ZÁPADOČESKÁ UNIVERZITA V PLZNI

FAKULTA STROJNÍ

KATEDRA ENERGETICKÝCH STROJŮ A ZAŘÍZENÍ

ZÁPADOČESKÁ UNIVERZITA V PLZNI

POWER SYSTEM ENGINEERING, THERMODYNAMICS & FLUID FLOW

2013

JEDNOTLIVÝ PŘÍSPĚVEK ZE SBORNÍKU

evropský sociální fond v ČR

EVROPSKÁ UNIE

MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY

OP Vzdělávání pro konkurenceschopnost

INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

1D MODELS OF THERMOHYDRAULIC SYSTEMS SUPPORTED BY CFD RESULTS

ČIBERA Václav, LÁVIČKA David, KŇOUREK Jindřich, KŮS Michal

This paper reports the way of thermohydraulic systems simulation as a whole real systems consisting of many parts. The crucial demand in our approach is to develop a software within the whole thermohydraulic system could be quickly designed and simulated under different conditions. Therefore new graphical thermohydraulic library under Simulink is being developed. Pivotal point in our approach is implementation of results obtained in CFD simulations and integration of measured data from calorimetric tunnel.

Keywords: thermohydraulic, Simulink, 1D models

Introduction

Last years, the simulations concerning fluid flow or thermal phenomena are in the vast majority of cases computed in FEM solvers. This approach gives undoubtedly the best results in the actual state of art. Disadvantage of this way is mostly a long time in preparation of models also as in the quick rearrangement of model usually because of geometry (re)meshing. Although for aim of this paper the results obtained in FEM solvers are crucial, the main target of this contribution is to show how to avoid these long computation. Therefore it was decided to investigate possibilities of models within are used only one spatial variable and time variable. Common mathematical models involving more than one spatial variable were replaced by: characteristics obtained in CFD /ANSYS computation, measured data in calorimetric tunnel [2] and empirical laws such as Coolenbrook-White equation for instance. The final mathematical background in this approach then consists “only” of a set of ordinary differential equations (ODEs) or of a set of ordinary differential – algebraic equations. This set of equation can be consequently quick solved by common ODEs solvers in MATLAB for example.

Methods

Seven variables play a key role in considered phenomena: volume flow rate [l/min], temperature [K] or [°C], heat flow through components [W], cooling fluid pressure or better pressure drops across element [Pa]. The resting three variables taken into account describe the basic properties of cooling fluid: density [kg/m^3], kinematic viscosity [m^2/s] and specific heat capacity [J / (kg K)]. For the better usage and better transparency in model arrangement, it was decided to develop a new graphical library under MATLAB and Simulink. In brief, Simulink offers graphical environment for MATLAB. In Simulink libraries can be found several specialized graphical blocks. Every of the blocks represents some mathematical expression or function. Mathematical models of the considered problem is assembled by connection of these blocks. Consequently is set duration of simulation, suitable solver and simulation can be launched. In presented library, the blocks represent not only mathematical function or expression but mainly real components and parts, which can be found in real thermohydraulical systems. Developed library is incorporated into Simulink which gives opportunity to use blocks from another libraries of Simulink also.

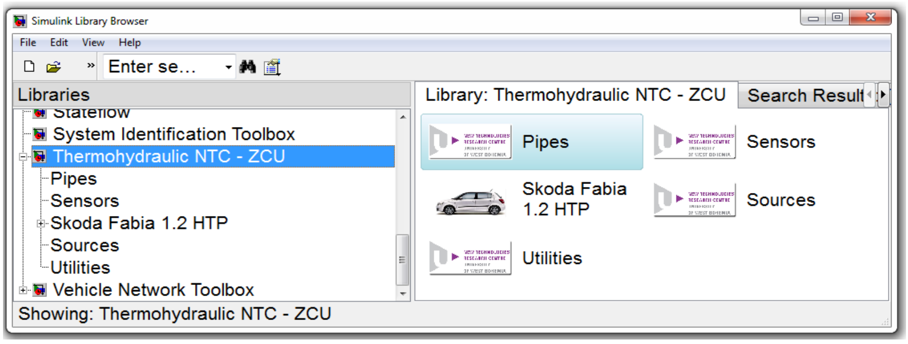

Fig. 1: Thermohydraulic library as a part of Simulink.

Till now library contains elements largely for design and simulation of cooling systems in a cars. As an example was assembled a part of cooling system in Škoda Fabia 1.2 HTP. The purpose of the designed system was to test behaviour of elementary parts such as thermostat, heat exchanger, heating etc. Elements are connected with pipes which could have different properties. In the system arranged like this could be investigated particularly thermal and pressure effect in elements depending on engine load, cooling fluid flow rate, cooling fluid temperature etc. Considered part of cooling system is depicted on Fig. 2.

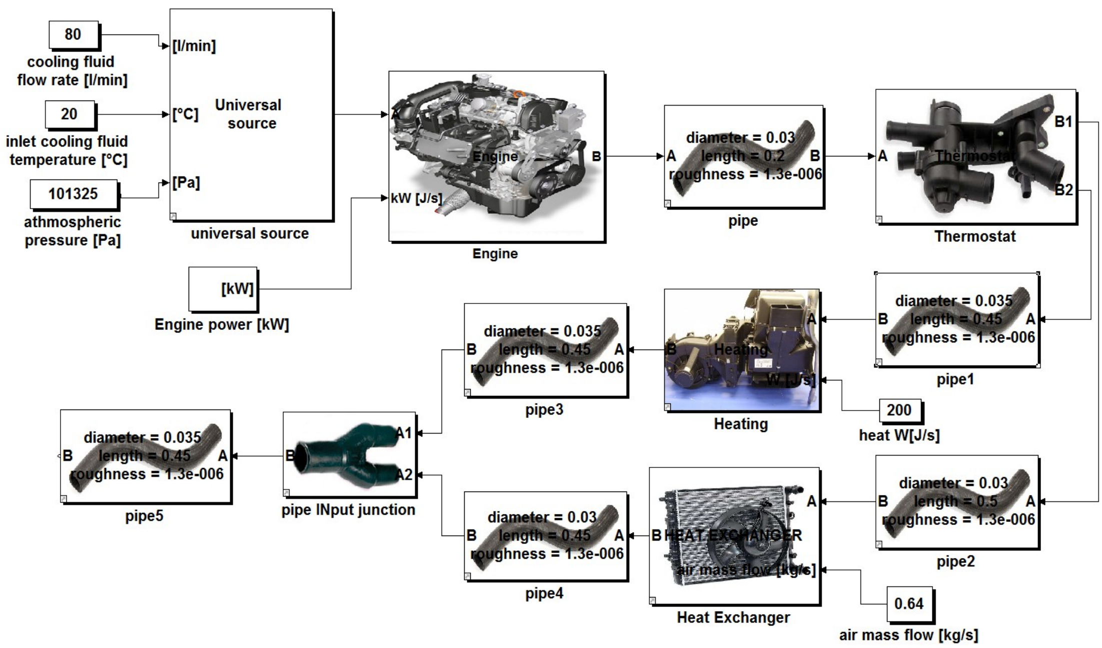

Fig. 2: A part of cooling circuit in Škoda Fabia 1.2 HTP.

Simulation

Several inputs could be set for simulations including characteristic data already incorporated in elements. At the "universal source" (see Fig. 2) block were set some cooling fluid properties as flow rate through system 80 [l/min], initial temperature 20 [°C] and atmospheric pressure 101325 [Pa]. At the block "engine" serve as input engine load in a course of time. This input is depicted on figure 3. Characteristic of engine load is crucial for heat generation inside engine and consequently for cooling fluid heating. In this case, this engine load should characterize driving in a traffic in town in first seconds and then driving out of the town. At “Heating” block was set heat (200 W) removed out of the system. At “Heat exchanger” block was set air mass flow (0.64 kg/s) through main heat exchanger, which is crucial for system cooling. At every “pipe” block was set values for diameter, length and hydraulic roughness. The simulation was conducted for duration of 600 s of simulation time.

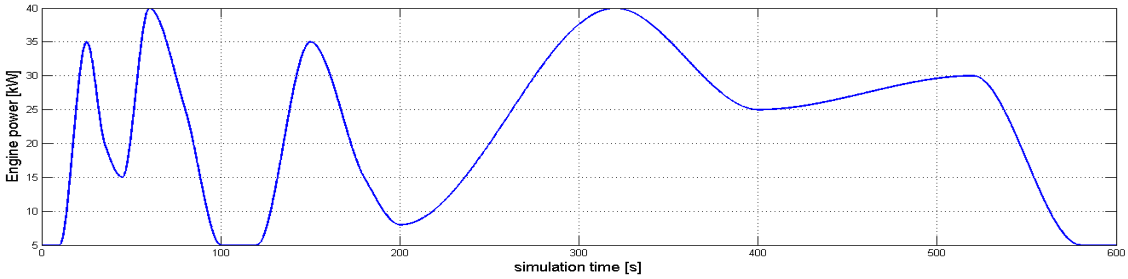

Fig. 3: Engine load [kW] in a course of time.

Results

At every used block on fig. 2 can be depicted graphs of every of seven variables stated above in a course of time. As an example for this paper were chosen two results. The first results presented on Fig. 4 shows pressure drop across element named “pipe” on Fig. 2.

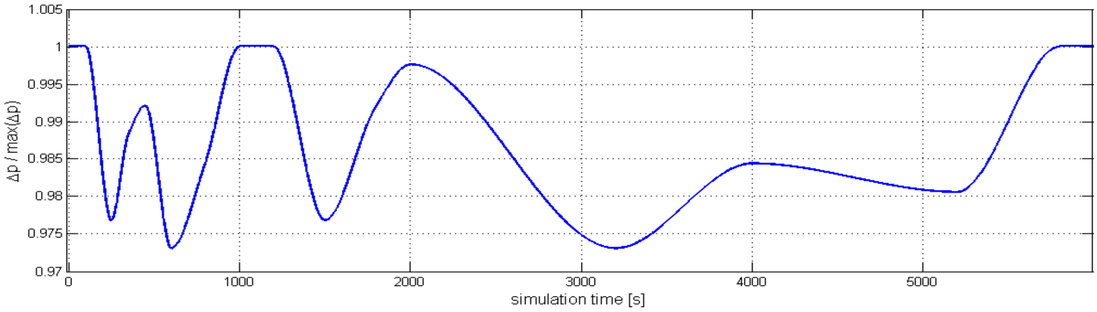

Fig. 4: Pressure drop across element "pipe".

As the second presented result here was chosen the behaviour of variable temperature in block “Heat exchanger”. Temperature in a course of time at this block is depicted on Fig. 5. Both results are presented in relative values.

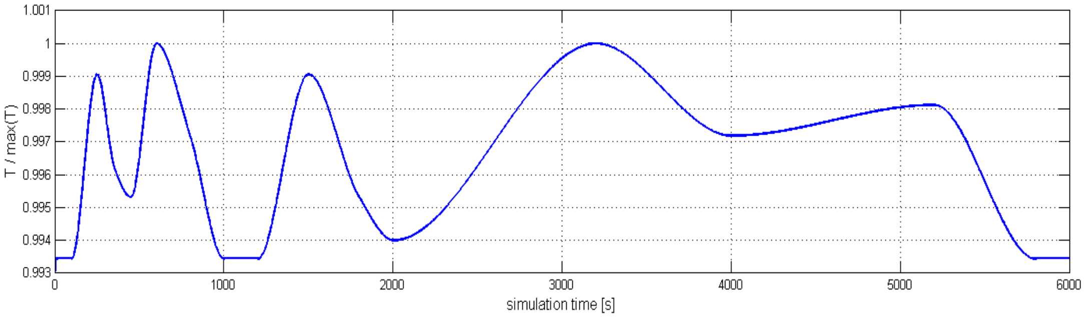

Fig. 5: Temperature in element "Heat Exchanger".

Conclusion

The purpose of the work was to investigate possibilities in quick simulations of cooling systems especially in a car. Easy and quick rearrangement of the whole system played a key role in our approach. Measured (calorimetric tunnel) and obtained (CFD - ANSYS) characteristic were integrated into the used parts of thermohydraulic system. Empirical and 1-D mathematical models were used for computing of flows and pressure drops. In comparison with finite element methods and solvers, this way of simulations has great advantage in short simulation time and flexibility in changes of system topology. On the other hand, there is a less accuracy in results. But for the first attempts in thermohydraulic system design, this approach is found to be useful and sufficient. Thermohydraulic library is further being developed to serve as a standalone part of Simulink.

Acknowledgement

The work was supported by internal grant SGS-2012-072.

Literature

[1] ČIBERA V., LÁVIČKA D., Investigation of the thermohydraulic systems in MATLAB & Simulink using developed library, In Experimental Fluid Mechanics 2012 Volume 45, p.123 – 126, 2012.

[2] KŇOUREK J., KŮS M., SYKA T., Measuring thermal characteristic of the heat exchanger, In Experimental Fluid Mechanics 2011 Volume 25, 2011

[3] MATLAB version R2011 User guide, help (The Mathworks Inc.)

Ing. ČIBERA Václav, ZČU v Plzni, NTC, KME , Univerzitni 8, 306 14 Plzeň, +420726914723, vacibera@ntc.zcu.cz
Ing. LÁVIČKA David PhD, ZČU v Plzni, NTC, Univerzitni 8, 306 14 Plzeň,
Ing. KŇOUREK Jindřich PhD, ZČU v Plzni, NTC, Univerzitni 8, 306 14 Plzeň,
Ing. KŮS Michal PhD, ZČU v Plzni, NTC, Univerzitni 8, 306 14 Plzeň.