Tikrit Journal of Engineering Sciences (2025); 32(1): 1131
DOI: http://doi.org/10.25130/tjes.32.1.31

IRAQI Academic Scientific Journals

ISSN: 1813-162X (Print); 2312-7589 (Online)

Tikrit Journal of Engineering Sciences

available online at: http://www.tj-es.com

TJES Tikrit Journal of Engineering Sciences

Fatigue Life Prediction for an Extrusion Die Based on the Experimental Factors

Vivean Hassan Hammed [a], Mohammed Ali Nasser *[a], Ali Hassan Salih [b]

a Department of Applied Mechanics, Technical Engineering College – Baghdad, Middle Technical University, Baghdad, Iraq.
b Department of Power Mechanics Technologies, Basrah University College of Science and Technology, Basrah, Iraq.

Keywords:
Axial extrusion load; Extrusion die; Die fatigue life; Friction temperature; Product material.

Highlights:

- *Numerical predict the fatigue life performed using ANSYS 2019-R3 for an extrusion die depending on the highest loading factors obtained from the experimental work.*
- *These factors were generated during the variation of extrusion speed and changing product materials.*
- *Extrusion die used to convert a cylindrical bar with (19 mm) diameter into a square bar with (13 × 13) mm with an extrusion rate of 1.67.*

ARTICLE INFO

Article history:

Received	19 May 2023
Received in revised form	28 Dec. 2023
Accepted	21 Mar. 2024
Final Proofreading	20 Oct. 2024
Available online	28 Mar. 2025

Citation: Hammed VH, Nasser MA, Salih AH. **Fatigue Life Prediction for an Extrusion Die Based on the Experimental Factors**. *Tikrit Journal of Engineering Sciences* 2025; **32**(1): 1131.
http://doi.org/10.25130/tjes.32.1.31

***Corresponding author:**

Mohammed Ali Nasser

Department of Applied Mechanics, Technical Engineering College – Baghdad, Middle Technical University, Baghdad, Iraq.

Abstract: According to the fact that the practical method for finding the working fatigue life is very expensive and takes a longer time with a huge amount of samples till the failure initiation of the die. So, the present research describes the main purpose of predicting the fatigue life of the die numerically based on the highest loading factors getting from the experimental work, indicating how many numbers of product samples can be produced during the whole life of this designed die depending on these loading or over. This fatigue life analysis and assessment were adopted by ANSYS Workbench 2019 R3 software depending on the loading factors (axial extrusion load and friction temperature difference) affecting the die strength and fatigue life during the experimental extrusion process. These factors were generated according to the variation of extrusion speed and changing product materials. The extrusion die was used to convert a cylindrical bar with (19) mm diameter into a square bar with (13 × 13) mm with an extrusion rate of 1.67. The results showed that the highest stress intensity factor reached about (251.4 MPa) at the critical converge-diverge section due to the die profile design, and the die is safe and has an infinite life reaching up to (10^9 cycles), using Lead or Aluminum products taking the maximum extrusion load of (90 kN) and temperature difference of (2 °C) at 150 mm/min extrusion speed. While changing the product billet into harder materials, the die life decreased and might fail later.

التنبأ بعمر الاعياء لقالب بثق بالاعتماد على العوامل التجريبية

فيفيان حسن حميد، محمد علي ناصر، علي حسن صالح

[1] قسم هندسة الميكانيك التطبيقي/ الكلية التقنية الهندسة - بغداد / الجامعة التقنية الوسطى / بغداد – العراق.

[2] قسم تقنيات ميكانيك القوى / كلية البصرة الجامعة للعلوم والتكنولوجيا / البصرة – العراق.

الخلاصة

استنادا الى الحقائق التي تؤكد بأن الغرض من ايجاد عمر الكلال لقالب بثق عمليًا يكون مكلف جدًا ماديًا ويأخذ وقتًا طويلاً، ويتطلب استخدام عينات كثيرة لحين الحصول على بداية فشل القالب. وعليه فإن هذا البحث يقدم وصف للهدف الرئيسي من التنبأ عددياً بالعمر التشغيلي لقالب عدديًا بالاعتماد على أعلى العوامل المؤثرة من الاحمال الناتجة المستحصلة من التجارب العملية. وبذلك يُعطي انطباعاً اوليًا حول عمر الاعياء الكلي لتصميم القالب بالاعتماد على تلك الاحمال أو أعلى. هذا التحليل لعمر الاعياء والتقييم بني باستخدام برنامج الانسس ٢٠١٩, معتمدًا على الأحمال المؤثرة عمليًا عليه من قوة البثق المحورية والتغير في درجات حرارة الاحتكاك المؤثرة على متانة وعمر الاعياء للقالب خلال عملية البثق التجريبية. هذه العوامل تولدت استنادًا لتغير سرعة عملية البثق وتغيير نوع معدن المنتج. قالب البثق استخدم لغرض تغيير قضيب دائري المقطع بقطر ١٩ملم الى قضيب مربع المقطع بابعاد (١٣ملم - ١٣ملم) بمعدل تغير بثق ١,٦٧. أوضحت النتائج أن أعلى عامل شدة اجهاد يصل الى قرابة (251.4 ميكا باسكال) في مقطع التضيق والتوسع لشكل التصميمي للقالب وأن القالب أمين وبعمر تشغيلي طويل يصل إلى $١٠^٩$ دورة مع استخدام الرصاص والالمنيوم كمنتجات وبأخذ أعلى قوة بثق تصل الى (٩٠) كيلونيوتن وتغير بالحرارة درجتين مئوية خلال سرعة بثق (١٥٠) ملم/دقيقة. في حين تغيير عينات المنتج لمواد اكثر قساوة يقلل عمر القالب واحتمالية فشله لاحقًا.

الكلمات الدالة: حمل البثق المحوري، قالب البثق، عمر تعب القالب، درجة حرارة الاحتكاك، ومادة المنتج.

1.INTRODUCTION

According to the extrusion process, the die is the main part of the extrusion system, suffering from many factors during the process, such as mechanical loading, temperature rising, and friction wear. The cyclic accumulation of these factors during many extrusion processes can cause damage or fatigue failure to the die. In reality, die life is an important factor in designing and optimizing extrusion processes. Fatigue failure is one of the most important difficulties in die design engineering since it rises cumulatively with the number of applied loading cycles and can lead to fracture and damage of the considered part. As a result, fatigue life prediction is critical and must be taken into account during the design phase of the die system component because to assess practically the fatigue life of the die, it should be taken a huge amount of product till the die fails, and this takes a long time, which is very expensive in cost and high wasting of time. Many researchers spent efforts to numerically study and analyze the fatigue concept of the dies by conducting a finite element method analysis to obtain the contact pressure distribution at the workpiece–die interface for different geometries. The stress and strain analyses were conducted to evaluate the crack propagation at each loading cycle at each extrusion process. The Paris law and the values assumed by the stress concentration coefficient for different crack dimensions were used, and the die life for each tested extrusion die geometries was compared. In addition, the effectiveness of using an external ring to change the die stress state during the procedure was studied. The results of numerical simulations showed that optimizing the die's design significantly improved fatigue behavior. Sharp changes in material flow must be avoided because of the high contact pressure values, which have detrimental effects on fatigue behavior and surface wear. Moreover, excessive reduction ratios should be avoided. A multi-step procedure with low reduction ratios would be preferable. Using a stress ring improves the matrix fatigue behavior even more, particularly for longitudinal flaws. Finally, the die design, the most critical step in the manufacturing process through the design parameters, must be improved to reduce the impacts of damage mechanisms and prevent early failures [1]. Some researchers predicted the fatigue life of an extrusion die by Monte Carlo simulation. Paris law model has been used to estimate the die life in terms of the number of cycles until the failure, which is the number of extruded billets. The value of fracture toughness (KIC) of H13 hot work die steel under normal tempering and operating temperature circumstances was determined using a previous prediction model created by researchers. A Weibull probability model with shape parameter (m = 4.4) and scale parameter (850 cycles) appropriately represented the simulated die-life observations denoted by the billets. Also, the discussed technique may readily be extended to predict the life of extrusion dies made of tool steels that have undergone various tempering procedures and operated at various operating temperatures [2]. Others presented a technique for optimizing the die fatigue life using a rational metal-forming system design so that the die stress was optimal and the die design was the best in terms of its service life. The S–N technique was used to evaluate die fatigue life. To realize this idea, combined billet plastic flow and die deformation modeling throughout the forming process was used to determine the die stress. The die stress is optimized by combining metal-formed product, die, and process configuration in a reasonable way. As a result, the ideal die life was calculated. In addition, a framework for implementing this technique was established, and case studies were utilized to verify and validate the methodology [3]. Other researchers employed Morrow's stress and strain-life damage models for axisymmetric flat extrusion dies. The extrusion process was simulated with the aid of the finite element software ABAQUS, and dynamic stress and strain values were derived by first locating the probable fatigue region in the die. The damage model's applicability was assessed for a hot extrusion die consisting of H13 steels and billet material Al-6063. Different simulation runs were done to study the influence of process and design factors on the useable die life, using temperature and strain rate as process parameters and bearing length and fillet radius of the die as geometric features. Morrow's stress life model perfectly matched between computed and real die failure. Die life was observed to increase with higher temperatures and decrease with higher strain rates, most likely due to billet flow stress, which decreased with temperature and increased with strain rate. High-flow stress causes high die stresses/strains, making the product more prone to short life. Because there is higher resistance to flow at the die entry, a small die fillet radius drastically lowers die life due to billet flow stress, decreasing with temperature and increasing with strain rate. Also, long bearing length reduces the die life [4]. Researchers performed a FEM model that has been built for the extrusion of a square cross-section from a round billet using different die profiles. Aluminum-1100 type was used as the work material. The extrusion load discovered in the cosine die was low, and the increase in load was more gradual than in the taper die. The die profile ~~is~~ important in ensuring a smooth material flow and developing a homogeneous microstructure. So, it was noticed that the quality of the extruded products was higher in converging dies than in flat-faced square dies. Non-linear converging dies provided various advantages over linear converging dies in terms of extrusion load reduction, flow characteristics improvement, and microstructure evolution. With zero velocity discontinuities at the exit and entrance planes, the cosine die profile was the most perfect die profile. Moreover, a cosine die profile for the extrusion of square sections from circular billet was constructed, and it proposed that lead and aluminum materials be numerically modeled by utilizing DEFORM-3D software [5]. Moreover, researchers used hot extrusion punches where the loading history of the dangerous part of the die was obtained by finite element numerical simulation. The load spectrum modified and normalized through the Neuber algorithm was used as the loading curve of fatigue analysis. The tensile properties of the die material at service temperature were obtained by experiments based on the Manson-Coffin model. The material's strain-life curve was obtained through theoretical deduction. Aiming at specific working conditions of aluminum alloy extrusion parts, the die's fatigue life prediction and model validation were conducted. The results showed that the die life predicted by the model was 1318 times, compared to the actual die life was 1120 times. The die's failure position and service life agreed with the predicted results. A new way was provided for optimizing the matching of heat treatment and service conditions of the die and has managerial significance for actual production. Moreover, the actual production verification of the prediction results was in good consistent. The numerical model was suitable for predicting the fatigue life of the hot extrusion die, and it was applied to the fatigue analysis of the hot forging die and cold extrusion die [6]. Newly, researchers predicted the load life of cold-forging dies by combining the experimental fatigue life behavior with numerical analysis. The experimental work designed and manufactured the upper and lower jaws considering the geometrical shapes that contained a stress concentration. Also, low and high cycle fatigue curves were obtained for the die jaws material (CK 45). The fatigue life of the cold forging die was numerically simulated depending on the highest experimental applied load used to form the circular sheet metal types (lead, Brass, AL, and steel) and their thickness (varied from 2 to 6 mm). The fatigue simulation adopted by ANSYS workbench 2019 was based on the fatigue life behavior of the jaws' materials obtained experimentally under the forging operation conditions in the simulation solution. The results showed that the upper jaw's fatigue life was higher than the lower jaw due to its geometrical design shape, and the fatigue life of the jaws decreased with increasing the applied load. The lower jaw showed that the bottom zone of the inner dome might be the region of starting failure in the die. The predicted life is influenced by the sheet metal, thickness, and jaw shape. Finally, the numerical analysis can indicate the failure locations of the die and its fatigue life [7]. In this research, predicting the fatigue life of an extrusion die is numerically obtained, based on the affecting factors obtained from the experimental extrusion process. The main factors (axial extrusion load and friction temperature) influence the die's strength and fatigue life. These factors were generated during the variation of extrusion speed and changing product materials using a die to convert a round bar with (19 mm) diameter into a square bar with (13 × 13mm).

2.DIE SYSTEM AND PRODUCT DESIGN

2.1.Die System Design

The extrusion die system assembles components using a cold manufacturing process to form a square cross-section from the circular section bar without heating. This process is done by applying localized compressive load. The die system should be withstanding against the applied load depending on the complicated shape and selected material of the parts. The designed and manufactured die system comprised four main parts, as shown in Fig. 1.

1- **Ram:** It is used as a punch to push the billet into the container, which is connected directly to the die.
2- **Container:** It occupied the product sample and fixed with the die to give the path toward the die by the ram pushing.
3- **Die:** It is the main important part that produced the shape and form of the product, giving a bulk deformation transformation from a cylindrical bar to a square bar.
4- **Bolts:** An auxiliary component consisted of four M8 bolts. L-key types were used to fix the die with the container.

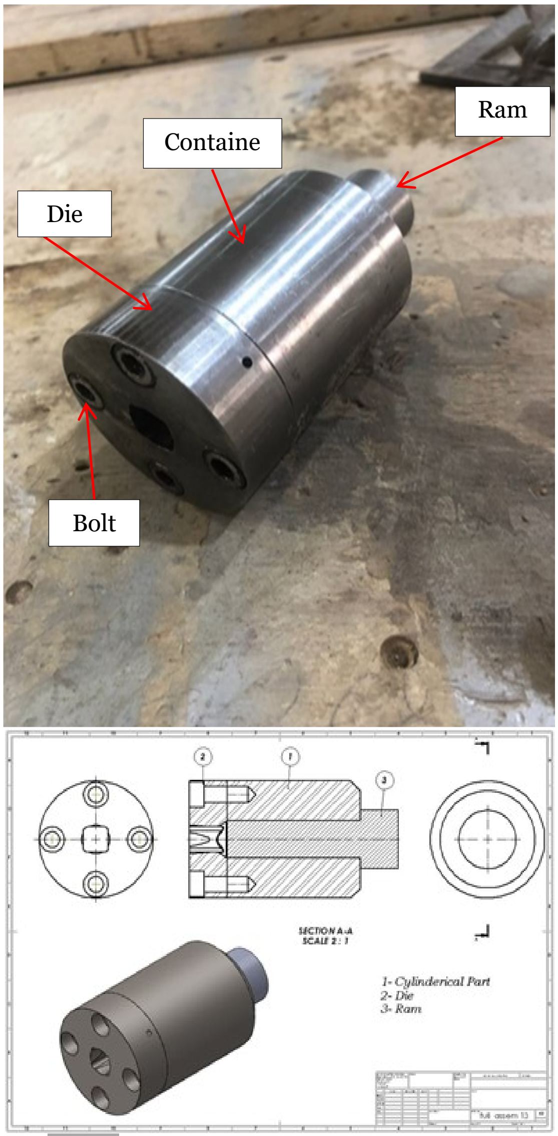


Fig. 1 The Designed and Manufactured Assembly of the Extrusion Die System.

2.2.*Product Design*

The product is designed to be a square bar generated from the bulking deformation of a cylindrical shaft during the extrusion process, which is the main goal of selecting this product shape design, to give the extrusion die a complicated design containing corners, flat edges, and fillets among the geometrical shape, which is the source of stress concentration in the die design. Therefore, according to this complicated die design, predicting the fatigue life is very important. The extrusion product design with dimension and the real manufacture product are shown in Fig. 2.

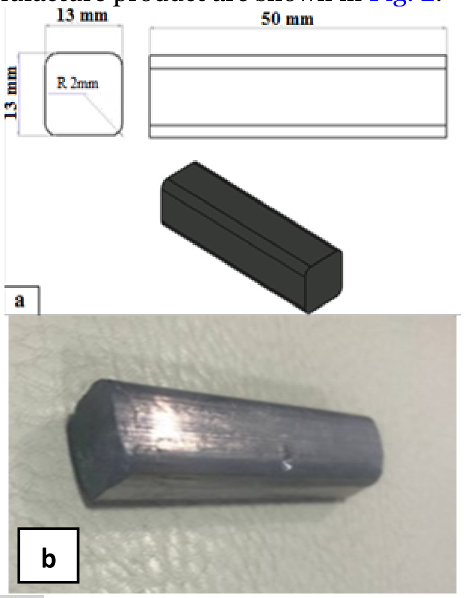


Fig. 2 The Extrusion Product Design (a) The Dimension of the Product, (b) the Real Manufacture Product for Al Material.

3.MATERIAL SELECTION

Selecting the material is made for the die system components and the product metal types. The material used in this research for manufacturing the die components was (XT 125 W Co 10 Cr Mo 4 V 3) according to IS: 7291-1981 Indian standard designation of high-speed tool steel (HSS) [8] and (High-speed tool steel grade C in German standards) [9]. Two types of material were used for extrusion products, cast Lead and pure Aluminum, due to their good mechanical properties and low cost. The verification of the materials was confirmed by the chemical composition and tensile testing to determine the practical mechanical properties of the materials. The tensile test specimens were cut from a steel rod with dimensions based on the testing standards E8/E8M-16a of the General Company for Engineering Inspection and Qualification. The chemical composition analysis is shown in Table 1 for the die material and Table 2 for product materials, while the mechanical characteristics analysis is shown in Table 3 for the die material and Table 4 for extrusion materials.

Table 1 The Standard and Tested the Chemical Composition of the Die Material (Tool Steel HSS) IS 7291-1981 Indian Standard.

Wt%	C%	Si%	Mn%	P%	S%	Cr%	Mo%	Ni%	Al%	Cu%	Co%	V%	W%	Fe%
Standard [6]	1.2 - 1.3	0.15 - 0.40	0.20 - 0.40	0.030	0.030	3.75 - 4.75	3.0 - 4.0				8.80 - 10.7	2.80 - 3.50	8.80 - 10.7	-
Tested	1.2	0.35	0.27	0.025	0.027	4.17	3.79	0.18	0.01	0.09	9.34	2.94	10.6	Bal

Table 2 The Tested Chemical Compositions of the Extruded Material (Al).

Wt%	Si%	Fe%	Cu%	Mn%	Mg%	Cr%	Ni%	Zn%	Ti%	V%	Pb%	Al%
Pure Al	0.134	0.167	0.0002	0.0018	0.011	0.002	< 0.001	< 0.001	0.005	0.005	0.005	99.7

Table 3 The Standard and Mechanical Properties of the Die Material (Tool Steel HSS).

Sample	Ultimate stress (σ_u) MPa	Yield stress (σ_y) MPa	Hardness Test HRC
Standard [9]	1200	1000	30
Tool steel test average	892	825	26

Table 4 The Mechanical Properties of the Selected Materials of the Extruded Product.

Material	Ultimate stress (σu) MPa	Yield strength (σy) MPa	Material
Pure Lead [10]	16.2	5.5	Pure Lead [10]
Pure Al Test average	112.5	84	Pure Al Test average

4.EXTRUSION PROCESS

The experimental work has been conducted for numerous rods made of Lead and Aluminum materials, called billets. The specimens were prepared for direct cold extrusion by machining a cylindrical solid rod with a length of 40 mm and a diameter of 19 mm. The machined specimens were directly cold extruded using a 30-ton vertical compression machine (UTM Universal testing machine) in the metallurgy Laboratory of the Technical Institute, as shown in Fig. 3. The extrusion die was made to generate rod product with a square cross-sectional area of (13×13 mm^2) and length of 70 mm from the cylindrical solid rod billet. The extrusion ratio was 1.67. The extrusion experiments were achieved at different ram speeds of 10, 50, and 150 mm/min. A cylindrical adapter was used for centering the container assembly and die with the ram to achieve the right alignment during the process. Also, the adapter had holes from two sides to remove the final product from the die system. The extrusion load during the vertical universal compression machine working was achieved by a load scale connected with the grip head used for fixing the ram. The load was computerized records and monitored graphically during the extrusion process depending on the pushing speed of the machine. The temperature generated during the extrusion, process resulting from the friction between the die and billets, was measured and monitored throughout the experimental work procedure. The measurement was done using a thermocouple type K (-50-1300 °C) in a hole made from outside at the critical section of the die. Also, a gun thermometer was pointed at the product's surface when it came out of the die to measure the surface temperature of the product. Moreover, a video recording monitor was made for the thermometer reading during all the extrusion processes to record the temperature developments with time, as shown in Fig. 4. The extrusion speed and friction temperature were the main experimental factors that influenced the extrusion process based on the variation of extrusion speed and types of product material, consequently, on the die strength as a combined effective load. This combined load can be transferred as the highest load into the numerical simulation to predict the die's fatigue life. The description of the extrusion tests showed that the highest loads and temperature obtained during the extrusion operation was using an Aluminum product with high speed at (150 mm/min), where the most detailed extrusion tests are shown in Table 5.

5.NUMERICAL ANALYSIS

It is one of the important methods for predicting and assessing the fatigue strength life of the cold extrusion die. The numerical analysis recommended specific structural modeling and simulation software, which are important to making the difficult design prediction possible. The die was modeled using Solidworks 2018 software, which has good facilities and tools for 3D drawings. While, ANSYS Workbench 2019R3 was utilized for meshing and fatigue life simulation.

5.1.Die Design Modelling

The modeling of this work concentrated on the extrusion die because it is the main part carrying almost all of the combined applied load during the extrusion process. The modeling of the geometrical shape of the die was generated using Solidworks 2018 software. The model is shown in Fig. 5. When the modeling file was completed, it was imported by the ANSYS simulation software.

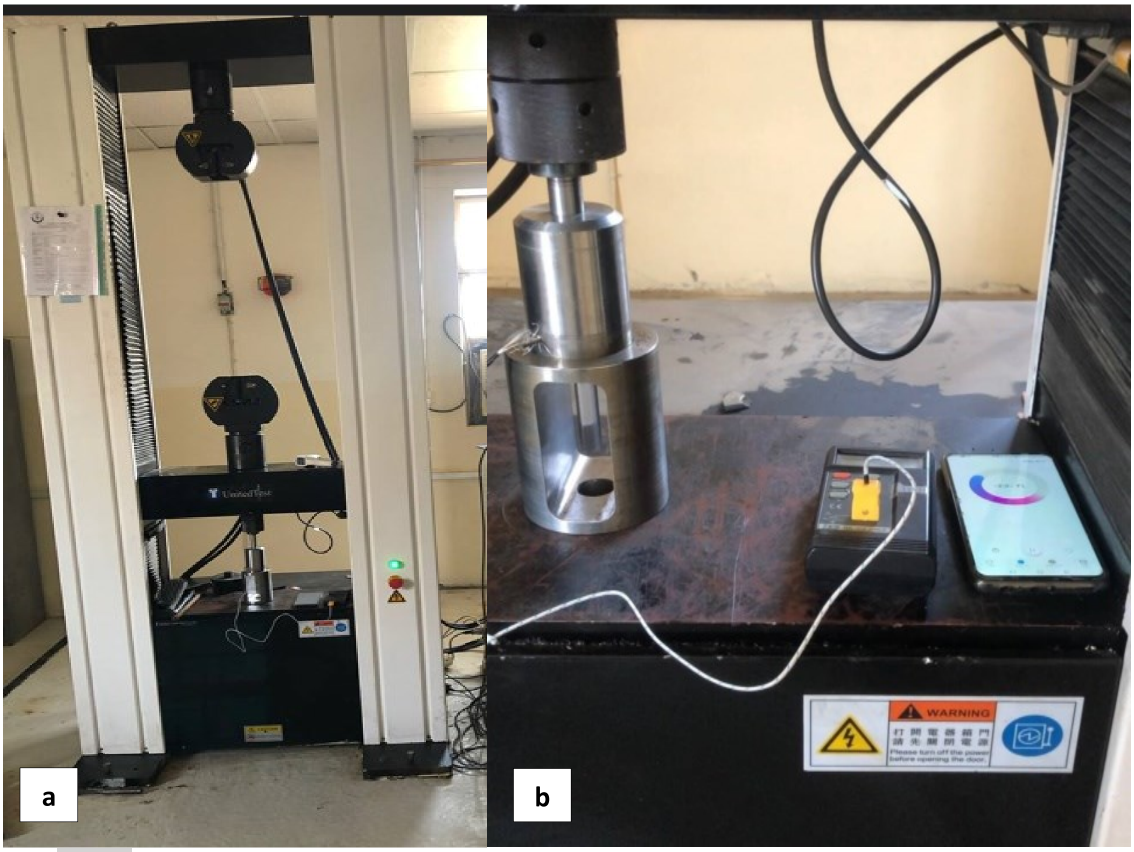


Fig. 3 (a) The UTM Testing Machine, (b) Extrusion the Die with a Cylindrical Adapter.

Fig. 4 The Thermometer and Timer for Temperature Recording with Time During the Extrusion Process.

Table 5 The Description of the Extrusion Tests with their Obtained Loads and Temperatures.

Test No.	Material	Speed mm/min	No. of samples	Max. Extrusion Load (kN)	Max. Temperature Difference (℃)	Process decision
1	Pure cast lead	10	4	25.22	1	Succeed
2		50	4	26.52	2	Succeed
3		150	4	27.01	3	Succeed
4	Pure AL with Grease	10	1	48.28	2	Succeed
5		50	1	85.03	2	Succeed
6		150	1	90	2	Succeed
7	Pure AL without Grease	10	1	-	-	The process failed

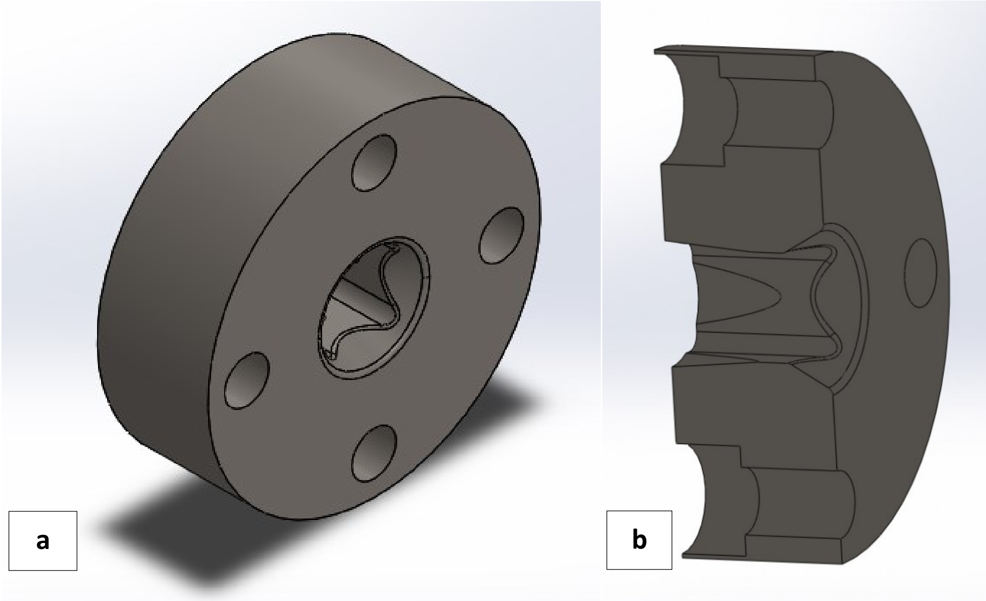


Fig. 5 The Geometrical Model of the Extrusion Die Generated by Solidworks Software (a) Front View, (b) Section View.

5.2.Meshing Process

The meshing process for the whole geometric part was made by describing the part regions into smaller discrete cells. Meshing divides the model into parts (or cells or zones) to solve the equations, allowing the solution to be approximated over a broader domain. The cut cell approach was employed in this study due to the die design's complexity, which is the general meshing method intended for Workbench ANSYS 2019 R3 software used for the die geometrical models, as shown in Fig. 6. Cut Cell and Tetrahedrons are the two main assembly meshing algorithms available [11]. The tetrahedral element was used. The number of nodes was 19491, and the number of elements was 11119.

5.3.Die Material Properties and Boundary Conditions

The physical and mechanical properties of the selective die material were obtained from the software as built-in data, especially the physical properties, while the mechanical strength data was obtained from the experimental testing. Material selection is an important step in the numerical analysis preprocessing process, which is the first step for giving the initial boundaries of the case study. Many novel materials and alloys properties are designed and upgraded in the simulation software these days. In general, having a unique combination of theoretical knowledge and actual experience data is required for material selection procedures. More material properties installed in the simulation, close to the real material properties as used in the experimental work, give more accurate and exact results received. In this numerical analysis, selecting the material properties is based on the mechanical stresses obtained from the experimental testing of the real material used in the manufacturing of the die, and other properties were obtained from the database of the Solidworks and Workbench softwares for AISI type A2 tool steel, as listed in Table 6. During the numerical simulation, boundary conditions must be given at each surface generated in the mesh creation process, which is very important because the solution of the simulation cannot exist if there were not any boundary conditions, eliminating or giving values for many numbers of equations indicated by the nodes of the mesh elements during the solution in this work using ANSYS workbench software in the simulation. The boundary conditions were made by specifying the fixing support regions of the die and the values of the applied load and temperature on the specific region of the die model, identical to those examined in the experimental solution of the problem. The descriptions of the boundary conditions of this work are shown in Fig. 7 for the fixing support regions of the die and Fig. 8 for the applied load and temperature regions on the die. According to the experimental extrusion process, many extrusion tests were made, as in Table 5. The highest load and temperature were measured using Aluminum billet with extrusion speed (150 mm/min), where the recorded highest load was (90 kN), and the highest temperature difference was (2 °C) during (13 sec) of the extrusion process duration.

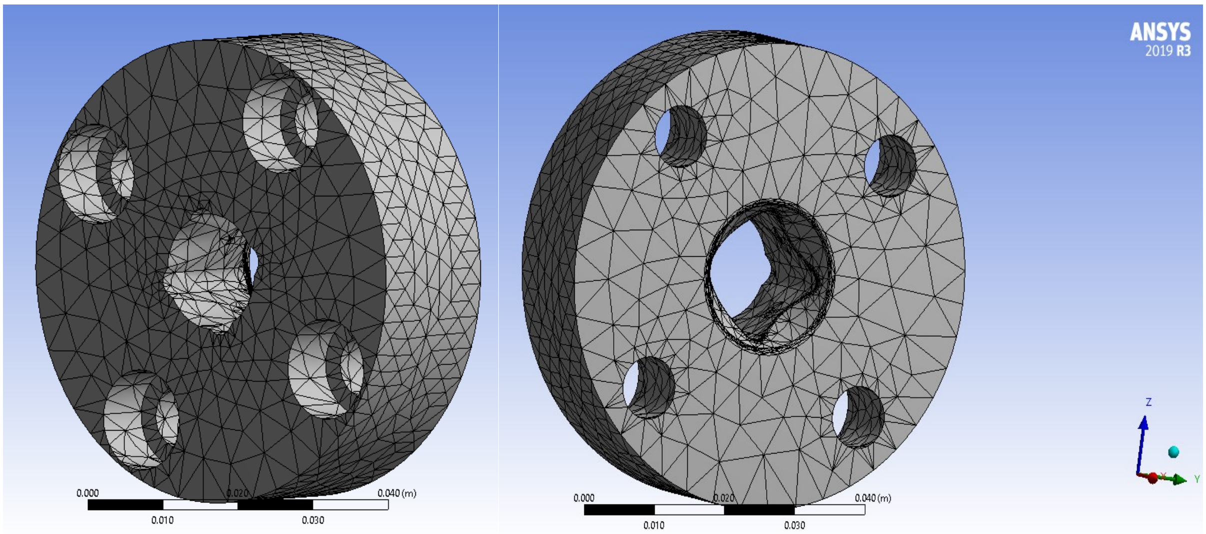


Fig. 6 The Meshing Distribution of the Die Model.

Table 6 The Physical and Mechanical Properties of the Die Material.

Ultimate stress (MPa)	Yield Stress (MPa)	Elastic Modulus of Elasticity (GPa)	Poisson's ratio	Specific Heat Capacity (J/kg*K)	Thermal Conductivity 20°C (W/m*K)	Thermal Expansion Coefficient (1/K)	Density (kg/m^3)
892	825	203	0.285	461	26	1.1e-5	7860
Experimental mechanical testing	Experimental mechanical testing	Solidworks database	Solidworks database	Solidworks database	Solidworks database	Solidworks database	Solidworks database

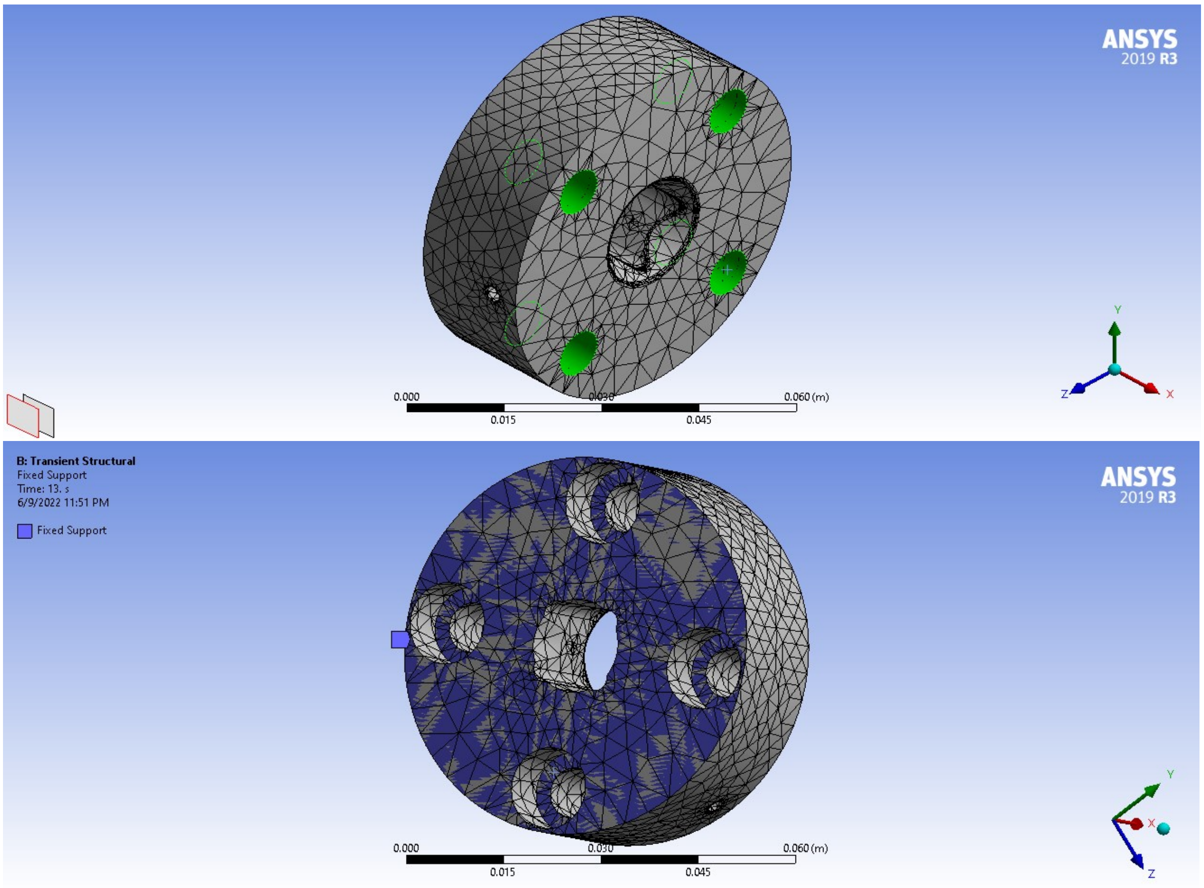


Fig. 7 The Fixing Support Regions Boundary Conditions on the Die Model.

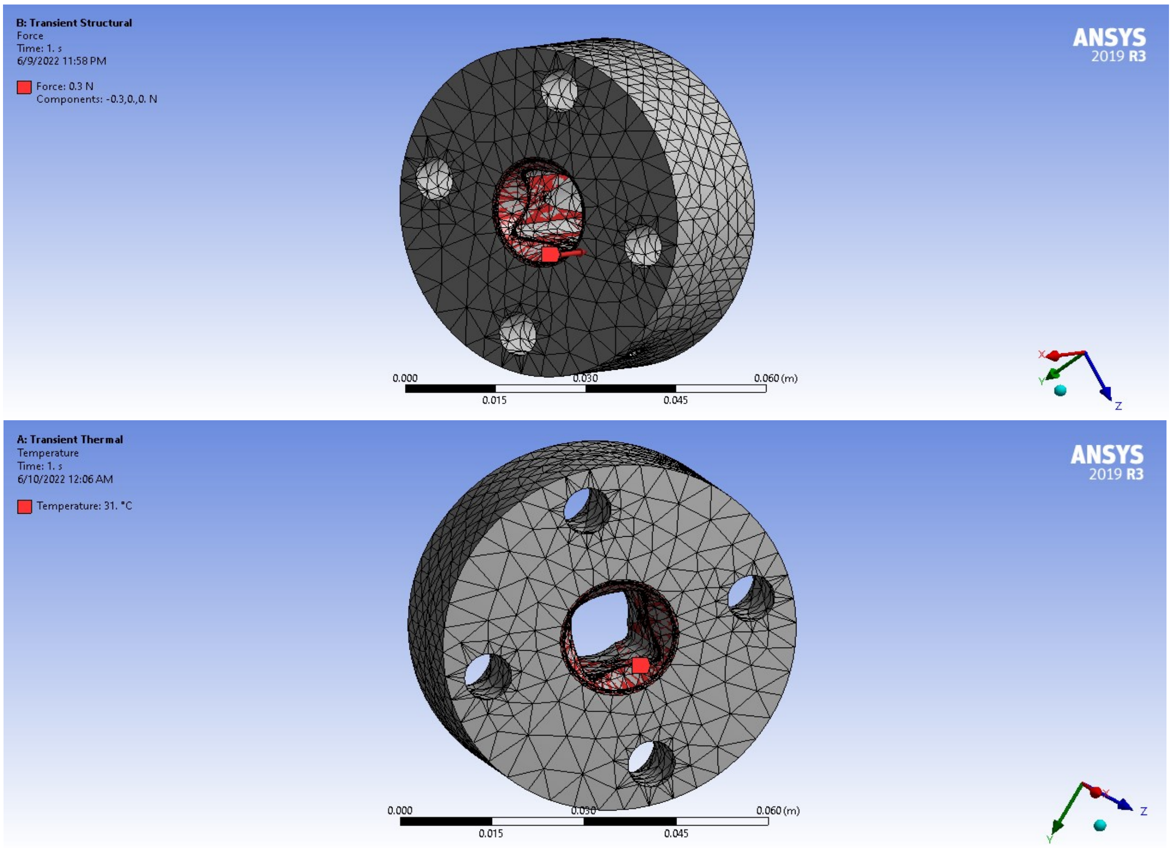


Fig. 8 The Applied Load and Temperature Regions on the Die Model.

5.4.Fatigue Life Simulation Setup

The simulation of fatigue life is a significant process to predict the working life of the whole die theoretically under different mechanical and thermal loadings. A thermo-mechanical simulation was built to produce the plastic stress expressed by equivalent von Mises stress, temperature distribution through the die model, and fatigue life. This simulation measured the fatigue life of the die based on the highest loads obtained during the extrusion tests were made experimentally by the die, where loads were the applied force and temperature difference, which can predict the fatigue life of the die; as a result, the designer will know the die failure mechanisms, the failure due to thermo-mechanical stresses obtained from different loading, different types of materials of the product and extrusion speed. Therefore, estimating the fatigue life of the die can give a good indication of the die failure mechanism, the number of products that can be produced till die failure, and finding the tools for improving the life of the die. In the fatigue simulation, many factors should be considered depending on the type of loading, supporting, die manufacturing, and the extrusion process experimentally. These factors will be considered as boundary conditions for the fatigue simulation obtained by ANSYS Workbench, which are;

I- ***Fatigue life method:*** In this numerical simulation, the strain-life method is adopted based on published research on the same type of die material [12]. This method depends on the strain-life equation of the die material dealing with low cycle fatigue behavior of the High-speed tool steel, as presented in the equations below and detailed in Table 7:

$$\frac{\Delta\varepsilon_t}{2} = \frac{\sigma'_f}{E}\left(2N_f\right)^b + \varepsilon'_f\left(2N_f\right)^c \qquad (1)$$

II- ***Loading type:*** It is the ratio of the minimum load to the maximum load ($R = \frac{\sigma min}{\sigma max} = 0$), meaning that the load applied in the fatigue test was a *repeated cyclic load*.

III- ***Applied stress:*** The applied stress components used in the fatigue life simulation were considered *(the equivalent of Von Mises stress).*

IV- ***Cyclic load frequency:*** It is considered *(1 cycle = 1 extrusion process experimentally*).

V- ***Infinite Life:*** It is considered *(10^9 cycles).*

Table 7 The Experimental S-N Data for the Same Die Jaws Material [12].

Cyclic strain hardening exponent (n')	Cyclic strength coefficient K' (MPa)	Fatigue strength coefficient σ'_f (MPa)	Fatigue strength exponent (b)	Fatigue ductility coefficient ε'_f	Fatigue ductility exponent (c)
0.10204	3854	4036	- 0.11715	0.46507	- 0.94526

6.RESULTS AND DISCUSSION

The simulation is created for the main part of the extrusion system, which is the die because it is subjected to severe combination loads consisting of direct axial mechanical force (billet pushing force) and friction temperature during the extrusion process. This analysis focused on presents clearly, the main factors in the analysis: temperature distribution, Von Mises stresses, stress intensity, and fatigue life occurred in the critical regions or edges of the die, depending on its geometrical design. This analysis considered the applied thermo-mechanical load based on the highest measured experimental extrusion test.

6.1.Temperature Distribution

The temperature distribution was numerically simulated with time through the die wall depending on the extrusion speed and product material types. The applied transient temperature on the die in the simulation was adopted from the highest temperature with time data recorded from the experimental test, where for an Aluminum product with the extrusion speed of (150 mm/min), the highest temperature reached (13 sec). Figure 9 shows the temperature distribution through the die wall, where the temperature increment was (2°C) during the extrusion time of about (13 sec). In this figure, the temperature is distributed from the high temperature at the inner surface, especially at the friction contact surfaces with the product during the extrusion process. This temperature reduced radial through the die wall. Moreover, from the back view of the die, the temperature of the divergent section was lower (yellow color) than the inner surface due to the gap obtained from the divergent design with the product.

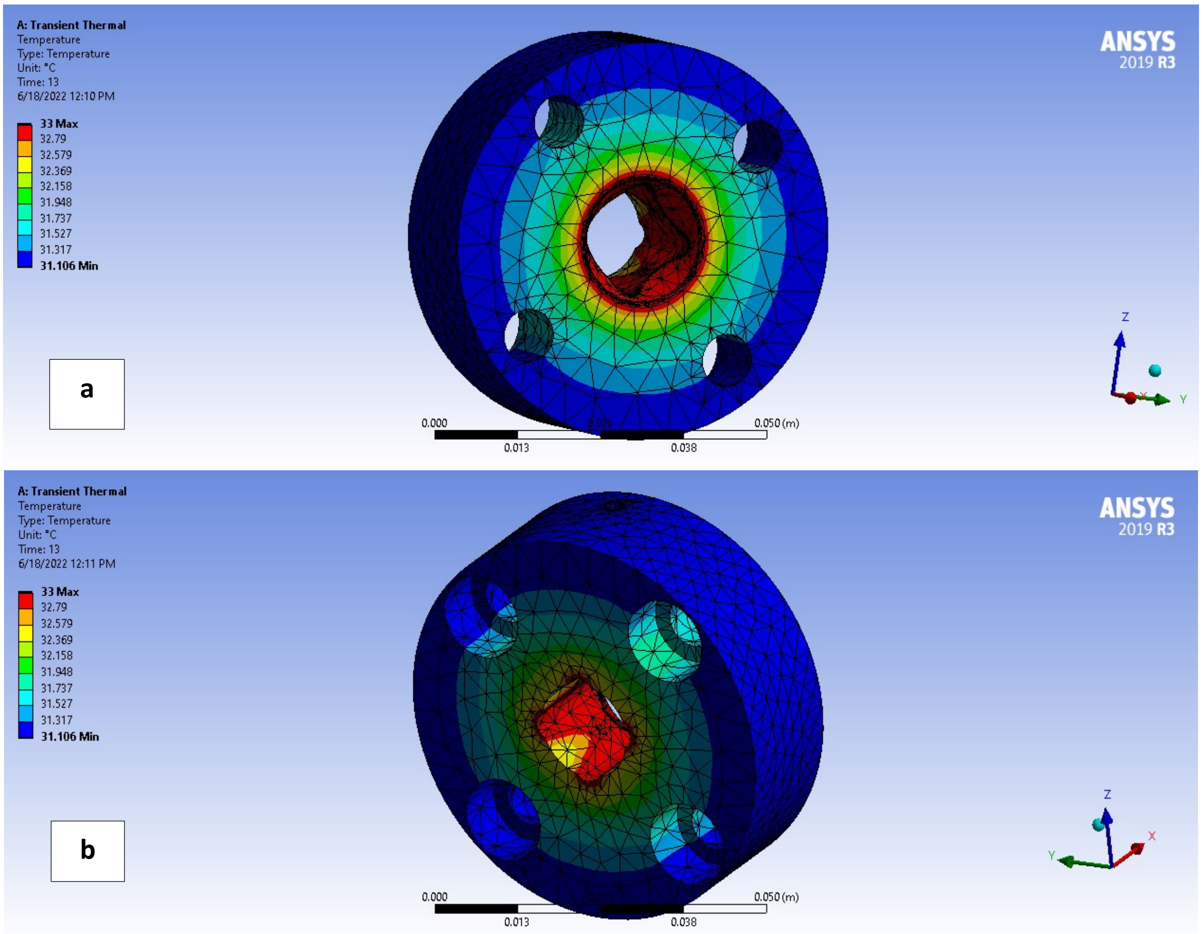


Fig. 9 The Transient Temperature Distribution Through the Die Wall for 13 Sec. (a) The Highest Temperature Distribution from Red to Blue, and (b) the Temperature Distribution for the Diverging Die Design.

6.2.Equivalent Stress and Stress Intensity

Calculating the equivalent Von Mises stress and the stress intensity in the die is very important because they are the basis of the strength behavior and fatigue life analysis. Also, they demonstrated the stress concentration in the critical sections or edges in the die profile depending on its geometrical design. According to the experimental extrusion process, the maximum applied thermo-mechanical loading was measured during the manufacturing of the Aluminum product at an extrusion speed (150 mm/min), which was (90 kN) axial force with (2 °C) temperature difference at (5.5 sec). In the simulation, this loading was subjected to the inner contact surface of the die with the product during the extrusion process. The finite element simulation employed by ANSYS Workbench 2019 R3 to determine the equivalent stresses and stress intensity and then directly predict the fatigue life of the die. The equivalent Von Mises result is shown in Fig. 10, and the stress intensity result is shown in Fig. 11. In Fig.10, the equivalent stress is clearly high on the inner surface of the die, especially in the contact areas, due to the reduction in the geometrical volume based on the extrusion rate of 1.67. The maximum stress reached about (230 MPa) at the inner edge of the converge-diverge connection section of the die, as shown in the last graph in Fig. 10. While the stress intensity was a fracture mechanics factor (K) used to predict the stress state as an intensity near the tip of a crack or notch or sometimes at the highest stress concentration zone due to the effects of combined load or residual stresses. It is a theoretical concept usually applied to the homogeneous, linear elastic material and is useful for providing the failure criterion of the materials. Figure 11 shows the maximum intensity of the stresses at the inner surface of the die, and the highest stress intensity factor reaches about (251.4 MPa) at the critical converge-diverge section due to the die profile design.

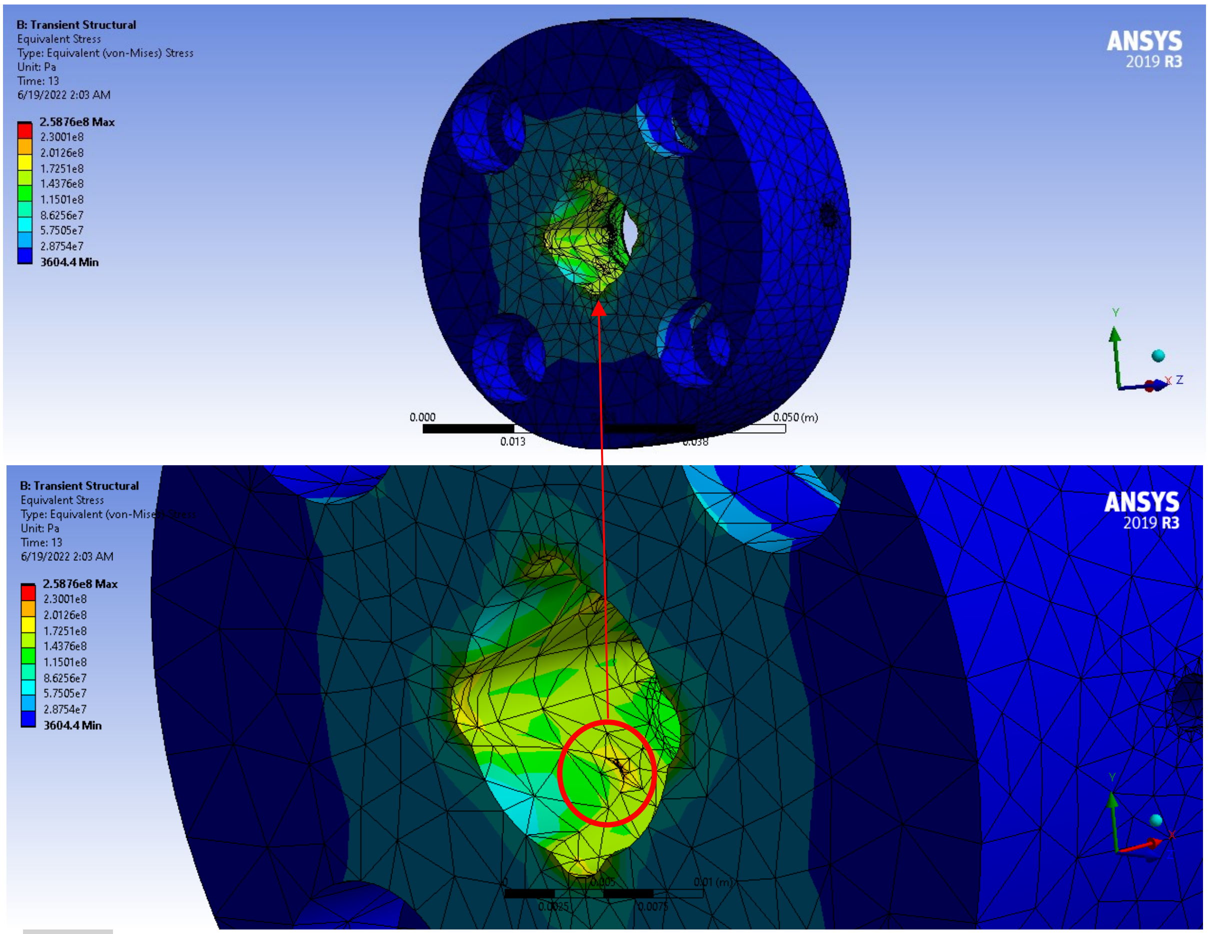
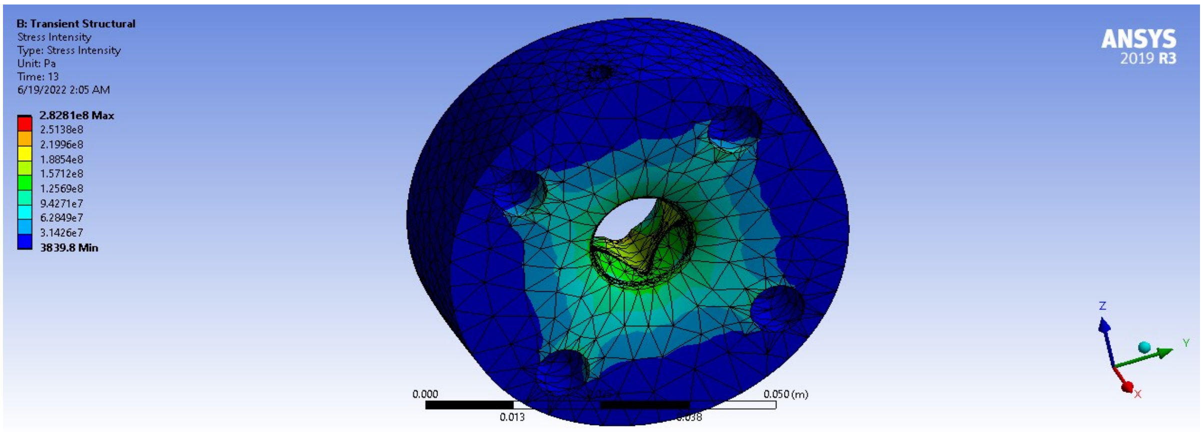


Fig. 10 The Equivalent Von Mises Stresses in the Die Using Aluminum Billet with Extrusion Speed (150 mm/min) Measured at (5.5 sec) at (90 kN) Maximum Applied Load.



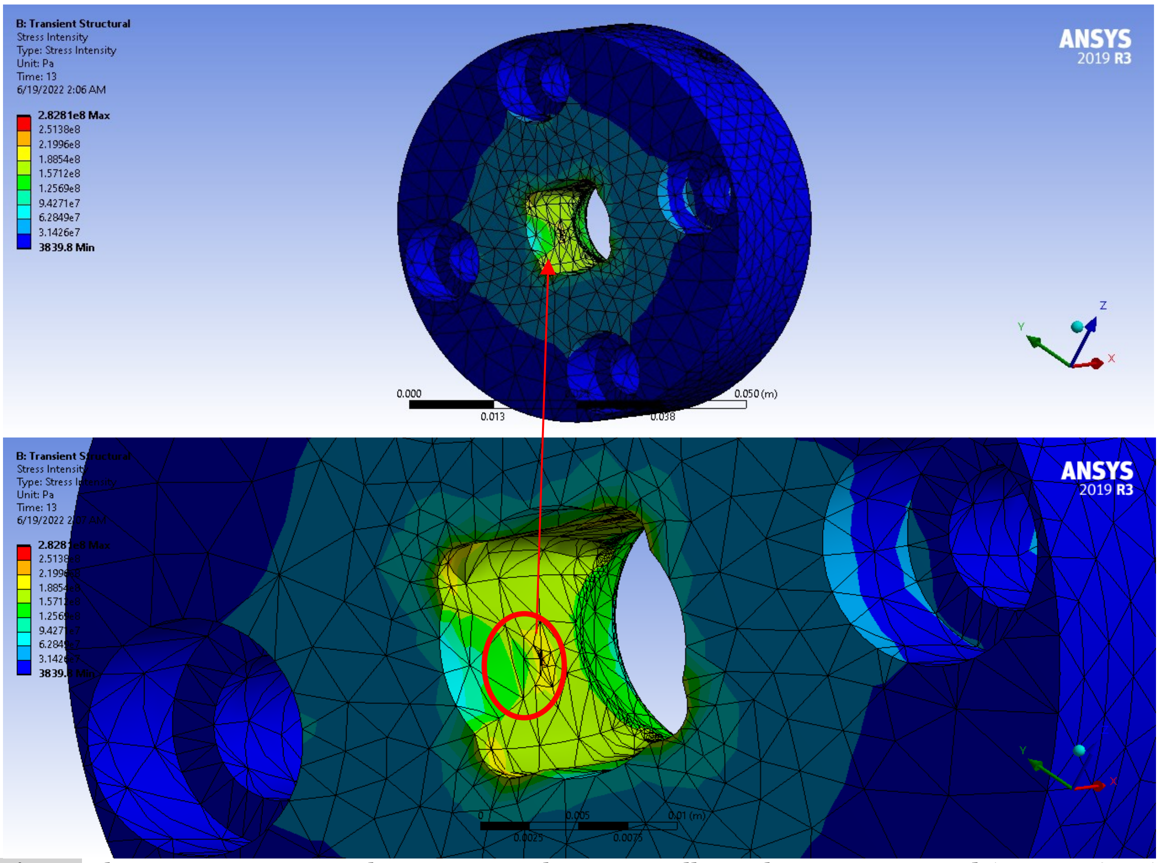
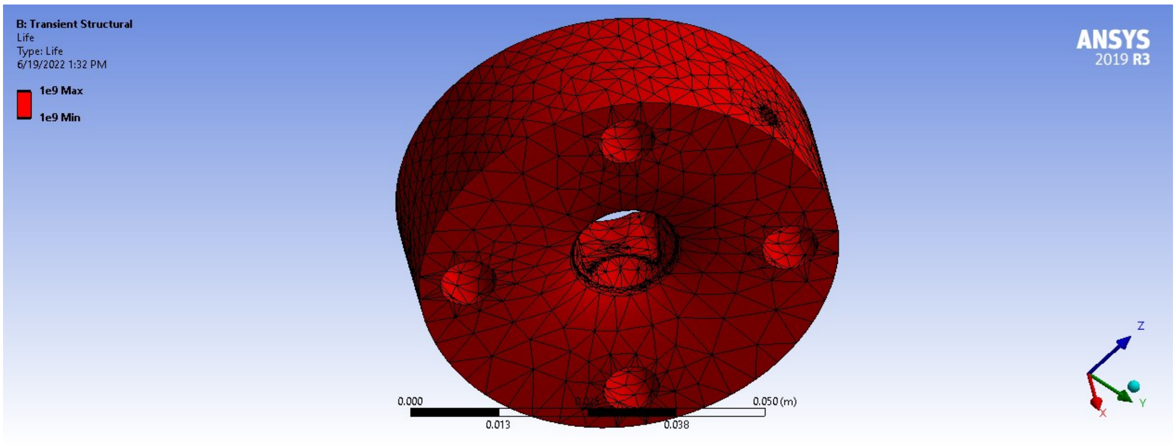


Fig. 11 The Stress Intensity in the Die Using Aluminum Billet with Extrusion Speed (150 mm/min) Measured at (5.5 sec) at (90 kN) Maximum Applied Load.

6.3. Fatigue Life

Simulating the fatigue life of the die is a significant process to predict the working life of the die theoretically under different mechanical and thermal loadings without the need to do destructive testing on the die. It can be done for the whole die or separate parts of the die, no matter the complexity of the design. According to the experimental extrusion process, thermo-mechanical loading was measured for manufacturing the Aluminum sample at speed (150 mm/min), considered the highest (peak) obtained load (90 kN), which was at (5.5 sec). The strain-life behavior of the die material was settled in the setting of the software simulation based on Zhang et al. [12]. From the highest load applied to the die during the extrusion of an Aluminum product with the high speed, the fatigue life result showed that the die was safe and had an infinite life reached (10^9 cycles) as it was settled in the software setting, as shown in Fig. 12, because the fatigue life analysis based on the Von Mises equivalent stresses measured during the simulation, which was less than the endurance total strain life of the die material behavior included in the solution of the analysis. Therefore, to assess and check the critical fatigue life of the die due to its geometrical design and material behavior, different loads were subjected to the inner surface of the die with a range of (200, 400, and 800 kN) to determine the critical fatigue life, as shown in Figs. 13 - 15.



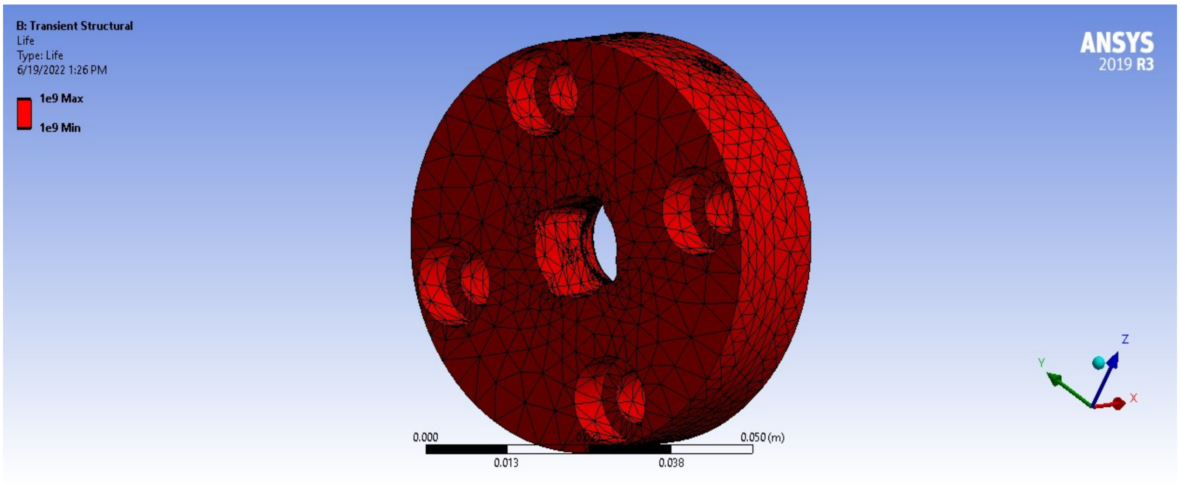


Fig. 12 Fatigue Life Simulation of the Die at Maximum Extrusion Process (90 kN).

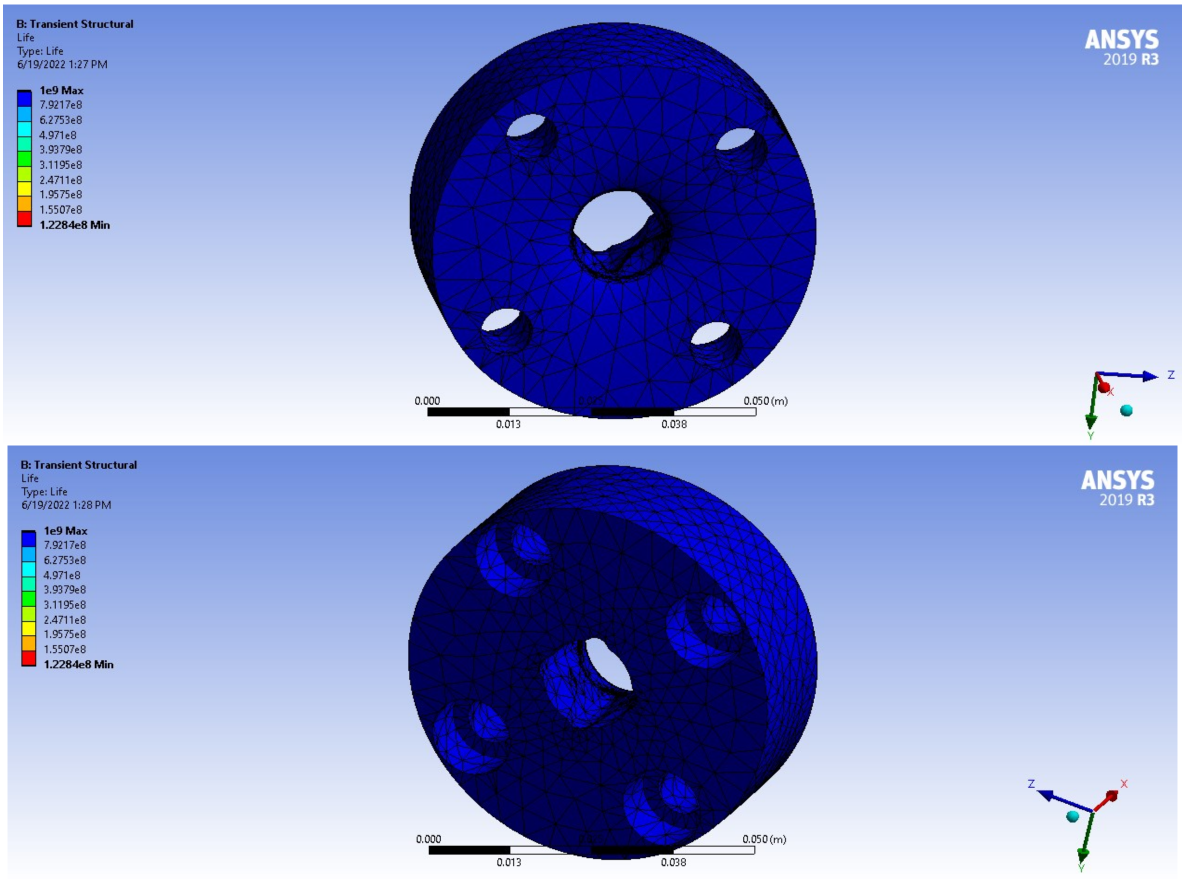
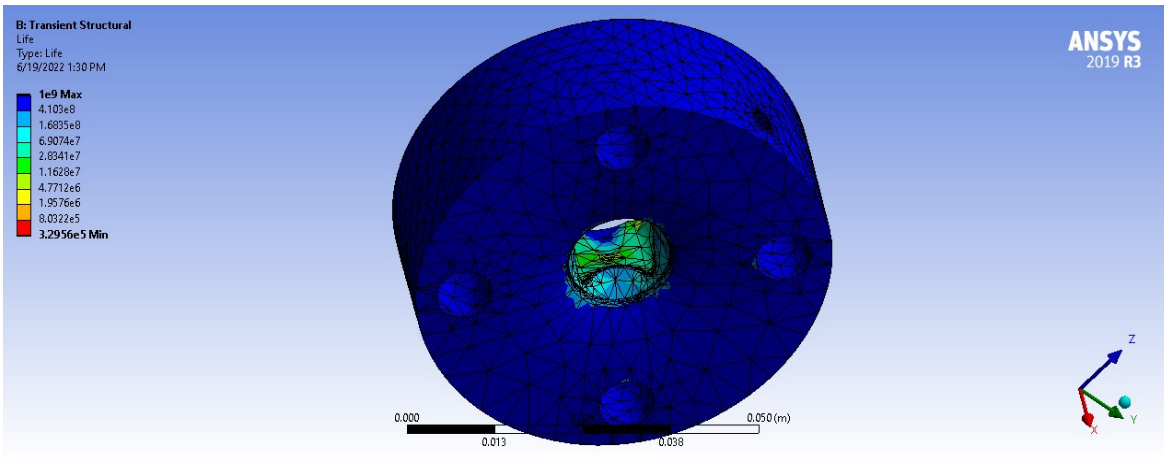


Fig. 13 Fatigue Life Simulation of the Die at Extrusion Load (200 kN).



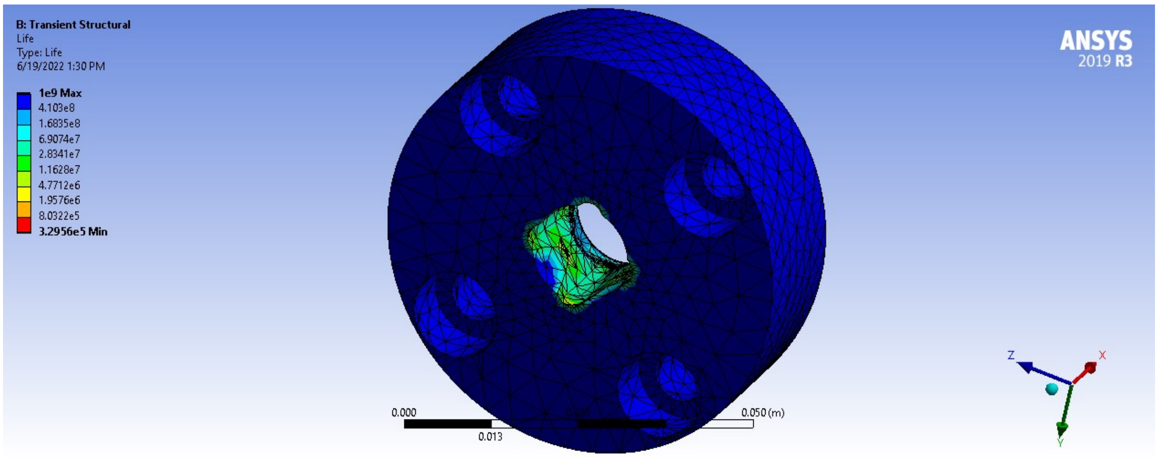


Fig. 14 Fatigue Life Simulation of the Die at Extrusion Load (400 kN).

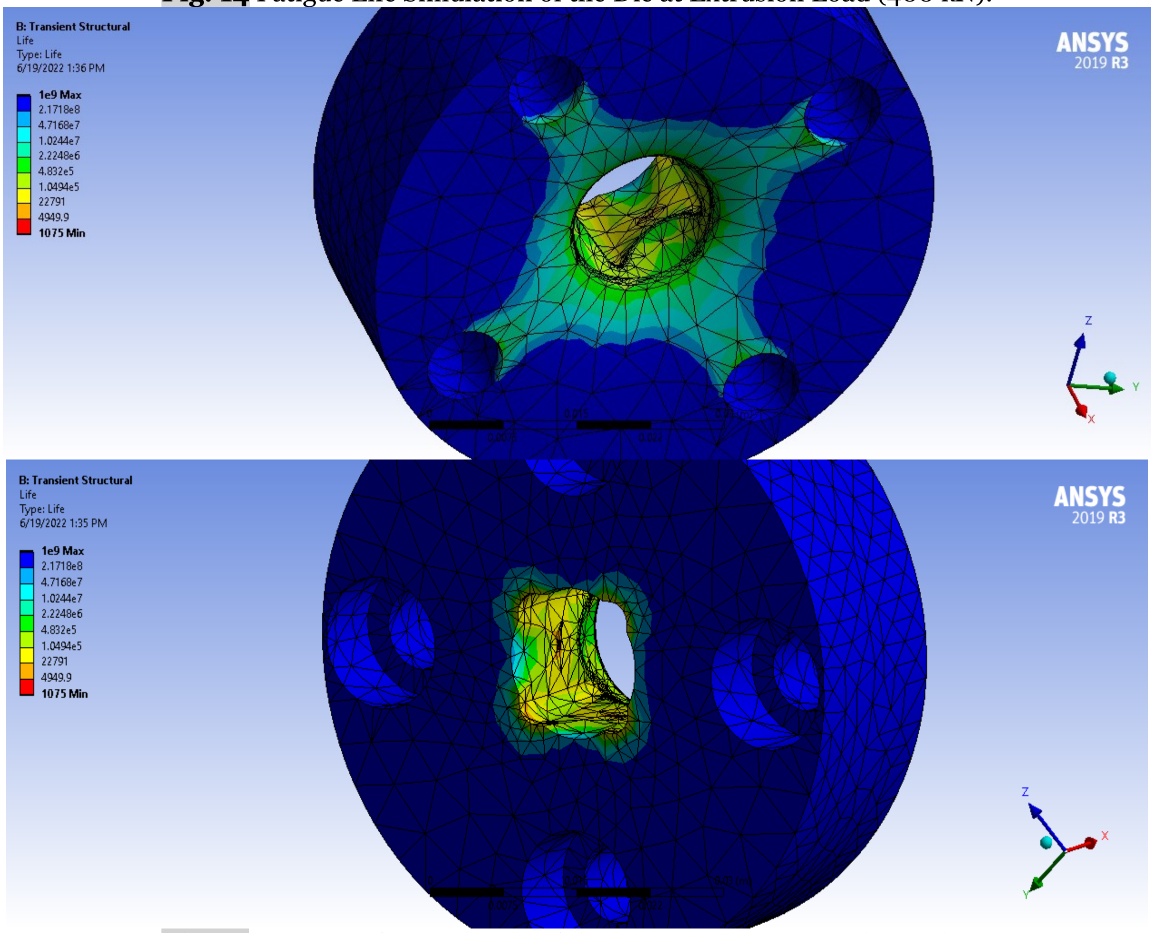


Fig. 15 Fatigue Life Simulation of the Die at Extrusion Load (800 kN).

Figure 13 shows that increasing the applied load to (200 kN), makes the fatigue life reaches into the critical total strain life of the die material. While increasing it to (400 kN), it is clearly appeared and the fatigue life start to reduce to about ($1.9576.10^6$ cycle), as shown in Fig. 14. This is significantly appeared in Fig. 15. When the applied load increased to (800 kN), the fatigue life decreased and reached (4949.9 cycle) at (800 kN) load, revealing that the die life would be reduced. The number of product samples returned to the number of extrusion processes would be minimized. Therefore, the numerical results showed that the die can withstand and safely use Lead or Aluminum with grease to produce extrusion products. However, changing the billet with harder materials (like Brass or Steel) would reduce the die life and might fail later. According to this numerical simulation, an understanding method can be concluded; through the die and product materials selection and experimental working and loading conditions, the fatigue or working life of the die also the amount of produced product can be predicted and provided for the manager a wide view about the process and what could be found finally, which is agreed with the final discovering of Zhang et al. [6] and Samya et al. [7].

7. CONCLUSIONS

According to the main factors influencing the die strength and fatigue life generated from the experimental extrusion process and numerically analyzed, it can be concluded that

- The temperature is distributed from the high temperature at the friction contact surfaces inside the die and reduced radial through the die wall. The highest temperature increment was (2°C) during the extrusion duration of about (13 sec).
- The equivalent stress was high at the inner contact surface of the die, and its maximum value reached about (230 MPa).
- The highest intensity factor reached about (251.4 MPa) at the inner surface of the die.
- The die was safe and had an infinite life that reached (10^9 cycles) using a maximum axial extrusion load of (90 kN) with a temperature difference of (2 °C) at 150 mm/min speed.
- The die was safe using Lead or Aluminum; however, changing the product billet into harder materials reduced the die's life and might fail later.
- The numerical simulation based on the working and loading data can predict the fatigue or working life of the die and the amount of product produced, giving the manager a wide view of the process and what can be found finally.

REFERENCES

[1] Cosenza C, Fratini L, Pasta A, Micari F. **Damage and Fracture Study of Cold Extrusion Dies**. *Engineering Fracture Mechanics* 2004; **71**(7–8):1021–1033.

[2] Qamar SZ, Sheikh AK, Arif AFM, Younas M, Pervez T. **Monte Carlo Simulation of Extrusion Die Life**. *Journal of Materials Processing Technolgy* 2008; **202**(1–3):96–106.

[3] Fu MW, Lu J, Chan WL. **Die Fatigue Life Improvement Through the Rational Design of Metal-Forming System**. *Journal of Materials Processing Technolgy* 2009; **209**(2):1074–1084.

[4] Akhtar SS, Arif AFM. **Fatigue Failure of Extrusion Dies: Effect of Process Parameters and Design Features on Die Life**. *Journal of Failure Analysis and Prevention* 2010; **10**(1):38–49.

[5] Maity K, Chaubey S, Rath S. **A Numerical Analysis of Extrusion of Square Section from Round Billet Through Mathmatically Contoured Die with Design of Die Profile**. *Journal for Technology of Plasticity* 2013; **38**(1):75-83.

[6] Zhang Y, Zhou Y, Guo X, Song K, Zhang X. **Fatigue Failure Prediction Model and Verification of Hot Extrusion Die**. *Procedia Manufacturing* 2019; **37** :66–72.

[7] Samya AN, Mohammed AN, Hani AA. **Fatigue Life Prediction Based on the Experimental Operating Conditions for a Cold Forging Die**. *International Journal of Mechanical Engineering* 2021; **6**(3):207-217.

[8] Khurmi J, Gupta RS. **A Textbook of Machine Design**. 1st ed., Eurasia Publishing House (PVT.) LTD., Ram Nagar, New Delhi-110 055, 2005.

[9] Bringas JE. **Handbook of Comparative World Steel Standards**. 3rd ed., ASTM International Standard Worldwide DS67B, USA, 2004.

[10] Boyer HE, Gall TL. **Metals Handbook**. 2nd desk edition, ASM International Handbook Commitee, 1985.

[11] Ansys Inc. **ANSYS Meshing User's Guide**. 2013; **15317**:724–746.

[12] Zhang Y, Hu CL, Zhao Z, Li AP, Xu XL, Shi WB. **Low Cycle Fatigue Behaviour of a Cr–Mo–V Matrix-Type High-Speed Steel Used for Cold Forging**. *Materials and Design* 2013; **44**: 612–621.