailed are provided below with test examples schematic, material properties, geometry properties and loading data. Dimension of the test example schematic drawing are in millimetre.

6.1 Test example 1

Rectangular plate with circular hole subjected to tensile loading

Reference:	J. E. Shigley, Mechanical Engineering Design, McGraw-Hill, 1st Edition, 1986, Table A-23, Figure A-23-1, p. 673
Analysis type:	Linear static analysis
Element Type	Solid

A rectangular plate with centre hole is subject to tensile pressure load over one of end face and which is fixed on opposite site. The geometric and material properties of the test example are given below. Also, the theoretical solution based on simplified mathematical model assumptions are mention below.

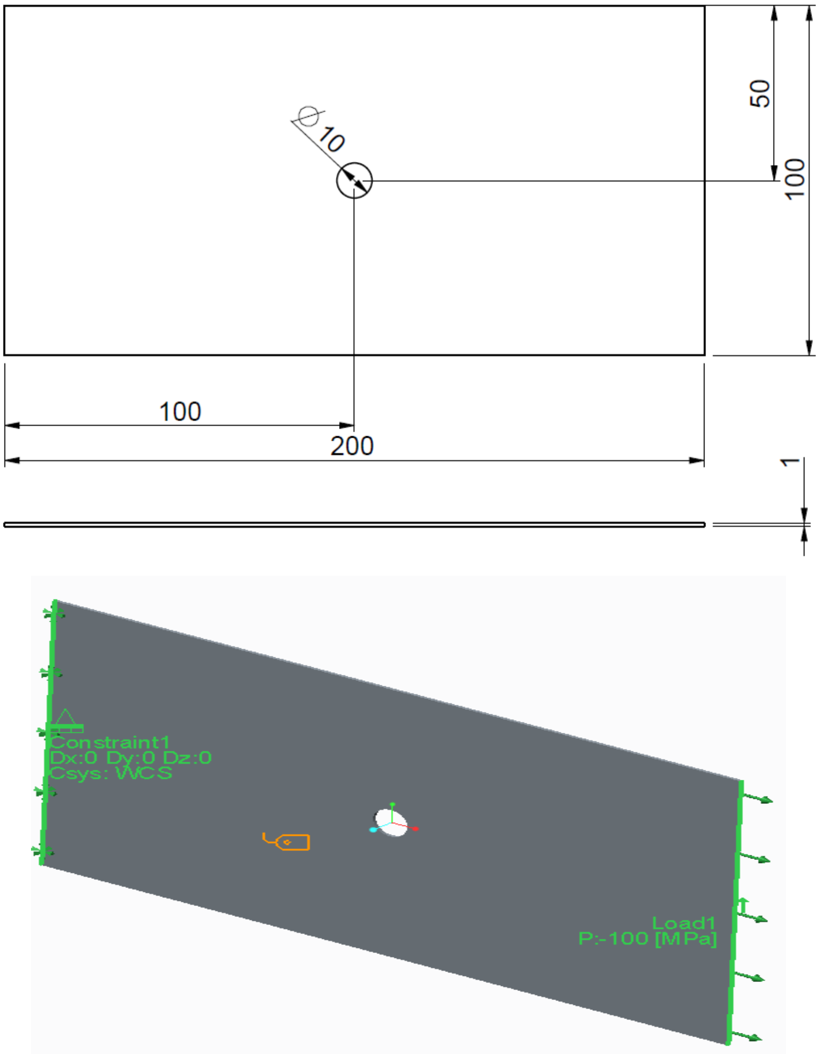


Figure 20 Schematic of test example 1

Table 15 Material, geometric properties and loading of Test Example 1

Material Properties	Geometric Properties	Loading
E= 210 GPa v=0.27	Length L=200 mm Width b = 100 mm Thickness t=1 mm Hole radius d= 10 mm	Pressure σ_0= 100 MPa

Theoretical solution

The results from theoretical solution are as follow. The detailed of theoretical solution are given in Appendix.

Table 16 Theoretical result of Test Example 1

Results(quantities)	Theoretical Results
Maximum Displacement (mm)	**0.09524**
Maximum Von Mises stress (MPa)	**302.349**

6.2 Test example 2

Square cross section cantilever beam with middle semicircle notched subjected to horizontal load at the end face

Reference:	Any Strength of Material Books
Analysis type:	Linear static analysis
Element Type	Solid

A square cross section cantilever beam with semicircle notched in the middle is subjected to horizontal forced over one of end face and is fixed on opposite face. The geometric and material properties of the test example are given below. Also, the theoretical solution based on simplified mathematical model assumptions are mention below.

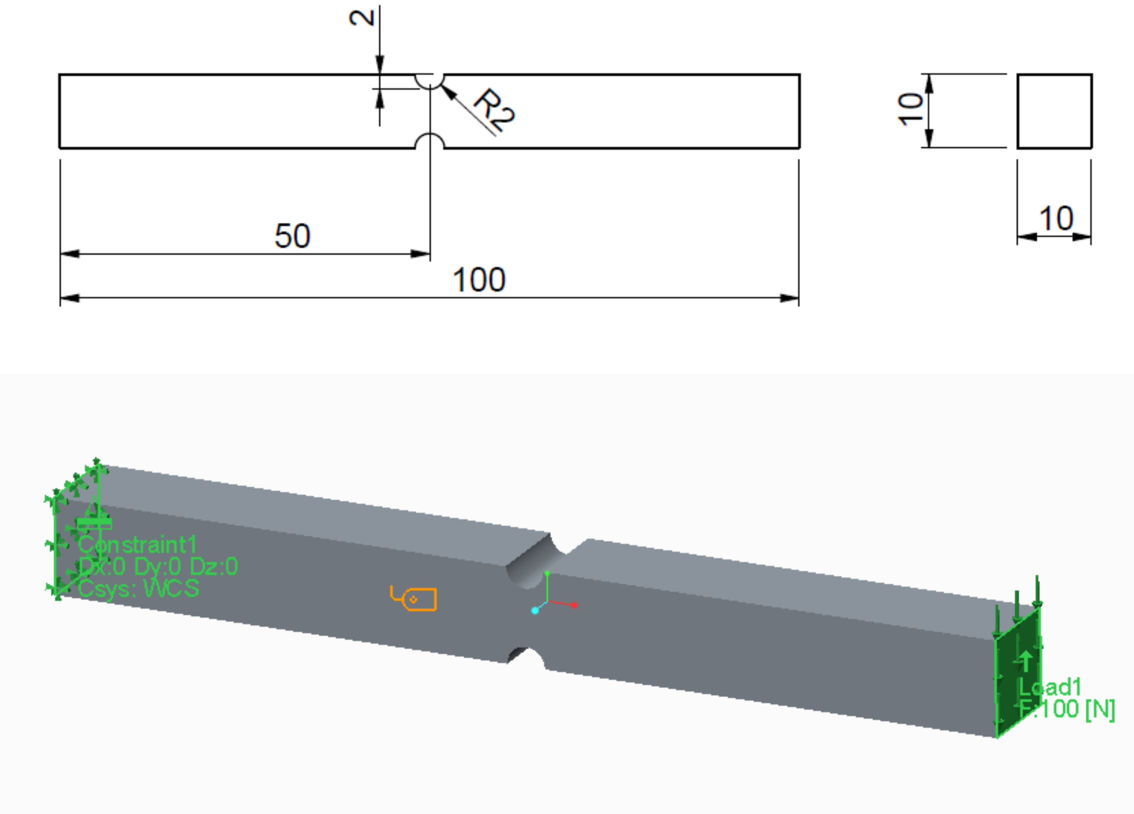


Figure 21 Schematic of test example 2

Table 17 Material, geometric properties and loading of Test Example 2

Material Properties	**Geometric Properties**	**Loading**
E= 210 GPa v= 0.27	Length of beam L= 100 mm Width= 10 mm Height =10 mm Radius of semicircle = 2 mm	Force F= 100 N

Theoretical solution

The results from theoretical solution are as follow. The detailed of theoretical solution is given in Appendix.

Table 18 Theoretical result of Test Example 2

Results(quantities)	Theoretical Results
Maximum Displacement (mm)	**0.19048**
Maximum Von Mises stress (MPa)	**117.041**

6.3 Test example 3

Cantilever I-beam subjected to distributed force.

Reference:	Any strength of material books
Analysis type:	Linear static analysis
Element Type	Solid

A cantilever I-beam length of 1000mm is subjected to distributed load and is fixed on one end. The geometric and material properties of the test example are given below. Also, the theoretical solution based on simplified mathematical model assumptions are mention below.

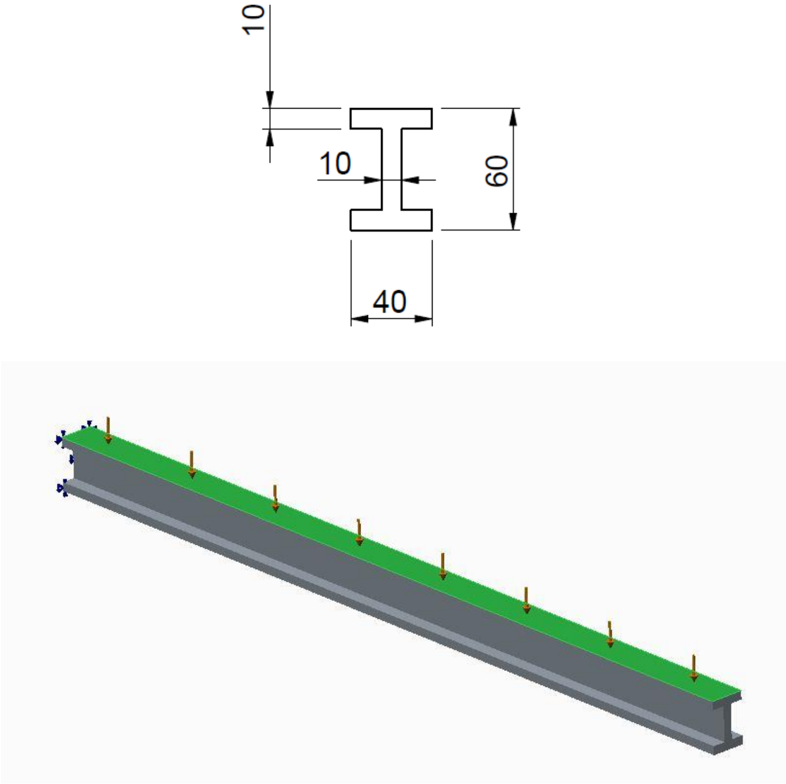


Figure 22 Schematic of test example 3

Table 19 Material, geometric properties and loading of Test Example 3

Material Properties	Geometric Properties	Loading
E= 210 GPa v=0.27	Length=1000 mm	Distributed load = 5 kN/m

Theoretical solution

The results from theoretical solution are as follow. The detailed of theoretical solution is given in Appendix.

Table 20 Theoretical result of Test Example 3

Results(quantities)	Theoretical Results
Maximum Displacement (mm)	**5.31463**
Maximum Von Mises stress (MPa)	**133.929**

6.4 Test example 4

Two part in a contact with each other.

Reference:	Sebestian, R. (n.d.). Avanced Calculix Tutorial. 1st ed. [ebook] Libremechics.com. Available at: http://www.libremechanics.com.
Analysis type:	Nonlinear static analysis
Element Type	Solid

A rotatory hook on a top base is fixed and loaded with a constant force in circular surface of hook core. The rotatory hook consists of two parts; hook base and hook core as shown in figure below. The contact area is formed by two parts on a uniform conic area. Downward force is applied on a surface of hook core. The material and loading properties of test example are given below.

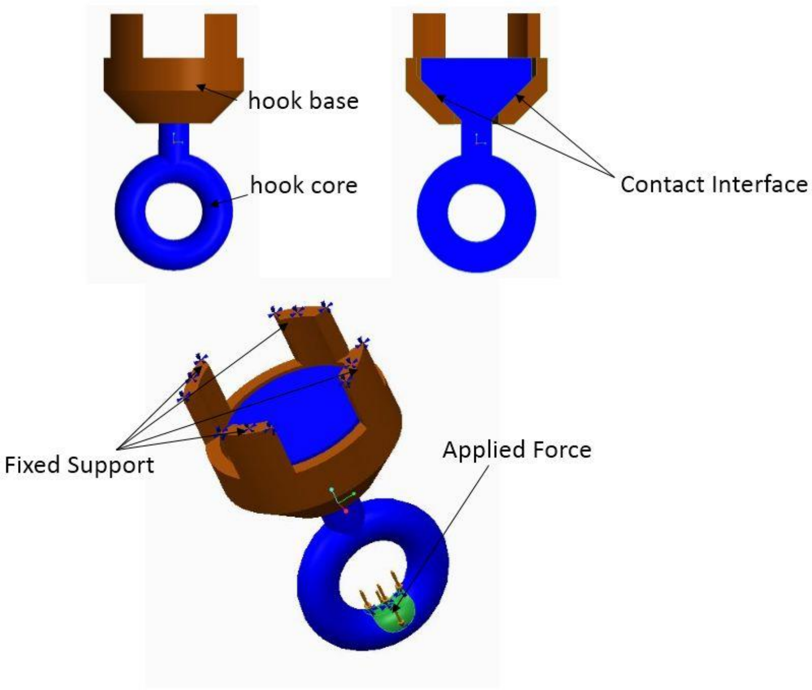


Figure 23 Schematic of test example 4

Table 21 Material, geometric properties and loading of Test Example 4

Material Properties	**Geometric Properties**	**Loading**
<u>Hook base</u> E= 110 GPa v=0.35 <u>Hook core</u> E = 200 GPa v = 0.26	See reference for the geometry	Force= 3000 N

6.5 Test example 5

<u>**Collapse load analysis of squared cross section cantilever beam**</u>

Reference: Any strength of material books
Analysis type: Nonlinear static analysis
Element Type Solid

A squared cross-section cantilever beam is loaded continuously until plastic hinge are formed and the structure collapsed. The geometric and material properties of the test example are given below. Also, the theoretical solution based on simplified mathematical model assumptions are mention below.

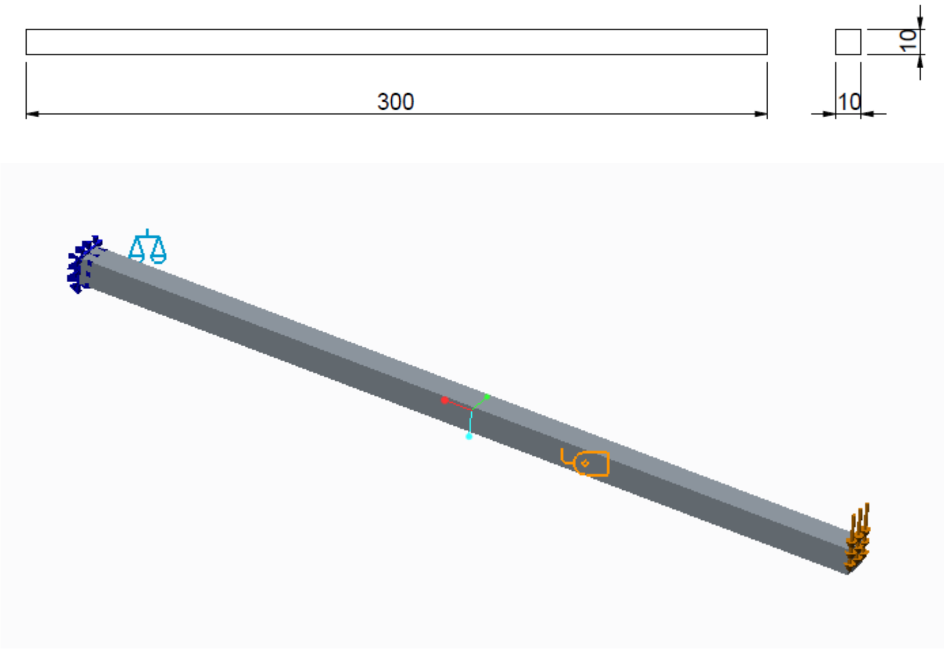


Figure 24 Schematic of test example 5

Table 22 Material, geometric properties and loading of Test Example 5

Material Properties	Geometric Properties	Loading
E= 210 GPa v = 0.27 Yield strength = 280 MPa Tangent Modulus = 50 MPa	Length=300 mm Width= 10 m Height =10 mm	Force applied up to 250N

Theoretical solution

The collapsed load or limit load at which the beam collapsed is **233.33 N**. The detailed solution is given in Appendix.

7 TEST RESULTS AND COMPARISON

Static analysis are performed for each test examples. In case of free FEM codes, external mesher program like Gmsh or Netgen are used to generate mesh. Most meshes for Calculix are generated using Netgen since calculix can read native Netgen files. Same unit system are used wherever applicable during analysis.

In this chapter, the results are compared to each other. A comparison between theoretical solution and numerical solution from each FEM codes are compared. The comparison quantities are maximum stress and maximum displacement. Maximum Von mises stress and maximum total displacement from each FEM codes result are included here. The contour plot comparison can be done as well. In some of the result, only the place where there is maximum stress are included in following snapshots. The result snapshots are taken from post processing window of each FEM codes.

At the end, the result variation between commercial FEM codes and free FEM codes are included in this chapter.

7.1 Test results of example 1

7.1.1 Test result data

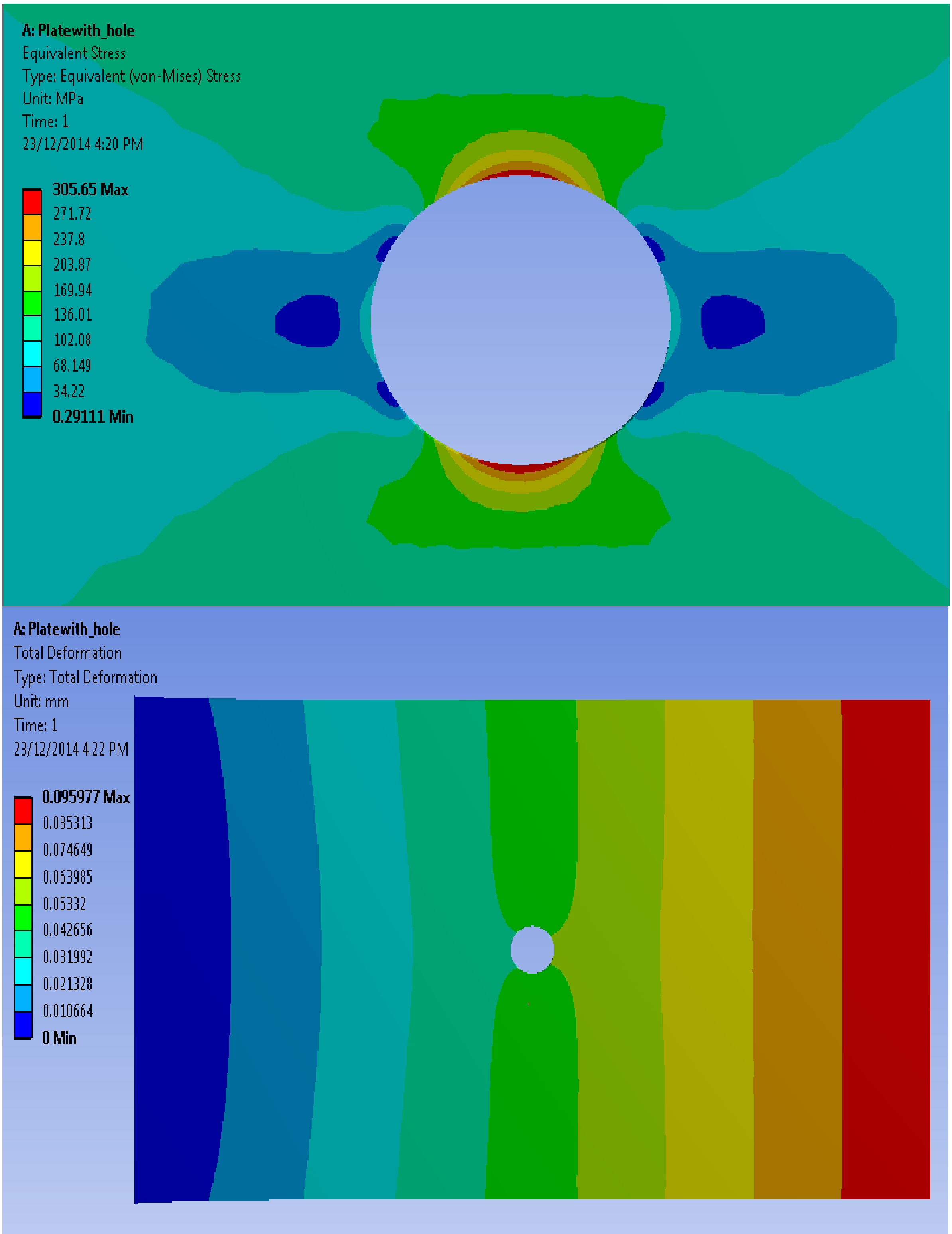


Figure 25 Stress and displacement plots from ANSYS for Test Example 1.

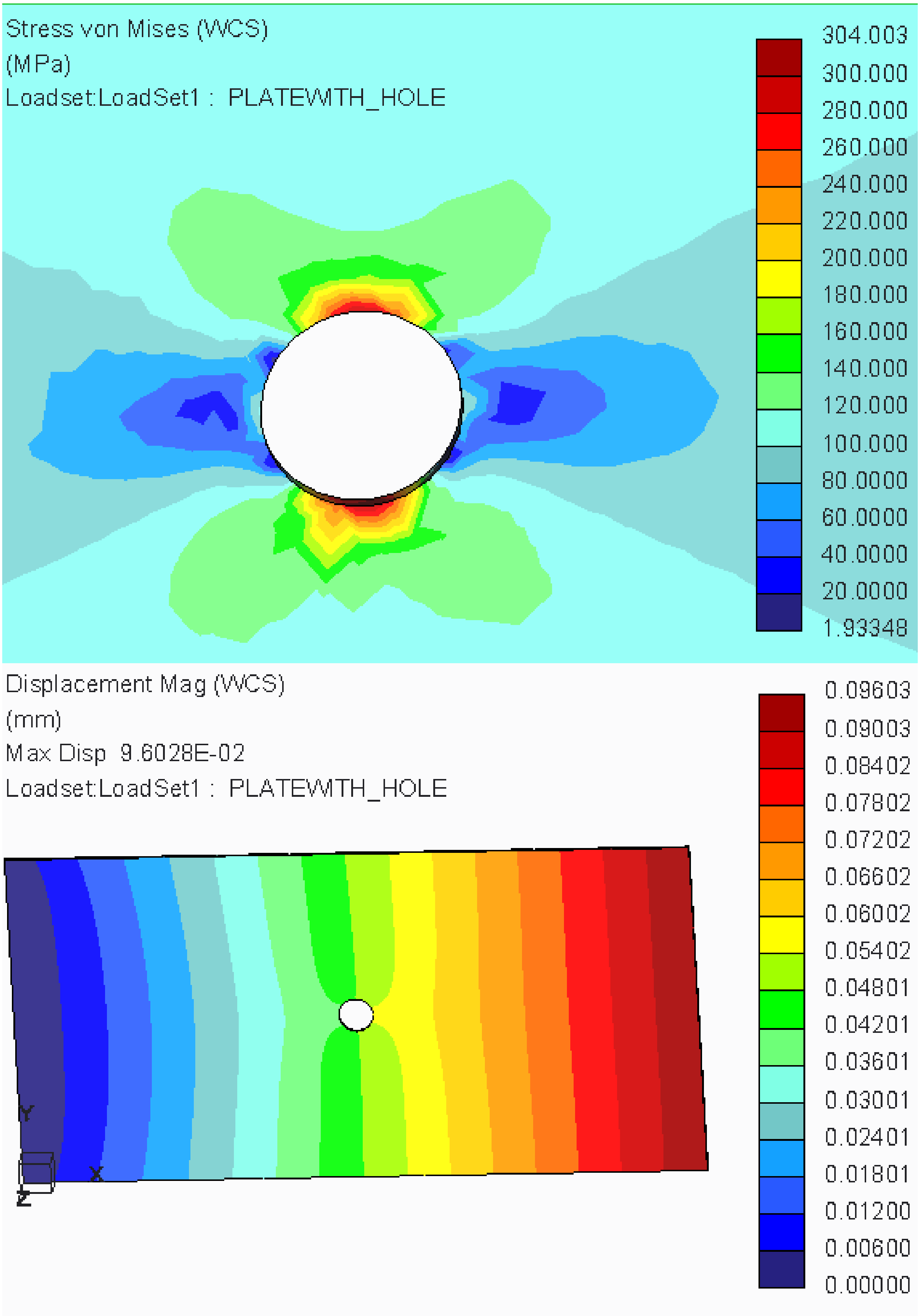


Figure 26 Stress and displacement plots from Creo Simulate for Test Example 1.

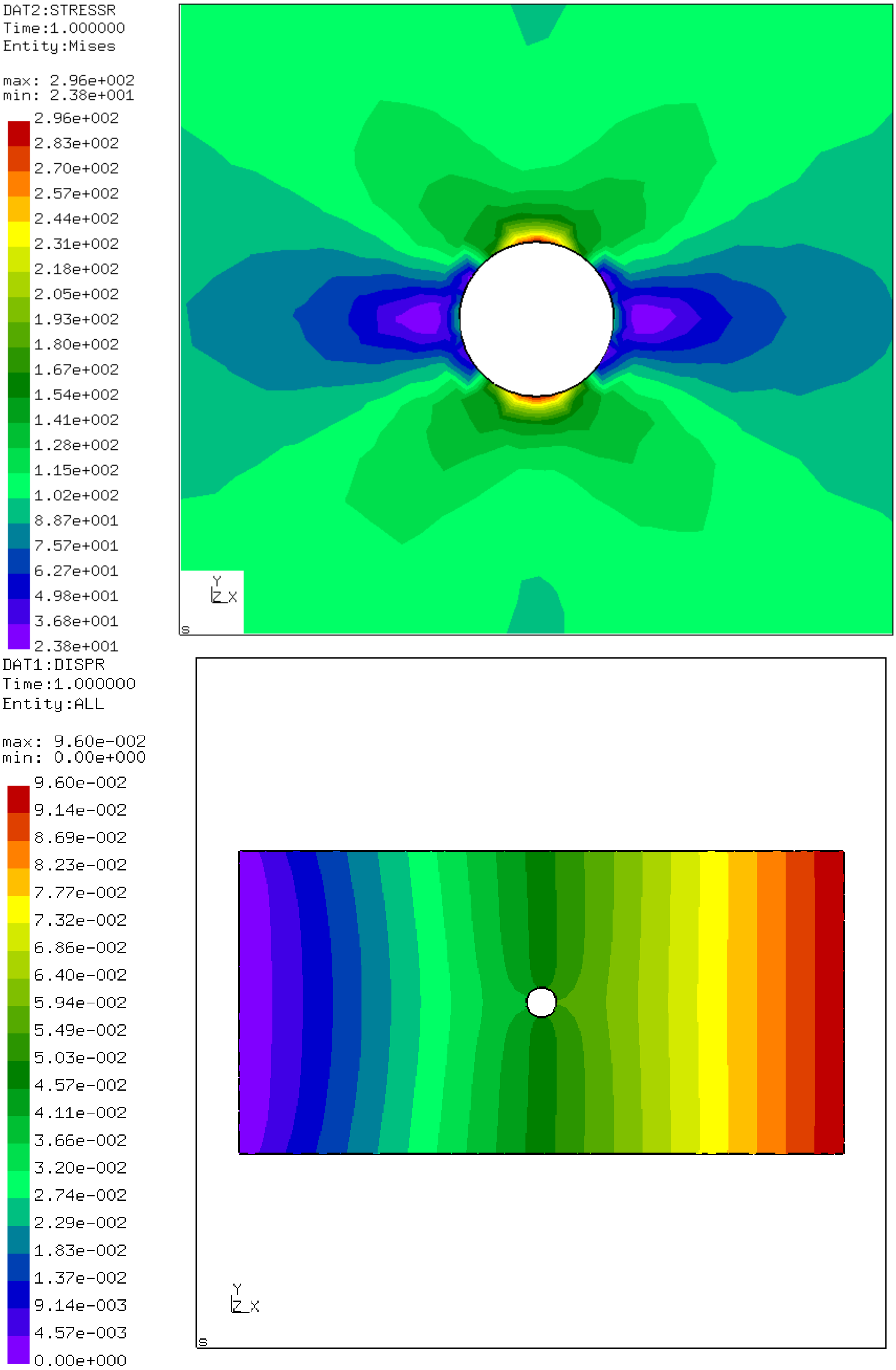


Figure 27 Stress and displacement plots from Calculix for Test Example 1.

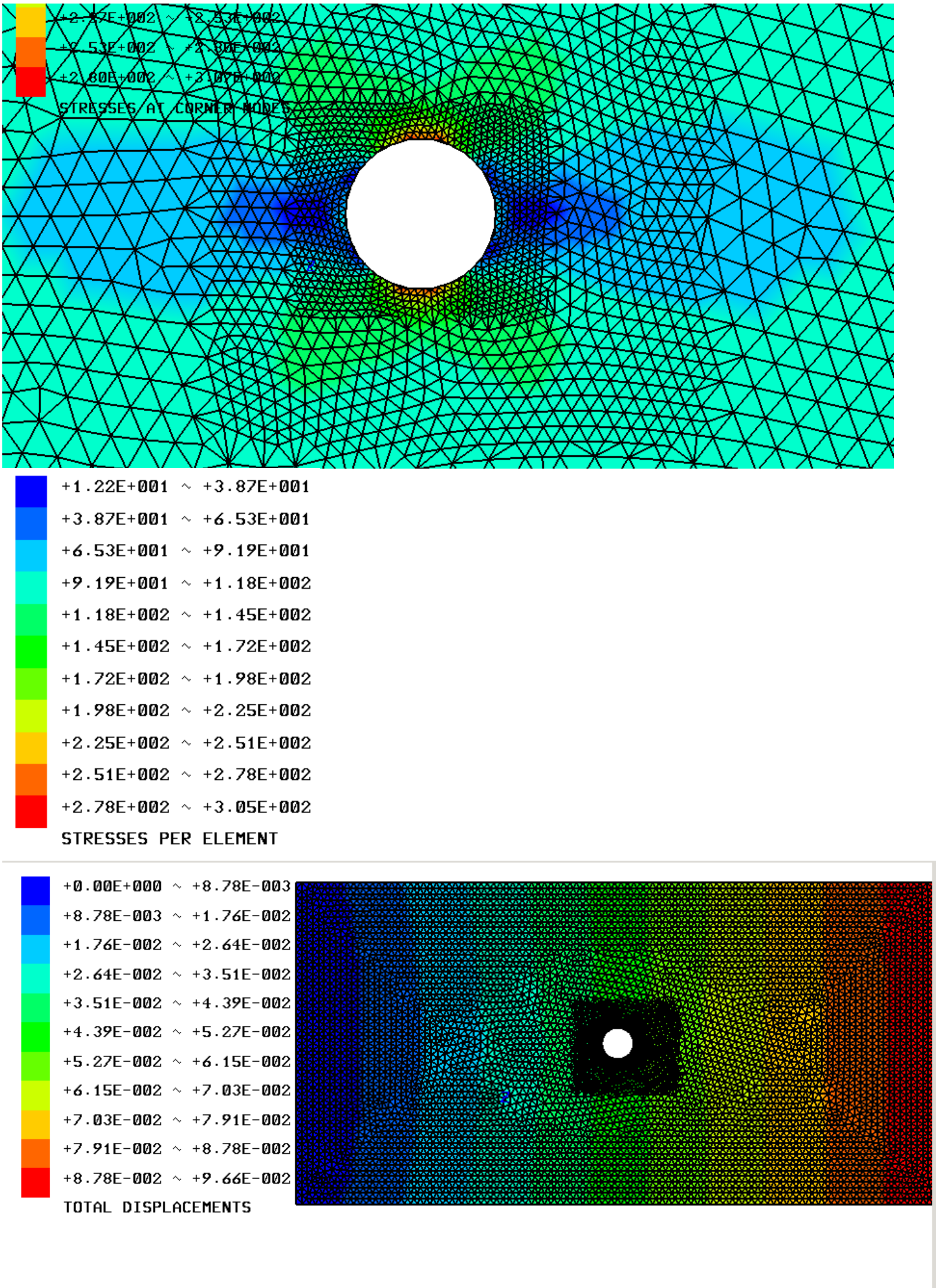


Figure 28 Stress and displacement plots from Z88Aurora for Test Example 1.

7.1.2 Comparison of result data

The following table show comparison of result obtained from theoretical solution and FEM codes. The results quantities are maximum total displacement and maximum Von mises stress. The results from each FEM codes are close to result obtained from theoretical solution with small percentage errors. Relative percentage errors has been calculated on the basis of theoretical values.

Table 23 Results comparison of test example 1

Results (quantities)	**Theoretical Results**	**FEM Codes**	**FEM Results**	**Relative Error in Percentage (%)**
Maximum Displacement (mm)	**0.09524**	Ansys	0.096022	**0.82%**
		Creo simulate	0.09603	**0.83%**
		Calculix	0.096	**0.80%**
		Z88Aurora	0.0966	**1.43%**
Max. Von Mises stress (MPa)	**302.349**	Ansys	305.65	**1.09%**
		Creo Simulate	304.003	**0.55%**
		Calculix	296	**2.10%**
		Z88Aurora	305	**0.88%**

The following table show comparison of result from commercial FEM code and result obtained from open source FEM codes. The idea of this comparison is to check how far the result from free or open source FEM codes deviate from commercial FEM codes. The average is taken from Ansys and Creo and the variation from average is calculated for free code from the average.

Table 24 Result comparison between commercial and open source FEM codes.

	Max. displacement (mm)	**Max. Von Mises stress (MPa)**
Ansys	0.096022	305.65
Creo Simulate	0.09603	304.003
Average	0.096026	304.8265
	Variation from average (%)	
Calculix	**0.03%**	**2.90%**
Z88 Aurora	**0.60%**	**0.06%**

Hence, all the results from each FEM codes for Test Example 1 is similar to each other. The result variation from commercial FEM codes (Ansys and Creo Simulate) with open source FEM codes (Calculix and Z88 Aurora) is very small.

7.2 Test results of example 2

7.2.1 Test result data

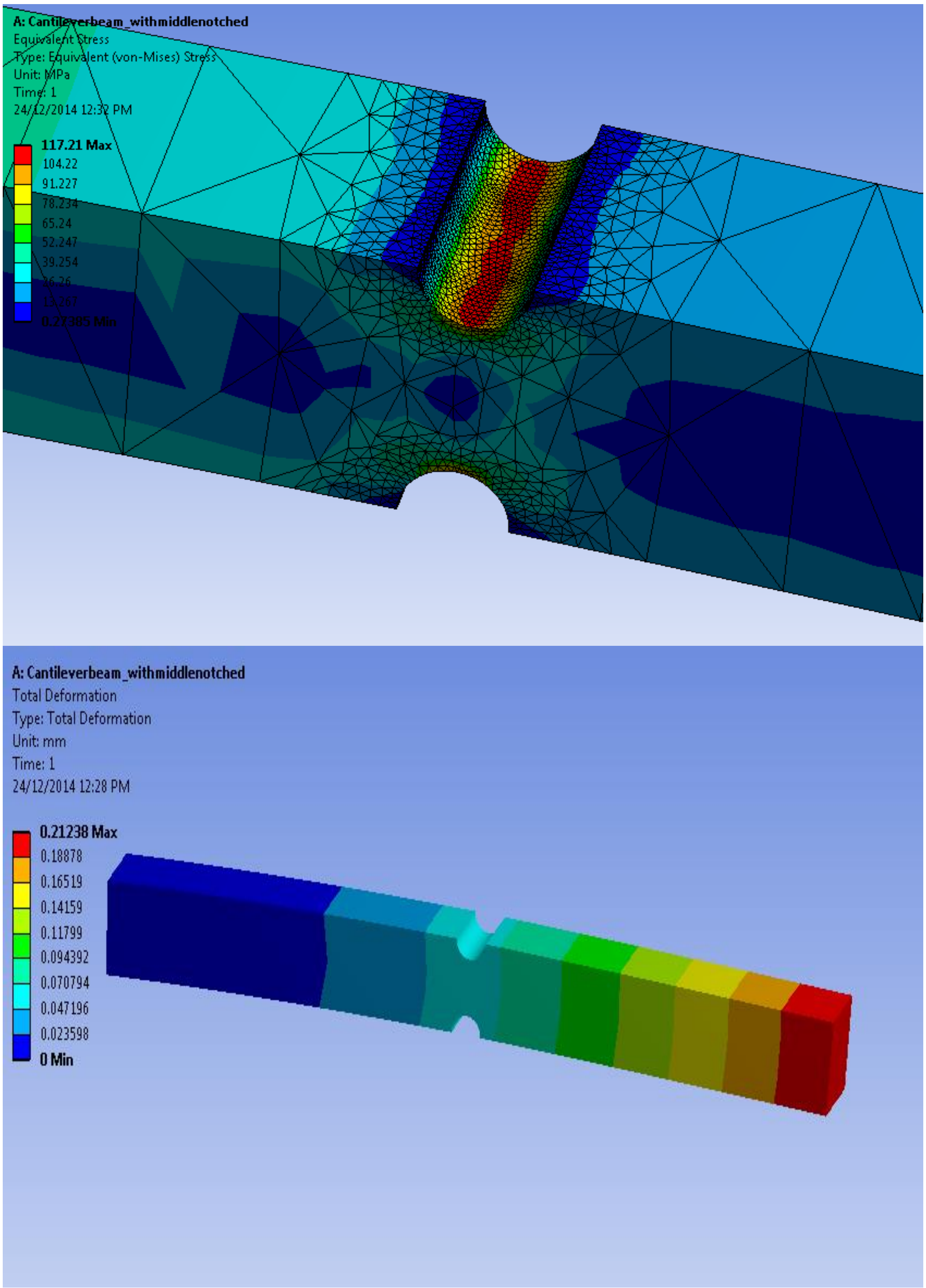


Figure 29 Stress and displacement plots from ANSYS for Test Example 2

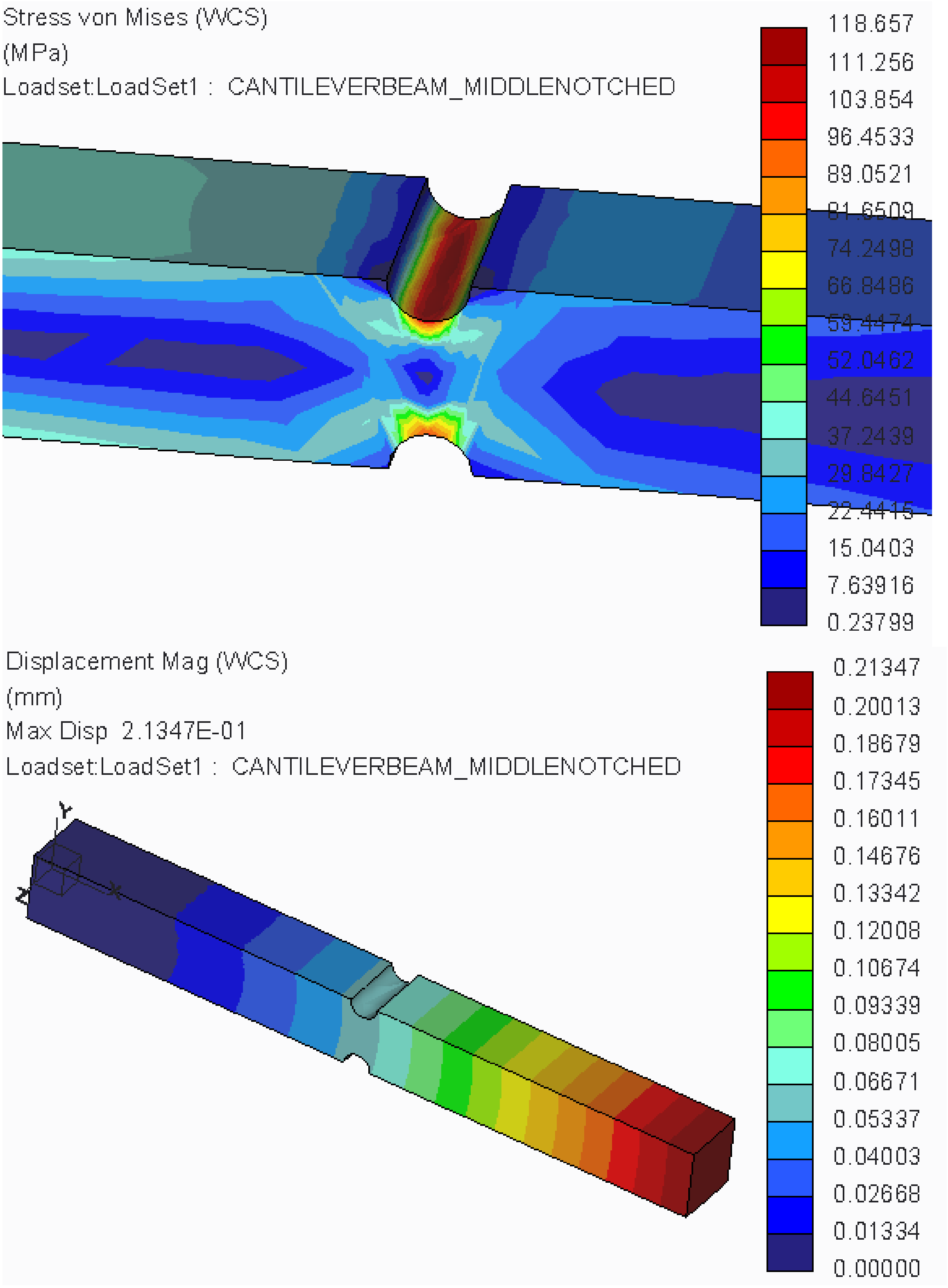


Figure 30 Stress and displacement plots from Creo Simulate for Test Example 2.

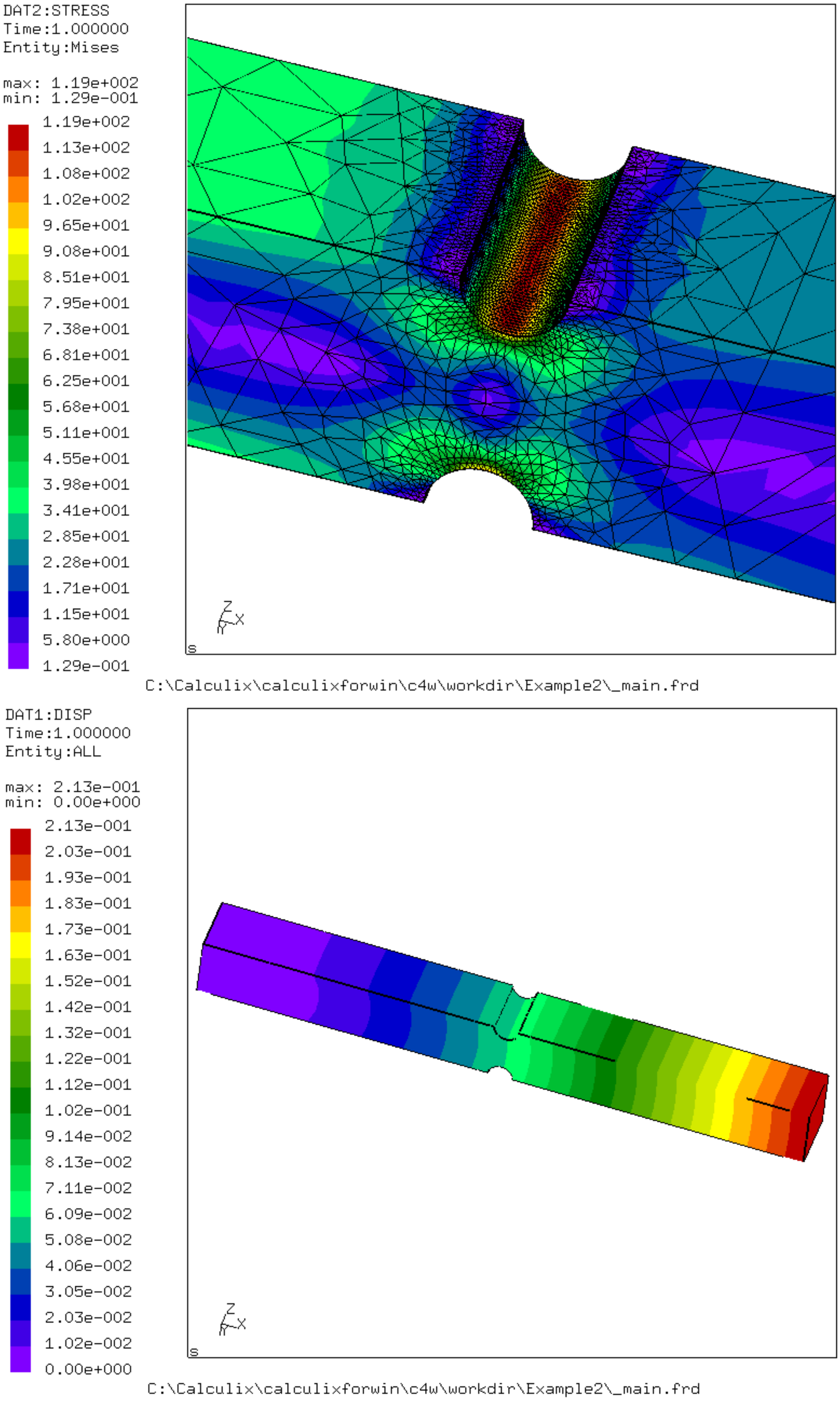


Figure 31 Stress and displacement plots from Calculix for Test Example 2.

Result plot from Z88

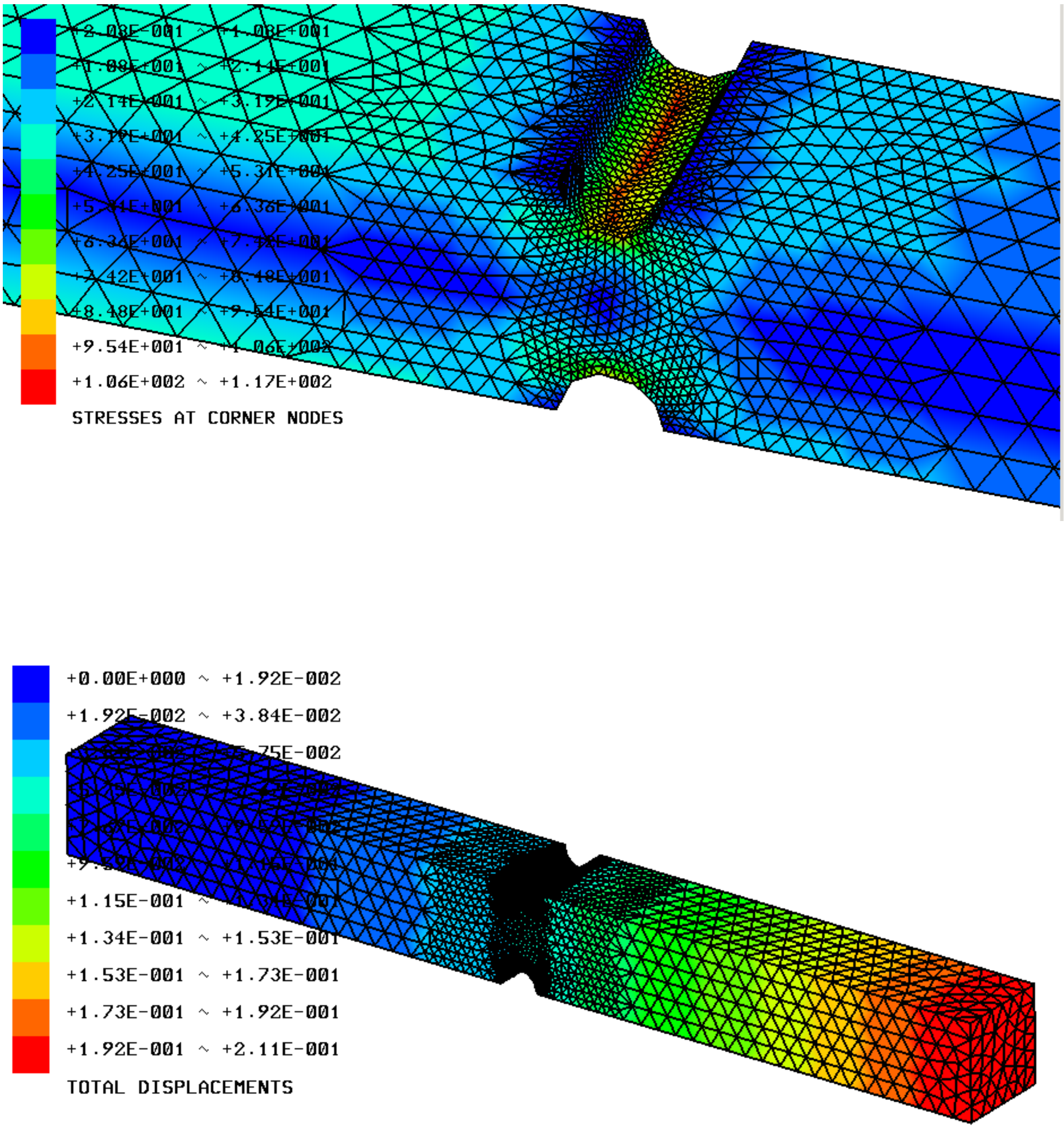


Figure 32 Stress and displacement plots from Z88Aurora for Test Example 2.

7.2.2 Comparison of results data

The following table show comparison of result obtained from theoretical solution and FEM codes. The results quantities are maximum total displacement and maximum Von mises stress. The results from each FEM codes are close to result obtained from theoretical solution with small percentage errors. Relative percentage errors has been calculated on the basis of theoretical values. Here, in this test example, theoretical value for maximum displacement was calculated assuming there is no semi-circle notched in the middle which leads more percentage error as compared to test example 1.

Table 25 Results comparison of test example 2

Results (quantities)	**Theoretical Results**	**FEM Codes**	**FEM Results**	**Relative Error in Percentage (%)**
Maximum Displacement (mm)	**0.19048**	Ansys	0.21238	**11.50 %**
		Creo simulate	0.21347	**12.07 %**
		Calculix	0.213	**11.82 %**
		Z88Aurora	0.211	**10.77 %**
Max. Von Mises stress (MPa)	**117.041**	Ansys	117.21	**0.14 %**
		Creo Simulate	118.657	**1.38 %**
		Calculix	119	**1.67 %**
		Z88Aurora	117	**0.04 %**

The following table show comparison of result from commercial FEM code and result obtained from open source FEM codes. The idea of this comparison is to check how far the result from free or open source FEM codes deviate from commercial FEM codes. The average is taken from Ansys and Creo and the variation from average is calculated for free code from the average.

Table 26 Result comparison between commercial and open source FEM codes.

	Max. displacement (mm)	**Max. Von Mises stress (MPa)**
Ansys	0.21238	117.21
Creo Simulate	0.21347	118.657
Average	0.212925	117.9335
	Variation from average (%)	
Calculix	**0.04 %**	**0.90 %**
Z88 Aurora	**0.90 %**	**0.79 %**

Hence, all the results from each FEM codes for Test Example 2 is similar to each other. The result variation from commercial FEM codes (Ansys and Creo Simulate) with open source FEM codes (Calculix and Z88 Aurora) is very small.

7.3 Test results of example 3

7.3.1 Test results data

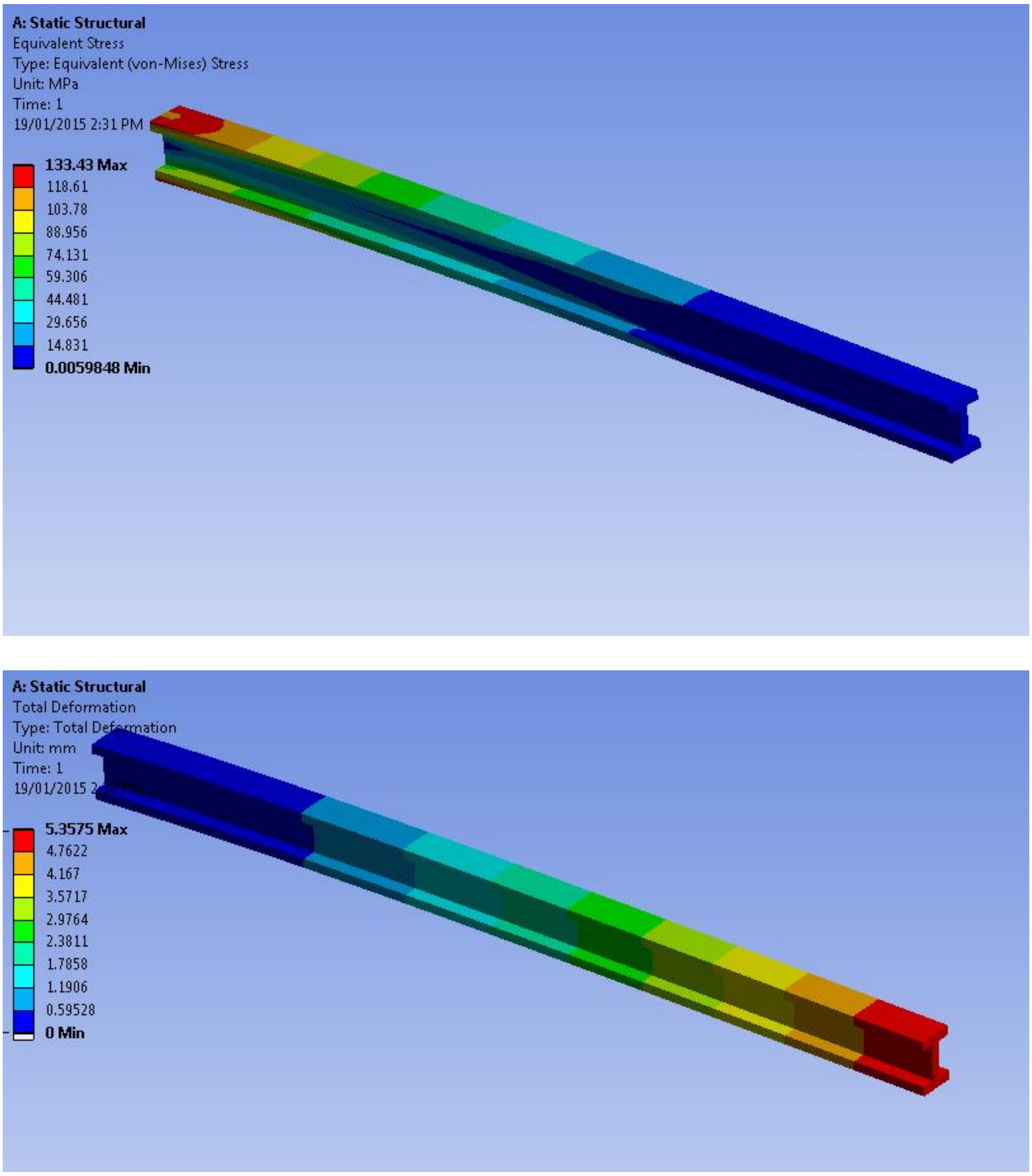
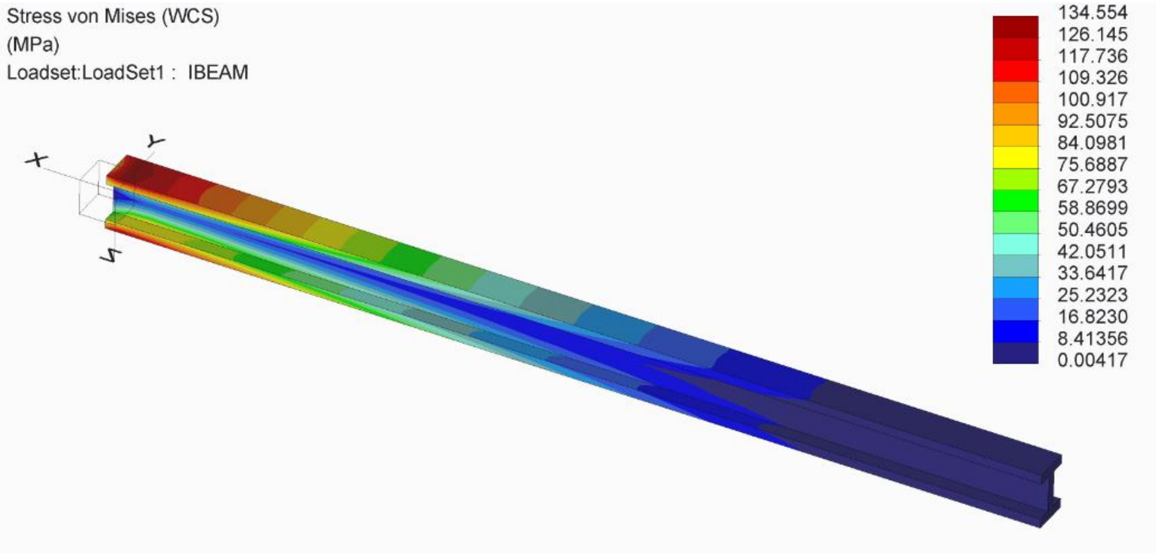


Figure 33 Stress and displacement plots from ANSYS for Test Example 3.



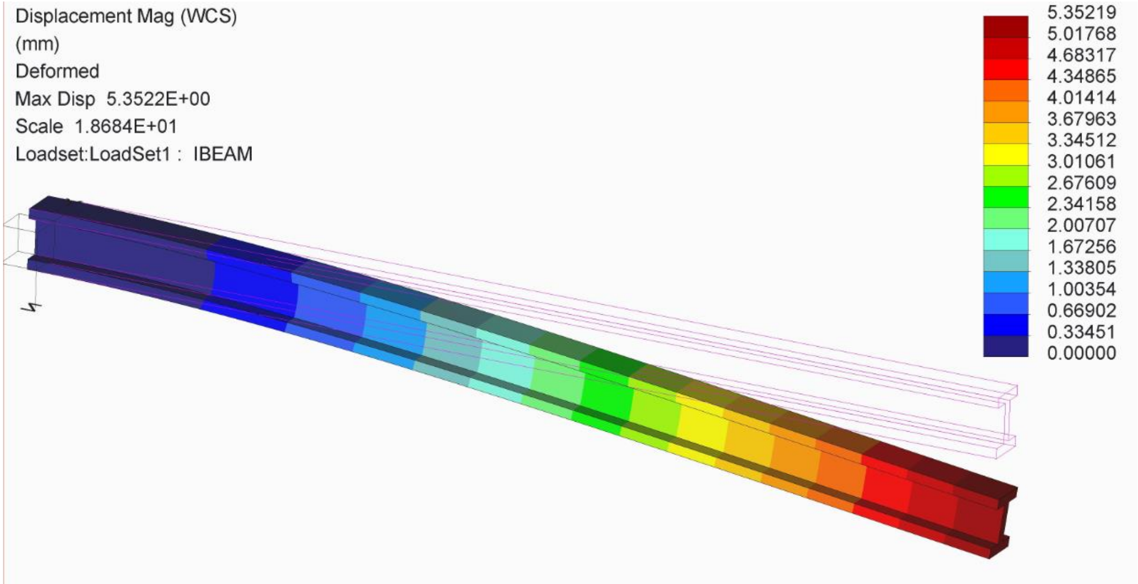


Figure 34 Stress and displacement plots from Creo Simulate for Test Example 3.

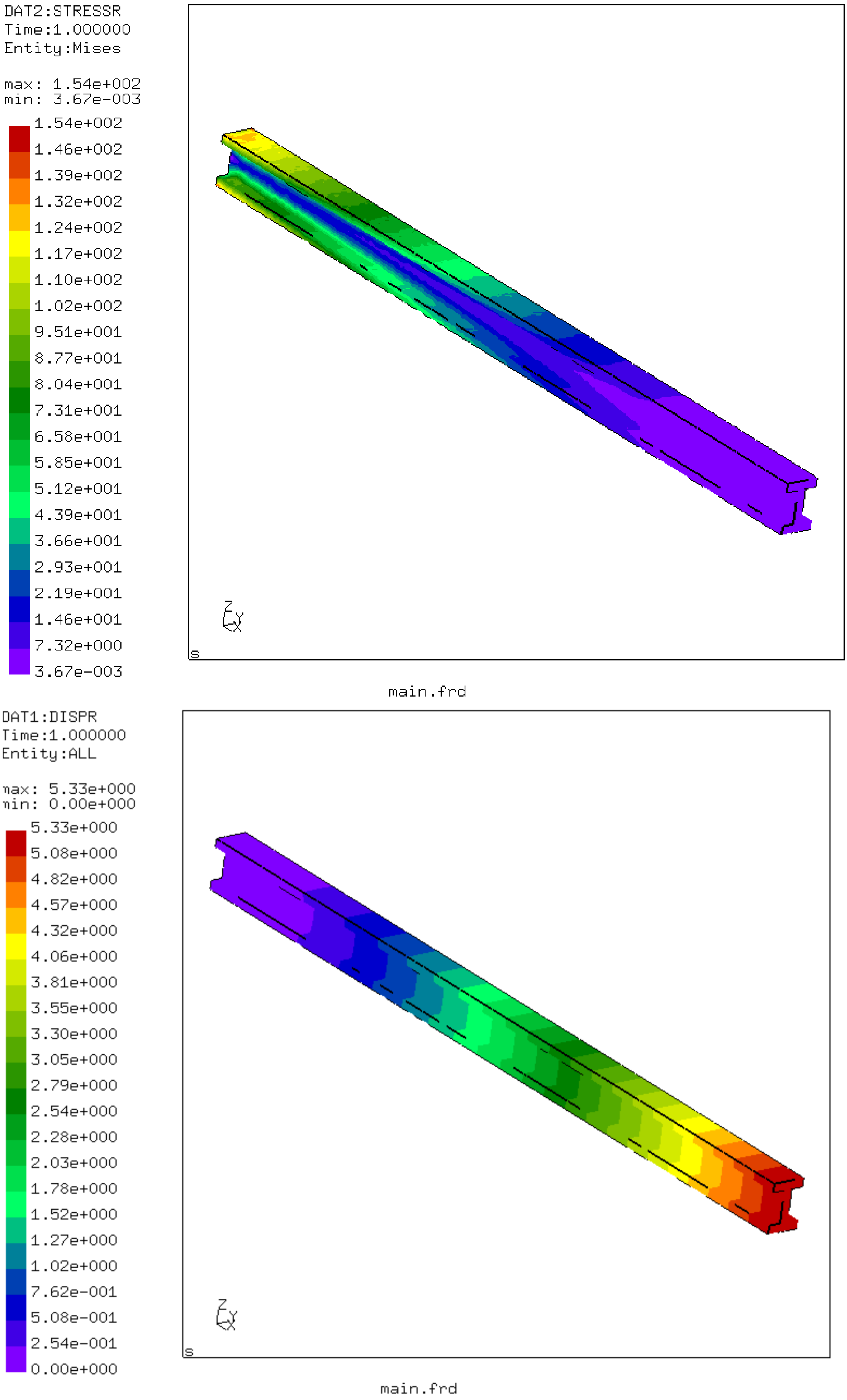


Figure 35 Stress and displacement plots from Calculix for Test Example 3.

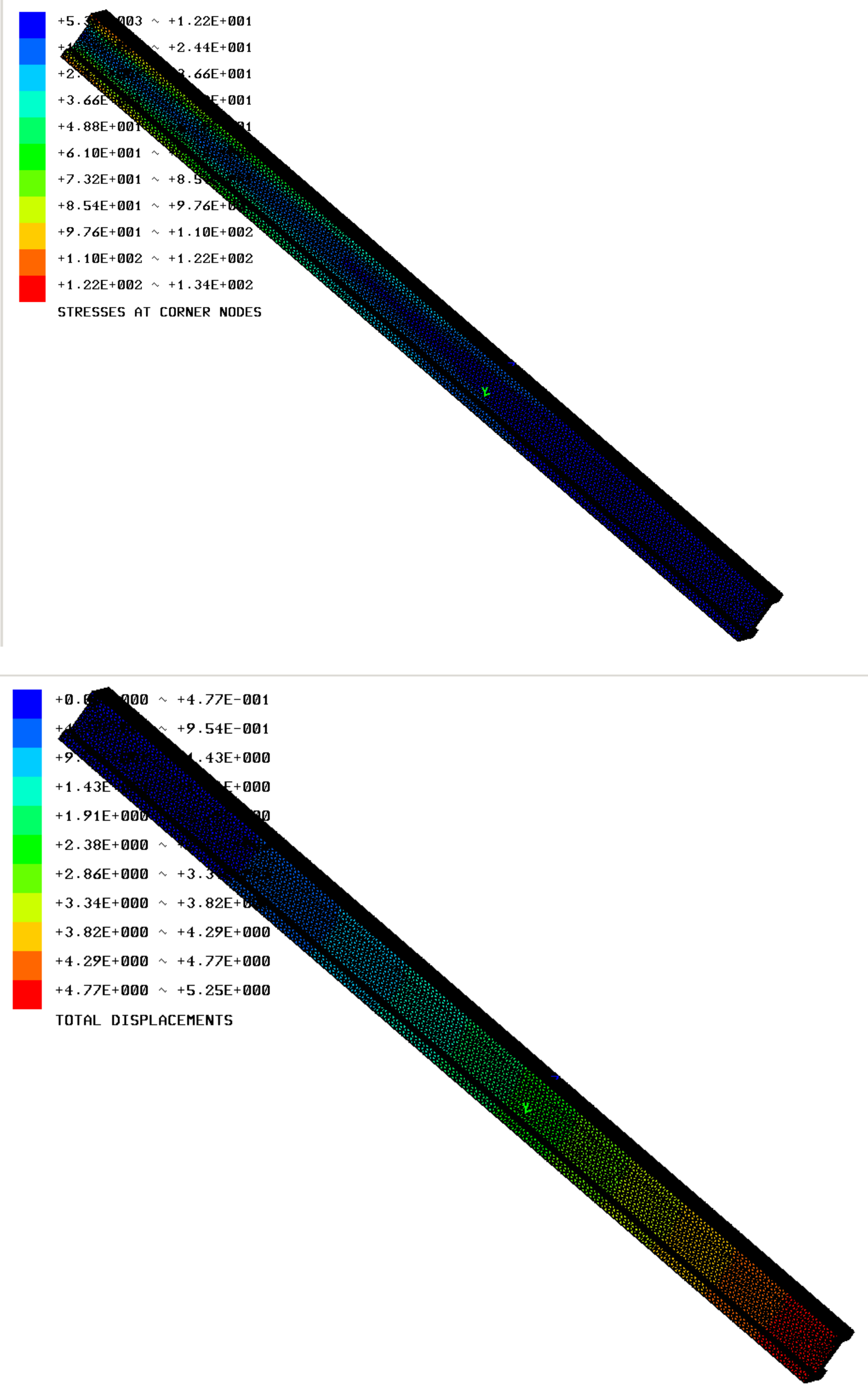


Figure 36 Stress and displacement plots from Z88 Aurora for Test Example 3.

7.3.2 Comparison of results data

The following table show comparison of result obtained from theoretical solution and FEM codes. The results quantities are maximum total displacement and maximum Von mises stress. The results from each FEM codes are close to result obtained from theoretical solution with small percentage errors. Relative percentage errors has been calculated on the basis of theoretical values. Here, maximum bending stress is the theoretical value.

Table 27 Results comparison of test example 3

Results (quantities)	**Theoretical Results**	**FEM Codes**	**FEM Results**	**Relative Error in Percentage (%)**
Maximum Displacement (mm)	5.31463	Ansys	5.3575	**0.81 %**
		Creo simulate	5.35219	**0.71 %**
		Calculix	5.33	**0.29 %**
		Z88Aurora	5.25	**1.22 %**
Max. Von Mises stress (MPa)	133.929	Ansys	133.43	**0.37 %**
		Creo Simulate	134.554	**0.47 %**
		Calculix	154	**14.99 %**
		Z88Aurora	134	**0.05 %**

The following table show comparison of result from commercial FEM code and result obtained from open source FEM codes. The idea of this comparison is to check how far the result from free or open source FEM codes deviate from commercial FEM codes. The average is taken from Ansys and Creo and the variation from average is calculated for free code from the average.

Table 28 Result comparison between commercial and open source FEM codes.

	Max. displacement (mm)	**Max. Von Mises stress (MPa)**
Ansys	5.3575	133.43
Creo Simulate	5.35219	134.554
Average	5.354845	133.992
	Variation from average (%)	
Calculix	**0.46 %**	**14.93 %**
Z88 Aurora	**1.96 %**	**0.01 %**

Here, all the result obtained from FEM codes are similar to each in total displacement but the result obtained from calculix have bigger percentage error. This is because tetra element were used during meshing with Netgen for calculix. The errors can be minimize with change in element type and mesh size.

7.4 Test results of example 4

7.4.1 Test result data

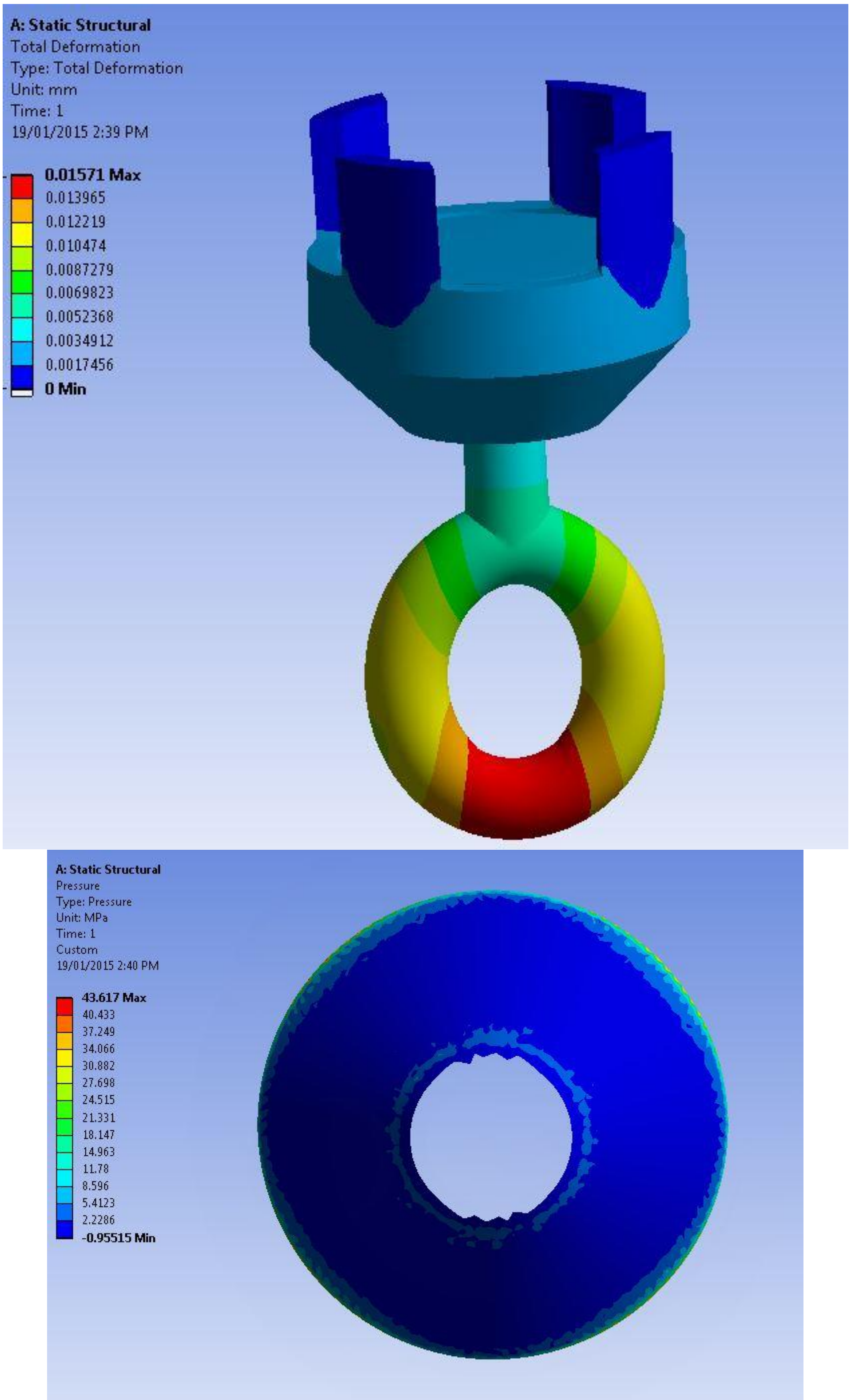


Figure 37 Total deformation and contact pressure plots from ANSYS for Test Example 4

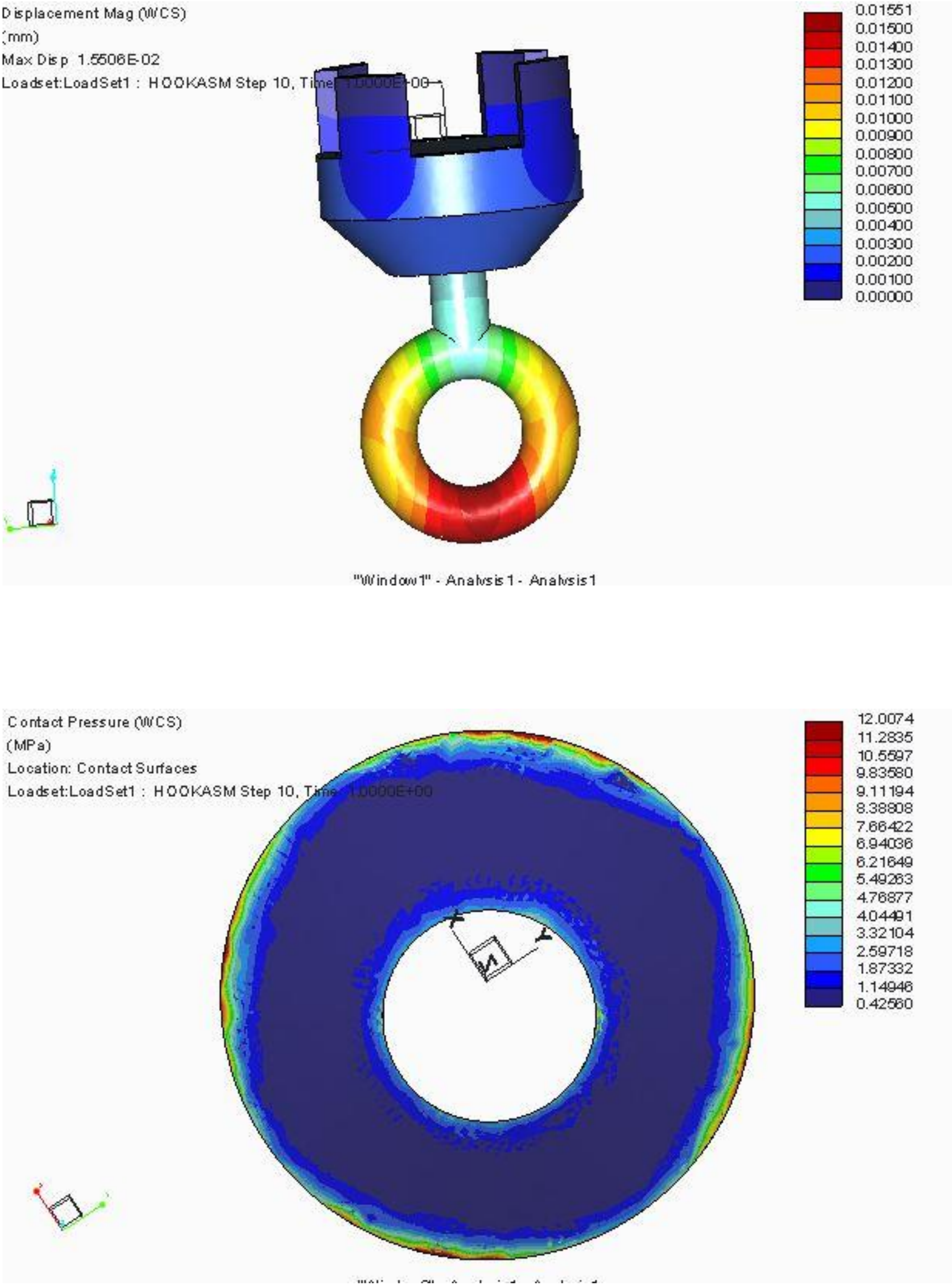


Figure 38 Total deformation and contact pressure plots from Creo for Test Example 4.

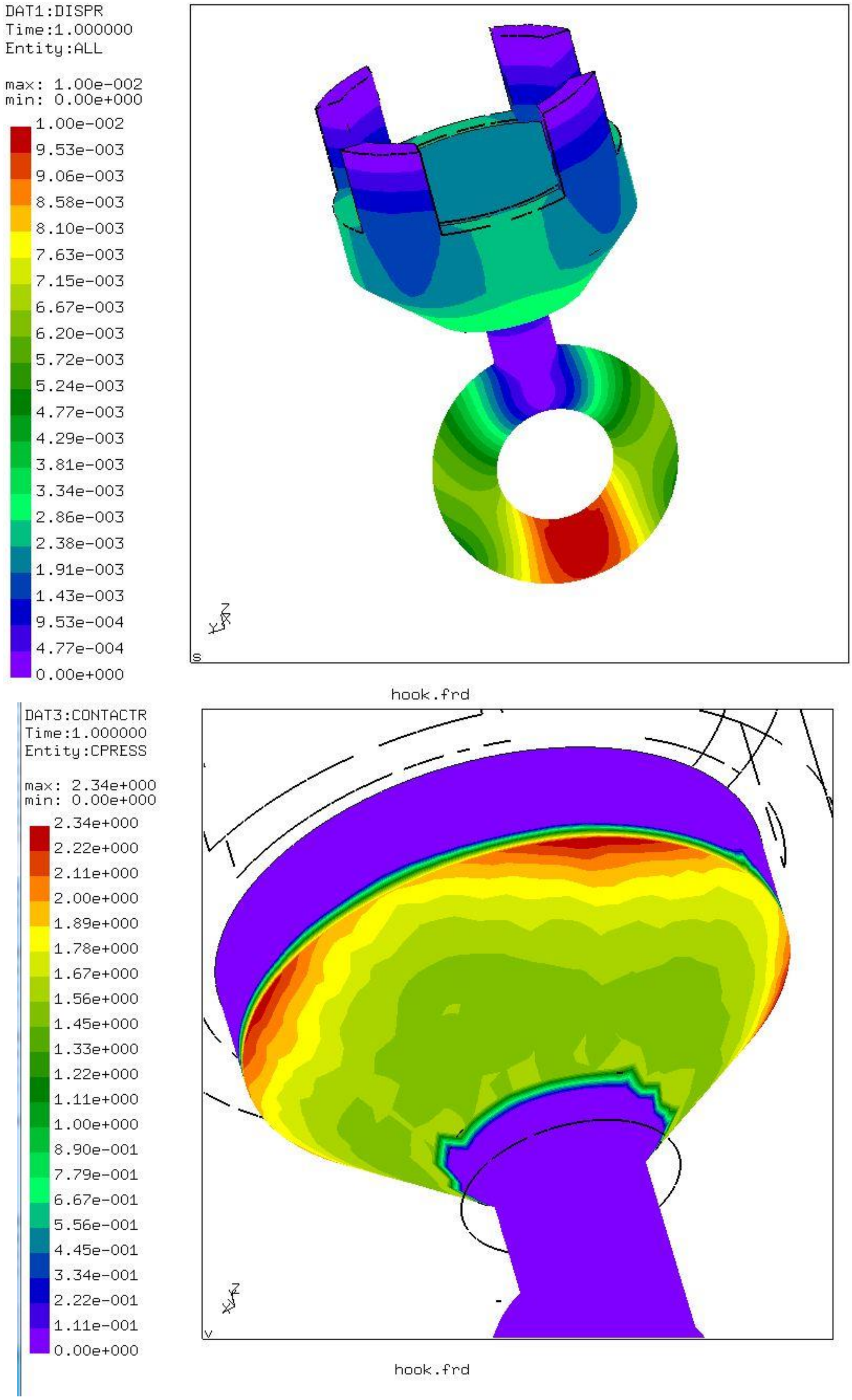


Figure 39 Total deformation and contact pressure plots from Calculix for Test Example 4.

7.4.2 Comparison of result data

The following table shows comparison between results obtained from different FEM codes for total displacement for test example 4.

Table 29 Results comparison of Test example 4.

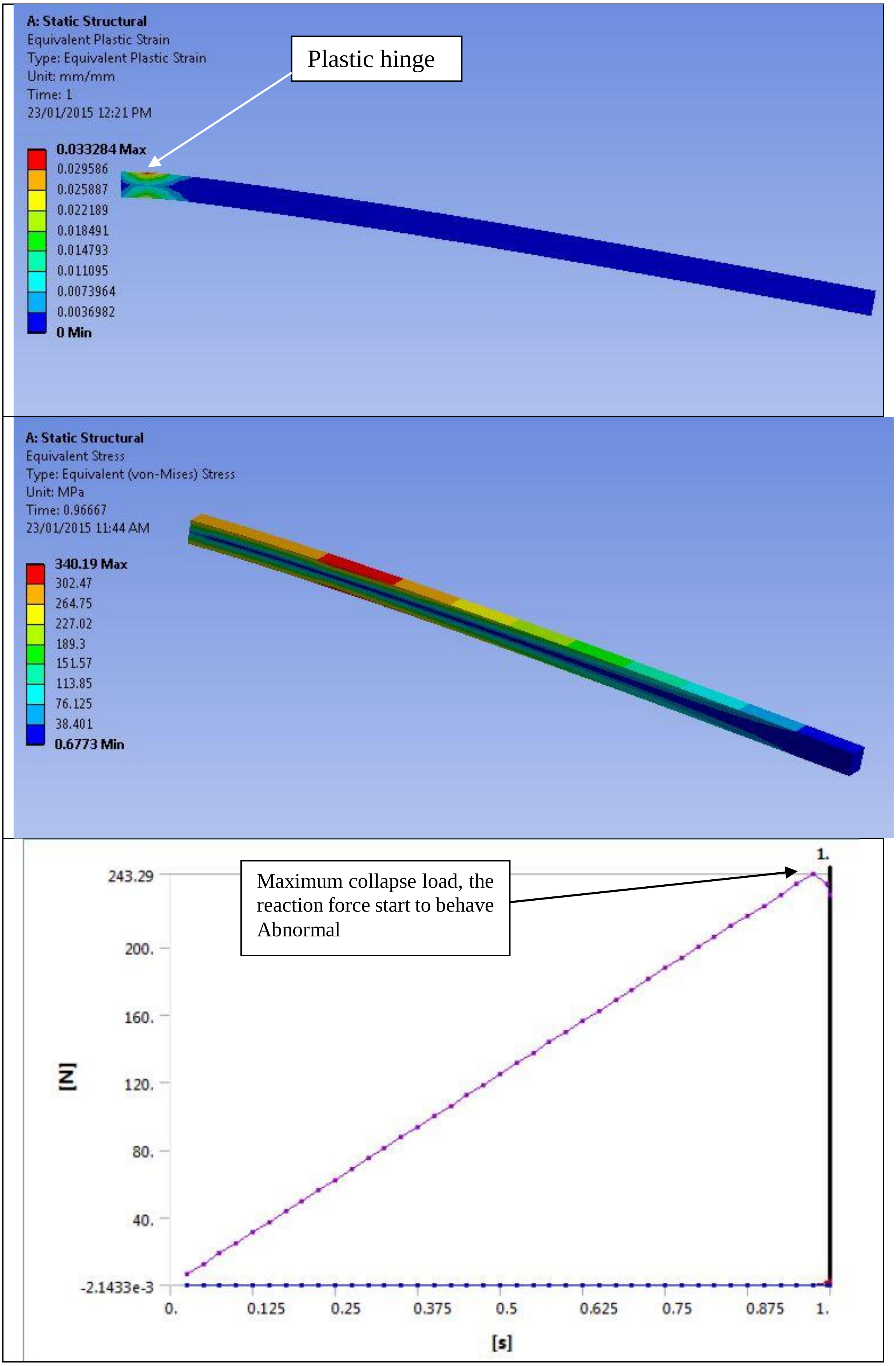
Results (quantities)	**FEM Codes**	**FEM Results**
Maximum Displacement (mm)	Ansys	0.01571
	Creo simulate	0.01551
	Calculix	0.01

Here, in this test example, the numerical results obtained from each FEM codes for quantities like stress and contact pressure are not similar to each other but the contour plot look similar to each other. The difference in the result is due to program default convergence criteria. The result can be obtained similar using same mesh size and same convergence criteria for each FEM codes. Also, the contact mechanism between each FEM codes is different here.

7.5 Test results of example 5

In this chapter, only equivalent plastic strain, total von mises stress and a graph of force reaction with time (load increment) are included. More results quantities can be found in appendix.

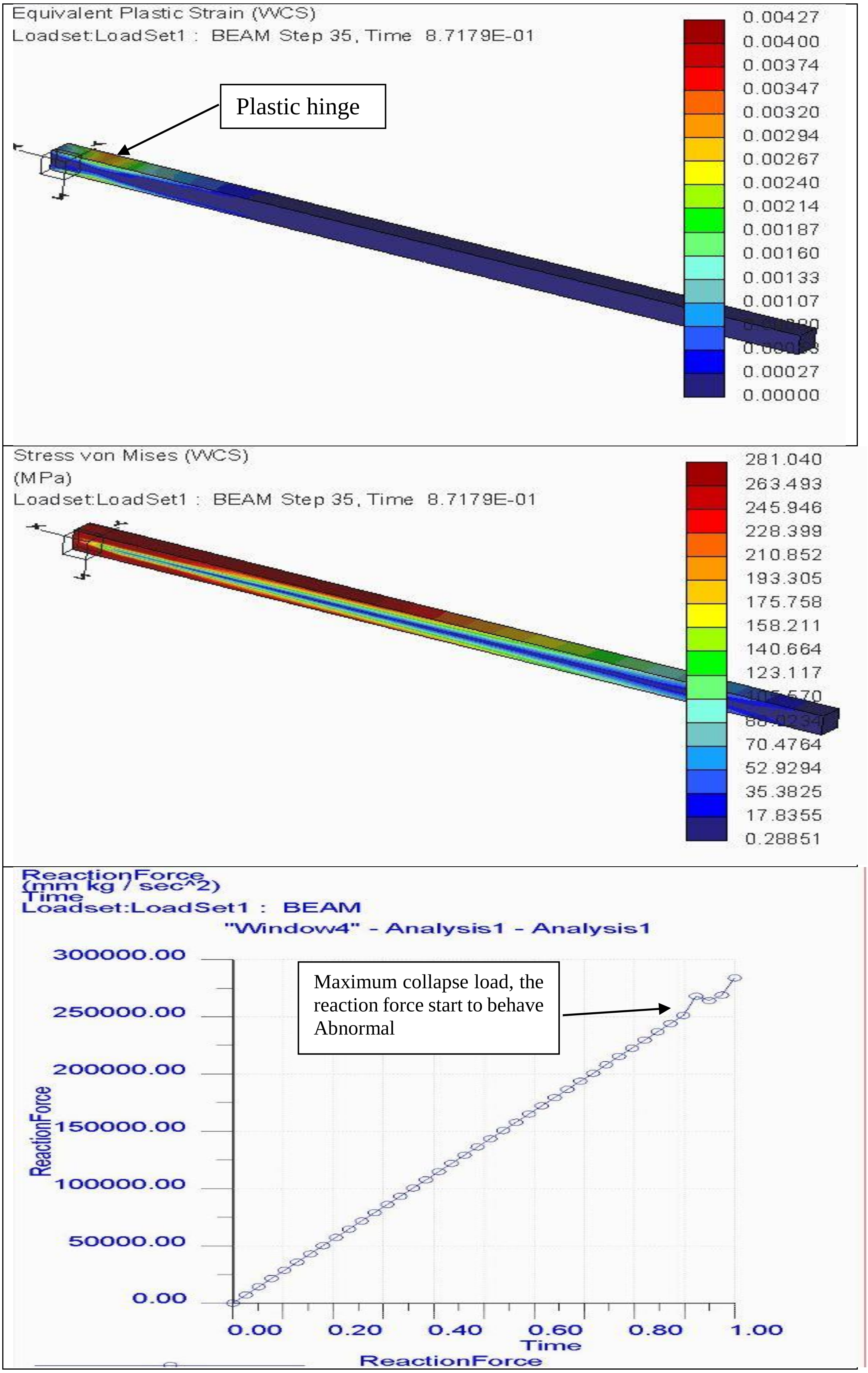
7.5.1 Test result data



Plastic hinge

Maximum collapse load, the reaction force start to behave Abnormal

Figure 40 Result plots from Ansys for plastic hinge, stress and Force reaction



Plastic hinge

Maximum collapse load, the reaction force start to behave Abnormal

Figure 41 Result plots from Creo for plastic hinge, stress and Force reaction

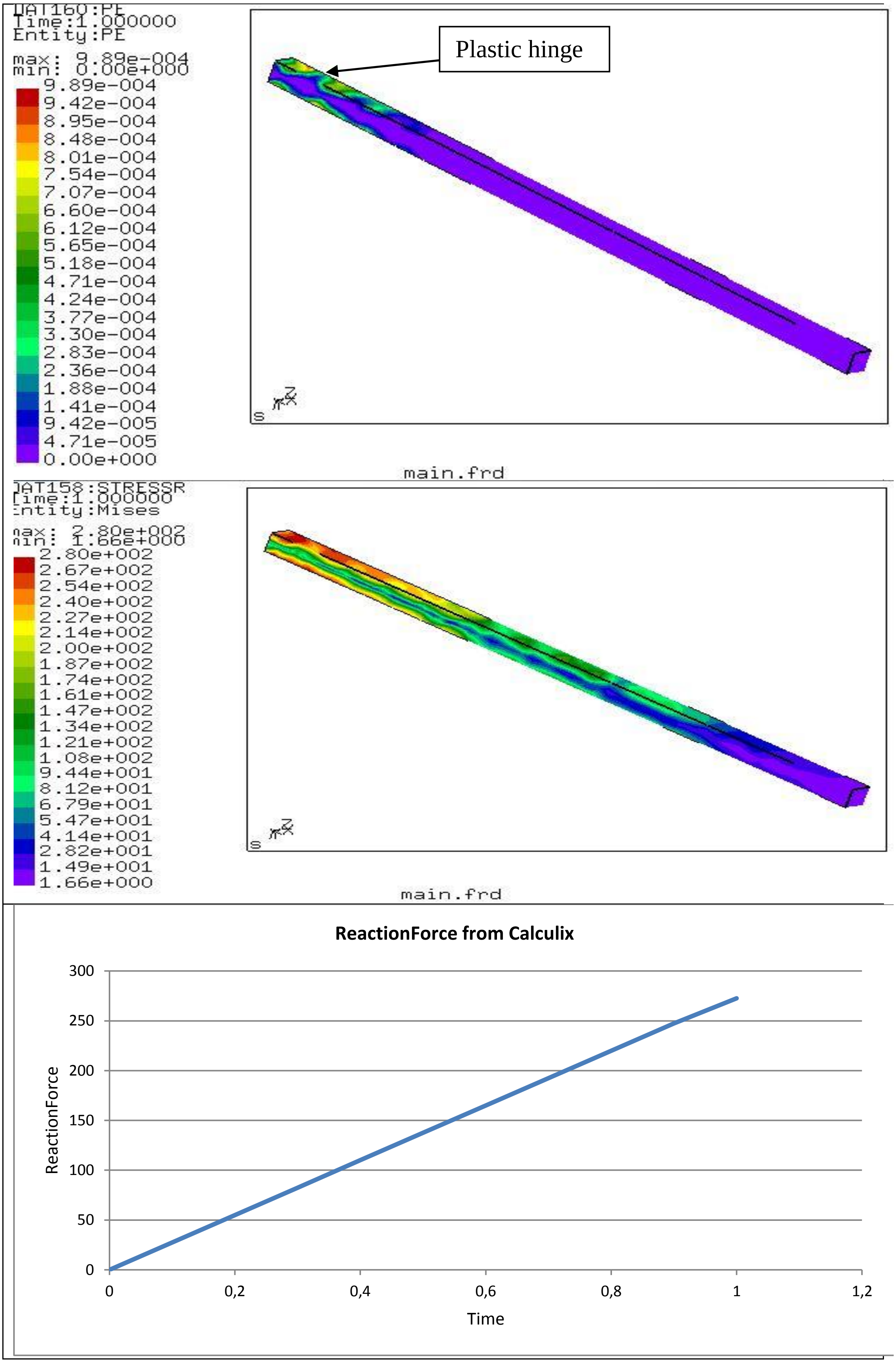


Figure 42 Result plots from Calculix for plastic hinge, stress and Force reaction

7.5.2 Comparison of Results Data

The following table shows the comparison between result obtained from theoretical solution and result obtained from FEM solution. The compared result is maximum collapse load. In each FEM codes, the solution does not converge while applying bigger loads. The FEM results are approximation solution which are obtained from reading graph by author. In the FEM solution procedure, few step size was selected and perfect mesh quality was not obtained during meshing. Definition of material property for plasticity was different due to program architecture and available material definition process. The result will be closer if experimental stress-strain curve are used for all FEM codes.

Table 30 Results comparison of test example 5

Results (quantities)	**Theoretical Results**	**FEM Codes**	**FEM Results (Approximation results from Graph)**	**Relative Error in Percentage (%)**
Maximum Collapse Load (N)	233.33	Ansys	243	**4.14 %**
		Creo simulate	251	**7.57 %**
		Calculix	272	**16.57 %**

The following table show comparison of result from commercial FEM code and result obtained from open source FEM codes. The idea of this comparison is to check how far the result from free or open source FEM codes deviate from commercial FEM codes. The average is taken from Ansys and Creo and the variation from average is calculated for free code from the average.

Table 31 Result comparison between commercial and open source FEM codes.

	Maximum Collapse Load (N)
Ansys	247
Creo Simulate	251
Average	249
	Variation from Average (%)
Calculix	**10.12%**

Hence, the maximum collapse load obtained from each FEM codes is closer to theoretical solution.

8 CONCLUSION

Static analysis includes the analyses of stress, strain and displacement under static loading. The analysis can be conducted as 1-D, 2-D or 3D. A simple structure such as a beam, frame, truss, etc can be analysed analytically but complicated 3-D structures cannot be analysed using an analytical solution technique which further requires numerical solutions. The static analysis can be linear or nonlinear and nonlinearity is characterized by geometry, material and contact or constraint. Different commercial or open source codes are implemented to solve these static problems. Computer codes are used to solve these problems (global equation) since the size of the matrix become very large in the case of a solid structure which is impossible to solve using hand calculations.

The general procedures of finite element methods include pre-processing, processing and post processing. Pre-processing includes defining the geometry, material and boundary conditions. Processing includes solving global equation and post processing includes displaying the graphical result from the solved raw data.

After a detailed study of the selected codes, it was discovered that all of the codes used h-element methods except Creo Simulate which used p-element methods. It was found that commercial codes were powerful compared to open source FEM codes. All of the studied codes could perform static linear and nonlinear analyses. Z88 Aurora could only perform nonlinear analysis caused by large deformations. Each code had their own material definition model. ANSYS had lots of nonlinear material models as compared to the other codes. It was also discovered that free code Calculix could perform simulation in more working fields compared to Creo Simulate (according table 10). It was also found that a simple one dimensional element could be modelled using Calculix and Z88 Aurora. Calculix pre-processor could model 3-D geometry as well but the modelling had to be done using its own input language. Calculix was found to be a unique but powerful tool. Tetra mesh cannot be generated by a Calculix pre-processor therefore an external mesher program such Gmsh or Netgen must be used for a good mesh. It was manifested that most of the free codes used an external mesher program. Z88 Aurora also used an external meshers (Netgen and Tetgen).

The input method for all the codes were similar accept for Calculix where the input had to be given in its own input programing language which was difficult to learn at the beginning. It used the Abaqus programing language. As compared to ANSYS and Creo Simulate, post processing was also weak with Z88Auora and Calculix since it was difficult to get desired graphical results. In the case of Z88 Aurora, it was observed that the element could not be hidden into the results which makes it difficult to read stress distribution contour plots.

After the results were obtained from each selected FEM code, it was discovered that all the selected codes gave similar results as compared to one other and also there were fewer relative errors found compared to theoretical solution in a linear static analysis. It was discovered that results from free codes for Test Example 1, Test Example 2 and Test Example 3 were similar to commercial codes. Based on this it was concluded that, these codes could perform simulation similarly as commercial FEM codes. In the case of a nonlinear analysis, only Calculix was compared to commercial codes. From the results of Test Example 4 which was contact analysis, the total deformation from each code was similar but the stress and contact pressure were different. Mesh and contact refinement could give similar solutions in this case. Similarly, for test example 5, the solution obtained from both commercial codes was different in quantities such as stress and strain. But the maximum collapse load was found similar. Free codes also gave similar results in this test example.

Finally, the finite element method is a powerful numerical solution technique used everywhere nowadays. There are lots of FEM codes which are free or commercial. Whether the codes are free or commercial, they must be used with sufficient knowledge in order to get good results. The overall conclusion from this thesis project was that each studied tool allowed a competent user to get to approximately similar results if used with a similar amount of care and knowledge. Also, using of open source FEM codes will save a lot of money and this can enhance the research and learning process for the development of new FEM codes since it provides all the source codes.

SOURCES

ANSYS Workbecnh User's Guide. (2009). 12th ed. [ebook] Canonsburg, PA: ANSYS, Inc. Available at: http://orange.engr.ucdavis.edu/Documentation12.1/121/wb2_help.pdf [Accessed 4 Dec. 2014].

ANSYS Workbench Verification Mannual. (2012). 14th ed. [ebook] Canonsburg, PA: ANSYS, Inc. Available at: http://orange.engr.ucdavis.edu/Documentation12.0/120/wb_vm.pdf [Accessed 4 May 2014].

Ansys.com, (2014). *ANSYS Workbench Platform*. [online] Available at: http://ansys.com/Products/Workflow+Technology/ANSYS+Workbench+Platform [Accessed 4 Dec. 2014].

Ashby, M. (2005). *Materials selection in mechanical design*. 3rd ed. Amsterdam: Butterworth-Heinemann, pp.471-505.

Bathe, K. (1996). *Finite element procedures*. Englewood Cliffs, N.J.: Prentice Hall, pp.695-765.

Bnmc.caltech.edu, (2014). *Finite Element Modeling Software - Caltech Biological Network Modeling Center*. [online] Available at: https://bnmc.caltech.edu/resources/finite_element [Accessed 8 Dec. 2014].

Budynas, R., Nisbett, J. and Shigley, J. (2008). *Shigley's mechanical engineering design*. 8th ed. Singapore: McGraw-Hill, pp.1002-1007.

Chandrupatla, T. and Belegundu, A. (2002). *Introduction to finite elements in engineering*. 3rd ed. Upper Saddle River, N.J.: Prentice Hall.

Chen, X. and Liu, Y. (2011). *Finite element modeling and simulation with ANSYS Workbench*. pp.56-80.

Cook, R. (1995). *Finite element modeling for stress analysis*. New York: John Wiley.

Deflection and Slopes of Beam. (2014). 1st ed. [ebook] Available at: http://www.iit.upcomillas.es/joctavio/Docs/Deflections&SlopesBeams.pdf [Accessed 1 Nov. 2014].

Den Hartog, J. (1987). *Advanced strength of materials*. New York: Dover Publications.

Dhondt, G. (2014). *CalculiX CrunchiX USER'S MANNUAL*. 2nd ed. [ebook] Available at: http://www.dhondt.de/ccx_2.7.pdf [Accessed 6 Aug. 2014].

Dhondt.de, (2014). *CalculiX:Overview of the finite element capabilities of CalculiX Version 2.7*. [online] Available at: http://www.dhondt.de/ov_calcu.htm [Accessed 1 Dec. 2014].

Element Reference. (2009). 12th ed. [ebook] Cannonsburg, PA: ANSYS, Inc. Available at: http://orange.engr.ucdavis.edu/Documentation12.1/121/ans_elem.pdf [Accessed 4 Aug. 2014].

EN175: Advanced Mechanics of Solid. (2014). 1st ed. [ebook] Brown Univeristy. Available at: http://www.simulia.com/academics/tutorial_pdfs/Tutorials/Circular%20Hole%20in%20Plate.pdf [Accessed 23 Dec. 2014].

Enes, S. (2009). *Shape functions generation, requirement, etc*. 1st ed. [ebook] Ruhr Universität Bochum. Available at: http://www.sd.ruhr-uni-bochum.de/downloads/Shape_funct.pdf [Accessed 28 Sep. 2014].

Galeano, C., Mantilla, J., Dugue, C. and Mejla, M. (2007). General Public License software tools to Finite Element Modeling. *Dyna*, Volume 74(Issuue 153), pp.313-324.

Help.ptc.com, (2014). [online] Available at: http://help.ptc.com/creo_hc/creo30_sim_hc/usascii/index.html#page/sim/simulate/overview_mechanica.html [Accessed 4 Dec. 2014].

Help.ptc.com, (2014). *Element type and Geometry entity*. [online] Available at: http://help.ptc.com/creo_hc/creo30_sim_hc/usascii/index.html#page/sim/simulate/modstr/mech_mesh/reference/el_types.html [Accessed 4 Dec. 2014].

Hokkanen, J. (2014). *Introduction of a segment-to-segment penalty contact formulation*. Master Thesis. Aalto University.

Lee, H. (2014). *Finite element simulations with ANSYS workbench 15*. Mission, KS: SDC Publications.

Madenci, E. and Guven, I. (2006). *The finite element method and applications in engineering using ANSYS*. New York: Springer, p.19.

Mittal, P. and Singh, J. (2013). Use of open source software in Engineering. *International Journal of Advanced Research in Computer Engineering and Technology*, Volume 2(Issue 3).

Mott, R. (n.d.). *Machine elements in mechanical design*. 5th ed.

Norton, R. (2006). *Machine design*. 3rd ed. Upper Saddle River, N.J.: Pearson Prentice Hall, pp.905-962.

PTC Creo Simulate. (2014). 1st ed. [ebook] PTC, Inc. Available at: http://www.ptc.com/File%20Library/Product%20Families/Creo/Validate/PTC_Creo_Simulate_DS.pdf [Accessed 4 Dec. 2014].

Rao, S. (2005). *The finite element method in engineering*. 4th ed. Amsterdam: Elsevier/Butterworth Heinemann, pp.282-305.

Rieg, F. (2014). *Z88 Aurora User Manual*. 2nd ed. [ebook] Z88 Aurora. Available at: http://www.z88.de/z88aurora/download/userguide.pdf [Accessed 1 Dec. 2014].

Sebestian, R. (n.d.). *Avanced Calculix Tutorial*. 1st ed. [ebook] Libremechics.com. Available at: http://www.libremechanics.com/ [Accessed 19 Jan. 2015].

Stress Concentration. (2014). 1st ed. [ebook] Available at: http://www.ewp.rpi.edu/hartford/~ernesto/Su2012/EP/MaterialsforStudents/Aiello/Roark-Ch06.pdf [Accessed 14 Dec. 2014].

Toogood, R. (2012). *Creo Simulate tutorial releases 1.0 & 2.0*. Mission, KS: Schroff Development Corp.

Valclav, S. (2008). About open-source solvers based on FEM. Bratislava: APLMAT 2008.

Web.mit.edu, (2015). *Contents*. [online] Available at: http://web.mit.edu/calculix_v2.7/CalculiX/ccx_2.7/doc/ccx/node1.html [Accessed 23 Nov. 2014].

Www-h.eng.cam.ac.uk, (2015). *CUED - ABAQUS*. [online] Available at: http://www-h.eng.cam.ac.uk/help/programs/fe/abaqus/faq68/abaqusf5.html [Accessed 18 Dec. 2014].

Z88.de, (2014). *Z88 - free Finite Element Analysis for Windows, OS X and Unix*. [online] Available at: http://www.z88.de/ [Accessed 6 Dec. 2014].

Z88.de, (2014). *Z88 AURORA*. [online] Available at: http://www.z88.de/z88aurora/wasistz88_e.htm [Accessed 6 Nov. 2014].

Appendix 1 Lists of FEM codes

Free or open source FEM codes lists

FEM codes
1. ADVENTURE
2. Aladdin
3. ALBERTA
4. Calculix
5. CMISS
6. Code_Aster
7. Deal.II
8. DOUG
9. Elmer
10. FEA(S)T
11. FENICS
12. FELIB
13. FEIt
14. FELYX
15. FEM_Object
16. FEMOCTAVE
17. FEMSET
18. FFEP
19. freeFEM
20. GetFEM++
21. Gmsh
22. HMD
23. Impact
24. IMS
25. Kaskade
26. KFEM
27. LUGR
28. MiniFEM
29. MODFE
30. MODULEF
31. NLFET
32. Netgen
33. OLEFI
34. OOFEM
35. Open FEM-miniFEM2D1
36. Z88/Z88 Aurora

Commercial FEM codes lists

FEM codes
1. ADINA
2. AGLOR
3. AxisVM
4. ANSYS
5. Cast3M
6. Cenaero
7. Creo Simulate
8. Compass
9. COMSOL
10. COSMOSWorks
11. ESI
12. Europlexus
13. FEAT
14. FEMAP
15. FesaWin
16. Go-Mesh
17. JL-Analyser
18. LISA
19. LS-Dyna
20. MARC
21. NEI
22. NISA
23. PERMAS
24. Range
25. SIMULA
26. Strand7
27. VisualFEA

THEORETICAL SOLUTION FOR TEST EXAMPLE 1

For maximum displacement

$$\varepsilon_{xx} = \frac{\sigma_{xx}}{E} \qquad \varepsilon_{xx} = \frac{\Delta L}{L}$$

$$\frac{\Delta L}{L} = \frac{\sigma_{xx}}{E} \qquad \Delta L = \frac{\sigma_{xx} \cdot L}{E} \qquad \sigma_{xx} = \sigma_0$$

$$\sigma_0 := 100MPa \qquad L := 200mm \qquad E := 210GPa$$

$$\Delta L := \frac{\sigma_0 \cdot L}{E}$$

$$\Delta L = 0.09524 \cdot mm$$

For maximum stress

Stress concentration

$$d := 10mm \qquad D := 100mm$$

$$K_t := 3 - 3.140\left(\frac{d}{D}\right) + 3.667\left(\frac{d}{D}\right)^2 - 1.527\left(\frac{d}{D}\right)^3$$

$$K_t = 2.721$$

maximum stress

$$\sigma_{nom} = \sigma_0 \cdot \frac{b}{(b - d)}$$

$$\sigma_{max} = K_t \cdot \sigma_{nom}$$

$$b := 100mm \qquad t := 1mm$$

$$\sigma_{nom} := \sigma_0 \cdot \frac{b}{(b - d)} \qquad \sigma_{nom} = 111.111 \cdot MPa$$

$$\sigma_{max} := K_t \cdot \sigma_{nom}$$

$$\sigma_{max} = 302.349 \cdot MPa$$

THEORETICAL SOLUTION FOR TEST EXAMPLE 2

For maximum displacement

$$\delta_{max} = \frac{F \cdot L^3}{3 \cdot E \cdot I}$$

$F := 100N$ $\quad L := 100mn$ $\quad b := 10mn$ $\quad h := 10mn$ $\quad E := 210GPa$

$I := \frac{b \cdot h^3}{12}$ $\qquad \delta_{max} := \frac{F \cdot L^3}{3 \cdot E \cdot I}$

$\delta_{max} = 0.19048mn$

For maximum stress

Stress concentration under bending

$h := 2mn$ $\qquad D := 10mn$

$$K_t := 3.065 - 6.637\left(\frac{2 \cdot h}{D}\right) + 8.229\left(\frac{2 \cdot h}{D}\right)^2 - 3.636\left(\frac{2 \cdot h}{D}\right)^3$$

$K_t = 1.494$

Maximum stress

The bending is taken near the notched at x=47mm

$M := F \cdot x$ $\qquad x := 47mn$

$$\sigma_{nom} = \frac{6 \cdot M}{d \cdot t^2}$$

$\sigma_{max} = K_t \cdot \sigma_{nom}$

$M = 4.7 \times 10^3 \cdot N \cdot mn$ $\qquad d := 10mn$ $\qquad t := 6mn$

$\sigma_{nom} := \frac{6 \cdot M}{d \cdot t^2}$ $\qquad \sigma_{max} := K_t \cdot \sigma_{nom}$

$\sigma_{max} = 117.041MPa$

THEORETICAL SOLUTION FOR TEST EXAMPLE 3

For maximum displacement

$$\delta_{max} = \frac{q\cdot L^{4}}{8\cdot E\cdot I}$$

$q := 5\frac{kN}{m}$ $L := 1000mn$ $I := 5.6\cdot 10^{5}\,mm^{4}$ $E := 210GPa$

$$\delta_{max} := \frac{q\cdot L^{4}}{8\cdot E\cdot I}$$

$$\delta_{max} = 5.31463mn$$

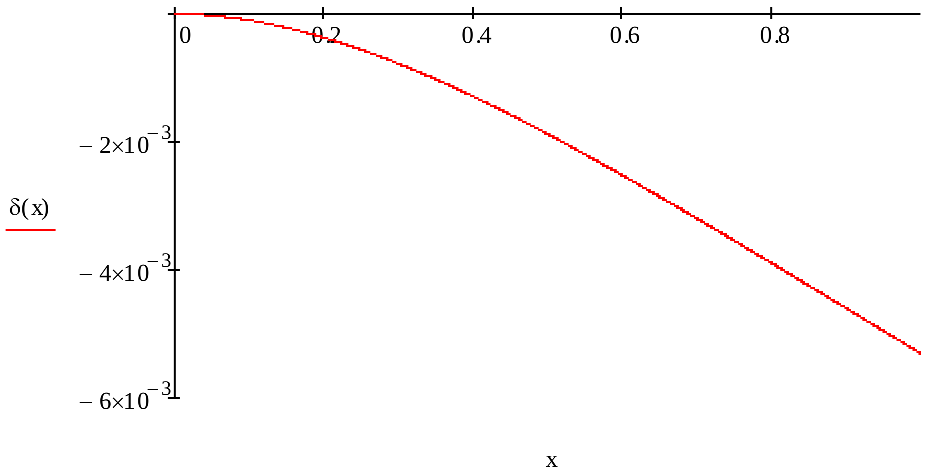
$$\delta(x) := \frac{-q\cdot x^{2}}{24\cdot E\cdot I}\cdot\left[2\cdot L^{2} + (2\cdot L - x)^{2}\right]$$

For maximum stress

$$M_{max} := \frac{q\cdot L^{2}}{2}$$ $c := 30mn$

$$\sigma_{max} := \frac{M_{max}\cdot c}{I}$$

$$\sigma_{max} = 133.929MPa$$



THEORETICAL SOLUTION FOR TEST EXAMPLE 5

Given parameters

$\sigma_y := 280\,\mathrm{MPa}$ $b := 10\,\mathrm{mm}$ $d := 10\,\mathrm{mm}$ $L := 300\,\mathrm{mm}$

Plastic section modulus

$$Z_P := \frac{b \cdot d^2}{4}$$

$$Z_P = 250 \cdot \mathrm{mm}^3$$

Elastic section modulus

$$Z_E := \frac{b \cdot d^2}{6}$$

$$Z_E = 166.667\,\mathrm{mm}^3$$

$$f := \frac{Z_P}{Z_E} \rightarrow \frac{3}{2} = 1.5$$

Plastic Moment

$$M_P := Z_P \cdot \sigma_y$$

$$M_P = 7 \times 10^4 \cdot \mathrm{N} \cdot \mathrm{mm}$$

Maximum collapse load (M=FL)

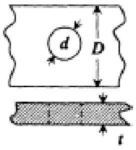
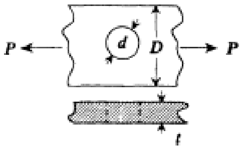
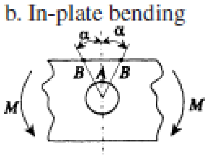
$$F := \frac{M_P}{L}$$

$$F = 233.333\,\mathrm{N}$$

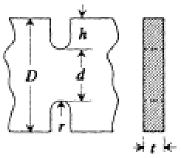
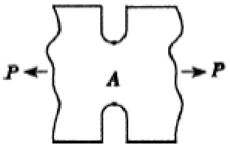
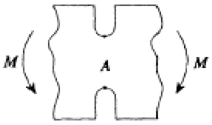
Appendix 3 Stress concentration table and equations

Stress concentration table and equation for test example 1 and 2

TABLE 6-1 (continued) STRESS CONCENTRATION FACTORS: Holes

2. Central single circular hole in finite-width plate 	a. Axial tension 	$\sigma_{\max} = \sigma_A = K_t \sigma_{\text{nom}}, \qquad \sigma_{\text{nom}} = P/[t(D-d)]$ $K_t = 3.000 - 3.140(d/D) + 3.667(d/D)^2 - 1.527(d/D)^3$ for $0 \le d/D \le 1$
	b. In-plate bending 	(1) At edge of hole, $\sigma_{\max} = \sigma_A = K_t \sigma_{\text{nom}}, \qquad \sigma_{\text{nom}} = 6Md/(D^3 - d^3)t$ $K_t = 2$ (independent of d/D) (2) At edge of plate, $\sigma_{\max} = \sigma_B = K_t \sigma_{\text{nom}}, \qquad \sigma_{\text{nom}} = 6MD/(D^3 - d^3)t$ $K_t = 2d/D(\alpha = 30°)$

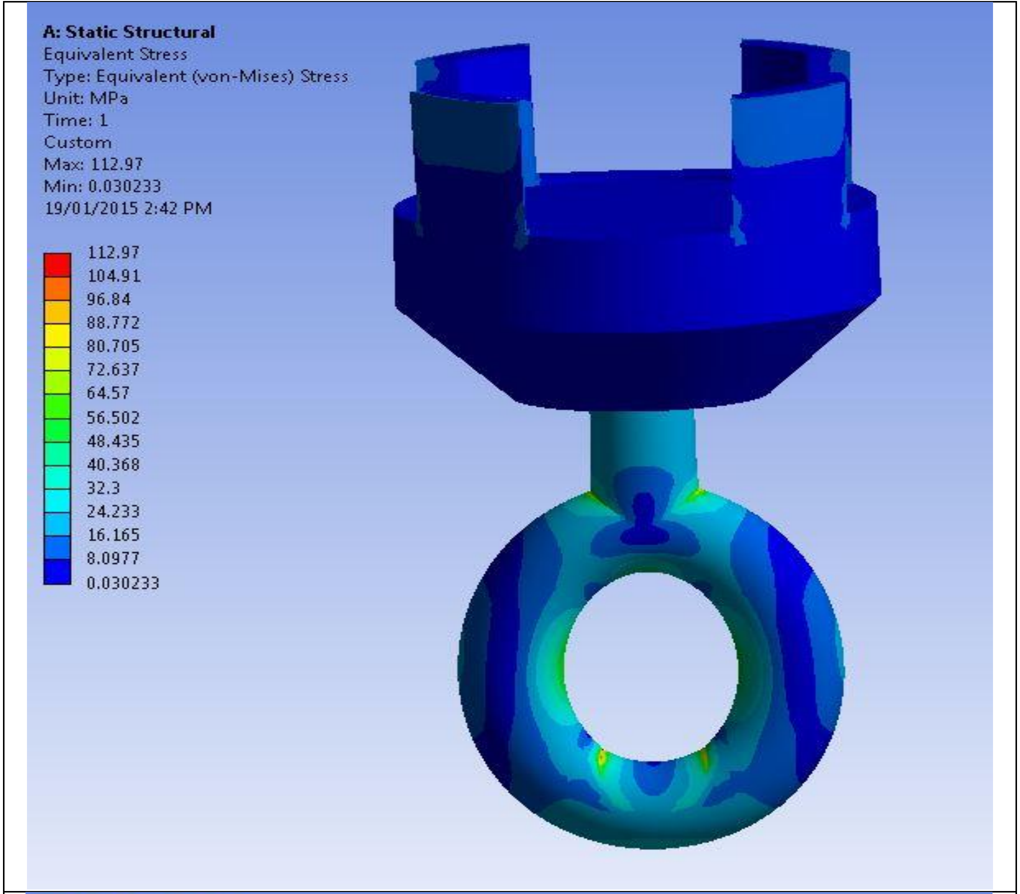
(Stress Concentration, 2014)

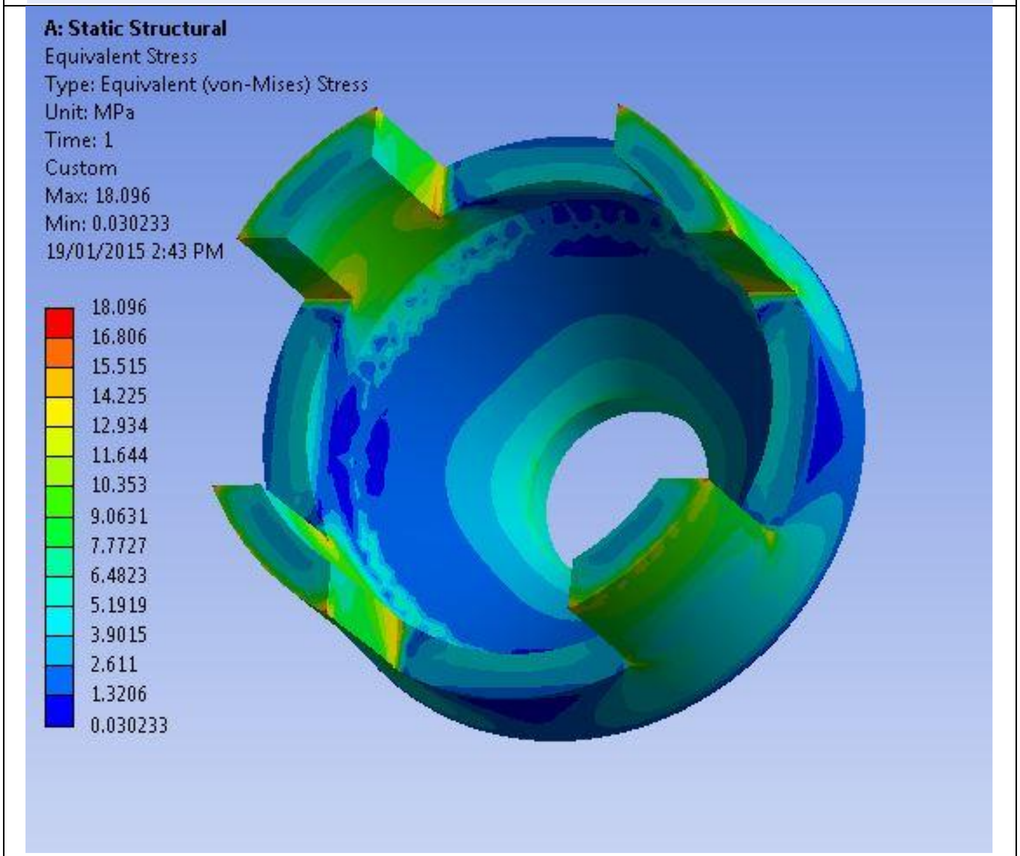
2. Opposite single U-shaped notches in finite-width plate 	a. Axial tension 	$\sigma_{\max} = \sigma_A = K_t \sigma_{\text{nom}}, \qquad \sigma_{\text{nom}} = P/td$ $K_t = C_1 + C_2 \frac{2h}{D} + C_3 \left(\frac{2h}{D}\right)^2 + C_4 \left(\frac{2h}{D}\right)^3$ $\begin{array}{c\|c\|c} & 0.1 \le h/r < 2.0 & 2.0 \le h/r \le 50.0 \\ \hline C_1 & 0.955 + 2.169\sqrt{h/r} - 0.081h/r & 1.037 + 1.991\sqrt{h/r} + 0.002h/r \\ C_2 & -1.557 - 4.046\sqrt{h/r} + 1.032h/r & -1.886 - 2.181\sqrt{h/r} - 0.048h/r \\ C_3 & 4.013 + 0.424\sqrt{h/r} - 0.748h/r & 0.649 + 1.086\sqrt{h/r} + 0.142h/r \\ C_4 & -2.461 + 1.538\sqrt{h/r} - 0.236h/r & 1.218 - 0.922\sqrt{h/r} - 0.086h/r \end{array}$ for semicircular notch ($h/r = 1.0$) $K_t = 3.065 - 3.472 \left(\frac{2h}{D}\right) + 1.009 \left(\frac{2h}{D}\right)^2 + 0.405 \left(\frac{2h}{D}\right)^3$
	b. In-plane bending 	$\sigma_{\max} = \sigma_A = K_t \sigma_{\text{nom}}, \qquad \sigma_{\text{nom}} = 6M/d^2t$ $K_t = C_1 + C_2 \frac{2h}{D} + C_3 \left(\frac{2h}{D}\right)^2 + C_4 \left(\frac{2h}{D}\right)^3$ $\begin{array}{c\|c\|c} & 0.1 \le h/r < 2.0 & 2.0 \le h/r \le 50.0 \\ \hline C_1 & 1.024 + 2.092\sqrt{h/r} - 0.051h/r & 1.113 + 1.957\sqrt{h/r} \\ C_2 & -0.630 - 7.194\sqrt{h/r} + 1.288h/r & -2.579 - 4.017\sqrt{h/r} - 0.013h/r \\ C_3 & 2.117 + 8.574\sqrt{h/r} - 2.160h/r & 4.100 + 3.922\sqrt{h/r} + 0.083h/r \\ C_4 & -1.420 - 3.494\sqrt{h/r} + 0.932h/r & -1.528 - 1.893\sqrt{h/r} - 0.066h/r \end{array}$ for semicircular notch ($h/r = 1.0$) $K_t = 3.065 - 6.637 \left(\frac{2h}{D}\right) + 8.229 \left(\frac{2h}{D}\right)^2 - 3.636 \left(\frac{2h}{D}\right)^3$

(Stress Concentration, 2014)

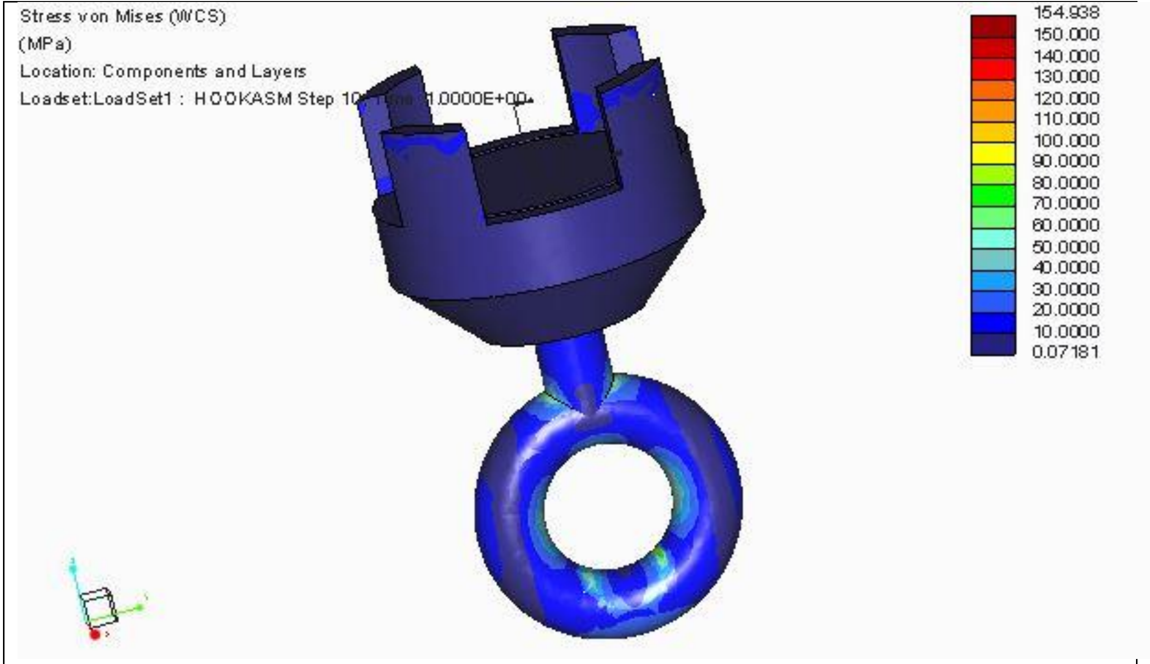
Appendix 4 Additional result plots

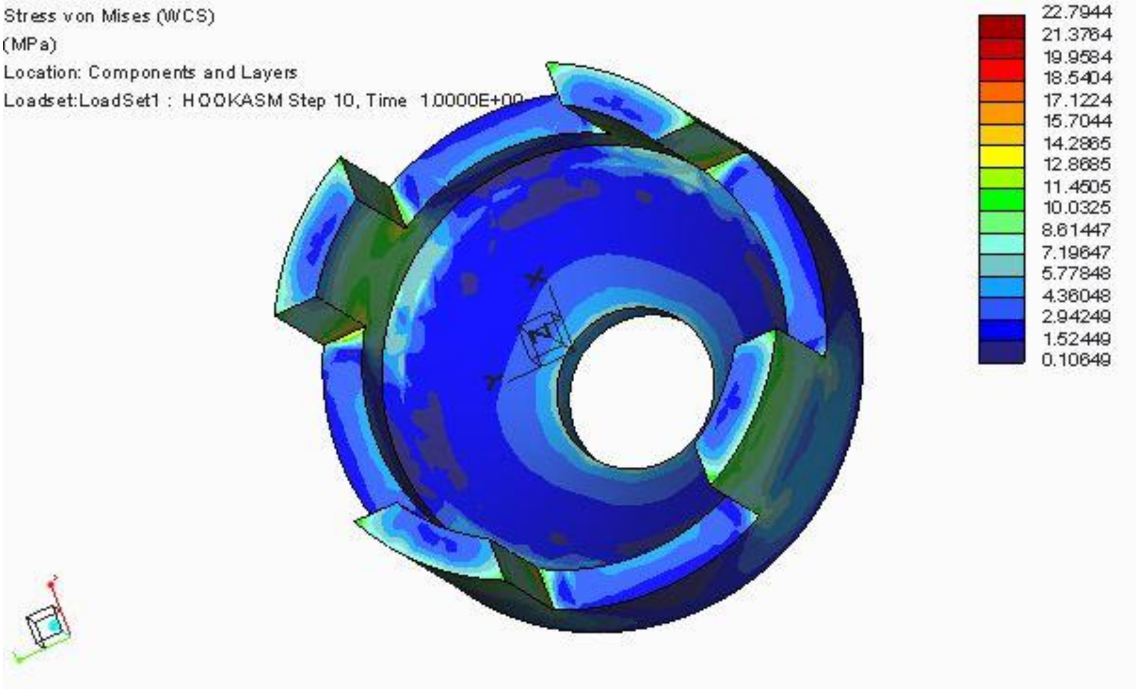
Result plot from ANSYS for stress in hook base and hook core





Result plot from Creo Simulate for stress in hook base and hook core





Result plot from Calculix for stress in hook base and hook core