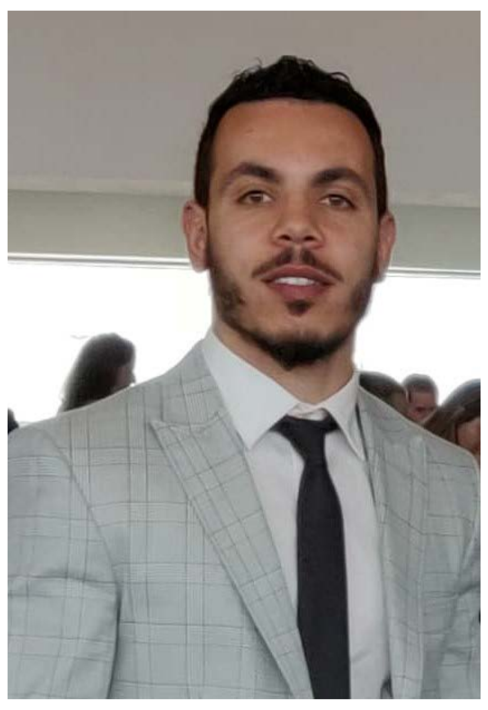
Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David



Tumble-Flow Development Through Port Geometry Optimisation and the Influence of Turbulence on GDI Engine Efficiency and Emissions.

Yousouf Boukra

Supervisor: Andrew Gibson

School *of* **Engineering**

BSc(Hons) Automotive Engineering

Introduction

The combustion process development is firmly related to cylinder-head design and port geometry, both emissions and combustion performance are connected to turbulent intensity, and the quality of mixture formation [1, 2].

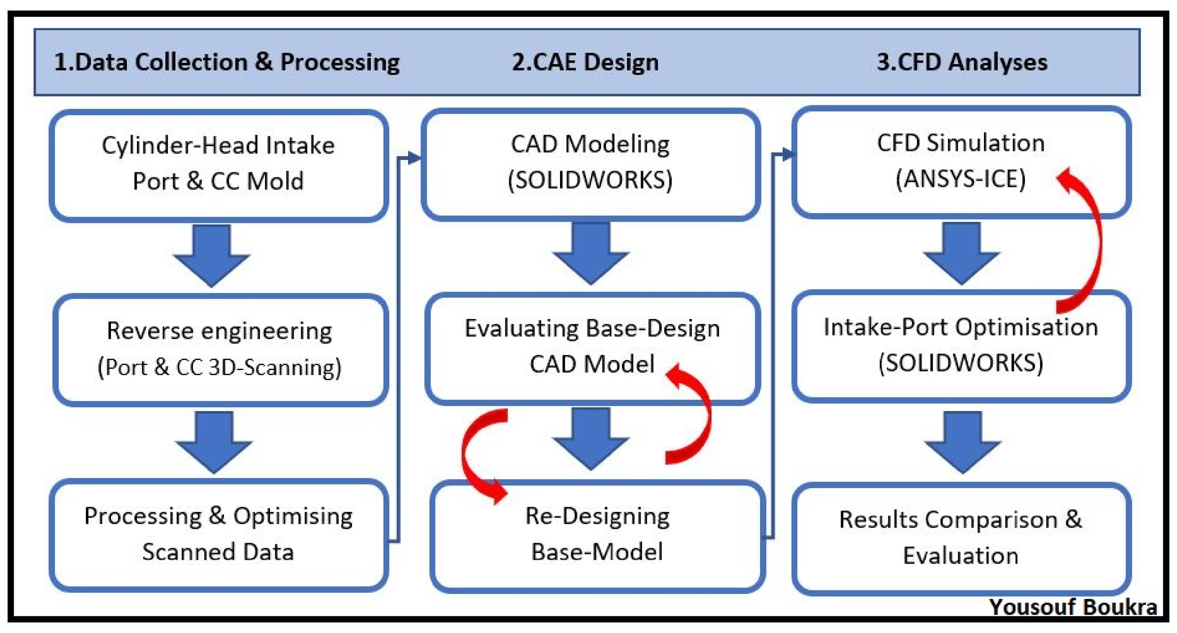
In the present study, a GDI intake port was analysed throughout the application of CAE tools and CFD simulation, to assess the tumble motion and in-cylinder flow structure. Solidworks package was implemented for the design and modelling, and the simulation process was conducted on ANSYS workbench. Extensive validation work was conducted using different turbulence models simulation, experimental flow-bench testing, and analytical verification. Further optimisation work was carried out to alter the tumble charge, by means of geometrical port modification, where the main design parameters comprised port profile angle, chimney height, and port masking.

Project Aims

- ➢ The main aim of this work was to critically assess the influence of intake-port geometry, on tumble generation, and in-cylinder turbulence. The complex turbulent flow occurring during the intake stroke, has motivated the implementation of CFD engine simulation, which was an imperative technology and has been utilised extensively during this project.
- ➢ The second aim included a comprehensive investigation, regarding turbulence modelling within a 3-D CFD domain, to discover the ideal turbulence model, which accurately predicts the fluid flow structure inside the cylinder.

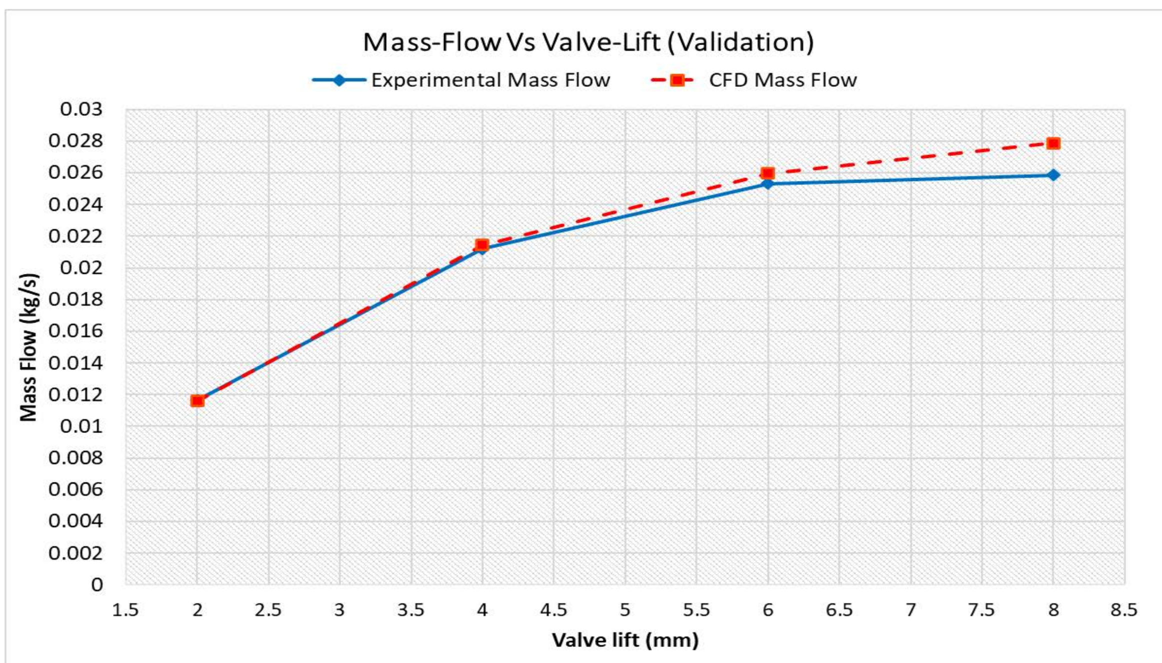
Methodology

The project was approached using a structured development planning which comprises of three main phases: **1**. collection of data throughout reverse engineering process; **2**. development and evaluation of CAD models; **3**. CFD simulation and flow validation against the experimental benchmark data, and theoretical calculation.



Yousouf, B (2020) 'Project Development Plan', UWTSD

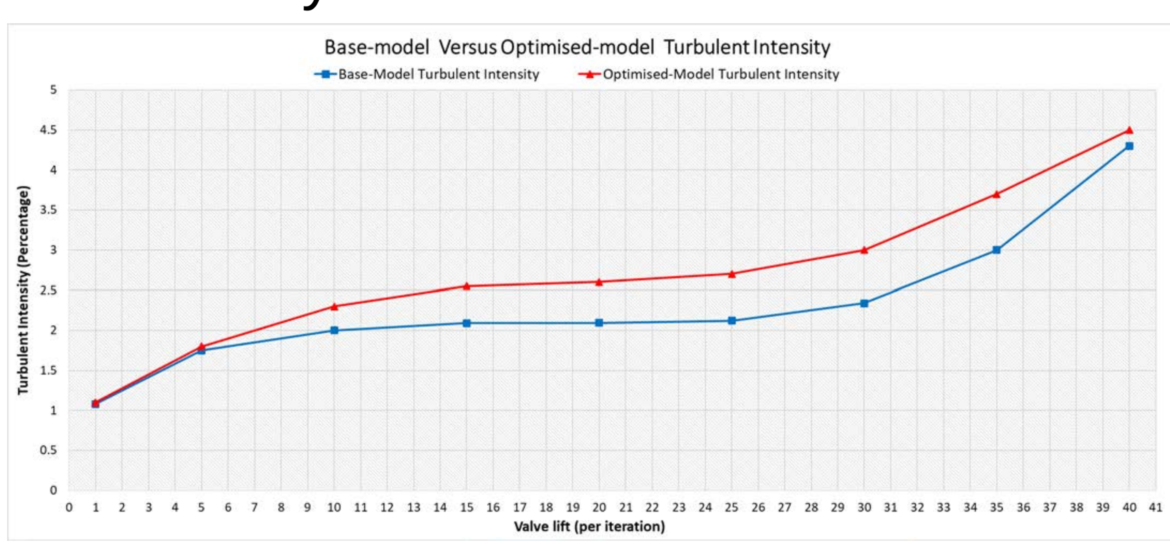
Both Solid-modelling and Surface-modelling techniques were utilised during the design procedure. Six '*K-ϵ*' turbulence models were applied to simulate the flow-charge during the validation process, of which, '***Realisable K-ϵ enhanced wall***' was the most accurate in term of mass-flow-rate and in-cylinder turbulence prediction (98.55% accurate).



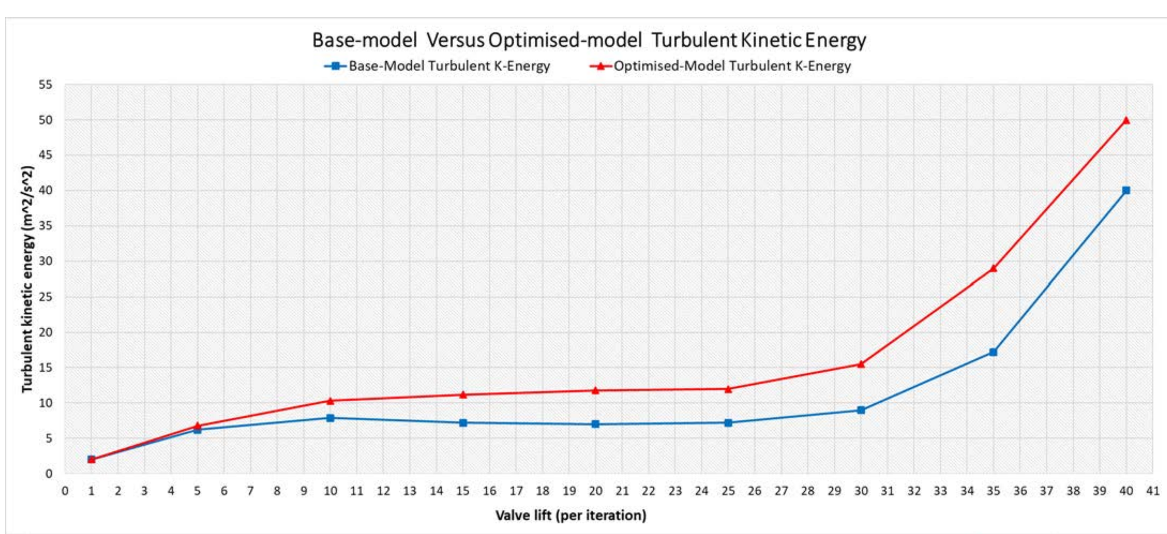
Yousouf, B (2020) 'Mass-Flow Validation', UWTSD

Results

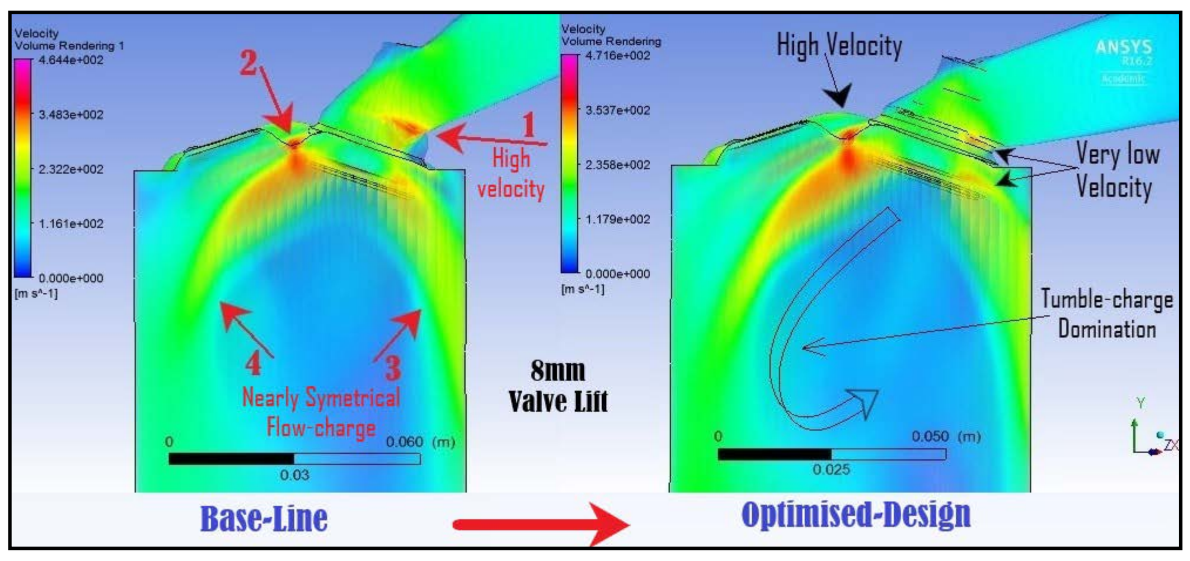
Despite the minimal decrease in the mass flow rate (2.3%), the optimised-port which prompt higher tumble-motion during the intake stroke, significantly enhanced turbulent intensity by 7.72%. Consequently, turbulent kinetic energy increased by 17.8%.



Yousouf, B (2020) 'TI Optimisation', UWTSD

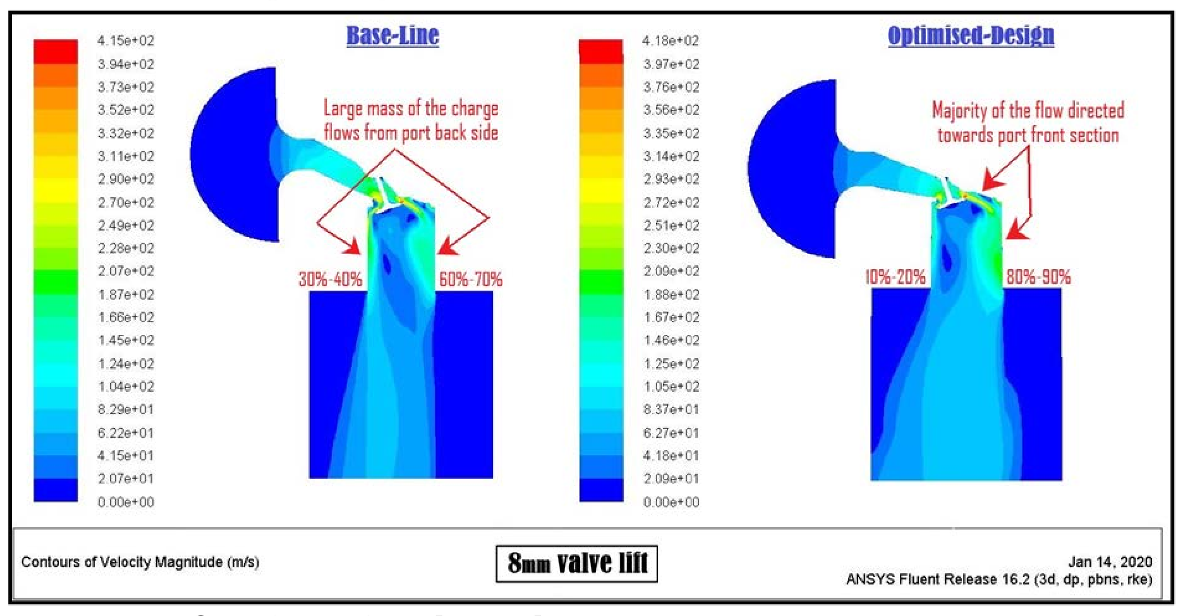


Yousouf, B (2020) 'TKE Optimisation', UWTSD

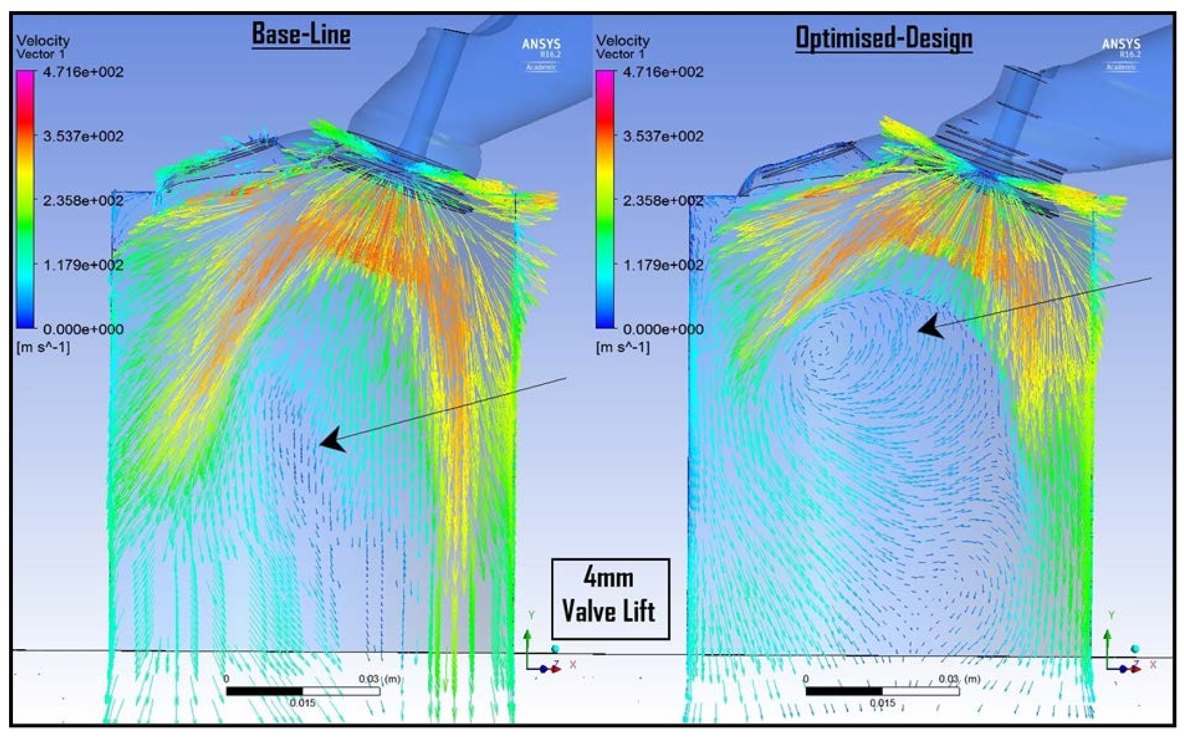


Yousouf, B (2020) 'Tumble-motion Development CFD', UWTSD

It was evident throughout this study, and by many researchers [3, 4], that an optimised high-tumble port, significantly improve the in-cylinder turbulence, and subsequently, the overall engine performance level.



Yousouf, B (2020) 'Flow-charge separation CFD', UWTSD



Yousouf, B (2020) 'In-Cylinder Turbulence Flow Structure Enhancement CFD', UWTSD

Conclusion

In the current research, a 1.6 Ford Ecoboost intake-port has been investigated, throughout the application of reverse engineering tools, CAE design, and CFD simulation, aided by experimental validation and theoretical work. The study aimed to analyse the influence of port geometry on the tumble-motion and introduce a new optimised design, where the results revealed, 7.72% and 17.8% increase in TI and TKE, respectively.

References

[1] Sakurai et al. ''Optimization of intake port for improvement of fuel consumption and torque'', SAE International Journal, 2017.
[2] Singh et al. ''Intake and Exhaust Ports Design for Tumble and Mass Flow Rate Improvements in Gasoline Engine'', SAE International Journal, 2019.
[3] Iyer and Yi. ''3D CFD upfront optimisation of the in-cylinder flow of the 3.5L V6 Ecoboost engine'', SAE International Journal, 2009.
[4] Cameretti et al. ''3D CFD analyses of intake duct geometry impact on tumble motion and turbulence production in SI engines'', SAE International Journal, 2017.

SolidWorks Geomagic 3D SYSTEMS Geomagic® for SOLIDWORKS® ANSYS FLUENT SuperFlow

School *of* **Engineering**