Mechanical Transmission Technology

Finite Element Analysis Report

Otter Quarks, Ivan Tregear, Mahesh Mottram, Doruk Dolen

Department of Mechanical Engineering,

Imperial College London

Date: 19/01/2023

Word Count: 6105

1. Abstract

With the high number of parameters involved in gear design, and the important consequences of transmission failure, it is crucial to have a good understanding of the effect that modifying a design parameter has on the stresses experienced by gears. Using Abaqus, an FEA study was conducted to evaluate the effect on stresses of changing gear parameters. A model of two spur gears meshing was first set up, on which a convergence study was performed to determine the mesh size at which the results become mesh independent, finding a mesh size of 0.03 mm to be satisfactory. A parametric study was carried out on four gear parameters: centre distance, material elastic modulus, number of teeth on the pinion, and tooth root fillet radius, and the effects of changing these parameters on root bending stress and contact pressure was studied. Contact pressure was found to be roughly invariant of centre distance and root fillet radius, while showing a logarithmic relationship for number of pinion teeth and elastic modulus. Root bending stress was found to increase with increasing centre distance and elastic modulus and decrease with increasing number of pinion teeth and root fillet radius. From these results, recommendations for transmission designers were made with respect to improving the safety of transmissions.

Contents

Transmission designers have a large degree of freedom when designing a gearbox, even under very well-defined requirements. This is in part due to the significant number of parameters that define a pair of gears, such as ratio, extension parameters, material, centre distance, number of teeth, thickness, etc… Without a detailed understanding of the effect of varying these parameters, it may be difficult to make an informed choice. On the other hand, the consequences of gearbox failure are so severe that transmission designers cannot afford to decide on these parameters arbitrarily.

This study aims to provide insight into the effect of variation of design parameters on the resultant stresses of a pair of meshing gears. To do so, a Finite Element Analysis (FEA) study is proposed, simulating the gears over a set time during which they will rotate throughout a meshing cycle. The FEA package used is Abaqus 2021, ran on an Academic license.

Gears have highly complex geometries and modelling the stresses of two gears through a meshing cycle rapidly becomes analytically difficult. Based on some assumptions, simple analytical results can be used to obtain an order of magnitude estimate, using for example the Lewis equation, however these solutions remain far from describing the full stress distribution. Evaluating the distribution through physical tests would be ideal, but would be extremely expensive and complex to implement, and would only produce results specific to the gears used. Numerical methods provide a good compromise, producing comprehensive results well adapted to parametric studies, while providing a decent approximation of reality.

FEA, as a computational method, does remain limited in how well it can describe real situations. A good and informed model setup is necessary to produce valuable data, including accurate boundary conditions, loading, supports, and a good meshing strategy. As such, the model setup and methodology will first be described, followed by an explanation of the meshing strategy. A convergence study will be run on the mesh size to ensure that the results of the following parametric study are independent of the mesh size. Finally, the parametric study itself will be carried out, varying four gear design parameters, and observing the effect on the resulting gear stresses.

3. Methodology

3.1. Model Setup

A set of spur gears was generated following the standard BS 436 basic rack, and standard gear geometry equations. The base set of gears had a module of 4.5, a wheel with 55 teeth and an addendum modification coefficient of 0.5, and a pinion with 35 teeth and an addendum modification coefficient of 0.1. From these parameters, the involute tooth profiles were generated for both wheel and pinion. Using the flank data, a set of 5 teeth was generated for both the pinion and wheel. This was estimated as the lowest number of teeth required such that the stresses far from the edge of the model are similar to those that would be observed for a full gear.

Having generated the outer profile of the gears, an inner rim was added to close the geometry. An inner web was also modelled, bounded at its outer diameter to the rim. Finally, a shaft was modelled as an infinitely thin, rigid surface, at the inner diameter of the web. The diameters were chosen to be far enough to not impact the accuracy of the stress in the area of interest. Thicknesses of 20 mm and 30 mm were attributed to the web and teeth respectively, thus completing the geometric setup of the model.

3.2. Boundary conditions

Despite having an assigned thickness, the model was conserved as 2D to significantly reduce the computational time required. In order to capture the 3D nature of the gears, the assumption of plane strain was made – since the teeth are significantly thicker out of plane than the area of contact in plane, out of plane strains may be assumed to be negligible. This allows 3D behaviour to be modelled in 2D with little loss of accuracy.

A number of boundary conditions were then applied to the model, to accurately represent the meshing cycle of the gears. Tie conditions were applied between the shaft, web and rim of each gear in order to connect the different components. The displacement of the shafts was fixed to avoid translation of the gears when under load. A torque loading condition was then applied, ramping from 0 to full torque of 2500 Nm over a duration of one second, during which the gears do not rotate. This progressive loading aids initial convergence of the model. Following this, the zero rotation of the gears is replaced with a continuous rotation of the wheel of 1 °/s over the remainder of the simulation, lasting for a total of 16 seconds.

3.3. Data extraction

Following each run, the Abaqus script then automatically generated output files. The output files, in CSV format, each contained a stress distribution over a certain number of nodes, for each time step. Four such files were generated for each the wheel and pinion, corresponding to contact pressure and maximum principal stress along the flank, as well as von Mises stress and maximum principal stress along the root fillet. The results were extracted only for the third tooth, to ensure a maximum distance between then area of interest and the less accurate edge of the model.

This entire process was repeated for each run presented in this study, although a large part of it was automated through the Abaqus script used to generate the model, and a Python script used to generate the gear geometries based on set parameters (geometries that were then fed as inputs to the Abaqus script).

4. Meshing Strategy

4.1. Model setup

Finite Element simulations rely on the discretisation of the modelled domain, dividing each component into small elements, and solving equations over the boundaries of each element. Finer meshes produce more accurate results, but generate a vast number of elements, and therefore a large amount of computational power is required to solve them. In order to achieve a balance between computational efficiency and accuracy, different mesh sizes were implemented at different locations in the model. Due to the complex and nonlinear stress distributions arising from the contact of teeth, a fine mesh was implemented along the flanks of the three central teeth, such that a majority of any stress distribution of interest could be captured within the fine mesh. A coarser mesh was implemented on the two outer teeth. Finally, an independent and constant coarse mesh size was implemented on the web of the gear.

To avoid large discontinuities within the mesh size, and to ensure that the mesh remained sufficiently fine around the areas of interest, the interfaces between the areas of coarse and fine mesh were seeded to produce a smooth transition gradient in element size. To avoid the gradient being too large, a fixed ratio between the coarse element size and the fine element size was used. Initially this was set to 5, which for relatively coarse meshes, appeared to function well. However, the Abaqus Academic license only allowed a maximum number of 250,000 nodes, which was rapidly exceeded for finer meshes. To overcome this, a larger mesh ratio of 20 was used.

A quadrilateral dominated mesh was used, employing a majority of quadrilateral elements with a small number of triangle elements. The inclusion of triangle elements allows the mesh to conform better to imposed geometries without resulting in elements with large angles or high aspect ratios. The resultant mesh is shown in Figure 1, where a fine element size of 0.03 mm has been used with a coarse element size of 0.6 mm. The web element size is 2 mm and was not varied throughout the study.

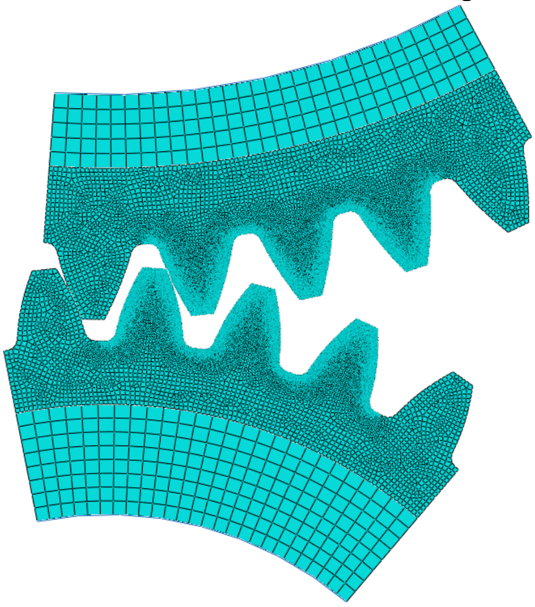

Figure 1: Mesh of model, fine element size of 0.03mm.

4.2. Convergence study

Before the parametric study could be conducted, a convergence study was proposed to ensure that the FE mesh was sufficiently refined. The purpose of this study was to determine the element size at which the results became independent of the mesh. Two stresses were examined to determine this: the tooth flank contact pressure, and the root fillet von Mises bending stress. The level of mesh refinement was then varied, keeping the constant ratio of coarse to fine mesh size of 20, with the aim of finding an element size at which the stress distributions no longer varied. The fine element size was varied between 1 and 0.022 mm, beyond which the maximum number of nodes was exceeded.

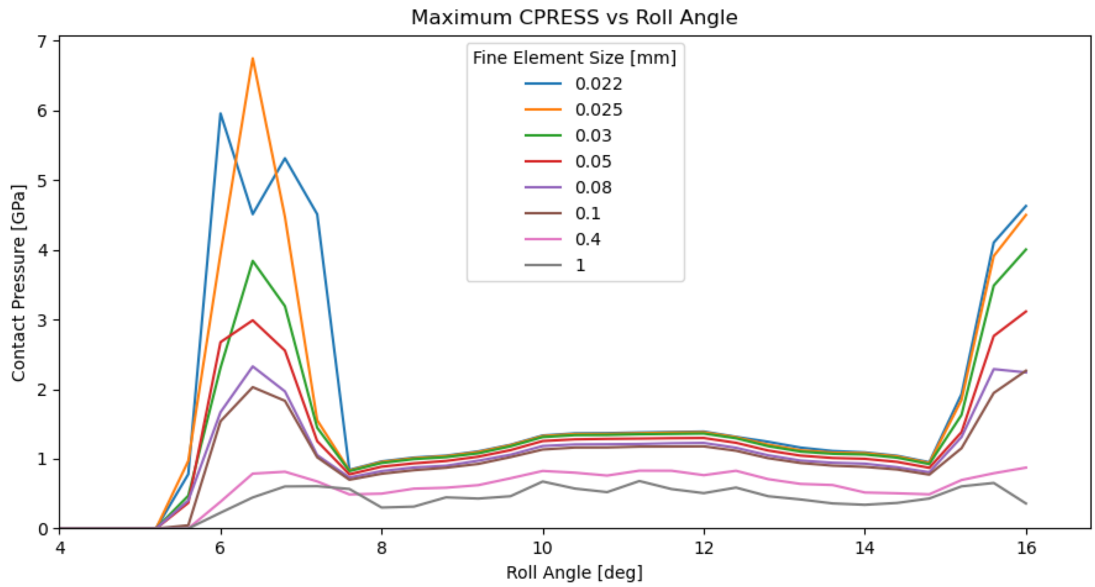

Figure 2: Maximum contact pressure along the flank against roll angle, for different fine element sizes.

Figure 2 shows the initially collected data. To reduce the number of dimensions, only the maximum contact pressure along the flank was recorded for each time step, which is displayed above. While the maximum contact pressure appears continuous for much of the meshing cycle, the results display a significant initial peak. The tips of the gears were modelled as having perfectly sharp corners, as the mirrored tooth flanks were simply joined by a straight line. At the beginning and end of each meshing cycle, the single node at the tip of a tooth therefore contacts the flank of another, causing a very large stress concentration. This was circumvented by evaluating the maximum contact pressure in the area away from the peak, between 9 and 13°.

The result for maximum contact pressure between 9 and 13° is shown in Figure 3, in which a logarithmic scale for the fine mesh size was used, as the absolute difference between mesh sizes is less illustrative than the relative sizes. The plot displays the contact pressure starting to plateau at the lowest mesh size, although the results do not seem to have perfectly converged. The mesh size however could not be reduced further due to the limitation of the Academic License preventing further refinement.

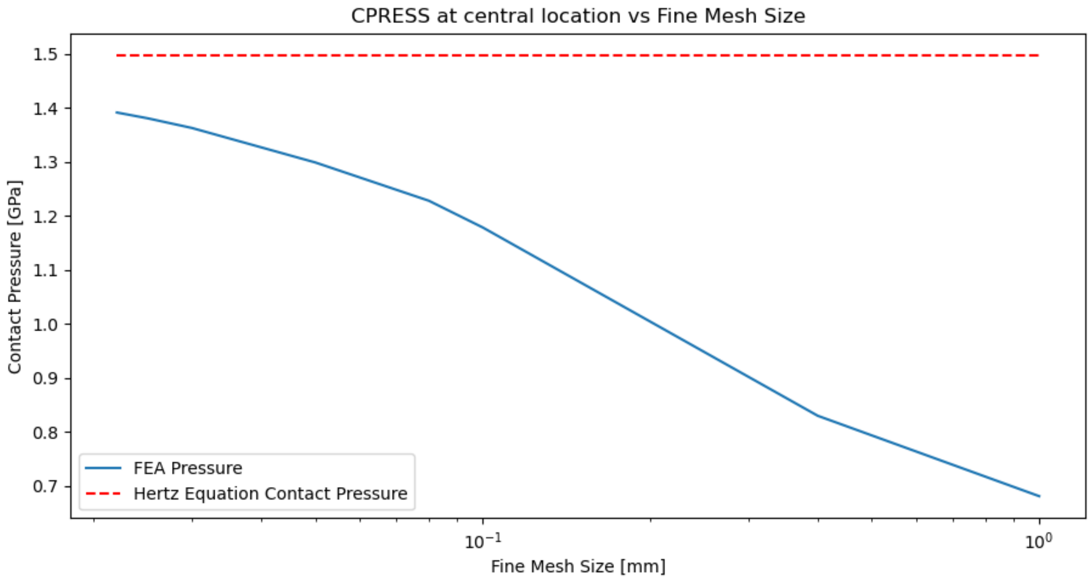

Figure 3: Maximum contact pressure over specified flank locations versus fine mesh size.

A theoretical value for the contact pressure as predicted by the Hertz Pressure equation is also included, as predicted by Equations 1, 2, and 3.

$$R_i = \frac{d_{bi}}{2}\tan(\alpha_{tw}) \tag{1}$$

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} \tag{2}$$

$$P_0 = \sqrt{\frac{PE^*}{\pi R}} \tag{3}$$

The contact pressure does not appear to converge to the predicted Hertz pressure, though this may be due to the centre distance used. The Hertz equation doesn't take the centre distance into account and assumes a tight mesh. While the tight mesh was calculated to be 205 mm, the study was conducted at a centre distance of 207 mm. As will be later discussed in Section 5.3, an increased centre distance results in a lowered contact pressure, thus explaining the discrepancy.

A similar process was also conducted for the root fillet stress. Once again the maximum root fillet stress was taken along the fillet edge for each time step, displayed in Figure 4, and the maximum value of this was taken across the range of roll angles, resulting in the second convergence plot shown in Figure 5. For the convergence plot, the overall maximum stress was taken. This was to allow for a comparison to the maximum analytical root stress predicted by the Lewis Equation. This is highlighted in Figure 5, with the Lewis stress shown in red. Note that the Lewis equation predicts the stress by modelling the tooth as a cantilever with tip loading. In reality the maximum loading occurs closer to the centre of the tooth, reducing the moment arm, hence the lower obtained stresses, even when approaching convergence.

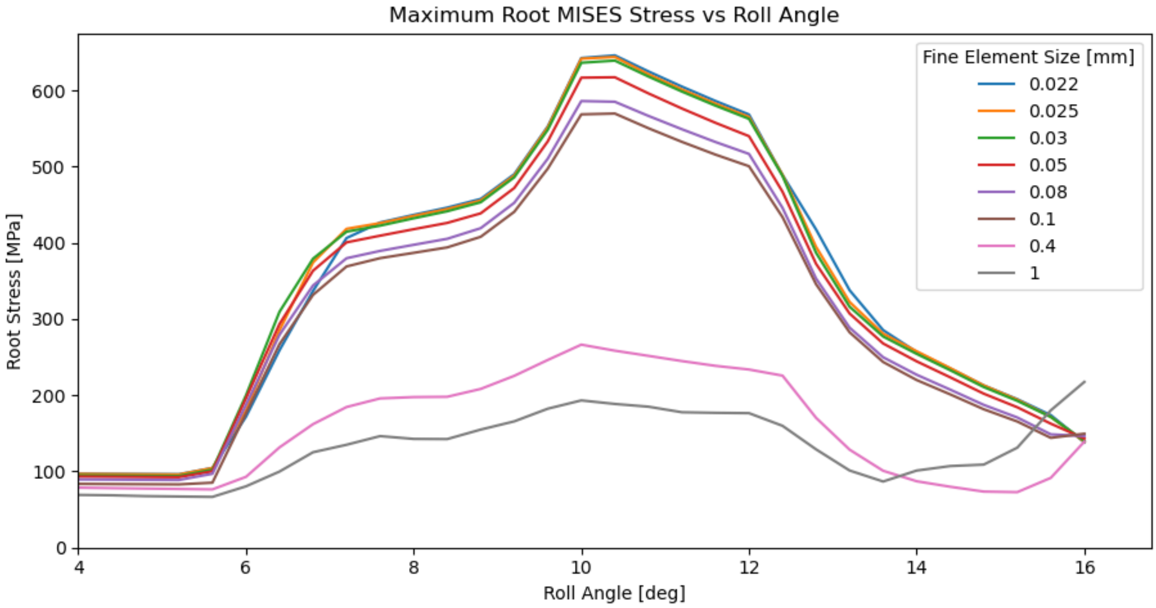

Figure 4: Maximum root fillet stress against roll angle, for varied fine element size.

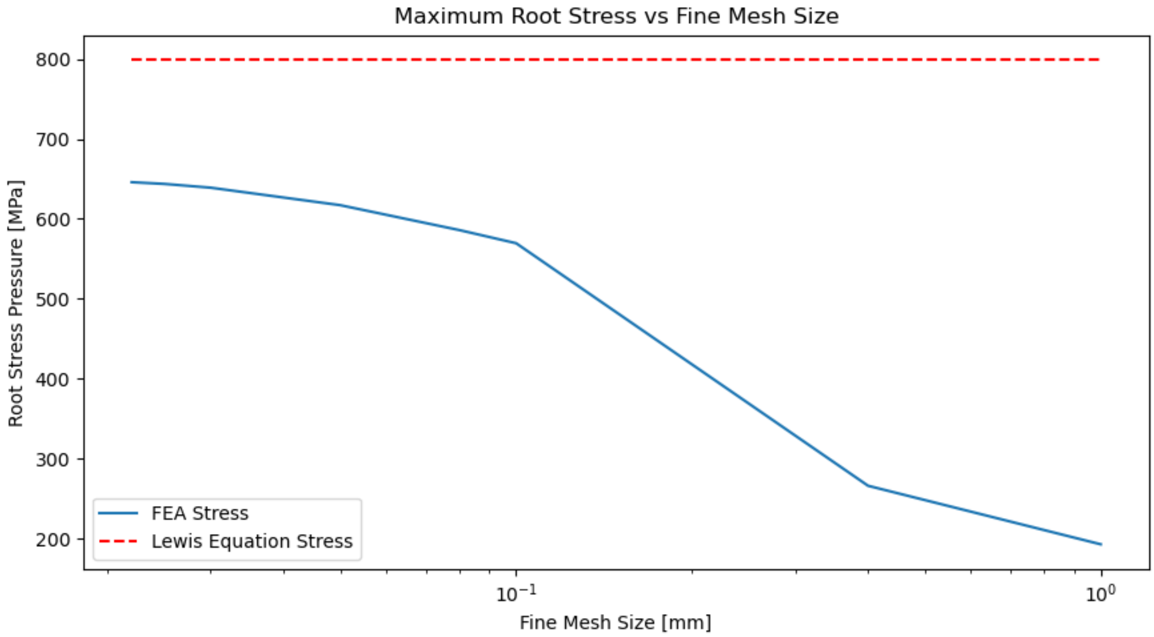

Figure 5: Maximum root fillet stress across temporal range, compared to the analytical result from the Lewis Equation.

The results shown above display a slightly higher degree of convergence when approaching the finer element sizes than observed for contact pressure. While full convergence was not reached, a fine mesh size of 0.03 mm and a coarse mesh size of 0.6 mm were chosen for the parametric study. The lowest possible element size (0.022 mm) was not chosen as it would not have allowed for any increase in gear size and corresponding number of nodes, e.g. increasing the number of teeth on the pinion. It was deemed that the results were close enough to convergence that the effect of mesh size would not vary while varying a given parameter. Therefore, while the absolute values found in the parametric study may be slightly under real stress values, the trends and relative values remain sufficiently accurate to depict the real behaviour of the gears with respect to parameter variation.

5. Results

Once a convergent mesh had been established, four parameters were chosen for the parametric study: centre distance, elastic modulus, number of pinion teeth and root fillet radius. For a fixed mesh size, these were varied, and their effect on contact pressure and root fillet bending stress was evaluated. The initial control model was taken with an elastic modulus of 207 GPa, a root fillet radius of 1.5 mm, 35 teeth on the pinion and a centre distance of 207 mm. It should be noted that the convergence study was initially run at tight mesh, with 205 mm centre distance, leading to no issues. However, at tight mesh, there was a large average number of teeth in contact at any given time, leading to incoherent and unpredictable deformations. This was remedied by rerunning both the convergence and parametric studies with a new centre distance of 207 mm. This is discussed further in the Section 6.1.3. The meshing achieved at this distance is shown in Figure 6 demonstrating single pair tooth contact.

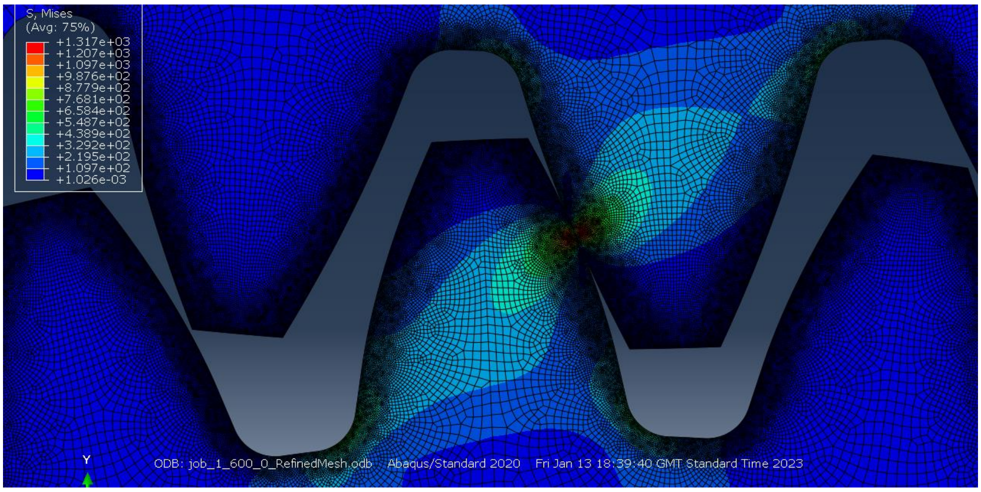

Figure 6: Stresses during contact, showing increased separation between gears and single tooth pair contact.

5.1. Elastic Modulus

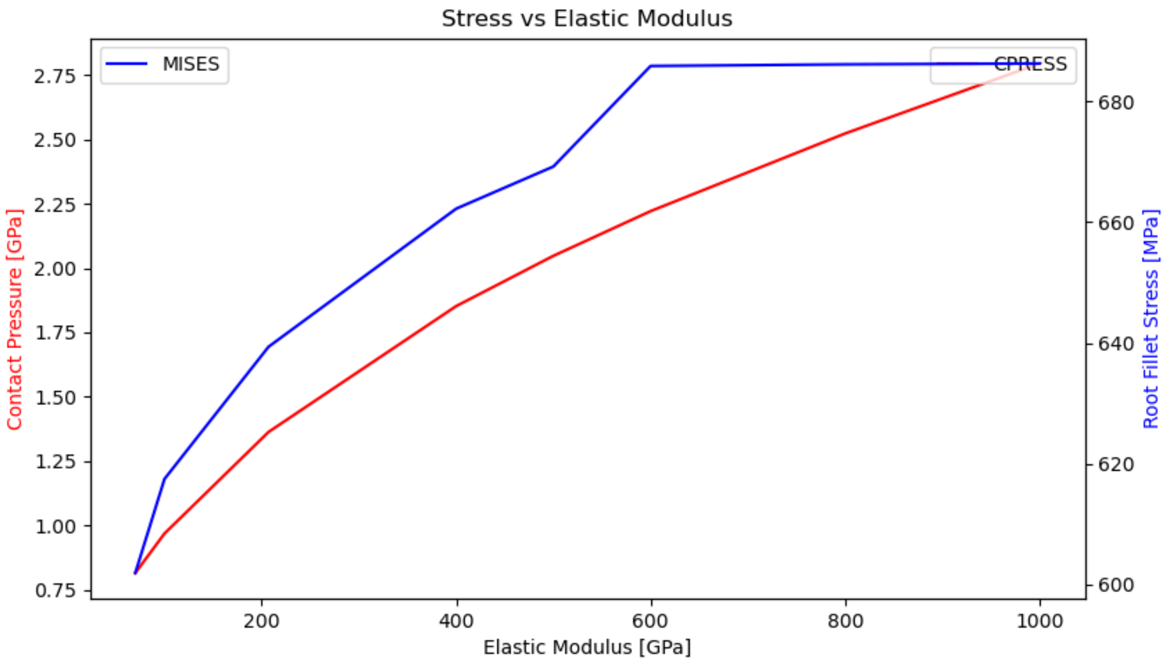

Figure 7: Contact pressure and root fillet stress versus elastic modulus.

The results for varying elastic modulus and the effect on contact pressure and root fillet stress are shown in Figure 7. It should be noted for this figure, and all subsequent figures that both root fillet bending stress and contact pressure are plotted on the same graph, although with different y-axes for easy visualisation – while the curves may appear similar, their orders of magnitude are significantly different.

The initial elastic modulus used in the convergence study was 207 GPa, and this was varied between 60 GPa and 1000 GPa. Prior to the study, it was predicted that there may be some non-linear behaviour in the low region of elastic modulus, as for sufficiently small elastic modulus, the bending of the teeth increases until multiple teeth may come into contact, or the geometry is significantly deformed. This is not necessarily evident in Figure 7 and no runs were able to converge for values below 60 GPa. The failure to converge didn't occur at the initial condition, instead occurring several seconds into the run; this may suggest that at a certain meshing point the deformation was indeed significant, causing the simulation to fail, however this is difficult to confirm as data could not be recorded for low values of elastic modulus.

Both contact pressure and root fillet bending stress increased with increasing elastic modulus, showing a rough logarithmic relationship. While this remains true across the entire elastic modulus domain for contact pressure, the root fillet stress appears to reach a constant value, and potential explanations for this are discussed in Section 6.1.1.

5.2. Pinion Teeth Number

Next, while maintaining the size of the wheel, the number of teeth on the pinion was varied. With 35 teeth initially, the pinion was varied between having 19 and 100 teeth. At lower than 19 teeth no model was able to converge correctly, presumably due to the more complex gear geometry. At 56 teeth the pinion becomes larger than the wheel and therefore technically becomes the wheel. Although at this point it is no longer the "pinion" teeth that are varied, points were still taken up to 100 teeth on the bottom gear to confirm that the behaviour remained continuous, independent of the denomination of the gear. After 100 teeth the model required other parameters to be adapted, as the default root fillet radius (1.5 mm) became too large.

While varying the number of teeth, other minor parameters needed adapting to correctly scale the gears. While most parameters were generated automatically, the shaft diameter and web diameter were set manually. These were kept approximately at:

$$d_{web} = d_{dedendum} - 20 \tag{4}$$

$$d_{shaft} = d_{web} - 20 \tag{5}$$

At these values the shaft and web were found to be far enough away to avoid having any significant impact. The centre distance was automatically calculated at tight mesh, and a further 2 mm was added for each run to match the control run (with a centre distance of 207 mm rather than 205 mm). The δ_2 (initial gear rotation) parameter was slightly adapted manually to ensure that the teeth were initially in contact, allowing the models to converge.

The results for contact pressure and root bending stress against number of pinion teeth are shown below in Figure 8.

Figure 8: Contact pressure and root fillet stress versus number of pinion teeth.

Both stresses can be seen to decrease nonlinearly with increasing number of pinion teeth, ranging from 140% to 50% of the initial value for contact pressure, and 183% to 30% for root fillet stress. Both stresses follow a similar trend, and seem to converge slowly at higher teeth numbers.

5.3. Centre Distance

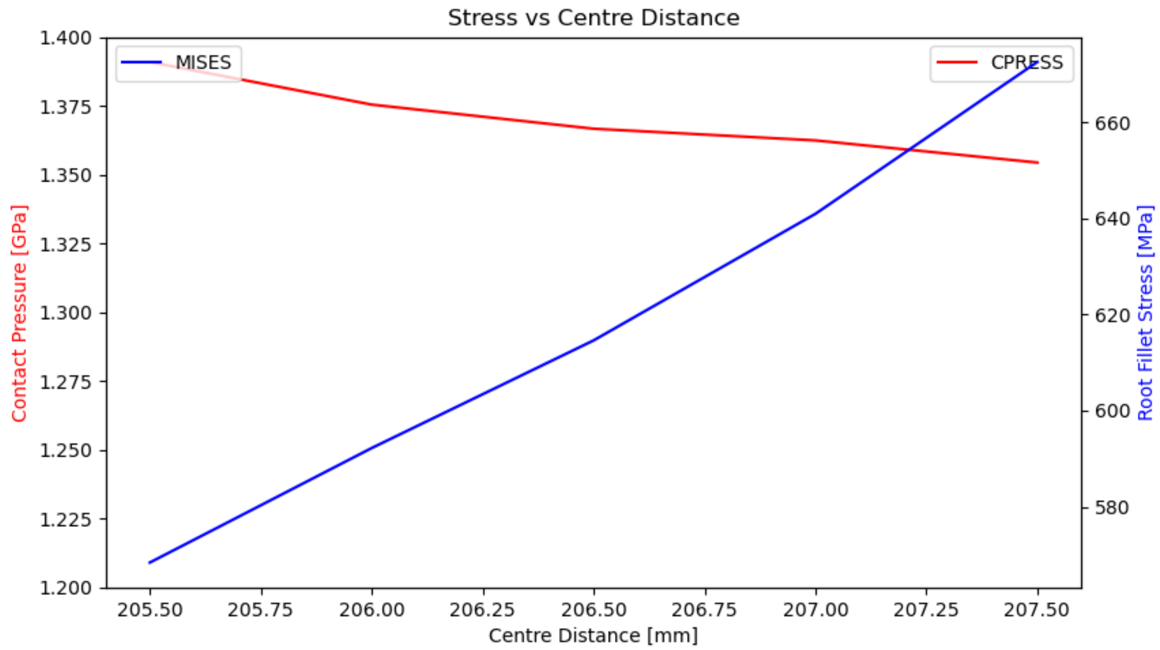

Figure 9: Contact pressure and root fillet stress versus centre distance.

The effects on the studied stresses due to varying centre distance are shown in Figure 9. Over the chosen range, the root fillet stress increased linearly by 18%, whilst the contact pressure decreased linearly by just 2%.

A range of 205.5 mm to 207.5 mm was selected to investigate the impact of centre distance. As discussed, the tight mesh centre distance, calculated using Equation 8 from the FEA task handout (MacLaren, 2022), was 205 mm for the gear geometry used. However, at this value, when the two gears were meshed, the approximation of the contact surface using the chosen elements was slightly larger than the input geometry, sufficiently so to bring multiple teeth into contact, and cause slight overlap. Effectively the teeth were now more tightly meshed than the minimum tight mesh point. As a result, non-linear behaviour and a spike in both contact pressure and root fillet stress around 205 mm was observed, with the linear trend seen in Figure 9 starting at approximately 205.5 mm. This was therefore taken as the "true" tight mesh centre distance of the model, and the parameter study was initiated at this value. This was seen as a limitation of the finite element model.

As previously mentioned, to ensure consistent results, upon discovery of this behaviour, a larger control centre distance of 207 mm was used in all other simulations in this report, in place of the calculated tight mesh. This was also used in the convergence study to verify the selection of mesh refinement.

5.4. Root Fillet Radius

Finally, the root fillet radius of the gears, initially at 1.5 mm, was varied to evaluate potential stress concentration effects. The radius was varied between 1.3 and 1.6 mm, and the results are shown in Figure 10. It is clear that there is little variation in contact pressure, which roughly remains constant across the change in fillet radius. As for root fillet stress, there is a relatively strong negative linear correlation, with root fillet stress decreasing by around 7% between 1.3 and 1.6 mm.

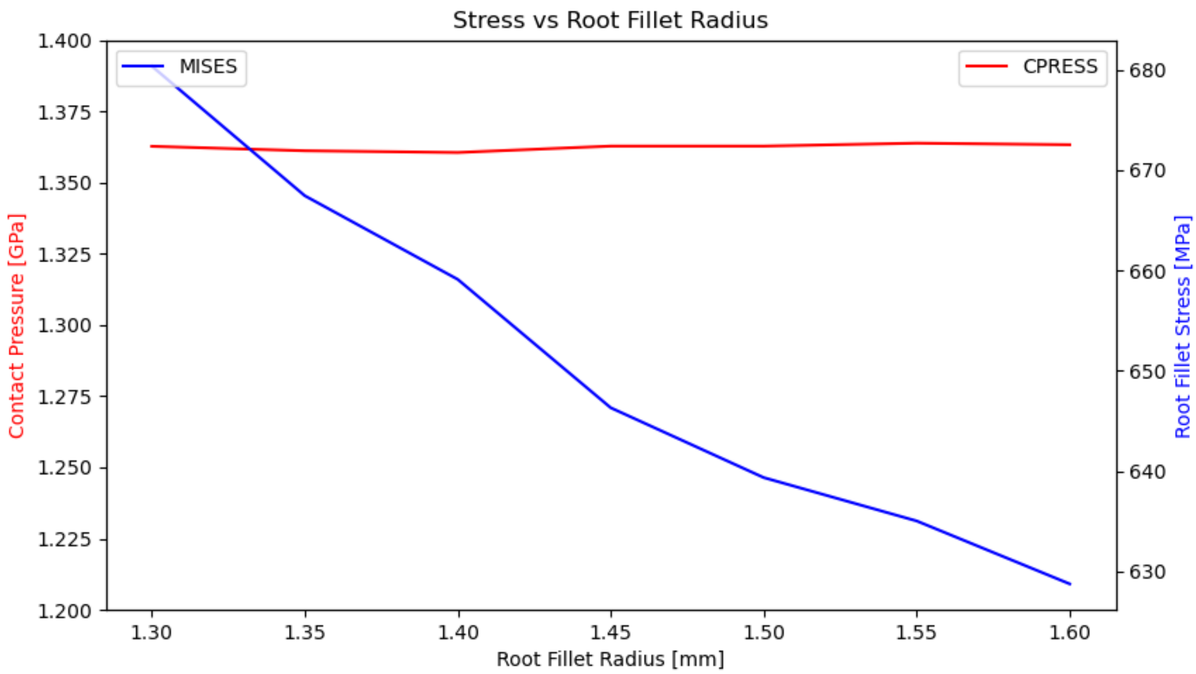

Figure 10: Contact pressure and root fillet stress versus root fillet radius.

6.1. Analysis of results

6.1.1. Elastic Modulus

The FE model is set up to only model linear elastic behaviour – plastic deformation is not considered. It was initially hypothesised that root fillet bending stress should be independent of elastic modulus, when considering Equation 6, modelling the tooth as a cantilever in bending.

$$\sigma = \frac{My}{I} \tag{6}$$

This does not appear to be the case, with a rough linear (or perhaps logarithmic) relationship between elastic modulus and root fillet stress observed in Figure 7 at lower elastic modulus values, and invariant behaviour beginning only around 600 GPa. Although stress should be invariant with elastic modulus, the deflection of the tooth is inversely proportional to the elastic modulus, as shown by Equation 7.

$$\delta_{max} = \frac{Fl^3}{3EI} \tag{7}$$

Expanding on this, additional insight can be gained by plotting the overall maximum root bending stresses against the roll angle at which it occurred, for different elastic moduli, as shown in Figure 11. Increasing elastic modulus is shown by a colour change from red to blue. As elastic modulus increases, the maximum stress point occurs progressively earlier during the meshing cycle, reaching a fixed time at 600 GPa.

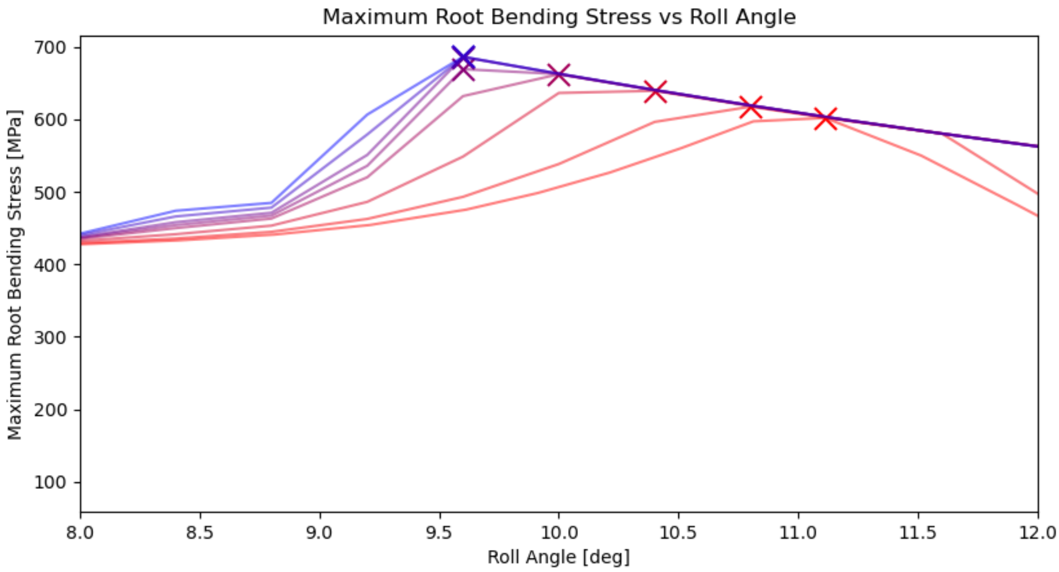

Figure 11: Maximum root bending stress along flank versus roll angle (against time), for different elastic modulus. Increasing modulus shown from red to blue, maximum stresses for elastic modulus shown with markers.

As force is applied to the tooth flank, it deflects. This causes the angle of the normal force to rotate slightly, decreasing the magnitude of the normal component of force, and therefore also reducing the bending moment. The level of deflection decreases with increasing elastic modulus, and therefore the magnitude of normal force component increases. This explains the increasing stress with increasing elastic modulus trend. As elastic modulus increases sufficiently, the behaviour of the gears becomes more similar to that of an infinitely stiff material, and deflection becomes sufficiently small that the change in normal component magnitude is negligible, explaining the convergence of results from 600 GPa.

The results shown in Figure 7 for contact pressure more closely align with the hypothesised trend. Although the normal force on the tooth flank is constant, the area over which it is applied changes. For a smaller elastic modulus, the application of the pressure results in a larger deformation. This increases the area over which the force acts, therefore reducing the pressure. Thus, increasing contact pressure with increasing elastic modulus is expected, and the parameter study reflects this well.

6.1.2. Pinion Teeth Number

The results observed in Figure 8 display a large decrease in both stresses as the number of teeth increases, with a non-linear curve. This trend could be expected, as several factors independently affect the stresses as the number of teeth varies. Firstly, the number of teeth is directly proportional to the diameter of the gear (as $d_f = z * m_n$). From this, for constant torque, the resulting force on the gears is given by Equation 8.

$$F = \frac{T}{r} = \frac{2T}{zm_n} \propto \frac{1}{z} \tag{8}$$

This factor is likely the most significant, as the variation of moment length is large over the range of 19 to 100 teeth. Accordingly, the stress curves have a gradient resembling an inversely proportional function.

Additionally, the thickness at the root of the teeth scales with number of teeth, an effect that is even more pronounced at low tooth numbers. This is illustrated in Figure 12, in which the pinion has 12 teeth, and the wheel has 55.

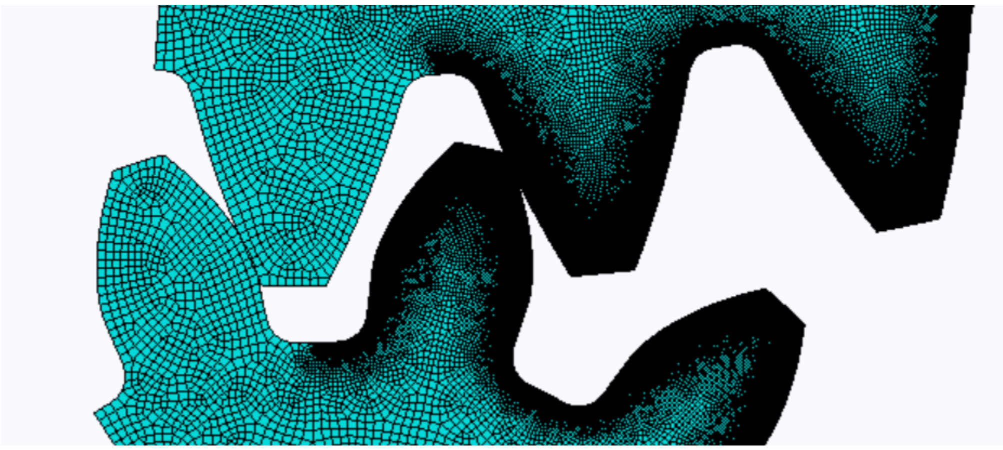

Figure 12: Different teeth profiles for z1=12 and z2=55.

This has a significant impact on the root bending stress, as when modelling the tooth as a cantilever, varying the thickness of the tooth varies its second moment of area cubically. The thickness is also varied at the location of highest bending stress along the tooth, therefore while the absolute value of the thickness variation is small, a non-negligible stress variation can still be expected.

This variation is unlikely to have a significant impact on contact pressure. However, accompanying the reduction of tooth thickness is a reduction in tooth flank radius (or increase in curvature). At high numbers of teeth, the gear becomes similar to the basic rack, and the tooth flanks are close to straight. At very low number of teeth, the gears become undercut and have a significant curvature. This effect may also be observed in Figure 12. This curvature reduces the area of contact, and consequently increases the contact pressure at that location. This effect is conversely unlikely to significantly vary the root bending stress.

The three identified effects of varying tooth number correspond well to the results of Figure 8, where the increase in stress for both contact pressure and root bending is significantly larger at low tooth numbers than at high. Finally, the results shown above also confirm that the stress behaviour is continuous with the number of teeth, and does not vary depending on whether the gear in question is the wheel or pinion. Although the model could not be run for higher teeth numbers without also varying other parameters, it is expected that the stress would slowly plateau as the relative change in teeth number becomes negligible.

6.1.3. Centre Distance

The results shown in Figure 9 show a strong positive linear relationship between root fillet stress and increasing centre distance, increasing 18% overall for a 2 mm increase from the "true" tight mesh point and a gradual negative linear relationship between contact pressure and centre distance, decreasing by 2% overall through the investigated range.

This was expected since a 2 mm increase in the centre distance represents approximately a 2% increase in total moment arm (from gear centre to contact point). Therefore, the contact pressure applied by the wheel would be expected to decrease by the same proportion, for the fixed wheel torque imposed on the model. This 2 mm increase also represents approximately 25% of the height of the pinion tooth and therefore, a large increase in moment arm about the root of the pinion's teeth, where the root fillet stress is measured. Accordingly, the root stress increases by a similarly large proportion.

Variation of centre distance required adjustment of the correction angle δ_2, previously calculated for a tight mesh centre distance, such that the simulation was initiated with a single pinion-wheel tooth contact. To do this, visual inspection, post mesh-generation, was used to iterate the δ_2 value. If the teeth were not in contact, it is likely that the initial step would not converge, whilst extreme overlap was avoided to represent contact accurately. δ_2 was then increased or decreased respectively to achieve a balance. This balance was also required since strong overlap often brought an additional tooth pair into contact, specifically the less mesh refined tooth on the edge of the 5-tooth segment. Both phenomena can be seen in Figure 13. This was avoided to keep different centre distance results as consistent as possible, and once again ensure initial convergence of the simulation.

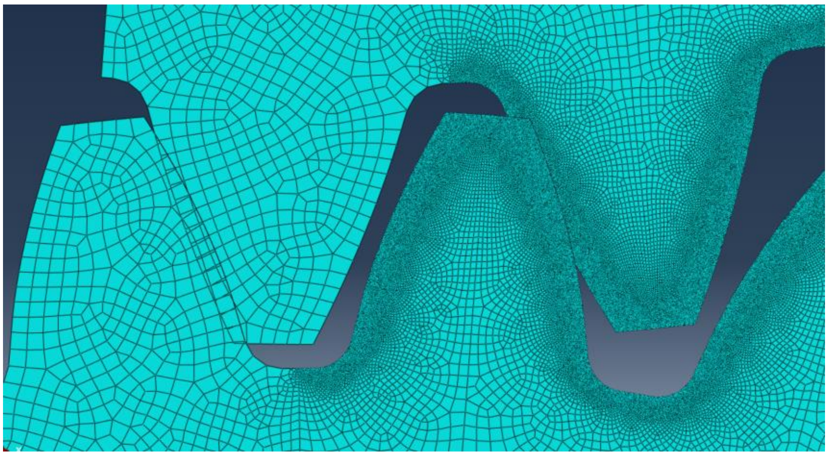

Figure 13: Correction angle too large, resulting in multiple teeth contacting, and extreme teeth mesh overlap.

Varying two parameters simultaneously had the potential to introduce error to the centre distance results if the contact was not consistently represented. However, there is little variation from the strong trends in Figure 9, which appear continuous, verifying the chosen methodology.

6.1.4. Root Fillet Radius

A smaller root fillet results in a sharper angle between the tooth flank and the gear dedendum. This sharp angle may cause an increase in stress concentration factor at the base of the teeth. This has little effect on contact pressure, as the teeth make little contact towards the base of the teeth as shown in Figure 12, where the teeth enter in contact far from the base. Additionally, the larger contact pressures generally occur at the midpoint of a meshing cycle, when only one tooth pair is in contact and the maximum pressure is close to the midpoint of each tooth. Therefore, it was expected that contact pressure should not vary with root fillet radius. This result is well reflected in Figure 10, where the contact pressure is constant across a range of root fillet radii.

While maximum contact pressure generally occurs far from the tooth base, maximum bending stress occurs precisely at the base. It is therefore expected that as the root fillet radius is decreased, stress is concentrated locally, and maximum bending stress increases. This trend can be seen in Figure 10, where root fillet stress decreases with increasing root fillet radius.

Additionally, increasing root fillet radius slightly increases the thickness at the base of the teeth. This has a similar effect on bending stress to increasing the number of teeth on the pinion, increasing the second moment of area cubically at the location of maximum stress. This also reduces stress as root fillet radius increases, although the effect is likely small as the variation still seems close to linear despite the expected non-linear effect of varying thickness. It should be noted that both effects have a comparatively small impact on the gear stresses over the range studied, as they only cause a maximum change of 8% in the bending stress experienced.

6.2. Comparison of results and recommendations for gear designers

With respect to contact pressure, the range of values experienced was significantly higher when varying elastic modulus and number of pinion teeth, with contact pressure varying from 0.6 GPa to 2.75 GPa. Varying both centre distance and root fillet radius, conversely, had little effect on contact pressure. In terms of root stress, the largest variation was obtained for number of pinion teeth, with a variation of close to 500% over the range of teeth explored, although it should be noted that the variation range of number of teeth was significantly larger than the variation range of, for instance, centre distance (which was only varied from 205.5 mm to 207.5 mm).

Since both root fillet radius and centre distance have an effect on bending stress but not on contact pressure, these parameters may be used by gear designers if they are concerned with root failure, but do not want to affect surface failure mechanisms. Centre distance may also be changed at a later stage in the design process, as it does not require any physical changes to the gears, although its variation is limited by geometric constraints, therefore it can only be used to make slight amendments to the stresses. If surface failure is also a concern, then increasing centre distance should be used with caution as it will lead to a slight increase in contact pressure.

Both elastic modulus and number of pinion teeth had a concurrent effect in terms of both stresses (either reducing or increasing them simultaneously). Therefore, these parameters may be varied when the overall safety factor of the gear pair needs to be increased for both failure modes. Elastic modulus has the advantage of only affecting stresses, and therefore would not require a full gear redesign, where varying the number of pinion teeth would influence most parameters of the gear, including a change in gear ratio.

Number of teeth is the parameter with the largest degree of freedom, as the number of teeth can always be increased further (subject to increasing the number of wheel teeth to maintain gear ratio), and only space constraints from the assembly will limit variation. Elastic modulus also has a high degree of freedom, but only a set number of values are commercially available, corresponding to commonly used materials. Root fillet radius is largely dictated by the manufacture method, and therefore may be difficult to vary accurately, and is also limited with an upper bound where the radii of two subsequent teeth intersect. Finally, centre distance may only be varied a small amount, between tight mesh and loss of gear contact.

Overall, most of the effects of the parametric study were similar to those expected, with the exception of variation of bending stress with elastic modulus. This illustrates that a good understanding of materials, stress analysis and basic gearing equations may provide transmission engineers with good intuition as to how different parameters may be selected during a preliminary design phase.

6.3. Limitations

The main limitation of this study was the limit number of nodes imposed by the Abaqus academic license. This impeded convergence during the meshing study, although it was estimated that any trends would still be clearly identifiable during the parametric study, with but slightly lower absolute stress values. Future works may wish to use a regular license to avoid this, although other alternatives may be explored, such as using a larger coarse to fine mesh ratio, or refining less areas of the gears. Additionally, it may be beneficial to use smaller gears for the study, as this would reduce the number of nodes and the computational time.

Some parameters were not explored fully, as runs would not converge past a certain point, for instance teeth numbers below 19 or elastic modulus below 60 GPa. Several runs were tried for these values, varying parameters such as initial gear rotation and centre distance, with little success. This may be a limitation simply of the software package used, therefore other FEA packages may be useful to run studies for the more extreme values.

Finally, the interpolation of the generated gear flanks was identified as slightly larger than the original input dimensions, leading to gear overlap at tight mesh centre distance. This was overcome by running all simulations at a larger centre distance, and it is recommended that other works also do this to avoid numerical errors.

7. Conclusion

To conclude, a pair of involute spur gears were modelled in Abaqus 2021, and a Finite Element Analysis was conducted to observe the stresses present during a meshing cycle. Care was initially taken to generate the gear profiles, ensuring that the flanks followed an involute curve, and the distribution of the mesh was informed based on the expected loading, with flanks of interest having a finer mesh. In order to evaluate what level of mesh refinement was appropriate, a convergence study was undertaken, comparing the change in both tooth flank contact pressure and root fillet bending stress, as mesh refinement was varied. The level of refinement was limited, due to the node limit imposed by the Abaqus license. Nonetheless, when comparing the stress values to the analytical values predicted by the Lewis equation and Hertz pressure equation, it was decided that a fine mesh size of 0.03 mm was sufficiently close to convergence to allow for valid and informative results.

Following this a parametric study was conducted, to explore the effect of varying centre distance, elastic modulus, pinion teeth number and root fillet radius, on both the contact pressure and root fillet bending stress. The results generally followed expected hypotheses, and the majority were well explained using basic stress analysis, although the effect of changing elastic modulus on root fillet bending stress differed from expectation, and an alternative hypothesis was presented. Several limitations were identified, such as parameter values that could not be assessed due to the FE solver failing to converge, as well as the node limit which prevented a more substantial conclusion for the convergence study.

References

MacLaren, A. (2022). MTT Gear FEA Coursework Handout. Imperial College London.