Jurnal Rekayasa Mesin
p-ISSN: 1411-6863, e-ISSN: 2540-7678
Vol.20, No.3, December **2025**, p. 359-368

Mechanical Contact Stress Analysis of Rollers at Various Hot Rolling Reductions Using Finite Element Approach

Agung Fauzi Hanafi[1], Ansor Salim Siregar*, Prabuditya Bhisma Wisnu Wardhana, Mega Lazuardi Umar
Mechanical Engineering, Politeknik Negeri Banyuwangi, Jl. Raya Jember No. KM13, Banyuwangi, East Jawa, 68461, Indonesia.
*E-mail: ansorsalim@poliwangi.ac.id

Submitted: 22-11-2025; Accepted: 04-12-2025; Published: 30-12-2025

Abstract

Hot rolling is a fundamental process in metal manufacturing; however, the operational integrity of the rollers is frequently threatened by excessive stress, which can lead to premature fatigue failure. This study aims to quantitatively analyze the effect of the cross-sectional reduction percentage on the distribution and magnitude of equivalent (von Mises) stress in hot rolling rollers. The Finite Element Analysis (FEA) method was employed to simulate the rolling process on rollers made of Ductile Cast Iron and a Structural Steel billet at a temperature of 800°C. Eleven simulation scenarios were executed by varying the reduction from 22% (industry standard) to 44%. The results indicate that the standard 22% reduction yields a very safe stress level (157.36 MPa), which is only 46.5% of the roller material's fatigue strength (338 MPa). It was found that roller stress increases non-linearly with increasing reduction, with a significant stress surge observed after a 41% reduction. The maximum safe operational limit was identified at a 43% reduction, which produced a stress of 275.43 MPa (81.5% of the fatigue limit). At a 44% reduction, the roller stress (369.37 MPa) exceeded the safe limit, indicating a high risk of component failure. This study provides a practical quantitative guide for the industry to optimize production throughput by establishing 43% as the maximum theoretical reduction limit.

Keywords: *ductile cast iron; fatigue analysis; finite element analysis (FEA);hot rolling; roller stress.*

Abstrak

Pengerolan panas (hot rolling) merupakan proses fundamental dalam manufaktur logam; namun demikian, integritas operasional rol sering kali terancam oleh tegangan berlebih, yang dapat menyebabkan kegagalan lelah (fatigue failure) dini. Penelitian ini bertujuan untuk menganalisis secara kuantitatif pengaruh persentase reduksi penampang terhadap distribusi dan besaran tegangan ekuivalen (von Mises stress) pada rol hot rolling. Metode Finite Element Analysis (FEA) digunakan untuk mensimulasikan proses pengerolan pada rol yang terbuat dari Besi Cor Nodular (Ductile Cast Iron) dan bilet Baja Struktural pada temperatur 800°C. Sebelas skenario simulasi dijalankan dengan memvariasikan tingkat reduksi mulai dari 22% (standar industri) hingga 44%. Hasil penelitian menunjukkan bahwa reduksi standar sebesar 22% menghasilkan tingkat tegangan yang sangat aman (157,36 MPa), yang hanya mencapai 46,5% dari kekuatan lelah material rol (338 MPa). Ditemukan bahwa tegangan pada rol meningkat secara non-linear seiring dengan peningkatan reduksi, di mana lonjakan tegangan yang signifikan teramati setelah reduksi mencapai 41%. Batas operasional aman maksimum teridentifikasi pada tingkat reduksi 43%, yang menghasilkan tegangan sebesar 275,43 MPa (81,5% dari batas lelah). Pada tingkat reduksi 44%, tegangan rol (369,37 MPa) melampaui batas aman, yang mengindikasikan risiko tinggi terhadap kegagalan komponen. Studi ini memberikan panduan kuantitatif praktis bagi industri untuk mengoptimalkan throughput produksi dengan menetapkan 43% sebagai batas reduksi teoritis maksimum.

Kata kunci: besi cor nodular; analisis kelelahan; *finite element analysis* (FEA); pengerolan panas; tegangan rol.

1. Introduction

The hot rolling process is a fundamental pillar of the modern metal manufacturing industry. This process is essential for transforming raw materials, such as billets or slabs, into semi-finished or finished products with desired shapes and mechanical properties, including rebar, plates, and structural profiles. The efficiency, precision, and reliability of the hot rolling process directly impact the final product quality and industrial profitability [1]. A primary challenge in hot rolling operations is maintaining the integrity and service life of the rollers themselves. During the process, rollers are subjected not only to extreme mechanical loads from reduction forces but also to severe thermal cycling (heat from the billet and subsequent cooling). This combination of mechanical and thermal stress can lead to rapid wear, fatigue cracking [2][3], or even catastrophic roller failure. Failure analysis studies indicate that thermal fatigue and overstress are common causes of roller damage, resulting in costly production downtime and increased maintenance expenses[4].

To understand and mitigate these risks, Finite Element Analysis (FEA) has been widely adopted as an effective method for predicting and analyzing stress conditions on rollers. Research in recent years has utilized FEA for various purposes. Some studies use 3D coupled thermal-mechanical simulations to analyze stress evolution and its impact on billet microstructure [1]. Other research focuses on modeling wear mechanisms and predicting stress concentration zones on the rollers to optimize maintenance schedules [4]. Furthermore, FEA has also been used to optimize the roller groove design to achieve more uniform material deformation and reduce roller stress . Other studies also focus on optimizing roll pass design to improve product geometric quality[5].

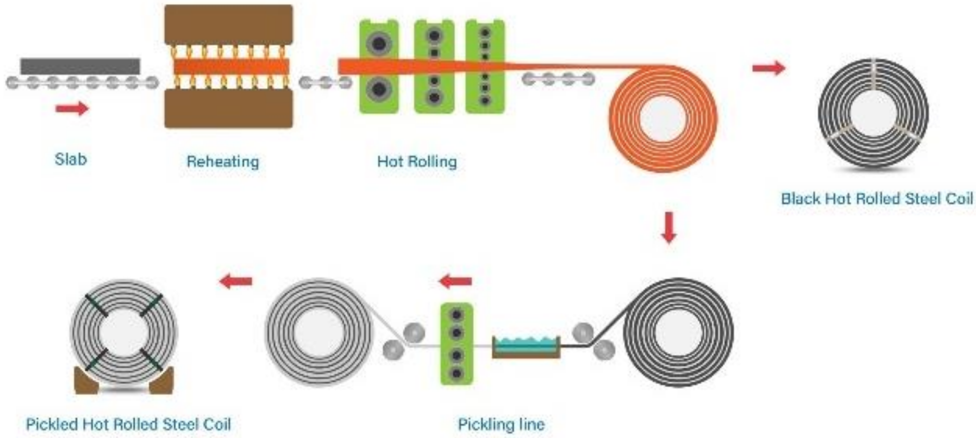


Figure 1. Schematic diagram of the hot rolling process.

Although these studies have provided valuable insights, many focus on optimizing the final product (billet) or analyzing specific failure mechanisms. There remains a need for quantitative analysis that directly links the key process parameter—namely, the percentage reduction—with the distribution of equivalent (von Mises) stress on the roller. Specifically, determining the safe operational reduction limit for a specific material combination, such as ductile cast iron rollers and a structural steel billet at 800°C, has not been deeply explored to find the optimal point between production efficiency (high reduction) and roller service life (safe stress).

Therefore, this study aims to fill this gap by systematically analyzing the effect of hot rolling product reduction variations on the stress experienced by the roller using Finite Element Analysis. This research specifically aims to: (1) Develop an accurate 3D FEA simulation model of the rolling process; (2) Analyze the distribution and magnitude of equivalent stress on the roller at the standard reduction rate (22%); and (3) Determine the maximum safe reduction percentage limit before the stress on the roller exceeds the material's fatigue strength. The results of this study are expected to provide practical recommendations for process [6] optimization in the metal manufacturing industry.

It is important to note that this study focuses specifically on the mechanical contact stresses generated by the reduction process. To isolate the effect of reduction percentage on roller integrity, the simulation assumes isothermal conditions, and thermal stresses due to temperature gradients are excluded from the current analysis scope.

2. Methods

This study utilizes a Finite Element Analysis (FEA) approach to simulate the hot rolling process and analyze the stress on the rollers. The research methodology is divided into three main stages: (1) Geometric and material modeling, (2) Finite Element (FEM) simulation setup, and (3) Analysis of reduction variation parameters.

2.1 Geometric Modeling and Materials

A 3D geometric model of the rollers and billet was created using Solidworks 2021 CAD software. The roller groove design and billet dimensions (input and output) were made to resemble actual conditions in industrial facilities. The 3D model used is shown in Figure 2 , while the detailed groove geometry and billet cross-sections are shown in Figure 3.

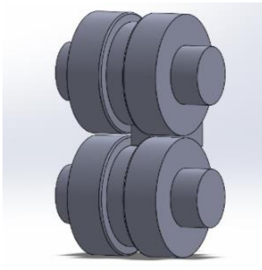
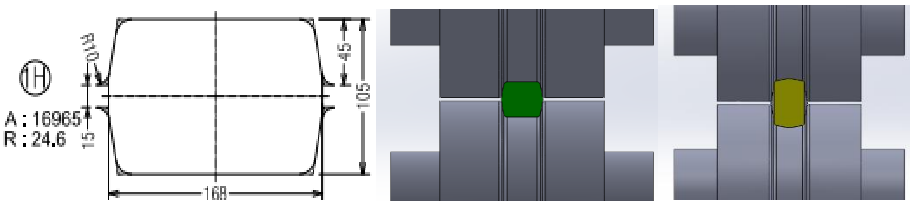


Figure 2. 3D geometric model of the rollers designed in Solidworks.



(a) (b) (c)

Figure 3. Detailed geometry definitions: (a) 2D schematic of the roll pass groove, (b) Input billet cross-section (A0), and (c) Output billet cross-section (A1).

Two main materials were defined in this simulation: (1) Roller: The roller material was defined as Ductile Cast Iron. This material selection is based on its common use in the rebar industry due to its good balance of strength, wear resistance, and cost. The mechanical properties of ductile cast iron used as simulation inputs are presented in Table 1, with the Fatigue Strength of 338 MPa being the critical reference parameter, and (2) Billet: The billet material was defined as Structural Steel at a process temperature of 800°C. The mechanical properties of this material at high temperatures, such as Yield Strength (40.425 MPa), are crucial for accurate plastic deformation modeling [7].

To accurately capture the material behavior, the Bilinear Isotropic Hardening plasticity model was used (shown in Figure 4), corresponding to the stress-strain curve of steel material at high temperatures. This model defines the stress-strain (σ-ε) relationship after it passes the yield strength (σ_y) using two linear slopes: one for the elastic area defined by the Modulus of Elasticity (*E*), and one for the plastic area defined by the Tangent Modulus (E_t) is presented in Equation (1).

$$\sigma = \begin{cases} E.\in & if\ \epsilon > \ \epsilon_y\ (Area\ Plastis) \\ \sigma_y + E_t.(\epsilon.\epsilon_y) & if\ \epsilon > \ \epsilon_y\ (Area\ Plastis) \end{cases} \quad (1)$$

Where the values for E, σ_y , and *E* for Structural Steel at 800°C are taken from Table 2.

Table 1. Mechanical Properties of Ductile Cast Iron (Roller).

Mechanical Property	Value
Yield Strength	627 MPa
Ultimate Tensile Strength	862 MPa

Mechanical Property	Value
Fatigue Strength	338 MPa
Hardness	300 HB
Density	7.300 Kg/m^3
Modulus of Elasticity	170.000 MPa

Table 2. Mechanical Properties of Structural Steel (Billet, 800°C).

Mechanical Property	Value
Yield Strength	40,425 MPa
Density	7.300 Kg/m^3
Modulus of Elasticity	18.900 MPa
Tangent Modulus	20

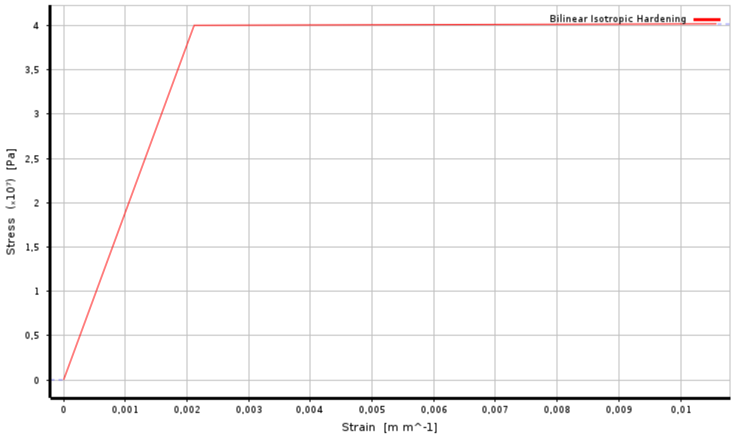


Figure 4. Bilinear Isotropic Hardening model defining the plastic behavior of structural steel at 800°C.

2.2 Finite Element Model (FEM)

SetupThis simulation aims to analyze the complex multi-axial stresses on the roller. To evaluate the material's safety limit and predict failure due to these combined stresses, the equivalent stress (von Mises) criterion was used as the primary output parameter. Von Mises stress (σ_v) calculates the total distortion energy in a material (Equation 2).

$$\sigma_V = \sqrt{\frac{1}{2}\left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_x\right)^2 + \left(\sigma_x - \sigma_y\right)^2 + 6\left(\tau_y^2 + \tau_{yz}^2 + \tau_{zz}^2\right)\right]} \qquad (2)$$

Where $\sigma_x, \sigma_y, \sigma_z$ is the normal stress component and $\sigma_{zy}, \sigma_{yz}, \sigma_{zx}$is the shear stress component. The analysis in this study will compare the maximum σ_V value generated on the roller with the material's Fatigue Strength (338 MPa).

The geometric model from Solidworks was imported into ANSYS Workbench software for Static Structural analysis. The FEM setup was as follows:

a) Contacts: Frictional contact was defined between the roller groove surfaces and the billet surfaces34. Accurate contact definition (shown in Figure 5) is critical in metal forming simulations to predict material flow and force transfer [8][9].

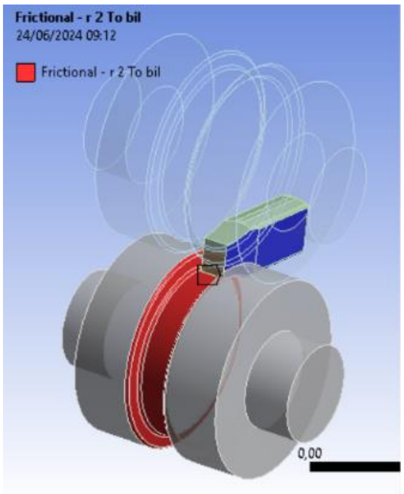


Figure 5. Frictional contact definition between the roller surface (target) and billet (contact).

b) Joints & Boundary Conditions: Joints were defined to replicate the machine's kinematics. As shown in Figure 6, a Joint-Rotation was applied to the rollers to simulate their rotation, while a Joint-Displacement was applied to the billet to simulate its forward motion into the rollers [10].

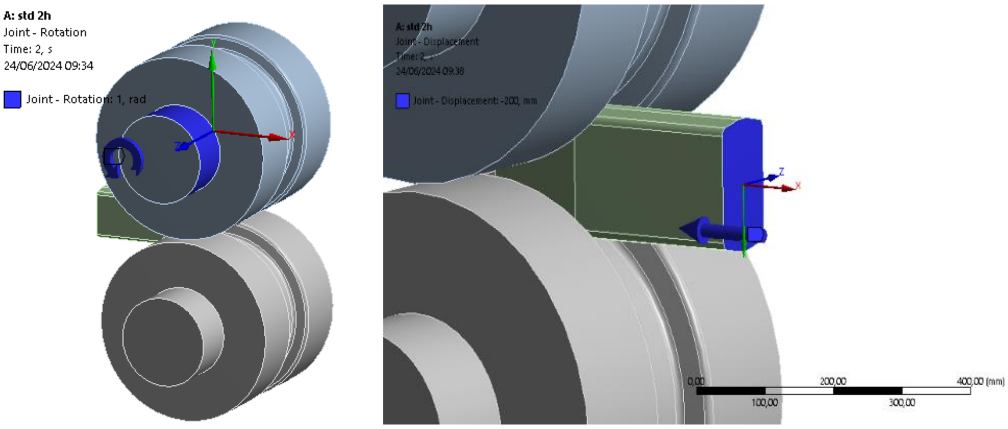


Figure 6. Boundary condition setup: (a) Rotation (Joint-Rotation) applied to the roller and (b) Displacement (Joint-Displacement) applied to the billet.

c) Meshing: A Hex Dominant meshing method with an element size of 15 mm was applied, particularly in the contact area between the roller and billet as seen in Figure 7. The use of hexahedral (hex) elements in critical deformation zones is known to improve the accuracy of stress results and reduce computational costs compared to pure tetrahedral elements.

The element size of 15 mm was selected to maintain a balance between computational efficiency and result accuracy. This discretization strategy aligns with recent large-scale hot rolling simulations [11], where mesh sensitivity analyses demonstrate that increasing mesh density beyond a critical point significantly increases computational cost (e.g., by over 90%) without providing substantial improvements in load prediction accuracy. Therefore, the chosen mesh size is considered sufficient to capture the global stress gradients in the billet.

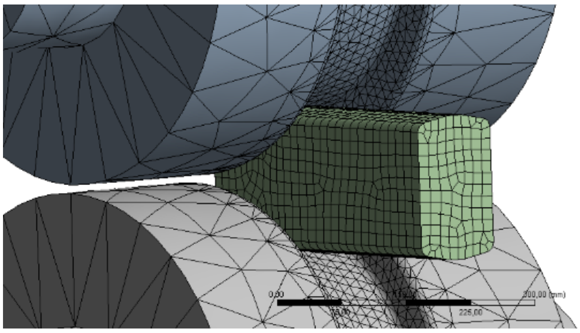


Figure 7. Hex Dominant meshing applied to the contact and deformation zones.

2.3 Simulation Parameters and Variations

The main independent variable investigated in this study is the percentage cross-sectional reduction (R). This parameter is mathematically defined as the ratio of the reduction in the initial billet cross-sectional area (A_0) to the final cross-sectional area after rolling (A_1), expressed as a percentage (Equation 3).

$$R = \frac{A_0 - A_1}{A_0} \text{ x } 100\% \quad (3)$$

Eleven simulation scenarios were run by varying the R value from 22% (industry standard) to 44%. The A_0 value was kept constant while the A_1 value was varied to achieve the desired reduction percentages, as summarized in Table 3.

3. Results and Discussion

3.1. Analysis of Standard Reduction (22%)

The first simulation was performed at the standard industrial reduction condition (R = 22%). The Static Structural simulation results show that the maximum equivalent (von Mises) stress occurring on the roller was 157.36 MPa. This finding is highly significant. When compared to the roller material's reference parameter (Table 1), this 157.36 MPa stress value is only about 46.5% of the material's Fatigue Strength (338 MPa). This indicates that the standard process operates with a very high Factor of Safety (FoS ≈ 2.15), which aligns with conservative industrial practices aimed at ensuring long component life and avoiding unexpected failures [12].

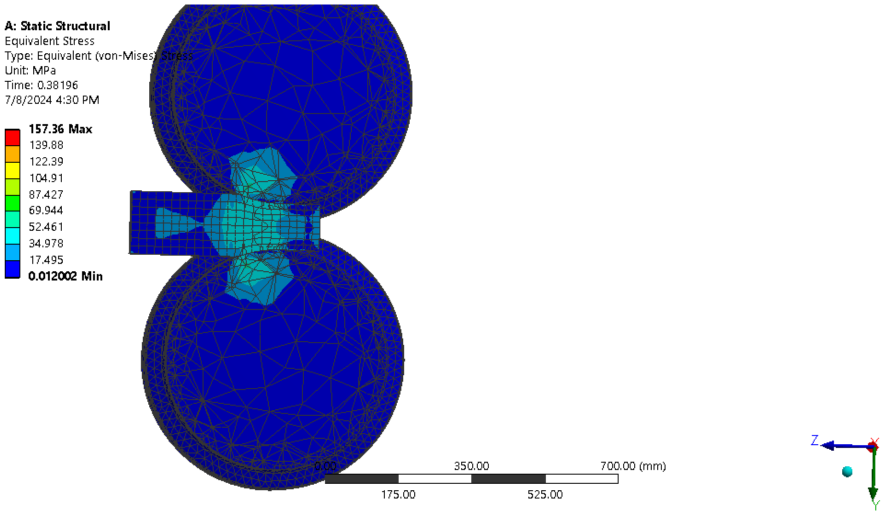


Figure 8. Equivalent (von-Mises) stress distribution on the roller at the standard 22% reduction (Max: 157.36 MPa).

3.2. Effect of Increased Reduction on Roller Stress

Eleven simulation scenarios were run to investigate the effect of increased reduction on roller stress. The quantitative results are summarized in Table 3 and graphed in Figure 9. The simulation results show a clear trend: the greater the material reduction value, the higher the stress generated on the roller [13]. However, this relationship is not linear.
The stress increase is relatively moderate in the 22% to 37% reduction range. A significant stress increase was observed after a 41% reduction. However, a significant stress surge was observed when the reduction exceeded 41%. Two critical data points were identified: 43% Reduction: Resulted in a roller stress of 275.43 Mpa and 44% Reduction: Resulted in a roller stress of 369.37 MPa.

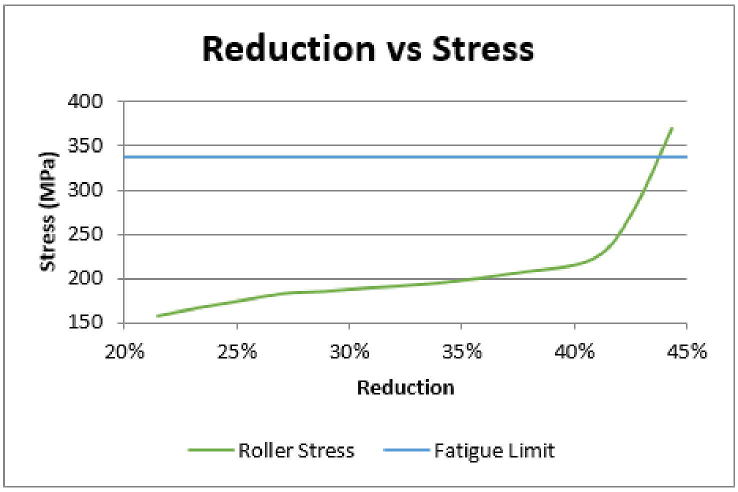


Figure 9. Effect of reduction percentage on maximum roller stress.

The physical mechanism driving this sudden stress surge at reductions exceeding 41% can be attributed to groove saturation and geometric constraint. As the reduction approaches 44%, the deforming billet material completely fills the available cross-sectional area of the roller groove. Since the structural steel behaves as an incompressible material during plastic deformation, the lack of remaining lateral space for material flow creates a condition similar to closed-die forging. This constraint leads to a sharp increase in hydrostatic pressure within the contact zone, resulting in the exponential spike in von Mises stress observed on the roller surface.

Table 3. Simulation Data Results for Reduction Variations.

No.	A_0 (mm^2)	A1 (mm^2)	R	σ (MPa)
1.	22422	17600	22%	157,36
2.	22422	17184	23%	166,84
3.	22422	16767	25%	174,91
4.	22422	16351	27%	182,79
5.	22422	15935	29%	185,4
6.	22422	15556	31%	188,75
7.	22422	14795	34%	194,85
8.	22422	14030	37%	206,02
9.	22422	13261	41%	222,54
10.	22422	12874	43%	275,43

No.	A_0 (mm^2)	A1 (mm^2)	R	σ (MPa)
11.	22422	12487	44%	369,37

3.3. Fatigue Safety Factor

Fatigue Safety Factor Assessment To address the cyclic nature of the mechanical loads on the rotating roller, a fatigue safety assessment was conducted using the Modified Goodman criterion. Since a specific point on the roller surface undergoes a load cycle from zero to maximum compressive stress during each rotation, the mean stress (σ_m) and alternating stress (σ_a) are defined as half of the maximum equivalent stress (σ_{max}).
Using the safety limit identified at 43% reduction (σ_{max} = 275.43 MPa) and the material properties of Ductile Cast Iron (S_s = 338 MPa, S_{ut} = 862 MPa), the Fatigue Factor of Safety (n) is calculated as follows Equation (4).

$$\frac{1}{n} = \frac{\sigma_a}{S_e} + \frac{\sigma_m}{S_{ut}} \tag{4}$$

Substituting the values where σ_a = σ_m = 137.71 Mpa,

$$\frac{1}{n} = \frac{137.71}{338} = \frac{137.71}{862} = 0.407 + 0.160 = 0.567, n \approx 1.76$$

This calculation confirms that even at the 43% reduction limit, the roller operates with a safety factor of 1.76 against fatigue failure, satisfying the standard engineering requirement ($n > 1.5$) for dynamic components. A theoretical validation using Hertzian contact theory was deemed unsuitable for this study. Recent analysis on roller bending processes [14] has highlighted that analytical formulas often fail to adequately describe material behavior during complex plastic deformation, leading to under- or overestimation of rolling forces. Since Hertzian solutions are limited to elastic contact, applying them to the hot rolling conditions in this study—which involve high plasticity and geometric confinement - would result in physically invalid predictions compared to the non-linear FEA approach.

4. Conclusion

This study successfully analyzed the effect of hot rolling reduction variations on the stress in ductile cast iron rollers using Finite Element Analysis. The standard process with 22% reduction produces 157.36 MPa of stress, which is very safe and well below the roller's fatigue strength (338 MPa). Roller stress increases non-linearly with increased reduction, with a significant stress surge observed after 41% reduction. The maximum safe operational limit was identified at a 43% reduction, which produces 275.43 MPa of stress (81.5% of the fatigue limit). Increasing the reduction to 44% causes the roller stress (369.37 MPa) to exceed the material's fatigue strength, potentially triggering component failure.
The implications of this study provide quantitative guidance for the industry to optimize production throughput. Increasing reduction from 22% to 41% can be done with relatively controlled risk. However, increases above 41% must be done with extreme caution, with an awareness of the drastically reduced roller service life. The main limitations of this study are its purely computational nature and its exclusion of thermal effects (thermal stress) [15]. Future research should focus on (1) Experimental validation of these simulation results through field measurements of roller wear [16][17], and (2) Development of a coupled thermal-mechanical simulation model to analyze the combined effects of mechanical and thermal stress [18].

References

[1] F. Yilmaz, M. A. Guvenc, and S. Mistikoglu, "Optimization of rolling forces in multi-pass hot rolling using MLR and ABC algorithm for enhanced product quality and energy efficiency," *Int. J. Adv. Manuf. Technol.*, vol. 139, no. 5–6, pp. 2409–2429, 2025, doi: 10.1007/s00170-025-16008-6.

[2] T. Moser, J. Seitz, E. Alp, and B. Kuhlenkötter, "Identification Of Investigation Procedures To Predict Work Roll Fatigue For Developing Machine Learning Applications – A Systematic Literature Review," *Proc. Conf. Prod. Syst. Logist.*, vol. 2, pp. 268–281, 2023, doi: 10.15488/15300.

[3] A. S. Siregar, M. Mulyadi, and S. Arief, "Analisis Kegagalan Laminasi Komposit Epoksi/Serat Karbon Pada Sayap Pesawat Tanpa Awak," *Pist. J. Tech. Eng.*, vol. 5, no. 2, p. 108, 2022, doi: 10.32493/pjte.v5i2.18596.

[4] K. Hu, R. Xue, Q. Shi, W. Han, F. Zhu, and J. Chen, "FEM simulation of thermo-mechanical stress and thermal fatigue life assessment of high-speed steel work rolls during hot strip rolling process," *J. Therm. Stress.*, vol. 45, no. 7, pp. 538–558, 2022, doi: 10.1080/01495739.2022.2080781.

[5] J. W. Lee, "A Design Study Using Simulation Techniques in Roll Form Production," Ph.D. dissertation, 2024, [Online]. Available: https://dspace.mit.edu/handle/1721.1/157160

[6] N. laras Agustina, "Metal Rolling by Computation Method : A Brief Review," *HBRP Publ.*, vol. 2, no. 3, pp. 1–9, 2020, doi: 10.5281/zenodo.3603763.

[7] Y. Yang, B. Zhang, Y. Wang, Z. Jiang, and K. Li, "Mechanical behaviors and constitutive model of structural steel influenced by strain aging," *J. Constr. Steel Res.*, vol. 192, no. November 2021, p. 107211, 2022, doi: 10.1016/j.jcsr.2022.107211.

[8] T. Pore, S. G. Thorat, and A. A. Nema, "Review of contact modelling in nonlinear finite element analysis," *Mater. Today Proc.*, vol. 47, no. xxxx, pp. 2436–2440, 2021, doi: 10.1016/j.matpr.2021.04.504.

[9] A. Zabala, E. S. de Argandoña, D. Cañizares, I. Llavori, N. Otegi, and J. Mendiguren, "Numerical study of advanced friction modelling for sheet metal forming: Influence of the die local roughness," *Tribol. Int.*, vol. 165, no. August 2021, 2022, doi: 10.1016/j.triboint.2021.107259.

[10] João Carlos L. Peixoto, Rafael L. Rangel, and Luiz F. Martha, "Interactive Modeling of NURBS for Isogeometric Analysis," 2024. doi: 10.55592/cilamce.v6i06.10229.

[11] A. Ojeda-López, M. Botana-Galvín, I. Collado-García, L. González-Rovira, and F. J. Botana, "Finite Element Simulation of Hot Rolling for Large-Scale AISI 430 Ferritic Stainless-Steel Slabs Using Industrial Rolling Schedules—Part 1: Set-Up, Optimization, and Validation of Numerical Model," *Materials (Basel)*, vol. 18, no. 2, 2025, doi: 10.3390/ma18020383.

[12] P. Odeyar, D. B. Apel, R. Hall, B. Zon, and K. Skrzypkowski, "A Review of Reliability and Fault Analysis Methods for Heavy Equipment and Their Components Used in Mining," *Energies*, vol. 15, no. 17, p. 6263, 2022, doi: 10.3390/en15176263.

[13] W. K. Liu, S. Li, and H. S. Park, "Eighty Years of the Finite Element Method: Birth, Evolution, and Future," *Arch. Comput. Methods Eng.*, vol. 29, no. 6, pp. 4431–4453, 2022, doi: 10.1007/s11831-022-09740-9.

[14] D. Boazu, I. Gavrilescu, and F. Stan, "Analytical and Finite Element Analysis of the Rolling Force for the Three-Roller Cylindrical Bending Process," *Materials (Basel)*, vol. 17, no. 21, 2024, doi: 10.3390/ma17215230.

[15] H. Liu, J. Zheng, Y. Guo, and L. Zhu, "Residual stresses in high-speed two-dimensional ultrasonic rolling 7050 aluminum alloy with thermal-mechanical coupling," *Int. J. Mech. Sci.*, vol. 186, no. May, p. 105824, 2020, doi: 10.1016/j.ijmecsci.2020.105824.

[16] U. S. Dixit, *Modeling of metal forming: a review*. Elsevier Series, 2020. doi: 10.1016/B978-0-12-818232-1.00001-1.

[17] E. Brusa, C. Delprete, and L. Giorio, “Smart Manufacturing in Rolling Process Based on Thermal Safety Monitoring by Fiber Optics Sensors Equipping Mill Bearings,” *Appl. Sci.*, vol. 12, no. 9, 2022, doi: 10.3390/app12094186.

[18] J. Kumar Singh Jadon, R. Singh, and J. Kumar Mahato, “Creep-fatigue interaction behavior of high temperature alloys: A review,” *Mater. Today Proc.*, vol. 62, pp. 5351–5357, 2022, doi: 10.1016/j.matpr.2022.03.487.