

University of Wales Trinity Saint David

Wales Institute of Science & Art



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

***Fatigue Analysis of a Connecting Rod using
ANSYS Mechanical***

Structural Component Analysis (ACHA7030)

Iván Moreno Fernández

2504057

Date (12/2025)

Contents

1. Methodology & Workflow.....	2
1.1 Engineering data	3
1.2 Rigid Dynamics	3
1.3 Steady-State Thermal.....	5
1.4 Static Structural	6
2. Boundary Conditions & Load Cases	6
3. Fatigue Life & Safety Factor Comparisons	9
4. Discussions & Critical Evaluation	14
5. References.....	15

1. Methodology & Workflow

The Fatigue Analysis of the connecting rod from a Ford Sigma engine has been developed with ANSYS software with the combination of different modules and system to obtain results and tendencies that can explain the behaviour of the engine part.

First, the geometry imported in our workspace it is composed of the connecting rod, one piston, gudgeon pin of the piston and the Crankshaft. It will be explained later why import the whole geometry for the study it improves the accuracy of the simulation and results.

The structure from the workflow and every module it is present in advance:

- Engineering Data

The module to define the materials for each part of the geometry.

- Rigid Dynamics

The module to calculate the inertia forces that are produced in the connecting rod due to the other elements.

- Steady-State Thermal

The module to calculate the temperature profile and study the influence on the results.

- Static Structural

The module to calculate the maximum and minimum stress to configure the fatigue analysis.

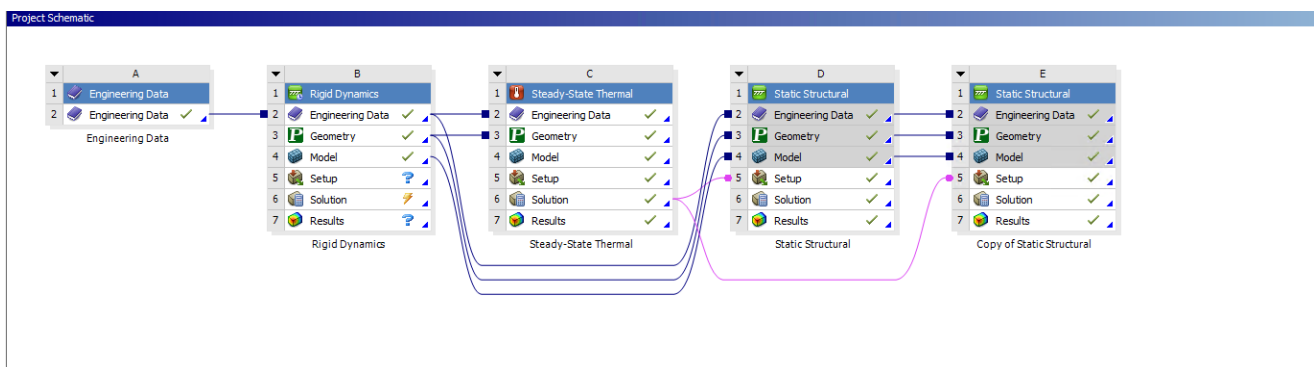


Figure 1.1 Workflow

1.1 Engineering data

In relation with this module, as it was mentioned before, here the materials for the study are defined. In this study it will be 3 different material, Aluminium 6061 T6, Low alloy steel AISI 4340 and structural steel.

The Aluminium 6061 will be use in the piston, the reason to use this material on the piston in because in the data from the specification of the Ford Sigma engine the pistons are made of Aluminium, head and block. This type of aluminium is usually use in automotive parts such as pistons due to its strength and light weight.

The AISI 4340 will be the material for the connecting rod, because after researching in different webpages for information and characteristics of the engine parts [2], this one is the material which connecting rods from Ford Sigma engines are made of.

The structural steel will be use for the crankshaft and the gudgeon pin. This material is the standard material in ANSYS and will be enough for the secondary part of our geometry that the aim of the analysis is not in them. These parts are usually made of some kind steel so for the simulation this simplification should not make a big difference.

1.2 Rigid Dynamics

The aim of rigid dynamics module is calculating the inertia forces generate because of the movement of the different components from the assembly. In this case it is necessary to define the different joints between each part, all of them revolute joint. In addition, with these joints, it must be defined the behaviour of the crankshaft and the piston with the “ground” (as it is defined by ANSYS the environment). This part will be critical because its determinate how correctly it is representing the movement from reality in the simulation.

For the crankshaft will be a revolute body-ground joint, as it represents the rotation of this part. In the case of the piston will be a body-ground joint that only let the translation in the vertical axis.

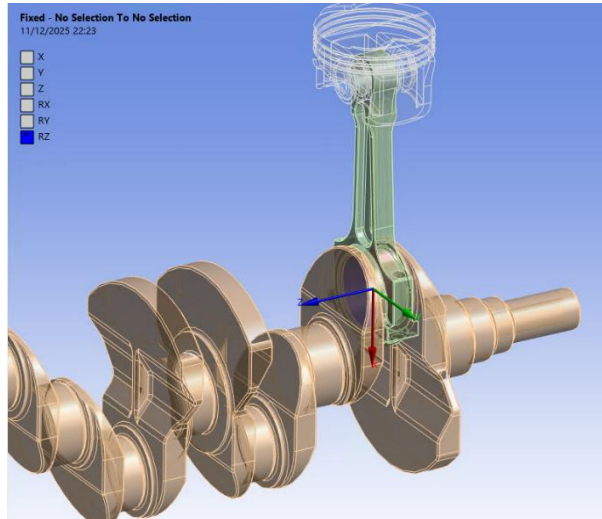


Figure 1.2 Rod and Crankshaft joint

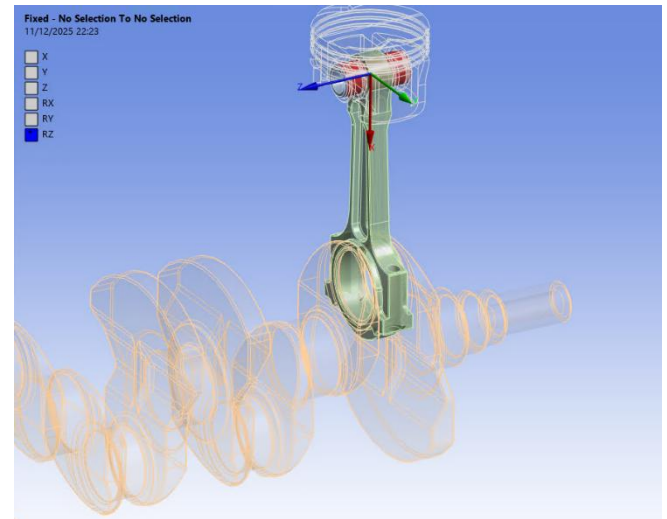


Figure 1.3 Rod and gudgeon pin joint

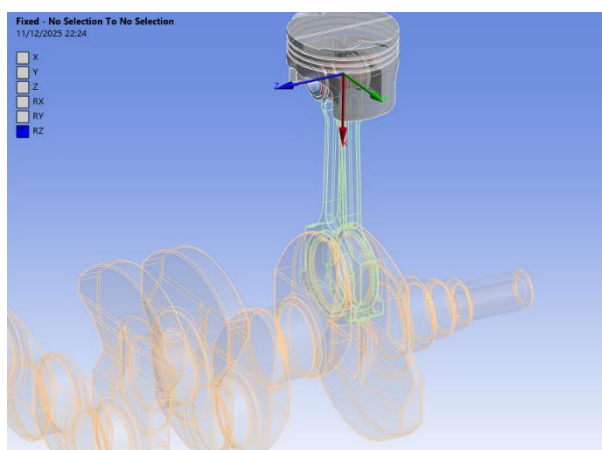


Figure 1.4 Piston and gudgeon pin joint

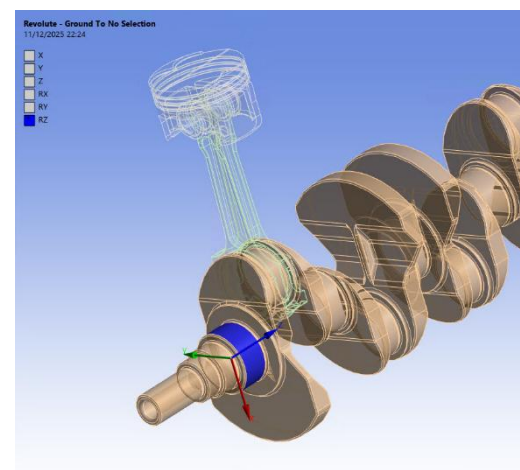


Figure 1.5 crankshaft- ground joint

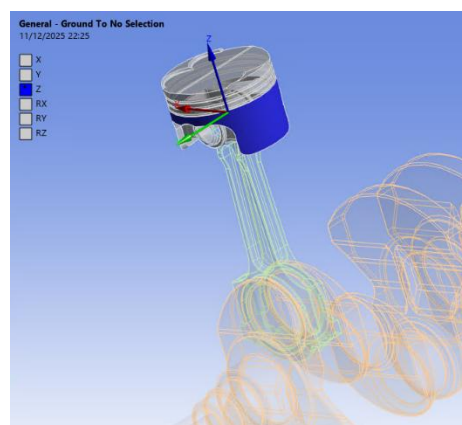


Figure 1.6 Piston-ground joint

With all this condition the real movement and behaviour of the geometry is well simulated. Defining the rotational velocity at 6500 rpm, it is possible to calculate the inertia forces produce in the connecting rod because of the crankshaft and piston. The reason to select this velocity is because that value is the highest of our engine which there is data, and the aim of the analysis is to evaluate the worst scenario for the Fatigue study, where the highest inertia forces are produced at the highest rpm. These inertia forces will be use as boundary condition in the static analysis.

1.3 Steady-State Thermal

In this module the aim will be to calculate the temperature distribution of the connecting rod. The reason for this data its because in some research [3], [4], [5] they include a Thermal study for the fatigue analysis while other mention that it affects in an important way to the results. This time, it will be included to get near the reality and contrast our research.

To build the temperature profile, it is defined with different convection condition throughout all the part, with higher initial temperatures near the small end and lower temperatures near the big end. In the following image it can be seen the profile.

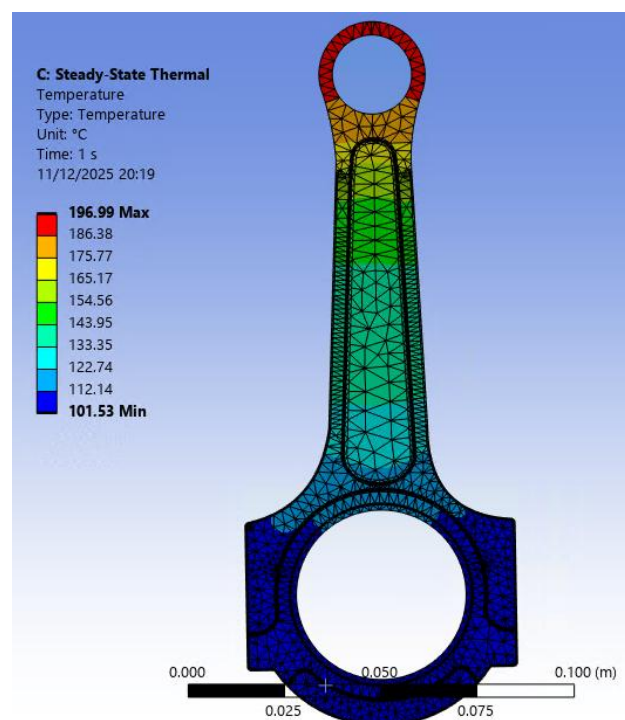


Figure 1.7 Temperature profile

1.4 Static Structural

This will be the final module of our workflow, where it will be defining our load case scenarios to build the fatigue module.

In this part of the workflow these scenarios are build up with the kinetic loads from the rigid dynamic module, the support and loads that will define the boundary conditions of the maximum and minimum cases. All these conditions will be explained in the part where it will be explaining their physics meaning and relation with the reality.

After calculating both load cases, with the solution combination, the two of them will be apply into the Fatigue tool of Ansys to determinate the values of life and safety factor.

For this fatigue analysis, it will be defined as non-proportional loading, based on the two scenarios calculated, with all the data of deformation and normal stress. In addition, the mean stress theory use will be Goodman, because it is the most common one for this type of analysis.

2. Boundary Conditions & Load Cases

For this fatigue analysis there will be define two load cases that are going to be explain why they are selected and their boundary conditions. As it was mentioned before for a fatigue study it is necessary to have two load cases, one with the maximum stresses and other one whit the minimum stresses that the part suffers.

The first scenario is the maximum tensile. This case is produced when the piston is near de TDC in the exhaust with the maximum tractions' stresses apply on the connecting rod. To be able to define these characteristics, the boundary conditions must represent the reality of the scenario. For this, the kinematic load generated in the rigid dynamics' module will define the forces apply on the connecting rod related to the movements of other components as it would be in the reality.

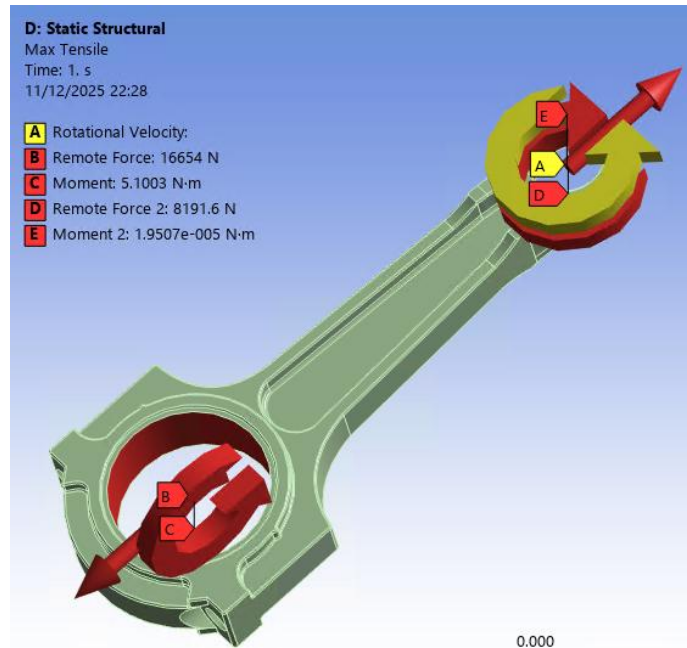


Figure 2.1 Kinetic loads maximum

In addition, with these one, it is necessary to consider the force due to the gas pressure leave it at the end of the exhaust. To calculate this one, use the top area of the piston (value of $4 \times 10^{-3} \text{ m}^2$) and divide the pressure by this. The pressure on this case will be 0.937 bar base on the datasheet provided, at 358° of crank angle.

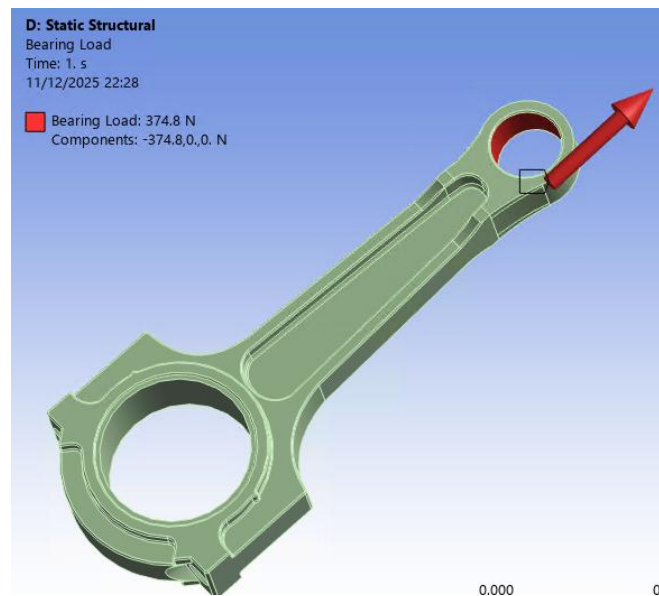


Figure 2.2 Pressure load at max tensile

Finally, it is necessary to simulate some kind of support for our part, so its represent that the connecting rod it is not floating in the environment. This will be resolved with a cylindrical support that only allows radial deformation, something near to reality than a fixed support.

The second scenario is the maximum compression, that will be the minimum value of stress. This case is produced at the TDC with the maximum pressure at the expansion stroke. As the first scenario, it is necessary to define the boundary conditions, for this case the kinematic loads will be use again but this time the minimum ones, because near the expansion stroke the inertia forces are lower due to the fact the piston is slowing before the ignition because of the gas pressure above.

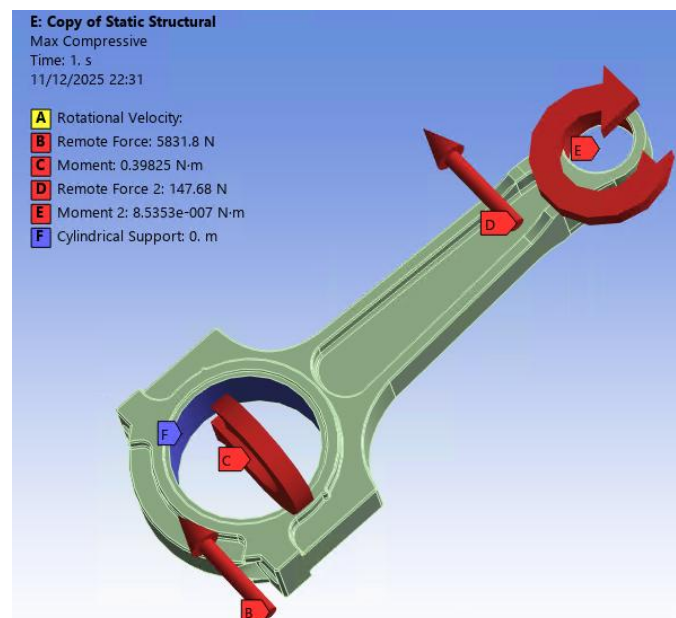


Figure 2.3 Kinetic loads minimum

The force due to the gas pressure in this case will be widely higher than in the previous one. To calculate this one, use the top area of the piston again (value of $4 \times 10^{-3} \text{ m}^2$) and divide the pressure by this. The pressure on this case will be 72.631 bar base on the datasheet provided, at 14° of crank angle.

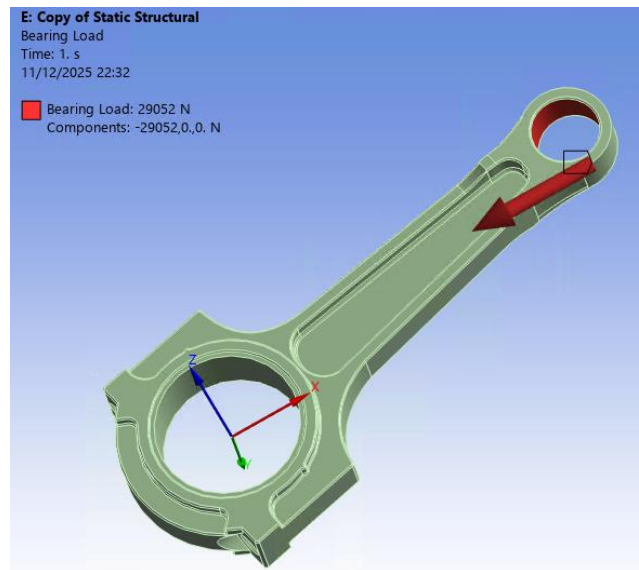


Figure 2.4 Pressure load at max compressive

As in the previous case, in this secondary scenario it is necessary a support, that will be the cylindrical support as before.

In relation with the boundary condition due to the thermal study, these loads are import on both scenarios as thermal loads.

3. Fatigue Life & Safety Factor Comparisons

In this part, the results will be presented. These results will be the total deformation and normal stress for each scenario and the values of life and safety factor from the fatigue analysis. At the end, it will be displaying the comparisons between the results with thermal load and without them.

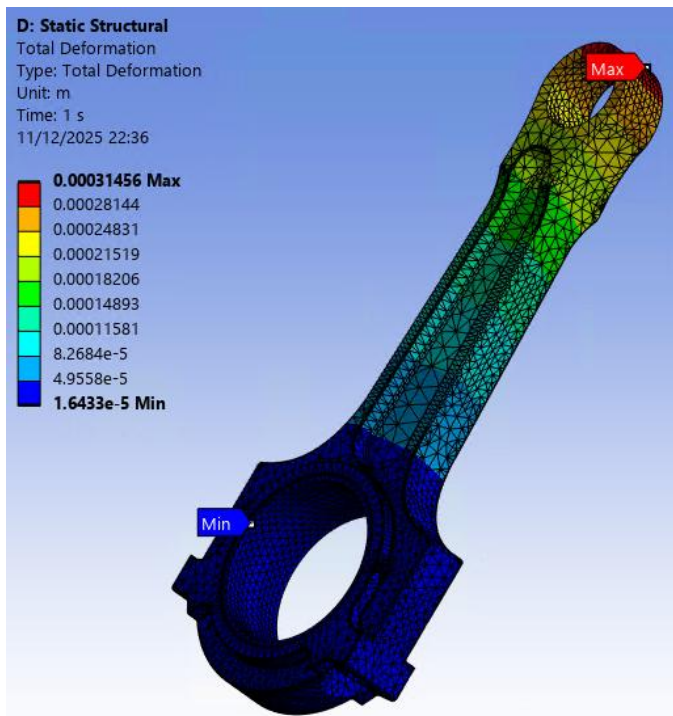


Figure 3.1 Deformation at max tensile

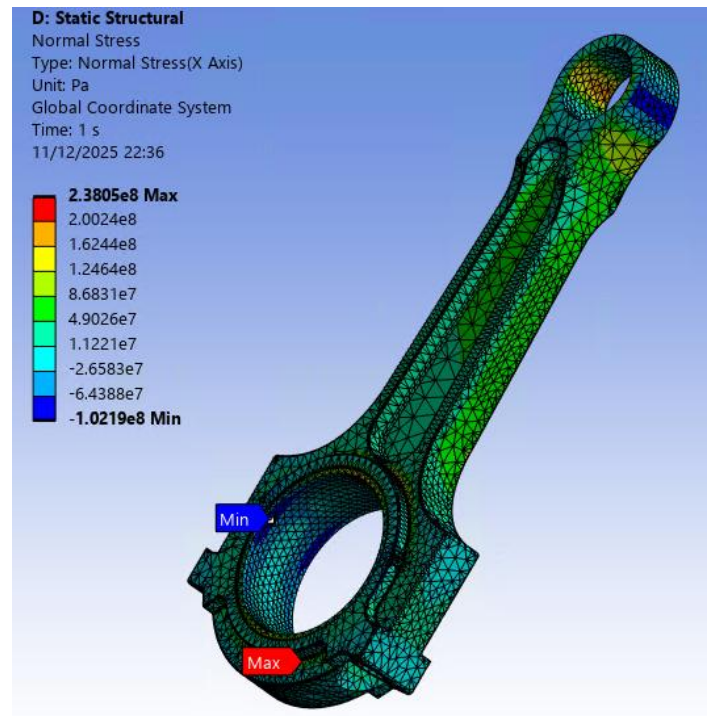


Figure 3.2 Normal stress at max tensile

As it can be seen in the above images, in the maximum tensile scenario the majority areas of the connecting rod are under tensile stress as it should be in the reality when the connecting rod is trying to slow down the piston and the reaction due to crankshaft retrain the connecting rod.

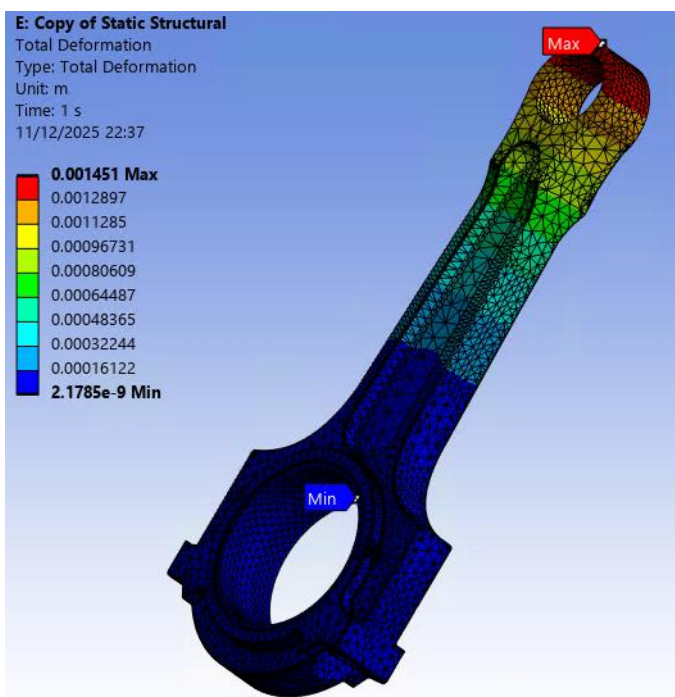


Figure 3.3 Deformation at max compressive

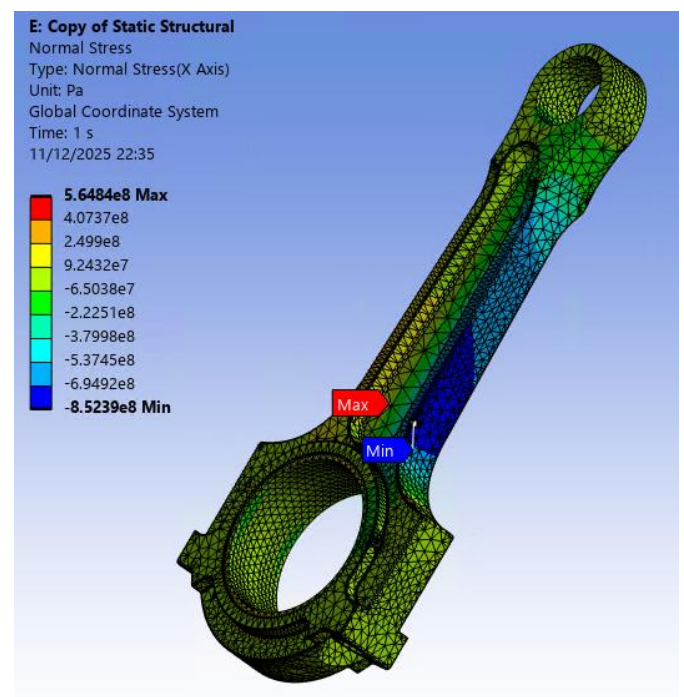


Figure 3.4 Deformation at max compressive

In images related to the maximum compression, one side of the beam is under compressive stress and the other under tensile stress. This happens because of the position of the connecting rod due to the crank angle. The force, as consequence of the gas pressure before the expansion stroke, its applied with a 4° deviation. These results seem nearly to how it should behave in reality the connecting rod in those scenarios.

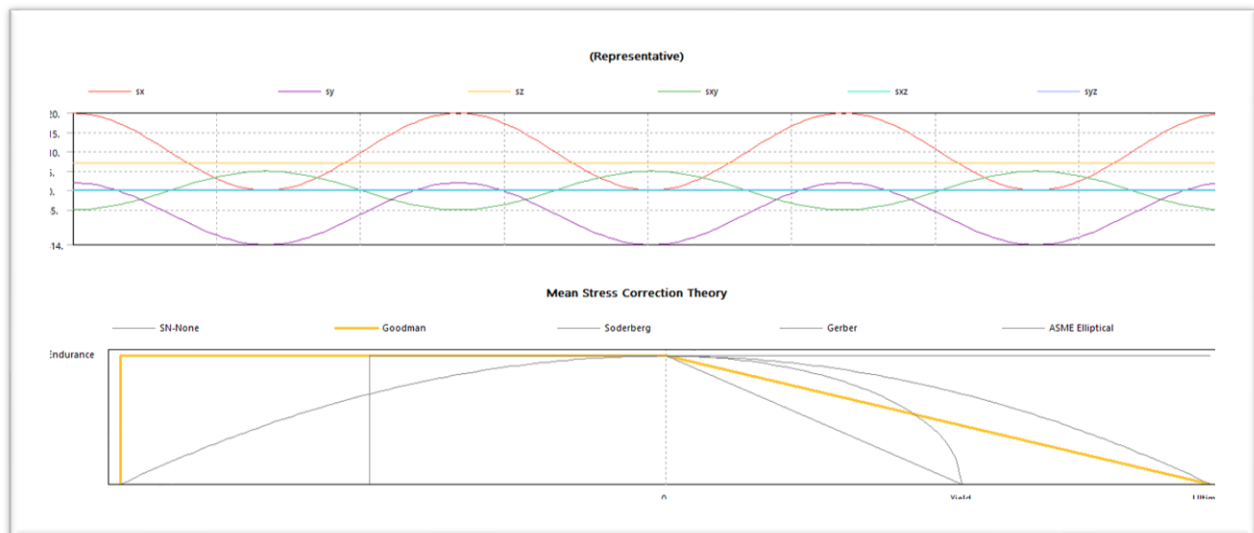


Figure 3.5 Fatigue tool

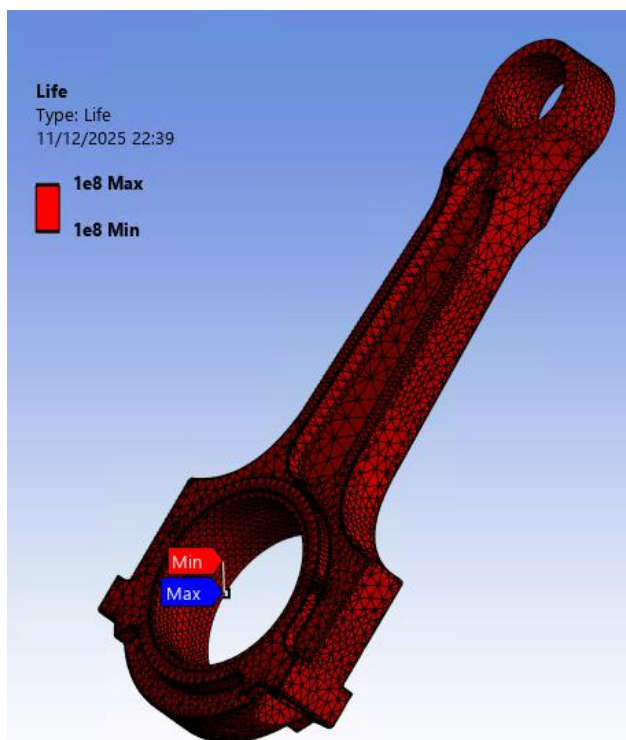


Figure 3.6 Life

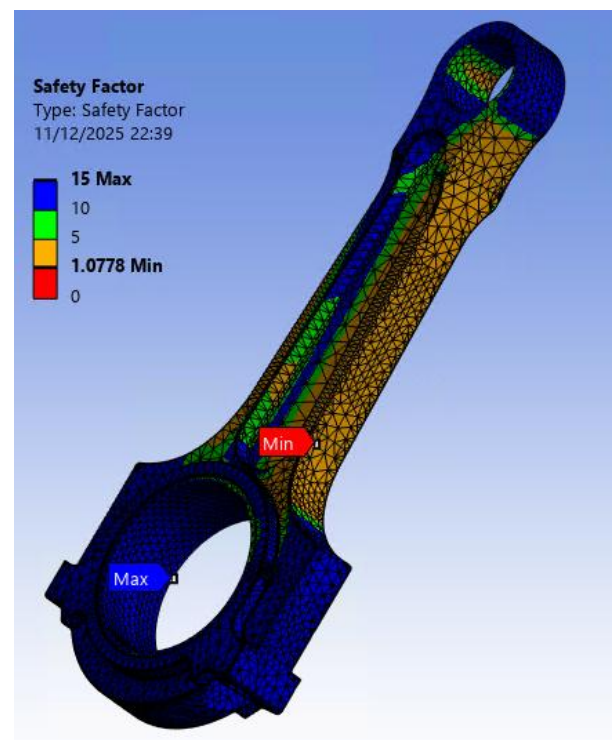


Figure 3.7 Safety Factor

As it can be seen in the Fatigue results, the connecting rod is expected to have an infinite life across the section. In relation to the safety factor, all the areas are above the value of 1, what indicates that the connecting rod is secure.

If these values are compared to an analytical value obtained with the same maximum and minimum strength it should be similar. The analytical value calculation will be:

$$\sigma_{\min} = -852.39 \text{ MPa and } \sigma_{\max} = 564.84 \text{ MPa}$$

$$\text{therefore } \sigma_{\text{mean}} = (\sigma_{\min} + \sigma_{\max}) / 2 = -143.775 \text{ MPa}$$

$$\text{And } \sigma_{\text{range}} = \sigma_{\max} - \sigma_{\min} = 1417.23 \text{ MPa}$$

$$\text{UTS} = 1462.3 \text{ MPa}$$

$$\sigma_{\text{limit}} = 0.5 * \text{UTC} = 731.15 \text{ MPa}$$

$$\sigma_{\text{allow}} = 2 * \sigma_{\text{limit}} * (1 - (\sigma_{\text{mean}} / \text{UTS})) = 1606.075 \text{ MPa}$$

$$\text{FSF} = \sigma_{\text{allow}} / \sigma_{\text{range}} = 1.1332$$

As it can be seen, our analytical value does not differ of the minimum value from our simulation.

However, it is important to consider that, the result for life seems to be a little strange. It can be possible that all the sections of the connecting rod are in infinite life status, but being the same exact number makes something can be wrong.

Apart from that, the safety factor is above one in every region, and it evolves in a tendency across the rod that makes sense. As the side with higher values of stress has the lower values.

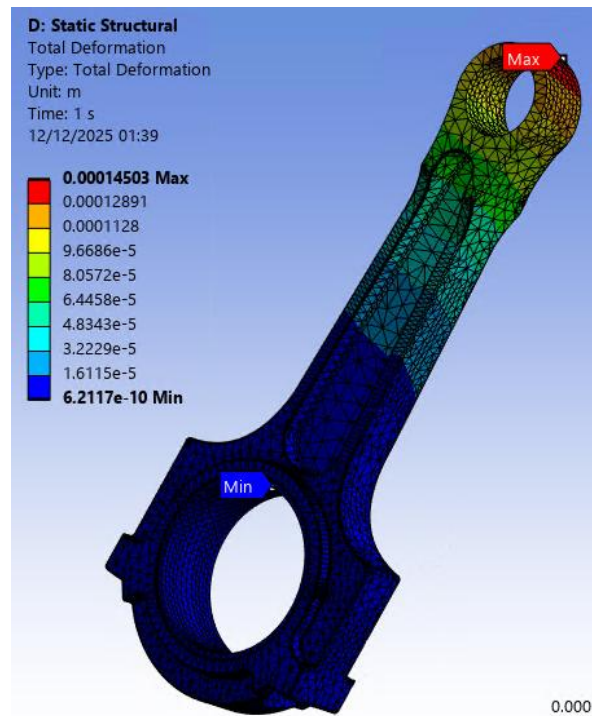


Figure 3.8 Total deformation without thermal load in Maximum Tensile

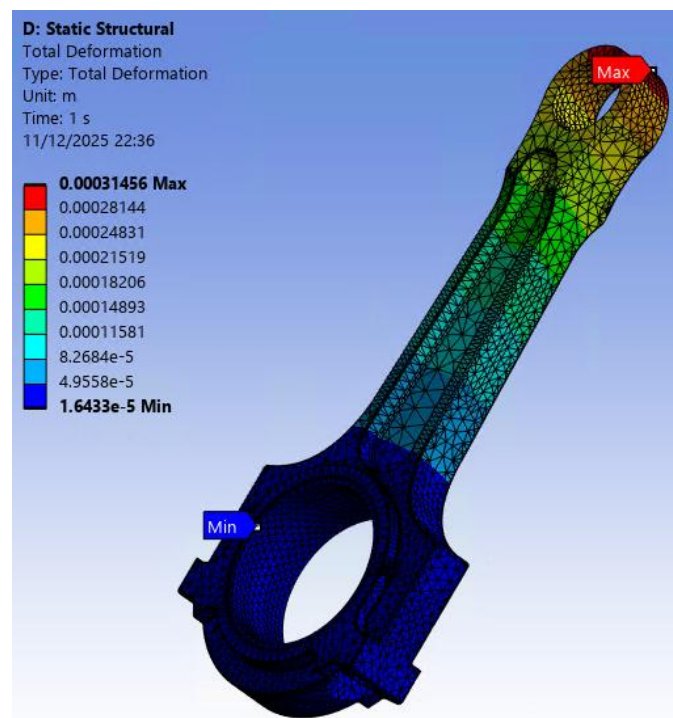


Figure 3.9 Total deformation with thermal load in Maximum Tensile

As it can be seen on the images above, because of the thermal load there is an important change in the deformation values. And we can assume this tendency can be applied to other values like the stresses.

This tendency is in accord with the different studies in the research that include the thermal analysis and included in the Fatigue study.

4. Discussions & Critical Evaluation

After developing all the fatigue analysis and obtain the results and compare them, it is possible to assume that the simulations of the FEA can give a good point of the tendencies and behaviours of the connecting rod during the different scenarios.

However, there are important limitations during the simulations and how to represent the reality. For instance, the cylindrical support on the big end to simulate that the rod is attached to something and cannot have any displacement, differ in an important way respect reality where due to the movements and inertias there can be some vibrations and stresses that alter the behaviour of the rod.

Other important simplification is how the kinetics loads are calculated. Normally these modules only can achieve some near value to reality due to assumption. In this case it would be interesting give to the crankshaft some kind of real material instead of the standard from ANSYS, this aspect for sure is going to affect to the inertia forces calculated.

It would be interesting to validate the refinement of the mesh, because this aspect can affect to the values and can be a really great point of improvement on the result.

In this case, like was said before, the tendencies seem pretty good and accurate, although the life results probably would need some revise before taking into consideration.

5. References

- [1] M. I. GmbH, *Cylinder Components: Properties, Applications, Materials*. Springer Vieweg. Springer Wiesb. GmbH, 2016.
- [2] RacingFuel, “Forged connecting rods Ford Fiesta ST-16 EcoBoost 182-200 hp K1 forged rods engine Sigma I4 HXJA HXJB JTJA,” RacingFuel.es. [Online]. Available: <https://racingfuel.es/forged-connecting-rods-ford-fieta-st-16-ecoboost-182-200cv-k1-forged-rods-engine-sigma-i4-hxja-hxjb-jtja>. [Accessed: Dec. 12, 2025].
- [3] JSME, “Temperature Measurements of the Connecting Rod, Piston Pin and Crankpin Bearing of an Automobile Gasoline Engine,” **JSME International Journal**, vol. 9, no. 33, pp. 181–189, 1966. [Online]. Available: https://www.jstage.jst.go.jp/article/jsme1958/9/33/9_33_181/_pdf. [Accessed: Dec. 12, 2025].
- [4] A. Mekonen, B. Yosiph, and J. B. Dawit, “Finite Element Analysis of Fatigue Life Prediction for Connecting Rod Using Aluminium and Steel Alloy Materials,” **International Journal of Innovative Science, Engineering & Technology (IJSET)**, vol. 6, no. 9, Sept. 2019. [Online]. Available: https://ijiset.com/vol6/v6s9/IJSET_V6_I9_35.pdf. [Accessed: Dec. 12, 2025].
- [5] R. Ranjan¹, Prof. D. S. Rawat and Dr. S. Rai, “Structural and Thermal Analysis of Piston, Connecting rod and Crank shaft assembly using Catia and Ansys Software,” **IJIRT**, no. 162161. [Online]. Available: https://ijirt.org/publishedpaper/IJIRT162161_PAPER.pdf. [Accessed: Dec. 12, 2025].