

University of Wales Trinity Saint David
School of Engineering

Motion Simulation and FEA
Advanced Computational Methods

M6X00408

Zhangwei Guo

1805078

25/01/2020

目录

1	Abstract.....	1
2	Introduction	2
3	Model	3
3.1	Related parts	4
3.2	Model Assembly	7
4	Motion simulation	8
5	FEA Analysis	10
6	Conclusion.....	13
7	Reference.....	14

1 Abstract

This paper focuses on FEA analysis and the comparison of the results from improved and the original parts. First of all, an assembly which is suitable to this study has been chosen. The assembly should be accessed which could be measured accurately so that the model and assemble could be established. After that, a motion study of the assembly should be finished from which the loads could be calculated and used as boundary conditions in FEA. Thirdly, the FEA analysis will be finished and the results will then be compared with the originals. Finally, the size of the rocker is optimized to achieve minimized weight as both of the strength and stiffness are the same level.

Keywords

Engine Design, finite element analysis(FEA), weight optimization

2 Introduction

As everyone knows, one of the primary objectives of designing an engine is to try to make the power sufficient throughout its common operating range which means the engine speed should be as higher as possible. Lightweight design of the valve has significant importance when engine speed is quite high as the mass moment of inertia will be a huge difference at different engine speeds as current knowledge suggests.

Using optimization techniques to optimize the size and shape to make the weight much more light but keep the strength and stiffness constrained is a trend.

The major target of this paper is to minimize the weight of the valve in an engine using FEA. This paper has been organized as follows. At the beginning, the assembly has been chosen and modified. After that, motion study has been done to get the loads and the original results. Thirdly, FEA has been used trying to optimize the valve. Finally, the results then be compared with the originals.

3 Model

For the model, a V6 engine with two turbo whose parameters could be found on the website has been chosen to finish the study as figure 1 shows.

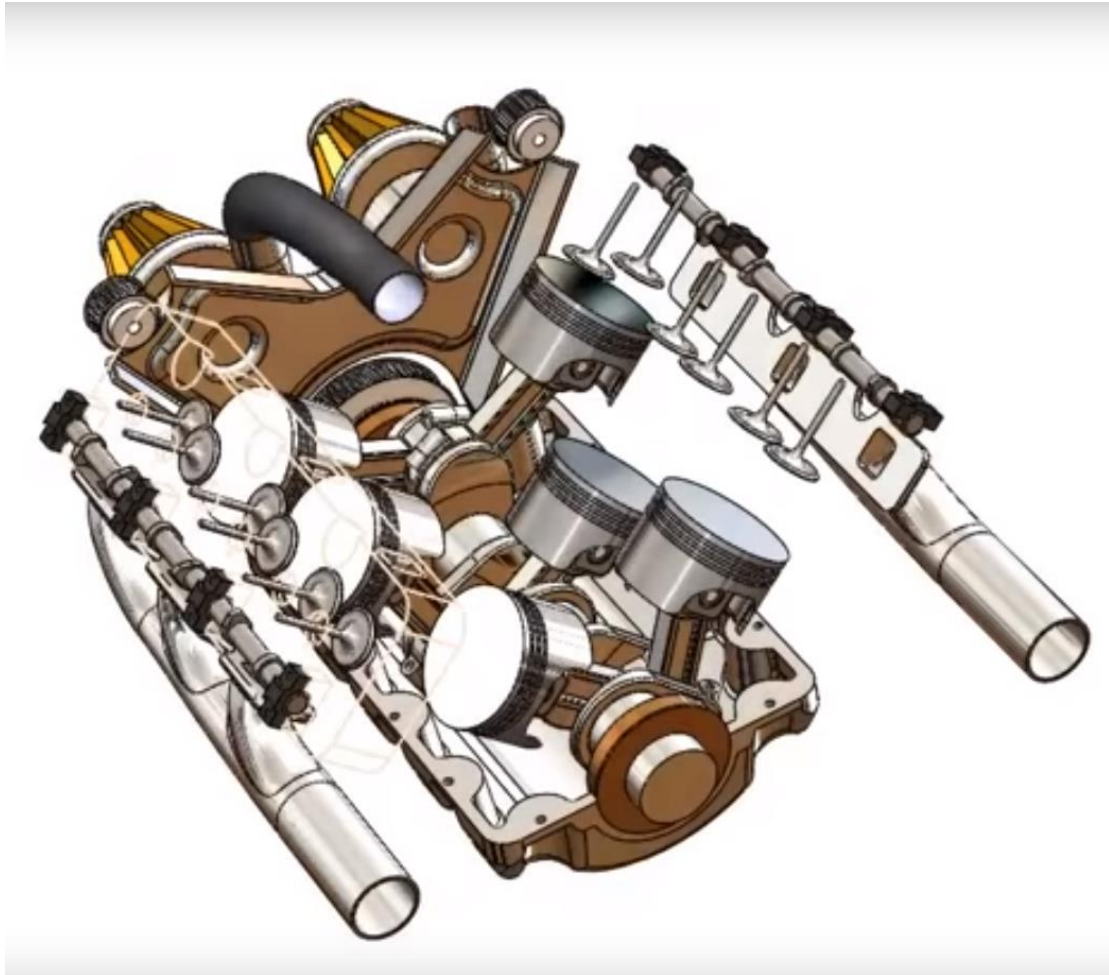


Figure 1. V6 Engine

In this paper, only several parts of this V6 engine which have been shown in figure 2 have been established as the analysis only aims to optimize the valve.

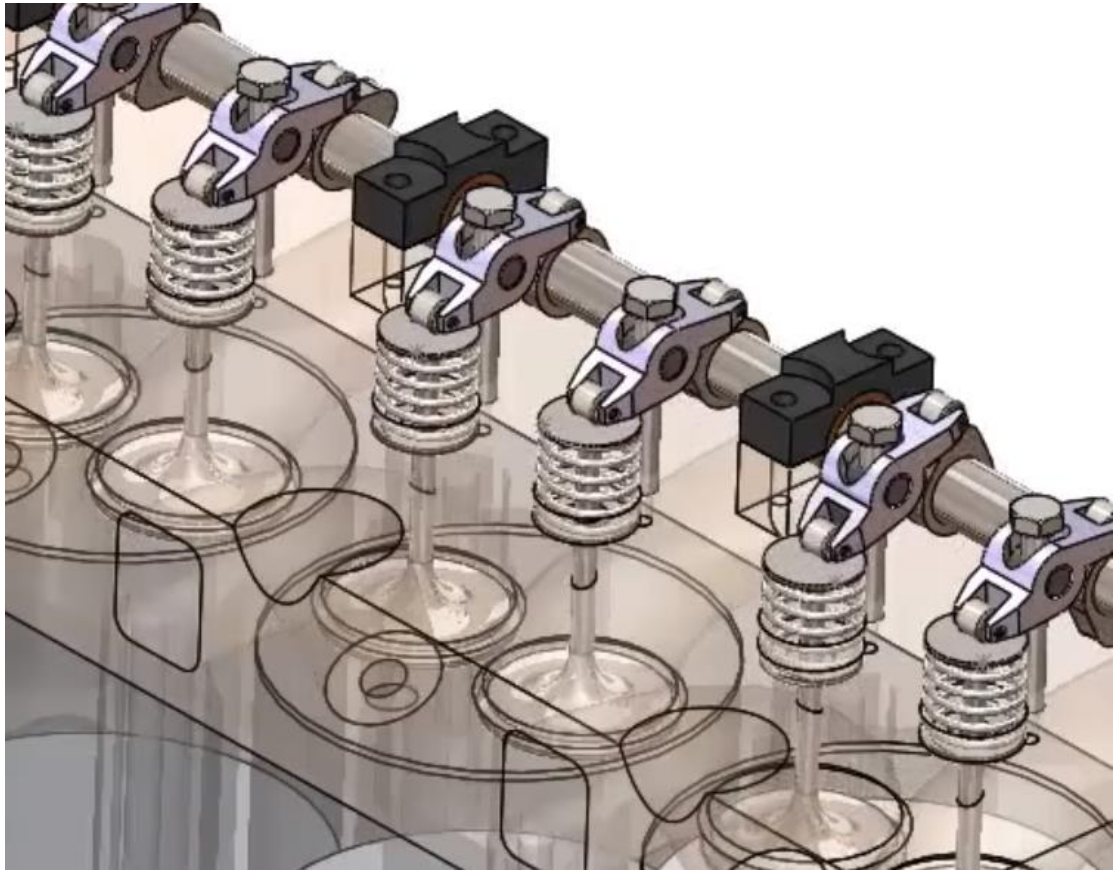


Figure 2. Related parts

3.1 Related parts

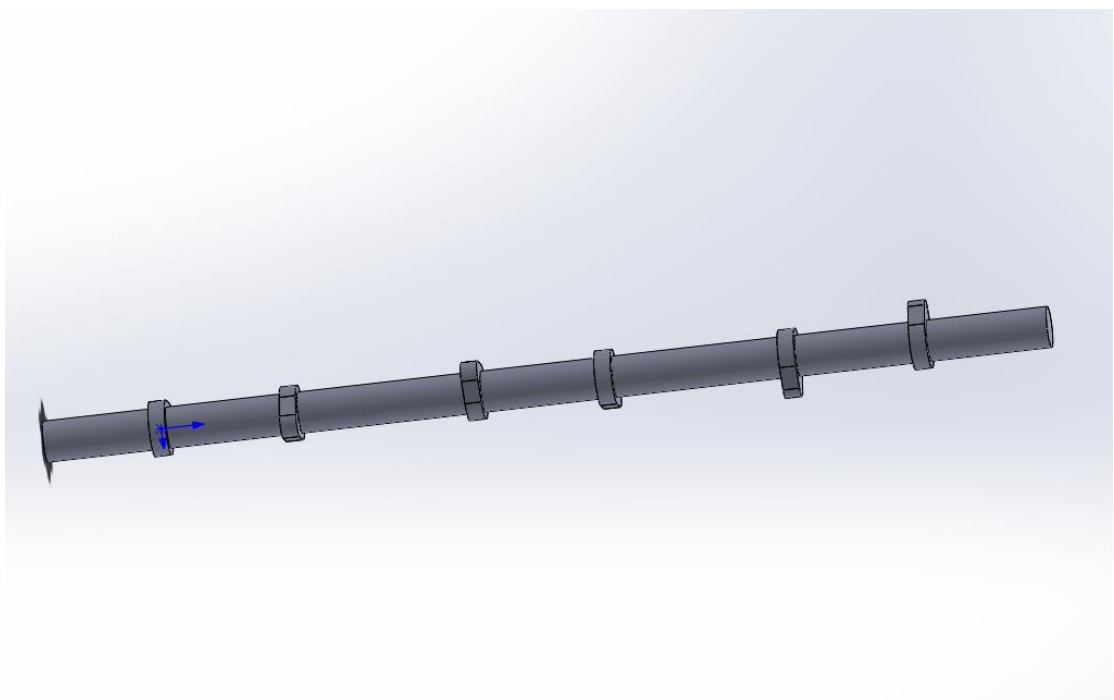


Figure 3. Cam shaft

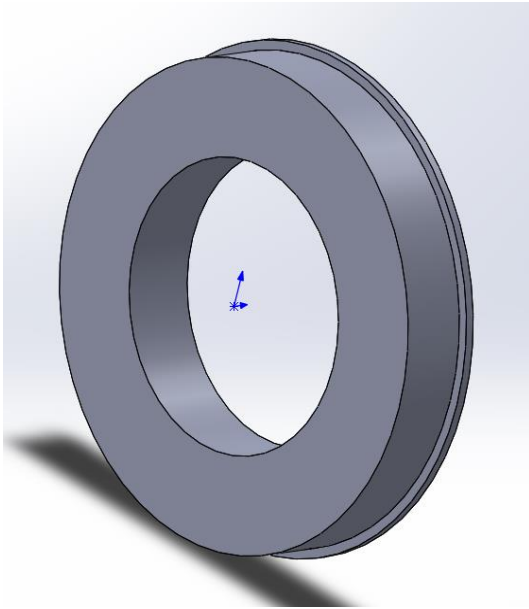


Figure 4. C.Bushings-1

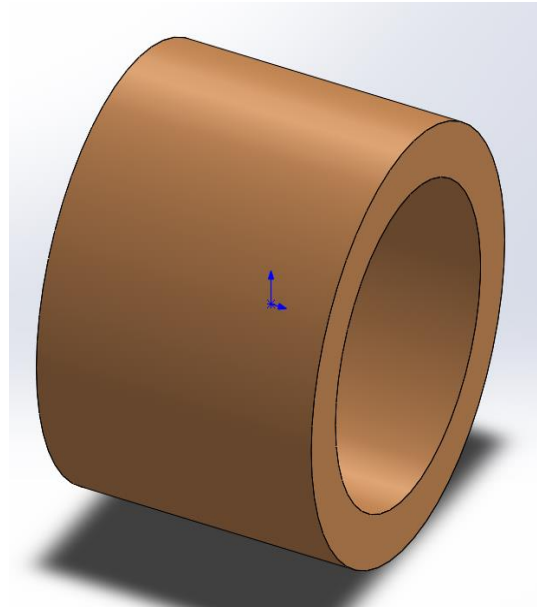


Figure 5. C.Bushings-2

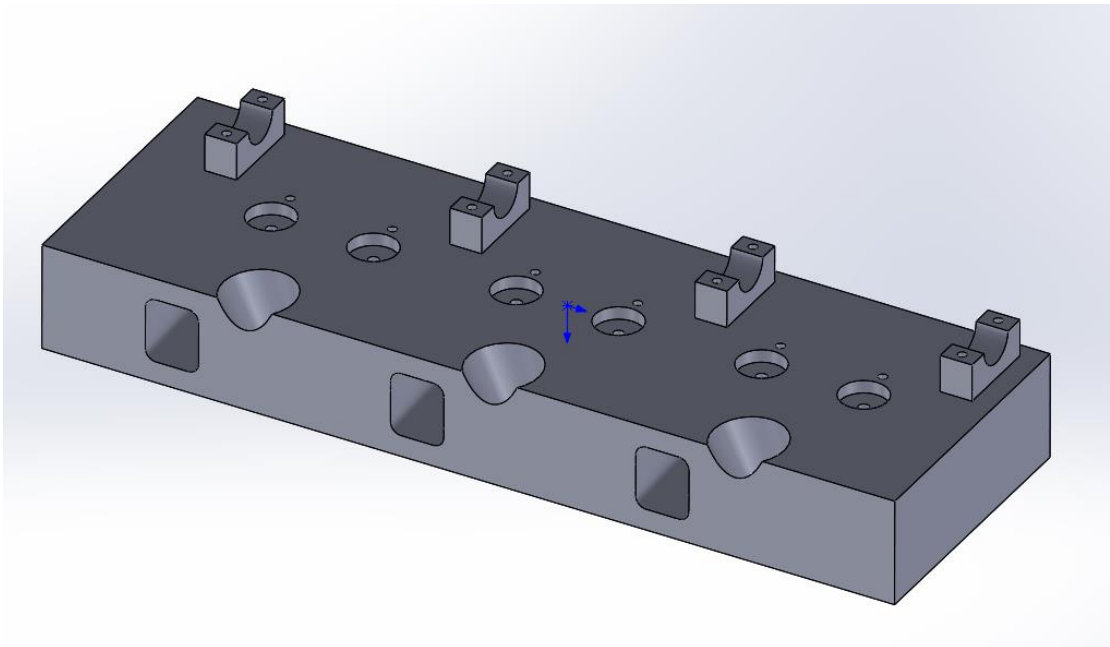


Figure 6. Cylinder Head

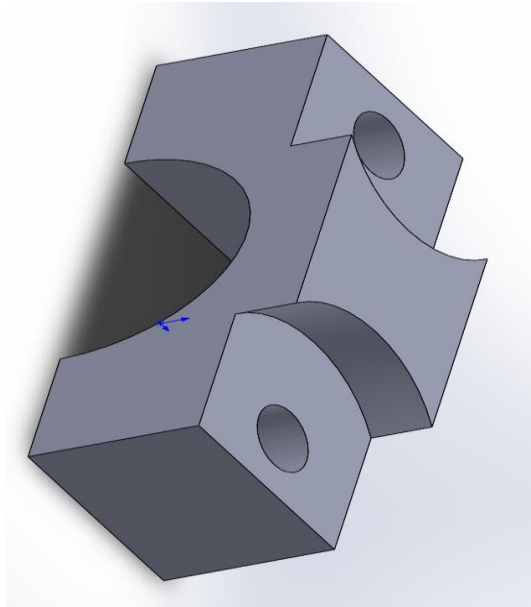


Figure 7. C.Retainer

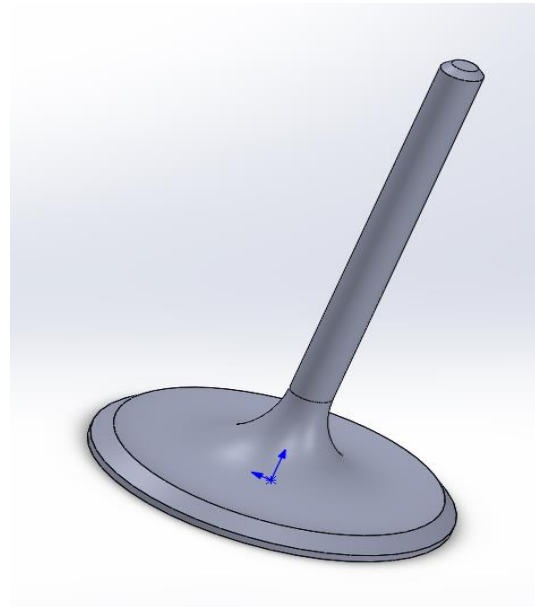


Figure 8. Valve

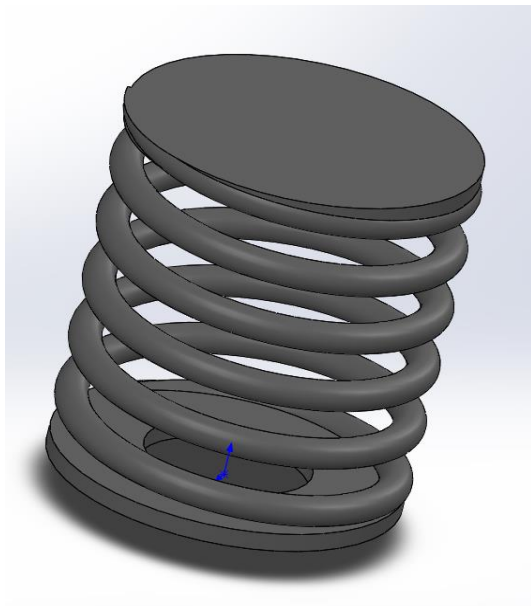


Figure 9. Rocker Spring

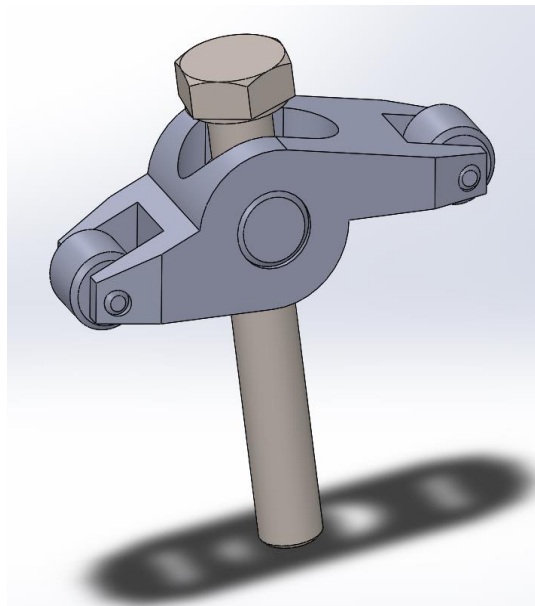


Figure 10. Rocker Arm Assembly

3.2 Model Assembly

For several simply parts, the mates in assembly are enough.

Table 1-Assembly and Mates

Part 1	Part 2	Mates
C . Bushings	Cam Shaft	Concentric, Coincidence
C . Retainer and Cylinder Head	Cam Shaft	Concentric, Coincidence
Valve	Rocker Spring	Concentric, Coincidence
Valve	cylinder head	Concentric
Rocker spring	Rocker Arm	Tangent
Rocker Arm	Cylinder Head	Concentric, Parallel, Distance

But for the mate of rocker arm and cam shaft, at the beginning, tried to make it with the cam-shaft mate within the mechanical mate, but it can only choose one face of the shaft when it has four. After that, change it to the feature called contact within the motion simulation tab and defined it as a solid contact with the material of steel greasy.

4 Motion simulation

After importing the geometric assembly into SOLIDWORKS, load the SOLIDWORKS simulation plug-in to set related motion parameters in the motion study tab. First of all, apply the gravity feature to the entire assembly using the built-in G value. After that, for the particularity of the model, add the contact feature as solid contacts between the shaft and the rocker and the rocker and the valve as well. The material properties steel greasy are given at the same time. And then a linear spring with an appropriate elastic coefficient is added between the upper surface of the valve and the upper surface of the cylinder head. Finally, apply a linear motor with a constant clockwise displacement on the camshaft and set the calculation time to 5 seconds to start the calculation. After the calculation is completed, the results and graphical features are used to define and record the contact forces between several parts of the assembly as follows.

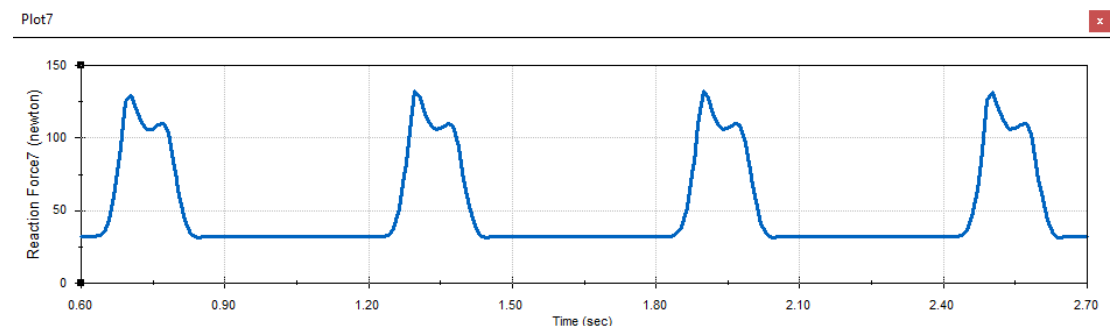


Figure 11. Contact Force Between Rocker and Camshaft

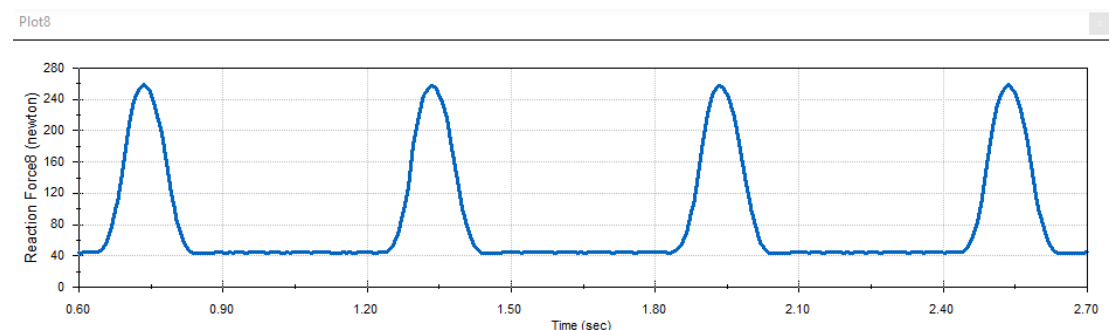


Figure 12. Contact Force Between Rocker and Valve

Before go for FEA, study properties needed to be set to 25 frames per second for Motion analysis up to 5 seconds. The load surfaces which records motion loads like

reaction forces, moments, acceleration and speed during analysis need to be defined before running the motion analysis on account of only forces transferred to faces can be exported from a motion study to simulation study. For the bearing surfaces, they could be defined as part of the definition of mates.

5 FEA Analysis

Before go for FEA, the calculation in motion analysis should be finished and saved so that loads from motion could be imported into SOLIDWORKS Simulation.

Using the feature of input motion load in the tab of simulation to get the load ready for FEA. Choose valve in the box of available assembly components and transfer it into the box of selected components. And then click on the button of multiscreen study with design case up to 10,000 sets. The start and end time could be input within 125 which means 5 seconds. After confirm all of these settings, SOLIDWORKS Simulation automatically generates new design studies with predefined design scenarios in the link part document. The load for each design scenario is formed during Motion in a specific instant of time in the relevant picture. The above specifications define 9 design scenarios. You can find critical time instants by analyzing the links of all design situations when the maximum stresses may form in the part.

There are several settings should be done before run the situation. Right-click the valve and select open part in the Feature Manager design tree first, then two simulation studies could be found are defined in the part document which is a static study and another one is a design study.

To review the definitions of motion loads like gravitational, centrifugal, and remote loads, right-click their respective icons in the static study tree and select edit definition. The gray areas are defined as link parameters. The most important thing is to assign material such as alloy steel to parts from the SOLIDWORKS material library otherwise could not go for running. Finally click the design study tab to review 9 active scenarios and the click run analysis activity scenario.

The design history charts could be generated for von Mises stresses to help to find moments in time and the associated design situations where maximum von Mises stresses may occur. Right-click results and graphs and select define design history graph in the design study tree, choose the constraint under the Y-axis and select

VON: von Mises stress. It is much easier to find the moment when the maximum von Mises stresses appear.

If more details about the results needed, In the design study tree, select the title row in one column when the pointer changes to an arrow.

The columns are highlighted in black, and the results for the design case are loaded as figure 13 &14 show.

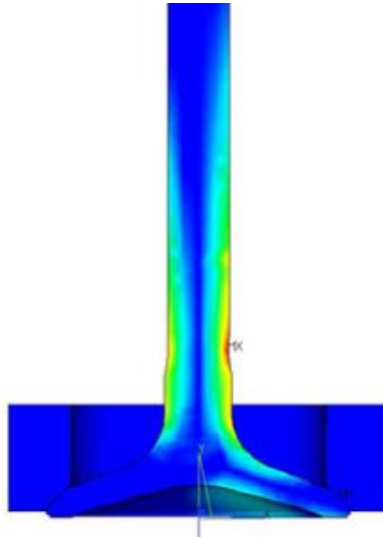


Figure 13. Von Mises Stress at 500RPM

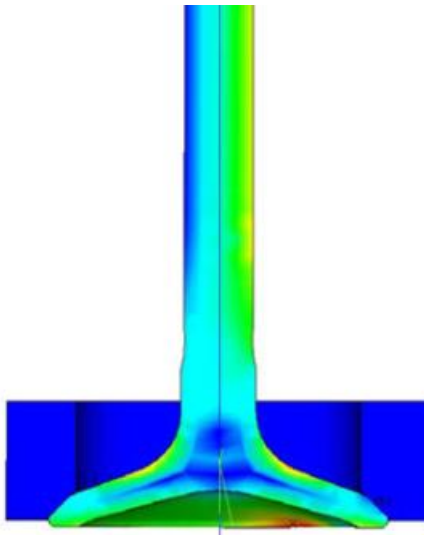


Figure 14. Von Mises Stress at 800RPM

The improvement of the valve is mainly carried out from two aspects, one of which is mainly for the optimization of the shape, and the other is for the optimization of the size. For the optimization of the shape, the selection of the angle from the chamfer to

the bottom surface has a significant effect on improving the local stress. The optimization of the geometry of the slender rod is not obvious for local stress changes, and it is of great help to improve the overall level of stress and strain. Both methods have a significant effect on the reduction of valve mass. Although the amount of mass reduction is small, considering that the valve has a very high speed at high speeds, it is meaningful to carry out this optimization.

6 Conclusion

The completion of this task not only deepens the understanding and application of the FEA process, but also cultivates the ability to apply the knowledge learned to practical problems. The weight of the valve is minimized through FEA which means achieved the original target of this paper.

7 Reference

- Varma Pakalapati, V. K. (2011, 04 12). CAE based 'Multi Objective Optimization approach for Spot Weld Connections Layout' in Automotive Structure. *SAE INTERNATIONAL*, pp. 50-62.
- Vlahopoulos, N. (2011, 05 17). Modeling of Stiffened Panels Using the Energy Finite Element Analysis. *SAE INTERNATIONAL*, pp. 60-28.
- Yang LeiLi, Chuxuan Wang and Yunqing ZhangQiang. (2017 年 03 月 28 日). Loads Analysis and Optimization of FSAE Race Car Frame. *SAE INTERNATIONAL*, 页 12-18.