How to increase the accuracy of analysis and reduce the computational time in ANSYS in the case of deformation study of orthopedic bone plates

Javad Malekani[a], Prasad KDV Yarlagadda[b,*], Beat Schmutz[c], YuanTong Gu[d], Michael Schuetz[e]

Queensland University of Technology (QUT), Brisbane, Queensland, Australia

[a]j.malekani@qut.edu.au, [b]y.prasad@qut.edu.au, [c]b.schmutz@qut.edu.au, [d]yuantong.gu@qut.edu.au, [e]m.schutz@qut.edu.au,

**Corresponding author*

Keywords: Finite Element Method (FEM), ANSYS, Computational time, Accuracy, Tibia, Bone plate, deformation

Abstract. Currently, finite element analyses are usually done by means of commercial software tools. Accuracy of analysis and computational time are two important factors in efficiency of these tools. This paper studies the effective parameters in computational time and accuracy of finite element analyses performed by ANSYS and provides the guidelines for the users of this software whenever they us this software for study on deformation of orthopedic bone plates or study on similar cases. It is not a fundamental scientific study and only shares the findings of the authors about structural analysis by means of ANSYS workbench. It gives an idea to the readers about improving the performance of the software and avoiding the traps. The solutions provided in this paper are not the only possible solutions of the problems and in similar cases there are other solutions which are not given in this paper. The parameters of solution method, material model, geometric model, mesh configuration, number of the analysis steps, program controlled parameters and computer settings are discussed through thoroughly in this paper.

1. Introduction

Nowadays numerical analysis techniques, especially finite element method, play a crucial role in engineering analyses e.g. stress, strain, safety and optimization. In practice, this kind of problems is mostly complicated and therefore performing each analysis needs thousands and maybe millions of iterations. Since it is not easy to perform them manually, computer codes have been developed. While computer codes were more common in past decades, currently available commercial software packages are preferred. Indeed, these packages are the advanced versions of the above mentioned codes. Software tools are useful in terms of simplicity of application, preparation time/energy/cost and accuracy of the results. However, they are like a black-box because the developers usually do not disclose scientific and technical basics in detail. Sometimes this issue can raise problems such as uncertainty of the results accuracy, repeatability of the analysis, error detection and correction. Software packages always provide a help system and explain how to work with it. But sometimes it is not comprehensive or helpful enough. For example, the Help of ANSYS software does not cover all details of this software. Authors of the current paper are using ANSYS software for years. While they have found it very useful tool in finite element analyses, they expect to confront an error or a problem in every second of working with it. In these cases they either have had to do a series of trial-and errors of different parameters to find out the solution, or

have found out the solution from the similar experiences of other users. For example it has been found that in 3D optimization by ANSYS the user has to delete the “Parameter Key” of ‘DS’ in ‘Geometry’ whenever imports the CAD model into ANSYS. Otherwise the optimization parameters cannot be defined. To date, it is not clear for the authors what ‘DS’ means; what other choices the user has; and what the technical or scientific reason of this choice is. Unfortunately the instruction of this software has not covered these kinds of problems and the Costumer Services of the product is as helpful as the Help of the product.
Publication through books and papers, internet forums, technical seminars and workshops are useful for sharing the knowledge. This paper discusses about the parameters affecting simulation of deforming a distal tibial orthopedic bone plate. These findings can be used in similar cases. Regarding the limitations, it does not cover the theoretical basics of the problem or mathematical principles of the finite element method. It only shares the results of numerous trials have been carried out to optimize the parameters of the simulations to achieve a desired accuracy and analysis time.

2. General Procedure of Finite Element Method (FEM)

Flowchart of Figure 1 is usually followed in analysis of a static solid mechanics problem by finite element method [1] and it is tried to solve the Equation 1 [2]. Details of each step and related equations are explained in the given references and in the similar resources.

Governing PDEs (Strong form of the problem)
↓
Establishment of the weak form
↓
Discritization of the problem domain
↓
Shape function construction (Creation of displacement function)
↓
Evaluation of the strain field
↓
Formation of the element stiffness matrices/vectors → Assembly of the global matrices/vectors
↓
Solution for the unknown nodal displacement
↓
Retrieval of strains and stresses

Fig 1 A macro flowchart of the basic FEM procedure [1]

$$\frac{\partial \sigma_{ij}}{\partial x_j} = 0 \qquad (1)$$

Where σ_{ij} denotes the true stress tensor and x_j the Cartesian coordinates. This equation can be solved by employing different calculation methods. Each calculation method ends in a specific group of equations and then forms a specific solution procedure. For example if it follows the equation (2) it forms the implicit solution technique. Similarly, the explicit technique is derived from the Equation (3) [2].

$$R_i^t = R_{i-1}^t + \left[\frac{\partial R}{\partial u}\right]_{i-1}^t \Delta u_i^t = 0$$
$$u_i^t = u_{i-1}^t + \Delta u_i^t \tag{2}$$
$$\dot{u}^{t+\frac{1}{2}} = M^{-1}(F^t - F_{int}^t)\Delta t^{t+\frac{1}{2}} + \dot{u}^{t-\frac{1}{2}}$$
$$u^{t+1} = u^t + \dot{u}^{t+\frac{1}{2}}\Delta t^{t+1} \tag{3}$$

Where *i* is iteration number, *R* is residual force vector, *t* is time, *F* is the generalized force vector and *u* is the displacement or velocity. According to the principles, finite element method provides an approximate solution by iteration of the solution [3]. With respect to Equation (2) and Equation (3), time and velocity/displacement increments (Δt and Δu) play an important role in the number of iterations and overall time of analysis. On the other hand increments affect on the accuracy of approximation. It means that the user should balance the analysis regarding the accuracy and computation time. In total, the solution can be optimized regarding the problem and solution factors. Both aspects are discussed in the following sections.

3. Problem: Deformation of an Orthopedic Bone Plate during Surgery

Nowadays, orthopedic bone plates are available in a variety of shapes and materials. They are usually precontoured to specific anatomic region. From clinical point of view, an anatomically well-fitted plate can greatly facilitate the process of closed reduction in terms of axial and rotational alignment of the main fragments [4]. Furthermore, such a plate may additionally protrude less with a nominal soft-tissue envelope, and therefore minimize soft-tissue impingement/irritations [5]. Studies on fit assessment of distal tibial plates show that only 19% initially fit to the underlying bone [5] as bone morphology is very patient specific. While, from mechanical point of view, a perfect fit between plate and the underlying bone is not necessary, it is still vital to attain the closest possible fit to ensure optimal load transfer [6]. Thus, some precontoured plates need to be deformed during surgery to ensure the appropriate fit. Deformation includes bending and twisting the plate at specific points with specific magnitudes [7] (Figure 1). It can be one or more depending on the fitting conditions. Stainless Steel AISI 316L (ASTM F138 [8] and F139 [9]) and Ti-6Al-4V ELI alloy (ASTM F136 [10]) are the most recommended biomaterials for bone plates [11, 12].

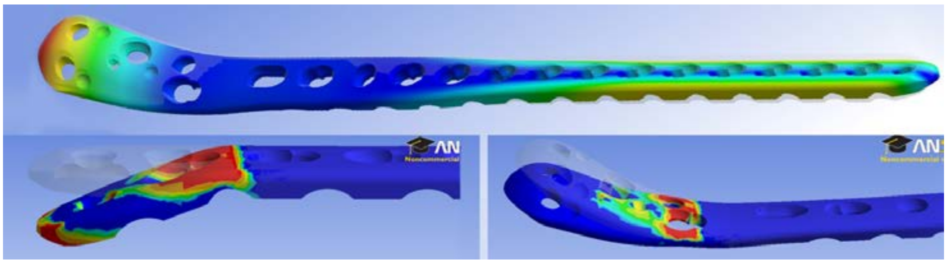

Fig 2 Three different deformations of distal tibial bone plate as a sample

In practice, deformations are done by pliers, irons and other mechanical apparatus in room temperature in the operation room. So the procedure is considered as cold metal forming.

4. Effective Factors in Solution of the Problem by Ansys

Important factors of solution method, material model, geometric model, mesh configuration, number of the analysis steps, program controlled parameters and computer settings affect on the results and therefore are discussed in this paper. Specifications of the workstation in which the models have been run are given for clarification.

4.1. Specifications of the Workstation

All deformations have been simulated by ANSYS workbench 13.0. The CAD model of the plate has been imported into ANSYS from Solidworks 2011 in SLDPRT format. Simulations have been run in a workstation with the following specifications:

- Processor: Quad core Intel Xeon CPU W3530 @ 2.80GHz
- Memory (RAM): 8.00 GB DDR3
- Operating System: Windows 7 64-bit

4.2. Solution Method

Since metal forming process happens in a low rate and the inertia contributions are negligible [13], it is considered quasi-static in finite element analysis. According to the classification of the deformation procedure of fracture fixation plates in metal forming, it is considered semi-static. Although some researchers believe that the results of implicit and explicit solutions are in good agreement in quasi-static analysis [14, 15], others believe that implicit codes can provide a more suitable procedure for effective analysis of metal forming problems [13].
Fundamentally, any of these methods has some advantages and disadvantages. For example in case of quasi-static implicit finite element formulations, main drawbacks are summarized as follows [2]:

- Requires the solution of linear systems of equations in each iteration;
- Requires high computation time and high memory;
- The computation time depends quadratically on the number of degrees of freedom if a direct solver is utilized;
- The stiffness matrix is often ill conditioned which can turn the solution procedure Unstable and deteriorate the performance of iterative solvers; and
- Experiences difficulties in dealing with complex non-linear contact and tribological boundary conditions that often lead to convergence problems

Unlike the implicit method, explicit analysis equations (Equation 3) can be solved independently and the equilibrium conditions are checked at each increment of time. Therefore overall solution can be performed independently and very fast for each degree of freedom. Also, computer programs do not present convergence problems [2]. However, the dynamic explicit formulations have the drawbacks as follows [2]:

- Requires the utilization of very small increments of time per step
- The equilibrium after each increment of time is not checked
- The assignment of the system damping is rather arbitrary
- Needs experienced users for adequately designing the mesh and choosing the scaling parameters for mass, velocity and damping. Otherwise it may lead to inaccurate solutions for the deformation, prediction of forming defects and distribution of the major field variables within the workpiece.
- Springback calculations are very time consuming and may lead to errors. This specific problem is frequently overtaken by combining dynamic explicit with quasi-static implicit analysis.

Analysis of springback effect is highly important in current project and therefore implicit method of analysis is more suitable. In other cases the user can choose each method based on the advantages, disadvantages and main application field of each method.

4.3. Material Model

There are several material models can be used for definition of material properties in ANSYS e.g. Bilinear Kinematic Hardening (rate-independent plasticity, BKIN), Multilinear Kinematic Hardening (rate-independent plasticity, MKIN), Multilinear Kinematic Hardening (rate-independent plasticity, KINH), Multilinear Isotropic Hardening (rate-independent plasticity, MISO), Bilinear Isotropic Hardening (rate-independent plasticity, BISO), Nonlinear Isotropic Hardening (rate-independent plasticity, NLISO), Anisotropic (rate-independent plasticity, ANISO), Multilinear Elastic (MELAS), Cast Iron Plasticity (CAST), Porous media (PM), User-defined materials (USER) or so on [16, 17].

Regarding the requirements in terms of application, biomechanical properties and the biological environment factors, orthopedic bone plates are made from highly isotropic biometals such as Stainless steel 316L and Ti-6Al-4V [12, 18]. Hence, the material models of Multilinear Isotropic Hardening (MISO), Bilinear Isotropic Hardening (BISO) have been tried in current study. It reveals that:

- Simulations are usually done in less than a minute (48-52 seconds) with BISO material model. However, the results are over %25 different from the experimental results and also former studies [19-24]
- MISO material model increases the computational time up to 15 minutes. But inaccuracy of the results can be under %2 which is excellent for most of numerical simulations.
- MISO material model has been tested with 4, 6 and 10 interval points in plastic zone.
- It has been found that the computational time changes slightly (less than %15) by increasing the intervals from four to ten but the accuracy of the analyses improves notably (over %10).

4.4. Geometric Model

The problems which are solved by finite element method can be approximated by three-dimensional (3D), two-dimensional (2D) or one-dimensional models. 2D models are simpler than 3D ones and then it is easier to make the model and solve the problem. Also less computational time is needed for 2D simulations. Previous studies reveal that the results of 2D models in stress analysis are higher than 3D models [25] and 2D simplification of 3D problems can introduce inaccuracy in results if it is not applied appropriately. In other words, while 2D modelling of some 3D problems, e.g. symmetric, axisymmetric and plane problems, can be satisfactorily approximated [16], in some other cases, e.g. asymmetric problems, it is necessary to model the problem three-dimensionally.

Geometric configuration of distal tibial bone plate is totally irregular and asymmetric. It includes many irregular curves. The holes on the plate do not any specific pattern. Width and thickness of the plate varies in different sections. Besides to geometric irregularity, boundary conditions of this problem are asymmetric. So, the problem necessarily has been simulated by 3D model and the efficiency of 2D and 3D FEM has not been comparatively examined in this problem. In other cases, the user should make decision accordingly.

4.5. Mesh Configuration

In performing analysis by finite element method the geometric model is divided into thousands of small regions called Element. Each element is bounded by some Nodes. Total number of nodes, degrees of freedom of each node and the formulation of each node define the properties of any element. So far, numerous types of elements have been developed to perform the analyses. Right choice of element type affect directly in the accuracy of results [26, 27]. Then, it is highly important to select the correct type of element. In former versions of ANSYS, the elements were chosen from the graphical menu or by the command of "ET". But in ANSYS workbench it is chosen automatically. Regarding to the experience of working with ANSYS Workbench in the past few years, element type proposed by ANSYS workbench works properly in almost all cases. However, it can be changed by inputting the above mentioned command in necessity.

Besides to the element type, the size of each element plays an important role in accuracy of the analysis and the computational time. Bigger element reduces the computational time. However it reduces the accuracy of analysis as well. It even can cause convergence problems if it is not defined appropriately [1, 2]. In iterative solution of a system of equations, convergence error is defined as the difference between the current iterate and the exact solution of the discretized equations [28]. It can be estimated accurately from data generated during an iteration process and is the best stopping criterion for an iterative process [28]. In ANSYS and other FEM software tools whenever the convergence error is too big the software fails the solution and shows error message. It usually happens in sharp edges and inflections. A sudden change in stiffness or sudden change in load can cause convergence problem as well [29]. Tolerances of acceptable convergence errors, time increments, solution sub-steps, number of iteration and mesh configuration are the most important parameters effect in such failures. The quality of mesh configuration can be checked by the feature of Mesh Metric in the recent versions of ANSYS and the other parameters can be controlled from the menu of Analysis Setting.

The Models of this paper have been meshed with the element of SOLID186. SOLID186 is a high order 3D 20-node solid element with three degrees of freedom on each node. Initially, the solution of the model was tried with 140K elements; but it was failed. As a solution, the meshing was refined and then the model was successfully run with 333K nodes and 227K elements. Since this solution has worked, all models have been simulated with these settings. The user can try either of above mentioned parameters whenever he/she faces with such errors.

4.6. Number of Analysis Steps

In some cases, e.g. in springback analysis, the finite element analysis should be inevitably performed in several steps because the analysis includes multiple consecutive steps of loading and unloading. Generally, it is advised to perform the finite element analysis in a single step whenever it can be done in either one step or multiple steps because performing a single step analysis in several steps decreases the performance, i.e. does not improve the accuracy but increases the computational time. It should be noted that there is a close concept in ANSYS named Sub-step. The setting of sub-step, which defines the number of increments during solution, is used when the user needs to modify or manually define the number of increments. Deformation of distal tibial bone plate was simulated through a one-step analysis and then a five-step analysis. Overall computational time was increased up to 35 minutes in a five-step analysis comparing with computational time of two minutes in one-step analysis. On the other hand, the results were exactly the same. It demonstrates that performing the analysis in multiple steps increases the computational time drastically but does not affect on precision and accuracy of the analysis. Thus, it is advised to perform such analyses in the least required steps.

4.7. Large Deformation

Some references note that large deformation happens whenever the strain is over 10% or the rotation is over 10° [30]; but some others refer to the strain of greater than 5% for introduction of large deformations [31]. However, all of them agree that the deformation is

small whenever strain-displacement relations are linear [31, 32] and it is large deformation if the mentioned relations become nonlinear [29, 31, 32].
Change in volume is assumed to be negligible in small deformations [31]. For structural mechanics problems under large deformations, the stiffness changes with deformation thus makes the problem non-linear [29]. Enabling or disabling the parameter of Large Deformation in ANSYS does not affect notably in the accuracy of the results whenever the problem involves with small deformations. However, the computational time increases drastically when this parameter is enabled. But the user has to enable it in the problems of large deformations, e.g. extrusion and deep-drawing. Otherwise the results will be totally wrong. In total the user should be very careful in studying the results and performing the analysis whenever the solution parameter of large deformation is enabled because the probability of facing with error is much higher.
In case of the problem of this paper, the authors have tried it and noticed that the computational time increases up to half an hour from almost two minutes. But the changes in results are not notable at all significant as it is less than <1%.

4.8. Program-Controlled Parameters

There are many parameters are defined automatically by the software. While advanced users can define these parameters manually it is not advised for the basic and intermediate users. Program-Controlled parameters are getting more in newer versions and it is easier to work with the recent versions of the software. However, automatically-defined parameters are a major source of errors and it is not easy to resolve the error whenever an error happens. In total it is advised to gain the necessary scientific and technical knowledge of working with the software in the field which this software is used to handle these types of errors better.

4.9. Operating System and Allocated Memory

ANSYS program requires a computing resource demand that spans every major component of hardware capability. Equation solvers that drive the simulation capability of Workbench and ANSYS analyses are computationally intensive, require large amounts of physical memory, and produce very large files which demand I/O capacity and speed [33]. The software developer, manufacturers of computer accessories and also researchers of this field have conducted numerous studies to find out how to obtain an optimal performance during running the ANSYS software [33-37]. As a result, they regularly advise the hardware requirements and the appropriate settings for each version of the software, Operating System and the workstation. For example the company of ANSYS has recently proven that the speed of core solver and the efficiency can be improved up to 8.83 times and 90% in order with a specific type of processor and applying suitable changes in settings of the software and workstation [38].
In total, while the users of this software need to apply the appropriate settings in order to achieve the optimal settings, it is recommended to do it very carefully as the software gives a lot of errors, e.g. memory errors, if the changes are not applied appropriately.

5. Conclusion

No doubt that the finite element method is an excellent solution method for complicated mechanical and structural engineering problems and ANSYS is an outstanding software tool for applying this method in practice. However, the software user has to have a good understanding of the software and related fields to utilize its capabilities appropriately. There are many parameters affect in the efficiency of this software. The parameters of solution methodology, material model, geometric model, mesh configuration, number of analysis steps, large deformation, program-controlled parameters, operating system and the allocated memory have been studied through this paper. In some cases, the personal experiences of other simulations and relevant findings of some other studies about this software were provided. In total, previously mentioned parameters should be chosen carefully as they directly affect on the accuracy of the results and the computational time. It should be added that sometimes reducing the computational time can be a pitfall by causing error or drastically reducing the accuracy. Sometimes it is necessary to do several trial-and-errors to find out the optimum approach. Using High Performance Computers (HPCs), installing a suitable Operating System (OS) and applying the optimal settings to hardware, operating system and the software can improve the performance of the software. It is advised to consult with the developer or authorized consultants to find out the most efficient conditions.

6. Acknowledgement

This work is supported by an Australian Research Council (ARC) Linkage Grant (ARC LP: LP0990250) in conjunction with faculty of Science and Engineering, and Institute of Health and Biomedical Innovation (IHBI), Queensland University of Technology (QUT).

7. References

[1] Liu, Gui-Rong and Nguyen, Thoi Trung, *The Finite Element Method*, in *Smoothed finite element methods*2010, CRC PressI Llc. p. 31-82.

[2] Tekkaya, AE and Martins, PAF, *Accuracy, reliability and validity of finite element analysis in metal forming: a user's perspective.* Engineering Computations, 2009. 26(8): p. 1026-1055.

[3] Zienkiewicz, Olgierd Cecil, Taylor, Robert Leroy, and Zhu, Jian Z, *The finite element method: its basis and fundamentals*. Vol. 1. 2005: Butterworth-Heinemann.

[4] Schmutz, B., Wullschleger, M.E., Noser, H., Barry, M., Meek, J., and Schütz, M.A., *Fit optimisation of a distal medial tibia plate.* Computer Methods in Biomechanics and Biomedical Engineering, 2011. 14(4): p. 5.

[5] Schmutz, B., Wullschleger, M.E., Kim, H., Noser, H., and Schütz, M.A., *Fit assessment of anatomic plates for the distal medial tibia.* Journal of orthopaedic trauma, 2008. 22(4): p. 258.

[6] Ahmad, M., Nanda, R., Bajwa, A. S., Candal-Couto, J., Green, S., and Hui, A. C., *Biomechanical testing of the locking compression plate: When does the distance between bone and implant significantly reduce construct stability?* Injury, 2007. 38(3): p. 358-364.

[7] Malekani, J., Schmutz, B., Gudimetla, P, Gu, Y.T., Schuetz, M., and Yarlagadda, P., K.D.V., *Studies on bending limitations for the optimal fit of orthopaedic bone plates.* advanced materials research, 2013. 602-604: p. 5.

[8] ASTM, *ASTM F138 - 08 Standard Specification for Wrought 18Chromium-14Nickel-2.5Molybdenum Stainless Steel Bar and Wire for Surgical Implants (UNS S31673)*, in *ASTM F138 (Medical Device Standards and Implant Standards)*2003, ASTM International: West Conshohocken, PA, .

[9] ASTM, *ASTM F139 - 08 Standard Specification for Wrought 18Chromium-14Nickel-2.5Molybdenum Stainless Steel Sheet and Strip for Surgical Implants (UNS S31673)*, 2003, ASTM International: West Conshohocken, PA, .

[10] ASTM, *ASTM F136 - 08 Standard Specification for Wrought Titanium-6Aluminum-4Vanadium ELI (Extra Low Interstitial) Alloy for Surgical Implant Applications (UNS R56401)*, 2003, ASTM International: West Conshohocken, PA, .

[11] Yarlagadda, P., K.D.V., Chandrasekharan, M., and Shyan, J.Y.M., *Recent advances and current developments in tissue scaffolding.* Bio-Med. Mater. Eng, 2005. 15: p. 159–177.

[12] Malekani, J., Schmutz, B., Gu, Y.T., Schuetz, M., and Yarlagadda, P., K.D.V., *Orthopedic bone plates: Evolution in Structure Implementation technique and biomaterial.* GSTF Journal of Engineering Technology, 2012. 1(1): p. 135-140.

[13] Onate, E. and de Saracibar, C.A., *Alternatives for finite element analysis of sheet metal forming problems.* Wood Chenot and Zienkiewicz, editors, Numerical Methods in Industrial Forming Processes, NUMIFORM, 1992. 92: p. 79–88.

[14] Rebelo, N., Nagtegaal, JC, Taylor, LM, and Passmann, R., *Comparison of implicit and explicit finite element methods in the simulation of metal forming processes.* Numerical Methods in Industrial Forming Process, 1992: p. 99-108.

[15] Soltani, B., Mattiasson, K., and Samuelsson, A., *Implicit and dynamic explicit solutions of blade forging using the finite element method.* Journal of materials processing technology, 1994. 45(1-4): p. 69-74.

[16] Research, ANSYS® Academic, *Help System*, 2012, ANSYS® Academic Research.

[17] Research, ANSYS® Academic, *Help System*, 2013, ANSYS® Academic Research.

[18] Malekani, J., Schmutz, B., Gu, Y.T., Schuetz, M., and Yarlagadda, P., K.D.V. *Biomaterials in orthopedic bone plates: a review*. 2011. Global Science and Technology Forum.

[19] DeTora, M. and Kraus, K., *Mechanical testing of 3.5 mm locking and non-locking bone plates.* Veterinary and comparative orthopaedics and traumatology: VCOT, 2008. 21(4): p. 318.

[20] Zahn, K., Frei, R., Wunderle, D., Linke, B., Schwieger, K., Guerguiev, B., Pohler, O., and Matis, U., *Mechanical properties of 18 different AO bone plates and the clamp-rod internal fixation system tested on a gap model construct.* Veterinary and comparative orthopaedics and traumatology: VCOT, 2008. 21(3): p. 185.

[21] Cesarone, DM and Disegi, JA, *Techniques in the application of ISO 9585 test method for the determination of bone plate bending properties.* Clinical and laboratory performance of bone plates, 1994: p. 65.

[22] Strom, A.M., Garcia, T.C., Jandrey, K., Huber, M.L., and Stover, S.M., *In Vitro Mechanical Comparison of 2.0 and 2.4 Limited Contact Dynamic Compression Plates and 2.0 Dynamic Compression Plates of Different Thicknesses.* Veterinary Surgery, 2010. 39(7): p. 824-828.

[23] Fujihara, K., Huang, Z.M., Ramakrishna, S., Satknanantham, K., and Hamada, H., *Performance study of braided carbon/PEEK composite compression bone plates.* Biomaterials, 2003. 24(15): p. 2661-2667.

[24] Oh, J. K., Sahu, D., Ahn, Y. H., Lee, S. J., Tsutsumi, S., Hwang, J. H., Jung, D. Y., Perren, S. M., and Oh, C. W., *Effect of fracture gap on stability of compression plate fixation: A finite element study.* Journal of orthopaedic research, 2010. 28(4): p. 462-467.

[25] Yao, Qizhou and Qu, Jianmin, *Three-dimensional vs. two-dimensional finite element modeling of flip chip packages*, 1998, Citeseer.

[26] Ramos, A and Simoes, JA, *Tetrahedral versus hexahedral finite elements in numerical modelling of the proximal femur.* Medical Engineering & Physics, 2006. 28(9): p. 916-924.

[27] Walz, Joseph E, Fulton, Robert E, and Cyrus, Nancy J, *Accuracy and convergence of finite element approximations*, 1968, DTIC Document.

[28] Ferziger, Joel H and Peric, Milovan, *Further discussion of numerical errors in CFD.* International Journal for Numerical Methods in Fluids, 1996. 23(12): p. 1263-1274.

[29] Venkatesh, K, Prakash, SV, and Srinivasa Murthy, PL, *IJERT IJERT*. International Journal of Engineering, 2013. 2(1).
[30] Bertram, Albrecht, *Plasticity*, in *Elasticity and plasticity of large deformations: an introduction*2011, Springer. p. 256-320.
[31] Hollister, SJ, *BME/ME 456 Biomechanics: Large Deformations Mechanics*, 2002, course notes. University of Michigan, College of Engineering.: USA.
[32] Dixit, M and Dixit, S, *Review of Stress, Linear Strain and Elastic Stress-Strain Relations*. Modeling of Metal Forming and Machining Processes: by Finite Element and Soft Computing Methods, 2008: p. 33-94.
[33] ANSYS, Inc, *White Paper – Obtaining Optimal Performance in ANSYS 11.0*, 2007, ANSYS Inc. p. 67.
[34] Bernstein, Joshua, *Higher Returns on the Simulation Investment.* ANSYS Advantage, 2008. II(3): p. 2.
[35] Schreiber, Olivier, Shaw, Scott, and Beisheim, Jeff, *ANSYS TM on Advanced SGIR Architectures*. 2010.
[36] Barbara Hutchings, ANSYS, *Cluster Computing with Windows CCS.* ANSYS ADVANTAGE, 2007. I(3): p. 2.
[37] Katajisto, Harri, *Finite Element modeling of the Big Wheel prototype*, 1999, B1 internal note.
[38] Inc, ANSYS, *Optimized Performance*, 2013, ANSYS Inc.