

Ö. HATİPOĞLU

ATILIM UNIVERSITY 2023

BEHAVIOR OF PIN CONNECTIONS UNDER TRANSVERSE LOAD

THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF
ATILIM UNIVERSITY

ÖZGE HATİPOĞLU

A MASTER OF SCIENCE THESIS
IN
THE DEPARTMENT OF CIVIL ENGINEERING

APRIL 2023

BEHAVIOR OF PIN CONNECTIONS UNDER TRANSVERSE LOAD

A THESIS SUBMITTED TO
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF
ATILIM UNIVERSITY

BY

ÖZGE HATİPOĞLU

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR
THE DEGREE OF MASTER OF SCIENCE
IN
CIVIL ENGINEERING

APRIL 2023

Approval of the Graduate School of Natural and Applied Sciences, Atilim University.

Prof. Dr. Ender Keskinliç
Director

I certify that this thesis satisfies all the requirements as a thesis for the degree of **Master of Science in Civil Engineering Department, Atilim University.**

Prof. Dr. Yakup Darama
Head of Department

This is to certify that we have read the thesis BEHAVIOR OF PIN CONNECTIONS UNDER TRANSVERSE LOAD submitted by ÖZGE HATİPOĞLU and that in our opinion it is fully adequate, in scope and quality, as a thesis for the degree of Master of Science.

Asst. Prof. Dr. Ertan Sönmez
Supervisor

Examining Committee Members:

Prof. Dr. Tolga Akış
Civil Eng. Department, Atilim University

Asst. Prof. Dr. Ertan Sönmez
Civil Eng. Department, Atilim University

Prof. Dr. Afşin Sarıtaş
Civil Eng. Dept., Middle East Technical University

Date: April 12, 2023



I declare and guarantee that all data, knowledge and information in this document has been obtained, processed and presented in accordance with academic rules and ethical conduct. Based on these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Name, Last Name : ÖZGE HATİPOĞLU

Signature :

ABSTRACT

Behavior of Pin Connections Under Transverse Load

Hatipoğlu, Özge

M.S., Department of Civil Engineering

Supervisor : Asst. Prof. Dr. Ertan Sönmez

April 2023, 92 pages

Pin connections are typically designed for longitudinal loads along the axis of the member or connection. Accordingly, design provisions for the pin connections primarily consider this case in structural standards. In this thesis, the behavior of pin connections under transverse load is investigated by analyzing parametric finite element models and studying the effects of pin-to-hole diameter ratio, eccentricity of hole (in axial direction), and extension of pin (beyond the connection plate). The contact interaction between the pin and the plate is modeled by face-to-face penalty contact formulation with linear pressure-overclosure relationship. The contact between the pin and plate is assumed to be frictionless. The accuracy of models are checked via mesh sensitivity analyses and comparison to analytical solutions given by Hertz for non-conformal cylindrical contacts and by Persson for conformal cylindrical contact.

The results of analyses performed in this study has shown that pin-to-hole ratio and bending of pin can have significant effects on stress concentration in pin connections under transverse load while axial eccentricity of plate hole has a limited effect. Although two-dimensional models are useful in preliminary assessment and model verification, they ignore the through-thickness effects that may be significant in some cases; therefore, three-dimensional models provide more accurate representation of stress distribution in pin connections. The analyses have also confirmed that the analytical solutions of both Hertz and Persson gave reasonable results for smaller contact angles when pin bending effect is not present while Hertz's solution deviates significantly from the results of finite element models with neat-fit pin having a more conformal contact surface with larger contact angles.

Keywords: Pin connections, contact mechanics, finite element analysis, Calculix



ÖZ

Mil Birleşimlerinin Enine Yük Altındaki Davranışı

Hatipoğlu, Özge

Yüksek Lisans, İnşaat Mühendisliği

Tez Yöneticisi : Asst. Prof. Dr. Ertan Sönmez

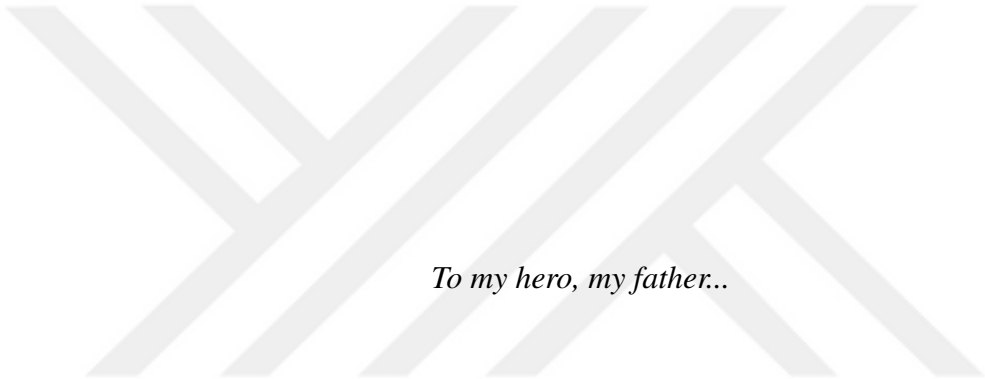
Nisan 2023, 92 sayfa

Mil birleşimleri genellikle eleman veya birleşim eksenine paralel yükler için tasarlanmaktadır. Buna göre, yapı standartlarında mil birleşimleri için verilen tasarım kuralları esas olarak bu durumu dikkate alır. Bu tezde, parametrik sonlu eleman modelleri kullanılarak mil birleşimlerinin enine yük altındaki davranışı araştırılıp, mil-delik çap oranı, deliğin eksenel yöndeki dışmerkezliği ve milin eğilme etkileri incelenmiştir. Mil ve plaka arasındaki temas etkileşimi, yüz yüze penaltı temas formülasyonu ile modellenmiştir. Mil ile plaka arasındaki temasın sürtünmesiz olduğu varsayılmıştır. Modellerin doğruluğu, ağ duyarlılığı analizleri ve Hertz'in uyumlu olmayan silindirik yüzeylerin teması için ve Persson'ın uyumlu silindirik temaslar için geliştirdiği analitik çözümlerle karşılaştırılarak kontrol edilmiştir.

Bu çalışmada yapılan analizlerin sonuçları, mil-delik çap oranı ve milin eğilme parametrelerinin enine yük altında olan mil bağlantılarındaki gerilme konsantrasyonu üzerinde önemli etkileri olabileceğini, deliğin eksenel yöndeki dışmerkezliğinin ise sınırlı bir etkisi olduğunu göstermiştir. İki boyutlu modeller, ön değerlendirme ve model doğrulamada yararlı olsa da, bazı durumlarda önemli olabilecek kalınlık boyu etkileri göz ardı ederler; bu nedenle, üç boyutlu modeller mil bağlantılarındaki gerilim dağılımının daha doğru temsili sağlamaktadırlar. Yapılan analizler ayrıca, hem Hertz hem de Persson'ın analitik çözümlerinin mil eğilme etkisinin olmadığı durumlarda daha küçük temas açıları için makul sonuçlar verdiğini, ama daha büyük temas açılarında Hertz'in çözümünün sonlu elemanlar çözümünden uzaklaştığını göstermiştir.

Anahtar Kelimeler: Mil birleşimleri, temas mekaniği, sonlu elemanlar analizi, Calculix





To my hero, my father...

ACKNOWLEDGMENTS

First and foremost, I owe a debt of gratitude to my supervisor Asst. Prof. Dr. Ertan Sönmez for his continuous advice, guidance and forbearance. There was not any second for him to hesitate if he had enough time to answer my questions or for the support which I really needed. I may have learnt everything I need owing to him. Most importantly I have learnt to know enough to come in out of the rain in life. Before it was decided that he was my supervisor, I had already wished to be his student and follow his path. Now, no matter how much I am grateful for his supervisorship, it will not fit in a few lines to describe.

I must express my sincere appreciation to Prof. Dr. Afşin Saritaş for his precious comments on this thesis.

My gratitude is also extended to Prof. Dr. Tolga Akış for his valuable advices and reassurance. I will always think of him with respect.

In particular, I extremely want to give my special thanks to Prof. Dr. Eray Baran who was my lecturer and my advisor in the past. Without him, I would never have been a civil engineer and may not have been able to get a master's degree. I have always felt his support and encouragement from my business life to my academic life.

I think it is essential to extend my thanks to my long term friends Merve and Bergütay who always made time to support me even I did not pick up my phone.

My acknowledgments is not said to be completed without expressing my sincere thanks going to my very respectful doctor Prof. Dr. Ali Şehirlioğlu who has improved the quality of my life. If life hadn't given me the chance to be his patient, my long hours of studies would have been painful.

Last but not least, my heartfelt thanks are due to my family. I am always proud to be a daughter, a sister and an aunt of them. Every breath they take is joy of my

life. More specifically, my warmest thanks go to my mother Zehra Hatipoğlu and my father İsmail Hatipoğlu. I always consider that they are the most incredible parents who always provide the help, guidance and much more love which any child needs.



TABLE OF CONTENTS

ABSTRACT	iii
ÖZ	v
DEDICATION	vii
ACKNOWLEDGMENTS	viii
TABLE OF CONTENTS	x
LIST OF TABLES	xii
LIST OF FIGURES	xiii
CHAPTER	
1. INTRODUCTION	1
1.1 Problem Statement	1
1.2 Objective and Scope	2
2. LITERATURE REVIEW	3
2.1 Analytical Studies	3
2.1.1 Hertz's Solution for Non-conformal Cylindrical Contact	4
2.1.2 Persson's Solution for Conformal Cylindrical Contact	7
2.2 Experimental Studies	9
2.3 Numerical Studies	13
2.4 Design Requirements for Pin Connections	15
2.4.1 AISC 360-16	15
2.4.2 EN 1993-1-8	18
3. MODELING OF CONTACT PROBLEMS	21
3.1 General Formulation of Contact Problems	21

3.1.1	Spring-Mass Frictionless Contact System	21
3.1.2	Contact Formulations	23
3.1.2.1	Lagrange Multiplier Method	23
3.1.2.2	Penalty Method	24
3.1.2.3	Augmented Lagrangian Method	26
3.1.3	Formulation in Continuum Mechanics	26
3.2	Finite Element Method	30
3.3	Finite Element Formulation of Contact Problems	37
3.4	Finite Element Analysis Software – Calculix	41
3.4.1	Calculix GraphiX (CGX)	41
3.4.2	Calculix CrunchiX (CCX)	43
4.	FINITE ELEMENT ANALYSIS OF PIN CONNECTIONS UNDER TRANSVERSE LOAD	48
4.1	Model Description	48
4.1.1	Model Geometry and Dimensional Parameters	48
4.1.2	Mesh and Boundary Conditions	50
4.1.3	Material Properties	53
4.1.4	Loading	53
4.2	Analysis Procedure	55
4.3	Model Verification	56
4.3.1	Sensitivity Analyses for Mesh Size	56
4.3.2	Comparison with Analytical Solutions	57
4.3.3	Comparison with Abaqus	59
4.4	Results of Two-Dimensional Finite Element Models	61
4.5	Results of Three-Dimensional Finite Element Models	70
5.	CONCLUSION	84
5.1	Summary	84
5.2	Concluding Remarks	84
	REFERENCES	92

LIST OF TABLES

TABLES

Table 2.1	Design criteria for pin connections in EN 1993-1-8 [2]	18
Table 2.2	Partial safety factors used in pin connection design	19
Table 4.1	Dimensional parameters	49
Table 4.2	Number of parametric models	49
Table 4.3	List of 2D models	50
Table 4.4	List of 3D models	51
Table 4.5	Number of elements and nodes used in some models	52
Table 4.6	Material properties for linear analysis	53
Table 4.7	Applied load and code unity checks for no eccentricity models ($a = c$) with $t_e/t = 0.5$ (assuming a yield strength of $\sigma_y = 225$ MPa for both plate and pin).....	55
Table 4.8	Number of elements and nodes used in 2D mesh sensitivity models .	57
Table 4.9	Number of elements and nodes used in 3D mesh sensitivity models .	58
Table 4.10	Comparison of Calculix 2D quarter model results with analytical solutions	59
Table 4.11	Comparison of Calculix 2D model results with Abaqus	60
Table 4.12	Comparison of Calculix 3D model results with Abaqus	62
Table 4.13	Summary of maximum von Mises stress results for all 2D models ...	73
Table 4.14	Summary of maximum von Mises stress results near the plate hole for all 3D models	82
Table 4.15	Summary of maximum von Mises stress results in the pin for all 3D models	83

LIST OF FIGURES

FIGURES

Figure 2.1 Non-conformal contact of two cylinders: cylinder in cylindrical cavity	7
Figure 2.2 Conformal contact of two cylinders	9
Figure 2.3 Comparison of normalized pressure profiles given by Persson and Hertz	10
Figure 2.4 Contact angle relation to normalized load	10
Figure 2.5 Typical pin connection configuration used in experimental studies [23, 24, 22].....	11
Figure 2.6 Pin loading in a three-plate connection	12
Figure 2.7 Pin loading model by Blake [30]	13
Figure 2.8 Pin loading model by Melcon and Hoblit [31]	13
Figure 2.9 Lug configuration in Ekvall [28]	14
Figure 2.10 AISC 360-16 dimensional requirements for pin-connected members	17
Figure 2.11 Bending moment in a pin	19
Figure 2.12 EN 1993-1-8 geometrical requirements for pin connections based on plate thickness (Type A)	20
Figure 2.13 EN 1993-1-8 geometrical requirements for pin connections based on plate geometry (Type B)	20
Figure 3.1 Mass-spring system in contact	22
Figure 3.2 Physical explanation of Penalty method	25
Figure 3.3 Equilibrium condition for a BVP	28
Figure 3.4 3D isoparametric solid element: (a) finite element, (b) reference element	35
Figure 3.5 Continuum and discrete contact conditions	37
Figure 3.6 Frictional interface model	40
Figure 3.7 Pre-processing with CGX	42
Figure 3.8 Post-processing with CGX	43

Figure 3.9 C3D20R quadratic brick element: (a) nodes, (b) $2 \times 2 \times 2$ reduced integration points	44
Figure 3.10 Expansion of 2D 8-node element into a 3D brick element: (a) original nodes, (b) expanded nodes	44
Figure 3.11 Linear pressure-overclosure relationship	46
Figure 4.1 Model geometry and dimensional parameters	49
Figure 4.2 Mesh division parameters for 2D and 3D models	52
Figure 4.3 Structured mesh for 2D models with $d_p/d = 0.9$: (a) $a/c = 1$, (b) $a/c = 0.5$, (c) $a/c = 1.5$	52
Figure 4.4 Structured mesh for 3D models with $d_p/d = 0.9$ and $a/c = 1$: (a) $t_e/t = 0$, (b) $t_e/t = 0.25$, (c) $t_e/t = 0.5$	53
Figure 4.5 Load application in 3D models: (a) model sketch, (b) model mesh .	54
Figure 4.6 Contact surfaces: MASTER (red), SLAVE (blue); (a) 2D model (2dm3), (b) 3D model (3dm9)	55
Figure 4.7 Mesh sensitivity results for 2D and 3D models	57
Figure 4.8 2D quarter model for comparison with analytical solutions	59
Figure 4.9 von Mises stresses for 2D model with $d_p/d = 0.9$ and $a/c = 1$: (a,b) Calculix, (c,d) Abaqus [model: 2dm3]	61
Figure 4.10 von Mises stresses for 3D model with $d_p/d = 0.9$, $a/c = 1$, and $t_e/t = 0$ (a,b) Calculix, (c,d) Abaqus [model: 3dm3]	62
Figure 4.11 von Mises stress results for 2D model with $d_p/d = 0.999$ and $a/c = 1$ [model: 2dm1]	63
Figure 4.12 von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.999$ and $a/c = 1$: (a) pin, (b) plate [model: 2dm1] ...	63
Figure 4.13 von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.999$ and $a/c = 1$ [model: 2dm1]	64
Figure 4.14 von Mises stress results for 2D model with $d_p/d = 0.98$ and $a/c = 1$ [model: 2dm2]	64
Figure 4.15 von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.98$ and $a/c = 1$: (a) pin, (b) plate [model: 2dm2]	65
Figure 4.16 von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.98$ and $a/c = 1$ [model: 2dm2]	65
Figure 4.17 von Mises stress results for 2D model with $d_p/d = 0.90$ and $a/c = 1$ [model: 2dm3]	66
Figure 4.18 von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.90$ and $a/c = 1$: (a) pin, (b) plate [model: 2dm3]	66

Figure 4.19 von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.90$ and $a/c = 1$ [model: 2dm3]	67
Figure 4.20 von Mises stress results for 2D model with $d_p/d = 0.999$ and $a/c =$ 0.5 [model: 2dm4]	67
Figure 4.21 von Mises stress in radial directions at $0, 45, 90, 135, 180^\circ$ for 2D model with $d_p/d = 0.999$ and $a/c = 0.5$: (a) pin, (b) plate [model: 2dm4] .	68
Figure 4.22 von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.999$ and $a/c = 0.5$ [model: 2dm4]	68
Figure 4.23 von Mises stress results for 2D model with $d_p/d = 0.98$ and $a/c =$ 0.5 [model: 2dm5]	69
Figure 4.24 von Mises stress in radial directions at $0, 45, 90, 135, 180^\circ$ for 2D model with $d_p/d = 0.98$ and $a/c = 0.5$: (a) pin, (b) plate [model: 2dm5] ...	69
Figure 4.25 von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.98$ and $a/c = 0.5$ [model: 2dm5]	70
Figure 4.26 von Mises stress results for 2D model with $d_p/d = 0.90$ and $a/c =$ 0.5 [model: 2dm6]	70
Figure 4.27 von Mises stress in radial directions at $0, 45, 90, 135, 180^\circ$ for 2D model with $d_p/d = 0.90$ and $a/c = 0.5$: (a) pin, (b) plate [model: 2dm6] ...	71
Figure 4.28 von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.90$ and $a/c = 0.5$ [model: 2dm6]	71
Figure 4.29 von Mises stress results for 2D model with $d_p/d = 0.999$ and $a/c =$ 1.5 [model: 2dm7]	72
Figure 4.30 von Mises stress results for 2D model with $d_p/d = 0.98$ and $a/c =$ 1.5 [model: 2dm8]	72
Figure 4.31 von Mises stress results for 2D model with $d_p/d = 0.90$ and $a/c =$ 1.5 [model: 2dm9]	73
Figure 4.32 von Mises stress results for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]	74
Figure 4.33 von Mises stress at the plate in radial directions at $0, 45, 90, 135, 180^\circ$ for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]	74
Figure 4.34 von Mises stress at the pin in radial directions at $0, 45, 90, 135, 180^\circ$ for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]	75
Figure 4.35 von Mises stress in thru-thickness direction for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]	75
Figure 4.36 von Mises stress along the pin/hole circumference for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]	76

Figure 4.37 von Mises stress results for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]	76
Figure 4.38 von Mises stress at the plate in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]	77
Figure 4.39 von Mises stress at the pin in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]	77
Figure 4.40 von Mises stress in thru-thickness direction for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]	78
Figure 4.41 von Mises stress along the pin/hole circumference for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]	78
Figure 4.42 von Mises stress results for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]	79
Figure 4.43 von Mises stress at the plate in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5] ..	79
Figure 4.44 von Mises stress at the pin in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5] ..	80
Figure 4.45 von Mises stress in thru-thickness direction for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]	80
Figure 4.46 von Mises stress along the pin/hole circumference for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]	81

CHAPTER 1

INTRODUCTION

1.1 Problem Statement

Pin connections are a common type of connection utilized in many engineering applications. Due to the rotation allowed through the pin, these connections do not resist to any bending moment about the pin axis. While this simplify the design and behavior of pin connections, the stress distribution in the connection elements are very complex due to the contact mechanics involved between the pin and connection plate(s). Although they are in general designed to transfer only axial loads in along the longitudinal axes of the connecting members, they may also be subjected to transverse or oblique loading in some applications.

The structural design standards and specifications usually have limited coverage for the design of pin connections with consideration of only axial loading. Two main references for structural engineering applications of pin connections are AISC 360-16 [1] specification and EN 1993-1-8 [2] standard. Both documents provide empirical equations for dimensional requirements without giving much detail on their development and consider only axial loading.

The lack of detailed information and consideration of only axial loading in most design standards indicate a need for additional research to have a better understanding of the behavior of pin connections under other loading directions.

1.2 Objective and Scope

The main objective of this study is to investigate how certain dimensional parameters affect the behavior of pin connections under transverse load by analyzing parametric finite element models that include frictionless contact interactions between the pin and connection plate. The main parameters that are examined include pin-to-hole diameter ratio, eccentricity of the hole in axial direction, and the pin extension beyond the plate thickness (for pin bending due to space between the connecting members through the pin). Finite element analyses are performed with both two-dimensional (2D) and three-dimensional (3D) models.

The thesis consists of five chapters. Chapter 1 presents the problem statement, the objective and scope of this study. A detailed literature review is presented in Chapter 2. Next, modeling of contact problems are discussed in Chapter 3. The results of parametric finite element analysis (FEA) models are presented in Chapter 4. And finally, the main conclusions of this study are summarized in Chapter 5. Finite element analyses of pin connections are performed using Calculix [3], which is an open source FEA software.

CHAPTER 2

LITERATURE REVIEW

A detailed review of past studies on pin connections along with the current structural design practice are presented in this chapter, which is divided into four sections as: (i) analytical studies, (ii) experimental studies, (iii) numerical studies, and (iv) design requirements for pin connections in AISC 360-16 [1] and EN 1993-1-8 [2].

2.1 Analytical Studies

A comprehensive review of history of pin connections and proposed analytical solutions can be found in Harms [4]. One of the first applications of pin connections was eye bars used for connection of cables in suspension bridges in late 19th century [4]. Since there was not much information about the stress distribution in the eye, first connections were designed with rough approximations and experiments. The experimental results of a reference eye were used to derive the first stress concentration factors.

Schaper [5], Bleich [6], Reißner and Strauch [7], Reidelbach [8], Poócza [9] developed analytical solutions for eye bars based on the elastic behavior and the distribution and orientation of the bearing load. All proposed analytical theories were based on elastic material behavior and certain assumptions on how the bearing load was distributed and oriented. The differences in the outcome of these theories were due to these assumptions, simplifications and calculation methods. Schaper [5] assumed radial orientated bearing load on the eye and derived the stresses in the eye using the resultant of these radial stresses. Bleich [6] assumed a radial orientated distributed bearing load and derived a set of equations from which the stress concentration fac-

tors can be calculated. Reißner and Strauch [7] assumed a sinusoidal distributed and radial orientated bearing load and derived a formula to calculate the stresses at the net section. Reidelbach [8] assumed that a shear stress is transferred via the contact in addition to a radial orientated bearing load and derived the stress concentration factors through whole section next to the eye and on top of the eye. Poócza [9] assumed a varying section height to include the effect of eccentricities and derived equations for stress concentration factors at the net section and at the top of the eye.

Hertz [10, 11, 12] conducted one of the first analytical studies on stresses between two contact surfaces. He studied several contact cases including non-conformal contact of a cylinder in a cylindrical cavity.

A closed form solution for conformal contacts of frictionless cylindrical surfaces made of identical materials is first obtained by Persson [13] in his Ph.D. thesis. Later, Noble and Hussain [14] developed an exact solution for an indentation problem where an infinite body with a smooth circular inclusion to which a concentrated force is applied. The authors formulated the problem in terms of dual series relations which in turn are transformed to an equivalent Cauchy-type or airfoil equation, leading to an exact solution for the case of perfect fit of the inclusion and elastically similar materials.

A general closed form solution for the conformal contacts of two-dimensional elastically similar cylindrical objects under frictionless contact were developed by Ciavarella and Decuzzi [15]. The authors also studied the general case of elastic dissimilarity in a companion paper [16] and developed an approximate closed form solution considering Dundurs' both material parameters (α, β) [17, 18], which can be related to the extent of contact between two bodies and the corresponding stresses.

The analytical solutions developed by Hertz and Persson are used to verify the results of finite element analyses performed in Chapter 4. They are summarized in more detail in the following sections.

2.1.1 Hertz's Solution for Non-conformal Cylindrical Contact

Hertz [10] formulated his theory by assuming each contact body could be considered

as an elastic half-space loaded over a small elliptical region of its plane surface [19]. This assumption allows the contact stresses to be treated separately from the general distribution of stresses in the two bodies. In addition, it is assumed that (i) the contact stresses are smooth and non-conforming, (ii) the strains are small, and (iii) the contact surfaces are frictionless.

The elastic solution of the contact problem can be obtained by assuming a pressure distribution $p(x, y)$ acting on the contact surface between the two elastic half-spaces. For the contact of two cylindrical bodies, the problem can be idealized as a two-dimensional one where the two cylinders contact over a long strip of width $2a$ parallel to the y -axis. The separation between two surface points on the cylinders can be obtained by using a common coordinate system and expressing the contact surface profiles (z_1, z_2) using second order polynomials with coefficients in terms radii of curvature (R_1, R_2) as

$$h = z_1 + z_2 = Ax^2 = \frac{1}{2} \left(\frac{1}{R_1} + \frac{1}{R_2} \right) x^2 = \frac{1}{2} \left(\frac{1}{R} \right) x^2 \quad (2.1)$$

where $1/R = 1/R_1 + 1/R_2$ is the relative curvature. It should be noted that a convex surface has a positive radius and a concave surface has a negative radius in the above equation.

Due to the contact pressure, each body's surface is displaced by $\bar{u}_{z1}$ and $\bar{u}_{z2}$ that are positive value into each body while the distant points of the two bodies move toward the contact point by displacements δ_1 and δ_2 , respectively, such that total displacement $\delta = \delta_1 + \delta_2$. For points within the contact area,

$$\bar{u}_{z1} + \bar{u}_{z2} = \delta - Ax^2 = \delta - \frac{1}{2} \left(\frac{1}{R} \right) x^2 \quad (2.2)$$

and for points outside the contact area,

$$\bar{u}_{z1} + \bar{u}_{z2} > \delta - \frac{1}{2} \left(\frac{1}{R} \right) x^2 \quad (2.3)$$

The surface gradients can be obtained by taking derivative of the above equation as:

$$\frac{\partial \bar{u}_{z1}}{\partial x} + \frac{\partial \bar{u}_{z2}}{\partial x} = - \left(\frac{1}{R} \right) x \quad (2.4)$$

Using the surface gradient expression determined for a pressure $p(x)$ acting on the strip $-a \leq x \leq a$ over an elastic half-space [19],

$$\frac{\partial \bar{u}_{z1}}{\partial x} + \frac{\partial \bar{u}_{z2}}{\partial x} = - \frac{2}{\pi E^*} \int_{-a}^a \frac{p(s)}{x-s} ds \quad (2.5)$$

where E^* is effective modulus of elasticity related to the modulus of elasticity (E_1, E_2) and Poisson's ratio (ν_1, ν_2) of the two contact bodies as

$$\frac{1}{E^*} = \frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \quad (2.6)$$

Using Eqs. 2.4 and 2.5,

$$\int_{-a}^a \frac{p(s)}{x-s} ds = -\left(\frac{\pi E^*}{2R}\right)x \quad (2.7)$$

from which $p(x)$ can be found as

$$p(x) = -\frac{\pi E^*}{2R} \frac{x^2 - a^2/2}{\pi(a^2 - x^2)^{1/2}} + \frac{P}{\pi(a^2 - x^2)^{1/2}} \quad (2.8)$$

As pressure must be positive in the contact region, $P \geq \pi a^2 E^* / (4R)$ for $x \leq a$. Also, the pressure (and the surface gradient) rises to an infinite value at $x = \pm a$ according to the above equation. Since such a deformed profile is inconsistent for the condition that contact should not occur outside the loaded area (Eq. 2.3), it must be concluded that:

$$P = \frac{\pi a^2 E^*}{4R} \quad (2.9)$$

$$a^2 = \frac{4PR}{\pi E^*} \quad (2.10)$$

and the pressure distribution is

$$p(x) = \frac{2P}{\pi a^2} (a^2 - x^2)^{1/2} \quad (2.11)$$

which reduces to zero at the edge of contact area. The maximum contact pressure is given as

$$p_0 = \frac{2P}{\pi a} = \frac{4p_m}{\pi} = \left(\frac{PE^*}{\pi R}\right)^{1/2} \quad (2.12)$$

where p_m is the mean pressure.

For an elastic half-space loaded over a strip ($-b < x < a$) by a normal pressure $p(x)$, the stresses at any point can be obtained as [19]:

$$\sigma_x = -\frac{2z}{\pi} \int_{-b}^a \frac{p(s)(x-s)^2 ds}{[(x-s)^2 + z^2]^2} \quad (2.13)$$

$$\sigma_z = -\frac{2z^3}{\pi} \int_{-b}^a \frac{p(s) ds}{[(x-s)^2 + z^2]^2} \quad (2.14)$$

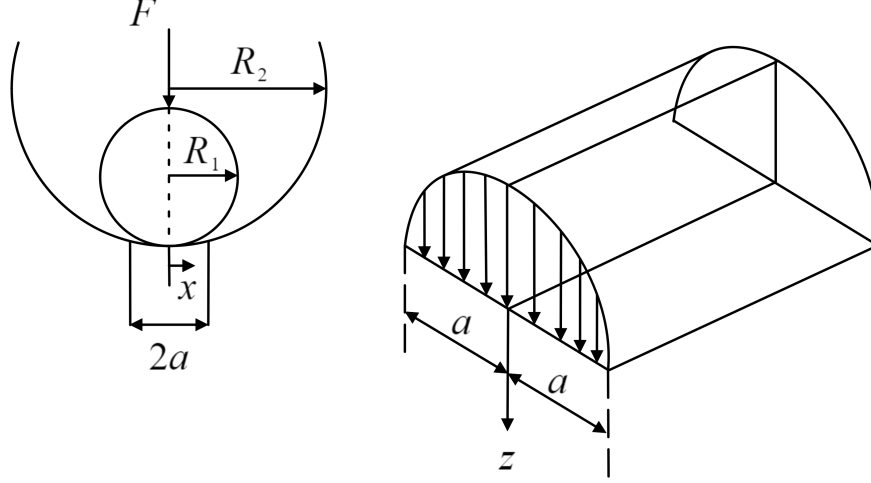


Figure 2.1: Non-conformal contact of two cylinders: cylinder in cylindrical cavity

$$\tau_{xz} = -\frac{2z^2}{\pi} \int_{-b}^a \frac{p(s)(x-s)ds}{[(x-s)^2 + z^2]^2} \quad (2.15)$$

Substituting Eq. 2.11 into the above equations, it is found that $\sigma_x = \sigma_y = -p(x)$ at the contact interface, and all stress components are zero at the surface outside the contact region. The principal stresses and the maximum shear stress along the z -axis can be expressed as:

$$\sigma_x = -\frac{p_0}{a} \left[(a^2 + 2z^2)(a^2 + z^2)^{-1/2} - 2z \right]^2 \quad (2.16)$$

$$\sigma_z = -p_0 a (a^2 + z^2)^{-1/2} \quad (2.17)$$

$$\tau_{xz,max} = \frac{1}{2}(\sigma_x + \sigma_z) = -\frac{p_0}{a} \left[z - z^2(a^2 + z^2)^{-1/2} \right] \quad (2.18)$$

The third principal stress can be calculated by $\sigma_y = \nu(\sigma_x + \sigma_z)$ for plane stress problems.

2.1.2 Persson's Solution for Conformal Cylindrical Contact

When the surfaces of two contacting bodies are closely confirming and the contact region becomes large enough to be comparable with the dimensions of bodies, the contact stresses become part of the general stress distributions within the bodies. In such cases, the contact stresses can no longer be expressed independent of the general stress distribution within the body as in the case of Hertz theory [20]. Steuermann [21] was the first to attempt a solution for the conformal contact of cylinders by rep-

representing the gap by a power series as

$$h = A_1 x^2 + A_2 x^4 + \cdots + A_n x^{2n} \quad (2.19)$$

where A_n are constants. He expressed the displacements in terms of integrals of line influence functions and solved for the contact pressure distribution using finite difference method as:

$$p_n(x) = nE^* A_n a^{2n-2} \times \left[\left(\frac{x}{a} \right)^{2n-2} + \frac{1}{2} \left(\frac{x}{a} \right)^{2n-4} + \cdots + \frac{1 \times 3 \times \cdots \times (2n-3)}{2 \times 4 \times \cdots \times (2n-2)} \right] (a^2 - x^2)^{1/2} \quad (2.20)$$

in which a is the contact half-width given by

$$a = \left(\frac{2 \times 4 \times \cdots \times 2n}{1 \times 3 \times \cdots \times (2n-1)} \frac{P}{n\pi A_n E^*} \right)^{1/2n} \quad (2.21)$$

The above equation reduces to Eq. 2.11 for $n = 1$. This method is more accurate than Hertz theory due to the higher-order terms included in the contact profile, but still in error because of the elastic half-space assumption for both bodies.

Persson [13] developed an analytical solution for conformal contact of cylindrical bodies using a contact criterion identical to Steuermann [21], but he assumed a cylindrical contact surface and expressed the contact criterion as an integrodifferential equation. Persson determined the pressure distribution due to conformal contact of two cylinders having same elastic constants as [13, 20]

$$p(\phi) = \frac{2F(c^2 - q^2)^{1/2}}{\pi R(1 + q^2)(c^2 + 1)^{1/2}} + \frac{F(c^2 + 1)^{-1}}{2\pi R c^2} \times \ln \left[\frac{(c^2 + 1)^{1/2} + (c^2 - q^2)^{1/2}}{(c^2 + 1)^{1/2} - (c^2 - q^2)^{1/2}} \right] \quad (2.22)$$

where $c = \tan(\alpha/2)$ and $q = \tan(\phi/2)$, and F is the applied load, ϕ is the angular position, 2α is the total contact angle and R is the contact radius as shown in Figure 2.2.

The normalized maximum pressure at the center of contact can be expressed as [13, 20]:

$$\frac{Rp_{\max}}{F} = \frac{2c}{\pi(c^2 + 1)^{1/2}} + \frac{\ln [(c^2 + 1)^{1/2} + c]}{\pi c^2(c^2 + 1)} \quad (2.23)$$

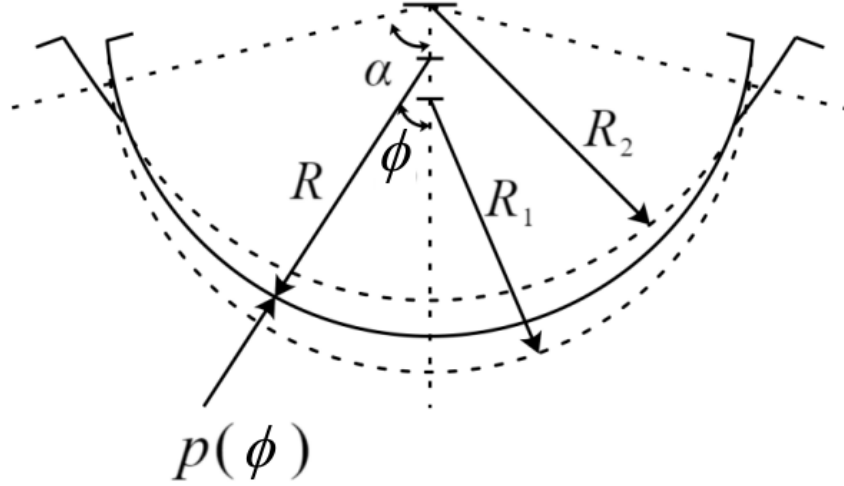


Figure 2.2: Conformal contact of two cylinders

The relationship between the contact half-angle α with the load F , radial clearance $\Delta R (= R_1 - R_2)$, elastic modulus E was derived by Persson [13] as:

$$\frac{E \Delta R}{F} = \frac{2(1 - c^2)}{\pi c^2} - \frac{I}{\pi^2 c^2 (c^2 + 1)} \quad (2.24)$$

where

$$c = \tan\left(\frac{\alpha}{2}\right) \quad (2.25)$$

$$I = \int_{-c}^c \frac{f(c, t)}{1 + t^2} dt \quad (2.26)$$

$$f(c, t) = \ln \left[\frac{(c^2 + 1)^{1/2} + (c^2 - t^2)^{1/2}}{(c^2 + 1)^{1/2} - (c^2 - t^2)^{1/2}} \right] \quad (2.27)$$

The normalized pressure profiles and contact angle relation to normalized load predicted by both Hertz and Persson's analytical solutions are plotted in Figure 2.3 and 2.4, respectively. Persson's solution has a limiting contact half-angle of about 84.83° since it is based on the assumption of a small-displacement linear solution [13, 20].

2.2 Experimental Studies

A comprehensive review of experimental studies on pin connections can be found in Duerr [22]. Pin connections have various failure limit states caused by the contact mechanics of connecting elements. Because of the complex stress distribution, design codes typically use empirical provisions based on experimental studies. Design

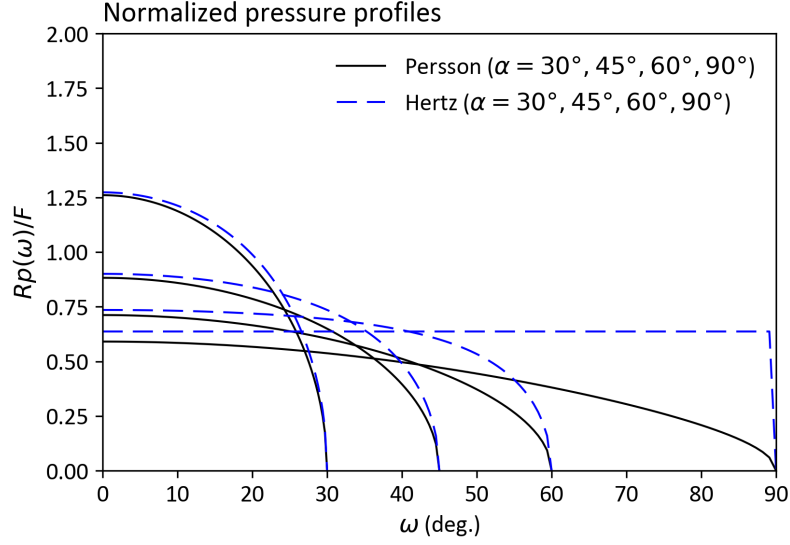


Figure 2.3: Comparison of normalized pressure profiles given by Persson and Hertz

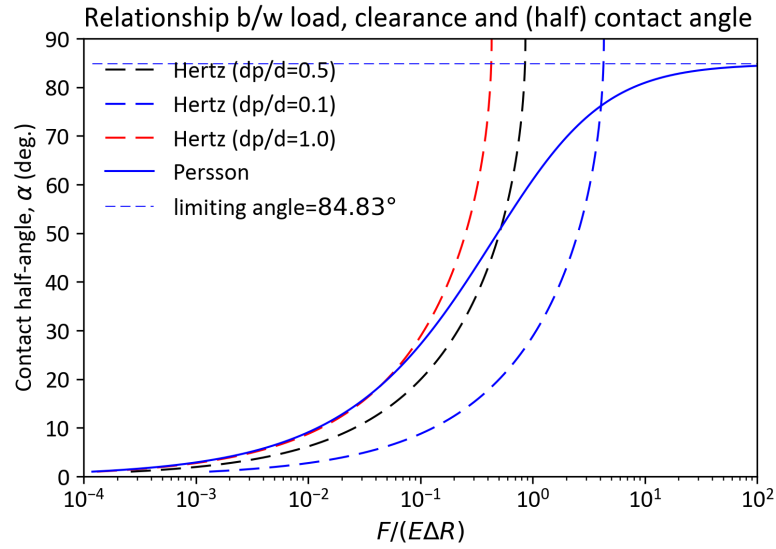


Figure 2.4: Contact angle relation to normalized load

requirements in AISC specifications are mainly based on an extensive test program conducted in 1930s by Johnston [23] and also the results of Blake [24]. For presentation of the previous experimental studies, the uniform notation used by [22] is adapted in this section. The configuration of the connection plate examined in these studies and dimensional parameters are shown in Figure 2.5.

Frocht and Hill [25] determined stress concentration factors (SCFs) for plates loaded through neat-fit pins with hole diameter to plate width ratios ranging from 0.086 to

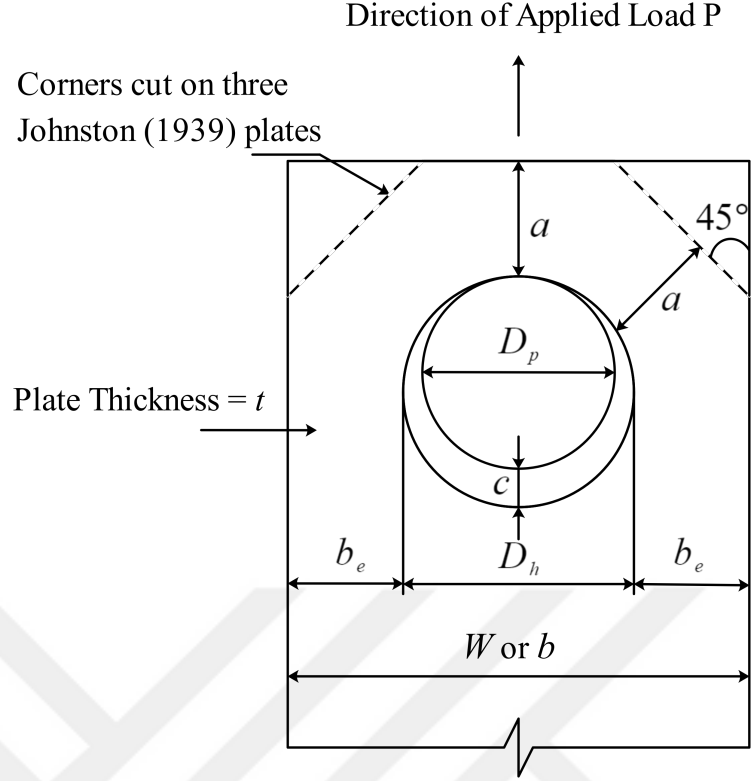


Figure 2.5: Typical pin connection configuration used in experimental studies [23, 24, 22]

0.76. The authors performed two sets of experiments on plates of finite width with central circular holes loaded through pins: one set of experiments using strain gages on aluminum specimens and another set of experiments using photoelastic measurements of small bakelite models.

Scott and Stone [26] studied the effect of clearance on stress concentration using photoelastic specimens of pin-connected lugs and found that the stress concentration factor was sensitive to pin clearance for neat-fit pins. The stress concentration factor was found to have a steep increase between $D_p/D_h \approx 1.00$ and $D_p/D_h \approx 0.98$ for $b_e/D_h = 0.5$. For $D_p/D_h < 0.98$, a further increase in clearance did not have a significant effect. They also found out that the increase from a neat-fit pin to the plateau of the curve is between 10 to 17% for common values of b_e/D_h used in practice.

Duerr [22] provided a comprehensive review of past theoretical and experimental studies of pin connections and examined various test data reported in the literature [23, 24, 27, 28]. He developed a set of equations for pin connection strength and

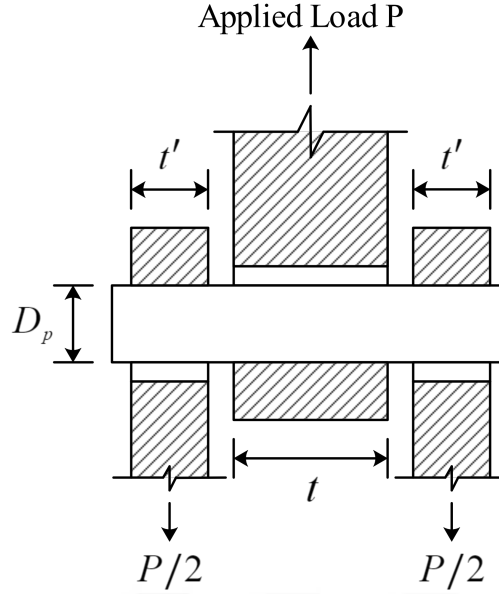


Figure 2.6: Pin loading in a three-plate connection

behavior, which are employed in ASME BTH-1 [29].

Loading in a typical configuration of a three-plate pin connection is shown in Figure 2.6. The most common pin loading model used in practice [30] is based on the assumption that the connection force to be transferred between plates and pin is uniform bearing across the full thicknesses of the plates as shown in Figure 2.7. The resulting shear and bending moment in the pin are then calculated using simple beam theory. This approach provides satisfactory results in many practical design situations. However, the actual bearing stresses are not uniform since bending deflection of the pin and bearing deformation of the plates create a varying (non-uniform) bearing stress distribution with the load concentrated toward the edges of the plates. Melcon and Hoblit [31] presented a pin analysis method that accounts for this behavior. An effective width ($\gamma t/2$) of bearing between the pin and the center plate of a symmetrical connection is defined as illustrated in Figure 2.8. The value of peaking factor γ is a function of the strength of the center plate relative to the applied load and the stiffness of the center plate, expressed as $r = a/t$. γ is determined by using the peaking factor curves given in Melcon and Hoblit [31].

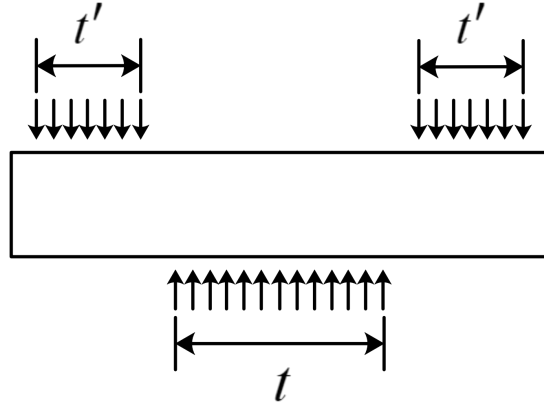


Figure 2.7: Pin loading model by Blake [30]

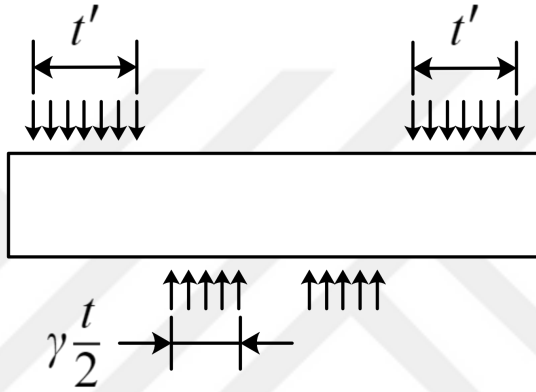


Figure 2.8: Pin loading model by Melcon and Hoblit [31]

2.3 Numerical Studies

Researchers started to employ finite element analysis for investigation of pin connections with the development of finite element method in the later half of the 20th century. Ekvall [28] determined the stress concentration factors as the ratio of the maximum tensile stress in the net section divided by the average bearing stress between the projected area of the pin and the plate hole for the plate configurations shown in Figure 2.9 using finite element analysis. Duerr [22] proposed the following stress concentration factor curve K_t as the ratio of maximum tensile stress to average tensile stress in the net section by using the data obtained from earlier studies (Frocht and Hill [25], Ekvall [28], Scott and Stone [26]):

$$K_t = 1.5 + 2.5 \frac{b_e}{D_h} - 0.27 \left(\frac{b_e}{D_h} \right)^2 \quad (2.28)$$

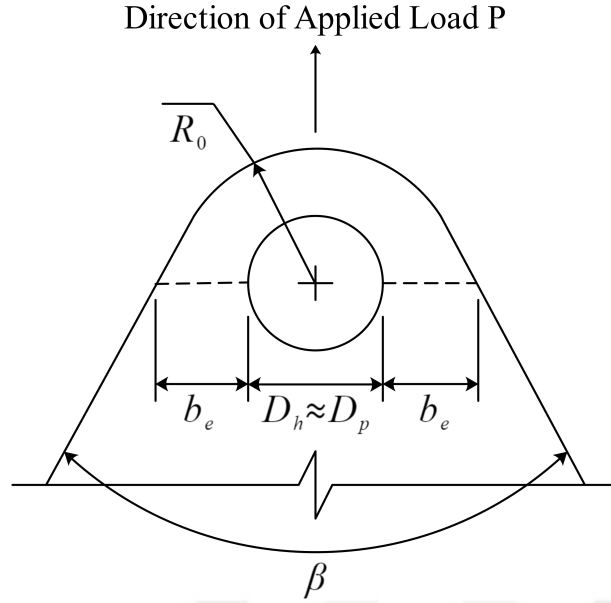


Figure 2.9: Lug configuration in Ekvall [28]

Harms [4] investigated three different geometries of rounded plate model including one which is critical for tension in the net section, one which is critical for fracture beyond the hole, and one which is critical for bearing according to EN 1993-1-8 [2]. The author performed nonlinear finite element analyses for 2D and 3D pin connection models with considered geometries, considering both a serviceability limit state (SLS) and an ultimate limit state (ULS) load as the input load on the finite element models for comparison with design code provisions. He observed that EN 13001-3-1 [32] and ASME BTH-1 [29] were the most accurate compared to the FEA results while considering most geometry parameters in their unity check equations. He also found that the elastic analysis results were sensitive to element types and mesh sizes whereas the plastic analysis provided more similar results for different element types and mesh sizes. His proposed formulas included all relevant geometry parameters (eye radius, eccentricity, pin diameter, hole diameter and thickness) to each limit state and predicted FEA results with an accuracy of 10%.

Fagan and McConnachie [20] examined the frictionless conformal cylindrical contact using finite element analysis of 2D models and compared the FEA results with the analytical solutions of Hertz and Persson. The authors observed that the Hertz theory provided reasonable results for total contact angles of the order of 40° , while the Persson contact theory gave similar results to FEA up to a total contact angle of 150° .

when similar materials were used for both the plate and the pin.

Grant et al. [33] used parametric 2D finite element models to study the elastic stress distribution in pin-loaded lugs. The authors considered various parameters including ratio of lug width (w) to hole diameter (d), the ratio of the distance of the hole center from the free end of the lug (h) to hole diameter (d), and the pin interference/clearance, and examined their effects on the magnitude and position of the maximum circumferential stress around the hole in the lug. They found that interference fit had beneficial effects over clearance fits. They also determined that the lug stresses were not very sensitive to the value of clearance. Later, Grant and Flipo [34] studied the effects of pin interference, amount of material around the hole, and the thickness of lug plate using both 2D and 3D finite element models. The authors concluded that through-thickness effects were significant and a 2D model could underestimate the true stress levels.

2.4 Design Requirements for Pin Connections

The design provisions given in AISC 360-16 [1] specification and EN 1993-1-8 [2] standard for pin connections are summarized in the following sections. It should be noted that the current Turkish Specification for Steel Structures [35] is primarily based on these two documents.

2.4.1 AISC 360-16

AISC 360-16 [1] has provisions for both Load and Resistance Factor Design (LRFD) and Allowable Strength Design (ASD). In LRFD, the required strength under factored (increased) loads should be lower than the design tensile strength ($\phi_t P_n$), where P_n is the nominal tensile strength and ϕ_t is the partial resistance factor. In ASD, the required strength under applied (service) loads should be lower than allowable tensile strength (P_n/Ω_t) where Ω_t is the safety factor. AISC 360-16 checks the pin-connected members against four limit states, which are tensile rupture, shear rupture, bearing and yielding. The lower values of design tensile strength and the allowable tensile strength determined for these limit states control the design.

(a) For tensile rupture on the net effective area:

$$P_n = F_u 2tb_e \quad [\text{AISC Eq. (D5-1)}] \quad (2.29)$$

with $\phi_t = 0.75$ (LRFD) and $\Omega_t = 2.00$ (ASD).

(b) For shear rupture on the net effective area:

$$P_n = 0.6F_u A_{sf} \quad [\text{AISC Eq. (D5-2)}] \quad (2.30)$$

with $\phi_{sf} = 0.75$ (LRFD) and $\Omega_{sf} = 2.00$ (ASD).

where

$A_{sf} = 2t(a + d/2)$ = area on the shear failure path.

a = shortest distance from edge of the pin hole to edge of the member, parallel to the direction of the force.

$b_e = 2t + 0.63$ in. ($b_e = 2t + 16$ mm), but not more than the actual distance from the hole edge to the edge of the part measured in the direction normal to the applied force.

d = diameter of the pin.

t = thickness of the plate.

(c) For bearing on the projected area of the pin:

$$R_n = 1.8 \times F_y A_{pb} \quad [\text{AISC Eq. (J7-1)}] \quad (2.31)$$

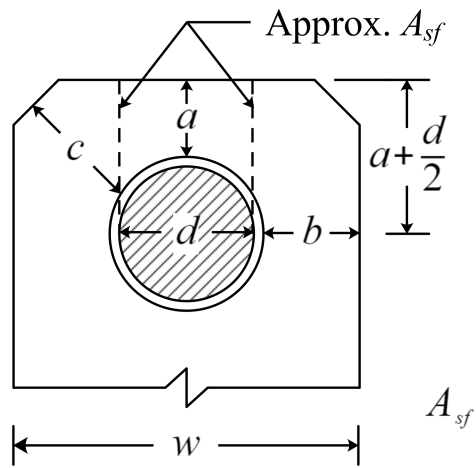
with $\phi_b = 0.75$ (LRFD) and $\Omega_b = 2.00$ (ASD).

(d) For yielding on the gross section:

$$P_n = F_y A_g \quad [\text{AISC Eq. (D2-1)}] \quad (2.32)$$

with $\phi_t = 0.9$ (LRFD) and $\Omega_t = 1.67$ (ASD).

Figure 2.10 shows the dimensional parameters and their minimum requirements for AISC 360-16.



$$a \geq 1.33b_e$$

$$w \geq 2b_e t + d$$

$$c \geq a$$

where

$$b_e = 2t + 0.63 \text{ in. (16 mm)} \leq b$$

Figure 2.10: AISC 360-16 dimensional requirements for pin-connected members

2.4.2 EN 1993-1-8

EN 1993-1-8 [2] has provisions for only Load and Resistance Factor Design (LRFD). Its design criteria for pin connections are given in Table 2.1. The moments in the pin are calculated depending on the basis of that connected steel parts in the pin form simple supports, and the reactions between the pin and the connected steel members are assumed to be uniformly distributed along the length of contact on each part, as illustrated in Figure 2.11.

If the pin is intended to be replaceable, the contact bearing stress should also satisfy:

$$\sigma_{h,Ed} \leq f_{h,Rd} \quad [\text{EN Eq. (3.14)}] \quad (2.33)$$

where

$$\sigma_{h,Ed} = 0.591 \sqrt{\frac{EF_{b,Ed,ser}(d_0 - d)}{d^2 t}} \quad [\text{EN Eq. (3.15)}] \quad (2.34)$$

$$f_{h,Rd} = 2.5f_y/\gamma_{M6,ser} \quad [\text{EN Eq. (3.16)}] \quad (2.35)$$

in which d is the pin diameter, d_0 is the diameter of the pin hole, and $F_{b,Ed,ser}$ is the design force to be transferred in bearing for serviceability limit states.

Table 2.1: Design criteria for pin connections in EN 1993-1-8 [2]

Failure Mode	Design Requirements
Pin shear resistance	$F_{v,Rd} = 0.6A f_{up}/\gamma_{M2} \geq F_{v,Ed}$
Bearing resistance of pin and plate	$F_{b,Rd} = 1.5td f_y/\gamma_{M0} \geq F_{b,Ed}$
This formula also needs to be satisfied if pin is desired to be replaced	$F_{b,Rd,ser} = 0.6td f_y/\gamma_{M6,ser} \geq F_{b,Ed,ser}$
Pin bending resistance	$M_{Rd} = 1.5W_{ef} f_{yp}/\gamma_{M0} \geq M_{Ed}$
This formula also needs to be satisfied if pin is desired to be replaced	$M_{Rd,ser} = 0.8W_{ef} f_{yp}/\gamma_{M6,ser} \geq M_{Ed,ser}$
Pin combined shear and bending resistance	$\left[\frac{M_{Ed}}{M_{Rd}}\right]^2 + \left[\frac{F_{v,Ed}}{F_{v,Rd}}\right]^2 \leq 1$
d = diameter of pin	f_y = lower yield strength of the connected members
f_{up} = ultimate tensile strength of pin	f_{yp} = yield strength of pin
t = thickness of connected part	A = cross section area of pin

The values of partial safety factors (γ) used in the above formulas are listed in Table 2.2. The bending moment in the pin is calculated using Eq. 2.36. The assumed load

Table 2.2: Partial safety factors used in pin connection design

Partial Safety Factor	Values
γ_{M0}	1
γ_{M2}	1.25
$\gamma_{M6,ser}$	1

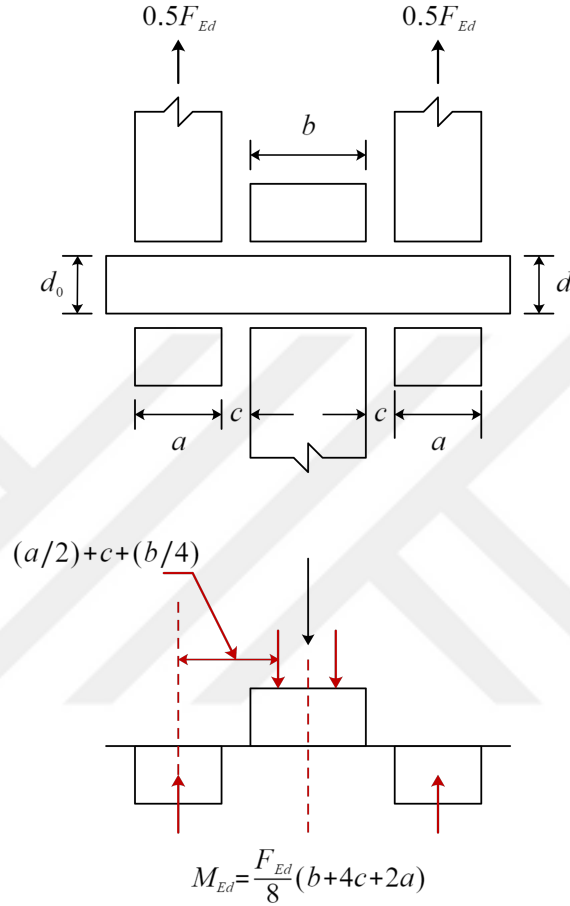
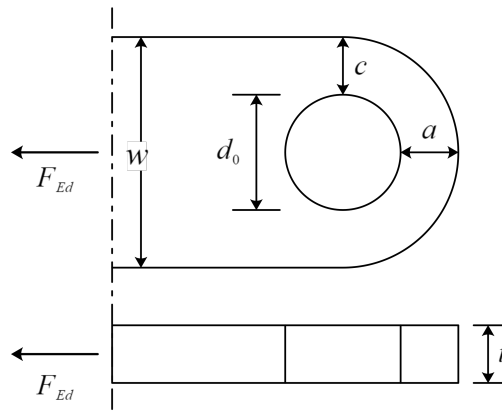


Figure 2.11: Bending moment in a pin

distribution and dimensional variables in the equation are shown in Figure 2.11.

$$M_{Ed} = \frac{F_{Ed}}{8}(b + 4c + 2a) \quad (2.36)$$

EN 1993-1-8 has two sets of geometrical requirements: one is based on plate thickness (called Type A) and another is based on given geometry of the connection plate (called Type B), as shown in Figures 2.12 and 2.13, respectively.



$$a \geq \frac{F_{Ed}\gamma_{M0}}{2tf_y} + \frac{2d_0}{3}$$

$$c \geq \frac{F_{Ed}\gamma_{M0}}{2tf_y} + \frac{d_0}{3}$$

F_{Ed} = design tensile force
 γ_{M0} = partial safety factor

Figure 2.12: EN 1993-1-8 geometrical requirements for pin connections based on plate thickness (Type A)



$$a \geq \frac{F_{Ed}\gamma_{M0}}{2tf_y} + \frac{2d_0}{3}$$

$$c \geq \frac{F_{Ed}\gamma_{M0}}{2tf_y} + \frac{d_0}{3}$$

F_{Ed} = design tensile force
 γ_{M0} = partial safety factor

Figure 2.13: EN 1993-1-8 geometrical requirements for pin connections based on plate geometry (Type B)

CHAPTER 3

MODELING OF CONTACT PROBLEMS

Formulation of contact problems and their modeling by finite element method are described in this chapter. After a brief introduction to general formulation of contact problems and the finite element method, finite element formulation for contact problems is summarized. Further details can be found in Johnston [23], Wriggers and Laursen [36], Kim [37], and Konyukhov and Izi [38]. An overview of Calculix, an open-source finite element analysis (FEA) software, used for the analyses of parametric finite element models is presented at the end of the chapter.

3.1 General Formulation of Contact Problems

3.1.1 Spring-Mass Frictionless Contact System

A simple contact problem can be defined by a spring-mass system where the point mass m suspended on a spring with stiffness k can get in contact with a rigid plane preventing the motion (or the spring deflection u), as shown in Figure 3.1 [36, 38].

The equilibrium of this system before the contact can be described by three equivalent expressions:

- (1) Strong formulation given by the force equilibrium equation as

$$F - mg = 0 \quad \implies \quad ku - mg = 0 \quad (3.1)$$

- (2) Weak or variational formulation by considering the work of forces over a small

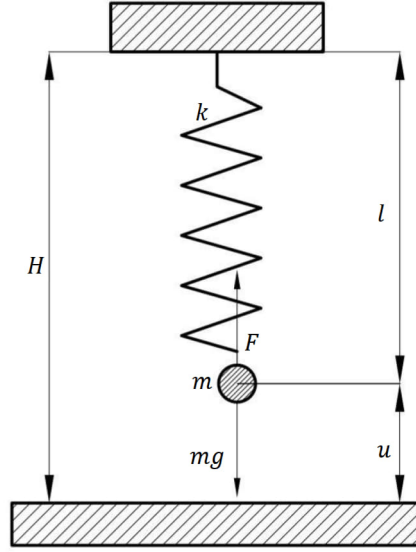


Figure 3.1: Mass-spring system in contact

(virtual) displacement δu (also called the variation of the displacement u).

$$\delta u(ku - mg) = 0 \quad (3.2)$$

The variation operation δ is similar to differentiation. Therefore, integrating the above equation leads to

$$\delta \left(\frac{1}{2}ku^2 - mgu \right) = \delta \Pi = 0 \quad (3.3)$$

where Π refers to the potential energy of the system.

(3) Weak or variational formulation in extremal form

$$\Pi(u) = \frac{1}{2}ku^2 - mgu \rightarrow \min \quad (3.4)$$

where the positive polynomial for the second order for Π indicates that the potential energy reaches its minimum when the necessary condition $\delta \Pi = 0$ is satisfied.

When the mass contacts with the rigid plane, the non-penetration condition can be expressed by the geometrical inequality using the total height H of the constrained spring-mass system and the undeformed length of the spring l as

$$l + u \leq H \quad (3.5)$$

The above equation can be expressed as a penetration function $p(u)$

$$p(u) := l + u - H \leq 0 \quad (3.6)$$

The minimization problem $\Pi \rightarrow \min$ subjected to the restriction of Eq. 3.6 could be solved by optimization methods. Expressing all contact situations in the context of optimization theory, Karush-Kuhn-Tucker conditions (or KKT-conditions) are formed as:

$$\text{contact condition:} \quad p = 0 \text{ and } N > 0$$

$$\text{no contact condition:} \quad p < 0 \text{ and } N = 0$$

$$\text{complimentary condition:} \quad pN = 0$$

where N is the Lagrange multiplier, physically corresponding to the normal contact force.

3.1.2 Contact Formulations

Various contact formulations can be obtained depending on how the inequality conditions are satisfied [38]. The most widely used three methods are described below.

3.1.2.1 Lagrange Multiplier Method

In the Lagrange multiplier method, a Lagrange functional L is constructed as the sum of potential energy Π and constraint equations times the Lagrange multiplier λ [38]. The weak formulation in extremal form is then expressed as

$$L(u, \lambda) = \Pi + \lambda p \rightarrow \min \quad (3.7)$$

The necessary condition for the above requirement is that the partial derivatives with respect to u and λ have to be zero such that:

$$\begin{aligned} \frac{\partial L}{\partial u} = 0 &\Rightarrow \frac{\partial}{\partial u} \left(\frac{ku^2}{2} - mgu \right) + \lambda \frac{\partial}{\partial u} (l + u - H) = 0 \\ \frac{\partial L}{\partial \lambda} = 0 &\Rightarrow p = l + u - H = 0 \end{aligned} \quad (3.8)$$

which lead to:

$$\begin{aligned} ku - mg + \lambda &= 0 \\ l + u - H &= 0 \end{aligned} \tag{3.9}$$

In the above equations, the Lagrange multiplier λ physically represents the normal contact force ($\lambda = N$) and the non-penetration condition $p = 0$ is satisfied exactly. In this method, the Karush-Kuhn-Tucker conditions are expressed as:

1. contact condition:

$$p = 0 \text{ and } \lambda > 0$$

2. no contact condition:

$$p < 0 \text{ and } \lambda = 0$$

3. complimentary condition:

$$p\lambda = 0$$

either ($p = 0$ and $\lambda \neq 0$), or ($p \neq 0$ and $\lambda = 0$)

3.1.2.2 Penalty Method

This method is based on construction of a penalty functional without introducing any extra unknown (i.e., a Lagrange multiplier λ) [38]. This new functional is obtained by adding a penalty functional $W_\varepsilon(p) = \frac{1}{2}\varepsilon p^2(u)$ to the potential energy.

$$\Pi_p(u) = \Pi + \frac{1}{2}\varepsilon p^2(u) \tag{3.10}$$

where ε is the penalty parameter and $p(u)$ is the penetration function. The contact problem is solved when the penalty parameter is increased to infinity ($p \rightarrow \infty$). The penalty functional should in general satisfy the following conditions:

$$1. W_\varepsilon(0) = 0$$

$$2. W_\varepsilon(p) > 0 \text{ and strictly increases for both } p \text{ and } \varepsilon$$

$$3. \lim_{p \rightarrow \infty} W_\varepsilon(p) = 0$$

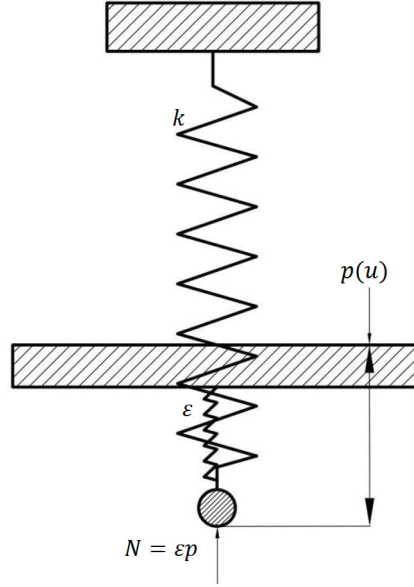


Figure 3.2: Physical explanation of Penalty method

One option to construct a penalty functional fulfilling the above requirements is using Taylor series in various forms, such as

$$W_\varepsilon(p) = \frac{1}{2}\varepsilon p^2 + \varepsilon_4 p^4 + \dots$$

The necessary condition for the minimum value of $\Pi_p(u)$ is

$$\frac{\partial \Pi_p}{\partial u} = ku - mg + \varepsilon(l + u - H) = 0 \quad (3.11)$$

from which the normal contact force N can be interpreted as

$$N = \varepsilon(l + u - H) = \varepsilon p(u) \quad (3.12)$$

The physical interpretation of the penalty method is illustrated in Figure 3.2. Another spring with a stiffness ε is added at the point of penetration to fulfill the non-penetration condition by providing a backward pulling force $N = \varepsilon p(u)$. A spring with infinite stiffness would lead to zero deformation of the additional spring (or zero penetration). This backward pulling force is a contact force that is represented by a spring (with a stiffness ε) attached to the rigid plane and acting on the deflection $p = l + u - H$. There is no further restriction for the non-penetration condition $p = 0$ and thus penetration $p(u) > 0$ is allowed.

3.1.2.3 Augmented Lagrangian Method

The augmented Lagrangian method is an alternative approach to satisfy the non-penetration condition exactly without introducing an additional variable (i.e., Lagrange multiplier λ). This method is based on the following functional:

$$L(u) = \Pi + \lambda p(u) + \frac{1}{2} \varepsilon p^2(u) \quad (3.13)$$

and a special recursive scheme constructed in such a way that the functional depends only on the primary variable u . This results in an recursive scheme to compute the Lagrange multiplier λ in an iterative manner. The scheme consists of two steps:

1. Initialization of the Lagrange multiplier λ :

$$\lambda_0 = 0$$

$$ku_0 - mg + \varepsilon(l + u_0 - H) = 0 \text{ (from Eq. 3.11)}$$

2. Solution of the minimization problem:

$$\frac{\partial L}{\partial u} = \frac{\partial \Pi}{\partial u} + \lambda + \varepsilon p = 0 \quad (3.14)$$

using the following recursive scheme

$$\begin{cases} \lambda_i = \lambda_{i-1} + \varepsilon(l + u_{i-1} - H) \\ ku_i - mg + \lambda_i + \varepsilon(l + u_i - H) = 0 \end{cases} \quad i = 1, 2, 3 \dots \quad (3.15)$$

where i is the augmentation counter. The above equation converges where $\varepsilon > 0$ in the following sense

$$\lim_{i \rightarrow \infty} p(u_i) = 0 \quad (3.16)$$

3.1.3 Formulation in Continuum Mechanics

The contact formulation for the simple spring-mass system can be extended into continuum mechanics problems. First, equilibrium equations are written in strong form and then converted to weak form. Contact conditions are then expressed as inequalities (i.e., Karush-Kuhn-Tucker (KKT) conditions) and the problem is formulated as a constrained optimization problem.

The equilibrium equation (strong form) for a 3D continuum domain with Ω representing the body is written for each point $x = (x^1, x^2, x^3) \in \Omega$ as a set of partial differential equations for stress tensor components σ_{ij} [38]

$$\nabla_i \sigma^{ij} = -f^j, \quad i, j = 1, 2, 3 \quad (3.17)$$

in which f^i is the pressure gradient, such as body force per unit of volume. The Nabla operator ∇_i is the gradient in the Cartesian coordinate system

$$\nabla_i(\dots) \equiv \frac{\partial(\dots)}{\partial x^i}, \quad i = 1, 2, 3$$

Using the Einstein notation,

$$\nabla_i \sigma^{ij} \equiv \nabla_1 \sigma^{1j} + \nabla_2 \sigma^{2j} + \nabla_3 \sigma^{3j} \quad (3.18)$$

The kinematic relations in the case of geometrically linear deformations can be described by the linear infinitesimal strain tensor:

$$\varepsilon_{ij} = \frac{1}{2} (\nabla_i u_j + \nabla_j u_i) \quad (3.19)$$

The constitutive equations for an anisotropic linear elastic material can be written as

$$\sigma^{ij} = A^{ijkl} \varepsilon_{kl}, \quad i, j = 1, 2, 3 \quad (3.20)$$

The following boundary conditions are considered for the boundary value problem (BVP) as illustrated in Figure 3.3,

1. Neumann boundary condition – forces $\sigma_{(n)}^i$ on the boundary surface Γ_σ :

$$\sigma^{ij} n_j = \sigma_{(n)}^i \text{ for } x \in \Gamma_\sigma \quad (3.21)$$

2. Dirichlet boundary condition – fixed or prescribed displacements of the boundary surface Γ_u :

$$u^i = u_0^i \text{ for } x \in \Gamma_u \quad (3.22)$$

The weak form of the equilibrium is obtained by multiplying the strong form equation (Eq. 3.17) with test functions δu_j that satisfy Dirichlet boundary conditions on the part of the surface Γ_u , summing over $j = 1, 2, 3$ and integrating over the domain Ω

$$\int_\Omega \nabla_i \sigma^{ij} \delta u_j d\Omega + \int_\Omega f^j \delta u_j d\Omega = 0 \quad (3.23)$$

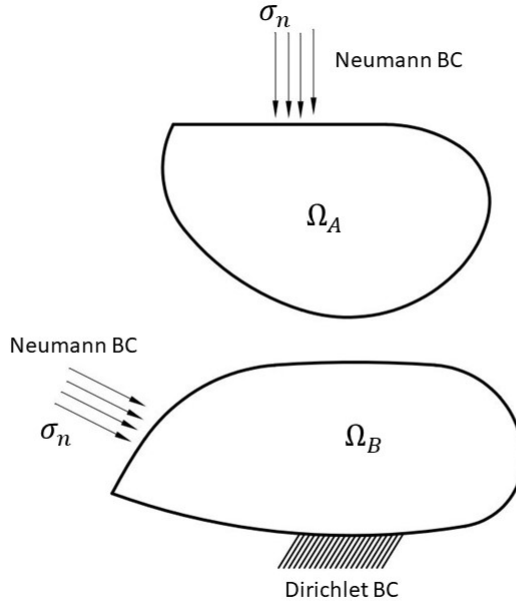


Figure 3.3: Equilibrium condition for a BVP

In the above weak formulation, the equilibrium condition is satisfied only in an average manner in the domain Ω , provided that the test functions are smooth and vanish at Dirichlet boundaries.

After some manipulation, for small deformations Eq. (3.23) can be written as [38]:

$$- \underbrace{\int_{\Omega} \sigma^{ij} \delta \epsilon_{ij} d\Omega}_{\delta W^i} + \underbrace{\int_{\Gamma_{\sigma}} \sigma_{(n)}^k \delta u_k d\Gamma}_{\delta W^{\Gamma_{\sigma}}} + \underbrace{\int_{\Omega} f^l \delta u_l d\Omega}_{\delta W^f} = 0 \quad (3.24)$$

where δW^i represents the variation of the work due to internal stresses, $\delta W^{\Gamma_{\sigma}}$ represents the variation of the work done by boundary forces $\sigma_{(n)}^k$, and δW^f represents the variation of the work done by the volume forces f^l .

Contact conditions are formulated such that a point on the boundary of deformable body (or softer body, called slave) is not allowed to penetrate the rigid body (or stiffer body, called master). This condition can be formulated using the position vectors r for the boundary of the slave body (r_{ref} for the initial undeformed shape) and r_{mas} for the boundary of the master body. After deformation, the current position vector r is updated by including the displacement vector u , such that:

$$\mathbf{r} = \mathbf{r}_{ref} + \mathbf{u} \quad (3.25)$$

The non-penetration condition can be expressed using the normal vector $\mathbf{n}$ on the boundary of the master body. The scalar product $\mathbf{u} \cdot \mathbf{n}$ projects the displacement vector onto the normal direction $\mathbf{n}$ giving the normal component $u_{(n)}$ of the vector u . Similarly, the initial gap $-g$ (g = initial penetration) between the slave and master boundaries along the normal direction can be calculated using the position vectors r and r_{mas} . Thus, the contact condition can be written as:

$$-u_{(n)} - g \leq 0 \quad (3.26)$$

If the initial penetration is zero (i.e., $g = 0$), the bodies are initially in contact and Eq. 3.26 reduces to:

$$-u_{(n)} \leq 0 \quad (3.27)$$

or using the normal vector to the slave boundary ($\mathbf{n}_{sla} = -\mathbf{n}_{mas} = -\mathbf{n}$),

$$u_{(n_{sla})} \leq 0 \quad (3.28)$$

which can geometrically be interpreted as the normal component of the displacement vector u to the contacting surface (i.e., slave) should be less than zero during the deformation with contact (penetration condition).

The above non-penetration condition correlates with the stresses defined to describe the contact conditions in the Karush-Kuhn-Tucker form. The normal force or stress vector $\sigma^{(n)}$ on the contact boundary can be resolved into normal and tangential (shear) components:

$$\sigma^{(n)} = \sigma_n^{(n)} \mathbf{n} + \sigma_\tau^{(n)} \boldsymbol{\tau} \quad (3.29)$$

where $\boldsymbol{\tau}$ is the unit tangential vector. For frictionless problems, the tangential (shear) forces are zero ($\sigma_\tau^{(n)} = 0$) and the KKT conditions then for the deformed (slave) body can be expressed as

$$\text{contact:} \quad u_{(n)} - g = 0 \text{ and } \sigma_n^{(n)} < 0$$

$$\text{no contact:} \quad u_{(n)} - g < 0 \text{ and } \sigma_n^{(n)} = 0$$

$$\text{complimentary condition:} \quad (u_{(n)} - g) \sigma_n^{(n)} = 0$$

The inequality conditions above are satisfied by expanding the weak form in Eq. 3.24 by the virtual work done by the stress vector on the contacting surface.

$$\delta W^i - \delta W^{\Gamma_\sigma} - \delta W^f + \delta W^c = 0 \quad (3.30)$$

where the virtual contact work for the frictionless problems are:

$$\delta W^c = \int_{\Gamma_c} \sigma_{(n)}^i \delta u_i d\Gamma = \int_{\Gamma_c} \boldsymbol{\sigma}^{(n)} \cdot \delta \mathbf{u} d\Gamma = \int_{\Gamma_c} \sigma_n^{(n)} (\mathbf{n} \cdot \delta \mathbf{u}) d\Gamma = \int_{\Gamma_c} \sigma_n^{(n)} \delta u_n d\Gamma \quad (3.31)$$

If the frictional interaction is considered, then the variational work done by the tangential stress component $\sigma_\tau^{(n)}$ should be included as follows:

$$\delta W^c = \int_{\Gamma_c} \sigma_n^{(n)} \delta u_n d\Gamma + \int_{\Gamma_c} \sigma_\tau^{(n)} \delta u_\tau d\Gamma \quad (3.32)$$

To describe the frictional interaction (i.e., a constitutive relationship for the tangential stress component), sticking and sliding cases should be defined using a friction law (e.g., Coulomb friction law) [38].

3.2 Finite Element Method

Many engineering problems can be mathematically modeled as boundary value problems by applying the force equilibrium to an infinitesimal element of the structure [37]. A boundary value problem is a differential equation with prescribed constraints (called boundary conditions) along the boundary of problem domain. It is also called the strong form of the problem. An axial bar subjected to a distributed axial force $p(x)$ is a simple, one-dimensional boundary value problem which can be described by:

$$\begin{aligned} \frac{d^2 u}{dx^2} + p(x) &= 0, \quad 0 \leq x \leq 1 \\ \left. \begin{aligned} u(0) &= 0 \\ \frac{du}{dx}(1) &= 0 \end{aligned} \right\} &\text{Boundary conditions.} \end{aligned} \quad (3.33)$$

The first boundary condition above is an essential boundary condition stating the value of the solution at a given point (e.g., $x = 0$), whereas the second boundary condition is called natural boundary condition giving the value of derivative du/dx at a given point (e.g., $x = 1$).

A more convenient form of the problem can be obtained by application of the principle of virtual work. Multiplying the differential equation by an arbitrary virtual displacement $\bar{u}(x)$, and then integrating over the domain gives:

$$\int_0^1 \left(\frac{d^2 u}{dx^2} + p \right) \bar{u} dx = 0 \quad (3.34)$$

where the arbitrary virtual displacement belongs to the space of kinematically admissible displacements. After taking the function $p(x)$ to the right hand side and applying integration by parts on the left hand side, the order of the differential equation is reduced for $u(x)$, leading to:

$$\left. \frac{du}{dx} \bar{u} \right|_0^1 - \int_0^1 \frac{du}{dx} \frac{d\bar{u}}{dx} dx = - \int_0^1 p \bar{u} dx \quad (3.35)$$

which is called the weak form of the problem.

It is generally difficult to find an analytical solution to boundary value problems. Instead, an approximate solution can be sought in a simple polynomial form expressed as a sum of a number of functions:

$$u(x) = \sum_{i=1}^n c_i \phi_i(x) \quad (3.36)$$

where n is the number of repeated terms, $\phi_i(x)$ is known as trial functions. The coefficients c_i are obtained by minimizing the error between correct and approximate solutions. The accuracy of the solution depends on the trial functions as it is a linear combination of them. The trial functions and coefficients are chosen so that $u(x)$ satisfy the essential boundary conditions of the problem. Such trial functions are called kinematically admissible displacements, $\mathbb{Z}$. Since it is difficult to obtain trial functions that satisfy the essential boundary conditions for the entire domain, the domain is divided into a set of simple sub-domains (finite elements) and the approximate solution is applied at the element level. Therefore, only the elements that include the essential boundary conditions need to have a special treatment. The finite elements are connected with adjacent elements by sharing their nodes and the solution is approximated using piecewise continuous linear (or higher-order) polynomials within each element.

Once the domain is discretized, the integrations in the weak form are performed over each element. For the one-dimensional axial bar problem, element e has two nodes at

$x = x_i$ and $x = x_{i+1}$. A linear polynomial with two unknowns is the appropriate choice to construct a trial function using the solution values at the two end nodes.

$$u(x) = a_0 + a_1x, \quad x_i \leq x \leq x_{i+1} \quad (3.37)$$

where the unknown coefficients a_0 and a_1 can be expressed in terms of nodal solutions $u(x_i)$ and $u(x_{i+1})$. Substituting these nodal values, the following expressions can be obtained:

$$\begin{cases} u(x_i) = u_i = a_0 + a_1x_i \\ u(x_{i+1}) = u_{i+1} = a_0 + a_1x_{i+1} \end{cases} \quad (3.38)$$

where u_i and u_{i+1} are values of the solution at the end nodes. Solving the above equations for the unknown coefficients a_0 and a_1 , and substituting them into Eq. 3.37, lead to the approximate solution given in terms of nodal solutions:

$$u(x) = \underbrace{\frac{x_{i+1} - x}{L^{(e)}}}_{N_1(x)} u_i + \underbrace{\frac{x - x_i}{L^{(e)}}}_{N_2(x)} u_{i+1} \quad (3.39)$$

where $L^{(e)} = x_{i+1} - x_i$ is length of each finite element e . The above equation can be re-written as

$$u(x) = N_1(x)u_i + N_2(x)u_{i+1}, \quad x_i \leq x \leq x_{i+1} \quad (3.40)$$

where $N_1(x)$ and $N_2(x)$ are called interpolation functions (or shape functions).

Finite element approximation to solve the weak form is described below in matrix form for 1D problems. The approximation solution within each element e is given by:

$$u^{(e)}(x) = u_i N_1(x) + u_{i+1} N_2(x) = \mathbf{N}^{(e)} \cdot \mathbf{d}^{(e)} \quad (3.41)$$

where $\mathbf{d}^{(e)} = [u_i, u_{i+1}]^T$ is the nodal solution vector, and $\mathbf{N}^{(e)} = [N_1, N_2]$ is the interpolation vector. It can be noted that the interpolation functions obtain the nodal solution as:

$$u^{(e)}(x_i) = u_i, \quad (3.42)$$

$$u^{(e)}(x_{i+1}) = u_{i+1}, \quad (3.43)$$

where u_i and u_{i+1} are nodal solutions at nodes i and $i + 1$, respectively. Since both the solution $u^{(e)}(x)$ and the virtual displacement $\bar{u}^e(x)$ belong to the same space $\mathbb{Z}$, the

same interpolation functions can be used to approximate the virtual displacement as

$$\bar{u}^{(e)}(x) = \bar{u}_i N_1(x) + \bar{u}_{i+1} N_2(x) = \mathbf{N}^{(e)} \cdot \bar{\mathbf{d}}^{(e)} \quad (3.44)$$

where $\bar{\mathbf{d}}^{(e)} = [\bar{u}_i, \bar{u}_{i+1}]^T$ is a vector of virtual displacements for element e . The derivative of $u^e(x)$ can be obtained as

$$\frac{du^{(e)}}{dx} = u_i \frac{dN_1}{dx} + u_{i+1} \frac{dN_2}{dx} \quad (3.45)$$

which can be expressed in matrix form as:

$$\frac{du^{(e)}}{dx} = \begin{bmatrix} dN_1 & dN_2 \\ dx & dx \end{bmatrix} \begin{Bmatrix} u_i \\ u_{i+1} \end{Bmatrix} = \left[-\frac{1}{L^{(e)}} \frac{1}{L^{(e)}} \right] \begin{Bmatrix} u_i \\ u_{i+1} \end{Bmatrix} = \mathbf{B}^{(e)} \cdot \mathbf{d}^{(e)} \quad (3.46)$$

where $L^{(e)}$ is the element length. The same equation can be utilized to approximate the derivatives of virtual displacement by replacing nodal displacements with virtual nodal displacements. Substituting the Eqs. 3.45 and 3.46 into the weak form (Eq. 3.35) leads to:

$$\bar{\mathbf{d}}^{(e)T} \left[\int_{x_i}^{x_j} \mathbf{B}^{(e)T} \mathbf{B}^{(e)} dx \right] \mathbf{d}^{(e)} = \bar{\mathbf{d}}^{(e)T} \int_{x_i}^{x_j} \mathbf{N}^{(e)T} p(x) dx + \bar{\mathbf{d}}^{(e)T} \begin{Bmatrix} -\frac{du}{dx}(x_i) \\ +\frac{du}{dx}(x_{i+1}) \end{Bmatrix} \quad (3.47)$$

The above variational equation needs to be satisfied for all virtual displacements $\bar{u}^{(e)}(x)$. If the element e does not belong to any boundary, the virtual displacements can be any smooth function. Arbitrary virtual displacements can be represented by selecting arbitrary nodal values of $\bar{u}_i$ and $\bar{u}_{i+1}$. Therefore, if the above equation is rearranged by $\bar{u}_i A + \bar{u}_{i+1} B = 0$, then A and B must vanish individually, from which the following equation is obtained for element e :

$$\left[\int_{x_i}^{x_j} \mathbf{B}^{(e)T} \mathbf{B}^{(e)} dx \right] \mathbf{d}^{(e)} = \int_{x_i}^{x_j} \mathbf{N}^{(e)T} p(x) dx + \begin{Bmatrix} -\frac{du}{dx}(x_i) \\ +\frac{du}{dx}(x_{i+1}) \end{Bmatrix} \quad (3.48)$$

The above equation can be expressed in the matrix form as:

$$[\mathbf{k}^{(e)}] \{\mathbf{d}^{(e)}\} = \{\mathbf{f}^{(e)}\} + \begin{Bmatrix} -\frac{du}{dx}(x_i) \\ +\frac{du}{dx}(x_{i+1}) \end{Bmatrix} \quad (3.49)$$

where $[\mathbf{k}^{(e)}]$ is the element stiffness matrix and $\{\mathbf{f}^{(e)}\}$ is the element force vector defined as:

$$[\mathbf{k}^{(e)}]_{2 \times 2} = \int_{x_i}^{x_{i+1}} \mathbf{B}^{(e)T} \mathbf{B}^{(e)} dx = \frac{1}{L^{(e)}} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad (3.50)$$

$$\{\mathbf{f}^{(e)}\} = \int_{x_i}^{x_{i+1}} p(x) \begin{Bmatrix} N_1(x) \\ N_2(x) \end{Bmatrix} dx \quad (3.51)$$

Combining all element matrix equations via assembly process leads to the matrix equations for the whole structure as

$$\begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & 0 & \dots & 0 \\ k_{21}^{(1)} & k_{22}^{(1)} + k_{11}^{(2)} & k_{12}^{(2)} & \dots & 0 \\ 0 & k_{21}^{(2)} & k_{22}^{(2)} + k_{11}^{(3)} & \dots & 0 \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & 0 & k_{21}^{(NE)} & k_{22}^{(NE)} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ \vdots \\ u_N \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_2^{(2)} \\ f_3^{(2)} + f_3^{(3)} \\ \vdots \\ f_N^{(NE)} \end{Bmatrix} + \begin{Bmatrix} -\frac{du}{dx}(x_1) \\ 0 \\ 0 \\ \vdots \\ +\frac{du}{dx}(x_N) \end{Bmatrix} \quad (3.52)$$

which can be written in compact form as

$$[\mathbf{K}]\{\mathbf{d}\} = \{\mathbf{F}\} \quad (3.53)$$

Since the global matrix $[\mathbf{K}]$ is singular, it cannot be solved directly, but it becomes nonsingular when the boundary conditions are imposed. Either the essential boundary condition or the natural boundary conditions must be specified at the boundaries ($x = x_1$ and $x = x_N$).

The above formulation can also be used for two- and three-dimensional problems. But it is not practical to develop interpolation functions for individual elements having different shapes. Instead, physical elements are mapped to a reference element so that different elements have the same interpolation function. Mapping to the reference element is illustrated for a three-dimensional, 8-node, isoparametric solid element in Figure 3.4. The physical element is defined in $x_1 - x_2 - x_3$ coordinates, whereas the reference element is defined in $\xi - \eta - \zeta$ coordinates. The reference element is a cube

with the length of each being 2 and has the center at the origin. The physical element can have their first node at any corner, but the first node of the reference element should always be located at $(-1, -1, -1)$.

For the isoparametric element, the coordinate and the displacement of the element can be expressed as:

$$\mathbf{x}(\xi) = \sum_{I=1}^8 N_I(\xi) \mathbf{x}_I \quad (3.54)$$

and

$$\mathbf{u}(\xi) = \sum_{I=1}^8 N_I(\xi) \mathbf{u}_I \quad (3.55)$$

where $\mathbf{x}_I$ is the nodal coordinate, $\mathbf{u}_I$ is the displacement vector at node $I = 1, \dots, 8$, and $N_I(\xi)$ is the isoparametric shape function defined as

$$N_I(\xi) = \frac{1}{8} (1 + \xi \xi_I) (1 + \eta \eta_I) (1 + \zeta \zeta_I) \quad (3.56)$$

in which (ξ_I, η_I, ζ_I) are the reference (or natural) coordinate corresponding to node I , as shown in Figure 3.4. The above shape functions are Lagrange interpolation functions (i.e., N_I is equal to 1 at node 1 and 0 at other nodes).

The eight corner nodes of the solid element having eight nodes are mapped into eight corner nodes for the reference element. Similarly, every point in the physical element has a one to one mapping relation in the reference element. This mapping relation is called isoparametric mapping since the same shape functions are used for interpolating both the geometry and solution (displacements).

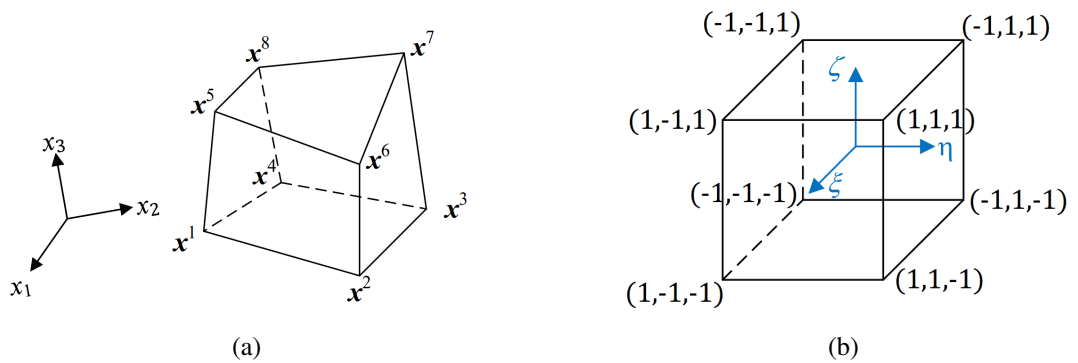


Figure 3.4: 3D isoparametric solid element: (a) finite element, (b) reference element

The mapping between physical and reference (natural) coordinates can be defined using a transformation matrix, called Jacobian. It can be obtained by taking the derivative of Eq. 3.54 as

$$\mathbf{J}_{3 \times 3} = \frac{\partial \mathbf{x}}{\partial \xi} = \sum_{I=1}^8 \mathbf{x}_I \frac{\partial N_I(\xi)}{\partial \xi} \quad (3.57)$$

The importance of Jacobian matrix is to make the derivatives of the shape functions correlating between physical and reference coordinates.

$$\left\{ \begin{matrix} \frac{\partial N_I}{\partial \xi} & \frac{\partial N_I}{\partial \eta} & \frac{\partial N_I}{\partial \zeta} \end{matrix} \right\} = \left\{ \begin{matrix} \frac{\partial N_I}{\partial x_1} & \frac{\partial N_I}{\partial x_2} & \frac{\partial N_I}{\partial x_3} \end{matrix} \right\} \begin{bmatrix} \frac{\partial x_1}{\partial \xi} & \frac{\partial x_1}{\partial \eta} & \frac{\partial x_1}{\partial \zeta} \\ \frac{\partial x_2}{\partial \xi} & \frac{\partial x_2}{\partial \eta} & \frac{\partial x_2}{\partial \zeta} \\ \frac{\partial x_3}{\partial \xi} & \frac{\partial x_3}{\partial \eta} & \frac{\partial x_3}{\partial \zeta} \end{bmatrix} \quad (3.58)$$

or

$$\frac{\partial N_I}{\partial \xi} = \frac{\partial N_I}{\partial \mathbf{x}} \cdot \mathbf{J} \quad (3.59)$$

And the spatial derivatives of the shape functions can be obtained as

$$\frac{\partial N_I}{\partial \mathbf{x}} = \frac{\partial N_I}{\partial \xi} \cdot \mathbf{J}^{-1} \quad (3.60)$$

The derivative of the shape function cannot be evaluated if the Jacobian is zero anywhere in the element. Therefore, the mapping relation is invalid if the Jacobian is zero or negative anywhere in the element.

An integration over physical element domain $\Omega^{(e)}$ can be converted to integration over the reference element as:

$$\iiint_{\Omega^{(e)}} d\Omega^{(e)} = \int_{-1}^1 \int_{-1}^1 \int_{-1}^1 |\mathbf{J}| d\xi d\eta d\zeta \quad (3.61)$$

The formulation requires integration over the domain or over the boundary for the construction of the element stiffness matrix and element force vector. Most integrals cannot be evaluated analytically and it is often faster to integrate them numerically. Gauss integration is commonly used due to its simplicity and accuracy. A general form of Gauss integration can be written as

$$\int_{-1}^1 f(\xi) d\xi \approx \sum_{i=1}^{NG} \omega_i f(\xi_i) \quad (3.62)$$

where N is the number of integration points, ξ_i is the integration point, and ω_i is the nonnegative integration weight. The integration points and weights are selected so that the summation of their products gives an accurate approximation. For integrations over 2D and 3D domains, the Gauss integration can be written as:

$$\int_{-1}^1 \int_{-1}^1 f(\xi, \eta) d\xi d\eta = \sum_{i=1}^{NG} \sum_{j=1}^{NG} \omega_i \omega_j f(\xi_i, \eta_j) \quad (3.63)$$

$$\int_{-1}^1 \int_{-1}^1 \int_{-1}^1 f(\xi, \eta, \zeta) d\xi d\eta d\zeta = \sum_{i=1}^{NG} \sum_{j=1}^{NG} \sum_{k=1}^{NG} \omega_i \omega_j \omega_k f(\xi_i, \eta_j, \zeta_k) \quad (3.64)$$

3.3 Finite Element Formulation of Contact Problems

Finite element formulation for contact problems is described in this section for the simpler case of contact between a flexible body and a rigid body in which case, the rigid body can be assumed to be fixed or having a prescribed motion [37]. The flexible body is chosen as the slave body (since it cannot penetrate the rigid body) and the rigid body is chosen as the master body. In this case, the master body is not governed by equilibrium and the contact variational form is only calculated on the slave boundary.

Figure 3.5 illustrates the contact of two bodies and penetration of a slave node into the master body for which the contact constraint can be defined using a pair that includes a slave node and a straight master segment. Thus, the contact pair can be defined using a slave node and two master nodes (defining the master segment), as $\mathbf{X} = \mathbf{x}_s, \mathbf{x}_1, \mathbf{x}_2^T$.



Figure 3.5: Continuum and discrete contact conditions

There are two objectives for a given contact pair $\mathbf{X}$: (i) find whether the pair is in

contact or separation (i.e., contact search), and (ii) to calculate the contact force and corresponding penetration in case of having contact. In a large contact model, many slave nodes can make contact with many master segments, therefore, the number of contact pairs is huge and significant computational time is spent in search of contact pairs. Once the actual contact pairs are identified, the contact force is calculated for these pairs. The unit normal and tangential vectors for a given contact pair are defined as:

$$\begin{aligned}\mathbf{t} &= \mathbf{x}_2 - \mathbf{x}_1 \\ \mathbf{e}_t &= \frac{\mathbf{t}}{\|\mathbf{t}\|} \\ \mathbf{e}_n &= \mathbf{e}_3 \times \mathbf{e}_t\end{aligned}\tag{3.65}$$

The gap between the slave and master can be expressed as

$$g_n = (\mathbf{x}_s - \mathbf{x}_1)^T \mathbf{e}_n\tag{3.66}$$

If $g_n > 0$, no contact occurs between the pair. If $g_n \leq 0$, it is necessary to check if the contact occur in this master segment (which is the case if the natural coordinate at the contact point is within $0 \leq \xi_c \leq 1$). The natural coordinate at the contact point is calculated as

$$\xi_c = \frac{1}{\|\mathbf{t}\|} (\mathbf{x}_s - \mathbf{x}_1)^T \mathbf{e}_t\tag{3.67}$$

If $\xi_c < 0$ or $\xi_c > 1$, the contact does not occur within this segment. In case of $0 \leq \xi_c \leq 1$, the contact occurs in this master segment, and the contact force then needs to be calculated. The normal contact force, which is proportional to the penetration, is calculated as:

$$\mathbf{f}_n^c = -\omega_n g_n \mathbf{e}_n\tag{3.68}$$

where ω_n is the penalty parameter for the normal contact. Since the rigid body is assumed to be fixed (or having prescribed linear motion), $\mathbf{e}_n$ is also fixed and only the gap needs to be linearized when calculating the tangent stiffness of the contact surface during nonlinear analysis. The contact stiffness can then be determined as

$$\mathbf{k}_n^c = \omega_n \mathbf{e}_n \mathbf{e}_n^T\tag{3.69}$$

In the penalty method, if a boundary region (Γ_c) violates the impenetrability condition (i.e., penetration occurs), then the potential energy is penalized using a penalty function. The contact penalty function for the penetrated region can be defined as

$$P = \frac{1}{2} \int_{\Gamma_c} g_n^2 d\Gamma + \frac{1}{2} \int_{\Gamma_c} g_t^2 d\Gamma \quad (3.70)$$

where ω_n and ω_t are the penalty parameters for normal contact and tangential slip, respectively. The variation of P yields the contact variational form defined by

$$\begin{aligned} b(\mathbf{u}, \bar{\mathbf{u}}) &\equiv \omega_n \int_{\Gamma_c} g_n \bar{g}_n d\Gamma + \omega_t \int_{\Gamma_c} g_t \bar{g}_t d\Gamma \\ &= b_N(\mathbf{u}, \bar{\mathbf{u}}) + b_T(\mathbf{u}, \bar{\mathbf{u}}) \end{aligned} \quad (3.71)$$

Using the slave-master pair and collocation integration (i.e., numerical integration with integration points at element nodes), the boundary integral is approximated by the summation for those violated slave nodes as

$$b_N(\mathbf{u}, \bar{\mathbf{u}}) \approx \sum_{I=1}^{NC} [\bar{\mathbf{u}}^T (\omega_n g_n \mathbf{e}_n)]_I = \sum_{I=1}^{NC} [\bar{\mathbf{u}}^T (-\mathbf{f}_n^c)]_I \equiv \bar{\mathbf{u}}_g^T (-\mathbf{F}_n^c) \quad (3.72)$$

where $\mathbf{F}_n^c$ is the contact force in the global coordinate. During the nonlinear analysis, the contact variational form is linearized as

$$b_N^*(\mathbf{u}; \Delta \mathbf{u}, \bar{\mathbf{u}}) = \sum_{I=1}^{NC} (\omega_n \bar{\mathbf{u}}^T \mathbf{e}_n \mathbf{e}_n^T \Delta \mathbf{u})_I = \sum_{I=1}^{NC} (\bar{\mathbf{u}}^T \mathbf{k}_n^c \Delta \mathbf{u})_I \equiv \bar{\mathbf{u}}_g^T \mathbf{K}_n^c \Delta \mathbf{u}_g \quad (3.73)$$

where $\mathbf{K}_n^c$ is denoted for the global contact stiffness in the global coordinate system, and $\Delta \mathbf{u}$ is the displacement increment. It should be noted that too large values for the penalty parameter can make the matrix ill-conditioned and cause numerical instabilities.

For frictional contact problems, the tangential slip requires information from both the current state and the reference configuration which can be the initial state or the previous time increment. The tangential slip g_t for the straight master segment is defined as

$$g_t = l^0 (\xi_c - \xi_c^0) \quad (3.74)$$

where the superscript 0 refers to the value obtained at the previous time increment, and l^0 is denoted for the master segment length. If the tangential slip is not zero, the

friction force in the penalty method is proportional to the tangential penalty parameter ω_t and can be written as

$$\mathbf{f}_t^c = -\omega_t g_t \mathbf{e}_t \quad (3.75)$$

The above equation represents the stick condition in which the tangential force vanishes as the tangential slip disappears and the two bodies stick together behaving similar to an elastic material. The stick condition can be considered as an elastic deformation before slip occurs. The magnitude of the tangential force is limited by the friction coefficient multiplied by the normal contact force. When the slip is larger than the limit, then the tangential force is limited by

$$\mathbf{f}_t^c = \mu \omega_n \operatorname{sgn}(g_t) g_n \mathbf{e}_t, \quad \text{if} \quad |\omega_t g_t| \geq |\mu \omega_n g_n| \quad (3.76)$$

where μ is the friction coefficient. This linearized friction law shown in Figure 3.6 reduces to the classical Coulomb friction law as $\omega_t \rightarrow \infty$. In case of stick condition, the tangent stiffness of frictional force is given as:

$$\mathbf{k}_t^c = \omega_t \mathbf{e}_t \mathbf{e}_t^T \quad (3.77)$$

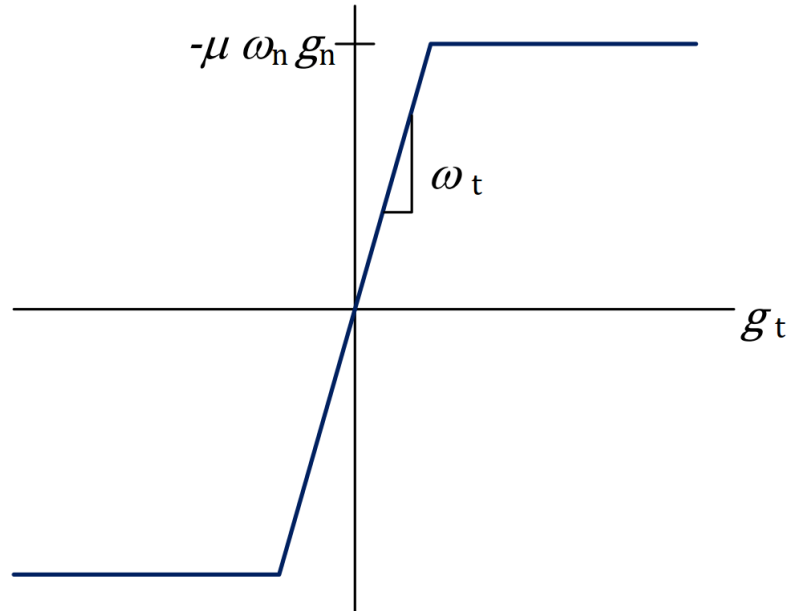


Figure 3.6: Frictional interface model

And the tangent stiffness of the contact frictional force for the slip condition is

$$\mathbf{k}_t^c = \mu \omega_n \operatorname{sgn}(g_t) \mathbf{e}_t \mathbf{e}_n^T \quad (3.78)$$

The tangential slip form can be discretized as

$$b_T(\mathbf{u}, \bar{\mathbf{u}}) \approx \sum_{I=1}^{NC} [\bar{\mathbf{u}}^T (-\mathbf{f}_t^c)]_I \equiv \bar{\mathbf{u}}_g^T (-\mathbf{F}_t^c) \quad (3.79)$$

where $\mathbf{F}_t^c$ is the frictional force in the global coordinate. The tangential slip variational form can be linearized as

$$b_T^*(\mathbf{u}; \Delta \mathbf{u}, \bar{\mathbf{u}}) \approx \sum_{I=1}^{NC} (\bar{\mathbf{u}}^T \mathbf{k}_t^c \Delta \mathbf{u})_I \equiv \bar{\mathbf{u}}_g^T \mathbf{K}_t^c \Delta \mathbf{u}_g \quad (3.80)$$

where $\mathbf{K}_t^c$ is the contact stiffness in the global coordinate.

3.4 Finite Element Analysis Software – Calculix

Calculix is an open source finite element software that can perform linear/nonlinear static analysis, frequency analysis, linear/nonlinear dynamic analysis, buckling analysis, heat transfer, coupled thermo-mechanical analysis, computational fluid dynamics, electromagnetic calculations and some other types of analyses [3]. The software consists of two separate programs: (1) an implicit and explicit solver called CrunchiX (or CCX), and (2) a pre- and post-processor called GraphiX (or CGX). Calculix is originally written for the Linux operating system by Guido Dhondt and Klaus Wittig, ported to the Windows operating system later by others. Calculix version 2.20 is used for the finite element analyses in this study.

3.4.1 Calculix GraphiX (CGX)

GraphiX (CGX) program is developed by Klaus Wittig [39] to generate and display the finite element mesh and also results computed by CCX. The program uses OpenGL library for visualization and glut library for window management and event handling. The program has simple commands to create the model geometry using point, line, surface, and body (volume) entities. The built-in mesher of CGX can then create structured mesh using line elements, quadrilateral shell elements and volume (brick) elements based on the model geometric entities. After a mesh is created, CGX can write the mesh to a text file for use with the solver program CCX. Similarly, CGX has other commands for generation of input data for the solver, such as definition of

boundary conditions, loads, etc., all written to text files to be used by CCX later. After the analysis is completed by CCX, the results data is written to an output file (binary or text) which can be read by CGX for display of the analysis results. CGX has also limited commands for simple calculations and programming with “if” and “while” statements. CGX can also make system calls to issue any shell command (Linux or DOS shell) via the “sys” command. The program can be run either in interactive mode or batch mode. In the interactive mode, all data is input via the graphical user interface (GUI) using interactive commands that start with the letter ‘q’. In batch mode, CGX reads all input commands from a script file. Script files having ‘.fbd’ extension are used for basic geometry commands, whereas it is recommended to use a different extension (e.g., .fbl) for scripts containing more than the basic geometry commands. In this study, ‘.fbd’ extension is used for pre-processing scripts, and ‘.fbl’ extension is used for post-processing scripts.

An example finite element mesh pre-processed by CGX is shown in Figure 3.7, where the model consists of two parts: (1) lug plate, and (2) pin. CGX can also assign different regions of model geometry and mesh into sets (groups) that can be shown separately and in different colors as in Figure 3.7.

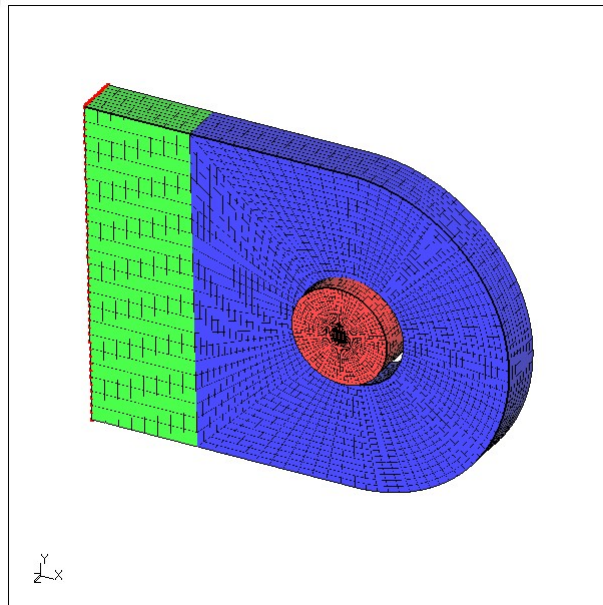


Figure 3.7: Pre-processing with CGX

An example von Mises stress plot of a model post-processed by CGX is shown in Figure 3.8. CGX can also plot two dimensional line graphs using Gnuplot program,

such as deformation or stress values along a path defined by a node set. Due to the parametric nature of this study, many FEA models had to be created, analyzed, and post-processed. Therefore, Python scripts were written to create all input files (.fbd, .inp, .fbl) used by CGX and CCX in batch mode, and also to extract data from CCX output file (.frd) to plot two-dimensional line graphs.

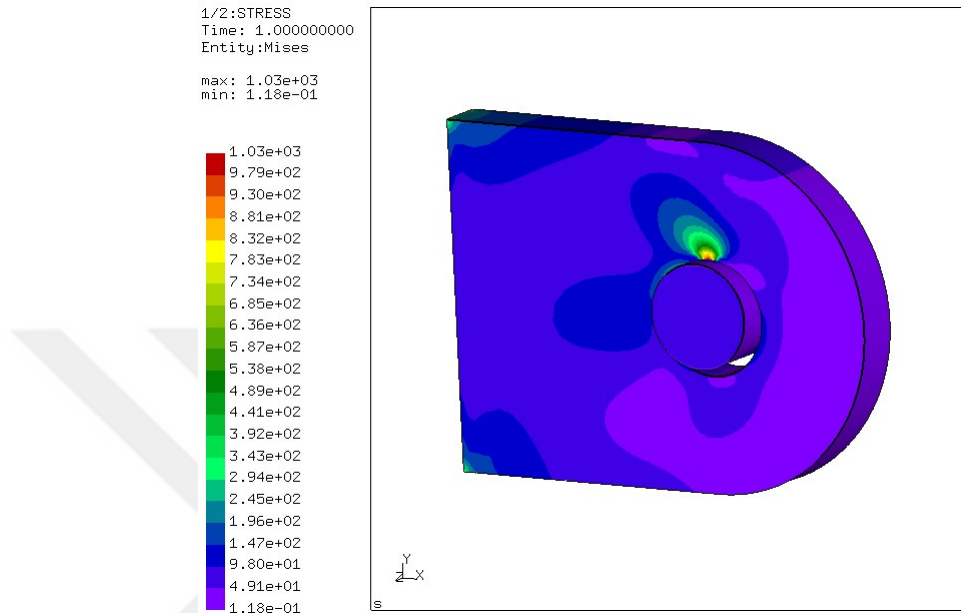


Figure 3.8: Post-processing with CGX

3.4.2 Calculix CrunchiX (CCX)

CrunchiX (CCX) program, which is developed by Guido Dhondt [40], is an implicit and explicit solver. CCX uses an input format similar to Abaqus for most commands. The input data is gathered in a single input file (with the extension '.inp') which can also add further input data for a specific command from other text files using '*INCLUDE' command. The user has to specify which results should be included in the output file (with the extension '.frd'). Upon completion of the analysis, CCX creates the output '.frd' file and CGX can be used to view the analysis results.

It should also be noted that although CCX allows use of 1D and 2D elements, the solver expands them to 3D elements internally and stores the results either in the expanded nodes (default) or in the original nodes.

CCX provides a large selection of element types, including linear (first-order) and

quadratic (second-order) elements in 1D, 2D, 3D with full or reduced integration and incompatible mode. The complete list of available element types can be found in [3]. Selection of the appropriate element type is very important for accuracy of the analysis results. To create a structured mesh of the pin connection, potential candidates are linear/quadratic brick elements C3D8, C3D8R, C3D20, C3D20R for 3D models, and linear/quadratic shell or plane stress elements S4, S4R, S8, S8R, CPS4, CPS4R, CPS8, CPS8R for 2D models.

Reduced integration elements CPS8R and C3D20R are selected to be used in this study, for 2D and 3D models, respectively. The element nodes and reduced integration points for C3D20R are shown in Figure 3.9. 8-node quadratic plane stress element (CPS8R) is expanded to a 3D brick element as shown in Figure 3.10.

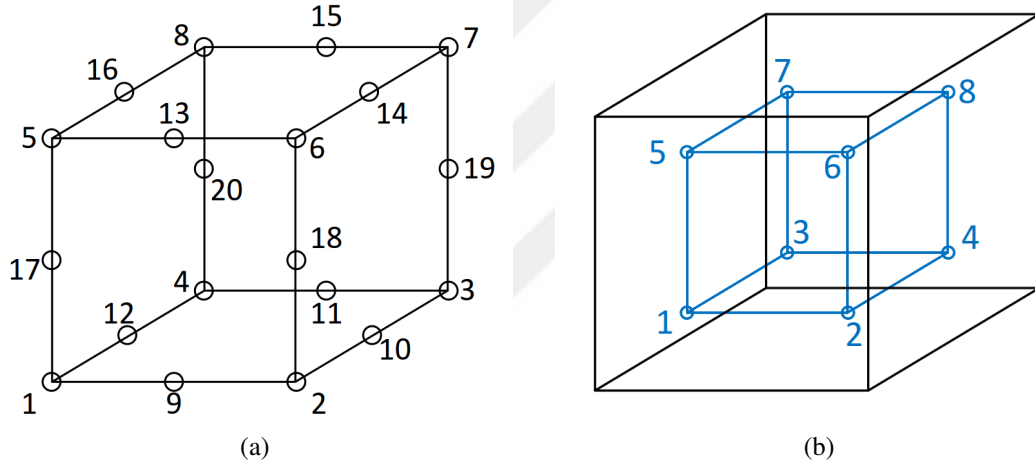


Figure 3.9: C3D20R quadratic brick element: (a) nodes, (b) $2 \times 2 \times 2$ reduced integration points

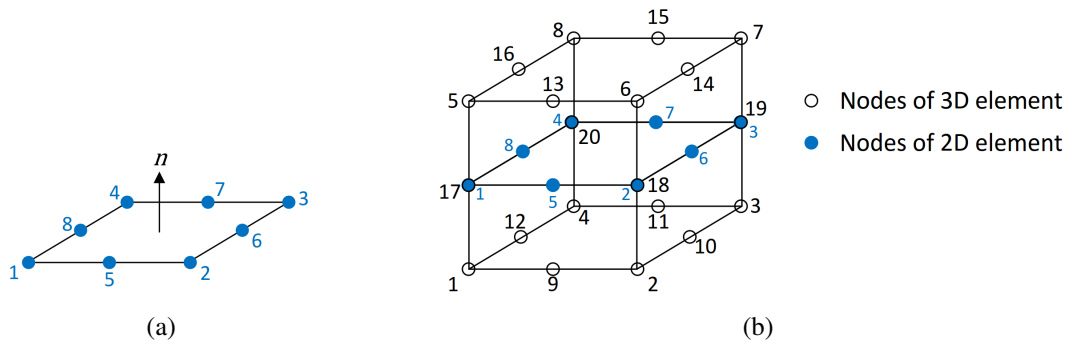


Figure 3.10: Expansion of 2D 8-node element into a 3D brick element: (a) original nodes, (b) expanded nodes

Calculix provides three options for contact modeling: (1) node-to-face penalty con-

tact, (2), face-to-face penalty contact, and (3) face-to-face mortar contact [40]. The first two options use the penalty method whereas the third option uses the Lagrange multiplier method for contact formulation. All three methods are based on a pairwise interaction of two surfaces: a dependent (slave) surface and an independent (master) surface. The dependent (slave) surface may be defined based on nodes or element faces, but the independent (master) surface must be defined by element faces.

In node-to-face penalty contact method, the slave surface consists of nodes whereas the master surface is defined by the element faces. The penalty contact is modeled by the nonlinear spring elements. For each node on the slave surface, a face on the master surface is localized such that it contains the orthogonal projection of the node. If such a face is found a nonlinear spring element is generated consisting of the slave node and all vertex nodes of the master element face. Depending on the element face, the contact spring contains 4, 5, 7, or 9 nodes. If the elements in slave surface have quadratic elements with quadrilateral faces, convergence may be slower as a uniform pressure on a 8-node face leads to compressive forces in the mid-nodes and tensile force in the vertex nodes, usually leading to divergence if the corner nodes belong to a node-to-face contact element. Thus, node-to-face contact is not recommended for quadratic elements.

In face-to-face penalty contact method, both slave and master surfaces are based on element faces. The contact spring element is now applied between an integration point of a slave face and a master face. The contact force at the integration point is subsequently transferred to the nodes of the slave face. This results in contact spring elements connecting a slave face with a master face. It should be noted that the integration points in the slave nodes are not the ordinary Gauss points. Instead, the common polygon areas of the master and slave mesh when put on top of each other are identified and triangulated. For each triangle a 7-node scheme is used which leads to a very smooth pressure distribution.

In face-to-face mortar contact method, Lagrange multipliers are used to model the contact stresses. The stress-strain relationship is satisfied in a weak sense due to the Lagrange multipliers, whereas in the penalty method the penetration uniquely leads to the contact stresses. The convergence of the mortar method is somewhat better (i.e.,

less iterations are needed) than the face-to-face penalty method due to this property. However, the cost of one iteration is higher.

The properties of the contact spring in penalty methods are defined by a ‘*SURFACE INTERACTION’ definition in the CCX input file, followed by ‘*SURFACE BEHAVIOR’ and ‘*FRICTION’ cards to define the interaction properties and friction between the contact surfaces. The pressure-overclosure behavior can be defined as exponential, linear, tabular, tied, or hard in CCX. Hard contact generally leads to slower convergence than soft contact, and a linear contact spring formulation converges faster than the exponential behavior. For a linear contact spring, the pressure-overclosure relationship is given by:

$$p = Kd \left[\frac{1}{2} + \frac{1}{\pi} \tan^{-1} \left(\frac{d}{\epsilon} \right) \right] \quad (3.81)$$

where ϵ is a small number and the term in the square brackets ensures that the value of p is very small for $d < 0$. The pressure curve for $K = 10^3$ and $\epsilon = 10^{-2}$ is plotted in Figure 3.11.

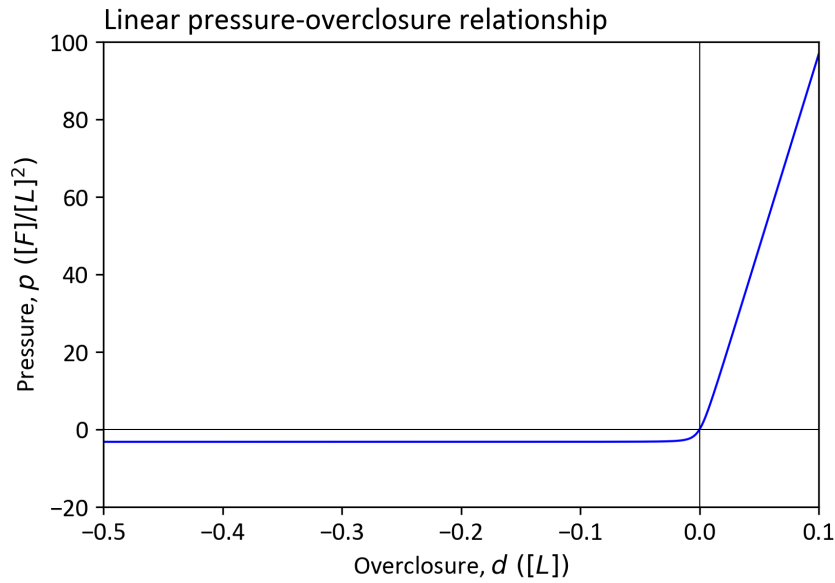


Figure 3.11: Linear pressure-overclosure relationship

As a final note, for selection of the master and slave surfaces, it is generally recommended that: (i) stiffer (or rigid) surface should be master, (ii) convex surface should be slave, and (iii) slave mesh should be at least as fine as (preferably finer than) master mesh.

Based on the discussion above, face-to-face penalty contact formulation with linear pressure-overclosure relationships is used for the finite element models studied in this thesis. Also, the plate hole surface and the pin outer surface are selected as the slave and master surfaces, respectively.



CHAPTER 4

FINITE ELEMENT ANALYSIS OF PIN CONNECTIONS UNDER TRANSVERSE LOAD

In this chapter, parametric finite element models used to study the behavior of pin connections under transverse load are described, and the analysis results for 2D and 3D models are presented. Effects of pin to hole diameter ratio, eccentricity of plate hole (in axial direction), and bending of pin (through pin extension) are examined. The finite element analyses were performed using Calculix.

4.1 Model Description

4.1.1 Model Geometry and Dimensional Parameters

Model geometry and dimensional parameters are illustrated in Figure 4.1. Definitions of dimensional parameters are also listed in Table 4.1. Some of the model parameters (i.e., t , d , c , L and h) are fixed as shown in Table 4.1, while others (i.e., a , d_p and t_e) are varied as shown in Table 4.2. A total number of 9 two-dimensional (2D) models and 27 three-dimensional (3D) models are considered in this study.

The parametric models are labeled using the following format:

pA_DdmX_aYYcZZ_dYYpZZ_tYYpZZ

where the uppercase letters are replaced by the (integer) values used in the models as follows:

A = 90 = loading direction (angle) from horizontal (in degrees)

D = model dimension (2 or 3)

X = model number (2D and 3D models are numbered separately)

YY = value used for dimensional parameters a , d , or t (in mm)

ZZ = value used for dimensional parameters c , d_p , or t_e (in mm) [ZZ value coming after letter ‘p’ refers to the corresponding value for the pin [i.e., pin diameter (d_p) or pin extension length (t_e) on each side of the lug plate]

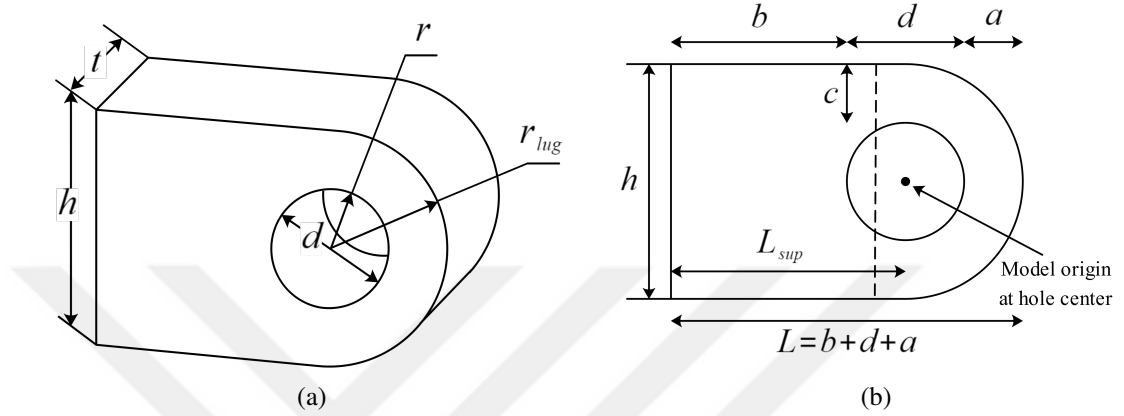


Figure 4.1: Model geometry and dimensional parameters

Table 4.1: Dimensional parameters

$t = 20\text{mm}$	thickness of lug plate
$d = 50\text{mm}$	plate hole diameter
a	edge distance from plate hole in axial direction
$c = 50\text{mm}$	edge distance from plate hole in transverse direction
$L = 4d = 200\text{mm}$	length of lug plate
$h = d + 2c = 150\text{mm}$	depth of lug plate
d_p	pin diameter
t_e	extension of pin beyond the lug plate
$L_p = t + 2t_e$	pin length

Table 4.2: Number of parametric models

Parameter	d_p/d	a/d (or a/c)	t_e/t	# of models
Values	0.999, 0.98, 0.90	1.0, 0.5, 1.5	0.0, 0.25, 0.5	
2D models	3	3	–	9
3D models	3	3	3	27

For example, “p90_2dm1_a50c50_d50p50_t20p0” is the name of 2D model no. 1 with $a = 50$ mm, $c = 50$ mm, $d = 50$ mm, $d_p = 49.95 \approx 50.0$ mm, $t = 20$ mm, and $t_e = 0$ mm. Similarly, “p90_3dm9_a50c50_d50p45_t20p20” label corresponds to 3D

model no. 9 with $a = 50$ mm, $c = 50$ mm, $d = 50$ mm, $d_p = 45$ mm, $t = 20$ mm, and $t_e = 20$ mm. All 2D models and 3D models are listed in Tables 4.3 and 4.4, respectively, along with their dimensional parameters. The models are referred with DdmX portion of the labels (i.e., 2dm1, 3dm1) in the text.

Table 4.3: List of 2D models

Model	a (mm)	c (mm)	d (mm)	d_p (mm)	t_e (mm)
p90_2dm1_a50c50_d50p50_t20p0	50	50	50	49.95	0
p90_2dm2_a50c50_d50p49_t20p0	50	50	50	49.0	0
p90_2dm3_a50c50_d50p45_t20p0	50	50	50	45.0	0
p90_2dm4_a25c50_d50p50_t20p0	25	50	50	49.95	0
p90_2dm5_a25c50_d50p49_t20p0	25	50	50	49.0	0
p90_2dm6_a25c50_d50p45_t20p0	25	50	50	45.0	0
p90_2dm7_a75c50_d50p50_t20p0	75	50	50	49.95	0
p90_2dm8_a75c50_d50p49_t20p0	75	50	50	49.0	0
p90_2dm9_a75c50_d50p45_t20p0	75	50	50	45.0	0

4.1.2 Mesh and Boundary Conditions

The models are discretized using a structured mesh based on given mesh division parameters along certain edges of the model. Mesh division parameters (i.e., nrp, nqh, nze, nr, nq, nx, nz) are shown in Figure 4.2 and listed in Table 4.5 for some models. For 2D models, 8-node plate stress element with reduced integration (CPS8R) is used while 20-node brick element with reduced integration (C3D20R) is used for 3D models. Since both elements are quadratic, the number of elements along an edge of the model (plate or pin part) is half of mesh division parameter.

Sample structured mesh divisions and corresponding mesh for 2D and 3D models are shown in Figures 4.3 and 4.4, respectively. The red nodes in the meshes indicate support nodes, and the paths along 0, 45, 90, 135, 180° directions (which are used to extract stress results) are indicated by blue color in the figure. The origin is defined as the center of plate hole in each model.

The mesh divisions are based on the number of elements used for the radial distance between the pin inner core and outer circumference (given by parameter nrp). The

Table 4.4: List of 3D models

Model	a (mm)	c (mm)	d (mm)	d_p (mm)	t_e (mm)
p90_3dm1_a50c50_d50p50_t20p0	50	50	50	49.95	0
p90_3dm2_a50c50_d50p49_t20p0	50	50	50	49.0	0
p90_3dm3_a50c50_d50p45_t20p0	50	50	50	45.0	0
p90_3dm4_a50c50_d50p50_t20p5	50	50	50	49.95	5
p90_3dm5_a50c50_d50p49_t20p5	50	50	50	49.0	5
p90_3dm6_a50c50_d50p45_t20p5	50	50	50	45.0	5
p90_3dm7_a50c50_d50p50_t20p10	50	50	50	49.95	10
p90_3dm8_a50c50_d50p49_t20p10	50	50	50	49.0	10
p90_3dm9_a50c50_d50p45_t20p10	50	50	50	45.0	10
p90_3dm10_a25c50_d50p50_t20p0	25	50	50	49.95	0
p90_3dm11_a25c50_d50p49_t20p0	25	50	50	49.0	0
p90_3dm12_a25c50_d50p45_t20p0	25	50	50	45.0	0
p90_3dm13_a25c50_d50p50_t20p5	25	50	50	49.95	5
p90_3dm14_a25c50_d50p49_t20p5	25	50	50	49.0	5
p90_3dm15_a25c50_d50p45_t20p5	25	50	50	45.0	5
p90_3dm16_a25c50_d50p50_t20p10	25	50	50	49.95	10
p90_3dm17_a25c50_d50p49_t20p10	25	50	50	49.0	10
p90_3dm18_a25c50_d50p45_t20p10	25	50	50	45.0	10
p90_3dm19_a75c50_d50p50_t20p0	75	50	50	49.95	0
p90_3dm20_a75c50_d50p49_t20p0	75	50	50	49.0	0
p90_3dm21_a75c50_d50p45_t20p0	75	50	50	45.0	0
p90_3dm22_a75c50_d50p50_t20p5	75	50	50	49.95	5
p90_3dm23_a75c50_d50p49_t20p5	75	50	50	49.0	5
p90_3dm24_a75c50_d50p45_t20p5	75	50	50	45.0	5
p90_3dm25_a75c50_d50p50_t20p10	75	50	50	49.95	10
p90_3dm26_a75c50_d50p49_t20p10	75	50	50	49.0	10
p90_3dm27_a75c50_d50p45_t20p10	75	50	50	45.0	10

remaining mesh divisions along the edges of plate and pin are calculated based on the ratio of their length to this reference radial distance. Mesh divisions and the corresponding number of elements and nodes for some models are listed in Table 4.5. n_{rp} is selected as 40 in 2D models and 18 in 3D models based on the mesh sensitivity analyses presented in Section 4.3.1.

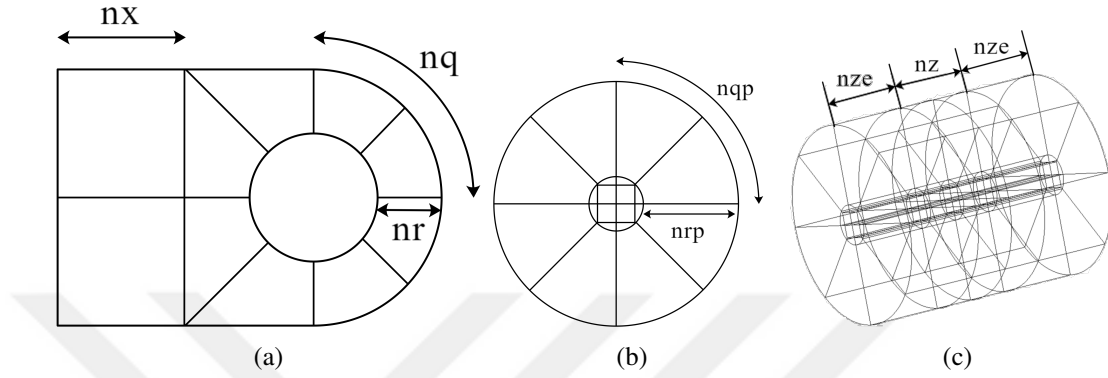


Figure 4.2: Mesh division parameters for 2D and 3D models

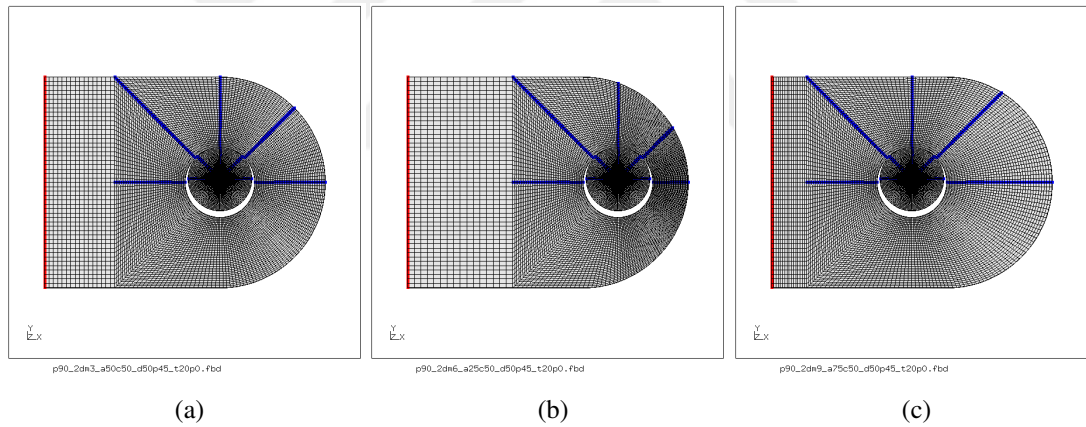


Figure 4.3: Structured mesh for 2D models with $d_p/d = 0.9$: (a) $a/c = 1$, (b) $a/c = 0.5$, (c) $a/c = 1.5$

Table 4.5: Number of elements and nodes used in some models

Model	n_{rp}	n_{qp}	n_{zp}	n_r	n_q	n_x	n_z	# of elements	# of nodes
2dm1	40	80	–	80	96	32	–	13408	40801
3dm1	18	36	16	36	44	16	16	22432	101589
3dm4	18	36	24	36	44	16	16	26608	118733
3dm7	18	36	32	36	44	16	16	30784	135877

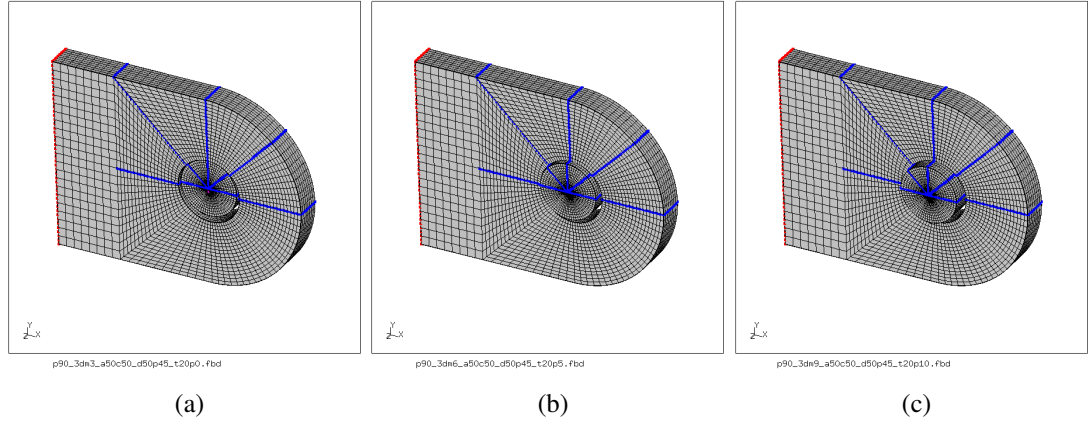


Figure 4.4: Structured mesh for 3D models with $d_p/d = 0.9$ and $a/c = 1$: (a) $t_e/t = 0$, (b) $t_e/t = 0.25$, (c) $t_e/t = 0.5$

4.1.3 Material Properties

In this study, both pin and plate are assumed to have identical linear elastic material properties as listed in Table 4.6.

Table 4.6: Material properties for linear analysis

Model Part	Modulus of Elasticity, E (MPa)	Poisson's Ratio, ν
Plate	210,000	0.3
Pin	210,000	0.3

4.1.4 Loading

The transverse load is applied on the side surfaces of the pin in 3D model as illustrated in Figure 4.5. The half of the load is applied on a reference point at either end of the pin at the center and then is distributed to the element nodes based on the area of elements on those surfaces. In 2D models, the load is applied by defining an acceleration in the loading direction. The nominal stress in the gross cross-sectional area for a general in-plane loading (having an axial and transverse component) can be expressed as

$$\sigma_{n,max} = \frac{P_x}{A} + \frac{M}{S} \quad (4.1)$$

$$\sigma_{n,max} = \frac{P}{A} \left[\cos \theta + \sin \theta \left(\frac{6L_{sup}}{h} \right) \right] \quad (4.2)$$

For transverse loading, $\theta = 90$

$$\sigma_{n,max} = \frac{P}{A} \left(\frac{6L_{sup}}{h} \right) \quad (4.3)$$

The applied transverse load on each model is chosen as 100 kN, which corresponds to a nominal tensile stress of 200 MPa in the gross area of the plate. If S235 grade steel plate having a thickness of 20 mm is used, the yield strength is 225 MPa and the nominal tensile stress of 200 MPa in plate corresponds to about 88.9 percent of the yield strength. Assuming same yield strength for both the plate and the pin, the LRFD code unity checks are calculated in accordance with AISC 360-16 [1] and EN 1993-1-8 [2] and listed in Table 4.7, along with the controlling limit state. It should be noted that these checks are done just to assess the level of loading rather than doing a rigorous connection design. Also, only code check results relevant to the contact region are presented in the table. Support region has actually higher (but still less than 1.0) unity check results for the applied transverse loading.

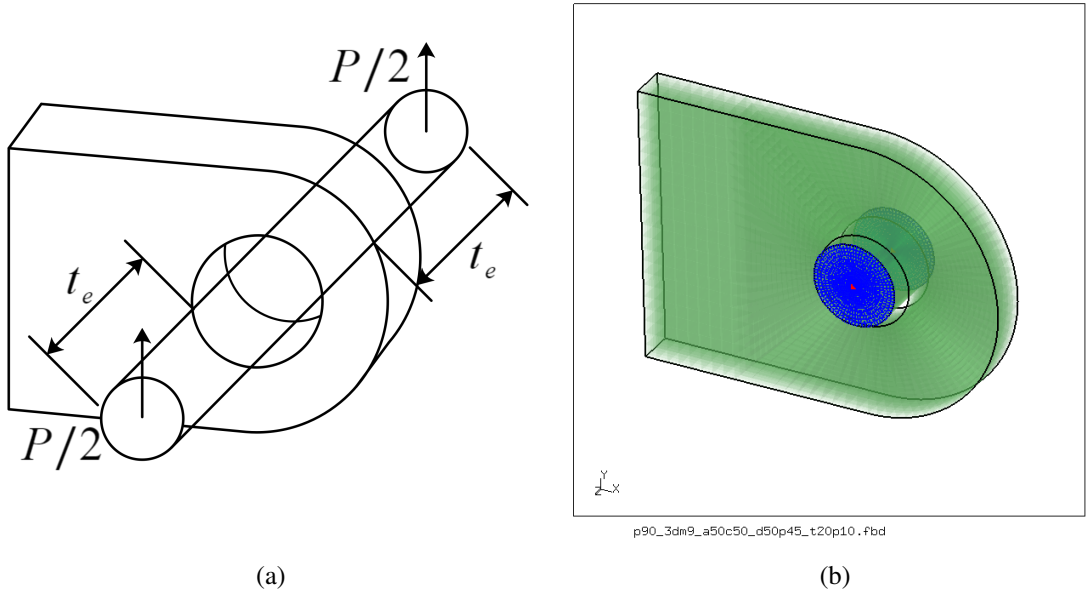


Figure 4.5: Load application in 3D models: (a) model sketch, (b) model mesh

Table 4.7: Applied load and code unity checks for no eccentricity models ($a = c$) with $t_e/t = 0.5$ (assuming a yield strength of $\sigma_y = 225$ MPa for both plate and pin)

$\frac{d_p}{d}$	Model Load (P) (kN)	Max. Nominal Stress (MPa)	LRFD Code Check* (UC)	
			AISC 360-16	EN 1993-1-8
0.999	100	200 (88.9% σ_y)	0.215 (bearing)	0.226 (pin shear)
0.98	100	200 (88.9% σ_y)	0.219 (bearing)	0.235 (pin shear)
0.90	100	200 (88.9% σ_y)	0.239 (bearing)	0.279 (pin shear)

* Only code check results relevant to the contact region are presented here.

4.2 Analysis Procedure

For each model, a nonlinear static contact analysis is performed using Calculix. A master contact surface is defined for the pin outer surface and a slave contact surface is defined for the plate hole surface, as illustrated in Figure 4.6. Since no interference fit is considered in the analyses, only half of the pin and plate hole surfaces are selected for contact search. For contact analysis, face-to-face penalty contact method is used with linear pressure-overclosure relationship. The stiffness of contact springs are sensitive to d_p/d ratio and mesh size. Accordingly, contact stiffness is selected as high as possible while ensuring convergence at a reasonable computation cost, usually within the range of $0.1 E - 5 E$ (where E = modulus of elasticity of the plate and pin).

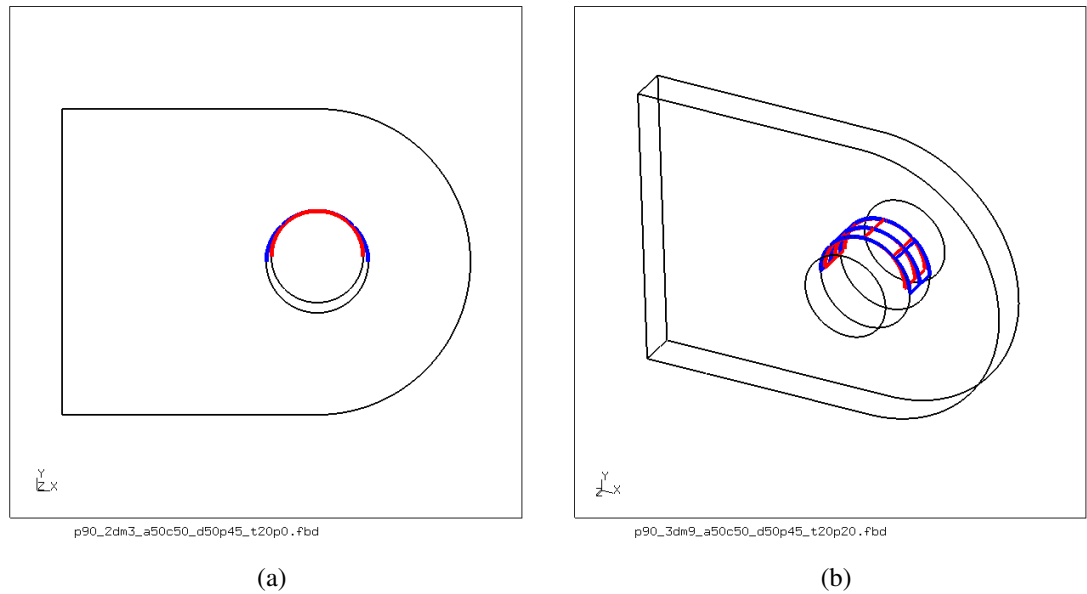


Figure 4.6: Contact surfaces: MASTER (red), SLAVE (blue); (a) 2D model (2dm3), (b) 3D model (3dm9)

4.3 Model Verification

The results of finite element models are verified using three approaches. First, a single 2D model (2dm1) and a single 3D model (3dm1) are analyzed with different mesh sizes to confirm that various result parameters approach to a certain value with increasing mesh size. Second, a 2D quarter model of a cylinder inside a cylindrical cavity is used to compare FEA results with the analytical solutions of Hertz and Persson. And lastly, three of 2D models and three of 3D models are analyzed in Abaqus with same mesh size and element type to compare Calculix results with a commercial FEA software.

4.3.1 Sensitivity Analyses for Mesh Size

Sensitivity analyses for mesh sizes are performed for one 2D model (2dm1) and one 3D model (3dm1). Eleven different mesh sizes are studied for 2D models with *nrp* parameter increasing from 4 to 50. The mesh divisions, corresponding element numbers and maximum contact stress (SY) are tabulated in Table 4.8 for 2D models. As seen in Table 4.8, the maximum SY value is approached to 378.2 MPa with increasing mesh size. Accordingly, the actual 2D models are meshed by selecting the mesh division parameter *nrp* as 40. Eleven different mesh sizes are studied for 3D models with *nrp* parameter increasing from 4 to 24. The mesh divisions, corresponding element numbers and maximum contact stress (SY) are tabulated in Table 4.9 for 3D models. As seen in Table 4.9, the maximum SY value is approached to 388.5 MPa with increasing mesh size. Due to the number of 3D models and computational cost, the actual 3D models are meshed by selecting the mesh division parameter *nrp* as 18. For 3dm1 model, the difference in maximum SY value between *nrp*=18 mesh and *nrp*=24 mesh is only 0.2%. Sensitivity analysis results are presented graphically for three result parameters (i.e., contact angle, normalized maximum pressure (Rp_{max}/F) and maximum penetration) in Figure 4.7 for 2D and 3D models.

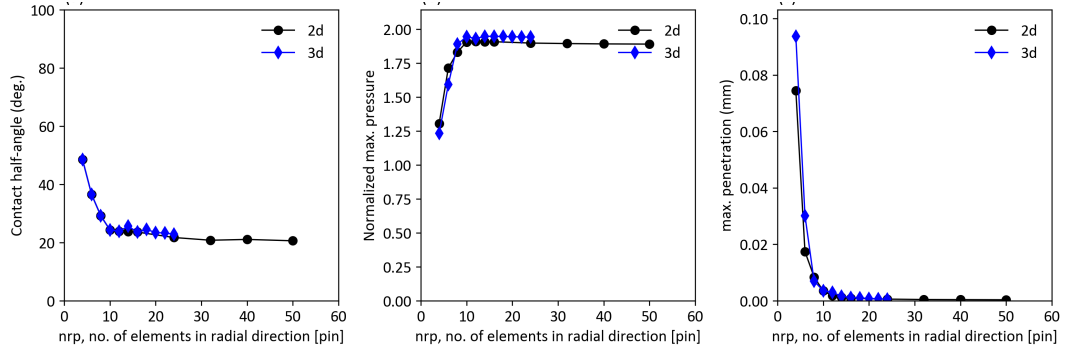


Figure 4.7: Mesh sensitivity results for 2D and 3D models

Table 4.8: Number of elements and nodes used in 2D mesh sensitivity models

Model	nrp (nr)	nqp (nq)	nx	# of elements	SYT (MPa)	Δ (%)
2dsm1	4 (8)	8 (12)	4	172	-261.2	—
2dsm2	6 (12)	12 (16)	6	348	-342.9	31.2%
2dsm3	8 (16)	16 (20)	8	584	-366.5	6.9%
2dsm4	10 (20)	20 (24)	8	868	-381.2	4.0%
2dsm5	12 (24)	24 (32)	12	1344	-381.8	0.2%
2dsm6	14 (28)	28 (36)	12	1760	-381.7	-0.0%
2dsm7	16 (32)	32 (40)	14	2252	-381.7	0.0%
2dsm8	24 (48)	48 (60)	20	5004	-379.7	-0.5%
2dsm9	32 (64)	64 (80)	28	8880	-378.9	-0.2%
2dsm10	40 (80)	80 (96)	32	13408	-378.4	-0.1%
2dsm11	50 (100)	100 (120)	40	20900	-378.2	-0.1%

4.3.2 Comparison with Analytical Solutions

Another two-dimensional model, similar to the one in Fagan and McConnachie [20], is used to compare FEA results with analytical solutions of Hertz and Persson. The model consists of a quarter model of pin inside a half model of a cylindrical cavity as shown in Figure 4.8. This model is selected as it is closer to the infinite two-dimensional elastic half-space assumption in analytical solutions and it allows comparison with the results of Fagan and McConnachie [20].

Eight models are created based on the same hole diameter ($d = 50.0$ mm) used in the actual analysis models and various pin diameters ($d_p = 49.995, 49.95, 49.0, 48.0, 47.0, 46.0, 45.0, 40.0$ mm) with larger values of dimensional parameters a and c (i.e.,

Table 4.9: Number of elements and nodes used in 3D mesh sensitivity models

Model	nrp (nr)	nqp (nq)	nzp (nz)	nx	# of elements	SY (MPa)	Δ (%)
3dsm1	4 (8)	8 (12)	4 (4)	4	344	-246.6	–
3dsm2	6 (12)	12 (16)	8 (8)	6	1392	-318.5	29.1%
3dsm3	8 (16)	16 (20)	8 (8)	8	2336	-378.0	18.7%
3dsm4	10 (20)	20 (24)	8 (8)	8	3472	-389.1	3.0%
3dsm5	12 (24)	24 (32)	12 (12)	12	8064	-386.3	-0.7%
3dsm6	14 (28)	28 (36)	12 (12)	12	10560	-389.7	0.9%
3dsm7	16 (32)	32 (40)	16 (16)	14	18016	-389.5	-0.0%
3dsm8	18 (36)	36 (44)	16 (16)	16	22432	-389.4	-0.0%
3dsm9	20 (40)	40 (48)	16 (16)	16	27136	-389.0	-0.1%
3dsm10	22 (44)	44 (56)	20 (20)	20	42840	-388.7	-0.1%
3dsm11	24 (48)	48 (60)	20 (20)	20	50040	-388.5	-0.1%

$a = c = 600.0$ mm). The thickness of the plate and pin is set to 20 mm. Since the model included half of the cylindrical cavity, half of the load (i.e., $P/2 = 50$ kN) is applied at the center node of the pin corresponding to a distributed thru-thickness load of $F = P/t = 5$ kN/mm.

The mesh for the seventh model with $d_p = 45$ mm (or $d_p/d = 0.90$) is plotted in Figure 4.8(a,b). The normalized load ($F/(E\Delta R)$), contact half-angle (α) and the magnitude of contact stress ($|S_{YY}|$) are listed for three of the models (model no. 2, 3 and 7 having $d_p = 49.95, 49.0, 45.0$ mm, respectively) in Table 4.10, along with the analytical solutions by Hertz and Persson. In addition, the contact half-angle (α) versus the normalized load ($F/(E\Delta R)$) values calculated from FEA results are plotted along with Hertz and Persson solution curves in Figure 4.8. For the third and seventh models with $d_p/d = 0.98$ and $d_p/d = 0.90$, the contact angle is relatively small and therefore both Hertz and Persson solutions are close to the FEA results as seen in Table 4.10 and Figure 4.8(c). For the second model with $d_p/d = 0.999$, the contact angle is relatively large due to conformal contact and therefore Hertz solution has significant error while FEA results are very close to Persson's solution.

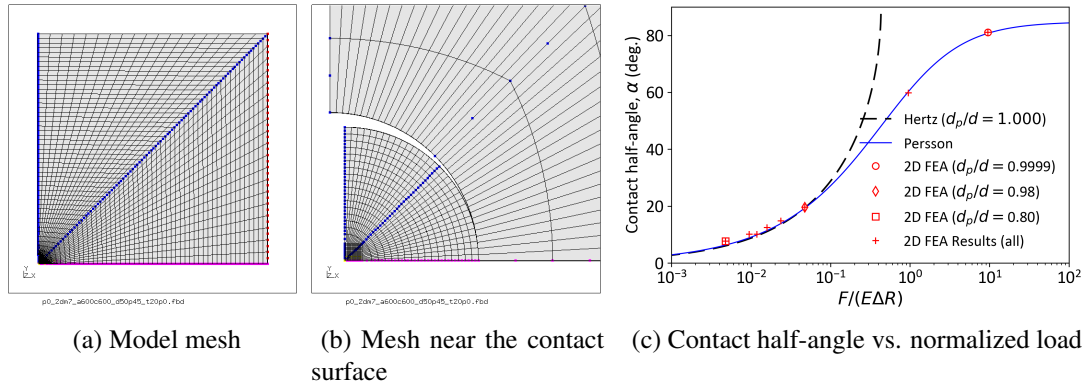


Figure 4.8: 2D quarter model for comparison with analytical solutions

Table 4.10: Comparison of Calculix 2D quarter model results with analytical solutions

Model	d_p (mm)	d (mm)	a, c (mm)	$F/(E\Delta R)$	α (°)	$ SY Y $ (MPa)
CCX					59.8	145.2
2 Hertz	49.95	50.0	600	0.9524	90.0	127.3
Persson					60.2	142.0
CCX					19.5	393.2
3 Hertz	49.0	50.0	600	0.0476	19.2	387.2
Persson					19.3	382.9
CCX					10.1	831.9
7 Hertz	45.0	50.0	600	0.0095	8.1	903.4
Persson					8.9	825.4

4.3.3 Comparison with Abaqus

Three of 2D models (2dm1, 2dm2, 2dm3) and three of 3D models (3dm1, 3dm2, 3dm3) are also modeled and analyzed in Abaqus, to compare Calculix results with a commercial FEA software. The results for maximum contact stress magnitude ($|SY Y|$) and maximum von Mises stress (SVM) are compared with corresponding models in Abaqus in Tables 4.11 and 4.12. The analytical solutions for maximum contact stress ($|SY Y|$) given by Hertz and Persson are also listed in the tables.

The contact stiffness ratio (k_c/E) in Abaqus is adjusted to give a similar maximum penetration value (COPEN) as calculated in Calculix. As can be seen from Tables 4.11 and 4.12, 3D models in Abaqus could be modeled with same contact stiffness values as in Calculix models giving similar COPEN values whereas 2D models required about 20 times higher contact stiffness to have similar COPEN values with Calculix

models. This discrepancy is most likely due to the conversion of 2D elements to 3D elements internally in Calculix.

Table 4.11: Comparison of Calculix 2D model results with Abaqus

Model		$\frac{k_c}{E}$	COPEN (mm)	SY Y (MPa)				SVM (MPa)	
				Plate	Pin	Hertz	Persson	Plate	Pin
2dm1	CCX	4.5	1.59e-4	132.9	150.6	85.7	140.4	283.0	133.4
	Abaqus	90	1.59e-4	132.9	149.5			287.9	149.5
	CCX Δ	–	–	+0.0%	+0.7%	+55.1%	-5.3%	-1.7%	-10.8%
2dm2	CCX	4.0	3.99e-4	378.5	377.8	387.2	383.6	307.0	290.1
	Abaqus	88	4.03e-4	372.9	372.0			341.8	371.5
	CCX Δ	–	–	+1.5%	+1.6%	-2.3%	-1.3%	-10.2%	-21.9%
2dm3	CCX	3.0	1.39e-3	888.1	883.8	903.4	906.4	582.3	604.4
	Abaqus	59	1.38e-3	860.5	856.1			813.8	849.2
	CCX Δ	–	–	+3.2%	+3.2%	-1.7%	-2.0%	-28.5%	-28.8%

Results presented in Table 4.11 for 2D models indicate that Calculix maximum |SY Y| results are 0.0% to 3.2% higher than Abaqus results, and the difference increases with the pin-to-hole diameter ratio. On the other hand, Calculix maximum SVM results are 1.7 to 28.8% less than the corresponding values in Abaqus. This discrepancy is again most likely due to the conversion of 2D elements to 3D elements internally in Calculix. The von Mises stress results obtained by Calculix and Abaqus for model 2dm3 (with $d_p/d = 0.90$) are compared graphically in Figure 4.9. In Table 4.11, Calculix maximum |SY Y| results are also compared with the maximum contact stress given by analytical solutions of Hertz and Persson. For the model with neat-fit pin (2dm1), Calculix maximum |SY Y| result in the plate is 55.1% higher than Hertz's solution and 5.3% less than Persson's solution. For the larger clearance models (2dm2, 2dm3), Calculix maximum |SY Y| result in the plate is 1.7 to 2.3% less than Hertz's solution and 1.3 to 2.0% less than Persson's solution.

For 3D models, the results presented in in Table 4.12 indicate much better match between Calculix and Abaqus models for both |SY Y| and SVM values. Calculix maximum |SY Y| results are within -0.6% and +0.9% of Abaqus results, while Calculix maximum SVM results are within -1.2% and +1.4% of the corresponding values in Abaqus. The von Mises stress results obtained by Calculix and Abaqus for model 3dm3 (with $d_p/d = 0.90$) are compared graphically in Figure 4.10. In Table 4.12, Calculix maximum |SY Y| results are also compared with the maximum contact stress

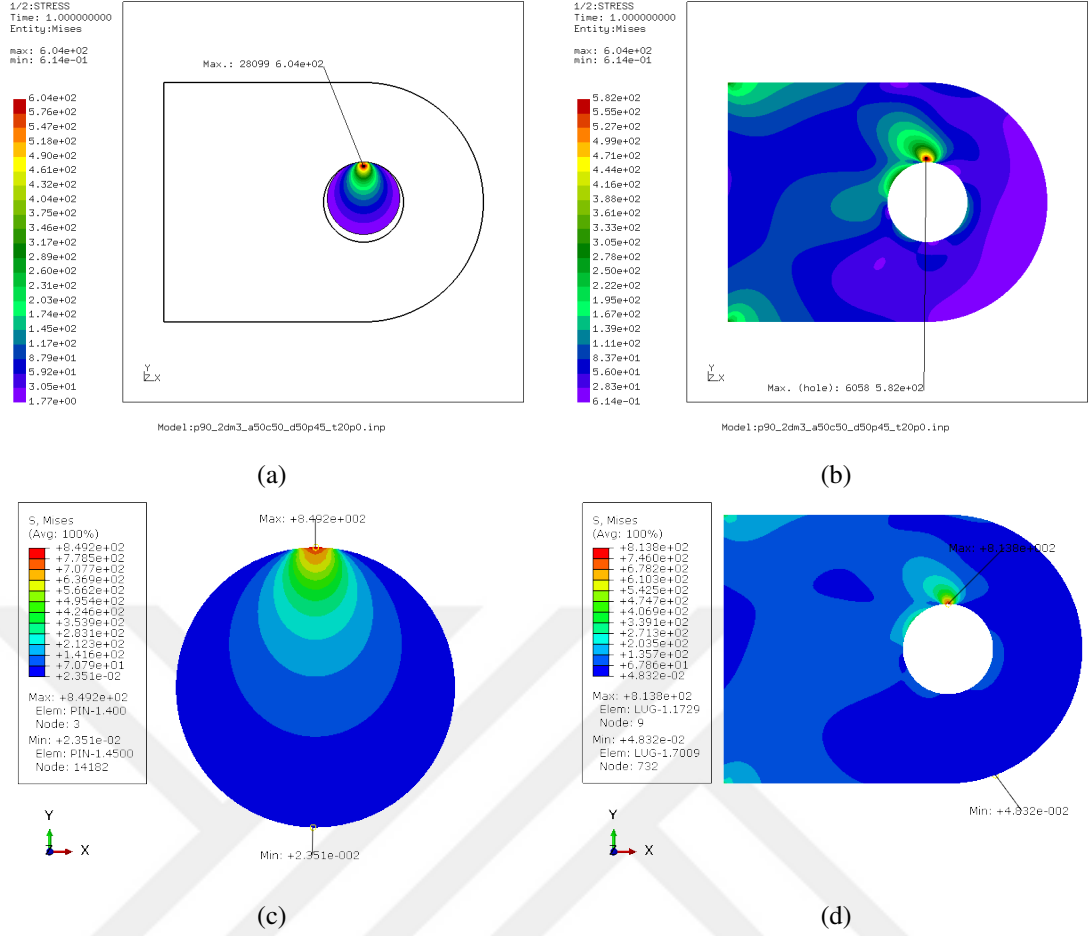


Figure 4.9: von Mises stresses for 2D model with $d_p/d = 0.9$ and $a/c = 1$: (a,b) Calculix, (c,d) Abaqus [model: 2dm3]

given by analytical solutions of Hertz and Persson. For the model with neat-fit pin (3dm1), Calculix maximum $|S_{YY}|$ result in the plate is 86.6% higher than Hertz's solution and 13.9% higher than Persson's solution. For the larger clearance models (2dm2, 2dm3), Calculix maximum $|S_{YY}|$ results in the plate are within -1.5% and $+0.6\%$ of Hertz's solution and within -1.8% and $+1.5\%$ of Persson's solution.

4.4 Results of Two-Dimensional Finite Element Models

Two-dimensional (2D) models are analyzed for different values of the parameters d_p and a . The von Mises stress results are shown graphically in Figures 4.11 through 4.19 for the first three models (2dm1, 2dm2, 2dm3) with $a/c = 1$ and in Figures 4.20 through 4.28 for the next three models (2dm4, 2dm5, 2dm6) with $a/c = 0.5$. For the

Table 4.12: Comparison of Calculix 3D model results with Abaqus

Model		$\frac{k_c}{E}$	COPEN (mm)	SYY (MPa)				SVM (MPa)	
				Plate	Pin	Hertz	Persson	Plate	Pin
3dm1	CCX	1.8	4.64e-4	159.9	162.1	85.7	140.4	290.0	151.5
	Abaqus	1.8	4.88e-4	158.8	160.6			289.1	149.4
	CCX Δ	–	–	+0.7%	+0.9%	+86.6%	+13.9%	+0.3%	+1.4%
3dm2	CCX	1.8	1.02e-3	389.4	386.9	387.2	383.6	328.0	309.1
	Abaqus	1.8	1.02e-3	391.2	387.2			329.8	309.2
	CCX Δ	–	–	-0.5%	-0.1%	+0.6%	+1.5%	-0.5%	-0.03%
3dm3	CCX	0.7	5.88e-3	889.8	884.5	903.4	906.4	665.2	669.5
	Abaqus	0.7	5.98e-3	894.9	888.0			671.8	677.8
	CCX Δ	–	–	-0.6%	-0.4%	-1.5%	-1.8%	-1.0%	-1.2%

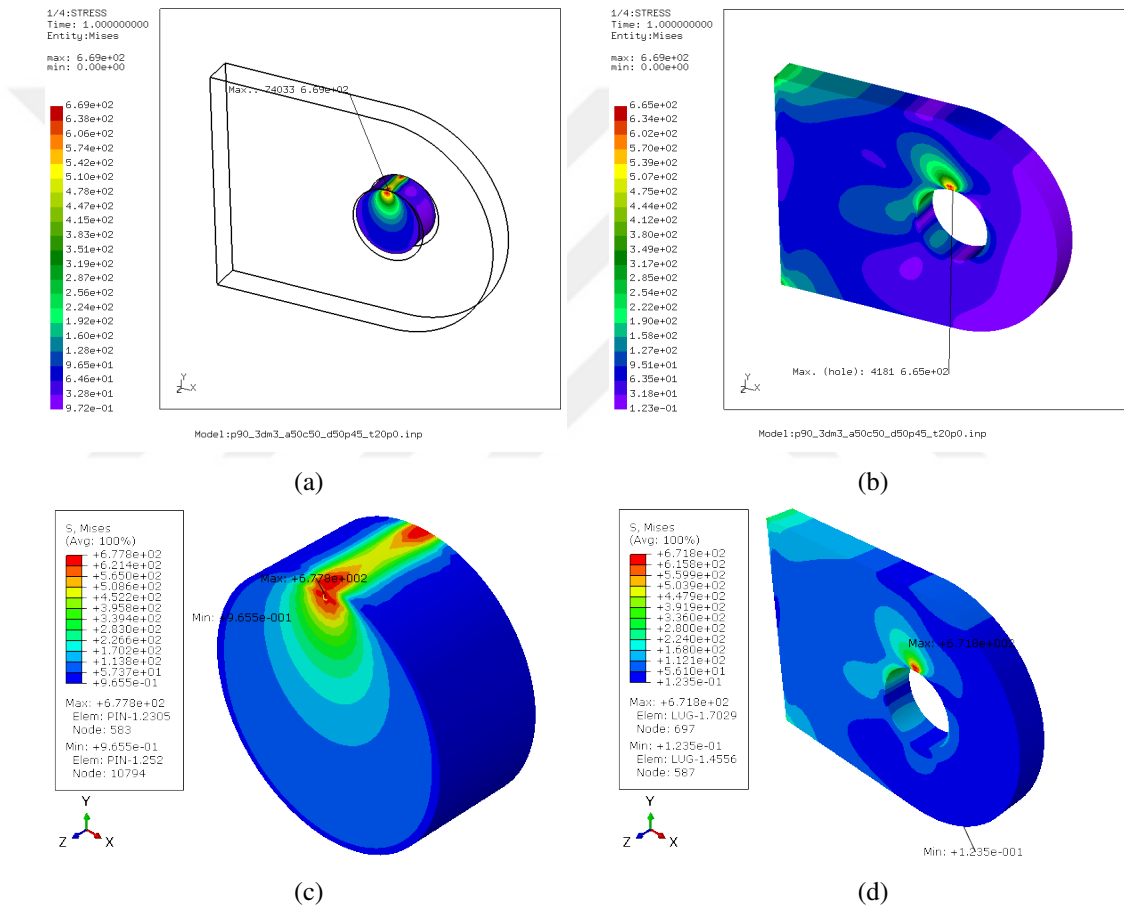


Figure 4.10: von Mises stresses for 3D model with $d_p/d = 0.9$, $a/c = 1$., and $t_e/t = 0$ (a,b) Calculix, (c,d) Abaqus [model: 3dm3]

last three models (2dm7, 2dm8, 2dm9), only von Mises stress contour plots are presented in Figures 4.29 through 4.31. Stress results for all 2D models are summarized in Table 4.13, along with their direction with respect to the model origin (center of the plate hole) and the location of maximum stress from the contact surface. It should

be noted that in some models maximum stress in the plate occurs near the support, however only the maximum stress near the contact region (hole) is reported in Table 4.13.

The von Mises stress contour plots for model 2dm1 with $d_p/d = 0.999$ and $a/c = 1$ are shown for the pin and the plate in Figure 4.11. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the pin and the plate (starting from the contact surface) are presented in Figure 4.12, along with the direction at which von Mises stress has maximum value. For model 2dm1, maximum von Mises stress occurs at 114.7° in the pin and at 146.2° in the plate. The variation of von Mises stress along the circumference of the pin and the hole is shown in Figure 4.13.

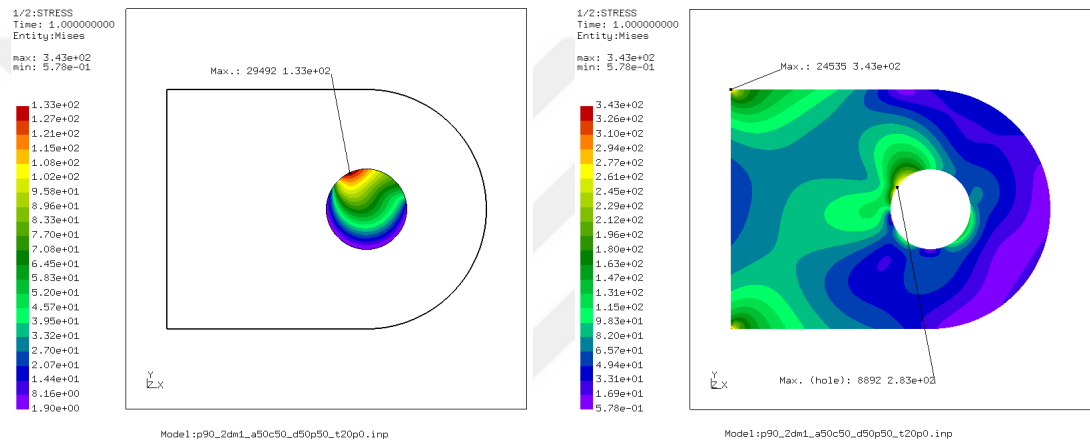


Figure 4.11: von Mises stress results for 2D model with $d_p/d = 0.999$ and $a/c = 1$ [model: 2dm1]

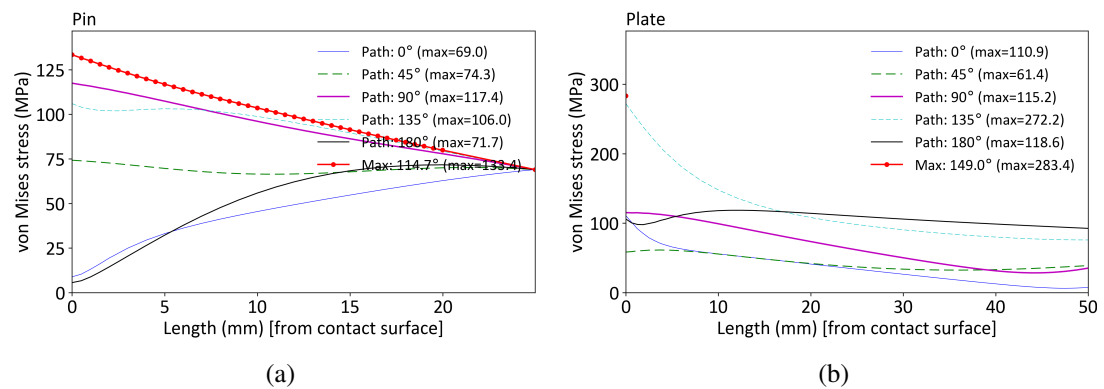


Figure 4.12: von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.999$ and $a/c = 1$: (a) pin, (b) plate [model: 2dm1]

The von Mises stress contour plots for model 2dm2 with $d_p/d = 0.98$ and $a/c = 1$ are shown for the pin and the plate in Figure 4.14. The variation of von Mises stress

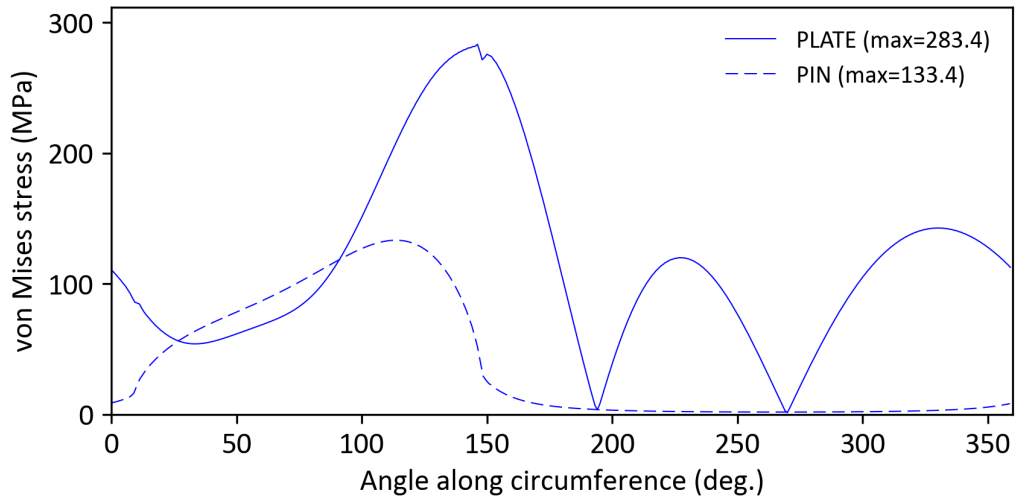


Figure 4.13: von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.999$ and $a/c = 1$ [model: 2dm1]

in radial directions at 0, 45, 90, 135, 180° in the pin and the plate (starting from the contact surface) are presented in Figure 4.15, along with the direction at which von Mises stress has maximum value. For model 2dm2, maximum von Mises stress occurs at 90.0° in the pin and at 140.6° in the plate. The variation of von Mises stress along the circumference of the pin and the hole is shown in Figure 4.16.

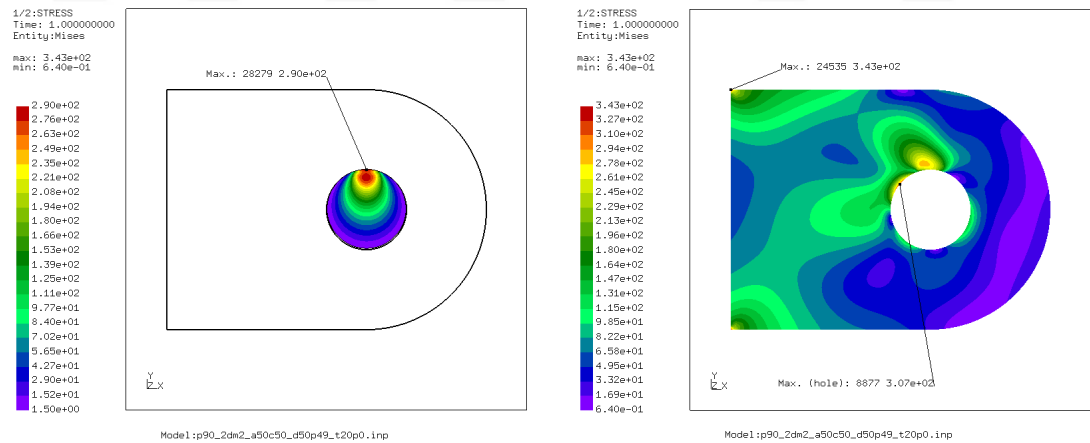


Figure 4.14: von Mises stress results for 2D model with $d_p/d = 0.98$ and $a/c = 1$ [model: 2dm2]

The von Mises stress contour plots for model 2dm3 with $d_p/d = 0.90$ and $a/c = 1$ are shown for the pin and the plate in Figure 4.17. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the pin and the plate (starting from the contact surface) are presented in Figure 4.18, along with the direction at which von

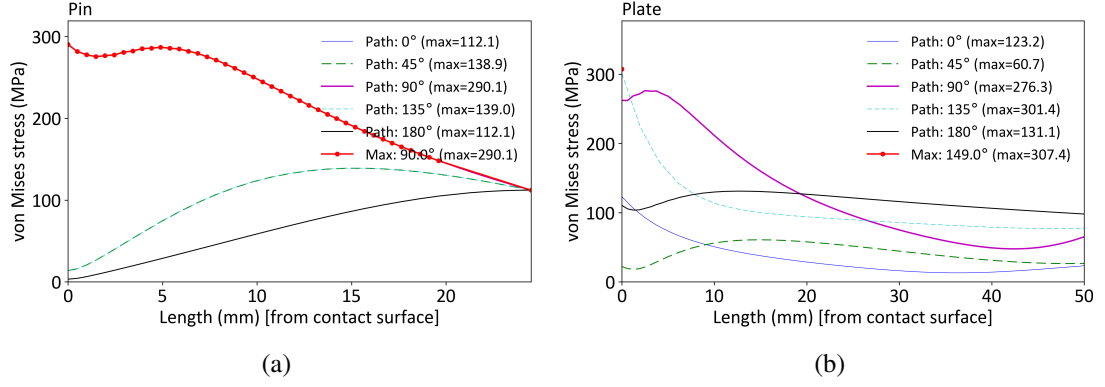


Figure 4.15: von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.98$ and $a/c = 1$: (a) pin, (b) plate [model: 2dm2]

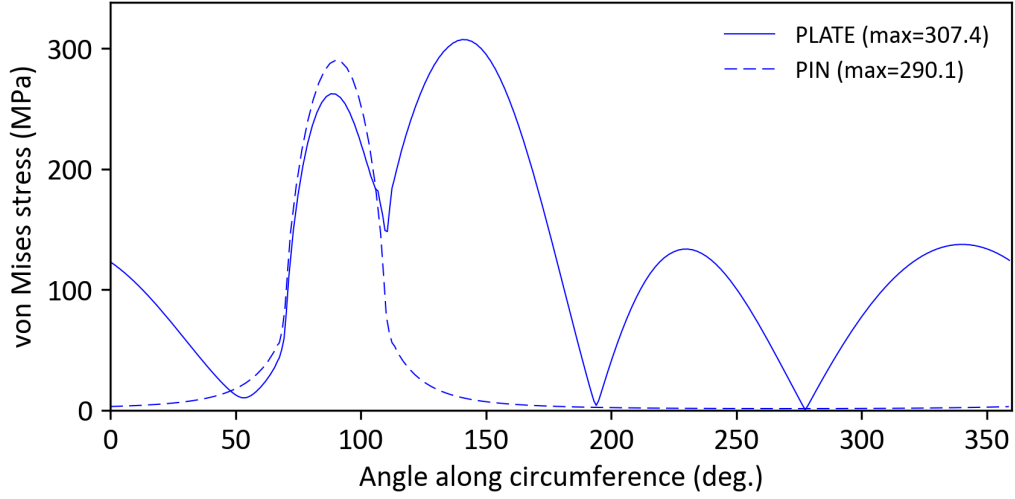


Figure 4.16: von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.98$ and $a/c = 1$ [model: 2dm2]

Mises stress has maximum value. For model 2dm3, maximum von Mises stress occurs at 90.0° in the pin and at 91.9° in the plate. The variation of von Mises stress along the circumference of the pin and the hole is shown in Figure 4.19.

The von Mises stress contour plots for model 2dm4 with $d_p/d = 0.999$ and $a/c = 0.5$ are shown for the pin and the plate in Figure 4.20. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the pin and the plate (starting from the contact surface) are presented in Figure 4.21, along with the direction at which von Mises stress has maximum value. For model 2dm4, maximum von Mises stress occurs at 112.5° in the pin and at 144.4° in the plate. The variation of von Mises stress along the circumference of the pin and the hole is shown in Figure 4.22.

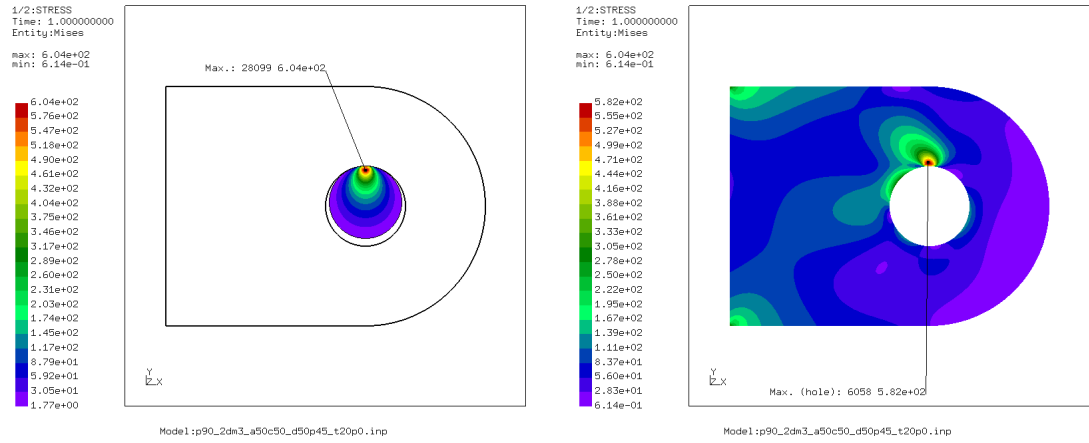


Figure 4.17: von Mises stress results for 2D model with $d_p/d = 0.90$ and $a/c = 1$ [model: 2dm3]

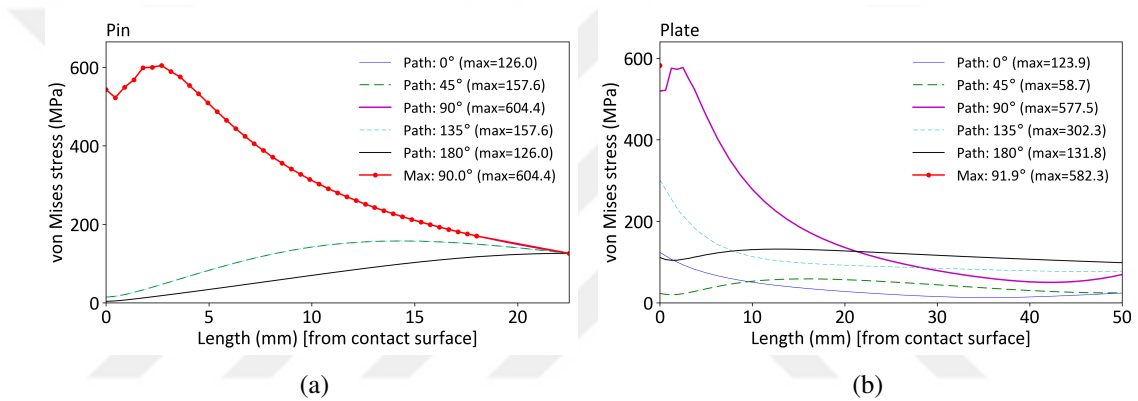


Figure 4.18: von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.90$ and $a/c = 1$: (a) pin, (b) plate [model: 2dm3]

The von Mises stress contour plots for model 2dm5 with $d_p/d = 0.98$ and $a/c = 0.5$ are shown for the pin and the plate in Figure 4.23. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the pin and the plate (starting from the contact surface) are presented in Figure 4.24, along with the direction at which von Mises stress has maximum value. For model 2dm5, maximum von Mises stress occurs at 90.0° in the plate and 140.6° in the pin. The variation of von Mises stress along the circumference of the pin and the hole is shown in Figure 4.25.

The von Mises stress contour plots for model 2dm6 with $d_p/d = 0.90$ and $a/c = 0.5$ are shown for the pin and the plate in Figure 4.26. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the pin and the plate (starting from the contact surface) are presented in Figure 4.27, along with the direction at which von

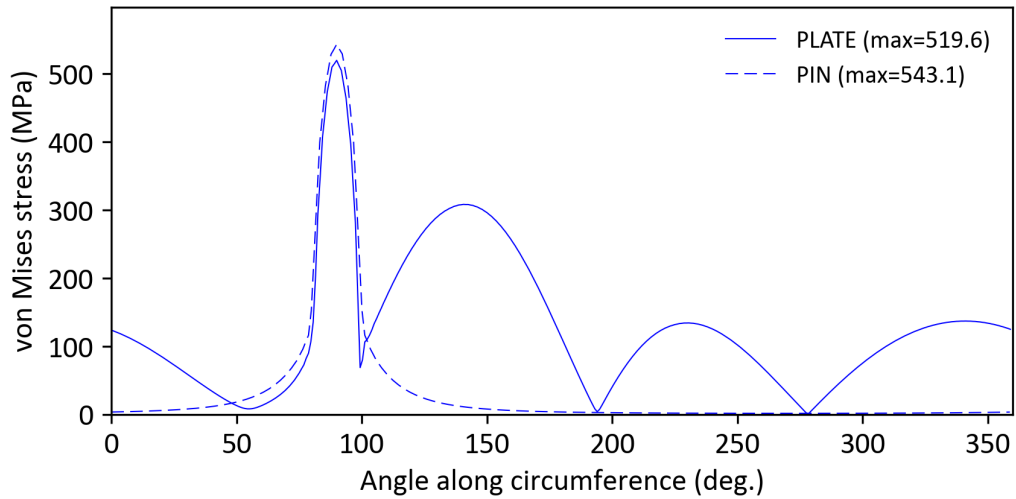


Figure 4.19: von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.90$ and $a/c = 1$ [model: 2dm3]

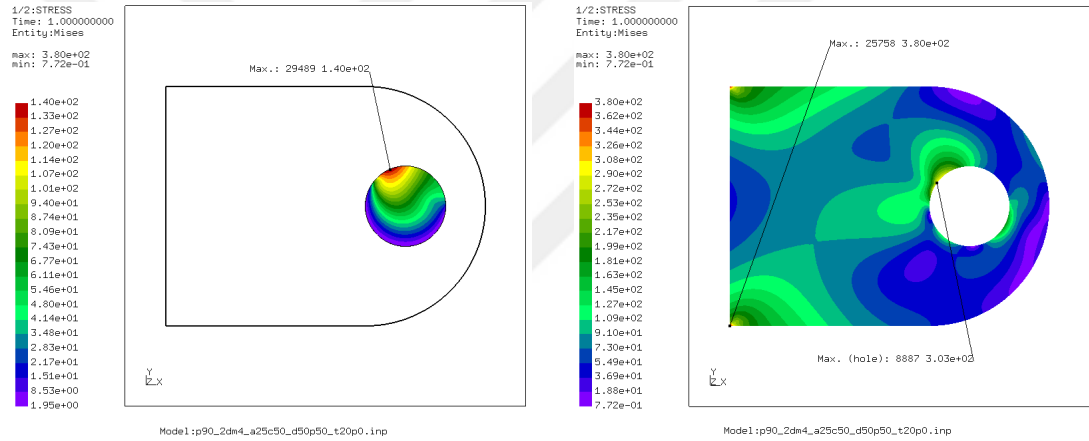


Figure 4.20: von Mises stress results for 2D model with $d_p/d = 0.999$ and $a/c = 0.5$ [model: 2dm4]

Mises stress has maximum value. For model 2dm6, maximum von Mises stress occurs at 90.0° in the pin and at 91.9° in the plate. The variation of von Mises stress along the circumference of the pin and the hole is shown in Figure 4.28.

For the remaining 2D models with $a/c = 1.5$, only the von Mises stress contour plots are shown in Figures 4.29 through 4.31. Comparing the stress contour plots of all 2D models, it is observed that for models with neat-fit pin, the stress contours in the plate follow the hole to a larger extent whereas the stress contours are more localized in the models with larger clearance between the plate hole and the pin. Axial eccentricity (a/c) has a limited effect on the stress results since c value is kept constant

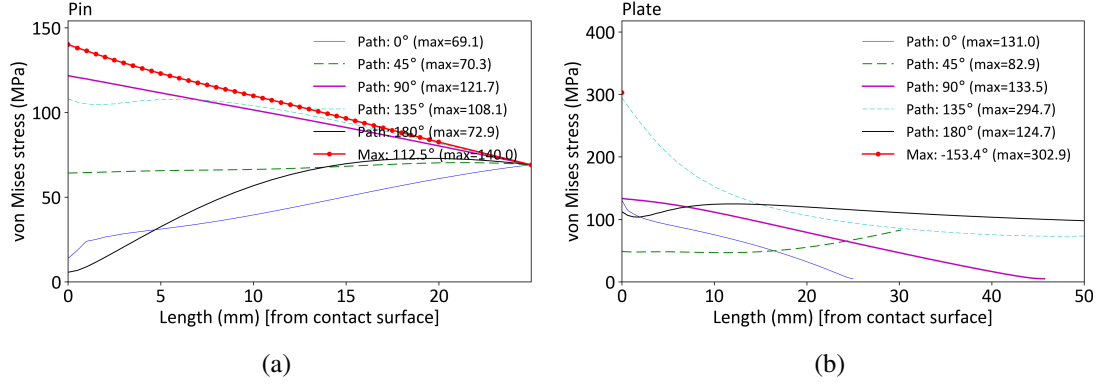


Figure 4.21: von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.999$ and $a/c = 0.5$: (a) pin, (b) plate [model: 2dm4]

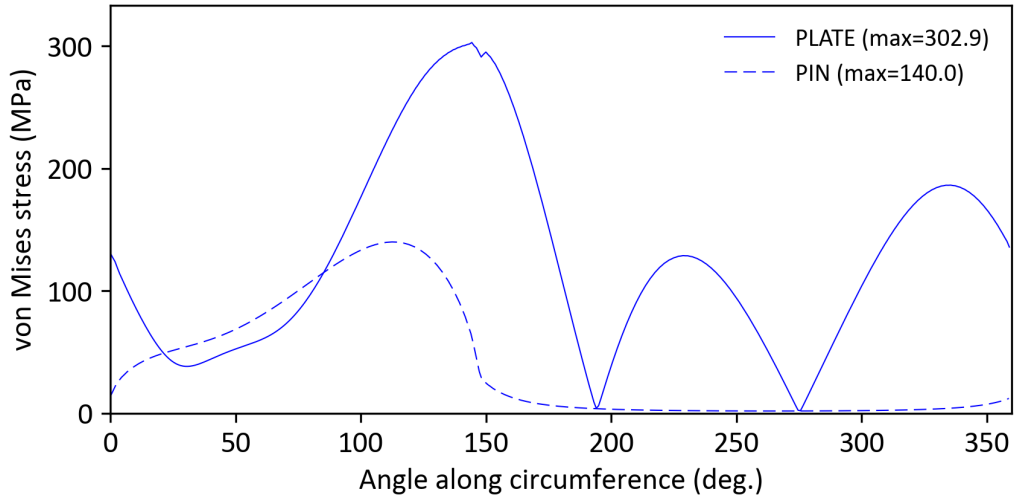


Figure 4.22: von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.999$ and $a/c = 0.5$ [model: 2dm4]

and only transverse loading is considered in the models. Its effect is more pronounced in models with smaller a/c and higher d_p/d values.

The maximum von Mises stress results in the pin and the plate are summarized in Table 4.13 for all 2D models, along with the direction and radial distance from the contact surface. Considering the first model (2dm1 with $d_p/d = 0.999$ and $a/c = 1$) as the base model, the ratio of maximum von Mises stress in other models to the one in the base model is also reported in Table 4.13 for both the pin and the plate. Changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.98$ increases the maximum SVM stress in the plate by a factor of 1.08 – 1.11 and in the pin by a factor of 2.07 – 2.20 depending on the value of a/c . And changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.90$

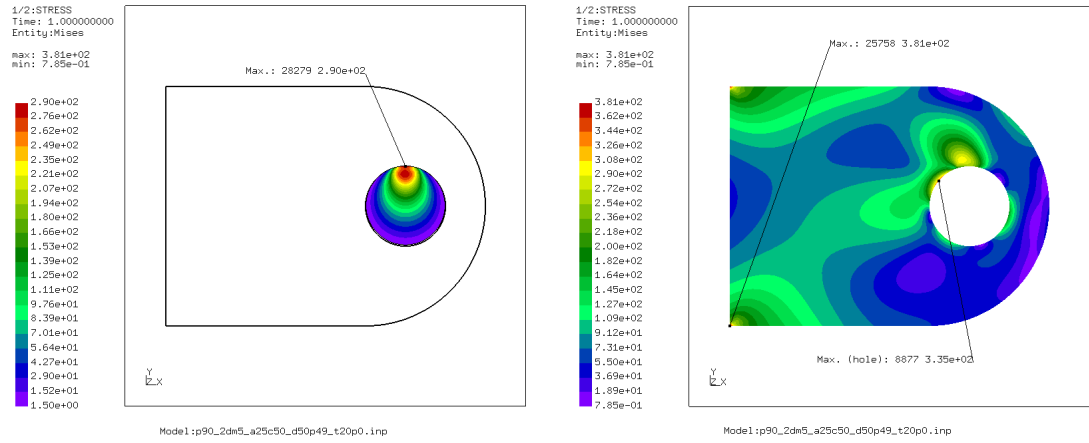


Figure 4.23: von Mises stress results for 2D model with $d_p/d = 0.98$ and $a/c = 0.5$ [model: 2dm5]

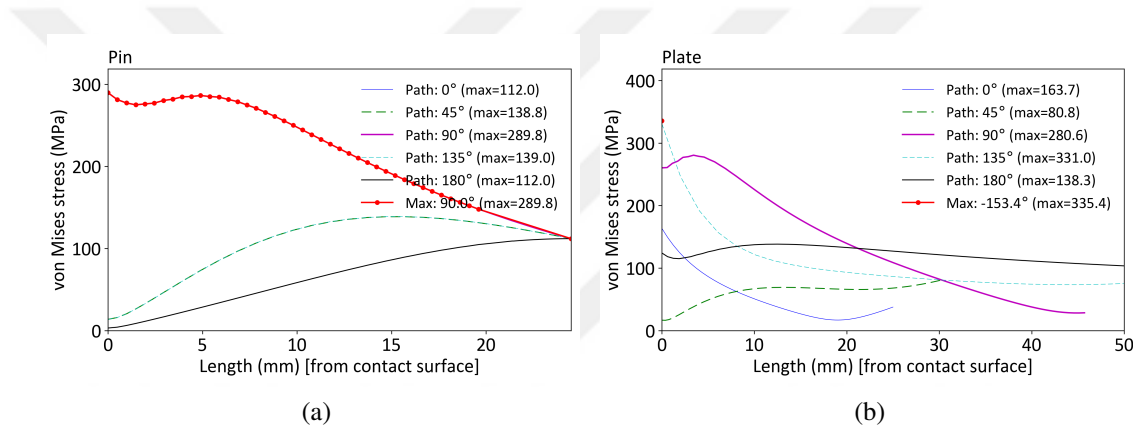


Figure 4.24: von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.98$ and $a/c = 0.5$: (a) pin, (b) plate [model: 2dm5]

increases the maximum SVM stress in the plate by a factor of 1.95 – 2.11 and in the pin by a factor of 4.32 – 4.58 depending on the value of a/c .

For models with neat-fit pin (2dm1, 2dm4, 2dm7), the maximum von Mises stress in the plate occurs at 146.2°, 144.4°, 146.2° on the contact surface. In the same models, the maximum von Mises stress in the pin occurs at 114.7°, 112.5°, 114.7° on the contact surface. Increasing clearance (or decreasing d_p/d ratio) shifts the location of maximum von Mises stress toward 90° in all models. The maximum von Mises stresses in both plate and pin are located on the contact surface in all models except the ones with a large clearance ($d_p/d = 0.90$) where the maximum stresses in the plate and pin are located 2.3 – 2.5 mm and 2.7 mm from the contact surface, respectively.

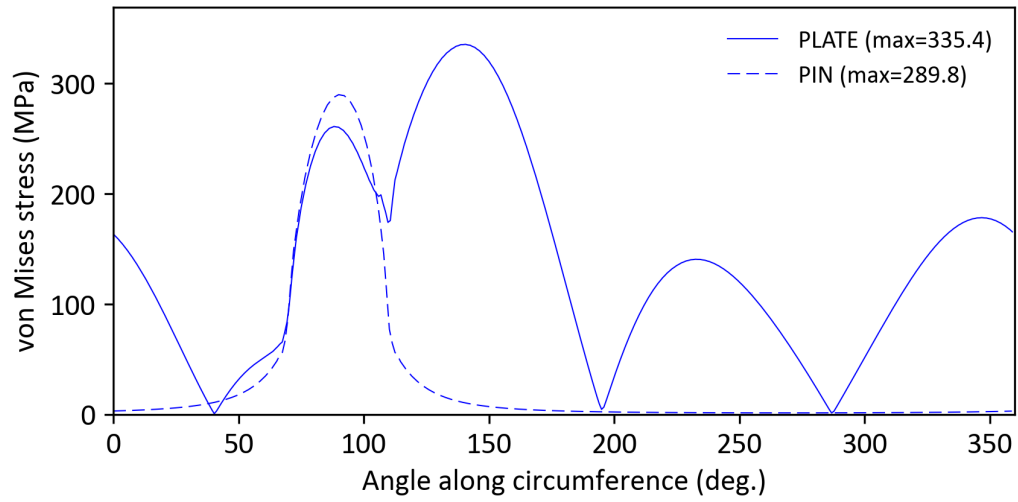


Figure 4.25: von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.98$ and $a/c = 0.5$ [model: 2dm5]

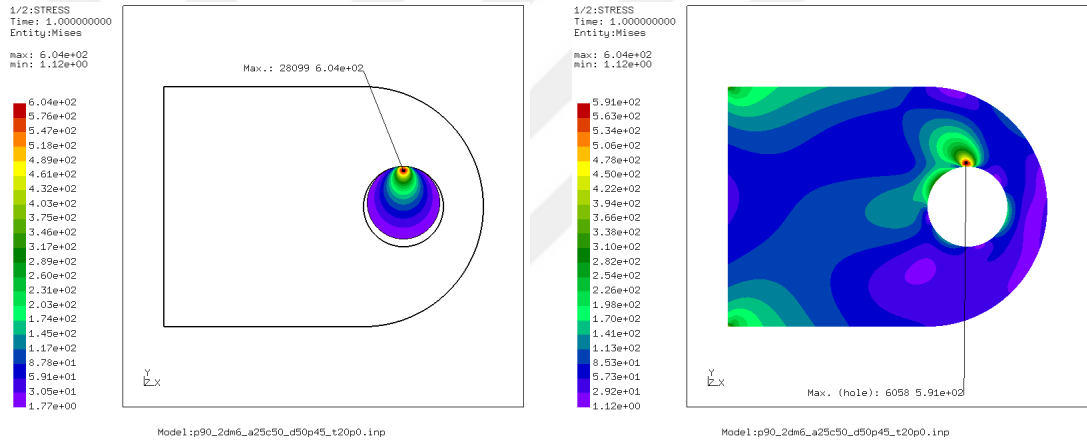


Figure 4.26: von Mises stress results for 2D model with $d_p/d = 0.90$ and $a/c = 0.5$ [model: 2dm6]

4.5 Results of Three-Dimensional Finite Element Models

Three-dimensional (3D) models are analyzed for different values of the parameters d_p , a , and t_e . In order to limit the number of figures, the detailed results are plotted only for three models (3dm1, 3dm2, 3dm5) in Figures 4.32 through 4.46. Maximum von Mises stress results for all 3D models are summarized in Tables 4.14 and 4.15, along with their direction with respect to the model origin (center of plate hole) and the location of maximum stress from the contact surface. In both tables maximum von Mises stress results in the plate (near the hole) and the pin are given at their middle

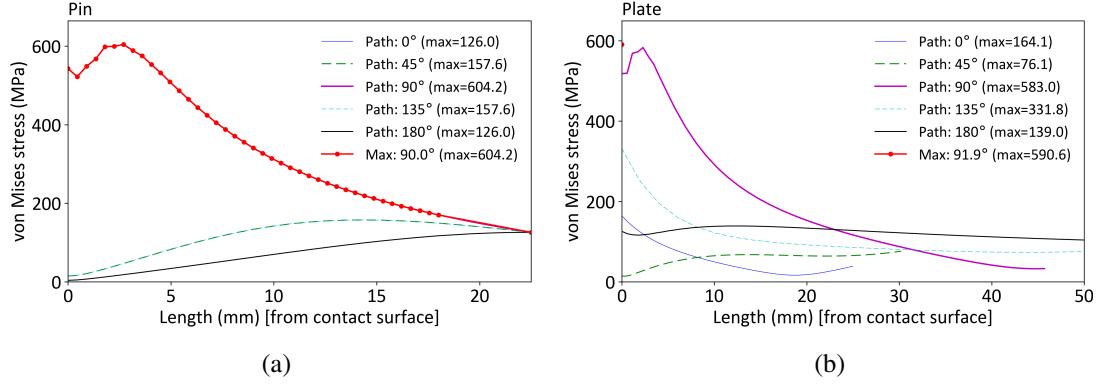


Figure 4.27: von Mises stress in radial directions at 0, 45, 90, 135, 180° for 2D model with $d_p/d = 0.90$ and $a/c = 0.5$: (a) pin, (b) plate [model: 2dm6]

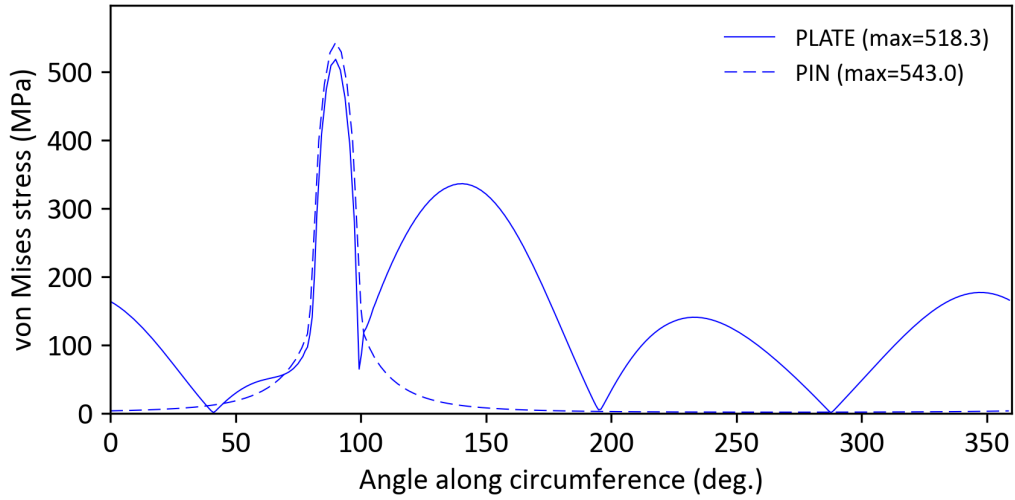


Figure 4.28: von Mises stress along the pin/hole circumference for 2D model with $d_p/d = 0.90$ and $a/c = 0.5$ [model: 2dm6]

surface and at the surface corresponding to the front of the plate. It should be noted that in some models maximum stress in the plate occurs near the support, however only the maximum stress near the contact region (hole) is reported in Table 4.14.

The von Mises stress contour plots for model 3dm1 with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ are shown for the pin and the plate in Figure 4.32. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the front and middle surfaces of the plate (starting from the contact surface) are presented in Figure 4.33, along with the direction at which von Mises stress has maximum value. Similar plots for the pin are shown in Figure 4.34 at the surfaces corresponding to the front and middle of the plate. The variation of von Mises stress in thru-thickness direction for

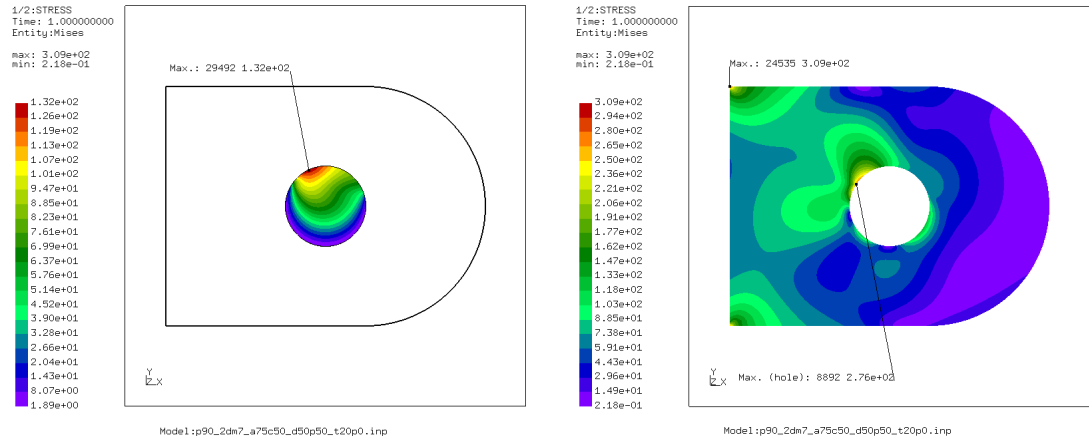


Figure 4.29: von Mises stress results for 2D model with $d_p/d = 0.999$ and $a/c = 1.5$ [model: 2dm7]

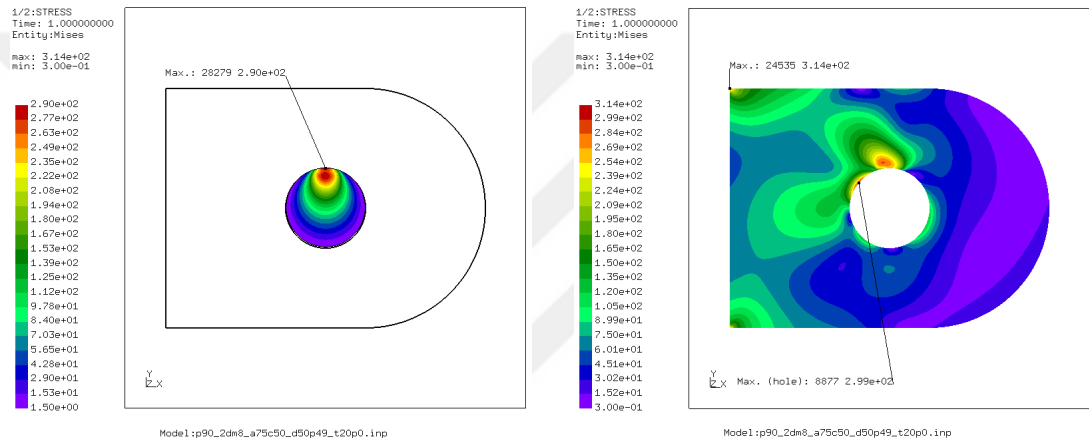


Figure 4.30: von Mises stress results for 2D model with $d_p/d = 0.98$ and $a/c = 1.5$ [model: 2dm8]

0, 45, 90, 135, 180° radial paths are presented in Figure 4.35 for both the pin and the plate. The variation of von Mises stress along the circumference of the pin and the hole are shown in Figure 4.36 for both the front and middle surfaces of the plate and the corresponding surfaces of the pin. For model 3dm1, maximum von Mises stress is located at 139.1° in the plate front surface and at 143.2° in the plate middle surface. The maximum von Mises stresses at the corresponding surfaces of the pin are located 115.0° direction. In thru-thickness direction, the maximum von Mises stress values are higher at the front and back surfaces of the plate.

The von Mises stress contour plots for model 3dm2 with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ are shown for the pin and the plate in Figure 4.37. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the front and middle

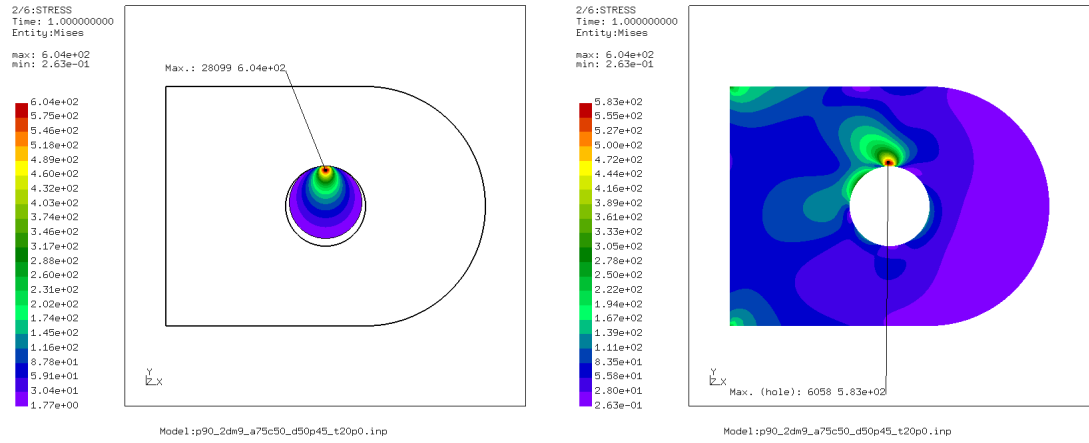


Figure 4.31: von Mises stress results for 2D model with $d_p/d = 0.90$ and $a/c = 1.5$ [model: 2dm9]

Table 4.13: Summary of maximum von Mises stress results for all 2D models

Model	$\frac{a}{c}$	$\frac{d_p}{d}$	Plate				Pin			
			SVM (MPa)	Dir. (°)	Dis. (mm)	Fac. (base)	SVM (MPa)	Dir. (°)	Dis. (mm)	Fac. (base)
1 (base)	1.00	0.999	283.4	146.2	0.0	1.000	133.4	114.7	0.0	1.000
2	1.00	0.980	307.4	140.6	0.0	1.085	290.1	90.0	0.0	2.175
3	1.00	0.900	582.3	91.9	2.5	2.055	604.4	90.0	2.7	4.531
4	0.50	0.999	302.9	144.4	0.0	1.069	140.0	112.5	0.0	1.050
5	0.50	0.980	335.4	140.6	0.0	1.184	289.8	90.0	0.0	2.172
6	0.50	0.900	590.6	91.9	2.3	2.084	604.2	90.0	2.7	4.529
7	1.50	0.999	276.2	146.2	0.0	0.975	131.8	114.7	0.0	0.988
8	1.50	0.980	298.9	140.6	0.0	1.055	290.3	90.0	0.0	2.176
9	1.50	0.900	582.9	91.9	2.5	2.057	603.7	90.0	2.7	4.525

surfaces of the plate (starting from the contact surface) are presented in Figure 4.38, along with the direction at which von Mises stress has maximum value. Similar plots for the pin are shown in Figure 4.39 at the surfaces corresponding to the front and middle of the plate. The variation of von Mises stress in thru-thickness direction for 0, 45, 90, 135, 180° radial paths are presented in Figure 4.40 for both the pin and the plate. The variation of von Mises stress along the circumference of the pin and the hole are shown in Figure 4.41 for both the front and middle surfaces of the plate and the corresponding surfaces of the pin. For model 3dm2, maximum von Mises stresses are located at 102.7° and 143.2° in the plate front and middle surfaces, respectively. Maximum von Mises stresses are located at 90° in the corresponding surfaces of the pin. In thru-thickness direction, the maximum von Mises stresses in the plate are

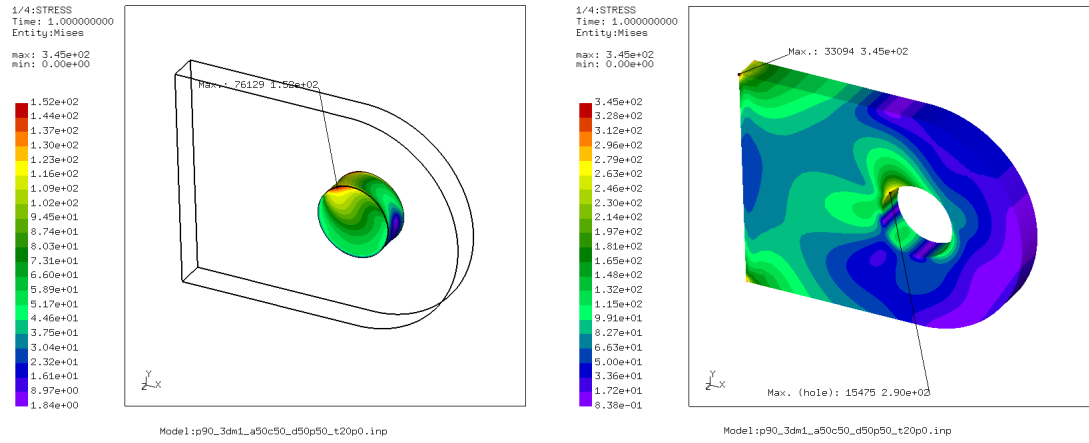


Figure 4.32: von Mises stress results for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]

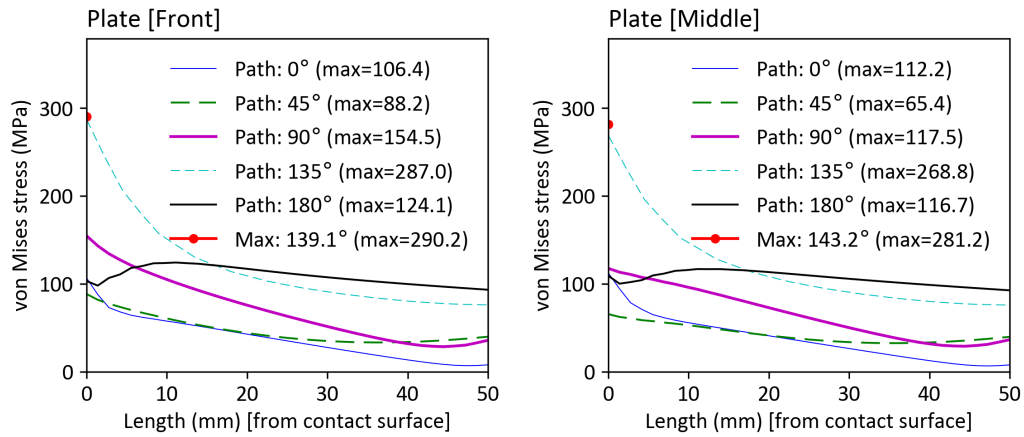


Figure 4.33: von Mises stress at the plate in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]

located on the front and back surfaces of the plate whereas in the pin they are located within the thickness of pin but still near the front and back surfaces of the plate.

The von Mises stress contour plots for model 3dm5 with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ are shown for the pin and the plate in Figure 4.42. The variation of von Mises stress in radial directions at 0, 45, 90, 135, 180° in the front and middle surfaces of the plate (starting from the contact surface) are presented in Figure 4.43, along with the direction at which von Mises stress has maximum value. Similar plots for the pin are shown in Figure 4.44 at the surfaces corresponding to the front and middle of the plate. The variation of von Mises stress in thru-thickness direction for 0, 45, 90, 135, 180° radial paths are presented in Figure 4.45 for both the pin and the

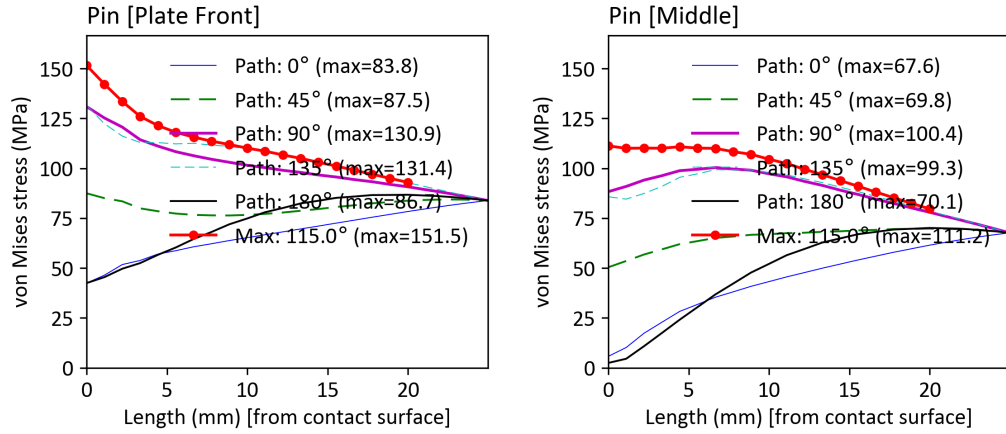


Figure 4.34: von Mises stress at the pin in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]

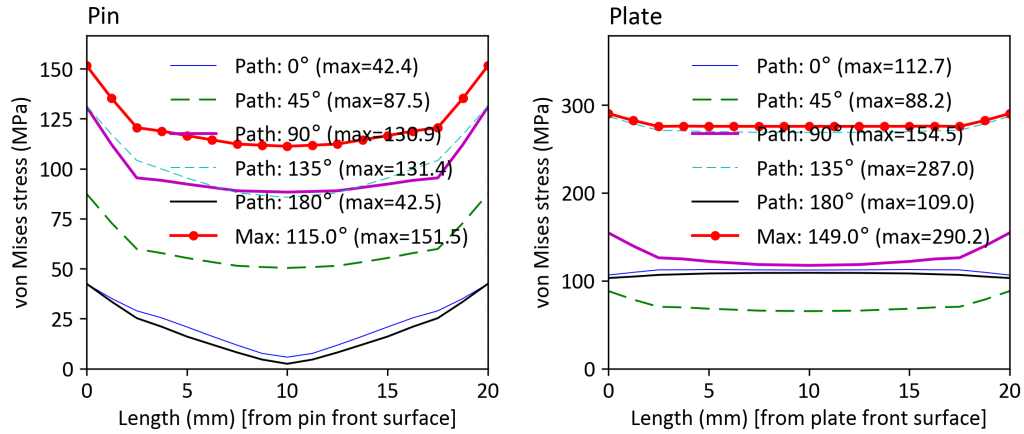


Figure 4.35: von Mises stress in thru-thickness direction for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]

plate. The variation of von Mises stress along the circumference of the pin and the hole are shown in Figure 4.46 for both the front and middle surfaces of the plate and the corresponding surfaces of the pin. For model 3dm5, maximum von Mises stresses are located at 90° and 143.2° in the plate front and middle surfaces, respectively and are located at 90° in the corresponding surfaces of the pin. In thru-thickness direction, the maximum von Mises stress values are at the front and back surfaces of the plate and at the corresponding surfaces of the pin.

The maximum von Mises stresses near the contact region in all 3D models are summarized in Table 4.14 for the plate front and middle surfaces and in Table 4.15 for the corresponding surfaces of the pin, along with the direction and radial distance

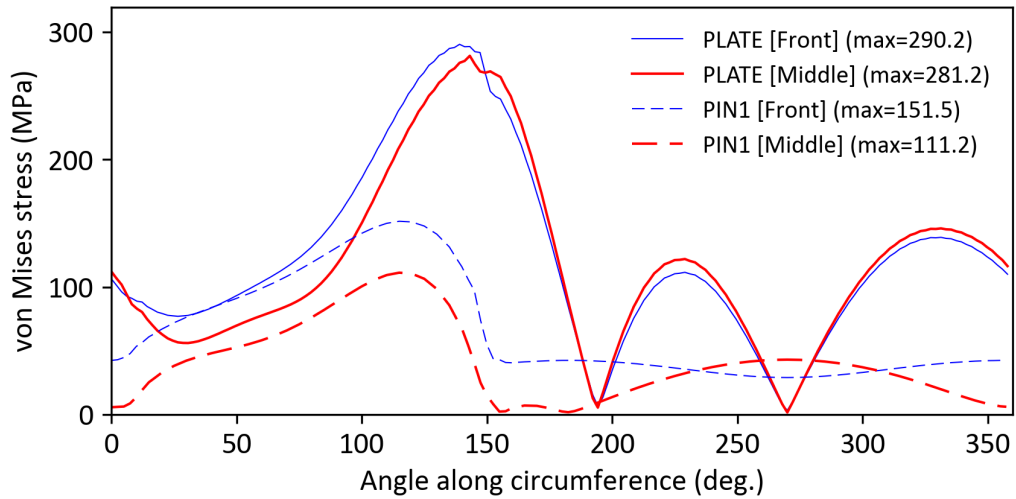


Figure 4.36: von Mises stress along the pin/hole circumference for 3D model with $d_p/d = 0.999$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm1]

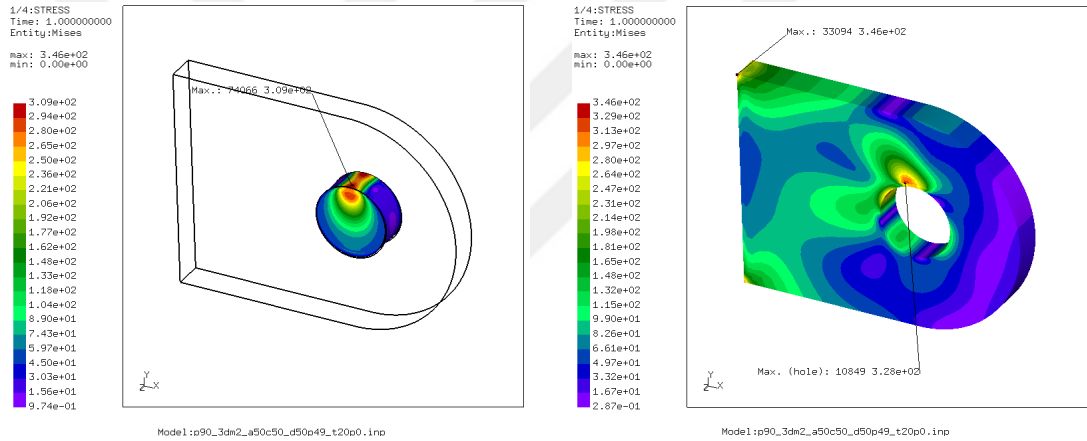


Figure 4.37: von Mises stress results for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]

from the contact surface. Fifth column in the tables lists overall maximum values which often match the value in plate front (and back) surface and the corresponding pin surface. In some models, the difference between the overall maximum von Mises stress and the reported maximum values in the front and middle surfaces is up to 1.6% for the plate and 9.2% for the pin, indicating that the maximum value occurs somewhere between the front and middle surface. Considering the first model (3dm1 with $d_p/d = 0.999$, $a/c = 1$ and $t_e/t = 0$) as the base model, the ratio of maximum von Mises stress in other models to the one in the base model is also reported in the tables for both the pin and the plate (under the column header “Fac. (base)”).

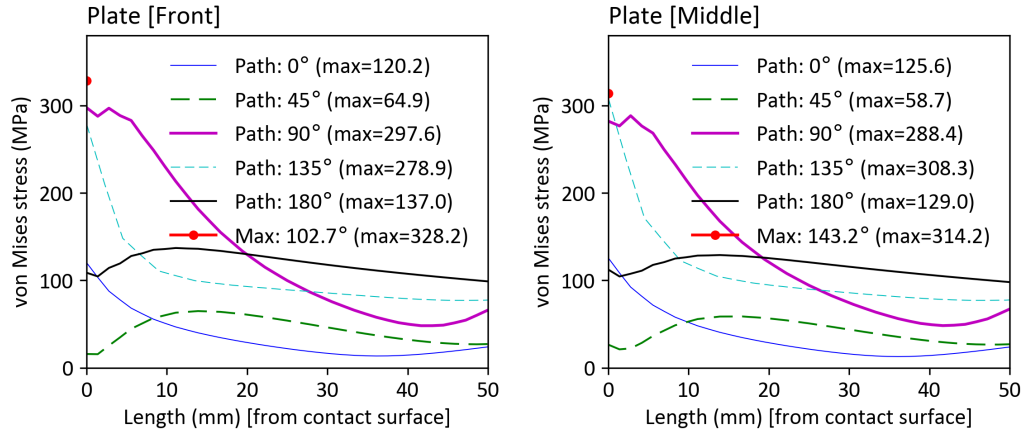


Figure 4.38: von Mises stress at the plate in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]

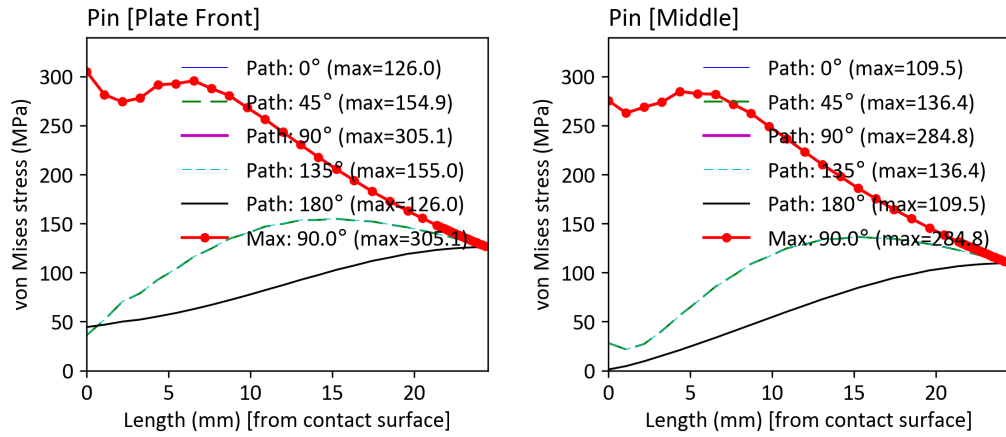


Figure 4.39: von Mises stress at the pin in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]

Comparing the results of 3D models with no pin extension (i.e., 3dm1-3, 3dm10-12 and 3dm19-21) with the results of 2D models, it is observed that the stresses in the 2D models are generally lower than 3D models. The maximum von Mises stress in the pin of 2D models are within the 3D model results reported at the front and middle surfaces of the plate in 3D models. A similar trend is observed in stress results of the plate, but only in neat-fit models ($d_p/d = 0.999$). The maximum von Mises stress at the middle surface of the plate is within -0.8% and $+2.3\%$ of the values reported for 2D models in Table 4.13. The maximum von Mises stress at the middle surface of the pin is within -2.6% and -1.8% of the values reported for 2D models in Table 4.13 for larger clearance models with $d_p/d = 0.98$ and $d_p/d = 0.9$ and about

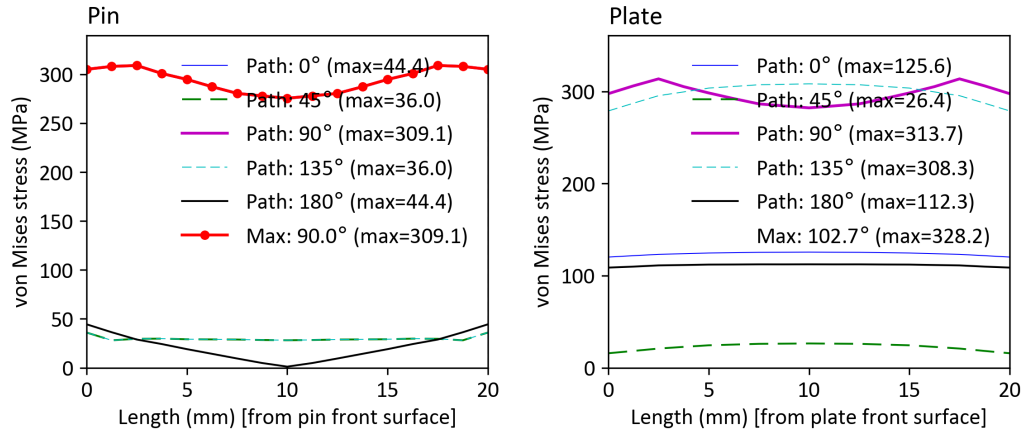


Figure 4.40: von Mises stress in thru-thickness direction for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]

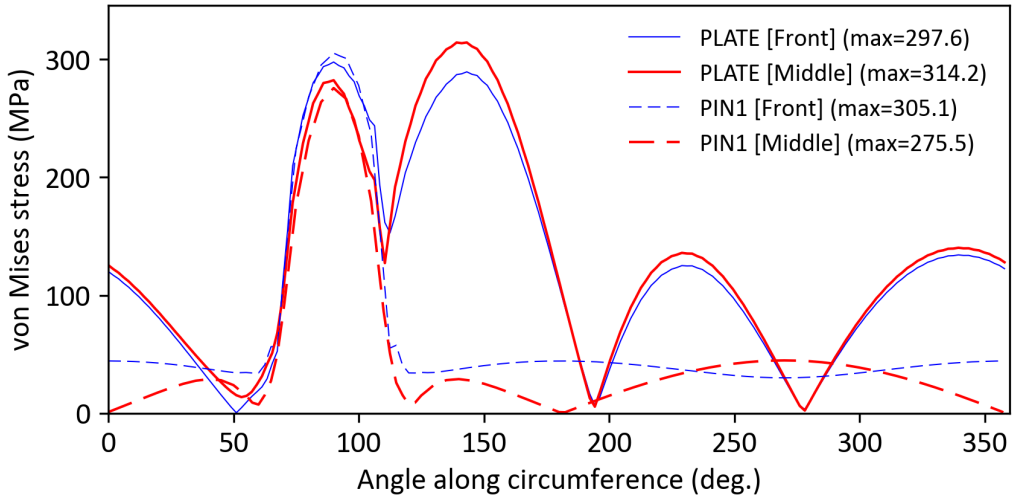


Figure 4.41: von Mises stress along the pin/hole circumference for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0$ [model: 3dm2]

–17% for the models with neat-fit pin with $d_p/d = 0.999$. Changing clearance in 3D models from $d_p/d = 0.999$ to $d_p/d = 0.98$ increases the maximum SVM stress in the plate middle surface by a factor of 1.12 – 1.17 and in the pin middle surface by a factor of 2.33 – 2.66 depending on the value of a/c . And changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.90$ increases the maximum SVM stress in the plate by a factor of 1.94 – 2.17 and in the pin by a factor of 4.76 – 5.68 depending on the value of a/c .

For models with neat-fit pin (3dm1,4,7,10,13,16,19,22,25), the maximum von Mises

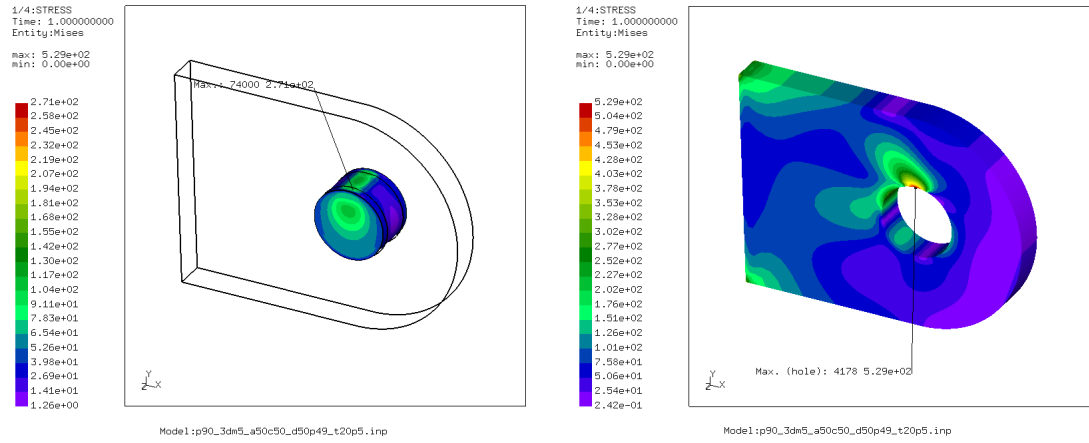


Figure 4.42: von Mises stress results for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]

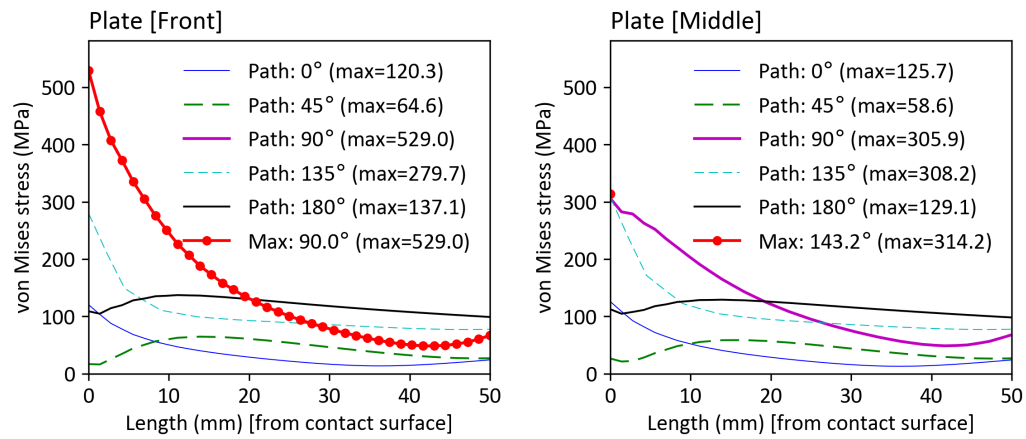


Figure 4.43: von Mises stress at the plate in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]

stress occurs on the contact surface at 143.2° in the plate middle surface (except in 3dm16 it is 139.1°) and at varying directions within 126.8° and 139.1° in the plate front surface. Increasing clearance (or decreasing d_p/d) decreases this direction angle to a lower value and increasing pin extension (t_e) brings the location of maximum von Mises stress in the plate to 90° direction. In models with no pin extension, the maximum von Mises stresses are located 2.7 – 2.9 mm from the contact surface in the plate front surface for $d_p/d = 0.90, 0.98$ and 2.7 – 2.8 mm from the contact surface in the plate middle surface for $d_p/d = 0.90$. For models with neat-fit pin with no extension, the maximum von Mises stresses occur on the contact surface at 115° direction in the pin middle surface and in the surface corresponding to the plate front (or back). Increasing pin extension (t_e) brings the location of maximum von Mises

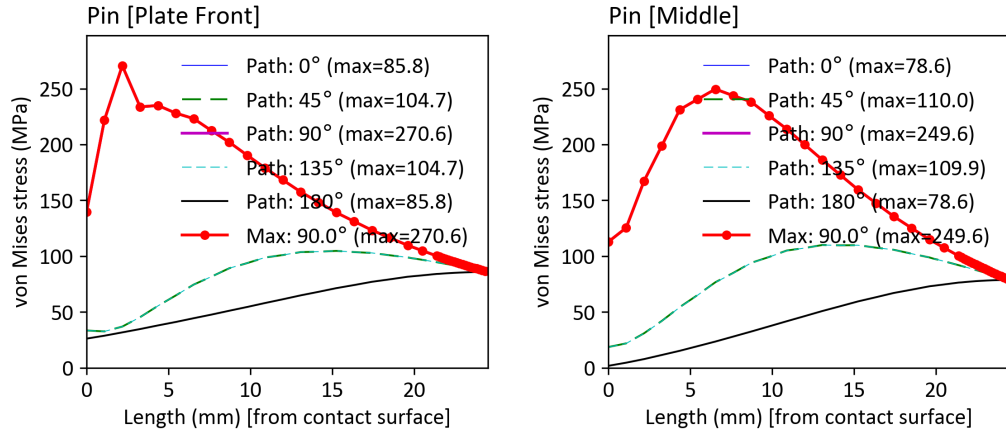


Figure 4.44: von Mises stress at the pin in radial directions at 0, 45, 90, 135, 180° for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]

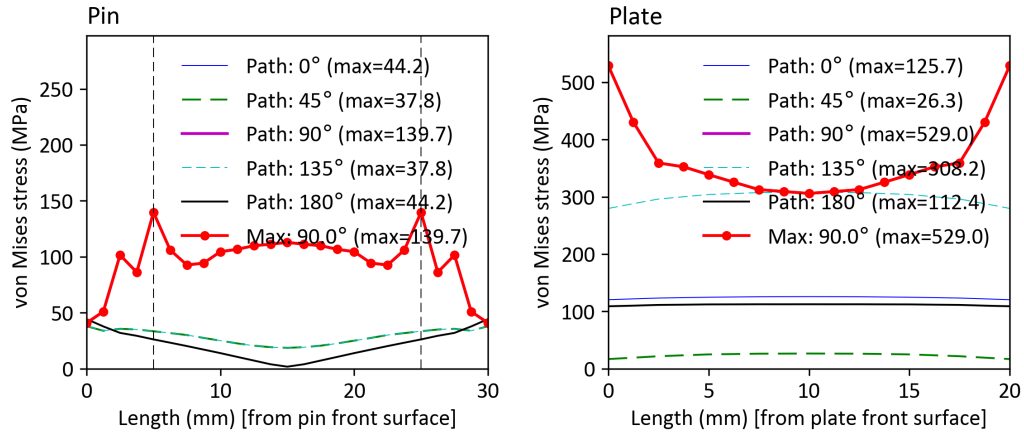


Figure 4.45: von Mises stress in thru-thickness direction for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]

stress to 110 – 120° direction about 2.0 – 11.1 mm away from the contact surface in the pin middle surface and to 115 – 120° direction about 2.0 – 2.2 mm away from the contact surface in the surface of the pin corresponding to the plate front (or back). Increasing clearance (or decreasing d_p/d) brings the location of maximum von Mises stress in the pin to 90° direction.

For models with pin extension ($t_e > 0$), bending of the pin over the plate edges causes significant stress increase at those locations. The stress concentration is more significant for models with larger clearance (i.e., $d_p/d = 0.98$ and $d_p/d = 0.9$). When other parameters are fixed, the difference between the results of $t_e/t = 0.25$ and $t_e/t = 0.5$ is observed to be not much. But compared to models with no pin extension ($t_e = 0$),

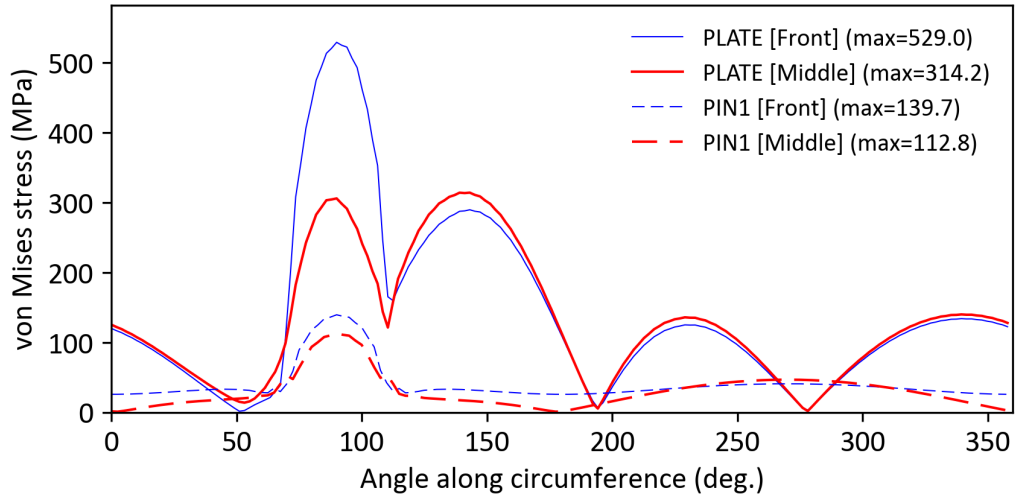


Figure 4.46: von Mises stress along the pin/hole circumference for 3D model with $d_p/d = 0.98$, $a/c = 1$, and $t_e/t = 0.25$ [model: 3dm5]

the maximum von Mises stress in the plate front surface increases about a factor of 1.18 – 1.20 for neat-fit models with $d_p/d = 0.999$, about a factor of 1.55 – 1.68 for clearance models with $d_p/d = 0.98$, and about a factor of 1.51 – 1.57 for large clearance models with $d_p/d = 0.90$. Pin extension has a less effect on maximum von Mises stress in the plate middle surface, within -5.1% and $+0.5\%$ of the maximum stress values of the corresponding models with no pin extension. Having pin extension decreases the maximum von Mises stress results about 8.4 – 20.8% in the pin middle surface and about 8.1 – 24.4% in the surface corresponding to the front of the plate with respect to the models having no pin extension.

Changing a/c ratio has a more pronounced effect on the plate. The maximum von Mises stress in the plate increases in general when a/c is decreased to 0.5, but this increase becomes less significant as d_p/d ratio decreases from 0.999 and 0.90. The change in the maximum von Mises stress in the plate front surface is between $+7.2\%$ and $+7.5\%$ for models with $d_p/d = 0.999$, between $+0.1\%$ and $+4.4\%$ for models with $d_p/d = 0.98$, and between $+0.2\%$ and $+2.1\%$ for models with $d_p/d = 0.90$. The change in the maximum von Mises stress in the plate middle surface is between $+6.4\%$ and $+7.1\%$ for models with $d_p/d = 0.999$, $+9.2\%$ for models with $d_p/d = 0.98$, and $+1.2\%$ for models with $d_p/d = 0.90$. When a/c is increased to 1.5, the maximum von Mises stress decreases for the model with neat-fit pin ($d_p/d = 0.999$) and no pin

extension ($t_e/t = 0$) whereas it may decrease or increase to a lesser degree for the other models. The change in the maximum von Mises stress in the plate front surface is between -2.0% and -2.3% for models with $d_p/d = 0.999$, between -0.7% and $+0.2\%$ for models with $d_p/d = 0.98$, and between $+0.1\%$ and $+0.2\%$ for models with $d_p/d = 0.90$. The change in the maximum von Mises stress in the plate middle surface is -2.6% for models with $d_p/d = 0.999$, between -2.2% and -2.7% for models with $d_p/d = 0.98$, and $+0.2\%$ for models with $d_p/d = 0.90$.

Table 4.14: Summary of maximum von Mises stress results near the plate hole for all 3D models

Model	$\frac{a}{c}$	$\frac{d_p}{d}$	$\frac{t_e}{t}$	Max. SVM (MPa)	Plate [Front]				Plate [Middle]			
					SVM (MPa)	Dir. (°)	Dis. (mm)	Fac. (base)	SVM (MPa)	Dir. (°)	Dis. (mm)	Fac. (base)
1 (base)	1.0	0.999	0.0	290.2	290.2	139.1	0.0	1.00	281.2	143.2	0.0	1.00
2	1.0	0.980	0.0	328.2	328.2	102.7	2.9	1.13	314.2	143.2	0.0	1.12
3	1.0	0.900	0.0	665.2	654.6	94.3	2.8	2.26	593.0	90.0	2.8	2.11
4	1.0	0.999	0.25	343.7	343.7	130.9	0.0	1.18	276.5	143.2	0.0	0.98
5	1.0	0.980	0.25	529.0	529.0	90.0	0.0	1.82	314.2	143.2	0.0	1.12
6	1.0	0.900	0.25	1010.5	1010.5	90.0	0.0	3.48	566.5	90.0	2.8	2.01
7	1.0	0.999	0.5	347.5	347.5	126.8	0.0	1.20	275.7	143.2	0.0	0.98
8	1.0	0.980	0.5	548.0	548.0	90.0	0.0	1.89	314.2	143.2	0.0	1.12
9	1.0	0.900	0.5	1027.8	1027.8	90.0	0.0	3.54	563.0	90.0	2.8	2.00
10	0.5	0.999	0.0	311.0	311.0	139.1	0.0	1.07	301.1	143.2	0.0	1.07
11	0.5	0.980	0.0	343.2	342.5	102.7	2.9	1.18	343.2	139.1	0.0	1.22
12	0.5	0.900	0.0	668.6	668.6	94.3	2.7	2.30	600.2	94.3	2.7	2.13
13	0.5	0.999	0.25	369.0	369.0	126.8	0.0	1.27	294.2	143.2	0.0	1.05
14	0.5	0.980	0.25	529.5	529.5	90.0	0.0	1.82	343.2	139.1	0.0	1.22
15	0.5	0.900	0.25	1012.1	1012.1	90.0	0.0	3.49	573.3	94.3	2.7	2.04
16	0.5	0.999	0.5	373.6	373.6	126.8	0.0	1.29	293.6	139.1	0.0	1.04
17	0.5	0.980	0.5	548.5	548.5	90.0	0.0	1.89	343.2	139.1	0.0	1.22
18	0.5	0.900	0.5	1029.4	1029.4	90.0	0.0	3.55	569.9	94.3	2.7	2.03
19	1.5	0.999	0.0	283.5	283.5	139.1	0.0	0.98	274.0	143.2	0.0	0.97
20	1.5	0.980	0.0	326.0	326.0	102.7	2.9	1.12	305.8	139.1	0.0	1.09
21	1.5	0.900	0.0	664.0	656.0	90.0	2.8	2.26	594.3	90.0	2.8	2.11
22	1.5	0.999	0.25	336.9	336.9	130.9	0.0	1.16	269.5	143.2	0.0	0.96
23	1.5	0.980	0.25	530.1	530.1	90.0	0.0	1.83	305.8	139.1	0.0	1.09
24	1.5	0.900	0.25	1011.2	1011.2	90.0	0.0	3.48	567.7	90.0	2.8	2.02
25	1.5	0.999	0.5	340.5	340.5	126.8	0.0	1.17	268.7	143.2	0.0	0.96
26	1.5	0.980	0.5	549.2	549.2	90.0	0.0	1.89	307.4	90.0	0.0	1.09
27	1.5	0.900	0.5	1028.5	1028.5	90.0	0.0	3.54	564.3	90.0	2.8	2.01

It should be noted that the models with $d_p/d = 0.90$, $a/c = 0.5$ and $t_e/t = 0.5$ are considered for more of an academic interest to study extreme conditions as such values are unlikely to be used in an actual design. These models also indicate very

Table 4.15: Summary of maximum von Mises stress results in the pin for all 3D models

Model	$\frac{a}{c}$	$\frac{d_p}{d}$	$\frac{t_e}{t}$	Max.	Pin [Plate Front]				Pin [Middle]			
				SVM (MPa)	SVM (MPa)	Dir. (°)	Dis. (mm)	Fac. (base)	SVM (MPa)	Dir. (°)	Dis. (mm)	Fac. (base)
1 (base)	1.0	0.999	0.0	151.5	151.5	115.0	0.0	1.00	111.2	115.0	0.0	1.00
2	1.0	0.980	0.0	309.1	305.1	90.0	0.0	2.01	284.8	90.0	4.4	2.56
3	1.0	0.900	0.0	669.5	665.4	90.0	2.0	4.39	588.6	90.0	2.0	5.29
4	1.0	0.999	0.25	115.5	115.5	120.0	2.2	0.76	101.4	110.0	11.1	0.91
5	1.0	0.980	0.25	270.8	270.6	90.0	2.2	1.79	249.6	90.0	6.5	2.24
6	1.0	0.900	0.25	616.6	564.7	90.0	2.0	3.73	509.9	90.0	2.0	4.59
7	1.0	0.999	0.5	114.9	114.9	120.0	2.2	0.76	88.7	120.0	11.1	0.80
8	1.0	0.980	0.5	280.1	280.1	90.0	2.2	1.85	232.6	90.0	6.5	2.09
9	1.0	0.900	0.5	600.4	579.5	90.0	2.0	3.82	496.9	90.0	2.0	4.47
10	0.5	0.999	0.0	158.8	158.8	115.0	0.0	1.05	118.8	115.0	0.0	1.07
11	0.5	0.980	0.0	308.4	304.2	90.0	0.0	2.01	284.6	90.0	4.4	2.56
12	0.5	0.900	0.0	669.0	664.5	90.0	2.0	4.39	588.5	90.0	2.0	5.29
13	0.5	0.999	0.25	121.7	121.7	115.0	2.2	0.80	107.1	115.0	8.9	0.96
14	0.5	0.980	0.25	270.3	270.2	90.0	2.2	1.78	249.4	90.0	6.5	2.24
15	0.5	0.900	0.25	616.3	564.3	90.0	2.0	3.72	509.9	90.0	2.0	4.59
16	0.5	0.999	0.5	121.6	121.6	115.0	2.2	0.80	94.1	120.0	11.1	0.85
17	0.5	0.980	0.5	279.6	279.6	90.0	2.2	1.85	232.4	90.0	6.5	2.09
18	0.5	0.900	0.5	600.1	579.1	90.0	2.0	3.82	496.8	90.0	2.0	4.47
19	1.5	0.999	0.0	149.8	149.8	115.0	0.0	0.998	109.3	115.0	0.0	0.98
20	1.5	0.980	0.0	309.6	305.7	90.0	0.0	2.02	285.1	90.0	4.4	2.56
21	1.5	0.900	0.0	669.7	665.8	90.0	2.0	4.39	588.7	90.0	2.0	5.29
22	1.5	0.999	0.25	114.0	114.0	120.0	2.2	0.75	100.1	110.0	11.1	0.90
23	1.5	0.980	0.25	271.1	271.1	90.0	2.2	1.79	249.8	90.0	6.5	2.25
24	1.5	0.900	0.25	616.9	565.0	90.0	2.0	3.73	510.0	90.0	2.0	4.59
25	1.5	0.999	0.5	113.3	113.3	120.0	2.2	0.75	87.5	120.0	11.1	0.79
26	1.5	0.980	0.5	280.5	280.5	90.0	2.2	1.85	232.8	90.0	6.5	2.09
27	1.5	0.900	0.5	600.7	579.7	90.0	2.0	3.83	497.0	90.0	2.0	4.47

high von Mises stresses that are significantly higher than the yield strength of typical structural steels. For a more proper evaluation of these cases, the analysis requires consideration of material nonlinearity to allow re-distribution of stresses and to assess the extent of plastic strains.

CHAPTER 5

CONCLUSION

5.1 Summary

The behavior of pin connections under transverse loading is investigated in this study by analyzing parametric finite element models. A total of 9 two-dimensional (2D) and 27 three-dimensional (3D) models are created by varying pin diameter, eccentricity of hole (in axial direction), and bending of the pin (by extension of the pin beyond the plate thickness) to examine the effects of these parameters on the stress distribution in the plate and the pin. The analyses are performed by using Calculix, an open source finite element software. The contact interaction between the pin and the plate is included in the models by face-to-face penalty contact formulation. The contact between the pin and the plate is assumed to be frictionless. The accuracy of models are checked via mesh sensitivity analyses and comparison to analytical solutions given by Hertz for non-conformal cylindrical contacts and by Persson for conformal cylindrical contact. Three of the 2D models and three of the 3D models are also analyzed in Abaqus to compare the Calculix results with a commercial FEA software.

5.2 Concluding Remarks

The main conclusions of this study can be summarized as:

- The convergence of the contact analysis based on face-to-face penalty formulation is sensitive to the stiffness of contact springs, which itself depends on pin-to-hole diameter ratio and mesh size. For the models used in this study (including the

sensitivity analyses for different mesh size), the contact stiffness is typically within $0.1 - 5 E$ (E = modulus of elasticity of contacting parts, pin and plate).

- Comparing the stress contour plots of all 2D models, it is observed that for models with neat-fit pin, the stress contours in the plate follow the hole to a larger extent whereas the stress contours are more localized in the models with larger clearance between the plate hole and the pin. Axial eccentricity (a/c) has a limited effect on the stress results since c value is kept constant and only transverse loading is considered in the models. Its effect is more pronounced in models with smaller a/c and higher d_p/d values.
- Comparison between 2D and 3D models (with no pin extension, i.e., $t_e/t = 0$) indicates similar trends, with higher stresses in 3D model due to more accurate modeling of stress distribution in thru-thickness direction. The maximum von Mises stress in the pin of 2D models are within the 3D model results reported at the front and middle surfaces of the plate in 3D models. A similar trend is observed in stress results of the plate, but only in neat-fit models ($d_p/d = 0.999$). The maximum von Mises stress at the middle surface of the plate is within -0.8% and $+2.3\%$ of the values reported for 2D models in Table 4.13. The maximum von Mises stress at the middle surface of the pin is within -2.6% and -1.8% of the values reported for 2D models in Table 4.13 for larger clearance models with $d_p/d = 0.98$ and $d_p/d = 0.9$ and about -17% for the models with neat-fit pin with $d_p/d = 0.999$.
- For 2D models with neat-fit pins ($d_p/d = 0.999$), the maximum von Mises stress in the plate occurs at 146.2° , 144.4° , 146.2° on the contact surface. In the same models, the maximum von Mises stress in the pin occurs at 114.7° , 112.5° , 114.7° on the contact surface. For 3D models with neat-fit pins, the maximum von Mises stress occurs on the contact surface at 143.2° in the plate middle surface (except in 3dm16 it is 139.1°) and at varying directions within 126.8° and 139.1° in the plate front surface.
- As the pin-to-hole diameter decreases, the stress concentration increases at the contact origin point and the maximum stress location in the plate shifts toward 90° from horizontal axis in both 2D and 3D models. The maximum von Mises stress in the pin is located at 90° for models with $d_p/d = 0.98$ and $d_p/d = 0.90$.
- Increasing clearance between the pin and the plate results in similar but slightly

larger increases in 3D models compared to 2D models. In 2D models, changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.98$ increases the maximum SVM stress in the plate by a factor of 1.08–1.11 and in the pin by a factor of 2.07–2.20 depending on the value of a/c . And changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.90$ increases the maximum SVM stress in the plate by a factor of 1.95 – 2.11 and in the pin by a factor of 4.32 – 4.58 depending on the value of a/c . In 3D models, changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.98$ increases the maximum SVM stress in the plate middle surface by a factor of 1.12 – 1.17 and in the pin middle surface by a factor of 2.33 – 2.66 depending on the value of a/c . And changing clearance from $d_p/d = 0.999$ to $d_p/d = 0.90$ increases the maximum SVM stress in the plate middle surface by a factor of 1.94 – 2.17 and in the pin middle surface by a factor of 4.76 – 5.68 depending on the value of a/c .

- In 2D models, the maximum von Mises stresses in both plate and pin are located on the contact surface in all models except the ones with a large clearance ($d_p/d = 0.90$) where the maximum stresses in the plate and pin are located 2.3 – 2.5 mm and 2.7 mm from the contact surface, respectively. For 3D models with neat-fit pin, the maximum von Mises stresses occur on the contact surface of the plate and pin. For other models, the maximum von Mises stresses in the plate are located 0 – 2.8 mm from the contact surface in the plate middle surface and 0 – 2.9 mm from the contact surface in the plate front surface whereas the maximum von Mises stress in the pin is located about 2.0 – 11.1 mm away from the contact surface in the pin middle surface and about 0 – 2.2 mm away from the contact surface in the surface of the pin corresponding to the plate front (or back).
- Changing a/c ratio has a more pronounced effect on the plate in both 2D and 3D models. In 3D models, the maximum von Mises stress in the plate increases in general when a/c is decreased to 0.5, but this increase becomes less significant as d_p/d ratio decreases from 0.999 and 0.90. The change in the maximum von Mises stress in the plate front surface is between +7.2% and +7.5% for models with $d_p/d = 0.999$, between +0.1% and +4.4% for models with $d_p/d = 0.98$, and between +0.2% and +2.1% for models with $d_p/d = 0.90$. The change in the maximum von Mises stress in the plate middle surface is between +6.4% +7.1% for models with $d_p/d = 0.999$, +9.2% for models with $d_p/d = 0.98$, and +1.2%

for models with $d_p/d = 0.90$. When a/c is increased to 1.5, the maximum von Mises stress decreases for the model with neat-fit pin ($d_p/d = 0.999$) and no pin extension ($t_e/t = 0$) whereas it may decrease or increase to a lesser degree for the other models. The change in the maximum von Mises stress in the plate front surface is between -2.0% and -2.3% for models with $d_p/d = 0.999$, between -0.7% and $+0.2\%$ for models with $d_p/d = 0.98$, and between $+0.1\%$ and $+0.2\%$ for models with $d_p/d = 0.90$. The change in the maximum von Mises stress in the plate middle surface is -2.6% for models with $d_p/d = 0.999$, between -2.2% and -2.7% for models with $d_p/d = 0.98$, and $+0.2\%$ for models with $d_p/d = 0.90$.

- For 3D models with pin extension ($t_e > 0$), bending of the pin over the plate edges causes significant stress increase at those locations. The stress concentration is more significant for models with larger clearance (i.e., $d_p/d = 0.98$ and $d_p/d = 0.9$). When other parameters are fixed, the difference between the results of $t_e/t = 0.25$ and $t_e/t = 0.5$ is observed to be not much. But compared to models with no pin extension ($t_e = 0$), the maximum von Mises stress in the plate front surface increases about a factor of 1.18 – 1.20 for neat-fit models with $d_p/d = 0.999$, about a factor of 1.55 – 1.68 for clearance models with $d_p/d = 0.98$, and about a factor of 1.51 – 1.57 for large clearance models with $d_p/d = 0.90$. Pin extension has a less effect on maximum von Mises stress in the plate middle surface, within -5.1% and $+0.5\%$ of the maximum stress values of the corresponding models with no pin extension. Having pin extension decreases the maximum von Mises stress results about 8.4 – 20.8% in the pin middle surface and about 8.1 – 24.4% in the surface corresponding to the front of the plate with respect to the models having no pin extension.
- Three of the 2D models (2dm1, 2dm2, 2dm3) and three of the 3D models (3dm1, 3dm2, 3dm3) that have no pin extension $t_e/t = 0$ and have the same edge distance in the axial and transverse directions ($a/c = 1$) are also analyzed in Abaqus to compare the Calculix results with a commercial FEA software and analytical solutions of Hertz and Persson. Maximum values of S_{YY} and von Mises stress are examined for the comparison. The first 3D model (3dm1) has 20.3% and 7.6% higher maximum S_{YY} stress and 2.5% and 13.6% higher maximum von Mises stress in the plate and in the pin, respectively, compared to 2D models. For 3dm2, maximum

SYY is higher about 2.9% in the plate and 0.2% in the pin while maximum von Mises stress is higher about 6.8% in the plate and 6.6% in the pin, compared to 2D models. For 3dm3, maximum SYY is higher about 0.2% in the plate and 0.1% in the pin while maximum von Mises stress is higher about 14.2% in the plate and 10.8% in the pin, compared to 2D models. For 2D models, Calculix maximum |SYY| results are 0.0% to 3.2% higher than Abaqus results, and the difference increases with the pin-to-hole diameter ratio. On the other hand, Calculix maximum SVM results are 1.7 to 28.8% less than the corresponding values in Abaqus. This discrepancy is again most likely due to the conversion of 2D elements to 3D elements internally in Calculix. Calculix maximum |SYY| results are also compared with the maximum contact stress given by analytical solutions of Hertz and Persson. For the model with neat-fit pin (2dm1), Calculix maximum |SYY| result in the plate is 55.1% higher than Hertz's solution and 5.3% less than Persson's solution. For the larger clearance models (2dm2, 2dm3), Calculix maximum |SYY| result in the plate is 1.7 to 2.3% less than Hertz's solution and 1.3 to 2.0% less than Persson's solution. For 3D models, the results have a much better match between Calculix and Abaqus models for both |SYY| and SVM values. Calculix maximum |SYY| results are within -0.6% and $+0.9\%$ of Abaqus results, while Calculix maximum SVM results are within -1.2% and $+1.4\%$ of the corresponding values in Abaqus. Calculix maximum |SYY| results are also compared with the maximum contact stress given by analytical solutions of Hertz and Persson. For the model with neat-fit pin (3dm1), Calculix maximum |SYY| result in the plate is 86.6% higher than Hertz's solution and 13.9% higher than Persson's solution. For the larger clearance models (2dm2, 2dm3), Calculix maximum |SYY| results in the plate are within -1.5% and $+0.6\%$ of Hertz's solution and within -1.8% and $+1.5\%$ of Persson's solution.

Based on the results of analyses performed in this study, it is found that pin-to-hole ratio and bending of pin can have significant effects on stress concentration in pin connections under transverse loading while axial eccentricity (a/c) has a limited effect. Although two-dimensional models are useful in preliminary assessment and model verification, they ignore thru-thickness effects that may be significant in some cases; therefore, three-dimensional models provide more accurate representation of stress

distribution in the pin connections. The analyses also confirm that the analytical solutions of both Hertz and Persson gave reasonable results for smaller contact angles when pin bending effect is not present while Hertz's solution deviates significantly from the results of finite element models with neat-fit pin having a more conformal contact surface with larger contact angles.



REFERENCES

- [1] AISC 360-16, *Specification for structural steel buildings*, American Institute of Steel Constructions, Chicago, 2016.
- [2] EN 1993-1-8, *Eurocode 3: Design of steel structures - part 1-8: Design of joints*, European Committee for Standardization, Brussels, 2006.
- [3] Calculix, “A free software three-dimensional structural finite element program,” July 2022. [Online]. Available: <http://www.calculix.de/>
- [4] I. Harms, “Pinned connections,” Master’s thesis, Delft University of Technology, Delft, the Netherlands, 2015.
- [5] G. Schaper, *Eisernen Brücke*. Berlin: Verlag Wilhelm Ernst & Sohn, 1908.
- [6] F. Bleich, *Theorie und Berechnung der eisernen Brücken*. Springer, 1924.
- [7] H. Reißner and F. Strauch, “Ringplatte und augenstab,” *Ingenieur-archiv*, vol. 4, no. 5, pp. 481–505, 1933.
- [8] W. Reidelbach, “Der spannungszustand in einem durchbohrten, durch eine bolzenkraft auf zug beanspruchten stangenkopf,” *Das Industrieblatt*, vol. 62, no. 11, 1962.
- [9] A. Poócza, *Zur Festigkeitsberechnung geschlossener Stangenköpfe*. Berlin: Springer, 1967.
- [10] H. Hertz, “On the contact of elastic solids,” *Journal fur die reins and angewandte Mathematik*, no. 92, pp. 156–171, 1881.
- [11] H. Hertz, “Ueber die berührung fester elastischer körper.” 1882.
- [12] H. Hertz, *Miscellaneous papers*. Macmillan, 1896.
- [13] A. Persson, “On the stress distribution of cylindrical elastic bodies in contact,” Ph.D. dissertation, Chalmers University of Technology, Tehniska, Goteborg, Sweden, 1964.
- [14] B. Noble and M. Hussain, “Exact solution of certain dual series for indentation and inclusion problems,” *International Journal of Engineering Science*, vol. 7, no. 11, pp. 1149–1161, 1969.
- [15] M. Ciavarella and P. Decuzzi, “The state of stress induced by the plane frictionless cylindrical contact. i. the case of elastic similarity,” *International Journal of Solids and Structures*, vol. 38, no. 26-27, pp. 4507–4523, 2001.

- [16] M. Ciavarella and P. Decuzzi, "The state of stress induced by the plane frictionless cylindrical contact. ii. the general case (elastic dissimilarity)," *International Journal of Solids and Structures*, vol. 38, no. 26-27, pp. 4525–4533, 2001.
- [17] J. Dundurs, "Discussion: Edge-bonded dissimilar orthogonal elastic wedges under normal and shear loading," *Journal of Applied Mechanics*, pp. 460–466, 1969.
- [18] J. Dundurs, "Properties of elastic bodies in contact," *The mechanics of the contact between deformable bodies*, pp. 54–66, 1975.
- [19] K.L. Johnson, *Contact mechanics*. Cambridge university press, 1985.
- [20] M. Fagan and J. McConnachie, "A review and detailed examination of non-layered conformal contact by finite element analysis," *The Journal of Strain Analysis for Engineering Design*, vol. 36, no. 2, pp. 177–195, 2001.
- [21] E. Steuermann, "To hertz's theory of local deformations in compressed elastic bodies," in *CR (Dokl.) Acad. Sci. URSS*, vol. 25, no. 5, 1939, pp. 359–361.
- [22] D. Duerr, "Pinned connection strength and behavior," *Journal of Structural Engineering*, vol. 132, no. 2, pp. 182–194, 2006.
- [23] B.G. Johnston, "Pin-connected plate links," *Transactions of the American Society of Civil Engineers*, vol. 104, no. 1, pp. 314–336, 1939.
- [24] G. Blake, "Structural tests of large pin-connected links," *US Steel Research, Monroeville, Pa*, 1981.
- [25] M. Frocht and H. Hill, "Stress-concentration factors around a central circular hole in a plate loaded through a pin in the hole," *Journal of Applied Mechanics*, vol. 7, no. 1, pp. A5–A9, 1940.
- [26] R. Scott and J. Stone, "Effects of design variables on the critical stresses of eye bars under load: An evaluation by photoelastic modeling," Lawrence Livermore National Lab., CA (USA), Tech. Rep., 1982.
- [27] C.O. Rex and W.S. Easterling, "Behavior and modeling of a bolt bearing on a single plate," *Journal of Structural Engineering*, vol. 129, no. 6, pp. 792–800, 2003.
- [28] J. Ekvall, "Static strength analysis of pin-loaded lugs," *Journal of Aircraft*, vol. 23, no. 5, pp. 438–443, 1986.
- [29] ASME BTH-1, *Design of Below-the-Hook Lifting Devices*, The American Society of Mechanical Engineers, New York, 2017.
- [30] A. Blake, "Structural pin design," *DESIGN NEWS*, vol. 29, no. 2, p. 88, 1974.
- [31] M. Melcon and F. Hoblit, "Developments in the analysis of lugs and shear pins," *Product Engineering*, vol. 24, pp. 60–170, 1953.

- [32] EN 13001-3-1, *Cranes - General Design - Part 3-1: Limit States and proof competence of steel structure*, Nederlands Normalisatie-instituut, Delft, 2012.
- [33] R. Grant, J. Smart, and P. Stanley, “A parametric study of the elastic stress distribution in pin-loaded lugs,” *The Journal of Strain Analysis for Engineering Design*, vol. 29, no. 4, pp. 299–307, 1994.
- [34] R. Grant and B. Flipo, “Parametric study of the elastic stress distribution in pin-loaded lugs modelled in two and three dimensions and loaded in tension,” *The Journal of Strain Analysis for Engineering Design*, vol. 44, no. 6, pp. 473–489, 2009.
- [35] *Specification for the Design, Calculation and Construction of Steel Structures*, Turkish Ministry of Environment and Urban Planning, 2018.
- [36] P. Wriggers and T.A. Laursen, *Computational contact mechanics*. Springer, 2006, vol. 2.
- [37] N.H. Kim, *Introduction to nonlinear finite element analysis*. Springer Science & Business Media, 2014.
- [38] A. Konyukhov and R. Izi, *Introduction to computational contact mechanics: a geometrical approach*. John Wiley & Sons, 2015.
- [39] K. Wittig, *CalculiX GraphiX User’s Manual - Version 2.20*, 2022.
- [40] G. Dhondt, *CalculiX CrunchiX User’s Manual - Version 2.20*, 2022.