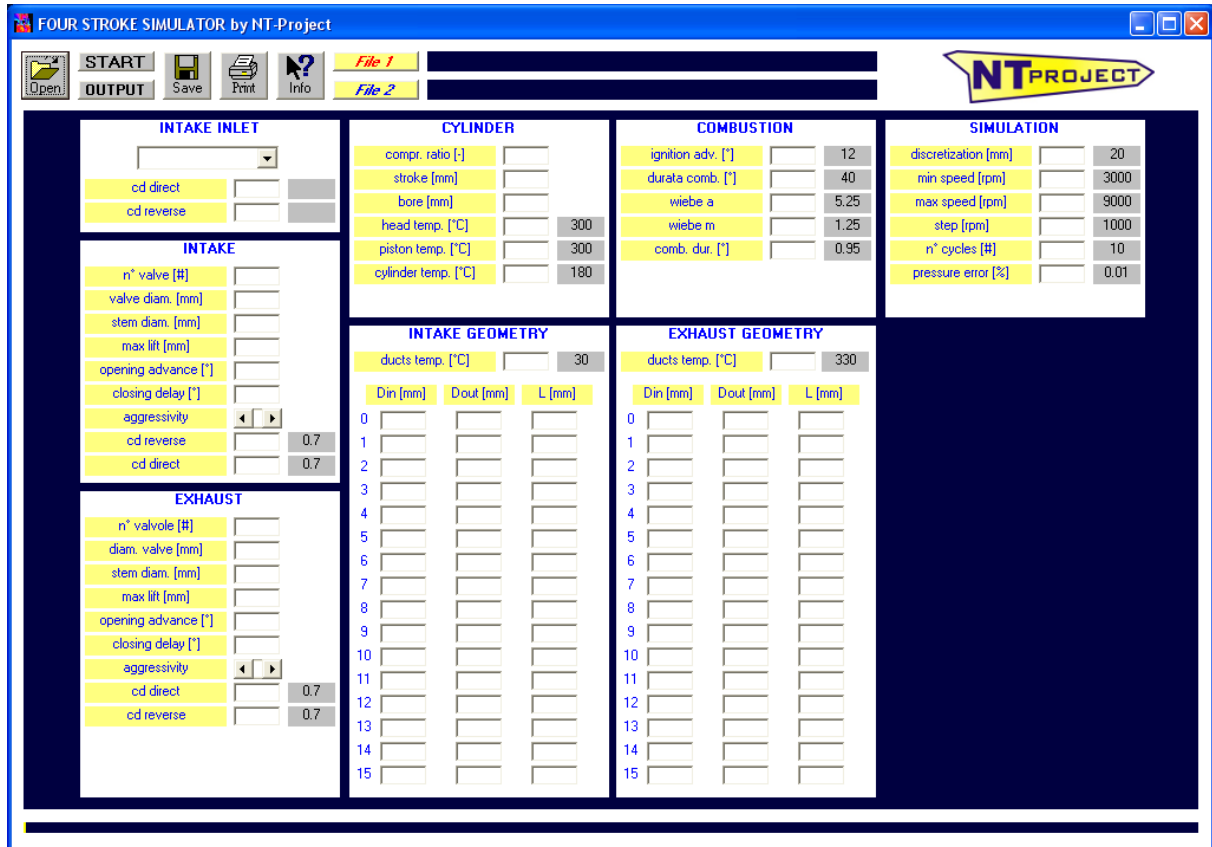


PRESENTATION SOFTWARE FOUR STROKE SIMULATOR

When opening the software appears this screenshot:





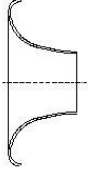
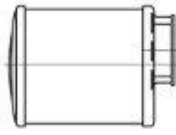
data entering

In the column on the left you must enter the features of the elements that define:

- air inlet in the intake system INTAKE INLET;
- air inlet in the crankcase CRANKCASE INLET;
- inlet and outlet of the air in the cylinder TRANSFER and EXHAUST PORTS.



You must select how the air enters in the intake system:

airbox	airbox-bellmouth	bellmouth	filter
			

INTAKE	
n° valve [#]	<input type="text"/>
valve diam. [mm]	<input type="text"/>
stem diam. [mm]	<input type="text"/>
max lift [mm]	<input type="text"/>
opening advance [°]	<input type="text"/>
closing delay [°]	<input type="text"/>
aggressivity	<input type="range"/>
cd reverse	<input type="text"/> 0.7
cd direct	<input type="text"/> 0.7
EXHAUST	
n° valvole [#]	<input type="text"/>
diam. valve [mm]	<input type="text"/>
stem diam. [mm]	<input type="text"/>
max lift [mm]	<input type="text"/>
opening advance [°]	<input type="text"/>
closing delay [°]	<input type="text"/>
aggressivity	<input type="range"/>
cd direct	<input type="text"/> 0.7
cd reverse	<input type="text"/> 0.7

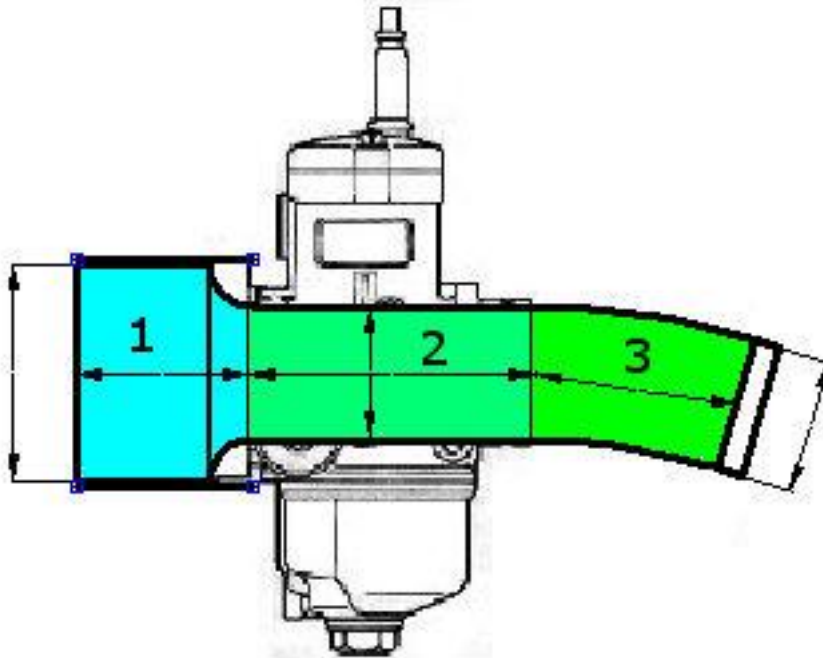
For intake and exhaust you must enter these data. For the valve diameter we refer to the seat diameter of the valve, instead for the timing (advance and delay) the measurement is referred with lift at the valve clearance. The stem diameter can be omitted, the software will use a value the measure wiPer aspirazione e scarico vanno inseriti questi dati. Per il diametro delle valvole si intende quello della sede valvola, mentre per le fasature (anticipo e ritardo) si intendono misurate a gioco valvola. Il diametro dello stelo può essere omesso, il software considererà a value equal to 0.2 times the valve diameter. The lift law hypothesized in the software is polydyne type, and moving the "aggressivity" bar you'll have a lift more aggressive.

The software has already preset the direct and reverse flow coefficient values for the ducts inside the head, anyway if specific flowbench tests are carried out, it is possible to enter manually the values obtained experimentally.

Then in the software must be entered the geometry of the ducts that go from:

- intake inlet to head inlet INTAKE GEOMETRY;
- head outlet to exhaust outlet EXHAUST GEOMETRY;

INTAKE GEOMETRY



The geometry of the intake system must be inserted considering a series of sections that start from the filter side and which corresponds to section 1, until to the crankcase side.

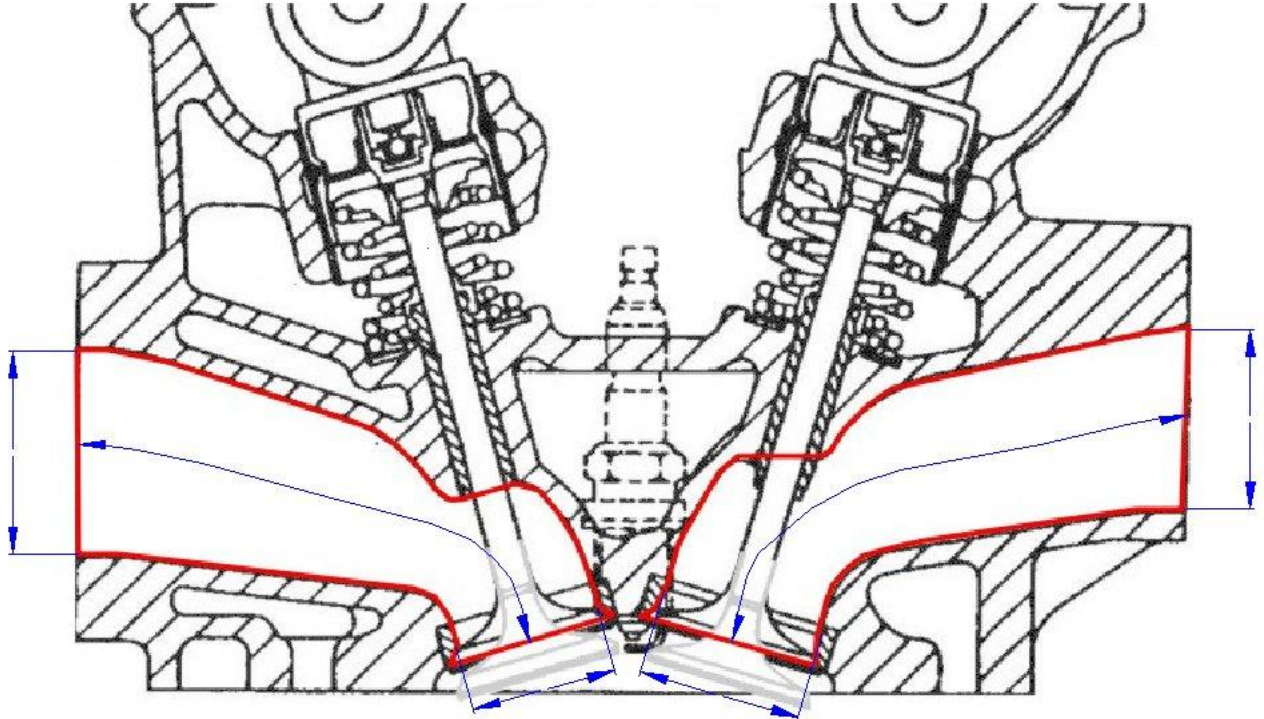
		INTAKE GEOMETRY		
<p>For each section must be entered:</p> <ul style="list-style-type: none"> - initial diameter D_{in} [mm]; - final diameter D_{out} [mm]; - length (considered on the axis of the duct) L [mm] <p>In addition to these data, it is possible to enter the value of the average wall temperature of the ducts; if you leave the field empty, the software will use the default value of 30 °C</p>		ducts temp. [°C]		30
		D _{in} [mm]	D _{out} [mm]	L [mm]
1	<input type="text"/>	<input type="text"/>	<input type="text"/>	
2	<input type="text"/>	<input type="text"/>	<input type="text"/>	
3	<input type="text"/>	<input type="text"/>	<input type="text"/>	
4	<input type="text"/>	<input type="text"/>	<input type="text"/>	
5	<input type="text"/>	<input type="text"/>	<input type="text"/>	
6	<input type="text"/>	<input type="text"/>	<input type="text"/>	
7	<input type="text"/>	<input type="text"/>	<input type="text"/>	
8	<input type="text"/>	<input type="text"/>	<input type="text"/>	
9	<input type="text"/>	<input type="text"/>	<input type="text"/>	

A typical case, shown in the figure above, will have:

- section 1 – sleeve between airbox/filter and carburetor or throttle body
- section 2 – carburetor or throttle body
- section 3 – manifold between carburetor/ throttle body and head

In addition to the sections external to the head, in the intake geometry must be entered also the dimensions of duct inside the head.

You must enter the equivalent diameter of head inlet, the equivalent diameter of the seat valves, and the duct length (considered on the axis).



EXHAUST GEOMETRY

The first section that you need to consider is the exhaust duct inside the head. You must enter the equivalent diameter of the seat valves, the equivalent diameter of head outlet, and the duct length (considered on the axis).

Then you must enter the geometry of the exhaust manifold, and of the exhaust muffler until the silencer.

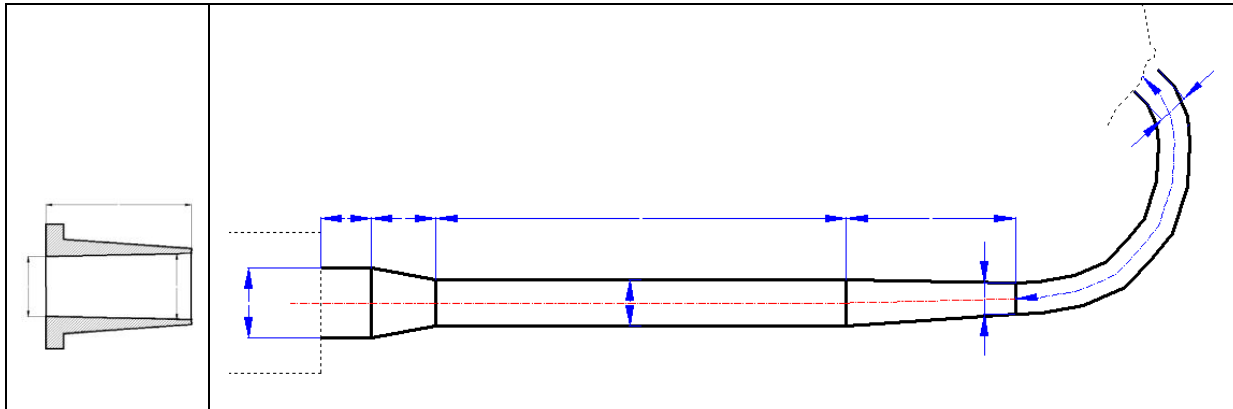
For each section must be entered:

- initial diameter D_{in} [mm];
- final diameter D_{out} [mm];
- length (considered on the axis of the duct) L [mm]

In addition to these data, it is possible to enter the value of the average wall temperature of the ducts; if you leave the field empty, the software will use the default value of 330 °C

EXHAUST GEOMETRY			
ducts temp. [°C]	<input type="text" value="330"/>		
	D_{in} [mm]	D_{out} [mm]	L [mm]
0	<input type="text"/>	<input type="text"/>	<input type="text"/>
1	<input type="text"/>	<input type="text"/>	<input type="text"/>
2	<input type="text"/>	<input type="text"/>	<input type="text"/>
3	<input type="text"/>	<input type="text"/>	<input type="text"/>
4	<input type="text"/>	<input type="text"/>	<input type="text"/>
5	<input type="text"/>	<input type="text"/>	<input type="text"/>
6	<input type="text"/>	<input type="text"/>	<input type="text"/>
7	<input type="text"/>	<input type="text"/>	<input type="text"/>
8	<input type="text"/>	<input type="text"/>	<input type="text"/>
9	<input type="text"/>	<input type="text"/>	<input type="text"/>

exhaust manifold and exhaust muffler



At this point, only the data relating to the cylinder remain to be entered.

For the cylinder you must enter the main specifications (stroke and bore), and the compression ratio.

In addition to these data, it is possible to enter the value of the average wall temperature of:

- head (default 300 °C)
- piston (default 300 °C)
- cylinder (default 180 °C)

CYLINDER

compr. ratio [-]	<input type="text"/>	
stroke [mm]	<input type="text"/>	
bore [mm]	<input type="text"/>	
head temp. [°C]	<input type="text"/>	300
piston temp. [°C]	<input type="text"/>	300
cylinder temp. [°C]	<input type="text"/>	180

Finally, in the software it is possible to manage the ignition and combustion parameters.

For ignition you can enter the advance value considered as crank angle degrees before top dead centre TDC. The default value is 12 °.

For combustion you can enter:

- duration (default 40 °);
- efficiency (default 0.95)

and furthermore it is possible to manage two parameters that define the combustion trend according to a well-known law defined by Wiebe:

- parameter "a" which identifies the efficiency, higher is the value and greater will be the efficiency (default value 5.25);
- parameter "m" which identifies the combustion speed (default value 1.25)

COMBUSTION

ignition adv. [°]	<input type="text"/>	12
comb. dur. [°]	<input type="text"/>	40
wiebe a	<input type="text"/>	5.25
wiebe m	<input type="text"/>	1.25
comb. eff. [-]	<input type="text"/>	0.95

Before starting the calculation in the software it is possible to define the characteristics of the simulation.

The discretization, the number of cycles, and the pressure error allowed, values influence the precision and the speed of the calculation. For the discretization the default value is 20 mm. For the number of cycles defines the default value is 10. For the pressure error allowed between two cycles the default is 0.01%.

Finally, you must enter the engine speed values from which you want to start calculating torque and power (default 3000), and the final one (default 9000), indicating every how many revolutions (default 1000) you want to carry out the calculation between initial speed and final speed.

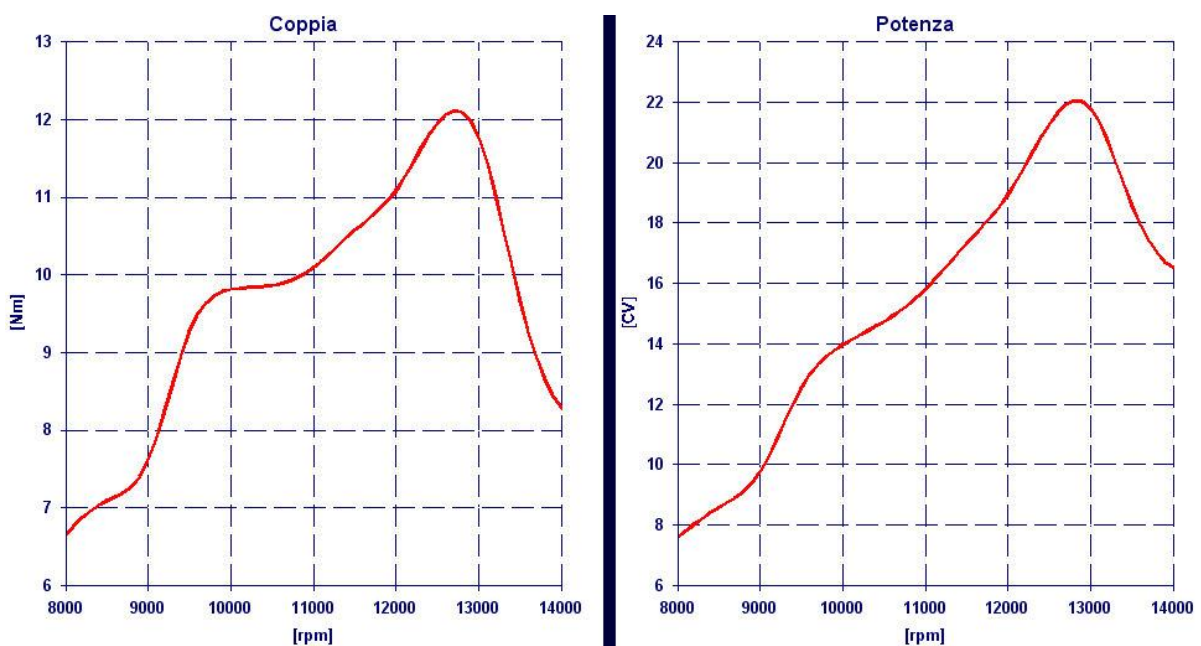
SIMULATION		
discretization [mm]	<input type="text"/>	20
min speed [rpm]	<input type="text"/>	3000
max speed [rpm]	<input type="text"/>	9000
step [rpm]	<input type="text"/>	1000
n° cycles [#]	<input type="text"/>	10
pressure error [%]	<input type="text"/>	0.01

After entering the data described above, you can start the calculation by pressing the START button. The software shows the progress of the calculation with a yellow bar at the bottom which indicates how much remain to be completed.



results

At the end of the calculation, pressing the OUTPUT button you can see the torque and power curves.

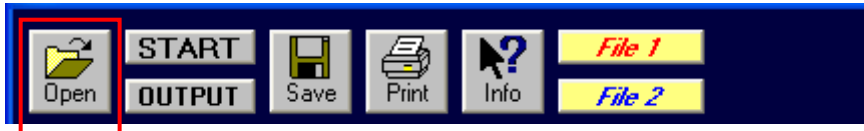


open and save data file

The data can be saved in a file * .tss through the "SAVE" button.



The stored data can be easily retrieved through the button "OPEN".



comparison results

The software allows you to compare the results with those of a previously saved solution.

To compare two results, after the calculation and saved the results, you must open the first file that you want compare from the button "Open" or the button "File1", and the second file from the button "File2".



The software will automatically show you the comparison (those of File1 in red, and those of File2 in blue). If you have loaded the File2, this will be maintained even when you perform a new calculation, so if you want to take a result as a reference, you can load it as File2, and then make the modifications on the File1, to start the calculation, and directly to compare the results.

print

At the end of the calculation you can print the input and output through the PRINT button.

