AYOUB MOFTAH MILAD YAAHY

AN INVESTIGATION ON THE OPTIMIZATION DOMAIN OF BIOLOGICAL GROWTH METHOD

NUE 2107

AN INVESTIGATION ON THE OPTIMIZATION DOMAIN OF BIOLOGICAL GROWTH METHOD

A THESIS SUBMITTED TO THE GRADUATE SCHOOL OF APPLIED SCIENCES OF NEAR EAST UNIVERSITY

By
AYOUB MOFTAH MILAD YAHYA

In Partial Fulfilment of the Requirements for the Degree of Master of Science in Mechanical Engineering

NICOSIA 2017

AN INVESTIGATION ON THE OPTIMIZATION DOMAIN OF BIOLOGICAL GROWTH METHOD

A THESIS SUBMITTED TO THE GRADUATE SCHOOL OF APPLIED SCIENCES OF NEAR EAST UNIVERSITY

By
AYOUB MOFTAH MILAD YAHYA

In Partial Fulfilment of the Requirements for the Degree of Master of Science in Mechanical Engineering

NICOSIA 2017

AYOUB MOFTAH MILAD YAHYA: AN INVESTIGATION ON THE OPTIMIZATION DOMAIN OF BIOLOGICAL GROWTH METHOD

Approval of Director of Graduate School of Applied Sciences

Prof. Dr. Nadire CAVUS

We certify that this thesis is satisfactory for the award of the degree of Master of Science in Mechanical Engineering

Examining Committee in Charge:

Prof. Dr. Mahmut A. Savas	Committee Chairman, Department of Mechanical Engineering, NEU
Assist. Prof. Dr. Ali Evcil	Supervisor, Department of Mechanical Engineering, NEU
Assist. Prof. Dr. Ehsan Kiani	Department of Automotive Engineering, NEU

I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Name, Last name: AYOUB YAHYA
Signature:
Date:

ACKNOWLEDGEMENTS

First, thanks to God alone for letting me to complete my Master Thesis success.

I would like to thanks to my supervisor, Assist. Prof. Dr. Ali Evcil for his patient guidance, encouragement and advice he has provided throughout my time as student. I have been extremely lucky to have a supervisor who cared so much about my work, and who responded to my questions and queries so promptly. In particular, I would like to thank Dogukan Evcil for his contribution during the software development.

Finally, I must express my very profound gratitude to my parents for providing me with unfailing support and continuous encouragement throughout my years of study and through the process of researching and writing this thesis. This accomplishment would not have been possible without them. Thank you.

To my parents…

ABSTRACT

The aim of the study was to investigate the effect of domain thickness in Biological Growth Method which is a tool used in structural shape optimization. The method was implemented by using MARC-MENTAT student version as the finite element code, pre- and post-processor. A small software called Biological Growth Interface (BGI) was developed to control and modify the data in the input and output files. The procedure was verified by conducting the parametric study of the plate with a hole problem discussed in the literature. The analyses were extended up to 40 mm domain thickness. It was observed that the number of iterations required for optimization decreased as the magnification factor and domain thickness increased. However, satisfactory results were obtained from the analyses resulted after more than 5 iterations.

It can be concluded that the method works with reasonable accuracy with an automatic mesh with large enough elements to prevent distortion and aspect ratio problems, an optimization domain selected roughly but including the remarkable stress changes around the hole boundary, a reference stress equal to the stress level far away from the hole and a low magnification factor to guarantee enough number of iterations for acceptable results.

Keywords: Biological growth method; domain thickness; finite element analysis; shape optimization; plane with a hole

ÖZET

Çalışmanın amacı, yapısal şekil optimizasyonunda kullanılan Biyolojik Büyüme Metodunda, etkinlik alan kalınlığının etkisini araştırmaktır. Metod MARC-MENTAT sonlu eleman paketinin öğrenci versiyonu kullanılarak uygulanmıştır. Biological Growth Interface (BGI) olarak isimlendirilen küçük bir ara yazılım girdi ve çıktı dosyalarındaki bilgileri kontrol etmek ve düzenlemek amacı ile oluşturulmuştur. Yöntem, literatürde yer alan delikli plakanın parametrik çalışması ile doğrulanmıştır. Etkin alanın kalınlığı 40 mm'ye kadar artırılarak sonuçlar incelenmiştir. Büyüklük faktörü ve etkin alan kalınlığı arttıkça iterastion sayısının azaldığı, ancak 5 iterasyondan fazla süren analizlerin tatminkar sonuç verdiği gözlenmiştir.

Metodun, otomatik sonlu eleman ağı kullanarak kabul edilebilir sonuçlar verebileceği gösterilmiştir. Bunun için elemanların boyutları şekillerindeki bozulmaları tolere edecek büyüklükte olmalı ve etkin alan seçimi delik çevresindeki gerilim yığılmalarını içine alacak şekilde yapılmalıdır. Referens gerilme delikten uzakta yer alan nominal gerilme olarak alınabilir. Yeterli sayıda iterasyon ise büyüklük faktörünü azaltarak elde edilebilir.

Anahtar Kelimeler: Biyolojik büyüme metodu; alan kalınlığı; sonlu eleman analizi; şekil optimizasyonu; delikli plaka

TABLE OF CONTENTS

APPENDICES

LIST OF TABLES

LIST OF FIGURES

List of Abbreviations and Symbols

2D	Two Dimension
3D	Three Dimension
BEM	Boundary Element Method
BGI	Biological Growth Interface
BGM	Biological Growth Method
D	Optimization domain
FEM	Finite Element Method
E	Actual Young's modulus
E_{red}	Reduced Young's modulus
k	Magnification factor
u, v, w	Displacement components
x, y, z	Cartesian coordinates
$\varepsilon_x, \varepsilon_y, \varepsilon_z$	Normal components of the infinitesimal strain tensor
$\sigma_x, \sigma_y, \sigma_z$	Normal components of Cauchy stress tensor
σ_{vm}	Equivalent von Mises stress
σ_{ref}	Reference stress
$\Delta\vartheta$	Temperature difference
Δt	Time span
β	Proportionality factor
Γ	Optimization boundary
α	Thermal expansion coefficient
ζ	Conversion factor
ν	Poisson's ratio

CHAPTER 1

INTRODUCTION

Optimization is known as the is a way through which a function can either be minimized or maximized. Optimization problems inserted are in any modeling and as well as in the designing. For identifying a model, there is a need of minimizing the distance that is between the model predictions made and the experiments which take place. Modelling can regularly be explained or expressed as a minimization of energy. For instance, the balance of a preservationist framework can be acquired by limiting its aggregate potential energy. What's more, obviously, is that the ideal plan is additionally concerned regarding the criteria of performance which is to be maximized.

Structural optimization is one of the most important because it looks for the best option out of all the designs for structure and it looks at both extremes of the design while selecting which are of minimization and maximization. Its function is to minimize the cost and the usage of material which is used for the project, and at the same time it is to make sure that the safety is taken into consideration and kept at maximum level, and also another concern is of maximizing the performance. For the design of the structure to be optimized in engineering, there are three different types structural optimization which is size, shape and topology optimization their detailed explanation is as follow:

Size optimization process selects the domain of the structure which is to be fixed, fixes it and once the process takes place, it cannot change the domain of the structure. The variables of design sizing can be in two states meaning that it can either be continuous or discrete. This process of size optimization is mostly known the application of optimization which takes place at the stage of design details.

Shape of the exterior boundary surfaces or arches is selected in the shape optimization. Examples which are known for this problem comprise of locating the border of the structure, locating the area of junctions of a skeletal structure, locating the best standards for parameters, which characterize the center surface of a shell structure. This process of shape optimization is known the application of optimization, and it is the initial design stage.

For finding the best layout for the structure according to the defined design topology optimization is used. Unlike the other optimization methodologies, typology optimization uses a grand or universal structure as its preliminary design. The issues which are identified are conditions of support, applied loads, structure volume which is to be constructed and other restrictions which might be considered by the designer of the structure. This optimization type is most tough amongst other two types (Tang, 2011).

Biological Growth Method (BGM) was introduced by Mattheck (1990), who had carried out observations in nature to come up with this method. According to BGM, if optimization were to be applied in nature, it would be done via swelling or shrinking of the outermost layer that produces the leveling of the local stress of the material. Then again, Mattheck characterizes ideal shape as the one that demonstrates a condition of consistent stress at part of, or the entirety of, the surface of the material (Cardona et al., 2006).

Hrennikoff, McHenry, and Newmark were the first ones who had started the development of Finite Element Method (FEM) in structural mechanics in 1940s. They made use of a mesh created by rods and beams for the solution of stresses in ongoing solids. Conrant5 which was in a lecture from 1941, it had given proposition which was a method for problems of the torsional model, it recommended for making use of piecewise polynomial interpolation over triangular sub-regions. As the development in the technological fields progressed, computers had come into being, and through the use of a computer it became possible for writing and solving the stiffness equations in the form of a matrix. The matrix of stiffness equation for the beam, truss, and various elements had been presented in a study carried out by Turner, Clough, Topp, and Martin in 1956. Clough was the one who had come up with the finite element and was credited for it. A great deal of work had been put into the development of finite element method. This work has been carried into the fields related to the formulation of the elements and as well as the implementation of a computer. There are number of developments which have been achieved in the computer technology such as the hardware, accurate solutions for matrix, efficiency in matrix solver, graphics which help to ease the visual of the process stages, generation of mesh, and as well as in the stages which take place after the processing (Budynas & Nisbett, 2008).

Boundary Element Method (BEM) is a technique which is used for conversion of equations governing into equivalent integrals. It uses the associations from vector calculus which relate to Gauss-Green or the divergence theorem, which include both surface and volume integrals, are converted to integral equations which do not consist of volume integrals concerning the unknown response. The last conversion includes few known solutions (fundamental solutions) related to the original differential equation.

The aim of the study was to implement the biological growth method using finite element software MSC Marc-Mentat student version and to investigate the effects of domain thickness on the method. A parametric study, also including the domain thickness among others, was conducted. A much more simpler and faster analysis technique was the expected outcome.

CHAPTER 2

LITERATURE REVIEW

2.1 Biological Growth Method

Biological Growth Method (BGM) was introduced by Mattheck (1990), who had carried out observations in nature on trees, their joints, deer antlers etc., in order to come up with the method. According to him natural substances are able to optimize their shapes and structures depending on the load themselves. He has defined optimum shape as “the one that shows a state of constant stress at part of or the whole of the surface of the component.” According to BGM, if optimization was to be applied in nature, it would be done via swelling or shrinking of most outer layer that produces the levelling of local stress of the material (Wessel et al., 2004).

2.2 History

The best example for shape optimization in a natural and simple state would be of bones and trees. They tend to bring change in their structure according to the external loads which are put upon them, this change takes place to reduce the stress.

Computer –Aided Shape Optimization (CAO) was developed by Mattheck and Burkhardt (1990), algorithm by simulating tree growth to optimize mechanical engineering structures. The method assumed that in all structures considered, a state of constant stress at the surface of the biological 'component' was always given the natural loading case applied. This technique is therefore equivalent to a procedure which material is added at overloaded places in the structure and is not added (or even removed) at places with stresses below the reference stress until the optimal shape attained (Mattheck, 1991).

An optimization algorithm known as Soft Kill Option (SKO) was proposed by Baumgartner et al. (1992), this algorithm was developed in order to locate the optimum structural topology depending on the replication of reconciling bone mineralization by having to change Young’s modulus depending on the calculated stress distribution. According to Mattheck and Burkhardt (1990), the optimum topology which is obtained can be made use of in order to create a new model of finite element for the subsequent shape optimization with the help of CAO to even out the contours and for the reduction of stress which remain (Baumgartner et al., 1992).

It was Chen and Tsai (1993) who had broadened the approaches for simulated biological growth with the help of fabricated temperature loading in order to lessen the stress concentration which was subjected to area limitations or to lessen area (weight) subjected to stress limitations.

According to Tekkaya and Guneri (1996), implementation of biological growth methodology was a part of experiential method and calculated systematically the impact of parameters that manage the process of optimization, on the procedure of optimization when minimizing the concentration of stress of a squared plate that initially contained circular hole under biaxial tensions.

A mixed method which was of experimental and evolutionary methods was proposed by Le Rice Le Riche and Cailletaud (1998), it was created to come up with the solution for the problems of shape optimization. In improvement of designs biological growth had been measured as an efficient approach yet the problem it faced was that it was not able to produce a global optimum shape. Evolutionary or genetic algorithms (Hajela, 1990; Jenkins, 1991; Rajeev and Krishnamoorthy, 1992) are able to manage problems related to nonconvex and find the global optimal shape but as large problems are in question then the calculated cost would be very high. Hence, mixture of the evolutionary approach and biological growth method was considered to an efficient and cost effective approach. As the outcomes were in agreement with the results of Le Riche and Cailletuad (Le Riche, & Cailletaud, 1998).

Cai, et al., (1998) developed and proposed a method which for the structural shape optimization, this method added the Boundary Element Method (BEM) with biological growth optimization method. The method proposed was considered to be correct as it had proven couple of examples. It came out to be an efficient, simple and effective method for shape optimization. Carolina et al., (2004) noted the implementation of BGM with BEM. Boundary-only along with the accuracy for the dislocation and stress solutions are the most special and known intrinsic characteristic of BEM which make this method efficient and effective for the solutions of shape optimization problems (Wessel et al., 2004).

An adjusted approach of biological growth method was presented by Tian & Shangjin (2004), this approach is able to get the shape optimization of structure though a complex geometry solution. This solution has three parts to it. First, there is no use of node coordinates in the modification of FEM model, the structure’s boundary is defined with the help of B-spline curve. Second, there is a cost function which is created in order to allow the structural weight to be decreased to its minimal level, which is subject to limitations of stress and geometrical. Therefore, there is an improvement in the biological growth method which allows it to optimize the design of the complex geometry. Third, as the evolution of shape optimization takes place, there is a method which is related to the penalty, it deals with anyone who violates the constraint settings. This adjusted approach had been tested and was successfully implemented for the shape optimization of centrifugal impellers (Tian & Shangjin, 2004).

A new approach had been presented for the shape optimization for three dimensional and damage tolerant structures by Peng & Jones (2008). This approach makes use of a new method, which is known as Failure Analysis of Structures (FAST), it is applied to get the estimation of the stress-intensity factor for the cracks at a notch. CAD and FAST codes are made use of in the development of methodology and software which are used for the automation damage-tolerance calculations. In order to find the location of worst cracks, modeling of number of cracks by the fractured critical edges of the structure is done by the help of FAST. FAST is later used for the evaluation of damage-tolerance objective functions for the algorithms of optimization. To understanding the problem which is being faced by optimization with fatigue life is done via stress-based biological growth method. Hence, by the help of numerical examples this has proven that a stress-optimized structure is not essentially going to provide the longest fatigue life (Peng & Jones, 2009).

Over the past years there has been various methods proposed, adaptive biological method is an example which was proposed for the reduction of cost and for improving accuracy (Zehsaz, Torkpanpouri & Paykani, 2013). In the study carried out by Zehsaz, Tokpanpouri, and Paykani (2013), influences of step factor, control points coordination and number of control points in the convergence rate were taken into consideration. ANYSYS Parametric Design language (APDL) was used for writing the codes, In APDL, parameters being studied are taken as inputs and it gives the best shape for the components which are being studied. The results of the study had shed light upon attaining successful optimization showed that step factor must be kept within a certain range in order to attain the successful optimization. Another way for attaining optimized shape is by making use of any coordinate system which is used for defining control points and as well as having to select any direction for stimulus vector of algorithm. Moreover, if the number of control points are increased, it can cause creation of non-uniformities in the studied boundaries. Having to attain the acceptable accuracy is impossible because of the formation of saw form at the studied boundary known as "saw position" (Zehsaz, Torkpanpouri & Paykani, 2013).

2.3 The Fundamental Procedure for the Method

Biological Growth Method (BGM) function is defined as:

$$\text{Minimize } [\sigma_{vm}(x,y,z) - \sigma_{ref}] \ \forall \ (\text{x,y,z}) \ \epsilon \ D \tag{2.1}$$

where $\sigma_{vm}(x,y,z)$ is the von Mises stress at any point at the optimization domain D and σ_{ref} is known as the reference stress. And through the reference stress the von Mises stress distribution tends to clear away. In correspondence to the growth of biological structures under loads, it is proposed that (2.1) can be satisfied if the optimization domain changes its shape according to:

$$\dot{\varepsilon_{sw}}(x,y,z) = \beta[\sigma_{vr}(x,y,z) - \sigma_{ref}] \tag{2.2}$$

Where $\dot{\varepsilon_{sw}}(x,y,z)$ is the volumetric swelling strain-rate which is proportional to the driving function, i.e. the deviation of the von. Mises stress from the reference stress at a generic location in the optimization domain. The proportionality factor is given by β. The volumetric swelling scheme can be attained with the use of an Euler integration scheme for a timespan of Δt as shown below:

$$\varepsilon_{sw}(x,y,z) = \beta[\sigma_{vr}(x,y,z) - \sigma_{ref}]\Delta\text{t} \qquad \forall(x,y,z) \ \epsilon \ D \tag{2.3}$$

An elegant method to implement the swelling equation (2.3) is by means of a thermal analogy. It can be shown that this analogy is based on the generalized Hooke's law (shear strains are discarded):

$$\varepsilon_x = \frac{1}{E}\,[\sigma_x - \nu\,(\sigma_y + \sigma_z)] + \alpha\,\Delta\vartheta$$

$$\varepsilon_y = \frac{1}{E}\,[\sigma_y - \nu\,(\sigma_x + \sigma_z)] + \alpha\,\Delta\vartheta \tag{2.4}$$

$$\varepsilon_z = \frac{1}{E}\,[\sigma_z - \nu\,(\sigma_y + \sigma_x)] + \alpha\,\Delta\vartheta$$

Here, ε_x, ε_y and ε_z are strain components, normal components of stresses are depicted by σ_x, σ_y *and* σ_z that are part of the Cauchy stress tensor, Poisson's ratio is depicted by ν, the coefficient of thermal expansion is represented by α and Δϑ represents the change in the temperature. Now, if the mechanical loads on the structure to be optimized are removed and a great reduction in the Young's modulus *E* of the optimization domain is made, then abandonment of the first parts of the strains can be done even by keeping the same boundary conditions of the real problems, Therefore,

$$\varepsilon_x \approx \varepsilon_y \approx \varepsilon_z \approx \alpha\,\Delta\vartheta \tag{2.5}$$

In the optimization domain D, if thermal expansion is just defined to be as non-zero, then, from equation (2.5)

$$\varepsilon_{sw}(x,y,z) = \alpha\,\Delta\vartheta(x,y,z) \quad \forall\,(x,y,z) \in D \tag{2.6}$$

Comparison of equation (2.3) with equation (2.6) indicates the correspondence

$$\Delta t \quad \Leftrightarrow \quad \alpha \qquad (2.7)$$

$$\beta\left[\sigma_{vm}(x,y,z) - \sigma_{ref}\right] \quad \Leftrightarrow \quad \Delta\vartheta(x,y,z) \qquad (2.7)$$

Equations (2.6) and (2.7) are the basic relations of the thermal analogy for the swelling phenomenon given in equations (2.2) and

(2.3)

Mathematical framework shown above for the optimization and its parameters is explained below:

1. First the optimization boundary has to be selected which is depicted by Γ.

2. Size of the region D which has to be optimized has to be decided that is constrained from by Γ.
3. The mechanical analysis has to be conducted via:
    a. Provided the conditions of mechanical loading;
    b. Provided the conditions of essential boundary;
    c. And properties of the original material.

    Von Mises stress distribution of the optimization region D is found through this analysis.

4. The mechanical analysis can be carried via:
    a. Thermal loads

$$\Delta\vartheta(x, y, z) = \zeta\, [\sigma_{vm}(x, y, z) - \sigma_{ref}] \tag{2.8}$$

Conversion factor is depicted by ζ, with having unit's degrees temperature per stress. And reference stress is depicted by σ_{ref}

    b. For the optimization, non-zero α which is the thermal expansion coefficient must be used and for other regions, zero expansion coefficient can be used.
    c. For the D (optimization domain), Young's modulus E_{red} is significantly reduced.

Through this calculation the displacements u, v, w can be provisioned along the optimization surface Γ which bounds D.

5. The optimization boundary Γ must be update with:

$$X = x + k\,u(x,y,z)$$
$$Y = y + k\,v(x,y,z) \quad (2.9)$$
$$Z = z + k\,w(x,y,z)$$

Here K represents a magnification factor that is essential for the acceleration convergence.

It is necessary to repeat steps 3 to 5 until there is no change noted and detected in the driving function. It must be noted that this procedure enables interfering of the user at the steps 1 and 2.

There has to be total of seven parameters which are to be set during the implication of the method. From these parameters, similar results are shown by ζ, α and k: All of these parameters tend to behave like magnification factors. Hence, this study, has only taken k into consideration, while ζ considered as unity, and the definite thermal expansion coefficient is α. The reduced Young's modulus has only a minor effect on the results as long as it is considerably small preventing any constraints owing to static indeterminacy. For this reason, the value of E_red is set equal to 1/400 of the actual Young's modulus of the material. On the other hand, selection of the optimization boundary Γ is an engineering decision and depends on the problem in hand, so that it is selected intuitively for the analysis problem described in the next section. (Tekkaya & Güneri, 1996).

2.4 Some Different Applications (2D, 3D)

In this section some applications from previous scientific papers were discussed.

2.4.1 Applications (2D)

- **Cantilever beam under top shear loading**

A cantilever beam under top uniform distributed shear loading, as shown below in Figure 1, is chosen as the first example. The length and the width of the beam are 5 m and 1.2 m, respectively. 6 MN\m is the value of the top shear loading. 210 GPa and 0.3 is the value for the Young's modulus and Poisson (Chen, & Tsai, 1993).

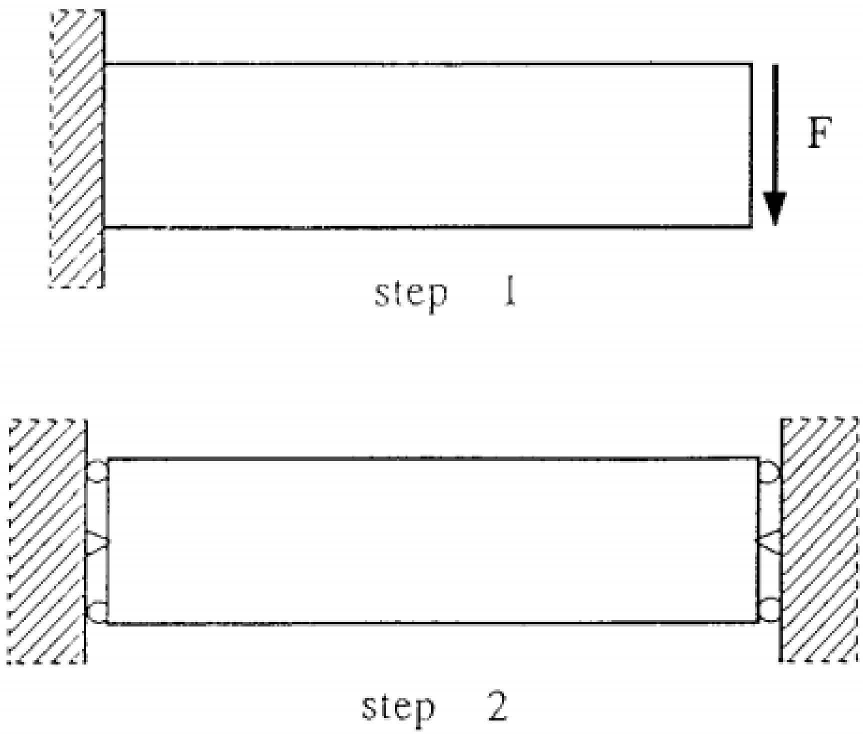


Figure 2.1: A cantilever beam under end shear load

- **Square plate with a hole under biaxial tension**

It can be seen in the figure 2, there is a square plate which has a hole that is there to even tensile loads by its edges. The emphasis of the stress is on the pinnacles of the hole. The main aim of the optimization is to come up with a shape which could be given to the hole in order to minimize the stress which is in the boundary hole elements. The plate is of 12 in length and the hole is of 2 in length. Young's modulus is 30 x 10^6 psi (69 GPa), Poisson's ratio is 0.3 and load P is 10 lb/in (1750 N/m) (Chen, & Tsai, 1993).

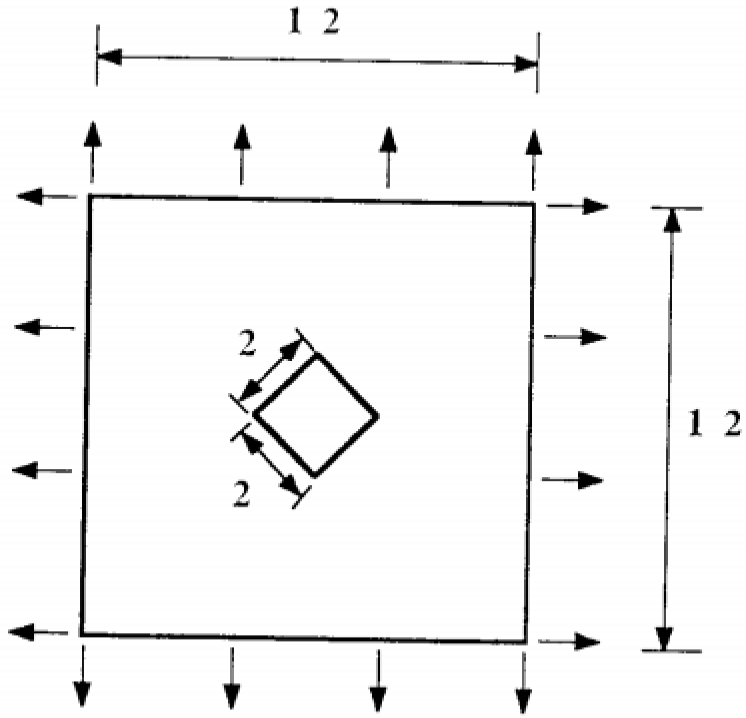


Figure 2.2: A square plate with hole under biaxial loading

- **Plate-with-a-hole problem**

The plate considered is a square with dimensions 300× 300mm as shown in Figure 3 with a center hole of diameter 80mm and thickness of 5mm. The material is presumed to be a standard steel which has the Young's modulus of 210 GPa and its Poisson's ratio is 0.3. The applied stress along the sides perpendicular to the x-axis is taken as 45MPa and along the sides perpendicular to they-axis is taken as 22.5 MPa. The stress state is taken two-dimensional (Tekkaya & Güneri, 1996).

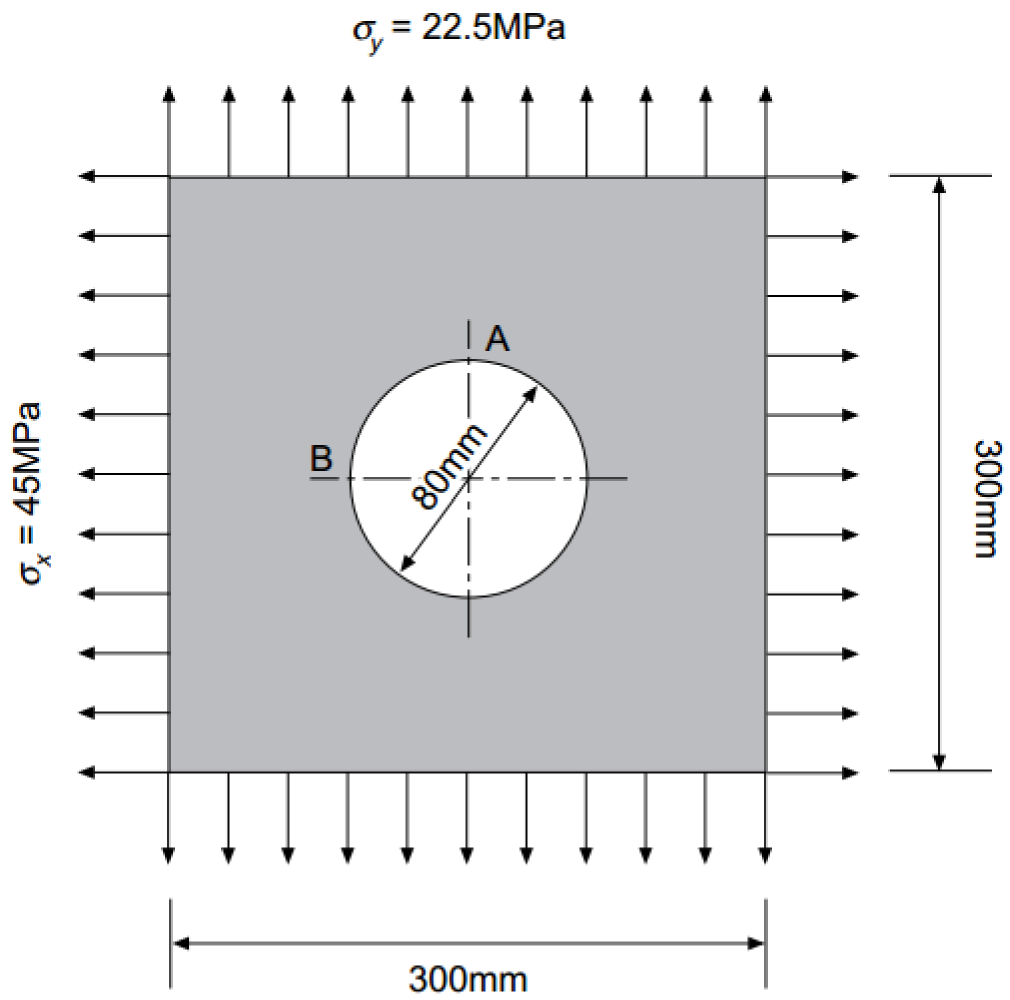


Figure 2.3: Plate-with-a-hole

2.4.2 Applications (3D)

- **Plate-with-a-hole problem**

Figure 4 given below shows a 3-D plate with a hole and it has a continuous in-plane tension of 100 MPa in the direction of x. As there is a symmetry, only a quarter part of the plate is ideal. In this example, the stress concentration factor is of 3 which is located at curve of the hole where it crosses y-axis. The externally applied tensile stress and the reference Mises stress are set to be the same, and for the ending of the loop criterion 2 was selected. Criterion 2 was selected because criterion 1 was not a good option because it could reach only zero-driving force as the hole grows together. And value of Poisson ratio which would require a fine mesh at the vertex on the main axis of the transient ellipse during the remolding phase is of v=0.0 (Mattheck, & Moldenhauer, 1990).

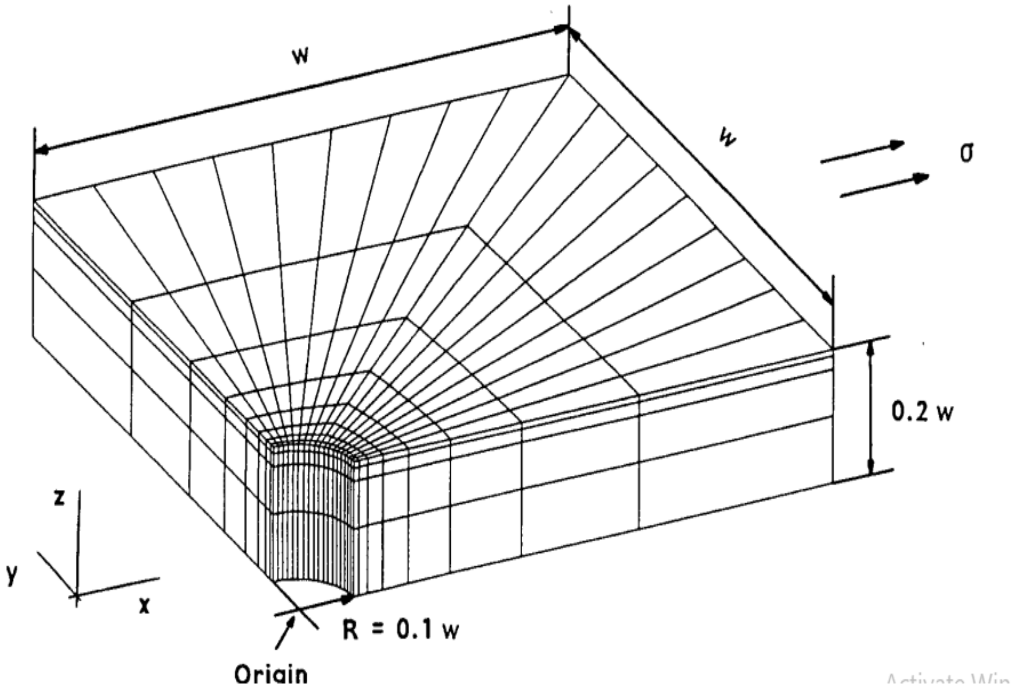


Figure 2.4: Plate-with-a-hole (3D)

CHAPTER 3

METHODOLOGY

The methodology used in the study involves the adaptation of the biological growth method for shape optimization to the student version of a commercial finite element code with the aid of a software developed. The flowchart of the procedure is shown in Figure 3.1 together with the tools used.

3.1 Optimization Tools

3.1.1 Marc-Mentat student version (2016.0.0.SE)

MARC-MENTAT student version is a limited and combined application of MARC finite element software and MENTAT pre- and post-processor. MENTAT is a powerful tool to generate finite element models, run MARC and interpret the results obtained. MARC can be run externally if the required data file is readily available.

3.1.2 Biological Growth Interface

Biological Growth Interface (BGI) is a software developed during the study using Java to transfer the required data between the input and output files created by MARC and MENTAT. It is also used to input the optimization parameters during the optimization process.

3.2 Modelling

MARC-MENTAT student version (2016.0.0 SE) were used as pre-processor to form the models. The models could be saved as *.mud or *.mfd files. The mesh, displacement boundary condition, geometric properties and element types of structural and thermal analyses models were the same. The elements and nodes of the domain to be optimized were defined. The two models deviate from each other as described below.

In the stress analysis model, force boundary conditions were applied. The material properties were given as it is for the material under consideration. von Mises stresses were selected to be given as output.

In the thermal analysis model, the material properties were defined. However, the Young's modulus was defined as 525 MPa (softer) for the domain elements. Displacements were selected to be given as output.

The data files (*.dat), needed to run MARC finite element software externally, were generated by running MARC via the application MARC-MENTAT student version (2016.0.0 SE) using the model files (*.mud or *.mfd) formed. The data file for the stress analysis is ready to run within the first iteration and therefore its name was given as *V1.dat. However, thermal boundary conditions were missing in the thermal analysis data file and must be added during the first iteration. The name of the thermal analysis file therefore was given as *V0.dat.

3.3 Optimization

The optimization iterations were conducted by Biological Growth Interface (BGI) software developed. After each iteration BGI stops and waits for new data set for the next iteration. At the beginning of each iteration it is required to enter the data files and three optimization parameters, namely, stress reference, stress-temperature factor and magnification factor. There is no need to re-enter the parameters if they will remain the same. However, data files for stress analysis (*stress*V*i*.dat) and thermal analysis (*thermal*V*i-1*.dat) must be updated after each iteration. Before the first iteration, BGI does the necessary changes to *thermal*V0.dat file to include the thermal boundary conditions, assigned as $\Delta T = 0$ to the nodes defined in the set Domain Nodes.

BGI then calls MARC to conduct the stress analysis using the file *stress*V*i*.dat. The files *stress*V*i*.out, *stress*V*i*.t16, *stress*V*i*.t19 are created as outputs. *.t16 (binary) and *.t19 (ASCII) files can be used to visualize the results using MENTAT as post-processor. The results obtained are also listed in *.out file.

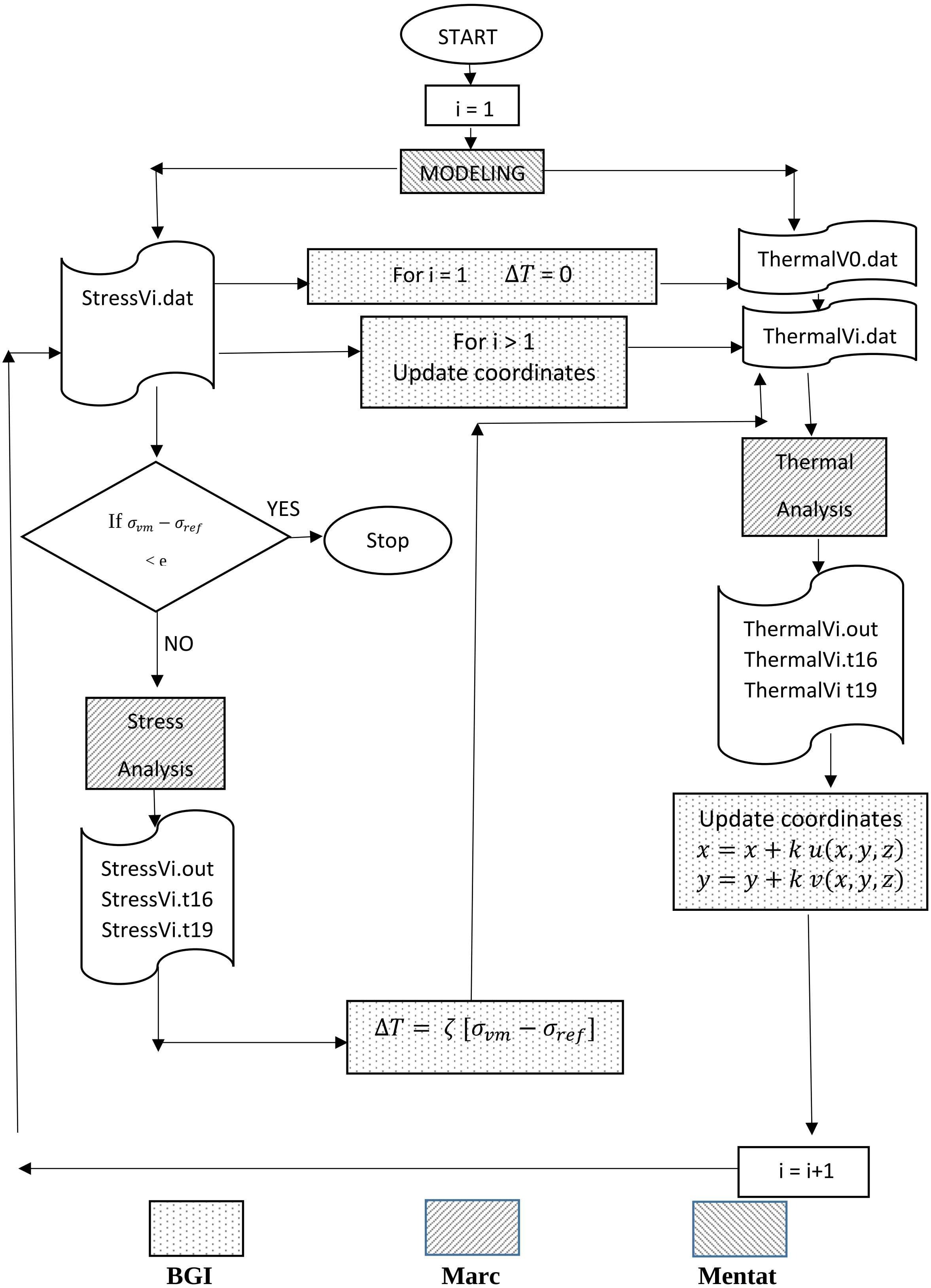


Figure 3.1: Flow chart of Biological Growth Method used

BGI then opens the *stress*Vi.out file and reads the von Mises stresses at every integration point. BGI finds the integration points around each node listed in the set Domain Nodes and takes the averages of their von Mises stresses to calculate the nodal von Mises stresses. The differences between the von Mises stresses and the reference stress multiplied by stress-temperature factor are assigned as temperature differences ($\Delta T = \zeta\ [\sigma_{vm} - \sigma_{ref}]$) to the nodes in the set Domain Nodes in the file *thermal*Vi*-1*.dat and the file is saved as *thermal*Vi.dat.

BGI calls MARC again to run *thermal*Vi.dat and the files *thermal*Vi.out, *thermal*Vi.t16, *thermal*Vi.t19 are created as outputs similar to that of stress analysis. The deflections of the nodes in the set Domain Nodes are obtained from the *thermal*Vi.out file and the coordinates of these nodes were updated in *stress*Vi.dat file to form *stress*Vi*+1*.dat file.

BGI now pauses and waits for a command for further optimization. The user is now expected to analyze the results and decide to continue or to stop. To continue, it is required to change the file names as *stress*Vi*+1*.dat and *thermal*Vi.dat and click on the run button.

The format of the *stress*Vi.dat, *stress*Vi.out, *thermal*Vi.dat and *thermal*Vi.out files are given in Table 3.1, Table 3.2, Table 3.3 and Table 3.4, respectively. Sample files are also presented in Appendix A1 to A4

Table 3.1 :Format of file StressVi.dat

Title	Line	Column	Explanation
sizing		1	--
		2	Total number of elements
		3	Total number of nodes
		4	--
connectivity	1		--
	others	1	Element number
		2	Element type
		3	1st elemental node
		4	2nd elemental node
		5	3rd elemental node
		6	4th elemental node
coordinates	1		--
	others	1	Nodes Numbers
		2	The coordinates of the point in the axis X
		3	The coordinates of the point in the axis Y
		4	The coordinates of the point in the axis Z
define node set apply#_nodes		1-N	Nodes defined in the in apply#-nodes set
Define ndsq set Domain_Nodes		1-N	Nodes defined in the Domain nodes set
Define element set Domain_elements		1-N	Elements defined in the Domain elements set
isotropic	1		Material type
	2		--
	3	1	Young's modulus
		2	Poisson's ratio
	7		Nodes numbers
geometry	1		---
	2	1	Thickness
	3		Nodes numbers
fixed temperature	1-6		Data about displacement boundary conditions
fixed disp	1-6		Data about displacement boundary conditions

Table 3.2 :Format of file stressVi.out

Title	Line	Column	Explanation
sizing		1	--
		2	Total number of elements
		3	Total number of nodes
		4	--
elements		1	Element type
tresca mises	1	2 4	Element no Integration point
	2	2	Section thickness
	3	3	Values von Mises stress
total displacements	1		--
	2		--
	3	1	Node number
		2	Displacement in x-direction
		3	Displacement in y-direction
		4	Node number
		5	Displacement in x-direction
		6	Displacement in y-direction
		7	Node number
		8	Displacement in x-direction
		9	Displacement in y-direction
total equivalent nodal forces	1		--
	2		--
	3	1	Node number
		2	Result in x-direction
		3	Result in y-direction
		4	Node number
		5	Result in x-direction
		6	Result in y-direction
		7	Node number
		8	Result in x-direction
		9	Result in y-direction
reaction forces at fixed boundary conditions	1		--
	2		--
	3	1	Node number
		2	Result in x-direction
		3	Result in y-direction
		4	Node number
		5	Result in x-direction
		6	Result in y-direction
		7	Node number
		8	Result in x-direction
		9	Result in y-direction

Table 3.3 :Format of file thermalVi.dat

Title	Line	Column	Explanation
Sizing		1	--
		2	Total number of elements
		3	Total number of nodes
		4	--
Connectivity	1		--
	others	1	Element number
		2	Element type
		3	1^{st} elemental node
		4	2^{nd} elemental node
		5	3^{rd} elemental node
		6	4^{th} elemental node
coordinates	1		--
	others	1	Nodes numbers
		2	The coordinates of the point in the axis X
		3	The coordinates of the point in the axis Y
		4	The coordinates of the point in the axis Z
define node set apply#_nodes		1-N	Nodes defined in apply#-nodes set
define node set applyT_nodes		1-N	Nodes defined in applyT#-nodes set for thermal BG
define ndsq set Domain_Nodes		1-N	Nodes defined in the Domain nodes set
define element set Domain_elements		1-N	Elements defined in the Domain elements set
isotropic	1		Material type
	2		--
	3	1	Young’s modulus
		2	Poisson’s ratio
		3	--
		4	thermal expansion
	7		Nodes numbers
geometry	1		---
	2	1	Thickness
	3		Nodes numbers
fixed temperature	1-6		Data about displacement boundary conditions
fixed disp	1-6		Data about displacement boundary conditions

Table 3.4 :Format of file thermalVi.out

Title	Line	Column	Explanation
sizing		1	--
		2	Total number of elements
		3	Total number of nodes
		4	--
elements		1	Element type
Tresca mises	1	2 4	Element no integration point
	2	2	section thickness
	3	3	Values von mises stress
total displacements	1		--
	2		--
	3	1	Node number
		2	Displacement in x-direction
		3	Displacement in y-direction
		4	Node number
		5	Displacement in x-direction
		6	Displacement in y-direction
		7	Node number
		8	Displacement in x-direction
		9	Displacement in y-direction
total equivalent nodal forces	1		--
	2		--
	3	1	Node number
		2	Result in x-direction
		3	Result in y-direction
		4	Node number
		5	Result in x-direction
		6	Result in y-direction
		7	Node number
		8	Result in x-direction
		9	Result in y-direction
reaction forces at fixed boundary conditions			(Same as total equivalent nodal forces)
total nodal temperatures	1		--
	2	1	Node number
		2	Temperature of node
		3	Node number
		4	Temperature of node
		5	Node number
		6	Temperature of node
		7	Node number
		8	Temperature of node
		9	Node number
		10	Temperature of node

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Verification of Biological Growth Method

Verification of the method was done by using the plane with a hole problem under bi-axial loading as shown in Figure 4.1 and described by Tekkaya (1996). The thickness of the plane was taken as 5 mm. Due to symmetry one-fourth of the plane was modeled and symmetry boundary conditions were applied as shown in Figure 4.2.

Optimization parameters are given in Table 4.1. Domain thickness of 10 mm is used for the verification of the model. The analyses with domain thicknesses from 20 to 40 mm were further examined and will be discussed after the verification section.

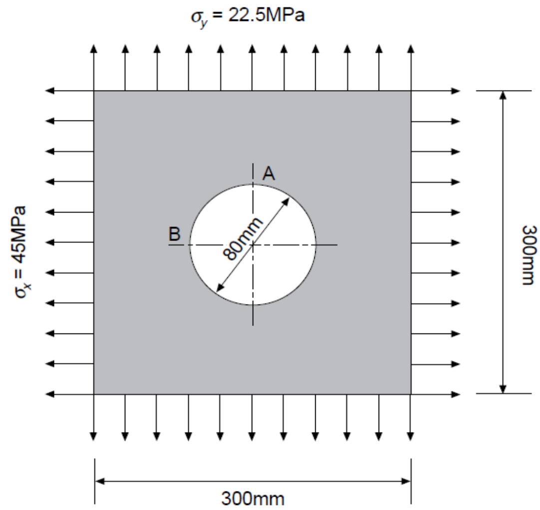


Figure 4.1: Description of the problem

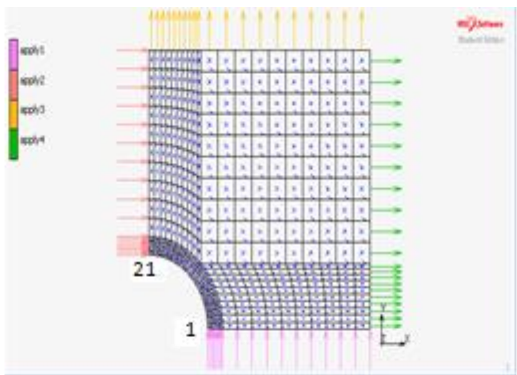


Figure 4.2: Finite element discretization of one-quarter of the plate

Table 4.1: Optimization parameters

Optimization boundary, Γ	Hole boundary
Stress-temperature factor, $\boldsymbol{\zeta}$	1°C/MPa
Reduced Young's modulus, $\boldsymbol{E_{red}}$	525 MPa (1/400 of original E)
Thermal expansion coefficient, $\boldsymbol{\alpha}$	0.0000108 m/m/°C
Reference stress, $\boldsymbol{\sigma_{ref}}$	10, 40, 60 MPa
Magnification factor, **K**	250, 275, 500, 750, 1000
Domain thickness, **D**	10, 20, 30, 40 mm

The finite element models and the boundary conditions for stress and thermal analysis are shown in Figure 4.3 for domain thickness 10 mm. The von Mises stresses of stress analysis and total deflections of the thermal expansion analysis of the original shape can be seen in Figure 4.4. The stress far away from the concentration zones is about 40 MPa. This value of stress would exist in the plate without the hole. The maximum and minimum von Mises stresses were around 130 MPa and 10 MPa respectively.

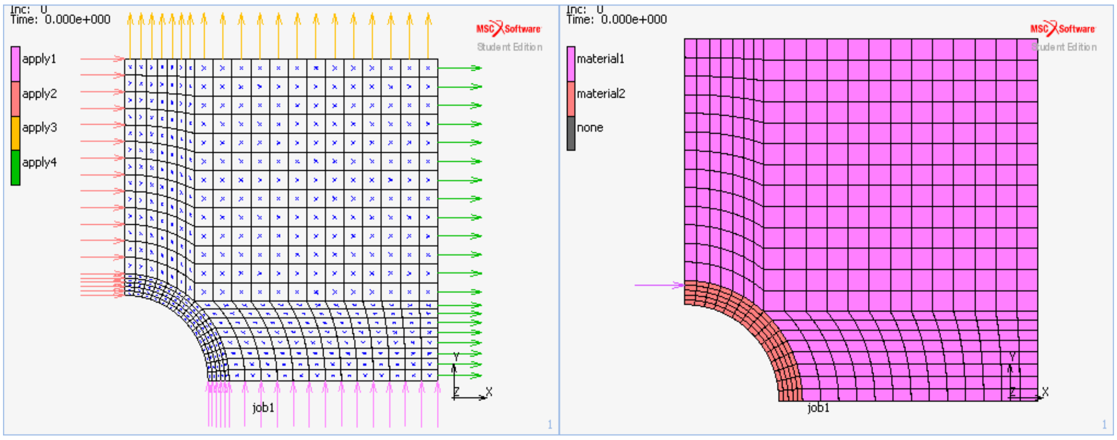


Figure 4.3: Finite element model for stress (left) and thermal (right) analysis (D=10mm)

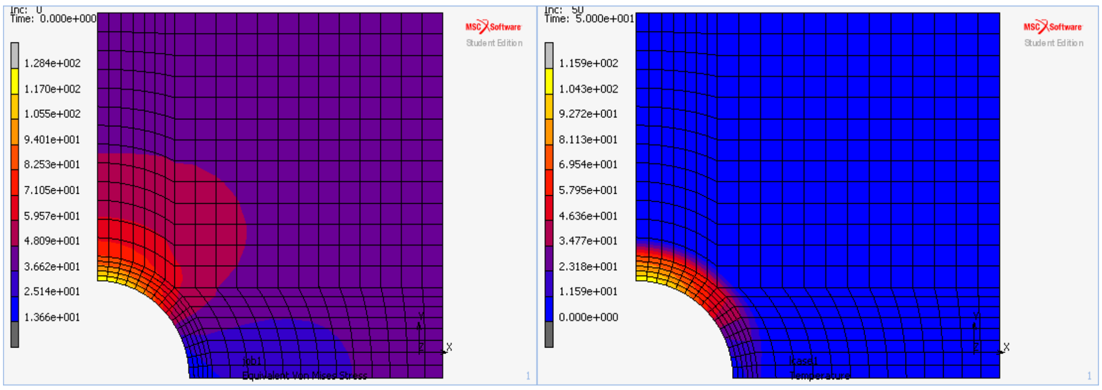


Figure 4.4: von Mises stresses (left) and thermal deformations (right) of the original shape

For domain thickness D = 10 mm, fifteen optimization analyses were conducted including the combinations of reference stresses σ_{ref} = 10, 40 and 60 MPa and magnification factor k = 250, 275, 500, 750 and 1000. The results obtained including (a) von Mises stress distributions of the plate after first and last iterations, (b) the change of von Mises stresses by iterations along the hole boundary, (c) the change of ellipse axes ratio by iterations are presented in Figures 4.5 to 4.19.

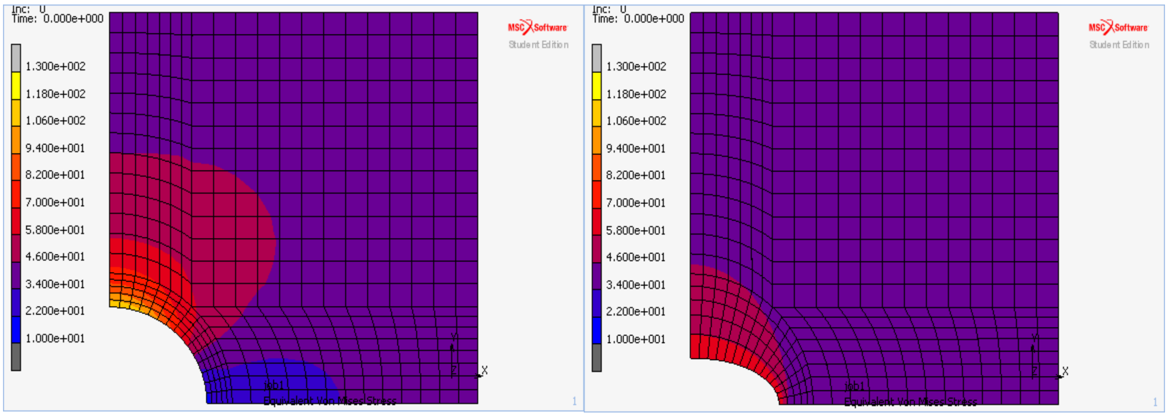
Generally, it was observed that the maximum von Mises stress of about 130 MPa at the beginning of the optimization analysis close to the hole boundary dropped down to the values around 70 MPa as the hole changed its shape to an ellipse with an ellipse axes ratio of around 2.

The results obtained were summarized in Table 4.3 together with results obtained by Tekkaya (1996) for comparison. It should be noted at this point that the main difference between the present study and Tekkaya (1996) was in the first the coordinated of the nodes in the domain set were only modified after every iteration according to the thermal deflections. However, in the second the coordinates of the nodes on the boundary of the hole were modified. A new mesh was regenerated after each iteration keeping the thickness of the domain as constant. In the present study, the thickness of the domain does not remain constant but changes during the optimization process, as it can also be seen in the figures. Even with this remarkable difference between the two studies, the results are still in good agreement with each other, verifying the methodology used.

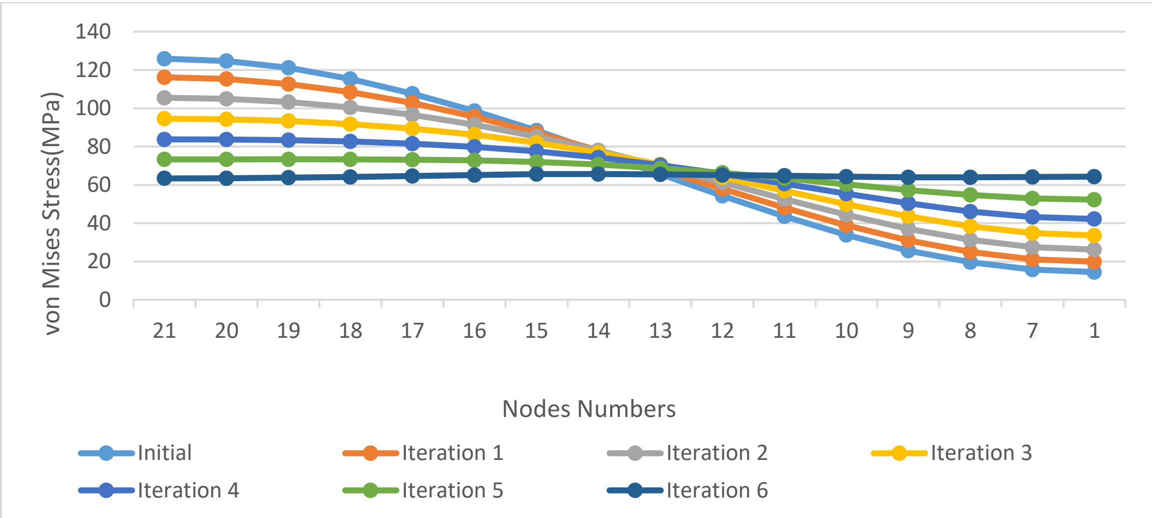
Number of iterations required for convergence decreased with increasing value of magnification factor. Very high magnification values caused iteration numbers as low as 2 for convergence. These values were considered as not trustable since the method does not have enough number of steps to regulate the optimum shape. This might be the reason why the analyses were not conducted by Tekkaya (1996) for σ_{ref} = 10 MPa and k = 750 and 1000. The results of these combinations showed that the convergence occurs in 2 iterations. Even the results for k = 750 were reasonable, the result for k = 1000 could not be accepted.

As the reference stress value was increased towards the expected final stress value, the number of iterations were increased. The final maximum von Mises stress values were obtained around 70 MPa for σ_{ref} = 40 MPa and σ_{ref} = 60 MPa. For σ_{ref} = 10 MPa these values are lower and vary between 65 and 70 MPa. It may be concluded that even though a σ_{ref} equal to the expected final maximum stress will give the best result, a value around the stress level some distance away from the stress concentration will be satisfactory.

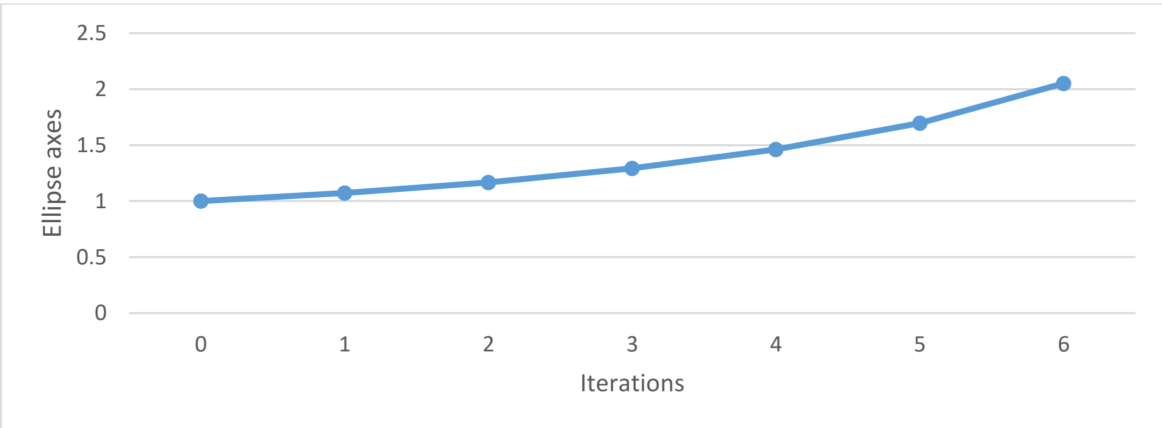
Having determined the effect of reference stress and magnification factor on the performance of the optimization procedure, it was challenging to examine the effect of domain thickness which was not investigated by Tekkaya (1996). In the following sections domain thicknesses 20, 30 and 40 mm were examined. The aim was to determine if it was possible to develop a way to simplify the modelling procedure.



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 6 (right)

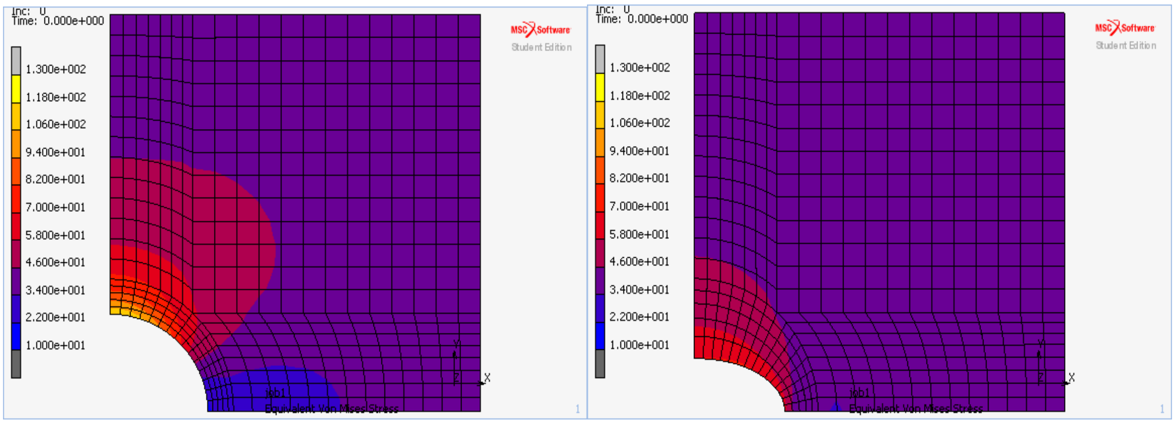


(b) Variation of von Mises stress distribution along the hole boundary

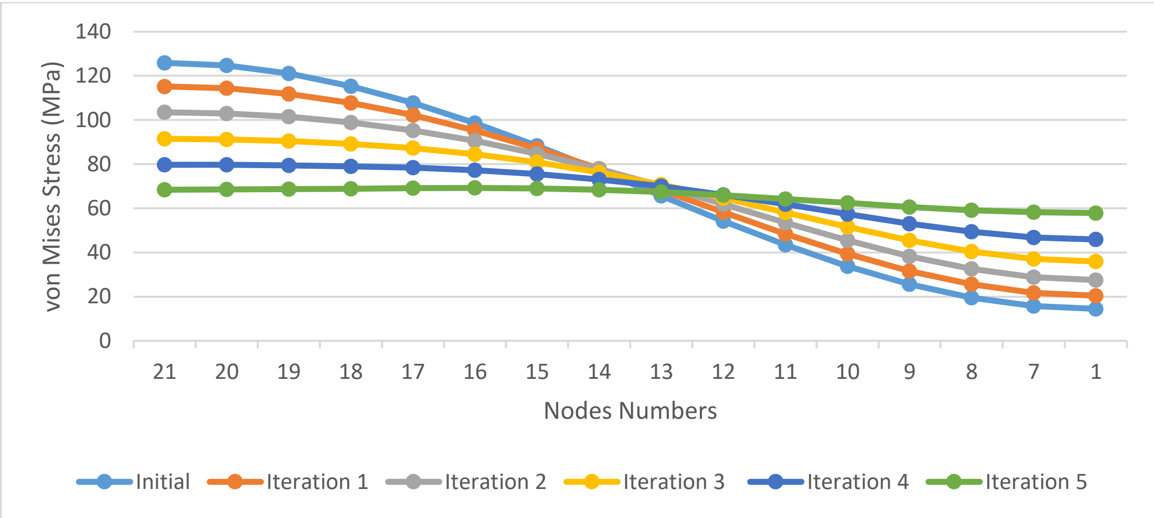


(c) Variation of ellipse axes ratio

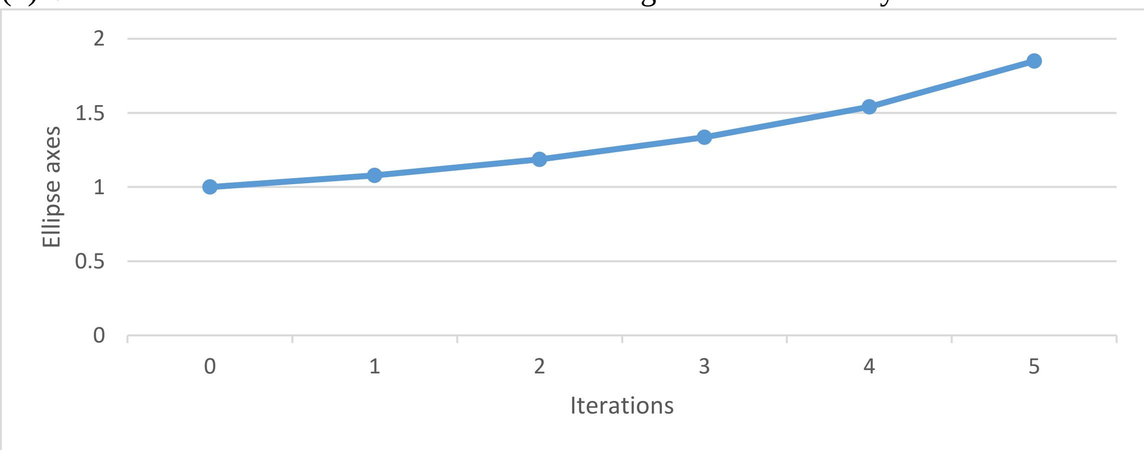
Figure 4.5: Optimization results of a plate with a hole (D=10 mm, K = 250, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)

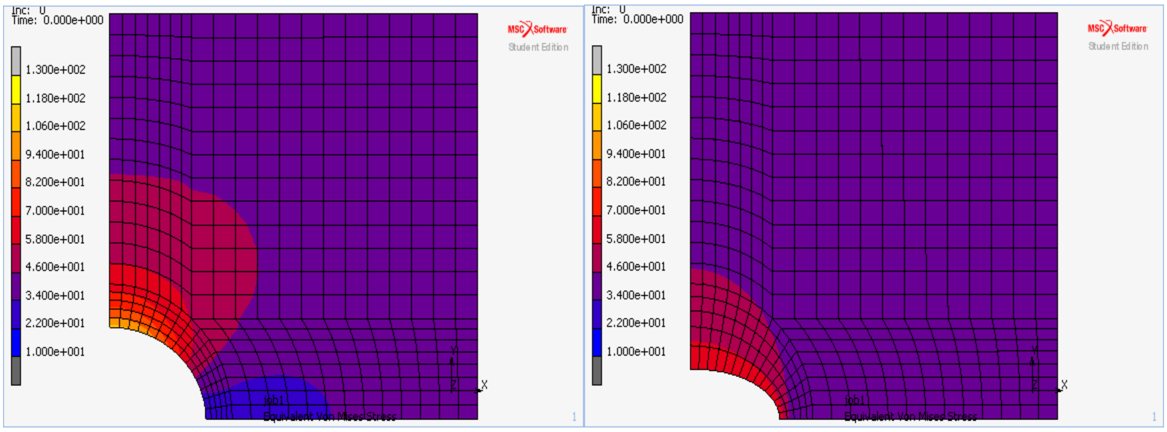


(b) Variation of von Mises stress distribution along the hole boundary

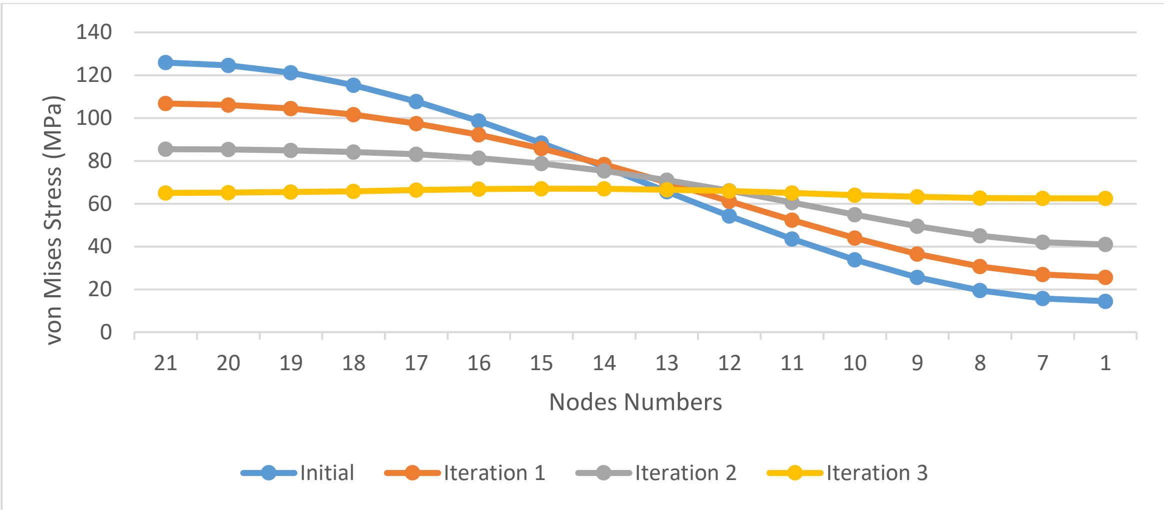


(c) Variation of ellipse axes ratio

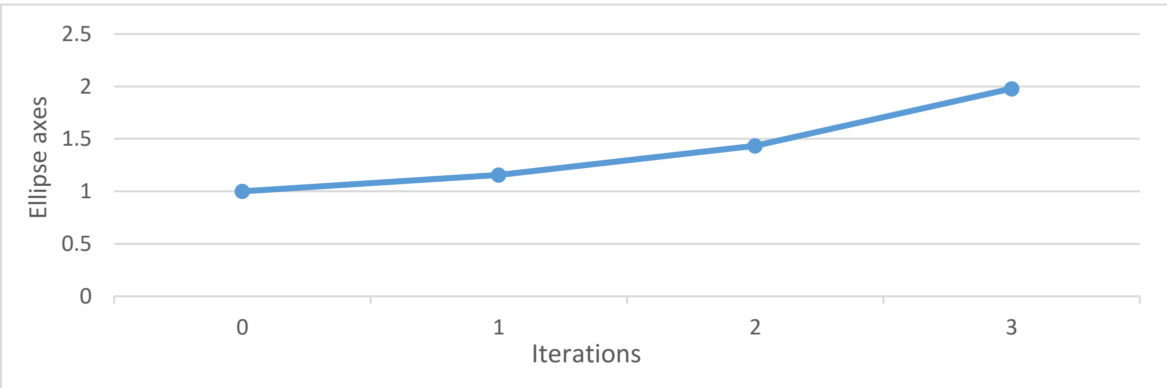
Figure 4.6: Optimization results of a plate with a hole (D=10 mm, K = 275, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

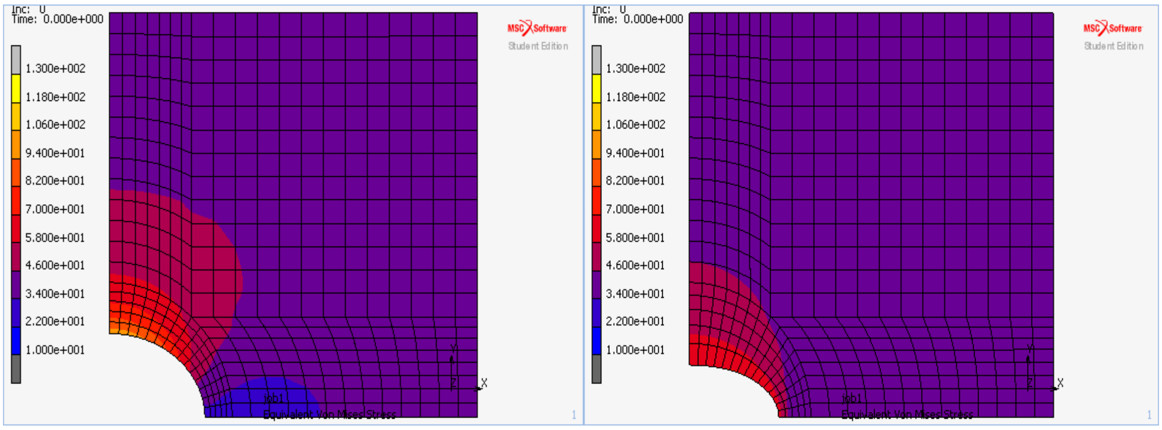


(b) Variation of von Mises stress distribution along the hole boundary

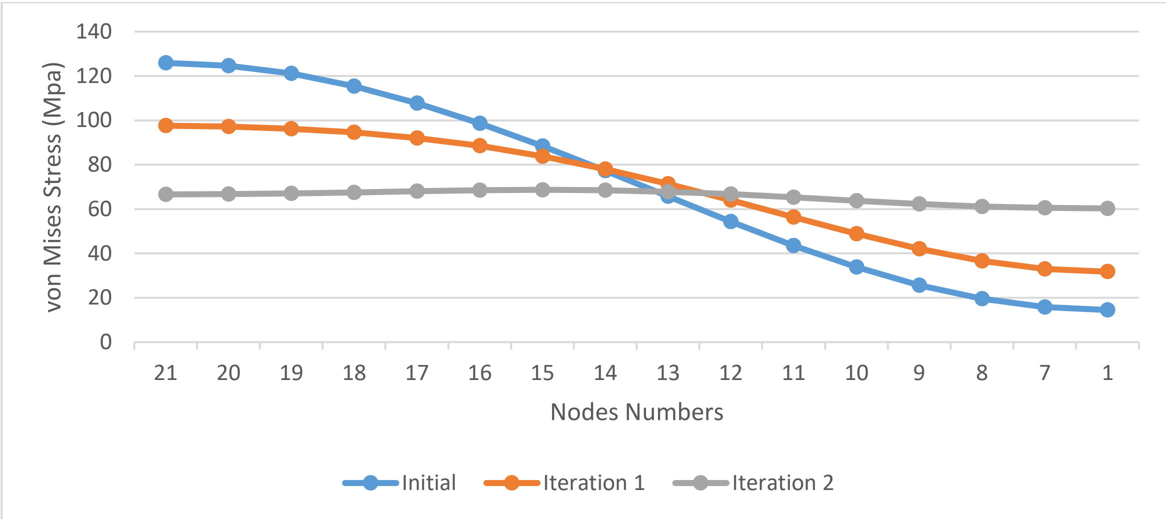


(c) Variation of ellipse axes ratio

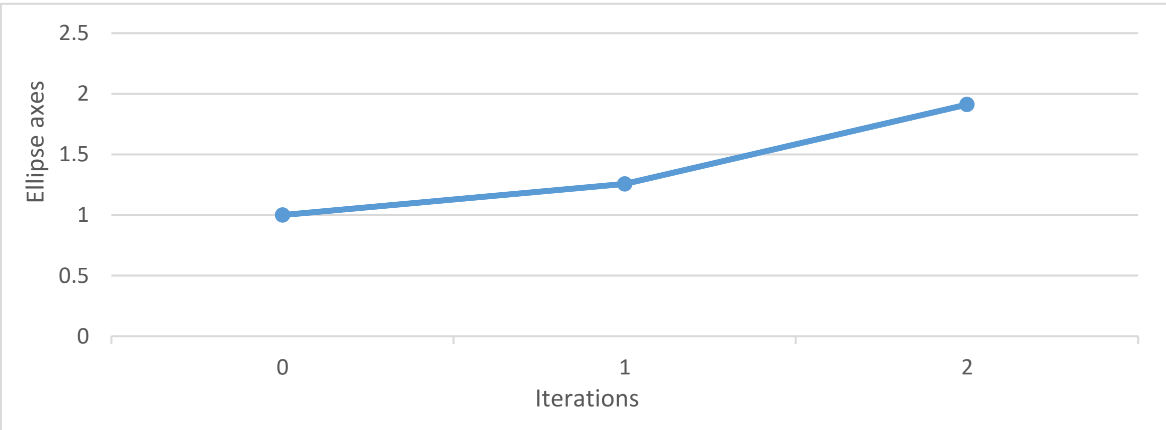
Figure 4.7: Optimization results of a plate with a hole (D=10 mm, K = 500, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

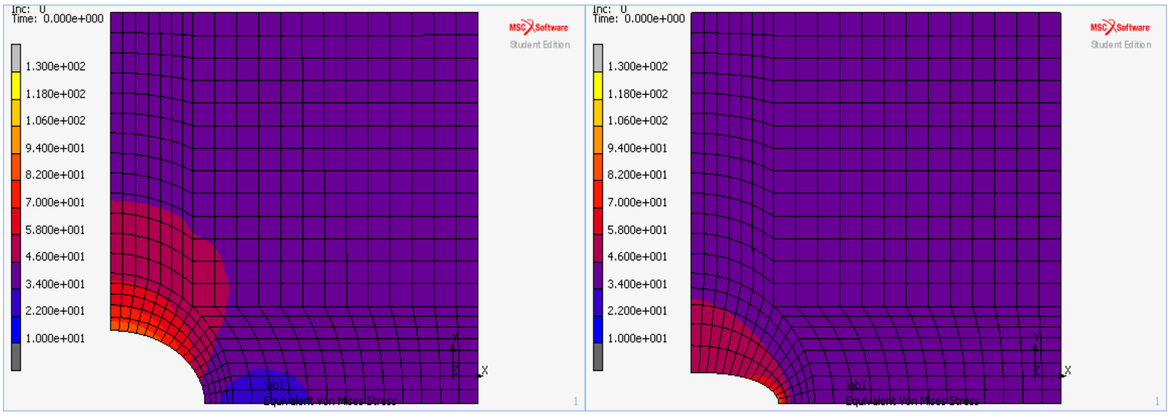


(b) Variation of von Mises stress distribution along the hole boundary

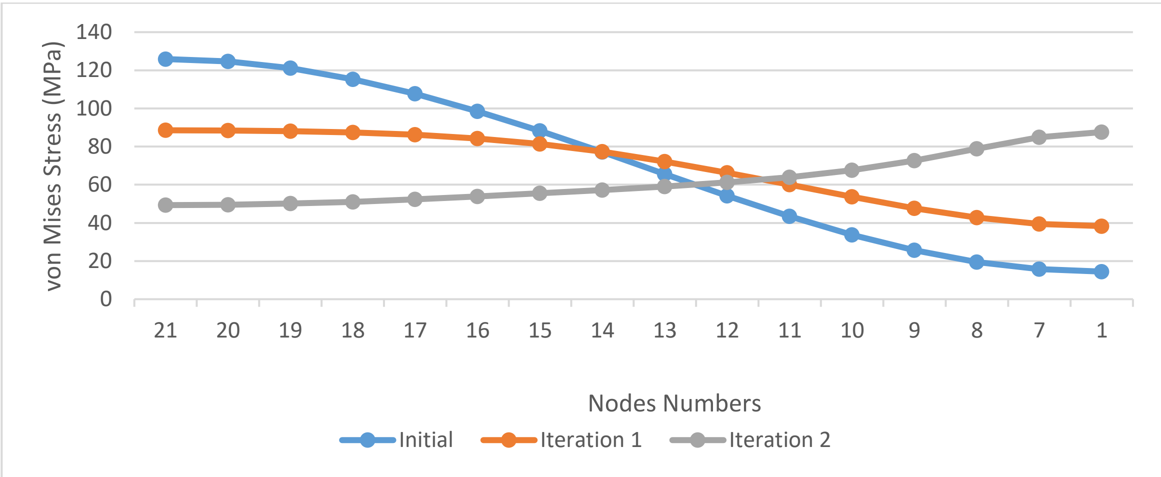


(c) Variation of ellipse axes ratio

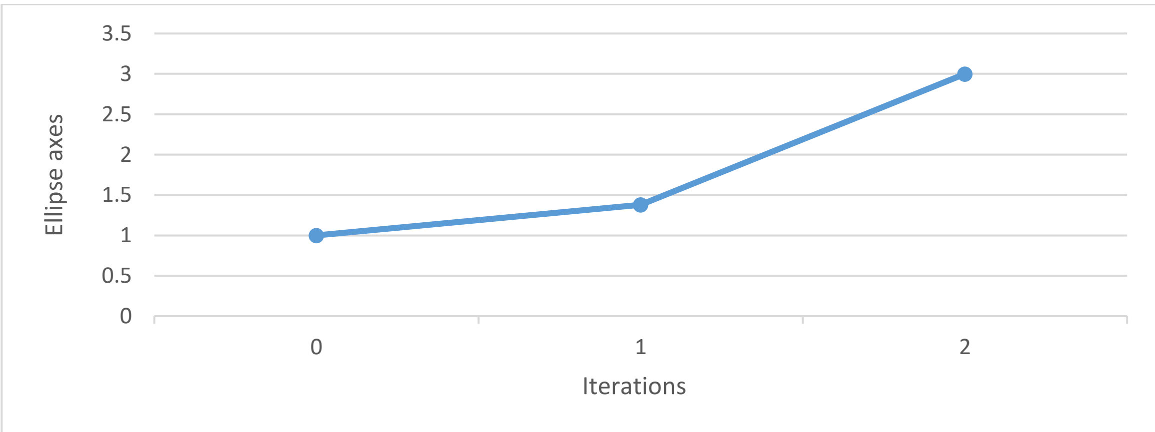
Figure 4.8: Optimization results of a plate with a hole (D=10 mm, K = 750, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 2 (right)

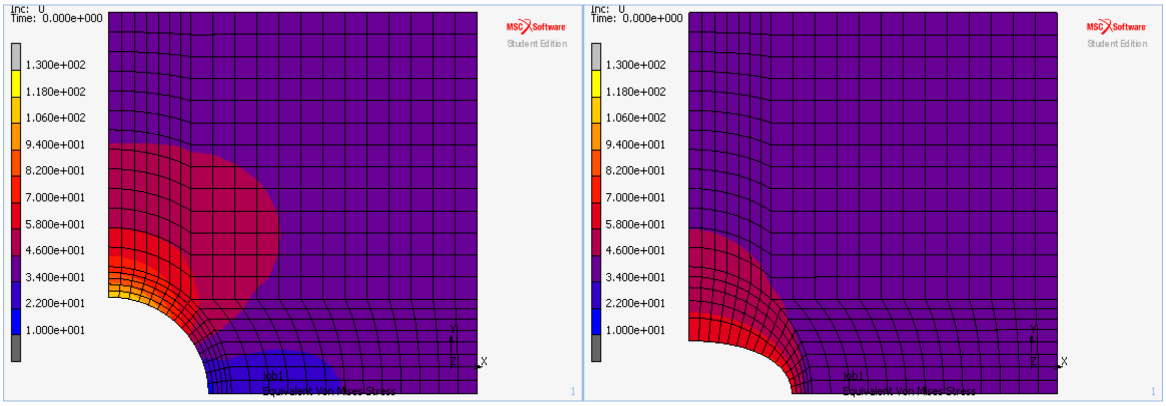


(b) Variation of von Mises stress distribution along the hole boundary

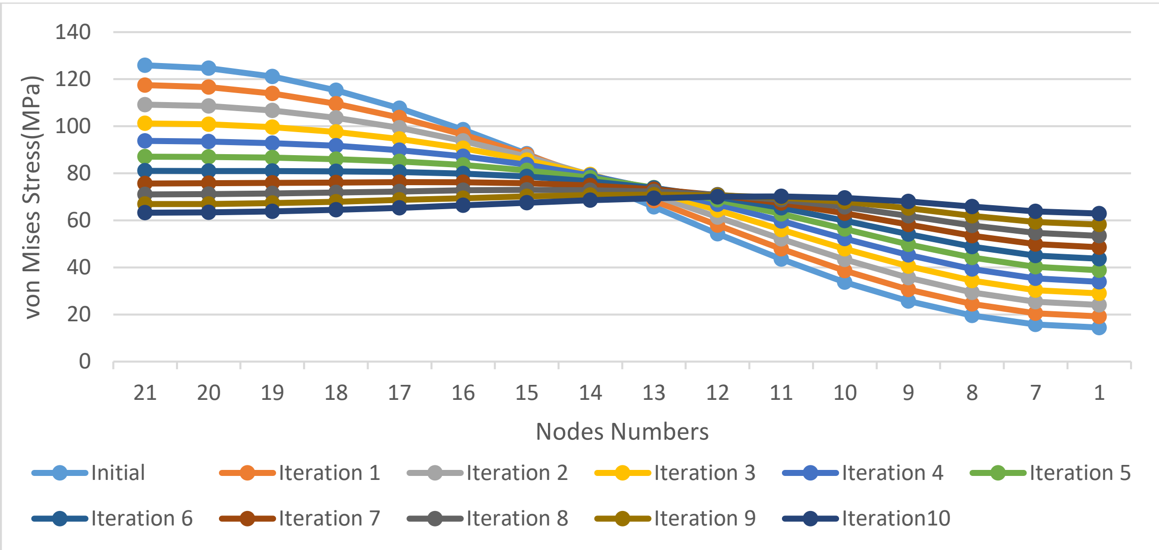


(c) Variation of ellipse axes ratio

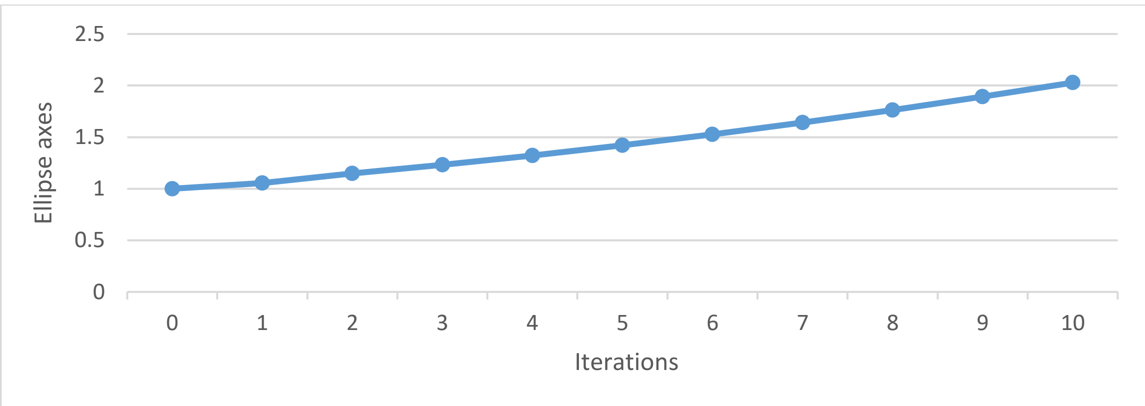
Figure 4.9: Optimization results of a plate with a hole (D=10 mm, K=1000, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 10 (right)

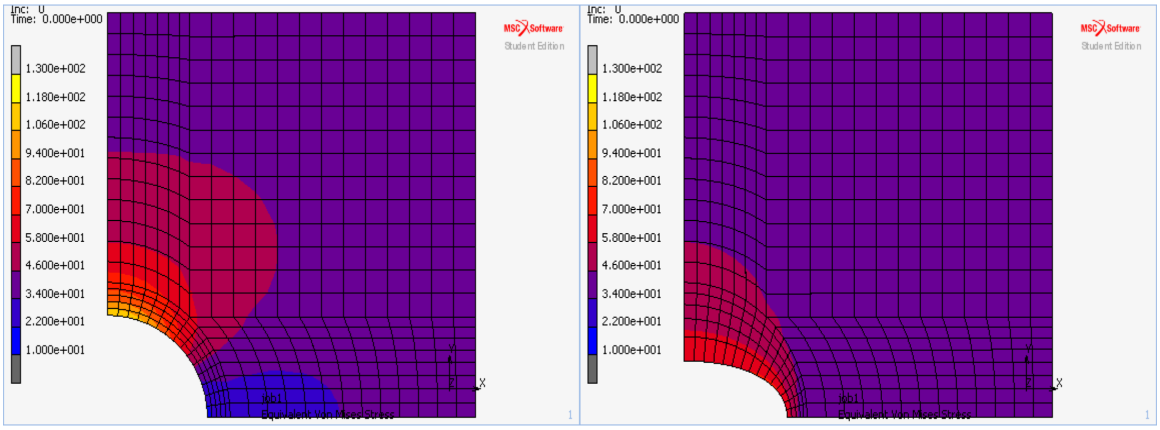


(b) Variation of von Mises stress distribution along the hole boundary

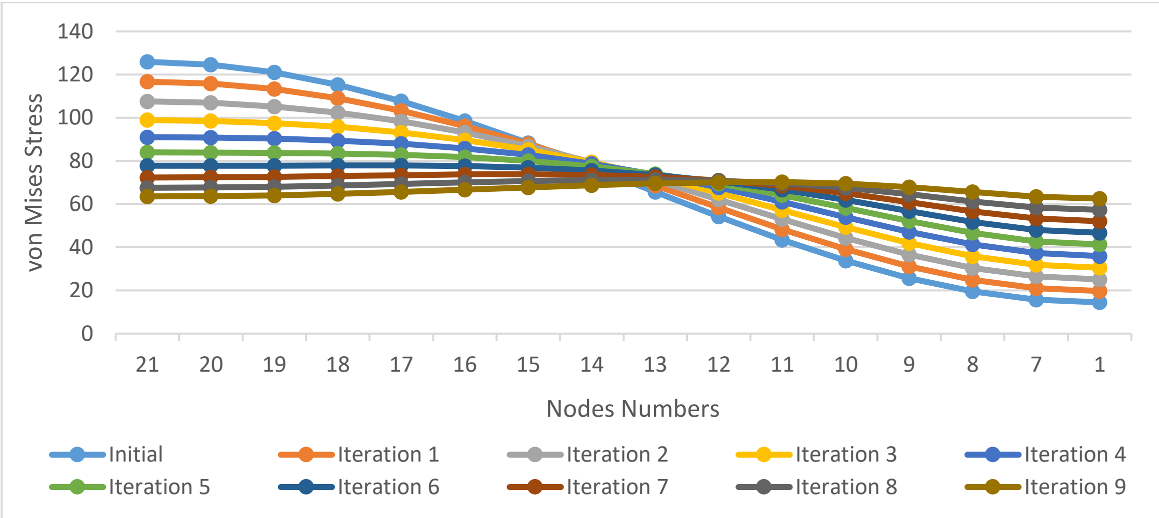


(c) Variation of ellipse axes ratio

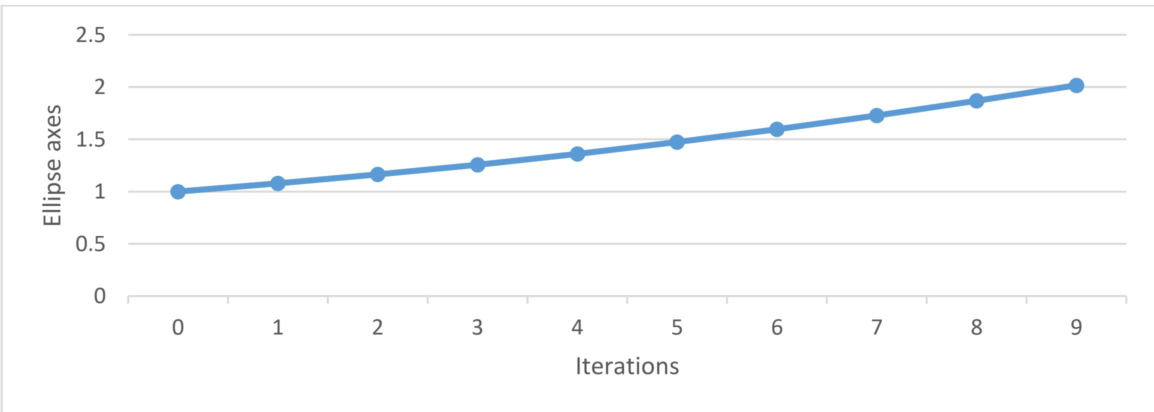
Figure 4.10: Optimization results of a plate with a hole (D=10 mm, K=250, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 9 (right)

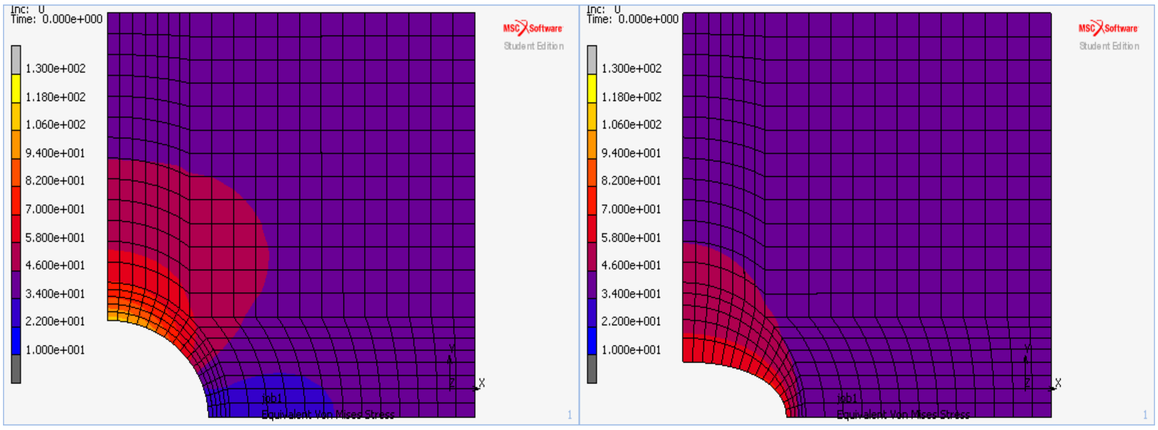


(b) Variation of von Mises stress distribution along the hole boundary

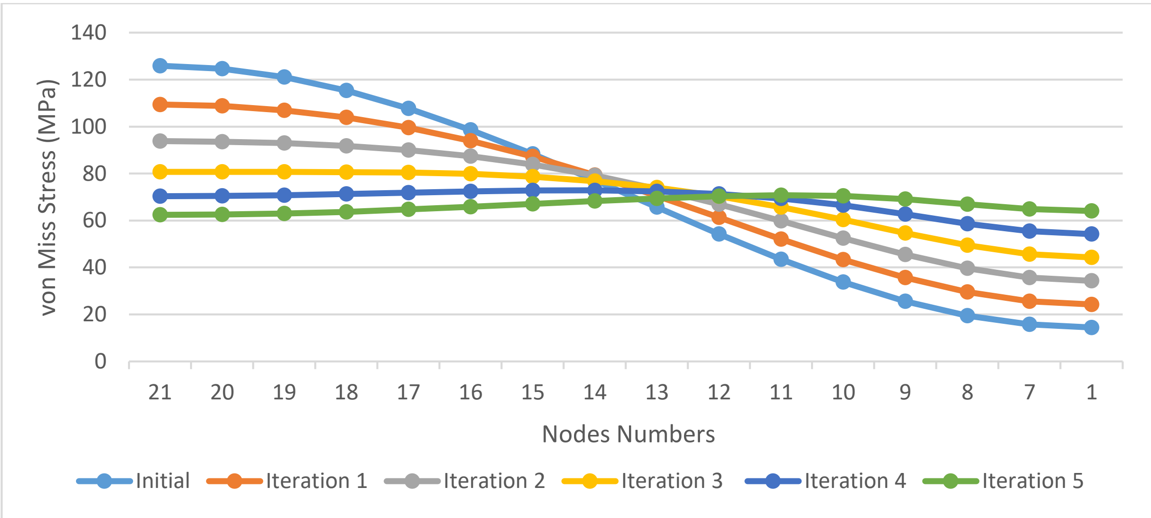


(c) Variation of ellipse axes ratio

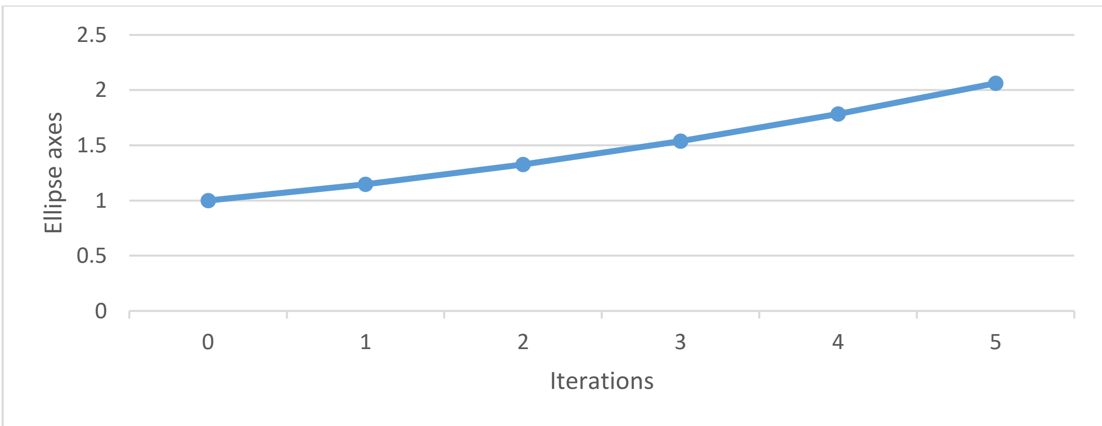
Figure 4.11: Optimization results of a plate with a hole (D=10 mm, K=275, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)

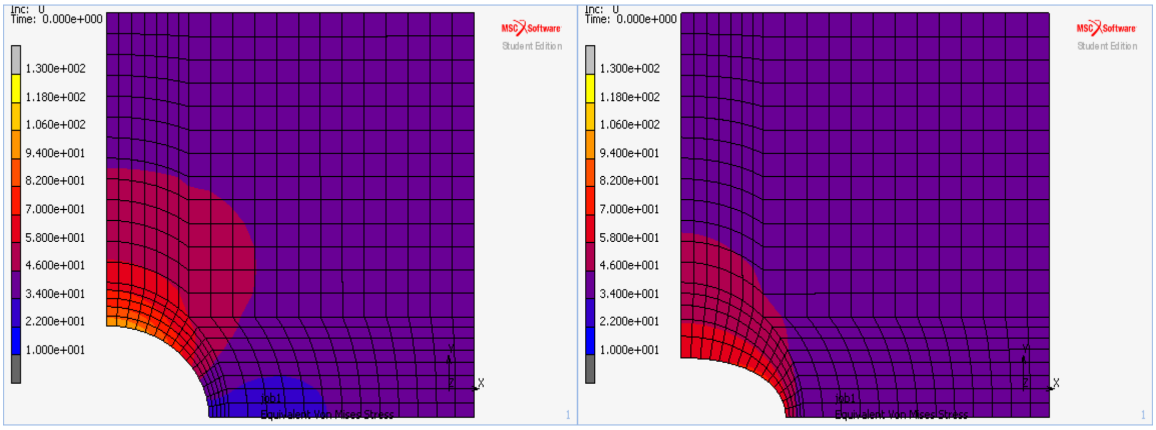


(b) Variation of von Mises stress distribution along the hole boundary

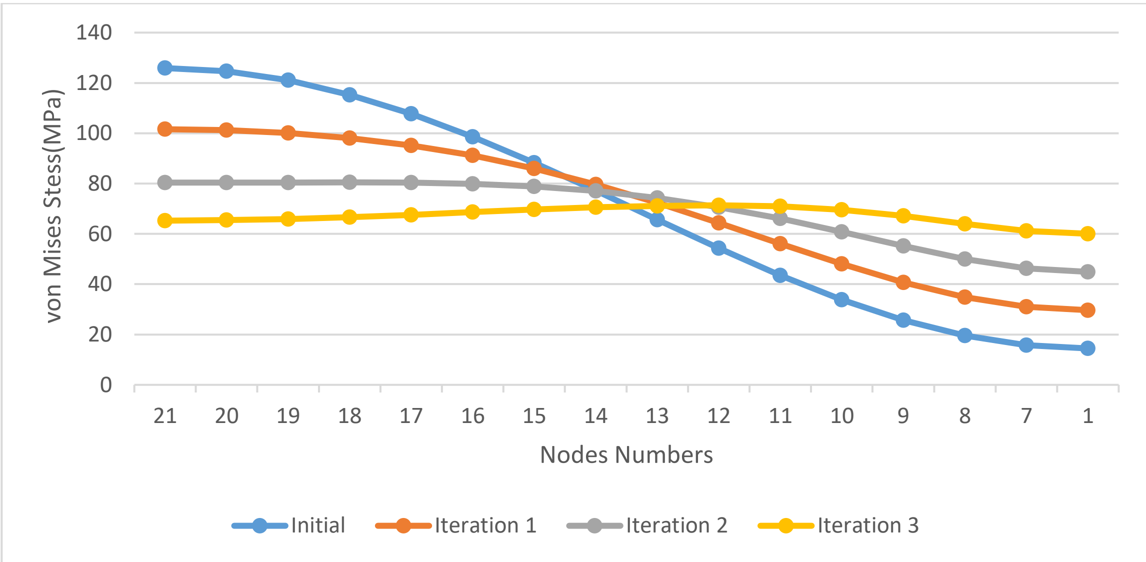


(c) Variation of ellipse axes ratio

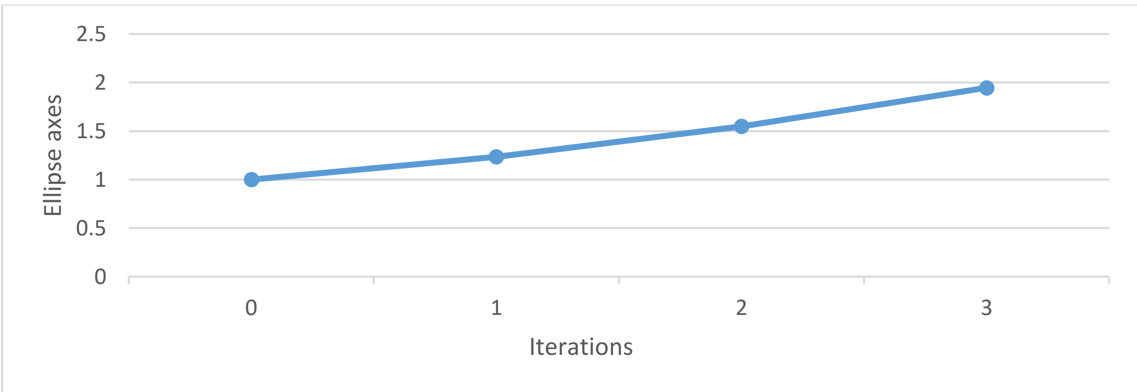
Figure 4.12: Optimization results of a plate with a hole (D=10 mm, K=500, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

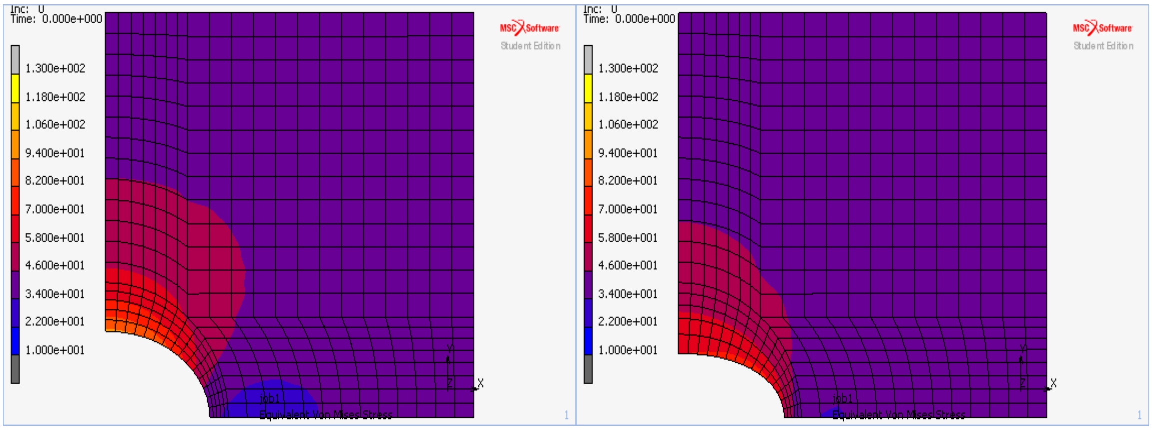


(b) Variation of von Mises stress distribution along the hole boundary

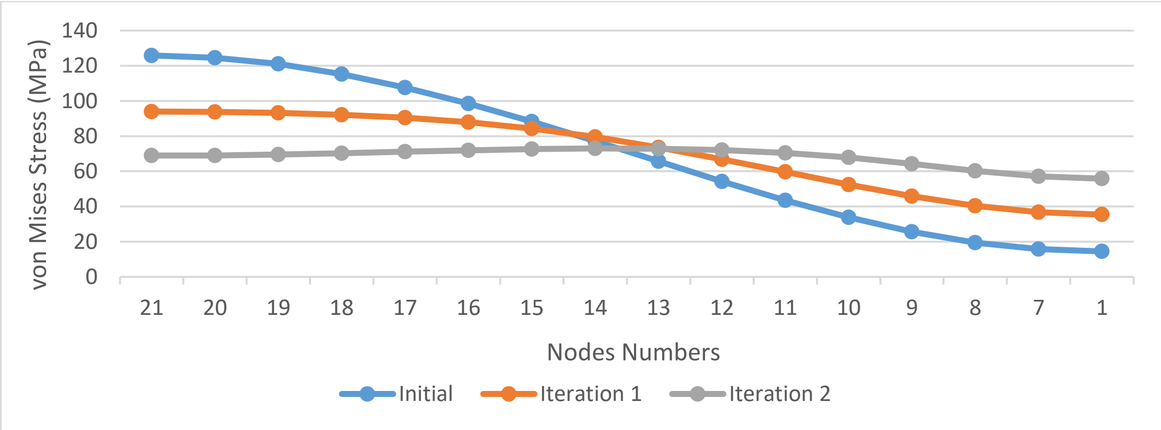


(c) Variation of ellipse axes ratio

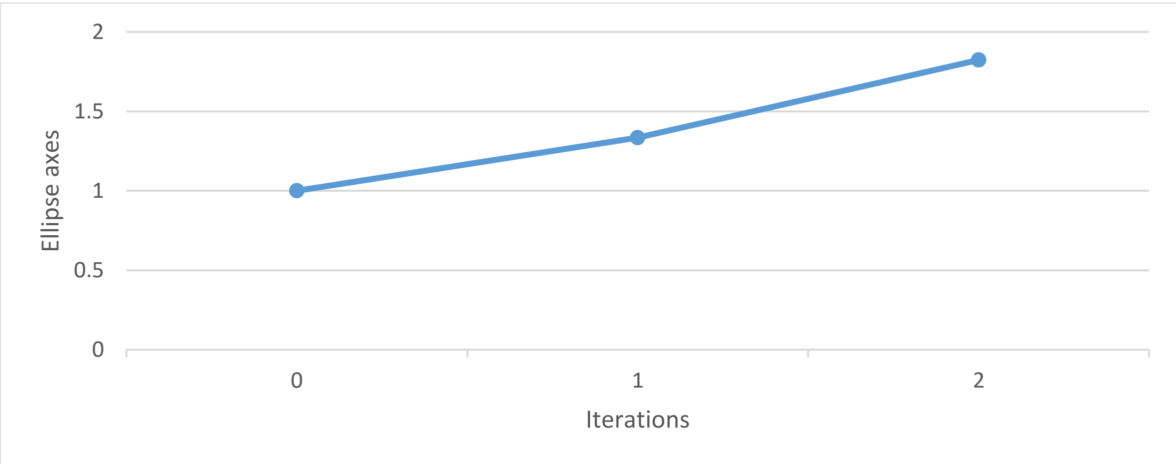
Figure 4.13: Optimization results of a plate with a hole (D=10 mm, K=750, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 2 (right)

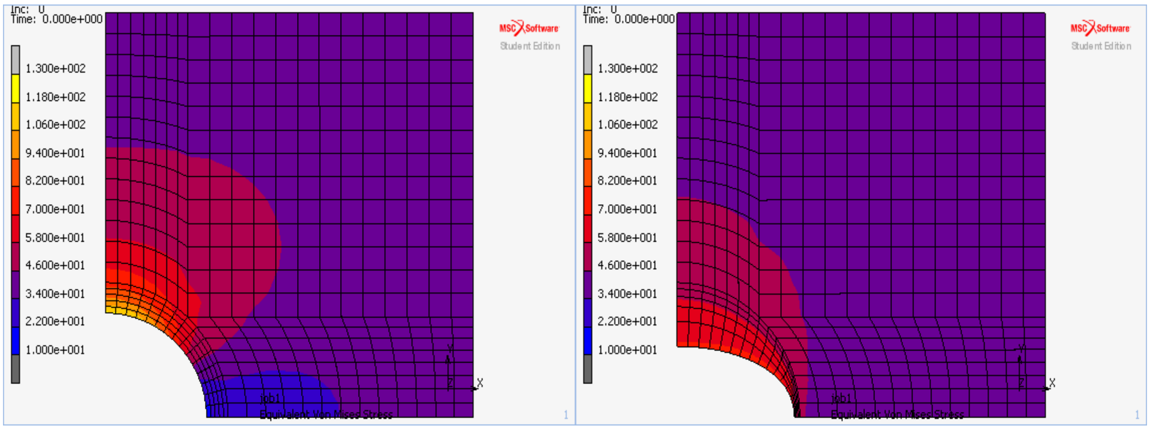


(b) Variation of von Mises stress distribution along the hole boundary

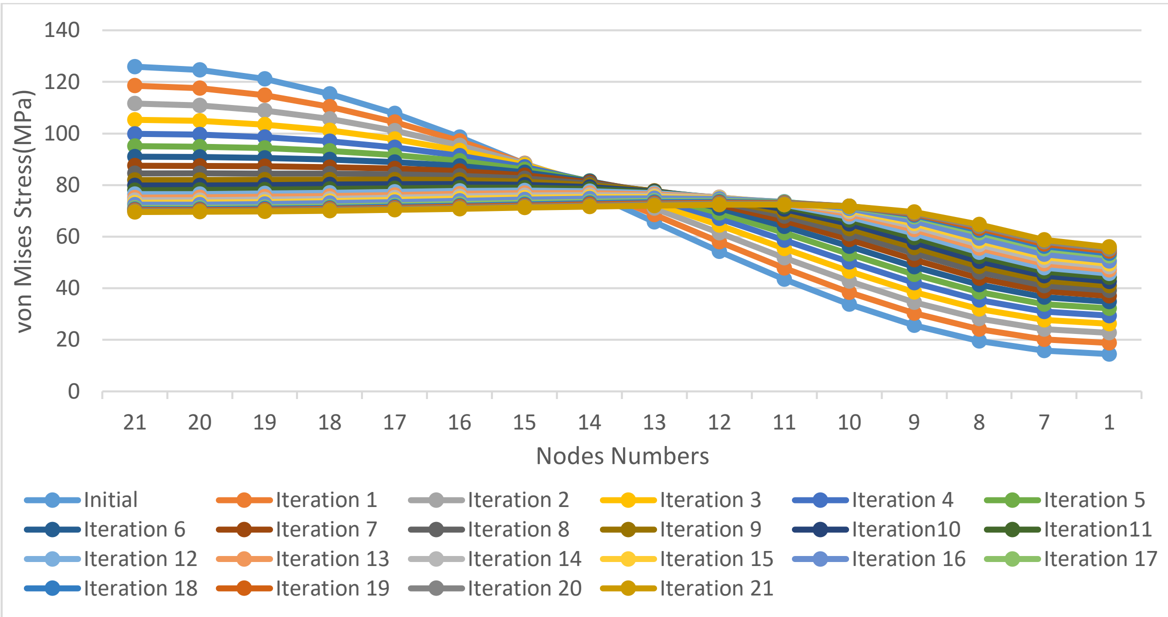


(c) Variation of ellipse axes ratio

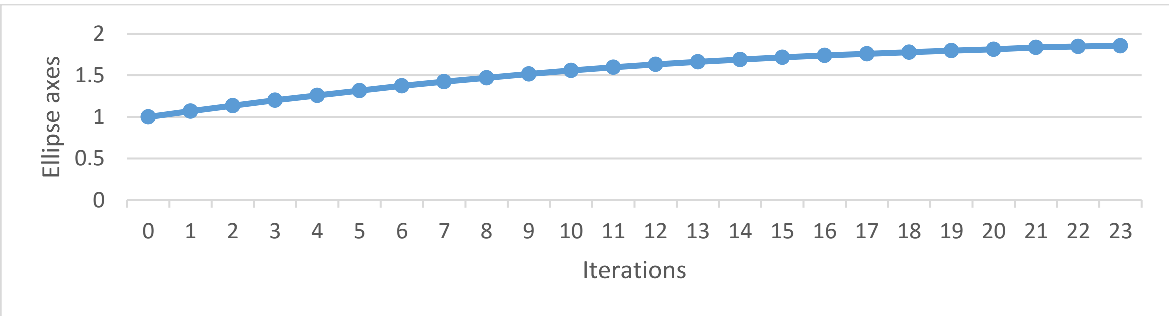
Figure 4.14: Optimization results of a plate with a hole (D=10 mm, K=1000, σ_{ref}=40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 21 (right)

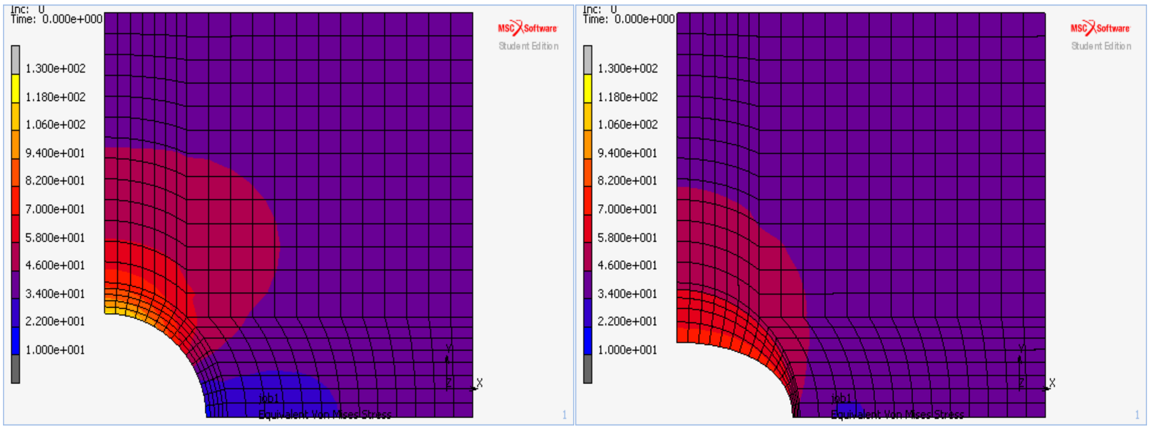


(b) Variation of von Mises stress distribution along the hole boundary

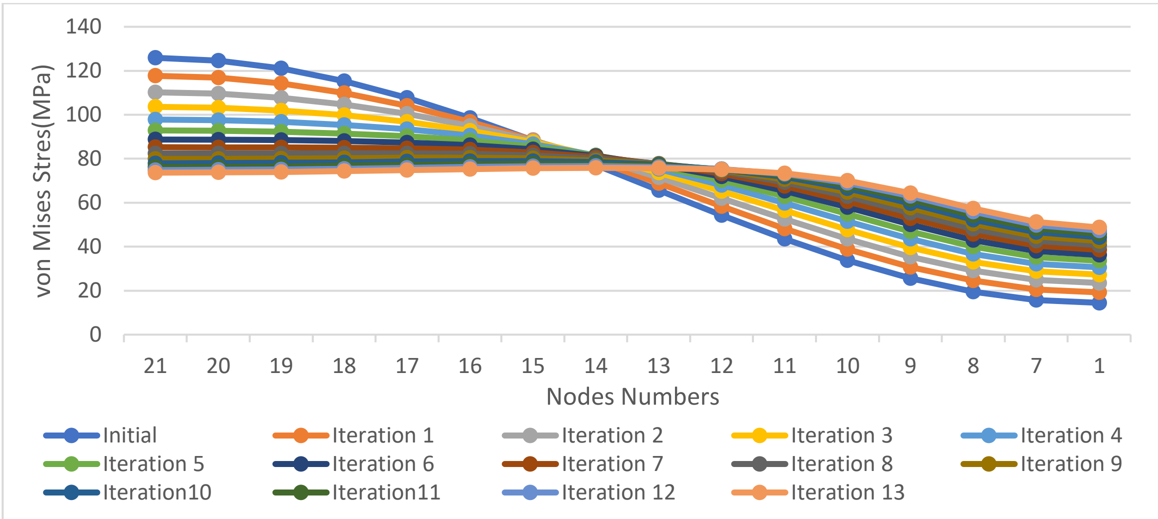


(c) Variation of ellipse axes ratio

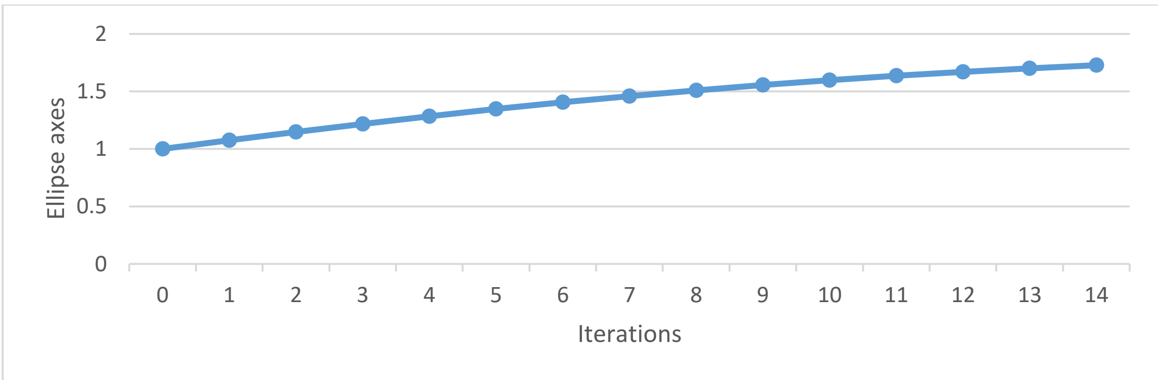
Figure 4.15: Optimization results of a plate with a hole (D=10 mm, K=250, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 13 (right)

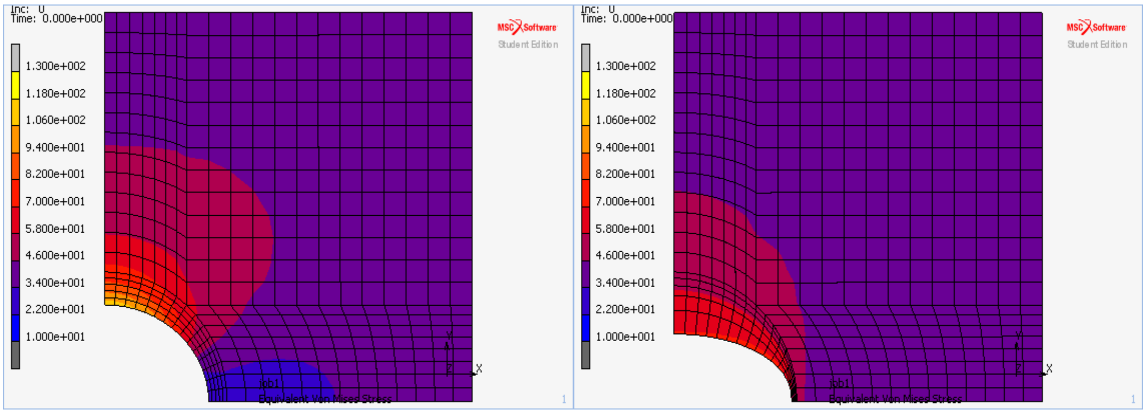


(b) Variation of von Mises stress distribution along the hole boundary

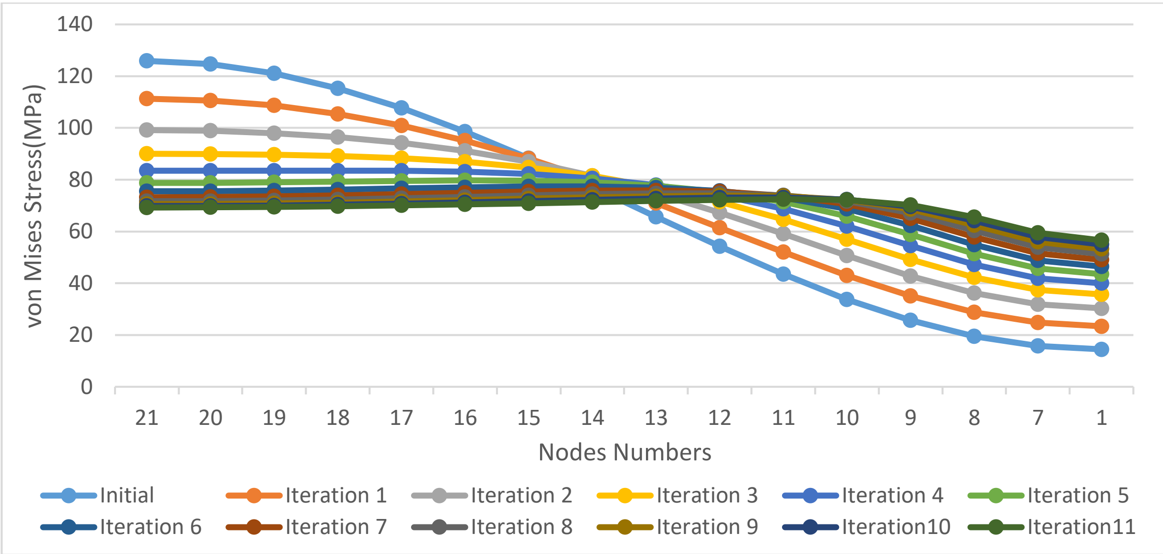


Variation of ellipse axes ratio

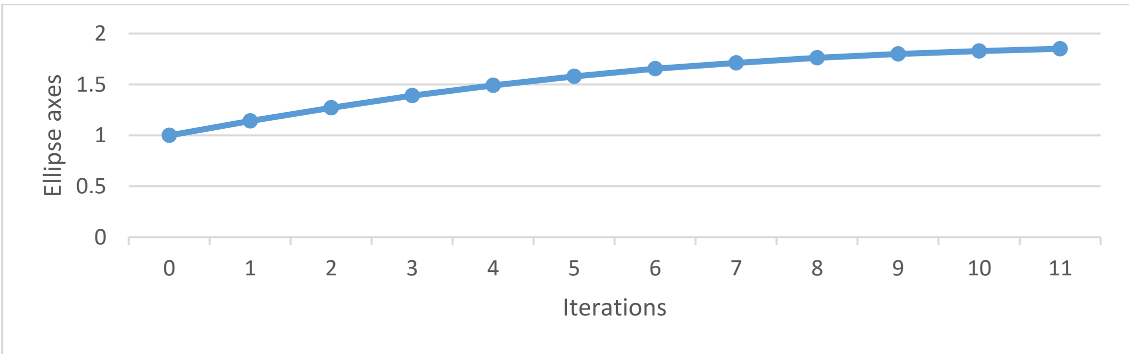
Figure 4.16: Optimization results of a plate with a hole (D=10 mm, K=275, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 11 (right)

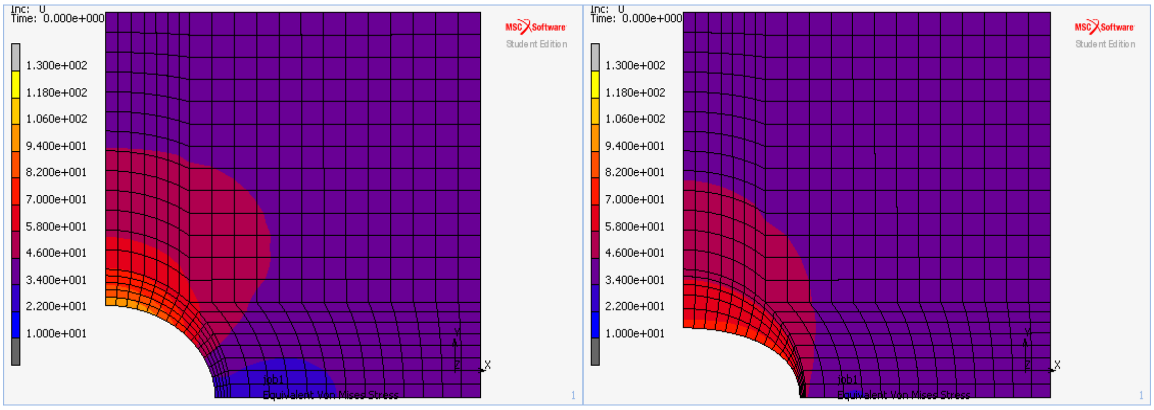


(b) Variation of von Mises stress distribution along the hole boundary

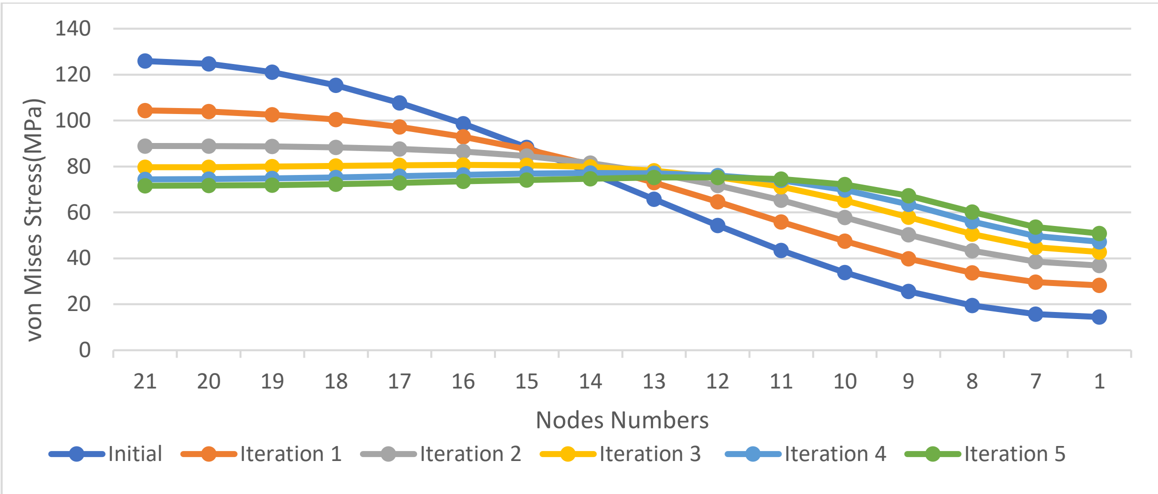


(c) Variation of ellipse axes ratio

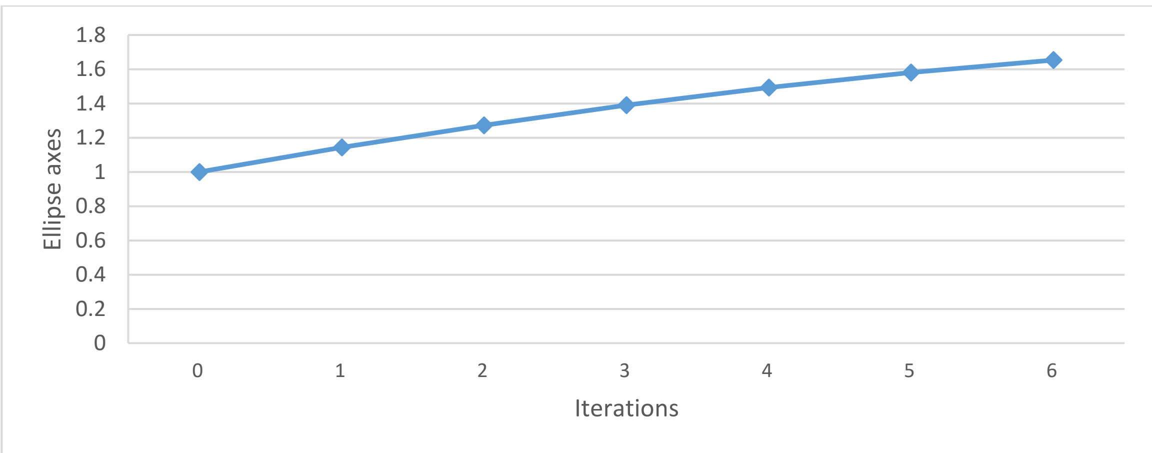
Figure 4.17: Optimization results of a plate with a hole (D=10 mm, K=500, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)

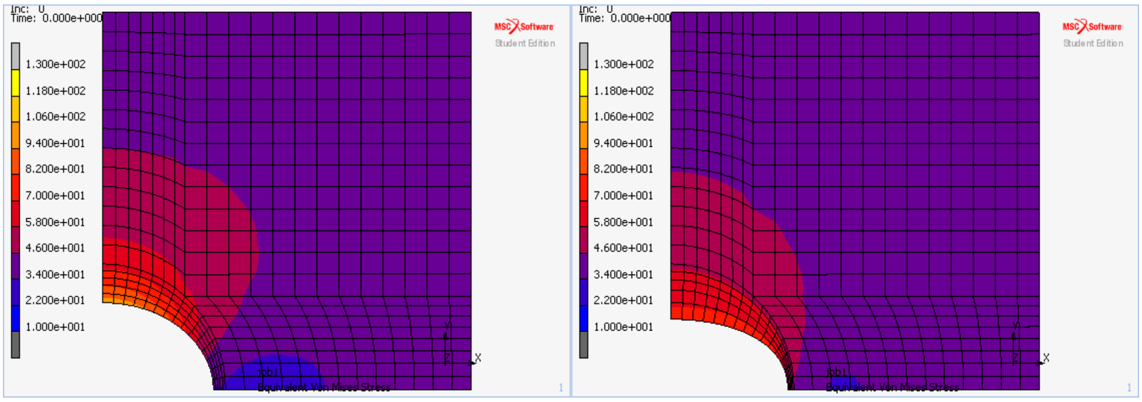


(b) Variation of von Mises stress distribution along the hole boundary

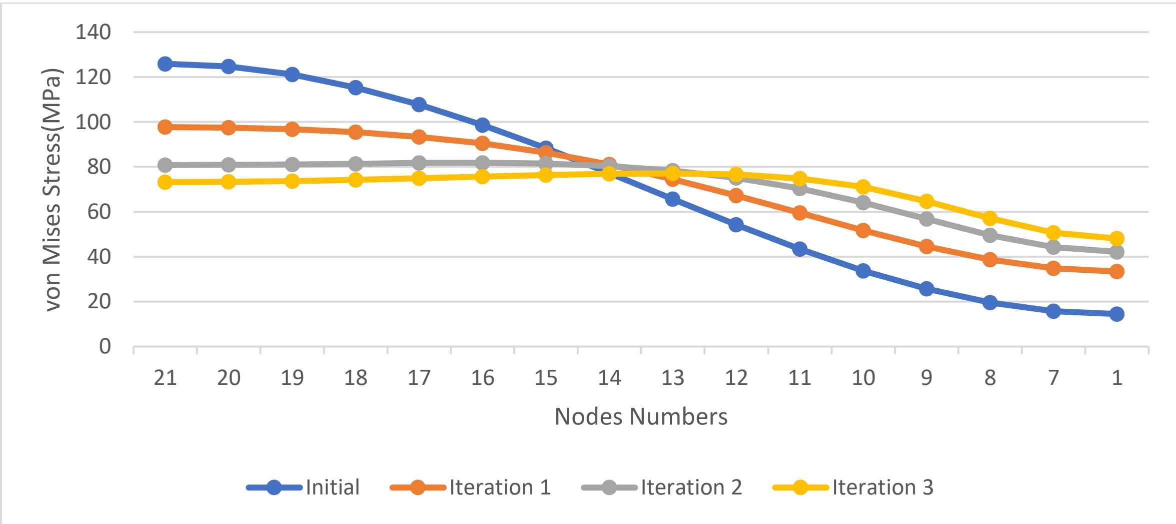


(c) Variation of ellipse axes ratio

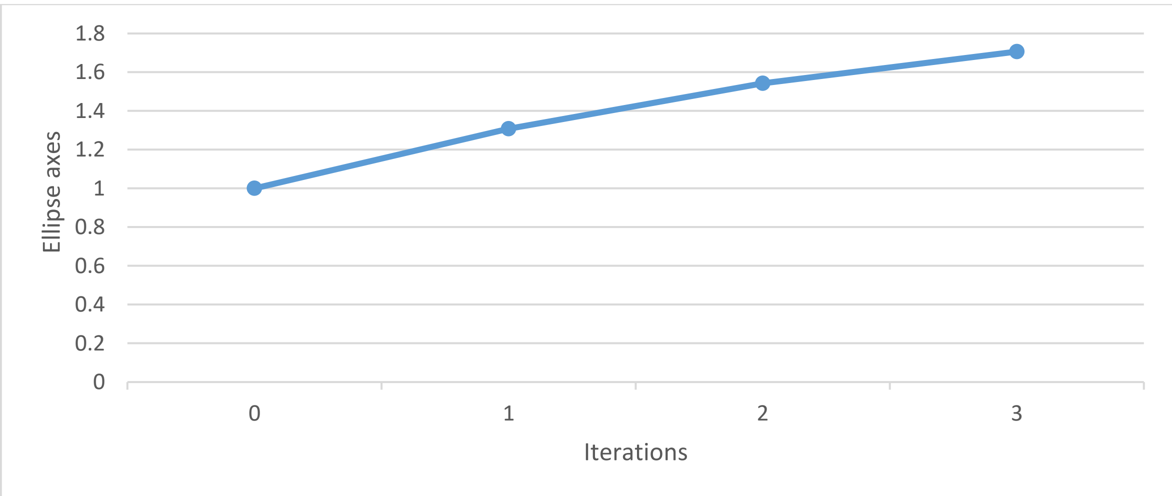
Figure 4.18: Optimization results of a plate with a hole (D=10 mm, K=750, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)



(b) Variation of von Mises stress distribution along the hole boundary



(c) Variation of ellipse axes ratio

Figure 4.19: Optimization results of a plate with a hole (D=10 mm, K=1000, σ_{ref}=60 MPa)

Table 4.2: Summary and comparison of results with domain thickness 10mm

		Magnification factor k									
σ_{ref}		250		275		500		750		1000	
		AY	ET	AY	ET	AY	ET	AY	ET	AY	ET
10	Iterations	6	7	6	4	3	4	2	--	2	--
	Max stress	65	67.2	69.22	67.2	67.02	69.2	68.69	--	87.62	--
	Ellipse axes ratio	2.05	2.03	1.99	2.1	1.97	1.91	1.91	--	2.9	--
40	Iterations	10	12	9	9	5	6	3	3	2	2
	Max stress	70.18	69.2	70.22	70	70.78	69.2	71.37	70	71.04	70.4
	Ellipse axes ratio	2.02	1.94	2.01	1.96	2.06	1.99	1.94	1.77	1.84	1.73
60	Iterations	21	19	13	13	11	8	5	5	3	4
	Max stress	71.14	71.6	72.15	71.2	71.31	70.8	71.98	70.8	70.27	71.6
	Ellipse axes ratio	1.85	1.97	1.82	1.94	1.85	1.94	1.90	2.00	1.80	1.99

AY: Ayoub Yahya, ET: Erman Tekkaya

4.2 Optimization of a Plate with a Hole with Domain Thickness 20 mm

The finite element models and the boundary conditions for stress and thermal analysis are shown in Figure 4.20 for domain thickness 20 mm. The von Mises stresses of stress analysis and total deflections of the thermal expansion analysis of the original shape can be seen in Figure 4.21. Similar to the analysis conducted for domain thickness 10 mm, the stress far away from the concentration zones is about 40 MPa. The maximum and minimum von Mises stresses were also around 130 MPa and 10 MPa respectively.

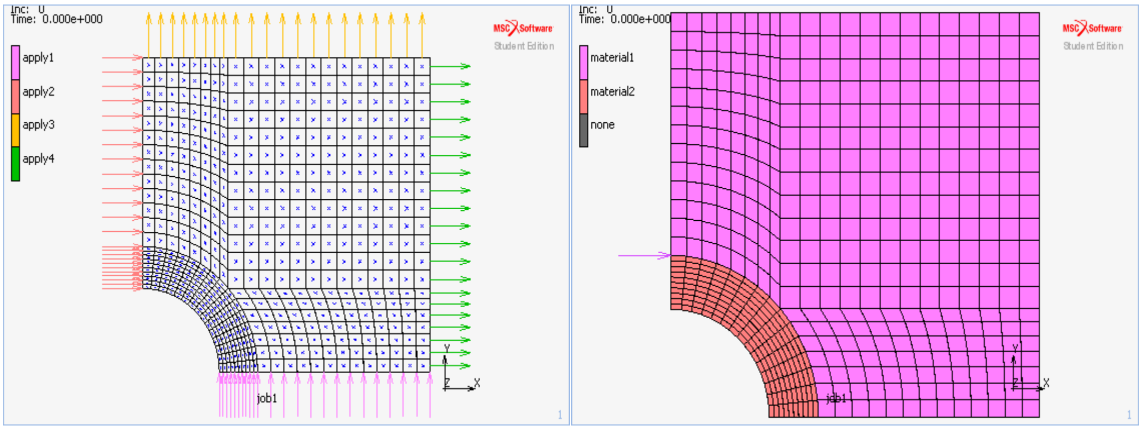


Figure 4.20: Finite element model for stress (left) and thermal (right) analysis (D=20mm)

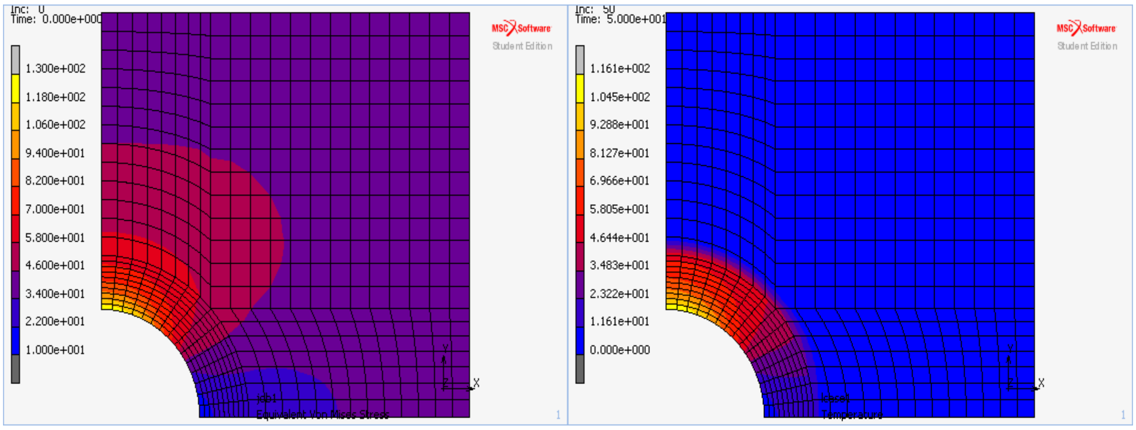
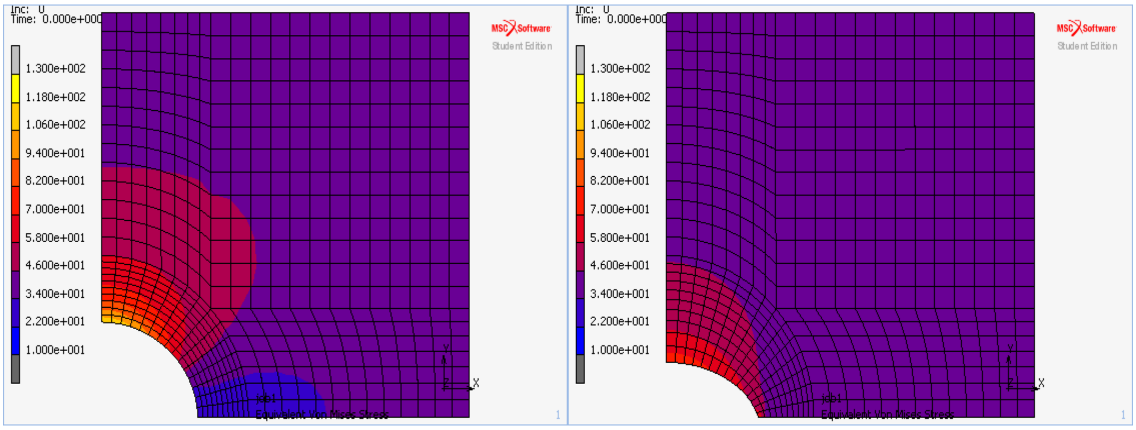
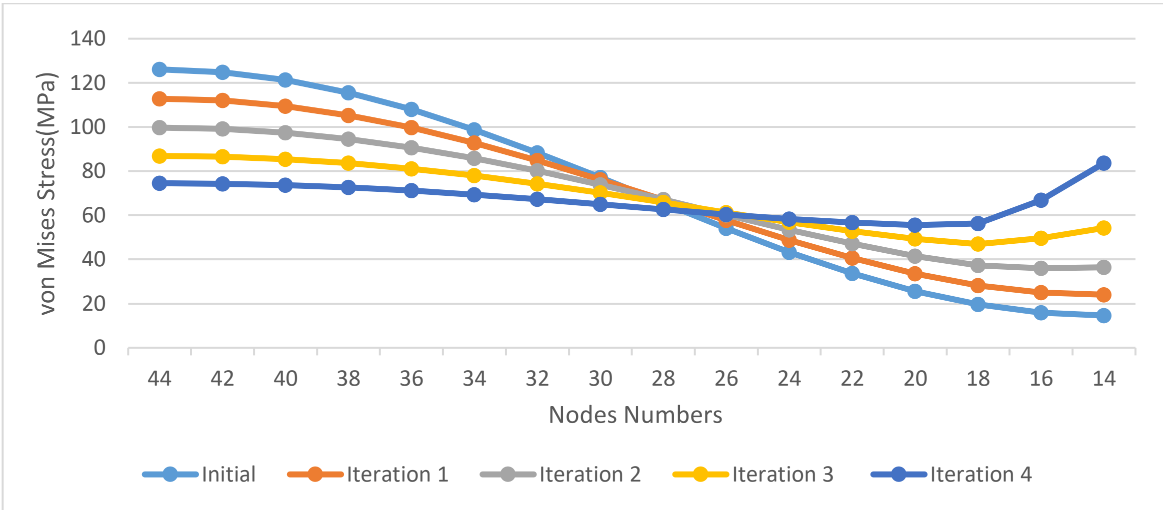


Figure 4.21: von Mises stress (left) and thermal deformations (right) of the original shape (D =20mm)

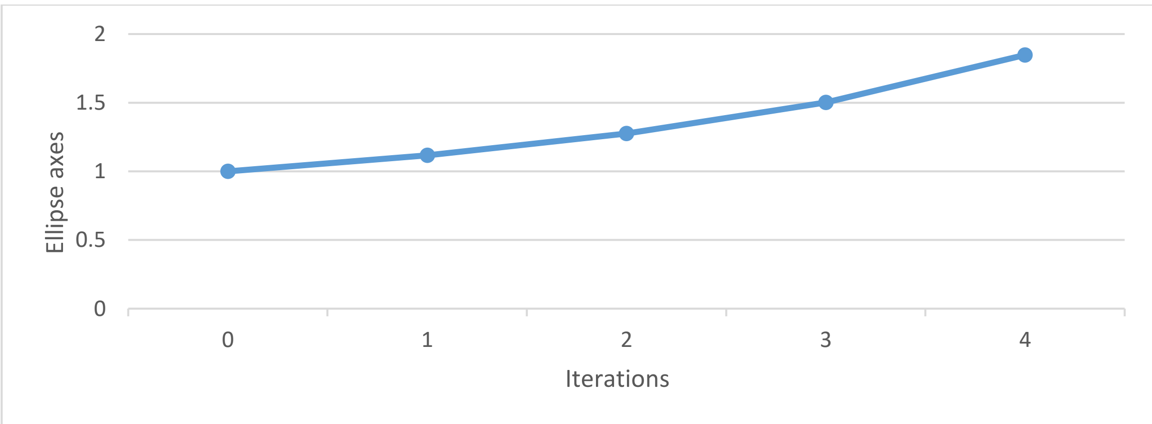
For domain thickness D = 20 mm, twelve optimization analyses were conducted including the combinations of reference stresses σ_{ref} = 10, 40 and 60 MPa and magnification factor k = 250, 275, 500, 750 and 1000. The results obtained including (a) von Mises stress distributions of the plate after first and last iterations, (b) the change of von Mises stresses by iterations along the hole boundary, (c) the change of ellipse axes ratio by iterations are presented in Figures 4.22 to 4.33. A numerical dis-order was observed almost in all of the analyses towards the end of the iterations at the side of the ellipse on the x-axis. However, reasonable results were obtained, close to the results of domain thickness 10 mm, except reference stresses σ_{ref} = 10 MPa as given in Table 4.4.



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 4 (right)

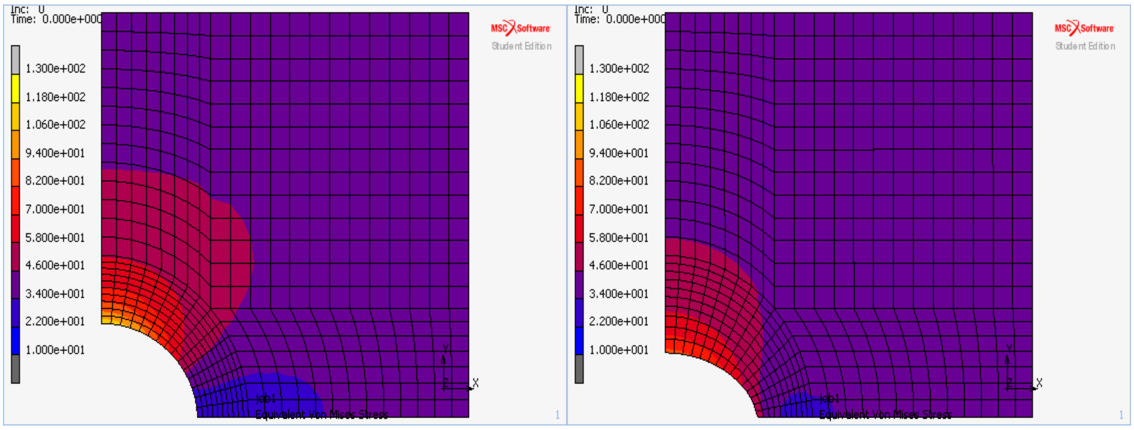


(b) Variation of von Mises stress distribution along the hole boundary

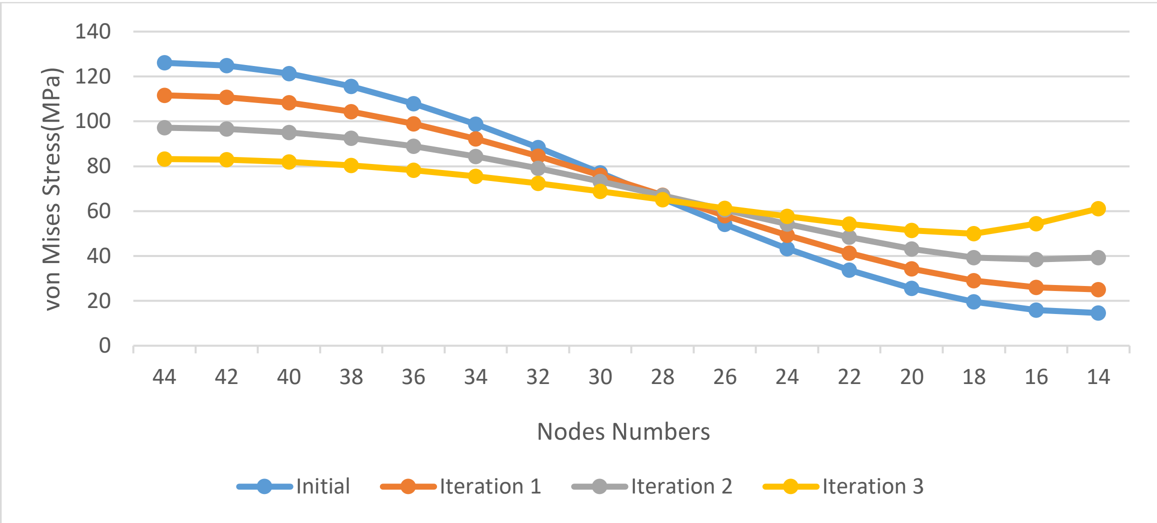


(c) Variation of ellipse axes ratio

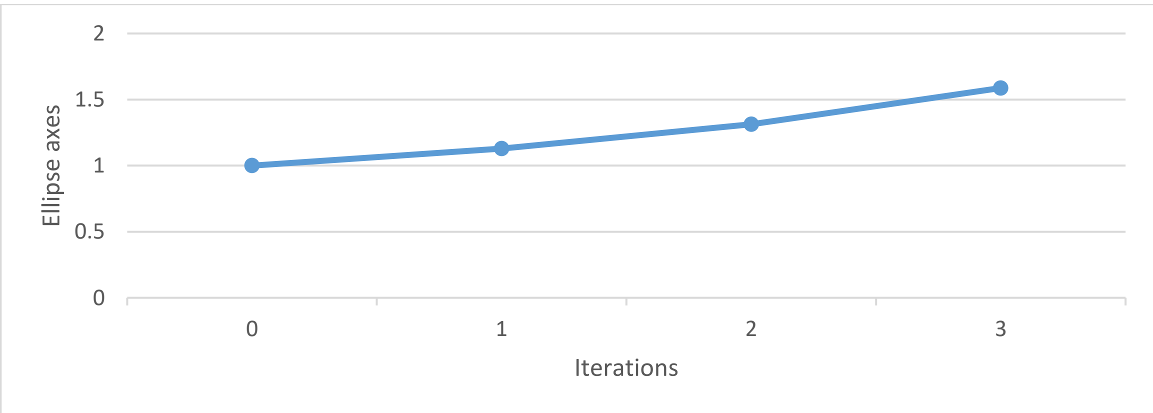
Figure 4.22: Optimization results of a plate with a hole (D=20 mm, K=250, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

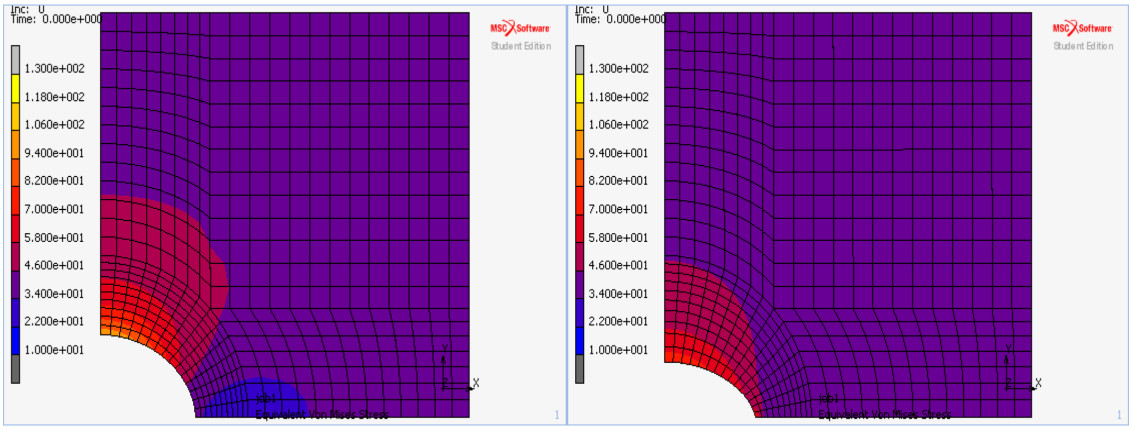


(b) Variation of von Mises stress distribution along the hole boundary

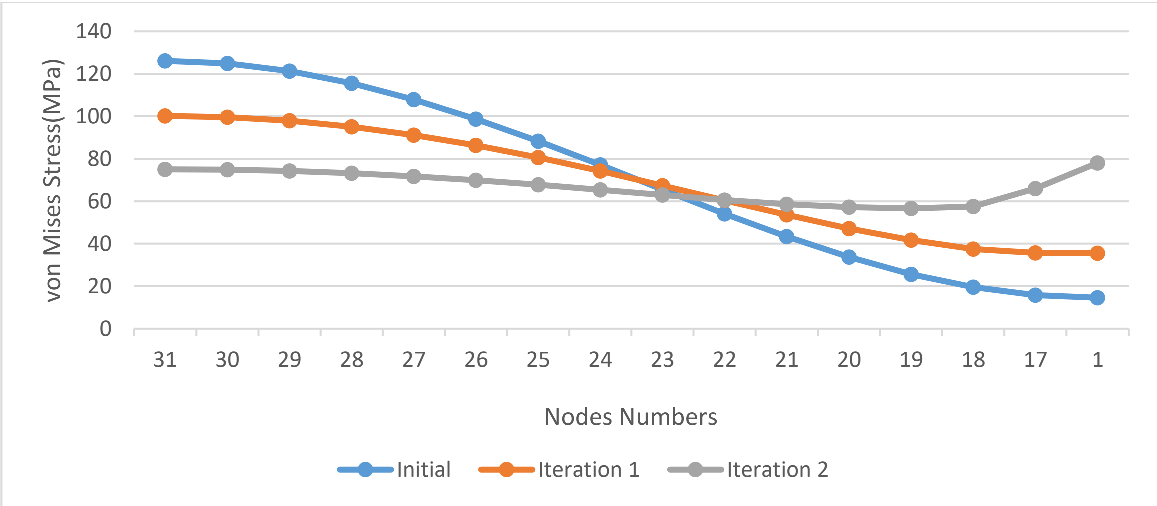


(c) Variation of ellipse axes ratio

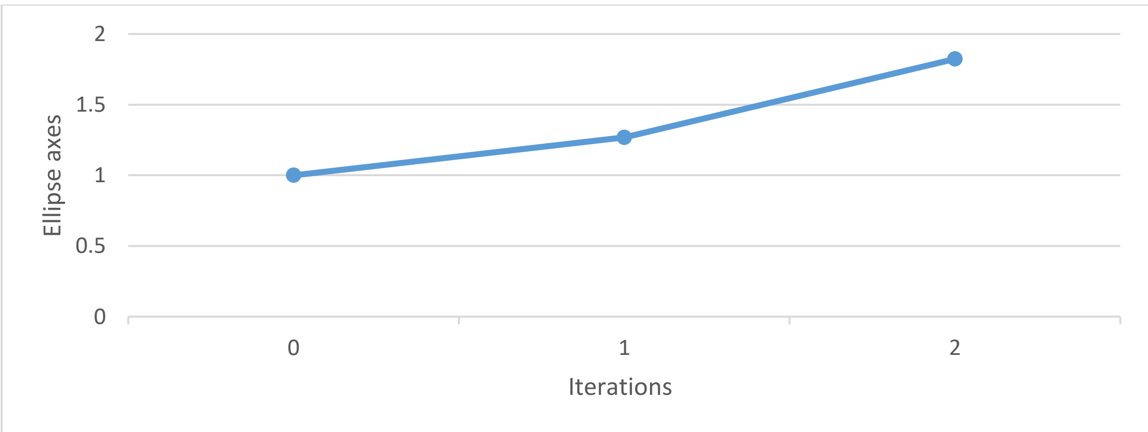
Figure 4.23: Optimization results of a plate with a hole (D=20 mm, K=275, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 2 (right)

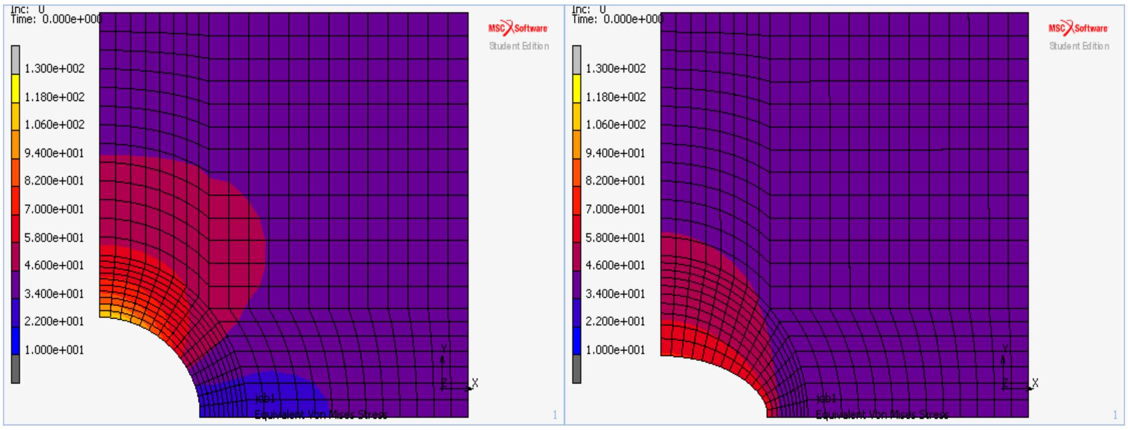


(b) Variation of von Mises stress distribution along the hole boundary

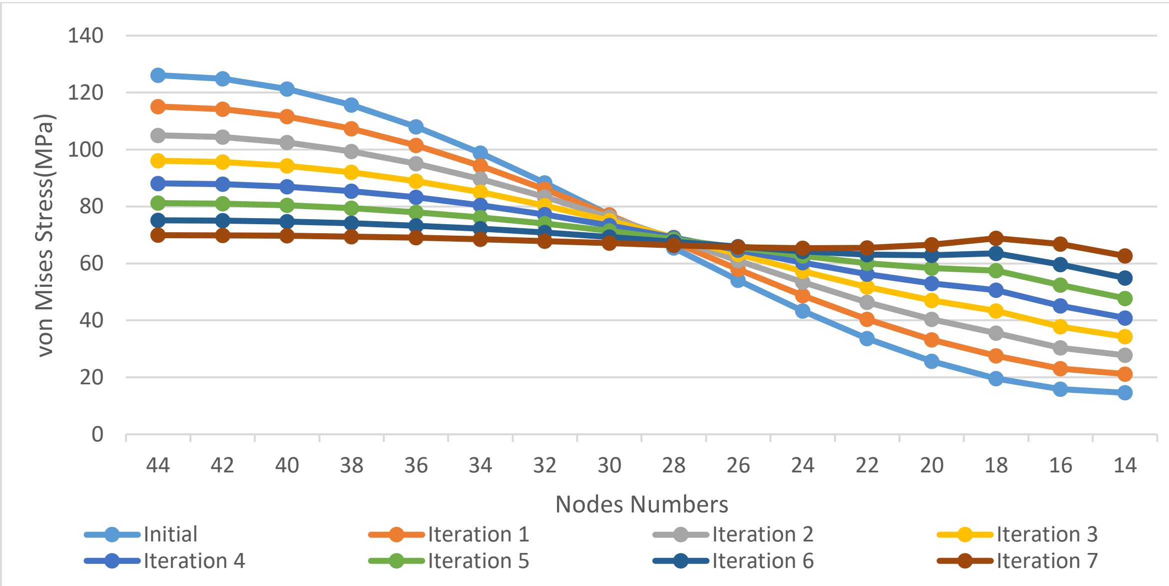


(c) Variation of ellipse axes ratio

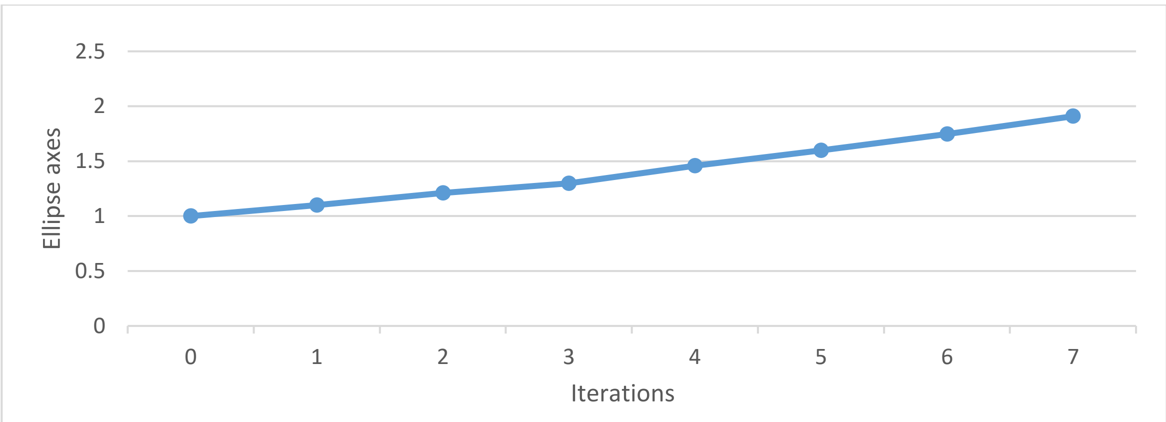
Figure 4.24: Optimization results of a plate with a hole (D=20 mm, K=500, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 7 (right)

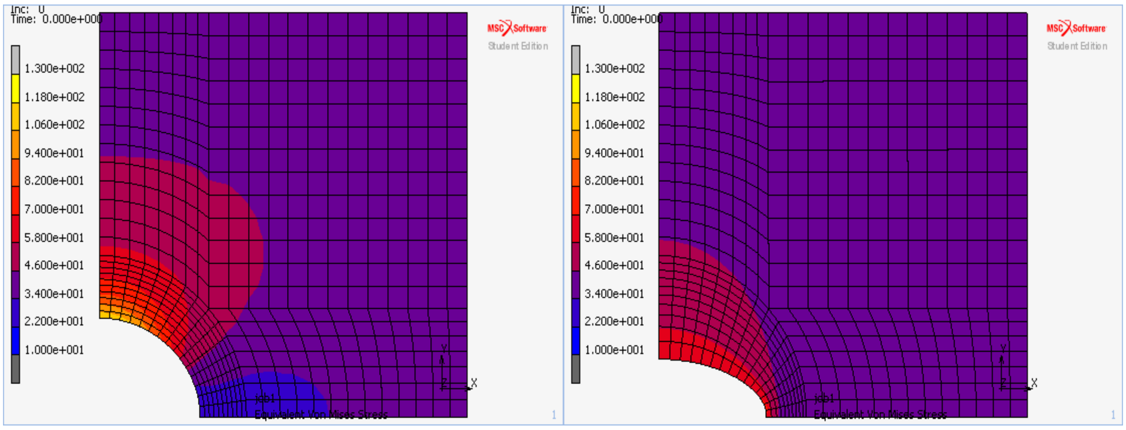


(b) Variation of von Mises stress distribution along the hole boundary

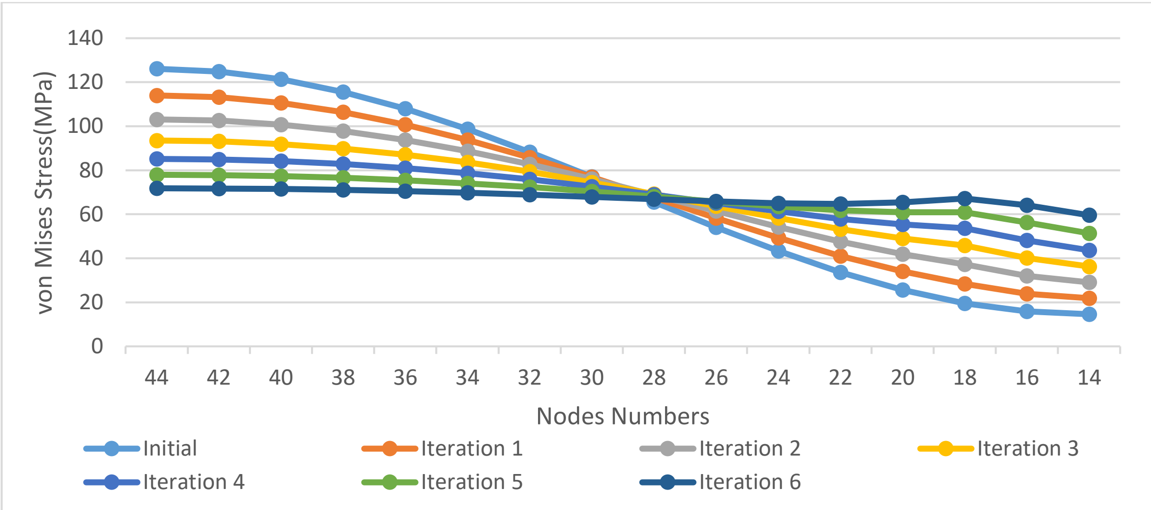


(c) Variation of ellipse axes ratio

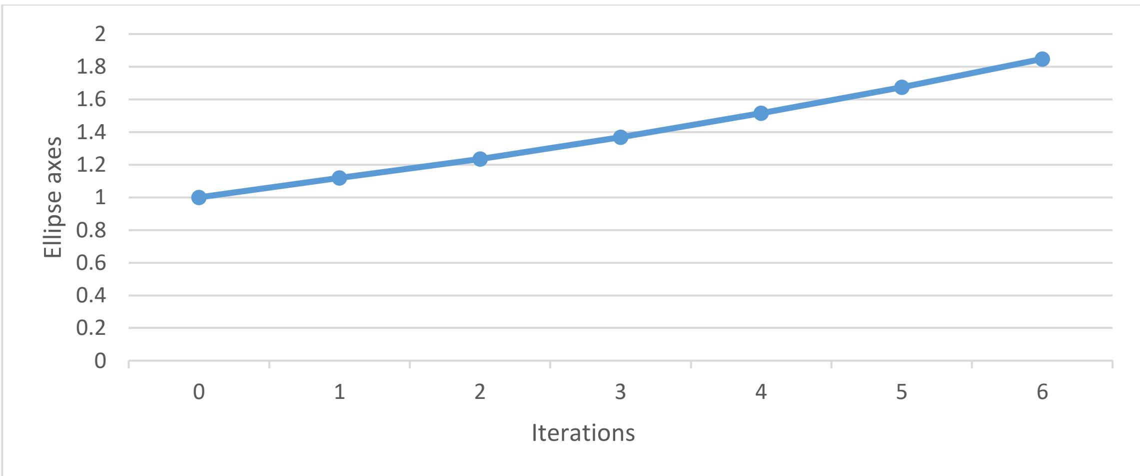
Figure 4.25: Optimization results of a plate with a hole (D=20 mm, K=250, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 6 (right)

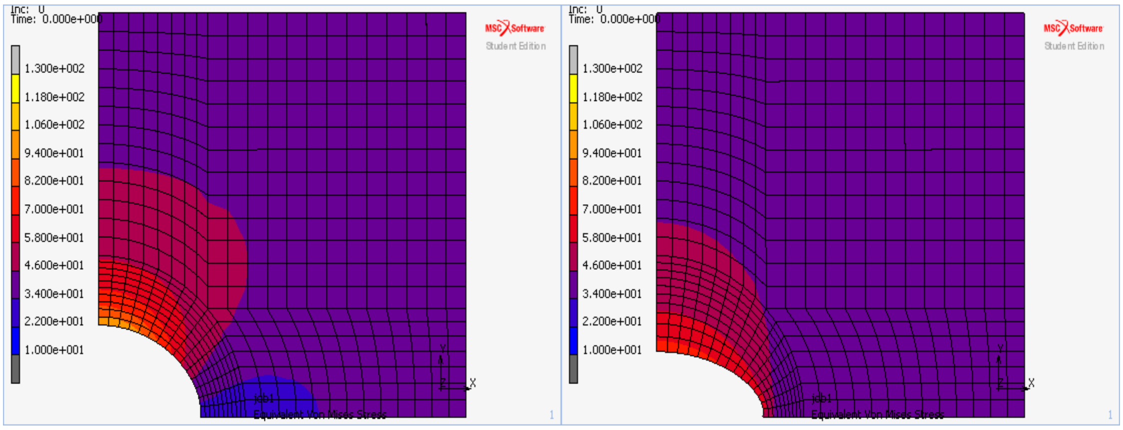


(b) Variation of von Mises stress distribution along the hole boundary

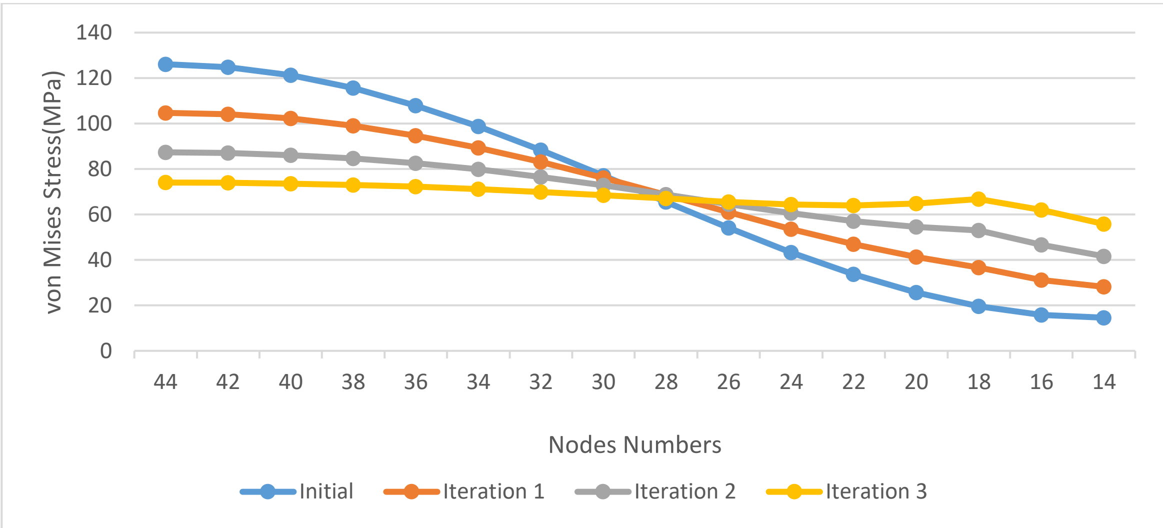


(c) Variation of ellipse axes ratio

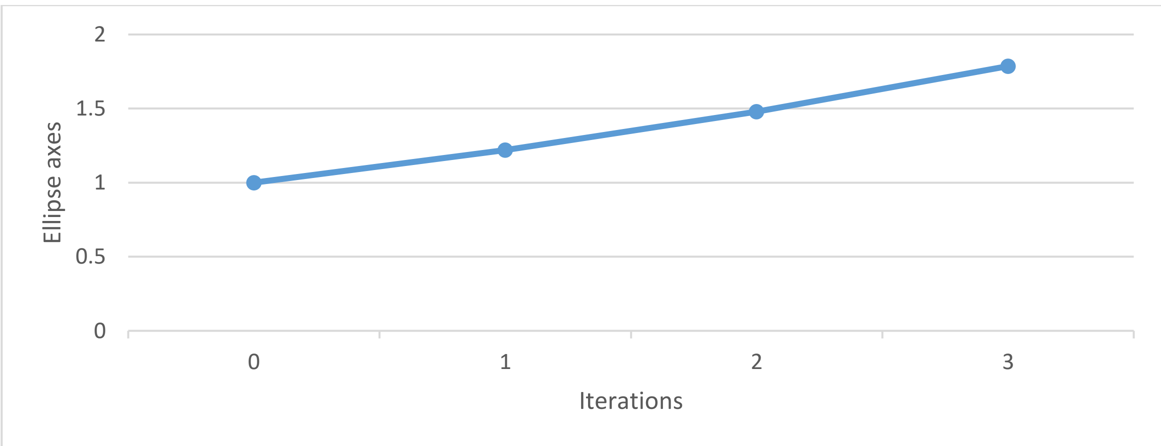
Figure 4.26: Optimization results of a plate with a hole (D=20 mm, K=275, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

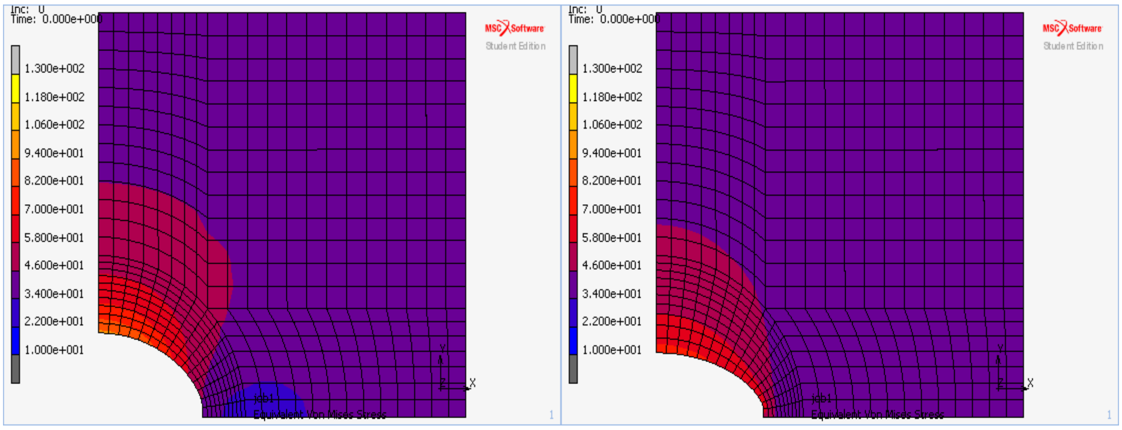


(b) Variation of von Mises stress distribution along the hole boundary

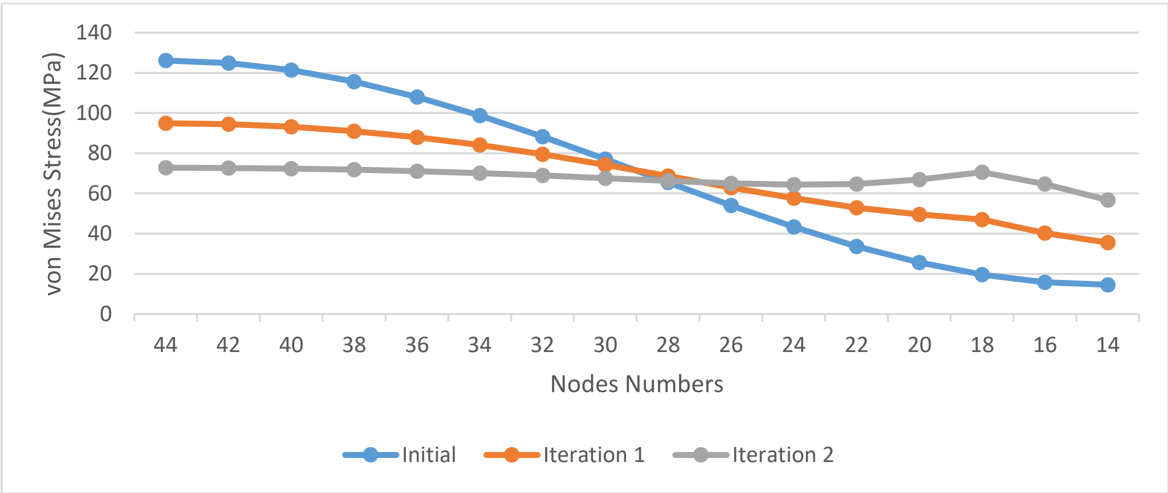


(c) Variation of ellipse axes ratio

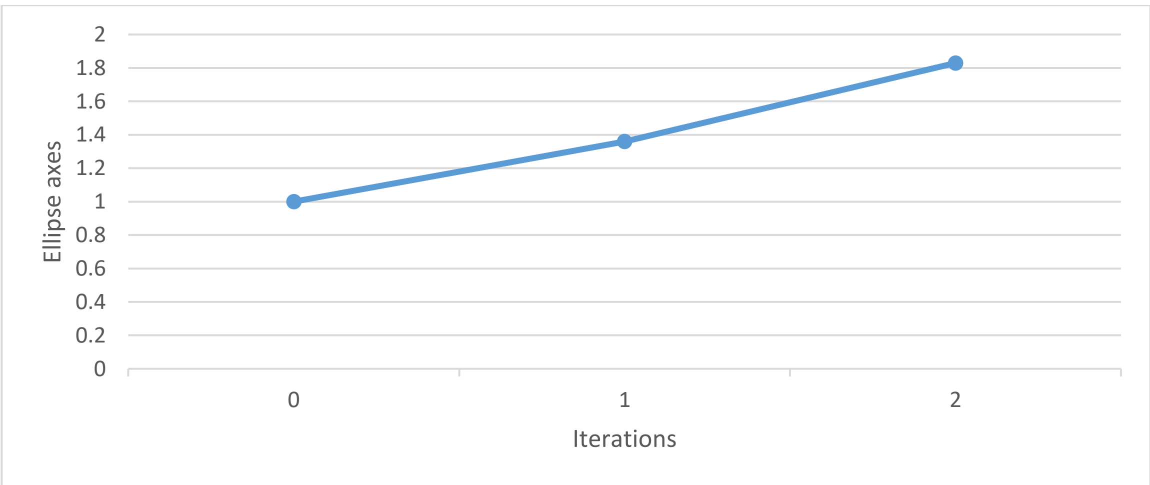
Figure 4.27: Optimization results of a plate with a hole (D=20 mm, K=500, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 2 (right)

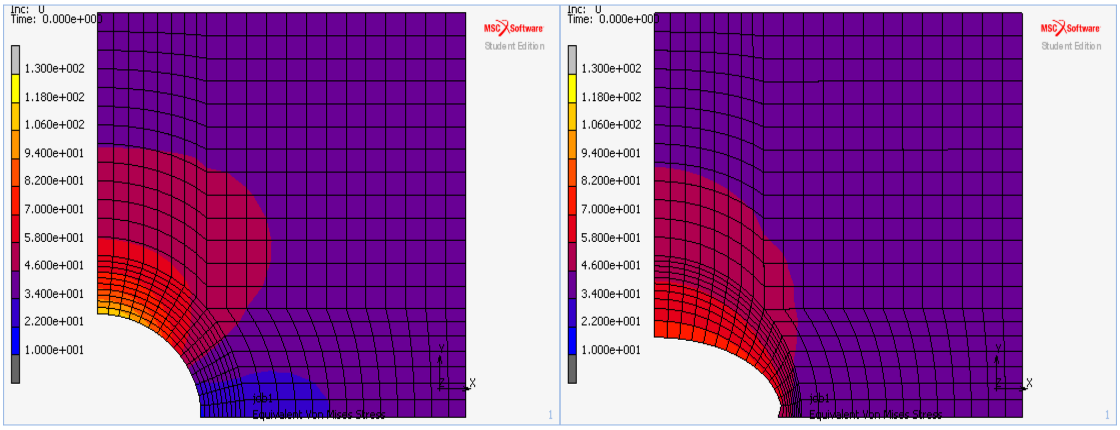


(b) Variation of von Mises stress distribution along the hole boundary

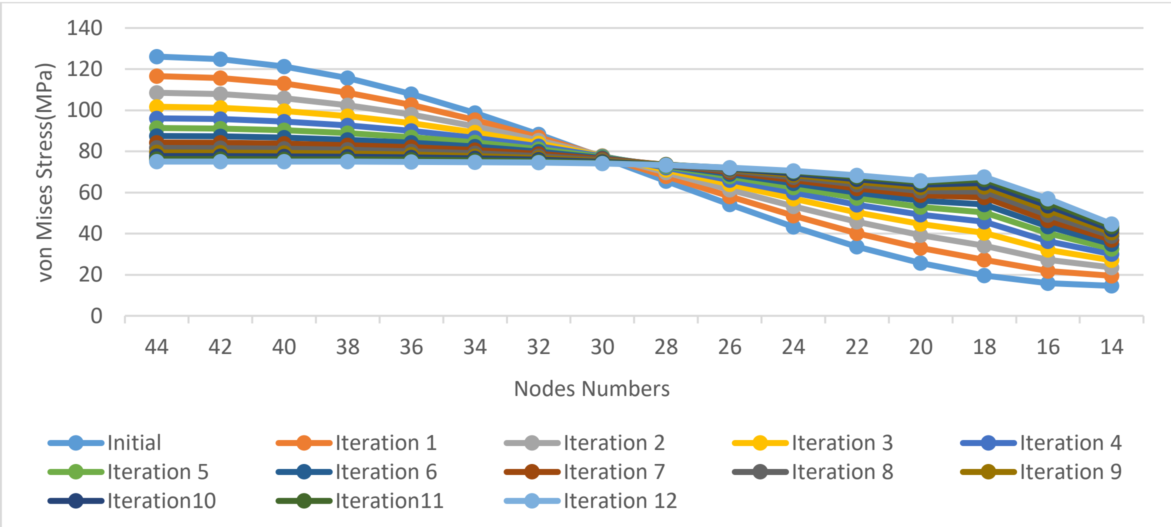


(c) Variation of ellipse axes ratio

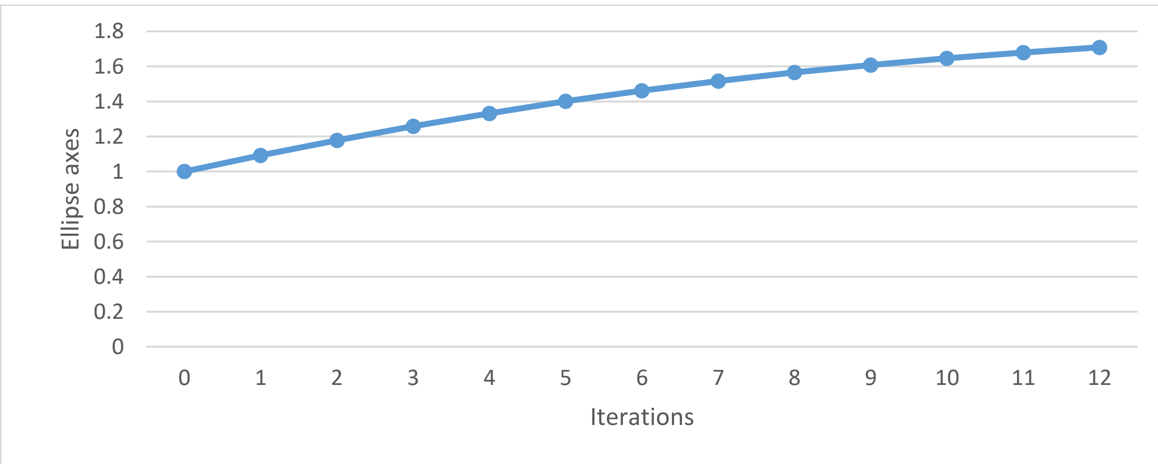
Figure 4.28: Optimization results of a plate with a hole (D=20 mm, K=750, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 12 (right)

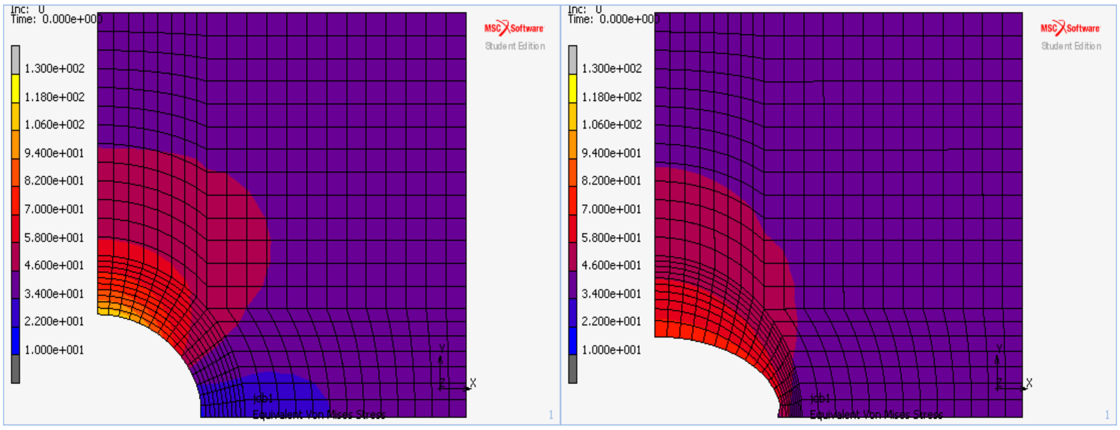


(b) Variation of von Mises stress distribution along the hole boundary

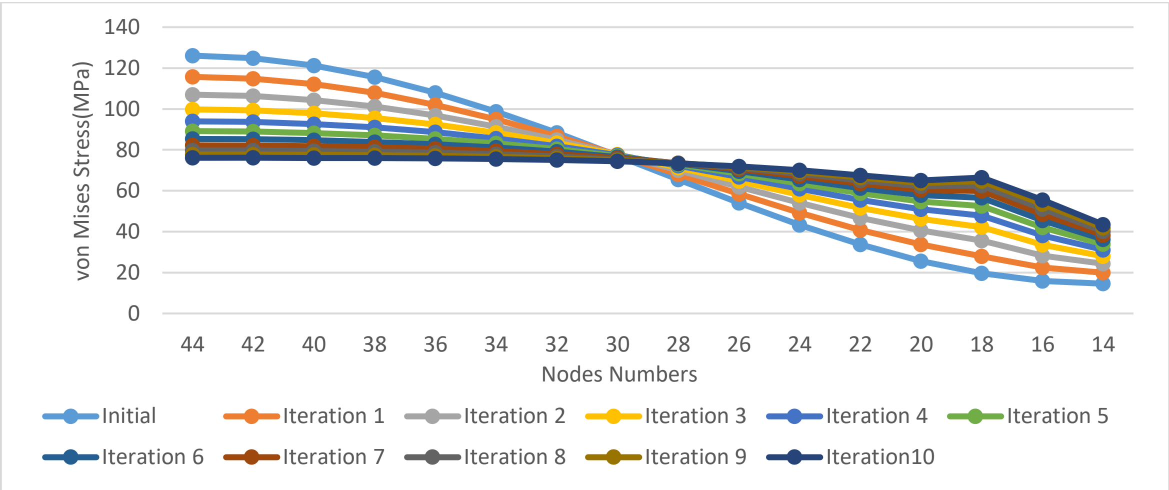


(c) Variation of ellipse axes ratio

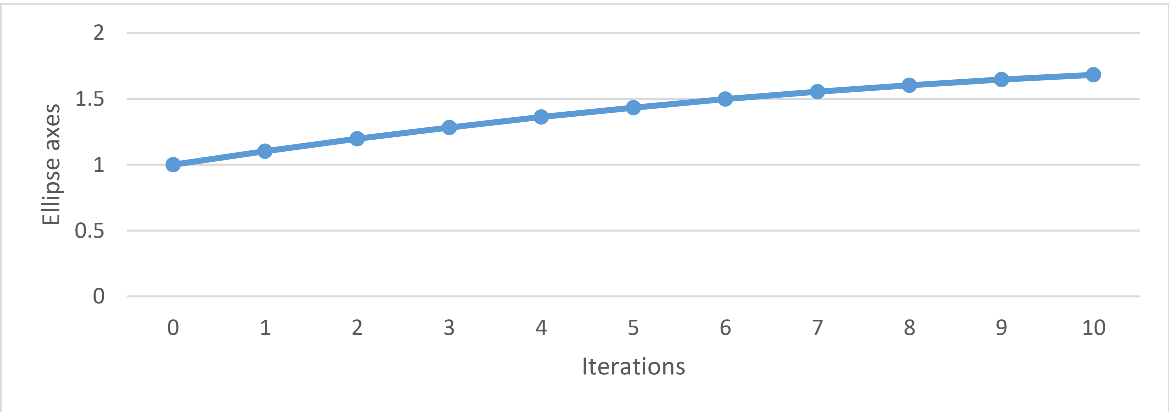
Figure 4.29: Optimization results of a plate with a hole (D=20 mm, K=250, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 10 (right)

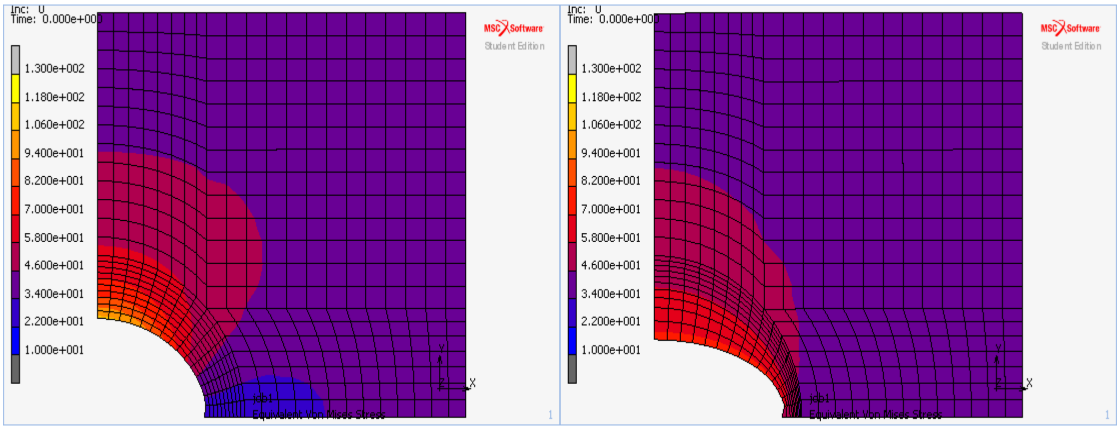


(b) Variation of von Mises stress distribution along the hole boundary

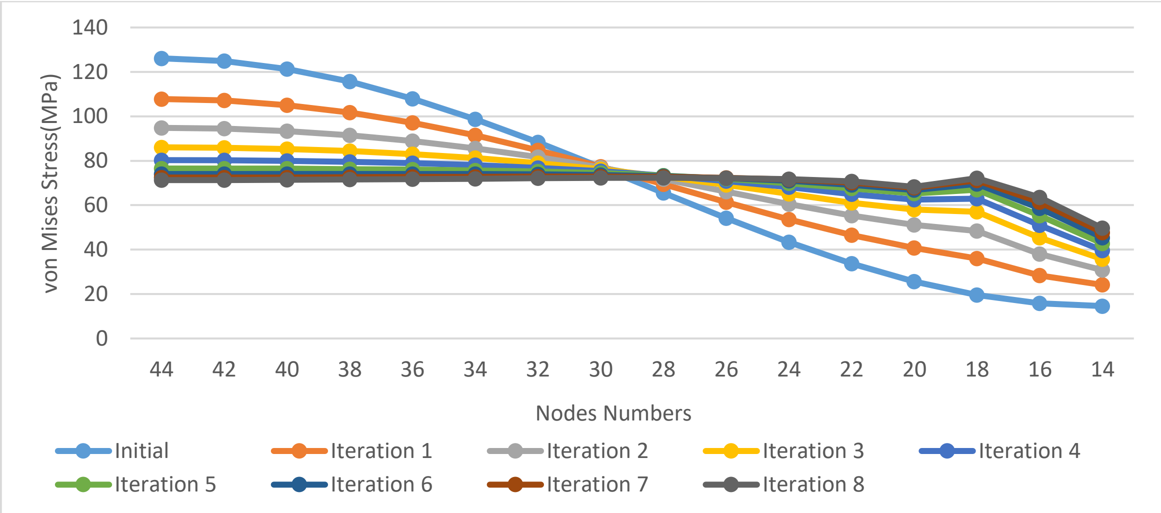


(c) Variation of ellipse axes ratio

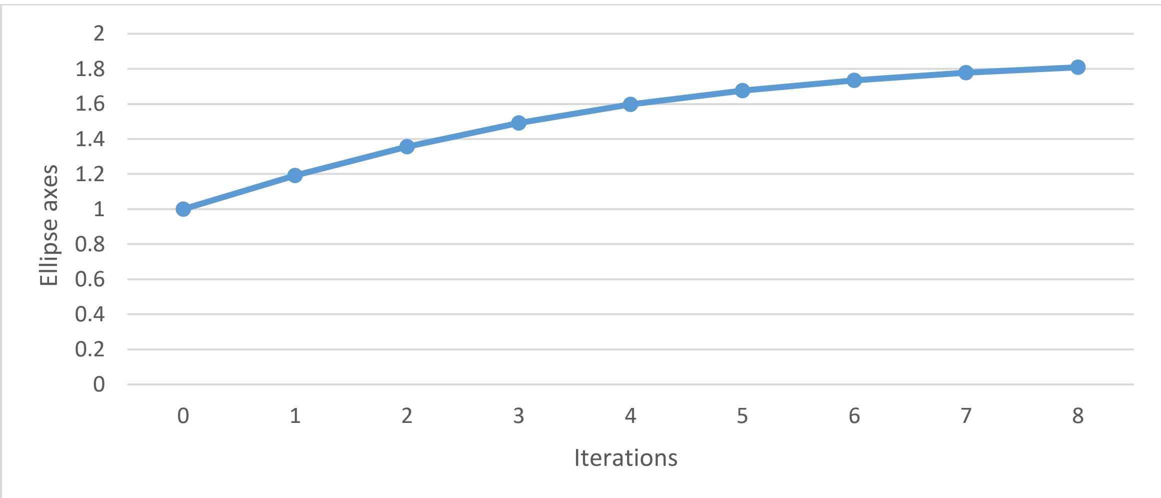
Figure 4.30: Optimization results of a plate with a hole (D=20 mm, K=275, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 8 (right)

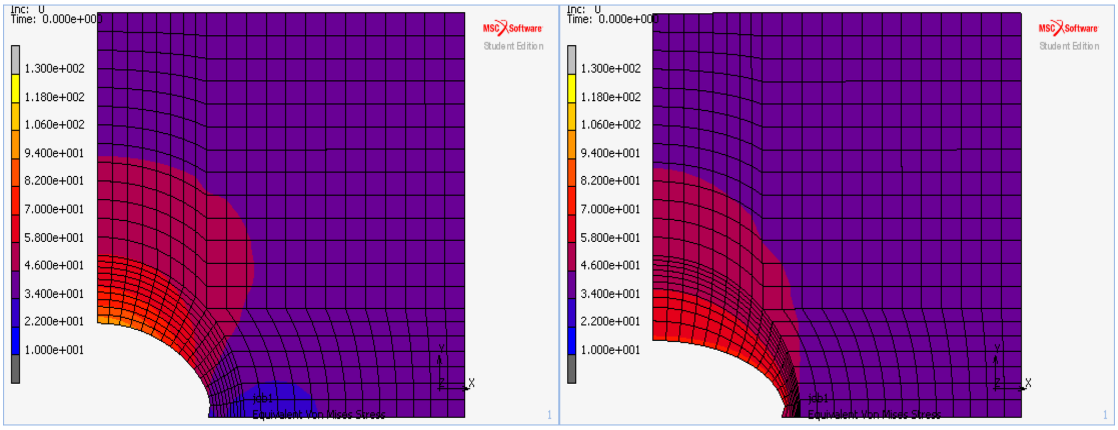


(b) Variation of von Mises stress distribution along the hole boundary

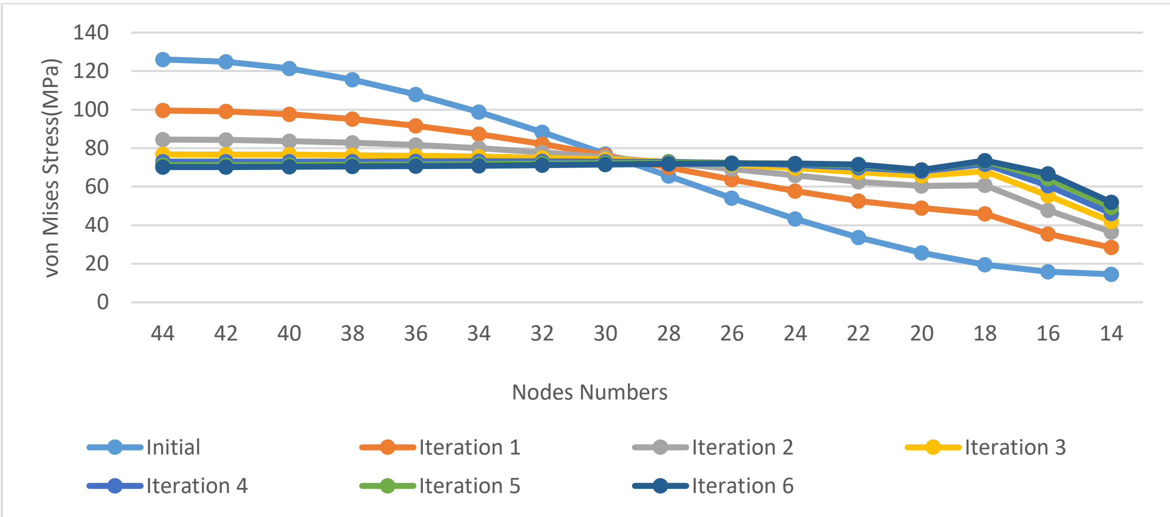


(c) Variation of ellipse axes ratio

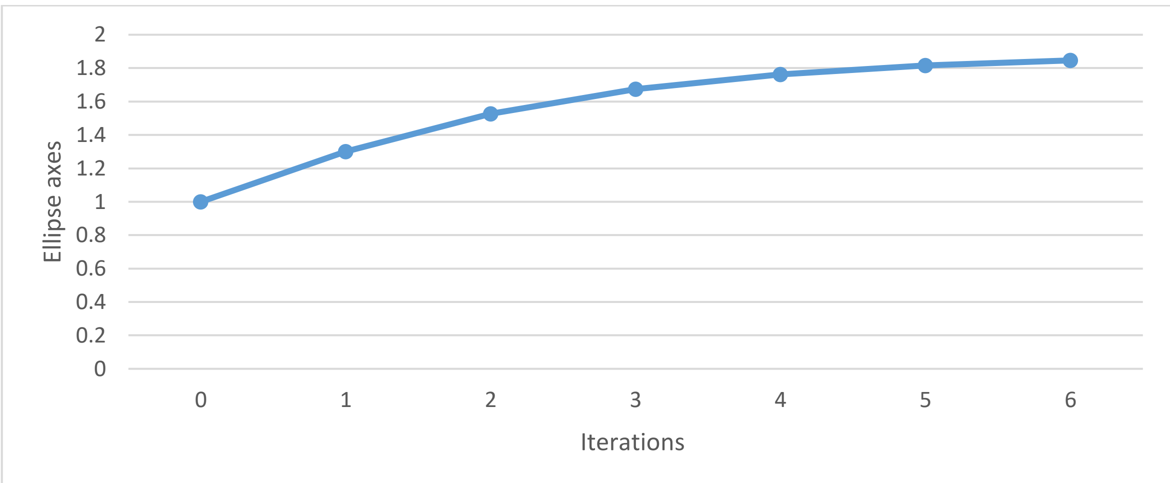
Figure 4.31: Optimization results of a plate with a hole (D=20 mm, K=500, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 6 (right)

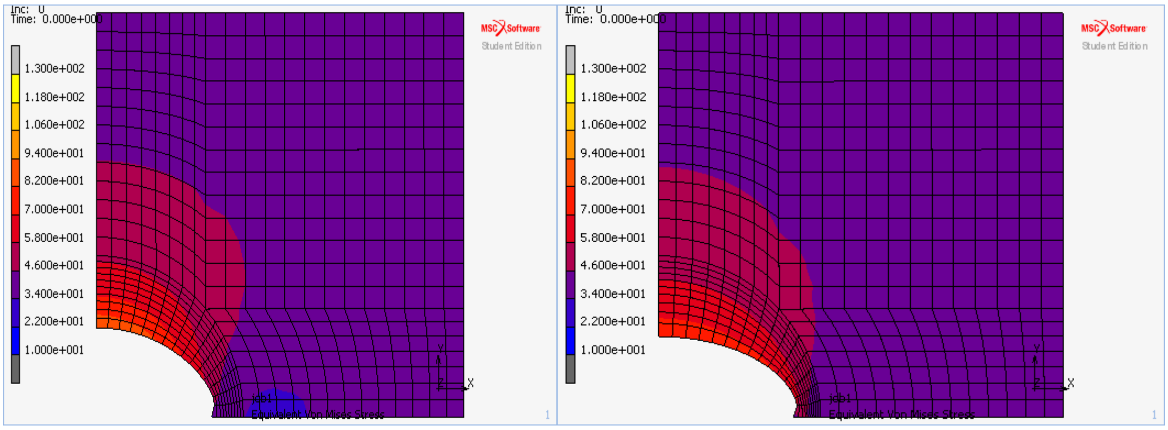


(b) Variation of von Mises stress distribution along the hole boundary

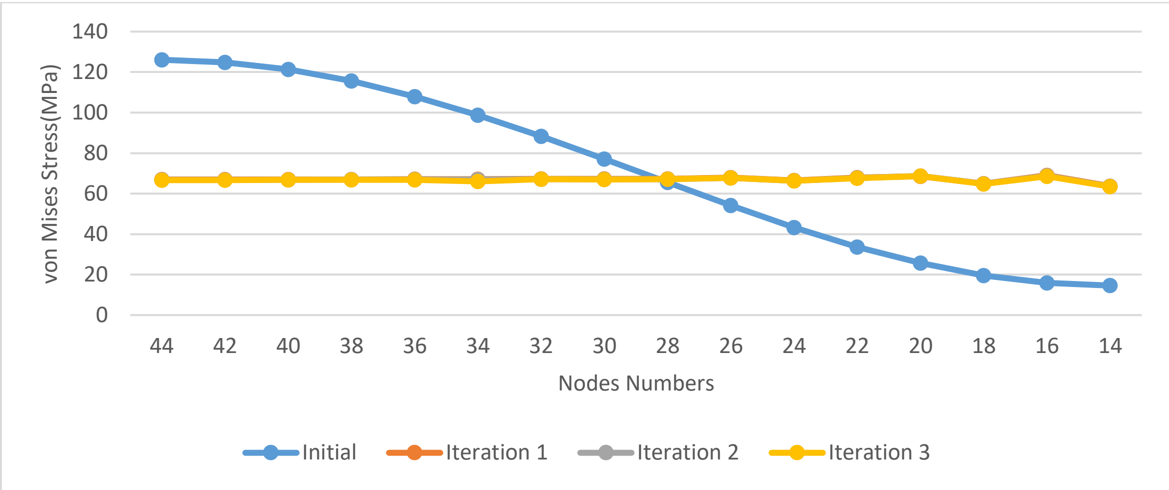


(c) Variation of ellipse axes ratio

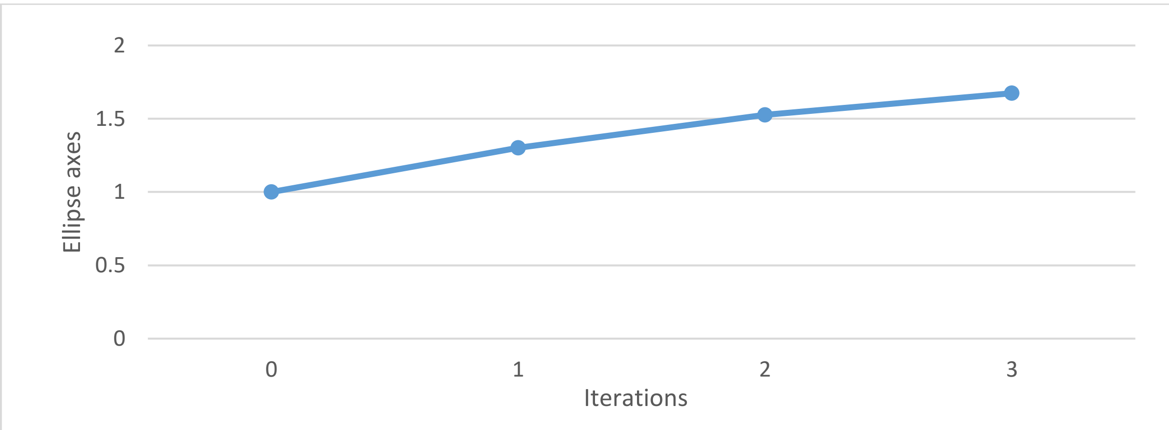
Figure 4.32: Optimization results of a plate with a hole (D=20 mm, K=750, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)



(b) Variation of von Mises stress distribution along the hole boundary



(c) Variation of ellipse axes ratio

Figure 4.33: Optimization results of a plate with a hole (D=20 mm, K=1000,σ_{ref} =60 MPa)

Table 4.3: Summary of results with domain thickness 20mm

		Magnification factor k				
σ_{ref}		*250*	*275*	*500*	*750*	*1000*
10	Iterations	4	3	2	--	--
	Max stress	83.64	83.19	87.06	--	--
	Ellipse axes ratio	1.84	1.58	1.82	--	--
40	Iterations	7	6	3	2	--
	Max stress	69.92	71.82	74.04	72.82	--
	Ellipse axes ratio	1.90	1.84	1.78	1.82	--
60	Iterations	12	10	8	6	3
	Max stress	75.03	74.82	72.34	73.20	68.65
	Ellipse axes ratio	1.70	1.68	1.86	1.84	1.67

4.3 Optimization of a Plate with a Hole with Domain Thickness 30 mm

The finite element models and the boundary conditions for stress and thermal analysis are shown in Figure 4.34 for domain thickness 30 mm. The von Mises stresses of stress analysis and total deflections of the thermal expansion analysis of the original shape can be seen in Figure 4.35. Similar to the analysis conducted for domain thickness 10 mm, the stress far away from the concentration zones is about 40 MPa. The maximum and minimum von Mises stresses were also around 130 MPa and 10 MPa respectively.

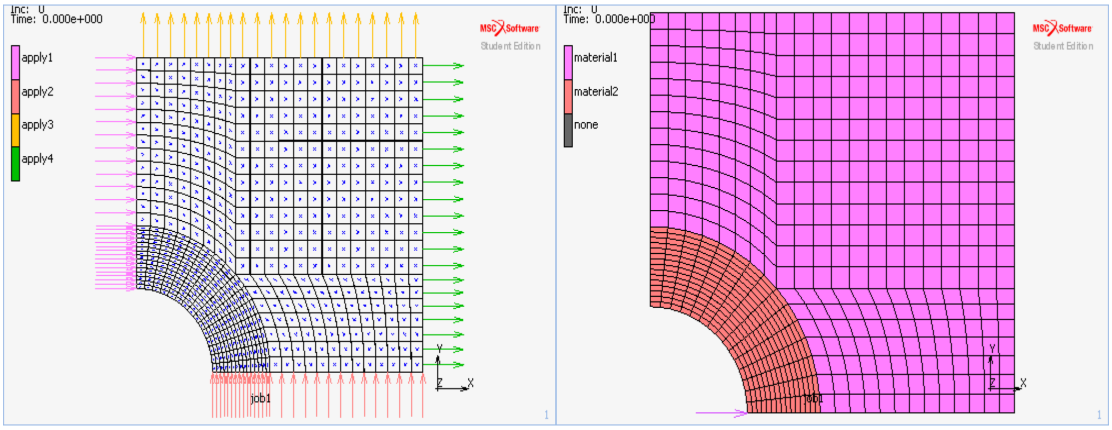


Figure 4.34: Finite element model for stress (left) and thermal (right) analysis (D=30mm)

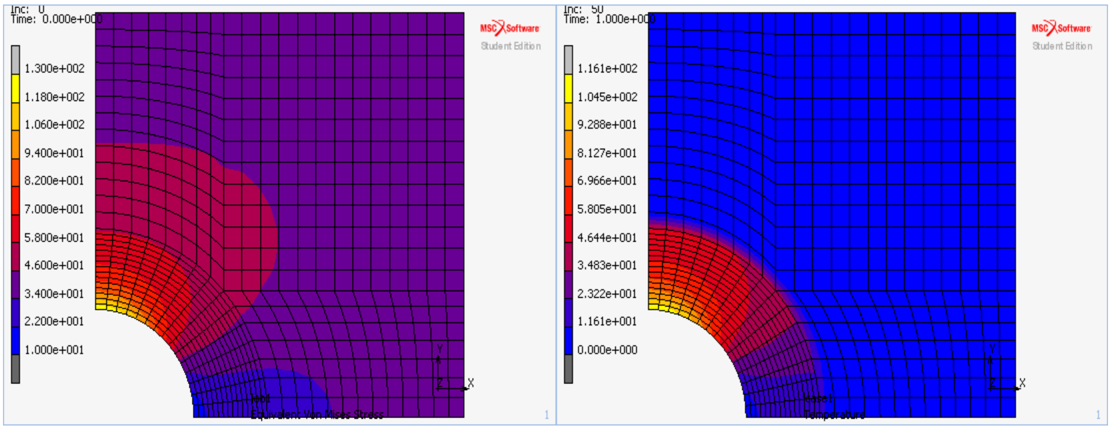
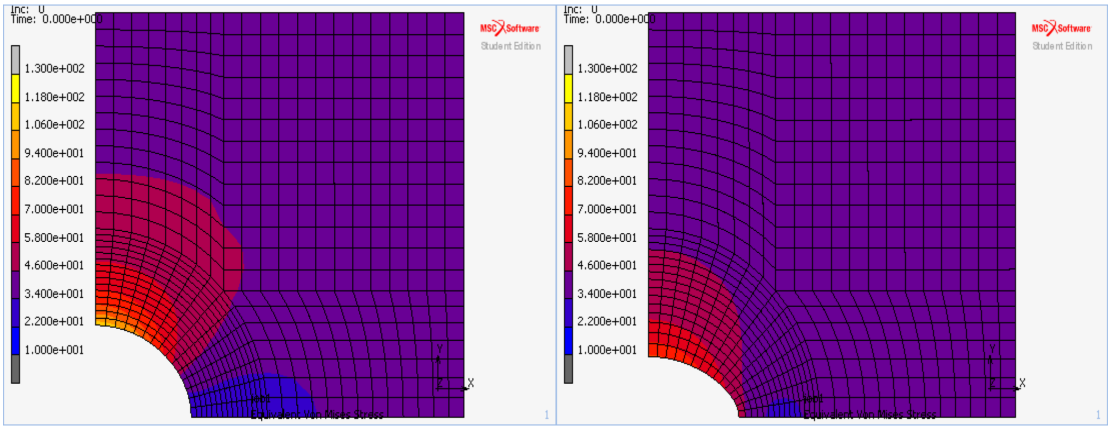
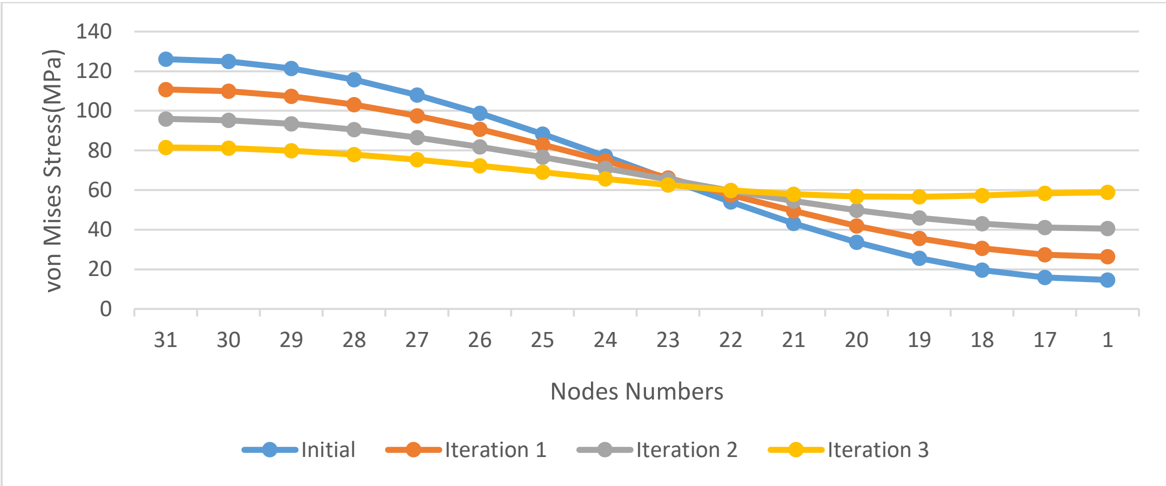


Figure 4.35: von Mises stresses (left) and thermal deformations (right) of the original shape (D=30mm)

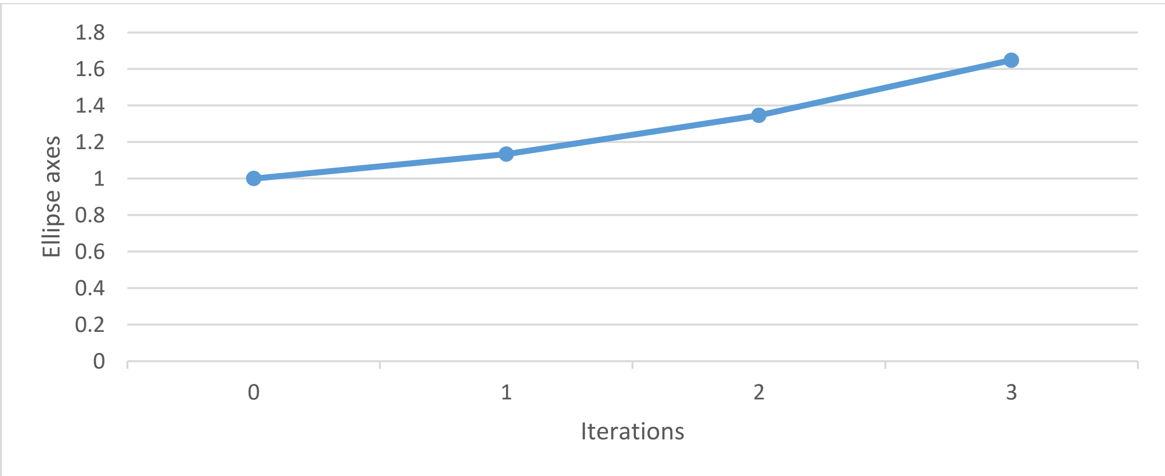
For domain thickness D = 30 mm, eleven optimization analyses were conducted including the combinations of reference stresses σ_{ref} = 10, 40 and 60 MPa and magnification factor k = 250, 275, 500, 750 and 1000. The results obtained including (a) von Mises stress distributions of the plate after first and last iterations, (b) the change of von Mises stresses by iterations along the hole boundary, (c) the change of ellipse axes ratio by iterations are presented in Figures 4.36 to 4.46. No remarkable difference was observed from the previous domain 10 and 20 mm analyses results as given in Table 4.4.



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

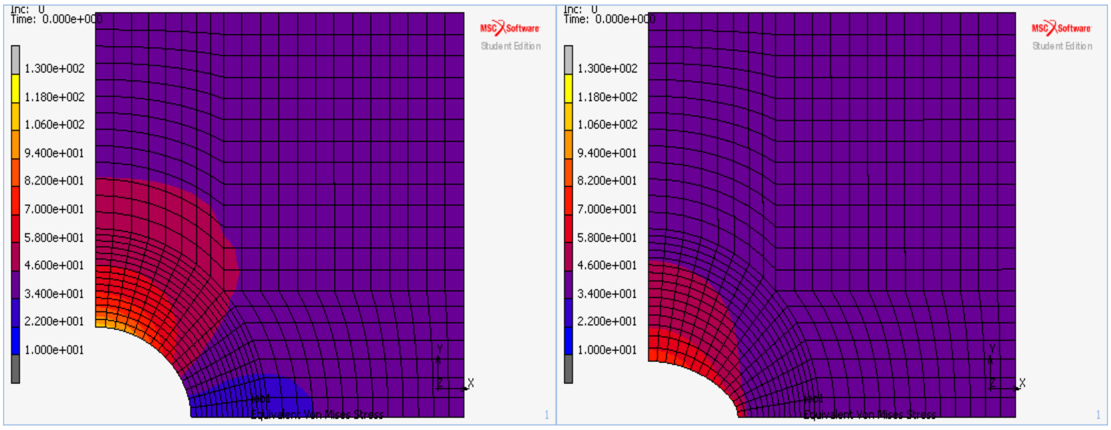


(b) Variation of von Mises stress distribution along the hole boundary

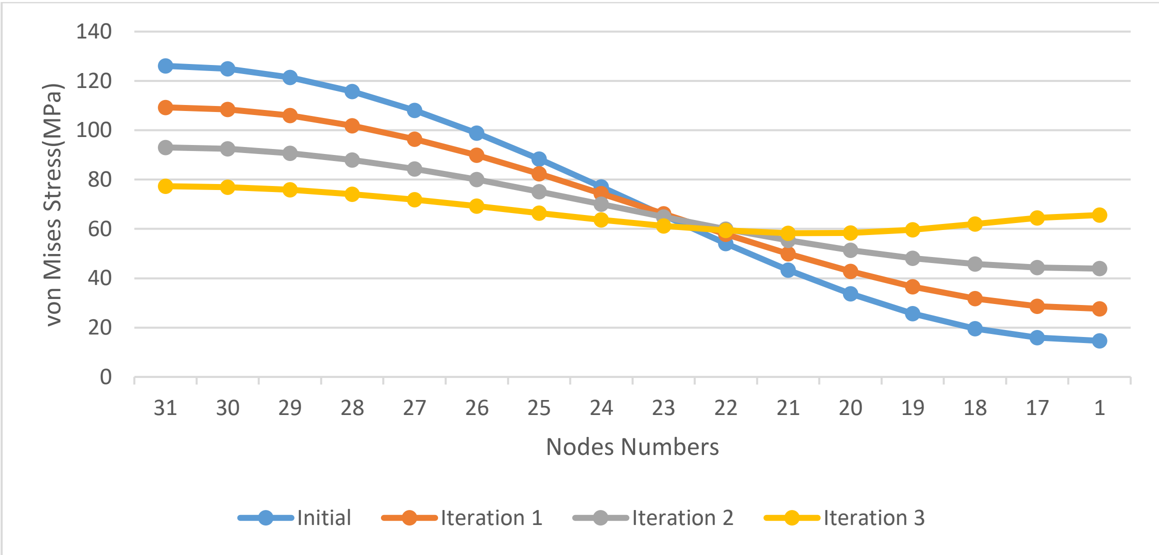


(c) Variation of ellipse axes ratio

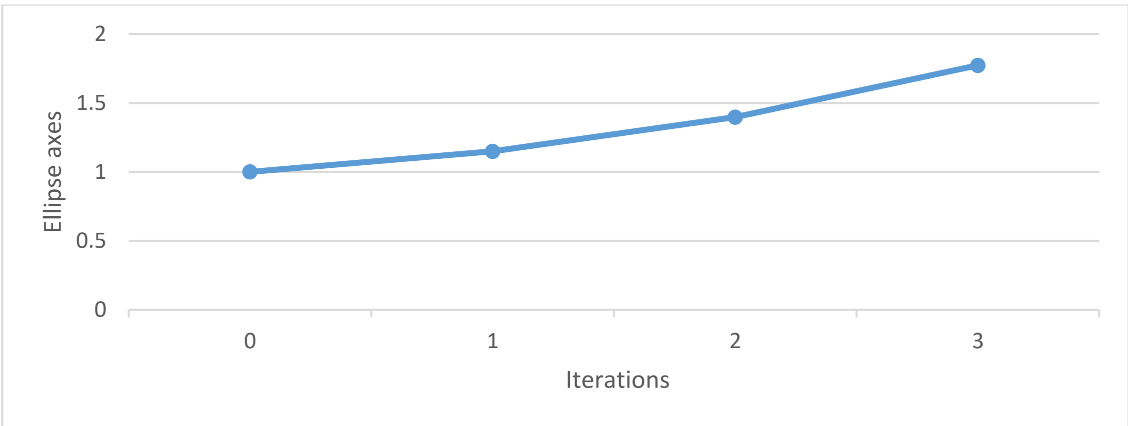
Figure 4.36: Optimization results of a plate with a hole (D=30 mm, K=250, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

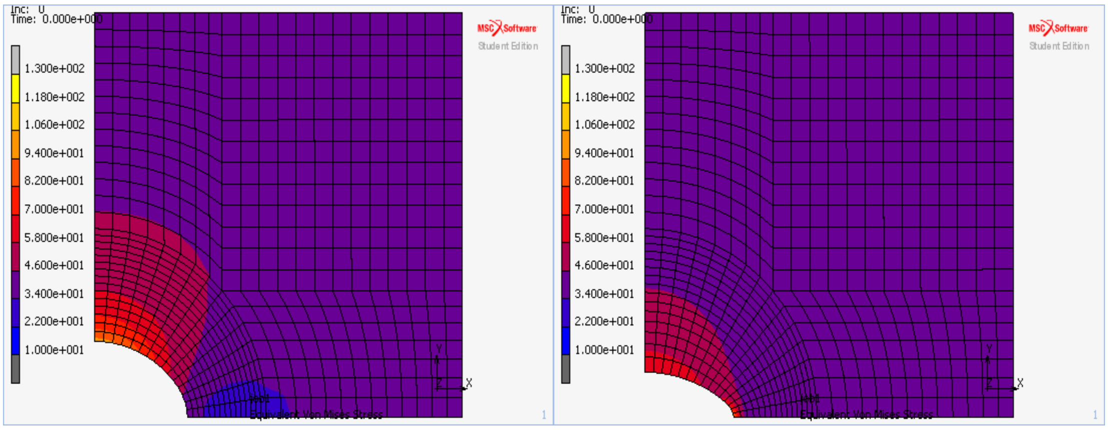


(b) Variation of von Mises stress distribution along the hole boundary

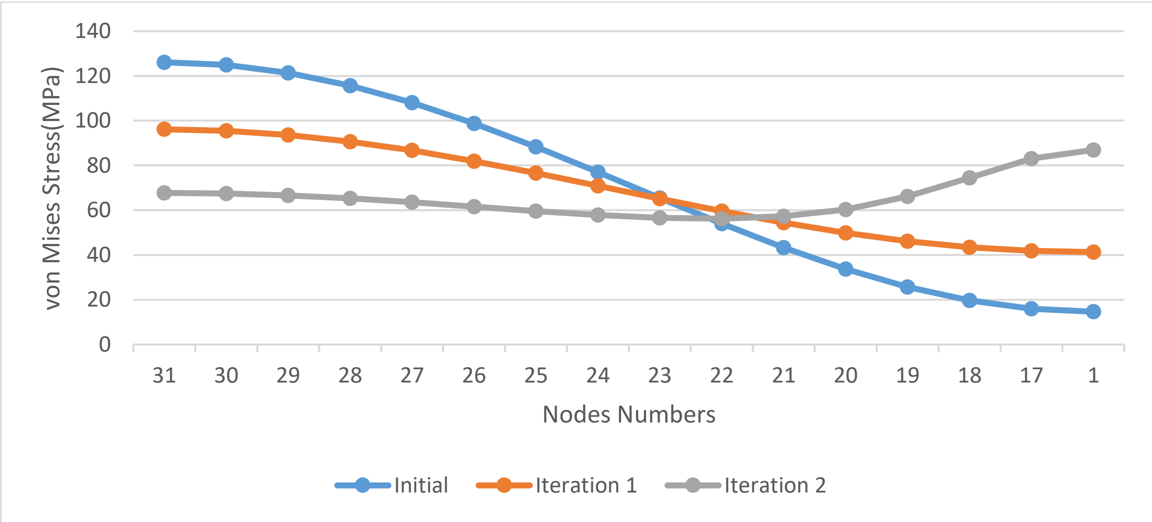


(c) Variation of ellipse axes ratio

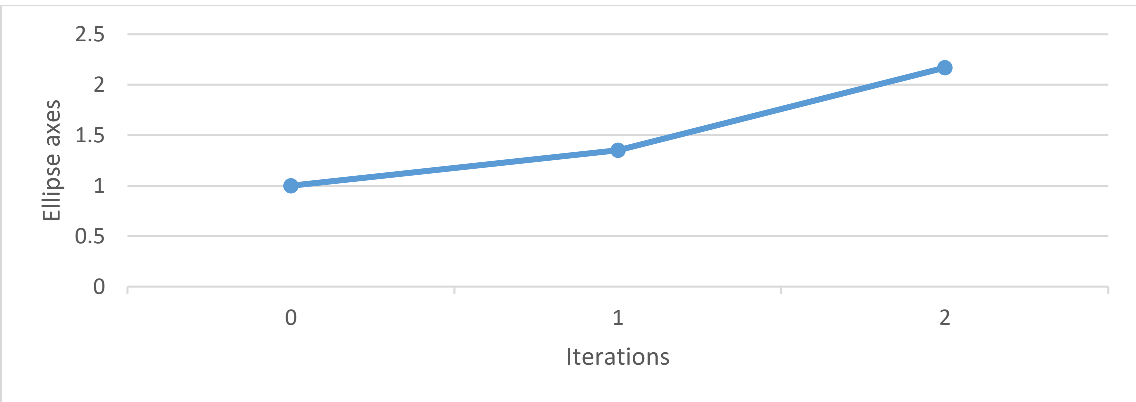
Figure 4.37: Optimization results of a plate with a hole (D=30 mm, K=275, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 2 (right)

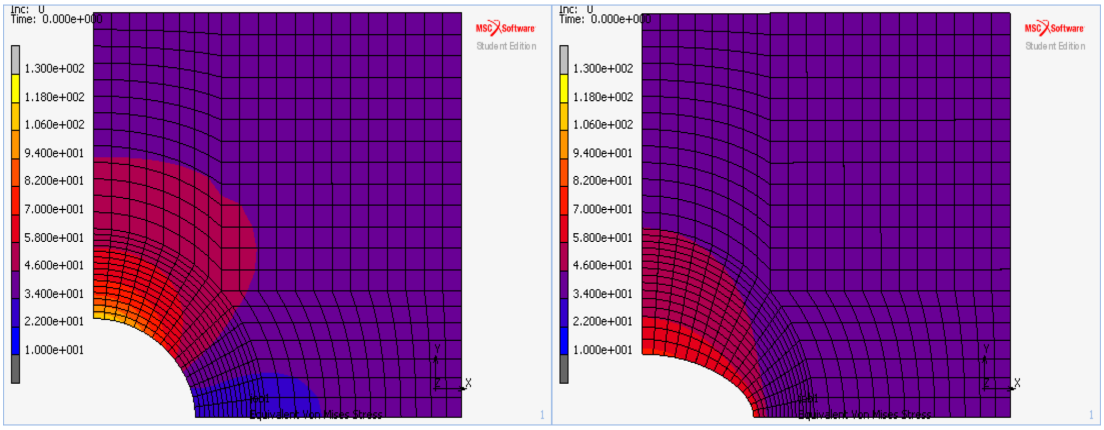


(b) Variation of von Mises stress distribution along the hole boundary

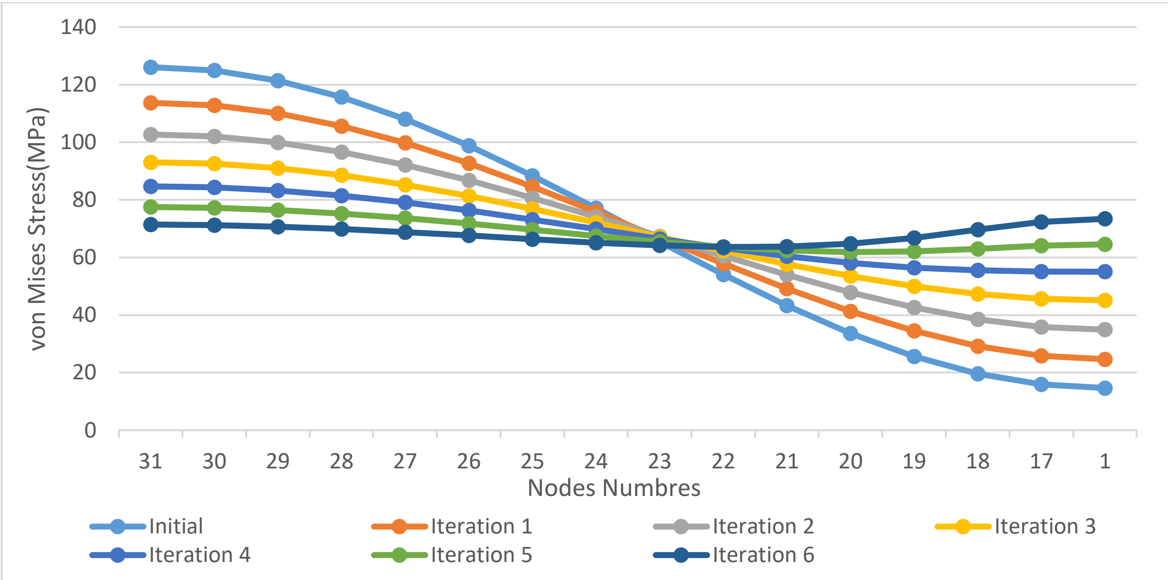


(c) Variation of ellipse axes ratio

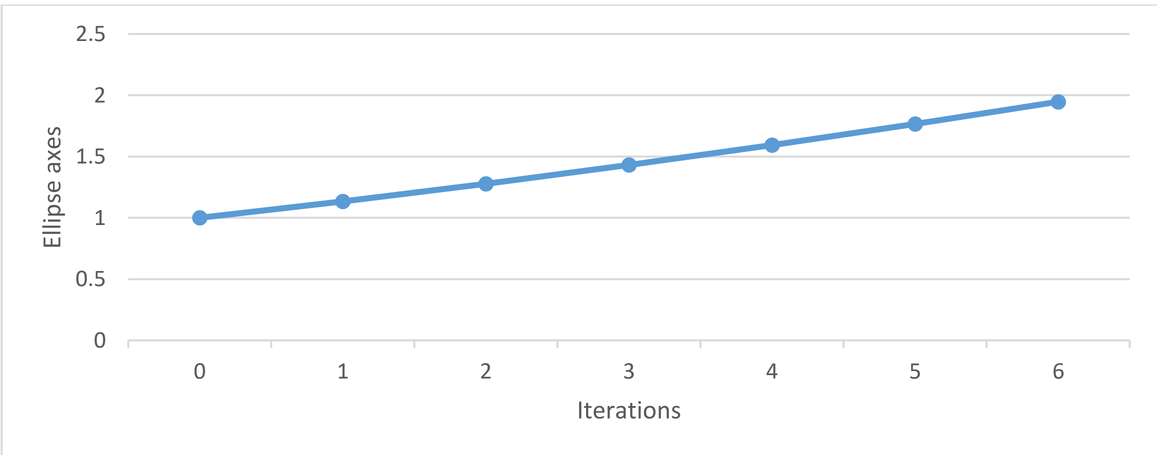
Figure 4.38: Optimization results of a plate with a hole (D=30 mm, K=500, σ_{ref} = 10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 6 (right)

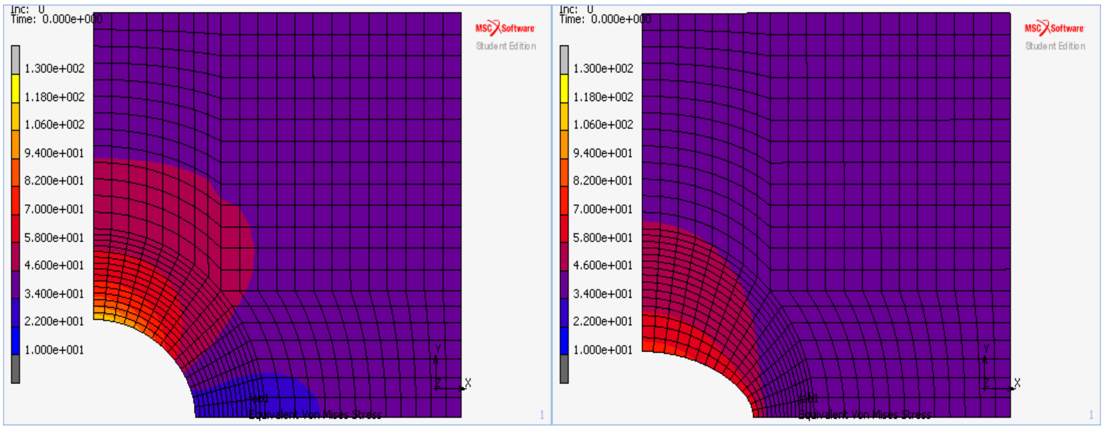


(b) Variation of von Mises stress distribution along the hole boundary

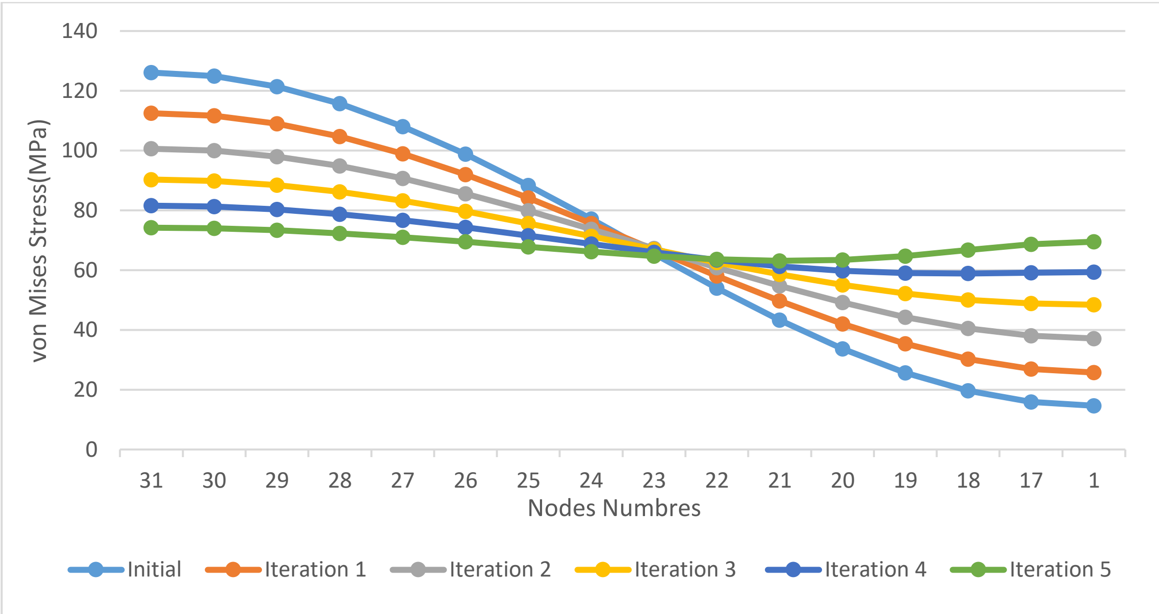


(c) Variation of ellipse axes ratio

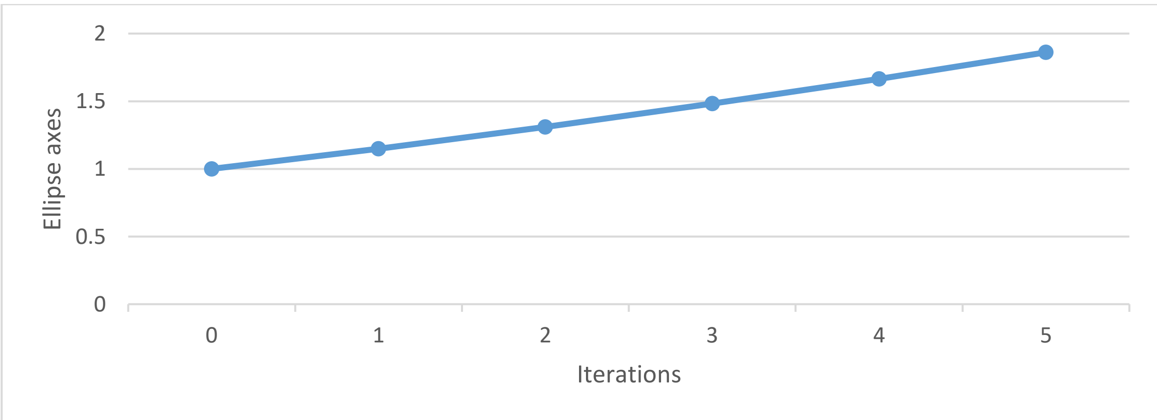
Figure 4.39: Optimization results of a plate with a hole (D=30 mm, K=250, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)

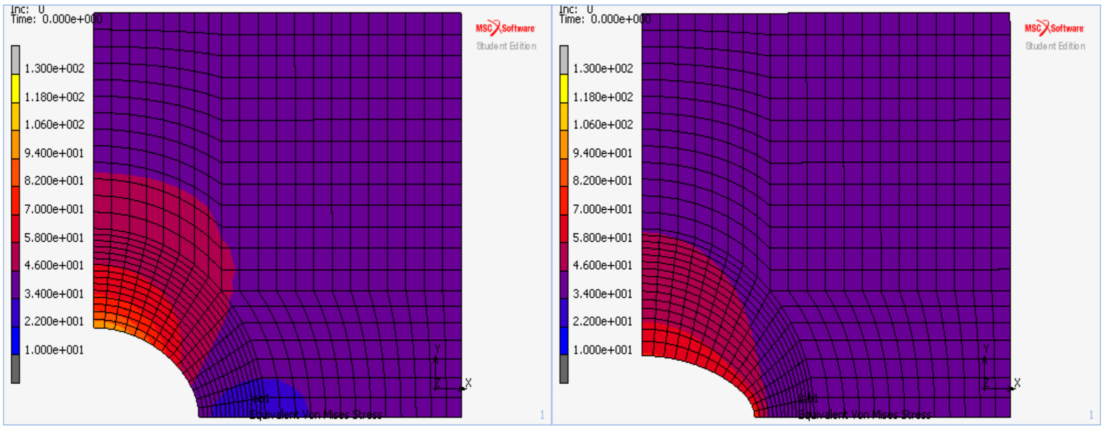


(b) Variation of von Mises stress distribution along the hole boundary

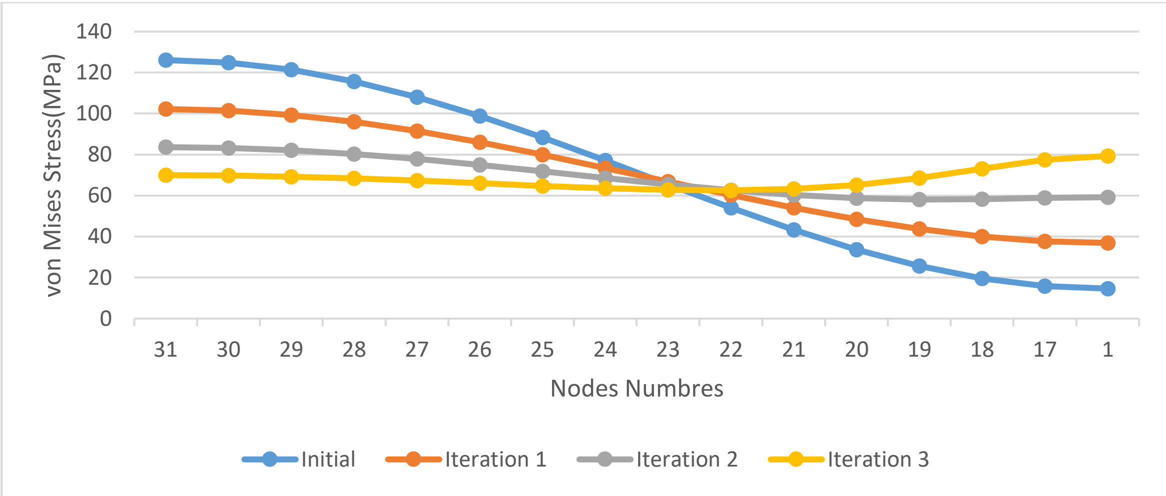


(c) Variation of ellipse axes ratio

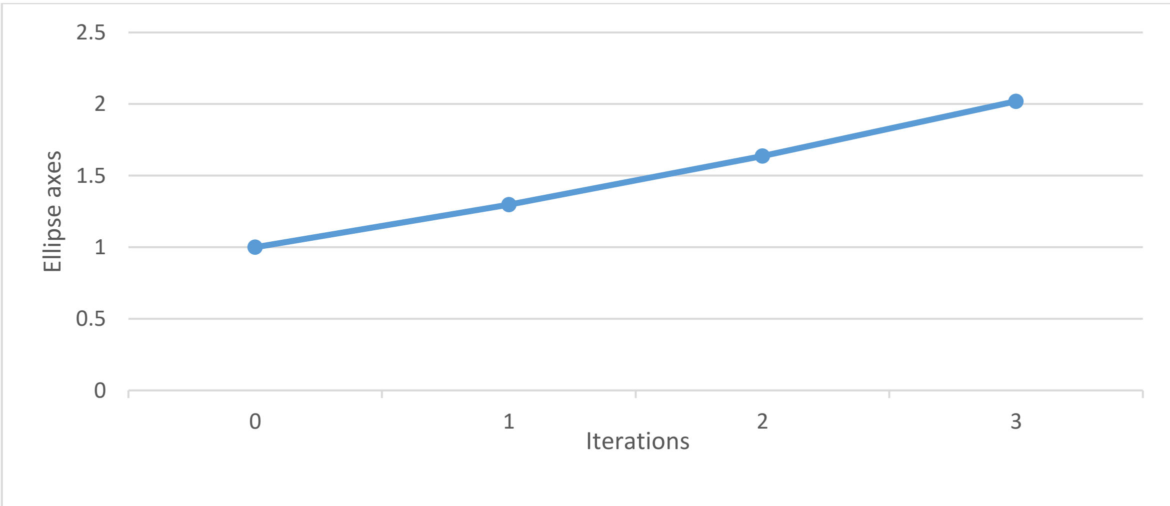
Figure 4.40: Optimization results of a plate with a hole (D=30 mm, K=275, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

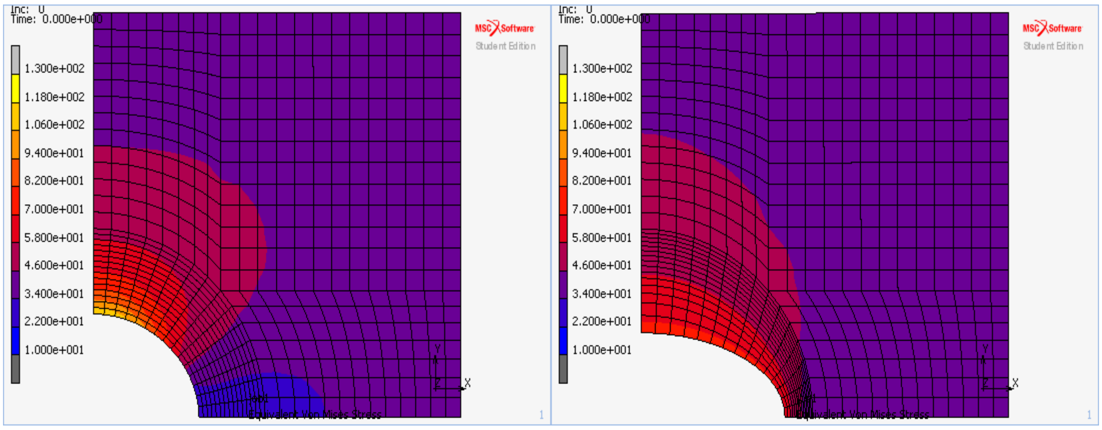


(b) Variation of von Mises stress distribution along the hole boundary

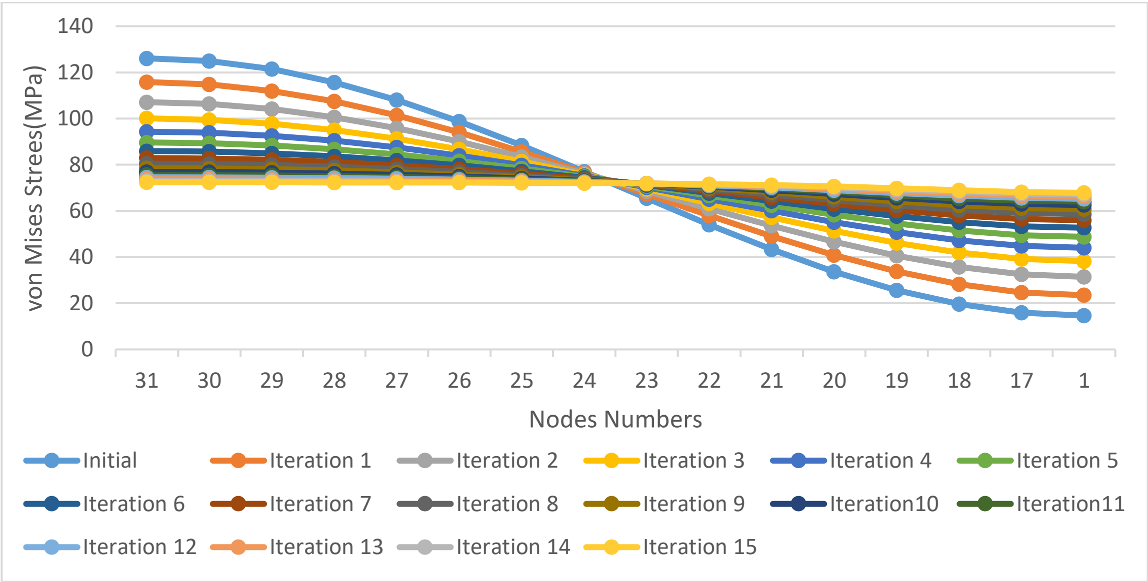


(c) Variation of ellipse axes ratio

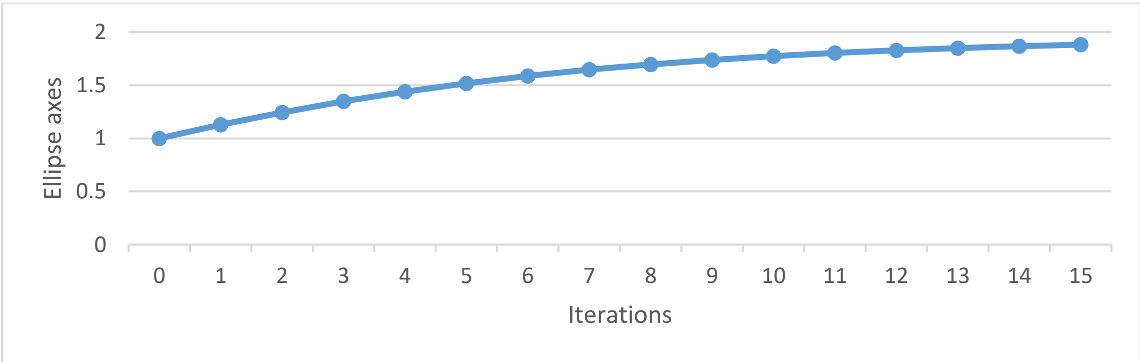
Figure 4.41: Optimization results of a plate with a hole (D=30 mm, K=500, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 15 (right)

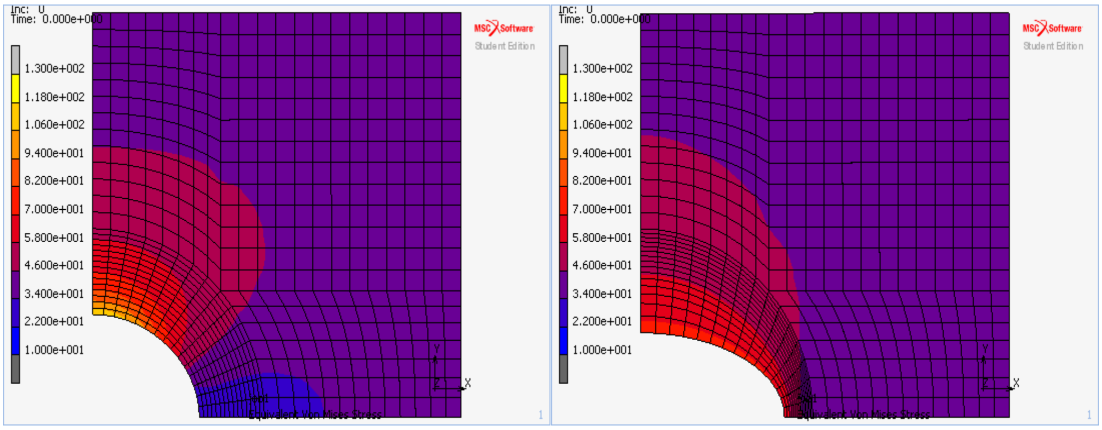


(b) Variation of von Mises stress distribution along the hole boundary

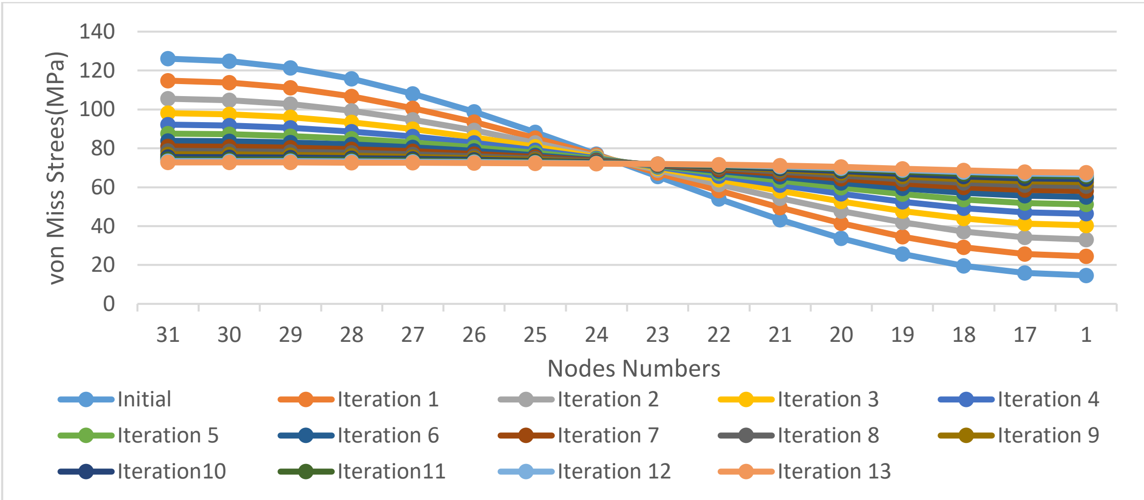


(c) Variation of ellipse axes ratio

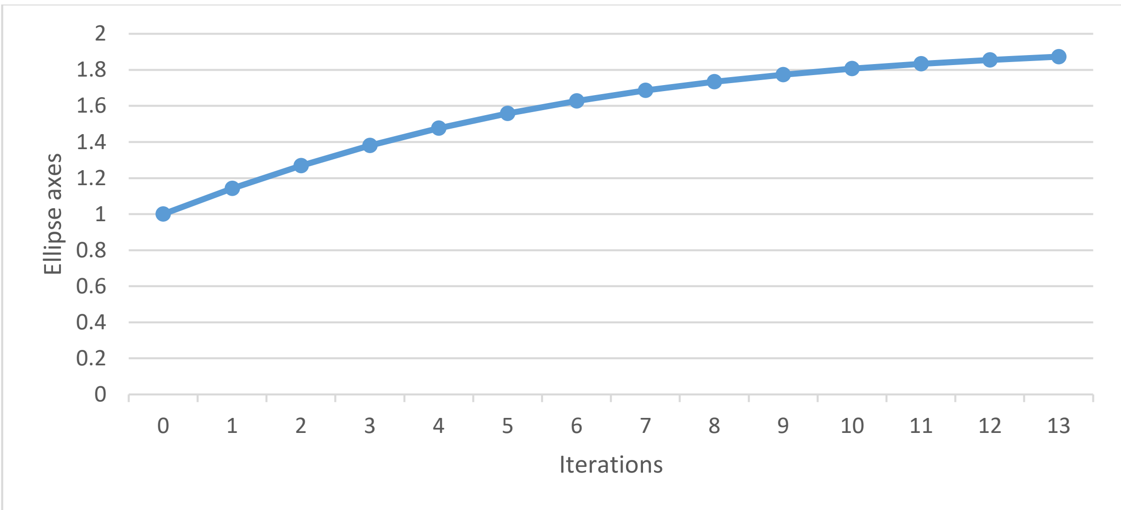
Figure 4.42: Optimization results of a plate with a hole (D=30 mm, K=250, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 13 (right)

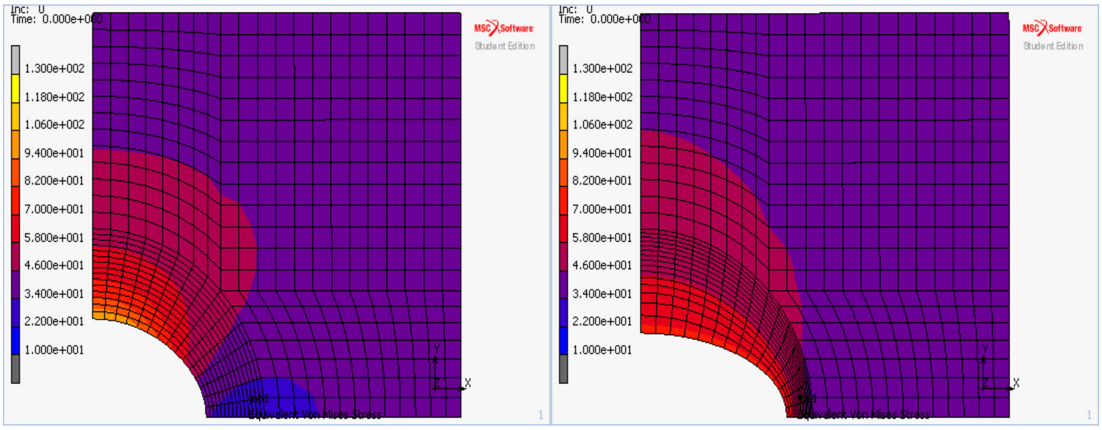


(b) Variation of von Mises stress distribution along the hole boundary

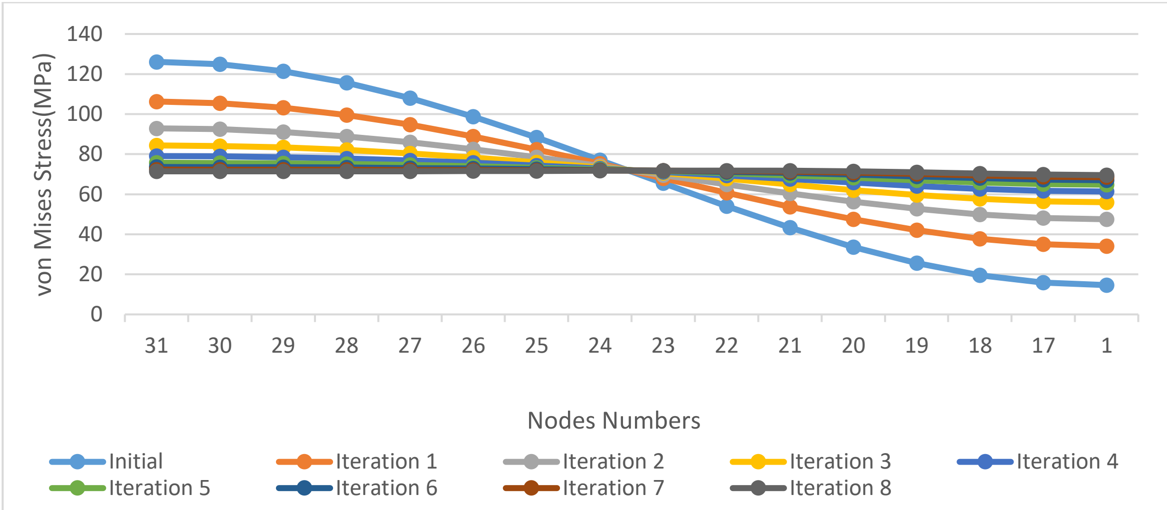


(c) Variation of ellipse axes ratio

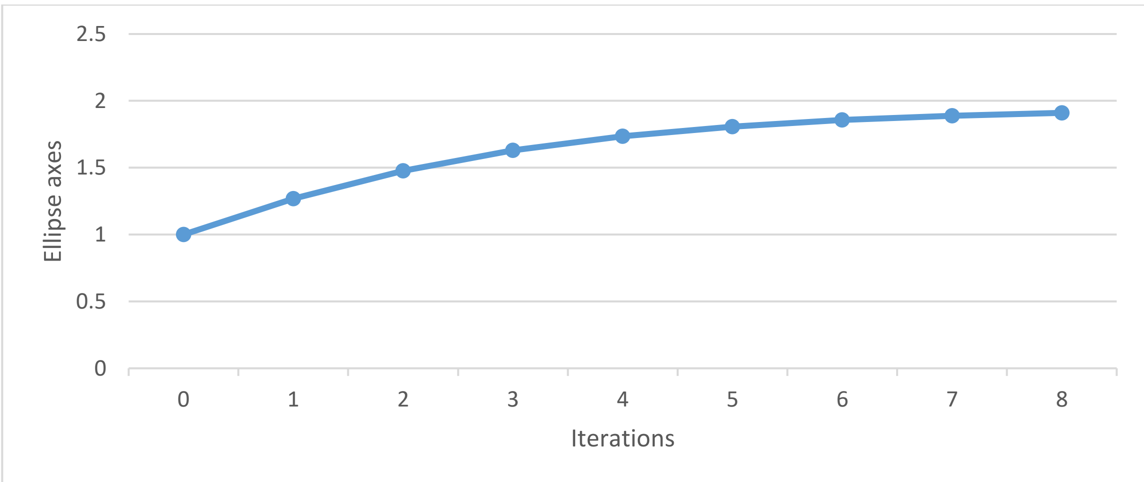
Figure 4.43: Optimization results of a plate with a hole (D=30 mm, K=275, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 8 (right)

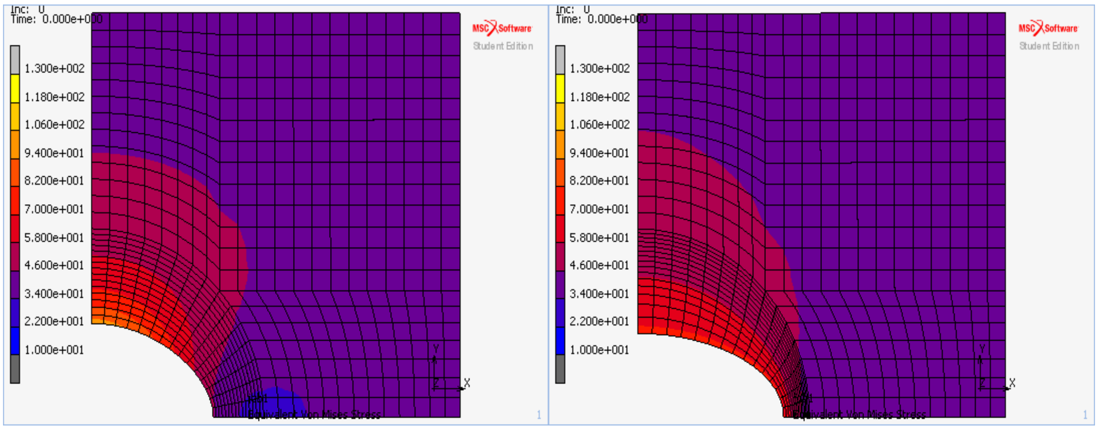


(b) Variation of von Mises stress distribution along the hole boundary

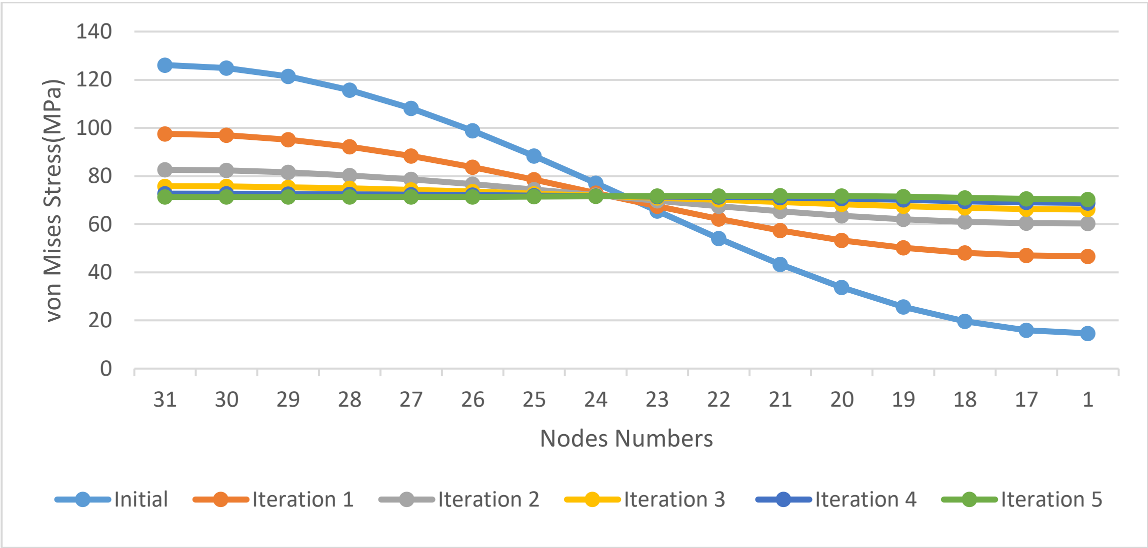


(c) Variation of ellipse axes ratio

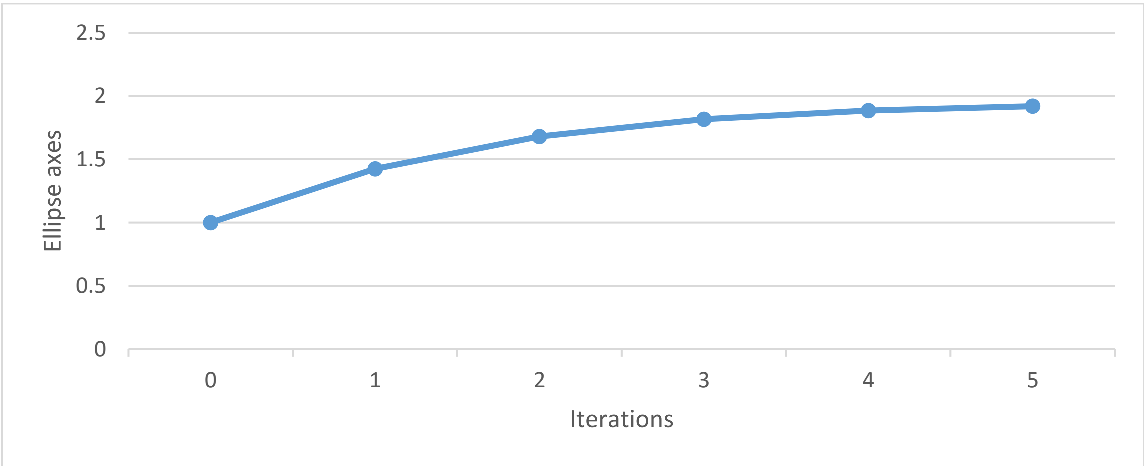
Figure 4.44: Optimization results of a plate with a hole (D=30 mm, K=500, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)

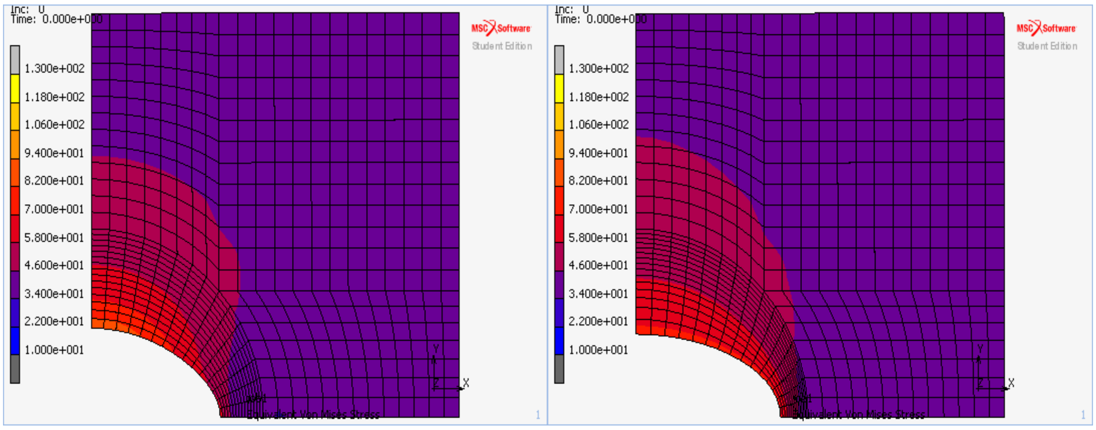


(b) Variation of von Mises stress distribution along the hole boundary

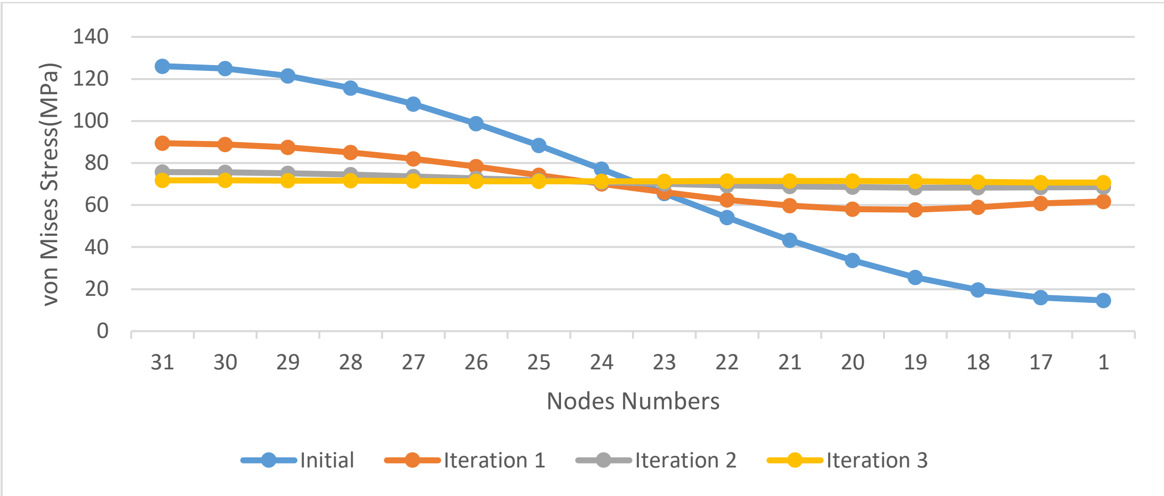


(c) Variation of ellipse axes ratio

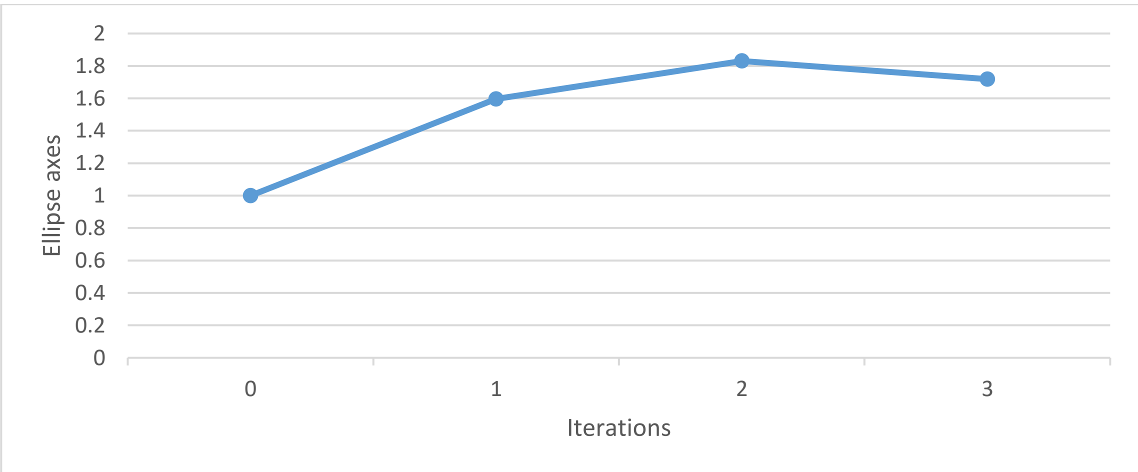
Figure 4.45: Optimization results of a plate with a hole (D=30 mm, K=750, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)



(b) Variation of von Mises stress distribution along the hole boundary



(c) Variation of ellipse axes ratio

Figure 4.46: Optimization results of a plate with a hole (D=30 mm, K=1000, σ_{ref}=60 MPa)

Table 4.4: Summary of results with domain thickness 30mm

		Magnification factor k				
σ_{ref}		*250*	*275*	*500*	*750*	*1000*
10	Iterations	3	3	2	--	--
	Max stress	81.53	77.33	86.93	--	--
	Ellipse axes ratio	1.64	1.77	2.17	--	--
40	Iterations	6	5	3	--	--
	Max stress	73.41	74.2	79.3	--	--
	Ellipse axes ratio	1.94	1.86	2.01	--	--
60	Iterations	15	13	8	5	3
	Max stress	72.42	72.75	71.78	71.88	70.71
	Ellipse axes ratio	1.64	1.77	1.91	1.91	1.80

4.4 Optimization of a Plate with a Hole with Domain Thickness 40 mm

The finite element models and the boundary conditions for stress and thermal analysis are shown in Figure 4.47 for domain thickness 40 mm. The von Mises stresses of stress analysis and total deflections of the thermal expansion analysis of the original shape can be seen in Figure 4.48. Similar to the analysis conducted for domain thickness 10 mm, the stress far away from the concentration zones is about 40 MPa. The maximum and minimum von Mises stresses were also around 130 MPa and 10 MPa respectively.

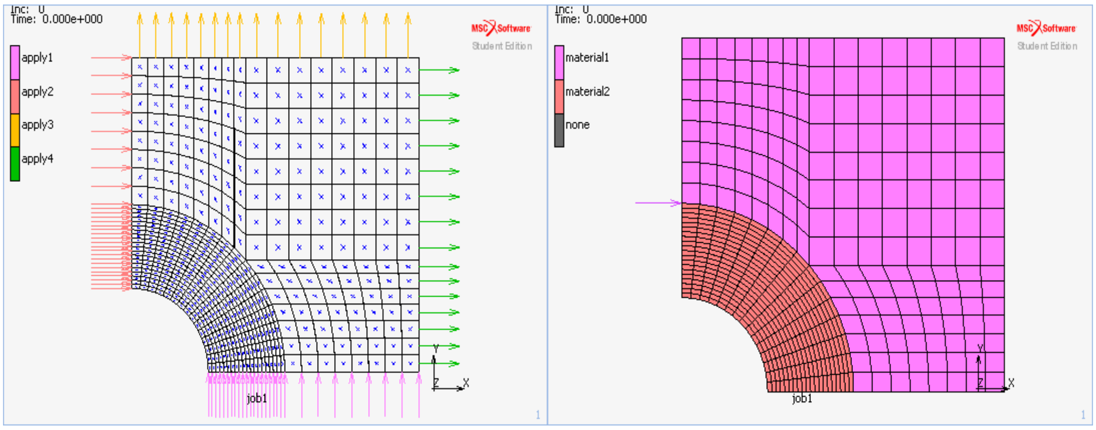


Figure 4.47: Finite element model for stress (left) and thermal (right) analysis (D=40mm)

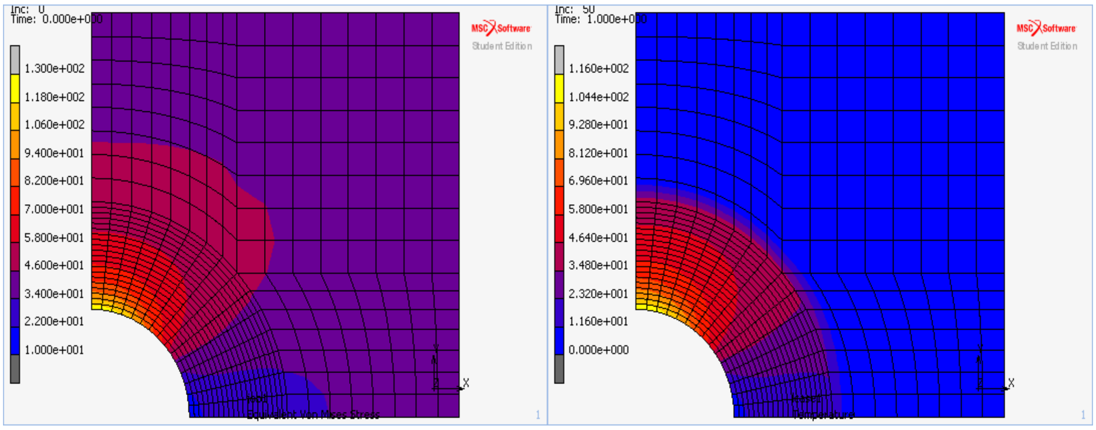
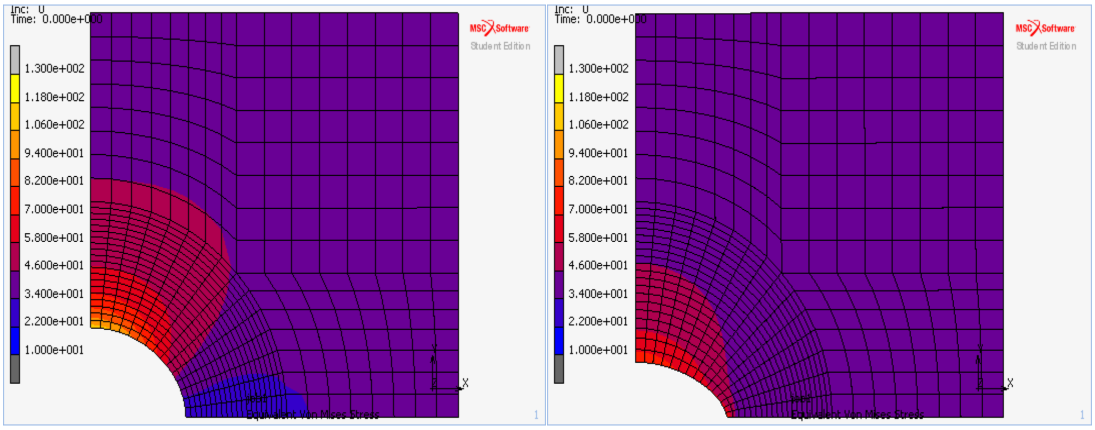
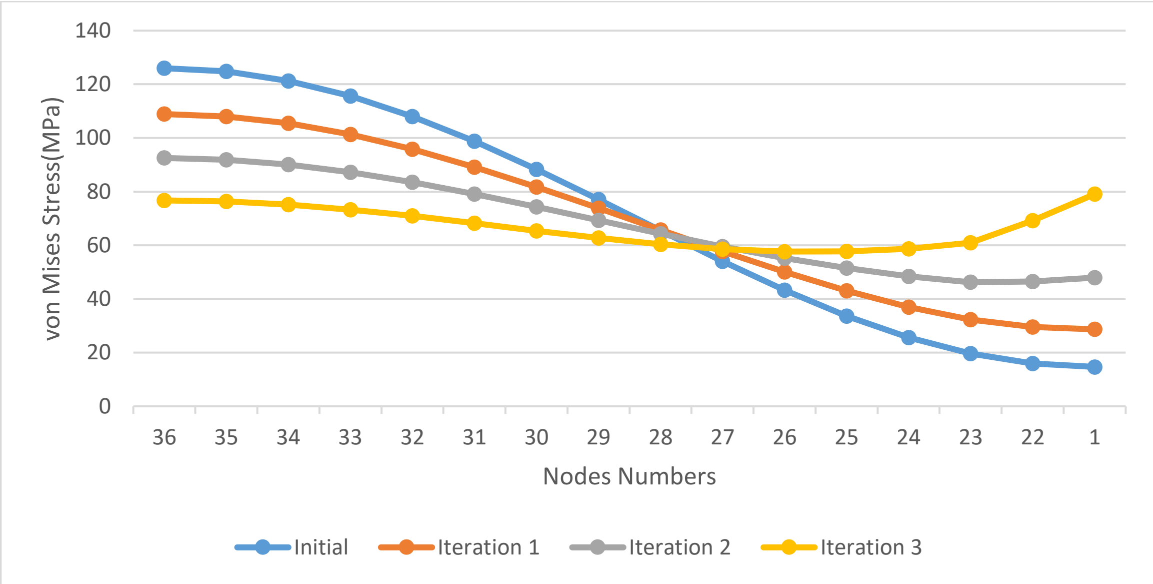


Figure 4.48: von Mises stresses (left) and thermal deformations (right) of the original shape (D =40mm)

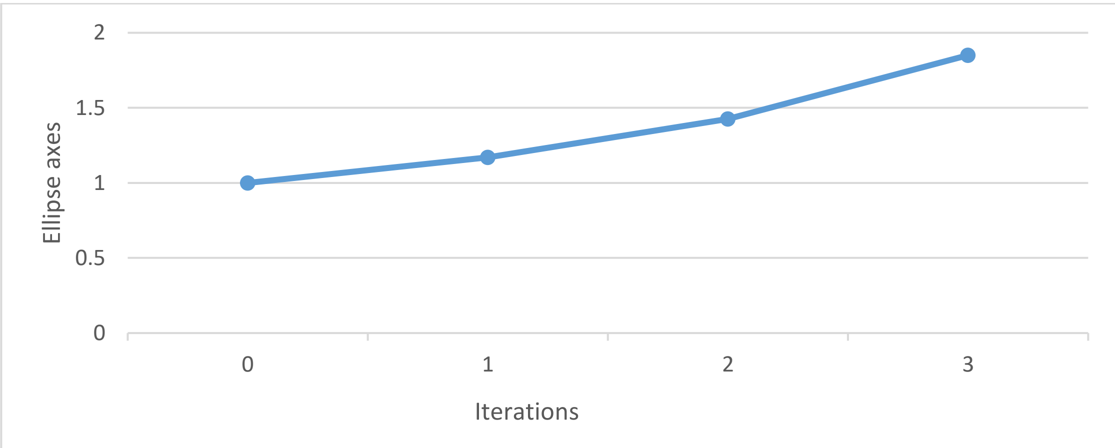
For domain thickness D = 40 mm, nine optimization analyses were conducted including the combinations of reference stresses σ_{ref} = 10, 40 and 60 MPa and magnification factor k = 100, 200, 250, 275 and 500. The results obtained are given in Figures 4.49 to 4.57. The results, summarized in Table 4.5, showed that σ_{ref} = 40 MPa with magnification factors around 200 and 250 were trustable for further investigations. In the following section the method was extended to optimization of the geometry modeled by automatic mesh generation.



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 3 (right)

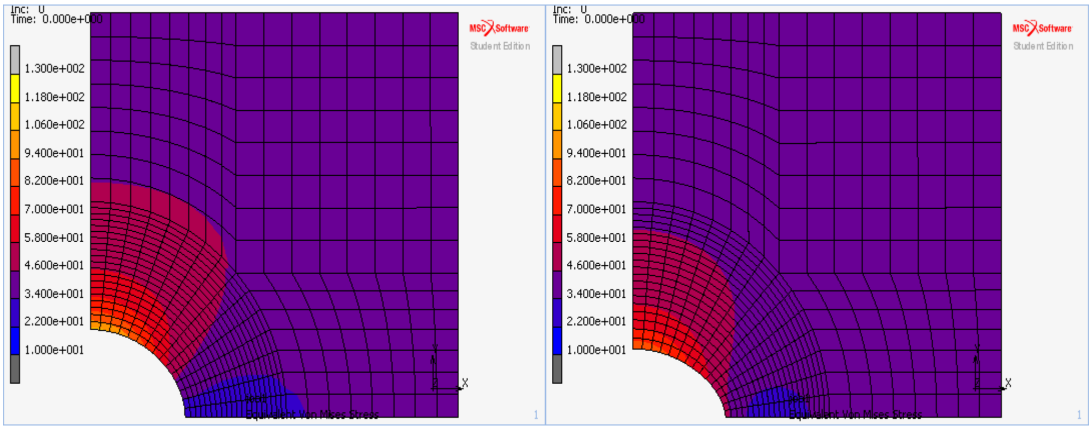


(b) Variation of von Mises stress distribution along the hole boundary

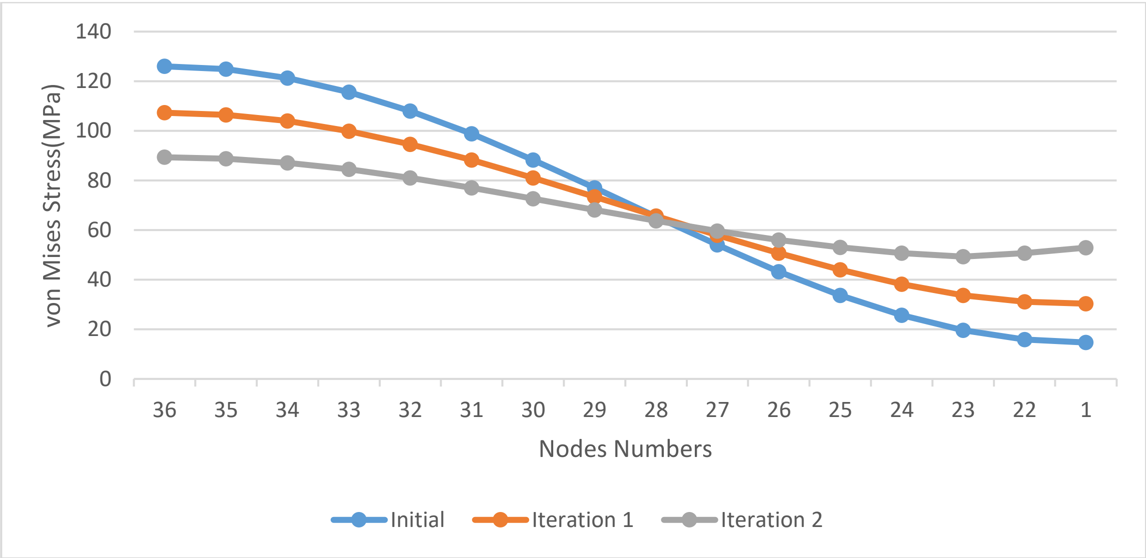


(c) Variation of ellipse axes ratio

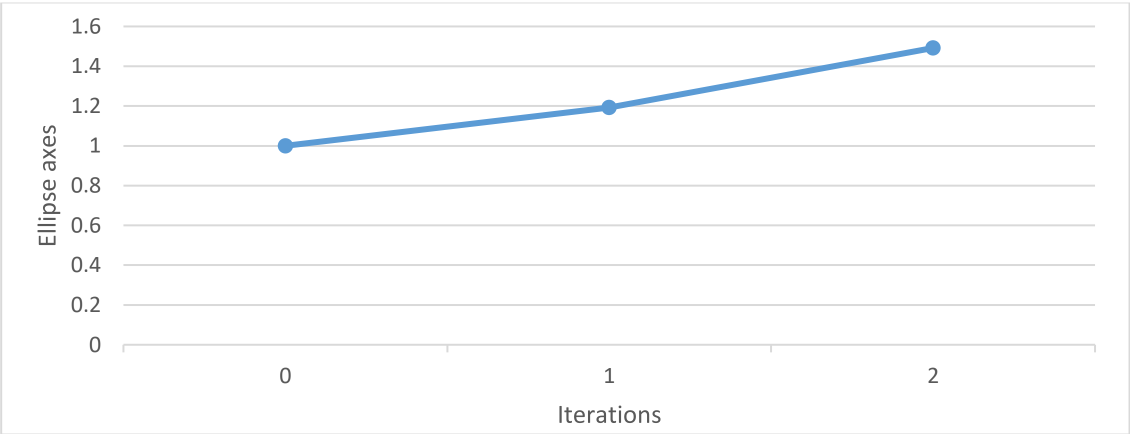
Figure 4.49: Optimization results of a plate with a hole (D=40 mm, K=250, σ_{ref} =10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 2 (right)

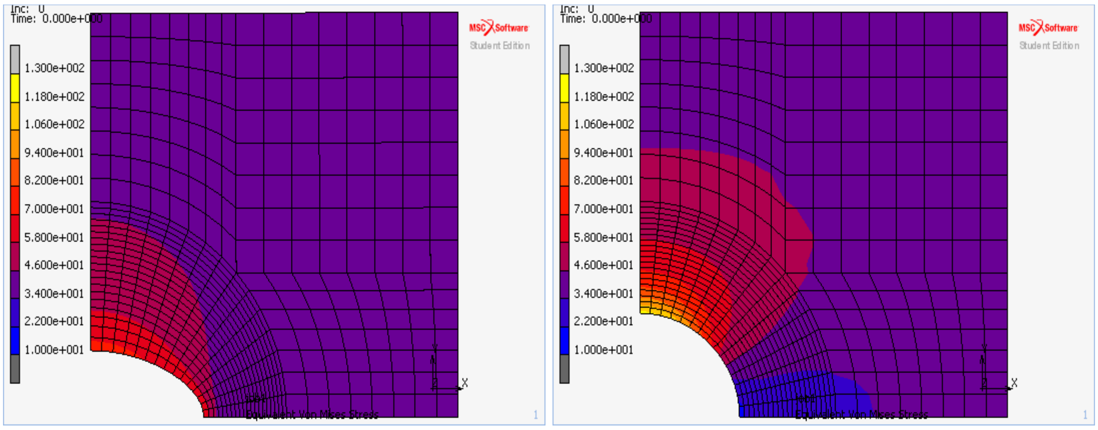


(b) Variation of von Mises stress distribution along the hole boundary

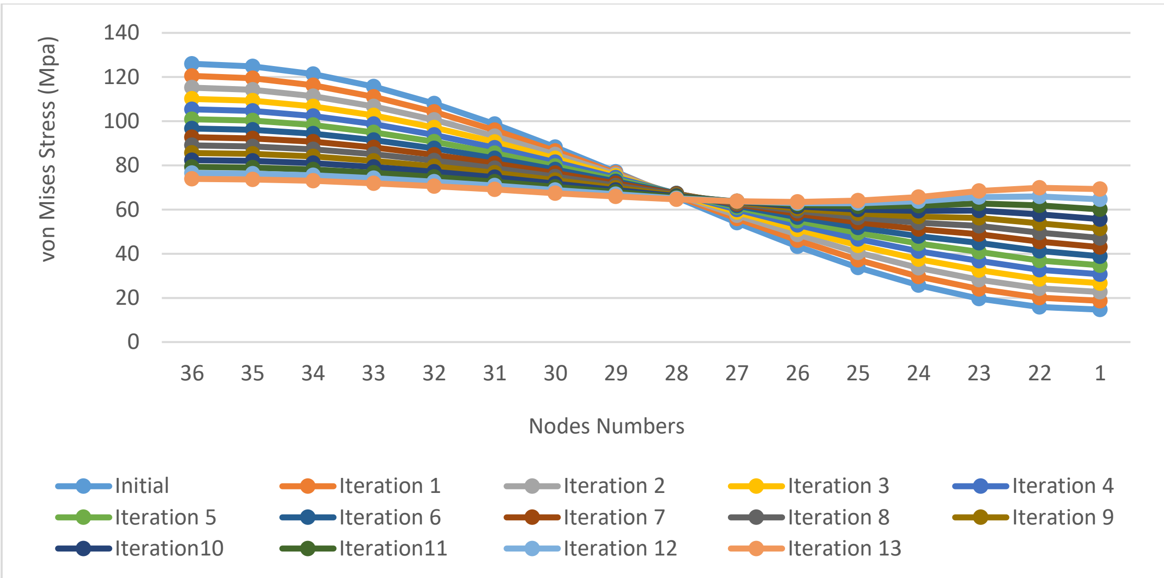


(c) Variation of ellipse axes ratio

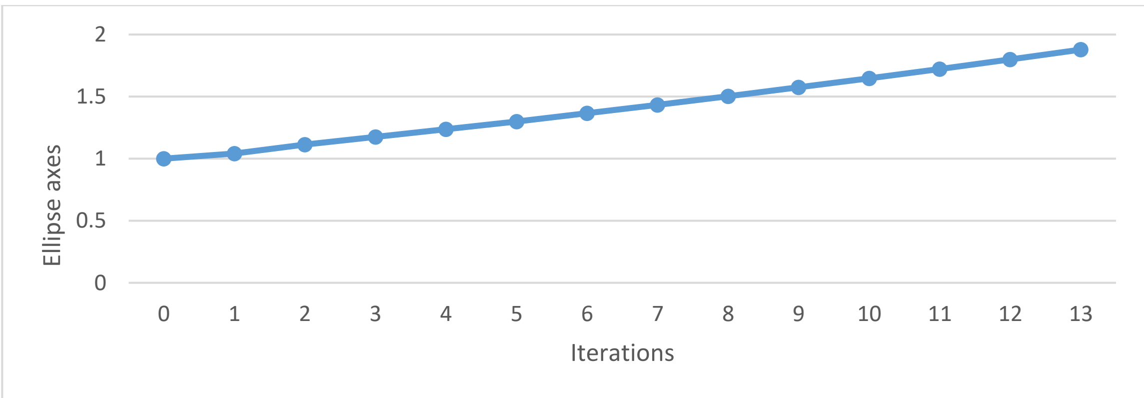
Figure 4.50: Optimization results of a plate with a hole (D=40 mm, K=275, σ_{ref} =10 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 13 (right)

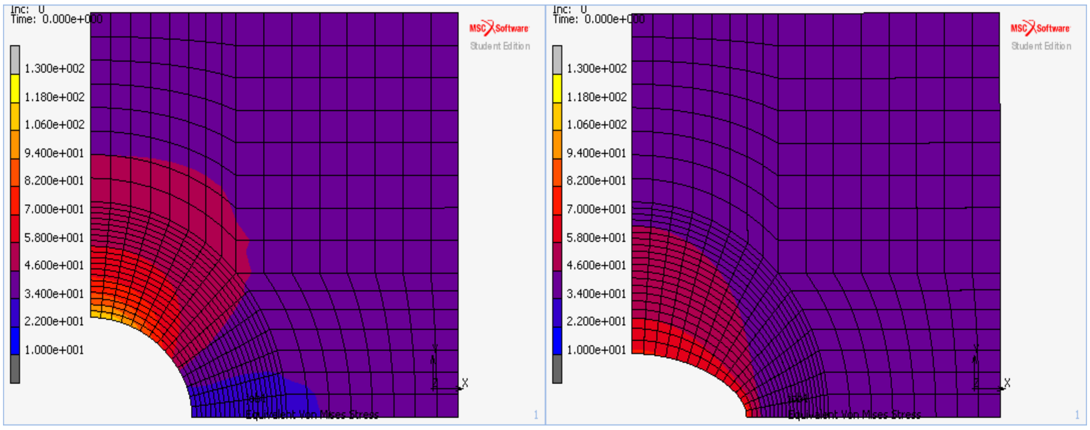


(b) Variation of von Mises stress distribution along the hole boundary

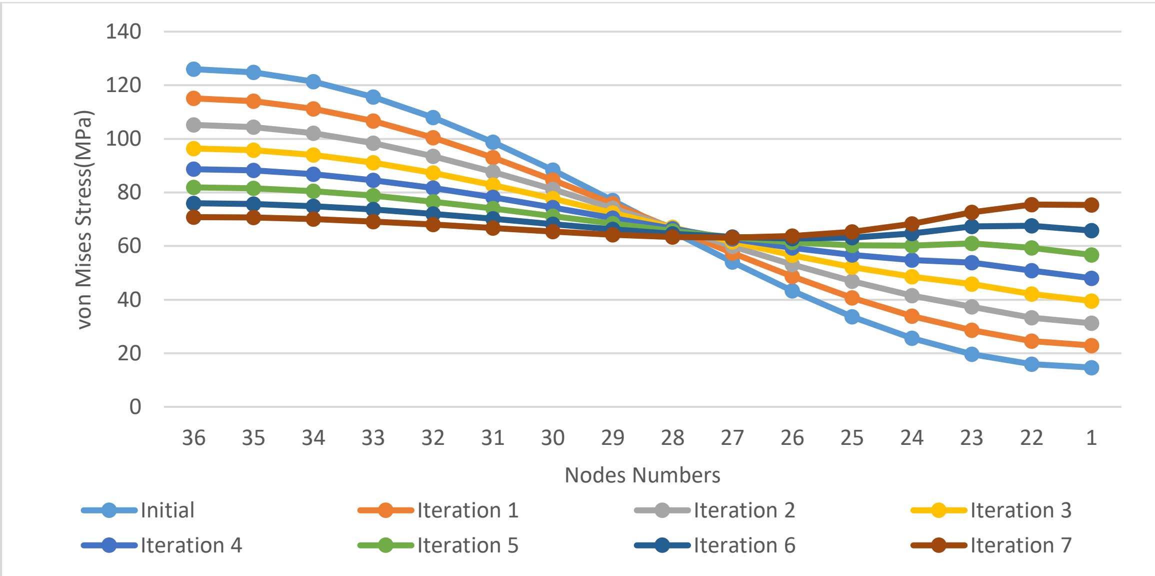


(c) Variation of ellipse axes ratio

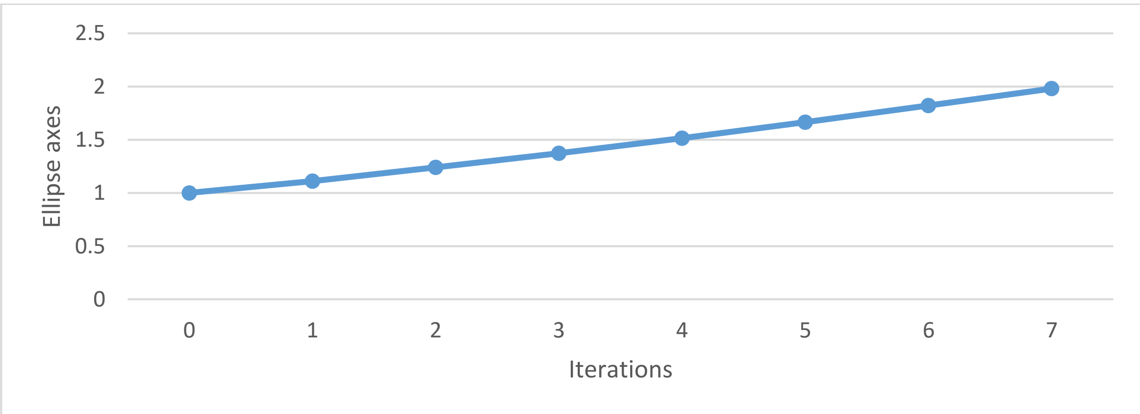
Figure 4.51: Optimization results of a plate with a hole (D=40 mm, K=100, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 7 (right)

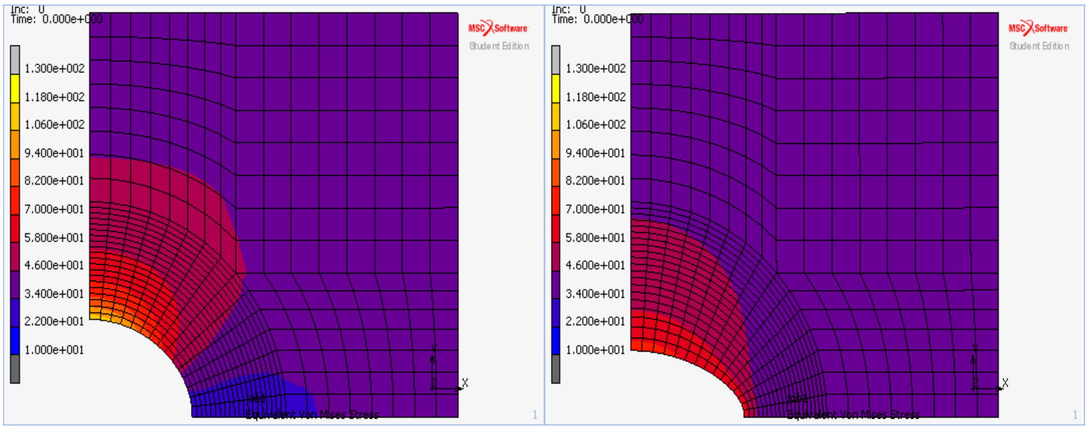


(b) Variation of von Mises stress distribution along the hole boundary

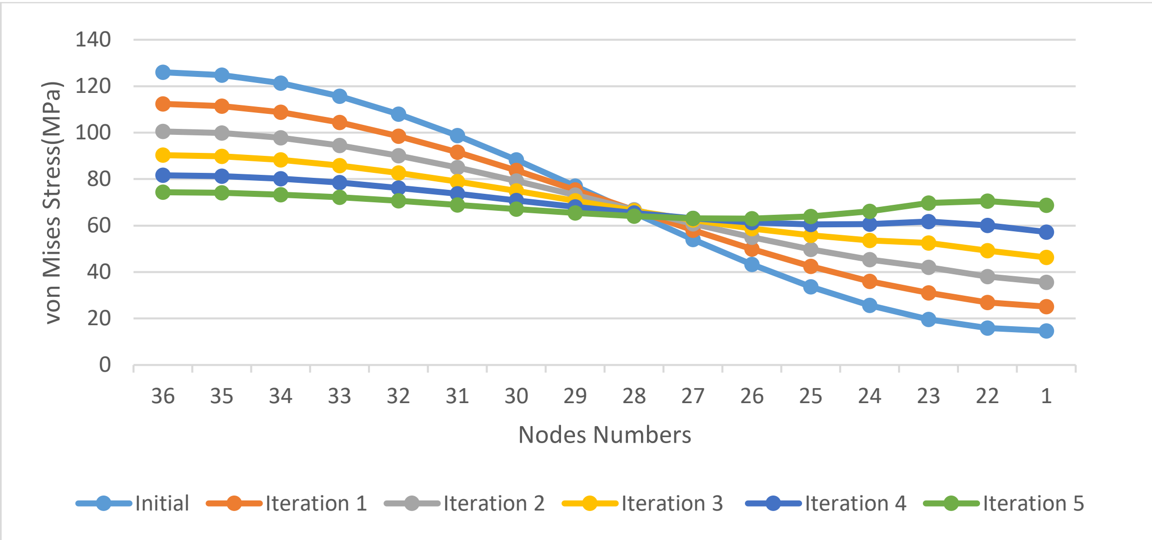


(c) Variation of ellipse axes ratio

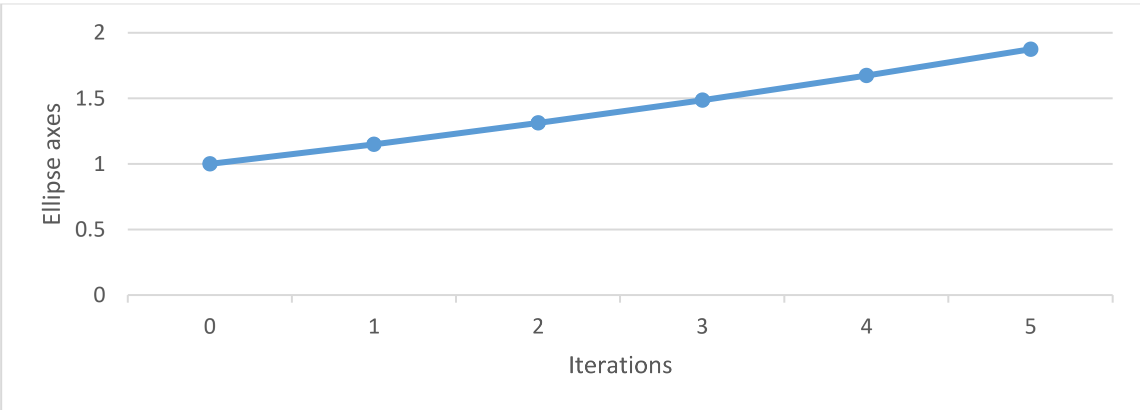
Figure 4.52: Optimization results of a plate with a hole (D=40 mm, K=200, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)

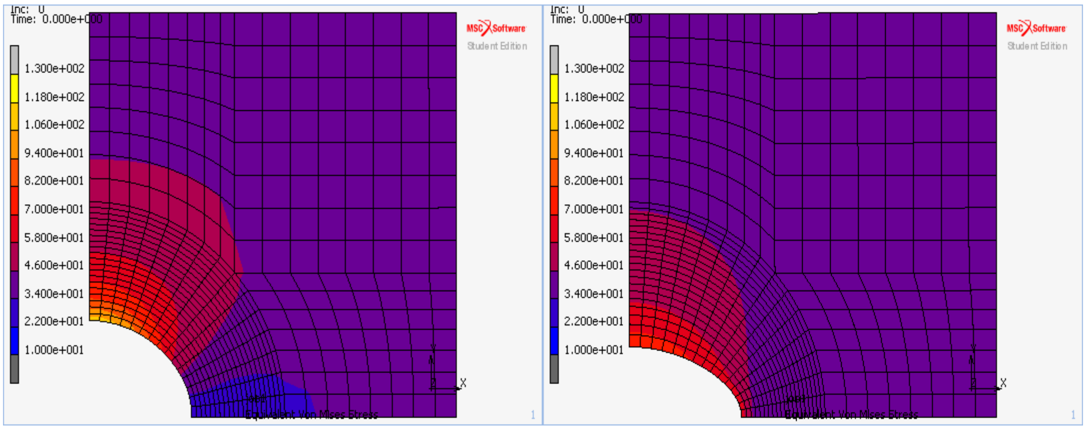


(b) Variation of von Mises stress distribution along the hole boundary

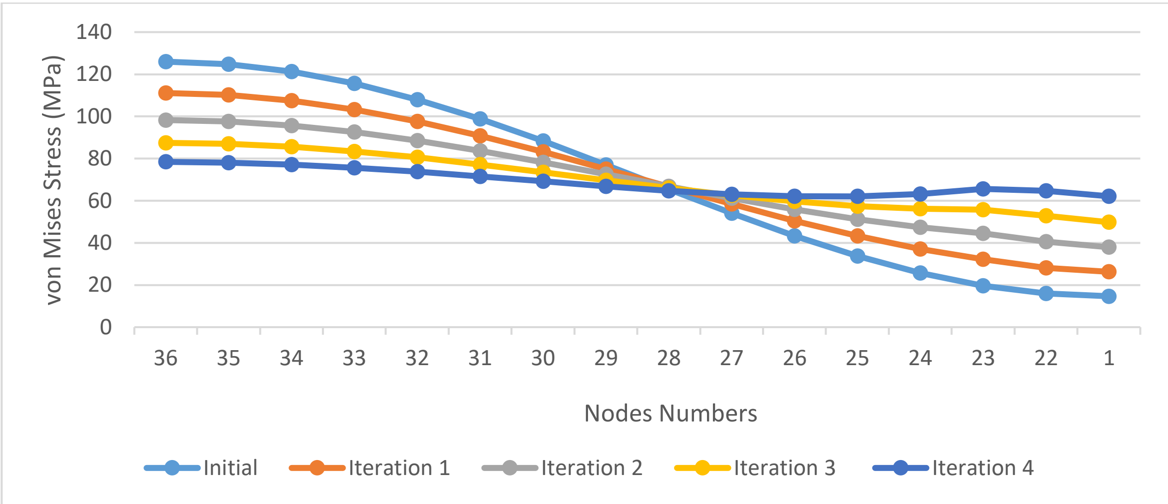


(c) Variation of ellipse axes ratio

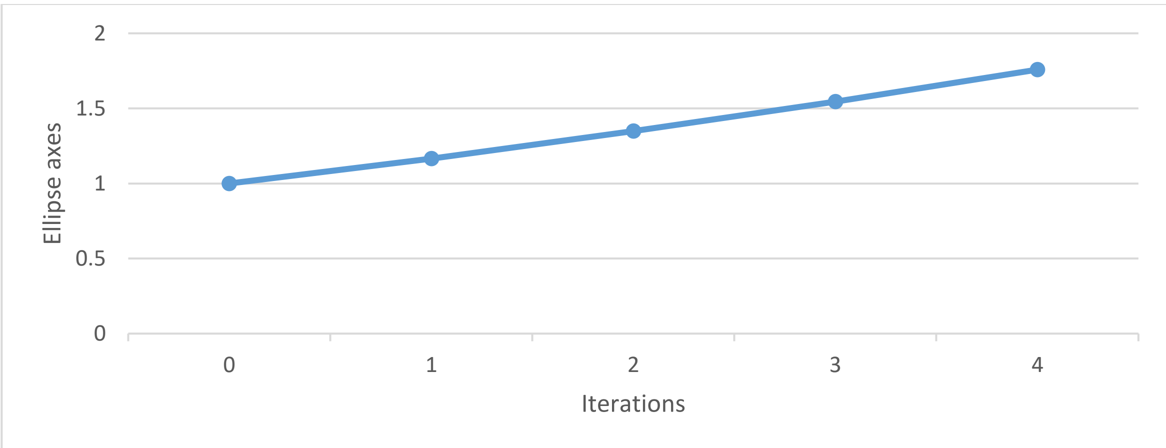
Figure 4.53: Optimization results of a plate with a hole (D=40 mm, K=250, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 4 (right)

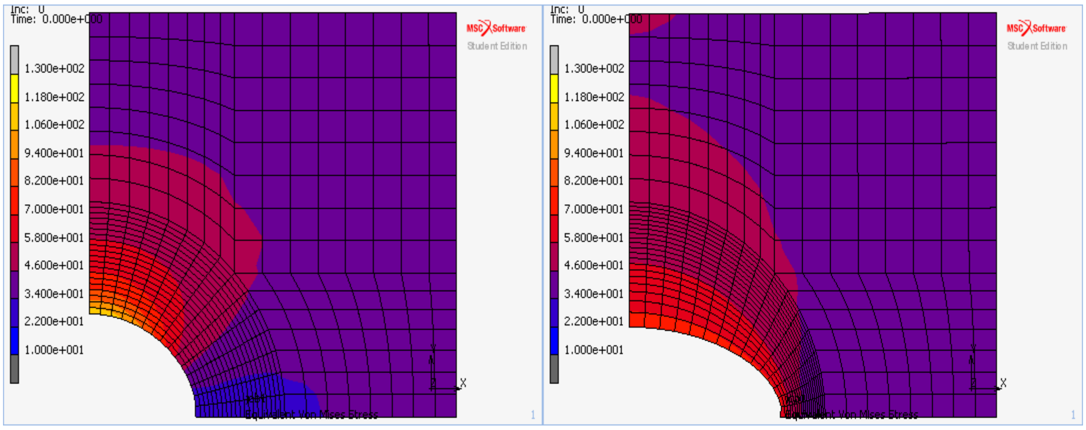


(b) Variation of von Mises stress distribution along the hole boundary

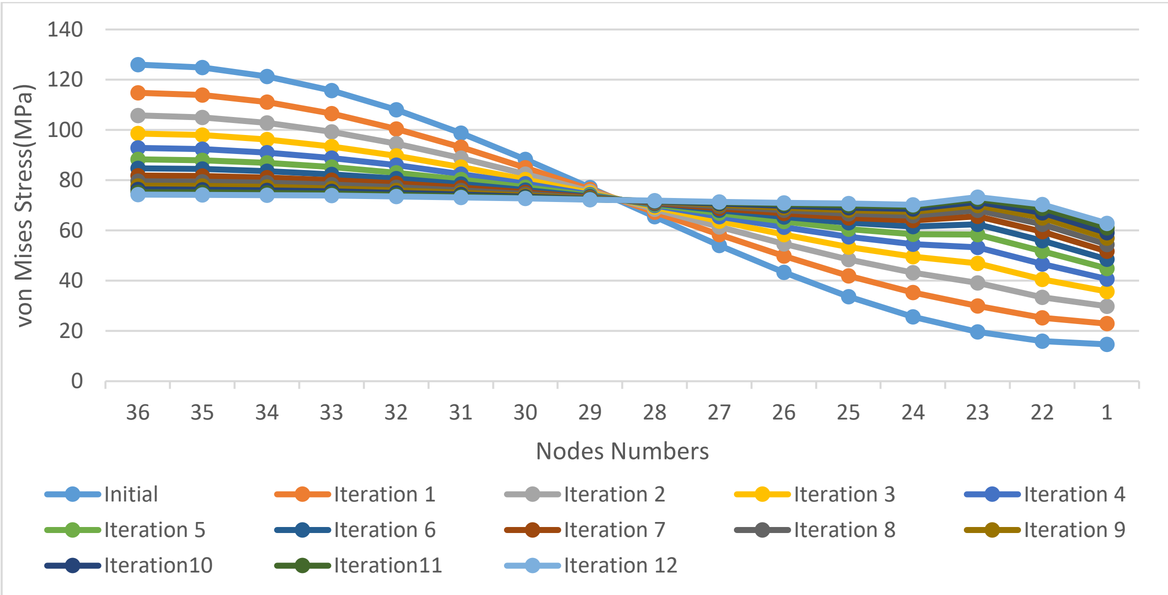


(c) Variation of ellipse axes ratio

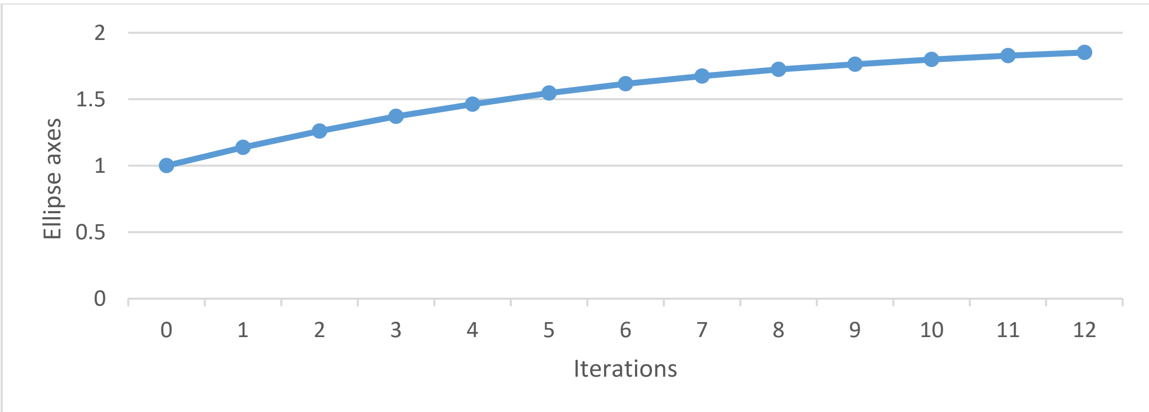
Figure 4.54: Optimization results of a plate with a hole (D=40 mm, K=275, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 12 (right)

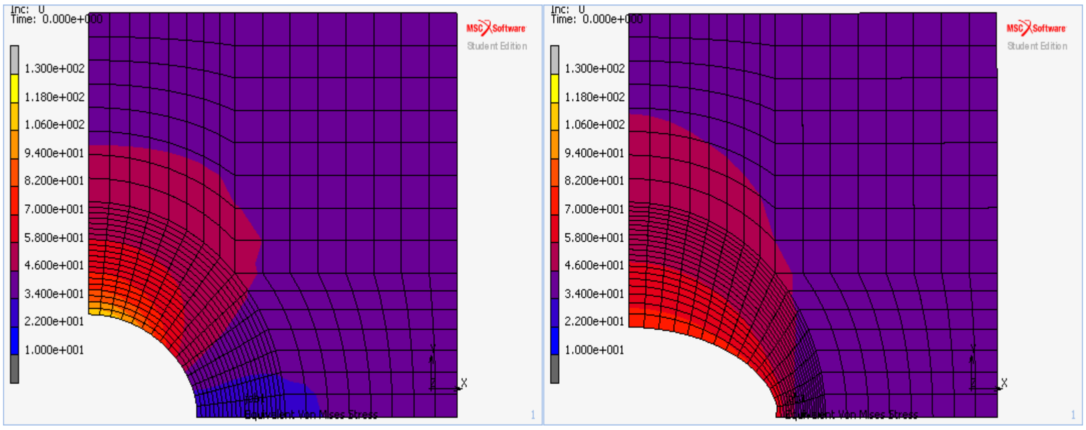


(b) Variation of von Mises stress distribution along the hole boundary

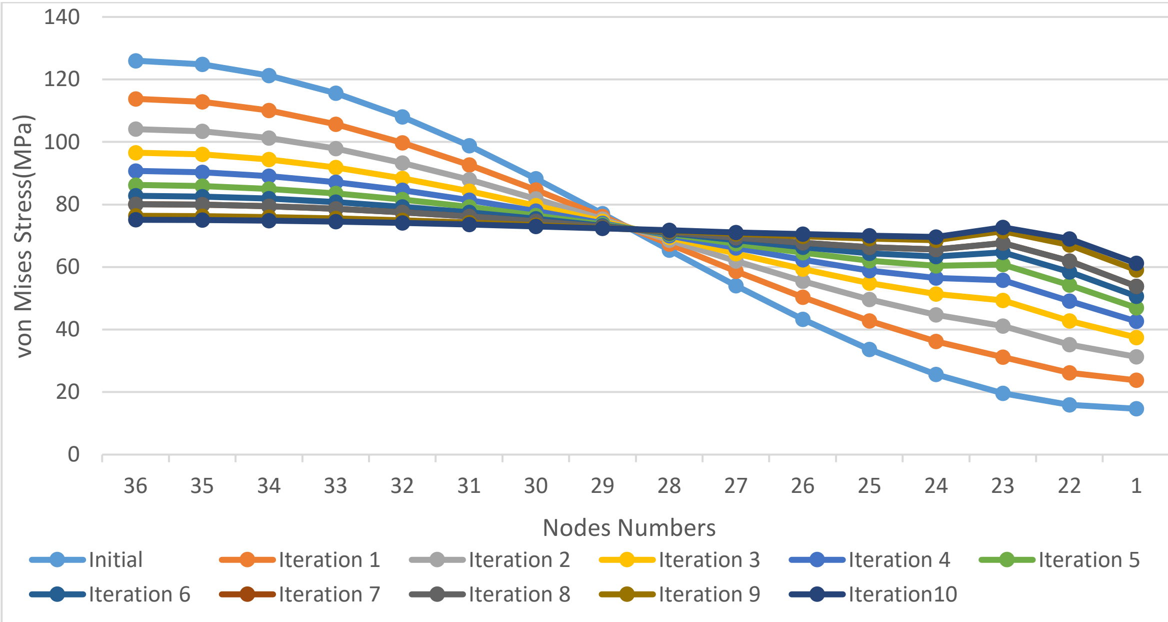


(c) Variation of ellipse axes ratio

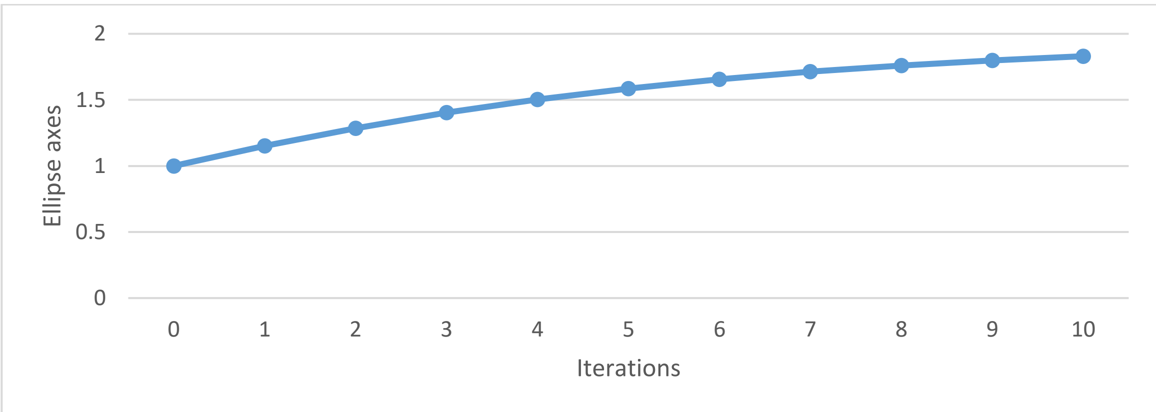
Figure 4.55: Optimization results of a plate with a hole (D=40 mm, K=250, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 10 (right)

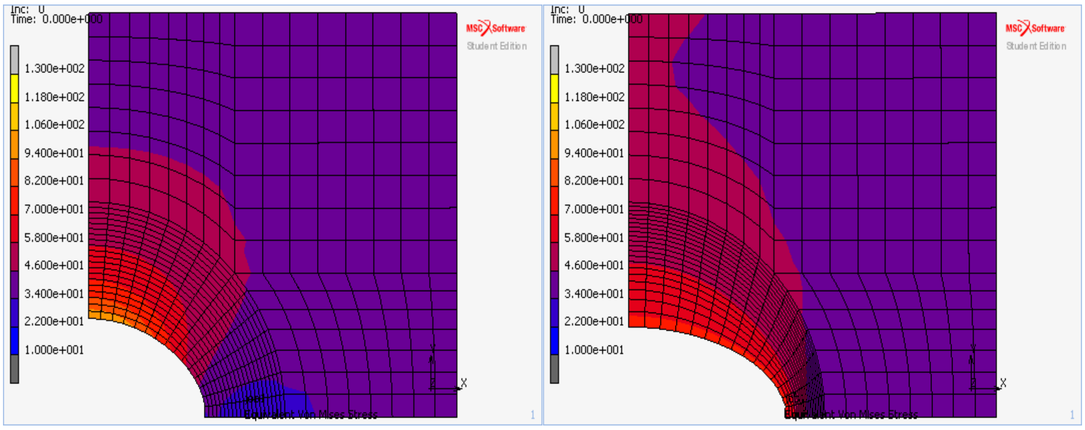


(b) Variation of von Mises stress distribution along the hole boundary

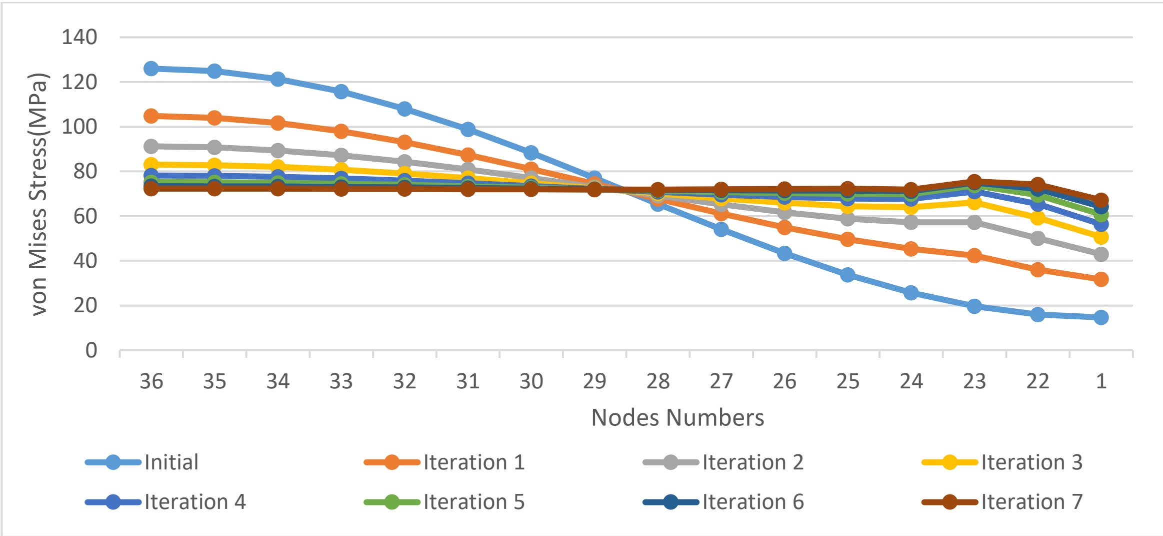


(c) Variation of ellipse axes ratio

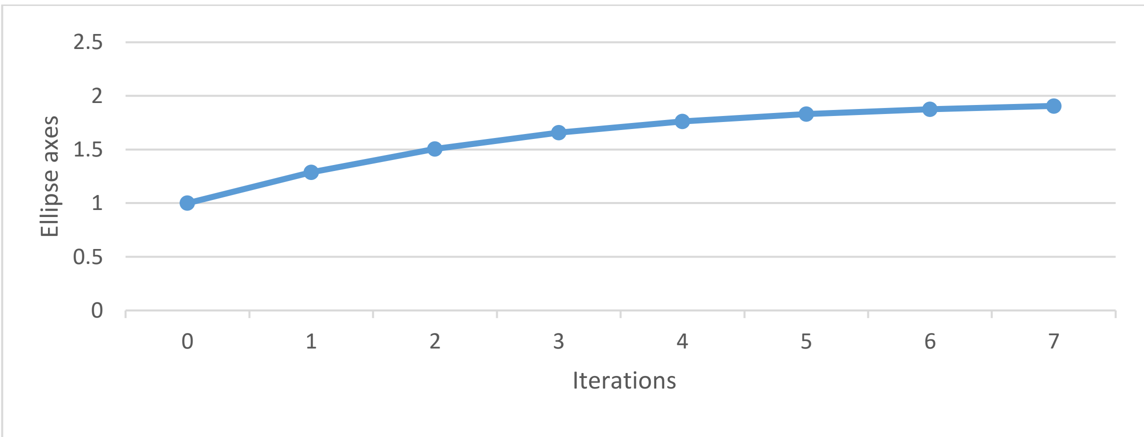
Figure 4.56: Optimization results of a plate with a hole (D=40 mm, K=275, σ_{ref} = 60 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 7 (right)



(b) Variation of von Mises stress distribution along the hole boundary



(c) Variation of ellipse axes ratio

Figure 4.57: Optimization results of a plate with a hole (D=40 mm, K=500, σ_{ref} = 60 MPa)

Table 4.5: Summary of results with domain thickness 40mm

		Magnification factor k				
σ_{ref}		*100*	*200*	*250*	*275*	*500*
10	Iterations	--	--	3	2	--
	Max stress	--	--	79.05	89.36	--
	Ellipse axes ratio	--	--	1.85	1.49	--
40	Iterations	13	7	5	4	--
	Max stress	73.88	75.48	74.35	76.03	--
	Ellipse axes ratio	1.87	1.98	1.87	1.75	--
60	Iterations	--	--	12	10	7
	Max stress	--	--	74.89	75.11	80.09
	Ellipse axes ratio	--	--	1.85	1.87	1.90

4.4 Optimization of a Plate with a Hole with Auto-Mesh

The finite element models and the boundary conditions for stress and optimization domain in the thermal analysis are shown in Figure 4.58. The area indicated by material2 was considered as the optimization domain and a reduced Young's modulus was assigned as given in Table 4.1. Optimization domain was defined as the area effected by the dislocations in the continuum, i.e. the area effected in the plane by the hole, and determined from the results of initial stress analysis of the geometry as shown in Figure 4.59. The stress far away from the hole was about 40 MPa and therefore σ_{ref} = 40 MPa was selected.

The results obtained for magnification factor k = 200, 250, given in Figures 4.60 to 4.61 and summarized in Table 4.6, were very close to the analyses conducted before in the present study and also by Tekkaya (1996).

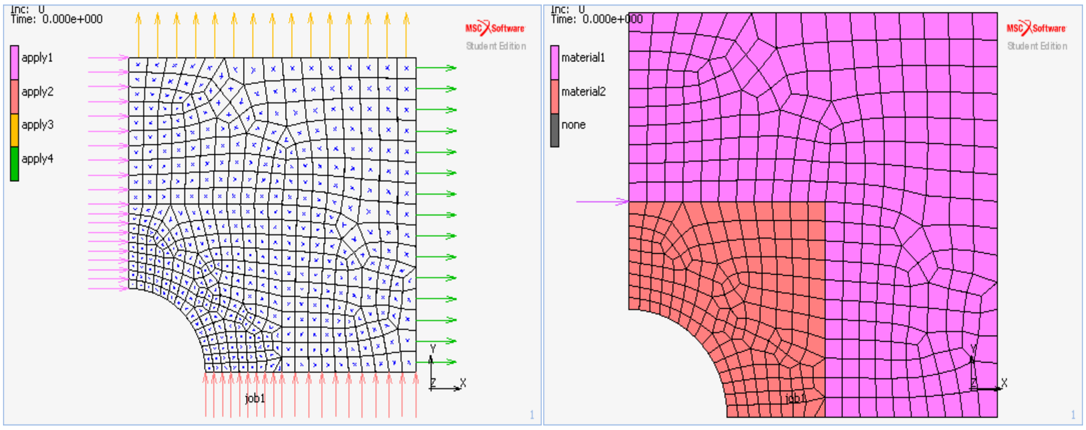


Figure 4.58: Finite element model for stress (left) and thermal (right) analyses (Auto-mesh)

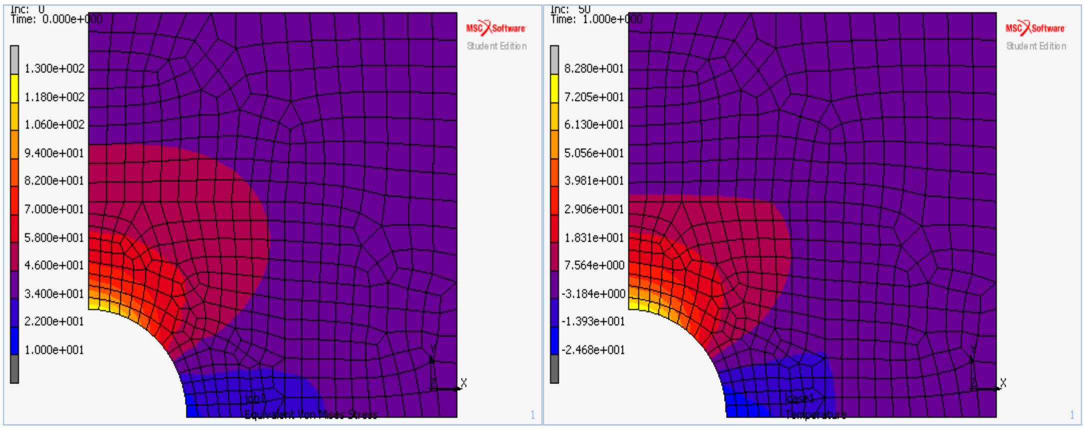
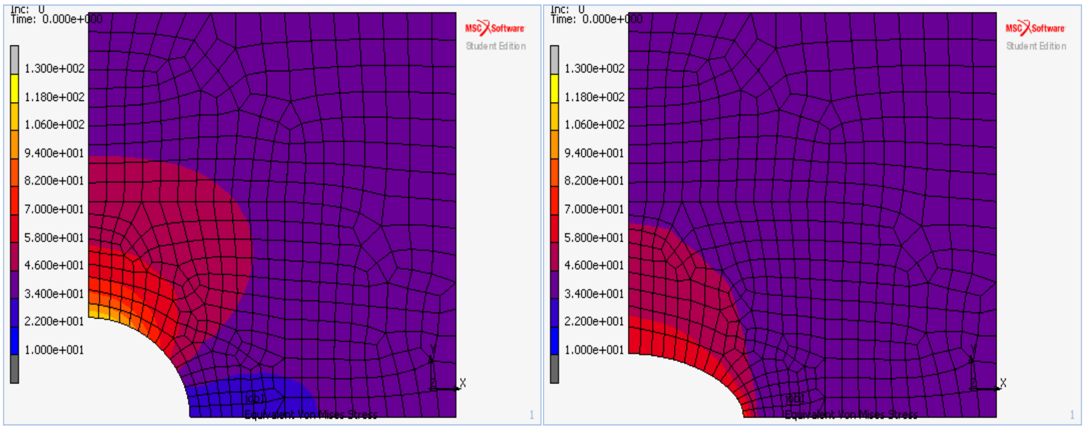
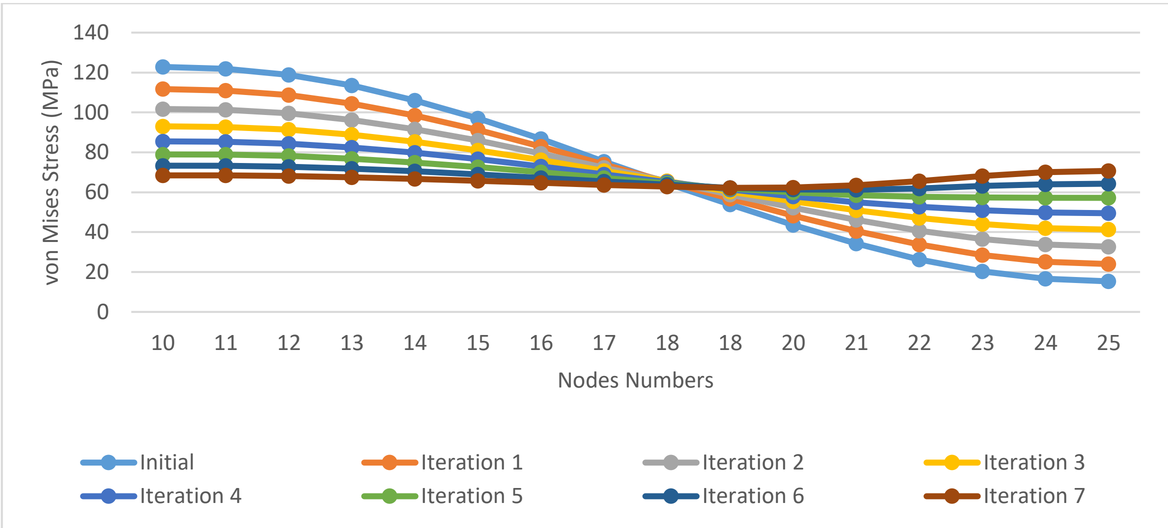


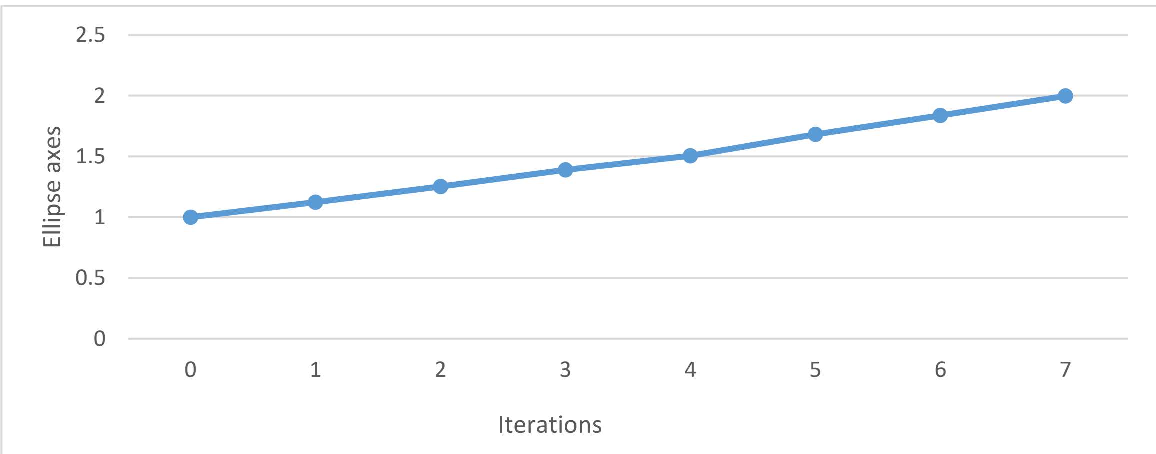
Figure 4.59: von Mises stresses (left) and thermal deformations (right) of the original shape (Auto-mesh)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 7 (right)

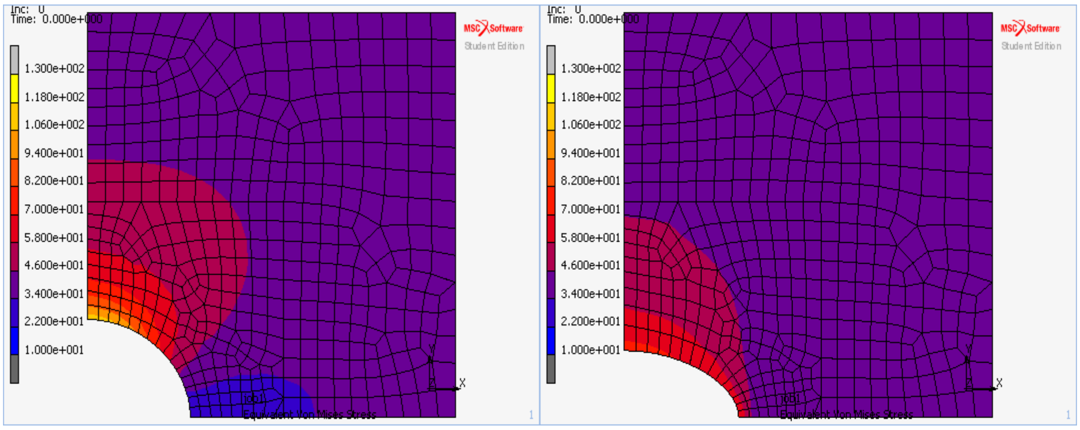


(b) Variation of von Mises stress distribution along the hole boundary

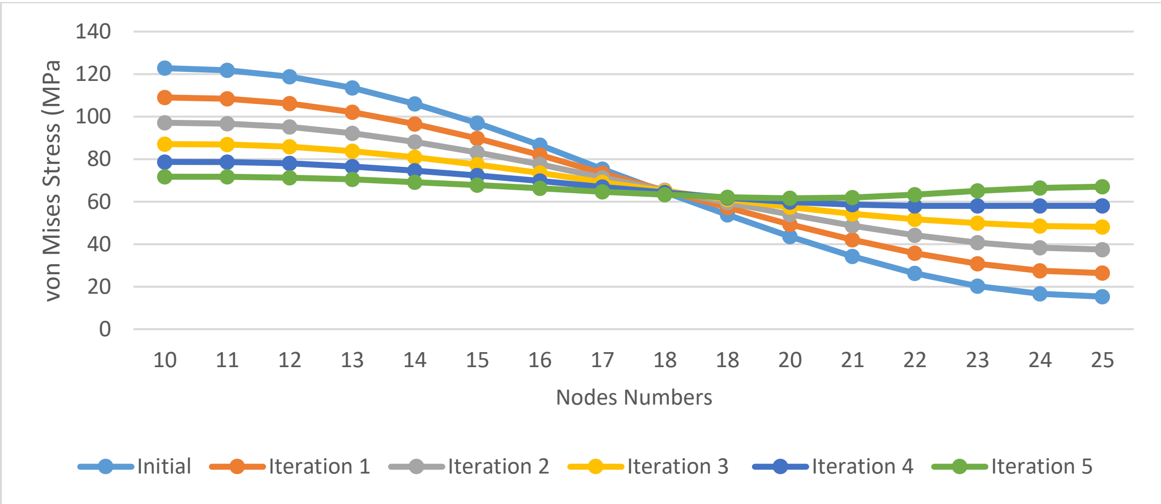


(c) Variation of ellipse axes ratio

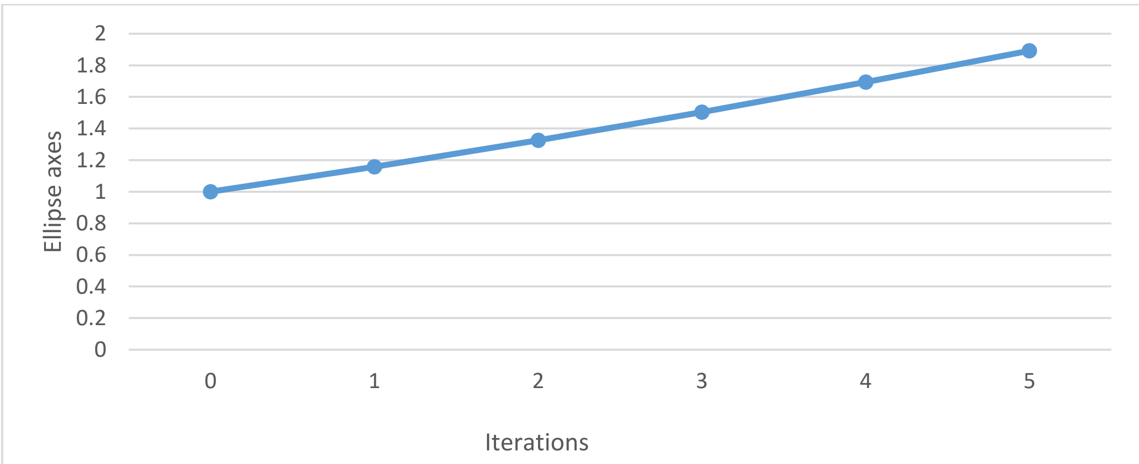
Figure 4.60: Optimization results of a plate with a hole (Auto-mesh K=200, σ_{ref} = 40 MPa)



(a) von Mises stresses for first and last iterations: Iteration 1 (left), Iteration 5 (right)



(b) Variation of von Mises stress distribution along the hole boundary



(c) Variation of ellipse axes ratio

Figure 4.61: Optimization results of a plate with a hole (Auto-mesh K=250, σ_{ref} = 40 MPa)

Table 4.6: Summary of results with auto-mesh

		Magnification factor k	
σ_{ref}		*200*	*250*
40	Iterations	7	5
	Max stress	70.68	71.73
	Ellipse axes ratio	1.99	1.89

CHAPTER 5

CONCLUSIONS

Biological Growth Method (BGM) is a structural shape optimization method, generally based on the natural optimization of trees under their own weights or external loads. Swelling or shrinking takes place at their outer layers decreasing the localized stresses in their body. In BGM, the outer layer that will shrink or swell is called the optimization domain. The von Mises stresses developed in the optimization domain during structural stress analyses were related to temperature differences in the consequent thermal expansion analysis to determine the magnitudes of swelling and shrinking. The study aimed to investigate the effect of domain thickness on BGM.

During the application of the method the finite element code, pre- and post-processor MARC-MENTAT student version was used. A software named Biological Growth Interface (BGI) was developed to interactively control and modify the data in the input and output files. The procedure was verified by conducting the parametric study of the plane with a hole problem of Tekkaya (1996). In the study, the effects of magnification factor and the reference stress were investigated with a constant domain thickness of 10 mm.

From the interpretation of the results of the verification analyses, it was concluded that as the magnification factor increased the number of iterations required for the optimization decreased. On the other hand, increasing the reference stress towards the maximum stress level expected after the optimization, increased the number of iterations. As the number of iterations increased the method gave better results. A minimum of 5-6 iterations seemed to be a must for the plane with a hole problem. 130 MPa of maximum von Mises stress around the hole before the optimization dropped down to about 70 MPa, the ellipse axes ratio being 1 for the circular hole resulted around 2 after the optimization procedure was applied.

The maximum stress before the optimization was at the hole boundary and generally the stress concentration was in the optimization domain of thickness 10 mm. Far away from the hole, the stress level drops down to the values that the plane would have without the hole. However, the stress levels in the vicinity of the 10 mm domain were still remarkable. To include the effect, the method was applied to the same problem with domain thicknesses 20, 30 and 40 mm. The maximum von-Mises stresses and ellipse axes ratios were determined to be between 70-75 MPa and 1.7-2.0, respectively. The number of iterations required decreased. Still, a minimum of 5-6 iterations seemed to be a must for better results.

It should be noted that the main difference between the present study and that of Tekkaya (1996) was that in the second, the mesh was re-generated after each iteration to keep the domain thickness at 10 mm and to eliminate the distortion of elements. In the present study, the mesh was not re-generated but the coordinates of the nodes were changed according to the deflections, i.e. so-called swelling and shrinking action, resulting in distorted or high aspect ratio elements effecting the results in a negative way.

The problem of mesh distortion can be resolved by adopting an automatic mesh generation scheme after each iteration and may be considered as a future work. However, the outcomes of the study turned the attention to form a model with larger elements to prevent distortion and aspect ratio problems with automatic mesh generation to have the benefit of easiness in modelling. The optimization domain roughly selected to include remarkable stress changes around the hole boundary. Reference stress was assigned as the stress level far away from the hole. Since the optimization domain is large magnification factor was taken as low to have enough number of iterations for acceptable results. A magnification factor of 200 resulted with a maximum von Mises stress of 70.68 MPa and an ellipse ratio of 1.99 after 7 iterations.

REFERENCES

Baumgartner, A., Harzheim, L., & Mattheck, C. (1992). SKO (soft kill option): the biological way to find an optimum structure topology. *International Journal of Fatigue*, 14(6), 387-393.

Budynas, R. G., & Nisbett, J. K. (2008). *Shigley's mechanical engineering design* (Vol. 9). New York: McGraw-Hill.

Cai, R., Cai, S., Yang, X., & Lu, F. (1998). A novel method of structural shape optimization coupling BEM with an optimization method based on biological growth. *Structural and Multidisciplinary Optimization*, 15(3), 296-300.

Cardona, A., Nigro, N., Sonzogni, V., & Storti, M. (2006). The coupling of the biological growth method and the boundary element method for structural shape optimization. *Mecánica Computacional*, *25*, 2827-2834

Chen, J. L., & Tsai, W. C. (1993). Shape optimization by using simulated biological growth approaches. *AIAA journal*, 31(11), 2143-2147.

Le Riche, R., & Cailletaud, G. (1998). A mixed evolutionary / heuristic approach to shape optimization. *International Journal for Numerical methods in Engineering*, 41, 1463-1484.

MARC-MENTAT Finite Element Software, Newport Beach, CA 92660, USA. Retrieved November 19, 2017 from http://www.mscsoftware.com/page/msc-software

Mattheck, C., Burkhardt, S., & Erb, D. (1991). Shape optimization of engineering components by adaptive biological growth. *In Engineering optimization in design processes* (pp. 15-24). Berlin: Proceedings of the International Conference

Mattheck, C., Moldenhauer, H. (1990). An intelligent CAD method based on biological growth. *Fatigue and Fracture of Engineering Materials & Structures*, 13(1), 41-51.

Peng, D., & Jones, R. (2009). A CAD-based on biological method for designing optimal fatigue life. *Structural and Multidisciplinary Optimization*, 37(3), 295-304.

Tang, J. (2011). Developing evolutionary structural optimization techniques for civil engineering applications Retrieved November 19, 2017 from https://researchbank.rmit.edu.au/view/rmit:160087.

Tekkaya, A. Erman, & Güneri, A. (1996). Shape optimization with the biological growth method: a parameter study. *Engineering computations*, 13(8), 4-18.

Tian, Z. M. H., & Shangjin, W. (2004). Modified biological growth method and its application to shape optimization of centrifugal impellers. *Chinese Journal of Mechanical Engineering*, 12, 1-6.

Wessel, C., Cisilino, A., & Sensale, B. (2004). Structural shape optimization using boundary elements and the biological growth method. *Structural and Multidisciplinary Optimization*, 28(2), 221-227.

Zehsaz, M., Torkanpouri, K. E., & Paykani, A. (2013). Convergence of shape optimization calculations of mechanical components using adaptive biological growth and iterative finite element methods. *Journal of Central South University*, 20(1), 76-82.

APPENDICES

APPENDIX 1

StressV1.dat

```
title               job1
$....MARC input file produced by Marc Mentat 2016.0.0 (64bit) Student Edition
$....................................
$....input file using extended precision
extended
$....................................
sizing                                  0       629       680         0
alloc                        25
elements                      3
version                      11
table                         0         0         2         1         1         0         0         1
processor                     1         1         1         0
$no list
elastic                       0
all points
no echo                       1         2         3
end
$..................
solver
         8         0         0         0         0         0         0         0         0         0
0         0         0         0         0         0
optimize                     11
connectivity
         0         0         1
         1         3        28        29        85        27
         2         3        29        30        86        85
         3         3        30        31        87        86
         ;         ;         ;         ;         ;         ;
         ;         ;         ;         ;         ;         ;
         ;         ;         ;         ;         ;         ;
       627         3       704       684       701       710
       628         3       688       709       703       700
       629         3       709       688       680       656
coordinates
         3       680         0         1
         1 1.000000000000000+2 0.000000000000000+0 0.000000000000000+0
         2 1.100000000000000+2 0.000000000000000+0 0.000000000000000+0
```

```
        3 1.200000000000000+2 0.000000000000000+0 0.000000000000000+0
        ;          ;        ;         ;         ;         ;
        ;          ;        ;         ;         ;         ;
        ;          ;        ;         ;         ;         ;
       709 3.990607755390000+1 5.560678049811000+1 0.000000000000000+0
       710 5.830118186922000+1 4.358156739948000+1 0.000000000000000+0
       711 5.258083798993000+1 5.388790137404000+1 0.000000000000000+0
define             node               set                apply1_nodes
         50          51          52          53          54          55          358          359
360        361         362         363         364   c
        365         366         367         368         369          370          371          372
define             node               set                apply2_nodes
          1           2           3           4           5           6          312          313
314        315         316         317         318   c
        319         320         321         322         323          324          325          326
define             edgemt             set                apply3_edges
              1:0               2:0               3:0               4:0               5:0
6:0               7:0               8:0   c
              9:0              10:0              11:0              12:0              13:0
14:0              15:0              16:0   c
             17:0              18:0              19:0              20:0              21:0
22:3
define             edgemt             set                apply4_edges
              1:3              53:0              54:0              55:0              56:0
57:0              58:0              59:0   c
             60:0              61:0              62:0              63:0              64:0
65:0              66:0              67:0   c
             68:0              69:0              70:0              71:0              72:0
73:0
define             ndsq               set                Domain_Nodes
          1          55          56          57          58          59           60           61
62         63          64          65          66   c
         67          68          69          70          71          72           73           74
75         76          77          78          79   c
         80          81          82          83          84         312          313          314
315        316         317         318         319   c
        663         664         665         666         667          668          669          670
```

```
671         672         673         674         675   c
        676         677         678         679         680         681         682         683
684         685         686         687         688   c
        689         690         691         692         693         694         695         696
697         698         699         700         701   c
        702         703         704         705         706         707         708         709
710         711
define              element             set                 Domain_Elements
        269           to         629
isotropic

        1elastic                                          10         0         0         0material1
 2.100000000000000+5 3.000000000000000-1 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
        0         0         0         0         0         0         0         0
          1           to         629
geometry

 5.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0
          1           to         629
fixed disp

        1         0         0         0         1         0apply1
 0.000000000000000+0
        0
        1
        2
apply1_nodes
        1         0         0         0         1         0apply2
 0.000000000000000+0
        0
        2
        2
apply2_nodes
dist loads
```

```
         1         0         0         0         0         0apply3
-2.250000000000000+1
         0
         0         1
        13
apply3_edges
         1         0         0         0         0         0apply4
-4.500000000000000+1
         0
         0         1
        13
apply4_edges
loadcase            job1
         4
apply1
apply2
apply3
apply4
post
         1        16        17         2         0        19        20         0         1         0
0         0         0         0         0         0
        17         0
parameters
 1.000000000000000+0 1.000000000000000+9 1.000000000000000+2 1.000000000000000+6 2.500000000000000-1
5.000000000000000-1 1.500000000000000+0-5.000000000000000-1
 8.625000000000000+0 2.000000000000000+1 1.000000000000000-4 1.000000000000000-6 1.000000000000000+0
1.000000000000000-4
 8.314000000000000+0 2.731500000000000+2 5.000000000000000-1 0.000000000000000+0 5.670510000000000-8
1.438769000000000-2 2.997900000000000+8 1.00000000000000+30
 0.000000000000000+0 0.000000000000000+0 1.000000000000000+2 0.000000000000000+0 1.000000000000000+0-
2.000000000000000+0 1.000000000000000+6 3.000000000000000+0
 0.000000000000000+0 0.000000000000000+0 1.256637061000000-6 8.85418781700000-12 1.200000000000000+2
1.000000000000000-3 1.600000000000000+2 0.000000000000000+0
 3.000000000000000+0
end option
$...................
```

APPENDIX 2

StressV1.out

```
        M               M
        W               W
      MMMMM           MMMMM
      WWWWW           WWWWW
    MMMMMMMMM       MMMMMMMMM
    WWWWWWWWW       WWWWWWWWW
  MMMMMMMMMMMMM   MMMMMMMMMMMMM
  WWWWWWWWWWWWW   WWWWWWWWWWWWW
MMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMM
WWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWW
MMMMMMMM MMMMMMMMMMMMMMM MMMMMMMM
WWWWWWWW WWWWWWWWWWWWWWW WWWWWWWW
MMMMMM     MMMMMMMMMMM     MMMMMM
WWWWWW     WWWWWWWWWWW     WWWWWW
MMMM         MMMMMMM         MMMM
WWWW         WWWWWWW         WWWW
MM             MMM             MM
WW             WWW             WW
M               M               M
W               W               W
MM             MMM             MM
WW             WWW             WW
MMMM         MMMMMMM         MMMM
WWWW         WWWWWWW         WWWW
MMMMMM     MMMMMMMMMMM     MMMMMM
WWWWWW     WWWWWWWWWWW     WWWWWW
MMMMMMMM MMMMMMMMMMMMMMM MMMMMMMM
WWWWWWWW WWWWWWWWWWWWWWW WWWWWWWW
MMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMM
WWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWW
  MMMMMMMMMMMMM   MMMMMMMMMMMMM
  WWWWWWWWWWWWW   WWWWWWWWWWWWW
    MMMMMMMMM       MMMMMMMMM
    WWWWWWWWW       WWWWWWWWW
```

```
          MMMMM          MMMMM
          WWWWWW         WWWWWW
            M              M
            W              W

version: Marc - Student Edition 2016.0.0, build 430850 (2016/08/12)

machine type: WINDOWS

integer*8 version: integers are 64-bits

date: Tue Oct 31 19:43:26 2017


Student Edition

  Maximum number of nodes in the model:       5000

  Expiration date: July 15, 2018

(c) COPYRIGHT 2016 MSC Software Corporation, all rights reserved

       Marc - Student Edition - W i n d o w s

        i n p u t   d a t a


   p a g e         1

                     10        20        30        40        50        60        70
80        90        100        110        120        130        140        150        160
                    -----------------------------------------------------------------------------
----------------------------------------------------------------------------------
```

```
                     title               job1
                     $....MARC input file produced by Marc Mentat 2016.0.0 (64bit) Student Edition
                     $....................................
                     $....input file using extended precision
    card         5   extended
                     $....................................
                     sizing                              0       629       680         0
                     alloc                     25
                     elements                   3
    card        10   version                   11
                     table                      0         0         2         1         1
0         0         1
                     processor                  1         1         1         0
                     -----------------------------------------------------------------------------
---------------------------------------------------------------------------------
                             10        20        30        40        50        60        70
80        90       100       110       120       130       140       150       160
                     -----------------------------------------------------------------------------
---------------------------------------------------------------------------------
                     -----------------------------------------------------------------------------
---------------------------------------------------------------------------------
1
             general memory increasing from      25 MByte to     106 MByte


             memory increased to        28003864 words during
             pre-reading of sets



   MSC Customer Entitlement ID
        N/A
```

```
*************************************************
*************************************************

 program sizing and options requested as follows



 element type requested*************************          3
 number of elements in mesh*********************        629
 number of nodes in mesh************************        680
 load correction was suppressed*****************
 elastic re-analysis flagged********************
 values stored at all integration points********
 tape no.for input of coordinates + connectivity          5
 no.of different materials    1 max.no of slopes          5
 number of points on shell section *************         11
 new style input format will be used************
 requested number of element threads************          1
 requested number of solver threads*************          1
 extended precision input is used **************
 Marc input version ****************************         11
 maximum number of boundary conditions *********          4
 suppress echo of list items *******************
 suppress echo of bc summary *******************
 suppress echo of nurbs data *******************


 end of parameters and sizing
*************************************************
*************************************************
```

```
                 key to stress, strain and displacement output



                           element type         3


                 4-node isoparametric plane stress

                 stresses and strains in global directions
                   1=xx
                   2=yy
                   3=xy

                 displacements in global directions
                   1=u global x direction
                   2=v global y direction

 allocated      76172 words of memory due to nodal vectors


 allocated       1940 words of memory due to boundary conditions


 allocated         12 words of memory due to geometric points


 allocated         46 words of memory due to geometric curves


 allocated         28 words of memory due to geometric surfaces
```

```
allocated       2720 words of memory due to shell, user or contact transformations


workspace needed for input and stiffness assembly:          34746


internal core allocation parameters
degrees of freedom per node (ndeg)  2
max. number of coordinates per node  2
max. nodes per element (nnodmx)  4
max. invariants per int. points (neqst)  1
max.stress components per int. point (nstrmx)   3
strains per integration point (ngens)  3


flag for element storage (ielsto)  0
element data in core

memory usage per element group
   group   # elements     nelsto      MByte            words
     1          629         410          1            257890
   ----------------------------------------------------------
   total        629                      1            257890

                 internal element variables



                 internal element number         1  library code type    3
                 number of nodes=  4
                 stresses stored per integration point =  3
```

```
                 direct continuum components stored =  2
                 shear continuum components stored =  1
                 shell/beam flag = 0
                 curvilinear  coord. flag = 0
                 int.points for elem. stiffness  4
                 number of local inertia directions  2
                 int.point for print if all points not flagged  5
                 int. points for dist. surface loads (pressure)  2
                 library code type =     3
                 large disp. row counts     4    4    7


                 residual load correction is switched off


$...................

solver
----------


multifrontal direct sparse solver invoked

optimize                   11
----------



metis nested dissection algorithm

connectivity
----------
```

```
          meshr1,  iprnt
                5      1
          elem no., type,        nodes

       coordinates
       ----------

       ncrd1 ,meshr1,iprnt
              2         5         1
             node      coordinates

       define              node                set                 apply1_nodes
       ----------

       a list of nodes given below
              50        51        52        53        54        55       358       359
      360       361       362       363       364
               echo suppressed for         8 items
               total number of items read:       21

       define              node                set                 apply2_nodes
       ----------

       a list of nodes given below
               1         2         3         4         5         6       312       313
      314       315       316       317       318
               echo suppressed for         8 items
               total number of items read:       21

       define              edgemt              set                 apply3_edges
       ----------

               1: 0         2: 0         3: 0         4: 0         5: 0         6: 0
```

```
7: 0         8: 0
                echo suppressed for       14 items
                total number of items read:       22

             define            edgemt            set                 apply4_edges
             ----------

                1: 3         53: 0        54: 0        55: 0        56: 0        57: 0
58: 0        59: 0
                echo suppressed for       14 items
                total number of items read:       22

             define            ndsq              set                 Domain_Nodes
             ----------

             a list of nodes given below
                1        55        56        57        58        59        60        61
62       63       64        65        66
                echo suppressed for      387 items
                total number of items read:      400

             define            element           set                 Domain_Elements
             ----------

             from element        269 to element        629 by          1

             isotropic
             ----------


             isotropic material  material id =          1
              elastic    yield criteria
              isotropic  hardening rule
```

```
material name is:       material1

    structural  property                              value       table id


Youngs modulus                                      2.10000E+05         0
Poissons ratio                                      3.00000E-01         0
mass density                                        0.00000E+00         0
shear modulus                                       8.07692E+04         0
coefficient of thermal expansion                    0.00000E+00         0
Yield stress                                        1.00000E+20         0
cost per unit volume                                0.00000E+00         0
cost per unit mass                                  0.00000E+00         0


from element          1 to element        629 by          1

geometry
----------

from element          1 to element        629 by          1

geometry id =          1

geometric parameter       value
          1             5.00000E+00
          2             0.00000E+00
          3             0.00000E+00
          4             0.00000E+00
          5             0.00000E+00
          6             0.00000E+00
          7             0.00000E+00
          8             0.00000E+00
```

```
fixed disp
----------

read data from unit    5

name of boundary condition apply1

Displacements are applied incrementally relative to current position

Prescribed Displacement for dof  1 =      0.00000E+00  table id is        0

applied to node ids

apply1_nodes

name of boundary condition apply2

Displacements are applied incrementally relative to current position

Prescribed Displacement for dof  2 =      0.00000E+00  table id is        0

applied to node ids

apply2_nodes

dist loads
----------

read data from unit    5

name of boundary condition apply3
```

```
Load Type     1

Prescribed Distributed Pressure =     -2.25000E+01  table id is        0

Prescribed Distributed Pressure =      0.00000E+00  table id is        0

Prescribed Distributed Pressure =      0.00000E+00  table id is        0

applied to elem mn-edge

apply3_edges

name of boundary condition apply4

Load Type     1

Prescribed Distributed Pressure =     -4.50000E+01  table id is        0

Prescribed Distributed Pressure =      0.00000E+00  table id is        0

Prescribed Distributed Pressure =      0.00000E+00  table id is        0

applied to elem mn-edge

apply4_edges

loadcase            job1
----------


activate boundary condition apply1

activate boundary condition apply2
```

```
activate boundary condition apply3

activate boundary condition apply4

post
----------

number of element variables on post file:    1

both binary and formatted post file will be used

initial output frequency of post file:       1

Marc 2005 style post file (default)

post variable     1 is post code   17 =
maximum record length on binary post file=           680
approximate no. of words per increment on file=          2532

maximum record length on formatted post file=           80
approximate no. of records per increment on file=          3200


parameters
----------


parameters set as follows

predictor used for stress-strain calculation    1.00000E+00
penalty factor for boundary conditions          1.00000E+09
penalty for incompressibility - r-p flow        1.00000E+02
penalty for incompressibility - fluid flow      1.00000E+06
```

```
beta parameter for Newmark operator              2.50000E-01
gamma parameter for Newmark operator             5.00000E-01
gamma1 parameter for Single-Step-Houbolt         1.50000E+00
gamma parameter for Single-Step-Houbolt         -5.00000E-01
sharp angle for sticking/separating - 2D         8.62500E+00
sharp angle for sticking/separating - 3D         2.00000E+01
initial strain rate for r-p flow                 1.00000E-04
lowest strain rate cut-off for r-p flow          1.00000E-06
fraction of dilatational stress neglected        1.00000E+00
factor for drilling d.o.f for shells             1.00000E-04
factor for displacement after rezoning           1.00000E+00

universal gas constant                           8.31400E+00
absolute temperature offset                      2.73150E+02
thermal properties evaluation weight             5.00000E-01
surface projection factor in ssh dynamics        0.00000E+00
Stefan Boltzmann constant                        5.67051E-08
Planck's second radiation constant               1.43877E-02
Speed of light in vacuum                         2.99790E+08
Permeability of vacuum                           1.25664E-06
Permittivity of vacuum                           8.85419E-12
maximum iterative displacement component         1.00000E+30
initial stiffness to simulate sticking           0.00000E+00
minimum angle between the normal vectors of
contacting segments                              1.20000E+02
radiation reflection cut-off                     1.00000E-03
angle for averaging adjacent beams (s2s)         1.60000E+02
stabilizer stiffness for model sections          0.00000E+00

maximum change in temperature per iteration      1.00000E+02
maximum rbe3 conditioning number                 0.10000E+07

end option
----------
```

```
            wall time =           0.00

            wall time =           0.00


            direct symmetric multi-frontal sparse solver is invoked for region   1


            number of element groups used:         1

               group     # elements     element type     material
formulation

                  1        629             3                1
S

              formulation:
                    S:   small displacement


     **** note ****
     can not find the following flow-data file
     in job directory or in the material library:
     C:\MSC.Software\Marc_Student_Edition\2016.0.0\marc2016\AF_flowmat\material1.mat
     This is ok if no separate flow-data file is required.
     Otherwise please provide the data file or result
     will be wrong.



            maximum connectivity in stiffness matrix is         10 at node        704
```

```
maximum half-bandwidth is        373  between nodes          1  and        404

number of profile entries excluding fill-in is            3246


total workspace needed with in-core matrix storage =           92122
  part of solver workspace is allocated separately

allocated        840 words of memory due to kinematic boundary conditions





load increments for each degree of freedom
summed over the whole model

from distributed loads
dist. loads on undeformed configuration - increments for dist. loads
increments for point loads
3.375000E+04   1.687500E+04


point loads
0.000000E+00   0.000000E+00

start of assembly     cycle number is     0
wall time =           0.00
```

```
             solver workspace for phase1 (matrix input)    (64bit words) =            8546 (
0.07 MByte)
             Metis nested dissection ordering used
             solver workspace for phase2 (nodal ordering)  (64bit words) =           25194 (
0.19 MByte)
             solver workspace for phase3 (symbolic factor) (64bit words) =           35511 (
0.27 MByte)
             solver workspace for phase4 (factorization)   (64bit words) =           59606 (
0.45 MByte) (MINIMUM)
             solver workspace for phase4 (factorization)   (64bit words) =          110322 (
0.84 MByte) (MAXIMUM)
             solver workspace available and used           (64bit words) =         1000000 (
7.63 MByte)


             start of matrix solution
             wall time =           0.00



             singularity ratio    2.9111E-01


             end of matrix solution
             wall time =           0.00



             element with highest stress relative to yield is        283
             where equivalent stress is 1.235E-18  of yield
```

```
   WINDOWS version
   Marc - Student Edition 2016.0.0
   output for increment       0. "          job1"


              total strain energy         is    6.07713E+02
                within which:
                elastic strain energy     is    6.07713E+02
              total ext-force work        is    6.07713E+02
                within which:
                work by appl. force/disp. is    6.07713E+02
                work by frictional forces is    0.00000E+00
          tresca     mises    mean   p r i n c i p a l   v a l u e s         p h y s i c a l   c
o m p o n e n t s
        intensity intensity  normal   minimum intermediate maximum     1         2         3
4         5         6
                            intensity



 element          1  point   1       integration pt. coordinate=     0.149E+03    0.149E+03
 section thickness = 0.500E+01
 engsts  4.497E+01 3.895E+01 2.248E+01 0.000E+00 2.247E+01 4.497E+01 4.497E+01 2.247E+01 1.548E-02
 engstn  2.784E-04 1.718E-04 0.000E+00-9.635E-05 4.277E-05 1.821E-04 1.821E-04 4.277E-05 1.916E-07

 element          1  point   2       integration pt. coordinate=     0.145E+03    0.149E+03
 section thickness = 0.500E+01
 engsts  4.498E+01 3.895E+01 2.249E+01 0.000E+00 2.249E+01 4.498E+01 4.498E+01 2.249E+01 2.205E-02
 engstn  2.784E-04 1.718E-04 0.000E+00-9.639E-05 4.285E-05 1.821E-04 1.821E-04 4.285E-05 2.730E-07
```

```
 element          1  point   3       integration pt. coordinate=     0.149E+03    0.145E+03
 section thickness = 0.500E+01
 engsts  4.499E+01 3.896E+01 2.249E+01 0.000E+00 2.248E+01 4.499E+01 4.499E+01 2.248E+01 2.227E-02
 engstn  2.785E-04 1.718E-04 0.000E+00-9.639E-05 4.277E-05 1.821E-04 1.821E-04 4.277E-05 2.758E-07

 element          1  point   4       integration pt. coordinate=     0.145E+03    0.145E+03
 section thickness = 0.500E+01
 engsts  4.500E+01 3.897E+01 2.250E+01 0.000E+00 2.250E+01 4.500E+01 4.500E+01 2.250E+01 2.882E-02
 engstn  2.786E-04 1.719E-04 0.000E+00-9.642E-05 4.285E-05 1.821E-04 1.821E-04 4.285E-05 3.568E-07
     ;            ;                          ;                          ;          ;
     ;            ;                          ;                          ;          ;
     ;            ;                          ;                          ;          ;
 element        629  point   1       integration pt. coordinate=     0.390E+02    0.545E+02
 section thickness = 0.500E+01
 engsts  6.383E+01 5.769E+01 2.642E+01 0.000E+00 1.542E+01 6.383E+01 6.173E+01 1.752E+01-9.864E+00
 engstn  3.952E-04 2.485E-04 0.000E+00-1.132E-04-1.777E-05 2.819E-04 2.689E-04-4.764E-06-1.221E-04

 element        629  point   2       integration pt. coordinate=     0.362E+02    0.544E+02
 section thickness = 0.500E+01
 engsts  6.377E+01 5.774E+01 2.627E+01 0.000E+00 1.505E+01 6.377E+01 6.164E+01 1.718E+01-9.956E+00
 engstn  3.948E-04 2.486E-04 0.000E+00-1.126E-04-1.942E-05 2.822E-04 2.690E-04-6.247E-06-1.233E-04

 element        629  point   3       integration pt. coordinate=     0.391E+02    0.517E+02
 section thickness = 0.500E+01
 engsts  6.361E+01 5.752E+01 2.629E+01 0.000E+00 1.526E+01 6.361E+01 6.145E+01 1.742E+01-9.989E+00
 engstn  3.938E-04 2.477E-04 0.000E+00-1.127E-04-1.821E-05 2.811E-04 2.677E-04-4.839E-06-1.237E-04

 element        629  point   4       integration pt. coordinate=     0.364E+02    0.517E+02
 section thickness = 0.500E+01
 engsts  6.354E+01 5.756E+01 2.614E+01 0.000E+00 1.488E+01 6.354E+01 6.135E+01 1.707E+01-1.009E+01
 engstn  3.933E-04 2.478E-04 0.000E+00-1.120E-04-1.990E-05 2.813E-04 2.677E-04-6.342E-06-1.249E-04
1
```

```
                    n o d a l   p o i n t   d a t a


                    t o t a l   d i s p l a c e m e n t s

              1  2.90842E-02  4.03022E-13                 2  3.05130E-02  5.85636E-13
3  3.20504E-02  6.06750E-13
              4  3.36153E-02  6.40419E-13                 5  3.52212E-02  6.86812E-13
6  3.68172E-02  3.64991E-13
           ;           ;                                    ;          ;
;
           ;           ;                                    ;          ;
;
           ;           ;                                    ;          ;
;
            704  2.00663E-02  3.16076E-03               705  1.71486E-02  3.27788E-03
706  1.52546E-02  3.14671E-03
            707  1.47720E-02  3.19445E-03               708  1.63233E-02  3.33722E-03
709  1.24337E-02  2.77517E-03
            710  1.88602E-02  3.23357E-03               711  1.58645E-02  3.24955E-03


          total equivalent nodal forces (distributed plus point loads)

              1   0.0000      0.0000                    2   0.0000      0.0000
3   0.0000      0.0000
              4   0.0000      0.0000                    5   0.0000      0.0000
6   767.05      0.0000
              7   1534.1      0.0000                    8   1534.1      0.0000
9   1534.1      0.0000
```

```
                               ;                                    ;            ;
;
                               ;                                    ;            ;
;
                               ;                                    ;            ;
;
                704   0.0000       0.0000                    705   0.0000       0.0000
706   0.0000       0.0000
                707   0.0000       0.0000                    708   0.0000       0.0000
709   0.0000       0.0000
                710   0.0000       0.0000                    711   0.0000       0.0000


                  reaction forces at fixed boundary conditions, residual load correction
elsewhere


                  1 -5.00222E-12  -1061.3                      2 -5.22959E-12  -1542.2
3 -4.78906E-12  -1597.8
                  4  2.95586E-12  -1686.4                      5 -1.25056E-12  -1808.6
6  3.29692E-12  -961.14
                  7  3.18323E-12 -1.13687E-12                  8 -1.81899E-12 -1.13687E-13
9 -1.15961E-11  4.88853E-12
                               ;                                     ;         ;
;                  ;
                               ;                                     ;         ;
;                  ;
                               ;                                     ;         ;
;                  ;
                701  1.21076E-11  5.11591E-13                702  1.81899E-12  6.53699E-13
706 -1.25056E-12 -2.55795E-12
                707 -3.29692E-12  1.25056E-12                708  3.36797E-12  1.36424E-12
709  1.13687E-12 -1.36424E-12
```

```
        710  9.54969E-12  7.95808E-13                711  6.36646E-12  2.55795E-13

                      summary of externally applied loads

  3.37500E+04    1.68750E+04

                      summary of reaction/residual forces

 -3.37500E+04   -1.68750E+04


          memory usage:                        MByte       words  % of total

           within general memory:
            element stiffness matrices:           0       30126         0.1
            solver: first part                    0       57376         0.2
            overallocation initial allocation   106    27911746        74.5
            other:                                0        4616         0.0
           allocated separately:
            solver 8                              8     1982908         5.3
            nodal vectors:                        0       77532         0.2
            defined sets:                         0        2068         0.0
            transformations:                      0        2720         0.0
            kinematic boundary conditions:        0        2780         0.0
            points, curves and surfaces:          0          86         0.0
            mem_none:                             0       89366         0.2
            element storage:                      1      280862         0.8
            material properties:                  0        3568         0.0
            executable and common blocks:        27     7000000        18.7
            miscellaneous                         0         212         0.0
            ---------------------------------------------------------------
            total:                              143    37445966

            general memory allocated:           107    28003864
```

```
  general memory used:                     0       92118

  peak memory usage:                     159    41557520

e n d   o f   i n c r e m e n t      0

binary post data at increment       0.  subincrement    0.  on file 16

formatted post data at increment       0.  subincrement    0.  on file 19
wall time =           1.00


$...................


                 *** end of input deck - job ends


memory usage:                          MByte       words  % of total

 within general memory:
  element stiffness matrices:              0       30126         0.1
  solver: first part                       0       57376         0.2
  overallocation initial allocation      106    27911746        74.5
  other:                                   0        4616         0.0
 allocated separately:
  solver 8                                 8     1982908         5.3
  nodal vectors:                           0       77532         0.2
  defined sets:                            0        2068         0.0
  transformations:                         0        2720         0.0
  kinematic boundary conditions:           0        2780         0.0
  points, curves and surfaces:             0          86         0.0
  mem_none:                                0       89366         0.2
  element storage:                         1      280862         0.8
```

```
  material properties:                   0      3568        0.0
  executable and common blocks:         27   7000000       18.7
  miscellaneous                          0       212        0.0
  ---------------------------------------------------------------
  total:                               143  37445966

  general memory allocated:            107  28003864
  general memory used:                   0     92118

  peak memory usage:                   159  41557520


timing information:                          wall time      cpu time

  total time for input:                           0.05          0.05
  total time for stiffness assembly:              0.02          0.02
  total time for stress recovery:                 0.01          0.00
  total time for matrix solution:                 0.06          0.02
  total time for output:                          0.05          0.06
  total time for miscellaneous:                   0.09          0.09
  ---------------------------------------------------------------
  total time:                                     0.28          0.23


**************************************************************************
```

This is a successful completion to a Marc simulation, indicating that no additional incremental data was found and that the analysis is complete.

```
***************************************************************************

                           Marc - Student Edition 2016.0.0

                           Exit number 3004
```

APPENDIX 3

ThermalV1.dat

```
title               job1
$....MARC input file produced by Marc Mentat 2016.0.0 (64bit) Student Edition
$....................................
$....input file using extended precision
extended
$....................................
sizing                                  0       585       633         0
alloc                       25
elements                     3
version                     11
table                        0          0         2         1         1         0         0         1
processor                    1          1         1         0
$no list
couple                       0
all points
no echo                      1          2         3
end
$...................
solver
         8         0         0          0         0         0         0         0         0         0
0         0         0         0          0         0
optimize                    11
connectivity
         0         0         1
         1         3        55         56        91        54
         2         3        56         57        92        91
         3         3        57         58        93        92
         ;         ;         ;          ;         ;         ;
         ;         ;         ;          ;         ;         ;
         ;         ;         ;          ;         ;         ;
         ;         ;         ;          ;         ;         ;

       583         3        55         54       647       612
       584         3       666        649        52        51
       585         3       649        666       607       608
coordinates
         3       633         0          1
```

```
         1 4.000000000000000+1 0.000000000000000+0 0.000000000000000+0
         2 4.500000000000000+1 0.000000000000000+0 0.000000000000000+0
         3 5.000000000000000+1 0.000000000000000+0 0.000000000000000+0

       666 3.389361435619000+1 1.449949264039000+2 0.000000000000000+0
       667 1.449969631442000+2 2.552391971165000+1 0.000000000000000+0
       668 1.449985816417000+2 1.703941592972000+1 0.000000000000000+0
define              node                set                 applyT1_nodes
          1
define              node                set                 applyT2_nodes
          2
define              node                set                 applyT3_nodes
          3
   ;       ;         ;                   ;                        ;
   ;       ;         ;                   ;                        ;
   ;       ;         ;                   ;                        ;
define              node                set                 applyT572_nodes
        572
define              node                set                 applyT573_nodes
        573
define              node                set                 applyT574_nodes
        574
define              node                set                 apply1_nodes
         55          56          57          58          59          60          61          62
63          64          65          66          67   c
         68          69          70          71          72          73          74          75
611         612
define              node                set                 apply2_nodes
          1           2           3           4           5           6           7           8
9          10          11          12          13   c
         14          15          16          17          18          19          20          21
576         577
define              ndsq                set                 Domain_Nodes
          1           2           3           4           5           6           7           8
9          10          11          12          13   c
         14          15          16          17          18          19          20          21
22          23          24          25          26   c
```

```
         27          28          29          30          31          32          33          34
35          36          37          38          39   c



         534         535         536         537         538         539         540         541
542         543         544         545         546   c
         547         548         549         550         551         552         553         554
555         556         557         558         559   c
         560         561         562         563         564         565         566         567
568         569         570         571         572   c
         573         574
define              element             set                 Domain_Elements
           1          to         528
isotropic

         1elastic                                                   10          0          0          0material1
 2.100000000000000+5 3.000000000000000-1 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
         0          0          0          0          0          0          0          0
 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0
         0          0          0          0          0          0          0
         529          to         585
isotropic

         2elastic                                                   10          0          0          0material2
 5.250000000000000+2 3.000000000000000-1 0.000000000000000+0 1.080000000000000-5 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
         0          0          0          0          0          0          0          0
 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0
         0          0          0          0          0          0          0
           1          to         528
geometry

 5.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
```

```
0.000000000000000+0 0.000000000000000+0
          1          to         585
fixed temperature

         1         0         0         0         0         0applyT1
 0.000000000000000+0
         0
         1
         2
applyT1_nodes
         1         0         0         0         0         0applyT2
 0.000000000000000+0
         0
         1
         2
applyT2_nodes
         1         0         0         0         0         0applyT3
 0.000000000000000+0
         0
         1
         2
applyT3_nodes



         1         0         0         0         0         0applyT574
 0.000000000000000+0
         0
         1
         2
applyT574_nodes
fixed disp

         1         0         0         0         1         0apply1
 0.000000000000000+0
         0
```

```
         1
         2
apply1_nodes
         1         0         0         0         1         0apply2
 0.000000000000000+0
         0
         2
         2
apply2_nodes
loadcase            job1
       576
applyT1
applyT2
applyT3



applyT574
apply1
apply2
post
         0        16        17         2         0        19        20         0         1         0
0         0         0         0         0         0
parameters
 1.000000000000000+0 1.000000000000000+9 1.000000000000000+2 1.000000000000000+6 2.500000000000000-1
5.000000000000000-1 1.500000000000000+0-5.000000000000000-1
 8.625000000000000+0 2.000000000000000+1 1.000000000000000-4 1.000000000000000-6 1.000000000000000+0
1.000000000000000-4
 8.314000000000000+0 2.731500000000000+2 5.000000000000000-1 0.000000000000000+0 5.670510000000000-8
1.438769000000000-2 2.997900000000000+8 1.00000000000000+30
 0.000000000000000+0 0.000000000000000+0 1.000000000000000+2 0.000000000000000+0 1.000000000000000+0-
2.000000000000000+0 1.000000000000000+6 3.000000000000000+0
 0.000000000000000+0 0.000000000000000+0 1.256637061000000-6 8.85418781700000-12 1.200000000000000+2
1.000000000000000-3 1.600000000000000+2 0.000000000000000+0
 3.000000000000000+0
end option
$...................
```

```
$....start of loadcase lcase1
title               lcase1
loadcase            lcase1
       576
applyT1
applyT2
applyT3



applyT574
apply1
apply2
control
     99999        10         0         0         0         1         0         0         1         0
1         0         0         0
 1.000000000000000-1 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
0.000000000000000+0 0.000000000000000+0 0.000000000000000+0
 2.000000000000000+1 1.000000000000000+2 0.000000000000000+0 1.000000000000000+2 1.000000000000000-1
1.000000000000000-1 1.000000000000000-1 1.0000000000000+30
parameters
 1.000000000000000+0 1.000000000000000+9 1.000000000000000+2 1.000000000000000+6 2.500000000000000-1
5.000000000000000-1 1.500000000000000+0-5.000000000000000-1
 8.625000000000000+0 2.000000000000000+1 1.000000000000000-4 1.000000000000000-6 1.000000000000000+0
1.000000000000000-4
 8.314000000000000+0 2.731500000000000+2 5.000000000000000-1 0.000000000000000+0 5.670510000000000-8
1.438769000000000-2 2.997900000000000+8 1.0000000000000+30
 0.000000000000000+0 0.000000000000000+0 1.000000000000000+2 0.000000000000000+0 1.000000000000000+0-
1.000000000000000+0 1.000000000000000+6 3.000000000000000+0
 0.000000000000000+0 0.000000000000000+0 1.256637061000000-6 8.85418781700000-12 1.200000000000000+2
1.000000000000000-3 1.600000000000000+2 0.000000000000000+0
 3.000000000000000+0
transient non auto
 2.000000000000000-2 1.000000000000000+0        50         0         0         0 0.000000000000000+0
continue
$....end of loadcase lcase1
$...................
```

APPENDIX 4

ThermalV1.out

```
        M               M
        W               W
      MMMMM           MMMMM
      WWWWW           WWWWW
    MMMMMMMMM       MMMMMMMMM
    WWWWWWWWW       WWWWWWWWW
  MMMMMMMMMMMMM   MMMMMMMMMMMMM
  WWWWWWWWWWWWW   WWWWWWWWWWWWW
MMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMM
WWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWW
MMMMMMMM MMMMMMMMMMMMMMM MMMMMMMM
WWWWWWWW WWWWWWWWWWWWWWW WWWWWWWW
MMMMMM     MMMMMMMMMMM     MMMMMM
WWWWWW     WWWWWWWWWWW     WWWWWW
MMMM         MMMMMMM         MMMM
WWWW         WWWWWWW         WWWW
MM             MMM             MM
WW             WWW             WW
M               M               M
W               W               W
MM             MMM             MM
WW             WWW             WW
MMMM         MMMMMMM         MMMM
WWWW         WWWWWWW         WWWW
MMMMMM     MMMMMMMMMMM     MMMMMM
WWWWWW     WWWWWWWWWWW     WWWWWW
MMMMMMMM MMMMMMMMMMMMMMM MMMMMMMM
WWWWWWWW WWWWWWWWWWWWWWW WWWWWWWW
MMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMMM
WWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWWW
  MMMMMMMMMMMMM   MMMMMMMMMMMMM
  WWWWWWWWWWWWW   WWWWWWWWWWWWW
    MMMMMMMMM       MMMMMMMMM
    WWWWWWWWW       WWWWWWWWW
```

```
                    MMMMMM          MMMMMM
                    WWWWWW          WWWWWW
                      M               M
                      W               W

             version: Marc - Student Edition 2016.0.0, build 430850 (2016/08/12)

             machine type: WINDOWS

             integer*8 version: integers are 64-bits

             date: Sun Aug 13 17:48:29 2017

             Student Edition

               Maximum number of nodes in the model:       5000

               Expiration date: July 15, 2018
```



```
                  Marc - Student Edition - W i n d o w s

                   i n p u t   d a t a

                p a g e         1

                                 10        20        30        40        50        60        70
80        90       100       110       120       130       140       150       160
                     ----------------------------------------------------------------------------
-----------------------------------------------------------------------------------
```

```
                    title               job1
                    $....MARC input file produced by Marc Mentat 2016.0.0 (64bit) Student Edition
                    $....................................
                    $....input file using extended precision
    card        5   extended
                    $....................................
                    sizing                             0       439       486         0
                    alloc                   25
                    elements                 3
    card       10   version                 11
                    table                    0         0         2         1         1
0         0         1
                    processor                1         1         1         0
                    -----------------------------------------------------------------------------
-----------------------------------------------------------------------------------
                           10        20        30        40        50        60        70
80        90       100       110       120       130       140       150       160
                    -----------------------------------------------------------------------------
-----------------------------------------------------------------------------------
                    -----------------------------------------------------------------------------
-----------------------------------------------------------------------------------
1
            general memory increasing from      25 MByte to     106 MByte


            memory increased to        28003866 words during
            pre-reading of sets



   MSC Customer Entitlement ID
        N/A
```

```
*************************************************
*************************************************

 program sizing and options requested as follows



 element type requested*************************         3
 element type requested*************************        39
 number of elements in mesh*********************       439
 number of nodes in mesh************************       486
 thermal stress analysis flagged****************
 load correction flagged or set*****************
 values stored at all integration points********
 tape no.for input of coordinates + connectivity         5
 no.of different materials    2 max.no of slopes         5
 heat transfer analysis,  extrapolation flag, **         1
 gradient of scaler field printed***************
 number of points on shell section *************        11
 new style input format will be used************
 coupled thermal-mechanical analysis flagged****
 requested number of element threads************         1
 requested number of solver threads*************         1
 extended precision input is used **************
 Marc input version ****************************        11
 maximum number of boundary conditions *********        98
 suppress echo of list items *******************
 suppress echo of bc summary *******************
 suppress echo of nurbs data *******************


 end of parameters and sizing
*************************************************
```

```
 ************************************************


                    element type         3


          4-node isoparametric plane stress

          stresses and strains in global directions
            1=xx
            2=yy
            3=xy

          displacements in global directions
            1=u global x direction
            2=v global y direction


                    element type        39


          4-node heat transfer planar element

          1 degree of freedom per node - temperature

 allocated      64164 words of memory due to nodal vectors


 allocated      51552 words of memory due to boundary conditions
```

```
allocated         12 words of memory due to geometric points

allocated         46 words of memory due to geometric curves

allocated         28 words of memory due to geometric surfaces

allocated       1944 words of memory due to shell, user or contact transformations


workspace needed for input and stiffness assembly:           26218


internal core allocation parameters
degrees of freedom per node (ndeg)  2
max. number of coordinates per node  2
max. nodes per element (nnodmx)  4
max. invariants per int. points (neqst)  1
max.stress components per int. point (nstrmx)   3
strains per integration point (ngens)  3
max. gradients per int. point  3


flag for element storage (ielsto)  0
element data in core

memory usage per element group
   group   # elements     nelsto      MByte            words
     1         364         554           1            201656
     2          75         554           0             41550
```

```
----------------------------------------------------------
total        439                        1            243206


              internal element variables



              internal element number         1  library code type    3
              number of nodes=  4
              stresses stored per integration point =  3
              direct continuum components stored =  2
              shear continuum components stored =  1
              shell/beam flag = 0
              curvilinear  coord. flag = 0
              int.points for elem. stiffness  4
              number of local inertia directions  2
              int.point for print if all points not flagged  5
              int. points for dist. surface loads (pressure)  2
              library code type =     3
              large disp. row counts     4    4    7
              number of nodes  4
              number of gradient components at each int. point  3
              integration points for conductivity  4
              integration point for print-out  5
              integration points for surface b.c.s  2

              internal element number         2  library code type   39
              number of nodes=  4
              stresses stored per integration point =  2
              direct continuum components stored =  2
              shear continuum components stored =  0
              shell/beam flag = 0
              curvilinear  coord. flag = 0
```

```
                 int.points for elem. stiffness  4
                 number of local inertia directions  1
                 int.point for print if all points not flagged  5
                 int. points for dist. surface loads (pressure)  2
                 library code type =    39
                 large disp. row counts     0    0    0
                 number of nodes  4
                 number of gradient components at each int. point  2
                 integration points for conductivity  4
                 integration point for print-out  5
                 integration points for surface b.c.s  2


                 residual load correction is invoked


$...................

solver
----------


multifrontal direct sparse solver invoked

optimize                   11
----------



metis nested dissection algorithm

connectivity
```

```
----------

   meshr1,  iprnt
         5      1
   elem no., type,        nodes

coordinates
----------

ncrd1 ,meshr1,iprnt
       2         5         1
      node      coordinates

define              node              set              applyT17_nodes
----------

a list of nodes given below
       19

define              node              set              applyT18_nodes

----------
      95

   ;                   ;                                         ;
   ;                   ;                                         ;
   ;                   ;                                         ;
   ;                   ;                                         ;


define              node              set              apply1_nodes
----------

a list of nodes given below
```

```
                  1         2         3         4         5         6        97       403
404       405       406       407       408
                  echo suppressed for         6 items
                  total number of items read:        19

          define              ndsq                set                 apply2_nodes
          ----------

          a list of nodes given below
                 21        36        51        66        81        96       106       307
308       309       310       311       312
                  echo suppressed for         6 items
                  total number of items read:        19

          define              node                set                 apply2_nodes_1
          ----------

          a list of nodes given below
                 21        36        51        66        81        96       106       307
308       309       310       311       312
                  echo suppressed for         6 items
                  total number of items read:        19

 define              ndsq                set                 Domain_nodes
          ----------

          a list of nodes given below
                  1         2         3         4         5         6         7         8
9        10        11        12        13
                  echo suppressed for        83 items
                  total number of items read:        96


          define              element             set                 Domain_elements
```

```
 ----------

 a list of elements given below
         1         2         3         4         5         6         7         8
 9         10        11        12        13
         echo suppressed for        78 items
         total number of items read:        91

 isotropic
 ----------


 isotropic material  material id =         1
  elastic    yield criteria
  isotropic  hardening rule

 material name is:       material1


     property                                          value       table id


 Youngs modulus                                    2.10000E+05         0
 Poissons ratio                                    3.00000E-01         0
 mass density                                      0.00000E+00         0
 shear modulus                                     8.07692E+04         0
 coefficient of thermal expansion                  0.00000E+00         0
 Yield stress                                      1.00000E+20         0
 cost per unit volume                              0.00000E+00         0
 cost per unit mass                                0.00000E+00         0
 thermal conductivity                              0.00000E+00         0
 specific heat                                     0.00000E+00         0
 mass density - heat transfer                      0.00000E+00         0
 emissivity                                        0.00000E+00         0
```

from element 76 to element 439 by 1

isotropic
----

isotropic material material id = 2
elastic yield criteria
isotropic hardening rule

material name is: material2

property	value	table id
Youngs modulus	5.25000E+02	0
Poissons ratio	3.00000E-01	0
mass density	0.00000E+00	0
shear modulus	2.01923E+02	0
coefficient of thermal expansion	1.08000E-05	0
Yield stress	1.00000E+20	0
cost per unit volume	0.00000E+00	0
cost per unit mass	0.00000E+00	0
thermal conductivity	0.00000E+00	0
specific heat	0.00000E+00	0
mass density - heat transfer	0.00000E+00	0
emissivity	0.00000E+00	0

from element 1 to element 75 by 1

```
geometry
----------

from element          1 to element        439 by           1

geometry id =          1

geometric parameter       value
          1             5.00000E+00
          2             0.00000E+00
          3             0.00000E+00
          4             0.00000E+00
          5             0.00000E+00
          6             0.00000E+00
          7             0.00000E+00
          8             0.00000E+00

fixed temperature
----------


read data from unit    5

name of boundary condition applyT1

Prescribed Temperature  for dof  1 =      7.92350E+01  table id is        0

applied to node ids

applyT1_nodes

name of boundary condition applyT2

Prescribed Temperature  for dof  1 =      8.63000E+01  table id is        0
```

```
applied to node ids

applyT2_nodes

fixed disp
----------

read data from unit    5

name of boundary condition apply1

Displacements are applied incrementally relative to current position

Prescribed Displacement for dof  2 =      0.00000E+00  table id is        0

applied to node ids

apply1_nodes

name of boundary condition apply2

Displacements are applied incrementally relative to current position

Prescribed Displacement for dof  1 =      0.00000E+00  table id is        0

applied to node ids

apply2_nodes_1

loadcase            job1
----------
```

```
activate boundary condition apply1

activate boundary condition apply2

activate boundary condition applyT1

;                  ;              ;
;                  ;              ;
;                  ;              ;
;                  ;              ;

activate boundary condition applyT95

activate boundary condition applyT96

post
----------

both binary and formatted post file will be used

initial output frequency of post file:       1

Marc 2005 style post file (default)

maximum record length on binary post file=           486
approximate no. of words per increment on file=           16

maximum record length on formatted post file=           80
approximate no. of records per increment on file=           490


parameters
```

----

parameters set as follows

predictor used for stress-strain calculation	1.00000E+00
penalty factor for boundary conditions	1.00000E+09
penalty for incompressibility - r-p flow	1.00000E+02
penalty for incompressibility - fluid flow	1.00000E+06
beta parameter for Newmark operator	2.50000E-01
gamma parameter for Newmark operator	5.00000E-01
gamma1 parameter for Single-Step-Houbolt	1.50000E+00
gamma parameter for Single-Step-Houbolt	-5.00000E-01
sharp angle for sticking/separating - 2D	8.62500E+00
sharp angle for sticking/separating - 3D	2.00000E+01
initial strain rate for r-p flow	1.00000E-04
lowest strain rate cut-off for r-p flow	1.00000E-06
fraction of dilatational stress neglected	1.00000E+00
factor for drilling d.o.f for shells	1.00000E-04
factor for displacement after rezoning	1.00000E+00
universal gas constant	8.31400E+00
absolute temperature offset	2.73150E+02
thermal properties evaluation weight	5.00000E-01
surface projection factor in ssh dynamics	0.00000E+00
Stefan Boltzmann constant	5.67051E-08
Planck's second radiation constant	1.43877E-02
Speed of light in vacuum	2.99790E+08
Permeability of vacuum	1.25664E-06
Permittivity of vacuum	8.85419E-12
maximum iterative displacement component	1.00000E+30
initial stiffness to simulate sticking	0.00000E+00
minimum angle between the normal vectors of contacting segments	1.20000E+02

```
radiation reflection cut-off                      1.00000E-03
angle for averaging adjacent beams (s2s)          1.60000E+02
stabilizer stiffness for model sections           0.00000E+00

maximum change in temperature per iteration       1.00000E+02
maximum rbe3 conditioning number                  0.10000E+07

end option
----------

wall time =           1.00

wall time =           1.00


direct symmetric multi-frontal sparse solver is invoked for region   1


number of element groups used:         2

   group     # elements     element type     material
formulation

      1        364              3               1
S
      2         75              3               2
S

  formulation:
        S:   small displacement


**** note ****
can not find the following flow-data file
```

```
in job directory or in the material library:
C:\MSC.Software\Marc_Student_Edition\2016.0.0\marc2016\AF_flowmat\material1.mat
This is ok if no separate flow-data file is required.
Otherwise please provide the data file or result
will be wrong.


**** note ****
can not find the following flow-data file
in job directory or in the material library:
C:\MSC.Software\Marc_Student_Edition\2016.0.0\marc2016\AF_flowmat\material2.mat
This is ok if no separate flow-data file is required.
Otherwise please provide the data file or result
will be wrong.



        maximum connectivity in stiffness matrix is          9 at node        390



        maximum half-bandwidth is        402  between nodes          2  and        403

        number of profile entries excluding fill-in is           2288



        total workspace needed with in-core matrix storage =          89594
          part of solver workspace is allocated separately

        allocated        760 words of memory due to kinematic boundary conditions


        allocated       1920 words of memory due to kinematic boundary conditions
```

```
load increments for each degree of freedom
summed over the whole model

from distributed loads
dist. loads on undeformed configuration - increments for dist. loads
increments for point loads
0.000000E+00   0.000000E+00


point loads
0.000000E+00   0.000000E+00

start of assembly     cycle number is     0
wall time =           1.00




solver workspace for phase1 (matrix input)    (64bit words) =            6048 (
0.05 MByte)
Metis nested dissection ordering used
solver workspace for phase2 (nodal ordering)  (64bit words) =           18782 (
0.14 MByte)
solver workspace for phase3 (symbolic factor) (64bit words) =           25341 (
0.19 MByte)
solver workspace for phase4 (factorization)   (64bit words) =           41381 (
0.32 MByte) (MINIMUM)
```

```
            solver workspace for phase4 (factorization)   (64bit words) =           74047 (
0.56 MByte) (MAXIMUM)
            solver workspace available and used           (64bit words) =         1000000 (
7.63 MByte)


            start of matrix solution
            wall time =          1.00



            singularity ratio    1.6829E-01


            end of matrix solution
            wall time =          1.00



            element with highest stress relative to yield is          1
            where equivalent stress is 1.000E-20  of yield





  WINDOWS version
  Marc - Student Edition 2016.0.0
  output for increment       0. "          job1"


            total strain energy         is    0.00000E+00
```

```
              within which:
              elastic strain energy     is    0.00000E+00
            total ext-force work        is    0.00000E+00
              within which:
              work by appl. force/disp. is    0.00000E+00
              work by frictional forces is    0.00000E+00
          tresca     mises    mean   p r i n c i p a l   v a l u e s         p h y s i c a l   c
o m p o n e n t s
         intensity intensity  normal   minimum intermediate maximum     1         2         3
4         5         6
                              intensity



 element          1  point   1       integration pt. coordinate=     0.404E+02    0.893E+00
 section thickness = 0.500E+01

 element          1  point   2       integration pt. coordinate=     0.415E+02    0.918E+00
 section thickness = 0.500E+01

 element          1  point   3       integration pt. coordinate=     0.402E+02    0.333E+01
 section thickness = 0.500E+01

 element          1  point   4       integration pt. coordinate=     0.414E+02    0.343E+01
 section thickness = 0.500E+01

    ;                                           ;                            ;
    ;                                           ;                            ;
    ;                                           ;                            ;

element        439  point   1       integration pt. coordinate=     0.143E+03    0.364E+02
 section thickness = 0.500E+01

 element        439  point   2       integration pt. coordinate=     0.143E+03    0.342E+02
```

```
 section thickness = 0.500E+01

 element        439  point   3       integration pt. coordinate=     0.148E+03    0.364E+02
 section thickness = 0.500E+01

 element        439  point   4       integration pt. coordinate=     0.148E+03    0.342E+02
 section thickness = 0.500E+01
1
                              n o d a l   p o i n t   d a t a


                              i n c r e m e n t a l   d i s p l a c e m e n t s


                   1   0.0000       0.0000                            2   0.0000       0.0000
3   0.0000       0.0000
                   4   0.0000       0.0000                            5   0.0000       0.0000
6   0.0000       0.0000
                   7   0.0000       0.0000                            8   0.0000       0.0000
9   0.0000       0.0000

                        ;              ;                             ;              ;
;             ;

                        ;              ;                             ;              ;
                 481   0.0000       0.0000                          482   0.0000       0.0000
483   0.0000       0.0000
                 484   0.0000       0.0000                          485   0.0000       0.0000


                              t o t a l   d i s p l a c e m e n t s
```

```
                  1   0.0000       0.0000                          2   0.0000       0.0000
3   0.0000       0.0000
                  4   0.0000       0.0000                          5   0.0000       0.0000
6   0.0000       0.0000
      ;               ;                ;                              ;              ;
      ;               ;                ;                              ;              ;
                481   0.0000       0.0000                        482   0.0000       0.0000
483   0.0000       0.0000
                484   0.0000       0.0000                        485   0.0000       0.0000
486   0.0000       0.0000
```

t o t a l   n o d a l   t e m p e r a t u r e s

```
               1   0.0000                   2   0.0000                    3   0.0000
4   0.0000                   5   0.0000
               6   0.0000                   7   0.0000                    8   0.0000
9   0.0000                  10   0.0000
              11   0.0000                  12   0.0000                   13   0.0000
14   0.0000                  15   0.0000
                    ;            ;                  ;                              ;
;      ;
;                                ;                  ;          ;
      ;              ;                            481   0.0000         482    0.0000
483   0.0000              484   0.0000           485   0.0000
              486   0.0000
```

t o t a l   n o d a l   f l u x e s

```
                 1   0.0000                    2   0.0000                     3   0.0000
4   0.0000                  5   0.0000
                 6   0.0000                    7   0.0000                     8   0.0000
9   0.0000                 10   0.0000
                11   0.0000                   12   0.0000                    13   0.0000
14   0.0000                 15   0.0000
                       ;                           ;                                 ;
;                          ;
                       ;                           ;                                 ;
;                          ;
                       ;                           ;                                 ;
               481   0.0000                  482   0.0000                   483   0.0000
484   0.0000                485   0.0000
               486   0.0000
```

total equivalent nodal forces (distributed plus point loads)

```
                    1   0.0000       0.0000                        2   0.0000       0.0000
3   0.0000       0.0000
                    4   0.0000       0.0000                        5   0.0000       0.0000
6   0.0000       0.0000
                    7   0.0000       0.0000                        8   0.0000       0.0000

                      ;              ;                               ;              ;
;             ;

                      ;              ;                               ;              ;
;             ;
```

```
                  481   0.0000        0.0000                    482   0.0000        0.0000
483   0.0000        0.0000
                  484   0.0000        0.0000                    485   0.0000        0.0000
486   0.0000        0.0000


                     reaction forces at fixed boundary conditions, residual load correction
elsewhere


                    1   0.0000        0.0000                      2   0.0000        0.0000
3   0.0000        0.0000
                    4   0.0000        0.0000                      5   0.0000        0.0000
6   0.0000        0.0000
                          ;              ;                          ;              ;
;             ;

                          ;              ;                          ;              ;
;             ;
483   0.0000        0.0000
                  484   0.0000        0.0000                    485   0.0000        0.0000
486   0.0000        0.0000

                                  summary of externally applied loads

    0.00000E+00    0.00000E+00

                                  summary of reaction/residual forces

    0.00000E+00    0.00000E+00


            memory usage:                      MByte       words  % of total
```

```
 within general memory:
  element stiffness matrices:           0      21554         0.1
  solver: first part                    0      63376         0.2
  overallocation initial allocation   106   27914276        74.6
  other:                                0       4660         0.0
 allocated separately:
  solver 8                              8    1987904         5.3
  nodal vectors:                        0      64166         0.2
  defined sets:                         0       3620         0.0
  transformations:                      0       1944         0.0
  kinematic boundary conditions:        0      54232         0.1
  points, curves and surfaces:          0         86         0.0
  mem_none:                             0      42438         0.1
  element storage:                      1     259576         0.7
  material properties:                  0       5150         0.0
  executable and common blocks:        27    7000000        18.7
  miscellaneous                         0        212         0.0
  ----------------------------------------------------------------
  total:                              143   37423194

  general memory allocated:           107   28003866
  general memory used:                  0      89590

  peak memory usage:                  159   41557522

e n d   o f   i n c r e m e n t      0

binary post data at increment       0.  subincrement    0.  on file 16

formatted post data at increment       0.  subincrement    0.  on file 19
wall time =           1.00
```

```
$...................

$....start of loadcase lcase1

title               lcase1
----------


loadcase            lcase1
----------


activate boundary condition apply1

activate boundary condition apply2

activate boundary condition applyT1
   ;        ;        ;       ;

   ;        ;        ;       ;

   ;        ;        ;       ;
activate boundary condition applyT95

activate boundary condition applyT96

control
----------


control information for region   1

maximum number of increments     100000
maximum number of recycles           10
```

```
minimum number of recycles            0

control of residual convergence:
  relative tolerance:             1.00000E-01
  cut-off value     :             0.00000E+00
  absolute tolerance:             0.00000E+00
  relative moment tolerance:      0.00000E+00
  moment cut-off value     :      0.00000E+00
  absolute moment tolerance:      0.00000E+00



full newton-raphson technique chosen

convergence testing is automatically switched between
residual and displacement testing if reactions or displacements
become too small

maximum nodal temperature change per time step =    2.00000E+01

maximum nodal temperature change before reassembly =  1.00000E+02

control messages will be written to log file

parameters
----------


parameters set as follows

predictor used for stress-strain calculation    1.00000E+00
penalty factor for boundary conditions          1.00000E+09
penalty for incompressibility - r-p flow        1.00000E+02
penalty for incompressibility - fluid flow      1.00000E+06
```

```
beta parameter for Newmark operator               2.50000E-01
gamma parameter for Newmark operator              5.00000E-01
gamma1 parameter for Single-Step-Houbolt          1.50000E+00
gamma parameter for Single-Step-Houbolt          -5.00000E-01
sharp angle for sticking/separating - 2D          8.62500E+00
sharp angle for sticking/separating - 3D          2.00000E+01
initial strain rate for r-p flow                  1.00000E-04
lowest strain rate cut-off for r-p flow           1.00000E-06
fraction of dilatational stress neglected         1.00000E+00
factor for drilling d.o.f for shells              1.00000E-04
factor for displacement after rezoning            1.00000E+00

universal gas constant                            8.31400E+00
absolute temperature offset                       2.73150E+02
thermal properties evaluation weight              5.00000E-01
surface projection factor in ssh dynamics         0.00000E+00
Stefan Boltzmann constant                         5.67051E-08
Planck's second radiation constant                1.43877E-02
Speed of light in vacuum                          2.99790E+08
Permeability of vacuum                            1.25664E-06
Permittivity of vacuum                            8.85419E-12
maximum iterative displacement component          1.00000E+30
initial stiffness to simulate sticking            0.00000E+00
minimum angle between the normal vectors of
contacting segments                               1.20000E+02
radiation reflection cut-off                      1.00000E-03
angle for averaging adjacent beams (s2s)          1.60000E+02
stabilizer stiffness for model sections           0.00000E+00

maximum change in temperature per iteration       1.00000E+02
maximum rbe3 conditioning number                  0.10000E+07

transient non auto
----------
```

```
   time      time     maximum   assembly  max iter
increment   period     steps    interval  mcreep
 1.000E+00 5.000E+01        50         0         5

continue
----------
```

auto control specified for time of 5.000E+01

s t a r t   o f   i n c r e m e n t   1

space needed for incremental backup: 63677

thermal pass

fluxes summed over the whole model

from distributed fluxes
magnitudes based upon undeformed configuration
0.000000E+00

```
            concentrated fluxes
            0.000000E+00

            start of assembly     cycle number is     0
            wall time =           1.00



            solver workspace for phase1 (matrix input)    (64bit words) =             6048 (
0.05 MByte)
            Metis nested dissection ordering used
            solver workspace for phase2 (nodal ordering)  (64bit words) =             2262 (
0.02 MByte)
            solver workspace for phase3 (symbolic factor) (64bit words) =             1761 (
0.01 MByte)
            solver workspace for phase4 (factorization)   (64bit words) =             2275 (
0.02 MByte) (MINIMUM)
            solver workspace for phase4 (factorization)   (64bit words) =             2276 (
0.02 MByte) (MAXIMUM)
            solver workspace available and used           (64bit words) =          1000000 (
7.63 MByte)


            start of matrix solution
            wall time =           1.00



            singularity ratio    1.0000E+00
```

```
end of matrix solution
wall time =           1.00


maximum nodal temperature change is  1.000E-20 at node          1

this is  5.000E-20 percent of change allowed on control option


automatic time stepping is switched off
init. thermal energy        is    0.00000E+00
total thermal energy        is    0.00000E+00

stress pass





load increments for each degree of freedom
summed over the whole model

from distributed loads
dist. loads on undeformed configuration - increments for dist. loads
increments for point loads
0.000000E+00   0.000000E+00


point loads
0.000000E+00   0.000000E+00

start of assembly     cycle number is     0
```

```
            wall time =           1.00



            solver workspace for phase1 (matrix input)    (64bit words) =            6048 (
0.05 MByte)
            Metis nested dissection ordering used
            solver workspace for phase2 (nodal ordering)  (64bit words) =           18782 (
0.14 MByte)
            solver workspace for phase3 (symbolic factor) (64bit words) =           25341 (
0.19 MByte)
            solver workspace for phase4 (factorization)   (64bit words) =           41381 (
0.32 MByte) (MINIMUM)
            solver workspace for phase4 (factorization)   (64bit words) =           74047 (
0.56 MByte) (MAXIMUM)
            solver workspace available and used           (64bit words) =         1000000 (
7.63 MByte)


            start of matrix solution
            wall time =           1.00



            singularity ratio    1.6829E-01


            end of matrix solution
            wall time =           1.00


            maximum residual force  at node       443 degree of freedom  1 is equal to    4.030E-
14
```

```
            maximum reaction force  at node       307 degree of freedom  1 is equal to
6.614E+00
            residual convergence ratio    6.093E-15


            dynamic change has reached time of 1.000E+00  of total time period 5.000E+01


            total transient time = 1.00000E+00



   WINDOWS version
   Marc - Student Edition 2016.0.0
   output for increment       1. "          lcase1"


            total strain energy         is    4.87682E-01
              within which:
              elastic strain energy     is    4.87682E-01
            total ext-force work        is    4.87682E-01
              within which:
              work by appl. force/disp. is    4.87682E-01
              work by frictional forces is    0.00000E+00

                             Thermal Results

       element     point      temp                    gradient components
flux components
                                                       1            2            3
1            2            3


            1         1      5.387                  1.521       0.3793
```

```
0.000         0.000
              1          2      7.152                    1.522       0.3591
;             ;
                         ;        ;                        ;              ;
;             ;
0.000         0.000
             11          4      89.92                   -3.401       -2.504
0.000         0.000

   WINDOWS version
   Marc - Student Edition 2016.0.0
   output for increment       1. "          lcase1"


       element     point      temp                   gradient components
flux components
                                                       1            2            3
1             2            3


             12          1      97.65                   -3.025       -2.992
0.000         0.000
             12          2      93.11                   -2.872       -3.056
                         ;        ;                        ;              ;
;             ;
                         ;        ;                        ;              ;
;             ;
0.000         0.000
            439          4      0.000                    0.000        0.000
0.000         0.000

                              Structural Results

           tresca     mises    mean   p r i n c i p a l   v a l u e s        p h y s i c a l   c
```

```
o m p o n e n t s
        intensity intensity  normal   minimum intermediate maximum     1         2         3
4         5         6
                              intensity


 element          1  point   1       integration pt. coordinate=     0.404E+02    0.893E+00
 section thickness = 0.500E+01
 engsts  1.407E-01 1.401E-01-4.601E-02-1.394E-01 0.000E+00 1.351E-03 1.304E-03-1.393E-01 2.582E-03
 engstn  3.485E-04 2.148E-04 0.000E+00-2.081E-04 7.888E-05 1.404E-04 1.403E-04-2.080E-04 1.279E-05
 thermal 5.818E-05 1.164E-04 0.000E+00 0.000E+00 5.818E-05 5.818E-05 5.818E-05 5.818E-05 0.000E+00

 element          1  point   2       integration pt. coordinate=     0.415E+02    0.918E+00
 section thickness = 0.500E+01
 engsts  1.392E-01 1.351E-01-4.927E-02-1.392E-01-8.602E-03 0.000E+00-8.673E-03-1.391E-01 3.042E-03
 engstn  3.234E-04 2.006E-04 0.000E+00-1.830E-04 8.446E-05 1.404E-04 1.402E-04-1.828E-04 1.506E-05
 thermal 7.724E-05 1.545E-04 0.000E+00 0.000E+00 7.724E-05 7.724E-05 7.724E-05 7.724E-05 0.000E+00

 element          1  point   3       integration pt. coordinate=     0.402E+02    0.333E+01
 section thickness = 0.500E+01
 engsts  1.446E-01 1.442E-01-4.770E-02-1.438E-01 0.000E+00 7.400E-04-5.680E-04-1.425E-01 1.369E-02
 engstn  3.580E-04 2.198E-04 0.000E+00-2.083E-04 8.178E-05 1.497E-04 1.464E-04-2.051E-04 6.780E-05
 thermal 6.608E-05 1.322E-04 0.000E+00 0.000E+00 6.608E-05 6.608E-05 6.608E-05 6.608E-05 0.000E+00

 element          1  point   4       integration pt. coordinate=     0.414E+02    0.343E+01
 section thickness = 0.500E+01
 engsts  1.436E-01 1.393E-01-5.086E-02-1.436E-01-8.976E-03 0.000E+00-1.041E-02-1.422E-01 1.384E-02

     ;                        ;                        ;                       ;

     ;                        ;                        ;                       ;

     ;                        ;                        ;                       ;
```

```
 element        439  point   3       integration pt. coordinate=     0.148E+03    0.364E+02
 section thickness = 0.500E+01
 engsts  4.391E-02 4.350E-02 1.491E-02 0.000E+00 8.206E-04 4.391E-02 9.511E-04 4.378E-02-2.367E-03
 engstn  2.718E-07 1.840E-07 0.000E+00-6.390E-08-5.882E-08 2.079E-07-5.801E-08 2.071E-07-2.931E-08

 element        439  point   4       integration pt. coordinate=     0.148E+03    0.342E+02
 section thickness = 0.500E+01
 engsts  4.378E-02 4.373E-02 1.462E-02 0.000E+00 9.855E-05 4.378E-02 3.176E-04 4.356E-02-3.086E-03
 engstn  2.710E-07 1.847E-07 0.000E+00-6.268E-08-6.207E-08 2.083E-07-6.071E-08 2.070E-07-3.820E-08
```

```
                           n o d a l   p o i n t   d a t a

                           i n c r e m e n t a l   d i s p l a c e m e n t s

                   1 -1.58077E-15  1.08567E-16                  2 -1.72493E-15  4.12865E-16
3 -1.60566E-15  1.69533E-16
                   4 -1.62224E-15  1.06395E-16                  5 -1.63117E-15  4.77400E-17
6 -1.64503E-15 -7.49572E-18
                        ;               ;                           ;             ;
;           ;
                        ;               ;                           ;             ;
;           ;

485 -2.27958E-21 -9.44609E-22                486 -2.26694E-21 -7.45146E-22
```

t o t a l   d i s p l a c e m e n t s

```
                 1 -1.40311E-03  3.06355E-34                     2  5.10222E-05 -2.49421E-32
3 -1.12704E-03 -2.51387E-33
                 4 -8.22878E-04 -9.81830E-34                     5 -5.11124E-04  2.27280E-33
6 -2.15670E-04 -3.25327E-34
;            ;
                      ;              ;                              ;            ;
;            ;
                      ;              ;                              ;            ;
;            ;
483  3.39817E-05  4.37533E-06
               484  3.35310E-05  5.04756E-06                   485  3.30855E-05  5.95247E-06
486  3.26170E-05  7.18174E-06
```

t o t a l   n o d a l   t e m p e r a t u r e s

```
              1   4.4700                2   14.325                 3   7.5450
4   10.330                5   12.150
              6   13.335                7   5.7550                 8   9.5400
9   15.655               10   23.770
;            ;
                      ;              ;                              ;            ;
;            ;
            481   0.0000              482   0.0000               483   0.0000
484   0.0000              485   0.0000
            486   0.0000
```

t o t a l   n o d a l   f l u x e s

```
               1   0.0000                    2   0.0000                    3   0.0000
4   0.0000                    5   0.0000
               6   0.0000                    7   0.0000                    8   0.0000
                      ;               ;                              ;             ;
;            ;
                      ;               ;                              ;             ;
;            ;
479   0.0000                  480   0.0000
             481   0.0000                  482   0.0000                  483   0.0000
484   0.0000                  485   0.0000
             486   0.0000
```

total equivalent nodal forces (distributed plus point loads)

```
                  1   0.0000       0.0000                        2   0.0000       0.0000
3   0.0000       0.0000
                  4   0.0000       0.0000                        5   0.0000       0.0000
;            ;
                      ;               ;                              ;             ;
;            ;
483   0.0000       0.0000
                484   0.0000       0.0000                      485   0.0000       0.0000
486   0.0000       0.0000
```

```
                    reaction forces at fixed boundary conditions, residual load correction
elsewhere


                     1  1.04083E-17  0.72903                            2   0.0000       2.2342
3  1.38778E-17   1.3763
                     4   0.0000       1.3266                            5   0.0000       1.2314
6  5.55112E-17   1.1027

                        ;               ;                               ;             ;
;            ;
                        ;               ;                               ;             ;
;            ;
                        ;               ;                               ;             ;
;            ;

                   478   0.0000      1.11022E-16                      479 -2.22045E-16  5.55112E-17
480 -3.33067E-16  1.38778E-17
                   481 -1.11022E-16   0.0000                          482  2.22045E-16  2.77556E-17
483  1.66533E-16 -3.33067E-16
                   484  2.22045E-16  1.11022E-16                      485  8.32667E-17  1.11022E-16
486  2.77556E-17  1.11022E-16

                              summary of externally applied loads

    0.00000E+00    0.00000E+00

                              summary of reaction/residual forces

    1.17796E-14   -3.10862E-15


             memory usage:                      MByte       words  % of total
```

```
 within general memory:
  element stiffness matrices:           0      21554         0.1
  solver: first part                    0      63376         0.2
  overallocation initial allocation   106   27914276        74.3
  other:                                0       4660         0.0
 allocated separately:
  incremental backup:                   0     127354         0.3
  solver 8                              8    1987904         5.3
  nodal vectors:                        0      64166         0.2
  defined sets:                         0       3620         0.0
  transformations:                      0       1944         0.0
  kinematic boundary conditions:        0      54232         0.1
  points, curves and surfaces:          0         86         0.0
  mem_none:                             0      43410         0.1
  element storage:                      1     259576         0.7
  material properties:                  0       5150         0.0
  executable and common blocks:        27    7000000        18.6
  miscellaneous                         0        212         0.0
  ---------------------------------------------------------------
  total:                              143   37551520

  general memory allocated:           107   28003866
  general memory used:                  0      89590

  peak memory usage:                  159   41557522

e n d   o f   i n c r e m e n t     50

binary post data at increment      50.  subincrement    0.  on file 16

formatted post data at increment      50.  subincrement    0.  on file 19
wall time =          10.00
```

```
$....end of loadcase lcase1

$..................

                 *** end of input deck - job ends

memory usage:                              MByte       words  % of total

 within general memory:
  element stiffness matrices:                  0       21554         0.1
  solver: first part                           0       63376         0.2
  overallocation initial allocation          106    27914276        74.3
  other:                                       0        4660         0.0
 allocated separately:
  incremental backup:                          0      127354         0.3
  solver 8                                     8     1987904         5.3
  nodal vectors:                               0       64166         0.2
  defined sets:                                0        3620         0.0
  transformations:                             0        1944         0.0
  kinematic boundary conditions:               0       54232         0.1
  points, curves and surfaces:                 0          86         0.0
  mem_none:                                    0       43410         0.1
  element storage:                             1      259576         0.7
  material properties:                         0        5150         0.0
  executable and common blocks:               27     7000000        18.6
  miscellaneous                                0         212         0.0
  ---------------------------------------------------------------
  total:                                     143    37551520

  general memory allocated:                  107    28003866
  general memory used:                         0       89590
```

```
  peak memory usage:                  159    41557522


timing information:                          wall time      cpu time

  total time for input:                           0.16          0.14
  total time for stiffness assembly:              1.32          1.19
  total time for stress recovery:                 1.24          1.12
  total time for matrix solution:                 1.64          1.39
  total time for output:                          3.85          3.45
  total time for miscellaneous:                   0.86          0.70
  ---------------------------------------------------------------
  total time:                                     9.08          8.00


**************************************************************************


This is a successful completion to a Marc simulation,
indicating that no additional incremental data was
found and that the analysis is complete.



**************************************************************************


                           Marc - Student Edition 2016.0.0

                           Exit number 3004
```