Chapter 6

FATIGUE CALCULATIONS

6.1 Scope

The previous chapter showed through various case-studies how the fatigue loads of the vehicle structures were measured. This chapter will show, using the different case-studies, the methodology in converting the measured data into a fatigue equivalent static load.

6.2 Aluminum Dry-bulk Tanker

The following methodology was used to calculate the fatigue equivalent stress that the $40m^3$ aluminium Spitzer tanker experienced during the measurement exercise. All the strain gauge measurement channels were used to calculate the damage experienced during the two trips. The spreading exercise was analysed separately from the Tzaneen excursion. The two separate damages will be used to calculate an accurate damage that a typical vehicle will experience throughout its life. The damages of all the data files were calculated as follows: the data generated by the pre-processing, is analysed with the use of the so-called *rain-flow* algorithm. The end result of this exercise is a table in which different stress amplitudes are displayed, with its various occurrences (refer to table 6.1). This table gives the following information: The table shows that

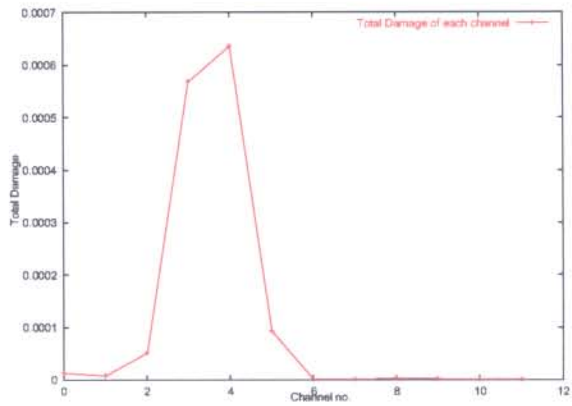


Figure 6.1: Damage measured during Tzaneen trip

a certain stress ($\Delta\sigma_i$) was repeated n_i times during the test trip. For example, the vehicle was subjected to an amplitude stress of 0.034 MPa that was repeated 268 times. The following step in the whole procedure is to evaluate the results of the rain-flow counting exercise. The stress-life equation [1] is used to calculate the amount of cycles (N_i) for each stress range ($\Delta\sigma_i$). Refer to equation (2.6), page 21.

The σ_f value, is selected arbitrary from the aluminium structural code [9] (as long as the value is bigger than the $\Delta\sigma_i$ value). The b value is selected using the assumption that the fatigue fracture will occur at a weld. The value of b is $-1/3$. The N_i and n_i values are now used in Miner's damage equation (2.12), page 29. Figure 6.1 displays the total damage of each channel for the measurement trip to Tzaneen. (This damage *does not* include the spreading operations data) The above-mentioned procedure was carried out with the help of custom programmed *Matlab* programs.

The assumption is made that while the vehicle travelled from Brakpan to Tzaneen, an average speed of 80 km/h was maintained. The total distance of measured data recorded is thus 110 km. The damage (D_{110km}) is subsequently calibrated of measurements during the Tzaneen trip. The data of the spreading exercise must, however, also be considered. The same procedure is

Table 6.1: Rain-flow results - dry bulk tanker

$\Delta\sigma_i$ [MPa]	n_i (cycles)
0.0168	42
0.0337	**268**
0.0506	181
0.0675	124
0.0844	102
0.1013	78
0.2365	10
0.3379	3
0.3886	1
0.4731	1

followed for the spreader measurement exercise to calculate its damage (D_{sprd}). The assumption is made that 5% of the vehicle's life is dedicated to similar spreading exercises. It is also assumed that the vehicle travels at an average speed of 10 km/h. The vehicle thus travels the equivalent of 23km during the measurement period.

The total damage (D_{tot}) can thus be calculated if it is assumed that the vehicle would travel 1 000 000 km during its lifetime. Refer to equation 6.1.

$$
\begin{aligned}
D_{tzan} &= \left(\frac{1\,000\,000km - 50\,000km}{110\,km}\right) \times D_{110km} \\
D_{sprd} &= \left(\frac{50\,000km}{23\,km}\right) \times D_{23km}
\end{aligned}
$$

$$D_{tot} = D_{tzan} + D_{sprd} \tag{6.1}$$

With the *total* damage known (D_{tot}), the equivalent stress range can now be calculated, using equation (6.2). Refer to table 6.3 to view the results.

Table 6.2: Total damage - dry bulk tanker

Channel no	D_{tzan}	D_{sprd}	D_{tot}
1	0.11347	0.00123	0.11470
2	0.07204	0.00195	0.07399
3	0.44689	0.01253	0.45943
4	4.90476	2.33337	7.23813
5	5.49477	0.33796	5.83273
6	0.81144	0.09027	0.90171
7	0.01332	0.00100	0.01432
8	0.00711	0.00057	0.00768
9	0.03029	0.00216	0.03245
10	0.02401	0.00193	0.02594
11	0.01050	0.00055	0.01105
12	0.01313	0.00079	0.01393

The σ_f value is arbitrarily chosen from the ECCS code [15], bearing in mind this value is cancelled out after further substitution in the fatigue equivalent equation. The σ_f value must be 'chosen' so that it is bigger than the $\Delta\sigma_i$ value.

$$\Delta\sigma_{eqv} = \left(\frac{n_{eqv}}{D_{tot}}\right)^b \sigma_f \tag{6.2}$$

where :

σ_{eqv} = equivalent stress

n_{eqv} = equivalent number of cycles (usually two million)

σ_f = fracture stress

D_{tot} = total damage

b = Basquin's fatigue strenght exponent

The finite element model was then subjected to a *unit vertical gravitational acceleration* load. The resultant stresses (S_{FEA}) of the finite element analysis are used to calibrate the results of the measurement exercise. Re-

Table 6.3: Equivalent stress results - dry bulk tanker

Channel no.	$\Delta\sigma_{eqv}$ $[MPa]$ (Tzaneen)	$\Delta\sigma_{eqv}$ $[MPa]$ (Spreading)	$\Delta\sigma_{eqv}$ $[MPa]$ (Combined)	S_{FEA} $[MPa]$ (Combined)	a_{eqv} $[g]$
1	11.62	2.57	11.66	24.8	0.47
2	9.98	3.00	10.07	28.0	0.36
3	18.35	5.57	18.51	23.5	0.79
4	42.35	16.71	43.20	18.2	2.55
6	22.38	10.76	23.18	18.8	1.23
10	6.92	2.98	7.10	14.3	0.50
11	5.25	1.96	5.34	6.03	0.89
12	5.66	2.22	5.77	12.1	0.48

fer to figures 6.2(a) and 6.2(b) to view the finite element calibration stresses. The equivalent acceleration load is calculated using equation 6.3. It should be noted that the stress comparison is done on the *same* geometric position on the vehicle. Not all the strain gauge channels were used to calculate the equivalent acceleration load. The rosette gauge (channels 7-9) and the strain gauge in the pocket of rear-boom cross-member (channel 5) are omitted in the calculations. The rosette gauge's data is used to calculate the principal stresses (verification purposes), while channel 5 measures the *shear* stress concentration in the rear cross-member pocket. The equivalent acceleration (a_{eqv}) represents a fatigue load that the vehicle would endure over 1 million kilometers. The equivalent acceleration load can now be used to evaluate the structure.

$$a_{eqv} = a_{cal} * \frac{\sigma_{eqv}}{S_{FEA}} \tag{6.3}$$

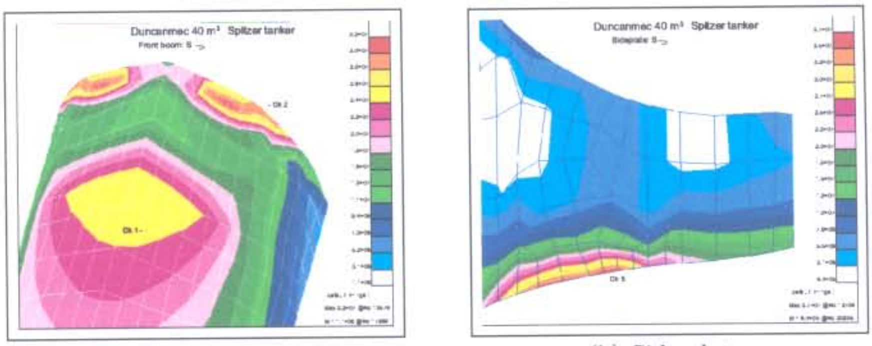
where :

a_{eqv} = equivalent acceleration load

a_{cal} = calibration acceleration load applied in the FEA to obtain S_{FEA}

σ_{eqv} = equivalent stress

S_{FEA} = Calibration or calculated stress using a FEA



(a) Front boom (b) Side plate

Figure 6.2: Bulk tanker - calibration stresses

The fatigue calculations of the data measurement exercises showed the following: A maximum load of $2.551g$ was calculated with the measurement data. This load was calculated at *channel 4*, on the outrigger of the compressor mounting (refer to table 6.3). The outrigger can be viewed as a cantilever beam with a large point load. The dynamic behaviour of the beam can therefore deform in higher order bending deformations. The Fatigue Equivalent Static Load method makes use of a calibration stress (using a FEA) in a first order bending mode. The FESL load calculated using these strain gauge measurements is therefore not accurate, and may be omitted. The following high quasi-static load occurred at *channel 6*, on the side plate. A load of $1.233g$ was calculated at the measurement point. This calculated load is near a *stress concentration*, and can therefore not be accurately interpreted. The rest of the channels gave results ranging between $0.36g$ and $0.89g$. The average of all the quasi-static loads is approximately $0.58g$.

The outcome of the fatigue calculations performed on the measurements gave excellent results regarding the usage of the FESL method. Channel 4 showed, as predicted, that higher order dynamic behaviour can not be used with the Fatigue Equivalent Static Load method. Channel 6 indicated the dangers of extrapolating measurements near high stress concentrations. The

Table 6.4: Final calibrated measurements - sub-frame of pick-up truck

Measurement position	Description	Damage (per file)	% of tot. life	Damage	a_{eqv} $[g]$
Right side bot. (channel 10)	Gravel, full	0.1111E-4	0.5	1.11	
	Secondary, full	0.1193E-4	0.5	0.99	
			Total DAMAGE	1.0531	**3.1**
Left wheel (channel 8)	Gravel, full	0.1037E-4	0.5	1.04	
	Secondary, full	0.0828E-4	0.5	0.69	
			Total DAMAGE	0.8642	**2.1**
Right wheel (channel 9)	Gravel, full	0.1393E-4	0.5	1.39	
	Secondary, full	0.0686E-4	0.5	0.57	
			Total DAMAGE	0.9825	**2.6**

rest of the channels showed an excellent correlation, with a narrow band of acceleration results. The narrow band of loads gives the designer confidence to extrapolate these loads to the whole structure.

6.3 Sub-frame of a pick-up truck

A profile of road usage was created so that the amount of damage can be calculated. The profile ratio is: 10:35:55 (gravel, secondary tar, highway). The high percentage chosen for highway usage includes usage on good secondary roads. The assumption was made that the total life-time of the vehicle will be 200 000 km. The second measurement exercise consisted only of gravel and secondary tar, while the vehicle was fully loaded. A ratio was therefore calculated to take the highway and empty damage into account (using the data obtained from the first exercise). The calculations showed a ratio of approximately two between the damage accumulated during gravel and secondary tar measurements, and the highway measurements. The *second* measurement exercise was also subjected to a fatigue calculation where the damage was calculated. The data of the two measurement exercises was processed and combined to give an accurate and realistic total damage figure (refer to table 6.4).

The total damage was used to calculate a fatigue equivalent vertical ac-

Table 6.5: Rain-flow results on left bracket - bus bracket

Amplitude Forces Δf_i $[N]$	**Mean Forces** $[N]$	**Reversals** n_i (cycles)
471	26304	70
941	25833	428
3295	23480	448
3765	23010	320
4236	22539	262
4706	22068	**130**
8001	18774	23
8472	18303	13
9884	16891	12
10354	16420	8
10825	15950	4
12708	14067	2
13178	13597	1
13649	13126	1

celeration cycle range (Δa_{eqv}), using equation 6.2, page 69. The equivalent load would cause the same damage as the modified measurements (if repeated 2×10^6 times). This vertical acceleration would therefore occur 2 million times during the 200 000 km (refer to table 6.4). An average vertical acceleration of $2.5g$ was applied on an FE model. The fatigue analysis was done using the stress results of the FE analysis. The chassis was evaluated using the European structural fatigue code [15], where the stresses calculated are treated as stress ranges, repeated 2×10^6 times during the life.

6.4 Suspension bracket of a large passenger bus

The following analysis was done to calculate the actual damage that the suspension bracket experienced during the measurement exercises.

Table 6.6: Damage Calculations on left bracket - bus bracket

File name	**Load Condition**	**Distance** $N_{meas}\,km$	**Road Surface**	$\mathbf{D_{file}}$
ctctle.mat	Empty	$5.5\,km$	Secondary Tar	397.6
ctctpful.mat	Laden	$5.5\,km$	Secondary Tar	407.9
ctchwful.mat	Laden	$6.5\,km$	Highway	152.4
ctcgrful.mat	Laden	$3\,km$	Gravel	555.0

6.4.1 Equivalent static force

The damage of the data files of the measured shock absorber forces are calculated as follows: the data is analysed with the use of the *rain-flow* algorithm [15]. The end result of this exercise is condensed in a table in which different force amplitudes, mean forces and their corresponding reversals are shown (refer to table 6.5). Table 6.5 gives the following information: the table shows that a certain force (Δf_i) was repeated n_i times during the test trip. For example, the bracket was subjected to an amplitude force of $4706N$ that was repeated 130 times.

The following step in the whole procedure is to evaluate the results of the rain-flow counting exercise. The stress-life equation [1] is used to calculate the amount of cycles (N_i) for each force range (Δf_i) (refer to equation (2.6), page 21). It should be noted that ECCS code contains stress data. The stress data is calibrated to a force by means of the finite element model. The b value is selected based on the assumption that the fatigue fracture will occur at a weld. The value of b is $-1/3$. The N_i and n_i values are now used to calculate the damage in Miner's damage equation (2.12). The above-mentioned procedure was carried out with the help of the *Matlab* programs.

The damages calculated by equation (2.12) are calibrated to constitute a damage equal to a lifetime of 3 000 000 km. Refer to equation (6.4).

$$D_{file} = \left(\frac{3\,000\,000km}{S_{meas}\,km}\right) D_{perfile} \tag{6.4}$$

where :

$$\begin{aligned} D_{file} &= \text{calibrated damage of the measurement file} \\ S_{meas} &= \text{distance of the measurement file} \\ D_{perfile} &= \text{actual damage of the measurement file} \end{aligned}$$

These damages are calculated for each measurement file. The damages (D_{file}) are then combined according to the user profile, as supplied by the client. The following equations show two user profiles. Refer to equations (6.5) and (6.6). Also refer to table 6.6 and table 6.7.

$$\begin{aligned} X &= (D_{ctctle} + D_{ctctpful})/11\,km \\ Y &= D_{ctchwful}/6.5\,km \\ Z &= D_{ctcgrful}/3\,km \end{aligned}$$

$$D_{up1} = (X \cdot 40\% + Y \cdot 50\% + Z \cdot 10\%) \times 3\,000\,000 km \tag{6.5}$$

$$D_{up2} = \left(\frac{D_{ctctle} + D_{ctctpful}}{11\,km}\right) \times 3\,000\,000 km \tag{6.6}$$

where :

$$\begin{aligned} D_{up1} &= \text{damage of user profile one} \\ D_{up2} &= \text{damage of user profile two} \end{aligned}$$

The following step is to calculate the *fatigue equivalent stress range* ($\Delta\sigma_{eqv}$) for 2×10^6 cycles, that will equal the combined damages. This is done using equation (6.2), page 69. The finite element model is subsequently employed to calibrate the FEM *force* relative to the fatigue equivalent static stress ($\Delta\sigma_{eqv}$). Table 6.7 displays the equivalent forces for the combined damages for the left

Table 6.7: Equivalent Forces - bus bracket

User Profile	Left		Right	
No.	Damage (D_{up*})	ΔF_{eqv} $[kN]$	Damage (D_{up*})	ΔF_{eqv} $[kN]$
1	178.55	56.21	0.82	9.36
2	219.69	60.23	1.55	11.6

and right bracket (D_{up1} & D_{up2}).

6.4.2 Data and analysis verification

During the measurement exercise, only the left suspension bracket input forces were measured. To ensure the integrity of the measurement exercise, especially regarding the right suspension bracket, the above mentioned exercise was verified. The verification was done using the following methodology:

- The finite element stresses at the strain gauge position are obtained from the FE model.
- The strains that cause these stresses are calculated through solving equations 6.7, 6.8 and 6.9 simultaneously.
- A relationship between a force and the strain at the measurement position is therefore established ($C_{cal} = F_{fem}/\epsilon_{fem}$).
- The following step is to compare the measured strains to the measured forces, using the multiplication factor (C_{cal}). The measured strains and forces clearly corresponded with each other.

$$\sigma_x = E\left(\epsilon_x - \nu(\epsilon_y - \epsilon_z)\right) \tag{6.7}$$

$$\sigma_y = E\left(\epsilon_y - \nu(\epsilon_x - \epsilon_z)\right) \tag{6.8}$$

$$\sigma_z = E\left(\epsilon_z - \nu(\epsilon_x - \epsilon_y)\right) \tag{6.9}$$

It is thus shown that the *calculated* finite element stress at the position of the strain gauge compares very favourably with the *measured* stresses. The resulting stresses on the FE model can therefore be linearly calibrated to the required fatigue equivalent stress range ($\Delta\sigma_{eqv}$). The fatigue equivalent static force for the right suspension bracket can subsequently also be calculated ($\Delta F_{eqv} = X_{FEA}\Delta\sigma_{eqv}$). Refer to table 6.7.

6.5 Suspension bracket of a 4x4 pick-up truck

6.5.1 Fatigue life prediction on existing design

In Chapter 5 it was shown how the measured force histories were converted to stress histories (refer to equation (5.1), p.64). Material fatigue properties were assumed, based on the tensile properties of the material. The damages calculated for each of the measured sections were appropriately accumulated to obtain the correct mix of tar, gravel, track and log sections for one cycle of the durability route (D_{cycle}). The distance to failure (s_{fail}) was subsequently calculated using equation (6.10). The original component failed at 35 000 km ($s_{fail} = 35\,000\,km$). The measurement data, as well as the FE model could therefore be calibrated to give a realistic representation of the component.

$$s_{fail} = \frac{1}{D_{cycle}} \times s_{cycle} \tag{6.10}$$

where :

s_{fail} = distance to failure

D_{cycle} = damage of one cycle

s_{cycle} = distance of one cycle of the durability route

Figure 6.3: Original suspension bracket - 4x4 pick-up truck

6.5.2 Fatigue criterion for modified design

The modified component must at least survive 120 000 km on the durability route ($s_{fail} = 120\,000\,km$). Based on the same calculations performed for the existing design, a fatigue criterion for a modified design was derived. Assuming that the component is modified with additional welded components, using the fatigue properties prescribed by BS 7608: 1993, the calculations indicated that the nominal stresses adjacent to the weld should, for a specific calibration input load, not exceed 4 MPa. For the parent metal, peak stresses away from welds should be below 10 MPa for the applied loading.

6.5.3 Qualification testing

Durability rig testing was performed on a *baseline* (original design) bracket (see figure 6.3), as well as a prototype modified bracket (see figure 6.4). The testing was performed using a servo-hydraulic actuator, inducing single amplitude sine

Figure 6.4: Modified suspension bracket - 4x4 pick-up truck

wave loading onto the bracket. The amplitude of loading was fixed to be equivalent to the average measured peak-to-peak loading experienced on the logs section. Based on fatigue calculations, it was estimated that it would be required to complete 1.6 million cycles of the rig testing to induce the same damage as would 120 000 km of durability route testing.

During the actual testing, the baseline specimen survived for 126 400 $cycles$ before failing. The modified design was only tested to 800 000 $cycles$ (without failing), implying a maximum life expectancy of 222 000 km in terms of a durability route distance.

6.6 Closure

The chapter illustrated how measurement data is processed into fatigue equivalent static loads. The case-studies indicated the various methods of calculating the damages that were measured. These damages were then used to calculated the FESL loads, that can be used to assess a vehicle structure. The chapter also showed the use of the finite element process to obtain these fatigue loads. The following chapter will describe how the FE models were created and used in solving the fatigue loads, and deal with various structural problems.

Chapter 7

FINITE ELEMENT STRUCTURAL ANALYSIS

7.1 Scope

The finite element analysis is an essential and valuable tool to assess the integrity of a structure. As mentioned in the previous chapters, the finite element analysis can be used during the various phases of the fatigue equivalent static load method. Chapter 5 illustrated how the data was measured. In Chapter 6 the resulting damages were used in conjunction with the FE calibration stresses to calculate the fatigue equivalent static load. This chapter deals with the finite element analyses of the various case studies.

7.2 Aluminum Dry-bulk Tanker

Geometry and Finite Element Modelling

The geometry of the finite element model was created using the manufacturer's drawings of the vehicle. These drawings included the shell assembly, item lists and assembly drawings of the front and rear assemblies. Unigraphics (UG), an advanced 3-D CAD program, was used to create all of the trailer parts. UG

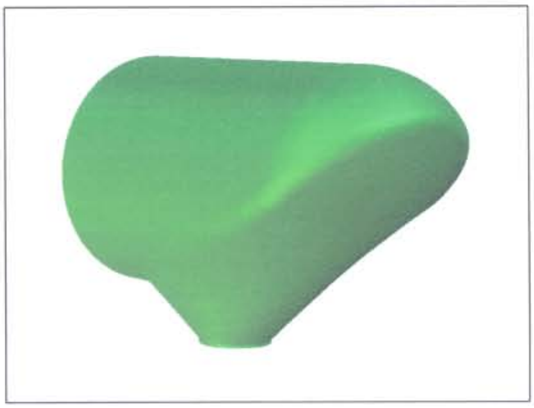


Figure 7.1: Unigraphics CAD model - hull

was especially useful to create the complex geometry of the tank shells (refer to figure 7.1). The 3-D geometric model was then imported into a finite element modeller, *MSC.Patran*. In the *Patran* package, the complex geometry was processed to create the finite elements that form the core of the analysis. Shell elements were used to create the Spitzer Tanker (refer to figure 7.2). The whole model, with all its different components, was analysed with a finite element analysis solver called PERMAS. After the model was analysed by PERMAS, the results were viewed in *Patran* for further evaluation.

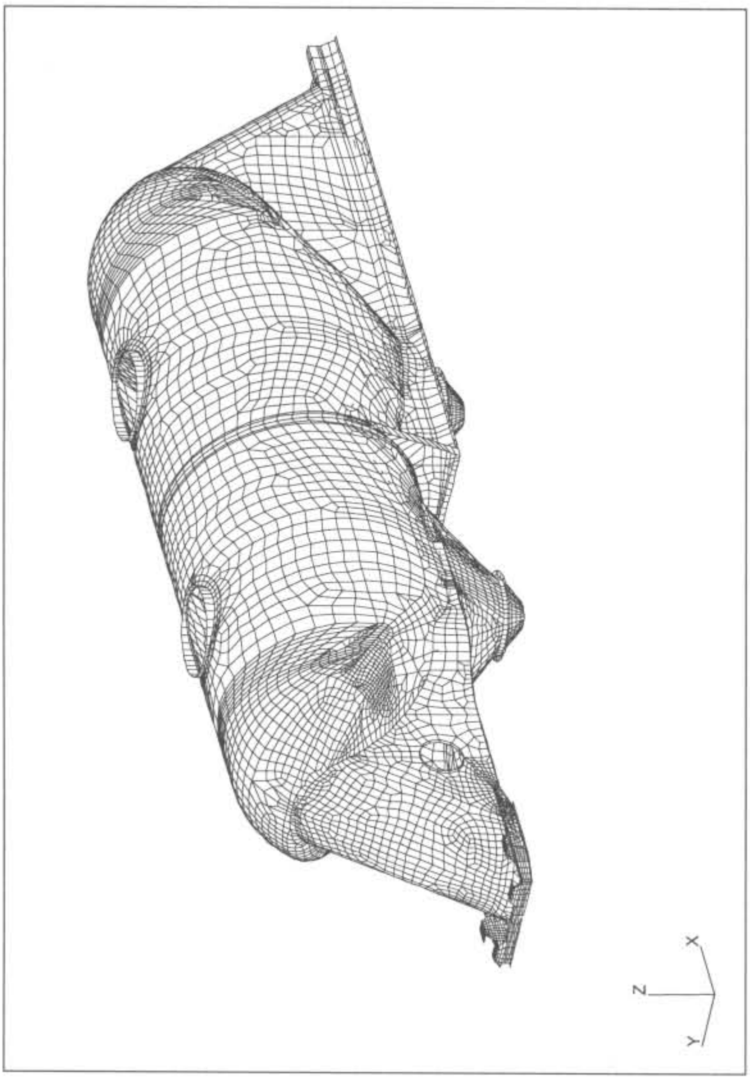
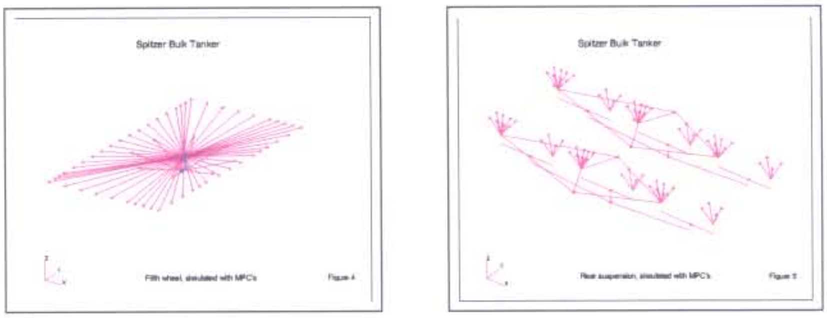


Figure 7.2: Bulk tanker - Finite Element Model



(a) Kingpin MPC, bulk tanker (b) Rear suspension

Figure 7.3: Multiple Point Constraints - bulk tanker

Materials and element properties

The material properties of the Spitzer tanker was aluminum. A Young's modulus of 80 GPa and the Poisson ratio of 0.3 was used. The properties were assigned to the mid-plane of the elements.

Loads and Boundary conditions

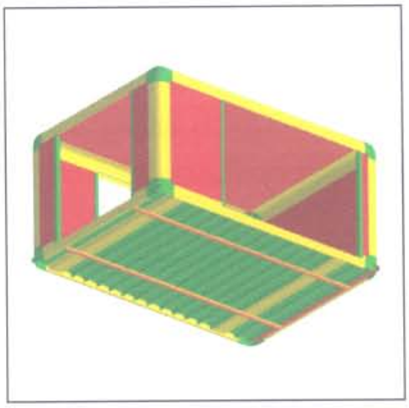
The model must however also be constrained. This is done at the fifth wheel and the suspension of the tanker. The fifth wheel, as well as the suspension is simulated with the use of MPC's[1] (refer to figure 7.3). The following equation was used to calculate the fatigue load case. The fatigue equation was based on the inputs obtained from the fatigue calculations stipulated in Chapter 5.

$$LC_{fat} = 0.58a_{vert} + 0.58P_{vert}$$

where :

$$a_{vert} = \text{unit load, vertical acceleration}$$
$$P_{vert} = \text{unit load, vertical pressure}$$

[1]Multiple Point Constraints



(a) Unigraphics CAD model - swop body

(b) Unigraphics CAD model - sub-frame

Figure 7.4: Unigraphics CAD models - sub-frame of pick-up truck

7.3 Sub-frame of a pick-up truck

Geometry and Finite Element Modelling

- *Chassis:* The chassis geometry was created using IGES and TIFF files that were supplied by the client. The chassis was subsequently modelled using a CAD package (Unigraphics) and exported to *MSC.Patran* for the finite element modelling.

- *Swap-body:* The geometry of the swap body was created in Unigraphics, using drawings supplied by the manufacturer (refer to figure 7.4). The finite element modelling was performed using *MSC.Patran*.

- *Cab:* Geometry of the cab was created in *MSC.Patran*. The geometry was created bearing in mind that the cab mounts to the chassis. The cab is only representative of the real article, and was created with the intention to achieve the correct stiffness of the whole structure.

- *Sub-frame:* The sub-frame was designed in Unigraphics (refer to figure 7.4). Special care needed to be taken to ensure that the sub-frame had the correct interface parameters with respect to the swap-body and the vehicle chassis. The manufacturing aspects also played an important part in the design of the sub-frame. The geometry of the model was generated using the advanced parametric capabilities of Unigraphics. This was necessary if iterations were to be done on the sub-frame.

The mesh was created using mainly QUAD4 and TRIA3 lower-order elements. The front part of the chassis was created using BECOS beam elements. The rubber mounts in the model were simulated with beam elements. The engine and gear-box of the pick-up were simulated with point mass elements that were positioned on the assumed centre of mass of the components (see figures 7.5 and 7.6). Two iterations were performed on the sub-frame. These iterations were necessary for the following reasons:

- weight reduction,
- compatibility to the swap-body that was still in a development phase, and
- manufacturing aspects.

Various items of the sub-frame had to be modified to make the manufacturing of the sub-frame more cost-effective. These modifications included the holes in the channels, the thickness of the materials and the shape and position of gussets.

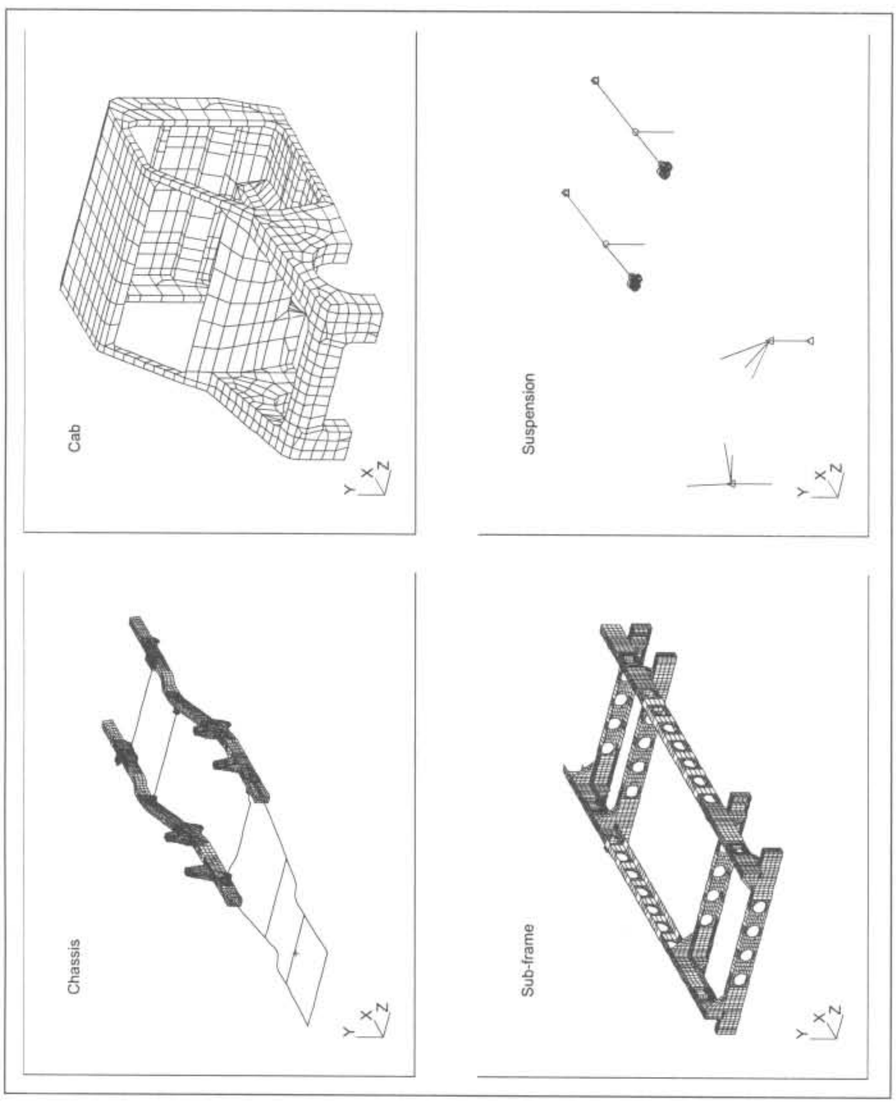


Figure 7.5: Finite Element Model: sub-frame of a pick-up truck

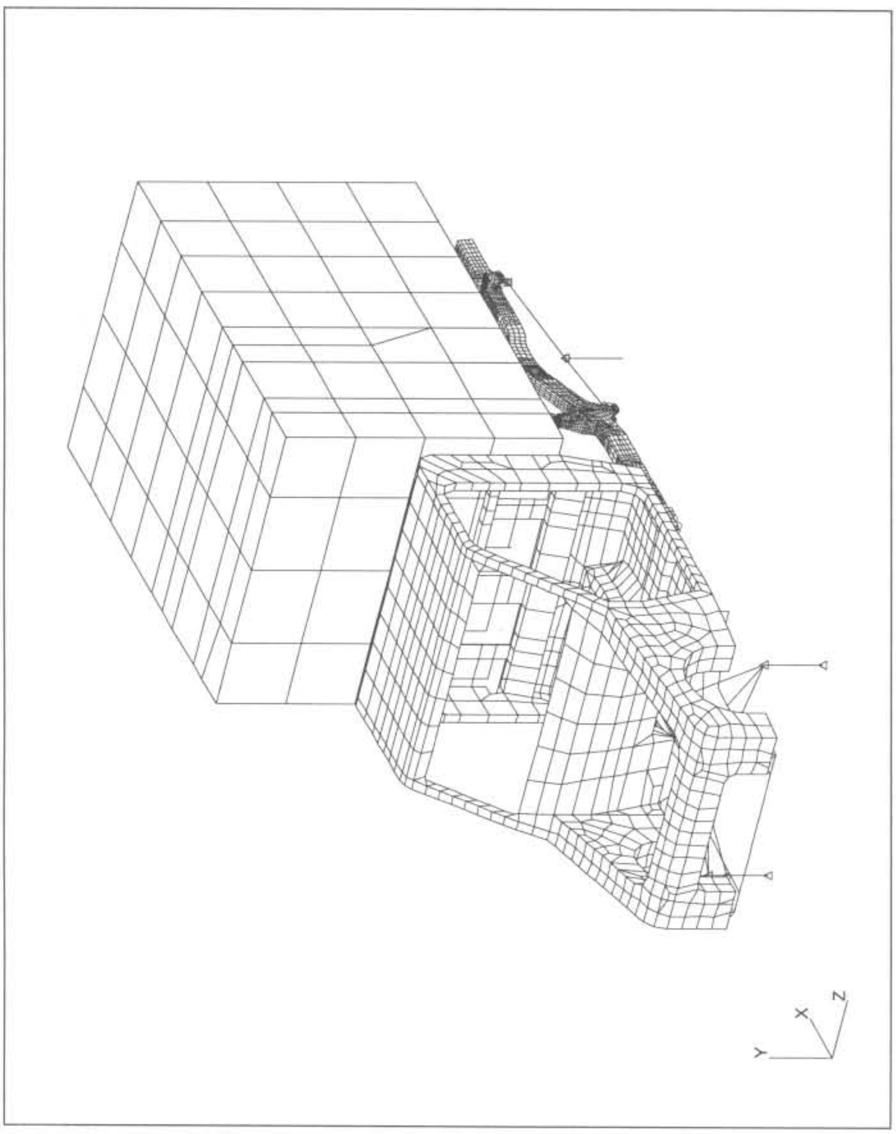


Figure 7.6: Finite Element Model: sub-frame of a pick-up truck

Materials and element properties

The steel chassis, sub-frame and cab elements were defined to have a Young's modulus of 210 GPa and Poisson ratio of 0.3. The prototype swap-body was constructed of aluminum. A Young's modulus of 80 GPa and the Poisson ratio of 0.3 was used. The properties were assigned to the mid-plane of the elements.

Load and Boundary conditions

The pick-up vehicle suspension was simulated with MPC's, beam and rod elements. The suspension was created to simulate the correct bending that the vehicle would have experienced due to vertical and longitudinal loading during the static and dynamic analyses. At the front, MPC's and rods were used to simulate the McPherson-type suspension. The McPherson suspension was connected at the wheel centres to the road surface. At the rear, MPC's were used to model the leaf springs and to connect the wheel centres to the road surface. The model was constrained at the road surface end of the MPC's.

The static analysis made use of a uniformly distributed pressure to simulate a 1 tonne load carried in the swap-body. The initial design iteration analyses had an assumed 2g vertical acceleration applied to the vehicle. The dynamic analyses required a different approach to apply the load. The mass of the swap-body floor elements were increased with a higher density to achieve the correct loading on the sub-frame and chassis. The purpose of the dynamic analysis was to obtain a design criteria for the sub-frame rubbers, so that the swap-body and sub-frame were isolated from the deformations that the chassis experienced. The suspension track data (at a speed of $20km/h$) displayed adequate vertical and torsional displacement on the chassis, and was therefore selected for the dynamic analysis. Due to the small effect of the lateral forces in regard to fatigue damage through the isolation of the sub-frame/swap-body assembly, the lateral loads were omitted from the analysis. The dynamic analysis therefore only used the longitudinal and vertical data to calculate the inputs to the vehicle.

The finite element model was analysed using the PERMAS7 solver for static analysis and PERMAS4 for the dynamic analysis. The *MSC.Patran* pre- and post-processing software package was used to process the results of the two analysis.

7.4 Suspension bracket of a large passenger bus

Geometry and Finite Element Modelling

The geometry of the suspension bracket was based on the supplied manufacturer's drawing. A bracket specimen was also supplied. The geometry as well as the mesh was created on the *MSC.Patran* FE package. The finite element analysis was done with the PERMAS7 package. The mesh consisted of QUAD4 and TRIA3 lower-order elements. The mesh was created on the centre plane positions of the various plates. Refer to figure 8.5, page 101.

Materials and element properties

The steel bracket was characterised by a Young's modulus of 200 GPa and Poisson ratio of 0.3. The properties were assigned to the mid-plane of the elements. The thickness of the suspension bracket plates were provided by the manufacturing drawings.

Load and boundary conditions

The suspension bracket was suppressed (using MPC's) in all the directions (translational and rotational) at the Huck bolt positions that connect the bracket to the chassis. Two point loads were applied, through MPC's, on the web where the shock absorber is connected to the bracket. A load of $1kN$ was applied on *each* web ($2kN$ in total), in the vertical direction, parallel to the angled plate.

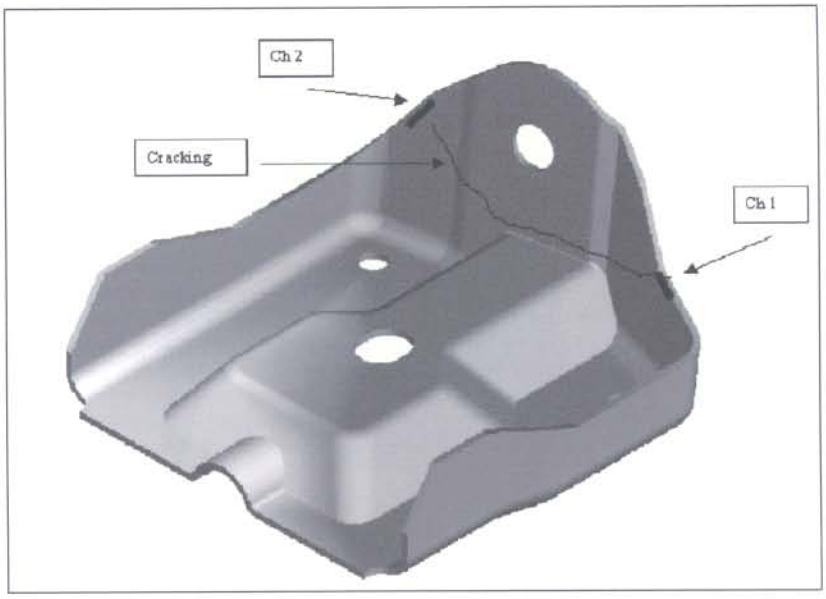


Figure 7.7: Unigraphics solid model - 4x4 pick-up bracket

7.5 Suspension bracket of a 4x4 pick-up truck

Geometry and Finite Element Modelling

The geometry was measured from a physical bracket, and modelled as a solid body in Unigraphics (a CAD package) (refer to figure 7.7). The finite element package *MSC.Patran* was used to create a model of the bracket. Shell elements were used for the model. Refer to the figure 7.8.

Materials and element properties

The steel bracket was characterised by a Young's modulus of 210 GPa and Poisson ratio of 0.3. The properties were assigned to the mid-plane of the elements. The thickness of the component was obtained from the actual component.

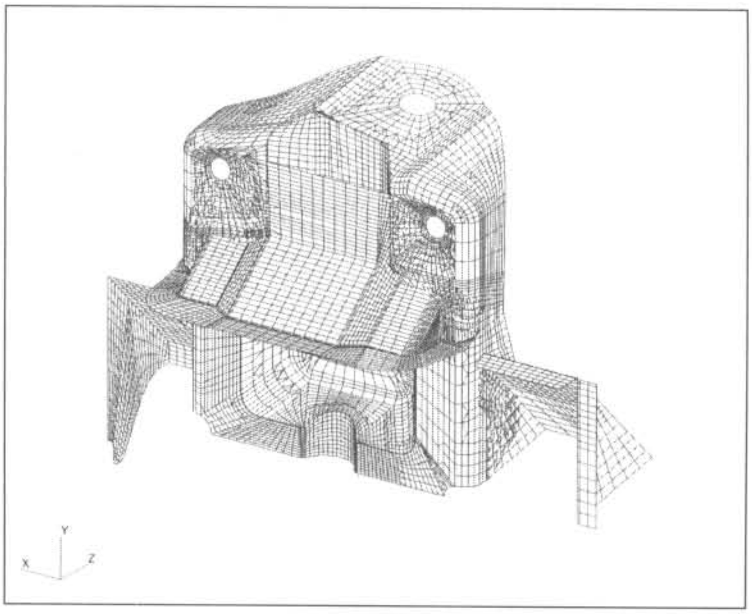


Figure 7.8: Finite element model: bracket of a 4x4 pick-up

Loads and Boundary Conditions

The model was restrained at the edges welded to the chassis beam. A pressure loading perpendicular to the bracket face, was applied to the ring of elements around the mounting hole to simulate the shock absorber force. The total force applied was 212 N. The finite element analysis package, PERMAS was used to analyze the model, performing a linear static analysis. The results of the PERMAS run were evaluated on *MSC.Patran*. Two iterations of design modifications were performed. Firstly, a gusset welded to the mounting face and the back face of the bracket was modelled and assessed. The second iteration involved a U-gusset, welded to the back faces, but only doubling up onto the mounting face, without welding.

7.6 Closure

The finite element models of the various case-studies were discussed in this chapter. The finite element analyses enabled the author to calculate the Fatigue Equivalent Static load. Equally importantly, the finite element tool enables the engineer to use the results of the fatigue analysis for assessment and design purposes.

The following chapter will discuss the results of the assessments of each case-study.