Numerical Investigation of In-chamber Flow inside a Wankel Rotary Engine

T Poojitganont1,*, J Sinchai2, B Watjatrakul2, and HP Berg1

1Chair of Combustion Engines and Flight Propulsion, Brandenburg University of Technology Cottbus - Senftenberg, Cottbus, 03046, Germany
2Department of Mechanical and Aerospace Engineering, King Mongkut’s University of Technology North Bangkok, Bangkok, 10800, Thailand

* Corresponding Author: Poojitth@B-Tu.De

Abstract. In this paper the in-house technique mesh moving function based on AVL-FIRE has been developed as a simulation tool to investigate the in-chamber flow phenomena of the Wankel rotary engine. The meshes from the starting rotor position at 165˚ BTDC until the ending rotor position at top dead center (TDC) have been created and connected together. The numerical simulations between an intake stroke and the end of a compression stroke have been successively performed. The results are validated with the selected publication on various engine speeds, at 675 rpm and 1170 rpm. In parallel three refinements of meshes have been carried out, in order to optimize the suitable meshes’ elements for the calculation of this engine type. In addition, the turbulence models, which are standard k-ε and Large Eddy Simulations (LES), have been varied for detailed investigation of their predictive capabilities. The simulation results show that the flow phenomena are well corresponding to the experimental data in both engine speeds, especially with the LES. It could be also identified that the LES model performs better on predicting the flow field both in directions and characteristics. Moreover, there is no evidently difference on the results between the medium (c.a. 100,000 elements) and fine (c.a. 1,000,000 elements) meshes comparing to the experimental results.

1. Introduction
The Wankel rotary engine is categorized as one type of internal combustion engine. Based on its simple and compact design, the Wankel rotary engine provides a high power to weight ratio compared to the reciprocating engines. Its advantage is profitable for automobile with Hybrid system and several kinds of aircrafts [1]. Furthermore, it generally generates flat torque characteristics and high performance at high speed [2, 3]. The exception from those advantages of the Wankel rotary engine, there are drawbacks, which are significant to be considered for improving this engine type. Typically, the engine has lower thermodynamics efficiency due to its long combustion-chamber design increasing possibility of quenching area. This causes the incomplete combustion to occur and releases emissions of NOx, CO, HC and unburned fuel [3].

In order to improve the combustion and mixture formation of the Wankel rotary engine, it is necessary to understand and be able to predict the flow behaviors inside the chamber. Therefore, the study of fluid mechanisms in inside the combustion chamber is apparently convincing in terms of optimizing the design of its geometries, fuel atomization, and fuel-air mixing including combustion...
efficiency [4-6]. Based on these ideas, the flow phenomena of the Wankel rotary engine has been investigated in this study by means of the CFD software, AVL-FIRE. The in-house mesh moving technique has been implemented to connect the group of meshes together. In order to validate various flow patterns, the numerical results has been compared with the experimental investigation from the publication on the airflow visualization of rotary engine assembly performed by F. Hamady, T. Stuecken and H. Schock [7].

2. CFD Modelling

2.1. Geometric Models Generation
In the investigation the flow field inside the Wankel rotary engine during an intake stroke to the end of compression stroke is focused, thus a group of meshes as same geometry as the referenced publication in small steps of rotor position between 165˚R BTDC until TDC has been performed, referencing on the absolute coordinate system. Examples of these meshes could be shown in figure 1.

![Figure 1. Example of calculating meshes.](image)

These meshes are connected and continuously calculated during the simulation, using the in-house mesh moving technique. Conservation of mass, momentum and thermal energy are utilized as the governing equation [8]. In this study, the $k-\varepsilon$ model and LES are chosen for capture turbulent behaviours furthered from the former research [9]. In addition, the variation of mesh quantity has been also achieved as well.

2.2. Turbulence Models
Regarding the flow pattern in the engine has sophisticated behaviors and high turbulence intensity, the turbulence models have to be implemented. Firstly, the $k-\varepsilon$ model is claimed to be suitable for this kind of study [10, 11]. The defined equation of standard, high-Re-number $k-\varepsilon$ model, integrated over a control volume is given based on AVL FIRE CFD solver [8] as follows,

\[
\rho \frac{\partial k}{\partial t} + \rho U_j \frac{\partial k}{\partial x_j} = P + G - \varepsilon + \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right)
\]  

(1)

\[
\rho \frac{\partial \varepsilon}{\partial t} = \left( C_{\varepsilon P} P + C_{\varepsilon G} G + C_{\varepsilon k} \frac{\partial k}{\partial x_j} - C_{\varepsilon T} T \right) \frac{\partial x_j}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu_t \frac{\partial \varepsilon}{\partial x_j} \right)
\]

(2)

Where $P$ is a production of $k$ by mean-flow deformation, $G$ is the production of $k$ by body force. Both parameters are given as follows,
\[ P = 2\mu_i S \cdot S - \left( \frac{2}{3} \right) [\mu_i (trS) + k](trS) \] (3)

\[ G = -\frac{\mu_i}{\rho \sigma} \nabla p \] (4)

\[ \mu_i = C_\mu \rho \frac{k^2}{\varepsilon} \] (5)

In order to investigate the predictive capability, LES model is also used in the simulation since it is proved more accurate by various studies [12-14]. The definition could be illustrated as following.

\[ \frac{\partial (\bar{u}_i)}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ B \frac{\partial (\bar{u}_i)}{\partial x_j} - \left( \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \right) \right] \] (6)

2.3. Mesh Quantity
Three different grid sizes are considered in this study including 2.4 mm, 0.8 mm, and 0.4 mm of hexahedral mesh. The mesh number example of those different sizes are shown in table 1.

| Grid size | 2.4 mm | 0.8 mm | 0.4 mm |
|-----------|---------|---------|---------|
| Mesh number at TDC | 7280 | 58240 | 465920 |
| Mesh number at BTC | 17900 | 143200 | 1145600 |

2.4. Initial and Boundary Condition
The working fluid is assumed to be air throughout the simulation which is defined as unsteady flow, viscous and compressible. All the leakage goes inward and outward the combustion room is neglected. The initial condition at 165°C BTDC shown in table 2 is based on experimental condition of the referent publication.

| Parameters | Value |
|------------|-------|
| Species: | Air |
| Pressure [pa]: | 1x10^5 |
| Density [kg/m^3]: | 1.161 |
| Temperature [K]: | 298 |
| Initial velocity U, V, W [m/s]: | 0, 0, 0 |

Figure 2. Definition of boundary condition.
Due to a symmetrical shape in Z-direction of the engine, the symmetry boundary condition could be applied, to reduce the global number of meshes. The inlet port has been added into and defined as inlet boundary condition shown in figure 2. The rest surfaces have been set as moving or fixed walls.

3. Simulation Results and Discussion

3.1. Mesh Validation
The simulation results on different grid sizes on both k–ε and LES cases are illustrated in figure 3 and 4. The results showed that the medium (0.8 mm) and fine (0.4 mm) grid size provide the closely results and capture full of details comparing to the rough one. The medium grid size has been selected to perform the further calculation since it provides adequate details and reasonable time spending.

Figure 3. Velocity profiles on different grid sizes with k-ε turbulence model.

Figure 4. Velocity profiles on different grid sizes with LES turbulence model.
3.2. *Turbulence Model Variation*

In this step, the comparison on the medium mesh with various turbulence models and experimental data has been performed as shown in figure 5. It is evident that the LES provides more accurate results comparing with k-ε model. Therefore, the investigation of flow validation on different engine speeds in the next chapter has been done based on LES model.

![Figure 5](image)

**Figure 5.** Velocity profiles with various turbulence models at 675 rpm.

3.3. *Flow Pattern Validation*

Starting from 165°R BTDC until the last position at TDC on the shaft speed of 675 rpm and 1170 rpm, the flow field inside the combustion chamber could be verified. During intake stroke, the flow is mainly governed by the momentum of intake and impingement on the rotor pocket, forming a counterclockwise recirculation in the main region. Since the effect of leakage at the apex seal is neglected, the simulation could not capture the counter flow along the housing adjacent to the leading edge as marked in figure 6. However, this flow seems to diminish rapidly due to the jet-like blow-by flow caused by the leakage at the apex seal [7].

![Figure 6](image)

**Figure 6.** Comparing results during intake stroke at 675 rpm.
When the rotor moves to the bottom dead center (90°R BTDC), the intake is started to close. After this position, the fluid is mainly governed by the movement of the rotor. The flow started to change its direction forming the clockwise recirculation as illustrated in figure 7. This phenomenon induces the mixing process between the oxidizer and fuel in this type of engine.

The reversed flow appeared in the experiment at 75°R BTDC could not be captured by the simulation. The assumption of discrepancy is the existence of the leakage of the apex seal at the tailing edge, which is ignored in the simulation and should be studied in the next paper. However, the main character of flow field could be well predicted with the LES model. In the final stage of compression, the vorticity is intensified by decreased volume.

On the higher shaft speed at 1170 rpm, the flow pattern has also the same behaviour as appeared in the slower case. However, there is a significant increase on the intensity of turbulence compared to the lower speed during intake stroke as shown in figure 8 and compression stroke in figure 9, respectively.

**Figure 7.** Comparing results during compression stroke at 675 rpm.
Figure 8. Comparing results during intake stroke at 1170 rpm.

Figure 9. Comparing results during compression stroke at 1170 rpm.
4. Conclusions
In this study, the dynamic meshes and turbulence models have been gathered and simulated to capture the flow phenomena inside the combustion chamber of the Wankel rotary engine. The simulation has been validated by comparison to the experimental results. The quantity of mesh has been considered and proved. Too rough mesh could not provide satisfactory details of flow for investigation, however, too fine mesh cost time and facility for calculation. Regarding the turbulence models variation, it could be concluded that the LES can predict the flow behavior more closely to the experimental data. In addition, the flow patterns occurred in each position inside the fluid domain correspond with the experimental result. It is also found out that during the intake stroke the flow field is mainly governed by the intake charge forming the counterclockwise circulation in the domain. After the intake port is closed, the flow field is dominated by the movement of the rotor and leakage at both leading and trailing edges only. These two parameters cause the change of circulation direction to clockwise. This specific phenomenon of this engine type influences the oxidiser and fuel to mix together as well.

Acknowledgments
This research is cooperatively done under the DAAD program, called "Praxispartnerschaften zwischen Hochschulen und Unternehmen in Deutschland und in Entwicklungsländern" between the BTU C-S and KMUTNB. The AVL-FIRE software is also supported by the AVL List GmbH under the "AVL AST UNIVERSITY PARTNERSHIP PROGRAM".

References
[1] Meng P, Hady W and Barrows R 1984 An overview of the NASA rotary engine research program NASA Tech. Memo. 83699 (Washington D.C.: National Aeronautics and Space Administration)
[2] Heywood J 1988 Internal Combustion Engine Fundamentals (McGraw Hill)
[3] Yamamoto K 1981 Rotary Engine (Hiroshima: Sankaido Co. Ltd.)
[4] Patiotis E, Kosmadakis G, Rakopoulos C 2012 Comparative analysis of three simulation models applied on a motored internal combustion engine Energy Conversion and Management 60 45-55
[5] Taghavifar H, Khalilarya S and Jafarmadar S 2014 Engine structure modifications effect on the flow behavior, combustion, and performance characteristics of DI diesel engine Energy Conversion and Management 85 20-32
[6] Szmilitscha B, Wang and Mihăescu M 2014 Flow effects due to pulsation in an internal combustion engine exhaust port Energy Conversion and Management 86 520-36
[7] Hamady F, Stuecken T and Schock H 1990 Airflow visualization and LDV measurements in a motored rotary engine assembly J. of Engines part 1 99 163-83
[8] AVL LIST GmbH 2017 FIRE™ CFD Solver (Graz)
[9] Poojitganont T, Berg H and Guo Z 2015 CFD simulation of flow field inside the Wankel rotary engine between intake and compression stroke AVL AST UC 2015 Graz Austria
[10] Han Z and Reitz R 1995 Turbulence modeling of internal combustion engines using RNG-k-ε models J. Combust. Sci. and Technol. 106 267-95
[11] Orszag S, Yakhot V, Flannery W, Boysan F 1993 Renormalization group modeling and turbulence simulations Int. Conf., Near-wall Turb. Flows Tempe Arizona
[12] Cheng Y, Lien F, Yee E and Sinclair R 2003 A comparison of large Eddy simulations with a standard k-ε Reynolds-averaged Navier–Stokes model for the prediction of a fully developed turbulent flow over a matrix of cubes J. of Wind Engineering and Ind. Aerodynamics 91 1301-28
[13] Murthy B and Joshi J 2008 Assessment of standard k-ε, RSM and LES turbulence models in a baffled stirred vessel agitated by various impeller designs Chem. Engineering Sci. 63 5468-95
[14] Khan Z and Joshi J 2015 Comparison of k-ε, RSM and LES models for the prediction of flow pattern in jet loop reactor Chem. Engineering Sci. 127 323-33