Stress Concentrations and Static Failure for Common Elements used in Finite Element Stress Analysis

Joseph J. Rencis, Sachin Terdalkar

University of Arkansas

Abstract

Element types that are commonly used in practice for a static finite element stress analysis include the truss element, beam element, plane stress/strain elements, axisymmetric elements, and solid elements. This paper focuses on stress concentrations and static failure associated with these elements and serves as a reference for instructors and practitioners. Stress concentrations and static failure for these elements has not been comprehensively addressed in any finite element textbook. The first author has integrated stress concentrations and static failure into introductory finite element undergraduate courses, graduate courses and industrial short courses. The authors have found through experience that students and practitioners are not familiar with the stress concentrations and static failure associated with these elements.

1. Introduction

The finite element method (FEM) has been used extensively during the past thirty years in industry and is now a standard engineering tool for both analysis and design. Years of experience with the method have shown that by understanding the fundamentals of the technique, real complex systems can be modeled with a high degree of reliability. It is important to emphasize, however, that the reliability of the process is highly dependent on the skill of the engineer in the application of the method. Modern finite element developments have become very sophisticated, and the available software developed for the user has become very easy to use. It has become more important than ever to insure that the analyst, in his/her search for the best modeling method, correctly uses the tools available.

Stress concentrations arise at locations where there are abrupt changes in geometry (e.g., holes, notches, fillets, grooves, etc.). Mechanics of materials and elasticity textbooks comprehensively address stress concentrations. Stress concentrations are not addressed in finite element textbooks and courses for the truss element and beam element. However, finite element textbooks and courses consider stress concentrations for plane stress/strain, axisymmetric, and solid elements. The truss and beam elements do not include stress concentrations, whereas, plane stress/strain elements, axisymmetric elements, and solid elements include stress concentrations.

An understanding of the failure modes associated with each component in a structure is required to ensure that it will not fail under loading. Static failure modes for different components can be readily found in mechanics of materials and machine design textbooks. The most common static failure criteria found in finite element textbooks is the von Mises applied to plane stress/strain elements, axisymmetric elements, and solid elements. A review of finite element textbooks reveals that there is no discussion on why the von Mises criterion was selected and other failure criterion are not considered. The authors are not aware of any finite element textbook that address the static failure modes associated with beam element and there is very limited discussion for the truss element.

The goal of this paper is to define the stress concentrations and static failure for finite elements commonly used to carry out a static, linear elastic, stress analysis. The elements considered include the truss element, beam element, plane stress/strain elements, axisymmetric elements, and solid elements as shown in Figure 1. Each element will be considered separately in this paper and the following will be discussed: overview that includes a literature review, stress concentrations, and static failure. This paper will serve as a reference for instructors and practitioners.

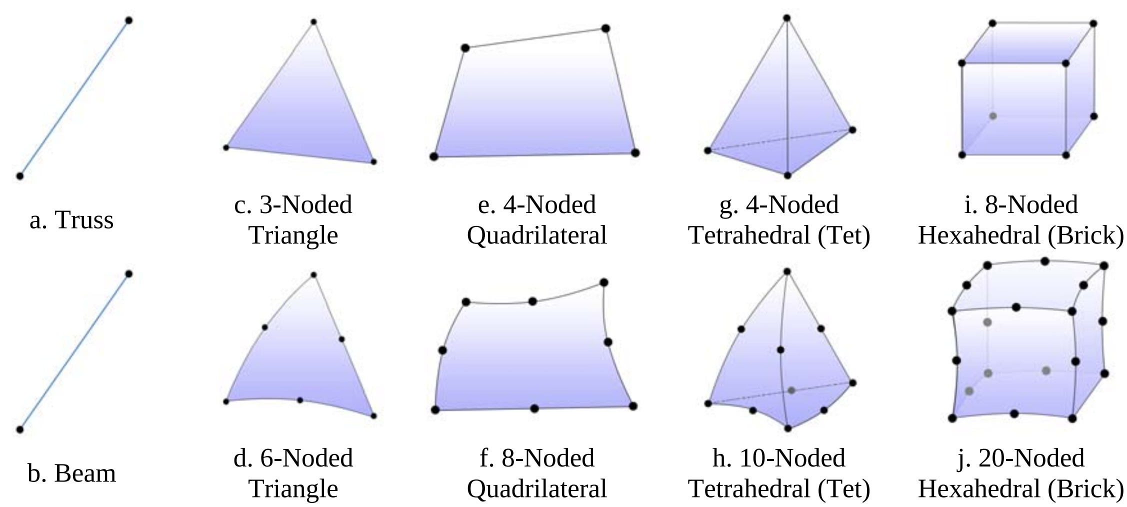


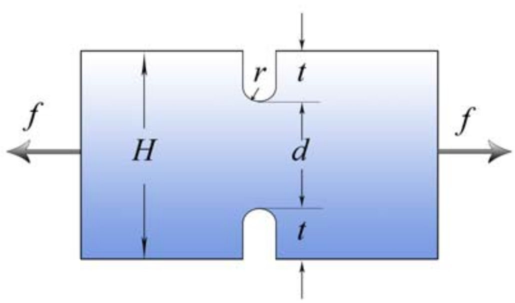
Figure 1. Common elements used to carry out a static, liner elastic, stress analysis.

2. Truss Element

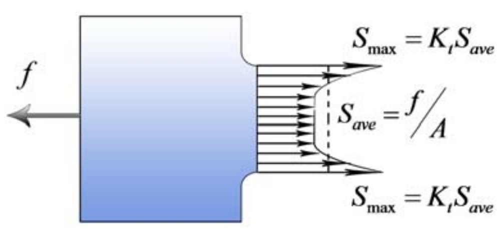
The truss element is one of the simplest elements used in practice for a finite element stress analysis. The theoretical development of the truss element is commonly found in finite element textbooks, undergraduate finite element courses, graduate finite element courses, commercial code short courses, and industrial short courses. Commercial finite element codes usually contain both the two-dimensional (plane) and three-dimensional (space) two-noded truss elements. The one-dimensional truss element cannot be found in commercial software and is only used as an academic exercise to introduce its theoretical development. The authors are not aware of any finite element textbook that addresses stress concentrations for the truss element. Static failure for the truss element is considered with an example in [1] and homework problems in [2, 3], however, there is no discussion regarding static failure. Adams and Askenazi [4] provide a discussion regarding static failure, however, they do not relate it to the truss element. The truss element considered in this paper is straight, uniform, linear elastic, homogeneous, isotropic and has two end nodes as shown in Figure 1a. Stress concentrations and static failure analysis addressed in this paper are carried out by hand or a spreadsheet using the results obtained from a commercial finite element code. The results from a commercial finite element code can be found in an output text file.

2.1. Stress Concentrations

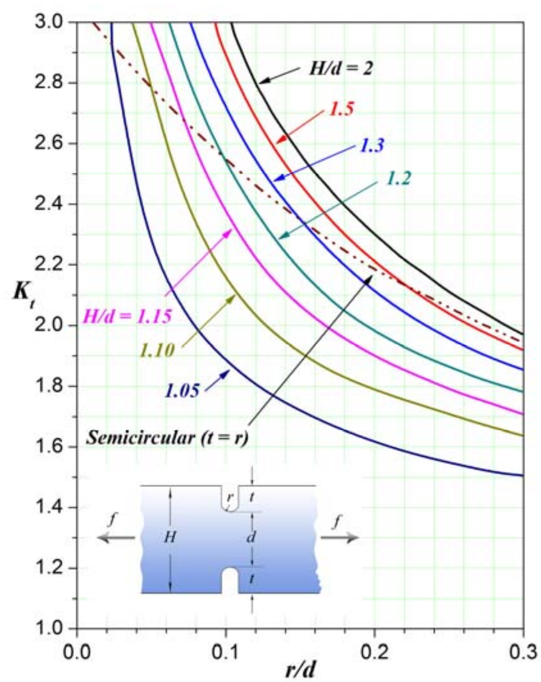
The truss element does not account for stress concentrations since it assumes a uniform stress distribution in the elements' cross-section. An abrupt change in the geometry, e.g., holes, notches, fillets, grooves, etc, results in a non-uniform stress pattern. Using the 'theoretical stress concentration factor (unitless)' for static loading the maximum stress can be determined at the location where there is a stress-riser. Stress concentration factors for common axial loading cases can be found in undergraduate mechanics of materials textbooks and a comprehensive list can be found in handbooks [5-7]. When the stress concentration factor cannot be found in a handbook, then electrical strain gauges, photoelasticity, or complex finite element models are used. Complex finite element models include the plane stress/strain, axisymmetric, and solid elements discussed in Section 4. Figure 2 shows theoretical stress concentration factor K_t for a flat bar with u-shaped notches subjected to axial loading.



a. Flat plate with u-shaped notches.



b. Stress distribution and maximum stress location.



c. Theoretical stress concentration factor K_t.

Figure 2. Theoretical stress concentration factor for a flat plate with u-shaped notches subjected to axial loading.

The maximum stress S_{max} in Figure 2a at a local stress-riser in the truss element is defined as follows:

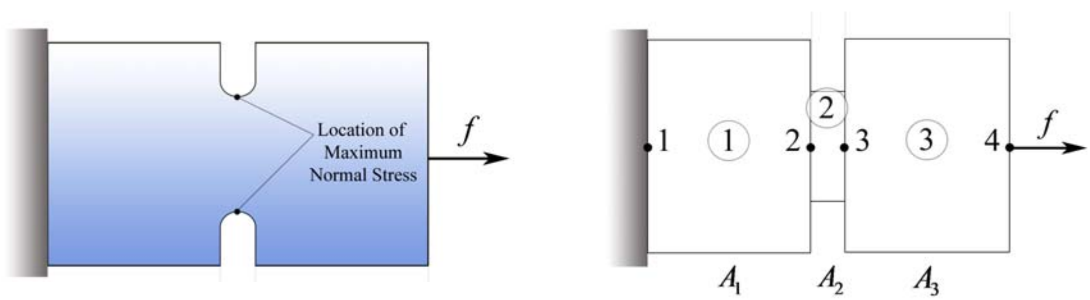
$$S_{\max} = K_t S_{ave} \tag{1}$$

where S_{ave} is the uniform normal stress in the truss element at the location of the stress-riser and K_t is the theoretical stress concentration factor (based on theoretical elastic, homogenous, isotropic material).

The application of the theoretical stress concentration factor K_t for static loading depends on the material type as follows:

- *Ductile Material* ($\varepsilon_f > 0.05$). A material is defined as ductile if the percentage of elongation to fracture ε_f based on the 2 inch gauge length is greater than 5%. The effect of K_t is ignored $(K_t = 1)$ since the material will yield locally at the stress-riser while the material farther away from the stress-riser remains below the yield strength.
- *Brittle Material* ($\varepsilon_f \le 0.05$). A material is defined as brittle if the percentage of elongation to fracture ε_f based on the 2 inch gauge length is less than or equal to 5%. The theoretical stress concentration factor K_t is used for a brittle material except for cast materials $(K_t = 1)$ since it has known defects throughout the interior and strength data includes stress concentrations.

Consider a plate with u-shaped notches subjected to an end tensile axial force as shown in Figure 3a. The corresponding finite element mesh with three truss elements is shown in Figure 3b. The maximum stress at the location of the stress-riser (notches) requires that K_t be determined from Figure 2c. The finite element uniform stress for element ② is $(S_{ave})_2$ and the maximum stress for element ② is determined using $(S_{\max})_2 = K_t (S_{ave})_2$.



a. Problem definition. b. Finite element model.

Figure 3. Example to determine maximum stress in a flat plate with u-shaped notches using truss elements.

2.2. Static Failure

The static failure criteria are based on whether the truss element is in tension or compression. Assuming that a truss element never exceeds the yield strength (S_y) of the material, then static failure is defined by Norton [8] as follows:

- *Tension Failure.* Static failure for a truss element in tension is based on the material yield strength (S_y). The factor of safety (FS) is defined as

$$FS = \frac{S_y}{S_a} \tag{2}$$

where S_a is the allowable stress in the truss element. When the material is brittle, except for cast material and ductile materials, the allowable stress S_a includes the theoretical stress concentration factor K_t, i.e., $S_a = K_t S_{ave}$.

- *Compression Failure.* Static failure for a truss element in compression is due to buckling or a combination of crushing and buckling. Since a truss element has frictionless pin connections (nodes), then the element is pinned-pinned supported as shown in Figure 4a. The effective length of the column is $l_{eff} = l$, where l is the truss element length. The failure strength is designated as S_{buck} and is based on the slenderness ratio S_r of the truss element as follows:

$$S_r \le (S_r)_D \qquad (S_r)_D = \pi\sqrt{\frac{2E}{S_{yc}}} \tag{3a}$$

$$S_{buck} = \frac{P_{cr}}{A} = S_{yc} - \frac{1}{E}\left(\frac{S_{yc}S_r}{2\pi}\right)^2 \qquad \text{Johnson Formula} \tag{3b}$$

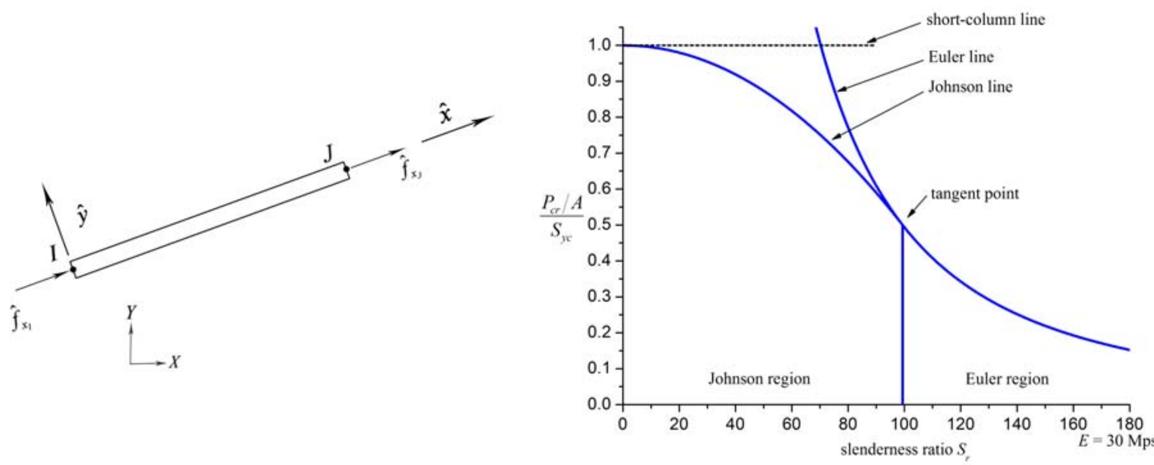
$$S_r > (S_r)_D \tag{4a}$$

$$S_{buck} = \frac{P_{cr}}{A} = \frac{\pi^2 E}{S_r^2} \qquad \text{Modified Euler Equation} \tag{4b}$$

with the factor of safety defined as

$$FS = \frac{S_{buck}}{S_a} \tag{5}$$

where E is Young's modulus, S_{yc} is the compressive yield strength, $S_r = l_{eff}/k$ is the slenderness ratio, $l_{eff} = l$ is the effective length of the truss element, $k = \sqrt{I/A}$ is the radius of gyration of the cross section, I is the smallest moment of inertia, A is the cross-sectional area, and S_a is the allowable stress in the truss element. Truss elements made of brittle material, except for brittle cast material and ductile materials, have an allowable stress S_a that includes the theoretical stress concentration factor K_t, i.e., $S_a = K_t S_{ave}$. Figure 4b shows the failure lines for the Johnson formula and the modified Euler equation in Equations (3b) and (4b), respectively.



a. Truss element. b. Column failure lines.

Figure 4. Truss element in compression and compression failure lines.

3. Beam Element

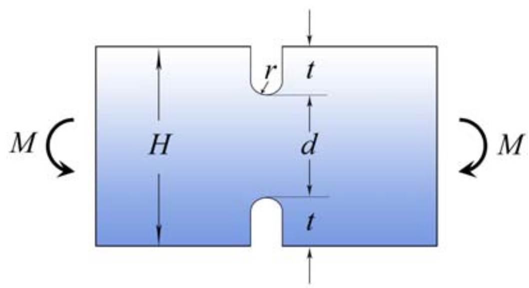
The beam element is one of the simplest elements used in practice for a finite element stress analysis. The theoretical development of the beam element is commonly found in finite element textbooks, undergraduate courses, graduate courses and short courses. The two-dimensional (plane) and three-dimensional (space) beam elements are commonly found in commercial finite element software. The one-dimensional beam element cannot be found in commercial software and is only used as an academic exercise to introduce its theoretical development. The authors are not aware of any finite element textbook that considers stress concentrations and static failure for the beam element. Potts and Oler [9] do state that "no stress concentration will be computed at the points of beam step change" and the reason is also discussed. However, there is no discussion on how to determine the maximum stress at the beam step change. Logan [3] does consider in a homework problem for static failure, however, there is no discussion regarding static failure for beam elements. Askenazi [4] provide a discussion regarding static failure, however, they do not relate it to the beam element. The beam element considered in this paper is straight, uniform, linear elastic, homogeneous, isotropic and has two end nodes as shown in Figure 1b. Both stress concentration and static failure for a beam element is carried out by hand or with a spreadsheet using the results from a commercial finite element code. The results from a commercial finite element code can be found in an output text file.

3.1. Stress Concentrations

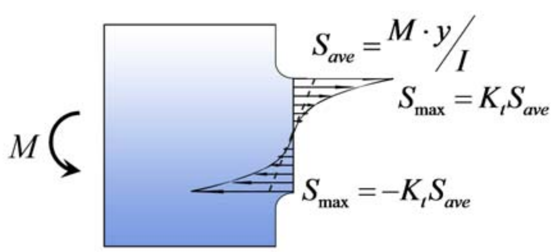
Stress concentrations are not present in the beam element. The theoretical stress concentration factor and the beam element average stress are used to determine the maximum stress at the location of the stress-riser. A three-dimensional beam element contains internal forces in terms of element (local) coordinates as follows: one axial force, two shear forces, two bending moments, and one torsional moment. Stress concentrations are based on a combination of six internal forces and moments. However, in practice for engineering (long) beams it is very common to consider that stress concentrations are only due to one axial normal stress and two normal bending stresses.

A study of commercial finite element codes reveals that only the normal stress components are used for the beam element.

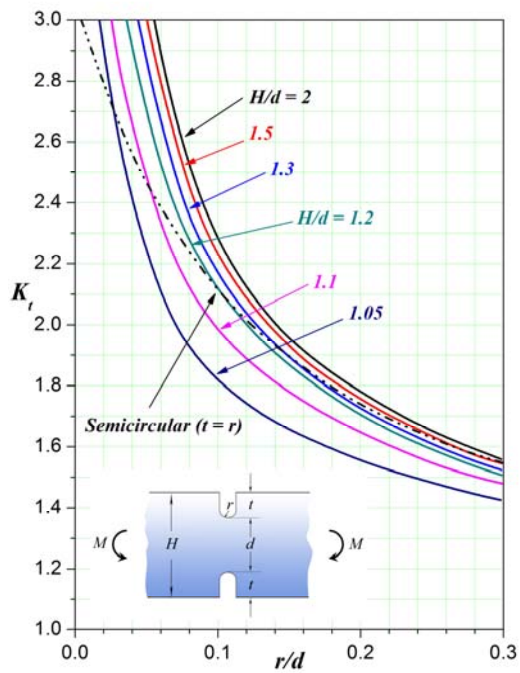
Stress concentration factors were discussed in Section 2.1 for axial loading. Stress concentration factors for pure bending of a beam can be found in undergraduate mechanics of materials textbooks for common cases and in handbooks for complex cases [5-7]. Figure 5 shows the theoretical stress concentration factor K_t for a flat bar with u-shaped notches subjected to pure bending.



a. Flat plate with u-shaped notches.



b. Stress distribution and maximum stress location.



c. Theoretical stress concentration factor K_t .

Figure 5. Theoretical stress concentration factor for a flat plate with u-shaped notches subjected to pure bending.

The application of the theoretical stress concentration factor K_t depends on whether the material is ductile or brittle as discussed in Section 2.1. The plane beam element is considered due to its simplicity and is shown in Figure 6. The normal stress due to axial and bending is shown in Figure 6 using the ANSYS® notation and sign convention [10]. Stress concentration factors are applied according to one of the following approaches:

1) *Theoretical Approach.*
   a) Determine the theoretical stress concentration factor K_{ta} based on the axial force.
   b) Determine the theoretical stress concentration factor K_{tb} based on the bending moment.
   c) Use the largest value of the maximum normal stress from the two values at the stress-riser:
      i) $S_{\max} = K_{ta} SDIR + K_{tb} SBYT$ (6a)

ii) $S_{\max} = K_{ta}SDIR + K_{tb}SBYB$ (6b)

where $SDIR$, $SBYT$, and $SBYB$ are defined in Figure 6.

2) *Conservative Approach.*
   a) Determine the theoretical stress concentration factor K_{ta} based on the axial force.
   b) Determine the theoretical stress concentration factor K_{tb} based on the bending moment.
   c) Use the largest of K_{ta} or K_{tb} and as K_t. Calculate the maximum stress at the stress-riser using

   $S_{\max} = K_t SMAX$ (7)

   where $SMAX$ is defined in Figure 6.

The theoretical approach is technically correct in comparison to the conservative approach. The conservative approach is easier to apply and is a more conservative since it predicts a greater maximum stress compared to the theoretical approach.

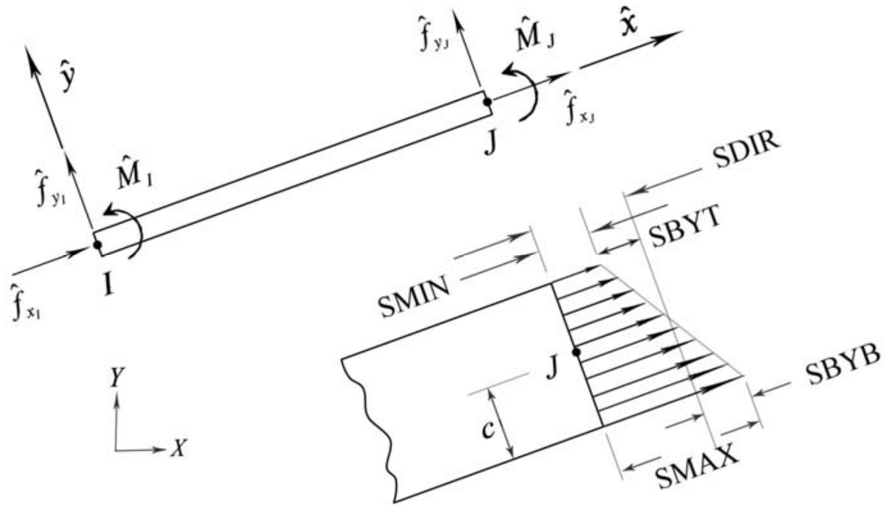


Figure 6. ANSYS® [10] plane beam element normal stress notation and sign convention for node J.

3.2. Static Failure

The bending moments and axial forces in a beam element yield a uniaxial state of stress. Beam elements found in commercial finite element codes also determine the normal stress due to bending moments and axial forces. Shear stresses due to torsion and shear force are neglected. Static failure for a beam element is based on yielding and buckling as follows:

- *Yield Failure.* All elements in the mesh are checked to determine if the factor of safety is satisfied as follows:

$$FS = \frac{S_y}{S_{\max}} \tag{8}$$

where S_y is the material yield strength, and $S_{\max}$ is the maximum stress in the beam element at a node as shown in Figure 6 (includes axial and bending). When the material is brittle, except for cast material and ductile materials, the maximum stress $S_{\max}$ includes K_t. When the state of stress at a point includes the shear stress due to torsional moment and shear force and a normal stress due to bending moment and axial force, then a complex static failure criterion is required as discussed in Section 4.2.

- *Compression Failure due to Buckling.* Elements that have a compressive normal stress due to axial force are checked, i.e., when $SDIR$ is negative in Figure 6. The beam element is assumed pinned at nodes and this is considered conservative. The actual connection at nodes results in a larger buckling load. Consider a column with an end load f shown in Figure 7a. The compressive axial forces f in Figure 7b is the same as the element nodal axial forces $\left(\hat{f}_{x_I}, \hat{f}_{x_J}\right)$ in Figure 7c. Furthermore, the bending moment M in Figure 7b is the same as the element nodal moments $\left(\hat{M}_I, \hat{M}_J\right)$ in Figure 7c. The failure stress for eccentric column buckling is defined by Norton [8] as follows:

$$E_r > 0.1 \qquad\qquad E_r = \frac{ec}{k^2}$$

$$S_{buck} = \frac{S_{yc}}{1 + \left(\frac{ec}{k^2}\right)\sec\left(\frac{l_{eff}}{k}\sqrt{\frac{f}{4EA}}\right)} \qquad \text{Secant Formula} \qquad (9a)$$

$$E_r \le 0.1$$

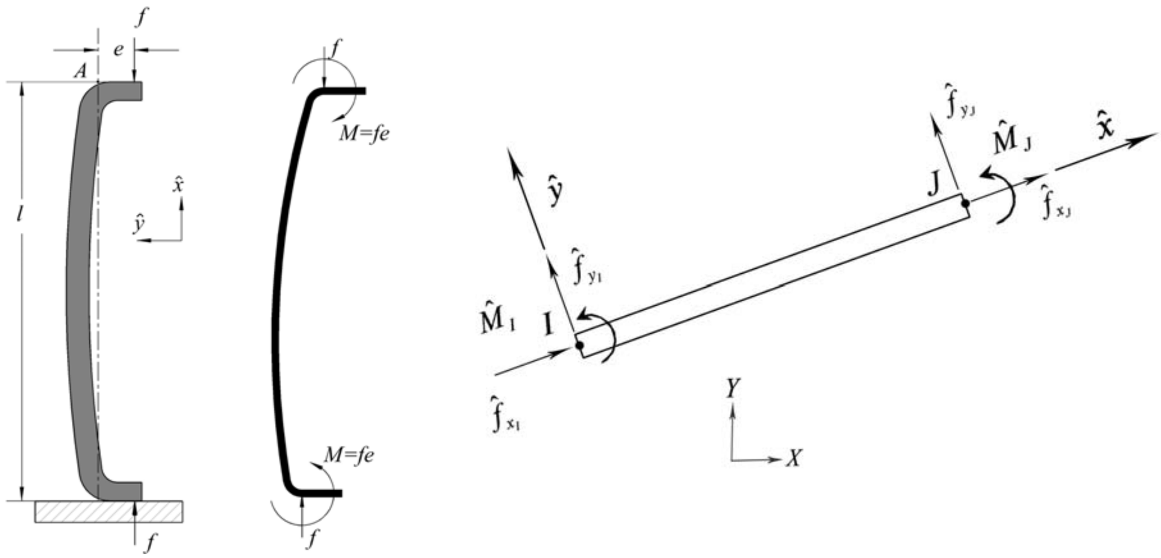
$$S_{buck} = S_{yc} - \frac{1}{E}\left(\frac{S_{yc}S_r}{2\pi}\right)^2 \qquad \text{Johnson Formula} \qquad (9b)$$

with the factor of safety defined as

$$FS = \frac{S_{buck}}{S_{\max}} \qquad (10)$$

where e is the eccentricity, c is the maximum distance from the centroid to the outer fiber of beam, $k = \sqrt{I/A}$ is the radius of gyration of the cross section, I is the least moment of inertia, A is the cross-sectional area, S_{yc} is the compressive strength, $l_{eff} = l$ is the effective length the beam element, E is Young's modulus, and $S_r = l_{eff}/k$ is the slenderness ratio. Beam elements made of brittle material, except for brittle cast material and ductile materials, have an allowable stress S_a that includes the theoretical stress concentration factor K_t, i.e., $S_a = K_t S_{ave}$. Figure 8 shows the column failure lines for secant and Johnson in Equations (9a) and (9b), respectively, for different eccentricity ratios E_r. The element eccentricity e is determined by using the element nodal forces $\left(\hat{f}_{x_I}, \hat{f}_{x_J}\right)$ and nodal moments $\left(\hat{M}_I, \hat{M}_J\right)$ in Figure 7c as follows:

$$e = Maximum\left(\frac{\hat{M}_I}{\hat{f}_{x_I}}, \frac{\hat{M}_J}{\hat{f}_{x_J}}\right) \tag{11}$$



a. Eccentrically loaded column. b. Free-body diagram of column. c. Beam element nodal forces and moments in terms of element coordinates.

Figure 7. Column and beam element.

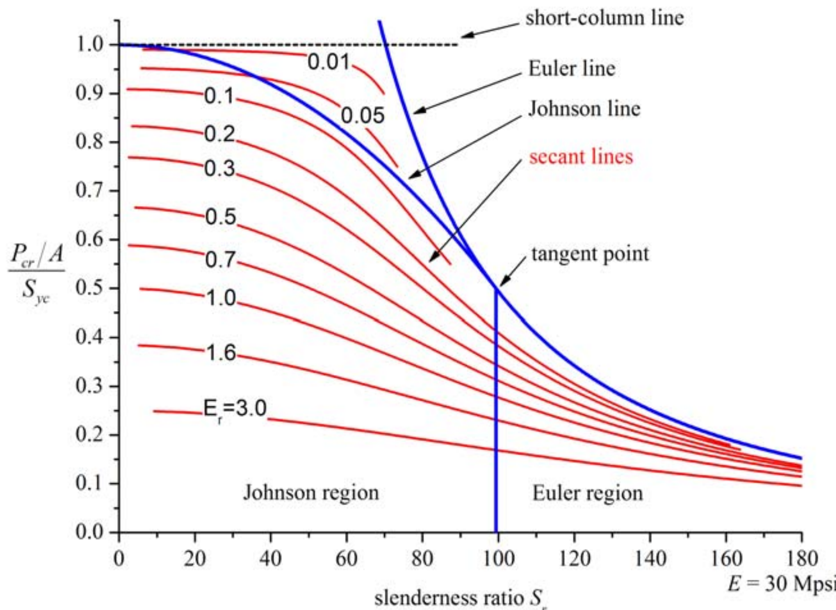


Figure 8. Secant and Johnson failure lines for different eccentricity ratios E_r.

4. Plane Stress/Strain Elements, Axisymmetric Elements, and Solid Elements

The plane stress/strain elements, axisymmetric elements, and solid elements are commonly found in finite element textbooks. Stress concentrations for these elements types are commonly addressed in finite element textbooks. Example problems and homework problems using the von Mises criteria are also commonly found in finite element textbooks. Discussion regarding the von Mises static failure criteria are addressed in [3, 11] and other failure criterion can be found in [9, 12]. The most comprehensive discussion regarding static failure can be found in Adams and Askenazi [4].

Triangular and quadrilateral shaped elements shown in Figures 1c-1f are found in commercial codes for modeling plane stress/strain and axisymmetric problems. These elements include the three- and six-noded triangles (Figures 1c and 1d) and four- and eight-noded quadrilaterals (Figures 1e and 1f). The solid element shapes found in commercial codes are tetrahedrals and bricks as shown in Figures 1g-1j. The tetrahedral has three and ten nodes (Figures 1g and 1h) and the brick has eight and twenty nodes (Figures 1i and 1j). The material for these elements is assumed linear elastic, homogeneous, and isotropic.

4.1. Stress Concentrations

The plane stress/strain elements, axisymmetric elements, and solid elements account for stress concentrations. These elements are formulated based on the theory of elasticity and determine a non-uniform state of stress throughout the body. Therefore, the theoretical stress concentration factor is not required to determine the maximum stress at a stress-riser. The stress value from the finite element analysis is used as the maximum stress at a stress-riser. Stress concentrations are commonly addressed in finite element textbooks for the plane stress/strain, axisymmetric, and solid elements.

The first author has applied the theoretical stress concentration factor to the finite element stress for a solid element mesh when only a coarse mesh could be generated. An example is where a gas turbine contains extremely small gas holes. These holes are so small that it becomes impossible, even with today's mesh generators, to create a fine solid mesh around the tiny hole. In this case the coarse mesh stress is multiplied by the stress concentration factor. This type of problem can only be modeled accurately using the boundary element method [13] to determine the actual stress concentration near the holes.

The first author has found that practitioners and students at times will carry out an incorrect finite element analysis when there is a sharp corner. Consider the L-bracket shown in Figure 9 that was considered in [14]. The exercise is to find the maximum von Mises stress in the L-bracket and to determine if the part fails when subjected to the uniformly distributed load of 1500 N/mm. A commercial finite element code is used to carry out a convergence study by solving the problem using successive mesh refinements as shown in Figure 10 (smaller four-noded quadrilateral elements are used in the vicinity of the re-entrant corner). A plot of the maximum von Mises stress versus the number of degrees of freedom is shown in Figure 11. From this graph the analyst determines that the stress will never converge. The reason is that theory of elasticity states that an infinite stress arises at a re-entrant corner. This is a common mistake where the practitioner will chase a stress that can never be obtained. Even more common is that students and practitioners do not carry out a proper convergence study and simply use the value of the von Mises stress for a given mesh. This type of application reinforces how important it is for students and practitioners to have an understanding of finite element theory and mechanics of materials theory. Furthermore, educating students on these slight yet often overlooked problems in finite element analysis instills a strong sense of practical and fundamental modeling skills. If the bracket had a fillet at the corner of interest then the stress will converge. This exercise demonstrates that "A lack of understanding finite element fundamentals can introduce the potential for erroneous stresses and deflections even in simple classical examples [15]."

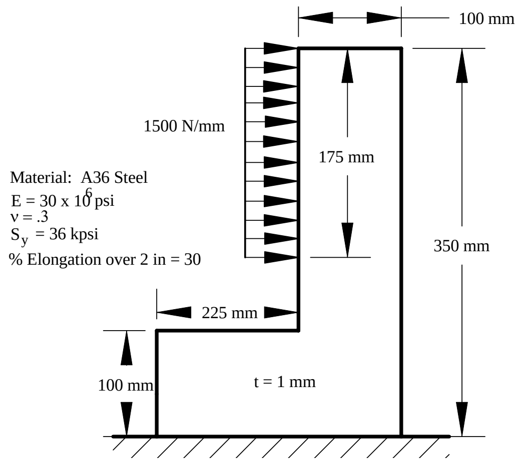
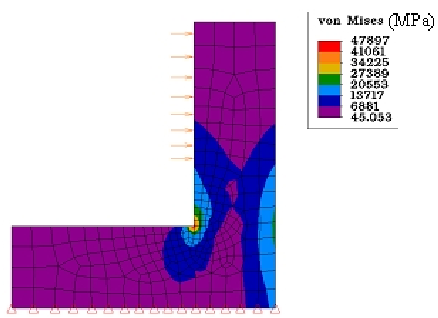
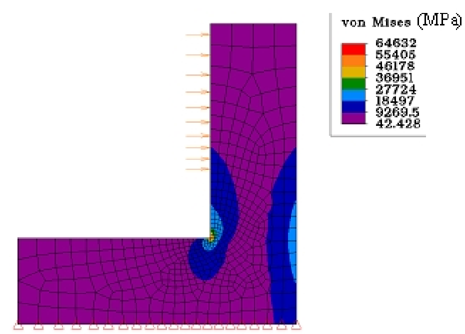


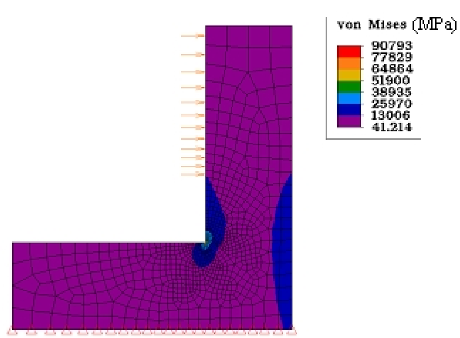
Figure 9. L-bracket problem definition.



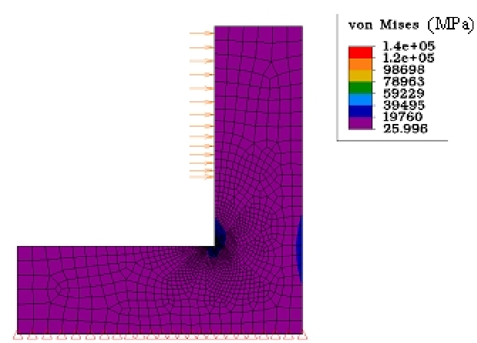
a. 248 Elements, 285 Nodes, 570 DOF.



b. 480 Elements, 527 Nodes, 1054 DOF.



c. 827 Elements, 885 Nodes, 1770 DOF.



d. 1672 Elements, 1751 Nodes, 3502 DOF.

Figure 10. L-bracket finite element plane stress solution for the von Mises stress at the re-entrant corner for four meshes with 4-noded quadrilateral elements.

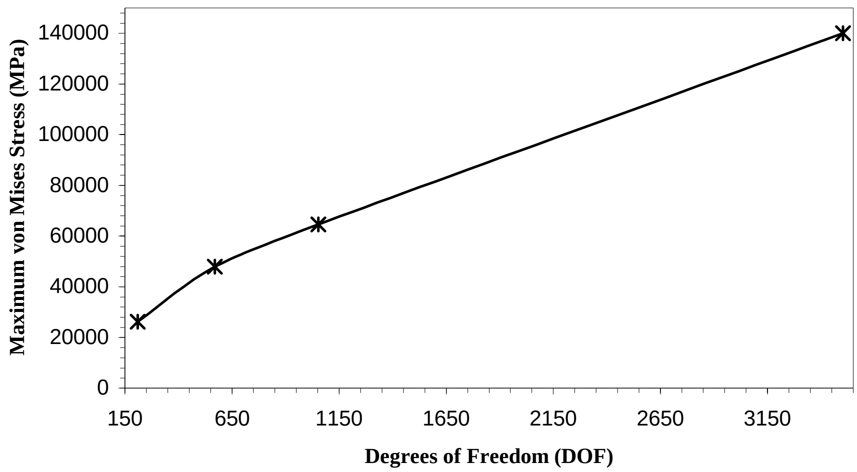


Figure 11. L-bracket non-convergence study of von Mises stress at re-entrant corner.

4.2. Static Failure

The state of stress at a point in plane stress/strain elements, axisymmetric elements and solid elements is complex such that failure theories are required. A failure theory is also known as failure criteria and addresses only material failure and not failure of the structure. Static failure criteria serve to predict whether a given state of stress will

1. cause the material to yield (ductile material), or
2. cause the material to fracture (brittle material).

The most common static failure criteria found in finite element textbooks is the von Mises failure criteria applied to plane stress/strain and axisymmetric problems. Unfortunately, finite element textbooks do not make it clear why the von Mises failure criteria is used.

The failure criteria for the plane stress problems are considered for brevity. The failure criteria selected is based on the material type as follows:

- *Ductile Material (*$\varepsilon_f > 0.05$*).* The definition of a ductile material was discussed in Section 2.1. The most common failure criteria used for ductile materials and found in commercial finite element codes are von Mises and Tresca as shown in Figure 12. These two criteria are as follows:

1. *von Mises (von Mises-Hencky) or Distortion-Energy Criteria.* This is considered the most accurate and preferred approach for ductile materials since it has very good correlation with experimental data. The von Mises theory states that yielding begins when the distortion energy equals the distortion energy at yield in simple tension. However, it is more convenient to state von Mises criteria in terms of an effective stress σ_e. The value of σ_e that defines yielding can be determined from a standard tensile test specimen. One can show that $\sigma_e = S_y$ is reached in uniaxial test, and for any other state of stress failure is defined as:

$$\sigma_e \geq S_y \tag{12a}$$

where S_y is the tensile yield strength. The effective stress σ_e in terms of principal stresses for a plane stress problem is:

$$\sigma_e = \sqrt{\sigma_1^2 + \sigma_1\sigma_2 + \sigma_2^2} \tag{12b}$$

or in terms of Cartesians stresses

$$\sigma_e = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + \tau_{xy}^2} \tag{12c}$$

The principal stresses (σ_1, σ_2) and Cartesians stresses (σ_x, σ_y, τ_{xy}) are determined from the finite element program. The safety factor for von Mises is defined as

$$FS = \frac{S_y}{\sigma_e} \tag{12d}$$

The effective stress σ_e is a fictitious stress that does not really act on any plane in a component, it is simply a number, always positive, representing an effective intensity of a stress equivalent to the three principal stresses. Since it is always positive it does not provide an indication of tension or compression. Figure 12 shows the von Mises ellipse failure envelope (yield surface or yield locus) for plane stress ($\sigma_3 = 0$) in terms of principal stresses (σ_1, σ_2).

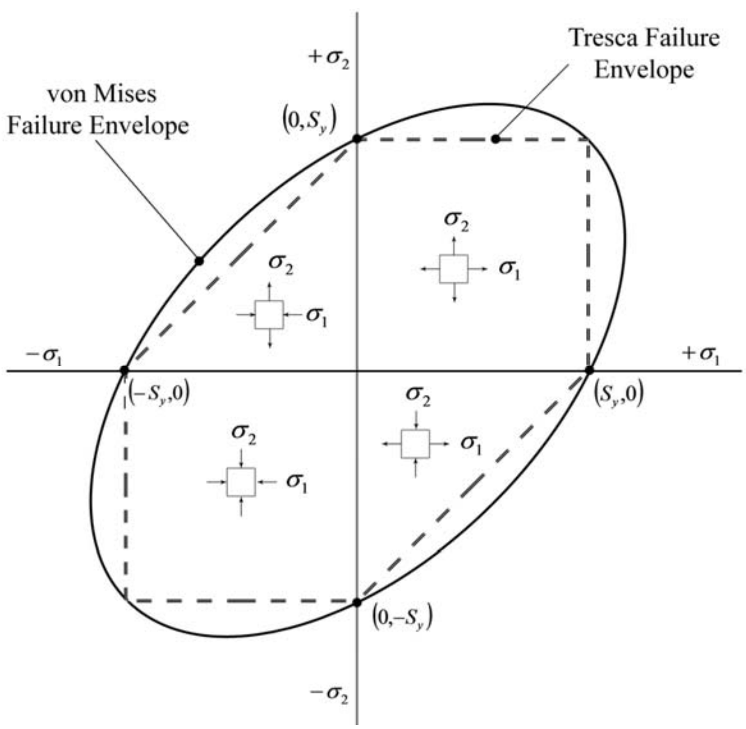


Figure 12. von Mises and Tresca failure envelopes for ductile materials.

2. *Tresca or Maximum-Shear Stress Criteria.* Tresca is mathematically simpler and more conservative than von Mises. The Tresca criteria states that yielding will begin at a point in a body when the maximum shear stress at that point equals the maximum shear stress in a standard tensile test specimen when the specimen begins to yield as follows:

$$\tau_{\max} \geq S_{y_s} = \frac{S_y}{2} \tag{13a}$$

where $\tau_{\max}$ is the maximum shear stress at a point and S_{y_s} is the shear strength of the material at yield that is one-half of the tensile yield strength S_y. Equation (13a) is a good relationship to remember in practice if only the yield strength is available. The factor of safety for Tresca is defined as

$$FS = \frac{S_{y_s}}{\tau_{\max}} = \frac{S_y}{2\tau_{\max}} \tag{13b}$$

Figure 12 shows the Tresca hexagonal failure envelope for plane stress ($\sigma_3 = 0$) in terms of principal stresses ($\sigma_1,\ \sigma_2$).

- *Brittle Material ($\varepsilon_f \leq 0.05$).* The definition of a brittle material was discussed in Section 2.1. The most common failure criteria used in practice includes the following:

  1. *Maximum Normal-Stress Criteria.*
  2. *Modified-Mohr Theory or Modified Internal Friction Criteria.*

  The finite element courses taught by the first author only provides an overview of these two criteria for brittle materials with no applications due to time limitations and therefore is not addressed in this paper.

Conclusion

This paper considers stress concentrations and static failure for elements that are commonly used in practice to carry out a static finite element stress analysis. The elements considered include the truss, beam, plane stress/strain, axisymmetric, and solid. Stress concentrations and static failure have not been comprehensively addressed in any finite element textbook. The first author has integrated these topics into his introductory finite element undergraduate, graduate, and industrial short courses. This was done when the first author started teaching industrial finite element short courses. The plate and shell elements are also commonly used element in practice. However, these elements were not addressed since stress concentrations and static failure are more complex then the elements considered in this paper.

References

1. Knight, C.E., *The Finite Element Method in Mechanical Design*, PWS-KENT Publishing Company, Boston, MA, 1993, p. 59.
2. Moaveni, S., *Finite Element Analysis: Theory and Application with ANSYS*, Prentice Hall, Upper Saddle River, NJ, 1999, p. 117.
3. Logan, D.L., *A First Course in the Finite Element Method*, Fourth Edition, Thompson Canada Limited, Toronto, Ontario, 2007, p. 149, pp. 291 & 297, pp. 341-342.
4. Adams, V. and Askenazi, A., *Building Better Products with Finite Element Analysis*, Onward Press, Sante Fe, NM, 1999, pp. 56-57 & 62-64, pp. 65-66, pp. 58-62.
5. Peterson, R.E., *Stress Concentration Factors*, John Wiley and Sons, New York, NY, 1974.
6. Roark, R.J. and Young, W.C., *Formulas for Stress and Strain*, Sixth Edition, McGraw-Hill, New York, NY, 1989.
7. Pilkey, W.D., *Peterson's Stress Concentration Factors*, John Wiley and Sons, New York, NY, 1991.
8. Norton, R.L., *Machine Design: An Integrated Approach*, Third Edition, Pearson Prentice Hall, Upper Saddle River, NJ, 2006.
9. Potts, J.F. and Oler, J.W., *Finite Element Applications with Microcomputers*, Prentice Hall, Englewood Cliffs, NJ, 1989, p. 132, pp. 88-90.
10. *ANSYS, Inc.*, Southpointe, 275 Technology Drive, Canonsburg, PA, http://www.ansys.com/.
11. Cook, R.D., Malkus, D.S., Plesha, M.E. and Witt, R.J., *Concepts and Applications of Finite Element Analysis*, Fourth Edition, John Wiley & Sons, Inc., New York, NY, 2002, pp. 116-117.
12. Spyrakos, C.C., *Finite Element Modeling in Engineering Practice*, Algor Publishing Division, Pittsburg, PA, 1996, pp. 29-32.
13. Brebbia, C.A. and Dominguez, J., *Boundary Elements: An Introductory Course*, Computational Mechanics, Southampton, UK and McGraw-Hill Book Company, New York, NY, 1989.
14. Rencis, J.J., Grandin, H.T. and Jolley, W.O., "A Finite Element Module for Undergraduates," *Proceedings of the* 41st *American Society of Engineering Education (ASEE) Midwest Section Meeting*, Kansas City, MO, September 13-15, 2006.
15. Kurowski, P., "Easily Made Errors Mar FEA Results," *Machine Design*, September 13, 2001, http://www.machinedesign.com/.

JOSEPH J. RENCIS
Joseph J. Rencis is Professor and Head of the Department of Mechanical Engineering at the University of Arkansas in Fayetteville. From 1985 to 2004 he was a Professor in the Mechanical Engineering Department at Worcester Polytechnic Institute. His research focuses on the development of boundary and finite element methods for analyzing solid, heat transfer, and fluid mechanics problems. He currently serves on the editorial board of Engineering Analysis with Boundary Elements and is associate editor of the International Series on Advances in Boundary Elements. Currently he serves as the Vice Chair of the

ASME Mechanical Engineering Department Heads Committee, Program Chair-Elect of the ASEE Mechanical Engineering Division, and ABET program evaluator. He has been the Chair of the ASEE Mechanics Division, received the 2002 ASEE New England Section Teacher of the Year, and is a fellow of the ASME. He received the 2004 ASEE New England Section Outstanding Leader Award and 2006 ASEE Mechanics Division James L. Meriam Service Award. He received his B.S. from the Milwaukee School of Engineering in 1980, a M.S. from Northwestern University in 1982, and a Ph.D. from Case Western Reserve University in 1985. V-mail: 479-575-4153; E-mail: jjrencis@uark.edu.

SACHIN TERDALKAR
Sachin Terdalkar is a Ph.D. candidate in the Department of Mechanical Engineering at the University of Arkansas in Fayetteville. His current research mainly focuses on finite element analysis and molecular dynamics. More recently has been working on molecular dynamic simulation of ion deposition induced curvature. He is also working on comparing a Virial stress formulation used in molecular dynamics with the continuum Cauchy stress formulation. He has worked as senior engineer in John Deere Technology Center, Pune INDIA. Responsibilities at John Deere included finite element analysis and fatigue analysis to determine life of the newly designed components for new generation tractors. He received a M.S. in Mechanical Engineering from the Worcester Polytechnic Institute in 2003. His M.S. research formulated and developed a new algorithm for interactive stress reanalysis in early stages of design using ANSYS®. He received his B.S. from the College of Engineering, Pune INDIA in 1999. V-mail: 479-575-6821; E-mail: sterdal@uark.edu.