UNIVERSITY OF THESSALY
SCHOOL OF ENGINEERING
DEPARTMENT OF MECHANICAL ENGINEERING

Numerical Calculation of Stress Concentration Factors in Shafts with Flat Bottom Grooves under Tension and Comparison with Theoretical Results.

by
Georgios Kompothanasis

Submitted in partial fulfillment of the requirements for the degree of Diploma in Mechanical Engineering at the University of Thessaly

Volos, 2021

© 2021 Georgios Kompothanasis

All rights reserved. The approval of the present D Thesis by the Department of Mechanical Engineering, School of Engineering, University of Thessaly, does not imply acceptance of the views of the author (Law 5343/32 art. 202).

Approved by the Committee on Final Examination:

Advisor	Dr. Sotiria Chouliara, Lab Teaching Personnel, Department of Mechanical Engineering, University of Thessaly
Member	Dr. Nikolaos Aravas, Professor, Department of Mechanical Engineering, University of Thessaly
Member	Dr. Alexis Kermanidis, Associate Professor, Department of Mechanical Engineering, University of Thessaly

Date Approved: [July 08, 2021]

Numerical Calculation of Stress Concentration Factors in Shafts with Flat Bottom Grooves and Comparison with Theoretical Results.

Georgios Kompothanasis

Department of Mechanical Engineering, University of Thessaly

Supervisor: Dr Sotiria Chouliara

Lab Teaching Personnel

Abstract

Theoretical and experimental measurements show that in a loaded structural member, near changes in the section, distributions of stress occur in which the peak stress reaches much larger magnitudes than does the average stress over the section. This increase in peak stress near holes, grooves, notches, sharp corners, cracks, and other changes in section is called stress concentration.

The purpose of this project is the numerical study of stress concentration factors (SCF) for shafts with geometrical discontinuities under tension. This is a classic topic of mechanical engineering and it is strongly connected with applications in machine elements. Any geometric discontinuity in a shaft or an axle, changes the stress allocation in the area, and prevents the accurate calculation of the stress with a simple equation. The stress concentration factor is a parameter that needs to be determined in order to get the ‘’exact’’ calculation of the stress load. This study refers to shafts with flat bottom grooves under tension. Finite Elements methods (ABAQUS) are applied to determine the corresponding stress concentration factors considering various parameters and, in the sequence, the obtained numerical results are compared with theoretical results available in literature.

Contents

Chapter 1. Introduction

1.1 Stress Concentration

The elementary stress formulas used in the design of structural members are based on the members having a constant section or a section with gradual change of contour (Figure 1). Such conditions, however, are hardly ever attained throughout the highly stressed region of actual machine parts or structural members. The presence of shoulders, grooves, holes, keyways, threads (Figure 2) and so on, results in modifications of the simple stress distributions given by Figure 1 so that localized high stresses occur.

A stress concentration is a location in an structural member where the stress is significantly greater than the surrounding region. In the presence of irregularities in the geometry or the material of a structural component stress concentrations occur that cause an interruption to the flow of stress. Stress concentrations may also occur from accidental damage such as nicks and scratches.

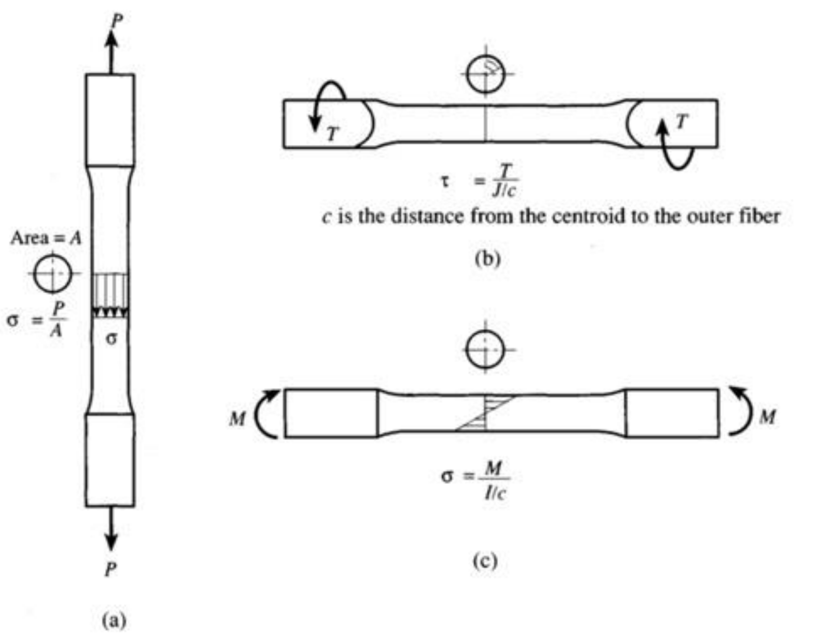


Figure 1 Elementary stress cases for specimens: a) Tension b) Torsion c) Bending

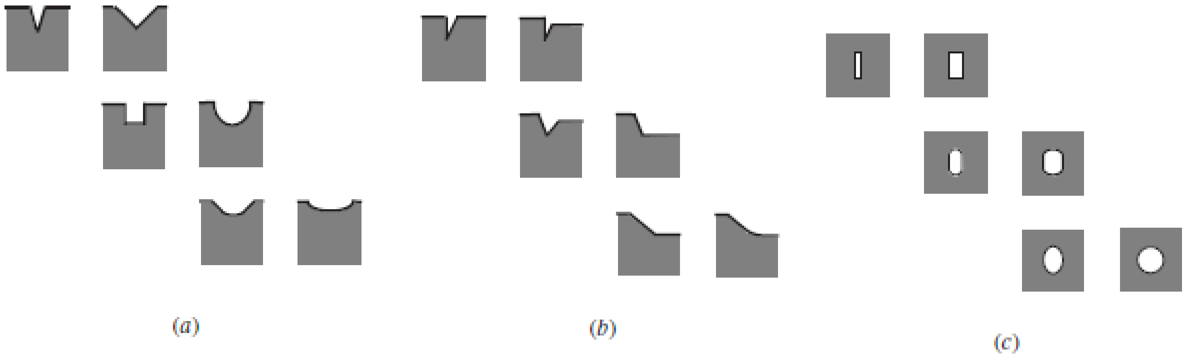


Figure 2 Reducing the effect of the stress concentration of notches and holes: (*a*) Notch shapes arranged in order of their effect on the stress concentration decreasing (moving from left to right and top to bottom) (*b*) asymmetric notch shapes, arranged in the same way (*c*) holes, arranged in the same way.[3]

1.2 Stress Concentration Factors

This localization of high stress (stress concentration) is measured by the *stress concentration factor (SCF).* The stress concentration factor K_t can be defined as the ratio of the peak stress in the body (or stress in the perturbed region) to some other stress (or stress like quantity) taken as a reference stress:

$$K_t = \frac{\sigma_{\max}}{\sigma_{nom}}$$ for normal stress (tension or bending)

$$K_t = \frac{\tau_{\max}}{\tau_{nom}}$$ for shear stress (torsion)

where the stresses $\sigma_{\max}$, $\tau_{\max}$ represent the maximum stresses to be expected in the member under the actual loads and the *nominal stresses* σ_{nom}, τ_{nom} are the reference normal and shear stresses, respectively.

It is noted that the value of this non-dimensional ratio K_t is in any case greater than or equal to 1 ($K_t \geq 1$).

1.3 Shafts and Shaft Materials

A *shaft* is a rotating member, usually of circular cross section, used to transmit power or motion. It provides the axis of rotation, or oscillation, of elements such as gears, pulleys, flywheels, cranks, sprockets, and controls the geometry of their motion. An *axle* is a non-rotating member that carries no torque and is used to support rotating wheels, pulleys, and the like. It is noted that the automotive axle is not a true axle; the term is a carryover from the horse-and-buggy era, when the wheels rotated on nonrotating members. A nonrotating axle can readily be designed and analyzed as a static beam, and will not warrant the special attention given in this chapter to the rotating shafts which are subject to fatigue loading.

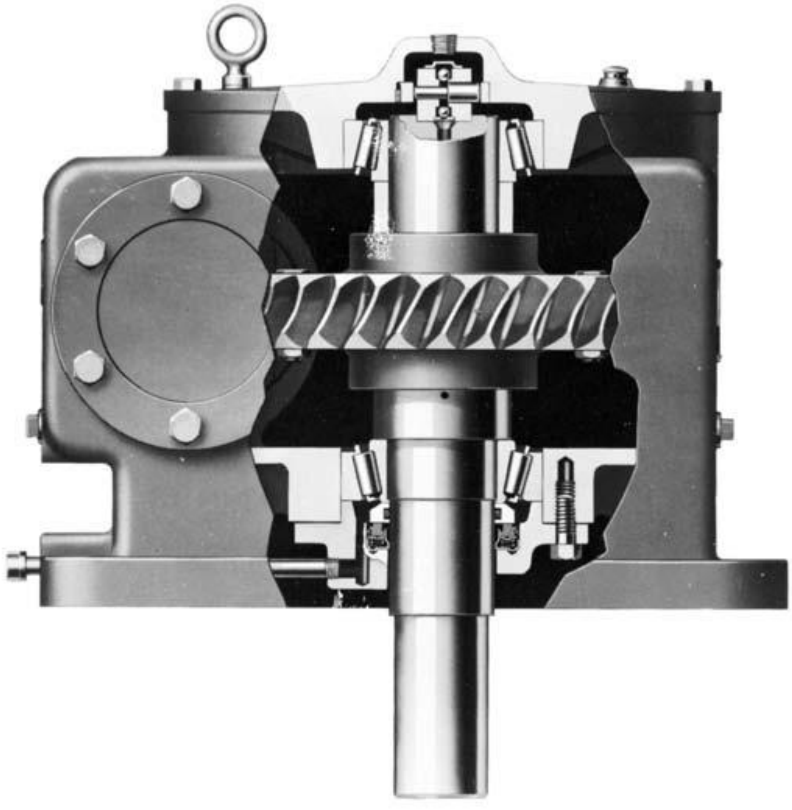


Figure 3 A vertical worm-gear speed reducer.[2]

Because of the ubiquity of the shaft in so many machine design applications, there is some advantage in giving the shaft and its design a closer inspection. A complete shaft design has much interdependence on the design of its components. The design of the machine itself is the one that dictates certain gears, pulleys, bearings, and other elements will have at least been partially analyzed and their size and spacing tentatively determined. In deciding on an approach to shaft sizing, it is necessary to realize that a stress analysis at a specific point on a shaft can be made using only the shaft geometry in the vicinity of that point.

Deflection is not affected by strength, but rather by stiffness as represented by the modulus of elasticity, which is essentially constant for all steels. For that reason, rigidity cannot be controlled by material decisions, but only by geometric decisions. Necessary strength to resist loading stresses affects the choice of materials and their treatments.

Many widely used shafts are made from low carbon, cold-drawn or hot-rolled steels, such as the AISI 1020-1050 steels. The properties of these steels are given by Figure 4. Significant strengthening from heat treatment and high alloy content are often not warranted. Fatigue failure is reduced moderately by increase in strength, and then only to a certain level before adverse effects in endurance limit and notch sensitivity begin to counteract the benefits of higher strength.

1 AISI No.	2 Treatment	3 Temperature °C (°F)	4 Tensile Strength MPa (kpsi)	5 Yield Strength, MPa (kpsi)	6 Elongation, %	7 Reduction in Area, %	8 Brinell Hardness
1030	Q&T*	205 (400)	848 (123)	648 (94)	17	47	495
	Q&T*	315 (600)	800 (116)	621 (90)	19	53	401
	Q&T*	425 (800)	731 (106)	579 (84)	23	60	302
	Q&T*	540 (1000)	669 (97)	517 (75)	28	65	255
	Q&T*	650 (1200)	586 (85)	441 (64)	32	70	207
	Normalized	925 (1700)	521 (75)	345 (50)	32	61	149
	Annealed	870 (1600)	430 (62)	317 (46)	35	64	137
1040	Q&T	205 (400)	779 (113)	593 (86)	19	48	262
	Q&T	425 (800)	758 (110)	552 (80)	21	54	241
	Q&T	650 (1200)	634 (92)	434 (63)	29	65	192
	Normalized	900 (1650)	590 (86)	374 (54)	28	55	170
	Annealed	790 (1450)	519 (75)	353 (51)	30	57	149
1050	Q&T*	205 (400)	1120 (163)	807 (117)	9	27	514
	Q&T*	425 (800)	1090 (158)	793 (115)	13	36	444
	Q&T*	650 (1200)	717 (104)	538 (78)	28	65	235
	Normalized	900 (1650)	748 (108)	427 (62)	20	39	217
	Annealed	790 (1450)	636 (92)	365 (53)	24	40	187

Figure 4 Mean Mechanical Properties of Some Heat-Treated Steels. Q&T stands for quenched and tempered.[2]

A good practice is to start with an inexpensive, low or medium carbon steel for the first time through the design calculations. If strength considerations turn out to dominate over deflection, then a higher strength material should be tried, allowing the shaft sizes to be reduced until excess deflection becomes an issue.

The cost of the material and its processing must be weighed against the need for smaller shaft diameters. Shafts usually don't need to be surface hardened unless they serve as the actual journal of a bearing surface.

Cold drawn steel is usually used for diameters under (about) 3 inches. The nominal diameter of the bar can be left unmachined in areas that do not require fitting of components. Hot rolled steel should be machined all over. For large shafts requiring much material removal, the residual stresses may tend to cause warping.

For low production, turning is the usual primary shaping process. An economic viewpoint may require removing the least material. High production may permit a volume conservative shaping method (hot or cold forming, casting), and minimum material in the shaft can become a design goal. Cast iron may be specified if the production quantity is high, and the gears are to be integrally cast with the shaft.

Conclusively, it can be said that properties of the shaft locally depend on its history—cold work, cold forming, rolling of fillet features, heat treatment, including quenching medium and agitation.

1.4 Shafts Layout

The general layout of a shaft is to accommodate shaft elements such as gears, bearings and pulleys. These elements are specified earlier in the design process in order to perform a free body force analysis and to obtain shear-moment diagrams. Shaft geometry is generally that of a stepped cylinder. The use of shaft shoulders is an excellent means of axially locating the machine elements and to carry any thrust loads.

The geometry configuration of a shaft to be designed is often simply a revision of existing models in which a limited number of changes must be made. If there is no existing design to use as a starter, then the determination of the shaft layout may have many solutions.

This problem is illustrated by the following two examples. In Figure 5-a a geared countershaft is to be supported by two bearings. In Figure 5-c a fanshaft is to be configured.

The solution shown in Figure 5-b and Figure 5-d are not necessarily the best ones, but they do illustrate how the shaft-mounted devices are fixed and located in the axial direction, and how provision is made for torque transfer from one element to another.

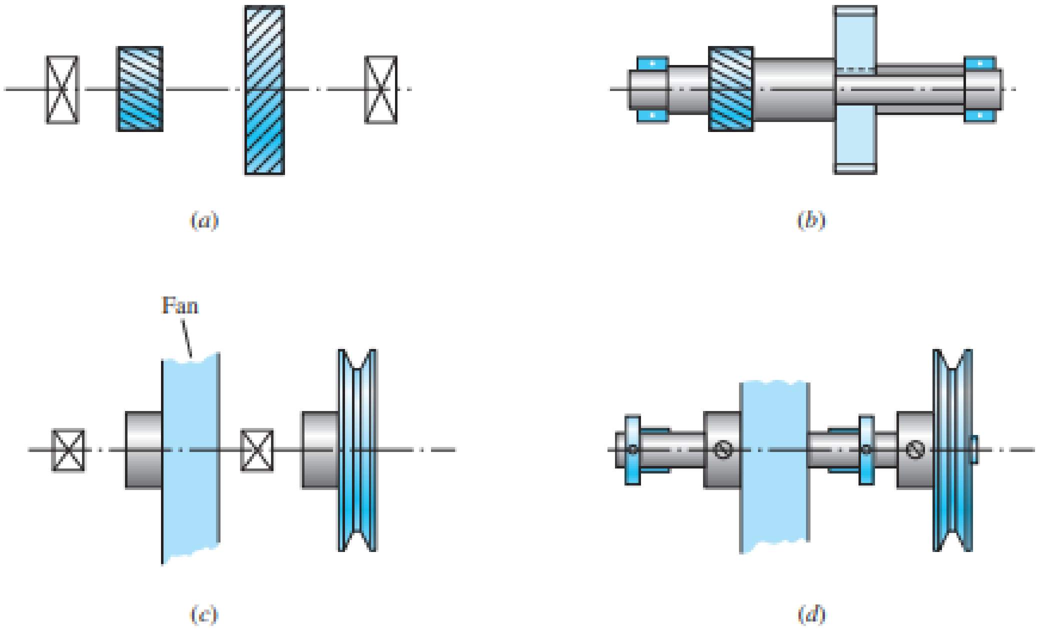


Figure 5 *a*) Choice of a shaft configuration to support and locate the two gears and two bearings. (*b*) Solution uses an integral pinion, three shaft shoulders, key and keyway, and sleeve. The housing locates the bearings on their outer rings and receives the thrust loads. (*c*) Choose fan-shaft configuration. (*d*) Solution uses sleeve bearings, a straightthrough shaft, locating collars, and setscrews for collars, fan pulley, and fan itself. The fan housing supports the sleeve bearings.[2]

1.5 Shoulder Fillet

The shoulder fillet is the type of geometric discontinuity that is commonly encountered in machine design practice. Shafts, axles, spindles, rotors, and so forth, usually involve a number of diameters connected by shoulders with rounded fillets replacing the sharp corners that were often used in former years. (Figure 6).

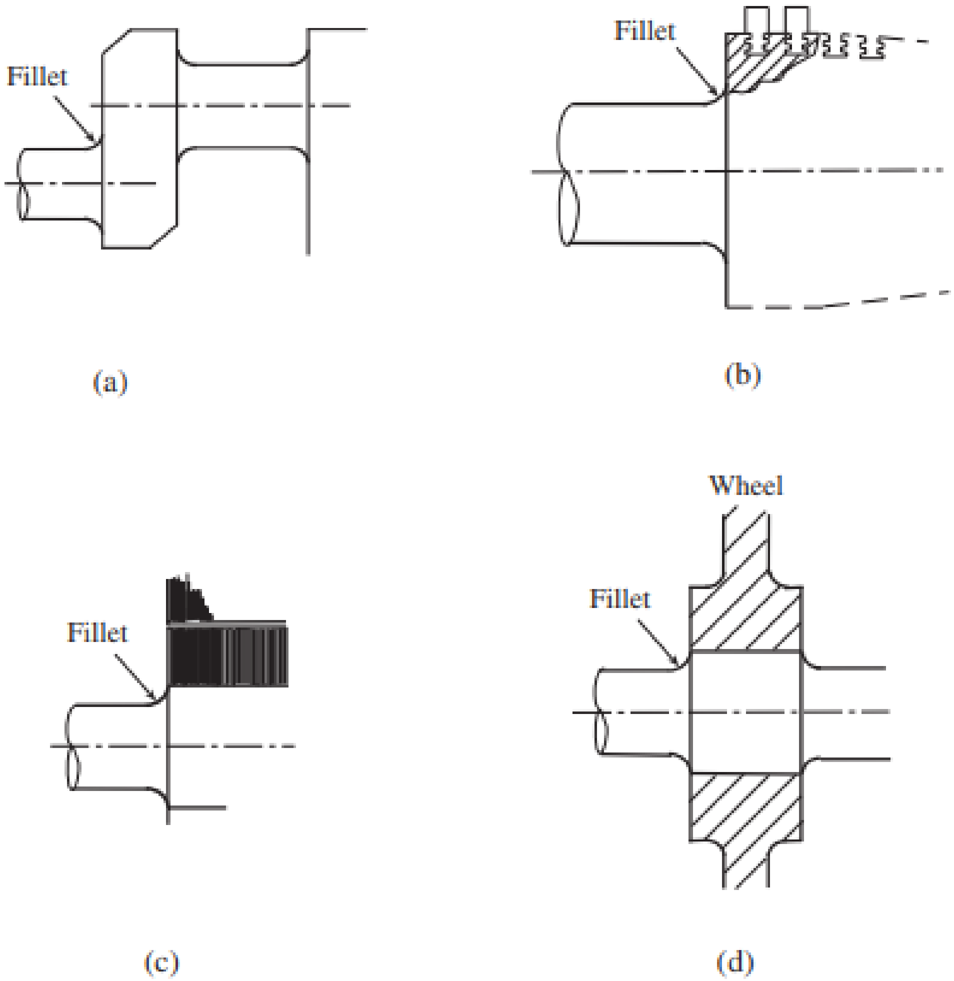


Figure 6 Examples of filleted members: (a) engine crankshaft, (b) turbine rotor, (c) motorshaft, (d) railway axle.[2]

1.6 Selection of the nominal stress

The definitions of the reference stresses σ_{nom} depend on the problem at hand. It is very important to properly identify the reference stress for the stress concentration factor of interest. The next example is used to explain the importance of selection of reference stress.

Example 1 Tension bar with a hole

Uniform tension is applied to a bar with a single circular hole, as shown in Figure 7. The maximum stress occurs at point A, and the stress distribution of $\sigma_{\theta\theta}(\theta = \frac{\pi}{2})$ can be shown to be as in Figure 7. It is supposed that the thickness of the plate is h, the width of the plate is H, and the diameter of the hole is d. The reference stress could be defined in two ways:

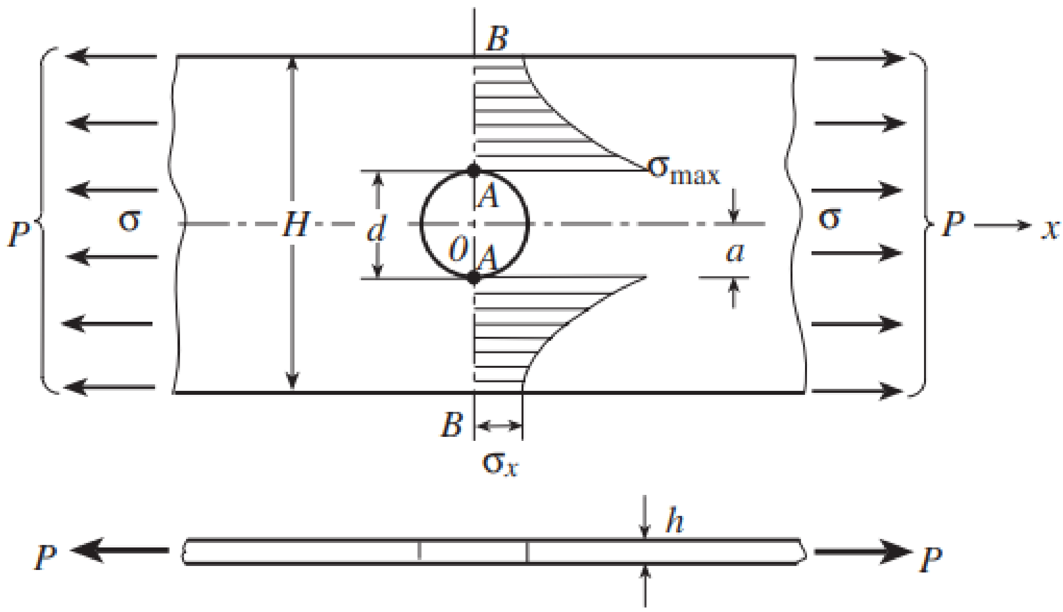


Figure 7 Tension bar with hole

a. Use of the stress in a cross section far from the circular hole as the reference stress. The area at this section is called the gross cross-sectional area. σ_{nom} is defined:

$$\sigma_{nom} = \frac{P}{Hh} = \sigma$$

So that the stress concentration factor becomes:

$$K_{tg} = \frac{\sigma_{\max}}{\sigma_{nom}} = \frac{\sigma_{\max}}{\sigma} = \frac{\sigma_{\max} Hh}{P}$$

b. Use of the stress based on the cross section at the hole, which is formed by removing the circular hole from the gross cross section. The corresponding area is referred to as the net cross-sectional area. If the stresses at this cross section are uniformly distributed and equal to σ_n:

$$\sigma_n = \frac{P}{(H-d)h}$$

The stress concentration factor based on the reference stress σ_n, namely, $\sigma_{nom} = \sigma_n$, is:

$$K_{tn} = \frac{\sigma_{\max}}{\sigma_{nom}} = \frac{\sigma_{\max}}{\sigma_n} = \frac{\sigma_{\max}(H-d)h}{P} = K_{tg}\frac{H-d}{H}$$

In general, K_{tg} and K_{tn} are different. It can easily be observed that as d/H increases from 0 to 1, K_{tg} increases from 3 to ∞, whereas K_{tn} decreases from 3 to 2. Either K_{tn} or K_{tg} can be used in calculating the maximum stress. It would appear that K_{tg} is easier to determine as σ is immediately evident from the geometry of the bar. But the value of K_{tg} is hard to read from a stress concentration plot for d/H > 0.5, since the curve becomes very steep. In contrast, the value of K_{tn} is easy to read, but it is necessary to calculate the net cross-sectional area to find the maximum stress. Since the stress of interest is usually on the net cross section, K_{tn} is the more generally used factor. In addition, in a fatigue analysis, only K_{tn} can be used to calculate the stress gradient correctly. In conclusion, normally it is more convenient to give stress concentration factors using reference stresses based on the net area rather than the gross area.

1.7 Stress Concentration Factor - Large Plate with Circular Hole

Figure 8 shows a large plate that contains a small circular hole. For an applied uniaxial tension the stress field is found from linear elasticity theory. In polar coordinates the circumferential component of stress at point *P* is given as

$$\sigma_{\theta\theta} = \frac{1}{2}\sigma\left[1+\frac{r^2}{\rho^2}\right] - \frac{1}{2}\sigma\left[1+3\frac{r^4}{\rho^4}\right]\cos 2\theta$$

The maximum stress occurs at the sides of the hole where $\rho = r$ and $\theta = \frac{1}{2}\pi$ or

$\theta = \frac{3}{2}\pi$. At the hole sides,

$$\sigma_{\theta\theta} = 3\sigma$$

The peak stress is three times the uniform stress σ. To account for the peak in stress near a stress raiser, the *stress concentration factor* or *theoretical stress concentration factor* K_t is defined as the ratio of the calculated peak stress to the nominal stress that would exist in the member if the distribution of stress remained uniform; that is,

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

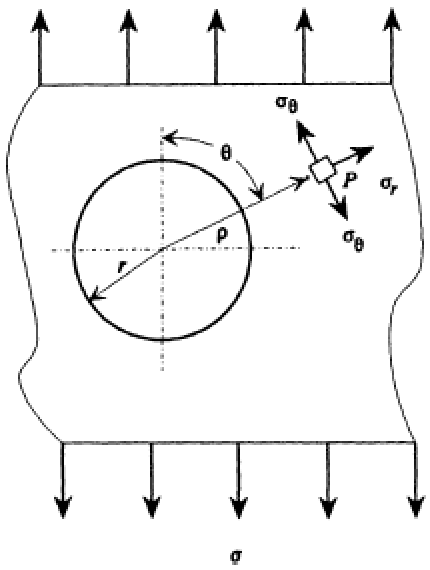


Figure 8 Large plate with small circular hole

In general, the nominal stress is found using basic strength-of-materials formulas, and the calculations are based on the properties of the net cross section at the stress raiser. Sometimes the overall section is used in computing the nominal stress and this was presented in previous section 1.6

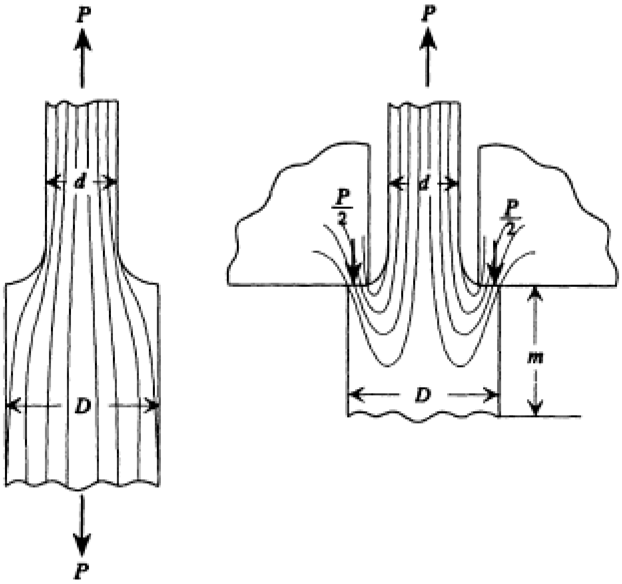
If applied stress σ is chosen as the nominal stress for the case shown in Figure 8, the stress concentration factor is

$$K_t = \frac{\sigma_{\max}}{\sigma_{nom}} = 3$$

The effect of the stress raiser is to change only the distribution of stress. Equilibrium requirements dictate that the average stress on the section is the same in the case of stress concentration as it would be if a uniform stress distribution was used. Stress concentration results not only in unusually high stresses near the stress raiser but also in unusually low stresses in the remainder of the section. When more than one loads act on a notched member (e.g., combined tension, torsion, and bending) the nominal stress due to each load is multiplied by the stress concentration factor corresponding to each load, and the resultant stresses are found by superposition. However, when bending and axial loads act simultaneously, superposition can be applied only when bending moments due to the interaction of axial force and bending deflections are negligible compared to bending moments due to applied loads.

1.8 Designing to Minimize Stress Concentration

As a general rule, force should be transmitted from point to point as smoothly as possible. The lines connecting the force transmission path are sometimes called the *force* (or *stress*) *flow*. Sharp transitions in the direction of the force flow should be removed by smoothing contours and rounding notch roots. When stress raisers are necessitated by functional requirements, the raisers should be placed in regions of low nominal stress if possible. Figure 2 of section 1.1 depicts forms of notches and holes in the order in which they cause stress concentration. Figure 9 shows how direction of stress flow affects the extent to which a notch causes stress concentration. The configuration in Figure 9-*b* has higher stress levels because of the sharp change in the direction of force flow.



(a) (b)

Figure 9 Two parts with the same shape (step in cross section) but differing stress flow patterns can give totally different notch effects and widely differing stress levels at the corner step: (*a*) stress flow is smooth; (*b*) sharp change in the stress flow direction causes high stress.[3]

When notches are necessary, removal of material near the notch can alleviate stress concentration effects (Figure 10).

(*a*) (*b*)

Figure 10 Guiding the lines of stress by means of notches that are not functionally essential is a useful method of reducing the detrimental effects of notches that cannot be avoided. These are termed relief notches. It is assumed here that the bearing surface of the step of (*a*) is needed functionally. Adding a notch as in (*b*) can reduce the hazardous effects of the corner of (*a*).[3]

1.9 Static Theory

A *static load* is a stationary force or moment, called load, applied to a member. To be stationary, the load must be unchanging in magnitude, point or points of application, and direction. A static load can produce axial tension or compression, a shear load, a bending load, a torsional load, or any combination of these. To be considered static, the load cannot change in any manner.

Failure can mean a part has separated into two or more pieces; has become permanently distorted, thus ruining its geometry; has had its reliability downgraded; or has had its function compromised, whatever the reason. A designer speaking of failure can mean any or all of these possibilities.

In strength-sensitive situations the designer must separate mean stress and mean strength at the critical location sufficiently to accomplish his or her purposes. Figures 11 to 14 are photographs of several failed parts. The photographs exemplify the need of the designer to be well-versed in failure prevention. Toward this end we shall consider one-, two-, and three-dimensional stress states, with and without stress concentrations, for both ductile and brittle materials.

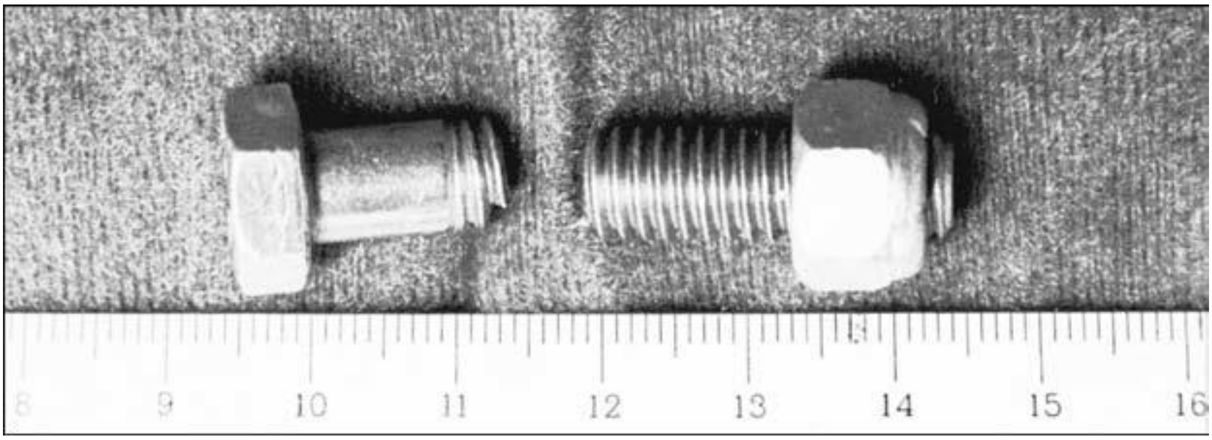


Figure 11 Failure of an overhead-pulley retaining bolt on a weightlifting machine. A manufacturing error caused a gap that forced the bolt to take the entire moment load.[2]

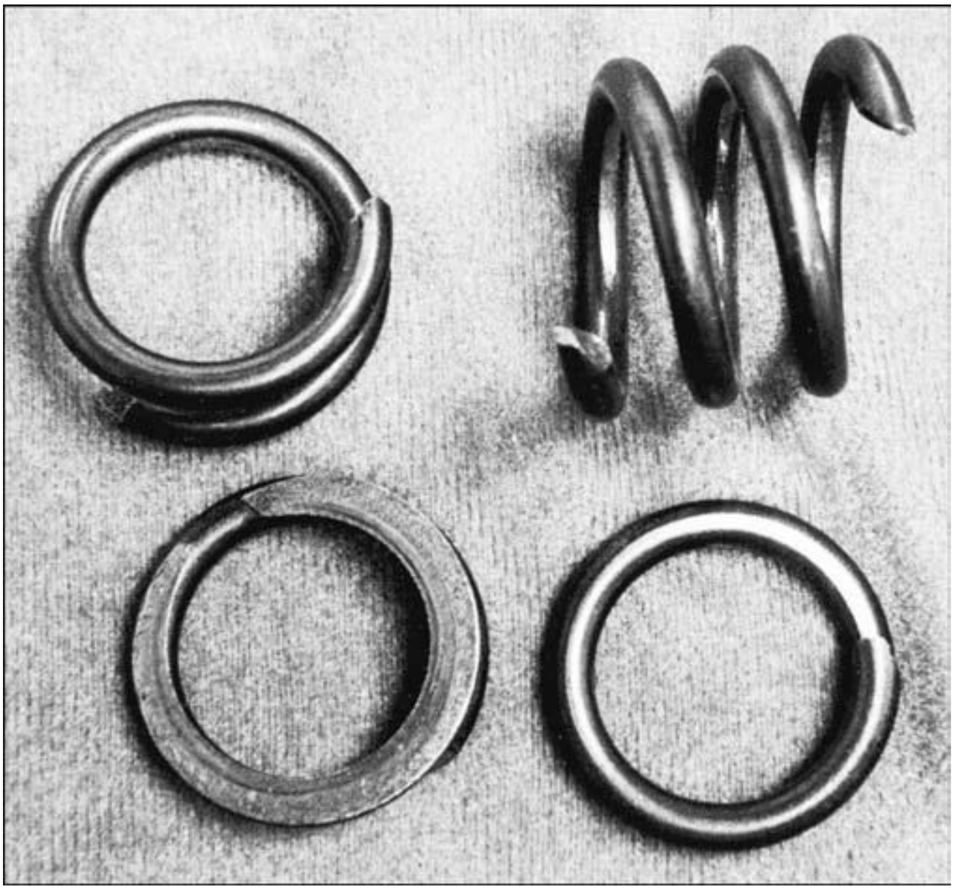


Figure 12 Valve-spring failure caused by spring surge in an oversped engine. The fractures exhibit the classic 45° shear failure.[2]

1.10 Notch Sensitivity

In section 1.1 it was pointed out that the existence of geometric irregularities or discontinuities, in a part increase the theoretical stresses significantly in the immediate vicinity of the discontinuity. A stress-concentration factor K_t is used with the nominal stress in order to obtain the maximum resulting stress due to the irregularity or defect. The use of stress concentration factor K_t (subscript t stands for theoretical) is combined with static load. However, when fatigue conditions are present a reduced value of stress concentration factor, given by K_f, is applied

$$\sigma_{\max} = K_f \sigma_{nom}$$

Or

$$\tau_{\max} = K_f \tau_{nom}$$

The factor K_f is commonly called a *fatigue stress-concentration factor,* and hence the subscript *f*. So it is convenient to think of K_f as a stress-concentration factor reduced from K_t because of lessened sensitivity to notches. The resulting factor is defined by the equation

$$K_f = \frac{\text{maximum stress in notched specimen}}{\text{stress in notch-free specimen}}$$

Another parameter considered in fatigue analysis is q, which expressed the so-called notch sensitivity and it is defined by

$$q = \frac{K_f - 1}{K_t - 1}$$

Parameter q usually takes values between zero and unity. The above equation shows that if $q = 0$, then $K_f = 1$, and the material has no sensitivity to notches at all. On the other hand, if $q = 1$, then $K_f = K_t$, and the material has full notch sensitivity. In analysis or design work, K_t *is* found first, from the geometry of the part. Then the material is specified, q is found, and K_f is solved from the equation

$$K_f = 1 + q(K_t - 1)$$

Notch sensitivities for specific materials are obtained experimentally. Published experimental values are limited, but some values are available for steels and aluminum. A typical graph of notch sensitivities is given by Figure 13. In using this graph it is well to know that the actual test results from which the curves were derived exhibit a large amount of scatter. Because of this scatter it is always safe to be used $K_f = K_t$ if there is any doubt about the true value of q. Also, it is noted that *q* is not far from unity for large notch radii.

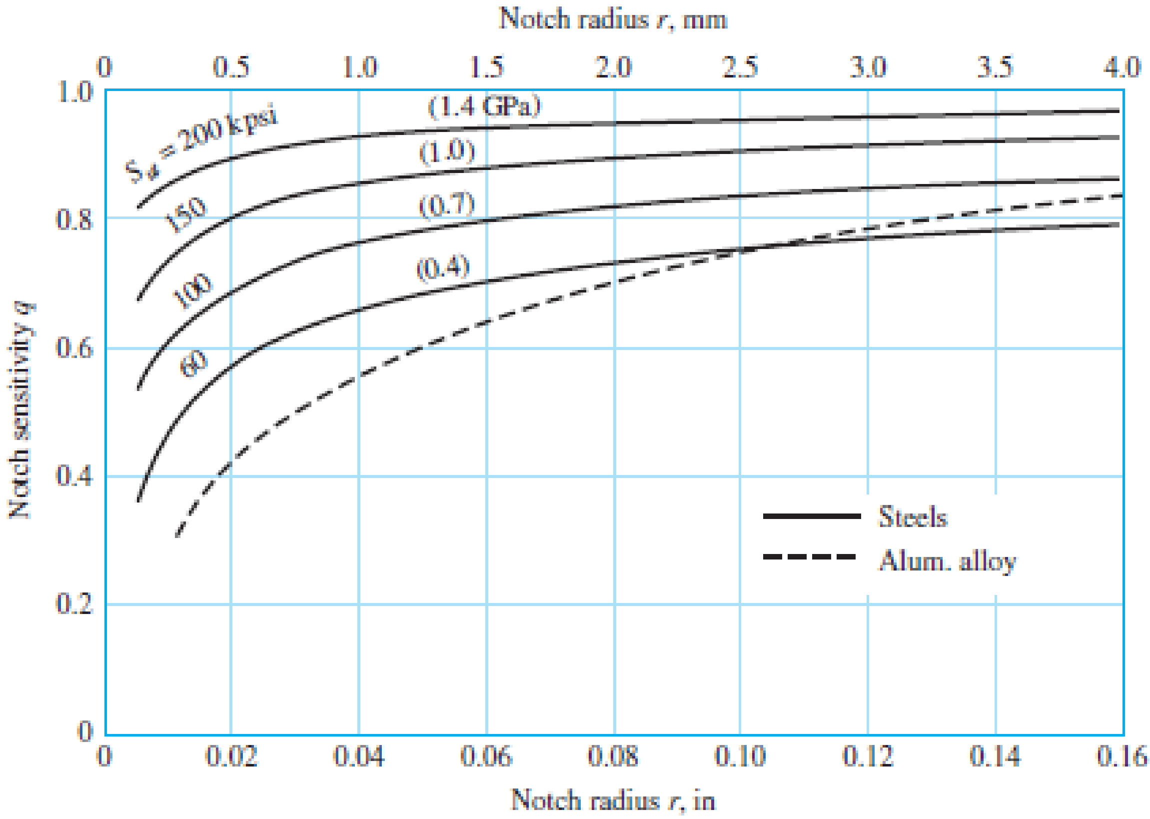


Figure 13 Notch-sensitivity charts for steels and UNS A92024-T wrought aluminum alloys subjected to reversed bending or reversed axial loads.[2]

The above figure has as its basis the *Neuber equation,* which is given by

$$K_f = 1 + \frac{K_t - 1}{1 + \sqrt{\frac{a}{r}}}$$

where $\sqrt{a}$ is defined as the *Neuber constant* and is a material constant. The notch sensitivity equation is

$$q = \frac{1}{1 + \sqrt{\frac{a}{r}}}$$

Chapter 2. FINITE ELEMENT METHOD (FEM)

2.1 Finite Elements

Mechanical components in the form of simple bars, beams, etc., can be analyzed quite easily by basic methods of mechanics that provide closed-form solutions. Actual components, however, are rarely so simple, and the designer is forced to less effective approximations of closed-form solutions, experimentation, or numerical methods.

There are a great many numerical techniques used in engineering applications for which the digital computer is very useful. In mechanical design, where computer-aided design (CAD) software is heavily employed, the analysis method that integrates well with CAD is finite-element analysis (FEA). The mathematical theory and applications of the method are vast.

The purpose of this chapter is only to expose the reader to some of the fundamental aspects of FEA, and therefore the coverage is extremely introductory in nature. Figure 14 shows a finite-element model of a connecting rod that was developed to study the effects of dynamic elastohydrodynamic lubrication on bearing and structural performance. There are a multitude of FEA applications such as static and dynamic, linear and nonlinear, stress and deflection analysis; free and forced vibrations; heat transfer (which can be combined with stress and deflection analysis to provide thermally induced stresses and deflections); elastic instability (buckling); acoustics; electrostatics

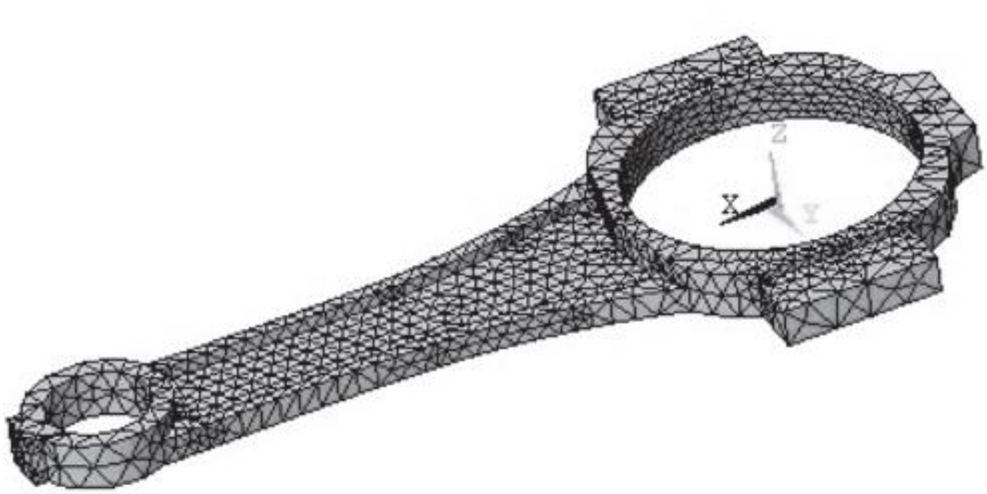


Figure 14 Model of a connecting rod using ANSYS finite-element software (Meshed model). [2]

Figure 15; Model of a connecting rod using ANSYS finite-element software (Stress contours).[2]

and magnetics (which can be combined with heat transfer); fluid dynamics; piping analysis; and multiphysics. An actual mechanical component is a continuous elastic structure (continuum). FEA divides (discretizes) the structure into small but finite, well-defined, elastic substructures (elements). By using polynomial functions, together with matrix operations, the continuous elastic behavior of each element is developed in terms of the element's material and geometric properties. Loads can be applied within the element (gravity,dynamic, thermal, etc.), on the surface of the element, or at the nodes of the element.

The element's nodes are the fundamental governing entities of the element, as it is the node where the element connects to other elements, where elastic properties of the element are eventually established, boundary conditions are assigned, and forces (contact or body) are ultimately applied. A node possesses degrees of freedom (dof's), which are the independent translational and rotational motions that can exist at a node. At most, a node can possess three translational and three rotational degrees of freedom. Once each element within a structure is defined locally in matrix form, the elements are then globally assembled (attached) through their common nodes (dof's) into an overall system matrix. Applied loads and boundary conditions are then specified and through matrix operations the values of all unknown displacement degrees of freedom are determined. Once this is done, the next step is the calculation of displacements knowing the field of displacements, the determination of strains and then of stresses is achieved.

2.2 The Method

The modern development of the finite-element method began in the 1940s in the field of structural mechanics with the work of Hrennikoff, McHenry, and Newmark, who used a lattice of line elements (rods and beams) for the solution of stresses in continuous solids. In 1943, from a 1941 lecture, Courant suggested piecewise polynomial interpolation over triangular subregions as a method to model torsional problems. With the advent of digital computers in the 1950s it became practical for engineers to write and solve the stiffness equations in matrix form. A classic paper by Turner, Clough, Martin, and Topp published in 1956 presented the matrix stiffness equations for the truss, beam, and other elements. The expression finite element is first attributed to Clough.

Since these early beginnings, a great deal of effort has been expended in the development of the finite element method in the areas of element formulations and computer implementation of the entire solution process. The major advances in computer technology include the rapidly expanding computer hardware capabilities, efficient and accurate matrix solver routines, and computer graphics for ease in the visual preprocessing stages of model building, including automatic adaptive mesh generation, and in the postprocessing stages of reviewing the solution results.

Since the finite-element method is a numerical technique that discretizes the domain of a continuous structure, errors are inevitable. These errors are:

1. Computational errors. These are due to round-off errors from the computer floating-point calculations and the formulations of the numerical integration schemes that are employed. Most commercial finite-element codes concentrate on reducing these errors, and consequently the analyst generally is concerned with discretization factors.
2. Discretization errors. The geometry and the displacement distribution of a true structure continuously vary. Using a finite number of elements to model the structure introduces errors in matching geometry and the displacement distribution due to the inherent mathematical limitations of the elements.

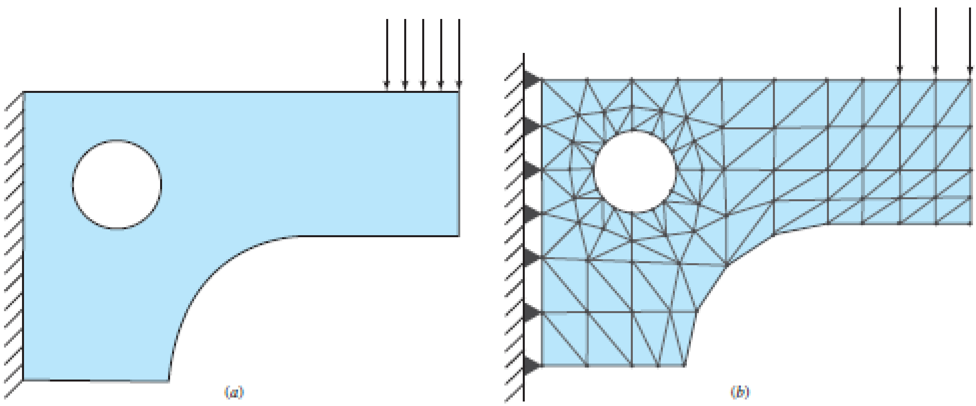


Figure 16 Structural problem. (a) Idealized model; (b) finite-element model.[2]

For an example of discretization errors, consider the constant thickness, thin plate structure shown in Figure 16 shows a finite-element model of the structure where three-node, plane stress, simplex triangular elements are employed.

This element type has a flaw that creates two basic problems. The element has straight sides that remain straight after deformation. The strains throughout the plane stress triangular element are constant. The first problem, a geometric one, is the modeling of curved edges. Note that the surface of the model with a large curvature appears poorly modeled, whereas the surface of the hole seems to be reasonably modeled.

The second problem, which is much more severe, is that the strains in various regions of the actual structure are changing rapidly, and the constant strain element will provide only an approximation of the average strain at the center of the element. So, in a nutshell, the results predicted by this model will be extremely poor. The results can be improved by significantly increasing the number of elements (increased mesh density).

Alternatively, the use of a more suitable element, such as an eight-node quadrilateral, will provide improved results. Because of higher order interpolation functions, the eight-node quadrilateral element can model curved edges and provides a higher-order function for the strain distribution.

In Figure 16-b, the triangular elements are shaded and the nodes of the elements are represented by the black dots. Forces and constraints can be placed only at the nodes. The nodes of the simplex triangular plane stress elements have only two degrees of freedom, corresponding to translation in the plane. Thus, the solid black, simple support triangles on the left edge represent the fixed support of the model. Also, the distributed load can be applied only to three nodes as shown. The modeled load has to be statically consistent with the actual load.

2.3 Mesh Generation

The network of elements and nodes that discretize a region is referred to as a mesh. The mesh density increases as more elements are placed within a given region. Mesh refinement is achieved when the mesh is modified from one analysis of a model to the next analysis to yield improved results. Results generally improve when the mesh density is increased in areas of high stress gradients and/or when geometric transition zones are meshed smoothly. Generally, but not always, the FEA results converge toward the exact results as the mesh is continuously refined. To assess improvement, in regions where high stress gradients appear, the structure can be remeshed with a higher mesh density at this location. If there is a minimal change in the maximum stress value, it is reasonable to presume that the solution has converged. There are three basic ways to generate an element mesh—manually, semiautomatically, or fully automatically.

1. **Manual mesh generation**. This procedure was applied in the early days of the finite-element method and it is a very labor intensive method of creating the mesh.
2. **Semiautomatic mesh generation.** Over the years, computer algorithms have been developed that enable the modeler to automatically mesh regions of the structure, using well-defined boundaries. Since the modeler has to define these regions, the technique is deemed semiautomatic.
3. **Fully automatic mesh generation.** Many software vendors have concentrated their efforts on developing fully automatic mesh generation, and in some instances, with automatic self-adaptive mesh refinement. The obvious goal is to significantly reduce the modeler's preprocessing time and effort to arrive at a final well-constructed FEA mesh (Figure 17).

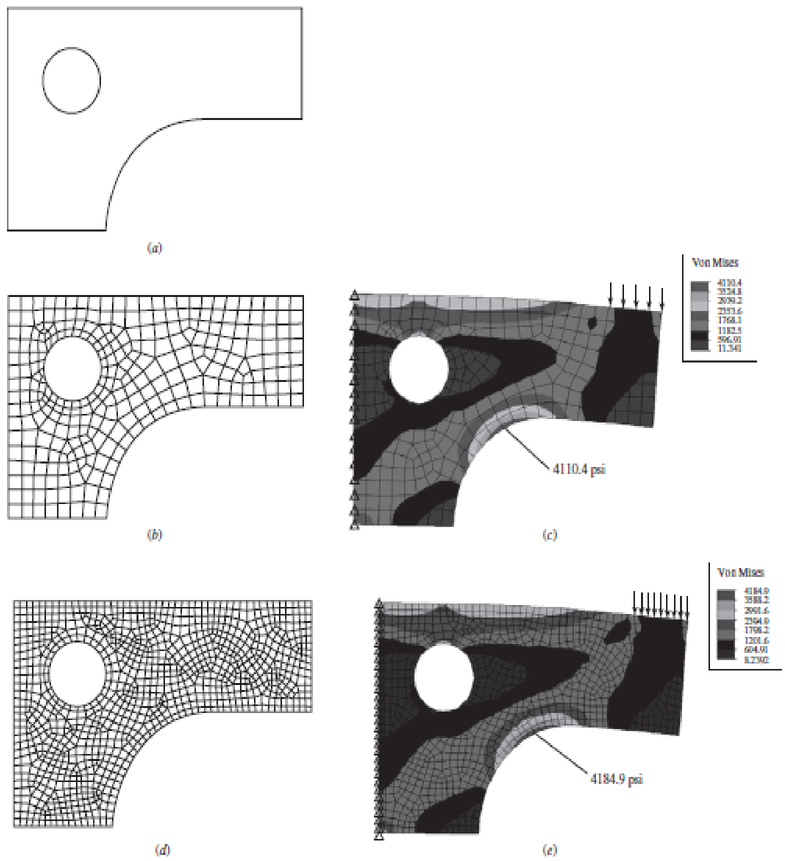


Figure 17 Automatic meshing the thin-plate model of Figure. (a) Model boundaries; (b) automesh with 294 elements and 344 nodes; (c) deflected (exaggerated scale) with stress contours; (d) automesh with 1008 elements and 1096 nodes, (e) deflected (exaggerated scale) with stress contours.[2]

2.4 Load Application

There are two basic forms of specifying loads on a structure—nodal and element loading. However, element loads are eventually applied to the nodes by using equivalent nodal loads. One aspect of load application is related to Saint-Venant's principle. If one is not concerned about the stresses near points of load application, it is not necessary to attempt to distribute the loading very precisely. The net force and/or moment can be applied to a single node, provided the element supports the dof associated with the force and/or moment at the node. However, the analyst should not be surprised, or concerned, when reviewing the results and the stresses in the vicinity of the load application point are found to be very large. Concentrated moments can be applied to the nodes of beam and most plate elements. However, concentrated moments cannot be applied to truss, two-dimensional plane elastic, axisymmetric, or brick elements.

They do not support rotational degrees of freedom. A pure moment can be applied to these elements only by using forces in the form of a couple. From the mechanics of statics, a couple can be generated by using two or more forces acting in a plane where the net force from the forces is zero. The net moment from the forces is a vector perpendicular to the plane of the forces and is the summation of the moments from the forces taken about any common point.

Element loads include static loads due to gravity (weight), thermal effects, surface loads such as uniform and hydrostatic pressure, and dynamic loads due to constant acceleration and steady-state rotation (centrifugal acceleration). As stated earlier, element loads are converted by the software to equivalent nodal loads and in the end are treated as concentrated loads applied to nodes. For gravity loading, the gravity constant in appropriate units and the direction of gravity must be supplied by the modeler. The gravity direction is normally toward the center of the earth. Surface loading can generally be applied to most elements. For example, uniform or linear transverse line loads (force/length) can be specified on beams. Uniform and linear pressure can normally be applied on the edges of two-dimensional plane and axisymmetric elements. Lateral pressure can be applied on plate elements, and pressure can be applied on the surface of solid brick elements. Each software package has its unique manner in which to specify these surface loads, usually in a combination of text and graphic modes.

2.5 Boundary Conditions

The simulation of boundary conditions and other forms of constraint is probably the single most difficult part of the accurate modeling of a structure for a finite element analysis. In specifying constraints, it is relatively easy to make mistakes of omission or misrepresentation. It may be necessary for the analyst to test different approaches to model esoteric constraints such as bolted joints, welds, etc., which are not as simple as the idealized pinned or fixed joints. Testing should be confined to simple problems and not to a large, complex structure. Sometimes, when the exact nature of a boundary condition is uncertain, only limits of behavior may be possible. For example, we have modeled shafts with bearings as being simply supported. It is more likely that the support is something between simply supported and fixed, and we could analyze both constraints to establish the limits. However, by assuming simply supported, the results of the solution are conservative for stress and deflections. That is, the solution would predict stresses and deflections larger than the actual.

Multipoint constraint equations are quite often used to model boundary conditions or rigid connections between elastic members. When used in the latter form, the equations are acting as elements and are thus referred to as rigid elements. Rigid elements can rotate or translate only rigidly. Boundary elements are used to force specific nonzero displacements on a structure. Boundary elements can also be useful in modeling boundary conditions that are askew from the global coordinate system.

Chapter 3. Numerical Calculation and Comparison with Theoretical Results

3.1 Diploma Thesis Subject

This study focusses on flat bottom grooves in shafts under tension. Finite Elements methods (ABAQUS) are applied to determine the corresponding stress concentration factors (SCF, K_t) considering various parameters and in the sequence the numerical results are compared with theoretical results available in literature.

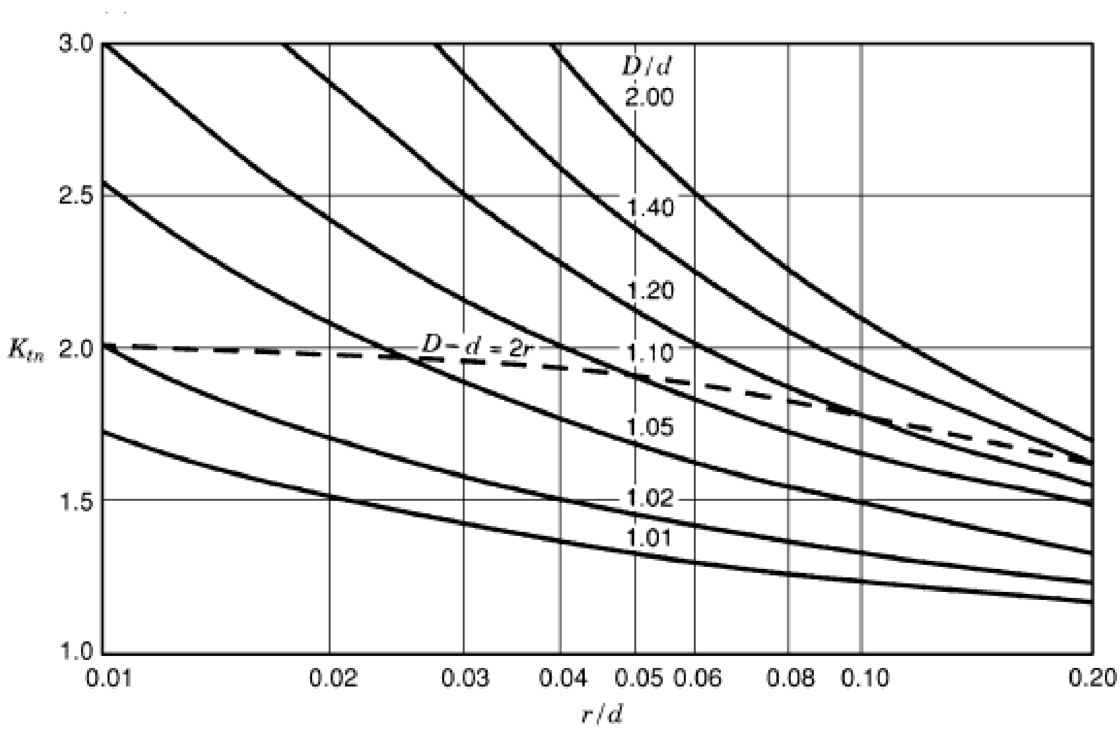


Figure 18 Stress concentration factors for flat bottom grooves in tension ($\frac{a}{d}=1$)[1]

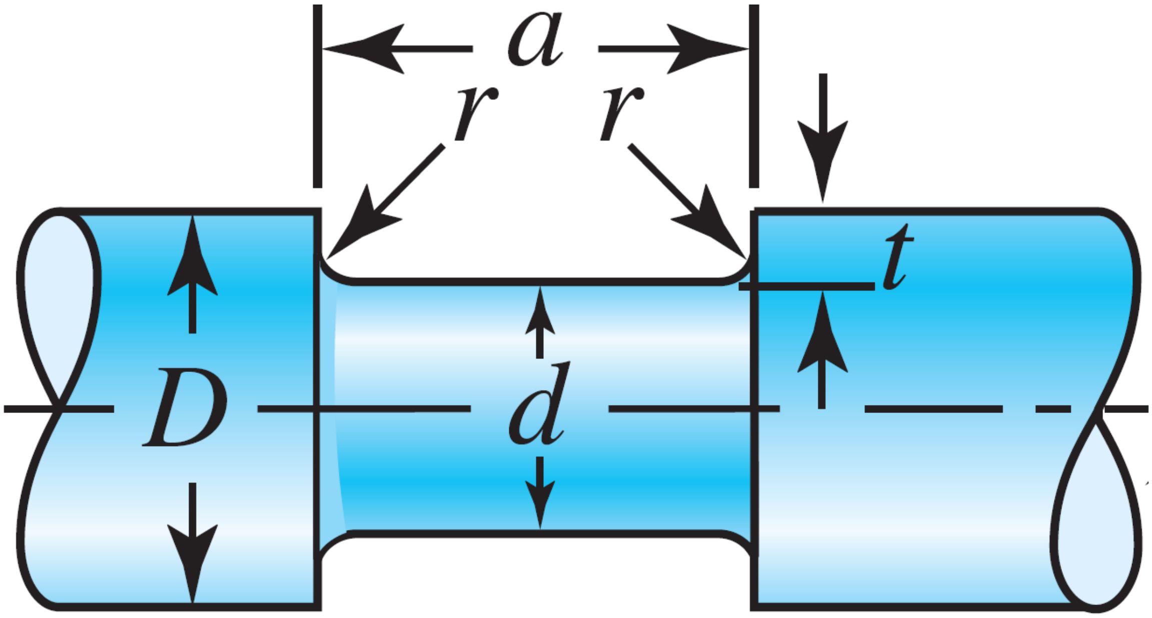


Figure 19 Shaft with Flat Bottom Grooves

3.2 Simulation with Computer Model and Mesh Generation

The shaft that is needed for the calculation has a length of 400mm and an outside diameter of 40mm (Figure 20). Calculations are made for 4 different geometries. The following table shows the geometric features of the shafts :

Specimen	D(mm)	d(mm)	r (mm)	D/d	r/d
3075	40	34,7826	2,6087	1,15	0,075
3015	40	30,7692	4,6154	1,3	0,15
3020	40	28,5714	5,7143	1,4	0,20
3010	40	33,3333	3,3333	1,2	0,10

Results are going to be compared with Neuber theory for notches from literature.

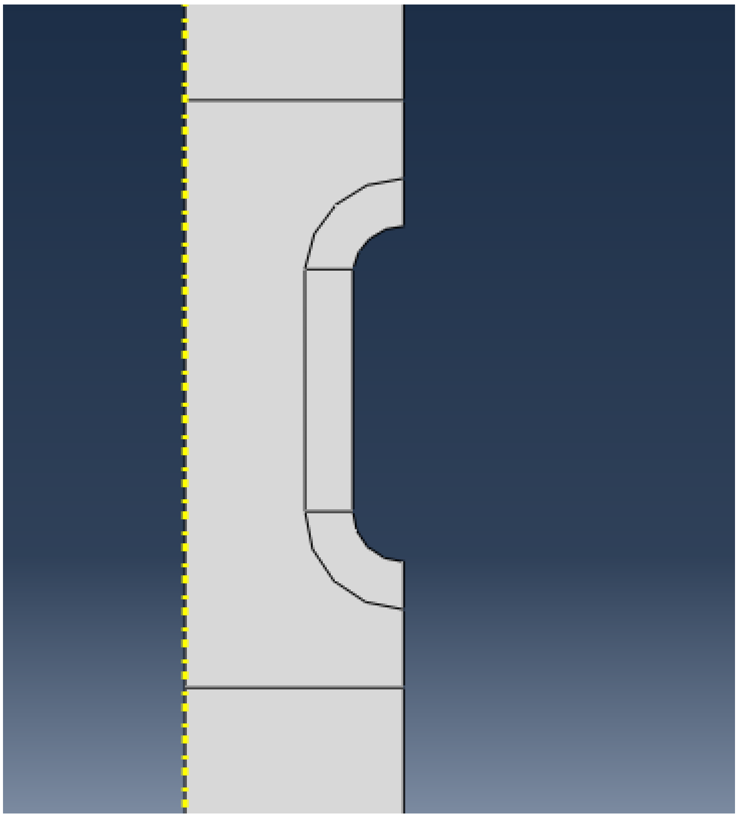


Figure 20 Specimen 3015 in ABAQUS environment.

The picture above shows one of the shafts being studied (3015). The program we use to solve the problem with the finite element method is ABAQUS.

The whole problem is axisymmetric so, we only need to discretize only a small part of the geometry. The shaft is subject to tension.

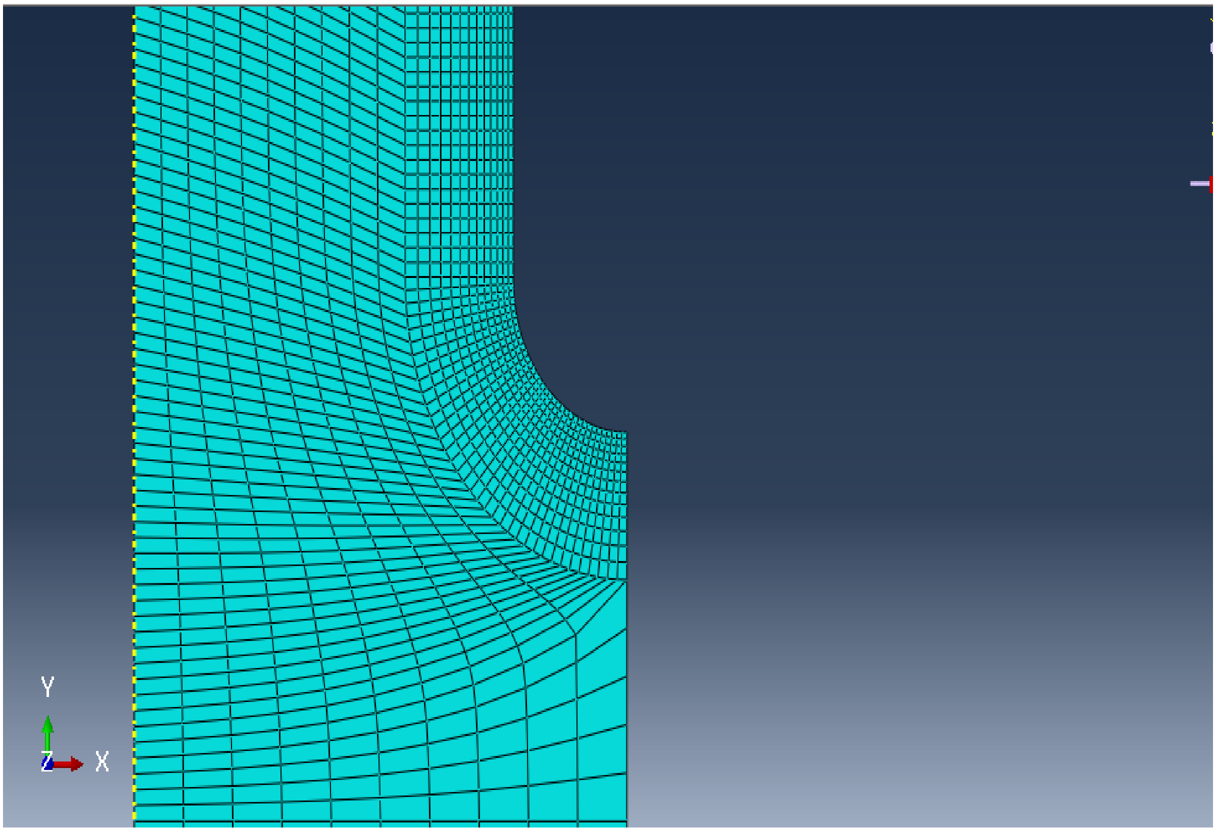


Figure 21 A mesh generation of 5383 elements.

The picture above shows the area we discretized as well as the areas that the way of discretization has changed. As we move closer to the notch we see a mesh generation that is more dense.

If the density of the grid were the same throughout the length of the geometry, it would have the effect of increasing the number of degrees of freedom and consequently the time of its resolution by the computer.

This is due to the fact that the focus is on greater accuracy of the stress value at the depth of the notch, where the stress concentration phenomenon occurs, and not in more remote areas where the phenomenon is less intense.

In the finite element method, the larger the number of elements, the more accurate the results. However, special care is required so that this increase occurs in the area that interests us and that is predicted to present a particular phenomenon, otherwise the flexibility and speed of the method is circumvented.

3.3 Boundary Conditions

The first boundary condition is applied at the axis of symmetry in the r direction:

$$U_1 = 0$$

The second boundary condition is applied on the bottom of the specimen which is anchored:

$$U_2 = 0$$

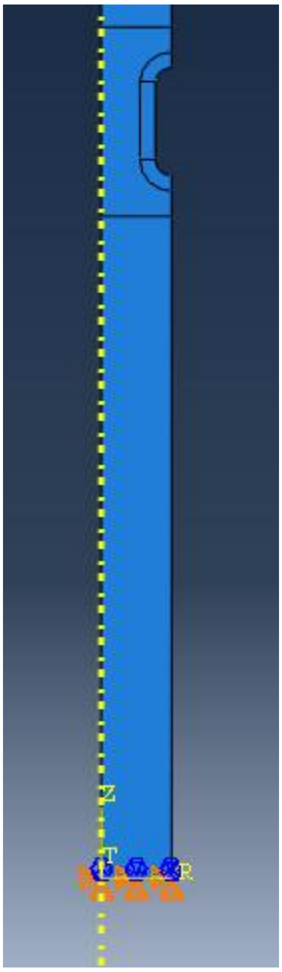


Figure 22 Boundary conditions in ABAQUS envirnoment.

3.4 Load Application

The picture below shows a load of 100 KN applied on the surface in order for the shaft to be subjected in tension.

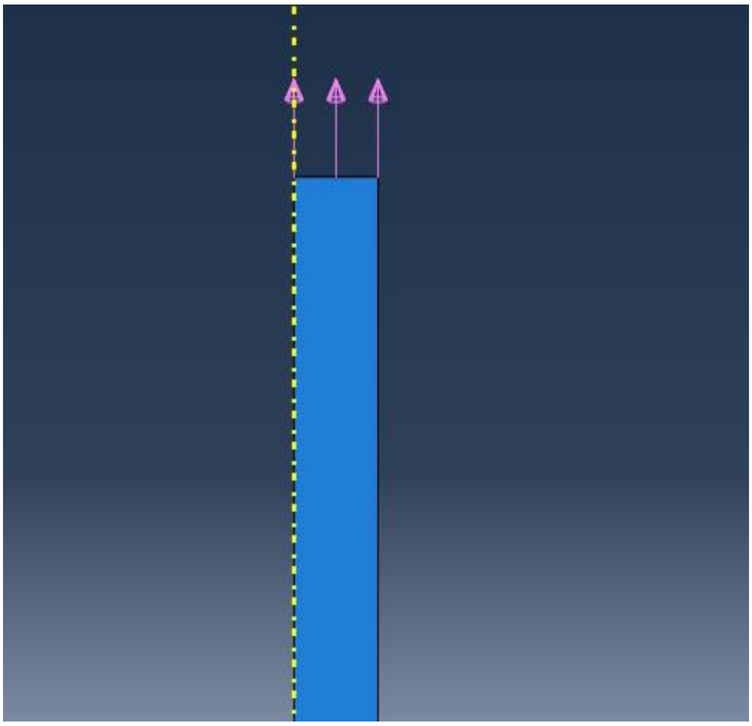


Figure 23 Load application in abaqus environment.

3.5 Solution to the Problem

The codes that have been used are listed in the Appendix (Input file). It should be mentioned that a different grid was used for each specimen. The grids used for each specimen are therefore different but have the same total number of elements and in the area near the notch have about the same density. The final computer model, three-dimensional grid, is obtained by rotating the two-dimensional around the x-axis. It should be mentioned the two-dimensional grid is type CAX4R.

After the design has been done and the appropriate boundary conditions have been applied, the loading is set. The material of the specimen is steel. But it does not need to be specified precisely because the problem is solved dimensionlessly. All that needs to be stated is that the tensile modulus E is the same for all steels. σ_{nom} has resulted from the following equation:

$$\sigma_{nom} = \frac{4P}{\pi D^2} = \frac{4P}{\pi D^2}\frac{d^2}{d^2} = \sigma \frac{d^2}{D^2}$$

Specimen	σ_{nom}	σ_{22}	$K = \frac{\sigma_{22}}{\sigma_{nom}}$	K_{Neuber}	Deviation %
3075	132,25	228	1.724	1,8	4,22
3015	169	293	1.735	1,75	0,86
3020	196	336	1.714	1,68	2,02
3010	144	243.36	1.69	1,75	3,42

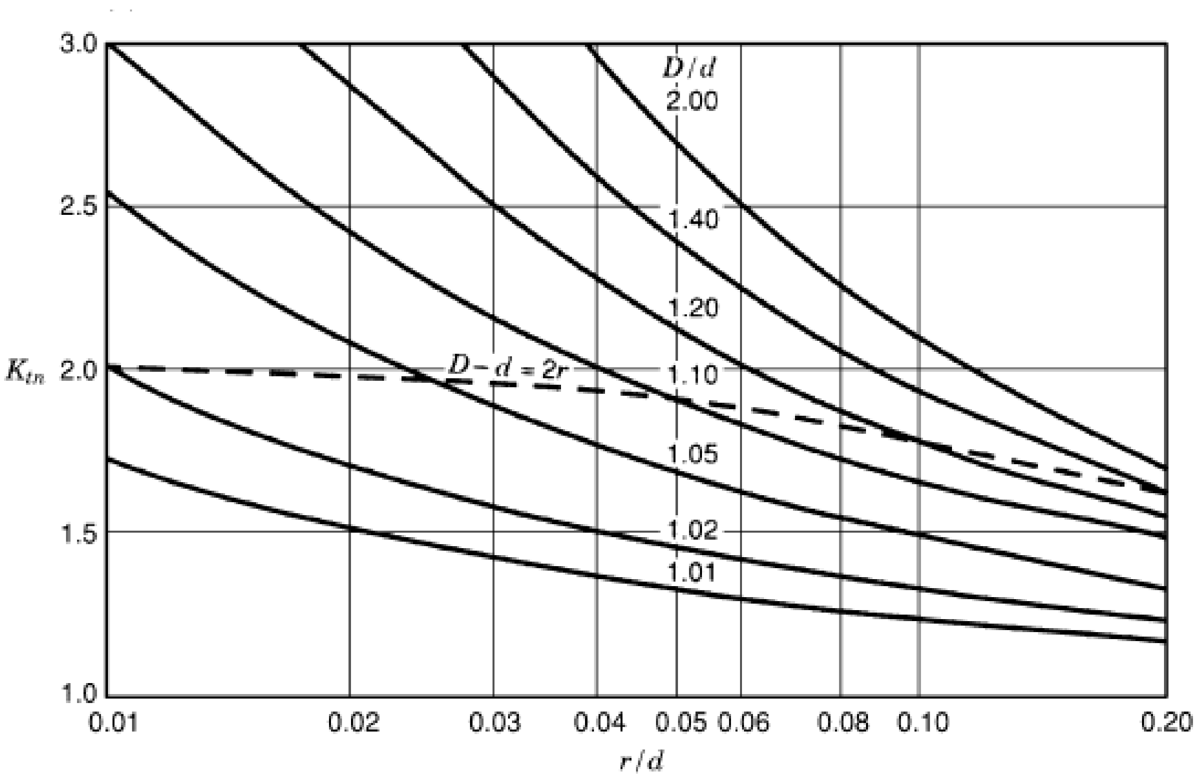


Figure 24 Stress concentration factors for flat bottom grooves in tension ($\frac{a}{d}=1$)[1]

The following figures shows the stress distribution for the specimen, as calculated after processing data from the ABAQUS software package. It is obvious that the maximum stress values are located at the depth of the notch.

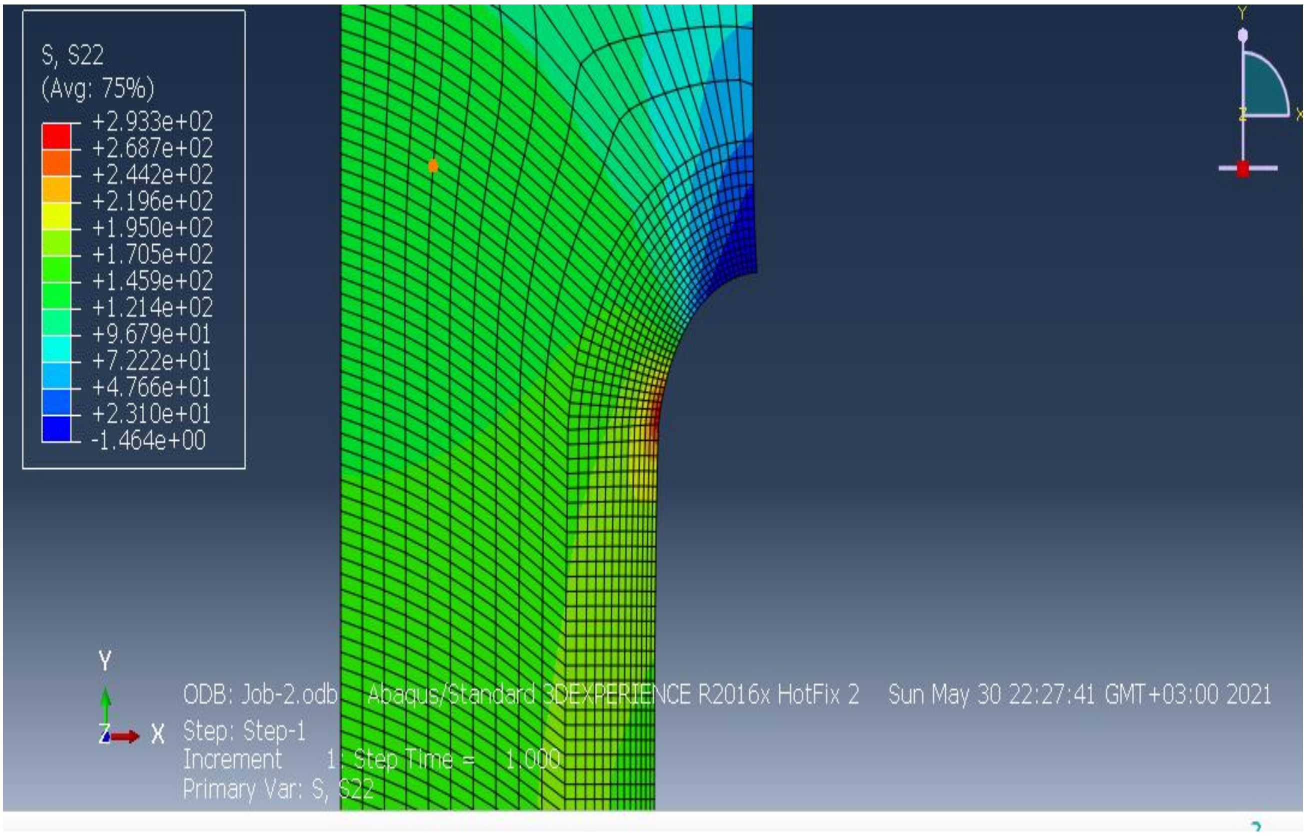


Figure 25 (Stress distribution across the 3015 specimen (2D))

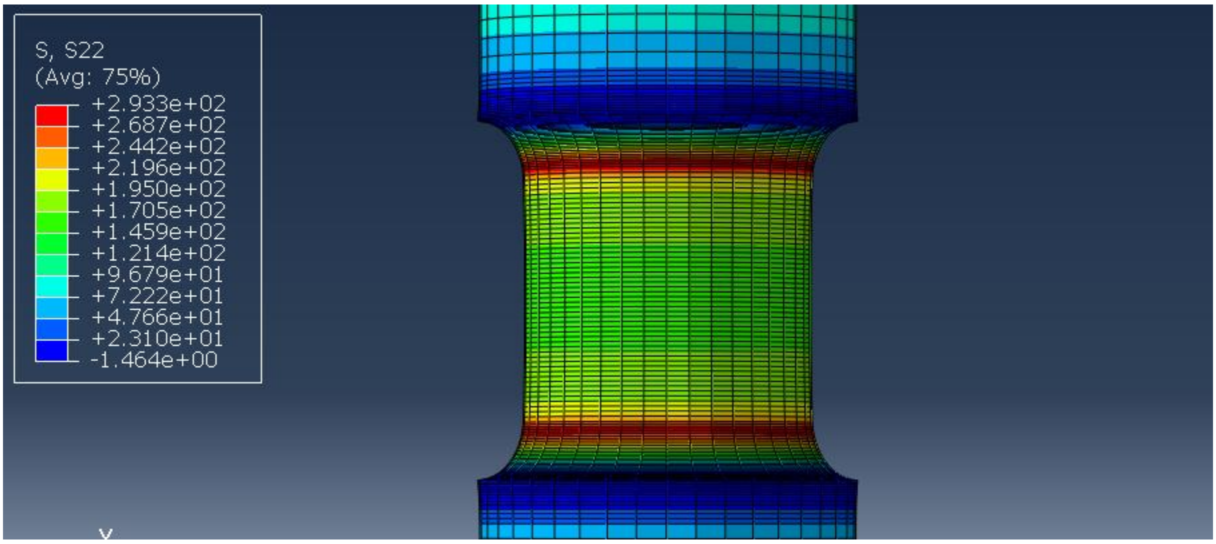


Figure 26 (Stress distribution across the 3015 specimen (3D))

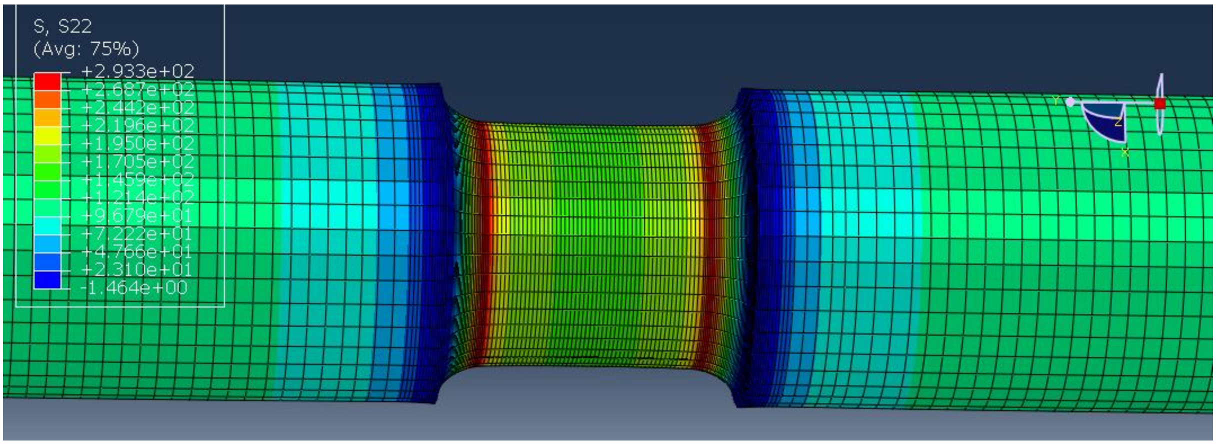


Figure 27 (Stress distribution across the 3015 specimen (3D))

3.6 Conclusions

The results are not the same in both cases, but they are very close to each other. We conclude that Neuber's estimates are very close to the actual value and the assumptions he has made in his theory have a negligible effect on the theoretical determination of the concentration factor.

In the next figure there is the graphical comparison between the stress concentration factor that is calculated by the theoretical graph of Neuber and the results of the finite elements method.

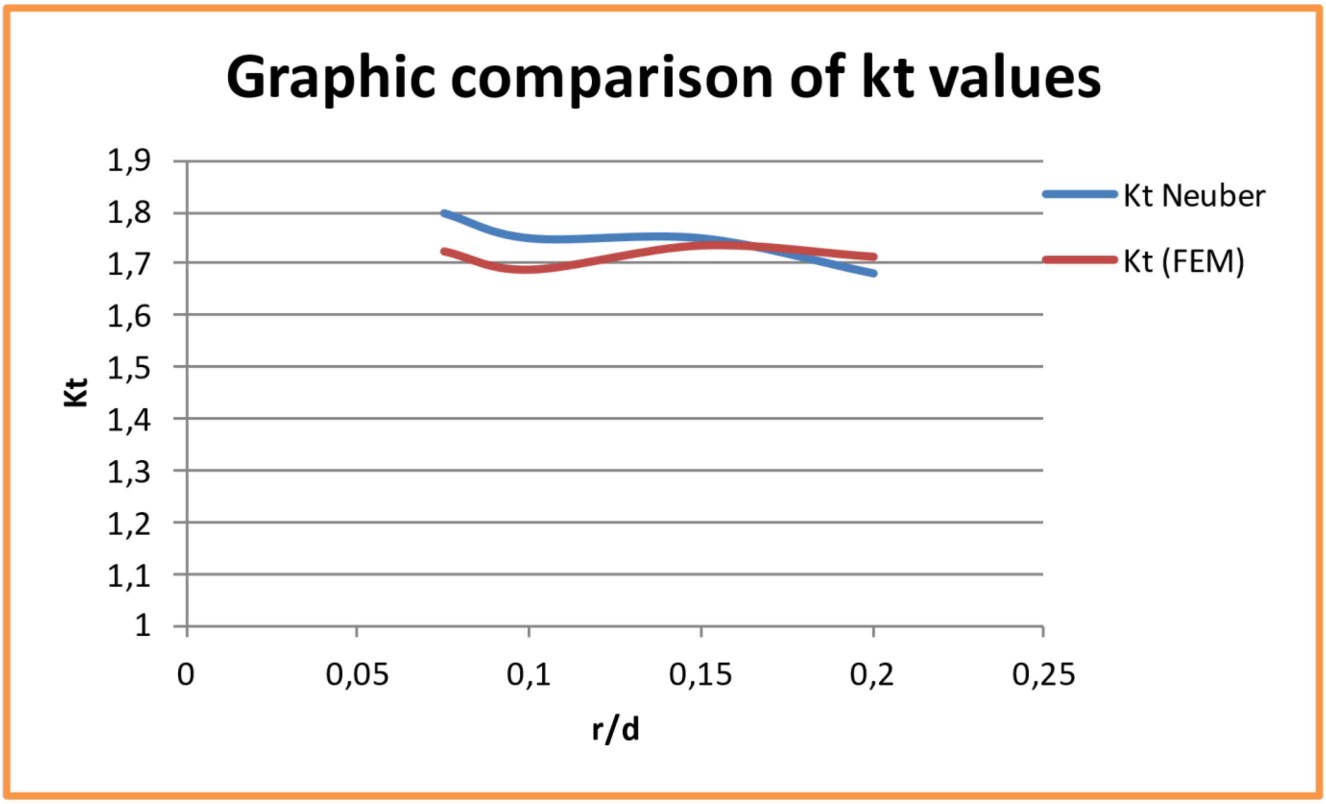


Figure 28

Stress concentration factors are calculated analytically by elastic theory, computationally by the finite element method and experimentally using methods such as photosensitivity. When the experimental procedure is performed with satisfactory accuracy the results coincide almost exactly with those of the stress concentration factors obtained analytically.

Unfortunately in practice, however, the use of stress concentration factors in analysis and design is not well established, as is the theoretical basis for approximating the factors.

This is because the solution based on the theory of elasticity, is based on assumptions that the material is homogeneous and isotropic that we rarely meet in practice.

More data are needed on materials, because due to lack of data, we often resort to statistical methods to have the required accuracy. Phenomena of conductivity within the material must also be taken into account.

The designer of course cannot have exact answers to all these questions. As always, existing information should be reviewed, and critical thinking should be used to develop reasonable approximation methods for design, which will lean in the safe direction in doubtful cases.

In the future it seems that there will be progress and a review of the use of stress concentration factors will be required. On the other hand we can say that our limited experience in using these methods is satisfactory.

REFERENCES

[1] WALTER D. PILKEY (1997) PETERSON'S STRESS CONCENTRATION FACTORS

[2] RICHARD G. BUDYNAS, J. KEITH NISBETT (2015) SHIGLEY'S MECHANICAL ENGINEERING DESIGN

[3] WALTER D. PILKEY (2005) FORMULAS FOR STRESS, STRAIN, AND STRUCTURAL MATRICES

Appendix

```
*Heading
** Job name: Job-2 Model name: Model-1
** Generated by: Abaqus/CAE 2016.HF2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=Part-1
*Node
      1,          20.,   235.153915
      2,   10.9765148,   226.818039
      3,   15.3845997,   226.818039
      4,          20.,   230.769196
      5,          11.,   204.615402
      6,   15.3845997,   204.615402
      7,          20.,   195.615402
      8,          20.,         200.
      9,          20.,    188.43428
     10,           0.,    188.43428
     11,           0.,           0.
     12,          20.,           0.
     13,           0.,    242.33429
     14,          20.,    242.33429
     15,          20.,   430.769196
     16,           0.,   430.769196
     17,   19.5903835,    235.14679
     18,   19.1815147,   235.121002
     19,   18.7742443,    235.07666
     20,   18.3694134,   235.013809
     21,   17.9678612,   234.932617
     22,   17.5704193,   234.833221
     23,   17.1779137,   234.715851
     24,   16.7911568,    234.58075
     25,   16.4109478,   234.428162
     26,   16.0380764,   234.258453
     27,    15.673316,   234.071945
     28,     15.31742,   233.869019
     29,   14.9711285,   233.650116
     30,   14.6351566,    233.41568
     31,   14.3102016,   233.166199
     32,   13.9969368,   232.902191
     33,   13.6960106,   232.624191
     34,   13.4080486,   232.332794
     35,   13.1336451,   232.028595
     36,   12.8733702,   231.712204
```

```
    37,   12.6277618,   231.384323
    38,   12.3973303,   231.045593
    39,   12.1825523,   230.696716
    40,   11.9838734,    230.33844
    41,   11.8017044,   229.971497
    42,   11.6364241,   229.596634
    43,   11.4883738,   229.214645
    44,   11.3578596,   228.826309
    45,   11.2451544,   228.432434
    46,   11.1504898,   228.033844
    47,   11.0740623,   227.631363
    48,   11.0160313,   227.225815
    49,    11.501977,   226.818039
    50,   11.9867983,   226.818039
….
5696,    11.166503,   235.956879
  5697,   12.8579187,   234.771072
  5698,   11.3196659,   236.511658
  5699,   13.0814524,   235.196472
  5700,   11.4736567,   237.156357
  5701,   13.3107138,   235.651779
  5702,   14.0719109,   236.347122
  5703,   14.6266155,   236.531845
  5704,   15.2063341,   236.673599
  5705,   15.7932892,   236.789871
  5706,    16.384161,   236.887131
  5707,   16.9786797,   236.967712
  5708,   17.5768032,   237.032059
  5709,   18.1781445,    237.07933
  5710,   18.7821121,   237.106995
  5711,   19.3882103,   237.109268
  5712,   12.3527718,    238.20285
  5713,   13.0865822,   238.401627
  5714,     13.84231,   238.528702
  5715,    14.604105,   238.619873
  5716,   15.3686838,   238.690155
  5717,   16.1355476,   238.745514
  5718,   16.9046249,   238.787781
  5719,   17.6757545,   238.816208
  5720,       18.4487,   238.827179
  5721,   19.2233429,   238.811325
*Element, type=CAX4R
 1,   1,  17,  844,  104
 2,  17,  18,  845,  844
 3,  18,  19,  846,  845
 4,  19,  20,  847,  846
 5,  20,  21,  848,  847
```

6, 21, 22, 849, 848
7, 22, 23, 850, 849
8, 23, 24, 851, 850
9, 24, 25, 852, 851
10, 25, 26, 853, 852
11, 26, 27, 854, 853
12, 27, 28, 855, 854
13, 28, 29, 856, 855
14, 29, 30, 857, 856
15, 30, 31, 858, 857
16, 31, 32, 859, 858
17, 32, 33, 860, 859
18, 33, 34, 861, 860
19, 34, 35, 862, 861
20, 35, 36, 863, 862
21, 36, 37, 864, 863
22, 37, 38, 865, 864
23, 38, 39, 866, 865
24, 39, 40, 867, 866
25, 40, 41, 868, 867
26, 41, 42, 869, 868
27, 42, 43, 870, 869
28, 43, 44, 871, 870
29, 44, 45, 872, 871
30, 45, 46, 873, 872
31, 46, 47, 874, 873
32, 47, 48, 875, 874
33, 48, 2, 49, 875
34, 104, 844, 876, 103
35, 844, 845, 877, 876
36, 845, 846, 878, 877
37, 846, 847, 879, 878
38, 847, 848, 880, 879
39, 848, 849, 881, 880
40, 849, 850, 882, 881
41, 850, 851, 883, 882
42, 851, 852, 884, 883
43, 852, 853, 885, 884
44, 853, 854, 886, 885
45, 854, 855, 887, 886
46, 855, 856, 888, 887
47, 856, 857, 889, 888
48, 857, 858, 890, 889
49, 858, 859, 891, 890
50, 859, 860, 892, 891
51, 860, 861, 893, 892
52, 861, 862, 894, 893

```
 53,  862,  863,  895,  894
 54,  863,  864,  896,  895
 55,  864,  865,  897,  896
 56,  865,  866,  898,  897
…
5360,   18,   17, 5711, 5710
5361,   17,    1,  723, 5711
5362, 4796, 5702, 5712, 4797
5363, 5702, 5703, 5713, 5712
5364, 5703, 5704, 5714, 5713
5365, 5704, 5705, 5715, 5714
5366, 5705, 5706, 5716, 5715
5367, 5706, 5707, 5717, 5716
5368, 5707, 5708, 5718, 5717
5369, 5708, 5709, 5719, 5718
5370, 5709, 5710, 5720, 5719
5371, 5710, 5711, 5721, 5720
5372, 5711,  723,  724, 5721
5373, 4797, 5712, 4667, 4666
5374, 5712, 5713, 4668, 4667
5375, 5713, 5714, 4669, 4668
5376, 5714, 5715, 4670, 4669
5377, 5715, 5716, 4671, 4670
5378, 5716, 5717, 4672, 4671
5379, 5717, 5718, 4673, 4672
5380, 5718, 5719, 4674, 4673
5381, 5719, 5720, 4675, 4674
5382, 5720, 5721, 4676, 4675
5383, 5721,  724,  725, 4676
*Nset, nset=Set-1, generate
    1,  5721,     1
*Elset, elset=Set-1, generate
    1,  5383,     1
** Section: Section-1
*Solid Section, elset=Set-1, material=Material-1
,
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Part-1-1, part=Part-1
*End Instance
**
*Nset, nset=Set-1, instance=Part-1-1
```

```
 11,  12, 401, 402, 403, 404, 405, 406, 407, 408, 409
*Elset, elset=Set-1, instance=Part-1-1, generate
 2491,  2500,     1
*Elset, elset=_Surf-1_S3, internal, instance=Part-1-1, generate
 4175,  4192,     1
*Surface, type=ELEMENT, name=Surf-1
_Surf-1_S3, S3
*End Assembly
**
** MATERIALS
**
*Material, name=Material-1
*Elastic
207000., 0.3
**
** BOUNDARY CONDITIONS
**
** Name: BC-1 Type: Displacement/Rotation
*Boundary
Set-1, 1, 1
Set-1, 2, 2
Set-1, 6, 6
** ----------------------------------------------------------------
**
** STEP: Step-1
**
*Step, name=Step-1, nlgeom=NO
*Static
1., 1., 1e-05, 1.
**
** LOADS
**
** Name: Load-1   Type: Pressure
*Dsload
Surf-1, P, -100.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**
*Output, history, variable=PRESELECT
```

*End Step