4 SOFTWARE VERIFICATION

In order to check the fatigue results and modelling techniques of ABAQUS and FE-Safe software, a test should be performed coupling the software to verify the properties and methods used in the simulation. Fatigue testing facilities are not available in Sri Lanka. Therefore other researchers' published tests have been used for modelling. This has been used primarily because the case study fatigue results cannot be verified through laboratory tests.

4.1 Simulation of uniaxial fatigue

Zhou, et al., 2008 have conducted an experimental study on the low cycle fatigue of stainless steel reinforcement bar specimens. An attempt is made to simulate the standard uniaxial fatigue test done by Zhou, et al using Abaqus/FEA and fe-safe commercial finite element software. Experiment is modelled in Abaqus to obtain the maximum stresses and strains under a specified loading condition first. Then the stress-strain data is imported to fe-safe to predict the fatigue behaviour under cyclic loading. The test apparatus and specimen dimensions are given below.

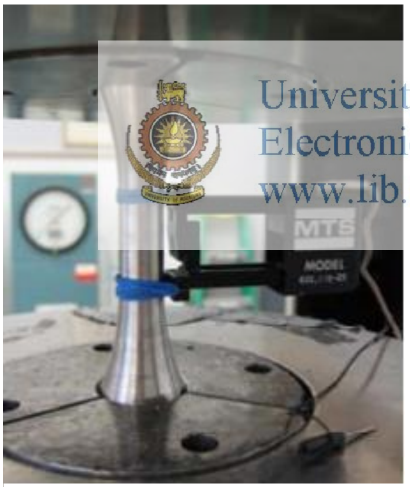


Figure 4.2: Test apparatus

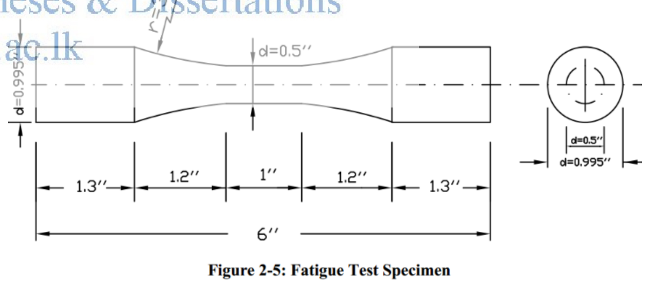


Figure 4.1: Specimen dimensions

4.1.1 Material Properties

The simulation was done for stainless steel 316LN test specimens. The cyclic properties and fatigue properties extracted from the literature are given in Table 4.1:

Table 4.1: Material properties of stainless steel 316LN test specimens

Mechanical Properties	**Cyclic Elastic properties** (Zhou, et al., 2008)	
	Young's modulus (MPa)	Poisson's ratio
	199817	0.33
	Cyclic Plastic properties (J.Shita, et al., 2013)	
	Yield stress(MPa)	Plastic strain
	270	0
	300	0.0025
	330	0.0075
	350	0.0125
	370	0.0175
	400	0.04
Fatigue Properties (Fe-safe, 2014)	B	-0.0835
	C	-0.5142
	ε_f'	0.476
	σf'	703.4

4.1.2 Material model

The hardening model is very important in simulating the fatigue behaviour. The intial transient behaviour is not important since the material comes to stabilzed cyclic response easily. However the stabilized cyclic response is presnt in majority of the fatiguelife. Therefore non-linear kinematic hardening model is used which is similar to that's described in (Staudinger & Reiter, 1997)

4.1.3 Modelling with ABAQUS

Abaqus standard is used for this analysis since the dynamic nature of the forces are negligible.3D20R solid elements with varying mesh size are used in order to model the stainless steel specimen. A finer mesh was used to model the central region to obtain a detail stress distribution.

General static analysis is performed neglecting the geometric nonlinearity. Bottom grip was restrained to move in all degrees of freedom and top grip was allowed to translate along the longitudinal axis of the specimen as shown in figure below.

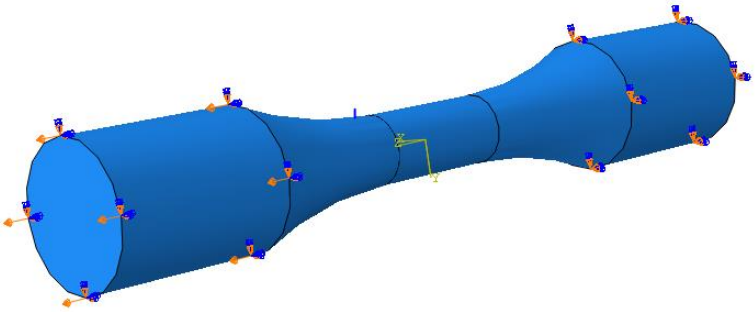


Figure 4.3: Idealized boundary condition in Abaqus

The model was run for different strain levels to construct the stress strain behaviour. Figure 4.4 shows strain distribution for a simulation subjected to 1.77% overall strain. Figure 4.5 illustrates the cyclic stress strain behaviour of a node at the centre of the specimen for the same 1.77% strain amplitude. The strain gauge in the experimental investigation performed by Zhou, et al., 2008 is located between the end points of the middle uniform cylindrical area. Therefore the strain values presented here refers to the same region.

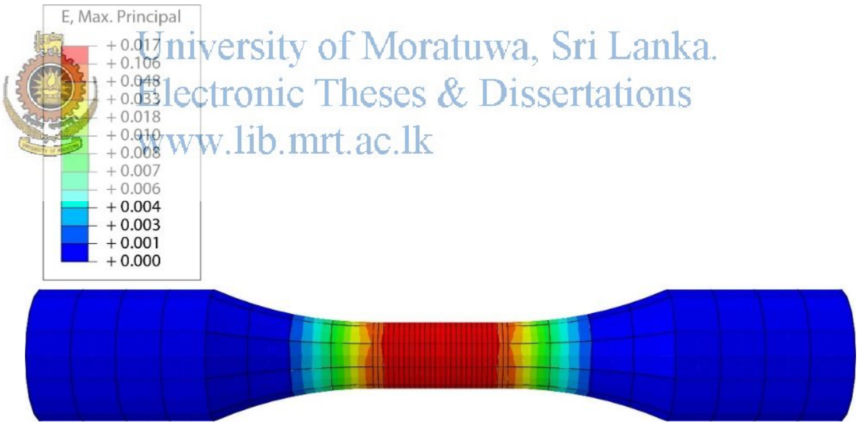


Figure 4.4: Abaqus simulation of 1.77% strain amplitude

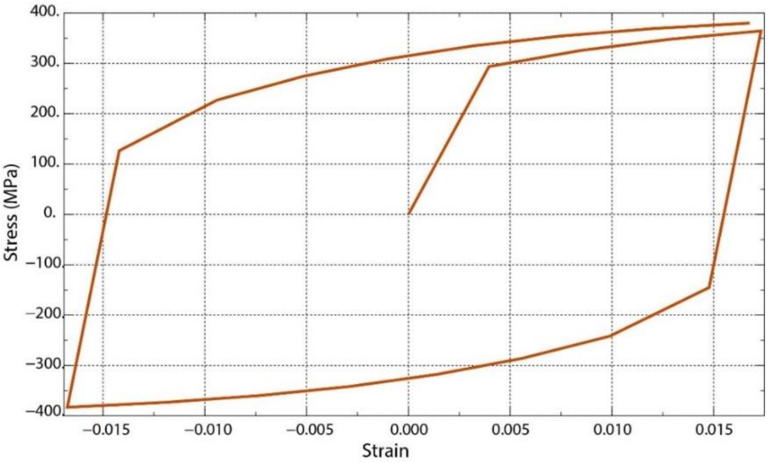


Figure 4.5: Principle stress strain response of an element in the central region

4.1.4 Fatigue analysis using FE-safe

Both the stress and strain datasets for each strain level are imported from Abaqus to the fe-safe software. By default Fe-safe assumes the stress datasets as elastic blocks. Elastic plastic analysis feature in fe-safe requires elastic plastic stress strain datasets for finite elements which is more accurate. Therefore an elastic plastic finite element analysis must be carried out. By combining stress dataset and strain dataset for each increment elastic plastic block was generated. The inbuilt fe-safe material database for S316 stainless steel is used for fatigue simulation and the surface finish was assumed as mirror polished.

Brown-miller equation with morrow's mean stress correlation was selected as the algorithm. Although the loading is uniaxial, uniaxial strain life equations cannot be used in fe-safe for 2D or 3D objects as stress and strains are not uniaxial.

4.1.5 Results and Discussion

Two sets of simulations were performed using an elastic plastic block but varying the surface finish to observe the sensitivity. Another simulation was performed with an elastic block. Table 4.2 compares the results obtained for the simulations against the experimental results obtained by (Zhou, et al., 2008).

Table 4.2: Comparison of fatigue life results

Number of reversals to failure (2Nf)				
Strain amplitude (%)	Experimental result (Zhou, et al., 2008)	Abaqus and fe-safe simulation		
		Using Elastic Plastic block		Elastic block with Neuber's rule
		Mirror polished	Rough(Ra-2um)	Mirror polished
0.99	1322	1198	552	54487
1.28	791	616	336	42369
1.5	576	411	225	35483
1.77	413	292	162	31438
2.1	293	216	120	28763
2.4	225	162	90	27064

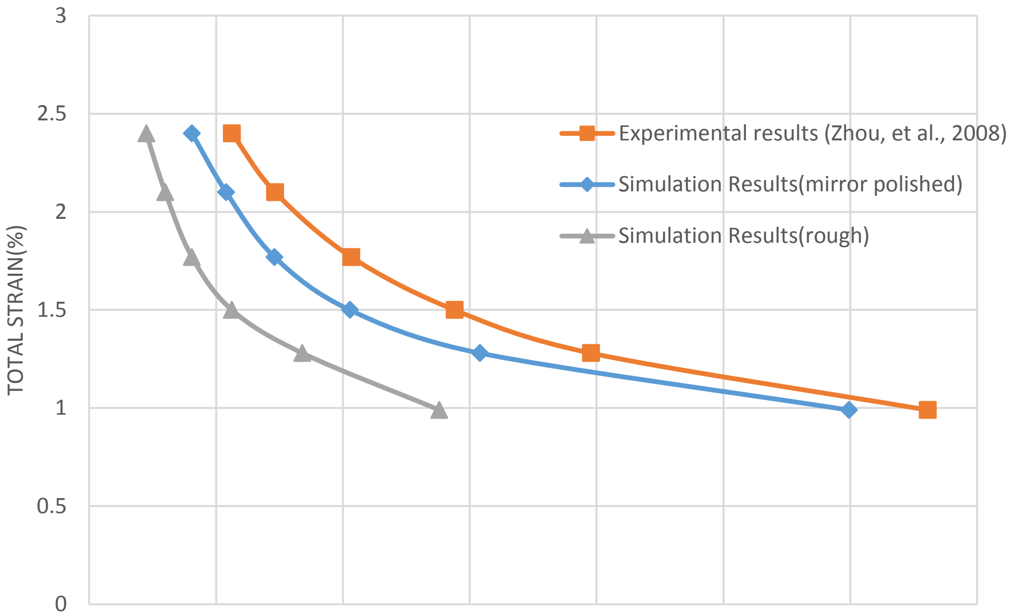


Figure 4.6: Comparison of S-N curves of simulation and experimental results

As shown in the Figure 4.6 the mirror polished surface finish which is the actual condition in the original test gave the closest behaviour to tested results. The simulation is within 25% range of the actual values. The original experimental results indidate that there is a 10%-20% variation of the experimental curve presented here. Therefore the simulation results is in an accpetable range.

The rough surface finish that is present in type 316 rebars (Kalpakjian, 6th edition) will give fatigue limits in the range of half that was for mirror polished. This indicates the surface finish is a primary factor for fatigue life of a material. As shown in Table 4.2 the simulation with an elastic block with Neuber’s rule gaves large fatigue lives which are not acceptable. The reason as explained in 4.1.4 is neuber’s rule being valid only when stress redistribution is insiginifcant. In this experiment whole central region of the speciemn is subjected to plastic stresses. Therefore elastic-plastic stress block method is preferred.

4.2 Simulation of multi-axial fatigue

Glodež & Knez have conducted an experimental study on the fatigue behaviour of high strength steel crawler crane arms. An attempt is made to simulate the experimental fatigue test done by Glodež & Knez using Abaqus/FEA and fe-safe commercial finite element software. Experiment is modelled in Abaqus to obtain the maximum stresses and strains under a specified loading condition first. Then the stress-strain data is imported to fe-safe to predict the fatigue behaviour under cyclic loading. The test apparatus and specimen dimensions are given in figure 4.7.

The simulation was done for S1100Q test specimens. The cyclic properties and fatigue properties extracted from the literature are given in the table below.

Table 4.3: Material properties of S1100Q test specimens

Mechanical Properties	**Cyclic Elastic properties**	
	Young's modulus (MPa)	Poisson's ratio
	194889	0.3
	Cyclic Plastic properties	
	Yield stress(MPa)	Plastic strain
	875	0
	1000	0.015
	1100	0.0766
Fatigue Properties (Fe-safe, 2014)	b	-0.0997
	c	-0.978
	ε_f'	9.93
	σf'	2076Mpa

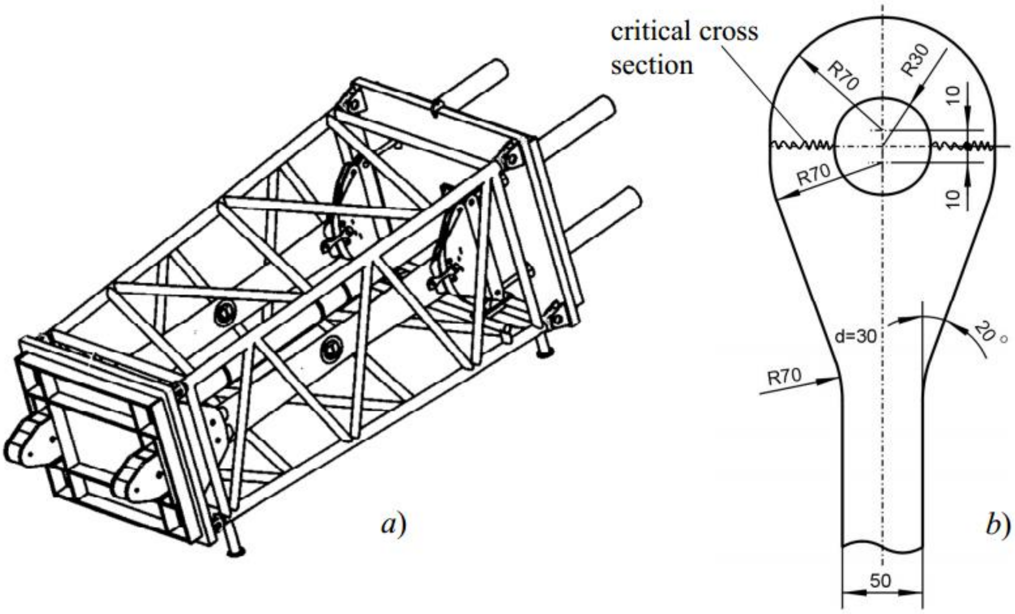


Figure 4.7: Fatigue testing machine (a) and testing bar (b)

4.2.1 Modelling with ABAQUS

Abaqus explicit is used for this analysis since the dynamic nature of the forces are significant while modelling the interaction with the bolt. The bolthole was idealized as analytical rigid shell assuming rigidity (figure 4.8). C3D8R: 8-node linear brick solid elements with varying mesh densities are used in order to model the specimen (figure 4.9). A finer mesh was used to model the central region around the opening to obtain a detail stress distribution.

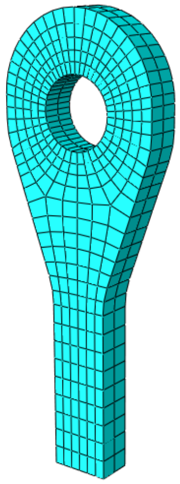


Figure 4.9: Mesh of the crane bar

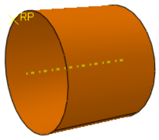
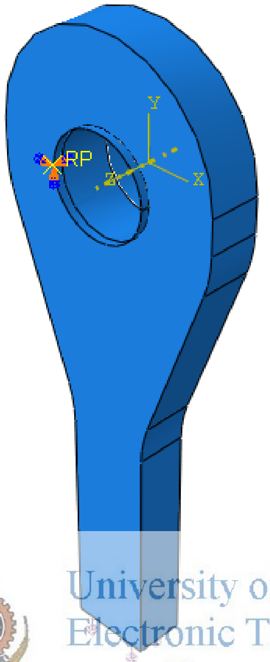


Figure 4.8: Analytical rigid shell for bolthole

Dynamic explicit analysis is performed considering the geometric nonlinearity. The bolt is restrained in all degrees of freedom. Surface to surface contact of Hard and frictionless contact using penalty contact method was used between the bolthole surface and bolt. The load expected in the bar was applied from the bottom end as shown in the figure 4.7 below.



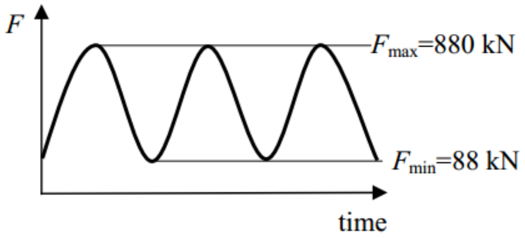
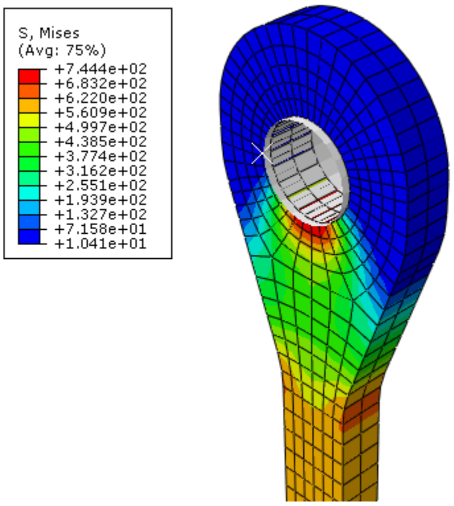


Figure 4.11: Applied cyclic loading

Figure 4.10: Loading and boundary conditions

The applied cyclic load is shown in the figure 4.8. This was applied as a periodic type amplitude with relevant parameters. Since the stabilized cyclic properties are given. The model comes to stabilized cyclic response in one cycle.



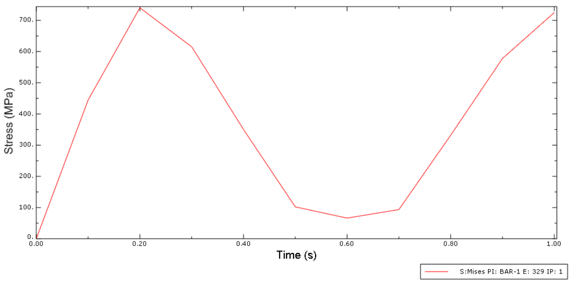


Figure 4.13: Misses stress distribution at the highest loading level

Figure 4.12: Stress variation of a node just below the bolt hole

4.2.2 Fatigue analysis using FE-safe

Both the stress and strain datasets for each strain level are imported from Abaqus to the fe-safe software. An elastic plastic finite element analysis was carried out. The fatigue material properties given by Glodež & Knez, 2007 was used for the analysis. Surface finish was assumed as fine machined (4 um< Ra < 16um). See Appendix 1

Brown-miller equation with morrow's mean stress correlation was selected as the algorithm.

4.2.3 Comparison of Results

Table 4.4: Results of the simulation and experiments

Experimental Results				FEA results
Test 1	Test 2	Test 3	Test 4	29979
38029	26727	24795	29036	

The results indicate that there is a closer relationship with simulated results and experimentally ested results. But it should be noted that the variations of surface finish idealisation can change the results by large amounts.

5 CASE STUDY

5.1 Introduction

Proposed tower is a 350m tall tower located in Colombo Sri Lanka.The tower mast at the top is reinforced concrete at lower half and steel at the its' upper half. The steel mast is 64.1m tall and it is made out of tubular sections (figure 5.1). The section shape and size change in 3 stages.

There are openings at all these stages. Out of these the openning at the bottom which is subjected to highest stress level and largest in size is valnerable to fatigue failure under cyclic loading.

Since the structure is at 300m height the wind induced vibrations are significant. There is a probability that the wind induced vibrations can cause fatigue. In order to check its vulnerability to fatigue and to compare the sensitivity to thickness, shape of the openning different finite element models are checked for fatigue.

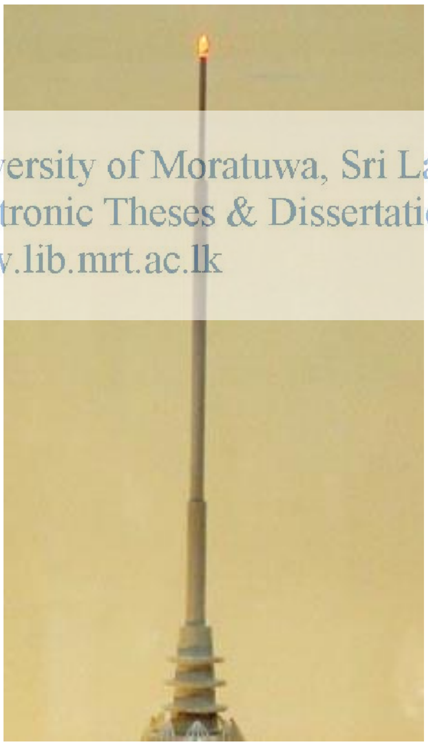


Figure 5.1: Proposed shape of the Steel tower