



University of Wales Trinity Saint David

School of Engineering

Motion Simulation & FEA

Advanced Computational Methods (M6X00408)

George Jones

P149914

(BEng) Motorsport Engineering

26.01.2020

Contents

1. Introduction	1
I. Aims.....	1
II. Objectives.....	1
Modelling Techniques.....	2
Accurate Camshaft Lobes	2
Components for Assembly.....	3
Material Properties.....	5
Assembly Methodology	6
Final Assembly	7
Motion Simulation Procedure.....	8
Initial Settings.....	8
Simulation for 6000rpm (Engine Speed).....	9
.....	10
Finite Element Analysis for Inlet Valve Closing at 6000rpm (Engine Speed)	11
Motion Loads	11
Finite Element Analysis Features in Solidworks.....	12
Von Mises Stress (N/m^2)	12
Resultant Displacement (URES)	12
Meshing.....	12
Assumptions Specific to this Study	13
Valve Study Settings.....	14
Original Valve Analysis.....	15
Iteration 1 Study Settings.....	18
Iteration 1 Analysis	19
Iteration 2 Study Settings.....	21
Iteration 2 Analysis	22
Iteration 3 Study Settings.....	24

Iteration 3 Analysis	25
Iteration 4 Study Settings.....	27
Iteration 4 Analysis	28
Comparison of Results	30
Conclusion.....	31
Computer Aided Engineering.....	31
Inlet Valve Study in Solidworks.....	31
Further Improvements.....	31
Appendix 1	32
Mesh Density Study	32
Appendix 2	34
Inclusion of Equations to the Design Phase.....	34
References	35

1. Introduction

I. Aims

- Accurately measure the Ford Duratec HE420 cylinder head assembly with attention to the specific valve train geometry and arrangement.
- Create individual parts referencing the real engine measurements and assemble in a 3D virtual environment.
- Simulate the kinematics of the real engine through motion analysis
- Investigate the loads experienced through the contact force of the inlet valve during a closing event at high engine speed and explore possible improvements to the system using the Finite Element Method.

II. Objectives

- Use suitable measuring apparatus to measure each physical component and achieve a geometric tolerance of $\pm 0.5\text{mm}$ (Camshaft Lobe Tolerance, $\pm 0.01\text{mm/}^\circ$)
- 3D modelling will be conducted in Solidworks where a final assembly will be produced using suitable construction and mating techniques.
- The final assembly will be introduced to the Solidworks Motion Analysis tool where a kinematic simulation can be carried out at 6000rpm engine speed to produce motion loads for further analysis.
- The loads produced from the Motion Study will be imported to the relevant component where Solidworks Finite Element Analysis (FEA) Simulation will support a design improvement investigation to reduce the mass of the inlet valve.

2. Modelling Techniques

I. Accurate Camshaft Lobes

The optical dividing head was used to generate the lift profiles for both inlet and exhaust camshaft lobes per degree. The resulting data (*figure 1*) was processed through Excel and imported as curves to form an accurate camshaft in Solidworks.

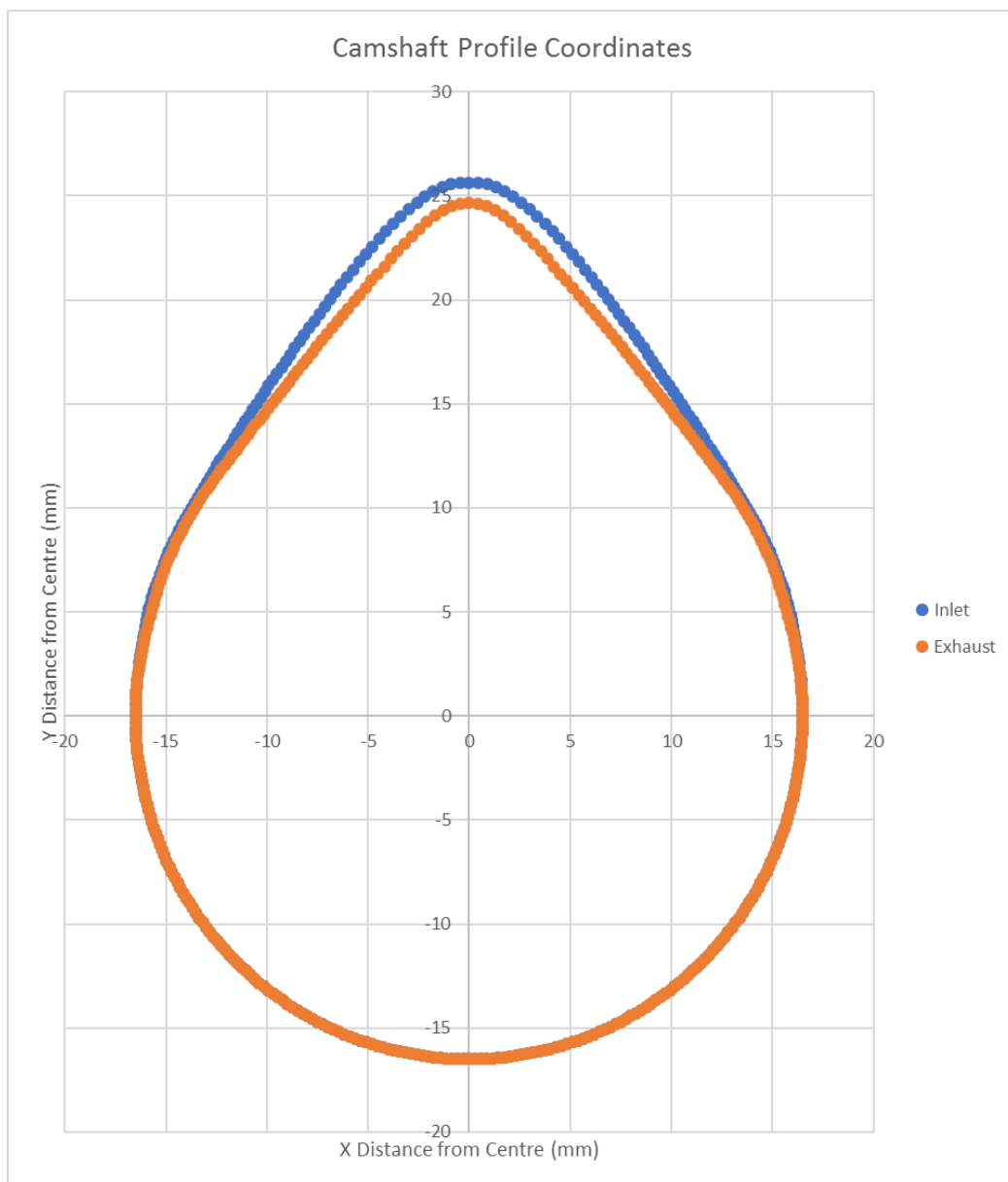


Figure 1 - Duratec HE420 Camshaft Profile Coordinates

II. Components for Assembly

Each component was measured using a Vernier Caliper, steel rule and micrometre. As the valve train components are symmetrical and circular, the preferred modelling technique is a revolved boss. This also improves the efficiency of producing iterations later in the investigation.

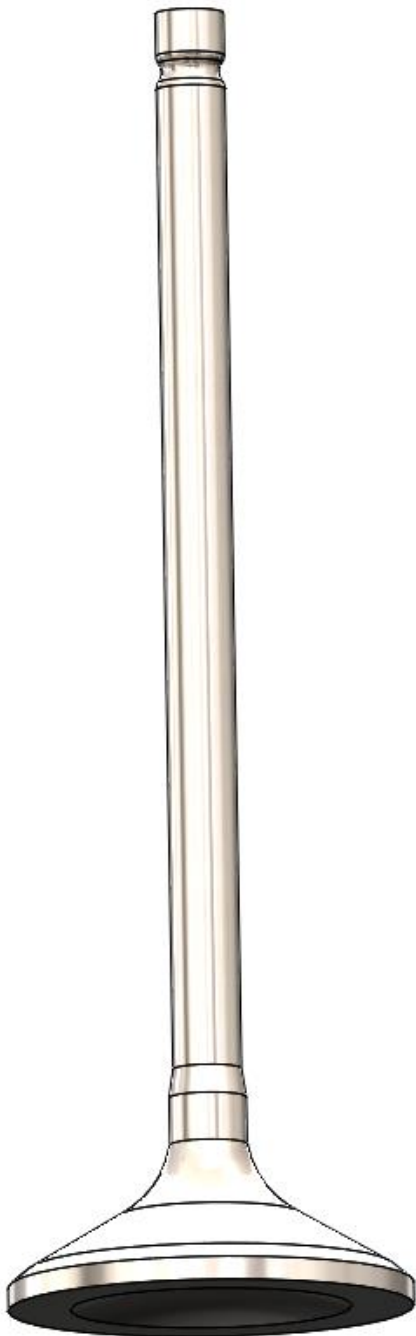


Figure 4 - Inlet Valve in Solidworks

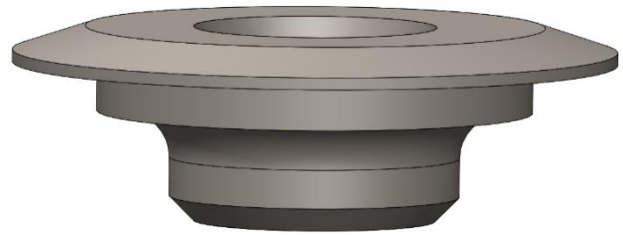


Figure 3 - Valve Spring Cap in Solidworks



Figure 5 - Valve Spring Collet in Solidworks

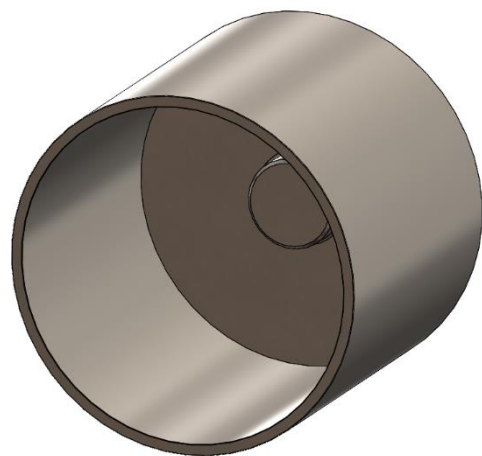


Figure 2 - Valve Shim (bucket type) in Solidworks

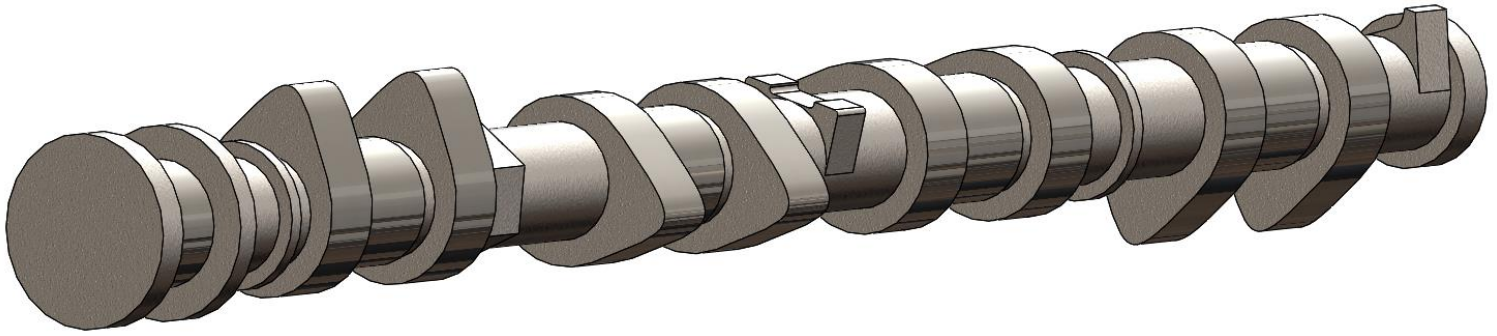


Figure 7 - Inlet Camshaft in Solidworks

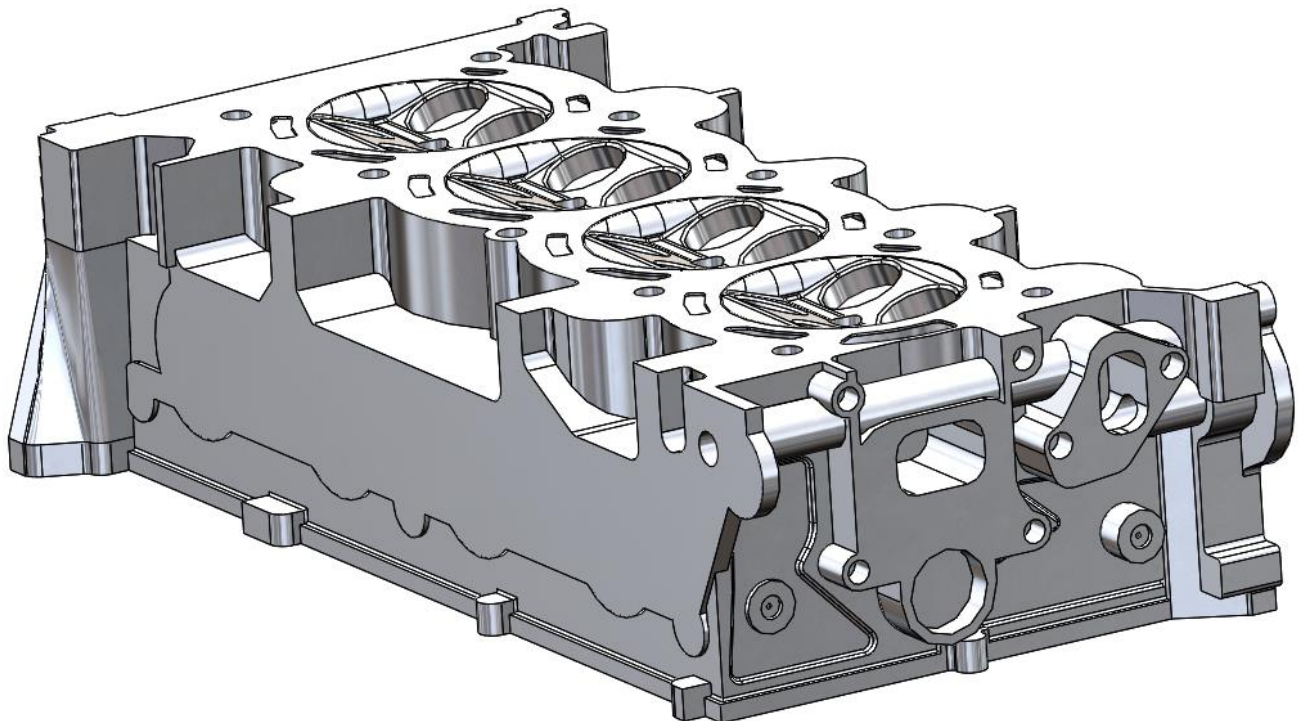


Figure 6 - Duratec Cylinder Head in Solidworks

The intricacies of the real cylinder head required significantly more sketches and features to replicate in Solidworks. The valve offset from the vertical was measured as 20° . (Brezina, 1997) suggests a more efficient method for modelling would be to use a Coordinate Measuring Machine (CMM) or X-ray to produce a point cloud for processing into a surface or solid.

III. Material Properties

The precise material compositions are not known for the real components, so the following assumptions have been made.

- All valve train parts are considered as Alloy Steel (*table 1*).
- Cylinder head is assumed to be 6061 Aluminium Alloy. However, the valve stem seats would be made from a different material or surface finish as discussed in *section 7*.
- Camshaft is assumed to be Cast Alloy Steel.

Table 1 - Material Properties for Alloy Steel in Solidworks

Property	Value	Unit
Mass Density	7700	Kg/m ³
Tensile Strength	7.238 x10 ⁸	N/m ²
Yield Strength	6.204 x10 ⁸	N/m ²
Elastic Modulus	2.1 x10 ¹¹	N/m ²
Shear Modulus	7.9 x10 ¹⁰	N/m ²

Table 2 - Mass Comparison between Real and Solidworks Parts

Component	Real Mass (g)	Solidworks Mass (g)
Inlet Valve	47.2	49.55
Inlet Shim Bucket	33	32.47
Collets (2)	1.8	1.62
Valve Spring Cap	9.8	9.41
Valve Spring	34.7	N/A

The mass for the total valve train (minus the spring) in Solidworks is *93.05g*. The ability to manually override the mass properties has not been used as there is only a 1.3% increase over that of the real parts when combined.

3. Assembly Methodology

The following assembly conditions have been considered to eliminate *all* potential redundant mates in the motion analysis.

- Valve, cap & collets assembled as locked mates as these can be assumed to not move in relation to each other.
- Valve to Bucket shim assumed as locked mates.
- Valve to valve guide in cylinder head is mated concentrically through the “*use for positioning*” function. A coincident mate is applied between the valve head and valve seat as an initial state in the motion simulation.
- Camshaft to cylinder head (camshaft bearing carrier) is mated coincidently with contacting edges. This defines concentricity and correct face contact.
- Camshaft lobe rotated to vertical (fully closed) ahead of motion simulation.

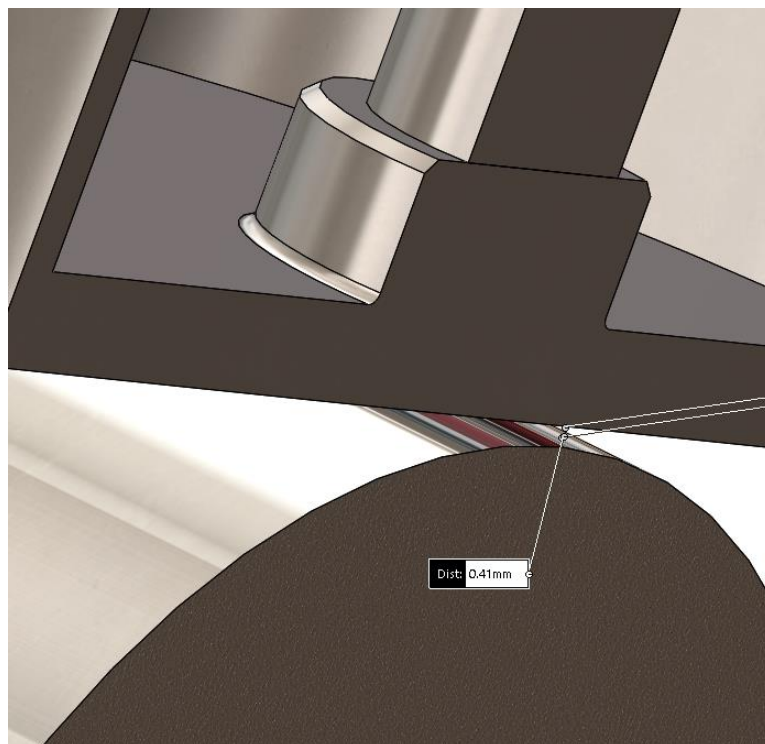


Figure 8 - Camshaft to Valve Shim Clearance (0.41mm)

Figure 8 illustrates the valve shim clearance, adjusted to match that of the real cylinder head.

I. Final Assembly

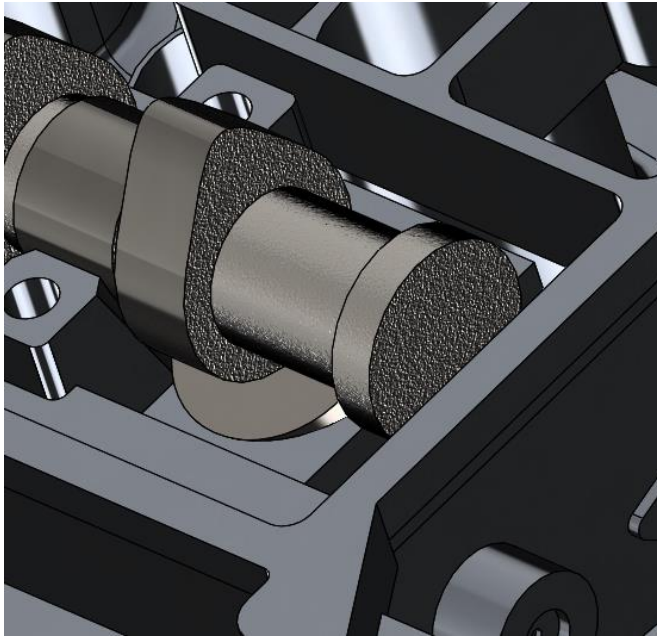


Figure 9 - Camshaft Assembly



Figure 10 - Valve Train Assembly

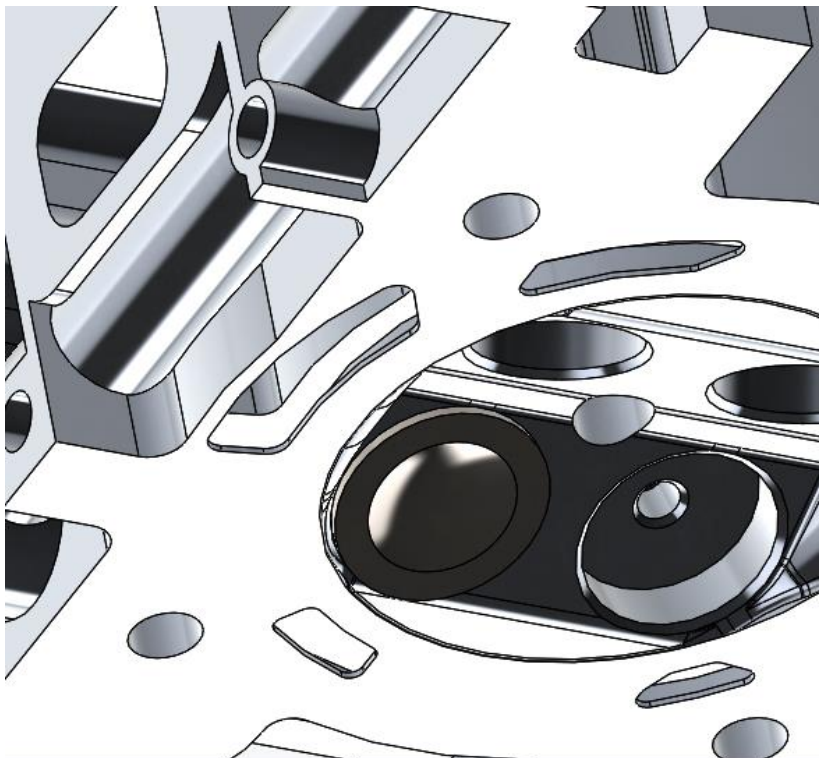


Figure 11 - Inlet Valve in Closed Position (against valve seat)

4. Motion Simulation Procedure

I. Initial Settings

These settings were implemented to the model to verify the assembly interactions.

- Gravity assigned to top plane in correct direction for the study ($9.806.65 \text{ m/s}^2$)
- Rotary motor applied to the camshaft @ 500rpm (half engine speed)
- Solid body contact groups used for interactions between camshaft, shim and cylinder head (steel greasy).
- Second Solid body contact groups applied for valve to valve seat interactions (steel & aluminium dry, respectively.)
- Valve seat mate removed from motion study ahead of calculations.

The actual spring rate was found by taking force measurements at 1mm intervals of compression and plotting the curve to find the rate coefficient.

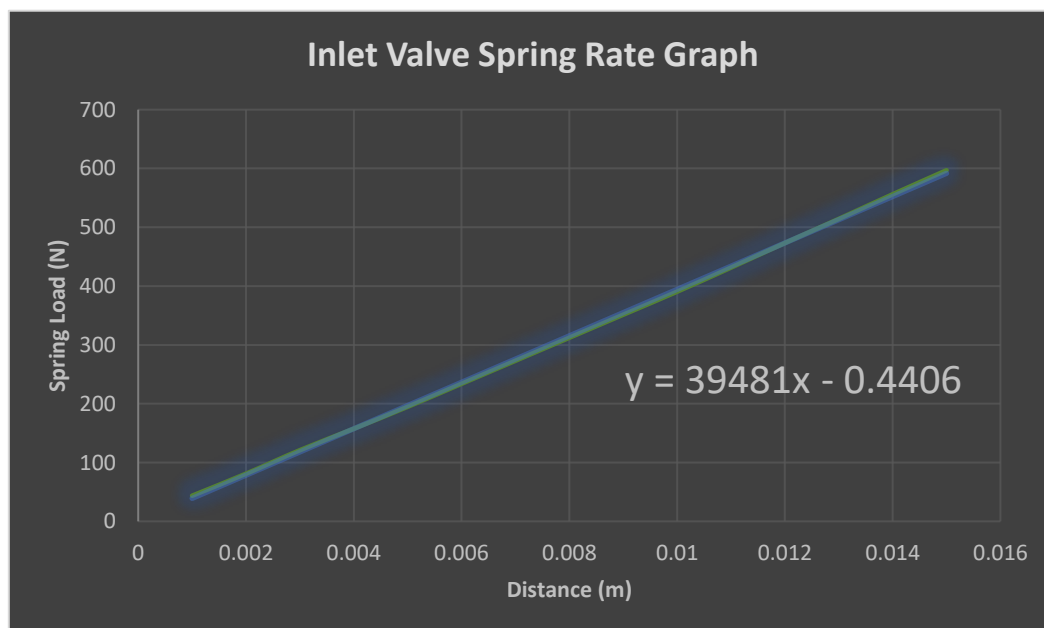


Figure 12 - Real Spring Stiffness Characteristics

Figure 12 expresses the valve spring to be acting in a linear fashion and produces a spring rate of approximately 39.5 N/mm .

This data was input to the spring component of the motion study along with the correct geometric restraints.

The simulation time was adjusted for two complete camshaft rotations to check stability of the model. *Figure 13* proves a sensible valve profile achieving 8mm @ maximum lift and is repeatable.

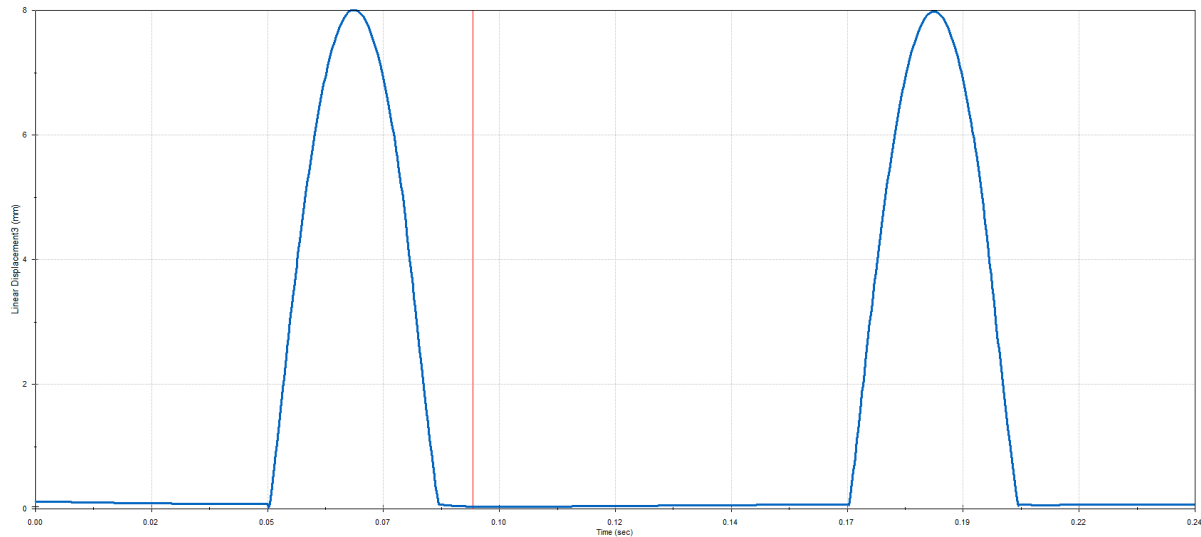


Figure 13 - Initial Valve Lift Profile @ 1000rpm Engine Speed

II. Simulation for 6000rpm (Engine Speed)

The camshaft motor was increased from 500rpm to 3000rpm and the simulation time adjusted accordingly. *Figure 14* matches the profile from the initial conditions proving the mechanical interactions are consistent. This test is necessary to validate the valve spring rates and masses through a change in acceleration. The phenomenon of valve bounce or float could occur at higher engine speeds if these values are inconsistent with reality as explained by (Tani *et al.*, 1993) and (Liu *et al.*, 1995).

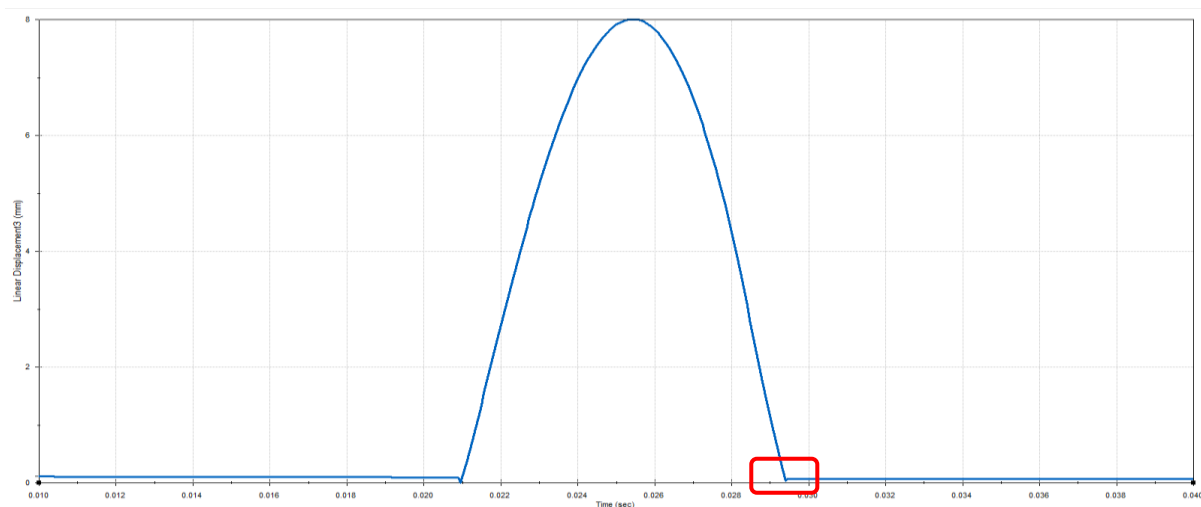


Figure 14 - Final Valve Lift Profile for 1 cycle @ 6000rpm Engine Speed

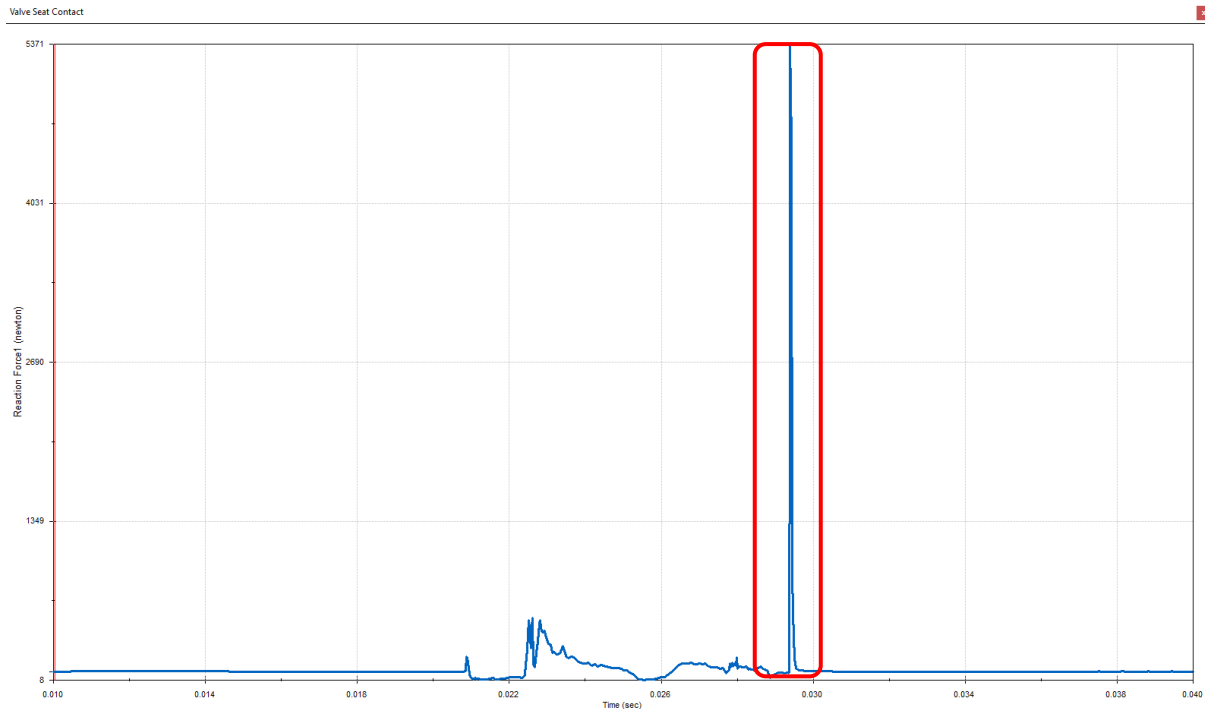


Figure 15 - Contact Force between Valve & Seat @ 6000rpm Engine Speed

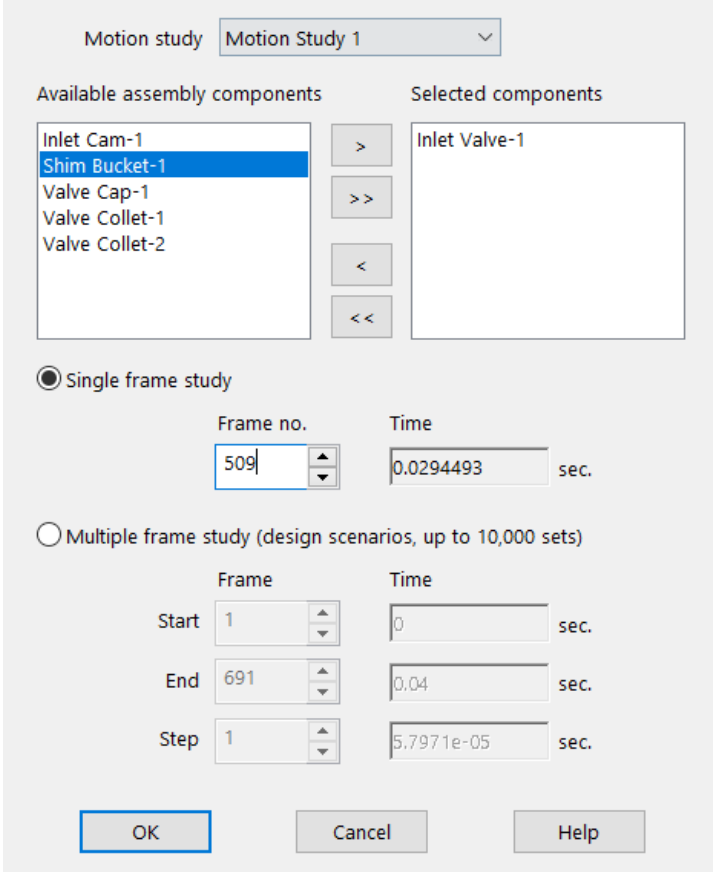
The contact force at the seating faces between the cylinder head and inlet valve can be seen in *figure 15*. A peak contact force of 5371 N occurs at the same instance as the valve displacement reaches zero in *figure 14*. However, there are anomalies in the plot where Solidworks records a rise in force where there is no obvious contact between the two referenced faces.

This implies there is an error in the way that Solidworks is applying the referenced faces or there is a fault in the motion study input parameters (Vavro *et al.*, 2017). The first small peak in the plot could be due to the cam hitting the bucket on the constant velocity ramp of the lobe causing a fraction of valve bounce before it begins to fully open. Various view orientations were applied to the study as well as using the section view function to attempt a diagnosis for the issue, but no plausible explanation was found. These anomalies can be ignored on this occasion as they occur prior to the valve closing event.

The plot is exported to as a .CSV file to locate the exact frame times for the peak contact force in excel.

5. Finite Element Analysis for Inlet Valve Closing at 6000rpm (Engine Speed)

I. Motion Loads



The dialog box is titled "Motion study" with a dropdown menu set to "Motion Study 1". It is divided into two main sections: "Available assembly components" and "Selected components".

Available assembly components:

- Inlet Cam-1
- Shim Bucket-1 (highlighted)
- Valve Cap-1
- Valve Collet-1
- Valve Collet-2

Selected components:

- Inlet Valve-1

Navigation buttons between the lists include: >, >>, <, and <<.

Study Type:

- ☒ Single frame study
 - Frame no.: 509
 - Time: 0.0294493 sec.
- ☐ Multiple frame study (design scenarios, up to 10,000 sets)
 - Frame
 - Start: 1
 - End: 691
 - Step: 1
 - Time
 - Start: 0 sec.
 - End: 0.04 sec.
 - Step: 5.7971e-05 sec.

Buttons at the bottom: OK, Cancel, and Help.

Figure 16 – Solidworks Import Motion Loads Dialog Box

Figure 16 creates an FEA study over the specified frames within the selected part file(s) using the loads produced in the motion simulation. As the valve is assumed to be geometrically constrained by the cylinder head and the valve spring via the collets simultaneously when peak force occurs, a single frame study is best suited.

II. Finite Element Analysis Features in Solidworks

The following studies will compare the Finite Element results in terms of Von Mises Stress and Resultant Displacement. The context for these terms in Solidworks is detailed below.

a) Von Mises Stress (N/m²)

Solidworks combines the individual principle and shear stresses into one equation (Von Mises). This yield criterion is widely used in material analysis (Gatto and Jammalamadaka, 2007).

$$\sigma = \sqrt{0.5 \left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 \right]} + \sqrt{+3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

Where σ = Principle Stress, τ = Shear Stress

Equation 1 - Von Mises Stress

b) Resultant Displacement (URES)

Solidworks calculates displacement through the X, Y and Z axis. The following equation is used to combine all displacements into a single value,

$$URES = \sqrt{\Delta x^2 + \Delta y^2 + \Delta z^2}$$

Where x, y and z = displacement (mm)

Equation 2 - Resultant Displacement

c) Meshing

For this study, Solidworks will calculate a global element size for the model with respect to the geometry of the part (unless the design criterion is close to breach). This includes the number of nodes and elements.

Solidworks uses tetrahedral solid elements for 3D studies.

The effects of changing mesh properties can be seen in *Appendix 1*.

III. Assumptions Specific to this Study

- The primary outcome for this study is to compare the stresses and displacements across the valve head during the valve closing event at a high engine speed. The stresses through the valve stem will be neglected.
- There would be an amount of pressure being exerted on the face of the valve from the initial compression of the charge. This will also be neglected through the study as it is likely to be small in comparison to contact force.
- The maximum stress exerted to the valve face through combustion pressure will not be investigated.
- One single inlet valve is to be tested. This is to improve computational efficiency as the study is investigating a force that has no real influence from the remainder of the valve train.
- The increase in temperature at the time of closing will not be considered as no sensible values can be produced to simulate this component of the event.

IV. Valve Study Settings

The imported motion loads include a gravity and centrifugal force acting on the valve, relative to the motion study in direction. To simplify the study, it is assumed that the valve collet faces are fixed geometries as shown in *Figure 17*. An additional roller/slider fixture is applied to the valve stem to replicate a valve stem guide.

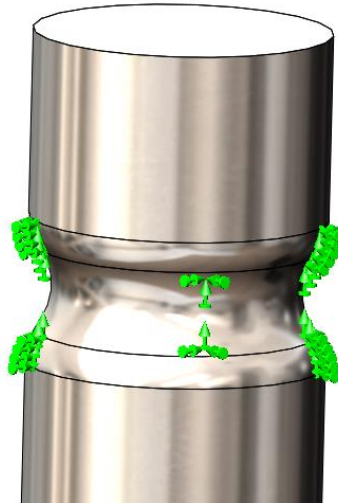


Figure 17 - Fixed Geometry around Collet Faces

The peak force (5371 N) acting on the seat face is included manually (*figure 18*) and a standard mesh is used. Inertial relief is deselected for this study.

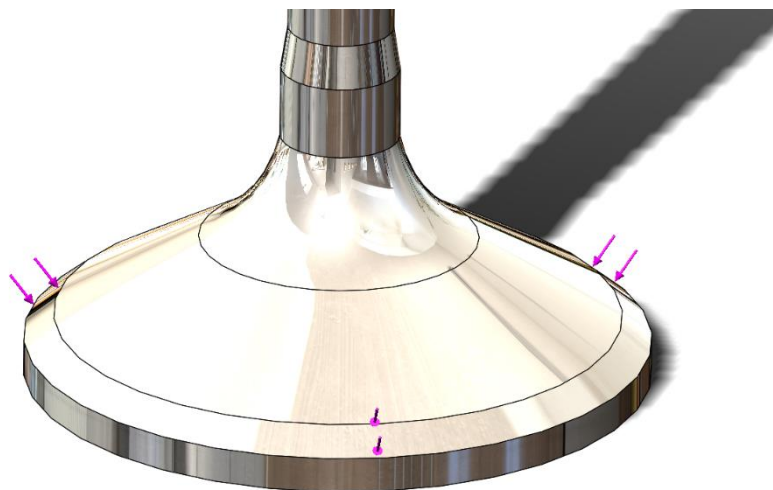


Figure 18 - Force Applied to the Valve Seating Face

V. Original Valve Analysis

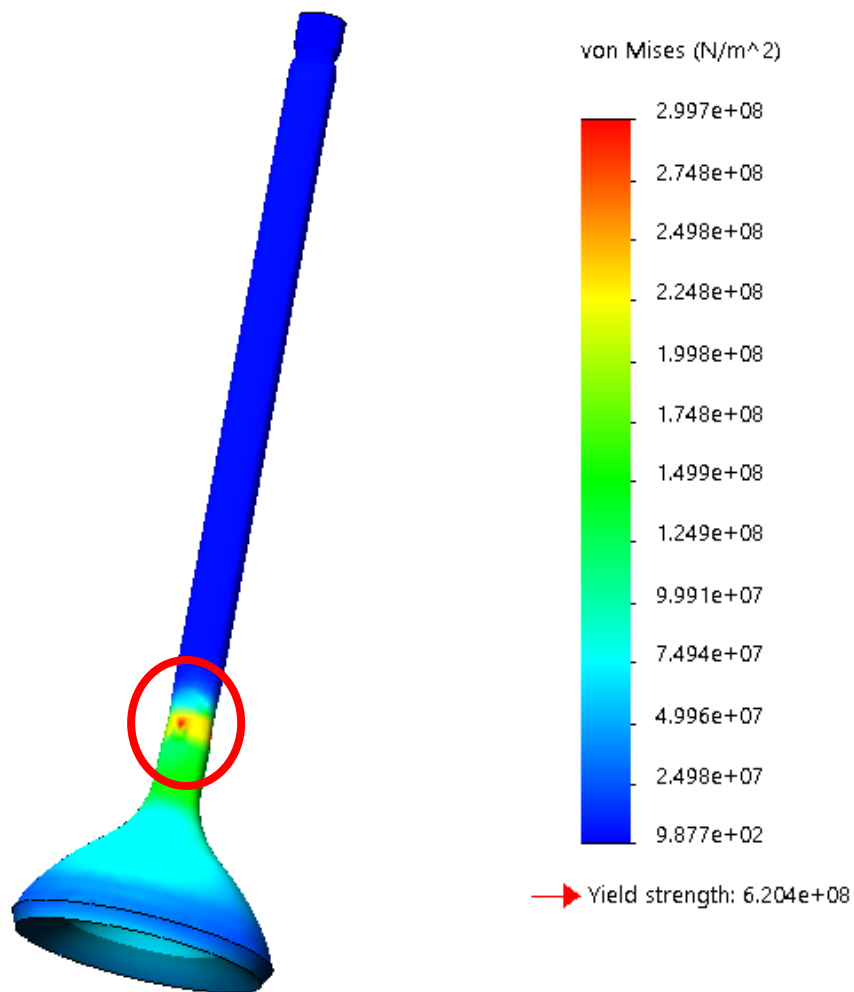


Figure 19 - Original Valve Von Mises Stress Plot

Figure 19 suggests maximum stress to be 48% of yield strength and located in a concentrated point on the valve stem, below the guide. This is due to the method of constraints being used through this study and offers a worst-case scenario for the stresses produced from the seating event.

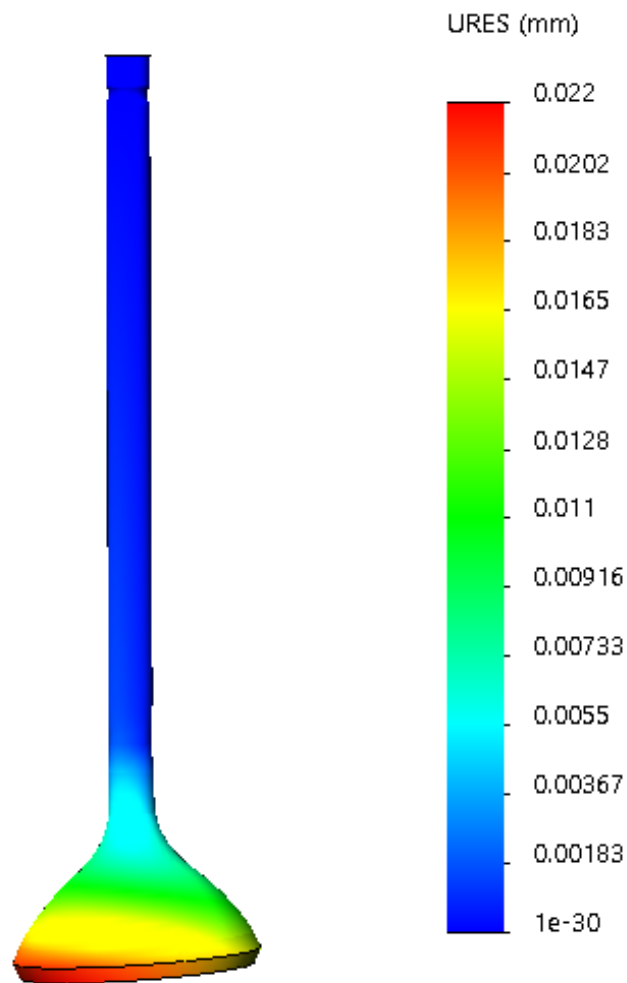


Figure 20 - Original Valve Resultant Displacement (including imported gravity and centrifugal forces) – Deformation scale is 519.991

The exaggerated resultant displacement reflects an unusual trait across the valve head. Through further analysis it was found that the imported gravity and centrifugal loads were causing the valve head to deform in an unusual manner. In theory, the deformation should occur vertically through the valve as the conical seat in the cylinder head would limit the displacement in any other direction.

There are various techniques for constraining the component to match those of the real conditions in Solidworks. The restraints that have been selected in this study are resulting in the unequal displacement phenomenon in *figure 20*. There is an option for virtual walls to be included to the model, but this only applies to flat contacting faces or by use of planes. Neither options are applicable to this chamfered valve seat. Further considerations to refine this study are explained in *section 7*.

Gravity and centrifugal forces from the motion study are responsible for an increase in maximum displacement of 30% and maximum stress of 5.2%.

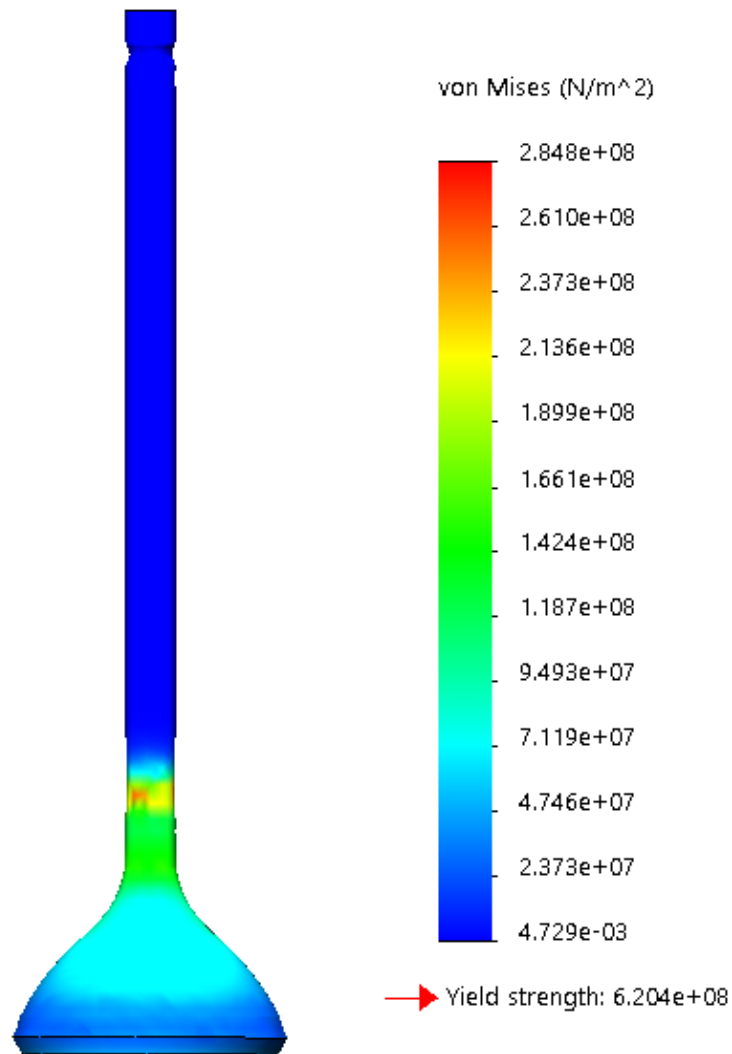


Figure 21 - Original Valve Von Mises Stress Plot (Corrected for Simplified FEA Study)

Figure 21 demonstrates how removing the imported loads has a more realistic influence on the deformation characteristics whilst the valve is in its seated position as well as maintaining a similar stress distribution. Therefore, the gravity and centrifugal forces will be neglected from this point and only the contact forces produced in the motion study are to be considered.

The target for the remainder of the design study will be no more than 90% of yield to ensure the material displacement is within elastic region (Ramberg and Osgood, 1943). A hypothetical maximum resultant displacement of 0.03mm will be considered through this investigation.

VI. Iteration 1 Study Settings

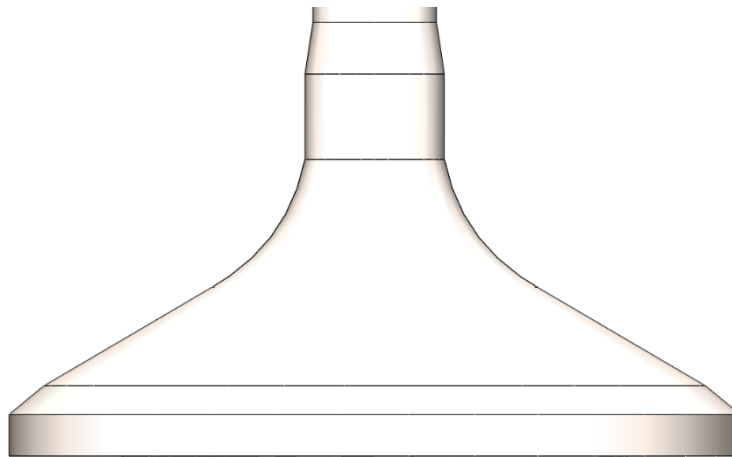


Figure 22 - Original Valve Side Profile

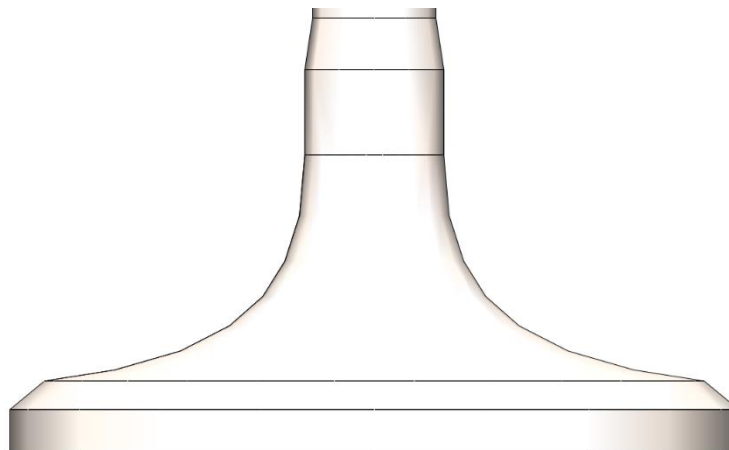


Figure 23 - Iteration 1 Valve Side Profile (aggressive port side spline)

The port side of the valve has been reprofiled from the standard geometry to reduce mass. A motion simulation was then produced to create updated contact force data for the FEA study.

Table 3 - Iteration 1 Parameters

Iteration 1 (Port Side Spline Profile)		
Original Mass	New Mass	Percentage Difference
49.55	41.97	-15.3%
Original Contact Force	New Contact Force	Percentage Difference
5371 N	4600 N	-14.4%

VII. Iteration 1 Analysis

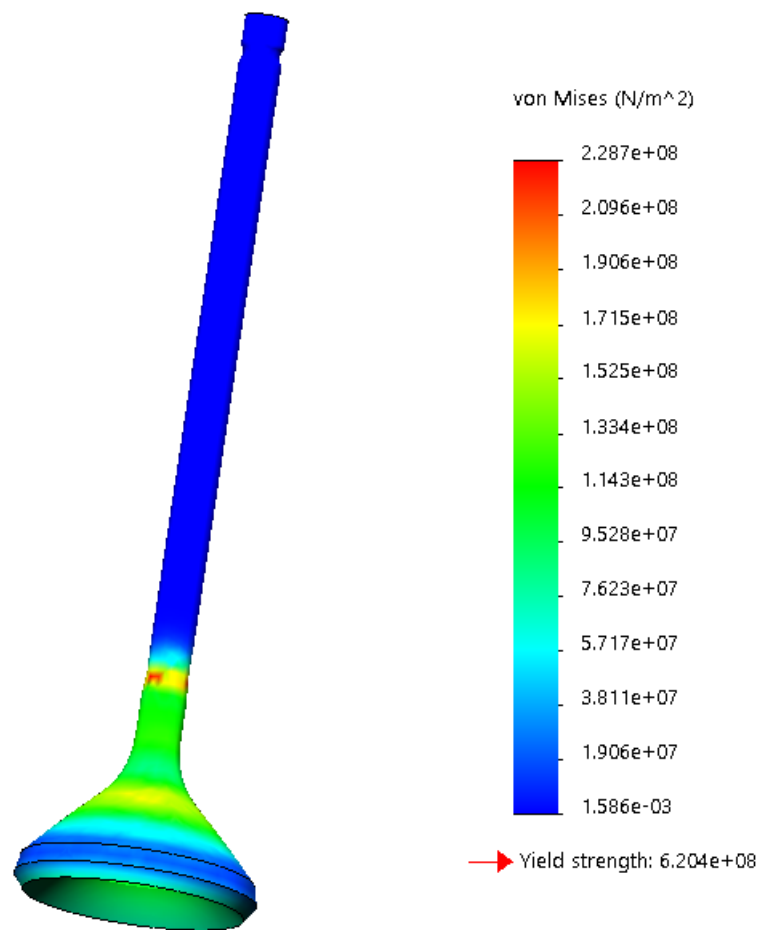


Figure 24 - Iteration 1 Von Mises Stress Plot

The reduction in mass from the more aggressive valve profile translates to a decline in peak stress by 25% when compared with the original valve. This iteration reaches 36.8% of the materials yield strength.

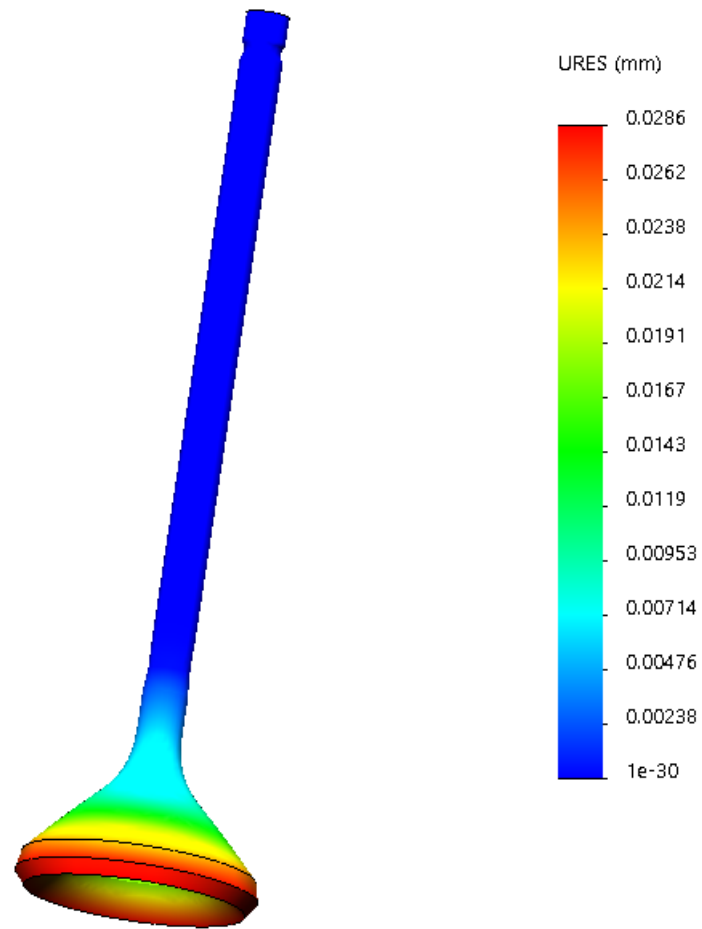


Figure 25 - Iteration 1 Resultant Displacement Plot - deformation scale is 376.606

An increase in displacement can be seen in *figure 25*. Reducing the material between the valve stem and the head influences the resistance to deformation in the region (Zienkiewicz, Taylor and Zhu, 2013). A 30% increase over the standard valve stretch is present but as the total deformation is below the theoretical maximum of $0.03mm$ it is considered safe.

VIII. Iteration 2 Study Settings

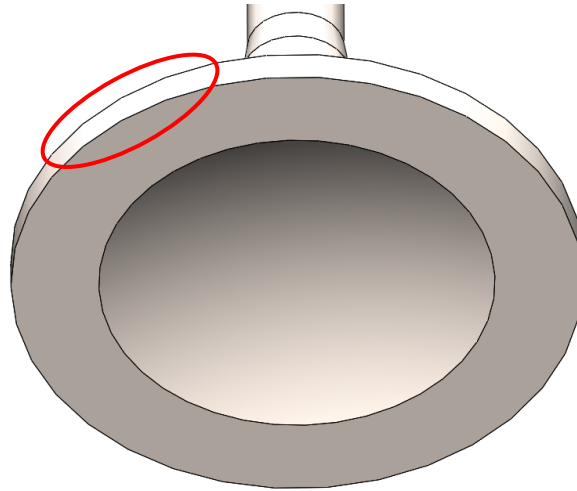


Figure 26 - Original Valve Head

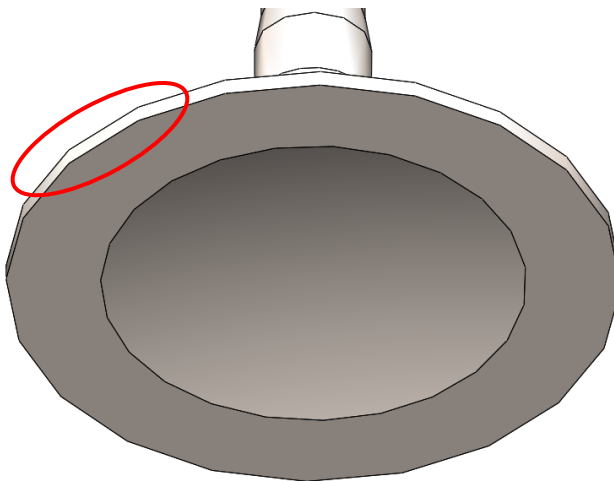


Figure 27 - Iteration 2 Valve Head

This iteration reverts to the original valve, with a 50% reduction in head height as detailed above.

Table 4 - Iteration 2 Parameters

Iteration 1 (Modified Head Profile)		
Original Mass	New Mass	Percentage Difference
49.55	44.68	-9.8%
Original Contact Force	New Contact Force	Percentage Difference
5371 N	5072 N	-5.6%

IX. Iteration 2 Analysis

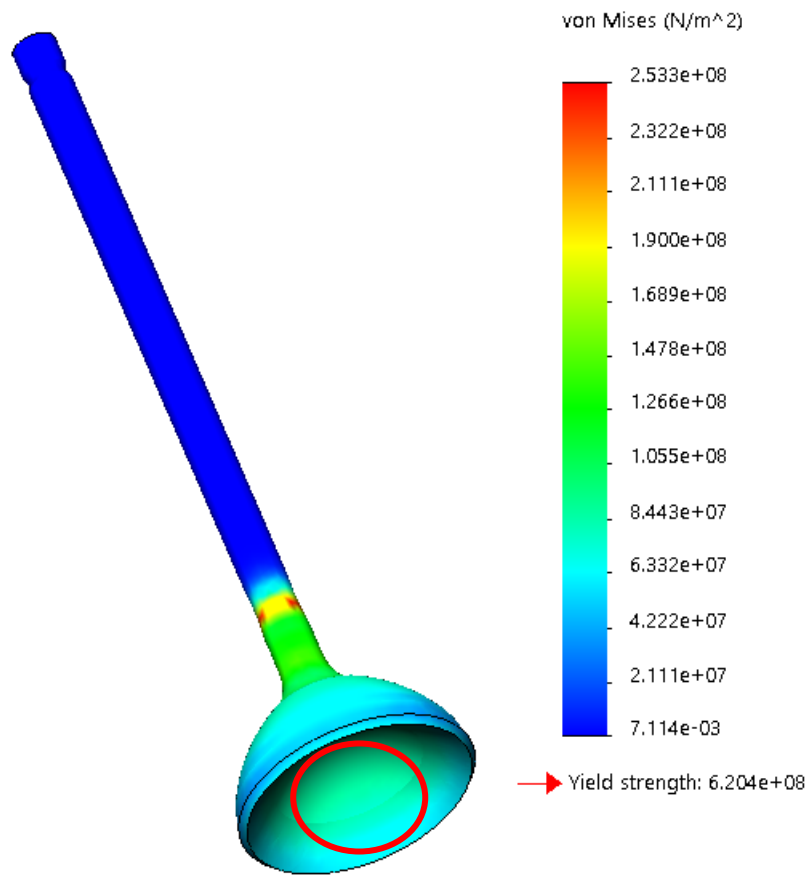


Figure 28 - Iteration 2 Von Mises Stress Plot

The effect on removing mass from the valve face causes a local decrease to stress over that found in Iteration 1 (highlighted in *figure 28*). The total stress experienced at the valve stem is between those from the original valve and the 1st iteration.

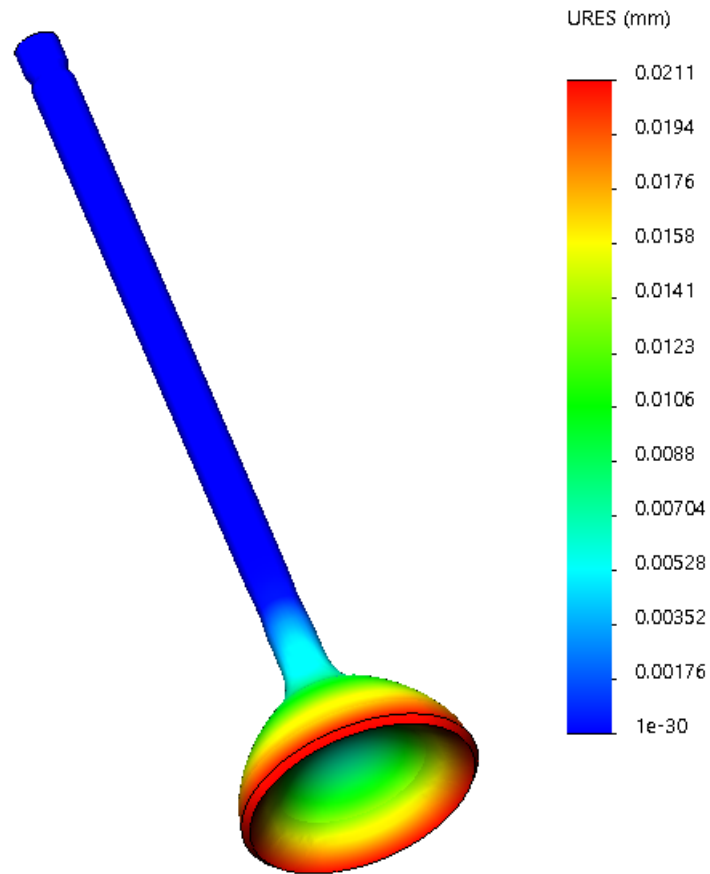


Figure 29 - Iteration 2 Resultant Displacement Plot - Deformation Scale is 501.948

Similarly, the deflection produced in this iteration is also between those found in the original valve and the first iteration. The deformation is more concentrated around the valve seating face than the previous studies but that is anticipated with the loss of material beneath the contacting surface.

X. Iteration 3 Study Settings

The material was changed from Alloy steel to Titanium (Ti-6Al-4V), reverting to the standard valve geometry.

Table 5 - Ti-6Al-4V Material Properties in Solidworks

Property	Value	Unit
Mass Density	4428.784	Kg/m ³
Tensile Strength	10.5 x10 ⁸	N/m ²
Yield Strength	8.273 x10 ⁸	N/m ²
Elastic Modulus	1.05 x10 ¹¹	N/m ²

This Titanium composition is approximately 40% lighter than the assumed original valve material (tables 5 & 6) but has approximately half of the elastic modulus.

Table 6 - Iteration 3 Parameters

Iteration 3 (Titanium - Standard Profile)		
Original Mass	New Mass	Percentage Difference
49.55	28.5g	-42.5%
Original Contact Force	New Contact Force	Percentage Difference
5371 N	3124 N	-41.8%

XI. Iteration 3 Analysis

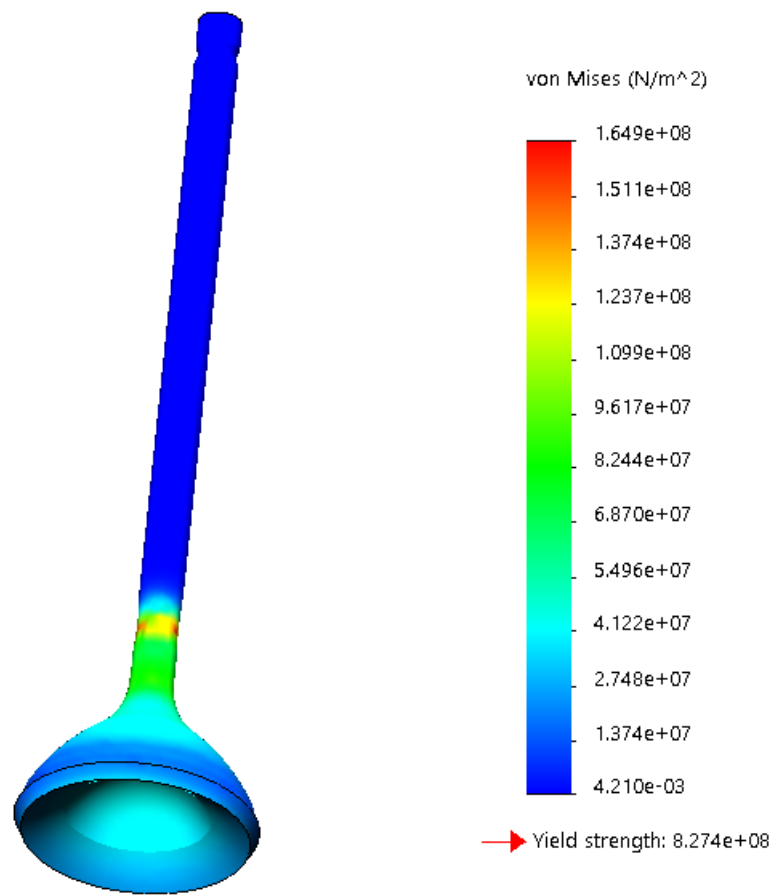


Figure 30 - Iteration 3 Von Mises Stress Plot

As expected, the Titanium valve experiences a reduction in stress overall. This is due to a drop in contact force being applied through the reduced density of the material. The increase in tensile strength is also responsible for the reduction in stress relative to its yield point.

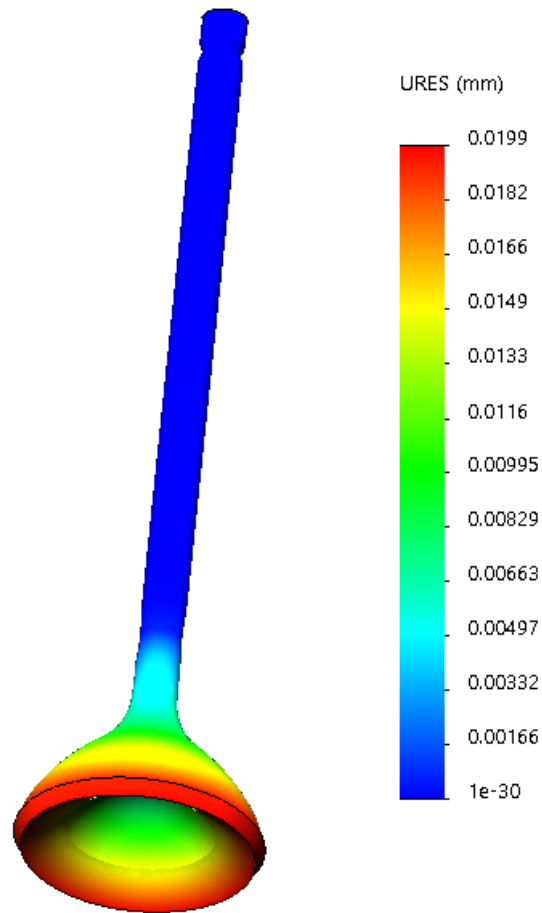


Figure 31 - Iteration 3 Resultant Displacement Plot

Iteration 3 displaces just under 35% more than the alloy steel equivalent.

A reduction in contact force translates to an increase in displacement that is lower than the ratio of elastic modulus when compared with the original valve.

XII. Iteration 4 Study Settings

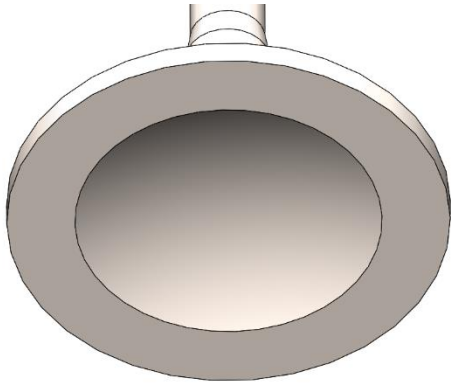


Figure 32 - Original Valve Face

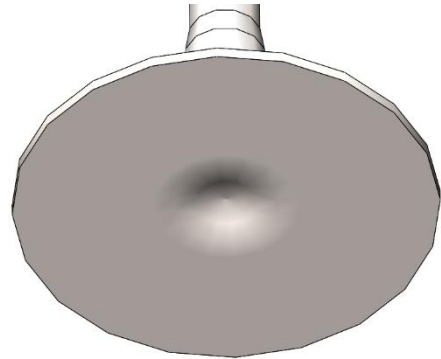


Figure 33 - Iteration 4 Valve Face

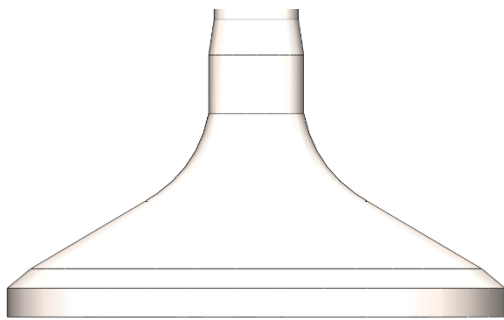


Figure 34 - Original Valve Side Profile

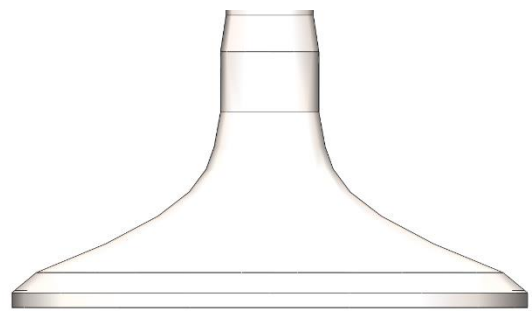


Figure 35 - Iteration 4 Side Profile (Shallow Port Side Spline)

The final iteration combines all elements of the previous studies except with a shallower port side spline and a less pronounced valve undercut. The Ti-6Al-4V material from Iteration 3 is carried over.

Table 7 - Iteration 4 Parameters

Iteration 3 (Titanium – Shallow Port Side Profile, Half Face Thickness & small Undercut)		
Original Mass	New Mass	Percentage Difference
49.55	24.8g	50%
Original Contact Force	New Contact Force	Percentage Difference
5371 N	2718 N	-49%

XIII. Iteration 4 Analysis

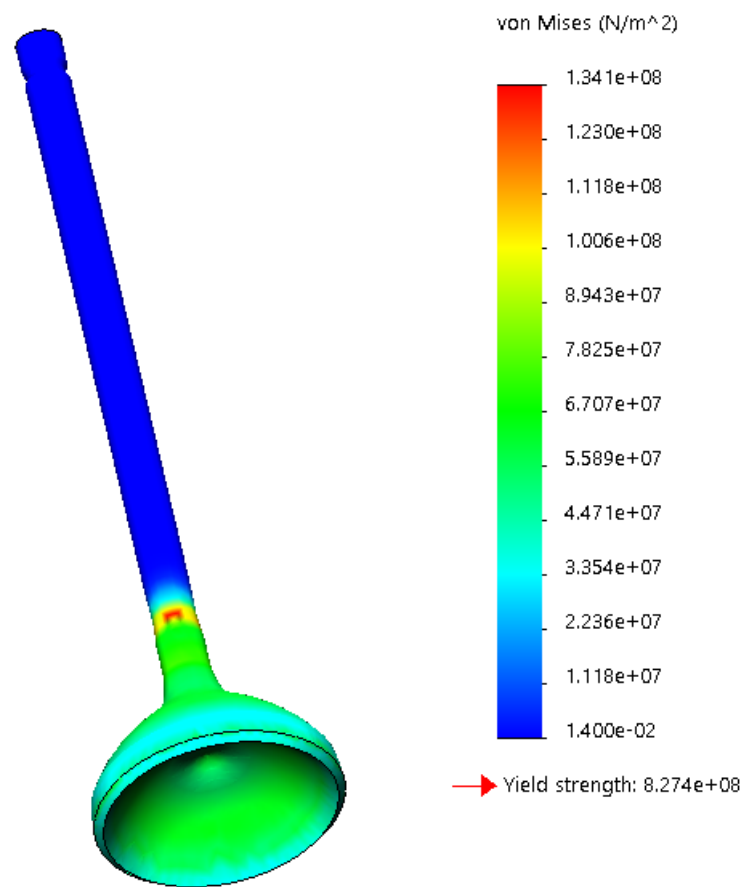


Figure 36 - Iteration 4 Von Mises Stress Plot

The combination of both modifications that were made to the alloy steel valve in *iteration 1* and *iteration 2* result in a further reduction in mass, thus reducing the contact force to half that of the original valve.

This converts to an overall reduction in peak stress at the valve stem and local stress around the valve head.

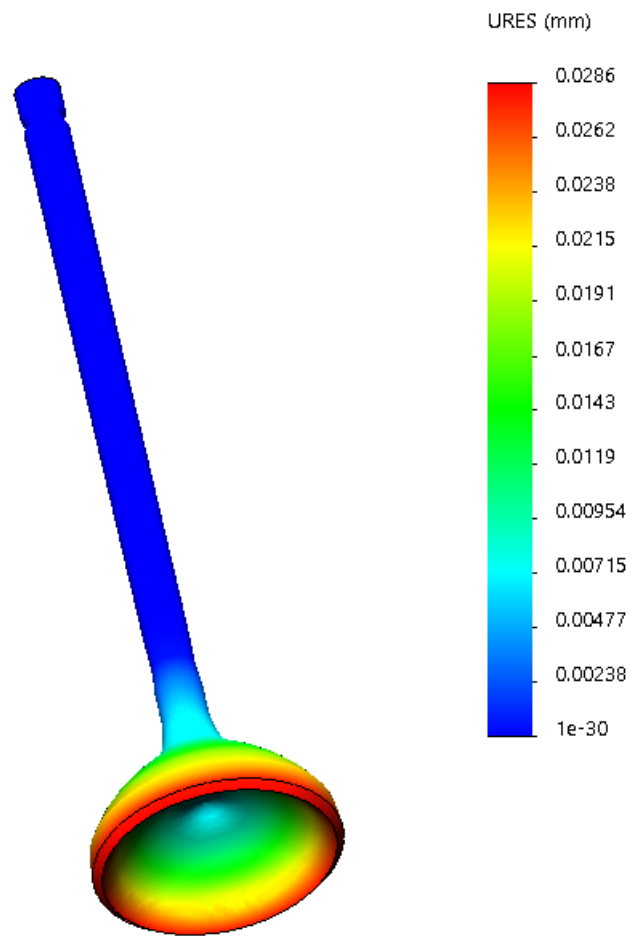


Figure 37 - Iteration 4 Resultant Displacement Plot

However, the removal of material above and below the contacting surface sees an increase in displacement across the edge of the valve head. This is below the hypothetical maximum value of *0.03mm* and is the greatest peak displacement of all iterations.

6. Comparison of Results

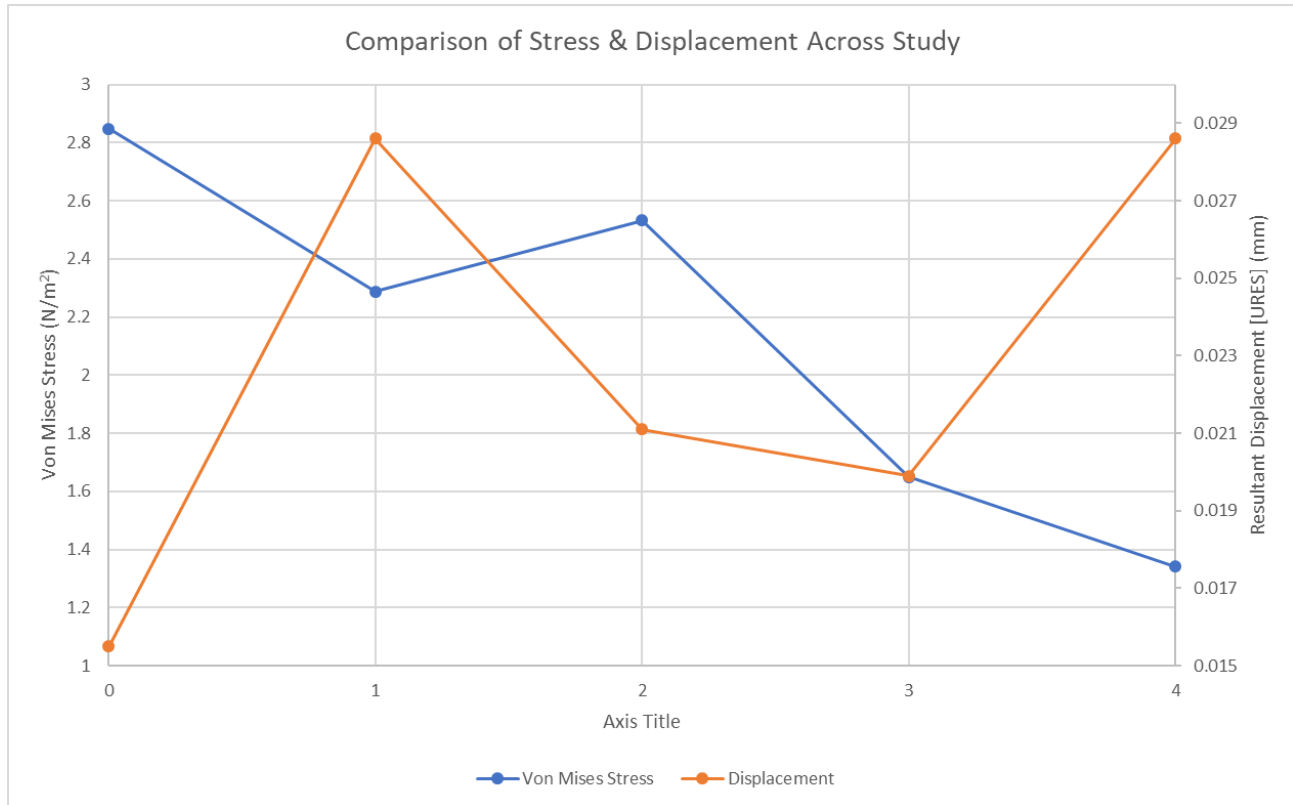


Figure 38 - Comparison Plot for Entire Study

Table 8 - X axis Notation for Comparison Plot

0	1	2	3	4
Original Valve Geometry Alloy Steel	Aggressive Spline Port Side Profile	Half Head Thickness	Ti-6Al-4V Original Valve Geometry	Ti-6Al-4V Modified Shallow Port Side Spline, Half Head Thickness, Small Undercut

The graph shows how changing the valve material from steel to titanium has the best compromise for Stress and Displacement overall. The 1st iteration sees the largest combined stress and displacement through the study.

7. Conclusion

I. Computer Aided Engineering

The Solidworks package allows the user to produce a variety of useful engineering tasks in a virtual environment. The ability to model, analyse motion assemblies and carry out Finite Element Analysis within one program is extremely beneficial. However, there are more sophisticated solutions for each problem in stand-alone form.

Although Solidworks is widely used in industry, the nature of design work has an influence on the type of software package that is required. For example, AutoCAD may be preferred for 2D technical drawings, Design Studio for FEA analysis and ANSYS ICEM for high quality hexahedral meshes.

In addition, materials selection programs should be considered for improvement and optimisation of individual components (Selector, 2020)

II. Inlet Valve Study in Solidworks

a) Further Improvements

- Simplify camshaft and cylinder head for increased computational efficiency
- Validate motion and FEA results with further physical component/material testing.
- Design a mesh study and validate to improve the quality of the results
- Include the cylinder pressures produced at time of inlet valve closing and peak combustion pressure.
- Include temperature data to the study as this is the main contributor to valve/seat wear (Mascarenhas *et al.*, 2015).
- (Chun *et al.*, 2007) proposes a difference in seat recession (due to a change in forces) with different materials. Finishing processes such as thermochemical nitriding can alter the stresses through the components as discussed by (Zhecheva *et al.*, 2005)
- Improve the constraint and connection methods for a more realistic simulation. One solution to overcome the issue is to include the cylinder head to the FEA study as a reference. Another would be to fix the valve seating surface and apply an equivalent contact force to the valve face, with a similar sliding restraint to the valve stem. The collet interface would have to be further considered through physical testing for more accurate results.
- Further consider how gravity and centrifugal forces influence the event.
- Explore the anomaly through the contact force plot as discussed in *section 4*

8. Appendix 1

I. Mesh Density Study

This section will explore how mesh settings influence the results for the design iteration with the highest combined stress and displacement.

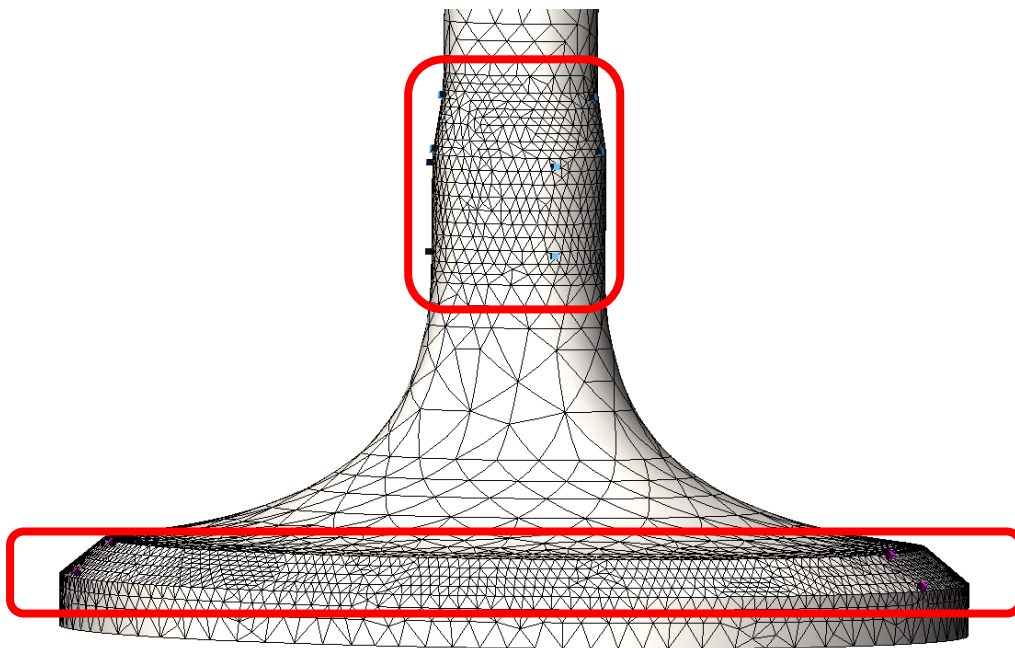


Figure 39 - Mesh Density Increase at Contact Face & Area of Peak Stress for Iteration 1

The Mesh Density was first adjusted all the way to fine for the contacting face and area of peak stress around the stem. This amplifies the peak stress by over 50% to $5.190e+08 \text{ N/m}^2$.

The increase in mesh density to these regions only increases the maximum resultant displacement by 0.3% to 0.0287 mm.

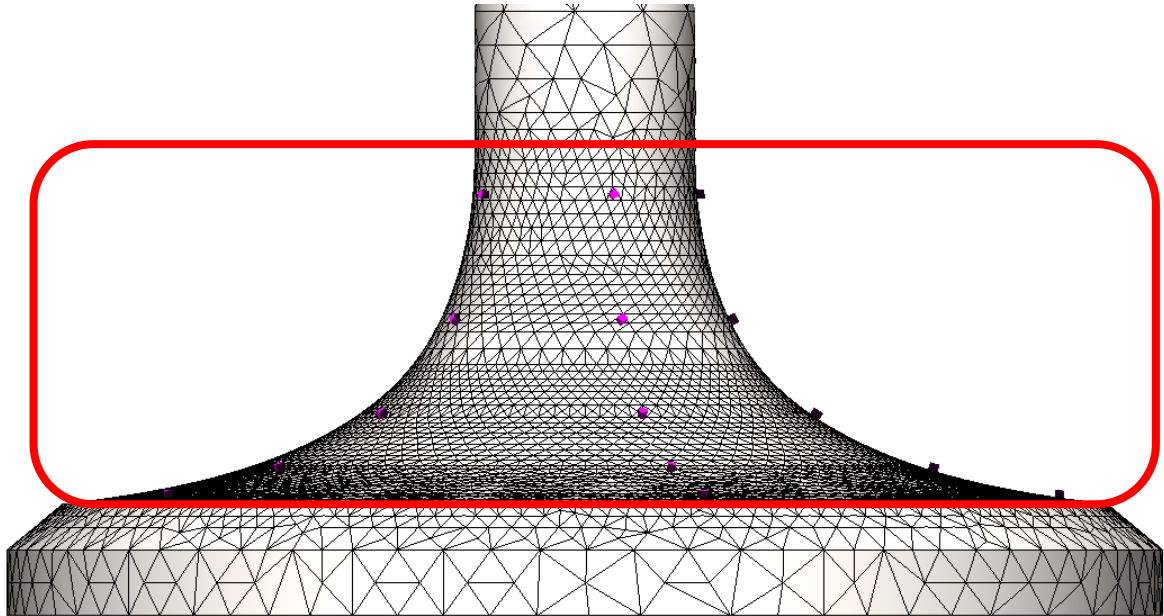
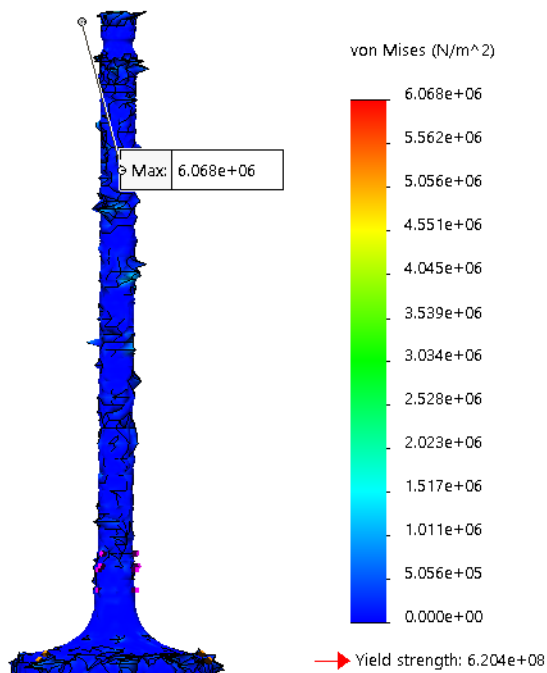


Figure 40 - Mesh Density Increase at Port Side Spline Profile for Iteration 1

Next, the port side spline profile was run at the finest mesh density. The increase to maximum displacement is equal to that of the mesh controls applied in *figure 39* with less than 1% change to the peak stress induced to the standard mesh originally used from Iteration 1.



Finally, the mesh density was reduced to the Coarsest option for the contact face and area of peak pressure. The annotation in *figure 41* shows how unstable the mesh is, migrating the peak stress to outside of the component. The peak stress is increased by over 60% of that produced from the standard mesh. The graphic rendering of the valve is also visually inconsistent due to the poor mesh quality.

This study proves the importance of mesh settings (Thompson and Thompson, 2017).

Figure 41- Mesh Density Reduction at Contact Face & Area of Peak Stress for Iteration 1

9. Appendix 2

I. Inclusion of Equations to the Design Phase

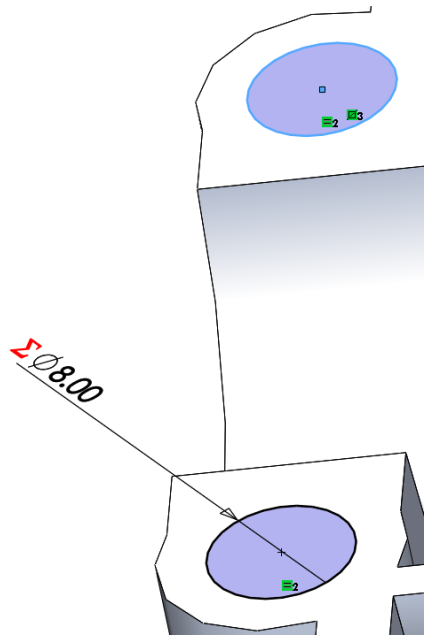


Figure 42 – Example of Sketch Equation with Equal Relations

Name	Value / Equation	Evaluates to	Comments
Global Variables			
"D1"	= 8.00mm	8mm	
D8	= 10.00	10.00mm	
D9	= 10.00	10.00mm	
Add global variable			
Features			
Add feature suppression			
Equations			
Add equation			

☐ Automatically rebuild ☐ Link to external file: Angular equation units: Radians ☒ Automatic solve order

OK Cancel Import... Export... Help

Figure 43 - Cylinder Head Global Variables

10. References

Selector, G., 2020. *Granta Design*. [Online]

Available at: <https://grantadesign.com/industry/products/ces-selector/>
[Accessed 23 January 2020].

Brezina, I. (1997) 'Coordinate measuring machines and systems', *Measurement*. doi: 10.1016/s0263-2241(97)00037-7.

Chun, K. J. *et al.* (2007) 'A study of wear in engine exhaust valve depending on valve materials using a laboratory simulator', in *SAE Technical Papers*. doi: 10.4271/2007-01-0944.

Gatto, R. and Jammalamadaka, S. R. (2007) 'The generalized von Mises distribution', *Statistical Methodology*. doi: 10.1016/j.stamet.2006.11.003.

Liu, A. H. *et al.* (1995) 'A comprehensive investigation of the high speed overhead-cam valve train dynamics in small internal combustion engine', in *SAE Technical Papers*. doi: 10.4271/951795.

Mascarenhas, L. A. B. *et al.* (2015) 'Design and operation of a high temperature wear test apparatus for automotive valve materials', *Wear*. doi: 10.1016/j.wear.2015.08.017.

Ramberg, W. and Osgood, W. R. (1943) 'Description of stress-strain curves by three parameters', *National Advisory Committee For Aeronautics*.

Tani, H. *et al.* (1993) 'Measurement and simulation of valve motion', in *SAE Technical Papers*. doi: 10.4271/931901.

Thompson, M. K. and Thompson, J. M. (2017) 'Meshing', in *ANSYS Mechanical APDL for Finite Element Analysis*. doi: 10.1016/b978-0-12-812981-4.00006-x.

Vavro, J. *et al.* (2017) 'Kinematic and Dynamic Analysis of Planar Mechanisms by Means of the Solid Works Software', in *Procedia Engineering*. doi: 10.1016/j.proeng.2017.02.248.

Zhecheva, A. *et al.* (2005) 'Enhancing the microstructure and properties of titanium alloys through nitriding and other surface engineering methods', *Surface and Coatings Technology*. doi: 10.1016/j.surfcoat.2004.07.115.

Zienkiewicz, O., Taylor, R. and Zhu, J. Z. (2013) *The Finite Element Method: its Basis and Fundamentals: Seventh Edition, The Finite Element Method: its Basis and Fundamentals: Seventh Edition*. doi: 10.1016/C2009-0-24909-9.

CAD Assembly Submission Link

https://uwotstd-my.sharepoint.com/:f:/g/personal/gjones_149914_student_uwotstd_ac_uk/EoLDOGIkYj5AvSnWN6TUIcQBzbr3sIMMGeNlpMLwtLHiw?e=E8Kpd4