3.1 Introduction

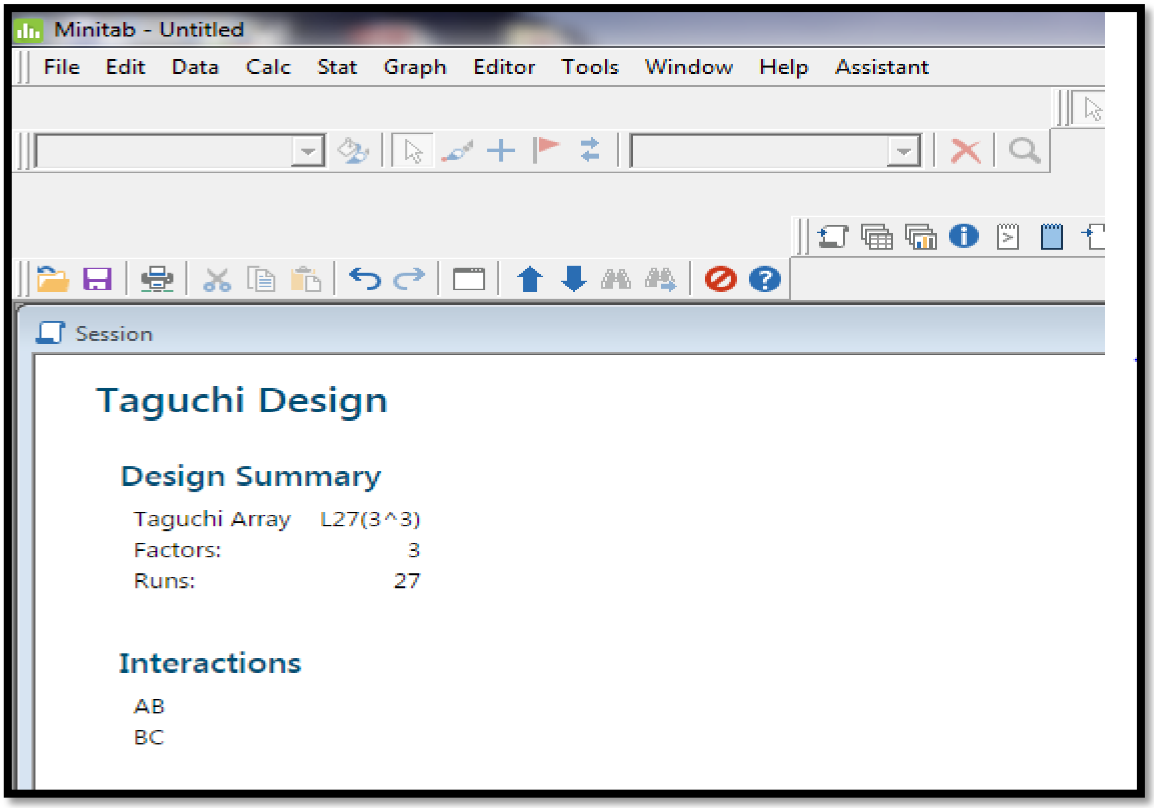
This chapter is divided into two parts, part one includes using Minitab 18 program to optimize the results by using the Taguchi method, and use ANOVA to calculate percentage contribution for each factor. Part two gives details to model generated by FEM Ansys workbench 18 and calculation total deformation equivalent stress, fatigue life, and safety of factor for weldment by FSW.

3.2 Minitab Program

3.2.1 Determining the Welding Parameters and Their Levels

In this work, the FSW parameters include rotation speed of (630, 1000 and 1600) rpm, welding speed of (20, 32, 45) mm/min, three angles of orientation of welding line (45^o, 60^o, 90^o) degree with the applied load and the tilt angle is constant at (2°).

When finding the FSW parameters and levels, the suitable orthogonal array (L27) was chosen. Selection the L27 array for three parameters and their nine levels is shown in figure **(3.1)** in the design of the experiments (DOE) for welded the similar aluminium alloys (6061-T6) to find the optimum welding parameters.



```
Taguchi Design

  Design Summary

   Taguchi Array    L27(3^3)
   Factors:                3
   Runs:                  27


  Interactions

   AB
   BC
```

Figure **(3-1)**: DOE -Taguchi method L27.

3.2.2 Signals to Noise Ratio (S/N)

The signals are indicators of the effect on average responses and the noises are evaluated of the influence on the deviations from the sensitiveness of the experiment. In this work, the S/N ratio was chosen according to the criterion of the larger the better, in order to maximize the response. In Taguchi method, the signal to noise ratio is used to determine the deviation of the quality characteristics from the desired value. The S/N ratio (larger-the-better) can be expressed as shown in Eq. (3.1) [65].

$$\mathbf{S/N = -10Log\ [\Sigma\ ((1/y^{2})/n)] \quad . . . . \ (3\text{-}1)}$$

Where:

y_i : is the average of observed data for each test.

n : is the number of conducted tests.

In the present work, 27 S/N ratios and 27 means were calculated.

3.2.3 Analysis of Variance

ANOVA is achieved to examine the importance of the parameters of FSW process which influence the tensile strength of joints. The F-test named after Fisher can also be utilized to identify which parameter has the largest effect on tensile strength. Usually, the alteration of the FSW parameters has important influence on quality characteristics of tensile strength of welded joints when F is large [69].

The total degree of freedom

$$DF = N - 1 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3\text{-}2)$$

Where N is the total number of levels

1. The total sum of square SS_T(total)

$$SS_T = \Sigma\,(y^2) - \frac{(\Sigma y)2}{n} \quad \ldots\ldots\ldots\ldots\ldots\ldots (3\text{-}3)$$

Where

n: is the total number of experiments

y: Sum of all the result

2. The sum of squares SS_F (factor)

$$SS_F = \frac{(\Sigma F1)2}{K_F} + \frac{(\Sigma F2)2}{K_F} + \frac{(\Sigma F3)2}{K_F} - \frac{(\Sigma y)2}{n} \quad \ldots\ldots\ldots (3\text{-}4)$$

Where:

SS_F: is the sum of squares for one factor.

$F_{(1,\ 2,\ \text{and}\ 3)}$: The average of tensile strength for one factor at three levels.

K_F: Represent a number of each level of the factor A_1, A_2, A_3.

3. Percentage contribution (P_A)

$$P_A = \frac{\mathrm{SSF}}{\mathrm{SST}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(3-5)}$$

4. Percentage contribution of parameters

$$P_p = \frac{P_A}{100-ERROR\%} * 100\% \ldots\ldots\ldots\ldots \text{(3.6)}$$

3.3 Numerical Procedure (ANSYS Program)

In this study, a commercial general-purpose FE program ANSYS 18.0 workbench is used for simulating of fatigue resistance of base martial and single pass (optimum case) for FSW joints and double pass (FSP).

1. Define element types.
2. Engineering data.
3. Geometry.
4. Mesh generation.
5. Boundary condition.
6. Fatigue solution.

3.3.1 Define Element Types

There are many various elements in ANSYS (18.0) utilized to analysis different problem, therefor the correct selection element type is very important. For fatigue analysis, SOLID45 was selection. A SOLID45 element has eight nodes and three degrees of freedom for each node. This element supports elasticity, plasticity, large strain, large deflection, stress stiffening and creep [70].

.

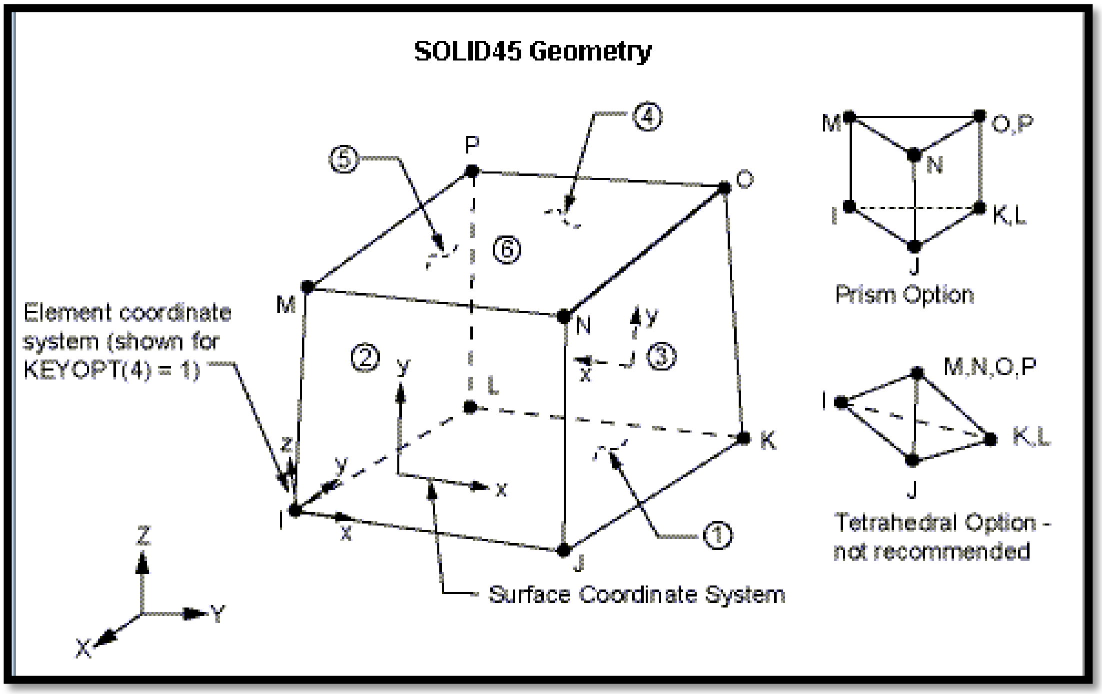
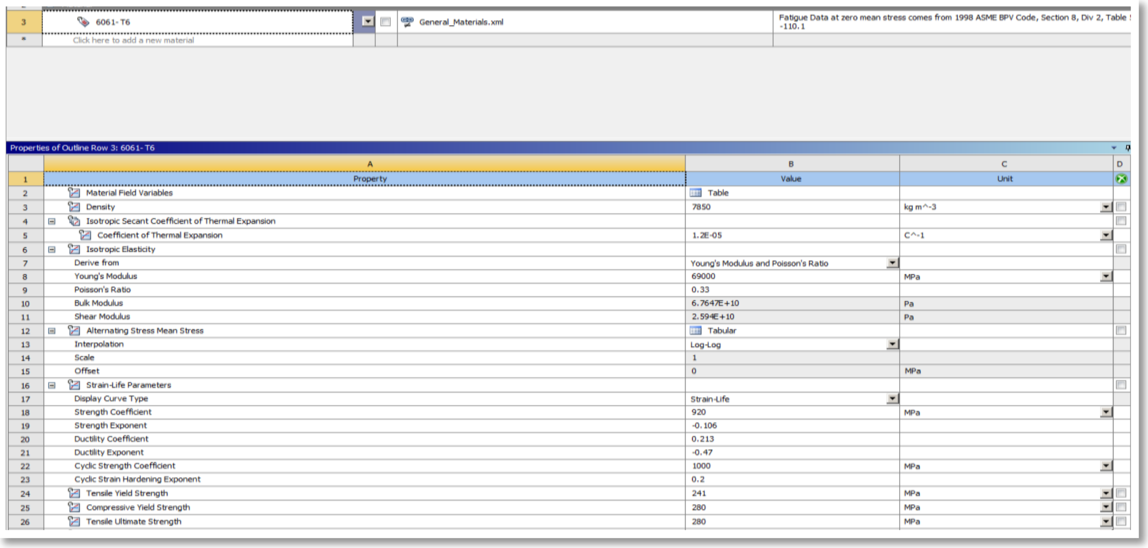


Figure (**3.2**): Three-dimensional structural solid element SOLID45 [70].

3.3.2 Engineering Data

By selection, the linear elastic isotropic material, the material properties such as Poisons ratio and modulus of elasticity will be illustrated as shown in figure **(3.3)**.



3 | 6061- T6 | General_Materials.xml | Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table -110.1

* | Click here to add a new material

Properties of Outline Row 3: 6061- T6

	A	B	C	D
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Density	7850	kg m^-3	☐
4	⊟ Isotropic Secant Coefficient of Thermal Expansion			☐
5	Coefficient of Thermal Expansion	1.2E-05	C^-1	
6	⊟ Isotropic Elasticity			☐
7	Derive from	Young's Modulus and Poisson's Ratio		
8	Young's Modulus	69000	MPa	
9	Poisson's Ratio	0.33		
10	Bulk Modulus	6.7647E+10	Pa	
11	Shear Modulus	2.594E+10	Pa	
12	⊟ Alternating Stress Mean Stress	Tabular		☐
13	Interpolation	Log-Log		
14	Scale	1		
15	Offset	0	MPa	
16	⊟ Strain-Life Parameters			☐
17	Display Curve Type	Strain-Life		
18	Strength Coefficient	920	MPa	
19	Strength Exponent	-0.106		
20	Ductility Coefficient	0.213		
21	Ductility Exponent	-0.47		
22	Cyclic Strength Coefficient	1000	MPa	
23	Cyclic Strain Hardening Exponent	0.2		
24	Tensile Yield Strength	241	MPa	☐
25	Compressive Yield Strength	280	MPa	☐
26	Tensile Ultimate Strength	280	MPa	☐

Figure **(3.3)**: Engineering date (material properties) in ANSYS

3.3.3 Geometry

The numerical model is presented in figure **(3.4)**. The dimensions of this model are length 60 mm, width 10 mm, and thickness 4 mm.

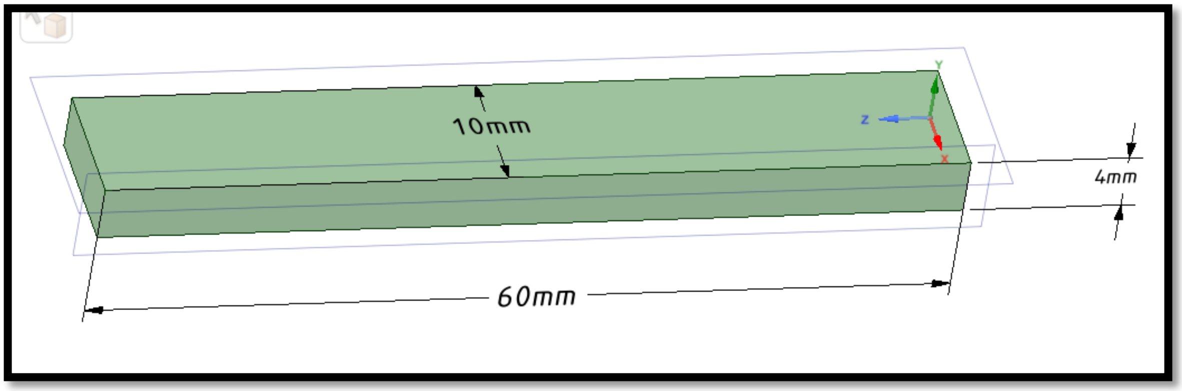


Figure **(3.4)**: ANSYS model used for fatigue life.

3.3.4 Mesh Generation

Molding meshing is very important to get reasonable results. The control of mesh size was performing by specifying the number of the element. In current work, the suitable mesh size was 1.2 mm as shown in table (3.1); the number of nodes was (17478) while the number of the element was (3520) as shown in figure (3.5). Skewness, aspect ratio, and element equality presented in figure (3.6), (3.7) and (3.8) respectively.

Table (**3.1**): Grid independent test (GIT)

Mesh size (mm)	5	4	3	2	1.5	1.2	1	0.5
Equivalent stress (MPa)	138.4	137.7	133.35	134.91	134	134.1	134.09	134.09

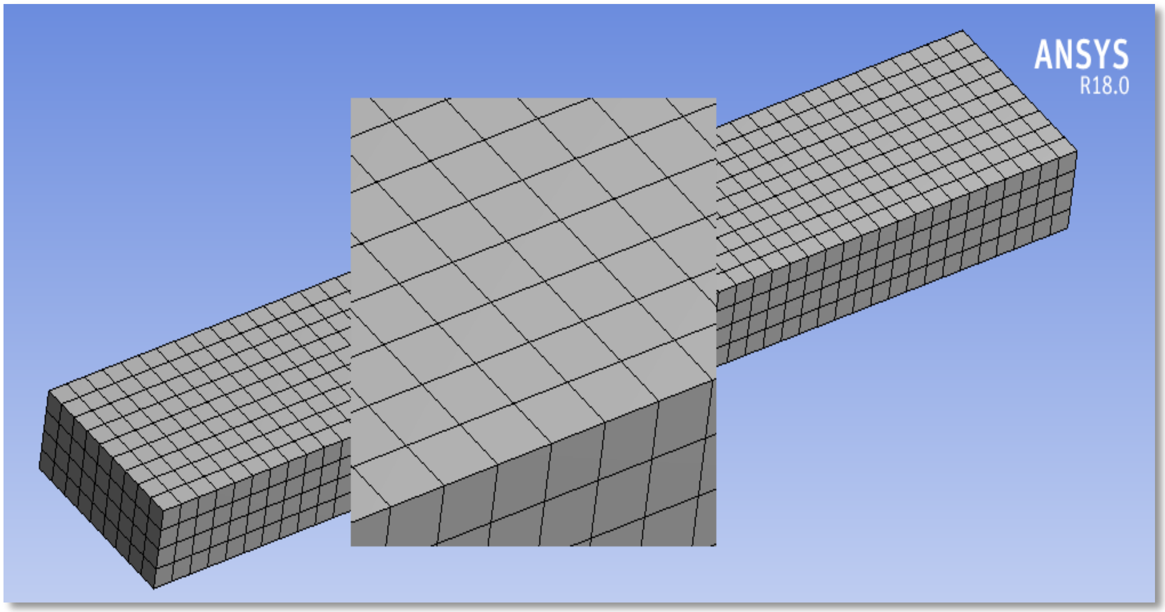
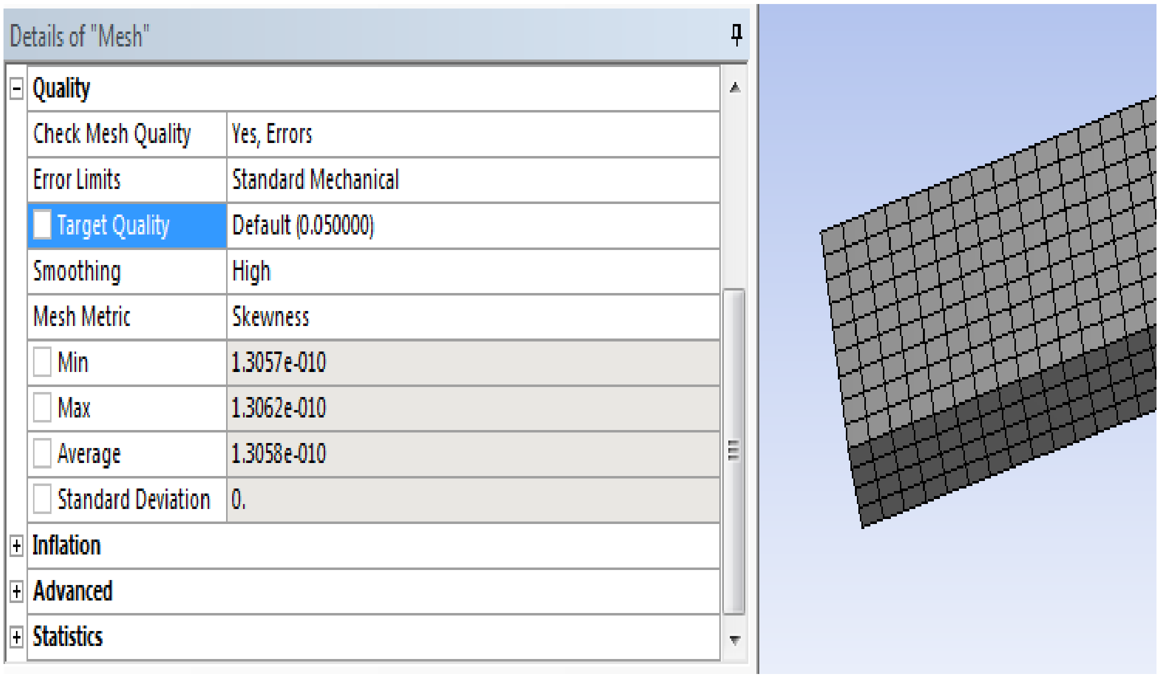


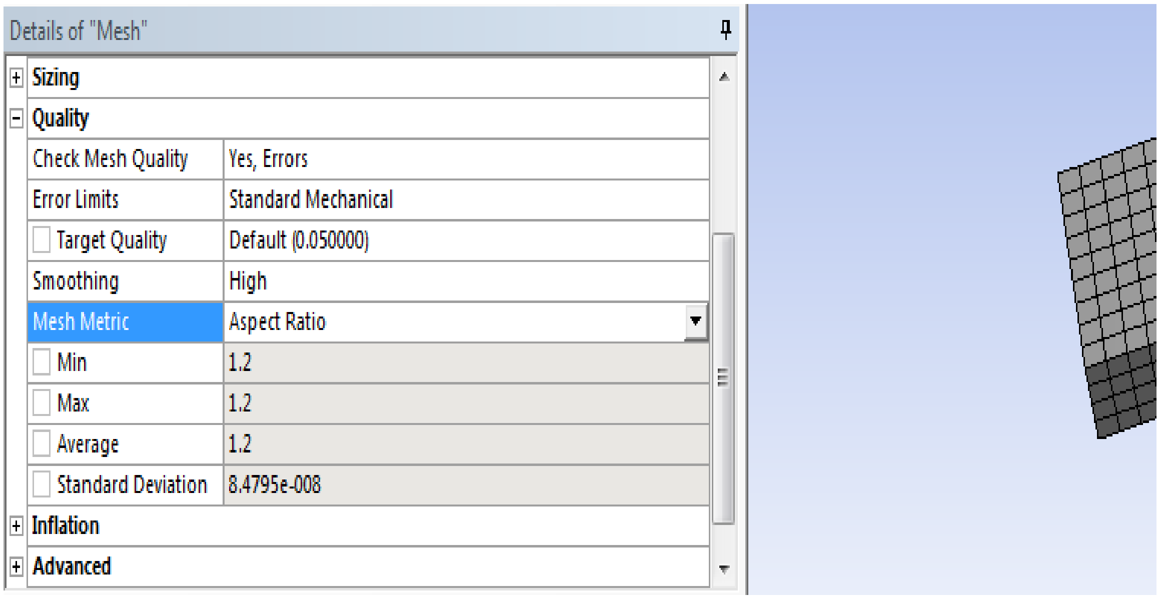
Figure (**3.5**): The mesh of the studied model.



Details of "Mesh"

⊟ **Quality**	
Check Mesh Quality	Yes, Errors
Error Limits	Standard Mechanical
☐ Target Quality	Default (0.050000)
Smoothing	High
Mesh Metric	Skewness
☐ Min	1.3057e-010
☐ Max	1.3062e-010
☐ Average	1.3058e-010
☐ Standard Deviation	0.
⊞ **Inflation**	
⊞ **Advanced**	
⊞ **Statistics**	

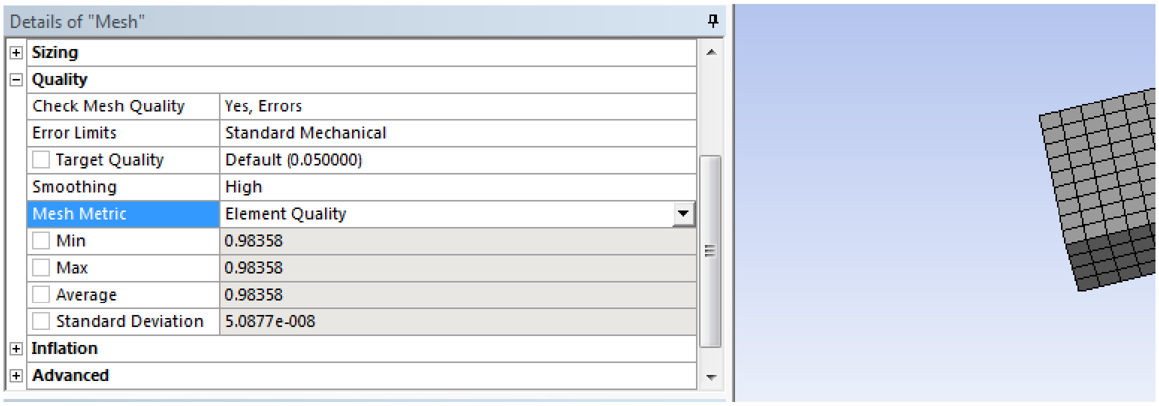
Figure (**3.6**): Skewness of the mesh that used in this study.



Details of "Mesh"

⊞ Sizing	
⊟ Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Standard Mechanical
☐ Target Quality	Default (0.050000)
Smoothing	High
Mesh Metric	Aspect Ratio
☐ Min	1.2
☐ Max	1.2
☐ Average	1.2
☐ Standard Deviation	8.4795e-008
⊞ Inflation	
⊞ Advanced	

Figure **(3.7)**: **A**spect ratio of the mesh that used in this study.



Details of "Mesh"

⊞ Sizing	
⊟ Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Standard Mechanical
☐ Target Quality	Default (0.050000)
Smoothing	High
Mesh Metric	Element Quality
☐ Min	0.98358
☐ Max	0.98358
☐ Average	0.98358
☐ Standard Deviation	5.0877e-008
⊞ Inflation	
⊞ Advanced	

Figure **(3.8):** Element quality of the mesh that used in this study.

3.3.5 Boundary Condition

The step inclined to apply suitable external loads and boundary conditions. The load in the ANSYS Workbench software will be at one side, and the other side was a fixed support, it presented in figure **(3.9).**

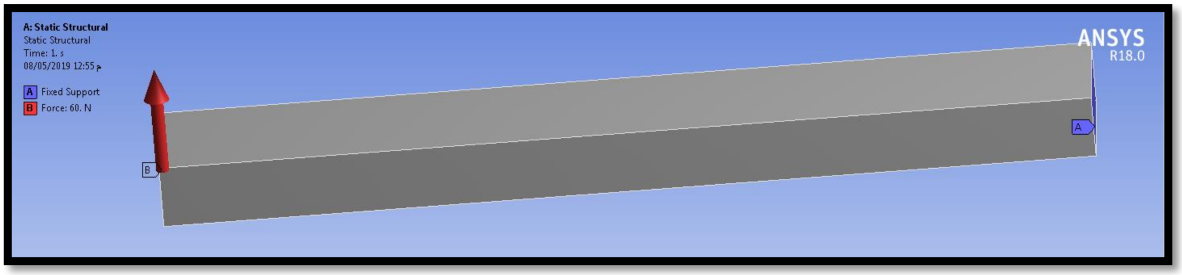


Figure **(3.9)**: The model with a fixed support and applying load.

3.3.6 Fatigue Solution

The fatigue results can be obtained from selection fatigue tool and insert life option at different loads in Ansys 18 workbench.

In order to complete the total solution of any case, other important parameters are added, such as mass density, Poisson ratio, ultimate stress, tensile yield stress and experimental S-N curves, and Young's modulus [23] as shown in the table **(3.2)**.

Table **(3.2)** Properties Insert to ANSYS (Obtain from Experimental Work)

Material	Proof strength (MPa)	Ultimate strength (MPa)	Modules of elasticity (GPa)	Poisson ratio
6061-T6	241	280	68.9	0.33
Single pass	168	242	59	0.33
Double pass	178.5	251	65.3	0.33

By using the alternating stress, it is important to mention that the fatigue analysis used in the present work is based on fully-reversed load (R = -1) and Soderberg theory, which is the most conservative theory, as shown in figure **(3.10)**.

$$\frac{\sigma_a}{\sigma_e}+\frac{\sigma_m}{\sigma_y}=1 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3.7)$$

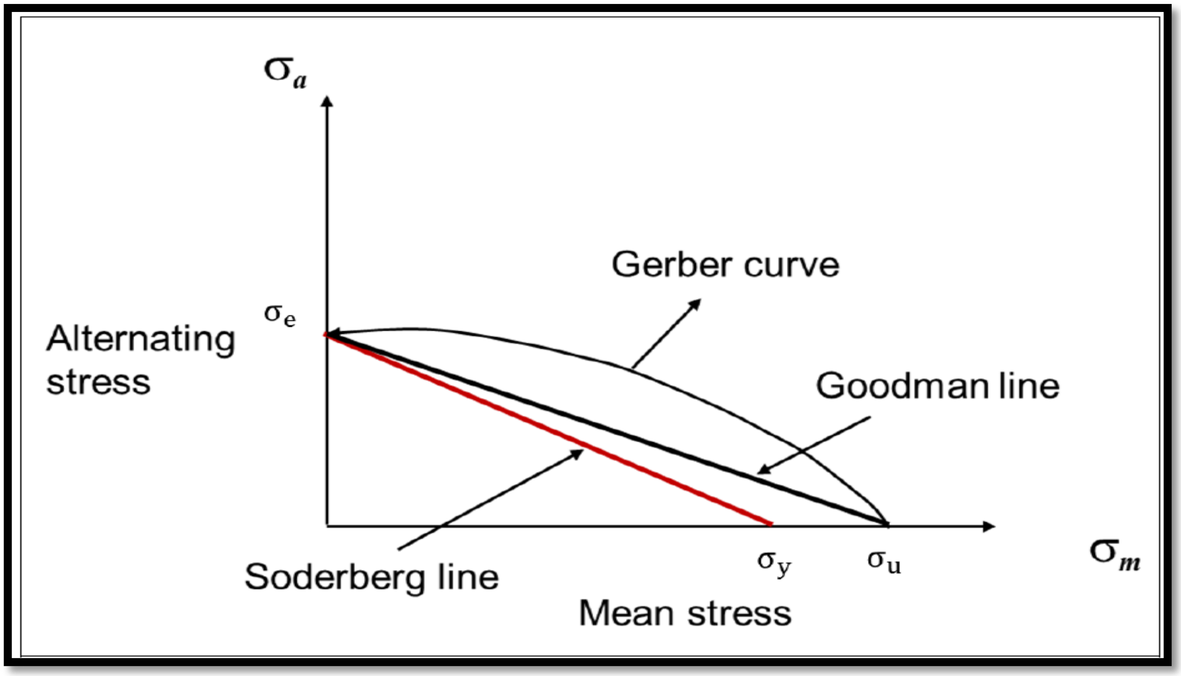


Figure **(3.10):** The effect of mean stress on fatigue life [71].

The endurance limit used for the numerical analysis is based on the 10^6 cycle life which is used in the ANSYS package.

After providing all the parameters mentioned above followed by running ANSYS software, the theory of failure of aluminium alloy is assumed by Von Misses theory. The final results will be obtained for up to 10^6 cycles, and the following results are obtained:

The maximum total deformation, fatigue life and safety of factor were obtain from ANSYS

The maximum equivalent stress tool depends on the maximum equivalent stress failure theory for ductile materials, as in equation **(3.8)** [72], also mentioned to as the Von Mises theory, maximum shear strain theory. The maximum equivalent stress failure theory supported by simulation, this theory is generally the best suitable for ductile materials, like as steel, aluminium, and brass [23], as presented in figure **(3.12)**.

$$\sigma_e = [\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}]^{0.5} \quad \ldots\ldots\ldots\ldots (3.8)$$