International Journal For Technological Research In Engineering
Volume 2, Issue 4, December-2014 *ISSN (Online): 2347 - 4718*

A REVIEW ON TURBULENCE ANALYSIS OF IC ENGINE THROUGH CFD APPROACH

Kishan Parmar[1], Prof. V. K. Jani[2]
[1]PG Scholar CAD/CAM Department
[2]Assistant Professor Mechanical Department
C.U. Shah. College Of Engineering and Technology, Wadhwan, Gujarat, India

Abstract: Internal combustion (I.C.) engines typically exhibit a characteristic efficiency profile which varies with operating load and engine speed, and it is widely known that the operating efficiency is poor under low loading conditions. The objective of this project is to investigate whether an energy storing and recovering process, involving compressing air and subsequently using it for propulsion, could be used to achieve better overall efficiency. An engine so modified would operate in two alternate modes. When using fuel, the engine operates as close to maximum efficiency as practicable, with the excess of engine output Over driving requirements being absorbed by air-compression loading - driving an external compressor, charging air into a receiver. Later, under low driving requirements, this air is expanded - using the engine cylinders - as a source of propulsion. Heat transfer from the exhaust gases to the stored compressed air Is used to improve engine efficiency. Through modeling and simulation, an overall efficiency improvement of 10% over standard engine operation is predicted to be realizable by applying this modification, and scope exists to further improve this figure through improved heat recovery from exhaust gases and improved loading capability

I. IC ENGINE SYSTEM: INTRODUCTION

IC Engine System
• A new Workbench Analysis System similar to Fluid-Flow (Fluent) or Fluid-Flow (CFX) Analysis Systems
• Reduces the setup time of ICE cold flow and port flow problems from many hours to few minutes
• First released ANSYS R14
• Supported on Windows and Linux platforms
• Standard feature included with ANSYS FLUENT

II. SCOPE OF IC ENGINE SYSTEM

Automated geometry preparation and mesh generation for all 4 stroke engines
- Any number of valves
- All standard shapes of piston at the given crank angle
• Automated case setup for “cold-flow” and “port-flow “type simulations based on the best practices
– including mesh motion
• User hooks for complex physics setup, e.g. spray injection, combustion simulation
• Automated report generation

III. COLD FLOW SIMULATION USING IC ENGINE SYSTEM

• Automatic preparation of geometry for meshing
• Automatic meshing including inflation layers and layering zones
• Automatic setup dynamic zones, events, and solve settings
• HTML report creation
• Reduces the turnaround time (CAD import to CFD setup) to less than an hour

A. IC Engine System will automatically setup theproblem

– Reads the valve and piston profile
– Create various dynamic mesh zones
– Create interfaces required for dynamic mesh setup
– Set up the dynamic mesh parameters
– Create all the required events, to model opening and closing ofvalves, and corresponding modifications in solver settings and Under-relaxations factors
– Set up the required models
– Set up the default boundary conditions and material
– Set up the default monitors
Initialize and patch the solution

IV. PORT FLOW SIMULATION USINGIC ENGINE SYSTEM

New feature in upcoming ANSYS R14.5 Release
• Prepares the geometry automatically
• Automatic meshing using hybrid and cut-cell Approaches
• Setup and solution strategy based on the best practices
• Automatic saving of important images and HTML report creation
• Reduces the turnaround time (CAD import to CFD setup) to less than an hour

A. Automatic Geometry

Preparation
• Moves the valve to appropriate position
• Deactivates the closed valve and deletes the port automatically
• Removes the piston-bowl (if needed) and extend the cylinder to appropriate length
• Create different shapes of inlet/outlet plenum
• Automatically creates the swirl/tumble planes at thegiven position

B. Automatic Meshing

Cut cell and hybrid meshing support

• Create proper mesh controls and sizing to get better mesh in the chamber and valve gap

• Boundary layers in both hybrid and cut cell meshing

• IC Engine System will setup the solver from the best practices for cut cell and Hybrid meshing

– Set appropriate solver methods and controls

– Set the boundary conditions

– Defines the default monitors

– Does the FMG initialization

• Automatically creates the default swirl plane from geometry information, and defines custom field functions for swirl

C. Regression and Time Statistics:

A strong regression suite

– More than 15 engines with various topologies from different customers are there in our

Regression suite, which runs on daily basis, to maintain the stability and high quality of Software

– For each of these engines geometry preparation, meshing, and setup for cold flow case is within 20min, and for port flow this is within 30 min

V. DOCUMENTATION

• Detailed explanation of all the features with tips on how one can modify the default behavior of the tool

• Trouble shooting chapters: All the knowledge gained since the release of 14 has been captured and documented. Separate sections for:

– Geometry check

– Geometry preparation

– Mesh generation

– Solver setting up

• Well documented process explaining how tool can be extended for some of the features which are not supported by automation

– Decomposing a straight valve engine with pockets for layered meshing

– Handling geometries in which solid valves are missing

• Detail steps for setting up and running the tutorials along with Video Tutorials

A. Advanced solver setup using journal customization.

User will be able to setup advanced physics using pre-iteration and post-iteration journal hooks

- Using pre-iteration journal hooks user should be able to setup combustion problem in IC Engine system:
- Define profile, udf, and chemking, file path and also other variables
- Compile and hook the udf, also define some udf related variables
- Deactivate port fluid zones
- Set up energy model, turbulence model, species model and dpm models
- Define injections

B. Key grid support

Automatic crank angle specific decomposition

– Create mesh as per the crank angle position

– Parametric support to get meshes at different

Crank angles

• You can setup up to the mesh once, and then you can create any number of design points with the exposed parameters like: crank angles, minimum lift, or connecting rod length and update the design points, you will have the appropriate mesh file ready at those given crank angles without any manual intervention

VI. USABILITY FEATURES

Animation of valve and piston motion for the cold flow simulation at geometry level

- Parameter support for port-flow solution and mesh generation in cold flow
- User can start the cold flow simulation from any crank angle; all the settings will be taken care automatically
- This saves a huge amount of time; earlier people use to reach the required crank angle by mesh motion which takes a lot of time
- Automatic cut planes and views in AMP for better visualization of the mesh

VII. CONCLUSION

New “standard” feature in ANSYS-FLUENT for In-Cylinder simulations

- Automates in-cylinder model creation
- Extensively tested on different engine Configurations
- Supported on Windows and Linux
- Quick to learn and easy to use!
- Provides hooks for custom in-cylinder simulations

REFERENCES

[1] J. Heywood, Internal Combustion Engine. Fundamentals. McGraw Hill,

[2] J. Ramos, Internal Combustion Engine Modeling, Hemisphere Publ. Corp.

[3] J. Horlock and D. Winterbone, The Thermodynamics and Gas Dynamics of Internal Combustion Engines., Clarendon Press, Vol II, Oxford,

[4] G. Blair, Design and simulation of four stroke engines, SAE Press,

[5] J. Corberan and M. Gascon, TVD schemes for the calculation of flow in pipes of variable cross-section, Math. Comput. Modelling, Vol. 21, pp. 85-92,

[6] G. Engl and P. Rentrop, Gas flow in a single cylinder internal combustion engine: A model and its numerical treatment, Int. J. Num. Meth. Heat Fluid Flow, Vol. 2,pp. 63-78

[7] N. Nigro, M. Storti and L. Ambroggi, Modelizacionnumerica de un motor de combustion

internamonocilindrico, Revista Internacional de MetodosNumericos para calculo y disegno enIngenieria, Vol. 15, Number 1, pp. 21-54,
[8] R. Benson, The Thermodynamics and Gas Dynamics of Internal Combustion Engines., Clarendon Press, Vol I, Oxford

