

POLITECNICO DI TORINO

Corso di Laurea Magistrale in Ingegneria Meccanica

Tesi di Laurea Magistrale

One-dimensional CFD analysis of a spark-ignition engine featuring a deep Miller Cycle and high-power density

**Relatori:**

Prof.ssa Daniela Anna Misul
Prof. Mirko Baratta

Candidato:

Luigi Siodambro

Company supervisor:

Ing. Pietro Bianchi

A.A. 2019/2020

Index

Index.....	2
Index of figures.....	4
Index of tables.....	7
1. Introduction.....	8
1.1 Previous work.....	9
2. Internal combustion engine.....	11
2.1 Spark ignition engine.....	13
2.1.1 Working cycle.....	14
2.1.2 Influence of geometrical-constructive parameters on the duration of the combustion process.....	16
2.1.3 Knocking phenomena.....	18
2.1.4 Supercharging.....	20
2.1.5 Miller cycle.....	21
3. Hybrid vehicle.....	24
3.1 Series HEV.....	27
3.2 Parallel HEV.....	29
3.3 Complex HEV.....	30
3.4 Motivation for HEVs.....	31
4. CFD Analysis.....	33
4.1 Discretization methods.....	34
5. Ricardo Wave software.....	37
5.1 Flow elements description.....	38
5.2 Simulation control & constant.....	39
5.3 Ambient element.....	42
5.4 Duct & Y-Junction.....	43
5.4.1 Y-Junction.....	45
5.5 Valves.....	46

5.6	Cylinder Panel.....	48
5.7	Engine general panel	51
5.8	Mapless Compressor	57
5.9	Mapless Turbine	59
5.10	Turbo shaft	61
5.11	Injector.....	62
5.12	Sensor	63
6.	Wave Model	64
6.1	Valve Timing	69
7.	Optimization process using HEEDS.....	75
7.1	HEEDS	75
7.1.1	Variables	76
7.1.2	Responses	78
7.1.3	Objectives	79
7.1.4	End of the run	82
8.	Results	83
9.	Conclusions.....	92
9.1	TZM Alloy	95
9.2	Hexoloy SE SiC.....	96
10.	Future works.....	98
10.1	Setting up the WAVE model	100
11.	Bibliography.....	102

Index of figures

Figure 1 Working cycle	14
Figure 2 Cylinder pressure versus crank angle for over advanced spark timing.....	15
Figure 3 Maximum brake torque timing	16
Figure 4 Pressure trend during the three phases.....	17
Figure 5 Location of the 50% mass fraction burned	17
Figure 6 Comparison among normal combustion, slight knock and intense knock	18
Figure 7 Miller cycle	22
Figure 8 EIVC vs LIVC	23
Figure 9 Forces acting on the vehicle	24
Figure 10 Engine operating map	25
Figure 11 Series HEV	27
Figure 12 Series HEV degree of hybridization	28
Figure 13 Parallel HEV	29
Figure 14 Parallel HEV degree of hybridization.....	30
Figure 15 Complex HEV	30
Figure 16 Simulation of aerodynamic package of a Porche Cayman	33
Figure 17 Ricardo Wave Software logo	37
Figure 18 Elements & Model panel.....	38
Figure 19 Simulation Control – General Parameters	39
Figure 20 Simulation Control - Fluid Properties.....	39
Figure 21 Simulation Control – Convergence.....	40
Figure 22 Constants Table.....	40
Figure 23 Ambient Panel – Ambient	42
Figure 24 Ambient panel – Initial Fluid Composition	42
Figure 25 Duct Panel	43
Figure 26 Pressure wave	44
Figure 27 Simple Y-junction Panel.....	45
Figure 28 Valve element.....	46
Figure 29 Lift valve editor.....	46
Figure 30 Valve Lift & Flow Coefficient Profile.....	47
Figure 31 Cylinder Panel – Liner Geometry	48
Figure 32 Cylinder Panel – Head Geometry	48
Figure 33 Cylinder Panel – Piston Geometry	49

Figure 34 Cylinder Panel – Initial Conditions & Valves.....	49
Figure 35 Cylinder Panel – Sub Models.....	50
Figure 36 Engine General Panel – Geometry	51
Figure 37 Engine General Panel – Combustion & SI Wiebe Combustion Model	53
Figure 38 Engine General Panel - Emissions	54
Figure 39 Peak cylinder pressure vs Knock intensity	56
Figure 40 Engine General Panel – Heat Transfer & Woschni Transfer Model	56
Figure 41 Turbocharger Mapless Compressor Panel	58
Figure 42 Turbocharger Mapless Turbine Panel	59
Figure 43 Turbocharger Mapless Compressor Panel	61
Figure 44 Proportional Injector Panel - Operating Point & Properties	62
Figure 45 Sensor Panel.....	63
Figure 46 Wave Model	64
Figure 47 Global Structural Conduction Panel	65
Figure 48 Engine General Panel – Conduction, Cylinder Walls Tab	66
Figure 49 Engine General Panel – Conduction, Cylinder Cooling Tab.....	67
Figure 50 Engine General Panel – Conduction, Valves Tab.....	68
Figure 51 Total Power vs IVO	69
Figure 52 Overall BSFC vs IVO	70
Figure 53 Total power vs Intake duration	71
Figure 54 Overall BSFC vs Intake duration	71
Figure 55 Total power vs EVO	72
Figure 56 Overall BSFC vs EVO	72
Figure 57 Total power vs Exhaust duration.....	73
Figure 58 Overall BSFC vs Exhaust duration.....	74
Figure 59 HEEDS - Variables	77
Figure 60 HEEDS – Study	77
Figure 61 HEEDS - Responses.....	78
Figure 62 Exporting the project to HEEDS.....	79
Figure 63 HEEDS Interface.....	79
Figure 64 Setting the method	80
Figure 65 Setting the objectives.....	80
Figure 66 Running the study	81
Figure 67 End of the run.....	82
Figure 68 Pareto curve	83

Figure 69 Design Table	84
Figure 70 Correlation table	85
Figure 71 Pareto curve, TEXH limit set to 1450 K.....	88
Figure 72 Pareto curve, TEXH limit set to 1550 K.....	89
Figure 73 Objective history, TEXH = 1350 K	90
Figure 74 Objective history, TEXH = 1450 K	90
Figure 75 Objective history, TEXH = 1550 K	91
Figure 76 Valve Lifts	93
Figure 77 Linear p-V diagram	93
Figure 78 Cylinder pressure trend.....	94
Figure 79 Cylinder temperature trend	94
Figure 80 Ricardo Vectis.....	98
Figure 81 Vectis work-flow.....	99
Figure 82 External CFD set-up.....	100
Figure 83 External CFD set-up panel	100

Index of tables

Table 1 Previous work – results	10
Table 2 Parameters of Chen-Flynn correlation	52
Table 3 First design results.....	85
Table 4 Second design results	87
Table 5 Design chosen.....	92

1. Introduction

The following is the result of the internship work done at Leonardo engineers for integration. The aim of this thesis work is to optimize the performance of a spark ignition engine, by looking for the right trade-off between the overall brake specific fuel consumption (Overall BSFC) and the total power, which will be used in a range-extender hybrid vehicle.

The model is designed in such a way that the power produced by the engine and the net power of the turbo-group, consisting of three compressors and one turbine, are used by two electric motors. The net power produced by the turbine has to be recovered by the system using the combustion gases energy in excess respect to that required by the compressors, so in order to do that, the electric motors are used as generators. The model features a deep Miller cycle to improve the engine performance by increasing the boost pressure and decreasing the exhaust gas temperature.

The first step was to study the model, in particular by looking for which parameters influenced the engine performance the most, once identified, the second step was to conduct an optimization process, in order to find the optimal working point in which the overall brake specific fuel consumption is minimized and the total power (calculated as the sum of the engine brake power and net power of the turbo-group) is maximized.

The software that has been used for this work is called “Wave”, we will discuss later about it in another chapter.

1.1 Previous work

In this chapter the previous work conducted by Salvatore Chierchia will be reassumed. Taking into consideration the same model that will be the subject of this thesis, Chierchia's objective was to minimize the overall brake specific fuel consumption (BSFC), knock intensity and exhaust gas temperature.

The optimization has been implemented through these five steps:

- use of the materials (taking into account all the thermal properties) of which the engine components are made;
- intake valve timing - law of real lift;
- implementation of the conduction model;
- appropriate sizing of the intake and exhaust ducts;
- exhaust duration.

After these five steps, the optimization process has been realized by using Heeds, which is an optimization software package that allows to search for better and more robust solutions within a given design space.

This analysis was conducted considering the objectives mentioned before, i.e. to minimize the overall BSFC, the knock intensity and the exhaust gas temperature. The following table shows the results found:

Brake power [kW]	20.8
Turbogroup Net power [kW]	1.57
Overall BSFC [g/kW/hr]	202
Turbine produced power [kW]	8.02
Turbine inlet temperature [K]	1306
Exhaust gas temperature [K]	1399
Knock intensity [-]	0.194

Peak cylinder pressure [bar]	108.7
Engine efficiency [%]	40.8

Table 1 Previous work – results

The greater problem is that the brake power of the engine isn't enough for traction. The same discussion can be done for the power produced by the turbine. Having a net power of the turbo-group equal to 1.57 kW means that almost all the power produced by the turbine is absorbed by the three compressors, giving little power to the electric machine. These low power values are a consequence of having minimized the overall BSFC. Another problem encountered concerns the intake valve lift value, which was very low, this was due to having found, at the end of the optimization process, a low intake duration value, which affects the lift values, worsening, in turn, the engine performances. This has happened because a relation between intake duration and intake valve lift, found by Antonio Tricarico in his work, has been considered. According to this relation the intake valve lift value changes with the intake duration, as the following equation shows:

$$y = 0,2517 \cdot e^{0,0122 \cdot x}$$

Where y is the maximum valve lift and x is the intake duration value.

Having found, at the end of the optimization process, an Intake duration value of 100,2 deg, the maximum intake valve lift value resulted, from the equation below, 0,85 mm

As a result, my work will be to find the right trade-off between the overall BSFC and power, and also to improve the intake valve lift value, by eliminating the relation mentioned above, and considering for both the intake and exhaust valve lift, the maximum value of 4,5 mm.

2. Internal combustion engine

Internal combustion engines, denoted by the acronym “ICE”, are the main protagonists of terrestrial propulsion.

These kinds of engines are able to convert the chemical energy of a fuel into mechanical energy for a variety of systems, through a combustion process, in fact the heat generated by the oxidation of the elements composing the fuel, such as carbon and hydrogen, produce an increasing in temperature, thus the enthalpy created is used to create mechanical work, thanks to the fuel expansion.

It is, therefore, possible to classify these engines as volumetric machines, in which the volume variation, necessary for the realization of the working cycle, is obtained by the alternating motion of a piston inside a cylinder closed, at its top, by a warhead. The movement of the piston is obtained through an ordinary and centered crank mechanism. When the crank performs a full 360-degree turn around the motor axis, the plunger performs a complete two runs between two extreme positions: top dead center (TDC), where the volume of dead space is delimited, and bottom dead center (BDC), where the plunger delimits the maximum volume that can be reached in the combustion chamber.

The cylinder is put in communication with the external environment through intake and exhaust valves, which are opened at each cycle and allow the change of the engine fluid. Initially, the fresh charge, consisting of a mix of fuel and combustion air taken from the environment, is introduced through the intake valves, then at the end of the cycle the flue gases are expelled through the exhaust valves back into the external environment.

It is possible to classify ICE according to different criteria such as:

- **ignition mode**: in this case the ICE is classified as: Spark ignition engine (SI), in which the combustion is generated by an external energy source, represented by a spark; Compression ignition engine (CI), where combustion is generated spontaneously by the compression of the fuel/air mixture without the aid of external energy sources.
- **duty cycle duration**: two-stroke engines, i.e. two piston strokes to describe a duty cycle; four-stroke engines, four piston strokes.
- **air or fuel supply system**: regarding the air supply mode, we can consider aspirated or supercharged engines; as for the fuel supply mode we have port fuel injection and direct gasoline injection engines.

The type of ICE considered in this work was a four-stroke supercharged SI engine.

Some engine performance parameters will be now reported:

- **Brake power:** $P_e = bmep \cdot (iV) \cdot \frac{n}{m} [\text{kW}]$
- **Brake mean effective pressure:** $bmep = \lambda_v \cdot \eta \cdot \rho_a \cdot \frac{H_i}{\alpha} [\text{bar}]$
- **Volumetric efficiency:** $\lambda_v = \frac{m_a}{\rho V i}$
- **Air/fuel ratio:** $\alpha = \frac{m_a}{m_f}$
- **Fuel conversion efficiency:** $\eta = \frac{P_e}{\dot{m}_f H}$
- **Brake specific fuel consumption:** $BSFC = \frac{1}{\eta H}$
- **Overall Brake specific fuel consumption:** $overall\ BSFC = \frac{\dot{m}_f}{(P_e + P_{turbine_net})}$
- **Compression ratio:** $CR = \frac{V + V_c}{V_c}$

With:

- n = rotational speed [rpm];
- i = number of cylinders;
- V = displacement [m³];
- V_c = clearance volume [m³];
- ρ = density [kg/m³];
- m_a = air mass [kg];
- m_f = fuel mass [kg];
- \dot{m}_f = fuel flow [kg/s];
- H = lower heating value [MJ/kg].

2.1 Spark ignition engine

A low reactivity fuel is used in these engines. This means that these fuels, mixed with combustion air, although under high pressure and temperature conditions (20÷40bar and 700K), do not spontaneously give rise to combustion reactions.

The fuels used have a common characteristic with regard to their chemical structure: they have an extremely rigid and compact molecule, such as methane CH₄, with all its derivatives (methanol, ethanol), and gasoline, which is a mixture of hydrocarbons produced by distillation of petroleum.

Because of the molecules' stiffness and compactness composing the fuel, the oxidation process takes place through a whole series of intermediate steps, the more rigid and compact the molecule and the more these intermediate oxidation processes take time (we always talk about milliseconds).

Given the impossibility of obtaining a spontaneous combustion process, resulting from the compression of the charge inside the cylinder, the only way to start it is to trigger it from the outside by setting off an electric spark between two electrodes, i.e. by applying at the appropriate time between the two electrodes a potential difference in the order of a few hundred volts. The electric arc created invests the air/fuel mixture and allows the first core to reach very high temperatures ($T \gg 10^3 \text{K}$), thus the oxidation process starts instantly. The process, once triggered, propagates to the rest of the charge in such a gradual and progressive way: the first core transmits heat to the mixture layer immediately adjacent, by convective heat exchange, this causes a raising in temperature to such a level to trigger the combustion reactions there as well, in this way, layer after layer, through a process that is indicated as flame front propagation, the whole charge is burned.

2.1.1 Working cycle

SI engines are also called “Otto cycle engines”, because they refer to its thermodynamic cycle. We can identify six phases of this cycle, as the following figure shows:

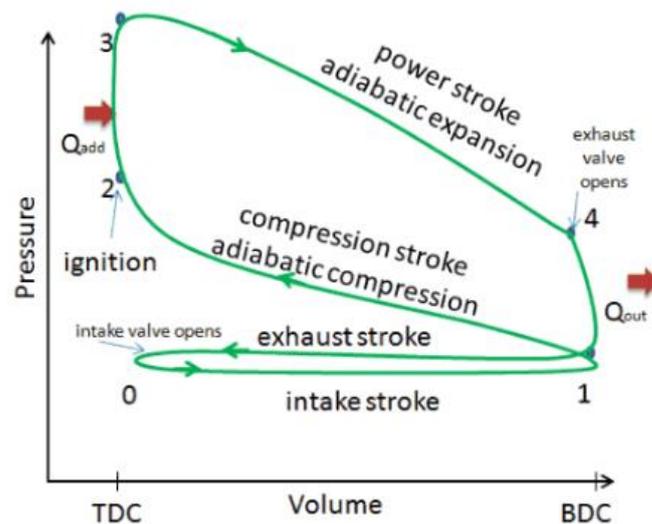


Figure 1 Working cycle

As regards the intake and exhaust phases, it should be noted that the valve lift profiles provide for gradual opening and closing, for this reason, since opening or closing are not instantaneous, during the intake phase, the inlet valve opens before the TDC and closes after BDC, the same happens to the exhaust phase: the opening takes place before BDC and the closing happens after TDC. After the intake stroke, the compression phase will take place, in which the residual burned gases of the previous cycle, will be compressed along with the previously air/fuel mixture aspirated; before the compression phase ends, the combustion process begins. Since, as mentioned before, the combustion process will take place through a series of intermediate steps, to take this into account, the spark plug must be triggered between 10 and 40 crank angle degrees before TDC. Due to the laminations during the exhaust and intake phase the fluid exchange process will produce a negative work, also called “pumping work”.

As it has been stated above, the spark that will trigger the combustion process, will have to be released before the TDC, that is due to the fact that we want the heat, generated from the flame, to be released as much as possible around the TDC.

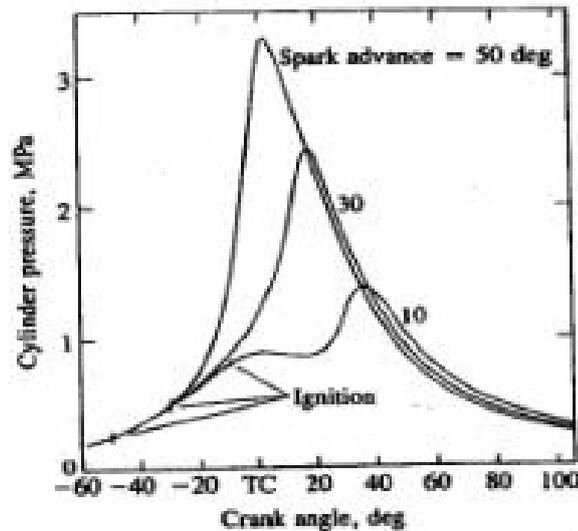


Figure 2 Cylinder pressure versus crank angle for over advanced spark timing

The figure above shows the pressure trend in the combustion chamber as a function of the crank angle, as the spark advance varies, in particular when the latter is 10 deg, 30 deg and 50 deg. It can be seen that if the advance is low (10 deg) the combustion process tends to develop during the expansion phase, when it won't be possible to gather work; moreover, lower pressure levels will be reached due to the fact that the volume is rising. It will be possible to increase the pressure in the combustion chamber and collect more work, by increasing the spark advance, this happens because the combustion process will take place during the compression phase, so when the volume decreases. As the spark advance increases, the pressures will tend to increase even during the final stages of the compression stroke, as a result, in this phase, there will be an increasing in compression work, i.e. a spent work. This effect is more pronounced as the spark advance keep on increasing (50 deg). It will be necessary, therefore, to find a trade-off, for the choice of the advance value, between the increase of the expansion work and the increase of the compression work.

In order to find the best trade-off, it's necessary to look at the following figure that shows the torque trend as a function of the spark advance:

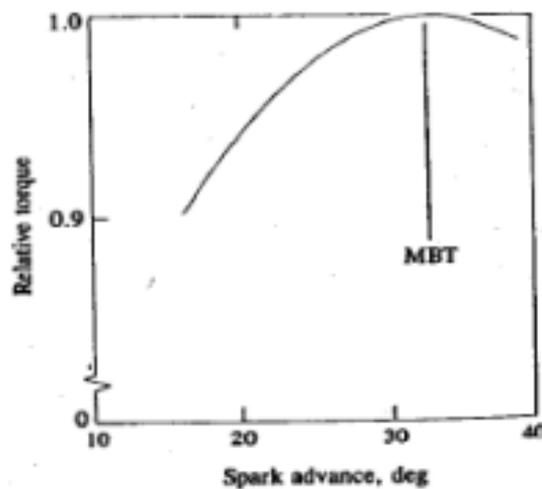


Figure 3 Maximum brake torque timing

It's possible to find out that the best trade-off is represented by the maximum reached by the curve, i.e. the maximum brake torque timing (MBT). It was tested, empirically, that the optimal timing should allow the cylinder pressure to reach its peak 8-10 deg after TDC.

2.1.2 Influence of geometrical-constructive parameters on the duration of the combustion process

It's possible to divide the combustion process in three angular intervals:

- **Development phase (θ_1):** in which the first mixture nucleus burns. This phase corresponds to the angular interval that goes from the spark bursting, to the instant in which the deviation of the pressure from the course of simple compression, becomes clearly perceptible. To define the end of this phase, conventionally we refer to the achievement of an appreciable quantity of burnt mass (5÷10% of the mass).
- **Rapid burning phase (θ_2):** from the first nucleus, the flame front proceeds in a progressive and gradual way, the combustion process spreads out layer after layer until the remaining part of the mixture. This phase ends when we will reach the 90% of the burnt mass.
- **Termination phase (θ_3):** At this stage the final phase of the combustion process takes place, which corresponds to the completion of the reactions after the flame front has reached the last fraction of mixture.

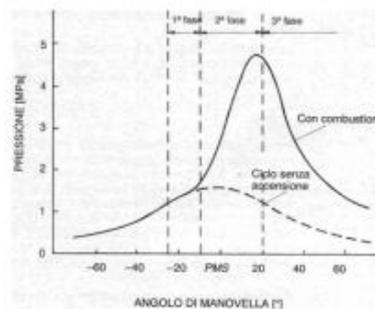


Figure 4 Pressure trend during the three phases

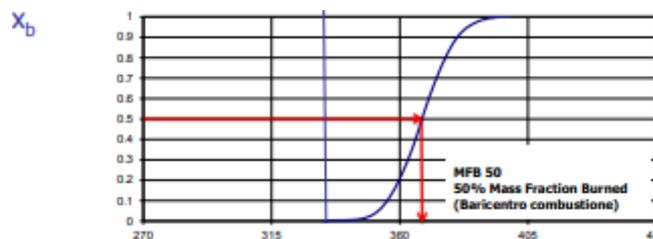


Figure 5 Location of the 50% mass fraction burned

The *figure 4* shows in detail the pressure trend in the combustion chamber as the three phases pass; The *figure 5* shows instead the mass fraction burned (X_b), in particular when the moment in which the 50% of the mass fraction is burned, that represent the barycenter of combustion process; it also provides information about how good is the combustion timing.

There are some geometrical-constructive parameters influencing the combustion process:

- **Compression ratio:** as its value increases, the combustion process will be faster and it will be possible to reach higher peak pressure and temperature. In the first analysis it's possible to say that the higher the value of the compression ratio, the better the combustion process will be, that is not true. The increase of the compression ratio will be limited by knock phenomena, which will occur because of the too high pressure and temperature reached during the combustion process.
- **Shape and size of the combustion chamber:** There is a limitation on the maximum size of the combustion chamber which, as far as the bore of automotive engines is concerned, is approximately 100 mm. This is because as the bore increases, the path that the flame front has to cover is lengthened, increasing the duration of the second phase of the combustion process discussed before. This will lead to a slower process and also to a greater risk that the end gas, i.e. the mixture portion which will be the last to burn, ignites spontaneously leading to

detonation. Regarding the chamber shape, it should be designed to be as small and compact as possible, with candle positions that minimize the distance that the flame front must cover. The most adopted solution is the pent-roof one with four valves placed on the roof flaps and spark plug in a central position.

2.1.3 Knocking phenomena

The most dangerous and most limiting anomaly in SI engines is the knocking phenomena. This is a combustion anomaly represented by the simultaneous self-ignition of the last fraction of the mixture, also called end-gas, before it is reached by the flame front, giving rise to a sudden increase in pressure inside the cylinder, which causes pressure oscillations that vibrate the walls of the combustion chamber, causing a characteristic metallic noise. The knock is responsible for damage to mechanical organs due to hot fatigue phenomena.

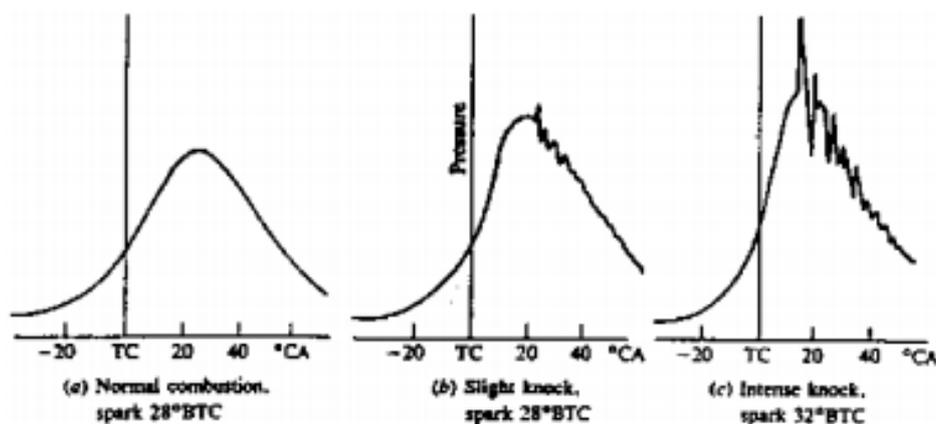


Figure 6 Comparison among normal combustion, slight knock and intense knock

The *figure 6*, in which the pressure trend as a crank angle function is represented, shows three cases of combustion a), b), c), in particular:

- a) This represents a normal combustion process in which it is possible to appreciate how the flame front progressively advances inside the chamber involving, layer after layer, the mixture, giving rise to a progressive and gradual release of energy;
- b) If the flame front takes too long to reach the farthest fractions of the mixture, or if too high pressure and temperature values are reached inside the chamber, reactions start to take place and the mixture starts to self-ignite. In this case we have a light detonation;
- c) This case represents a more marked detonation, also called intense detonation.

There are several geometric-constructive and operating parameters that affect detonation.

Regarding the first type, are included: size and shape of the combustion chamber. The greater the size of the chamber and the greater the length of the path that the flame front must cover from the candle until it reaches the end-gas, therefore greater risk that knock occurs. That is the reason why the bore must not exceed 100 mm. The chamber' shape must be as compact as possible. Chambers offering turbulent motions around the TDC are preferred (high squish chambers). The optimal choice are pent-roof chambers. The position of the candle must be designed in such a way as to minimize the path of the flame front, in this case central candles are adopted. The last parameter to consider is the compression ratio; excessive high values cause more stress on the end-gas, due to the high pressures and temperatures reached in the chamber, so the risk that knock occurs increases.

As for the operating parameters, we consider: the ignition timing; as the ignition timing increases, the pressures and temperatures in the chamber increase as well. An important contribution is provided by the air/fuel ratio, the closer it is to the stoichiometric value, the more reactive the mixture will be. The last one to be considered is the rotational speed. At low rpm there are low ignition advances and low fillings and therefore the knock is not particularly critical.

At intermediate rpm there is a particularly critical situation because we reach the maximum filling, and the advance that continues to grow. At high rpm the detonation naturally tends to decrease, along with the filling and, consequently, these effects prevail over the fact that the advance increases.

In order to detect the risk of knocking, it is possible to use a pressure sensor in the combustion chamber, or by means of measurements, made with accelerometers, of the vibrations induced on a monoblock, allowing to control several cylinders, with only one sensor, exploiting the phase shift of the cycles. When vibrations occur, it is easy to trace them back to the cylinder that emitted them, in this way the control unit, to reduce the risk of detonation, reduces the ignition timing on the cylinder where the anomaly occurred.

It is possible to measure the resistance to knock of a fuel by means of a procedure by which an index called octane number (ON) is determined. Since knock is extremely sensitive to a multitude of factors, it is important that the test is carried out under standardized conditions in order to allow repeatability and reproducibility of it, so that the measurement of engine knock is independent of the test conditions. For this reason, a standardized engine called *Committee fuel research* is used.

The only way to qualify the fuel in a repeatable and reproducible way is to comparison with reference fuels. These fuels are mixtures of Isottane (C₈H₁₈ -rigid and compact structure with high resistance

to detonation) and Normaleptane

2.1.4 Supercharging

The amount of mixture introduced into the cylinder, is a determining factor for the power that can be generated and depends on the engine displacement. In order to increase performance and, therefore, the power that can be produced, with the same displacement, supercharging is introduced.

Remembering that:

$$Pu = bmep \cdot (iV) \cdot \frac{n}{m}$$

It is possible to increase power by targeting the break mean effective pressure (bmep):

$$bmep = \eta_u \cdot \frac{\lambda_v \cdot \rho_a \cdot H_i}{\alpha}$$

Since η_u , λ_v , and H_i are constants, we must increase the air density, before it is introduced in the cylinder and this happens through the supercharging by using:

- **Mechanical compressor:** in this case, a mechanical compressor, which is driven directly from the drive shaft, is used in order to provide the compressed air. For this kind of solution volumetric compressors, such as Roots or screw are used.
- **Turbocharger:** this solution implies the coupling between engine and supercharging system by fluid dynamics. The compressors used, in this case, are the centrifugal ones, who are driven by a turbine moved by the residual energy of the exhaust gases still capable of expanding. In this way, power is not taken away from the crankshaft, thus increasing not only performance but also efficiency.

The latter case has different solutions:

- *constant supply:* the exhaust gases are conveyed into a single exhaust manifold in which they expand; in this manifold, which collects the flow impulses from the different cylinders, a constant pressure condition is established so that a practically stationary flow arrives at the turbine;
- *Impulse supply:* the exhaust pipes of the different cylinders remain separated from each other

until they are admitted into the turbine, in such a way that we have a less dissipation of the kinetic energy of the exhaust gases, however, while the turbine operates in continuous transient conditions.

The use of these two types of solutions depends on the characteristics that are required: for engines that do not require a rapid transient response, the most suitable solution is the one with constant supply; otherwise the response of the impulse supply is the fastest.

2.1.5 Miller cycle

Normally Otto cycle and Diesel cycle engines are characterized by having the same compression and expansion ratio. This design choice allows on the one hand, to maximize the filling by making the best use of the available volume, but on the other hand it is less efficient from a thermodynamic point of view than the choice to adopt a higher expansion ratio than the compression ratio. The increasing need to improve the efficiency of modern Otto cycle and Diesel cycle engines has led designers to choose two particular cycles called Atkinson cycle and Miller cycle. Both cycles aim to achieve a higher expansion ratio than the compression ratio but through different ways.

The engine designed by Ralph Miller (who patented it under number 2817322 on December 24, 1957), in contrast to the particular packing used by the Atkinson cycle, uses a simple Otto cycle engine to which a variable timing system has been applied at the intake. As mentioned above, the aim is to obtain a lower compression phase than the expansion phase in order to improve the engine efficiency thanks to the higher energy extracted from the combustion in the form of pressure. To do this, the intake valve is closed either very late (LIVC), or very early (EIVC), compared to the classical situation so, in the first case, it is possible to let out a part of the charge entering the cylinder while, in the second case, it is possible to prevent a part of the charge entering the cylinder. The efficiency, therefore, increases but at the expense of the power that decreases, because with the same displacement, the amount of fuel that enters in a Miller cycle engine is less than that which enters in an Otto or Diesel cycle engine.

In the case of SI engines, the Miller cycle is also used to reduce the risk of detonation, by trying to obtain a lower temperature at the end of the compression stroke, with the same compression ratio and pressure used. This lower temperature can be obtained by significantly anticipating the closure of the intake valve, which however entails a decrease in filling, which can be compensated for by an increase in the boost pressure.

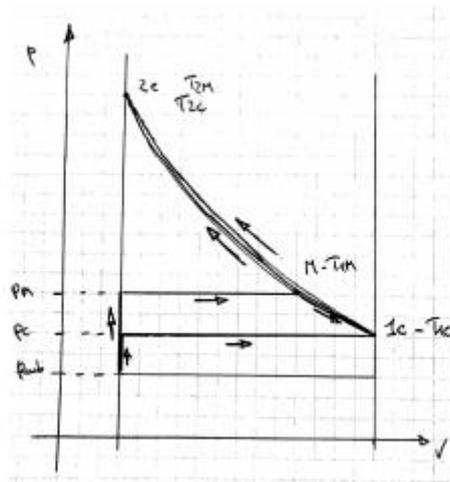


Figure 7 Miller cycle

With the help of the *figure 7*, let us see in detail how the Miller cycle works with an EIVC.

Starting from a supercharged engine, air is taken at room temperature and pressure and compressed to a pressure level p_c , which it will correspond to a temperature T_c upstream of an intercooler. At the exit from the intercooler there will be a temperature T_{1c} , which will be assumed, ignoring the effects of heat exchange with the walls during the intake phase, equal to that which is inside the cylinder, i.e. the inlet temperature of the engine. We now focus on the compression phase $1c-2c$, at the end of which, we will have reached a temperature T_{2c} . With the Miller cycle, the intake valve is closed early at point M, this will cause, as mentioned above, a decrease in filling, which will be compensated for by increasing the boost pressure to a p_M level. After that, the trapped charge will expand for the remaining part of the descending stroke of the piston following an expansion polytropic and ending at the same compression start point $1c$ of the original cycle, so that the compression phase begins again. As it can be seen, the pressure levels have not changed, as the increase in boost pressure occurs as a result of a decrease in filling, but the temperature reached at the end of compression phase will be lower, due to the lower compression stroke.

What has been dealt with applies to the EIVC, a similar discourse applies to the LIVC, but there are some differences:

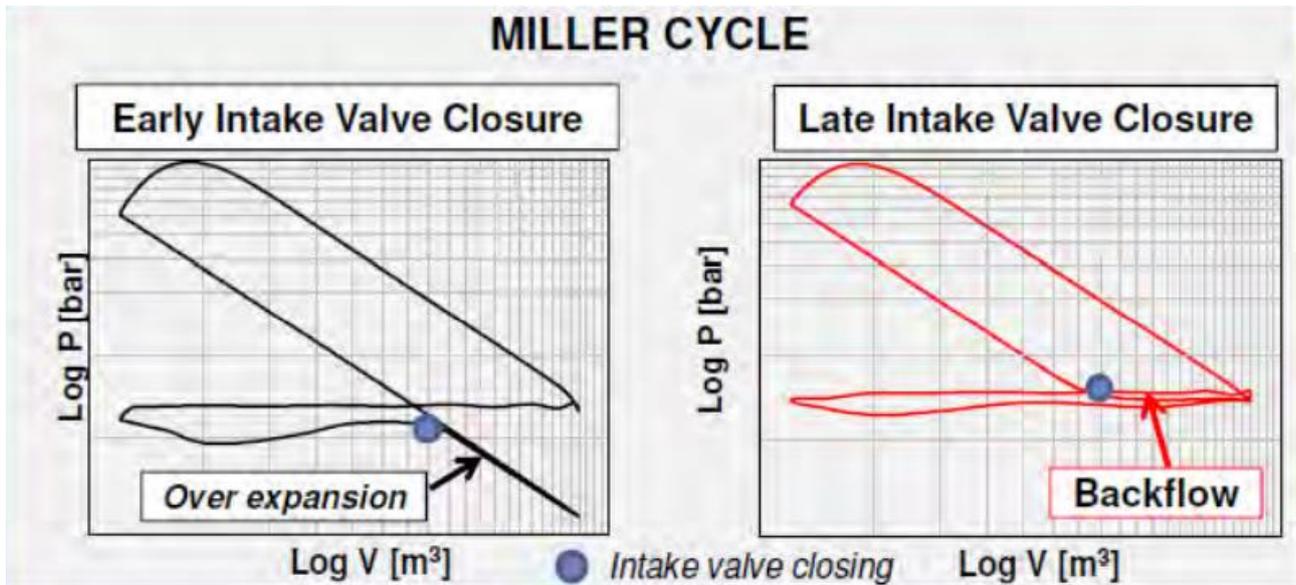


Figure 8 EIVC vs LIVC

- At high load, LIVC is superior over EIVC in improving fuel economy;
- The improvement with LIVC is due to advanced combustion phasing and increased pumping work;
- At low load, EIVC is better than LIVC in improving fuel economy;
- Pumping loss with EIVC is smaller than with LIVC at low load;
- But heat release rate with EIVC is slower than with LIVC.

3. Hybrid vehicle

Vehicles combining two or more sources of power, i.e. electric engine with ICE engine, that can directly or indirectly provide propulsion, are called *Hybrid vehicle* (HEV). The primary energy source is generally the chemical energy stored in the fuel of the internal combustion engine.

ICE engines provide good performance and long operating range by utilizing the high energy-density advantages of petroleum fuels. However, conventional ICE vehicles bear the disadvantages of poor fuel economy and environmental pollution.

In ICE vehicles, part of the chemical energy of the fuel is converted into mechanical energy, the remaining part consists of exhaust gases and heat. In addition, the mechanical energy produced must be able to win:

- Inertia;
- Gravity;
- Aerodynamic drag;
- Internal friction;
- Rolling resistance.

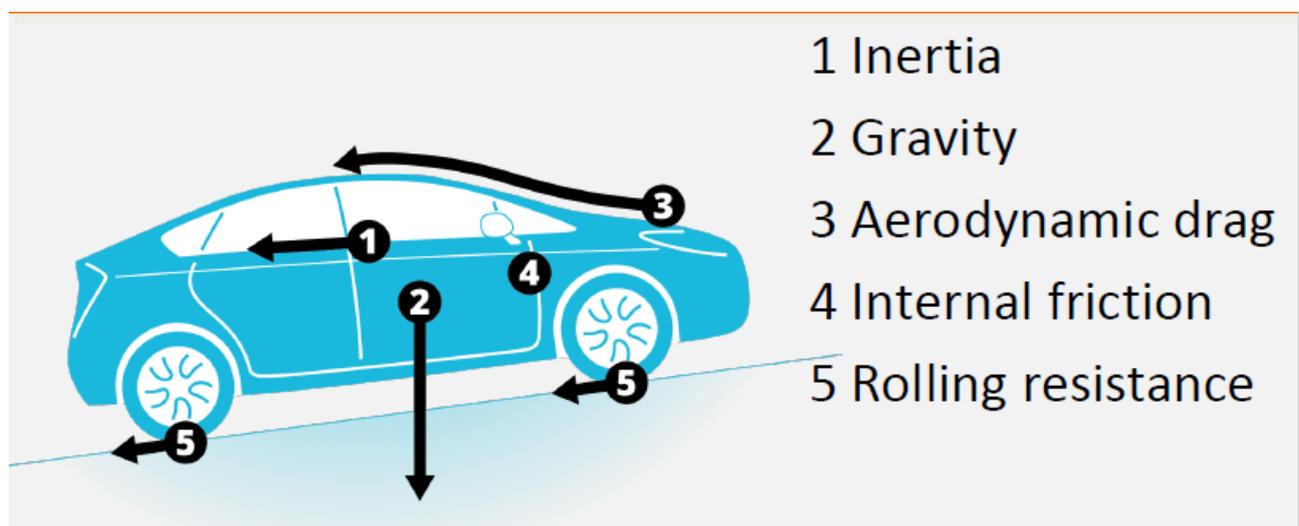


Figure 9 Forces acting on the vehicle

It is easy to understand that part of the mechanical energy produced by the combustion process is wasted.

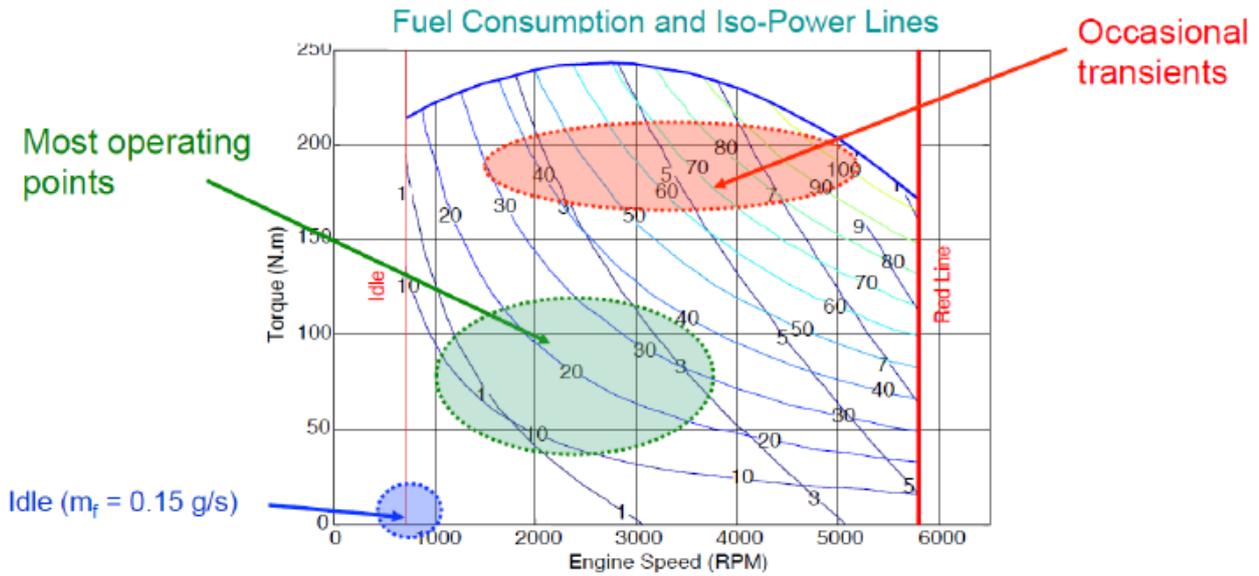


Figure 10 Engine operating map

As the *figure 10* shows, conventional vehicles operate in the lower-left quadrant of the engine operating map to have plenty of “reserve” power for fast accelerations. As a result, engines are typically oversized significantly compared to the average power required. This forces the engine to operate at relatively low efficiencies on the average, significantly lower than the peak efficiency.

Another problem encountered in ICE vehicles is represented by the defects inherent the stationary start. The stationary start is subject to the physical laws of inertia, which requires torque at almost zero speed, while the cyclic thermal engine needs a minimum speed to provide not zero torque.

Battery-powered electric vehicles (EV), possess some advantages over conventional ICE vehicles, such as high energy efficiency and zero environmental pollution. Nevertheless, the problem related to this type of vehicles concerns the weight of the battery pack: with the same mass of fuel on board, the same mileage is not guaranteed, due to the lower energy content of the batteries with respect to the energy content of gasoline.

Hybrid electric vehicles have the advantages of both ICE and EV vehicles and overcome their disadvantages. In principle, an electric transmission can completely decouple the speeds of ICE shaft and wheels; in addition, ICE engine does not have problem in terms of weight of the battery pack.

The HEV' components are the following:

- **ICE:** is the primary source of energy. ICE has the aim of power the vehicle, however, in HEV, this engine is smaller than on a conventional vehicle and uses advanced technologies to reduce emissions and increase efficiency;
- **Fuel tank:** it is the primary energy storage device on board;
- **Electric machine(s):** Advanced electronics allow them to act as motors or generators, so that they can both draw energy from the batteries to accelerate the car, or recovering energy from the car and return it to the batteries;
- **Batteries:** they are the secondary energy storage device on board. Unlike the fuel tank, which can only power the engine, the electric machines can draw or supply energy from/to the batteries;
- **Transmission:** The transmission is a key component of a hybrid vehicle, allowing for supplying power from the sources to the wheels.

There are different hybrid power train configurations: *Series HEV* (also called *range extender*), *Parallel HEV* and *Complex HEV*.

3.1 Series HEV

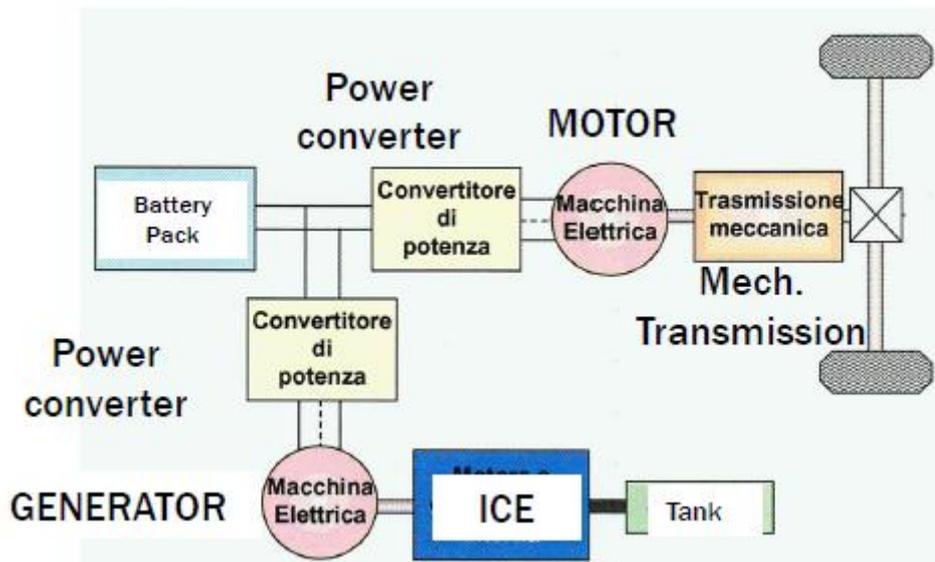


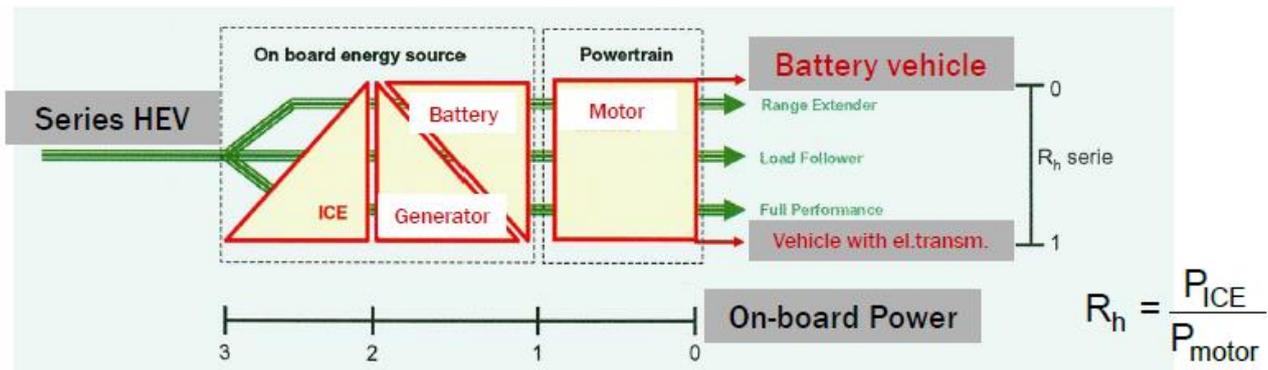
Figure 11 Series HEV

The vehicle is propelled by a traction motor. The traction motor is powered by a battery pack and/or an IC engine/generator unit. The engine/generator unit has a dual task, depending on the load power demand: if the latter is large, it helps the batteries to power the traction motor, or charges the batteries when load power demand is small. Mechanical transmission has usually a fixed transmission ratio. The motor and the generator are usually AC current machines (induction or synchronous), therefore traction converter is of the DC/AC type (inverter). The power converter of the generator is usually an inverter, too.

Since inverters are reversible electric machines, the power converter can be used to crank the ICE, whereas the traction converter can be used for regenerative brake.

Vehicle performance is completely determined by the size and characteristics of the traction motor drive. The determination of the size of the motor drive and gears of transmission is the same as in an electric vehicle design.

However, the drive train control is essentially different from the pure electric drive train due to the involvement of an additional engine/generator unit.



$R_h = 0$: pure electric vehicle; $R_h = 1$: vehicle with electric transmission.

Figure 12 Series HEV degree of hybridization

The Range extender vehicle, has a low degree of hybridization and it is a sort of thermally assisted electric vehicle, in which the ICE works at fixed point, in order to reach the maximum efficiency, and it is used to charge batteries. The ICE (and the electric machines connected to the ICE line) is designed to provide the average power during the scheduled driving cycle. The range is related to tank size.

Series hybrid systems optimize engine operation and efficiency. Because the ICE is not connected directly to the wheels, it can be run at its most efficient point and shut off when it is not needed. The system also eliminates the need for clutches and conventional transmissions.

Problems may occur when the ICE is used as the primary power source, that is because, in this case, the need to convert the output of the engine into electricity before driving the motor creates significant inefficiencies. As a result, in the range extender vehicles are set up with a small ICE that provide a range-booster to a large battery back, in this way it is possible to reduce emissions and excellent efficiency during stop-and-go driving are reached.

3.2 Parallel HEV

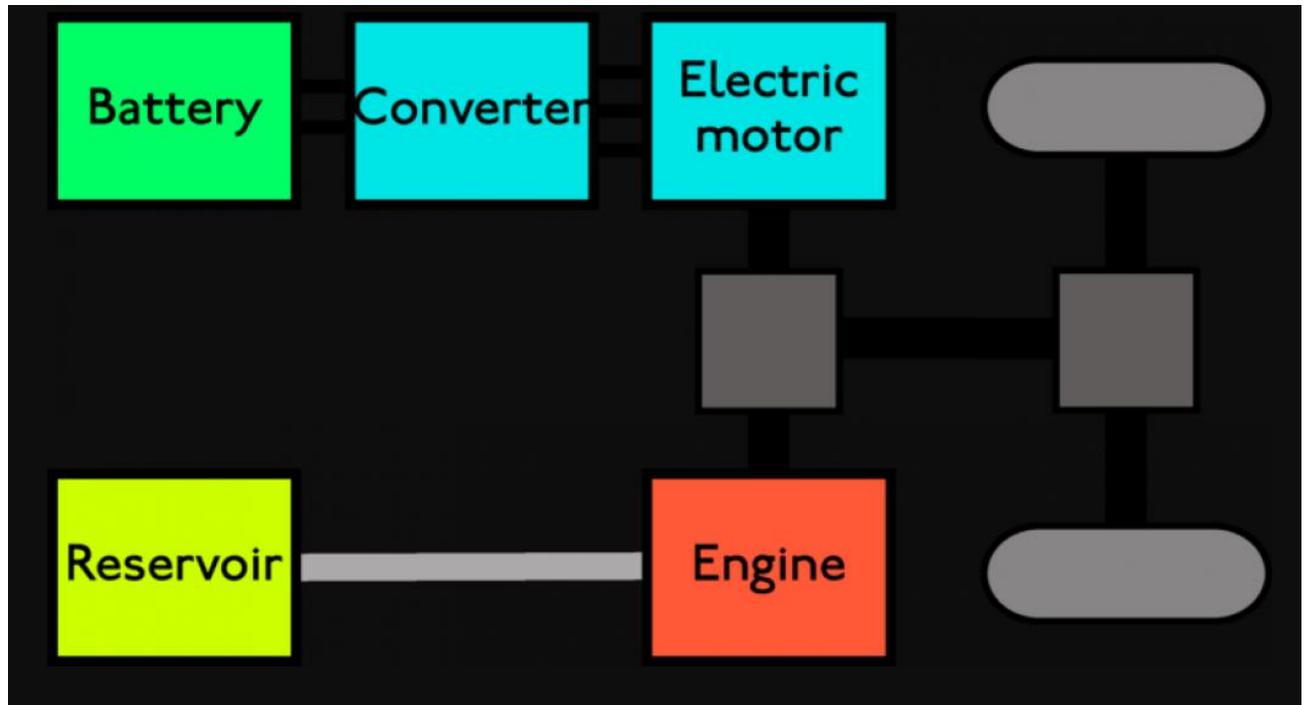


Figure 13 Parallel HEV

The parallel hybrid drive train has features that allow both the engine and traction motor to supply their mechanical power in parallel directly to the driven wheels. This hybrid type presents some advantages over the series configuration, that allow to have a better overall efficiency, for example: generator is not required, the traction motor is smaller and multi-conversion of the power from the engine to the driven wheels is not necessary.

On the other hand, the control of the parallel hybrid drive train is more complex than that of a series hybrid drive train, due to the mechanical coupling between the engine and the driven wheels.

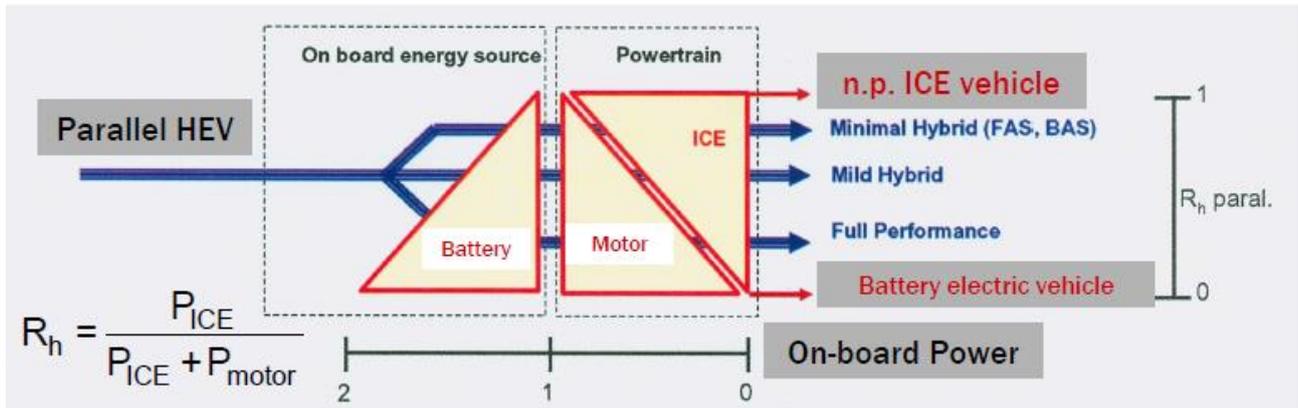


Figure 14 Parallel HEV degree of hybridization

As the degree of hybridization decreases, the on-board electric power increases. However, a wide range, such as 80-100 km on NEDC cycle, in pure electric operation requires large and heavy battery packs. Therefore, the battery pack should be designed for peak power source.

3.3 Complex HEV

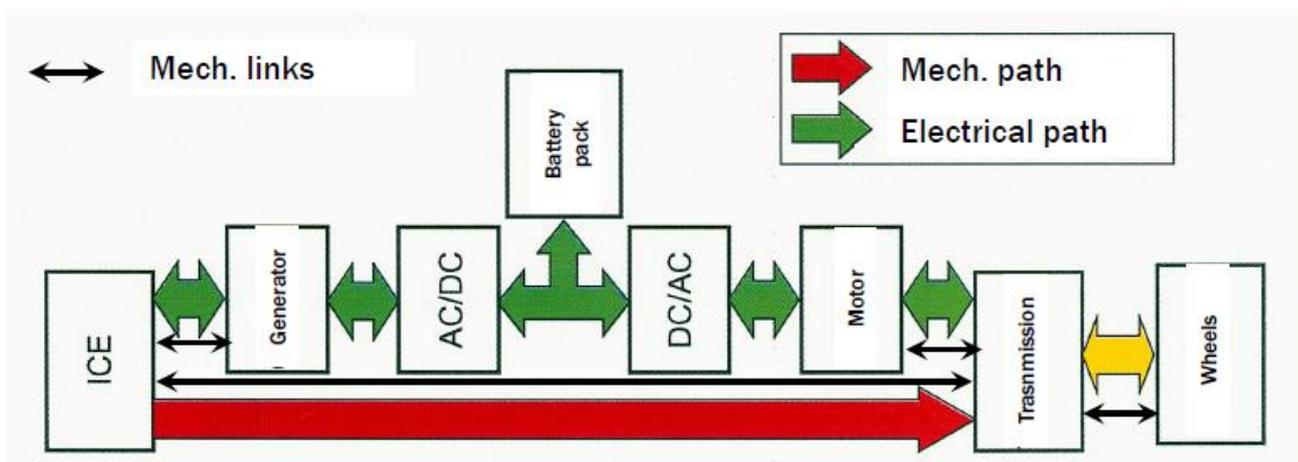


Figure 15 Complex HEV

Complex hybrids can be obtained with three different technology ways:

- By increasing the number of traction motors/ICEs;
- By increasing the number of energy and power sources;

- By coupling parallel and series concepts on the same powertrain architecture.

These HEV feature two possible energy paths from ICE to wheels:

- a direct mechanical one, in which ICE and motor can deliver their power to the wheels at the same time;
- an indirect electric path, in which ICE delivers its power to a motor through a double energy conversion.

3.4 Motivation for HEVs

The reasons why a HEV should be chosen will be now reassumed:

- **Regenerative Braking:** When the vehicle is braking, its kinetic energy can be recovered by a generator and stored on-board;
- **Idling Reduction:** Depending on the sizing of the secondary power source, the engine can be turned off at stops and lower speed conditions;
- **ICE Efficiency Improvement:** the secondary power source assists the ICE, preventing it from operating at inefficient conditions;
- **ICE Downsizing/Downspeeding:** due to the assistance of the secondary power source, a smaller ICE or a “longer” final drive can be chosen without compromising performance;
- **Eliminate or mitigate the “clutching” losses** by not engaging the engine until the speeds are matched and do not require any slip;
- **Potentially continuously varying gear ratio;**
- **Power auxiliary/accessories electrically;**
- In addition to energy considerations, which are perhaps the largest motivation behind hybrids, additional benefits can be achieved by **having more control over the engine operating point and engine transients.**

On the other hand, along with the pros, the cons regard:

- **The price:** in fact, HEVs are more expensive;
- **The weight:** HEVs are heavier;
- **Fuel efficiency:** more complex control systems are required to optimize it.

4. CFD Analysis

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering, fluid flows and heat transfer, and engine and combustion analysis.

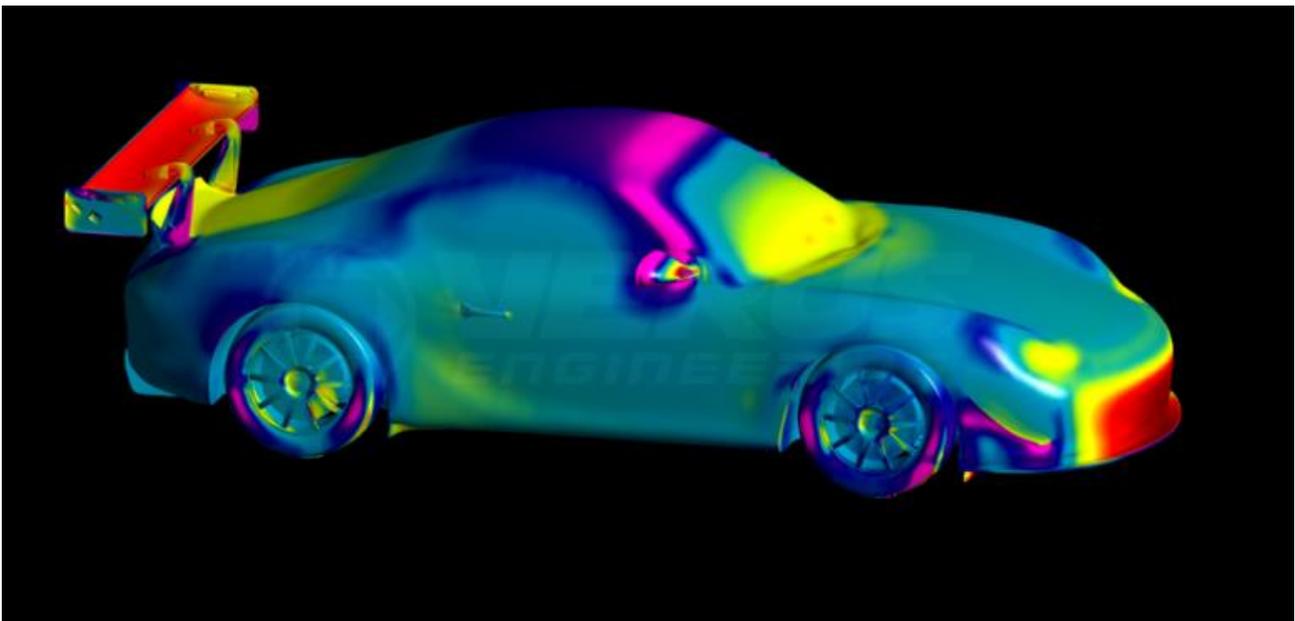


Figure 16 Simulation of aerodynamic package of a Porche Cayman

CFD analyses have a great potential to save time in the design process and are therefore cheaper and faster compared to conventional testing for data acquisition. Furthermore, in real life tests a limited amount of quantities is measured at a time, while in a CFD analysis all desired quantities can be measured at once, and with a high resolution in space and time.

Because CFD analyses approximate a real physical solution, it should be noted that these CFD analyses cannot fully exclude physical testing procedures. For verification purposes tests should still be performed.

A CFD analysis basically consists of the following three phases:

- **Pre-processing:** In this phase the problem statement is transformed into an idealized and discretized computer model. Assumptions are made concerning the type of flow to be modelled (viscous/inviscid, compressible/incompressible, steady/non steady). Other processes involved are mesh generation and application of initial- and boundary conditions;
- **Solving:** The actual computations are performed by the solver, and in this solving phase computational power is required. There are multiple solvers available, varying in efficiency and capability of solving certain physical phenomena;
- **Post-processing:** Finally, the obtained results are visualized and analyzed in the post processing phase. At this stage the analyst can verify the results and conclusions can be drawn based on the obtained results. Ways of presenting the obtained results are for example static or moving pictures, graphs or tables.

CFD is based on the Navier-Stokes equations. Arising from applying Newton's second law to fluid motion, together with the assumption that the stress in the fluid is the sum of a diffusing viscous term and a pressure term, these equations describe how the velocity, pressure, temperature, and density of a moving fluid are correlated.

4.1 Discretization methods

The stability of the selected discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

- **Finite volume method:** The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large

problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion). In the finite volume method, the governing partial differential equations are recast in a conservative form, and then solved over discrete control volumes. This discretization guarantees the conservation of fluxes through a particular control volume;

- **Finite element method:** The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. The FEM formulation requires special care to ensure a conservative solution. The FEM formulation has been adapted for use with fluid dynamics governing equations. Although FEM must be carefully formulated to be conservative, it is much more stable than the finite volume approach;
- **Finite difference method:** The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids;
- **Spectral element method:** Spectral element method is a finite element type method. It requires the mathematical problem to be cast in a weak formulation. This is typically done by multiplying the differential equation by an arbitrary test function and integrating over the whole domain. Purely mathematically, the test functions belong to an infinite-dimensional function space. Clearly an infinite-dimensional function space cannot be represented on a discrete spectral element mesh; this is where the spectral element discretization begins. The most crucial thing is the choice of interpolating and testing functions. In a spectral element method however, the interpolating and test functions are chosen to be polynomials of a very high order. This guarantees the rapid convergence of the method;
- **Lattice Boltzmann method:** The lattice Boltzmann method (LBM) with its simplified kinetic picture on a lattice provides a computationally efficient description of hydrodynamics. Unlike the traditional CFD methods, which solve the conservation equations of macroscopic properties numerically, LBM models the fluid consisting of fictive particles, and such particles perform consecutive propagation and collision processes over a discrete lattice mesh;
- **Boundary element method:** In the boundary element method, the boundary occupied by the fluid is divided into a surface mesh.

The importance of CFD analysis lies in the fact that without numerical simulations of fluid flow, it is very difficult to imagine how:

- Meteorologists can forecast the weather and warn of natural disasters;
- Vehicle designers can improve aerodynamic characteristics;
- Architects can design energy-saving and safe-living environments;
- Oil and gas engineers can design and maintain optimal pipes networks;
- Doctors can prevent and cure arterial diseases by computational hemodynamic.

5. Ricardo Wave software



Figure 17 Ricardo Wave Software logo

Wave is a 1D engine simulation software package, developed by Ricardo Software, used to analyze mass flows, energy losses in ducts, plenums and the manifolds of different machines and to study the dynamicity of pressure waves through ducts. It provides a fully integrated treatment of time-dependent fluid dynamics and thermodynamics by means of a one-dimensional formulation. The code provides detailed numeric and graphical output of a large number of important engineering parameters, such as cycle-averaged pressures, temperatures, and flow rates at many locations within the network; it also shows the time variation of important variables such as mass flow, velocity, composition and thermodynamic variables. There are several fields in which the use of Wave may be helpful, these include:

- engine performance;
- acoustic and noise;
- combustion and emissions;
- thermal analysis;
- dynamic system control;
- real time analysis;
- 1-D/3-D CFD Co-Simulation.

In the following chapters it will be possible to find an extensive overview of the software.

5.1 Flow elements description

As soon as the program will be open, the interface will show the model panel and the elements panel. The latter has all the elements that we will need to build the model, then once chosen they will be moved in the model panel.

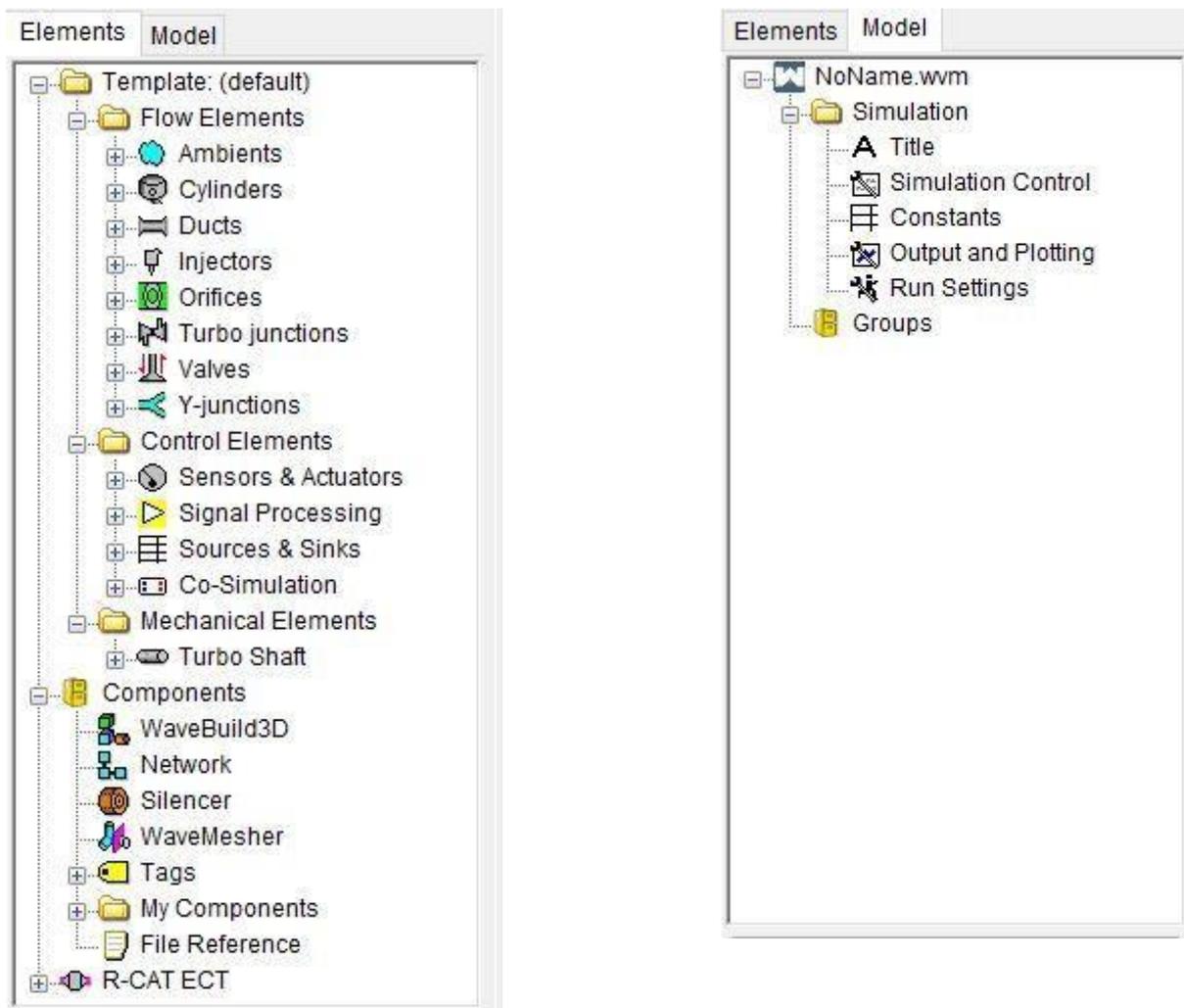


Figure 18 Elements & Model panel

5.2 Simulation control & constant

In Simulation control, it's possible for the user to control the model's behavior, through its six tabs.

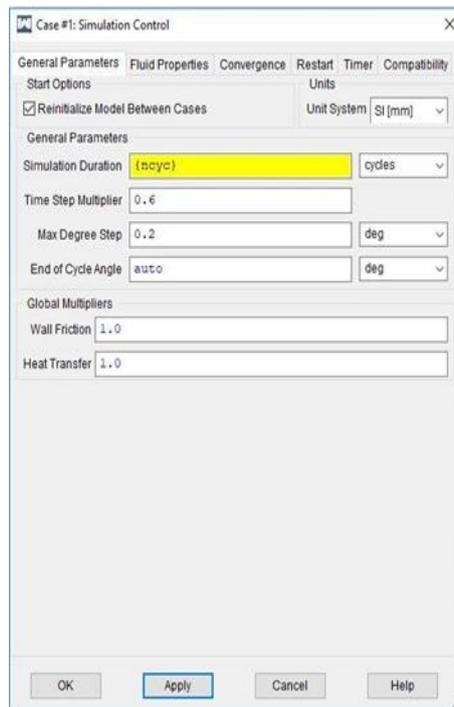


Figure 19 Simulation Control – General Parameters

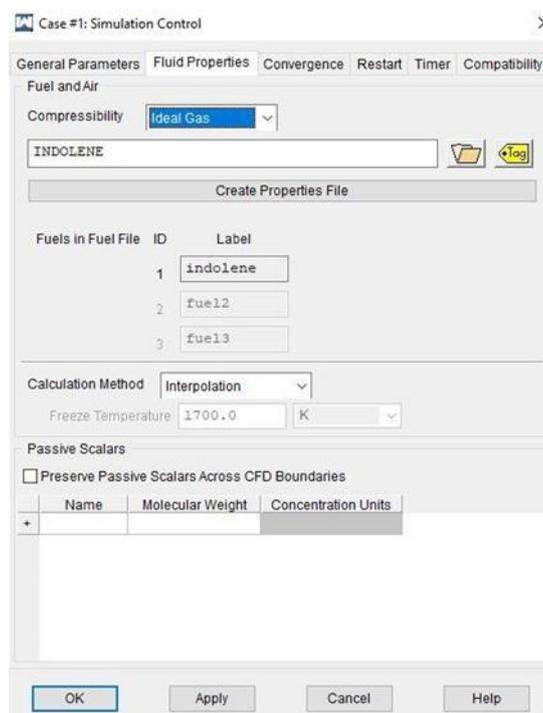


Figure 20 Simulation Control - Fluid Properties

In particular: the General Parameters tab (which will need to be edited for every models) controls the timesteps of the simulation;

In Fluid Properties tab, the user will be able to set up the fuel, in this case INDOLENE fuel has been chosen, with air to fuel ratio equal to 14.7. This tab also controls the definition of both active scalars and the passive scalars, the first one will be transported as mass throughout the model, the latter without mass instead.

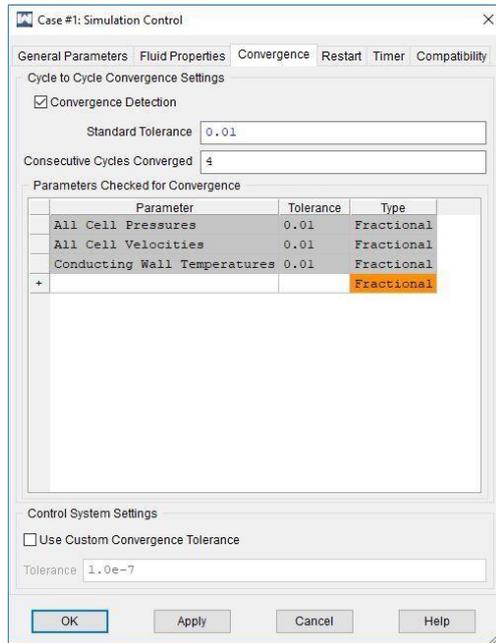


Figure 21 Simulation Control – Convergence

Convergence criteria will be specified in the convergence tab. Once the simulation has reached a converged condition, WAVE will stop the simulation earlier than the value set in “Simulation Control”, as it was mentioned before.

Constants				
Constants Table				
	Name	Units	Case 1	+
Status			Run	Run
Title			Case 1	
1	AF		14.7	
2	C_DUR	deg	20	
3	CA_50	deg	8	
4	DIA2	mm	90	
5	EVO	deg	120	
6	IVO	deg	330	
7	ncyc	cycle	250	
8	TIME	tr	200	

Figure 22 Constants Table

The constant table allows the user: first of all, to set up several run cases, then to modify the values belonging to elements without opening them.

5.3 Ambient element

Case #1: Ambient Panel

Template Name
Default Ambient

Name
amb1

Ambient Initial Fluid Composition

Diameter AUTO mm

Pressure 1.01325 bar

Temperature 300 K

Discharge Coefficient AUTO

Additional Acoustic End Correction 0 mm

Solution Type Fixed Floating

OK Apply Cancel Help

Figure 23 Ambient Panel – Ambient

The moment the user starts building the model, the ambient element, located in the element tree, is the starting point. As it is possible to see, there are different types of ambient elements, depending on the characteristics the user wants to attribute to the model. This element represents a connection to an infinite reservoir at a specified pressure, temperature, and composition. The user, through the ambient panel, can set all the boundary conditions, regarding the ambient and the initial fluid composition.

Case #1: Ambient Panel

Template Name
Default Ambient

Name
amb1

Ambient Initial Fluid Composition

Fresh Air 1.0

Vaporized Fuel 0.0

Burned Air 0.0

Burned Fuel 0.0

Liquid Fuel 0.0

Passive Scalars

	Name	Molecular Weight	Concentration	Units
+				

OK Apply Cancel Help

Figure 24 Ambient panel – Initial Fluid Composition

5.4 Duct & Y-Junction

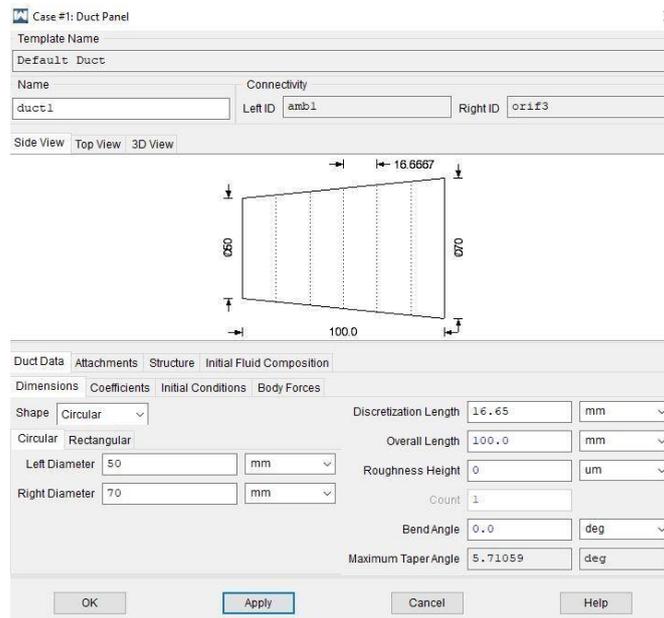


Figure 25 Duct Panel

Ducts, which are essentially a channel where flow enters at one end and exits at the other, are used to model the pipe geometry and to connect sub-volumes together when modelling large volumes (massless ducts). They are connected to ambients, cylinders, orifices, turbo, valves, y-junctions. The user has the task to set the length, the diameter, the boundary conditions (pressure and temperature) and the discretization length, which is the practice of taking the modeled geometric system and dividing the large single volume into smaller sub-volumes. The greater the discretization and the better the accuracy of the result will be, on the other hand computational time will get worse, that is because more elements means more calculations.

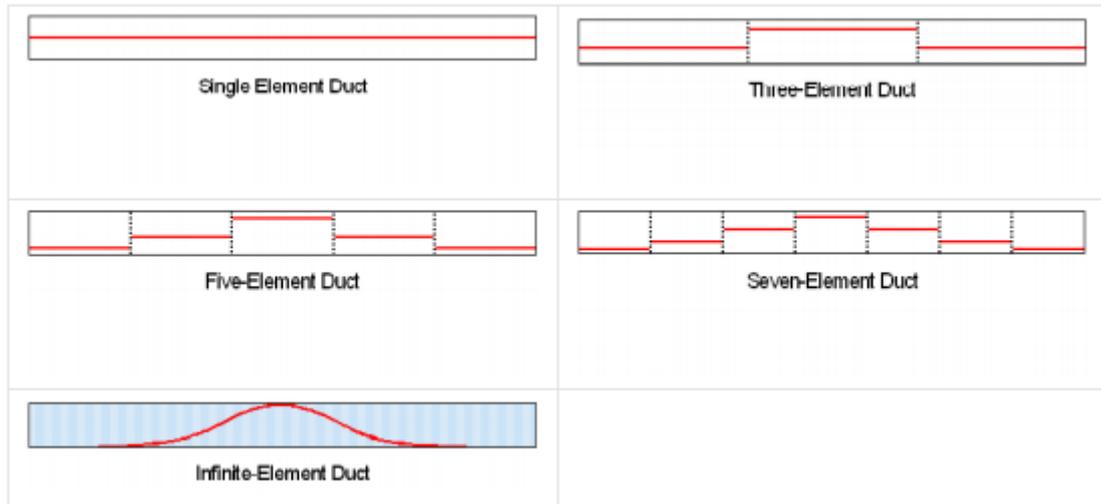


Figure 26 Pressure wave

In literature it is possible to find a standard formula that regulates the value of the discretization length (dx) according to the engine bore (B):

For the intake side: $dx = 0,45B$;

For the exhaust side: $dx = 0,55B$.

To achieve the best combination of speed and accuracy, it is recommended that the discretization length is set equal for all ducts of similar temperature within a system.

5.4.1 Y-Junction

Y-junctions represent an arbitrary volume where multiple flows may enter and exit, they are used to model junctions where more than two ducts are attached.

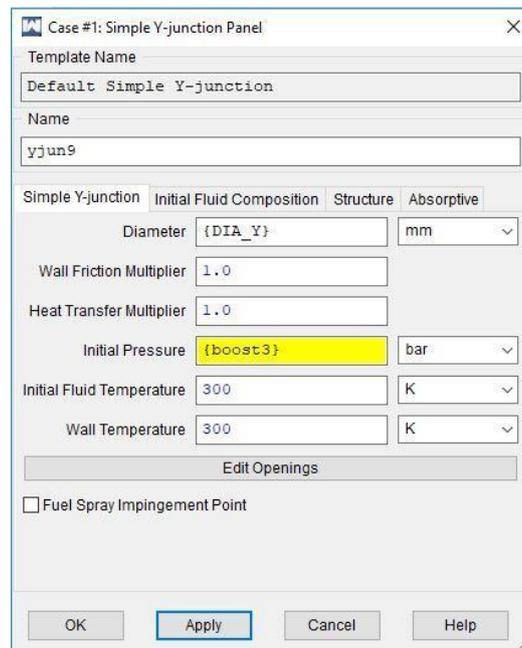


Figure 27 Simple Y-junction Panel

There are two different Y-junctions: Simple and Complex. The Simple Y-junction is assumed to be spherical in shape and requires minimal input, a diameter defines the volume, surface area, and characteristic flow values required for the junction. Complex Y-junctions are more flexible but also require more user-input. They are used to define any arbitrary shape desired.

5.5 Valves



Figure 28 Valve element

A Valve element must be attached to two ducts, or between a duct and a cylinder, or between a duct and a crankcase, or for a valved connection, between a duct and a Y-junction, in the flow system and represent variable area orifices. The valve behavior is defined by valve sub-models. The motion of the valve may be imposed as a function of time, driven by flow dynamics, or driven by an actuator.

In Wave we can have essentially four types of valve:

- ***Inline valve***
- ***Piston driven valve;***
- ***Throttle valve;***
- ***Valved connection.***

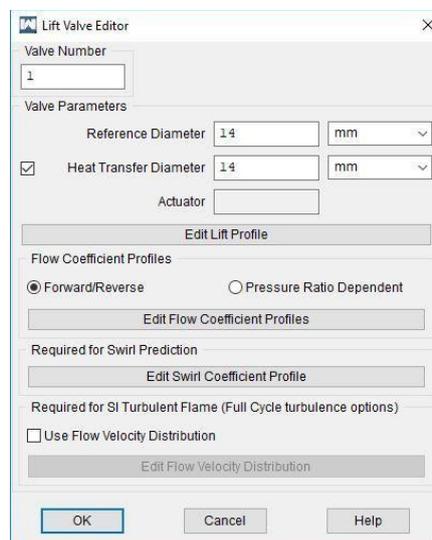


Figure 29 Lift valve editor

The user can set all the valve's parameters, including diameter, heat transfer diameter, lift profile, flow and swirl coefficient through the "Lift Valve Editor".

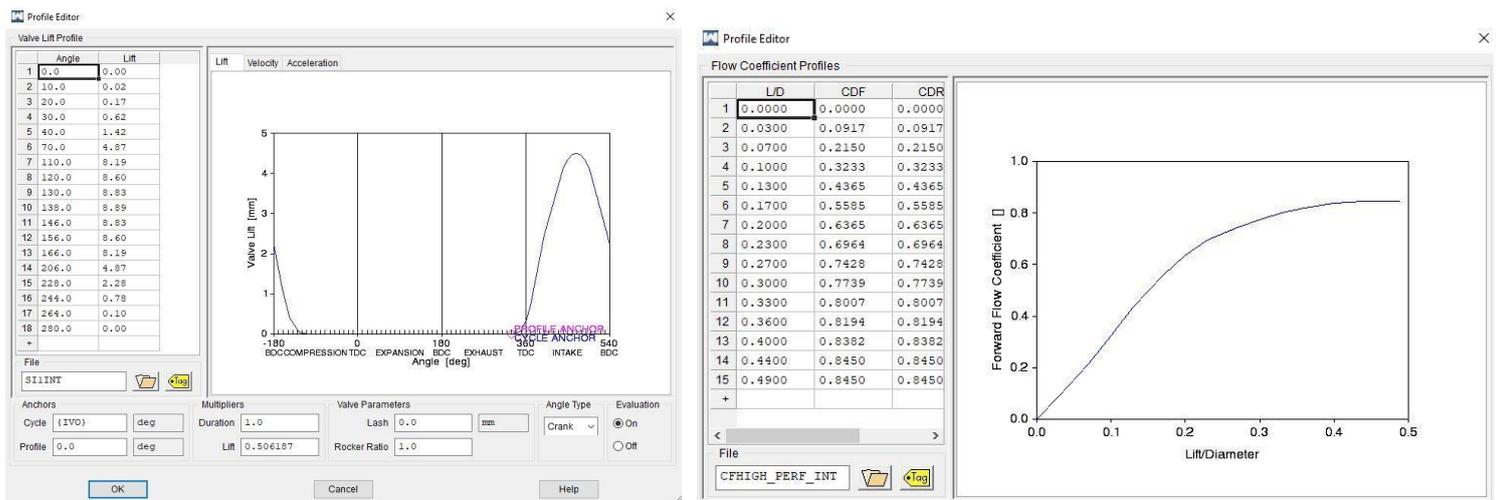


Figure 30 Valve Lift & Flow Coefficient Profile

In the Valve Lift Profile Editor, data must be entered for the behavior of the valve. Lift profile and Flow coefficient profile has been imported as default. In the Profile editor it is possible to find fields of primary importance, i.e. the Anchors, the Multipliers and the Valve Parameters.

The Anchors allow arbitrary alignment of the valve profile within the engine cycle. The Profile Anchor denotes a point specified in the array of angle data. The Cycle Anchor denotes another point in the engine cycle in crank angle degrees, and along with the profile anchor, they align to locate the valve lift array within the engine cycle. These anchors can be set as constants to allow variable valve events between cases.

The Multipliers are used to multiply every point in the valve array in either lift magnitude or angle duration. These can be parameterized to allow variable valve lift and duration between cases. Duration and lift multipliers are normally set to 1.0.

As for the Valve Parameters, Lash, also called tappet clearance, should be entered as hot lash, and its value doesn't change during the valve event. When the lift array contains cam lift data a Rocker Ratio scaling factor, which multiplies the Lift array of the valve lift profile, is needed. It is set to 1.0 if the lift array contains actual valve lift data.

5.6 Cylinder Panel

Case #1: Cylinder Panel

Template Name
Default Engine Cylinder

Name: cyl1 Cylinder Number: 1 Number of Valves: 4

Edit Engine General Properties

Geometry | Initial Conditions | Valves | Sub-Models | Initial Fluid Composition

Liner | Head | Piston

Bore: 37 mm

Stroke: 29 mm

Clearance Height: 7 mm

OK Apply Cancel Help

Figure 31 Cylinder Panel – Liner Geometry

The user can set all the cylinder's parameters through the "Cylinder Panel". First of all, in the Geometry tab it's possible to set the Liner, Head and Piston Geometry.

Regarding the Liner, parameters such as Bore, Stroke and Clearance Height are set.

Case #1: Cylinder Panel

Template Name
Default Engine Cylinder

Name: cyl1 Cylinder Number: 1 Number of Valves: 2

Edit Engine General Properties

Geometry | Initial Conditions | Valves | Sub-Models | Initial Fluid Composition

Liner | Head | Piston

Surface Area Multiplier: 1.0

Surface Area: 1075.21 mm²

OK Apply Cancel Help

Figure 32 Cylinder Panel – Head Geometry

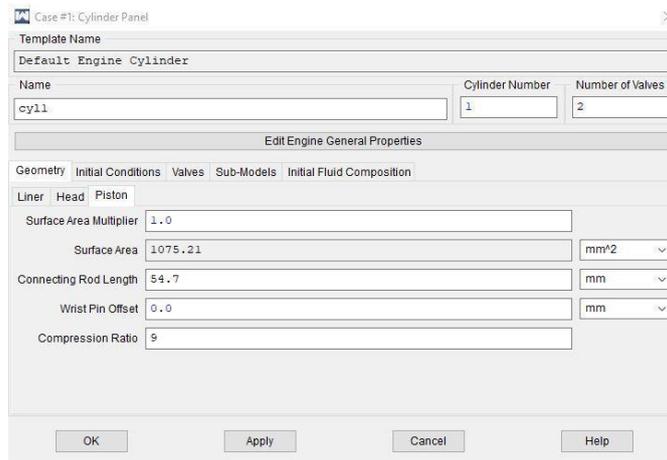


Figure 33 Cylinder Panel – Piston Geometry

As for the Head tab, parameters to be set are: Surface area for the head and Surface area Multiplier, expressed as a multiple of the bore area.

In the Piston tab, the user can set the Connecting rod length, i.e. the distance between the center lines of the wrist-pin and elbow pin on the connecting rod; the Wrist pin offset, which is the distance from the center line of the piston, to the center line of the wrist pin; and finally, the Compression Ratio, the geometric ratio of the volume of the combustion chamber when the piston is at BDC, to the volume of the combustion chamber when the piston is at TDC.

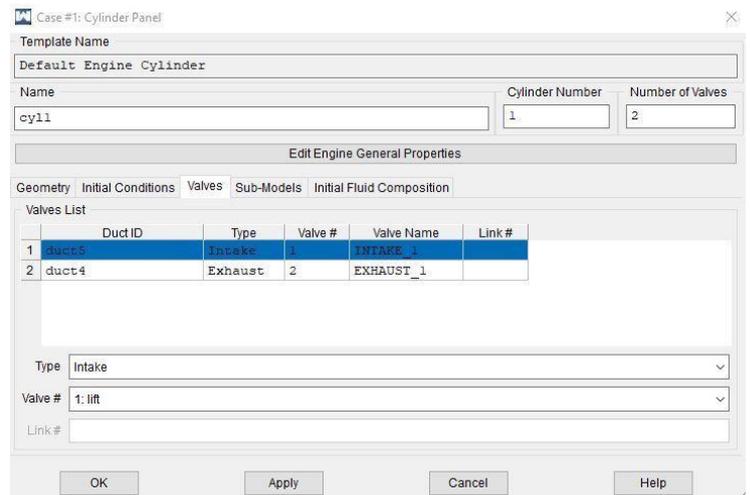
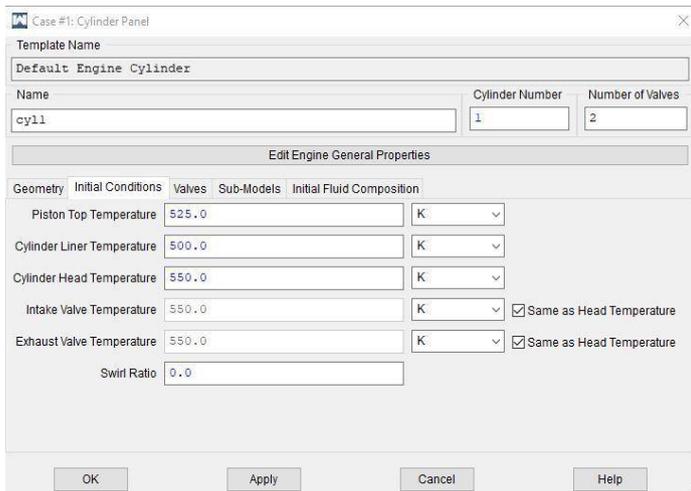


Figure 34 Cylinder Panel – Initial Conditions & Valves

In the Initial Conditions tab, boundary conditions, in terms of temperature, for the Piston, Cylinder, intake and exhaust valve, are set.

The valves tab shows the valve' type (intake or exhaust) and the ducts to which they are connected.

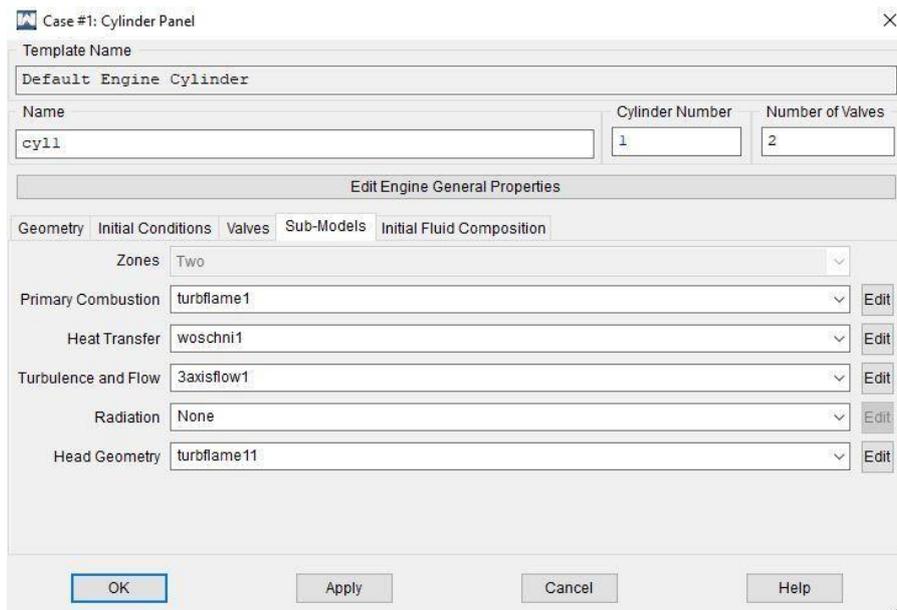


Figure 35 Cylinder Panel – Sub Models

The Sub-Models tab allows the user to specify all the sub-models, such as Combustion model, Heat Transfer model and Turbulence and Flow model, applied.

5.7 Engine general panel

Case #1: Engine General Panel

Geometry | Operating Parameters | Scavenge | Combustion | Conduction | Heat Transfer | Turbulence and Flow | Head Geometry | Slaved Models

Configuration

No. of Cylinders: 4

Strokes per Cycle: 4

Engine Type: Spark Ignition

Displacement: 0.124724

Imposed Piston Motion

Printout Flag

In-Cylinder State: 0

Port Conditions: 0

Friction Correlation

ACF: 0.5

BCF: 0.006

CCF: 600

QCF: 0.2

Firing Order and Relative TDC

	1	2	3	4
Cylinder	1	2	4	3
TDC	0	180	180	180

Required for Swirl Prediction

Piston Bowl Depth: 0.0

Piston Bowl Diameter: 0.0

Piston Bowl Rim Diameter: 0.0

Piston Bowl Volume: 0.0

OK | Apply | Cancel | Help

Figure 36 Engine General Panel – Geometry

The “Engine General Panel” allows the user to set all the engine parameters. In order to model friction, WAVE uses a modified form of Chen-Flynn correlation. The correlation formula has several terms for:

- **accessory friction:** a constant term (ACF), if the friction is to be entered directly, this input field should be used and the BCF, CCF, and QCF terms should be set to zero.
- **load dependence:** a term which varies with the peak cylinder pressure (BCF);
- **hydrodynamic friction:** a term linearly dependent on mean piston velocity (CCF);
- **windage losses:** a quadratic term with mean piston velocity (QCF).

The equation is given below:

$$FMEP = A_{cf} \frac{1}{ncyl} \sum_{i=1}^{ncyl} [B_{cf}(P_{max})i + C_{cf}(S_{fact})i + Q_{cf}(S_{fact})^2 i]$$

With: $S_{fact} = RPM * stroke/2$

Where:

A_{cf}	ACF user input
B_{cf}	BCF user input
C_{cf}	CCF user input
Q_{cf}	QCF user input
P_{max}	Maximum cylinder pressure
RPM	Cycle-average engine speed
stroke	Cylinder stroke

Table 2 Parameters of Chen-Flynn correlation

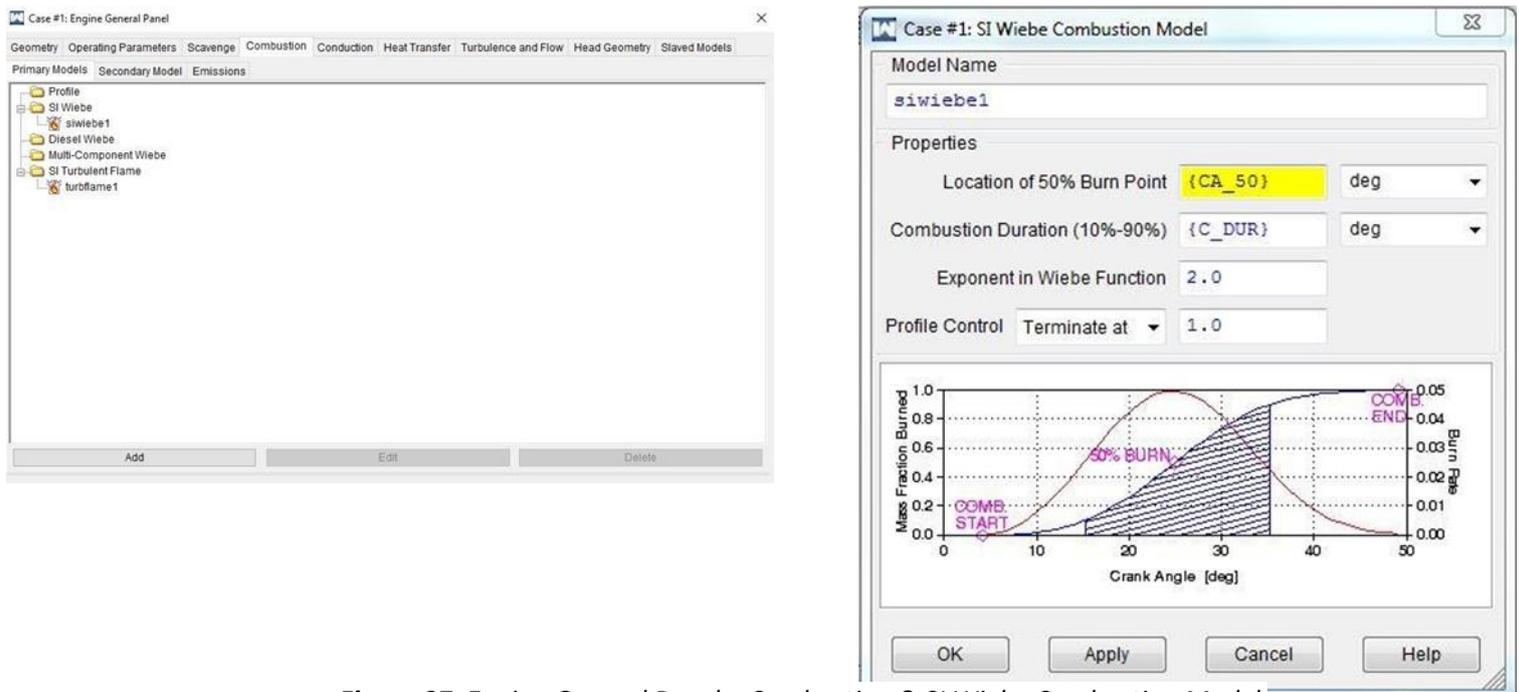


Figure 37 Engine General Panel – Combustion & SI Wiebe Combustion Model

In the combustion tab the user can set all the combustion parameters. Before analyze the SI Wiebe combustion model, it is important to point out to some combustion terminology. The terms “burn rate” and “heat-release rate” are often used interchangeably. There are however distinct differences between the two that are important in respect to their usage in WAVE. WAVE uses input in the form of burn rate for Wiebe functions or profile combustion. Heat release rate, although often times similar in profile to burn rate, should not be used. Burn rate can be defined as the rate at which the fuel mass in the cylinder is consumed in the combustion process to become products of combustion. Regardless of the final state of the air/fuel mixture the initial fresh air and fuel no longer exist in their natural states when combustion has completed. Heat release rate itself can have many different meanings but is most often intended as “apparent heat release rate”, which is usually calculated using analysis methods from a measured in-cylinder pressure trace. It is labeled “apparent” as the analysis methods of the in-cylinder pressure measurement cannot perfectly account for the heat transfer through the chamber walls and the unknown instantaneous specific heat ratio during the combustion event. The SI Wiebe function is a primary combustion model and the most commonly used combustion sub-model in SI engines. It is widely used to describe the rate of fuel mass burned in thermodynamic calculations. This relationship allows the independent input of function shape parameters and of burn duration. It is known to represent quite well the experimentally observed trends of premixed SI combustion. The Wiebe sub-model can be applied to all engine cylinder elements.

In the SI Wiebe Combustion Model tab, it's possible to set the:

- Location of 50% mass fraction burned (AT DC);
- Combustion Duration from 10% to 90% mass fraction burned points;
- Exponent in the Wiebe function, controlling the shape of the Wiebe curve; it may vary from 0.5 to 5. The lower its value, the more energy release there will be at the beginning, this will decelerate the process towards the end; likewise, the higher the value, the slower the combustion will be in the initial phase, accelerating until it will reach the end.

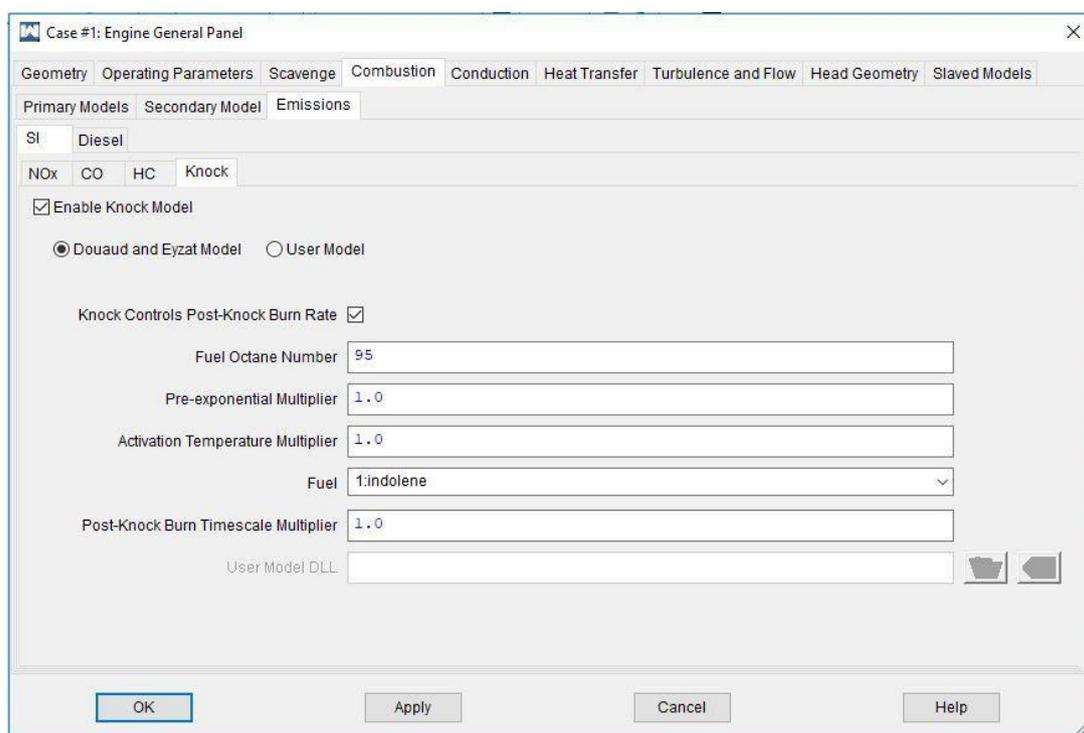


Figure 38 Engine General Panel - Emissions

As it has been said in previous chapters, in an SI engine, before the air/fuel mixture ignites, a series of intermediate steps are necessary, during which the formation of radicals takes place, after reaching and exceeding a critical threshold, the combustion process begins. The mixture is therefore only ignited after a certain delay, called ignition delay, or induction time (τ).

The Simple knock sub-model is based on the Douaud and Eyzat (1978) induction time correlation. It can be applied only to SI engines and is mutually exclusive with the User Knock sub-model. If activated, it is applied to all engine cylinders and requires that the cylinders have the “Zones” switch set to “Two”.

The induction time, in seconds, is calculated at every time-step using the following equation:

$$\tau = \frac{0,01869}{A_p} * \frac{ON^{3,4107}}{100} * p^{-1,7} \exp \frac{3800/A_t}{T}$$

Where:

- A_p is a user-entered pre-exponential multiplier;
- ON is the user-entered fuel Octane number;
- P is the cylinder pressure [kgf/cm²];
- A_t is a user-entered activation temperature multiplier;
- T is the unburned gas temperature [K].

In general, this induction time continually decreases as combustion progresses and the unburned zone temperature rises. The end-gas auto-ignites (knocks) if the induction time is less than the flame arrival time.

It is assumed that auto-ignition happens when:

$$\int_{t_0}^{t_i} \frac{d\tau}{\tau} = 1$$

With t_0 and t_i assumed respectively as: start of end-gas compression and time of auto-ignition.

When the knock condition is reached, the user can specify whether to continue with the remainder of the combustion event without change, or to force the remainder of the fuel mass in the cylinder to rapidly burn to simulate the knock event.

The work that was carried out by Mariani has detected a relationship between the Peak Cylinder Pressure (PCP) and Knock Intensity:

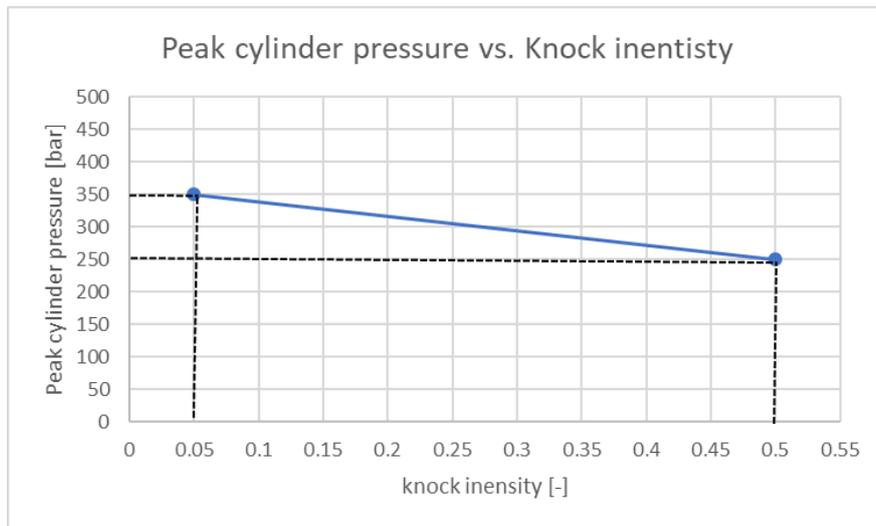


Figure 39 Peak cylinder pressure vs Knock intensity

As the *figure 39* shows, the peak cylinder pressure decreases as the knock intensity increases. Since WAVE doesn't take into account pressure fluctuations caused by the knock phenomena, the PCP value will be below the limit represented by the straight line.

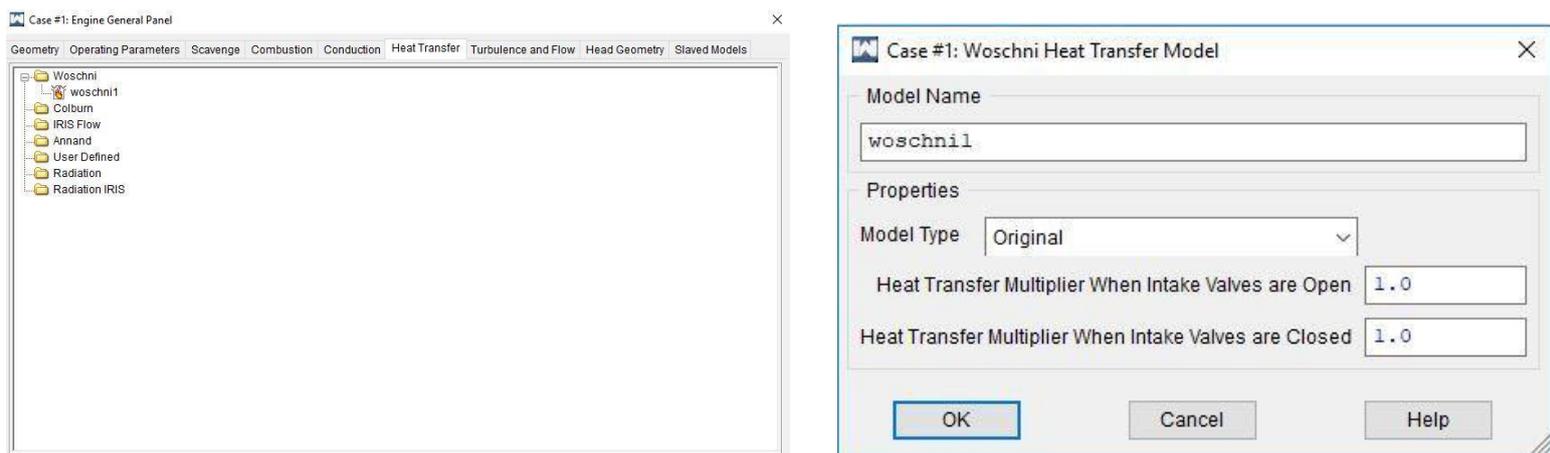


Figure 40 Engine General Panel – Heat Transfer & Woschni Transfer Model

In the Heat-Transfer tab it's possible to realize that the Woschni Model has been chosen, it is the most commonly used heat transfer sub-model and can be applied to all cylinder elements.

The Woschni heat transfer sub-model views the charge as having a uniform heat flow coefficient and

velocity on all surfaces of the cylinder and calculates the amount of heat transferred to and from the charge based on these assumptions.

The Woschni heat transfer coefficient is calculated using the following equation:

$$h_g = 0,0128D^{-0,20}P^{0,80}T^{-0,53}v_c^{0,8}C_{enht}$$

Where:

- D = Cylinder Bore;
- P = Cylinder Pressure;
- T = Cylinder Temperature;
- v_c = Characteristic velocity;
- C_{enht} = User-entered multiplier.

The characteristic velocity is the sum of the mean piston speed and an additional combustion-related velocity that depends on the difference between the cylinder pressure and the pressure that would exist under motoring conditions.

5.8 Mapless Compressor

In order to simulate the behavior of a turbocharged engine, mapless compressor, if compressor map data is not available, can be used. The compressor is modeled as a planar “rotor”, where the mass flow and outlet enthalpy are controlled, and an outlet flow passage. The outlet enthalpy is calculated from the pressure ratio and isentropic efficiency. The compressor responds to a target outlet pressure by controlling mass flow, whilst allowing for a reasonable amount of pulsation.

The compressor is modeled as a planar boundary between adjacent duct sub-volumes and requires at least two connected ducts: one at the blue inlet connection point and one or more at the red outlet connection point. It must be connected to a single turbo shaft from the hub of the compressor.

A mapless compressor can be used in isolation to model a supercharger. A sweep of the target outlet pressure will show how the engine responds and give the power requirements for the compressor, as

a function of the target pressure, for a particular engine operating point.

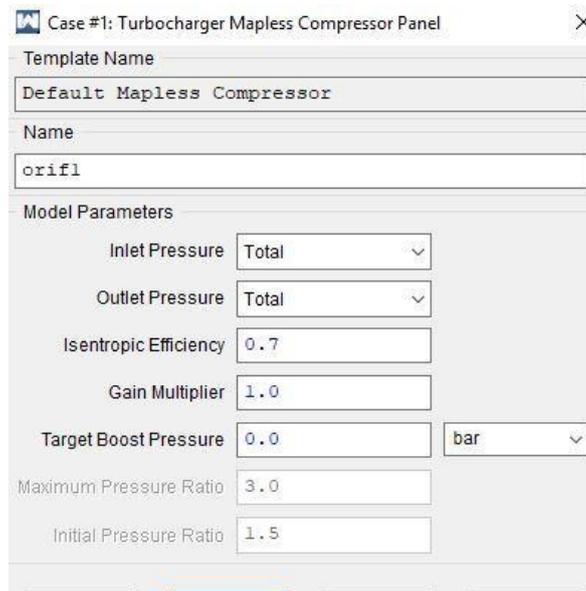


Figure 41 Turbocharger Mapless Compressor Panel

The gain multiplier is a parameter that the user can control, in order to account for differences in geometry (other than size) between the reference compressor and the model compressor. The default value of 1.0 is an adequate for this parameter in most cases.

The power consumed by the mapless compressor is calculated by the following equation:

$$W_c = \frac{C_r T_{ci}}{\eta_c} \left(PR^{\frac{\gamma-1}{\gamma}} - 1 \right) \dot{m}$$

Where:

- C_r = Specific heat of the gas;
- T_{ci} = Inlet temperature of the gas;
- η_c = Compressor isentropic efficiency;
- PR = Pressure ratio across the compressor.

5.9 Mapless Turbine

If turbine map data is not available, the Mapless Turbine can be used to represent a turbine. The turbine is modeled as a simple nozzle. The turbine is modeled as a planar boundary between adjacent duct sub-volumes and requires at least two connected ducts: one or more at the blue inlet connection point and one at the red outlet connection point. It must be connected to a single turbo shaft from the hub of the turbine.

A mapless turbine can also be used in isolation to model a turbo-compounding device. A sweep of the effective nozzle diameter will show how the engine responds, and give the power development for the turbine, as a function of the effective nozzle diameter, for a particular engine operating point.

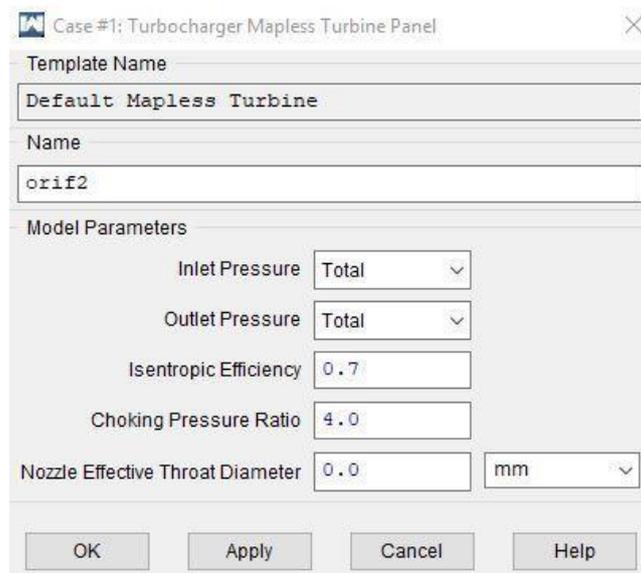


Figure 42 Turbocharger Mapless Turbine Panel

The mapless turbine is modeled as a simple nozzle, with the mass flow determined by the following equation:

$$\dot{m} = \frac{\pi}{4} D_{eff}^2 \frac{P_{0i}}{\sqrt{RT_{0i}}} PR_{ts}^{-\frac{1}{\gamma}} * \sqrt{\left\{ \frac{2\gamma}{\gamma-1} \left[1 - PR_{ts}^{\left(\frac{1-\gamma}{\gamma}\right)} \right] \right\}}$$

Where:

- D_{eff} = Nozzle effective throat diameter;
- P_{0i} = Inlet stagnation pressure;
- T_{0i} = Inlet stagnation temperature;
- PR_{ts} = Total-to-static pressure ratio.

With the critical pressure ratio of the nozzle calculated as:

$$PR^* = \left(\frac{2}{\gamma + 1} \right)^{\frac{\gamma}{1-\gamma}}$$

5.10 Turbo shaft

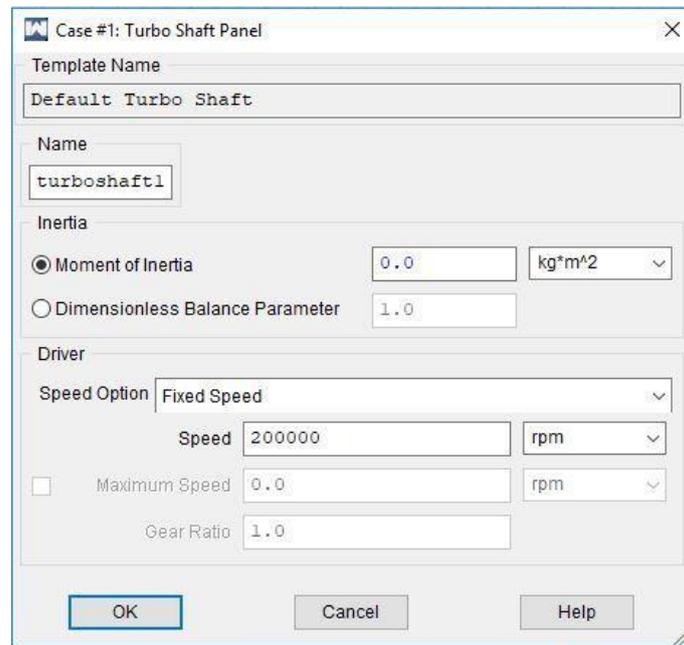


Figure 43 Turbocharger Mapless Compressor Panel

There is currently only one type of mechanical element in WAVE, the Turbo Shaft, used to create a mechanical link between one or more turbo elements from the flow network and, optionally, the engine's crankshaft. A turbo shaft element is considered a rigid body and contains the rotational speed, moment of inertia, and drive information.

There are several rules that have to be applied to turbo shaft elements and all turbocharger junctions, such as:

- Every turbo junction must be connected to one (and only one) shaft;
- Any number of and any combination of any type of turbocharger junction may be connected to a shaft;
- A turbo shaft can be connected to no junctions at all;
- A shaft cannot be actuated if every turbo junction attached to it is mapless.

5.11 Injector

WAVE fuel injectors can be attached to engine cylinders, ducts, and y-junctions. Each injector can attach to one element. Several different injectors can be attached to the same element.

For this model, a default proportional injector has been chosen.

A Proportional Injector element injects continuously into the connected flow element, according to a user-entered fuel/air ratio. WAVE automatically adjusts the fueling rate to be proportional to the instantaneous air mass at the injection point so that the fuel-air ratio is continuously controlled to the specified user input.

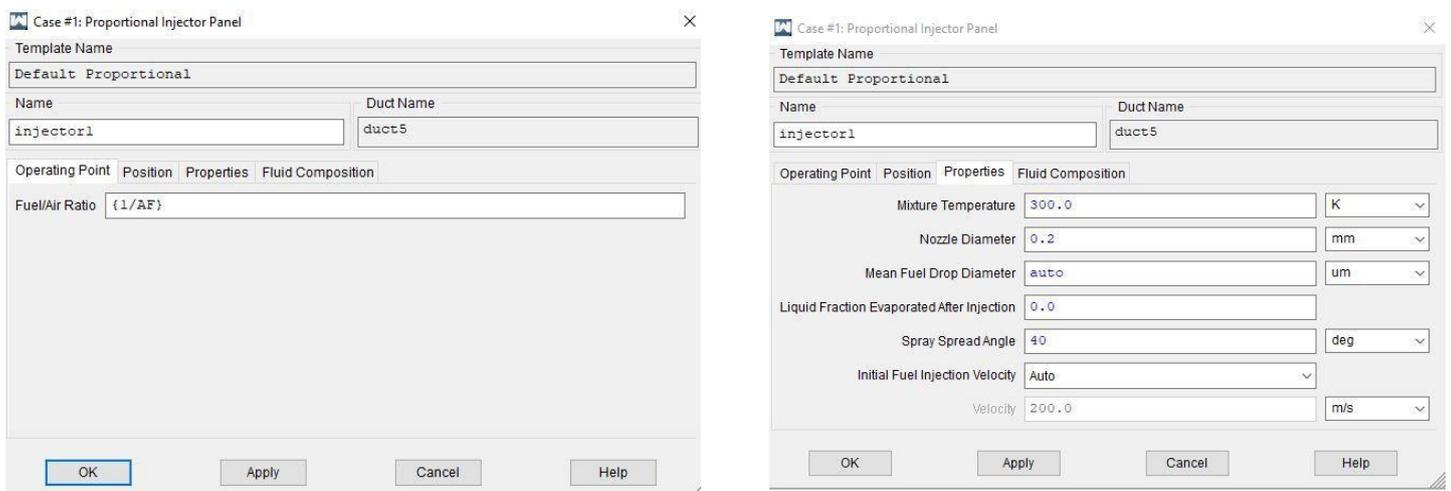


Figure 44 Proportional Injector Panel - Operating Point & Properties

5.12 Sensor

A Sensor element is used to take measurements of variables in the flow system and can be used as input to controllers built using WAVE's control elements or an external program. It is intended to model real hardware and includes a bandwidth for sampling frequency and a simple, 2nd-order filter. A sensor can connect to any one element in the flow system from which it will read its value. It can be connected as an input to any number of control elements.

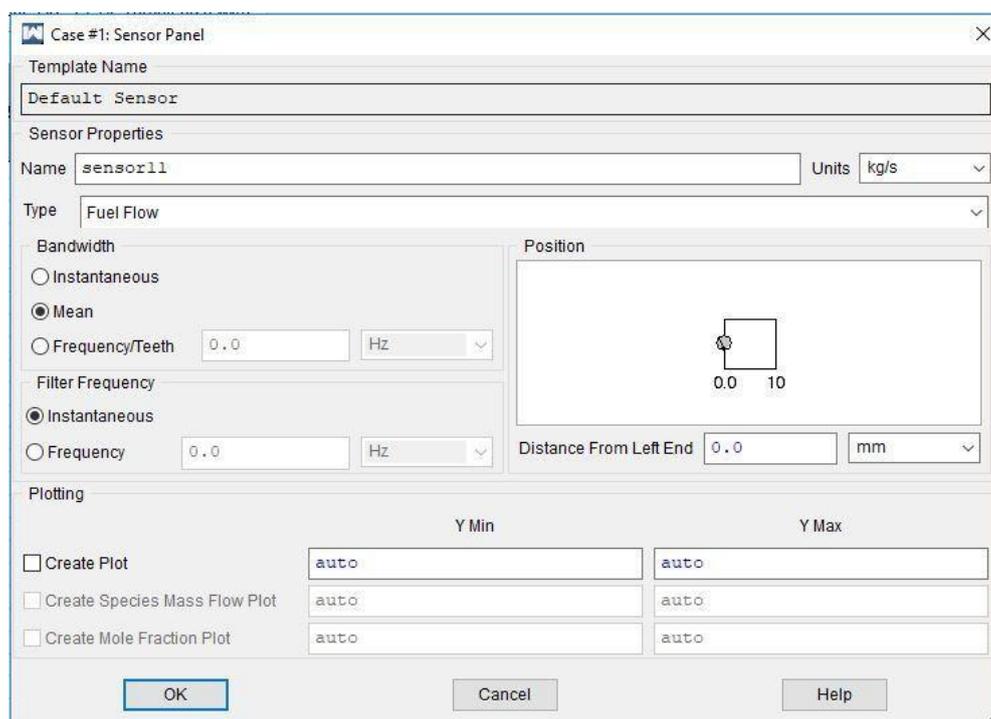


Figure 45 Sensor Panel

According to the kind of element the sensor is connected to, the first thing to do is to define its type, since sensor type determines the variable which is measured by the sensor.

Sensor bandwidth frequency describes the mechanical ability of the sensor to follow an input signal. The meaning of bandwidth and its allowed values depend on the sensor type. In this study the bandwidth will be set to “Mean”.

6. Wave Model

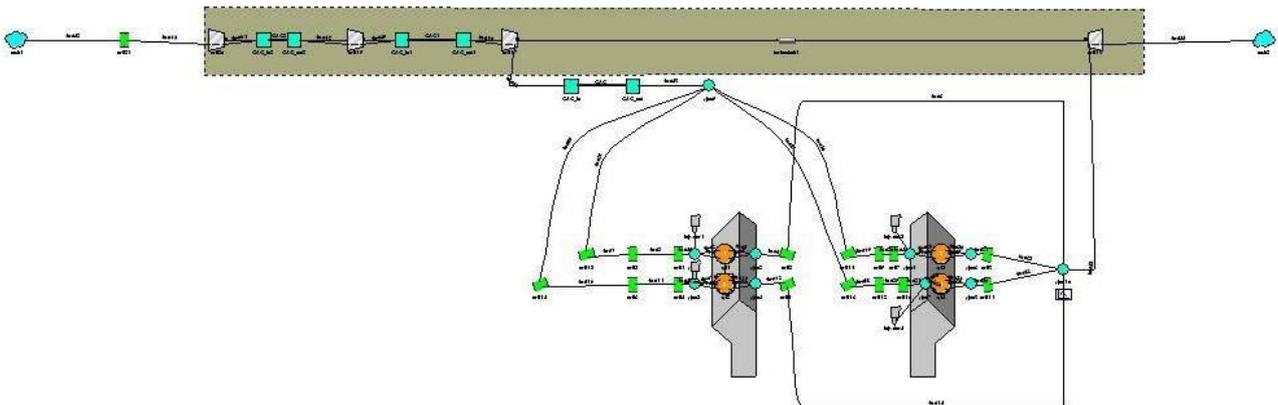


Figure 46 Wave Model

In this chapter it will be explained how the model has been built in WAVE. Since every single element used to build the model has been explained in the previous chapter, only the values will be now reported.

As it possible to see from the figure, two ambient (intake and exhaust) have been chosen. For the intake ambient, the following pressure and temperature values have been set: 1.01bar and 300 K.

The model is equipped with three maples compressors, for each of them an adiabatic efficiency of 0.7 has been chosen, in order to take into account the fact that the passage area is reduced and therefore the influence of the boundary layers towards the undisturbed flow increases. The pressure in the three compressors is increasing, it will be 3 bar in the first, 7.05 bar in the second and 12 bar in the third. Intercoolers are provided after each compressor, so complex Y-joints were chosen to model them. After each compressor the temperature of 400 K was reached and then entered the intercooler, which was released through it at 300 K. As for the intake and exhaust manifold simple Y-joints were used to shape them.

The turbine used in the model is axial type and has five stages. In order to properly evaluate the choking pressure ratio, it was assumed that the fifth stage, having the lowest inlet stator temperature, was the first to reach the choking condition limiting the axial speed for all other stages. The isentropic efficiency was assumed analytically equal to 0.85. The nozzle throat diameter has been set equal to 5 mm.

A turbo-shaft, with a rotation speed of 200000 rpm analytically calculated, was used to connect the turbine (compressors and turbine).

Considering a four-cylinder, 16-valve engine, the reference valve diameter was set at 14 mm. Antonio Tricarico's work has provided the maximum value of the valve lift. For this reason, in the profile editor the lift multiplier has been set to 0,506187, in order to obtain the desired maximum lift, i.e. 4,5 mm.

As for the geometrical parameters of the cylinder, these are set as follows: bore equal to 37 mm, stroke equal to 29 mm, clearance height equal to 7 mm, connecting rod length equal to 54.7 mm. The compression ratio was initially set to 10.

In the Engine panel, Wiebe primary combustion model and Whoschni heat transfer model are chosen. It is also possible to set the Conduction model and define the thermal properties of the engine elements, such as piston, cylinder head and cylinder barrel. Conduction sub-models are used to calculate in-cylinder surface temperatures. Accurate surface temperatures improve boundary conditions for the in-cylinder heat transfer sub-models and can be used to assist in engine component design. The simple engine conduction sub-model uses a pre-defined thermal network to represent the cylinder liner, head, piston, intake valves, and exhaust valves. This enables the prediction of the surface temperatures for the combustion chamber as well as heat rejection of the engine to the coolant. It is important to specify that in this sub-model, the combustion chamber is represented as a simplified thermal resistance network. In the thermal network, the piston, cylinder liner, head, intake valves, and exhaust valves are each made of a single type of material. The cylinder liner and head are treated as a one-dimensional thermal resistance, so are the intake and exhaust valves. The piston is presented as a Y-junction type of thermal resistance, that is, the heat is transferred from the cylinder gas via the gas-side surface to interior node, then is diverted to two directions. One part of the heat goes to the piston oil-cooled side, the other part to the piston skirt.

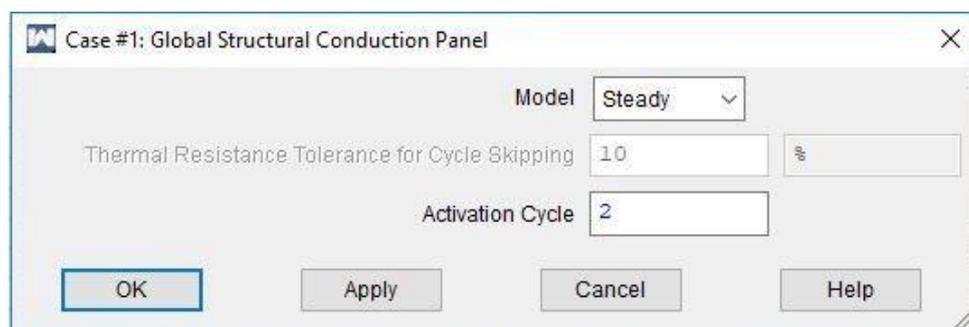


Figure 47 Global Structural Conduction Panel

When the steady-state model is activated, WAVE will solve a simplified conduction model with axial conduction turned off for until the user-specified activation cycle, when it will activate the full 2-D conduction model.

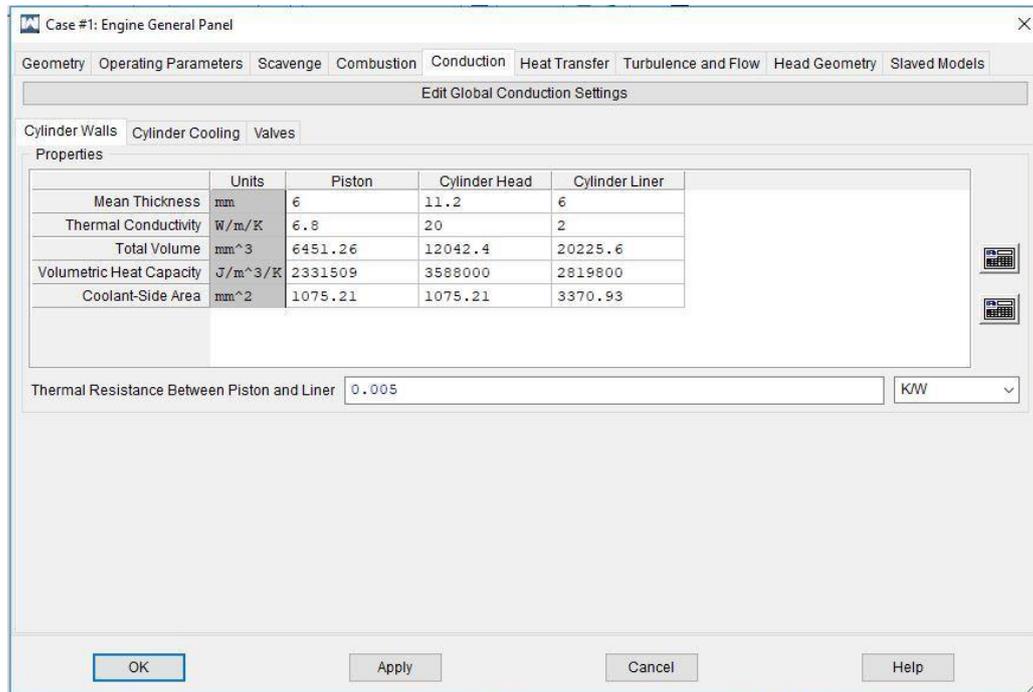


Figure 48 Engine General Panel – Conduction, Cylinder Walls Tab

In the Cylinder Walls tab, the user sets the physical properties of the piston, liner, and head for engine cylinders using the simple conduction model. The Thermal Resistance Between Piston and Liner value can be left at its default value of 0.005 [K/W].

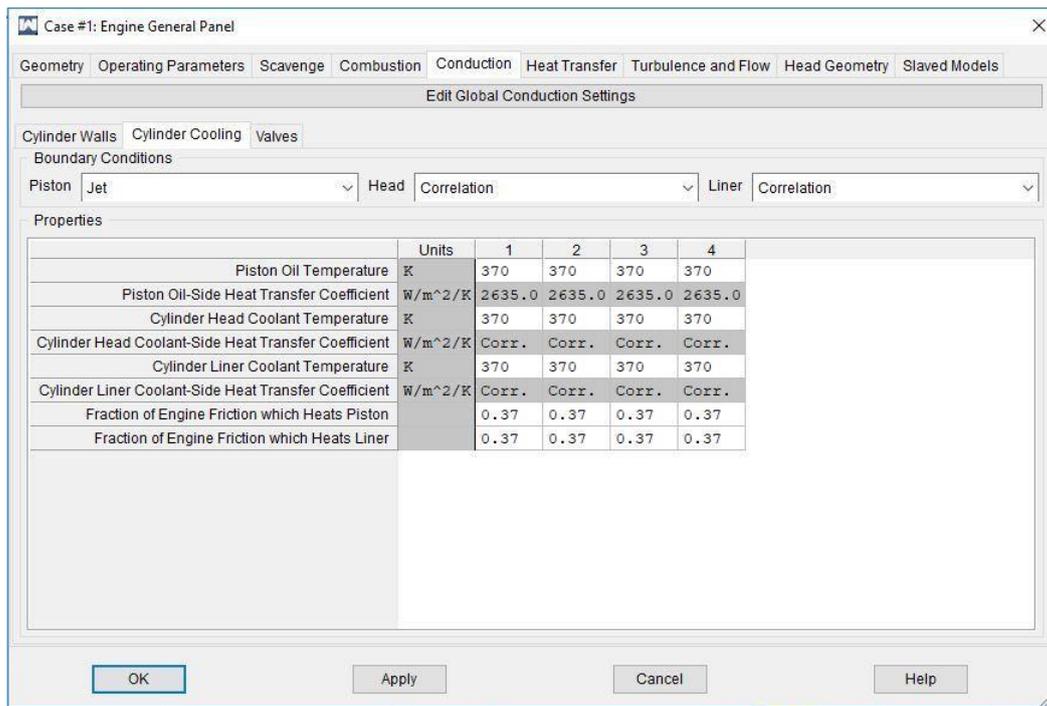


Figure 49 Engine General Panel – Conduction, Cylinder Cooling Tab

The Cylinder Cooling tab defines the coolant-side boundary conditions for engine cylinders using the simple conduction model. According to Taylor (1985), about three fourths of total engine friction loss is caused by friction between piston and liner. When friction heat is divided equally, their scaling factors are equal to about 0.37, respectively. For the cylinder head and liner, the heat transfer coefficient can be calculated from an empirical equation, in which the effect of nucleate boiling on heat convection is considered, see Howarth (1966):

$$h = 1,4846795 * Q^{0,644}$$

Where:

- h = Heat transfer coefficient [W/m² /K];
- Q = Heat flux [W/m²].

With:

$$50 \leq h \leq 15000 \text{ [W/m}^2 \text{ /K]}$$

This correlation is valid only if the engine is warmed up. The general heat conduction equation is discretized as an equation for the thermal network:

$$\sum_n \frac{T'_{m,n} - T'_{m,p}}{R_{m,n}} = \frac{C_m}{\Delta\tau} (T'_{m,p} - T_{m,p})$$

Where: $C_m = \rho V_m c_p$

With:

- T = Temperature at the previous time step;
- T' = Temperature at the current time step;
- V_m = Volume of metal;
- R_{m,n} = Thermal resistance.

Subscript m refers to solid wall and subscripts p and n designate interior and neighboring nodes, respectively.

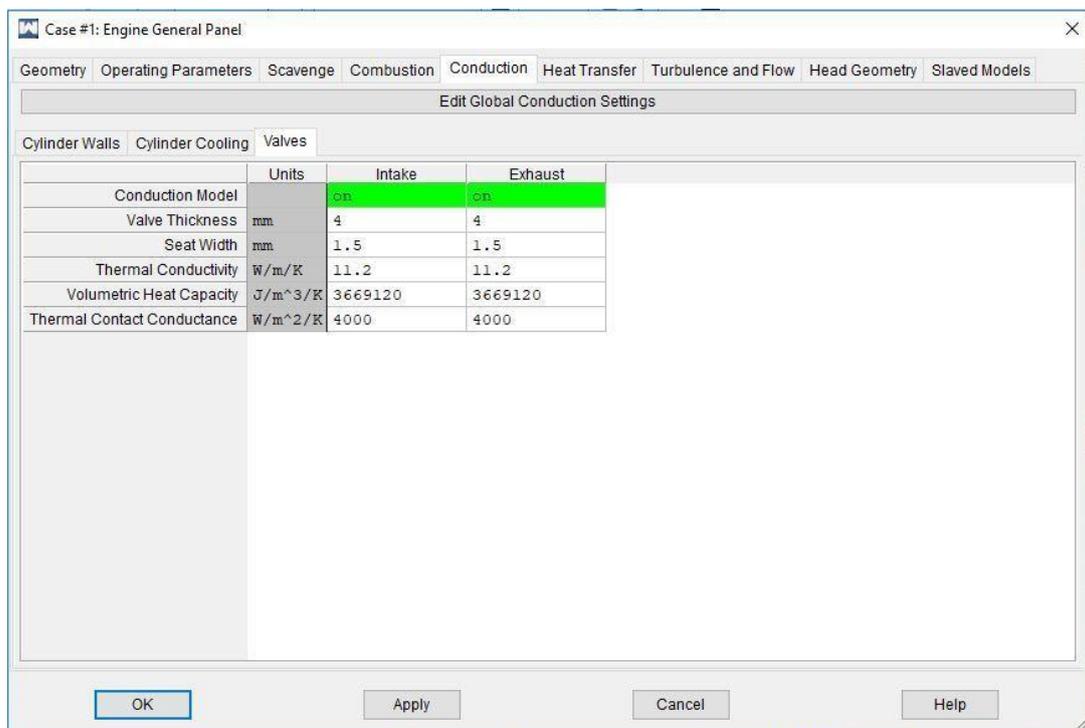


Figure 50 Engine General Panel – Conduction, Valves Tab

Finally, the Valves tab defines the physical properties of poppet valves for engine cylinders using the simple conduction model.

6.1 Valve Timing

In this chapter it will be explained how the valve timing has been set before the optimization process. In order to establish the optimal timing, changes in total power and specific consumption have been observed.

As for the intake event:

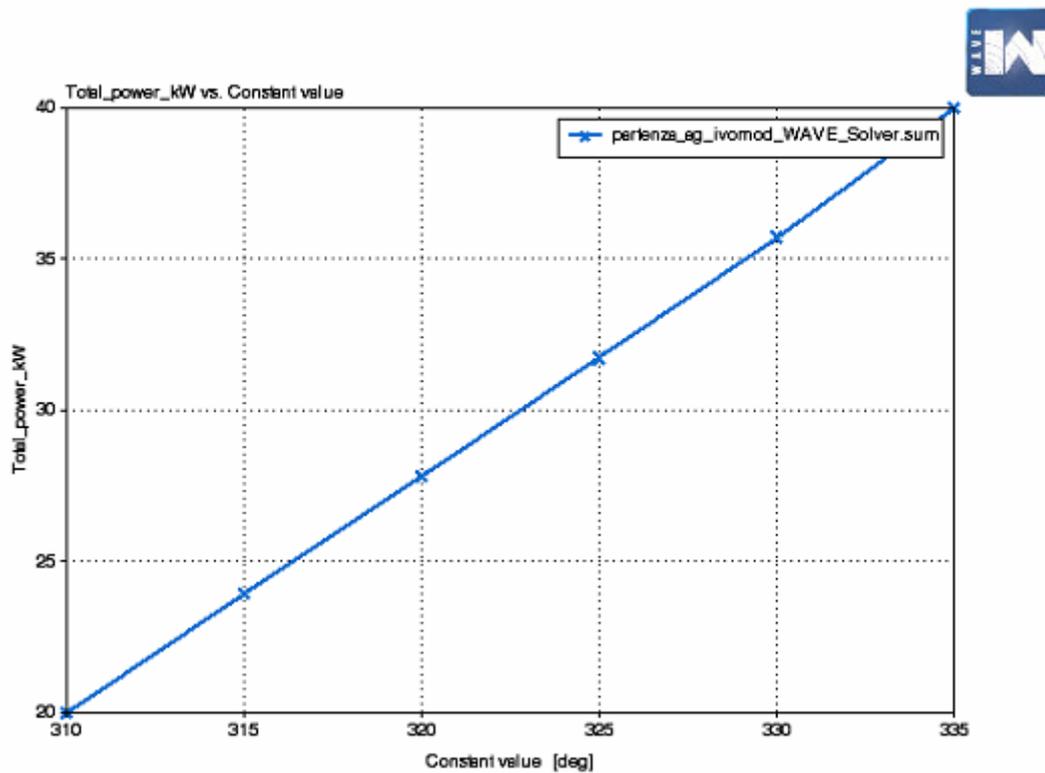


Figure 51 Total Power vs IVO

It is possible to observe how the total power increases with the IVO value; taking into account the overall BSFC:

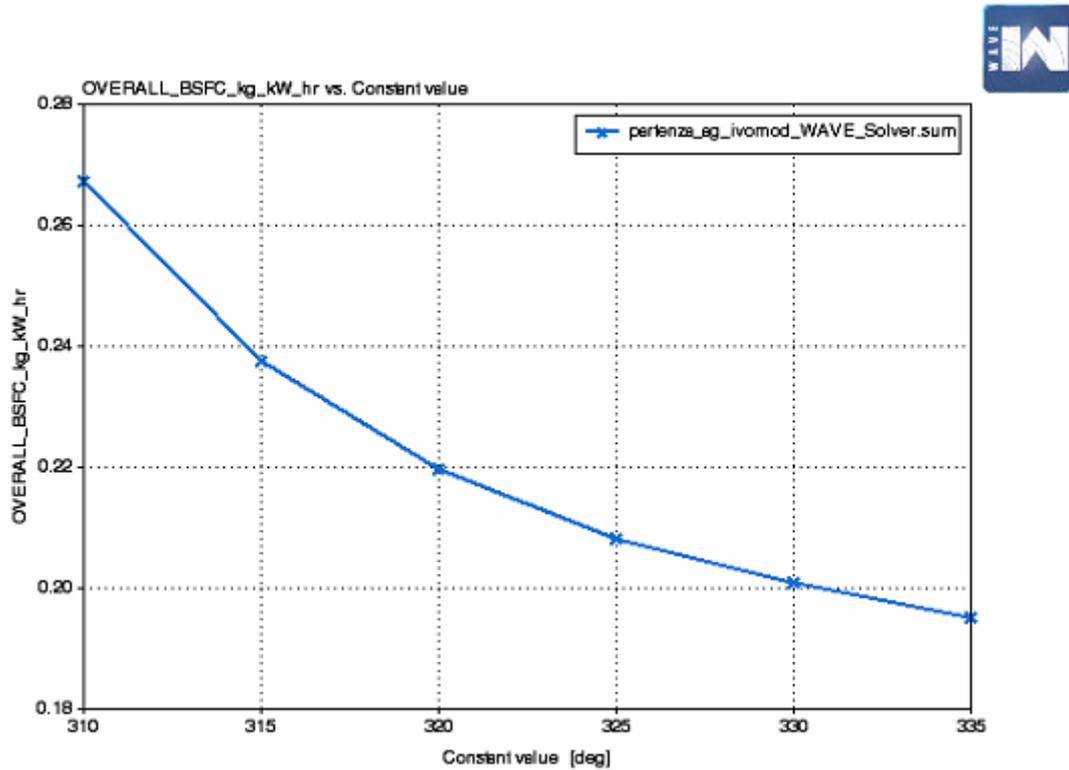


Figure 52 Overall BSFC vs IVO

In the end It can be said that increasing the value of IVO, there is a beneficial effect both in total power and overall BSFC, as the first increases and the latter decreases.

Therefore, the IVO value has been initially set to 330 deg.

Now, moving on to the intake duration, it is possible to find that the same discourse applies to it:

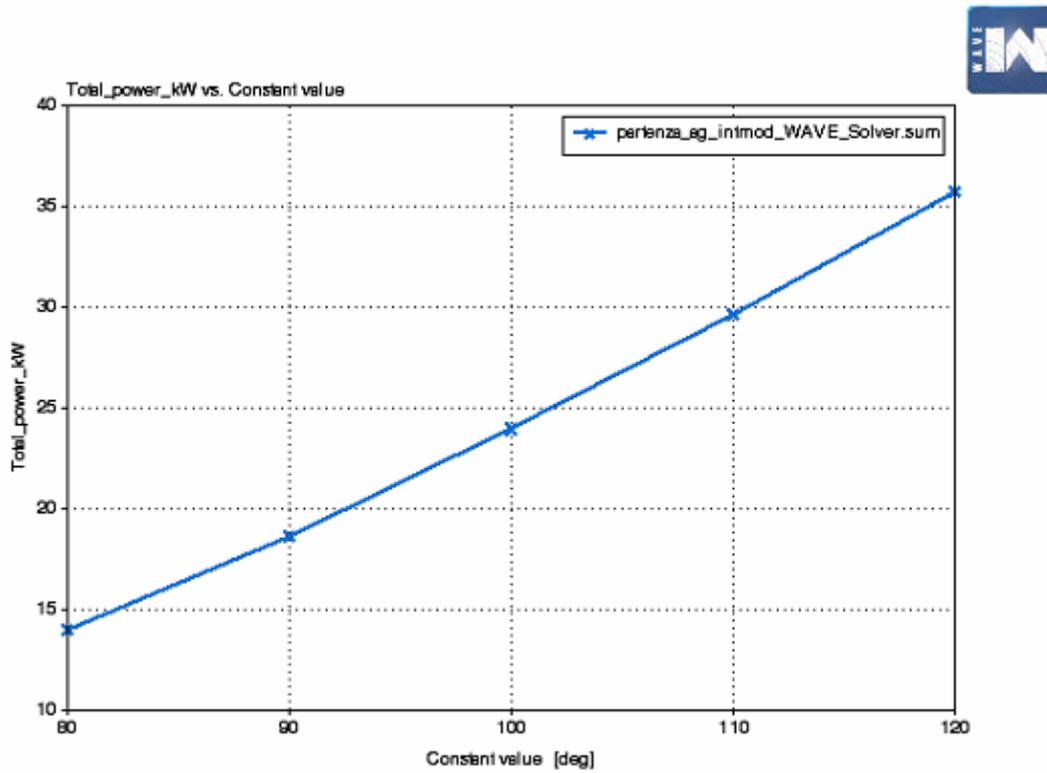


Figure 53 Total power vs Intake duration

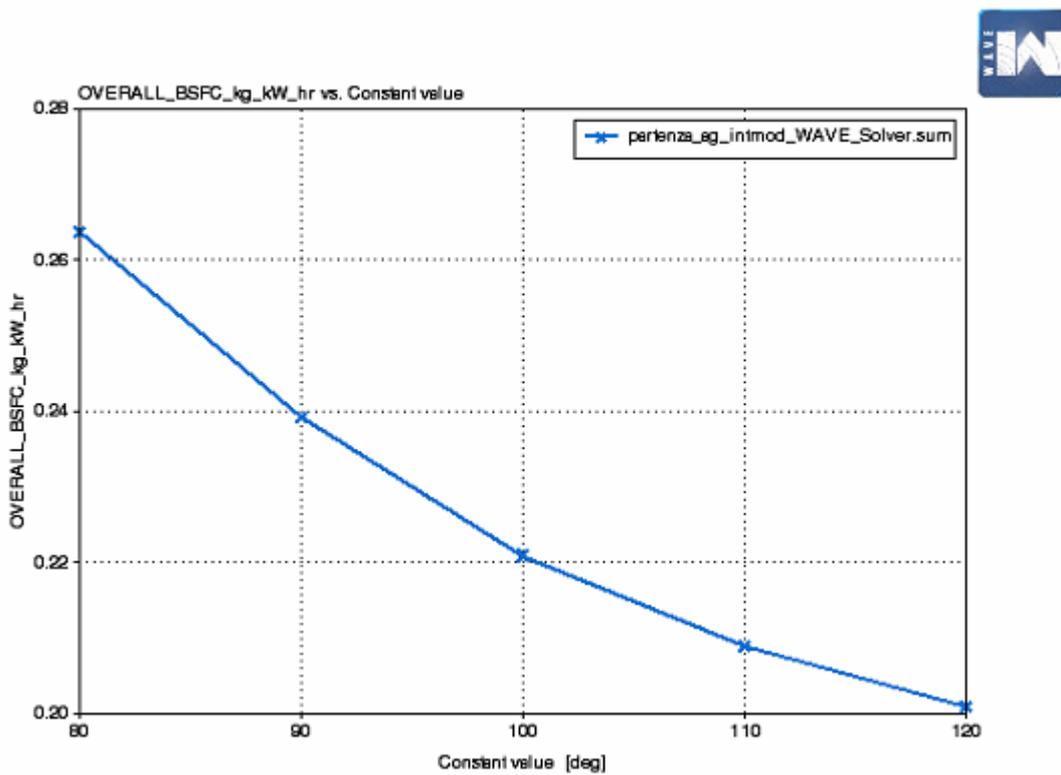


Figure 54 Overall BSFC vs Intake duration

The intake duration has been set to 120 deg, so with an IVO value to 330 deg, we will have a IVC to 450 deg.

As for the exhaust event:

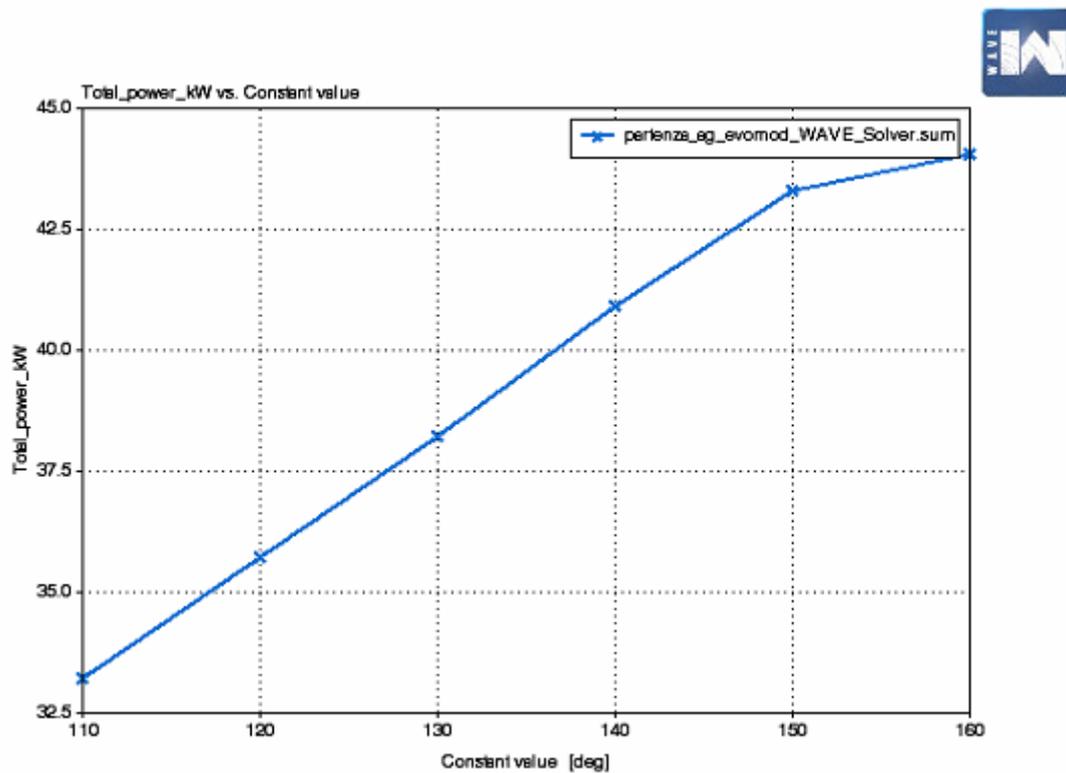


Figure 55 Total power vs EVO

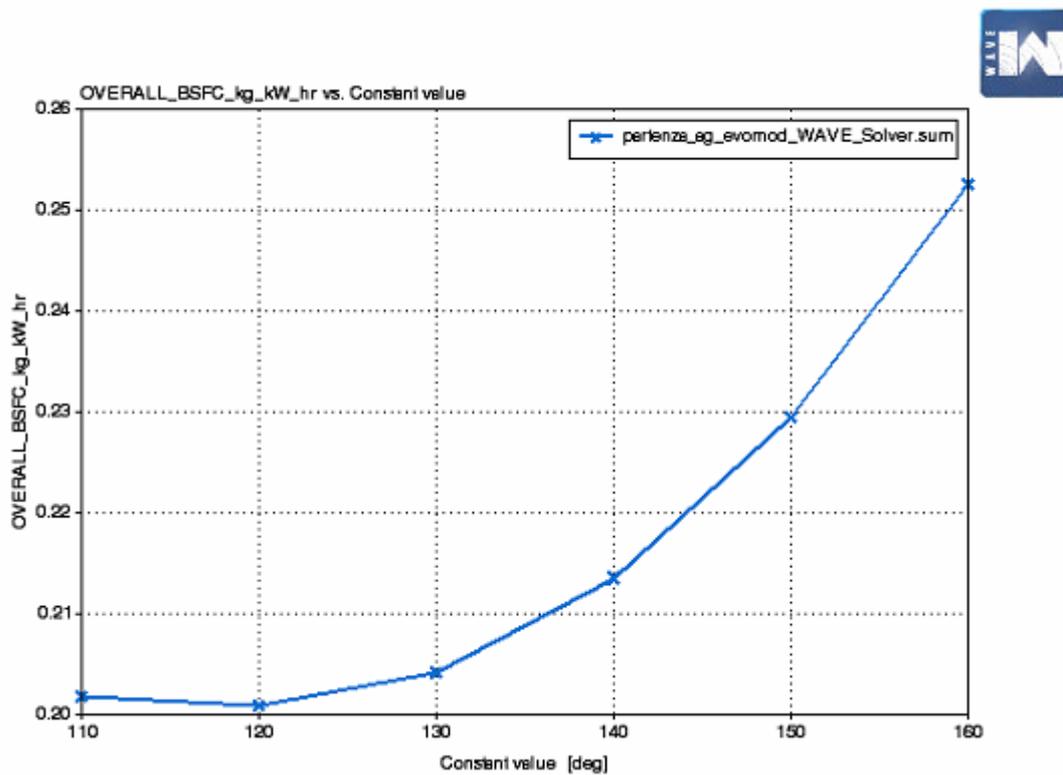


Figure 56 Overall BSFC vs EVO

Figure 55 and figure 56 show that, unlike the IVO, increasing the EVO has a beneficial effect only in total power, since even the overall BSFC augments.

As a result, the EVO value has been set to 120 deg.

Considering, now, the exhaust duration:

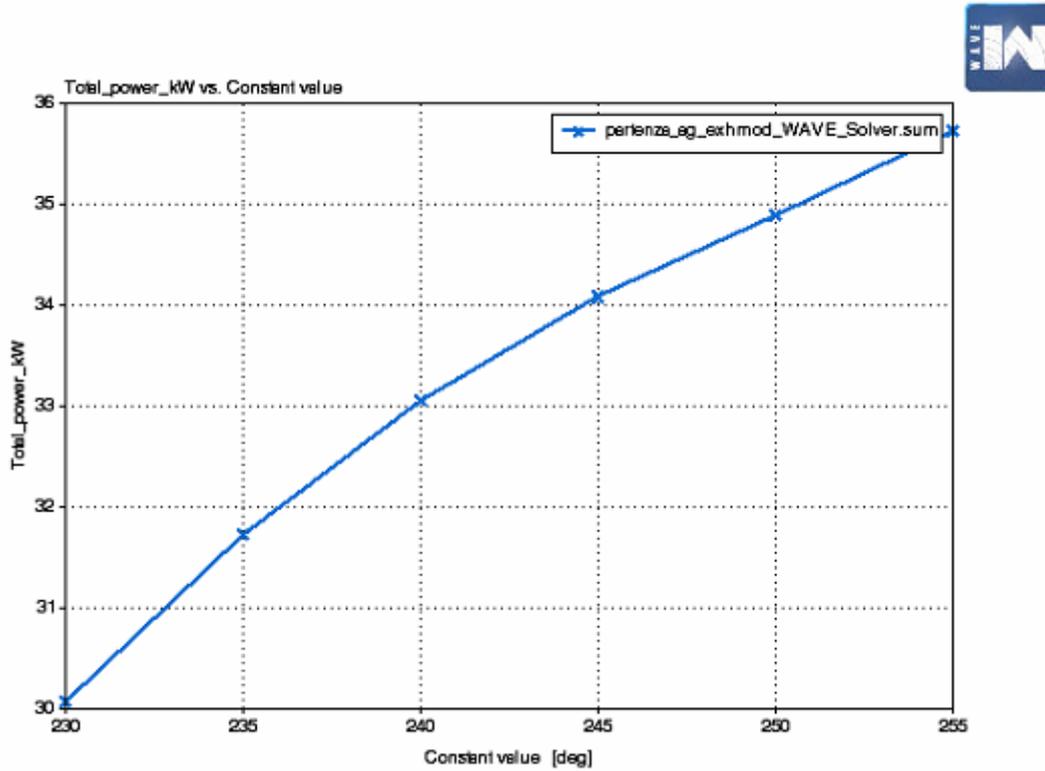


Figure 57 Total power vs Exhaust duration

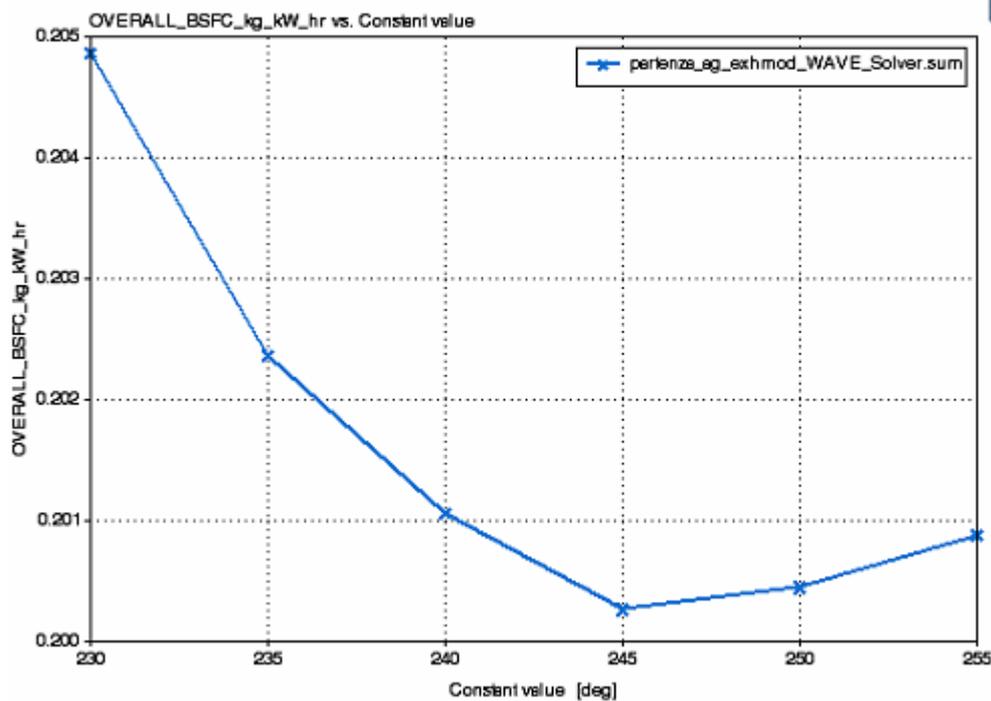


Figure 58 Overall BSFC vs Exhaust duration

As it is possible to find from the two graphs, as the exhaust duration increases, the total power increases; but as for the overall BSFC, we can observe an initial decrease, followed by an increase after the value of 245 deg.

With a compromise choice, the exhaust duration has been set to 255 deg, since the total power is maximized and the overall BSFC has a little significant increase. In the end, with an EVO to 120 deg, the EVC will be at 375 deg.

7. Optimization process using HEEDS

As it was stated in the "Previous work" sub-chapter, the results that Salvatore Chierchia obtained through his work, have shown a low engine and turbogroup net power and low valve lifts. The optimization process, that he conducted, was aimed at minimizing the overall BSFC, exhaust gas temperature and knock intensity, by varying the location of 50% burn point (CA₅₀), the compression ratio (CR), the nozzle effective diameter (D_n), the lift multiplier and the intake duration. Having obtained an intake duration value of 100,2 deg was punitive for the valve lift values, as it has been explained before; on the other hand, the aim of minimizing the overall BSFC, affect the total power value, which resulted low. Been said that, the optimization process that I conducted was aimed at finding a trade-off between the overall BSFC and the total power, resulted by the sum of the brake engine power and the turbogroup net power.

7.1 HEEDS

Optimization is generally a complex process requiring the user to set up the parameters to vary and the targets to try to achieve. To solve an optimization problem manually, the user needs to choose combinations of the variables and perform an analysis for each combination, this would take much time, given the nature of most engineering design environments, limitations on time and resources prevent manual search methods from reliably producing the best possible design; that is why a software like HEEDS is needed. HEEDS (Hierarchical Evolutionary Engineering Design System) is a robust design exploration and optimization software package that automates the search for better and more robust solutions within a given design space, all this is done in a very short space of time. HEEDS enables the user to find designs that perform extremely well according to multiple criteria, while simultaneously satisfying multiple constraints and using a large number of variables.

Ricardo software has partnered with Red Cedar Technology to integrate WAVE with their HEEDSTM engineering design exploration software product. The HEEDS Optimization tool in Ricardo Software is a "light" version of the full HEEDS optimization package. It runs only with Ricardo Software e.g. WAVE and IGNITE. It cannot be opened directly, but Ricardo Software can be used to open existing HEEDS files.

To optimize a design, the user needs to identify its measurable parts. Every optimization problem can be specified in terms of the following:

- A set of variables that specify what values can be changed from the baseline design in order to define a new design;
- A baseline design to use as the frame of reference;
- One or more objectives to use to judge how good a design is. The objective is what has to be maximized or minimized;
- A set of constraints that will determine whether or not a design is feasible.

Once the user has identified the variables, you need to determine their allowable values.

The combined allowable range of all variables determines the design space that will be searched.

7.1.1 Variables

The project variables represent the quantities or values in the input file that are to be varied during a design evaluation. To be used as a variable, a quantity that represents the variable must be present in at least one of the input files used in the project. When the user defines a variable, it has to be named. HEEDS will use that name to identify the variable in the evaluation, it is also possible to assign it a type. A variable may be classified as continuous, discrete, dependent, or parameter:

- ***Continuous variable:*** The values of continuous variables are chosen from a range of real numbers, the user specifies the minimum value and the maximum value of the range;
- ***Discrete variable:*** this type can take on any value among a specified set of choices. This set may contain specific numeric values. It may also contain non-numeric choices;
- ***Dependent variable:*** The value of a dependent variable is determined from other values supplied during the evaluation. Dependent variables can be based on the value of another variable (or several of them) and/or determined from the value in a response that has already been calculated for the design. A dependent variable can simply assume the value of another variable or response, or you may define a formula to calculate the value;
- ***Parameter variable:*** A parameter variable is a variable whose nominal value does not change. It can be used to assign a constant value to a variable for a given run.

Variables			
Responses			
Additional Files			
>> Model parameters			
	HEEDS variable name	Parameter	Optimize
1	ca50	CA_50	Whole run
2	cr	CR	Whole run
3	dn	D_n	Whole run
4	evo	EVO	Whole run
5	intd	INT_DUR	Whole run
6	ivo	IVO	Whole run

Figure 59 HEEDS - Variables

In this study, the variables which have been taken into account are: the location of 50% burn point (CA_50), the compression ratio (CR), the intake valve opening (IVO), the exhaust valve opening (EVO), the intake duration (INT_DUR) and the nozzle effective throat diameter (D_n).

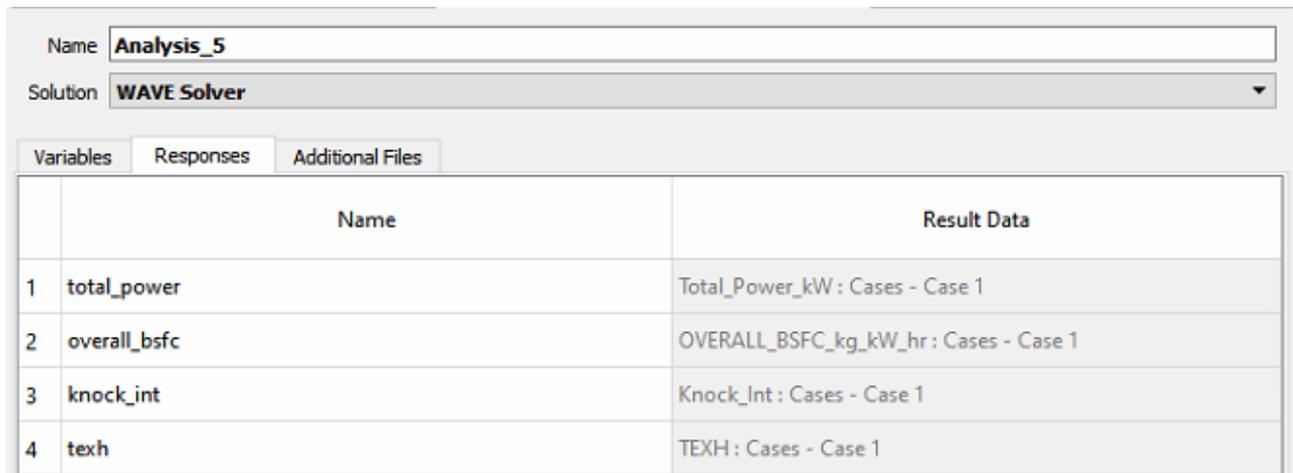
Study <input type="text" value="Study_1"/>				
	Variable	Min	Baseline	Max
1	ca50	10	25	30
2	cr	9	10	21
3	dn	3	5	6
4	evo	110	120	160
5	intd	80	120	140
6	ivo	310	330	340

Figure 60 HEEDS – Study

These variables were set as continuous, the *figure 60* shows their range.

7.1.2 Responses

Responses are either designated values in the output file or values that are calculated from other variables and responses.



	Name	Result Data
1	total_power	Total_Power_kW : Cases - Case 1
2	overall_bsfc	OVERALL_BSFC_kg_kW_hr : Cases - Case 1
3	knock_int	Knock_Int : Cases - Case 1
4	texh	TEXH : Cases - Case 1

Figure 61 HEEDS - Responses

In this study the summary quantity that were considered are: the total power, that we want to be maximized, the overall BSFC, that has to be minimized, the knock intensity and the exhaust gases temperature, both considered as constraint in the optimization process.

After the user has set all the variables, their variable range and the responses, will be able to export the study created in HEEDS.

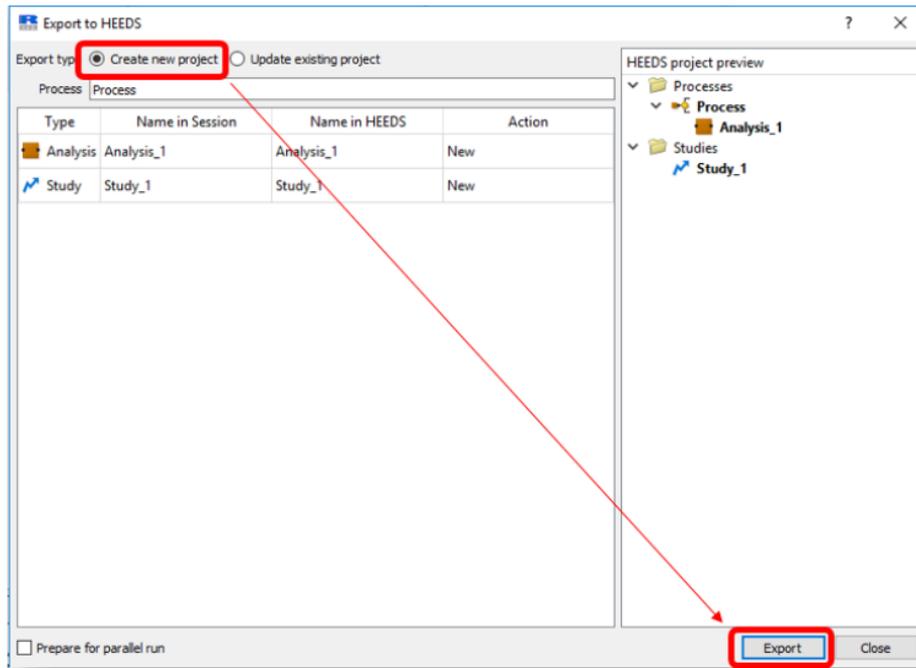


Figure 62 Exporting the project to HEEDS

7.1.3 Objectives

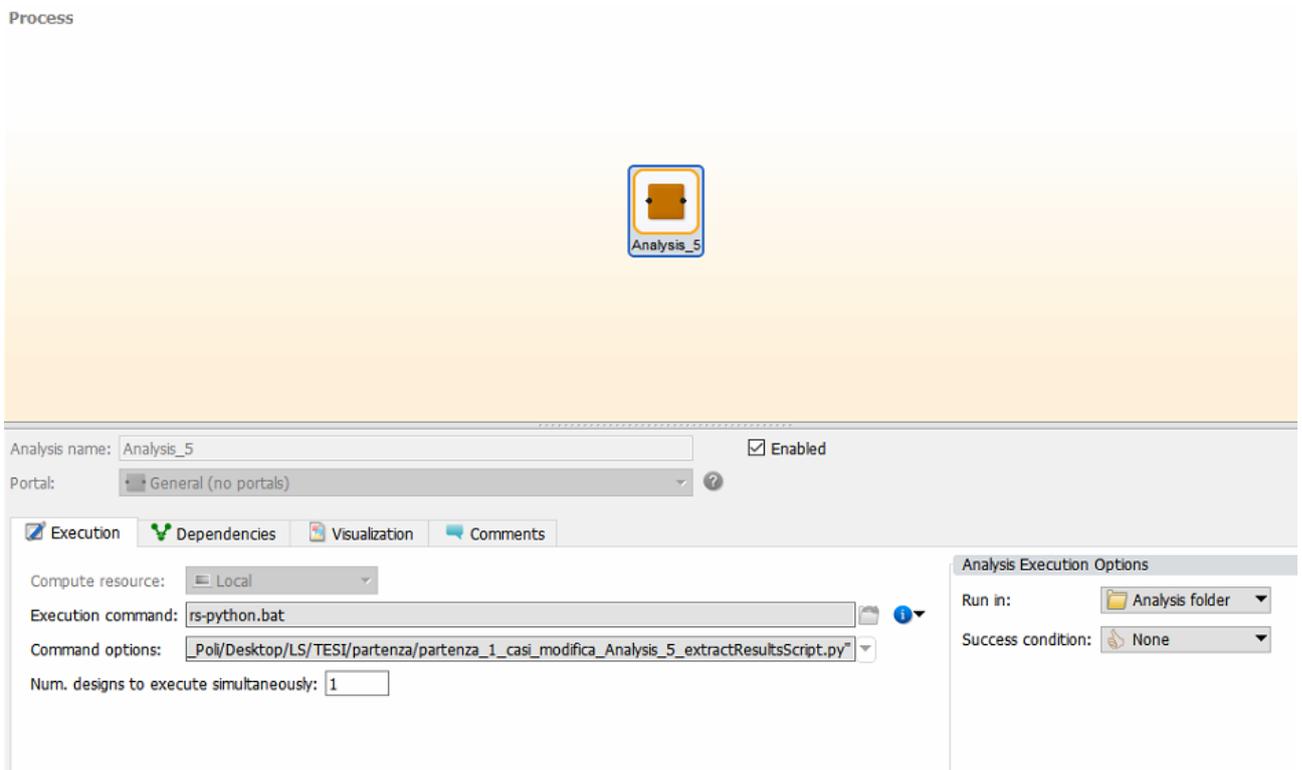


Figure 63 HEEDS Interface

The first thing the user needs to do is to set the number designs to execute simultaneously, in this case 1 has been set. Pareto Tradeoff Study was chosen within the optimization method tab.

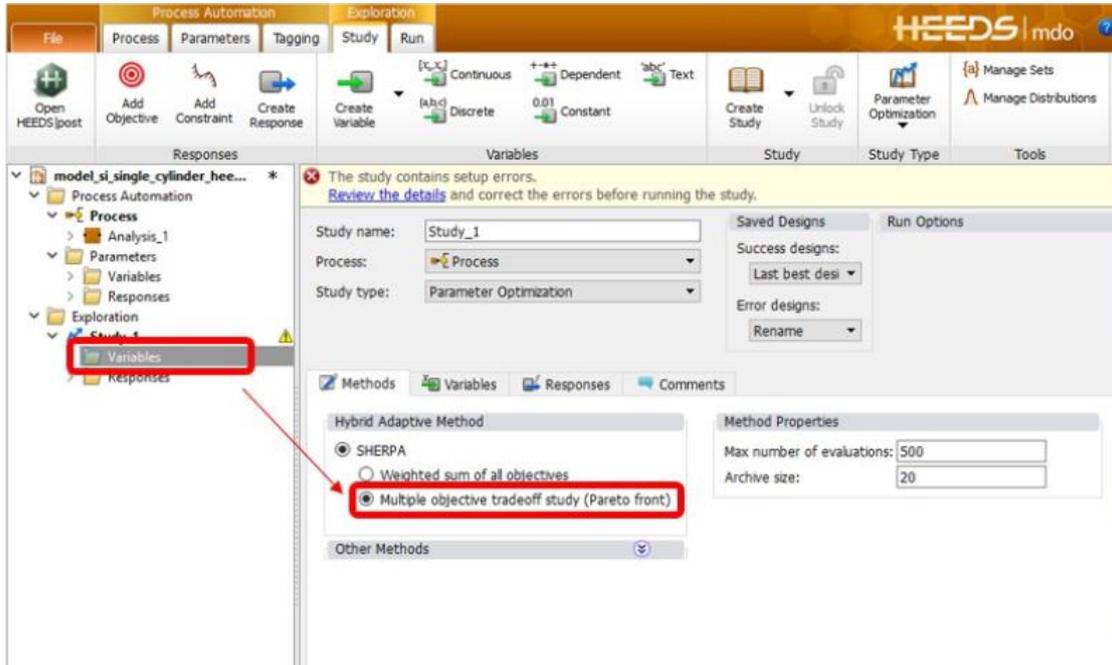


Figure 64 Setting the method

It is possible, now, to set the objectives:

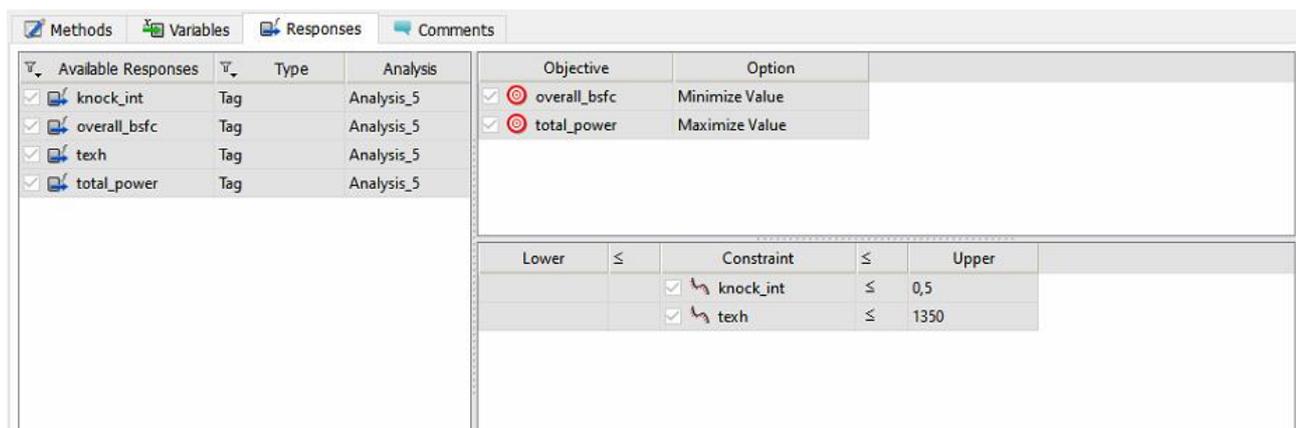


Figure 65 Setting the objectives

As it was stated before, the objectives are: maximize the total power and minimize the overall BSFC. Knock intensity and the exhaust gases temperature are set as constraint, in particular, we want HEEDS to consider as feasible, the designs which respect those constraints, i.e.:

- Knock intensity $\leq 0,5$
- Exhaust gases temperature ≤ 1350 K.

The optimization process is, now, ready to start. The results page is updated while the optimization is running.

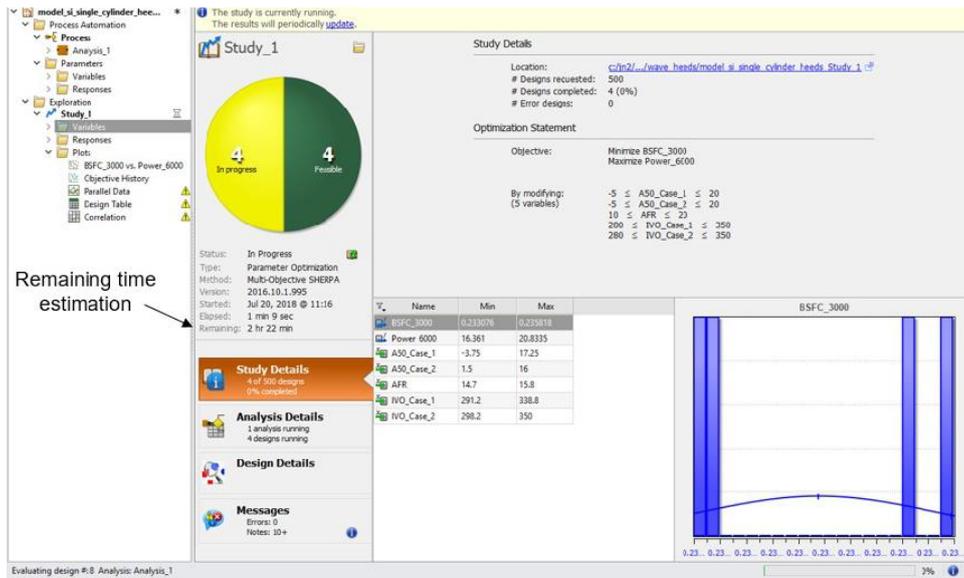


Figure 66 Running the study

7.1.4 End of the run

When the optimization process ends, all the feasible and unfeasible designs, based on whether the constraints have been violated or not, will be reported.

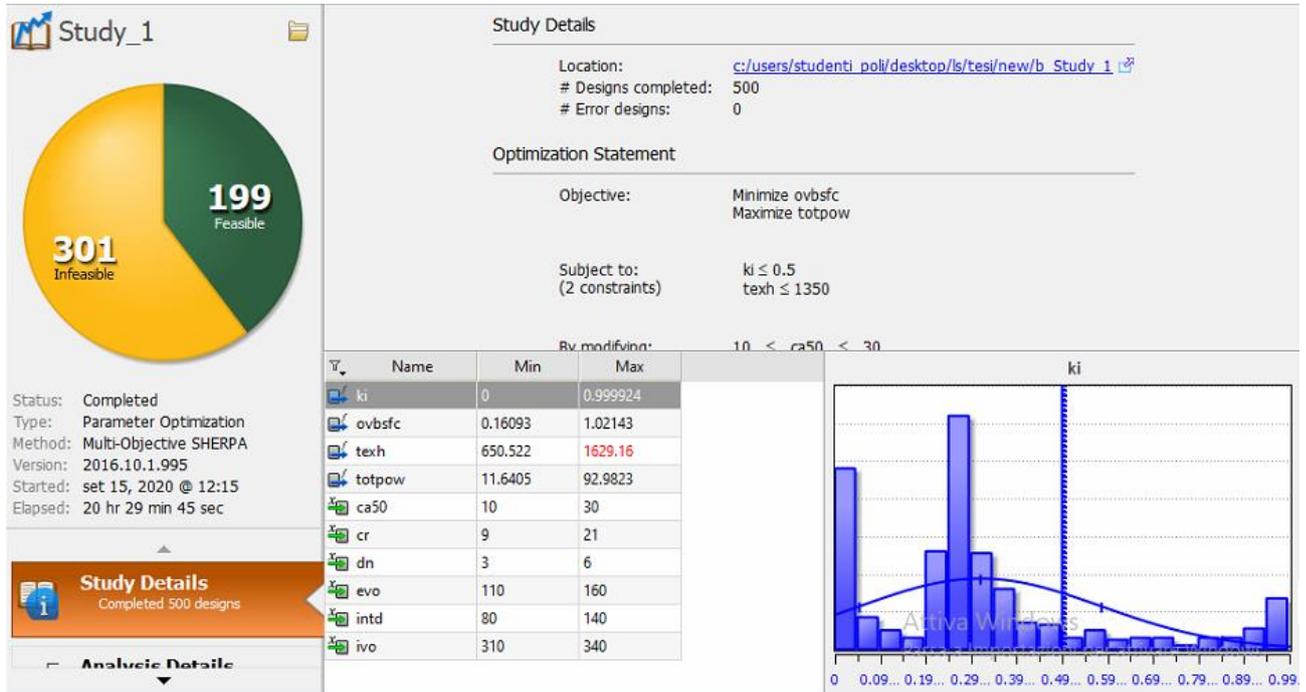


Figure 67 End of the run

8. Results

After the optimization process ends, it is possible to obtain a Pareto curve, showing the relation between the two objectives, i.e. the total power and the overall BSFC:

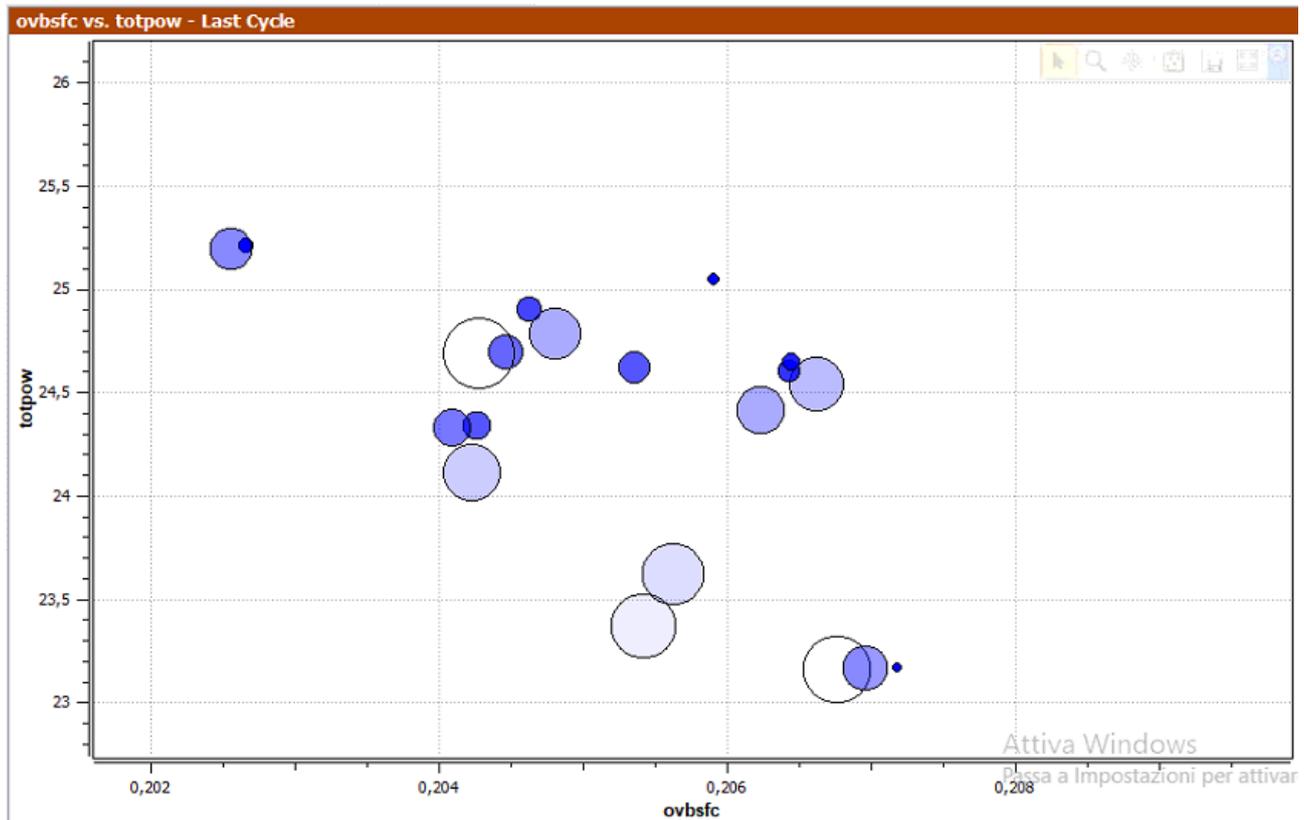


Figure 68 Pareto curve

The design table, shown in *figure 69*, allows us to look at all the feasible and unfeasible design, the latter, signed in red, i.e. those not respecting the constraint imposed, with all the variable values.

Study_1: Design Table									
Design ID	ki	ovbsfc	texh	totpow	ca50	cr	dn	evo	intd
1	0,634892	0,172696	1567,92	62,5119	25	10	5	120	120
2	0,978507	0,173381	1458,02	45,8302	11	13,44	3,78	128,5	105,2
3	0,999494	0,176759	1475,31	37,4722	21,6	14,04	3	149,5	120,8
4	0,999788	0,188072	1283,19	66,641	18,4	17,88	4,74	157,5	137
5	0,999416	0,192785	1431,28	29,895	15,2	19,08	3,15	112,5	117,8
6	0,999924	0,169603	1391,26	48,6958	24,8	20,4	3,63	144	127,4
7	0,738976	0,377398	958,528	26,2185	13,2	16,56	4,11	154,5	83
8	0,225665	0,289998	1143,85	47,3811	25,8	12,84	5,22	160	102,2
9	0	0,238231	1567,07	21,9367	30	11,52	3,33	133,5	86,6
10	0,910025	0,16908	1464,01	49,5216	29	17,16	4,59	118	99,2
11	0,564238	0,176614	1564,55	51,2102	20,6	9	4,89	110	111,8
12	0,927926	0,174363	1466,17	48,9232	26,8	14,64	3,96	120,5	133,4
13	0,300581	0,363068	982,769	42,0984	27,8	15,96	6	147	114,8
14	0	0,275319	1357,11	25,3117	17,4	9,6	3,48	152	95,6
15	0,995357	0,233425	1115,93	36,7762	19,4	19,68	5,67	139	92,6
16	0,564197	0,193626	1434,92	62,9907	23,6	10,32	5,37	131	124,4
17	0,972632	0,595511	785,257	18,7999	12,2	21	5,04	136,5	108,2
18	0,0554736	0,238685	1274,23	19,1569	14,2	12,12	5,52	115,5	89,6
19	0,999716	0,170309	1380,15	74,3674	16,4	15,36	5,85	123	130,4
20	0,983411	0,167071	1488,49	70,2241	10	10,92	4,41	141,5	140
21	0	0,248967	1395,72	21,9994	21,2	9,36	4,23	125	83,6
22	0	0,265601	1219,48	16,7934	16,2	12,72	5,82	118	84,8
23	0,854469	0,16801	1534,47	65,6326	25,2	11,04	4,8	118	125

Figure 69 Design Table

The first design that has been chosen, gave as output:

- CA₅₀ = 14.2 deg;
- CR = 12;
- D_n = 5.52 mm;
- EVO = 118.5 deg;
- IVO = 325.3 deg;
- INT_DUR = 92 deg.

The next step was to import these values in WAVE, and to start a run. The results calculated by the WAVE processor will be reported in the following table:

Total power [kW]	28.55
Overall BSFC [g/kW/hr]	201
Engine Power [kW]	25.57
Turbine produced power [kW]	11
Net power of the Turbogroup [kW]	3
Knock Intensity [-]	0.40

TEXH [K]	1352
Turbine Outlet Temperature [K]	918
Turbine Inlet Temperature [K]	1215
Turbine Inlet Pressure [bar]	5.52
Turbine Outlet Pressure [bar]	1.2
Mass flown through turbine [kg/hr]	102.73
Peak Cylinder Pressure [bar]	164.81
Brake thermal engine efficiency [%]	37

Table 3 First design results

An important output has been given by the correlation table, which shows how the variables influence the objectives.

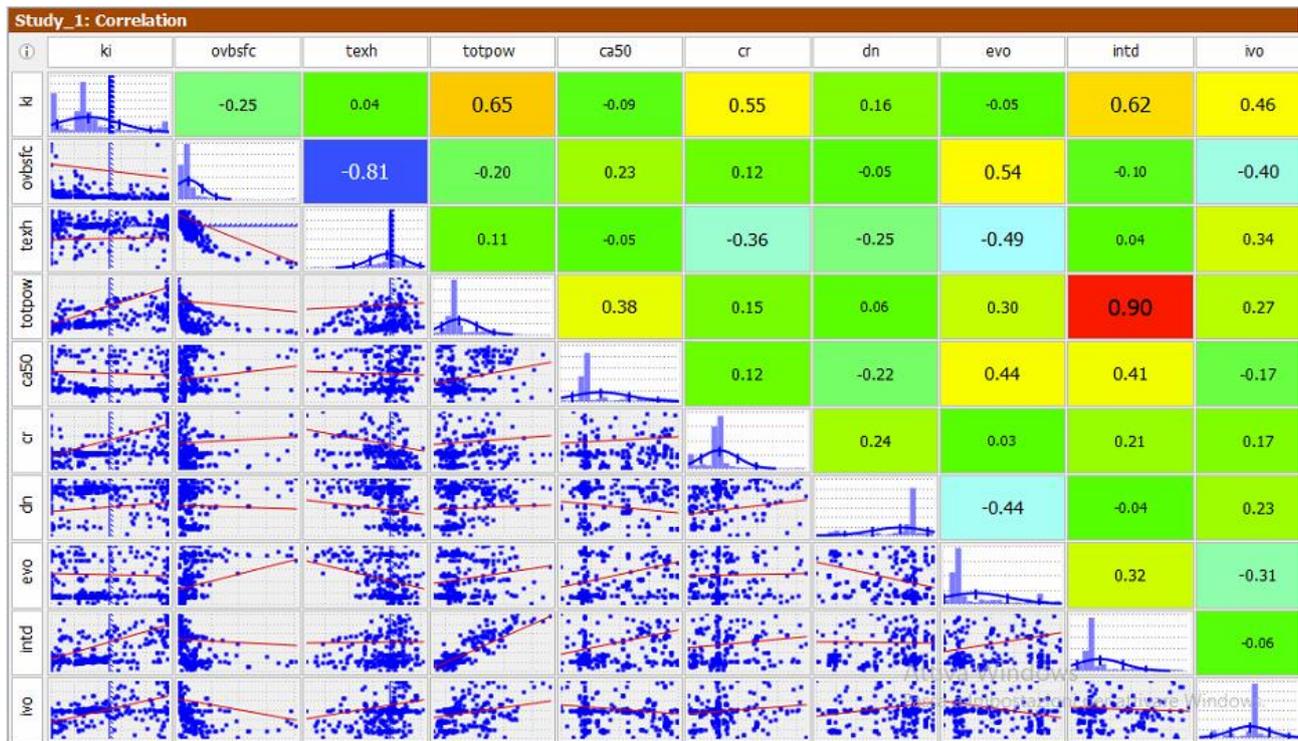


Figure 70 Correlation table

As it possible to notice, there is a strong correlation, indicated by a correlation index of 0.90, between the intake duration and the total power, this means that, the first is the variable that affects the most the total power. As a result, since there is a direct proportionality between these two factors, and since the objective is to maximize the total power, we need to augment the intake duration in order to obtain a greater power.

On the other hand, with the increase of the total power, it is possible to see a greater knock intensity, because, between them, there is another strong correlation. In order to solve this problem, the compression ratio will be reduced, there being a great impact on the knock intensity, and a weak correlation with the total power, this means that, reducing the compression ratio won't affect significantly the latter, but will reduce a lot the knock intensity.

Another strong correlation, with an index of -0.81, is showed by the overall BSFC and the exhaust gases temperature, this time represented by an undirected proportionality, meaning that an increase in the exhaust gases temperature is followed by a decreasing in the overall BSFC, which is one of the aims of this work.

For these reasons, a better design has been chosen:

- CA₅₀ = 20.6 deg;
- CR = 9;
- D_n = 4.89 mm;
- EVO = 110 deg;
- IVO = 332.2 deg;
- INT_{DUR} = 102.2 deg;
- IVC = 434.4 deg;
- EVC = 365 deg.

Importing these values in WAVE, the following results have been found:

Total power [kW]	42.41
Overall BSFC [g/kW/hr]	181
Engine Power [kW]	33.09
Turbine produced power [kW]	20.30
Net power of the Turbogroup [kW]	9.31

Knock Intensity [-]	0.36
TEXH [K]	1563.92
Turbine Outlet Temperature [K]	1029.94
Turbine Inlet Temperature [K]	1442.35
Turbine Inlet Pressure [bar]	8.21
Turbine Outlet Pressure [bar]	1.2
Mass flown through turbine [kg/hr]	131.22
Peak Cylinder Pressure [bar]	157.53
Brake thermal engine efficiency [%]	36

Table 4 Second design results

It is important to point out that the Peak Cylinder Pressure and the Knock Intensity are mean values of the four cylinders.

In order to evaluate how the Pareto curve changes as the exhaust gases temperature increases, other two HEEDS runs have been set, in each of them, the TEXH limit has been changed.

For the first run, the TEXH limit was set to 1350 K, as a consequence, for the second and the third run the constraints imposed have been: 1450 K and 1550 K respectively.

The Pareto curve showed the following trends:

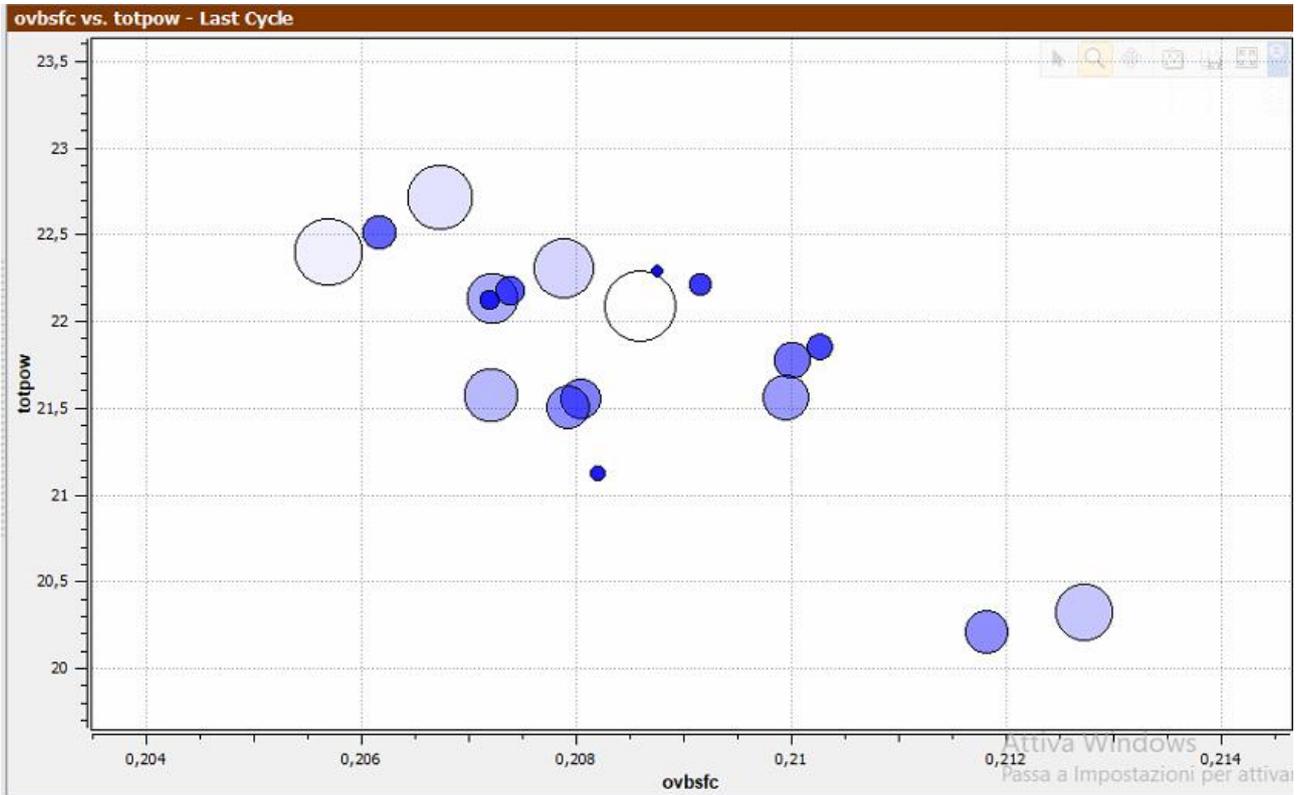


Figure 71 Pareto curve, TEXH limit set to 1450 K

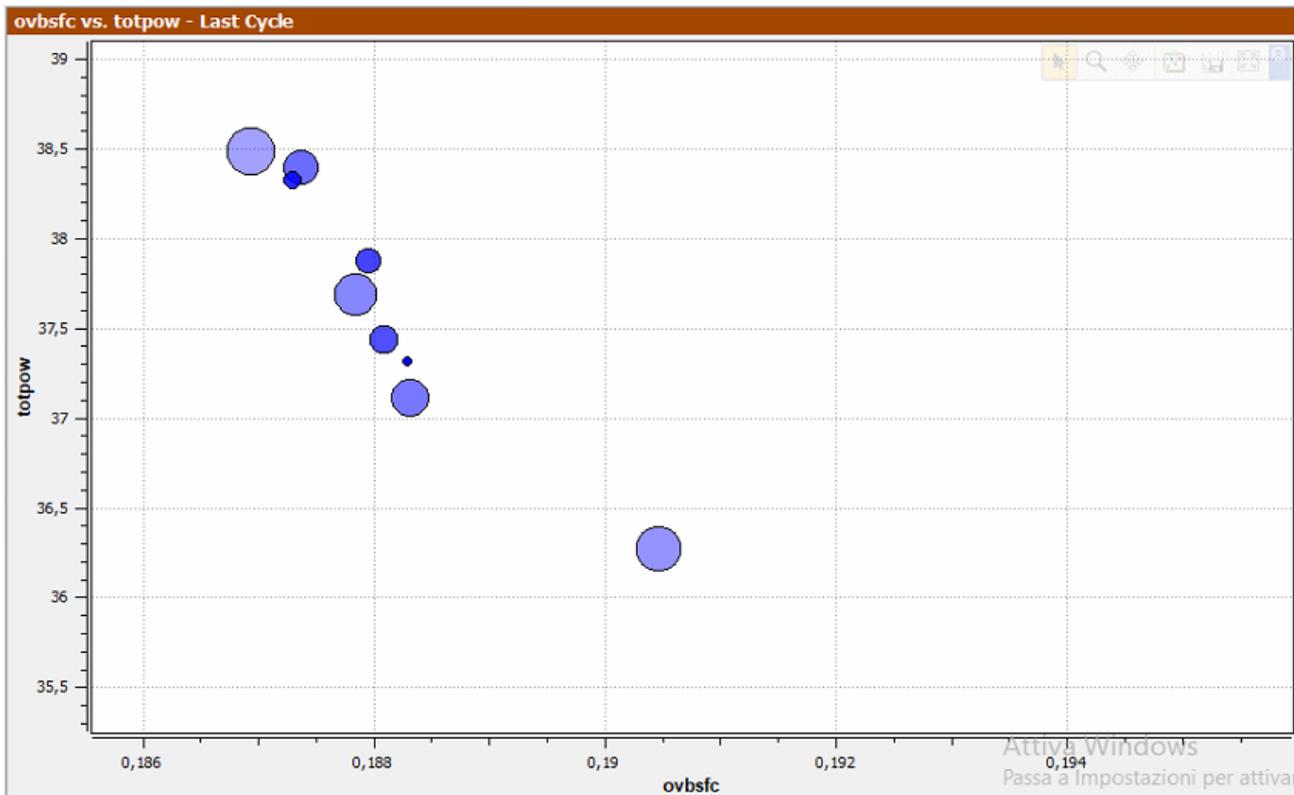


Figure 72 Pareto curve, TEXH limit set to 1550 K

By looking at the *Figure 68*, *Figure 71* and *Figure 72*, it is possible to notice that, considering the Pareto curve with the TEXH limit set to 1350 K and 1450K, there is no consistent change, but taking into account the case in which the limit was 1550 K, the changes are relevant, in fact the curve has shifted to higher power and lower BSFC values.

These results are also shown in the “objective history”:

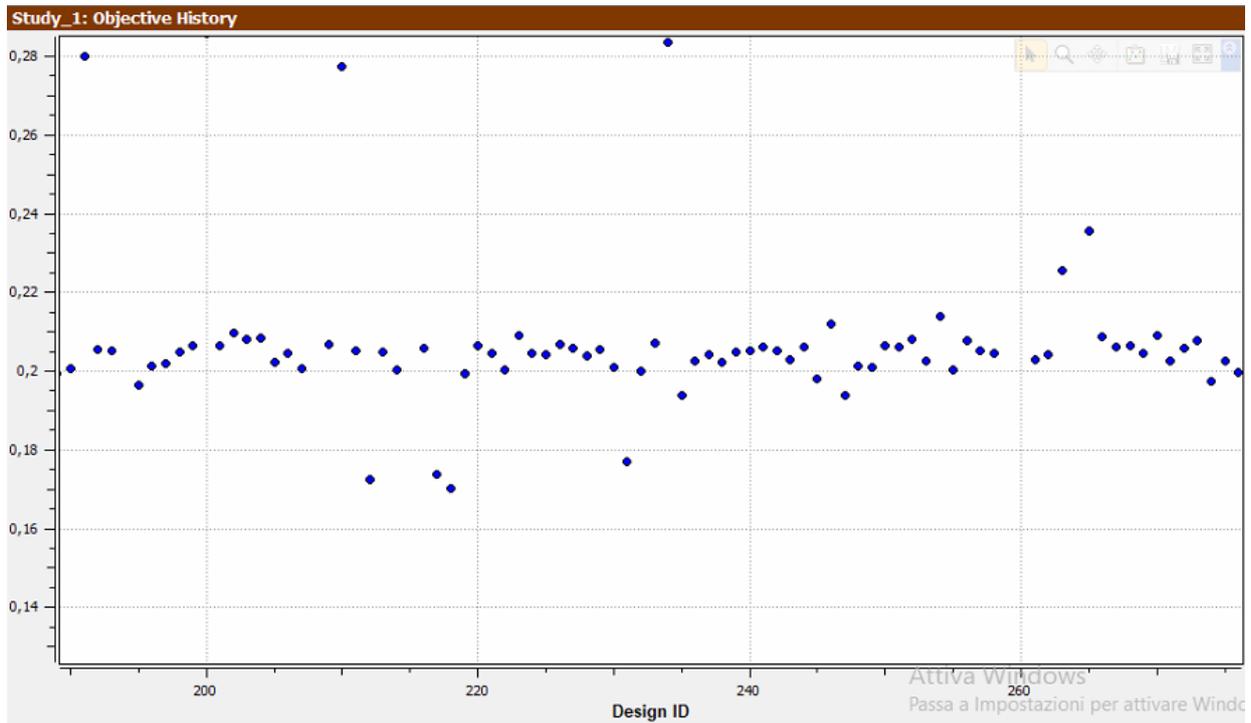


Figure 73 Objective history, TEXH = 1350 K

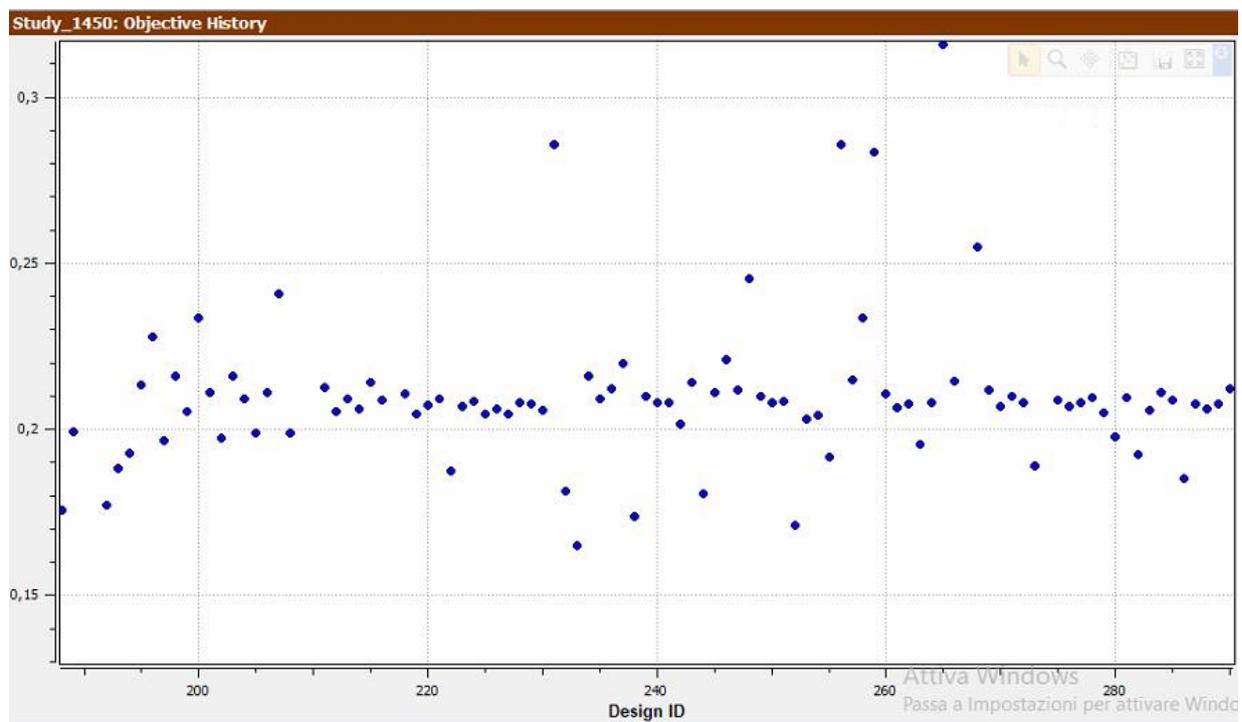


Figure 74 Objective history, TEXH = 1450 K

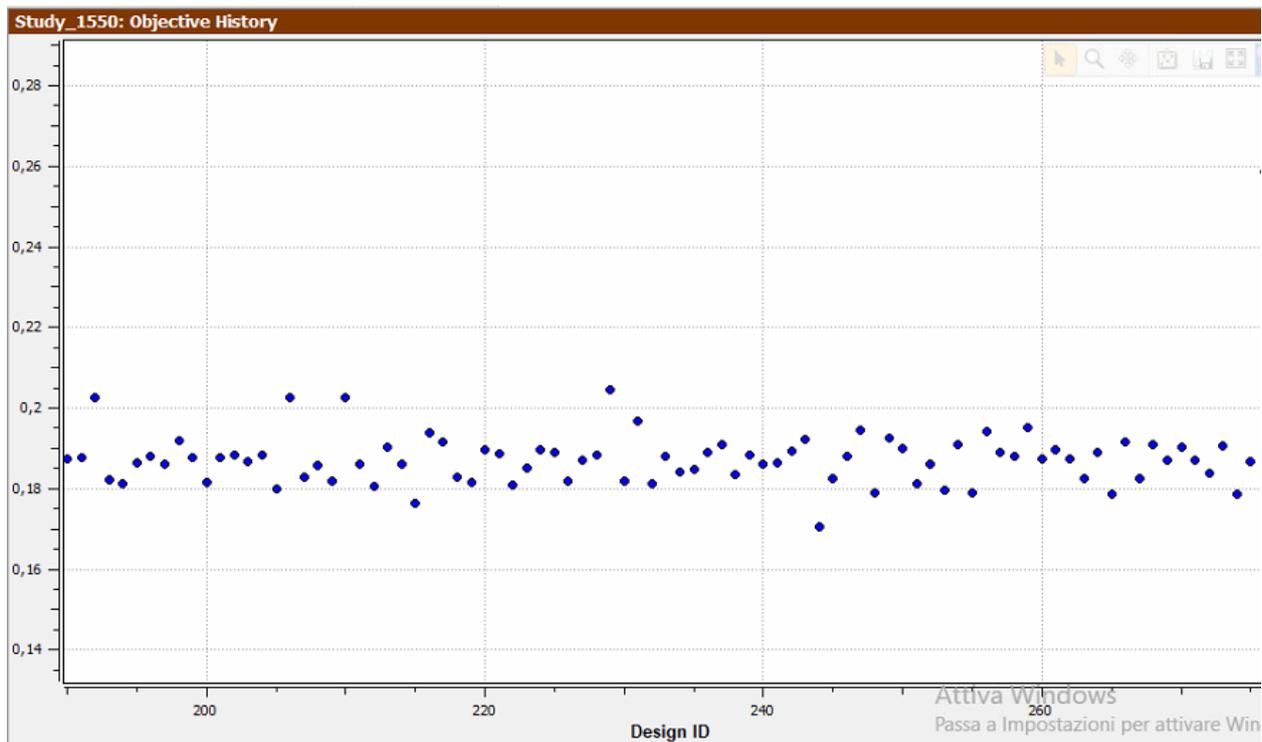


Figure 75 Objective history, $TEXH = 1550 K$

As it possible to see, the overall BSFC remains stable around the 200 g/kWh values, with some fluctuations of 5%, between 1350 K and 1450 K, but in the HEEDS run in which the TEXH limit has been set to 1550 K, the objective history shows a 10% decreasing in the overall BSFC.

9. Conclusions

The considerations made in the previous chapter, lead to the conclusion that the best design, i.e. the final design that has been chosen, was the second one, the results will be now reported again:

Total power [kW]	42.41
Overall BSFC [g/kW/hr]	181
Engine Power [kW]	33.09
Turbine produced power [kW]	20.30
Net power of the Turbogroup [kW]	9.31
Knock Intensity [-]	0.36
TEXH [K]	1563.92
Turbine Outlet Temperature [K]	1029.94
Turbine Inlet Temperature [K]	1442.35
Turbine Inlet Pressure [bar]	8.21
Turbine Outlet Pressure [bar]	1.2
Mass flown through turbine [kg/hr]	131.22
Peak Cylinder Pressure [bar]	157.53
Brake thermal engine efficiency [%]	36

Table 5 Design chosen

The results show that the aim of this work, i.e. the trade-off between the total power and the overall BSFC, in order to improve the value of the first factor, which was not enough for traction in the previous work, has been reached: the total power has doubled and the overall BSFC has decreased by 11% (see *table 1*).

Moreover, the valve lift value has also been improved, having eliminated the correlation with the intake duration, as a result, the value has increased from less than 1 mm to 4.5 mm.

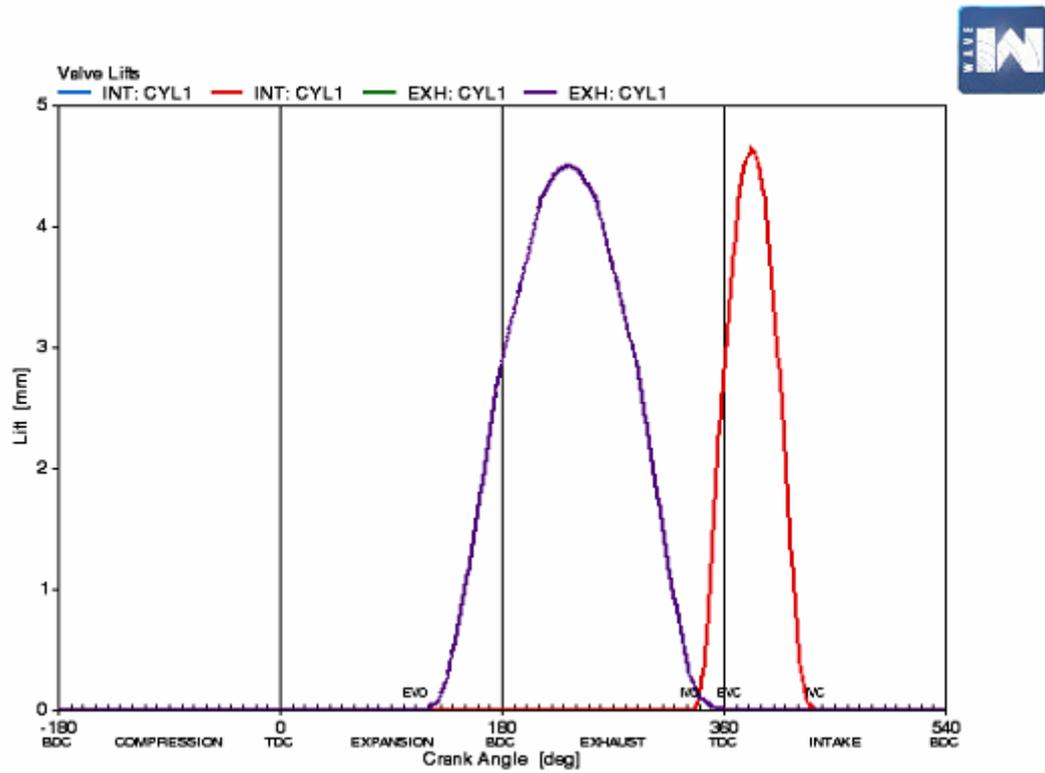


Figure 76 Valve Lifts

Linear p-V diagram, cylinder pressure and temperature trend will now be reported:

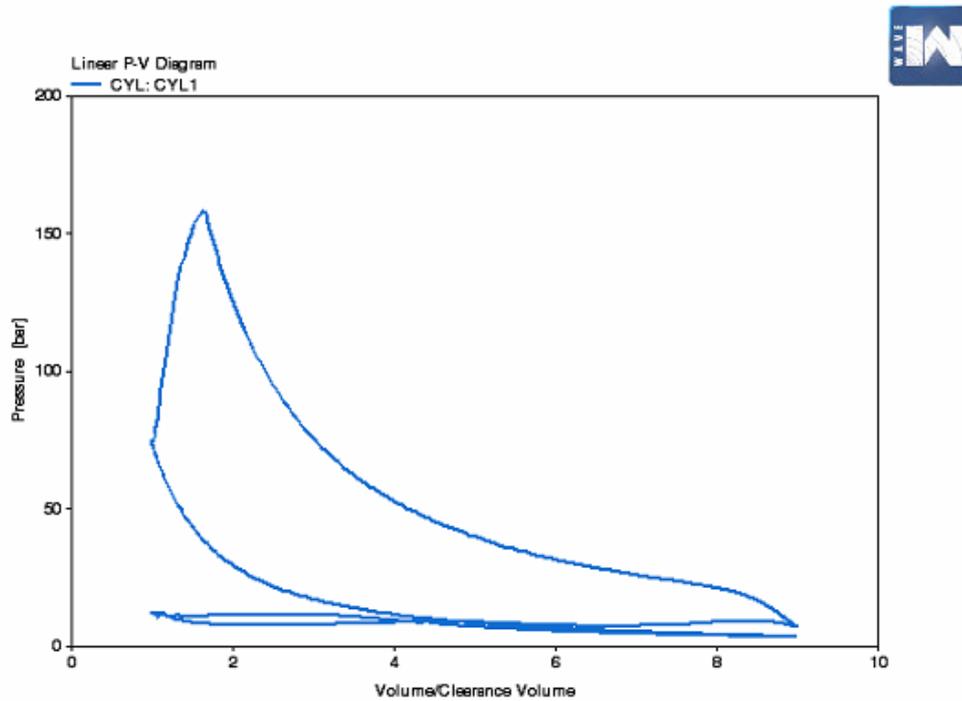


Figure 77 Linear p-V diagram

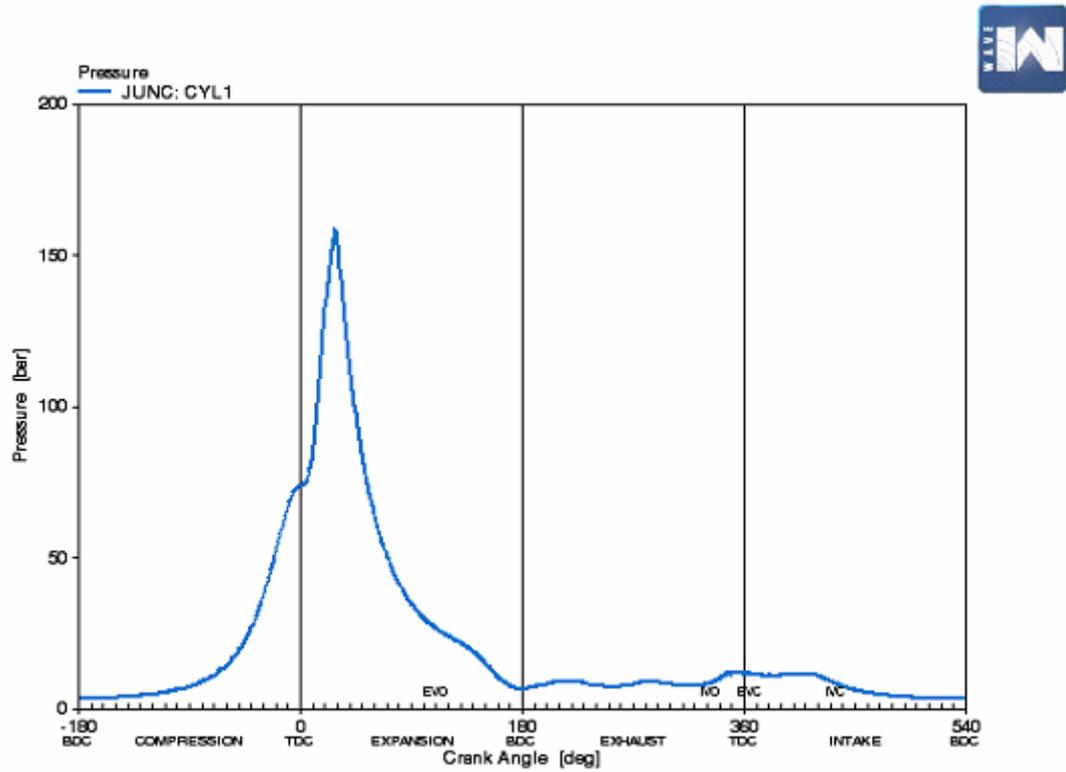


Figure 78 Cylinder pressure trend

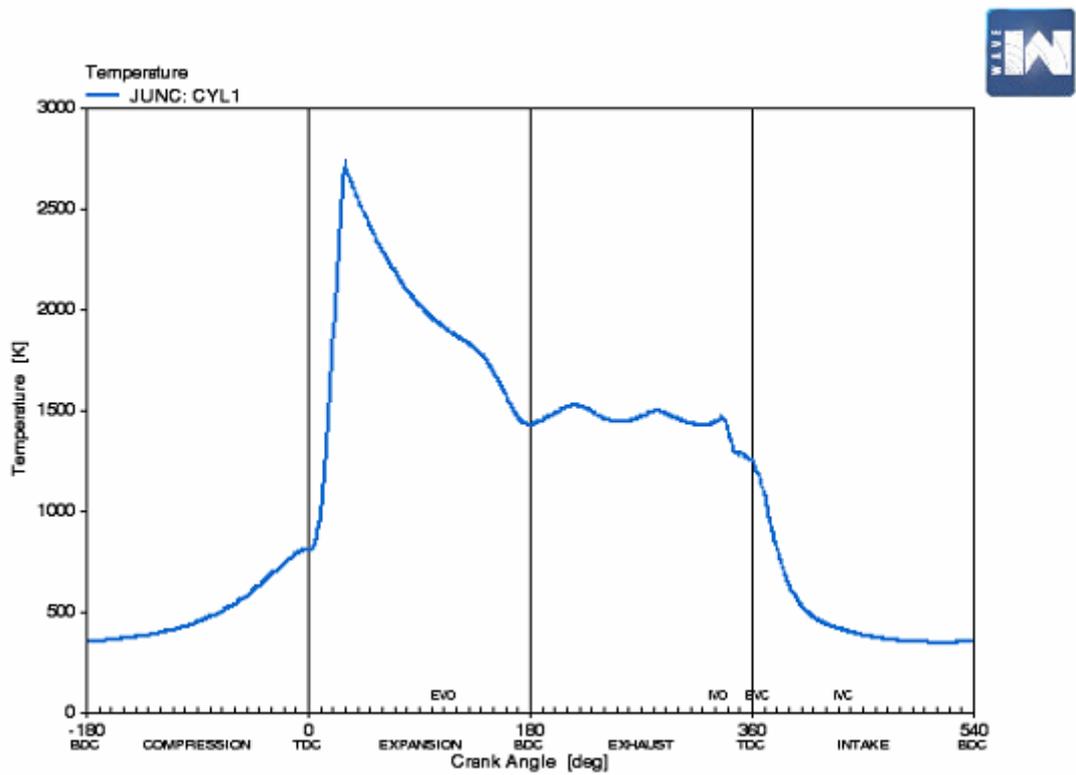


Figure 79 Cylinder temperature trend

Looking at *table 5*, what stands out is the TEXH value, which may seem too high, but considering the correlation table (*figure 70*), it is possible to see how, by choosing designs with much lower exhaust gases temperatures, the overall BSFC was 15-20% worse. Another important parameter to consider is the knock intensity, in this case much higher. Therefore, these designs provided high power, lower TEXH, but, on the other hand, worse overall BSFC and high knock intensity values.

This explains the choice of the final design, since it is possible to work only on the TEXH value, by selecting the right materials of which the exhaust valves should be made.

Using too expensive alloys could be a disadvantage, but considering the valves a very small part, in fact, the volume occupied by a valve is 2216.24 mm³, this disadvantage does not persist anymore.

Since the choice will have to fall on materials resistant to very high temperatures, the following have been taken into consideration: TZM alloy and Hexoloy SE SiC.

9.1 TZM Alloy

TZM and Molybdenum, molybdenum is similar to tungsten, with a melting point of 2620 °C, slightly lower than tungsten.

Molybdenum is transformed into TZM using small amounts of extremely fine carbides, it contains more than 99.2% up to 99.5% Mo, 0.50% and 0.08% titanium, zirconium and carbide.

TZM is more resistant than pure molybdenum and has a higher recrystallization temperature and better creep resistance. Recommended operating temperatures are between 700 and 1 400°C. TZM is used in high temperature applications involving high mechanical stress, e.g. in forging tools.

Due to its characteristics such as high temperature resistance and good thermal conductivity, TZM alloy is also used to produce mould parts for gravity casting and die-casting of aluminium castings or subject to erosion by liquid metal.

Chemical/physical characteristics:

- Melting point: 2620 °C
- Density: 10.15 g/cm³
- at 20°C - 6.0 m/m°C
- at 1000°C - 5.8 m/m°C
- at 1500°C - 6.5 m/m°C

Thermal conductivity:

- at 20°C - 125 w/m°C
- at 1000°C - 95 w/m°C
- at 1500°C - 85 w/m°C

Tensile strength: 590-790 MPa

Hardness: 230-320 DPH10

Elongation: 5-15%

Valve weight = 22,5 g

9.2 Hexoloy SE SiC

Hexoloy SE SiC, produced by Saint-Gobain, is the result of pressureless sintering of submicron silicon carbide powder in a proprietary extruding process. The sintering process results in a self-boned, fine grained (less than 10 micron) SiC product which is 95% dense.

Hexoloy SE SiC offers an excellent alternative material to metals, super-alloys and other ceramics for applications such as chemical processing, high temperature furnaces, and other demanding, severe environment applications. Hexoloy SE SiC provides a full range of exceptional properties in one package including:

- Extreme hardness;
- High strength;
- Virtually universal corrosion resistance;
- High temperature stability;
- High thermal conductivity.

Hexoloy SiC is one of the hardest high-performance materials available and is 50% harder than tungsten carbide. Its density is in excess of 95% of theoretical and it is completely impervious without the use of any impregnants, which means no contamination in high purity applications.

The single-phase composition of Hexoloy SE SiC enables it to reliably perform in air at temperatures in excess of 1650°C.

Where dimensional changes at high temperature are a concern, Hexoloy SE SiC has a consistently

low coefficient of thermal expansion. Because of its high thermal conductivity and low coefficient of thermal expansion, it is very resistant shock and will survive rapid thermal cycling as compared to other materials.

Typical properties:

- Maximum use temperature = 1900°C;
- Density = 3,05 g/cm³;
- Apparent Porosity = 5-10 %;
- Thermal Conductivity at 1200°C = 34,8 W/mK;
- Coefficient of thermal expansion = $4,02 \cdot 10^{-6} / ^\circ\text{C}$

Flexural Strength:

- Room Temperature = 280 MPa;
- At 1450°C = 270 MPa;
- At 1600°C = 300 MPa;

Modulus of Elasticity:

- At 20°C = 420 GPa;
- At 1300°C = 363 GPa

Valve weight = 6,76 g

10. Future works

The combustion process in an ICE is very complex, being WAVE a 1D simulation software, the analysis conducted in this work will need a 3D simulation software support, in order to evaluate the truthfulness of the values. This 3D analysis can be performed by another software developed by Ricardo, called VECTIS.



Figure 80 Ricardo Vectis

VECTIS is a next generation computational fluid dynamics (CFD) software product developed by Ricardo. It is a general-purpose tool for solving advanced 3D industrial fluid flow and heat transfer problems, with particular attention given to the requirements of automotive applications.

The product contains a fully automatic Cut-Cartesian mesh generator, a multi-domain solver capable of running on arbitrary unstructured meshes, and an advanced Graphical User Interface (GUI) to prepare geometry and display simulation results.

Typically, the multi-domain features are used to simulate Conjugate Heat Transfer (CHT) in automotive applications such as cooling of cylinder heads and engine blocks. VECTIS offers a Dynamic Cut-Cell capability for accurate simulations of problems involving moving boundaries in complex domains, such as Internal Combustion Engines.

The following figure provides a brief guide to the setting-up process (from start to finish) for a typical CFD simulation within VECTIS.

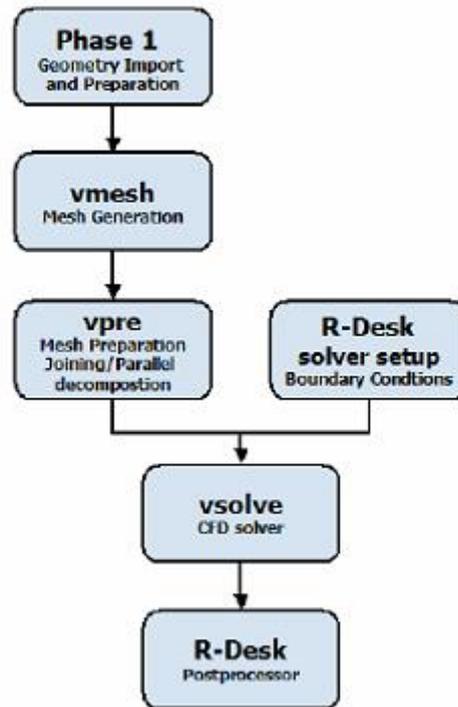


Figure 81 Vectis work-flow

At the end of the optimization process, a WAVE-VECTIS co-simulation will then be required. The purpose of coupled WAVE-VECTIS (1D-3D) engine performance simulations is to produce detailed 3D flow information for a specific component from the VECTIS analysis results whilst also predicting the performance of the complete engine using WAVE. Specific interfaces are setup in WAVE which allow WAVE and VECTIS to communicate at a time step level.

Flow information such as the fluid pressure, temperature, and velocity are passed between the two codes at these interfaces at each time step so as to allow for the prediction of transient flow fields in the VECTIS simulation.

10.1 Setting up the WAVE model

When the WAVE model has been setup, calibrated and run successfully as a standalone simulation it can then be modified to include the setup for the coupled interfaces and the Ricardo coupled analysis strategy.

The first setup requirement for a WAVE- VECTIS analysis is the EXTERNAL CFD junction run parameters:

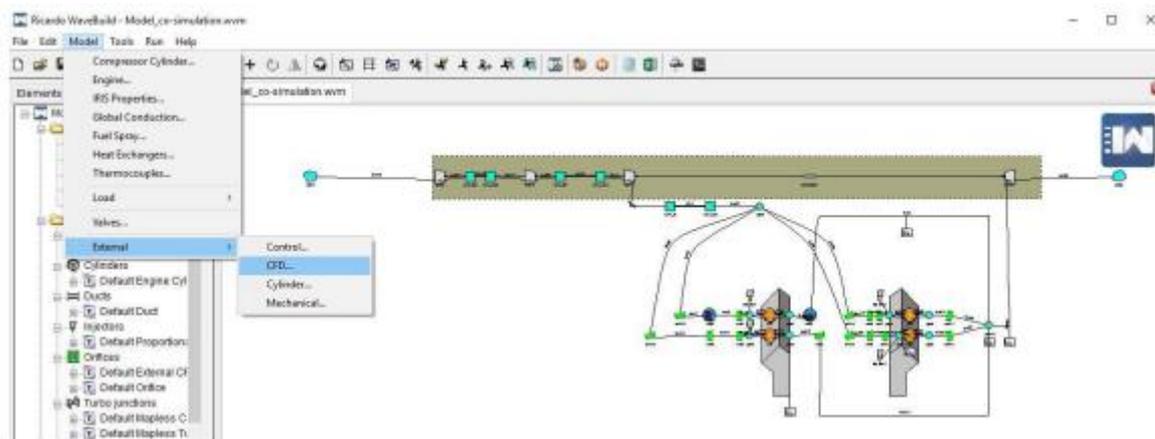


Figure 82 External CFD set-up

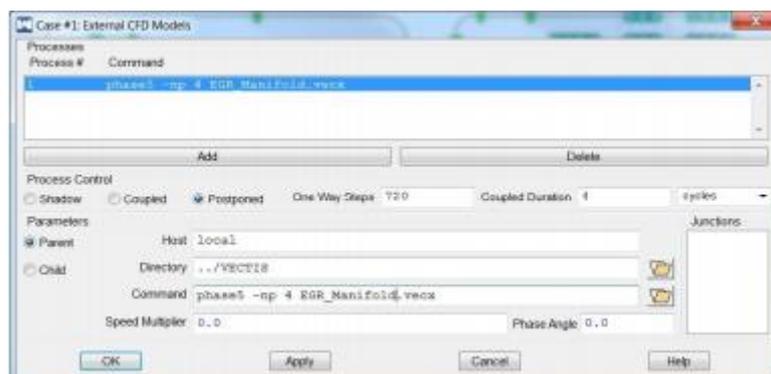


Figure 83 External CFD set-up panel

In the external CFD set-up panel, the user sets “postponed” radio bottom, in order to postpone the 1D-3D coupling until the WAVE simulation has reached a converged solution. When convergence has been reached in the WAVE only part of the simulation, the simulation will start running what is referred to as one-way coupling. The number that is input by the user in the “One Way Steps” input field represents the number of VECTIS iterations that will run until the simulation switches to full 1D-

3D coupling mode. The VECTIS simulation will use a time step of 0.25 crank angle degrees and 720 one-way steps are needed to achieve 180 degrees of engine motion.

During this one-way step time period, WAVE will pass VECTIS information but VECTIS will not pass WAVE any information. This will allow the VECTIS flow field to initialize before the two-way coupling.

After the 180 degrees of engine crank angle of both WAVE and VECTIS will pass each other information in what is known as two-way coupling. The total coupled duration, including the one-way steps is specified in the Coupled Duration input field.

11. Bibliography

1. Millo F., PROPULSORI TERMICI, Torino, (2017);
2. Heywood J. B., INTERNAL COMBUSTION ENGINES FUNDAMENTALS, McGraw-Hill Education, (1988);
3. Spessa E., HEV INTRODUCTION AND ARCHITECTURES, Torino, (2017);
4. <https://it.motor1.com/features/212607/auto-ibride-una-storia-secolare/>;
5. Mariani C. A., *ONE-DIMENSIONAL MODEL OF A RANGE EXTENDER WITH HIGH POWER DENSITY AND DEEP MILLER CYCLE DEVELOPED WITH RICARDO WAVE SOFTWARE*, Torino, (2018);
6. Tricarico A., *DESIGN LAYOUT OF VALVETRAIN AND BALANCING OF A FOUR STROKE SPARK IGNITION ENGINE FOR RANGE EXTENDER APPLICATION*, Torino, (2018);
7. Chierchia S., RANGE EXTENDER ENGINE WITH VERY HIGH POWER DENSITY, SUITABLE FOR THE USE IN A HYBRID PROPULSION SYSTEM: DEEP MILLER CYCLE OPTIMIZATION WITH ADVANCED CFD 1D-3D VIRTUAL VALIDATION, Torino, (2019);
8. Ricardo software, *WAVE USER MANUAL*, Van Buren Township, (2018);
9. Ricardo software, *VECTIS VSOLVE USER MANUAL*, Van Buren Township, (2019);
10. https://it.wikipedia.org/wiki/Ciclo_Miller;
11. <https://www.autotecnica.org/ciclo-atkinson-ciclo-miller-e-motore-hcci/>;
12. <https://www.quora.com/What-is-a-Miller-cycle>;
13. Science Direct: THE MILLER CYCLE EFFECTS ON IMPROVEMENT OF FUEL ECONOMY IN A HIGHLY BOOSTED, HIGH COMPRESSION RATIO, DIRECT-INJECTION GASOLINE ENGINE: EIVC VS. LIVC;
14. Red Cedar Technology, GETTING STARTED WITH HEEDS, (2007);
15. <https://www.ceramicsrefractories.saint-gobain.com/products/hexoloy-sic-materials>;
16. <https://www.teknosteelsrl.com/prodotti/molibdeno/lega-di-molibdeno/>;
17. <https://www.outils.it/tzm-molibdeno/>