HEAT DISTRIBUTION STUDY ON TURBOCHARGER TURBINE VOLUTE

MOHD IBTHISHAM BIN ARDANI

This thesis is submitted to the Faculty of Mechanical Engineering
in partial fulfillment of the requirements for the award of the
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

JANUARY, 2012

To my dearly mother, father, wife and all other family members …………….

ACKNOWLEDGEMENT

First and foremost I would like to express my greatest gratitude to my two supervisors of this project, which are Dr Srithar Rajoo and Prof. Amer Nordin Darus for the guidance during conducting this study. The continuous support, supervision and advice are highly appreciated. With the supervision, I gain abundance of knowledge that hard to find by books.

Also not forget to mention my fellow colleagues who willing to share some ideas which are related to this project. Also millions of thank you to Dr Khalid Saqr who guide me on giving FLUENT tutorials and comments on my simulation works.

Las but not least, I would like to acknowledge the support given by my family and those who contributed and involved directly or indirectly in this project.

ABSTRACT

The aimed of this project is to evaluate turbine's performance based on its actual condition. Holset H3B nozzles turbine geometry was used as simulation model. Turbine's actual working condition was simulated using common computational fluid dynamics analysis software which is FLUENT. Initial analysis was done by one-dimensional and two-dimensional analysis. Further investigation was done in three-dimensional with heat loss via turbine volute by the mode of convection. All the simulation results were compared with established data in order to confirm its validity. The parameters studied are corrected mass flow, turbine's efficiency at different heat cases, temperature distribution along turbine's volute and difference in temperature between inner and outer wall temperature. Temperature difference within turbine's volute is the major factor that deteriorates turbine's efficiency. Since turbine wall is thin, small temperature difference will result to high heat loss.

ABSTRAK

Analisis adalah bertujuan untuk menilai prestasi turbin berdasarkan keadaan sebenar. Geometri turbin model Holset H3B telah digunakan sebagai model simulasi. Keadaan kerja sebenar turbin disimulasi dengan menggunakan perisian analisis dinamik yang biasa iaitu FLUENT. Analisis awal telah dilakukan dengan analisis satu dimensi dan dua dimensi. Siasatan lanjut telah dilakukan dalam tiga dimensi dengan kehilangan haba melalui volut turbin oleh mod olakan. Semua keputusan simulasi dibandingkan dengan data simulasi dan ujikaji oleh penulis lain untuk mengesahkan kesahihannya. Parameter yang dikaji ialah perbetulan aliran jisim, kecekapan turbin pada kes-kes haba yang berbeza, suhu sepanjang volut dan perbezaan suhu turbin di antara suhu dinding dalam dan luar. Perbezaan suhu di dalam volut turbin adalah faktor utama yang menyebabkan kemerosotan kecekapan turbin. Disebabkan turbin mempunyai dinding yang nipis, perbezaan suhu yang kecil akan menyebabkan kehilangan haba yang tinggi. Pekali pemindahan haba juga memainkan peranan penting dalam menentukan kecekapan turbin. Semakin tinggi nilai pekali, olakan yang lebih kuat akan berlaku seterusnya menyebabkan kemerosotan kecekapan turbin.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDIX

LIST OF SYMBOLS

A	-	turbine's volute external area
C_p	-	air specific heat
h_{ext}	-	exhaust flow heat transfer coefficient
h_{amb}	-	ambient heat transfer coefficent
k	-	ratio of specific heat
K	-	thermal conductivity
$\dot{m}$	-	mass flow rate
$\dot{m}_{corr}$	-	corrected mas flow rate
P_{in}	-	turbine inlet pressure
P_{out}	-	turbine outlet pressure
P_{ref}	-	reference pressure
PR	-	pressure ratio
Q_{ext}	-	external heat loss
Q_{cond}	-	heat loss through conduction mode
Q_{conv}	-	heat loss through convection mode
Q_{rad}	-	heat loss through radiation mode
T_{in}	-	turbine inlet temperature
T_{out}	-	turbine outlet temperature
T_{inner}	-	turbine inner wall temperature
T_{outer}	-	turbine outer wall temperature
T_{∞}	-	ambient temperature

T_{ref}	-	reference temperature
V_x	-	velocity in x-direction
V_y	-	velocity in y-direction
V_z	-	velocity in z-direction
W_{act}	-	actual turbine work
W_{isen}	-	isentropic turbine work
ρ	-	air density
μ	-	air viscosity
η	-	turbine's efficiency

CHAPTER 1

INTRODUCTION

1.1 Background of the study

Nowadays, demand on powerful engine has increased enormously due to the ability of the engine to produce rapid acceleration. Power up engine is a method of increasing engine power beyond the ability of normal stock engine. There are several ways to power up engine such as, having bigger cylinder, which mean by increasing the size of bore and stroke, supercharger and turbocharger. Some of these devices assist engine to induce more air into intake manifold. Apparently, the least expensive, easy to main yet producing good output to vehicle is by having a turbocharger.

By the aid of turbocharger, engine can produce more power at the same speed of Naturally Aspirated (NA) engine. Technically speaking, turbocharger forcing more air into combustion chamber thus, this will increase and improve volumetric efficiency (Crouse and Anglin, 1993).

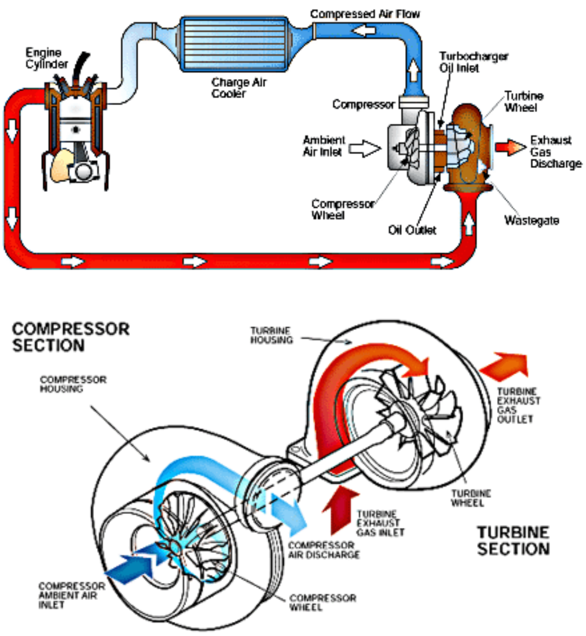

Figure 1.1 Operation of a turbocharger (Source: http://conceptengine.tripod.com)

In turbocharger system as illustrated in Figure 1.1, there are two main parts, which are compressor and turbine. Turbine acts as centrifugal air pump, which is driven by exhaust gas while compressor induced air, compressed it and forced the air to combustion chamber. Both of the parts are connected via main shaft, which is turned by turbine by the flow of exhaust gas that strike turbine's blade. The rotation speed of turbine depends on the speed of high temperature exhaust flow and normally it can achieve to more than hundred thousands RPM. Heat is distributed throughout the whole turbocharger components due to different of temperature between parts. Some of the heat loss through convection to ambient and some of the heat are conducted through components. Heat losses will deteriorate turbocharger performance and specifically on turbine side. Since turbine volute has larger area exposed to ambient, thus it acts as a main source of heat loss.

The main purpose of this research is to study of the temperature and heat distribution at turbocharger turbine's volute. This study will be focused on the temperature variation at the turbine volute. The investigation on heat flow throughout turbocharger components is well studied in this project. The author has made several literature reviews in this topic, which will be discussed in great details after this. Consequently, the author will develop simplified model of turbine volute in one-dimensional and two-dimensional and verified with experimental works. Furthermore, a three-dimensional model is made which mimics the real process that occurs at turbine volute. The analysis will be conducted by MATLAB and FLUENT.

1.2 Objective

To identify the effect of heat distribution and heat transfer within a turbine volute that influences a turbocharger performance.

1.3 Problems Statement

The problem statements for this thesis are

a) Heat loss can be portrayed as lost of energy that can be utilized.
b) Heat loss due to heat transfer should be reduced or minimized to obtain optimum work transfer.
c) Investigation or study of heat distribution is deemed necessary to capture the phenomenon of heat transfer that occurs.

1.4 Scopes

The scopes for this thesis are

a) Initial analysis will be based on numerical calculation in MATLAB
b) Three-dimensional model will be created in FLUENT
c) Volute modelling is based on HOLSET H3B nozzle less volute
d) Only steady state simulation will be conducted

1.5 Methodology

Methodology that have been created is applied throughout this research on the simulation model as showed in Figure 1.2:

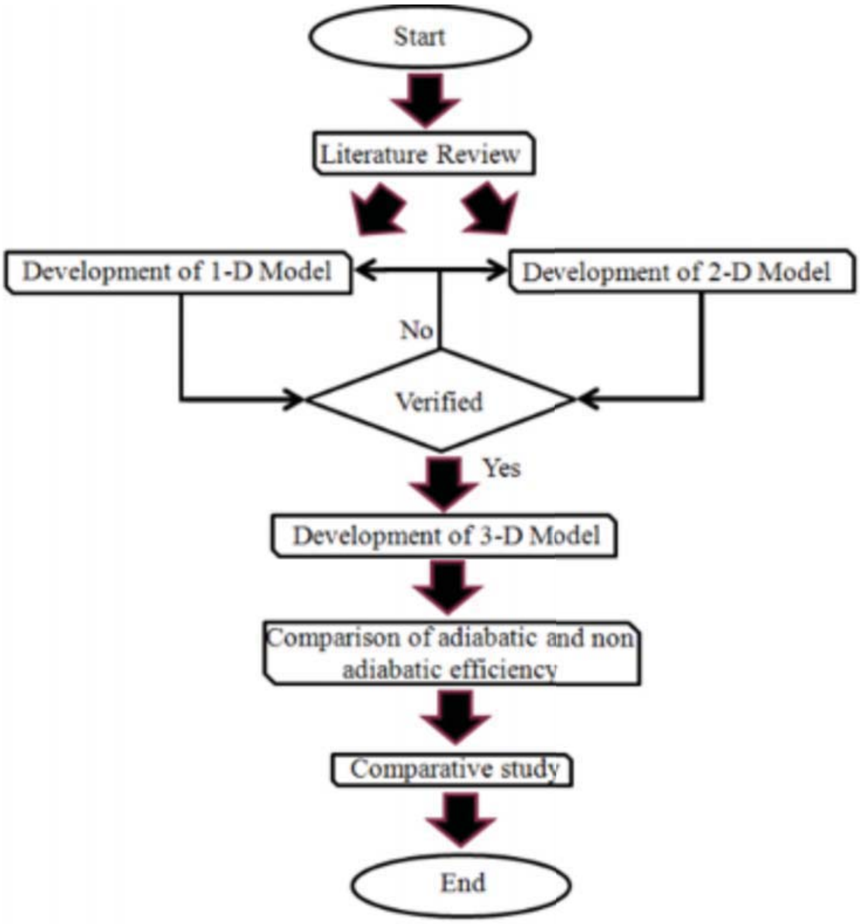

Figure 1.2 Methodology for heat distribution study on turbocharger turbine volute