

Multi-Dimensional Modeling of Direct Natural-Gas Injection and Mixture Formation in a Stratified-Charge SI Engine with Centrally Mounted Injector

Mirko Baratta, Andrea E. Catania and Ezio Spessa
Politecnico di Torino, Torino, Italy

Lothar Herrmann and Klaus Roessler
Daimler AG, Stuttgart, Germany

Copyright © 2008 SAE International

ABSTRACT

Direct injection (DI) of natural gas (NG) at high pressure conditions has emerged as a high-potential strategy for improving SI engine performance. Besides, DI allows an increase in the fuel economy, due to the possibility of a significant engine dethrottling at partial load. The high-pressure gas injection can also increase the turbulence level of mixture and thus the overall fuel-air mixing. Since direct NG injection is an emerging technology, there is a lack of experience on the optimum configuration of the injection system and the associated combustion chamber design. In the last few years, some numerical investigations of gas injection have been made, mainly oriented at the development of reliable numerical investigation tools.

The present paper is concerned with the development and application of a numerical Star-CD based model for the investigation of the direct NG injection process from a poppet-valve injector into a bowl-piston engine combustion chamber. Different aspects related to the reliability of prediction models are examined, such as: structure and size of the mesh, flow initial and boundary conditions. In this work, the multidimensional simulation tool is applied to the whole relevant volume, including the injector-nozzle internal path. A challenging outlook, concerning the best compromise between computational cost and accuracy, is the meshing strategy of the moving injector, mainly at the needle-seat passage.

The process of model development can be split in three steps. First, a basic numerical study was carried out, in order to characterize the fuel jet properties into an open environment, as functions of both the adopted meshing strategy and flow initialization. Second, an axisymmetric injector-nozzle configuration was designed, with the same jet-formation properties as the starting available design, in order to allow the moving needle simulation with a hand-made, high-quality, fully hexahedral computational mesh. Third, the numerical procedure was validated through the comparison of numerical results with Schlieren images of the jet profile in a constant-volume bomb.

Finally, the results of model application for simulating the methane direct-injection into the combustion chamber of an engine with centrally mounted injector are presented, in the cases of different injection strategies, injector-tip protrusions, piston and cylinder-head shapes. Indications concerning mixture-formation capabilities of the injector and chamber are analyzed.

INTRODUCTION

The air pollution control and energy supply security have increasingly turned out into two fundamental issues for industrialized countries. Natural gas (NG) emerged as the most promising energy source capable of matching both issues in the short and medium term, thanks to its intrinsic environmental features ([1-3]¹). Actually, CO₂ emissions with NG fuelling are about 25% smaller than those with gasoline, for the same efficiency ([4]). A better efficiency, as a result of the greater knock tolerance, is also a reason for choosing NG. On present port-injected engines fuelled with compressed natural gas (CNG), the increase in engine efficiency is of about 3 to 6% ([4]). Besides, the natural gas combustion produces the lowest greenhouse gas emissions among fossil fuels, along with negligible amounts of suspended particles and photochemical smog promoters. The natural gas also offers a number of technical advantages, such as zero evaporative emissions, reduced cold-start and low-temperature emissions (owing to the elimination of cold enrichment), as well as compatibility with lean-burn technology. Nevertheless, natural gas has largely been unexploited in the transportation sector yet, principally because of the fuel gaseous state, which either significantly reduces engine power output or limits the vehicle operating range due to the lower energy density of the stored fuel. In addition, NG fuelling requires both the development of a dedicated distribution network and the adoption of high-pressure tanks for on-vehicle storage. At present, the main mixture preparation technique for gaseous-fueled engines is to premix fuel with air in the intake manifold, just upstream

¹ Numbers in square brackets designate references at the end of paper.

from the inlet valve, i.e., port injection (PI). This practice generally determines lower pressure and temperature which the injector is subjected to, leading to an easier injector design development and also to economically advantageous manufacturing. On the other hand, it leads to lower volumetric efficiency and thus to worse engine performance with respect to direct injection.

NG engine performance can be further improved if direct injection is realized, mainly due to the lower detonation constraints and, hence, to the possibility of increasing the compression ratio and of turbocharging the engine. In addition, stratified combustion strategies allow reducing pumping losses at part load, thus increasing the engine efficiency ([5]).

Previous investigations have been made ([5-13]), aimed at better understanding the physics of gaseous injection in engines through multidimensional modeling. Such an approach is rather complicate, due to the fine grid required in order to capture the evolution of transient gaseous jets ([5, 6]) as well as to the inherent difficulties in turbulence modeling ([7, 8]).

The main parameters determining the type of flow exiting from a sonic nozzle are the pressure expansion ratio and the nozzle geometry ([9]). For simply convergent nozzles, as the injection pressure increases, a condition is reached where the exit flow velocity gets to the speed of sound. At higher pressure ratios, the Mach number at the seat-injector throat remains equal to one, and substantial gas post-expansion takes place from the exit pressure to the cylinder pressure downstream from the nozzle. It is thus important to compute the initial outflow from the nozzle accurately, as it governs the subsequent mixing of the gas jet with the cylinder air.

A virtual-nozzle approach has been used in modeling internal combustion engines as an alternative method for computing the detailed near-field fluid mechanics ([10-12]). In such a case, the computation of the flow field begins downstream from the Mach disk, i.e., the normal shock located at the end of the supersonic flow zone. The boundary conditions at the Mach disk are usually determined assuming a perfectly expanded nozzle with the same mass-flow rate and a nozzle area based on an isentropic expansion of the jet up to the cylinder pressure. This approach decreases the grid density requirements at the nozzle, but requires information about the Mach disk size and location. Besides, it is applicable only to two dimensional slot and hole injection geometries. The most accurate approach is to include the nozzle details in the computations. This is the procedure followed, for example, in [13]. In engines, the length scale difference between the nozzle and the chamber dimension is of order of 100 ([11]). To resolve the details of the velocity profile at the exit of the nozzle, computations should be performed at scales approximately 10 times smaller than the nozzle diameter. As a consequence, the numerical calculation of the nozzle flow details requires a significant computational effort.

PRESENT WORK

The present paper is mainly concerned with the work done by Politecnico di Torino and Daimler AG within the NICE (New Integrated Combustion system for future passenger car Engines) Integrated Project of the EC VI Framework Program. The work is principally focused on the study of the injector and chamber mixing capabilities in DI NG engines by means of numerical simulations. A numerical Star-CD based model was developed for the investigation of the direct injection process of NG from a poppet-valve injector into the combustion chamber of SI engines. The most accurate approach, i.e., the inclusion of the nozzle details into the computational mesh, was followed. The development process of the finite-volume, three-dimensional model consisted of three steps. First, a basic numerical study was carried out, in order to characterize the properties of the fuel jet into an open environment, as functions of both the adopted meshing strategy and flow initialization. Second, an axisymmetric injector-nozzle configuration was designed, with the same jet-formation properties as the starting available design, to allow the moving needle simulation with a specifically hand made, high quality, fully hexahedral computational mesh. Third, the adopted computational procedure was validated through the comparison of numerical results with Schlieren images of the jet in a constant-volume bomb. Finally, the validated numerical model was applied to the simulation of methane direct injection into the combustion chamber of engines with a centrally mounted injector: the effects of injection timing, injector-tip protrusion, piston and cylinder-head shapes on the mixture air-fuel-ratio distribution were analyzed.

RESEARCH SINGLE CYLINDER ENGINE

One of the topics of NICE project was the development of future gas SI IC engines with Diesel equivalent fuel consumption ([14]). Regarding the state-of-the-art of NG vehicles, many manufacturers offer production vehicles which are able to run either with NG or with gasoline. All of these bi-fuel engines work on a stoichiometric basis in order to comply with the required emission level by using conventional three-way catalyst (TWC) aftertreatment systems. The concept for the NICE CNG engines is to follow the lean approach in order to further reduce the CO₂ emission in comparison to the currently available stoichiometric CNG engines. One design key is to reach minimum NO_x emissions by ultra-lean homogeneous or stratified operation with high EGR rates. To meet these goals, Daimler developed and tested, in cooperation with AVL ([15]), a DI CNG engine based on a Mercedes-Benz four cylinder in-line gasoline engine. A prototype single-cylinder engine (SCE) was developed and built-up first for testing. This SCE, featuring a combustion chamber made up of a piston-bowl and a pent-roof cylinder head, was equipped with a centrally mounted, outward-opening poppet-valve injector. The engine compression ratio was adapted to the high knock resistance of CNG and set to a value of 12 for the start of investigations. The value of 13 was also explored for comparison of: performance,

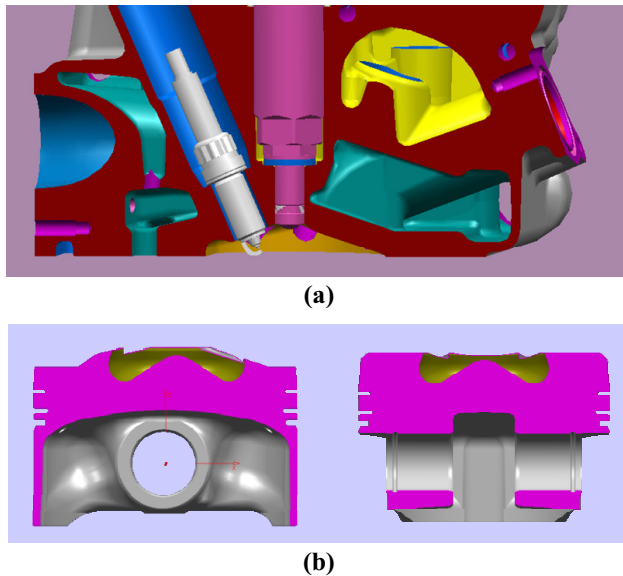


Figure 1 – Schematic representation of the NICE single cylinder engine combustion chamber
(a): Cylinder head; (b): Piston

fuel-consumption and emissions. A representation of the prototype engine can be observed in Fig. 1, while its main characteristics are summarized in Tab. 1.

NUMERICAL MODEL DEVELOPMENT

The simulation of methane DI into an engine combustion chamber is a challenging task, due to the substantial difference between the dimensions of the nozzle and the cylinder and to the high pressure ratios which are used. To resolve the details of the flow in the zone downstream from the nozzle, very fine grids are to be used, so that a very high computational effort is required to simulate the in-chamber injection. The simulation of the methane jet from a poppet valve injector into a relatively large-bore engine combustion chamber was numerically simulated in [13]. However, the injector needle was maintained at its maximum-lift position during the whole injection period, and the quite simple geometry taken for the combustion chamber allowed the use of a variable-size axisymmetric mesh for discretizing the computational domain. At the beginning of the simulation, the injection pressure was imposed in the whole volume of the nozzle from the inlet section to the throat. However, the evolution of pressure wave versus time is not the same as in the real injector, in such a case, and this can have a sensible influence on the simulation results.

In this work, the authors made a preliminary analysis in order to investigate the effects of the wave propagation phenomena inside the injector on the injected mass-flow rate time-history and on the jet shape downstream from the injector. The pressure-wave propagation inside the injector is affected by the inlet boundary-section location with respect to the nozzle-throat section, as well as by the initial flow conditions. In the following, the stages of

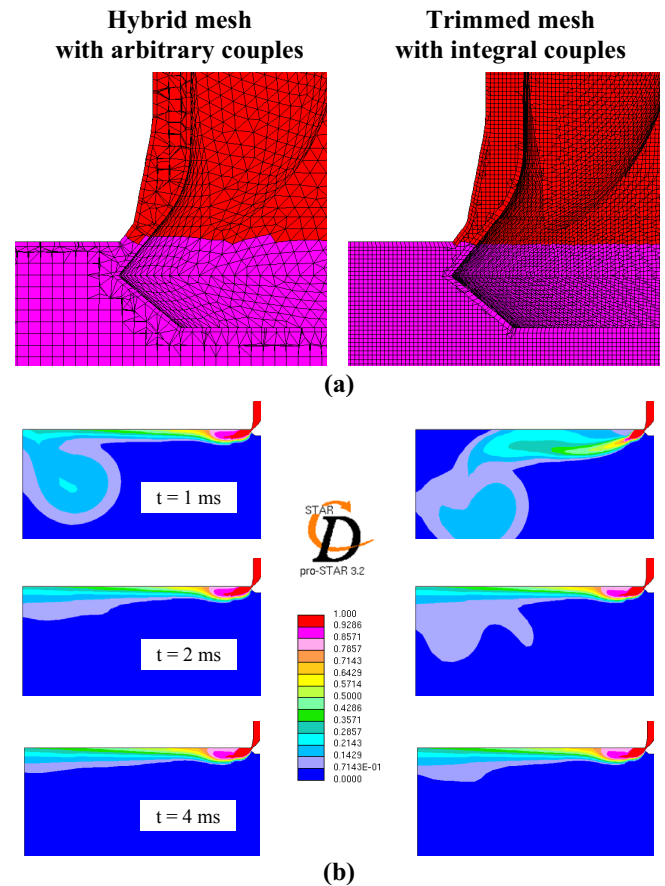


Figure 2 – Influence of mesh structure and size on methane concentration distribution
(a): Detail of the mesh topology at needle-seat passage
(b): Concentration contours at t = 1 ms, 2ms and 4 ms ASI

Table 1 – Single Cylinder Engine characteristics ([15])

Number of cylinders	1
Bore	82 mm
Stroke	85 mm
Conrod length	136.5 mm
Displacement	449 cm ³
Compression ratio	12 - 13

the development process of the numerical model within the present approach are described in detail. Main topics concerning numerical aspects of the computational model are summarized in Appendix 1.

STEP 1: BASIC NUMERICAL STUDY

As a first step of the model definition, a preliminary set of simulations of methane injection from a poppet valve injector into an open environment was carried out. Due to the complexity of the real injector tip geometry, it was initially necessary to fix the needle position during the computations, instead of implementing the actual needle lift variation with time. Furthermore, two different meshing

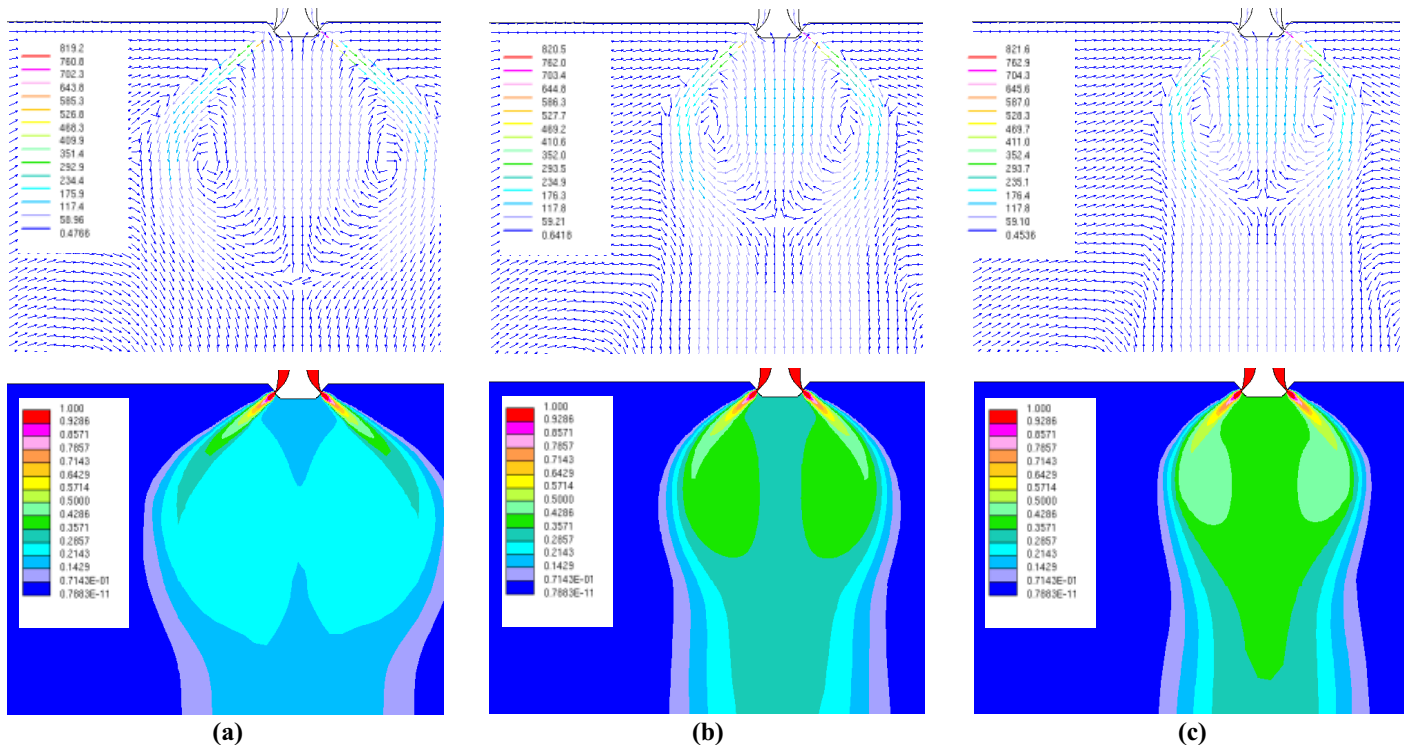


**Figure 3 – Influence of flow initialization strategy – (a) strategy 1; (b) strategy 2, $T = T_0$, (c) strategy 2, $T = 1.25 T_0$
 $t = 2$ ms ASI – Upper row: velocity fields, lower row: methane concentration isolines**

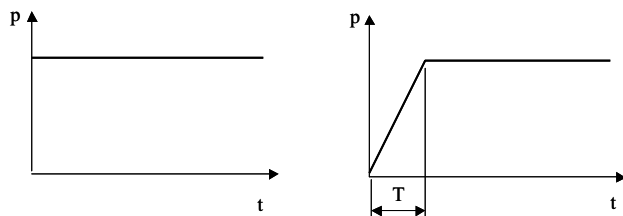


**Figure 4 – Pressure at injector inlet versus time for different flow initialization strategies
 Left: strategy 1, right: strategy 2**

strategies of the control volume in the needle proximity were compared (Fig. 2a). Both were built by means of the automatic mesh generator Pro-AM, which is included in the Star-CD package ([16, 17]). The first mesh featured a fully hexahedral core, which covered the major part of the simulated volume, and a number of tetrahedral-type cells at the core boundaries. This mesh will be indicated as “hybrid mesh” in the following. The interface with the downstream environment mesh was realized by means of “arbitrary couples” ([17]), i.e., a group of cells coupled with their neighbors by means of partial boundaries. This type of mesh is easy to build, particularly when the control volume under investigation presents a complex geometry. On the other hand, the transfer of properties between adjacent cells by means of partial boundaries gives rise to numerical viscosity, and the tetrahedral quality is often so poor that the model robustness can be compromised. The second meshing strategy adopted is the “trimmed mesh” one, in which an hexahedral-mesh is prepared first, so as to contain the relevant control

volume, then it is trimmed in order to obtain the desired shape. In such a case, the mesh does not include virtually any tetrahedral cell and can be merged into the mesh of the environment by means of “integral couples” (without partial boundaries), provided the vertices of the mesh are properly positioned. This kind of grid is more difficult to be prepared with respect to the hybrid mesh, and needs generally a trial-and-error procedure in order to reach a good accuracy and numerical robustness, especially in case of complex geometries.

Figure 2b shows the result of a preliminary test, in which the named meshing strategies are compared. Figure 2a reports the details of the mesh shape and size around the needle-seat passage. Figure 2b compares the computed methane concentration contours, for three different times after the start of injection (ASI). The left picture in Fig.2a shows that the “hybrid mesh” approach gives rise to a generally poor grid quality. In fact, several skewed cells are present, and the transition from the throat zone to the downstream environment is too sharp. Besides, the presence of “arbitrary couples” is a drawback with respect to the “trimmed mesh” approach. Therefore, the trimming meshing strategy should be adopted whenever possible, because of its lower numerical diffusion and increased quality of the interpolations that are used to compute the fluxes at finite-volume boundaries. These advantages are mainly due to the absence of arbitrary couples, and will in turn be beneficial for both the model accuracy and the numerical stability. Nevertheless, the numerical stability of the model is also strongly dependent on the degree of mesh distortion that is usually expressed by means of the

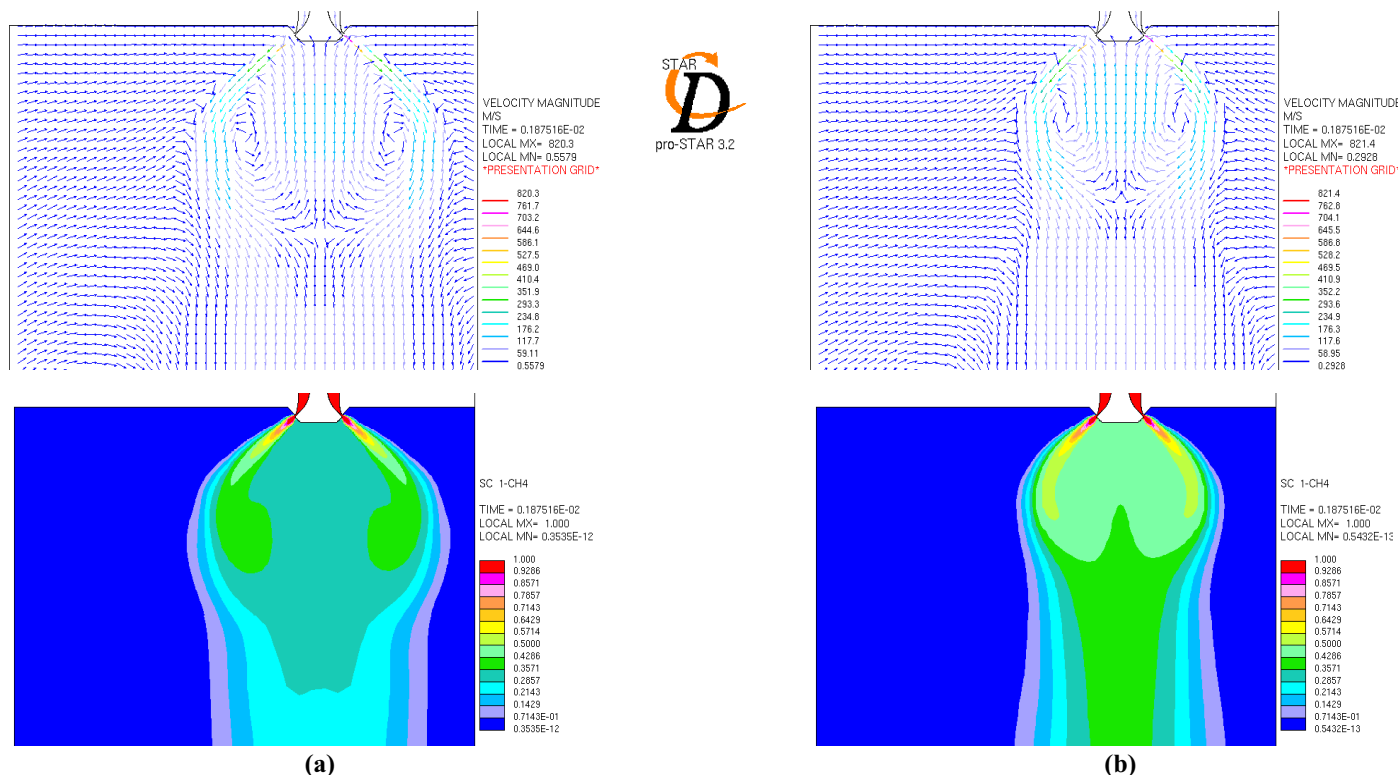


Figure 5 – Influence of the distance between the inlet and the throat section (“Injector length”, L)
Initialization strategy 2, $T = T_0 - t = 1.75$ ms ASI – (a) “Long” injector, $L = L_0$; (b) “Short” injector, $L = 0.45 L_0$
Upper row: velocity fields, lower row: methane concentration isolines

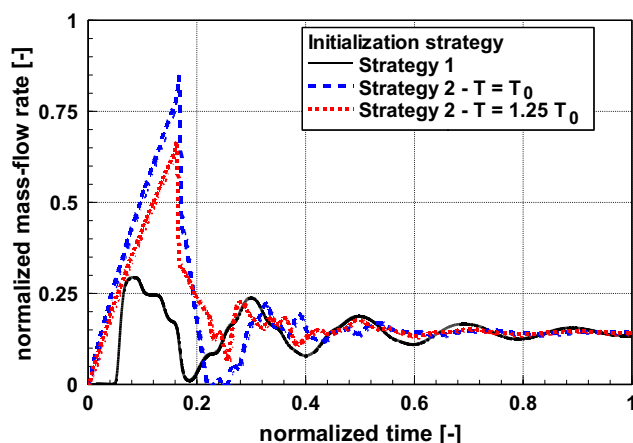


Figure 6 – Normalized mass-flow rate histories for different flow initialization strategies

“aspect ratio”, the “internal angle” and the “warp angle” parameters. Although indicative bound values for these parameters can be provided, the related effects of the mesh distortion also depend on the nature of the local flow. Consequently, since the flow in the region around the throat passage between the needle-head and seat is quite difficult to be numerically handled, the presence of several skewed tetrahedrals in the “hybrid mesh” can constitute a problem.

As can be inferred from the pictures in Fig. 2b, the model results are rather sensitive to the meshing strategy. It

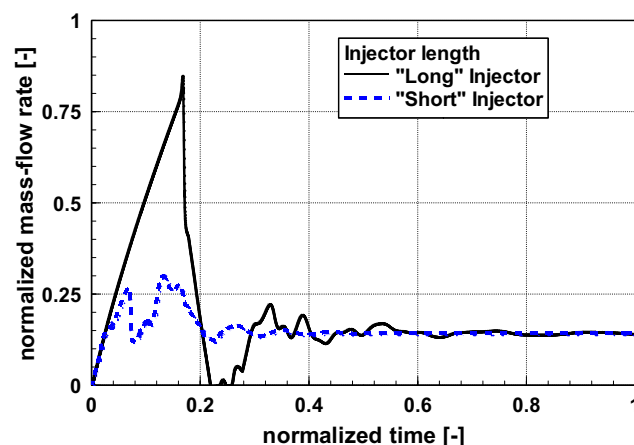


Figure 7 – Normalized mass-flow rate histories for different injector lengths (Initialization strategy 2, $T = T_0$)

should also be stressed that the sensitivity of the results gradually reduces as the injection proceeds, and that very small differences tend to appear when a steady-state condition is approached (Fig. 2b, $t = 4$ ms). However, owing to the fact that in CNG DI engines the duration of the initial transient phase is not negligible with respect to the whole injection duration, a “trimmed mesh” approach should be followed in order to obtain reliable results. In special cases, it should be worth slightly modifying the model geometry to overcome any possible difficulties that the mesh generator can induce at the boundaries of the control volume.

The influence of the initial flow condition on the injection phase evolution is illustrated in Fig. 3, for injection in an open environment. The necessity of such analysis arises when the needle is kept at a fixed position during the calculations, so that the simulation of the gradual injector opening has to be made by the representative simulation of a proper pressure variation at the inlet section of the flow domain. Therefore, two strategies (Fig. 4) have been compared: the “strategy 1”, that is, an already established pressure value inside the injector at the beginning of the simulation, and the “strategy 2”, in which the pressure at the inlet section of the injector is gradually raised from its value in the downstream environment to the nominal upstream pressure value, while keeping the needle lift at its target value. The results of these two strategies are reported in Figs. 3, 6, to show their significant influence on the wave propagation phenomena within the injector, and consequently on the jet shape in the downstream environment. Although a more direct comparison of the total injected methane amount and of the jet concentration contours is not feasible between the strategies, because

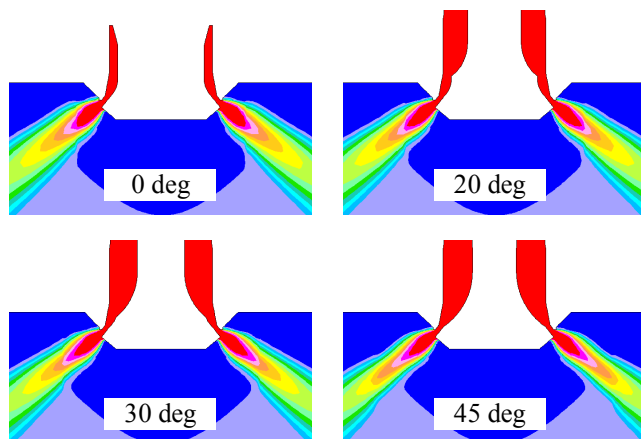
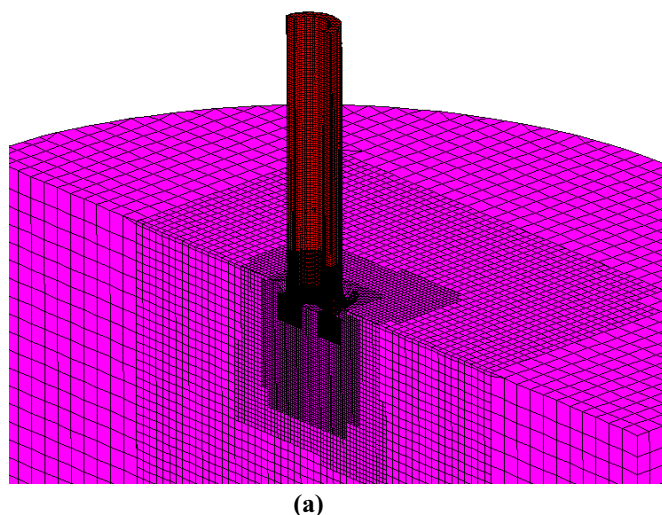
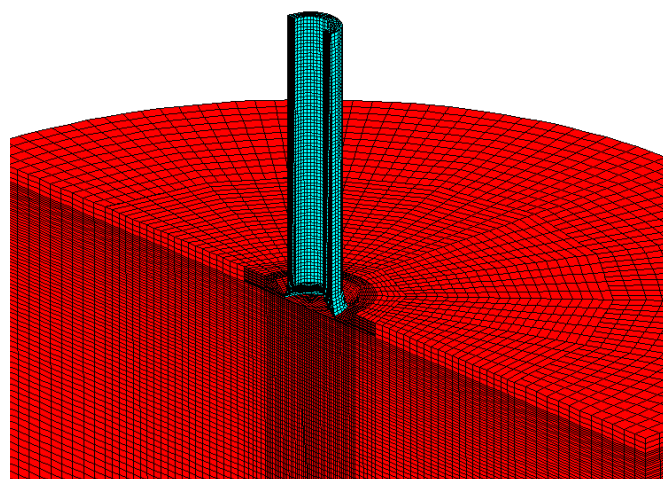


Figure 8 – Angular dependence of flow field for the original injector design



(a)



(b)

Figure 9 – Trimmed meshes, without arbitrary couples, used in the computations

- (a): “Cartesian-based”, automatically built: used in the preliminary analyses and for axisymmetric profile definition
- (b): “Cylindrical-based”, manually built, with controlled cell and vertex numbering: used for moving mesh implementation

of the differences in mass-flow rate time-histories shown by Fig. 6, a qualitative analysis of the flow initialization-strategy influence on the flow field pattern indicates that the jet is longer and narrower if strategy 2 is applied, and that the difference is more evident for larger durations of the pressure rise phase (Fig. 3).

Figure 5 shows the influence of the injector length on the jet methane concentration distribution, with reference to strategy 2. The configurations “long injector” and “short injector” refer to a higher and a lower distances between the inlet and throat sections of the injector, respectively. As can be observed, the jet shape is longer and narrower for the “short injector” configuration.

Figures 6 and 7 illustrate the effects of the flow switch-on strategy and of the injector length, respectively, on the time history of the mass-flow rate at the inlet section of the injector. If the “strategy 1” is applied, the injector is already filled with methane and the mass-flow rate undergoes an initial delay due to the time of propagation of the depression wave which starts from the throat section at injector opening. If the “strategy 2” is followed, the mass-flow rate undergoes a larger initial oscillation because of the injector filling process, but the global duration of the transient phase appears to be shorter. This is physically consistent with a more gradual start-up of the flow, which occurs for “strategy 2”. Concerning the injector length, in the “short injector” case, the variations of the initial mass-flow rate are less pronounced, because there is a smaller flow volume to be filled with methane.

STEP 2: AXISYMMETRIC CONFIGURATION DESIGN

The previous results show that the flow initialization strategy is an important aspect of the numerical model, because it influences the wave propagation phenomena inside the injector. Furthermore, the location of the inlet boundary-condition section is critical.