



University of Wales Trinity Saint David

School of Engineering

Motion Simulation & FEA

Advanced Computational Methods

M6X00408

Callum-James Hill

P148699

January 2020



Contents

1	CAD OneDrive Link.....	3
2	Acronyms.....	3
3	List of Equations.....	3
4	List of Graphs	3
5	List of Tables	3
6	List of Figures	4
7	Introduction.....	1
8	Background Research	1
9	Modelling Methodology	2
10	Assembly & Motion Simulation.....	6
11	FEA Results & Analysis.....	13
12	Optimization.....	15
12.1	Initial Optimization	15
12.1.1	Methodology	15
12.1.2	Results & Analysis	16
12.2	Final Optimization	18
12.2.1	Methodology	18
12.2.2	Results & Analysis	20
13	Conclusion.....	22
14	References	23
15	Appendix	24
15.1	Cam Profile	24
15.2	RPM Limit Regulations	25
15.2.1	BSB Motostar Moto3	25
15.2.2	FIM Moto3 Junior World Championship	25
15.2.3	World Moto3.....	25
15.3	Spring Rates	26
15.4	Friction Coefficients.....	27
15.5	Solid Body Contact Forces Graphs in SolidWorks	27

1 CAD OneDrive Link

To Access the SolidWorks Files used within this study, please follow the link below.

https://uowtsd-my.sharepoint.com/:f:/g/personal/chill_148699_student_uwtsd_ac_uk/EjyLW6ikHS5Fn6Ky9R8KER8BAOxTg2zO5umxkkA8CPOdhg?e=4NyHAN

2 Acronyms

FEA – Finite Element Analysis

FEM – Finite Element Method

CAE – Computer-aided Engineering

CAD – Computer-aided Design

ICE – Internal Combustion Engine

RPM – Revolutions Per Minute

3 List of Equations

Equation 1 Camshaft Speed	7
Equation 2 Revolutions Per Second.....	7
Equation 3 Time Taken per for One Revolution.....	7
Equation 4 Frames Needed at 6750 RPM.....	7
Equation 5 Safety Factor.....	20
Equation 6 ESTRN (SolidWorks, Strain Components, 2013)	21

4 List of Graphs

Graph 1 Solid Body Contact Forces	10
Graph 2 Typical Stress-Strain Curve (Omnexus, 2020)	17
Graph 3 Moto3 KTM Cam Lift.....	25
Graph 4 KTM Moto3 Spring Rates.....	26
Graph 5 Camshaft to Follower Contact Forces.....	27
Graph 6 Finger Follower to Shim Contact Force.....	28

5 List of Tables

Table 1 Component Masses	5
Table 2 Materials Used	5
Table 3 Maximum Forces Newtons	10
Table 4 Mesh Properties	12
Table 5 Maximum Stress and Displacement.....	20

Table 8 Friction Coefficients (The Engineering ToolBox, 2004).....	27
--	----

6 List of Figures

Figure 1 2014 KTM RC250R Moto3 Cylinder Head	2
Figure 2 2014 KTM RC250R Moto3 Finger Follower	2
Figure 3 Finger Follower SolidWorks Model	2
Figure 4 Equations, Global Variables & Dimensions	3
Figure 5 Sketch 1 of Finger Follower	3
Figure 6 2014 KTM RC250R Moto3 Intake Camshaft	3
Figure 7 Intake Camshaft SolidWorks Model	3
Figure 8 2014 KTM RC250R Moto3 Intake Valve	4
Figure 9 Intake Valve SolidWorks Model	4
Figure 10 2014 KTM RC250R Moto3 Finger Follower Shaft	4
Figure 11 Valve Hat SolidWorks Model	4
Figure 12 2014 KTM RC250R Moto3 Collett	4
Figure 13 135 Collett SolidWorks Model	4
Figure 14 2014 KTM RC250R Moto3 Shim	4
Figure 15 Shim SolidWorks Model	4
Figure 16 Valve Train Assembly	6
Figure 17 Follower Shaft Equation	6
Figure 18 Rotary Motor	7
Figure 19 Cam Shaft to Finger Follower Contact Force	8
Figure 20 Finger Follower to Shim Contact Force	8
Figure 21 Import Motion Loads	9
Figure 22 Processing scenarios	9
Figure 23 Finger Follower Forces	10
Figure 24 Maximum Force Point	11
Figure 25 finger follower All Forces	11
Figure 26 Coarse Mesh	12
Figure 27 Fine Mesh	12
Figure 28 Von Mises Stress Equation Expressed by the Six Stress Components (Reena Patel 1, 2019)	13
Figure 29 Von Mises Stresses Standard Finger Follower	13
Figure 30 Minimum Von Mises Stress Baseline Follower	14
Figure 31 Maximum Stress Standard Follower	14

Figure 32 Standard Finger Follower Displacement	14
Figure 33 Strain Standard Finger Follower.....	15
Figure 34 Sketch 8 of Initial Optimization	15
Figure 35 Finger Follower Initial Optimization.....	16
Figure 36 von Mises Stresses for Initial Optimization.....	16
Figure 37 Displacement for Initial Optimization.....	17
Figure 38 Strain Initial Optimization	18
Figure 39 Sketch 8 of Final Optimization.....	18
Figure 40 Final Optimization Fillet	19
Figure 41 Finger Follower Final Optimization	19
Figure 42 Von Mises Stresses Final Optimization.....	20
Figure 43 Displacement Final Optimization.....	21
Figure 44 Strain Final Optimization	21
Figure 45 Measuring Intake Cam Profile.....	24
Figure 46 BSB Moto3 Rev Limit (Motorsport Vision Racing, 2019)	25
Figure 47 Moto3 World Junior Championship Rev Limit (FIM, 2019)	25
Figure 48 World Moto3 Rev Limit (FIM, 2019)	25
Figure 49 Measuring Intake Valve Spring Rate.....	26

7 Introduction

This report will consist of a Finite Element Analysis (FEA) study and optimization of the finger followers within the Intake valve train of a 2014 KTM RC250R Moto3, proving that computational methods help further improve a component. With the use of SolidWorks, the appropriate components were modelled and assembled, then a motion study and an FEA analysis using the loads exerted on to the finger followers during the motion simulation was carried out. Furthermore, from analysing the FEA results, improvements were undertaken to the component in order to optimize its performance and compared to the original part to determine how effective the enhancements were.

8 Background Research

“FEA is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM)” (Sim Scale, n.d.). It is the furthestmost commonly used method for solving complications within engineering and mathematical models. Furthermore, the beginnings of FEA can be traced back as far as Euler in the 16th Century, and a more rigid definition tracing back to Schellbach in 1851, but it wasn’t until the mid-1950’s that the techniques of FEA *“had become advanced enough that engineers could start using it in real-world situations”* (English, 2019). Modern day FEA software is used to predict how a component will react to physical effects such as forces, fluid flow, heat transfer etc. It works by *“breaking down a real object into many finite elements. Mathematical equations help predict the behaviour of each element, adding up all the individual behaviours predicts the behaviour of the actual object”* (Autodesk, 2018).

Due to the ever-increasing complexity of mechanical products, the kinematics and dynamics of a component need to be determined, as well as a structural performance FEA analysis to test the feasibility of a potential product before a prototype can be manufactured. Motion simulation provides information about the kinematics including *“position, velocity & acceleration”*. (SolidWorks, 2008) It also provides information on the dynamics including *“joint reactions, inertial forces & power requirements of all the components of a moving mechanism”*. (SolidWorks, 2008).

The valve train within an ICE controls the operation of the valves governing the amount of air and fuel that enters the combustion chamber. When designing the components within the valve train there is *“generally a conflict between the demand for fast opening valves”* (Husselman, 2005) which in turn will result in large valve accelerations and *“the requirement of limiting loads”* (Husselman, 2005) which has the opposite affect reducing the valves acceleration. With the use of CAE simulation technology, the kinematic and dynamic characteristics of the components within the valve train can be analysed to see the feasibility during the early stages of development.

9 Modelling Methodology

Firstly, the valve train components that needed to be modelled in order to carry out the study were removed from the cylinder head and measured using a Vernier Calliper enabling them to be modelled accurately.



Figure 1 2014 KTM RC250R Moto3 Cylinder Head

The purpose of this report as previously stated is to carry out an FEA analysis on the finger follower, therefore, a crucial factor was ensuring the correct materials were used to ensure the most accurate results. The finger followers are made from a cast steel allowing them a *“very complex component geometry, which enables an especially high resilience”* (Schaeffler, 2017).



Figure 2 2014 KTM RC250R Moto3 Finger Follower



Figure 3 Finger Follower SolidWorks Model

When modelling the finger follower, it was acknowledged that if the follower were to be used in a different engine it may need to be larger or smaller. Global variables and equations were used to allow the follower to be scaled to different sizes whilst maintaining its original shape as shown in figures 4 & 5.

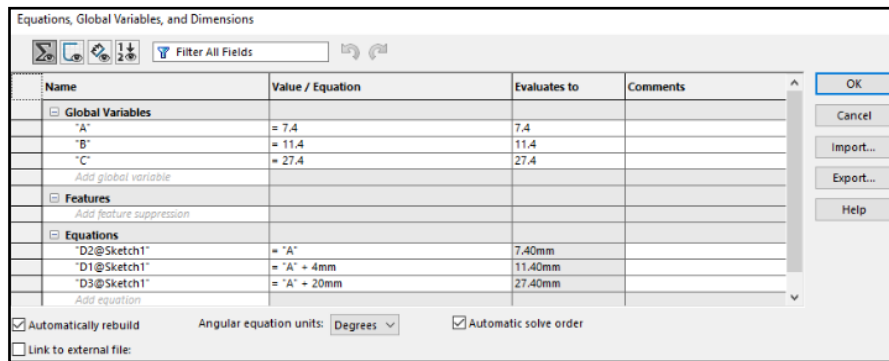


Figure 4 Equations, Global Variables & Dimensions

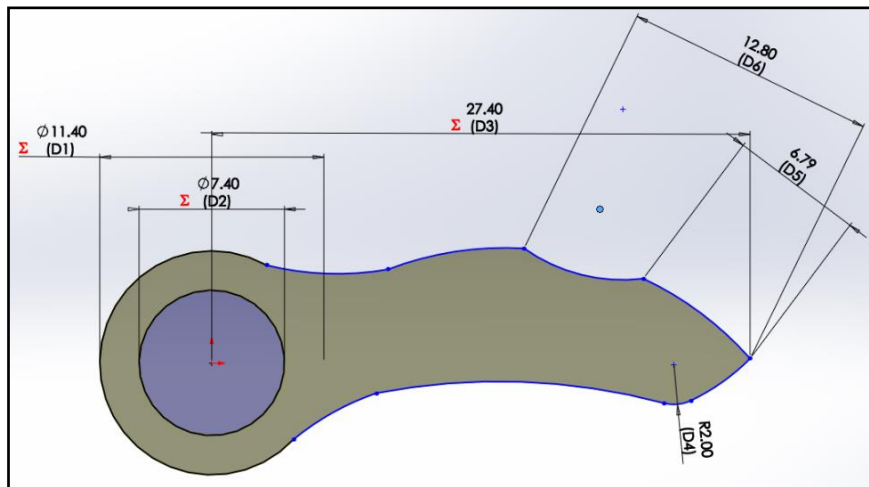


Figure 5 Sketch 1 of Finger Follower

Next, to be able to correctly model the camshaft the cam profile was measured with the use of a DTI gauge and an Optical Dividing Head, please refer to Appendix 15.1. An image that is perpendicular to the profile of the cam lobe was imported in to SolidWorks and used as a template to sketch around, then the dimensions of the lobe which had been previously measured were applied to the model.



Figure 6 2014 KTM RC250R Moto3 Intake Camshaft

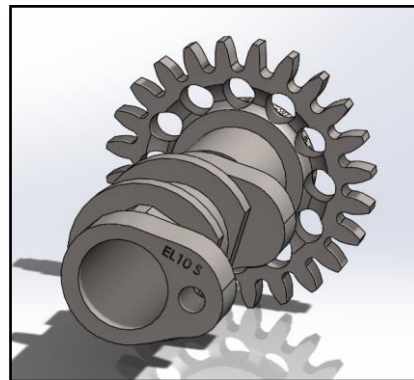


Figure 7 Intake Camshaft SolidWorks Model



Figure 8 2014 KTM RC250R Moto3 Intake Valve

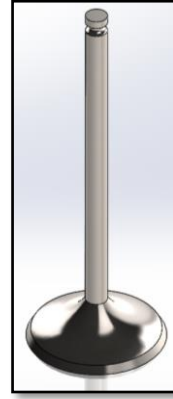


Figure 9 Intake Valve SolidWorks Model



Figure 10 2014 KTM RC250R Moto3 Finger Follower Shaft

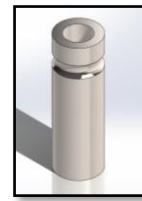


Figure 11 Finger Follower Shaft



Figure 12 2014 KTM RC250R Moto3 Valve Hat



Figure 11 Valve Hat SolidWorks Model



Figure 12 2014 KTM RC250R Moto3 Collett

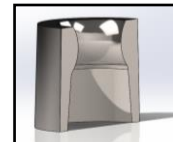


Figure 135 Collett SolidWorks Model



Figure 14 2014 KTM RC250R Moto3 Shim

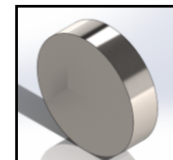


Figure 15 Shim SolidWorks Model

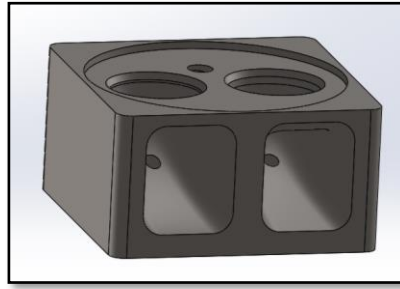


Figure 18 Cylinder Head SolidWorks Model

Next, the mass of each component was found with the use of a set of weighing scales. Subsequently, the materials of most of the components within the valve train were unknown and unattainable. Therefore, research was carried out to discover common materials used for each part, then as a validation tool the researched materials were applied to the SolidWorks models and the masses were then compared to that of the actual masses.

Component	Actual Mass (g)	SolidWorks Mass (g)	Difference
<i>Intake Camshaft</i>	242	233.54	3.56%
<i>Finger Follower</i>	7.7	7.50	2.63%
<i>Finger Follower Shaft</i>	5	4.66	7.04%
<i>Intake Valve</i>	24	24.70	2.87%
<i>Valve Hat</i>	7	7.82	11.06%
<i>Collett</i>	0.63	0.7	10.53%
<i>Shim</i>	0.8	0.92	6.74%

Table 1 Component Masses

For most components within the study the geometry was the most crucial factor, therefore the mass being within 15% was acceptable. On the other hand, with the camshaft and finger follower, the mass was an imperative factor to ensure the results produced within the FEA analysis were accurate. Both were within 5% of the actual mass, which was an acceptable tolerance.

Component	Material
<i>Intake Camshaft</i>	Cast Iron
<i>Finger Follower</i>	Cast Alloy Steel
<i>Finger Follower Shaft</i>	Stainless Steel
<i>Intake Valve</i>	Ti-6Al-4V Solution treated and aged (SS)
<i>Valve Hat</i>	Cast Alloy Steel
<i>Collett</i>	Plain Carbon Steel
<i>Shim</i>	Plain Carbon Steel

Table 2 Materials Used

10 Assembly & Motion Simulation

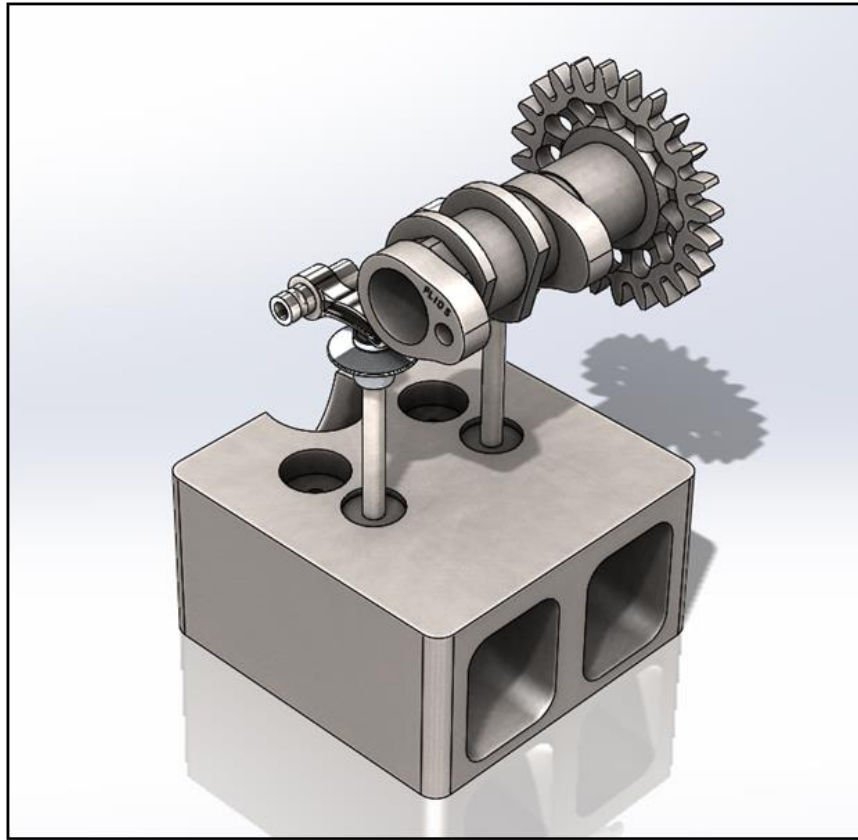


Figure 16 Valve Train Assembly

Figure 16 shows the modelled components when assembled, this was done using a method of mating the required components to one another. Mechanical mates were used when possible to reduce the amount of redundancies within the assembly. Furthermore, an equation was used within the assembly so that if the internal diameter of the hole in the finger follower were to be adjusted, the shaft that it sits on would re-calculate to be the same diameter.

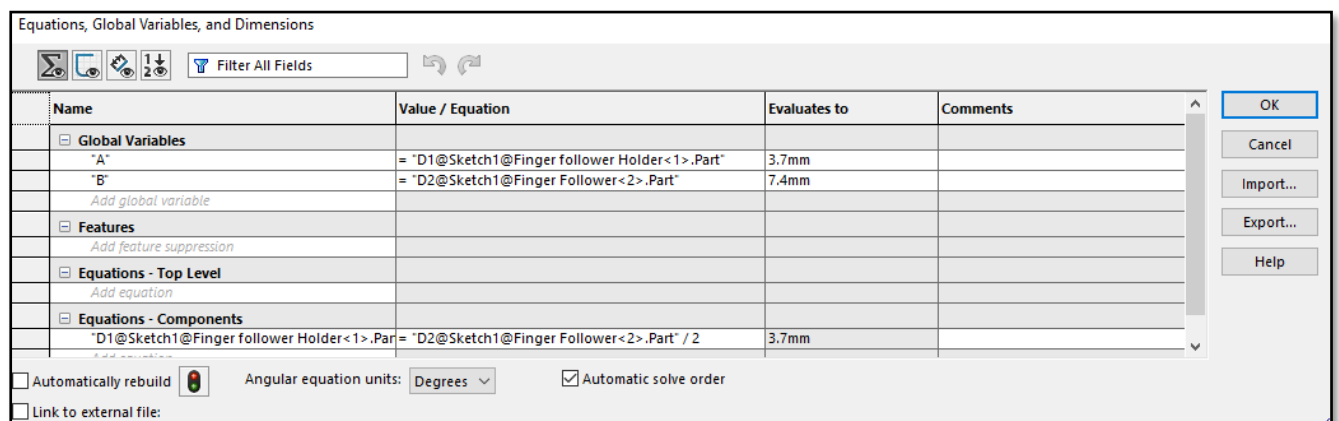


Figure 17 Follower Shaft Equation

The Motostar Moto3 support class in British Superbikes (Motorsport Vision Racing, 2019), FIM Moto3 Junior world Championship (FIM, 2019) and World Moto3 (FIM, 2019) regulations all state that the RPM must be limited to 13,500, please refer to Appendix 15.2. Therefore, for this study a rotary motor was applied to the camshaft running at a constant speed of 6,750 RPM as;

$$\text{Camshaft Speed} = \frac{\text{Crankshaft Speed}}{2}$$

Equation 1 Camshaft Speed

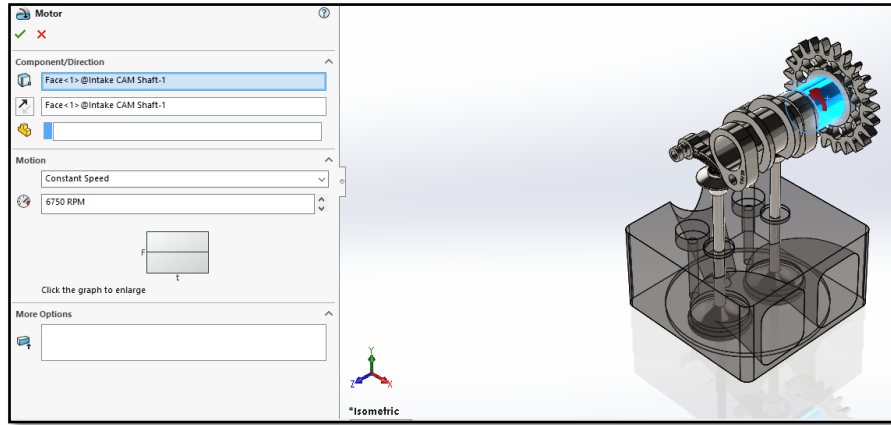


Figure 18 Rotary Motor

The simulation consists of 360° of camshaft rotation, as this was all that was needed to simulate all the forces exerted onto the finger followers and only took 0.0089 Seconds which saves on computational time.

$$\text{Revolutions Per Second} = \frac{6750}{60}$$

$$\therefore = 112.5$$

Equation 2 Revolutions Per Second

$$\text{Time Taken for One Revolution} = \frac{1}{112.5}$$

$$\therefore = 0.0089s$$

Equation 3 Time Taken per for One Revolution

With the use of equation 4 below the required number of frames within the motion study was calculated.

$$\text{Frames needed at 6750 RPM} = 112.5 \times 360$$

$$\therefore = 40,500$$

Equation 4 Frames Needed at 6750 RPM

Furthermore, a linear spring with a rate of 26 N/mm was applied to both valves. This figure was acquired from measuring the spring rates of the actual valve springs from the Moto3, please refer to appendix 15.3. Gravity was also added to the model with a rate of acceleration of 9806.65 mm/s² as without it the results would be inaccurate. The contact between the cam lobe and the finger followers was modelled using solid body contact with a kinetic friction coefficient of 0.133 and a static friction coefficient of 0.21. These values were founded after research was conducted on iron to steel friction coefficients, which can be found in appendix 15.4 (The Engineering ToolBox, 2004). Another important contact point within the simulation is that of the finger follower upon the shim. This was also modelled using a solid body contact, as both materials are steel SolidWorks assumed coefficient of friction values which are shown in figure 20.

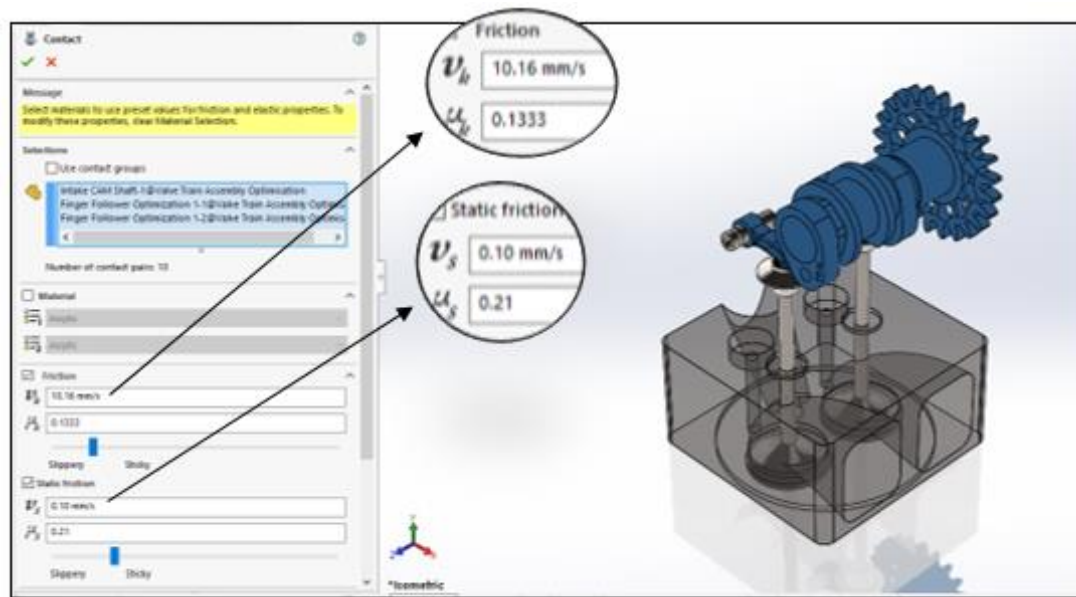


Figure 19 Cam Shaft to Finger Follower Contact Force

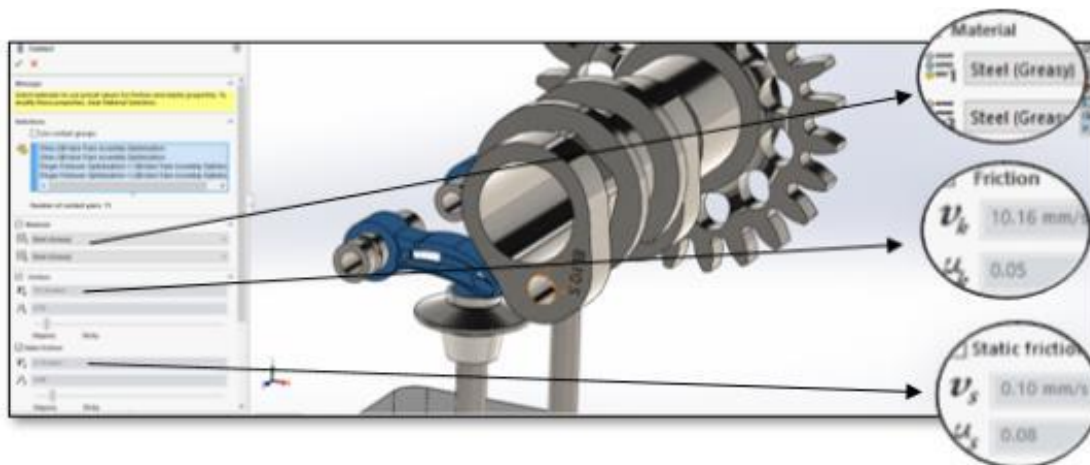


Figure 20 Finger Follower to Shim Contact Force

After running a motion study, the forces that take place within the simulation on individual components can be exported out to calculate their stresses and displacements. This is carried out with the use of the “*Import Motion Loads*” tool within SolidWorks Simulation, which allows the user to create a design scenario for every chosen time frame populated with motion loads developed at that particular frame. When the motion simulation had been calculated the import motion loads tools was then used on the finger follower.

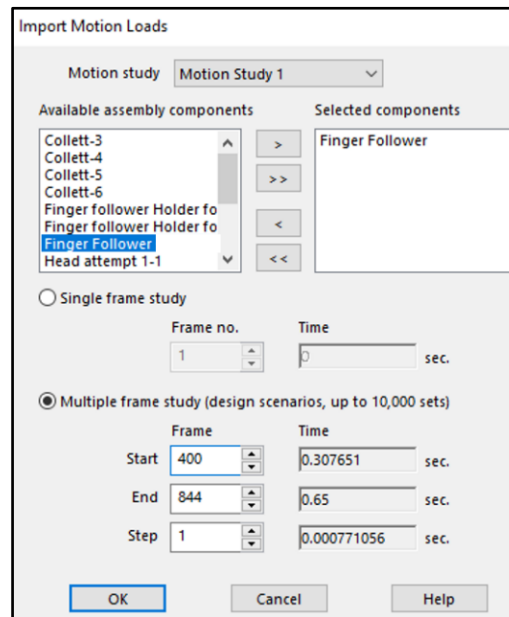


Figure 21 Import Motion Loads

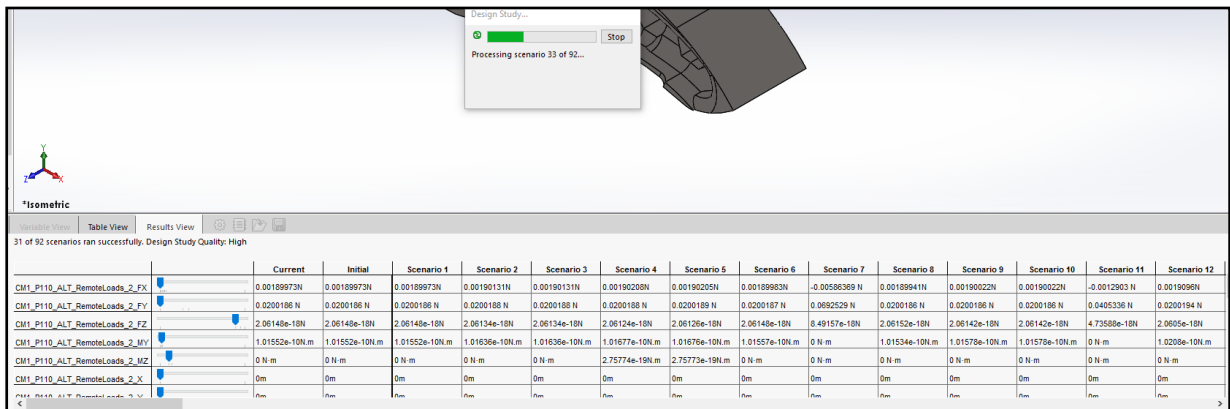


Figure 22 Processing scenarios

Upon completion of this all the scenarios were run; it was apparent that the contact forces placed onto the finger follower from the cam lobe and shim were not exported. Only Gravity, centrifugal and remote loads were displayed, please refer to figure 23.

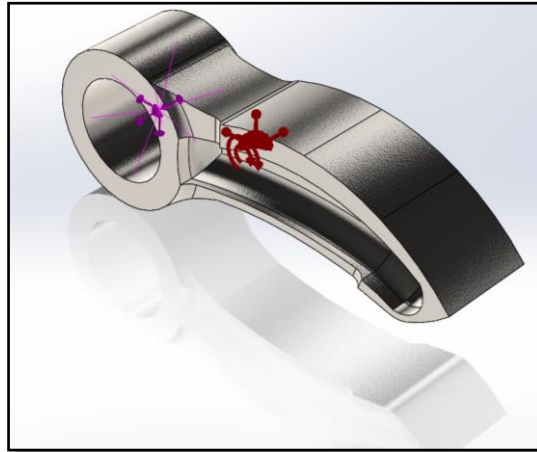


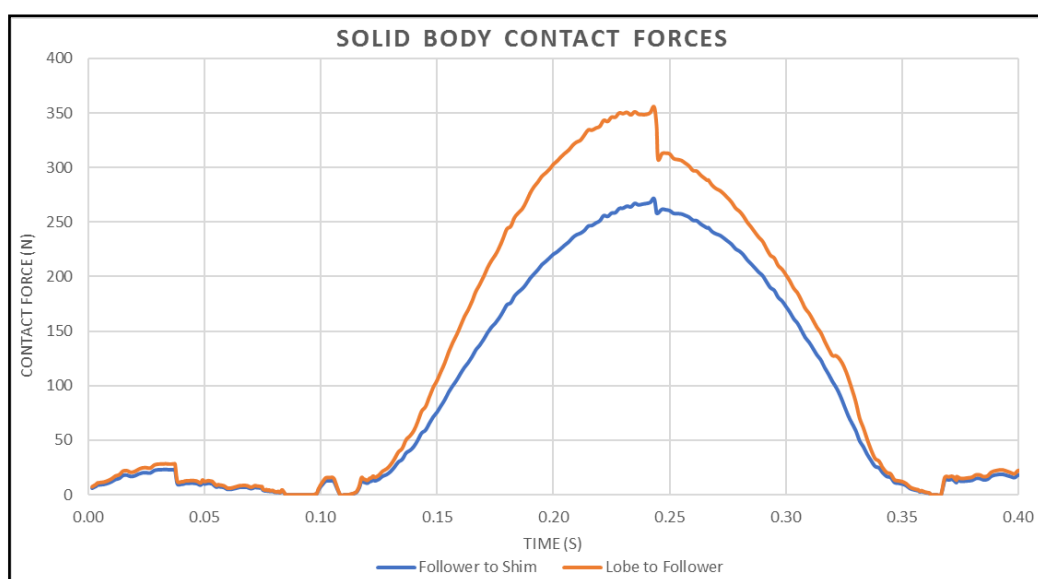
Figure 23 Finger Follower Forces

In order to rectify this, two separate force graphs were plotted within the motion study to see at what point the maximum force was being exerted onto the finger follower from the cam lobe and the shim, these graphs can be found in appendix 15.5. This data was then exported into excel where the maximum values were found as displayed in table 3, measured in newtons.

Maximum Force Exerted onto Shim	Maximum Force Exerted onto Follower
271.15	355.27

Table 3 Maximum Forces Newtons

Graph 1 shows that the amount of force being placed on the follower is not constant throughout the motion study.



Graph 1 Solid Body Contact Forces

The position on the follower where the maximum force occurs from the cam lobe needed to be located so it could be applied to the FEA model. The reason this point needed to be found was because of the contact point between the lobe and follower continuously changing, meaning it would be inaccurate to apply the maximum force to the entirety of the top surface of the follower, when maximum force is only being applied to one particular area. The Point was discovered from analysing the force graph generated within SolidWorks (Appendix 15.5), then a sketch was made within the assembly to mark where the lobe was sitting on the follower at this point.

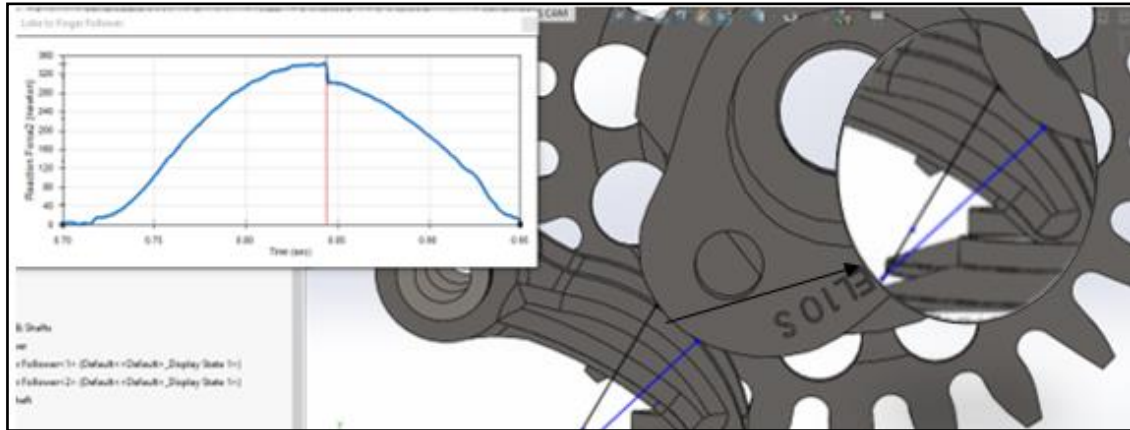


Figure 24 Maximum Force Point

From this sketch the “*smart dimension*” tool was utilised to evaluate where each point either side of the lobe and followers contact point, was in relation to the nose of the follower. The sketch was then re-created on the follower out of the assembly and the surface split, this process was then repeated for the contact point between the shim and follower where maximum force occurs.

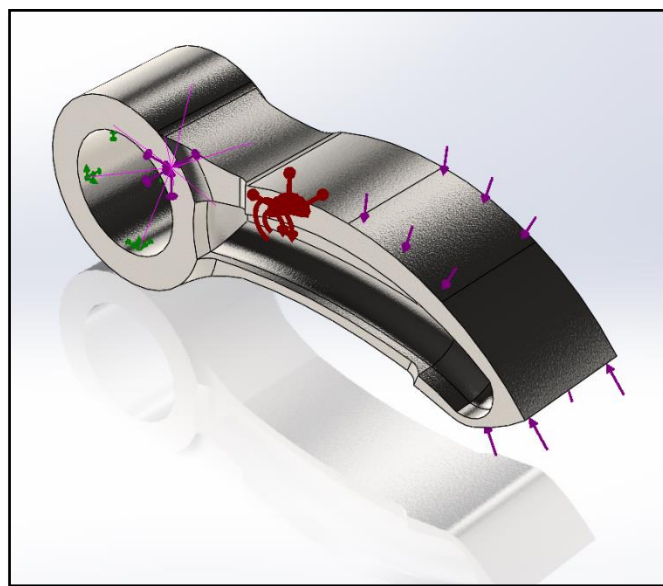


Figure 25 finger follower All Forces

The motion study was re-calculated and the “*import motion loads*” tool was used again to evaluate all scenarios, but this time due to the split surfaces the contact forces were applied within the FEA study. Furthermore, the inner diameter was fixed in space so that the geometry was a fixed constraint as shown in figure 25.

“*The accuracy that can be obtained from any FEA model is directly related to the finite element mesh that is used*” (Cosmol, 2017). The refinement of a mesh is one of the most crucial factors for accurate results, if a mesh is too coarse it can yield inaccurate results in analysis. With a coarse mesh the component is subdivided into larger elements, whereas a finer mesh breaks the component down into smaller elements, which takes more computational time but provides more accurate results “*FE models with fine mesh yield to highly accurate results*” (Liu, 2020).

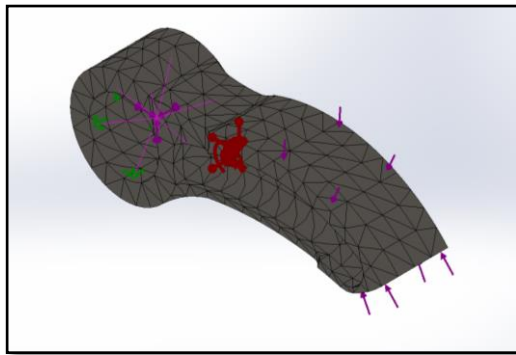


Figure 26 Coarse Mesh

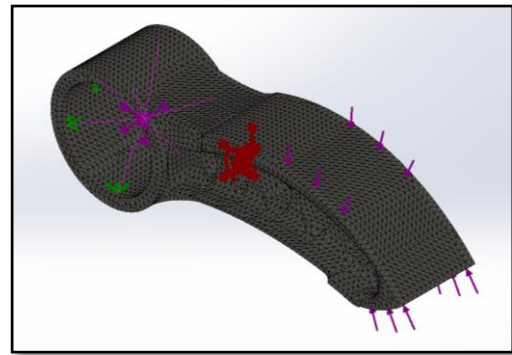


Figure 27 Fine Mesh

For this study the finest mesh available within SolidWorks was applied to the component, this was done on account of the importance of calculating the most accurate results. As previously stated, the finer the mesh the more computational time is taken, in this case that was not an issue due to the size of the component being small.

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points	4 Points
Element Size	0.493781 mm
Tolerance	0.0246891 mm
Mesh Quality Plot	High

Table 4 Mesh Properties

Table 4 shows that Jacobian points were used within the mesh, meaning quadratic elements were used as opposed to the linear elements within a draft quality mesh, it is also worth mentioning that both are tetrahedral.

11 FEA Results & Analysis

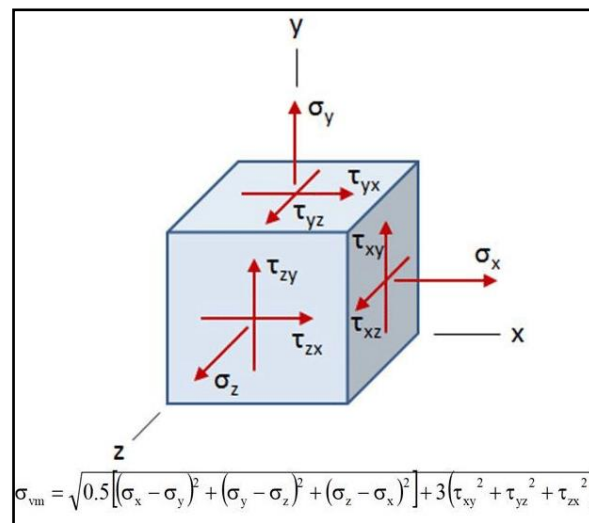


Figure 28 Von Mises Stress Equation Expressed by the Six Stress Components (Reena Patel 1, 2019)

Figure 28 displays the von mises stress equation. Von Mises is used to calculate when a material will yield and is mostly used for ductile materials such as metals. The concept of von mises stress comes from the distortion of energy theory which is the theory used to predict when a tough material will fail “yielding occurs when the distortion energy reaches a critical value” (Simscale , n.d.)

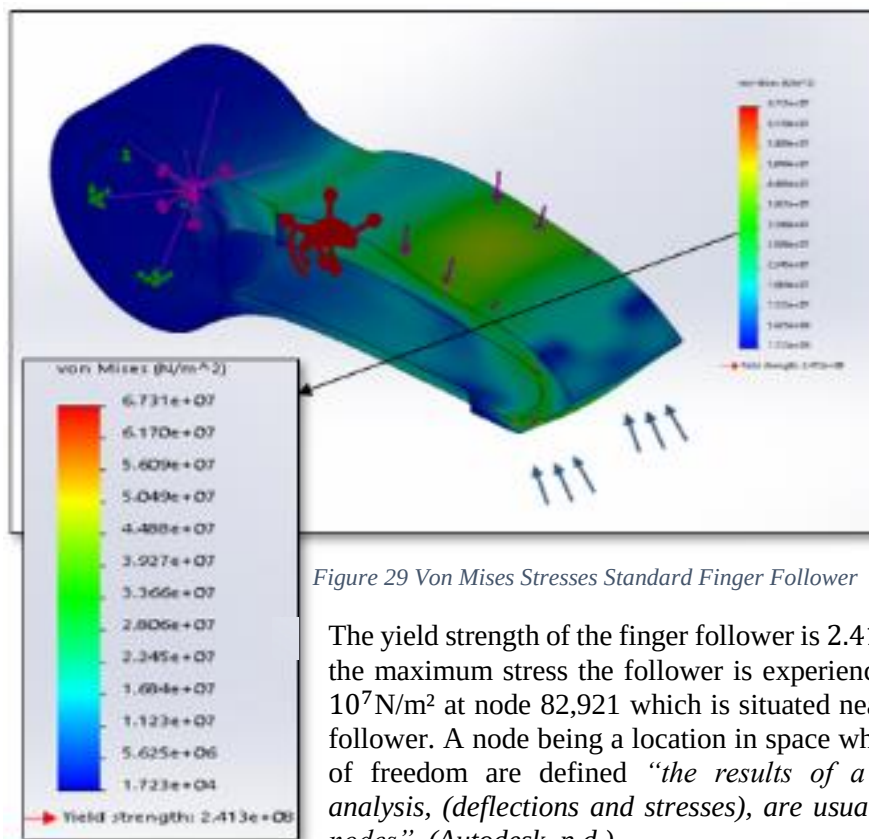


Figure 29 Von Mises Stresses Standard Finger Follower

The yield strength of the finger follower is 2.413×10^8 N/m², the maximum stress the follower is experiencing is 6.731×10^7 N/m² at node 82,921 which is situated near the tip of the follower. A node being a location in space where the degrees of freedom are defined “the results of a finite element analysis, (deflections and stresses), are usually given at the nodes”. (Autodesk, n.d.).

Furthermore, the minimum stress the component witnessed was $2.413 \times 10^8 \text{ N/m}^2$ at node 73,992 which is situated at the hole at the back of the follower where the shaft sits.

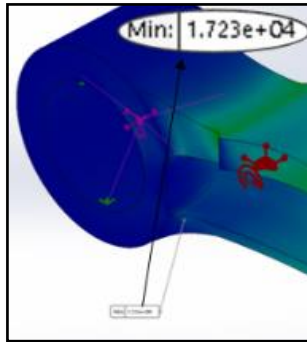


Figure 30 Minimum Von Mises Stress Baseline Follower

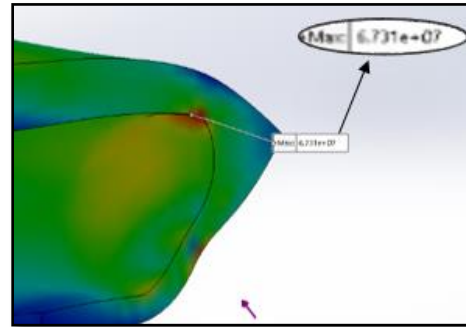


Figure 31 Maximum Stress Standard Follower

Within the FEA model there is a minimum displacement of $1 \times 10^{-30} \text{ mm}$ and a maximum displacement the follower sees is 0.0078 mm , both values are so minute they are negligible. Furthermore, as figure 32 below shows the maximum displacement occurs at the tip of the follower, this being on account of the contact force from the lobe pushing down on the follower and the valve spring pushing the valve and shim back up against the follower in the opposing direction.

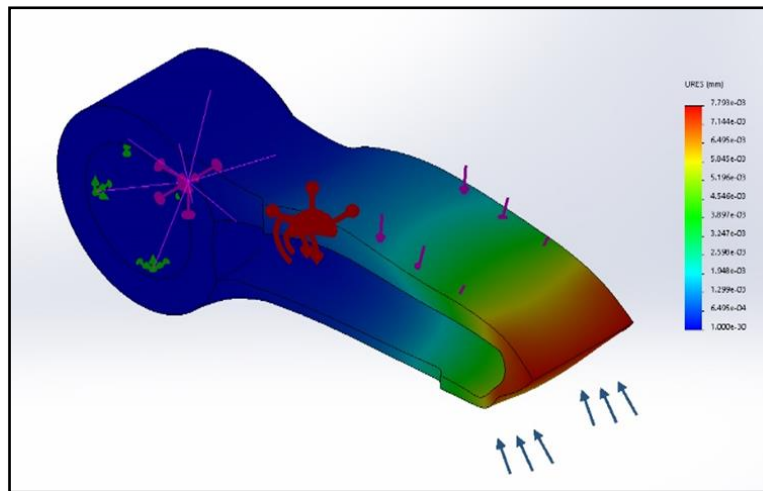


Figure 32 Standard Finger Follower Displacement

Strain is described as the change in length due to a force divided by the materials original length and is a dimensionless unit as it is the ratio of two different lengths “Engineering strain is defined as the amount of deformation in the direction of the applied force divided by the initial length of the material” (NDT Resource Center, n.d.). The minimum strain the finger follower experienced during the study was 1.624×10^{-7} and the maximum was 2.44×10^{-7} both of which are inconsequential. Finally, as displayed in figure 33 it is clearly visible that the minimum and maximum strains are taking place in the same areas as the minimum and maximum stresses.

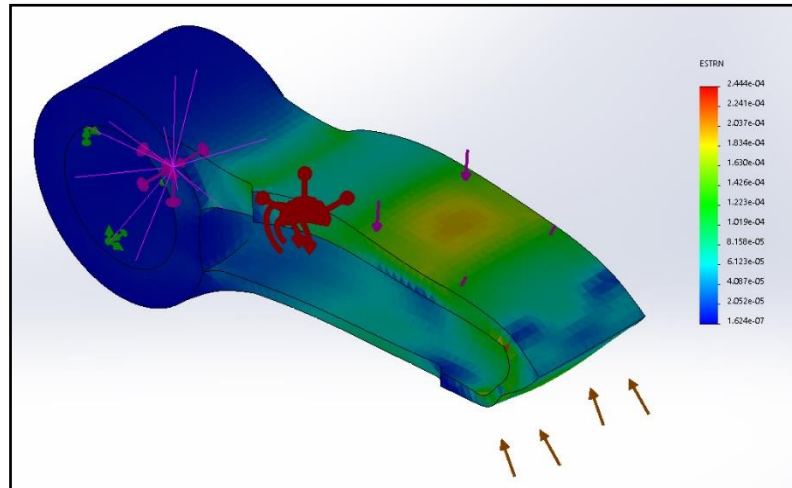


Figure 33 Strain Standard Finger Follower

12 Optimization

12.1 Initial Optimization

12.1.1 Methodology

The standard finger follower was massively below the point of yield even at the point of maximum stress there was a difference of 112.43%. Therefore, the aim was to optimise the component by a method of reducing its mass by removing un-necessary materials, the lighter the component the less energy it requires when moving. This was achieved by cutting two holes in the component but keeping the material at the centre to maintain the structural integrity *“Lightening holes, which first found their way into race cars in the 1920’s, are holes in structural components used by a variety of engineering disciplines to make structures lighter”* (Woods, n.d.). This reduced the mass of the component by 21.6% from 7.50 grams to 5.88 grams.

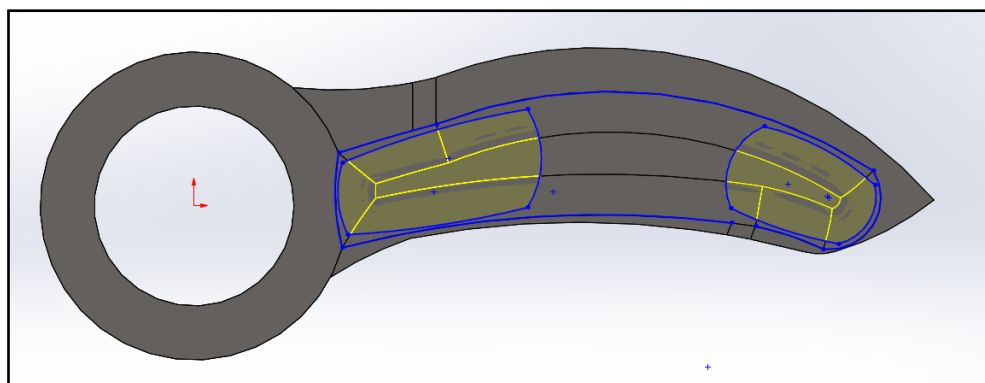


Figure 34 Sketch 8 of Initial Optimization



Figure 35 Finger Follower Initial Optimization

12.1.2 Results & Analysis

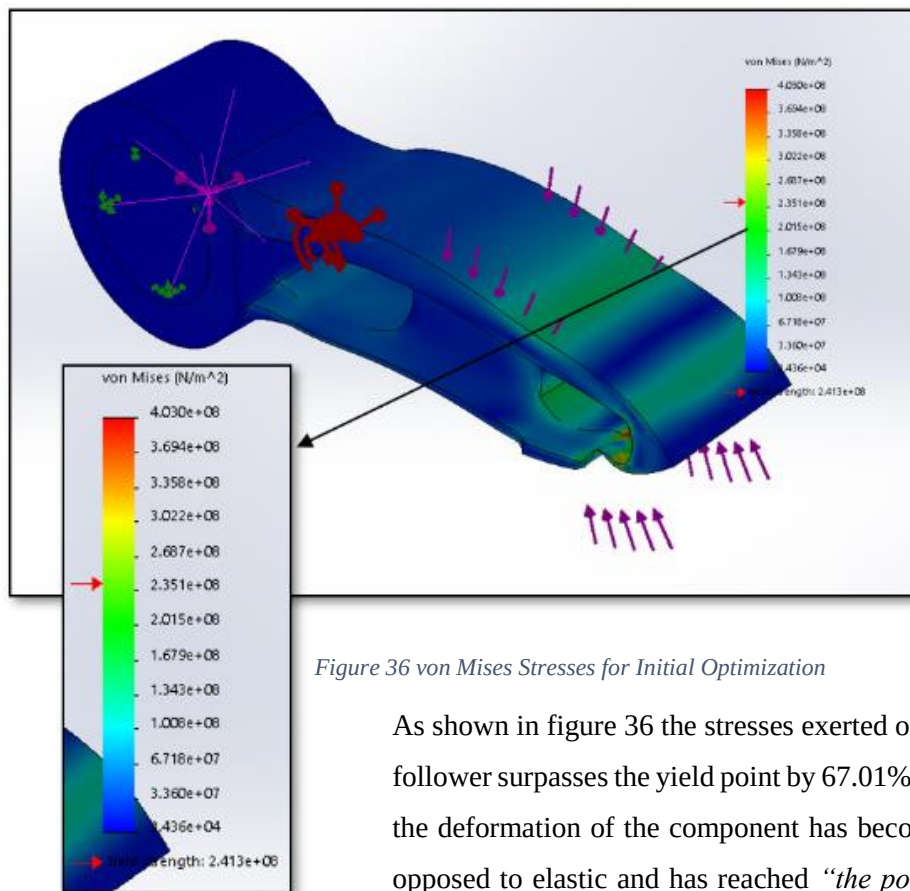
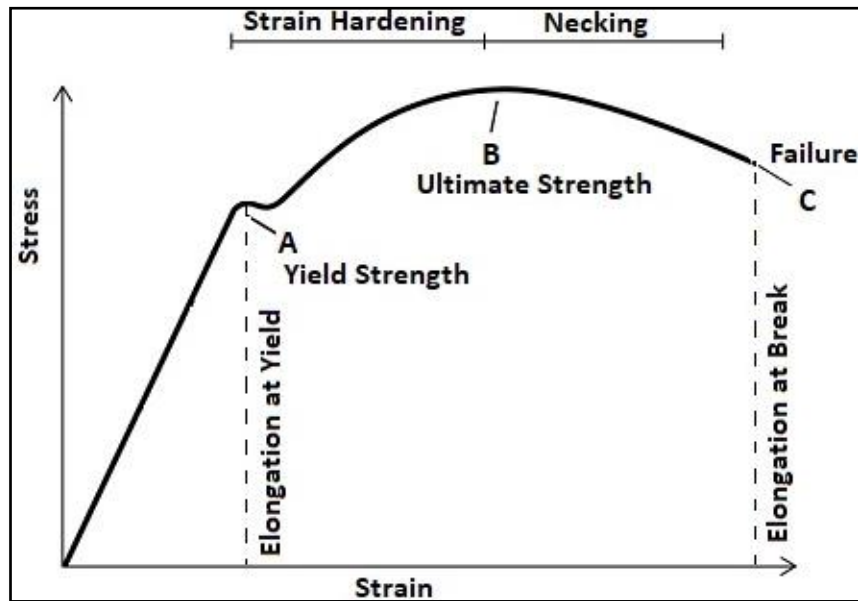


Figure 36 von Mises Stresses for Initial Optimization

As shown in figure 36 the stresses exerted onto the finger follower surpasses the yield point by 67.01% meaning that the deformation of the component has become plastic as opposed to elastic and has reached “the point where the material begins to have permanent deformation” (Leonghuat, n.d.).



Graph 2 Typical Stress-Strain Curve (Omnexus, 2020)

Furthermore, when comparing to the standard finger follower the maximum stress seen increased 498.72% which is a monumental escalation. This is a result of the hole cut into the finger follower situated at the front of the component being too large, not leaving enough material to withstand the forces acting upon it.

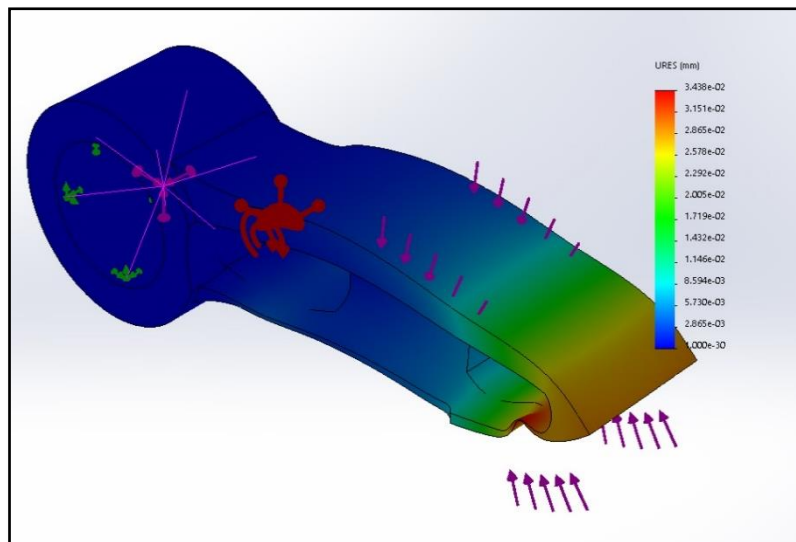


Figure 37 Displacement for Initial Optimization

Figure 37 shows that the maximum displacement is taking place at the same point on the follower where max stress arises. Also, even though the stress the component is experiencing is far past the point of linear deformation, a displacement of 0.0344mm is demonstrated which is rather insignificant. Finally, the component also experiences a maximum strain of 1.072×10^{-3} within the same area, and a

minimum strain value of 1.237×10^{-7} around the same area as can be seen on the standard finger follower.

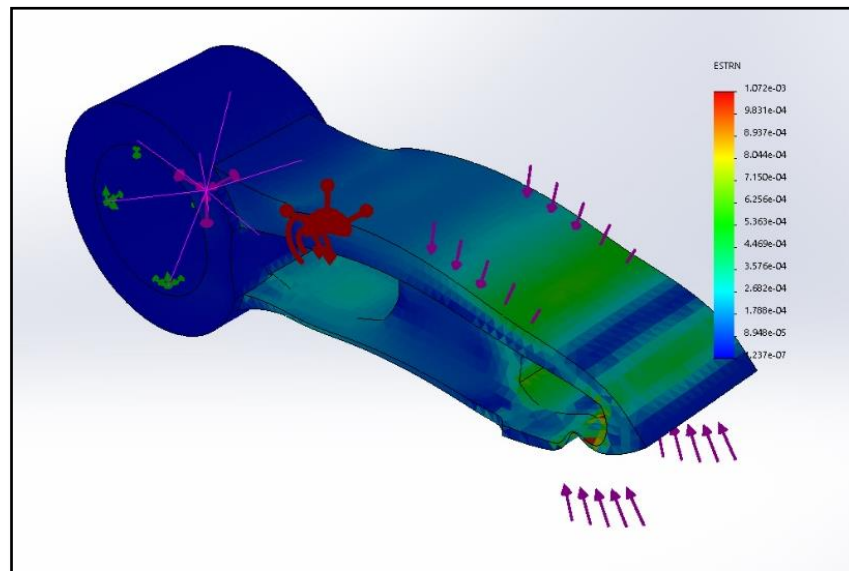


Figure 38 Strain Initial Optimization

12.2 Final Optimization

12.2.1 Methodology

Due to the initial optimization exceeding the point of yield, another variation of the component needed to be created in order to carry out a successful optimization. A slightly different approach was taken, where five smaller holes were spread out throughout the follower as opposed to the two larger holes. This was carried out, in the hopes that it would reduce the mass of the component but also maintain its structural integrity without passing the point of yield.

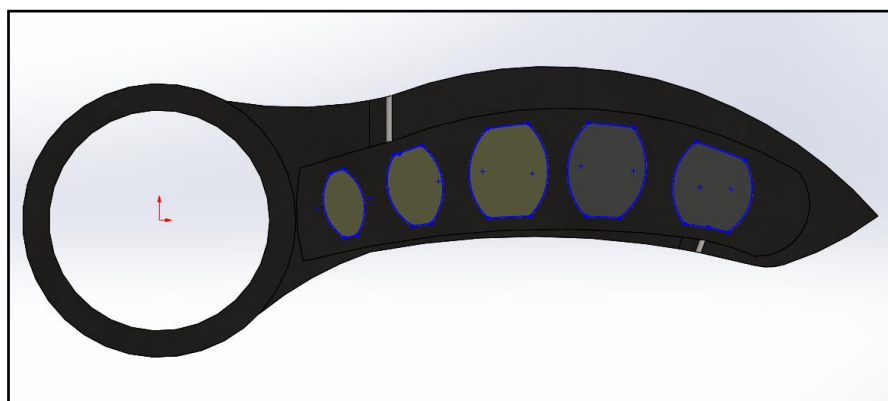


Figure 39 Sketch 8 of Final Optimization

Also, from reviewing figure 39 it is visibly clear that the inner diameter of the hole where the shaft sits has been increased by 1mm whilst keeping the dimensions of the outer diameter the same as the standard

follower reducing the thickness of the material in-between to 2mm in order to remove un-necessary material, this areas was selected on account of it encountering the least amount of displacement, strain and stresses for both the standard and initial optimization of the follower. Subsequently, fillets of 1mm were added to various edges around follower to further reduce mass.

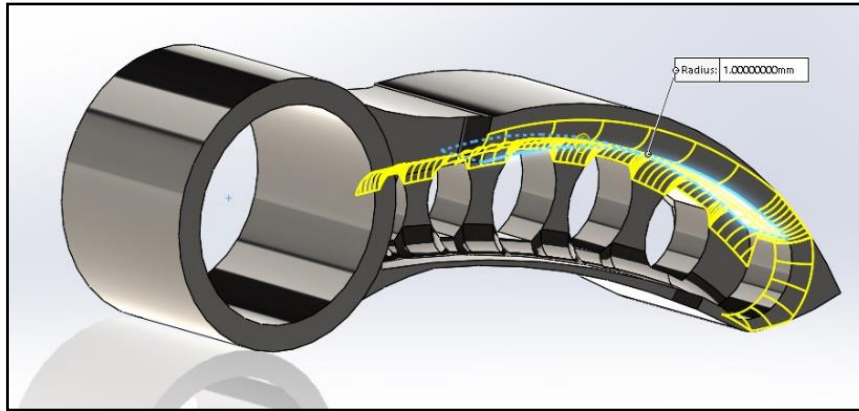


Figure 40 Final Optimization Fillet

These modifications caused a 39.33% decrease in mass going from 7.50 grams to 4.55 grams, which is a further improvement from the initial optimization which saw a decrease of 21.6%.



Figure 41 Finger Follower Final Optimization

12.2.2 Results & Analysis

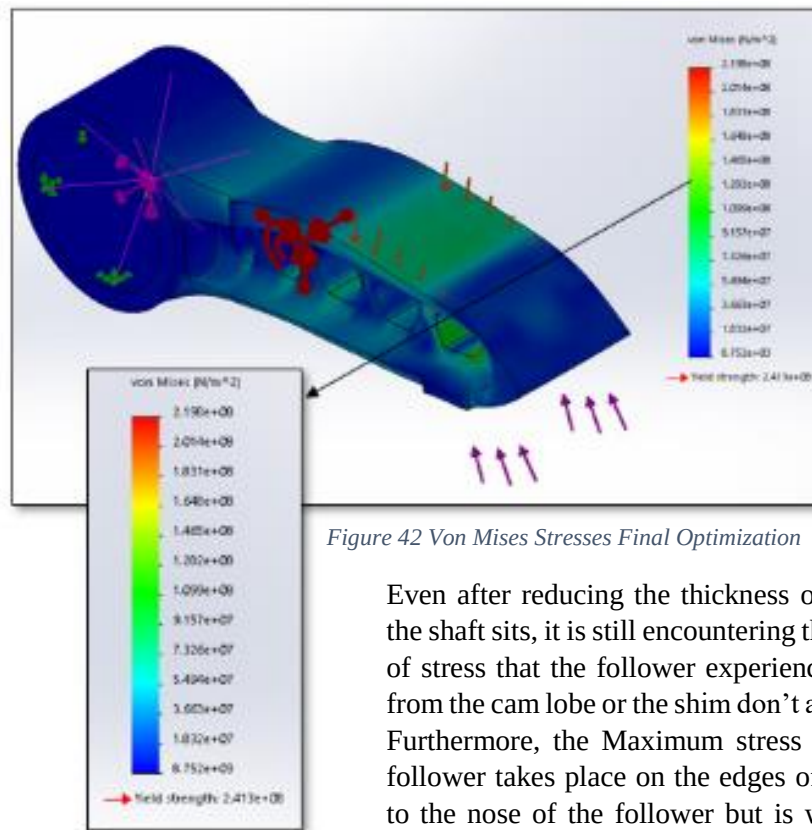


Figure 42 Von Mises Stresses Final Optimization

Even after reducing the thickness of the hole where the shaft sits, it is still encountering the lowest amount of stress that the follower experiences, as the forces from the cam lobe or the shim don't act upon this area. Furthermore, the Maximum stress exerted onto the follower takes place on the edges of the hole closest to the nose of the follower but is within 95% yield strength and has a factor of safety of 1.1.

$$\text{Safety Factor} = \frac{\text{Yield Stress}}{\text{Actual Stress}}$$

Equation 5 Safety Factor

	Standard	Final	Increase (%)		
Maximum Stress (N/m²)	6.76E+07	2.20E+08	224.99%	Yield Strength	2.41E+08
Displacement (mm)	0.0078	0.019	143.59%		

Table 5 Maximum Stress and Displacement

Table 5 shows the comparison between the standard finger follower and the final optimization. There was an increase in maximum stress exerted onto the follower of 224.99%, which may sound significant, but is still 9.11% below the point of yield meaning it is still experiencing linear deformation. In addition to this, table 5 also shows that there is a 143.19% increase in maximum displacement but is still only 0.019mm which is infinitesimal.

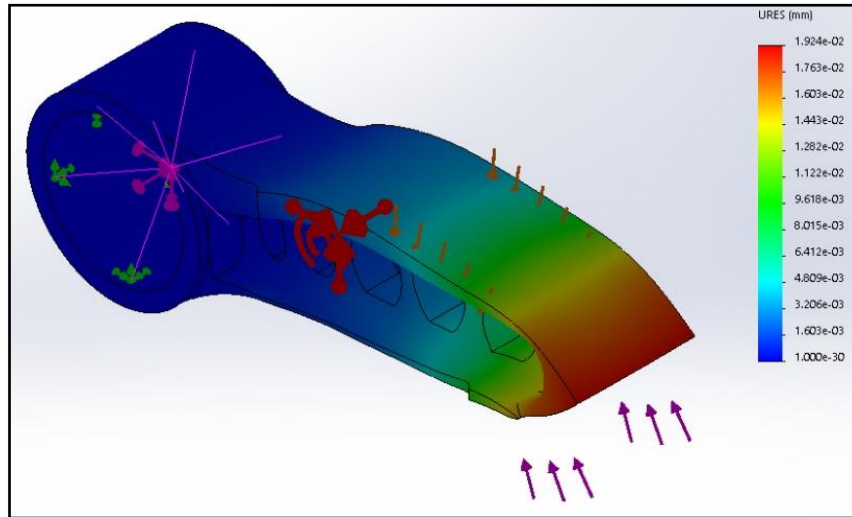


Figure 43 Displacement Final Optimization

Figure 44 shows the equivalent strain ESTRN which is defined as;

$$ESTRN = 2 \left[\frac{\varepsilon_1 + \varepsilon_2}{3} \right]^{\frac{1}{2}}$$

Equation 6 ESTRN (SolidWorks, Strain Components, 2013)

The maximum strain the follower is withstanding is 6.921×10^{-4} and is occurring around the same area where the maximum stress takes place and vice-versa with the minimum strain, which had a value of 9.253×10^{-8} .

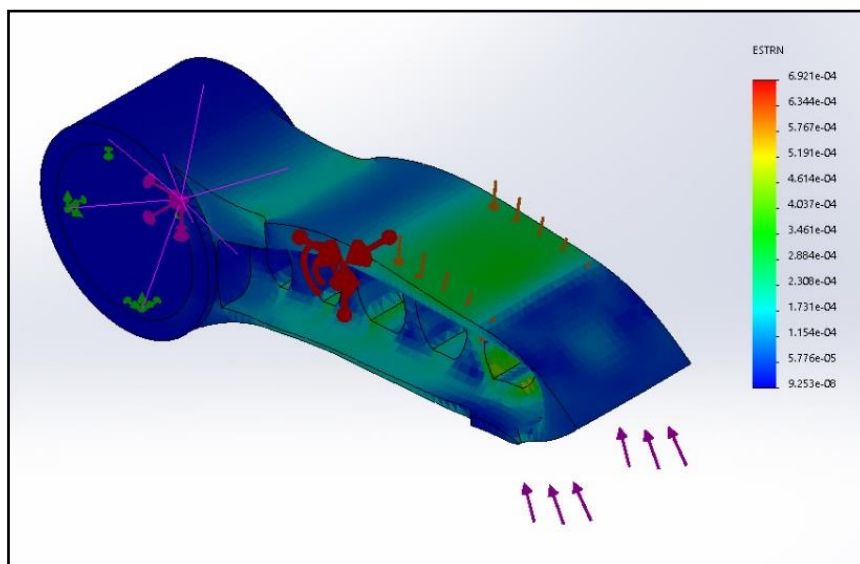


Figure 44 Strain Final Optimization

13 Conclusion

To conclude, the objective of this study was to optimize the finger follower within the Intake valve train of a Moto3 engine by a method of reducing the mass. This goal was achieved with a reduction in mass of 39.33% whilst, still maintaining the structural integrity of the component as proved within the FEA analysis.

Furthermore, this study has emphasized the influence and importance computer-aided engineering software has in the early stages of the design process, limiting potential problems, saving time, materials and money. This has been demonstrated through the many features within the SolidWorks package.

Finally, further work could consist of trialling different materials for the finger follower and trying to further reduce the redundant mates as *“having redundant mates is the equivalent of over-defining a SolidWorks model”* (SolidWorks, Redundant Mates, 2018). this therefore may affect the accuracy and validity of the results.

14 References

- (n.d.). Retrieved from <https://www.solidworks.com/>
- Autodesk. (2018). *Finite Element Analysis Software (FEA Software)*. Retrieved from Autodesk .
- Autodesk. (n.d.). *Nodes and Elements*. Retrieved from Autodesk :
http://download.autodesk.com/us/algorithmguides/mergedProjects/getting_started/Introduction_to_Algorithm/Nodes_and_Elements.htm
- Cosmol. (2017, February 21). *Finite Element Mesh Refinement*. Retrieved from Comsol Multiphysics
 Cyclopedia: <https://uk.comsol.com/multiphysics/mesh-refinement>
- English, T. (2019, November 7). *What is Finite Element Analysis & How Does It Work?* Retrieved
 from Interesting Engineering: <https://interestingengineering.com/what-is-finite-element-analysis-and-how-does-it-work>
- FIM. (2019). *FIM World Championship Grand Prix Regulations*. FIM.
- FIM. (2019). *Moto3 Junior World Championship Regulations 2019*. FIM.
- Husselman, M. (2005). *Modeling and Verification of Valve Train Dynamics in Engines*. Matieland,
 South Africa: Stellenbosch University.
- Leonghuat. (n.d.). *Importance of Yield Strength & Plastic Deformation to Civil Engineers*. Retrieved
 from Leonghuat: <https://www.leonghuat.com/articles/civil%20engineering.htm>
- Liu, Y. (2020). *Effects of Mesh Density on Finite Element Analysis*. Louisiana: SAE International .
- Motorsport Vision Racing. (2019). *HEL Motostar British Championship Technical Regulations 2019*.
 MSVR.
- NDT Resource Center. (n.d.). *Stress & Strain* . Retrieved from NDE Education : <https://www.nde-ed.org/EducationResources/CommunityCollege/Materials/Mechanical/StressStrain.htm>
- Omnexus. (2020). *Strength at Break (Tensile)*. Retrieved from Omnexus :
<https://omnexus.specialchem.com/polymer-properties/properties/strength-at-break-tensile>
- Reena Patel 1, *, G. (2019). *A Transdisciplinary Approach for Analyzing Stress*. MDPI.
- Schaeffler. (2017, November). *The Valve Train System* . Retrieved from Schaeffler Automotive
 Aftermarket: www.schaeffler.com
- Sim Scale. (n.d.). *What is FEA?* Retrieved from Sim Scale:
<https://www.simscale.com/docs/content/simwiki/fea/whatisfea.html>

Simscale . (n.d.). *What is Von Mises Stress?* Retrieved from SimScale:

<https://www.simscale.com/docs/content/simwiki/fea/what-is-von-mises-stress.html>

SolidWorks. (2008). *Understanding Motion Simulation*. Concord: Dassault Systems.

SolidWorks. (2013). *Strain Components*. Retrieved from Dassault Systemes :

https://help.solidworks.com/2013/english/SolidWorks/cworks/c_Strain_Components_output_LSA.htm

SolidWorks. (2018). *Redundant Mates*. Retrieved from Solidworks Help:

https://help.solidworks.com/2018/English/SolidWorks/motionstudies/c_Redundant_Mates_SWMotion.htm

The Engineering ToolBox. (2004). *Friction & Friction Coefficients* . Retrieved from The Engineering ToolBox: https://www.engineeringtoolbox.com/friction-coefficients-d_778.html

Woods, S. (n.d.). *A NEW PROCESS DETERMINES OPTIMAL SIZE, SHAPE AND PLACEMENT OF WEIGHT-SAVING LIGHTENING HOLES IN VEHICLES*. Retrieved from fsm direct:

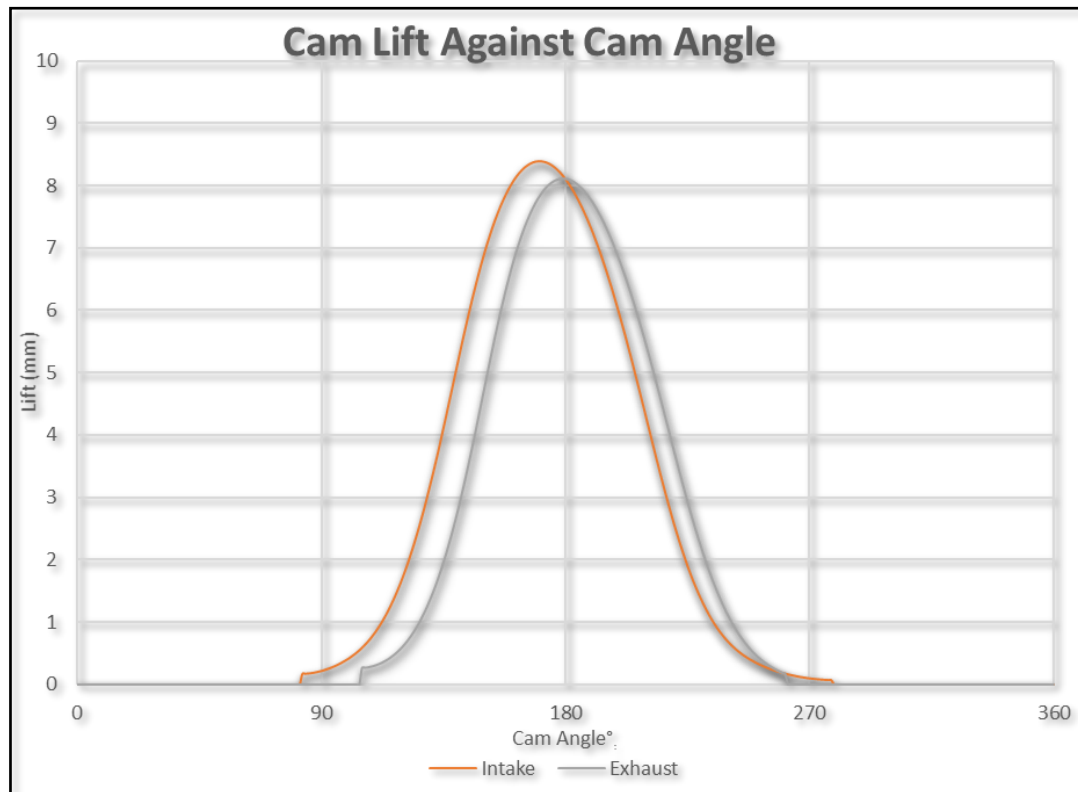
<https://fsmdirect.com/tools/automotive/450-lighten-up>

15 Appendix

15.1 Cam Profile



Figure 45 Measuring Intake Cam Profile



Graph 3 Moto3 KTM Cam Lift

15.2 RPM Limit Regulations

15.2.1 BSB MotoStar Moto3

5.3.14 Ignition/Engine Control System (ECU)

ECU is free with a maximum rev limit of 13,500rpm, this may be checked at any time during the event on a Dyno or be enforced by the compulsory fitment of approved logging device.

Figure 46 BSB Moto3 Rev Limit (Motorsport Vision Racing, 2019)

15.2.2 FIM Moto3 Junior World Championship

4. Maximum engine RPM in acceleration is controlled by the official ECU. The RPM Limiter strategy target is set at 13,500 rpm.

Figure 47 Moto3 World Junior Championship Rev Limit (FIM, 2019)

15.2.3 World Moto3

Moto3 Class Engines

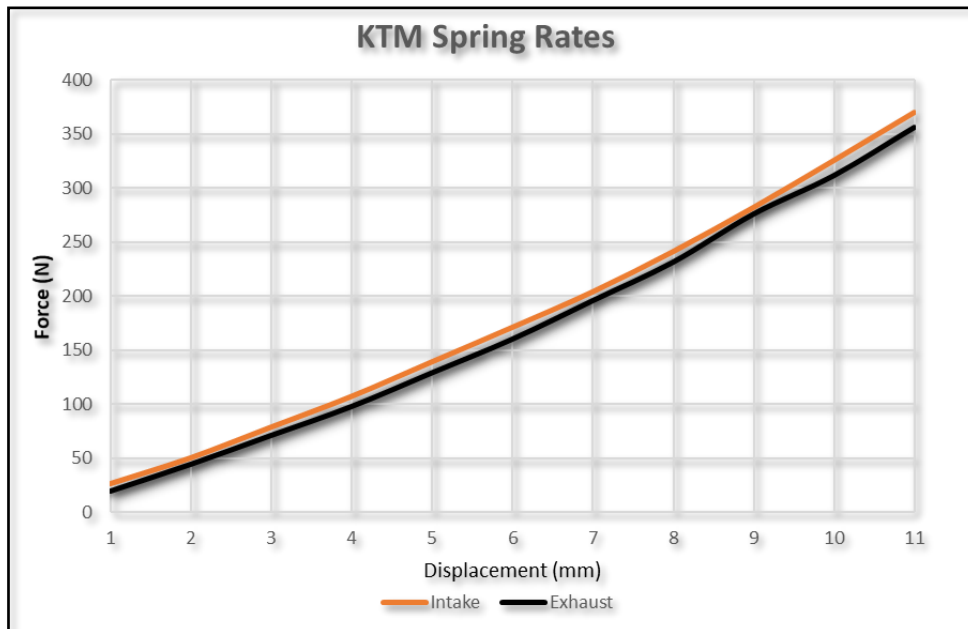
11. Maximum engine RPM in acceleration is controlled by the official ECU*. The RPM Limiter strategy target is set at 13,500 rpm.

Figure 48 World Moto3 Rev Limit (FIM, 2019)

15.3 Spring Rates



Figure 49 Measuring Intake Valve Spring Rate



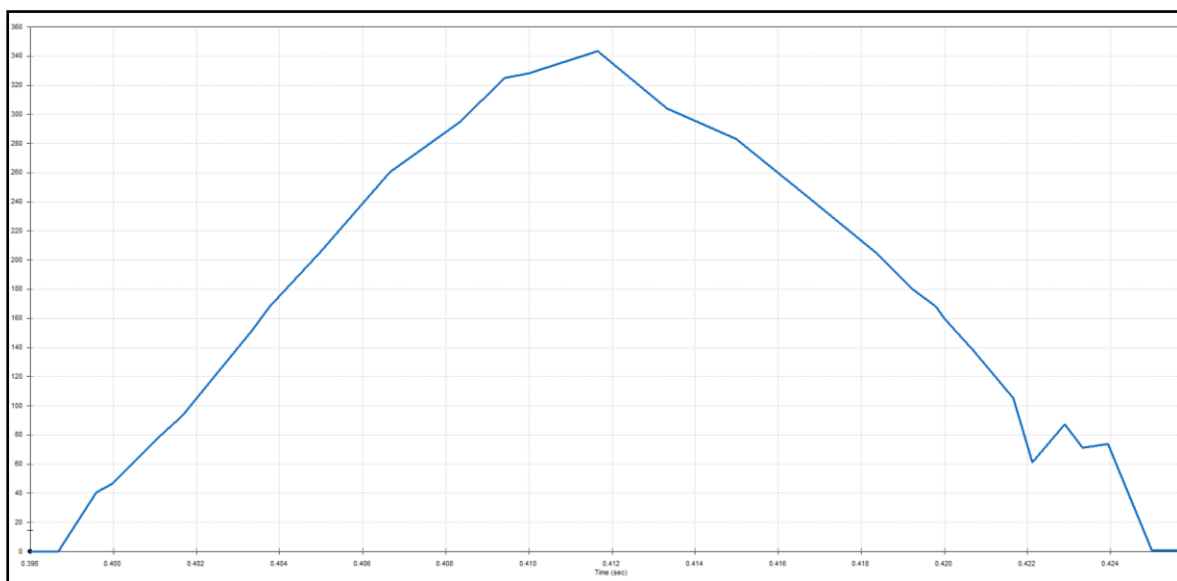
Graph 4 KTM Moto3 Spring Rates

15.4 Friction Coefficients

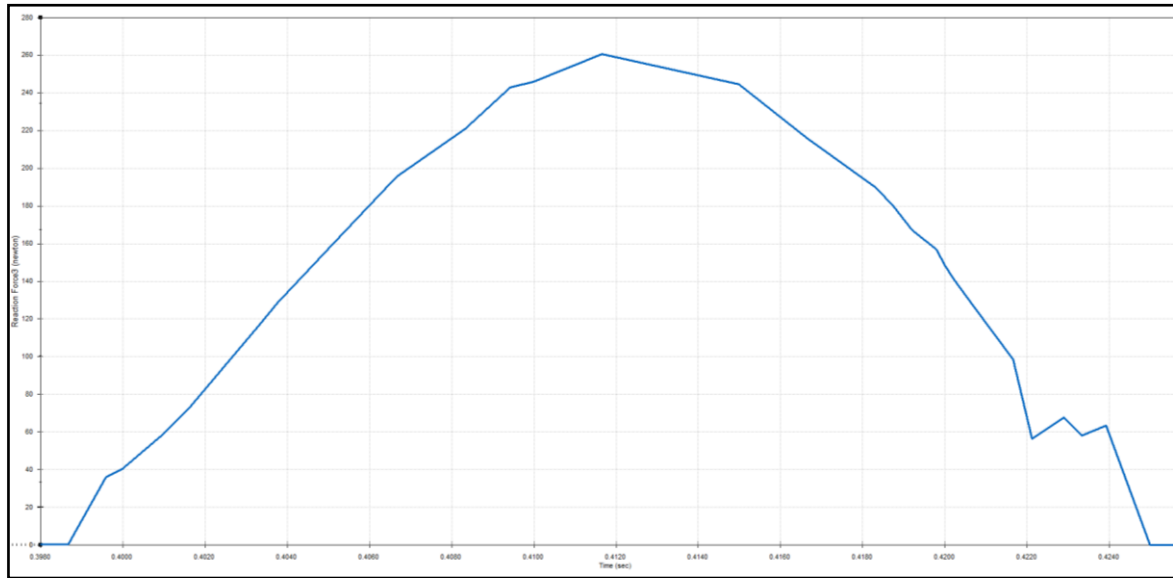
Materials and Material Combinations		Surface Conditions	Frictional Coefficient	
			Static - μ_{static} -	Sliding - $\mu_{sliding}$ -
Cast Iron	Cast Iron	Clean and Dry	1.1	0.15
Cast Iron	Cast Iron	Clean and Dry		0.15
Cast Iron	Cast Iron	Lubricated and Greasy		0.07
Cast Iron	Oak	Clean and Dry		0.49
Cast Iron	Oak	Lubricated and Greasy		0.075
Cast iron	Mild Steel	Clean and Dry	0.4	
Cast iron	Mild Steel	Clean and Dry		0.23
Cast iron	Mild Steel	Lubricated and Greasy	0.21	0.133
Steel	Steel	Clean and Dry	0.5 - 0.8	0.42
Steel	Steel	Lubricated and Greasy	0.16	
Steel	Steel	Castor oil	0.15	0.081
Steel	Steel	Steric Acid	0.005	0.029
Steel	Steel	Light mineral oil	0.23	
Steel	Steel	Lard	0.11	0.084
Steel	Steel	Graphite		0.058
Steel	Graphite	Clean and Dry	0.21	

Table 6 Friction Coefficients (The Engineering ToolBox, 2004)

15.5 Solid Body Contact Forces Graphs in SolidWorks



Graph 5 Camshaft to Follower Contact Forces



Graph 6 Finger Follower to Shim Contact Force